Advanced Geometry, Meshing, Customization, and Variable LBCs using Patran

PAT302 Course Notes

June 2012

MSC Software®
Simulating Reality, Delivering Certainty™

P3*V2012*Z*Z*Z*SM-PAT302-NT

Legal Information

MSC.Software Corporation reserves the right to make changes in specifications and other information contained in this document without prior notice. The concepts, methods, and examples presented in this text are for illustrative and educational purposes only, and are not intended to be exhaustive or to apply to any particular engineering problem or design. MSC.Software Corporation assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Copyright © 2012 MSC.Software Corporation. All Rights Reserved. This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MSC.Software Corporation is prohibited.

The MSC.Software corporate logo, Adams, Dytran, Easy5, Fatigue, Laminate Modeler, Marc, Mentat, MD Nastran, Patran, MSC, MSC Nastran, Mvision, Patran, SimDesigner, SimEnterprise, SimManager, SimXpert and Sofy are trademarks or registered trademarks of the MSC.Software Corporation in the United States and/or other countries. NASTRAN is a registered trademark of NASA. All other trademarks belong to their respective owners.

CONTENTS

Section		Page
1	**Overview**	1-1
	Objectives of the course	1-3
2	**Introductory Comments on Meshing**	2-1
	Overview of strategy	2-3
	Types of idealization	2-5
	Many possible paths through Patran	2-6
	Example: Geometry Cleanup	2-9
3	**Strategies in Geometric Modeling and Meshing, 2D Surface Meshing**	3-1
	Idealization Type	3-3
	2D Beginning to End	3-4
	Paver (2D) Meshing	3-7
	Case Study: Paver Parameters	3-8
	Meshing Surfaces Using Mesh Control	3-19
	Associated Geometry	3-26
	Case Study: 2D Meshing	3-38
	2D Meshing & Surface Topology	3-56
	Geometry Editing	3-64
	Case Study: Computer Panel	3-106
	Workshop 1: Surface Edit and Mesh, Instrument Panels	
	Geometry Editing, Composite Surface	3-120
	Case Study: Create Composite Surface	3-122
	Workshop 2: Composite Surfaces	
	Case Study: Trimmed Surface, Composite	3-183

Copyright© 2012 MSC.Software Corporation

CONTENTS

Section		Page
3	**Strategies in Geometric Modeling and Meshing, 2D Surface Meshing**	
	Tessellated Surface	3-218
	Case Study: Create 2D Mesh From Non-congruent Surfaces	3-221
	Workshop 3: Tessellated Surface	
	Non-congruent Meshes and Mesh Modification	3-246
	Sheet Body Mesher	3-248
	Case Study: Using Sheet Body Mesher	3-256
	Mesh On Mesh	3-261
	Case Study: Mesh on Mesh 1	3-262
	Workshop 4: Mesh On Mesh 2D	
	Case Study: Mesh on Mesh 2	3-279
	Iso-meshing of Parasolid Solid Faces	3-284
	Advanced Surface Mesher (ASM)	3-287
	Case Study: ASM Example	3-292
4	**Strategies in Geometric Modeling and Meshing, Combined 2D and 3D Meshing**	4-1
	Idealization Type	4-3
	3D Start Point, 2D Idealization	4-4
	Mid-surface Extraction	4-7
	Workshop 5: Mid-surface Creation, Simple Bracket	
	Workshop 6: Mid-surface Extraction, Wing Section	
	Mid-plane Mesher	4-26
	Workshop 7: Mid-plane Mesher	
	3D Start Point, 2D Idealization	4-42

Copyright© 2012 MSC.Software Corporation

CONTENTS

Section		Page
4	**Strategies in Geometric Modeling and Meshing, Combined 2D and 3D Meshing**	
	Case Study: Carrier 3D -> 2D	4-44
	Workshop 8: Mesh On Mesh 2D to 3D	
	3D Start Point, 2 1/2D Idealization	4-62
	Case Study: Caliper 3D -> 2D/2 ½ D	4-76
	Workshop 9: Hex Mesh Using Sweep Vector	
5	**Strategies in Geometric Modeling and Meshing, 3D Meshing**	5-1
	Idealization Type	5-3
	3D Beginning to End	5-4
	Case Study: Tetmeshing	5-9
	Avoid Negative Jacobian	5-30
	Control TetMesh at Detail Using 2D Elements	5-34
	Control TetMesh Using Proximity Based Meshing	5-45
	3D Mesh Editing	5-47
	Modify TetMesh Using Collapse Ratio and/or Jacobian Parameter	5-48
	Case Study: Improve TetMesh Using Modify	5-51
	3D IsoMesh & Solid Topology	5-63
	3D Beginning to end	5-69
	3D IsoMesh & Solid Topology (Cont.)	5-75
	Create Hexahedral Element Mesh for Complex Geometry	5-78
	Workshop 10: Hex & Tet Solid Element Mesh	
	Building B-rep Solids	5-82
	Workshop 11: Edit Solid Blade	

Copyright© 2012 MSC.Software Corporation

CONTENTS

Section		Page
5	**Strategies in Geometric Modeling and Meshing, 3D Meshing**	
	Primitive Solid Modeling Tools	5-91
	Case Study: Lamp Housing	5-121
	Workshop 12: Assembly Meshing, Tension Fitting	
	Workshop 13: Solid Modeling of K-T Joint	
	Workshop 14: Solid Modeling of Plastic Housing	
	Parasolid Solid Assembly Meshing Tools	5-138
	Case Study: Assembly Mesh Ex. 1	5-140
	Case Study: Assembly Mesh Ex. 2	5-159
	Assembly Meshing of Multiple Solids in One Operation	5-164
	Case Study: Assembly Mesh Ex. 3	5-167
	Gaps and Intersections in Assembly Meshing	5-178
	Sliver/Degenerate Solid Faces for Parasolid Assemblies	5-192
6	**Customization, Tailoring Work Environment**	
	Topics Discussed	6-3
	Files	6-5
	Command Line Arguments	6-28
	Environment Variables	6-36
	Using the GUI	6-43
	Shortcuts	6-61
	Change Solver Preference	6-71
	Workshop 15: Editing the Toolbar	

Copyright© 2012 MSC.Software Corporation

CONTENTS

Section		Page
6	**Customization, Tailoring Work Environment**	
	Workshop 16: Keymapping	
7	**Import CAD Geometry into Patran**	
	Geometry Import Methods	7-3
	Import Parasolid xmt File	7-5
	Import ACIS File	7-21
	Import CATIA Model File	7-37
	Import Pro/engineer Model File	7-55
	Import Unigraphics Model File	7-73
	Import IGES Model File	7-91
	Import STEP Model File	7-97
	Workshop 17: Geometry Import and Repair	
	Workshop 18: Editing Imported Geometry with the Feature Recognition Tool	
	Workshop 23: Introduction to Parameters and Features (Optional)	
	Workshop 24: Modifying Unigraphics Models and Paramters (Optional)	
8	**Fields**	
	Types of Fields	8-4
	Basics of Spatial Fields	8-5
	Case Study 1: Bolt Loading Represented by Cos^2 Field	8-11
	Case Study 2: Spatial/PCL Function Fields, Parametric	8-31
	Fields, Spatial/PCL Function, Vector	8-36
	Case Study 3: Varying Traction Load on a Spike	8-37
	Fields, Spatial/Tabular Input, Real	8-47
	Fields, Spatial/Tabular Input, Parametric	8-48
	Case Study 4: Radiation to Tubes	8-53

Copyright© 2012 MSC.Software Corporation

CONTENTS

Section		Page
8	**Fields**	
	Fields, Spatial/FEM, Discrete	8-65
	Fields, Spatial/FEM, Continuous	8-66
	Case Study 5: Modification of Mass Through Fields	8-69
	Case Study 6: FEM Field, Continuous, CFD Pressure to Patran Pressure	8-77
	Workshop 19: Global/Local Modeling Using FEM Fields	
	Fields, Material Property/Tabular Input	8-101
	Fields, Material Property/General	8-104
	Fields, Material Property/Model Variable	8-105
	Case Study 7: Material Property Field	8-109
	Fields, Non-spatial/Tabular Input	8-115
9	**PCL for Non-programmers**	
	Overview of PCL	9-3
	PCL Basics	9-4
	Variable Declaration	9-7
	Define the PCL Expressions	9-8
	Compile the PCL Function	9-9
	Case Study: Change Black to White	9-12
	Case Study: Making a Custom View	9-14
	Case Study: Function for Pressure	9-25
	More on Variable Declaration	9-31
	More on PCL Expressions	9-34
	PCL Directives	9-36

Copyright© 2012 MSC.Software Corporation

CONTENTS

Section		Page
9	**PCL for Non-programmers**	
	Starting Patran, Briefly	9-39
	Where to go for Help	9-41
	Workshop 20: Creating PCL Functions	
	Workshop 21: PCL via Session File, Pipe Intersection	
10	**Results**	
	Fringe Plots of Scalar Results	10-3
	Case Study: Fringe Plot Options	10-4
	Spectrums and Ranges	10-36
	Creating a Range	10-40
	Results Processing Settings	10-55
	Case Study: Visualize Open or Close of Gaps	10-59
	Using FEM Field With Results	10-73
	Case Study: Freebody of Simple Model	10-78
	XY Plot	10-90
	Case Study: XY Plot for Analysis Versus Test Data	10-91
	Read Results Using Neutral File	10-97
	Case Study: Read Foreign Results	10-98
	Results Templates	10-104
	Results Plot Sets	10-113
	Workshop 22: Postprocessing Using Plot Sets	

Copyright© 2012 MSC.Software Corporation

SECTION 1

INTRODUCTION

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

OBJECTIVES OF THE COURSE

- **Course Objectives**
  - One of the key objectives in this course is to make the students aware of the strategies available in geometric modeling and meshing, and how to implement these after having defined clear analysis objectives. Realistic case studies are used to discuss various topics. A review of handy geometry construction and direct mesh editing will be presented. Essential QA procedures will be presented for checking both geometry and mesh accuracy, and quality.
  - To give the students the ability to customize Patran to improve productivity via techniques ranging from creating toolbar icons through to writing simple applications in the Patran Command Language (PCL).

Copyright© 2012 MSC.Software Corporation

OBJECTIVES OF THE COURSE

- **Course Objectives**
  - Techniques for fixing CAD geometry, which may be corrupted or poorly defined for the requirements of the analysis model, will be described.
  - All types of fields will be discussed. Several case studies are presented to show how to use fields in various ways. The case studies cover such things as using PCL Function for bolt loads or beam cross-section definition, spatial tabular for temperature LBC definition, discrete FEM field to modify mass properties, and continuous FEM field to apply CFD pressure data to a structural model.
  - Provide students with a greater insight into the vital task of post-processing analysis results via the many tools available.

Copyright© 2012 MSC.Software Corporation

SIMCOMPANION

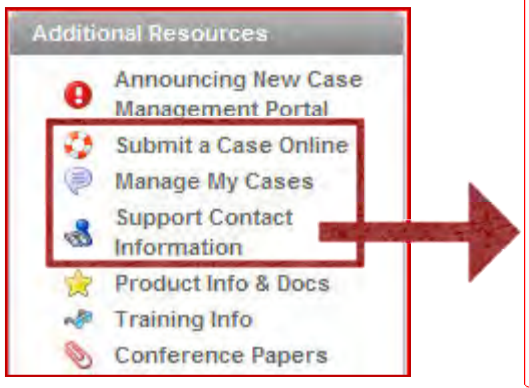
- **One stop for full online support**
- **Find answers to your questions**
- **Search across ALL content**
- **Subscribe to email notification**
- **Single sign-on to ALL content**
- **Access to other support resources**
  - Case Management Portal
  - Discussion Forums
  - Training Information

http://simcompanion.mscsoftware.com

Copyright© 2012 MSC.Software Corporation

SIMCOMPANION

- **Personalized Support via the following channels**
  - Web
    - Submit a Case Online
    - Manage My Cases
  - Email
    - List of Addresses in Support Contact Information
  - Phone
    - List of Phone Numbers in Support Contact Information



Web:

http://support.mscsoftware.com/servicerequest

Email:

English	
- MD Nastran or MSC Nastran	mscnastran.support@mscsoftware.com
- MD Adams or Adams	mscadams.support@mscsoftware.com
- Patran	mscpatran.support@mscsoftware.com
- Dytran	mscdytran.support@mscsoftware.com
- Easy5	easy5.support@mscsoftware.com
- Fatigue	mscfatigue.support@mscsoftware.com
- Marc	mscmarc.support@mscsoftware.com
- Mvision	mscmvision.support@mscsoftware.com
- Sinda	mscsinda.support@mscsoftware.com
- Sofy	mscsofy.support@mscsoftware.com
- SimDesigner	simdesigner.support@mscsoftware.com
- MSC SimManager	simmanager.support@mscsoftware.com
- MSC SimXpert	simxpert.support@mscsoftware.com
Chinese (Simplified)	support.cn@mscsoftware.com
Chinese (Traditional)	support.tw@mscsoftware.com
Dutch	support.be@mscsoftware.com
French	support.fr@mscsoftware.com
German	support.de@mscsoftware.com
Italian	support.it@mscsoftware.com
Japanese	support.jp@mscsoftware.com
Korean	support.kr@mscsoftware.com
Nordic	support.no@mscsoftware.com
Portuguese	suporte.mscbrasil@mscsoftware.com
Russian	support.ru@mscsoftware.com
Spanish	support.es@mscsoftware.com

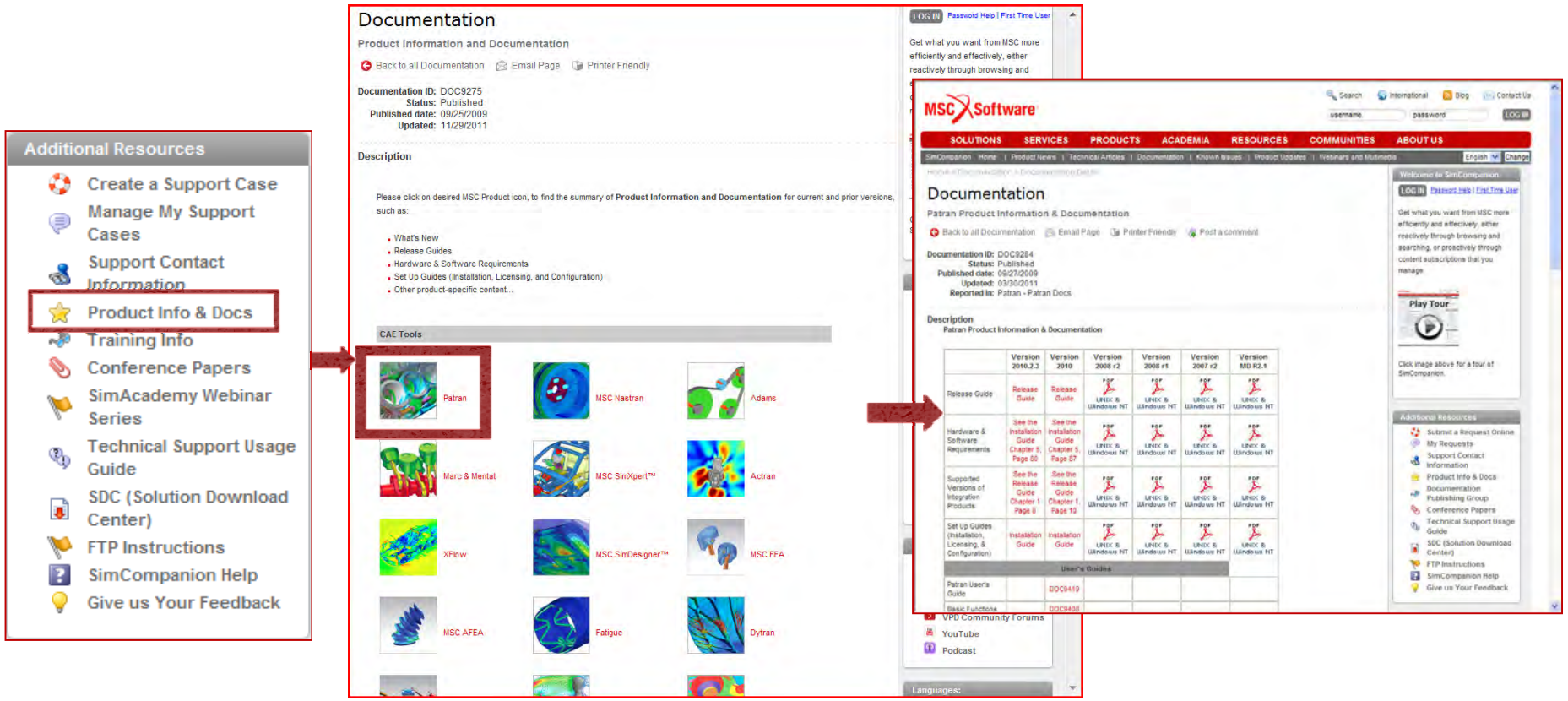
Phone:

Belgium and Luxembourg	+31 182 536444
Brazil	0800-891-4346
China - Beijing	+86-10-82607000
China - Shanghai	+86-21-63326655
China - Chengdu	+86-28-86199275/6
China - Shenzhen	+86-755-23811895
Czech Republic	+420 54517 6106
DACH	+49 89 431 987 277
Denmark	+45 61 22 32 00
Finland	0800-9-14709
France	+33 5 34 60 44 80

Copyright© 2012 MSC.Software Corporation

SIMCOMPANION

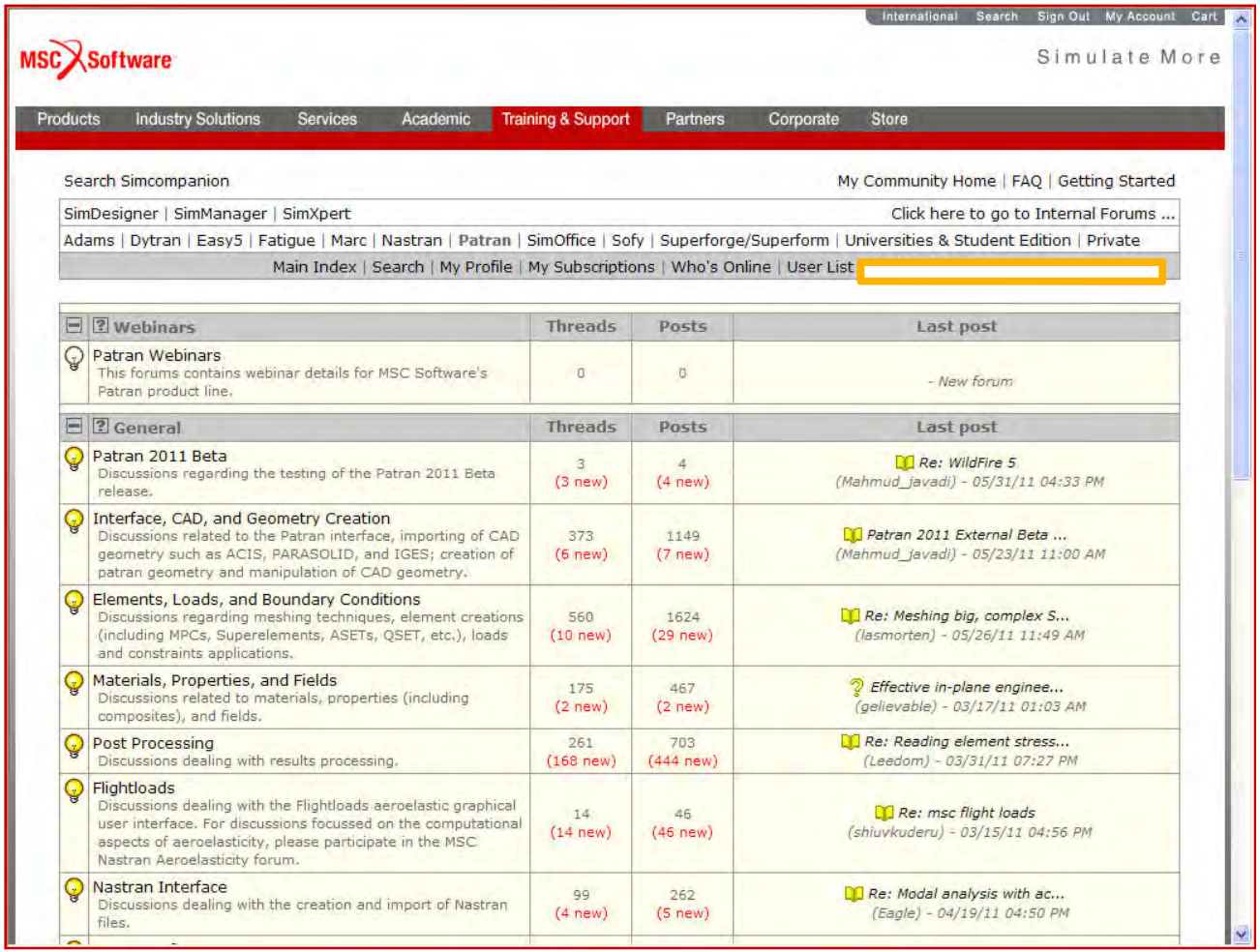
- **Product Info and Docs**
  - Access to all Product Documentation



Copyright© 2012 MSC.Software Corporation

SIMCOMPANION

- **Access to Communities**
  - VPD Community Discussion Forums
  - Subscribe to discussion communities of interest



Copyright© 2012 MSC.Software Corporation

SECTION 2

INTRODUCTORY COMMENTS ON MESHING

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

OVERVIEW OF STRATEGY

- **Strategies available in geometric modeling and meshing**
- **Both geometry modeling and meshing must be considered at this stage as they are intimately linked**
- **Assumption is that creating mesh(es) is the objective**
- **Factors to consider**
  - Analysis objective
  - CAD data that is available
  - Type of structure being analyzed
  - Assembly or part
  - Timescale
  - Resource
  - Industry or company accepted practice

Copyright© 2012 MSC.Software Corporation

OVERVIEW OF STRATEGY

- **Only by considering all these factors can the appropriate Patran strategy be evolved**
  - What type of idealization should be made?
  - What is the best set of Patran functionality to use?

Copyright© 2012 MSC.Software Corporation

TYPES OF IDEALIZATION

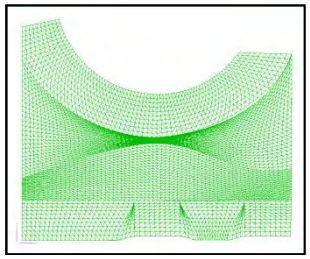
- **Breakdown of Idealization type:**
  - 2D beginning to end
  - 3D start point to 2D idealization
  - 3D start point to 2 ½ D idealization
  - 3D and 2D mix
  - 3D beginning to end

Copyright© 2012 MSC.Software Corporation

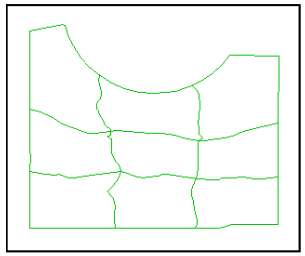
MANY POSSIBLE PATHS THROUGH PATRAN

- **Below are some example paths through Patran:**

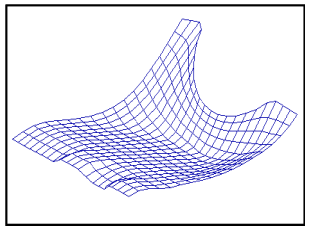
Tri, Surface, Quad



Non-congruent tri mesh

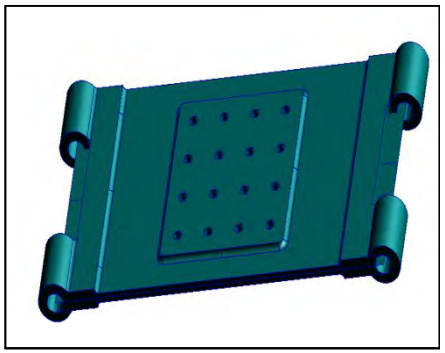


Surface from tri mesh

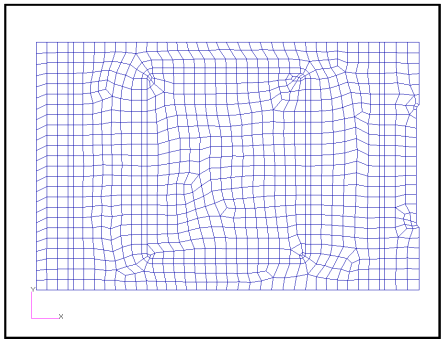


Quad mesh from surface

3D Object, 2D Model

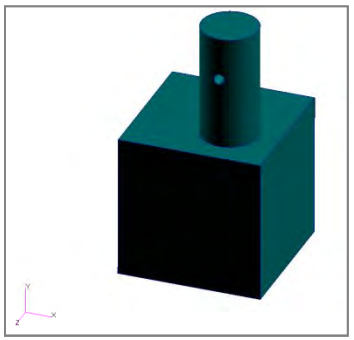


3D object

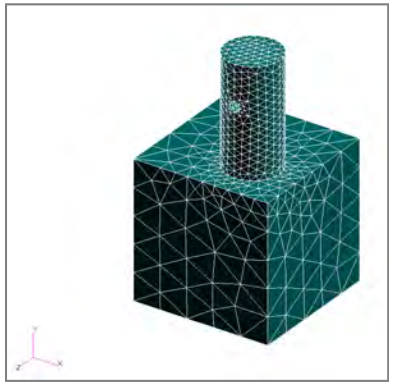


2D plate elements

3D Assembly, Congruent tet Mesh



3D Parasolid solids



Congruent assembly mesh

Copyright© 2012 MSC.Software Corporation

PATHS THROUGH PATRAN

- **Because of the many possible paths that can be taken in creating a model using Patran, it is important to have a simple way of visualizing the possible approaches.**
- **A method for easily keeping track of the possible modeling approaches is described in the following slides.**

Copyright© 2012 MSC.Software Corporation

DESCRIPTION OF PATHS THROUGH PATRAN

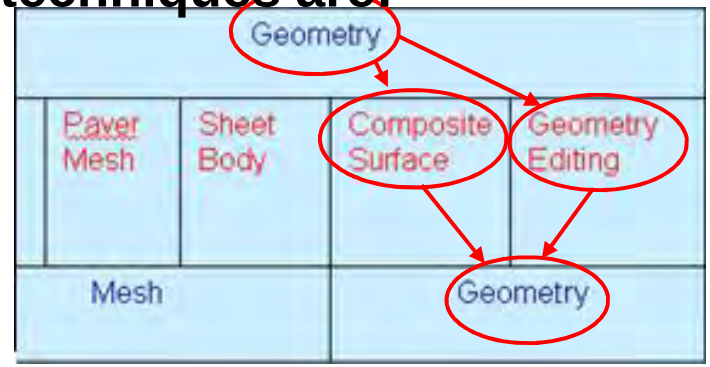
- The approach is to select the idealization starting point and desired end point
- The available Patran tools then fall into a logical set
- For example, the following is used for 2D beginning-to-end model creation:

Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated Surface	Iso Mesh	Paver Mesh	Sheet Body	Composite Surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	

Copyright© 2012 MSC.Software Corporation

EXAMPLE: GEOMETRY CLEANUP

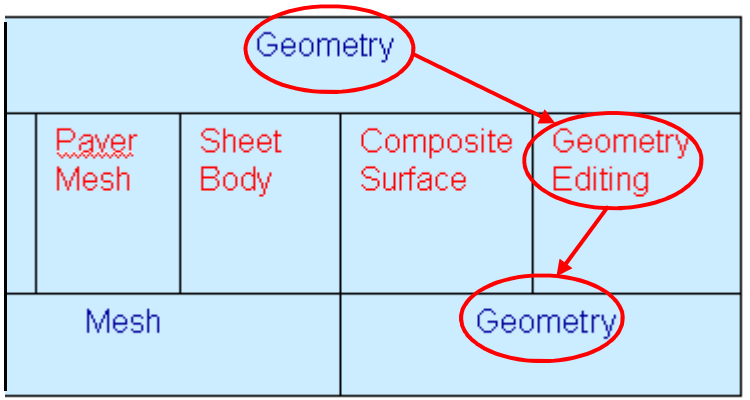
- **A typical scenario is where geometry has been imported and there are cracks (non-congruent geometry). This results in unacceptable meshes (unconnected meshes).**
- **Also, it may be desirable to de-feature the geometry to remove tooling holes and other unwanted details**
- **Can either:**
  - Clean up the geometry prior to building further geometry or meshing
  - Deal with the mesh after meshing the poor geometry
- **Look at geometry clean up; two main techniques are:**
  - Geometry Editing
  - Creating a Composite Surface
  - A combination of both may be possible



Copyright© 2012 MSC.Software Corporation

EXAMPLE: GEOMETRY CLEANUP

- **Geometry Editing methods include:**
  - Edge Match to remove cracks, forcing congruency
  - Sew – automates edge matching
  - Breaking surfaces to achieve congruency
  - Extending surfaces to fill gaps
  - Manually constructing intermediate surfaces
  - Associating surface vertex points to adjacent surface edges
  - Using advanced surface features to remove holes, etc.



Copyright© 2012 MSC.Software Corporation

PATH CAN BE SEQUENCE OF (SUB) PATHS

- **The actual modeling path can be a sequence of paths, each with its own start and end points.**
- **The following is an example that involves constructing intermediate geometry, and then Paver meshing**

Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated surface	Iso Mesh	Paver Mesh	Sheet Body	Composite surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	

Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated surface	Iso Mesh	Paver Mesh	Sheet Body	Composite surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	

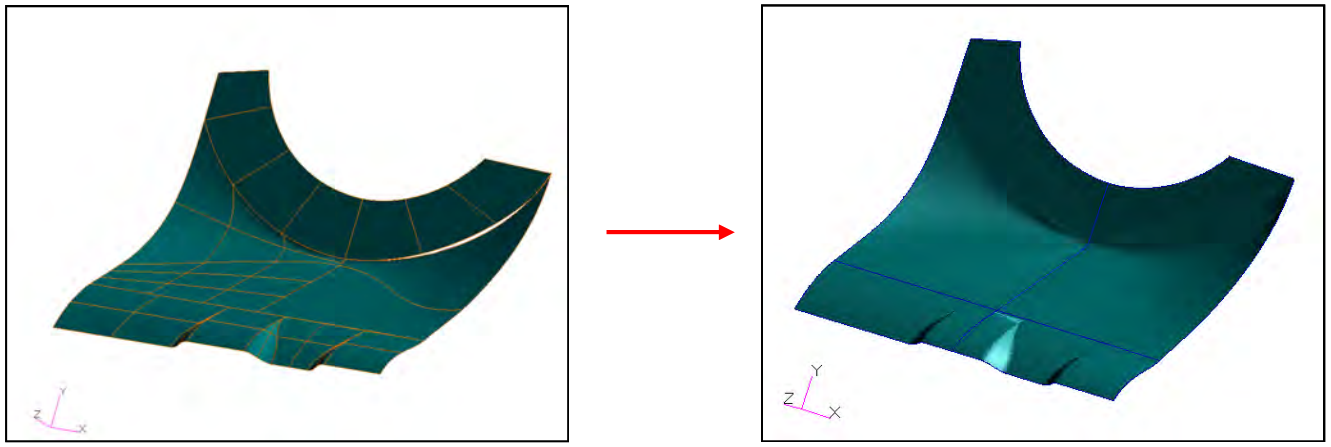
Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

SECTION 3

STRATEGIES IN GEOMETRIC MODELING AND MESHING

2D SURFACE MESHING



Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

IDEALIZATION TYPE

- **Breakdown of Idealization type:**
  - **2D beginning to end**
  - 3D start point to 2D idealization
  - 3D start point to 2 ½ D idealization
  - 3D and 2D mix
  - 3D beginning to end

Copyright© 2012 MSC.Software Corporation

2D BEGINNING TO END

- **Category: Only surfaces (2D) and 2D meshing beginning to end**

- **Structures falling in this category include:**
  - Vehicle body
  - Aircraft wing and fuselages
  - Ship hulls
  - Buildings
  - (Add 1D bar/beam idealizations for the above, as needed)

Copyright© 2012 MSC.Software Corporation

2D BEGINNING TO END

- **The approach is to select the 2D idealization starting point and desired 2D end point.**

- **The available Patran tools can then be used to create 2D geometry or 2D mesh.**

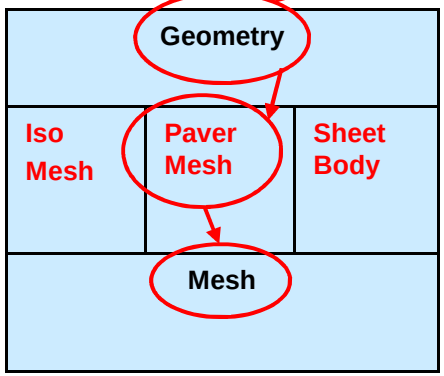
Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated Surface	Iso Mesh	Paver Mesh	Sheet Body	Composite Surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	

Copyright© 2012 MSC.Software Corporation

2D BEGINNING TO END

- **Many times, the actual modeling path will be a sequence of start and end points.**
- **This example involves constructing intermediate geometry (surface) and then Paver meshing it.**

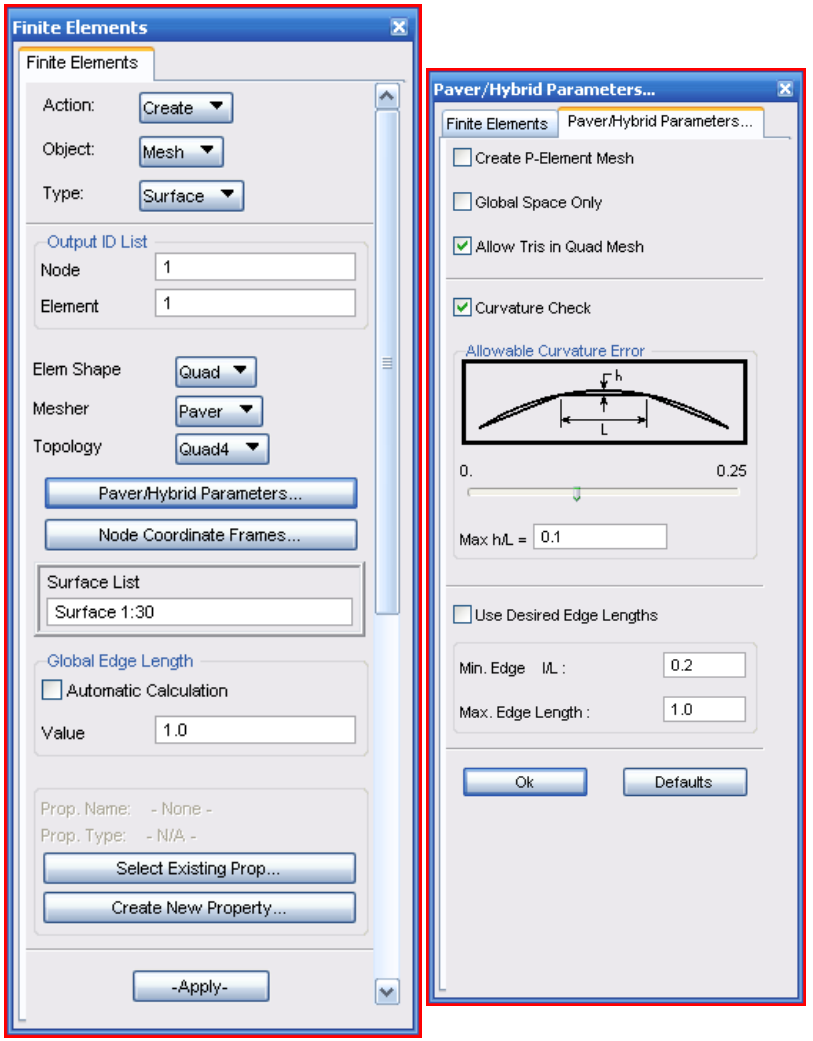
Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated Surface	Iso Mesh	Paver Mesh	Sheet Body	Composite Surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	



Copyright© 2012 MSC.Software Corporation

PAVER (2D) MESHING

- **There are several Paver parameters that can be used to influence the quality of the mesh created.**
  - Global Edge Length
  - Create P-Element Mesh
    - Allow element shapes to be poorer than they otherwise might be
  - Global Space Only
  - Curvature Check
    - Max h/L
  - Use Desired Edge Lengths
    - Min. Edge l/L
    - Max. Edge Length



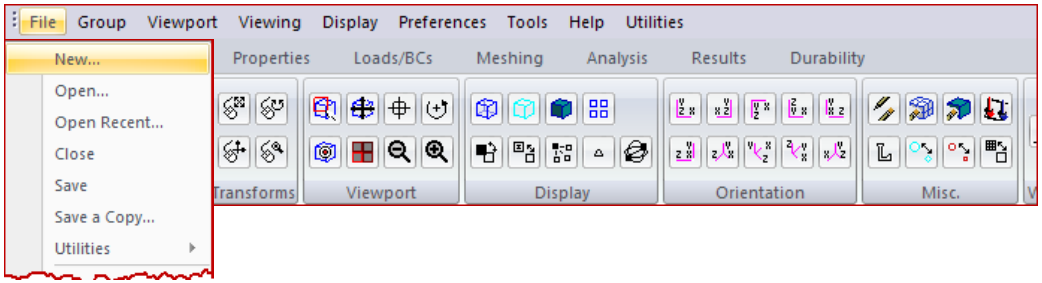
Copyright© 2012 MSC.Software Corporation

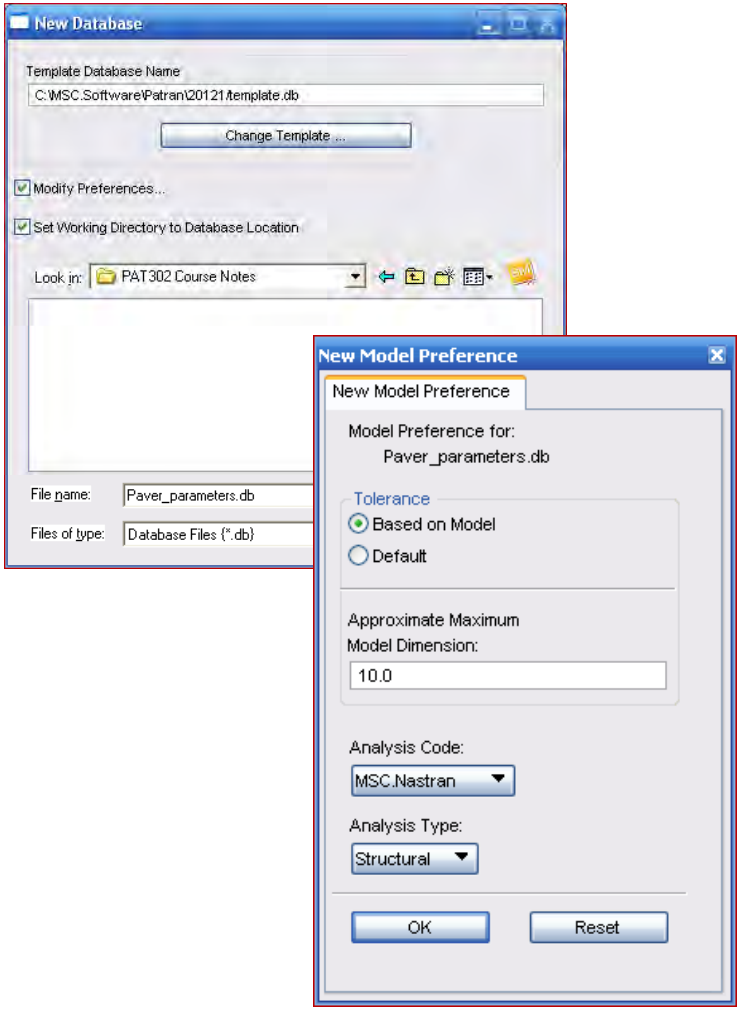
CASE STUDY: PAVER PARAMETERS

- **This case study deals with using the Curvature Check parameter Max h/L**
- **For a given (fixed) value of GEL, the value of MAX h/L is varied, and a Paver Mesh is obtained for each value**
- **The changes of the mesh are looked at and studied for changes in the value of Max h/L**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: PAVER PARAMETERS

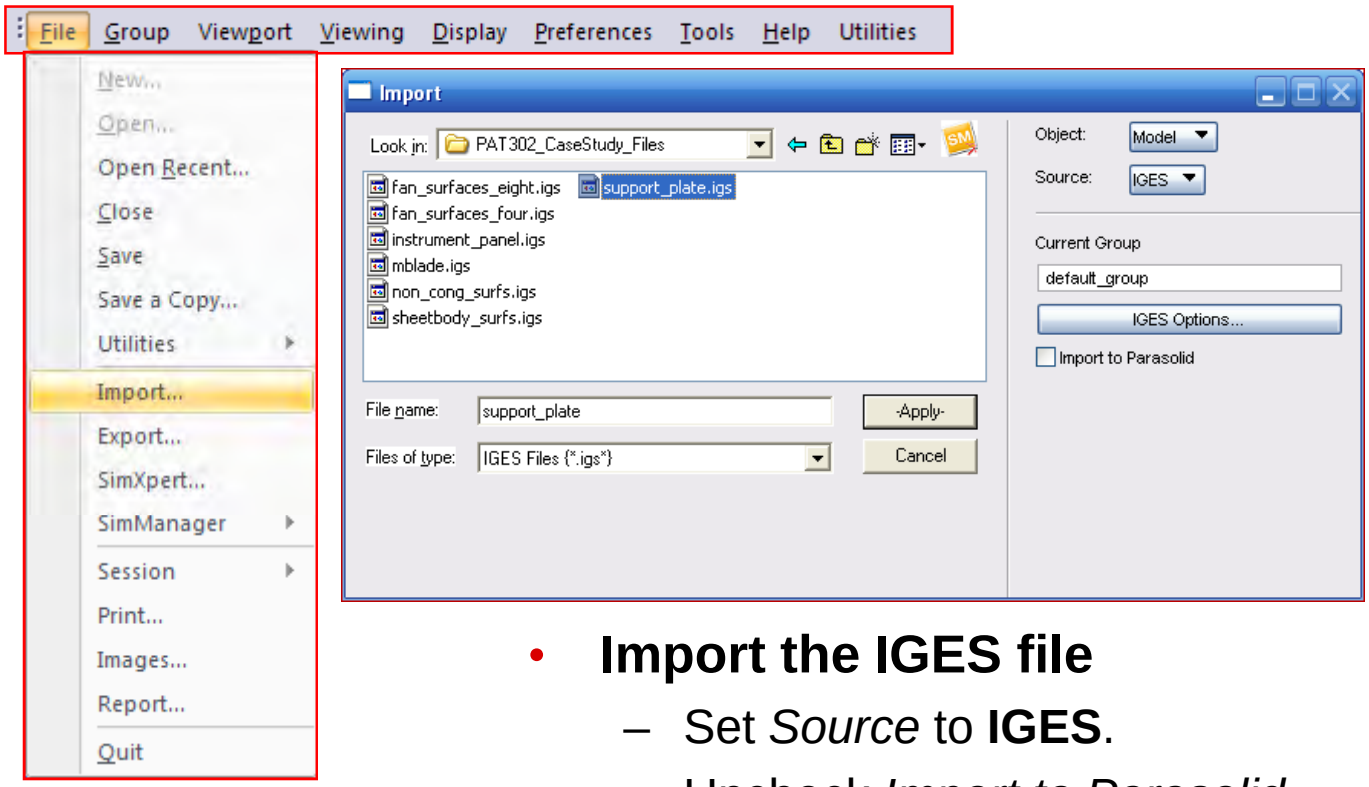




- **Create a new database**
  - Enter **Paver_parameters.db** for the file name.
  - Set the *New Model Tolerances* to **Based on Model**.
  - Set the *Analysis Code* and *Analysis Type* to **MSC.Nastran** and **Structural**, respectively.

Copyright© 2012 MSC.Software Corporation

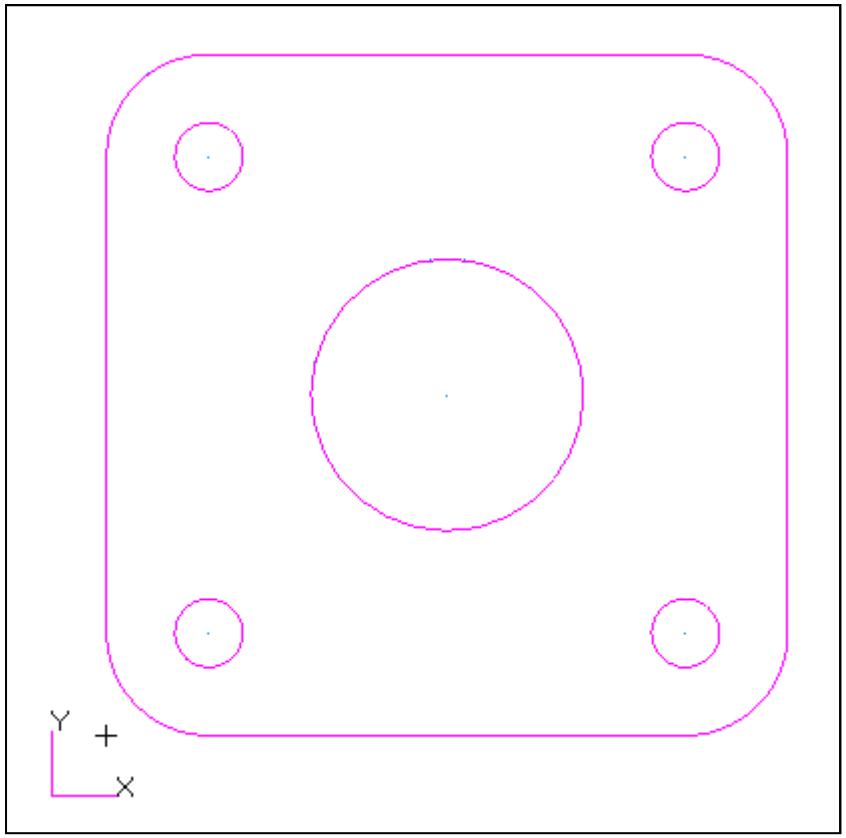
CASE STUDY: PAVER PARAMETERS

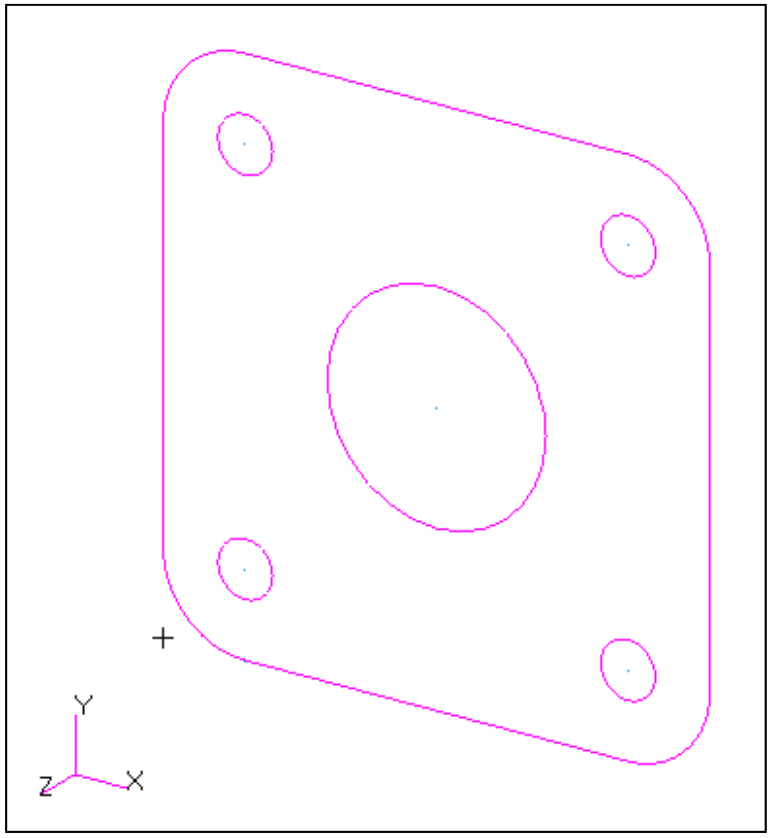


- **Import the IGES file**
  - Set *Source* to **IGES**.
  - Uncheck *Import to Parasolid*.
  - Select **support_plate.igs** and click **Apply**.
  - Click **OK** when the import summary appears.
  - Click on the **Front view** icon.
  - Then click on the **Iso 1 View** icon.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: PAVER PARAMETERS



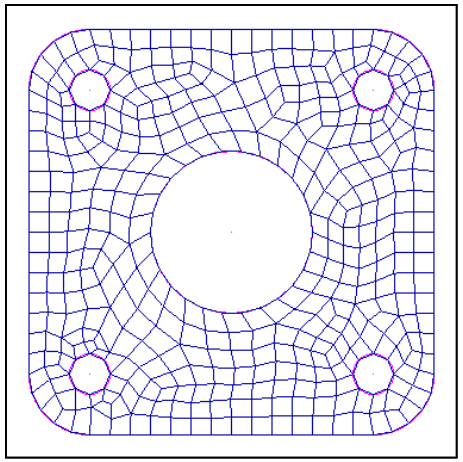


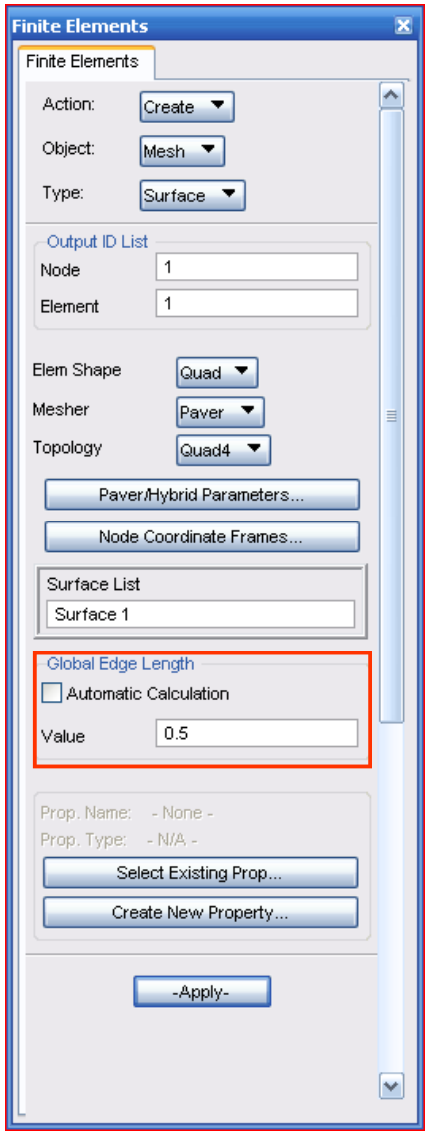
"Front" and "Iso 1" views of the model.

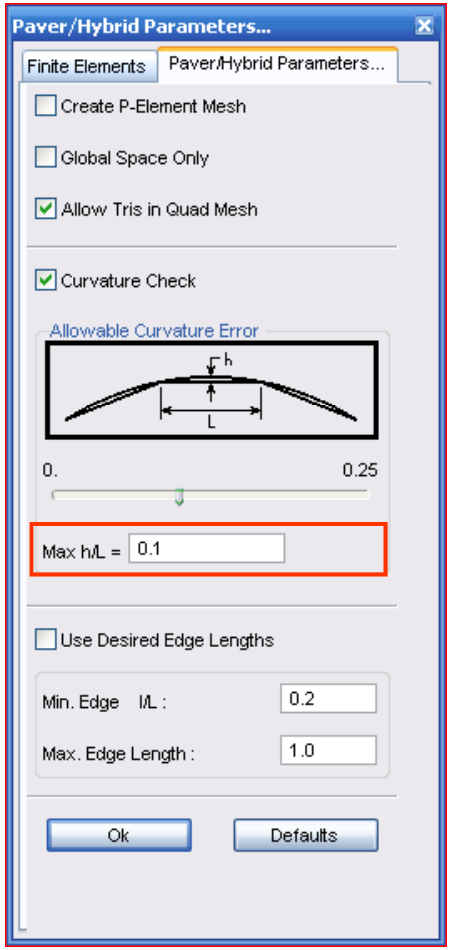
Copyright© 2012 MSC.Software Corporation

CASE STUDY: PAVER PARAMETERS

- **Problem Description**
  - See how changing the value of the Paver parameters affects the mesh obtained
  - The first mesh obtained is based on the following parameter values
    - GEL = 0.5
    - Max h/L = 0.1







Copyright© 2012 MSC.Software Corporation

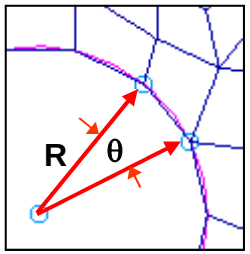
CASE STUDY: PAVER PARAMETERS

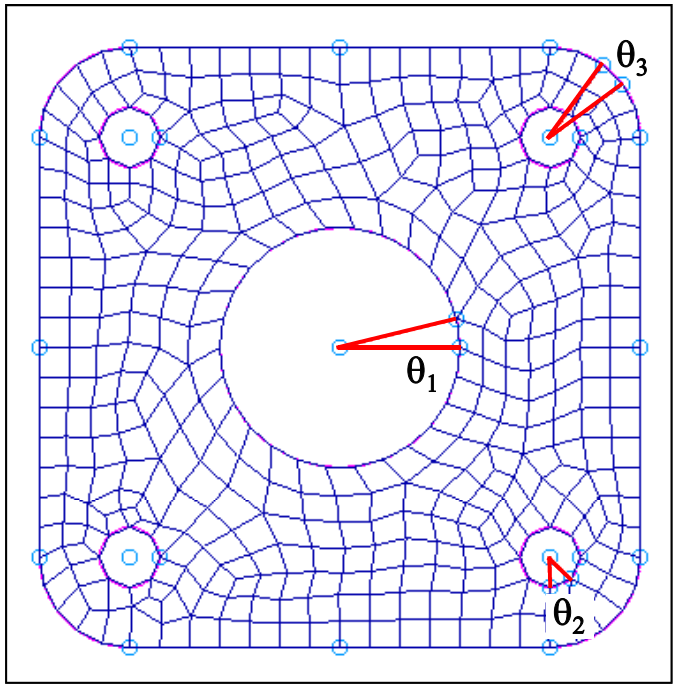
- **Determine the value of h/L for an element at each different radius on the surface edges.**

The simple expression for h/L is

$h/L = (1-\cos(\theta/2))/(2\sin(\theta/2))$

where h and L are defined in Patran, and θ is the angle between the two radii at the ends of an element edge, on a curved surface edge; the radii are from the center of curvature of the curved surface edge.





- **Now, determine the three angles identified in the figure, and the corresponding h/L.**

θ_i	h/L
θ_1 (13.84 deg)	0.030
θ_2 (45.01 deg)	0.099
θ_3 (18.00 deg)	0.039

Copyright© 2012 MSC.Software Corporation

CASE STUDY: PAVER PARAMETERS

- **Compare the value of h/L in the table to the value of Max h/L = 0.1 for the Paver mesher:**

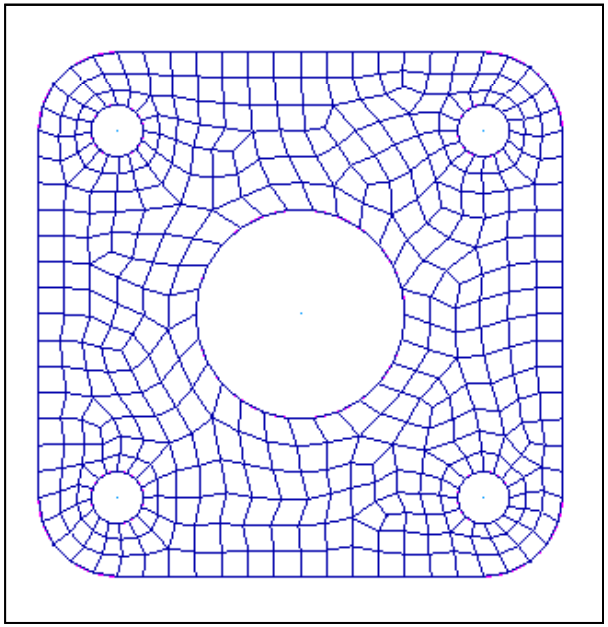
θ_i	h/L	Elem Edge Length
θ_1 (13.84 deg)	0.030 (< 0.1)	0.482
θ_2 (45.01 deg)	0.099 (~=0.100)	0.383
θ_3 (18.00 deg)	0.039 (< 0.1)	0.469

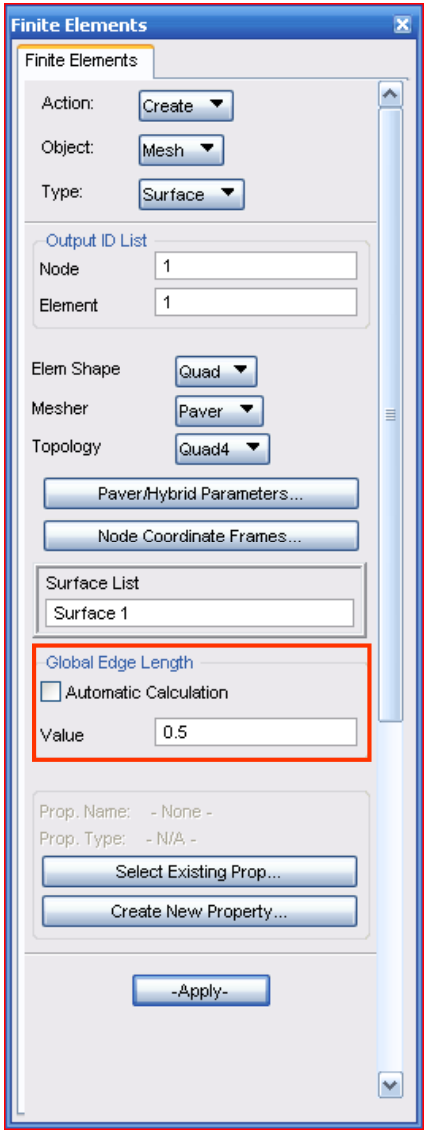
- **Because h/L is always < 0.1, for the most part the lengths of the element edges are controlled by the value of the GEL; GEL = 0.5. There is some variation of the element edge lengths from 0.5, so that well shaped elements are created.**

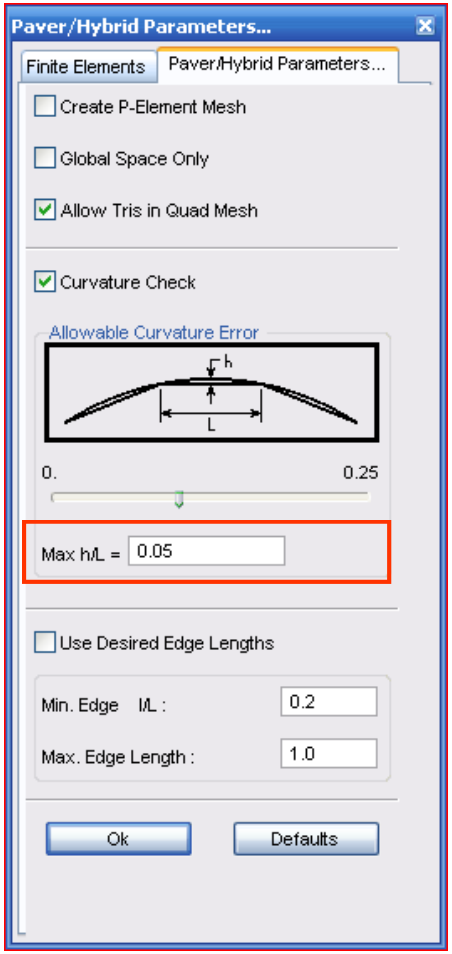
Copyright© 2012 MSC.Software Corporation

CASE STUDY: PAVER PARAMETERS

- **Now, reduce the value of Max h/L and remesh**
  - GEL = 0.5
  - Max h/L = 0.05



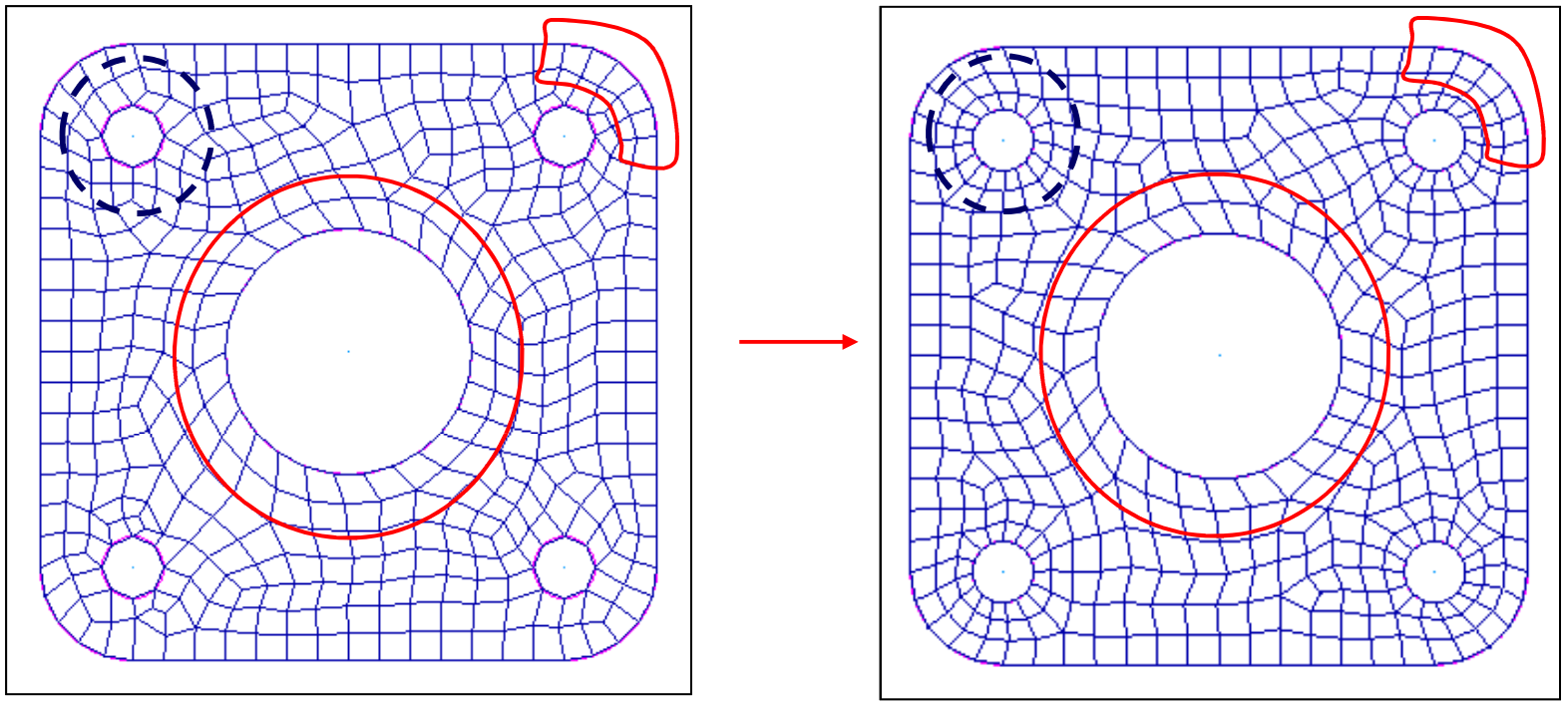




Copyright© 2012 MSC.Software Corporation

CASE STUDY: PAVER PARAMETERS

- **Compare the mesh for the two values of Max h/L**



Max h/L = 0.10

Max h/L = 0.05

- **Little change [inside solid boundary] where h/L < 0.05. Considerable change [inside dashed boundary] where h/L ~= 0.05**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: PAVER PARAMETERS

- **Further reduce the value of Max h/L and re-Paver mesh**
  - GEL = 0.5 (constant value for all meshes)
  - Max h/L = 0.031, 0.02

Max h/L = 0.100			Max h/L = 0.050			Max h/L = 0.031			Max h/L = 0.020		
θ_i	h/L	EEL	θ_i	h/L	EEL	θ_i	h/L	EEL	θ_i	h/L	EEL
θ_1 13.84 deg	0.030 (< 0.100)	0.48	θ_1 13.84 deg	0.030 (< 0.050)	0.48	θ_1 13.84 deg	0.030 (~= 0.031)	0.48	θ_1 9.16 deg	0.020 (= 0.020)	0.32
θ_2 45.01 deg	0.099 (~= 0.100)	0.38	θ_2 22.33 deg	0.049 (~= 0.050)	0.20	θ_2 13.28 deg	0.031 (~= 0.031)	0.12	θ_2 9.26 deg	0.021 (~= 0.020)	0.08
θ_3 18.00 deg	0.039 (< 0.100)	0.47	θ_3 17.99 deg	0.039 (< 0.050)	0.47	θ_3 12.74 deg	0.029 (~= 0.031)	0.34	θ_3 9.09 deg	0.020 (= 0.020)	0.24

Note: EEL refers to **E**lement **E**dge **L**ength.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: PAVER PARAMETERS

- **From the previous material, it is seen that**
  - For a given GEL
    - As the value of Max h/L is reduced, the element size at curved surface edges is progressively governed by it, starting with the surface edges with the smallest radius of curvature, and proceeding to the edges with greater radius of curvature.
    - The pattern of elements at curved surface edges does not change very much, for changes of the value of Max h/L, as long as h/L for the edges **<** Max h/L.
    - As Max h/L **->** h/L for an edge(s) the pattern of elements at that edge(s) changes, the pattern of elements near the edge(s) changes.
- **So, when Paver meshing,**
  - Specify the desired value of the GEL to obtain elements, with a size of about the GEL, away from curve surface edges.
  - Keep reducing Max h/L until the desired mesh density, element size, and element shape at curved surface edges are obtained.

Copyright© 2012 MSC.Software Corporation

MESHING SURFACES USING MESH CONTROL

- **There are several types of mesh control**
  - Mesh seed
  - Hard (associated) points and curves
    - One type of hard point is “Auto Hard Points”
  - Mesh control for surfaces (the GEL can be different for each surface)
- **The Sheet Body mesher has similar controls, i.e. feature curves and vertices.**

Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated Surface	Iso Mesh	Paver Mesh	Sheet Body	Composite Surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	

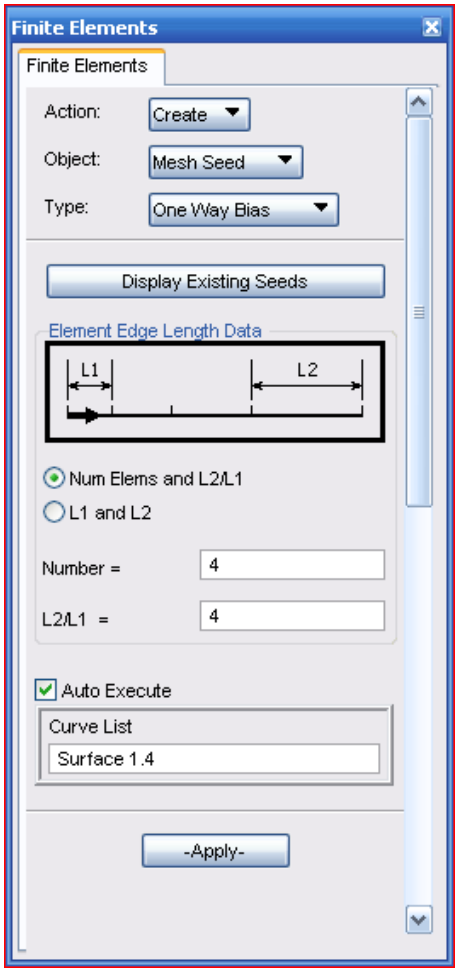
Copyright© 2012 MSC.Software Corporation

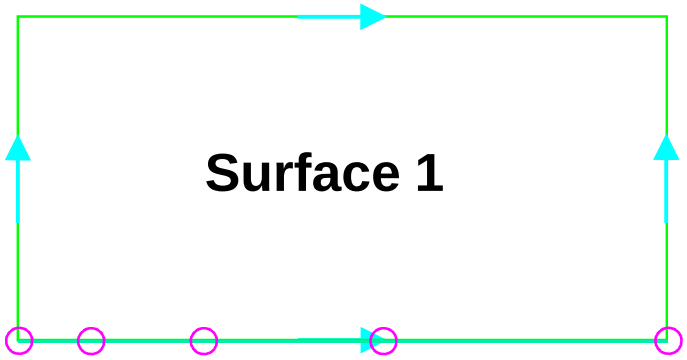
MESH SEEDING

- **Mesh seed on curves or edges is used to control the number and size of elements generated for the model.**
- **Also, it is used for transitioning a mesh between different densities.**
- **Patran has different methods to generate the seeding**
  - Uniform seed bias (equally spaced nodes)
  - Non-uniform seed bias (variable spacing)
  - Curve based seeding (automatic in highly curved regions)
  - Tabular, including using existing nodes
  - PCL function

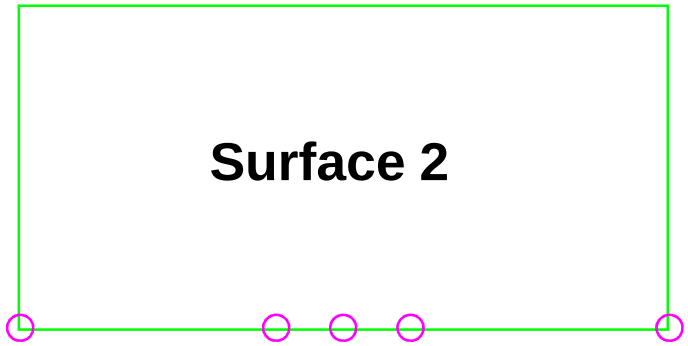
Copyright© 2012 MSC.Software Corporation

NON-UNIFORM MESH SEED BIAS

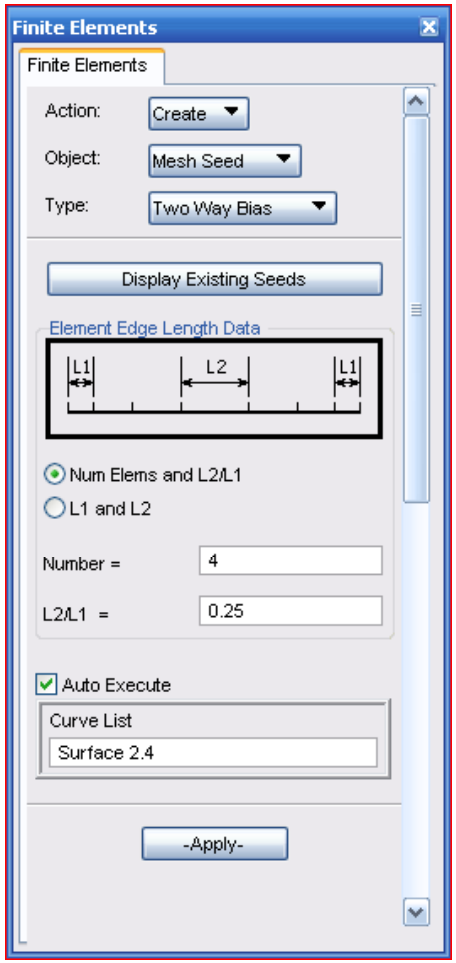




Mesh Ratio = 4



Mesh Ratio = 0.25 (or -4)

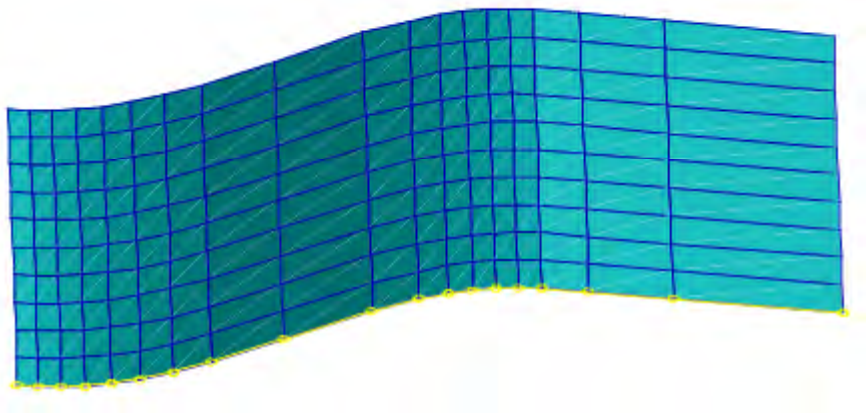


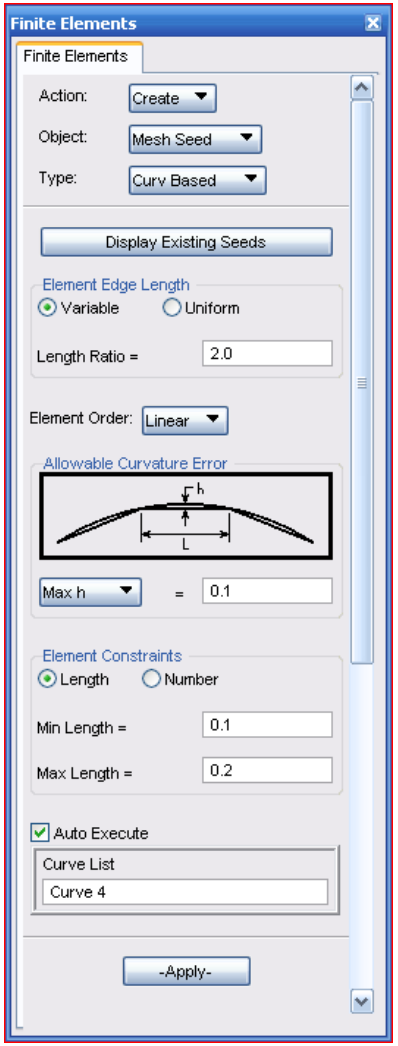
(*) Cyan arrows → indicate positive edge direction

Copyright© 2012 MSC.Software Corporation

CURVE BASED MESH SEEDING

- **Variable or Uniform distance along a curve**
  - Length Ratio dictates the ratio of the length of adjacent elements
- **Order of elements to be created**
  - Linear causes more mesh seeds to be created



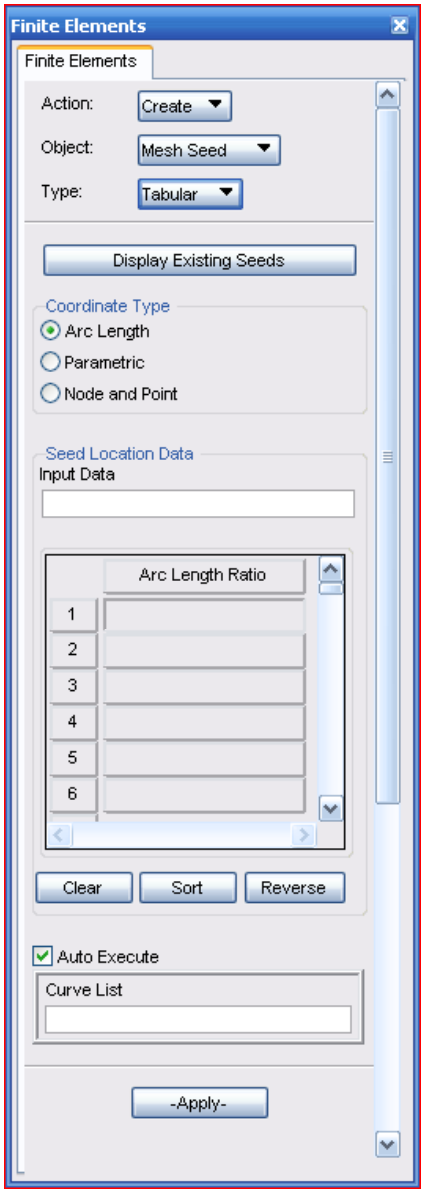


- **Refine mesh based on chordal tolerance**
  - Max h, or Max h/L
- **Specify the minimum and maximum element lengths, or minimum and maximum number of elements**

Copyright© 2012 MSC.Software Corporation

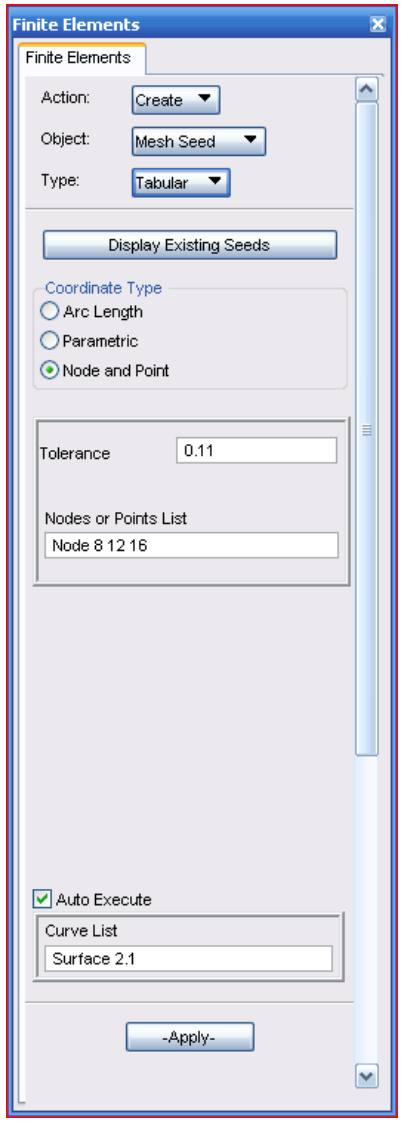
TABULAR MESH SEEDING

- **Arbitrary distribution of mesh seed along a curve/edge**
- **Location can be defined in real or parametric space**
- **Sort seed location in ascending order**
- **Reverse seed locations**
- **Create mesh seeds corresponding to existing nodes or points, and if desired, assign them to an edge of an adjacent surface or solid**

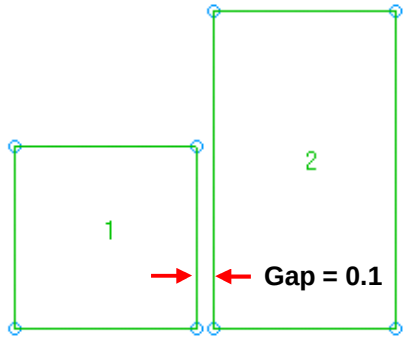


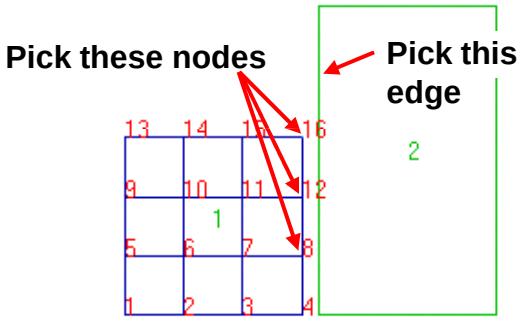
Copyright© 2012 MSC.Software Corporation

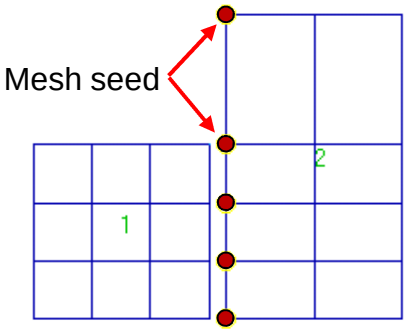
CONGRUENT MESHES VIA MESH SEED AT NODES



- **Create mesh seeds on adjacent surface edges**
- **Use nodes of existing meshes**
- **Tolerance must be > distance between adjacent surface edges, e.g. 0.11**
- **Created surface meshes will be aligned because they will follow seeds on their surface edges**







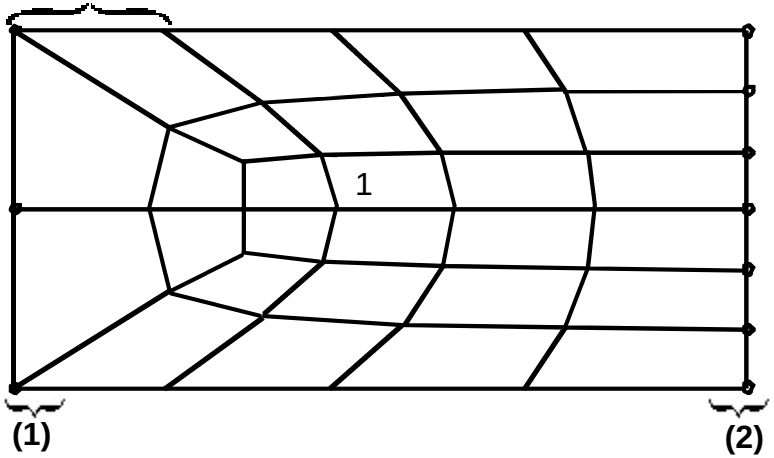
Copyright© 2012 MSC.Software Corporation

MESHING TRANSITION USING MESH SEEDS

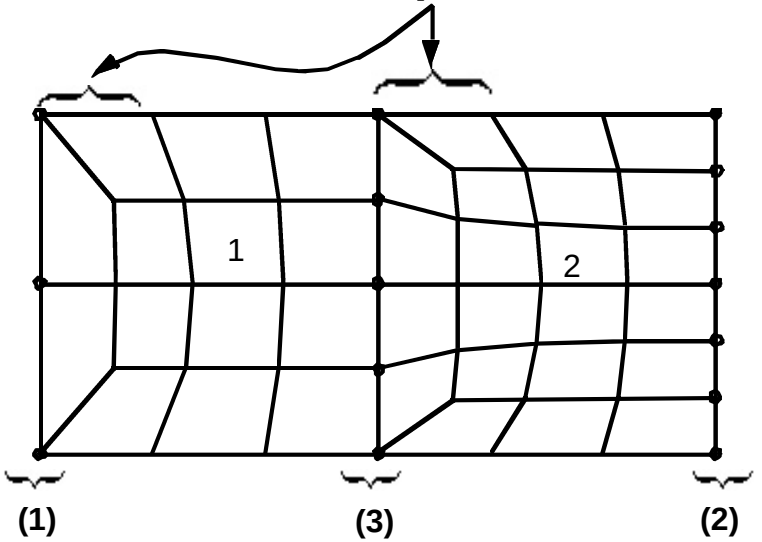
- Mesh seeds are useful in mesh transition

Transition control of IsoMesh with mesh seeds

Abrupt transition



Less abrupt transition



Two Surfaces

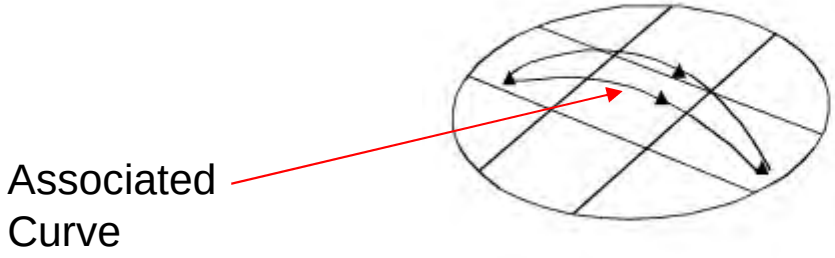
Note: (1) seeded for 2 elements

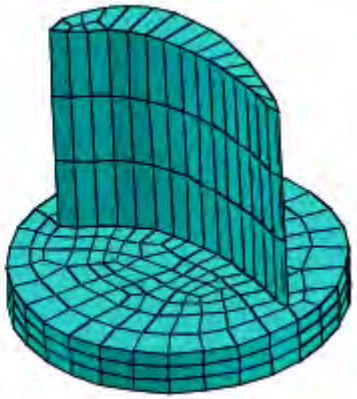
(2) seeded for 6 elements

(3) seeded for 4 elements

Copyright© 2012 MSC.Software Corporation

ASSOCIATED GEOMETRY



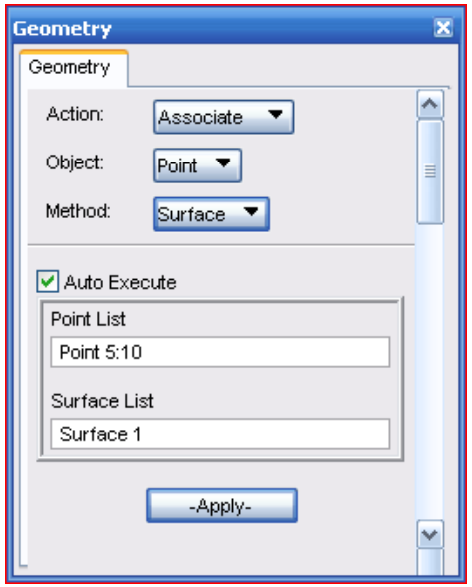


- **Also referred to as hard geometry**
- **Points and curves can be associated with other geometric entities, e.g. surface**
- **Hard Geometry can lie on the interior of geometry, e.g. inside surface**
- **The associated geometry (curve), shown above, outlines the joining (interface) mesh, providing a congruent quad mesh at the interface. The quad elements are then swept in opposite directions to provide a congruent hex mesh.**

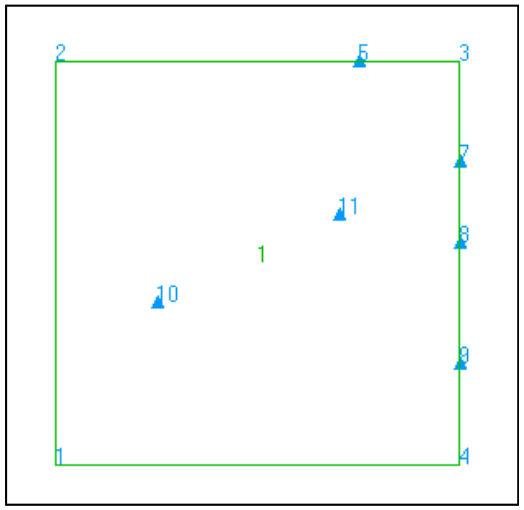
Copyright© 2012 MSC.Software Corporation

POINT ASSOCIATE / DISASSOCIATE

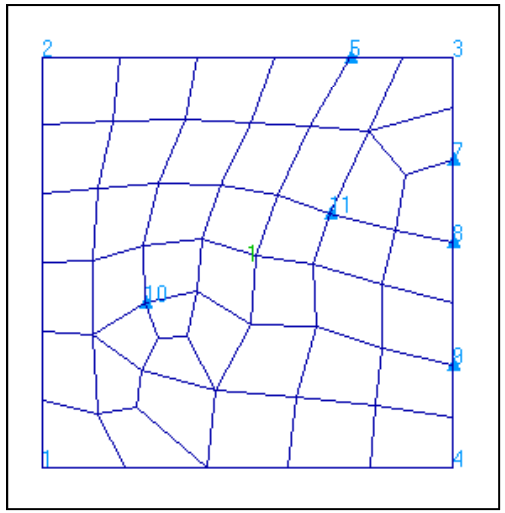
- Associated points are used to guide the meshers
- Points can be associated with curve and surface geometry
- It is only possible to associate points to curves or surfaces which are within the global model tolerance of the points
- Associated geometry is a restriction to the meshers



After Association



After Paver Meshing



- Only Paver uses associated points interior to surfaces
- Associated points can be disassociated

Copyright© 2012 MSC.Software Corporation

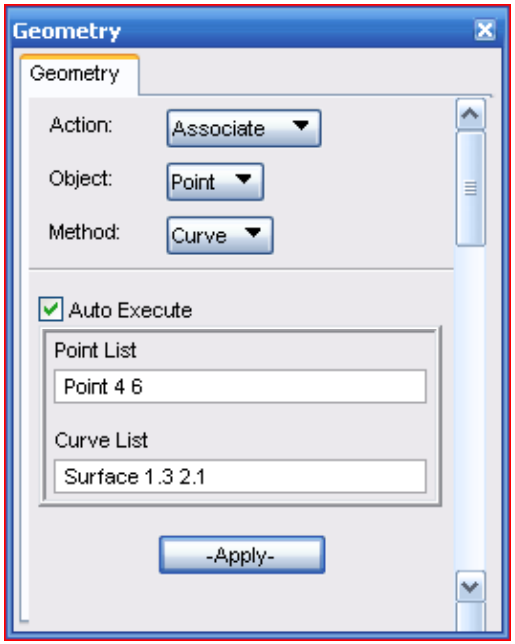
ASSOCIATED POINTS

- **Can define on edges of surface/solid or interior to surface/solid face.**
- **Geometry associated to geometric edges can be used with both, Paver and IsoMesher**
- **Load constraint sets and element properties can be assigned to hard points**

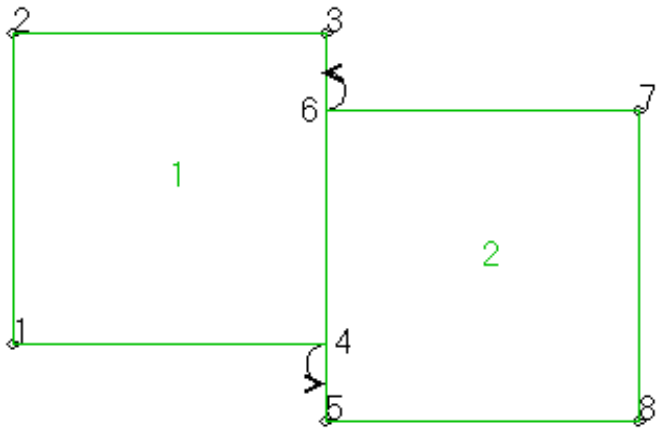
Object	Method	Description
o Point	**Curve**	Associate point to a curve.
	Surface	Associate point to a surface.
o Curve	**Curve**	Associate curve to a curve.
	Surface	Associate curve to a surface.

Copyright© 2012 MSC.Software Corporation

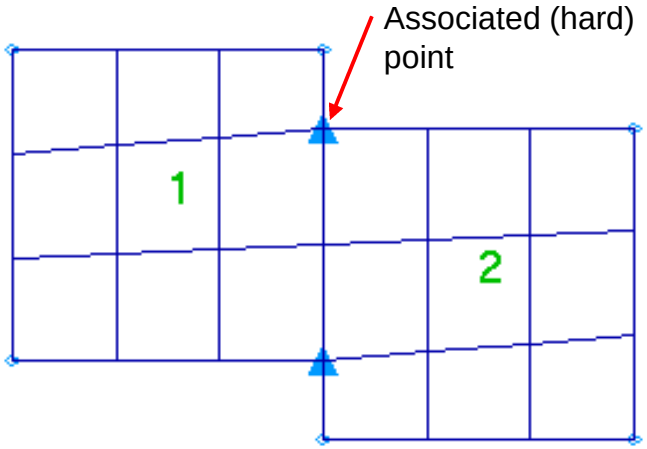
ASSOCIATING VERTICES TO EDGES



- **Associating points (vertices) to adjacent surface edges will cause created meshes to be congruent.**
- **Same mesher Global Edge Length must be used.**
- **Under the Meshing tab click on Auto Hard Points from the Mesh Control group can be used.**



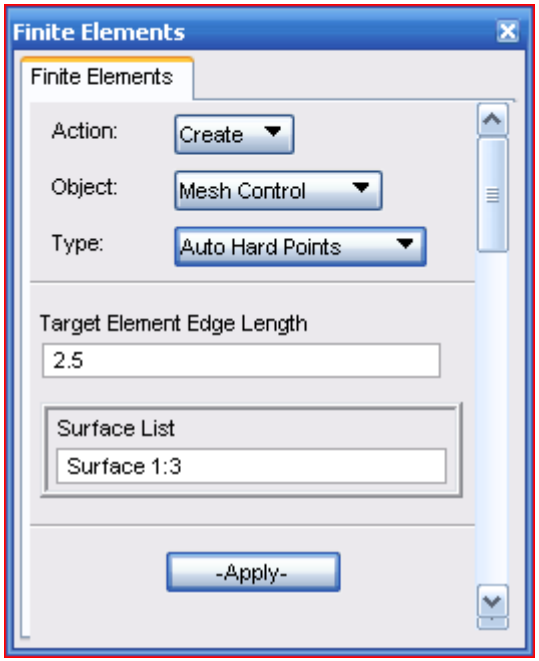
Associate vertex points



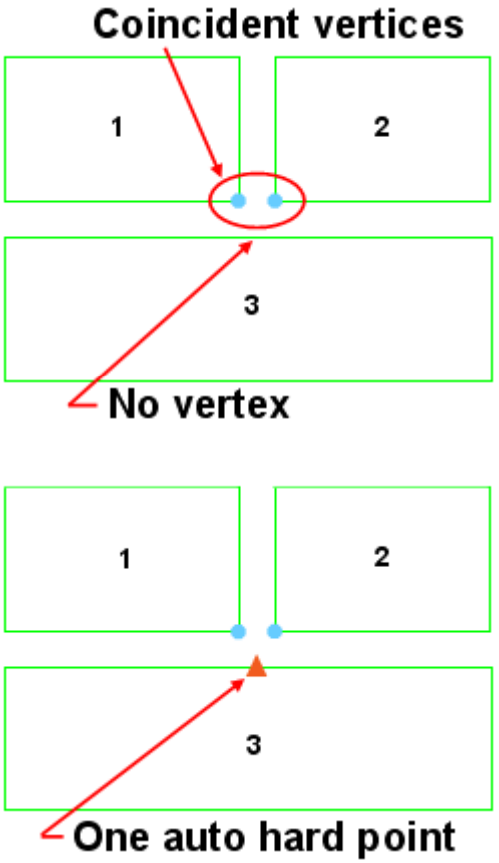
Congruent surface meshes

Copyright© 2012 MSC.Software Corporation

CONGRUENT SURFACE MESHES VIA [AUTO] HARD POINTS



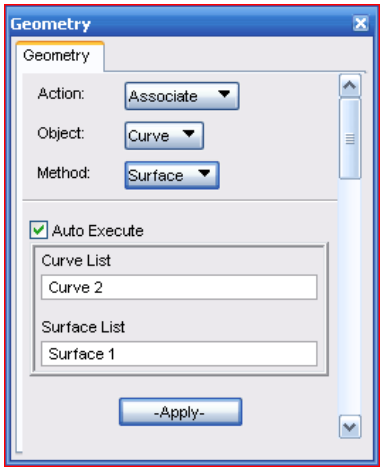
- **Create hard points automatically (quickly)**
- **Good if there is a need for many hard points – good if there are many areas of incongruency**
- **Specify the following**
  - "T" point tolerance is (Target Element Edge Length)/20
  - Surfaces for which auto hard points may be created



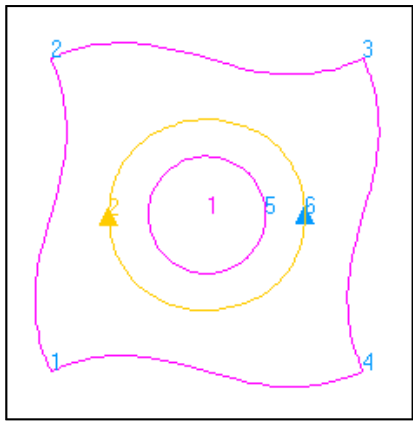
Copyright© 2012 MSC.Software Corporation

CURVE ASSOCIATE / DISASSOCIATE

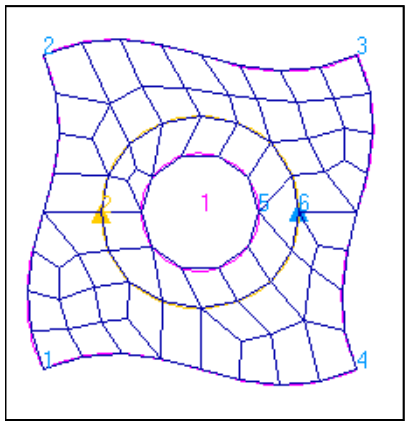
- Associated curves are used to guide the meshing of surface
- Can only associate curves which are within the global model tolerance
- Associated geometry is restriction to meshers
- The curves can be mesh seeded
- Only the Paver uses associated curves interior to surfaces
- Associated geometry can be disassociated



After Association



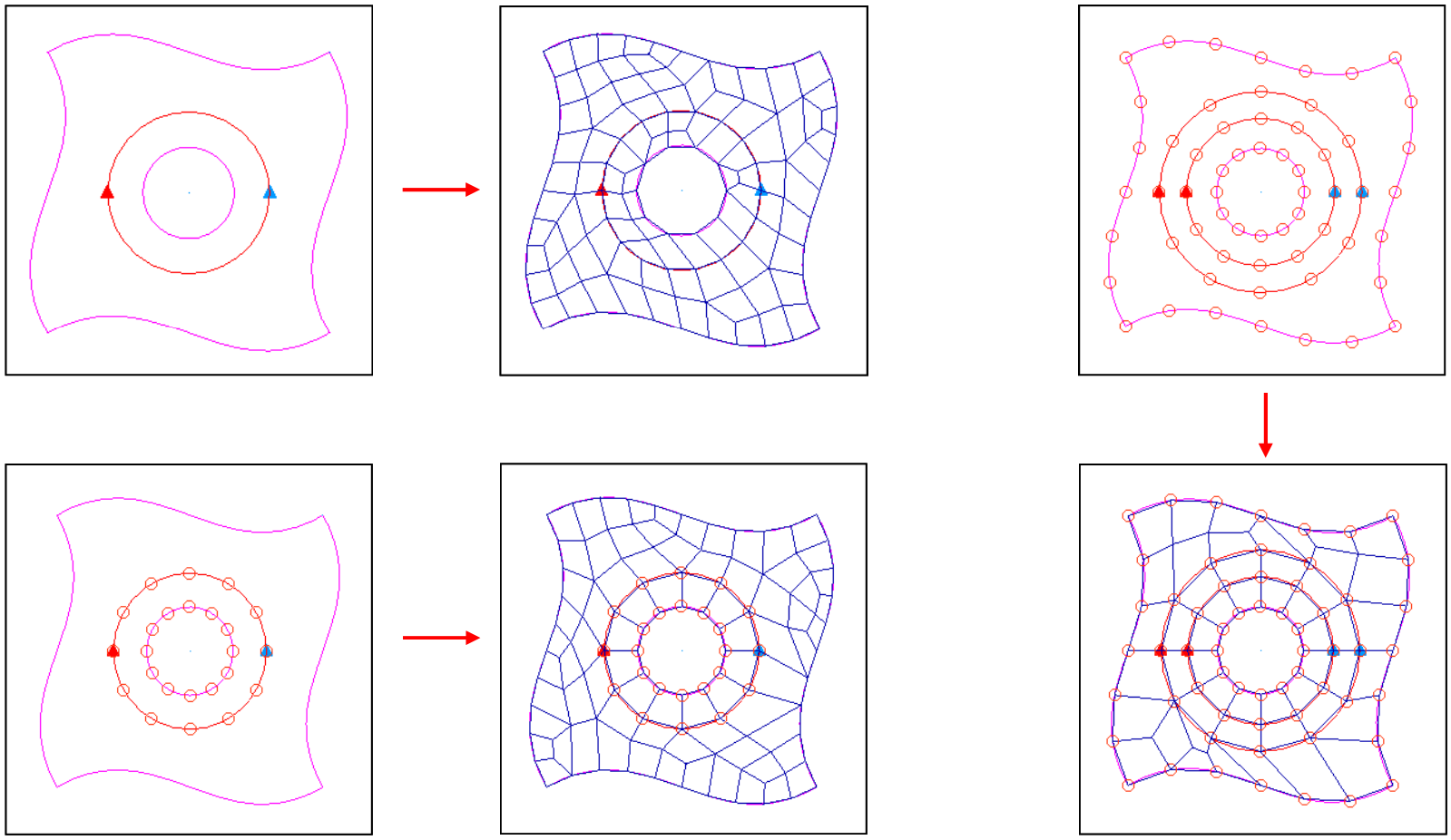
After Paver Meshing



Copyright© 2012 MSC.Software Corporation

CURVE ASSOCIATE / DISASSOCIATE

- **Paver mesh for associated curves and mesh seeds**



Copyright© 2012 MSC.Software Corporation

ASSOCIATED CURVES

- **Definable on edge of surface/solid and surface/solid face**
- **Can span multiple surfaces or faces**
- **Does not require end points to be on a physical boundary of geometric entity**
- **Can assign attributes such as load sets and mesh seeds**
- **Same capabilities for meshing as for associated points**

Object	Method	Description
o Point	**Curve**	Associate point to a curve.
	Surface	Associate point to a surface.
o Curve	**Curve**	Associate curve to a curve.
	Surface	Associate curve to a surface.

Copyright© 2012 MSC.Software Corporation

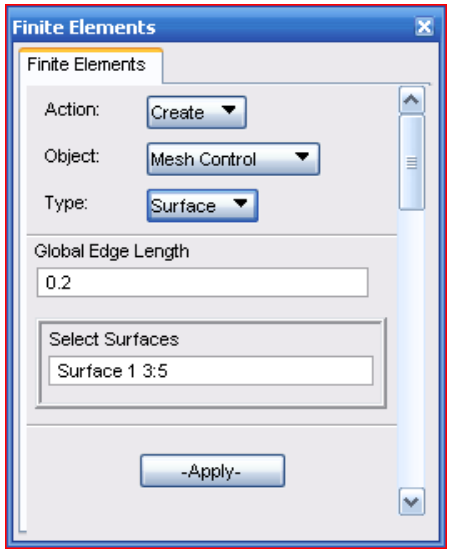
USES OF HARD GEOMETRY

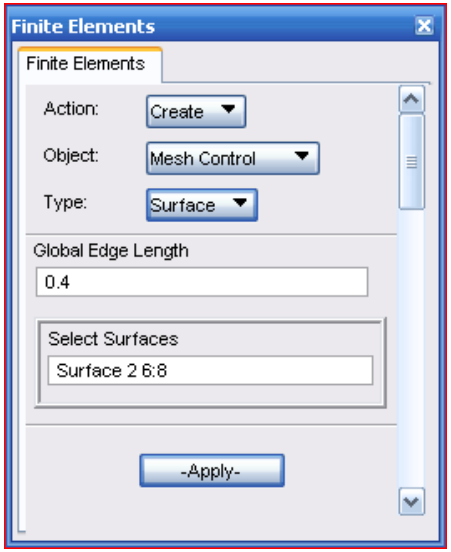
- **Guides the meshing process**
- **Useful with non-congruent geometry**
- **Modeling real features such as bolts and bolt faces**
- **Modeling stiffened panels**
- **Model weld points**

Copyright© 2012 MSC.Software Corporation

MESH CONTROL OF MULTIPLE SURFACES

- **A different meshing global edge length can be assigned to different surfaces**
- **Only used by the Paver mesher**
- **Must still specify the global edge length in the surface mesh form Under the Meshing tab click on Surface from the Mesh Control group.**

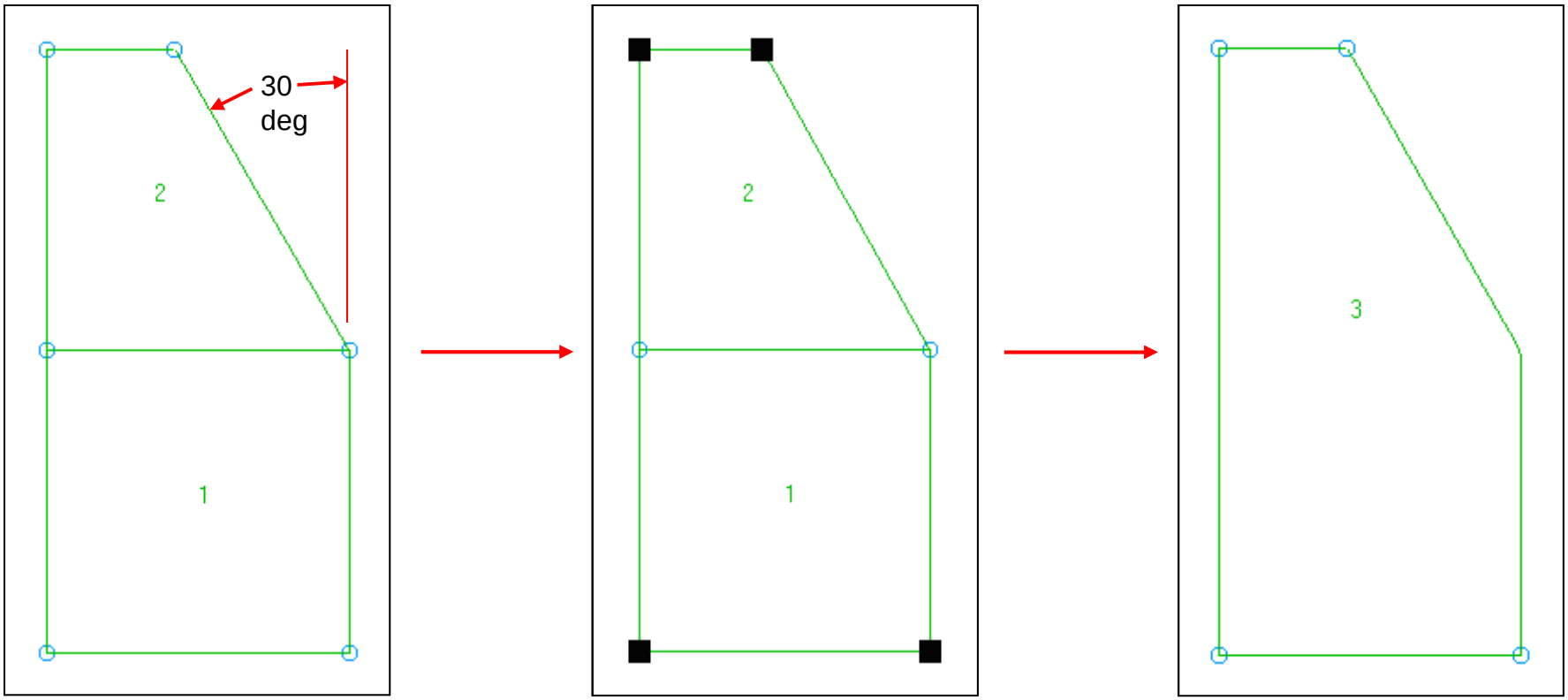




Copyright© 2012 MSC.Software Corporation

SNAP ANGLE FOR COMPOSITE SURFACE

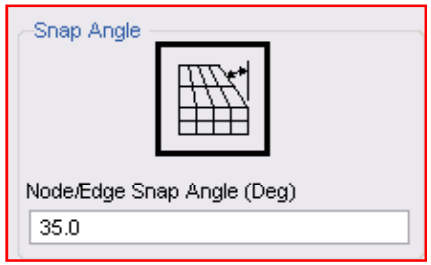
- **Specification of meshing node placement at change of angle of the edge of a composite surface (Preferences > Finite Element…)**
  - Create composite surface with four vertices > create simple surface

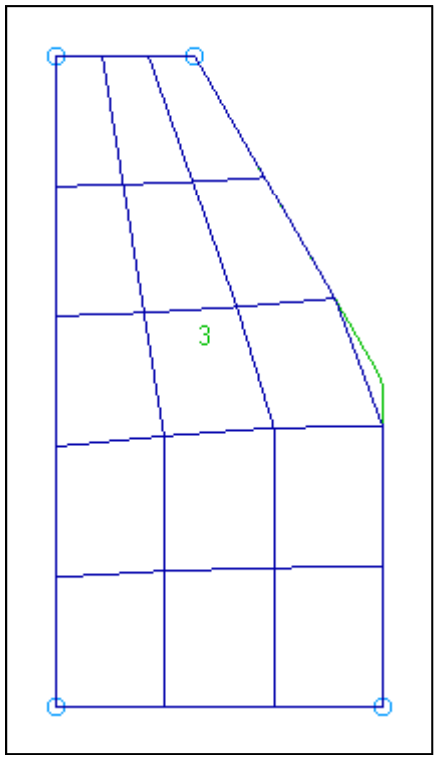


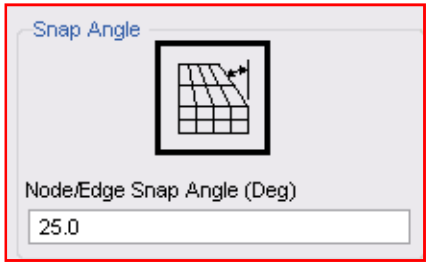
Copyright© 2012 MSC.Software Corporation

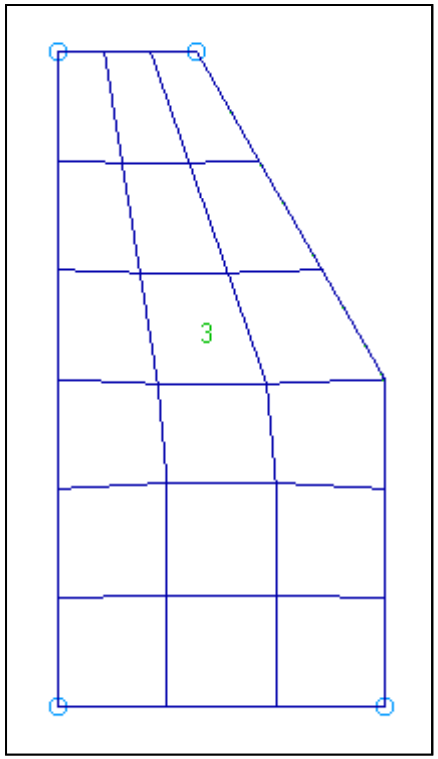
SNAP ANGLE FOR COMPOSITE SURFACE

- **Specification of meshing node placement at change of angle of the edge of the composite surface**
  - IsoMesh the composite surface with different values of Snap Angle
    - GEL = 0.4 for the creation of both meshes



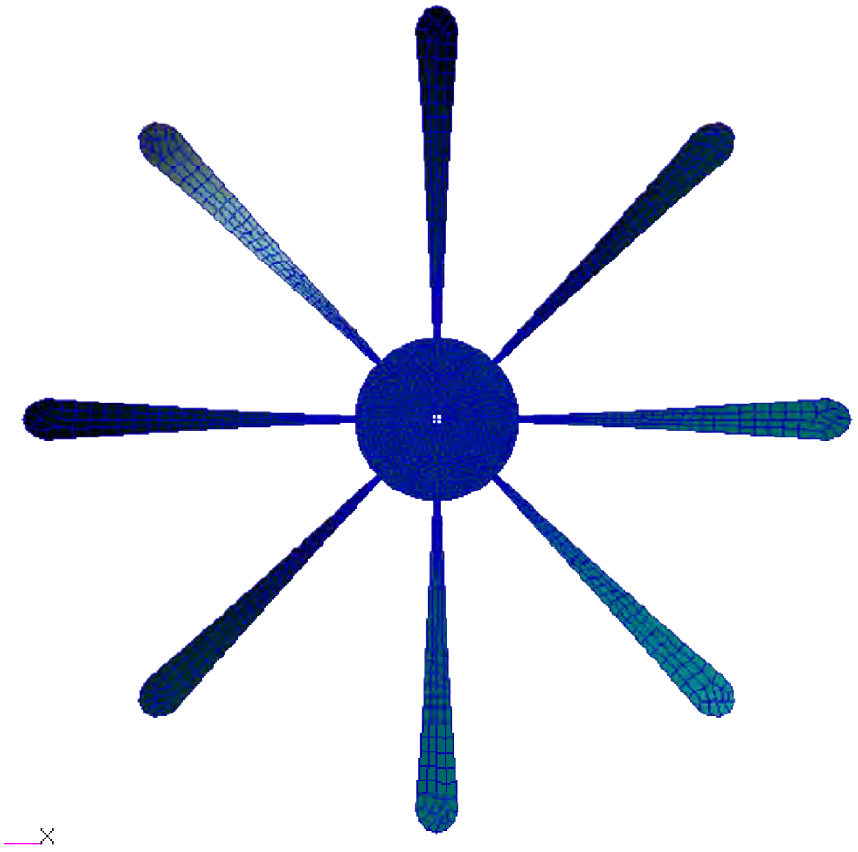






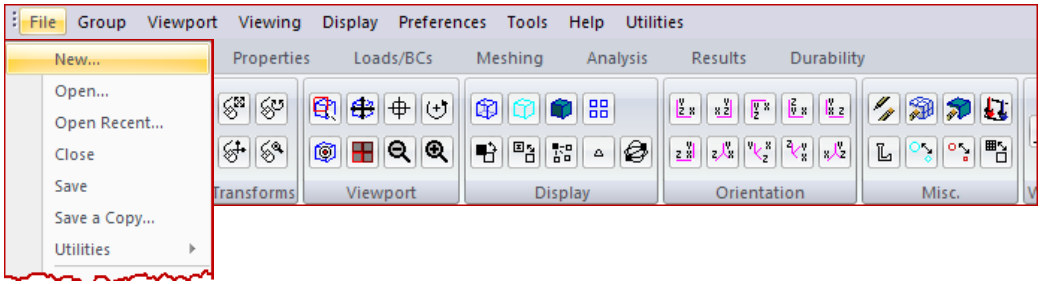
Copyright© 2012 MSC.Software Corporation

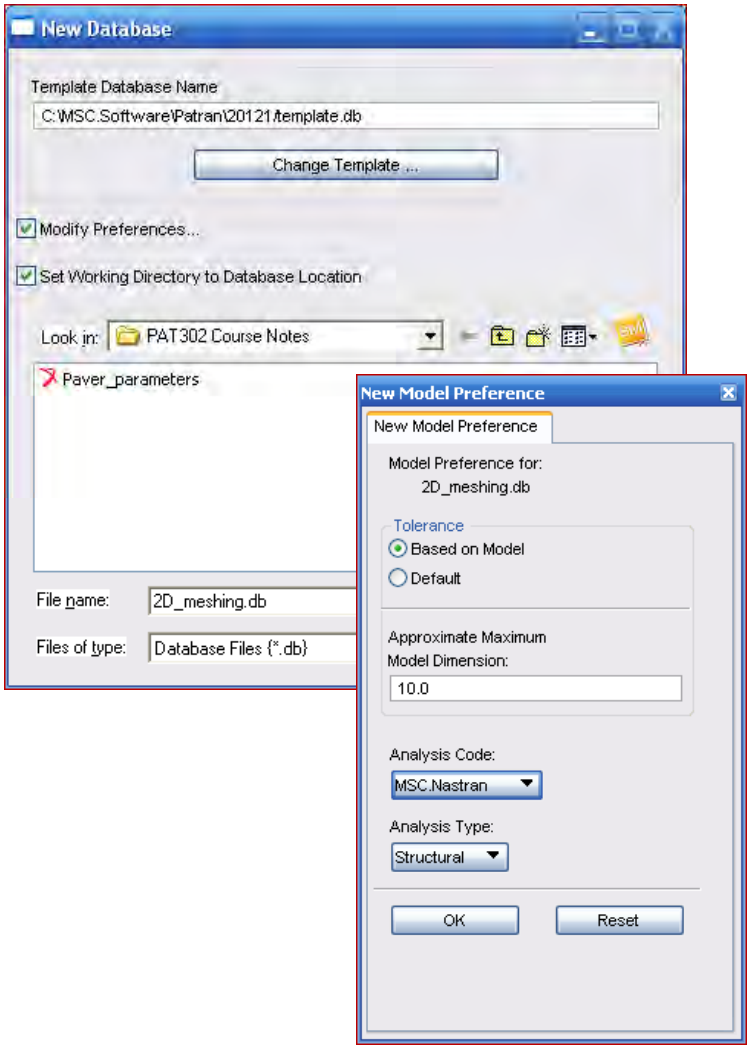
CASE STUDY: 2D MESHING



Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING

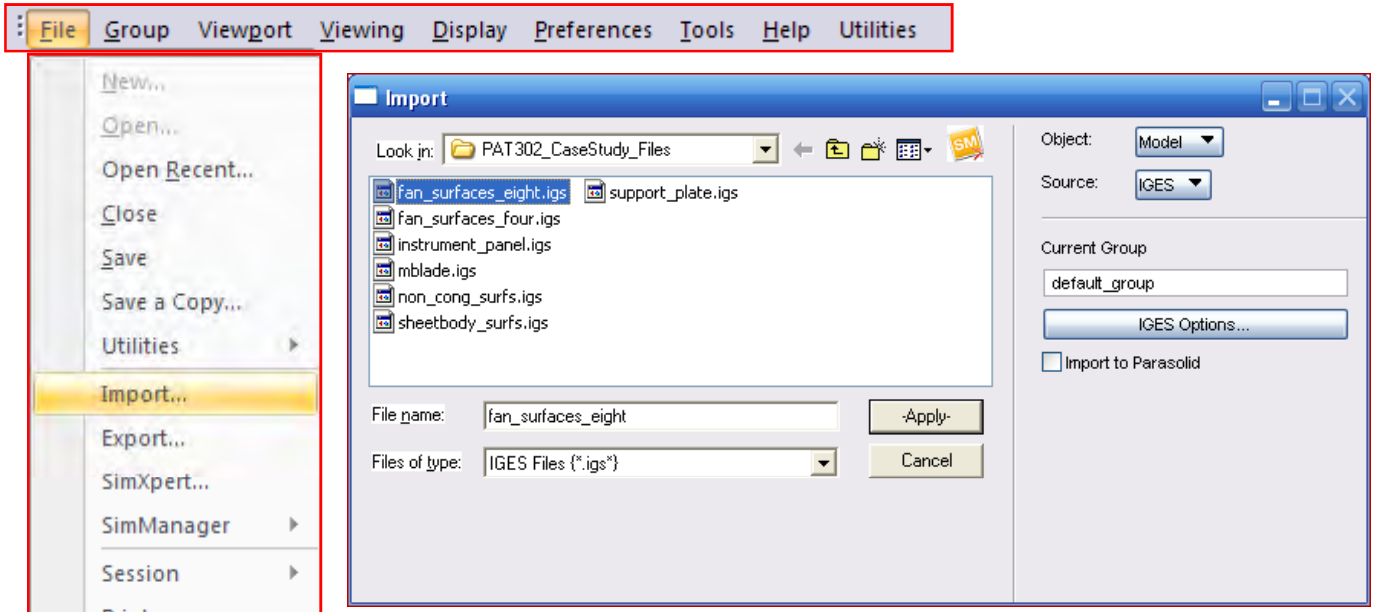




- **Create a new database**
  - Enter **2D_meshing.db** for the File name.
  - Set the *New Model Tolerances* to **Based on Model**.
  - Set the *Analysis Code* and *Analysis Type* to **MSC.Nastran** and **Structural**, respectively.

Copyright© 2012 MSC.Software Corporation

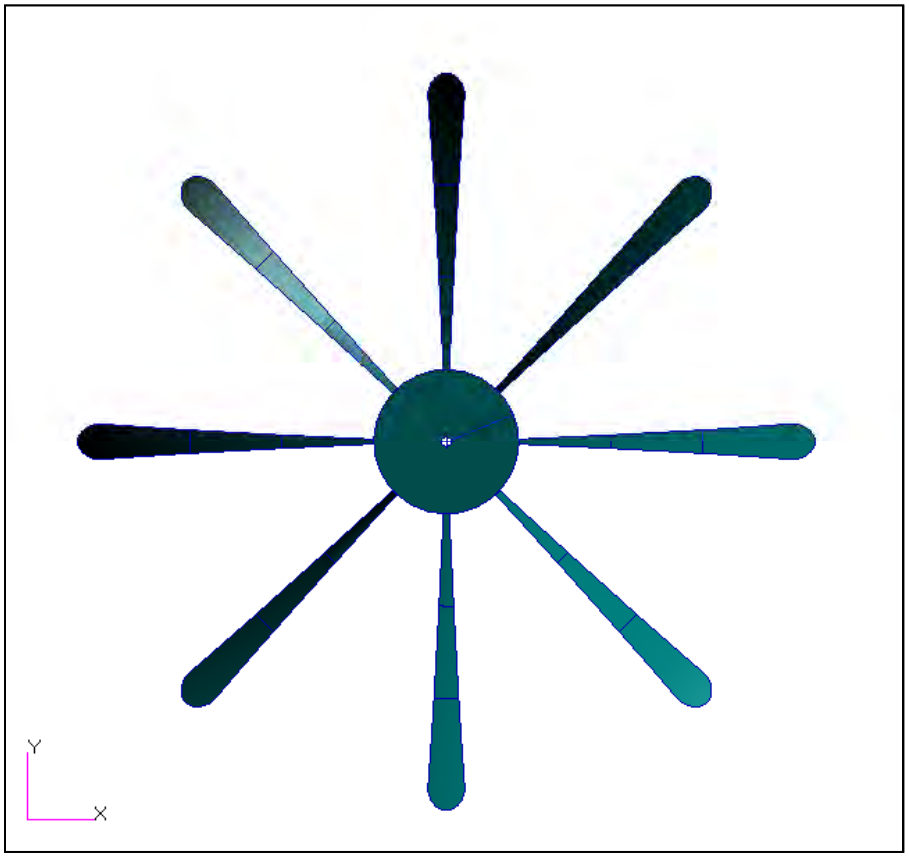
CASE STUDY: 2D MESHING

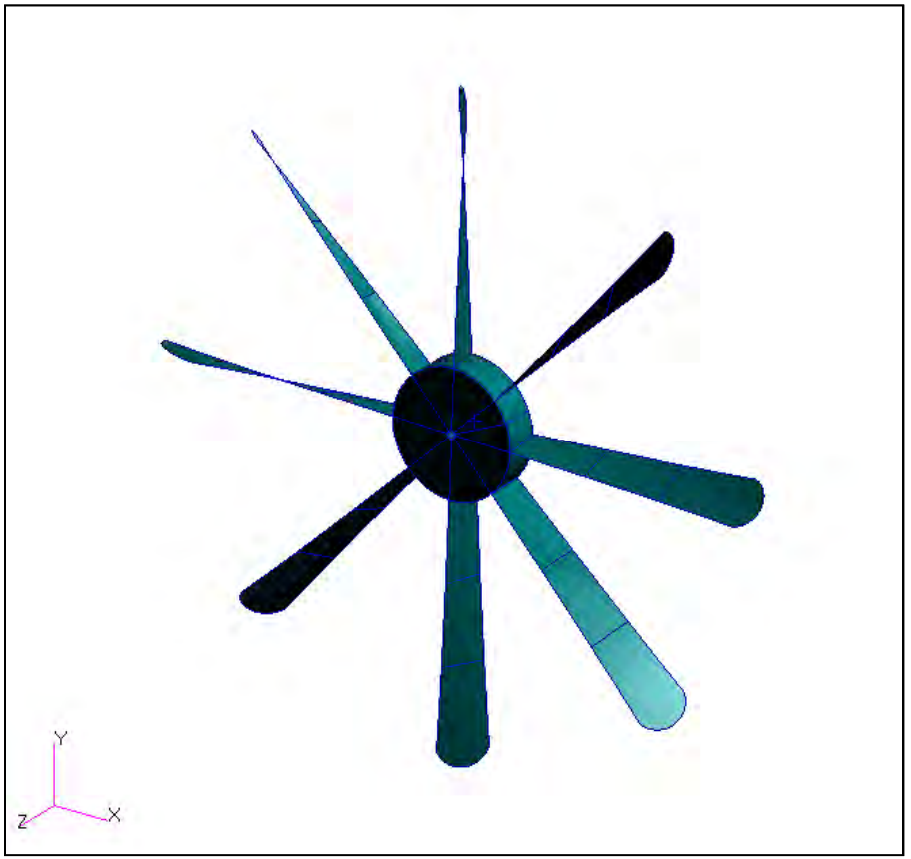


- **Import the IGES file:**
  - Set *Source* to **IGES**. **Import to Parasolid**
  - Uncheck **Import to Parasolid**.
  - Select **fan_surfaces_eight.igs** and click **Apply**.
  - Click **OK** when the import summary appears.
  - Click on the **Smooth shaded** and **Front view** icons.
  - Then click on the **Iso 1 View** icon.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING



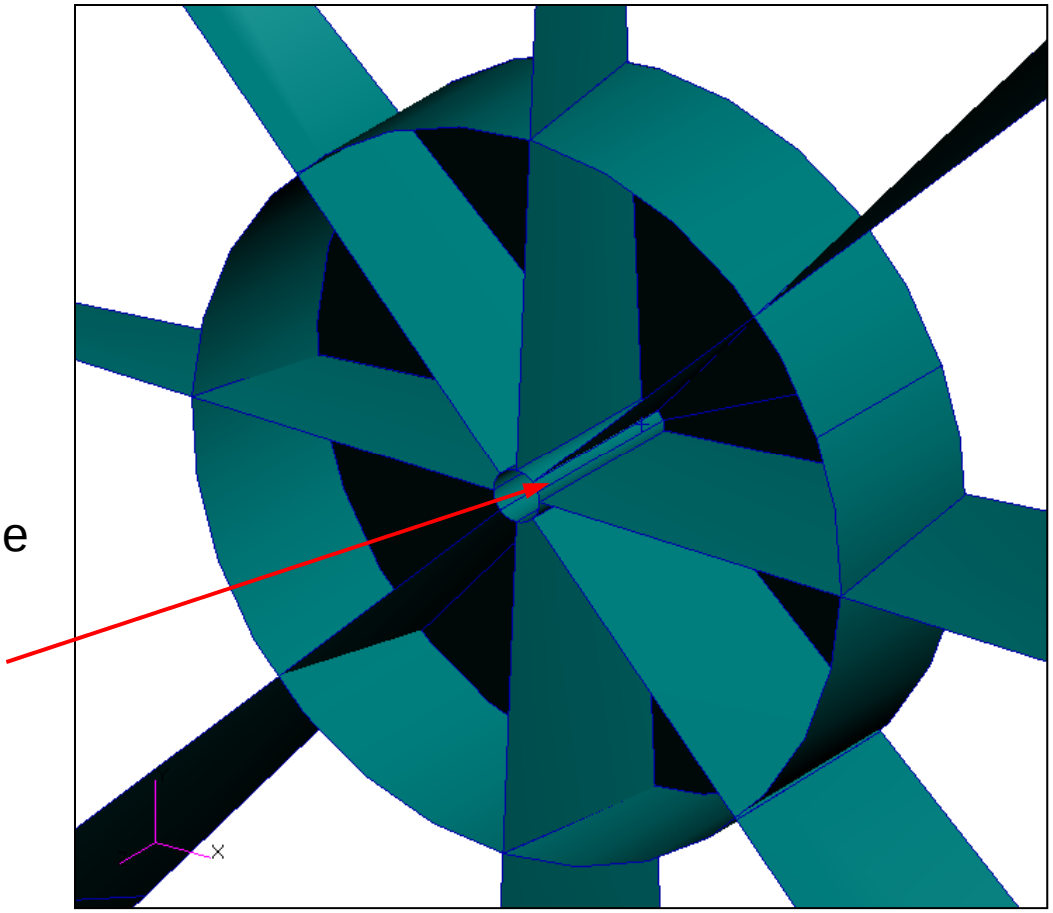


Illustrated here is the model from the front and from the *Iso 1 View* view.

Copyright© 2012 MSC.Software Corporation

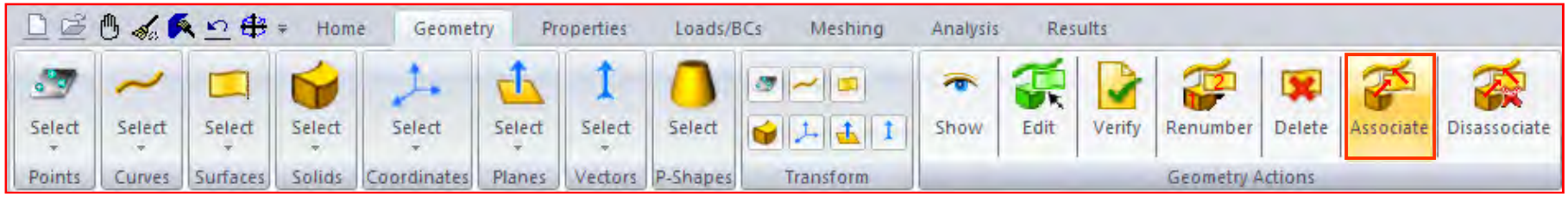
CASE STUDY: 2D MESHING

- **Problem Description**
  - The model consists of eight blades that pass through all the way to the center cylinder.
  - The task is to 2D mesh the surfaces using the Paver mesher with Quad 4 elements. Some of the surfaces are congruent, while others are not. Also, because there are some surface edges between the two adjacent fan blade surfaces at the center, there is not much room for well-shaped Quad4 elements.

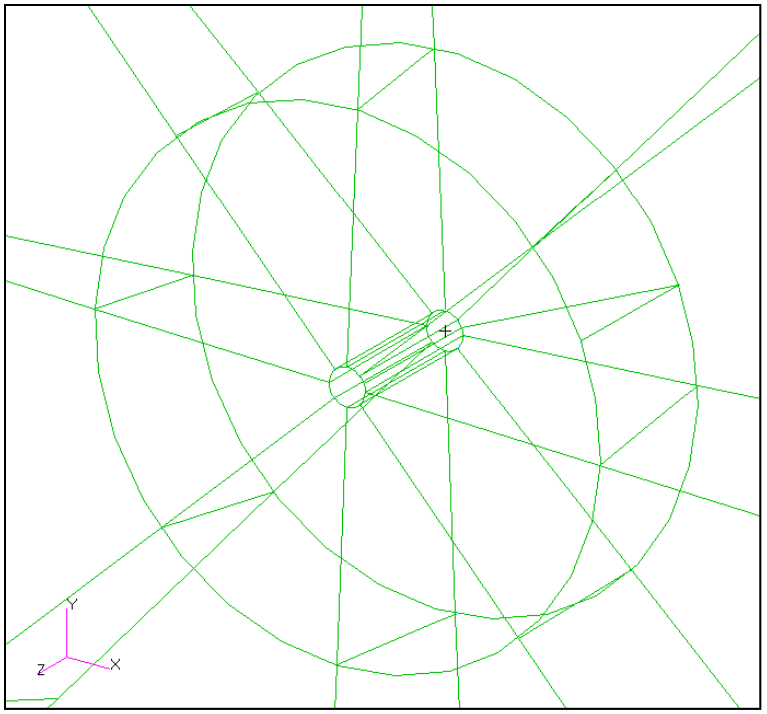


Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING

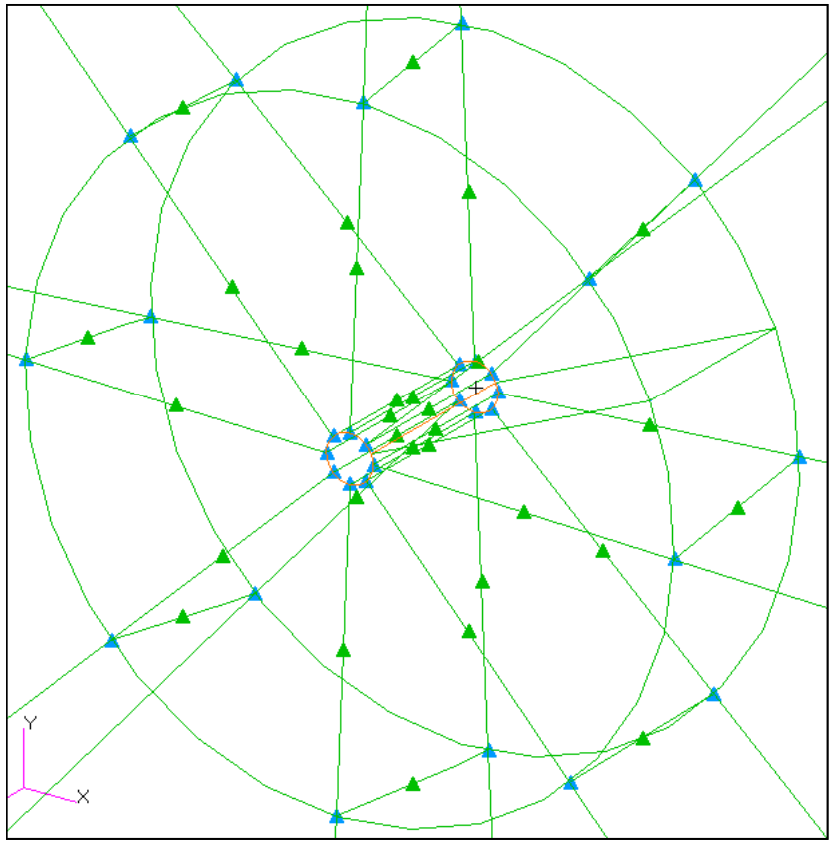


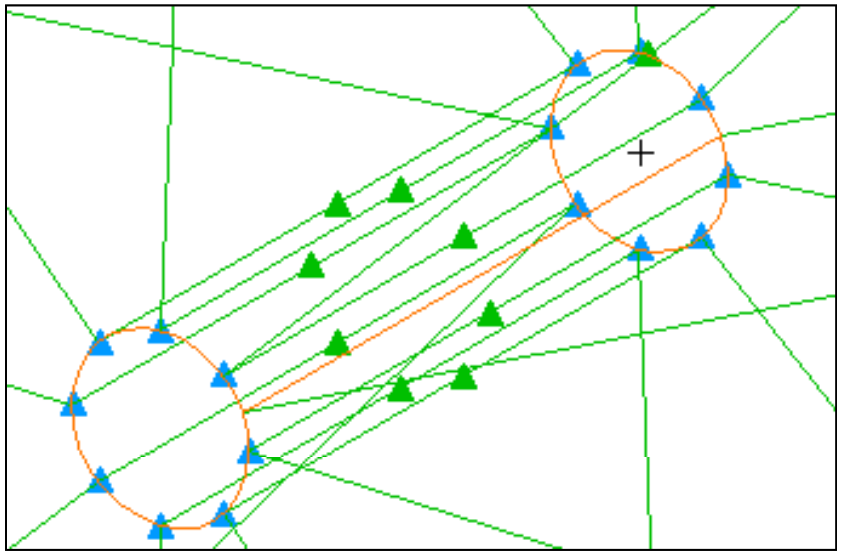
- **The first task is to associate some surface edges to surfaces**
  - All associations will be made at the central portion of the fan blade assembly, e.g. where the fan blades touch the cylinders.
  - Click on the **Wireframe** icon.
  - Under the *Geometry* tab, click on **Associate** in the *Geometry Actions* group. Set the *Object* and *Method* as **Curve** and **Surface** respectively
    - Associate every surface edge that touches the central assembly of the fan



Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING



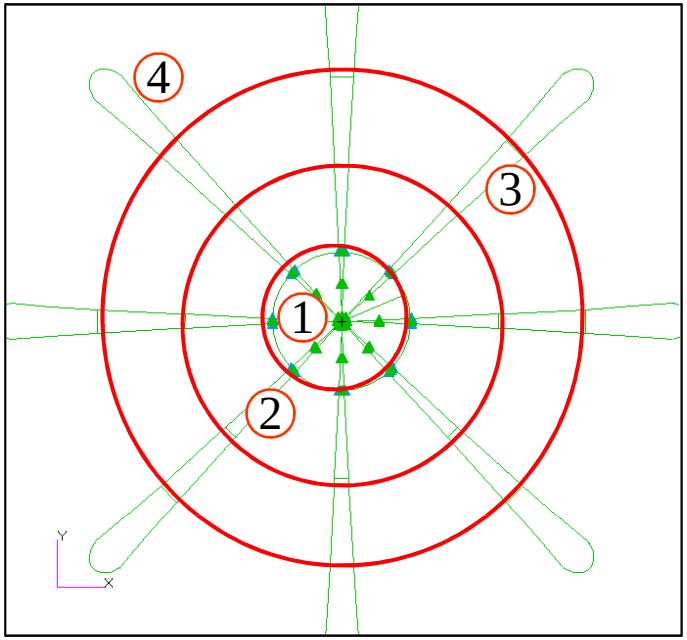


- **Associating the geometry**
  - Shown above are all the curves/edges that are associated to surfaces
  - This will ensure that the elements will be congruent where the surfaces intersect

Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING

- **Use the mesh control to assign different GELs to various surfaces**
  - More elements will be created on the sections of the fan blades closer to the center and fewer will be created near the tip of the blades
  - The following table lists the surface layers on the blade and the respective GELs:

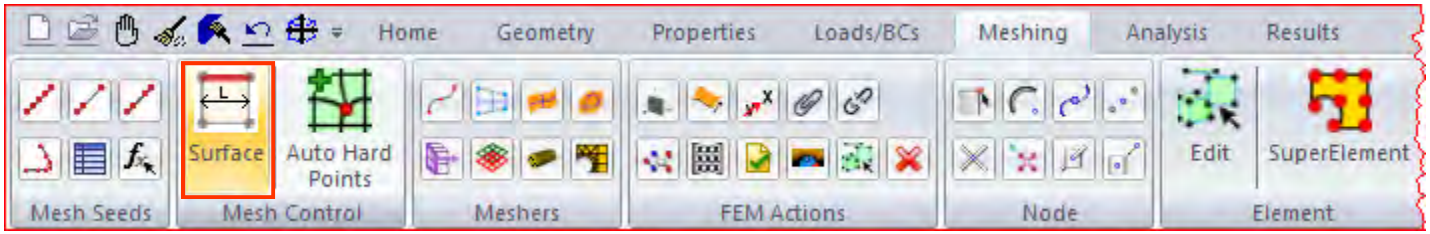


Radial Layer of Surfaces	Surfaces in Layer	Global Edge Length
1	Surface 5:33:4	0.50
2	Surface 6:34:4	0.75
3	Surface 7:35:4	1.00
4	Surface 8:36:4	1.25

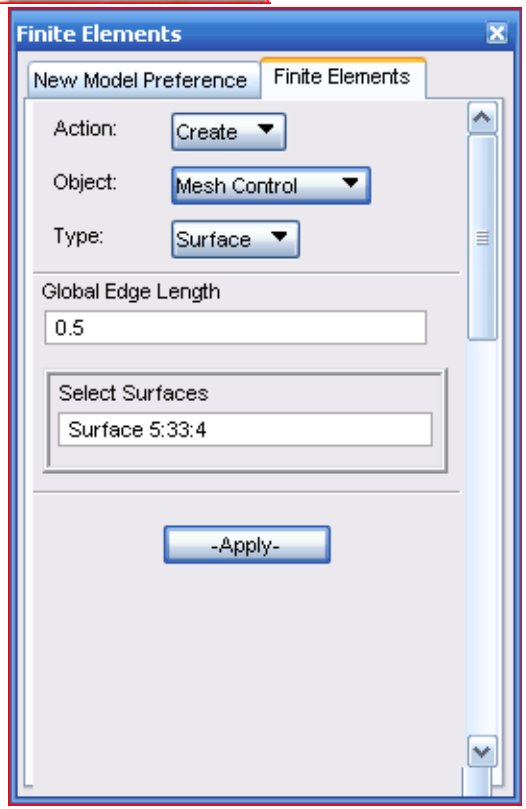
Illustrated here are the radial layers of the fan blades. Note that when selecting the section of the fan blades on the inner section of the fan assembly, do not include the cylinders.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING

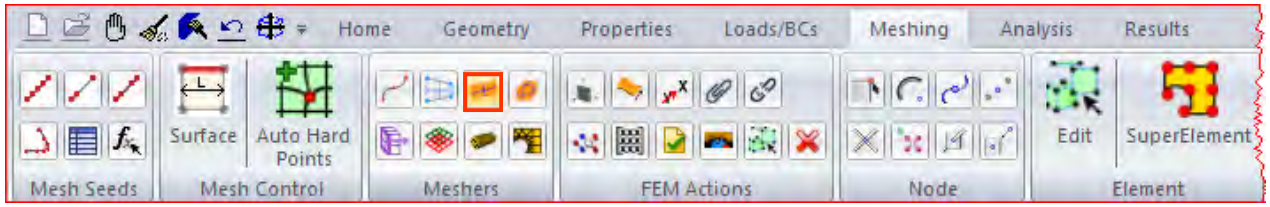


- **Create the mesh control:**
  - Under the *Meshing* tab click on **Surface** in the *Mesh Control* group.
  - Enter the first GEL, e.g. **0.5**
  - Enter, or select, the corresponding surfaces, e.g. **Surface 5:33:4** (shown in the previous table)
  - Repeat the procedure until mesh control has been created for each layer of the fan blades.

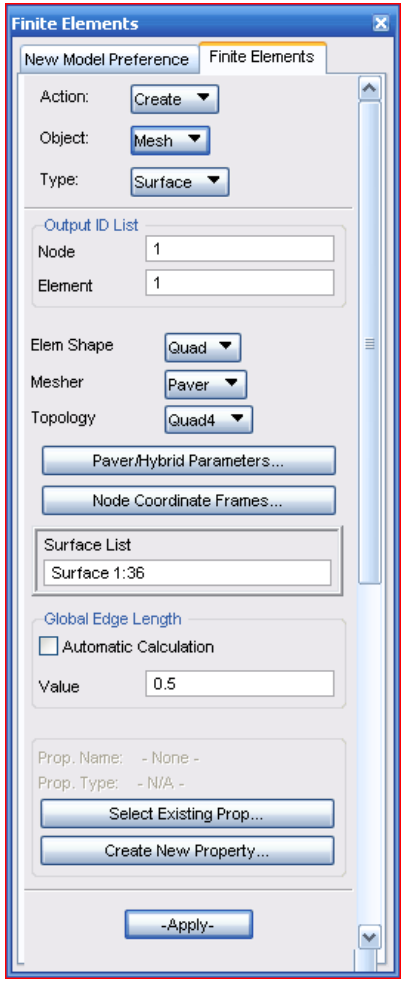


Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING

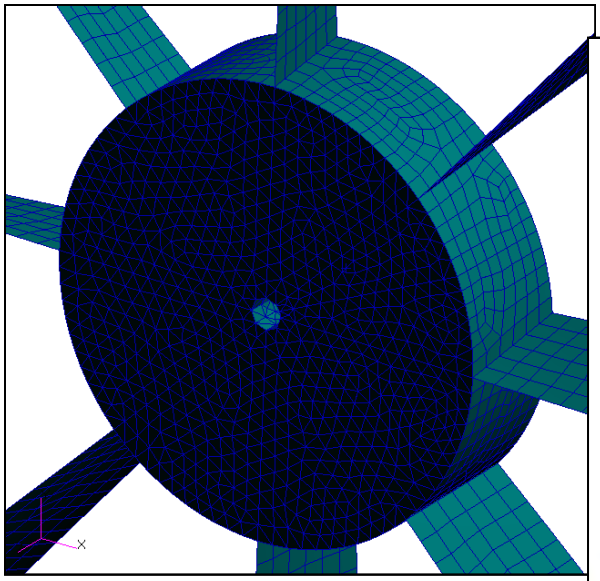


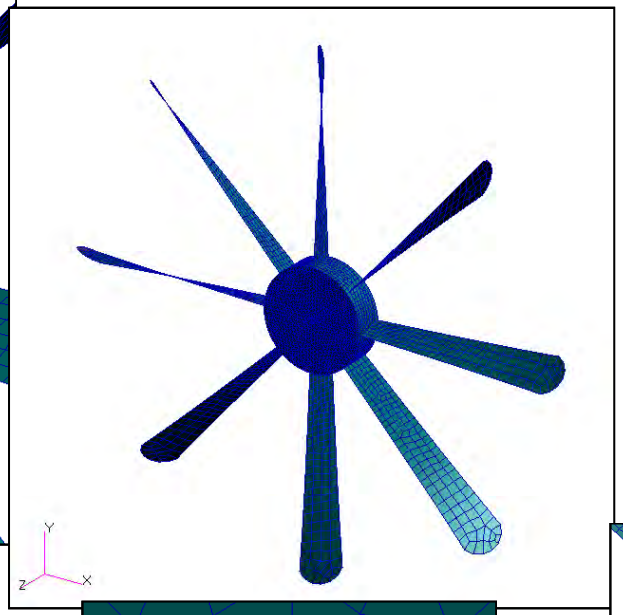
- **Paver mesh the model:**
  - Under the *Meshing* tab click on **Surface** in the *Meshers* group.
  - Set the *Elem Shape*, *Mesher*, and *Topology* to **Quad**, **Paver**, and **Quad4**, respectively.
  - Turn off the toggle for *Automatic Calculation* and enter **0.5** for the GEL.
  - Select the entire model for the *Surface List*.
  - Click **Yes for all** when asked to retry using tri elements.
  - Click on the **Smooth shaded** icon.
  - Click on the **Plot/Erase** icon and erase the geometry.

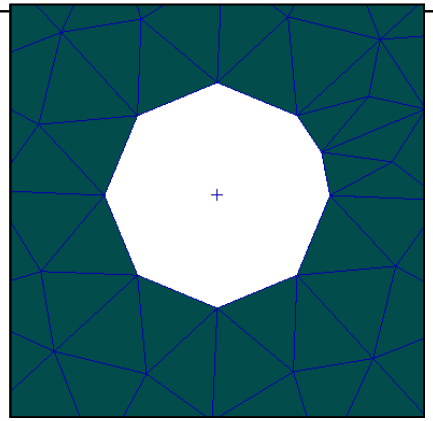


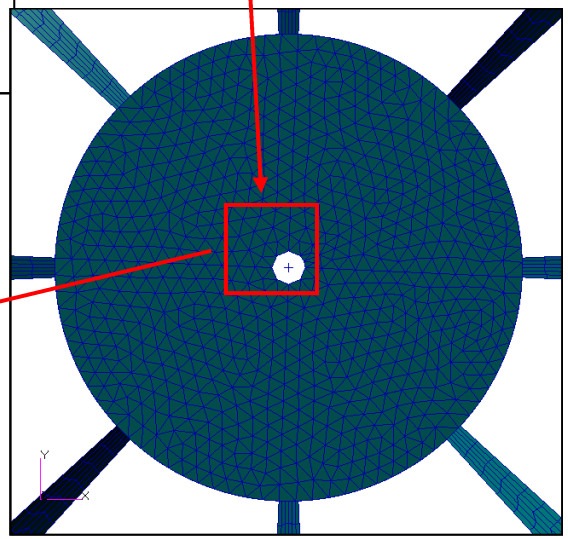
Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING









Shown here are several views of the meshed model. Note the tri elements on the front face of the cylinder in the front view.

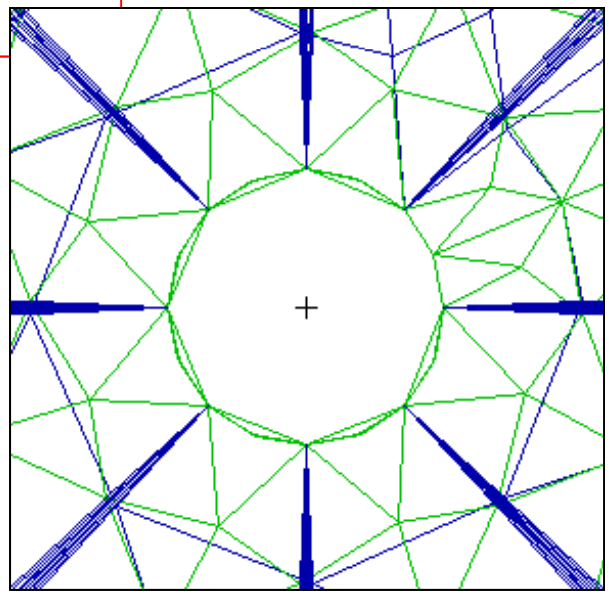
Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING

- **Observing the Paver mesh:**
  - During the mesh process, the following messages were issued by Patran:

```
fem_create_mesh_surf_4( "Paver", 49680, "Surface 1:36", 4,
["0.5", "0.1", "0.2", "1.0"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @
fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @
fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )
$# Warning reported from application MSH
$#     Trying to mesh in parametric space.
$# Question from application FEM
$#     The Mesher could not complete the mesh on Surface 1.
$# Would you like to retry using TRI elements?"
```

  - The Paver mesher was directed to use Quad4 element topology. However, as illustrated to the right, it is easy to see that there is only enough room for well-shaped tri elements.
  - In order to see the difference in the resultant mesh, another model with only four blades will be imported and meshed in the same manner.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING

Create a new database(db) and import another model:

a. Close the current db and create a new db called **2D_meshing_2.db**.

b. Import the IGES file:

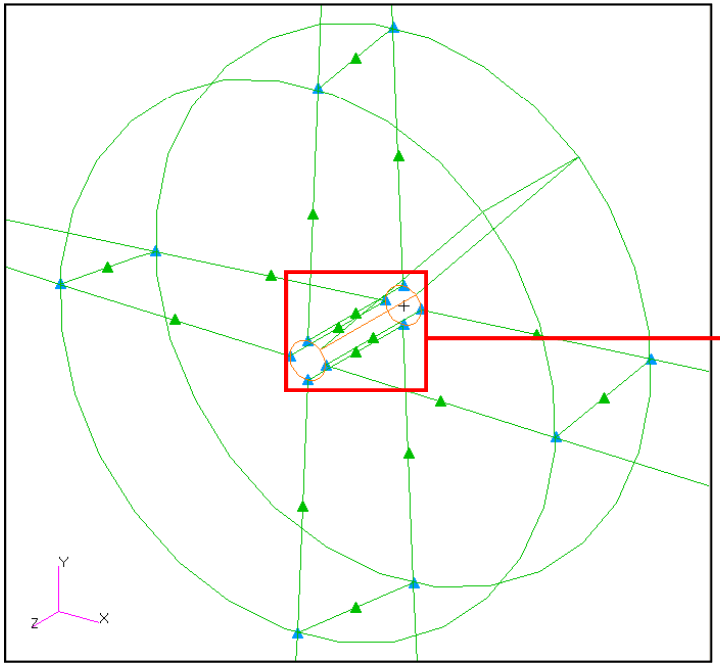
- Select **Import…** from the *File* menu
- Import the file **fan_surfaces_4.igs**.
- Click **OK** when the import summary appears.

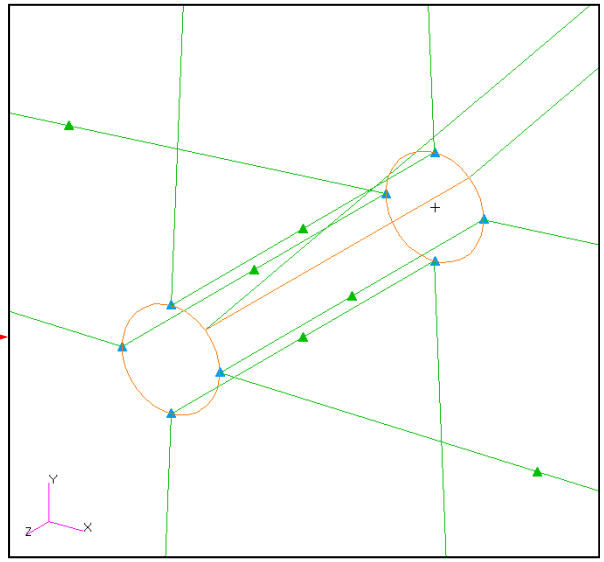
c. Click on the **Smooth shaded** and **Iso1 view** icons.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING

- **Associate central section of the fan blade assembly:**
  - Click on the **Wireframe view** icon.
  - Under the *Geometry* tab, click the **Associate** in the *Geometry Actions* group. Select **Curve** as the *Object* and **Surface** as the *Method.*
  - Use the View corners icon to zoom in on the central section of the model.





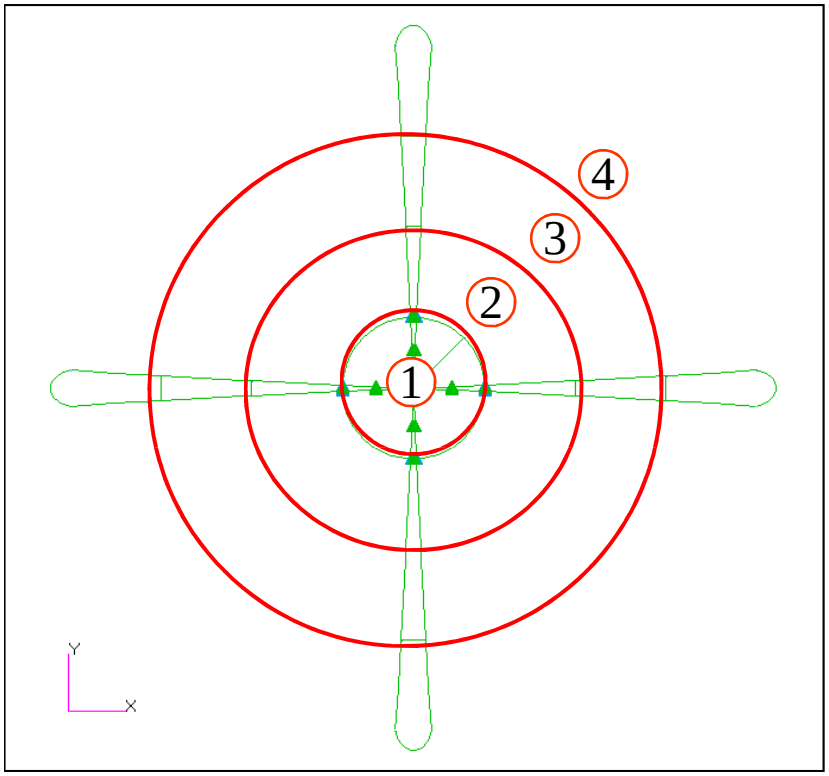
Shown here is the model with all of the edges associated to the central cylinder of the fan.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING

- **Create the mesh control for the model**
  - Under the *Meshing* tab click on **Surface** in the *Mesh Control* group.
  - Select or enter the following surfaces according to the table given:

Layer	Surfaces	*GEL*
1	5:17:4	0.50
2	6:18:4	0.75
3	7:19:4	1.00
4	8:20:4	1.25

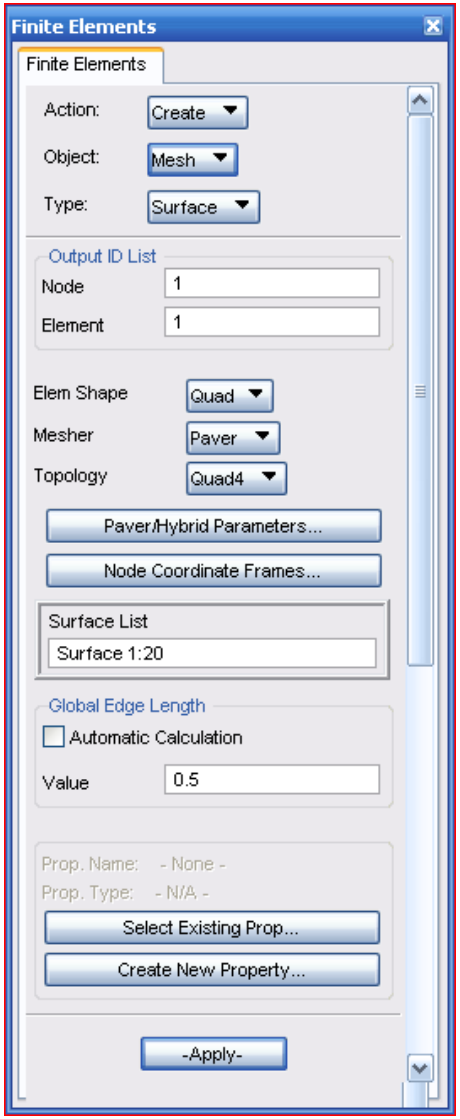


Illustrated here are the radial layers of the fan blades. Again, note that when selecting the section of the fan blades on the inner section of the fan assembly, do not include the cylinders.

Copyright© 2012 MSC.Software Corporation

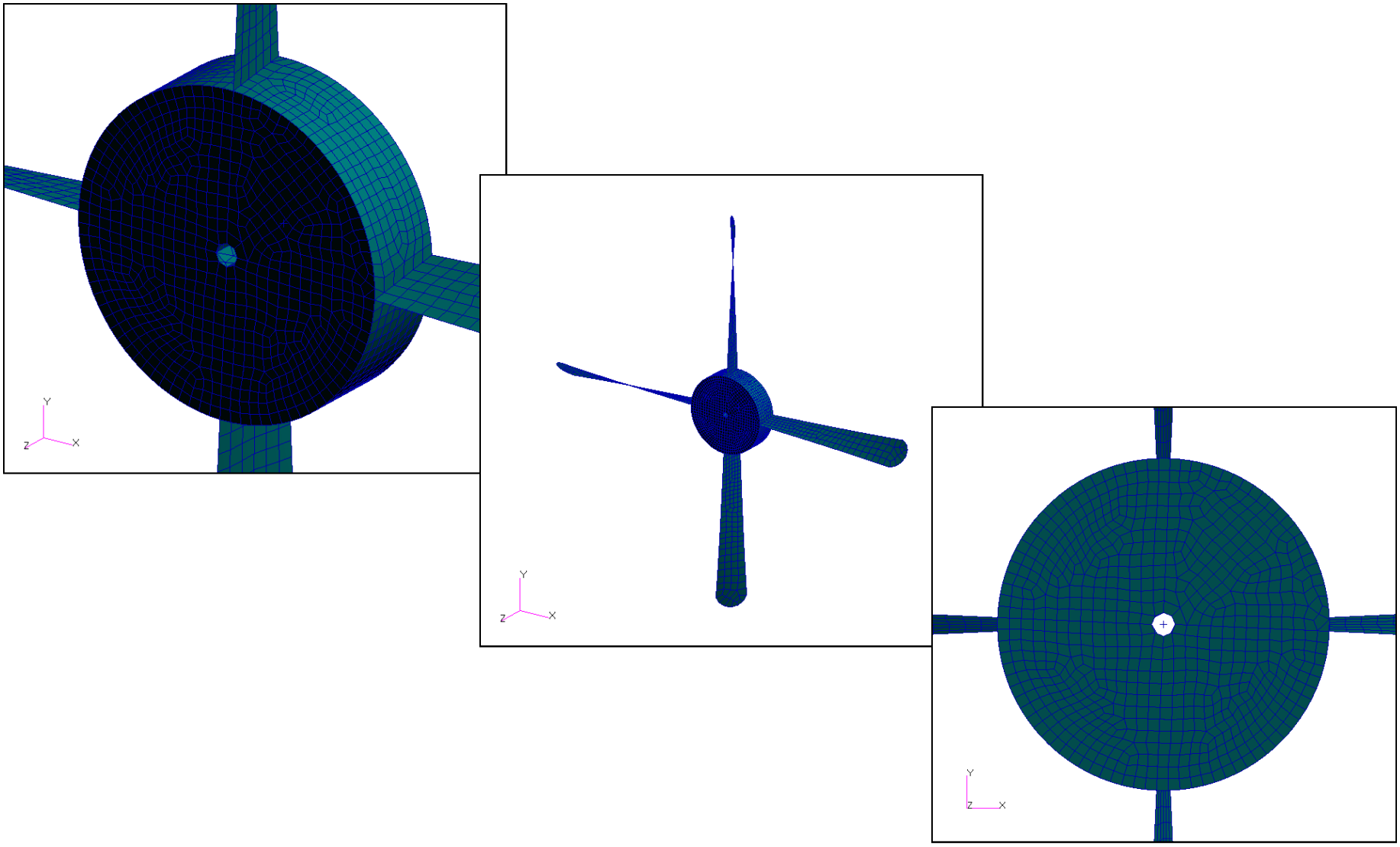
CASE STUDY: 2D MESHING

- **Paver Mesh the model:**
  - Under the *Meshing* tab click on **Surface** in the *Meshers* group.
  - Set the *Elem Shape*, *Mesher*, and *Topology* to **Quad**, **Paver**, and **Quad4**, respectively.
  - Turn off the toggle for *Automatic Calculation* and enter **0.5** for the *GEL*.
  - Under *Surface List*, select the entire model.
  - Select **Apply**.
  - Click on the **Smooth Shaded** and **Iso 1 View** icons.
  - Click on the **Plot/Erase icon** and erase the geometry.



Copyright© 2012 MSC.Software Corporation

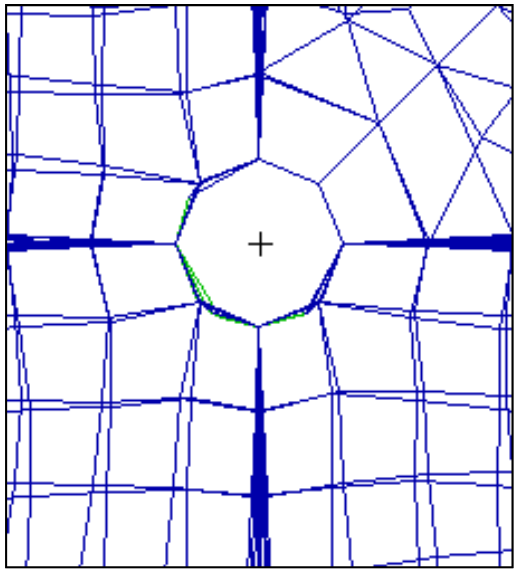
CASE STUDY: 2D MESHING

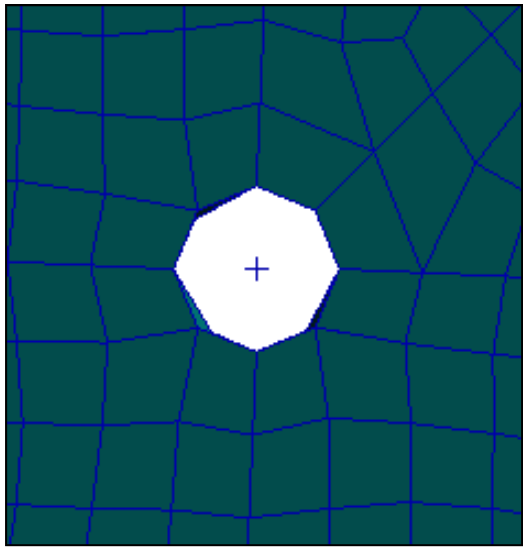


Copyright© 2012 MSC.Software Corporation

CASE STUDY: 2D MESHING

- **Observing the same general area as for the previous model :**
  - There are quad elements where earlier there were tri elements
  - There are still a few tri elements at the intersection of the fan blades and the inner cylinder
- **This shows that, to create a good mesh, enough space is needed for well-shaped elements.**



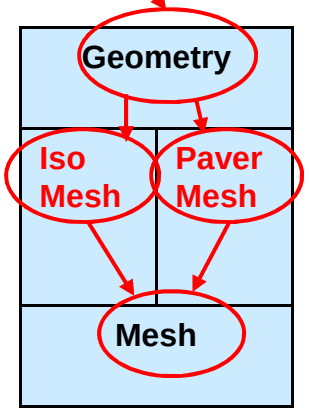


Copyright© 2012 MSC.Software Corporation

2D MESHING & SURFACE TOPOLOGY

- **The next set of techniques involve creating surfaces using different topological approaches. The surfaces are meshed using the IsoMesh and Paver meshers.**

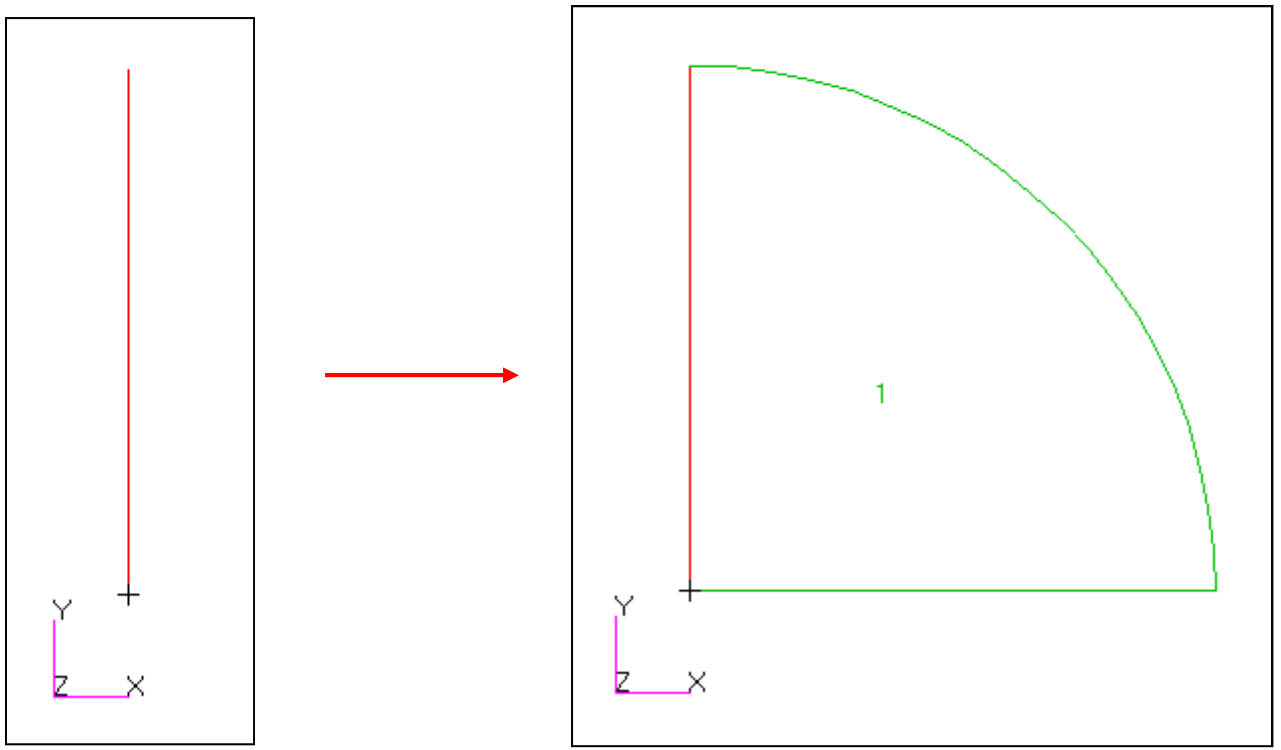
Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated Surface	Iso Mesh	Paver Mesh	Sheet Body	Composite Surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	



Copyright© 2012 MSC.Software Corporation

2D MESHING & SURFACE TOPOLOGY

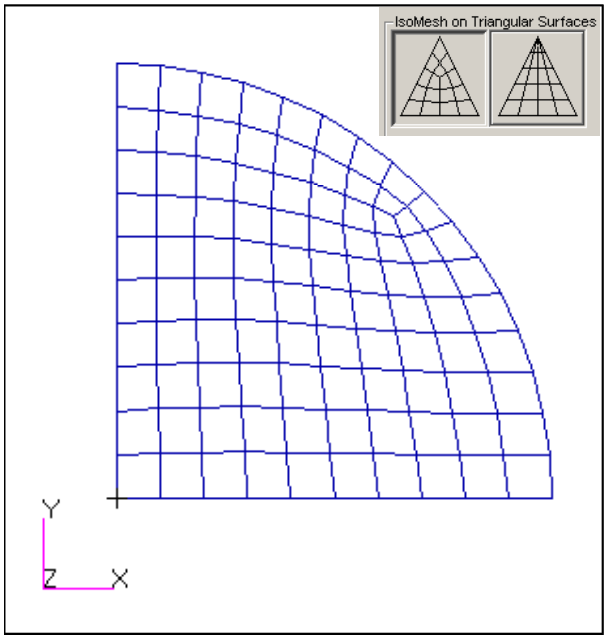
- **Create a curve and revolve it. Then IsoMesh and Paver mesh.**



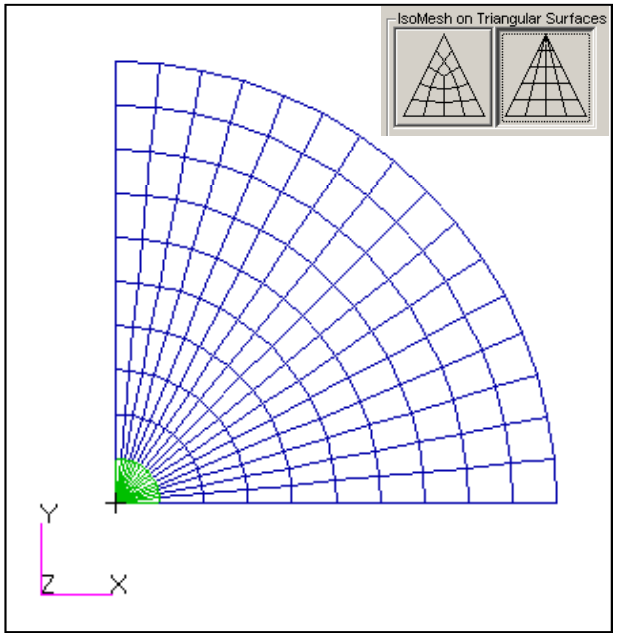
Copyright© 2012 MSC.Software Corporation

2D MESHING & SURFACE TOPOLOGY

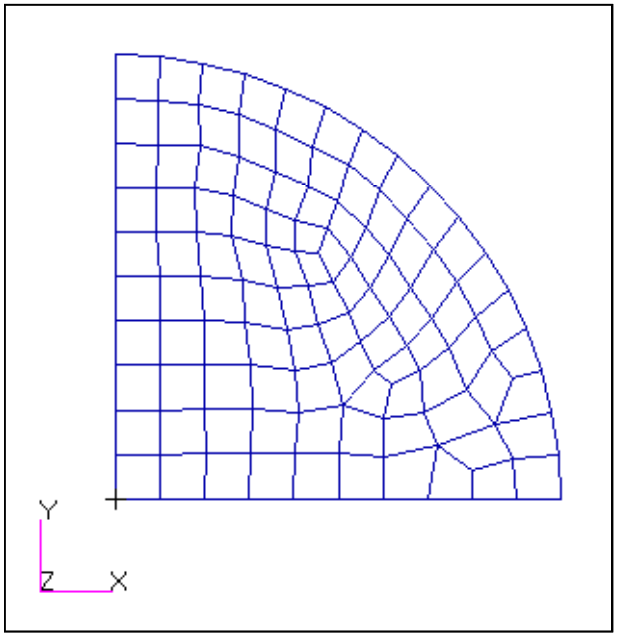
- **Results of IsoMesh and Paver meshing. Use of the IsoMesh parameter for IsoMesh on Triangular Surfaces.**



IsoMesh with Quads



IsoMesh with Quads and Tris

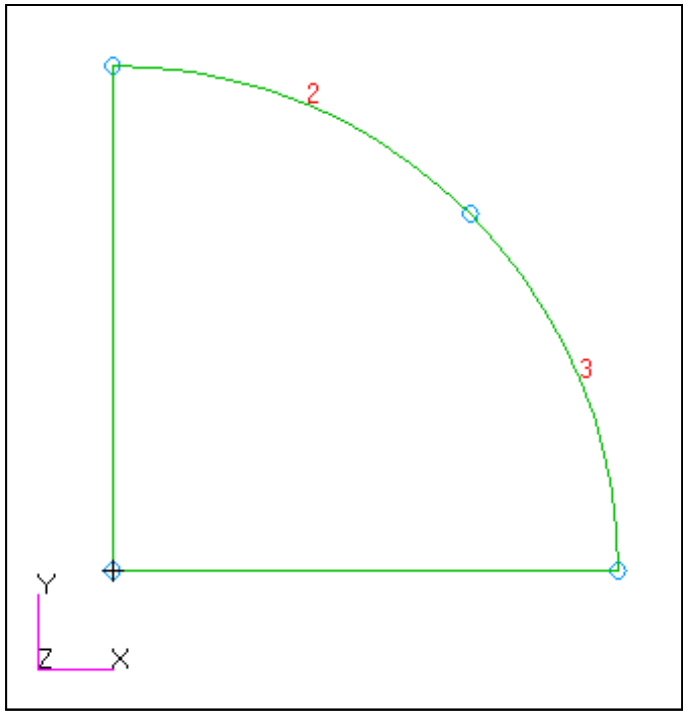


Paver with Quads

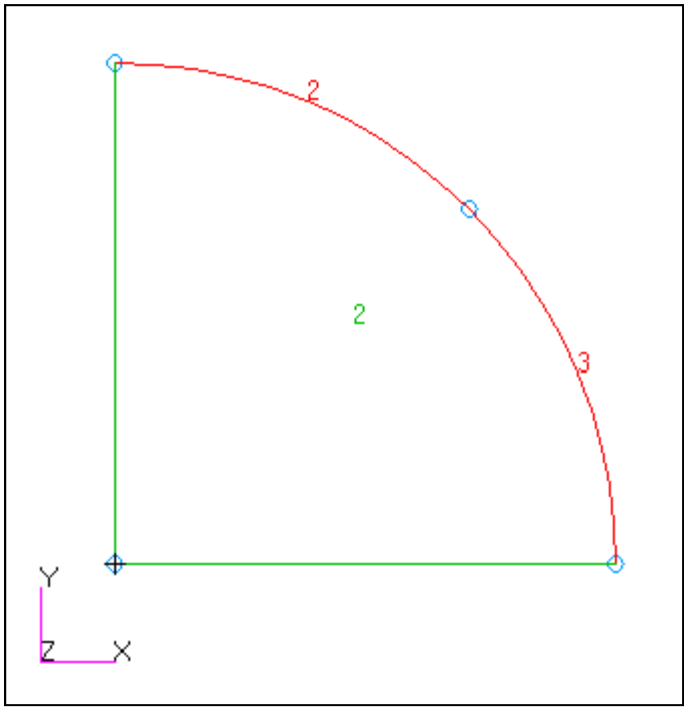
Copyright© 2012 MSC.Software Corporation

2D MESHING & SURFACE TOPOLOGY

- **Use different geometry topology for creating surface.**



Break Curved Edge Midway

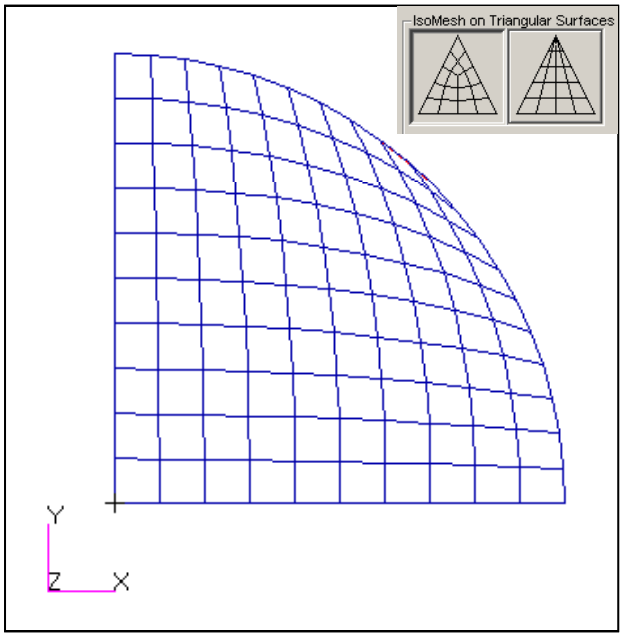


Create Surface using two Edges and two Curves

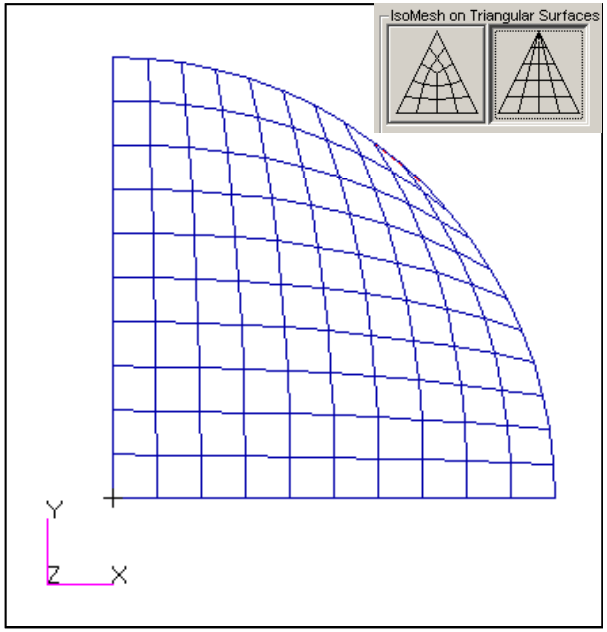
Copyright© 2012 MSC.Software Corporation

2D MESHING & SURFACE TOPOLOGY

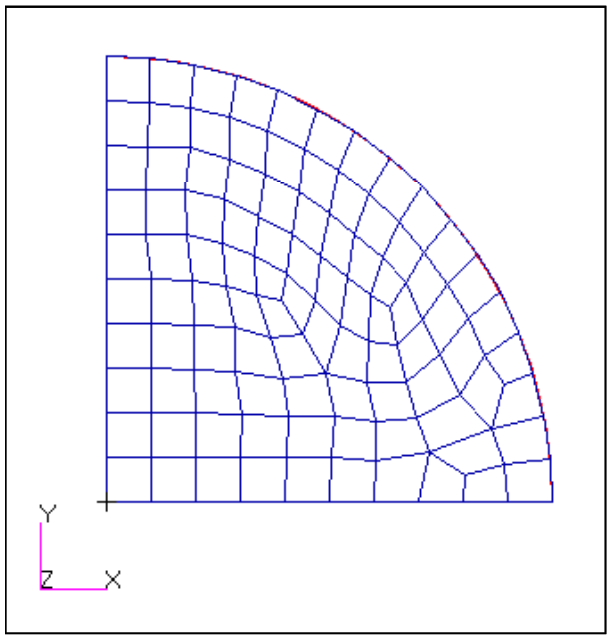
- **Results of IsoMesh and Paver meshing. Use of the IsoMesh parameter for IsoMesh on Triangular Surfaces.**



IsoMesh with Quads



IsoMesh with Quads and Tris

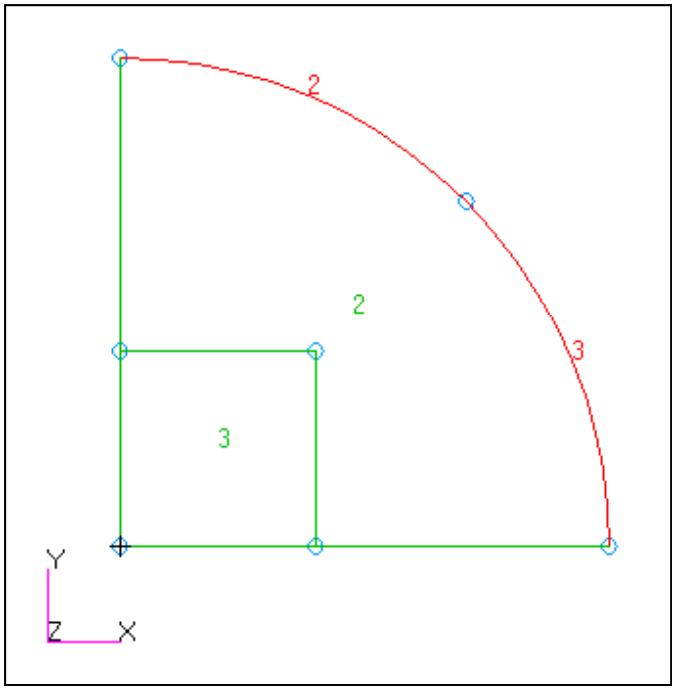


Paver with Quads

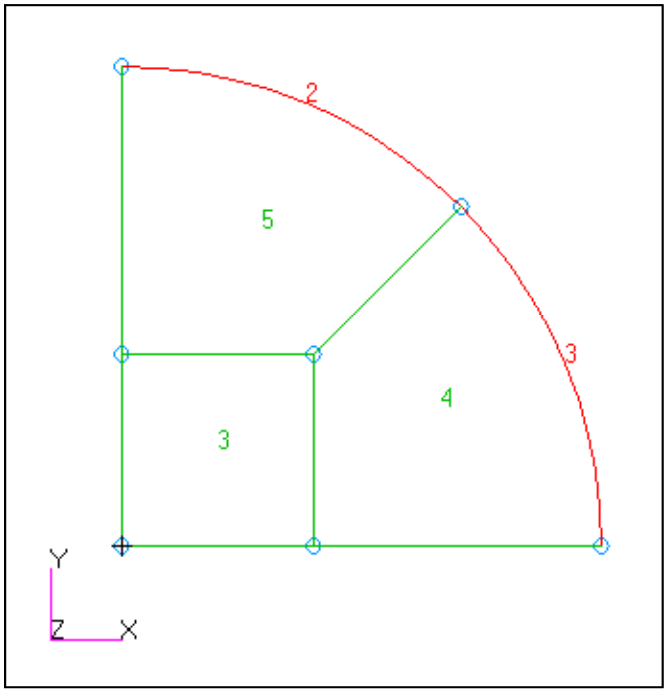
Copyright© 2012 MSC.Software Corporation

2D MESHING & SURFACE TOPOLOGY

- **Improve geometric topology by starting with the previous surface, then creating three improved surfaces.**



Add Square Surface

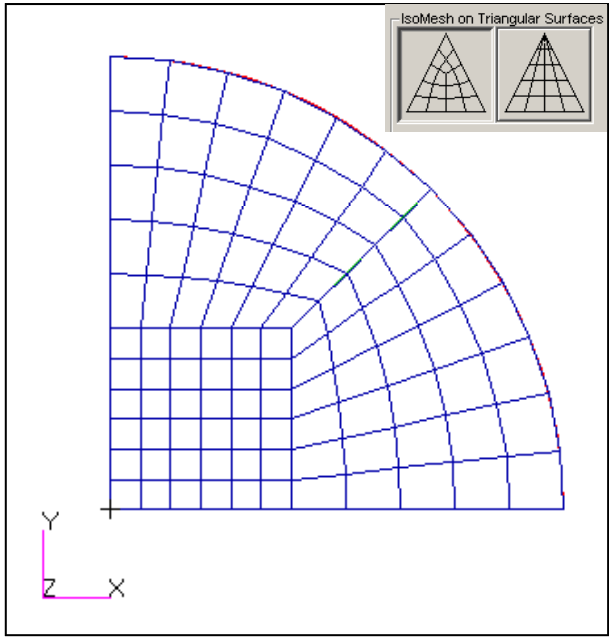


Create two Surfaces using Curve and Edge

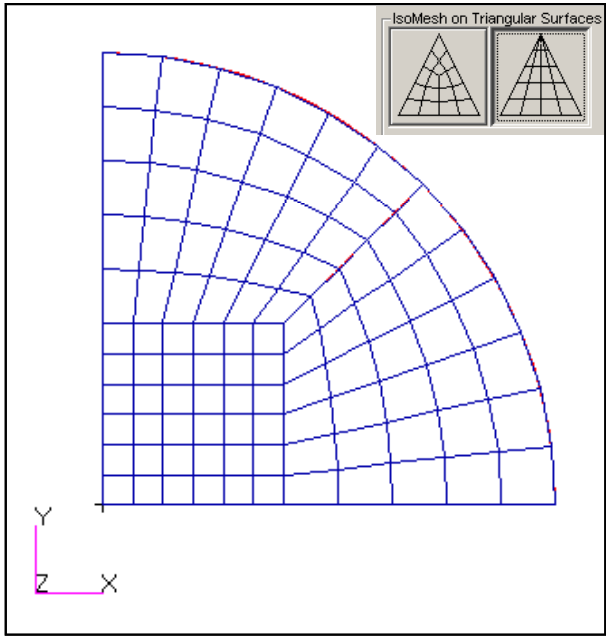
Copyright© 2012 MSC.Software Corporation

2D MESHING & SURFACE TOPOLOGY

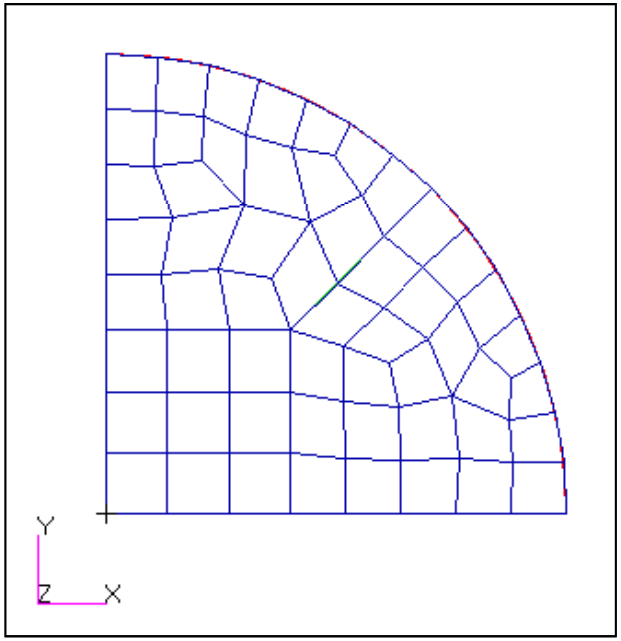
- **Results of IsoMesh and Paver meshing. Use IsoMesh parameter for IsoMesh on Triangular Surfaces.**



IsoMesh with Quads



IsoMesh with Quads and Tris

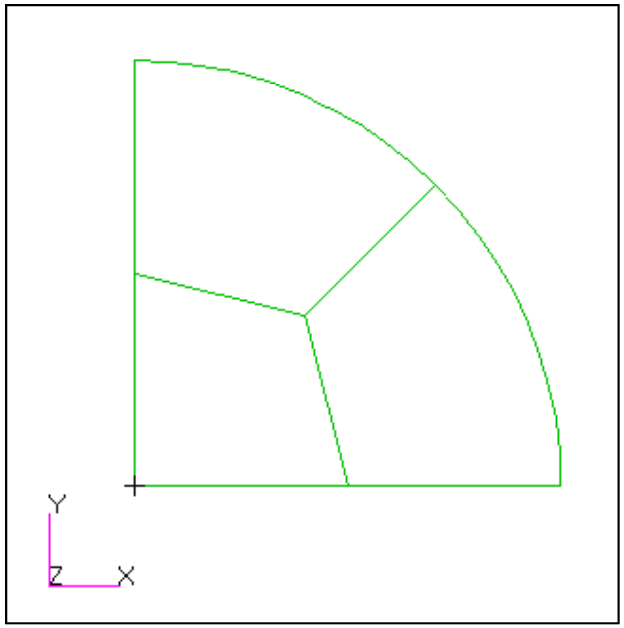


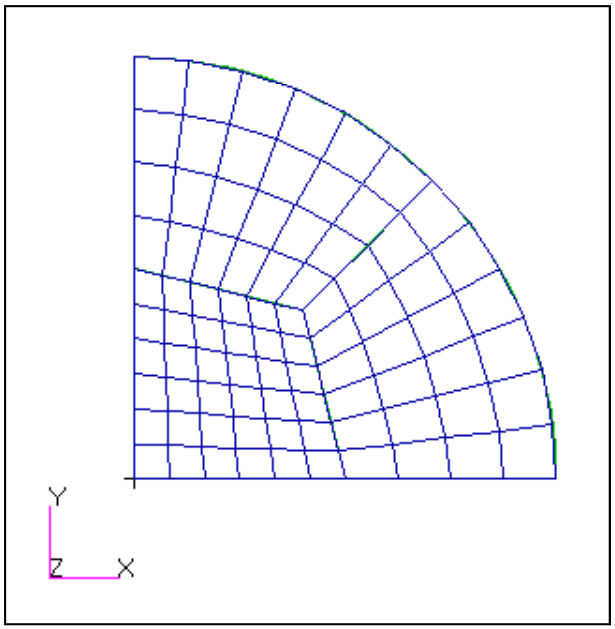
Paver with Quads

Copyright© 2012 MSC.Software Corporation

2D MESHING & SURFACE TOPOLOGY

- **It is clear that the quality (i.e. shape) of the elements is directly dependent on the shape of the geometry.**
- **What can be done to improve the surfaces used for the IsoMesh ?**





Copyright© 2012 MSC.Software Corporation

GEOMETRY EDITING

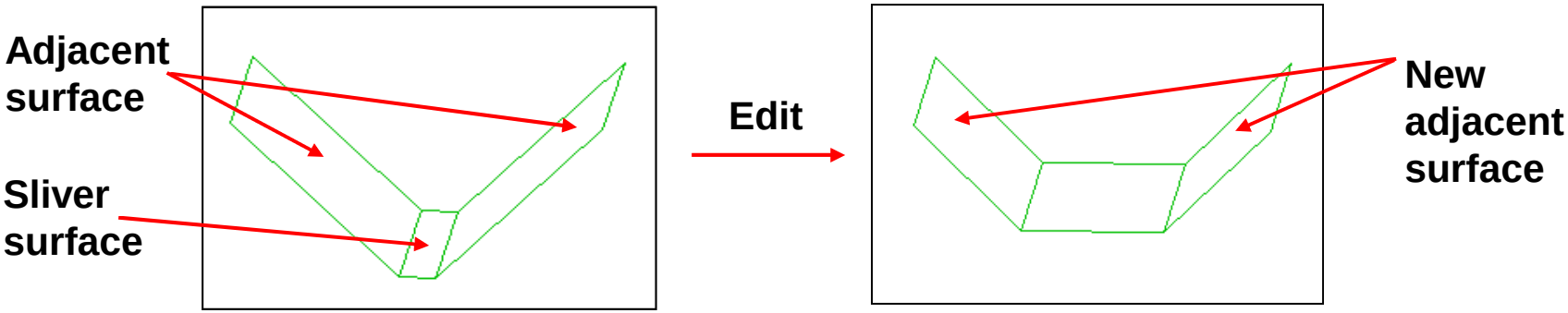
- **Sometimes the geometry available (imported from CAD, IGES, etc., or created in Patran) must be modified prior to meshing, i.e. make surfaces congruent or de-feature surfaces.**

Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated Surface	Iso Mesh	Paver Mesh	Sheet Body	Composite Surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	

Copyright© 2012 MSC.Software Corporation

GEOMETRY EDITING

- **When editing surfaces, consider the effect on the quality of meshes that will be created (e.g. are sharp surface corners being created?), and hence, on the quality of stress results.**
- **Sliver-like surfaces, e.g. sliver surface between two adjacent surfaces, must be dealt with.**
  - Trim back the surfaces adjacent to the sliver, delete the sliver, and create a new surface between the adjacent surfaces.
    - The new surfaces may not follow the original surfaces, but the inaccuracy may be acceptable.



Copyright© 2012 MSC.Software Corporation

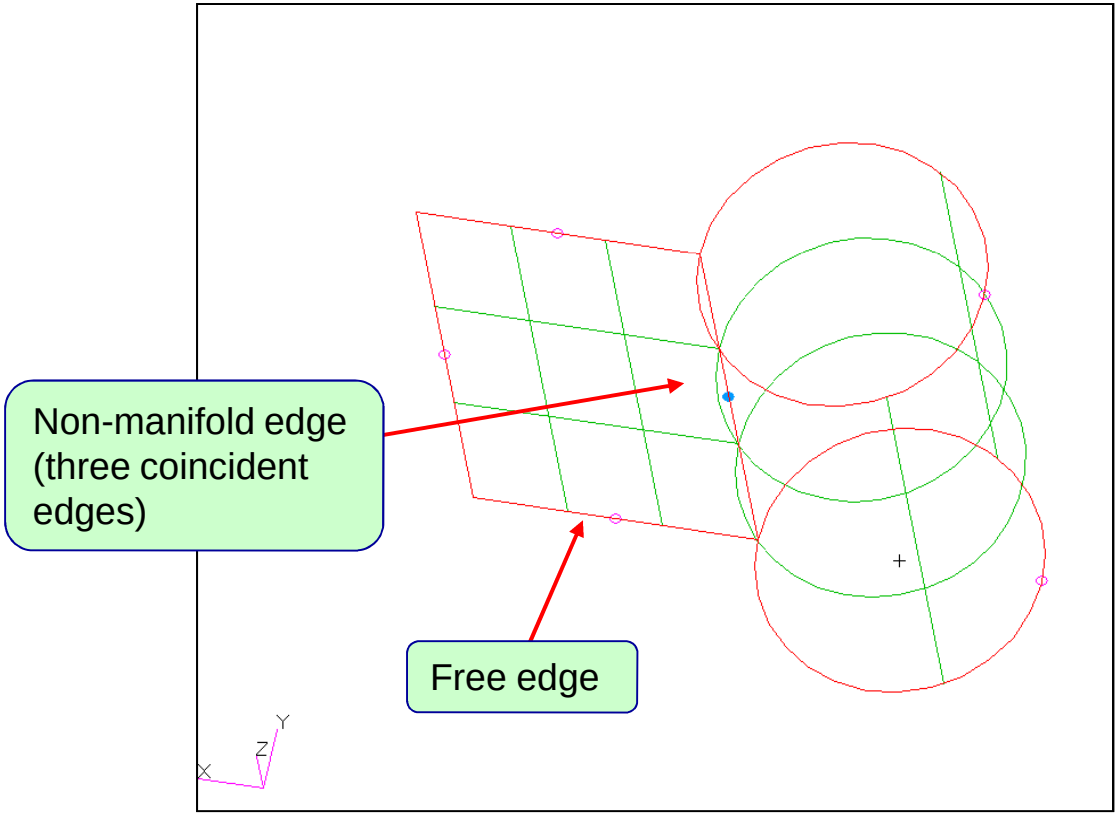
GEOMETRY EDITING TOOLS

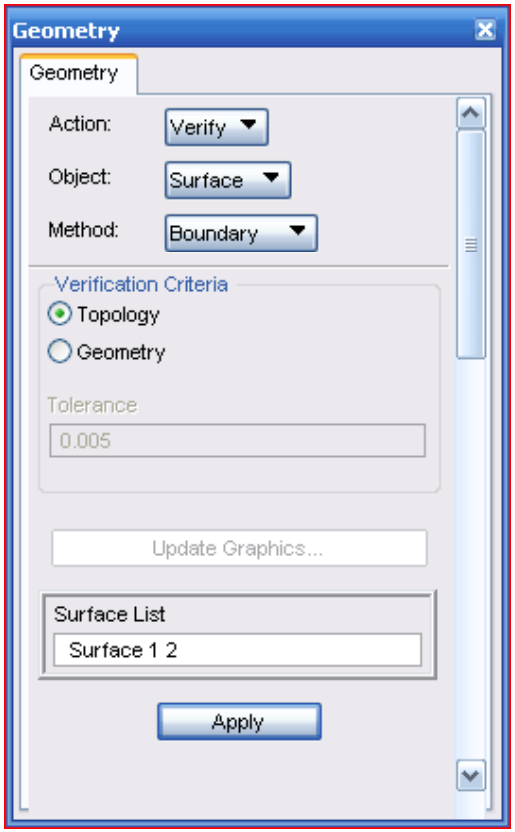
- **Several Geometry: Edit/Surface tools**
  - Break – break a surface into two parts
  - Edge Match – make two non-congruent surfaces congruent, or add a vertex to a surface
  - Extend – lengthen a surface using several different approaches
  - Refit – represent a single complex (in shape) surface with several simpler surfaces; this is only for parametric surfaces
  - Sew – add vertices to a set of surfaces to make them congruent
  - Add Vertex – add a vertex to a surface; the same as Edge Match
  - Remove Vertex – delete a surface vertex; this deletes a surface edge

Copyright© 2012 MSC.Software Corporation

LOOKING FOR CRACKS BETWEEN EDGES

- **First, verify geometry to find surface free edges**
- **Indicates free and non-manifold edges in model**
  - Free edge: no other edge that is coincident
  - Non-manifold edge: three or more coincident edges

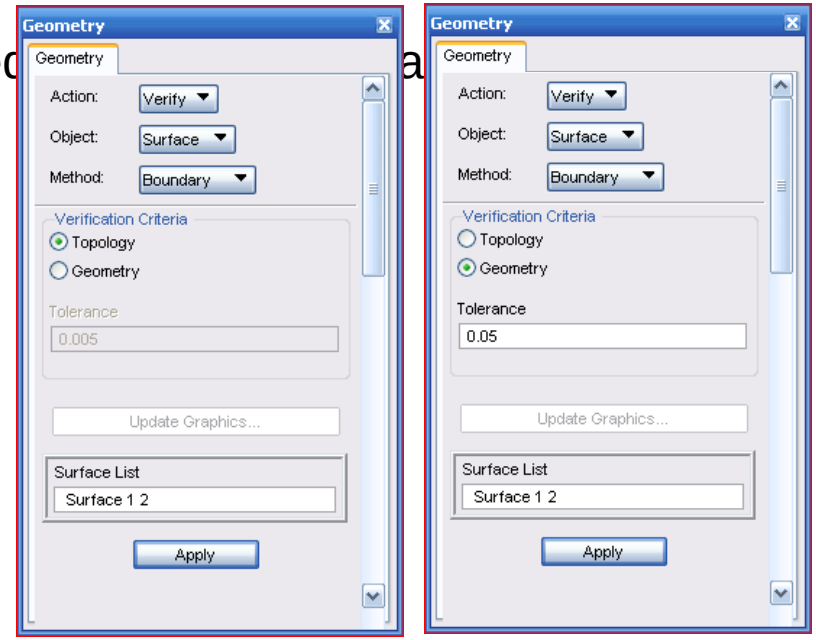




Copyright© 2012 MSC.Software Corporation

LOOKING FOR CRACKS BETWEEN EDGES

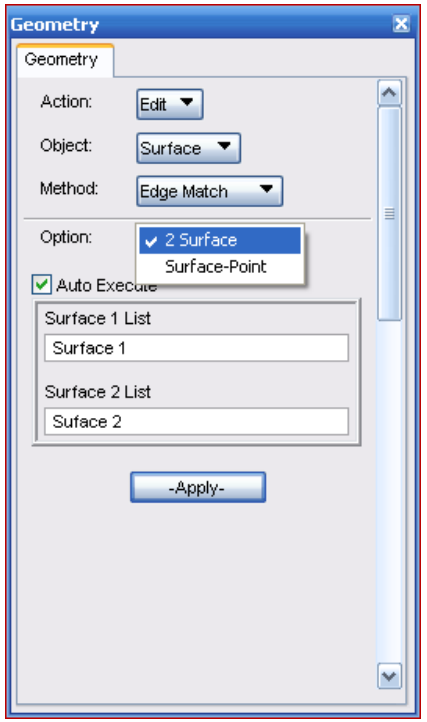
- **There are two ways to use Geometry : Verify / …**
- **Under Verification Criteria:**
  - Topology – looks in the Patran database for established relationships between adjacent surface edges, e.g. edges Surface 1.3 and Surface 2.1, including their vertices, are coincident. No calculations are done for Topology.
  - Geometry – space between adjacent ed a the tolerance;
    if the space is < tolerance
    there are no free edges
    at that tolerance
    - Reducing the tolerance until there are free surface edges, will provide an estimate of the numerical extent of the congruency.



Copyright© 2012 MSC.Software Corporation

MAKING SURFACES CONGRUENT- EDGE MATCH

- **Edge Match**
  - The 2 Surface option recreates the second surface of a specified pair, with congruent (coincident) vertices, but has a gap between adjacent edges.
  - The Surface-Point option recreates a specified surface as a trimmed surface
    - An additional cursor defined vertex point is created
    - Used to make surfaces congruent



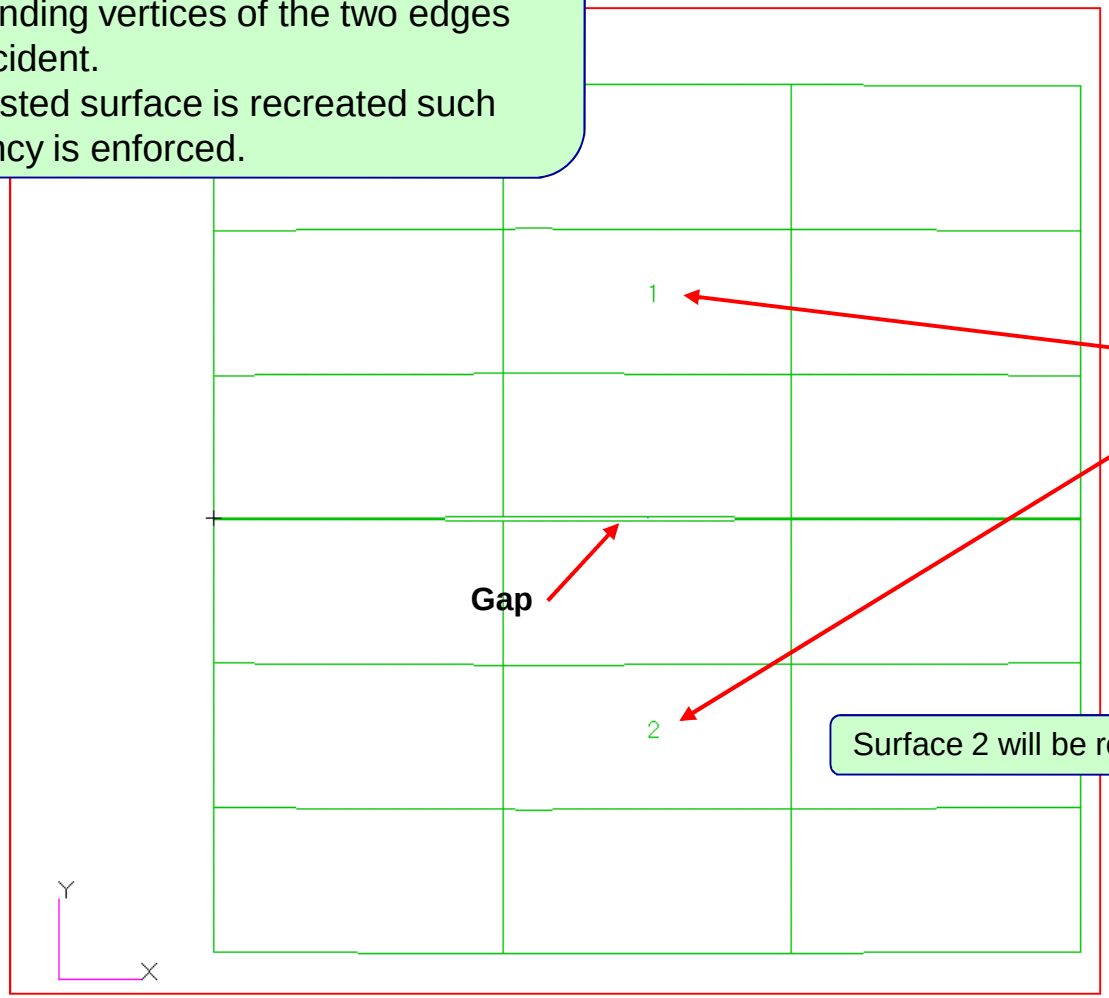
Copyright© 2012 MSC.Software Corporation

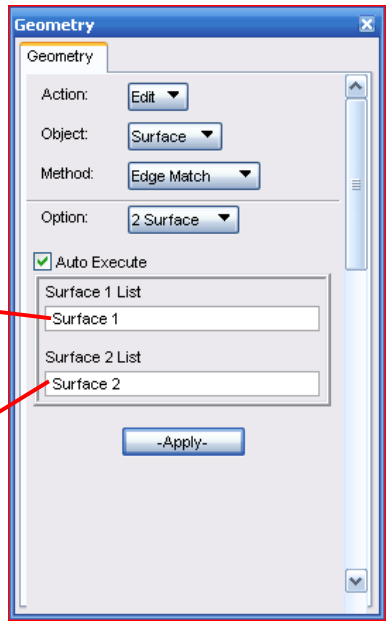
EDGE MATCH

Edge Match: 2 Surface option

Two surfaces are selected

- The corresponding vertices of the two edges must be coincident.
- The second listed surface is recreated such that congruency is enforced.





Surface 2 will be recreated to match Surface 1

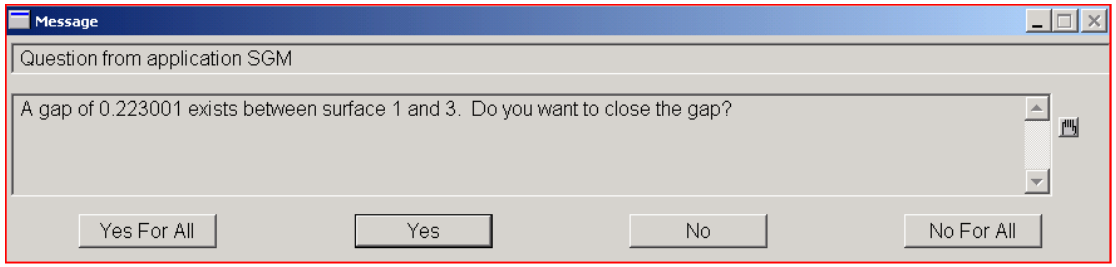
Copyright© 2012 MSC.Software Corporation

EDGE MATCH

- **Edge Match: 2 Surface Option**
  - Gaps between edges must be less than 10 times the Global Model Tolerance or Patran does not close the gap, and the following message is issued:

$# Unable to close gap between edges Surface 1.4 and Surface 2.4 since the gap of 0.052 is greater than the allowable gap closure tolerance of 0.05.

  - In this case, the Global Tolerance is 0.005, set by using an approximate maximum model dimension of 10.0.
  - If the gap is below the threshold, a confirmation of the gap closure is requested.



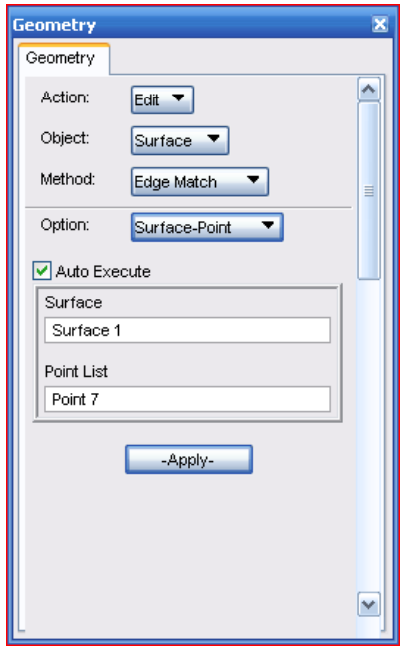
  - Gaps less than the Global Model Tolerance will not issue a message.
  - Note that some surfaces constructed in Patran, such as using the 2 curve method, cannot be edited in this manner.

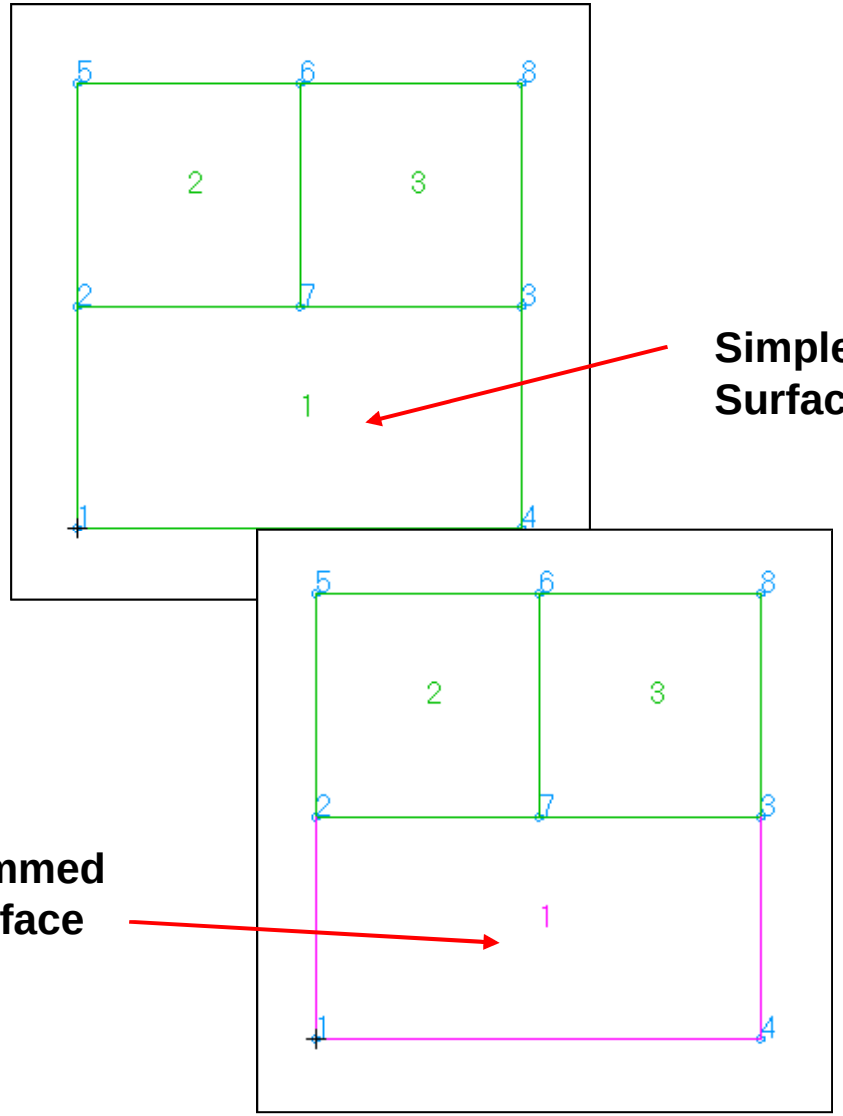
Copyright© 2012 MSC.Software Corporation

EDGE MATCH

Edge Match: Surface-Point option

- A surface is selected for which a vertex is to be added, e.g. Surface 1
- The location of the vertex to be added is cursor selected, Point 7 in this case.
- A vertex is added to the surface, making it trimmed





Copyright© 2012 MSC.Software Corporation

CHANGE FROM TRIMMED TO PARAMETRIC SURFACE

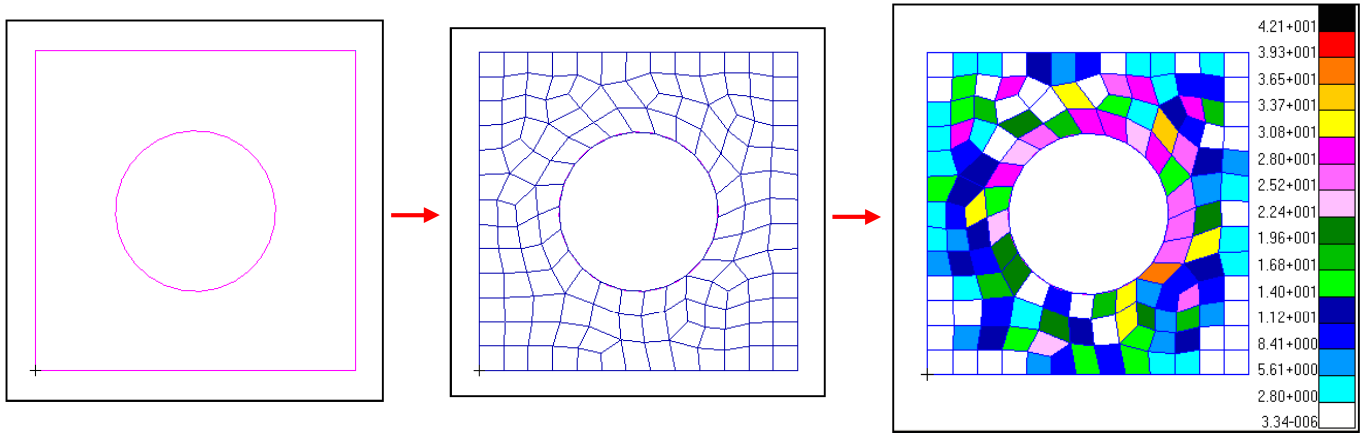
- **Remove a vertex from a trimmed surface to create a parametric surface**
- **Advantage is that can IsoMesh the parametric surface**
- **Disadvantage is may get poorer quality 2D elements because of the topology of the parametric surface**

Copyright© 2012 MSC.Software Corporation

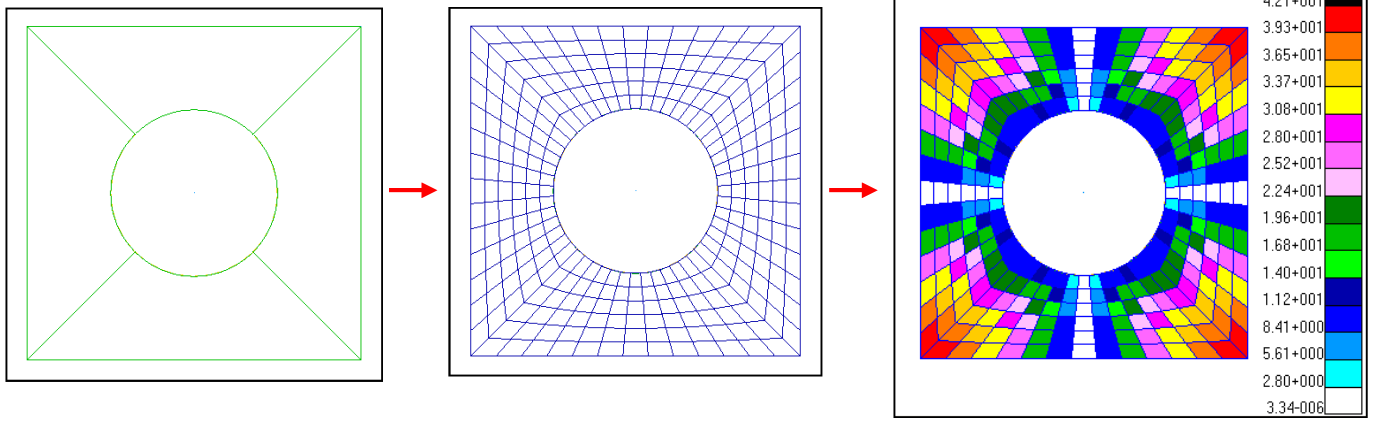
CHANGE FROM TRIMMED TO PARAMETRIC SURFACE

- **Typical surface with hole and Paver mesh vs. surface broken and IsoMesh**

Square trimmed surface with hole, Paver mesh, verify



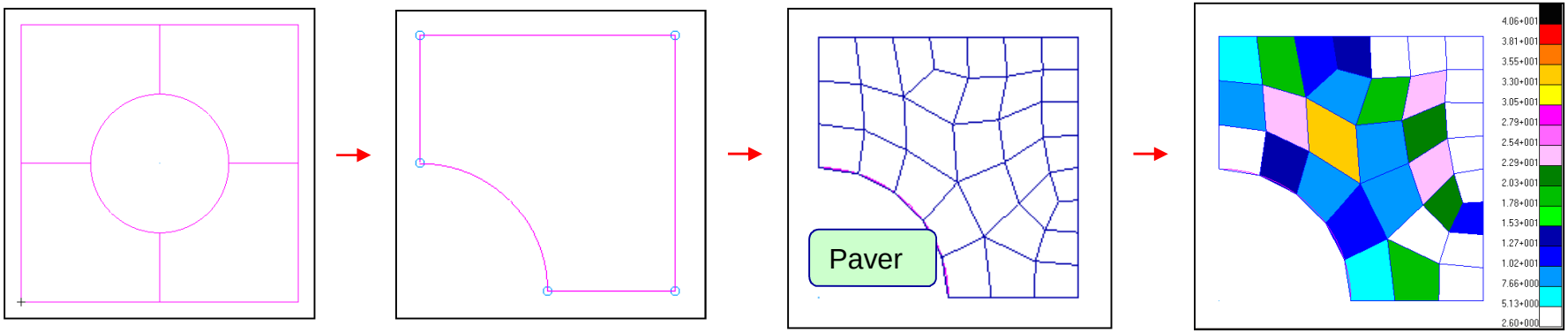
Set of parametric surfaces with hole, IsoMesh mesh, verify

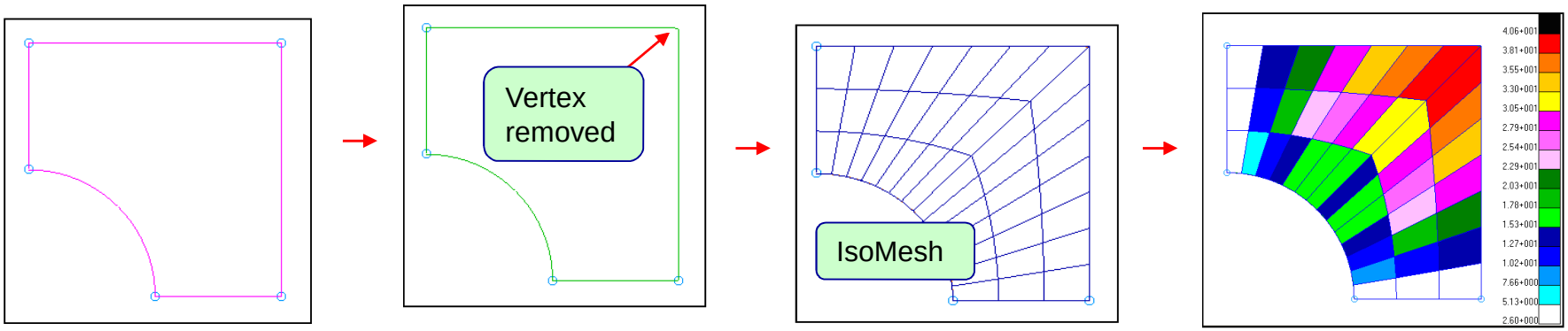


Copyright© 2012 MSC.Software Corporation

CHANGE FROM TRIMMED TO PARAMETRIC SURFACE

- **Another way of creating a parametric surface**
  - Break differently
  - Remove a vertex

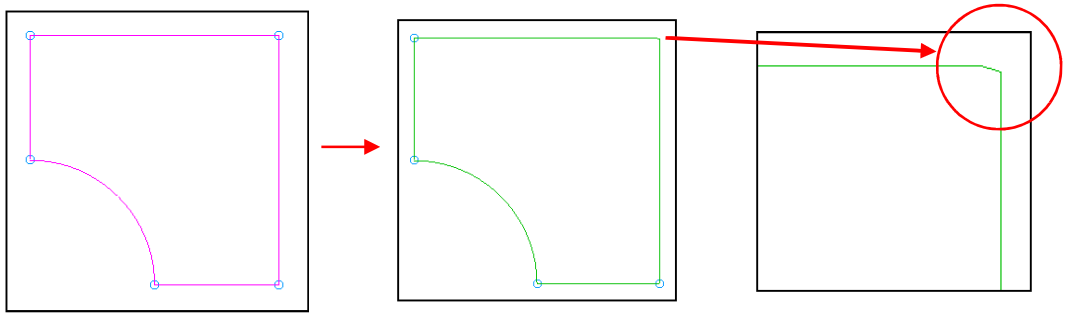




Copyright© 2012 MSC.Software Corporation

CHANGE FROM TRIMMED TO PARAMETRIC SURFACE

- **Other issues that must be dealt with in removing vertices**
  - Will the geometric edges at the locations where the vertices are removed have the required fidelity if the surface created is parametric ?



  - Having congruency with adjacent surface edges
  - Will extruding a parametric surface that was created by removing vertices result in parametric solids that are congruent with adjacent solids ?

Copyright© 2012 MSC.Software Corporation

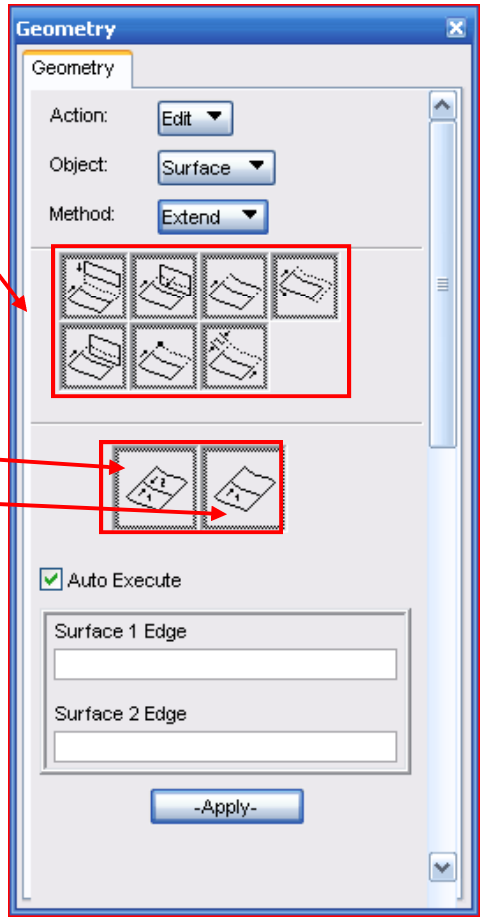
SURFACE EXTENSION

- **This set of tools offers multiple options to modify a surface by lengthening it.**
- **Options for extending a surface include**
  - To another surface edge
  - To an intersecting point, curve, surface or plane
  - To a fixed distance
  - By a percentage

Copyright© 2012 MSC.Software Corporation

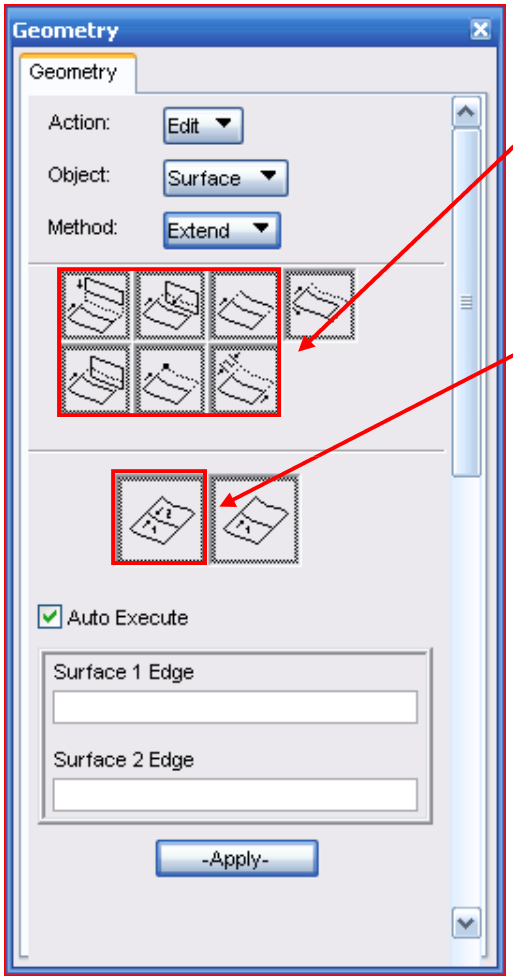
SURFACE EXTENSION

- **The seven different options for extending a surface are chosen through these icons**
- **For most of these picks, there is a further option between:**
  - Edit All Associated Surfaces
  - Edit Only Selected Surfaces

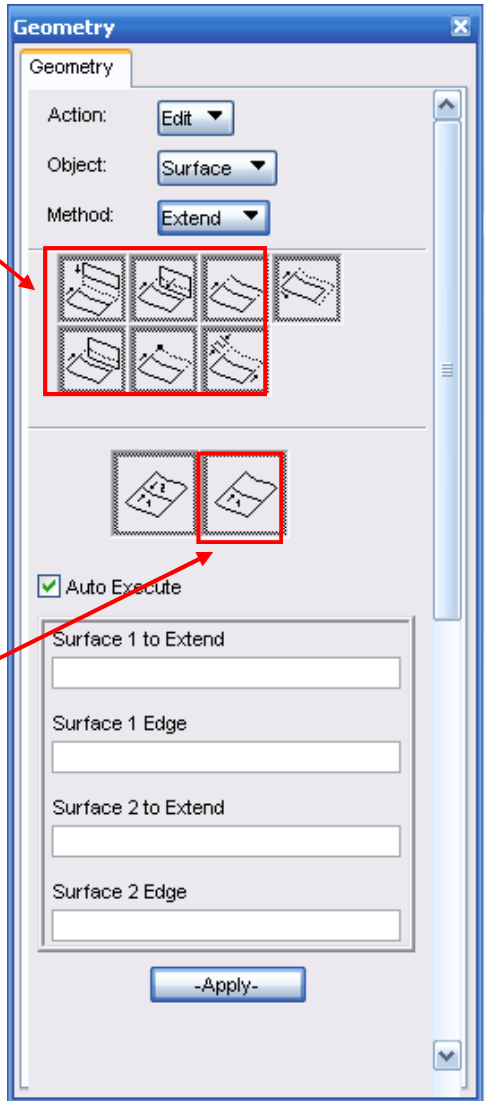


Copyright© 2012 MSC.Software Corporation

SURFACE EXTENSION



- **Six of these options have automatic or manual surface edge selection**
- **The Edit All Associated Surfaces form is more simplified when there is less ambiguity in the operation**
- **Edit Only Selected Surfaces form better specifies which surfaces and edges will be extended**

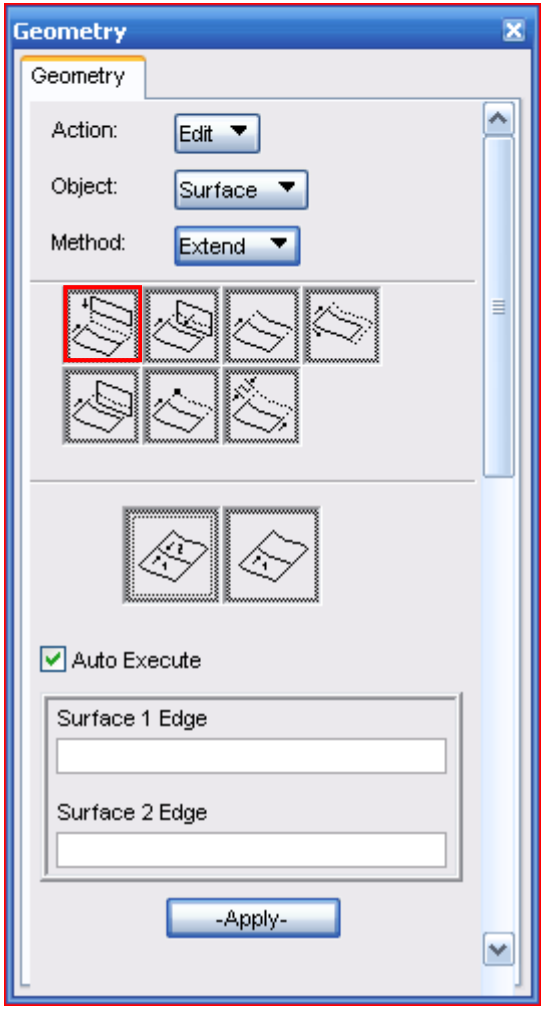


Copyright© 2012 MSC.Software Corporation

SURFACE EXTENSION 2 SURFACE

- **This option extends the two surfaces to a common intersection.**

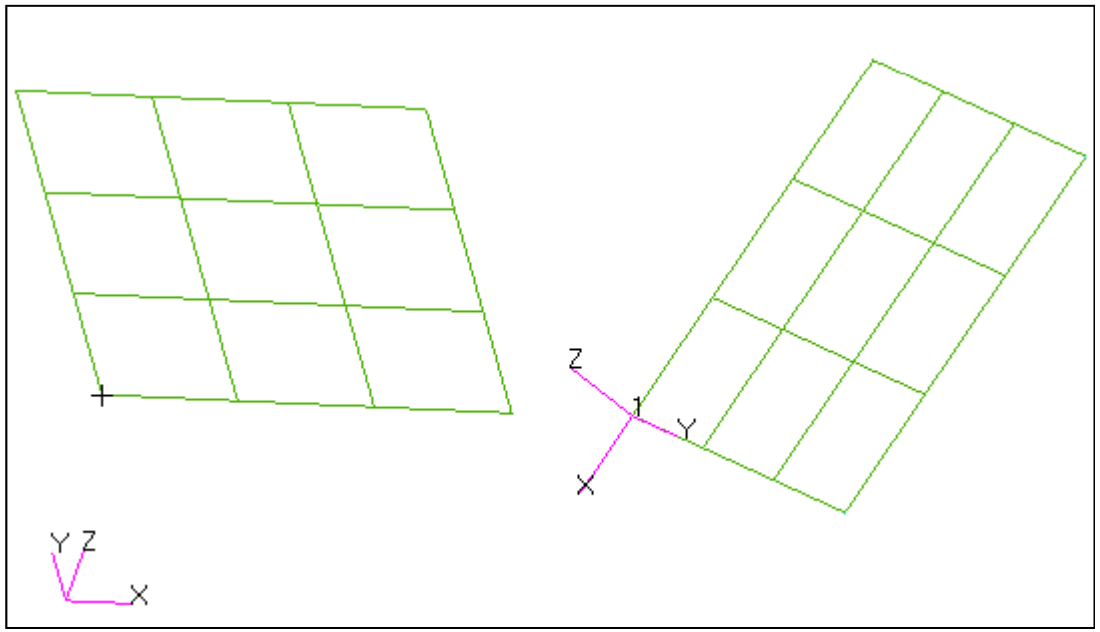
- **In this operation, both surfaces can be modified.**



Copyright© 2012 MSC.Software Corporation

EXAMPLE: SURFACE EXTENSION, 2 SURFACE

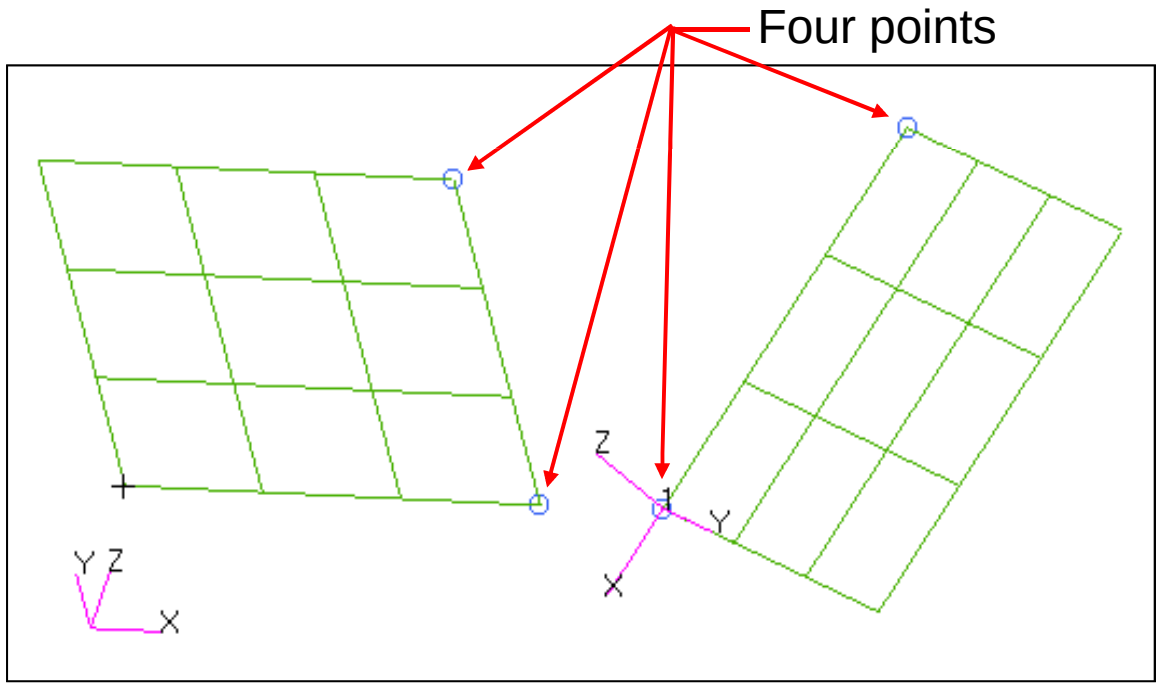
- **In this example, there are two surfaces that need to be extended to intercept one another.**
- **Both surfaces are 1 square unit and out of plane from one another.**



Copyright© 2012 MSC.Software Corporation

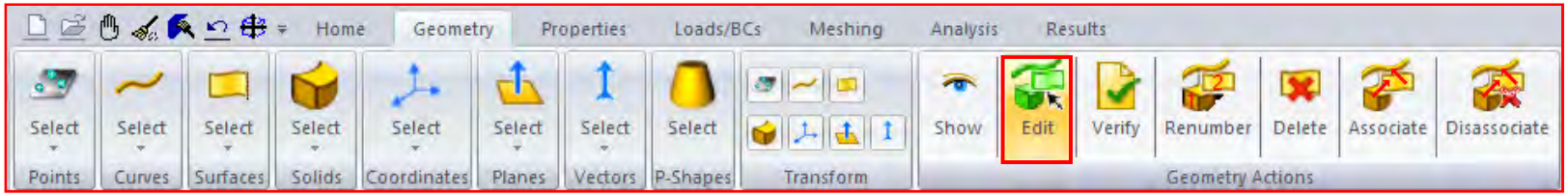
EXAMPLE: SURFACE EXTENSION, 2 SURFACE

- **To better view how the surfaces are changed, four new points are created as shown.**
- **These points are not associated with the surfaces and so are independent of the modification to follow.**

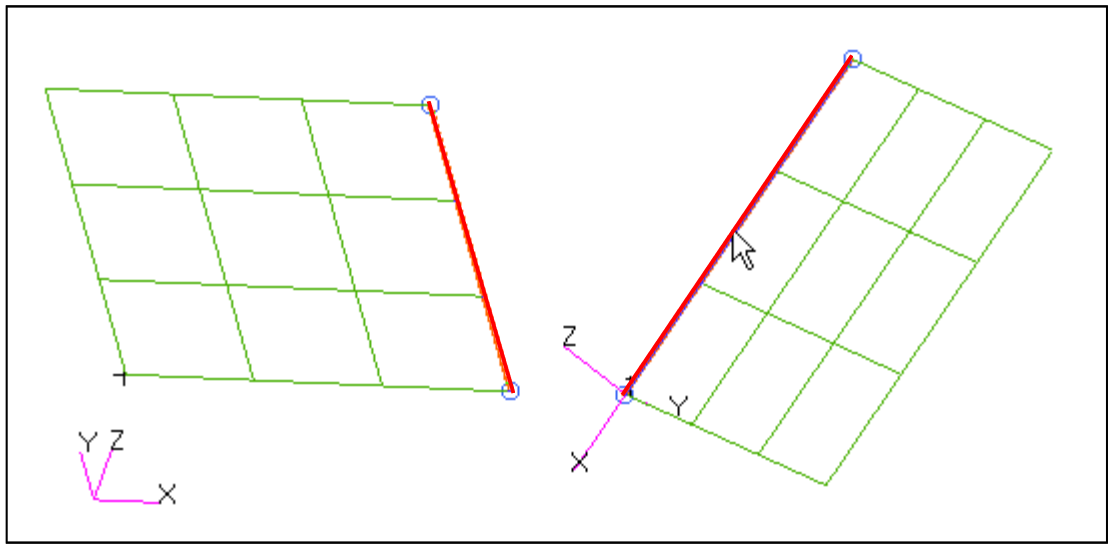


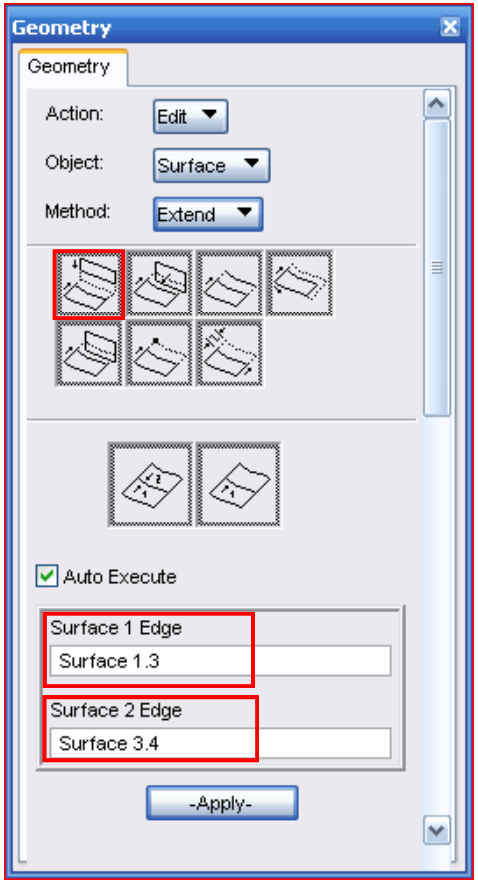
Copyright© 2012 MSC.Software Corporation

EXAMPLE: SURFACE EXTENSION, 2 SURFACE



Under the *Geometry* tab click on **Edit** from the *Geometry Actions* group. Select **Surface/Extend** as the *Object/Method*.
Under *Surface 1 Edge*, select Surface 1.3
Under *Surface 2 Edge*, select Surface 3.4
Click **Apply**

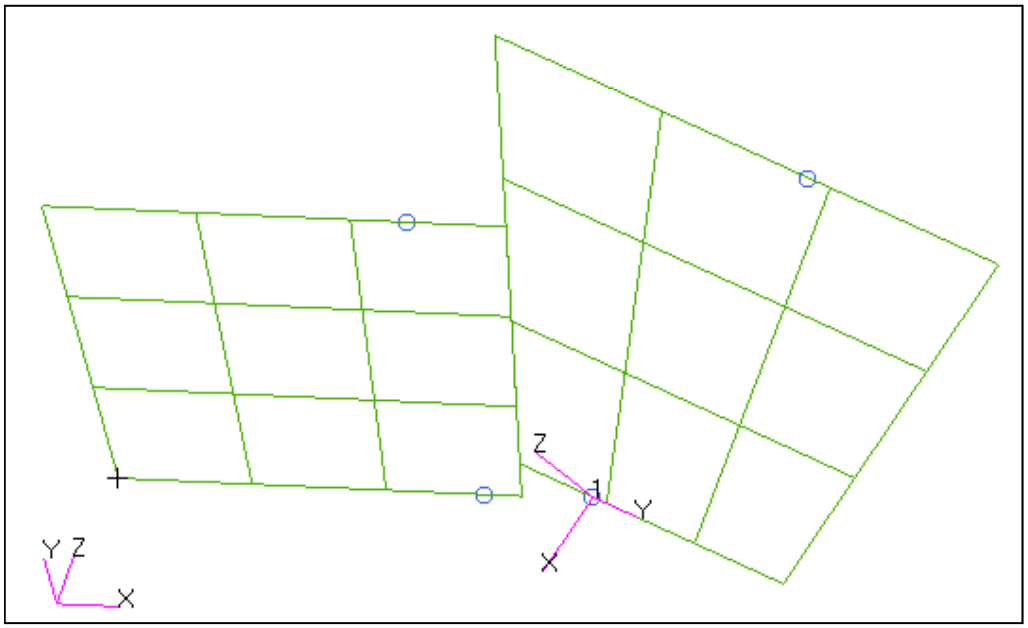




Copyright© 2012 MSC.Software Corporation

EXAMPLE: SURFACE EXTENSION, 2 SURFACE

- **By examining the reference points, it can be seen that both surfaces have been modified. Also, the two selected edges were re-created with new angles at the intersection, but the original adjacent edges did not change angle.**

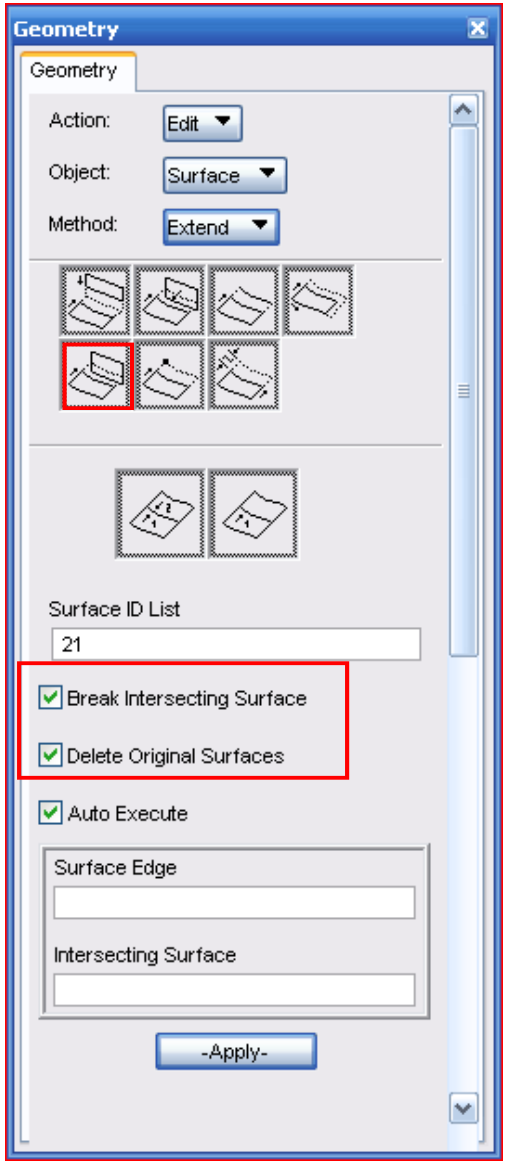


Copyright© 2012 MSC.Software Corporation

SURFACE EXTENSION TO A SURFACE

- **This option will extend a surface edge to another surface.**

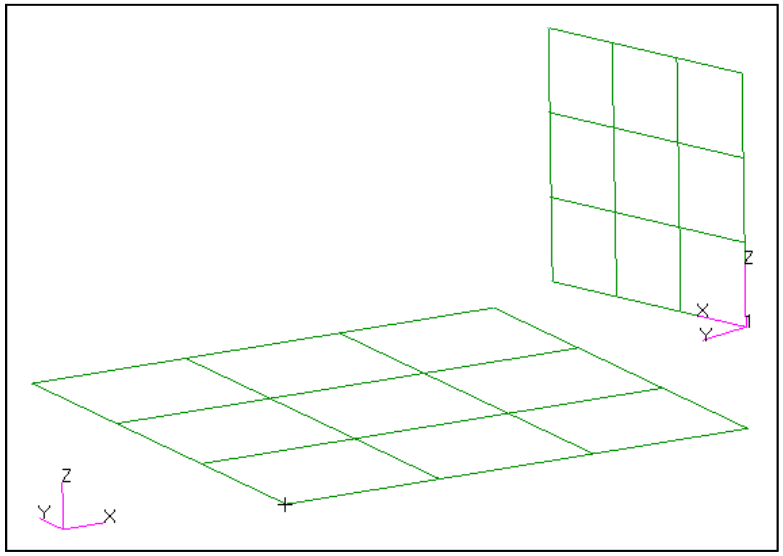
- **The additional options determine if the intersecting surface is to be broken and if the original surfaces are to be deleted.**



Copyright© 2012 MSC.Software Corporation

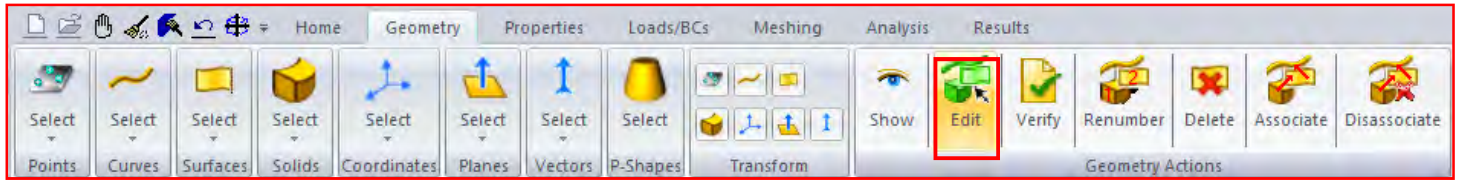
EXAMPLE: SURFACE EXTENSION, TO A SURFACE

- **In this example, the smaller surface is extended to meet the lower surface**
  - It is also desired to break the lower surface where the smaller surface intercepts it.
  - The alternate coordinate system on the smaller surface is a good reference to see the extension.



Copyright© 2012 MSC.Software Corporation

EXAMPLE: SURFACE EXTENSION, TO A SURFACE



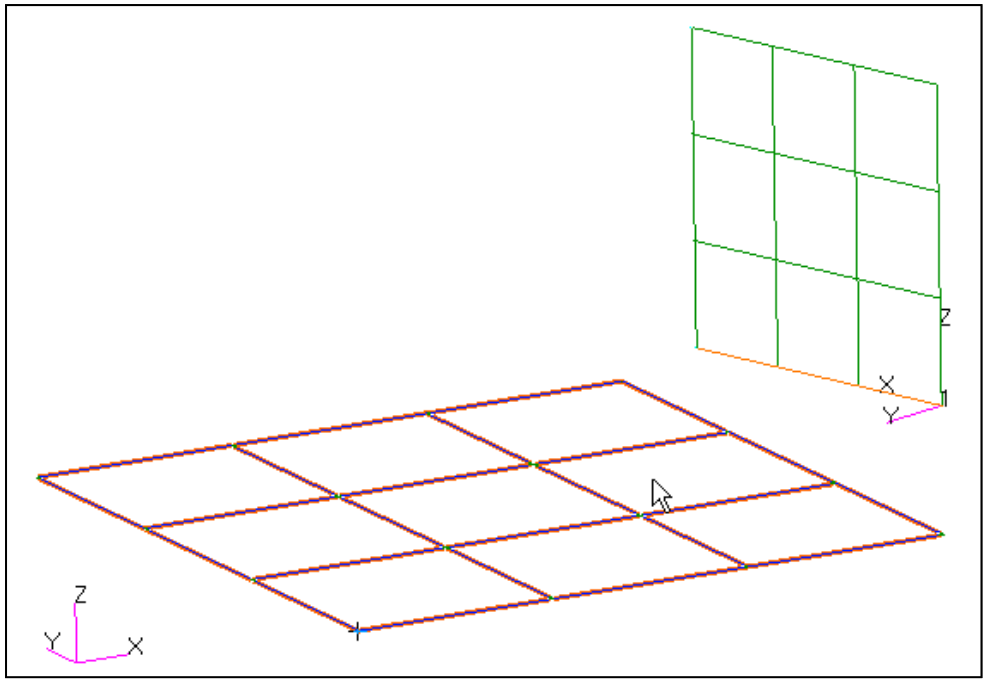
Under the *Geometry* tab click on **Edit** from the *Geometry Actions* group. Select **Surface/Extend** as the *Object/Method*.

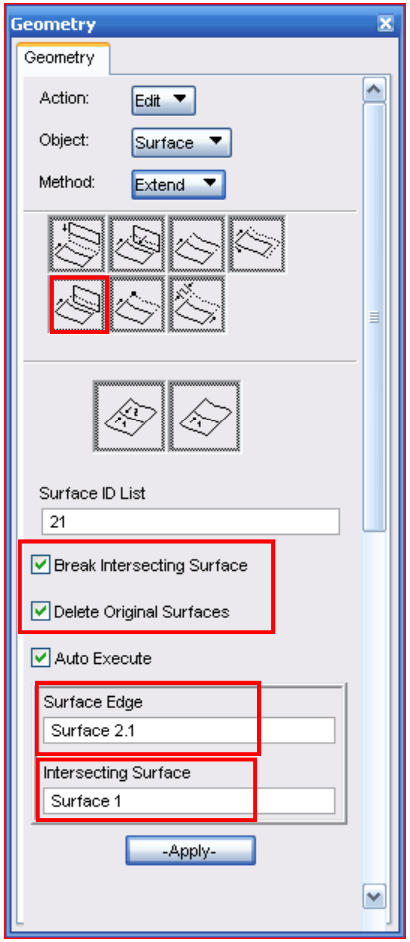
Select the options to break and delete the original surface.

For *Surface Edge*, select the smaller surface's edge (Surface 2.1).

For the *Intersecting Surface*, select Surface 1.

Click **Apply**

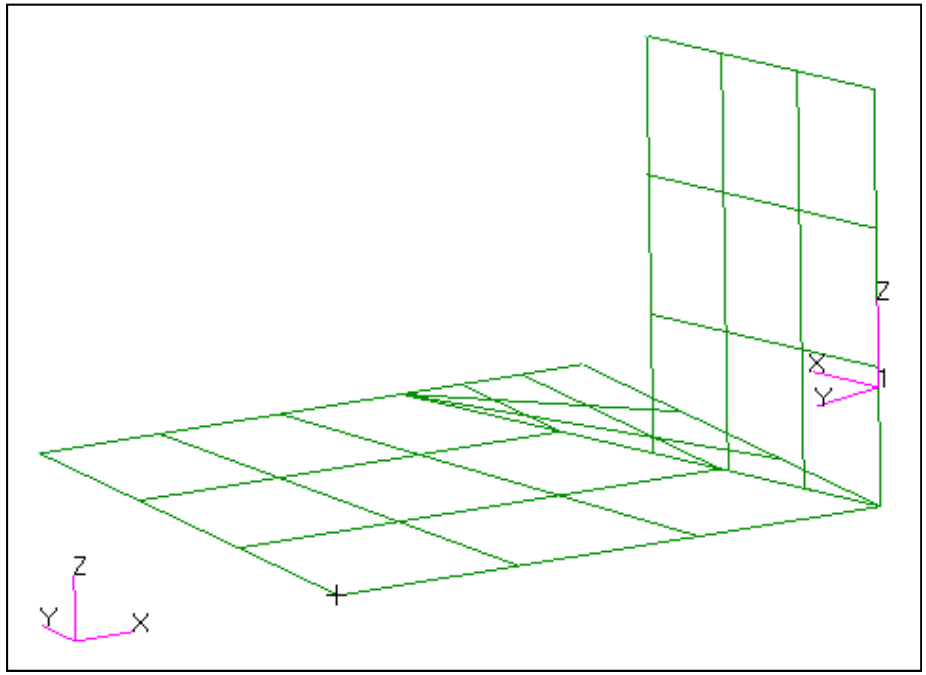




Copyright© 2012 MSC.Software Corporation

EXAMPLE: SURFACE EXTENSION, TO A SURFACE

- **The smaller surface has been extended down to the larger surface. Coordinate Frame 1 helps to illustrate this.**
- **The larger surface has been broken into two separate surfaces as specified by the options selected.**

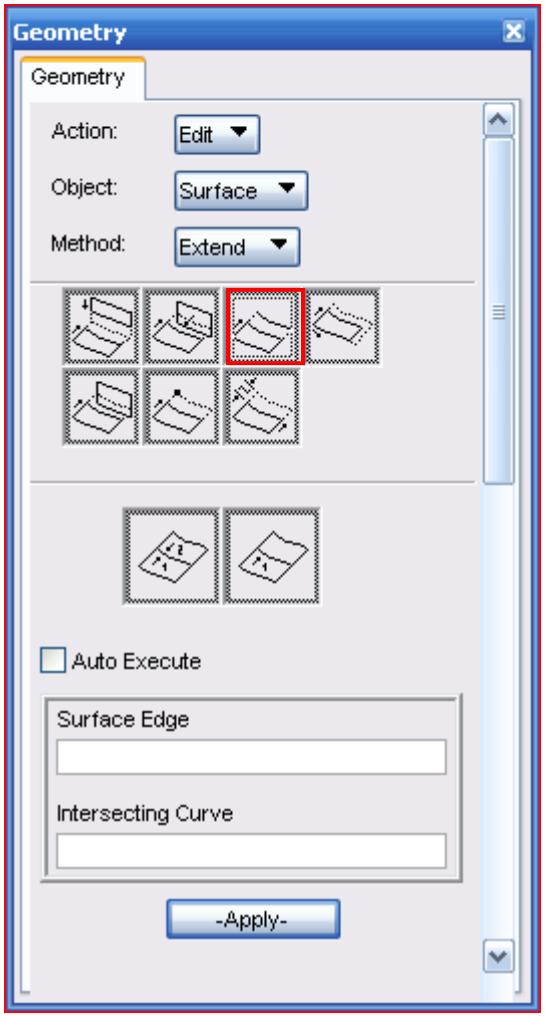


Copyright© 2012 MSC.Software Corporation

SURFACE EXTENSION TO A CURVE

- **This option will extend a surface edge, in the plane of the surface, to a curve.**

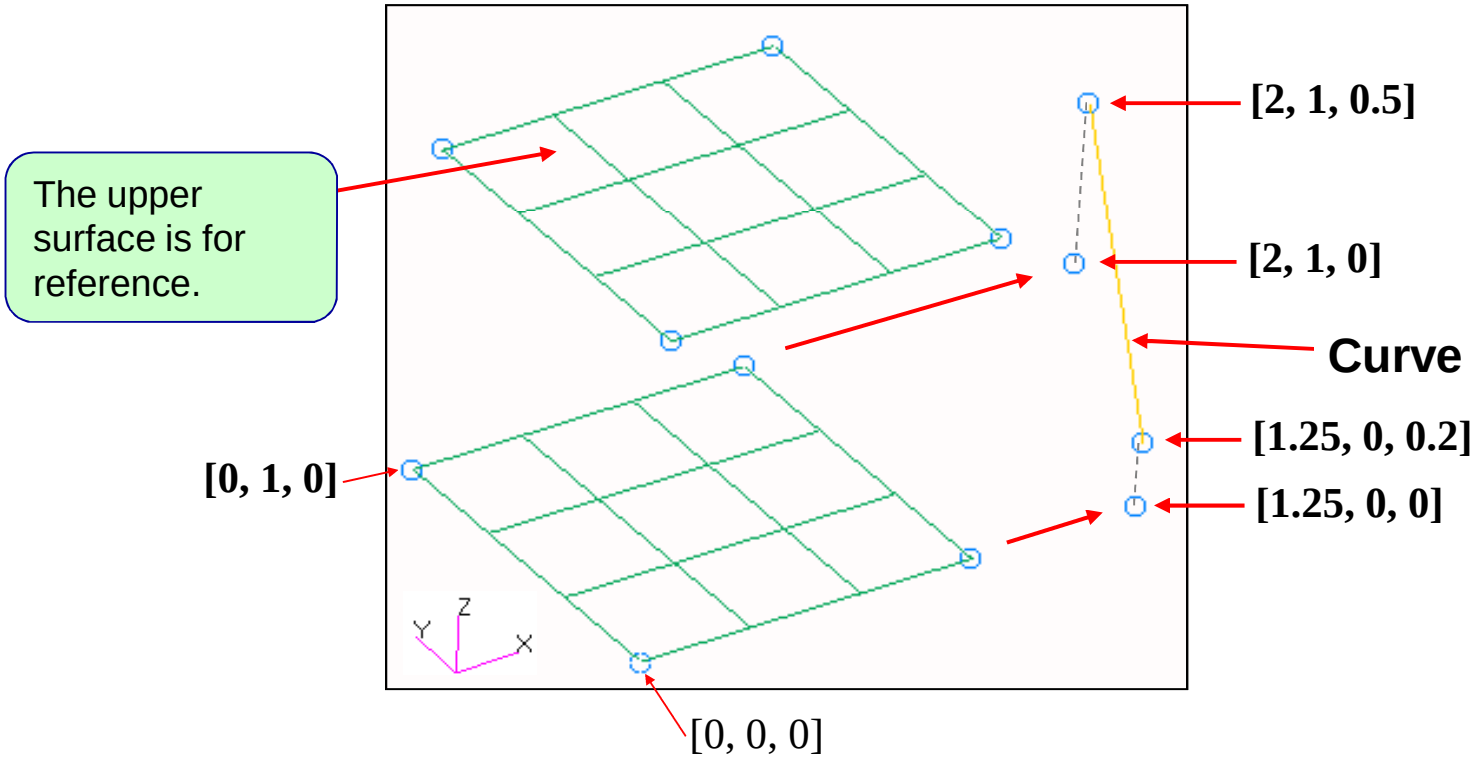
- **If the curve is not in the plane of the surface, the surface edge is extended to the projection of the curve to the plane of the surface.**



Copyright© 2012 MSC.Software Corporation

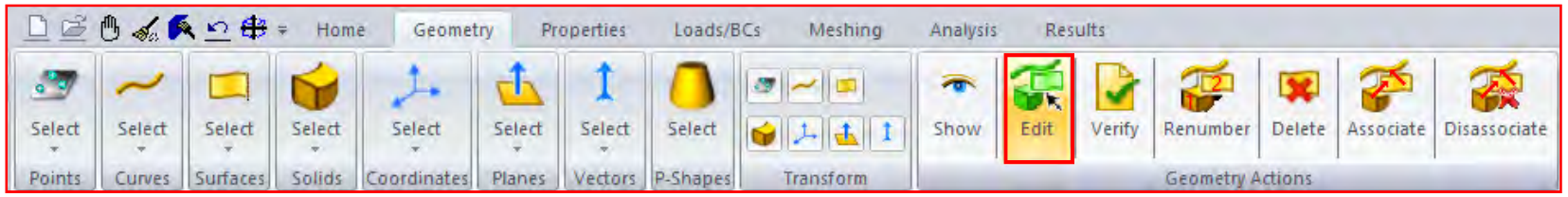
EXAMPLE: SURFACE EXTENSION TO A CURVE

- **There are two surfaces, one above the other, and each is one unit square. The goal is to extend the lower surface to the curve. The curve is above the lower surface.**



Copyright© 2012 MSC.Software Corporation

EXAMPLE: SURFACE EXTENSION TO A CURVE

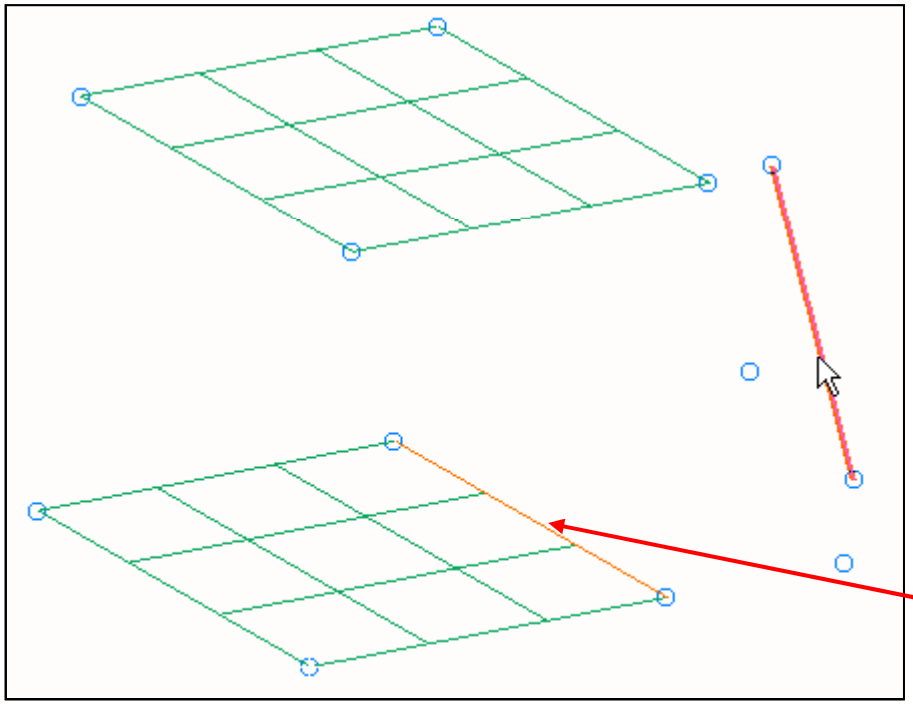


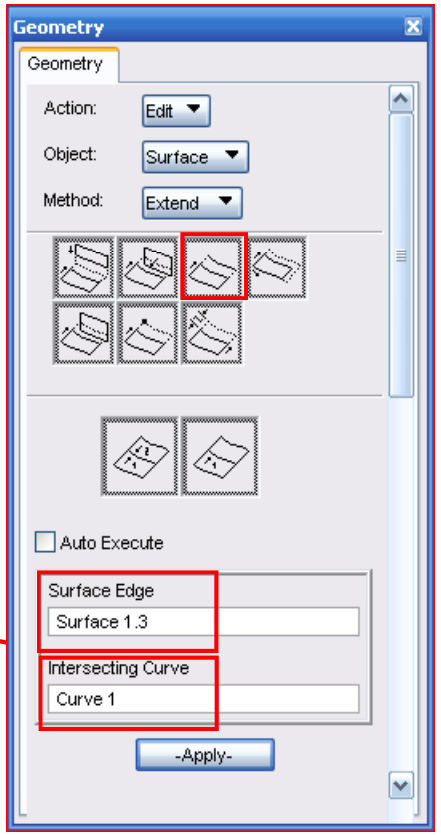
Under the *Geometry* tab click on **Edit** from the *Geometry Actions* group. Select **Surface/Extend** as the *Object/Method*.

Under *Surface Edge*, select Edge Surface 1.3.

Under *Intersecting Curve*, select Curve 1.

Click **Apply**

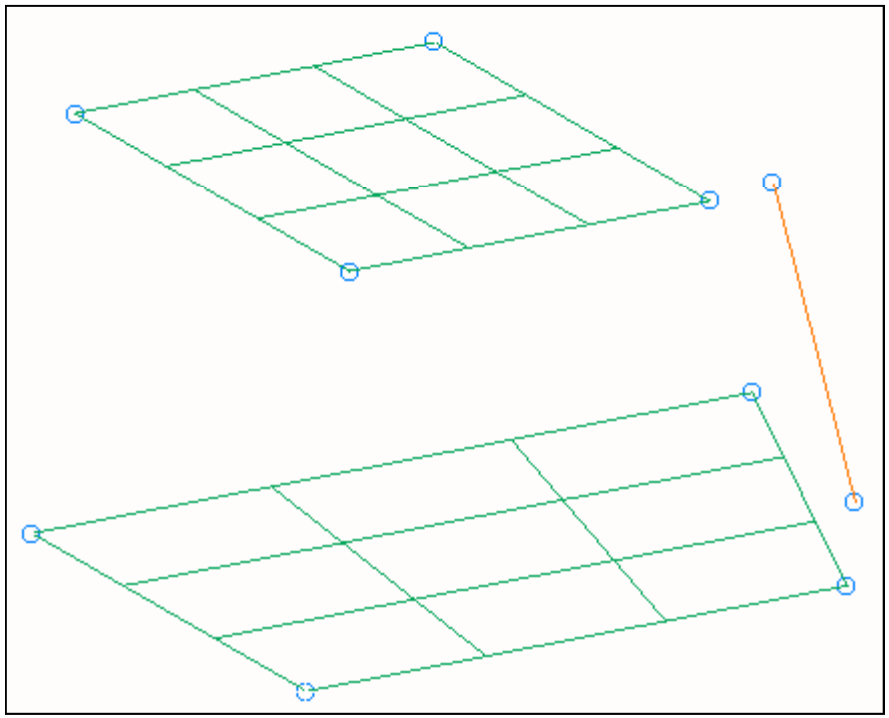




Copyright© 2012 MSC.Software Corporation

EXAMPLE: SURFACE EXTENSION TO A CURVE

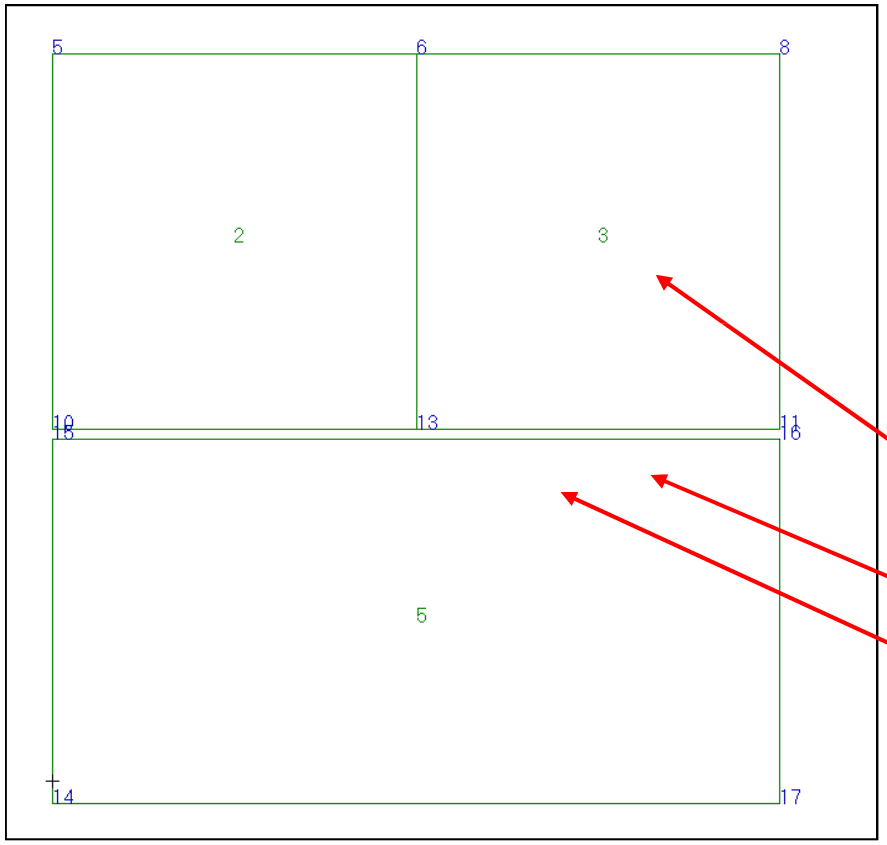
- **The surface has been extended in-plane to the projection of the curve.**

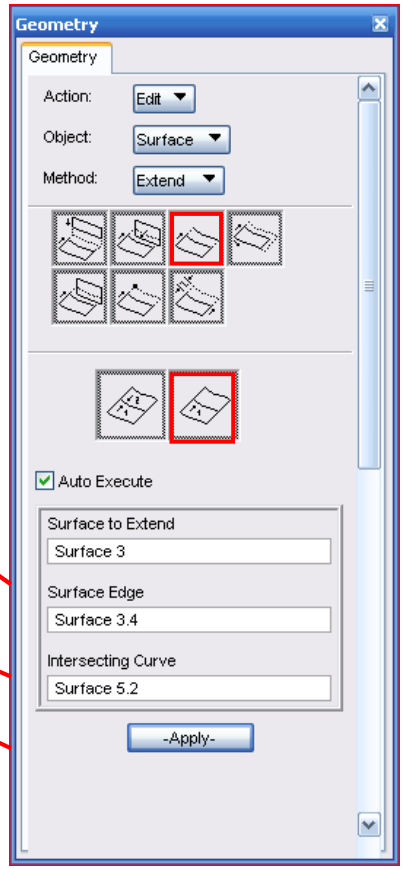


Copyright© 2012 MSC.Software Corporation

EXAMPLE: SIMPLE GEOMETRY EDITING

- **Close gaps using Surface Extend To a Curve**
- **Surface 3 is extended to butt against surface 5**

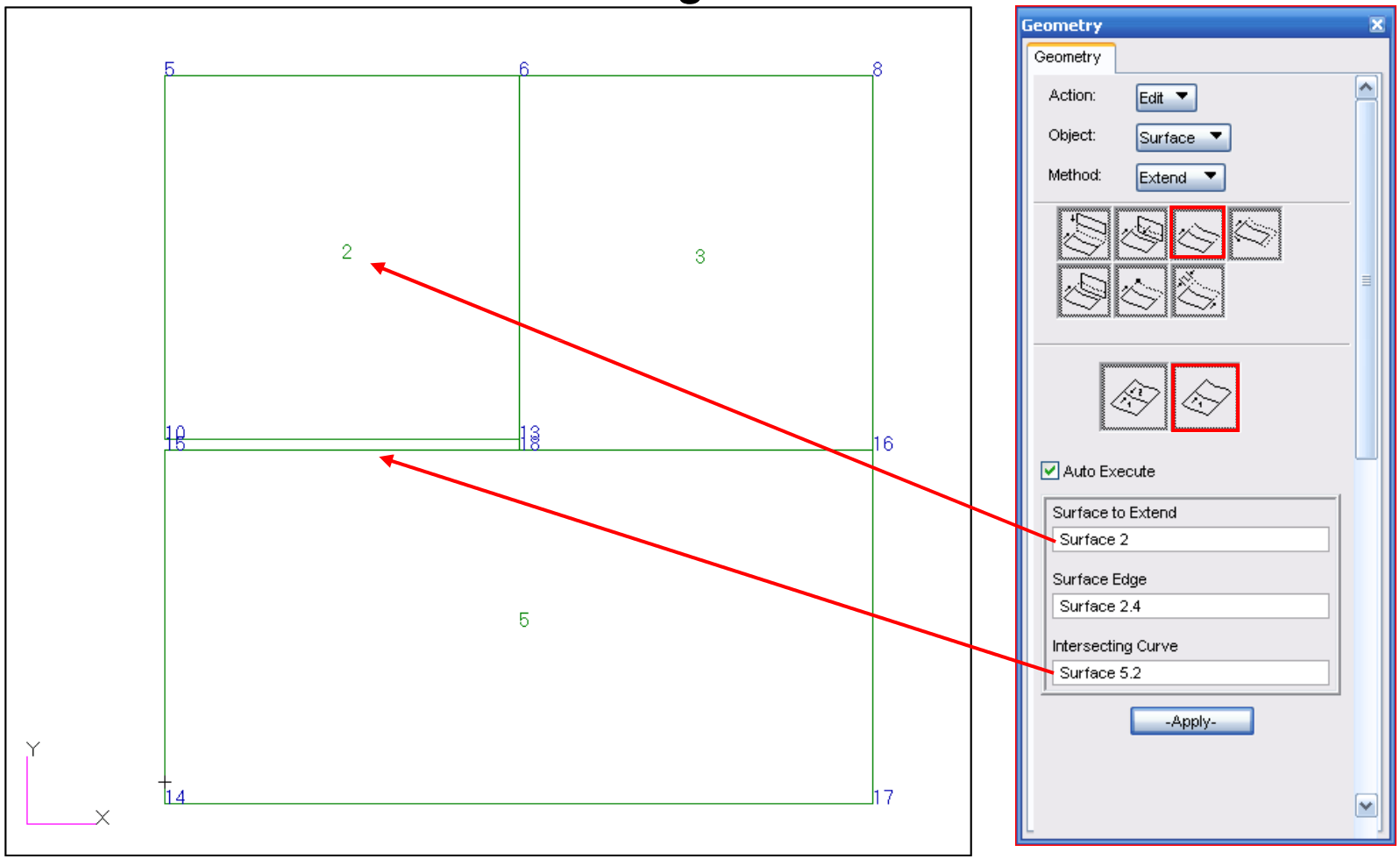




Copyright© 2012 MSC.Software Corporation

EXAMPLE: SIMPLE GEOMETRY EDITING

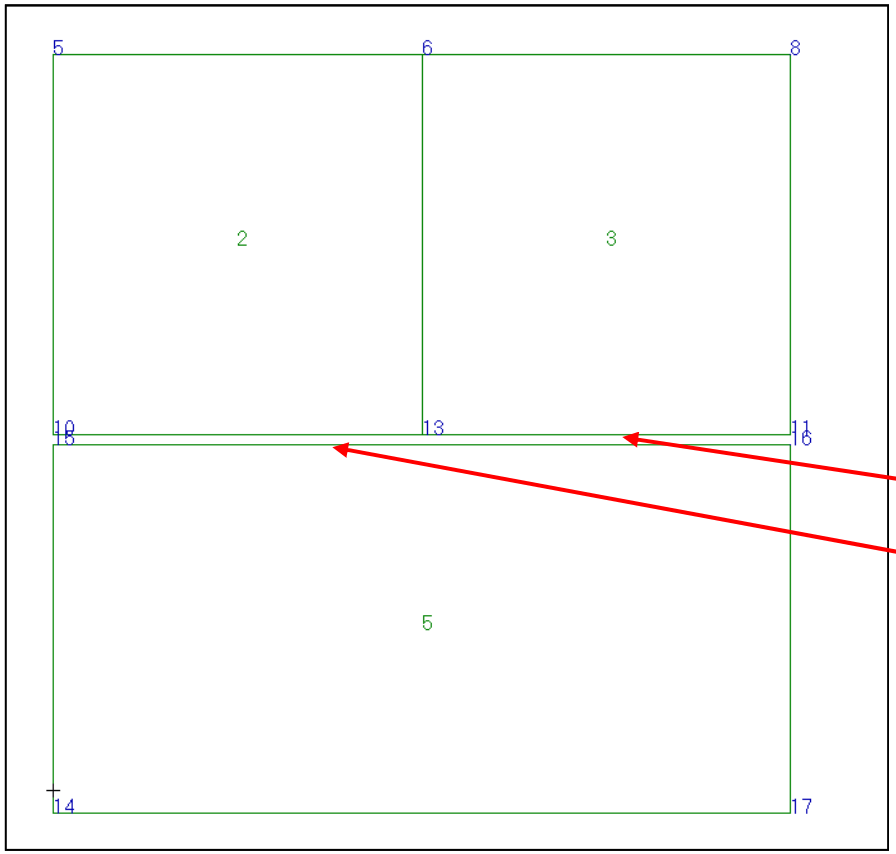
- **Surface 2 is extended to butt against Surface 5**

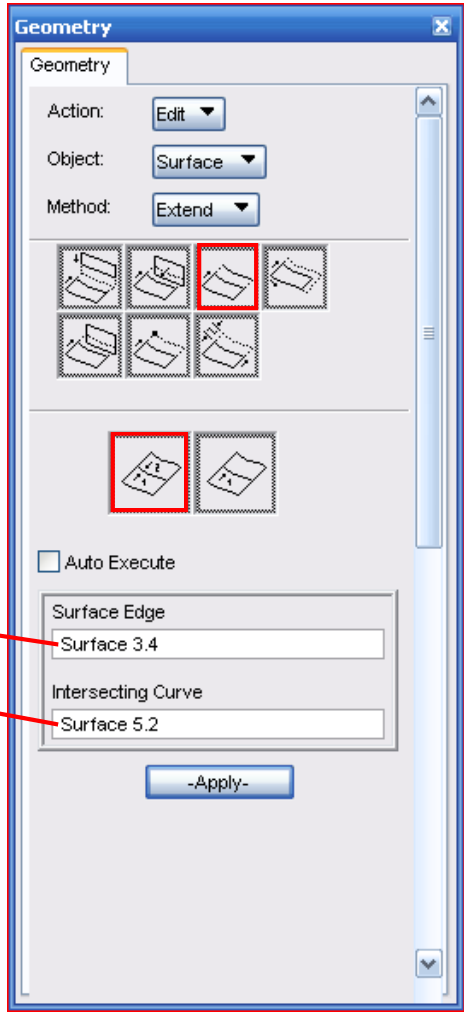


Copyright© 2012 MSC.Software Corporation

EXAMPLE: SIMPLE GEOMETRY EDITING

- An alternative is to use the option to edit all affected surfaces
- The same result is achieved in one operation

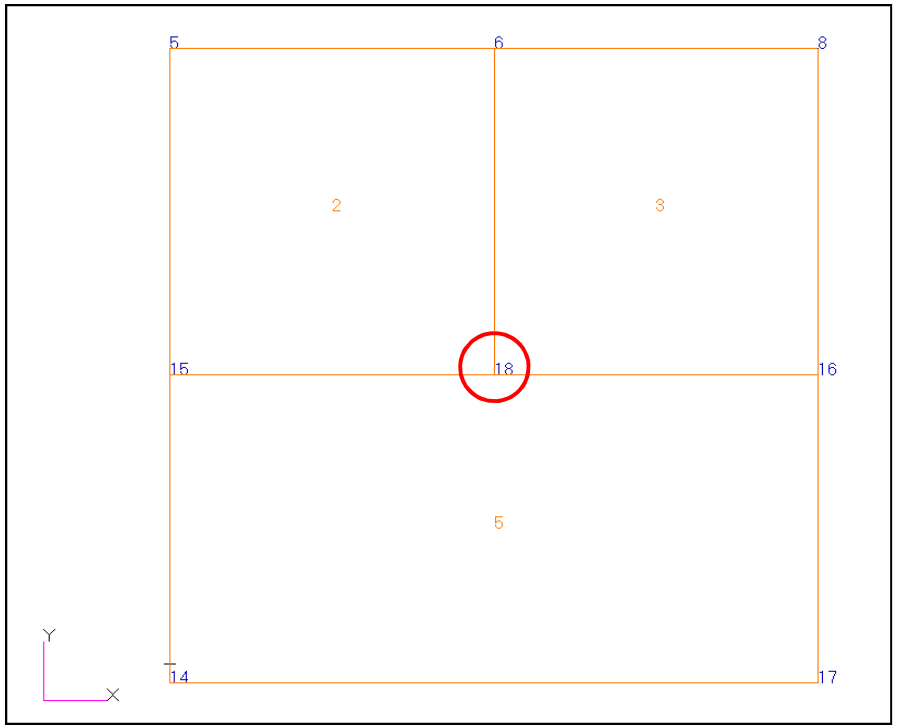


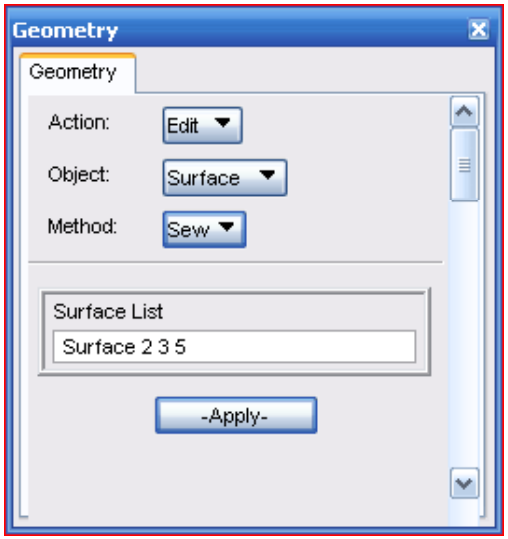


Copyright© 2012 MSC.Software Corporation

EXAMPLE: SIMPLE GEOMETRY EDITING

- **Finally, the use of Edit/Surface/Sew will turn Surface 5 into a trimmed surface by adding a vertex at Point 18 to the surface edge. The surfaces will then be congruent.**





This is similar to Geometry: Edit/Surface/Edge Match

Copyright© 2012 MSC.Software Corporation

EXAMPLE: SIMPLE GEOMETRY EDITING

- **The messages issued by Patran during the “Sew” operation are given as follows:**

  sgm_edit_surface_sew("Surface 2 3 5", …)

  $# T-Edge Surface 5.2 broken at Point 18 into edges Surface 3.4 and Surface 2.4.

  $# No Points Edited.

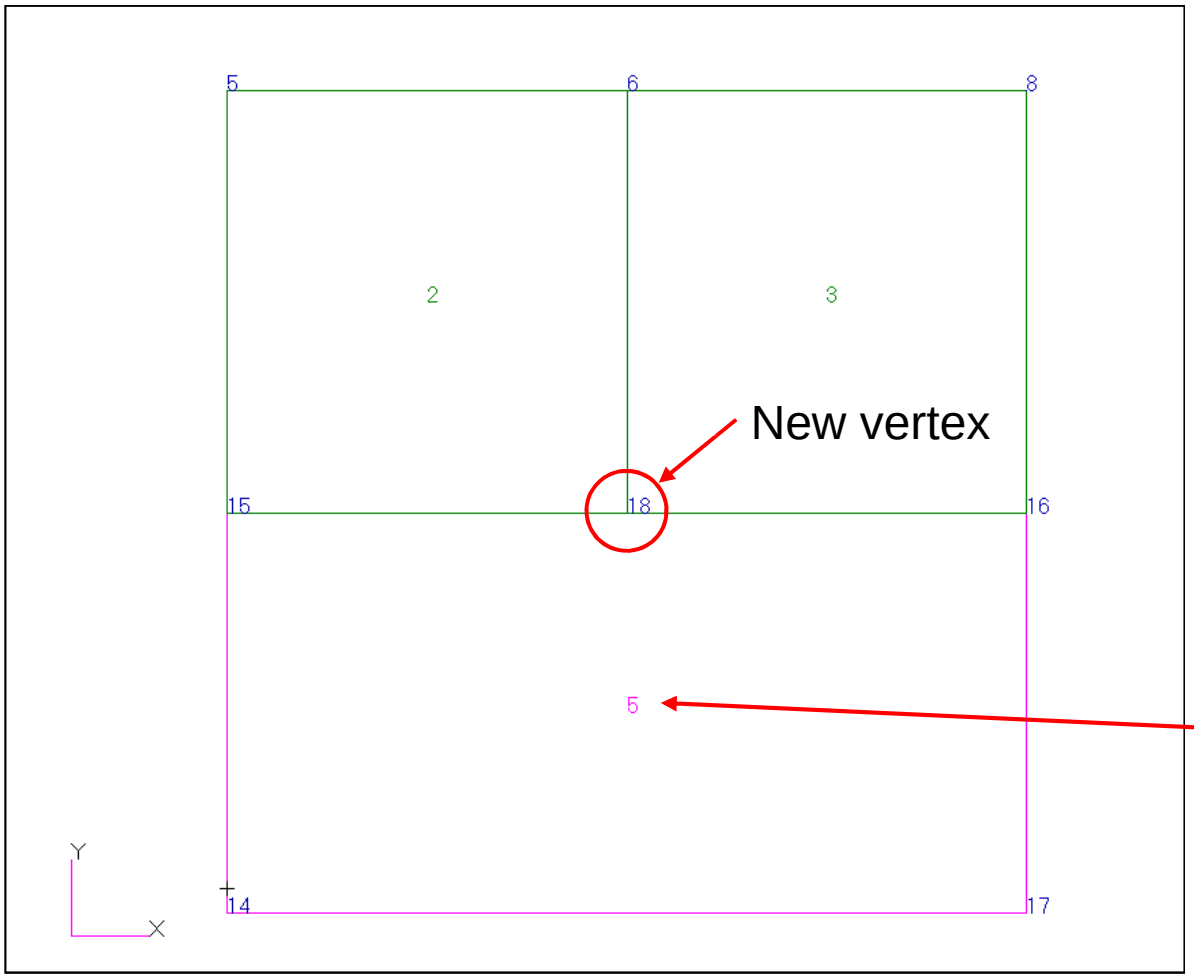
  $# No Points Deleted.

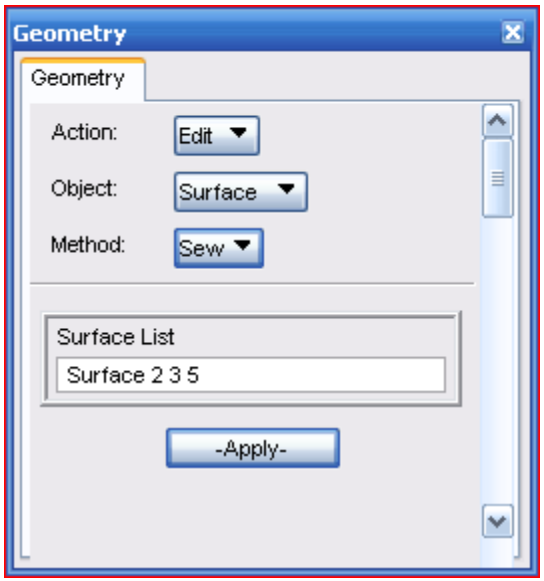
  $# 1 Surface Edited: Surface 5

- **Sew identifies “T” edges and creates a vertex on the long edges**
- **The editing of Surface 5 consists of adding one vertex at the location of Point 18**

Copyright© 2012 MSC.Software Corporation

EXAMPLE: SIMPLE GEOMETRY EDITING



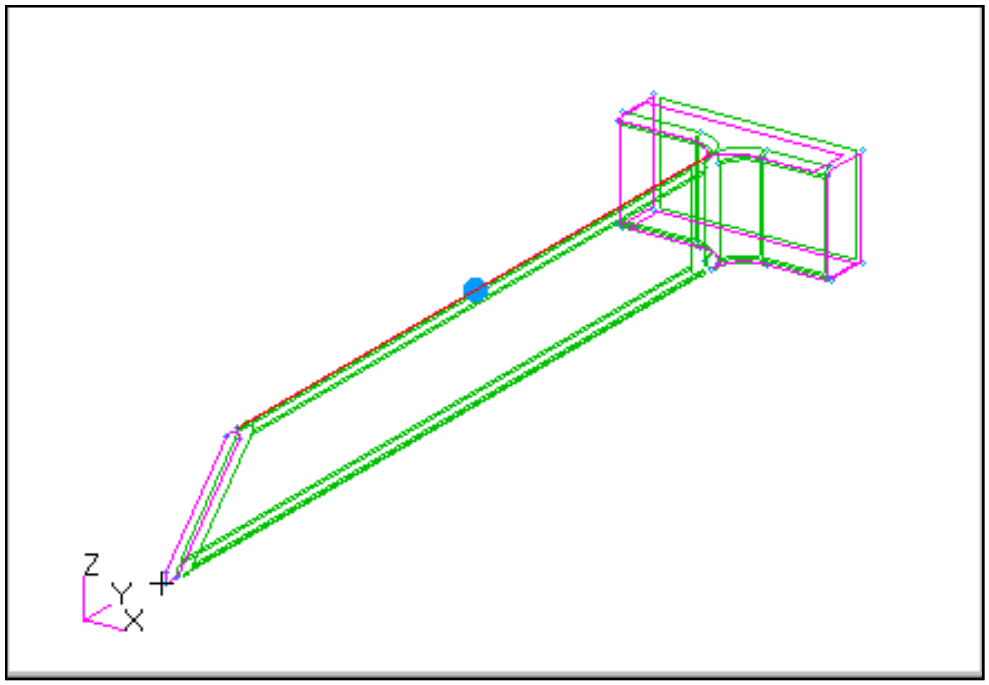


Trimmed surface created

Copyright© 2012 MSC.Software Corporation

EXAMPLE: MORE SIMPLE EDITING

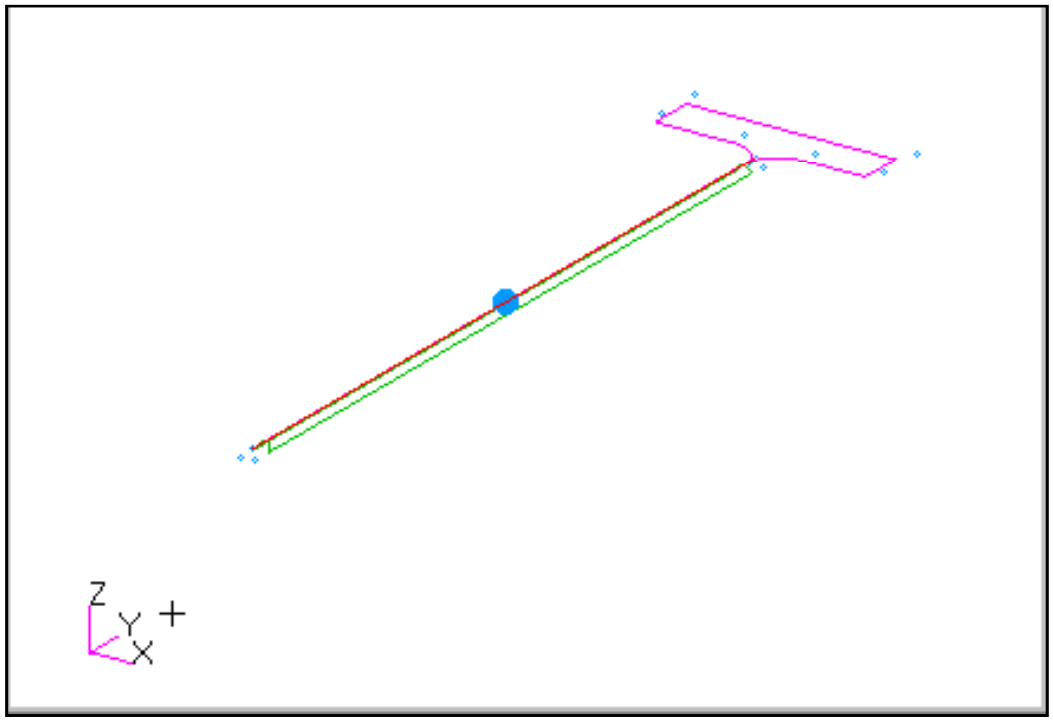
- **Shown below is a set of surfaces. They have been shrunk slightly. The Verify command was executed to establish the state of congruency. The blue dot indicates a congruency problem area.**



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MORE SIMPLE EDITING

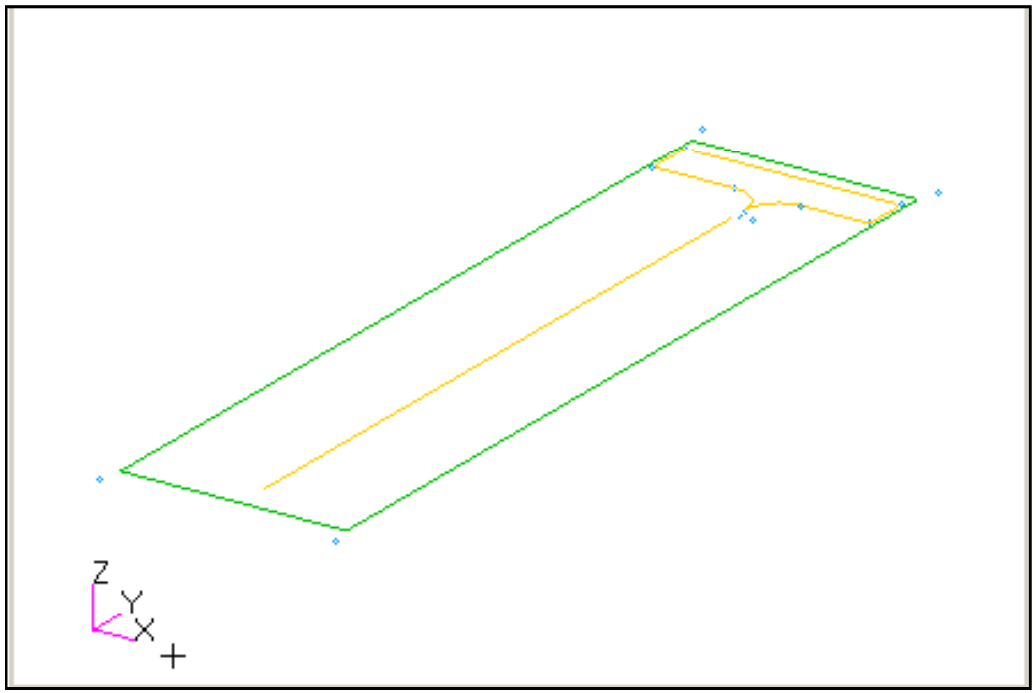
- **All the congruent surfaces have been erased from the viewport, leaving only non-congruent surfaces.**



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MORE SIMPLE EDITING

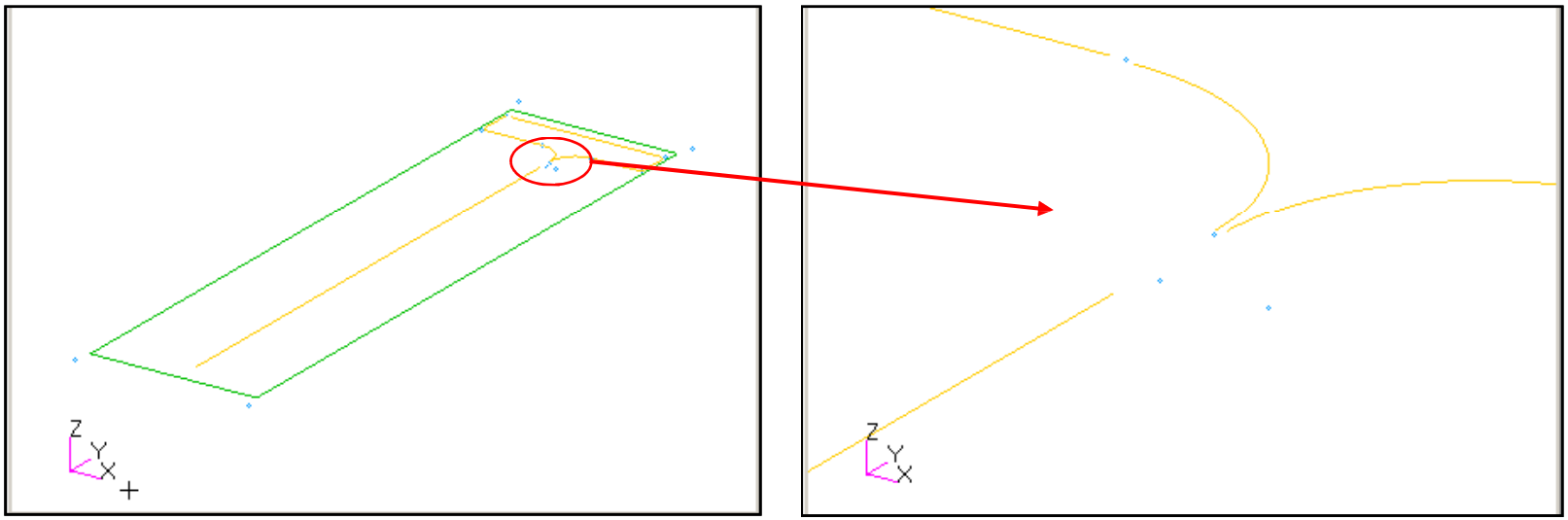
- **Display only the problem trimmed surface, disassembled into its boundary curves (yellow) and parent parametric surface (green).**



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MORE SIMPLE EDITING

- **Zoom into where the curves transition to a single long curve (‘tail’ curve).**

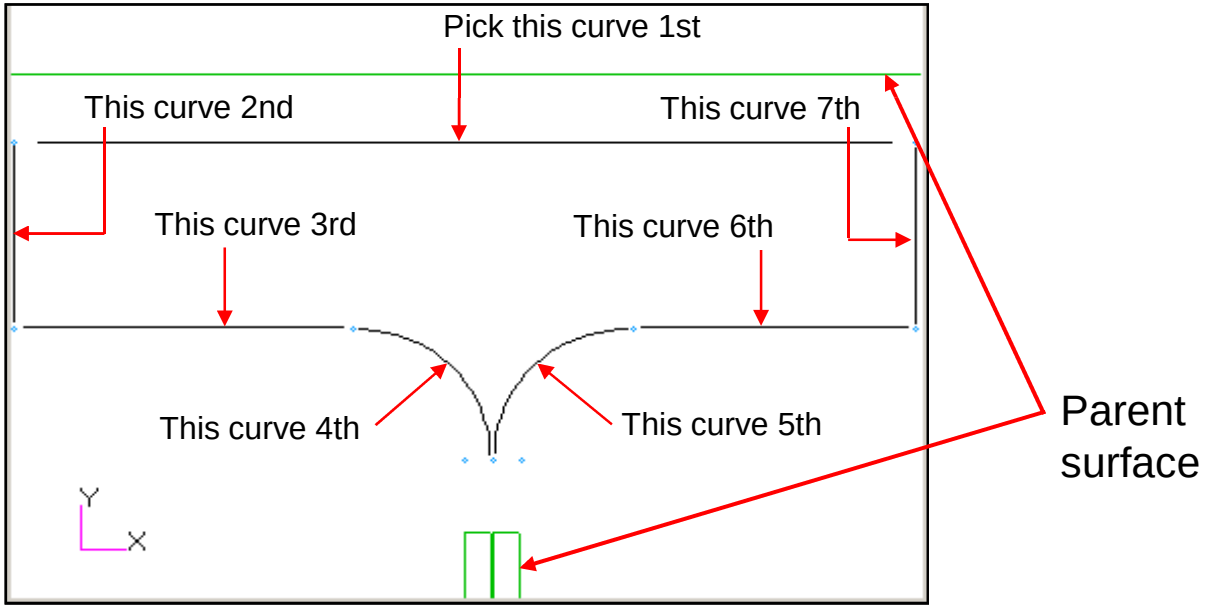


- **The ‘tail’ curves should not be included in the new chain curve that will be built.**

Copyright© 2012 MSC.Software Corporation

EXAMPLE: MORE SIMPLE EDITING

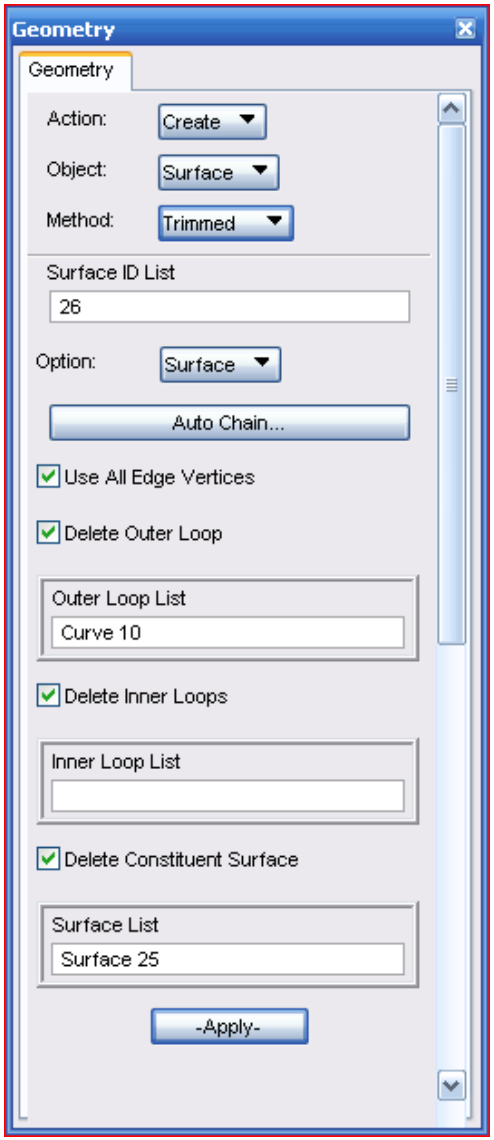
- **In fact, there are two curves that make-up the ‘tail’; Curves 1 & 2. Delete these curves.**
- **Zoom into the area where the chain curve is to be created, and create the chain curve.**



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MORE SIMPLE EDITING

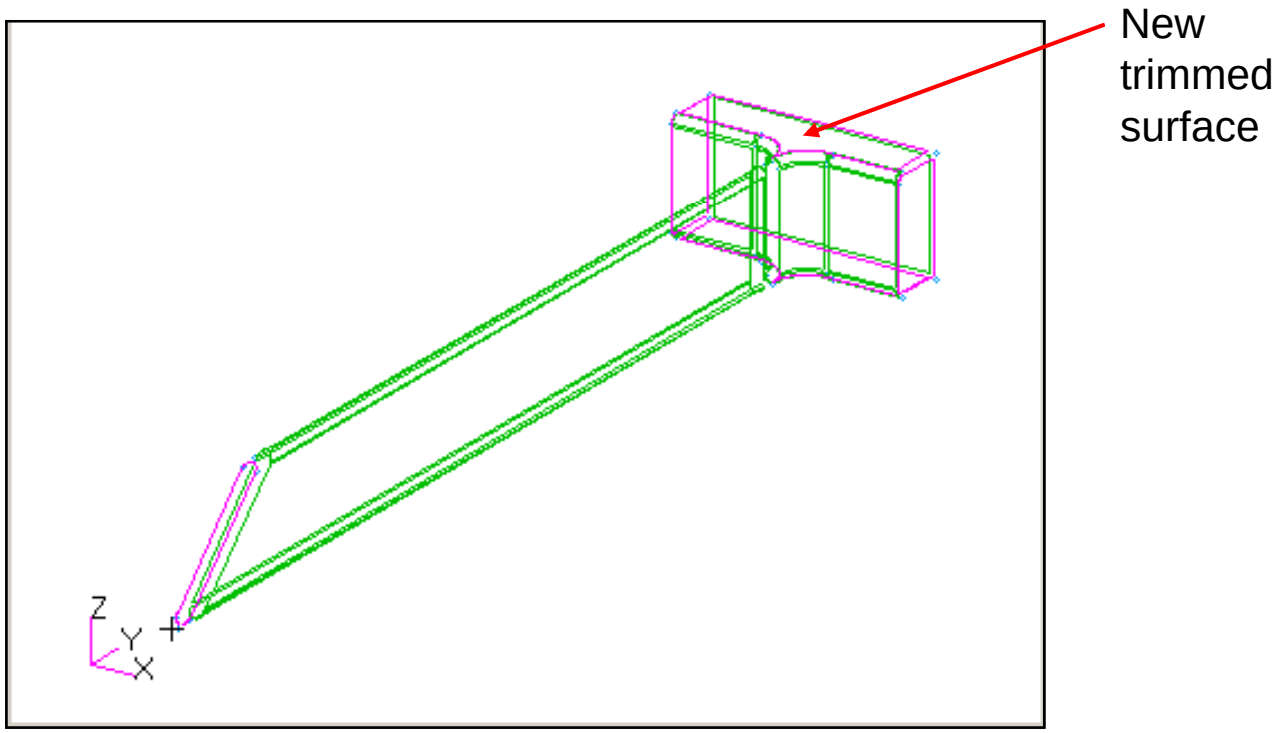
- **Create a new trimmed surface using the chain curve just created (Curve 10) and the parent parametric surface created during the disassembly of the original trimmed surface (Surface 25).**
- **Display this surface with all the original surfaces, without the original trimmed surface.**



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MORE SIMPLE EDITING

- **These surfaces are congruent.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

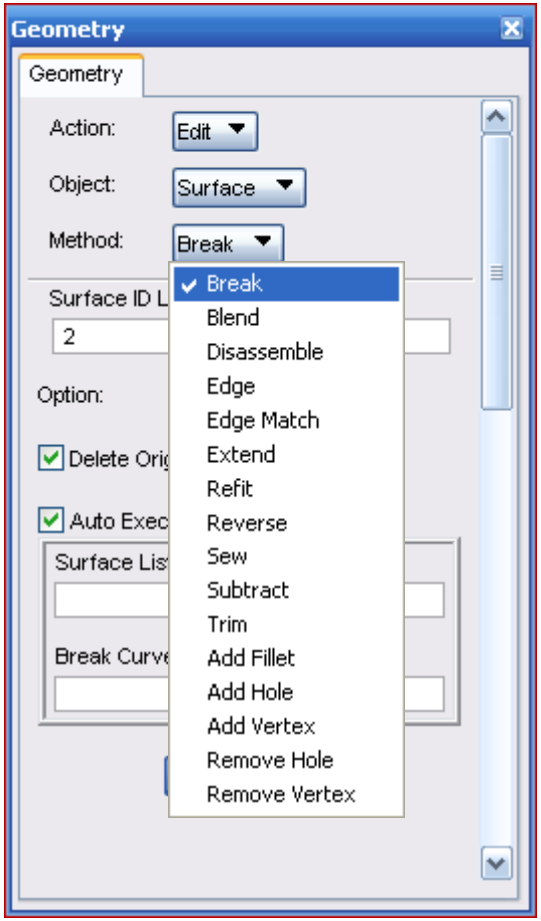
- **Here is a case study of a panel with numerous cutouts, sharp edges, and small features.**
- **Use surface modeling features such as: add hole, vertex, fillet, etc. to edit the surface to create a good Paver mesh.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

- **2D surface editing features used are**
  - Edit / Surface / Add Fillet
  - Edit / Surface / Add Hole
  - Edit / Surface / Add Vertex
  - Edit / Surface / Edge
  - Edit / Surface / Remove Hole
  - Edit / Surface / Remove Vertex

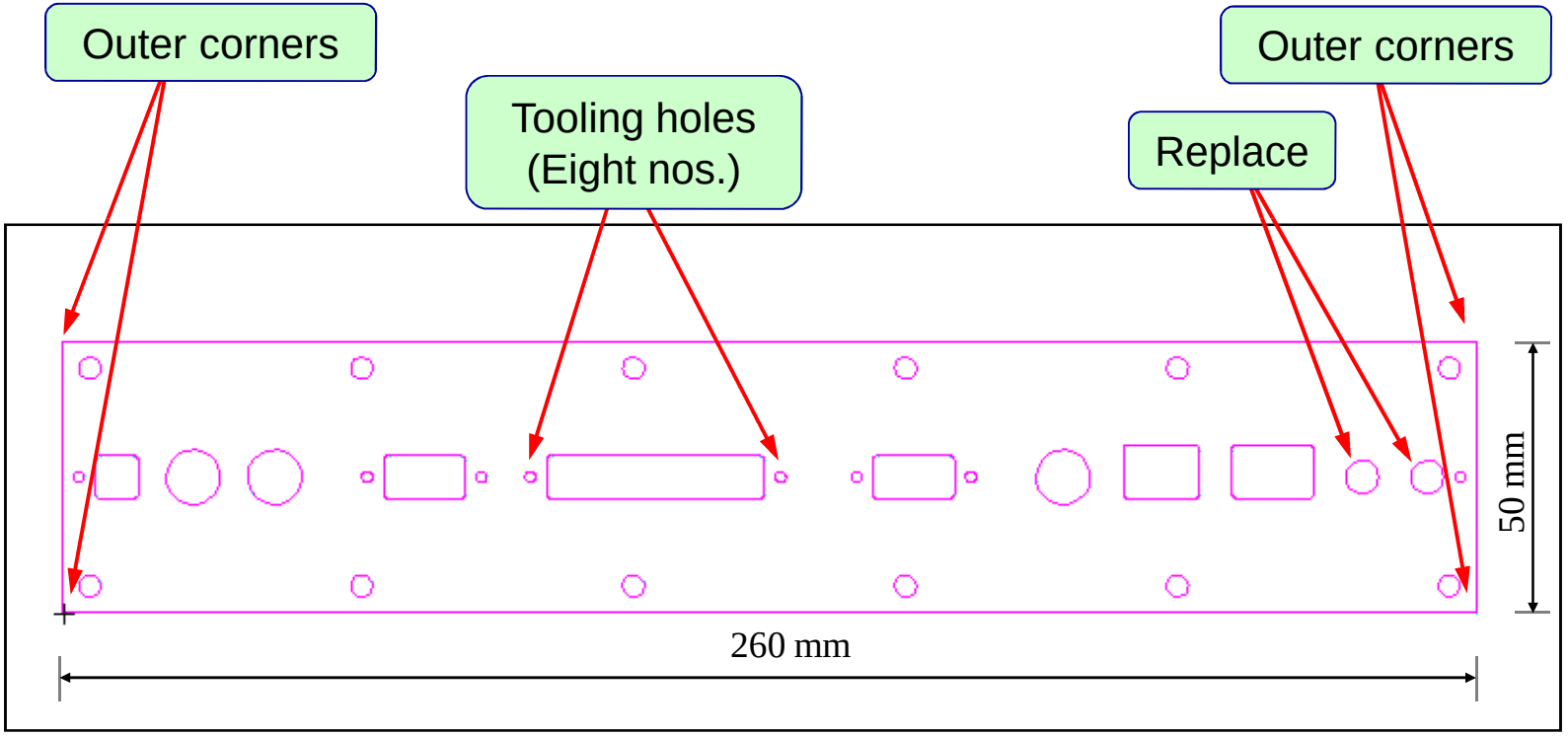
- **The above features are demonstrated in the following case study**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

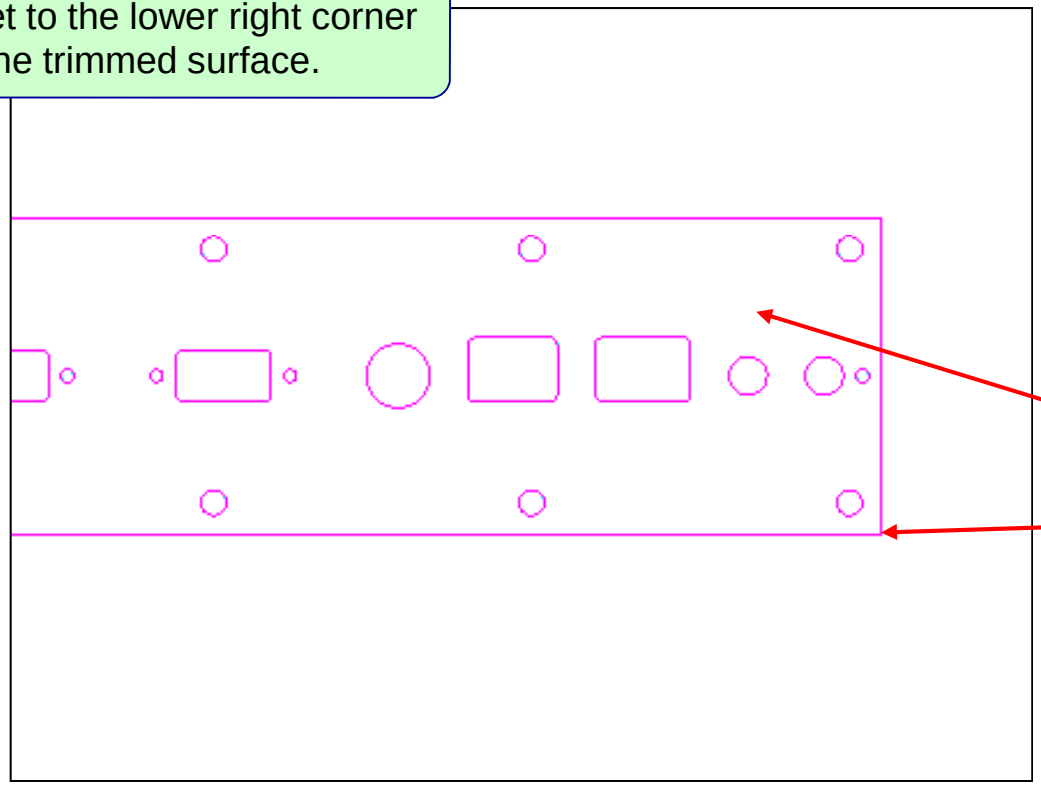
- **Import the STEP file, computer_panel.stp, with the panel model (surface).**

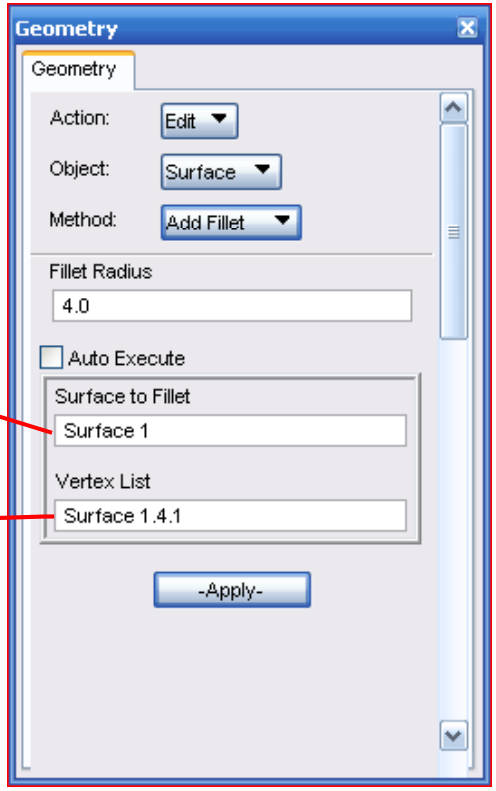


Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

Add a fillet to the lower right corner of the trimmed surface.

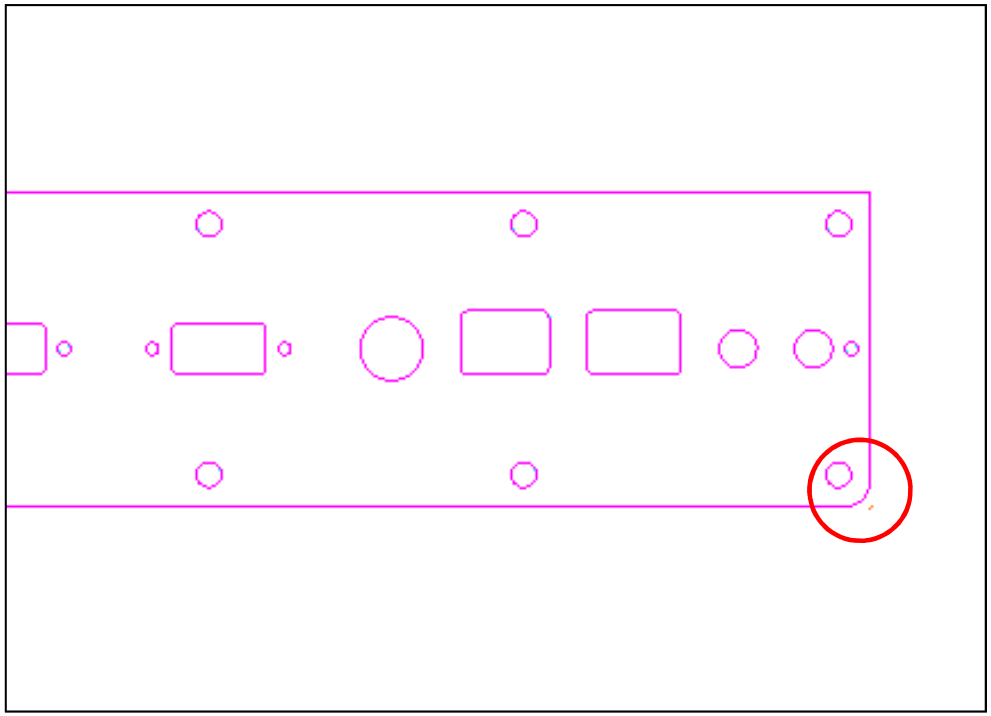




Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

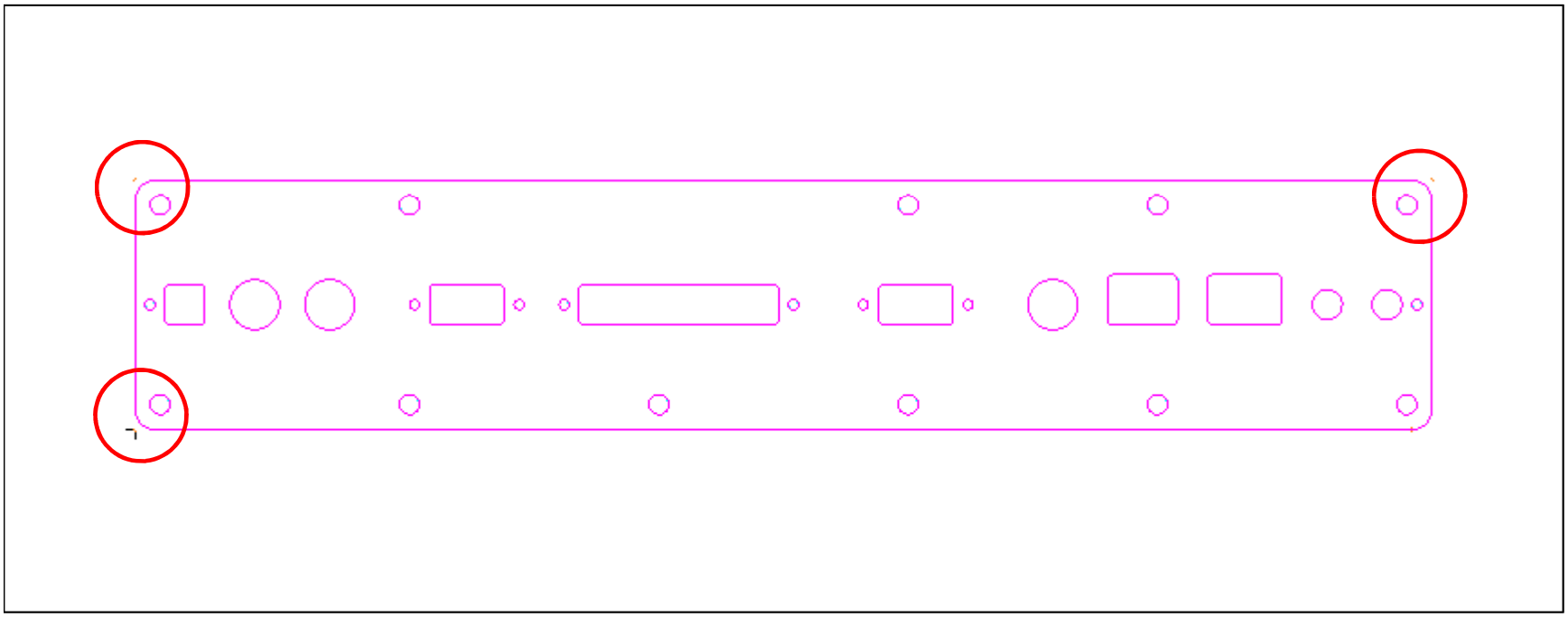
- **The result should look like the following:**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

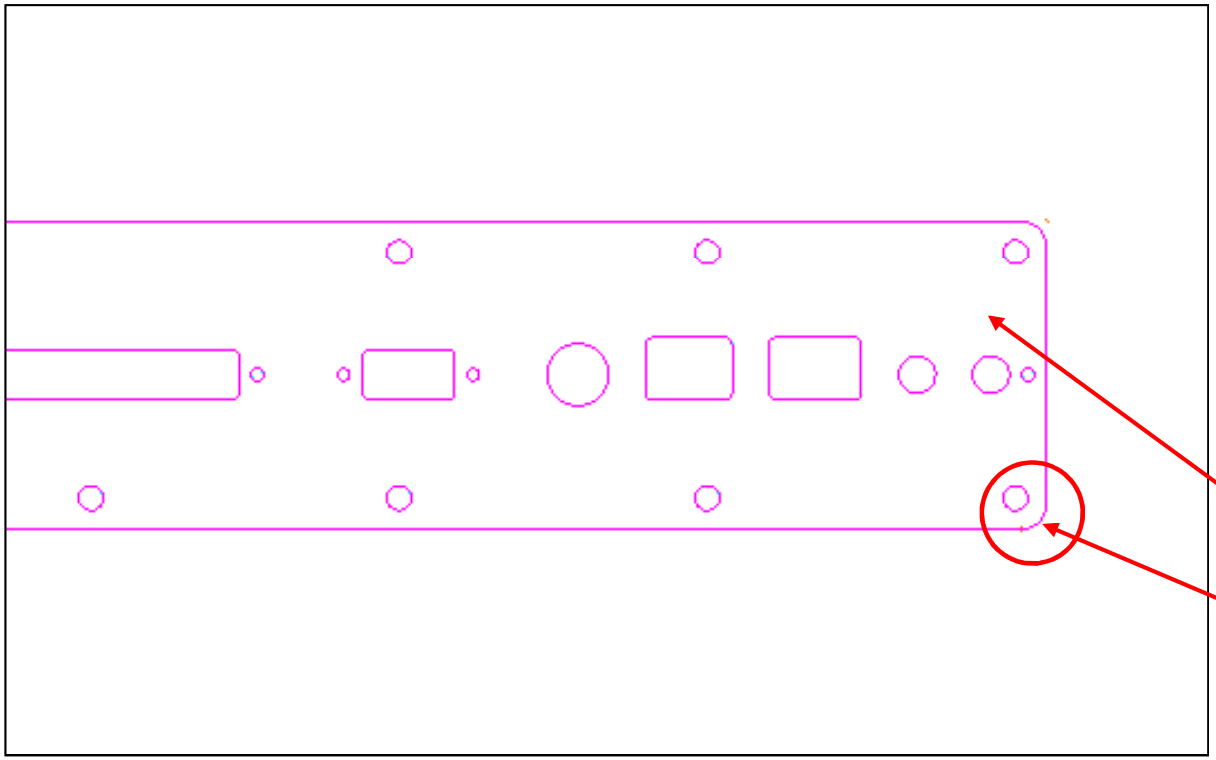
- **Add fillets to the remaining corners, following the same procedure.**

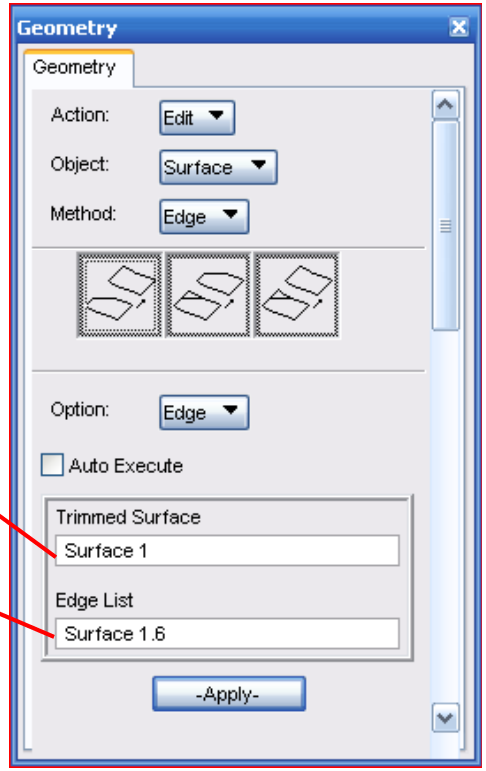


Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

- **Fillets can be removed just as easily if required.**

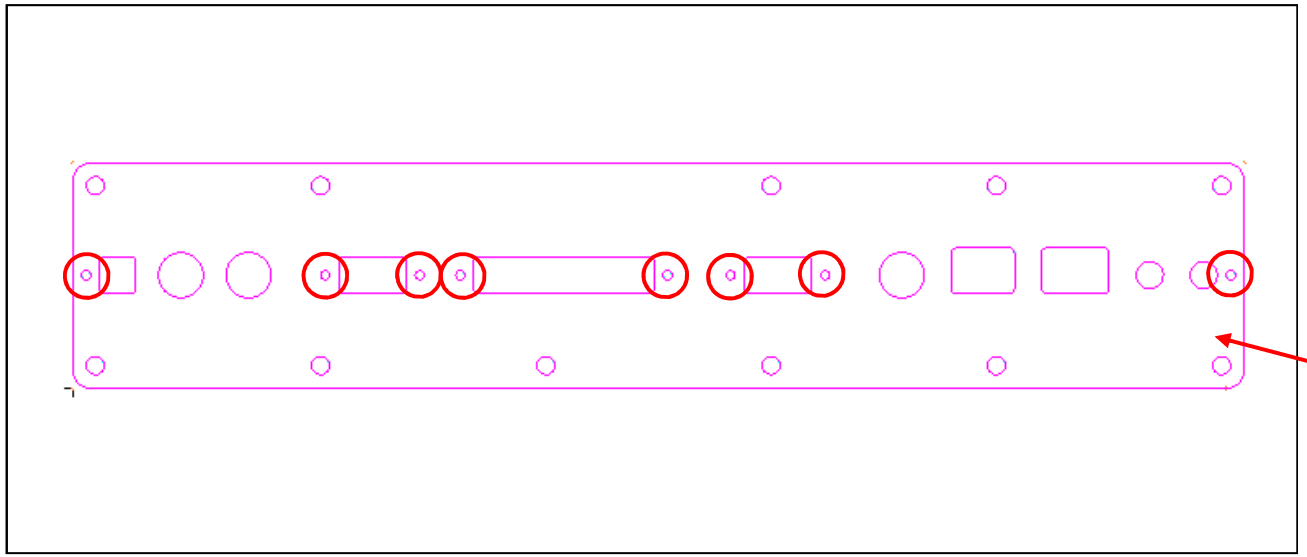


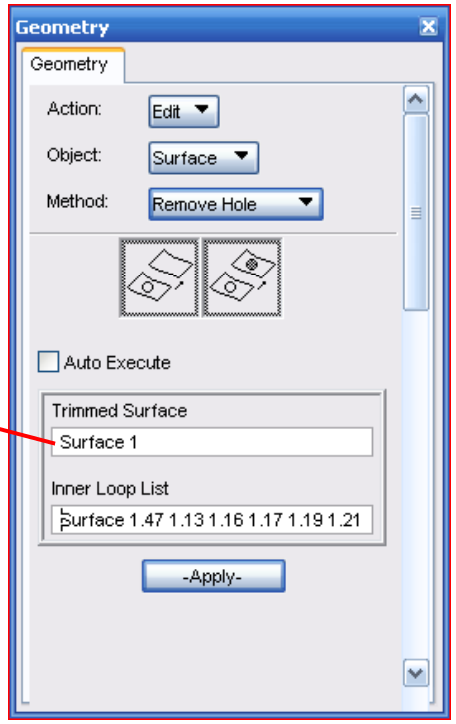


Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

- **Remove tooling holes from panel**

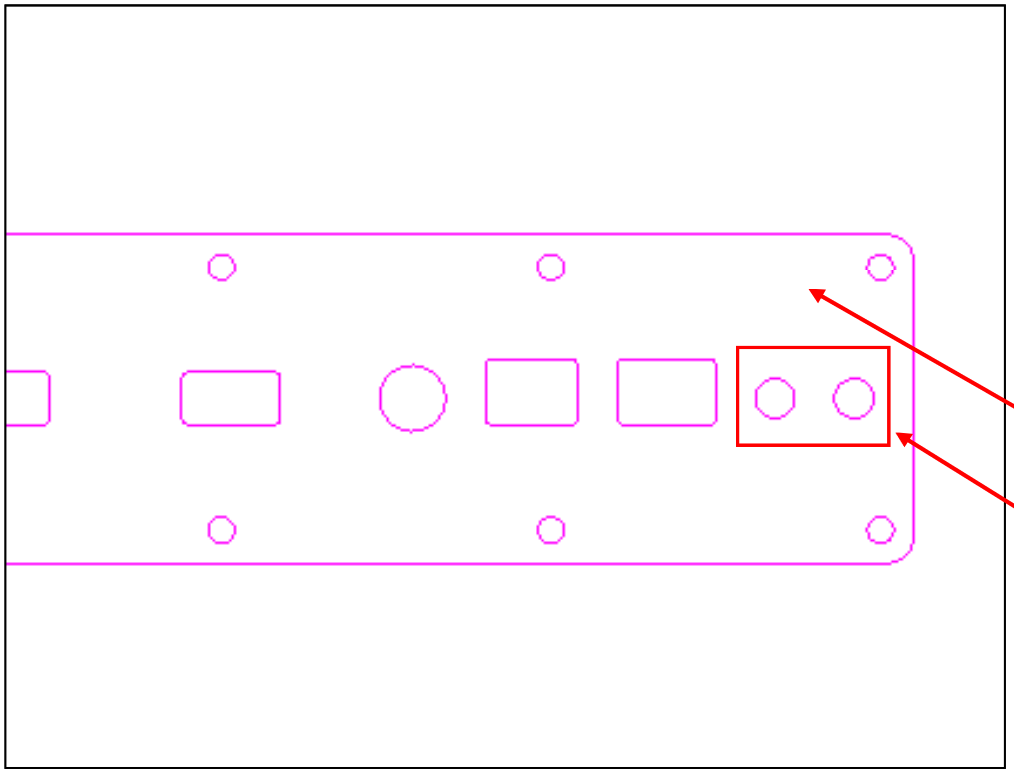


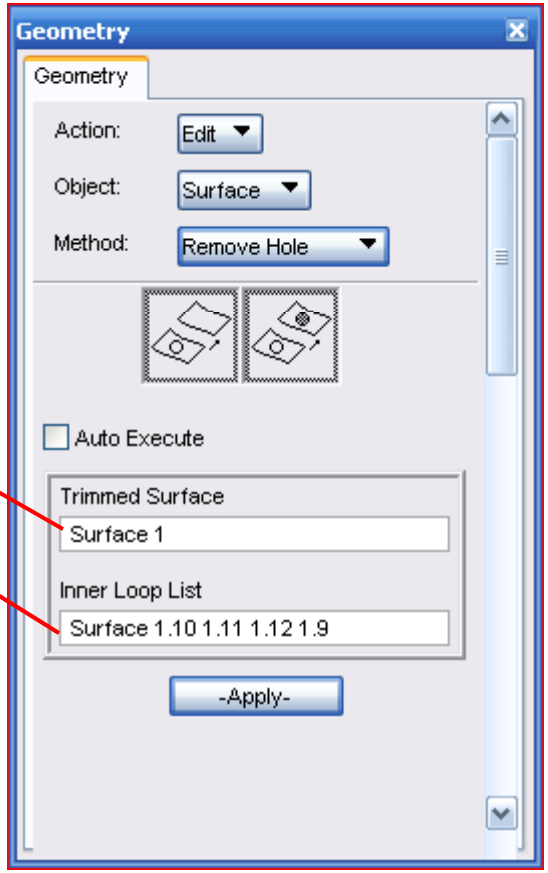


Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

- **Decrease the radius of the speaker plug holes: first, remove the existing holes**

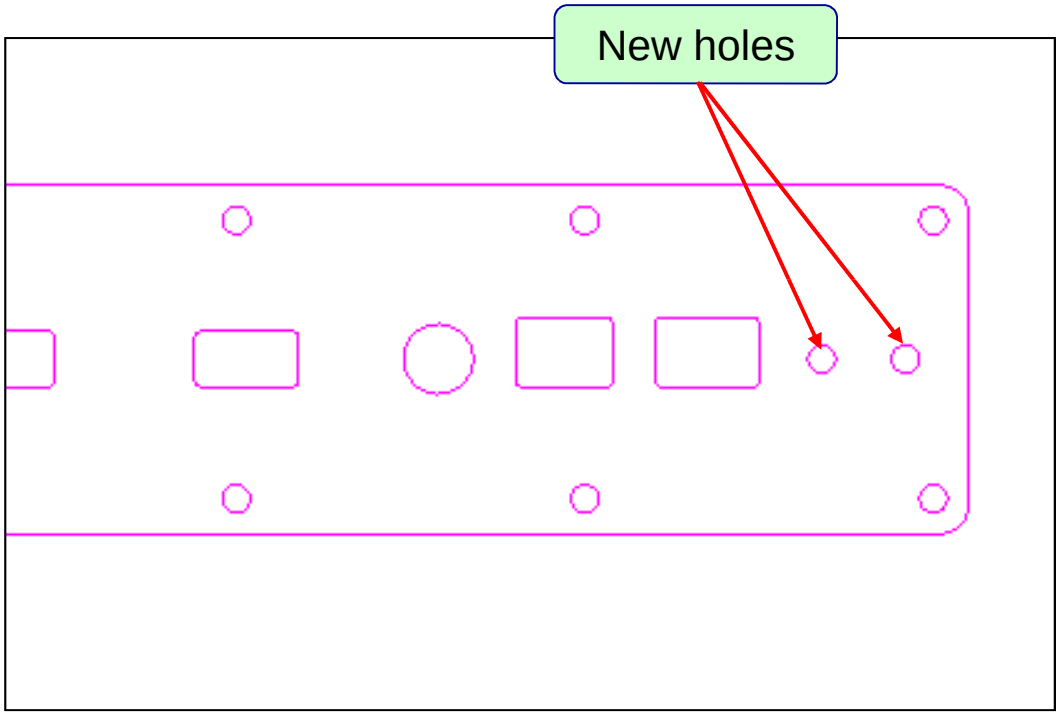


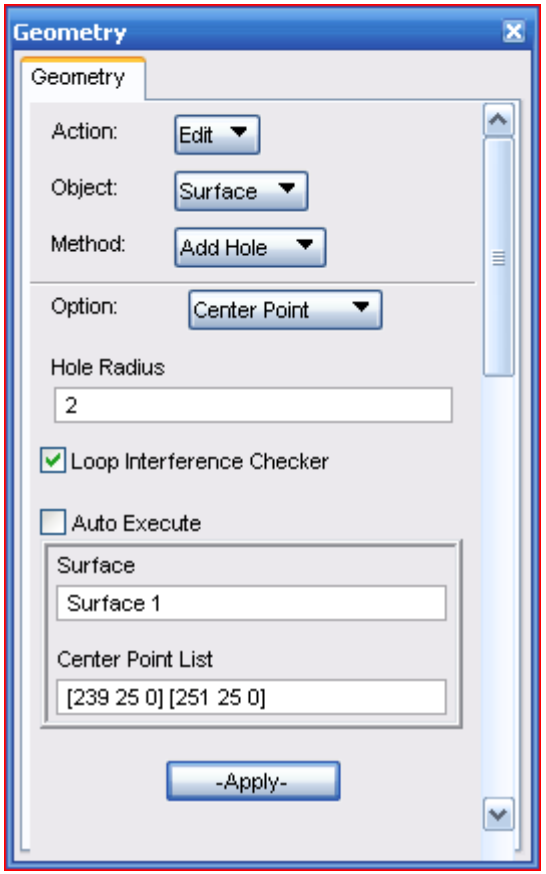


Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

- **Recreate the holes with the new radius**

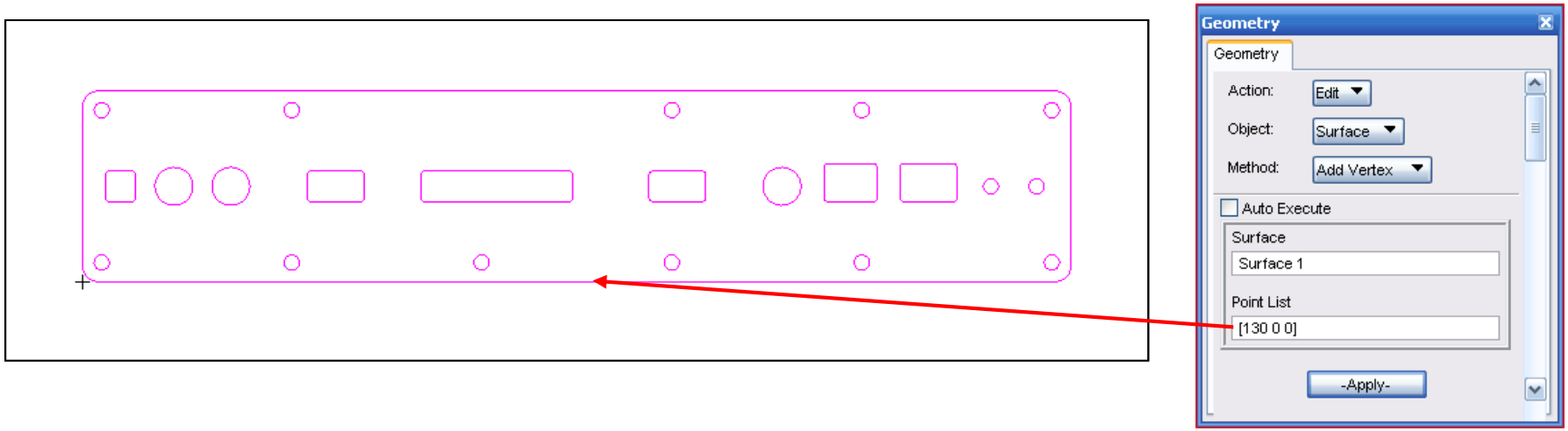




Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

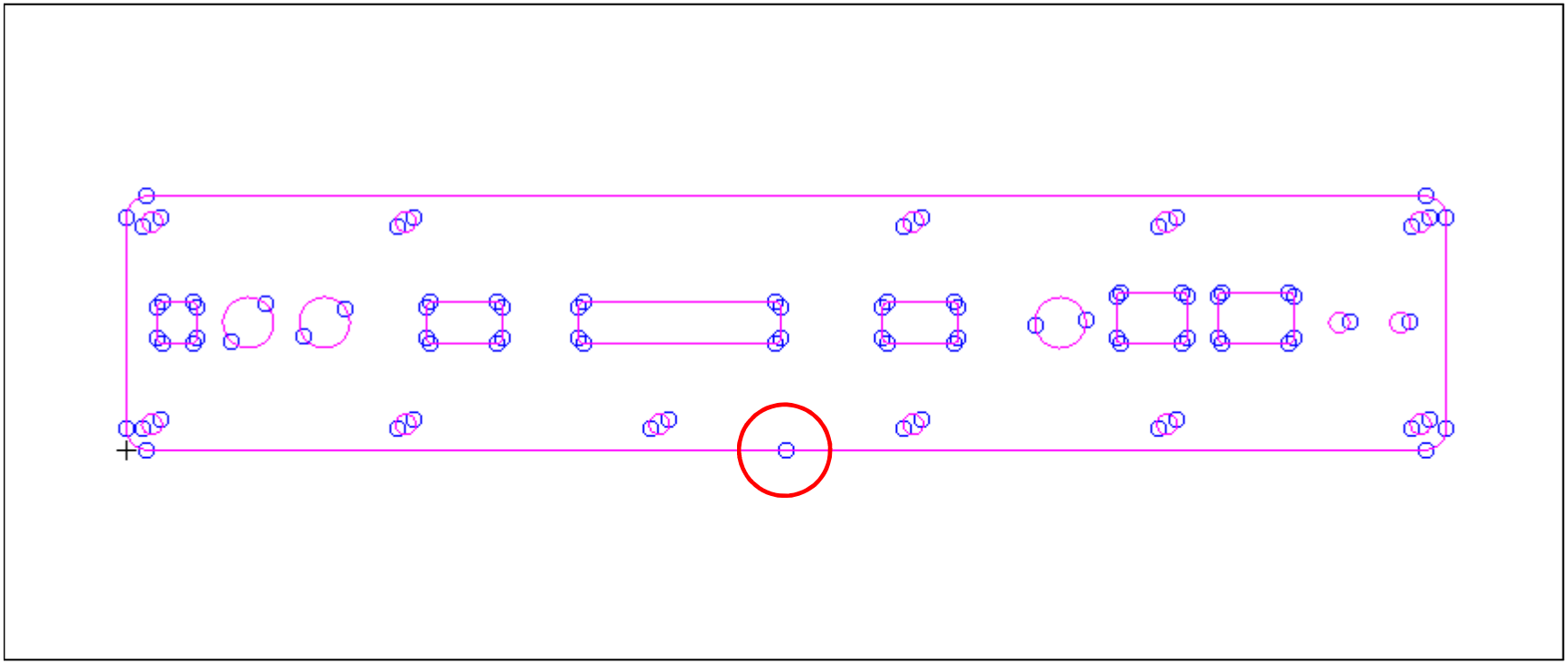
- **Before Paver meshing, add a new vertex to the surface at the location where the load is to be applied ([130,0,0]). This will force the mesher to place a finite element node at that location.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

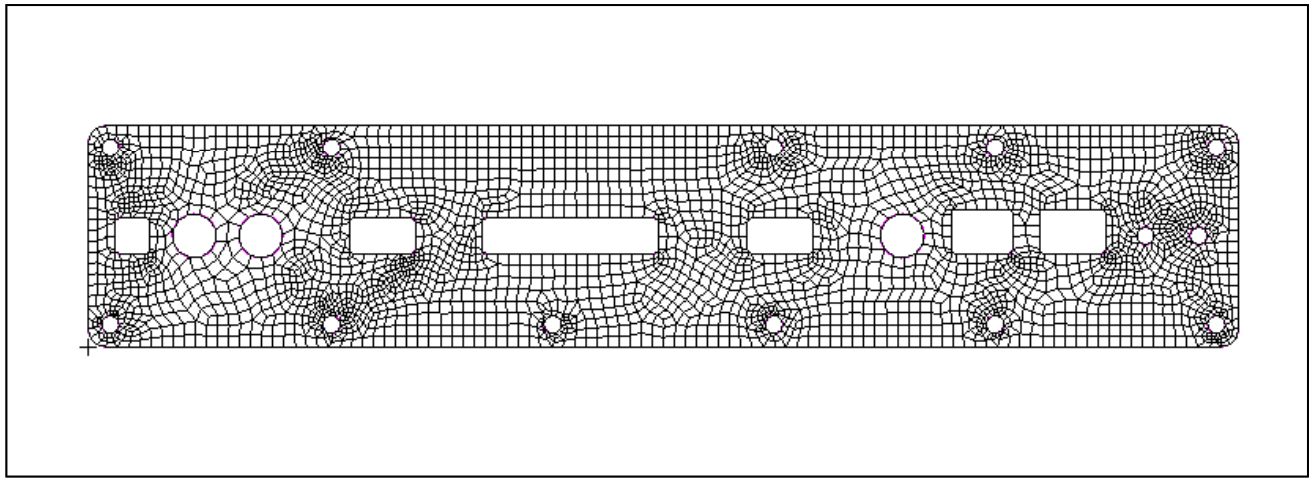
- **Observe the newly created point:**

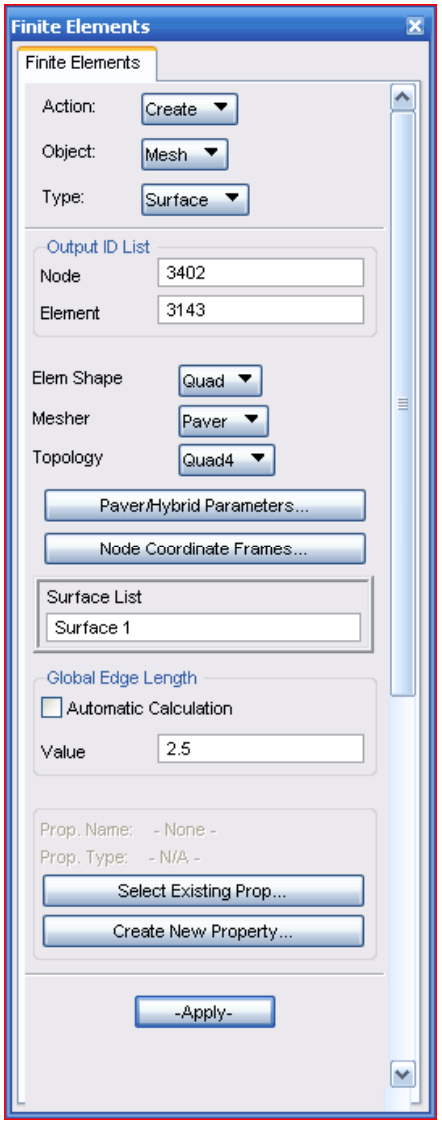


Copyright© 2012 MSC.Software Corporation

CASE STUDY: COMPUTER PANEL

- **Mesh the model with Quad4 elements, using the Paver mesher, and a Global Edge Length of 2.5.**





Copyright© 2012 MSC.Software Corporation

EXERCISES

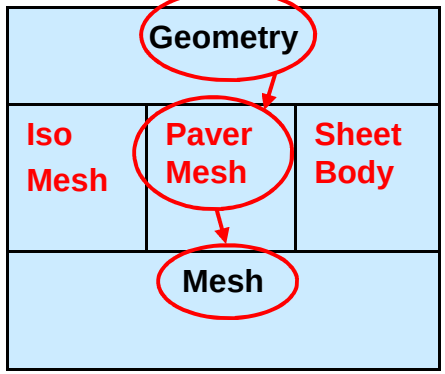
- Perform Workshop 1 “Edit and Mesh Instrument Panel Geometry” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

GEOMETRY EDITING COMPOSITE SURFACE

- **There are two methods:**
  Create / Surface / Composite
  Create / Surface / Trimmed / Option: Composite

Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated Surface	Iso Mesh	Paver Mesh	Sheet Body	Composite Surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	



Copyright© 2012 MSC.Software Corporation

GEOMETRY EDITING COMPOSITE SURFACE

- **Automatic boundary detection**
- **The user can define a final vertex configuration of the composite surface.**
- **Boundaries can be previewed with colored markers to help indicate problem areas.**
- **Clean-up tolerances can be used to close perimeter and internal gaps.**
- **Surface edges may be added or removed, or curves may be added as needed to define the outer boundary.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Create a composite surface using**
- **Composite from the Surfaces group under the Geometry tab.**

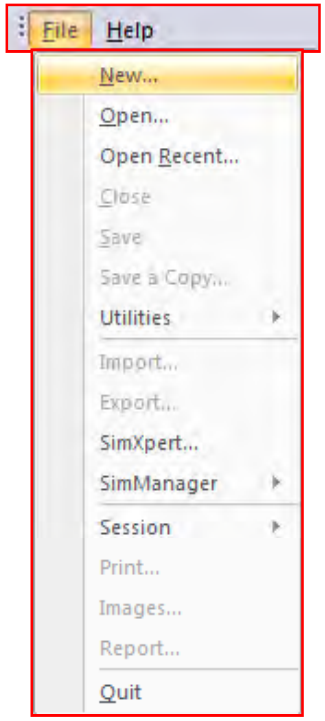
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

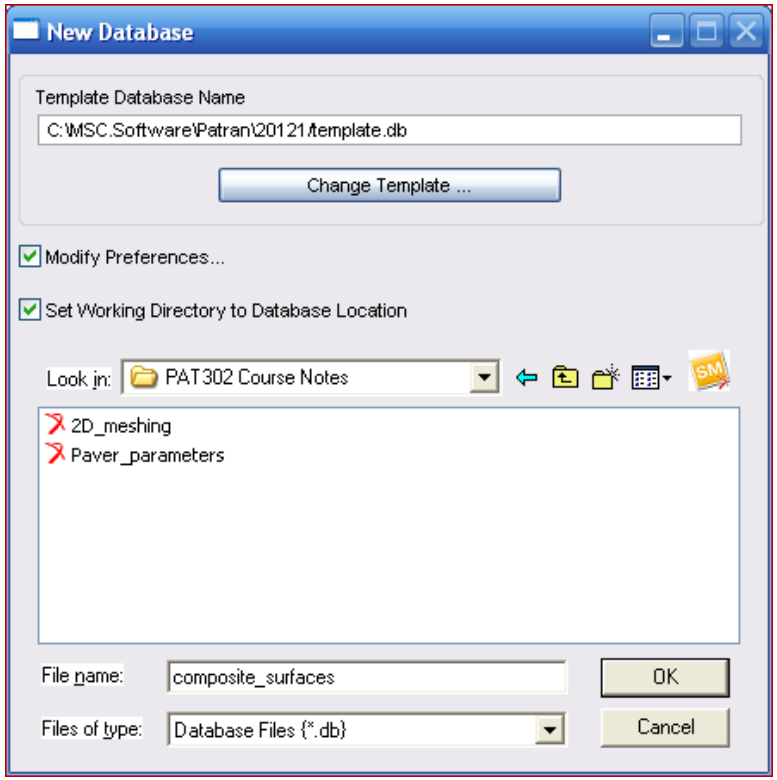
- **Problem Description**
  - In this exercise, a CAD file for a plate structure is imported into a database. However, the translation leads to certain gaps and discontinuities in the geometric model. In order to create a suitable mesh for analysis, the geometry needs to be “cleaned up”. This is where the concept of composite surfaces arises.
  - Take the many disjointed (non-congruent) surfaces which represent the plate structure and combine them to create a set of composite surfaces. These newly created composite surfaces will then be mesh seeded, then meshed.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE



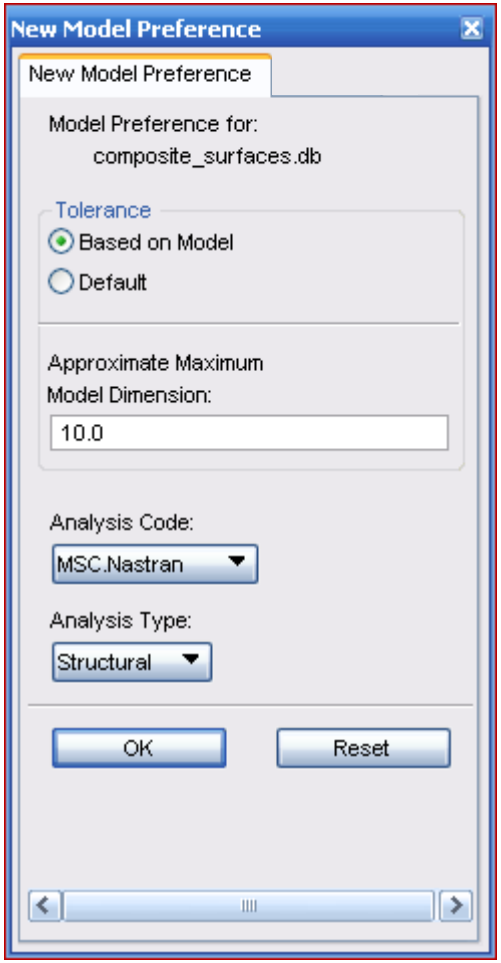
- **Create a new database**
  - Enter **composite_surfaces** for *File name.*
  - Click **OK**



Copyright© 2012 MSC.Software Corporation

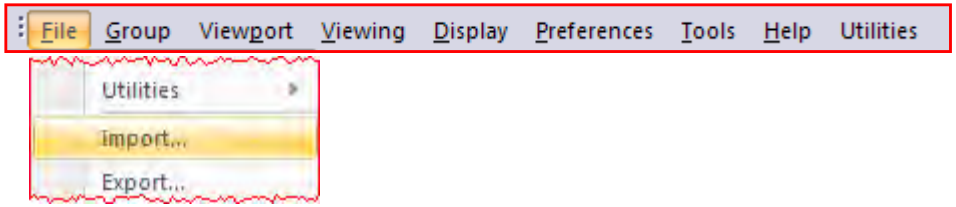
CASE STUDY: CREATE COMPOSITE SURFACE

- **Set the Tolerance to Based on Model**
- **Specify the Approximate Maximum Model Dimension as 10.0 (subsequent to this, the actual size of the model will be checked and the implications of using 10.0 as the maximum dimension will be investigated).**

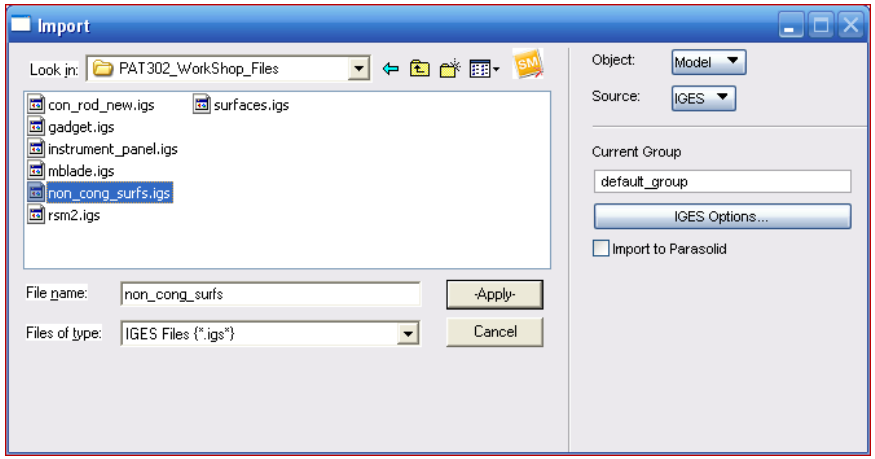


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE



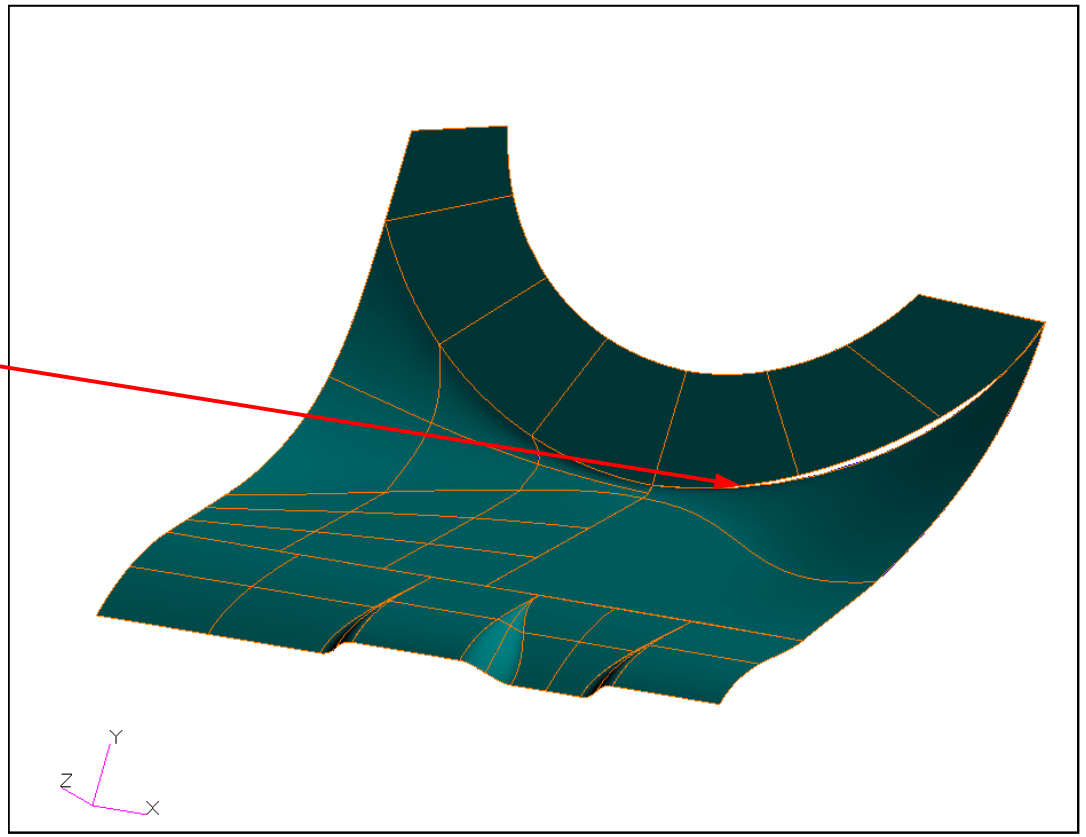
- **Import the Surface Geometry**
  - Pull down **Import…** from the *File* menu
  - Set *Object and Source* to **Model** and **IGES,** respectively.
  - Uncheck **Import to Parasolid**
  - Select **non_cong_surfs.igs**
  - Click **Apply**
- **An IGES import summary form appears. Look at the IGES entities there, and the Patran Geometry created.**
- **Click OK.**



Copyright© 2012 MSC.Software Corporation

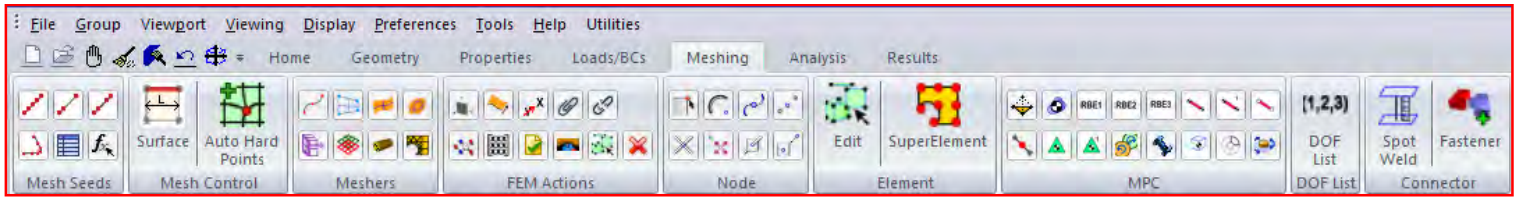
CASE STUDY: CREATE COMPOSITE SURFACE

- **The model shown here is a set of surfaces, all of which are parametric (green).**
- **There are a considerable number of surfaces.**
- **Several of the surfaces are sliver-like.**
- **A crack is evident.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

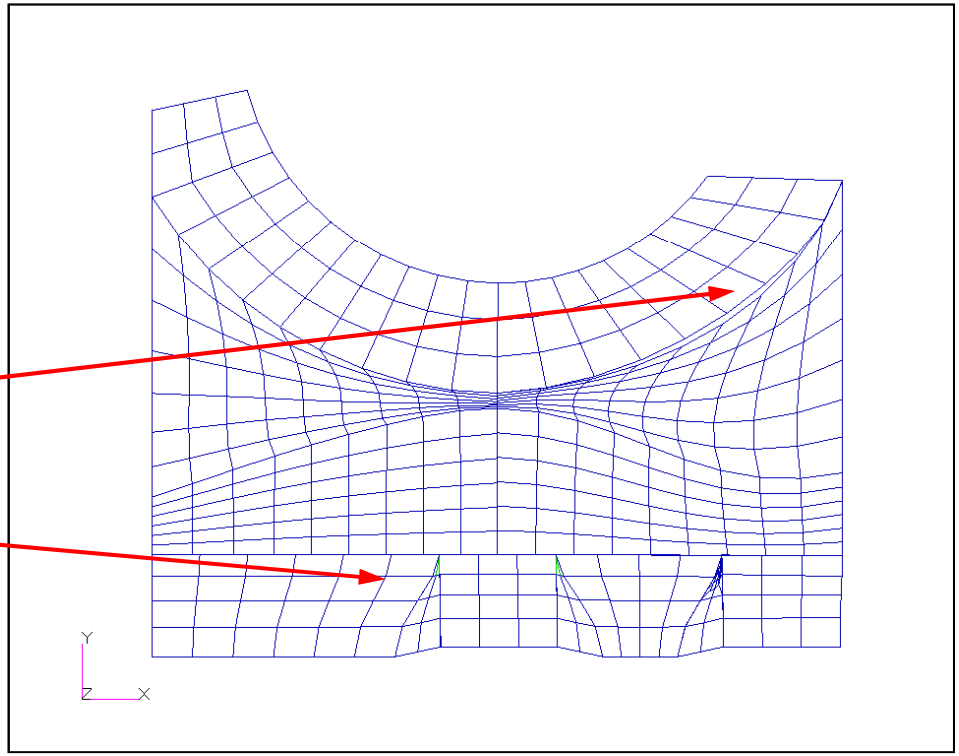


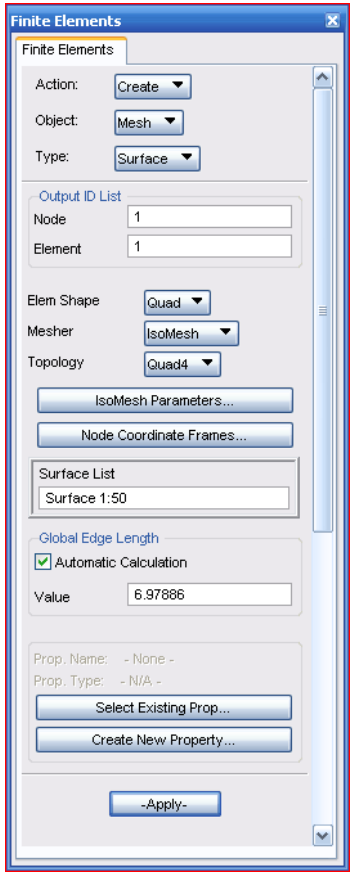
This is the mesh that would be obtained by IsoMesh, using the default parameter settings.

There exist two severe problems:

1. There are cracks in the mesh – the mesh is not congruent

2. The elements are highly distorted at sharp edge connections



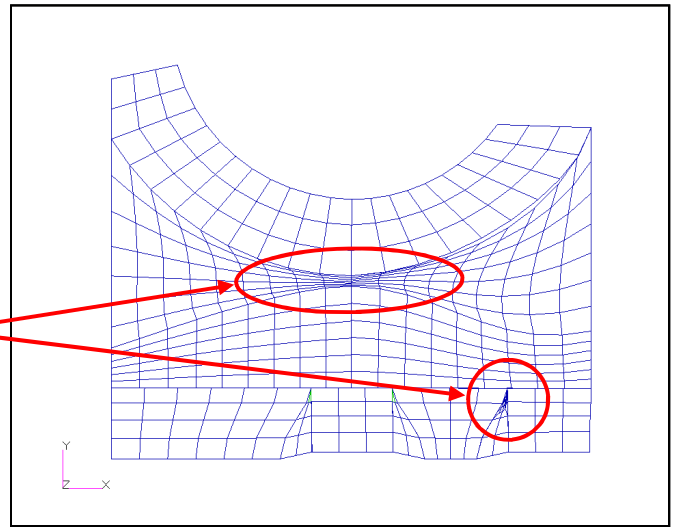


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **What are the options now ?**
  - Paver mesh – it has been seen that the IsoMesh mesh was not acceptable, so a Paver mesh would probably not be good
  - Paver mesh, then edit the mesh by hand
    - This may leave elements with sharp corners and destroy the relationship of mesh to geometry.

Sharp corners (intersection of edges) will dominate mesh

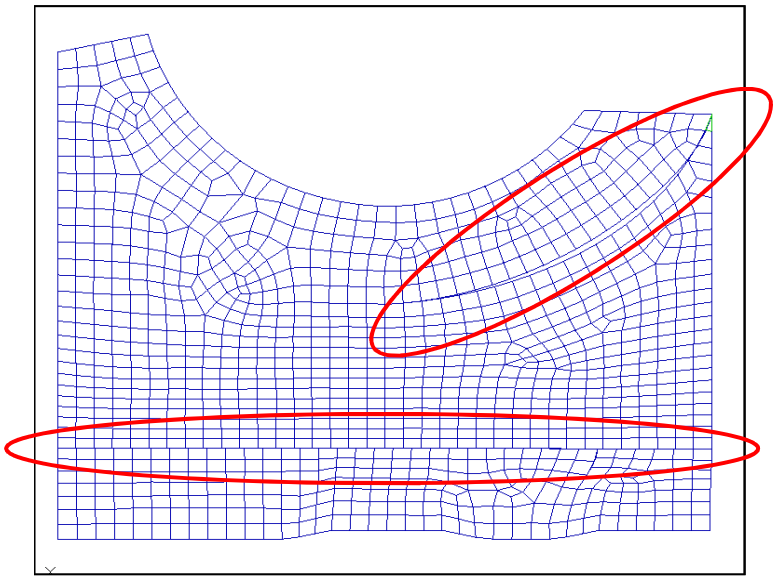


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- Can attempt to “Sew” the geometry, but this will not deal easily with cracks across multiple surfaces
- “Sheetbody Mesh” mesh (covered later) – requires a congruent set of surfaces; do not have that
- Build a composite surface
  - This has the major advantage of eliminating vertices and edges internal to the surfaces

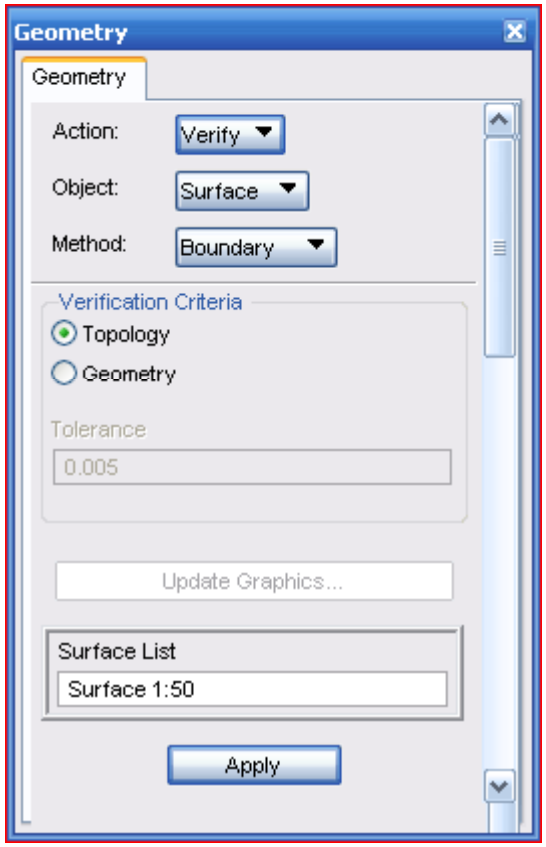
Result of Sheetbody Meshing



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Check the model for geometric incongruencies:**
  - Change the *Display* to **Wireframe**
  - Under the *Geometry* tab click on **Verify** from the *Geometry Actions* group. Set the *Object* and *Method* to **Surface** and **Boundary** respectively.
  - Select all surfaces in the viewport for *Surface List*.
  - Click **Apply**.
- **A warning message will appear in the history window stating that free edges and/or non-manifold edges exist.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

Copyright© 2012 MSC.Software Corporation

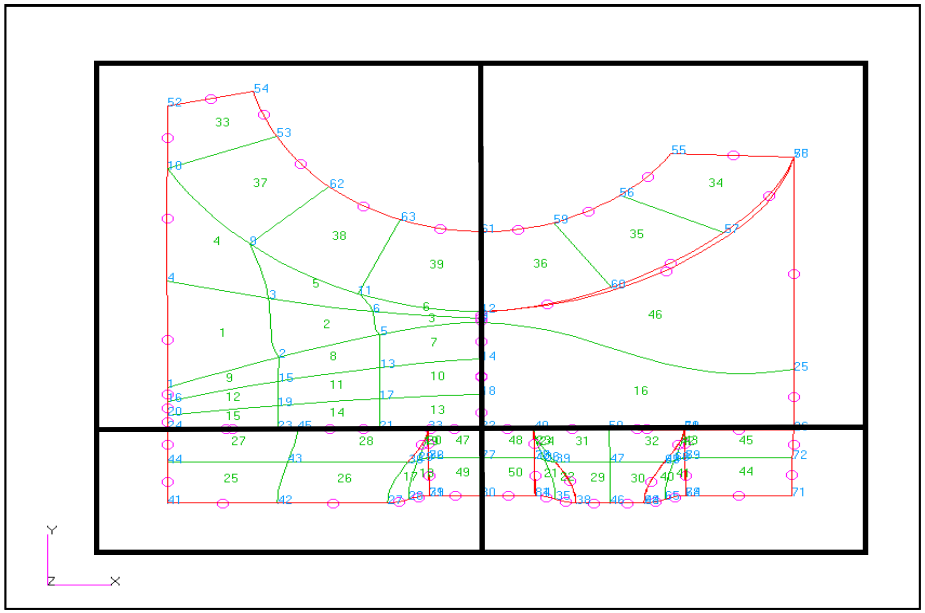
CASE STUDY: CREATE COMPOSITE SURFACE

- From the markers displayed in the interior of the model, it can be seen that there is a non-congruency problem which will affect the congruency of the meshes. That is, the meshes will not be compatible at the common boundary interfaces of non-congruent surfaces.
- This problem can be dealt with by using composite surfaces in Patran. For this, a set of several surfaces are combined into a new simple or trimmed surface whose perimeter is identical to that of the set of surfaces. A plan has to be developed to use composite surfaces. The following procedure outlines this plan.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

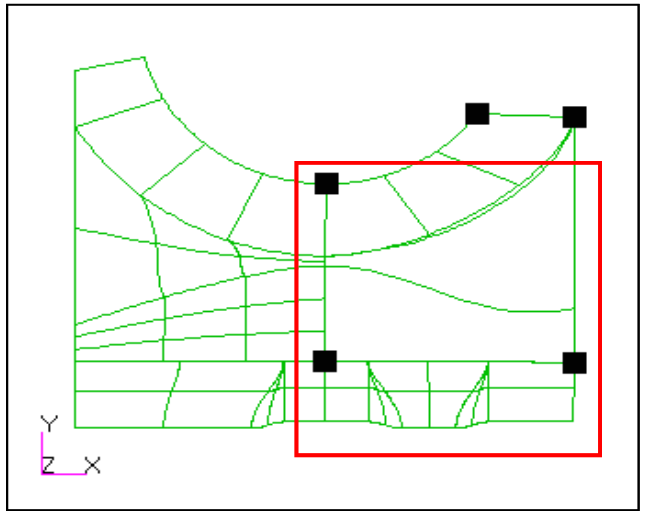
- **The model will be divided into four quarters. The surfaces in each of the quarters will be used to construct a single composite surface for each quarter of the model. The four newly created composite surfaces will then be meshed and equivalenced, creating a single, ‘fluid’, consistent mesh which can be used for analysis.**
- **Also, because there are no surface edges interior to a composite surface, when a composite surface is meshed the created nodes do not have to follow interior edges or vertices.**

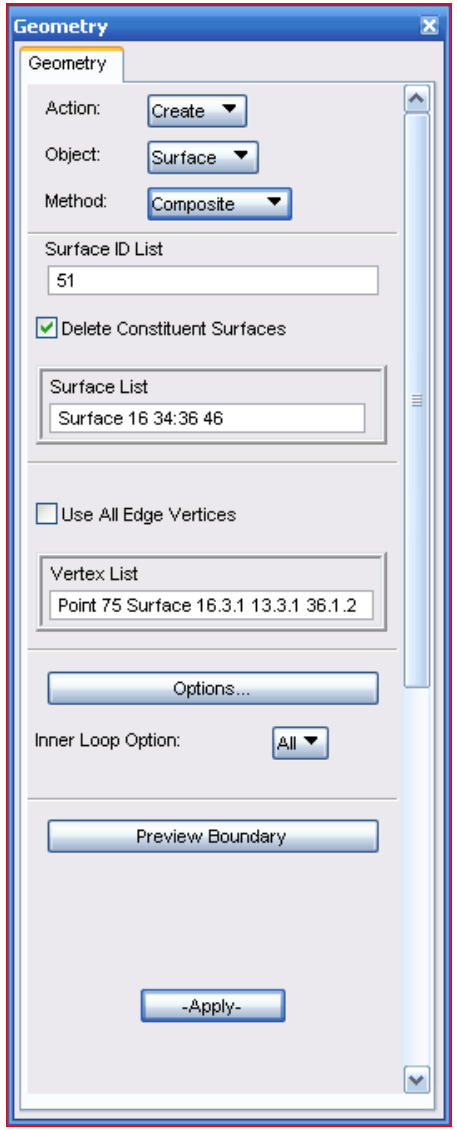


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Create the first composite surface**
  - Under the *Geometry* tab click on **Composite** in the *Surfaces* group.
  - Select the surfaces of the upper right corner for *Surface List*.
  - Select five points (points from the selected surfaces), to define the outside corners of the composite surface, for *Vertex List.*

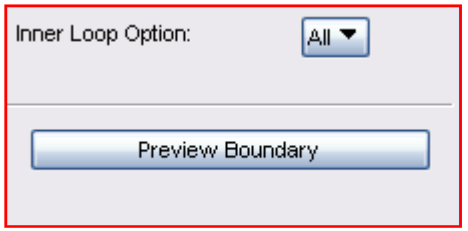


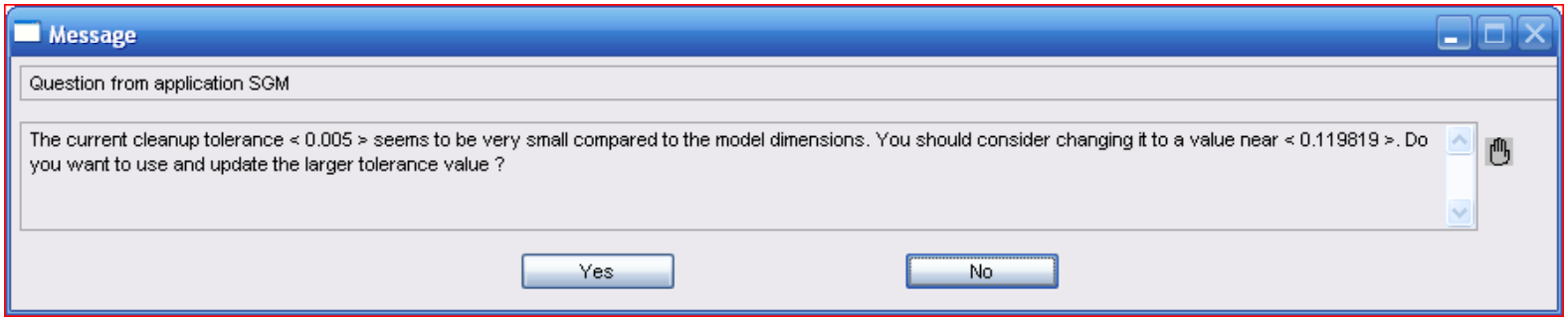


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **View the current perimeter of the set of surfaces, as seen by the composite surface application**
  - Click on **Preview Boundary**.
  - The following message appears:





  - Reply **No** to using the suggested tolerance.

Copyright© 2012 MSC.Software Corporation

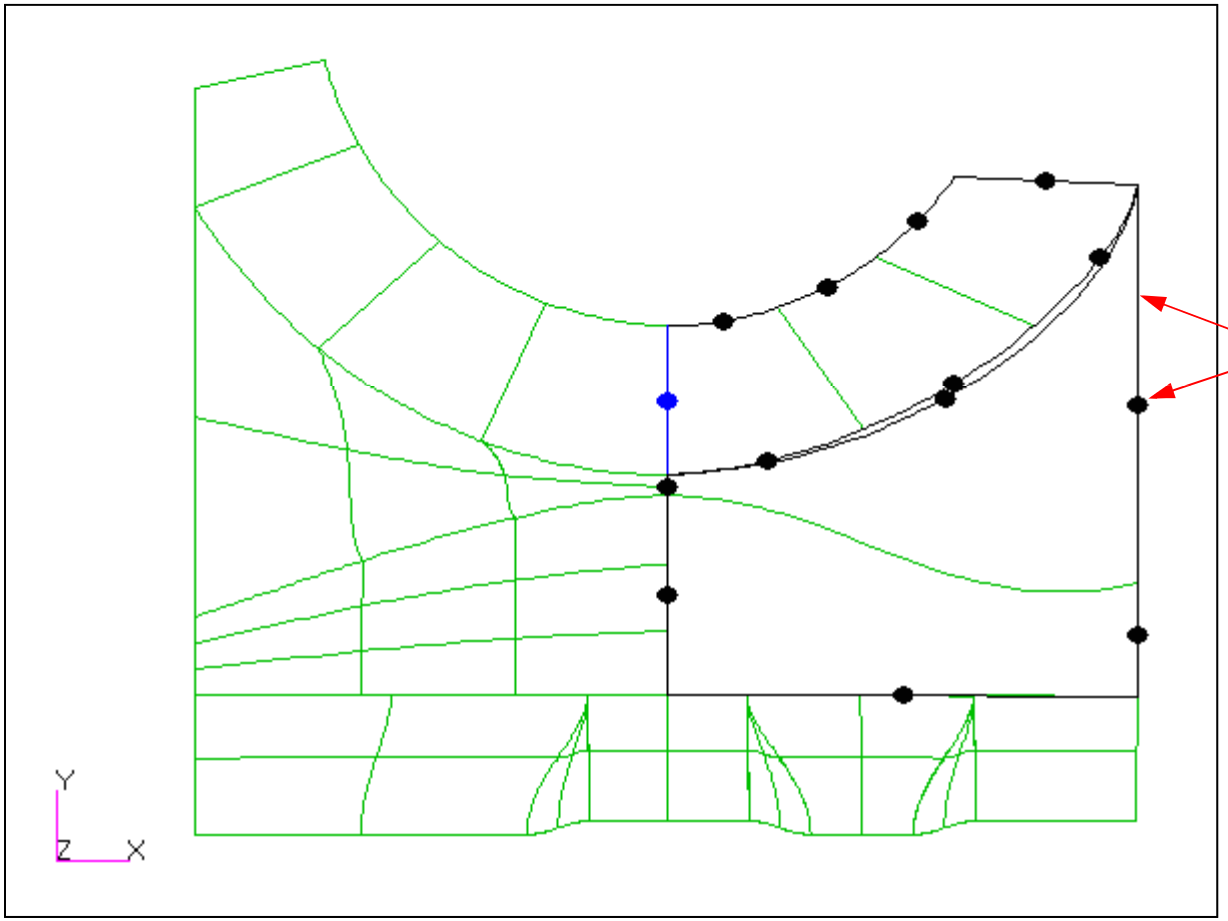
CASE STUDY: CREATE COMPOSITE SURFACE

- An informational message will appear explaining what the various colors mean with regards to the boundary
  - White – free edges within the current surface selection
  - Dark blue – two coincident surface edges
  - Light blue – more than two coincident surface edges
  - Red – surface edges that are not part of the boundary due to a gap

- **Continue to the next slide to see graphics.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

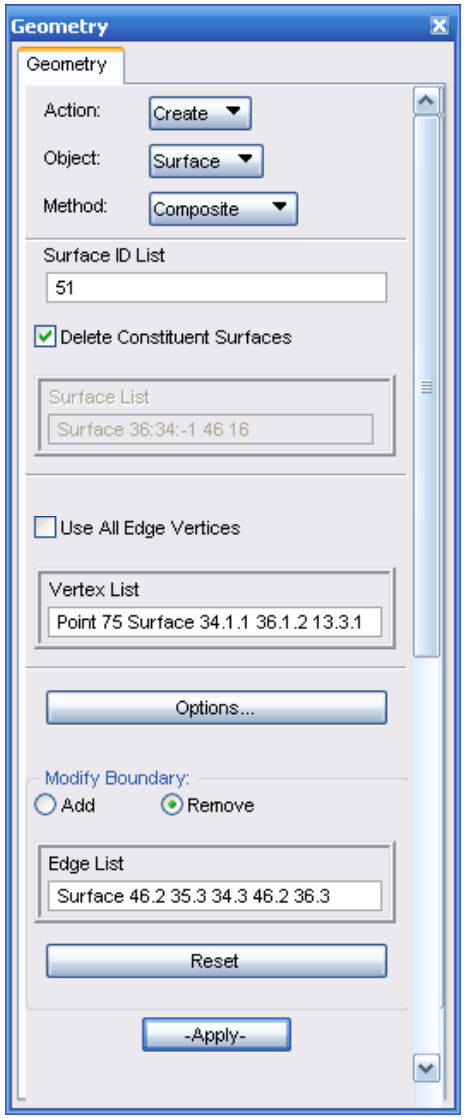


White lines and dots in Patran are shown here as black

Copyright© 2012 MSC.Software Corporation

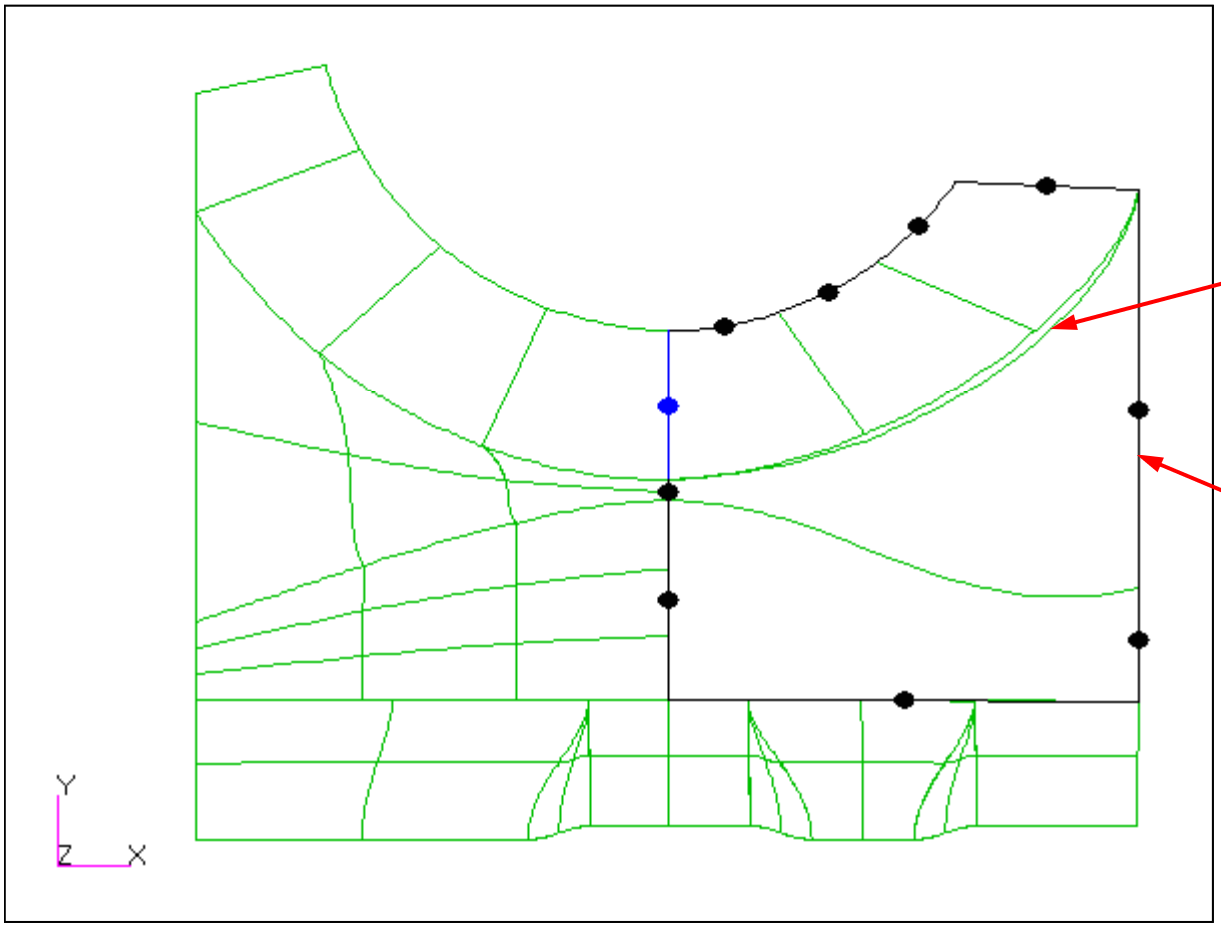
CASE STUDY: CREATE COMPOSITE SURFACE

- **The boundary definition (represented by white and dark blue lines with solid dots) needs to be modified so it will include only the desired surface edges. For this problem, only the surface edges on the outside (outer perimeter) of the set of surfaces in Surface List are to be used. The white surface edges with white dots interior to the surfaces must be removed from the boundary definition.**
  - Click on **Remove**.
  - Select the interior (white) edges (in any order).
  - Click **Apply**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE



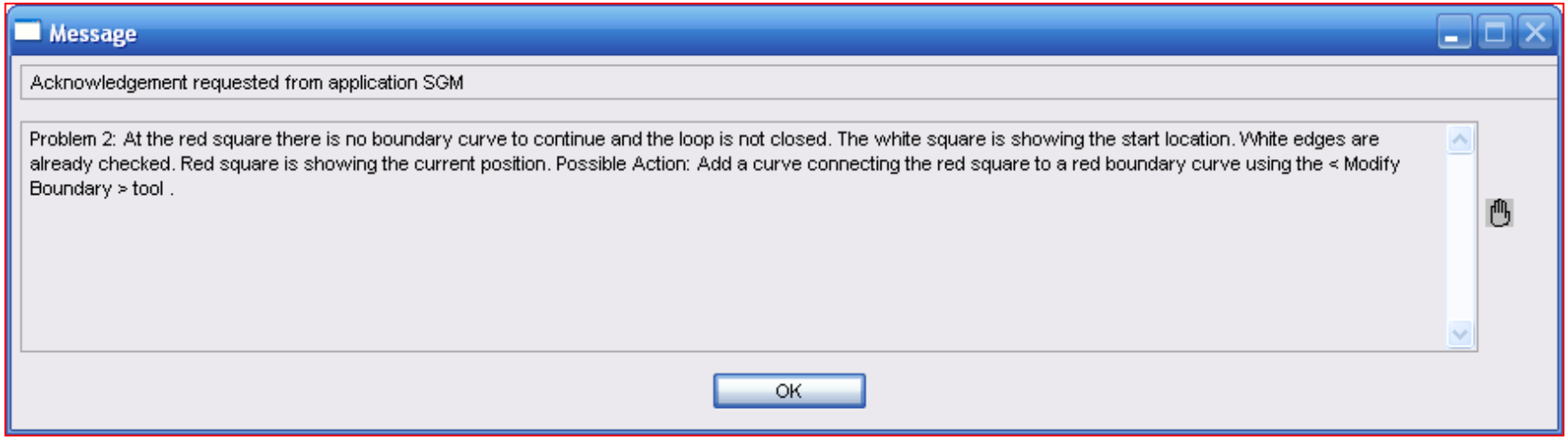
Interior edges removed from boundary

Boundary (outer)

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

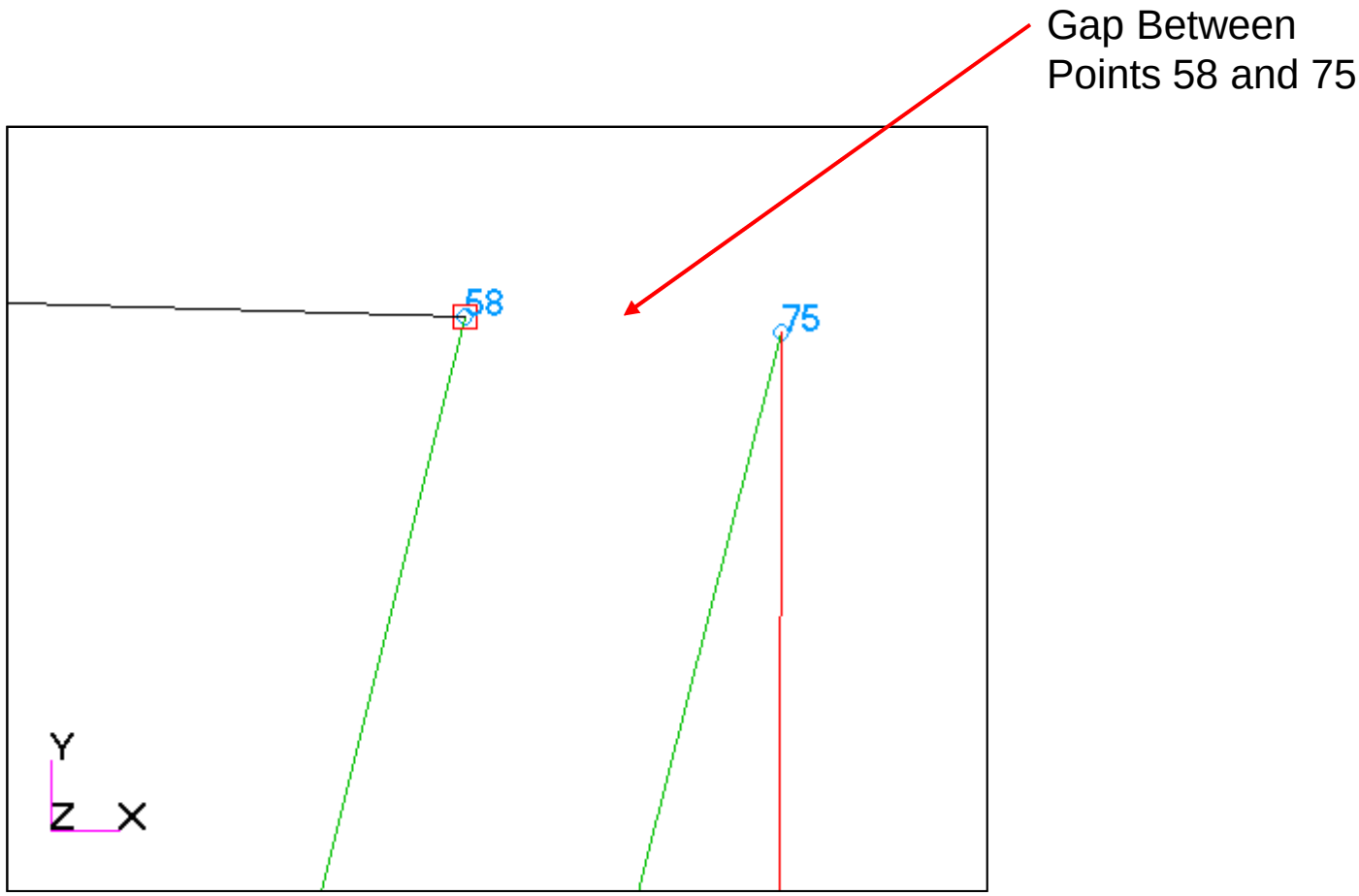
- **A message appears indicating that the boundary loop is not closed where the red square is.**



  - Click **OK**.
  - Zoom (closely) into the red square.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

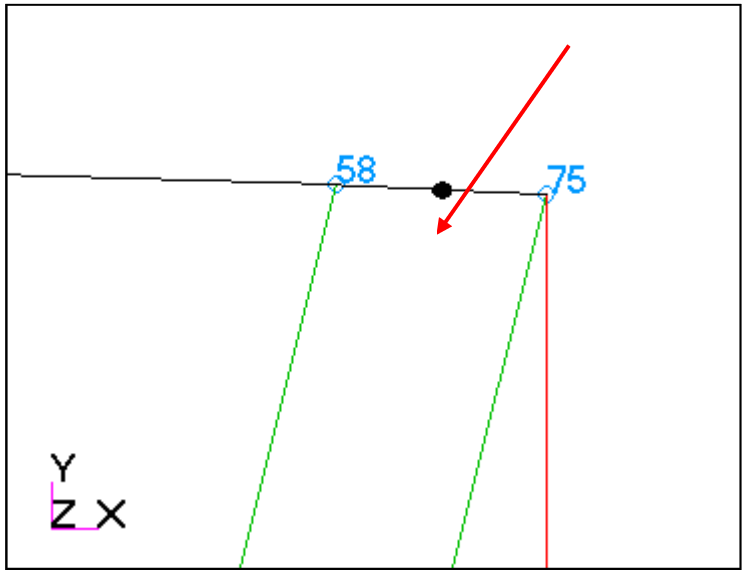


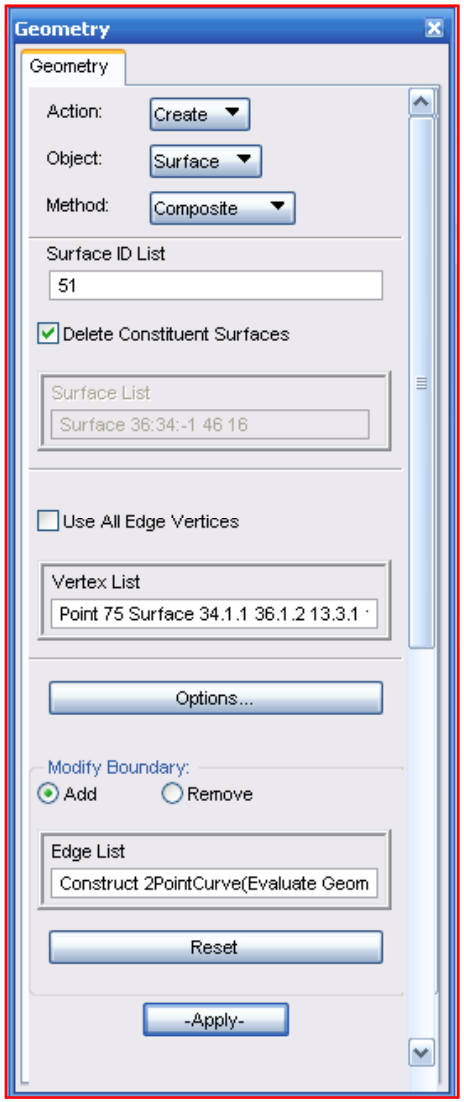
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Close the gap by adding a curve between the two vertices (points):**
  - Click **Add.**
  - Select the **Two Points** picking icon.
  - Select the two vertices (points).

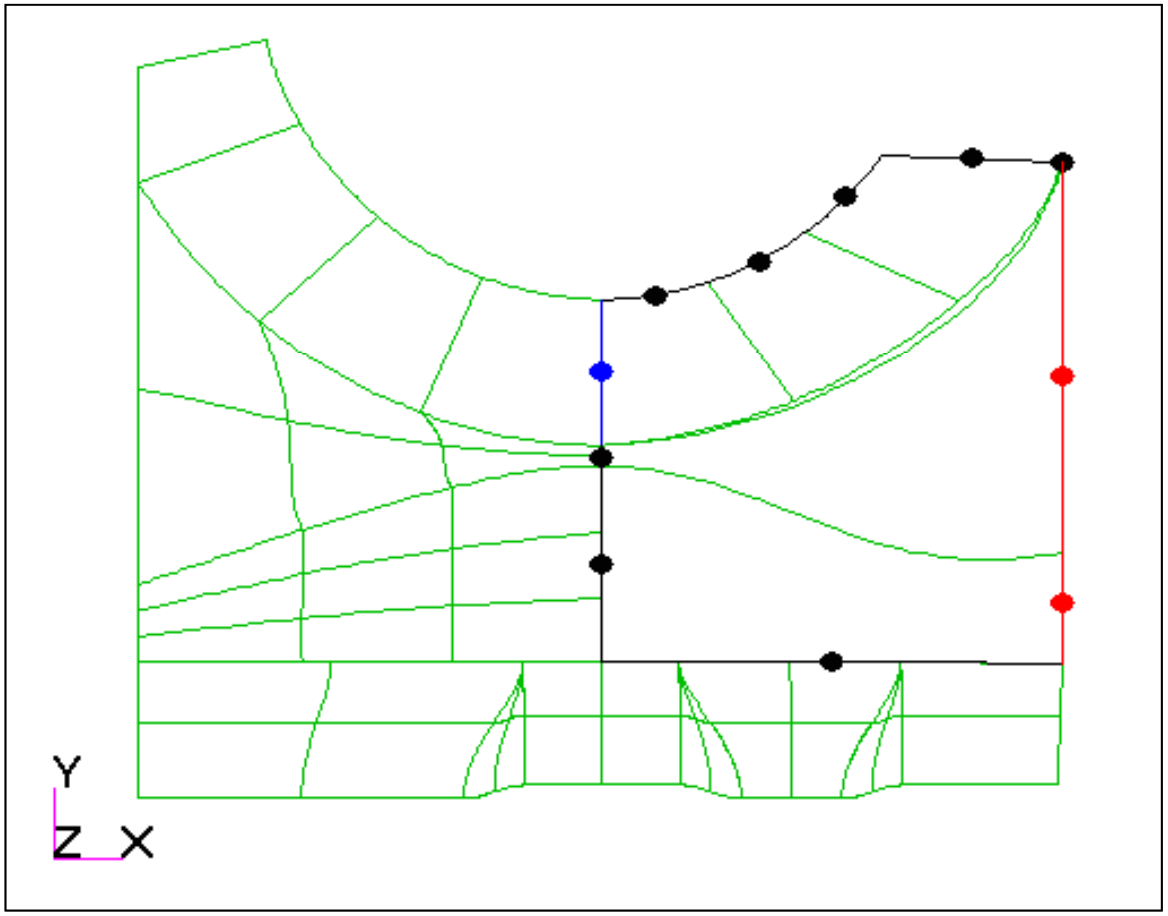
- **A curve is created (on the ‘fly’) between the two points.**
- **Go to Fit View.**





Copyright© 2012 MSC.Software Corporation

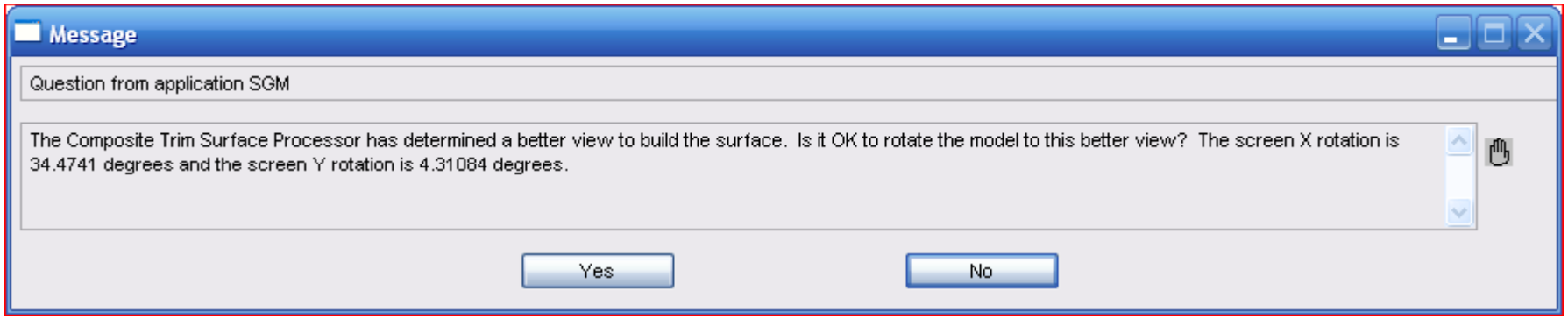
CASE STUDY: CREATE COMPOSITE SURFACE



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Click Apply.**
- **A message appears that says there is a better view to use for creation of the composite surface.**

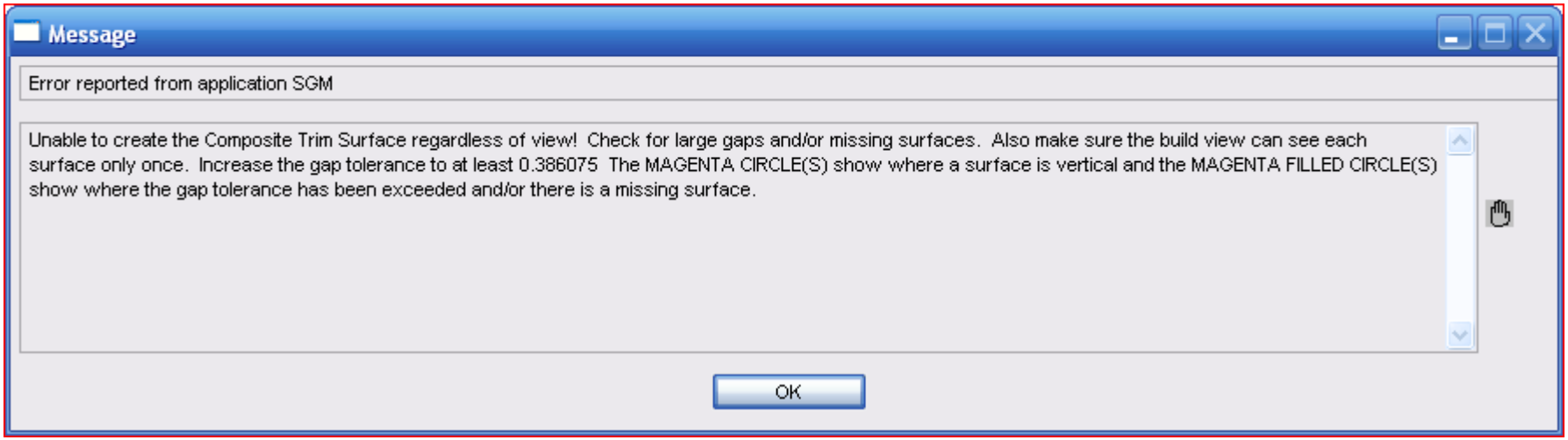


- **To create a composite surface all the surfaces in Surface List must be projected onto a plane. The view of the projection from each surface must be unobstructed by all other projections. Allow Patran to select the optimal view for projecting the surfaces. This issue will be discussed later.**
- **Click Yes.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **A message appears that says a composite surface cannot be created regardless of the view. Increase the gap tolerance to at least 0.386**

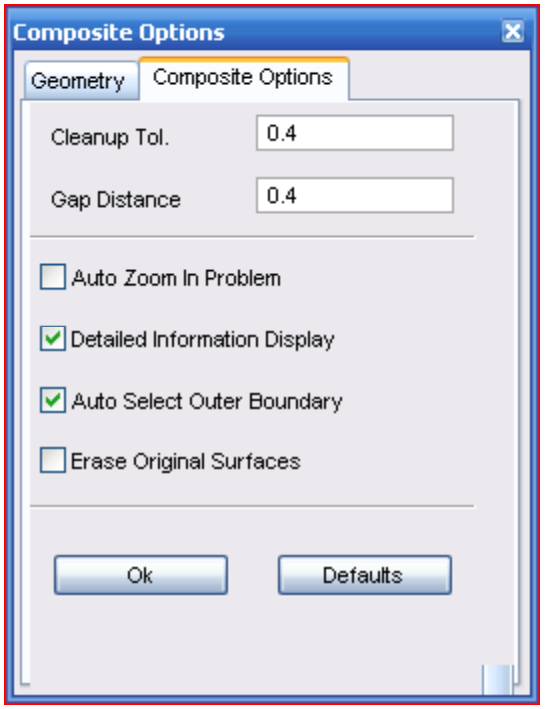


- **Click OK.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

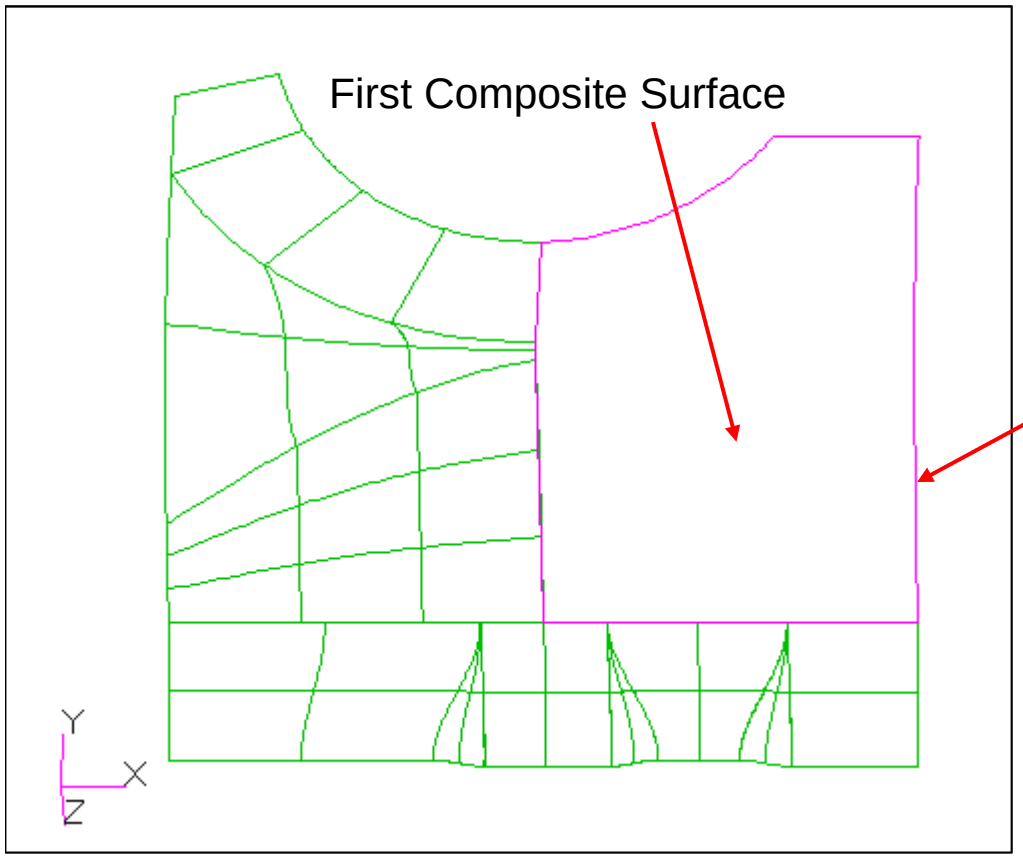
- **Options…**
  - Click **Options…**
  - Increase the *Cleanup Tol.* and *Gap Distance* to **0.4**.
  - Click **OK**, then **Apply**.
- **Reply Yes to Deleting the original Surfaces.**
- **Refresh the graphics.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **The model should now appear as follows:**

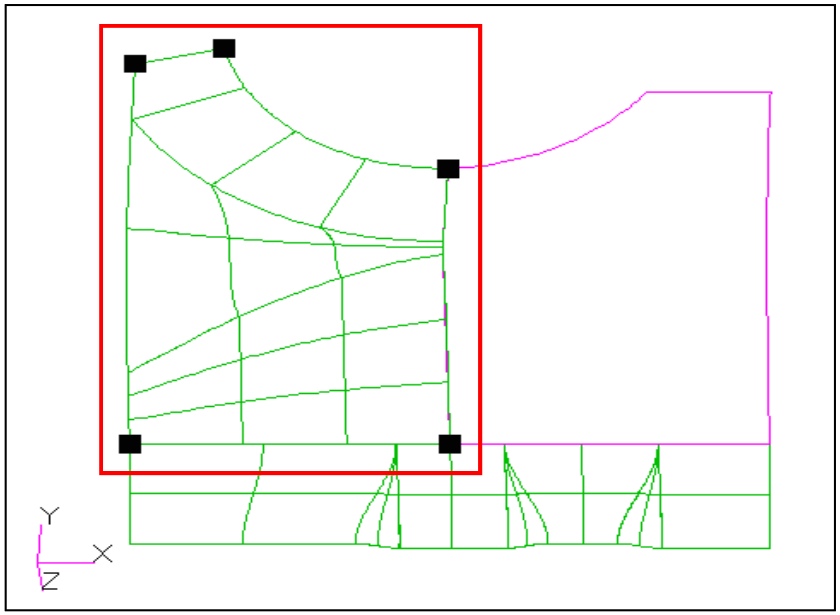


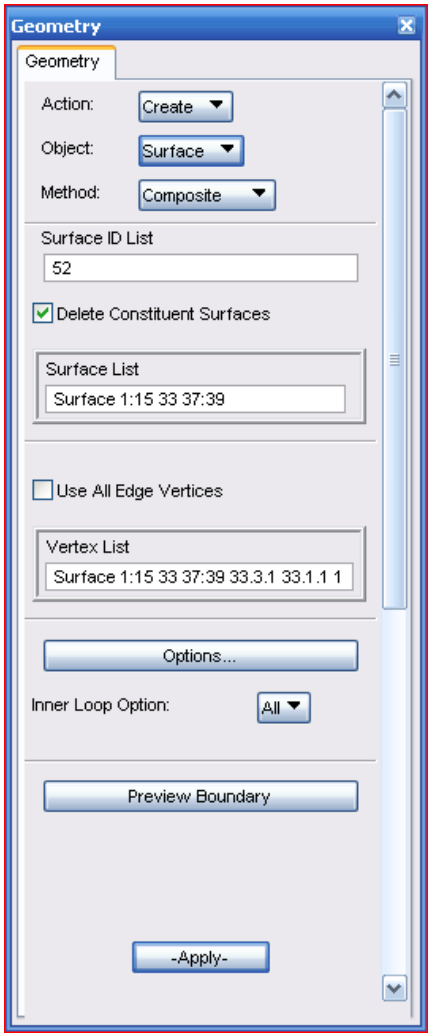
Magenta edges indicate that the surface has more than 4 edges

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Create a composite surface for the surfaces in the upper left quarter of the model**
  - Select surfaces in the upper left quarter of the model for Surface List.
  - Select the five points (as indicated), to define the outside corners of the composite surface, for Vertex List.

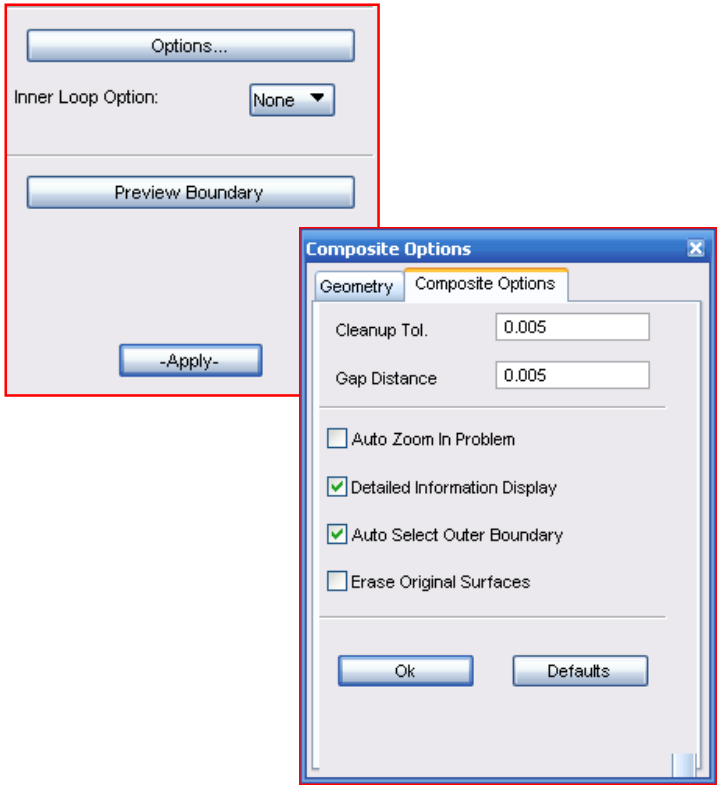


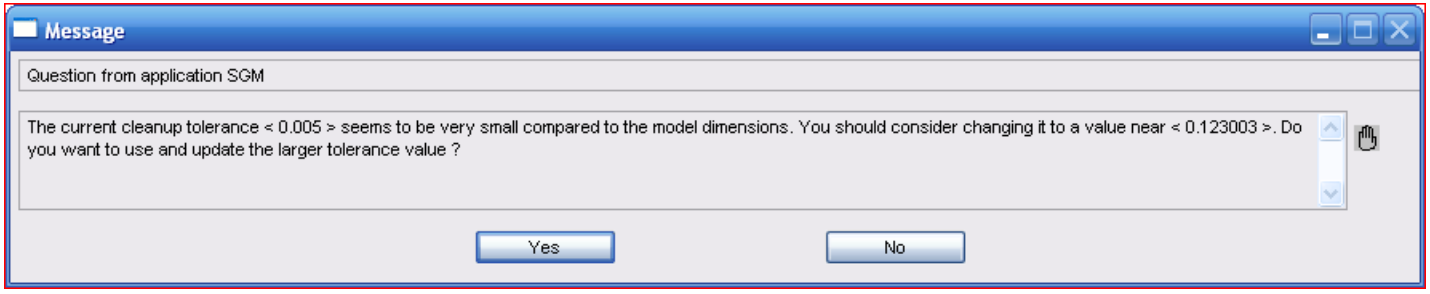


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Continue to create a composite surface for the surfaces in the upper left quarter of the model**
  - Click **Options…**
  - Click **Defaults**, then **OK.**
  - Click on **Preview Boundary**.
  - When the tolerance question appears reply **Yes.**
  - Click **Apply**.

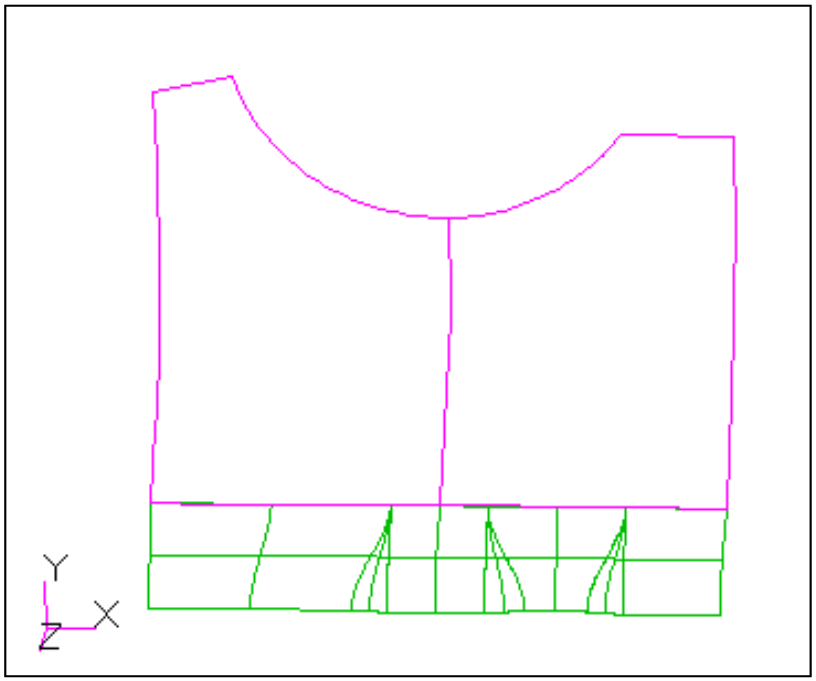




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

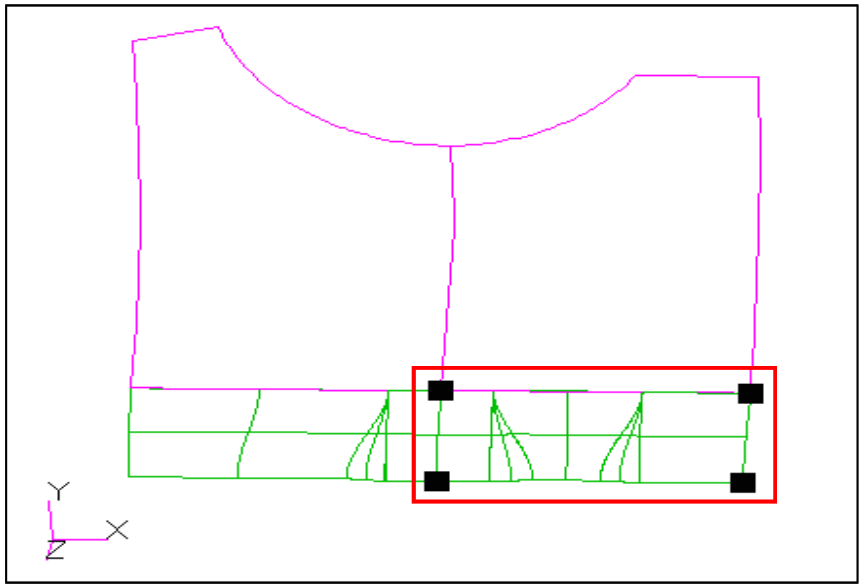
- **Reply Yes to having Patran change to the optimal view for projection.**
- **Another message appears, asking to delete the original surfaces. Click Yes.**
- **Refresh Graphics.**

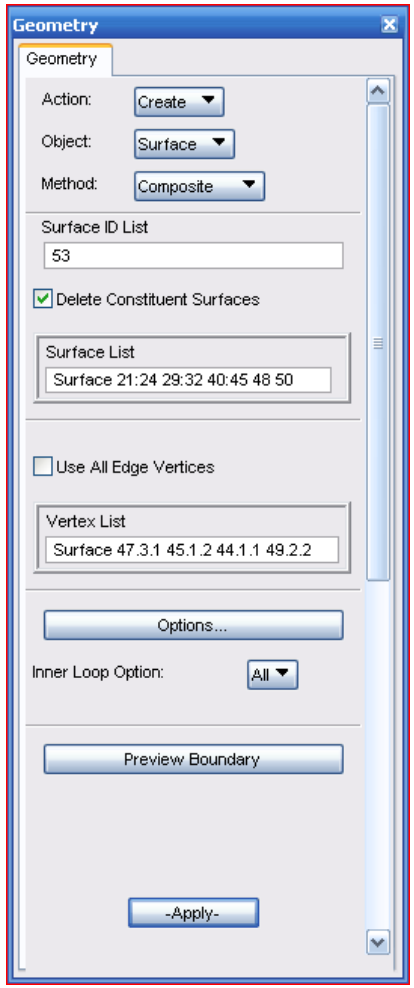


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Create a composite surface for the lower right quarter of the model**
  - Select the surfaces in the lower right quarter of the model for *Surface List*.
  - Select the four points indicated, for *Vertex List*.

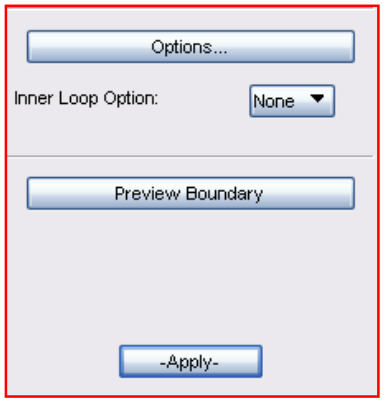


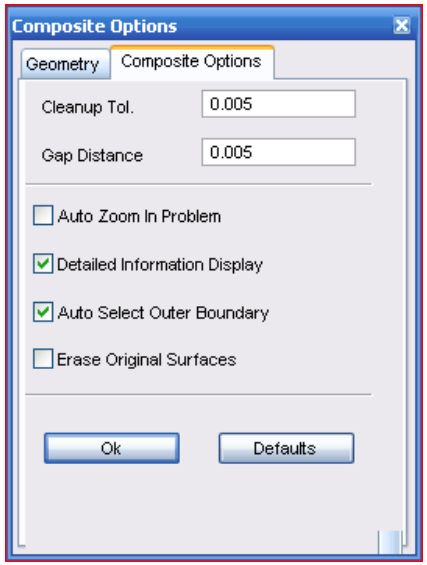


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Continue to create a composite surface for the lower right quarter of the model:**
  - Click on **Options…**
  - Click **Defaults**, then **OK.**
  - Click on **Preview Boundary.**
  - Click **Yes** when asked to change the tolerance.
  - Zoom into the lower right corner of the model.

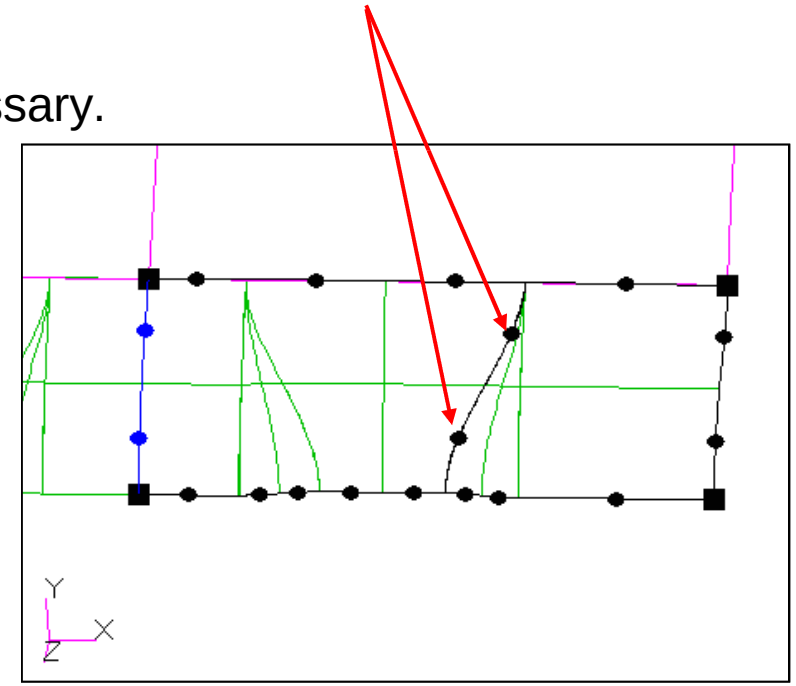




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

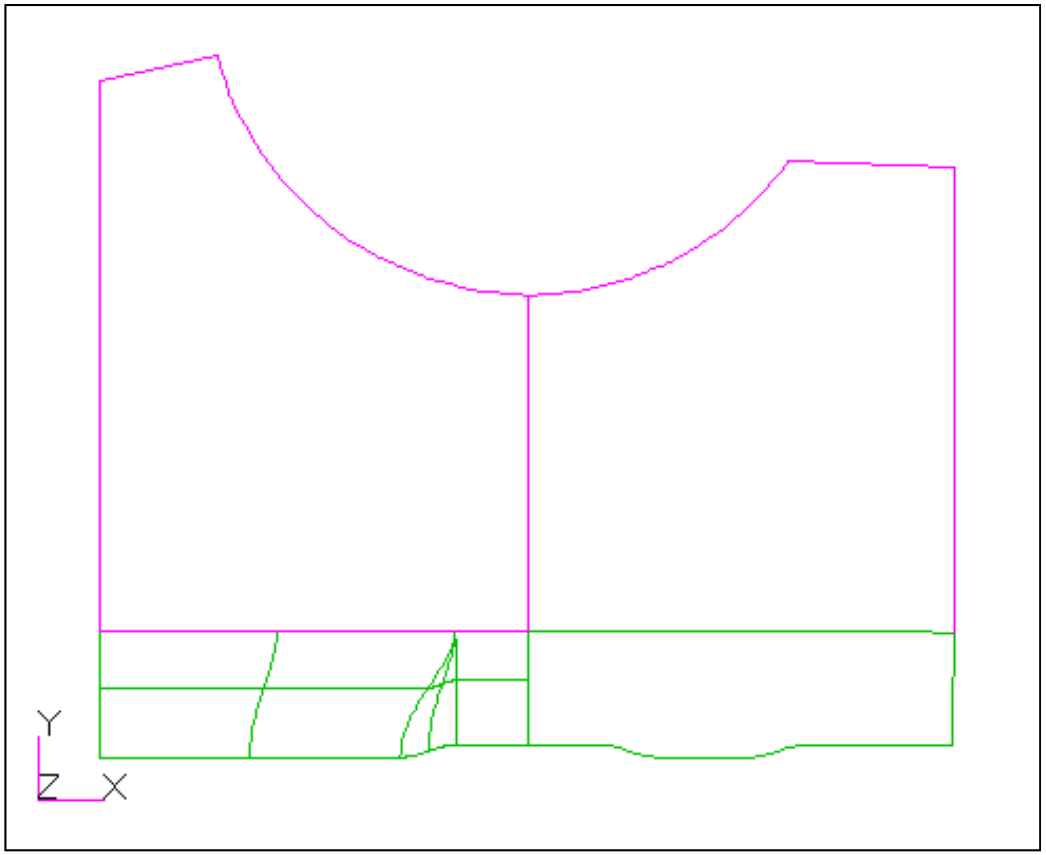
- **Continue to create a composite surface for the lower right quarter of the model.**
  - Click **Remove** under *Modify Boundary*.
  - Select the white edges along the internal gaps.
  - Click **Apply**.
  - Be sure to add boundaries as necessary.
  - Click **Yes** when asked to change the view.
  - Click **Yes** when asked to delete the original surfaces.
  - Refresh the graphics.
  - Go to **Fit View**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

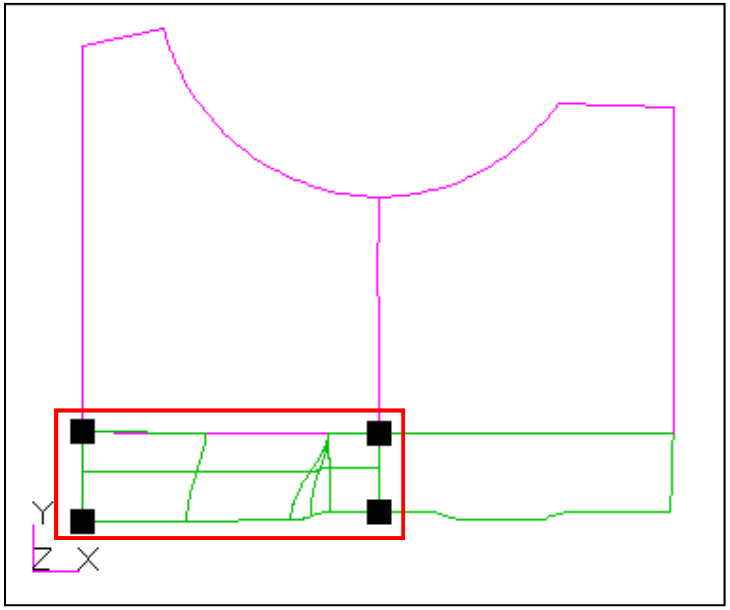
- **The model should now appear as follows:**

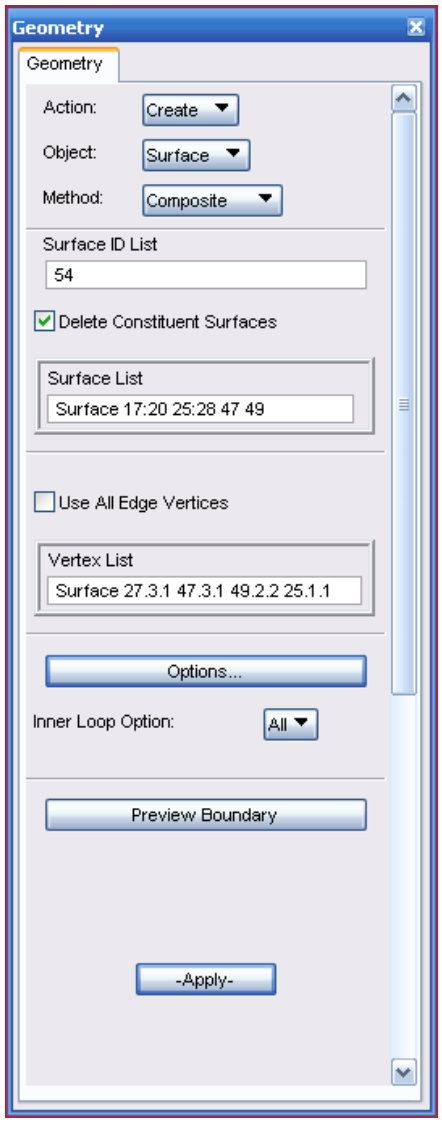


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Create a composite surface for the lower left corner of the model**
  - Select the surfaces in the lower left quarter of the model for *Surface List*.
  - Select the 4 points indicated for *Vertex List*.

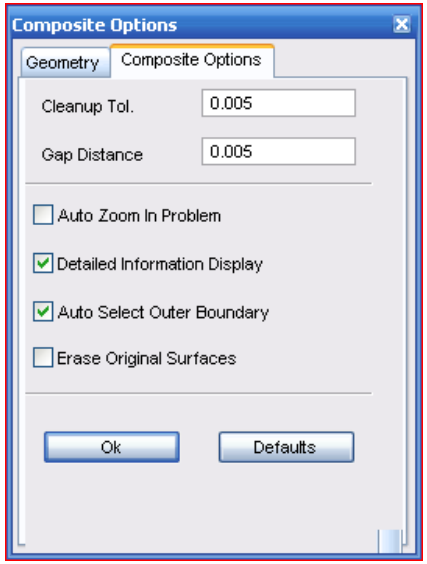




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

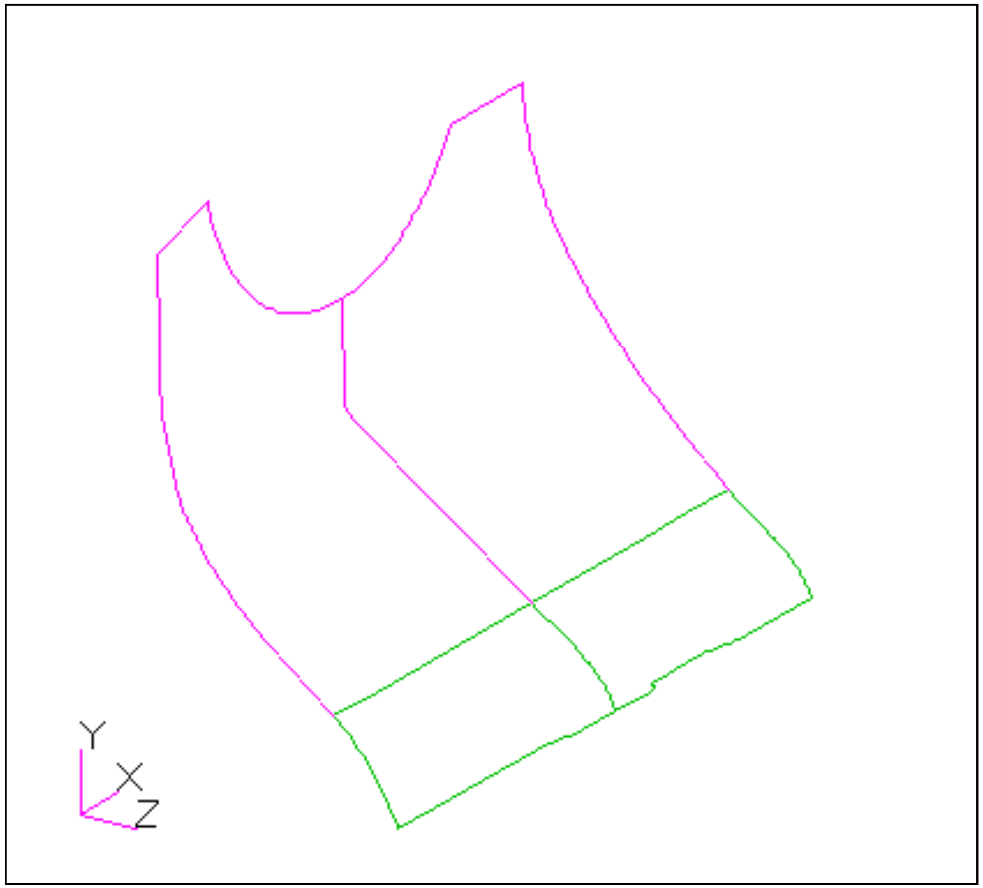
- **Continue to create a composite surface for the lower left corner of the model**
  - Click on **Options…** and select **Defaults.**
  - Click on **OK**, then select **Preview Boundary.**
  - Click **Yes** when asked about the Cleanup Tolerance.
  - (Note that there are no interior gaps.)
  - Click **Apply.**
  - Click **Yes** when asked to delete the original surfaces.
  - Refresh Graphics.
  - Change to **Iso 2View**.
  - Click **Fit View**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

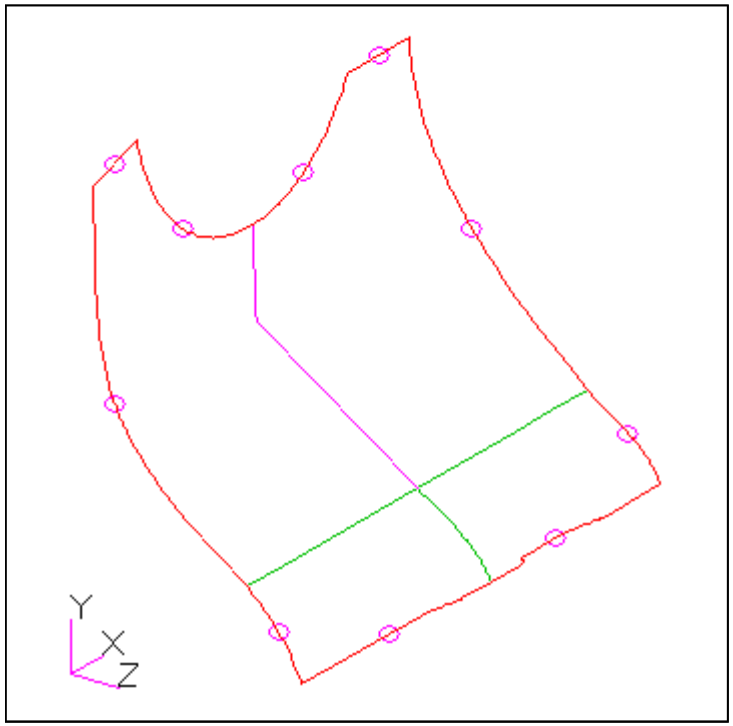
- **The model should appear as follows**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

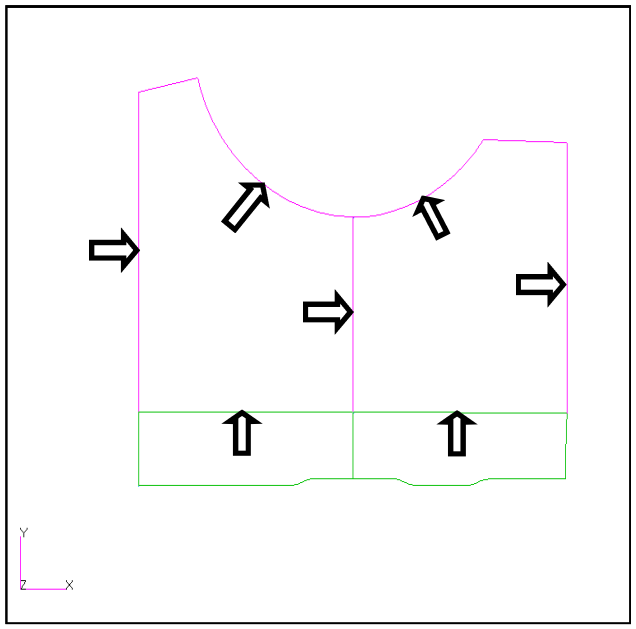
- **Check the model once again to verify that there are no geometric incongruities**
  - Under the *Geometry* tab click on **Verify**. Set *Object* and *Method* as **Surface** and **Method,** respectively.
  - Select all four surfaces on the screen for *Surface List*, and click **Apply.**
  - The four composite surfaces are congruent.
  - Click on **Update Graphics**… and **Erase Markers**.
  - Click **OK.**
  - Reset Graphics.

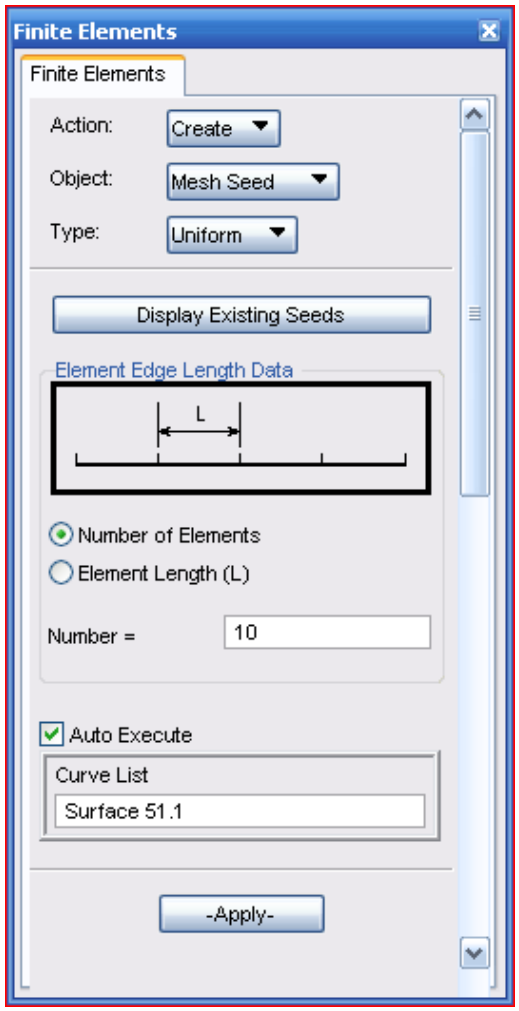


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Create mesh seeds for the model**
  - Under the *Meshing* tab click on **Uniform** in the *Mesh Seeds* group.
  - Enter **10** for *Number of Elements*.
  - Check **Auto Execute**.
  - For *Curve List*, select the 7 longest edges.

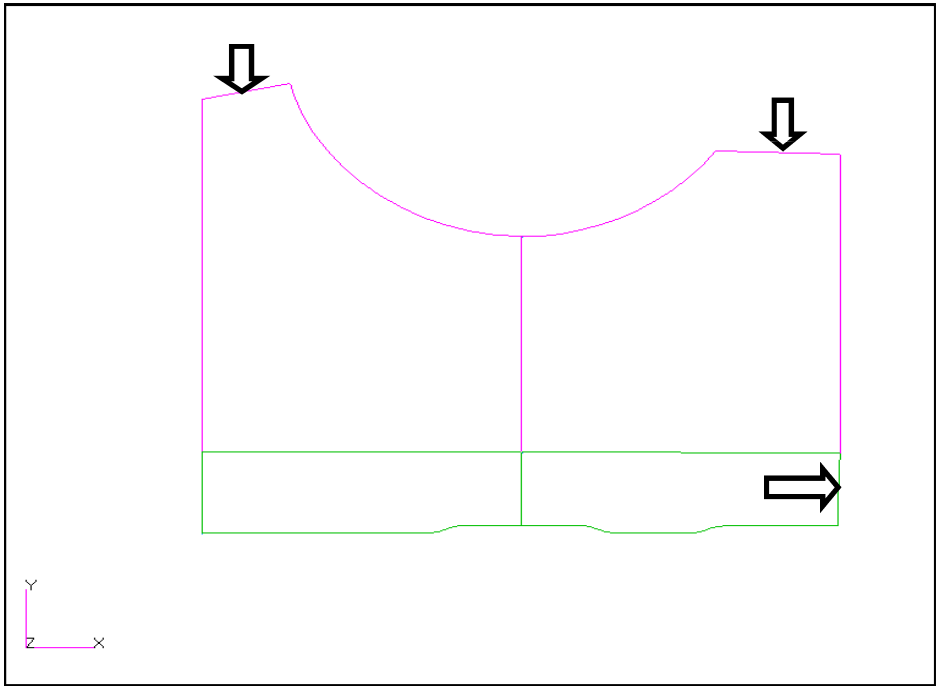


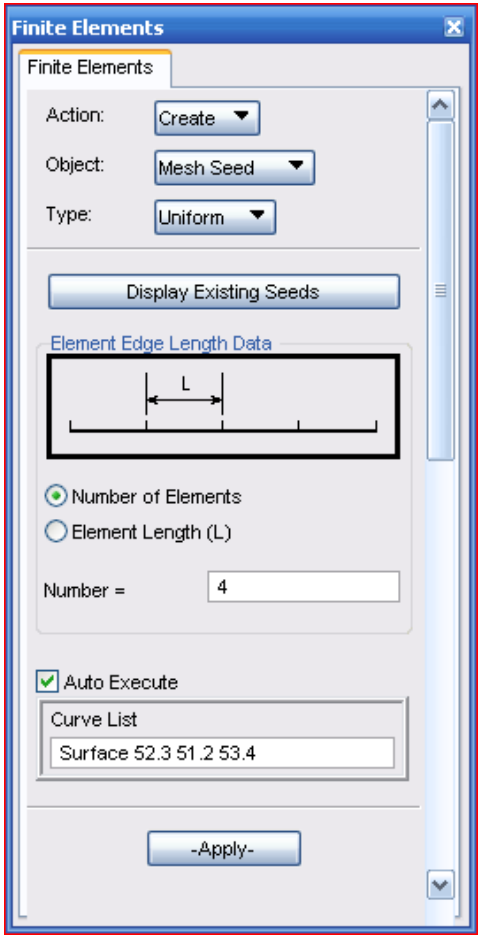


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Continue to create mesh seeds for the model**
  - Enter **4** for *Number of Elements*.
  - Select the following surface edges for *Curve List*.

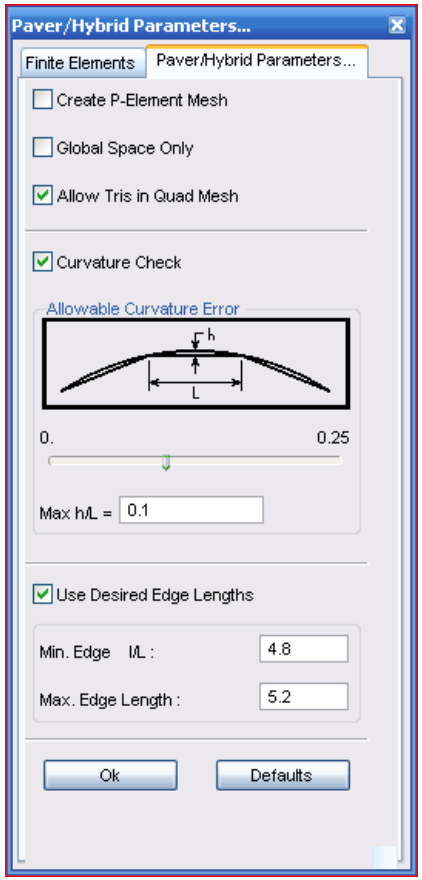


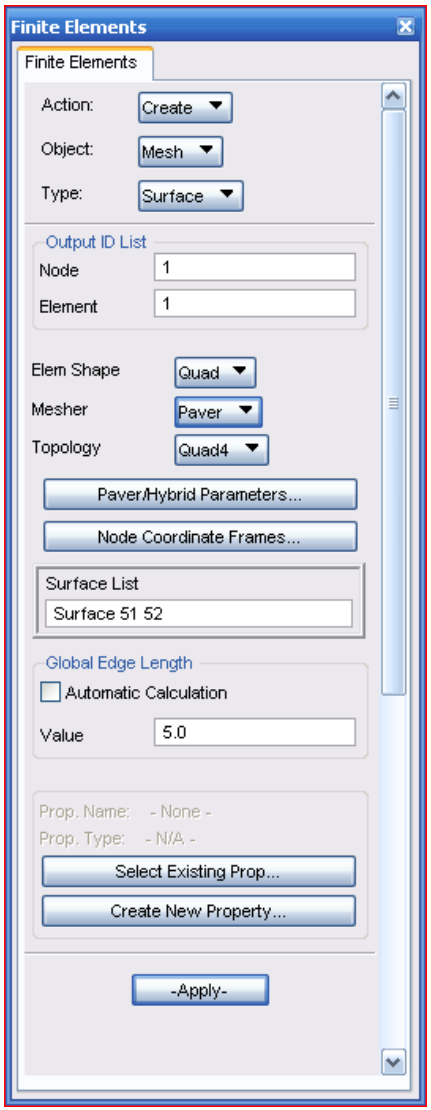


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Create the mesh for the model**
  - Under the *Meshing* tab click on **Surface** in the *Meshers* group.
  - Select **Paver** for *Mesher.*
  - Select the two magenta surfaces (at the top) for *Surface List*.
  - Enter **5.0** for *Global Edge Length*.
  - Click on *Paver/Hybird Parameters…*
  - Select **Use Desired Edge Lengths**.
  - Enter **4.8** for *Min.Edge Length*.
  - Enter **5.2** for *Max. Edge Length*.
  - Click **OK**, then **Apply**.

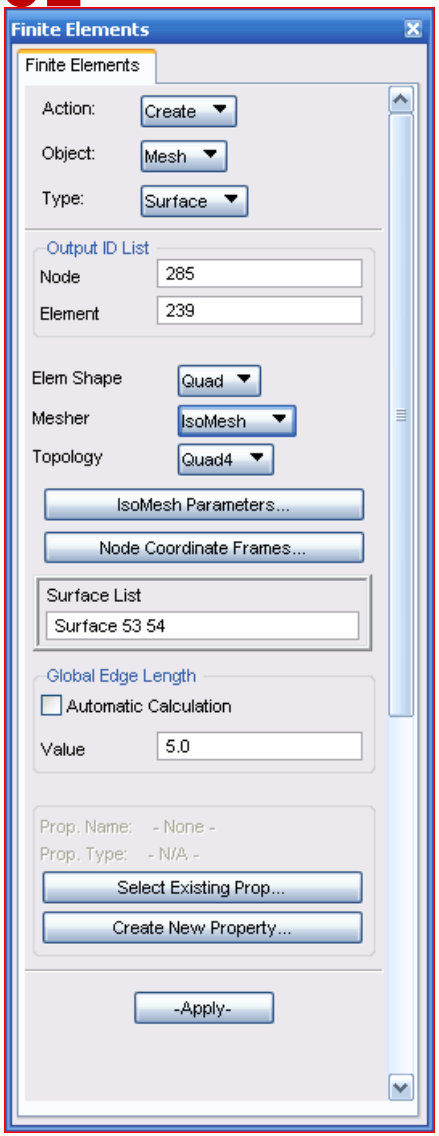




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

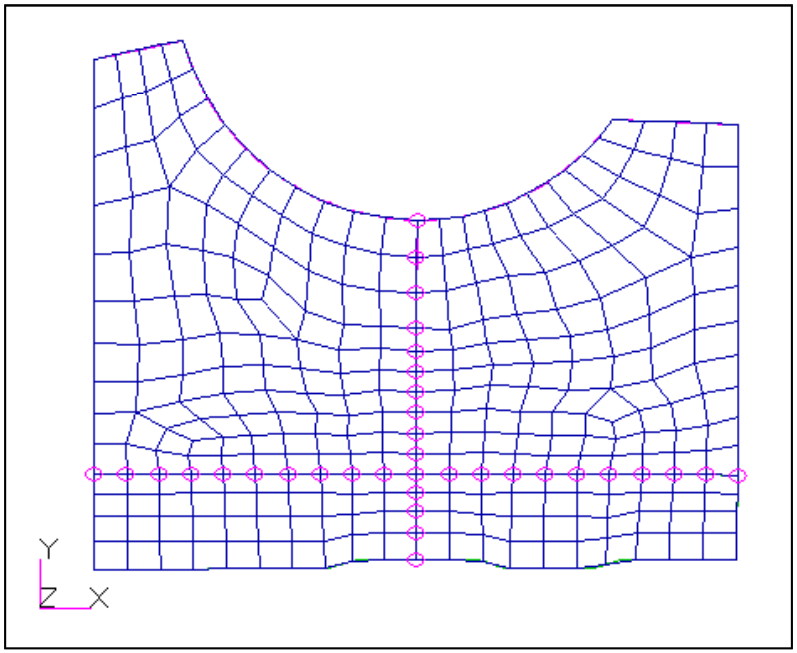
- **Continue meshing the composite surfaces. Use the IsoMesh for the two green surfaces (at the bottom).**
  - Select **IsoMesh** for *Mesher*.
  - Select the two bottom surfaces for *Surface List*.
  - Click **Apply**.

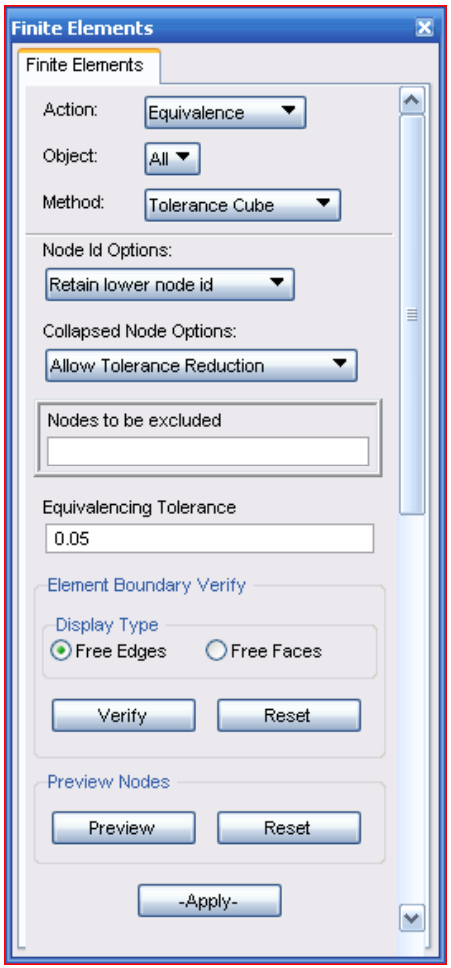


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

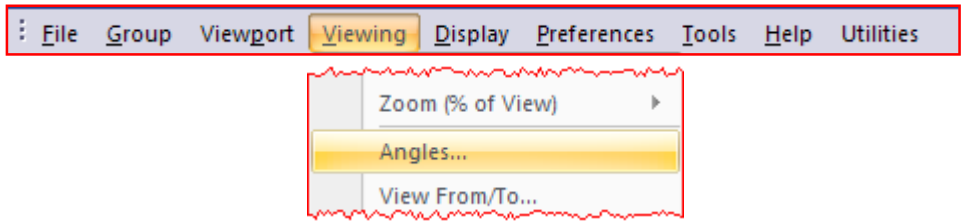
- **Equivalence the mesh**
  - Under the *Meshing* tab click on **Equivalence** in the *FEM Actions* group.
  - Change the *tolerance* to **0.05**, instead of using the default value of **0.005**.
  - Click **Apply**.



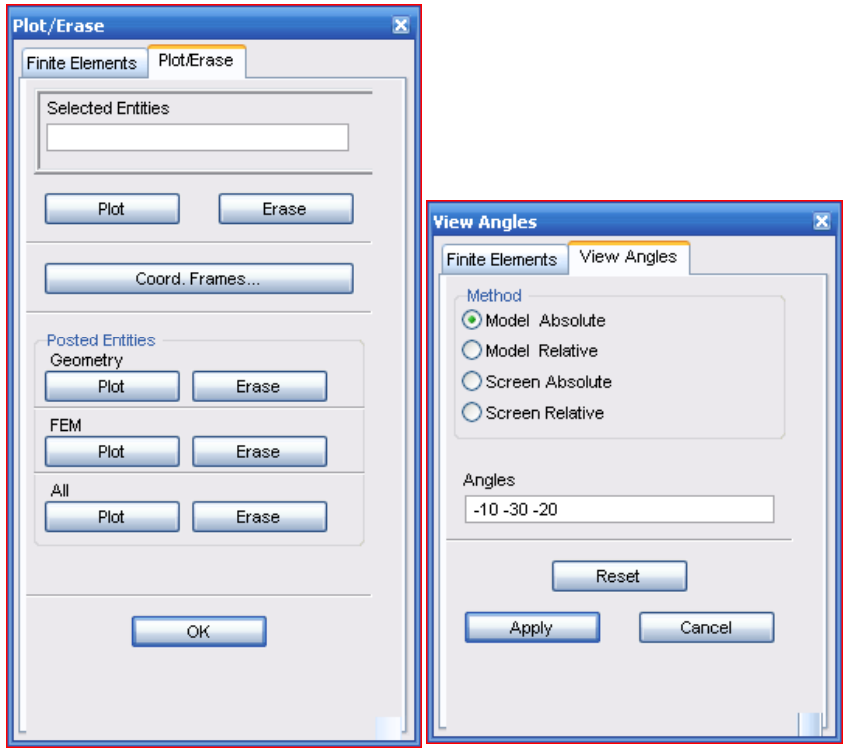


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

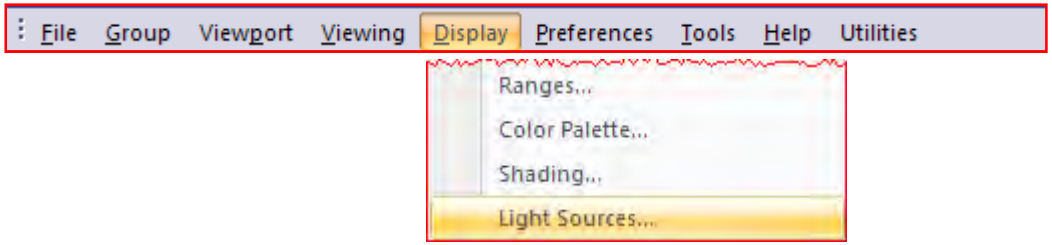


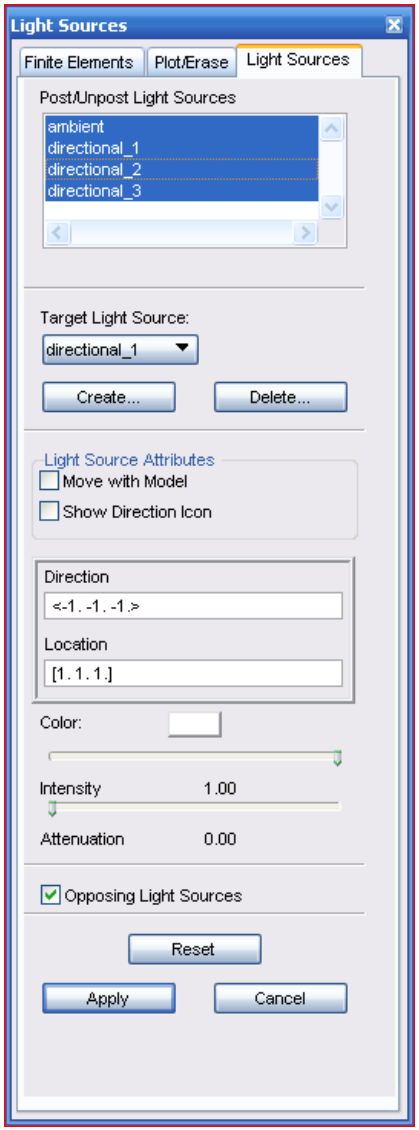
- **Change the display**
  - Change to **Smooth shaded**.
  - Select **Plot/Erase…** under the *Display* menu
  - Click **Erase under Geometry**, and click **OK.**
  - Select **Angles…** under the *Viewing* menu
  - Enter **-10 -30 -20** for *Angles*.
  - Click **Apply**, then click **Cancel**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE



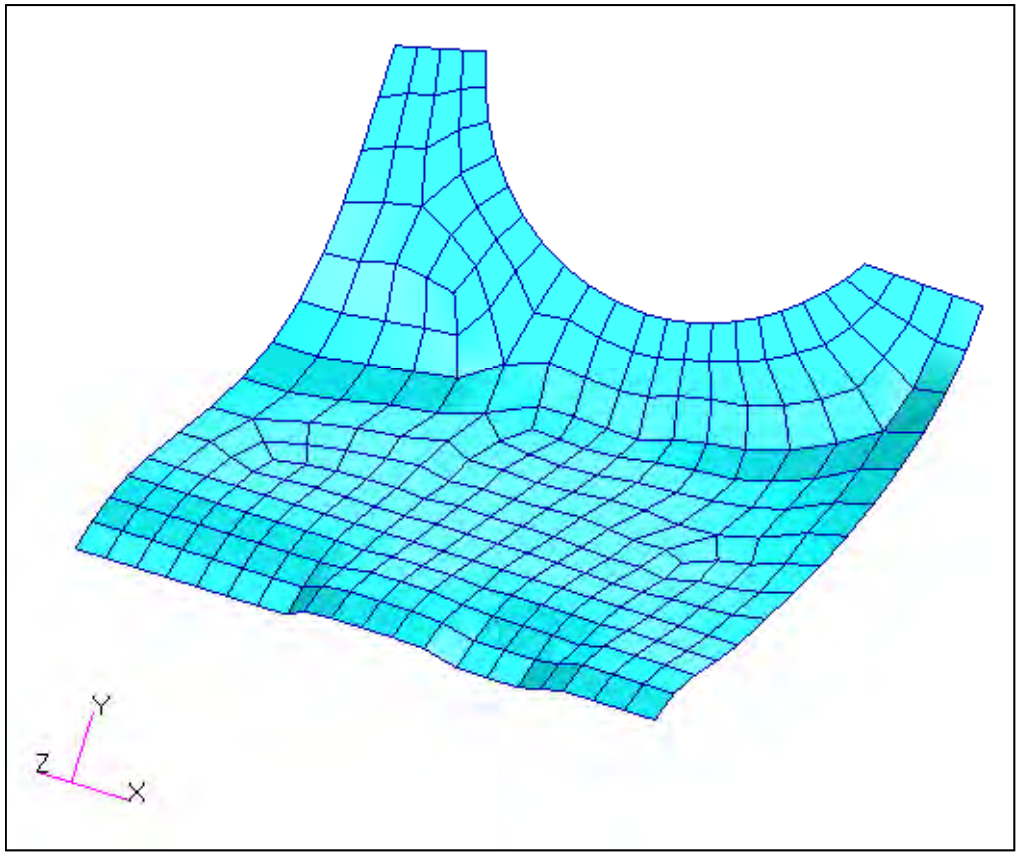


- **Change the lighting**
  - Select **Light Sources…** under the *Display* menu
  - Select **ambient**, **directional_1**, **directional_2**, and **directional_3**.
  - Click **Apply**, then click **Cancel**.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

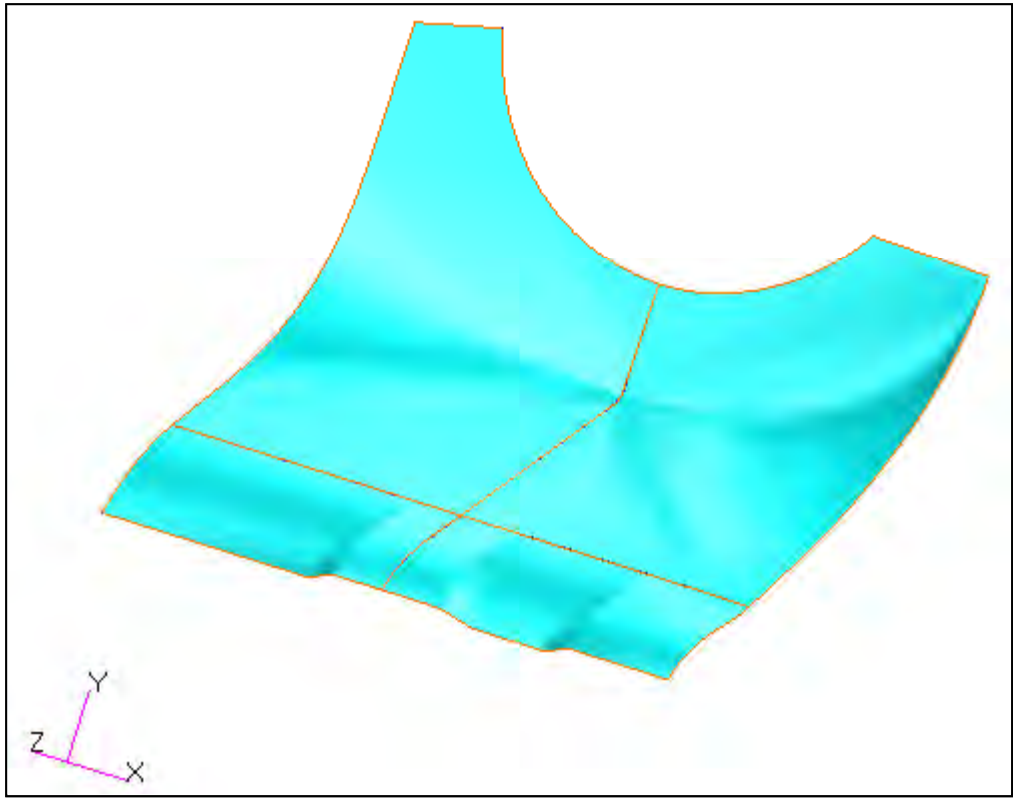
- **This is the final 2D quad mesh for the composite surfaces**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

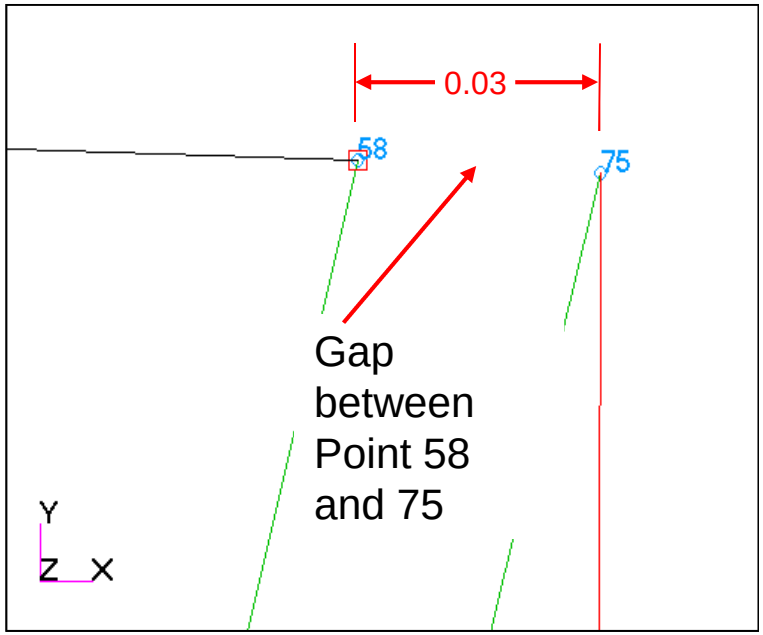
- **The four composite surfaces are displayed below:**
  - Click on the **Plot Erase** icon.
  - **Erase** the *FEM*, and **Plot** the *Geometry*.
  - Click **OK**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

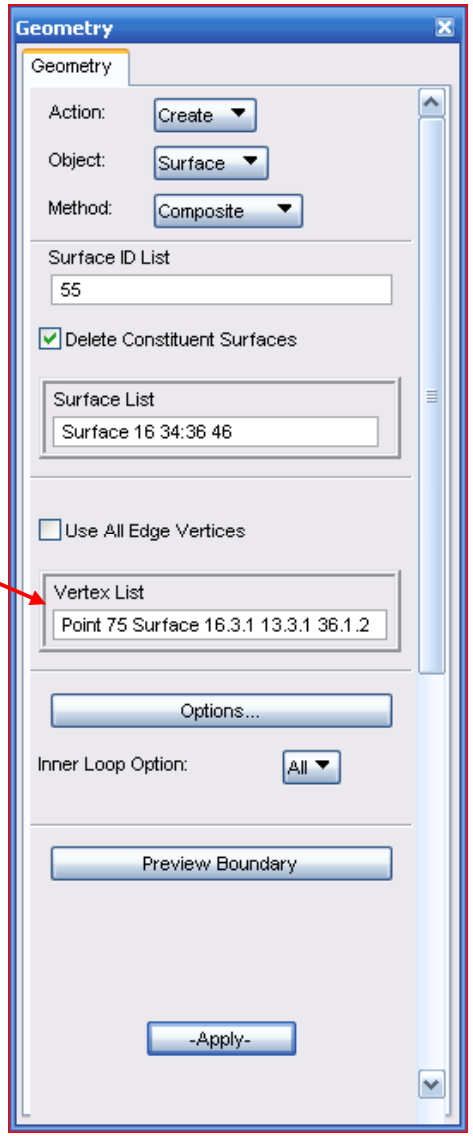
- **Previously, it was noticed that there are two vertices (at Points 58 and 75) that are close together. What occurred at import ?**
  - During the database creation the tolerance was set to 0.005 (=10.0/2000.0).
  - The vertices (in the IGES file) were separated by 0.03.
  - At import of the surfaces the tolerance (0.005) < gap(0.03), so the two vertices were not “equivalenced”, and a point was created at each vertex.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Previously, when the composite surface in the upper right corner was created, five vertices were selected. The vertex at Point 75 was used.**
- **Instead of using Point 75, Point 58 could have been used.**
- **How would using a close but different vertex affect the creation of the composite surface, with a tolerance of 0.005 ?**

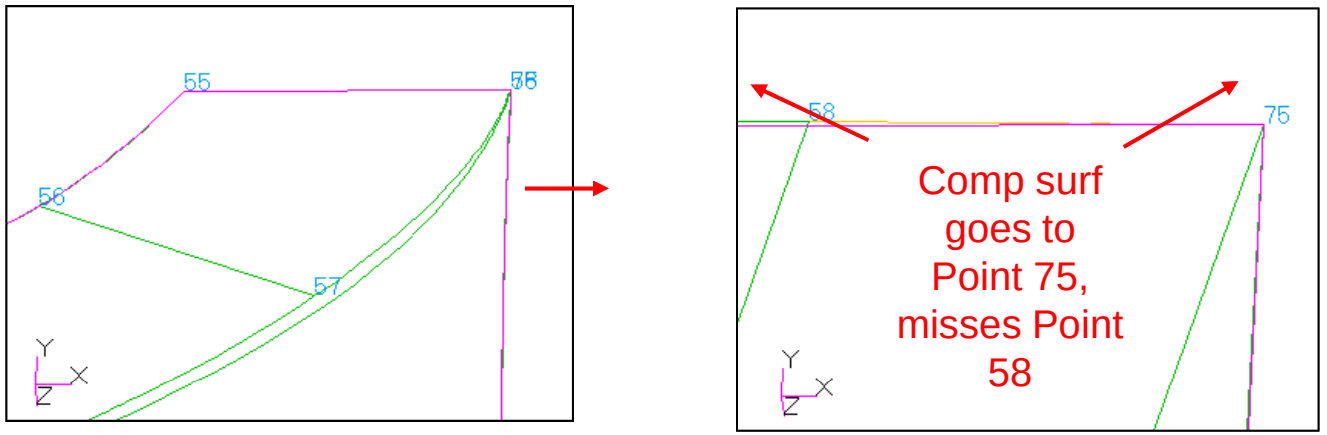


Copyright© 2012 MSC.Software Corporation

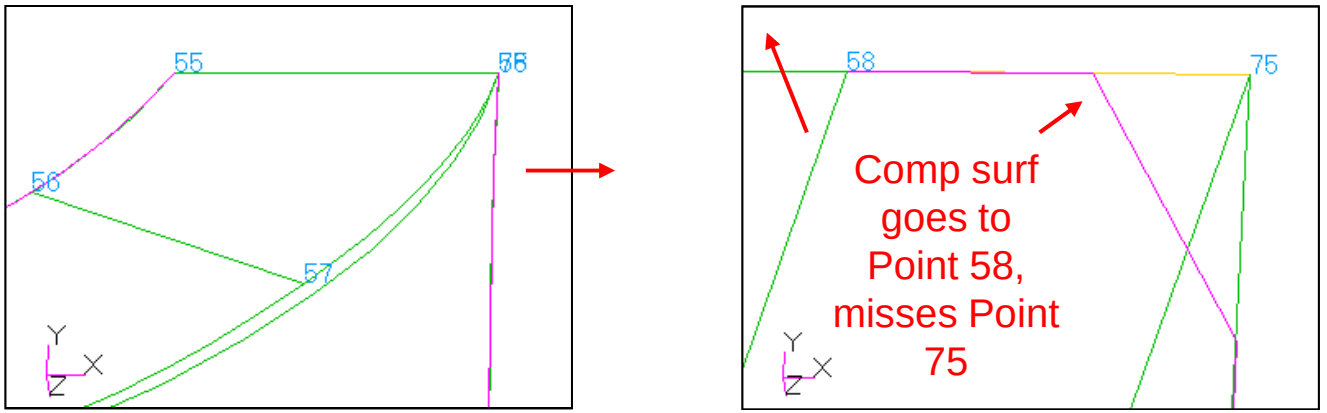
CASE STUDY: CREATE COMPOSITE SURFACE

- **Compare composite surface using either Point 75 or 58.**

Vertex at Point 75



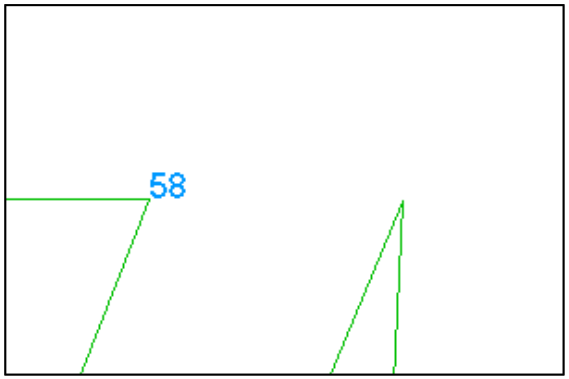
Vertex at Point 58



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

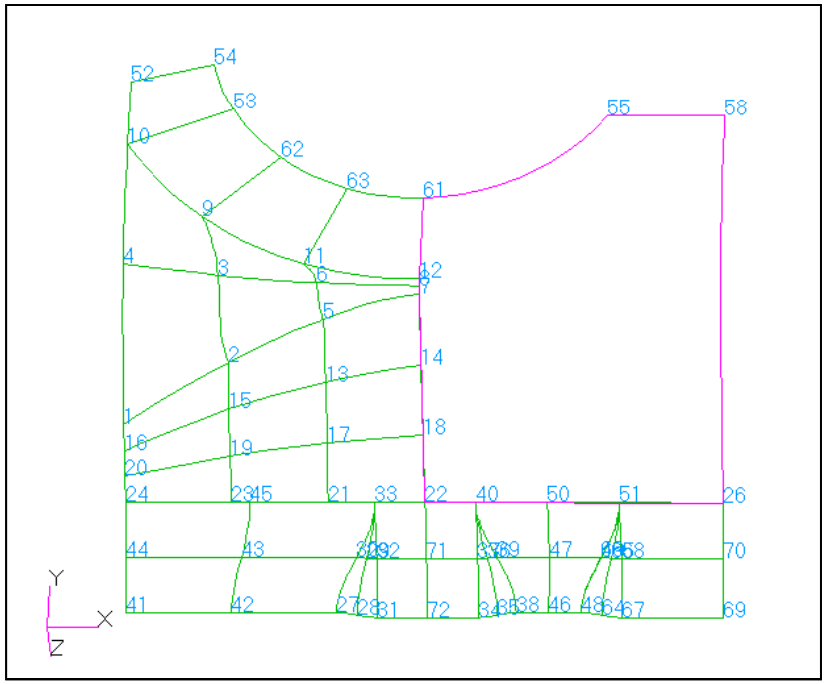
- **Re-import the IGES file with the surfaces.**
- **Use a larger tolerance – "Based on Model": Approximate Maximum Model Dimension = 100.0 → tolerance = 0.05**
- **Create a composite surface for the surfaces in the upper right corner of the model, following the same steps that were used previously.**
  - Notice that there is only one point, Point 58
  - This is because the two vertices were equivalenced during import



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

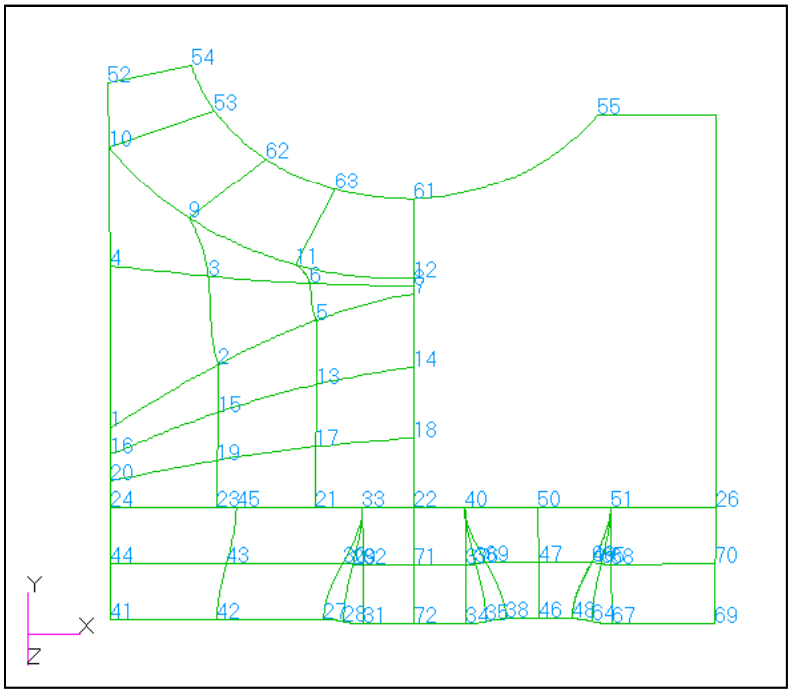
- Note that it is not necessary to create a curve between the “red square” and “red line”.
- The composite surface looks like the previous one, except for there being only one point, Point 58.
- There are some slight differences in the details at Point 58.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Create another composite surface, using only four vertices (exclude Point 58)**
  - Notice that the composite surface still goes through the area of Point 58. There are some differences in the details at Point 58.



Copyright© 2012 MSC.Software Corporation

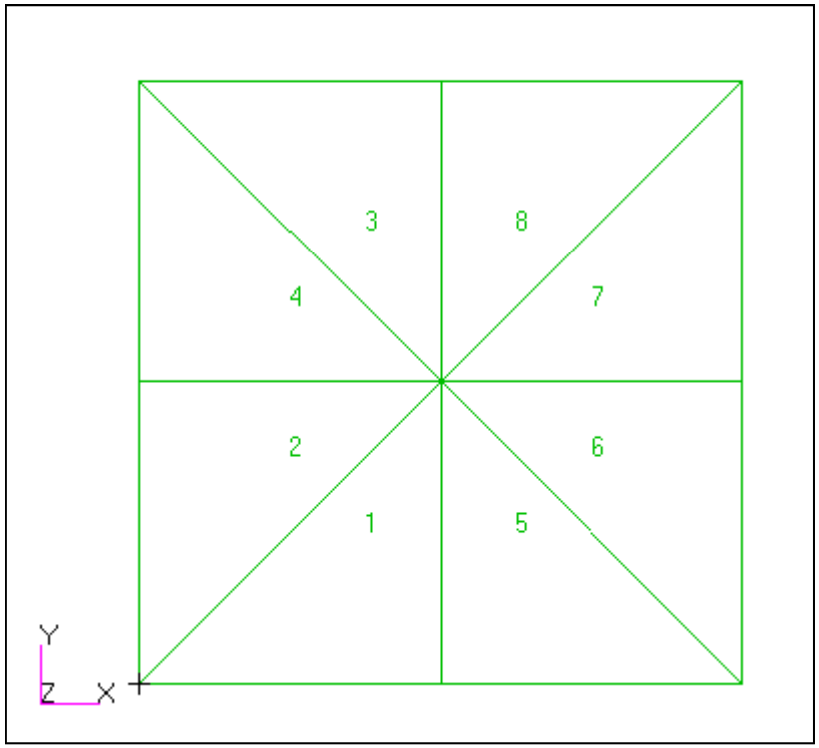
CASE STUDY: CREATE COMPOSITE SURFACE

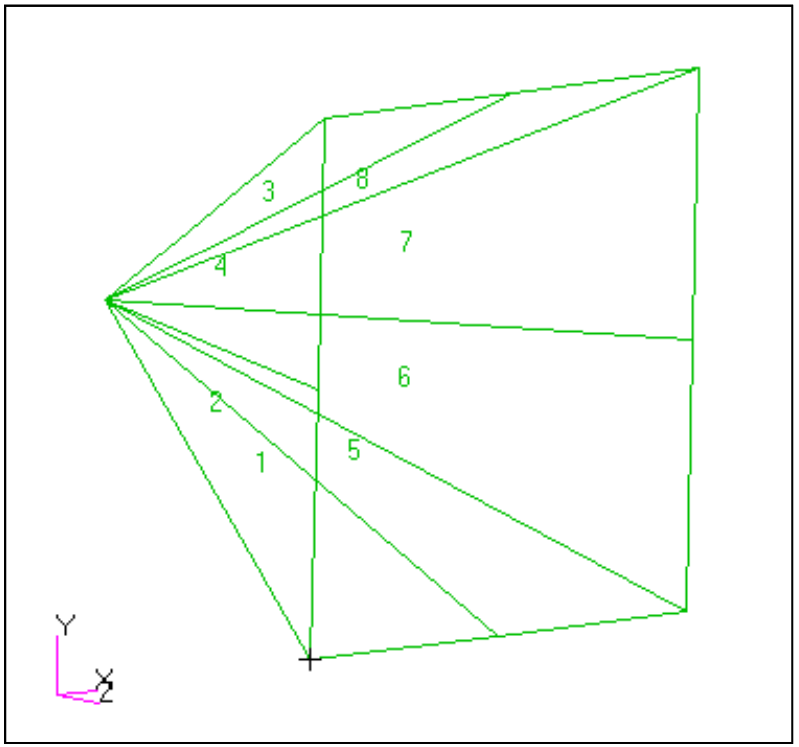
- **During creation of a composite surface, the model (surfaces in Surface List) is rotated to provide an improved view. It is done in a way so that the projection of each surface onto a plane can be seen without an overlap of the projections. This is demonstrated using a simple model consisting of eight triangular surfaces.**
- **Create a new database and import the simple surface model.**
  - Use default tolerance **0.005** (not an issue for this discussion)
  - Import neutral file **eight_tri_surfs.out**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Two views of the surfaces:**

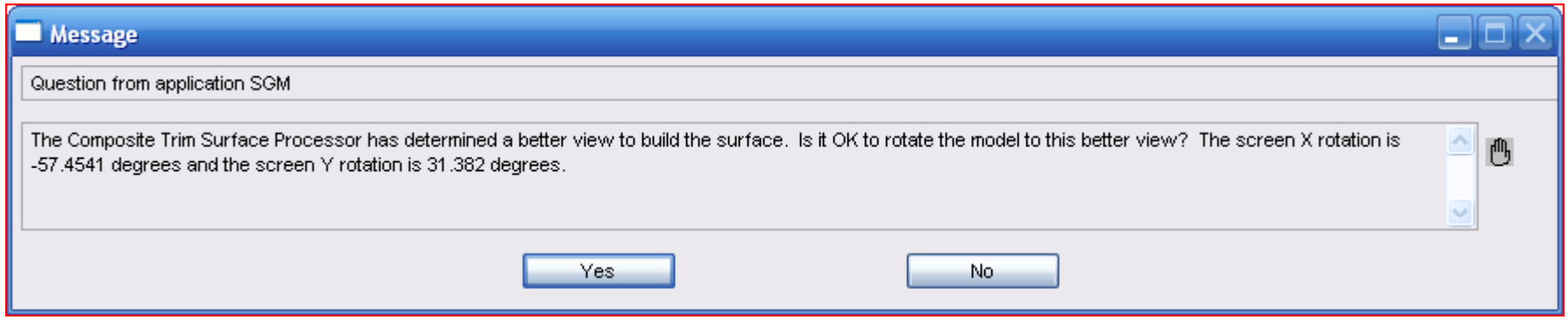




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Create a composite surface for two different views at the time of creation, e.g. … determined a better view to build the surface … Yes or No**

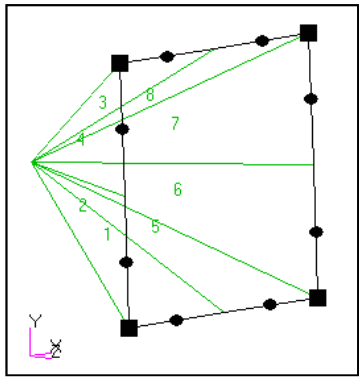


  - If **Yes**, use Patran view (optimal).
  - If **No**, use user view.

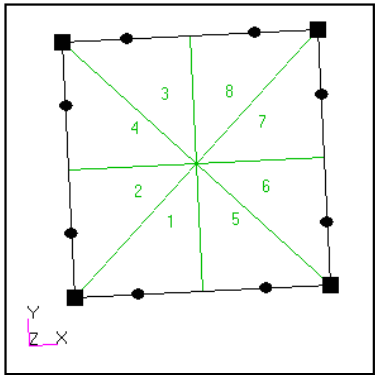
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

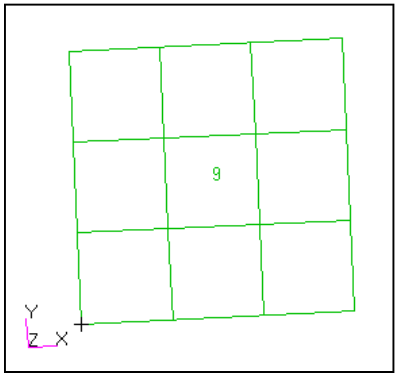
Patran Optimal View



Preview boundary

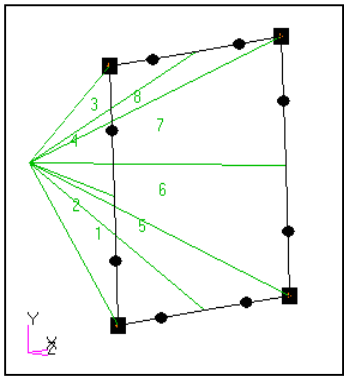


Use Patran view (**Yes**)

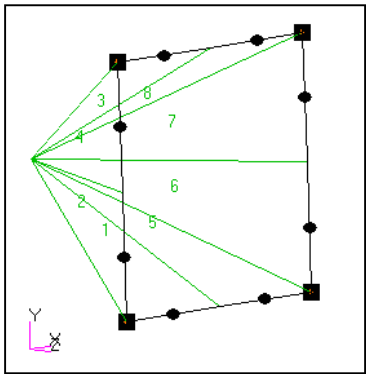


Comp surf w/ same view

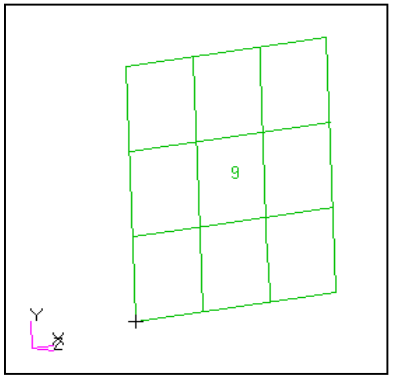
User View



Preview boundary



No Patran view (**No**)

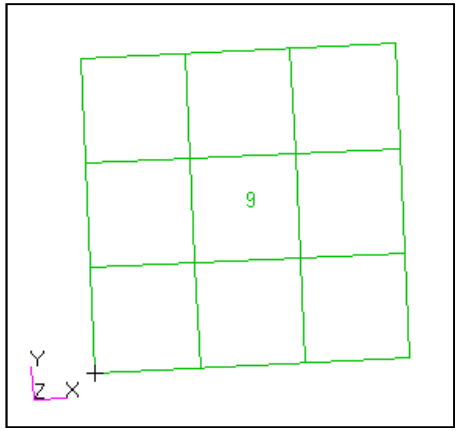


Comp surf w/ same view

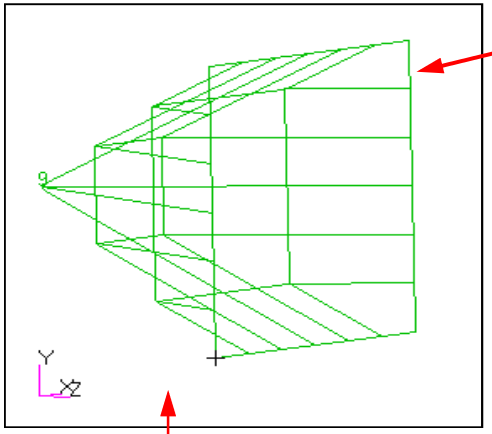
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

Patran Optimal View



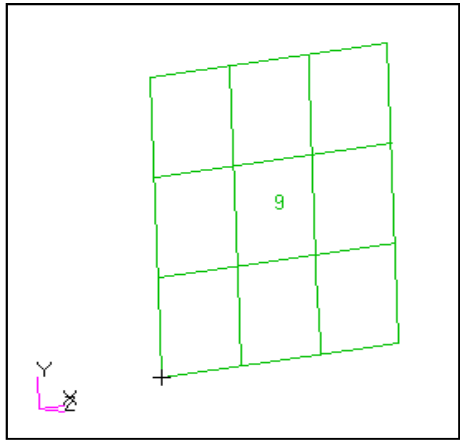
Comp surf w/ same view



Comp surf built correctly w/ good view

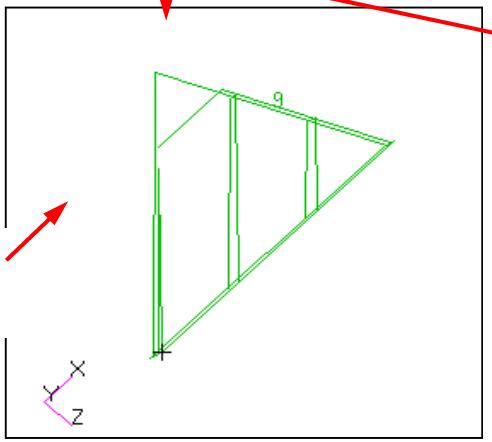
Comp surf w/ different view & 5 disp. lines

User View



Comp surf w/ same view

Missing part



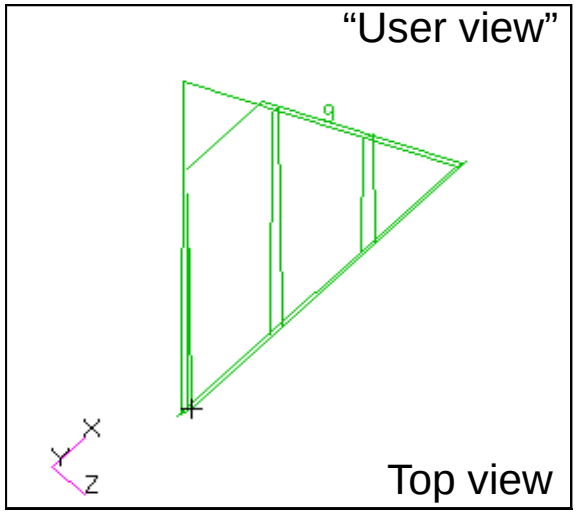
User view surfaces cut w/ plane at (0,0,0); **top view** shown

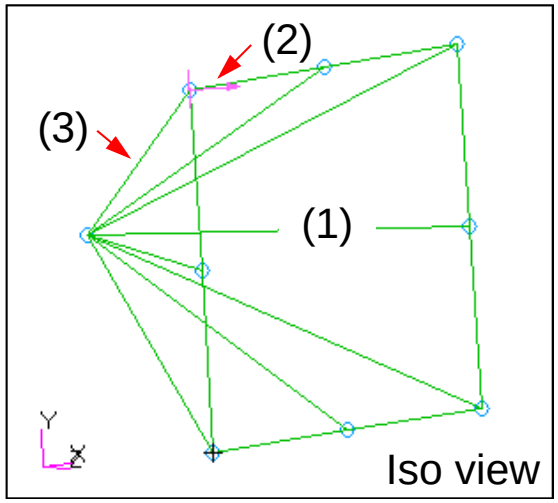
Comp surf w/ different view & 5 disp. lines

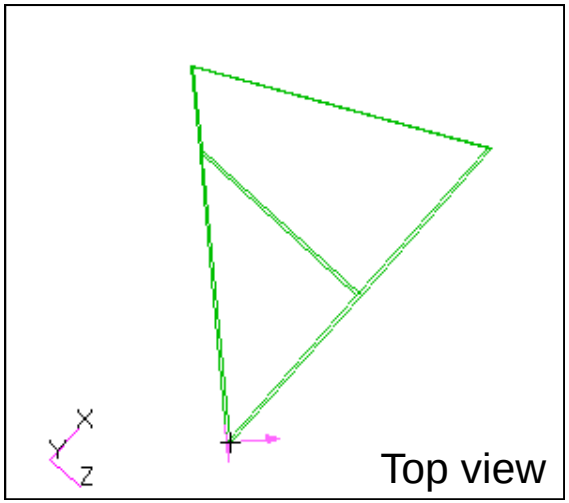
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **Break the original surfaces in Surface List (1) with a plane (2). Delete the new surfaces (3) to the left of the plane. The resulting surfaces look like the composite surface from the user view.**



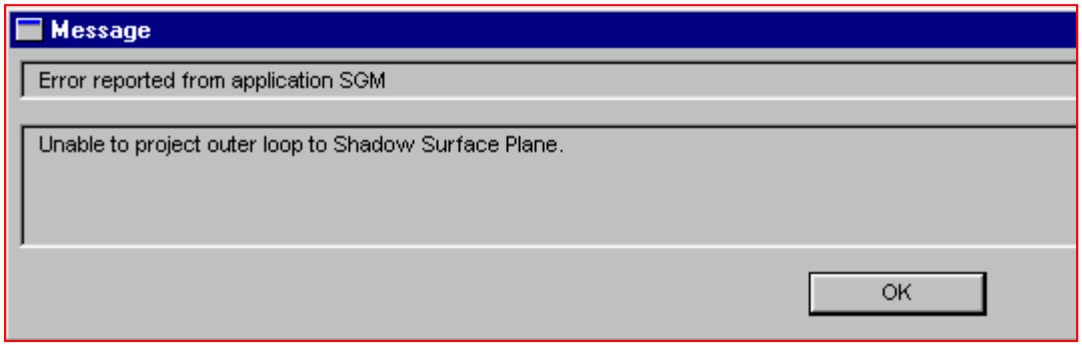




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE COMPOSITE SURFACE

- **If there are overlapping surfaces, a possible error message may be:**



- **This indicates that all the surfaces cannot be projected without any overlapping. The shadow surface plane is the plane that the surfaces are projected onto.**

Copyright© 2012 MSC.Software Corporation

EXERCISES

- Perform Workshop 2 “Composite Surfaces” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

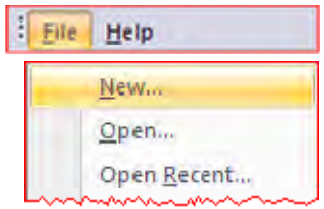
CASE STUDY: TRIMMED SURFACE, COMPOSITE

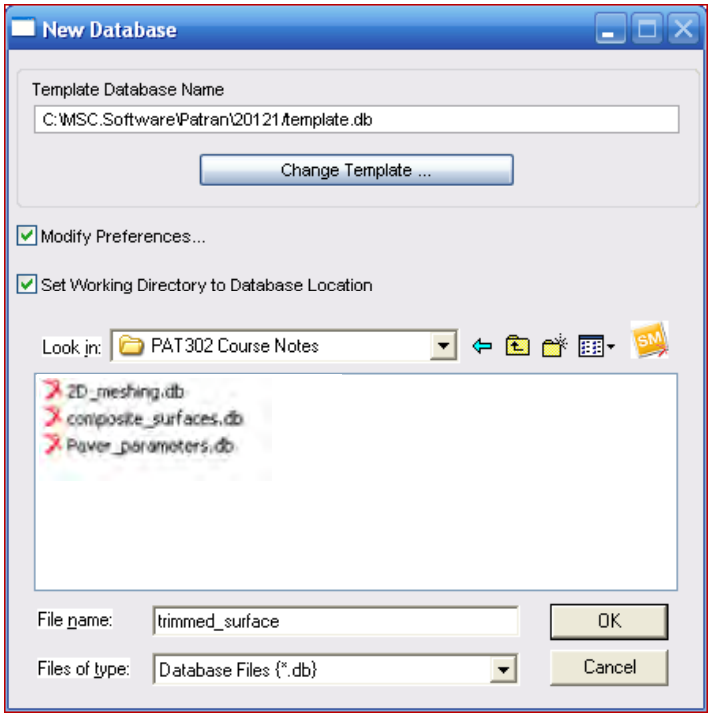
- **Create a trimmed composite surface using**
  - Under the *Geometry* tab, click **Trimmed** in the *Surfaces* group.
    - Select **Composite** for *Option*

Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

- **Create a new database**
  - Select **New** from the *File* menu
  - Enter **trimmed_surface** for *File name*.
  - Click **OK.**

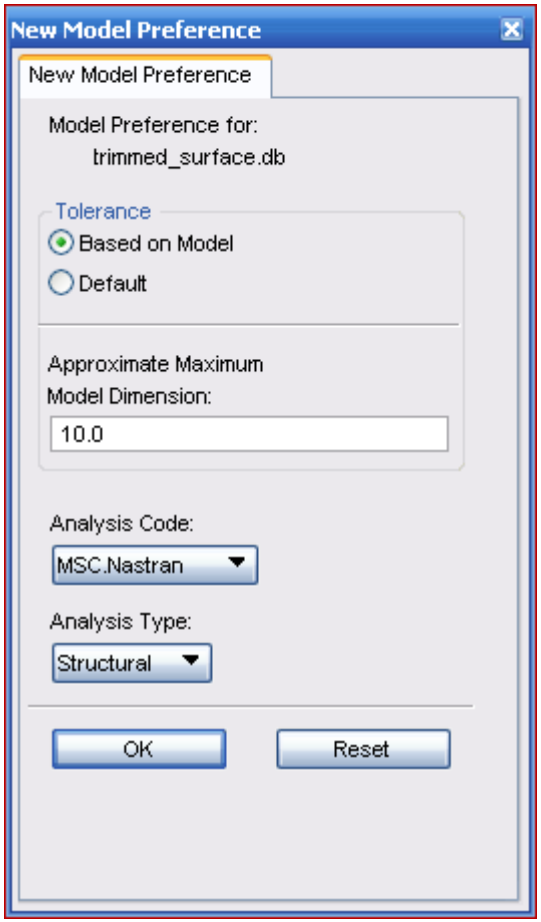




Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

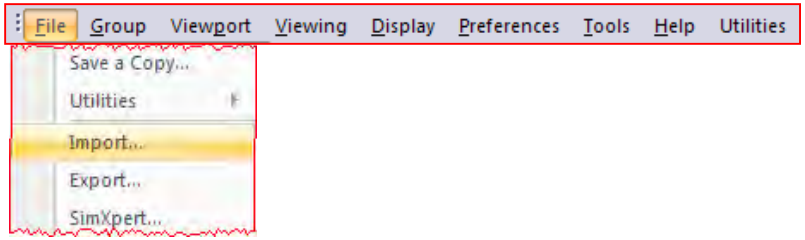
- **Set the New Model Preference**
  - Set the *Tolerance* to **Based on Model**
  - Enter **10.0** for *Approximate Maximum Model Dimension*.
  - Select **MSC.Nastran** for *Analysis Code*.
  - Select **Structural** for *Analysis Type*.
  - Click **OK**.



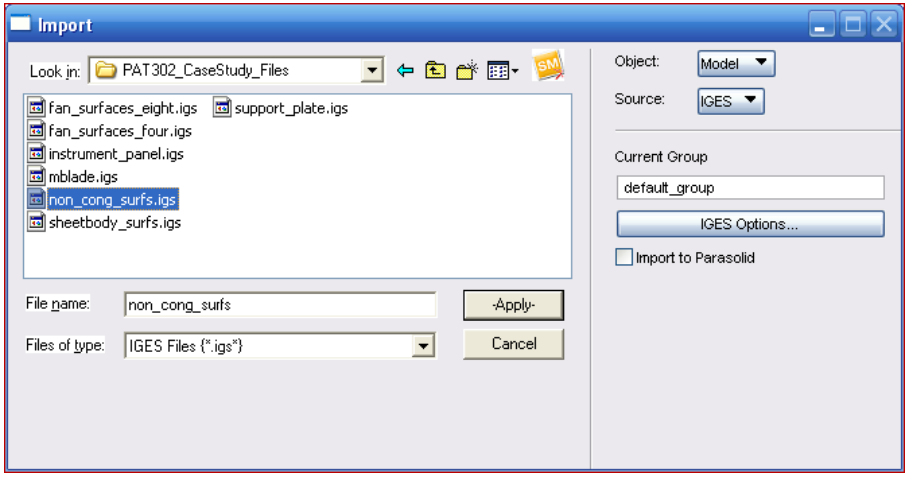
Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

- **Import the Surface Geometry**
  - Select **Import** under the *File* menu
  - Set *Object* as **Model** and *Source* as **IGES**
  - Uncheck **Import to Parasolid**.
  - Select **non_cong_surfs.igs**.
  - Click **Apply**.



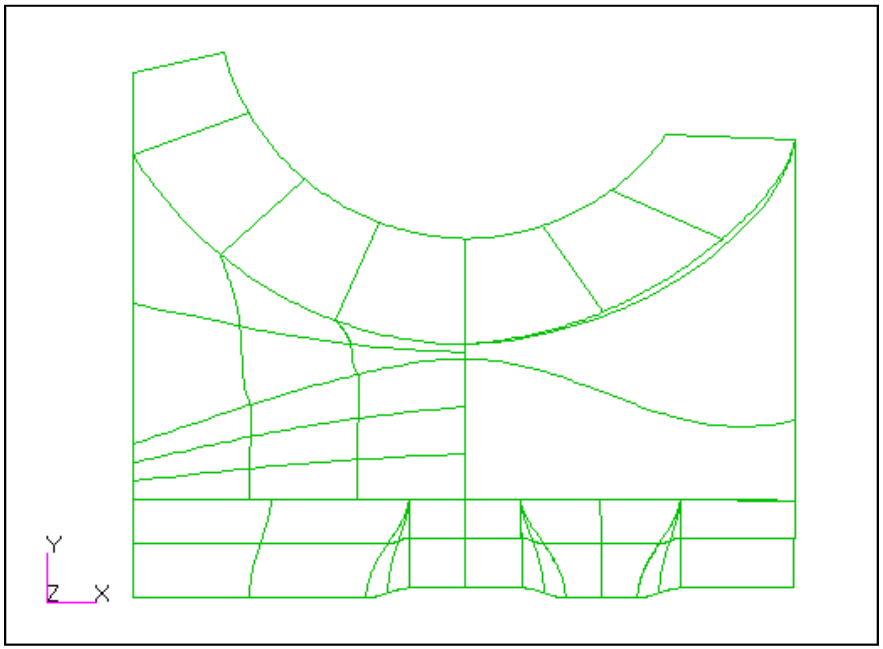
- **An IGES import summary from will appear**
  - See what types of surfaces are imported.
  - Click **OK**.

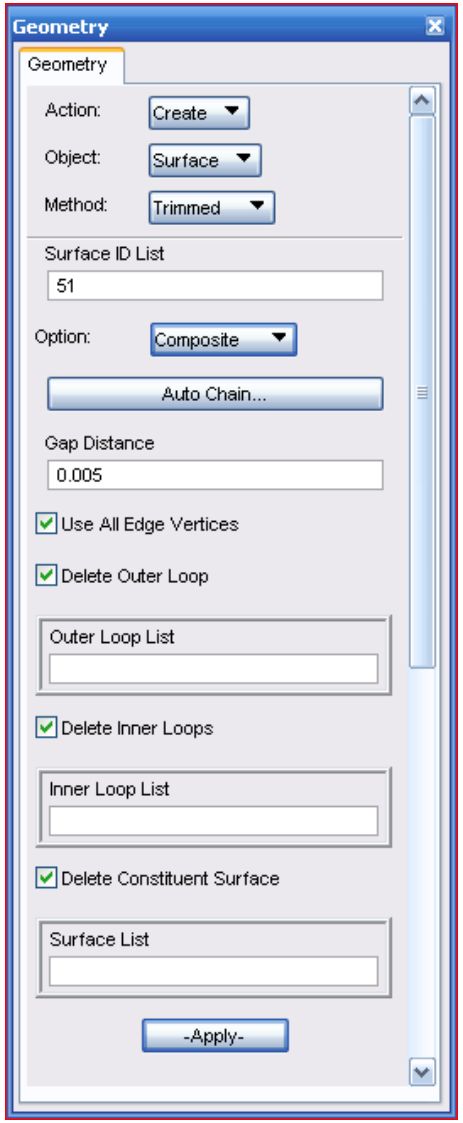


Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

- **Create the trimmed surface using the Composite option:**
  - Under the *Geometry* tab click on **Trimmed** in the *Surfaces* group.
  - Select **Composite** for *Option*.
  - Click **Auto Chain**.

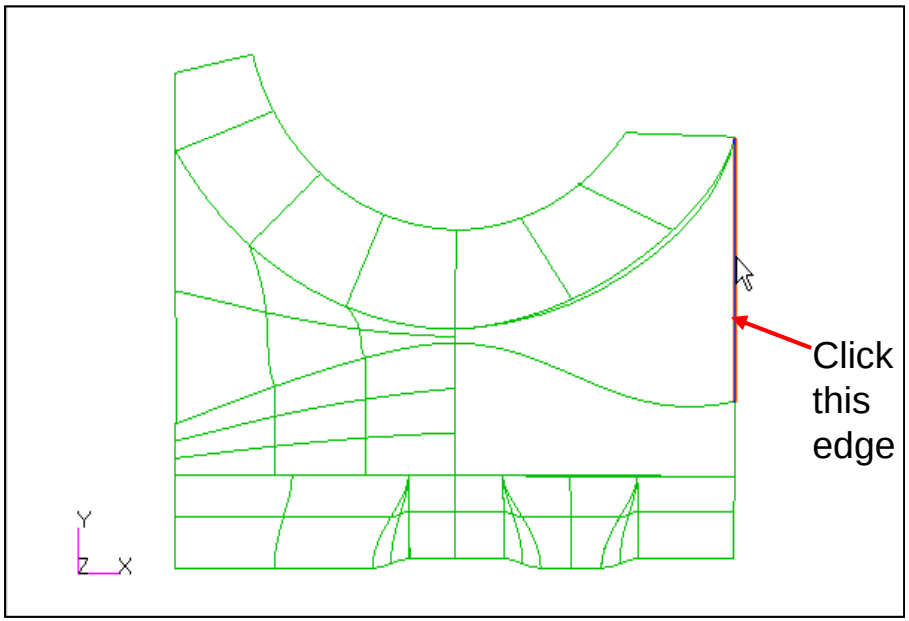


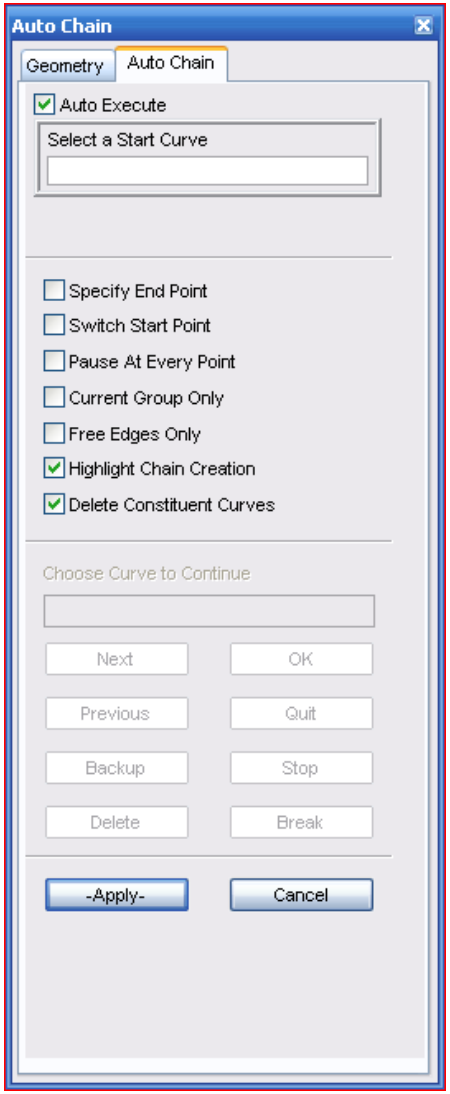


Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

– Select a surface edge to start creating the outer loop for the trimmed surface.

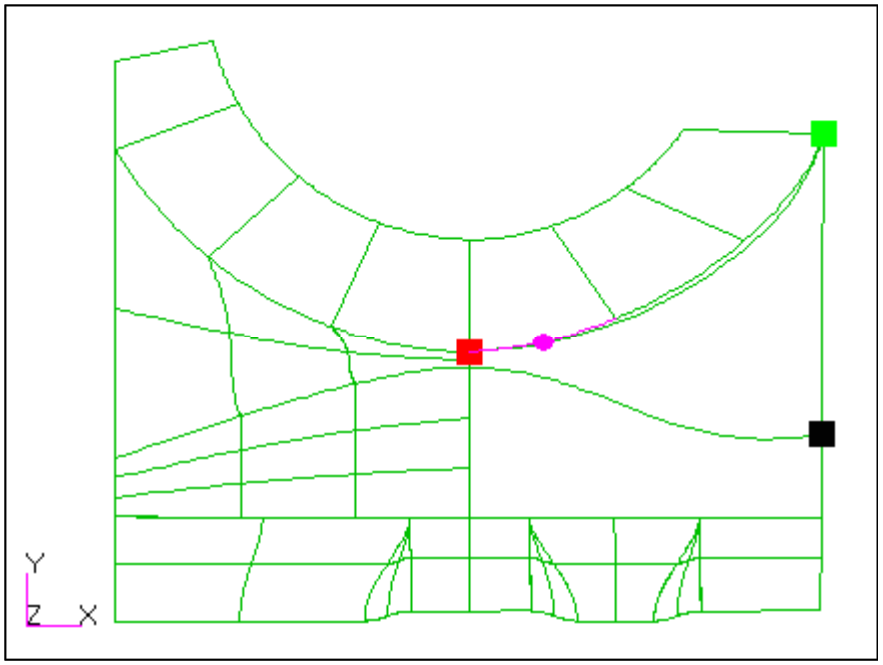




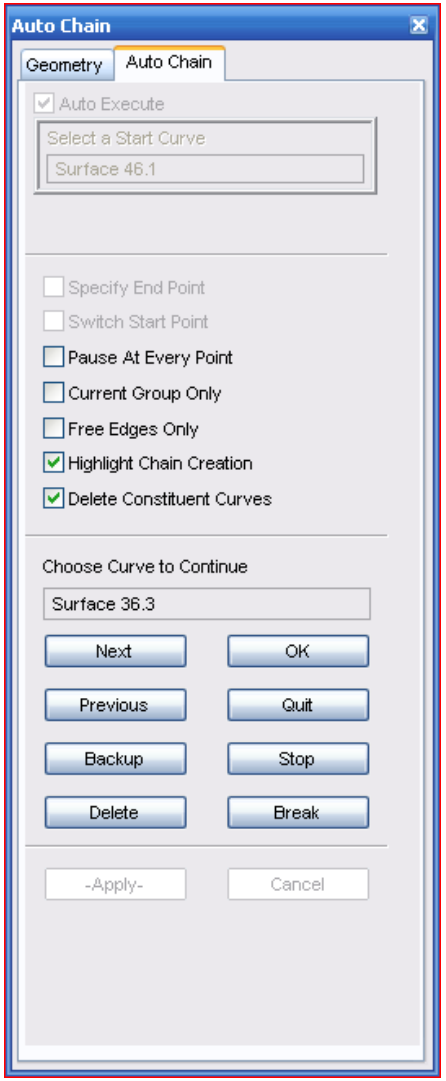
Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

- **The model should now appear as follows**



The autochain starts at the black square, and goes until the edge where the magenta circle is.

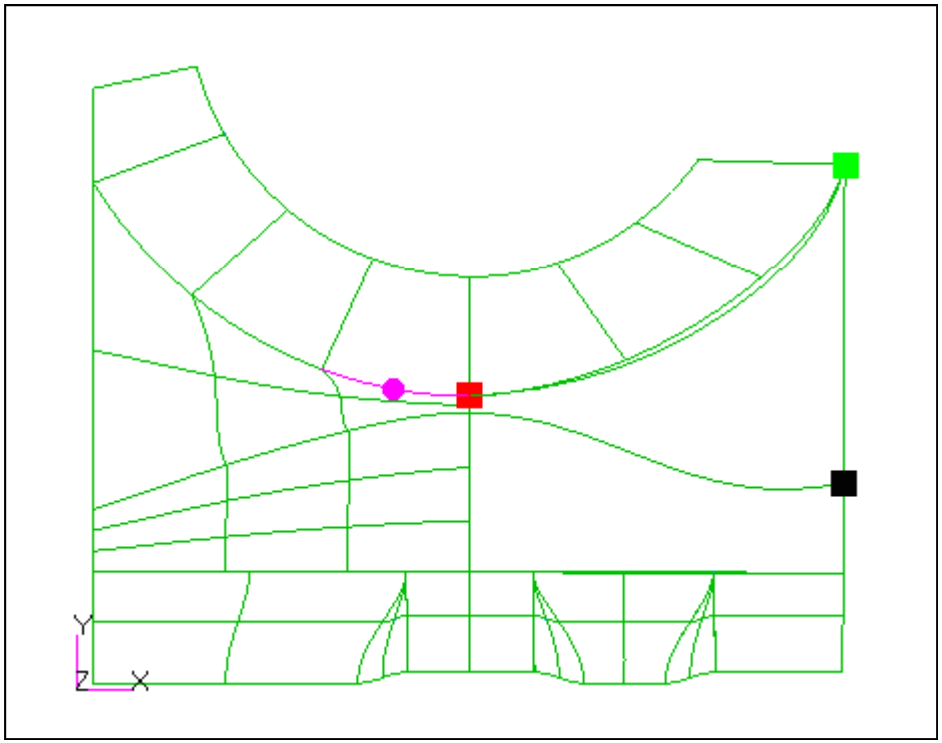


- **Because we want to create a surface from the external (outer) edges, this is not the desired path; so, click the Next button.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

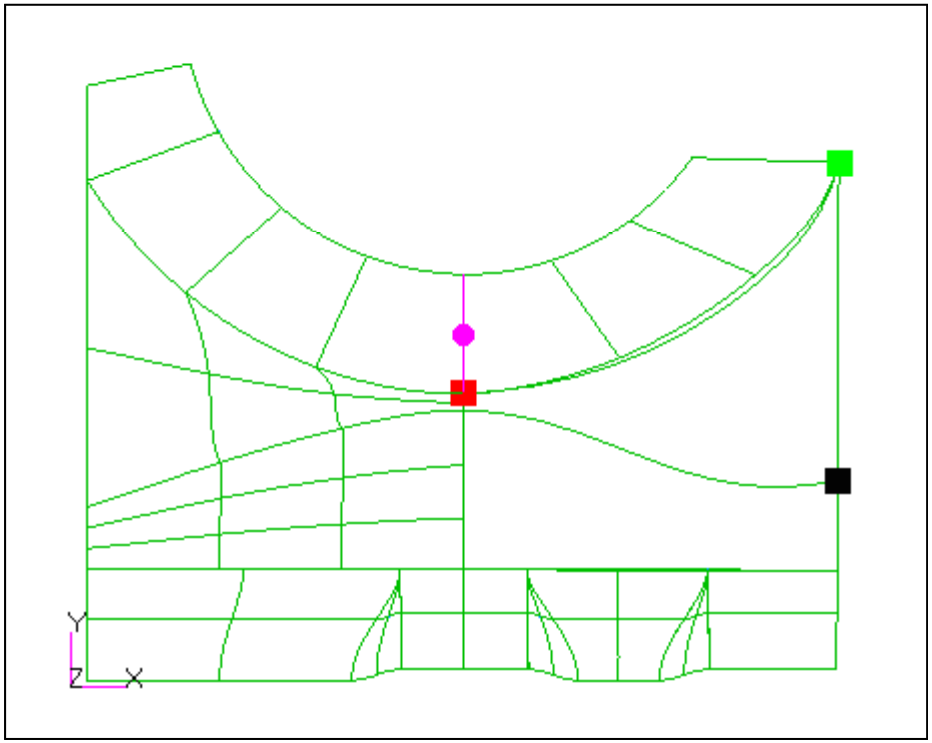
– The next edge found is shown below at the magenta dot, but again, we only want external edges; so, click Next once more.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

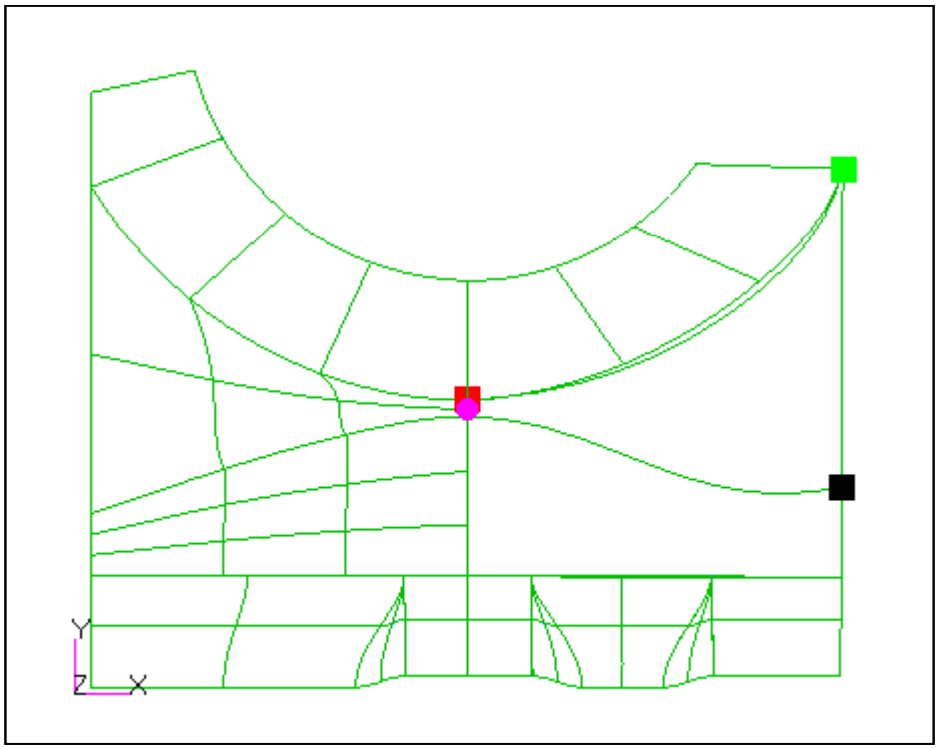
- The next edge found is shown below, but again, this is not the desired edge.
- Click **Next**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

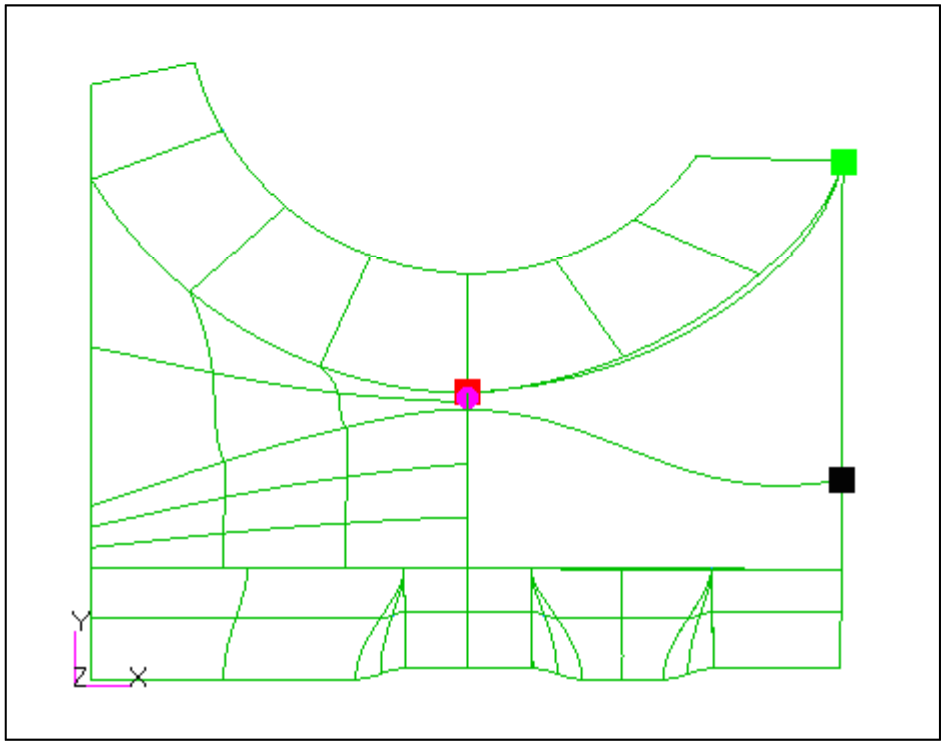
- The next edge found is given below, but again, it is not the external edge that is desired.
- Click **Next**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

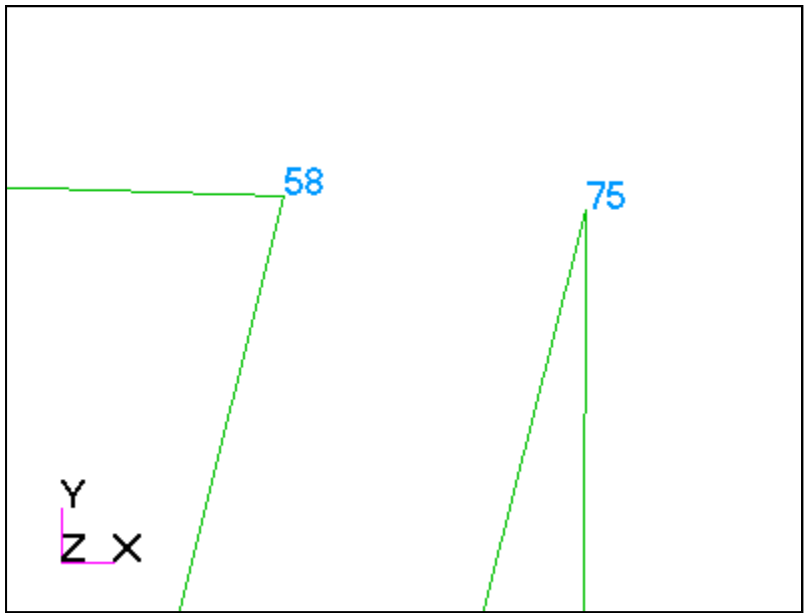
- Another edge is found, but not an external one.
- Click **Next**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

- **Note that the initial edge is found once again. Thus, it will not be possible to get to the next desired external edge to define the outer loop of the trimmed composite surface.**
  - Select **Quit**, then click **Cancel**.
  - Zoom into the upper right corner of the model.



There are two points (58, 75) / vertices. They are 0.03 units apart. The Global Model Tolerance is 0.005.

Copyright© 2012 MSC.Software Corporation

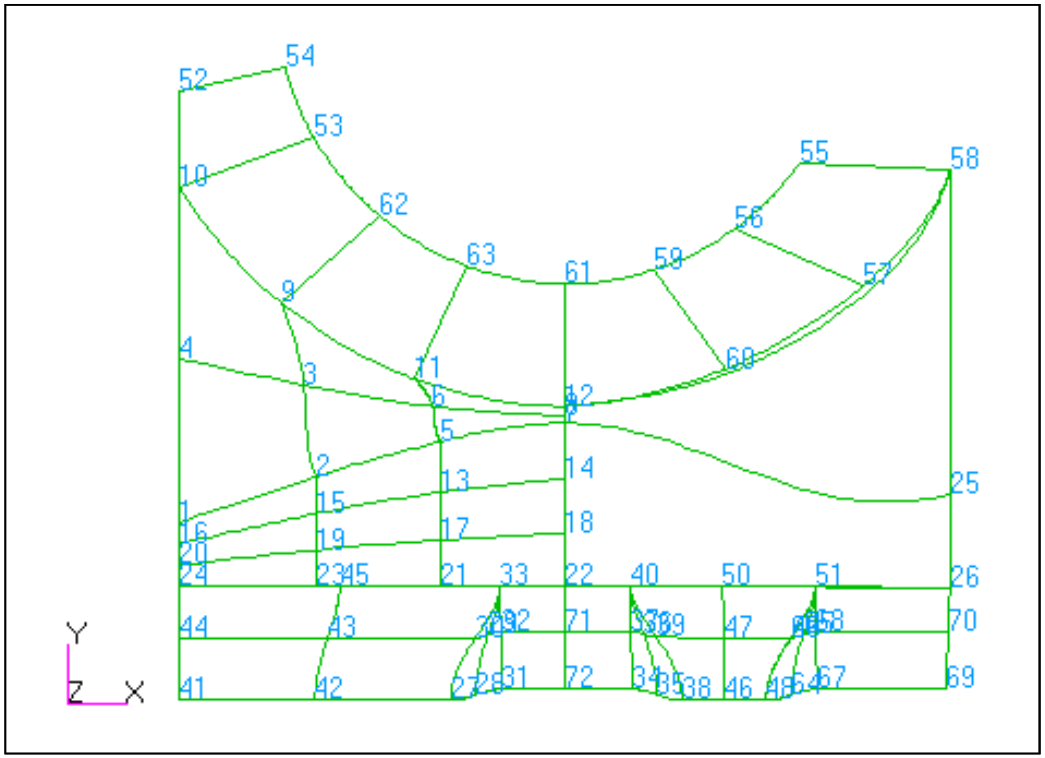
CASE STUDY: TRIMMED SURFACE, COMPOSITE

- **The two vertices were not equivalenced upon import. This is why it was not possible to "jump" (traverse) the gap during the attempted creation of the outer loop.**
- **Start over!**
  - Under the *Geometry* tab click on **Delete** in the *Geometry Actions* group.
    - Select all surfaces in the viewport for *Geometric Entity List*
    - Click **Apply** and **Refresh graphics**
  - Select **Global…** from the *Preferences* menu
    - Change the *tolerance* to **0.05**
    - Click **Apply**
  - Select **Import…** from the *File* menu
    - Select **Model** for *Object* and **IGES** for *Source*
    - Select the IGES file **non_cong_surfs.igs**
    - Make sure **Import to Parasolid** is unchecked.
    - Click **Apply**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

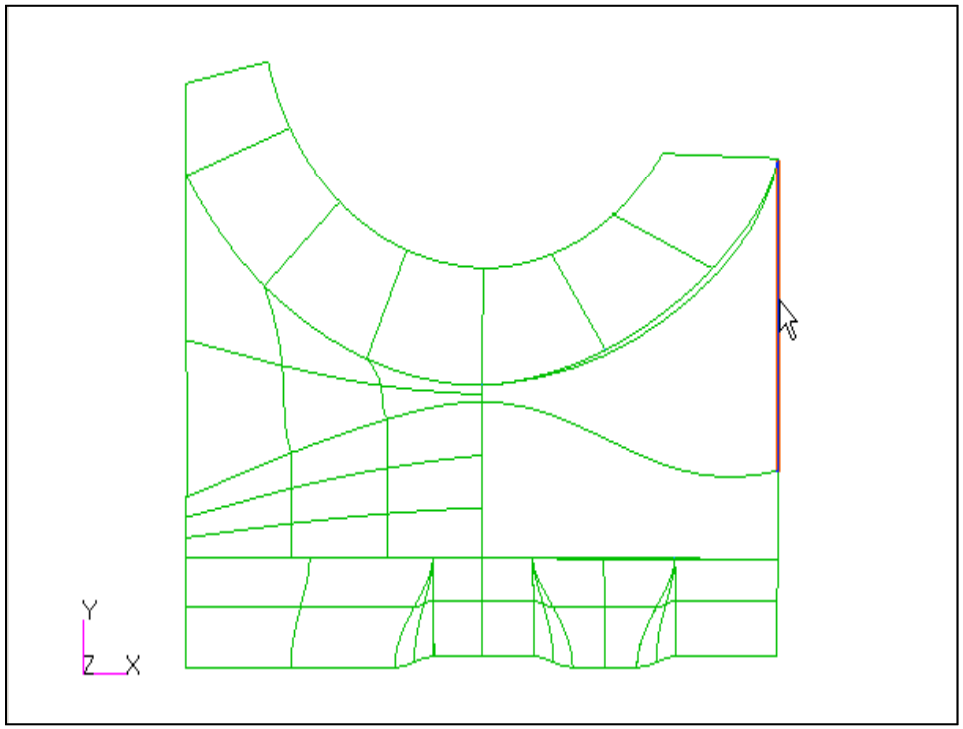
- **Now only one point (Point 58) is at the upper right corner vertex.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

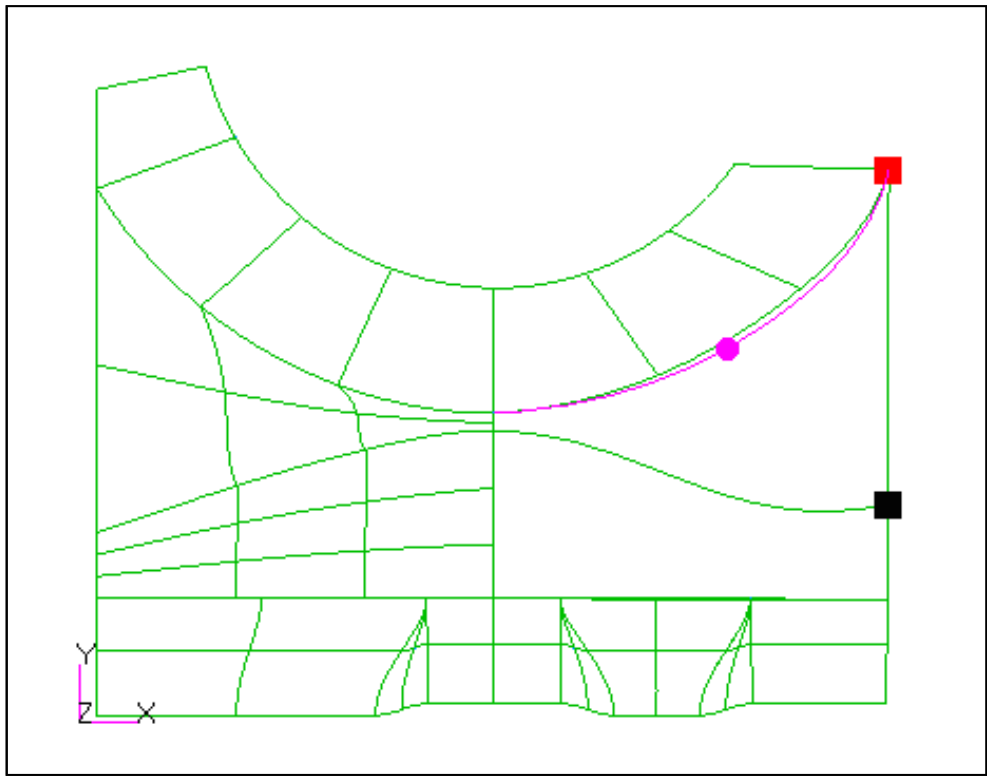
- **Try to create the trimmed composite surface using the Auto Chain feature again:**
  - Select **Composite** for *Option*.
  - Select **Auto Chain**.
  - Click the edge as shown in the given image.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

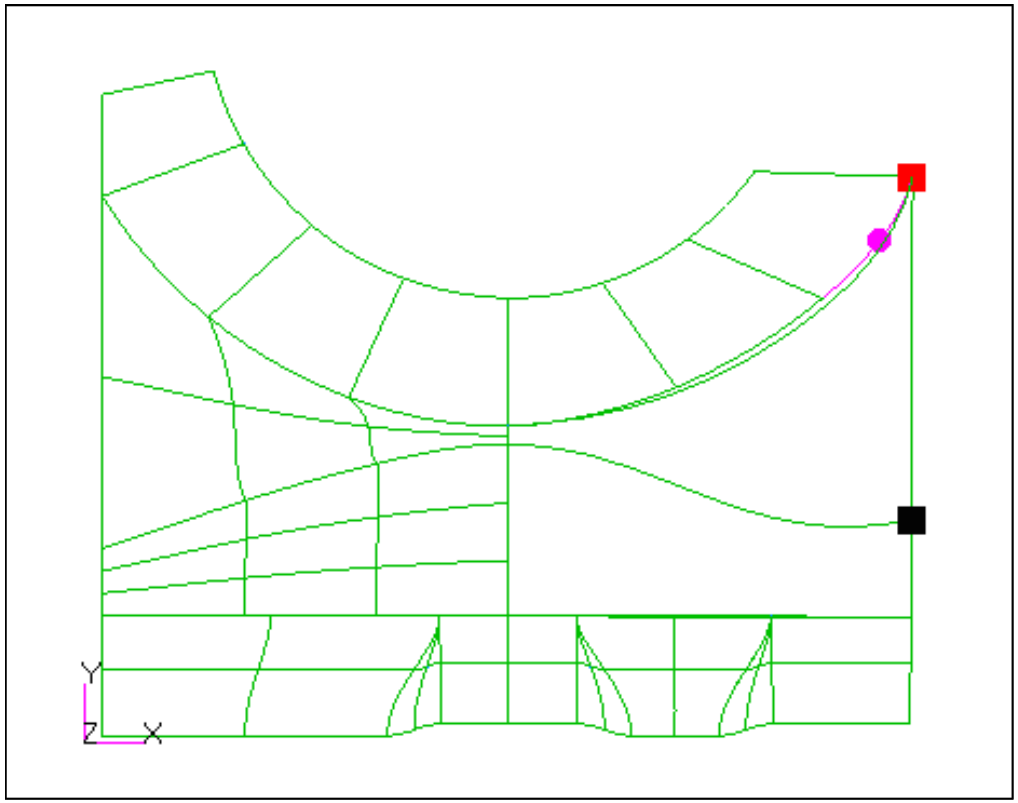
- **The first edge found is shown as follows**
  - Click **Next**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

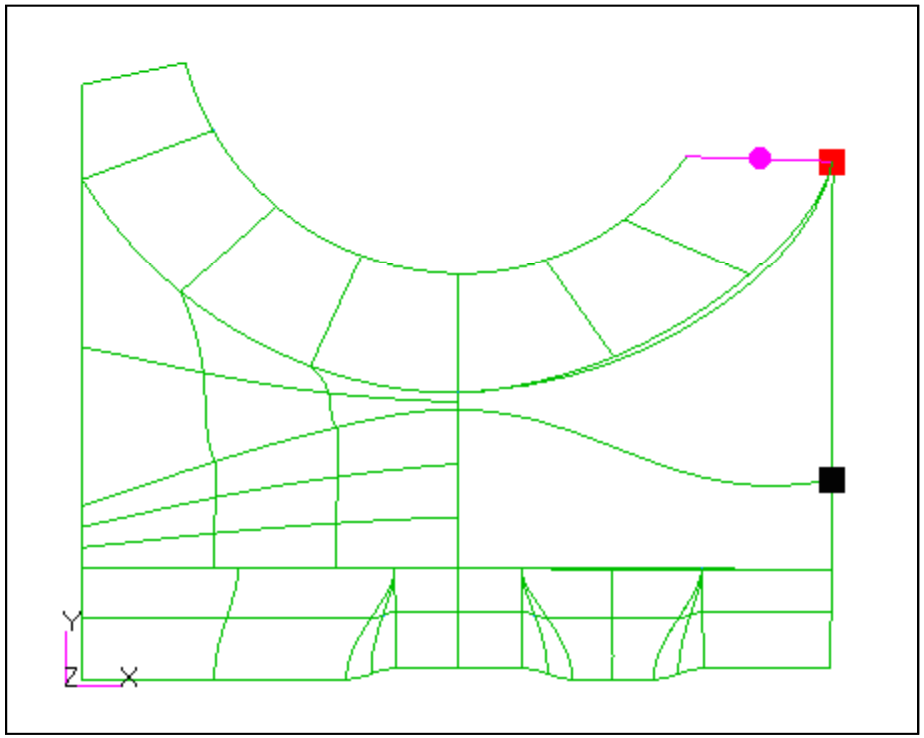
- **The second edge found is as follows**
  - Click **Next**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

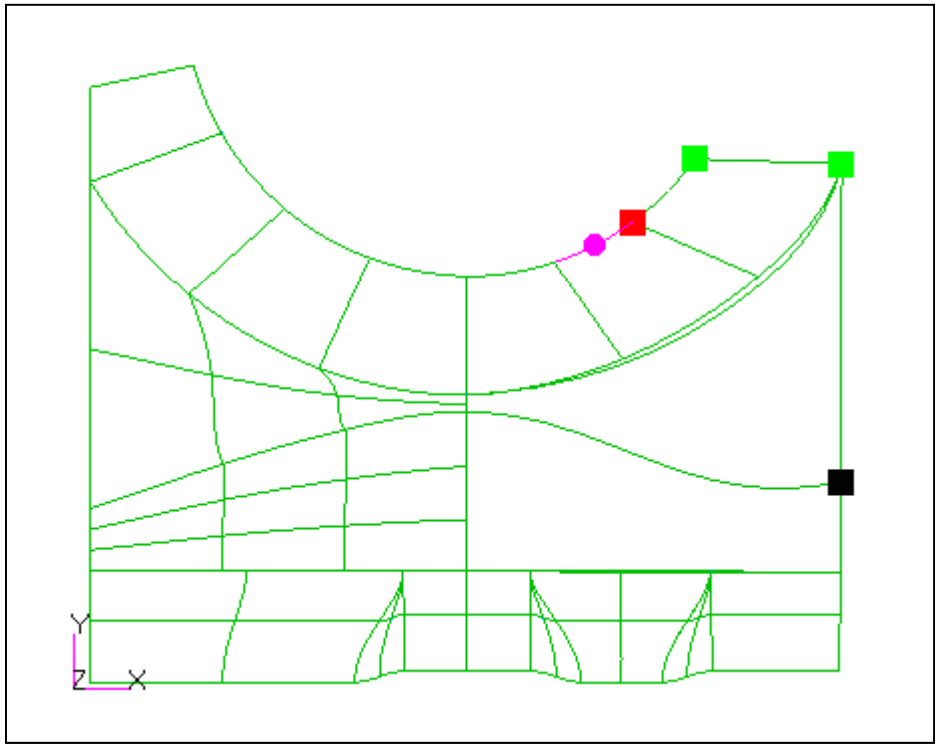
- **Finally, the desired edge is found**
  - Click **OK**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

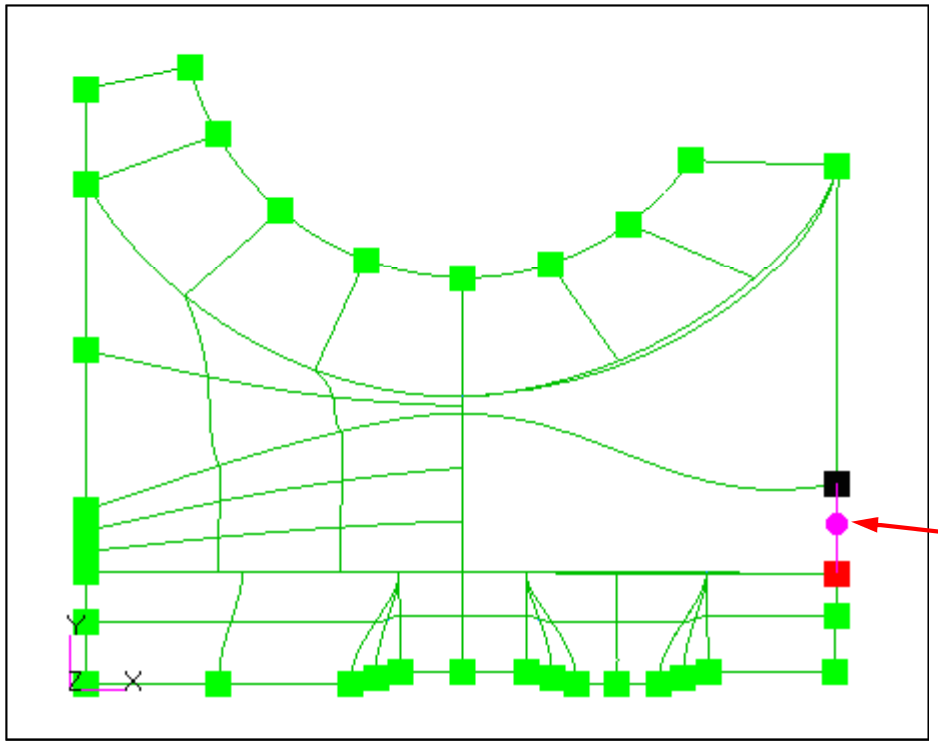
- **The following edge is then selected**
  - Click **OK**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

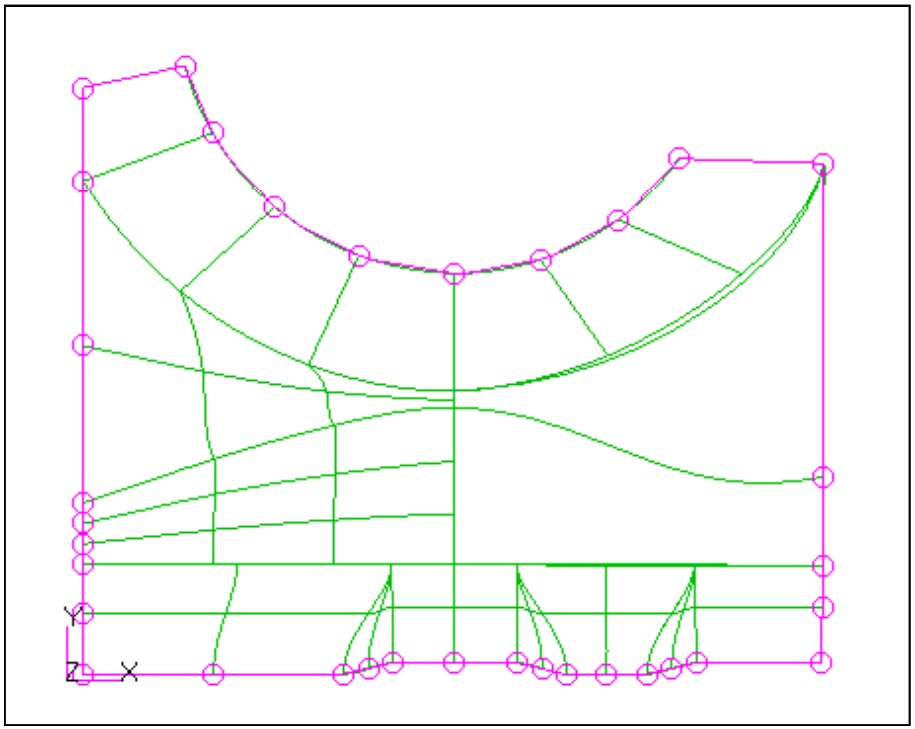
- **Keep using Next and OK, as needed, until the outer loop is defined as shown**
  - Click **OK**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

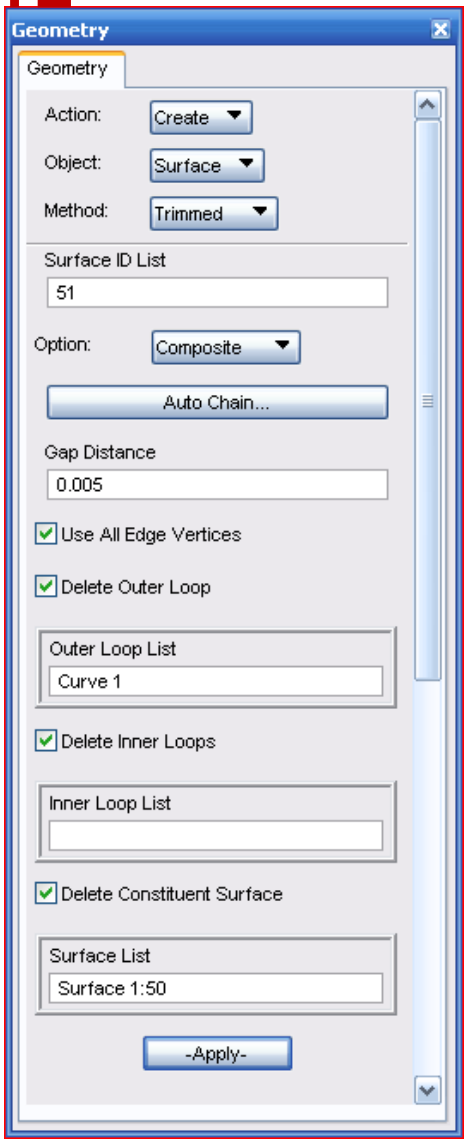
- **The chained curve has been created. The curve's color is yellow, but initially is shown as magenta.**
  - Click **Cancel**.

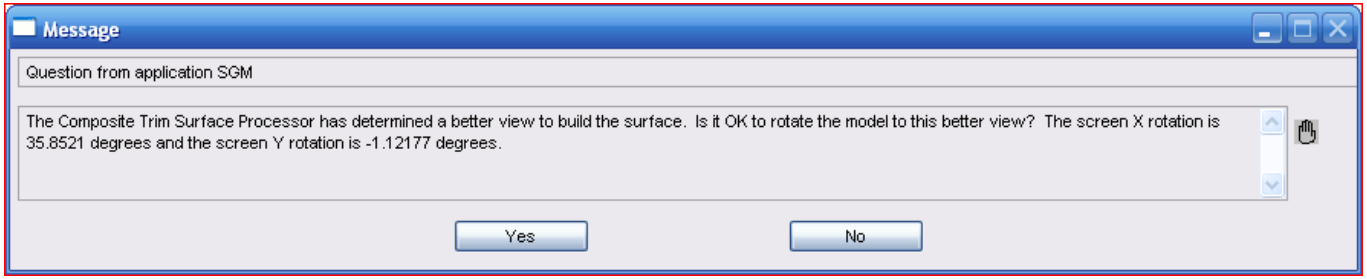


Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

- **Select the newly created chained curve for Outer Loop List.**
- **Select all the surfaces for Surface List.**
- **Click Apply.**
- **You will then receive a message stating that a better view exists.**
  - Select **Yes**.

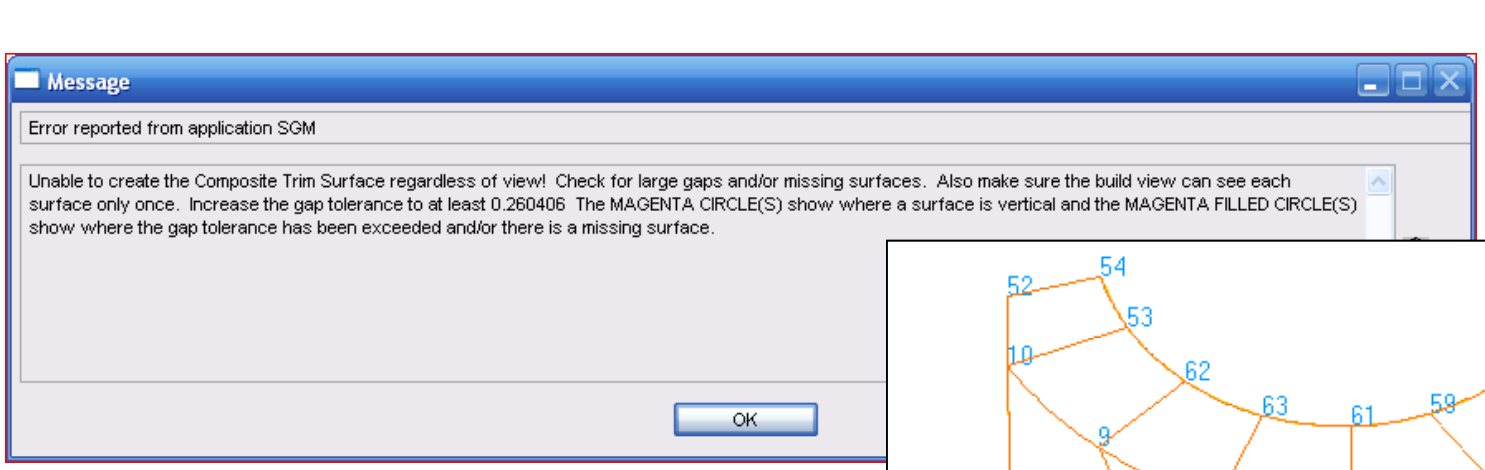


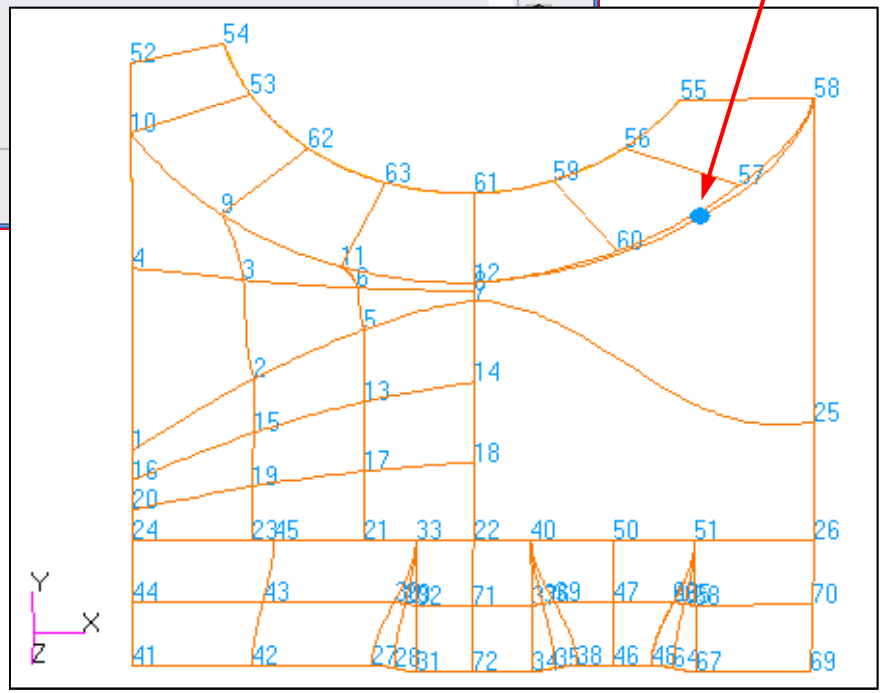


Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

- **Another message appears, stating that there is an internal gap.**



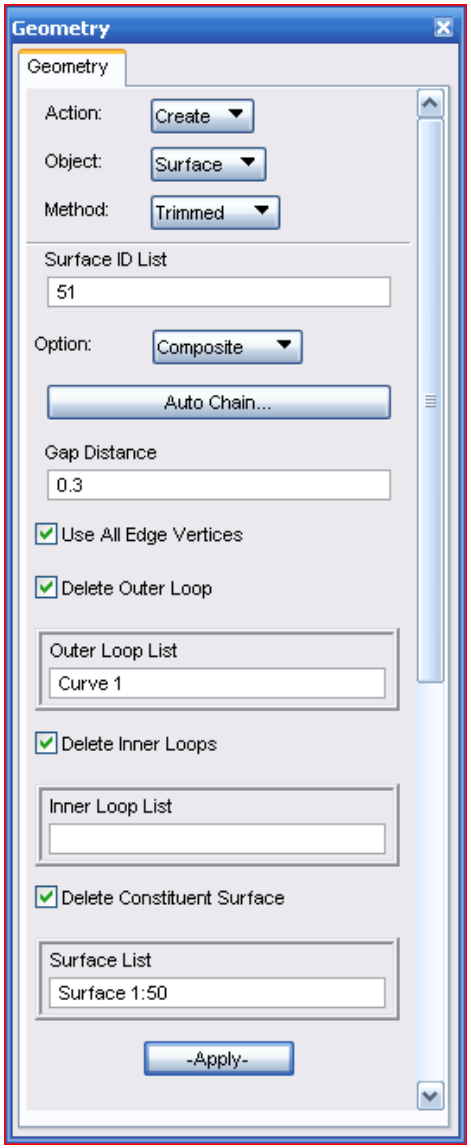


  - The blue dot indicates the location of the internal gap.
  - Click **OK**.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

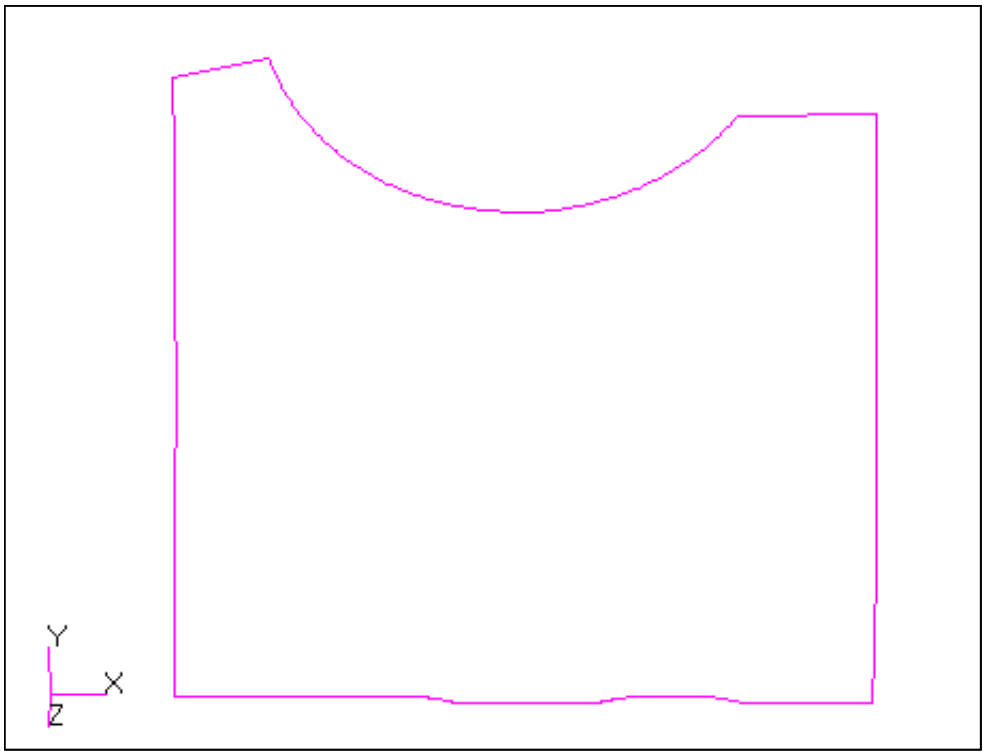
- **Change the Gap Distance to 0.3.**
- **Click Apply.**
- **Click Yes to rotating to a better view and Yes to Deleting original Curves and Surfaces.**
- **Refresh the graphics.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

- **The following trimmed composite surface should be obtained**



Copyright© 2012 MSC.Software Corporation

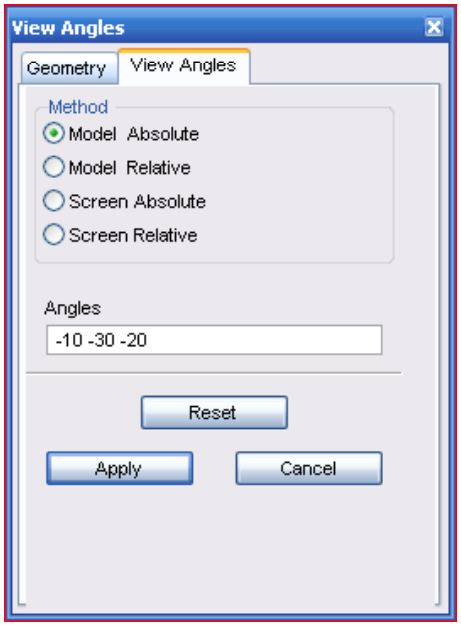
CASE STUDY: TRIMMED SURFACE, COMPOSITE

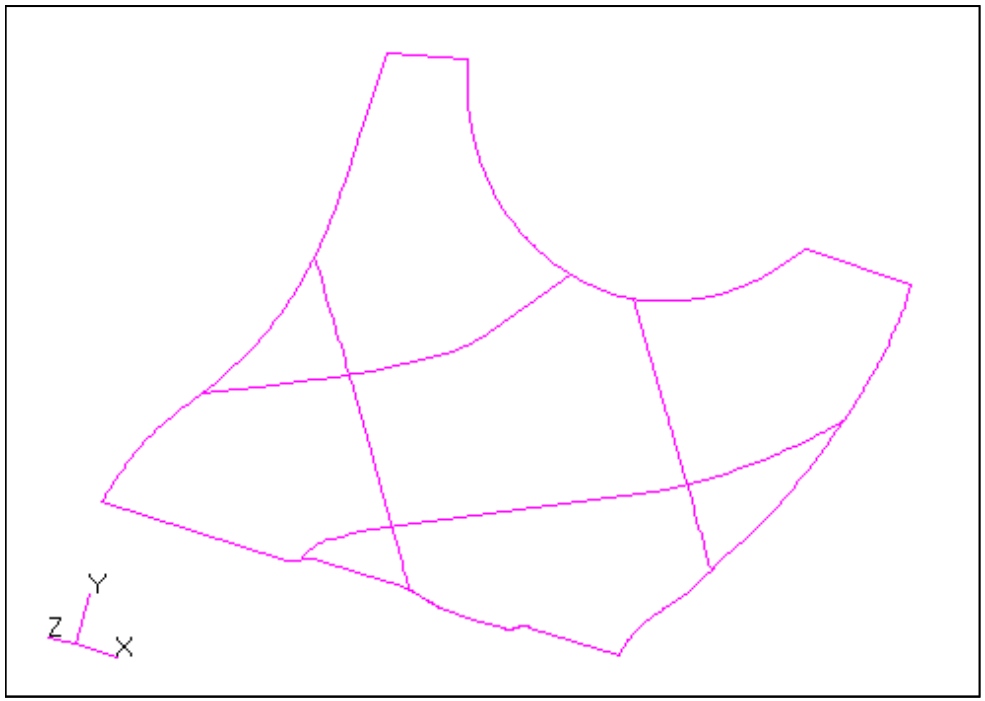
- **The following warning messages were issued by Patran:**
  - $# Warning reported from application SGM
  - $# WARNING -- Composite Trim Surface Evaluator has determined that a surface is approaching vertical. Angle=89.94569 As a result, the display of the surface may be abnormal and the surface may NOT be meshable!
- **This can occur if two surfaces in Surface List are almost at right angles to each other.**
- **Proceed to see what happens.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

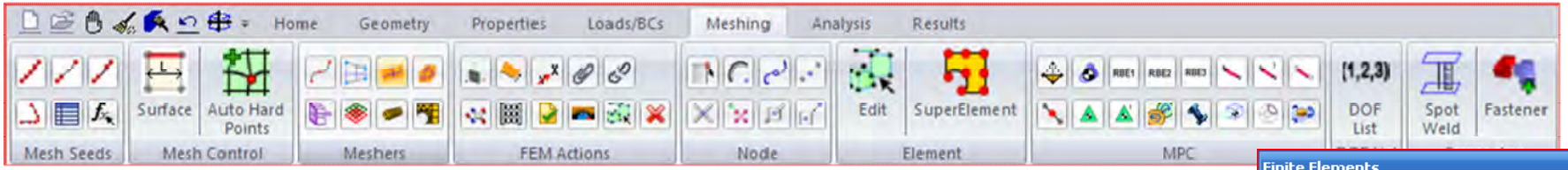
- **Show the display lines using the display lines icon**
- **Change the view:**
  - Select **Angles…** under the *Viewing* menu
  - Set angles to **-10 -30 -20**
  - Click **Apply**.

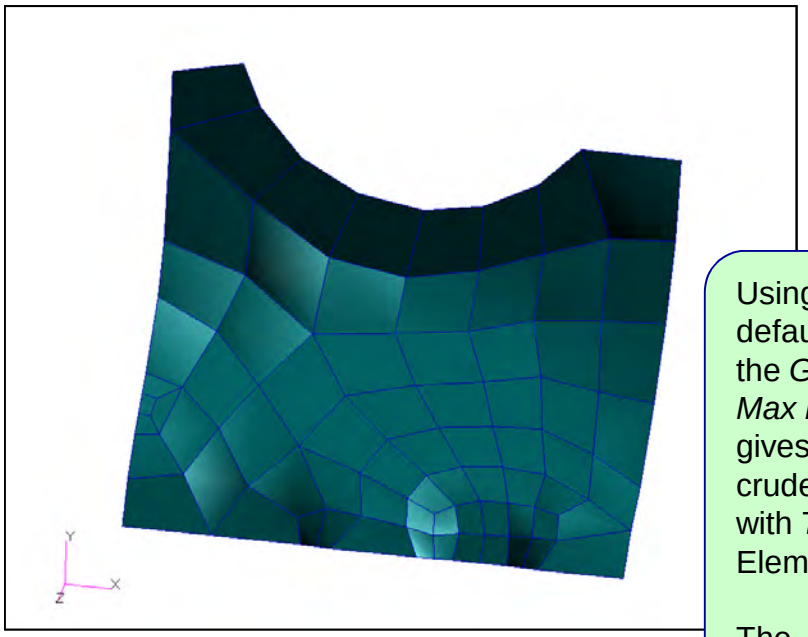




Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

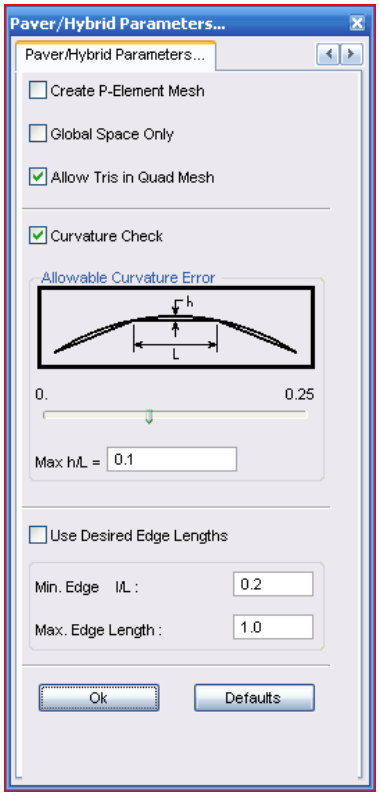


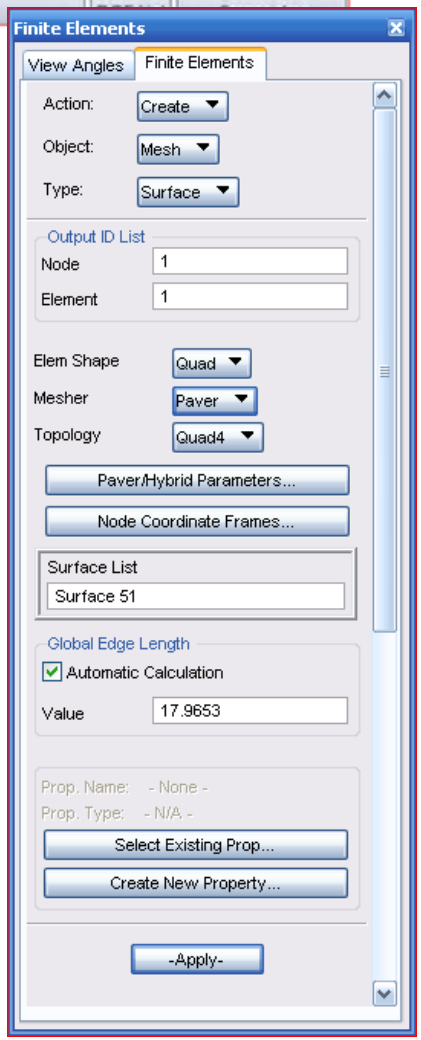


Using defaults for the *GEL* and *Max h/L* gives a very crude mesh with 73 Elements.

The question now is:

Is the mesh fit for the purpose?





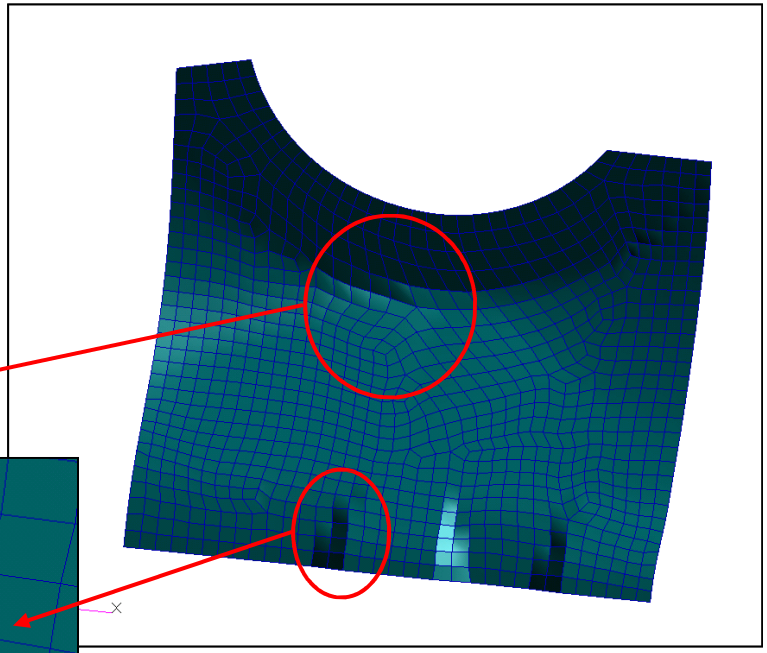
Copyright© 2012 MSC.Software Corporation

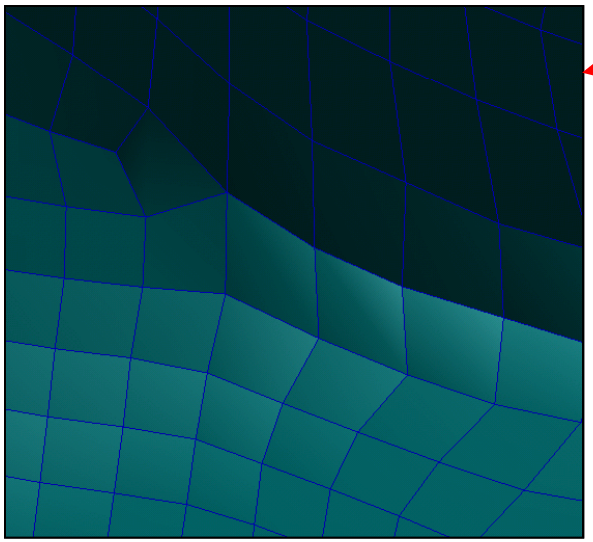
CASE STUDY: TRIMMED SURFACE, COMPOSITE

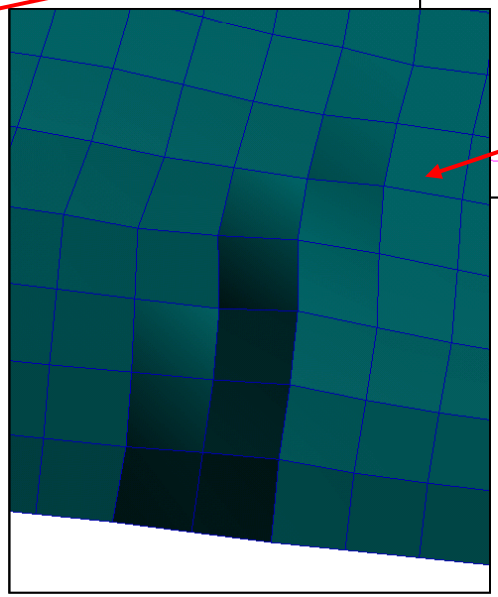
Using a *GEL* of 3.0 and default *Max h/L* gives a mesh with 1006 Elements.

The detail of the swages and the creases in the skin are poor.

Is the mesh fit for the purpose?







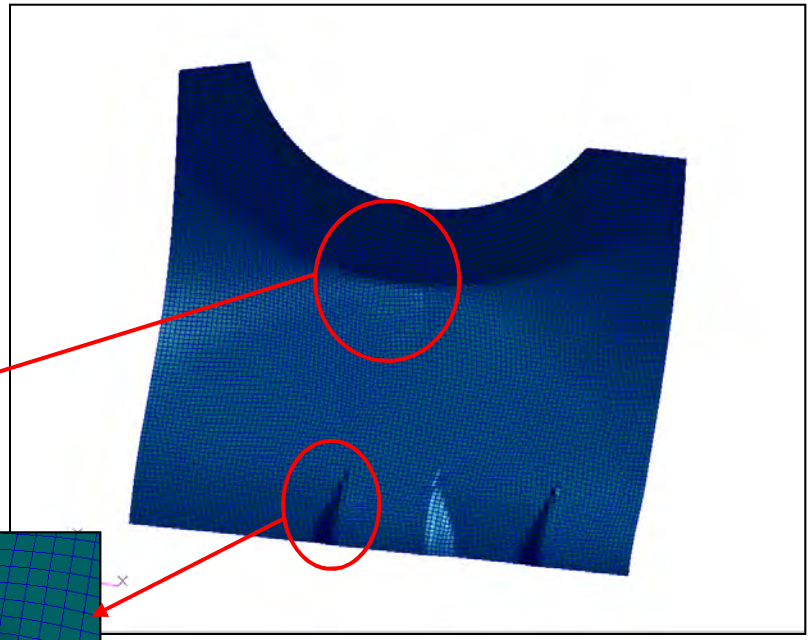
Copyright© 2012 MSC.Software Corporation

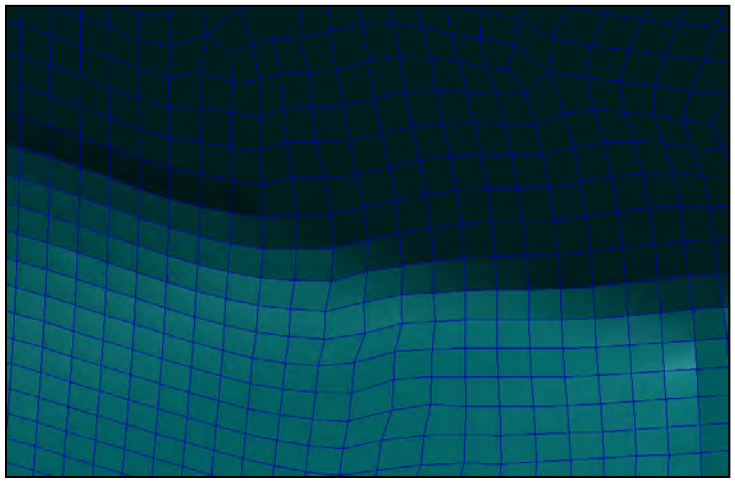
CASE STUDY: TRIMMED SURFACE, COMPOSITE

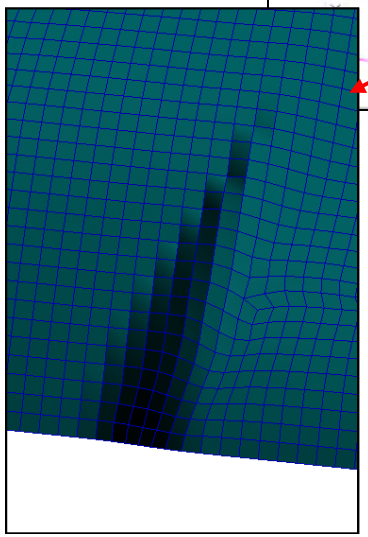
Using a GEL of 1.0 and default Max h/L gives a mesh with 9592 Elements.

The detail of the swages and the creases in the skin have improved.

Is the mesh fit for the purpose?







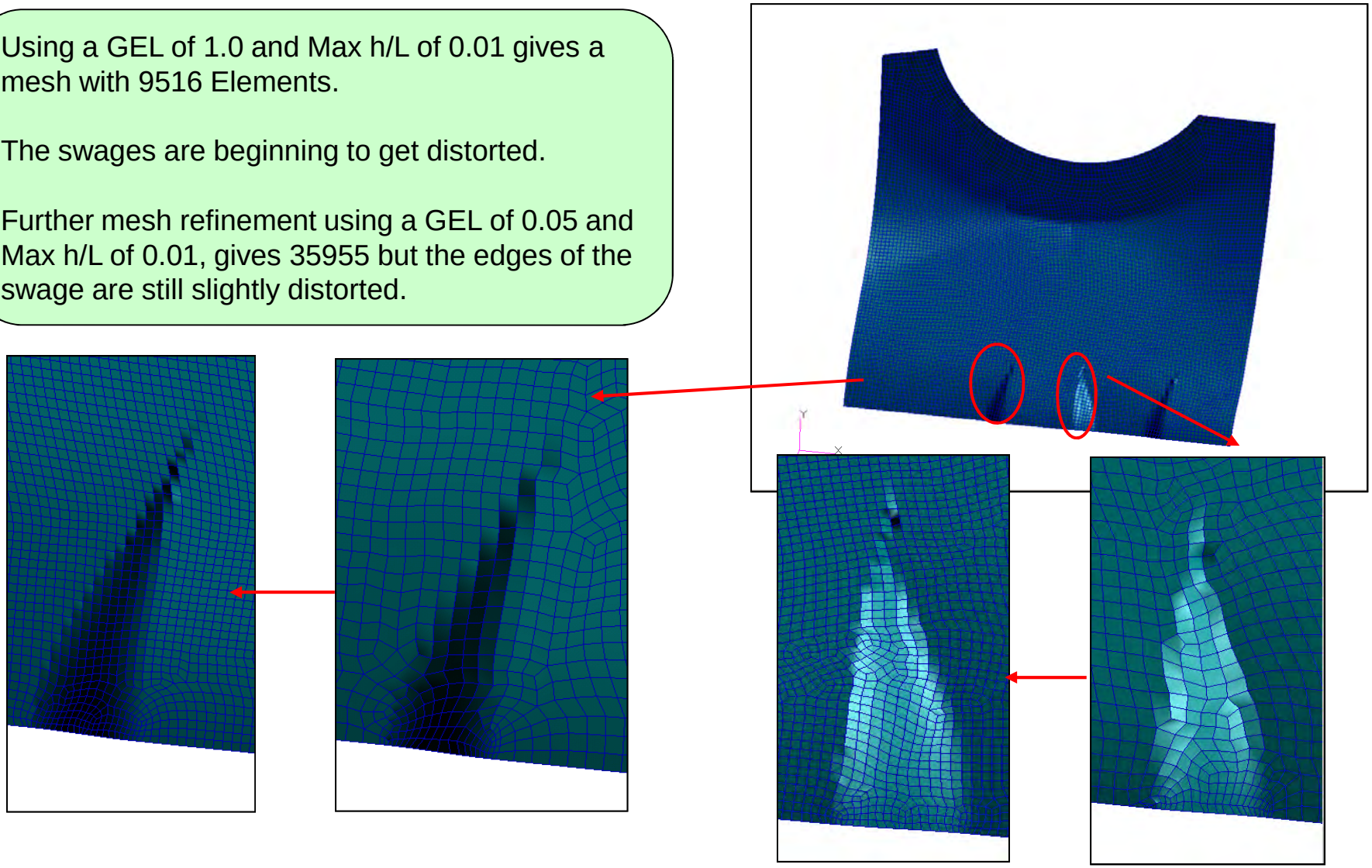
Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

Using a GEL of 1.0 and Max h/L of 0.01 gives a mesh with 9516 Elements.

The swages are beginning to get distorted.

Further mesh refinement using a GEL of 0.05 and Max h/L of 0.01, gives 35955 but the edges of the swage are still slightly distorted.



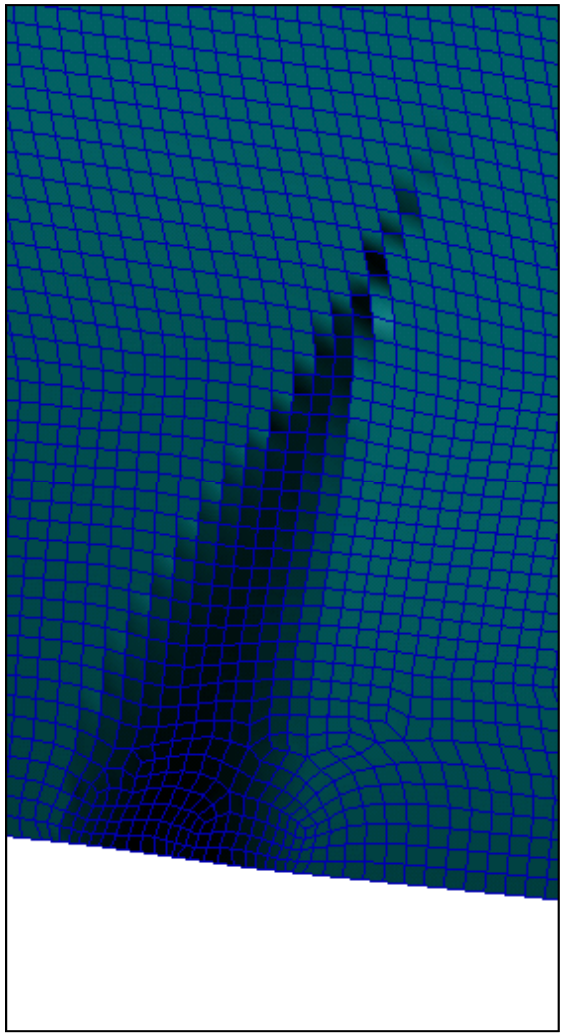
Copyright© 2012 MSC.Software Corporation

CASE STUDY: TRIMMED SURFACE, COMPOSITE

It may be possible to go further, using "Mesh on Mesh" to coarsen the mesh in areas away from local details such as the swage.

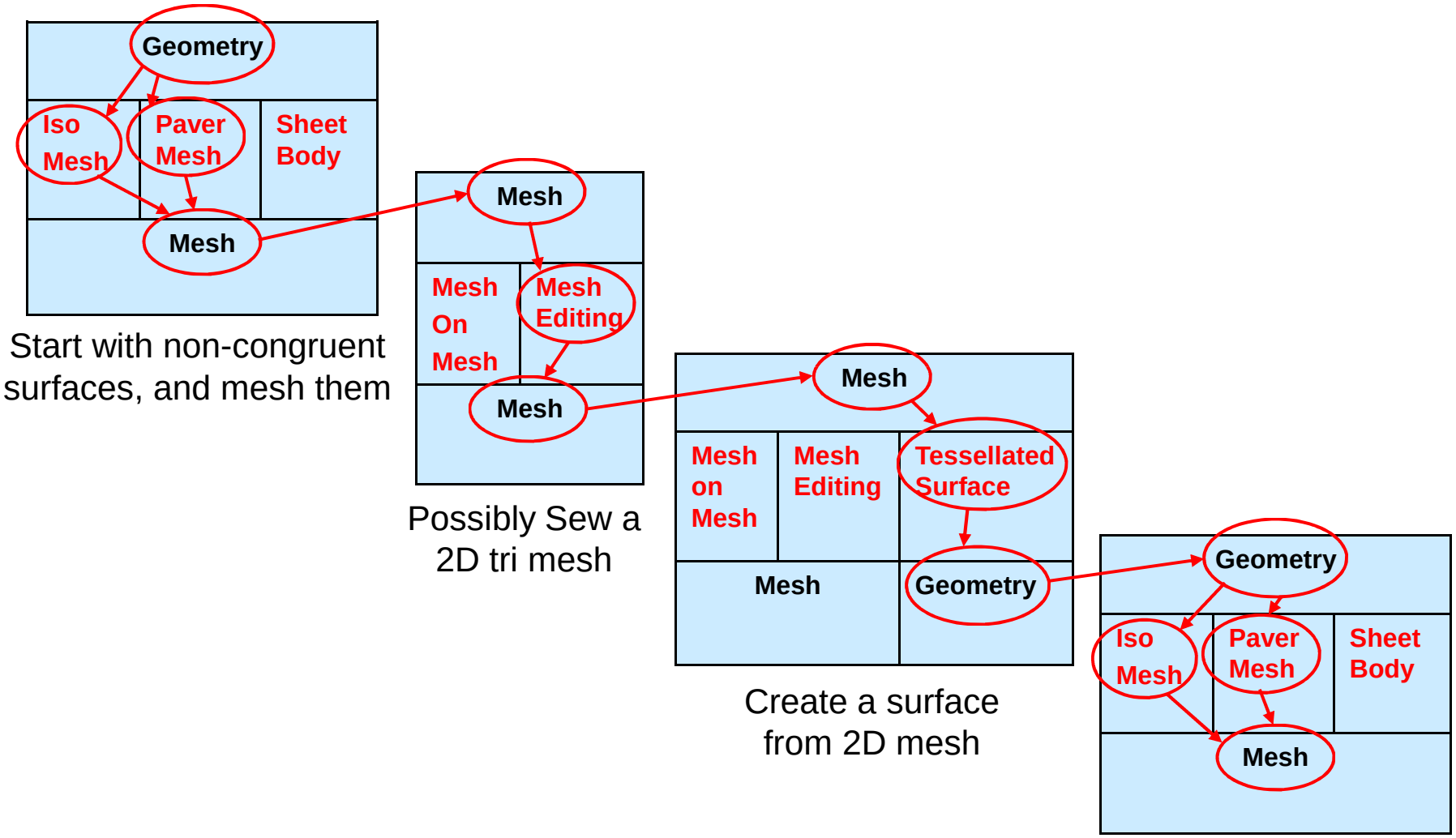
The target would be to reduce the element count in areas that are not of concern.

Will return to "Mesh on Mesh" later



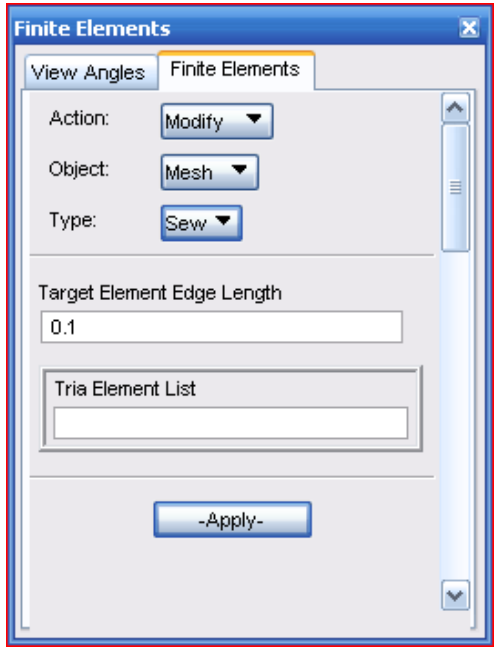
Copyright© 2012 MSC.Software Corporation

CONTINUOUS MESH FROM DISCONTINOUS SURFACES

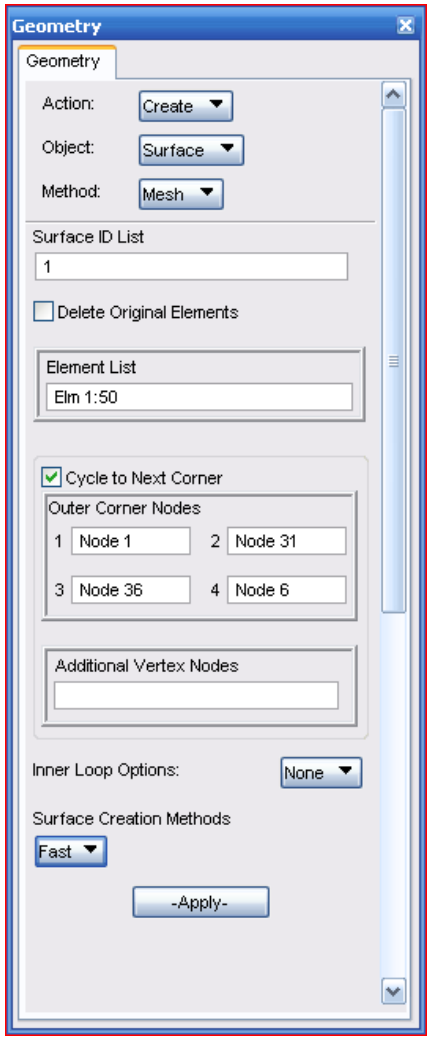


Copyright© 2012 MSC.Software Corporation

CREATE CONTINUOUS MESH FROM DISCONTINUOUS TRIA MESHES

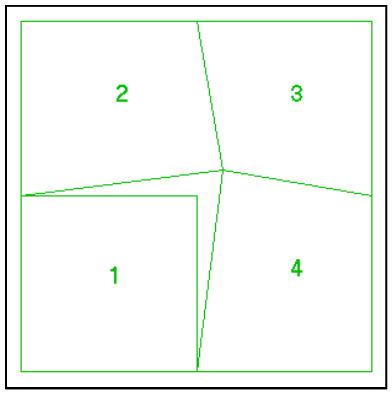


- Given a number of non-congruent surfaces, mesh them with tri elements
- Tie the non-congruent tri elements together using the Sew command under Finite Elements (Similar to Equivalence).
- Create a single surface using the congruent tri elements – create a tessellated surface.
- Finally, create a mesh by meshing the tessellated surface.

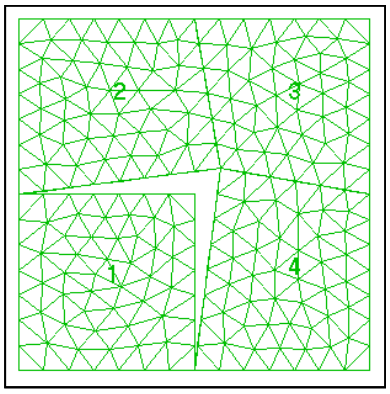


Copyright© 2012 MSC.Software Corporation

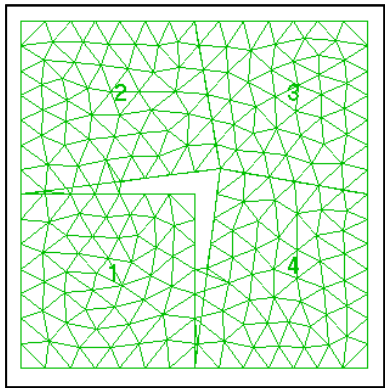
EXAMPLE: CREATE CONTINUOUS MESH FROM DISCONTINUOUS TRIA MESHES



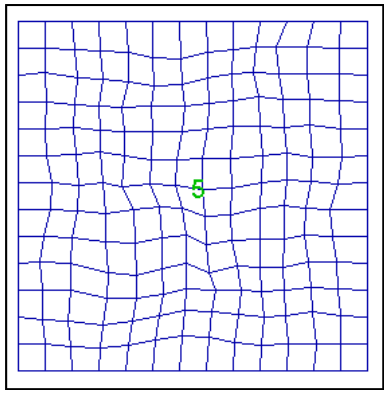
Four surfaces with gap



Surfaces with tri mesh (gap at the center)



Mesh after **Modify/Mesh/Sew**



Tessellated surface mesh (no gap present)

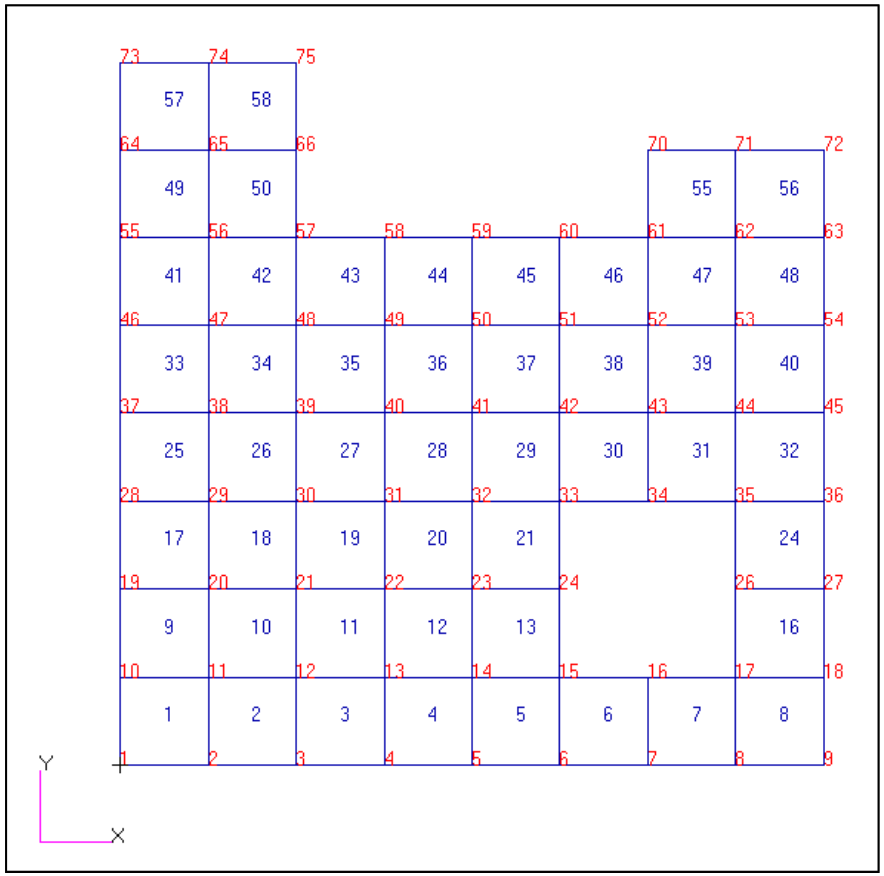
Copyright© 2012 MSC.Software Corporation

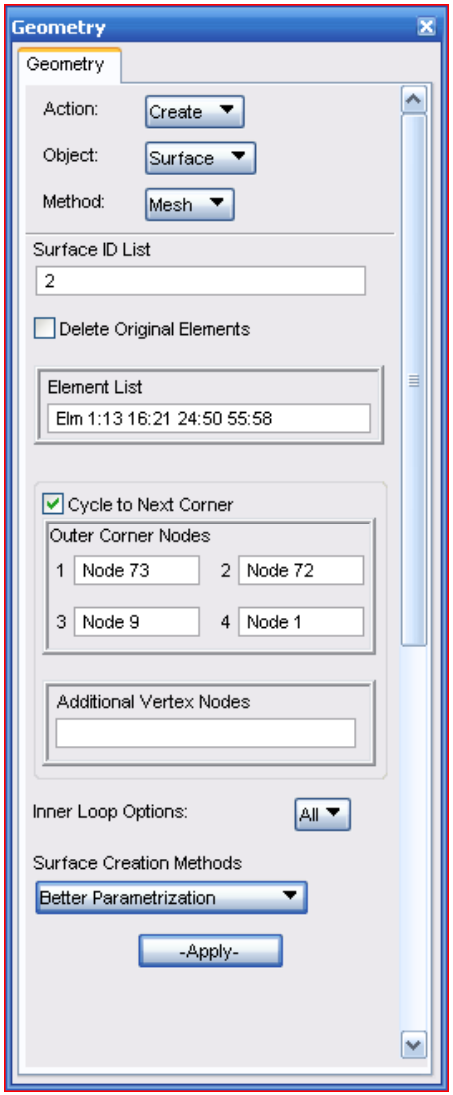
EXAMPLE: TESSELLATED SURFACE

A mesh is selected, which can be
- IsoMesh or Paver
- Trias or Quads

If a cutout is present, use the **Inner Loop Option** as needed.

Surface parameterization can be switched between speed and smoothness.

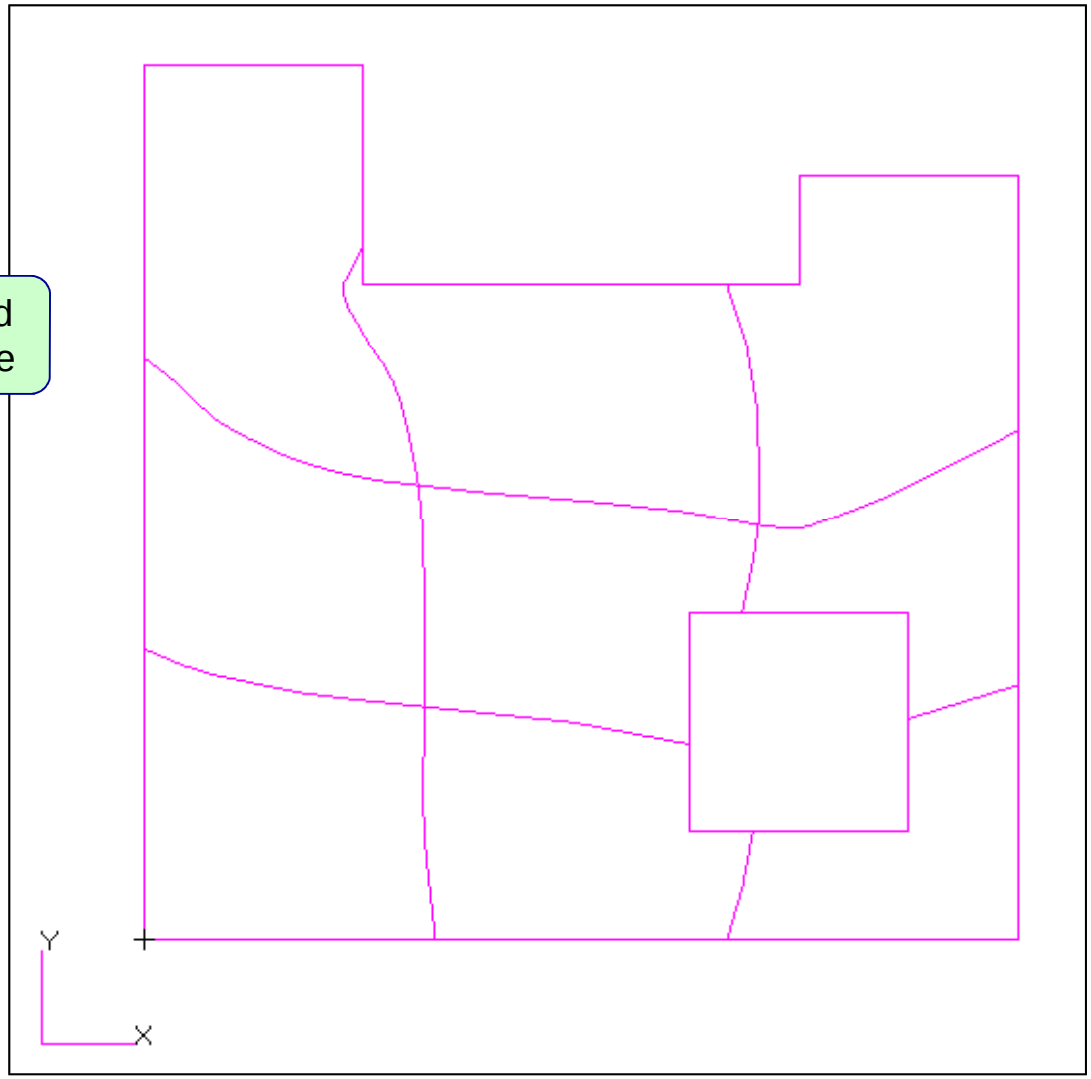




Copyright© 2012 MSC.Software Corporation

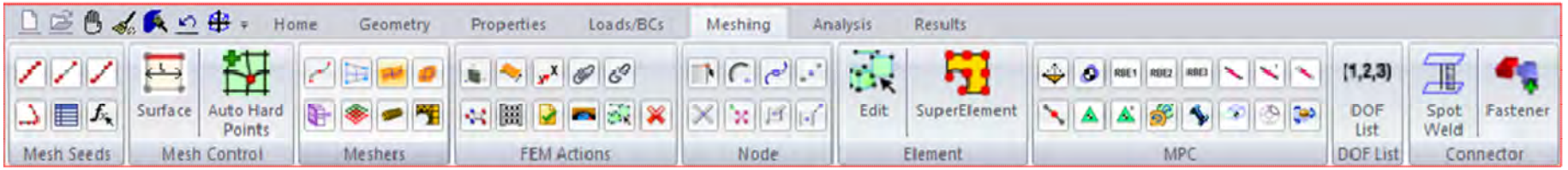
EXAMPLE: TESSELLATED SURFACE

The surface created is a trimmed surface

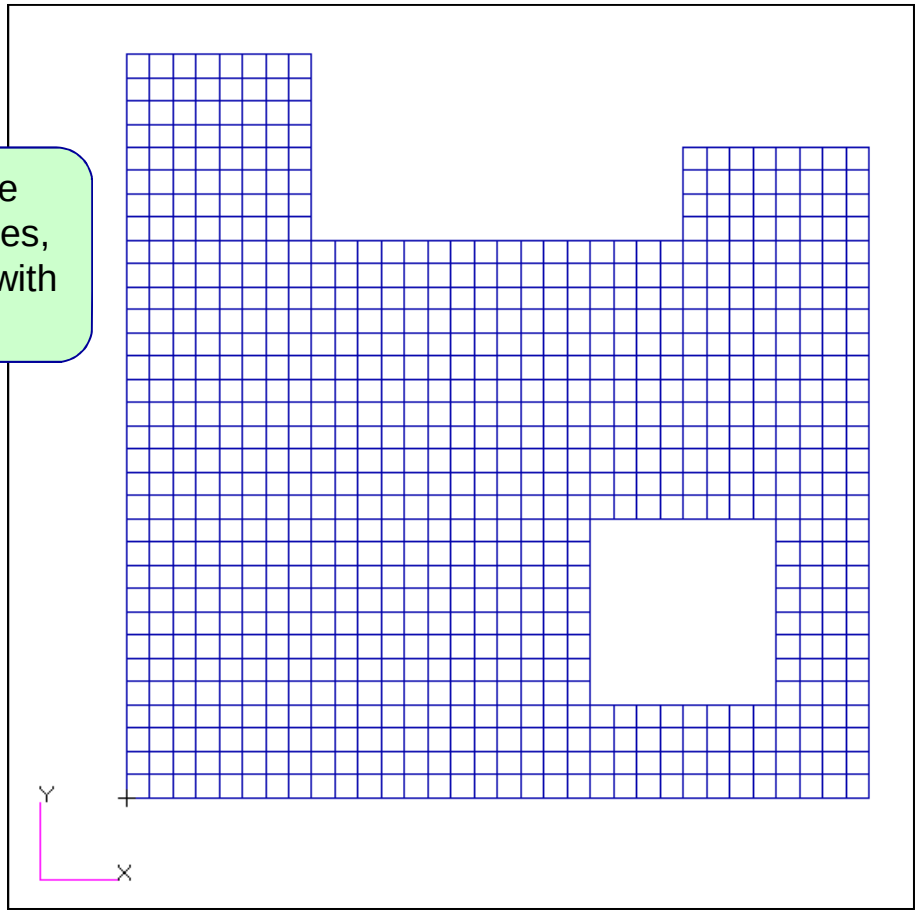


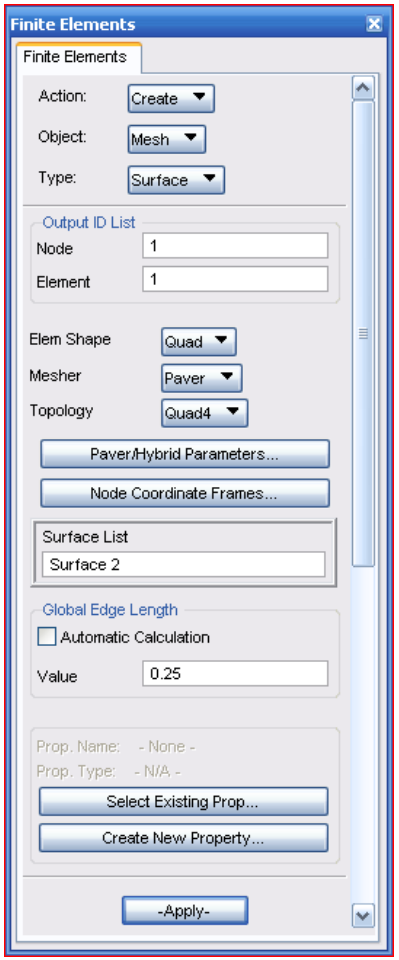
Copyright© 2012 MSC.Software Corporation

EXAMPLE: TESSELLATED SURFACE



This surface can be used as other surfaces, including meshing it with a finer mesh





Copyright© 2012 MSC.Software Corporation

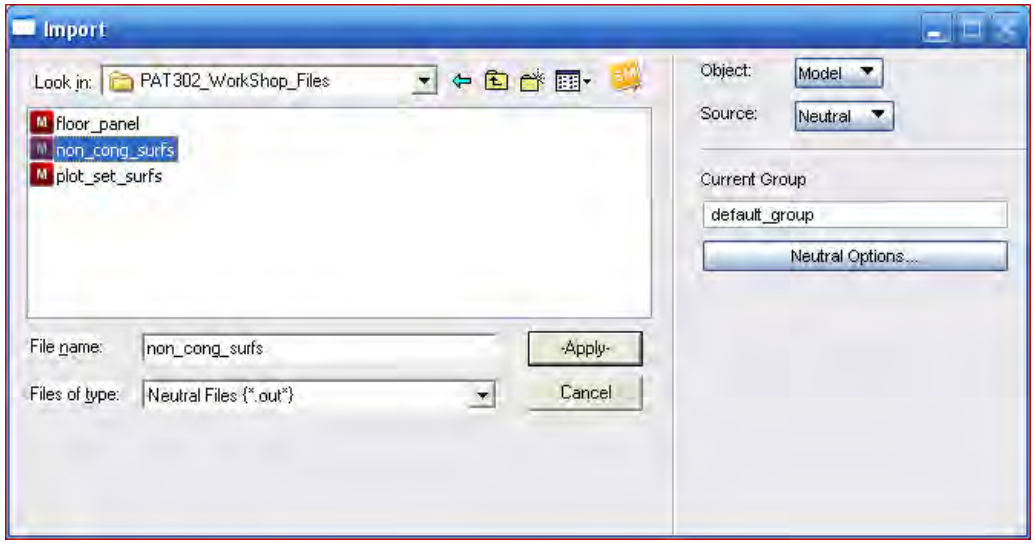
CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

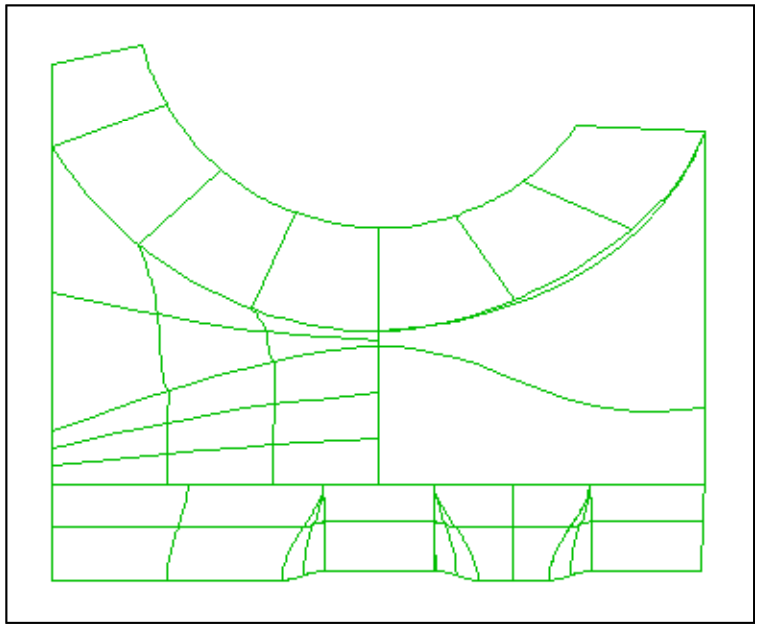
- **Create an Patran database**
  - Enter **tessellated.db** for *File name*
  - Set the *Tolerance* to **Based on Model**
  - Set the *Approximate Maximum Model Dimension* to **10.0**
  - Click **OK**
  - Note: The tolerance created is 0.005

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

- **Import the following Geometry**
  - Select **Import…** under the *File* menu
  - Select **Neutral** as the *Source*.
  - Select **non_cong_surfs.out**.
  - Click **Apply**.

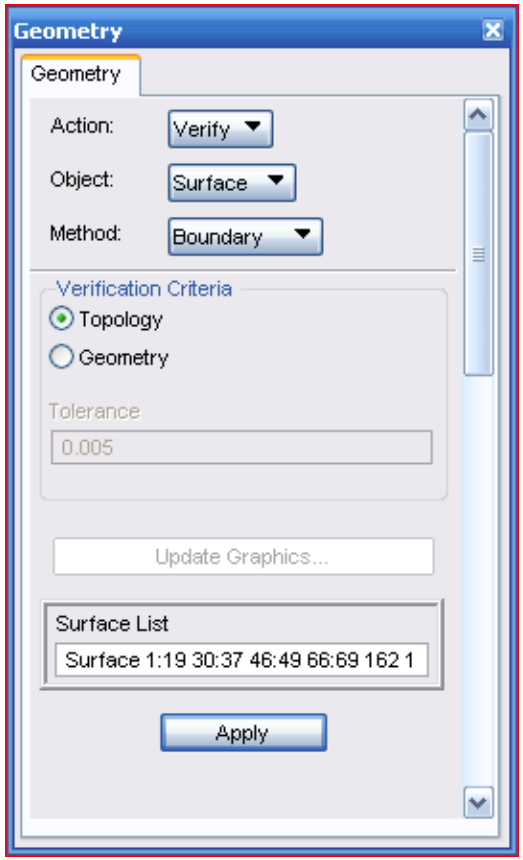




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

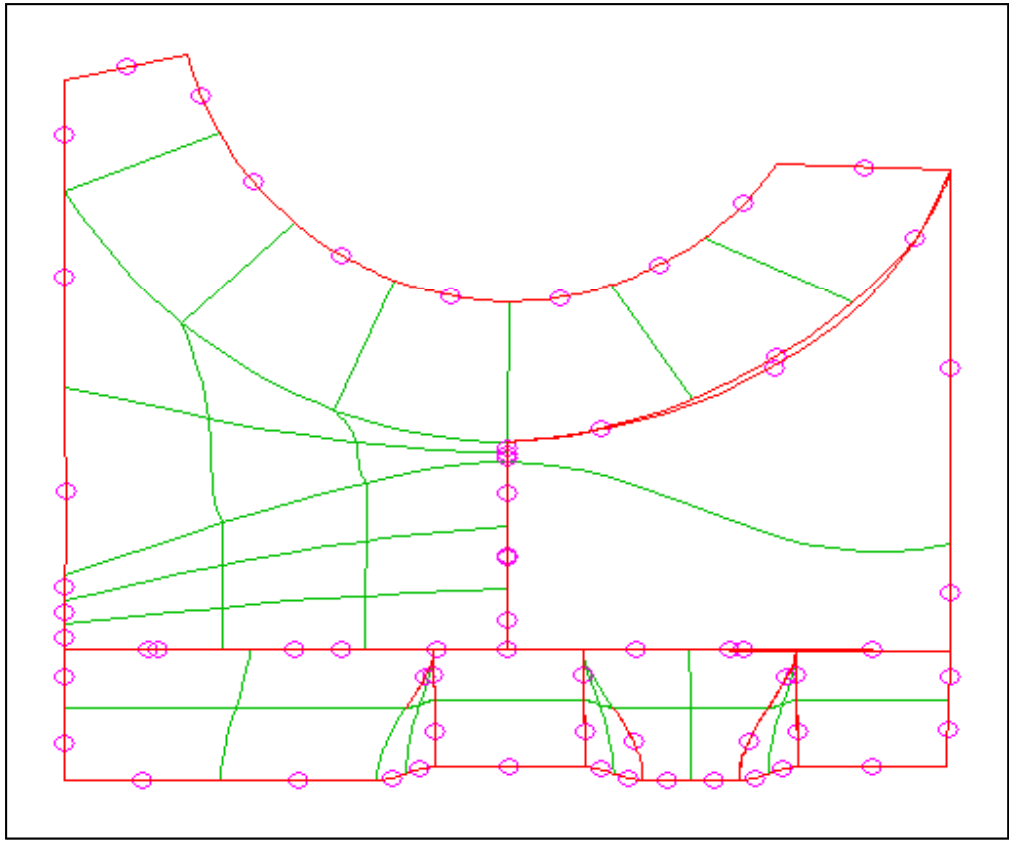
- **Determine where the surface free edges are**
  - Under the *Geometry* tab click on **Verify** from the *Geometry Actions* group.
  - Select all the surfaces for *Surface List*.
  - Click **Apply**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

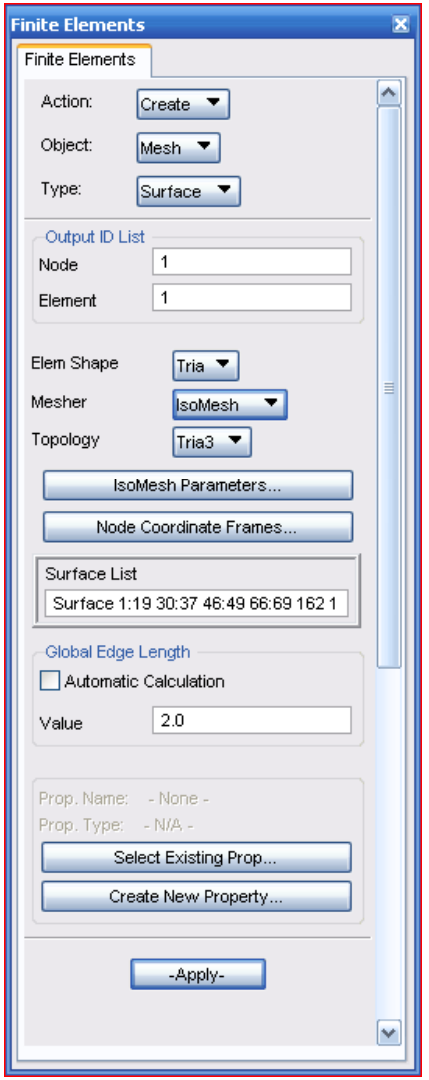
- **Note that there are numerous locations where there are surface free edges (shown by a Magenta circle).**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

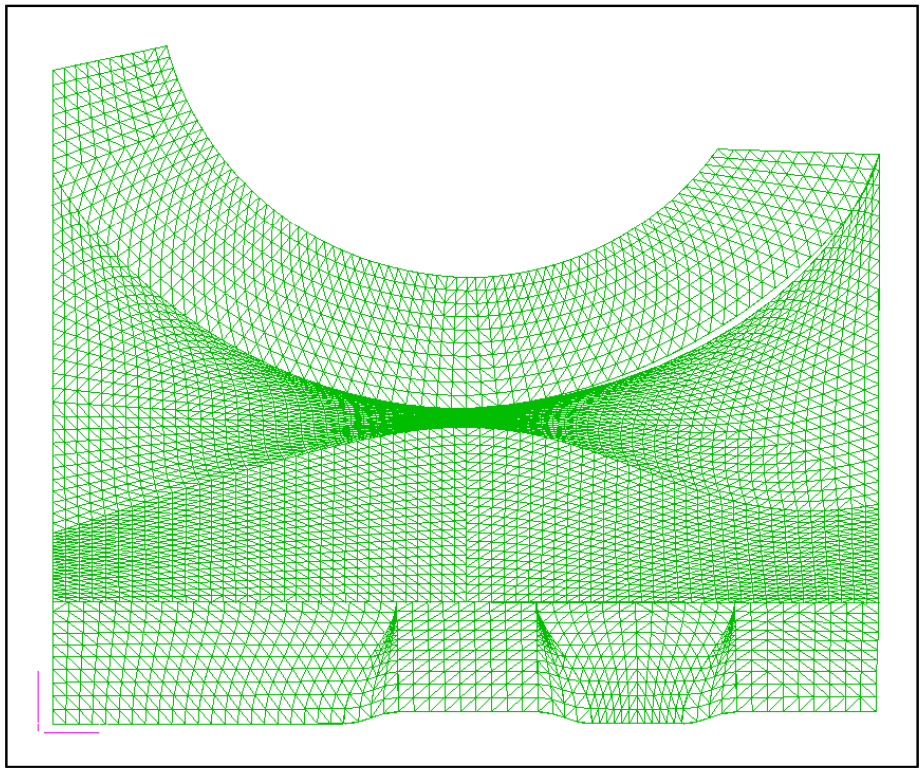
- **Create Tria3 mesh using the IsoMesh (IsoMesh can be used because all the surfaces are parametric)**
  - Under the **Meshing** tab click on *Surface* in the **Meshers** group.
  - Select **Tria** for *Elem Shape*.
  - Select **IsoMesh** for *Mesher*.
  - Select **Tria3** for *Topology*.
  - Select all the surfaces in the display for *Surface List*.
  - Deselect **Automatic Calculation**.
  - Enter **2.0** for *Global Edge Length*.
  - Click **Apply**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

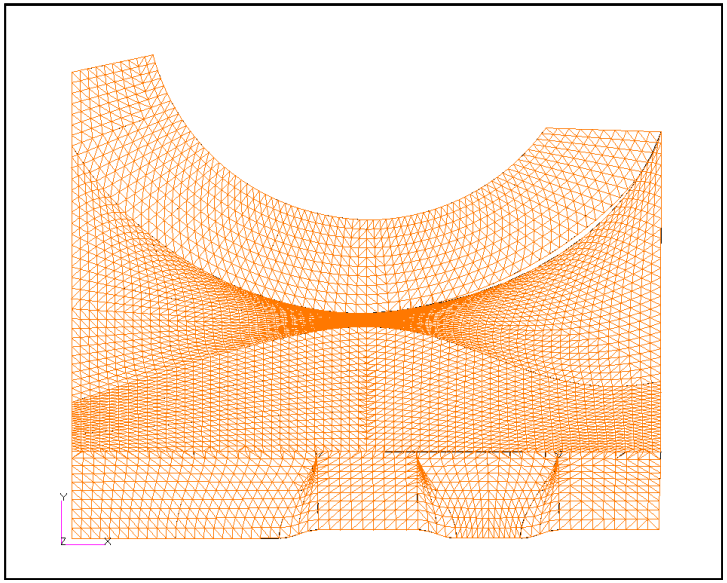
- **All surfaces were meshed with 2D tria3 elements.**

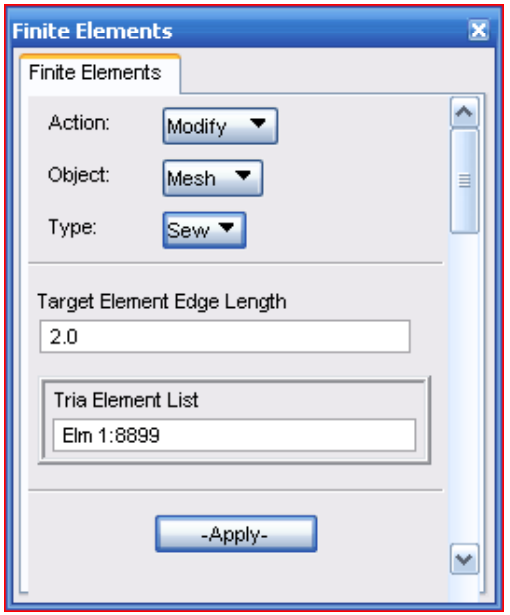


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

- **Sew the elements together. This command is similar to the Equivalence Command.**
  - Under the *Meshing* tab click on **Modify** in the *FEM Actions* group. Set the *Object* and *Type* to **Mesh** and **Sew** respectively.
  - Enter **2.0** for *Target Element Edge Length* (TEEL).
  - Select all the tri elements for *Tria Element List*.
  - Click **Apply**.





Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

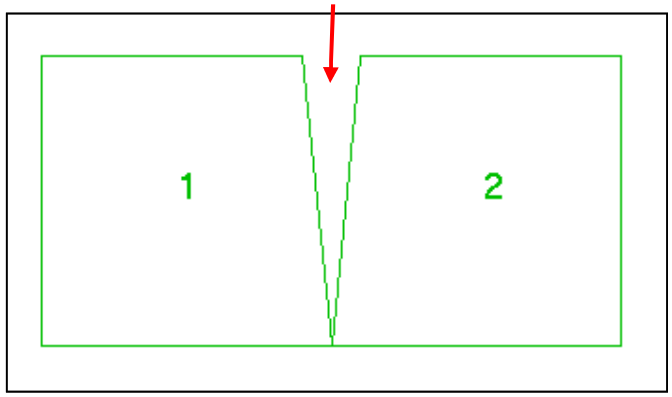
- **What does “Sew” do?**
  - Equivalences finite element nodes
    - If the target element edge length is approximately less than the gap size the only thing done is that nodes are equivalenced
  - Creates Tria elements to fill gaps
    - If the target element edge length is large enough, tria elements are created to fill gaps
  - The quality of the Tria elements created may not be very good, but it is acceptable because a surface is to be created from them and the original Tria-s; after the surface is created, the Tria mesh will be deleted.

Copyright© 2012 MSC.Software Corporation

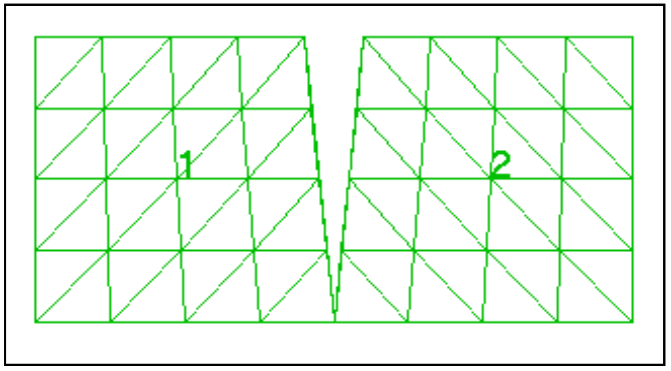
CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

- **Sew to just equivalence**

Gap (0.2)

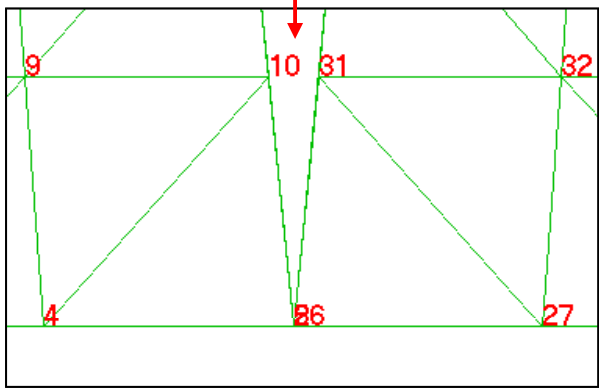


1) Two Surfaces w/ Gap

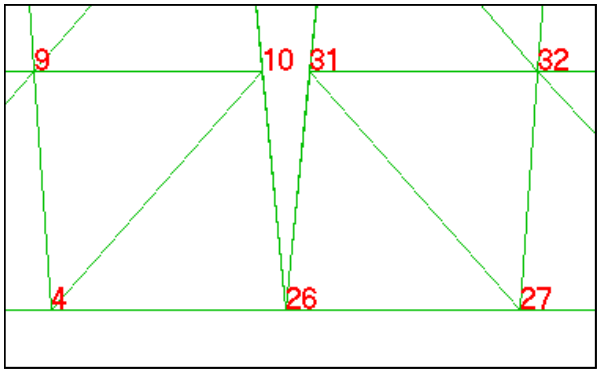


2) Tri Mesh on Surfaces (GEL = 0.25)

Gap



3) Before Sew (TEEL = 0.1)



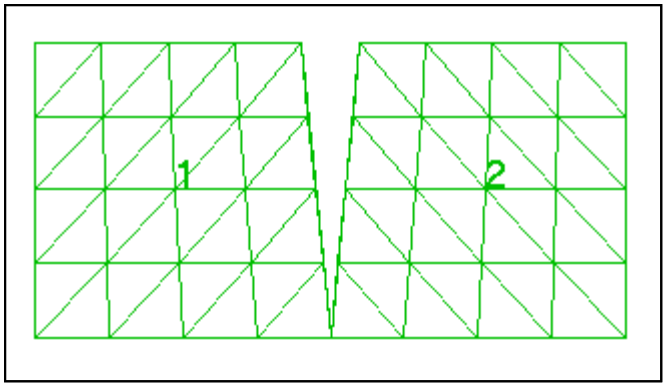
4) After Sew

Copyright© 2012 MSC.Software Corporation

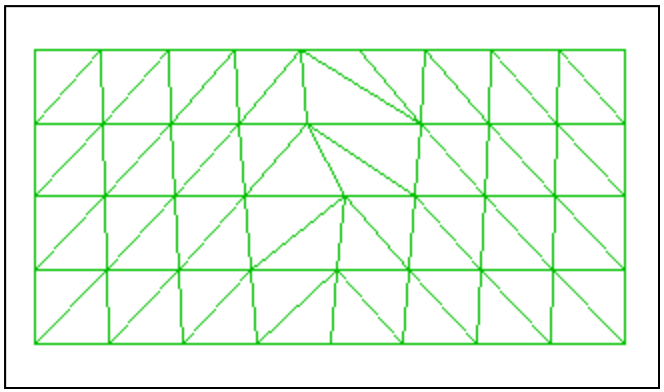
CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

- **Sew to equivalence and create tri elements**

Before Sew (TEEL = 2.0)



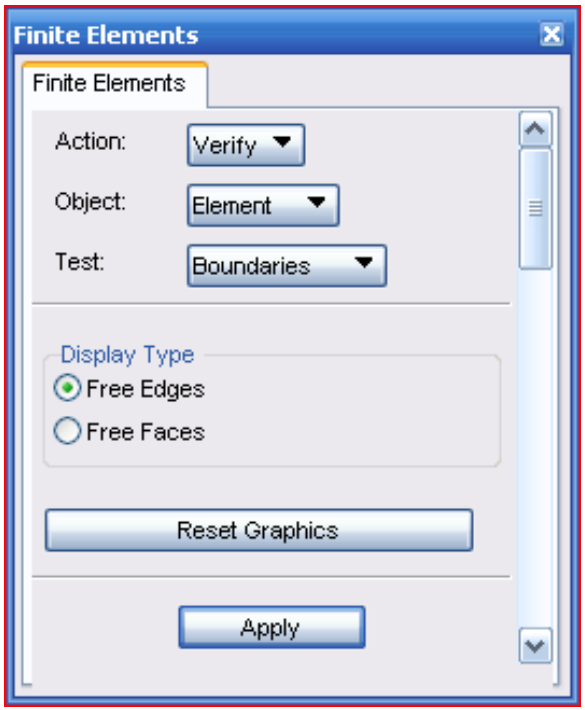
After Sew (Geometry Erased)

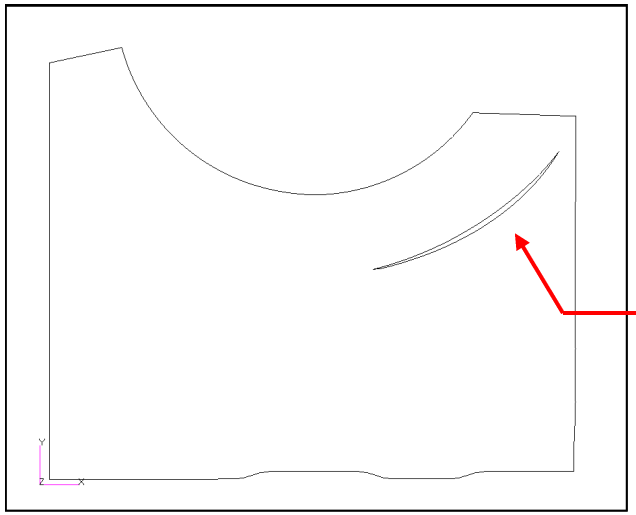


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

- **Now, see where the element free edges are**
  - Under the *Meshing* tab click on **Verify** in the *FEM Actions* group.
  - Select **Free Edges** for *Display Type*.
  - Click **Apply**.
  - The effect of having a hole in the mesh is discussed later.



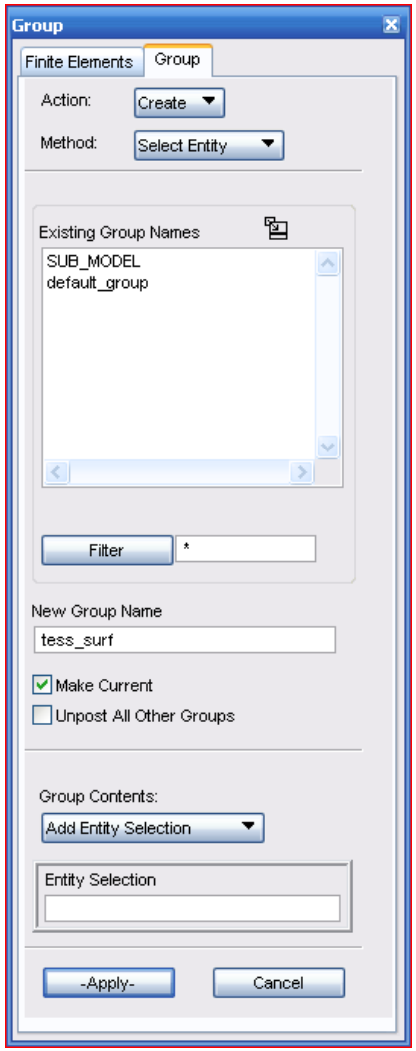


Hole in Tria3 Mesh

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

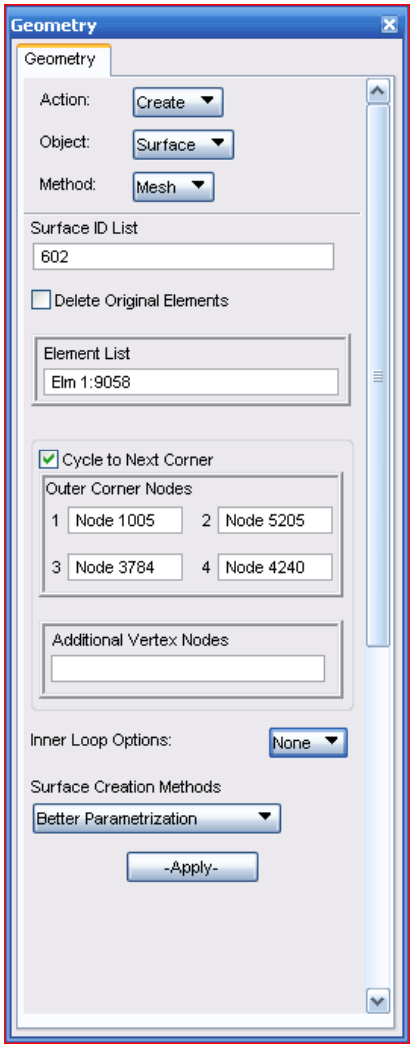
- **Create a group for the surface to be created from the tria3 elements. Make the group "current", so when the surface is created it will be placed in it.**
  - Select **Create…** under the *Group* menu
  - Enter **tess_surf** for the *New Group Name*.
  - Check **Make Current**.
  - Click **Apply**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

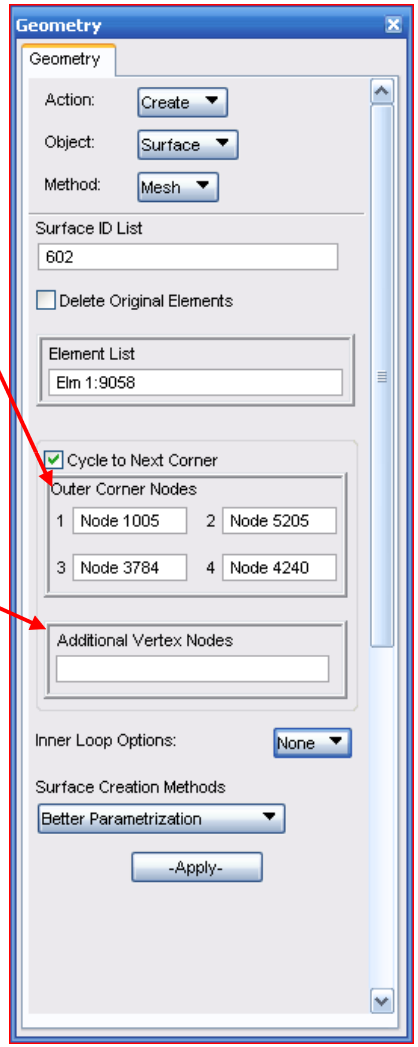
- **Create a surface from the 2D tria3 mesh**
  - Under the *Geometry* tab click on **Mesh** from the *Surfaces* group.
  - Select the tri3 elements for *Element List*.
  - Select four corner nodes for *Outer Corner Nodes*. No additional nodes are selected, so it will create a parametric surface.
  - Select **None** for *Inner Loop Options*.
  - Click **Apply**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

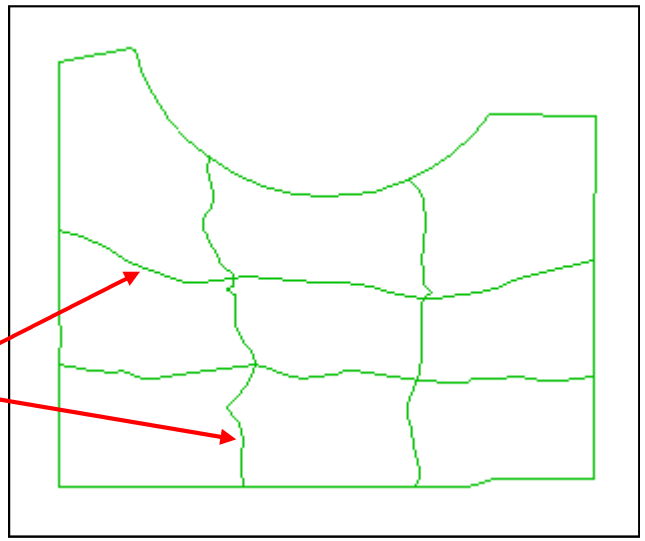
- **Select (control point) nodes and additional vertex nodes**
  - Four (control point) nodes are required
    - They should be physically spaced apart from each other
  - Additional vertices may be needed to adequately define the edges of the surface being created
    - If the necessary vertex nodes are not included, the resulting surface may be of poor quality – the surface edges may not follow the tria mesh.

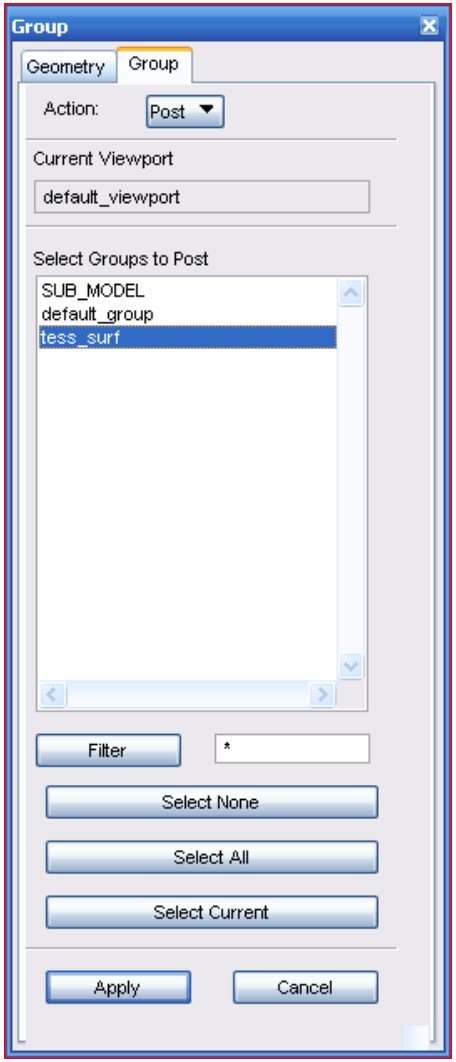


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

- **Display only the group with the new surface, tess_surf:**
  - Select **Post…** under the *Group* menu
  - Select **tess_surf** for *Select Groups to Post*.
  - Click **Apply**.
  - Turn on the display lines.

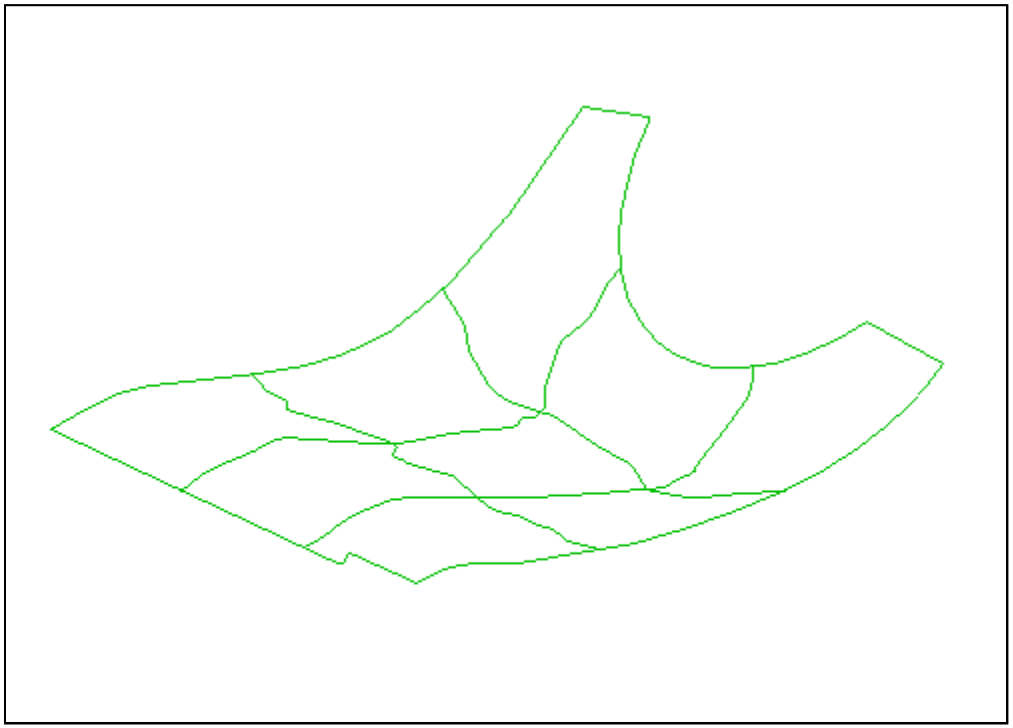




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

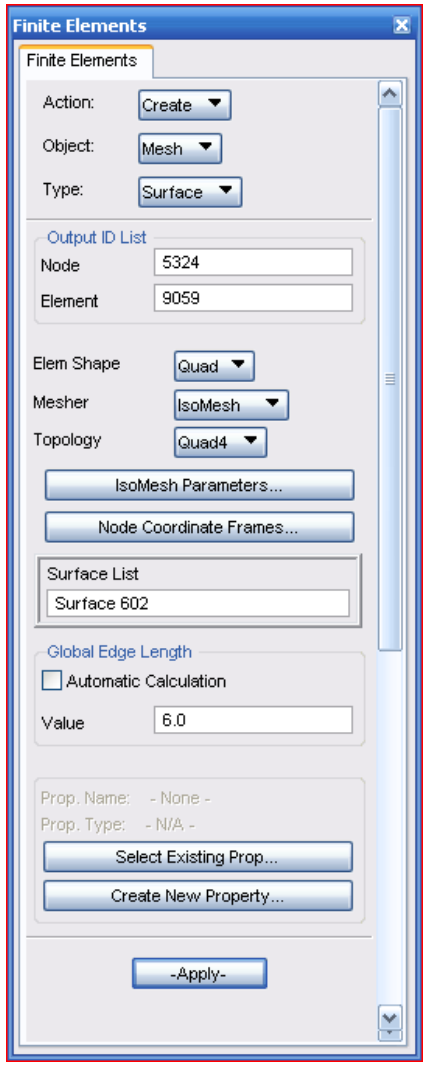
- **Change the view by holding down the middle mouse button, and rotating the model.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

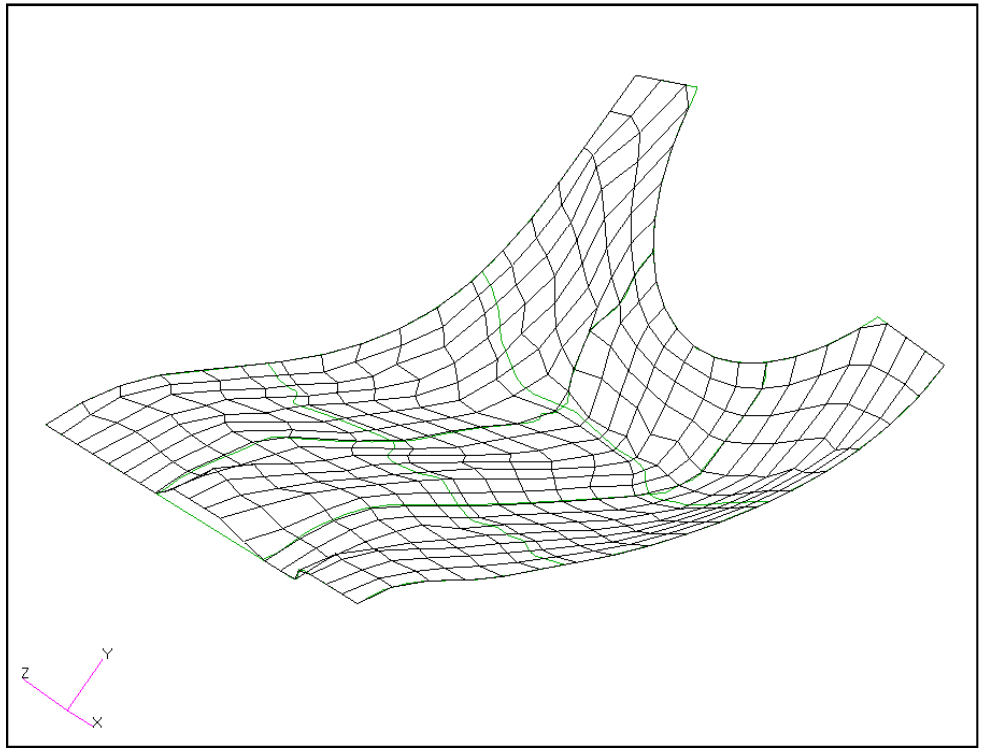
- **Mesh the new surface using IsoMesh (use larger element size)**
  - Under the *Meshing* tab click on **Surface** from the *Meshers* group.
  - Select **Quad** for *Elem Shape*.
  - Select **IsoMesh** for *Mesher*.
  - Select **Quad4** for *Topology*.
  - Select **Surface 602** for *Surface List*.
  - Deselect **Automatic Calculation** and enter **6.0** for *Global Edge Length*.
  - Click **Apply**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

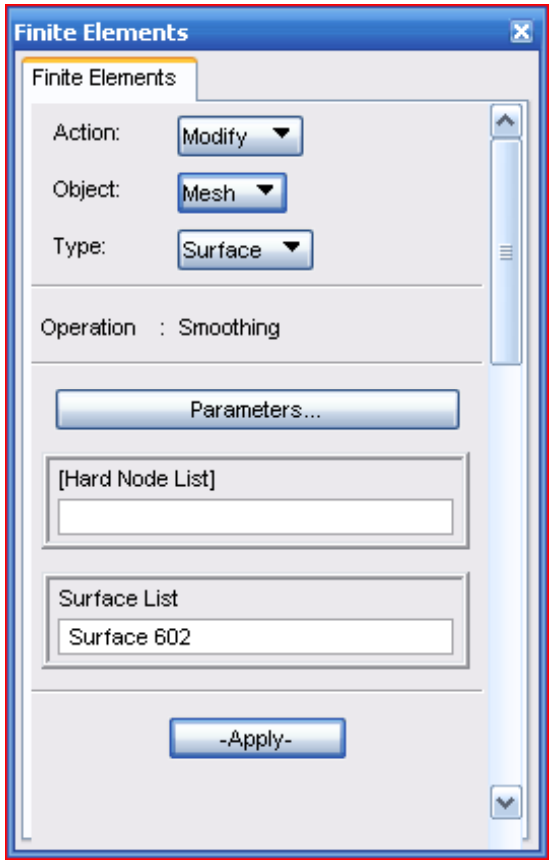
- **The model should now appear as follows:**



Copyright© 2012 MSC.Software Corporation

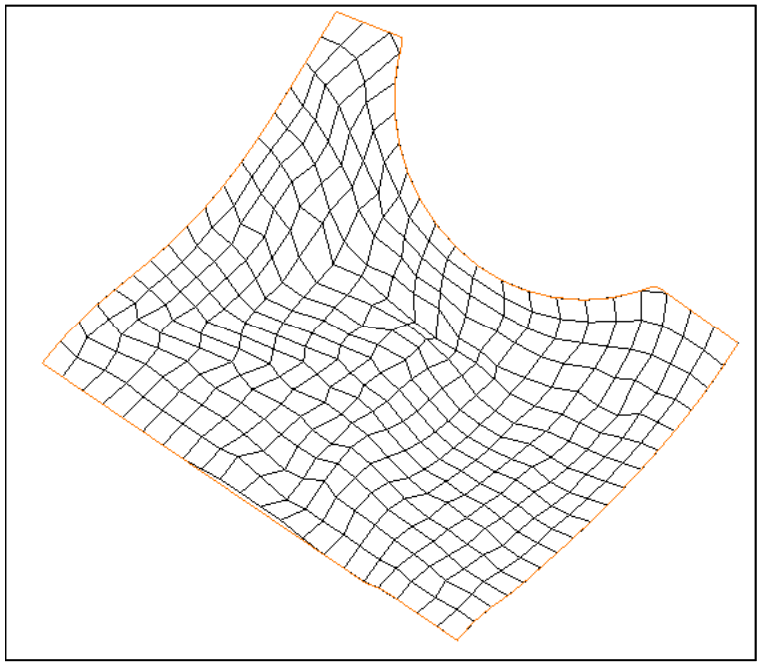
CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

- **Smooth the new 2D surface mesh using an application in Finite Elements**
  - Under the *Meshing* tab click on **Modify** from the *FEM Actions group*.
  - Select the newly created tessellated surface, **Surface 602**.
  - Click **Apply**.
  - Erase the geometry in order to get a better view of the mesh.

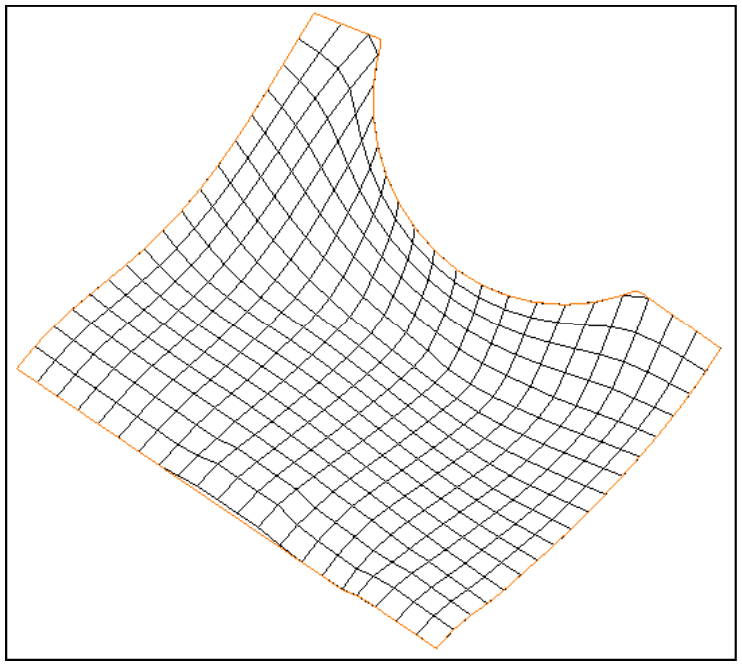


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES



Before Smoothing



After Smoothing

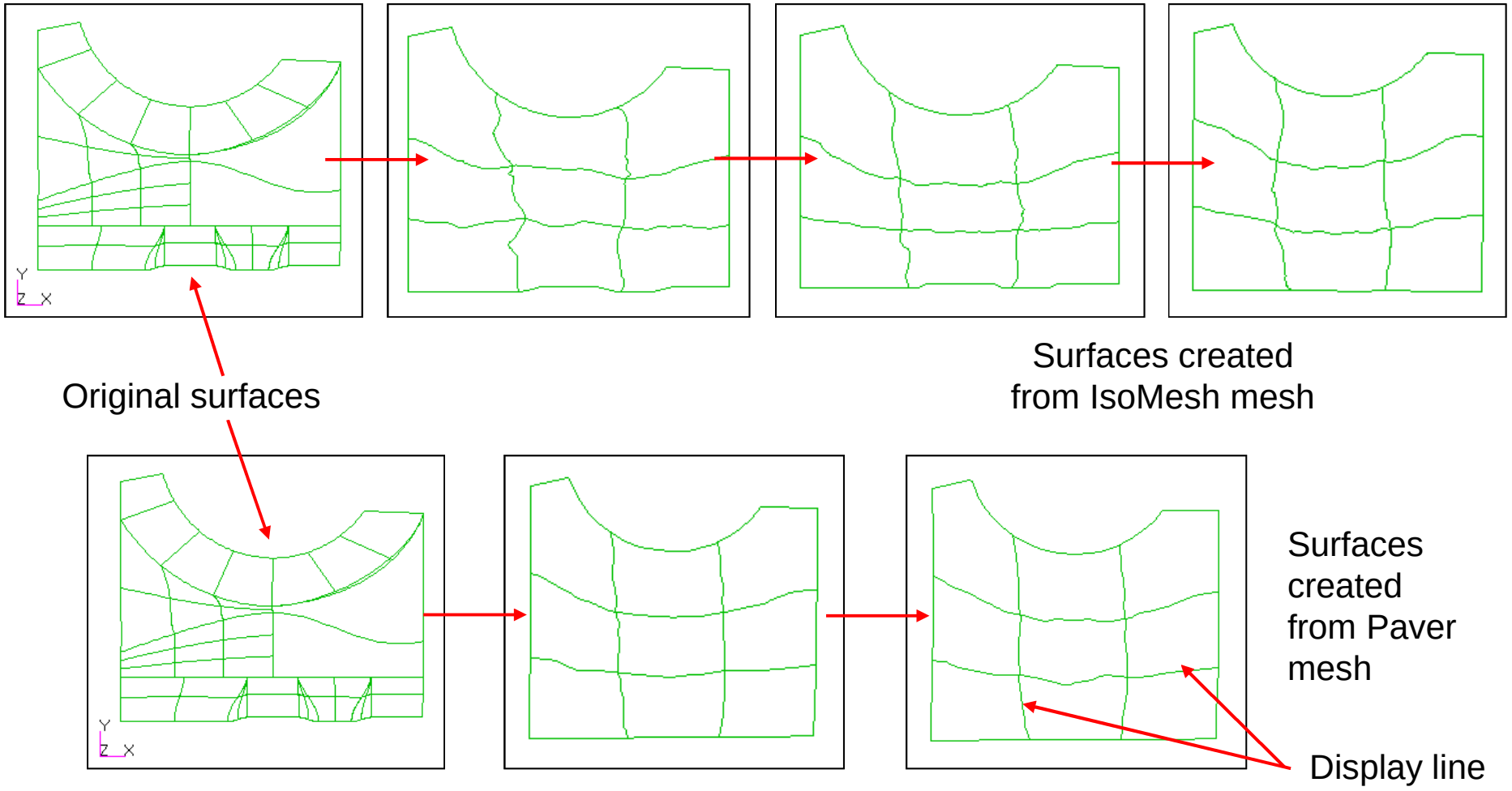
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

- **There is more than one way to improve a 2D mesh for a tessellated surface.**
- **Use an iterative approach to create an improved tessellated surface:**
  - Start with a 2D mesh
  - Create the surface
    - Under the *Geometry* tab, select **Mesh** from the *Surfaces* Group.
  - Mesh the new surface, deleting the previous 2D mesh.
  - Check the surface and mesh quality.
  - Repeat the process until the desired quality is obtained.
  - Then, delete the surface.

Copyright© 2012 MSC.Software Corporation

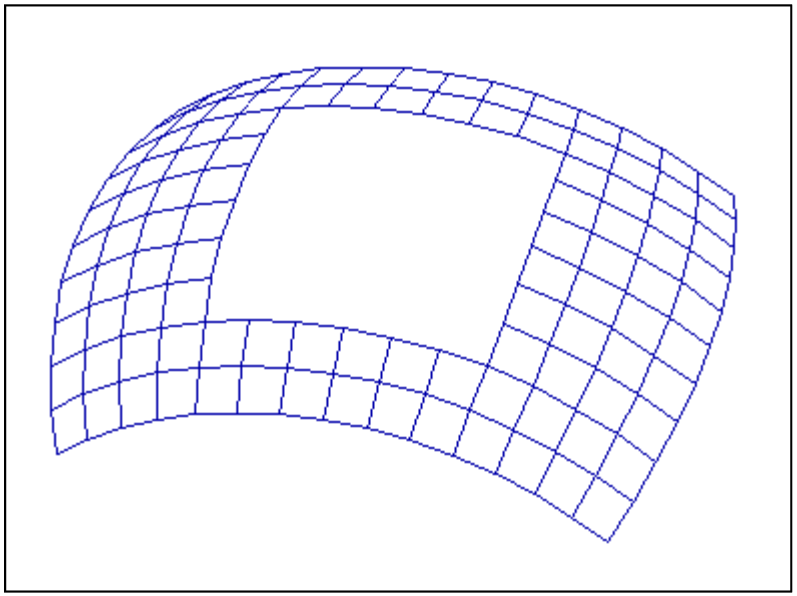
CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

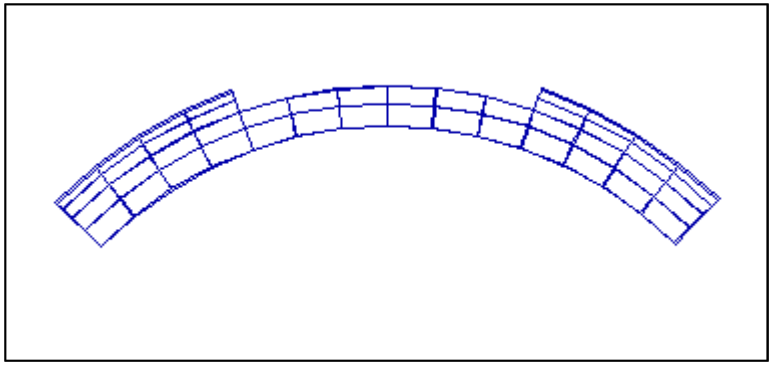


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

- **Create a surface from a mesh with holes in it**
  - Can include or exclude the holes in the mesh
- **For highly curved meshes, the created surface may be flat where the hole is**
  - May be necessary to close hole in mesh using elements

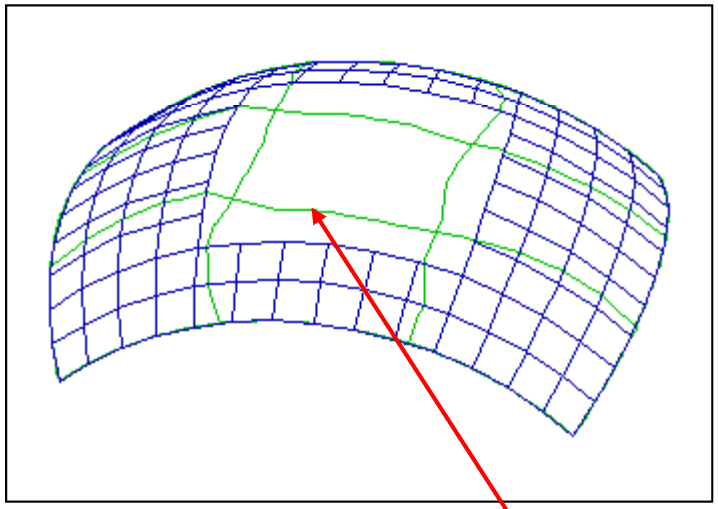




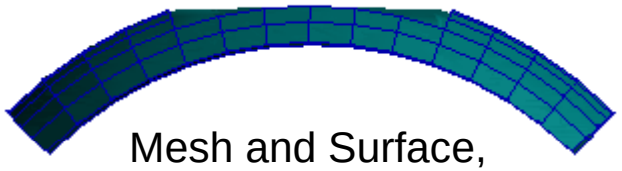
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CREATE 2D MESH FROM NON-CONGRUENT SURFACES

- **Surface created from mesh with hole, showing flat area**
  - Curvature too great for the size of the hole



Mesh and Surface (Green Display Line)



Mesh and Surface, Showing Flat Area

Surface, Showing Flat Area on Top

Copyright© 2012 MSC.Software Corporation

EXERCISES

- Perform Workshop 3 “Tessellated Surface” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

NON-CONGRUENT MESHES AND MESH MODIFICATION

- **Non-congruent geometry or geometry with severe curvature issues can produce poor meshes.**
- **Sometimes, a few finite elements must be modified in light of verification checks or analyzing stress results.**
- **Patran provides tools to edit elements and/or nodes.**

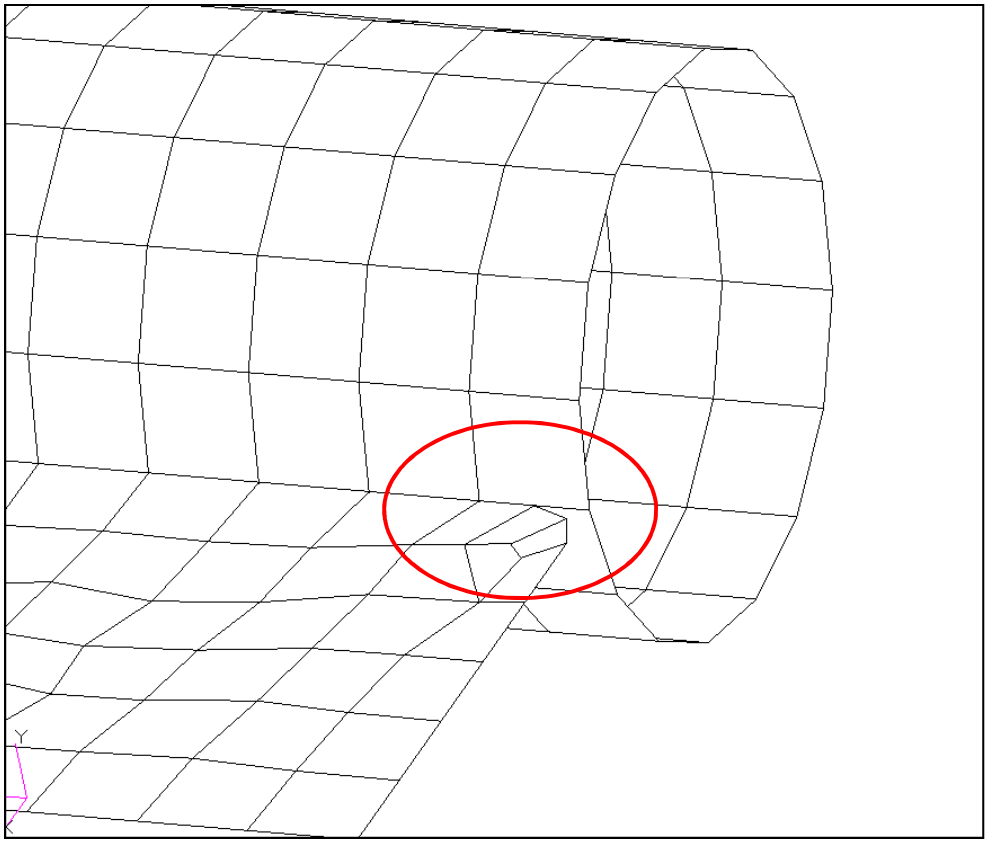
Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated Surface	Iso Mesh	Paver Mesh	Sheet Body	Composite Surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	

Copyright© 2012 MSC.Software Corporation

NON-CONGRUENT MESHES AND MESH MODIFICATION

Consider a case where a filleted corner has been modeled on one assembly but not the other.

Rather than rebuild the geometry, it may be expedient to use direct mesh editing techniques as shown in the following slides.



Copyright© 2012 MSC.Software Corporation

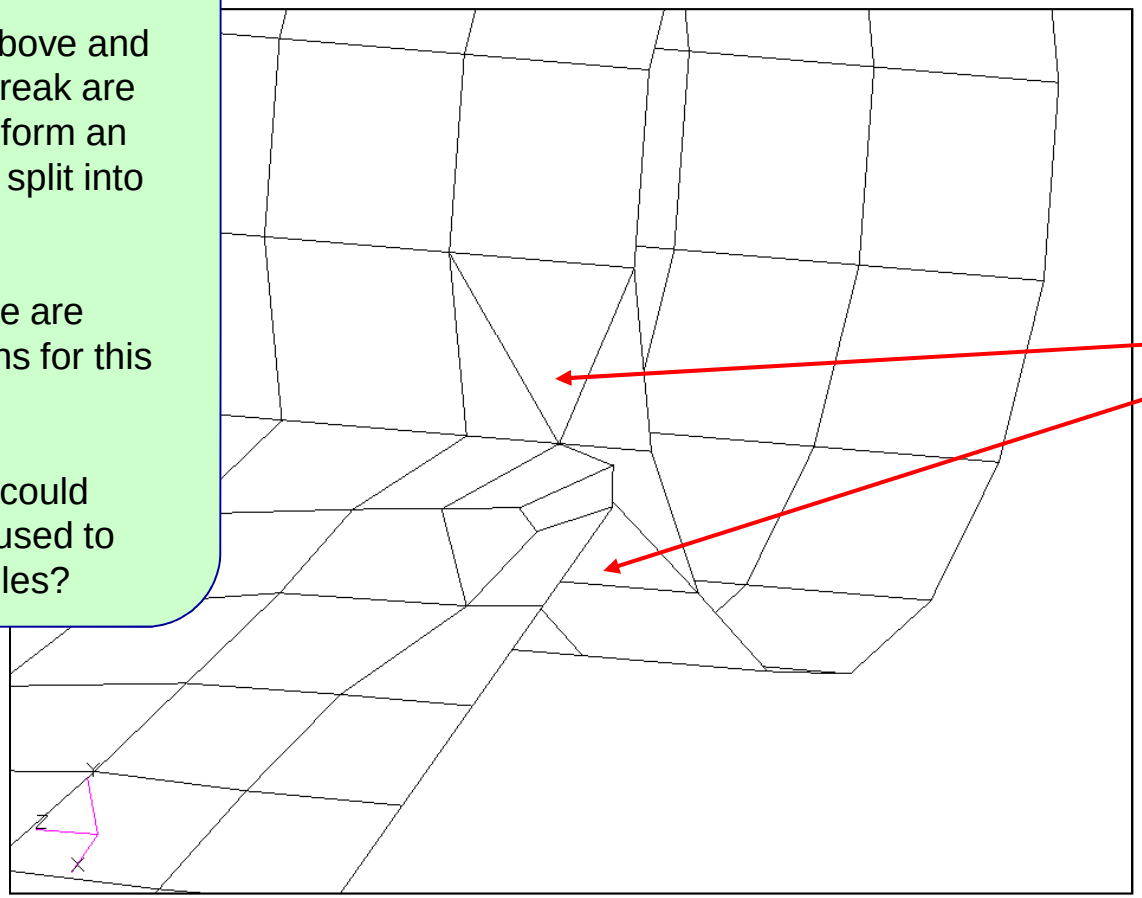
NON-CONGRUENT MESHES AND MESH MODIFICATION

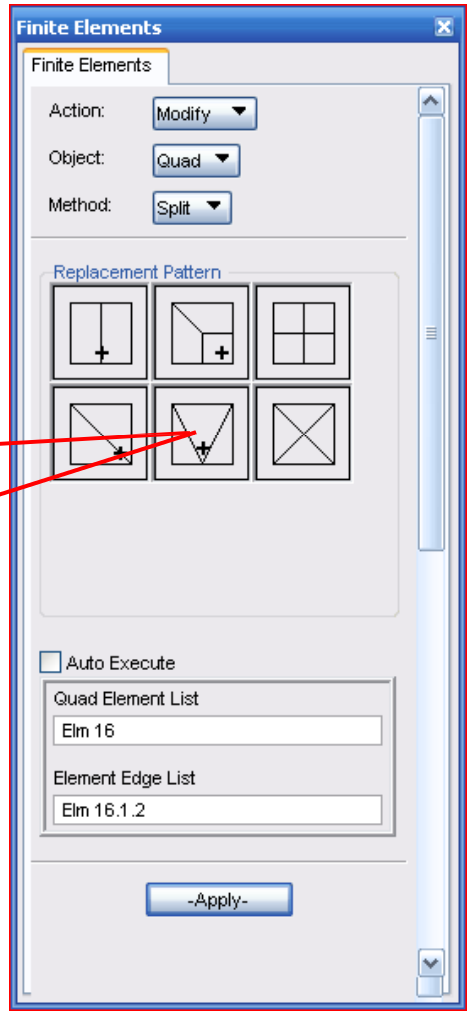
First Stage:

The quad above and below the break are modified to form an appropriate split into triangles.

Notice, there are many options for this method.

Which split could have been used to avoid triangles?





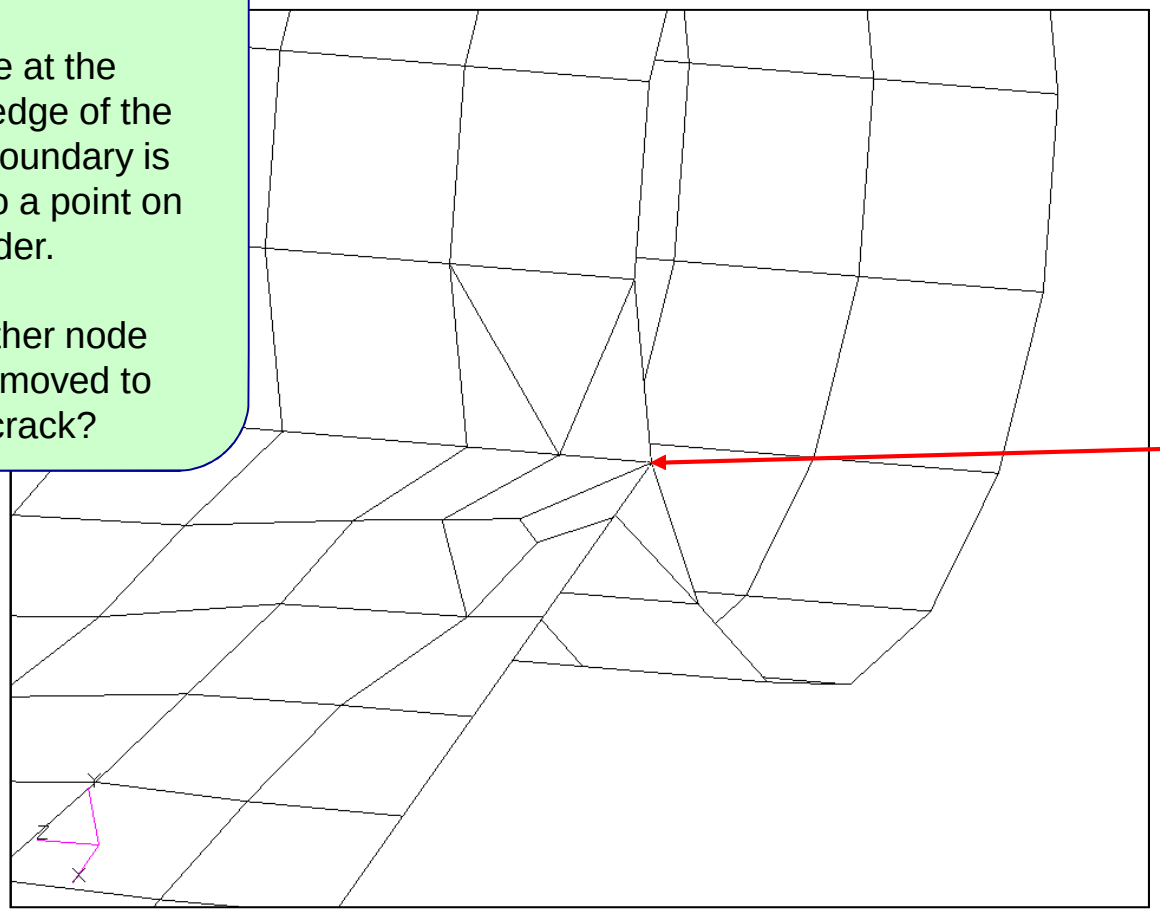
Copyright© 2012 MSC.Software Corporation

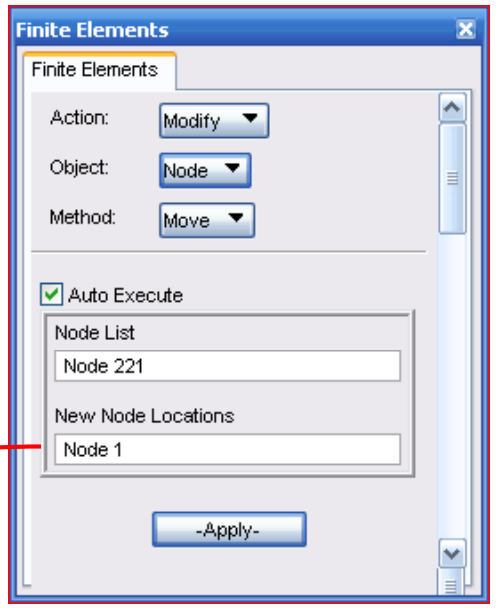
NON-CONGRUENT MESHES AND MESH MODIFICATION

Second Stage:

The node at the original edge of the curved boundary is moved to a point on the cylinder.

Which other node must be moved to avoid a crack?





Copyright© 2012 MSC.Software Corporation

SHEET BODY MESHER

- **The Sheet Body mesher can create a high quality mesh for a set of geometric surfaces that contain sliver and/or skewed surfaces.**
  - This mesher is for meshing a set of congruent surfaces. A single mesh is created for the domain of all the surfaces.
  - The mesher can be controlled to ignore select surface edges internal to the perimeter of the set of surfaces.
  - A conventional mesher may create poorly-shaped elements because of poorly shaped surfaces.
  - The effort to modify congruent yet complex surfaces, to create better shaped surfaces, can be time consuming.

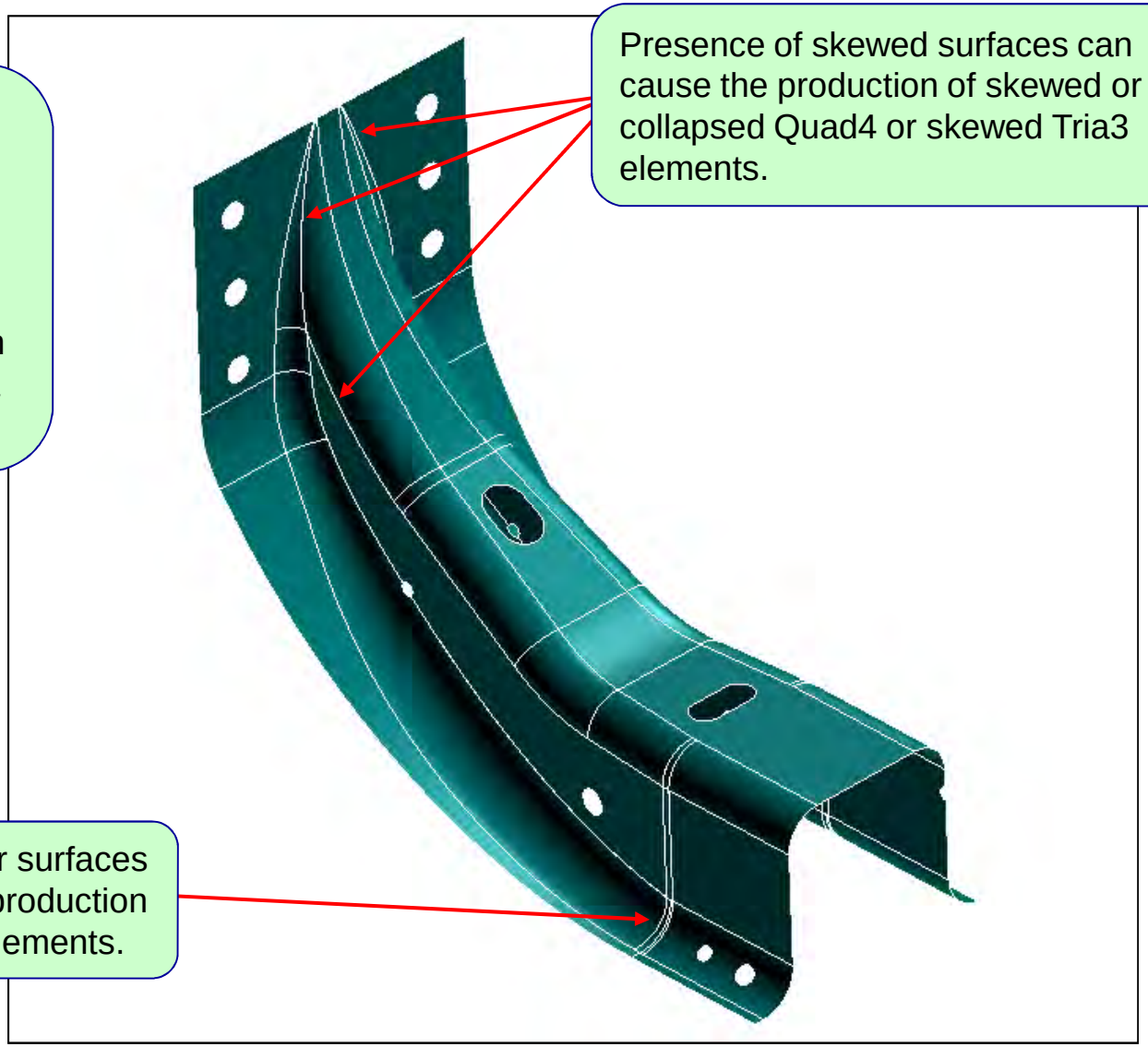
Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh on Mesh	Mesh Editing	Tessellated Surface	Iso Mesh	Paver Mesh	Sheet Body	Composite Surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	

Copyright© 2012 MSC.Software Corporation

EXAMPLE: SHEET BODY MESHER

The geometry shown has several problem areas.

Use the Sheet Body mesher to mesh across these problem features to produce a good quality mesh.

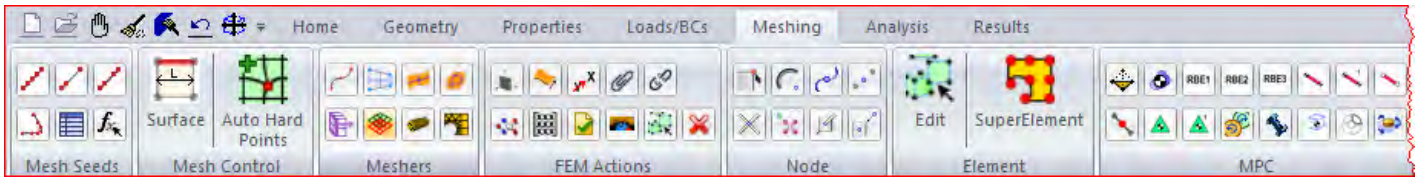


Presence of skewed surfaces can cause the production of skewed or collapsed Quad4 or skewed Tria3 elements.

Presence of sliver surfaces can result in the production of sliver Quad4 elements.

Copyright© 2012 MSC.Software Corporation

EXAMPLE: SHEET BODY MESHER



Use the Sheet Body mesher:

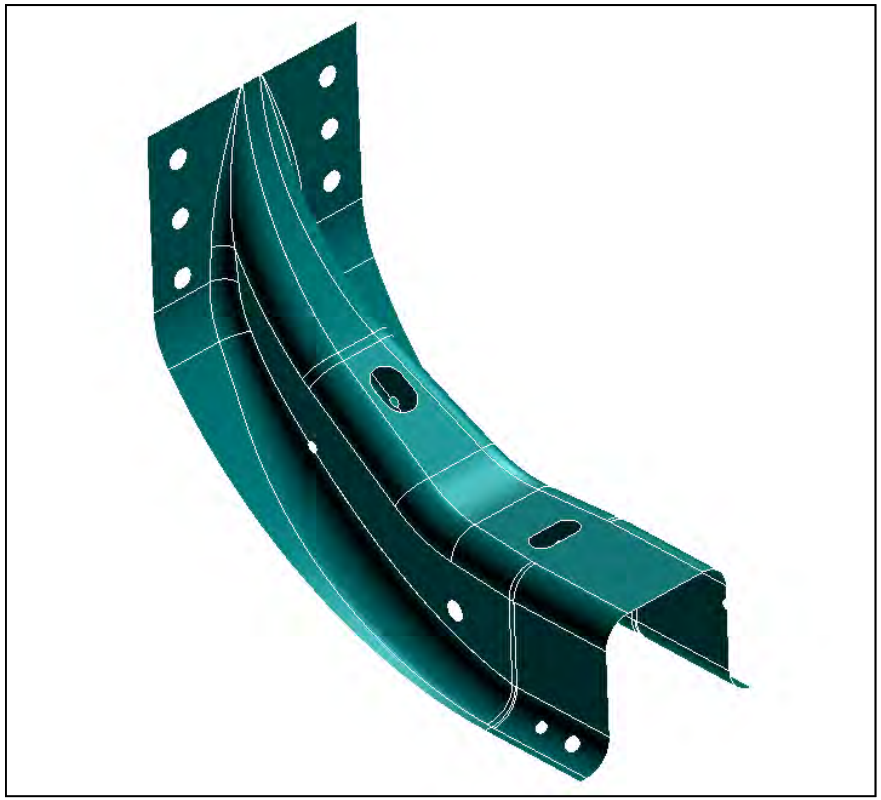
Under the Meshing tab click on **Sheet Body** from the Meshers group.

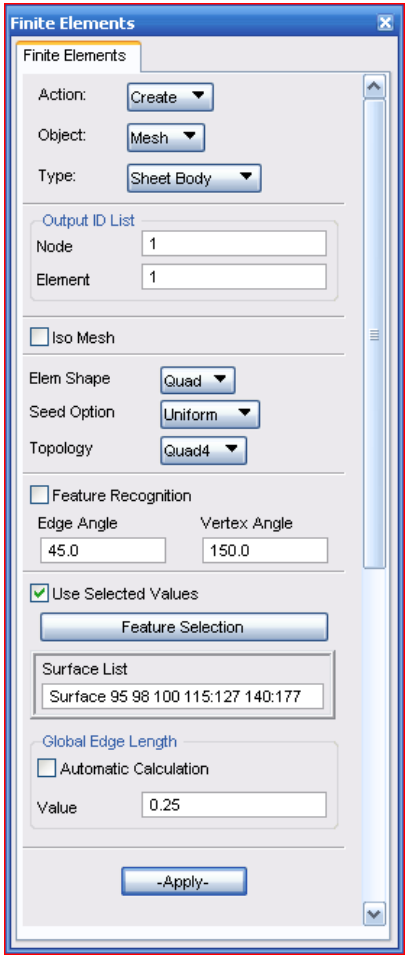
Select the element type.

Select the surfaces.

Enter the desired element size.

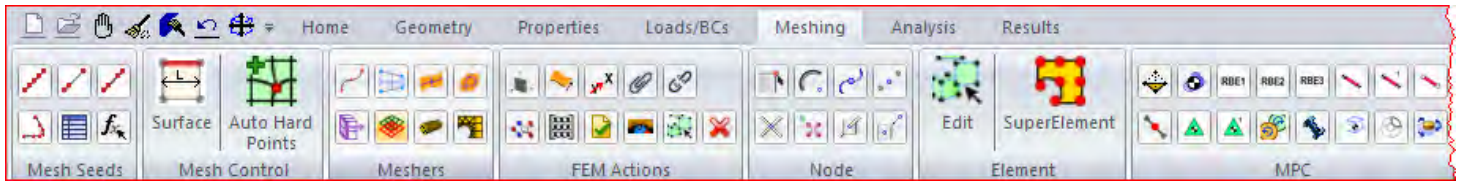
Click **Apply**.



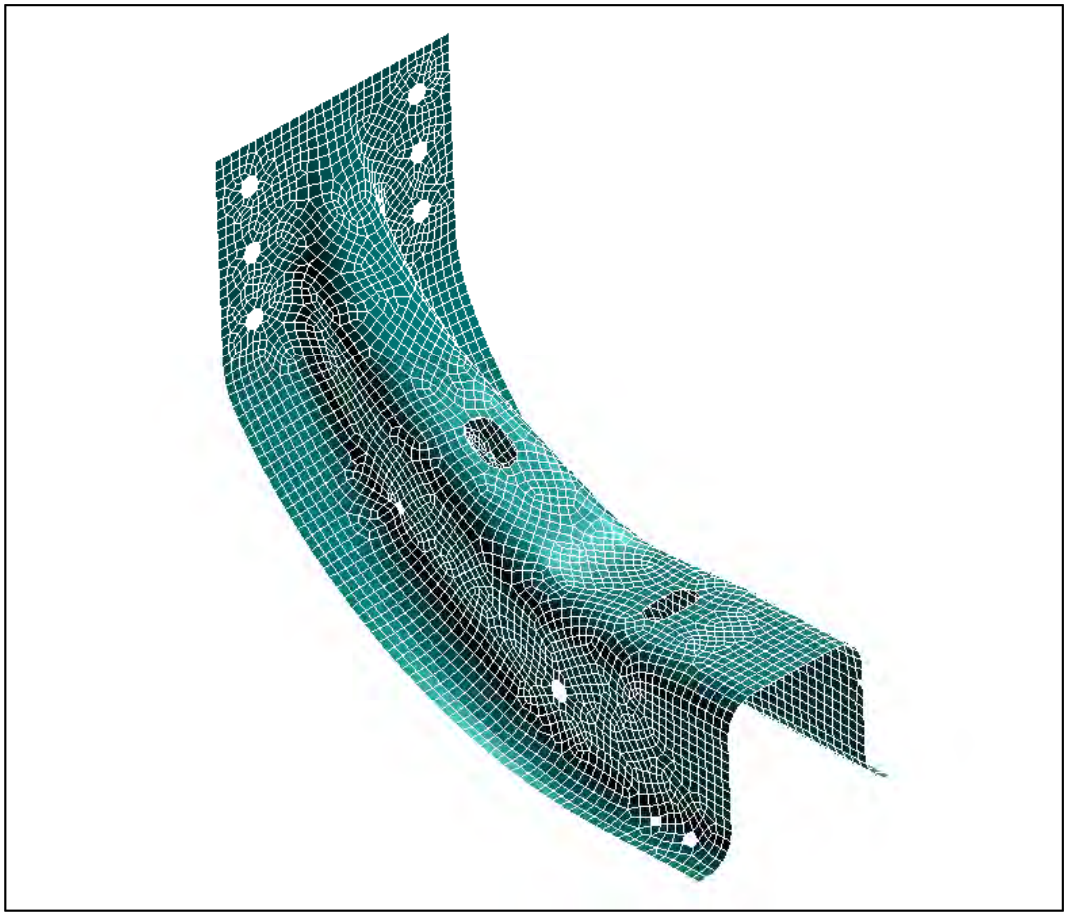


Copyright© 2012 MSC.Software Corporation

EXAMPLE: SHEET BODY MESHER

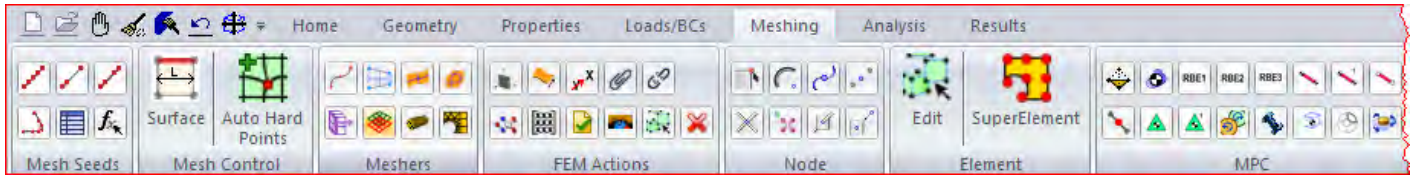


The Sheet Body mesher ignored edges of skewed and sliver surfaces, preventing the creation of skewed and sliver elements.



Copyright© 2012 MSC.Software Corporation

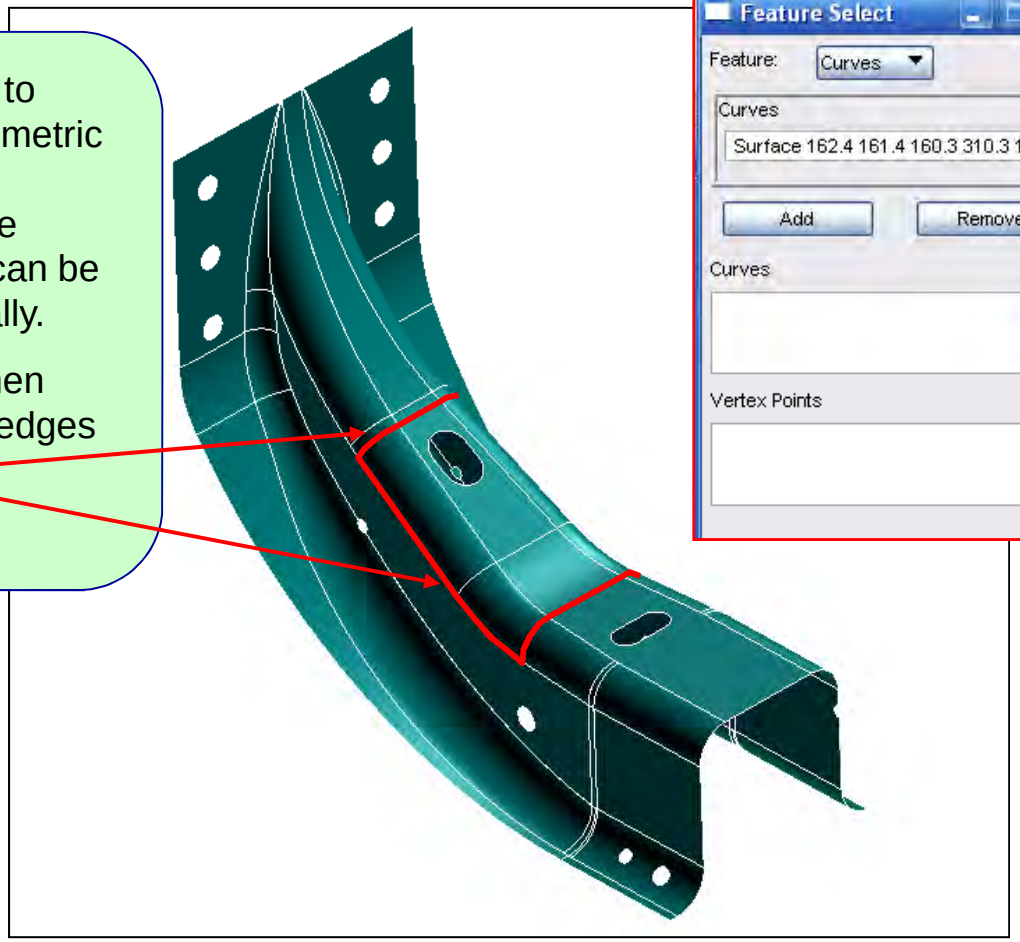
EXAMPLE: SHEET BODY MESHER

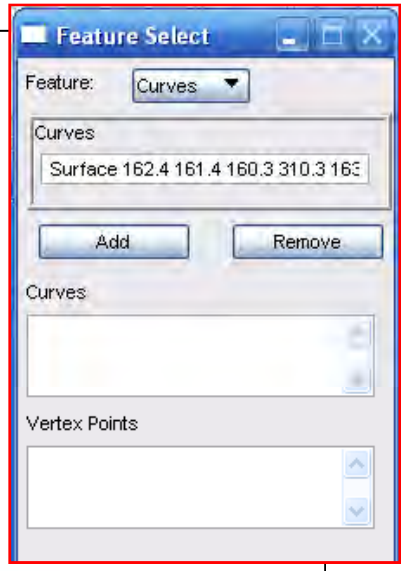


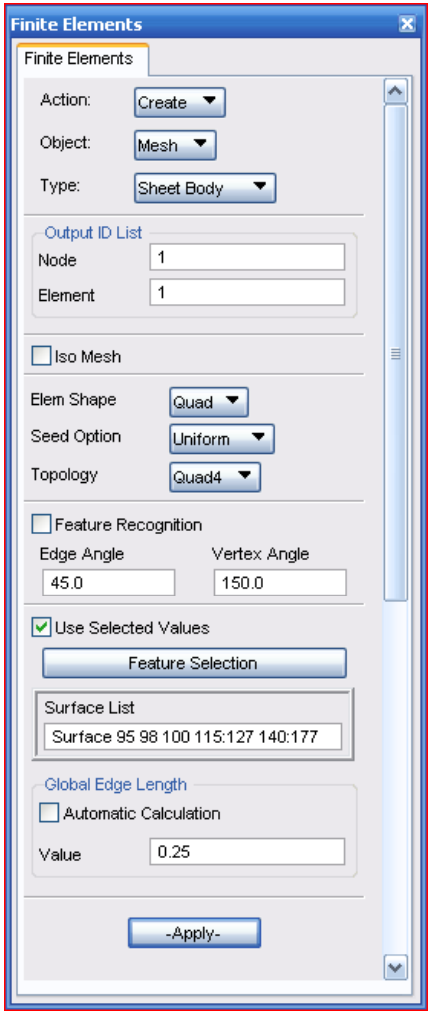
If the mesh needs to follow specific geometric features, Feature Recognition can be used, or features can be specified individually.

Undo the mesh, then select the feature edges as shown.

Click **Apply**.

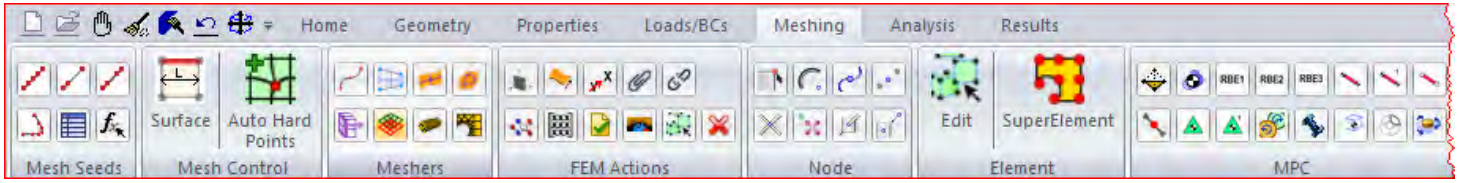


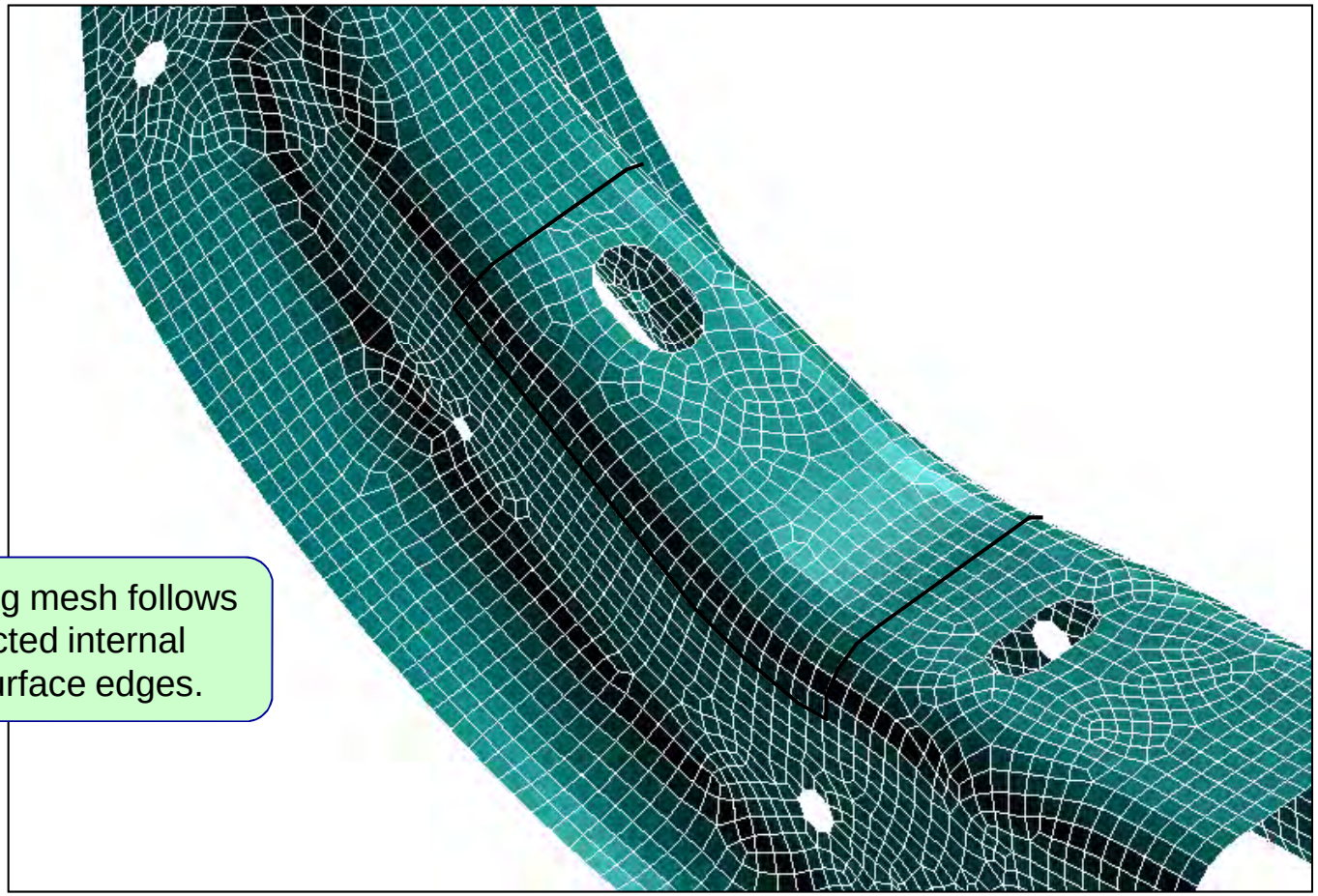




Copyright© 2012 MSC.Software Corporation

EXAMPLE: SHEET BODY MESHER



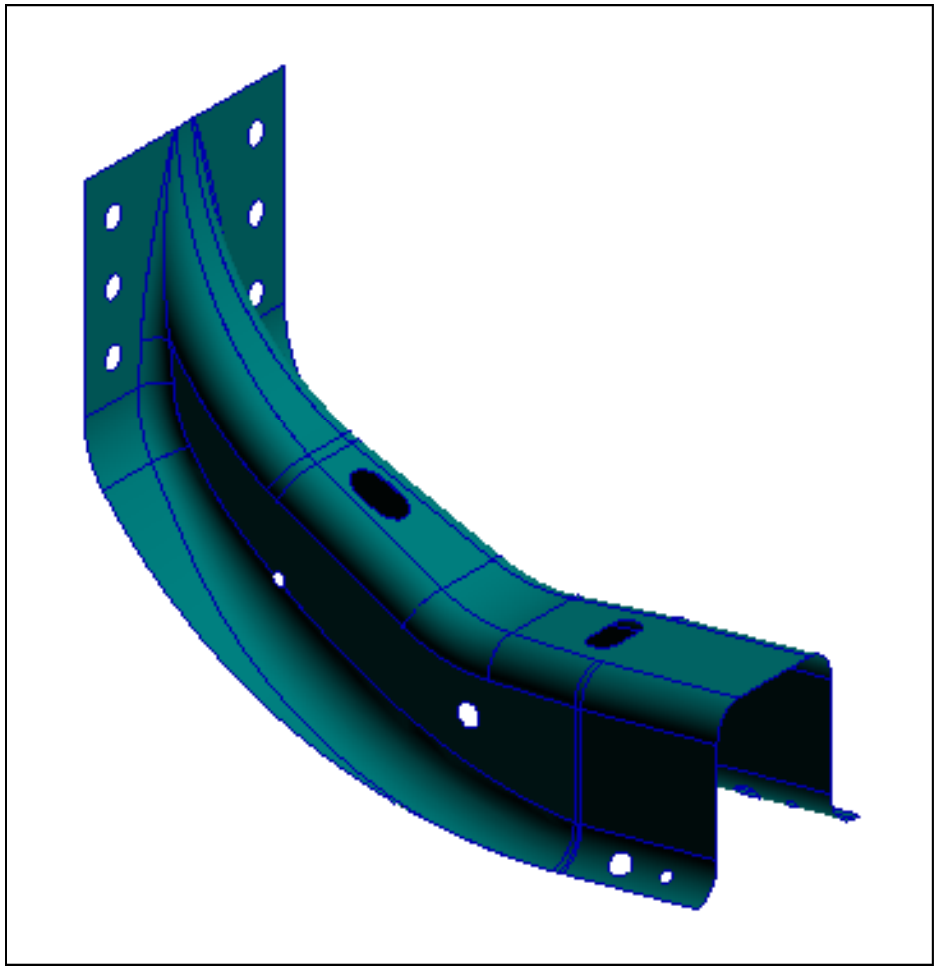


The resulting mesh follows the selected internal feature surface edges.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: SHEET BODY MESHER

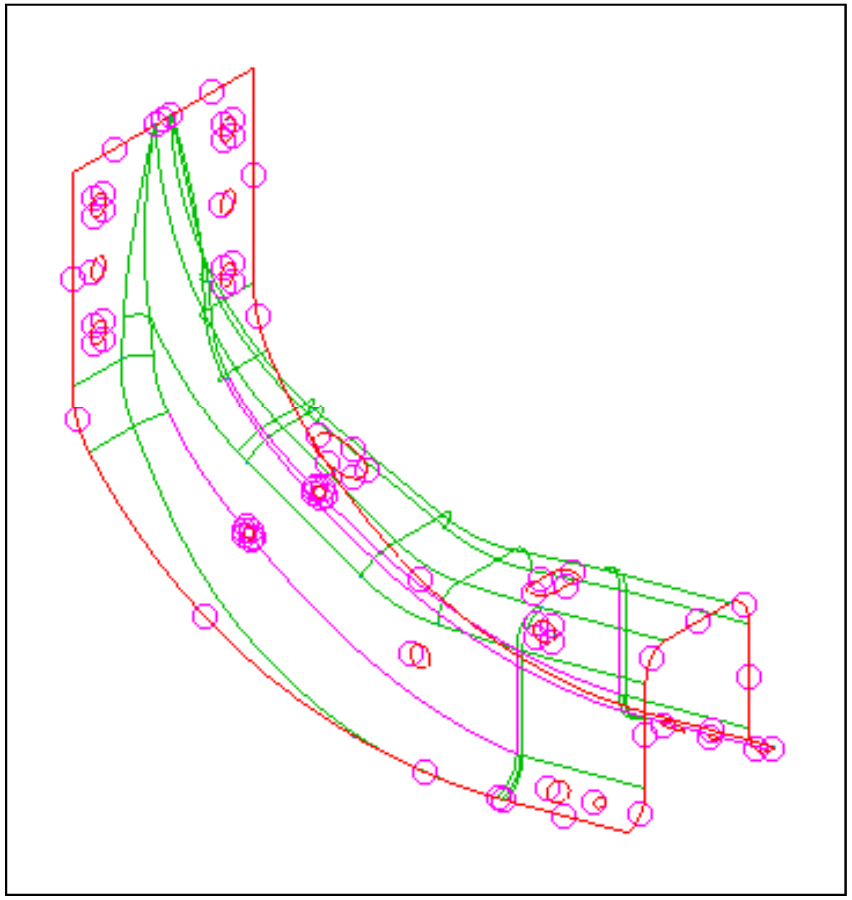
- Import the geometric surfaces file, **sheetbody_surfs.igs**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: SHEET BODY MESHER

- **Look for areas of non-congruency**

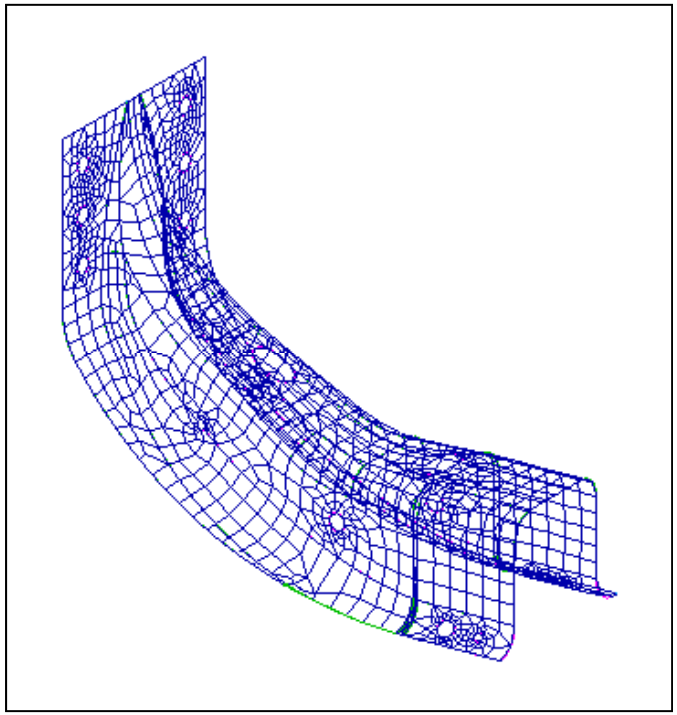


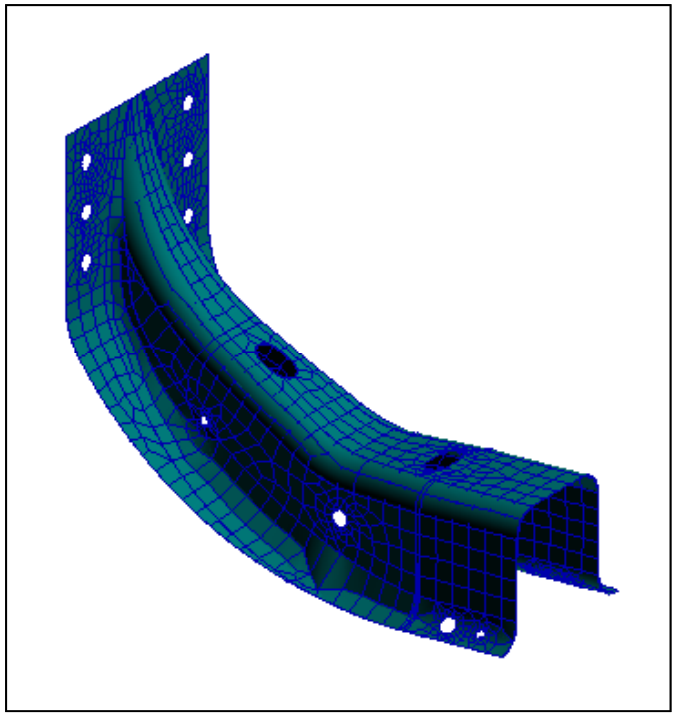
- **There are none.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: SHEET BODY MESHER

- **Paver mesh the set of congruent surfaces**
  - Use the default value for the global edge length, 0.556217.



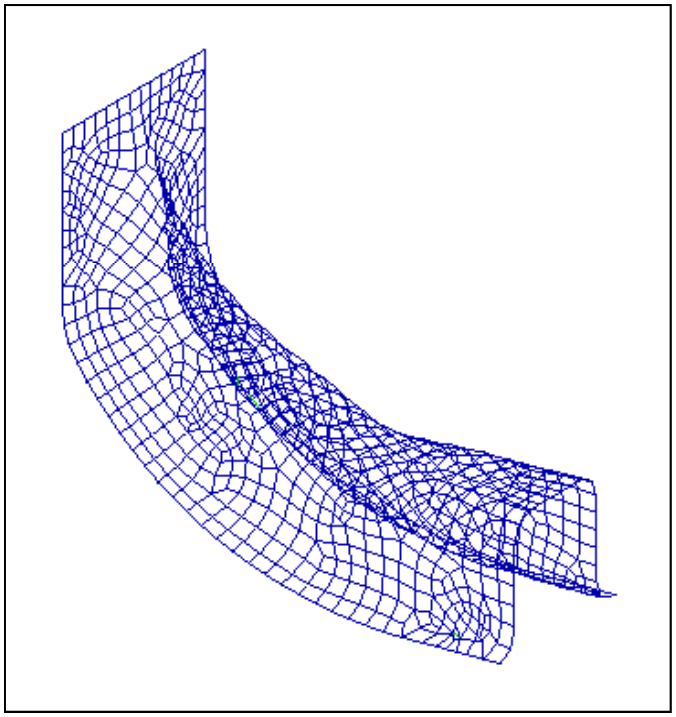


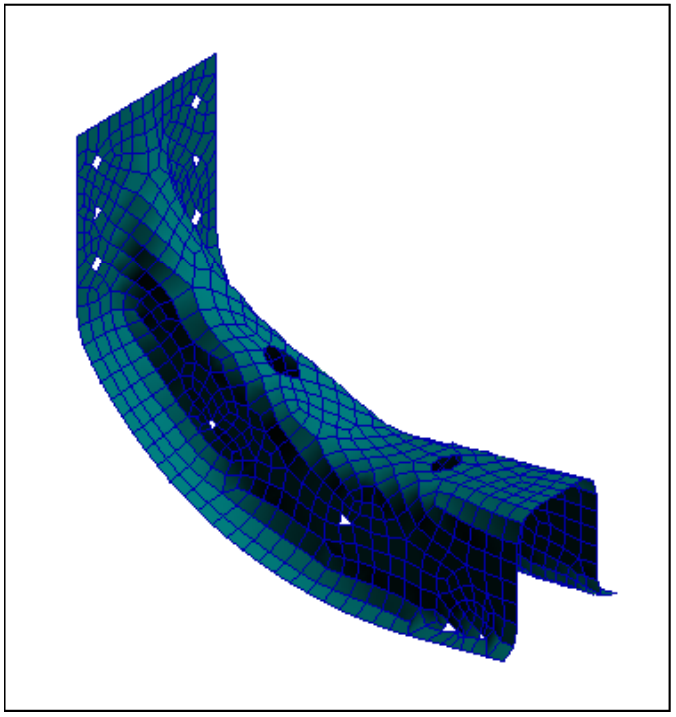
- **Look for areas where there are bad elements**
  - Look at the corresponding geometry.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: SHEET BODY MESHER

- **Mesh using the Sheet Body mesher**
  - Use the same value for the global edge length.
  - Do not use any mesher parameters.

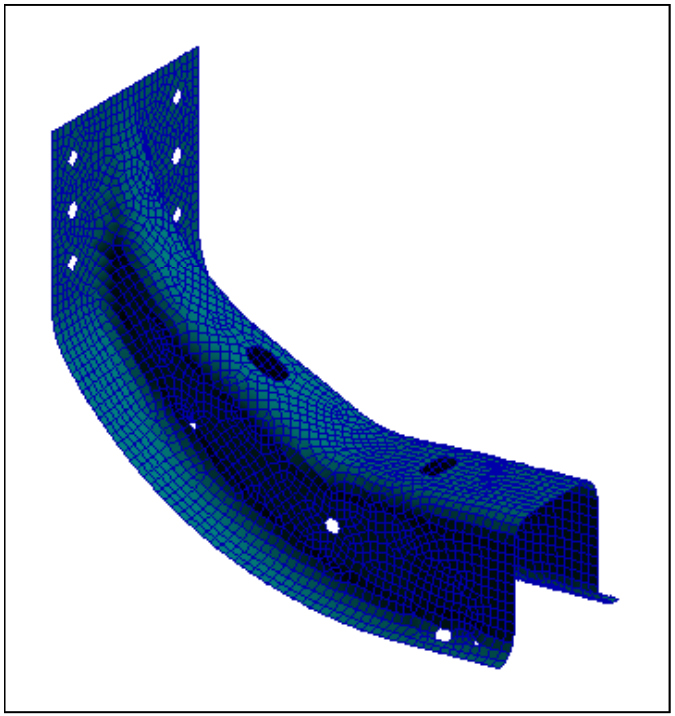


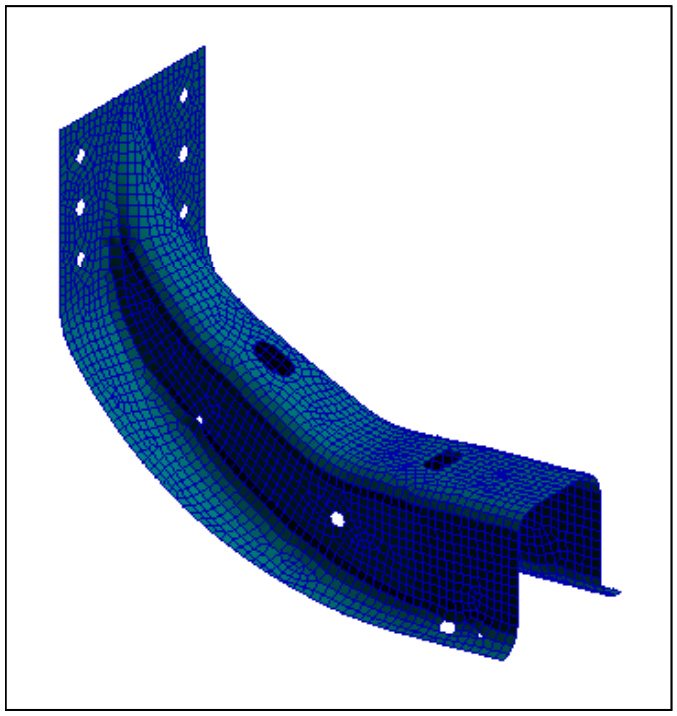


Copyright© 2012 MSC.Software Corporation

CASE STUDY: SHEET BODY MESHER

- **Use a smaller value for the global edge length.**
- **Use mesher parameters.**





Copyright© 2012 MSC.Software Corporation

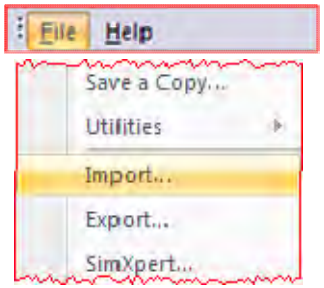
MESH ON MESH

- **Meshes using an existing 2D mesh**
- **Uses existing mesh, not geometry**
- **Many features for controlling the meshing:**
  - Mesh with QUAD4 or TRIA3 elements
  - Feature recognition
  - Hard node, hard bar, soft bar
  - Mesh refinement of a region
  - IsoMesh a region

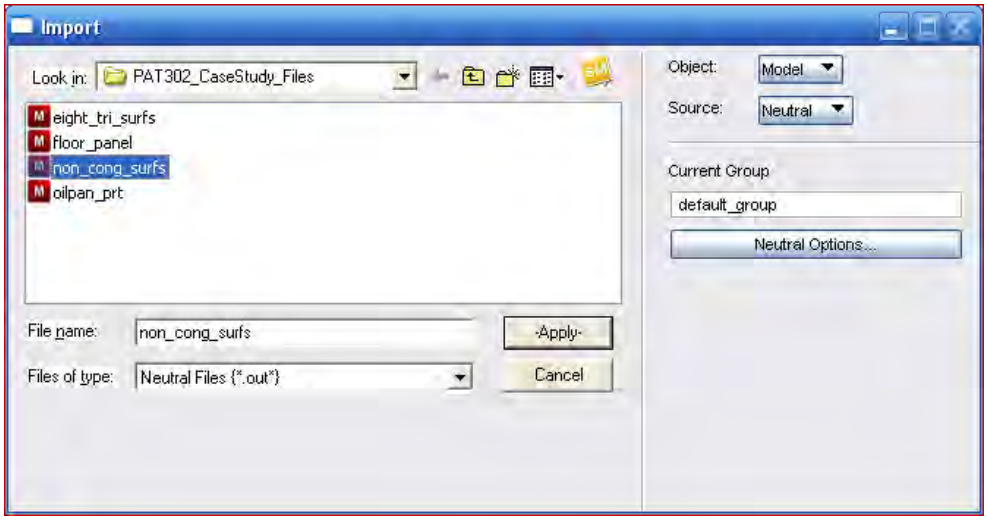
Start Point 2D	Nothing		Mesh			Geometry				
Method	Direct Nodes Elems	Geometry Construction	Mesh On Mesh	Mesh Editing	Tessellated Surface	Iso Mesh	Paver Mesh	Sheet Body	Composite Surface	Geometry Editing
End Point 2D	Mesh	Geometry	Mesh		Geometry	Mesh			Geometry	

Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1

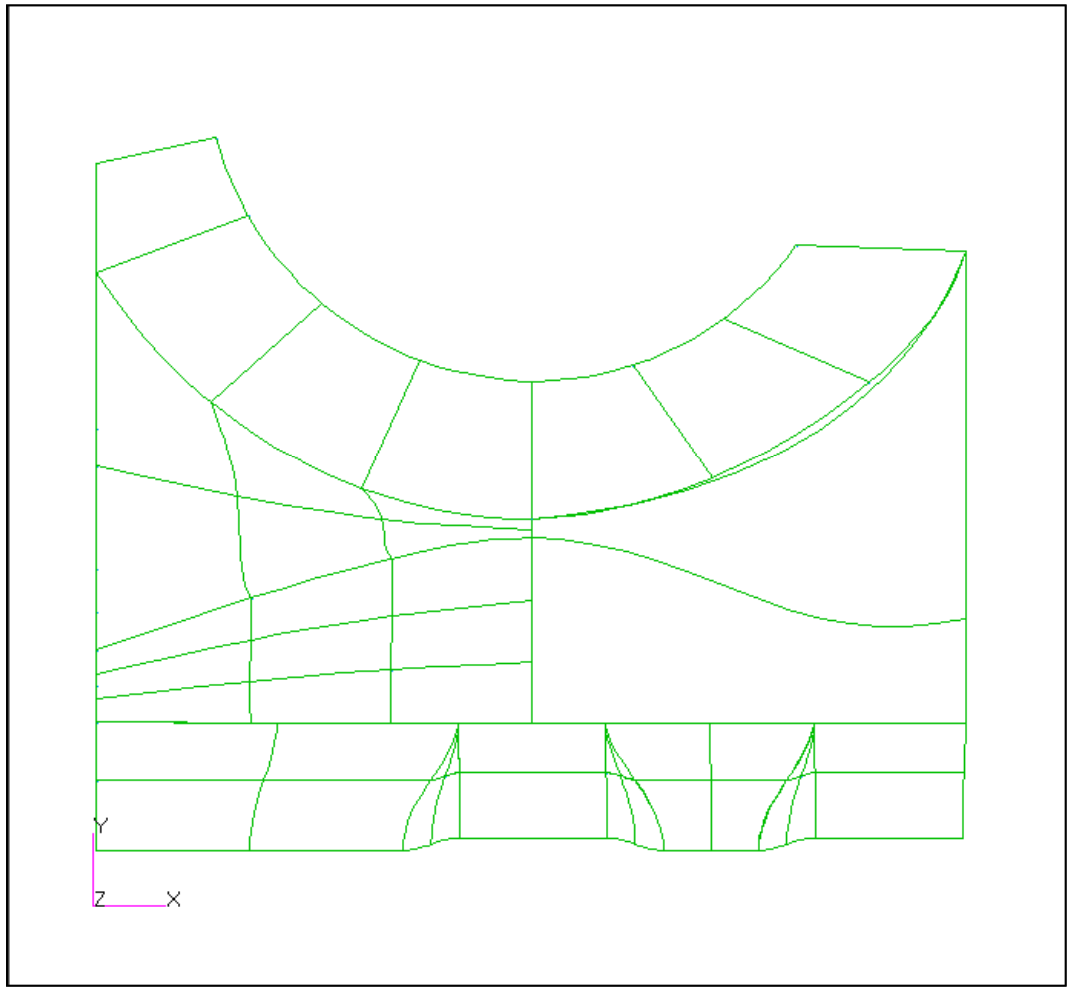


- **Import the Tria element mesh of the floor panel**
  - Select **Import…** from the *File* menu
  - Set *Object* as **Model** and *Source* as **Neutral.**
  - Select the file, **non_cong_surfs.out**.
  - Click **Apply**.



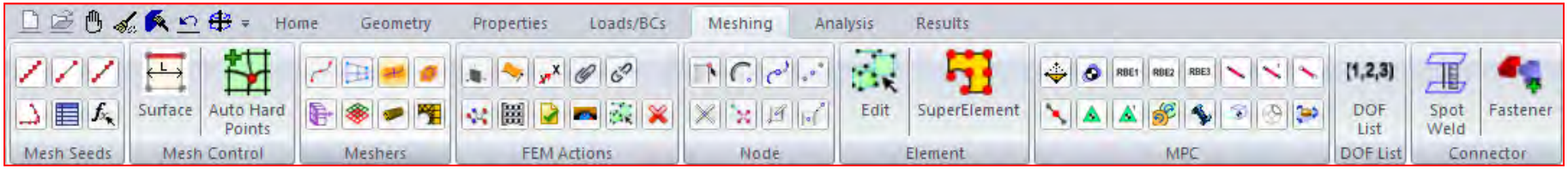
Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1

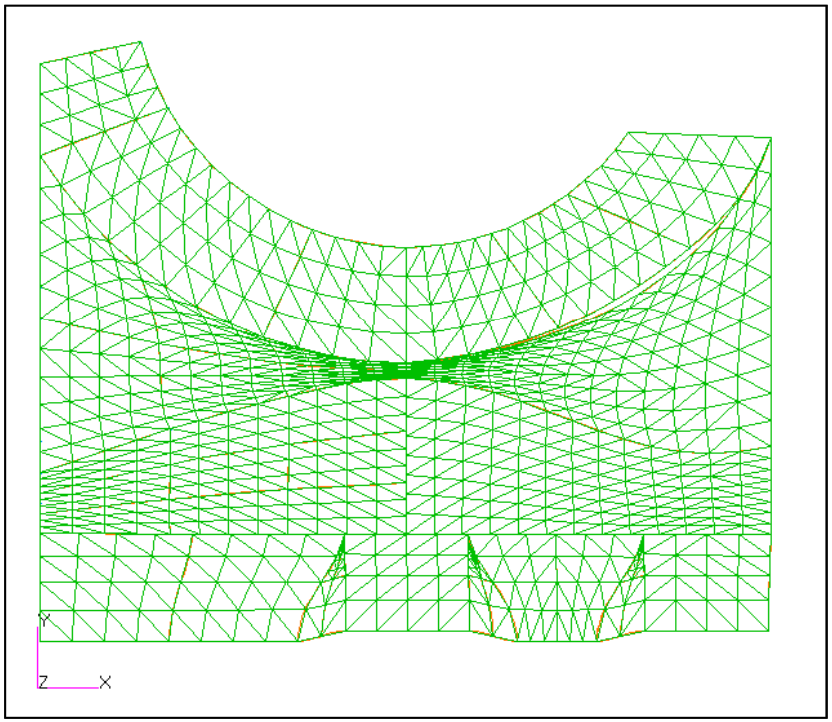


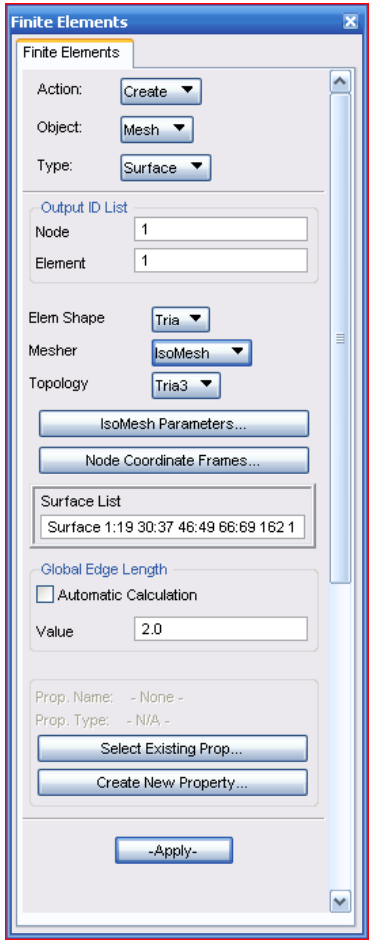
Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1



- **A single 2D congruent mesh that is coincident to the surfaces is needed.**
- **First, IsoMesh all the surfaces using tri3 elements**

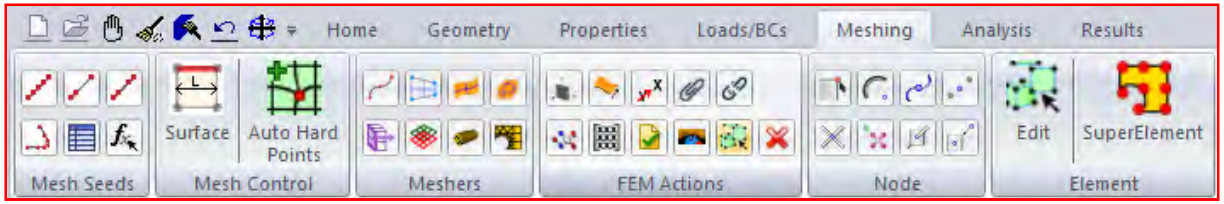


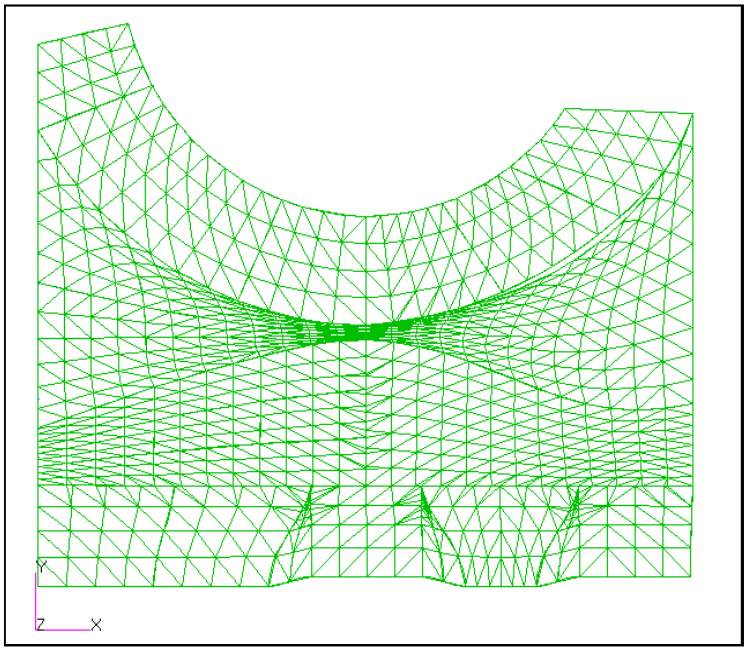


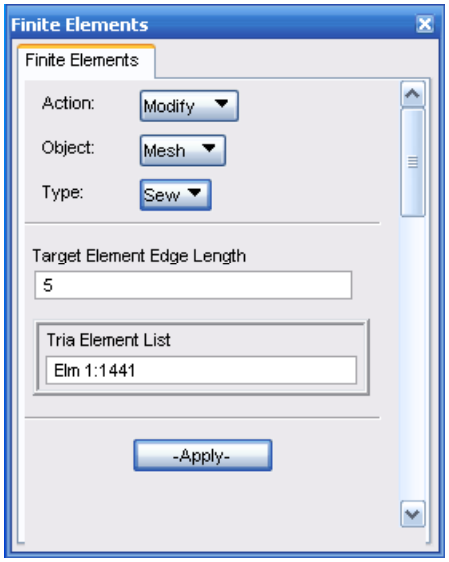
Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1

- **Equivalence and create tri elements by sewing the mesh:**
  - Under the *Meshing* tab click on **Modify** in the *FEM Actions* group. Select **Mesh** as the *Object* and **Sew** as the *Type*
  - Enter **5** for the *Target Element Edge Length*.
  - Under *Tria Element List*, select the entire model
  - Click **Apply**.



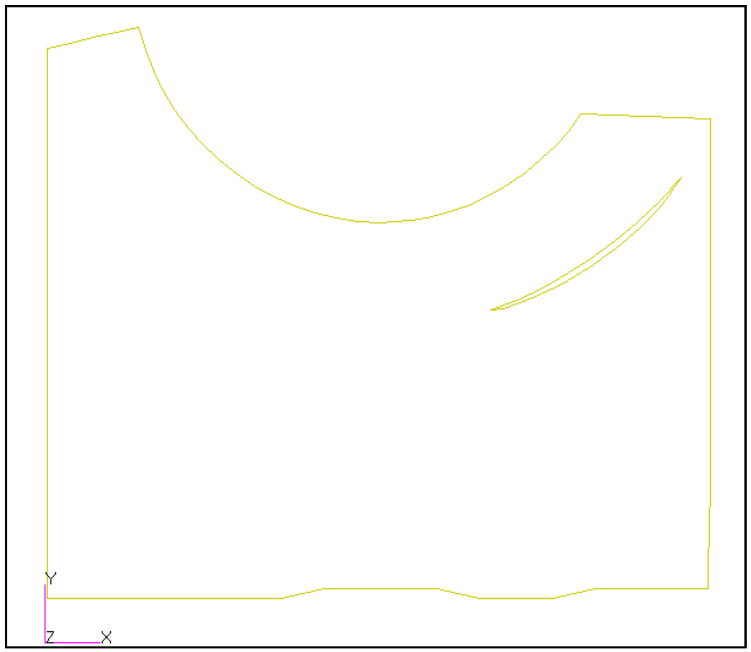


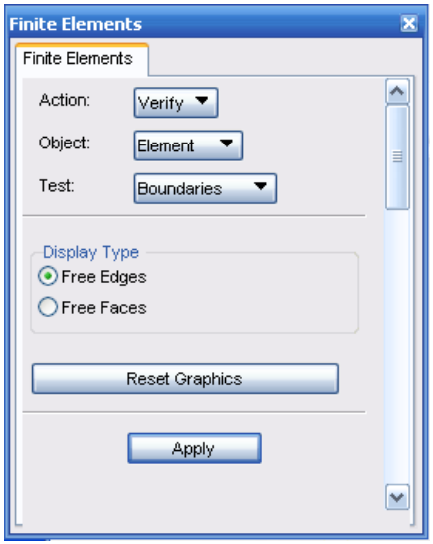


Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1

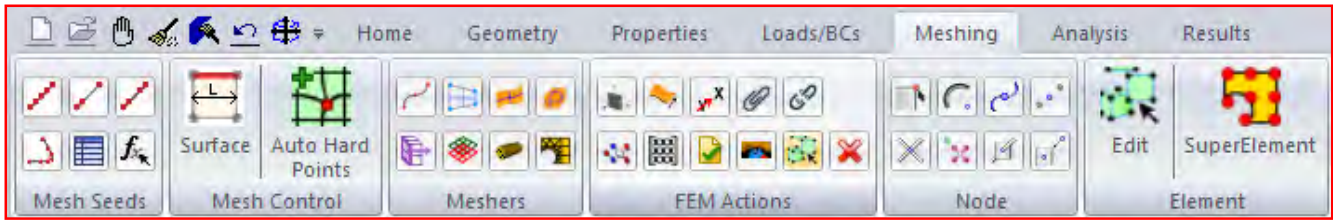
- **Check the free edges of the model to see if the sewing technique worked:**
  - Under *the Meshing tab* click on **Verify** in the **FEM Actions** group.
  - Set the *Display Type* to **Free Edges**.
  - Click **Apply**.
  - Click on the **Reset Graphics** button.



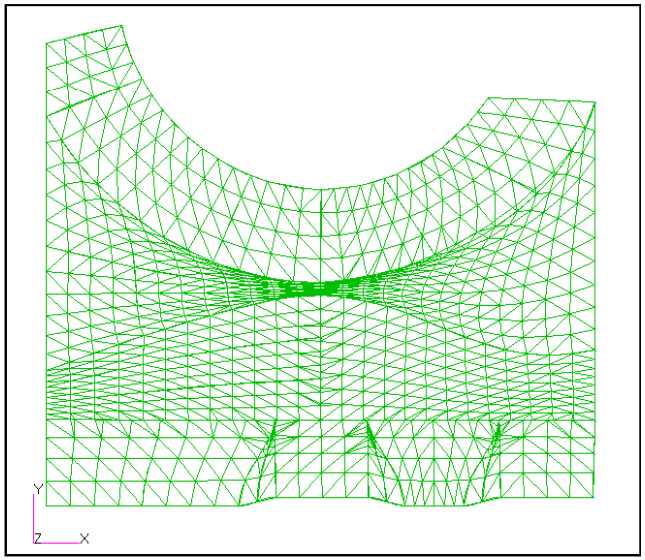


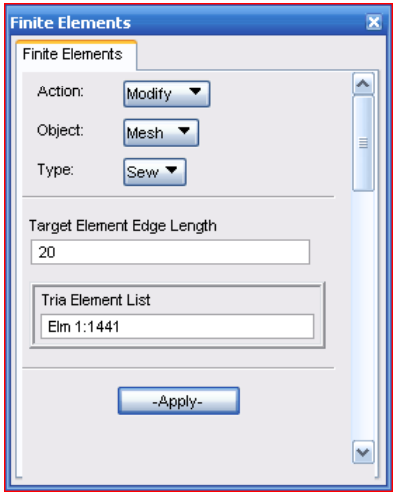
Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1



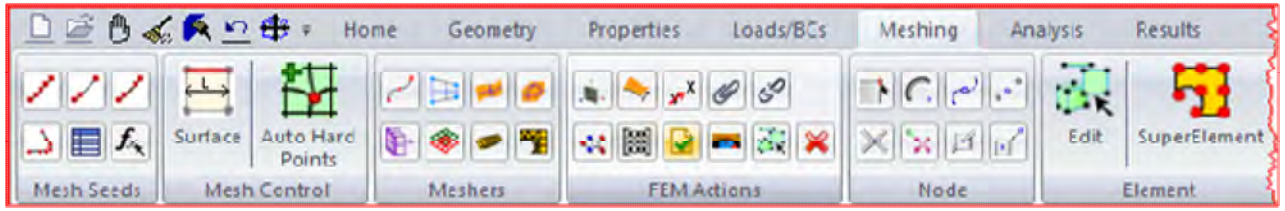
- **Undo, then repeat the sew command using a different Target Element Edge Length:**
  - Click on the **Undo** icon.
  - Under the *Meshing* tab click on **Modify** in the *FEM Actions* group and select **Mesh** as the *Object* and **Sew** as the *Type*
  - Enter **20** for the *Target Element Edge Length*.
  - Under *Tria Element List*, select the entire model.
  - Click **Apply**.



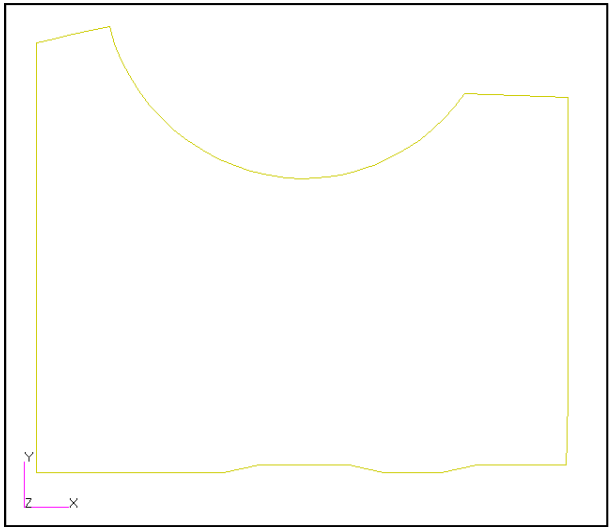


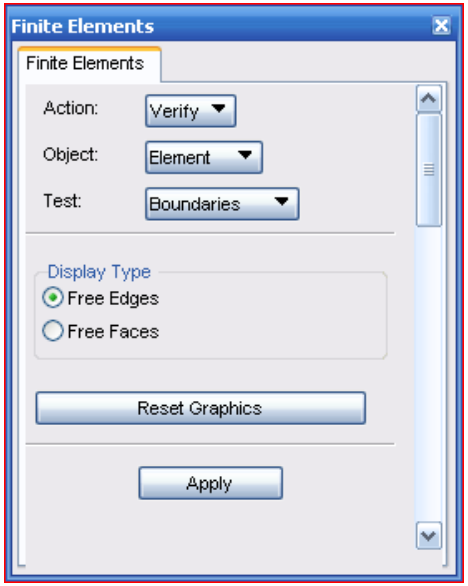
Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1



- **Check the free edges of the model to see if the sewing technique worked:**
  - Under the *Meshing* tab click on **Verify** in the *FEM Actions* group.
  - Set the *Display Type* to **Free Edges**.
  - Click **Apply**.

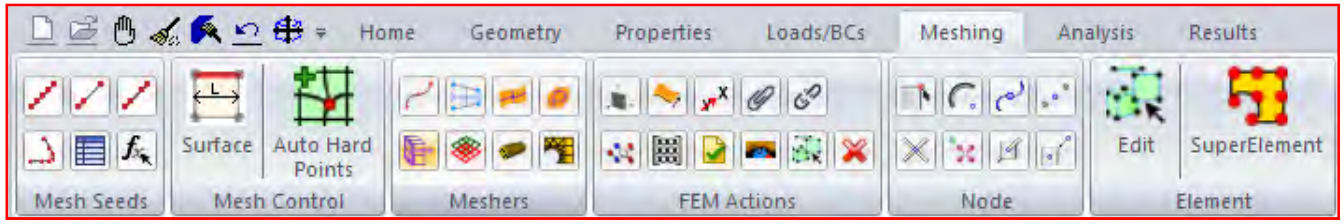




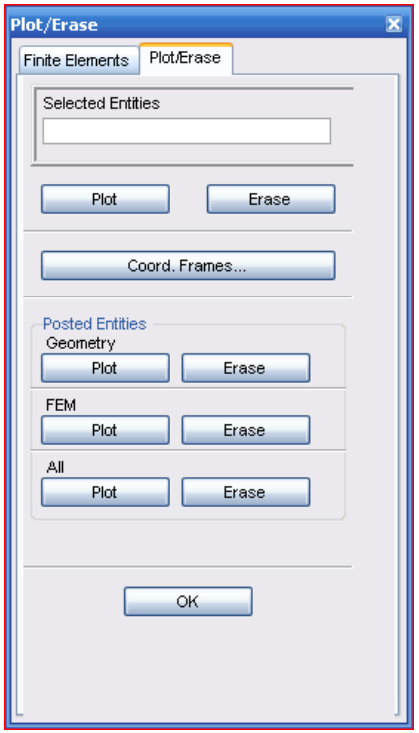
There are no element free edges internal to the mesh. This tri mesh is not acceptable for performing a finite element analysis. However, it is acceptable for using the Patran Mesh On Mesh capability to create a mesh that can be used to perform an analysis.

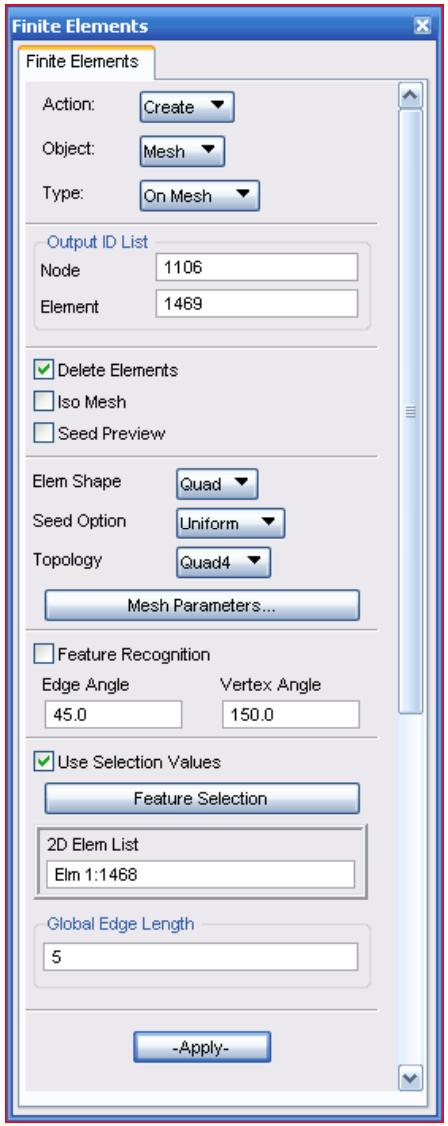
Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1



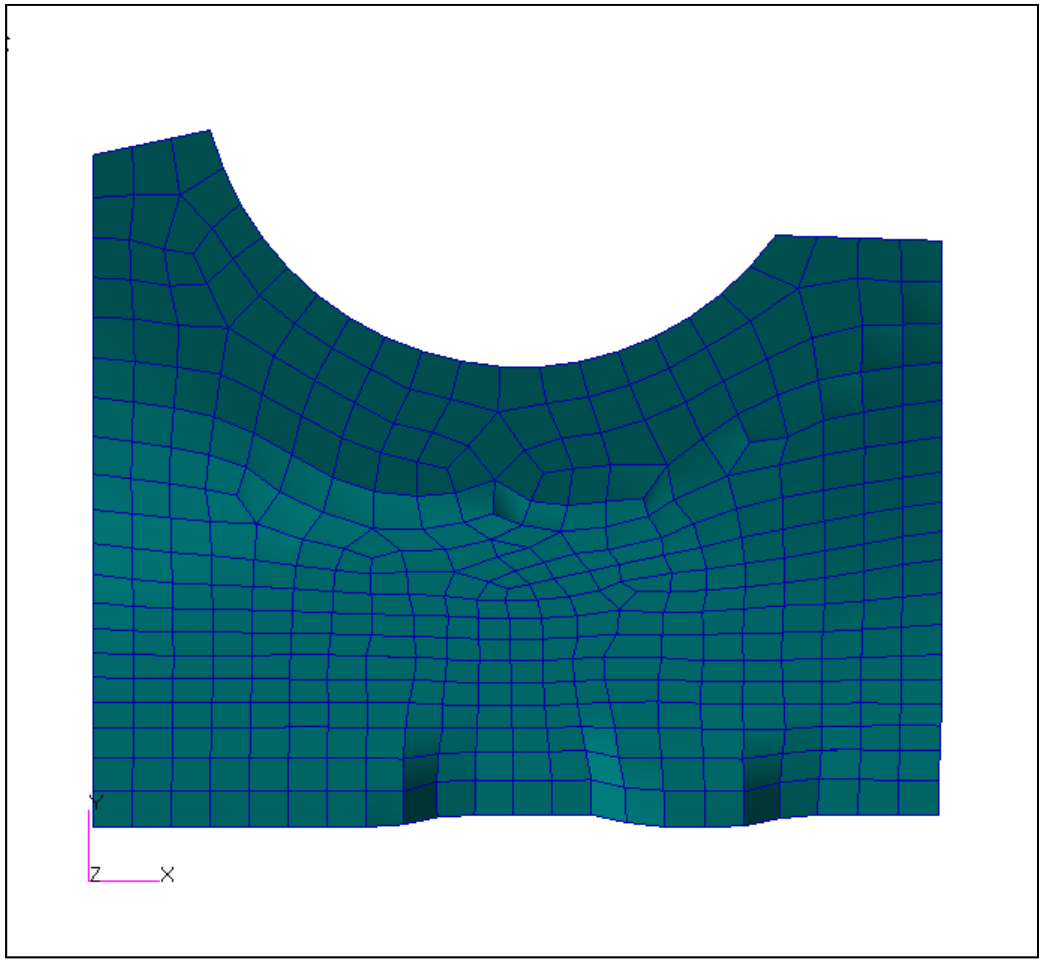
- **Use the Mesh On Mesh tool to create a mesh that can be used for an analysis:**
  - Under the *Meshing* tab click on **On Mesh** in the *Meshers* group.
  - Select the checkbox for **Delete Elements**.
  - Under *2D Elem List*, select the entire model.
  - Enter **5** for *Global Edge Length*.
  - Click **Apply**.
  - Click on the **Plot/Erase** icon.
  - Click on **Erase** under *Geometry*.
  - Click on the *Smooth shaded* icon.





Copyright© 2012 MSC.Software Corporation

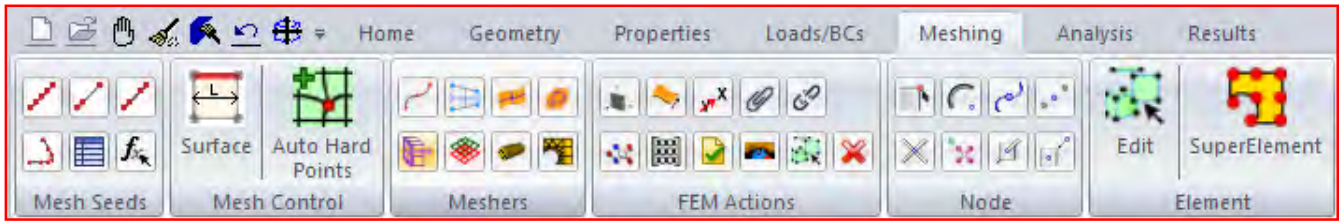
CASE STUDY: MESH ON MESH 1



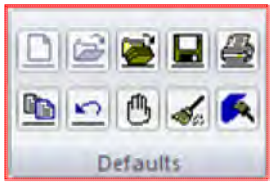
This mesh is too coarse. Therefore it will be necessary to undo the previous step and run the Mesh On Mesh application again, this time with a smaller element size.

Copyright© 2012 MSC.Software Corporation

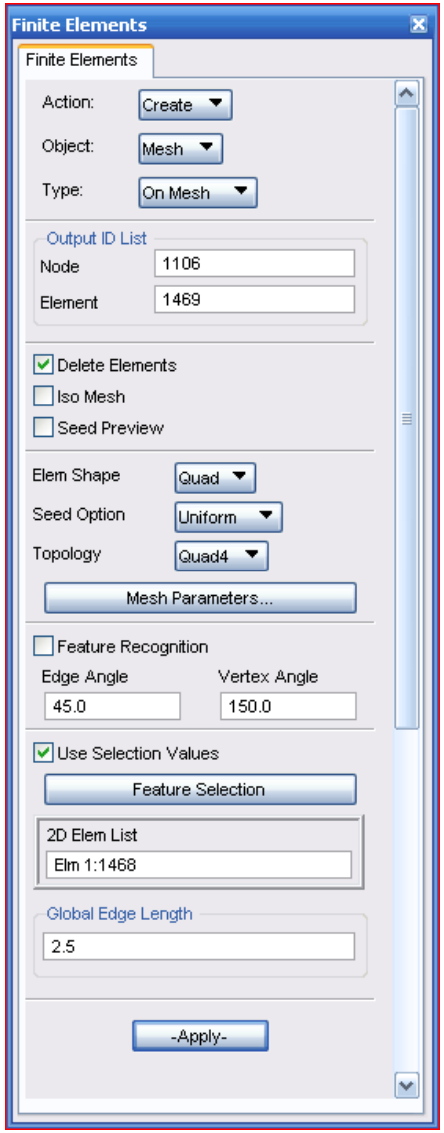
CASE STUDY: MESH ON MESH 1



- **Undo the previous Mesh On Mesh creation, and create a new one with a smaller element size:**



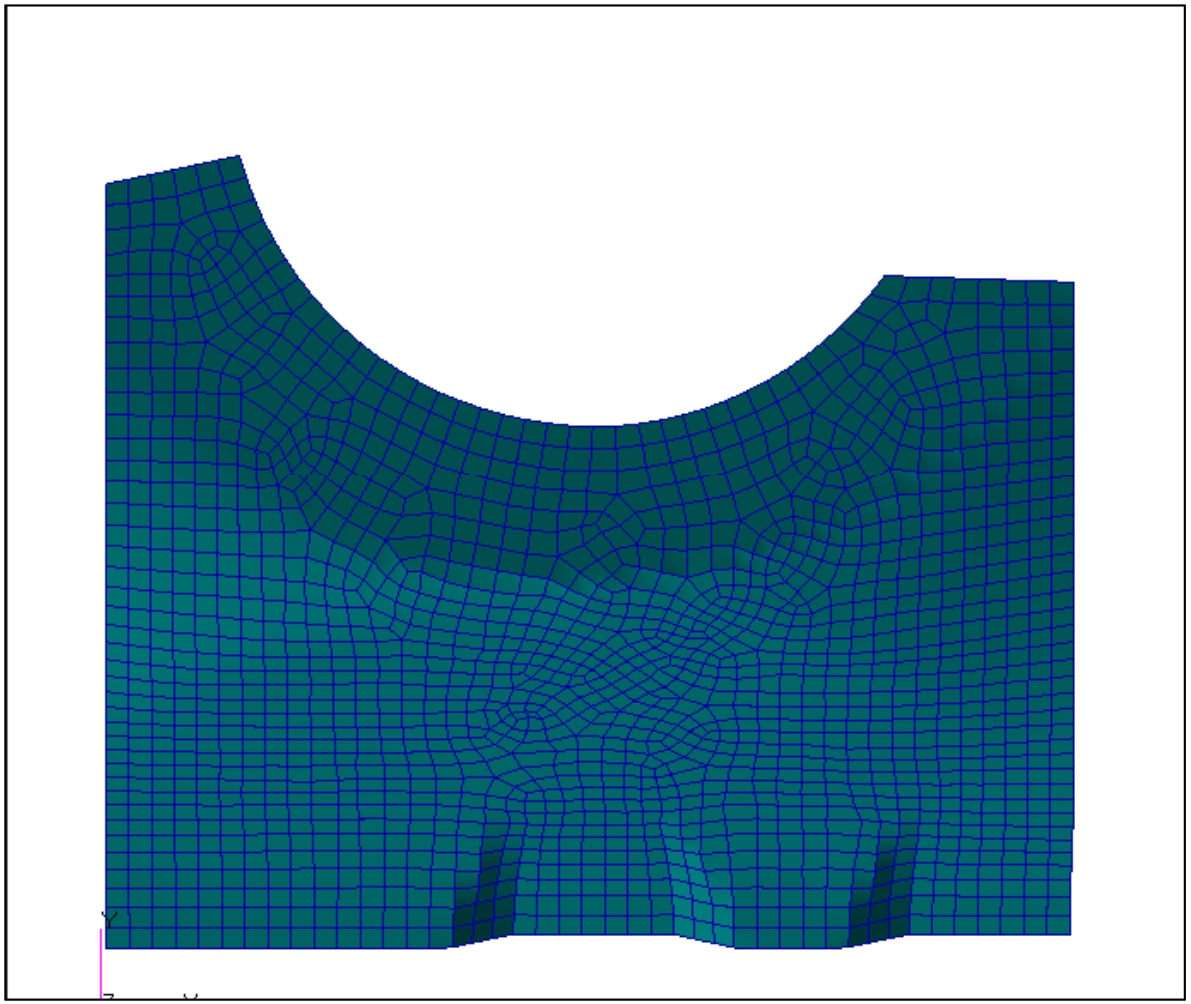
  - Click on the **Undo** icon.
  - Under the *Meshing* tab click on **On Mesh** in the *Meshers* group.
  - Make sure **Delete Elements** is checked.
  - Under *2D Elem List*, select the entire model.
  - Enter **2.5** under *Global Edge Length*.
  - Click **Apply**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1

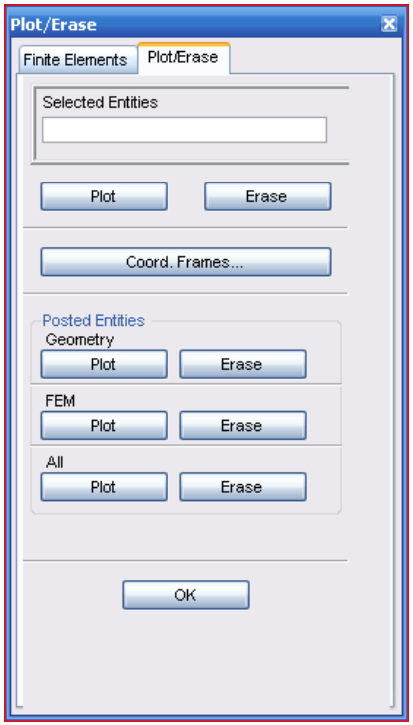
Although this mesh is much finer, it still does not follow the original model geometry closely enough. It will be necessary to do another Mesh On Mesh operation, with a slightly smaller element size and with control, using 1D elements on some of the surface edges. The control on the edges is achieved by using 1D Bar2 elements shown in the following step.

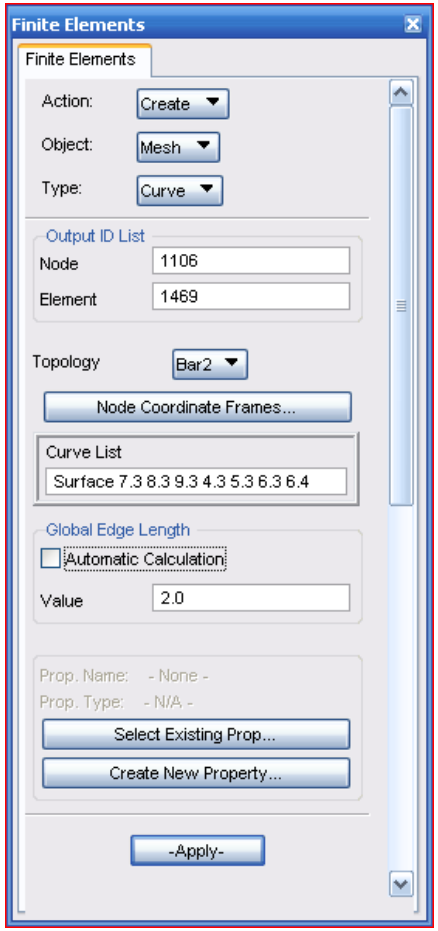


Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1

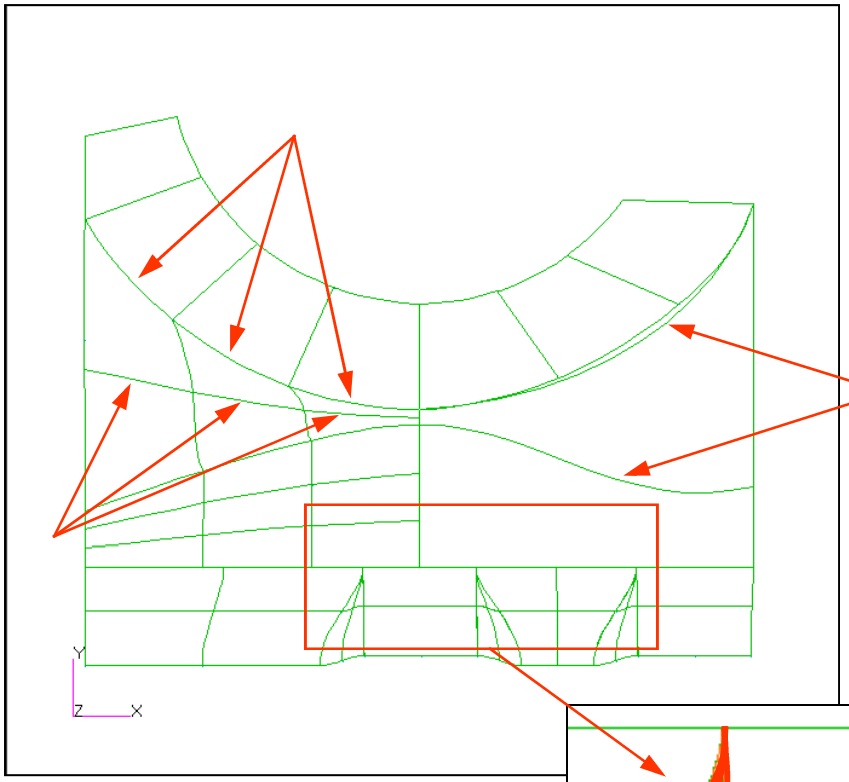
- **Undo the mesh created by the Mesh/ On Mesh application, and re-mesh using select edges with Bar2 elements:**
  - Click on the **Undo** icon.
  - Click on the **Plot/Erase** icon.
  - Click on **Erase** under *FEM*.
  - Click on **Plot** under *Geometry*.
  - Click **OK**.
  - Under the *Meshing* tab click on **Curve** in the *Meshers* group.
  - Change the *Topology* to **Bar2**.
  - Under *Curve List*, select the indicated curves (shown in the next slide).
  - Uncheck **Automatic Calculation** and enter **2.0** for *Global Edge Length*.
  - Click **Apply**.



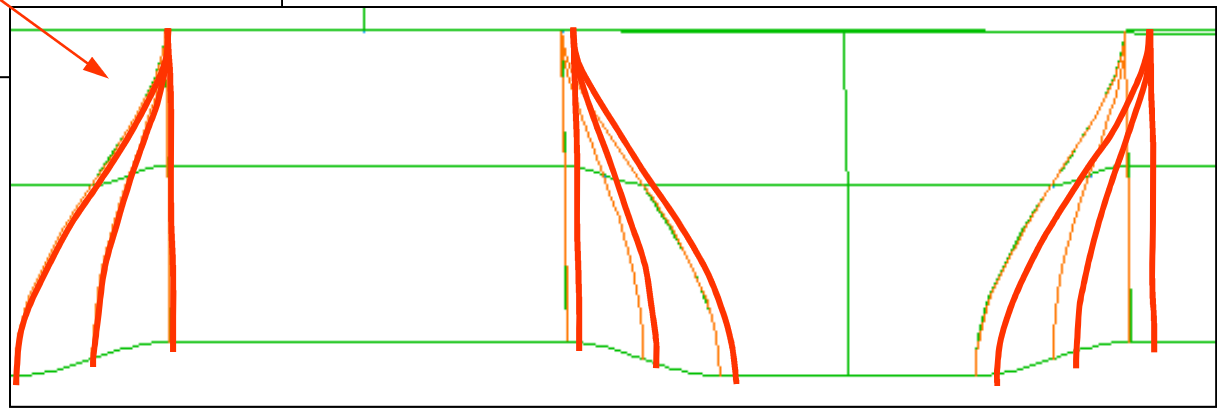


Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1

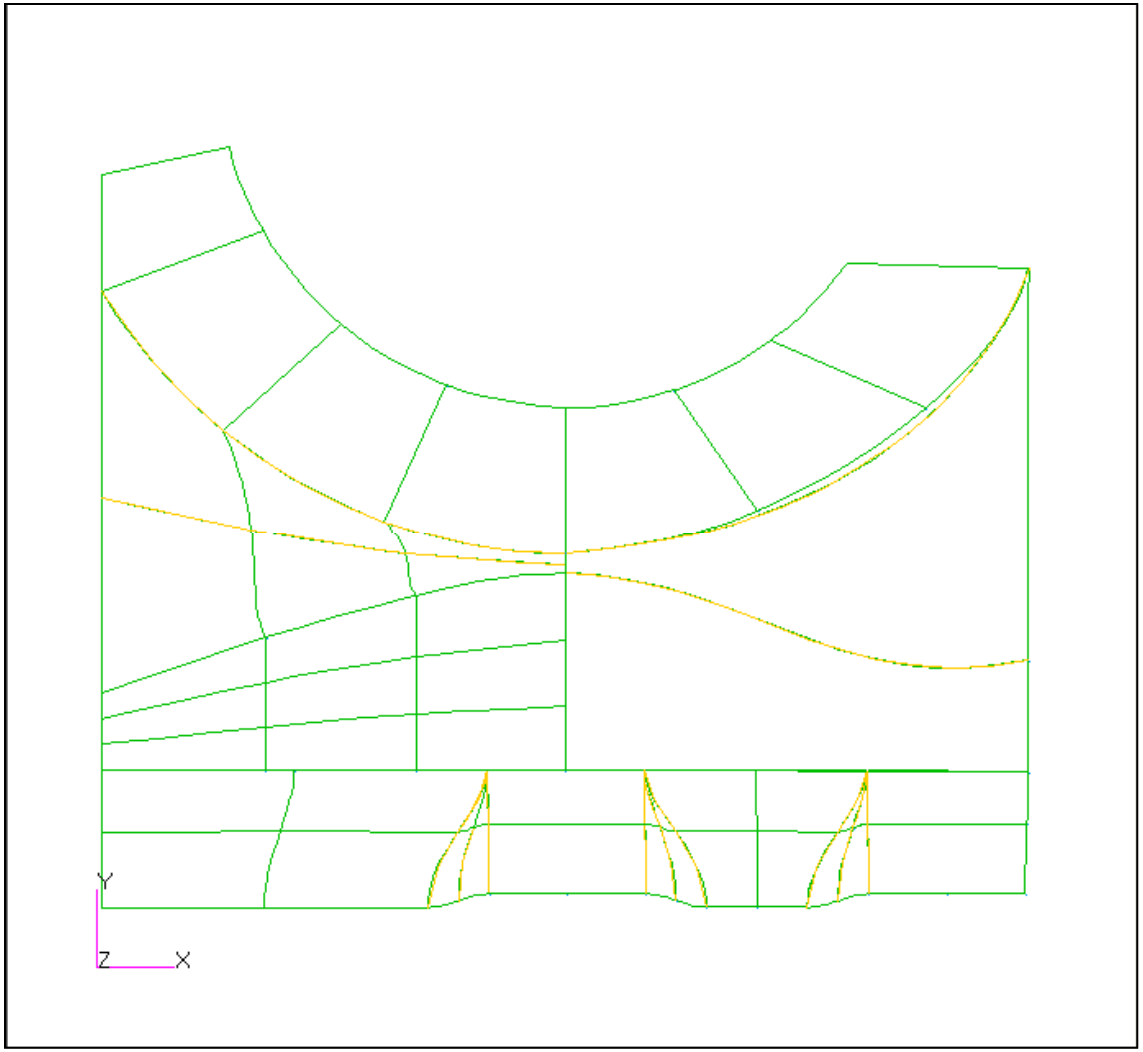


Illustrated here, are the surface edges that are to be meshed with Bar2 elements. There are 26 edges. To the left, the arrows point to 8 of the edges. Below, 18 of the edges are shown as thick lines.



Copyright© 2012 MSC.Software Corporation

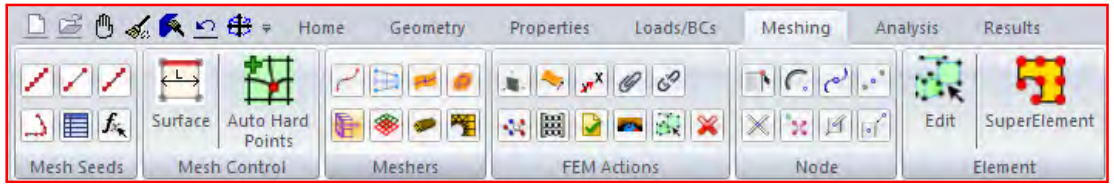
CASE STUDY: MESH ON MESH 1



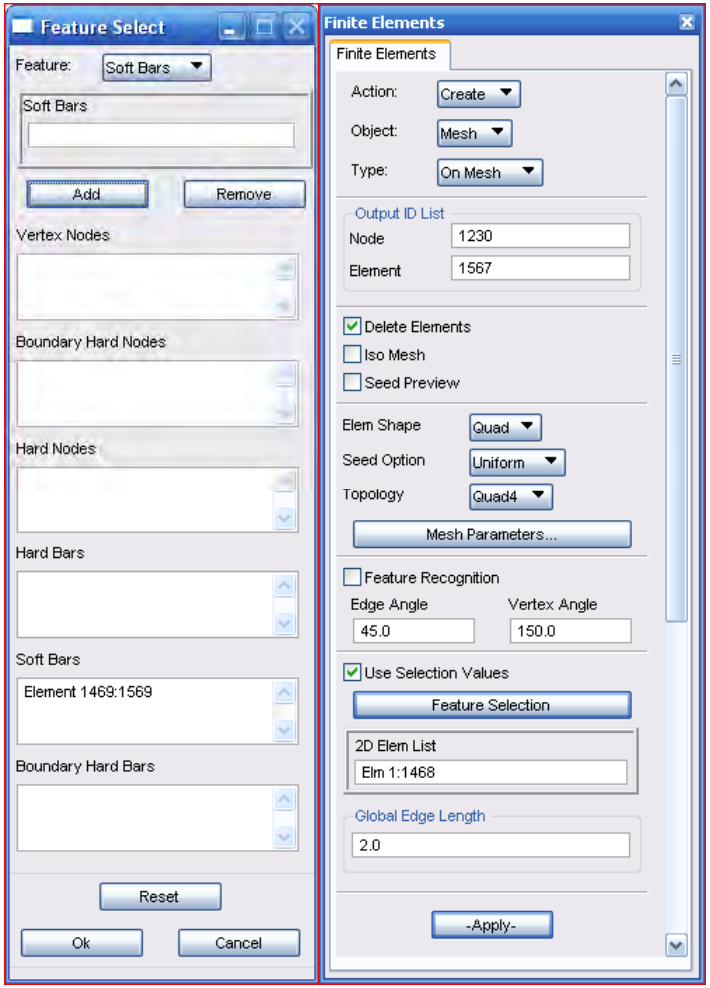
The Bar2 elements are shown in yellow.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1



- **Use the Mesh/On Mesh application to create a final mesh that should follow the geometry:**
  - Under the *Meshing* tab click on **On Mesh** in the *Meshers* group.
  - Make sure **Delete Elements** is checked.
  - Click on **Feature Selection**.
  - Under *Soft Bars*, select all the 1D bar elements and click **ADD** and then **OK**.
  - Under *2D Elem List*, select the entire model.
  - Enter **2.0** for *Global Edge Length*.
  - Click **Apply**.
  - Click on the **Plot/Erase** icon.
  - Click on **Erase** under *Geometry*.
  - Click on the **Smooth shaded** icon.

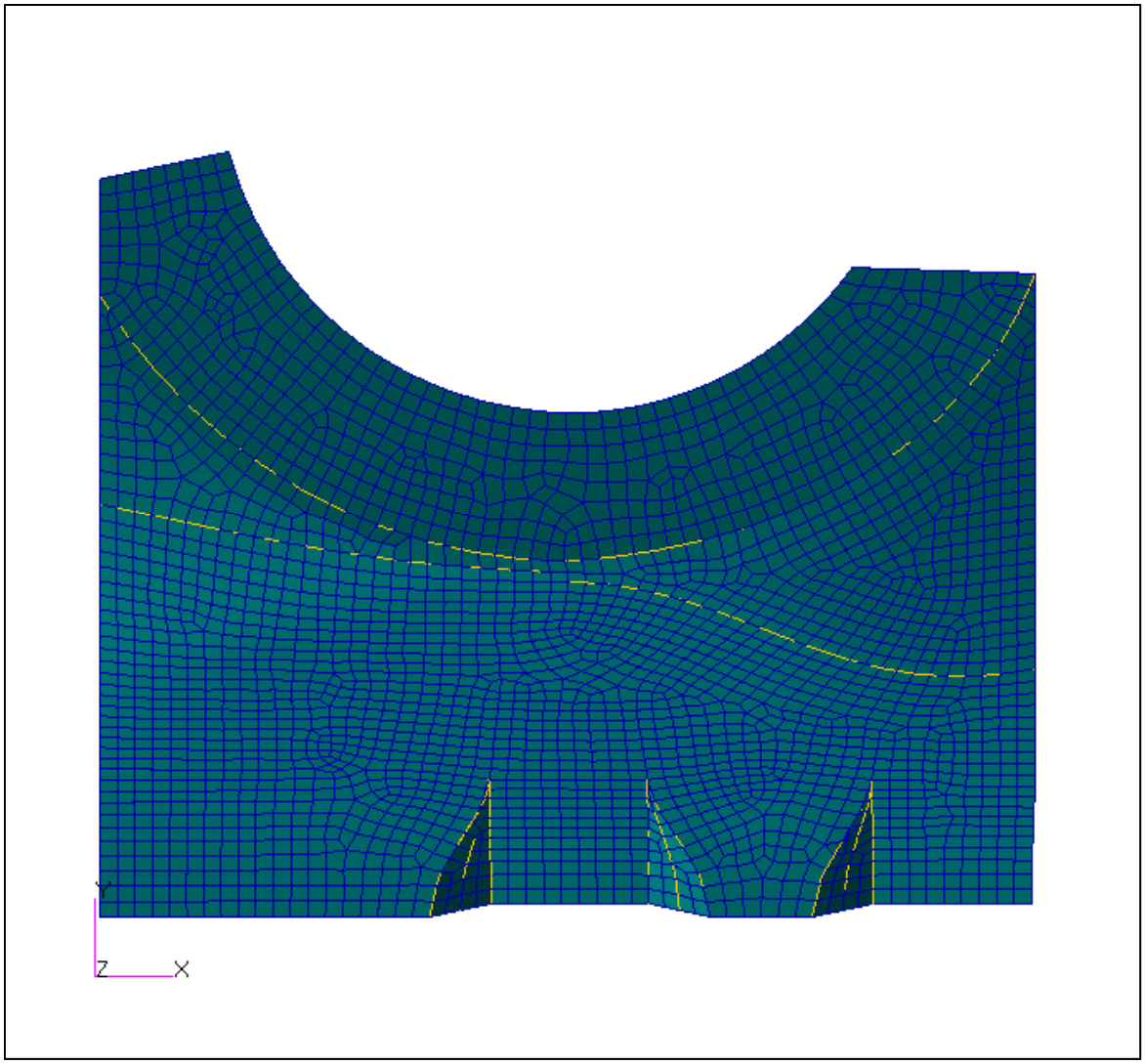


Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 1

Illustrated here is the final mesh created by the Mesh On Mesh application.

Note how the mesh follows the model geometry far better than for the previous meshes.



Copyright© 2012 MSC.Software Corporation

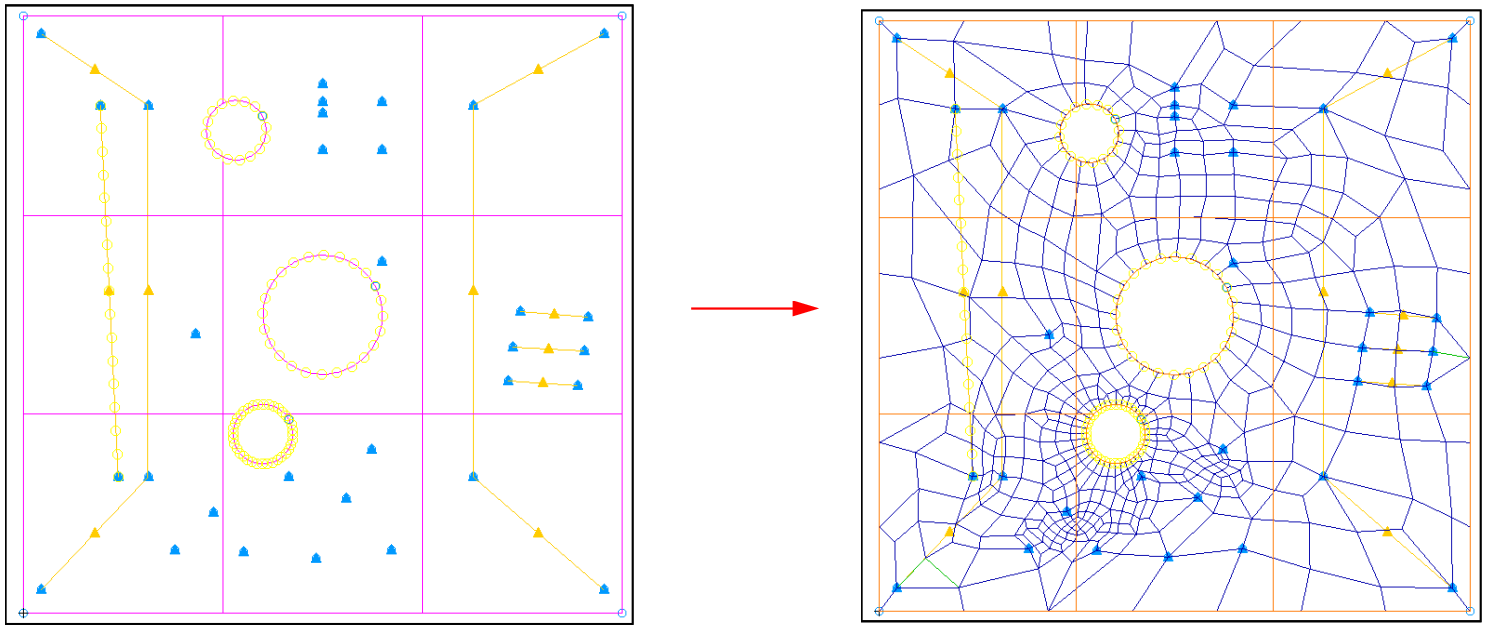
EXERCISES

- Perform Workshop 4 “Mesh on Mesh 2D” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 2

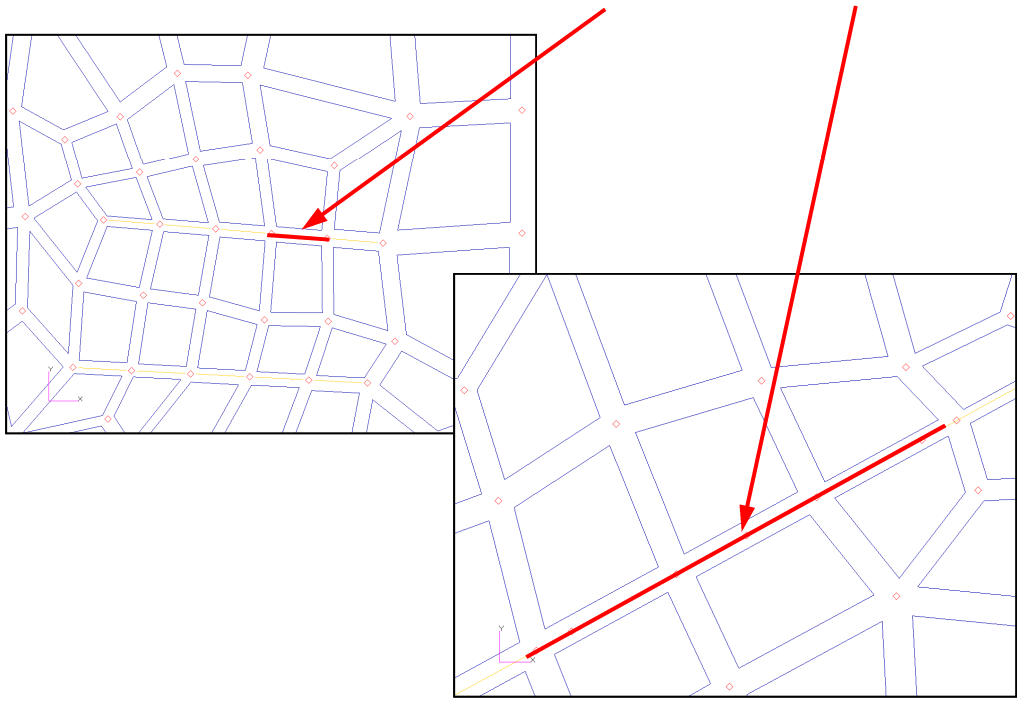
- **Using a meshing procedure is useful for complex flat surface meshes**
  - Many hard points, many hard curves, many mesh seeds
  - Paver or Hybrid may fail or produce an unacceptable mesh

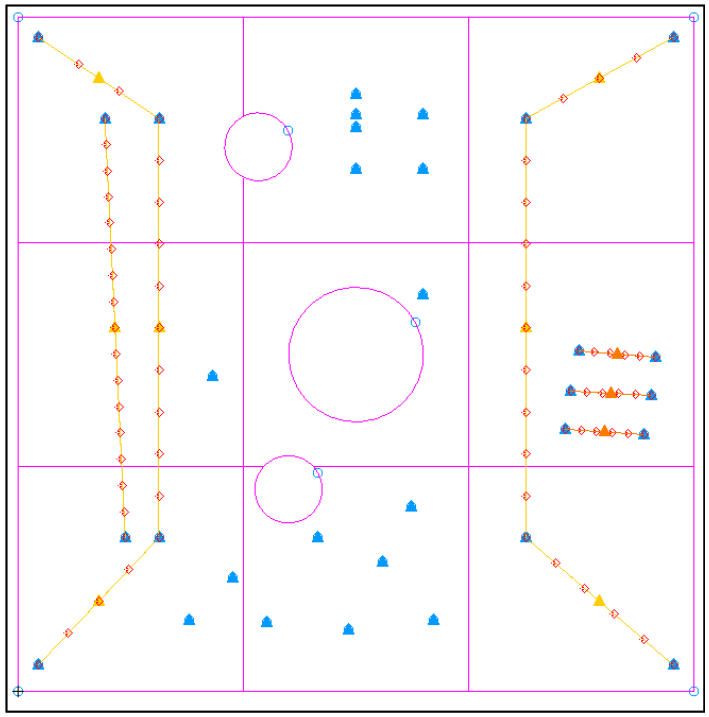


Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 2

- **Step one: Mesh all hard curves**
  - Resulting bar elements will be used later to guide re-meshing
  - Two uses: hard bar, soft bar

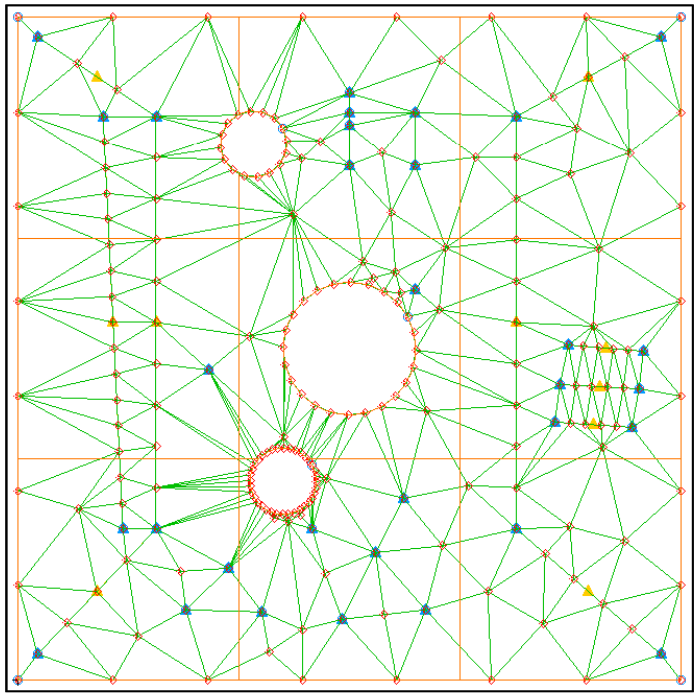




Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 2

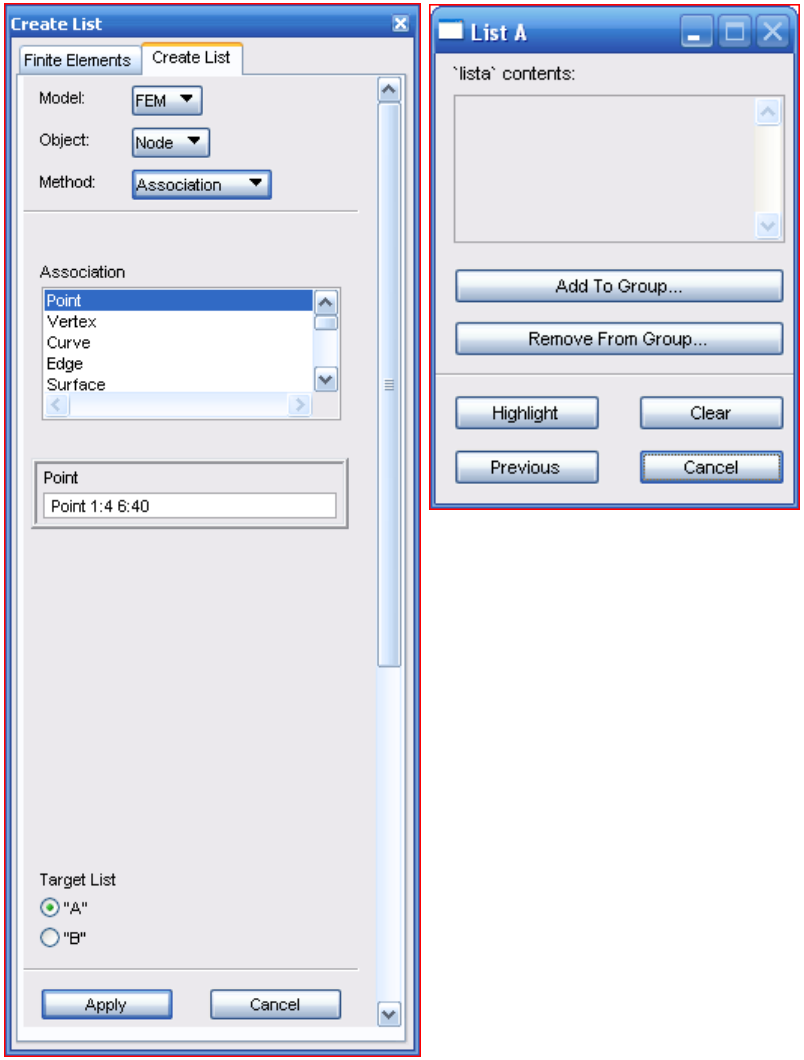
- **Step two: Mesh the surface using triangle elements**
  - Meshing with triangles is more reliable, as there are fewer failures
  - Create a tria3 mesh using Paver or Hybrid
  - Nodes will be created at and associated with the hard points
  - Equivalence at duplicates



Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 2

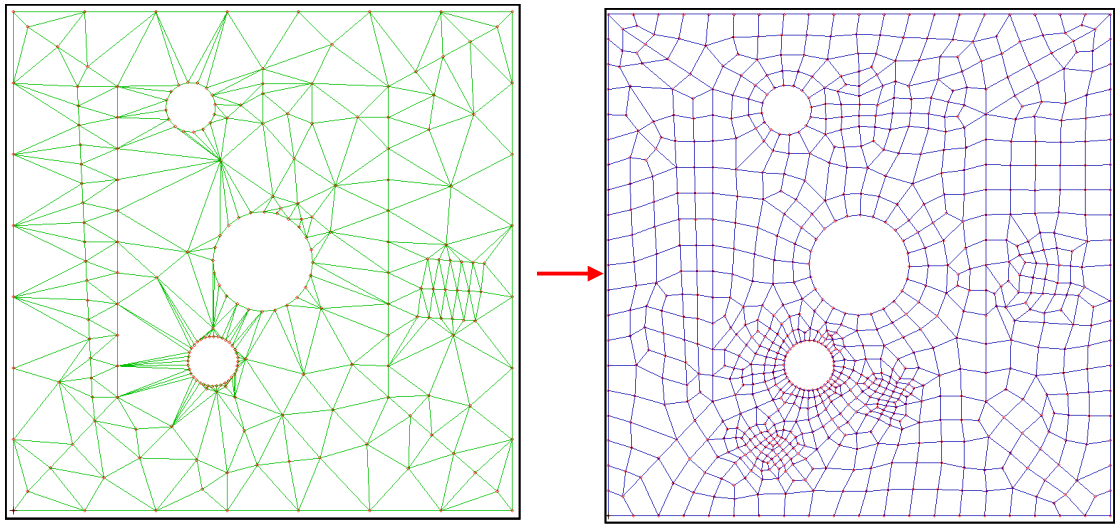
- **Step three: Create node list**
  - Create a list of all nodes created at hard points
  - These node locations will be preserved in the re-meshing operation

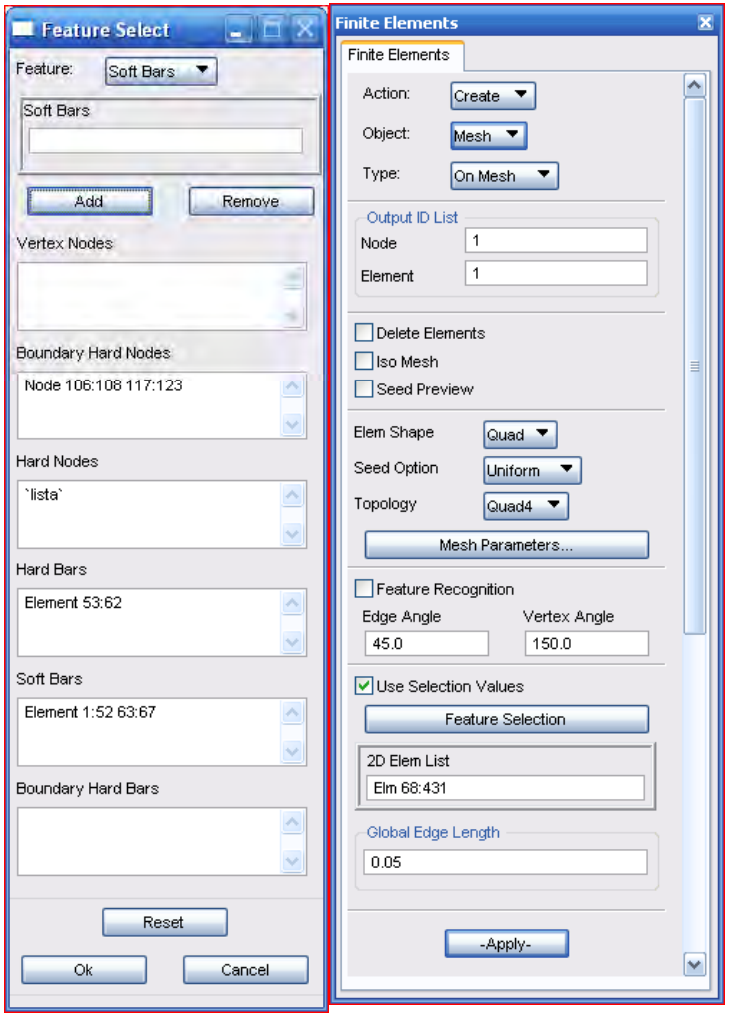


Copyright© 2012 MSC.Software Corporation

CASE STUDY: MESH ON MESH 2

- **Step four: Create new Quad4 mesh using Mesh/On Mesh**
  - Specify Seed Option:
    Uniform, Existing Boundary, Defined Boundary
  - Select boundary seeds
  - Specify hard nodes using `lista`
  - Select existing bar elements for use as either hard bars or soft bars
  - Select the tria3 elements
  - Specify element size (GEL)

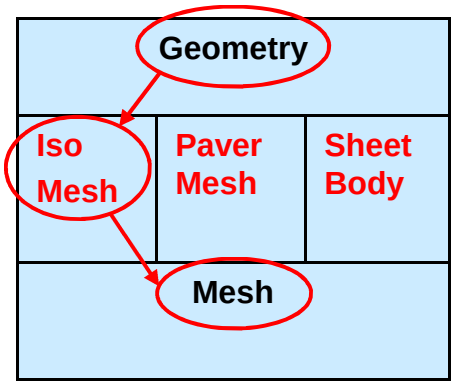




Copyright© 2012 MSC.Software Corporation

ISO-MESHING OF PARASOLID SOLID FACES

- **Parasolid solid faces can now be isomeshed.**
- **These faces must be 3- or 4-sided with no inner boundaries.**



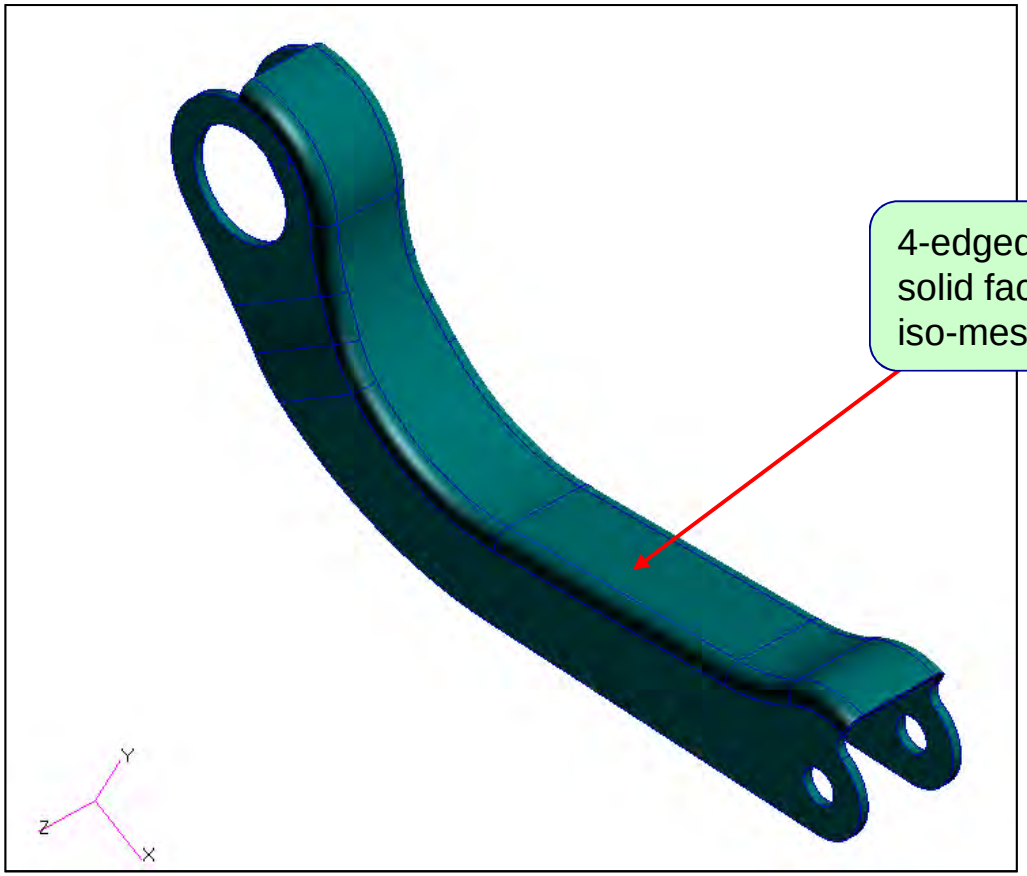
Copyright© 2012 MSC.Software Corporation

EXAMPLE: ISO-MESHING OF PARASOLID SOLID FACE

A parasolid solid model is shown.

All 3- or 4-sided faces are iso-meshable.

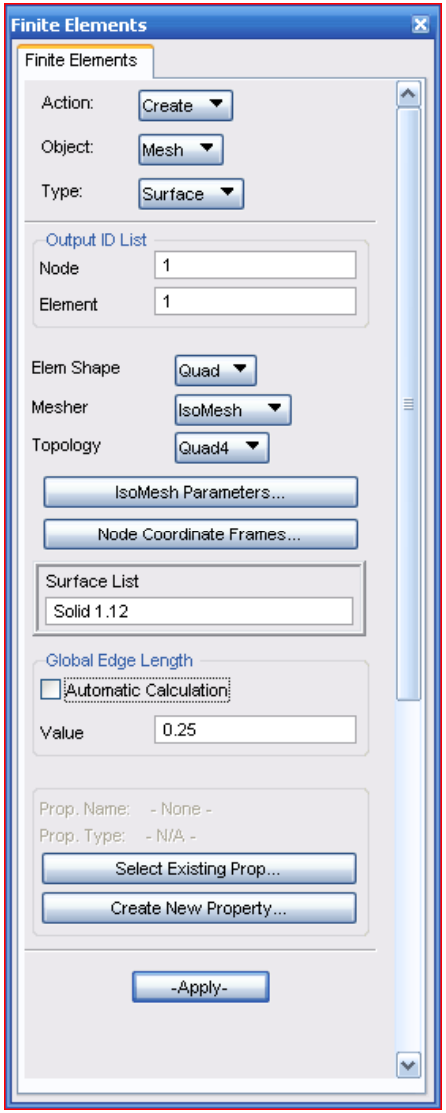
Select one face as shown.



Copyright© 2012 MSC.Software Corporation

EXAMPLE: ISO-MESHING OF PARASOLID SOLID FACE

The parasolid solid face is iso-meshed



Copyright© 2012 MSC.Software Corporation

ADVANCED SURFACE MESHER

- **Advanced Surface Mesher (ASM) Capabilities**
  - Handles complex surface models
    - Non-congruencies
    - Gaps
    - Overlaps
    - Slivers
    - Small edges
  - Provides optional global and local editing tools to accommodate meshing specific requirements

Copyright© 2012 MSC.Software Corporation

ADVANCED SURFACE MESHER

- **ASM Capabilities**
  - Automatic and manual tools for a set of surface meshes
    - Fill holes and gaps
    - Make congruent
    - Stitch gaps
    - Break face
    - Merge faces
    - Delete faces
    - Auto correct
  - Added meshing enhancements to
    - Recognize features
    - Support hard points and curves

Copyright© 2012 MSC.Software Corporation

ASM BENEFITS

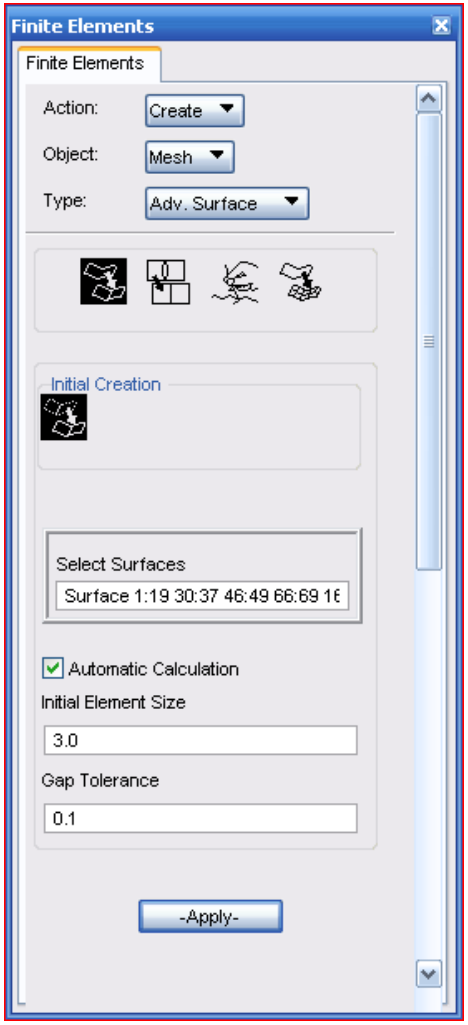
- **Benefits**
  - Drastically reduced analysis modeling time
    - The part (see surface) can now be meshed in less than a minute
    - With other methods, it would take at least 30 minutes

  - Reduced cost
  - Increased productivity
  - Build quality finite element models with minimal user interaction

Copyright© 2012 MSC.Software Corporation

ASM DESIGN SPECIFICATION

- **Design of Advanced Surface Mesher**
  - Original geometric model surfaces can be imported or created in Patran
  - The geometric surfaces are used to create surfaces of elements, Elm Surfaces (2D tria mesh on each geometric surface)
  - The Elm Surfaces are segregated by Patran groups, and group mode is used for display
  - Access to the functionality is through the Advanced Surface Meshing form



Copyright© 2012 MSC.Software Corporation

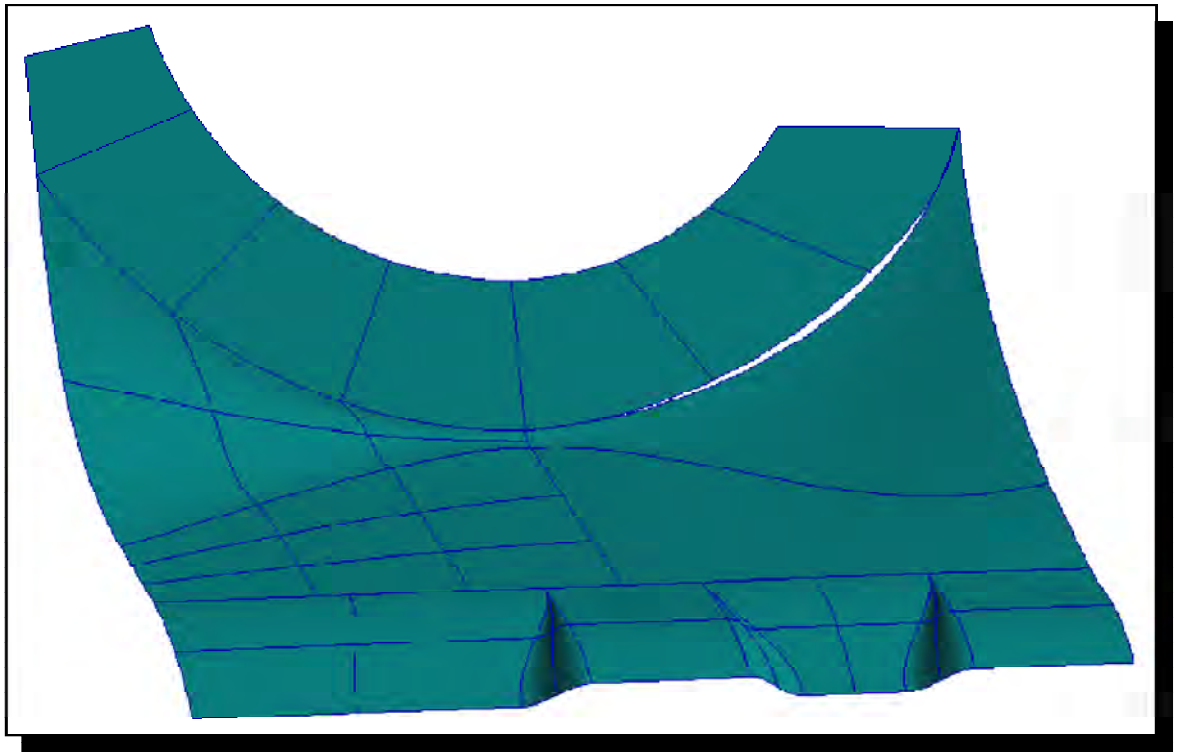
ASM DESIGN SPECIFICATION

- **Design of Advanced Surface Mesher (Cont.)**
  - An Elm Surface can be modified using manual and automated operations such as:
    - Fill holes and gaps
    - Stitching gaps
    - Break a face (Elm Surface) with a curve
    - Make congruent
    - Merge faces
    - Auto correct
    - Delete faces
  - The “Mesh / On Mesh” algorithm is used to mesh the Elm Surface geometry
    - It recognizes features
    - It supports hard points and hard curves

Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

- **oilpan_prt.out (Patran neutral file)**
- **This surface model contains**
  - Non-congruent surface boundaries
  - Overlaps
  - Gaps



Copyright© 2012 MSC.Software Corporation

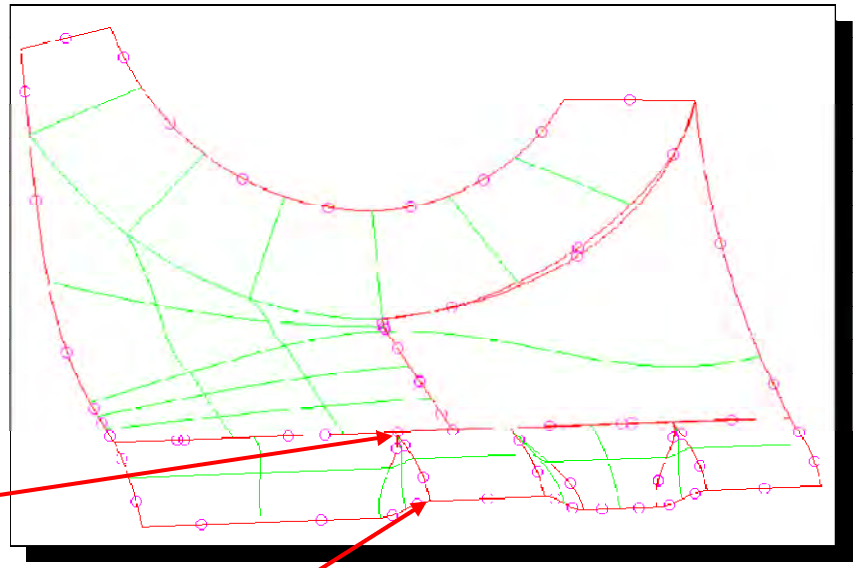
CASE STUDY: ASM EXAMPLE

- **Under the Geometry tab click on Verify from the Geometry Actions group.**
- **Many internal free edges are found**

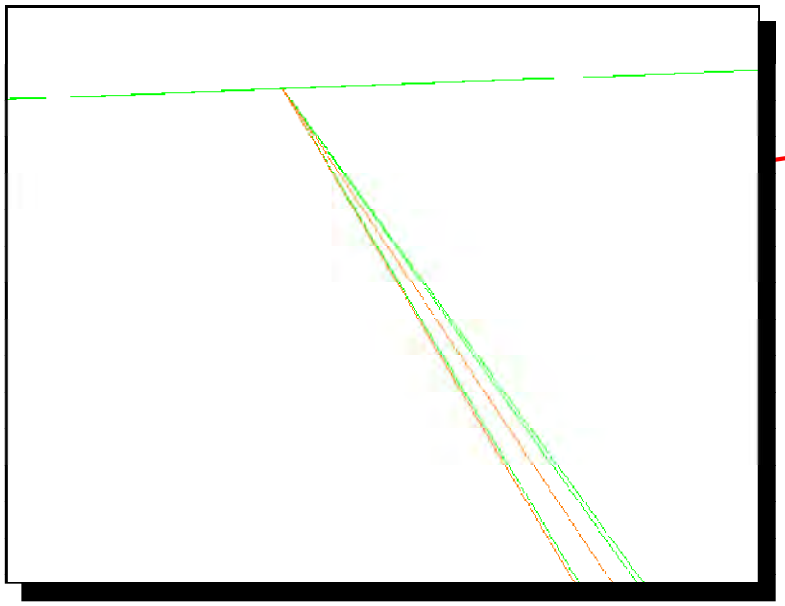
Copyright© 2012 MSC.Software Corporation

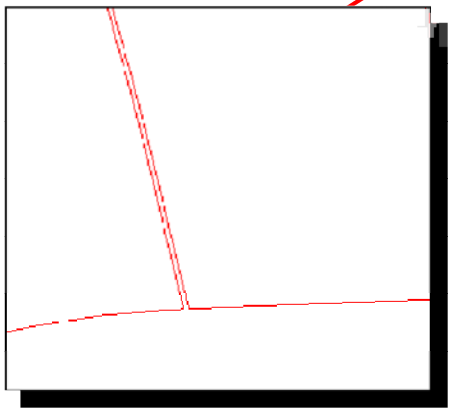
CASE STUDY: ASM EXAMPLE

- **Close-up view of problem areas**



Overlapped
surface boundaries





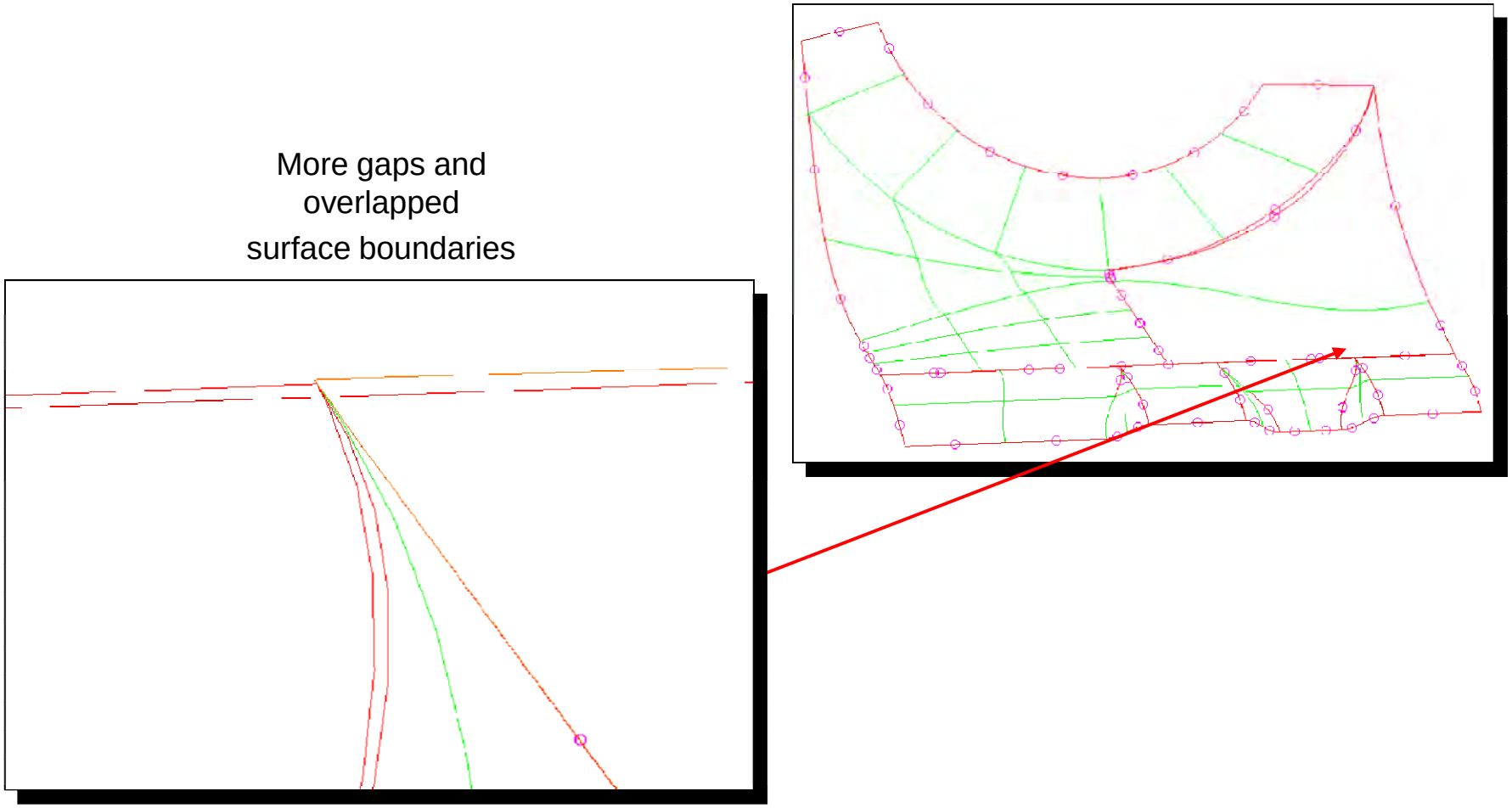
Gap

Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

- **Close-up view of problem areas (Cont.)**

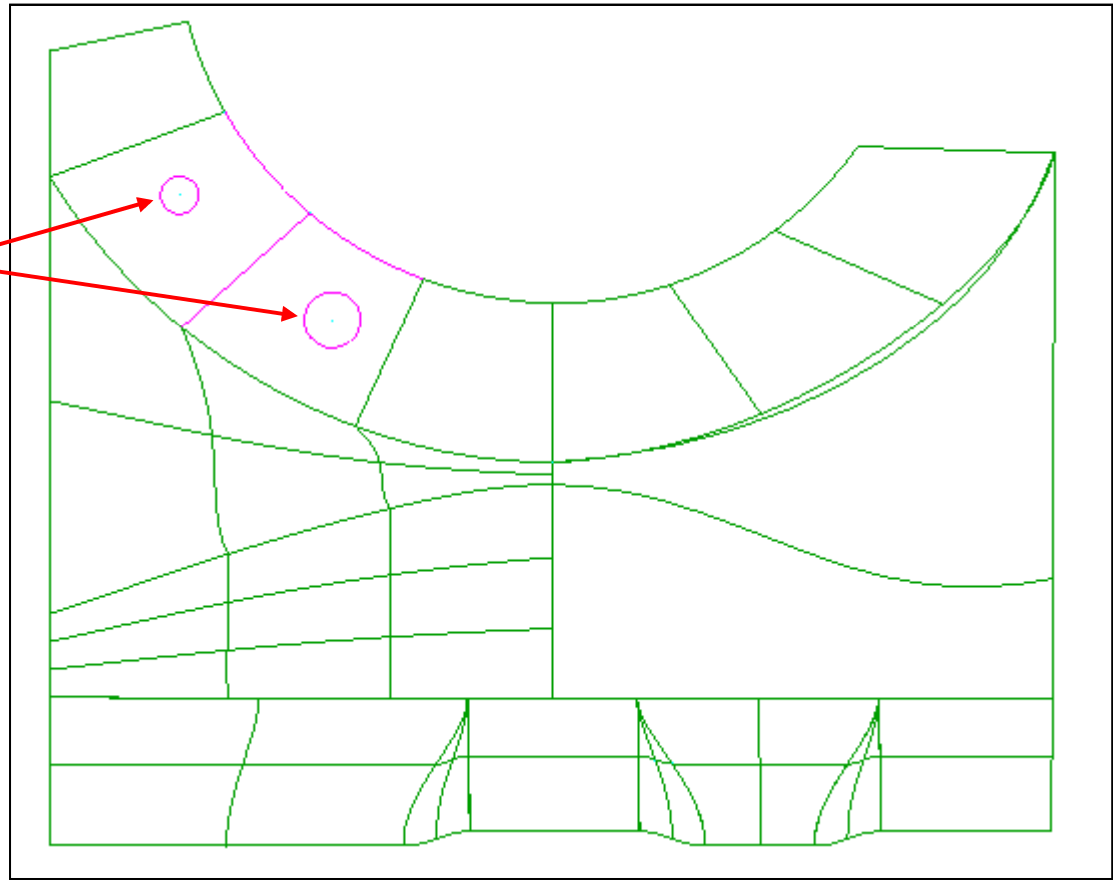
More gaps and
overlapped
surface boundaries



Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

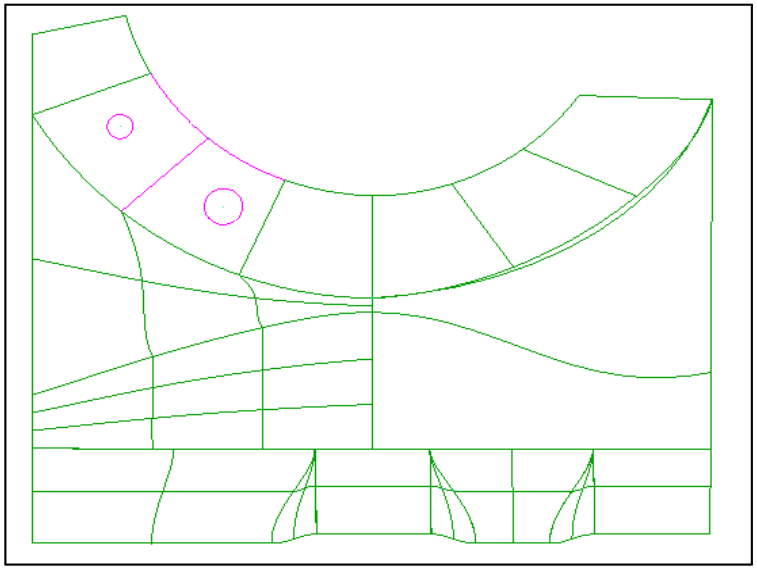
Geometry:
Edit/Surface/Add Hole to add two holes to the geometric surface model.
Radii are 2.0 or 3.0 units.



Copyright© 2012 MSC.Software Corporation

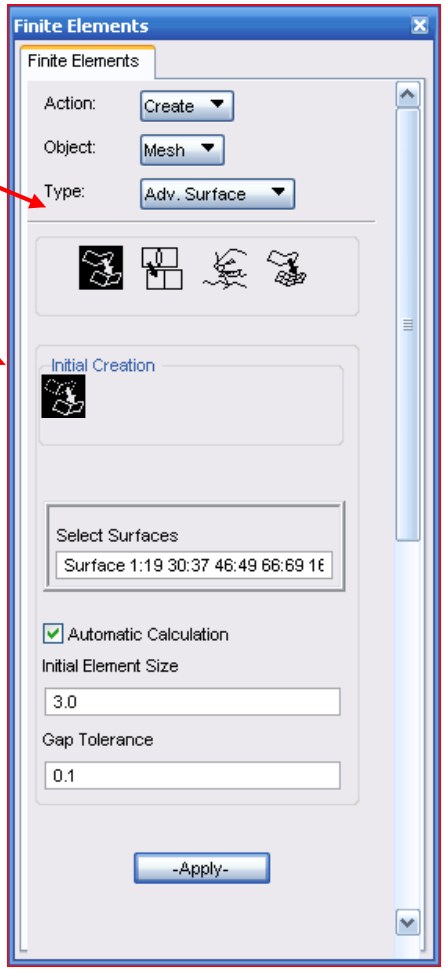
CASE STUDY: ASM EXAMPLE

Create initial tessellated surfaces:
Click on the **Create Surfaces** icon.
Select **Auto Tessellated Surface** icon.
For Select Surfaces select all geometric surfaces.
Enter an initial element size of **3.0**
Click **Apply**.



Create Surfaces

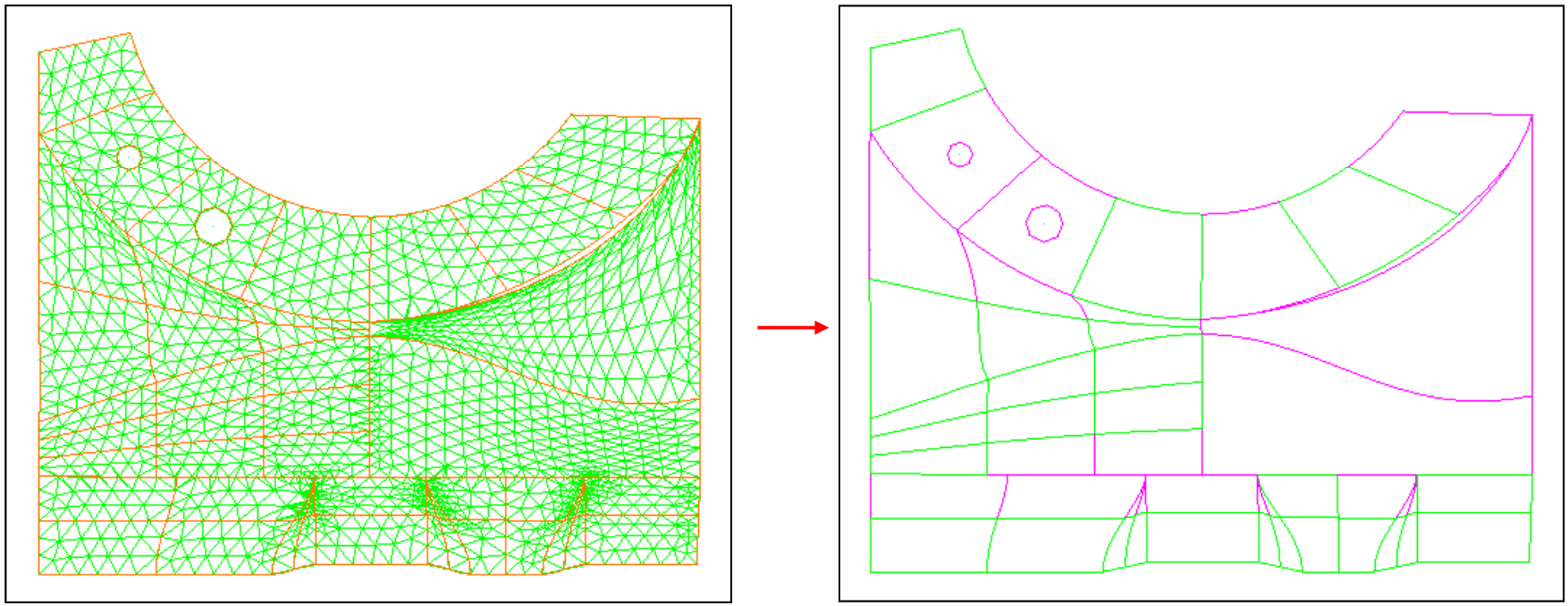
Auto Tessellated Surface



Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

Creates a tria element mesh for each original surface
Then, create a tessellated surface for each tria element mesh



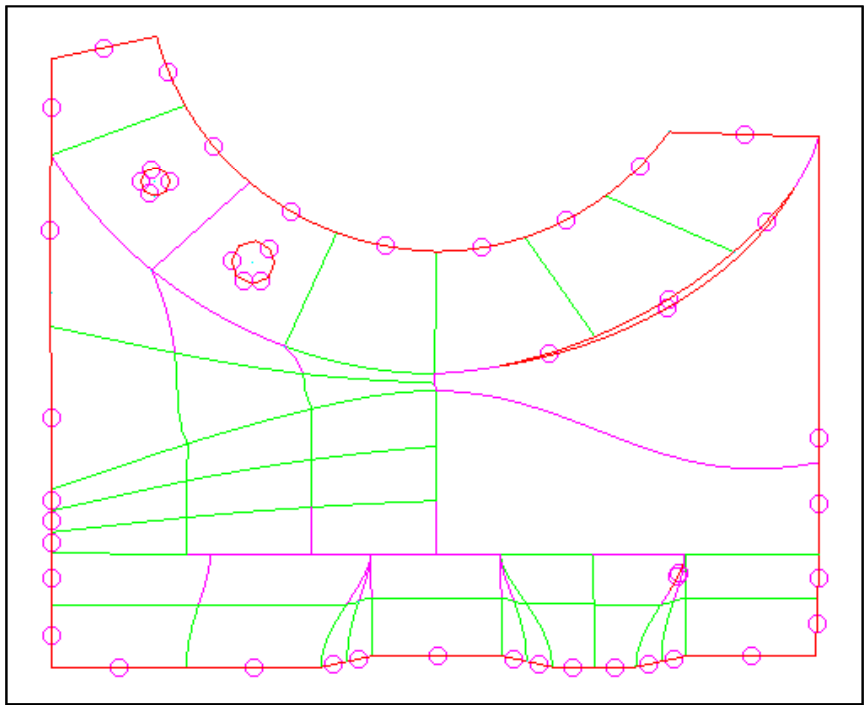
About 2460 Tria Elements

Tessellated Surfaces 1:48

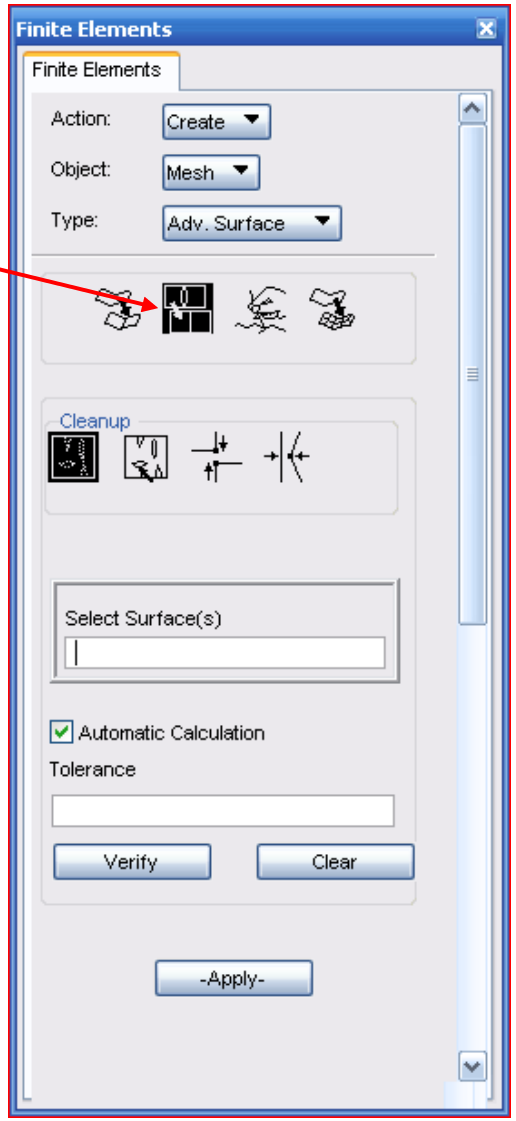
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

Click on the **Cleanup** icon, and click **Verify** to show free edges.



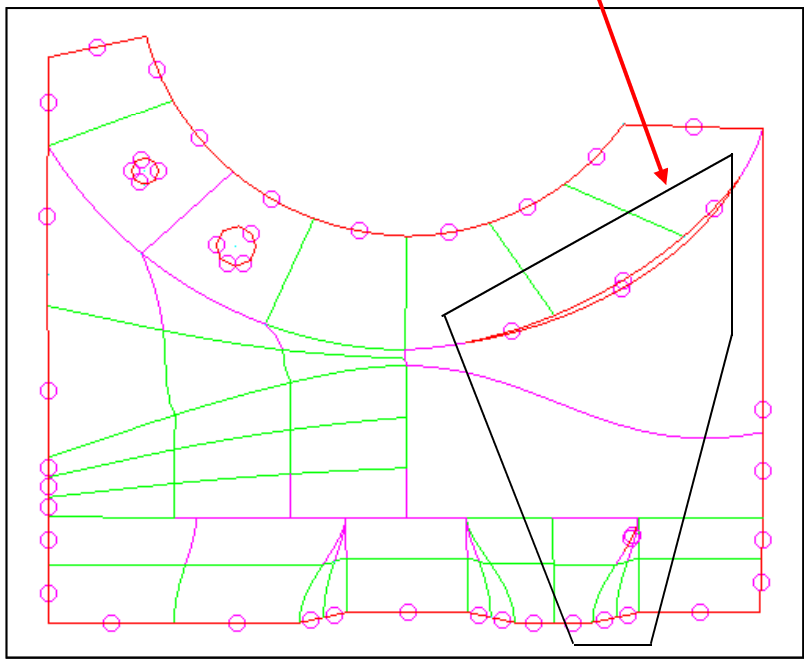
Cleanup

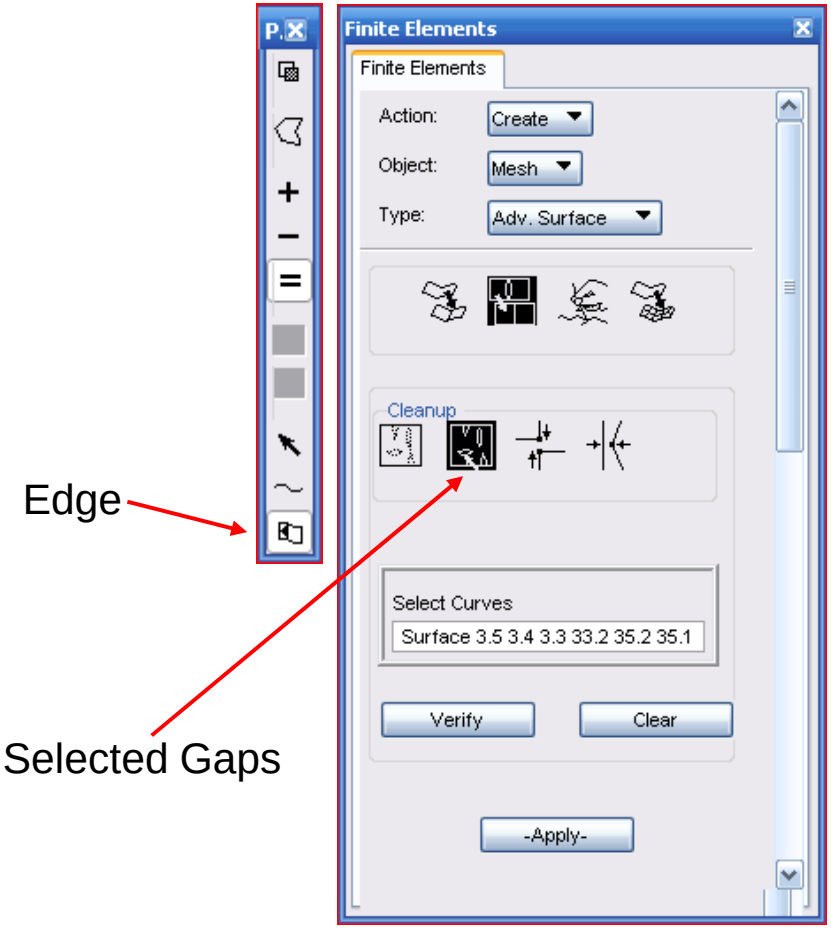


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

Click on the **Selected Gaps** icon under Cleanup.
Select the **Edge** icon from the filter select menu
Polygon pick the two gaps
Click **Apply**

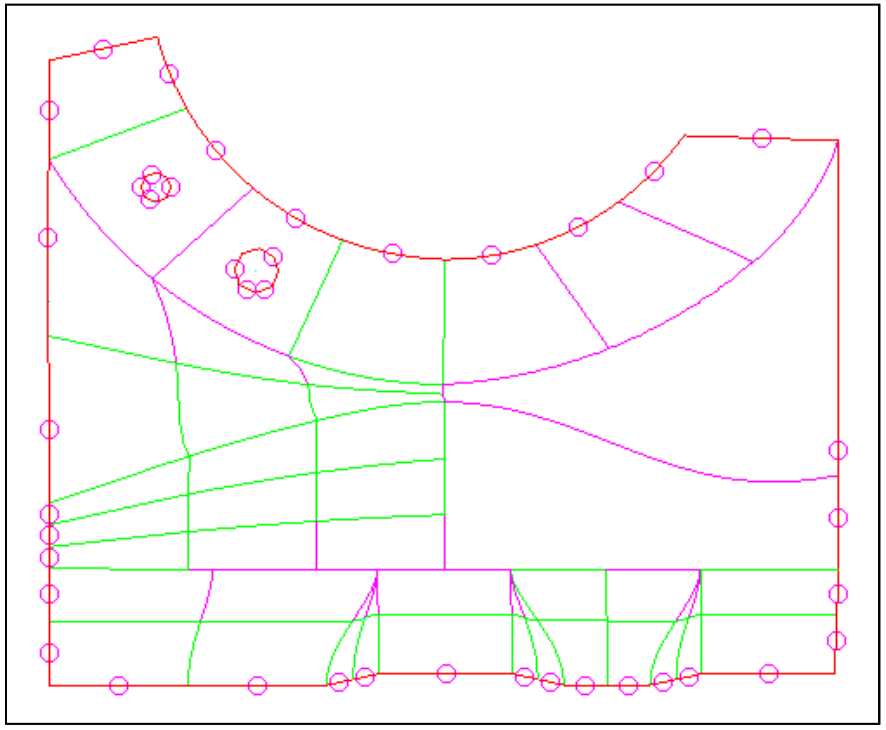


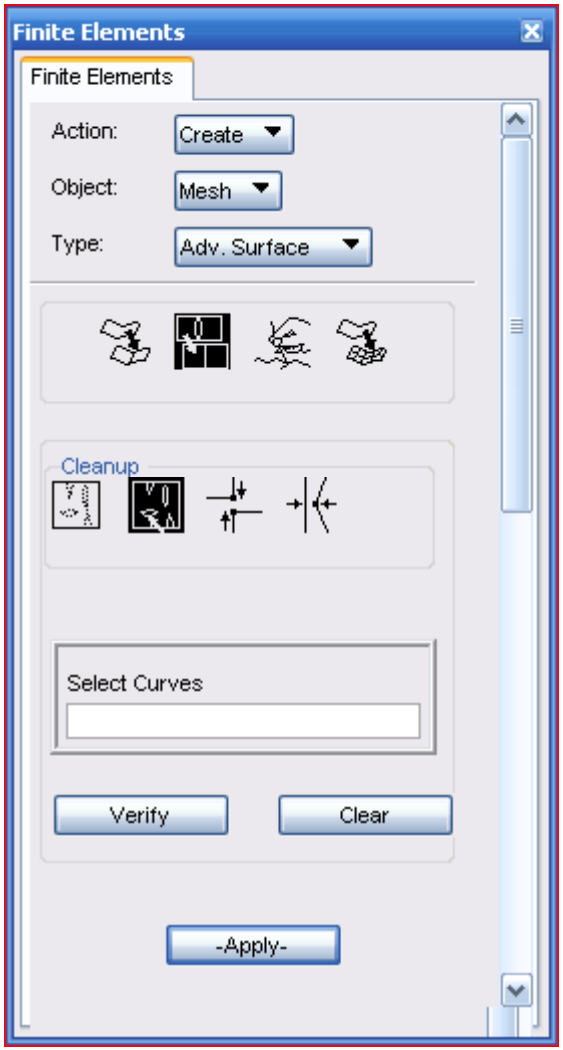


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

Click **Verify**
The two gaps have been closed by stitching

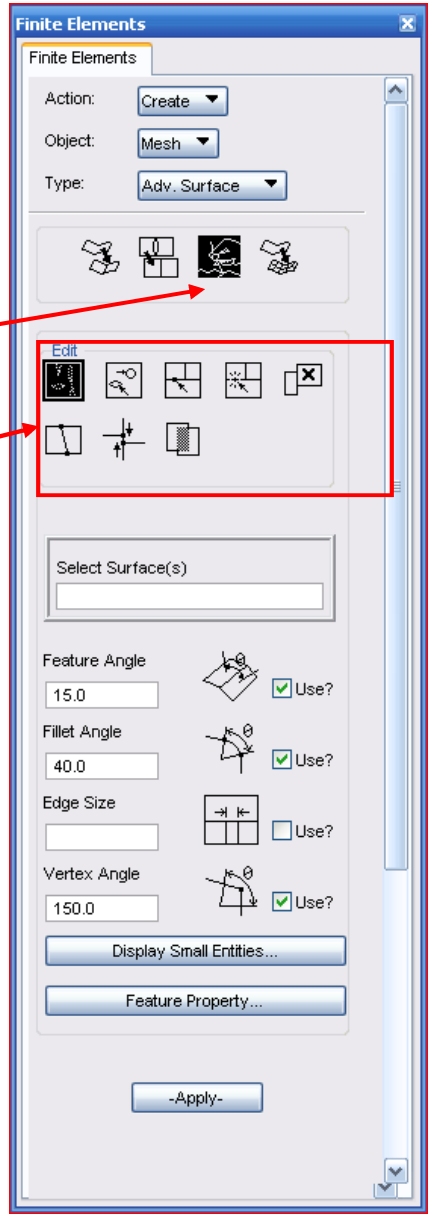




Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

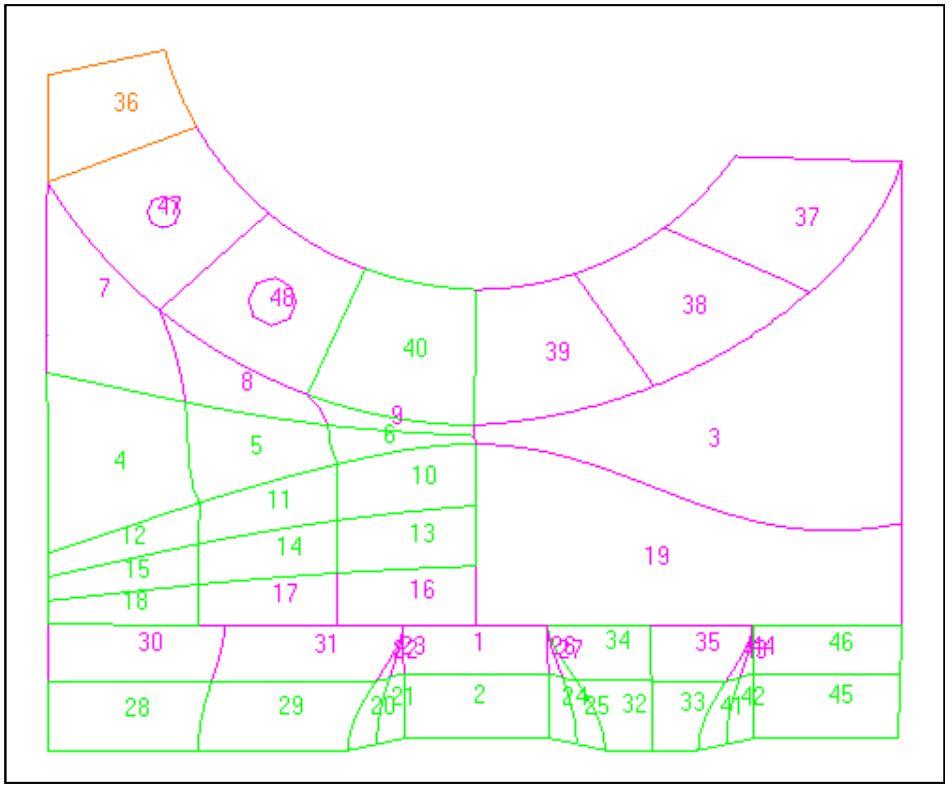
Next, click on the Edit icon
There are 8 Edit tools
- Auto Merge
- Fill Hole
- Insert Vertex
- Delete Point
- Delete Surface
- Split Surface
- Collapse Curve
- Merge Surfaces



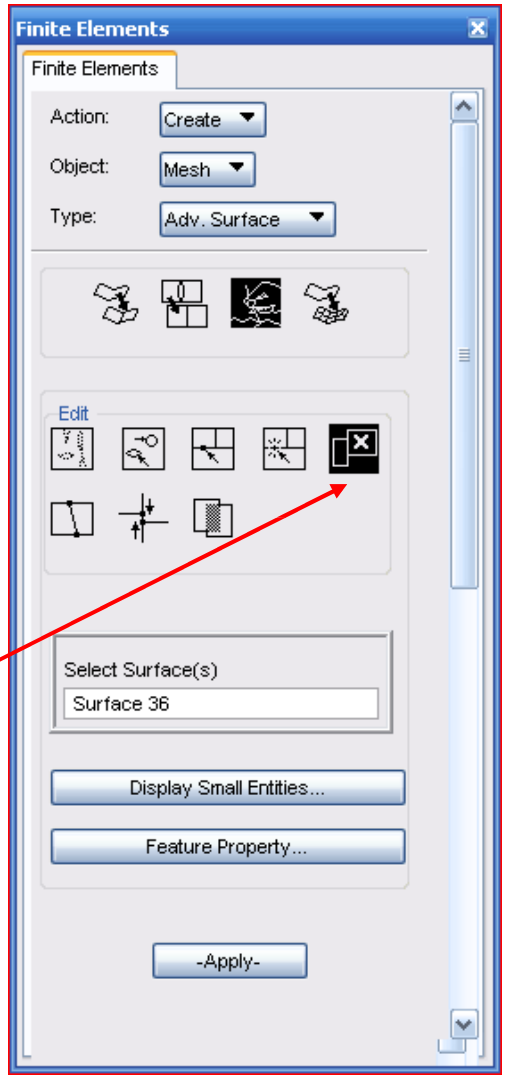
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

Delete a surface:
Click on the **Delete Surface** icon under Edit
Select **Surface 36**
Click **Apply**

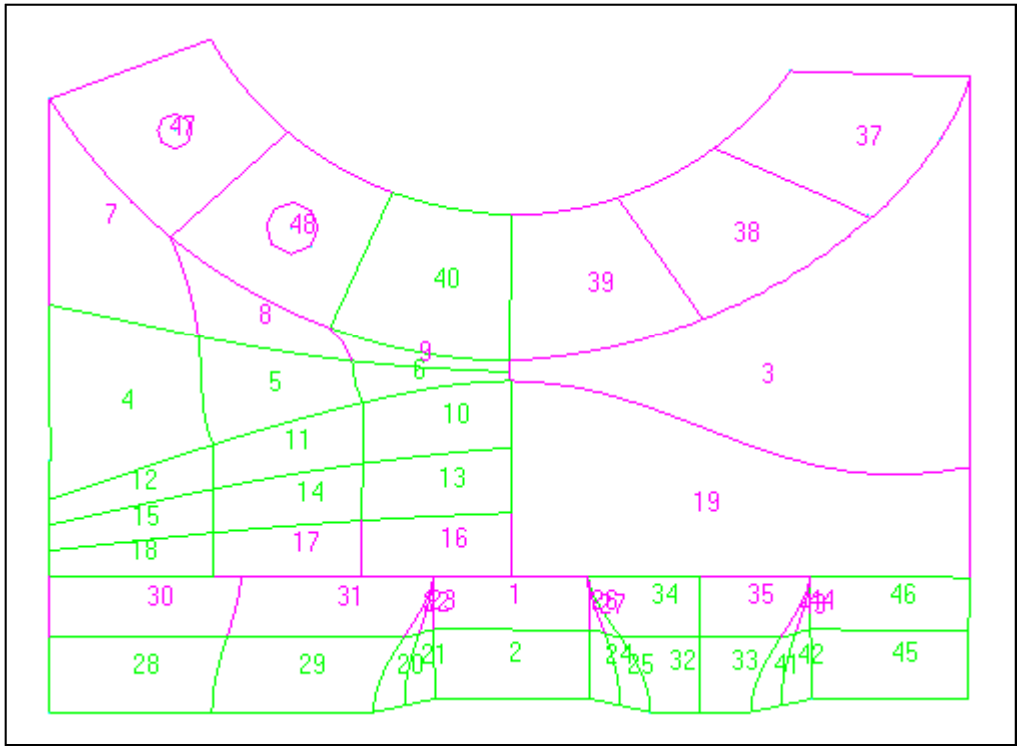


Delete Surface



Copyright© 2012 MSC.Software Corporation

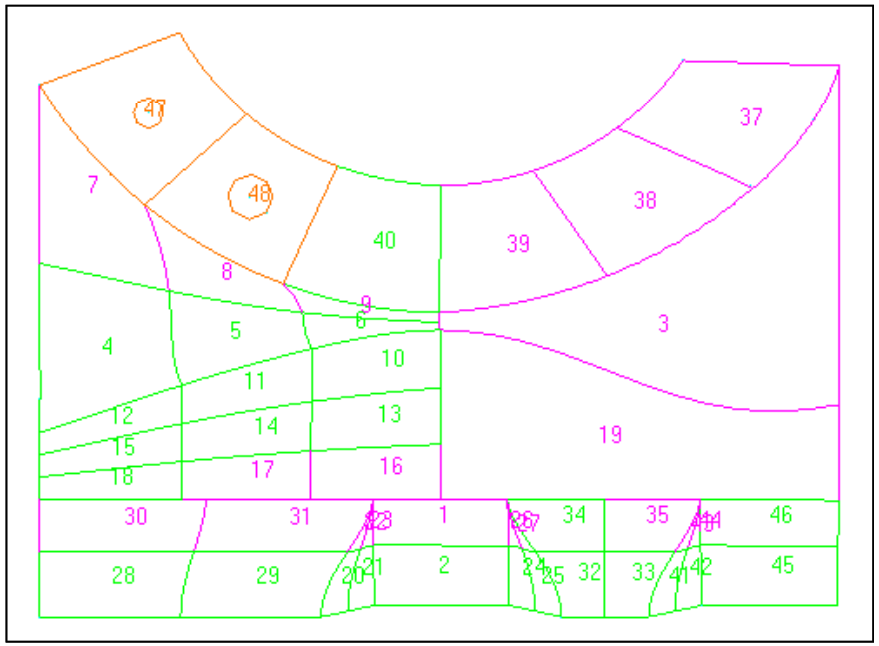
CASE STUDY: ASM EXAMPLE



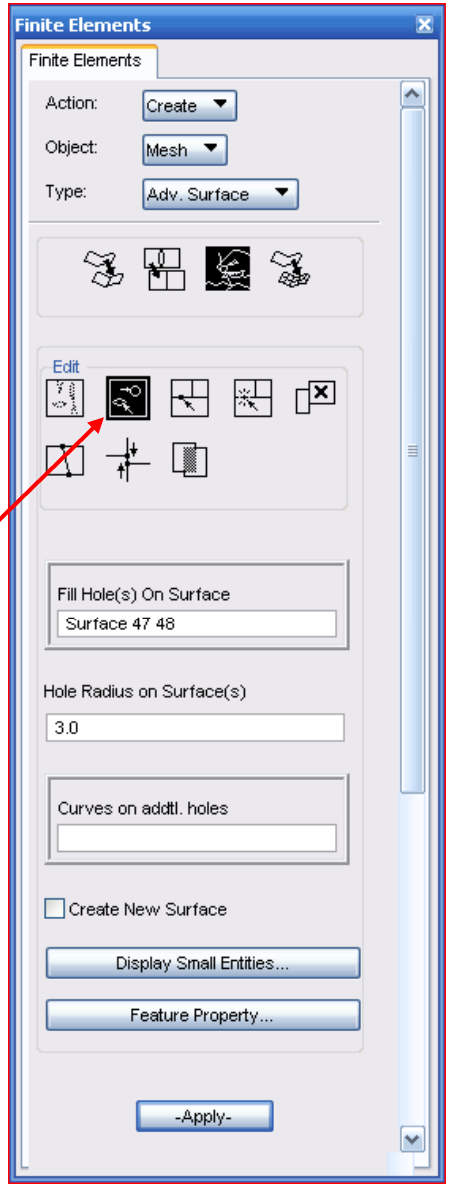
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

Next, remove the two holes:
Click on the **Fill Hole** icon under Edit
Select **Surface 47, 48** containing the holes
For the maximum hole radius enter **3.0**
Click **Apply**



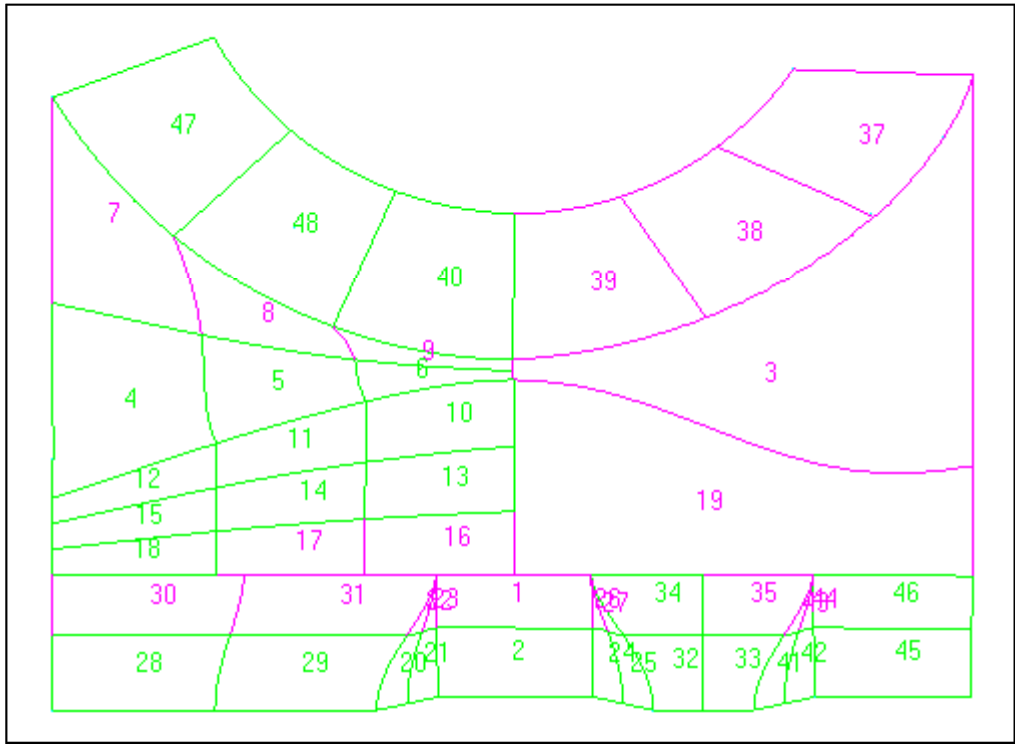
Fill Hole



Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

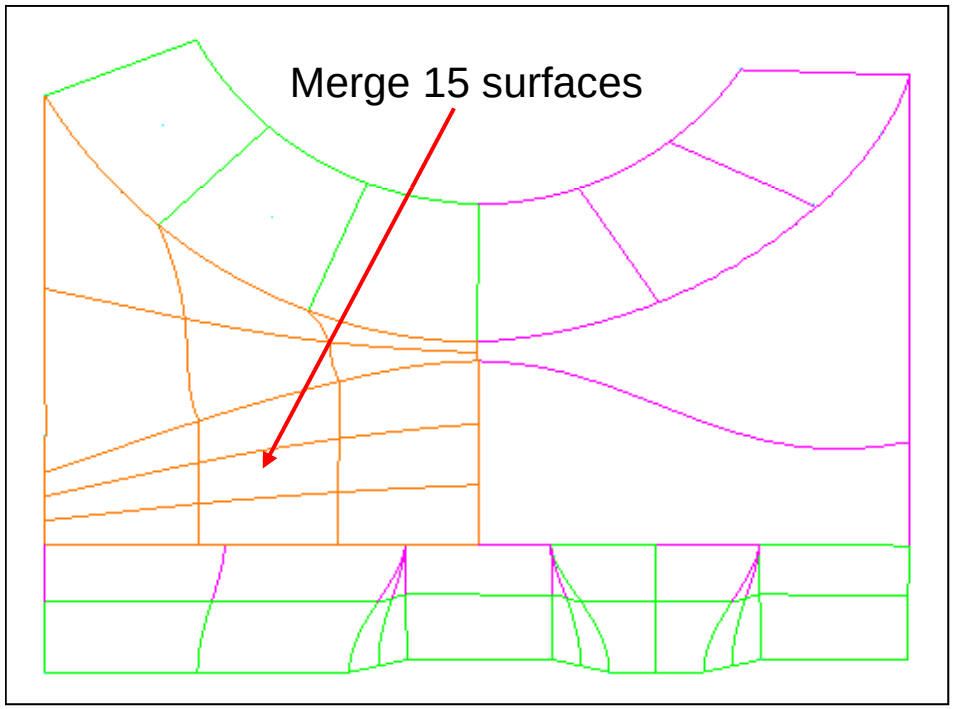
The holes have been removed



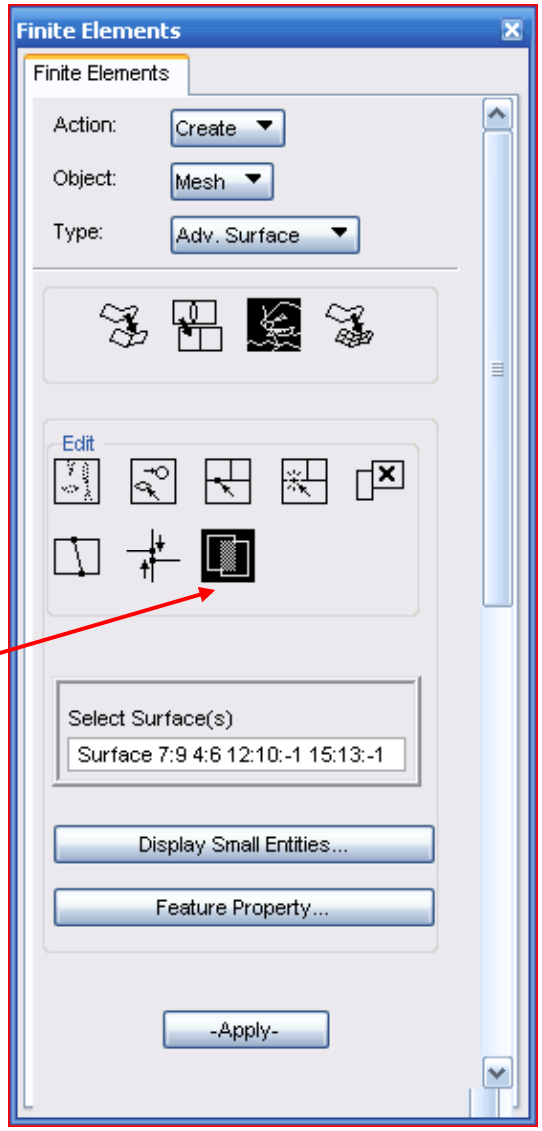
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

Next, merge surfaces to eliminate skewed or sliver surfaces:
Click on the **Merge Surfaces** icon under Edit
Select the 15 surfaces to be merged
Click **Apply**



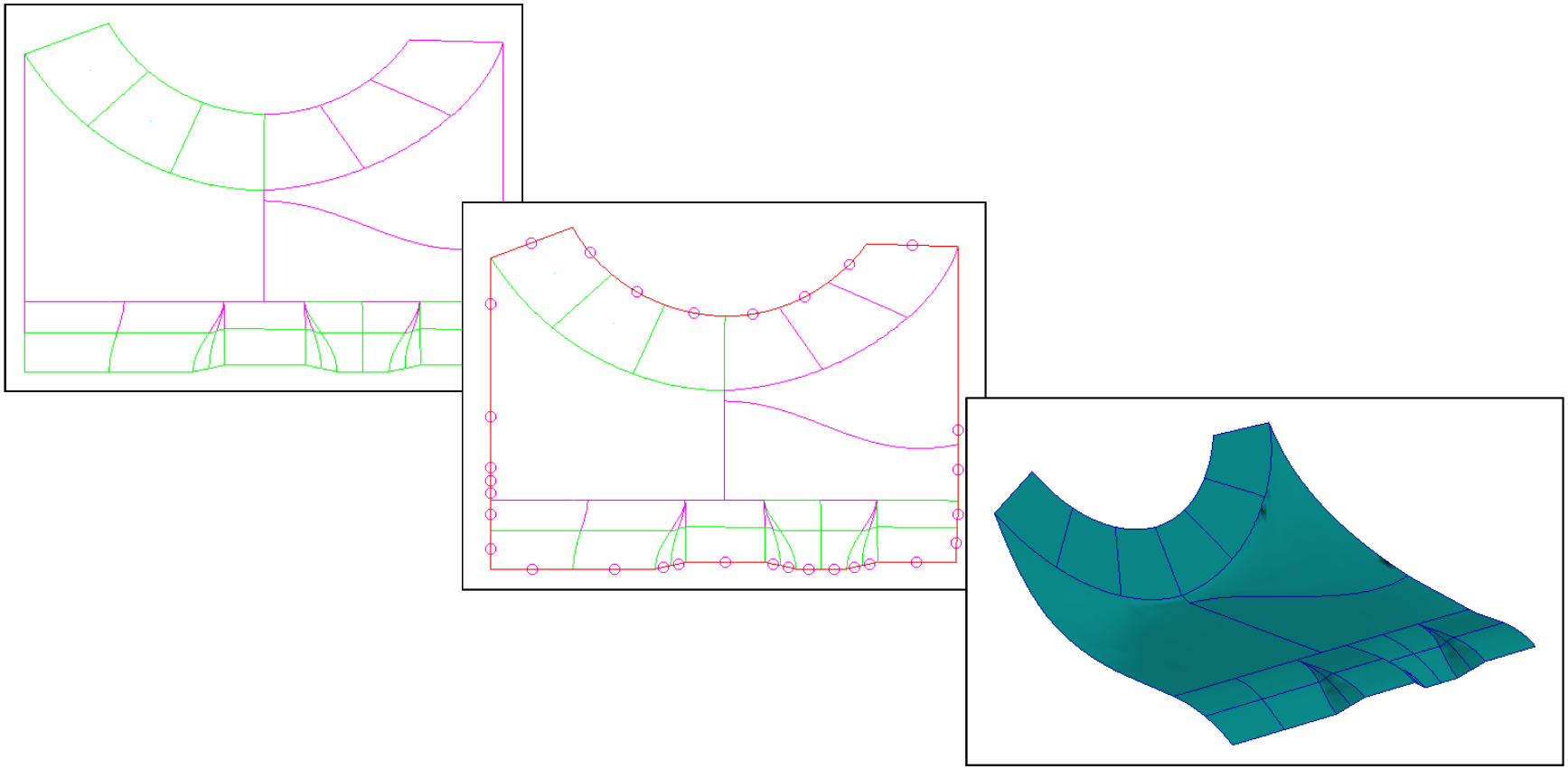
Merge Surfaces



Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

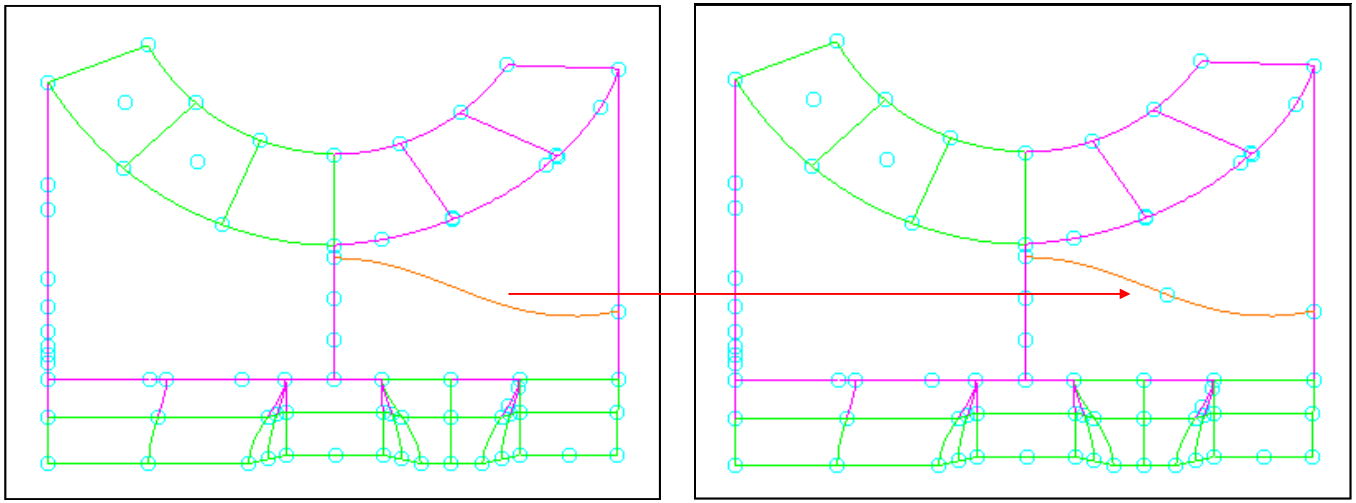
15 surfaces have been merged into one surface

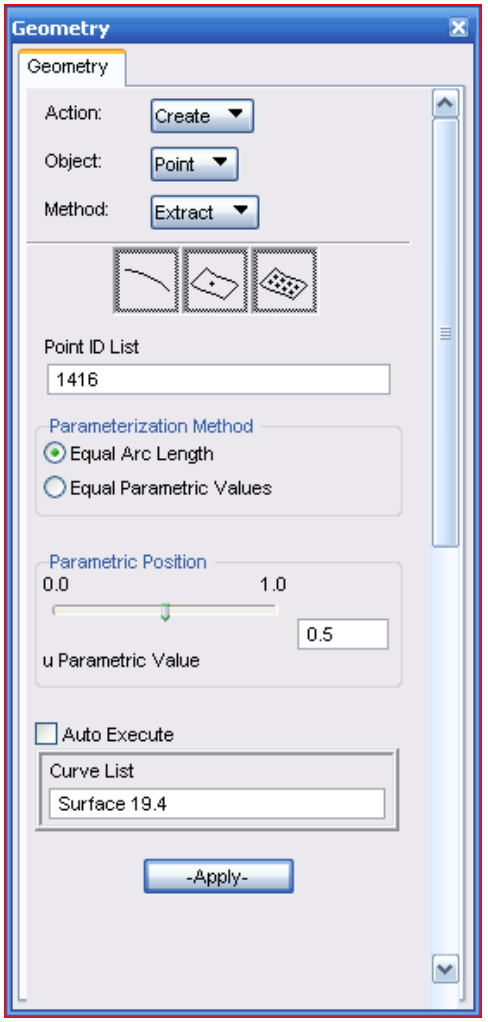


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

Next, split a surface:
First, it is necessary to add a vertex to a surface edge
Under the Geometry tab click on **Extract** from the Points group.
Select the indicated surface edge
Click **Apply**

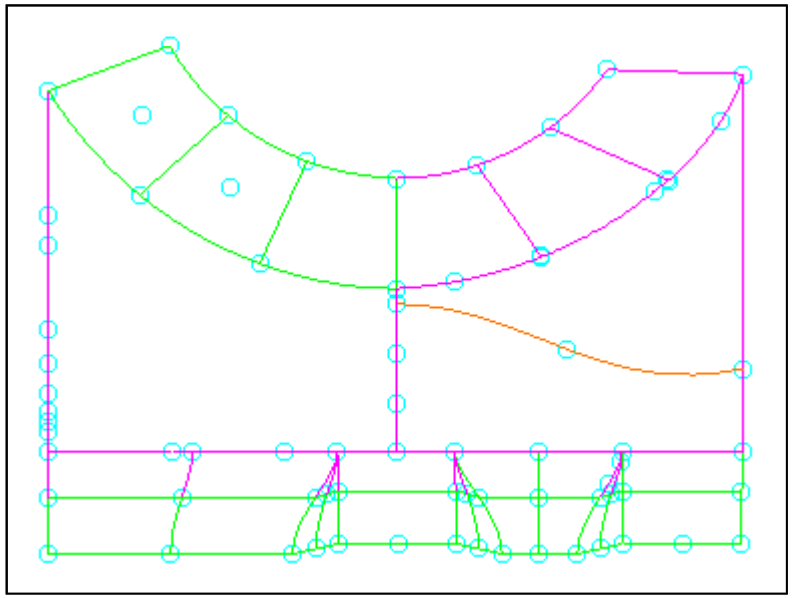




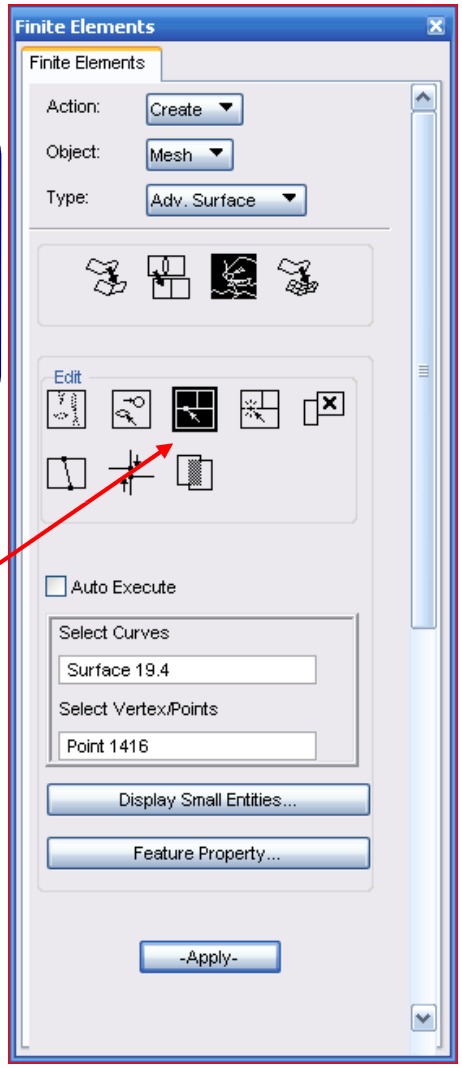
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

Using the ASM, click on the **Insert Vertex** icon under Edit
Select the surface edge where the vertex is to be created, **Surface 19.4**
Select the Point that was just created, **Point 1416**
Click **Apply**



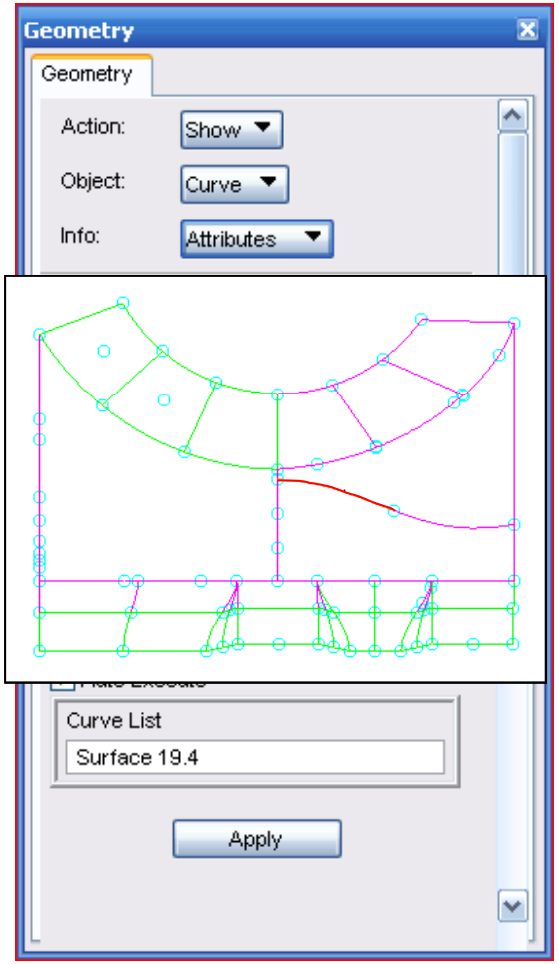
Insert Vertex

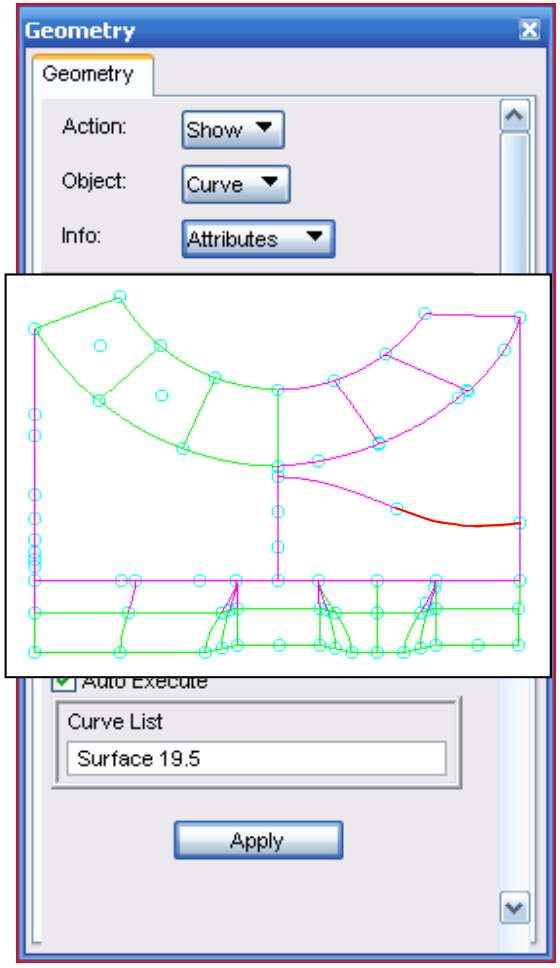


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

A vertex has been added to the surface edge

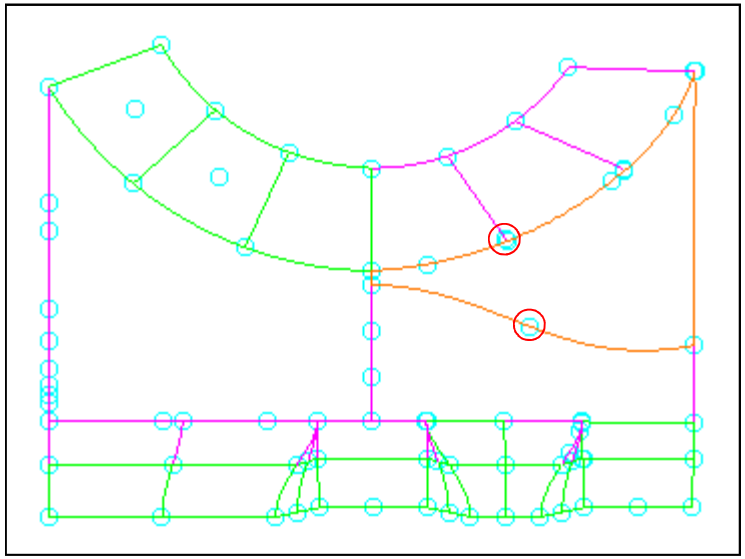




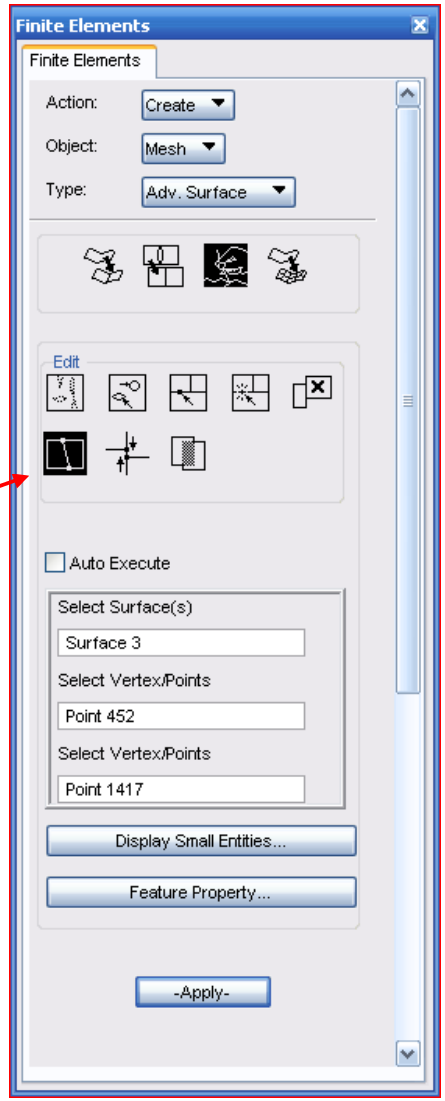
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

Now, split a surface:
Click on the **Split Surface** icon under *Edit*
Select the indicated surface
Select a point on one side of the surface
Select a point on the other side of the surface
Click **Apply**



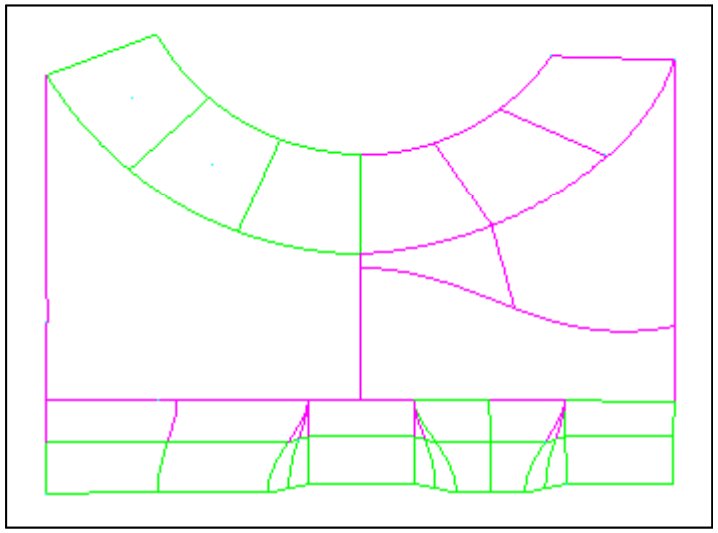
Split Surface

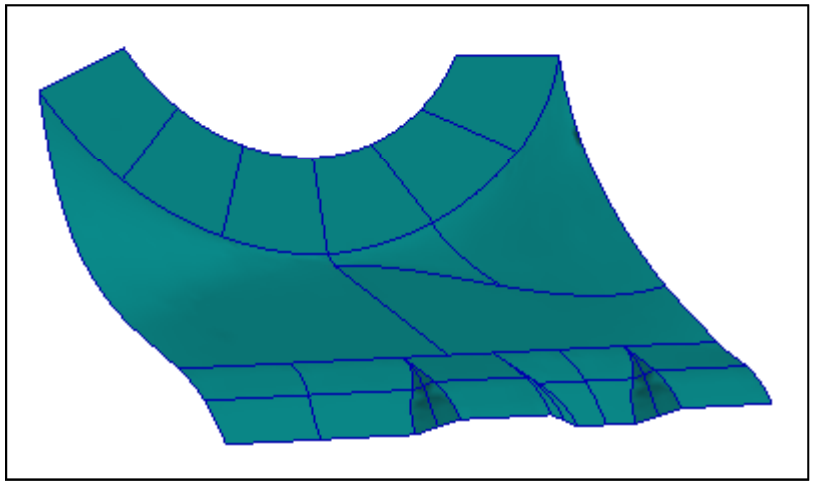


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

The surface has been split

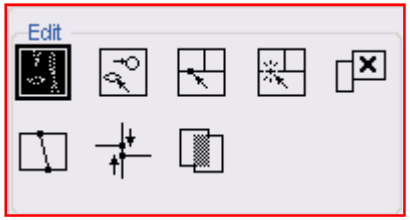




Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

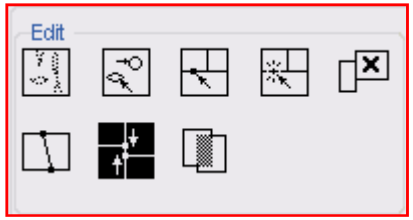
Next, can use the **Auto Merge** icon under *Edit*: This tool automatically merges the selected surfaces based on the criteria set by the user



Copyright© 2012 MSC.Software Corporation

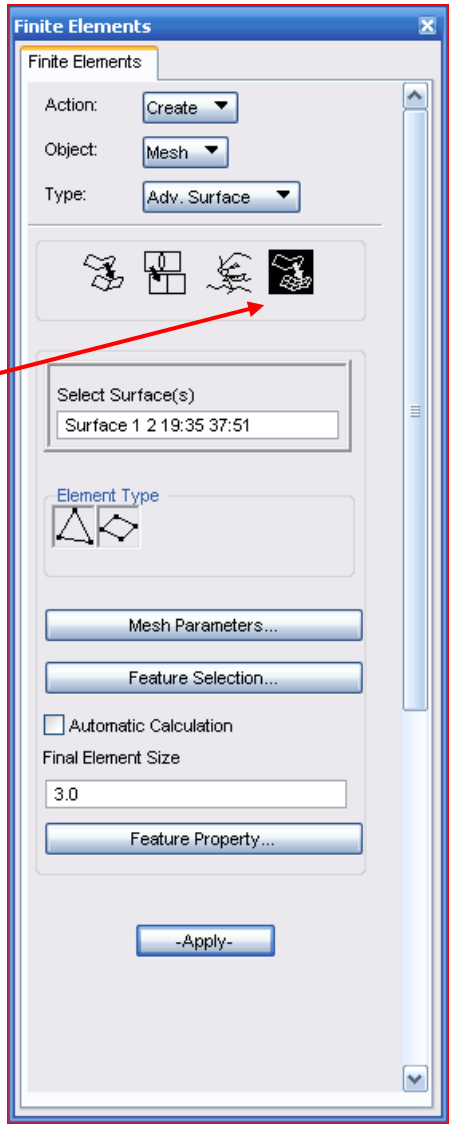
CASE STUDY: ASM EXAMPLE

Next, can use the Collapse Curve icon under Edit:
Select the curves to be collapsed
Click **Apply**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE



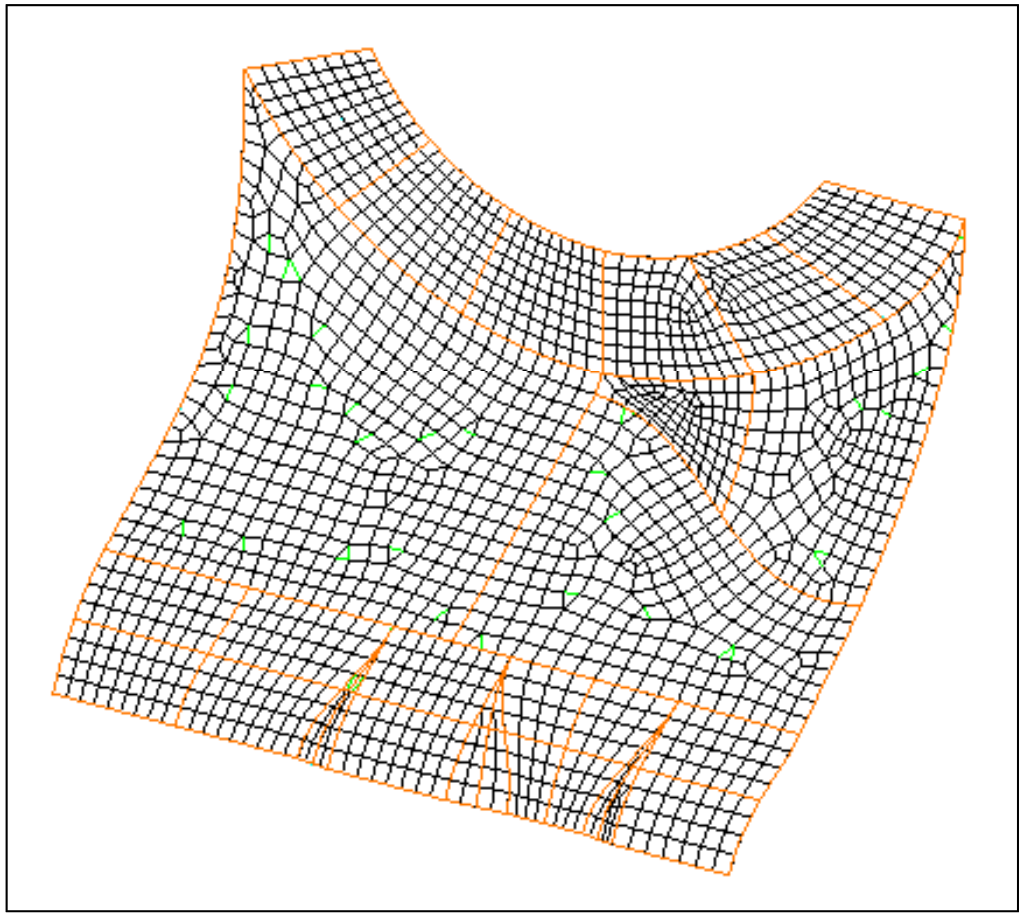
Final Mesh

Create a mesh for the set of final tessellated surfaces:
Click on the **Final Mesh** icon to create the final quad mesh
Select all the tessellated surfaces
Enter the final element size of **3.0**
Click on **Mesh Parameters**, **Feature Selection**, and/or **Feature Property** to specify Curvature Check, to add Hard Nodes, Hard Bars, etc.
Click **Apply**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASM EXAMPLE

The final mesh has been created

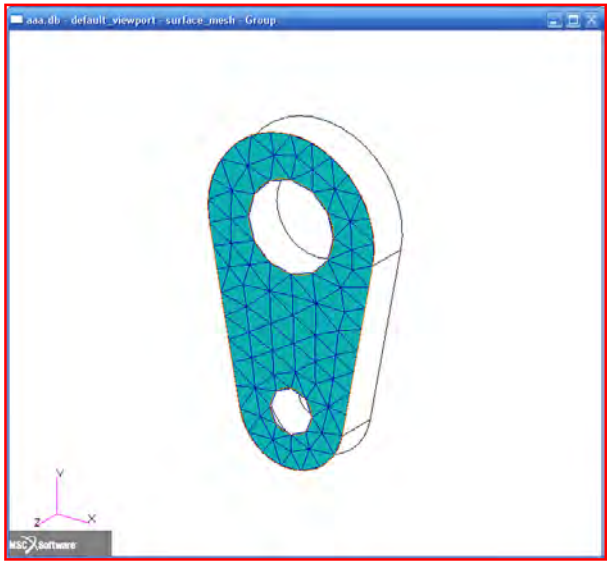


Copyright© 2012 MSC.Software Corporation

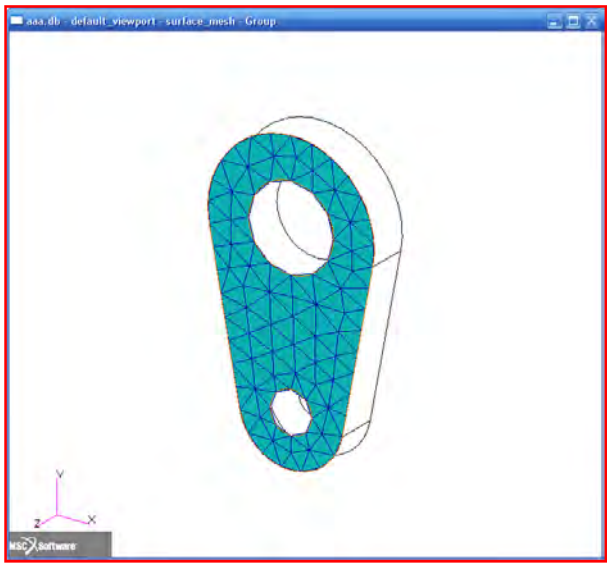
HYBRID SURFACE MESHER

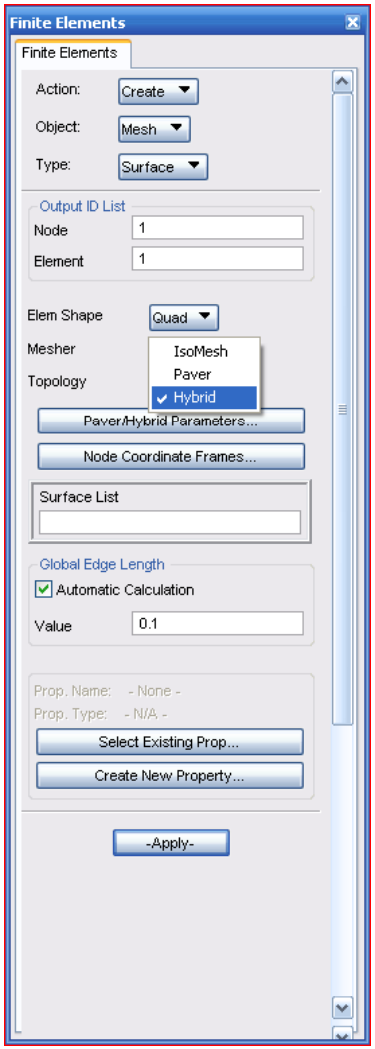
- **The Hybrid mesher is the surface mesher that the Solid Tetmesher uses internally as part of the tetmeshing process**
- **As such, it is more robust than the normal paver mesher and usually does very well with Solid Faces**
  - Tip: When Paver mesh fails to mesh a solid face, use Hybrid as an alternative
  - Tradeoff: Potential reduced element quality

Paver Mesher



Hybrid Mesher



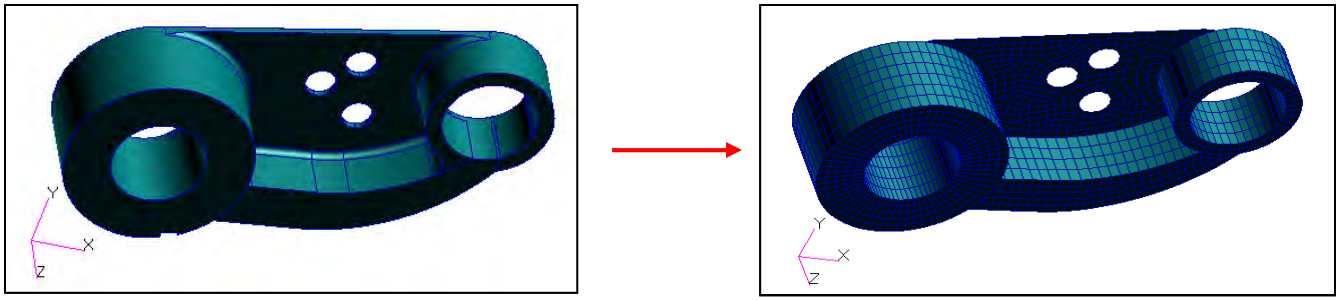


Copyright© 2012 MSC.Software Corporation

SECTION 4

STRATEGIES IN GEOMETRIC MODELING AND MESHING

COMBINED 2D AND 3D MESHING



Copyright© 2012 MSC.Software Corporation

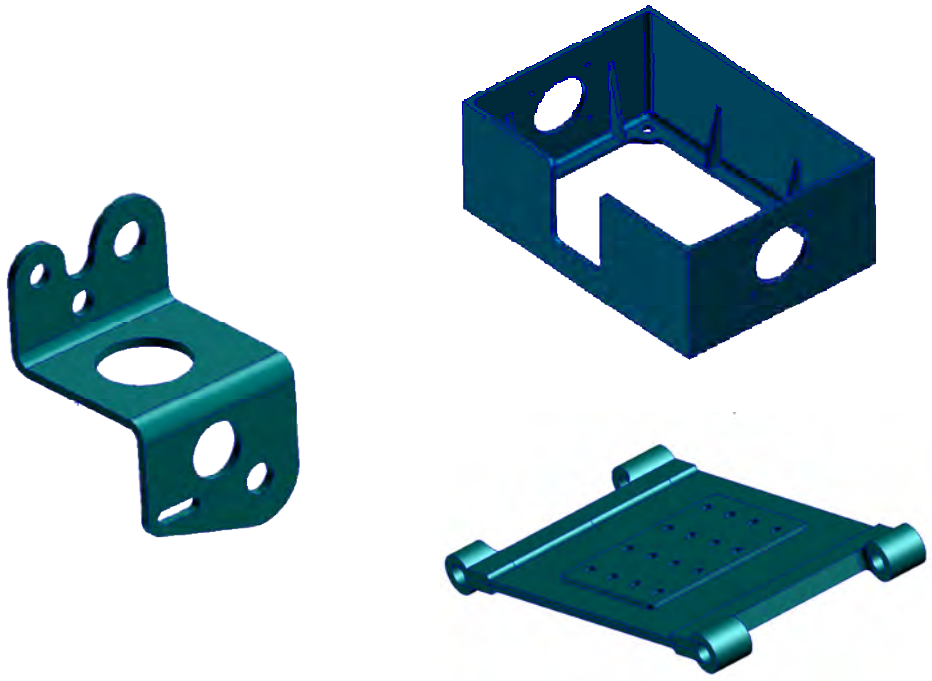
IDEALIZATION TYPE

- **Idealization types for this section**
  - 2D beginning to end
  - **3D start point to 2D idealization**
  - **3D start point to 2 ½ D idealization**
  - **3D and 2D mix**
  - 3D beginning to end

Copyright© 2012 MSC.Software Corporation

3D START POINT, 2D IDEALIZATION

- **Category: 3D start point, 2D idealization**
- **Structures falling in this category are typified by:**
  - Dominance of thin wall sections
  - 3D details like fillets may be present but are not of interest to the analyst
- **Structures include:**
  - Packaging
  - Brackets
  - Heavy frames
  - Pipes



Copyright© 2012 MSC.Software Corporation

3D START POINT, 2D IDEALIZATION

- **Motivation**
  - The number of DOFs is significantly reduced
    - Disc storage is reduced
    - CPU time is reduced
    - Results interpretation may be simplified
    - Results data storage is reduced
- **Justification**
  - The structural behavior is assumed
    - Dominated by membrane, bending, and in-plane shear
    - Local 3D effects can be neglected

Copyright© 2012 MSC.Software Corporation

3D START POINT, 2D IDEALIZATION

- **Focus on 3D geometry starting point here**
  - It is unlikely to want to transform a 3D mesh into a 2D mesh.
  - If starting from nothing, it would build surface geometry.

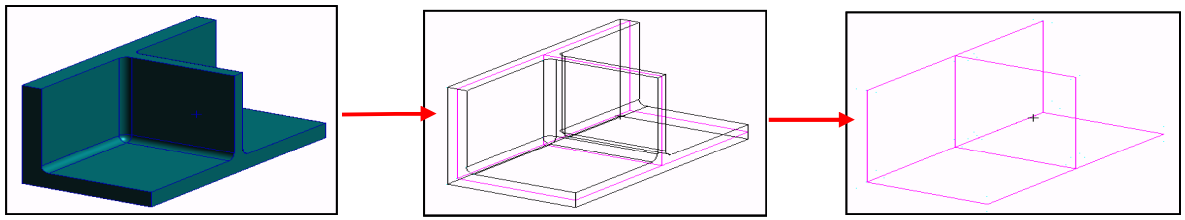
- **Methods available:**
  - 3D geometry break it down and rebuild to 2D geometry
  - Mid-surface tool extracts mid-surfaces from geometric solids
  - Mid plane mesher meshes geometric solids directly

Start Point 3D	Nothing		Mesh		Geometry		
Method	N/A	N/A	N/A	N/A	Mid Plane Mesher	Mid Surface Extraction	Geometry breakdown and rebuild methods
End Point 2D	Mesh	Geometry	Mesh	Geometry	Mesh	Geometry	

Copyright© 2012 MSC.Software Corporation

MID-SURFACE EXTRACTION

- **The new Mid-surface tool is used to extract mid-surfaces from solids with thin walls.**
- **The user has the choice of using the single-click automatic extraction or manual extraction.**



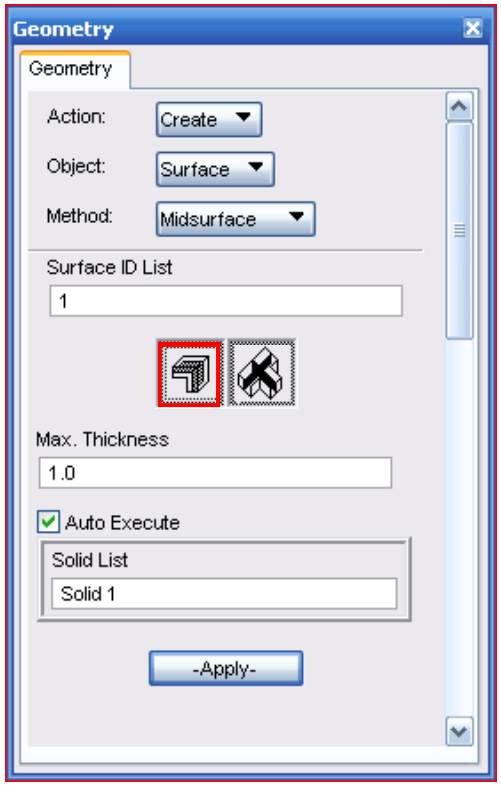
Start Point 3D	Nothing		Mesh		Geometry		
Method	N/A	N/A	N/A	N/A	Mid Plane Mesher	Mid Surface Extraction	Geometry breakdown and rebuild methods
End Point 2D	Mesh	Geometry	Mesh	Geometry	Mesh	Geometry	

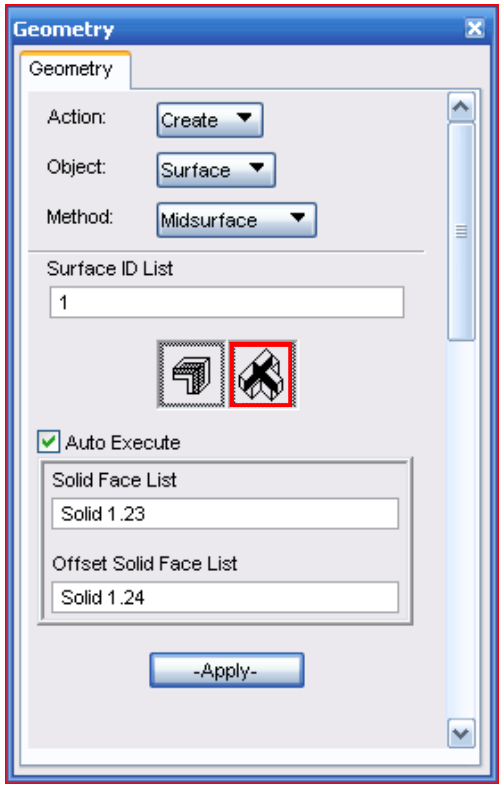
Copyright© 2012 MSC.Software Corporation

MID-SURFACE EXTRACTION

- **Mid-surface extraction uses two approaches:**

Automatic **Manual**

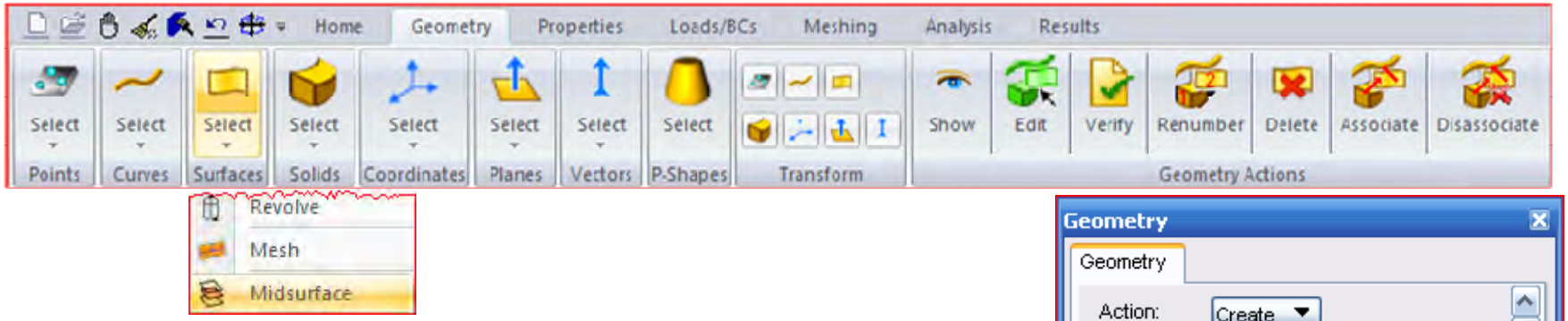


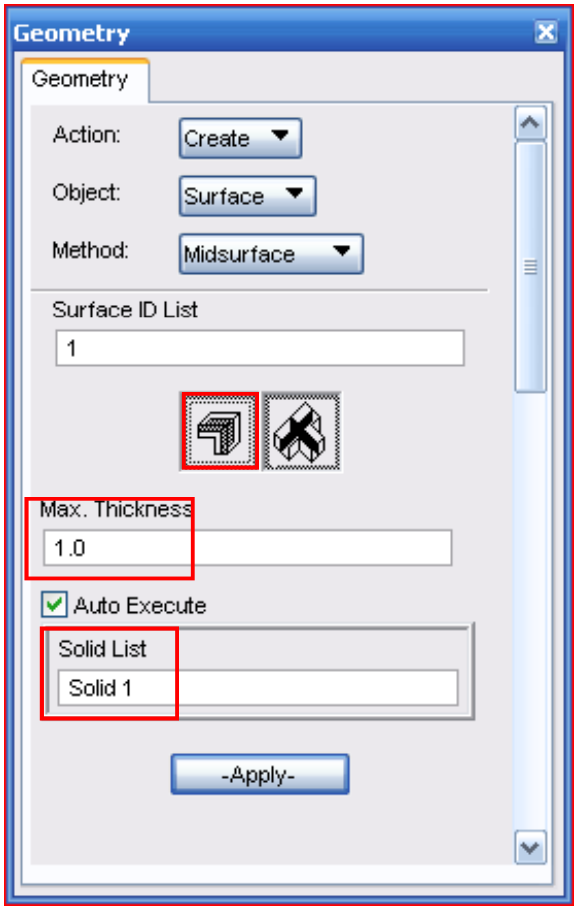


- **Parasolid Modeling license is required for this operation.**

Copyright© 2012 MSC.Software Corporation

MID-SURFACE EXTRACTION, AUTOMATIC



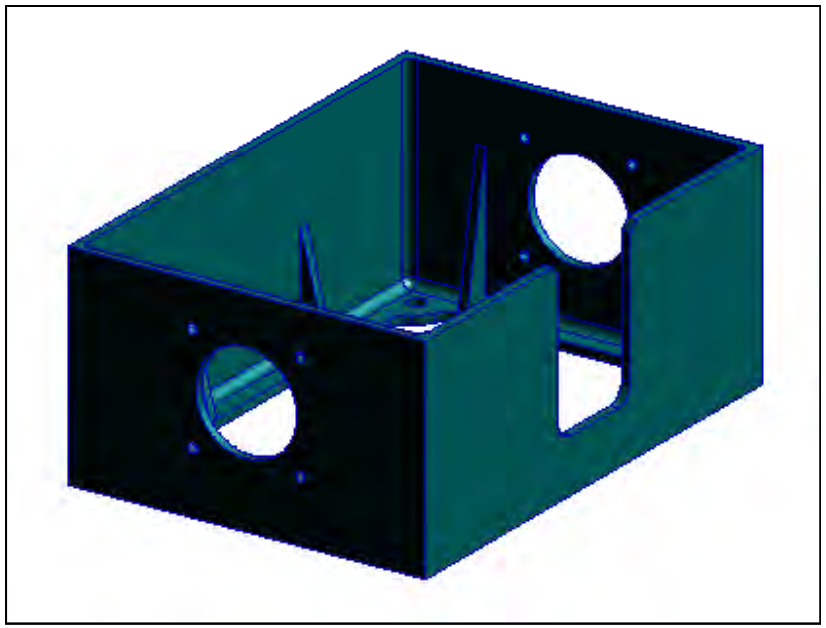


- **Automatic Mid-surface extraction works simply with the selection of a solid and a maximum thickness.**

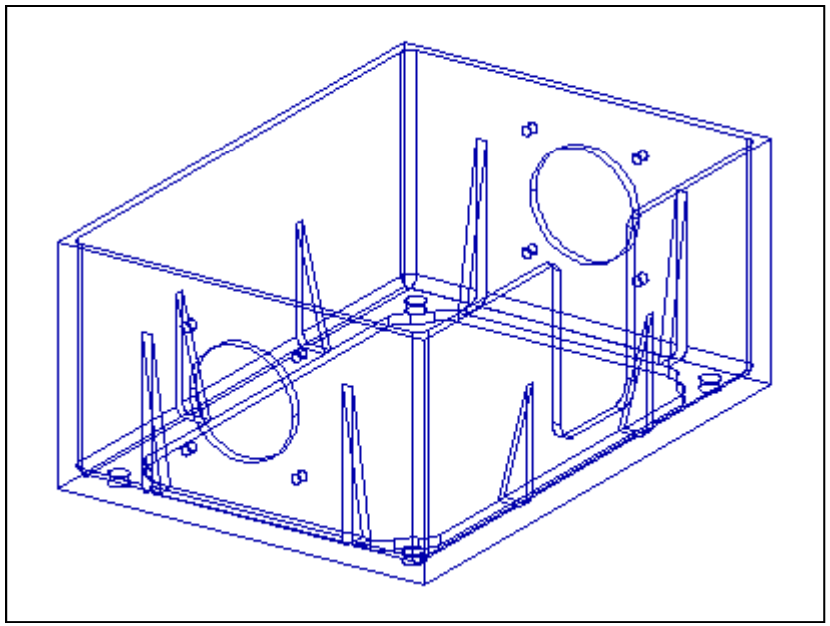
Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-SURFACE EXTRACTION, AUTOMATIC

- **Below, is a parasolid solid for which mid-surfaces are to be created:**
  - The part is a thin walled box with multiple cutouts and gussets.



Smooth Shaded View

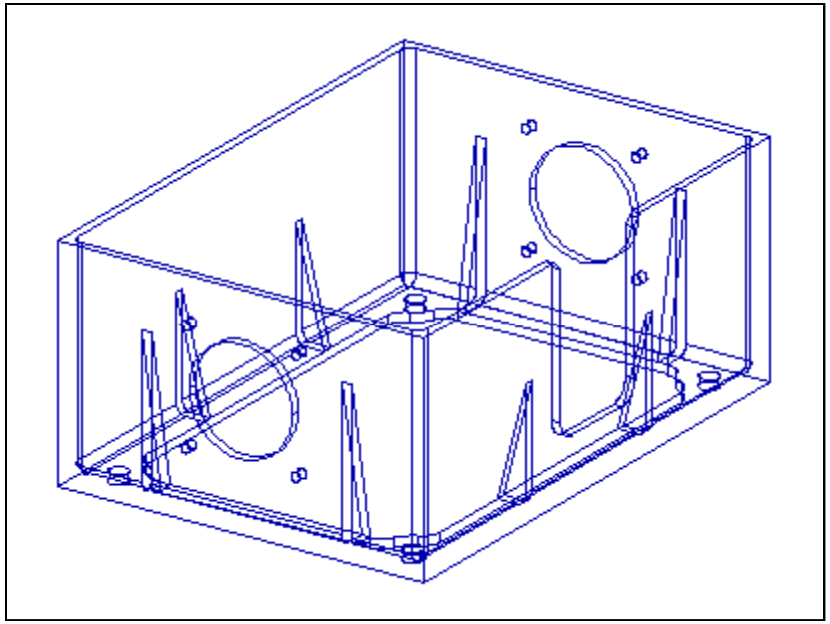


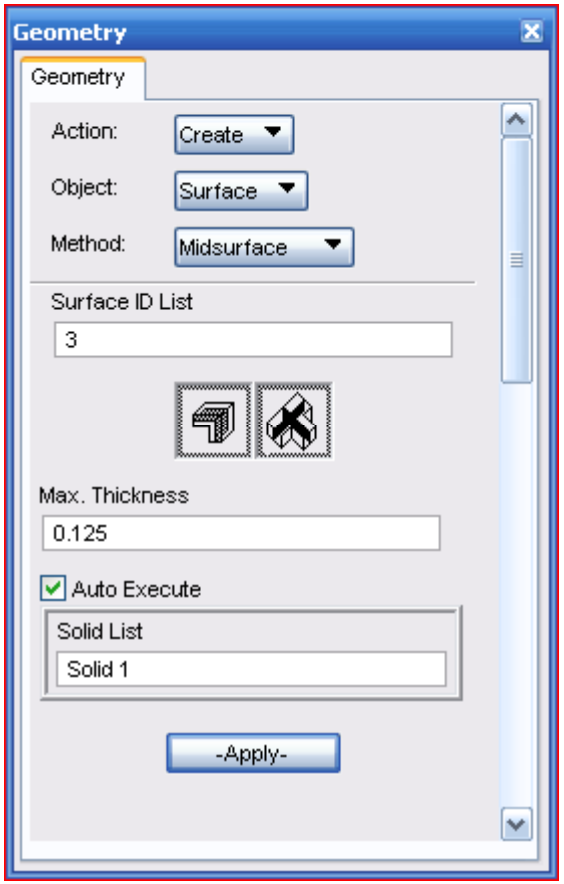
Wire Frame View

Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-SURFACE EXTRACTION, AUTOMATIC

To create the mid-surfaces, simply set the *Max. Thickness* and select a set of solids from which the mid-surfaces are desired.

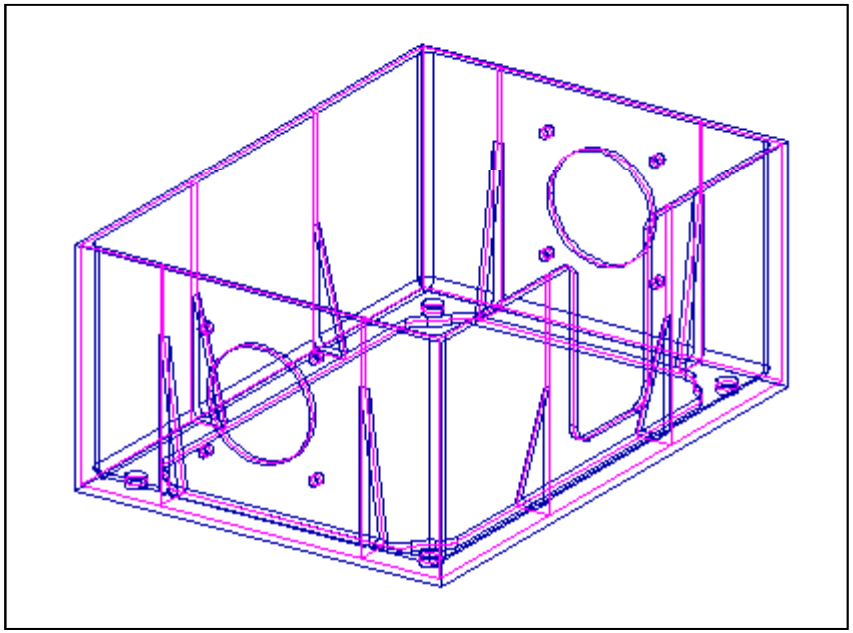




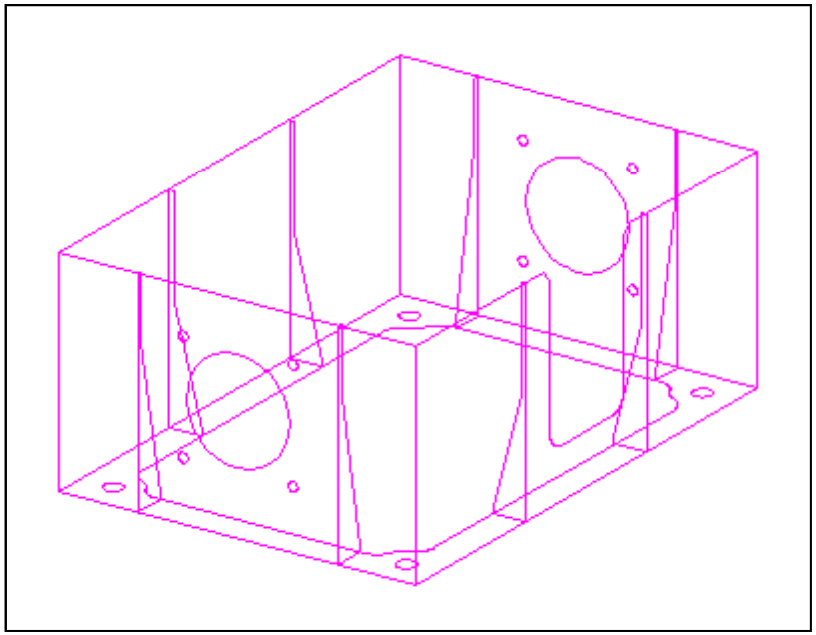
Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-SURFACE EXTRACTION, AUTOMATIC

- **After selecting the solid, and setting Max. Thickness to 0.125, the following mid-surfaces were created:**



Mid-surfaces and Original solid

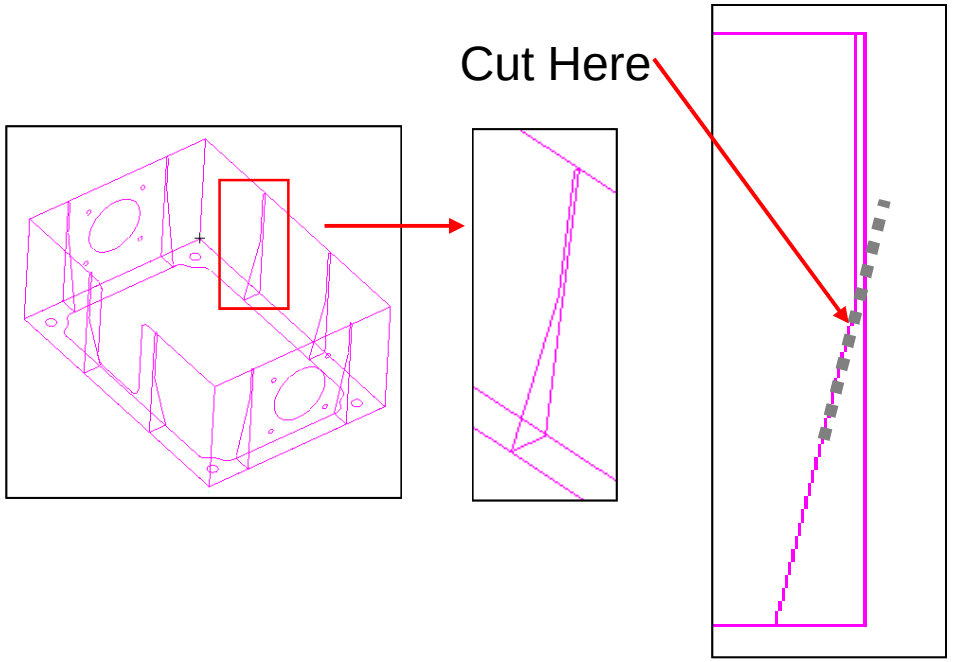


Mid-surfaces displayed (only)

Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-SURFACE EXTRACTION, AUTOMATIC

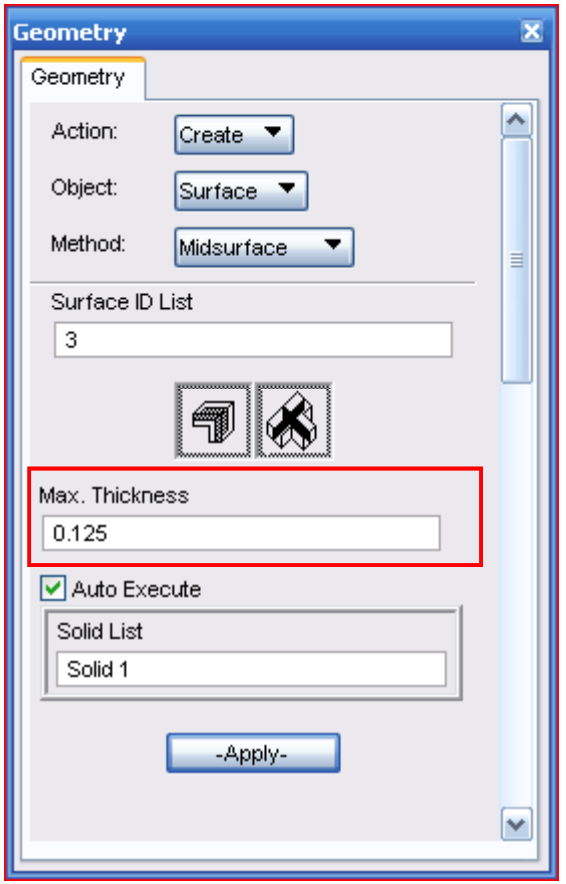
- **Inspection of the mid-surfaces shows that the gussets were created, but they have a thin “tail” that needs to be eliminated -- the additional piece off the triangular surface must be removed.**
- **The additional piece of the gusset comes from the mid-surface tool using the full height of the solid.**
- **The width of the tail (excess surface) will be half the thickness of the original solid wall.**
- **This will be discussed in greater detail in the mid-surface trimming section.**



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-SURFACE EXTRACTION, AUTOMATIC

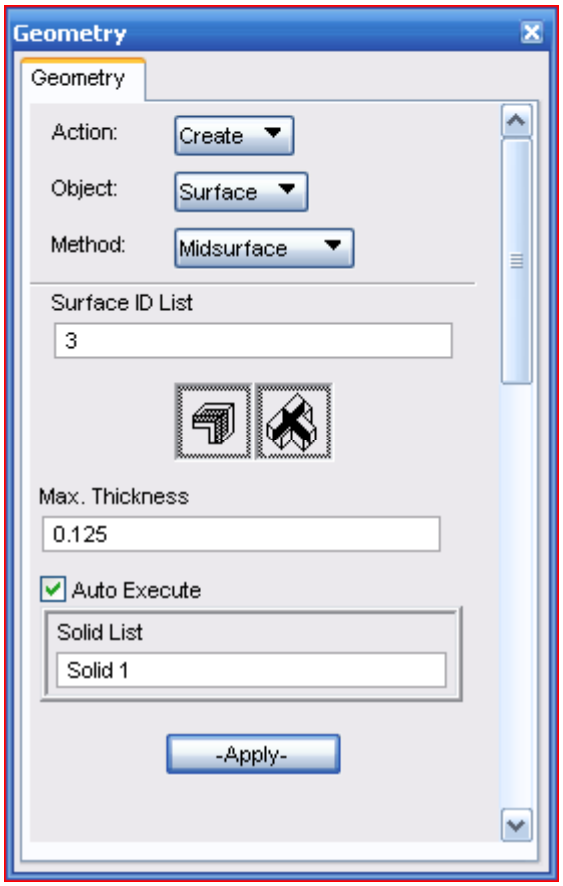
- **The model's wall thickness is 0.125. Based on this information, a slightly higher value of 0.126 can be used for Max. Thickness. A value of 0.125 would have produced the same results for this case.**
- **If the default of 1.0 for this particular model is used, none of the gussets will be mid-surfaced. The walls and cutouts would be created.**
- **If a value of 0.12 is used, then no mid-surfaces will be created.**



Copyright© 2012 MSC.Software Corporation

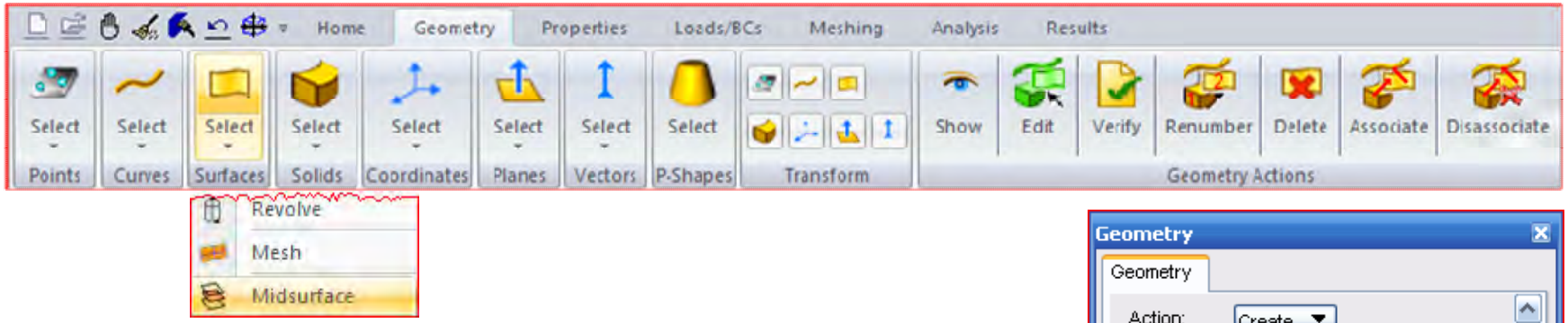
MID-SURFACE EXTRACTION, AUTOMATIC – FEATURES AND OPTIONS

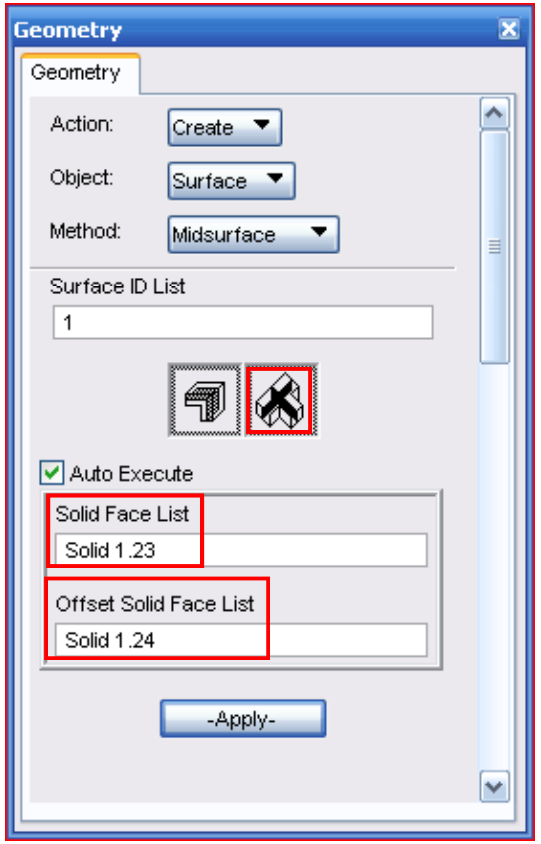
- **In the automatic mode, choosing the maximum thickness is important. This value is used to create specific mid-surfaces at/or under the specified maximum thickness.**
  - Too large a thickness can produce skewed surfaces. The mid-surfaces may not be in the wall of the solid selected or may not look as expected.
  - Too small a thickness may miss some contours or features created by varying wall thickness.
- **The mid-surfaces created by this option are not always congruent. Additional surface editing may be required to ensure a congruent mesh.**



Copyright© 2012 MSC.Software Corporation

MID-SURFACE EXTRACTION





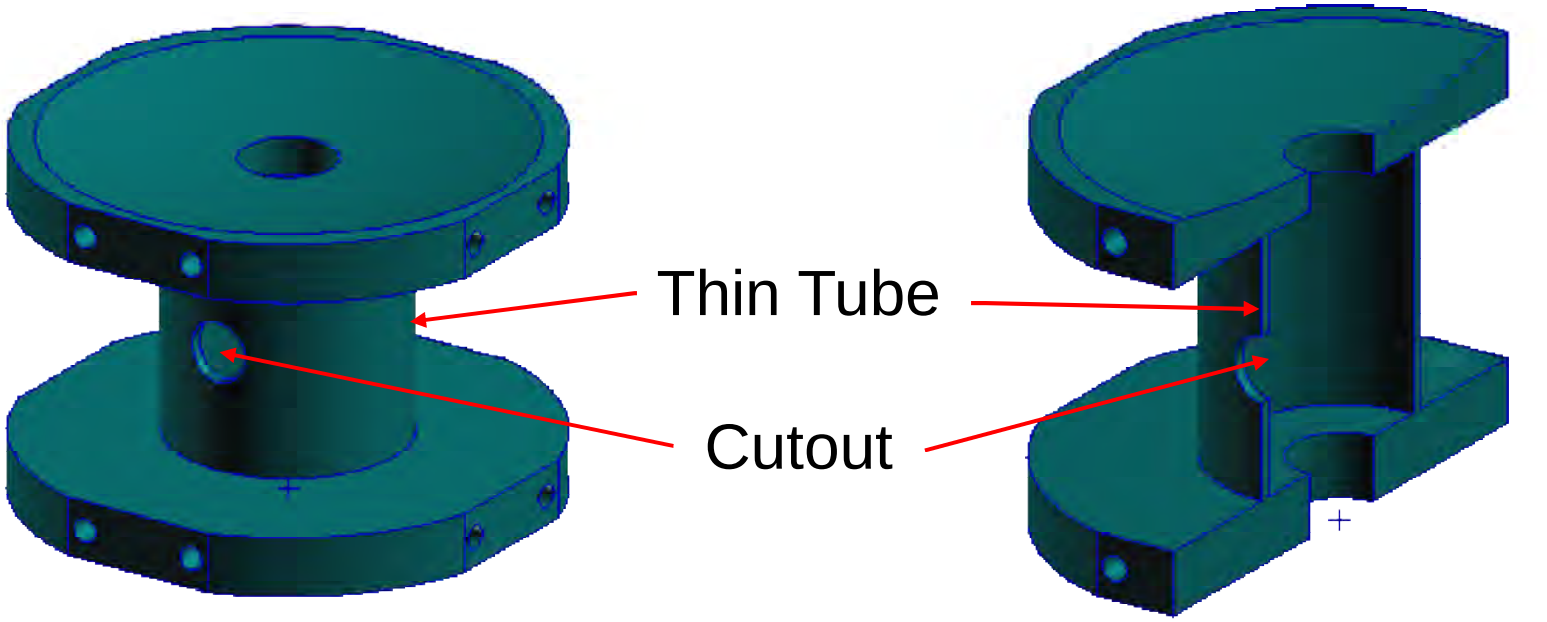
- **The Manual mode allows more control over the creation of the mid-surfaces.**

- **Manual mid-surface extraction works by selecting the two solid faces between which the mid-surface is desired.**

Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-SURFACE EXTRACTION, MANUAL

- **Below is a support lug. Most of the structure is thick except a thin walled tube connecting the two solids.**
  - The cutout in the thin tube section is an added complexity.

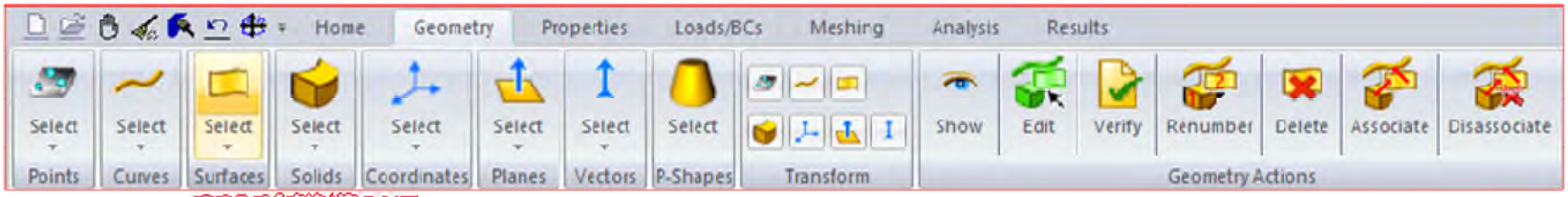


Support Lug

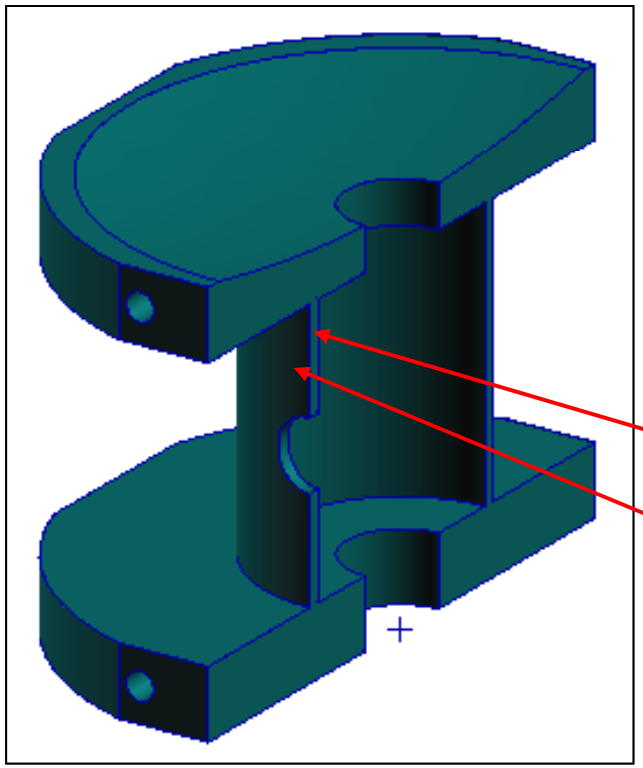
Support Lug cross-section

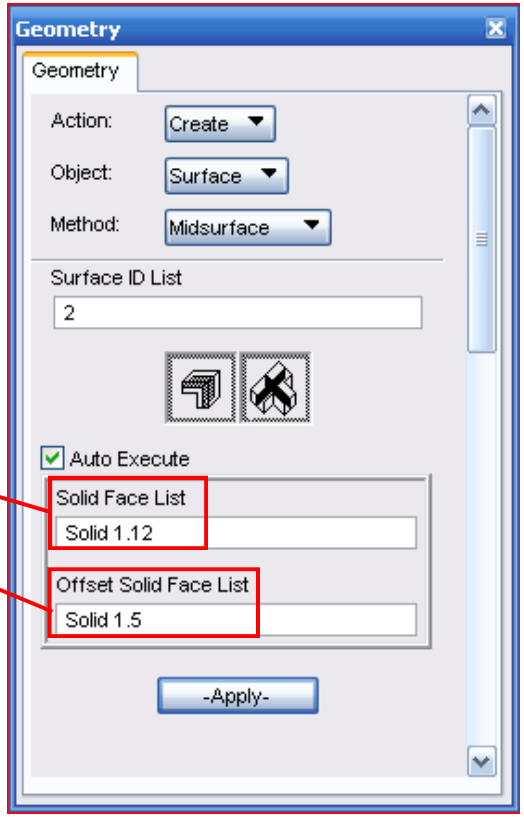
Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-SURFACE EXTRACTION



Simply select the inner and outer walls of the tube.

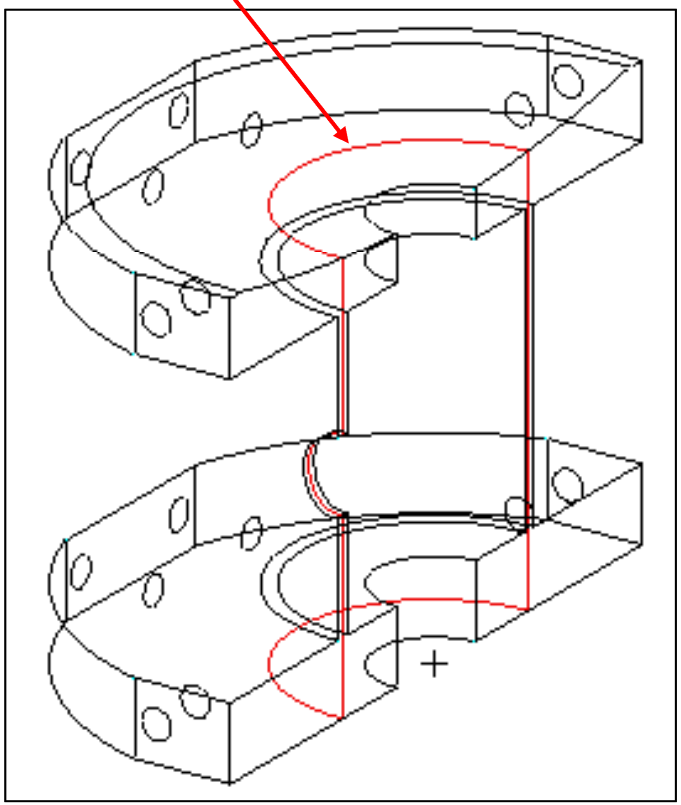




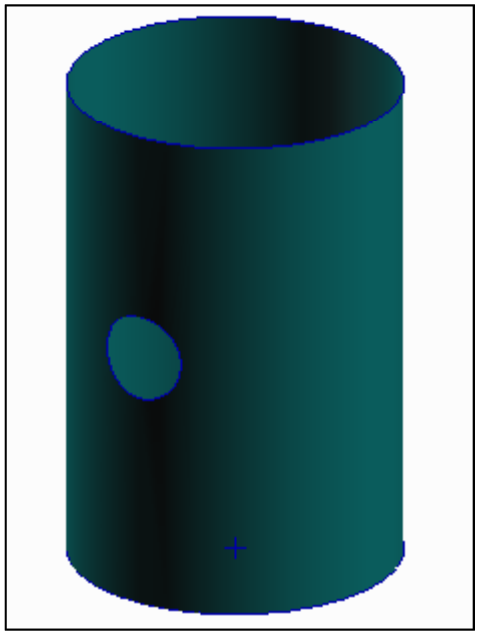
Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-SURFACE EXTRACTION, MANUAL

Half solid with magenta mid-surface

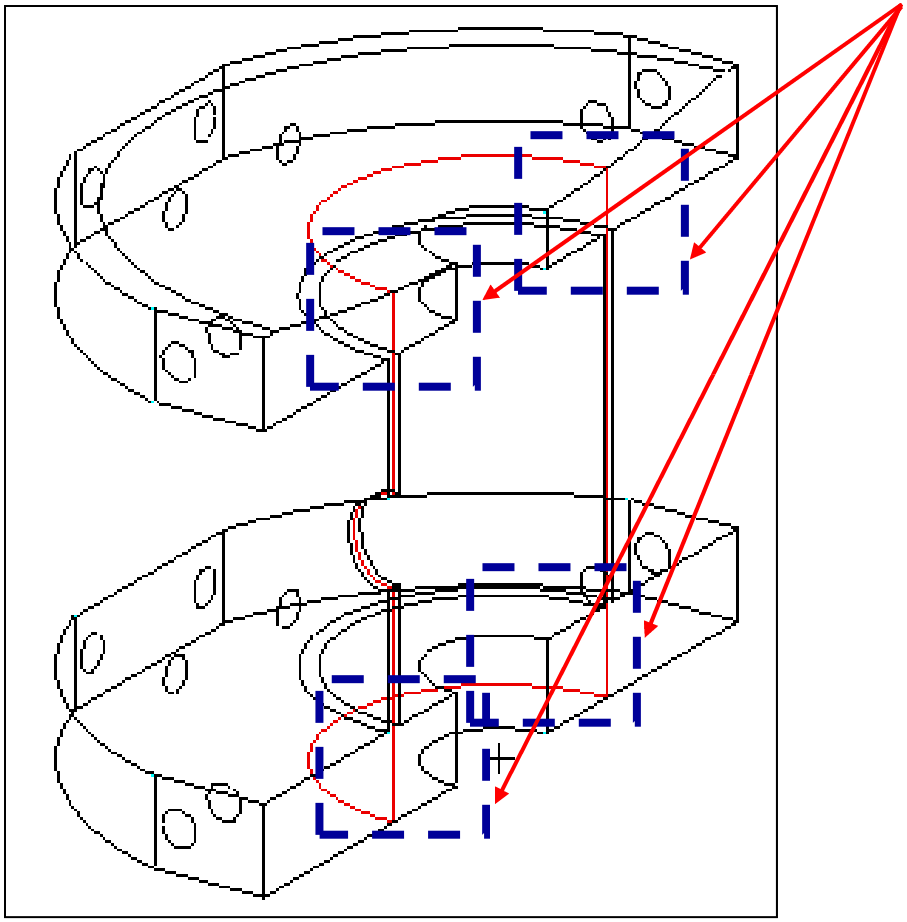


Smooth shaded mid-surface without solid



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-SURFACE EXTRACTION, MANUAL

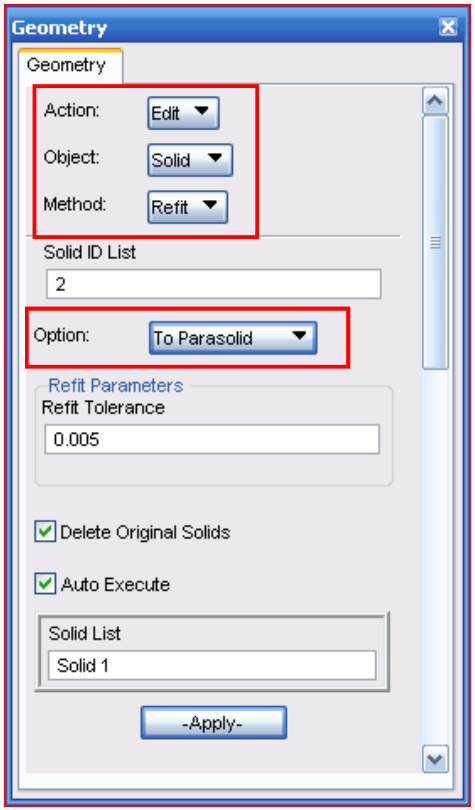


- **Note that the mid-surfaces go to the extent of the solid.**
- **Trimming of the mid-surface by surfaces may be required if this manual mode is used.**
  - Use: Edit/Surface/Break or Edit/Surface/Extend to modify the mid-surfaces.

Copyright© 2012 MSC.Software Corporation

MID-SURFACE EXTRACTION, PARASOLID GEOMETRY NEEDED

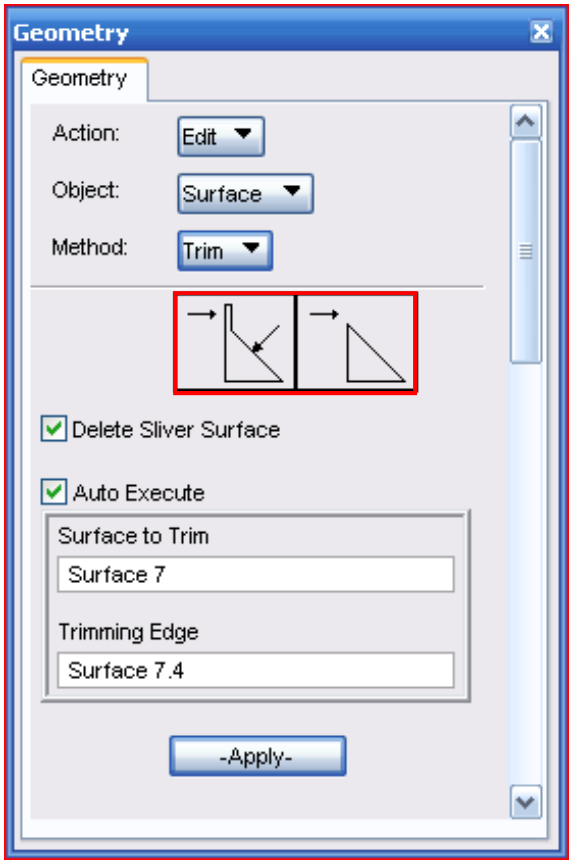
- **During automatic mid-surface extraction, Patran will automatically convert (refit) non-parasolid geometry to parasolid geometry before creating the mid-surfaces.**
- **During manual mid-surface extraction, Patran will not automatically refit non-parasolid geometry to parasolid geometry. The user needs to convert non-parasolid geometry to parasolid geometry before performing this operation.**



Copyright© 2012 MSC.Software Corporation

MID-SURFACE TRIMMING

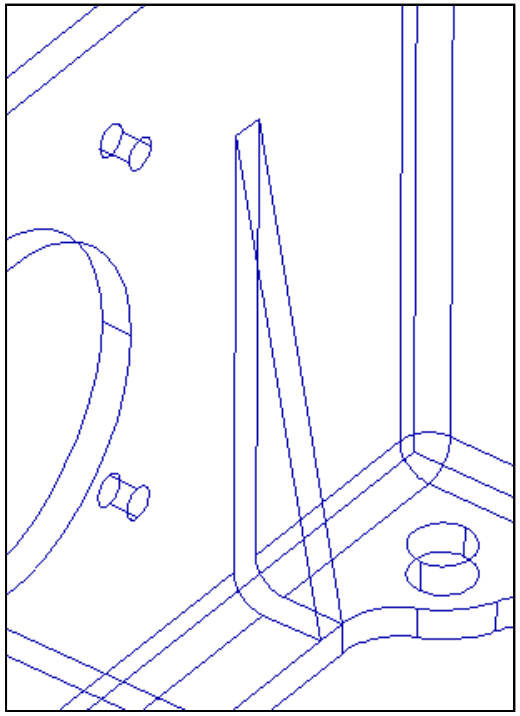
- **Mid-surface trimming allows the modification of common mid-surface protrusions.**
  - The Trimming Edge must be part of the surface that is being trimmed.



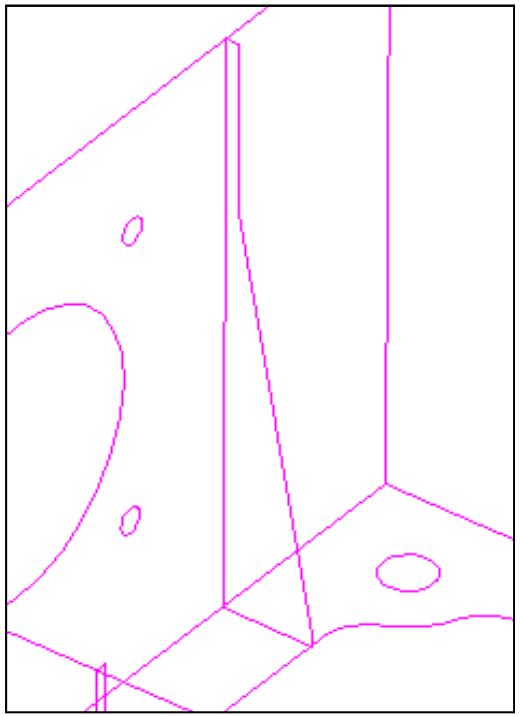
Copyright© 2012 MSC.Software Corporation

MID-SURFACE TRIMMING

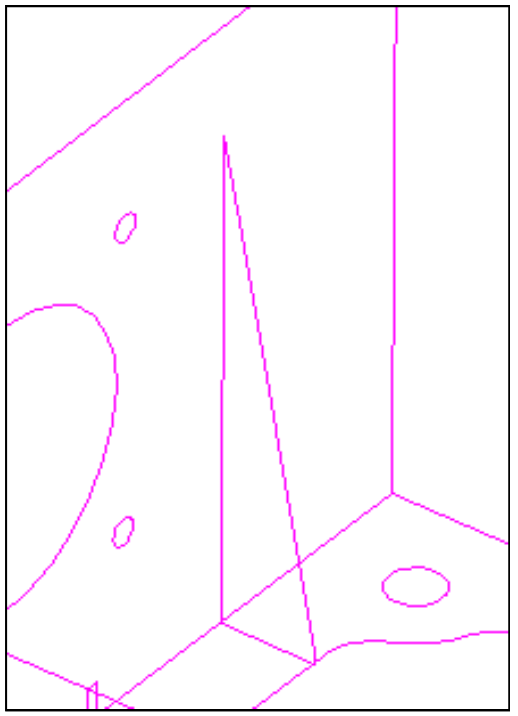
- **Trimming the mid-surface created from a gusset is a common application.**



Imported model with solid gusset



Automatic mid-surface from the solid gusset

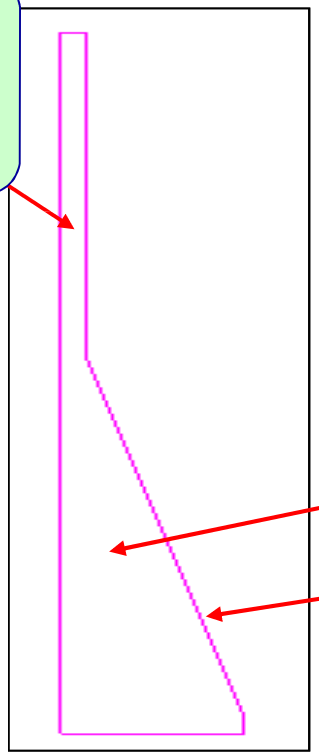


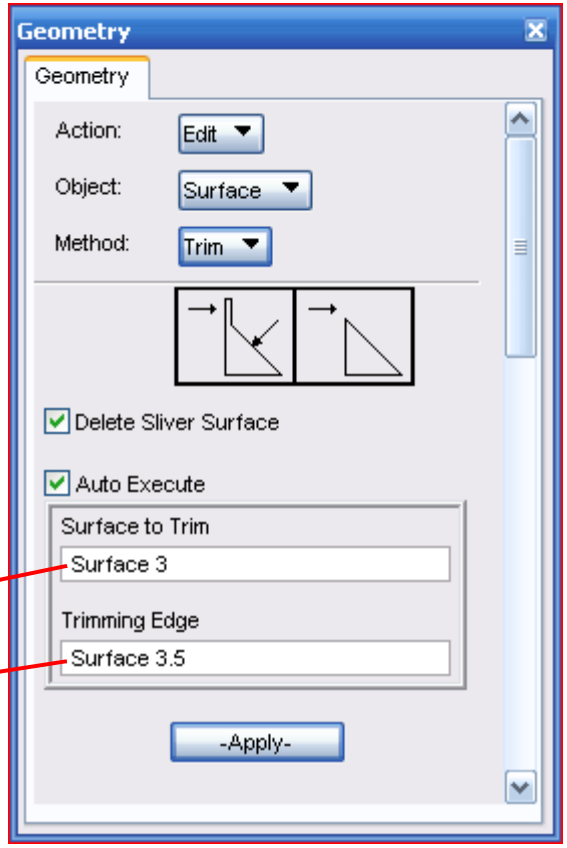
Mid-surface after trimming

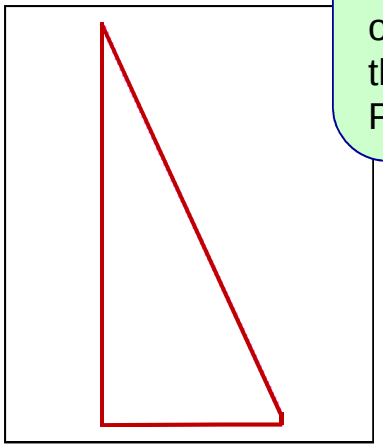
Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-SURFACE TRIMMING

Remove this sliver from the gusset surface.







What will remain is a surface representation of the gusset that can be Paver meshed.

Copyright© 2012 MSC.Software Corporation

EXERCISES

- Perform Workshop 5 “Mid-surface Creation, Simple Bracket” in your exercise workbook.
- Perform Workshop 6 “Mid-surface Extraction, Wing Section” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

MID-PLANE MESHER

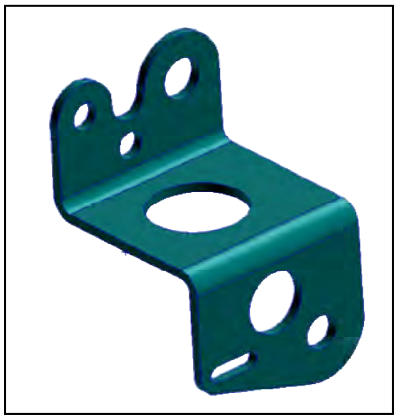
- **Single-step creation of a midplane mesh in a solid part**
- **Intended for bent sheet material**
- **Does not work for “tee” type junctions**

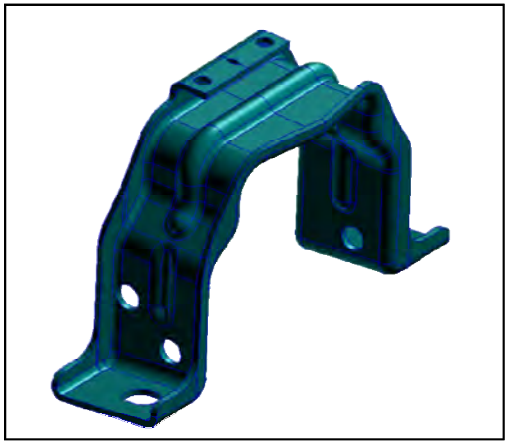
Start Point 3D	Nothing		Mesh		Geometry		
Method	N/A	N/A	N/A	N/A	Mid Plane Mesher	Mid Surface Extraction	Geometry breakdown and rebuild methods
End Point 2D	Mesh	Geometry	Mesh	Geometry	Mesh	Geometry	

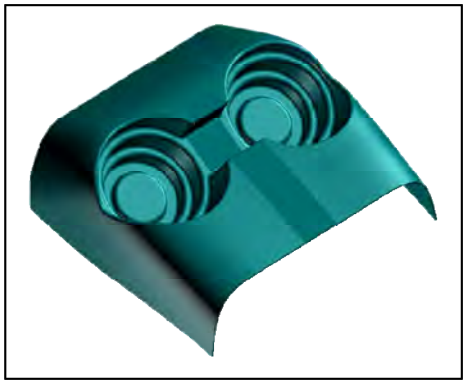
Copyright© 2012 MSC.Software Corporation

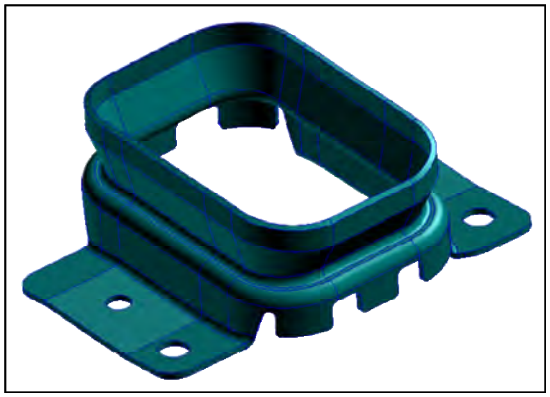
MID-PLANE MESHER

- **Examples of parts that can be midplane meshed**









Copyright© 2012 MSC.Software Corporation

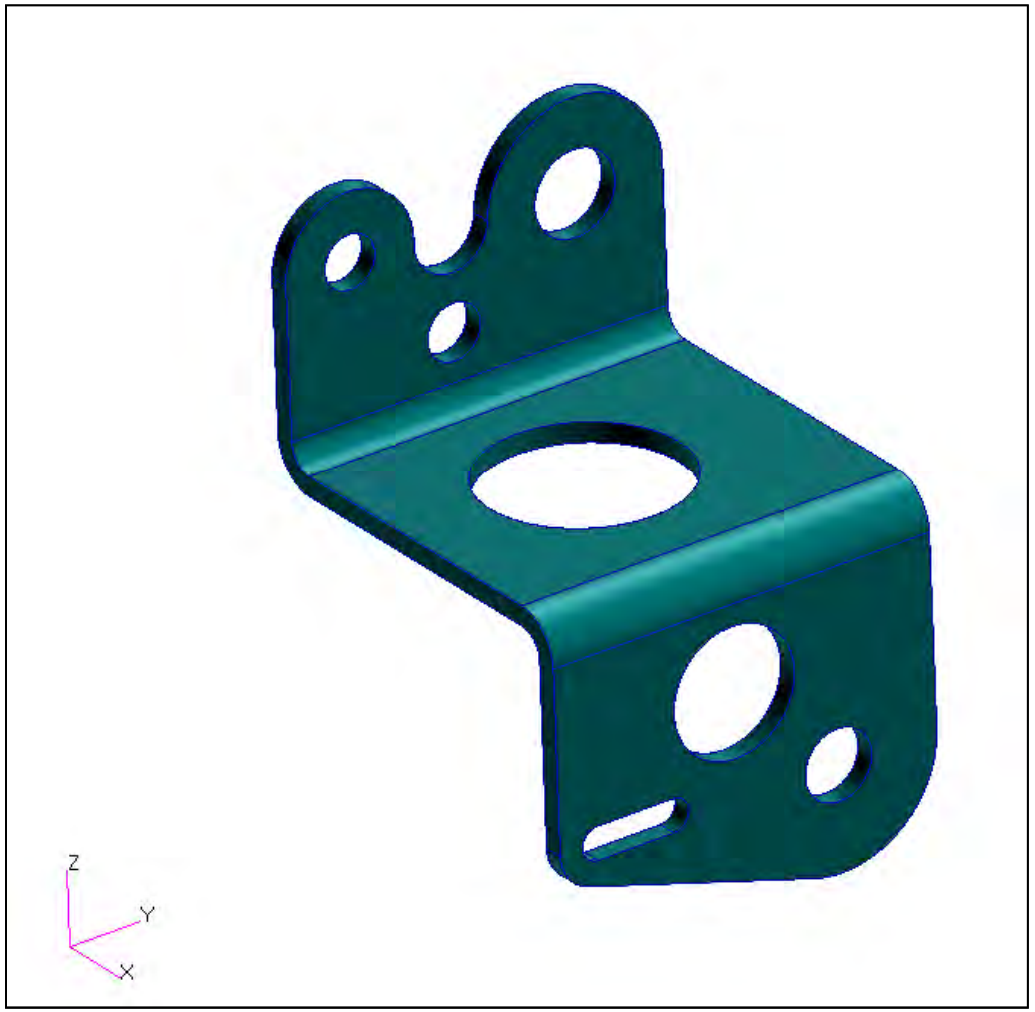
MID-PLANE MESHER

- **Comparison between midplane meshing and mid-surface extraction**
  - Midplane meshing produces a mesh that is not associated with any geometry. This means that element properties, loads, and boundary conditions must be applied directly to elements, not geometry.
  - Midplane meshing does not work for parts with tee junctions.
  - The Midplane mesher is more robust in creating a mesh than mid-surface extraction with subsequent 2D meshing
    - It will work for some solid geometry for which the mid-surface extraction method will fail

Copyright© 2012 MSC.Software Corporation

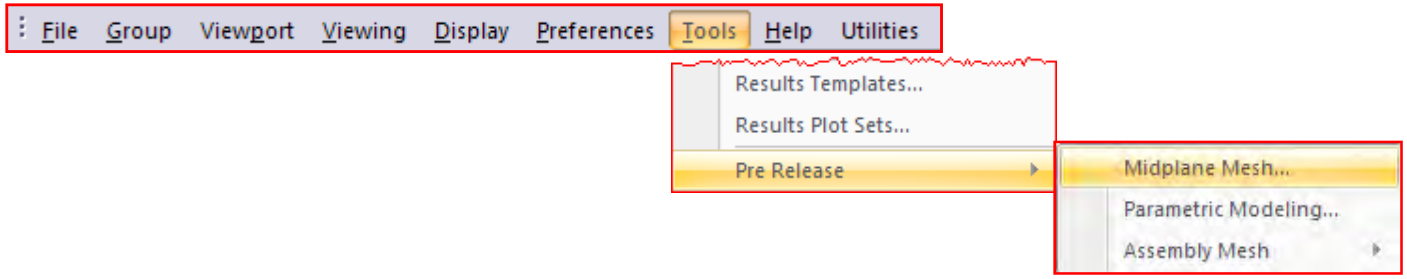
EXAMPLE: MID-PLANE MESHER, SIMPLE BRACKET

A simple sheet metal bracket is illustrated to the right. Use the midplane mesher to mesh the part.



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-PLANE MESHER, SIMPLE BRACKET



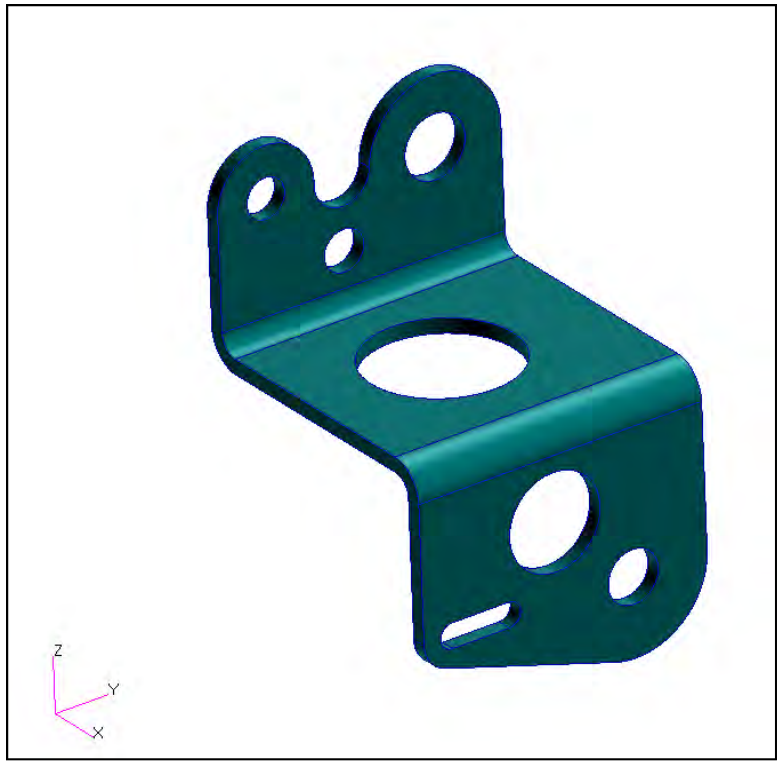
Tools / Pre Release / **Midplane Mesh…**

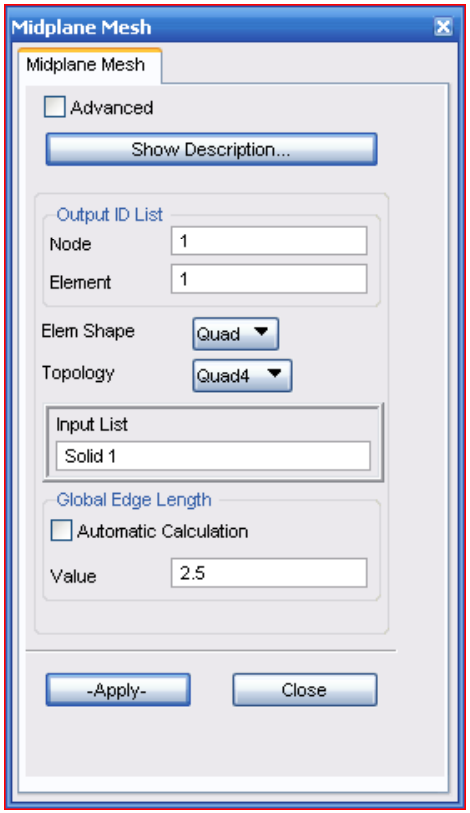
Uncheck the **Advanced** toggle.

Select the solid.

Enter Global Edge Length (*GEL*).

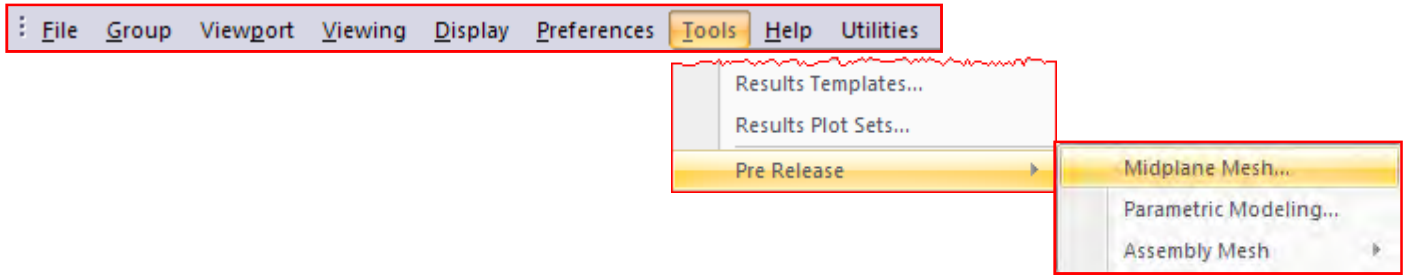
Click **Apply**.



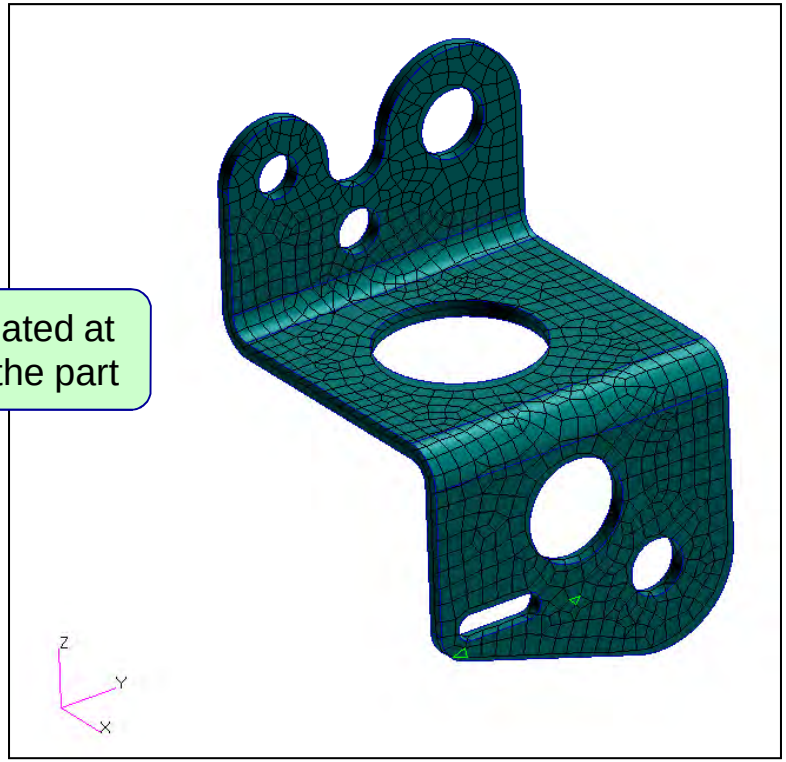


Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-PLANE MESHER, SIMPLE BRACKET

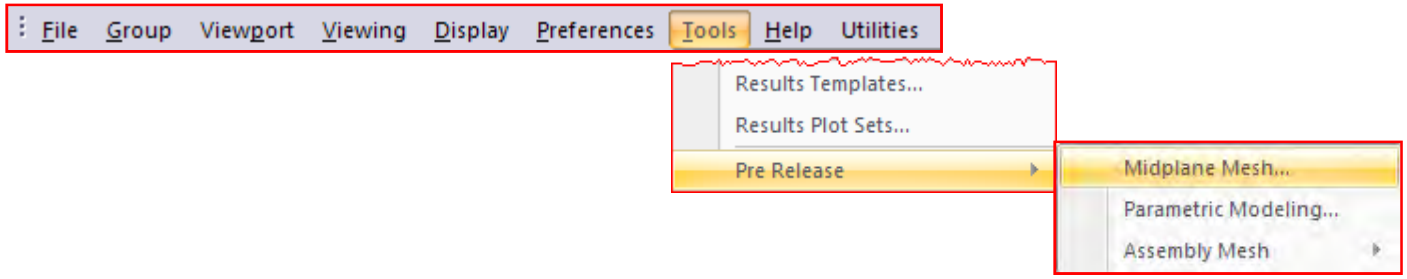


2D mesh created at midplane of the part

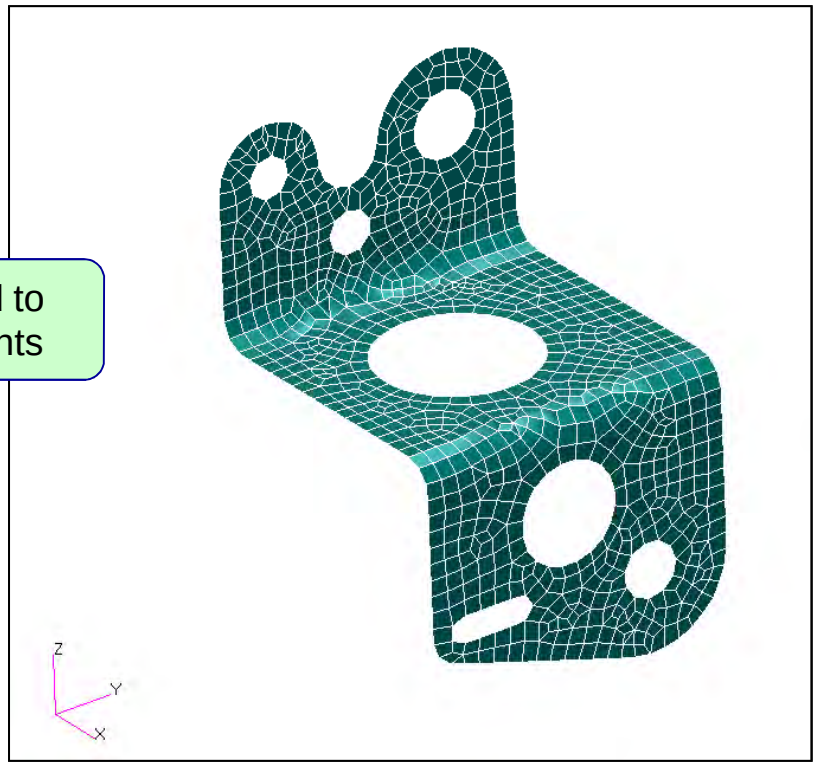


Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-PLANE MESHER, SIMPLE BRACKET



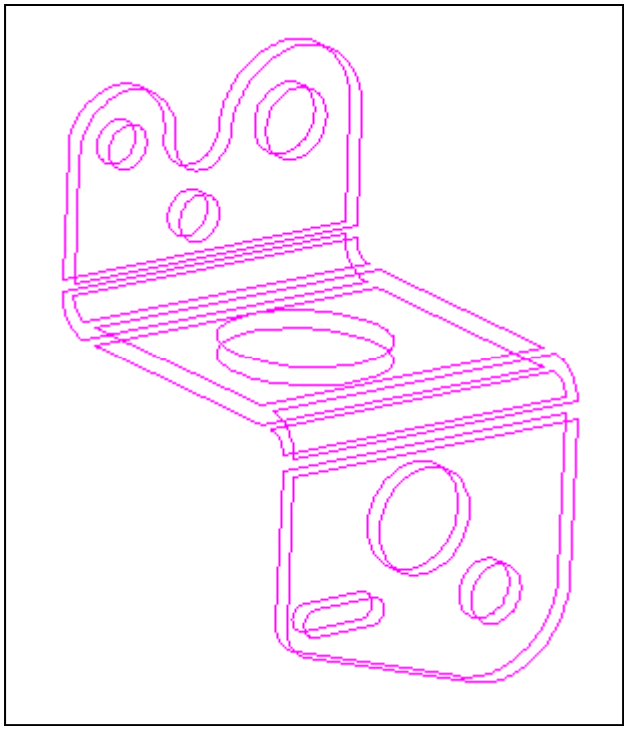
Geometry erased to show only elements



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-PLANE MESHER, SIMPLE BRACKET

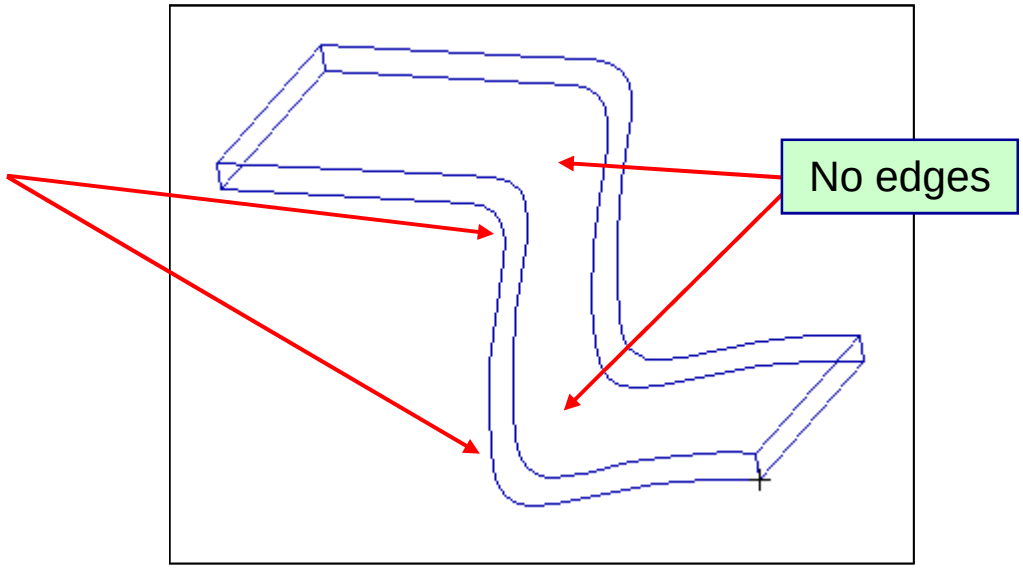
- **If the meshing is accomplished by using the mid-surface extraction, and then 2D meshing, the process would be successful.**
- **The mid-surfaces that would be created between the opposing front and back faces (largest faces) of the solid would be planar or have simple single curvature, making it very numerically tractable to create them.**
- **For this simple geometry, it may be best to use this approach.**



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-PLANE MESHER, SIMPLE BRACKET

- **However, had the solid been like this simple one, it would not have been possible to extract a mid-surface.**
  - The following messages will appear:
    $# Warning reported from application Geometry
    $# A mid-surface cannot be created from solid with the current parameters
    $# No Geometry Created
  - Midsurface would have had complex curvature



Copyright© 2012 MSC.Software Corporation

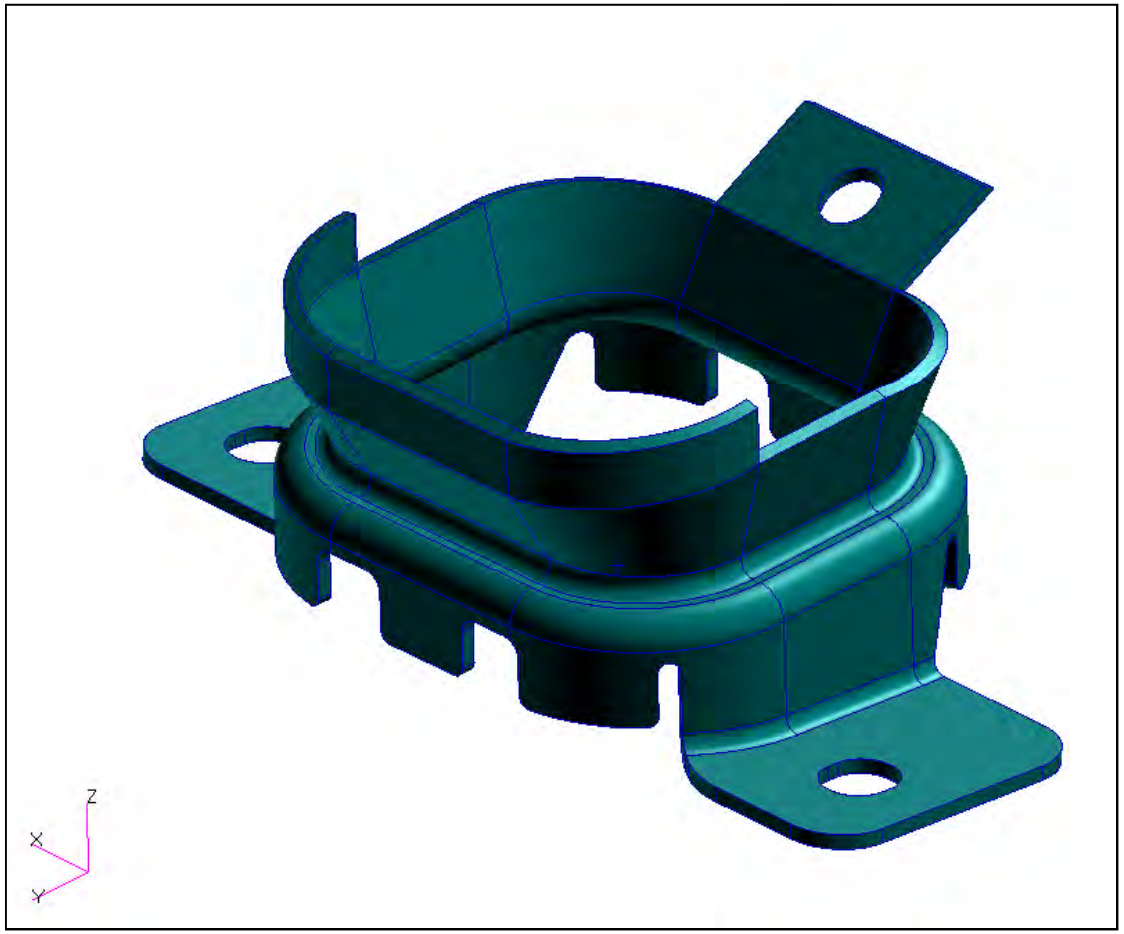
EXERCISES

- Perform Workshop 7 “Mid-plane Mesher” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

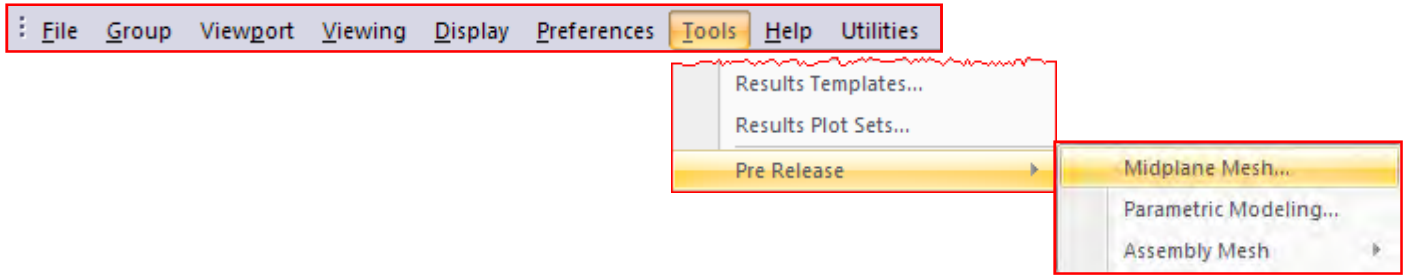
EXAMPLE: MID-PLANE MESHER, COMPLEX HOUSING

This part is even more complex than the previous example. It is a housing produced by drawing and stamping. Again, the midplane mesher will be used to mesh the part.



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-PLANE MESHER, COMPLEX HOUSING

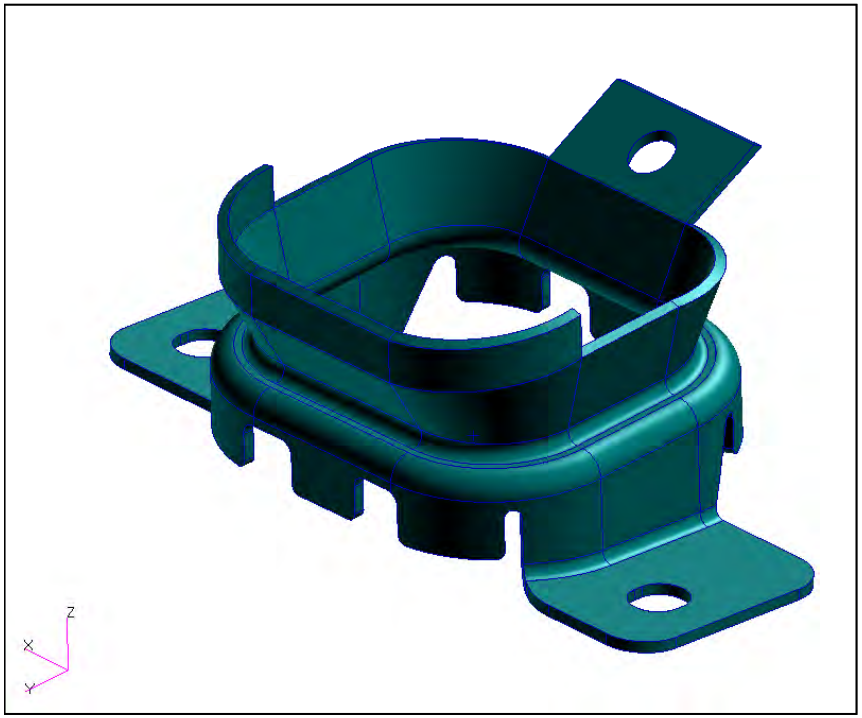


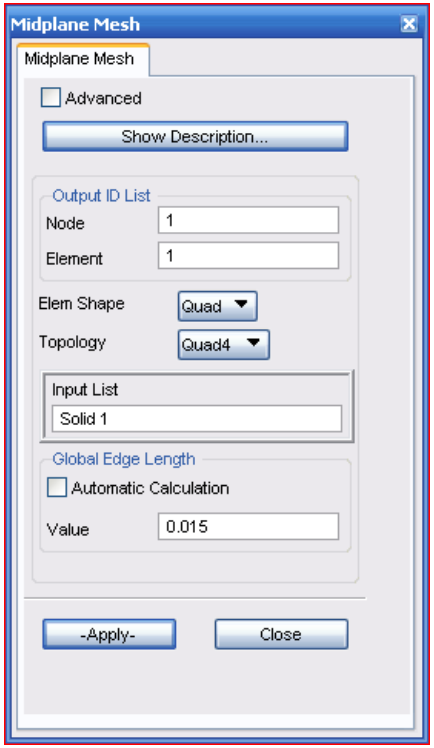
Tools / Pre Release / **Midplane Mesh…**

Select the solid.

Enter the global edge length (*GEL*).

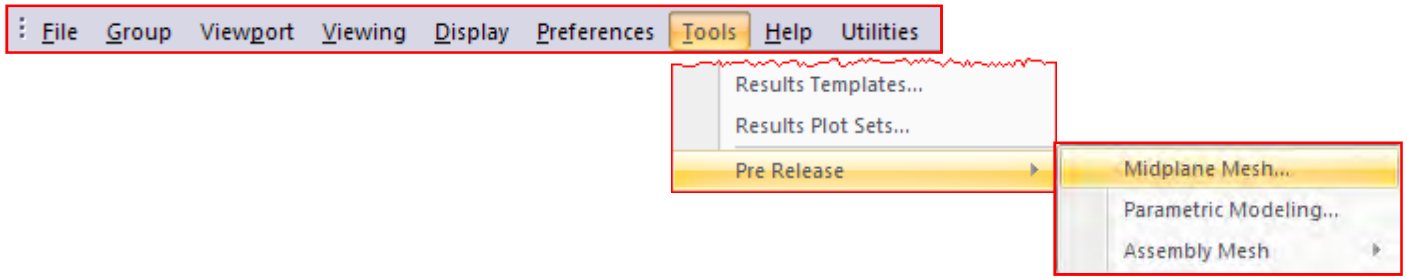
Click **Apply**.

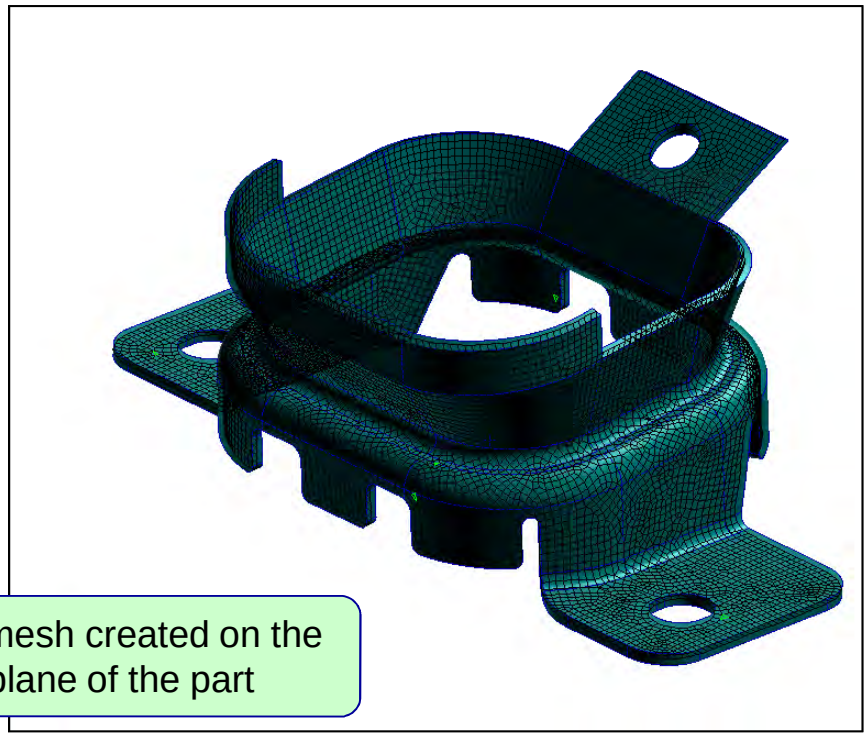




Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-PLANE MESHER, COMPLEX HOUSING

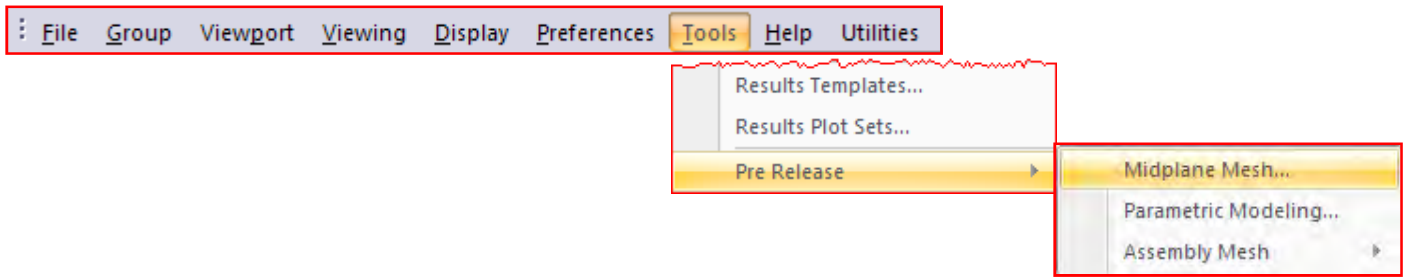


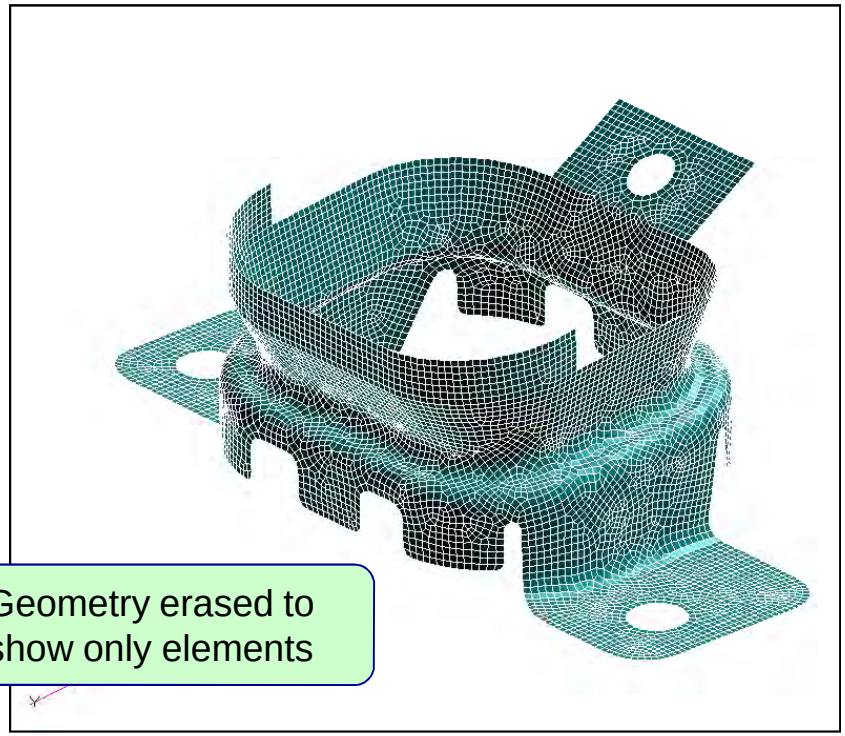


2D mesh created on the midplane of the part

Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-PLANE MESHER, COMPLEX HOUSING

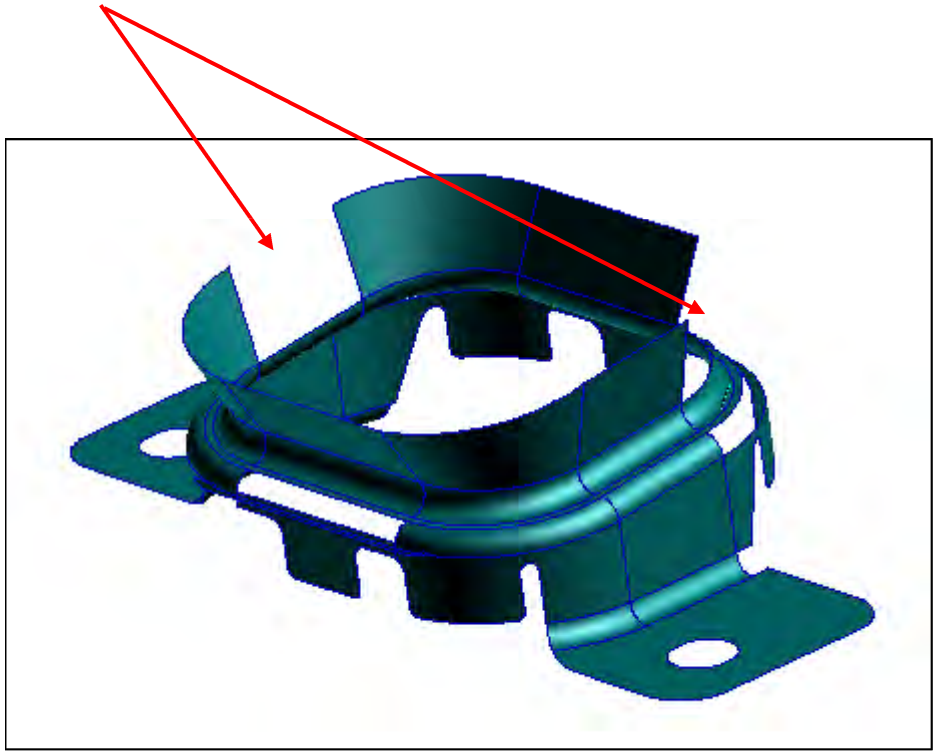




Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-PLANE MESHER, COMPLEX HOUSING

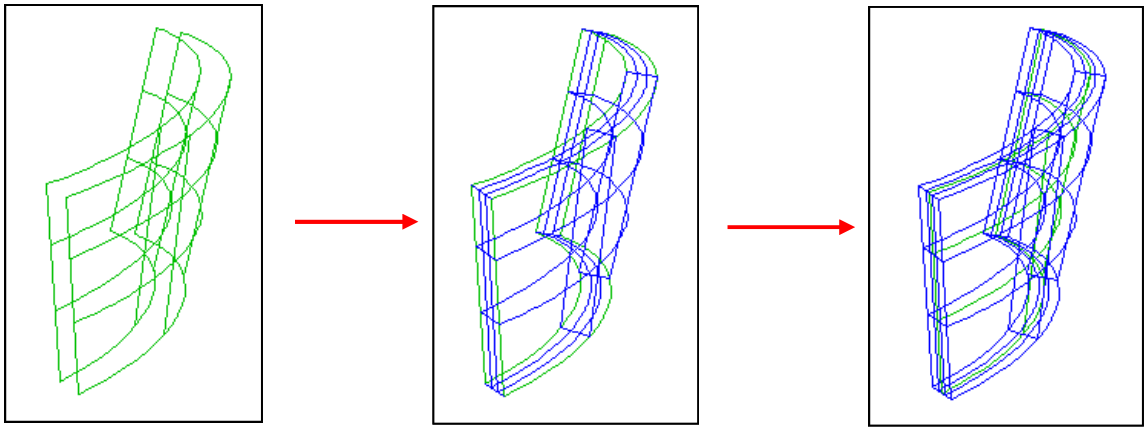
- **Had the mid-surface extraction approach been used, it would not have been successful.**
  - There are missing surfaces



Copyright© 2012 MSC.Software Corporation

EXAMPLE: MID-PLANE MESHER, COMPLEX HOUSING

- **The missing surfaces can be created manually**
  - For this solid, using the manual approach under Create / Surface / Midsurface fails
    - $# Warning reported from application Geometry
    - $# A mid-surface could not be created between two surfaces.
    - $# No Geometry Created.
  - It is necessary to find an alternative approach, such as create a parametric solid using two opposing faces, then create a parametric surface at parametric 0.5 through the solid.



Copyright© 2012 MSC.Software Corporation

3D START POINT, 2D IDEALIZATION

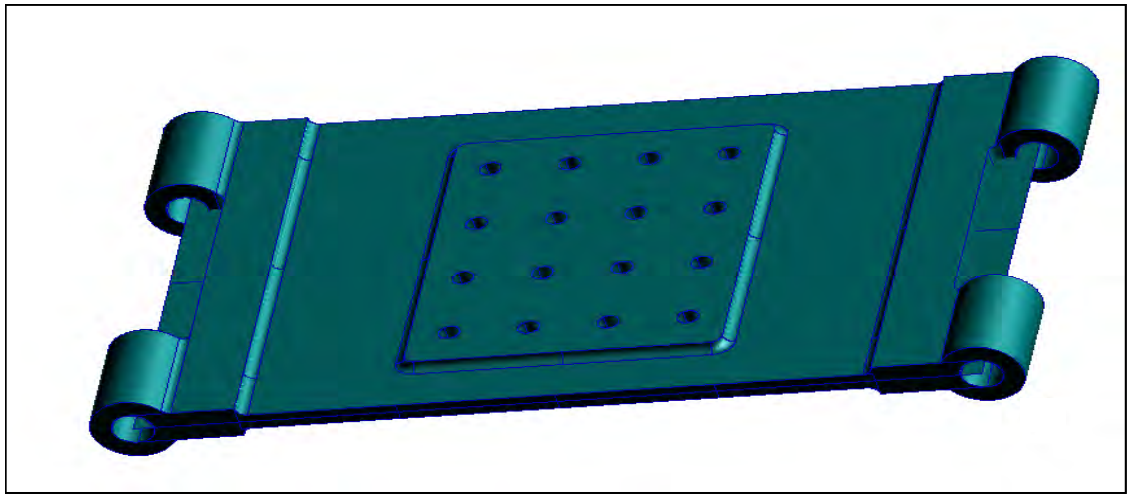
- **3D geometry break-down, then rebuild to create 2D geometry.**

Start Point 3D	Nothing		Mesh		Geometry		
Method	N/A	N/A	N/A	N/A	Mid Plane Mesher	Mid Surface Extraction	Geometry breakdown and rebuild methods
End Point 2D	Mesh	Geometry	Mesh	Geometry	Mesh	Geometry	

Copyright© 2012 MSC.Software Corporation

3D START POINT, 2D IDEALIZATION

- **Consider the CAD model shown below**
  - It represents a carrier (support) tray on which measuring equipment is bolted
  - The tray is connected to the backing structure via four brackets
  - It is assumed that the structure can be adequately represented by plate elements, with the brackets modelled as rigid elements



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

Disassemble the solids:

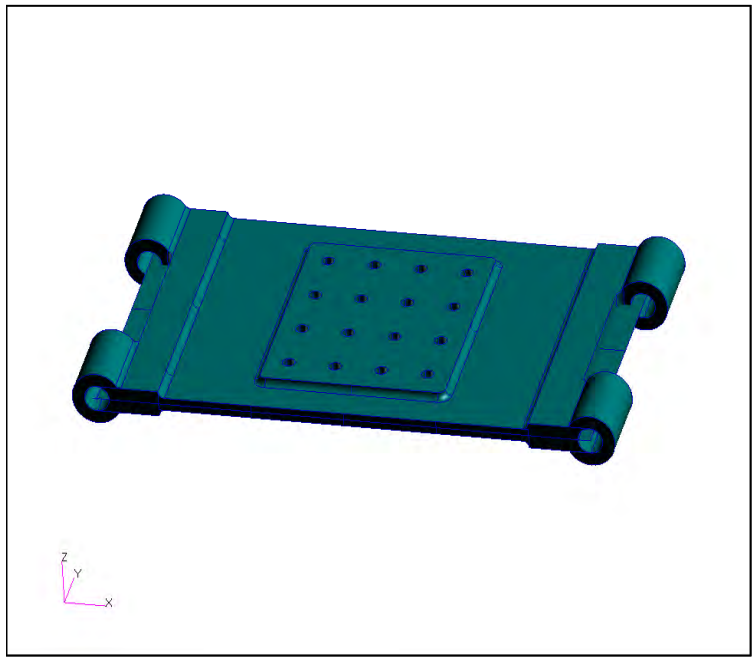
Geometry : **Edit** > **Solid** > **Disassemble**

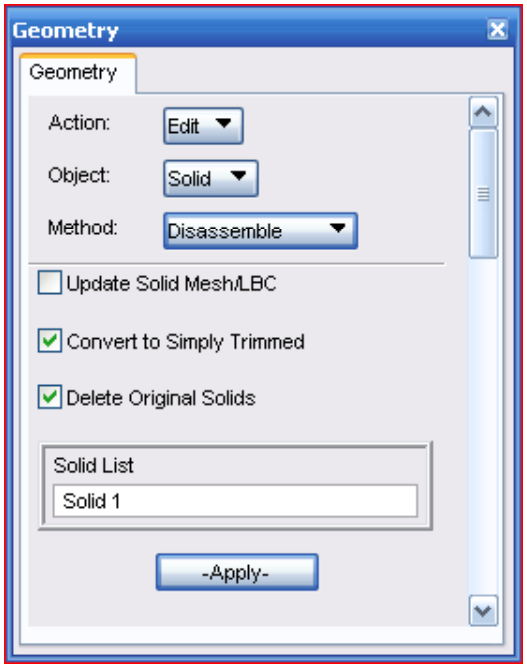
Select the toggle for **Convert to Simply Trimmed** (Green surface, where possible). The other option is to leave as trimmed.

Select the **Delete Original Solids** toggle.

Under *Solid List*, select the solid.

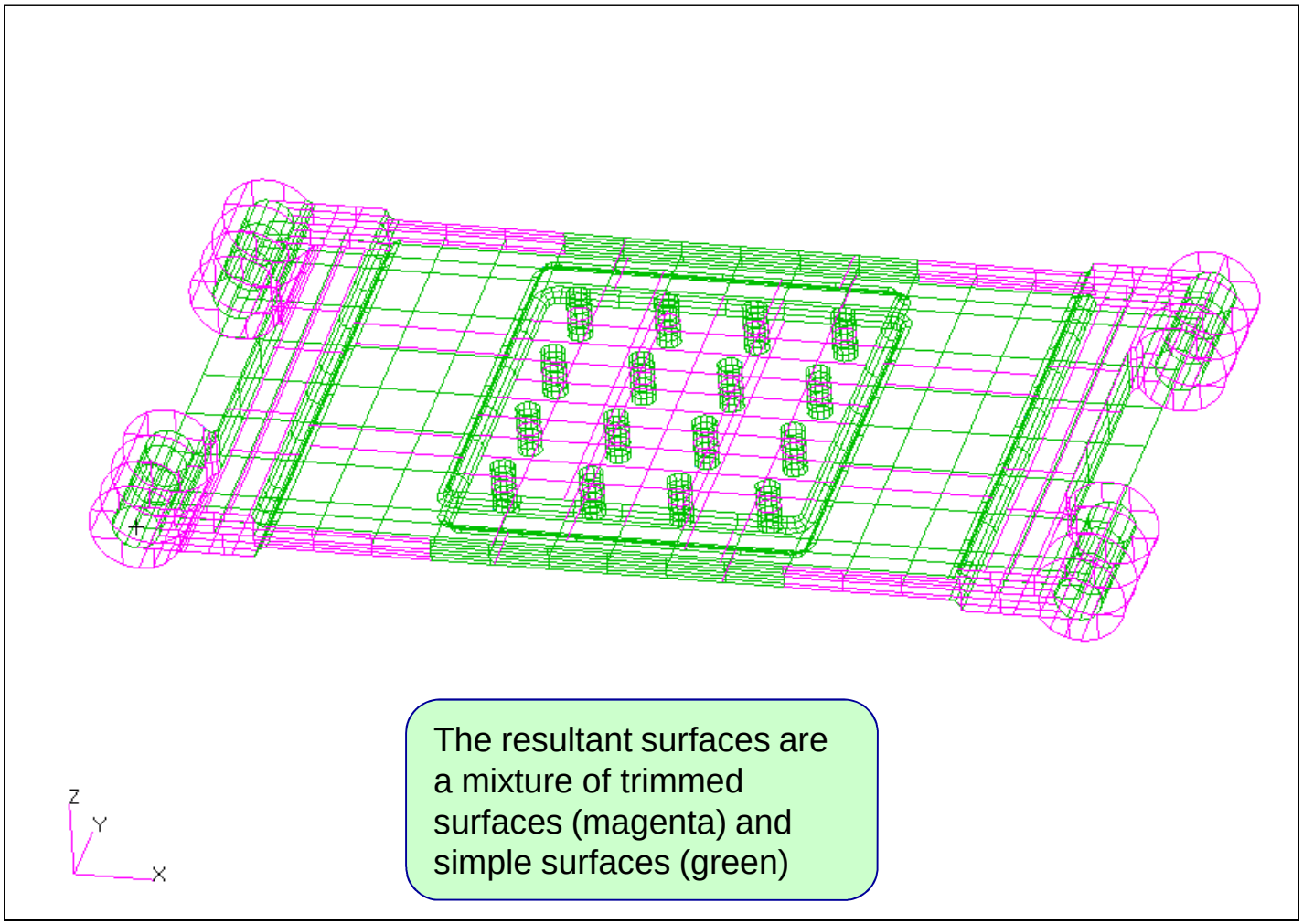
Click **Apply**.





Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

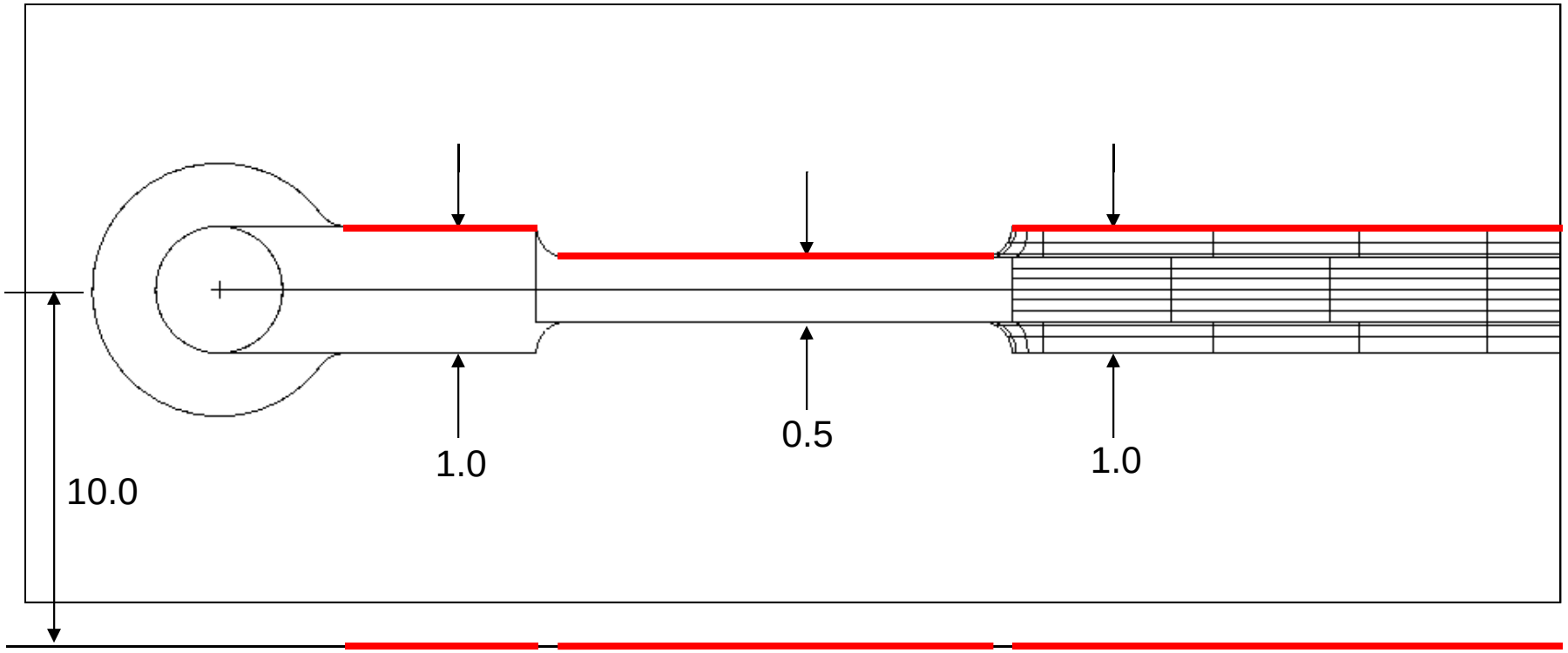


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

The strategy to be used is to copy select surfaces to a common plane. (This is just so the new surfaces will be out of the way of the original model.) For convenience, this plane is positioned 10.0 inches below the center plane of the solid.

The blend radii will be ignored in the analysis.



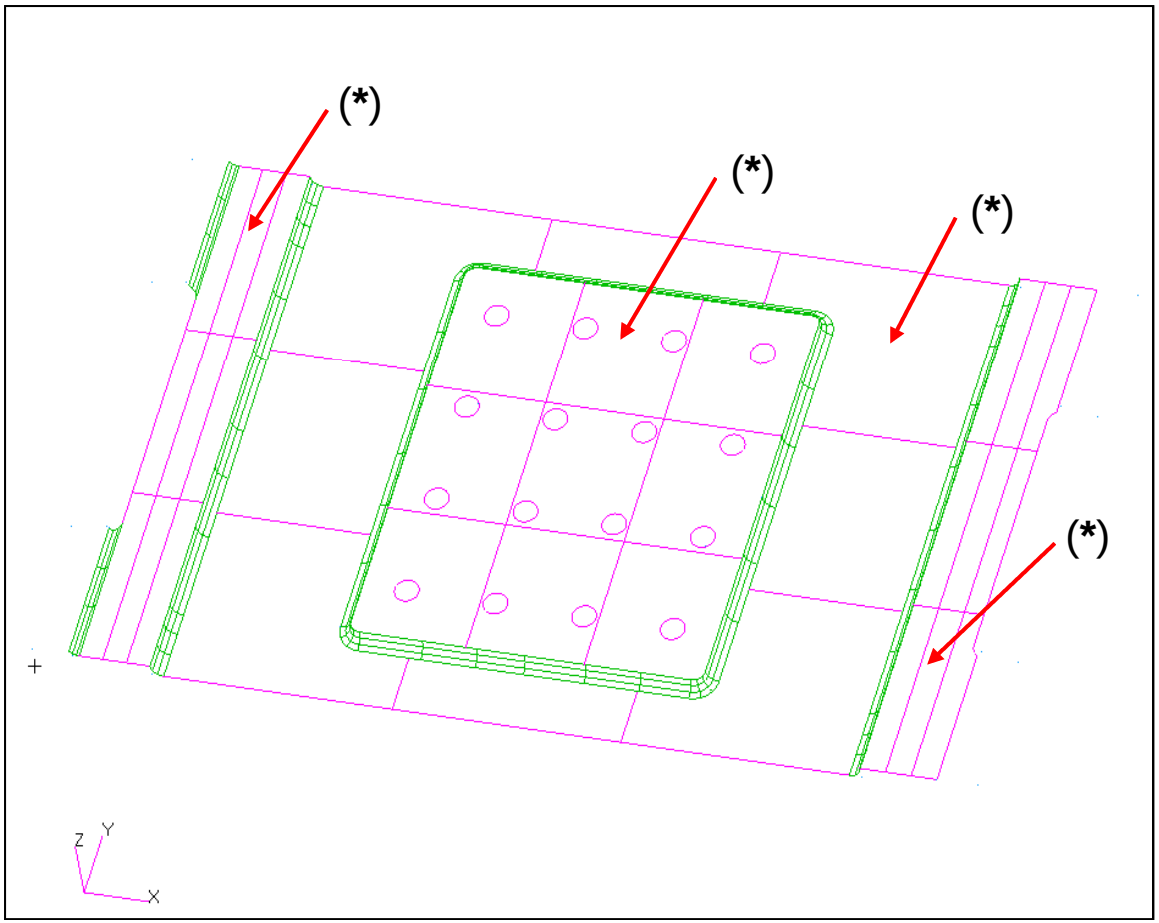
Copyright© 2012 MSC.Software Corporation

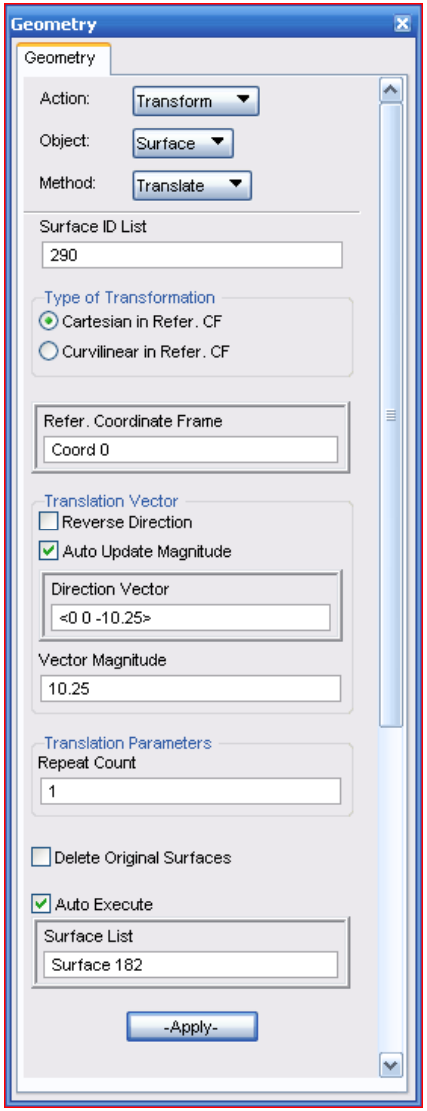
CASE STUDY: CARRIER 3D -> 2D

The top surfaces (*) are displayed together for easy selection.

The surfaces are copied to a common plane, using the appropriate translation vector.

The fillet surfaces are ignored.

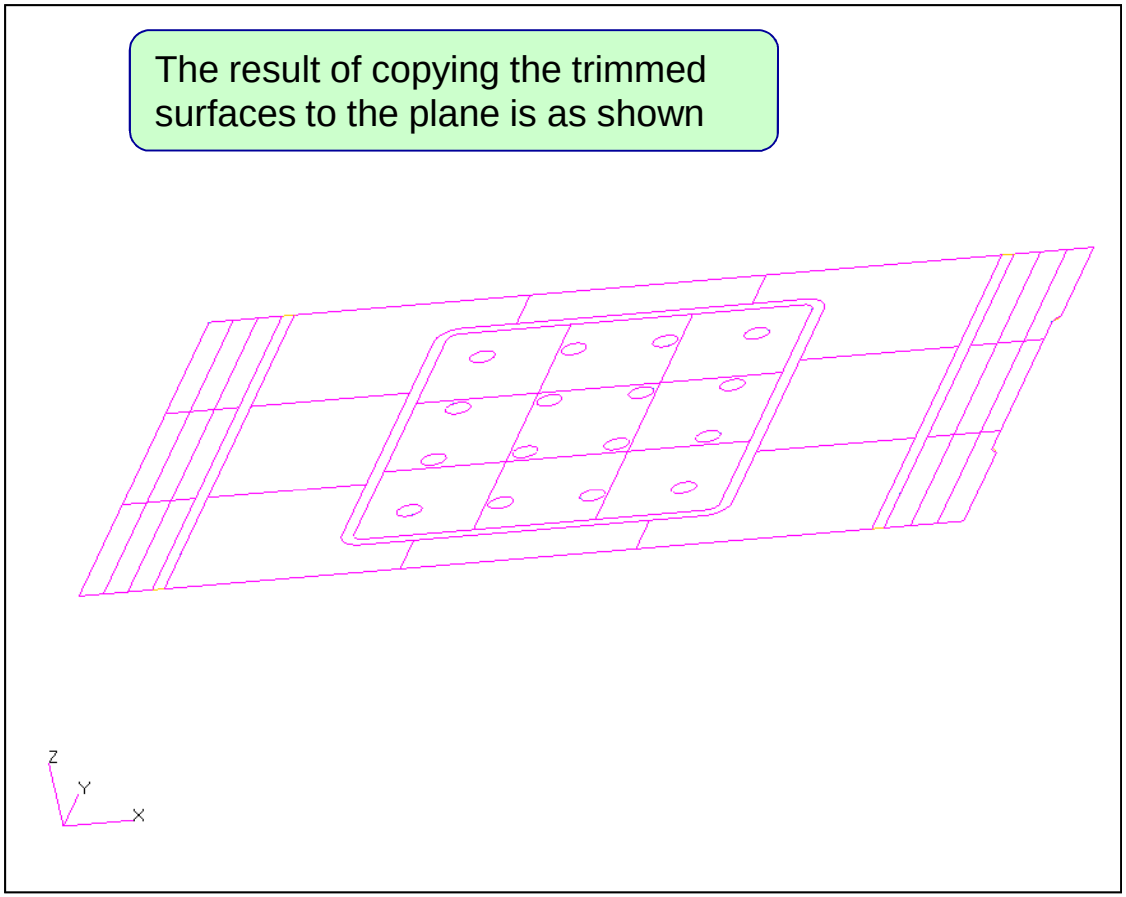


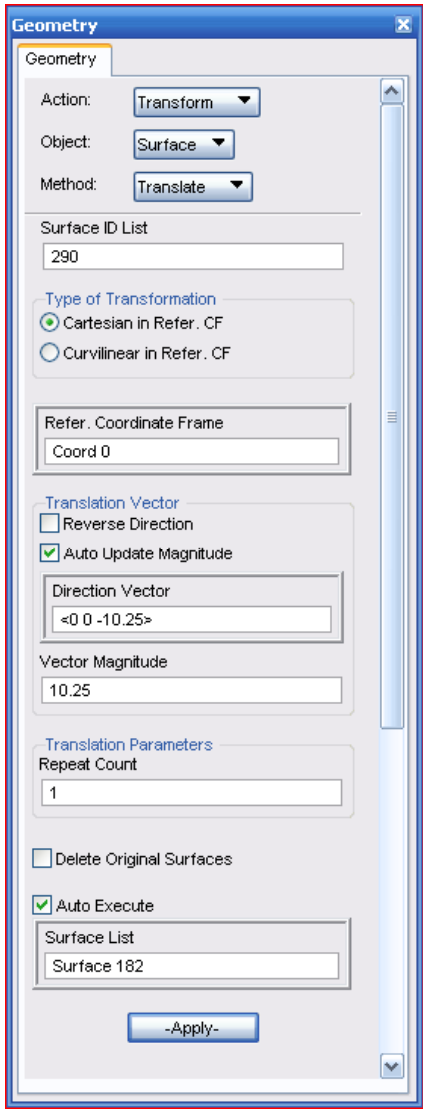


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

The result of copying the trimmed surfaces to the plane is as shown



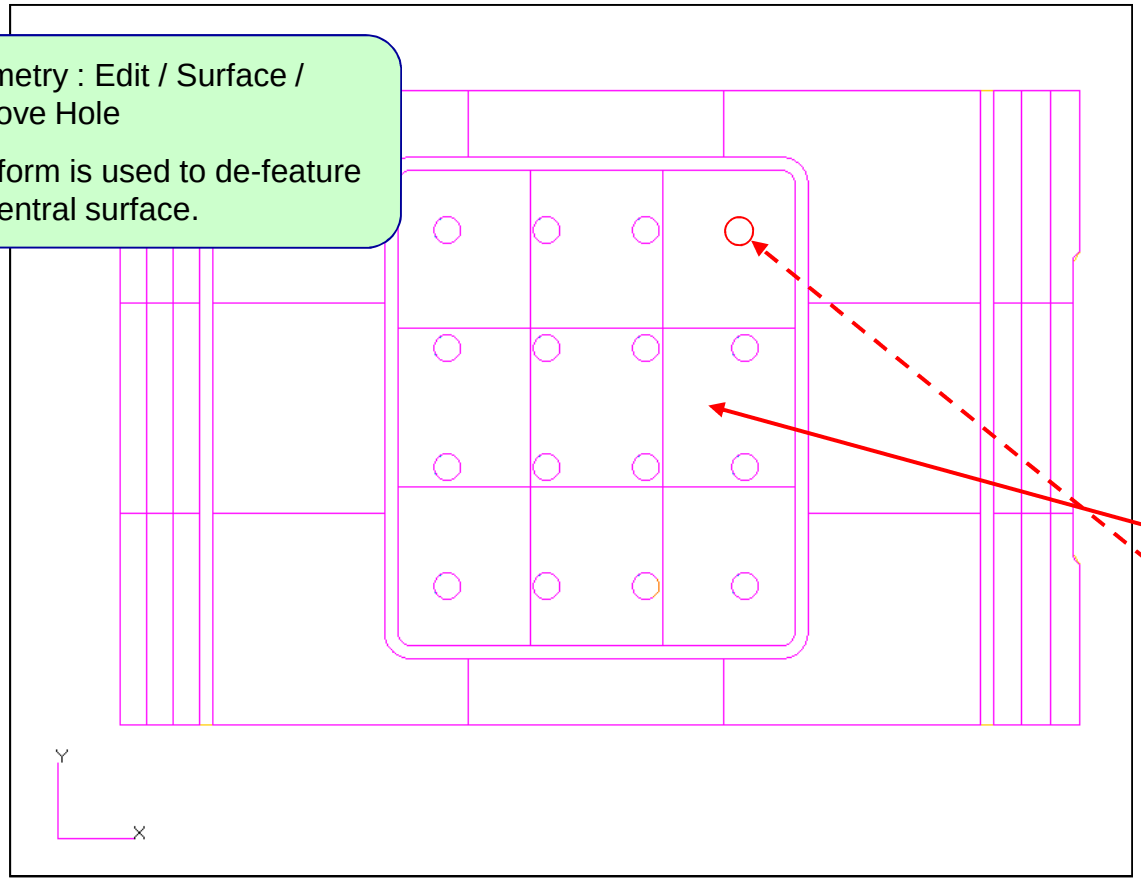


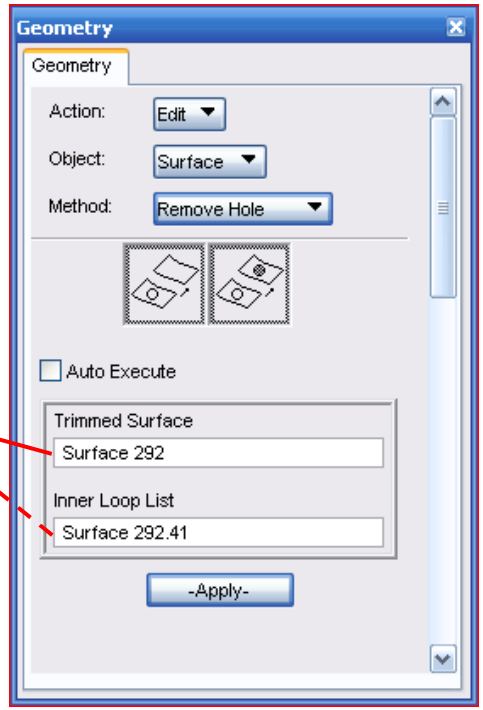
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

Geometry : Edit / Surface / Remove Hole

This form is used to de-feature the central surface.

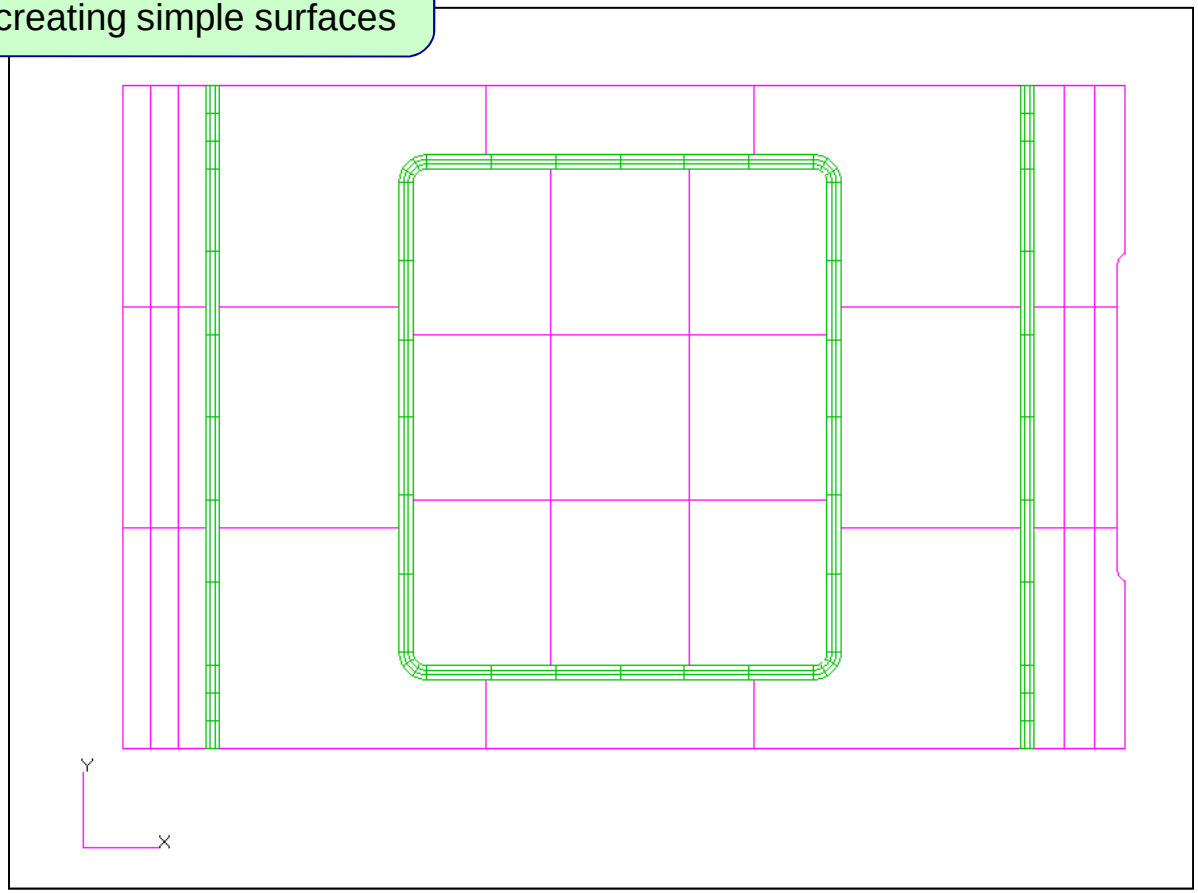


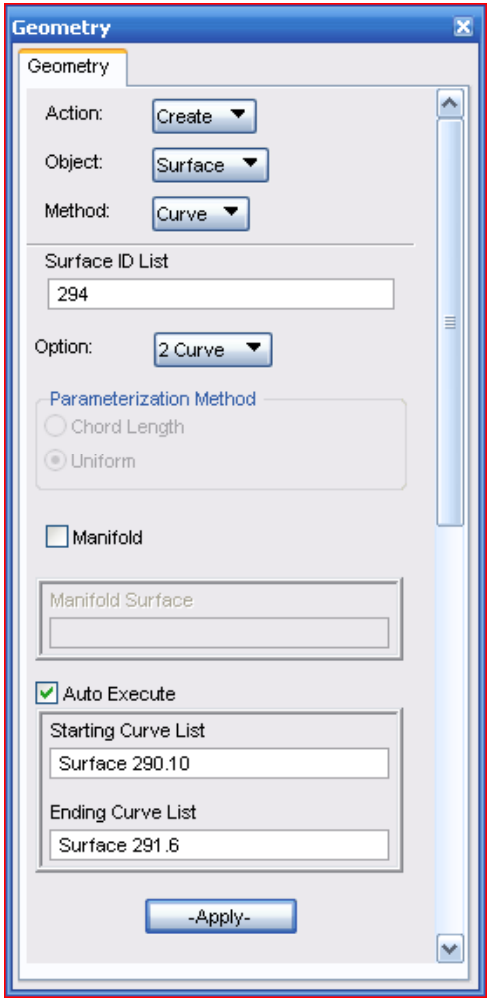


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

The gaps between the trimmed surfaces are filled by creating simple surfaces





Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

- **The set of surfaces are congruent because the sliver surfaces (new surfaces created to fill gaps) were created using existing trimmed surface edges**
- **Because the surfaces are congruent it is possible to use a variety of ways to carry out the meshing**
  - Paver mesh
  - Paver mesh followed by Mesh on Mesh
  - Sheet Body mesh
- **The various techniques are now shown**

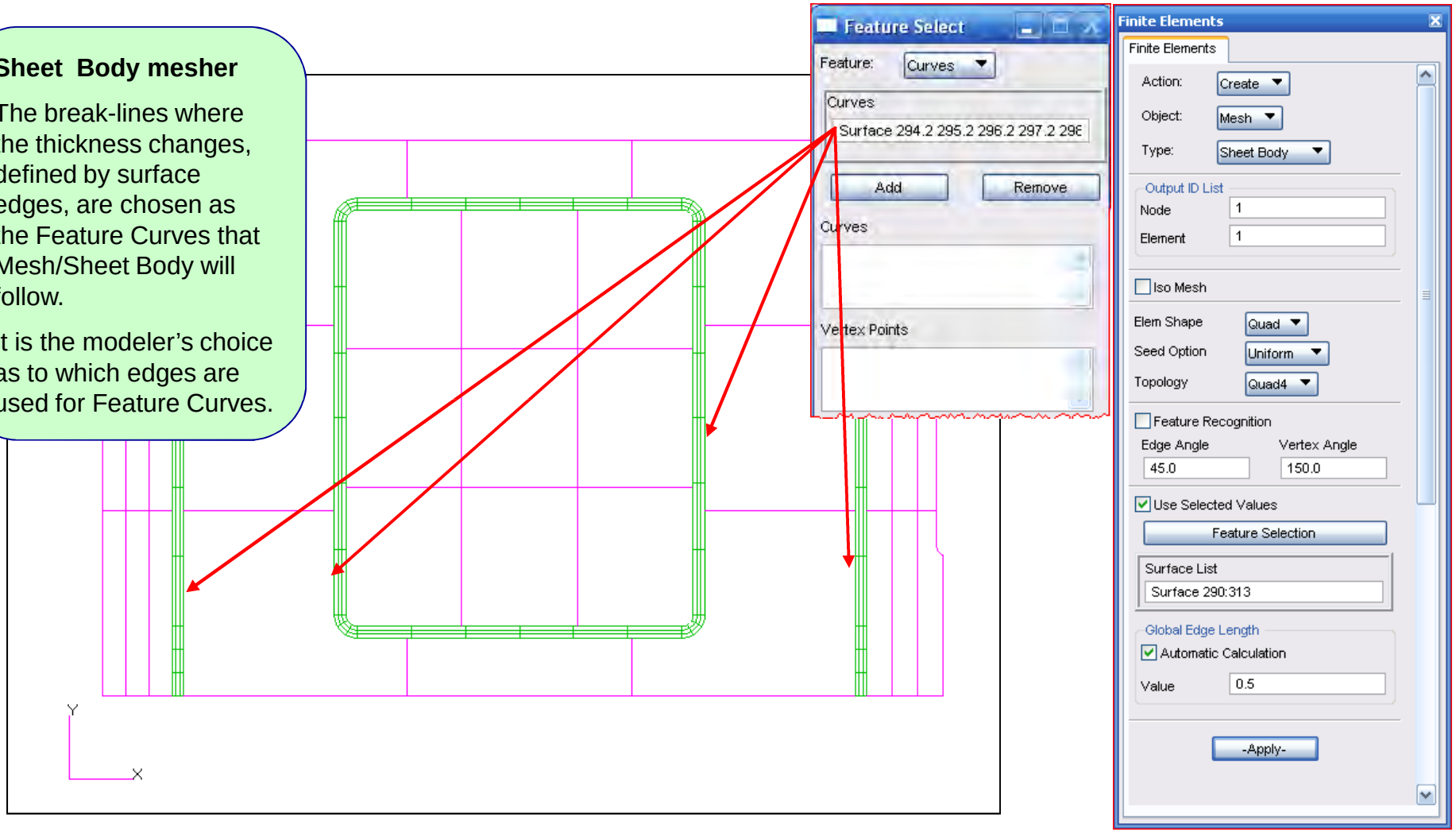
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

Sheet Body mesher

The break-lines where the thickness changes, defined by surface edges, are chosen as the Feature Curves that Mesh/Sheet Body will follow.

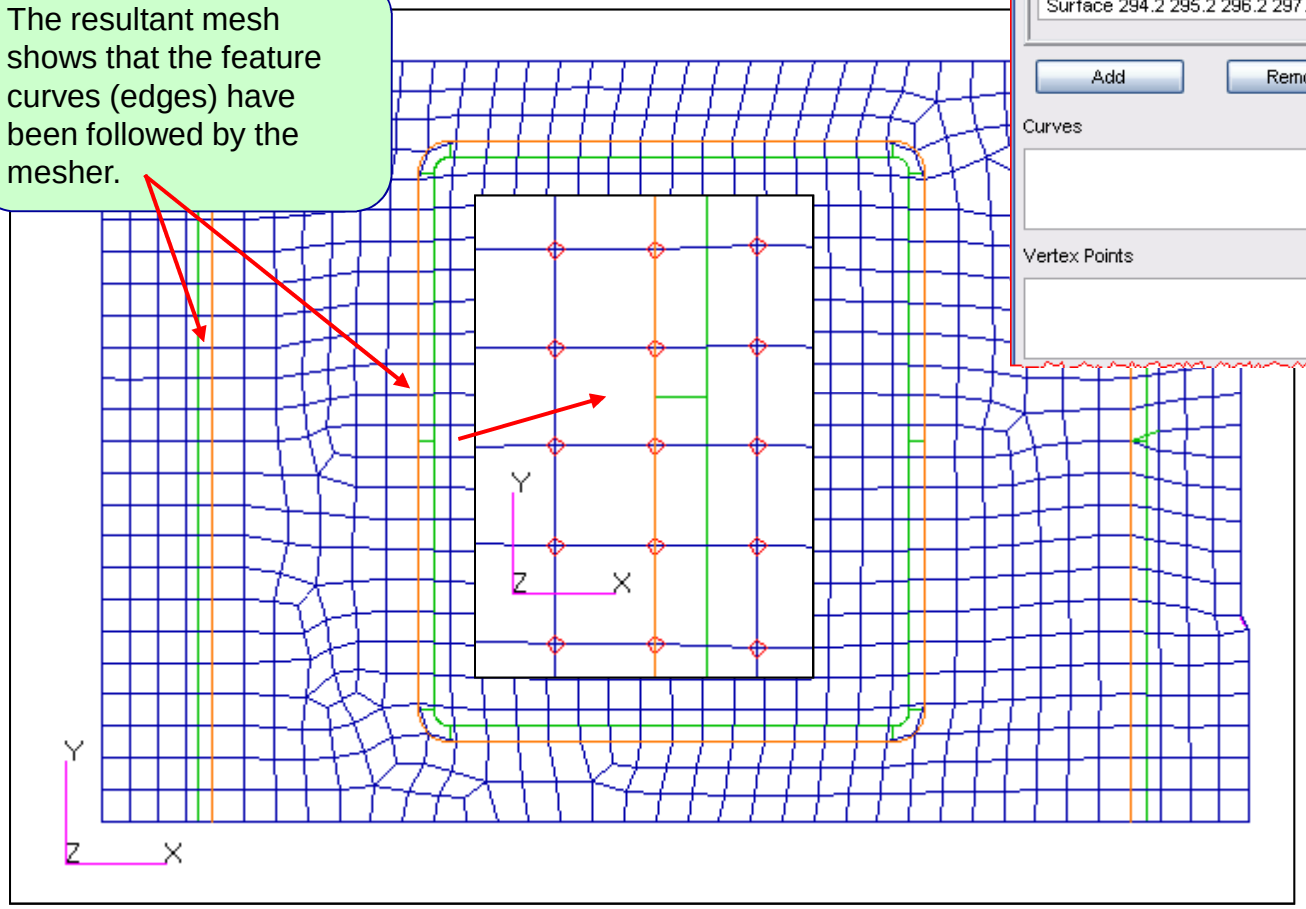
It is the modeler's choice as to which edges are used for Feature Curves.

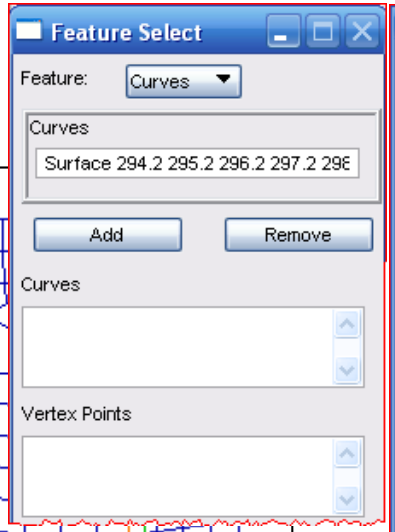


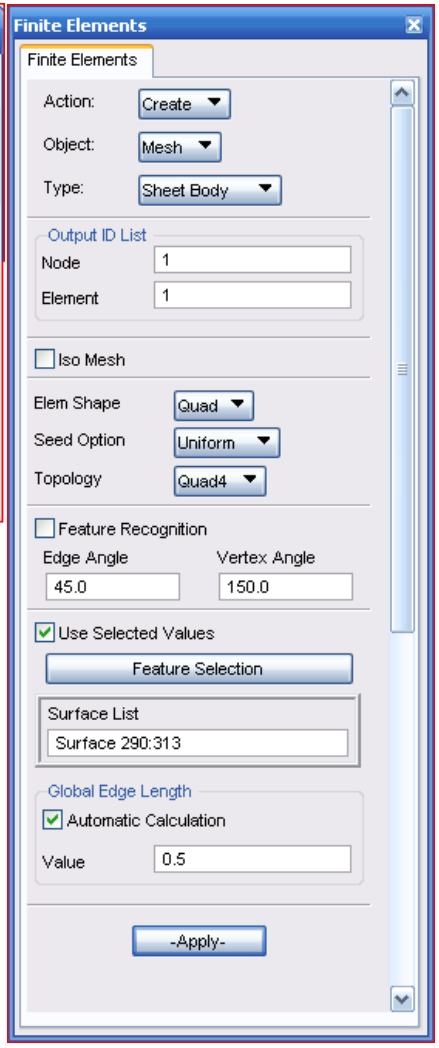
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

The resultant mesh shows that the feature curves (edges) have been followed by the mesher.





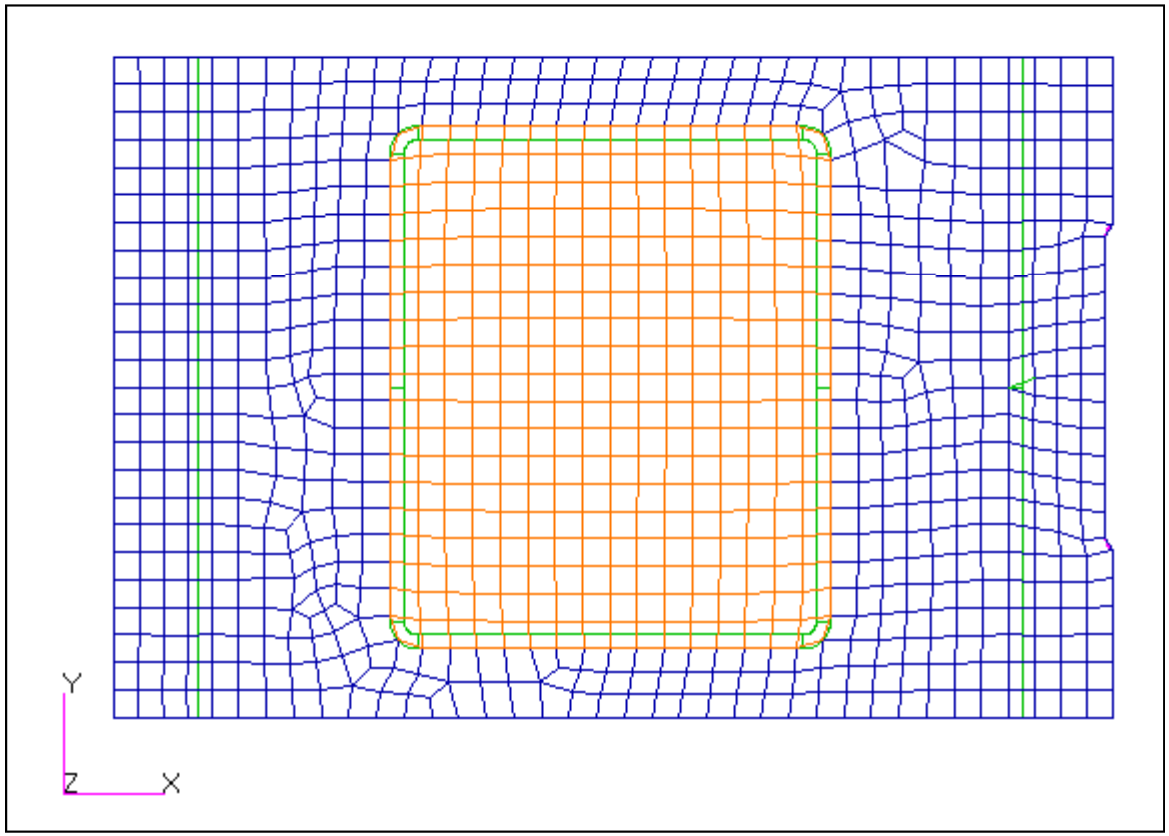


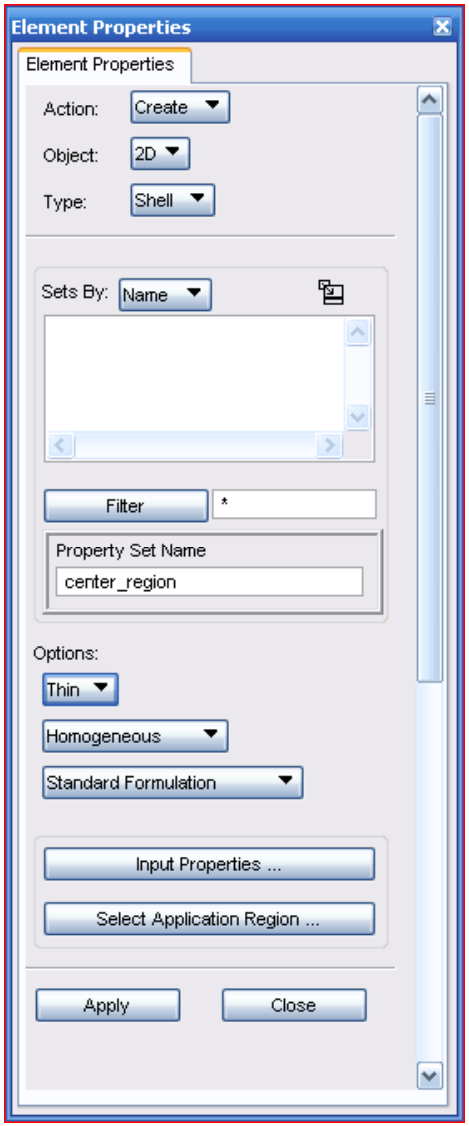
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

The center region plate thickness property is set to 1.00 inch.

The other plate regions are given appropriate thickness.

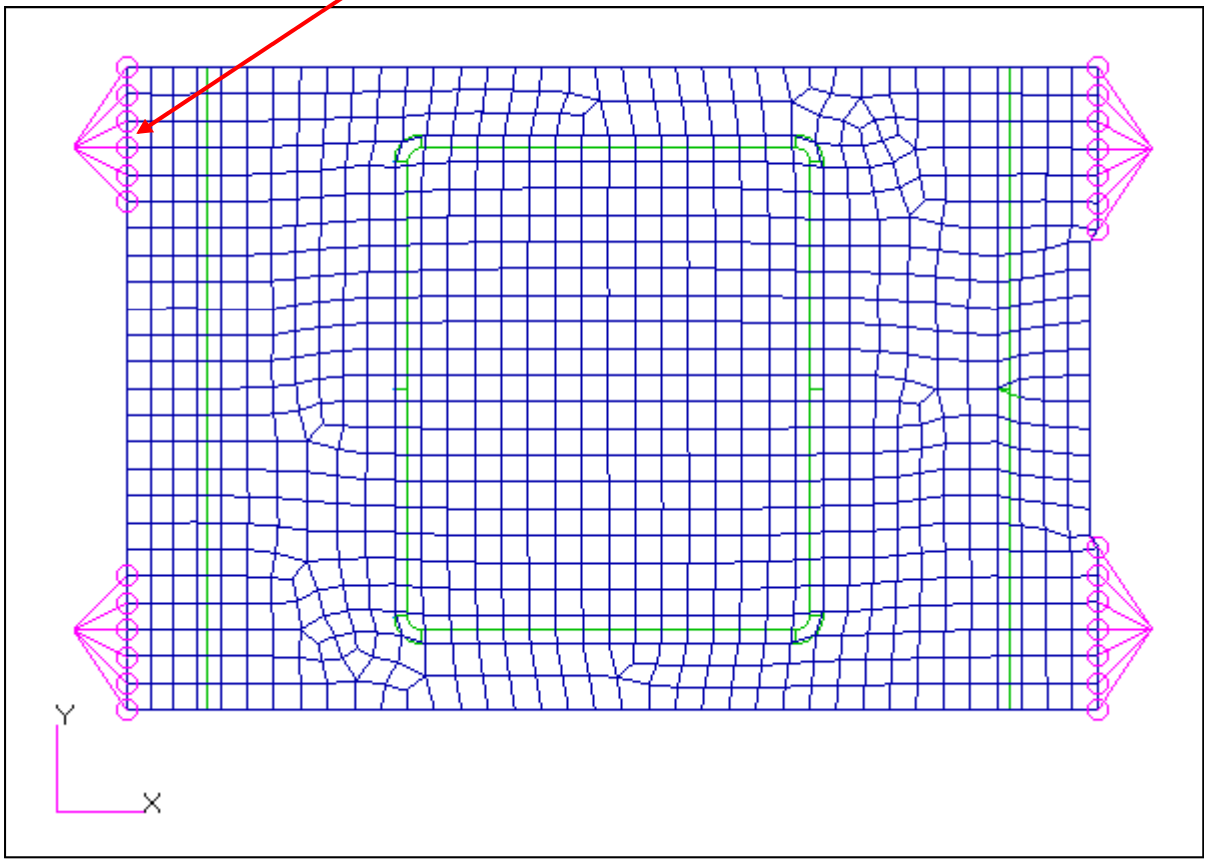


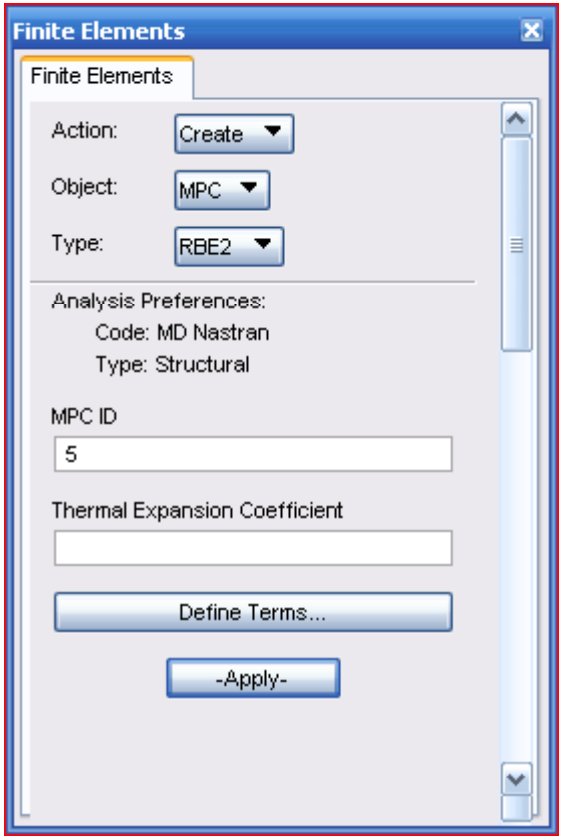


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

Finally, RBE2 rigid “elements” (links) are created to represent the stiff bracket region and its connection to the adjacent structure



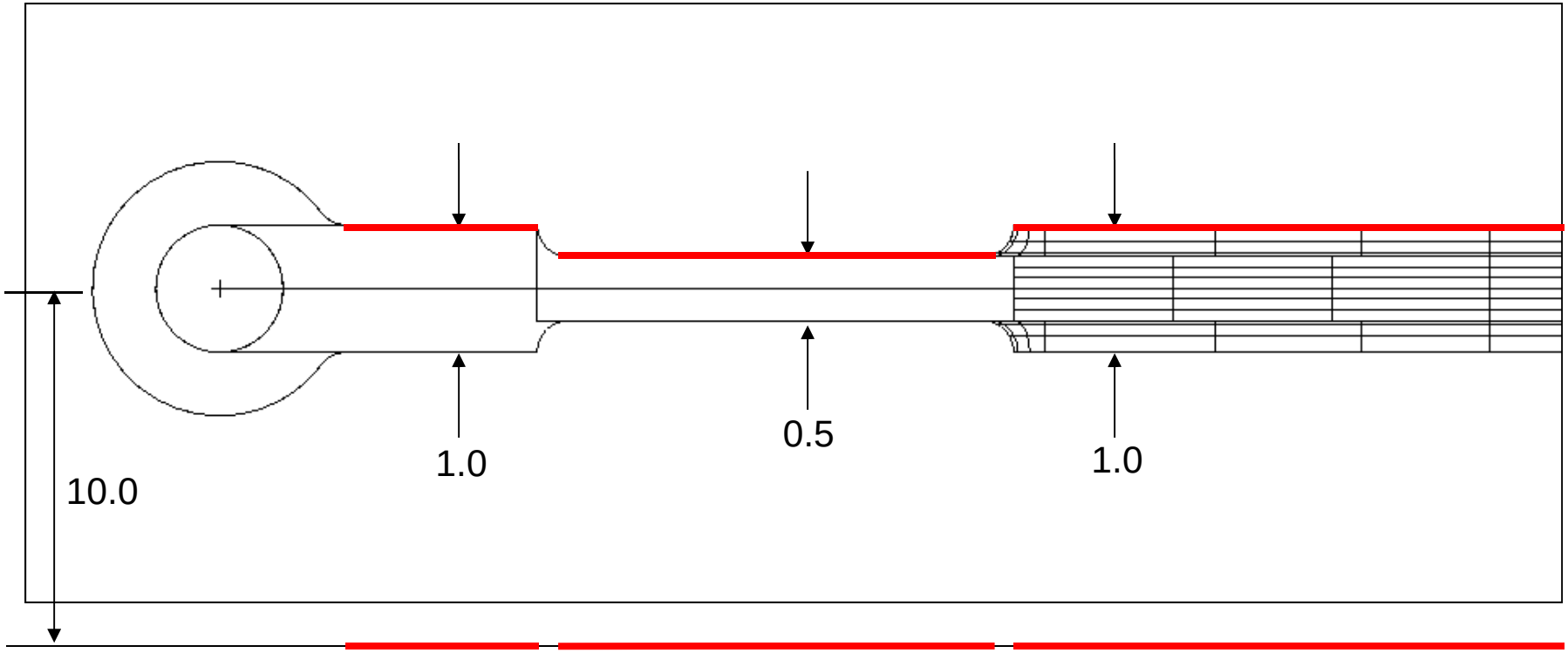


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

Another idealization is the same as previously discussed, but the Paver mesher is used to mesh the congruent surfaces. Shell elements represent the plate-like regions of the support tray, and RBE2s represent the brackets.

The effects of the radii are ignored.

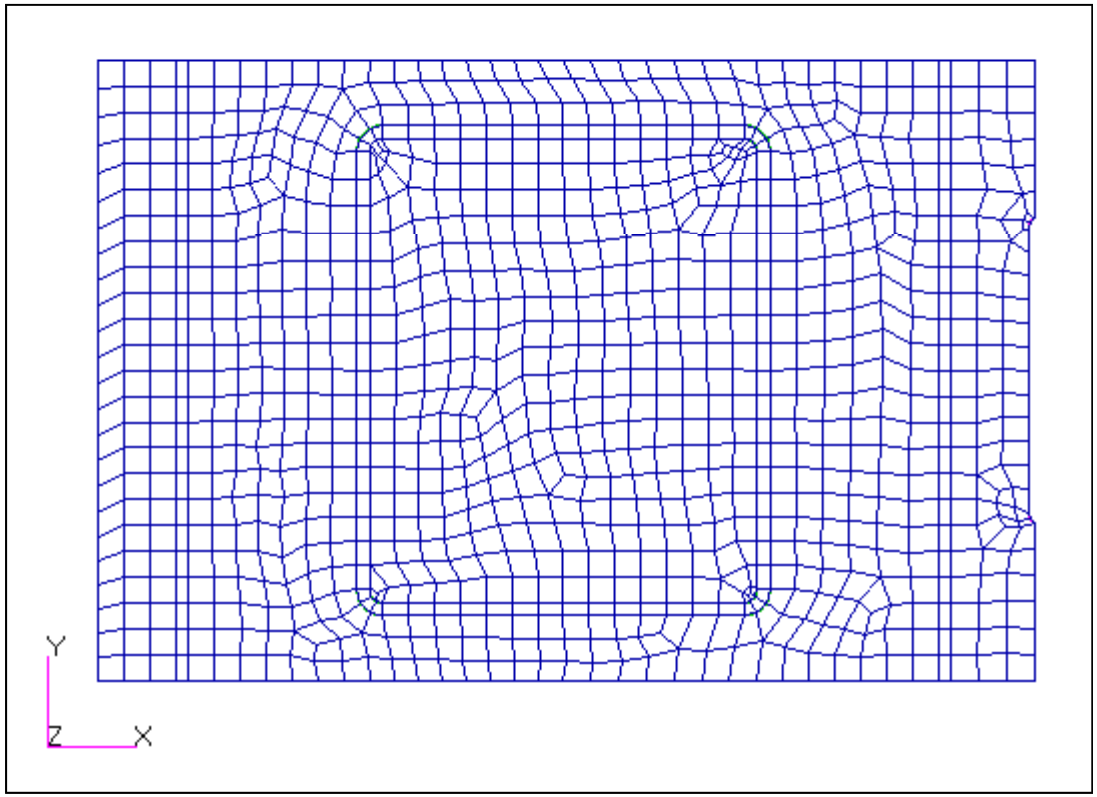


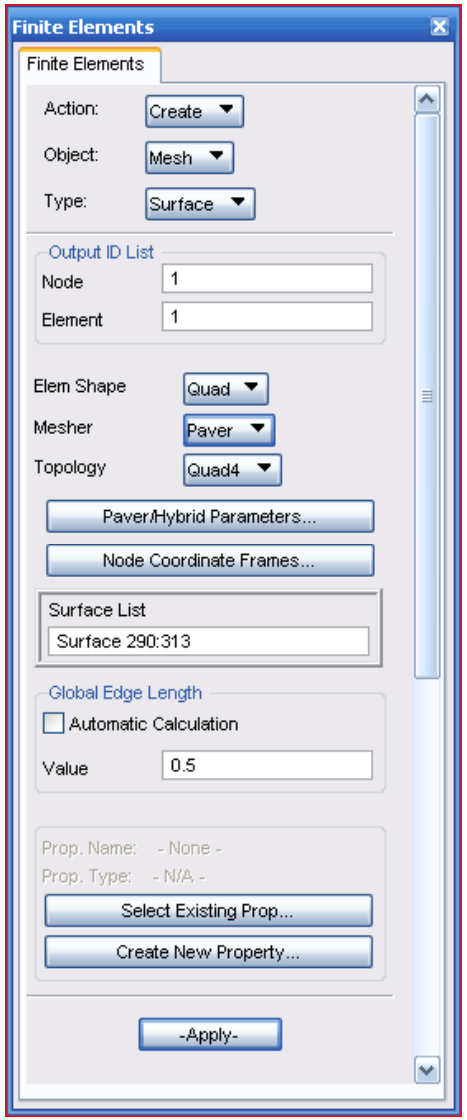
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

The geometry is congruent.

Therefore, as an alternative, use the Paver mesher using the same GEL of 0.5





Copyright© 2012 MSC.Software Corporation

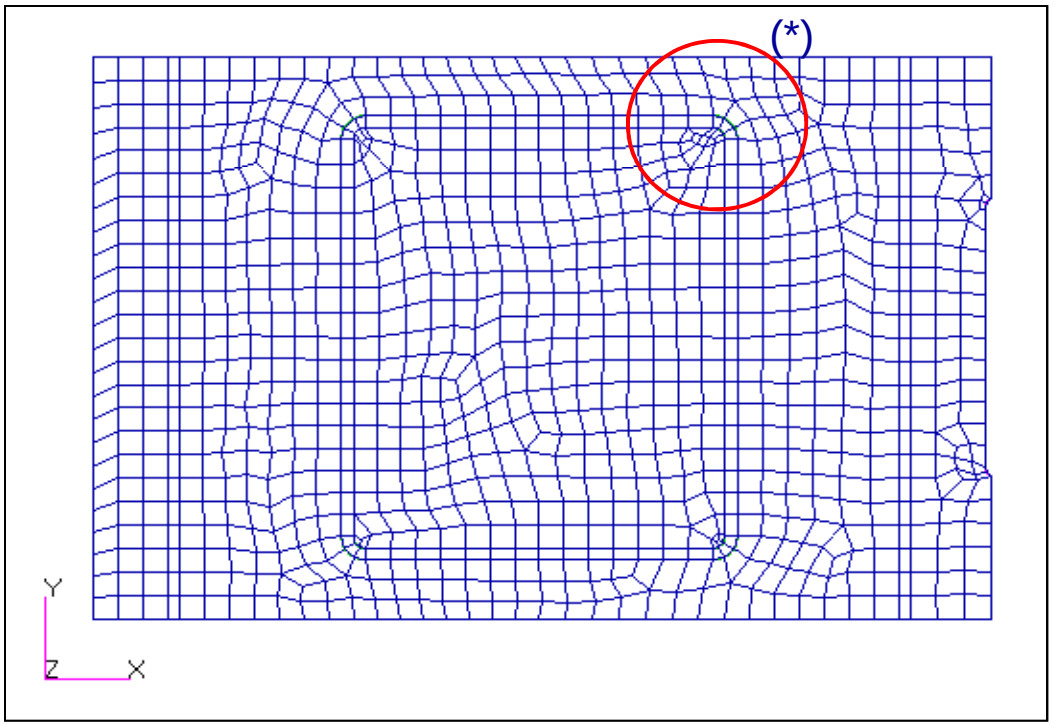
CASE STUDY: CARRIER 3D -> 2D

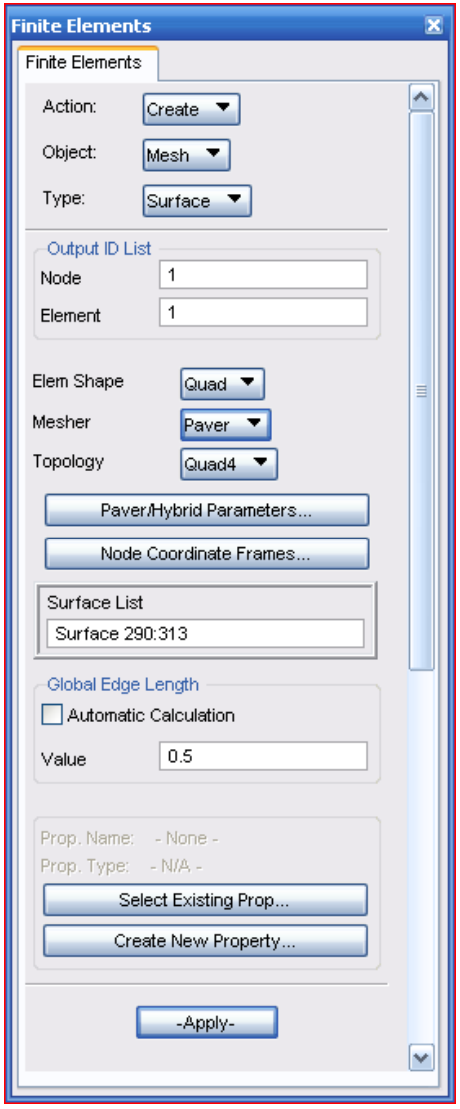
The meshing was guided by **all** the geometry, e.g. surface edges

The presence of all the geometric features may be a disadvantage, e.g. there may be small elements in some regions (*****).

In this case, the regions with different thickness are bounded by surface edges; this makes it easy to specify each thickness.

Also, this Paver mesher can be used as the starting point for using Mesh / On Mesh.



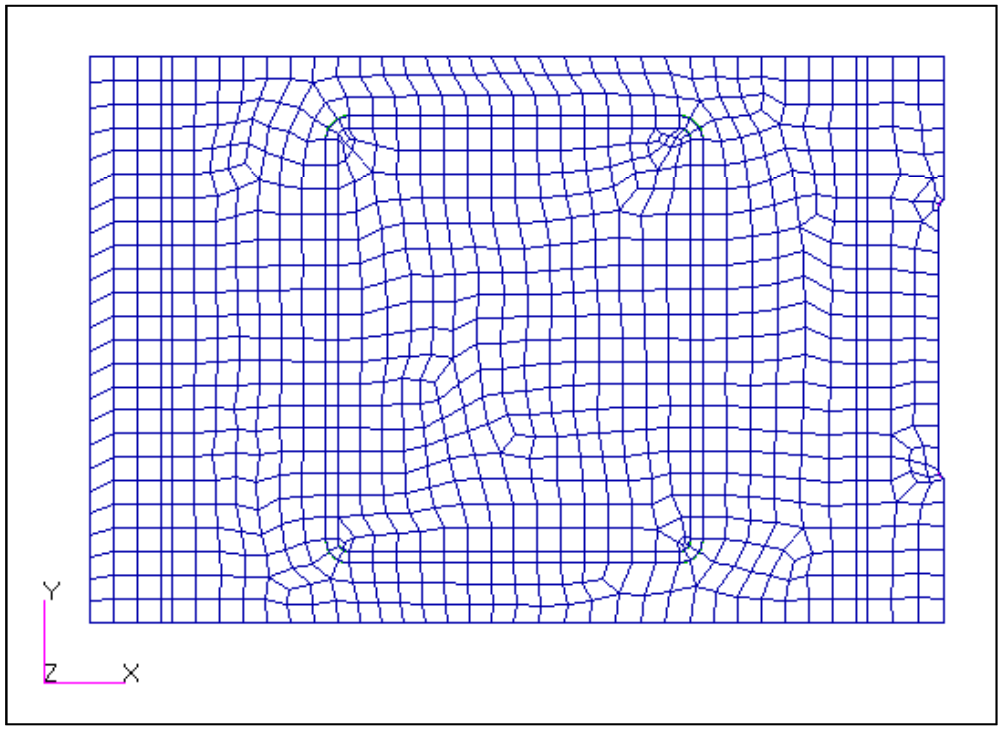


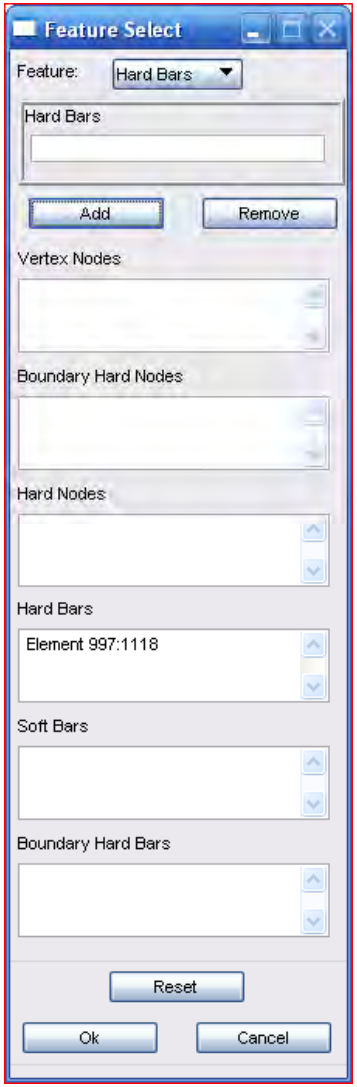
Copyright© 2012 MSC.Software Corporation

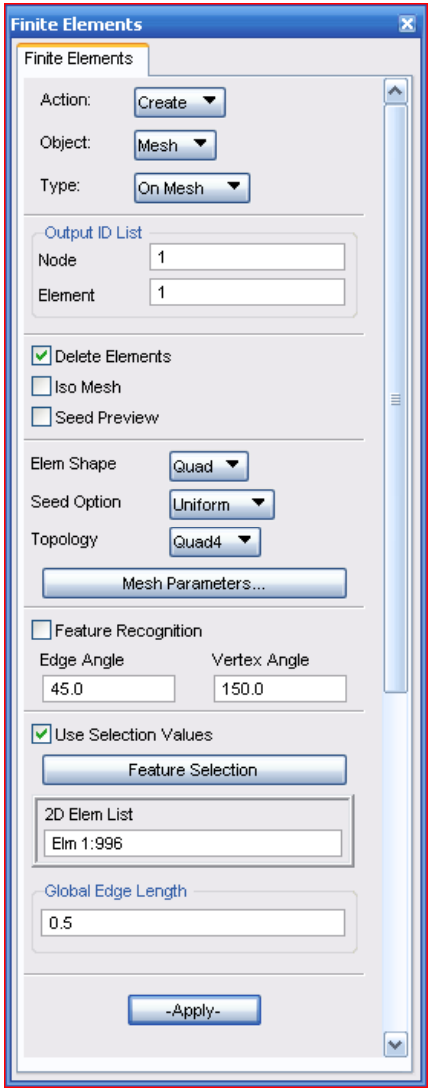
CASE STUDY: CARRIER 3D -> 2D

Equivalence the 2D meshes before using **On Mesh** from the Meshers group under the Meshing tab.

Create Bar2 elements on some of the internal surface edges; these will be used as Hard Bars for Mesh / On Mesh.



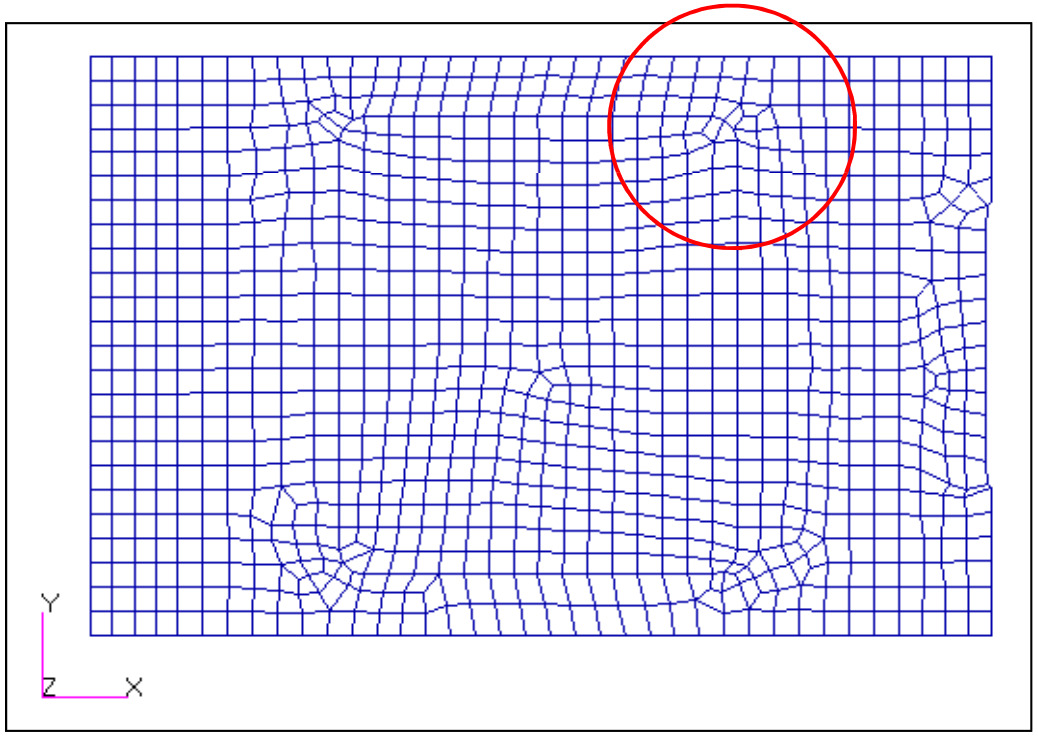


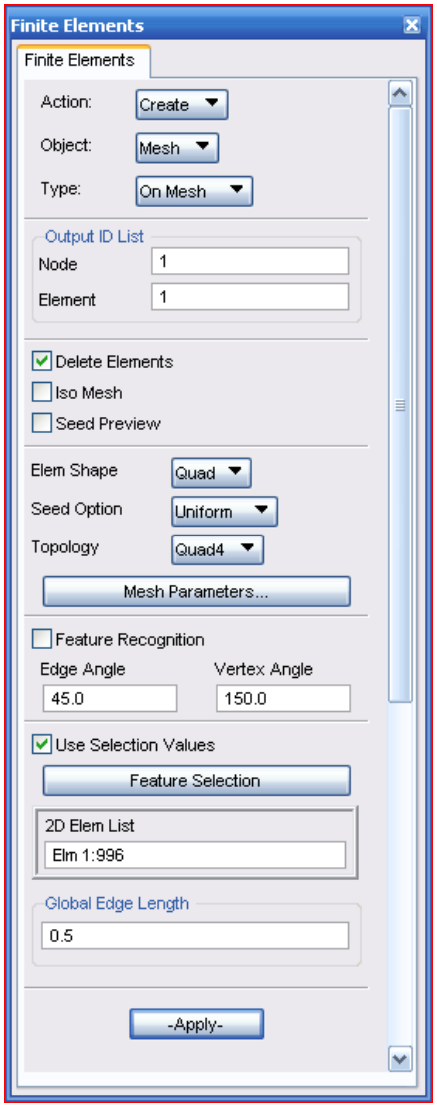


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CARRIER 3D -> 2D

- **This mesh is not as good as the one created using Mesh / Sheet Body.**
- **However, with some additional work (e.g. IsoMesh a sub-region(s)), the mesh can be improved.**





Copyright© 2012 MSC.Software Corporation

EXERCISES

- Perform Workshop 8 “Mesh on Mesh 2D to 3D” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

3D START POINT, 2 1/2D IDEALIZATION

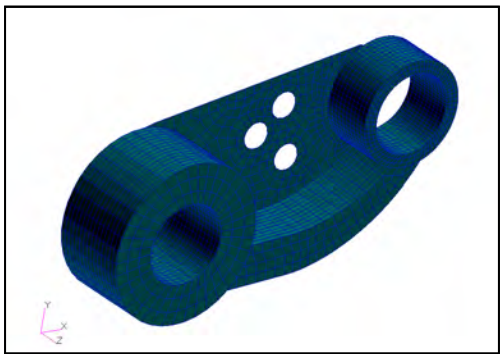
- **Need a 3D model (mesh) or combined 3D and 2D model (mesh)**
  - Prefer or must have hexahedral elements
    - Create a set of congruent parametric solids, and IsoMesh them
    - Create 2D Quad element mesh(es) and extrude them to create Hex element mesh(es)
- **Start with 3D geometry**
  - Create a set of parametric solids, and IsoMesh
    - This is discussed in the next section
- **Start with 3D geometry and breakdown to 2D, or start with 2D geometry**
  - Mesh 2D geometry with Quad elements
  - Sweep Quads to create Hex elements
    - Several methods of sweeping, i.e. Extrude, Glide

Start Point 3D	Nothing		Mesh		Geometry		
Method	N/A	N/A	N/A	N/A	Mid Plane Mesher	Mid Surface Extraction	Geometry breakdown and rebuild methods
End Point 2D	Mesh	Geometry	Mesh	Geometry	Mesh	Geometry	

Copyright© 2012 MSC.Software Corporation

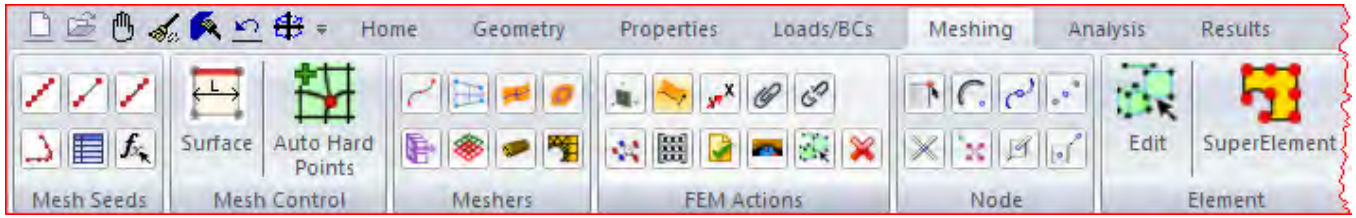
3D START POINT, 2 1/2D IDEALIZATION

- **2 ½D model – 3D elements created by sweeping 2D elements**
- **Sophistication of 3D mesh is usually in the directions normal to the sweep direction. The swept 2D elements can be created by Paver meshing general trimmed surfaces of general shape**
- **Sweeping (normal to the 2D elements) uses simple generator rules, i.e. Extrude, Glide, etc.**
- **Idealization methods, e.g. how to represent structure with simplifying finite element mesh**
- **Features in the sweep dimension, such as transverse holes and complex shape changes, may not be modeled by sweeping. It may be necessary to sweep in another direction to deal with them. In certain circumstances, it may be possible to assume that these types of areas are unimportant concerning their effects on stress, and not model them.**
- **Construction technique for 2 ½ D**
  - Disassembly of 3D geometry into 2D (surfaces)
  - 2D Meshing
  - 3D Mesh creation using sweep meshing

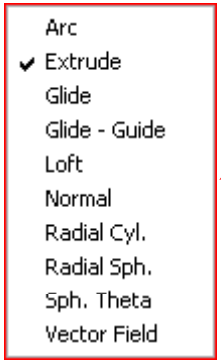


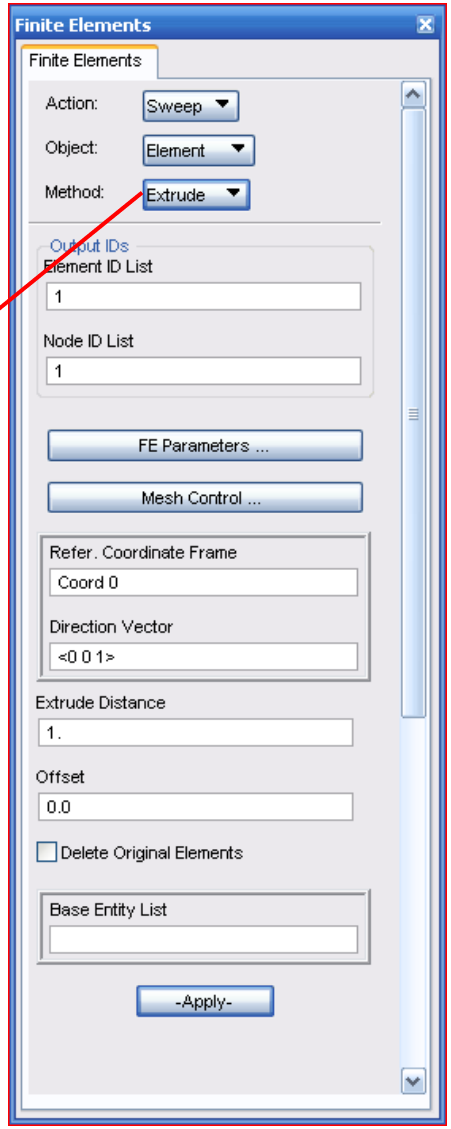
Copyright© 2012 MSC.Software Corporation

3D START POINT, 2 1/2D IDEALIZATION



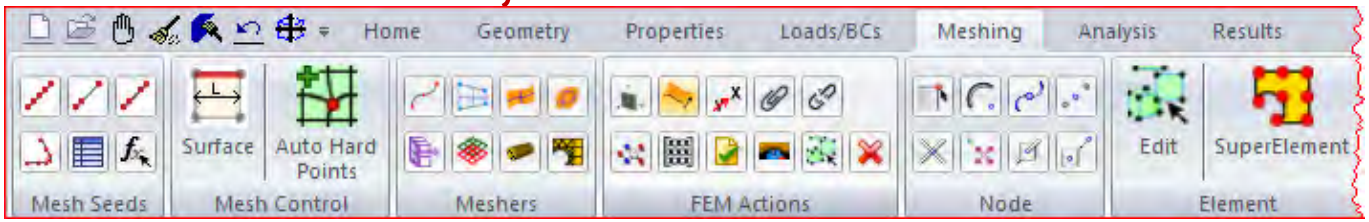
- **3D mesh creation methods reviewed herein are:**
  - Extrude
  - Glide
  - Loft
  - Vector Field





Copyright© 2012 MSC.Software Corporation

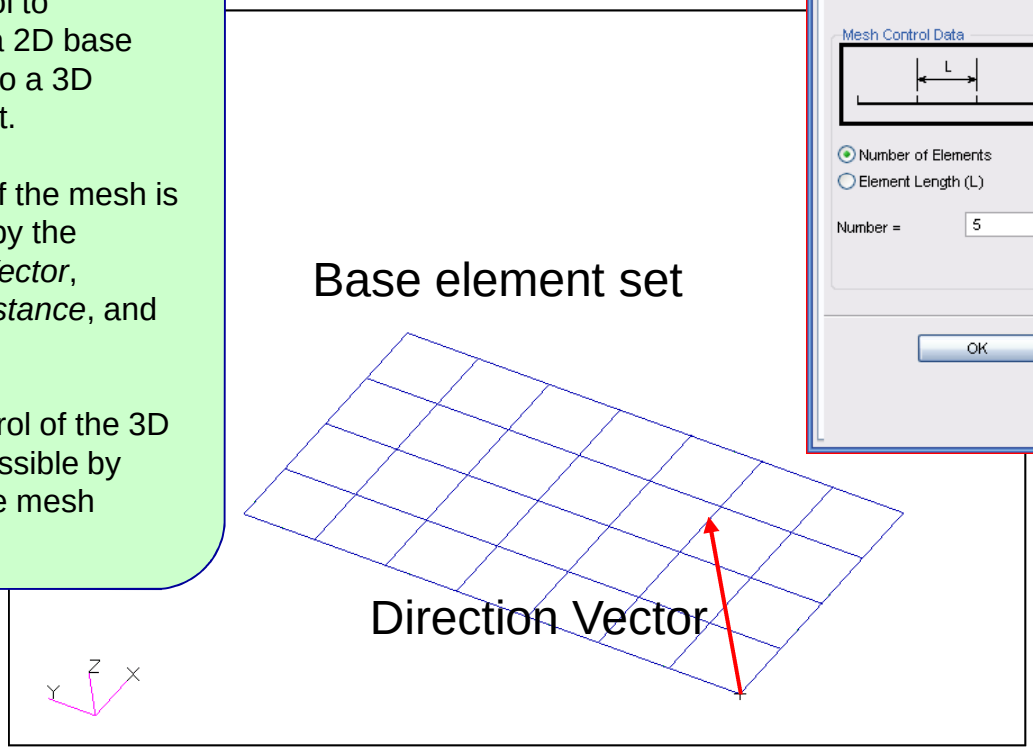
3D START POINT, 2 1/2D IDEALIZATION

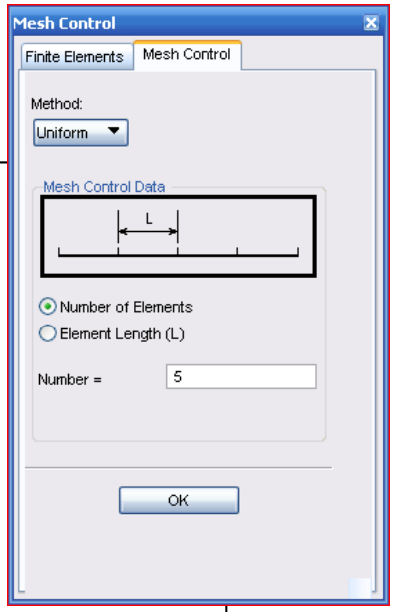


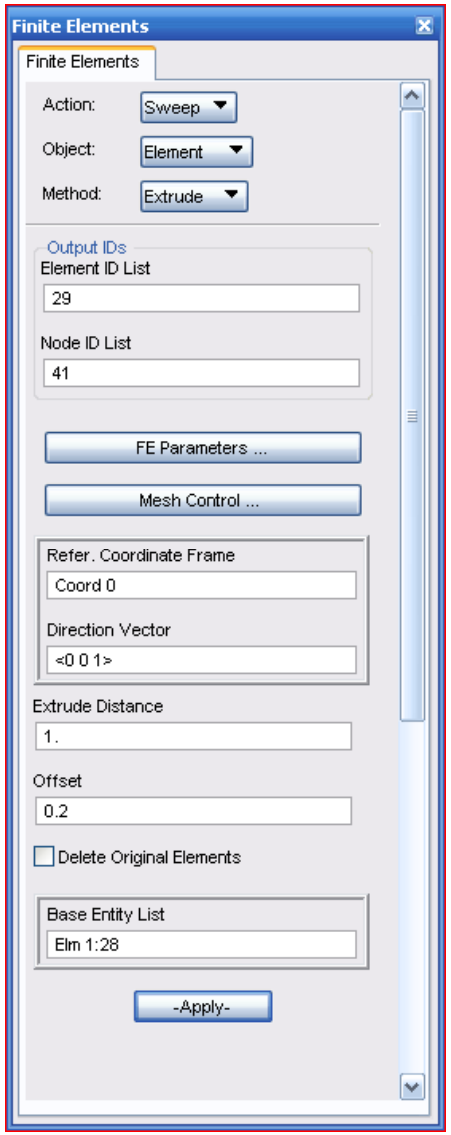
In this example, use the Extrude tool to transform a 2D base element into a 3D element set.

The form of the mesh is controlled by the *Direction Vector*, *Extrude Distance*, and *Offset*.

Some control of the 3D mesh is possible by defining the mesh grading.

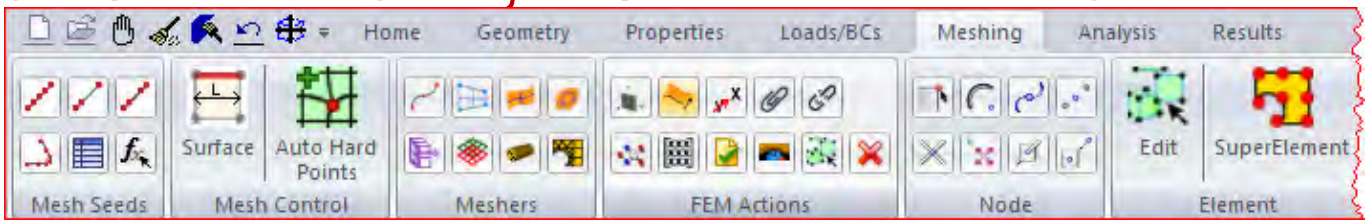




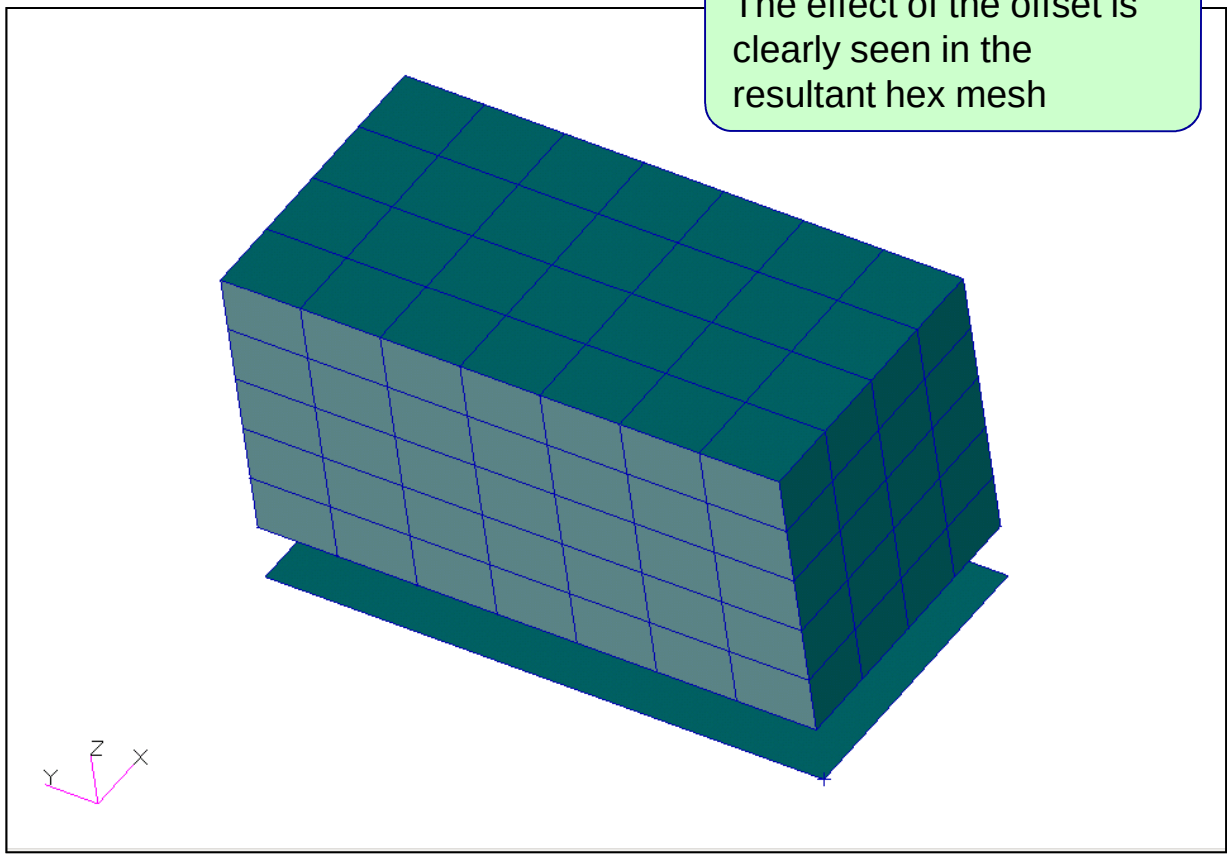


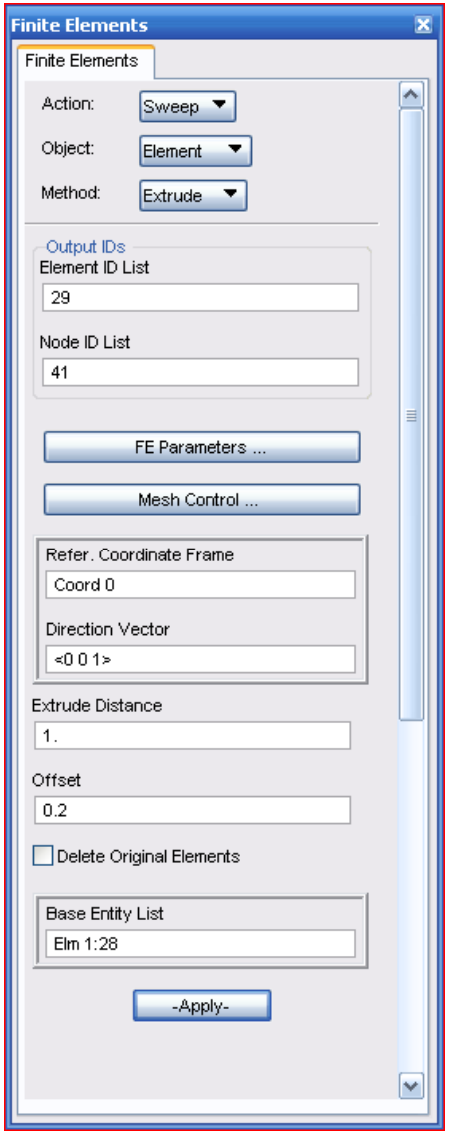
Copyright© 2012 MSC.Software Corporation

3D START POINT, 2 1/2D IDEALIZATION



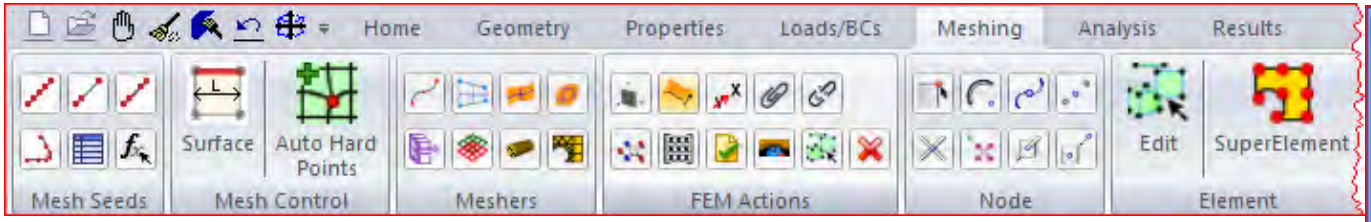
The effect of the offset is clearly seen in the resultant hex mesh





Copyright© 2012 MSC.Software Corporation

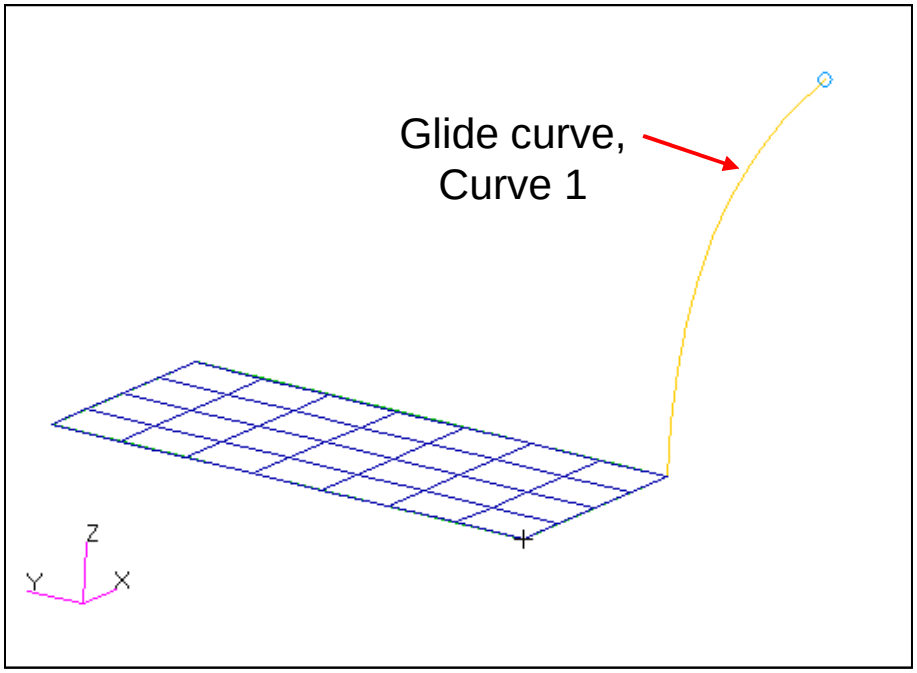
3D START POINT, 2 1/2D IDEALIZATION

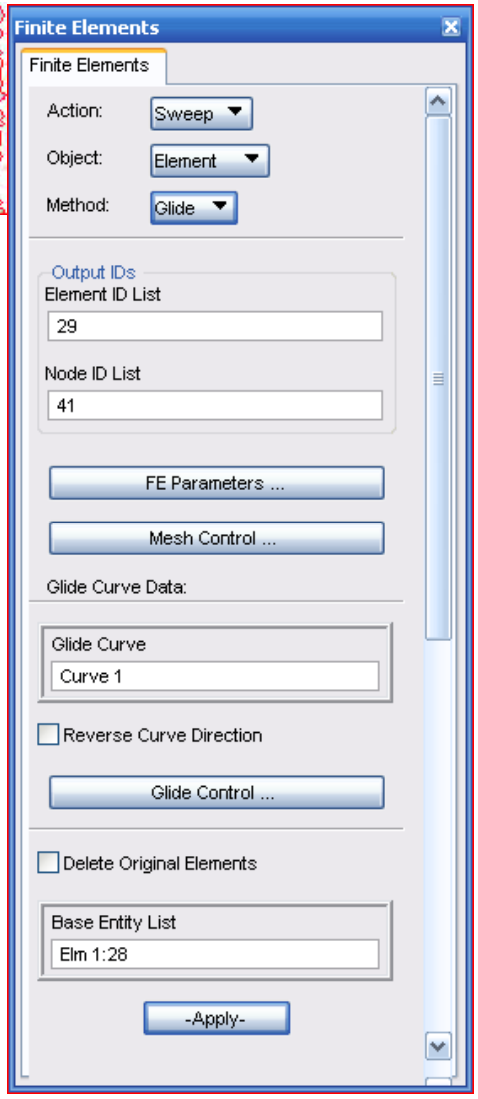


It is possible to carry out a more complex operation using Glide rather than Extrude.

A glide curve is a specified curve which controls the form of the 3D shape.

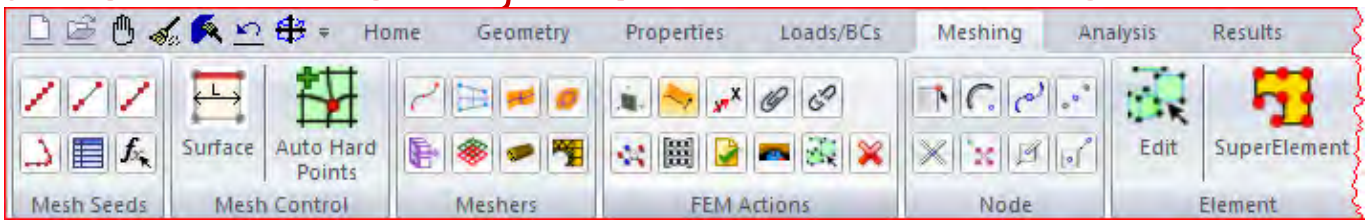
A typical glide curve is shown.





Copyright© 2012 MSC.Software Corporation

3D START POINT, 2 1/2D IDEALIZATION

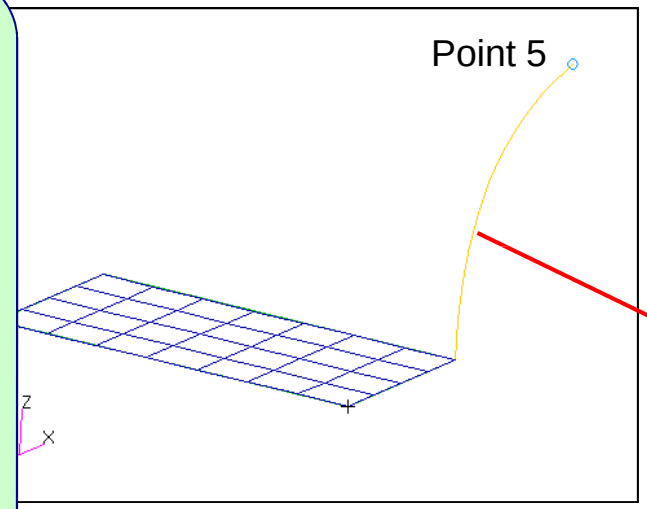


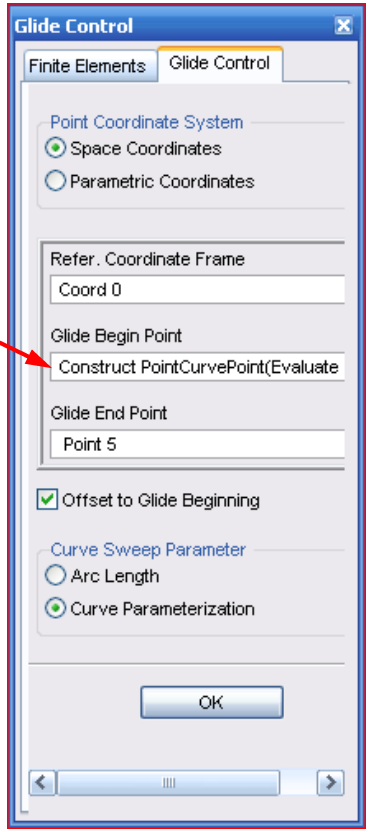
Further refinement is possible using Glide Control…

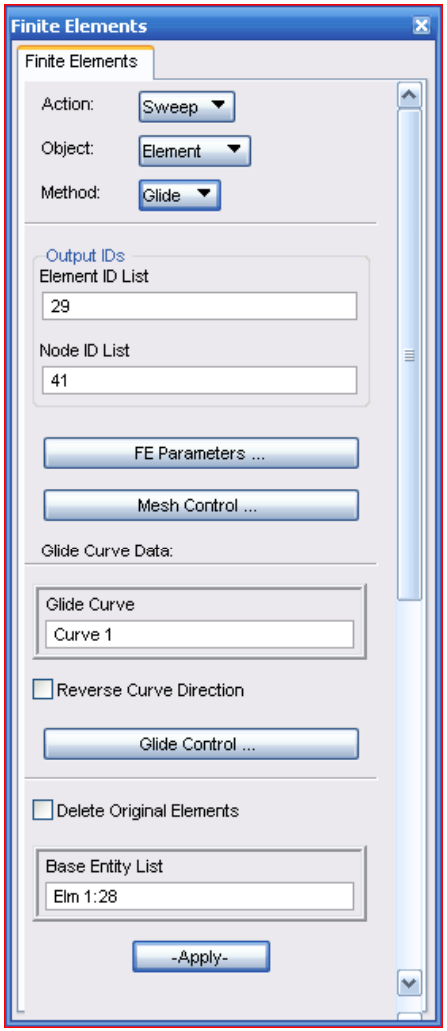
In this case, use space coordinates to define a start and end point for the glide.

The start is at a user defined point on the curve.

The end is at an existing point.

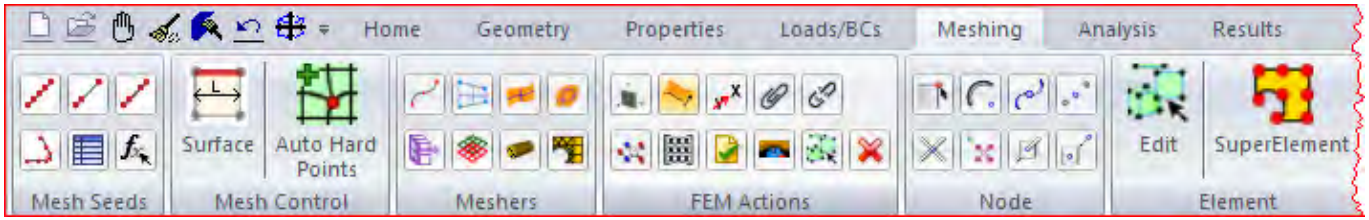




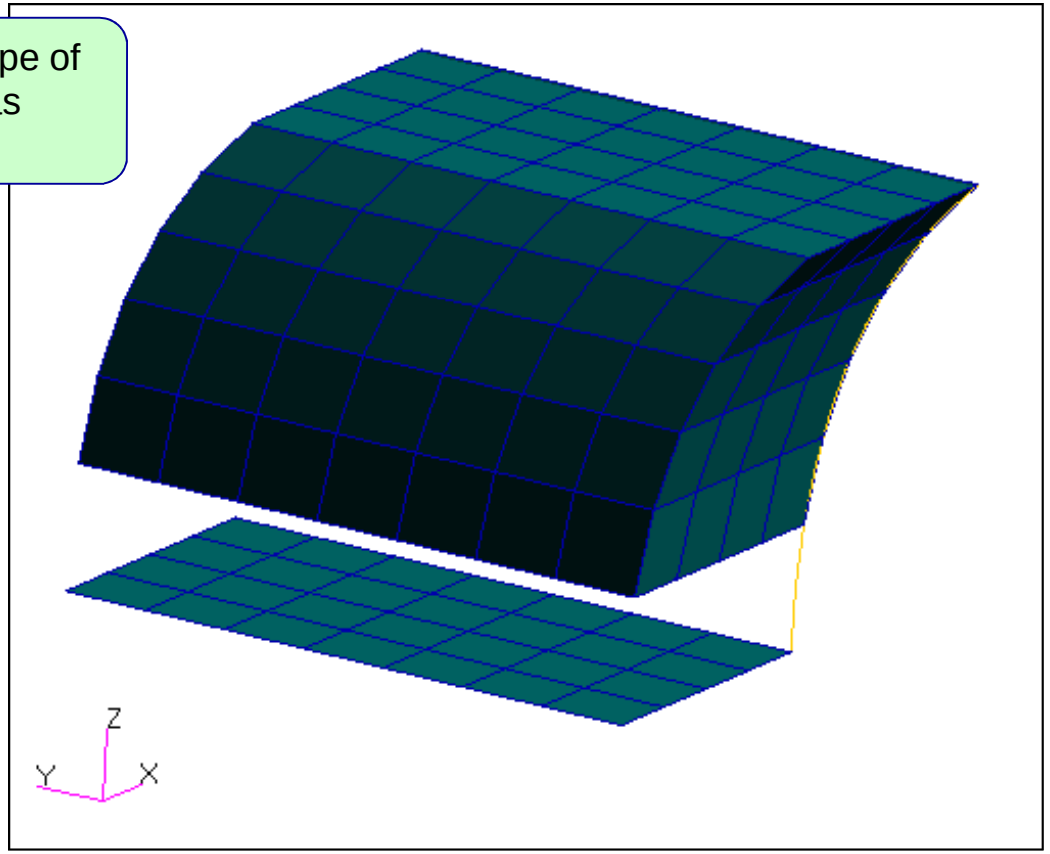


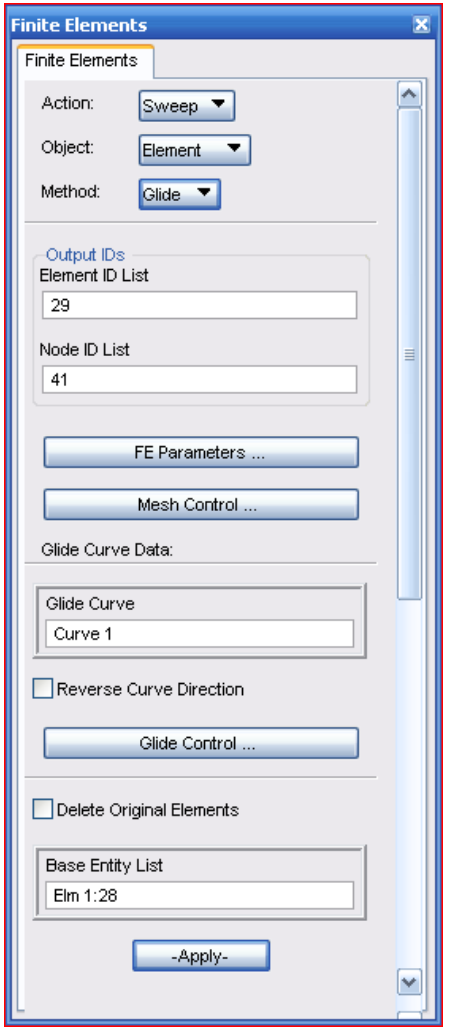
Copyright© 2012 MSC.Software Corporation

3D START POINT, 2 1/2D IDEALIZATION



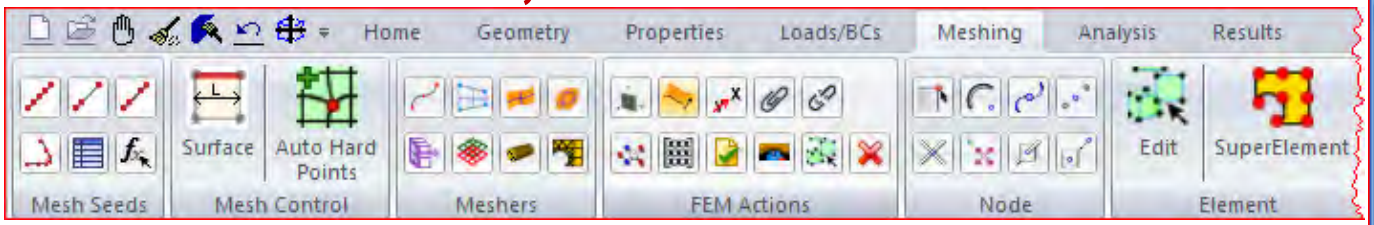
The resultant shape of the hex mesh is as shown





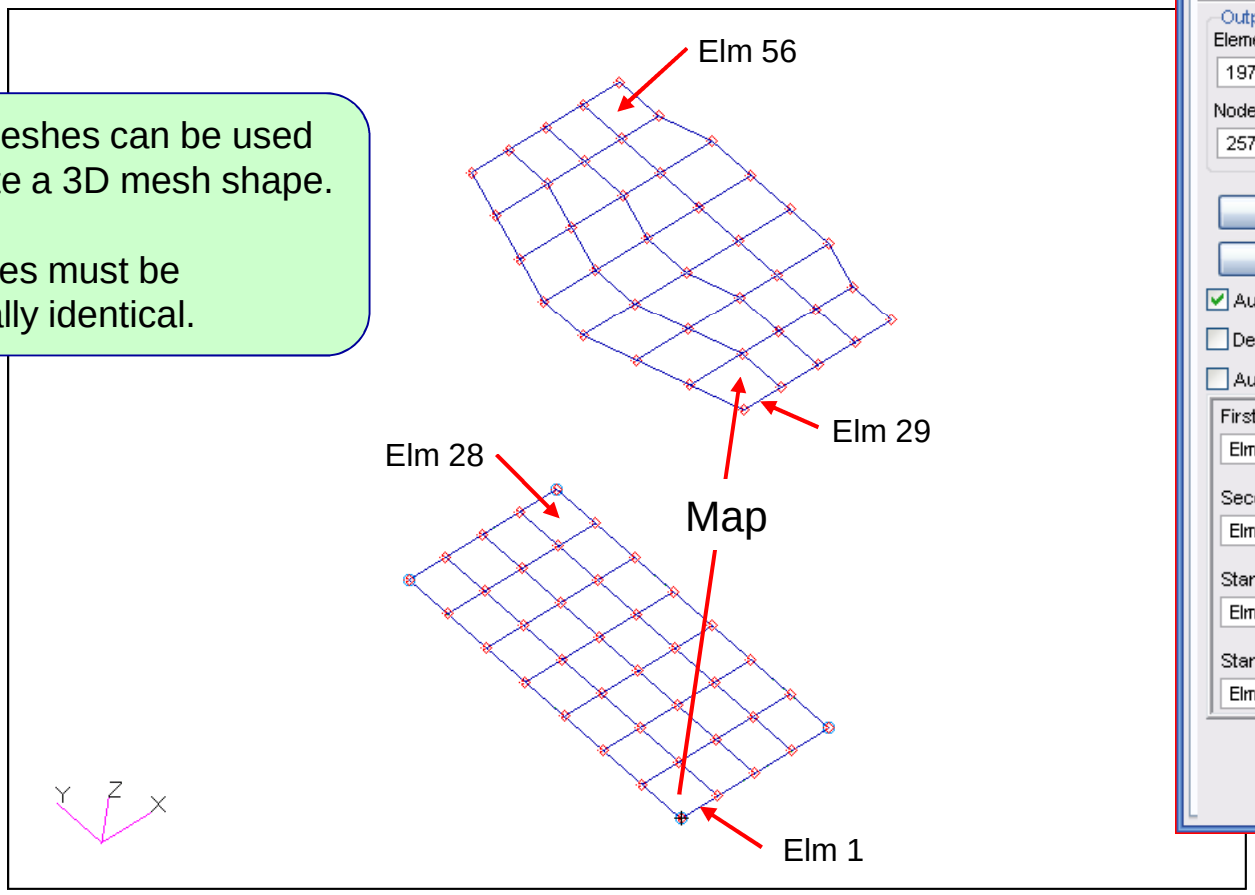
Copyright© 2012 MSC.Software Corporation

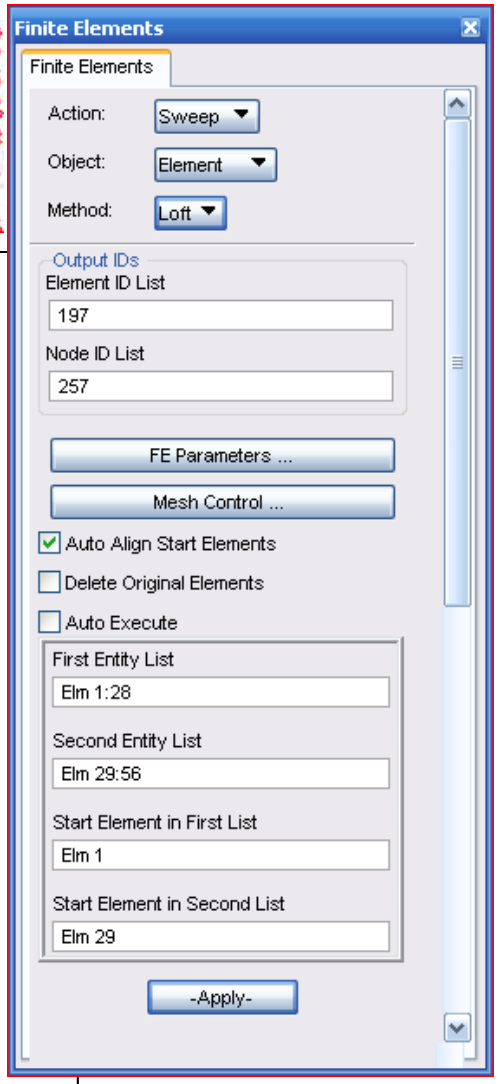
3D START POINT, 2 1/2D IDEALIZATION



Two 2D meshes can be used to generate a 3D mesh shape.

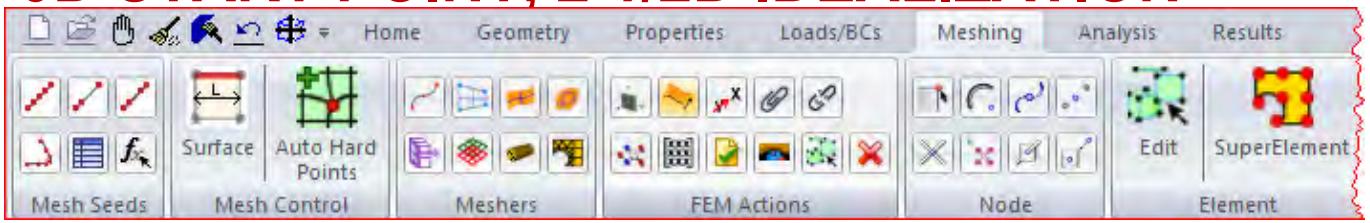
The meshes must be topologically identical.



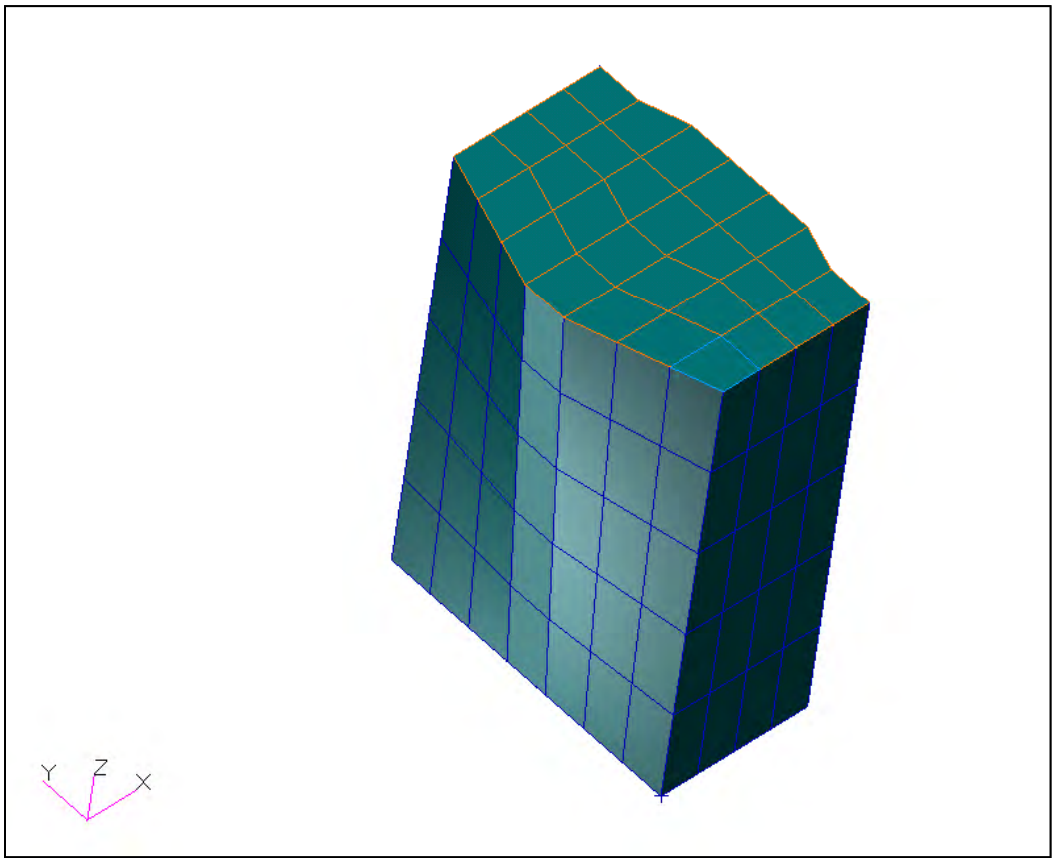


Copyright© 2012 MSC.Software Corporation

3D START POINT, 2 1/2D IDEALIZATION

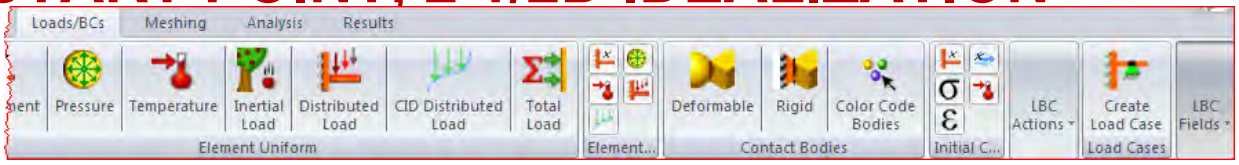


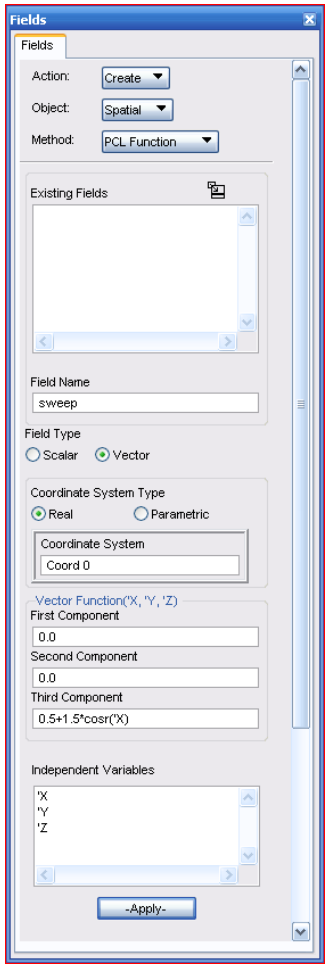
The resultant hex mesh is as shown.

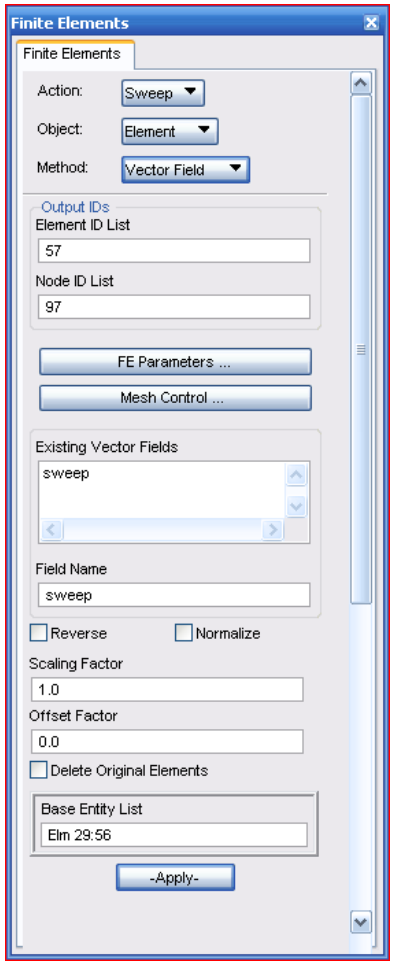


Copyright© 2012 MSC.Software Corporation

3D START POINT, 2 1/2D IDEALIZATION





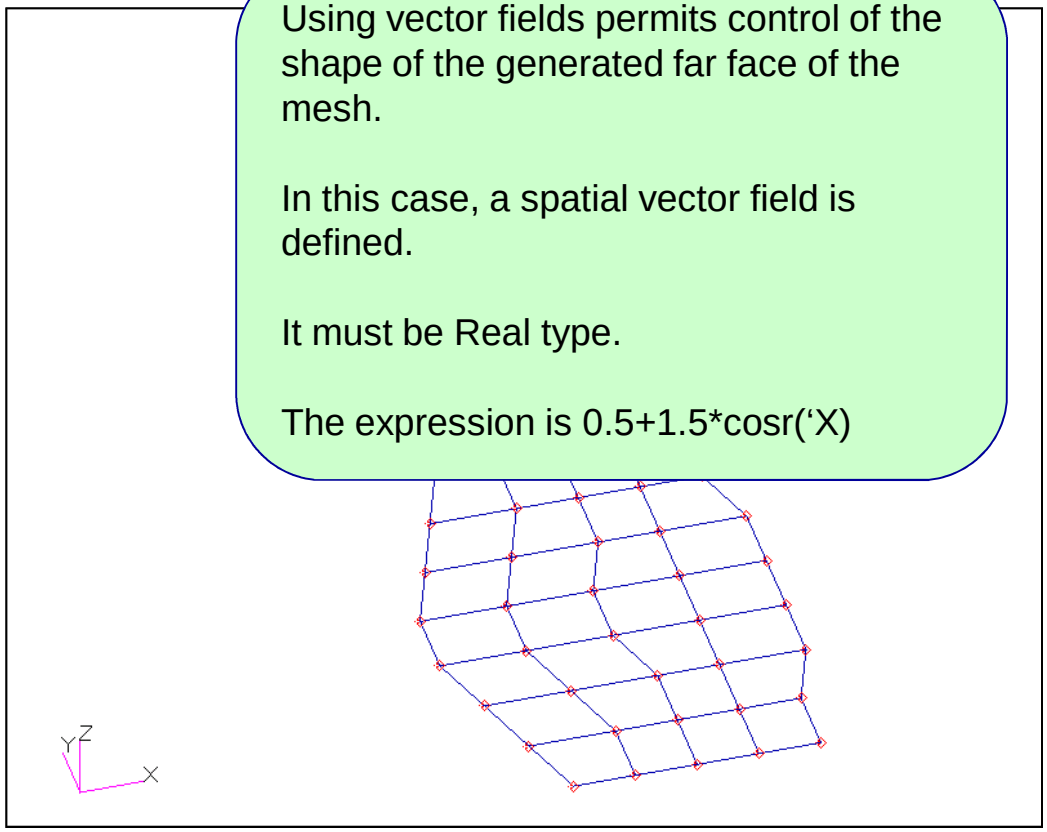


Using vector fields permits control of the shape of the generated far face of the mesh.

In this case, a spatial vector field is defined.

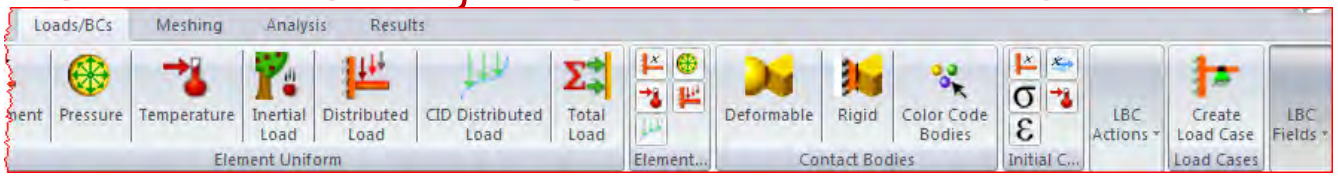
It must be Real type.

The expression is 0.5+1.5*cosr('X)

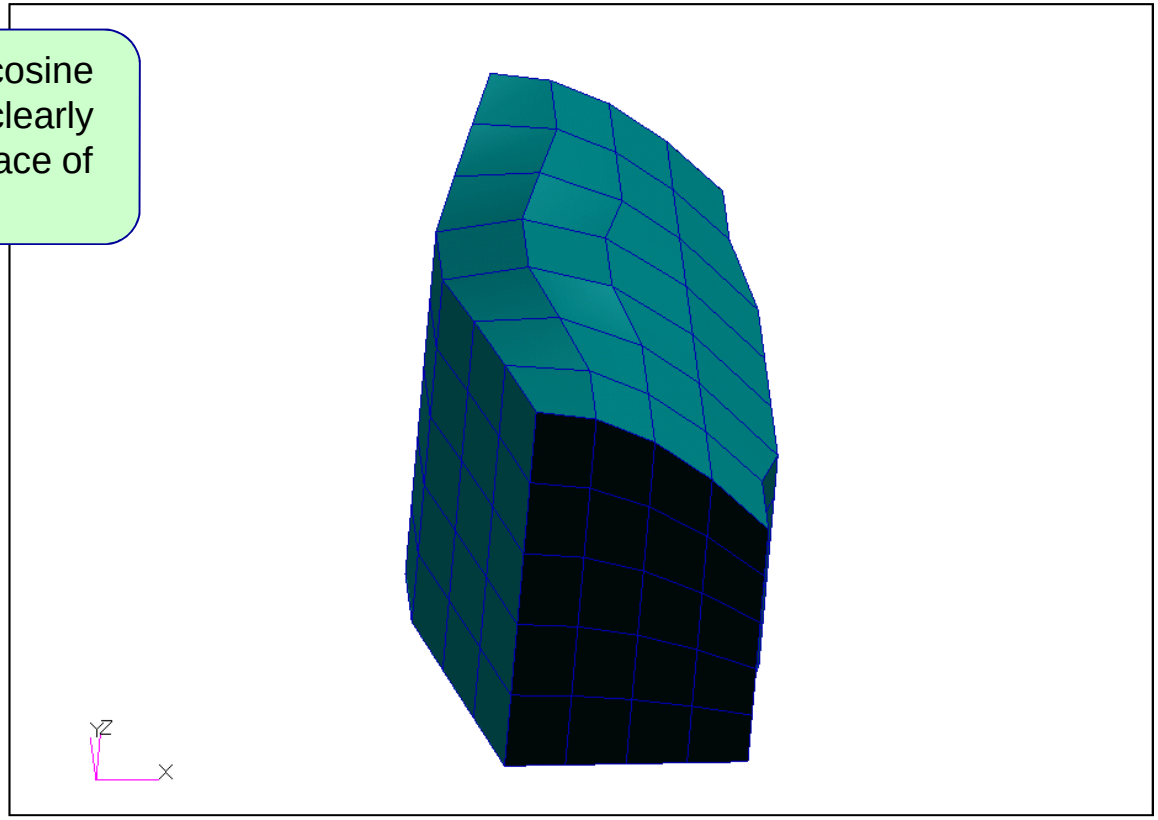


Copyright© 2012 MSC.Software Corporation

3D START POINT, 2 1/2D IDEALIZATION

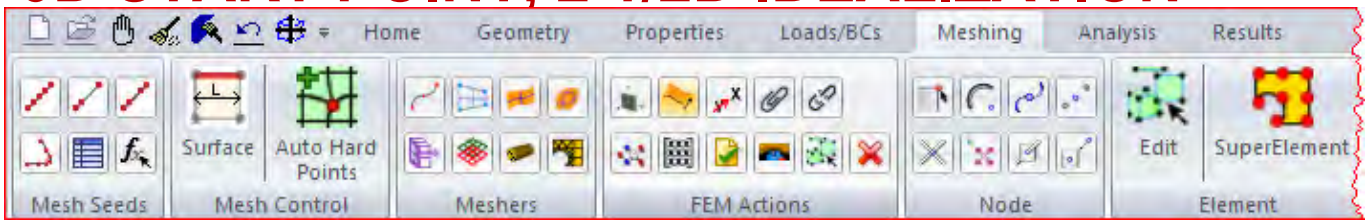


The form of the cosine function can be clearly seen on the far face of the hex mesh.

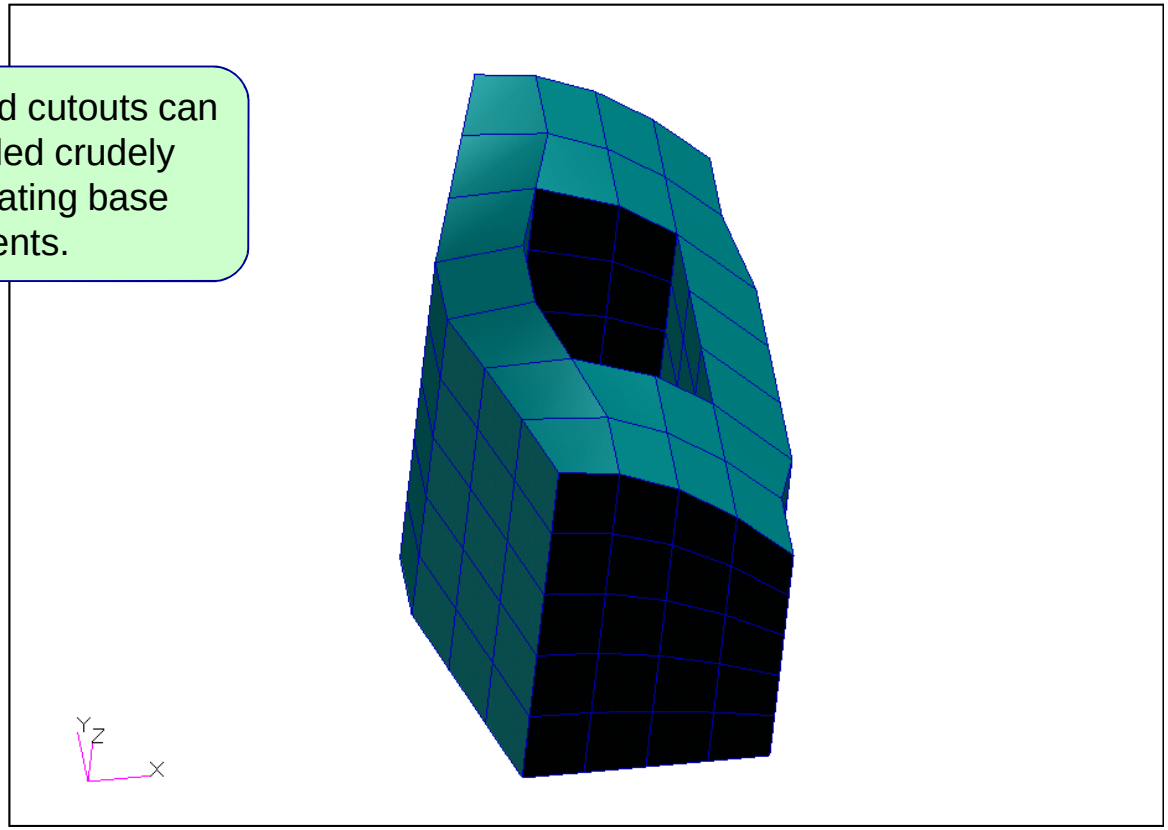


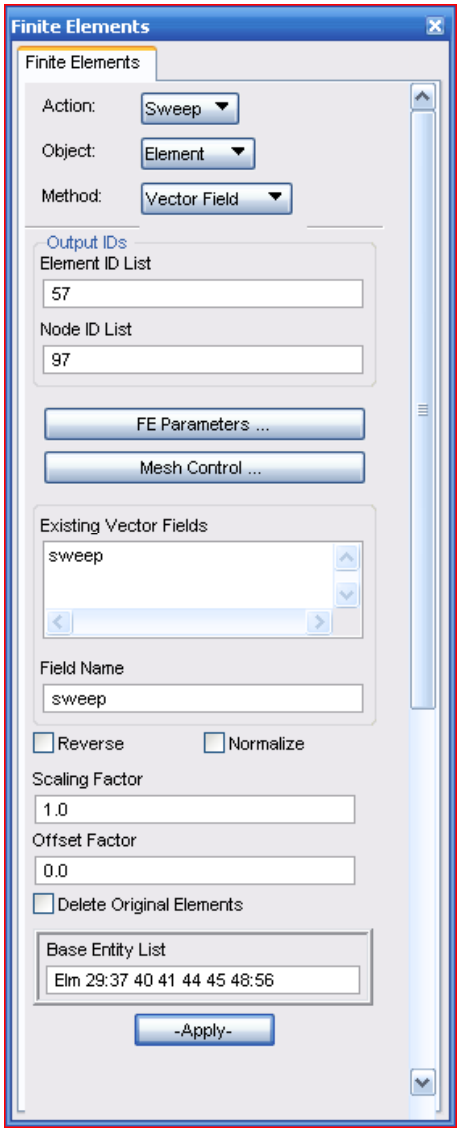
Copyright© 2012 MSC.Software Corporation

3D START POINT, 2 1/2D IDEALIZATION



Holes and cutouts can be modeled crudely by eliminating base 2D elements.



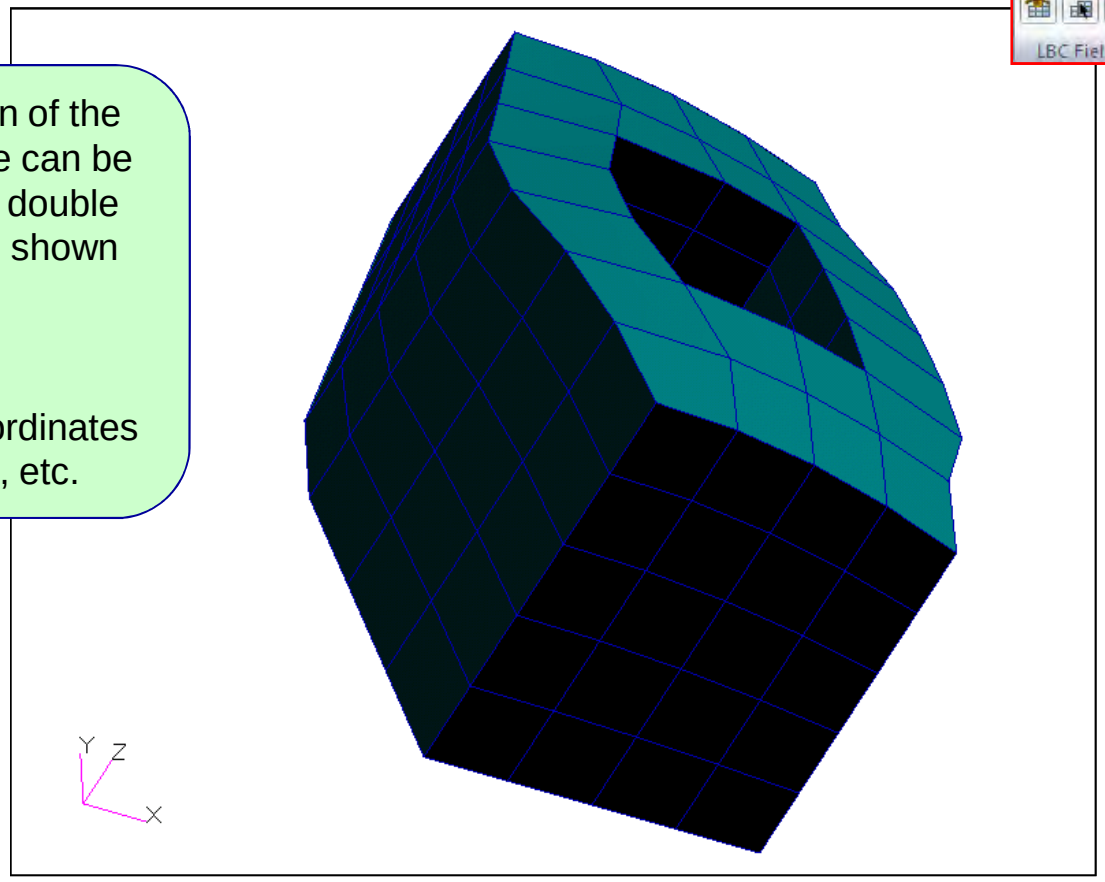


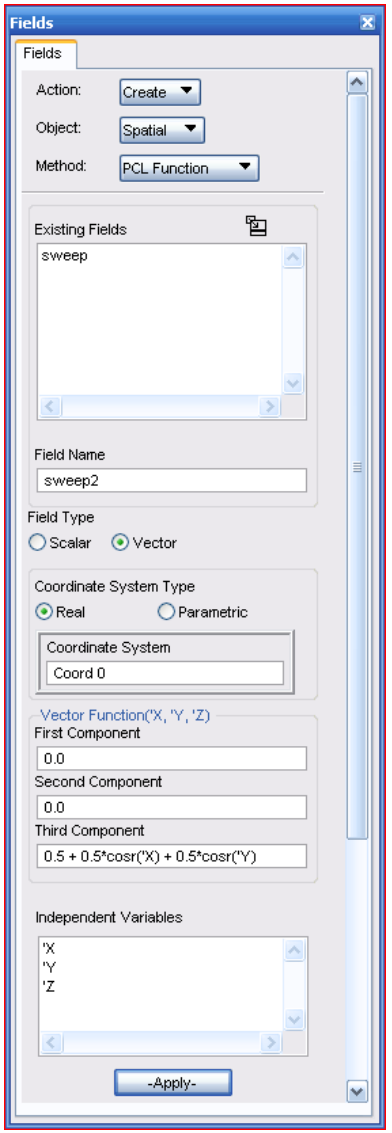
Copyright© 2012 MSC.Software Corporation

3D START POINT, 2 1/2D IDEALIZATION

Sophistication of the far mesh face can be increased by double curvature, as shown here.

Alternatively, spherical coordinates may be used, etc.

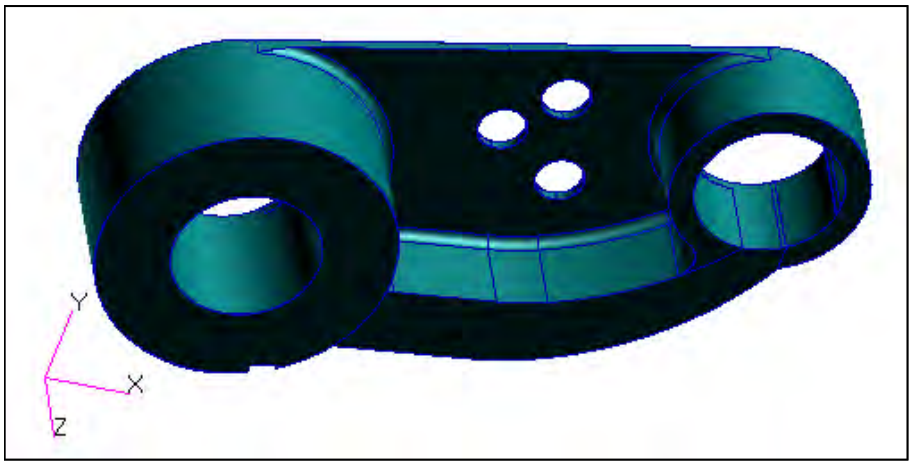


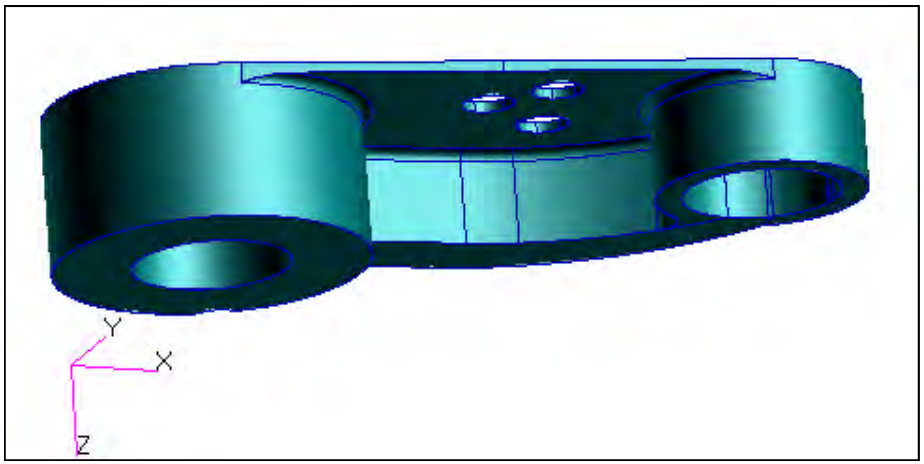


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

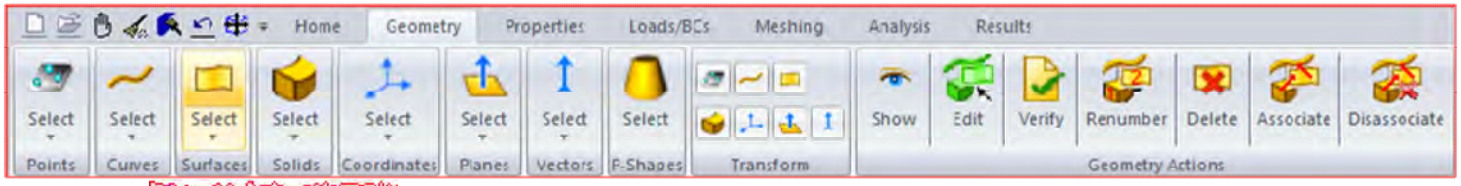
- **An example of the approach previously described is shown by using the CAD model (given below), which represents a caliper type structure.**
  - The boss regions will be modeled using 2 ½ D Hex mesh (swept 2D mesh)
  - The flange will be modeled using 2D plate elements





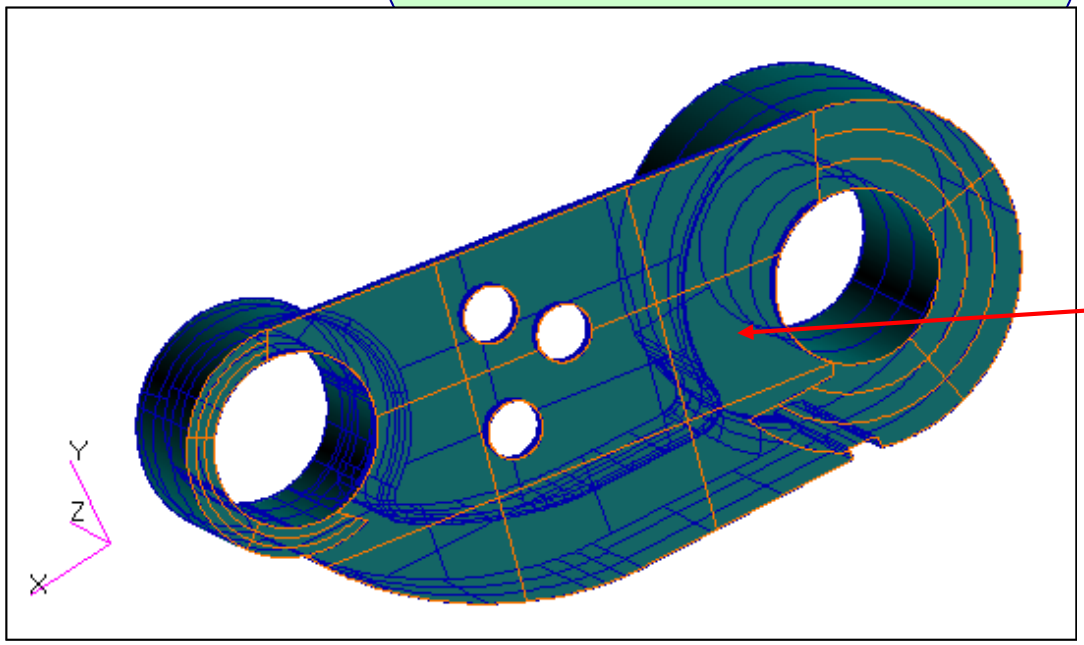
Copyright© 2012 MSC.Software Corporation

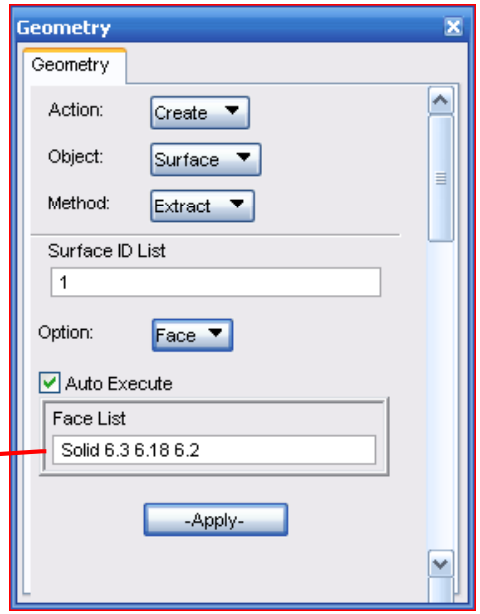
CASE STUDY: CALIPER 3D -> 2D/21/2D



The first step is to selectively extract surfaces from the faces of the B-rep solid.

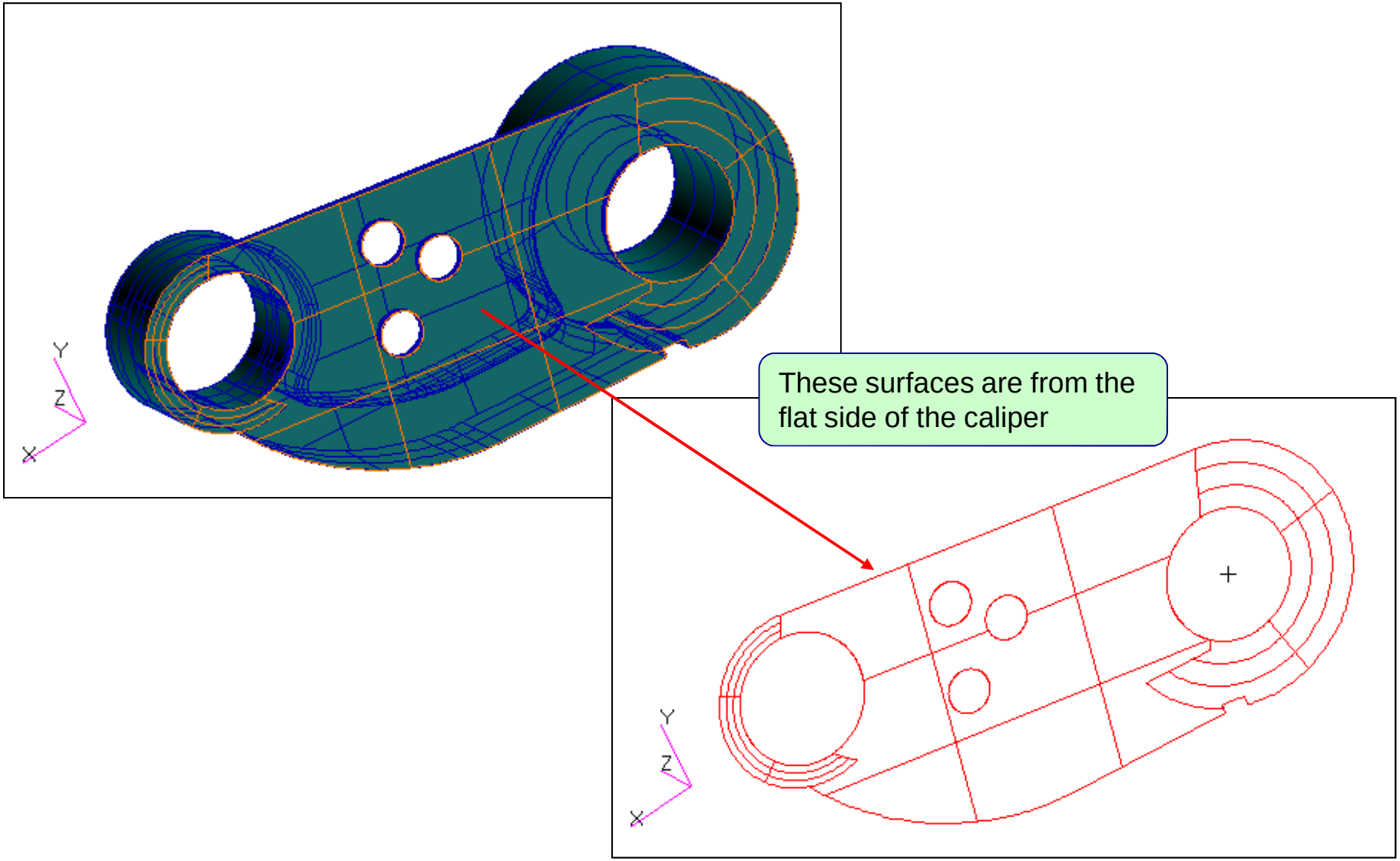
These will be used for further construction.





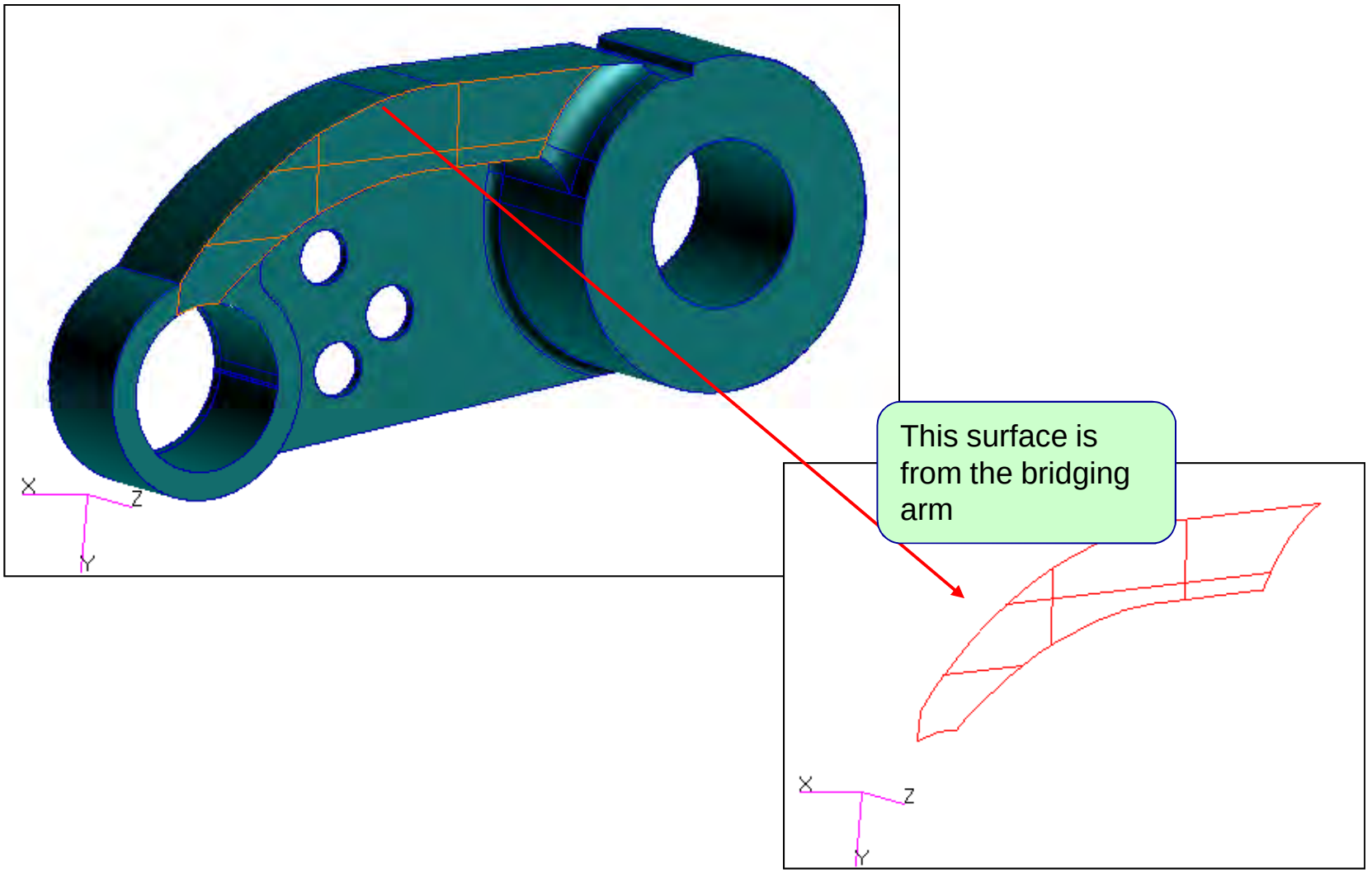
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D



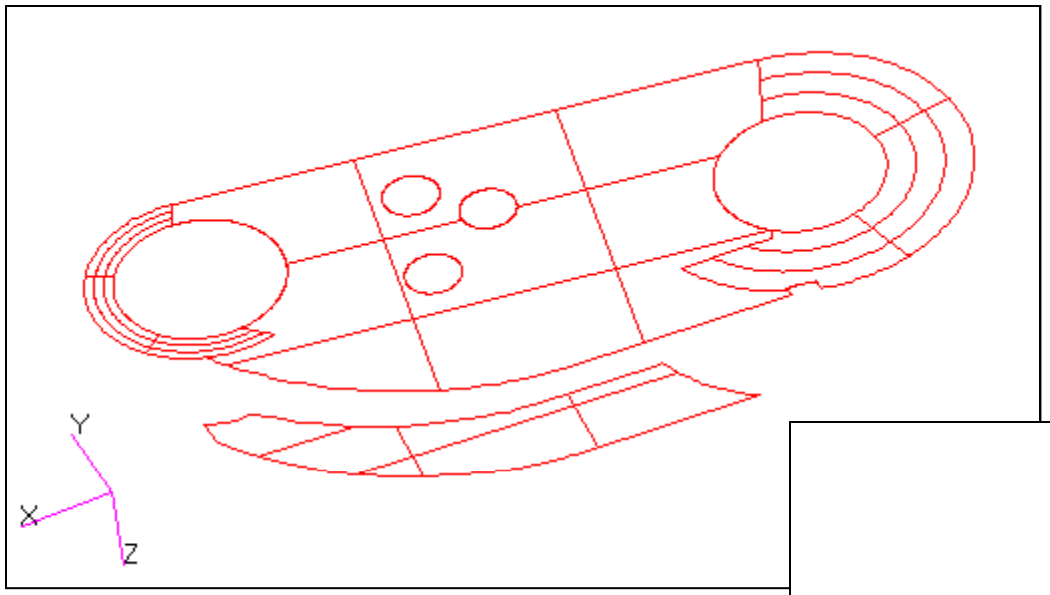
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

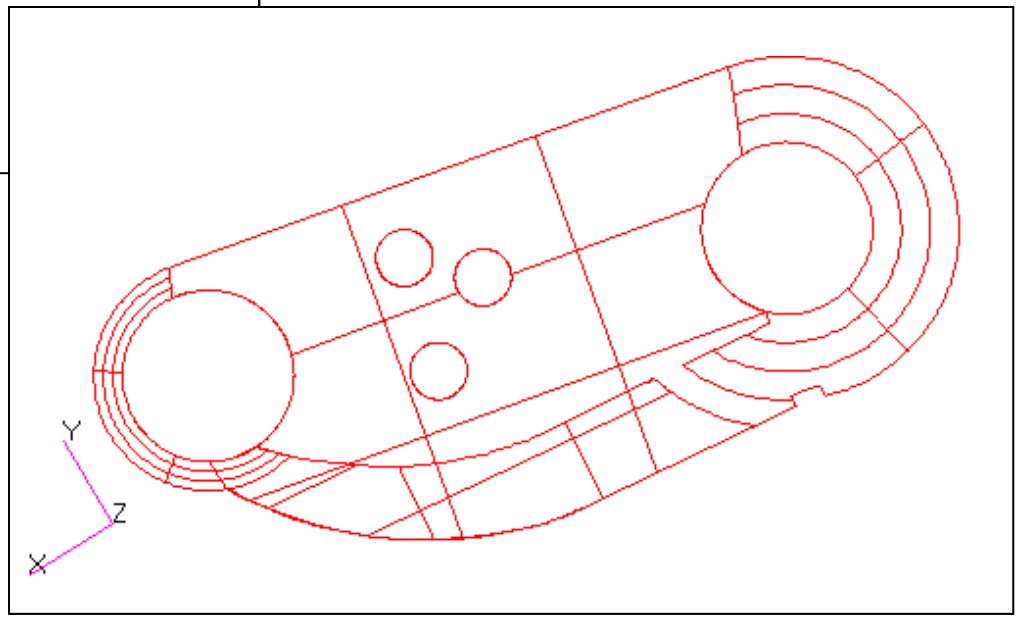


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

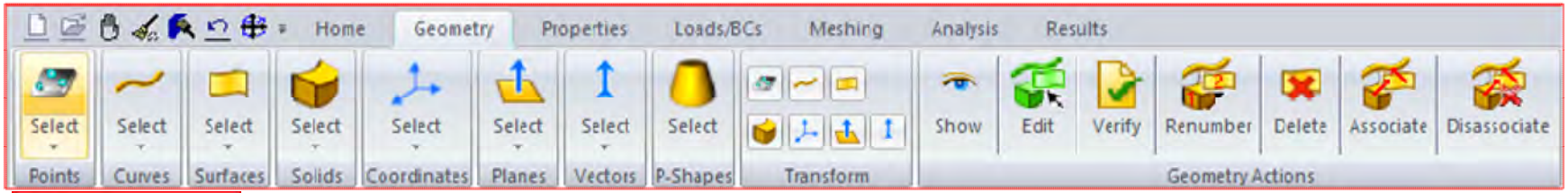


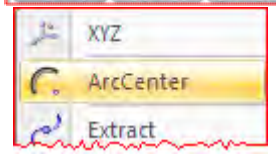
Illustrated here is the spatial relationship between the surfaces on the flat side of the caliper and the bridging arm.



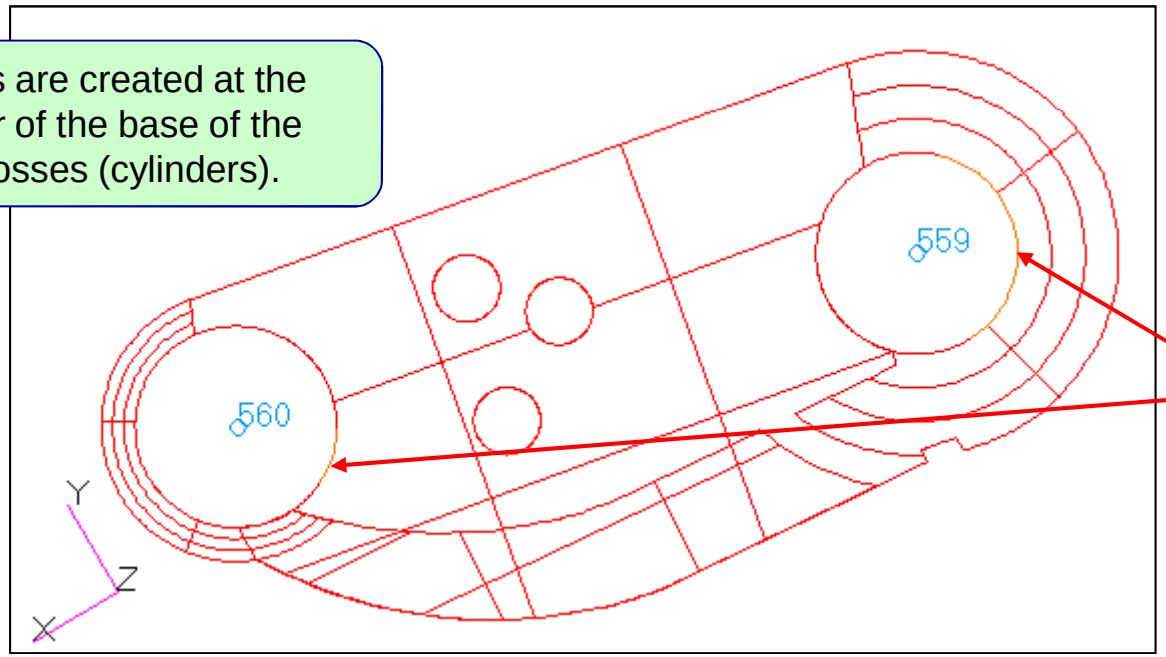
Copyright© 2012 MSC.Software Corporation

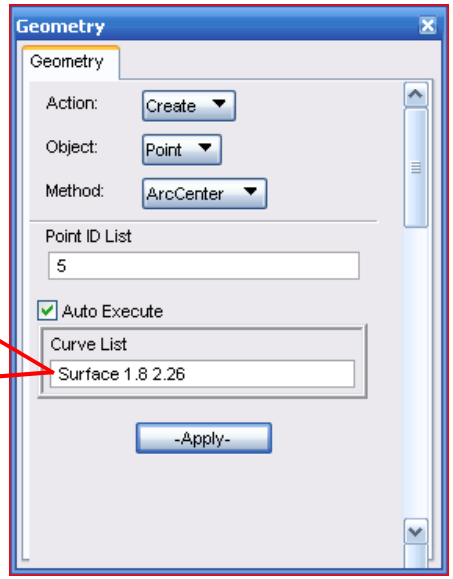
CASE STUDY: CALIPER 3D -> 2D/21/2D





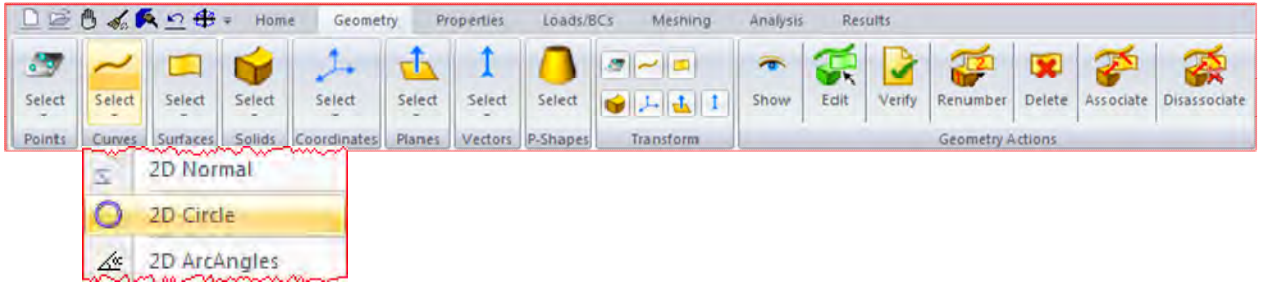
Points are created at the center of the base of the two bosses (cylinders).



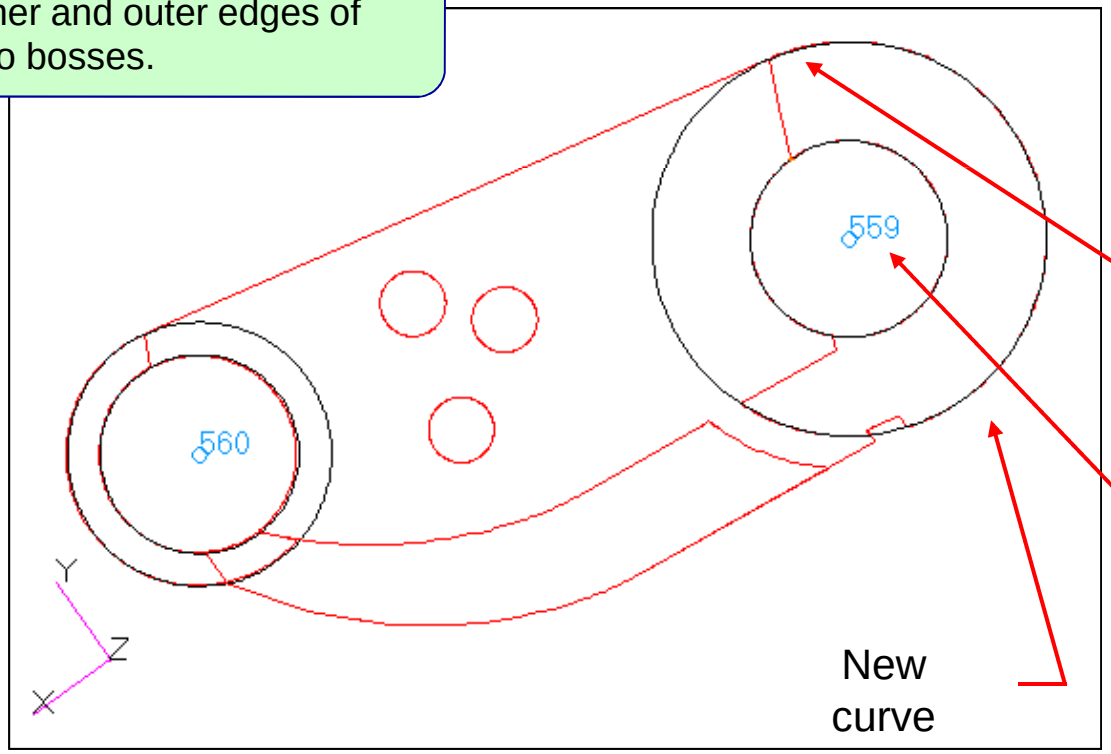


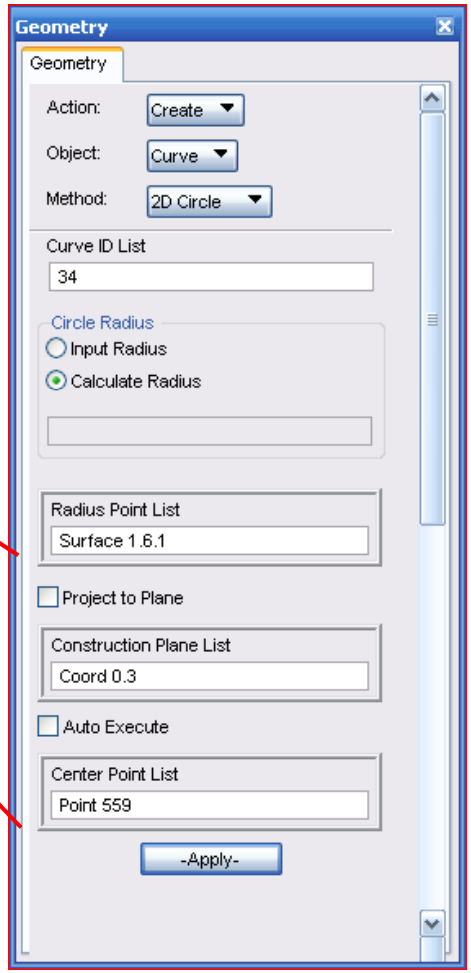
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D



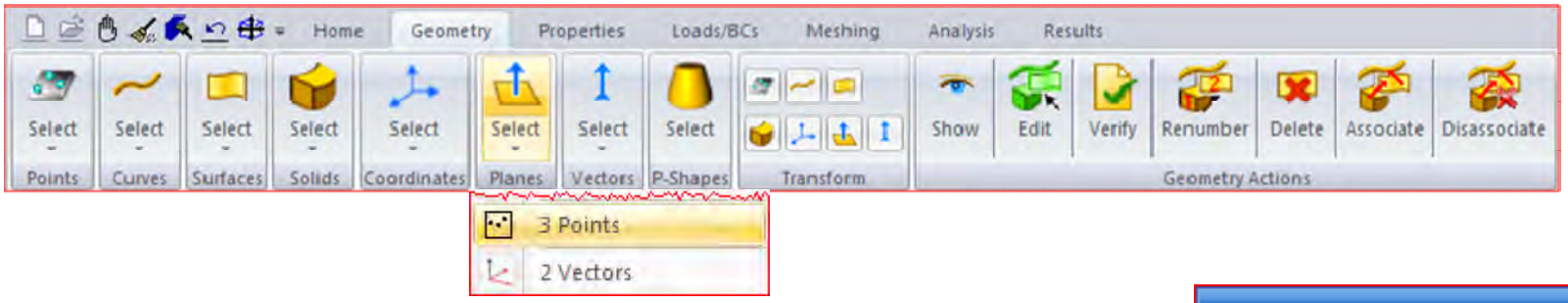
A circular curve is created at the inner and outer edges of the two bosses.



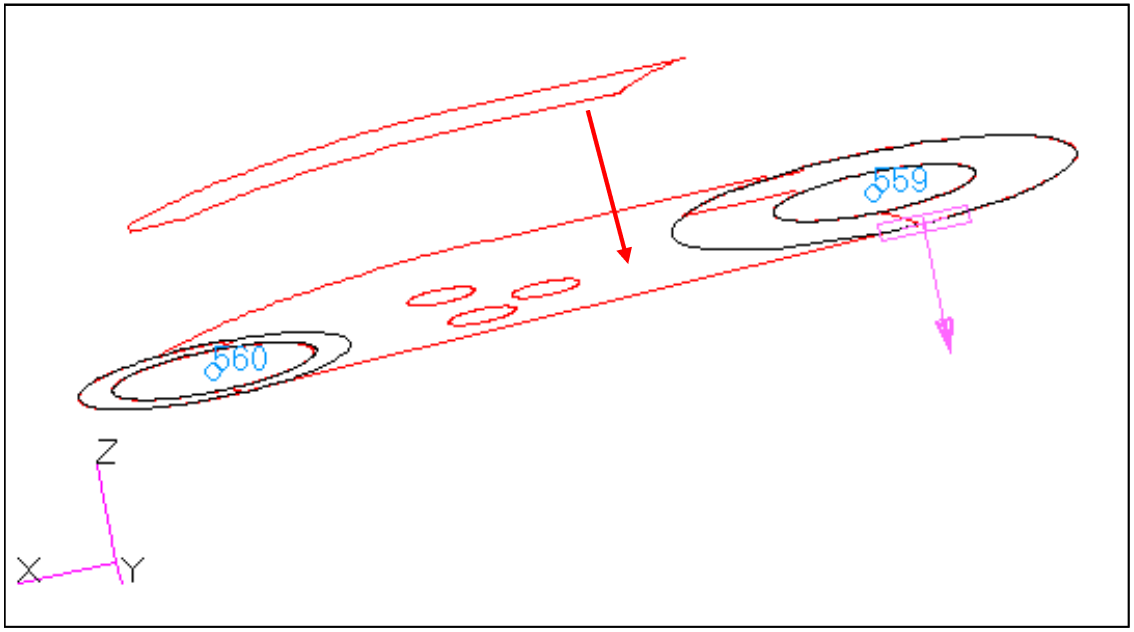


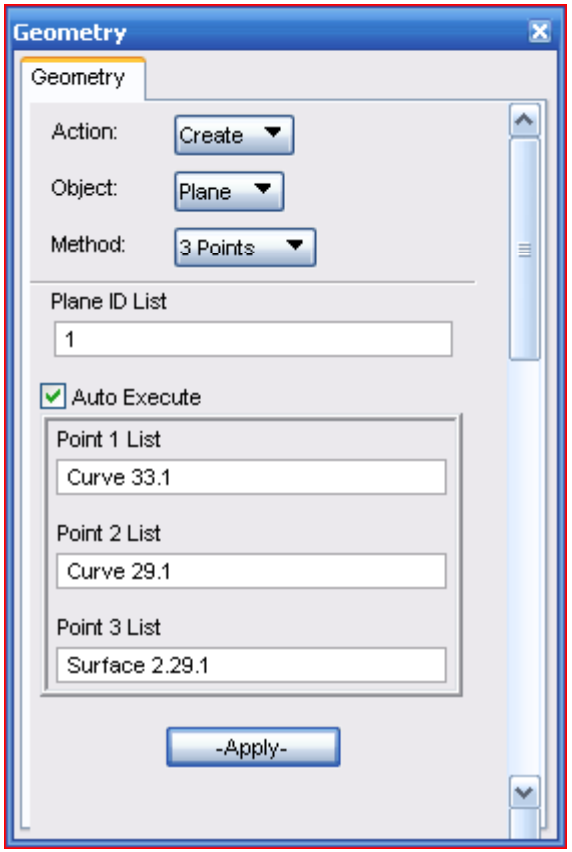
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D



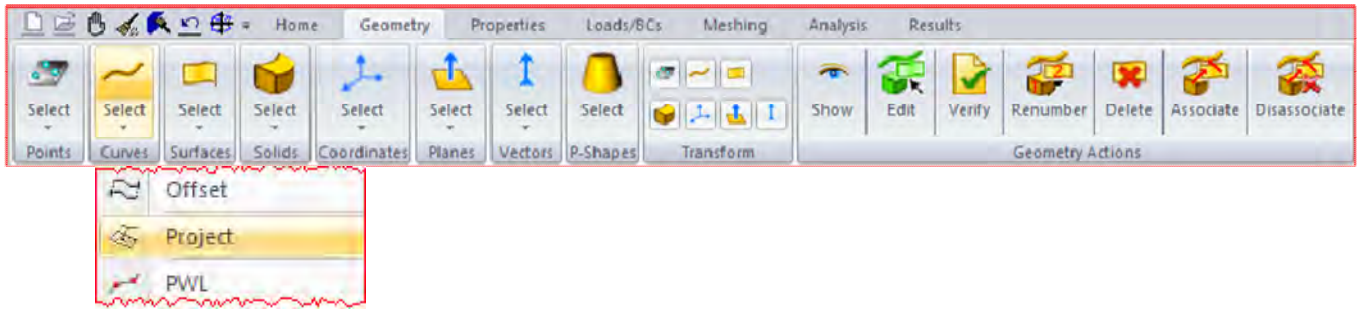
Create a plane along the surfaces from the flat side of the caliper (plane of trimmed surface with three holes). This Plane is to be used for projecting the edges of the bridge arm surface.



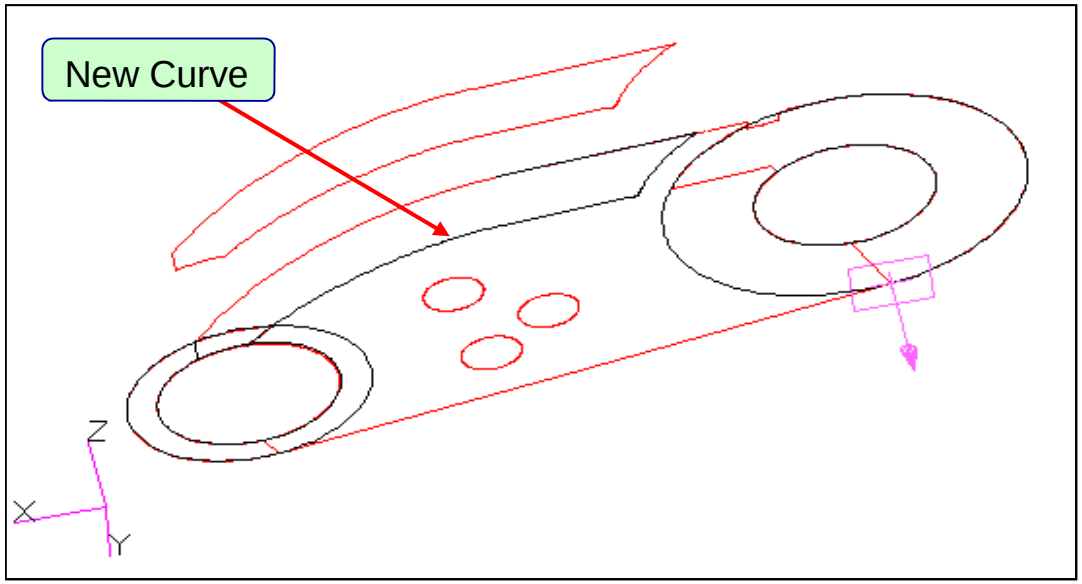


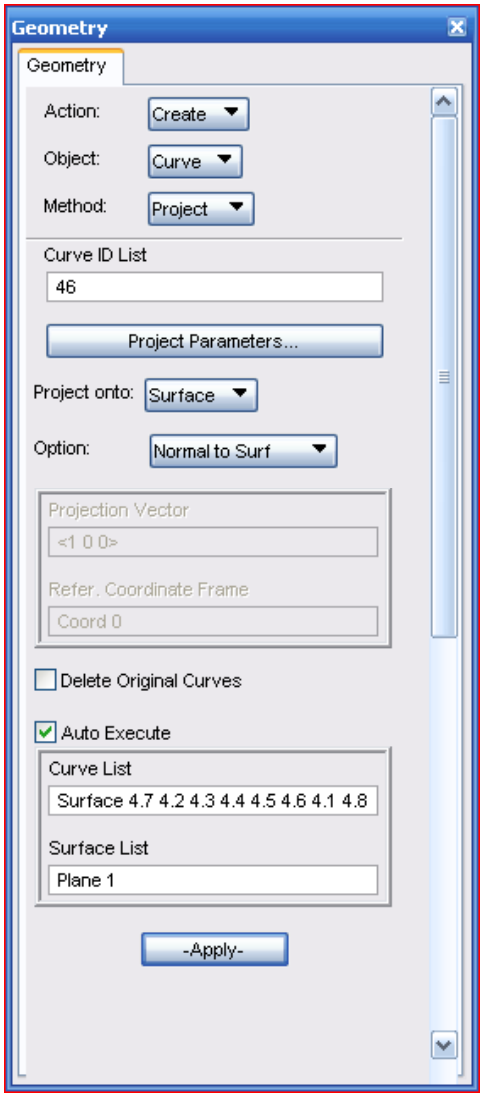
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D



The edges of the bridging arm surface are projected onto the Plane.

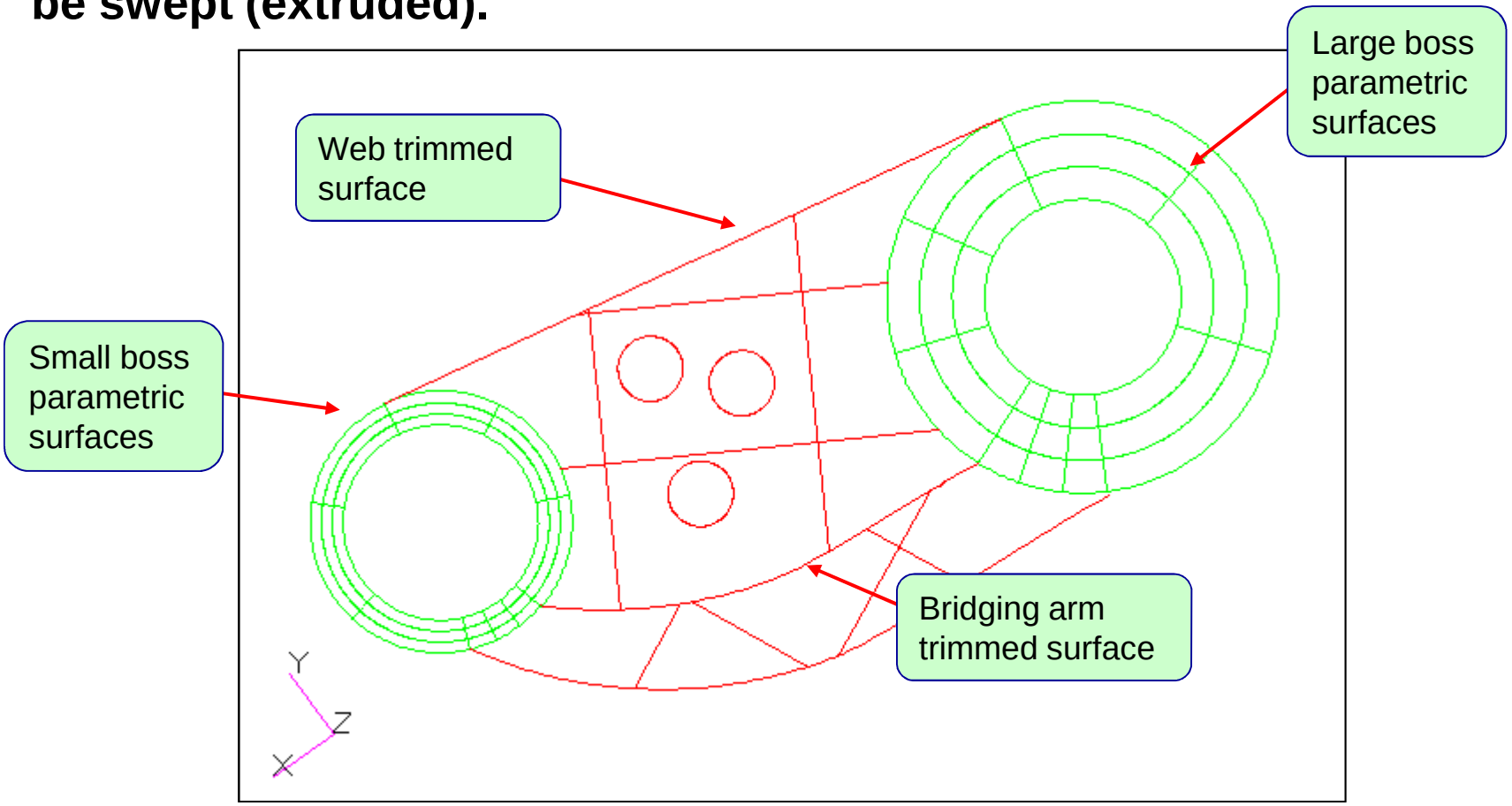




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

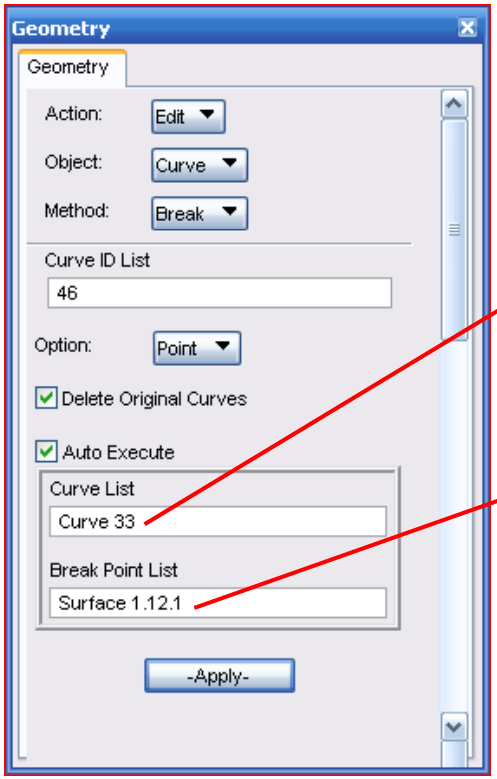
- **The steps described in the following slides lead to the creation of a set of surfaces that can be 2D meshed. Then the 2D meshes can be swept (extruded).**

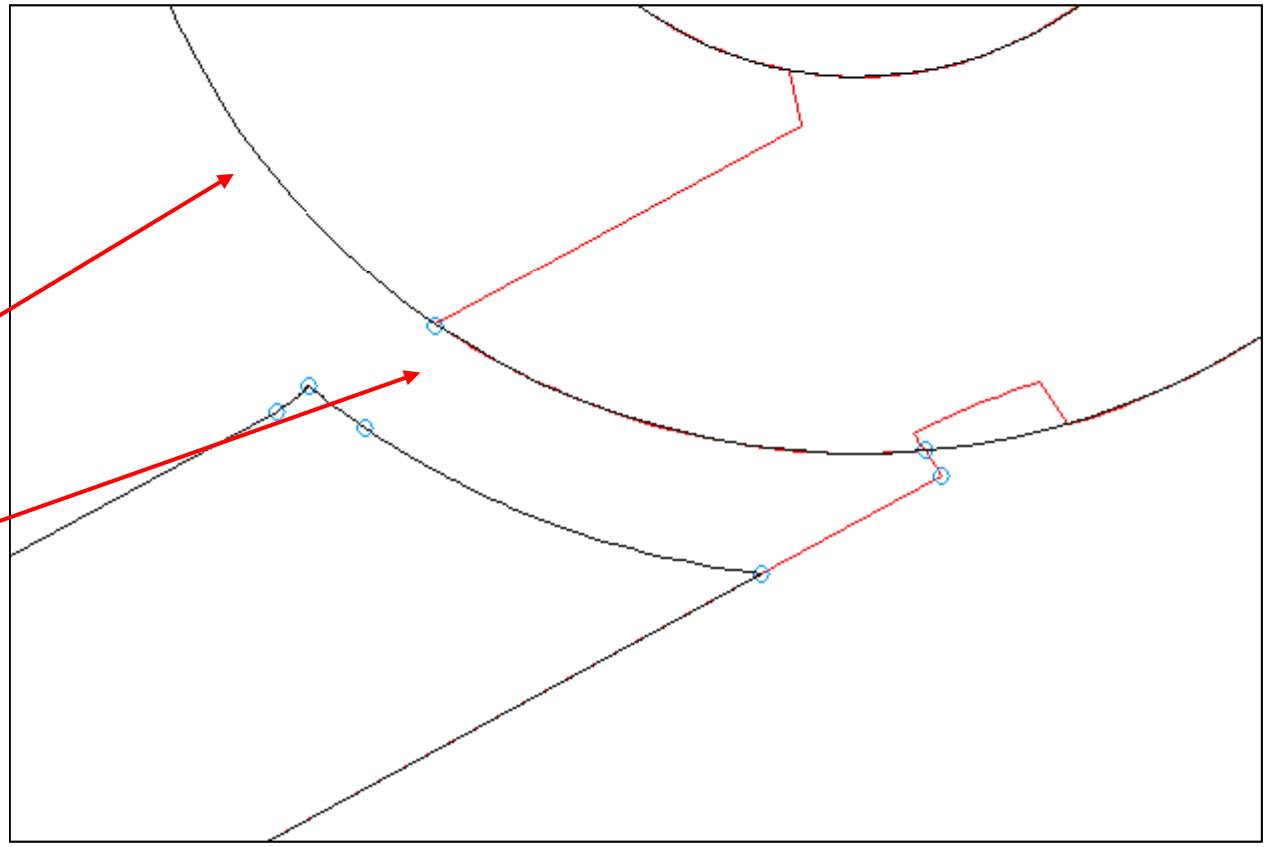


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

- **Breaking the curve is part of the process of creating a chain curve, which will then be used to create a trimmed surface.**

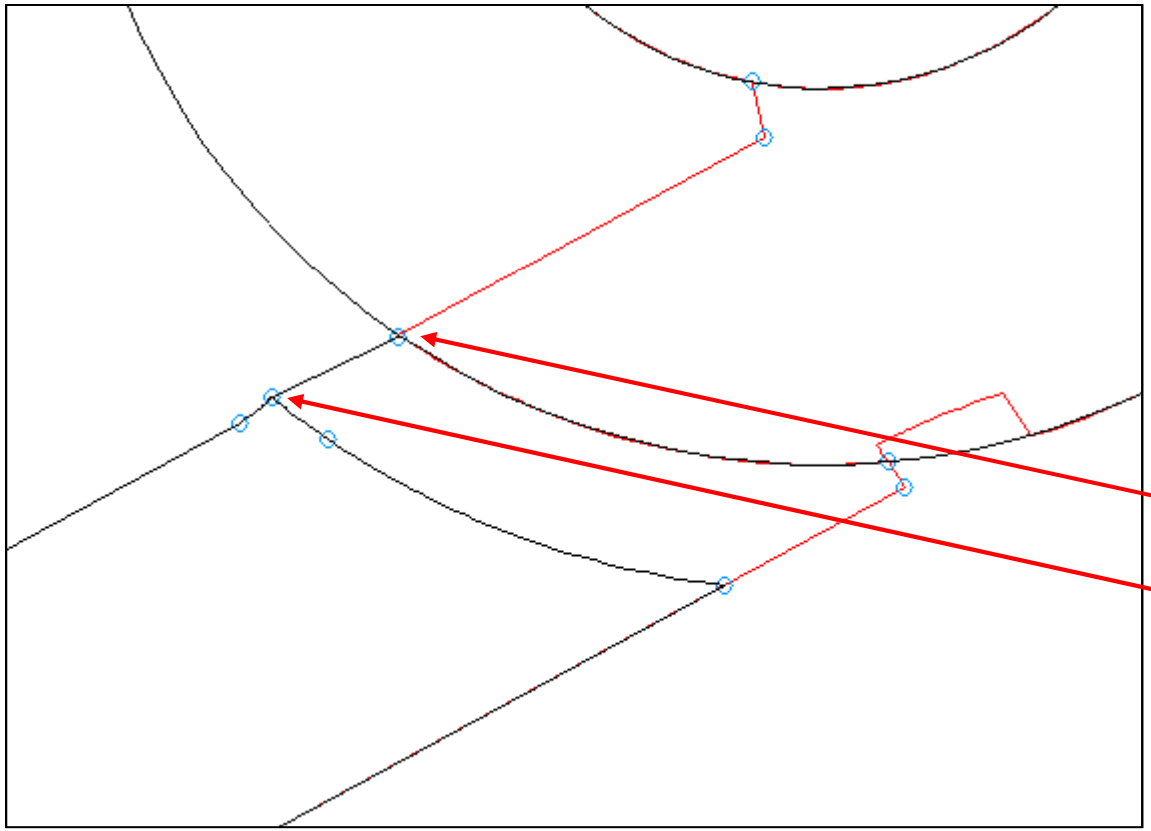


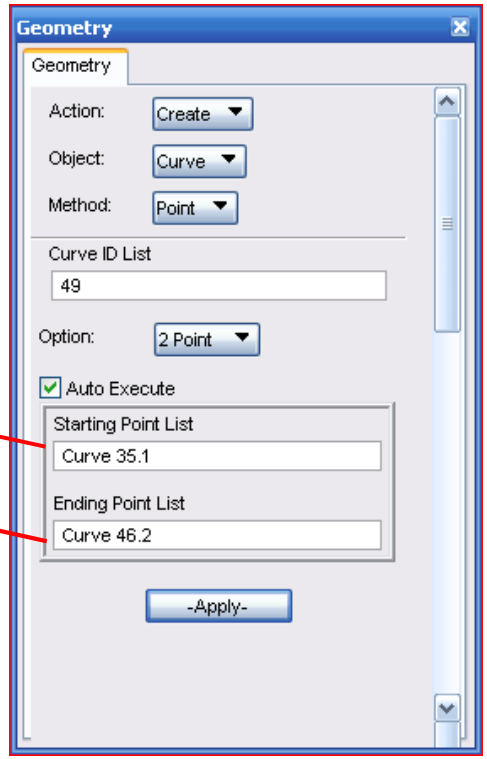


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

- **Create a curve using two vertices. This curve will then be used to create two trimmed surfaces, which will be 2D Paver meshed.**

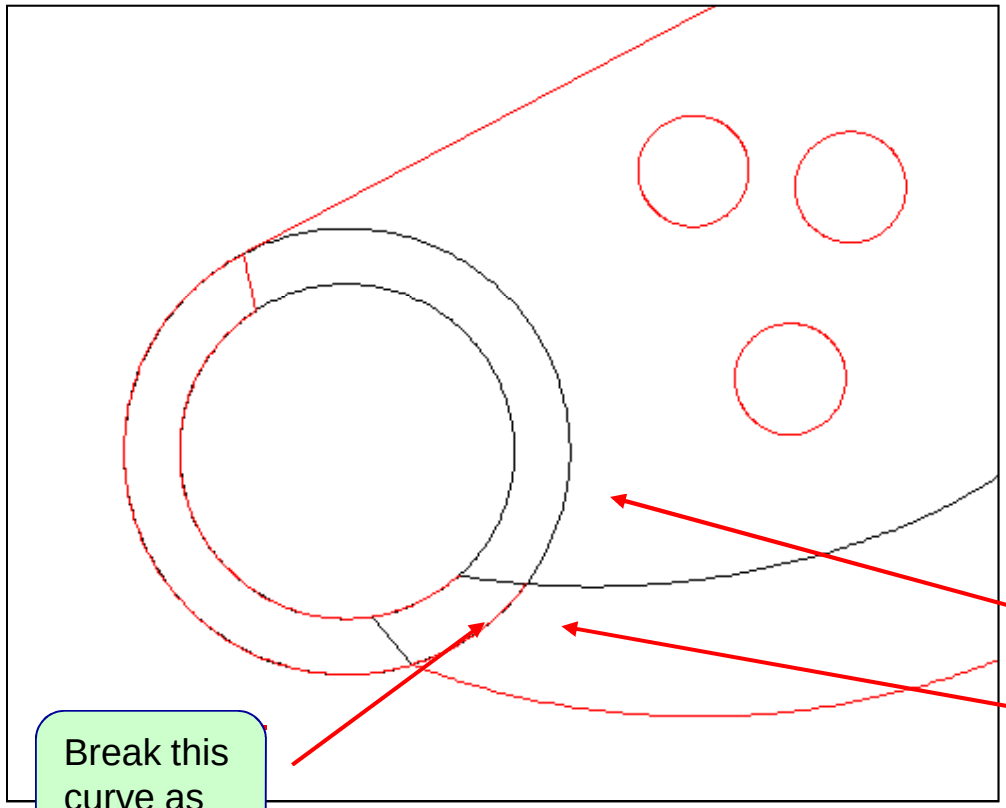


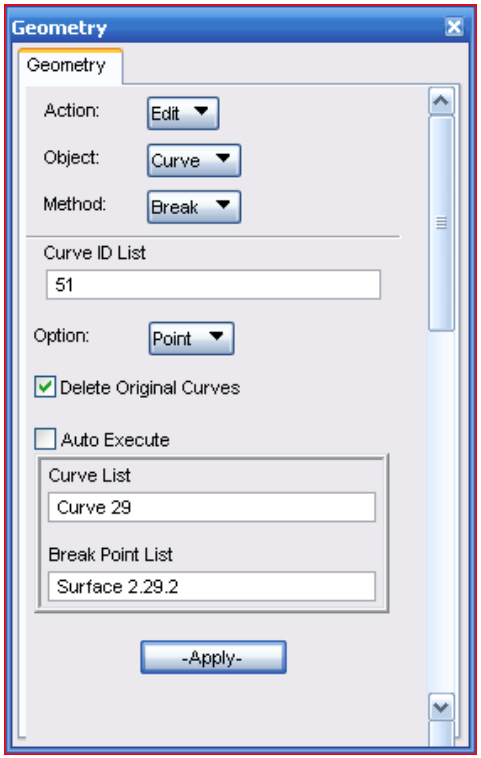


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

- **Break two additional curves as part of creating a chain curve, which will be used to create a trimmed surface.**

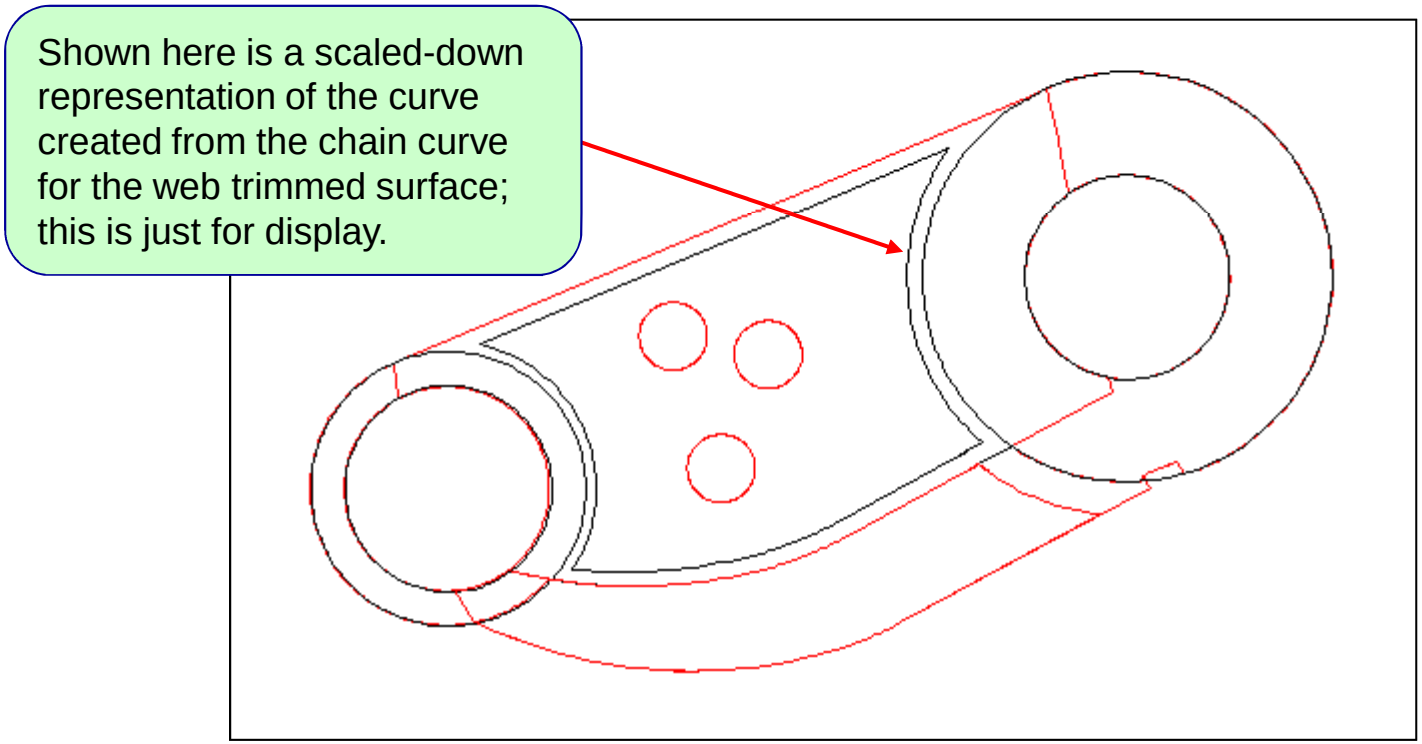




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

- **Create a chain curve using the separate but congruent curves/edges where the web with three holes is located.**

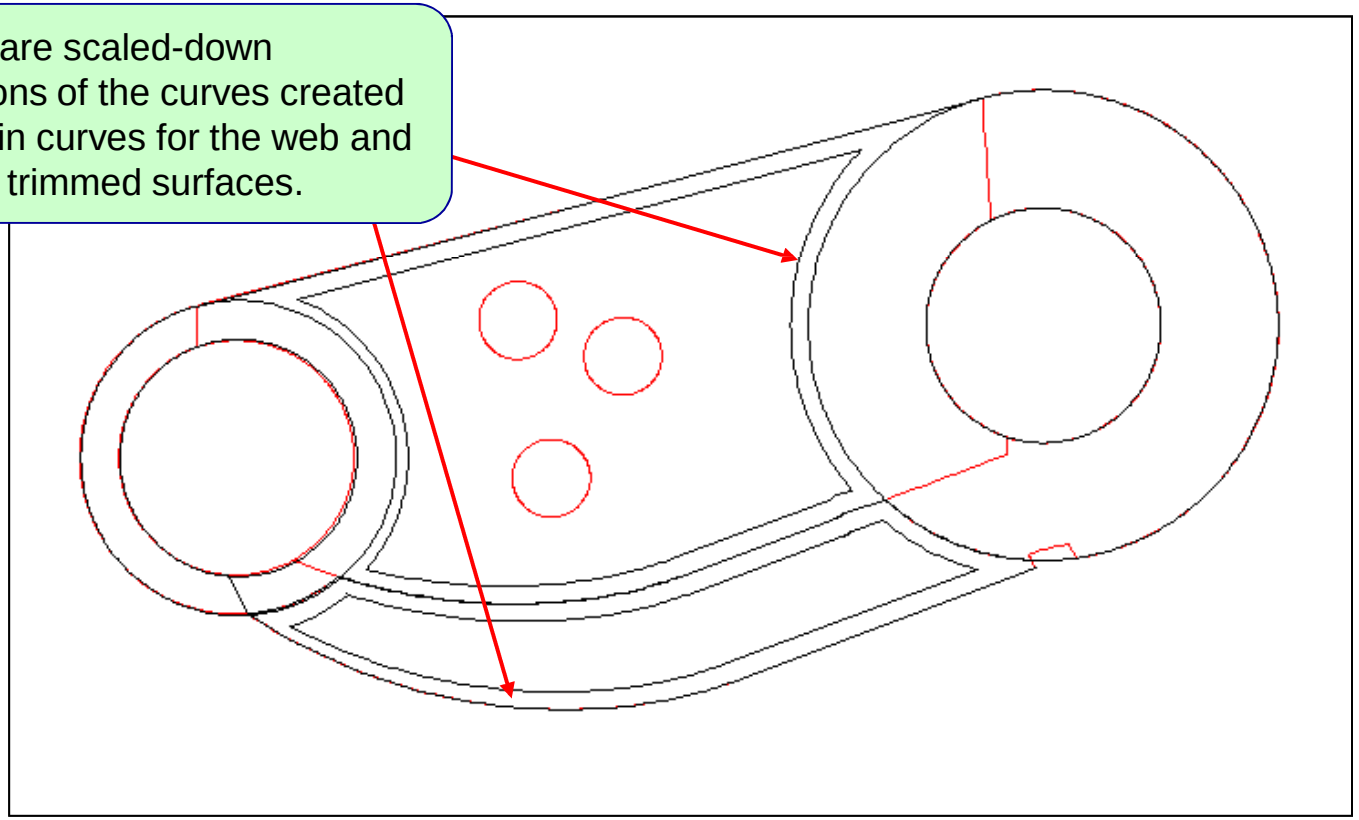


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

- **A chain curve is created for both the web (with three holes) and the area/region of projection of bridging arm on the plane.**

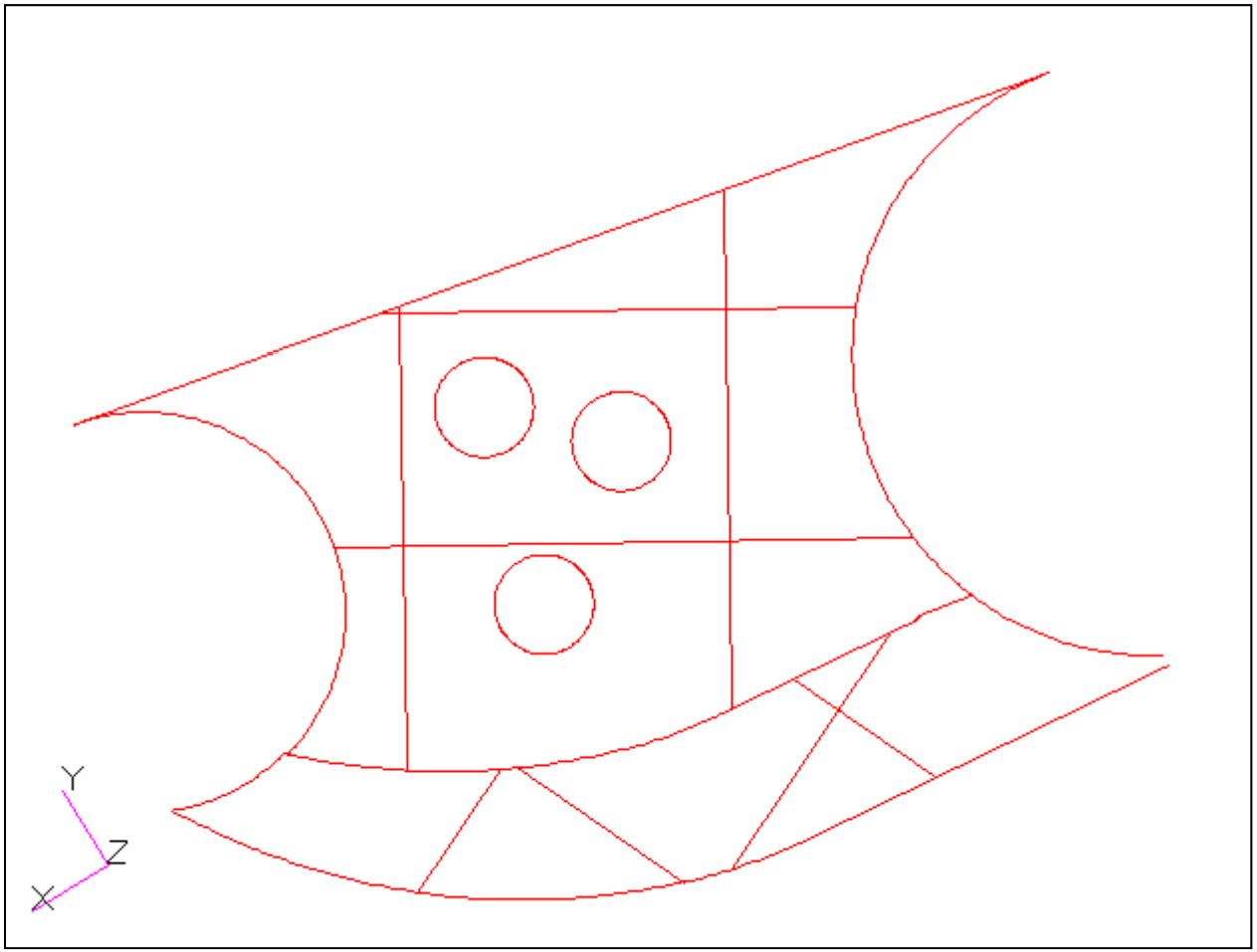
Shown here are scaled-down representations of the curves created from the chain curves for the web and bridging arm trimmed surfaces.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

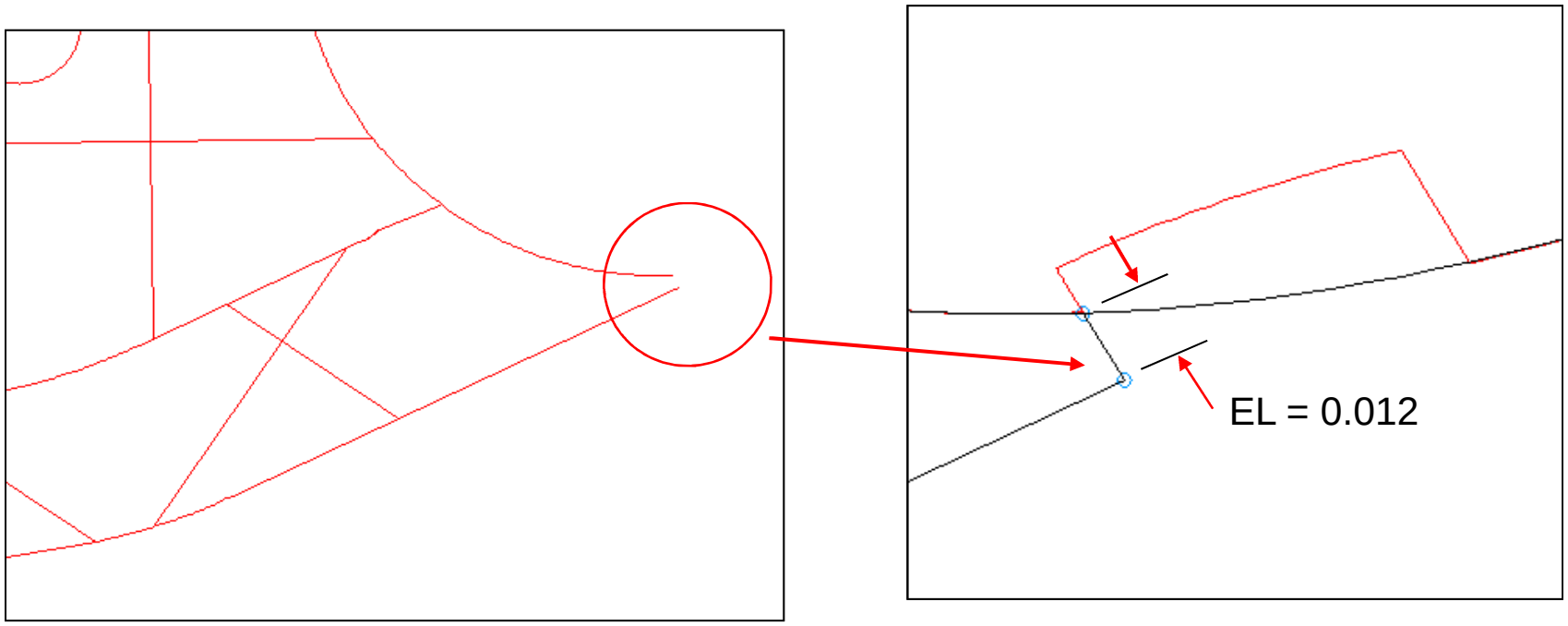
- **The two trimmed surfaces are created**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

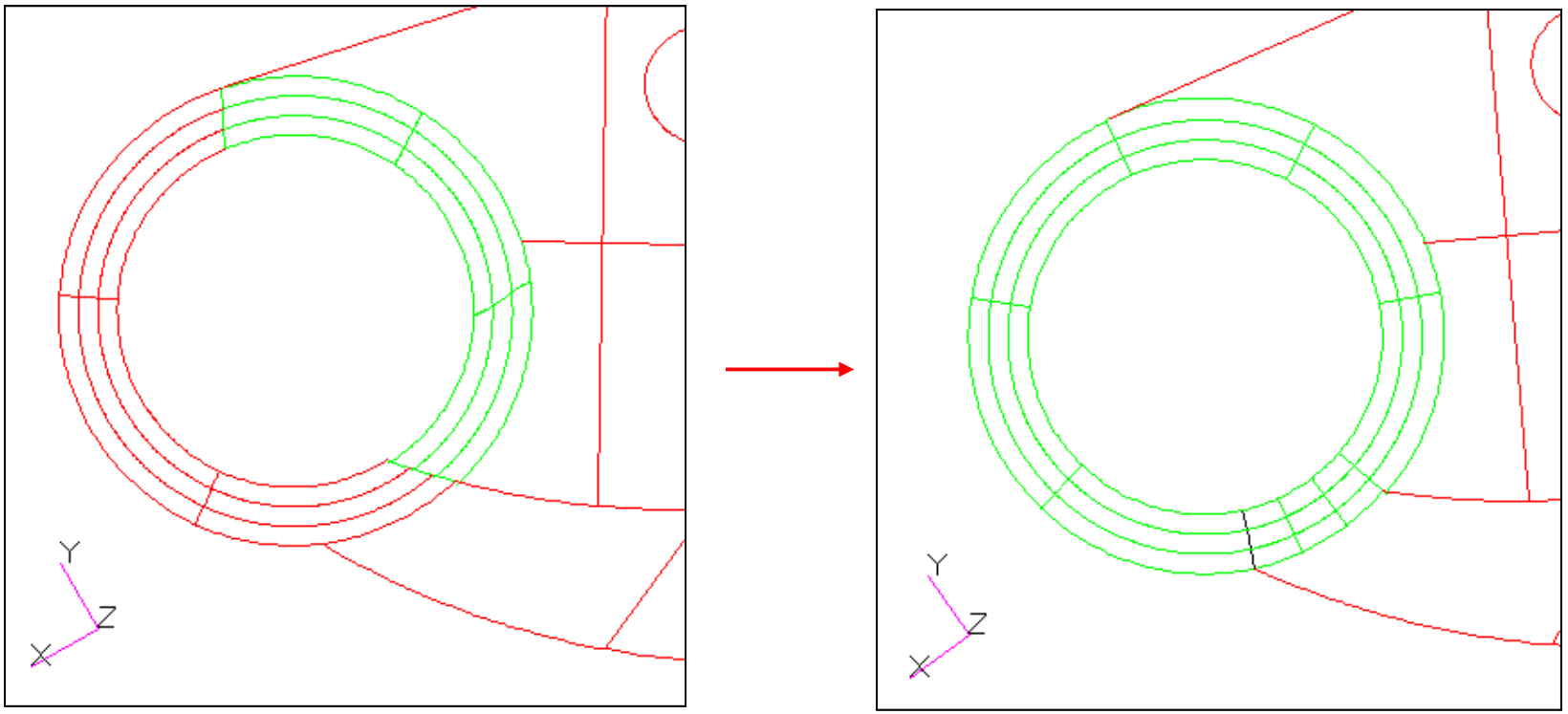
- **Note: There is a missing edge in the display of the bridging arm trimmed surface; edge length (EL) = 0.012 < GMT = 0.015 => the edge is degenerate.**
- **It is still possible to create a chain curve and trimmed surface.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

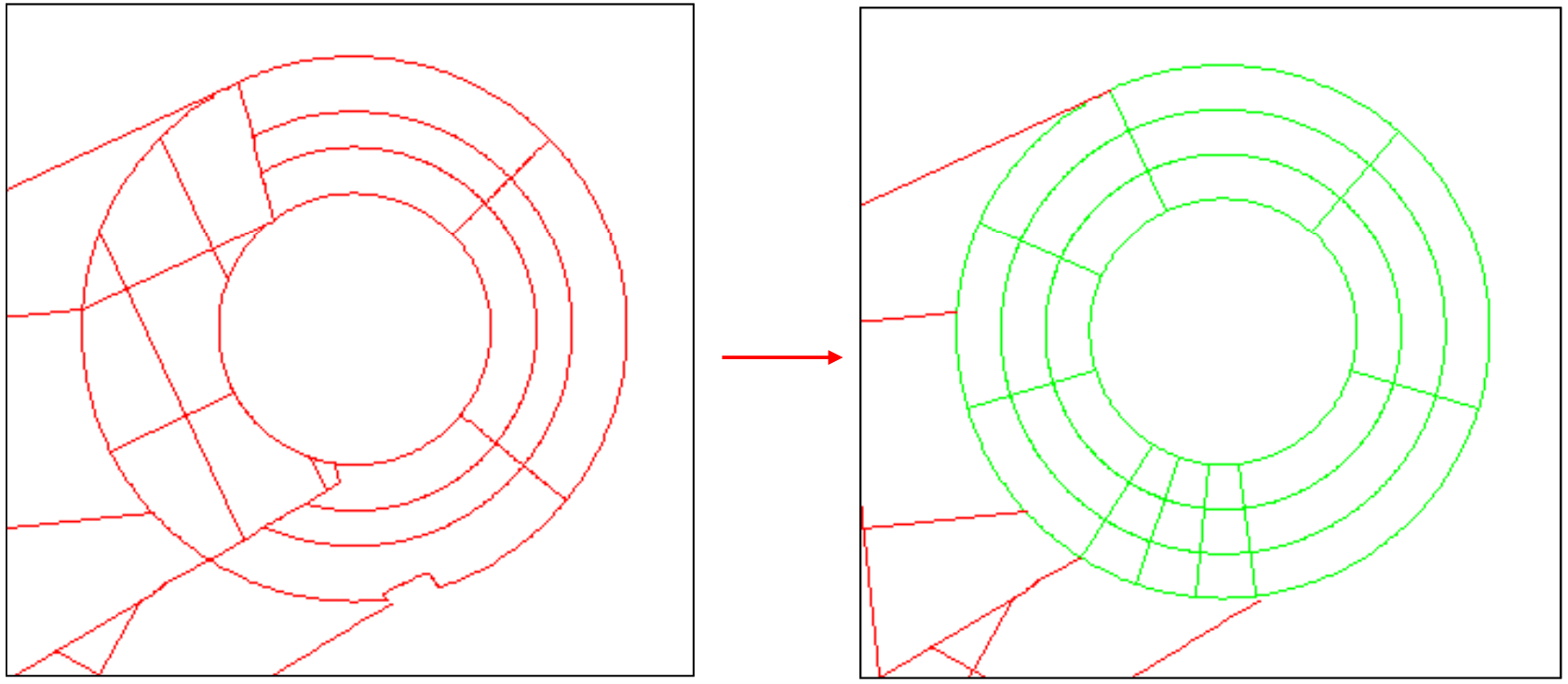
- **It is useful to clean up/recreate the surfaces that represent the base of the bosses, so the surfaces are better shaped and congruent. Shown here is the smaller boss:**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

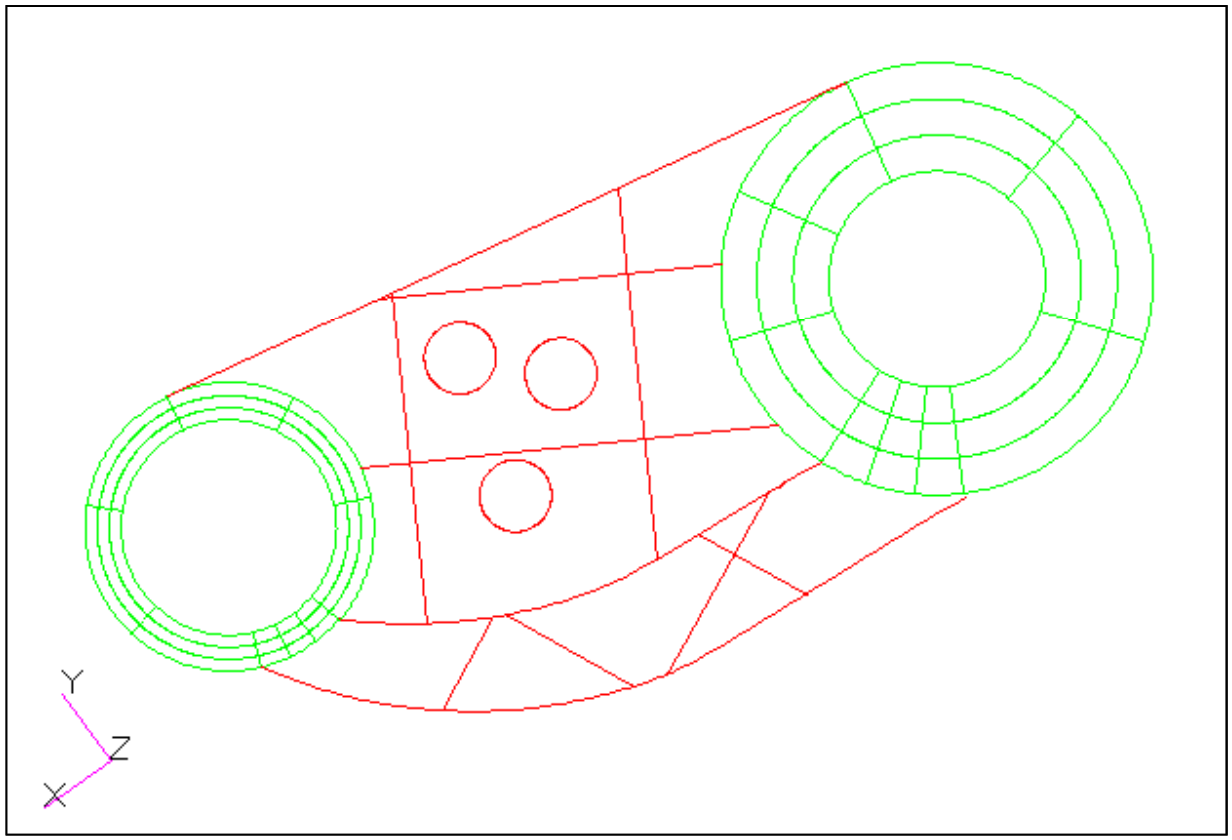
- **Shown below is the larger boss**



Copyright© 2012 MSC.Software Corporation

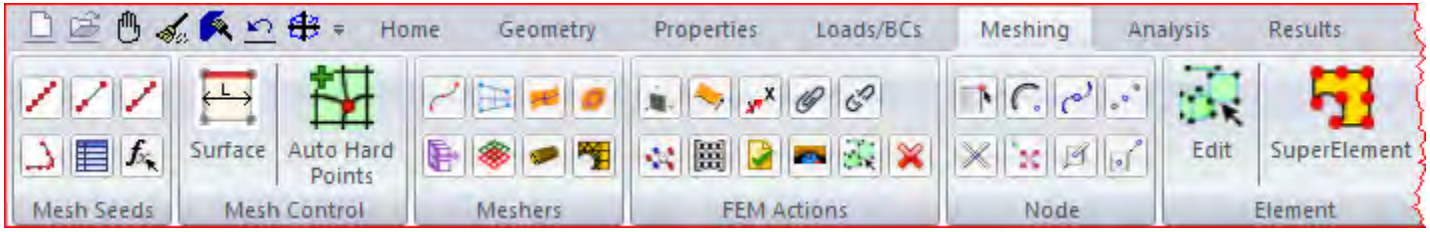
CASE STUDY: CALIPER 3D -> 2D/21/2D

- **The completed surfaces (trimmed or parametric) for the model are shown below**



Copyright© 2012 MSC.Software Corporation

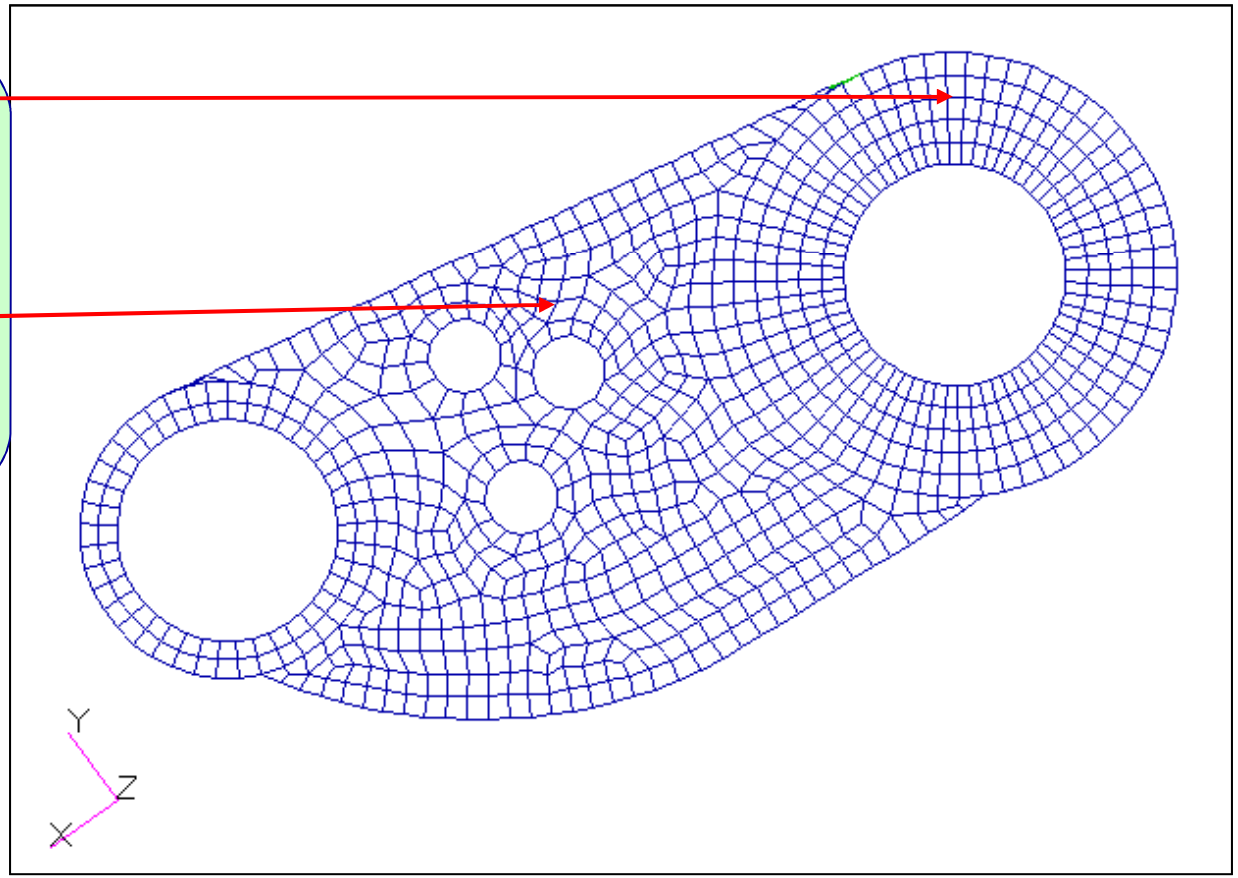
CASE STUDY: CALIPER 3D -> 2D/21/2D



The bases of the bosses were IsoMesh-ed

The web and arm were Paver meshed

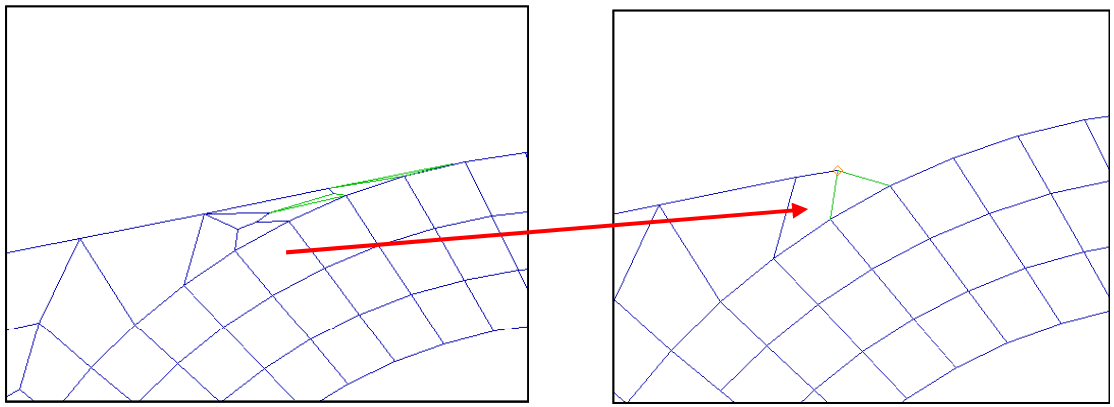
The resulting 2D mesh is as shown

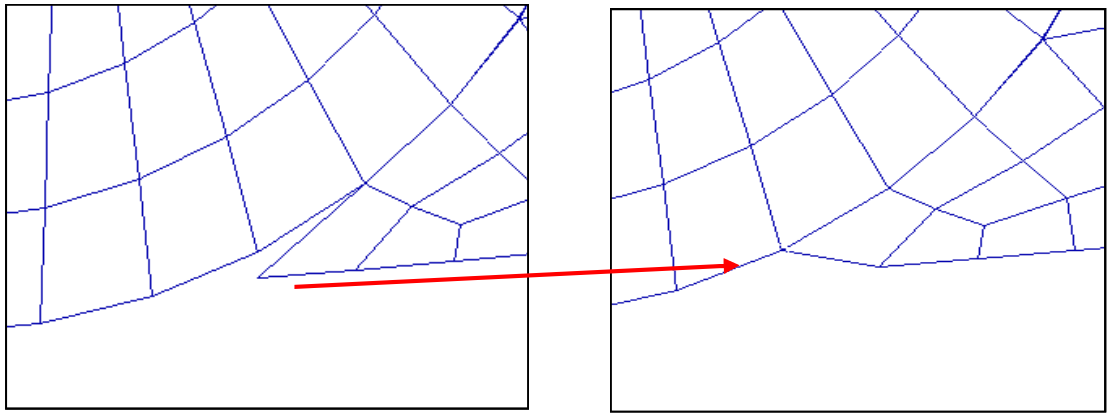


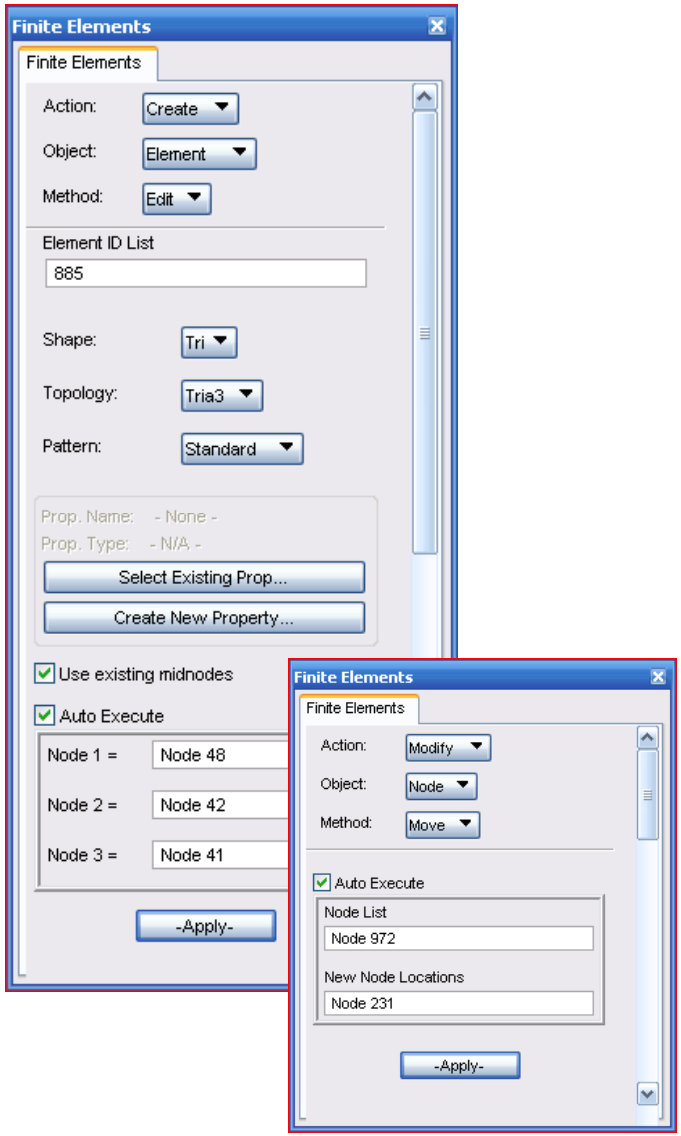
Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

- **Corrections need to be made to details of the mesh, i.e.**



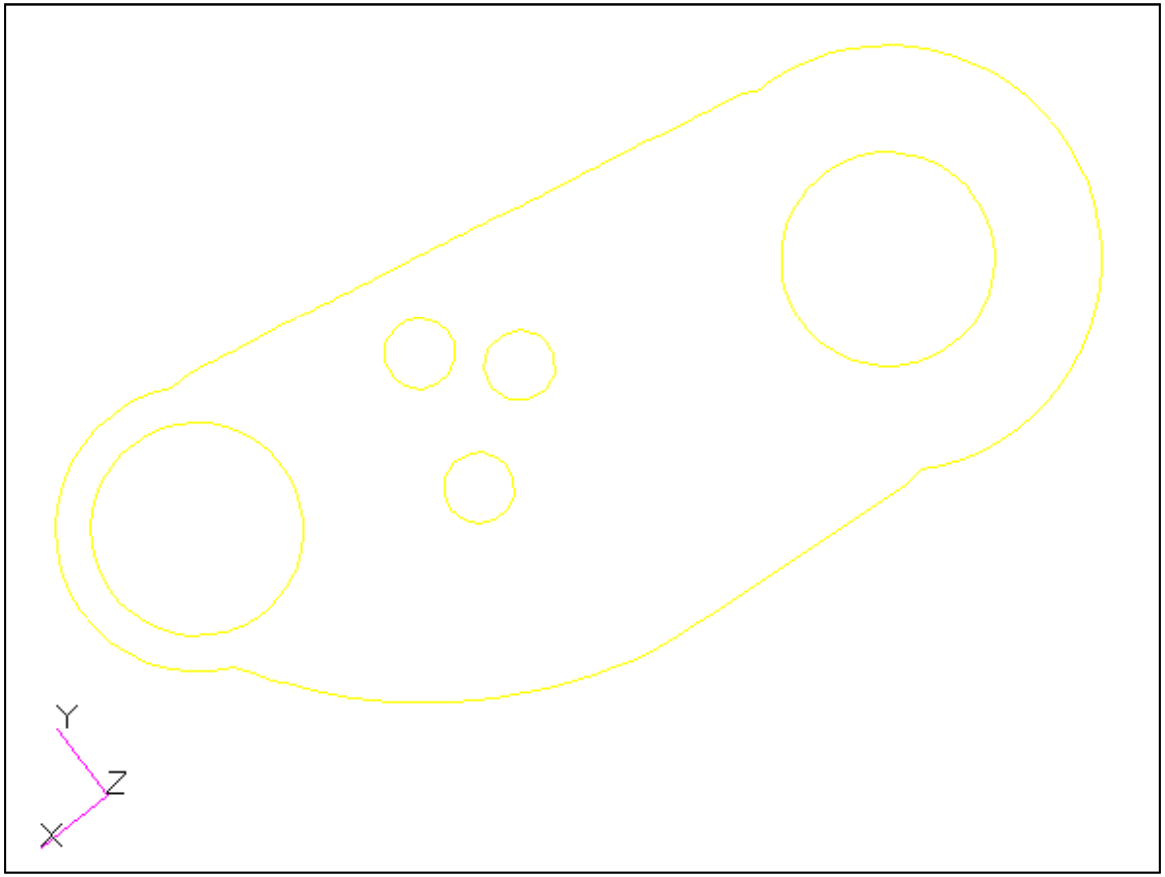




Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

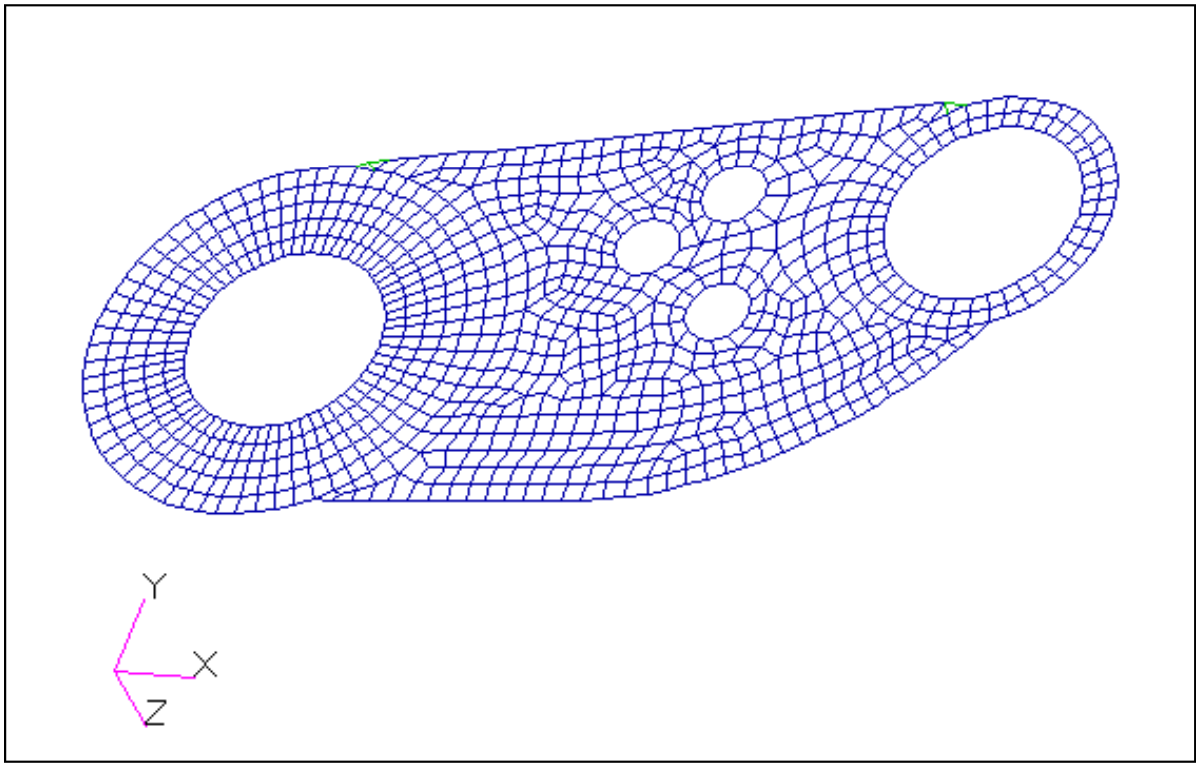
- **Check the 2D mesh for any unconnected elements.**

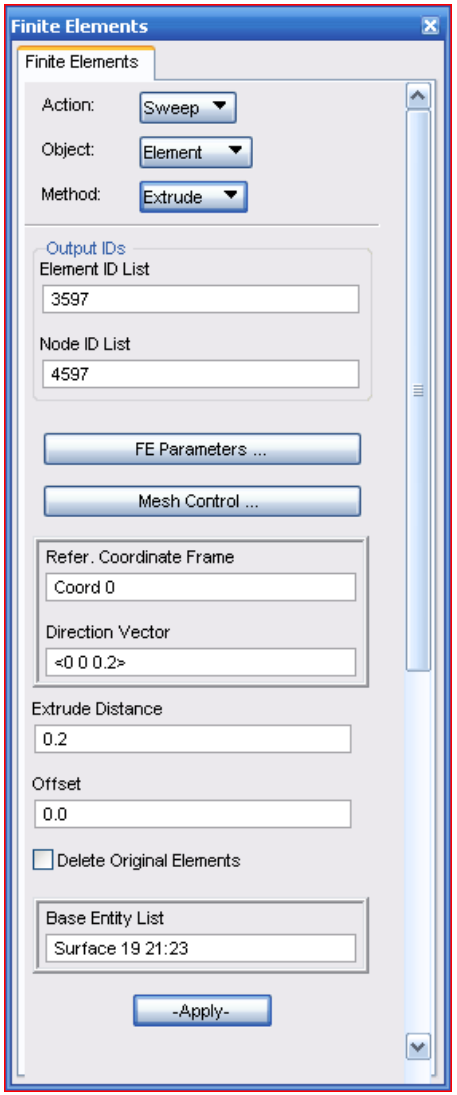


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

- **The final 2D mesh is used for the extrusion, except that for the web.**

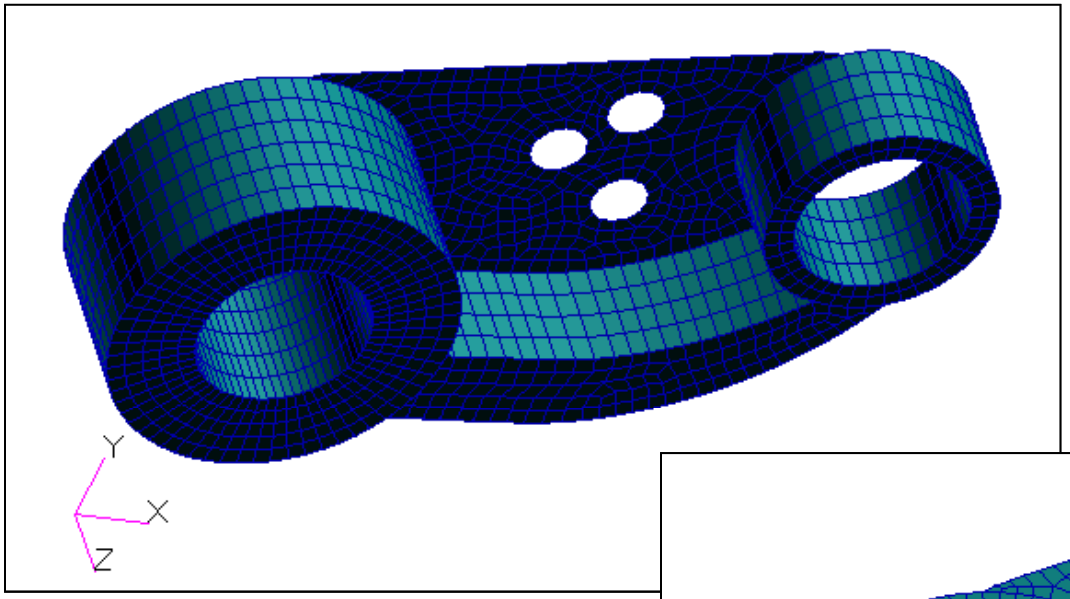


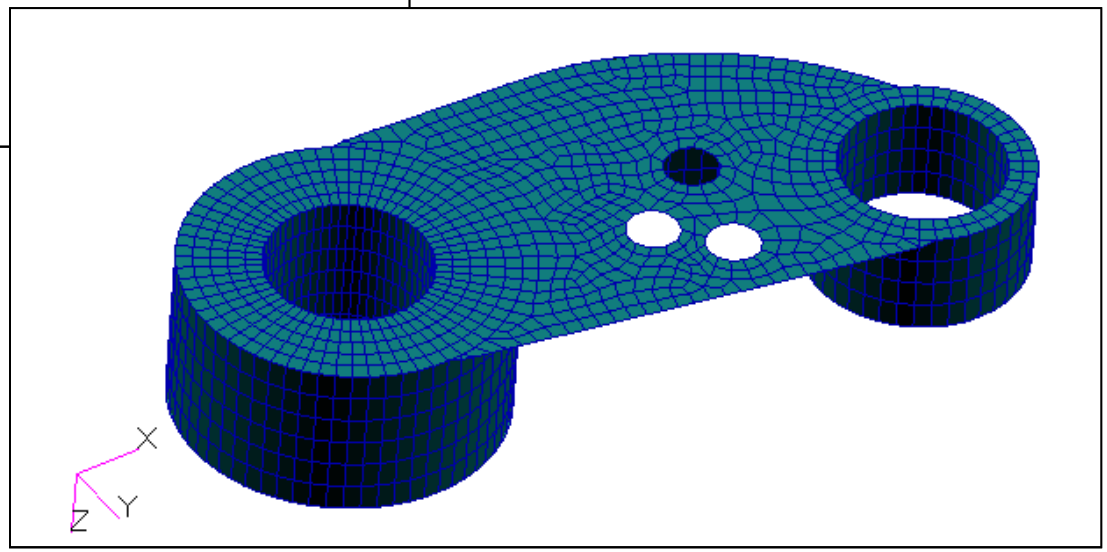


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CALIPER 3D -> 2D/21/2D

- **The final 2D/3D mesh for the caliper**





Copyright© 2012 MSC.Software Corporation

EXERCISES

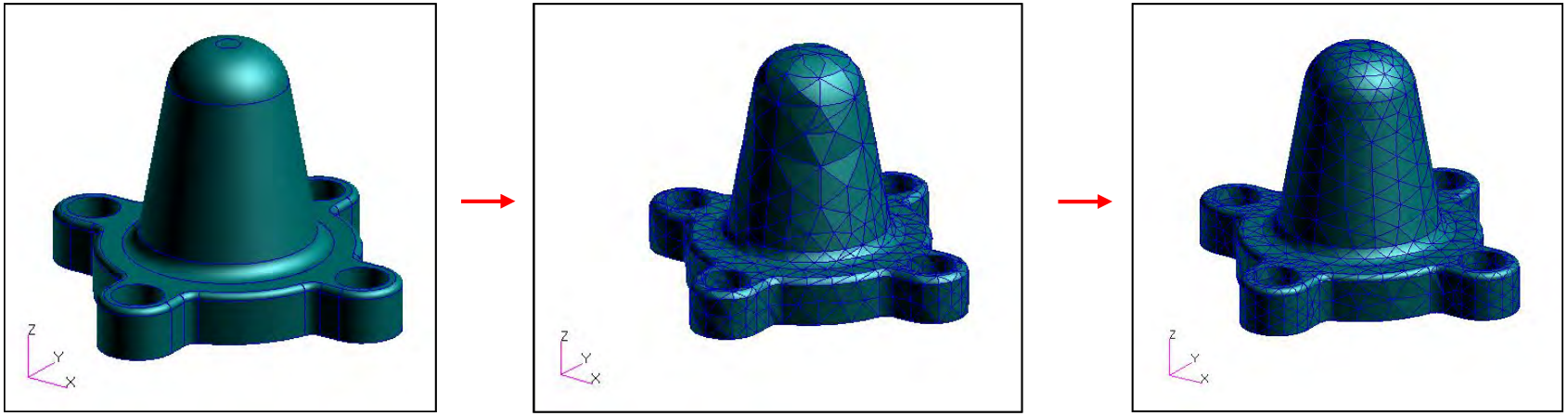
- Perform Workshop 9 “Hex Mesh Using Sweep Vector” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

SECTION 5

STRATEGIES IN GEOMETRIC MODELING AND MESHING

3D MESHING



Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

IDEALIZATION TYPE

- **Idealization type for this section:**
  - 2D beginning to end
  - 3D start point to 2D idealization
  - 3D start point to 2 ½ D idealization
  - 3D and 2D mix
  - **3D beginning to end**

Copyright© 2012 MSC.Software Corporation

3D BEGINNING TO END

- **The main categorization of methods in Patran is given in the table below.**
- **3D idealization is dominated by two factors**
  - What type of mesh element is acceptable, i.e. tetrahedral or hexahedral?
  - Is the geometry simple (parametric) or complex (B-rep)?

Start Point 3D	Nothing		Mesh	Geometry		
Method	Extrude, Glide, etc. Methods	B-Rep or Primitive Construction	Mesh Editing	TetMesh	IsoMesh	Geometry breakdown and rebuild methods
End Point 3D	Simple Geometry	Complex Geometry	Mesh	Mesh		Geometry

Copyright© 2012 MSC.Software Corporation

3D BEGINNING TO END

- **What is an acceptable mesh element type, i.e. tetrahedral or hexahedral?**
  - Tetrahedral elements:
    - Fast and easy to mesh with, in general
    - Can result in a poor mesh distribution if careless
    - A large number of elements is generated
    - Tet4 elements are very poor for structural models
    - Tet10 elements are good
  - Hexahedral elements:
    - Can be complicated to mesh complex shapes
      - Need a set of parametric solids to represent a complex shape
    - Gives excellent mesh distribution
    - Minimum number of elements are generated
    - Hex8 and Hex20 are both good

Copyright© 2012 MSC.Software Corporation

3D BEGINNING TO END

- **Is the geometry simple (parametric) or complex (B-rep)?**
  - Simple Geometry:
    - Blue parametric solid
    - Can be IsoMesh-ed using Hex, Wedge, or Tet elements
    - For complex geometric shapes, a set of blue solids is needed
      - Labor intensive
  - Complex Geometry:
    - Patran native or parasolid solid B-rep solid
    - Can only be meshed with the TetMesh-er
    - Ideal for complex shapes
      - Often comes from CAD

Copyright© 2012 MSC.Software Corporation

3D BEGINNING TO END

- **The choice of whether to use hexahedral or tetrahedral elements, or create simple or complex geometry is dependent on the following issues**
  - Allotted time
  - Desired accuracy
  - Modeling skill of Patran and/or CAD user
  - Finite element solver requirements
  - Certification or client dictates

Copyright© 2012 MSC.Software Corporation

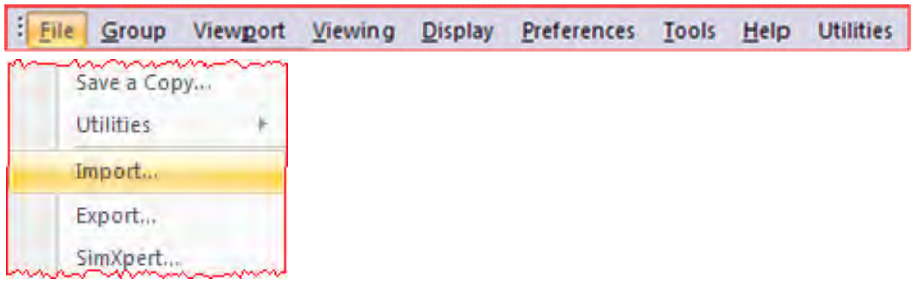
3D BEGINNING TO END

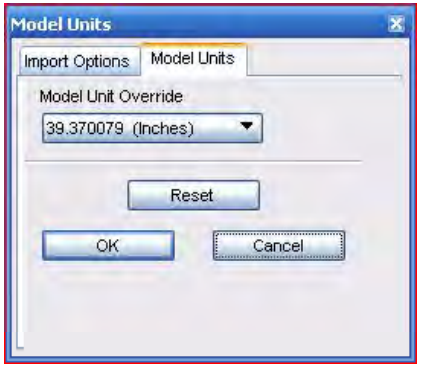
- **The solids created in CAD and/or Patran can be meshed using the appropriate solid mesher: IsoMesh or TetMesh**
  - IsoMesh elements: Hex8, Hex20, Tet4, Tet10, etc.
  - TetMesh elements: Tet4, Tet10, etc.

Start Point 3D	**Nothing**		**Mesh**	**Geometry**		
Method	**Extrude, Glide, etc. Methods**	**B-Rep or Primitive Construction**	**Mesh Editing**	**TetMesh**	**IsoMesh**	**Geometry breakdown and rebuild methods**
End Point 3D	**Simple Geometry**	**Complex Geometry**	**Mesh**	**Mesh**		**Geometry**

Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING



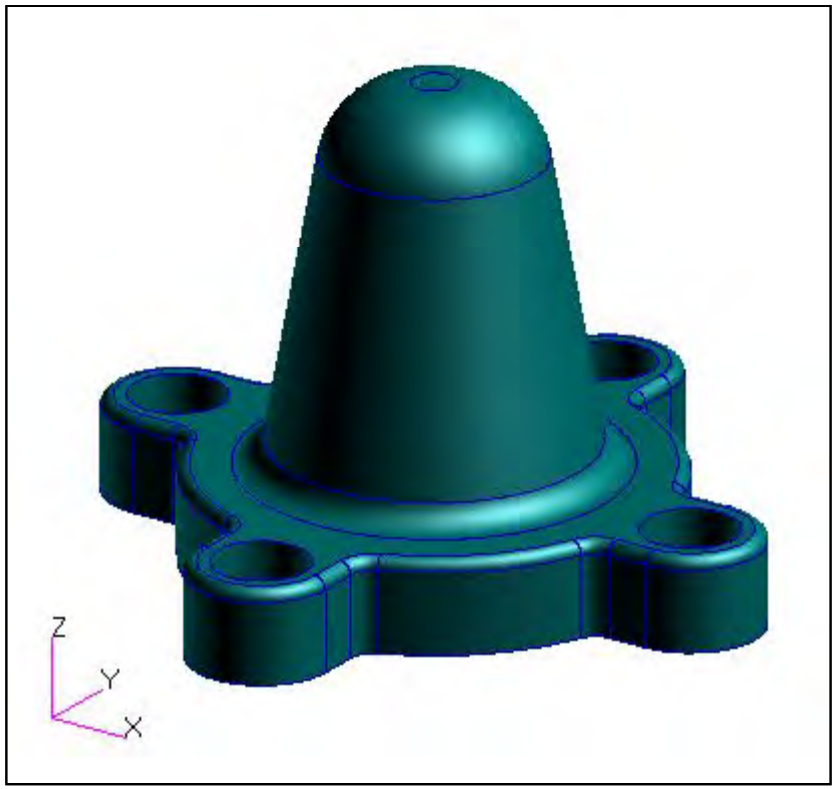


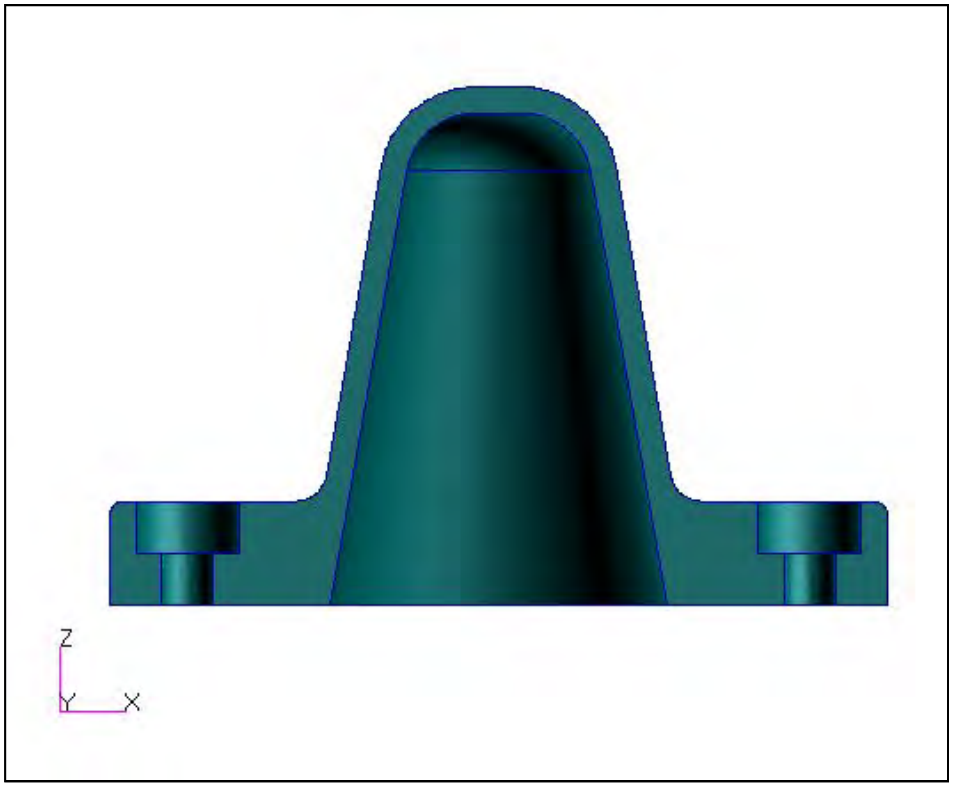
- **Import the parasolid xmt file:**
  - Select **Import** from the *File* menu
  - Set the *Source* to **Parasolid xmt**.
  - Click on **Parasolid xmt Options**.
  - In the *Import Options* form, click on **Model Units**.
  - Change the *Model Unit Override* to **39.370079 (Inches)** and then click **OK** for both forms.
  - Import parasolid solid file **parasolid_1.xmt**.

Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

- **Here are two views of the solid; notice the internal structure:**

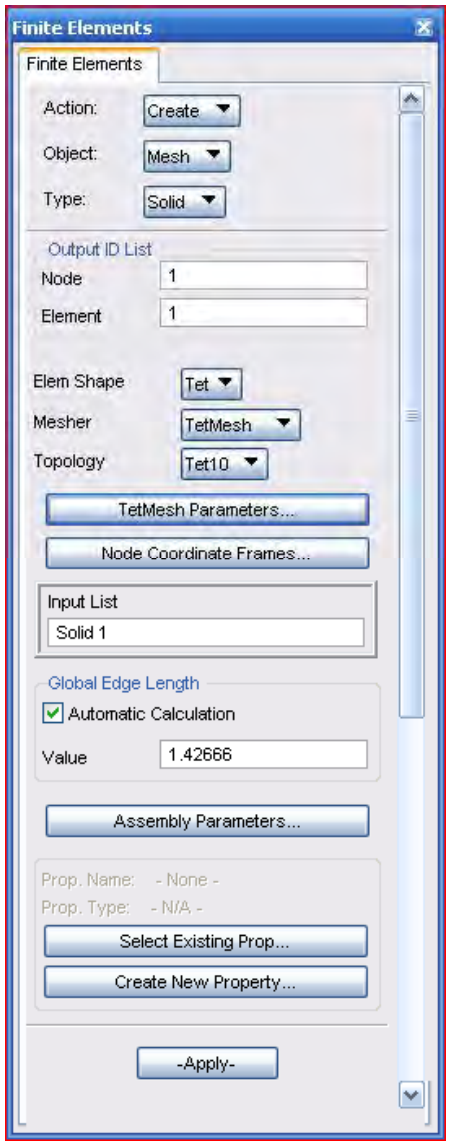


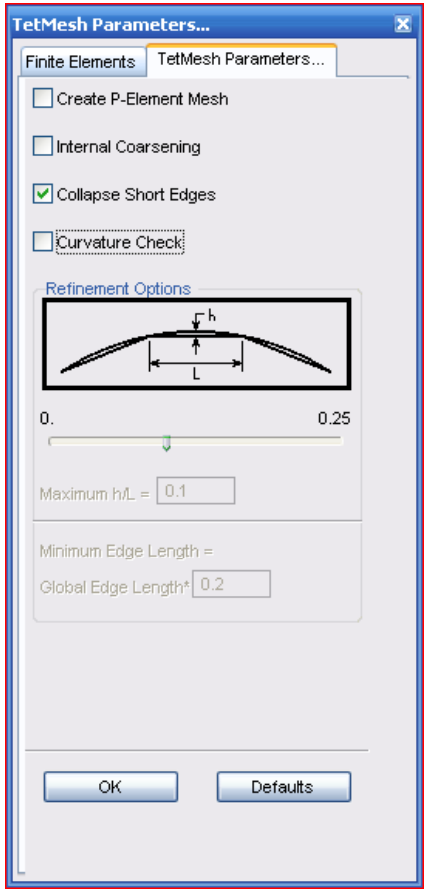


Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

- **TetMesh the solid:**
  - Under the *Meshing* tab click on **Solid** in the *Meshers* group.
  - Set *Elem Shape*, *Mesher*, and *Topology* to **Tet**, **TetMesh**, and **Tet10**, respectively
  - Select **Solid 1** for *Input List*
  - Click on **TetMesh Parameters**, and remove toggle for **Curvature Check**.
  - Click **OK**, then click **Apply**.

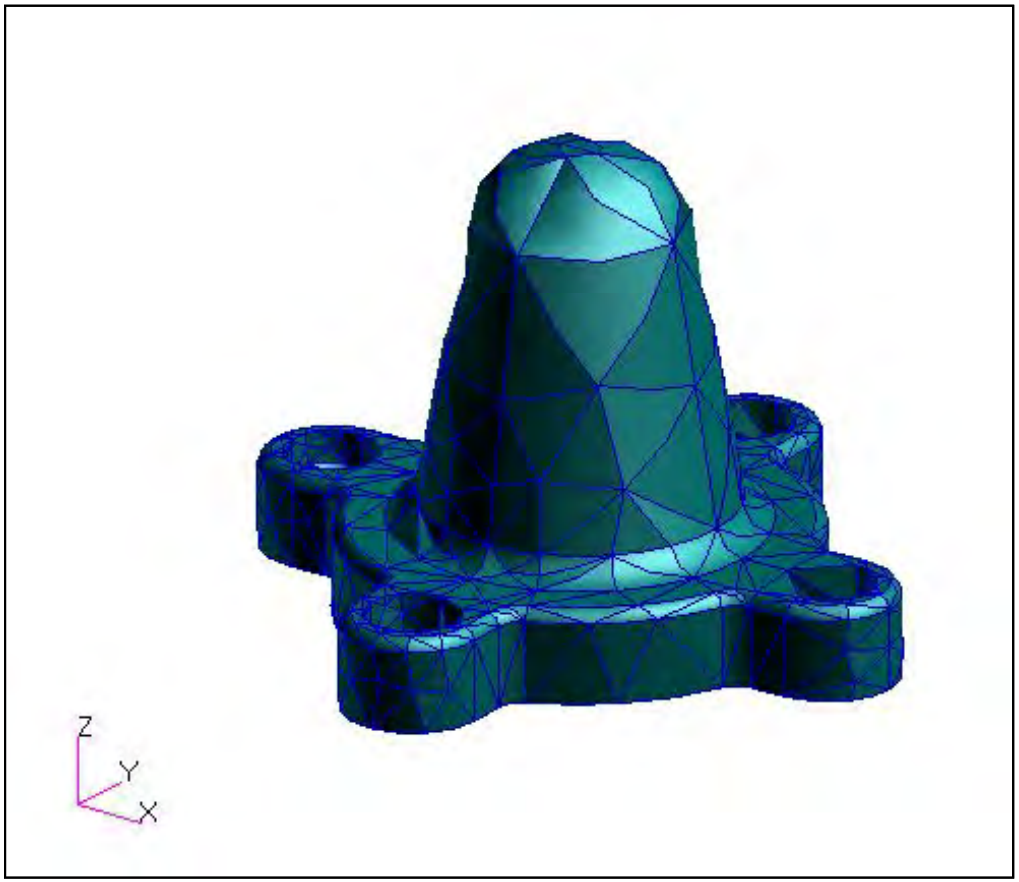




Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

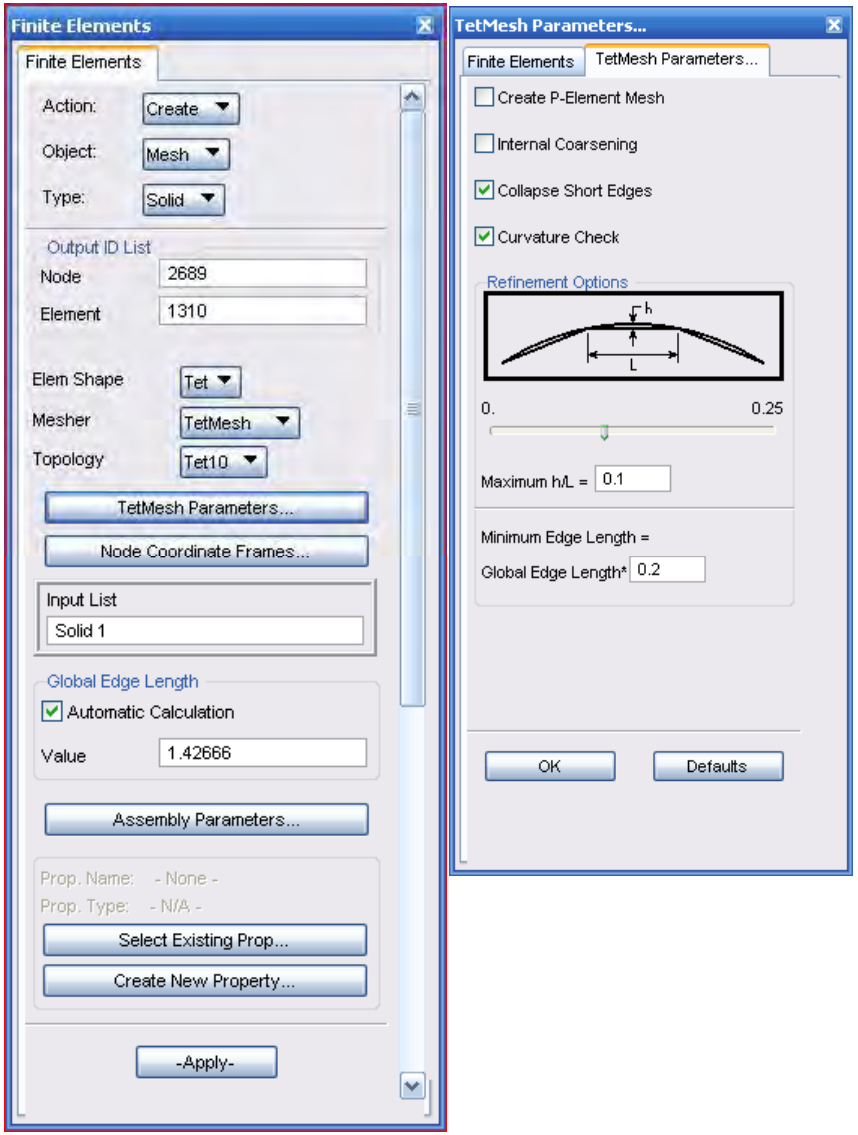
- **This is a very poor tet mesh. The solid will be meshed again, this time using Curvature Check.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

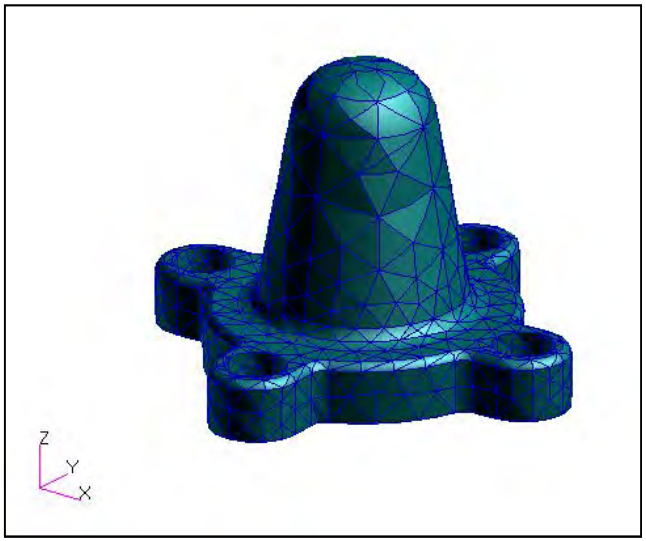
- **TetMesh the solid:**
  - Click on **TetMesh Parameters** and click the checkbox for **Curvature Check.**
    - Use the default values for the *Maximum h/L* and *GEL factor* (**0.1** and **0.2**, respectively).
  - Click **OK**, then **Apply**.
  - Click **Yes For All**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

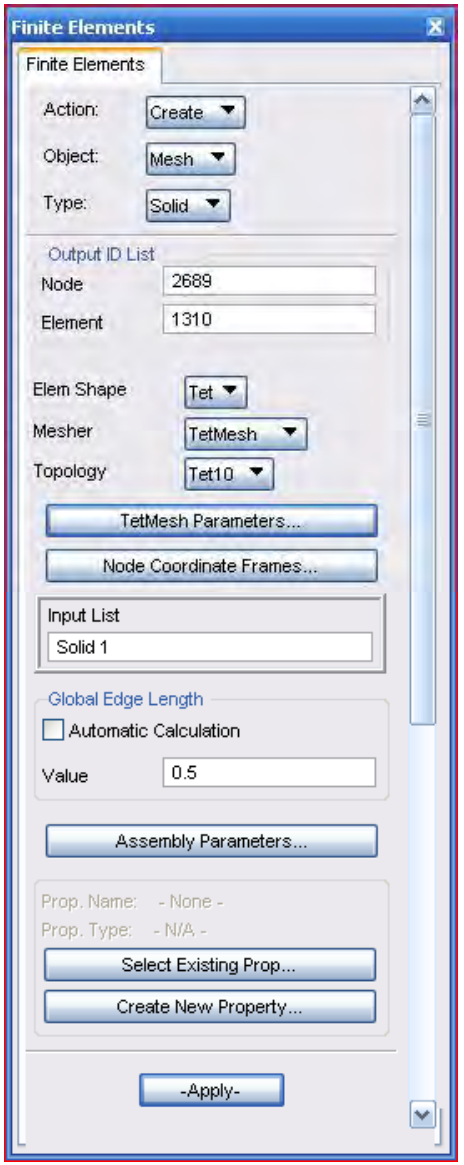
- This mesh, created using the Curvature Check, is definitely better than the previous mesh. However, it is still quite poor. Even though a large GEL was used, smaller elements were created in areas of geometric curvature. It is desirable to use a reasonably large GEL so areas of non-curved geometry will have larger elements, minimizing the number of elements. However, by using the Curvature Check, smaller elements are created at curved areas. It is important to use Curvature Check when there are highly curved edges and faces.



Copyright© 2012 MSC.Software Corporation

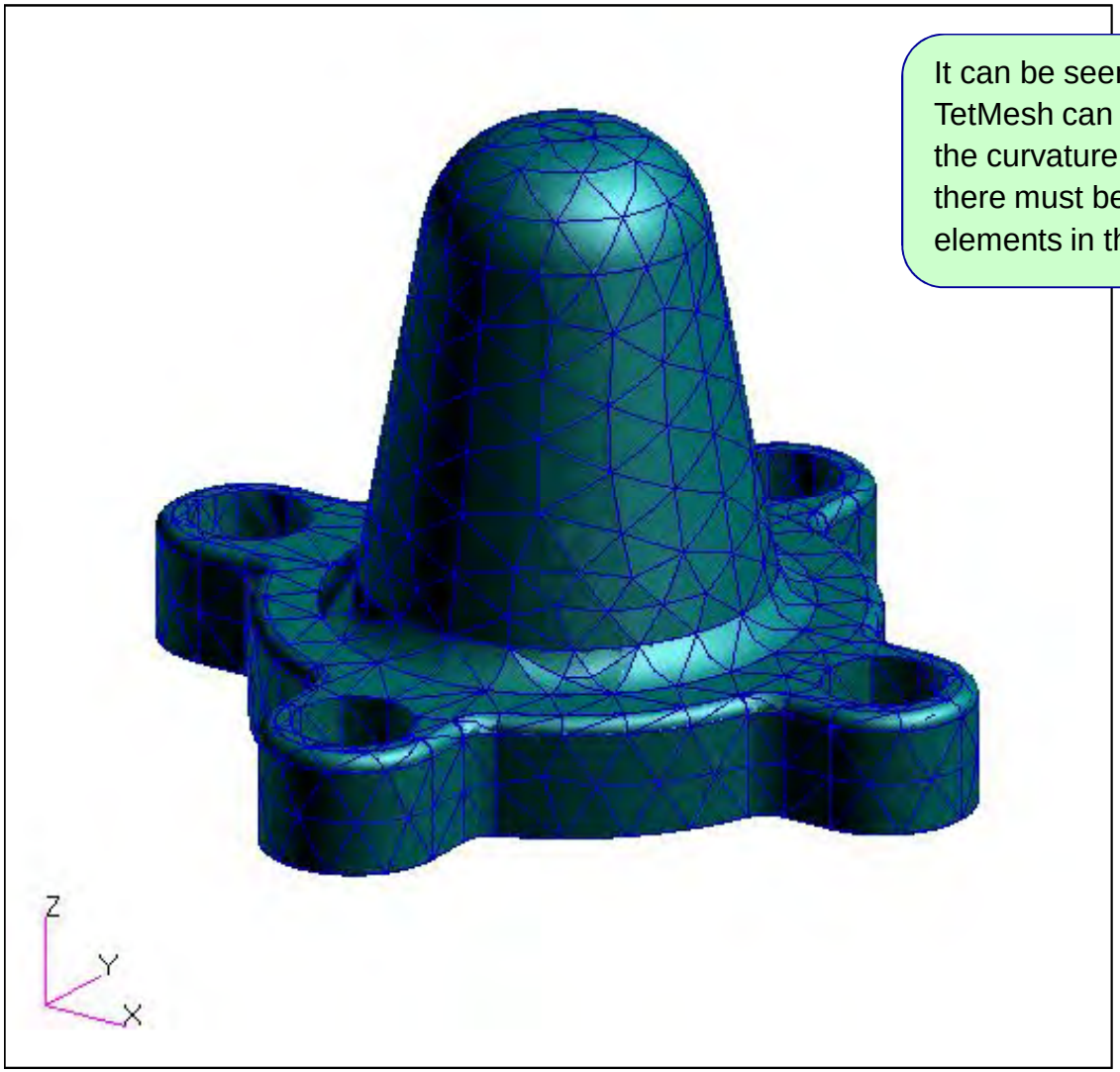
CASE STUDY : TETMESHING

- **Mesh the model again, using a much smaller GEL and no curvature check:**
  - Click on **TetMesh Parameters** and uncheck the checkbox for **Curvature Check**, and click **OK**.
  - Turn the toggle off for *Automatic Calculation* and enter **0.5** for the *GEL*.
  - Click **Apply**.
  - Click **Yes For All**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

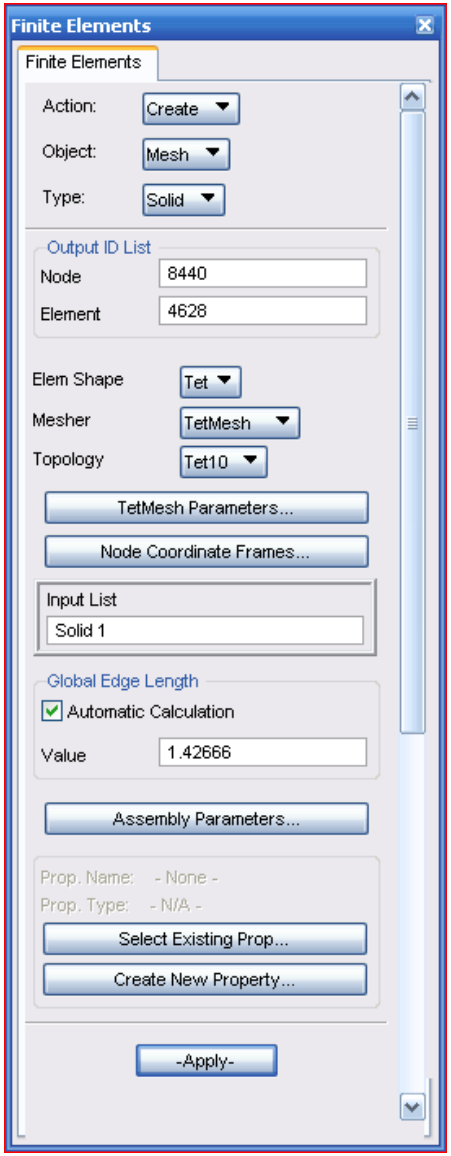


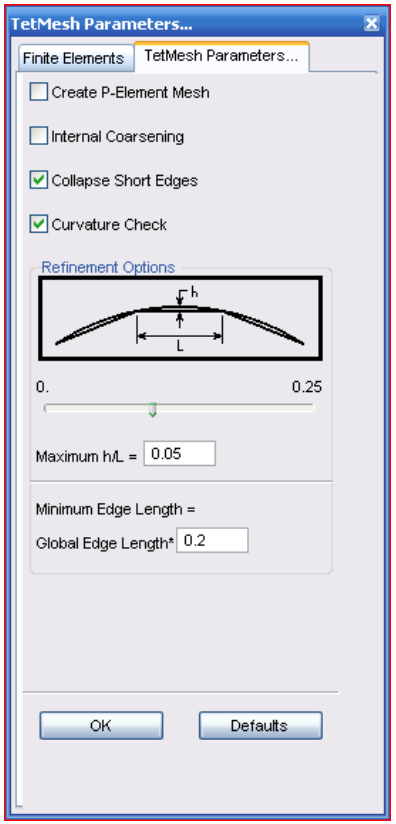
It can be seen that a high quality TetMesh can be created without the curvature check. However, there must be a larger number of elements in that mesh.

Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

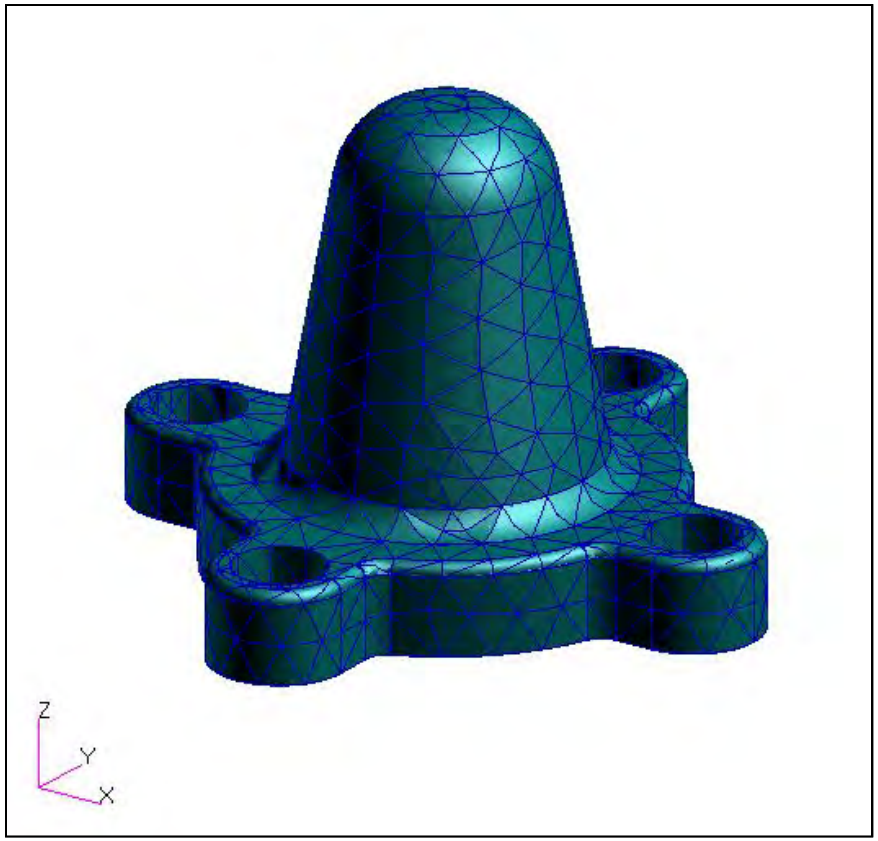
- **TetMesh again, changing only the Maximum h/L and observe the effects**
  - Click on **TetMesh Parameters** and turn on the toggle for **Curvature Check.**
    - Enter **0.05** for the Maximum h/L and click **OK**.
  - Turn the toggle on for **Automatic Calculation** and use the default value for the *GEL*.
  - Click **Apply**.
  - Click **Yes For All**.

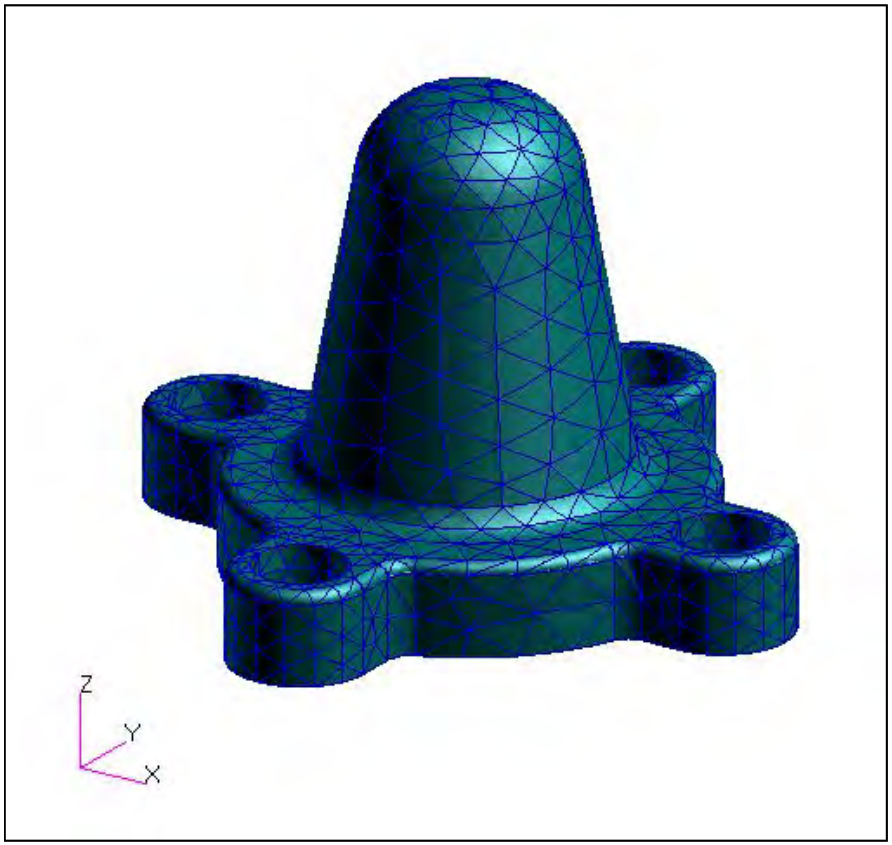




Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING





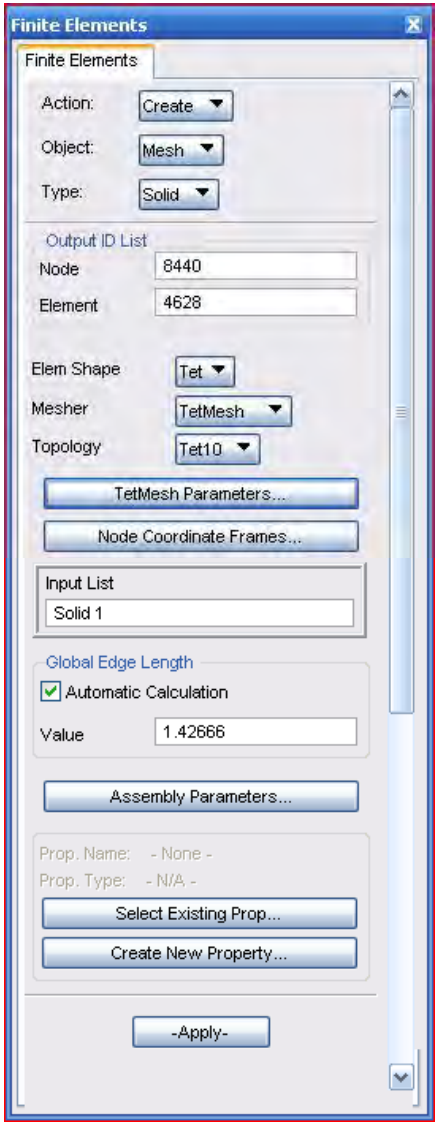
Shown here, is the previous mesh with 0.5 GEL and no curvature check.

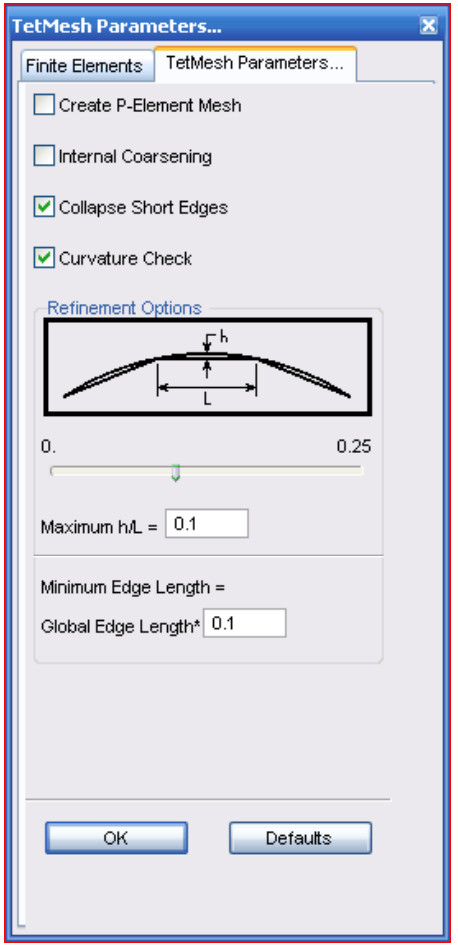
Here is the current mesh. Obviously, it is a better mesh because there are more elements at the fillet area between the dome and the base.

Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

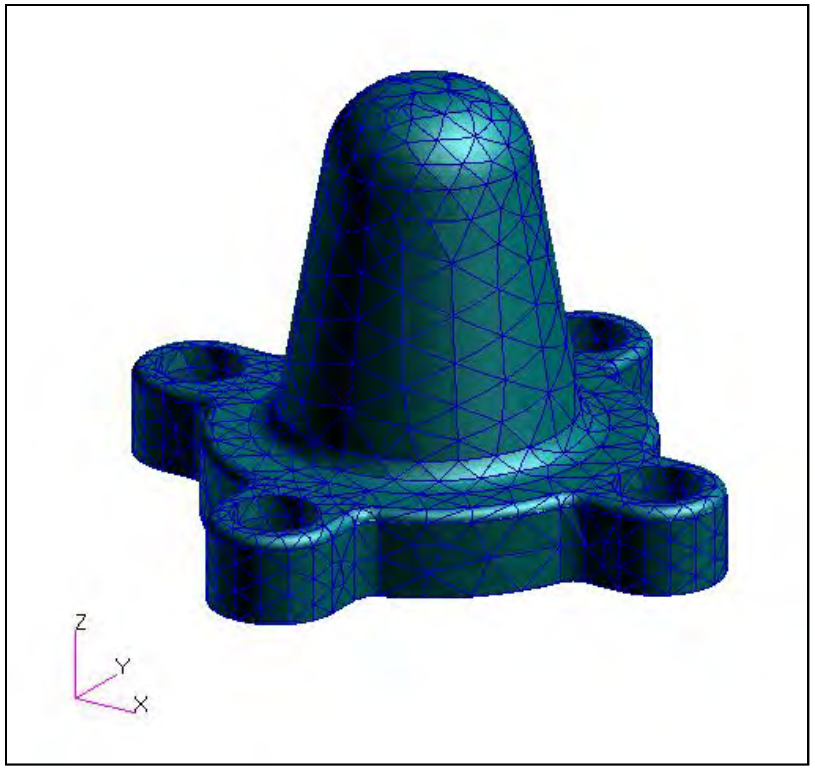
- **Mesh the model again, changing the Minimum Edge Length.**
  - Click on **TetMesh Parameters** and turn on the toggle for **Curvature Check**.
  - Use the default value (**0.1**) for *Maximum h/L*, and enter **0.1** for the *Minimum Edge Length factor*, and click **OK**.
  - Turn the toggle on **for Automatic Calculation** and use the default value for the *GEL*.
  - Click **Apply**.
  - Click **Yes For All**.



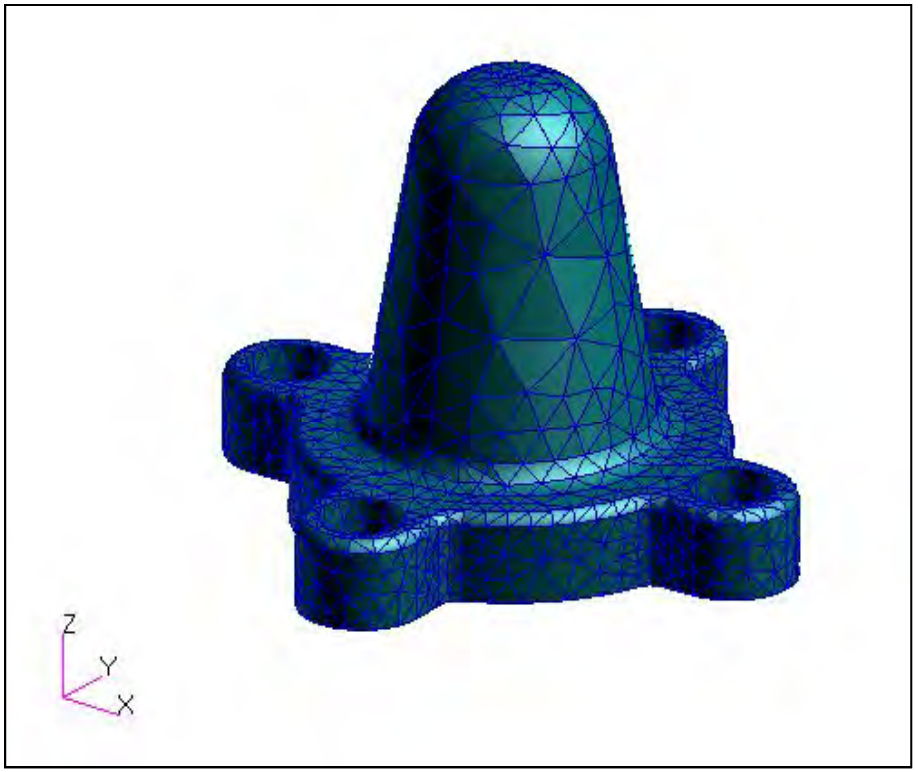


Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING



Here is the previous mesh, with Maximum h/L 0.05, Minimum Edge Length GEL*0.2, and GEL 1.42665

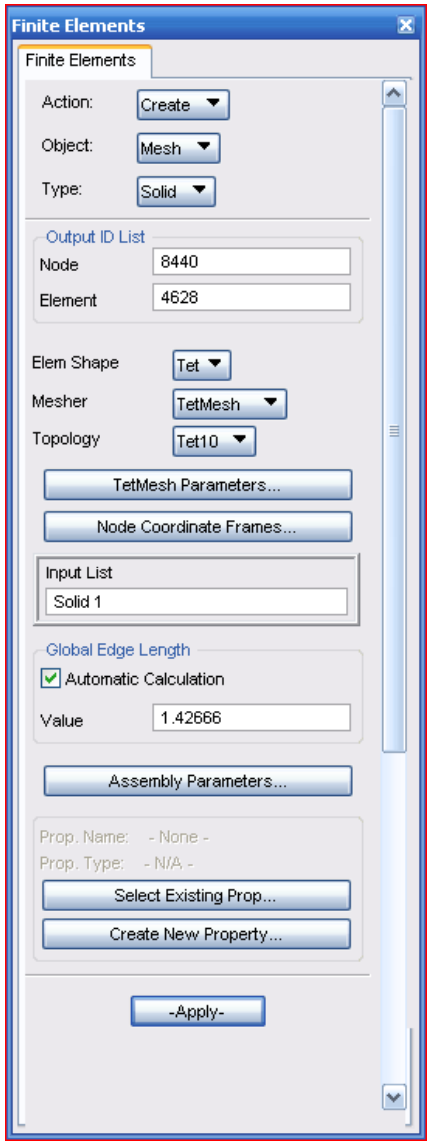


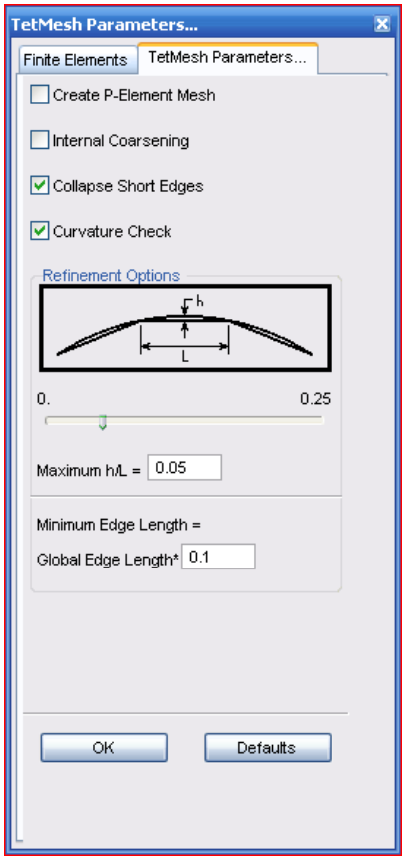
Shown here, is the current mesh, with Maximum h/L 0.1, Minimum Edge Length GEL*0.1, and GEL 1.42665. It is a regression from the quality of the previous TetMesh.

Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

- **Redo the TetMesh**
  - Click on **TetMesh Parameters** and turn on the toggle for **Curvature Check.**
    - Enter **0.05** for the *Maximum h/L* and enter **0.1** for the *Minimum Edge Length factor*, and click **OK**.
  - Turn the toggle on for **Automatic Calculation** and use the default value for the *GEL*.
  - Click **Apply**.
  - Click **Yes For All**.

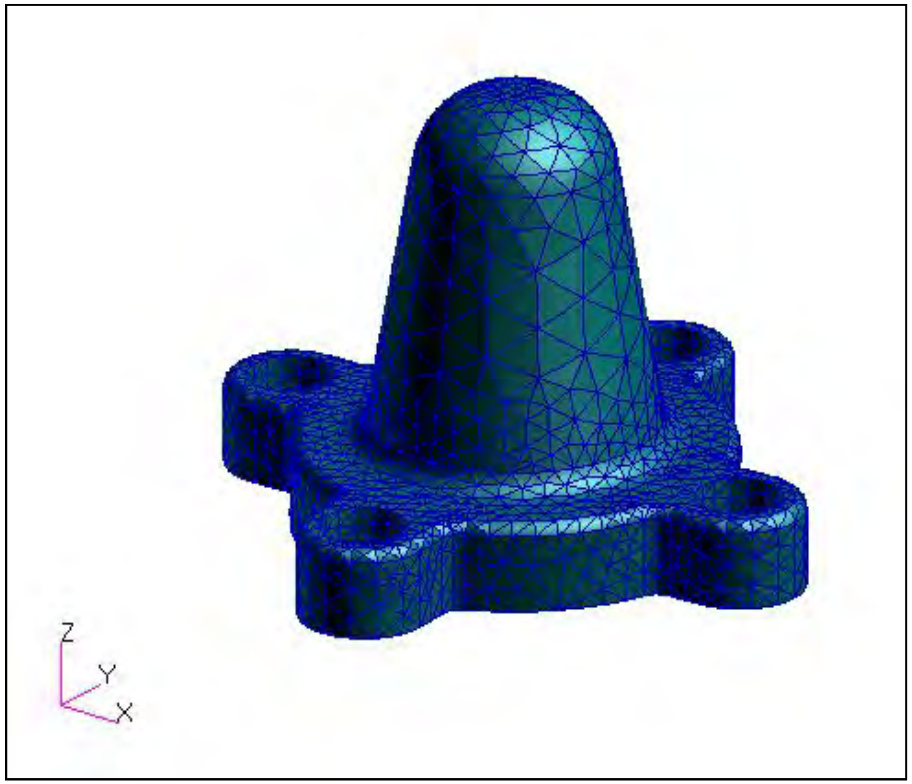


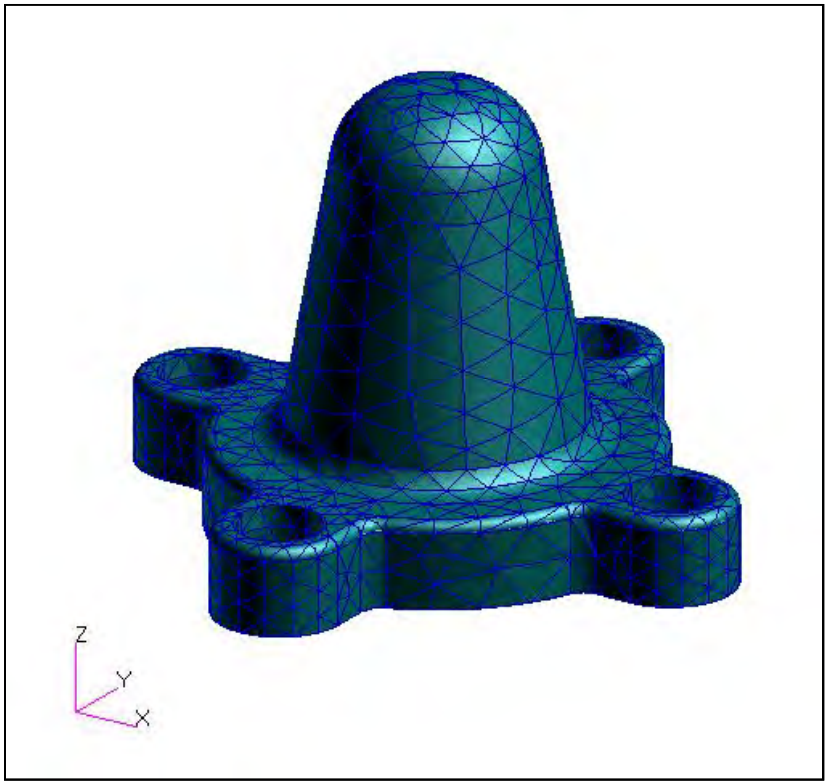


Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

- **Compare the new mesh with a previous mesh**





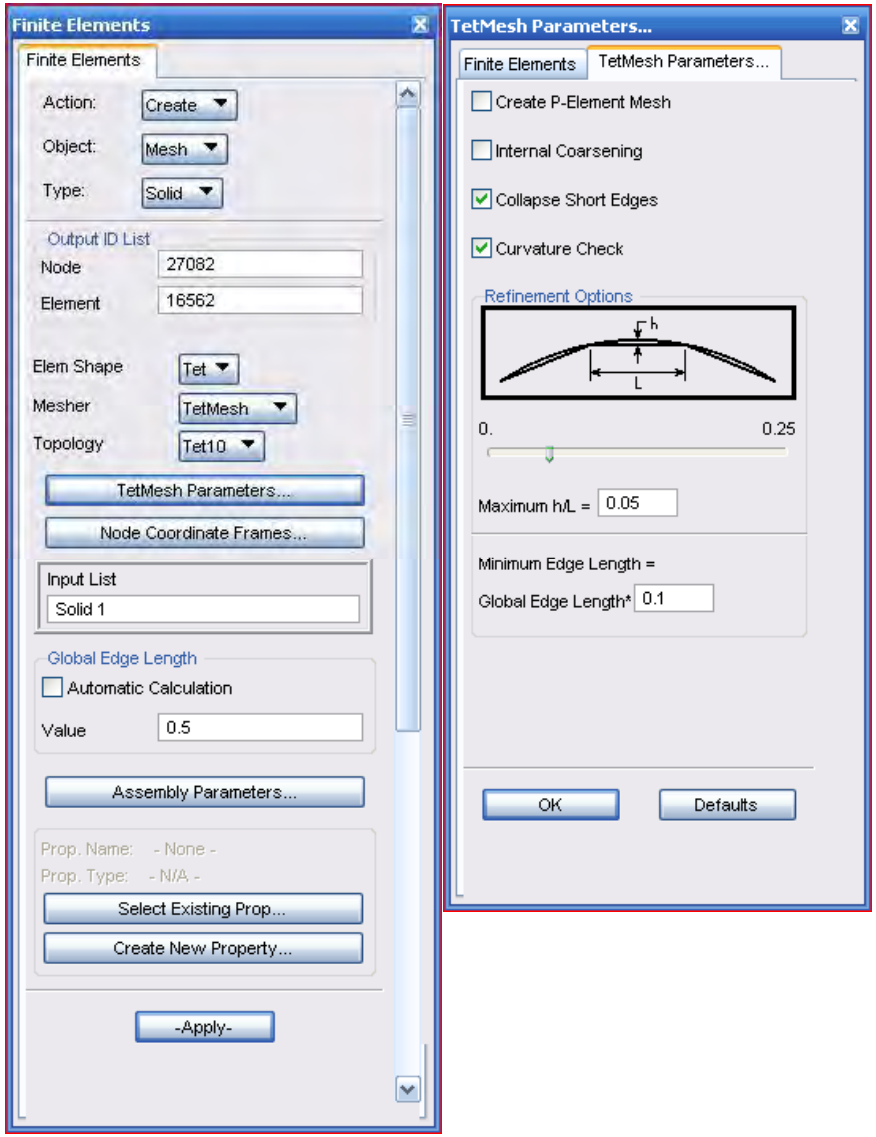
Shown here is the current mesh, with the following values:
Maximum h/L = **0.05**
Minimum Edge Length = GEL* **0.1**
GEL = **1.42665** (default value)

Here is a previous mesh, with the following values:
Maximum h/L = **0.05**
Minimum Edge Length = GEL***0.2** (default value)
GEL = **1.42665** (default value)

Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

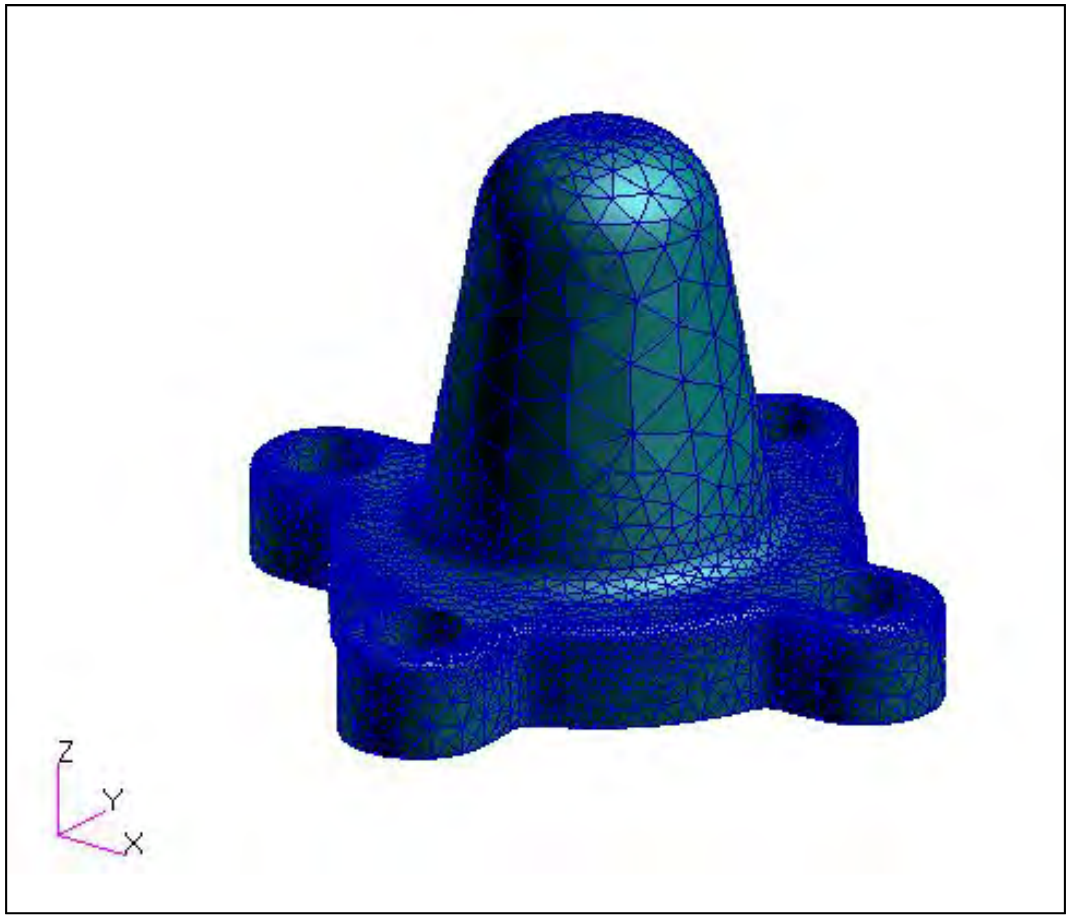
- **TetMesh the model with different values:**
  - Click on **TetMesh Parameters** and turn **on** the toggle for **Curvature Check**.
    - Enter **0.05** for the *Maximum h/L* and enter **0.1** for the *Minimum Edge Length* factor, and click **OK**.
  - Turn the toggle off for **Automatic Calculation** and enter **0.5** for the *GEL.*
  - Click **Apply**.
  - Click **Yes For All**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

- **This TetMesh is very good, and probably much more dense than is necessary.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

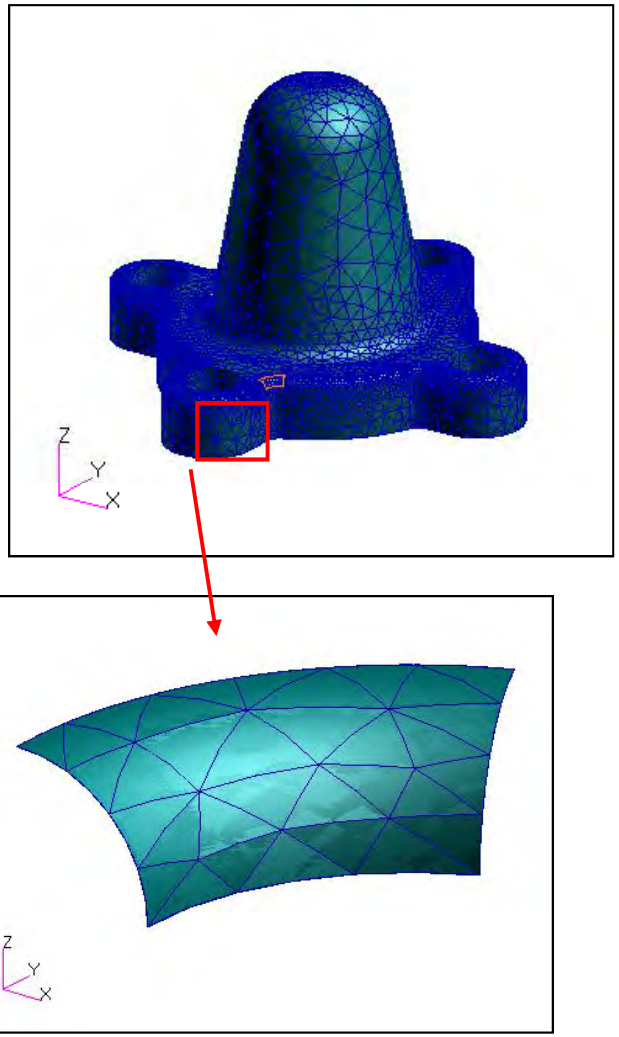
- **During this and the previous Tet mesh creations, the following messages appeared in the Patran session file**

```
fem_create_mesh_sol_5( "Solid 1", "TetHybrid", "Tet10", 4, ["0.5", "0.05",  @
"0.1", "0.0"], 49168, 0, 1, 0, 1, 0.0047185398, "", "#", "#", "Coord 0",  @
"Coord 0", fem_create_mesh_solid_num_nodes, fem_create_mesh_solid_num_elems,  @
fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )
$# Question from application FEM
$#   Do you wish to delete the existing mesh on Solid 1?
$? YESFORALL 2001103
$# Processing the geometry...
$# The current "Minimum Edge Length" restricts the mesher from creating a fine mesh for : Solid 1.10
$# The current "Minimum Edge Length" restricts the mesher from creating a fine mesh for : Solid 1.25.3
…
$# The current "Minimum Edge Length" restricts the mesher from creating a fine mesh for : Solid 1.3
$# Completed Meshing the Vertices...
$# Completed Meshing the Edges...
$# Completed Meshing the Faces...
$# Completed Meshing the Solids...
$# 72187 nodes and 45859 elements created for Solid 1.
$# === 72187 nodes created.  IDs  = 1:72187.
$# === 45859 elements created.  IDs  = 1:45859.
```


Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

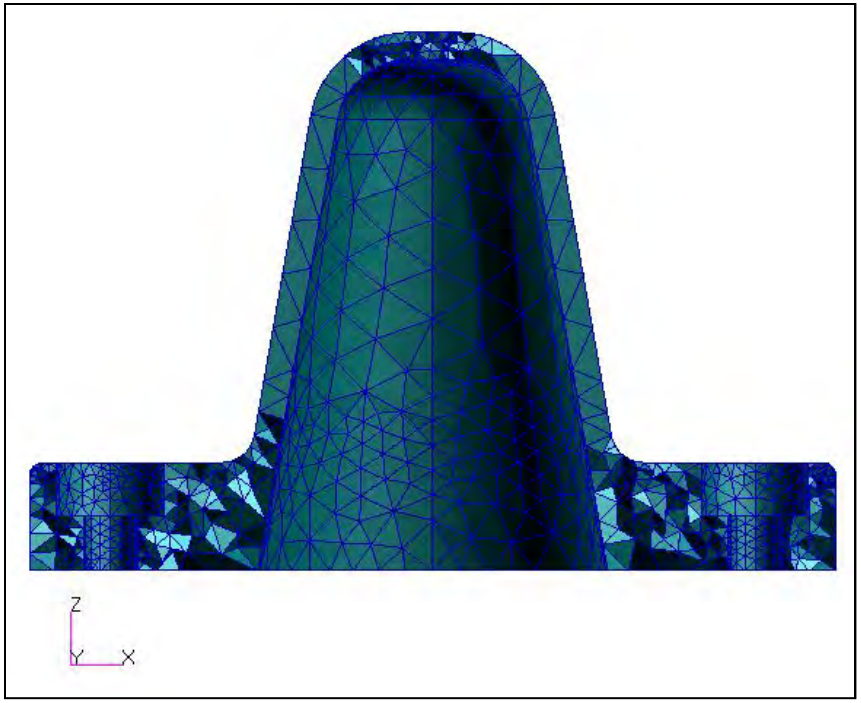
- **Studying the mesh:**
  - The messages from the history window indicate that the meshing was completed successfully. However, the messages also indicate that there were some edges and faces for which the element size was restricted from becoming finer (“Minimum Edge Length” is specified under TetMesh Parameters). Shown here is a surface extracted from Solid 1.3, and a tri element mesh that was created from the free faces of the tet elements at the surface.
  - There are probably enough tet elements at this surface at Solid 1.3 to adequately represent the spatial variation of stress. The message is not significant for all models.



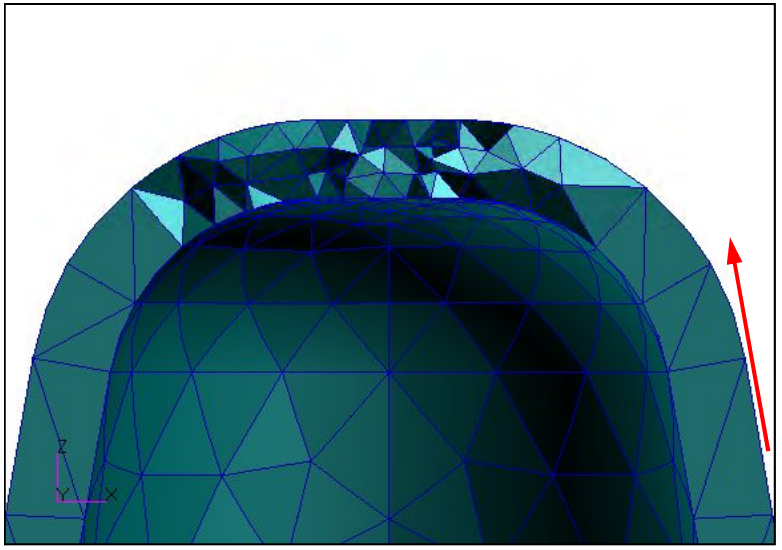
Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

Look at the cross section of the latest tet mesh:



Here is the mesh at the top of the dome:



Notice that the number of elements through the thickness increases going up the dome

Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

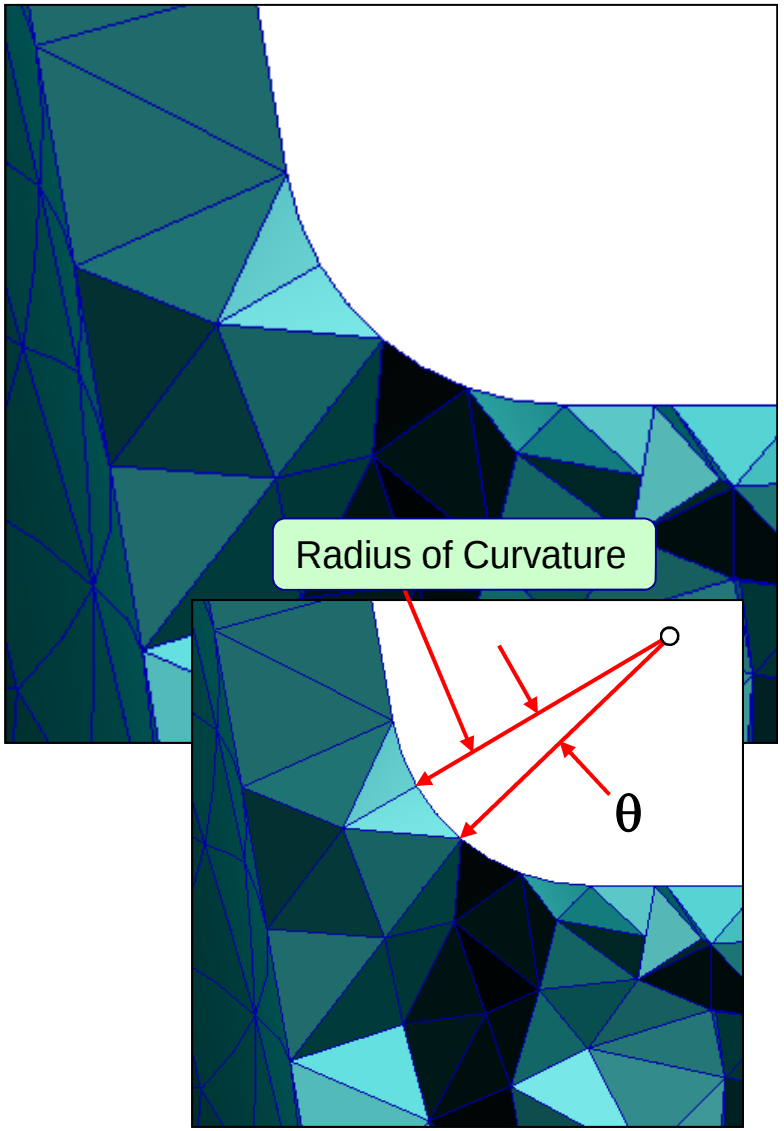
- **Take another look at the fillet area between the dome and the base.**

There are four tet10 elements along the same solid edge as was discussed previously. The value for *h/L* and *L* is **0.047** and **0.109**, respectively. Of course, *h/L*(**0.047**) < Maximum *h/L*(**0.05**), and *L*(**0.109**) > *Minimum Edge Length*(*GEL****0.1** = **0.05**). Here, the *GEL* was small enough, so the *Minimum Edge Length*(*GEL****0.1**) was small enough to cause the creation of tet elements with a size that was bracketed by the **TetMesh Parameter** values.

The simple expression for h/L is

$$h/L = (1-\cos(\theta/2))/(2\sin(\theta/2)) \qquad (1)$$

where h and L are defined in Patran, and θ is the angle between the two radii at the ends of an element on a curved edge; the radii are from the center of curvature of the curved edge.



Copyright© 2012 MSC.Software Corporation

CASE STUDY : TETMESHING

- **Estimating Maximum h/L and Minimum Edge Length**
  - This can be used to estimate the desired value of Maximum h/L.
  - For this case study θ = 20 degrees, and from the previous expression (1) h/L = 0.044. The following table can be used to help estimate Maximum h/L and Minimum Edge Length for Curvature Check:

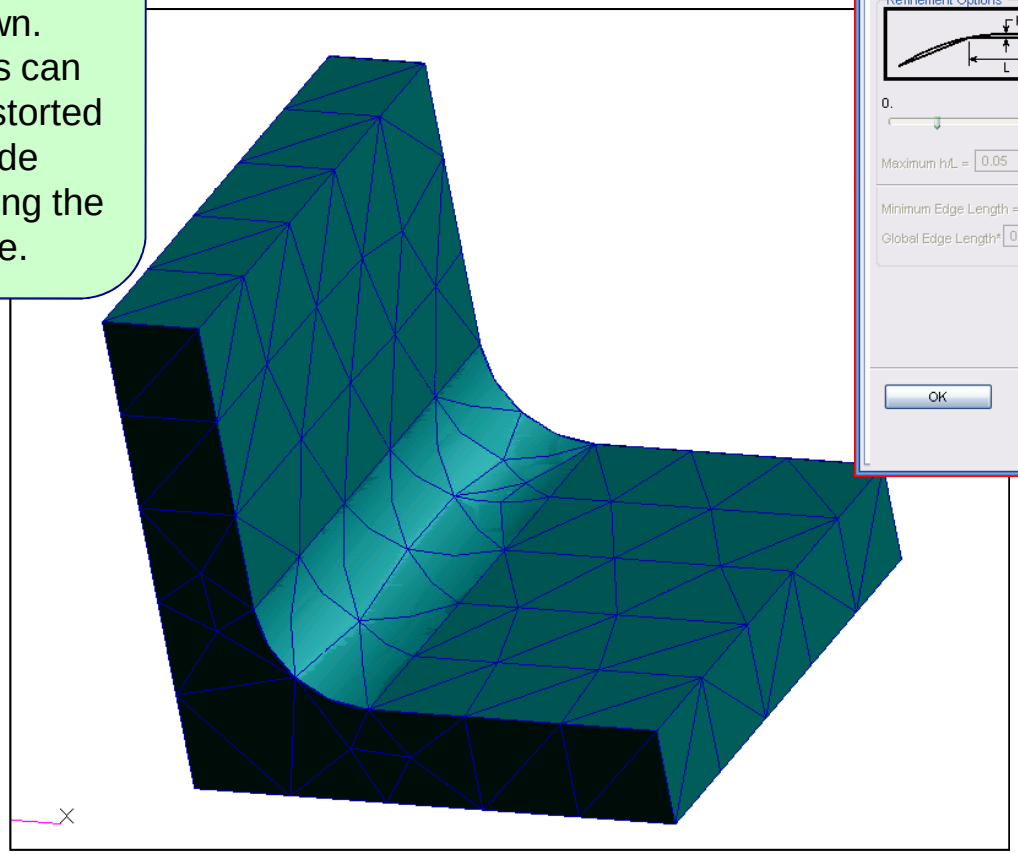
	h/L and L/R		
	Number of Elements on Curved Edge		
Edge Angle(deg)	**2**	**4**	**8**
45	0.049 and 0.390	0.025 and 0.196	0.012 and 0.098
90	0.099 and 0.765	0.049 and 0.390	0.025 and 0.196

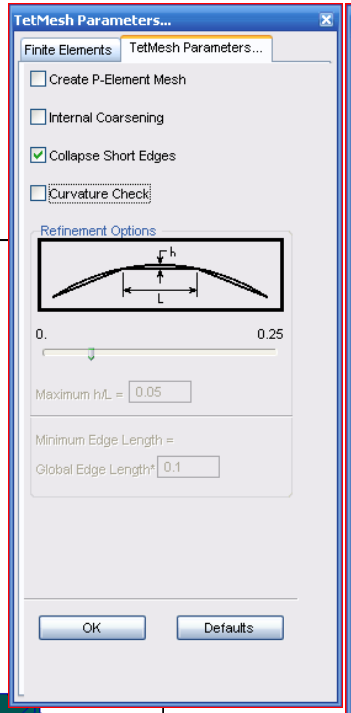
Where **R** is the radius of curvature, and **h** and **L** are defined in the Patran GUI. The *GEL* should be chosen to minimize the total number of tet elements created, but correctly capture the deformation field of the model.

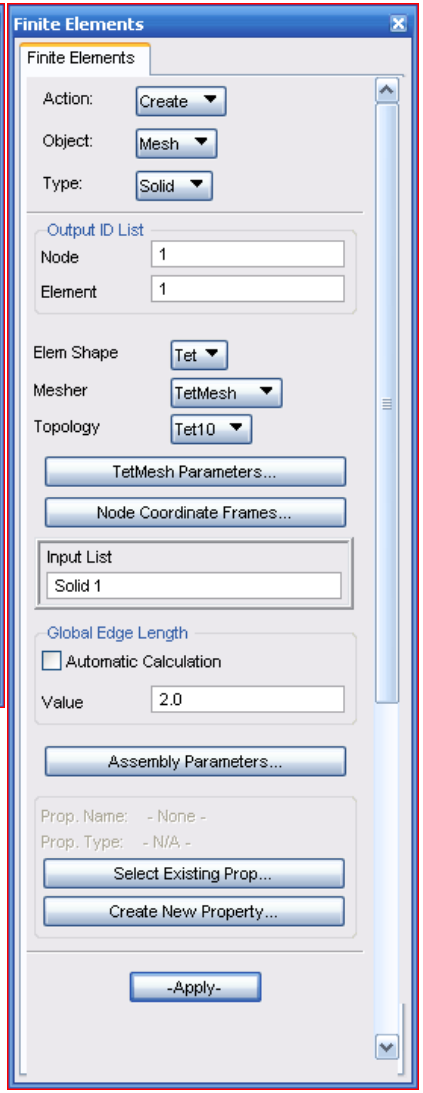
Copyright© 2012 MSC.Software Corporation

AVOID NEGATIVE JACOBIAN

Tet10 elements are recommended, but care must be taken in areas such as the fillet shown. The elements can be greatly distorted by the mid-side nodes following the fillet curvature.





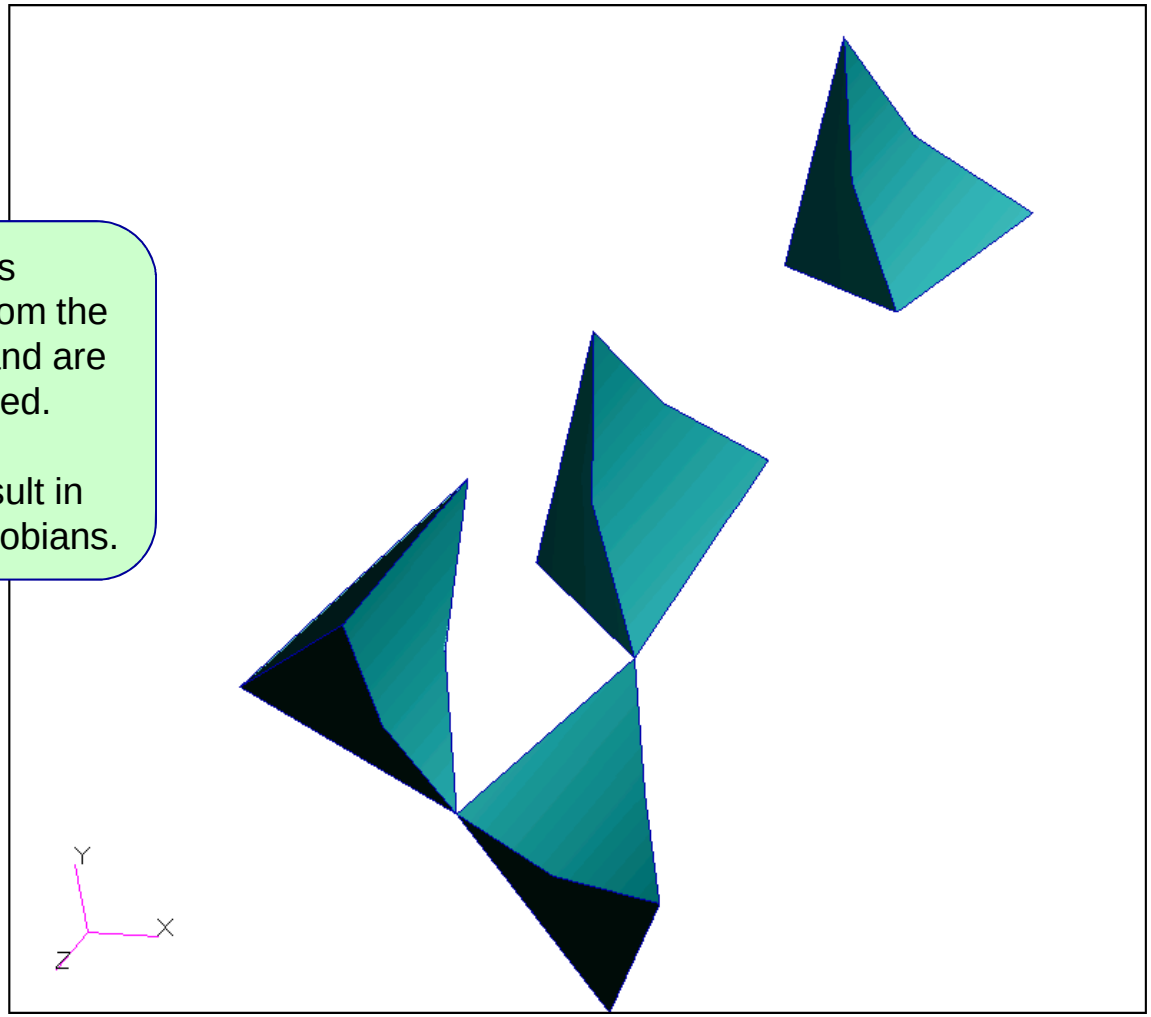


Copyright© 2012 MSC.Software Corporation

AVOID NEGATIVE JACOBIAN

The elements shown are from the fillet region and are highly distorted.

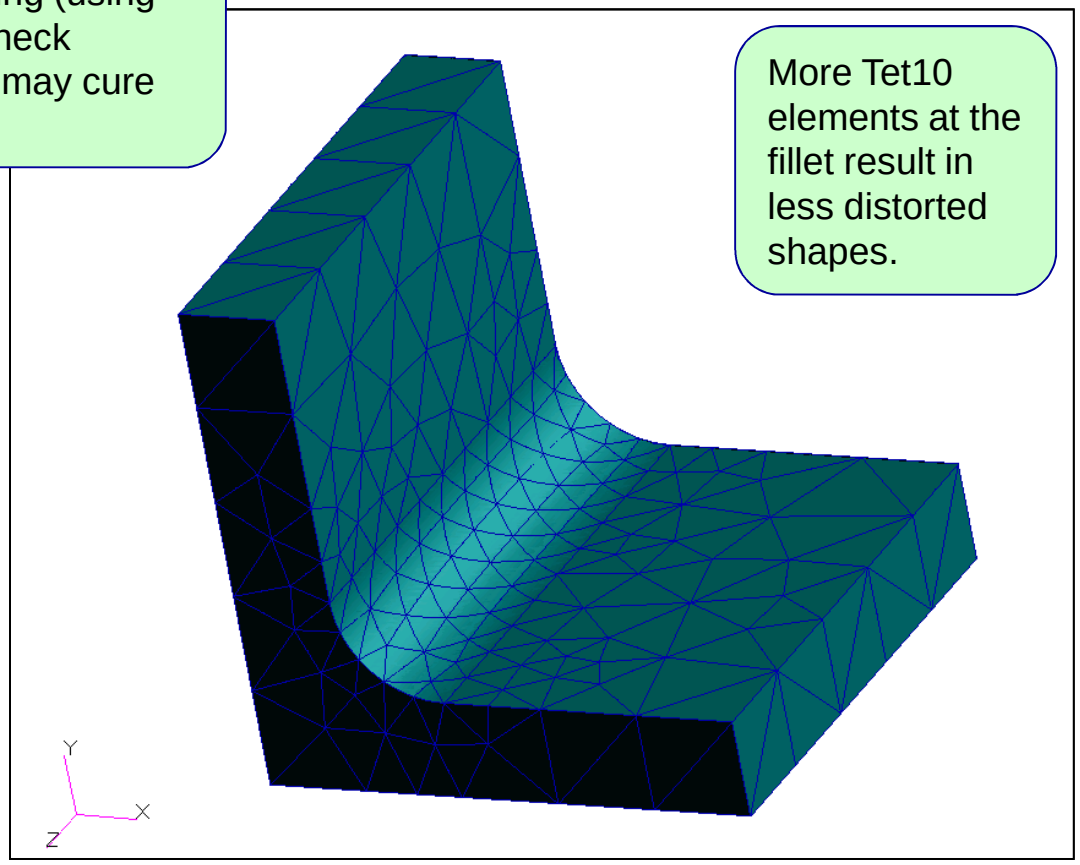
They can result in negative Jacobians.



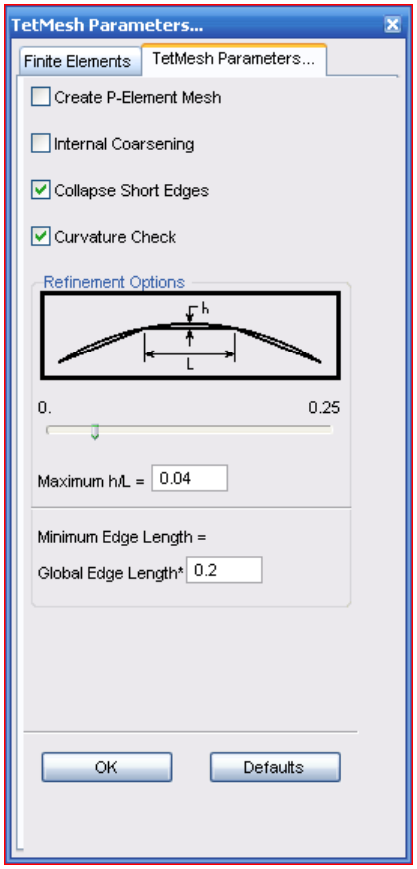
Copyright© 2012 MSC.Software Corporation

AVOID NEGATIVE JACOBIAN

Using the curvature-based meshing (using Curvature Check parameters) may cure the problem.



More Tet10 elements at the fillet result in less distorted shapes.

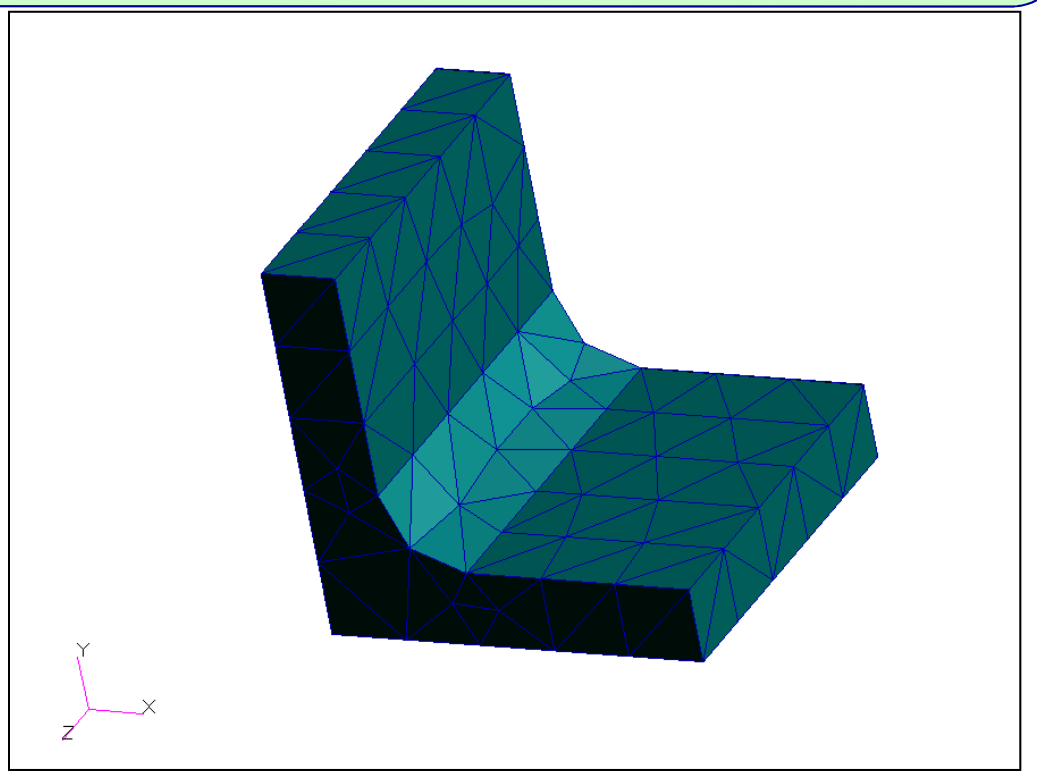


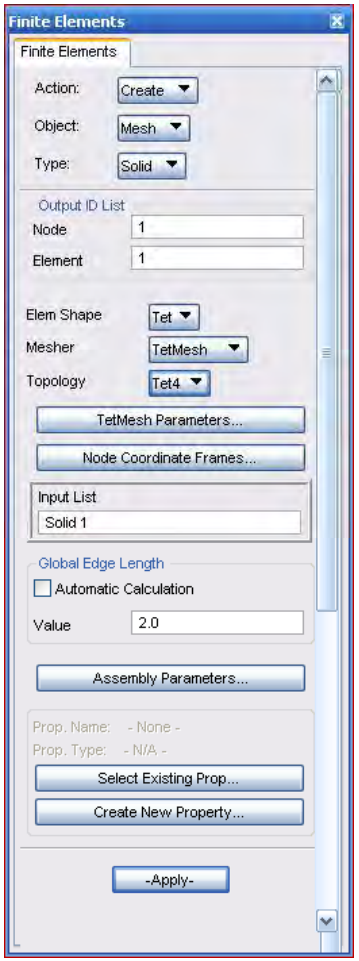
Copyright© 2012 MSC.Software Corporation

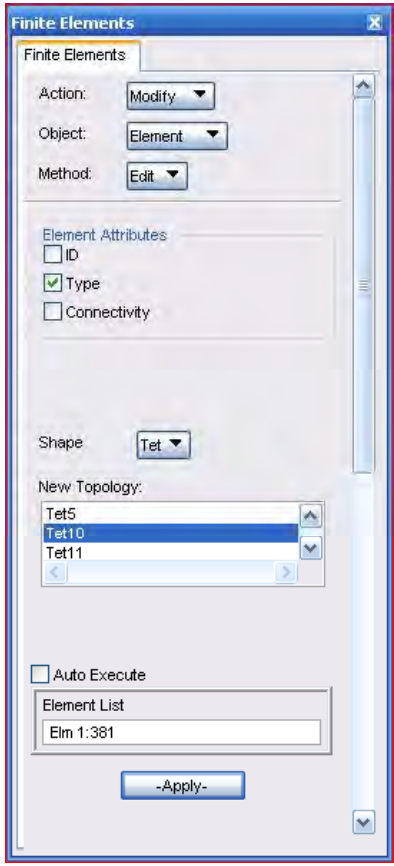
AVOID NEGATIVE JACOBIAN

Another technique is to mesh with Tet4s, disassociate the Tet4s from the geometry, then modify them to be Tet10s. This will keep the Tet10 edges straight.

The accuracy of representing the geometry will be reduced, but the location of the midside nodes will not cause problems.



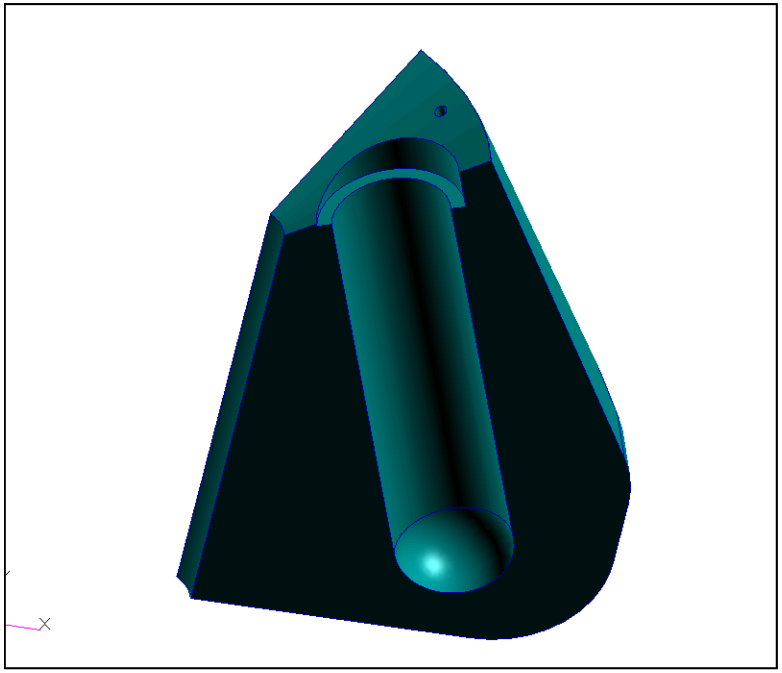




Copyright© 2012 MSC.Software Corporation

CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

- **This structure is used in an example of how to control the tet meshing of a B-rep or parasolid solid using 2D surface meshing around structural details.**

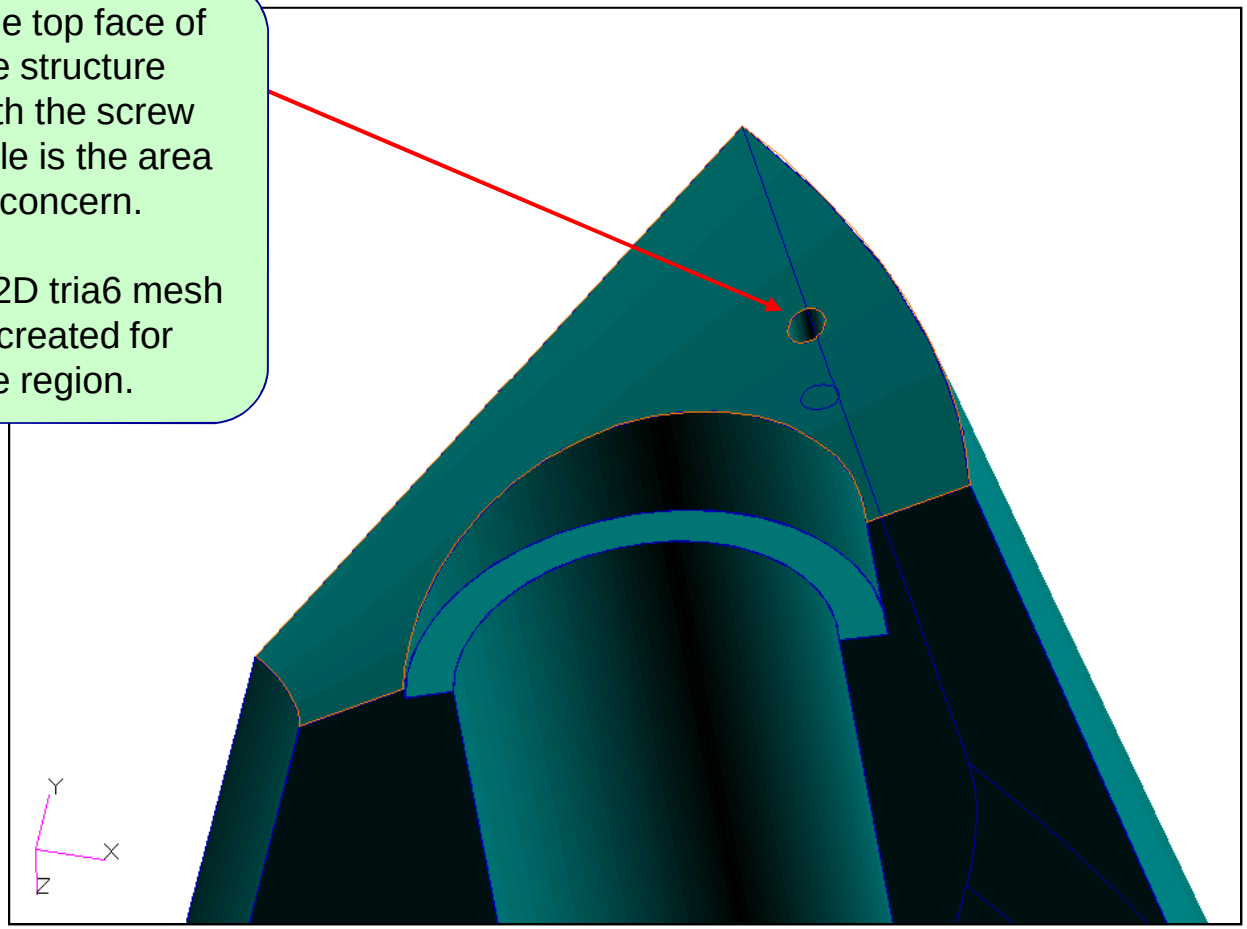


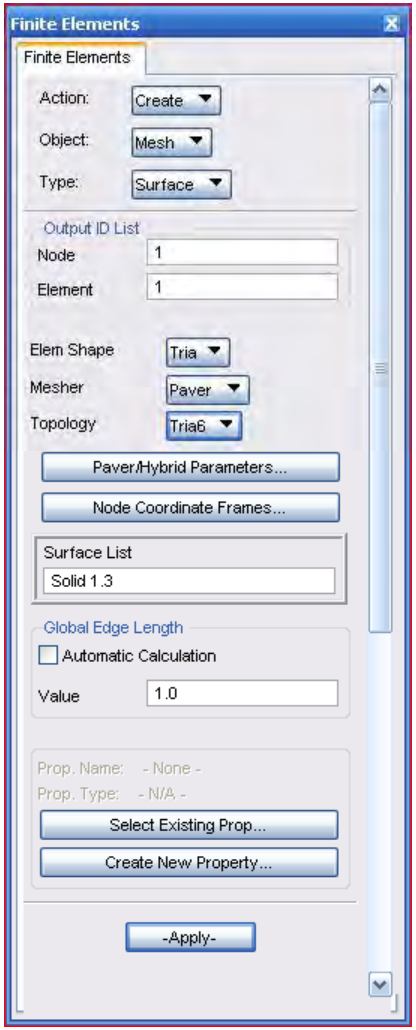
Copyright© 2012 MSC.Software Corporation

CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

The top face of the structure with the screw hole is the area of concern.

A 2D tria6 mesh is created for the region.

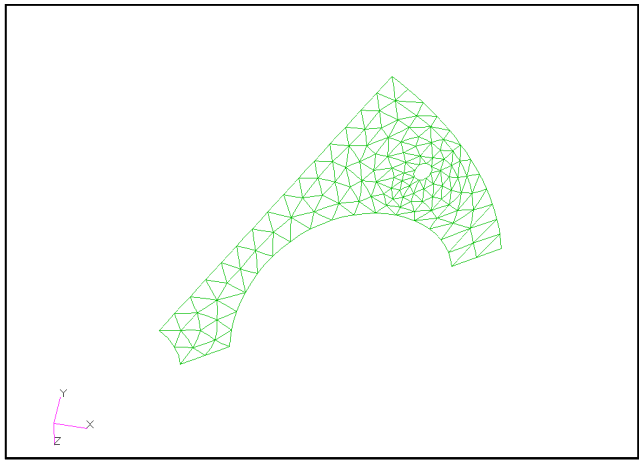


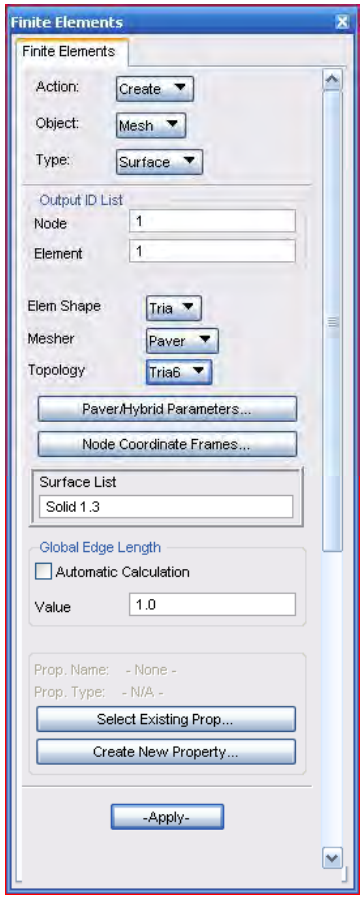


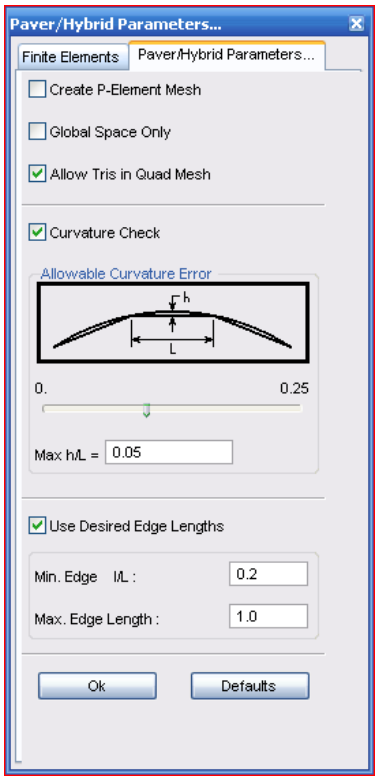
Copyright© 2012 MSC.Software Corporation

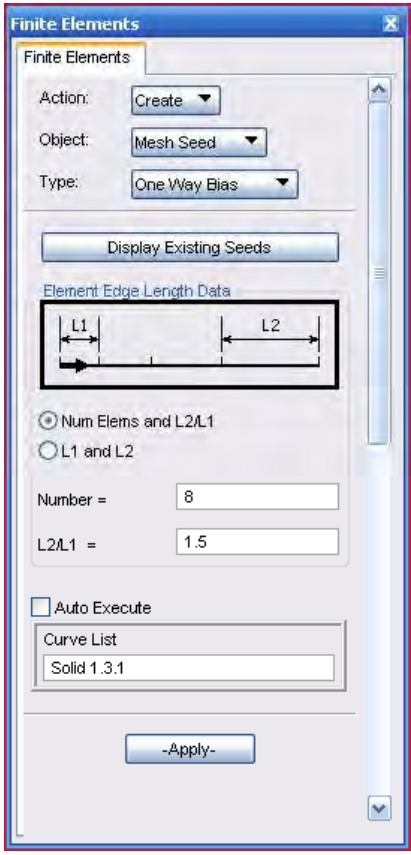
CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

- **"Trial and error" effort using**
  - Paver Parameters control
  - Mesh seeding on the hole edge
  - Other, e.g. hard geometry
- **Suitable 2D tria6 mesh is obtained**





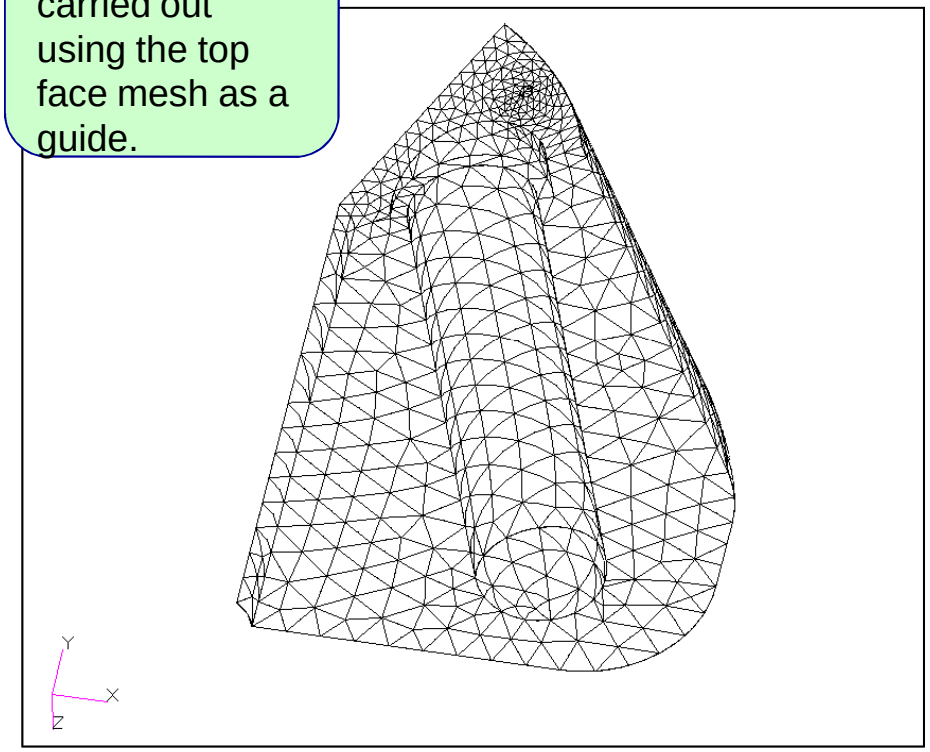


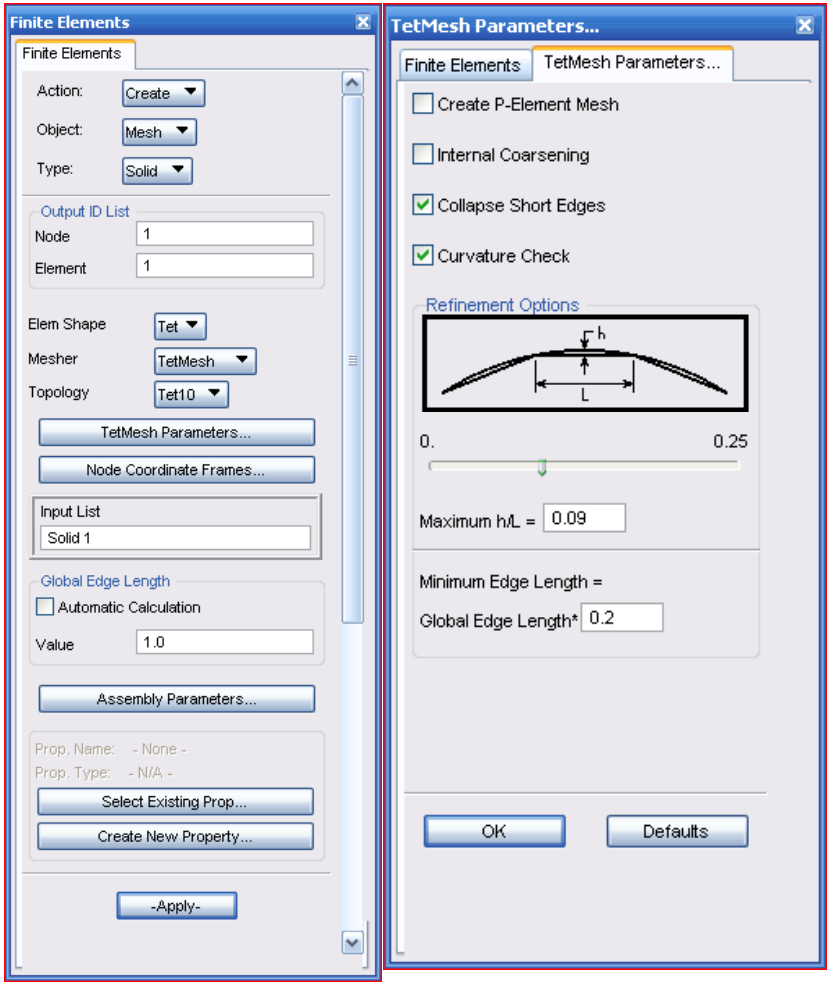


Copyright© 2012 MSC.Software Corporation

CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

The 3D meshing is now carried out using the top face mesh as a guide.

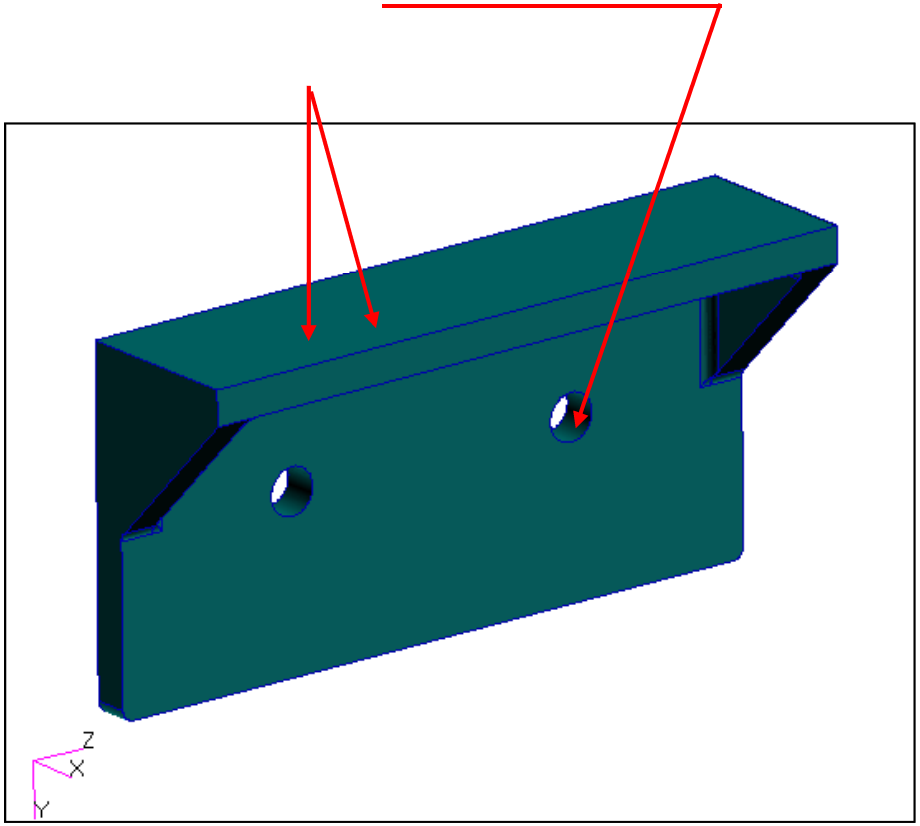




Copyright© 2012 MSC.Software Corporation

MORE ON CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

- **Apply load and constraint to model**
  - Known constraint location
  - Only approximate load location known



Copyright© 2012 MSC.Software Corporation

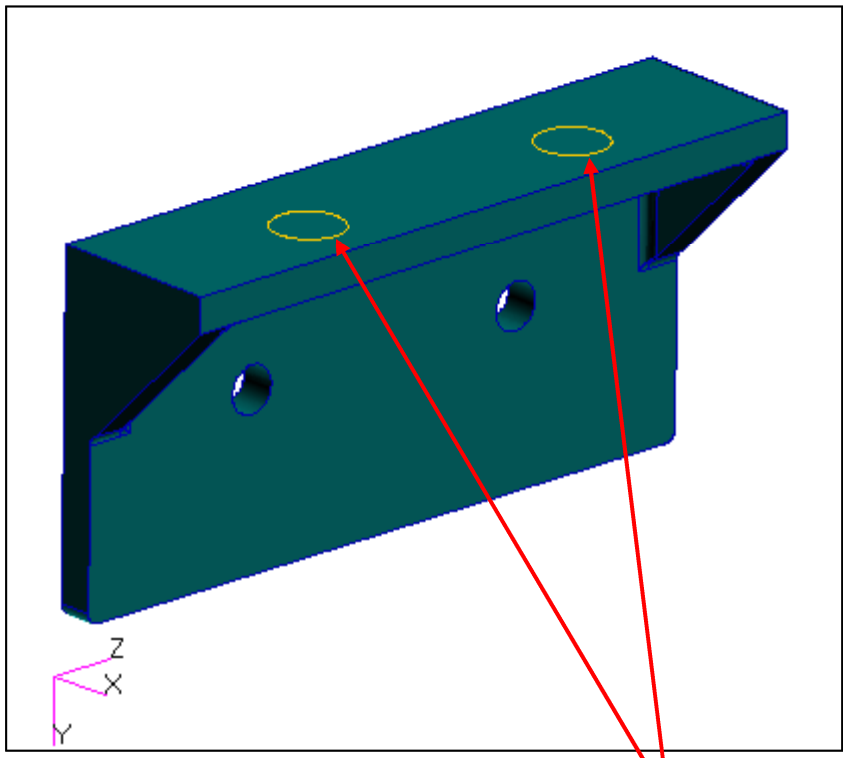
CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

- **A finer mesh is needed at areas of stress concentration, e.g. load, constraint, and/or load paths areas. Coarser meshes elsewhere are acceptable.**
- **For this example, the exact location of loading is not known**
  - Must approximate the region of loading; make the region larger to allow for uncertainty
- **Determine approximate load paths**
  - Where are the areas of constraint?
  - Where are the areas of load?
  - Where are the areas of stress concentration?
  - How are the areas related by the structure?
- **With the information given above, estimate the load paths for this model.**

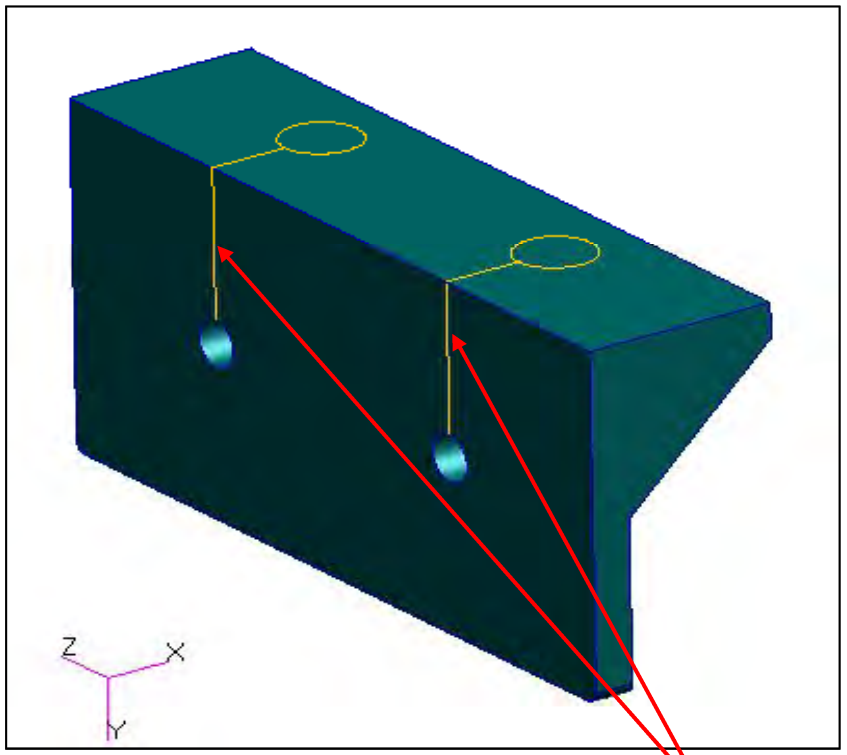
Copyright© 2012 MSC.Software Corporation

CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

- **Shown below are approximate loading areas**
- **Also shown are qualitative (approximate) load paths**



Loading areas

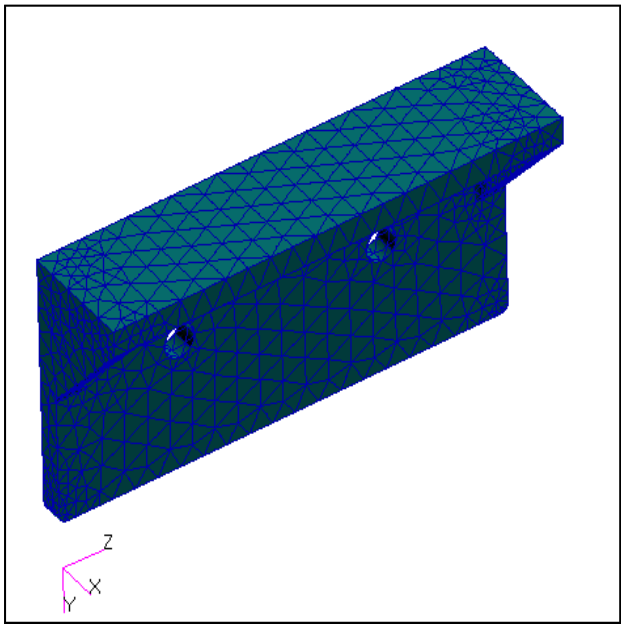


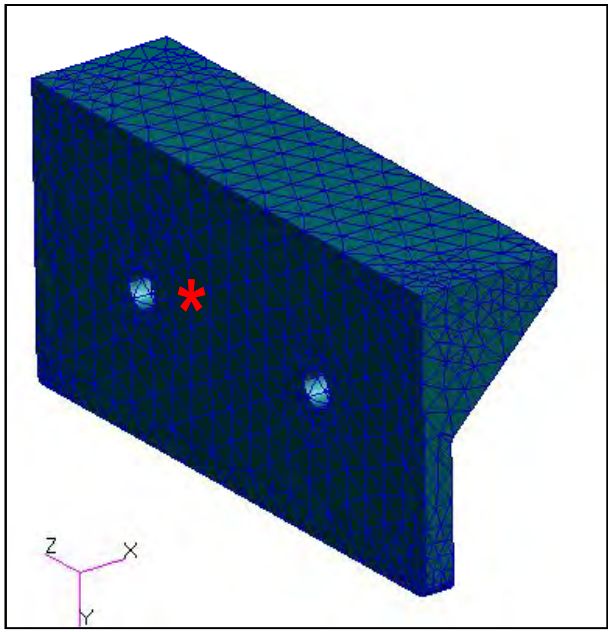
Approximate load paths

Copyright© 2012 MSC.Software Corporation

CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

- **TetMesh without any control of mesh, other than Curvature Check.**
  - The only increase in mesh density is at the holes where the constraints (*) will be applied.





Copyright© 2012 MSC.Software Corporation

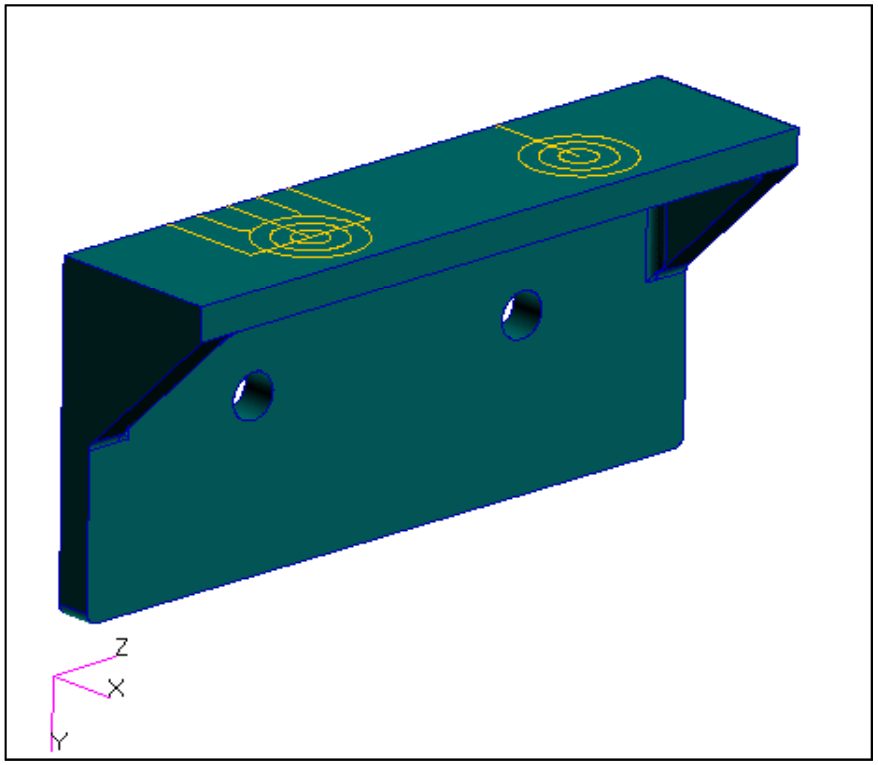
CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

- **Increase tet mesh density at areas of stress concentration**
- **First approach -- create solid faces at areas**
  - Disassemble solid into surfaces
  - Break the surfaces into smaller surfaces
  - Re-assemble the solid, e.g. create B-rep solid
  - This can be labor intensive for complicated solids
- **Second approach – associated curves at areas**
  - Create a set of curves at/along areas
  - Associate the curves to current solid faces
  - Paver mesh the solid faces
    - GEL for creating larger elements away from associated curves
    - Associated curves control local element size, so they are smaller
      - Use mesh seeds on associated curves
  - TetMesh the solid

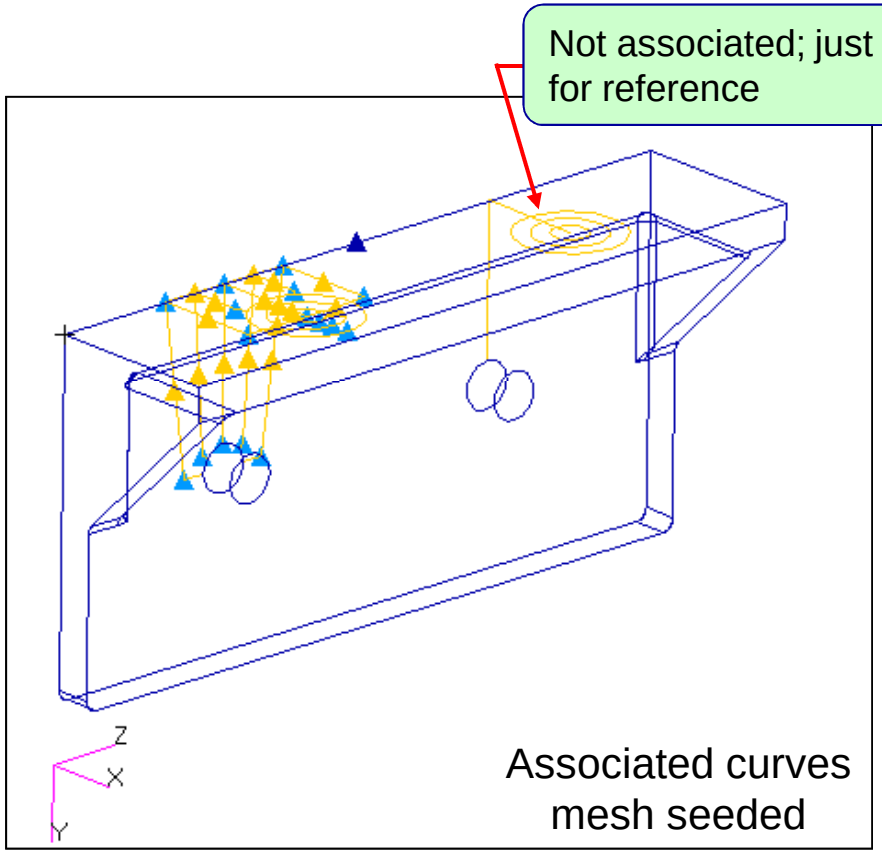
Copyright© 2012 MSC.Software Corporation

CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

- **Geometric solid with curves associated to two solid faces (top and back):**



Shaded solid with curves

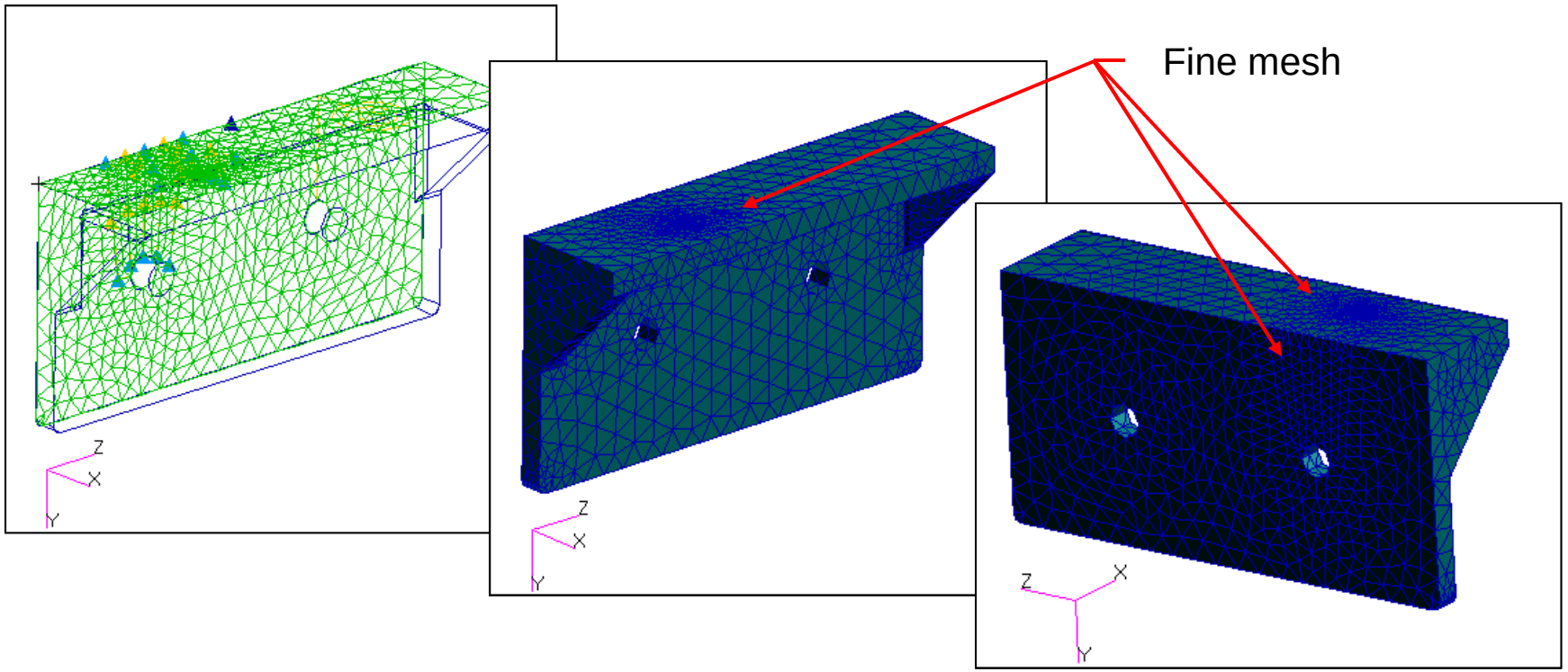


Solid with associated curves

Copyright© 2012 MSC.Software Corporation

CONTROL TETMESH AT DETAIL USING 2D ELEMENTS

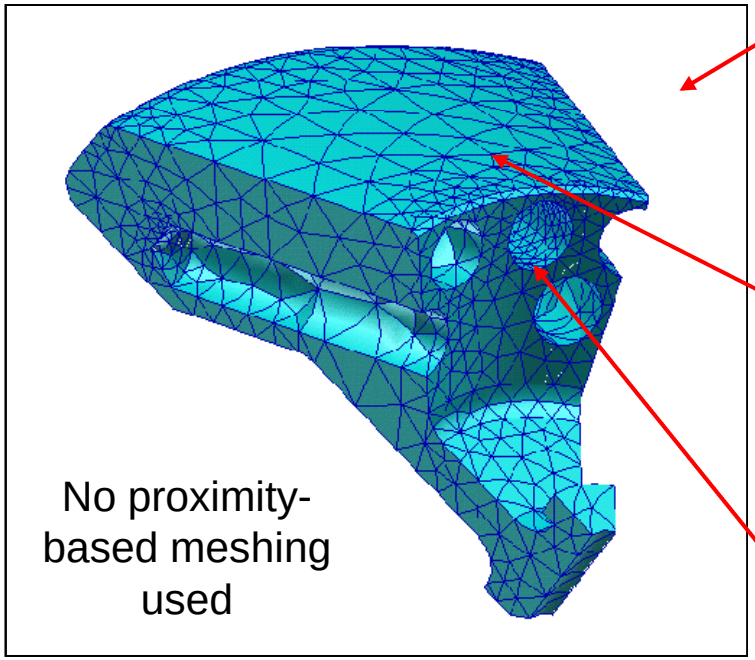
- **Paver mesh two surfaces with Tri elements, and TetMesh solid with Tet elements:**



Copyright© 2012 MSC.Software Corporation

CONTROL TETMESH USING PROXIMITY BASED MESHING

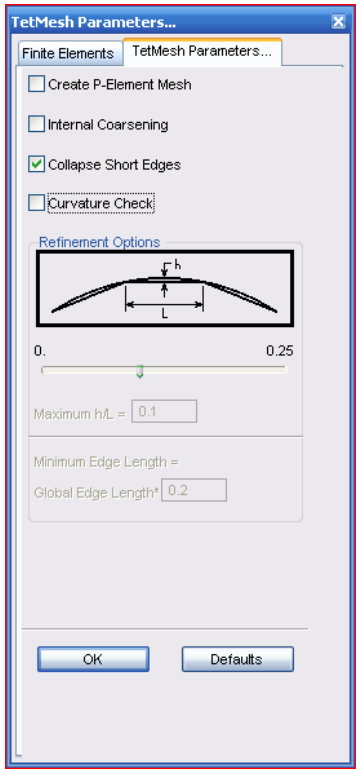
- **Proximity-based meshing**
  - Control the density of tet meshes using fine tri element meshes on solid face(s)
    - Forces fine tet mesh on face(s) opposite fine tri element mesh



No Curvature Check, no proximity-based meshing

Coarse tet mesh does not “match” fine tri mesh

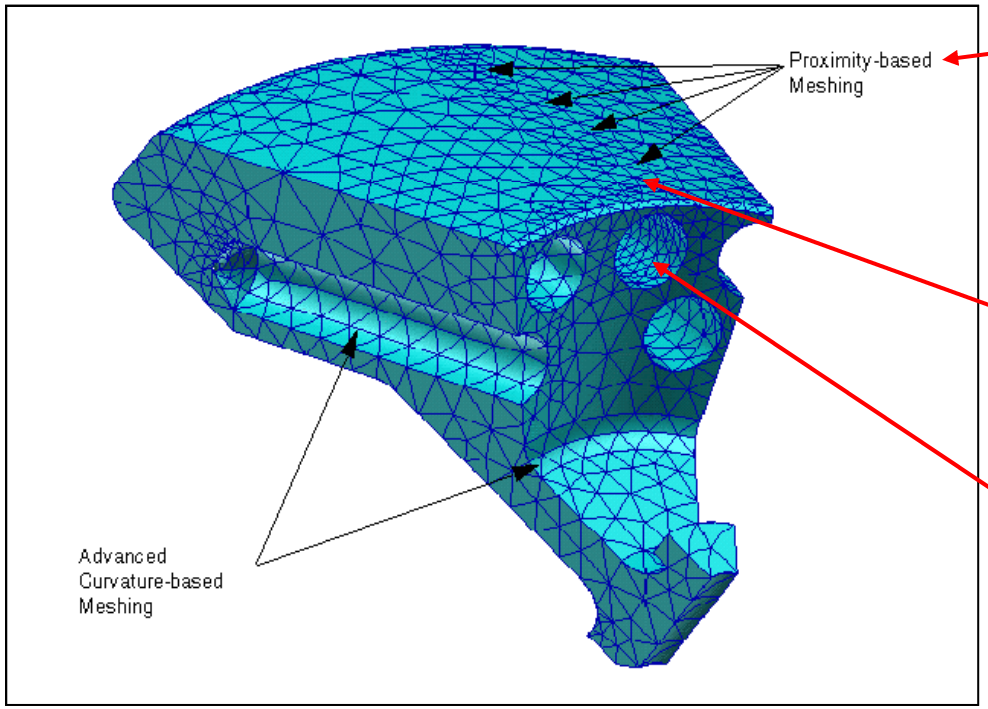
Fine tri mesh inside hole



Copyright© 2012 MSC.Software Corporation

CONTROL TETMESH USING PROXIMITY BASED MESHING

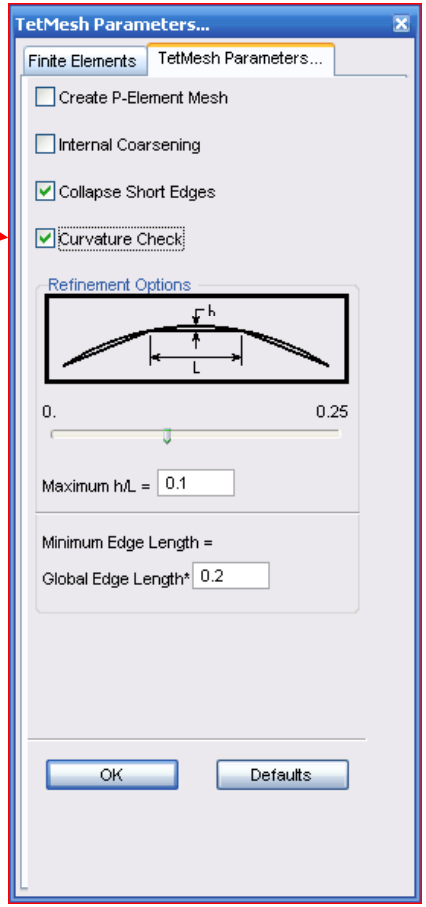
- **Under TetMesh Parameters, using Curvature Check causes proximity-based meshing to be used**



Curvature Check enables proximity-based meshing

Fine tet mesh “matches” fine tri mesh

Fine tri mesh inside hole



Copyright© 2012 MSC.Software Corporation

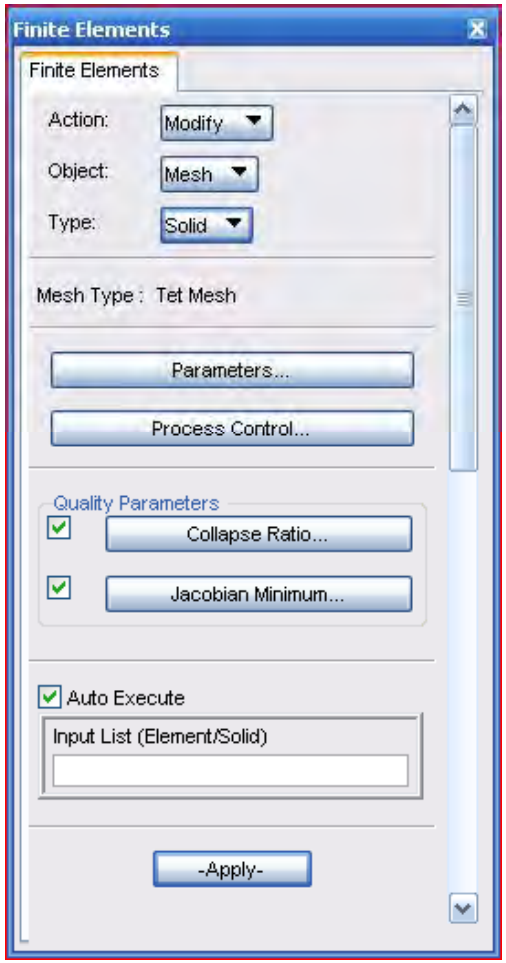
3D MESH EDITING

- **Start with a 3D mesh, since it can be modified as needed using tools such as Modify/Node/Move (or Offset or Edit or Project), or Modify/Element/Edit (or Reverse or Separate), e.g. Modify/Element/Edit Type to change the topology of the elements.**
- **A tetrahedral mesh can be modified automatically to improve the element shapes.**

Start Point 3D	**Nothing**		**Mesh**	**Geometry**		
Method	**Extrude, Glide, etc. Methods**	**B-Rep or Primitive Construction**	**Mesh Editing**	**TetMesh**	**IsoMesh**	**Geometry breakdown and rebuild methods**
End Point 3D	**Simple Geometry**	**Complex Geometry**	**Mesh**	**Mesh**		**Geometry**

Copyright© 2012 MSC.Software Corporation

MODIFY TETMESH USING COLLAPSE RATIO AND/OR JACOBIAN PARAMETER



- **After a tet mesh has been created, elements can be modified if some elements have too small a collapse ratio and/or minimum Jacobian value.**
- **Specify Hard Nodes (not to be moved)**
- **FEM Volume Control**
  - Can allow volume of boundary elements to change or force it to remain constant
- **Use of either Collapse Ratio or Jacobian Minimum, or both.**
- **User must specify an Acceptance Value for each.**

Copyright© 2012 MSC.Software Corporation

MODIFY TETMESH USING COLLAPSE RATIO AND/OR JACOBIAN PARAMETER

Issue with Jacobian: when creating a TetMesh, it is possible to generate an element with a negative Jacobian due to complex geometry (highly curved edges and/or faces).

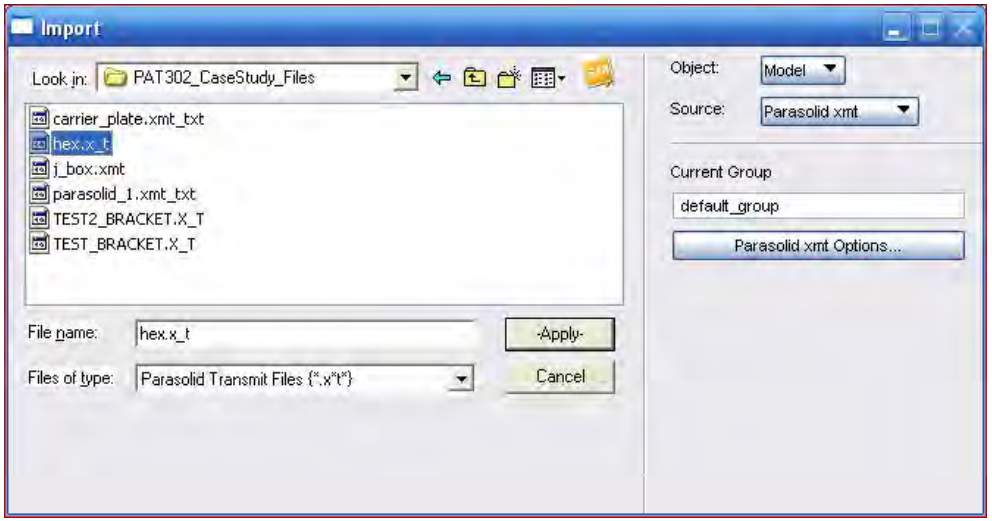
To deal with this problem, the Elements: Modify / Mesh / Solid Jacobian Minimum changes the edges of the elements so they are straighter.

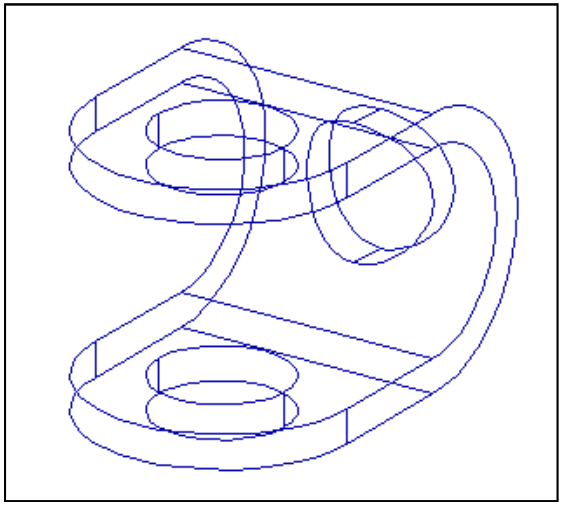
This may resolve the problem of the negative Jacobian, but the element edges may no longer follow the curved geometry. It *may* be best to create local mesh control (mesh seeds) and re-mesh with a smaller element size.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

- **Import the following Geometry:**
  - Pull down **Import…** under the *File* drop down menu
  - Set *Source* to **Parasolid xmt**
  - Select **hex.x_t**.
  - Click **Apply**.
  - Switch to **Iso 3** view using the given icon

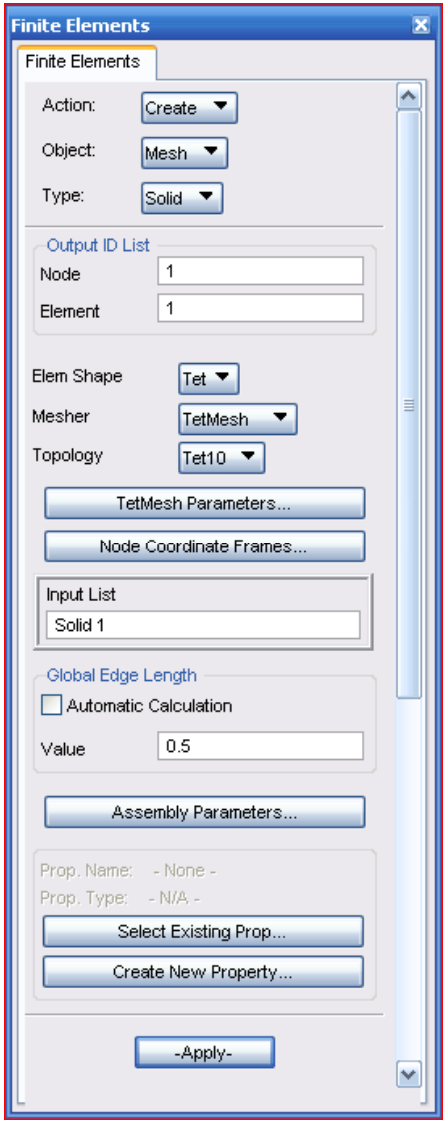




Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

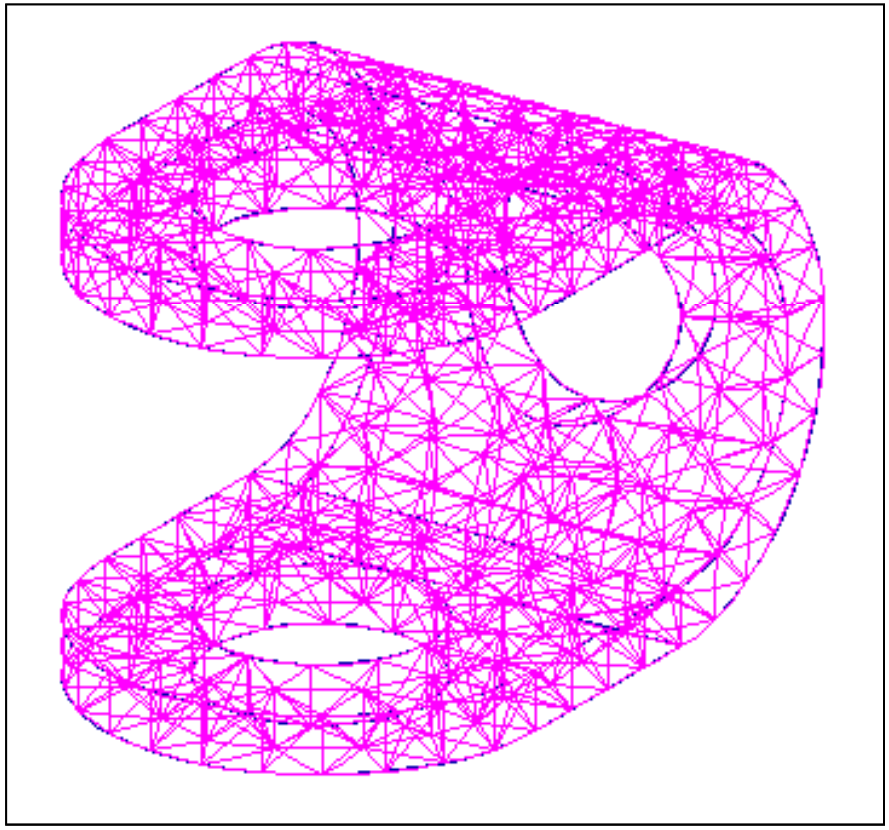
- **TetMesh the parasolid solid using default parameter values:**
  - Under the *Meshing* tab click on **Solid** from the *Meshers* group.
  - Select **Tet** for *Elem Shape*.
  - Select **TetMesh** for *Mesher*.
  - Select **Tet10** for *Topology*.
  - Select **Solid 1** for *Input List*.
  - Enter **0.5** for *Global Edge Length*.
  - Click **Apply**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

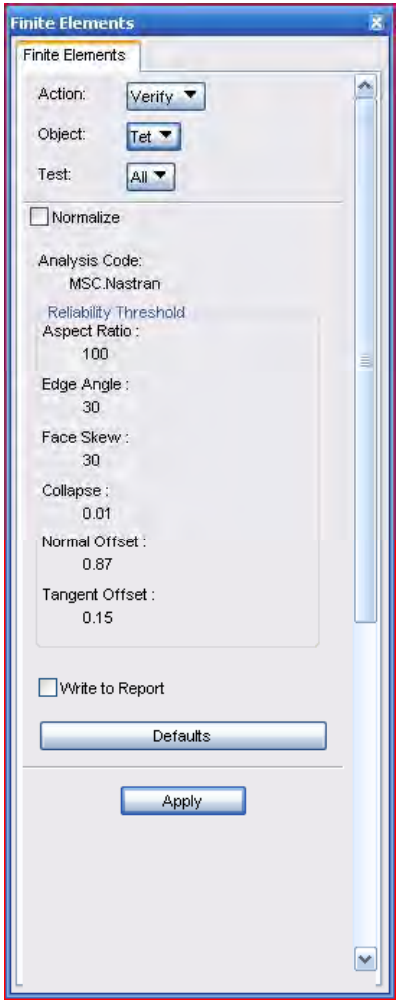
- **Your model should now appear as follows.**

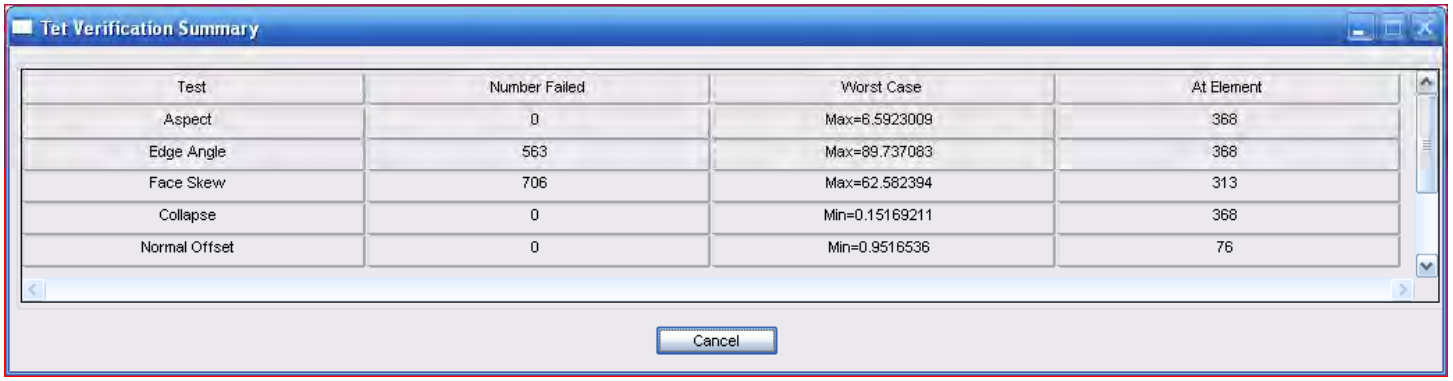


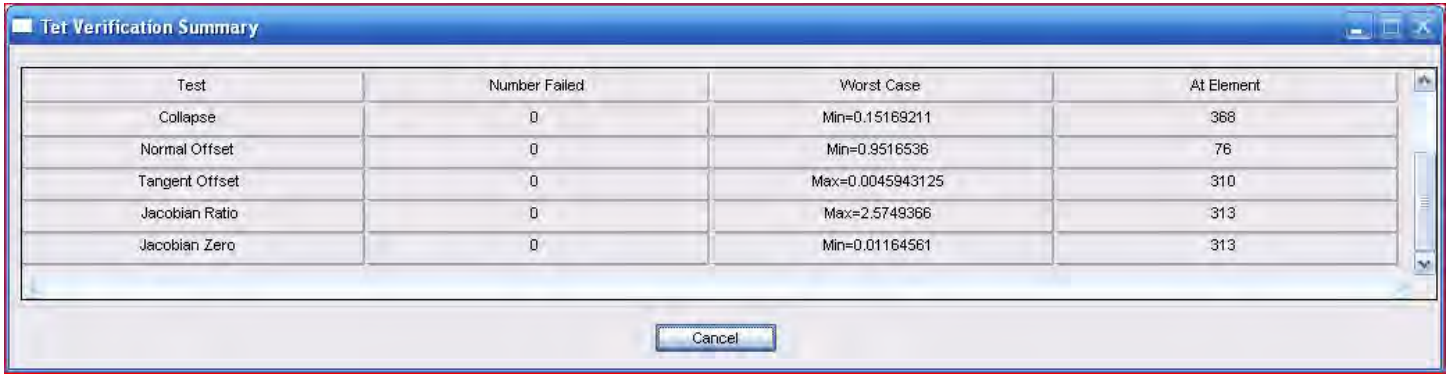
Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

- **Determine the quality of the tet10 Mesh**
  - Under the *Meshing* tab click on **Verify** from the *FEM Actions* group. Set *Object* and *Test* as **Tet** and **All**, respectively.
  - Click **Apply**



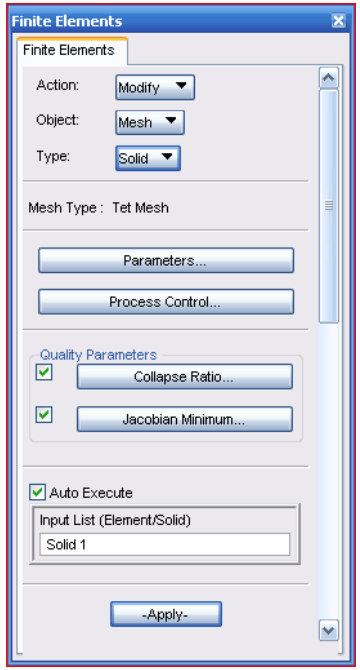


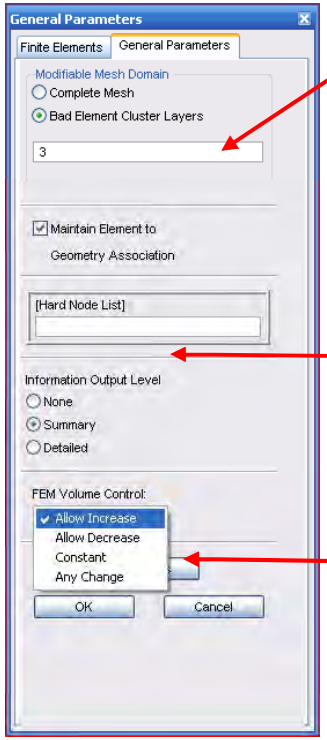


Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

- **Modify the tet10 mesh by specifying the minimum collapse ratio:**
  - Under the *Meshing* tab: Set *Action, Object,* and *Type* as **Modify, Mesh,** and **Solid,** respectively.
  - Click on **Parameters…**
    - Use default settings





Number of layers of elements around bad tet element to be used as modifiable region (region where tets can be changed)

List of nodes that are not to be moved

Only for element edges on mesh boundary not associated to geometry

Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

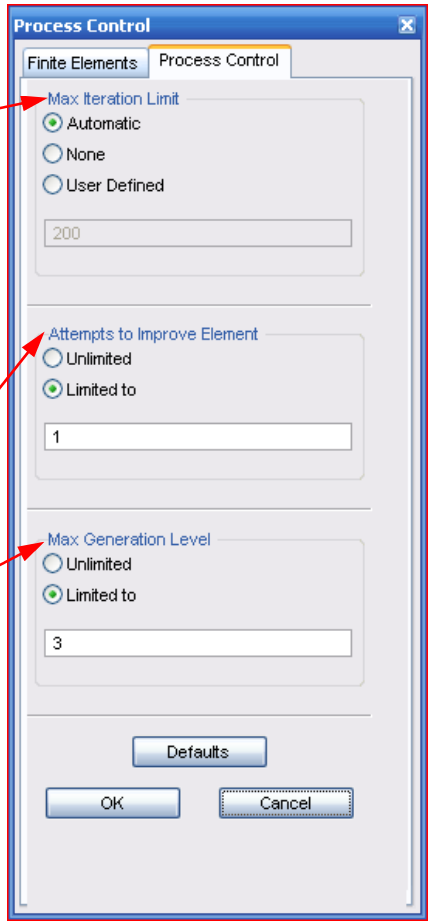
- Click **Process Control…**
  - Use default settings.

Maximum number of allowed iterations

Automatic – limit based on number of bad els = 100 * number bad els

User Defined – if = 0, mesh evaluated but no change; if N(> 0) maximum of N modifications

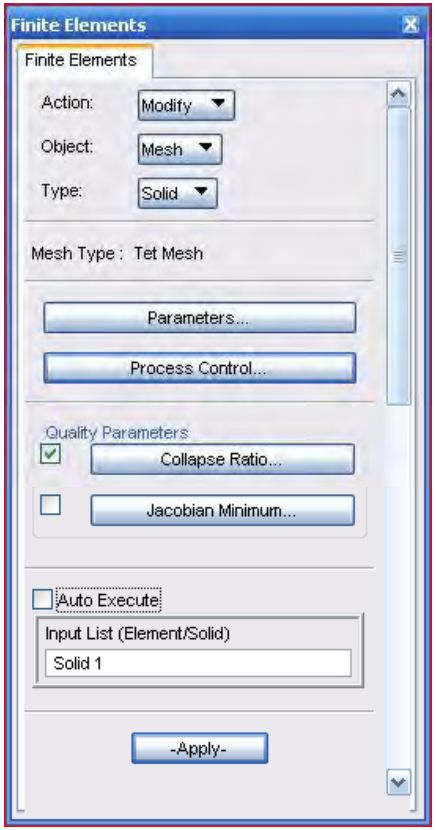
Has to do with number of times an element can be replaced by a better shaped element

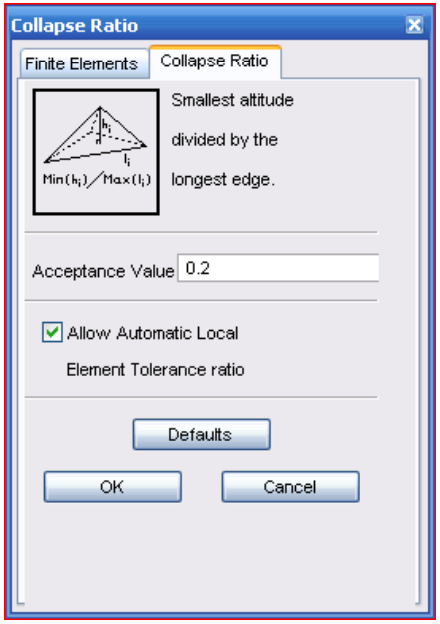


Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

- **Modify the tet10 mesh**
  - Click **Collapse Ratio…** and enter **0.2** for *Acceptance Value*. There is a limit of 0.2 for the algorithm used in Patran.
  - Deselect **Jacobian Minimum…**
  - Select **Solid 1** for *Input List (Element/Solid)*.
  - Click on **Apply**.

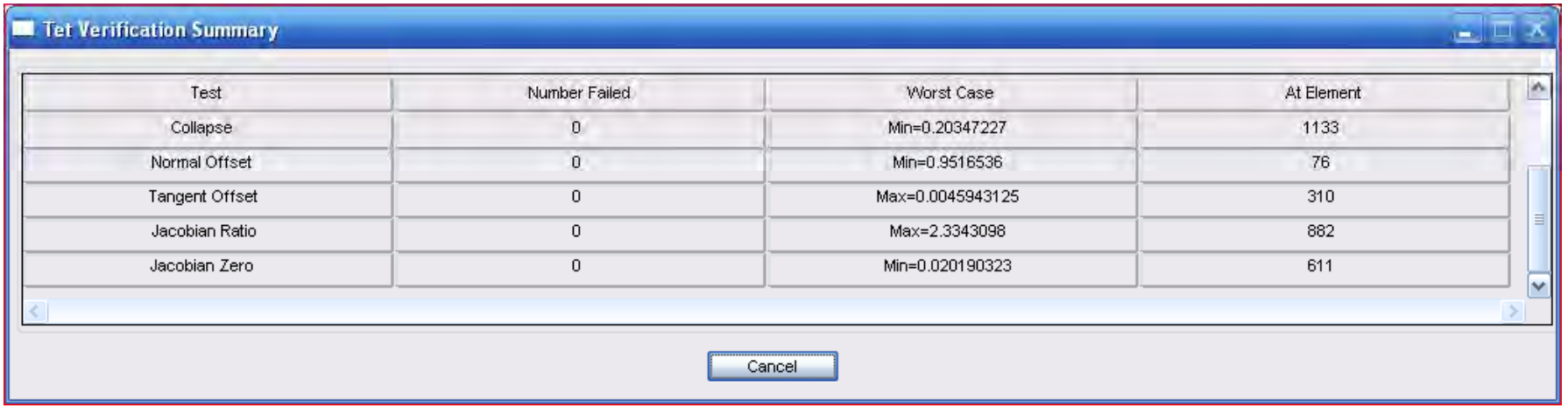




Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

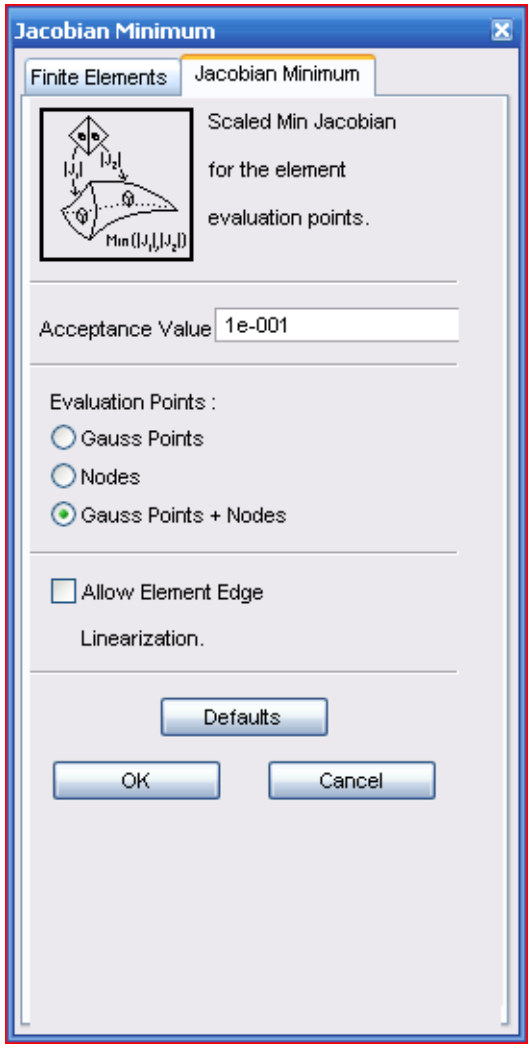
- **Verify results for collapse ratio only.**
  - The minimum collapse ratio is now closer to 0.2



Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

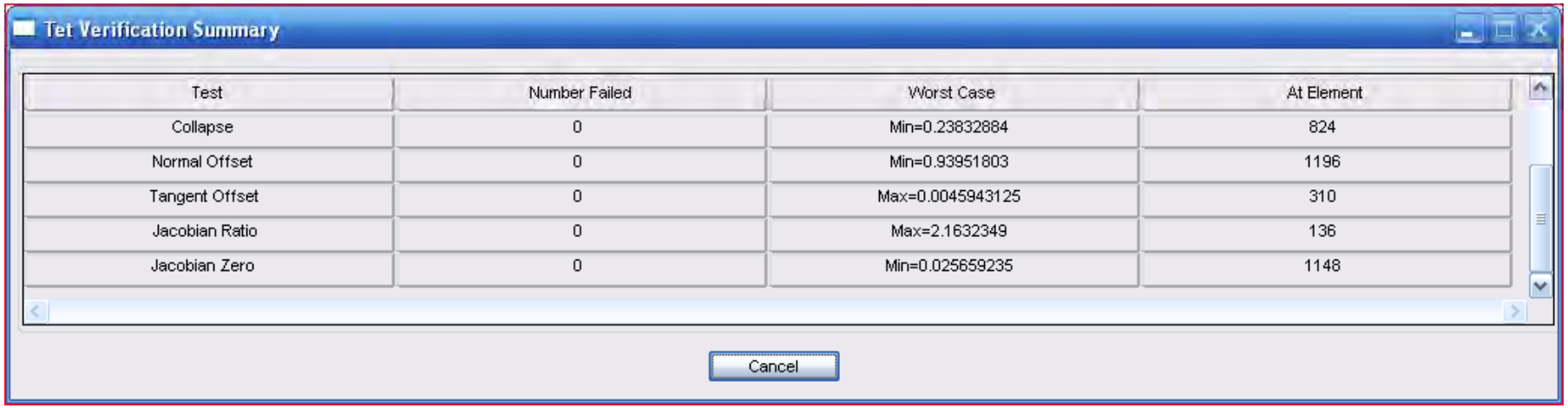
- **Continue improving the tet mesh**
  - Click on **UNDO** button.
  - Turn the **Collapse Ratio** toggle off.
  - Toggle on **Jacobian Minimum**, and click on button.
    - Set the *Acceptance Value* as **0.1**
  - Click **Apply**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

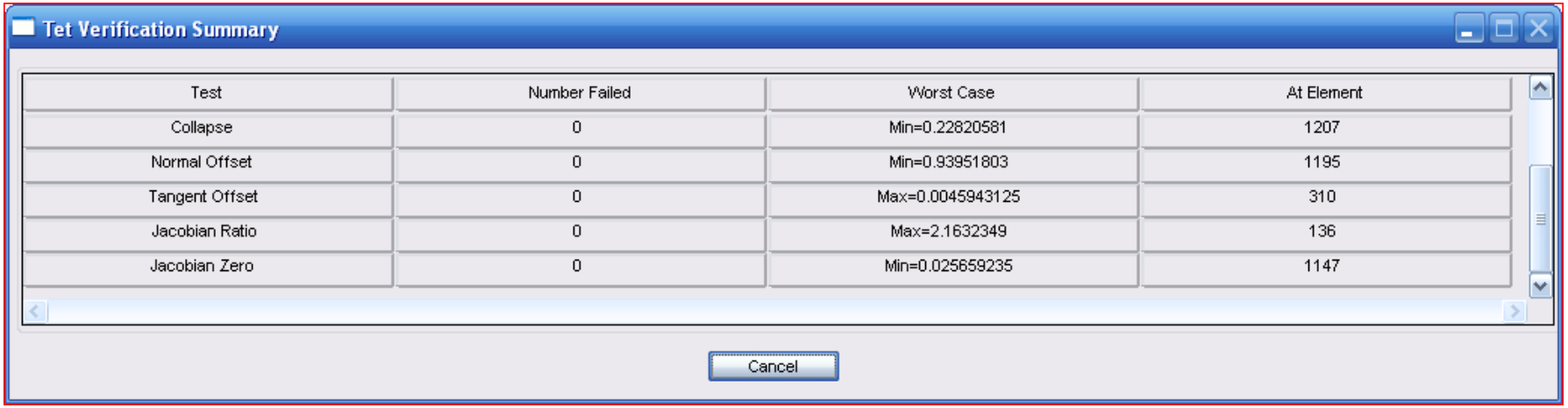
- **Verify results for Jacobian minimum only.**
  - Notice that the collapse ratio > that for the prior modification.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

- **Continue improving the tet mesh:**
  - Click on **UNDO** button.
  - Toggle on both the **Collapse Ratio** and **Jacobian Minimum**, and set both Acceptance Values as before.
  - Click **Apply**.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: IMPROVE TETMESH USING MODIFY

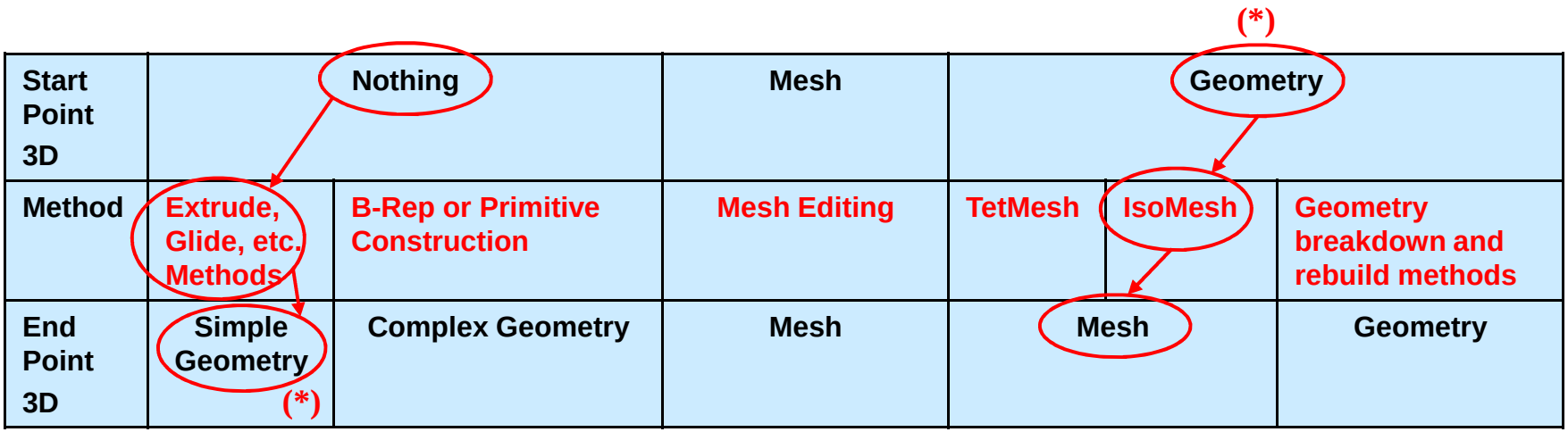
- **Comparison of Verify results for all four combinations of modify parameter values**

Test	Neither	Collapse	Jacobian	Both
Aspect	6.592	4.915	4.196	4.382
Edge Angle	89.737	78.169	78.169	78.169
Face Skew	62.582	62.734	54.192	54.920
Collapse	0.152	0.203	0.238	0.228
Normal Offset	0.952	0.952	0.940	0.940
Tangent Offset	0.005	0.005	0.005	0.005
Jacobian Ratio	2.575	2.334	2.163	2.163
Jacobian Zero	0.012	0.020	0.026	0.026

Copyright© 2012 MSC.Software Corporation

3D ISOMESH & SOLID TOPOLOGY

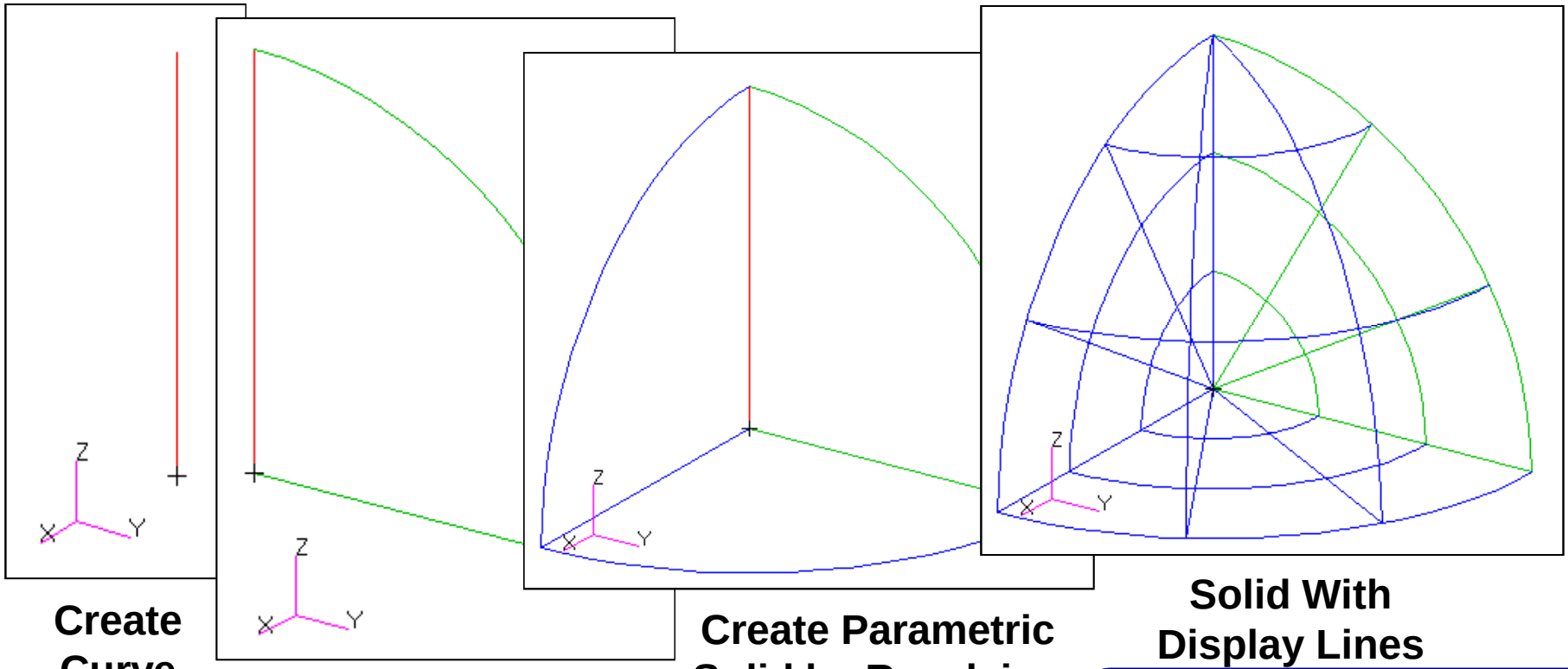
- **The next set of techniques involve creating solids using different topological approaches. The solids are meshed using IsoMesh.**



Copyright© 2012 MSC.Software Corporation

3D ISOMESH & SOLID TOPOLOGY

- **Create 1/8 of sphere for IsoMesh with Hex elements**



Create Curve

Create Surface by Revolving

Create Parametric Solid by Revolving

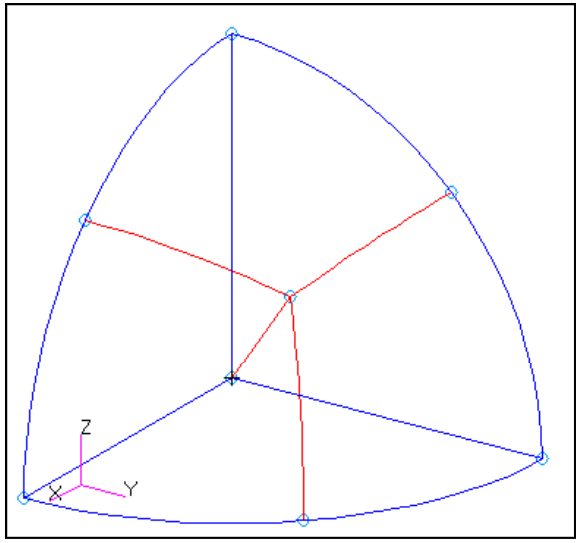
Solid With Display Lines

Cannot Hex mesh this solid with the IsoMesh because of geometry topology; only IsoMesh with Tet elements is possible.

Copyright© 2012 MSC.Software Corporation

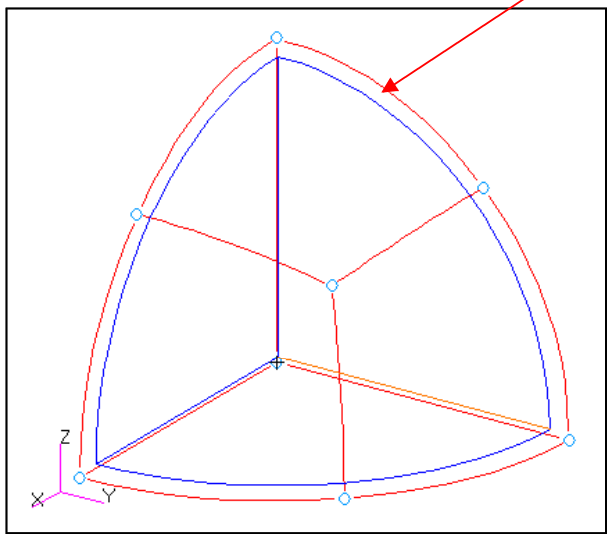
3D ISOMESH & SOLID TOPOLOGY

- **Create 1/8 of sphere for IsoMesh with Hex elements**



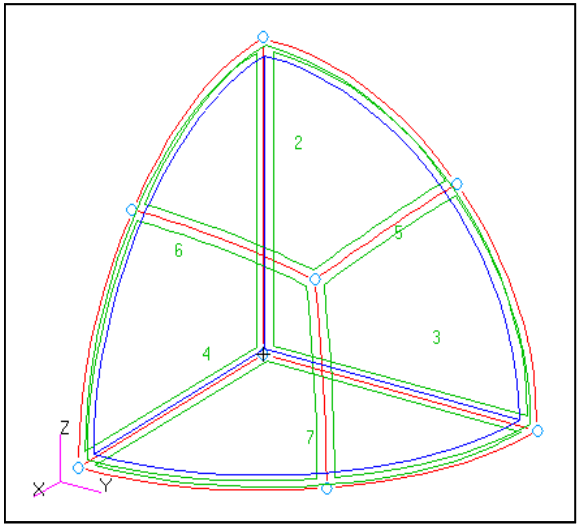
Divide current 1/8 solid into equal 1/3 regions

Create a curve from [0,0,0] to [1,1,1], intersect curve with solid face, extract point at parametric 0.5 on three curved edges, and create three curves on the face between points.



Create curves for four-edged surfaces

Break three edges at parametric 0.5 using existing points



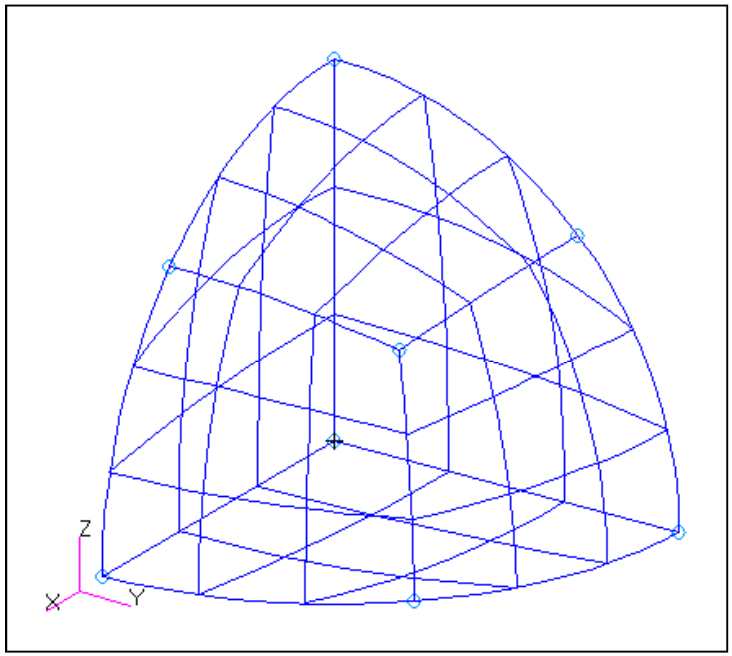
Create six four-edged surfaces

Use Geometry: Create/Surface/Edge Option: 4 Edge

Copyright© 2012 MSC.Software Corporation

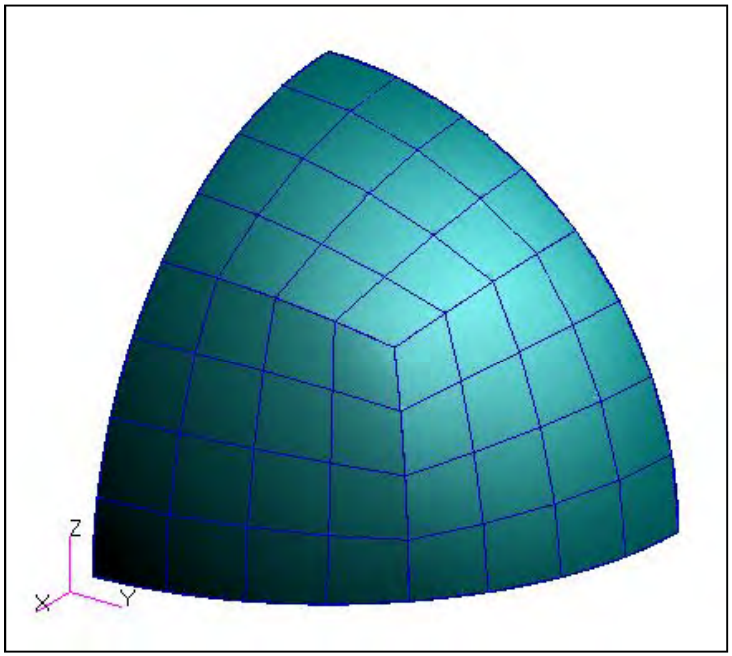
3D ISOMESH & SOLID TOPOLOGY

- **Create 1/8 of sphere for IsoMesh with Hex elements**



Create six-faced parametric solid

Under the Geometry tab: Create / Solid / Face, Option: 6 Face



Hex mesh of solid using IsoMesh

Hex-meshable because of topology

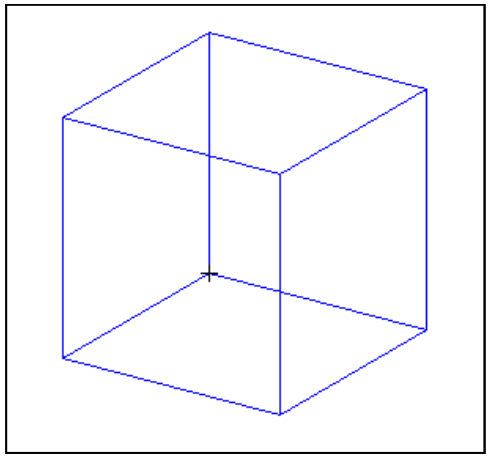
Copyright© 2012 MSC.Software Corporation

3D ISOMESH & SOLID TOPOLOGY

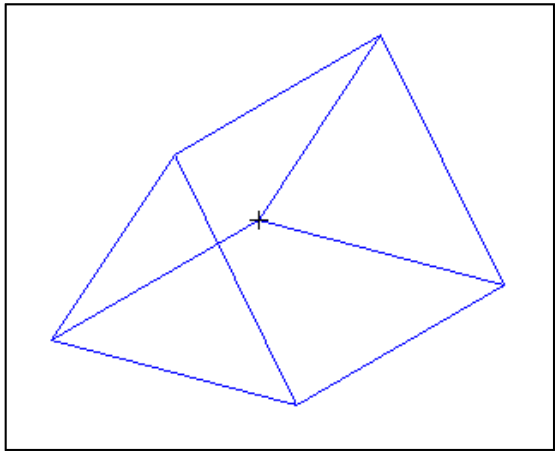
- **To create a parametric solid that will be IsoMeshable with the desired element topology (e.g. Hexahedral), it is necessary to use a procedure that will produce the required geometric topology.**
- **Following are three parametric solids, each with a different topology condition. They are IsoMeshed.**
  - It was possible to use Hex elements for two of the solids.
  - The solid with two degenerate faces could only be meshed with Tet elements.

Copyright© 2012 MSC.Software Corporation

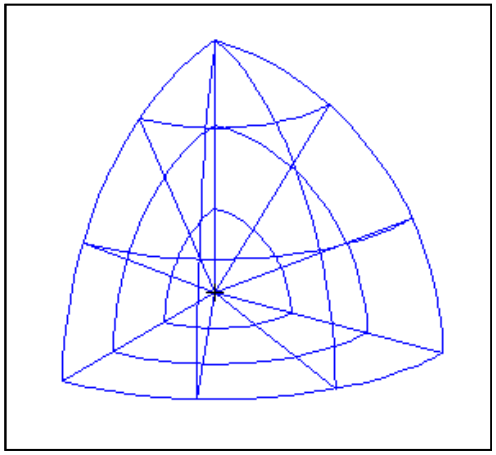
3D ISOMESH & SOLID TOPOLOGY



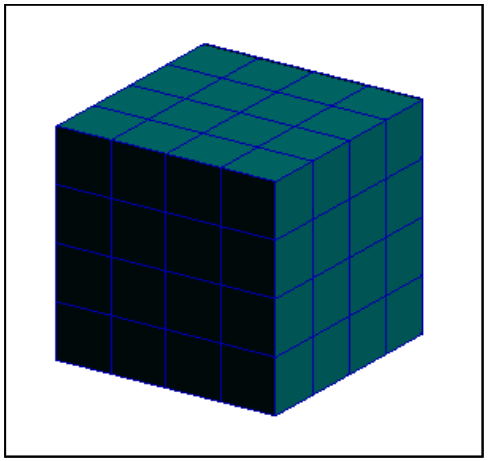
Zero Degenerate Faces



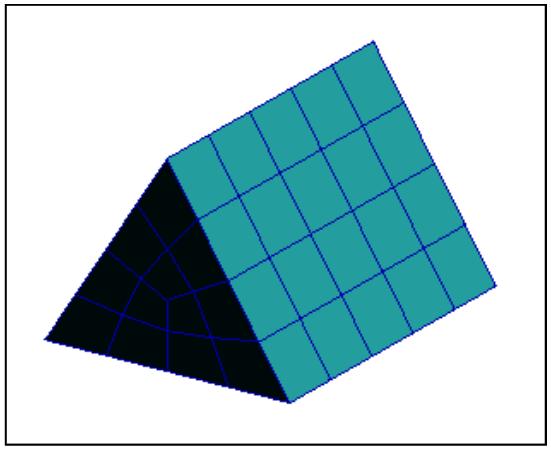
One Degenerate Face



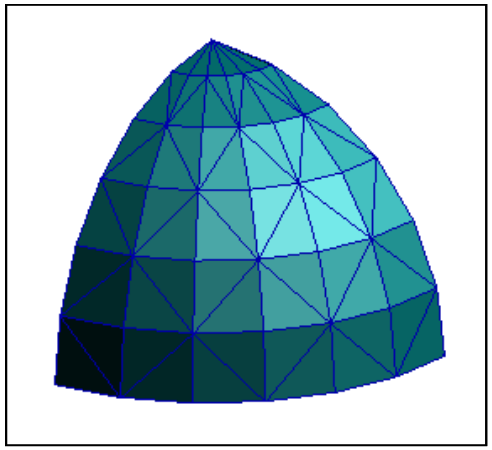
Two Degenerate Faces



Hex Mesh



Hex Mesh

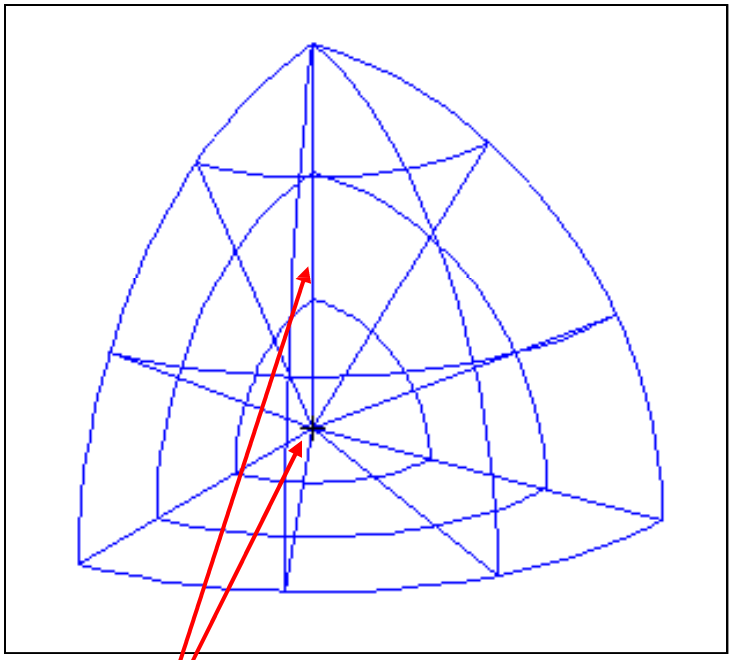


Tet Mesh (Only)

Copyright© 2012 MSC.Software Corporation

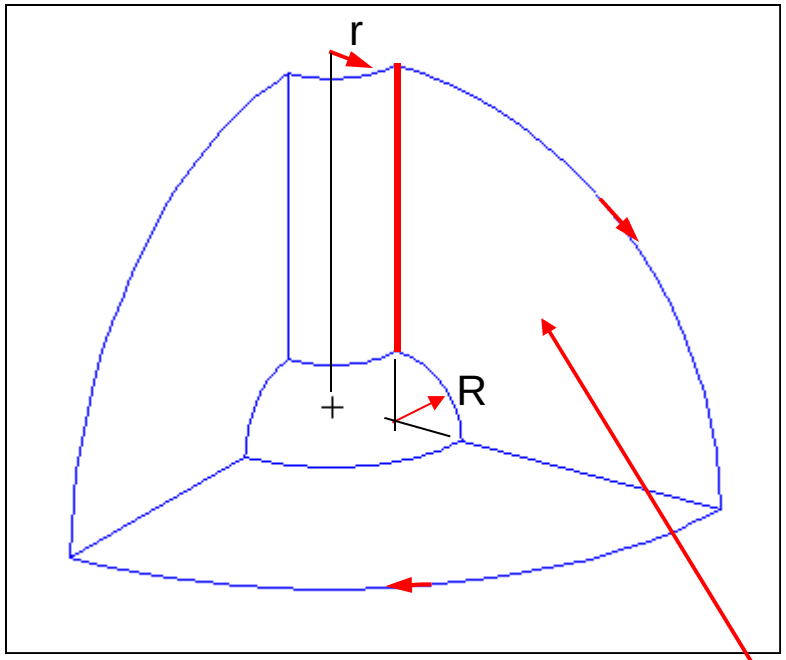
3D ISOMESH & SOLID TOPOLOGY

- **Where are the degenerate faces for the solid with two degenerate faces?**



Degenerate Face

VS

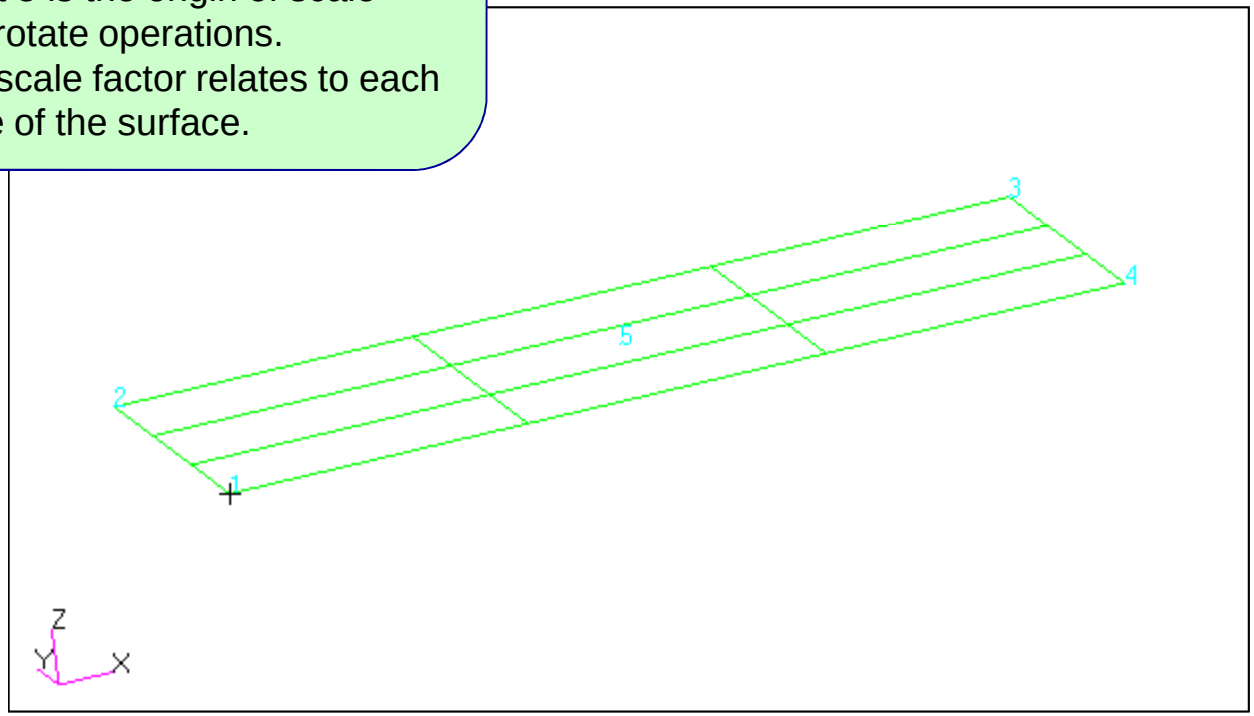


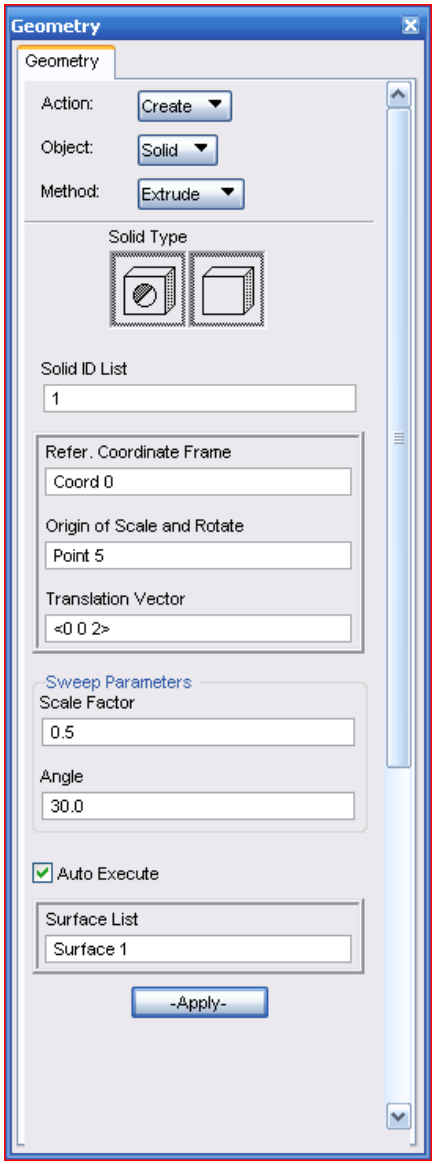
Solid Created:
1) Rotate Curve to Create Surface
2) Rotate Surface to Create Solid

Copyright© 2012 MSC.Software Corporation

3D BEGINNING TO END

The **Solid/Extrude** method can be used for simple or trimmed surfaces.
In this example, a parametric surface is used.
Point 5 is the origin of scale and rotate operations.
The scale factor relates to each edge of the surface.



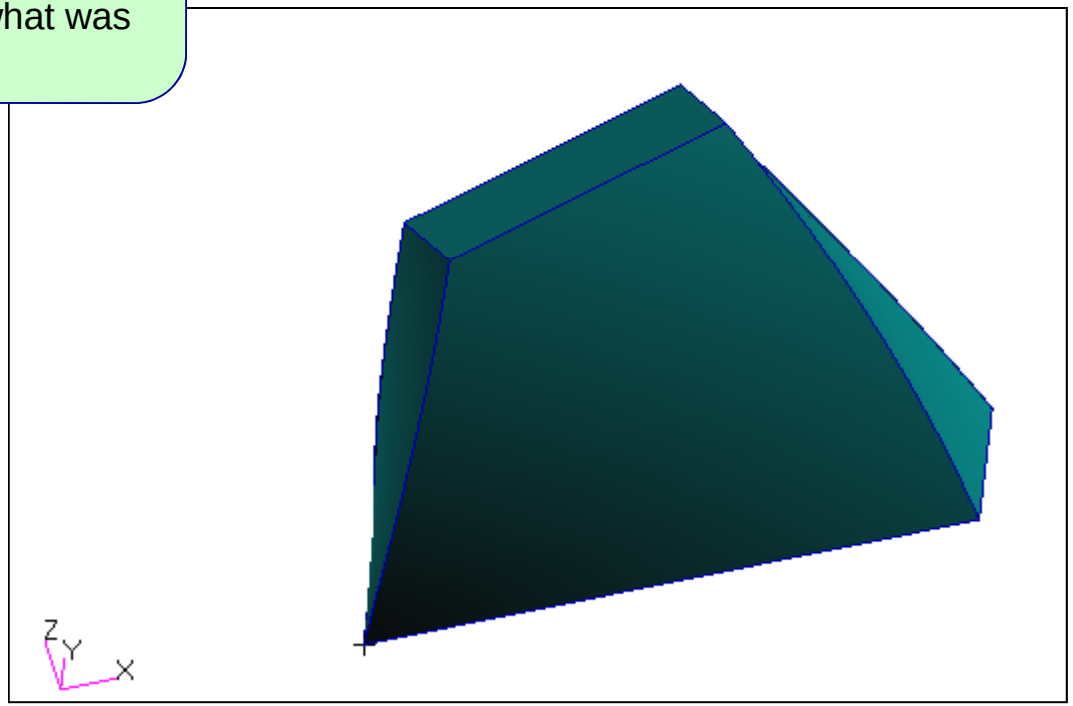


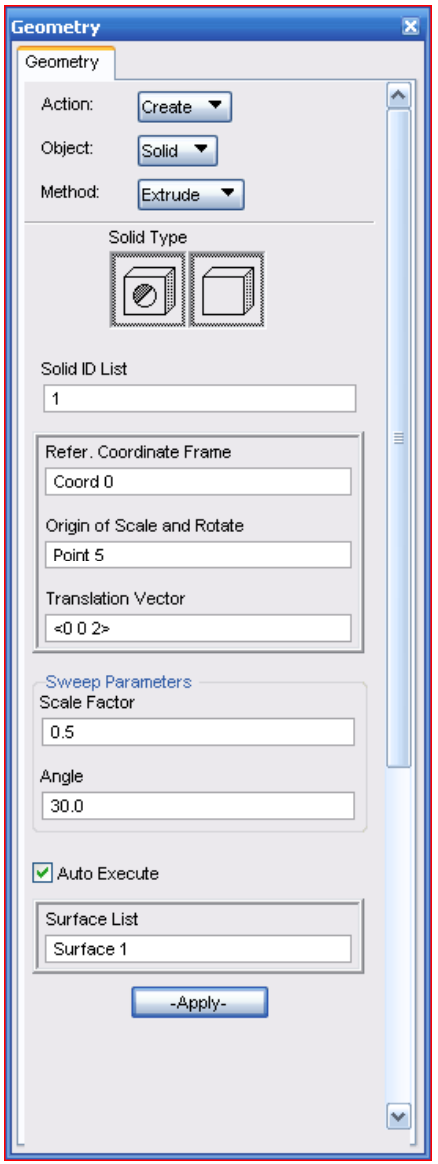
Copyright© 2012 MSC.Software Corporation

3D BEGINNING TO END

The resultant shape is as shown.

The next slide explains what was done.

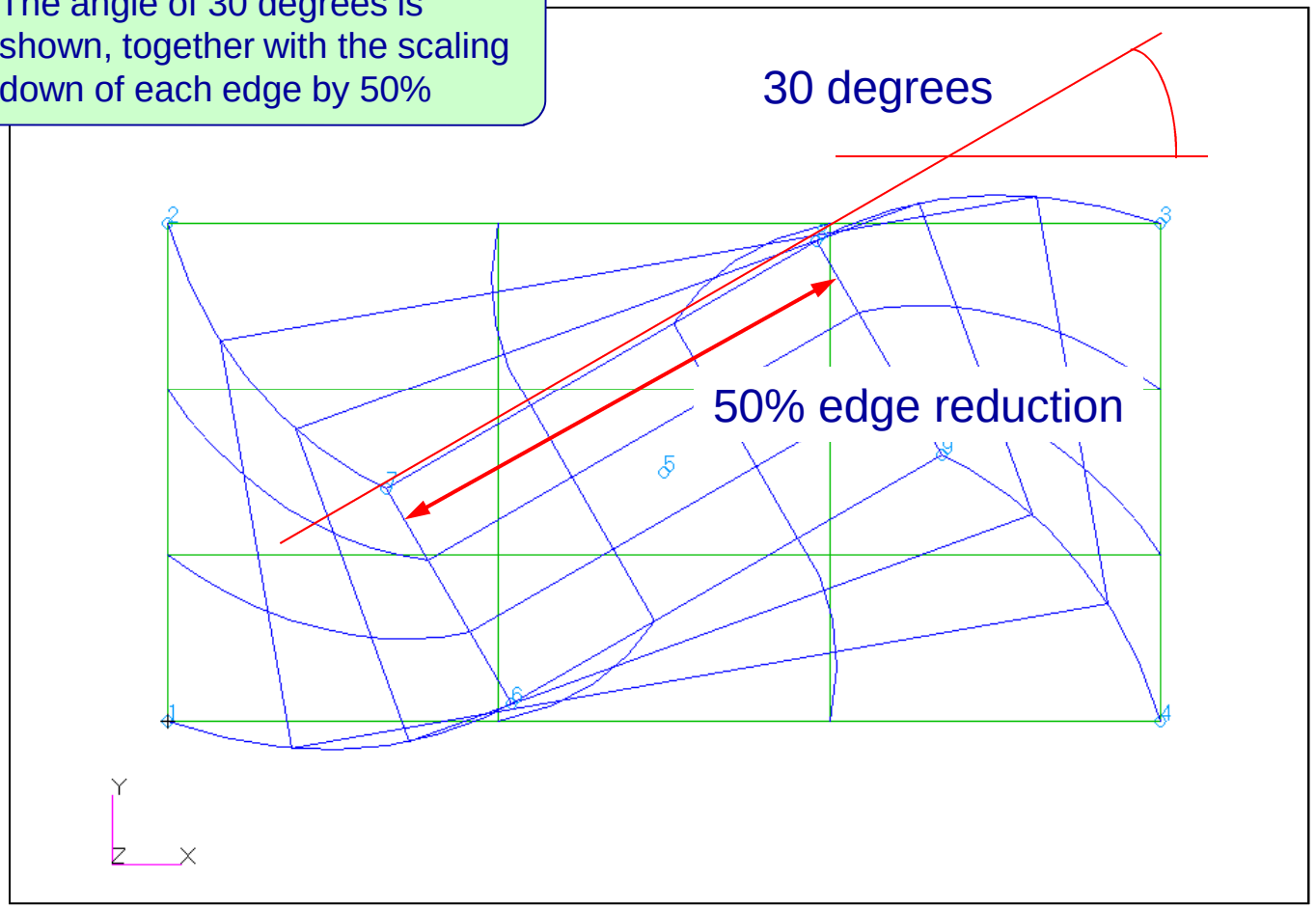




Copyright© 2012 MSC.Software Corporation

3D BEGINNING TO END

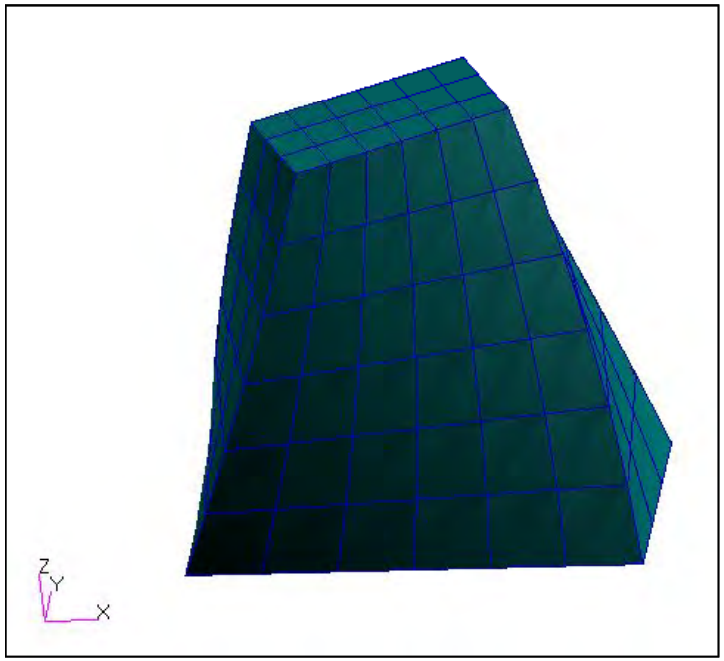
The angle of 30 degrees is shown, together with the scaling down of each edge by 50%



Copyright© 2012 MSC.Software Corporation

3D BEGINNING TO END

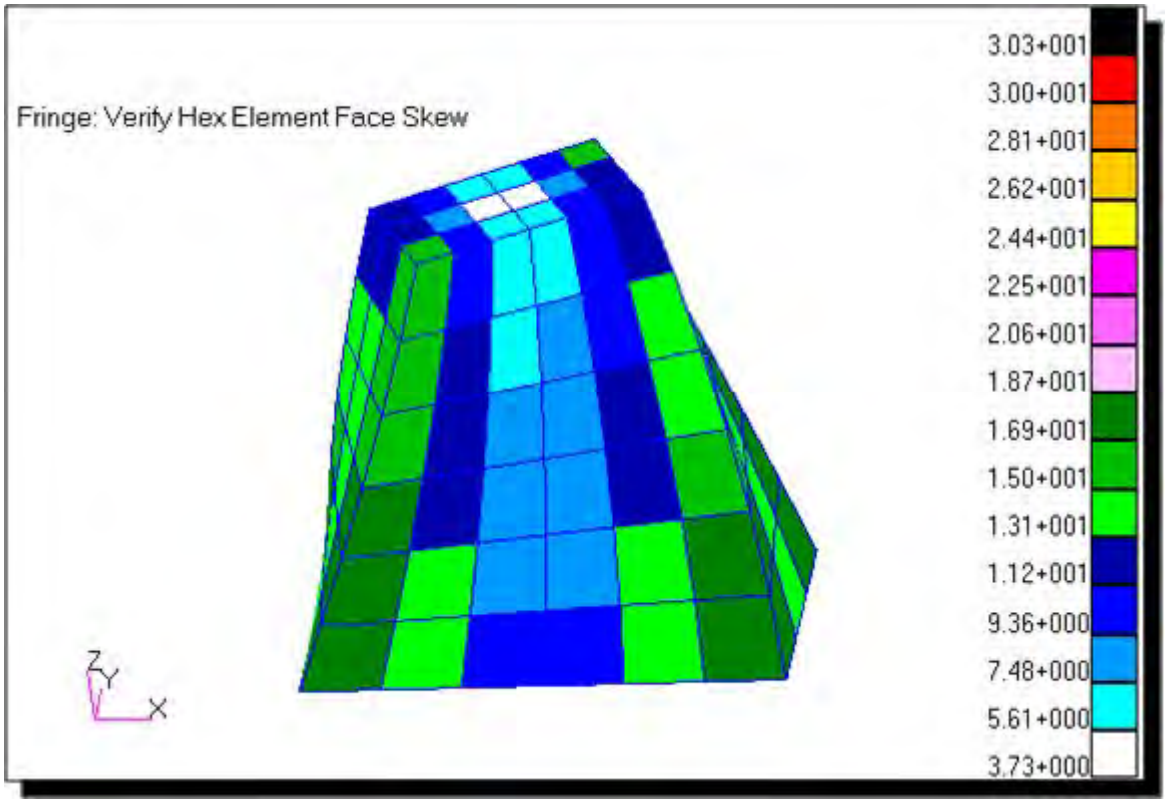
- **IsoMesh the solid with Hex8 elements**
  - It was expected that Hex8 elements would be created because the geometric solid has six non-degenerate faces.
  - However, it was also expected that the Hex8 elements would be skewed because the geometry is skewed.

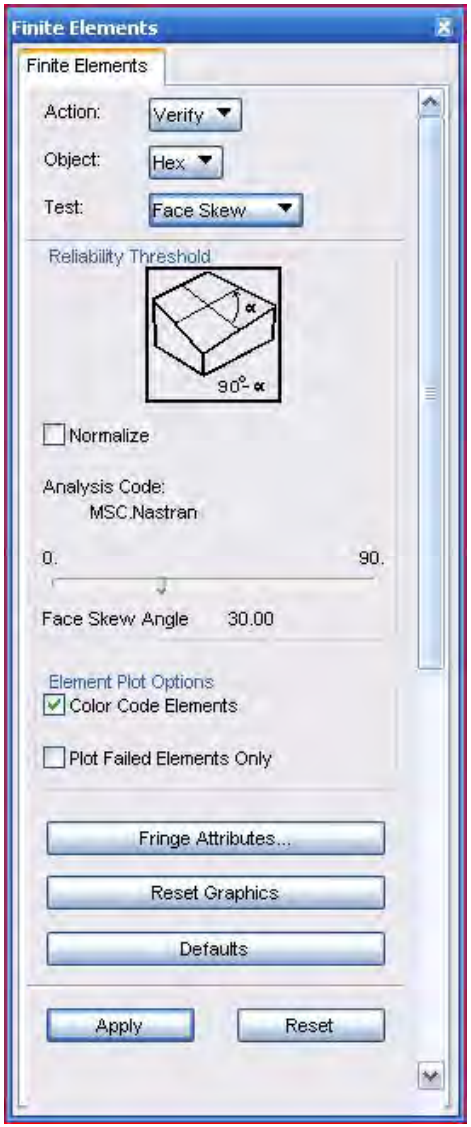


Copyright© 2012 MSC.Software Corporation

3D BEGINNING TO END

- **Check the Hex8 elements for face skew using Verify/Hex/Face Skew under the Meshing tab.**



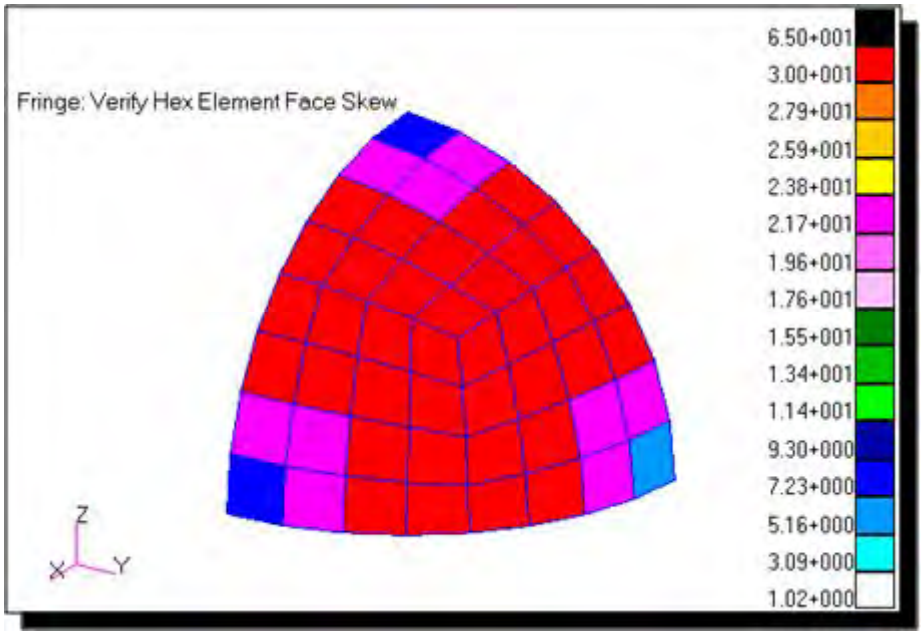


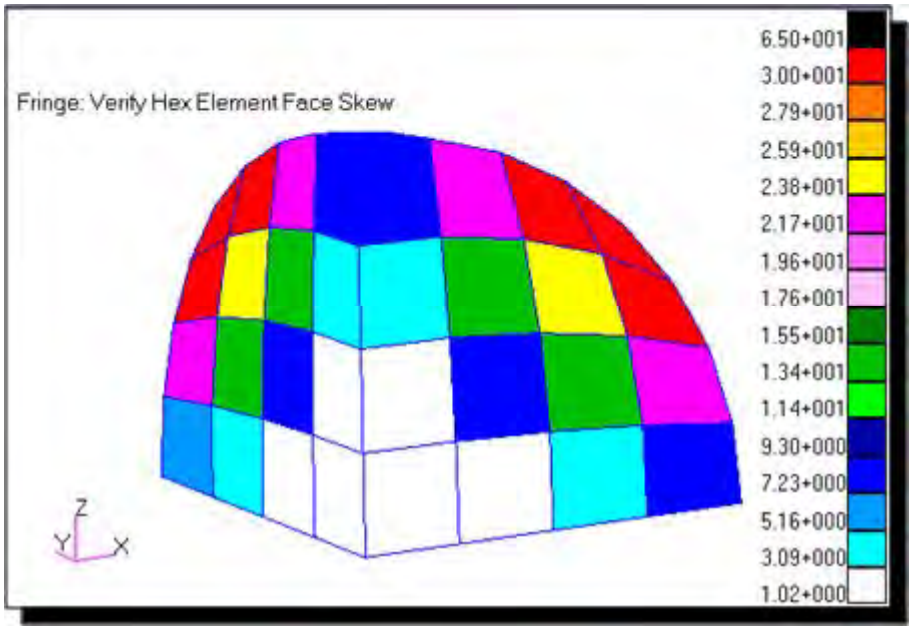
- **The largest face skew is 18.4 degrees**

Copyright© 2012 MSC.Software Corporation

3D BEGINNING TO END

- **Look at the face skew for the previous Hex mesh for the 1/8 sphere**





- **The largest value for face skew is 65.0 degrees**

Copyright© 2012 MSC.Software Corporation

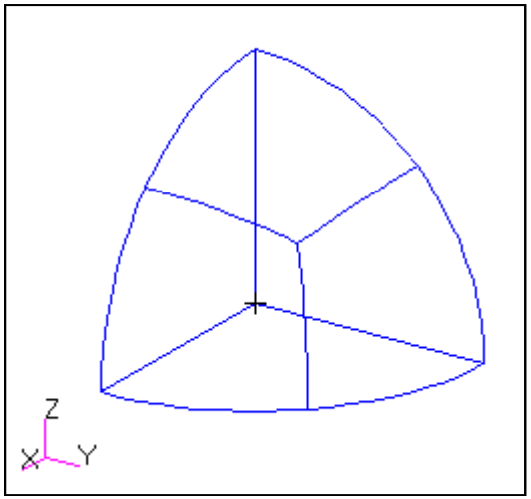
3D ISOMESH & SOLID TOPOLOGY

- **The topology condition may be acceptable for IsoMesh to create Hex elements.**
- **However, the shape of the geometry may be poor for creating Hex elements.**
- **It is important to create well shaped simple solids for Hex meshing.**
- **Look at different ways of representing the 1/8 sphere geometry, using four congruent parametric solids.**

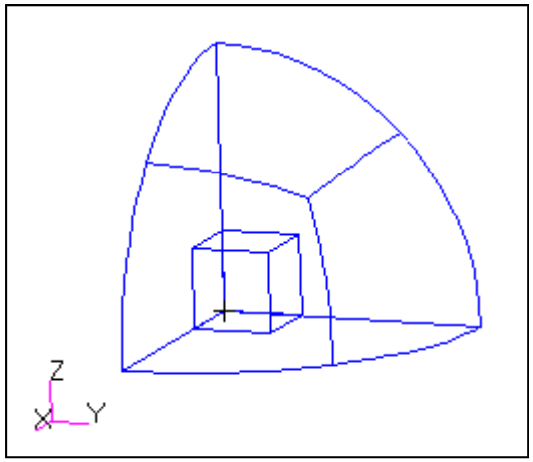
Copyright© 2012 MSC.Software Corporation

3D ISOMESH & SOLID TOPOLOGY

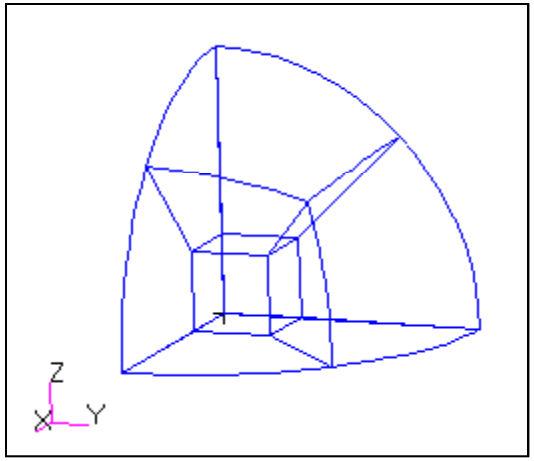
- **Begin with the last solid created; it has six non-degenerate faces:**



Start from Last Solid



Create a Cube Solid at Origin

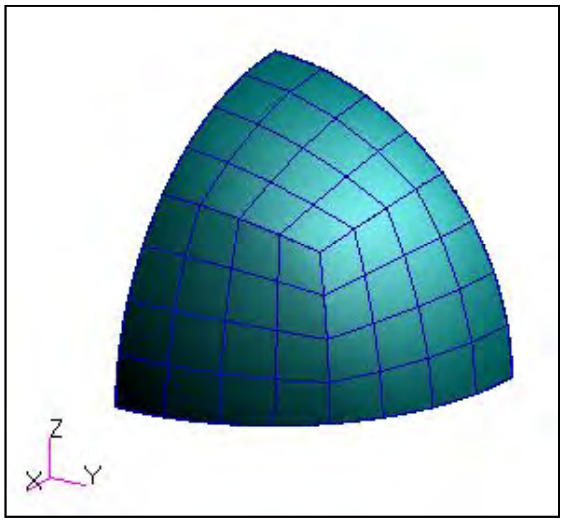


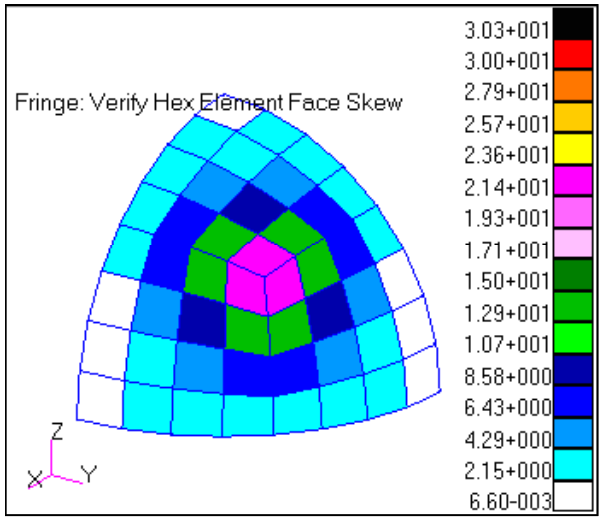
Create Three Solids from Solid Faces

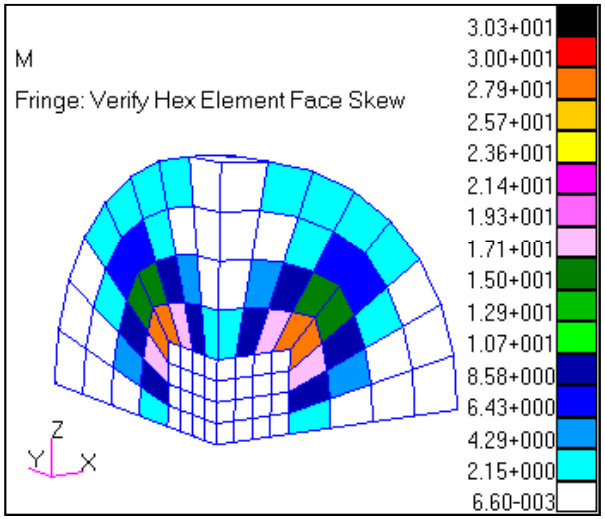
Copyright© 2012 MSC.Software Corporation

3D ISOMESH & SOLID TOPOLOGY

- **Hex elements from IsoMesh, and verify of Hex elements for face skew**







- **The largest value for face skew is 30.3 degrees**

Copyright© 2012 MSC.Software Corporation

CREATE HEXAHEDRAL ELEMENT MESH FOR COMPLEX GEOMETRY

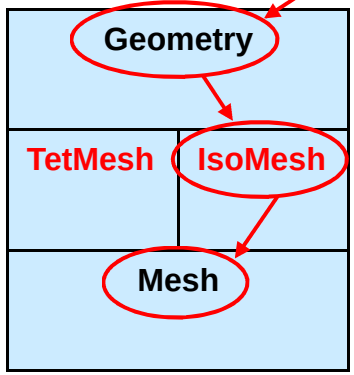
- **If there is need to create a hexahedral element mesh for a complex model, several approaches can be used**
  - Represent the geometry with a set of parametric solids.
    - These can be IsoMeshed or HexMeshed to create hex elements.
    - Use mesh seeds, and hard geometry on geometric edges.
    - Faces of parametric solids can be IsoMeshed.
  - Sweep the 2D element mesh normal to the mesh in order to create 3D elements.
    - Approaches are Extrude, Glide, Vector Field, etc.

Copyright© 2012 MSC.Software Corporation

CREATE HEXAHEDRAL ELEMENT MESH FOR COMPLEX GEOMETRY

- **Follow path through Patran that starts with 3D geometry (e.g. parasolid solid), disassemble it into surfaces, edit the surfaces, and create a set of congruent parametric solids.**

Start Point 3D	Nothing		Mesh	Geometry		
Method	Extrude, Glide, etc. Methods	B-Rep or Primitive Construction	Mesh Editing	TetMesh	IsoMesh	Geometry breakdown and rebuild methods
End Point 3D	Simple Geometry	Complex Geometry	Mesh	Mesh		Geometry



Copyright© 2012 MSC.Software Corporation

CREATE HEXAHEDRAL ELEMENT MESH FOR COMPLEX GEOMETRY

- **This is for starting with 3D geometry (e.g. parasolid solid), disassemble it into surfaces, edit the surfaces, 2D mesh the surfaces, and sweep the 2D mesh.**

Start Point 3D	Nothing		Mesh		Geometry		
Method	N/A	N/A	N/A	N/A	Mid Plane Mesher	Mid Surface Extraction	Geometry breakdown and rebuild methods
End Point 2D	Mesh	Geometry	Mesh	Geometry	Mesh	Geometry	

Start Point 2D	Mesh
Method	Sweep
End Point 3D	Mesh

Start Point 2D	Geometry
Method	IsoMesh or Paver
End Point 2D	Mesh

Copyright© 2012 MSC.Software Corporation

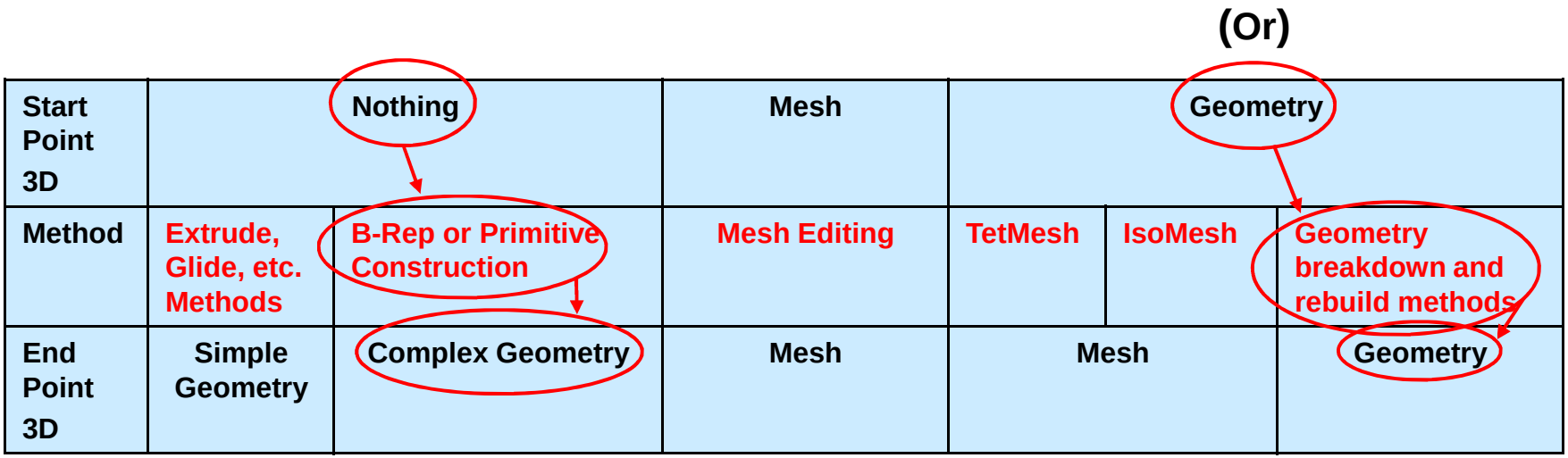
EXERCISES

- Perform Workshop 10 “Hex & Tet Solid Element Mesh” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

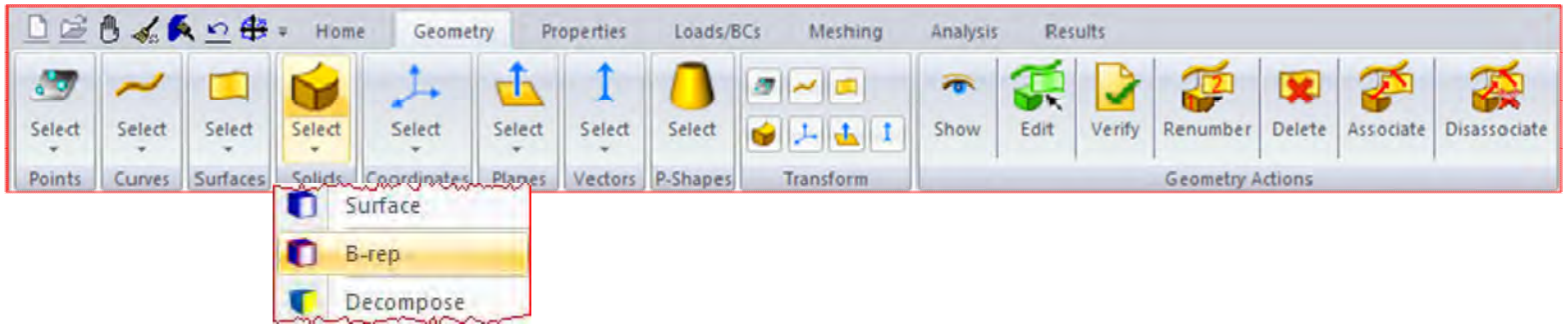
BUILDING B-REP SOLIDS

- **There are two rules that must be followed in creating a B-rep Solid**
  - The group of surfaces must completely enclose the volume.
  - The surfaces must be topologically congruent. That is, adjacent surfaces must have coincident edges and vertices.

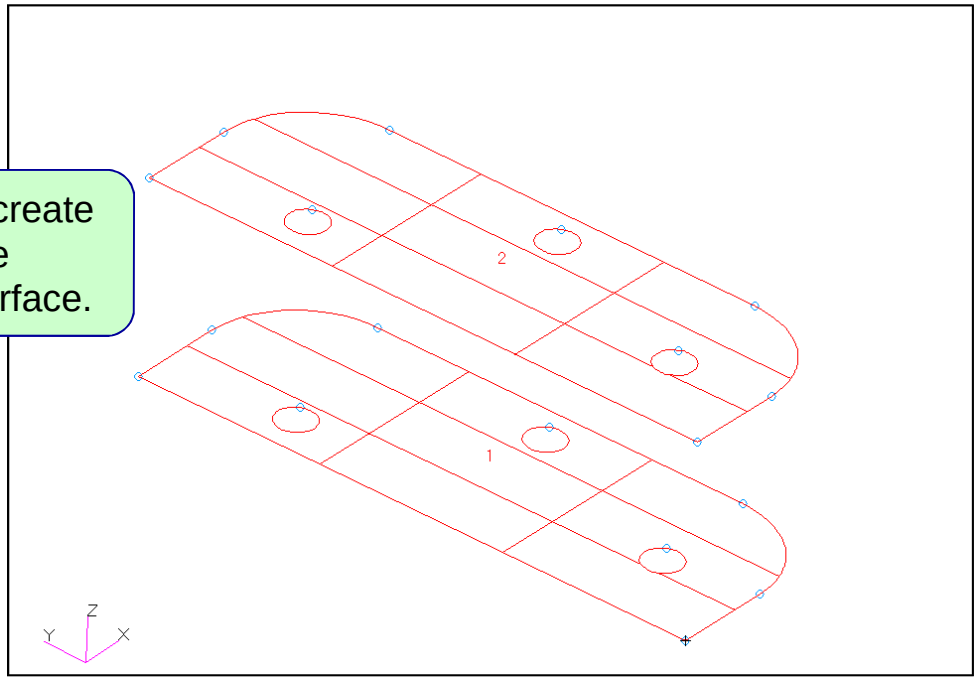


Copyright© 2012 MSC.Software Corporation

EXAMPLE: BUILDING B-REP SOLID

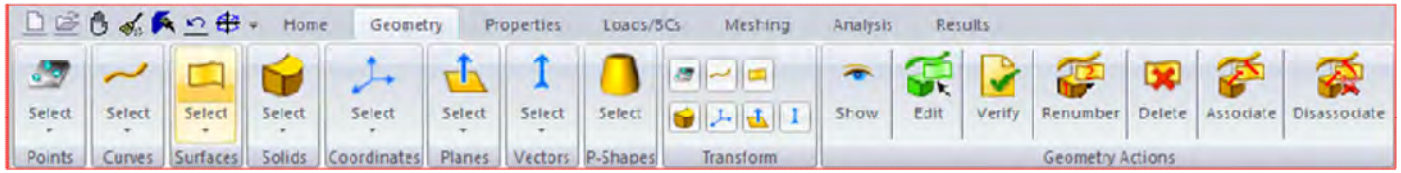


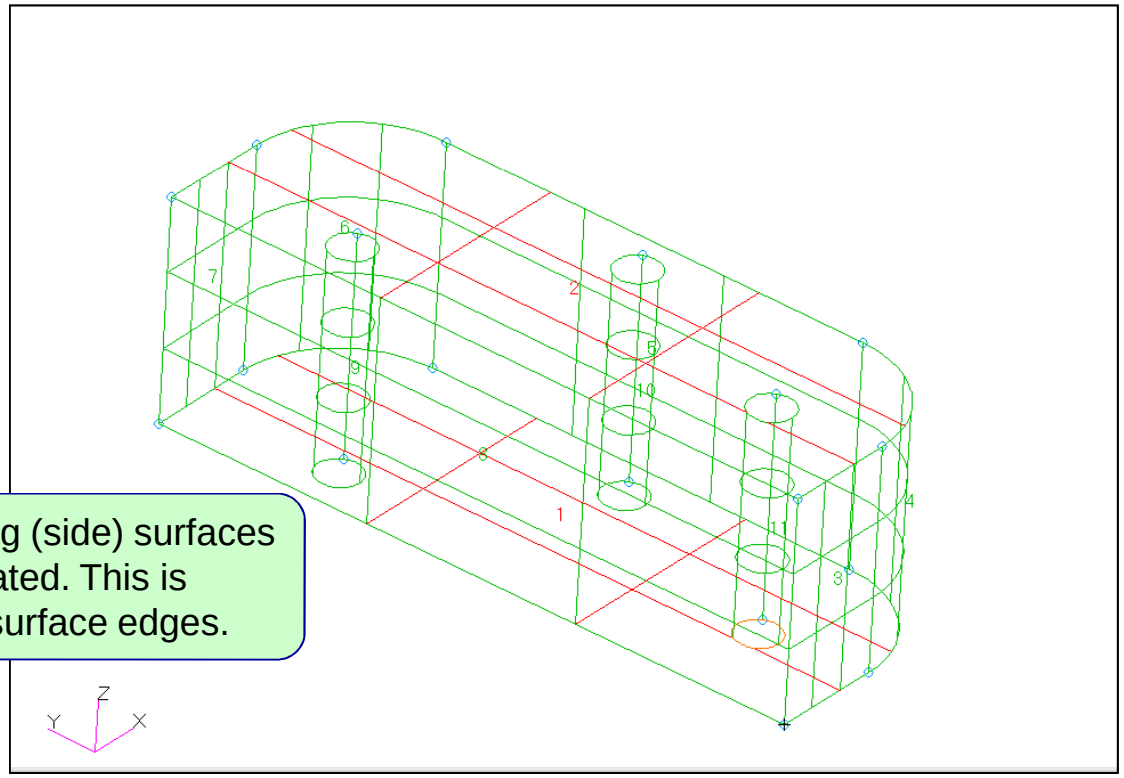
The requirement is to create a B-Rep solid from one surface to the other surface.



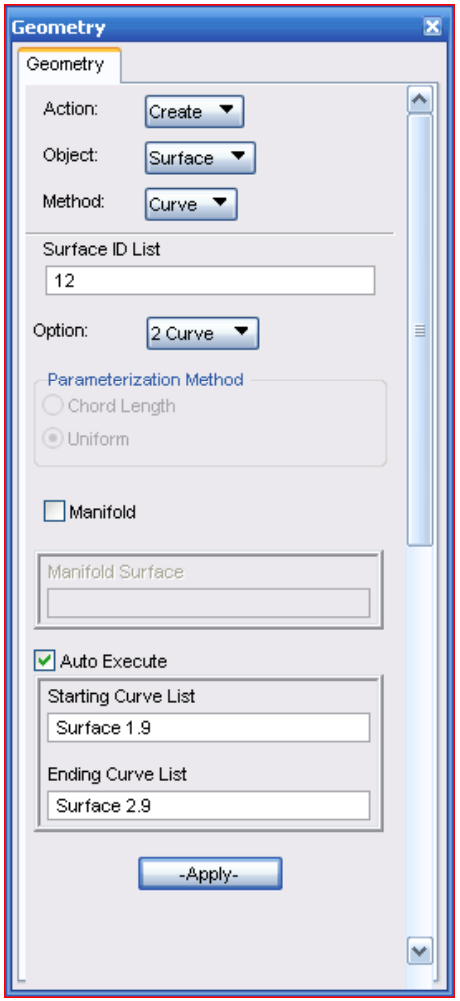
Copyright© 2012 MSC.Software Corporation

EXAMPLE: BUILDING B-REP SOLID



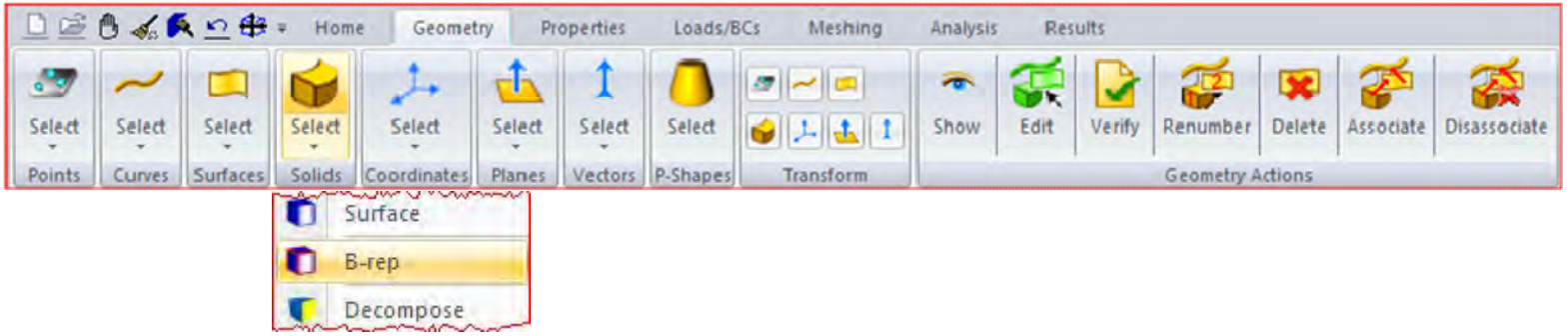


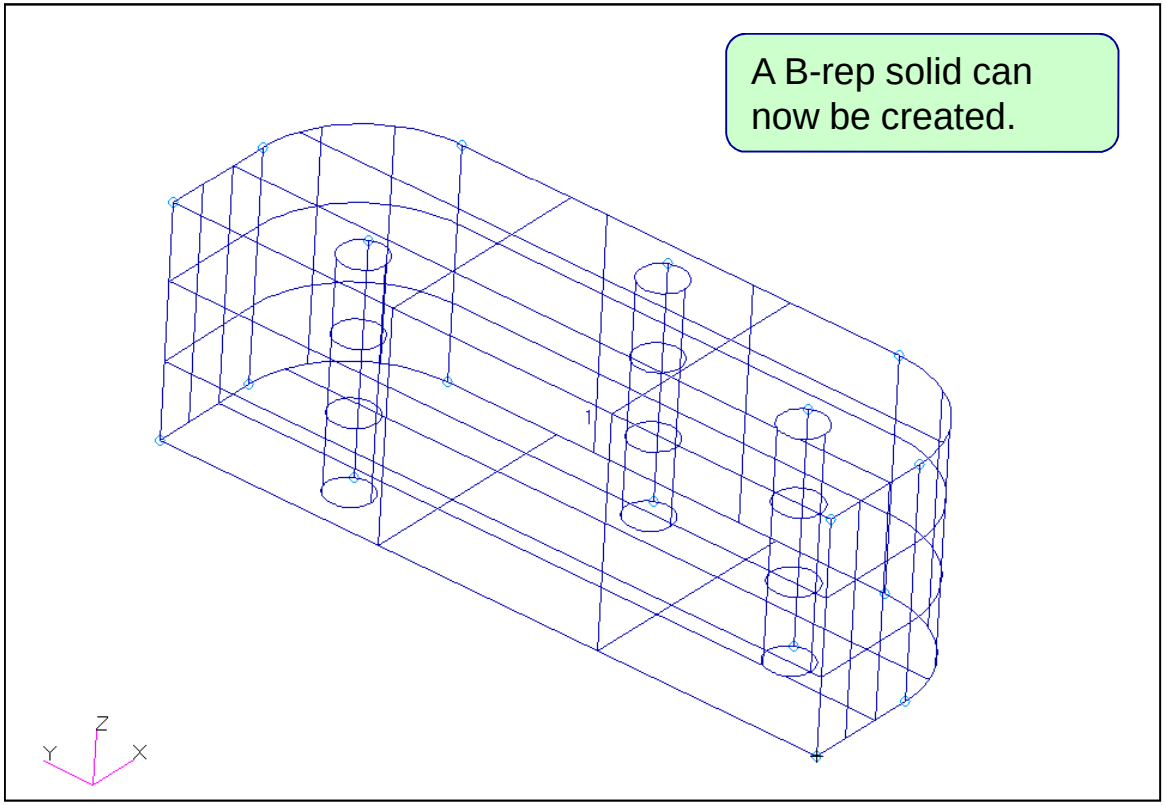
The bounding (side) surfaces must be created. This is done using surface edges.

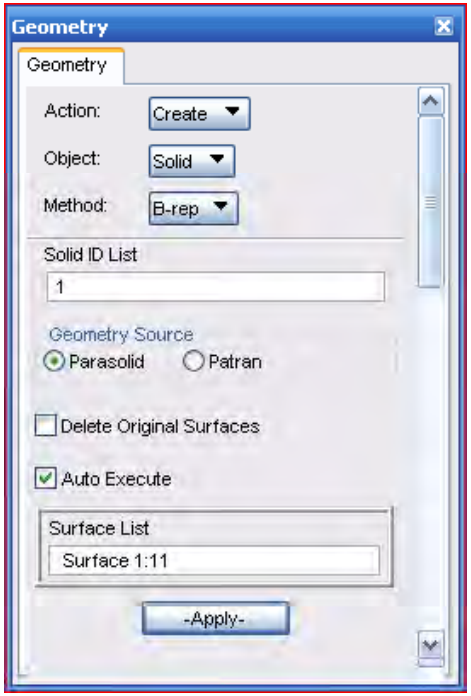


Copyright© 2012 MSC.Software Corporation

EXAMPLE: BUILDING B-REP SOLID







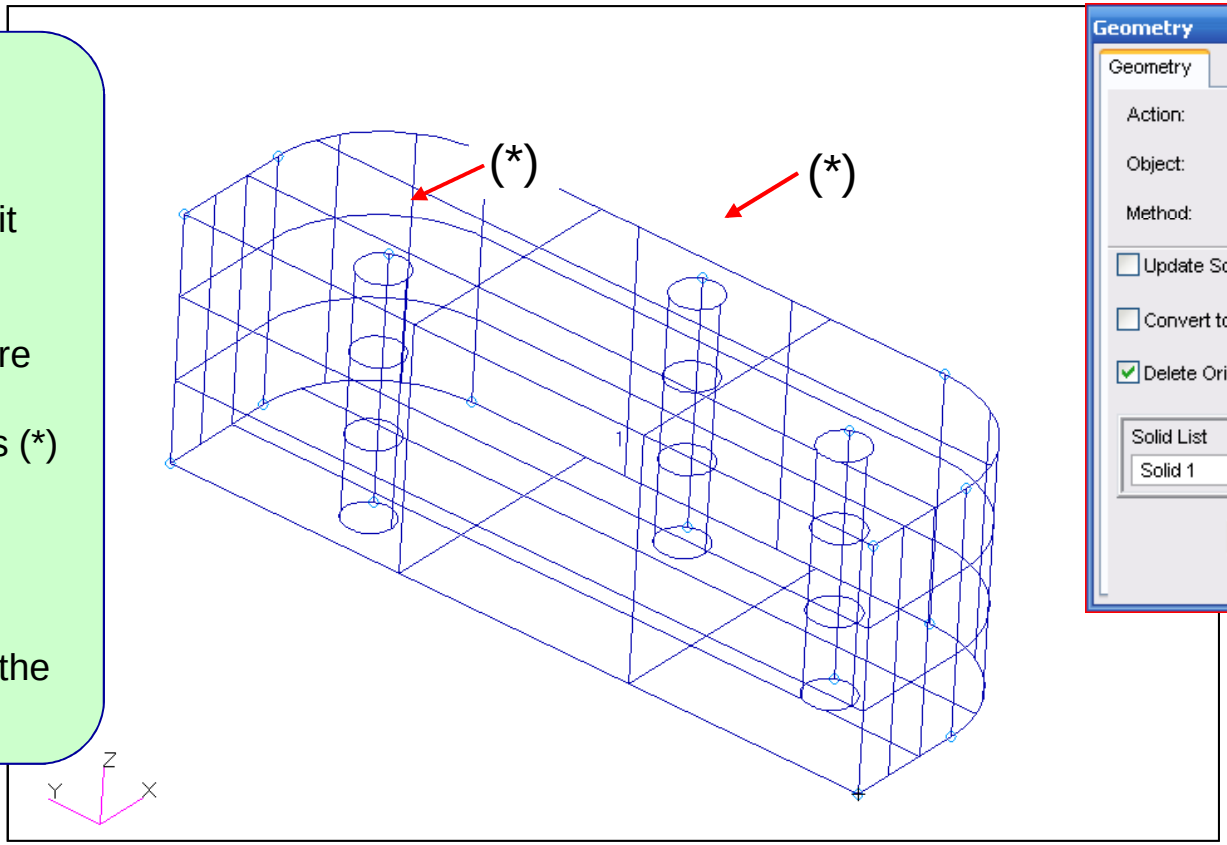
Copyright© 2012 MSC.Software Corporation

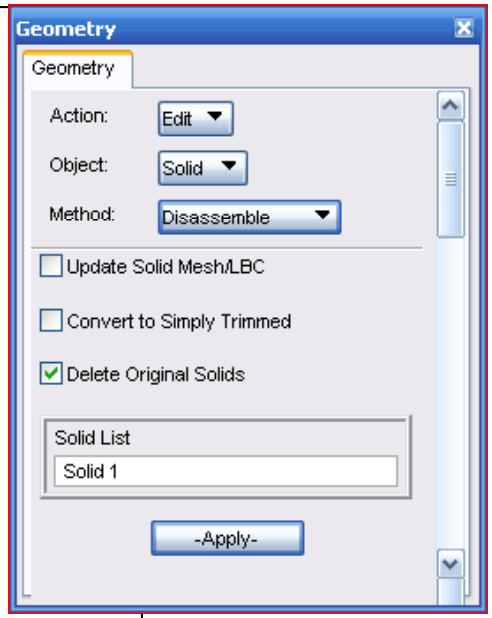
EXAMPLE: BUILDING B-REP SOLID

De-featuring a B-rep solid involves disassembling it into surfaces.

The surfaces are edited; in this case, two holes (*) are to be eliminated.

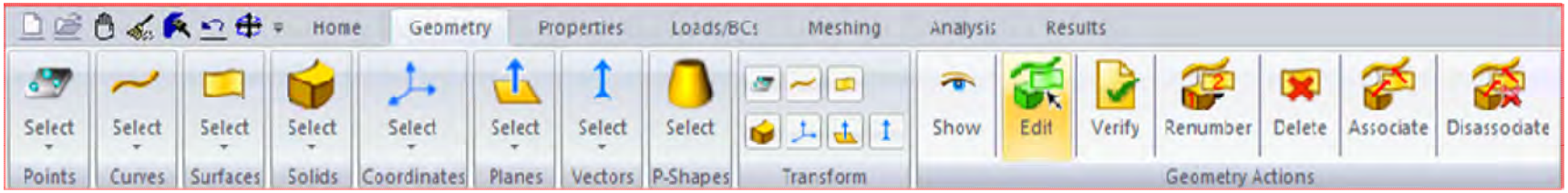
Start by disassembling the solid.



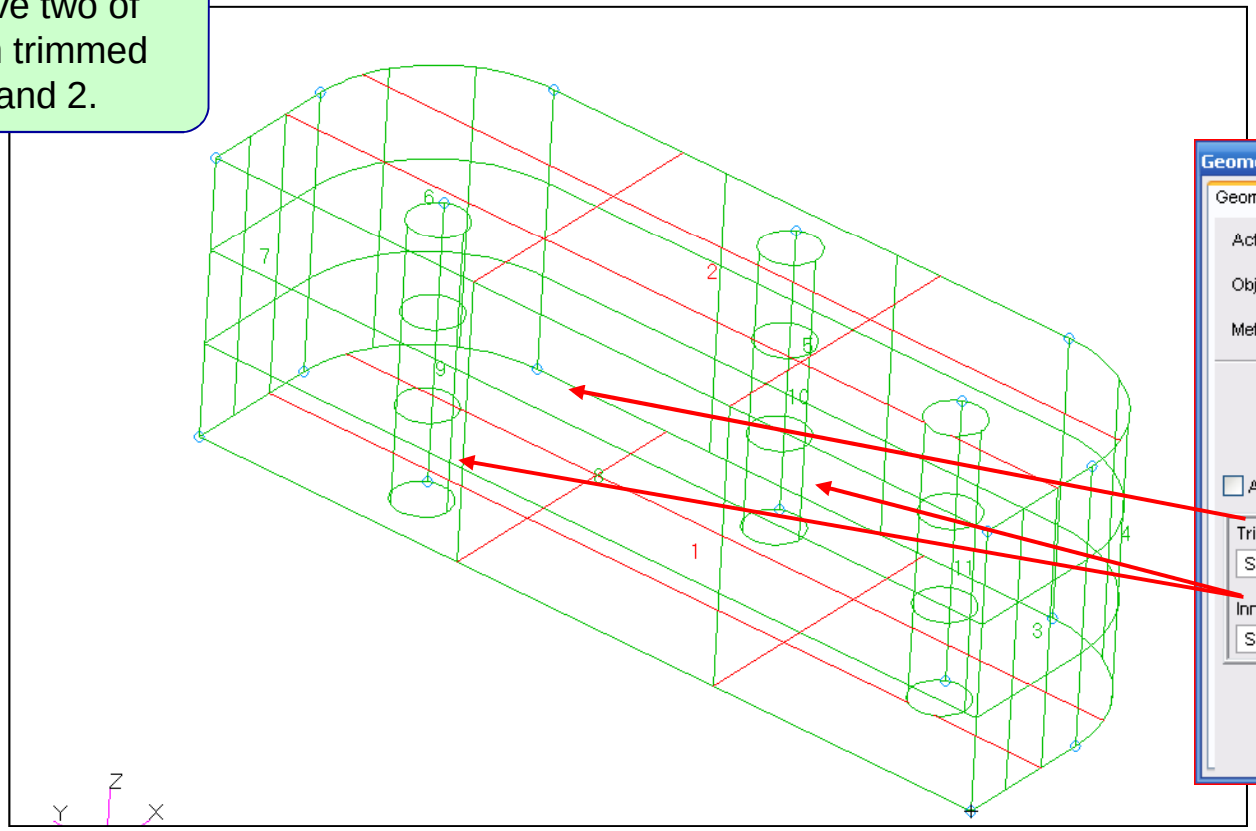


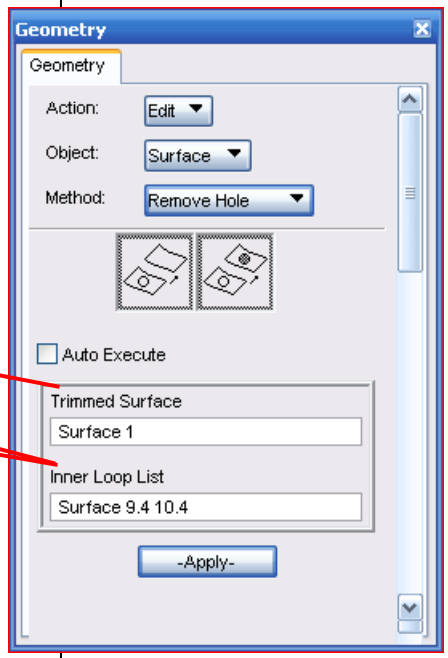
Copyright© 2012 MSC.Software Corporation

EXAMPLE: BUILDING B-REP SOLID



Now, remove two of the holes in trimmed surfaces 1 and 2.

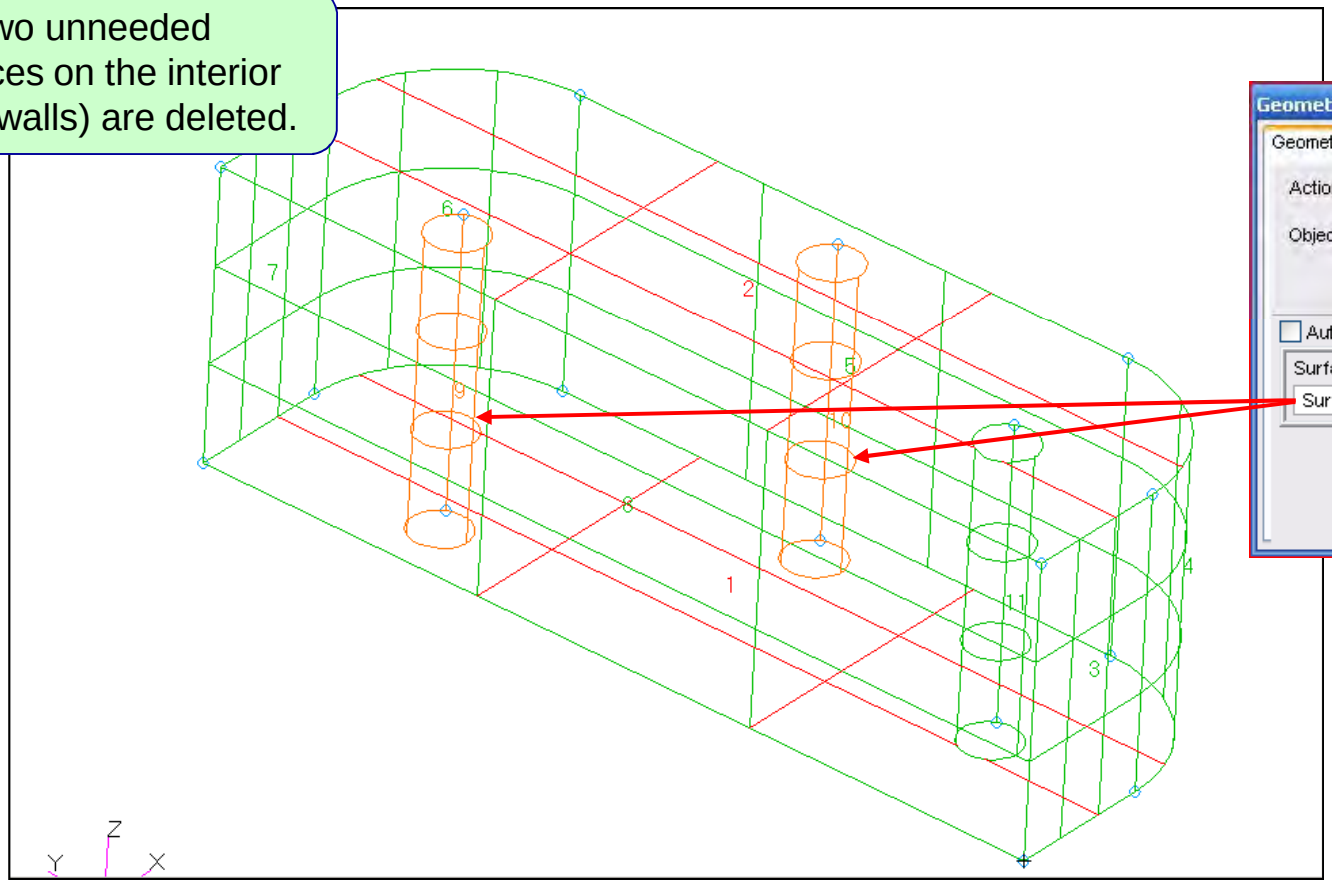


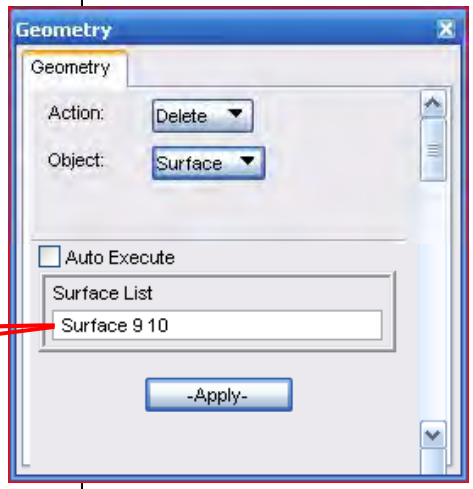


Copyright© 2012 MSC.Software Corporation

EXAMPLE: BUILDING B-REP SOLID

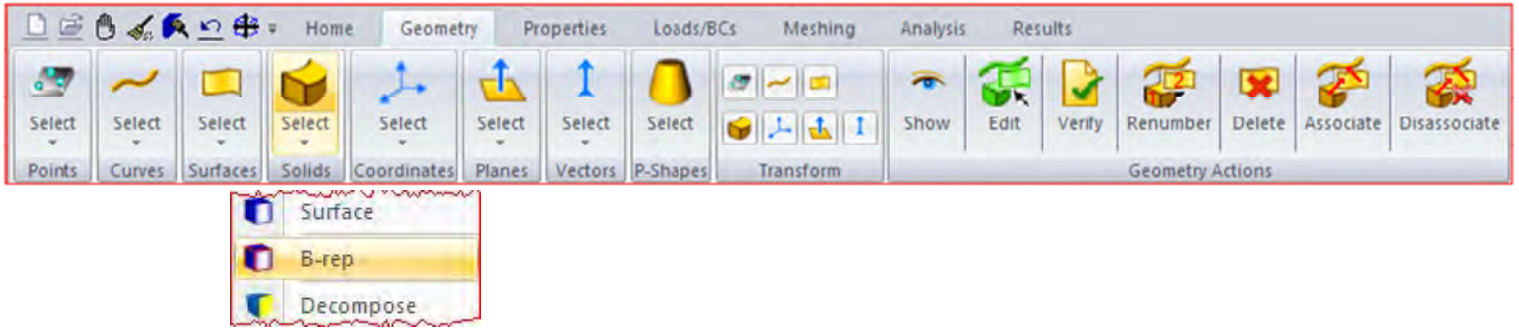
The two unneeded surfaces on the interior (hole walls) are deleted.

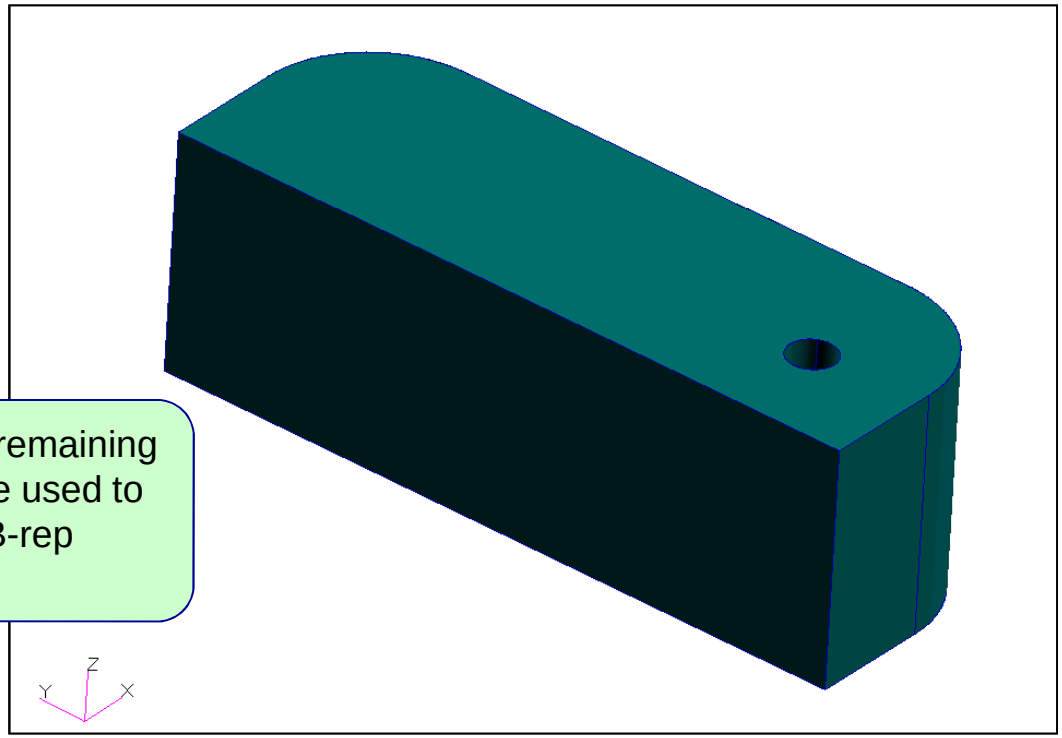


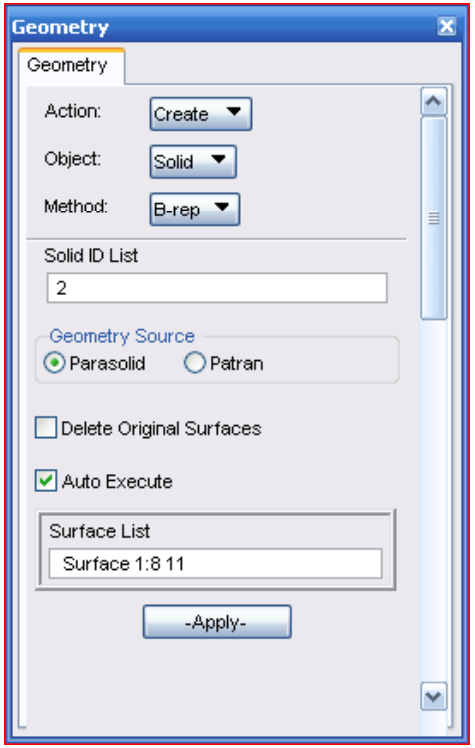


Copyright© 2012 MSC.Software Corporation

EXAMPLE: BUILDING B-REP SOLID







Finally, the remaining surfaces are used to create the B-rep solid.

Copyright© 2012 MSC.Software Corporation

EXERCISES

- Perform Workshop 11 “Edit Solid Blade” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

PRIMITIVE SOLID MODELING TOOLS

- **Primitive solid creation**
  - Rectangular box, cylinder, cone, sphere, torus
  - Optional on-the-fly (automatic) Boolean operation
- **Solid creation operations**
  - Extrude
  - Revolve
- **Solid editing operations**
  - Boolean operations: add, subtract, intersect
  - Edge blend: constant radius, chamfer
  - Shell: create thin walled solids
  - Imprint: solid on solid
  - Refit to Parasolid
  - Auto update of CAE data after a solid editing operation

Copyright© 2012 MSC.Software Corporation

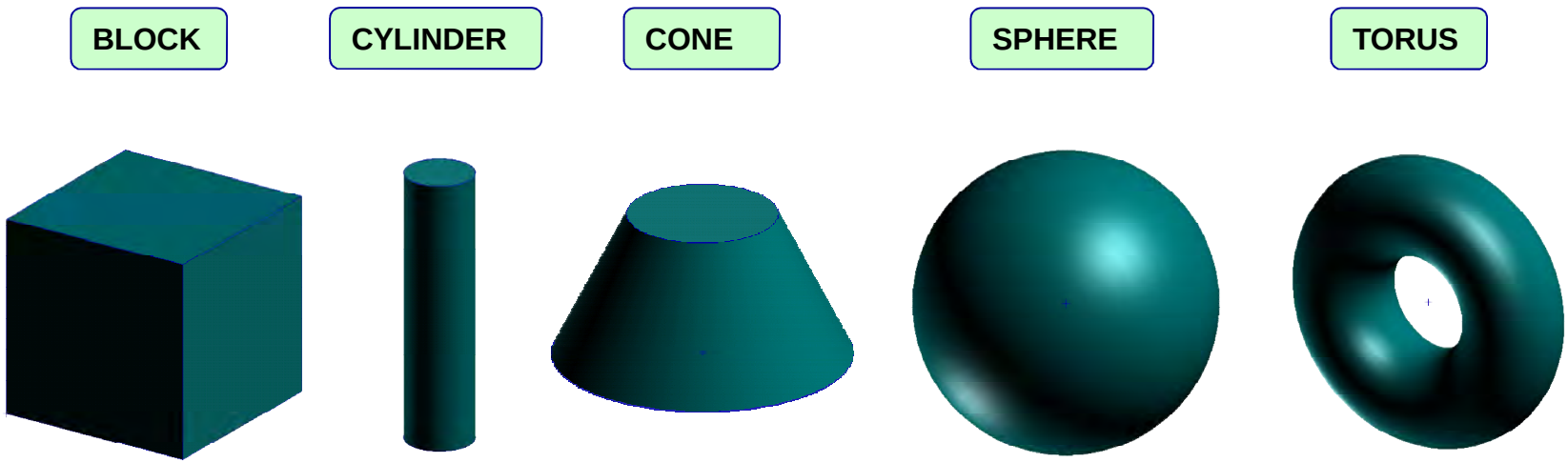
PRIMITIVE SOLID MODELING TOOLS

- **The parasolid solid modeling tools described in this section use the parasolid kernel which requires the parasolid modeling license.**

Copyright© 2012 MSC.Software Corporation

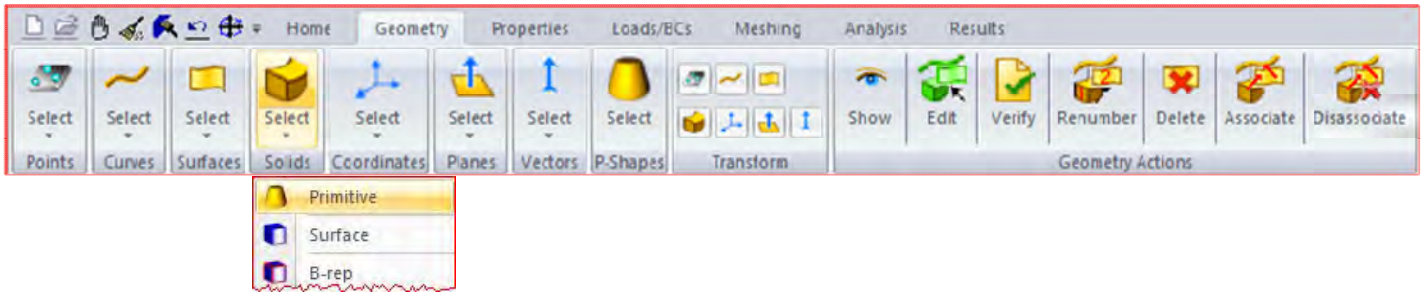
PRIMITIVE (SOLID) CREATION

- **A primitive is a solid that can be defined with only a few parameters.**
- **Patran has five different types of primitives:**



Copyright© 2012 MSC.Software Corporation

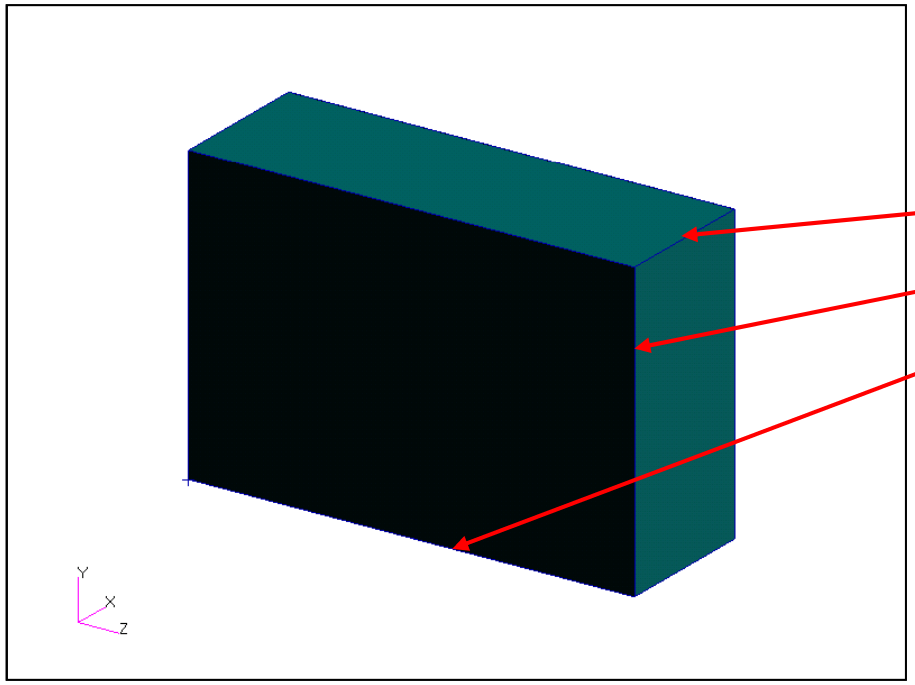
PRIMITIVE CREATION BLOCK

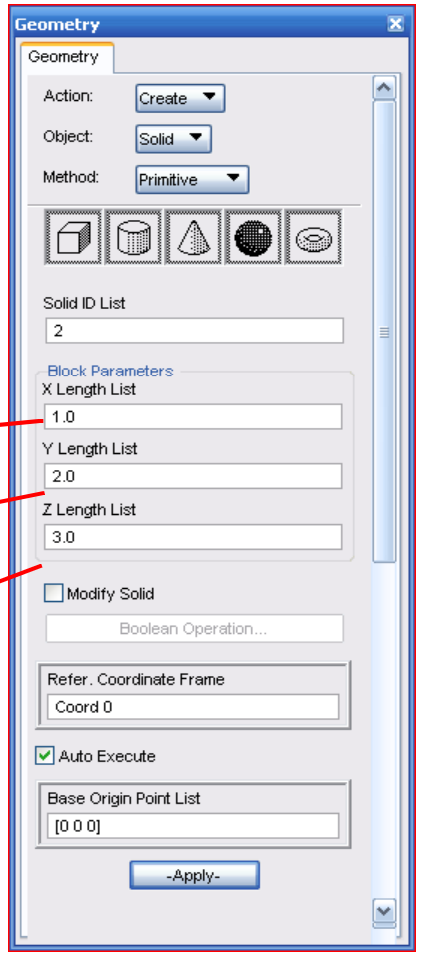


Create a rectangular solid by specifying side lengths.

The direction of the X, Y, and Z length are determined by the coordinate frame in Refer. Coordinate Frame. Users can specify a negative side length to create the block in the opposite direction.

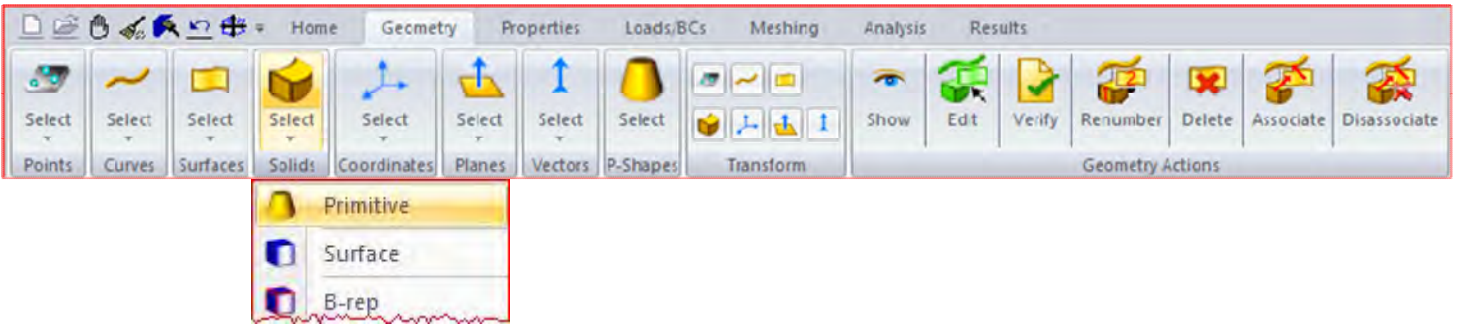
The Base Origin defines the corner of the block and also depends on the reference coordinate frame.





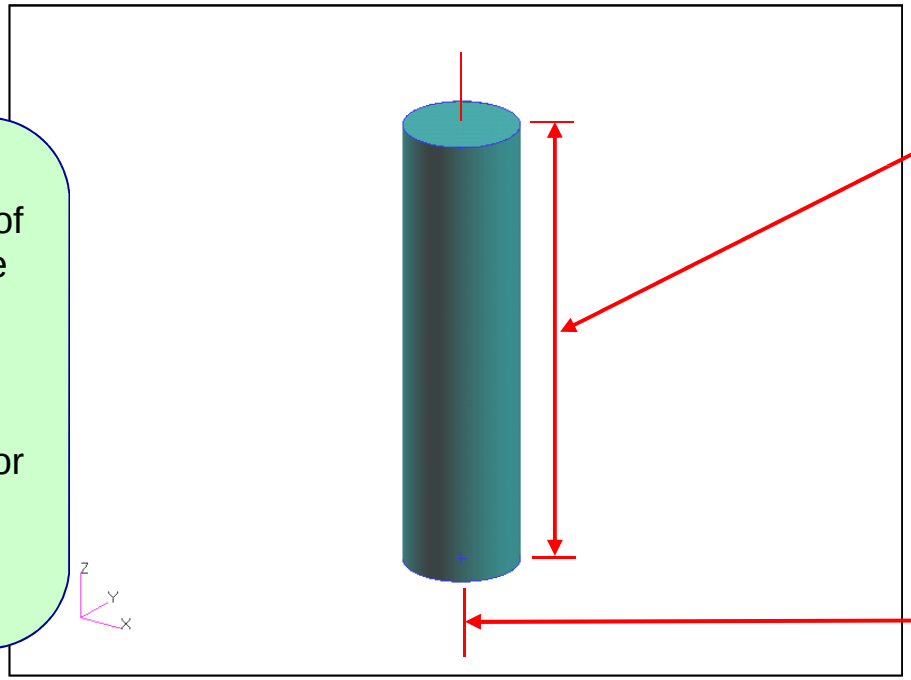
Copyright© 2012 MSC.Software Corporation

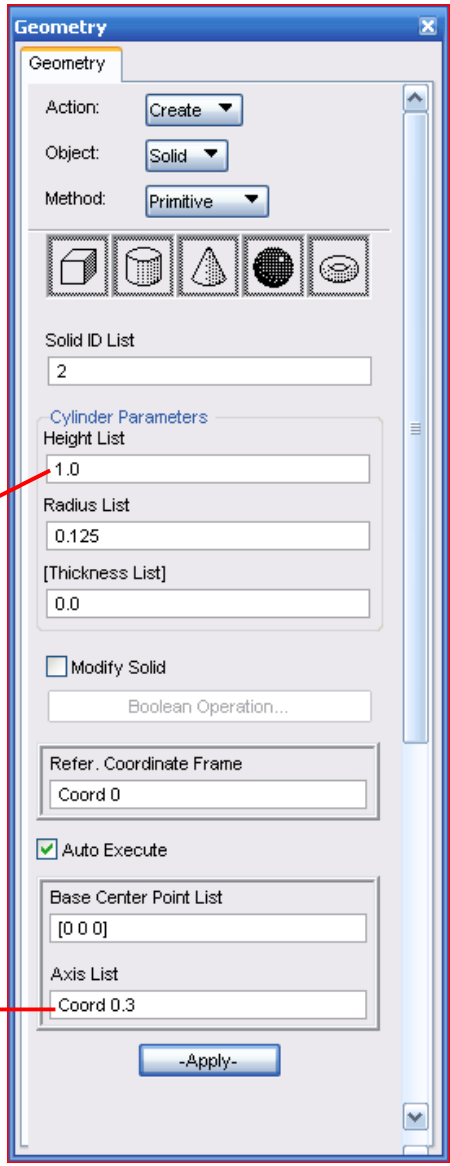
PRIMITIVE CREATION CYLINDER



Define a cylinder using height, radius, location of the base, and centerline orientation.

The height can be defined along the X, Y, or Z axis. This direction is defined by the Axis List entry.

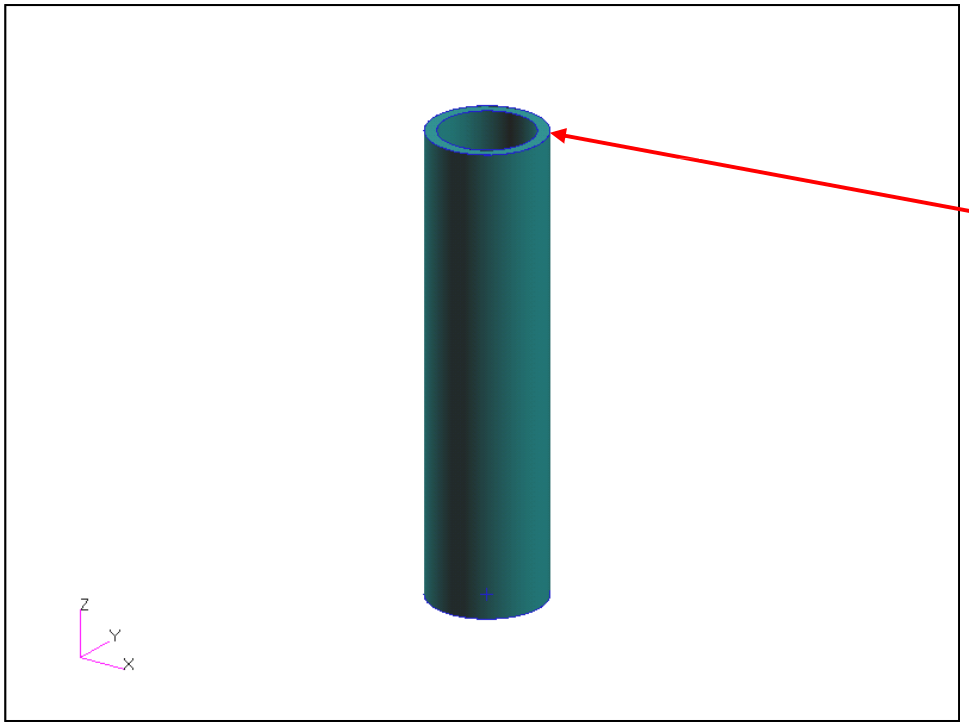


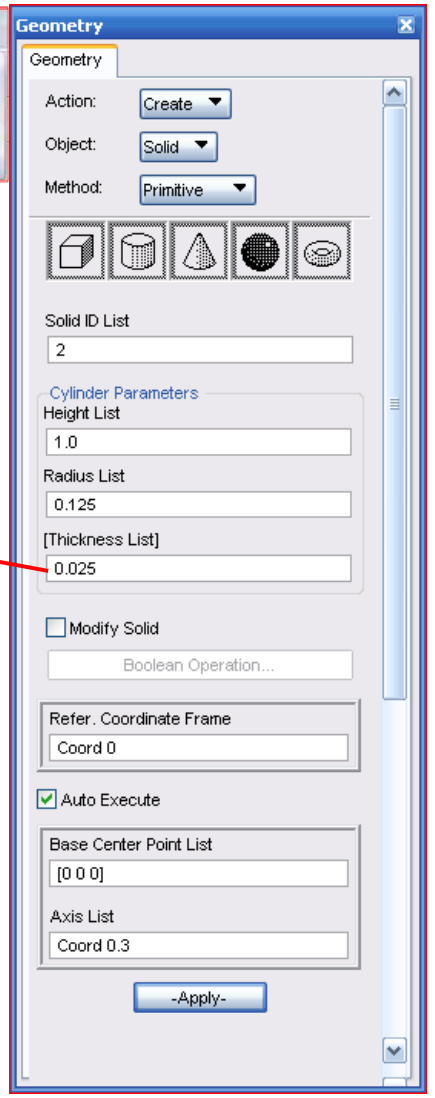


Copyright© 2012 MSC.Software Corporation

PRIMITIVE CREATION CYLINDER

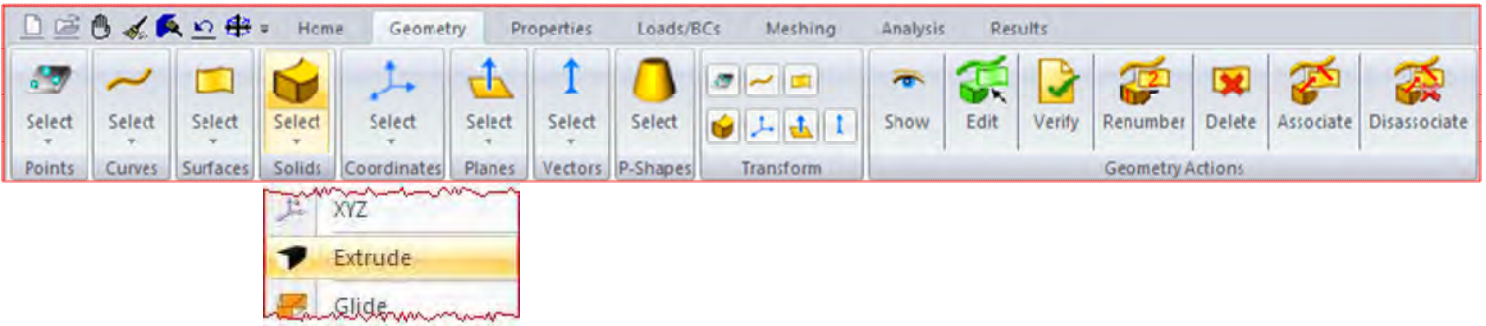
By entering a wall thickness, a tube can be created.





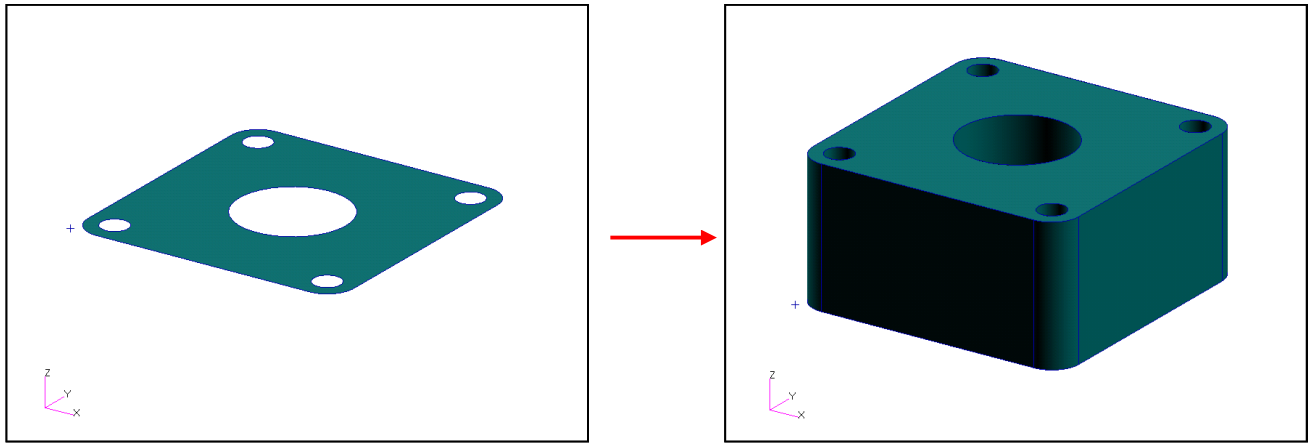
Copyright© 2012 MSC.Software Corporation

SOLID CREATION EXTRUDE

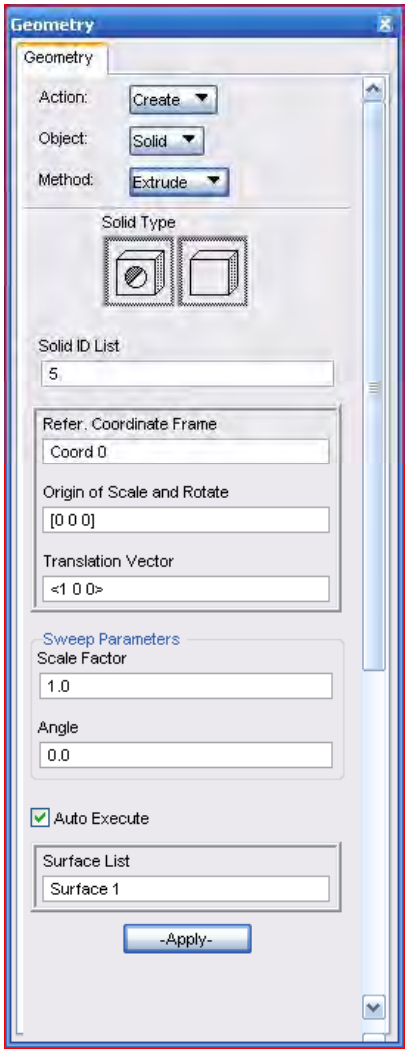


Extruding Surfaces:

Using this option, any type of surface can be extruded to create a complex B-rep. (white) solid.

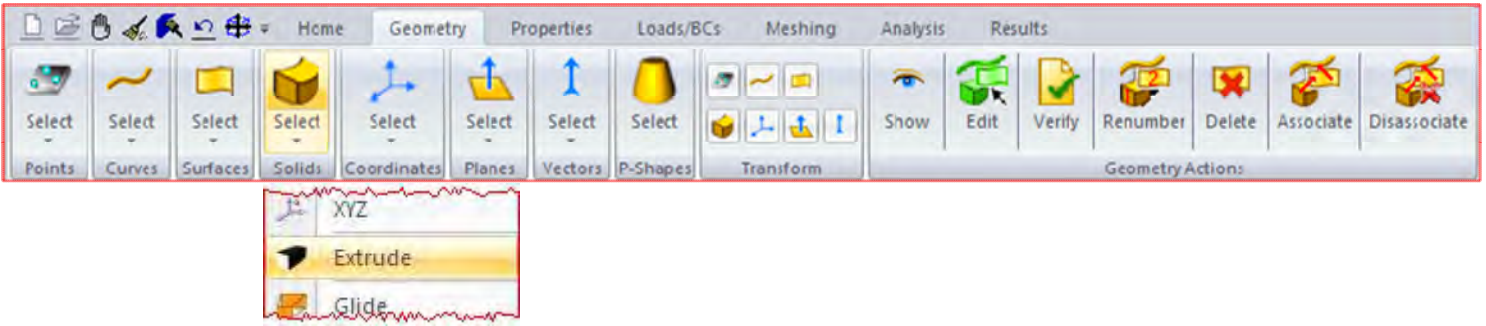


Surface 1



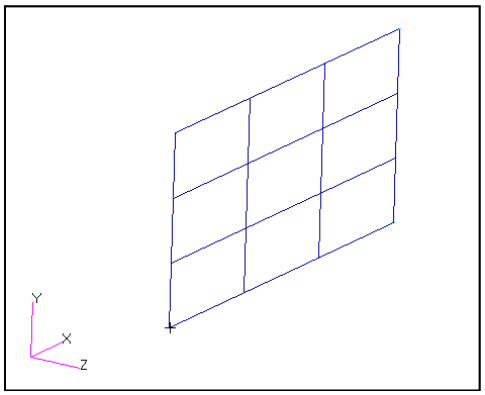
Copyright© 2012 MSC.Software Corporation

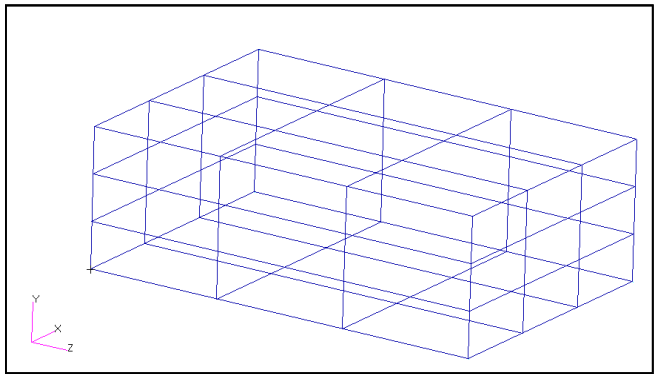
SOLID CREATION EXTRUDE



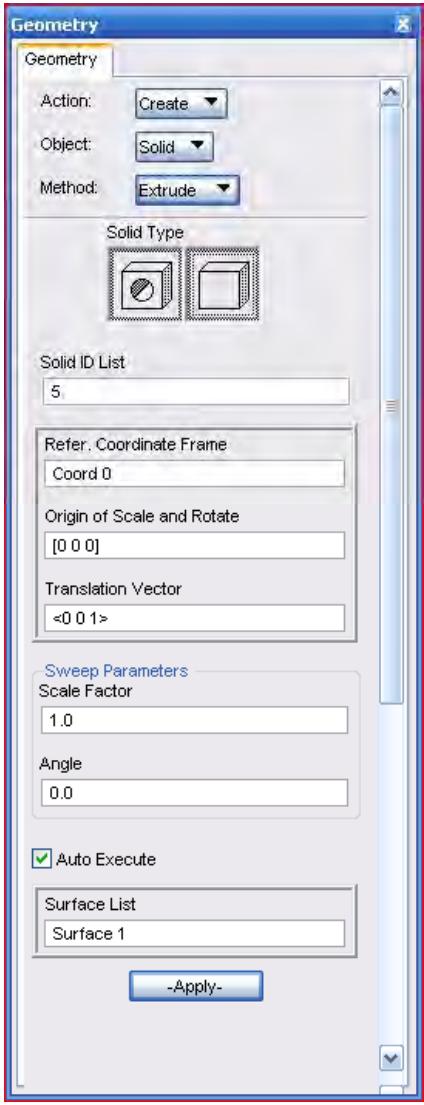
Extruding Simple Surfaces:

This option works only with simple (parametric) surfaces, which are extruded to create parametric (blue) solids.





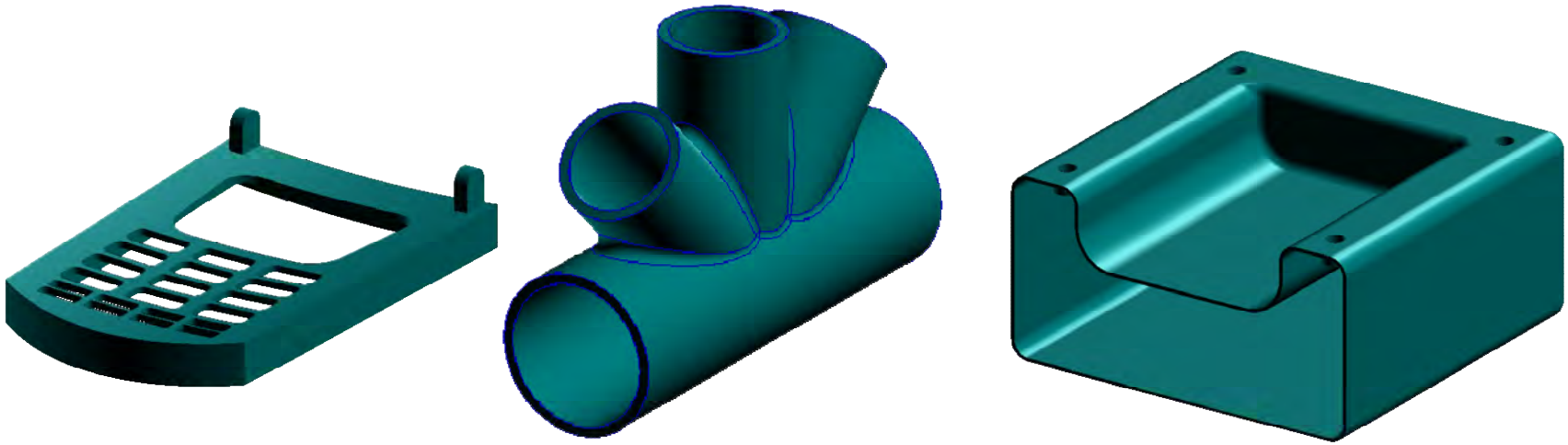
Surface 1



Copyright© 2012 MSC.Software Corporation

SOLID EDITING

- **The following solid editing tools are available in Patran:**
  - **Boolean operations**: add, subtract, intersect
  - **Edge blend**: fillet, chamfer
  - **Imprint**: solid on solid
  - **Shell**: create thin-walled solids
  - **Refit**: to parasolid solid, or parametric solid
  - **Break**: use plane and surface



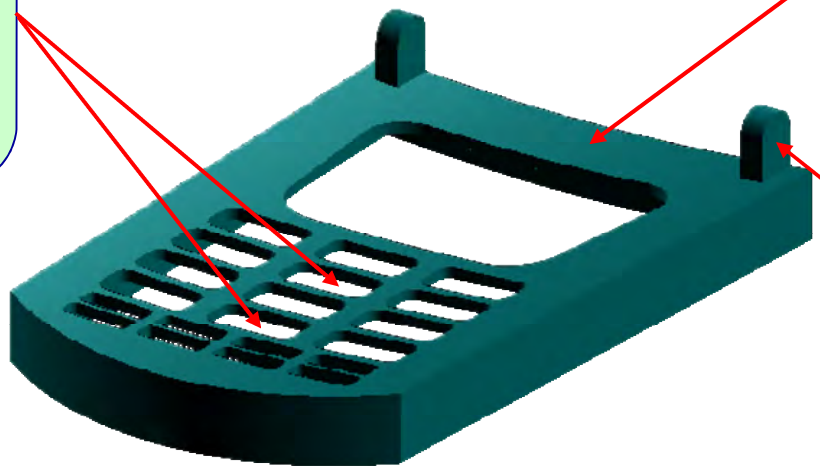
Copyright© 2012 MSC.Software Corporation

SOLID EDITING, BOOLEAN OPERATIONS OVERVIEW

- **This solid model represents the top face of a cell phone. It was created using three types of Boolean operations, add, subtract, and intersect**

The holes for the keyboard were made using multiple solids for subtraction.

This is much faster than subtracting the solids one at a time.

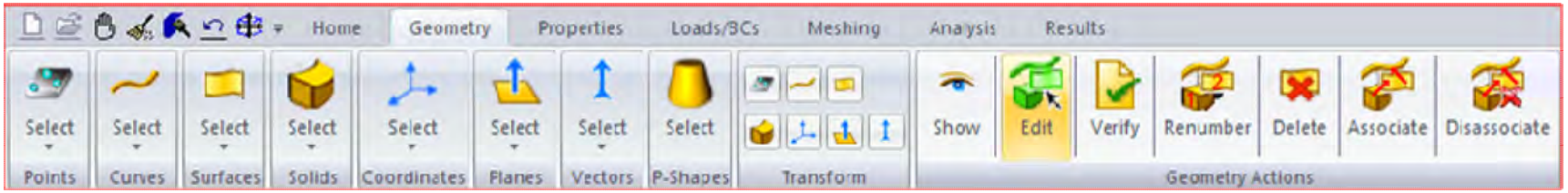


The main body of this solid was made by intersecting a thin walled cylinder with a simple block.

The tabs on the top of this part were included using Boolean add.

Copyright© 2012 MSC.Software Corporation

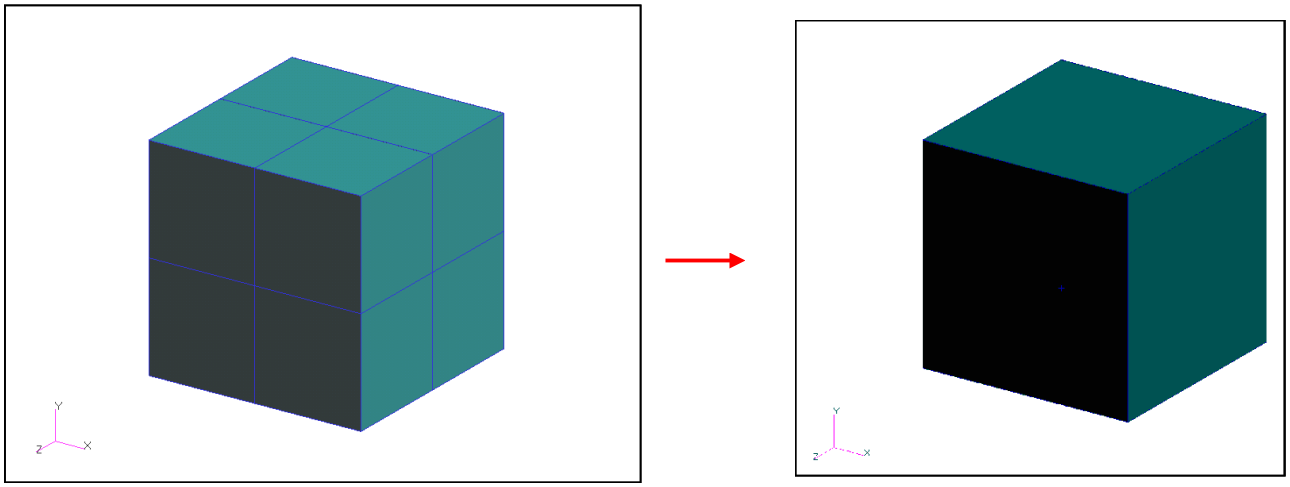
SOLID EDITING, BOOLEAN OPERATIONS

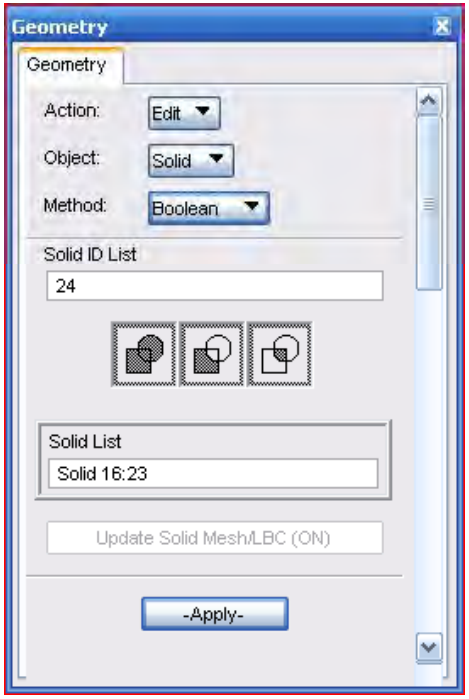


Boolean Add option:

Patran allows users to perform this operation on more than two solids at once.

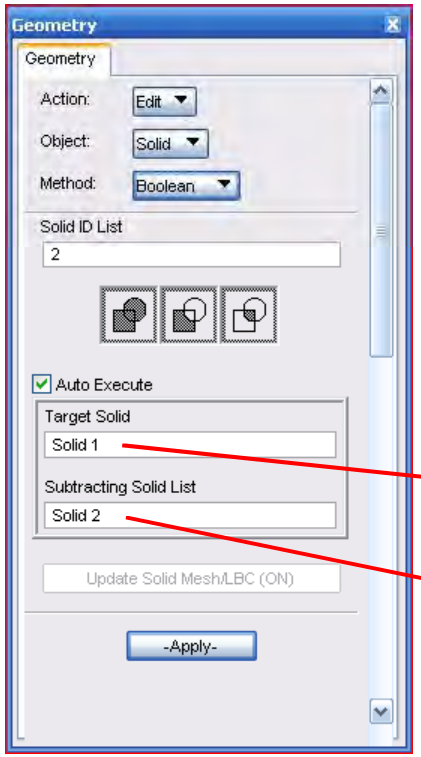
This picture shows individual solids before an Add operation is performed.





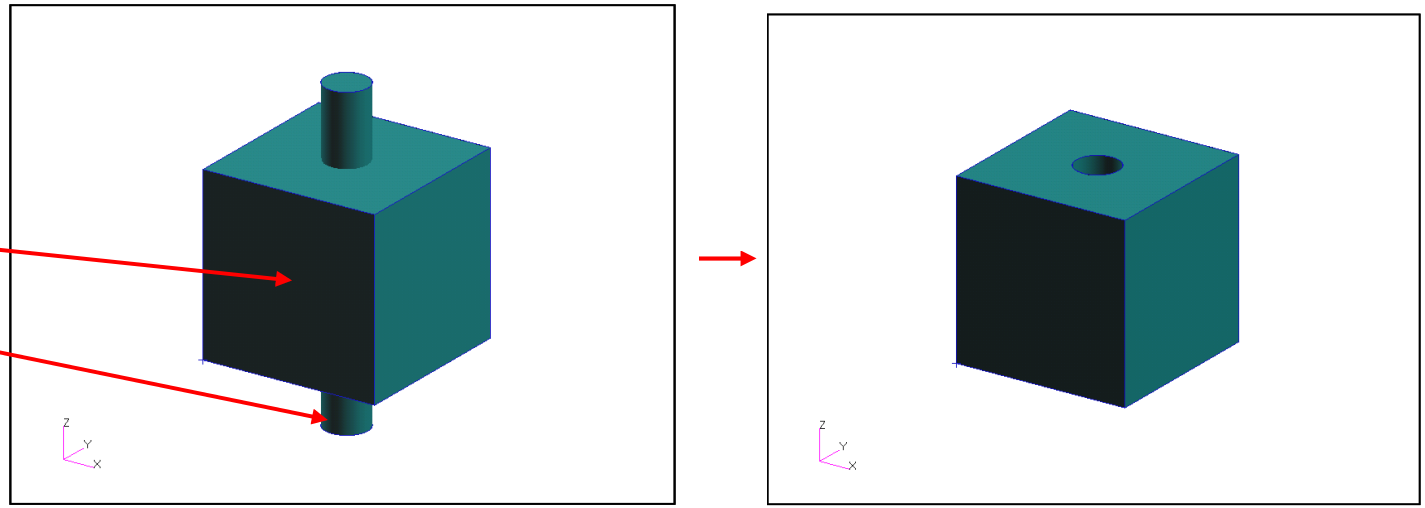
Copyright© 2012 MSC.Software Corporation

SOLID EDITING, BOOLEAN OPERATIONS



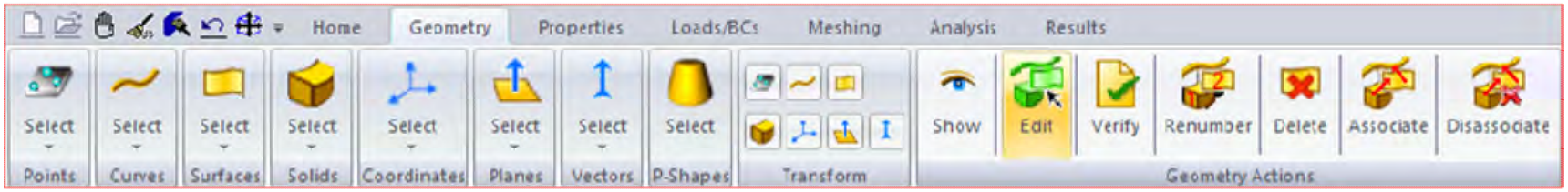
Boolean Subtract option:

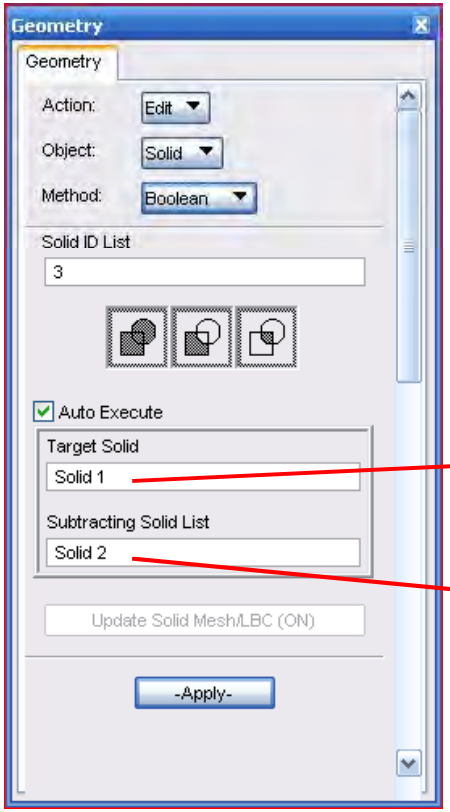
In this example the cylinder is subtracted from the block, which is the target solid.



Copyright© 2012 MSC.Software Corporation

SOLID EDITING, BOOLEAN OPERATIONS

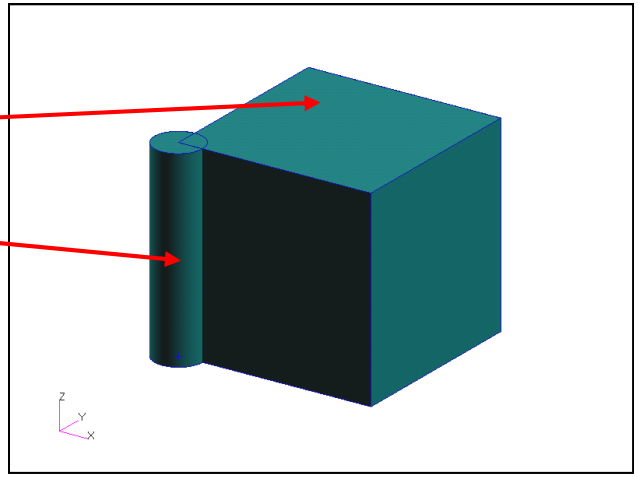


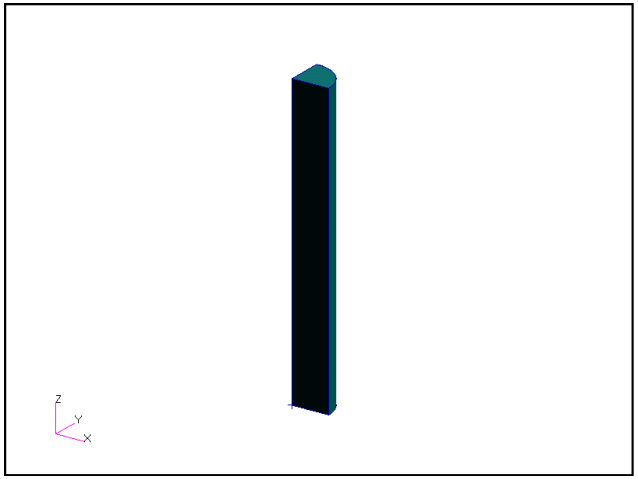


Boolean Intersect option:

Creates a new solid from what was common between the Target Solid and the Intersecting Solids.

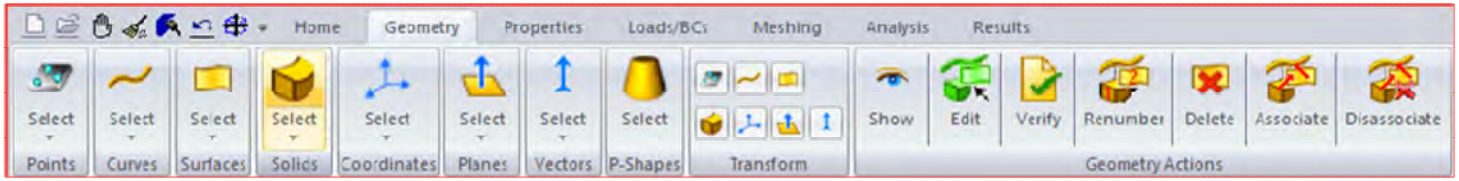
Patran automatically deletes the solids used for this operation. If the user still wants to use these solids, it will be necessary to copy them BEFORE the operation.

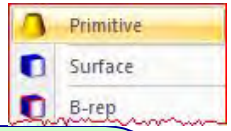




Copyright© 2012 MSC.Software Corporation

PRIMITIVE CREATION, AUTOMATIC BOOLEAN OPERATION





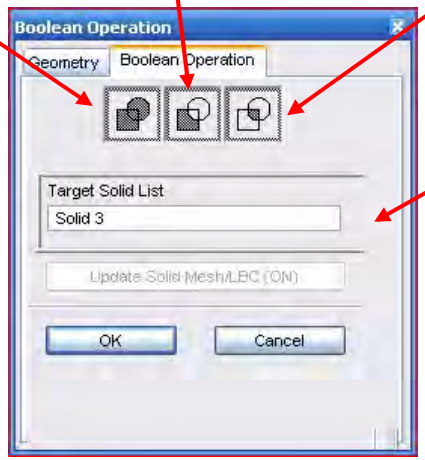
The Boolean operation menu controls what kind of Boolean to perform using the created solid and target solid.

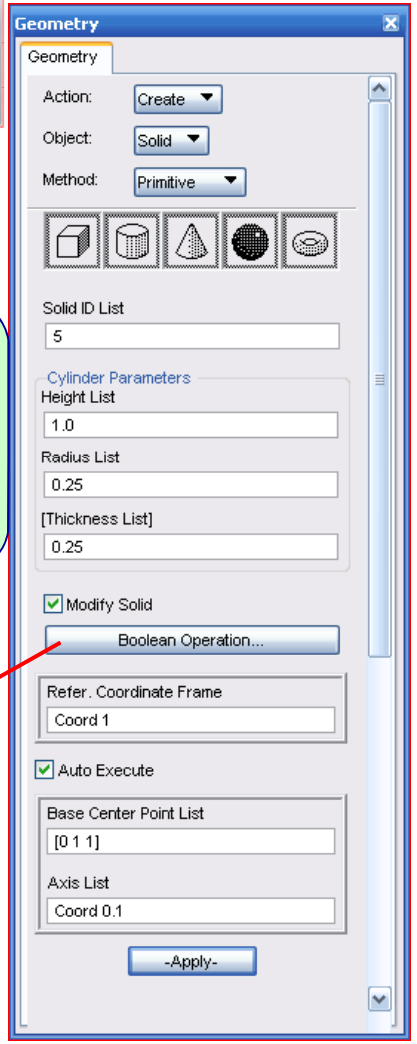
This menu appears when the user selects the Modify Solid option toggle in any of the primitive solid menus.

The created solid is *added* to the target solid to give the final solid.

The created solid is *subtracted* from the target solid to give the final solid.

The final solid is the *intersection* of the created solid and the target solid.





Copyright© 2012 MSC.Software Corporation

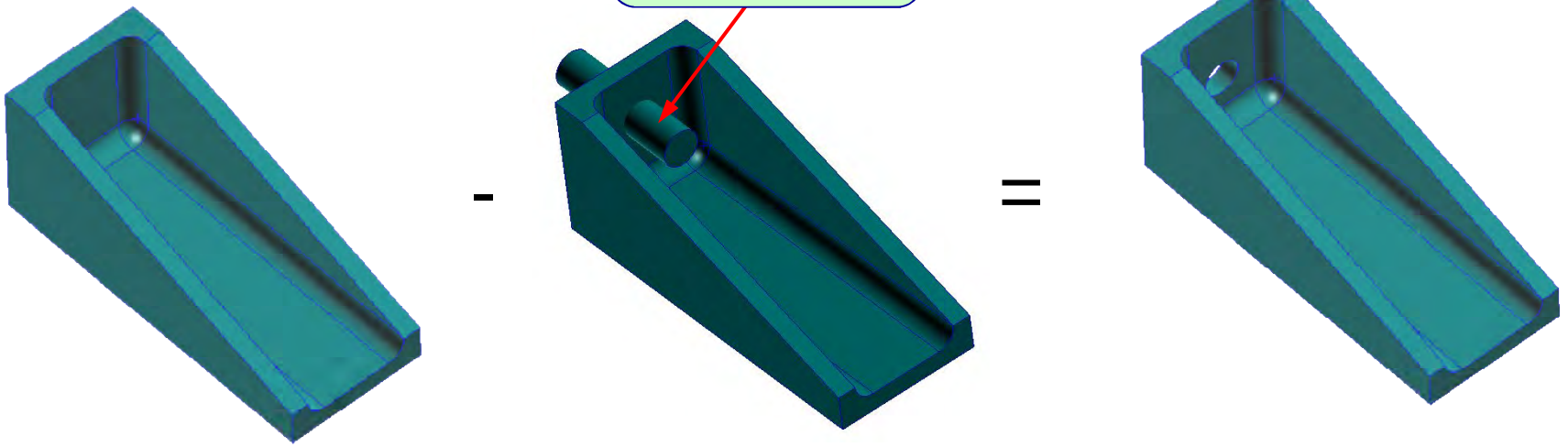
PRIMITIVE CREATION, AUTOMATIC BOOLEAN OPERATION

In this example the cylinder created will be automatically subtracted from the target solid, creating the hole in the target solid in one step.

Target Solid

Newly Created Primitive Solid (Cylinder)

Finished Part

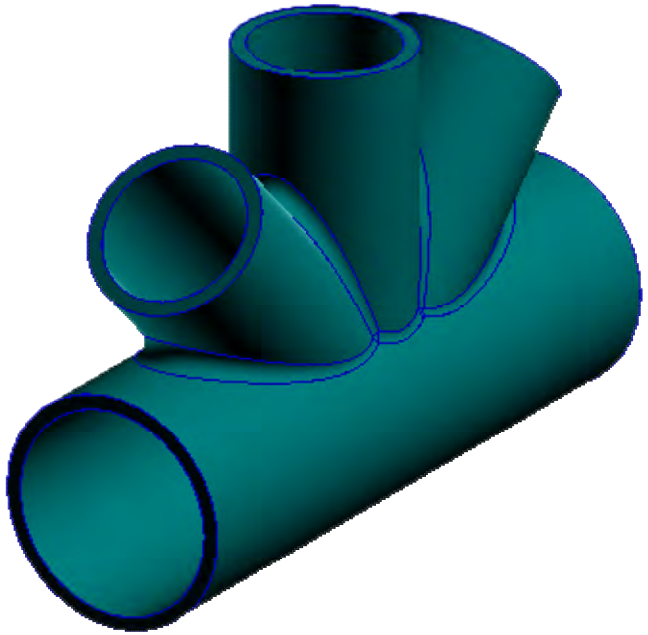


Copyright© 2012 MSC.Software Corporation

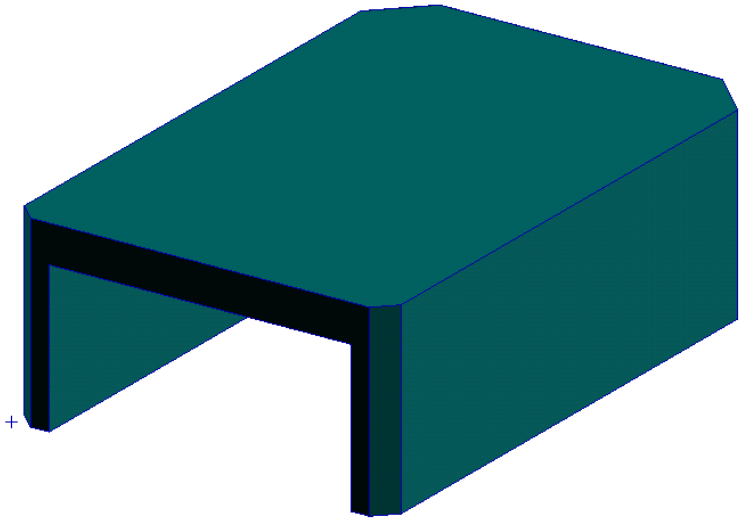
SOLID EDITING, EDGE BLEND OVERVIEW

- **Users can also add fillets and chamfers to a solid using the Edge Blend tool.**

For this model, an intersection of pipes and edge fillets were added using the Edit/Solid/Edge Blend tool under the Geometry menu.

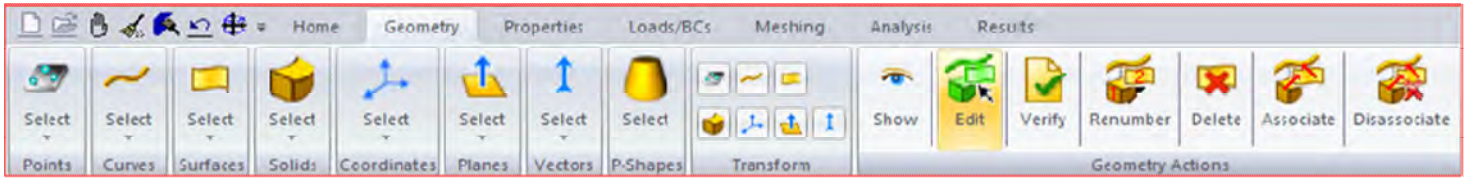


Chamfers were added to this solid using the same menu.



Copyright© 2012 MSC.Software Corporation

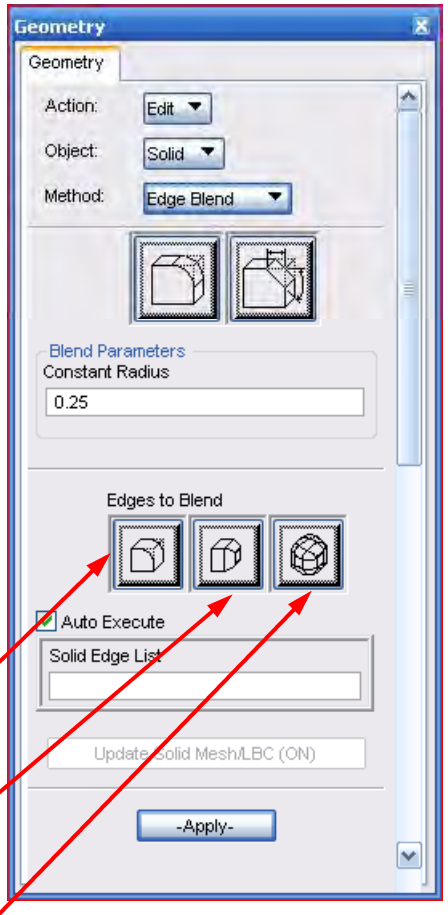
SOLID EDITING, EDGE BLEND



Fillet Option:

With this option, the user can add a fillet to a single edge of a solid, to all the edges of a face, or to all the edges of a solid.

Consider the solid illustrated here and add fillets to all the edges



Edge(s) of solid

All edges of face

All edges of solid

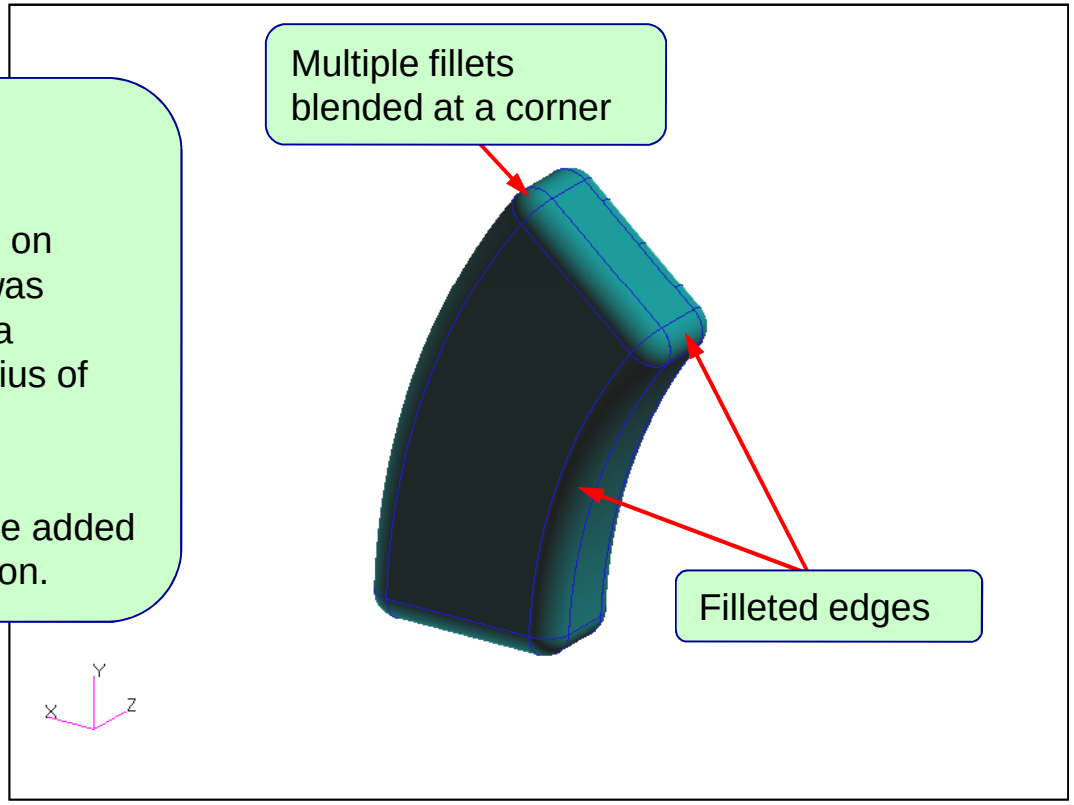
Copyright© 2012 MSC.Software Corporation

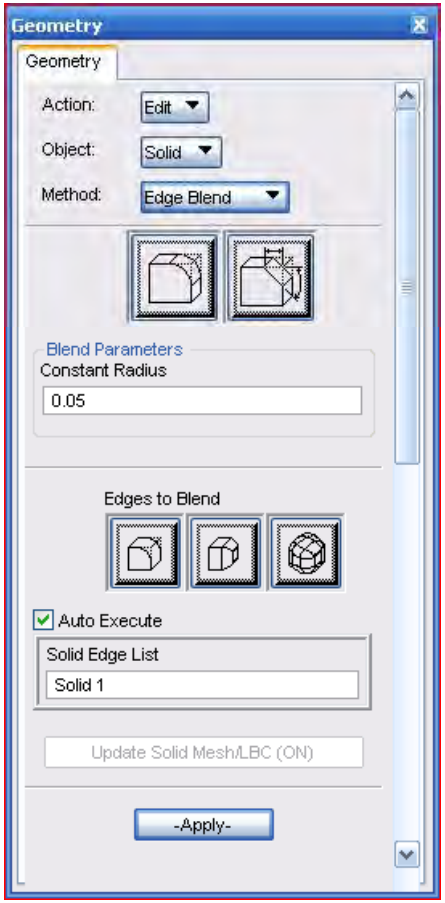
SOLID EDITING, EDGE BLEND

Fillet option:

The All Edges on Solid button was selected and a Constant Radius of 0.05 chosen.

The fillets were added in one operation.



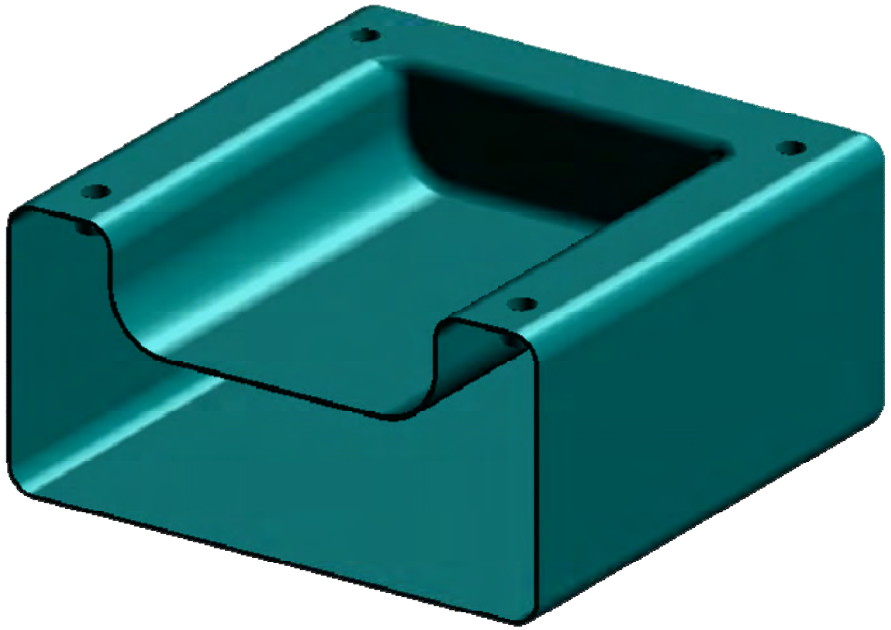


Copyright© 2012 MSC.Software Corporation

SOLID EDITING, SHELL OVERVIEW

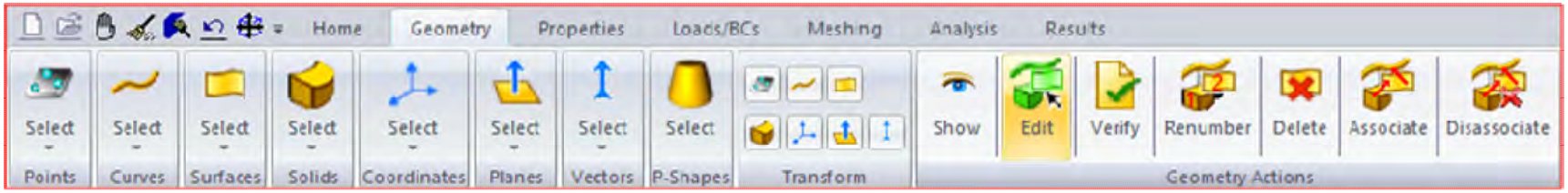
- **Users can remove the material on the interior of a solid using the Edit/Solid/Shell tool under the Geometry menu.**

The model illustrated to the right was obtained from a solid that was hollowed out utilizing the shell tool.



Copyright© 2012 MSC.Software Corporation

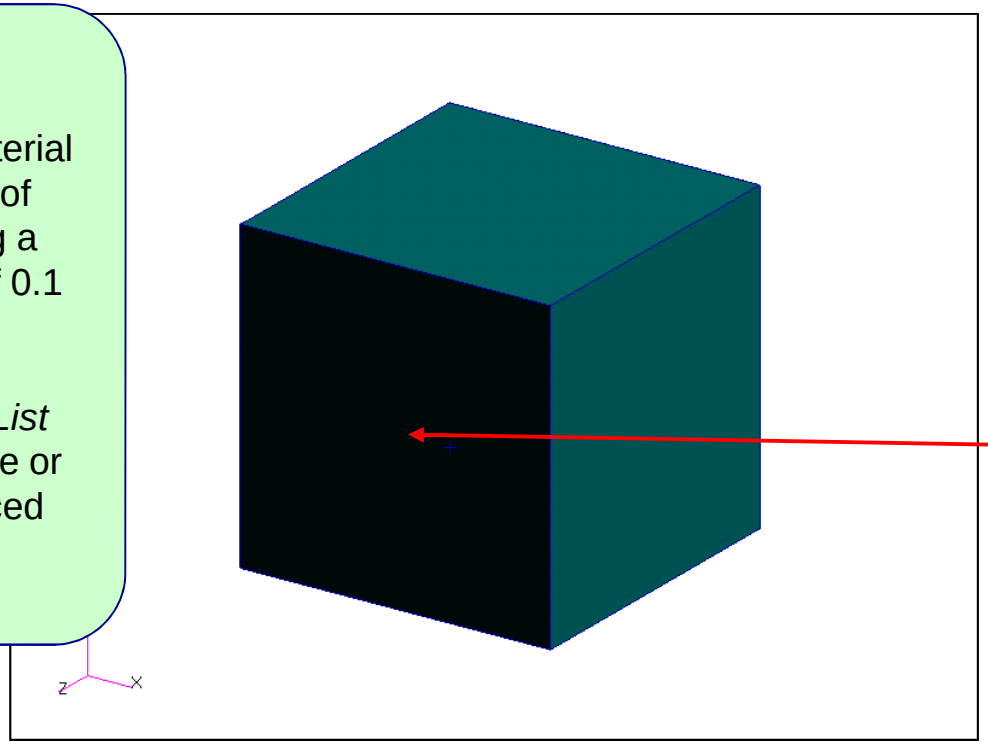
SOLID EDITING, SHELL OVERVIEW

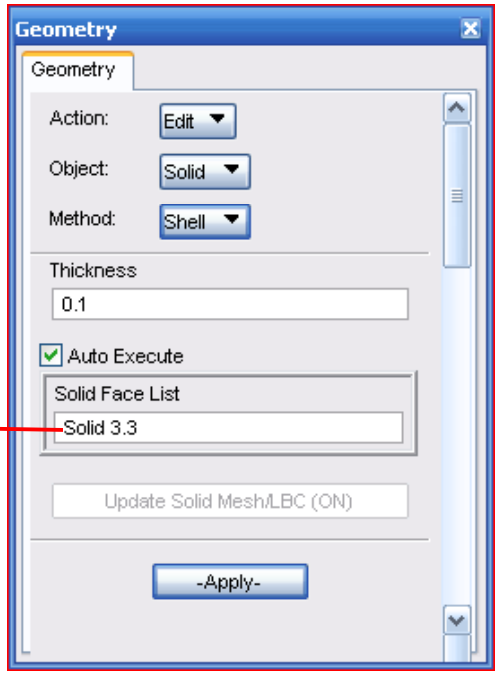


Shell option:

Remove the material from the interior of the solid, leaving a wall thickness of 0.1 inches.

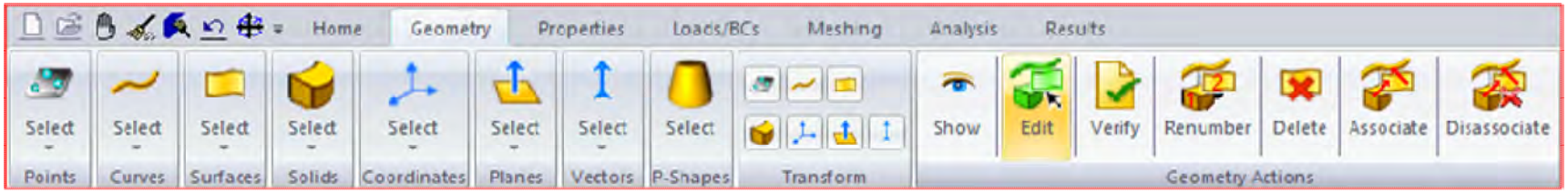
The *Solid Face List* identifies the face or faces to be pierced to remove the "material."





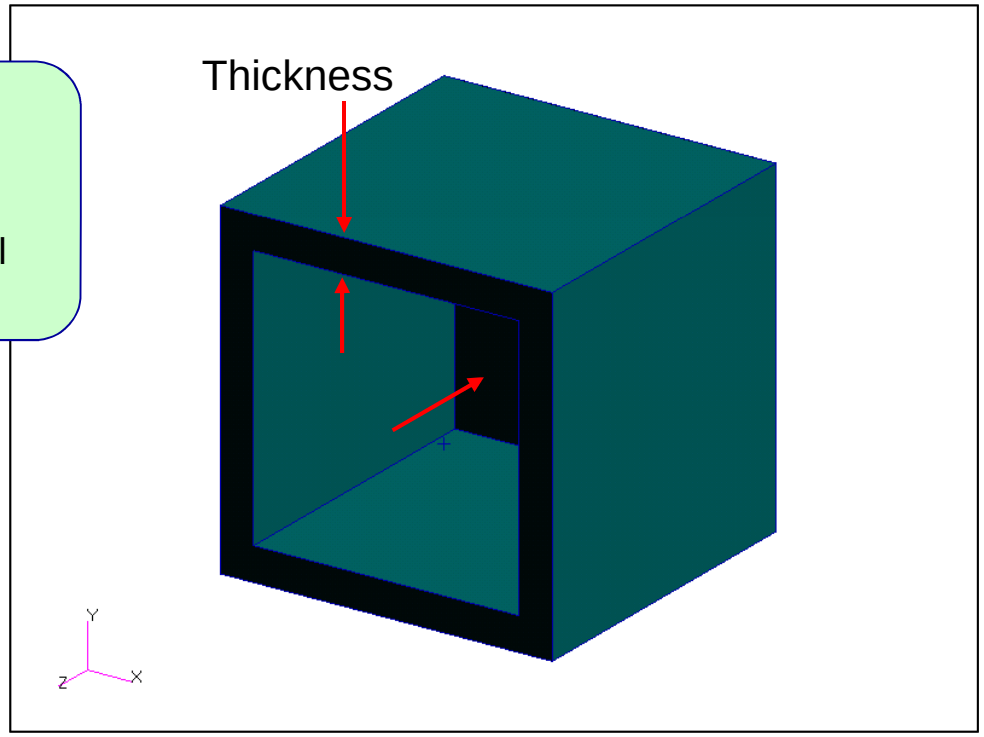
Copyright© 2012 MSC.Software Corporation

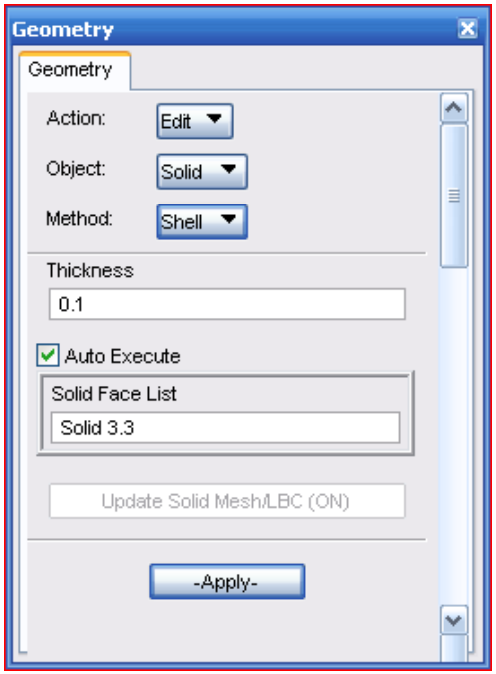
SOLID EDITING, SHELL OVERVIEW



Shell option:

The “material” is removed from the solid up to the wall thickness of 0.1

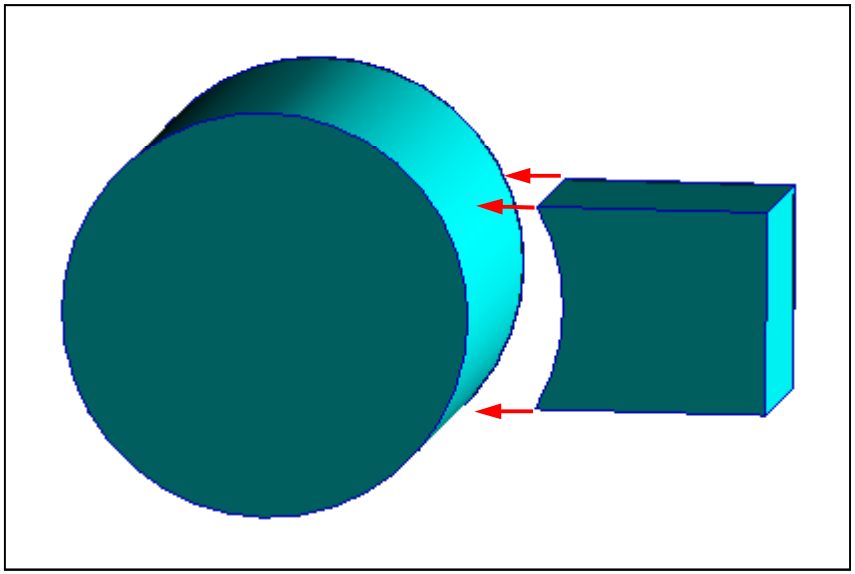




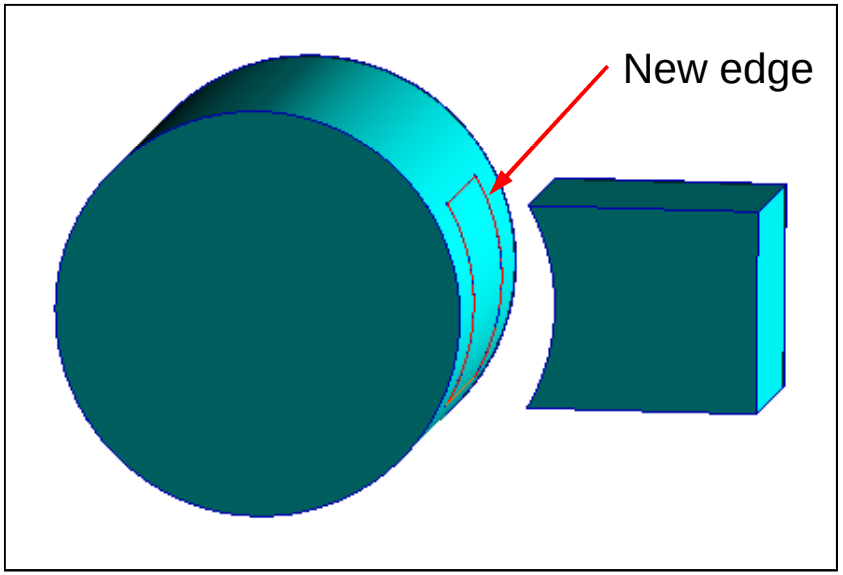
Copyright© 2012 MSC.Software Corporation

SOLID EDITING, IMPRINT OVERVIEW

- **The Imprint tool imprints one or more solids onto other solids.**



Before Imprint



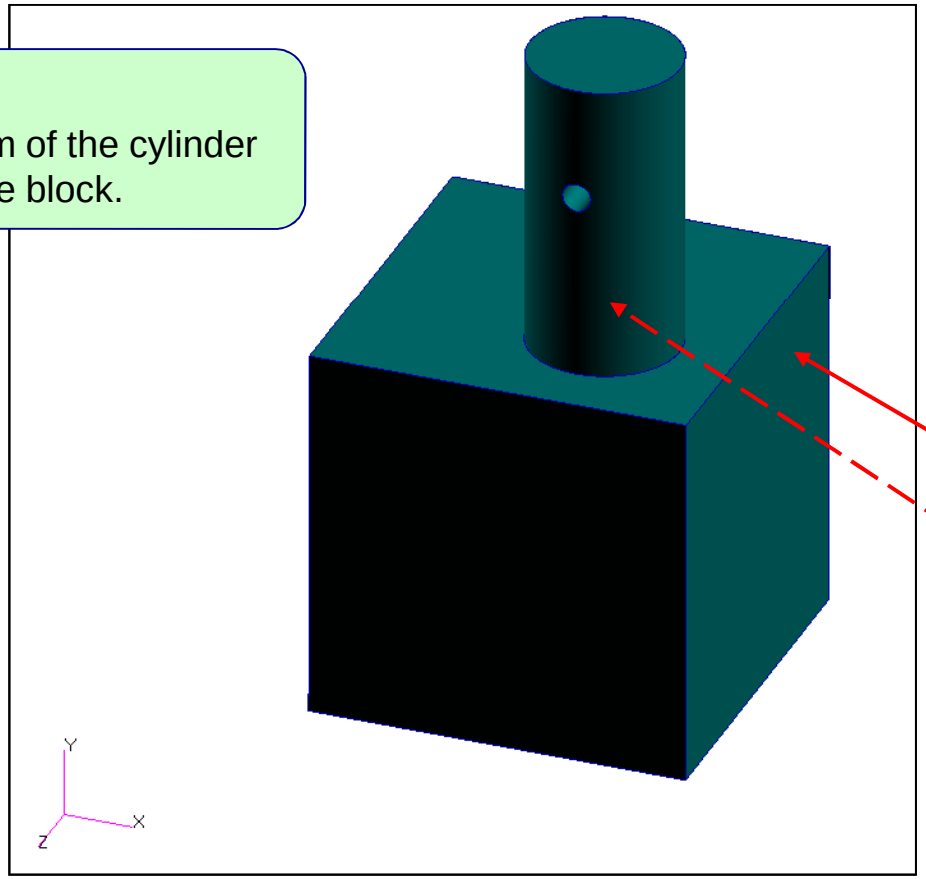
After Imprint

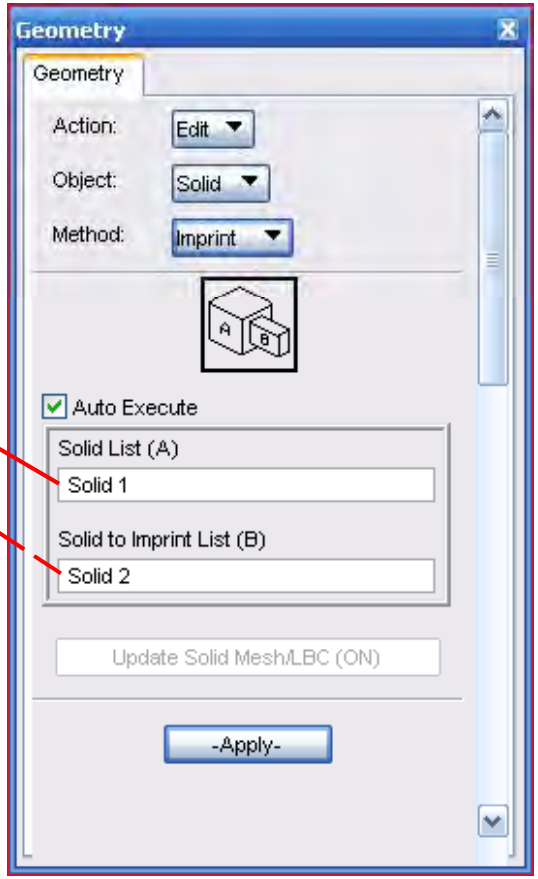
Copyright© 2012 MSC.Software Corporation

SOLID EDITING IMPRINT OVERVIEW

Imprint option:

Imprint the bottom of the cylinder onto the top of the block.





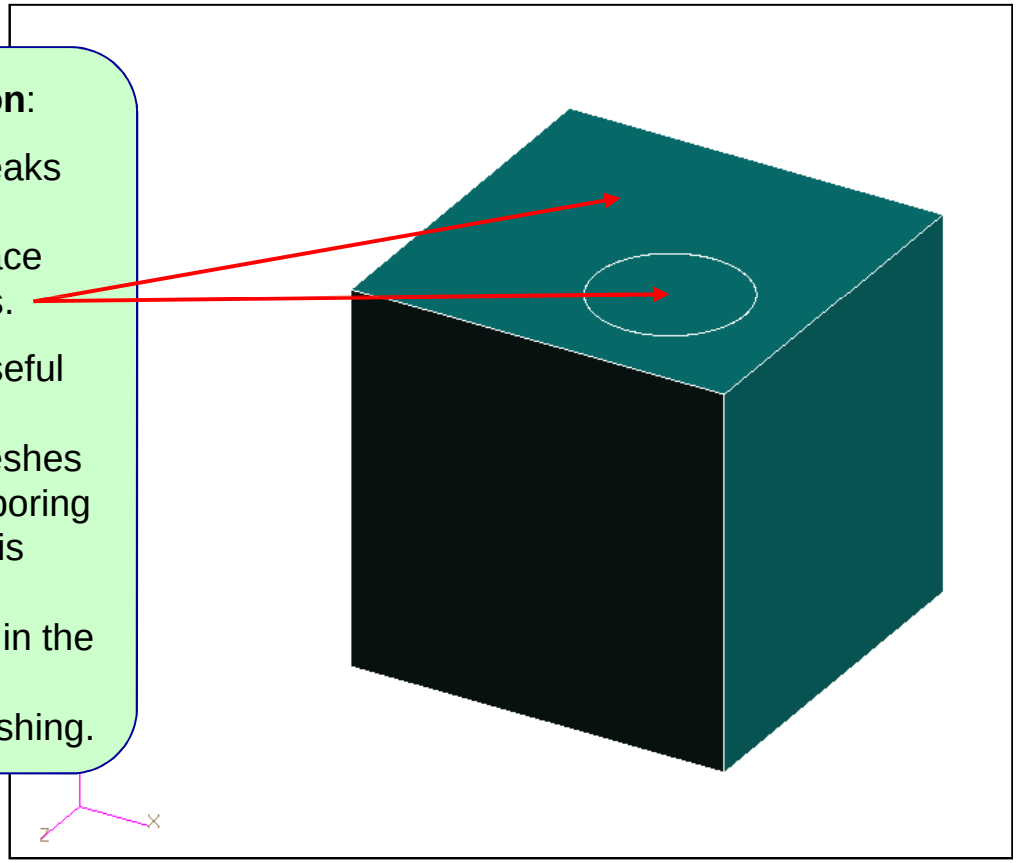
Copyright© 2012 MSC.Software Corporation

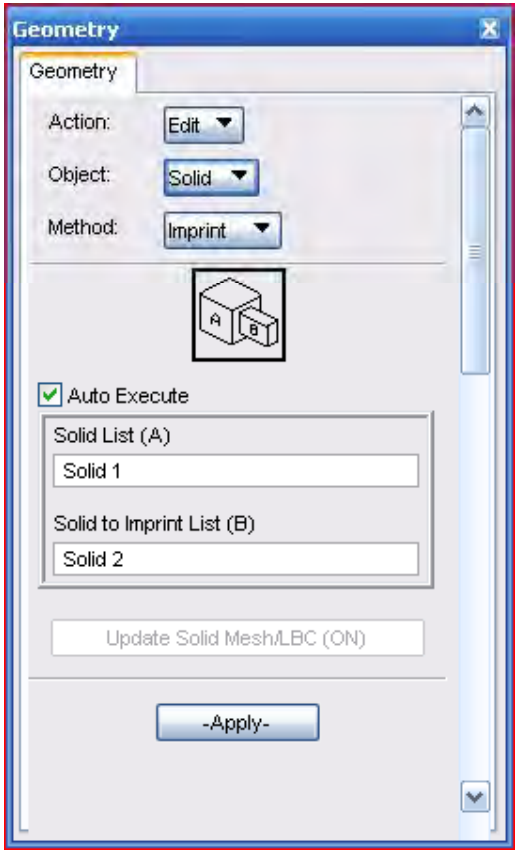
SOLID EDITING IMPRINT OVERVIEW

Imprint option:

Imprinting breaks the larger rectangular face into two faces.

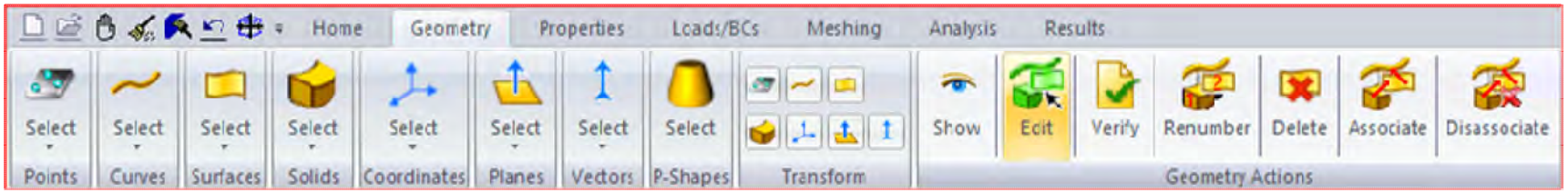
This tool is useful for creating congruent meshes across neighboring solids, which is discussed in greater detail in the section on assembly meshing.



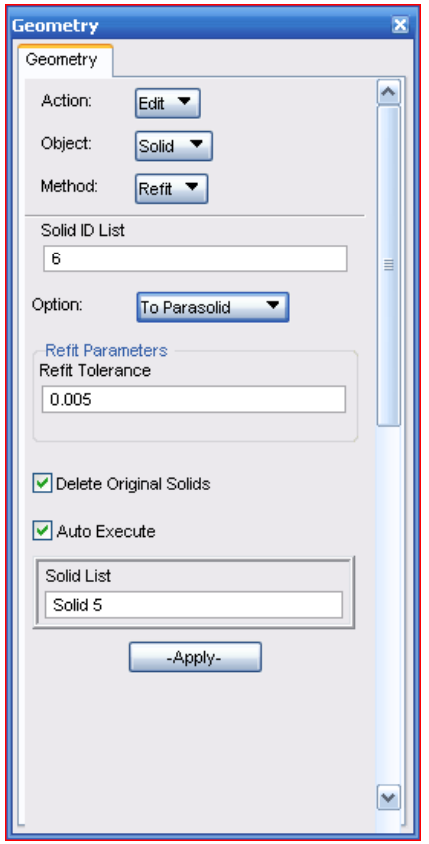


Copyright© 2012 MSC.Software Corporation

SOLID EDITING, REFIT TO PARASOLID

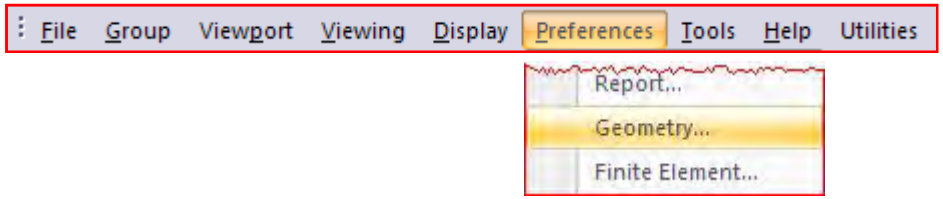


- **Non-parasolid geometry can be converted into parasolid geometry by using the Refit Option: To Parasolid tool.**
- **Once the geometry is converted to parasolid geometry, the user can take advantage of the new parasolid editing tools described in this section.**
- **Alternatively, the user may choose to let Patran automatically refit the geometry to parasolid geometry. This automatic refit occurs whenever a parasolid editing operation is requested and Patran detects that the solids involved are not parasolid solids.**

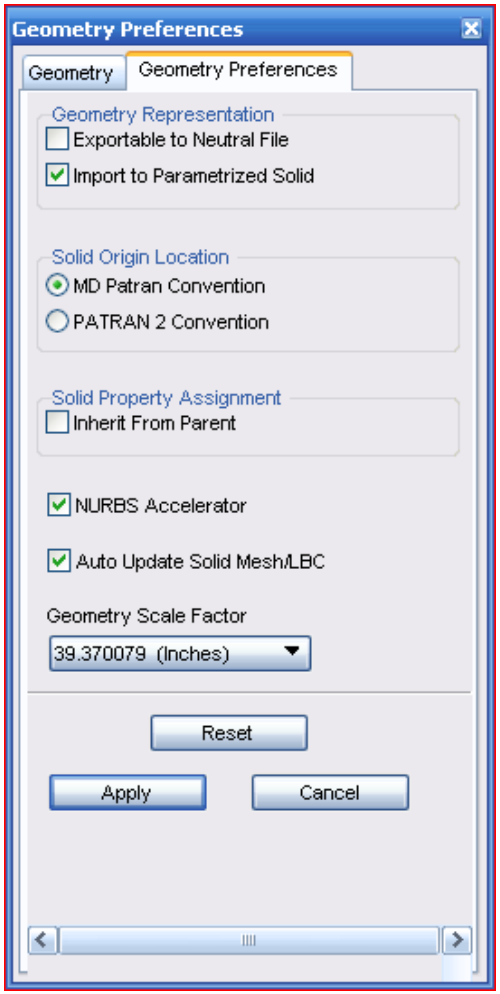


Copyright© 2012 MSC.Software Corporation

SOLID EDITING, AUTO UPDATE OF CAE DATA



- **The Auto Update Solid Mesh/LBC toggle (set to yes), from the Preferences / Geometry… form, causes re-meshing using previously specified mesh parameters, and re-applying of loads and boundary conditions after geometry modification (such as a Boolean operation).**

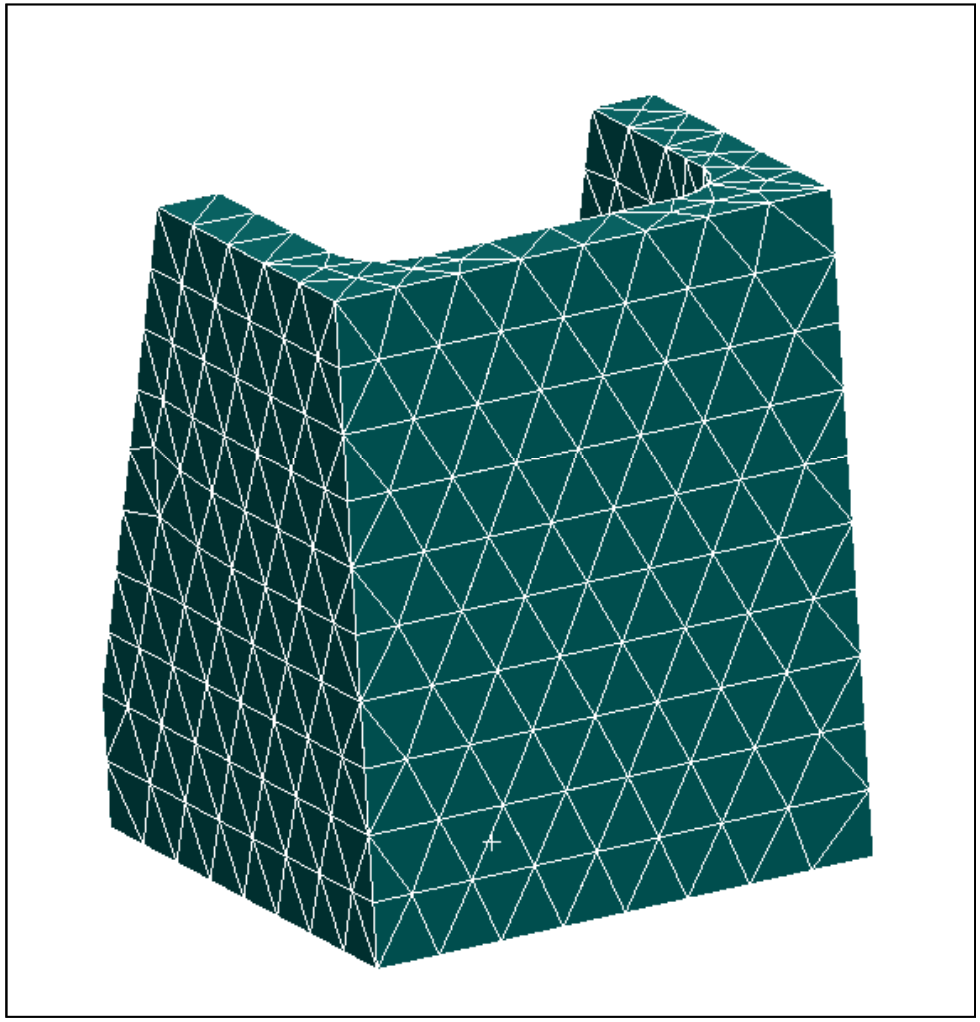


Copyright© 2012 MSC.Software Corporation

SOLID EDITING, AUTO UPDATE OF CAE DATA

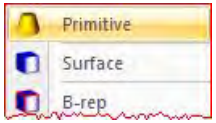
For example, the parasolid solid on the right has been meshed with Tet10 elements.

It is desired to put a hole through the solid using Boolean subtract.

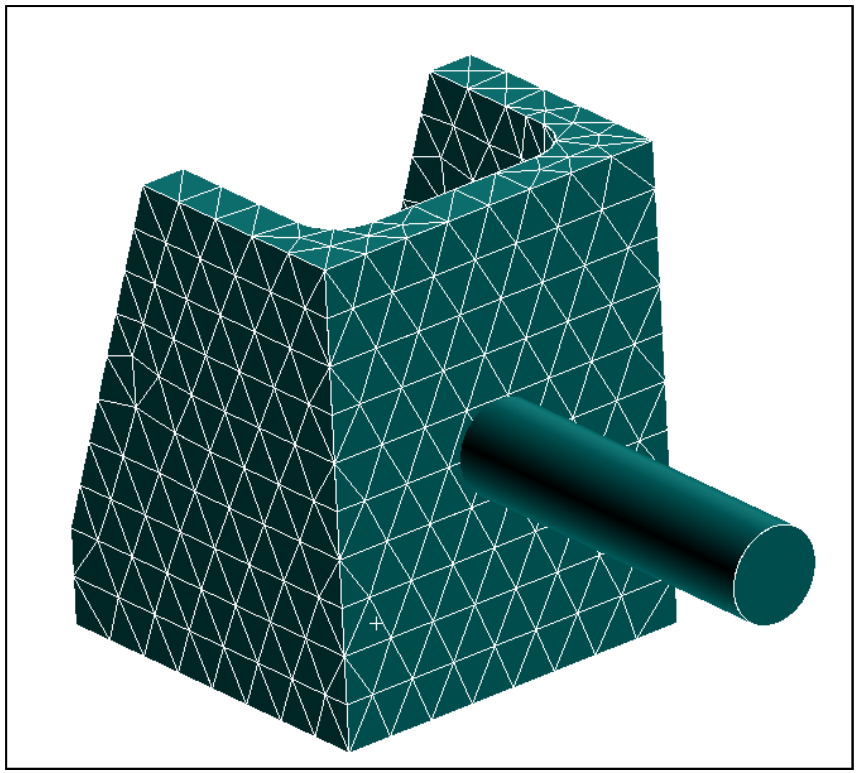


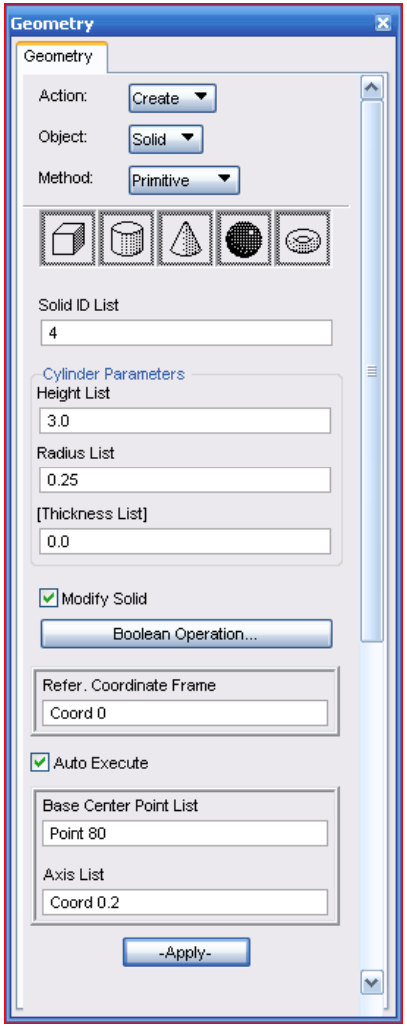
Copyright© 2012 MSC.Software Corporation

SOLID EDITING, AUTO UPDATE OF CAE DATA



A cylinder is created and will be used in the creation of the hole.



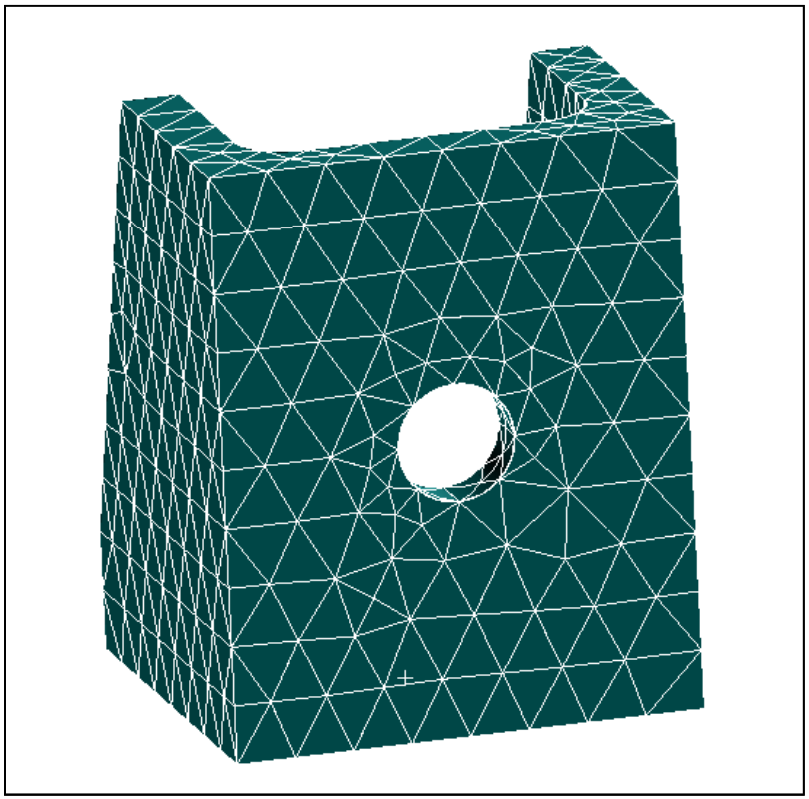


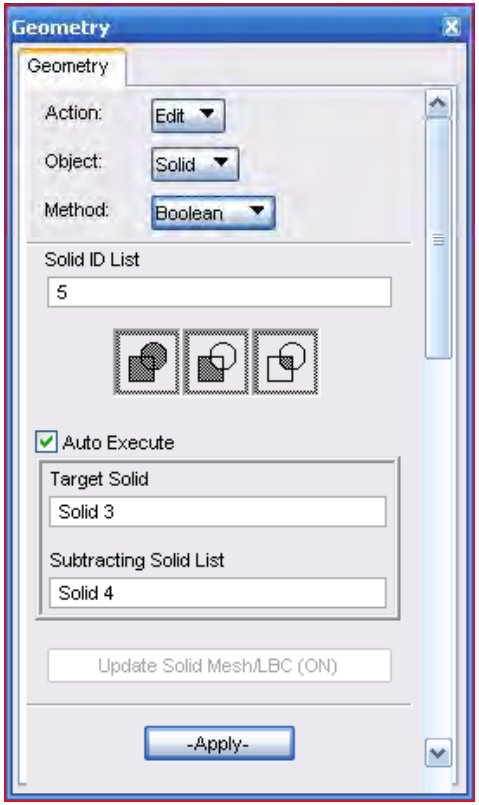
Copyright© 2012 MSC.Software Corporation

SOLID EDITING, AUTO UPDATE OF CAE DATA

Use the Boolean subtract tool to create the hole.

The resulting solid is modified, and the tet mesh is automatically updated.





Copyright© 2012 MSC.Software Corporation

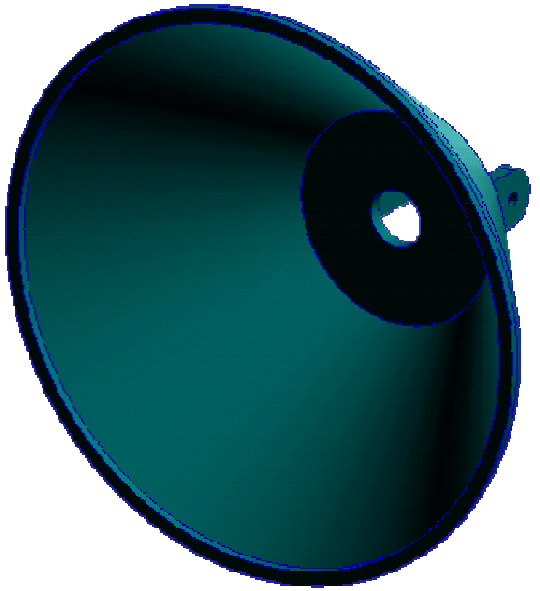
PRIMITIVE SOLID MODELING EXAMPLE

- **The following case study, Lamp Housing, will demonstrate the primitive solid modeling tools.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING

- **Model the lamp housing shown below using primitive geometry, shell, fillets, and Booleans.**



Copyright© 2012 MSC.Software Corporation

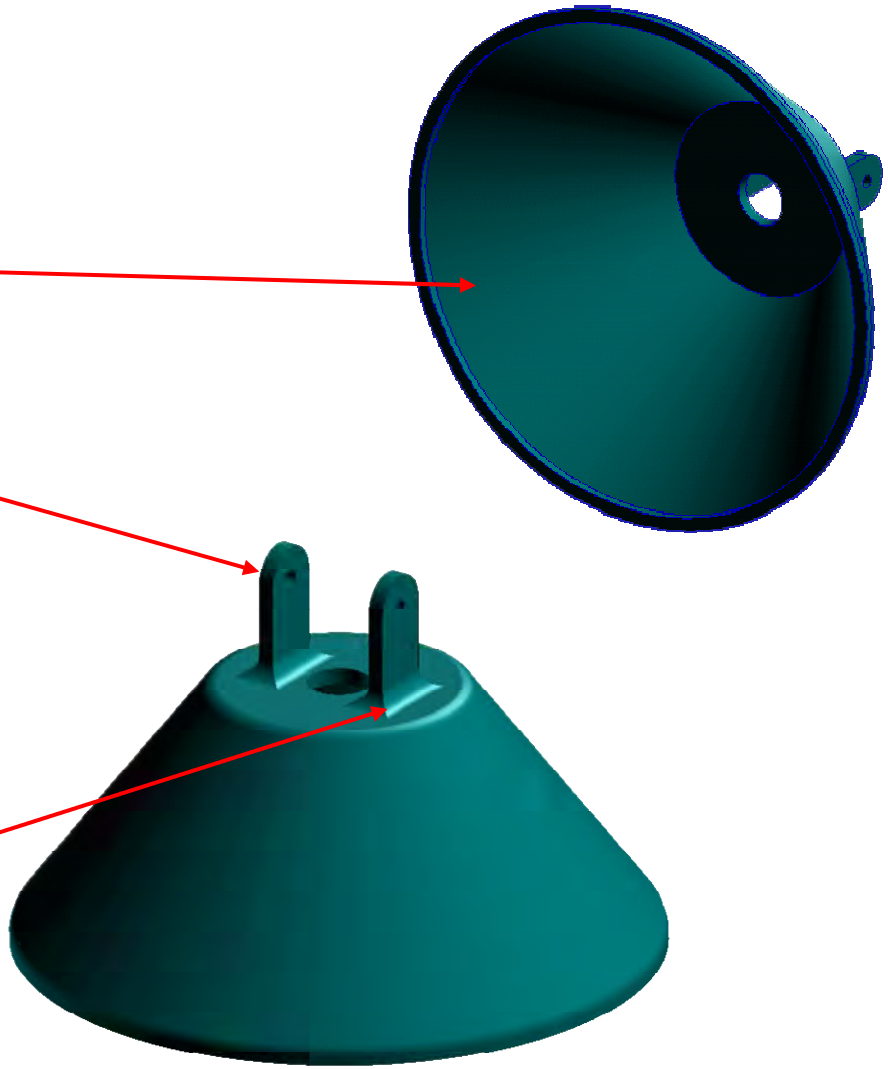
CASE STUDY: LAMP HOUSING

Step 1: Model the base geometry using a primitive cone.

Step 2: Use a shell operation to hollow out the cone.

Step 3: Use a trimmed surface to model the outline of a tab. Extrude the surface to create a solid, create a second tab, and use a Boolean Add to join the tabs to the main solid.

Step 4: Add the fillets to the solid.

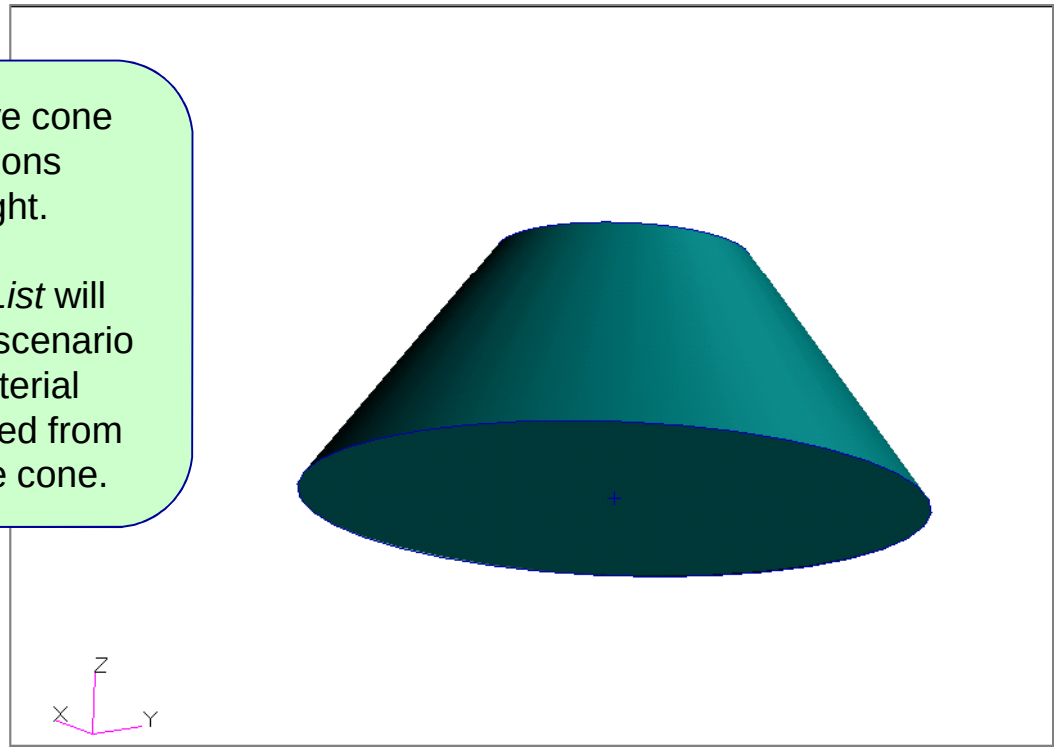


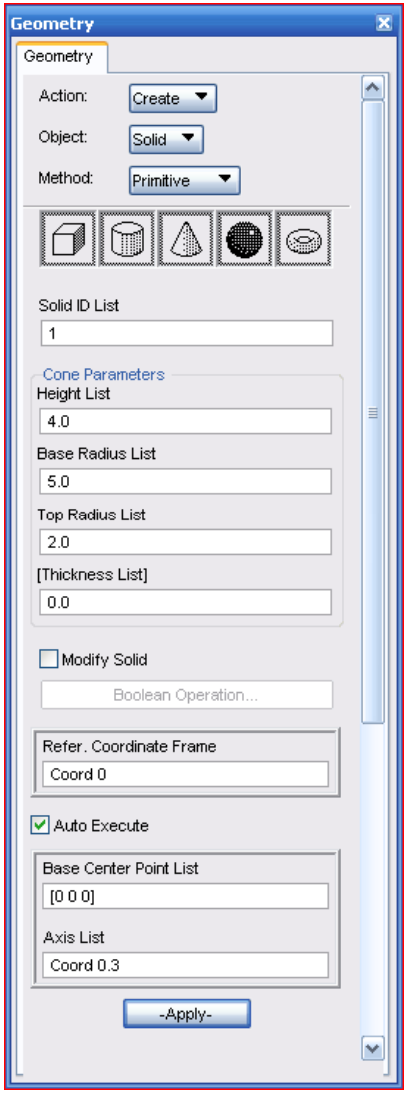
Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING

Define a primitive cone with the dimensions shown on the right.

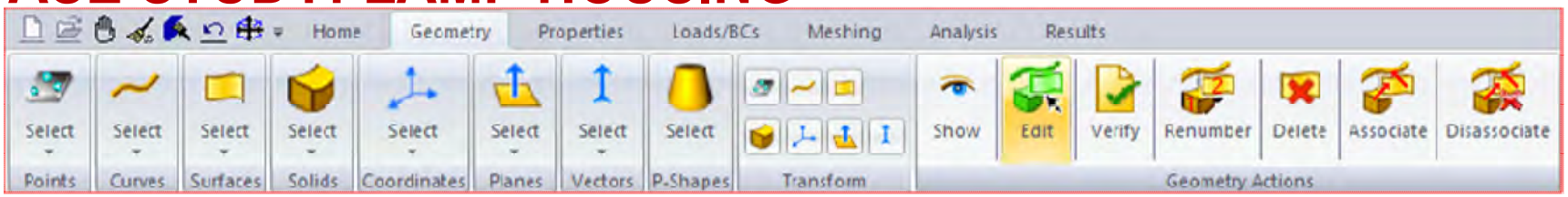
The *Thickness List* will not work in this scenario because the material would be removed from both sides of the cone.



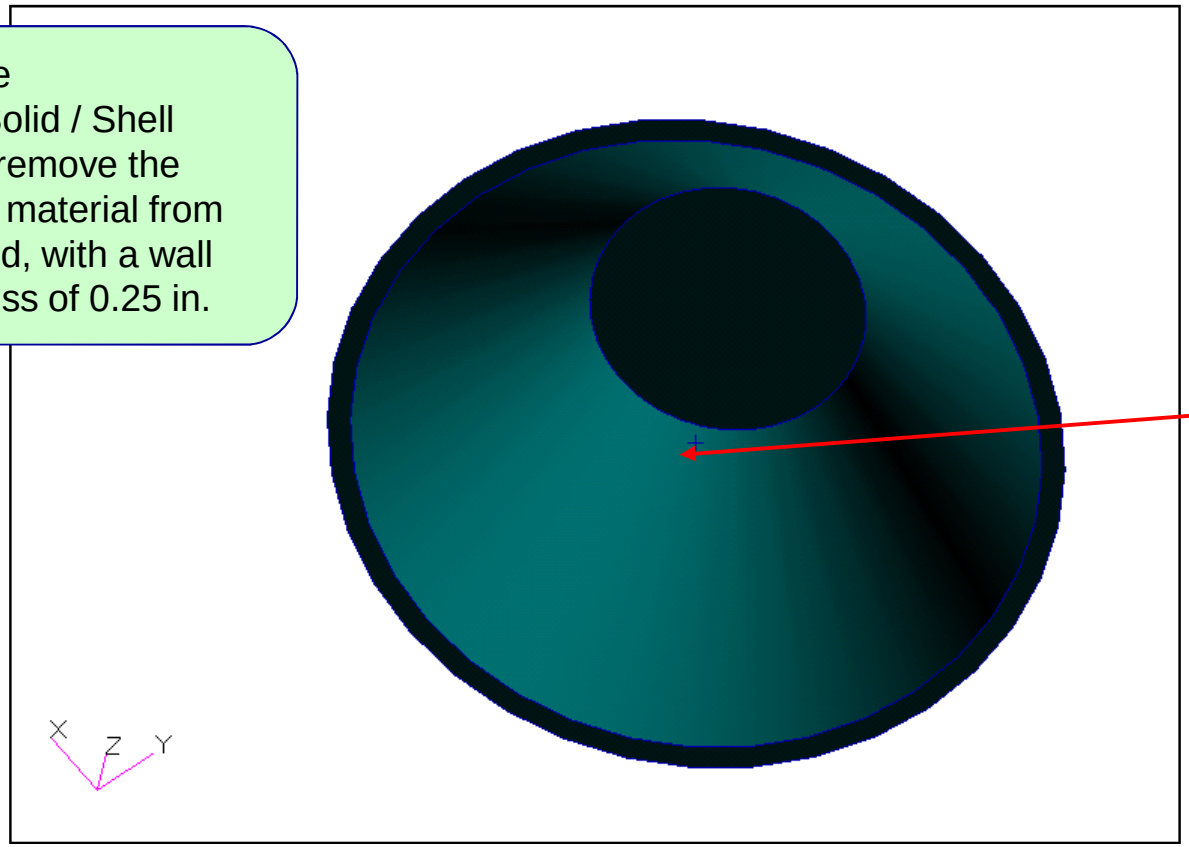


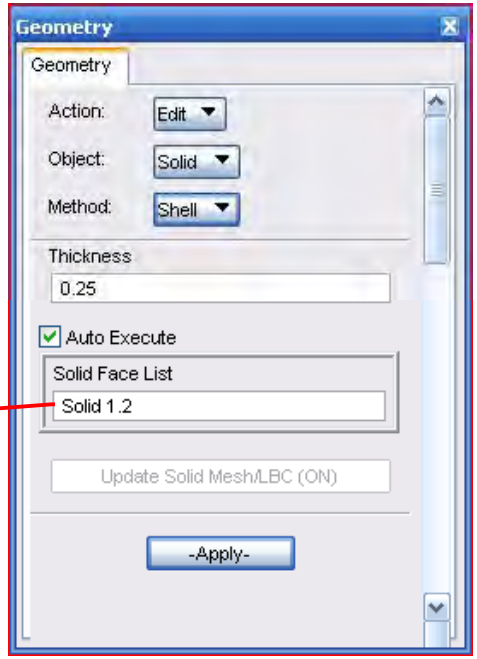
Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING



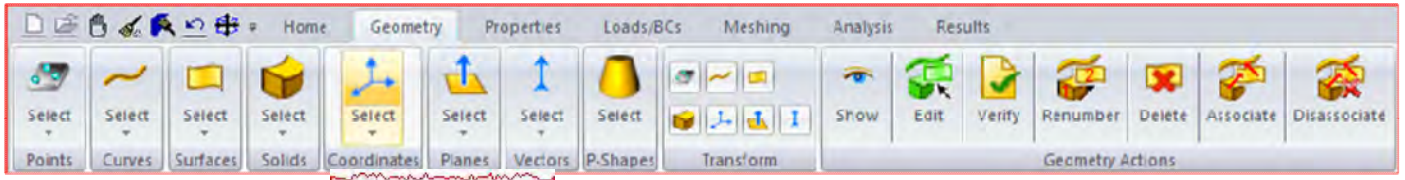
Use the Edit / Solid / Shell tool to remove the interior material from the solid, with a wall thickness of 0.25 in.

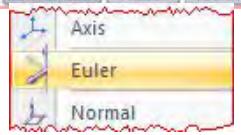




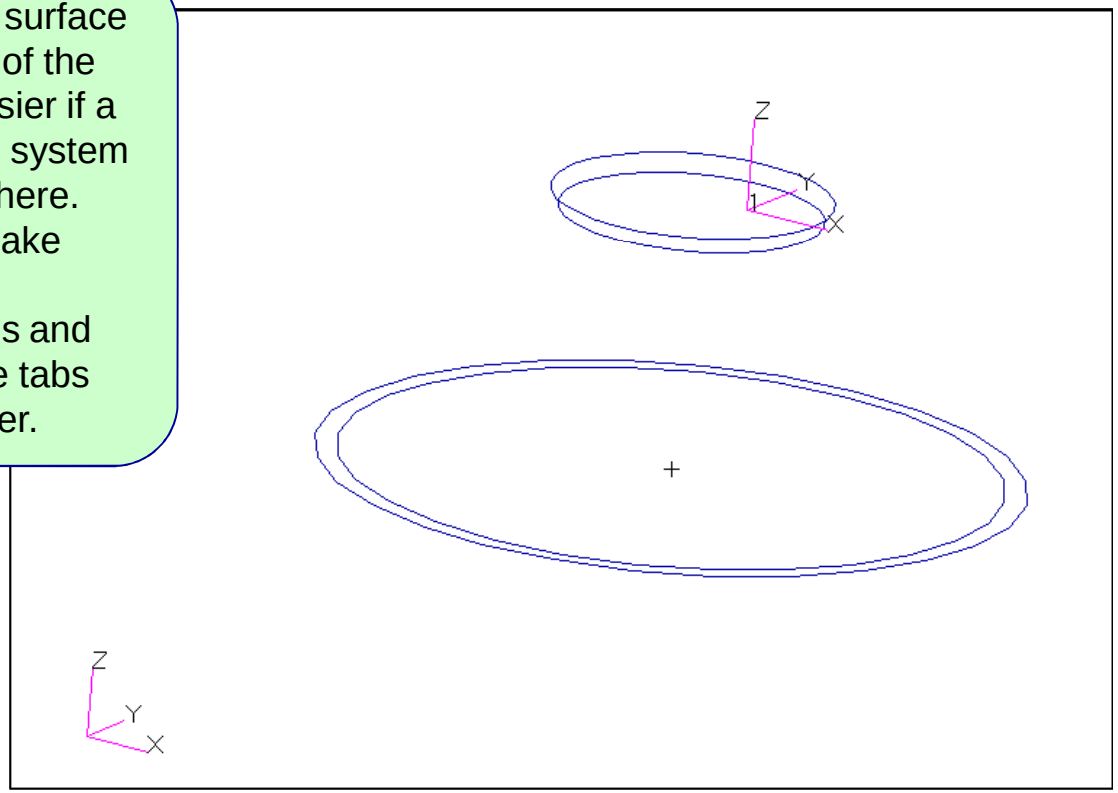
Copyright© 2012 MSC.Software Corporation

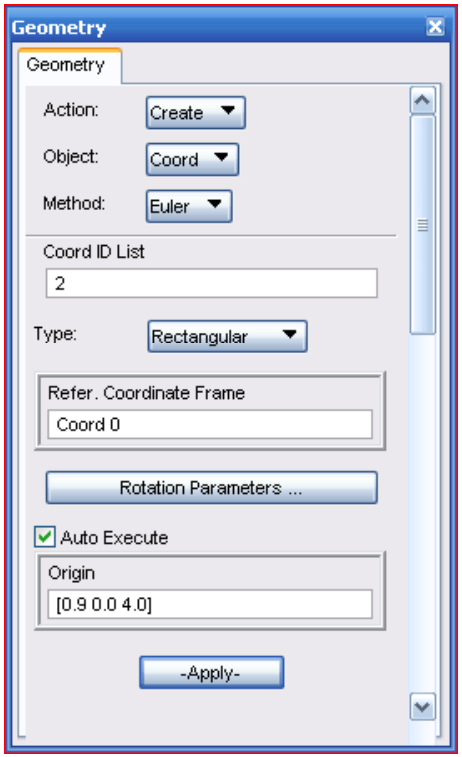
CASE STUDY: LAMP HOUSING





Defining a surface on the top of the base is easier if a coordinate system is placed there. This will make defining coordinates and moving the tabs much easier.



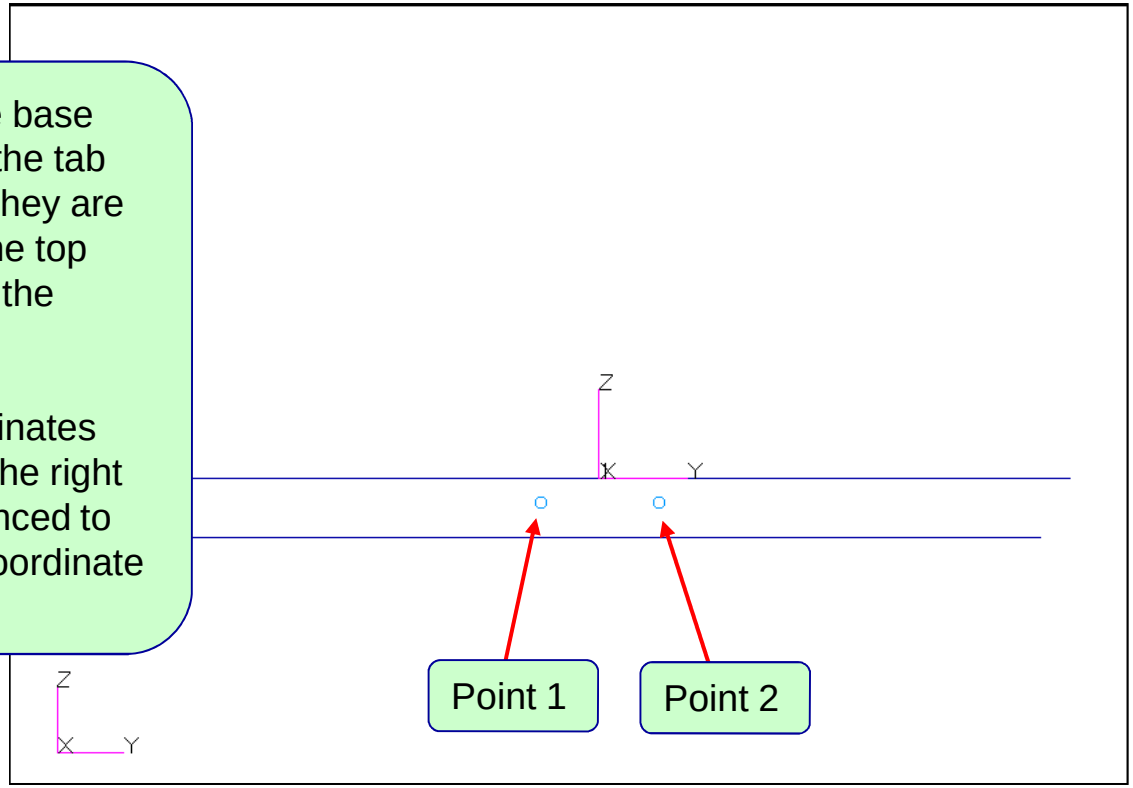


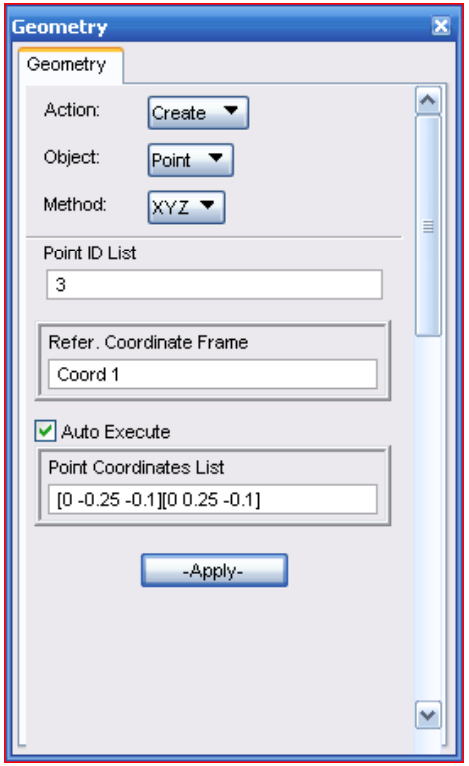
Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING

Create the base points for the tab such that they are beneath the top surface of the cone.

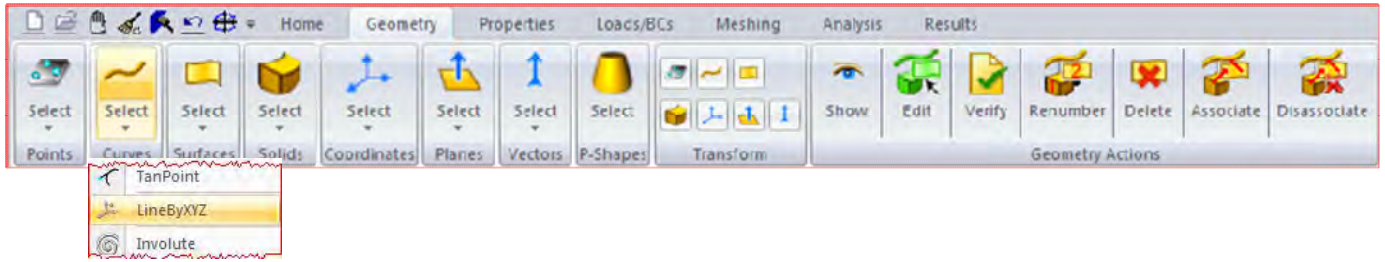
The coordinates shown to the right are referenced to the new coordinate system.

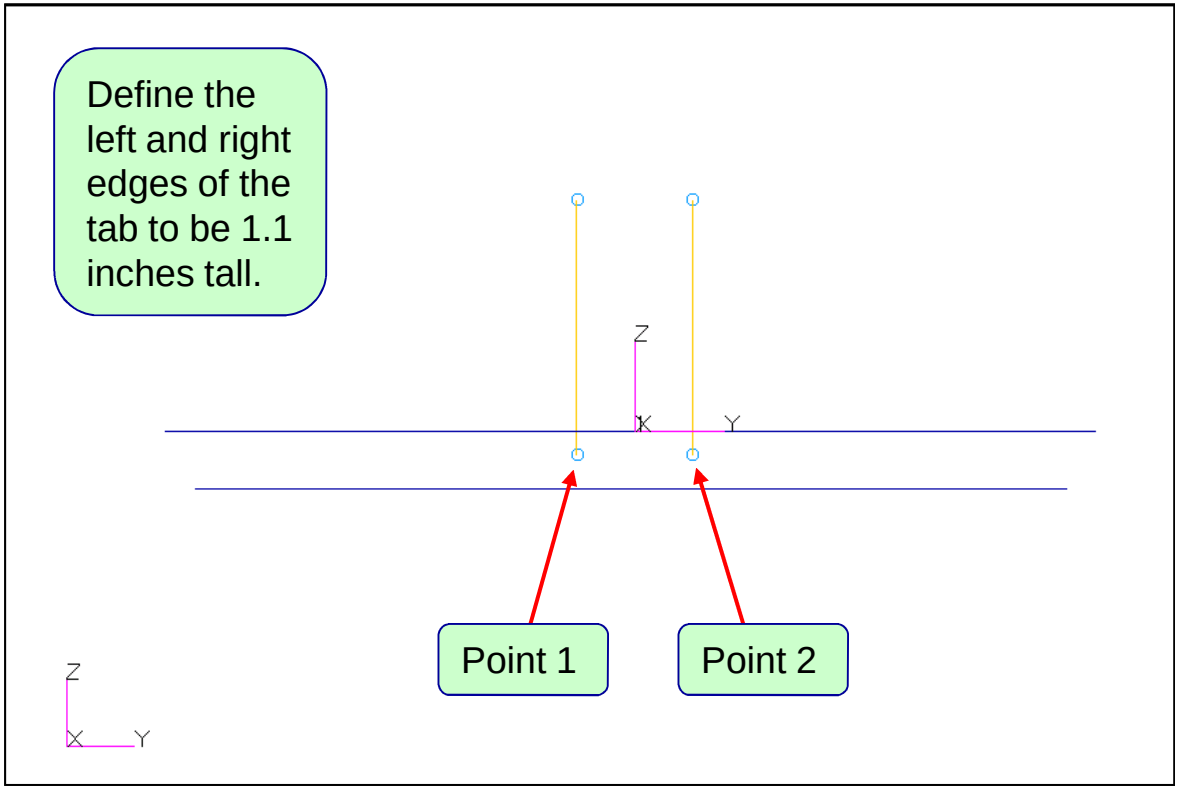


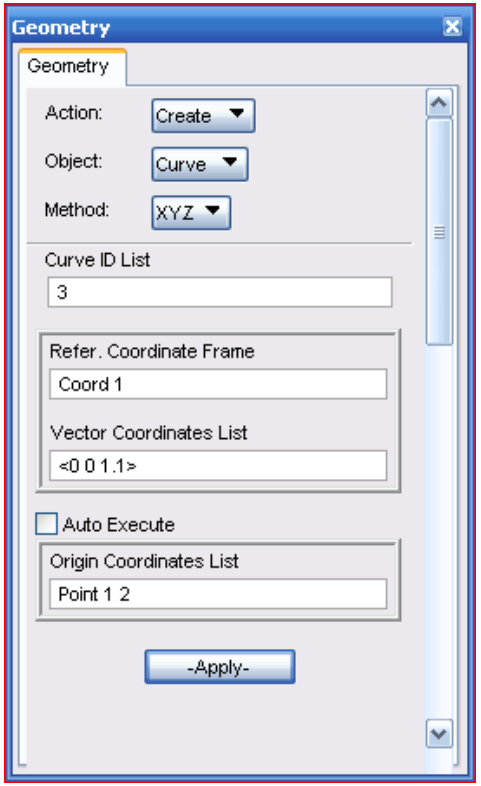


Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING



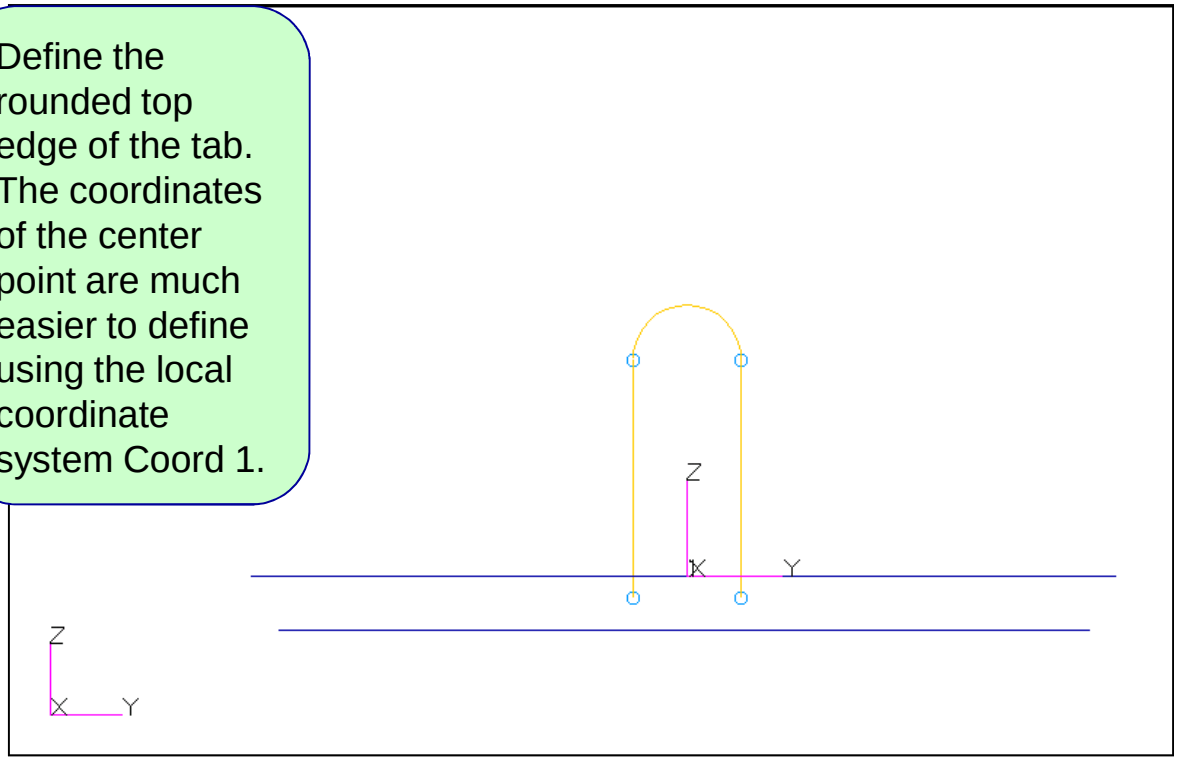


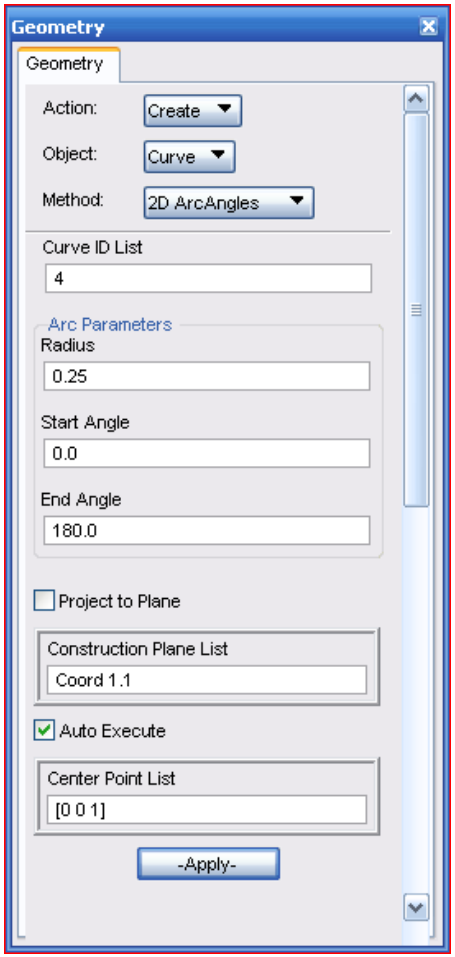


Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING

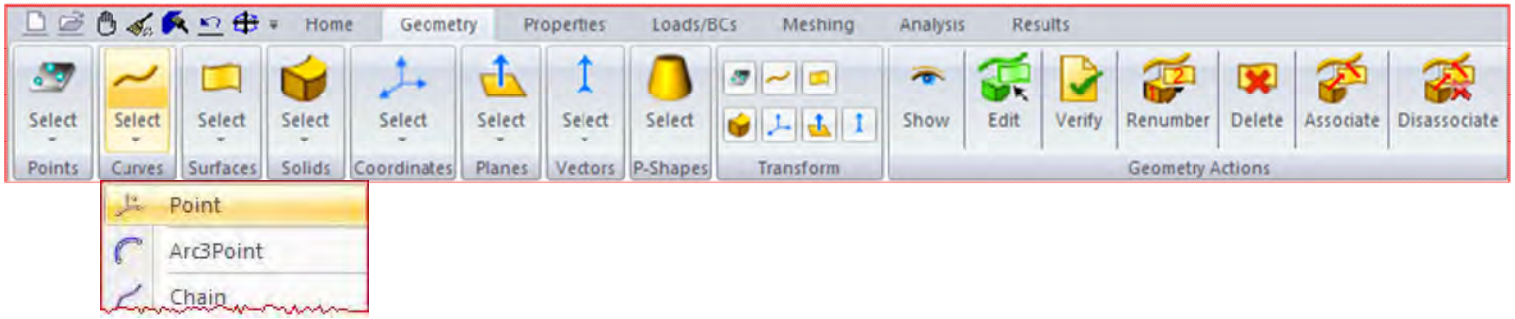
Define the rounded top edge of the tab. The coordinates of the center point are much easier to define using the local coordinate system Coord 1.



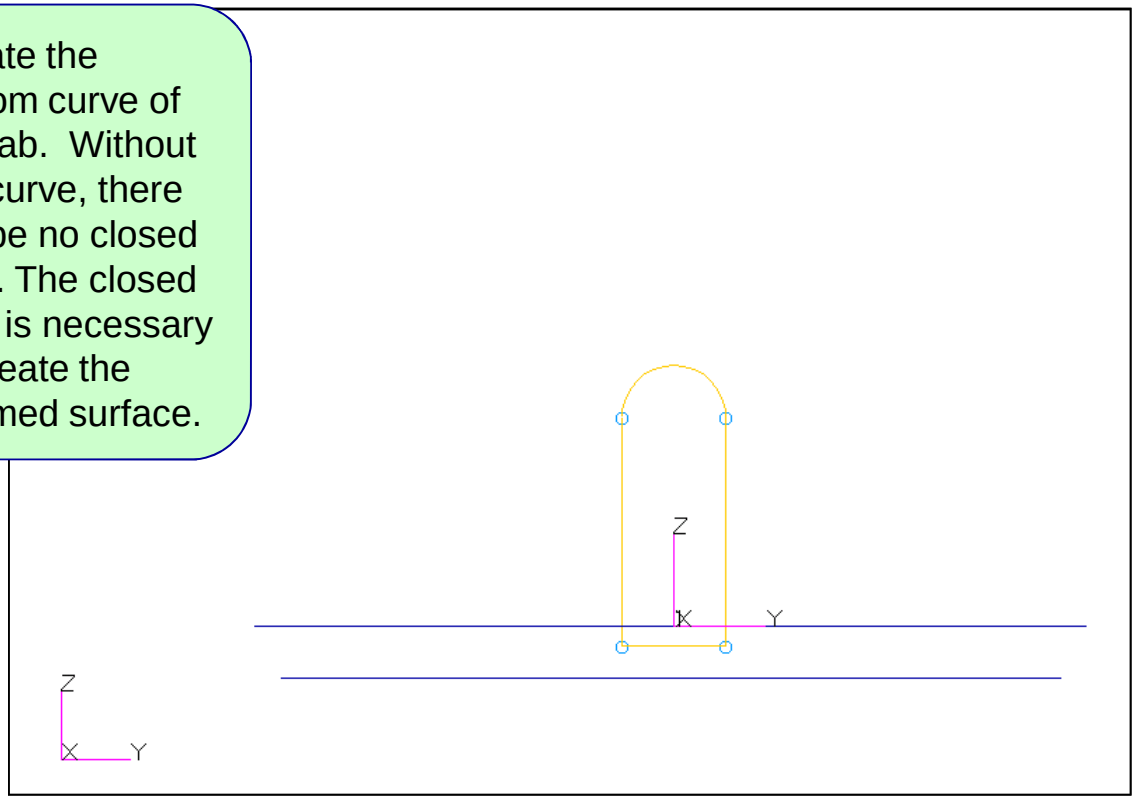


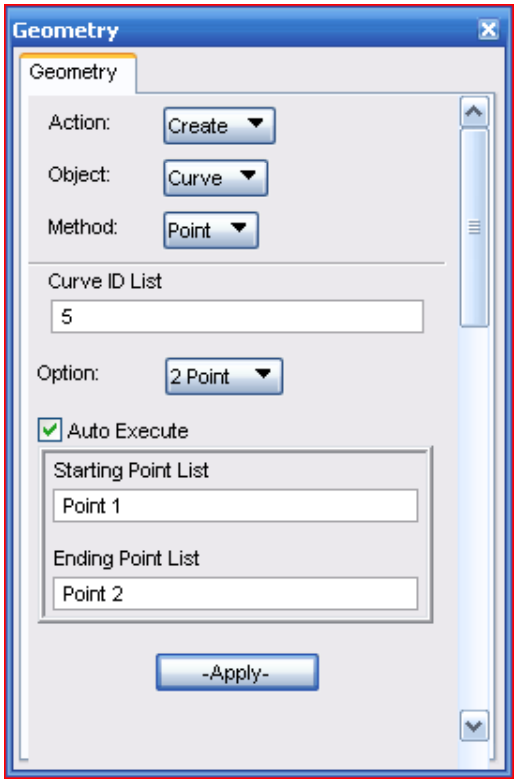
Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING



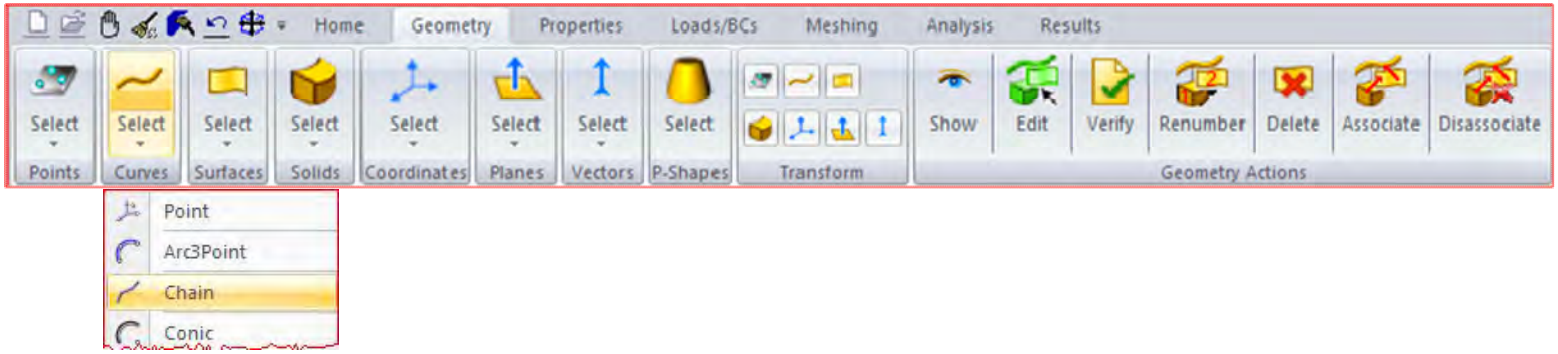
Create the bottom curve of the tab. Without the curve, there will be no closed loop. The closed loop is necessary to create the trimmed surface.



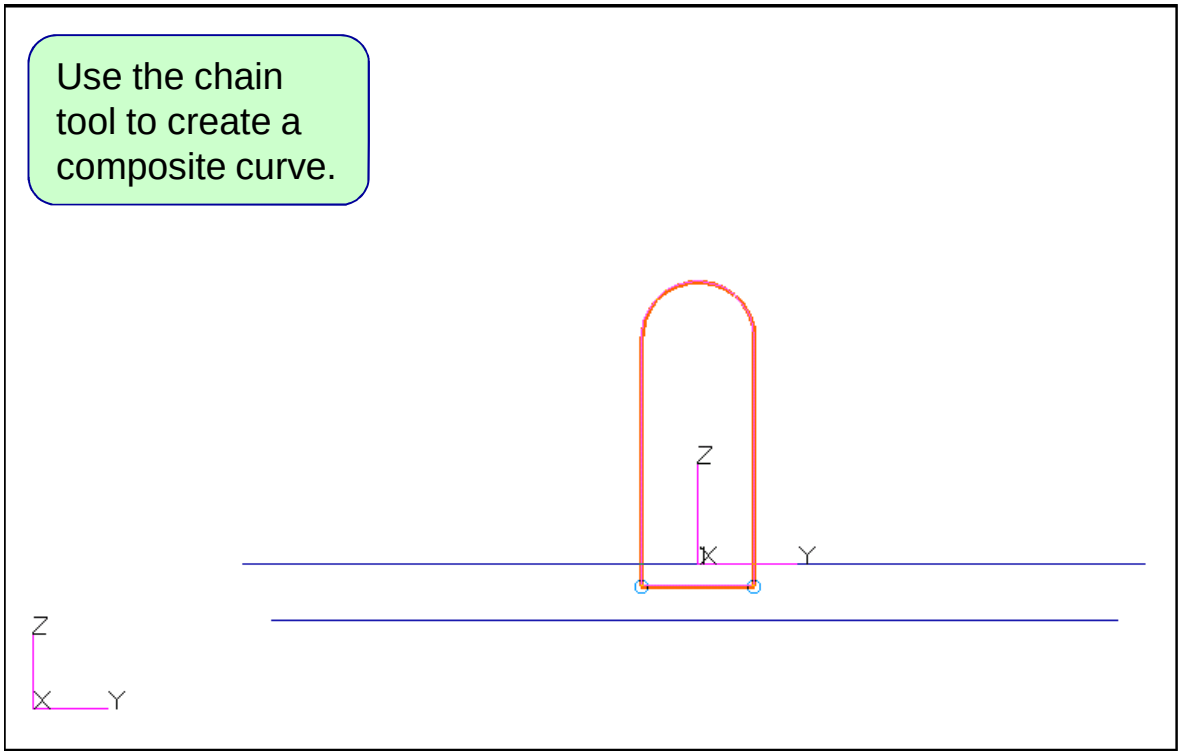


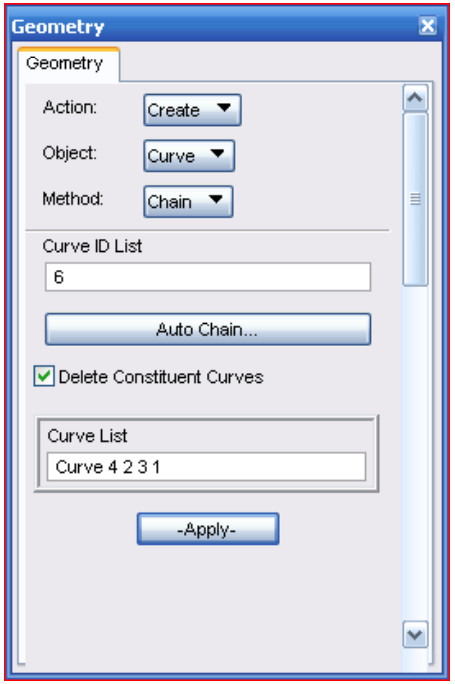
Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING



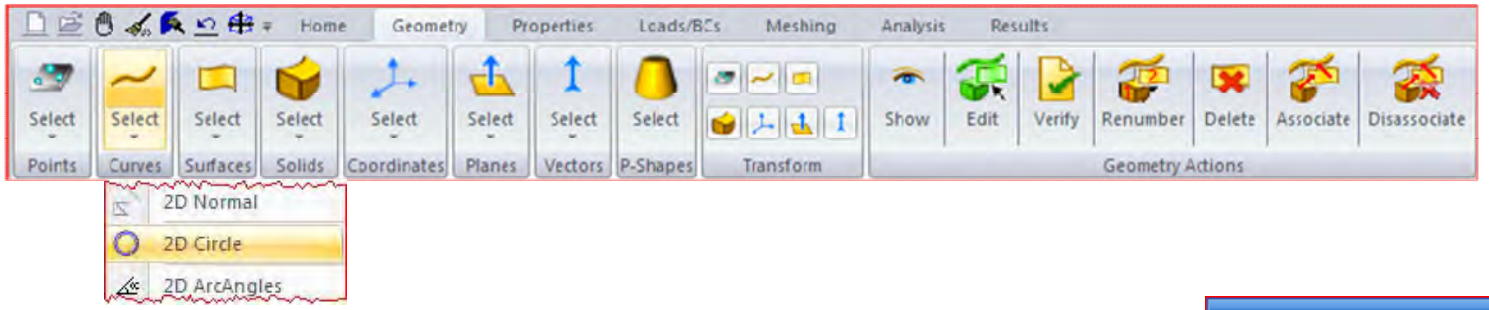
Use the chain tool to create a composite curve.



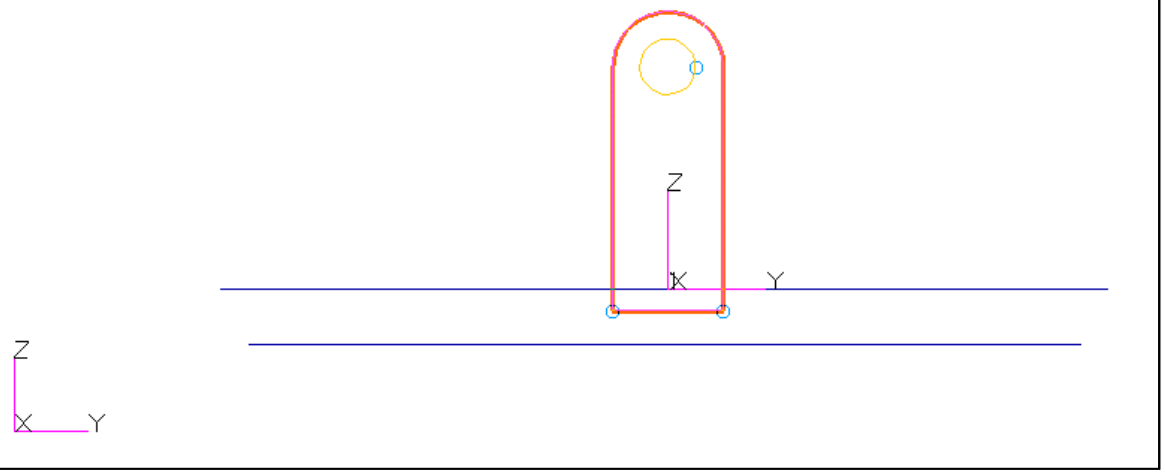


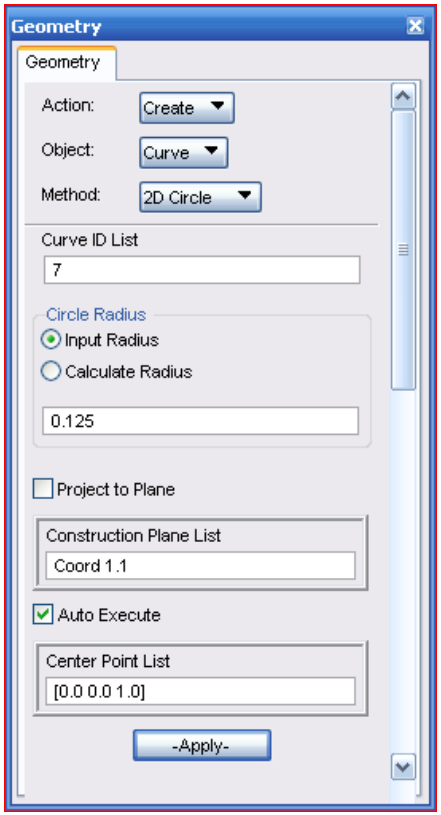
Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING



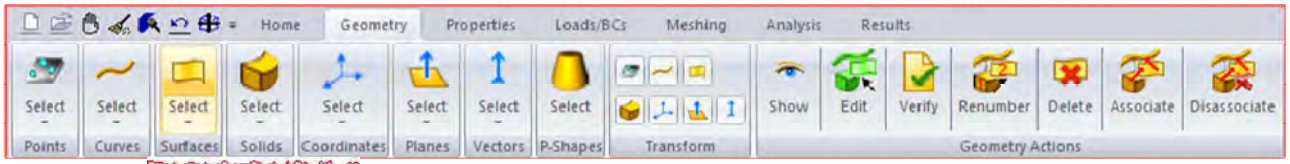
Define the hole for the tab with a radius of 0.125 inch.

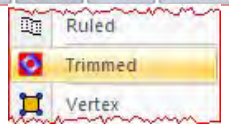




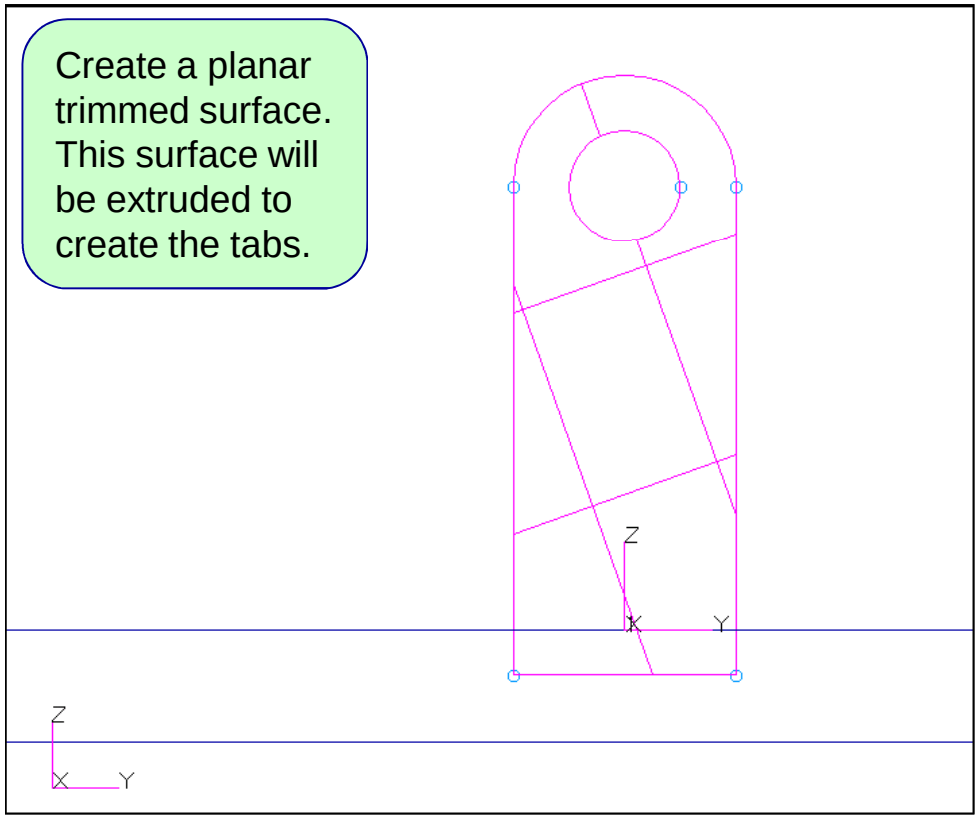
Copyright© 2012 MSC.Software Corporation

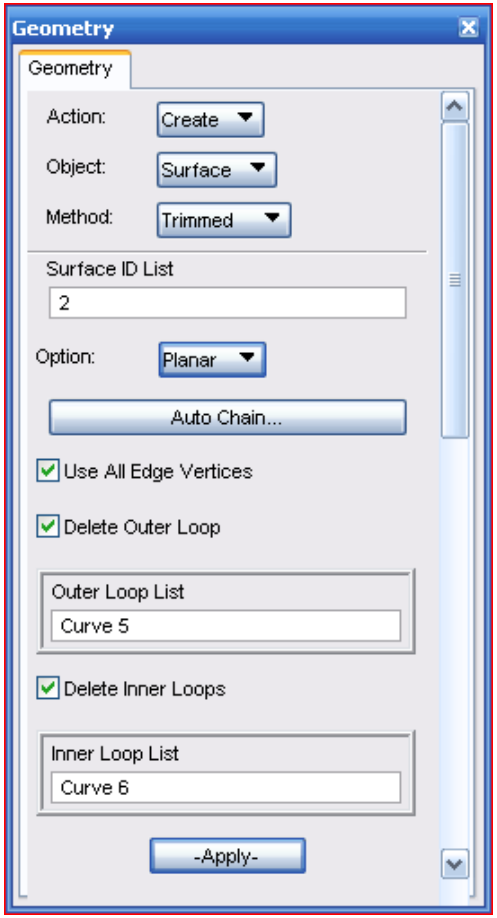
CASE STUDY: LAMP HOUSING





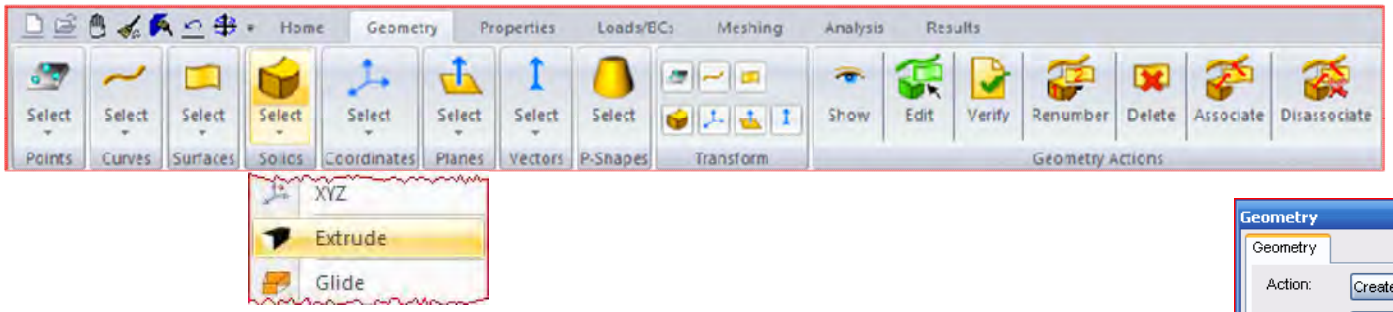
Create a planar trimmed surface. This surface will be extruded to create the tabs.



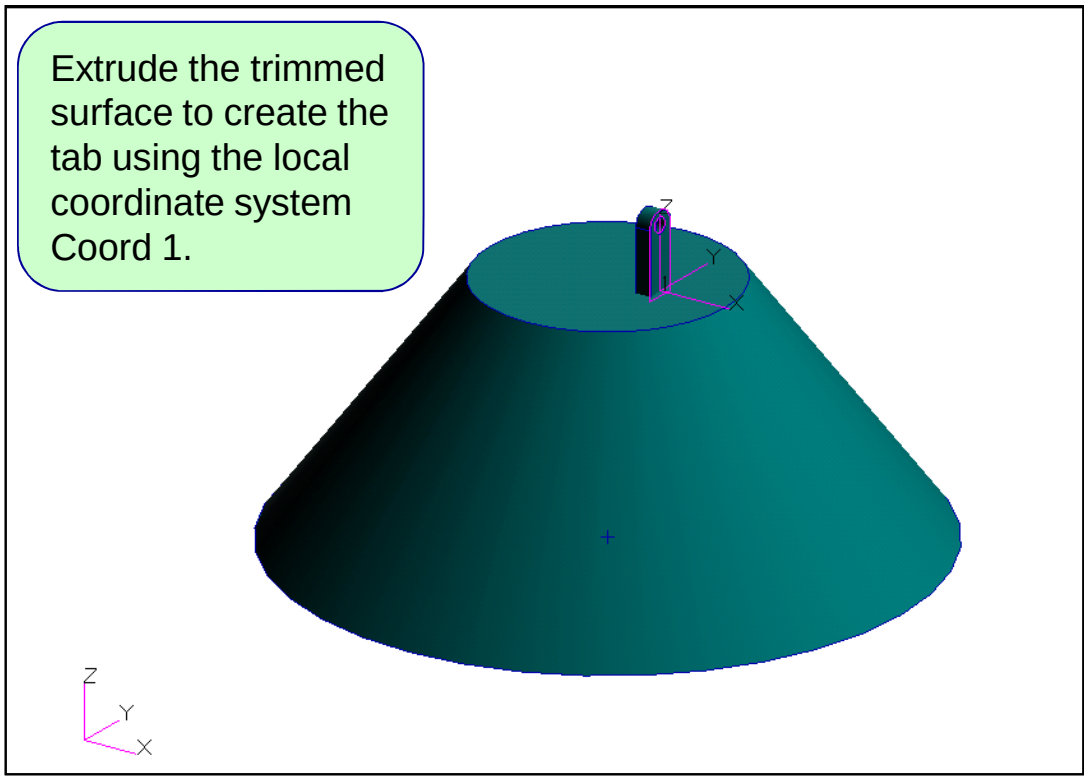


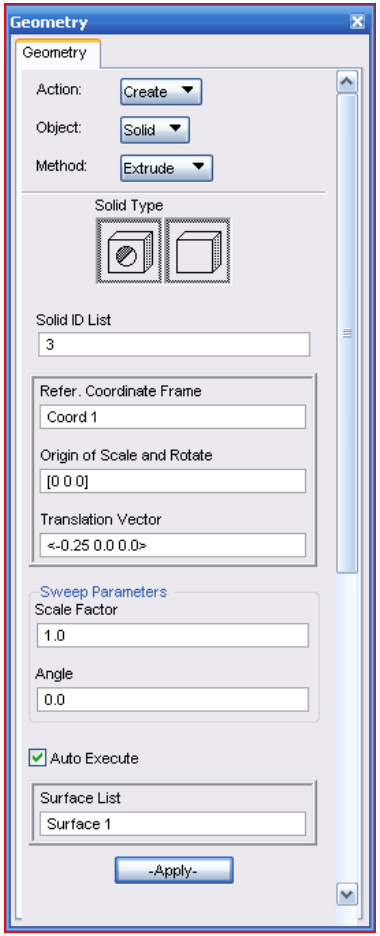
Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING



Extrude the trimmed surface to create the tab using the local coordinate system Coord 1.

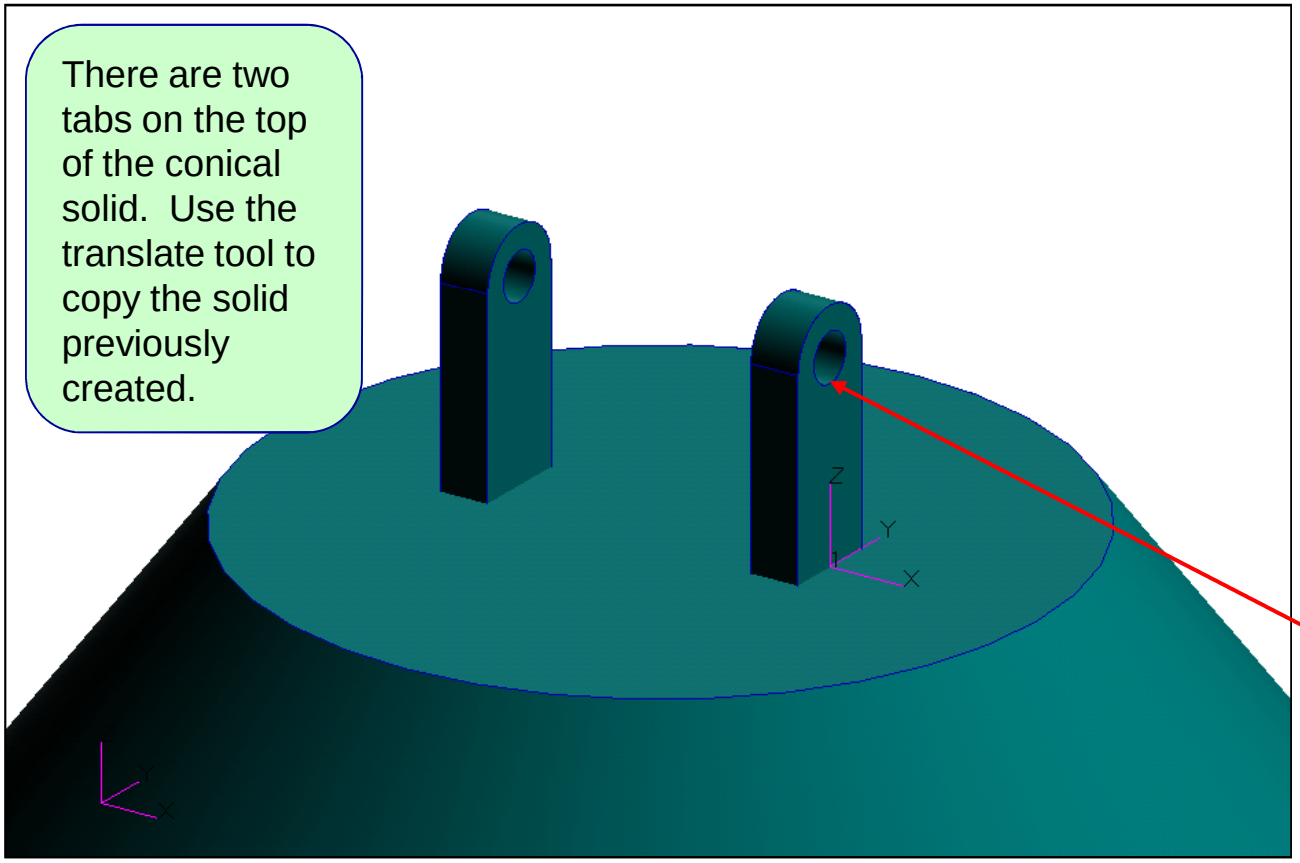


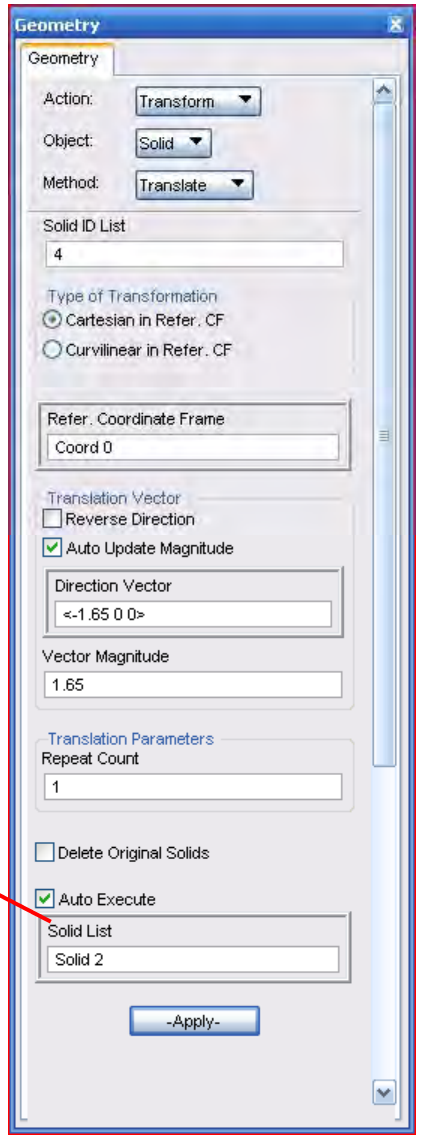


Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING

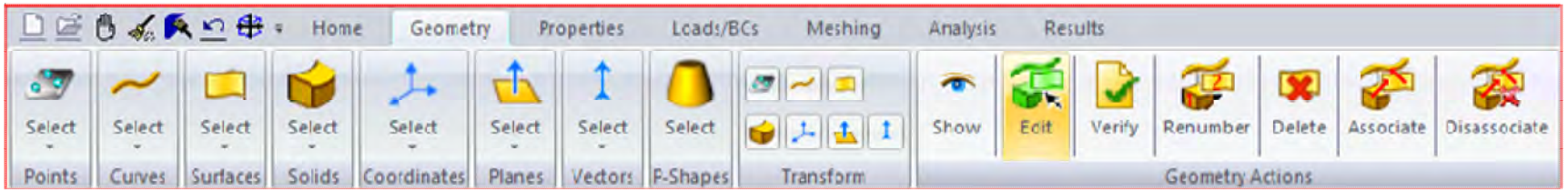
There are two tabs on the top of the conical solid. Use the translate tool to copy the solid previously created.





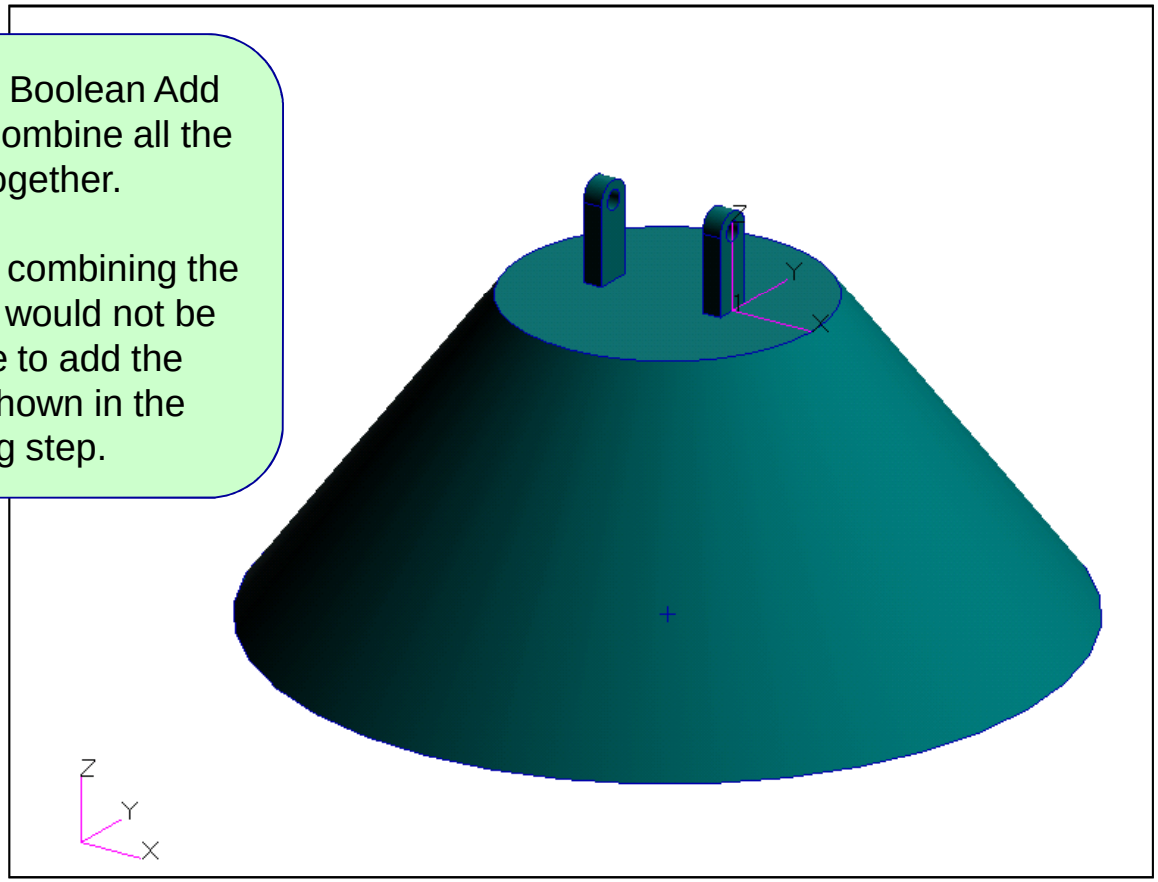
Copyright© 2012 MSC.Software Corporation

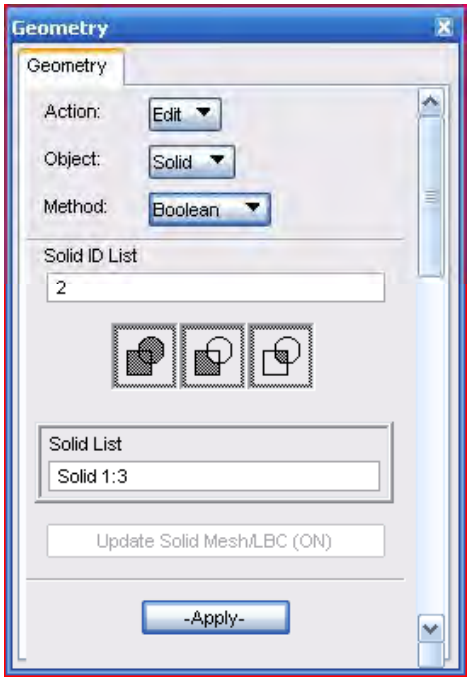
CASE STUDY: LAMP HOUSING



Use the Boolean Add tool to combine all the solids together.

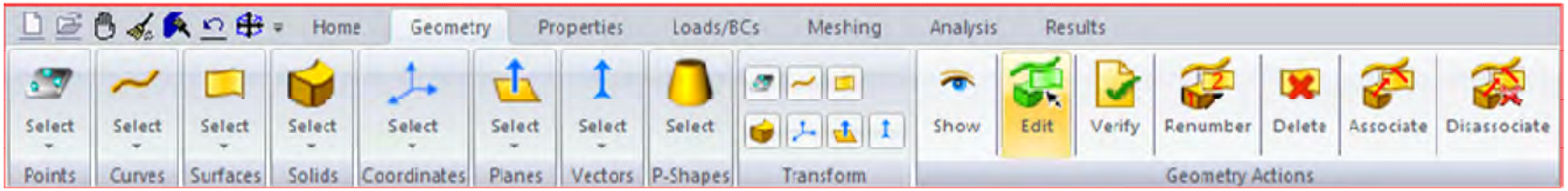
Without combining the solids it would not be possible to add the fillets, shown in the following step.

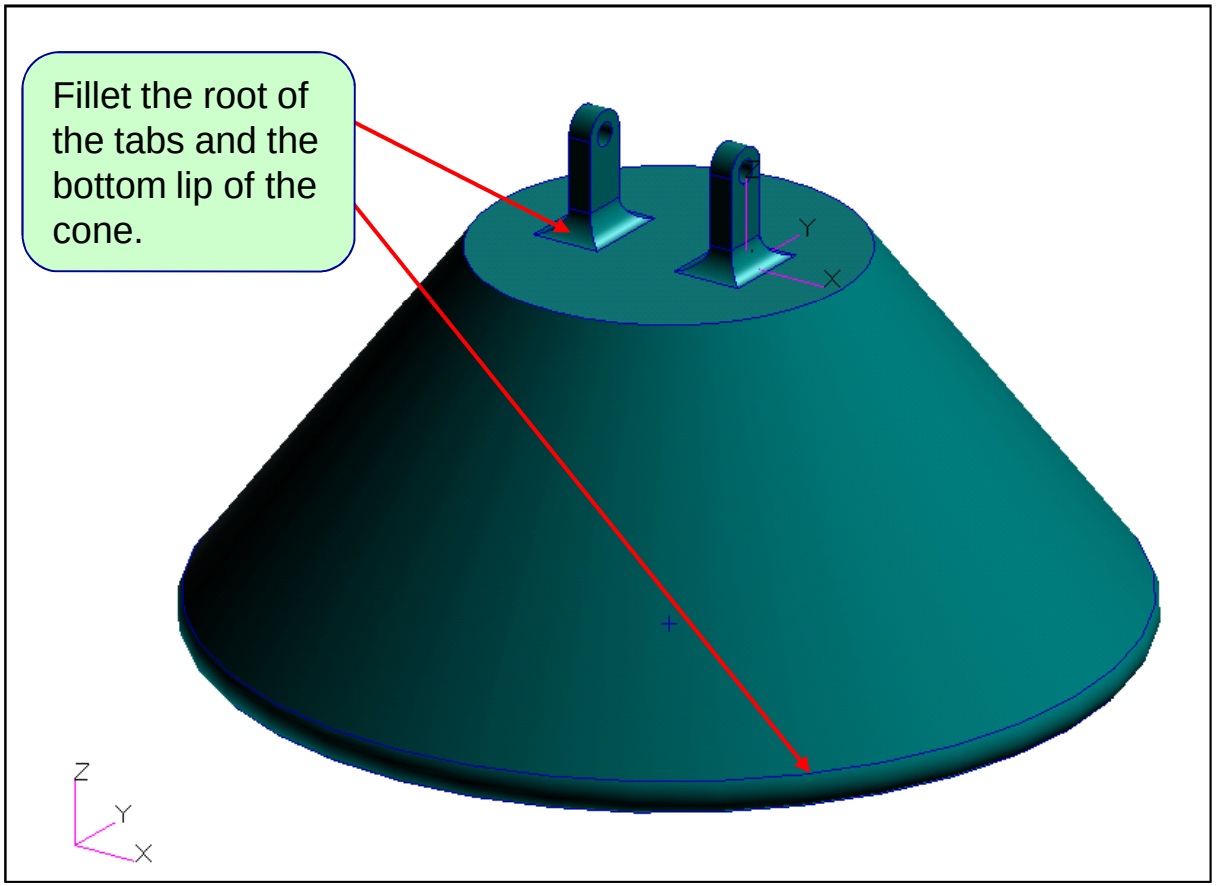


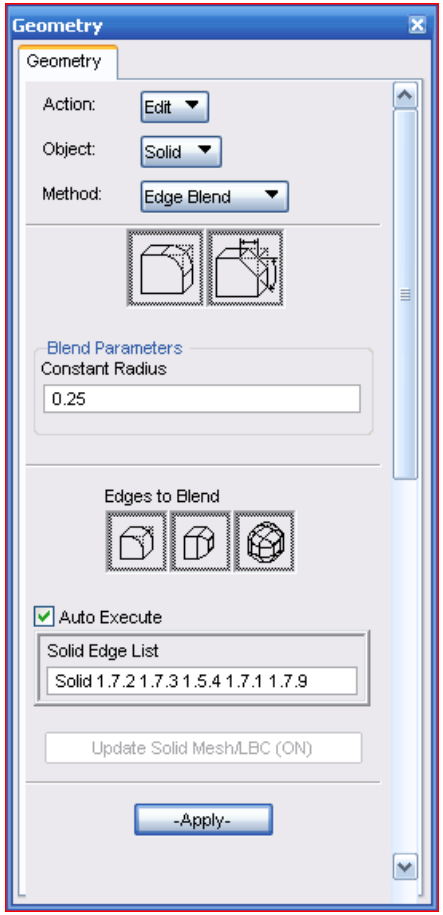


Copyright© 2012 MSC.Software Corporation

CASE STUDY: LAMP HOUSING







Copyright© 2012 MSC.Software Corporation

EXERCISES

- Perform Workshop 12 “Assembly Meshing, Tension Fitting” in your exercise workbook.
- Perform Workshop 13 “Solid Modeling of K-T Joint” in your exercise workbook.
- Perform Workshop 14 “Solid Modeling of Plastic Housing” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

PARASOLID SOLID ASSEMBLY MESHING TOOLS

- **The parasolid solid assembly mesher is capable of meshing neighboring solids in an assembly with different mesh densities while matching the meshes across the solids.**
- **The assembly mesher can also mesh an entire assembly in one operation using a single mesh size.**

Start Point 3D	Nothing		Mesh	Geometry		
Method	Extrude, Glide, etc. Methods	B-Rep or Primitive Construction	Mesh Editing	TetMesh	IsoMesh	Geometry breakdown and rebuild methods
End Point 3D	Simple Geometry	Complex Geometry	Mesh	Mesh		Geometry

Copyright© 2012 MSC.Software Corporation

PARASOLID SOLID ASSEMBLY MESHING TOOLS

- **The key concepts in assembly meshing are**
  - Congruent Faces
  - Matched Faces
- **These key concepts will be demonstrated in the next few examples**

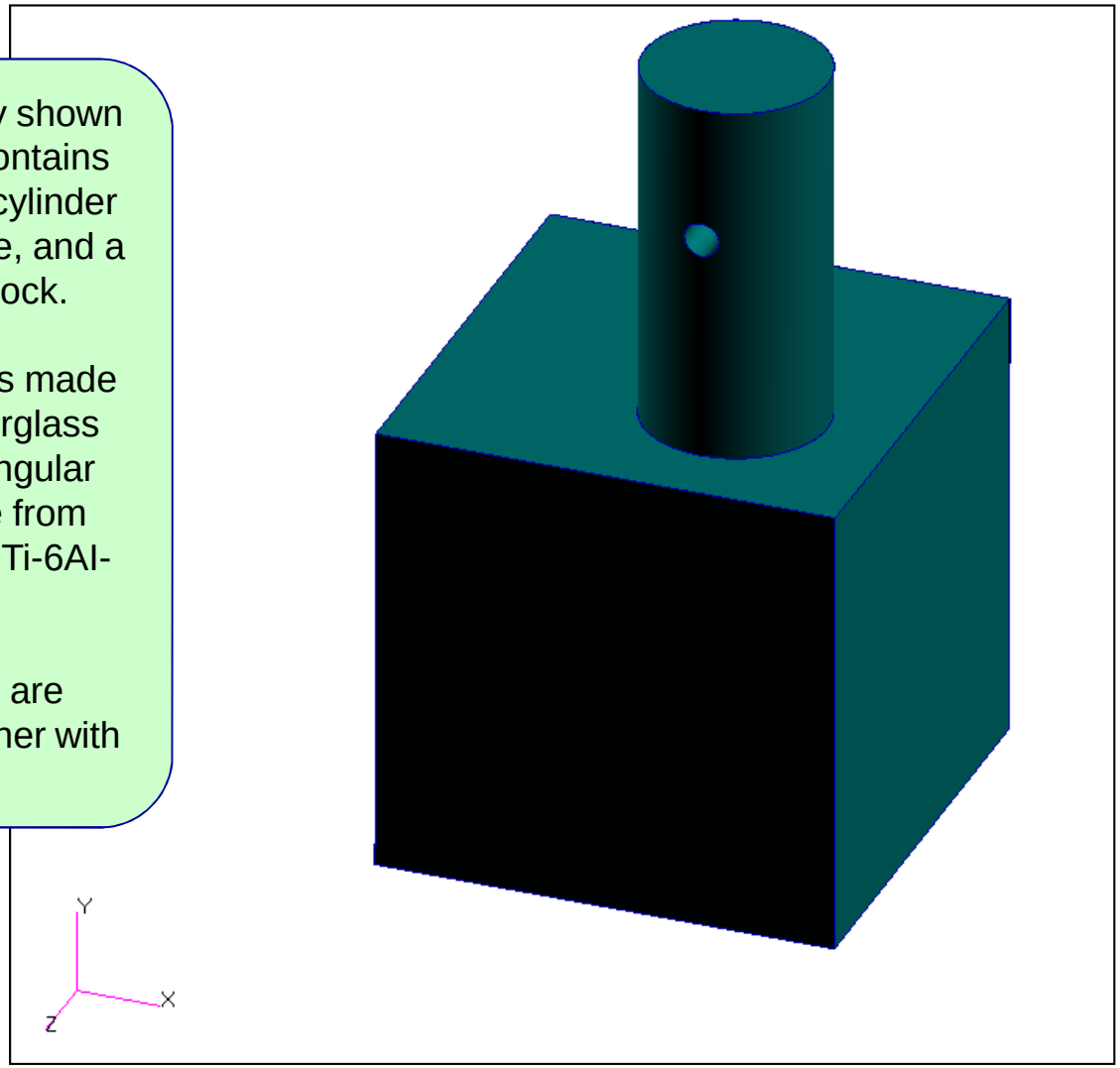
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1

The assembly shown on the right contains two solids: a cylinder with a pin hole, and a rectangular block.

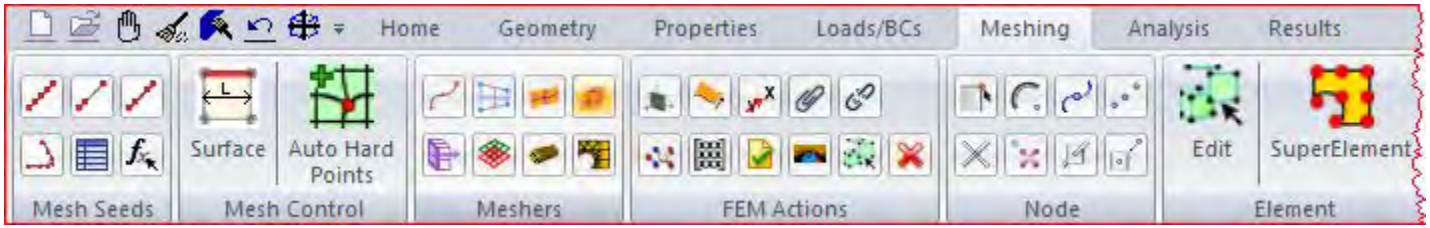
The cylinder is made from G10 fiberglass and the rectangular block is made from titanium alloy Ti-6Al-4V.

The two parts are bonded together with an adhesive.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1



TetMesh the rectangular block:

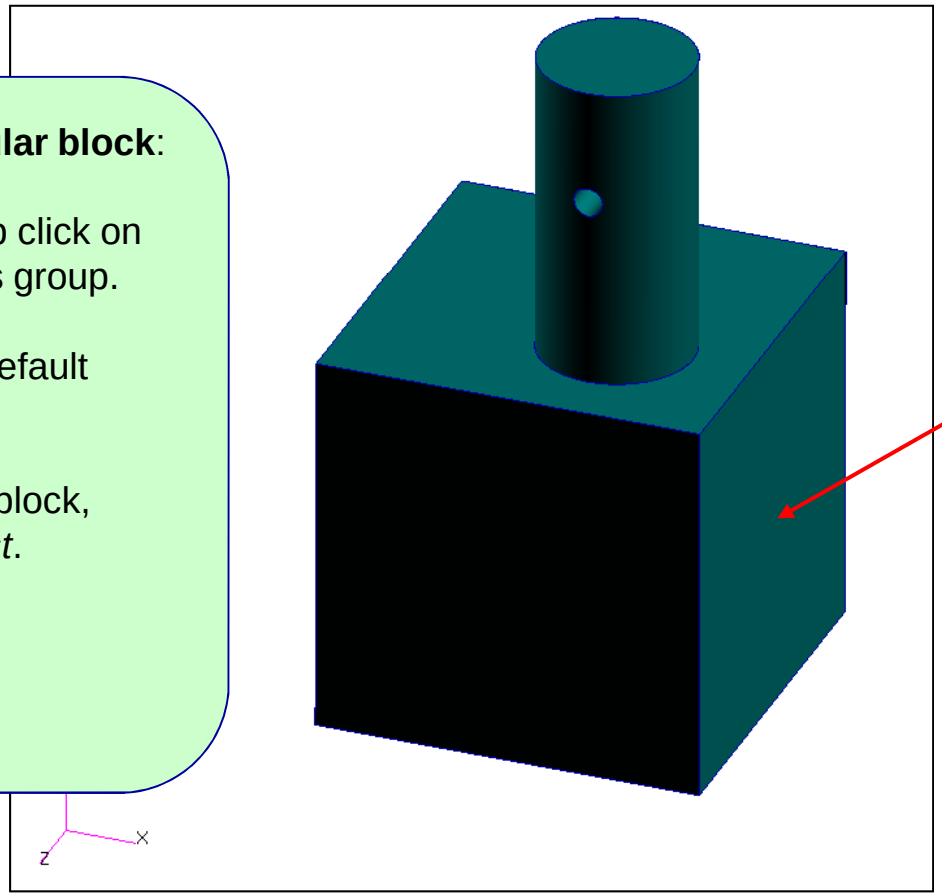
Under the Meshing tab click on Solid from the Meshers group.

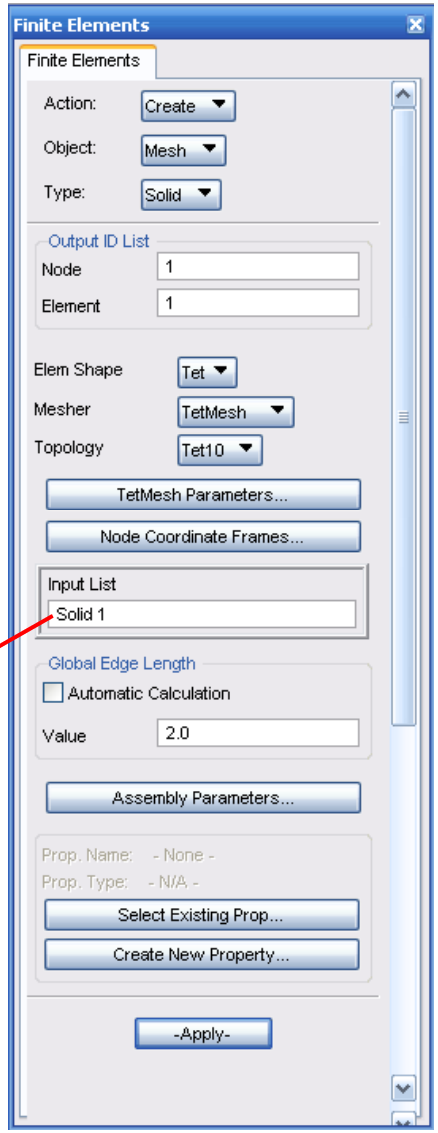
Use **Tet10** elements (default setting).

Select the rectangular block, Solid 1, under *Input List*.

Use a *GEL* of **2.0**.

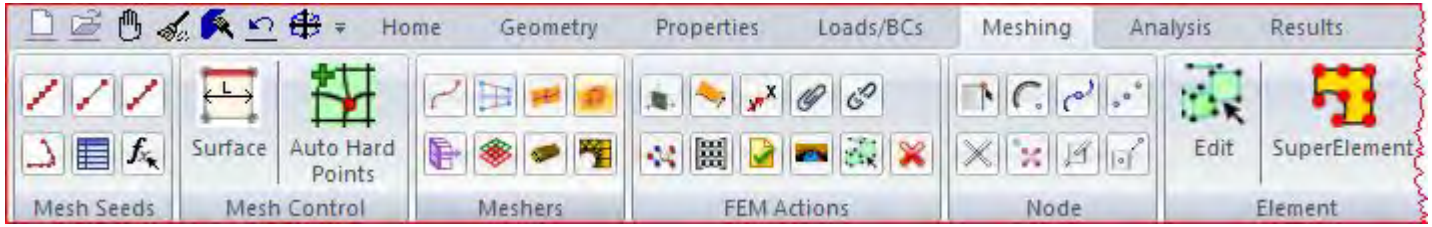
Click **Apply**.



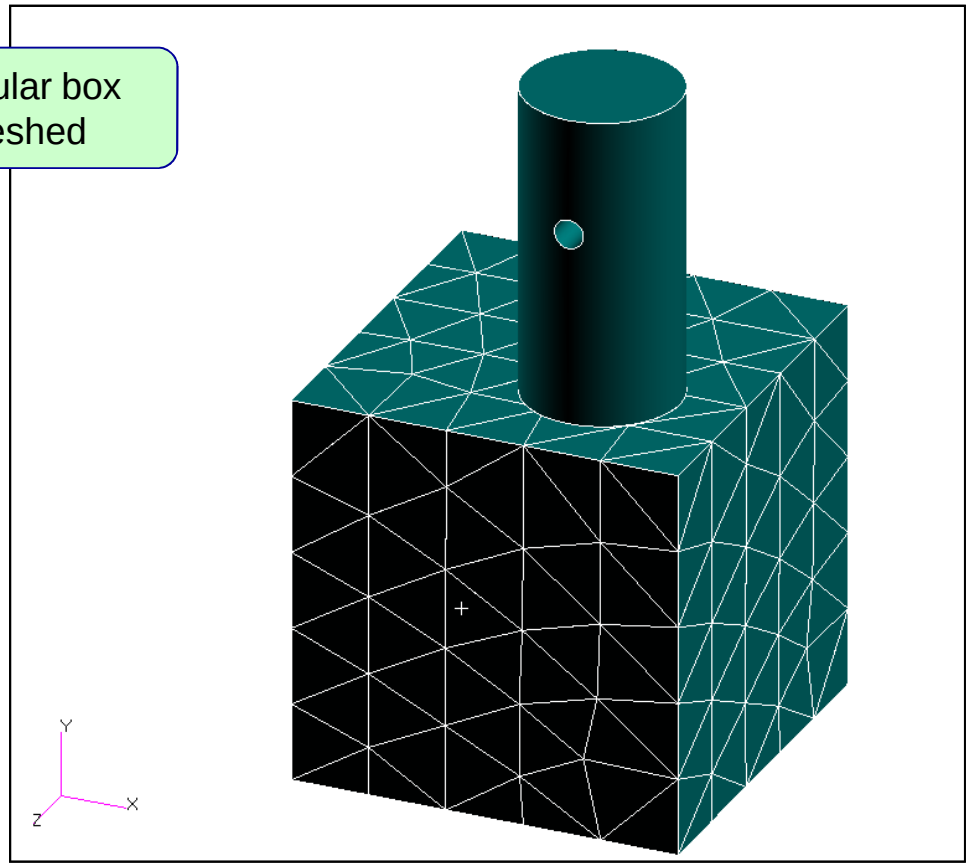


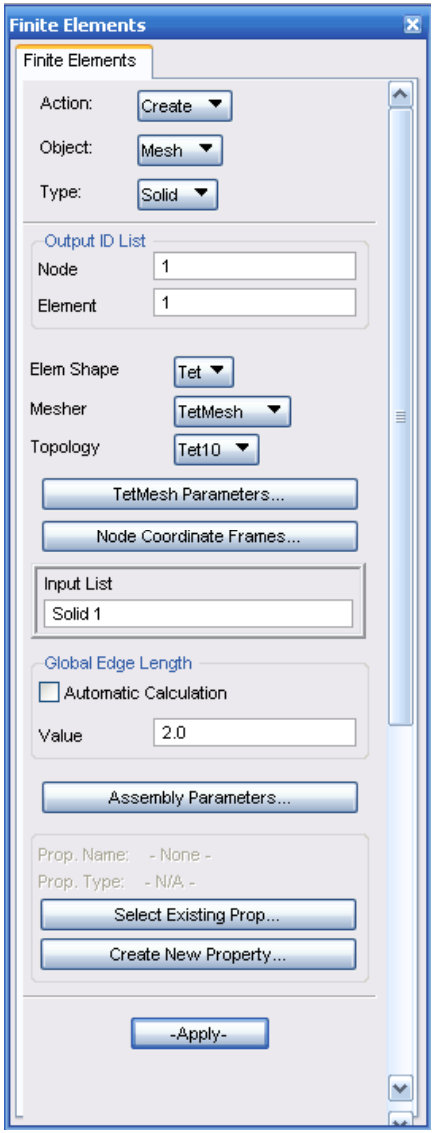
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1



The rectangular box has been meshed

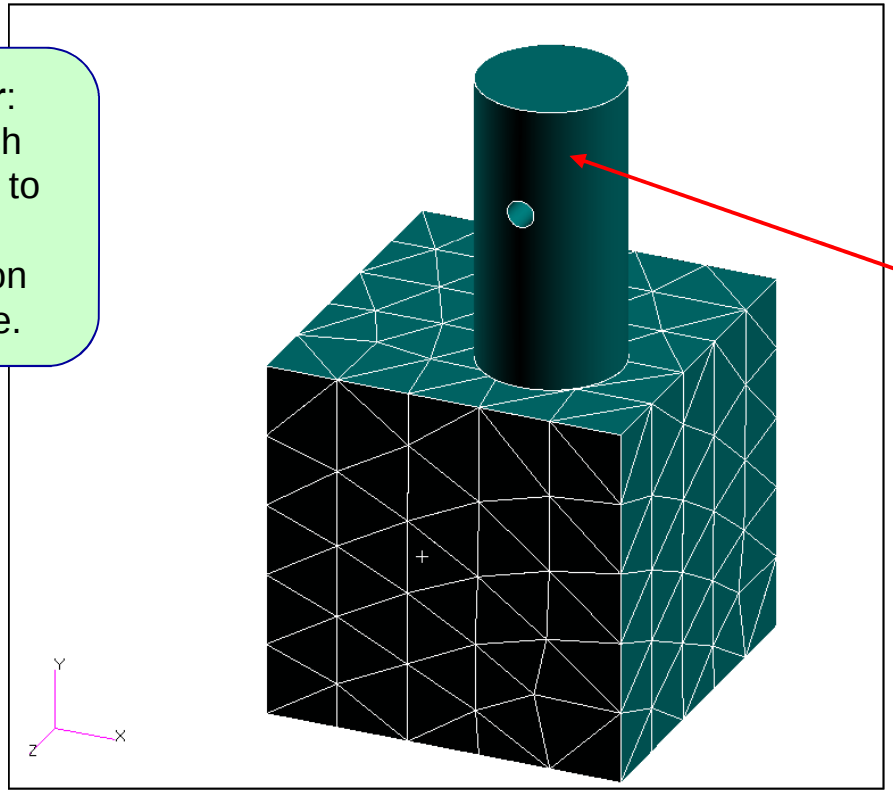


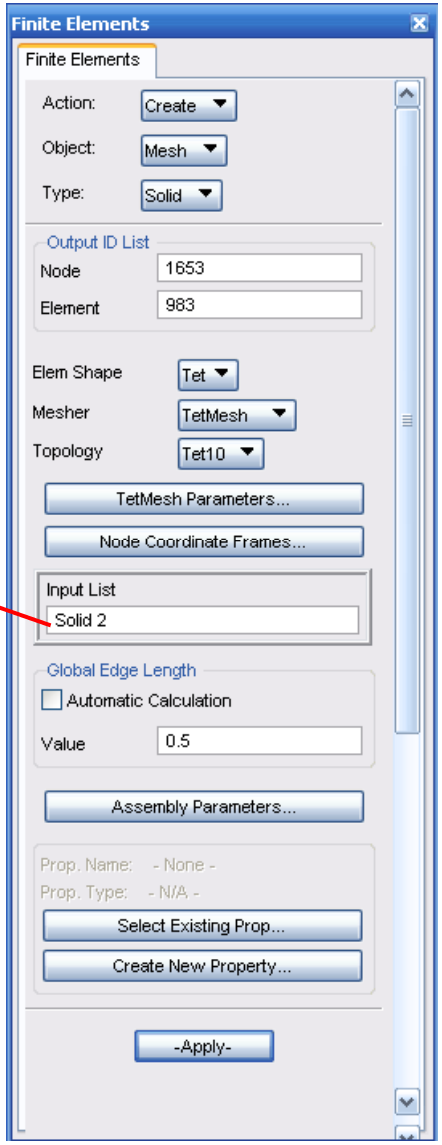


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1

Mesh the cylinder: Use a smaller mesh size of 0.5 in order to better capture the stress concentration around the pin hole.



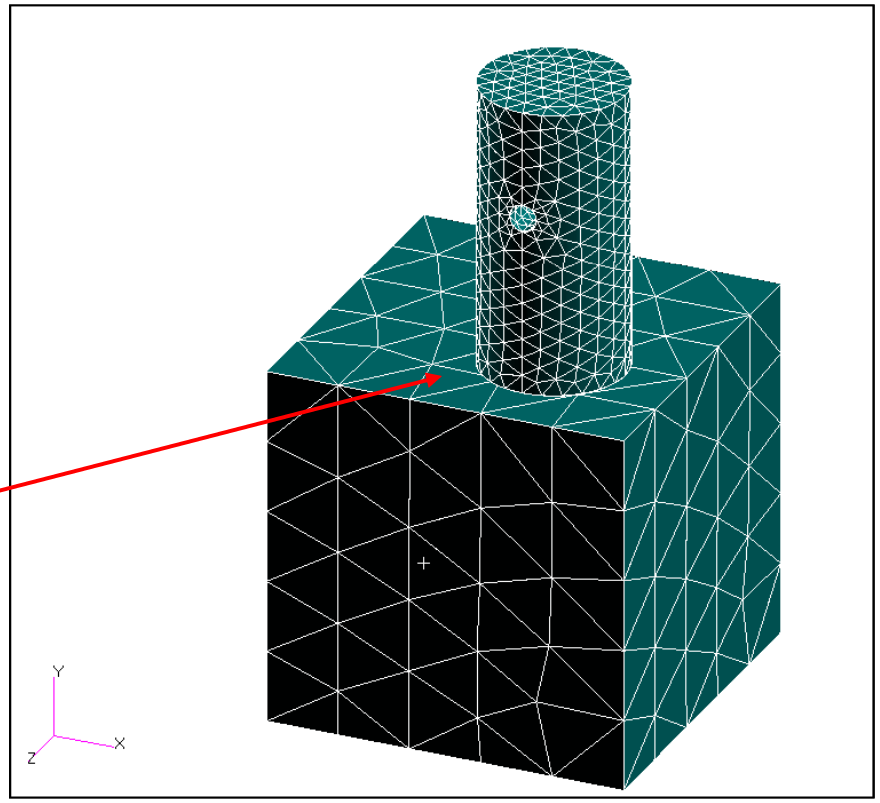


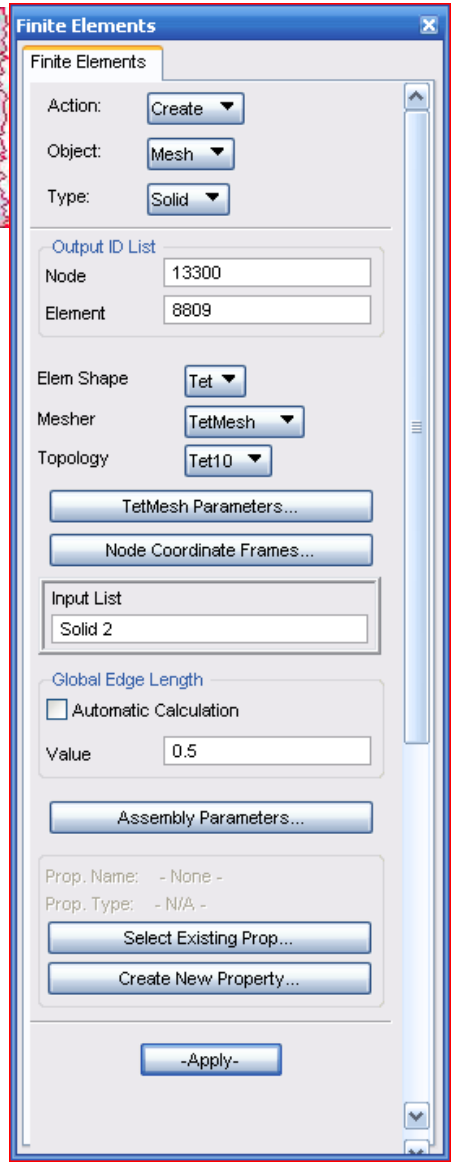
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1

The cylinder has now been meshed.

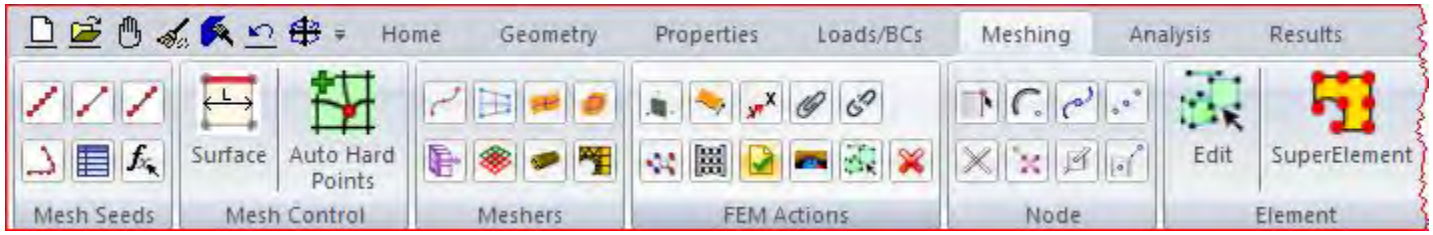
A visual inspection of the meshed assembly illustrates that the mesh in the cylinder does not match the mesh in the rectangular block at the interface of the two solids.





Copyright© 2012 MSC.Software Corporation

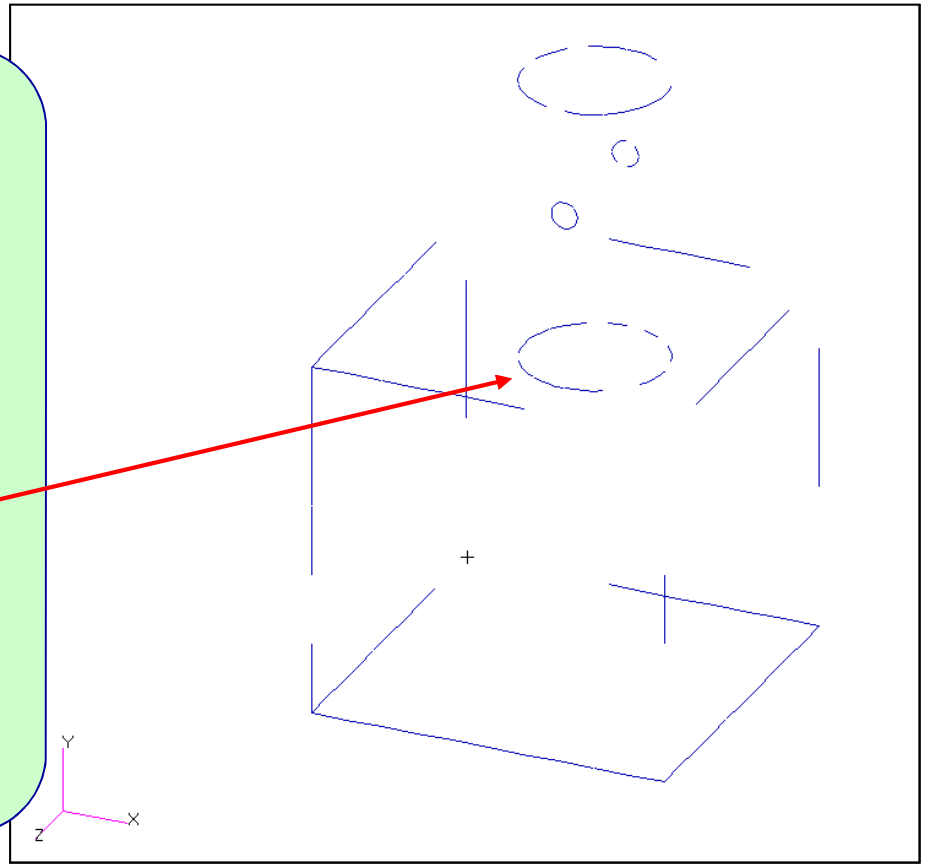
CASE STUDY: ASSEMBLY MESH EX. 1

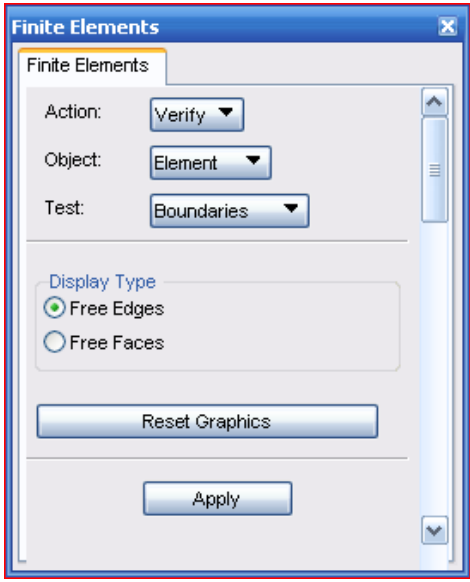


Find the problem areas in this assembly model: Under the Meshing tab click on Verify from the FEM Actions group.

Set the Display Type to **Free Edges**. The free edges in the model are then displayed.

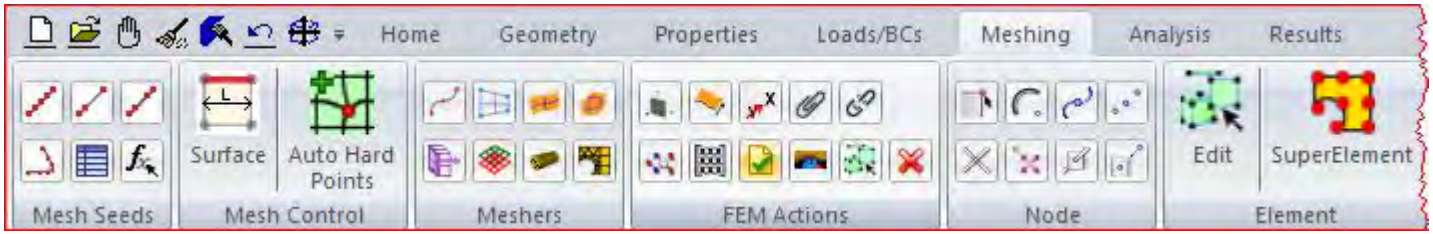
Free edges are shown at the perimeter of the interface area, indicating unconnected elements.





Copyright© 2012 MSC.Software Corporation

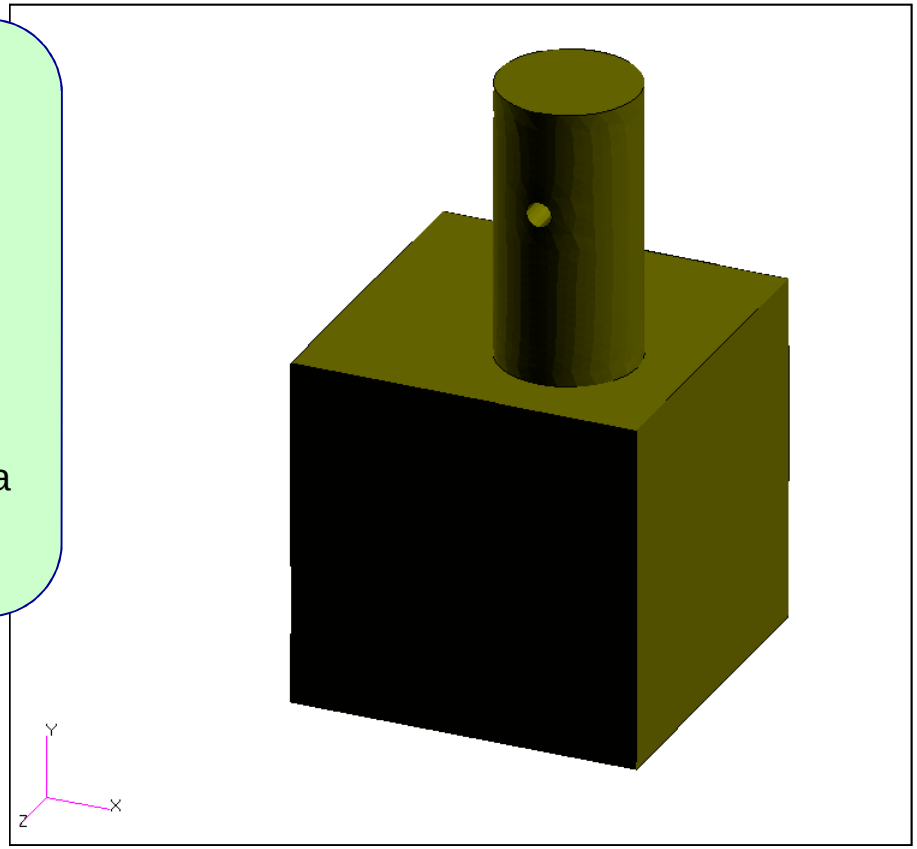
CASE STUDY: ASSEMBLY MESH EX. 1

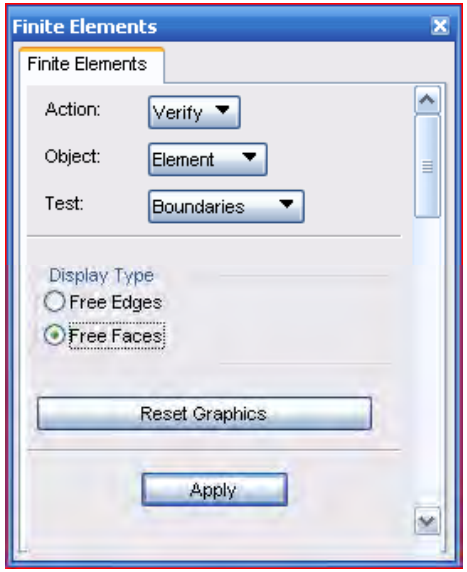


Next, set the display type to **Free Faces**.

All the free faces in the model are now displayed.

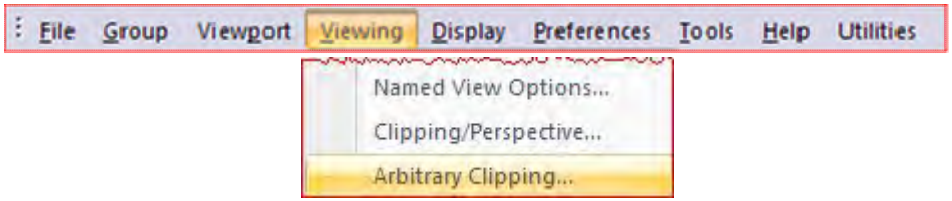
In order to check for internal free faces, the model needs to be sliced open. Use a clipping plane to do this.





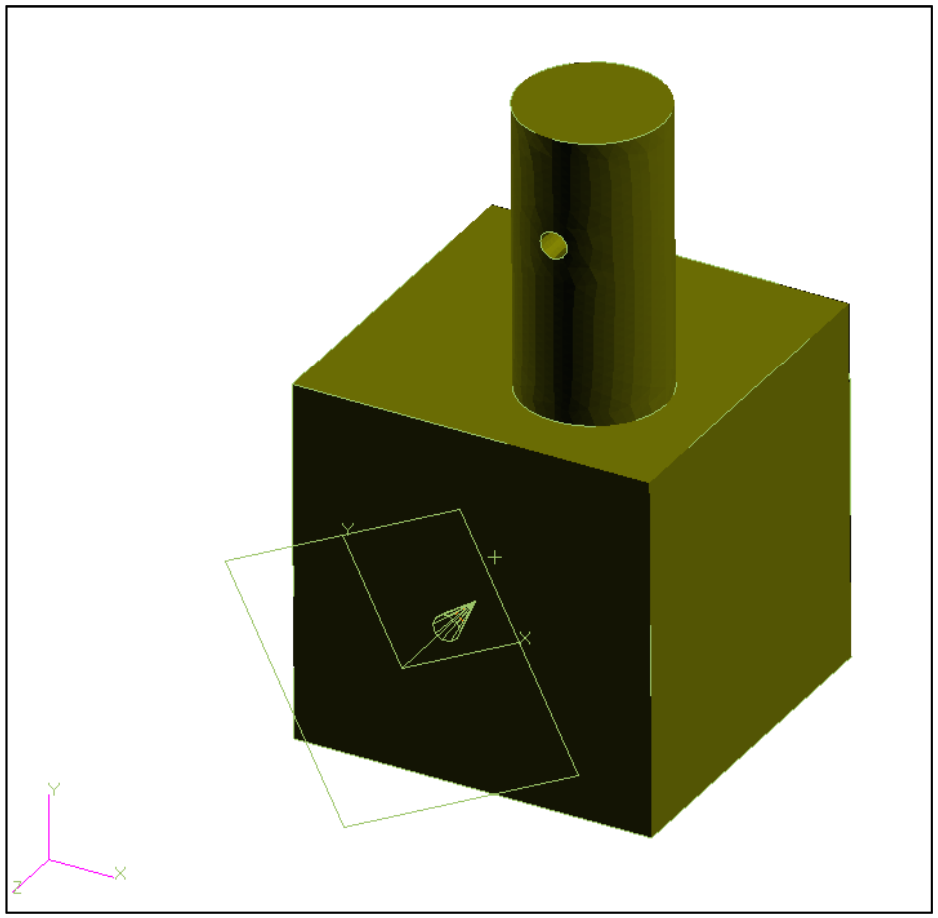
Copyright© 2012 MSC.Software Corporation

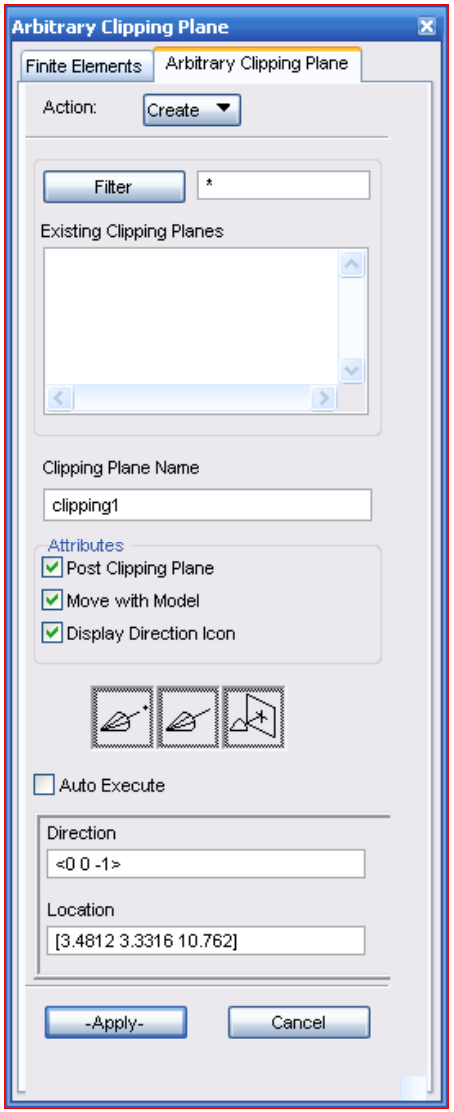
CASE STUDY: ASSEMBLY MESH EX. 1



Create and post a clipping plane, called “clipping1”.

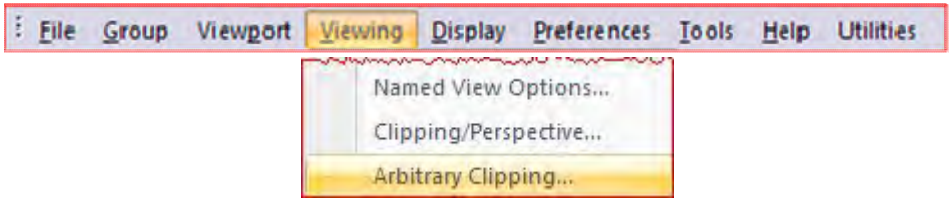
The clipping plane normal is to be in the -Z-direction.





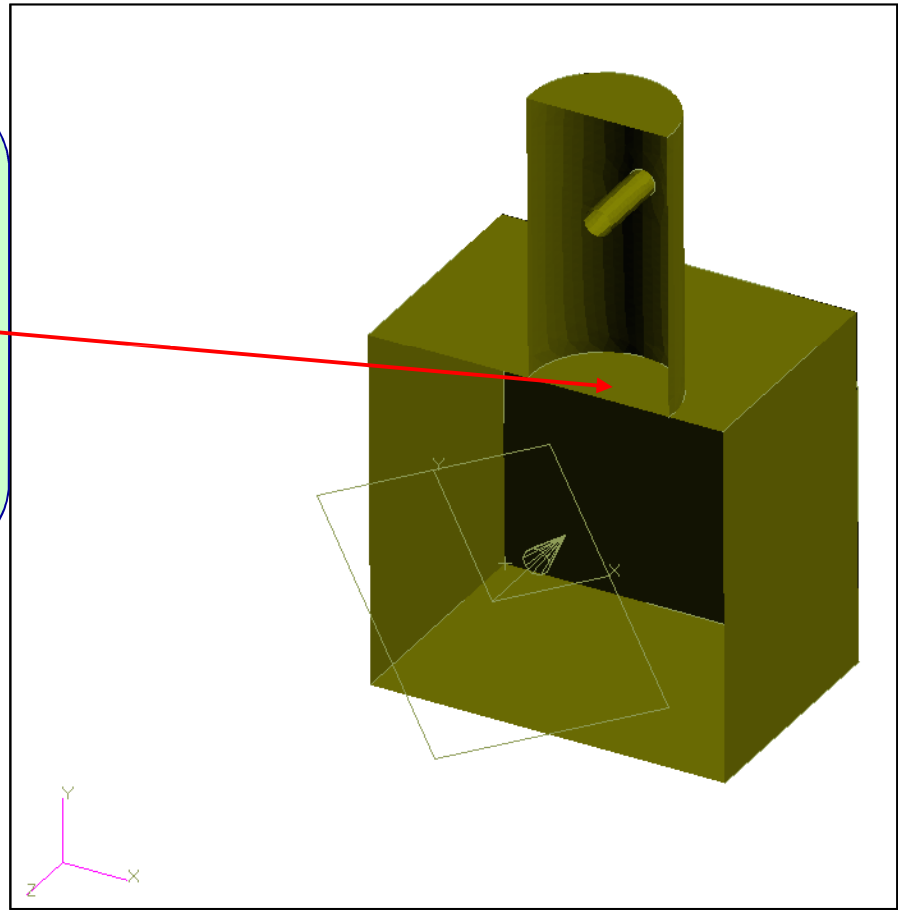
Copyright© 2012 MSC.Software Corporation

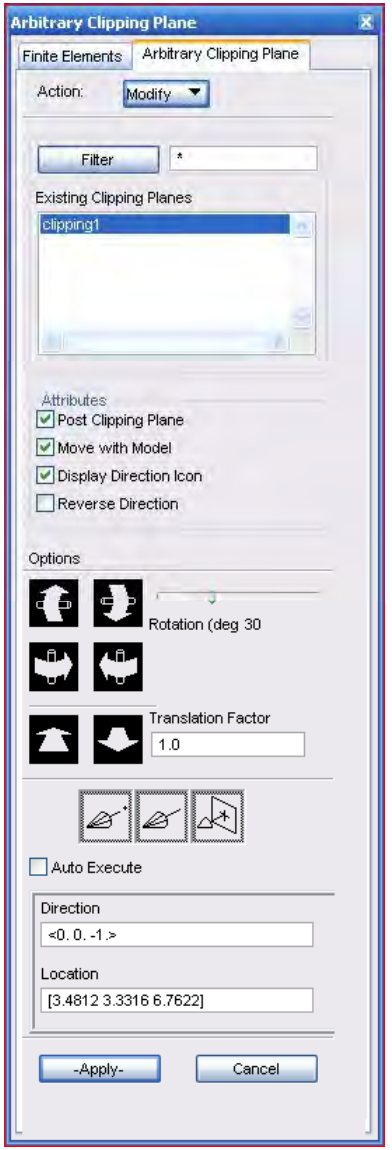
CASE STUDY: ASSEMBLY MESH EX. 1



Move the clipping plane until it cuts through both solids.

Free faces are displayed at the interface between the two solids, indicating that the meshes are not connected.





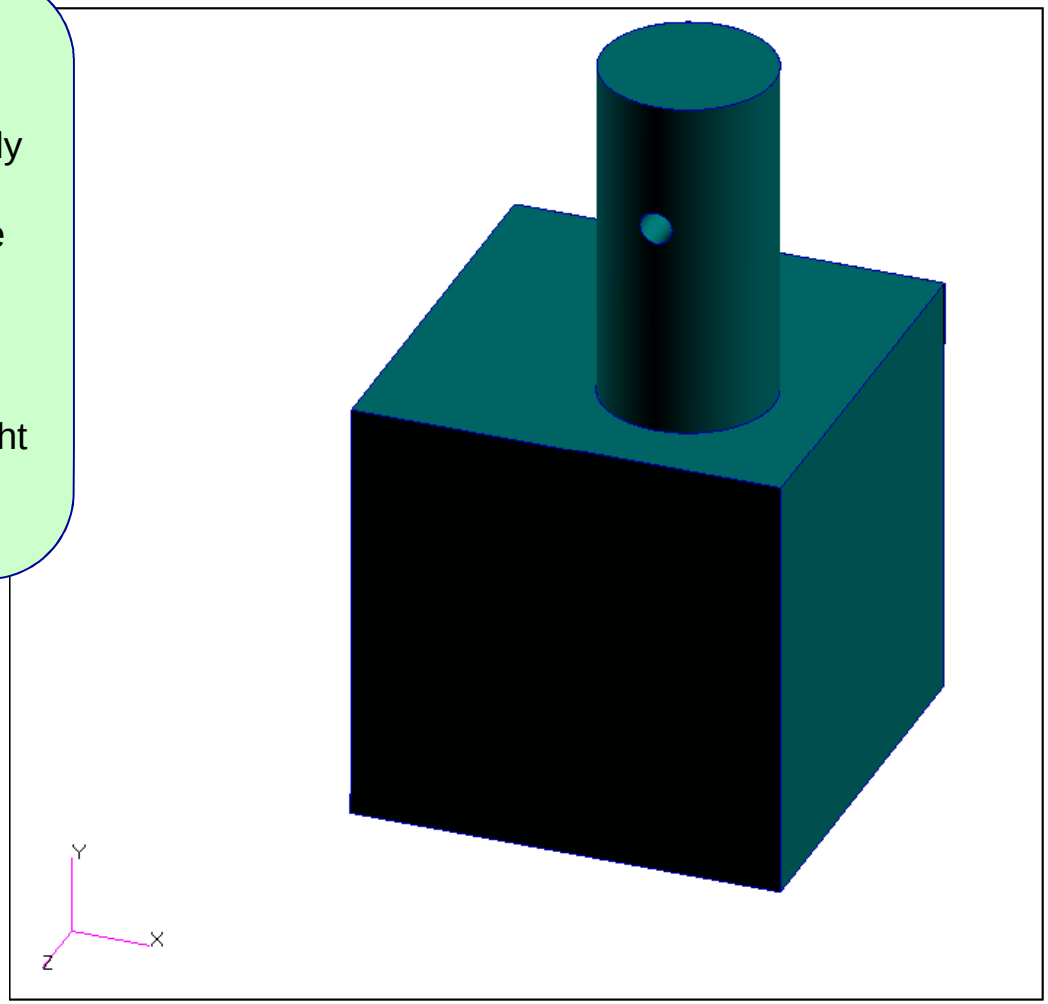
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1

Start again with the two solids.

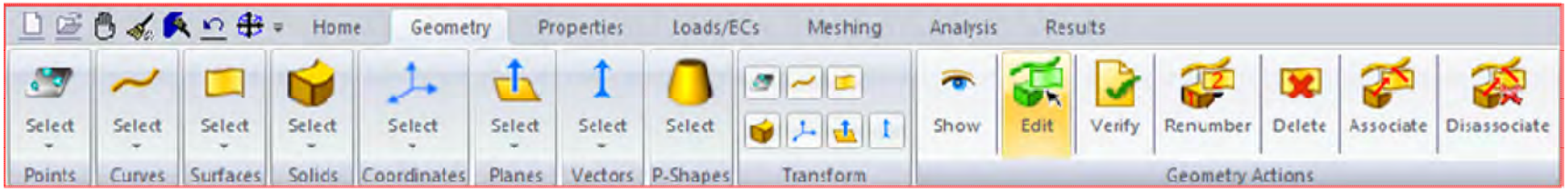
A crucial step in assembly meshing, **one solid at a time**, is to make sure the adjacent solids have **congruent (matching)** faces at their interface.

The two solids on the right do not have congruent faces at the interface.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1

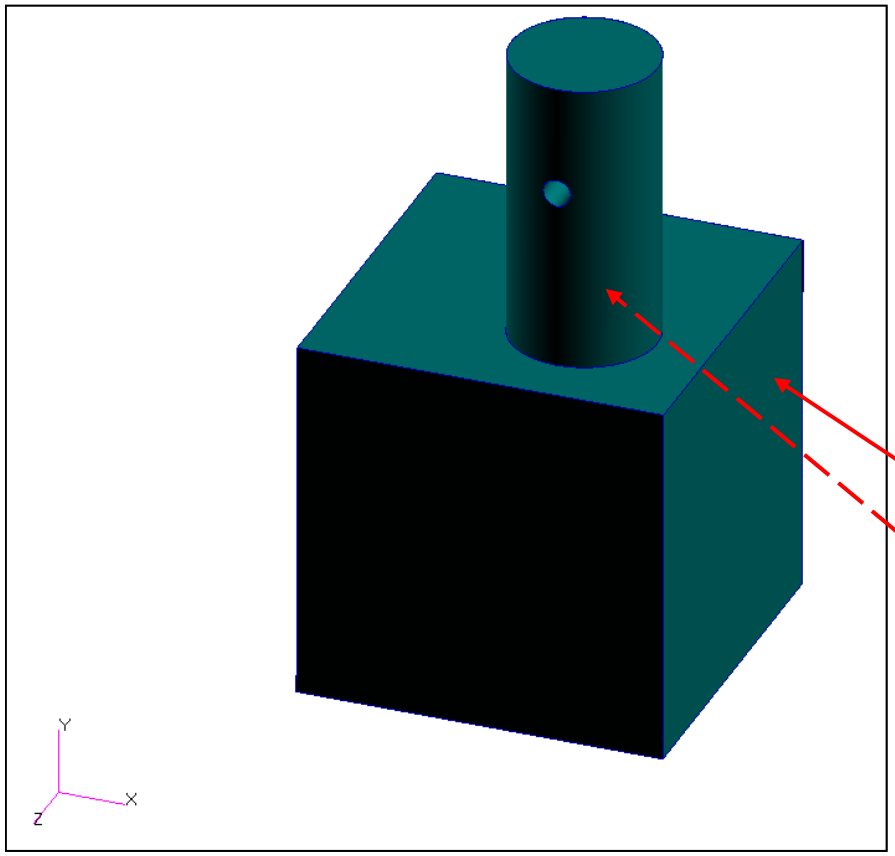


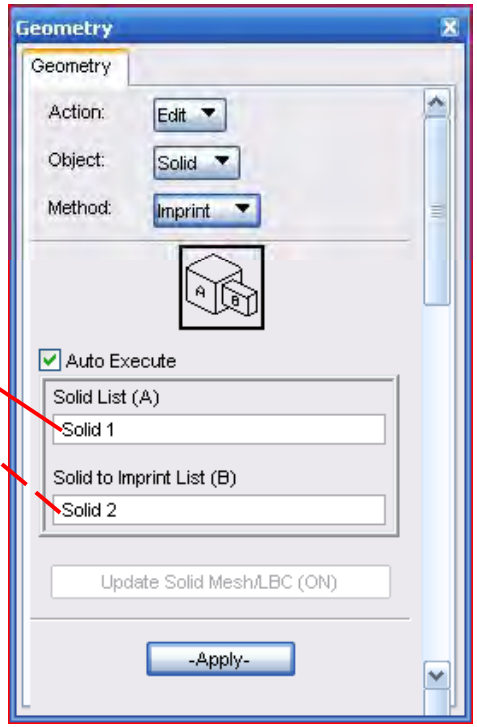
Create congruent faces at the solids interface by using the imprint tool:

Under the Geometry tab: Edit / Solid / Imprint

Imprint the cylinder onto the rectangular block.

Another approach is to disassemble the block into surfaces, edit the surface(s), and create a new block solid.





Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1

First, erase the cylinder.

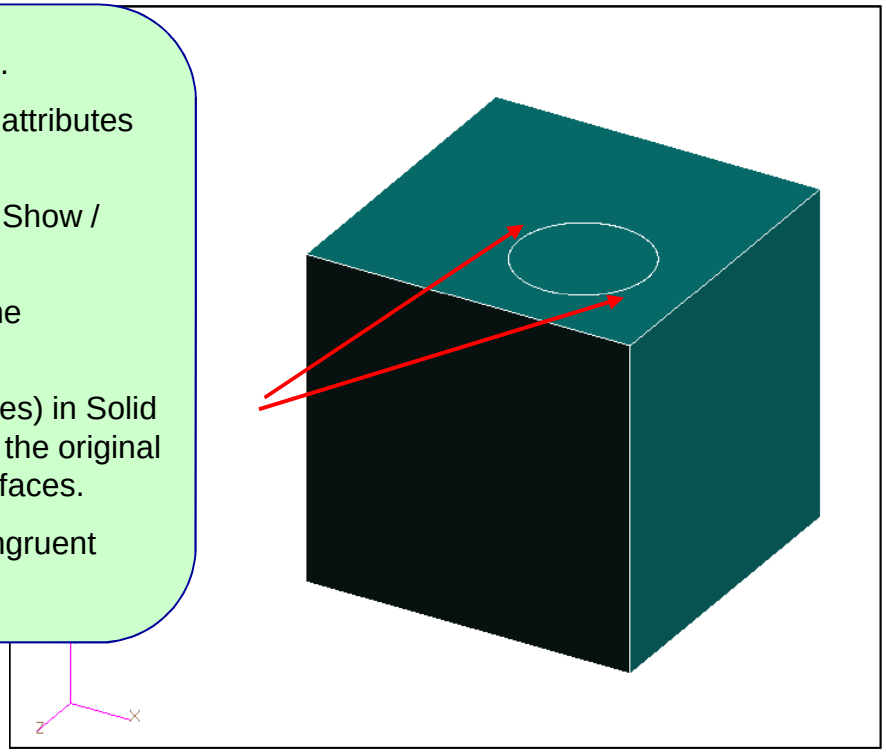
Then determine the solid attributes using

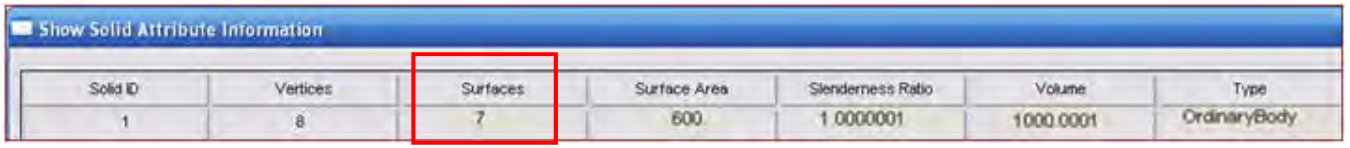
Under the *Geometry* tab: Show / Solid / Attributes

Under *Solid List*, select the rectangular solid, Solid 1.

There are 7 surfaces (faces) in Solid 1. The imprinting divides the original rectangular face into two faces.

Now both solids have congruent faces.



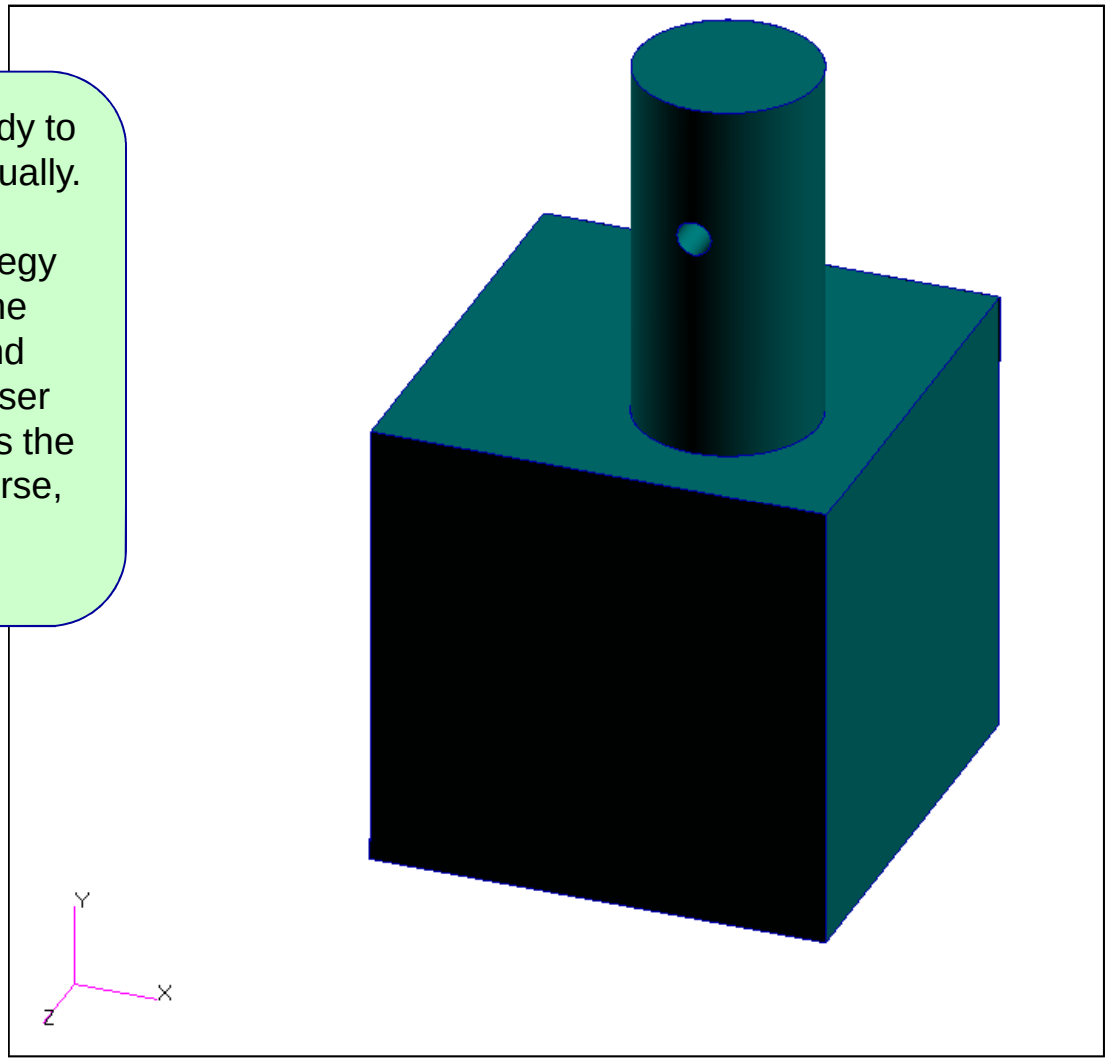


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1

The solids are ready to be meshed individually.

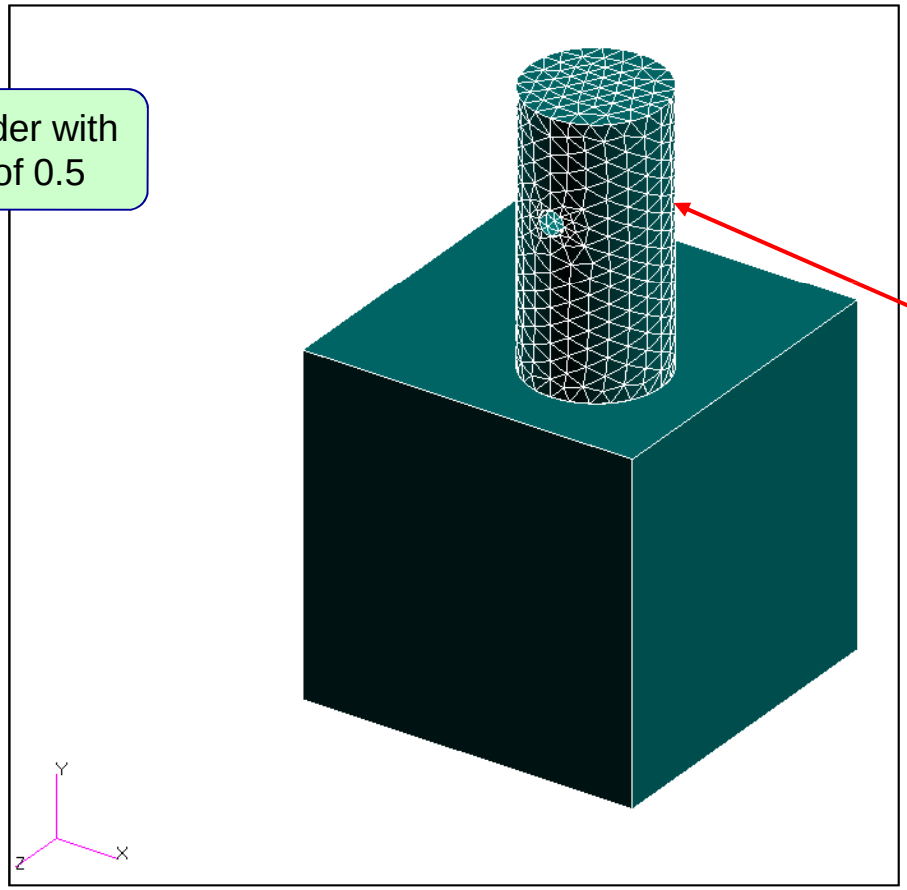
The meshing strategy here is to create the finer mesh first, and then create a coarser mesh that matches the first mesh. Of course, the order does not matter.

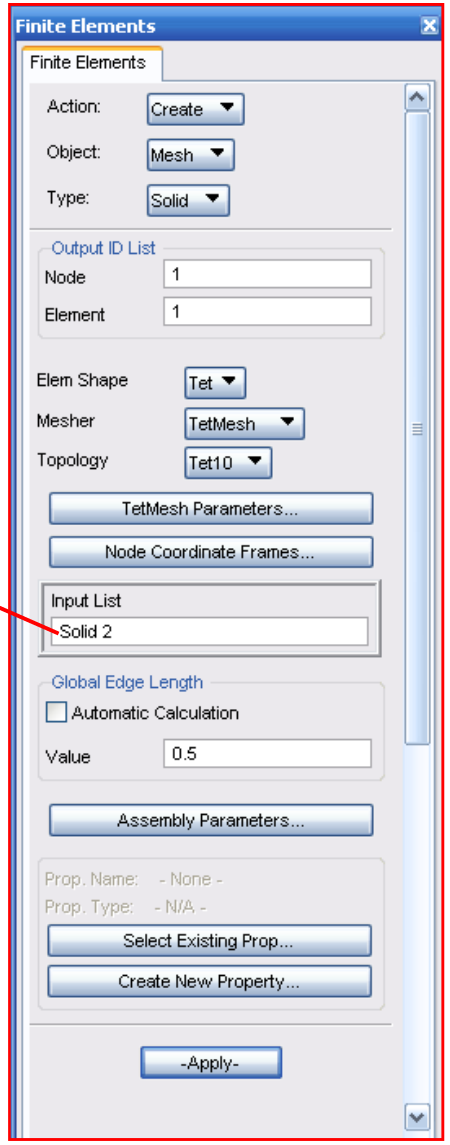


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1

Mesh the cylinder with a mesh size of 0.5





Copyright© 2012 MSC.Software Corporation

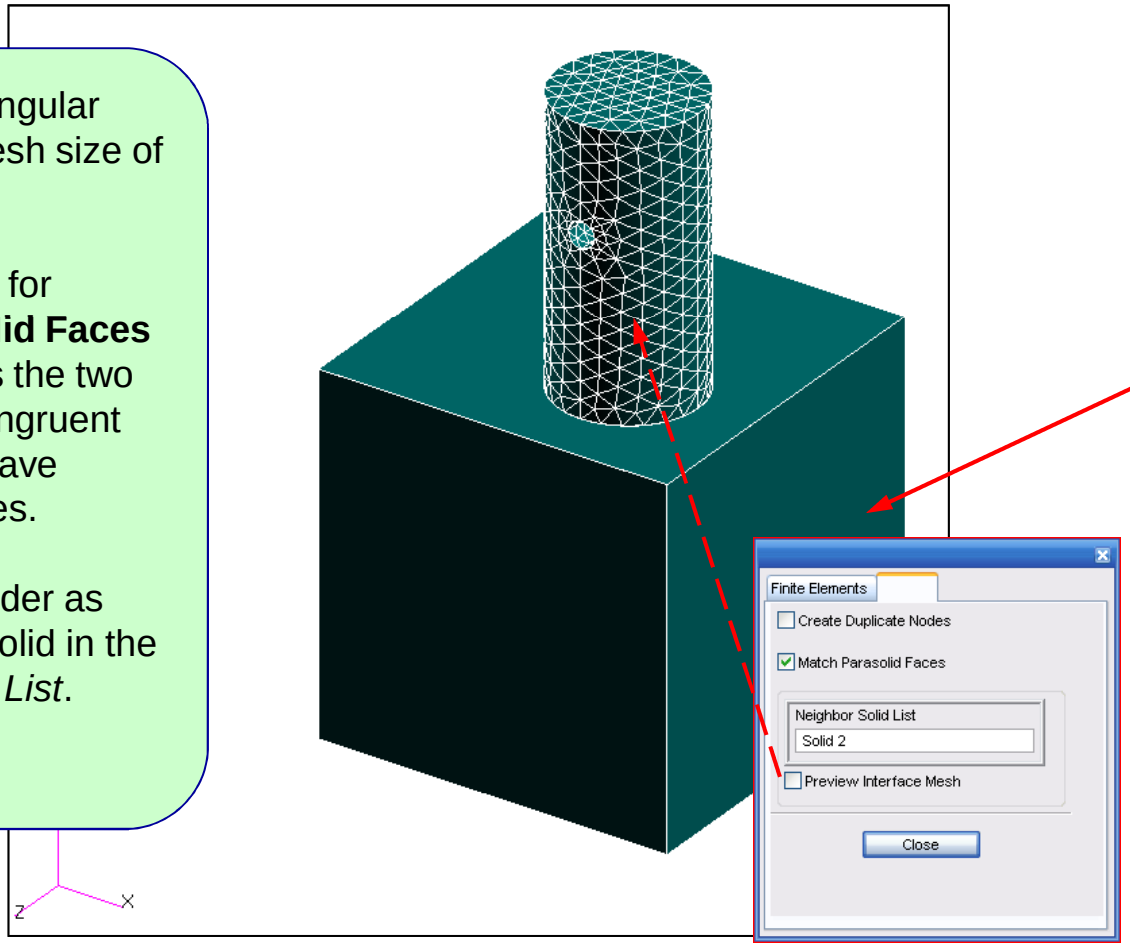
CASE STUDY: ASSEMBLY MESH EX. 1

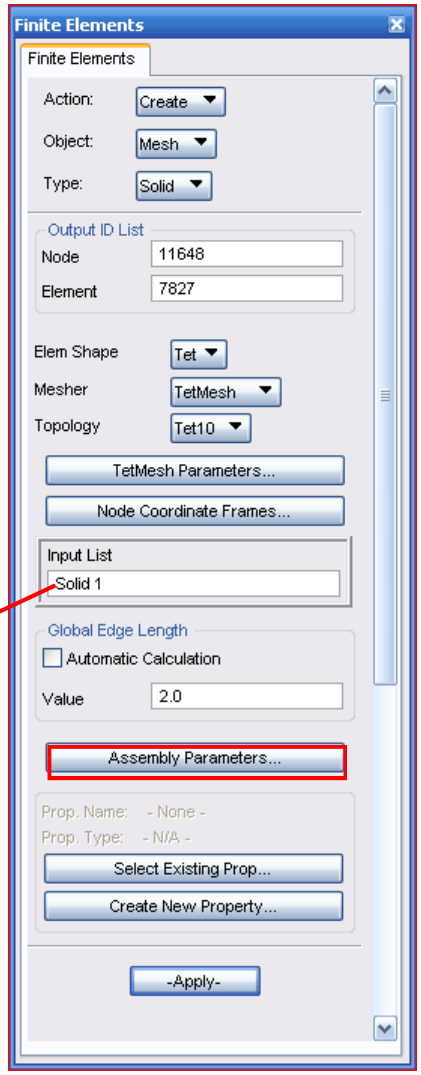
Mesh the rectangular block with a mesh size of 2.0

Turn the toggle for **Match Parasolid Faces** on. This forces the two neighboring congruent solid faces to have identical meshes.

Select the cylinder as the (meshed) solid in the *Neighbor Solid List*.

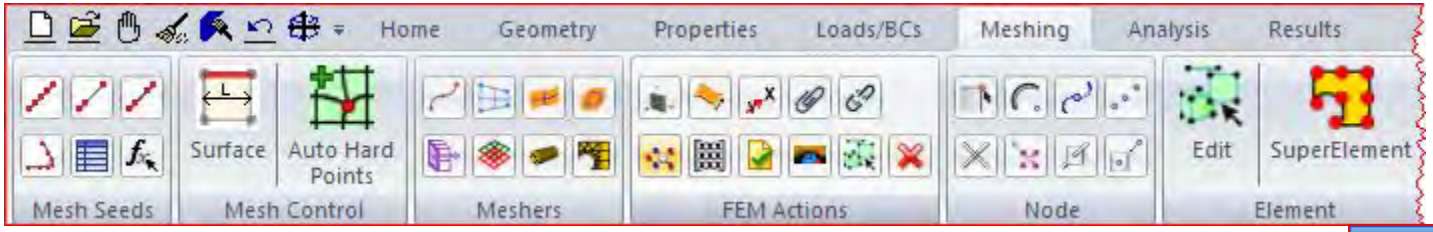
Click **Apply**.





Copyright© 2012 MSC.Software Corporation

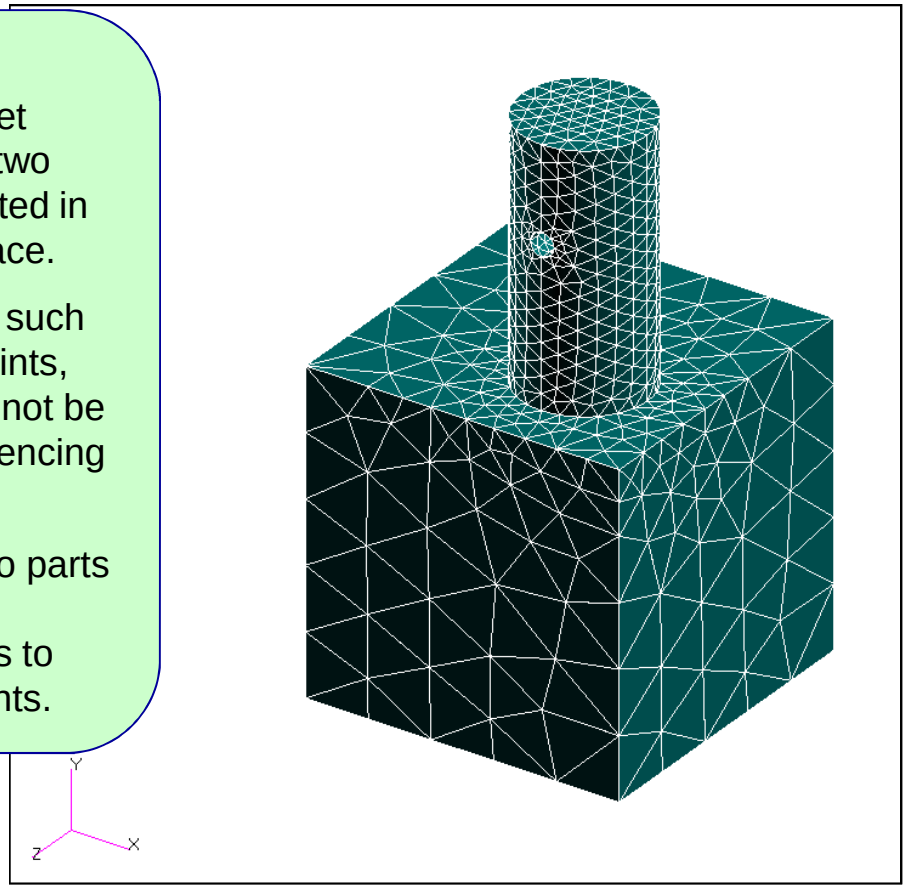
CASE STUDY: ASSEMBLY MESH EX. 1

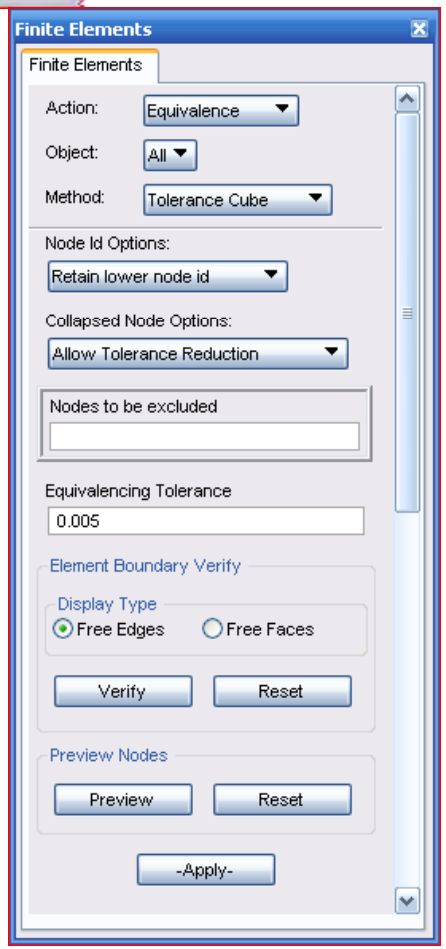


Both solids have been meshed. Coincident tet element faces for the two solids have been created in the plane of the interface.

In modeling situations such as contact or bolted joints, the tet elements must not be connected, so equivalencing is not done.

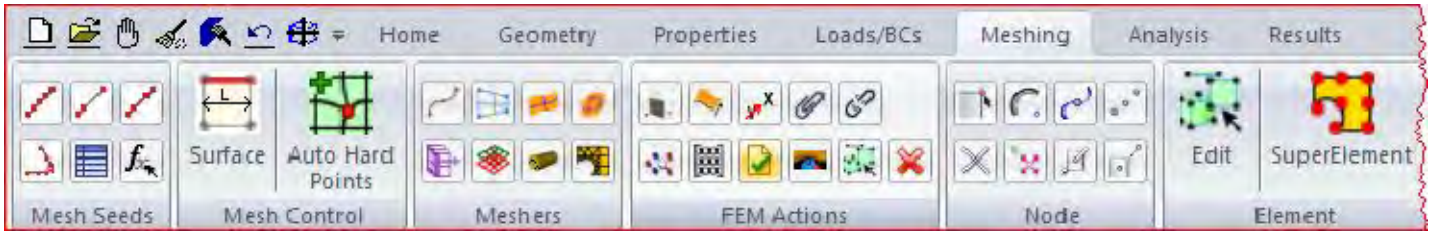
In this example the two parts are bonded together. Equivalence the nodes to connect the tet elements.





Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1

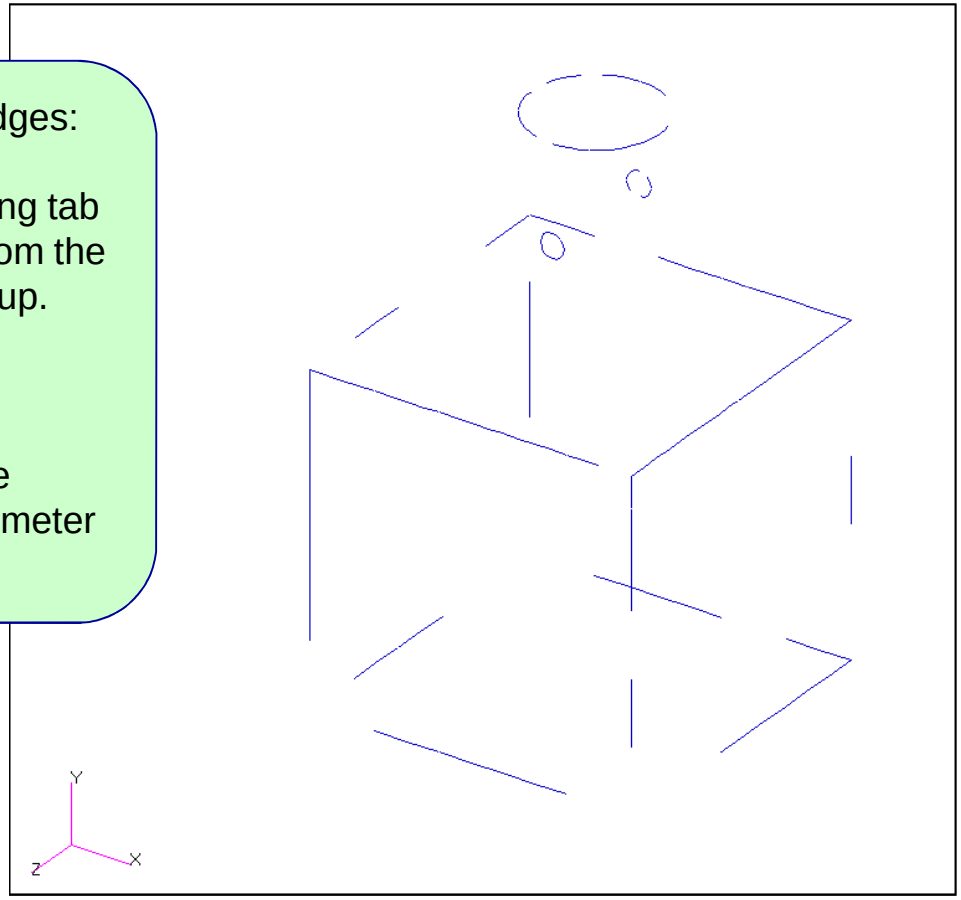


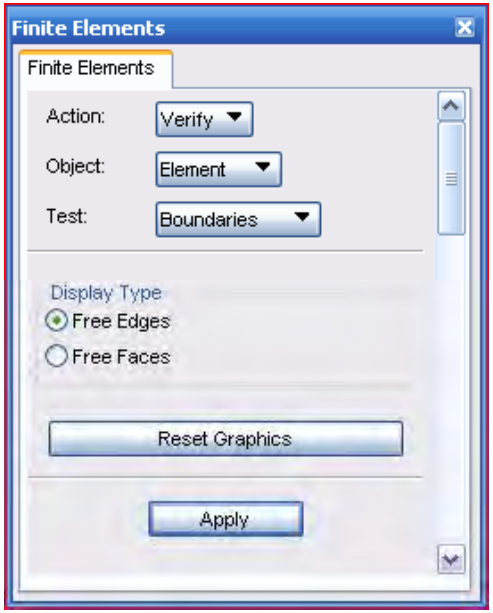
Check for free edges:

Under the Meshing tab click on **Verify** from the FEM Actions group.

Click **Apply**.

There are no free edges at the perimeter of the interface.

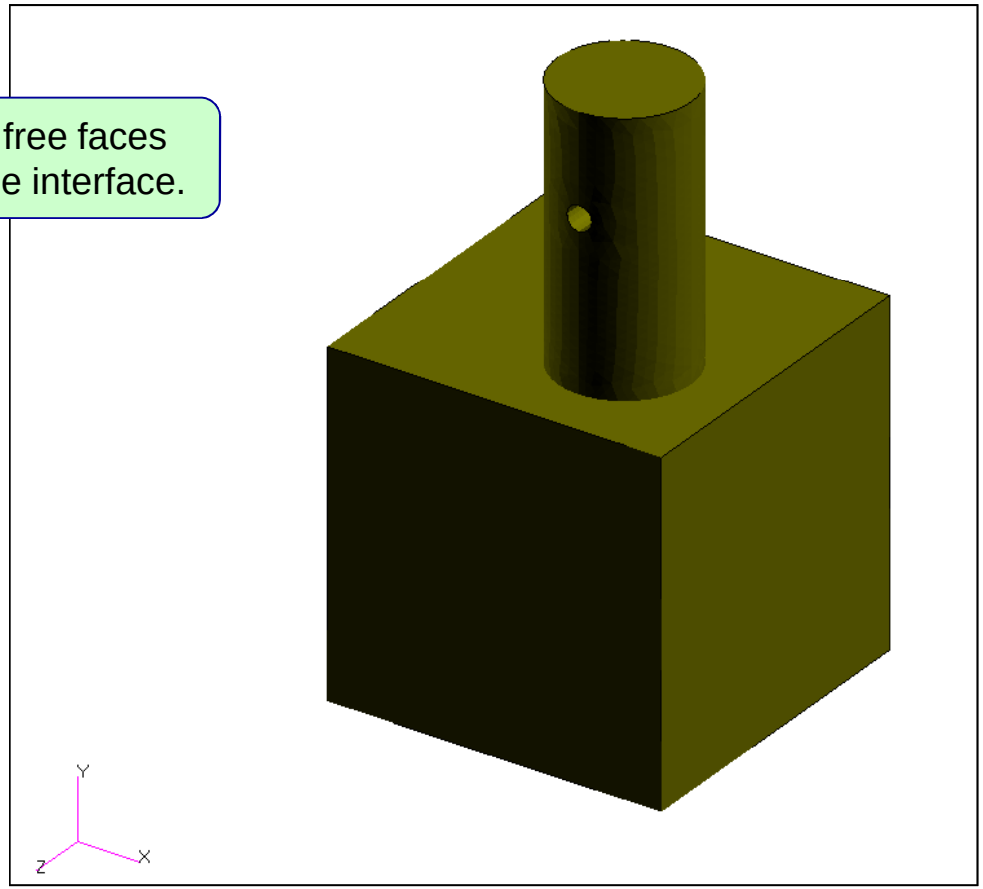


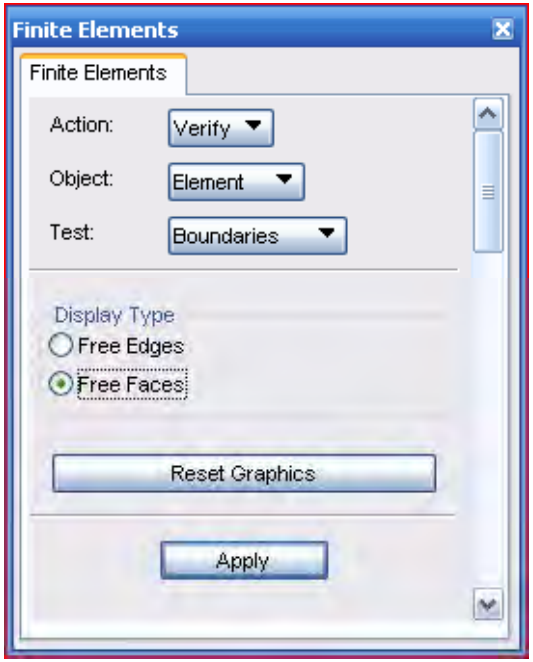


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1

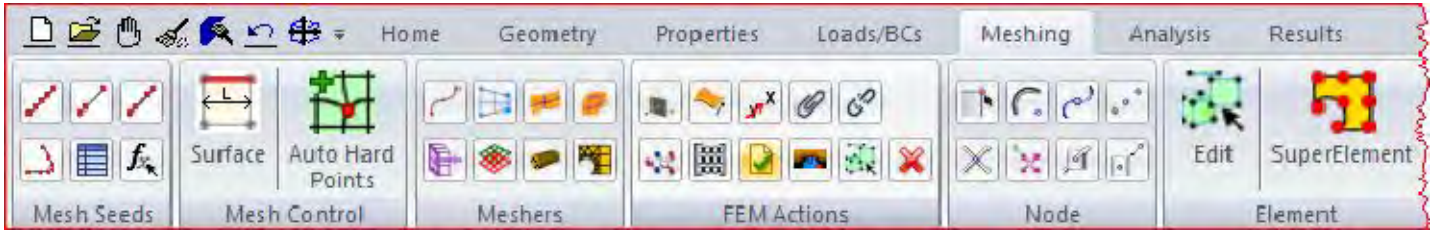
Check for free faces through the interface.



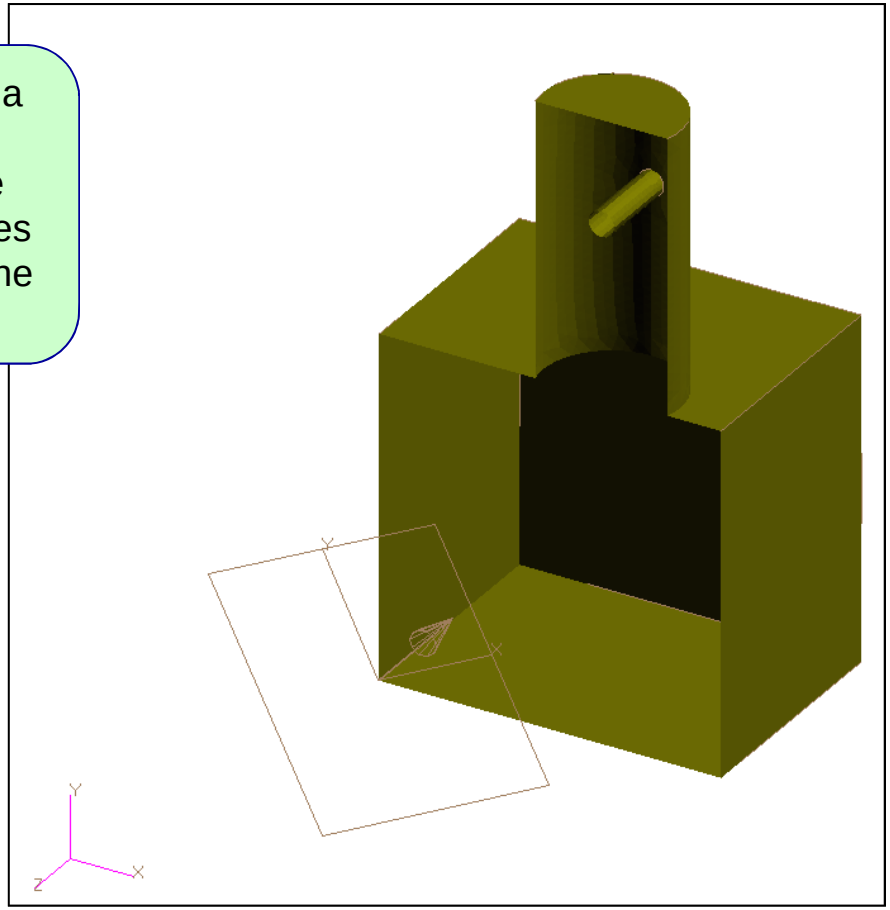


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 1



Cut the model with a clipping plane to verify that there are no internal free faces at the interface of the two solids.



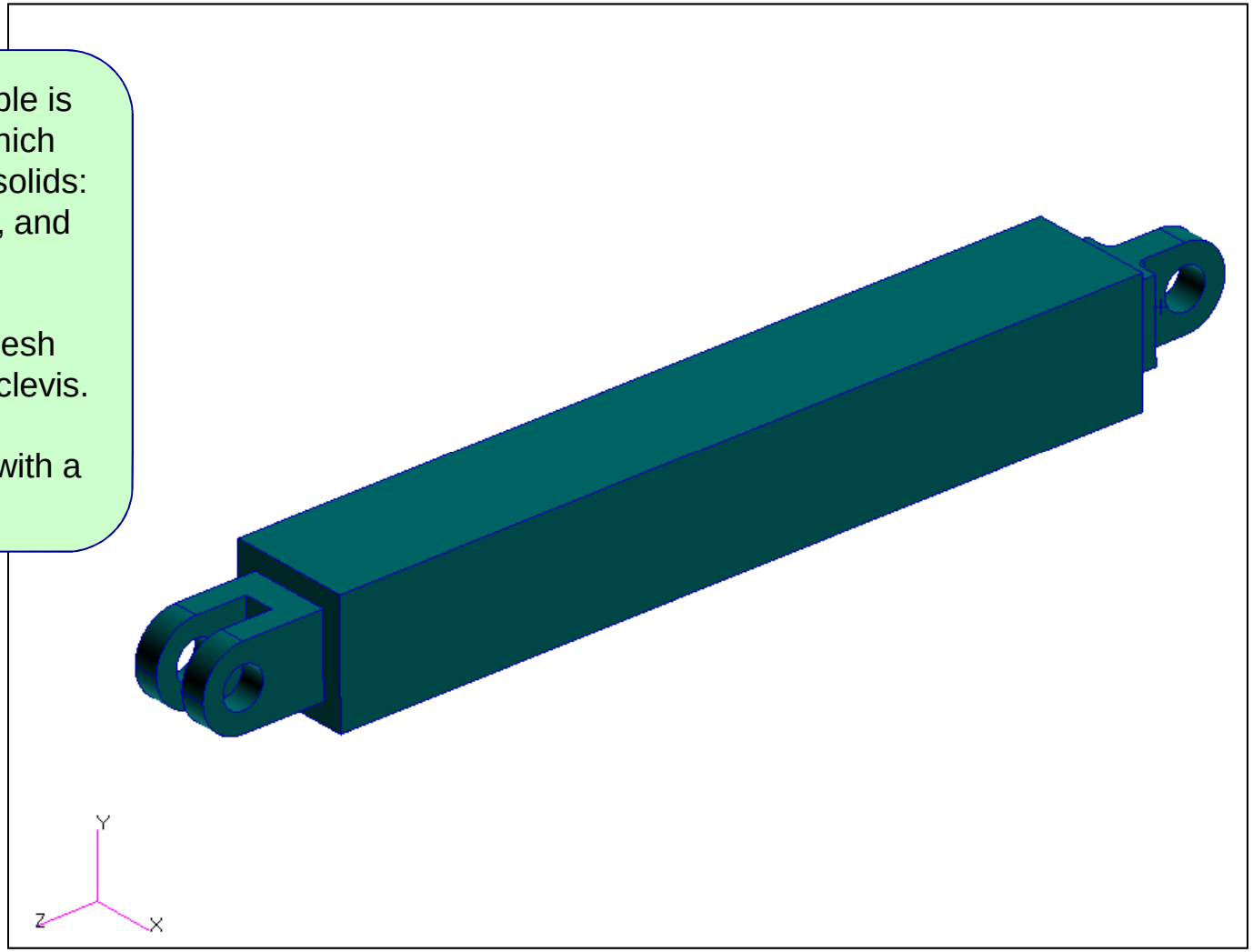
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 2

The next example is an assembly which contains three solids: a clevis, a strut, and a lug.

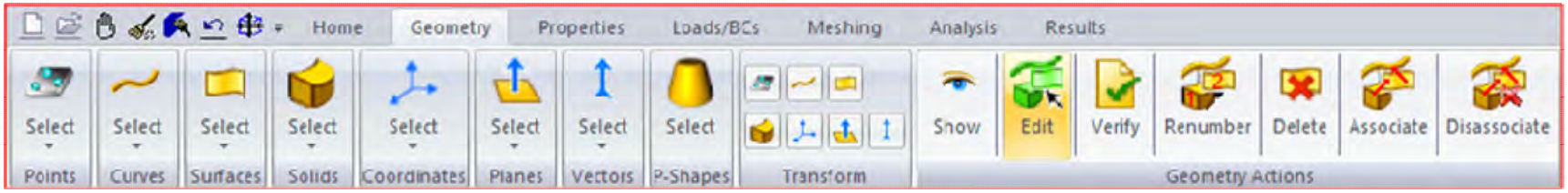
Create a fine mesh for the lug and clevis.

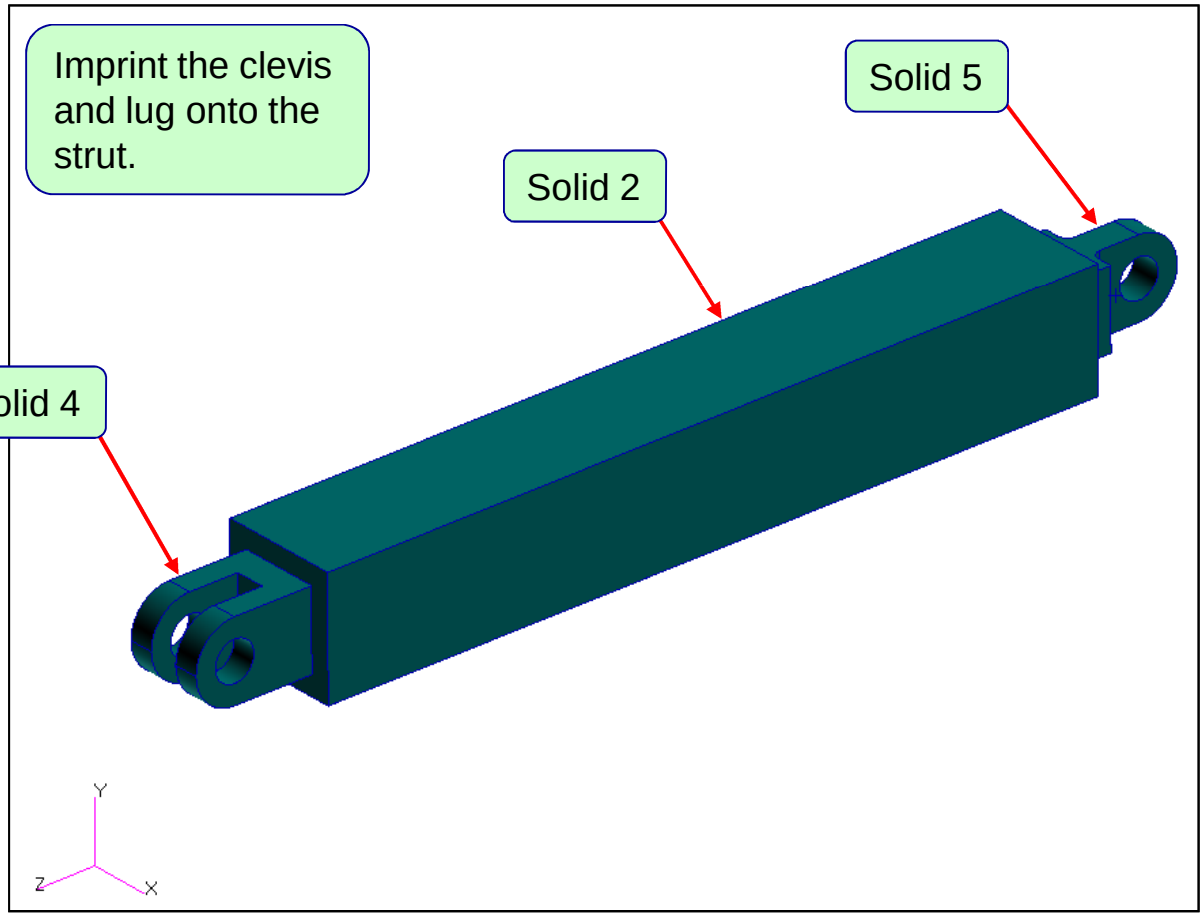
Mesh the strut with a coarser mesh.

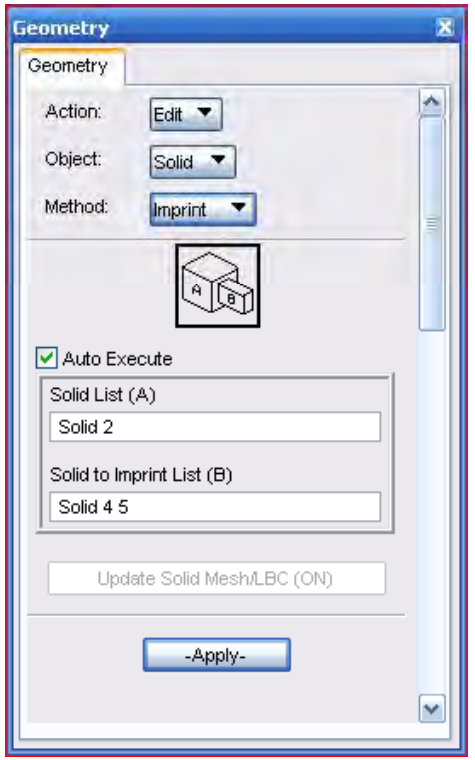


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 2

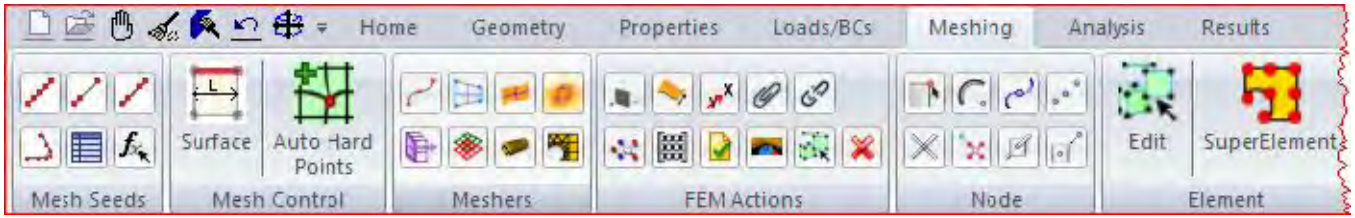




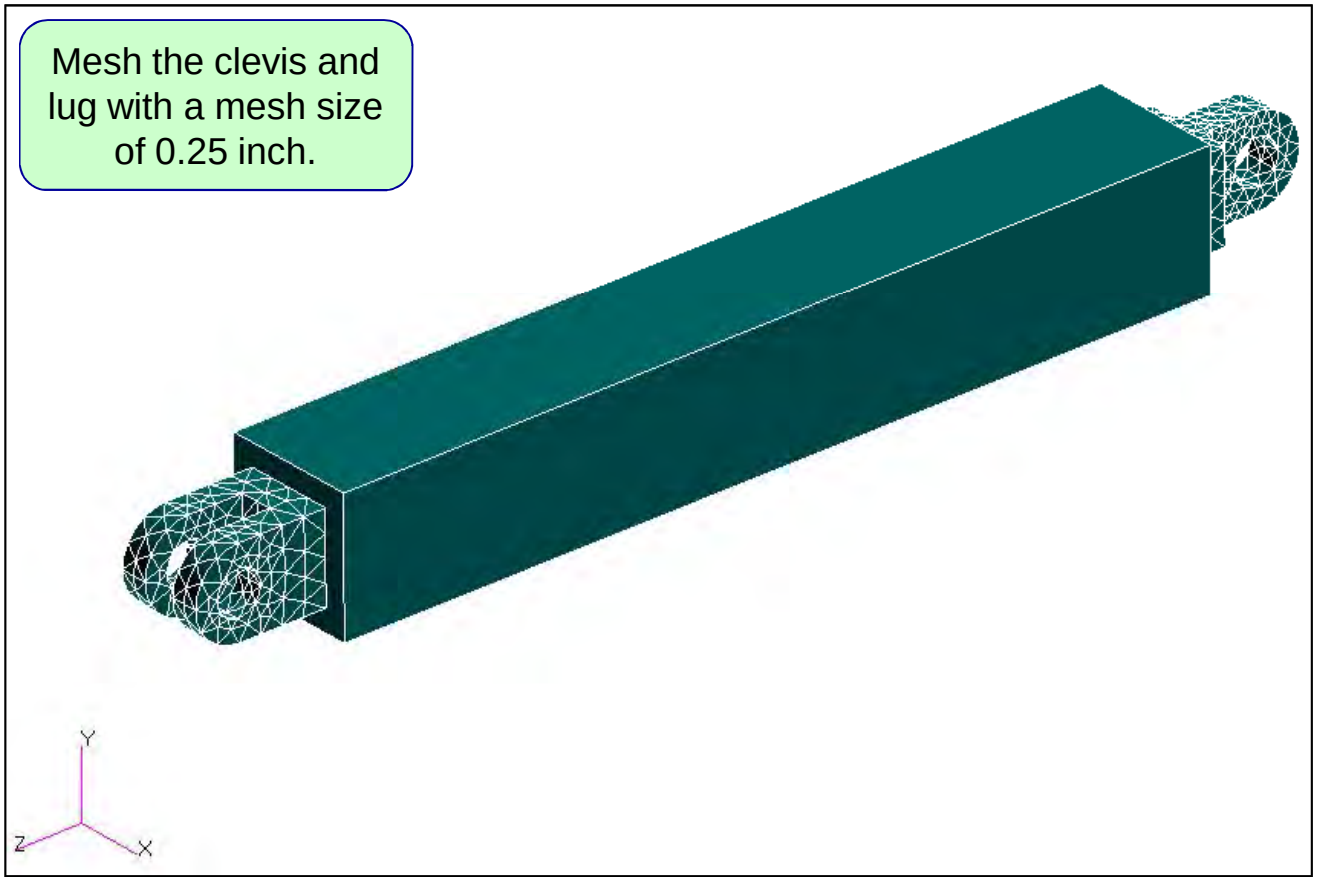


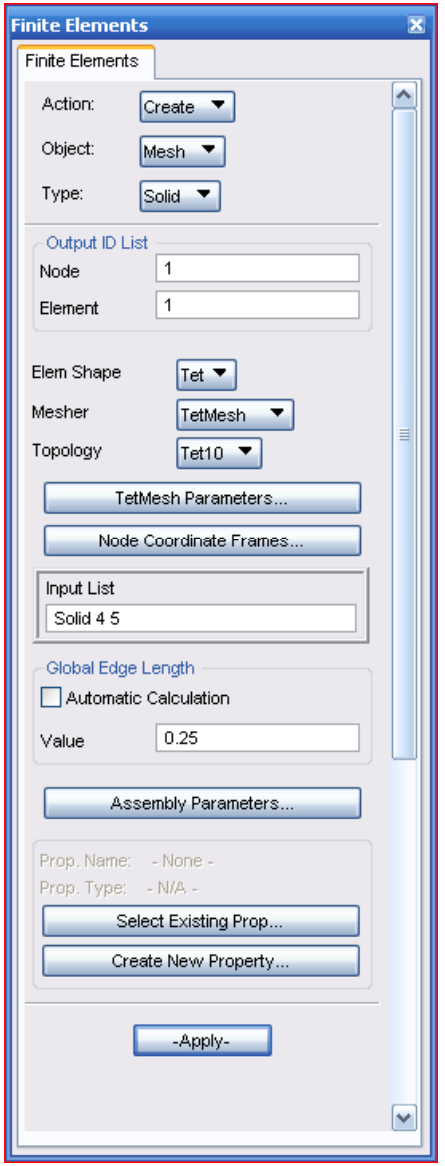
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 2



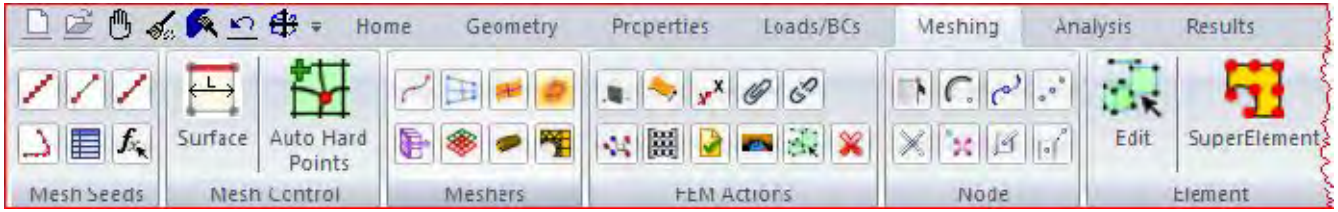
Mesh the clevis and lug with a mesh size of 0.25 inch.





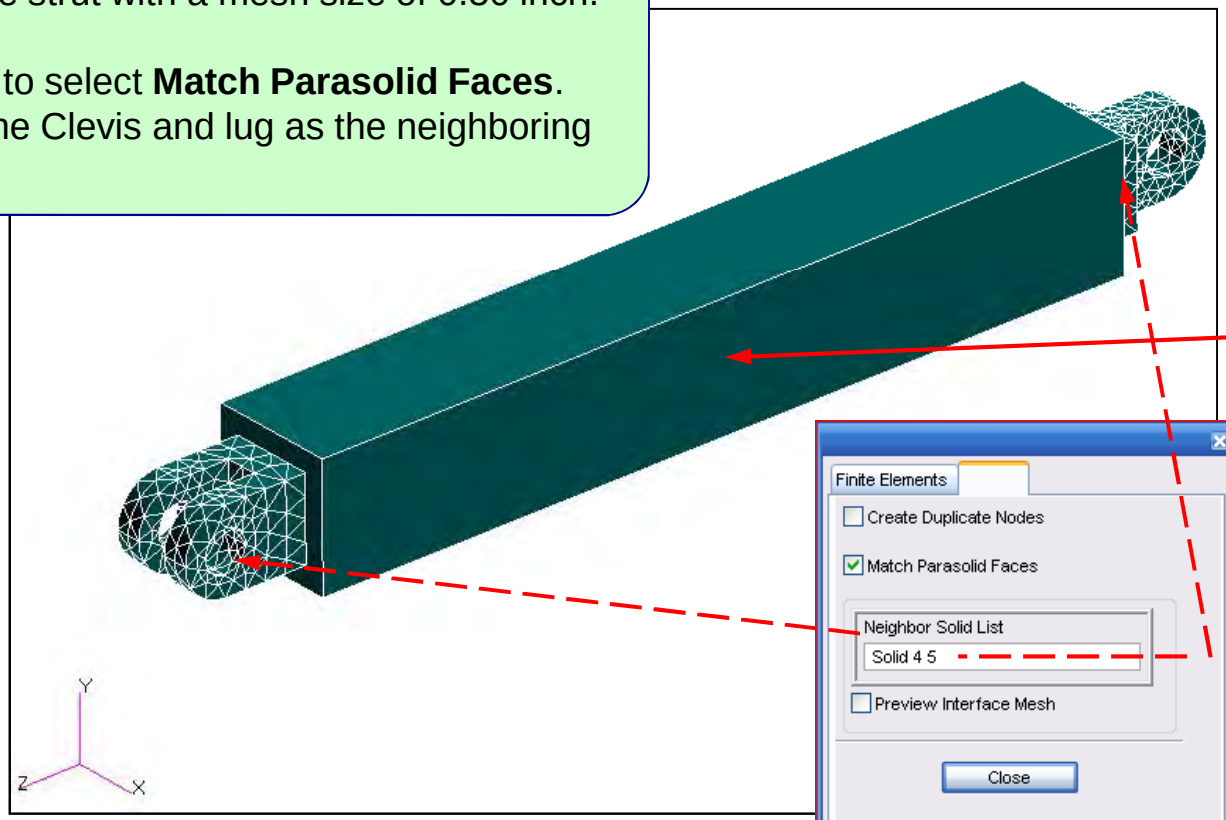
Copyright© 2012 MSC.Software Corporation

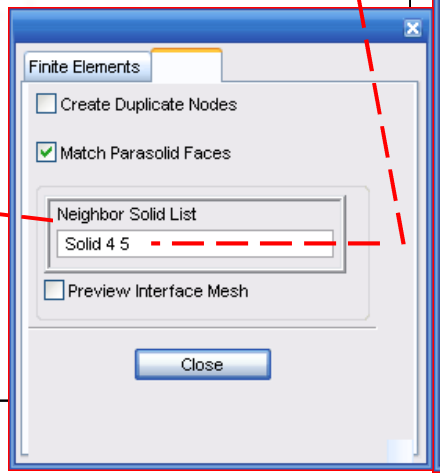
CASE STUDY: ASSEMBLY MESH EX. 2

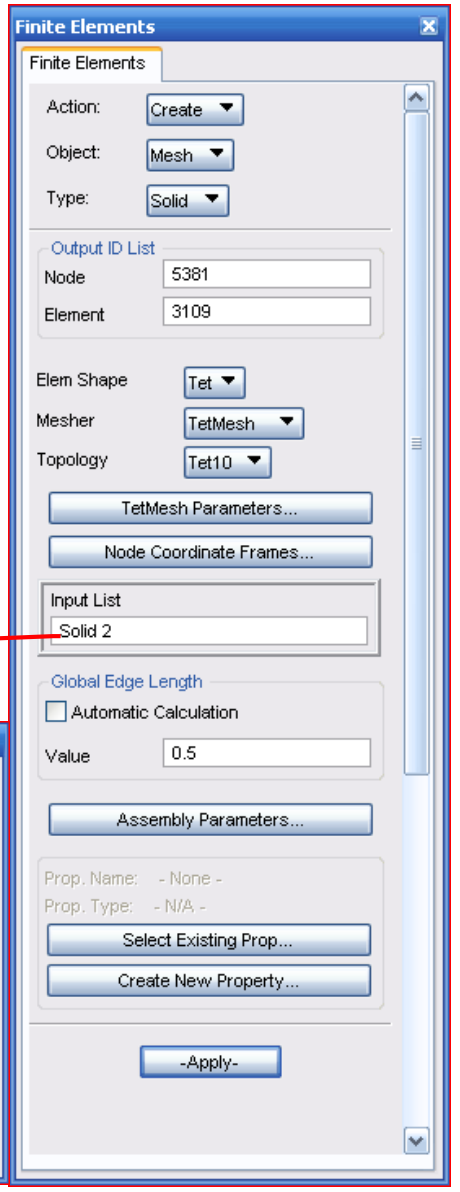


Mesh the strut with a mesh size of 0.50 inch.

Be sure to select **Match Parasolid Faces**. Select the Clevis and lug as the neighboring solids.

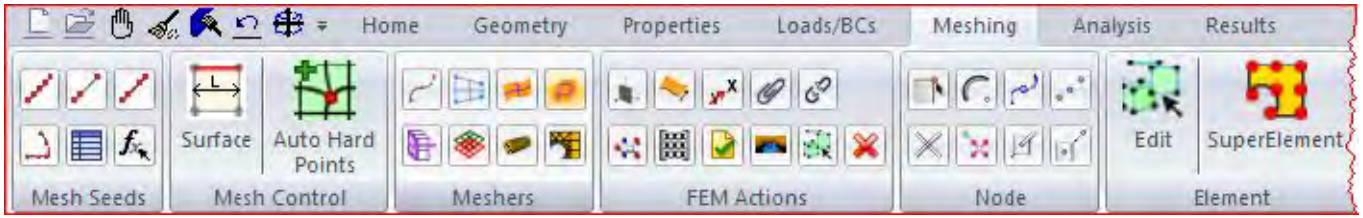




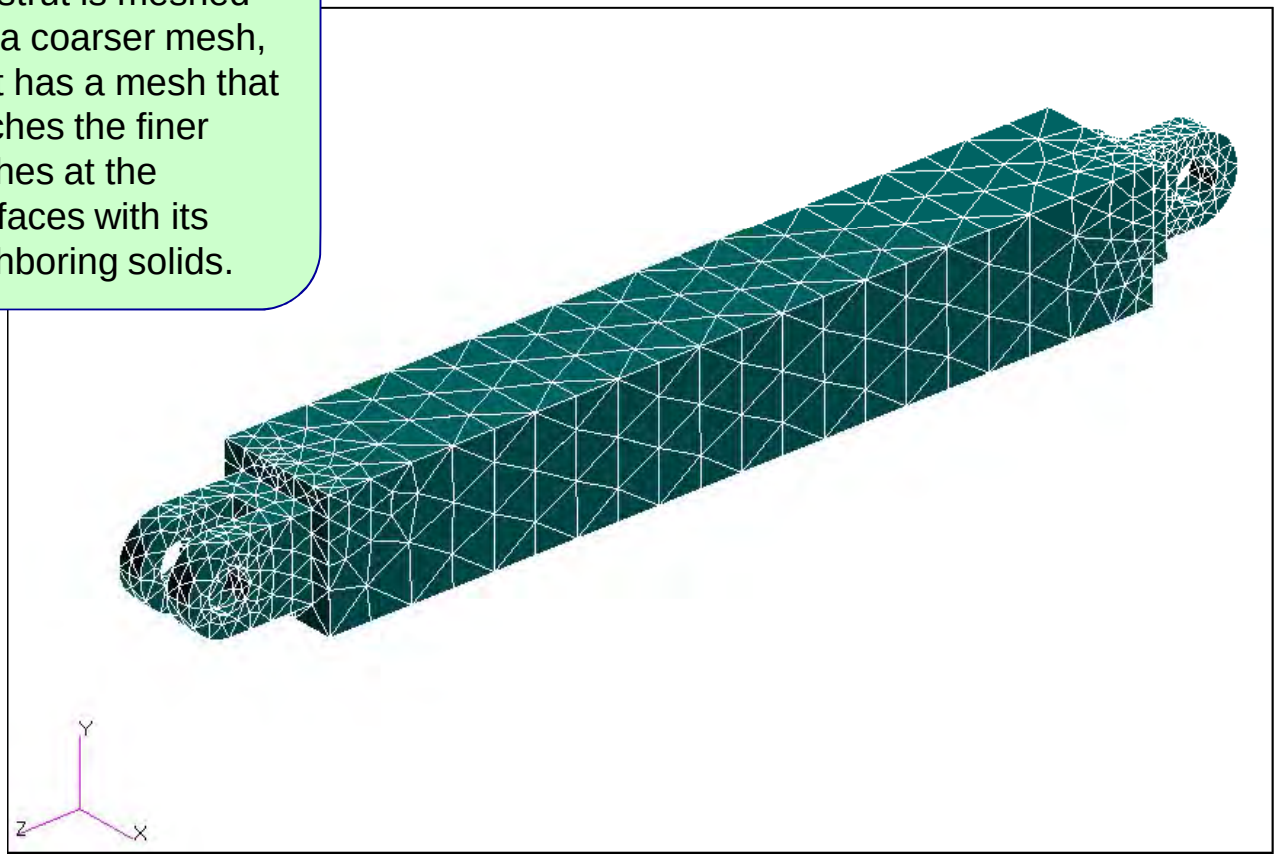


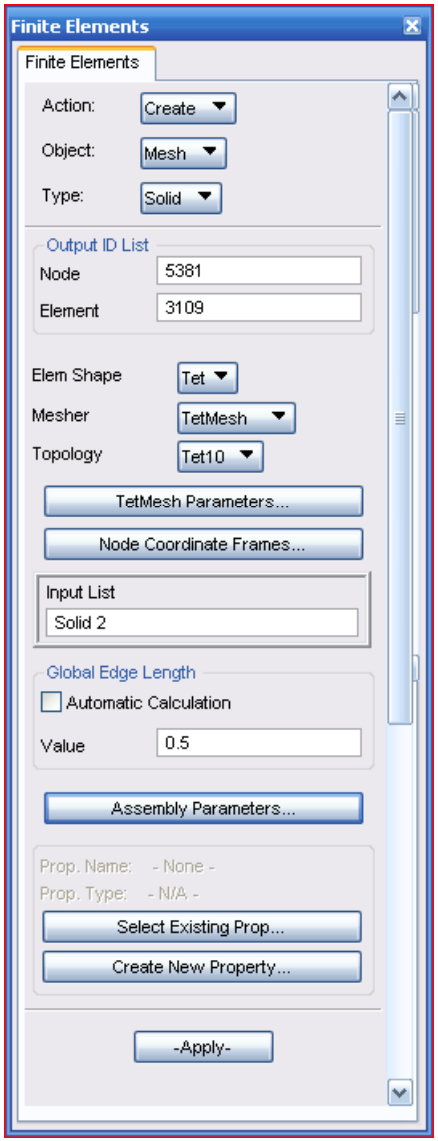
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 2



The strut is meshed with a coarser mesh, but it has a mesh that matches the finer meshes at the interfaces with its neighboring solids.

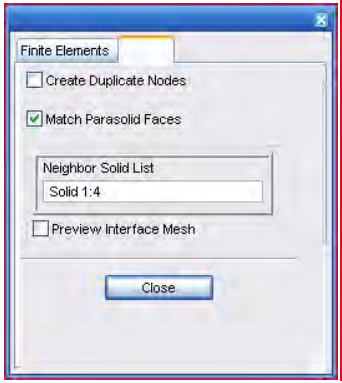


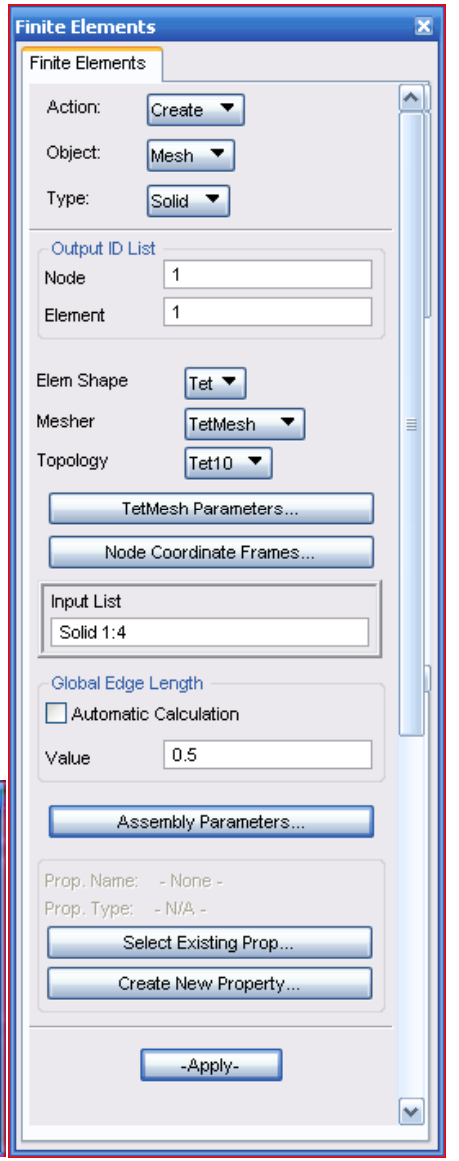


Copyright© 2012 MSC.Software Corporation

ASSEMBLY MESHING OF MULTIPLE SOLIDS IN ONE OPERATION

- **The assembly mesher also provides the capability to mesh multiple solids in an assembly in one operation, with one mesh density.**
- **The steps to be performed are as follows:**
  - Under the *Meshing* tab: Set *Action, Object,* and *Type* as **Create, Mesh,** and **Solid,** respectively
  - Use **Tet**, **TetMesh**, and **Tet10**
  - Select **all** solids for *Input List,* e.g. **Solid 1:4**
  - Specify a value for *Global Edge Length*
  - Select the check box for **Match Parasolid Faces**
    - Select **all** solids for *Neighbor Solid List*, e.g. **Solid 1:4**
  - Click **Apply**

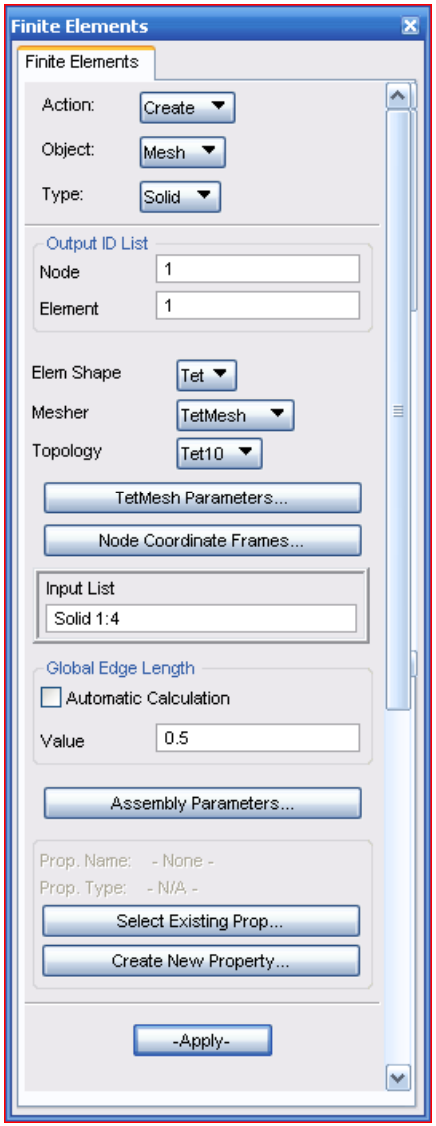


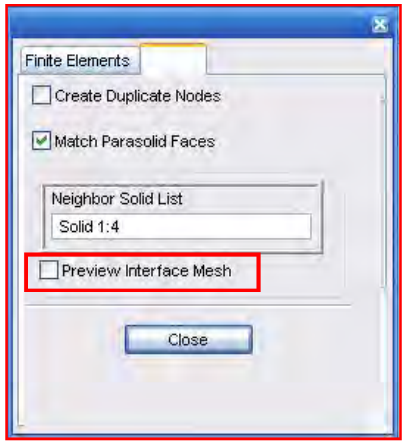


Copyright© 2012 MSC.Software Corporation

ASSEMBLY MESHING OF MULTIPLE SOLIDS IN ONE OPERATION

- **The “Match Parasolid Faces” option provides congruent meshes across non-congruent solid faces at interfaces of solid assemblies.**
  - This is done by **temporarily imprinting** solid faces onto adjacent solid faces at interfaces. After the meshing is complete, the imprinting is removed.
- **The interface mesh can be viewed using the Preview Interface Mesh toggle on the same form.**

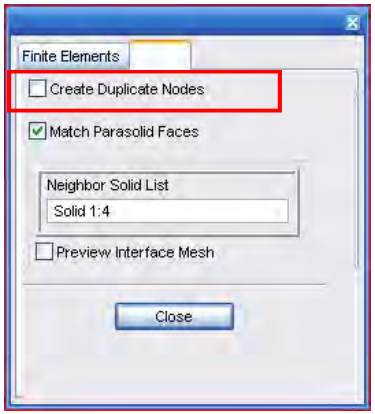


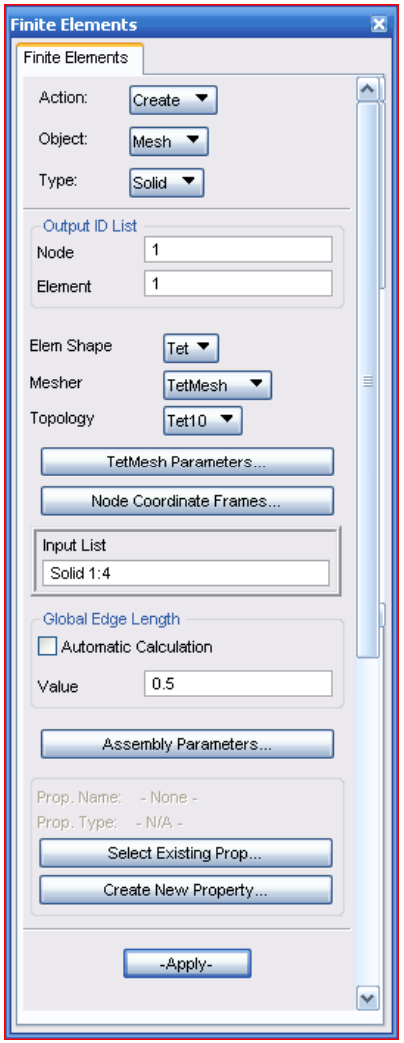


Copyright© 2012 MSC.Software Corporation

ASSEMBLY MESHING OF MULTIPLE SOLIDS IN ONE OPERATION

- **Patran TetMesh has a toggle, Create Duplicate Nodes, that can be used to control equivalencing during meshing. If checked “off”, the nodes at all interfaces in the assembly are automatically equivalenced. If checked “on”, no equivalencing is performed during meshing.**
- **In some modeling situations, duplicate nodes are required for subsequent creation of bolted or contact type joints. The “Create Duplicate Nodes” option can be toggled on to achieve this.**



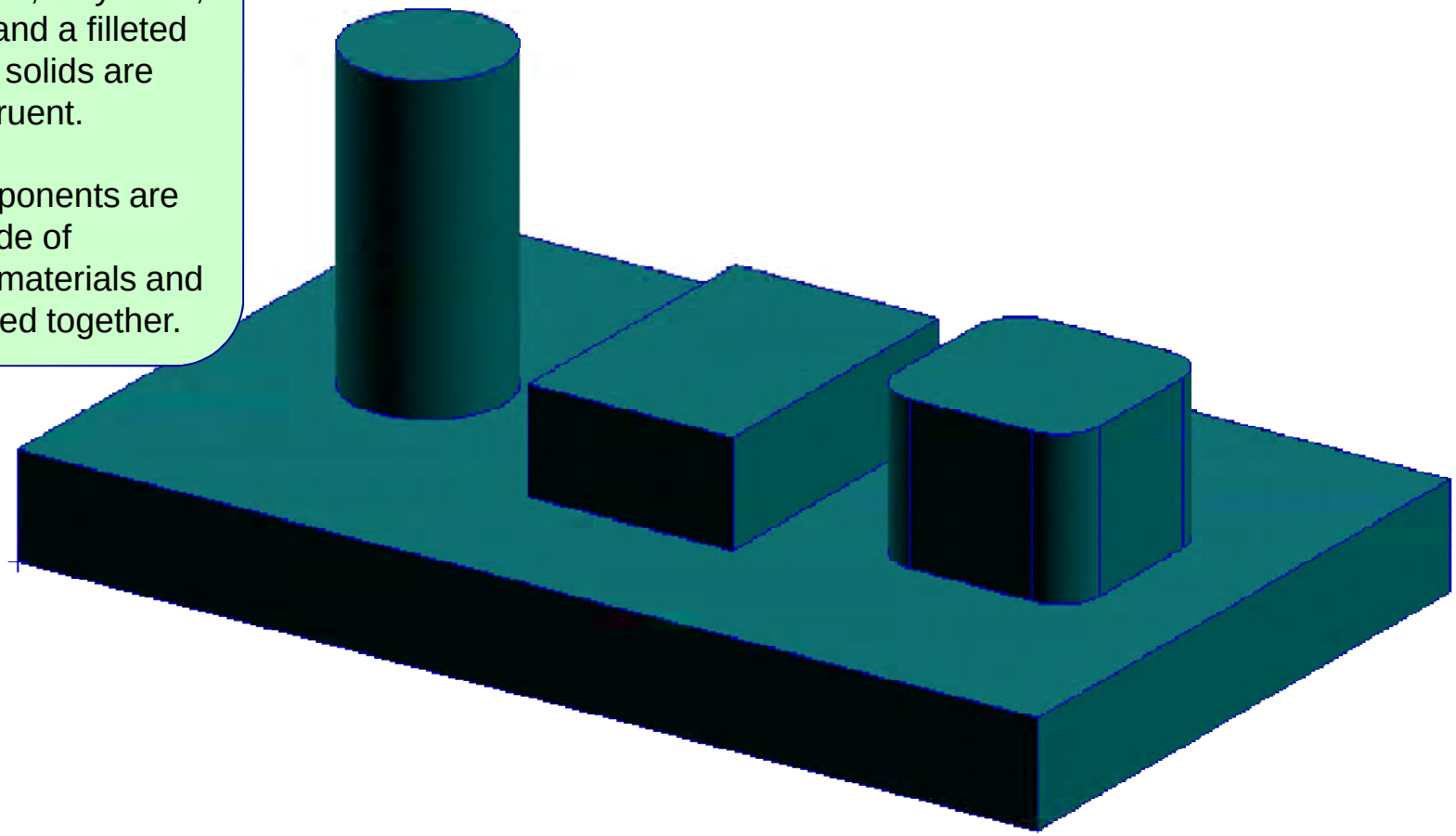


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 3

The assembly illustrated to the right contains four solids: a base place, a cylinder, a block, and a filleted box. The solids are not congruent.

The components are each made of different materials and are bonded together.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 3

Mesh the assembly:

Under the Meshing tab: Create / Mesh / Solid

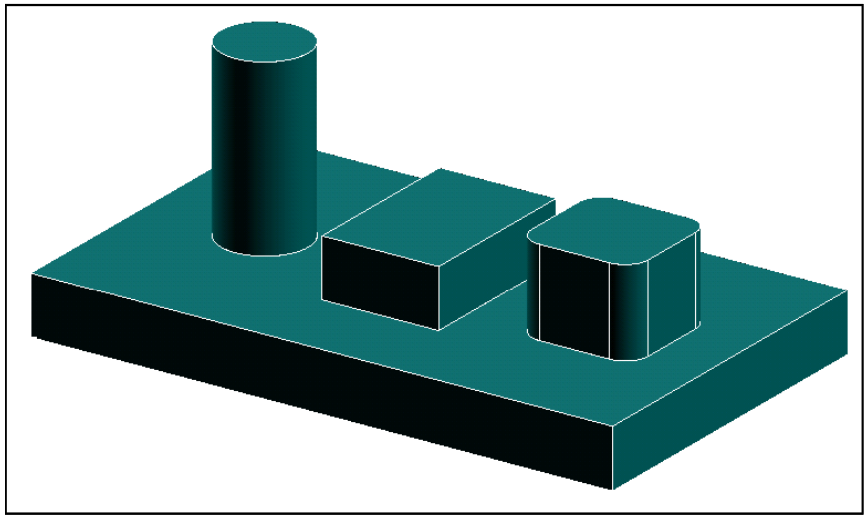
Use Tet10 elements.

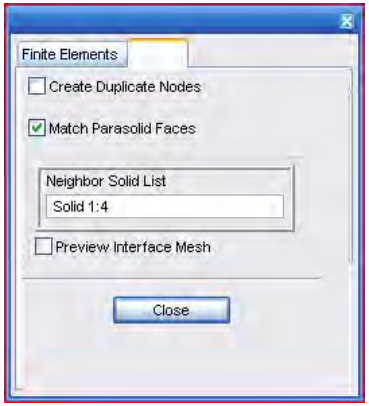
Under *Input List*, select the entire model.

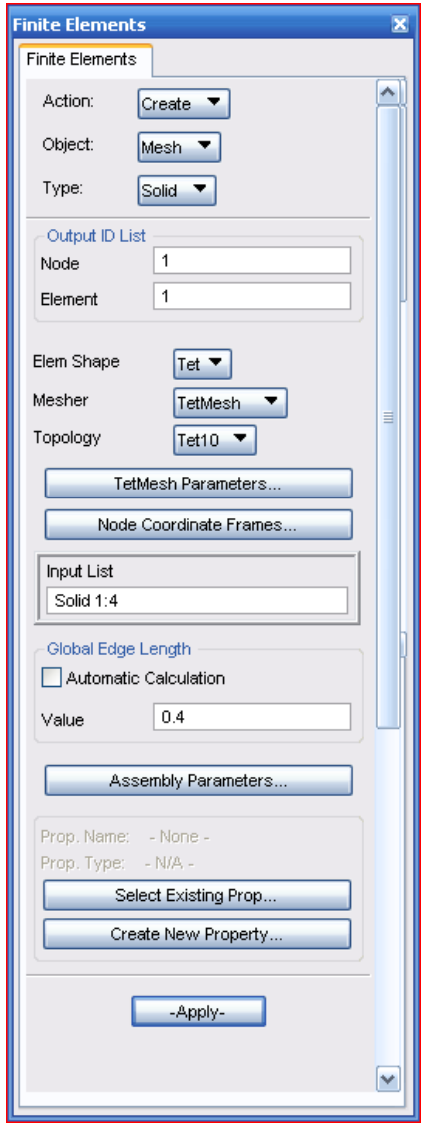
Use a *GEL* of **0.4** and be sure to select **Match Parasolid Faces**.

Select the entire model for the *Neighbor Solid List.*

Do **NOT** click **Apply**.







Copyright© 2012 MSC.Software Corporation

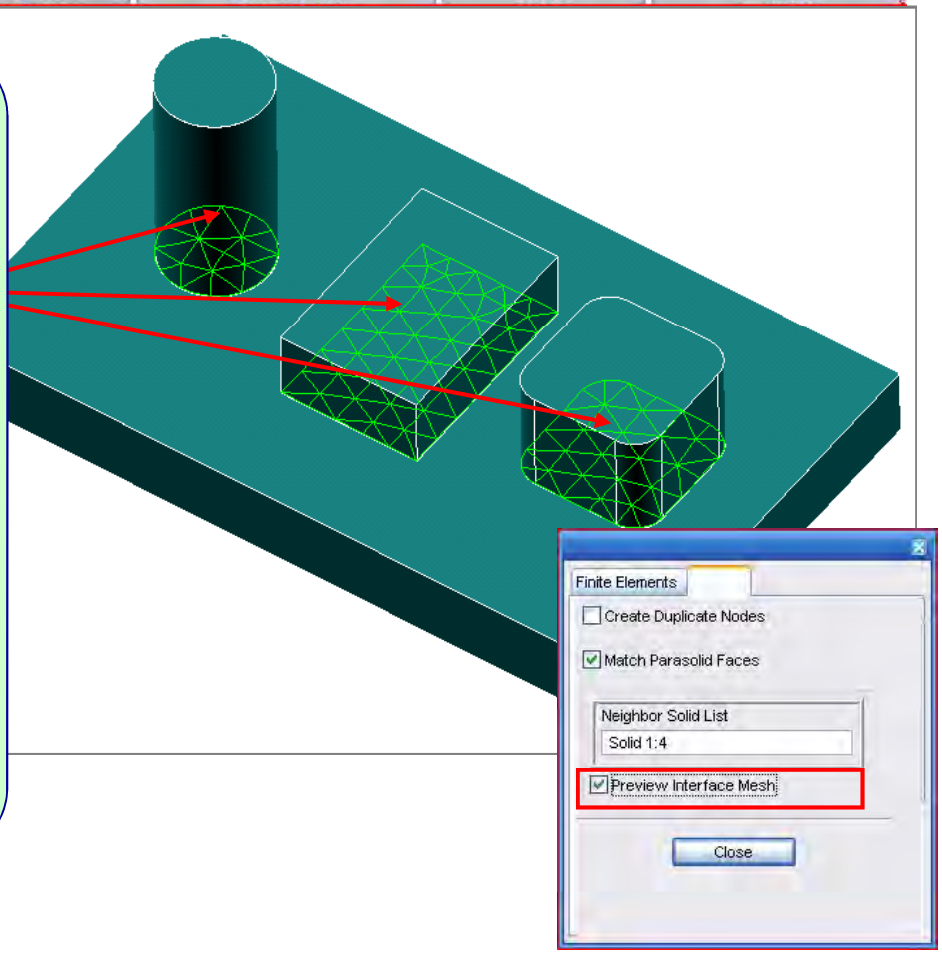
CASE STUDY: ASSEMBLY MESH EX. 3

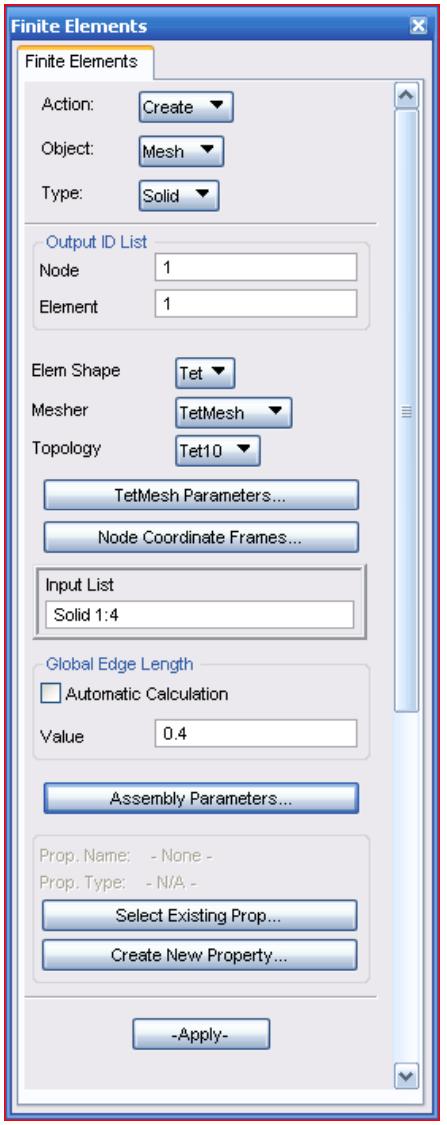
Display the tet mesh pattern at the interface of the solids:

Select the check box for **Preview Interface Mesh**.

Click **Apply**.

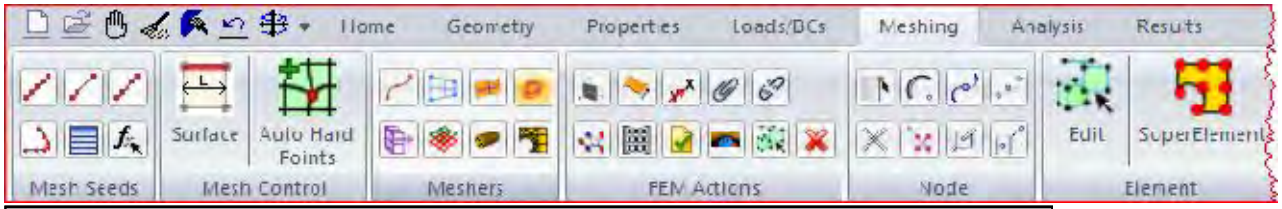
Note that Patran automatically imprints (temporarily) the 3 small solids onto the large base plate, and shows the tria mesh pattern for all solids.

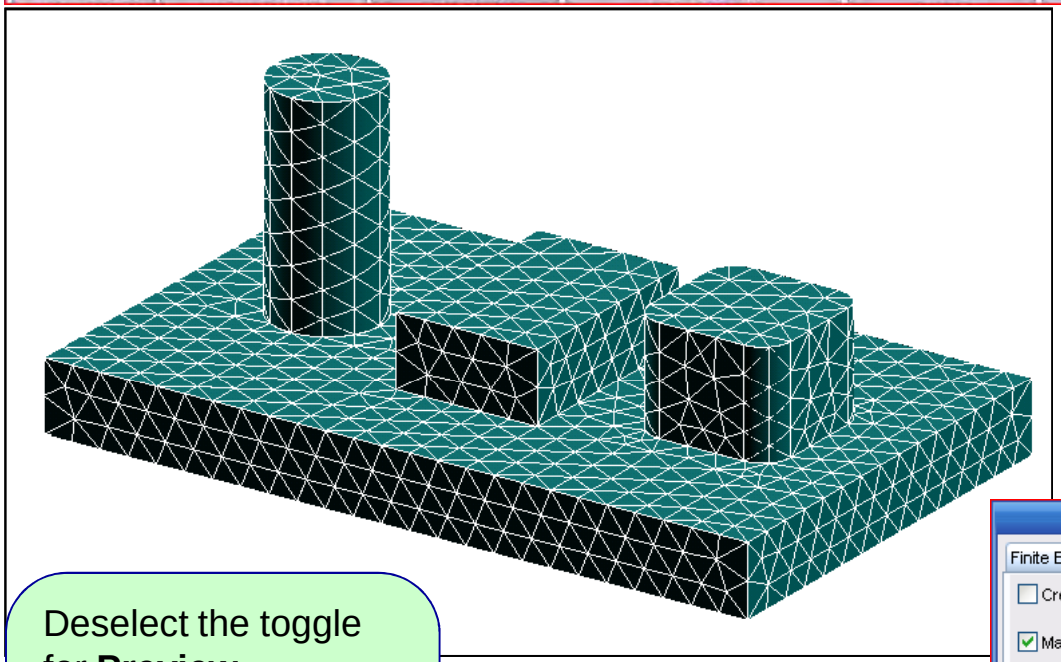




Copyright© 2012 MSC.Software Corporation

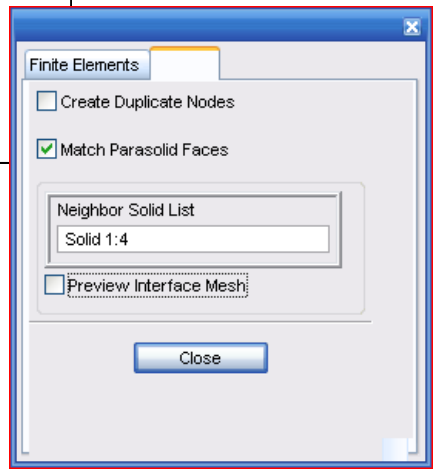
CASE STUDY: ASSEMBLY MESH EX. 3

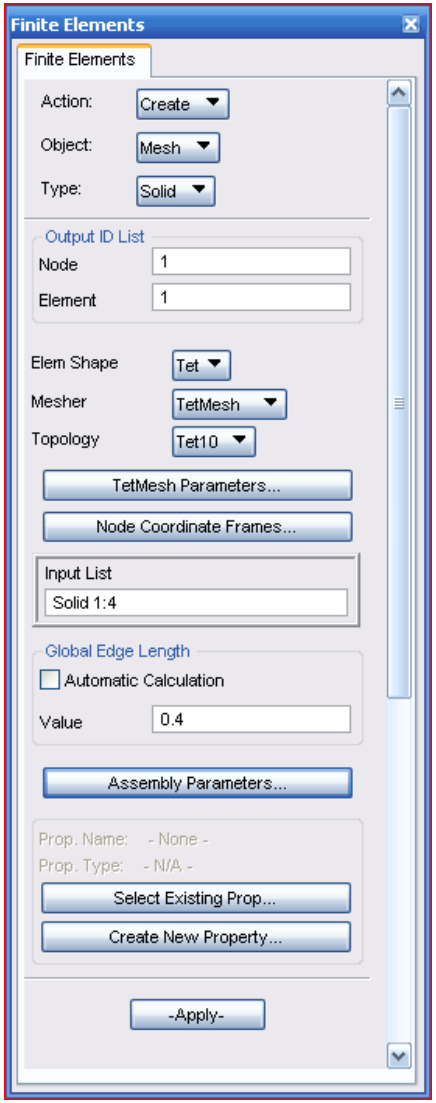




Deselect the toggle for **Preview Interface Mesh**.

Click **Apply** and the entire assembly mesh is created.

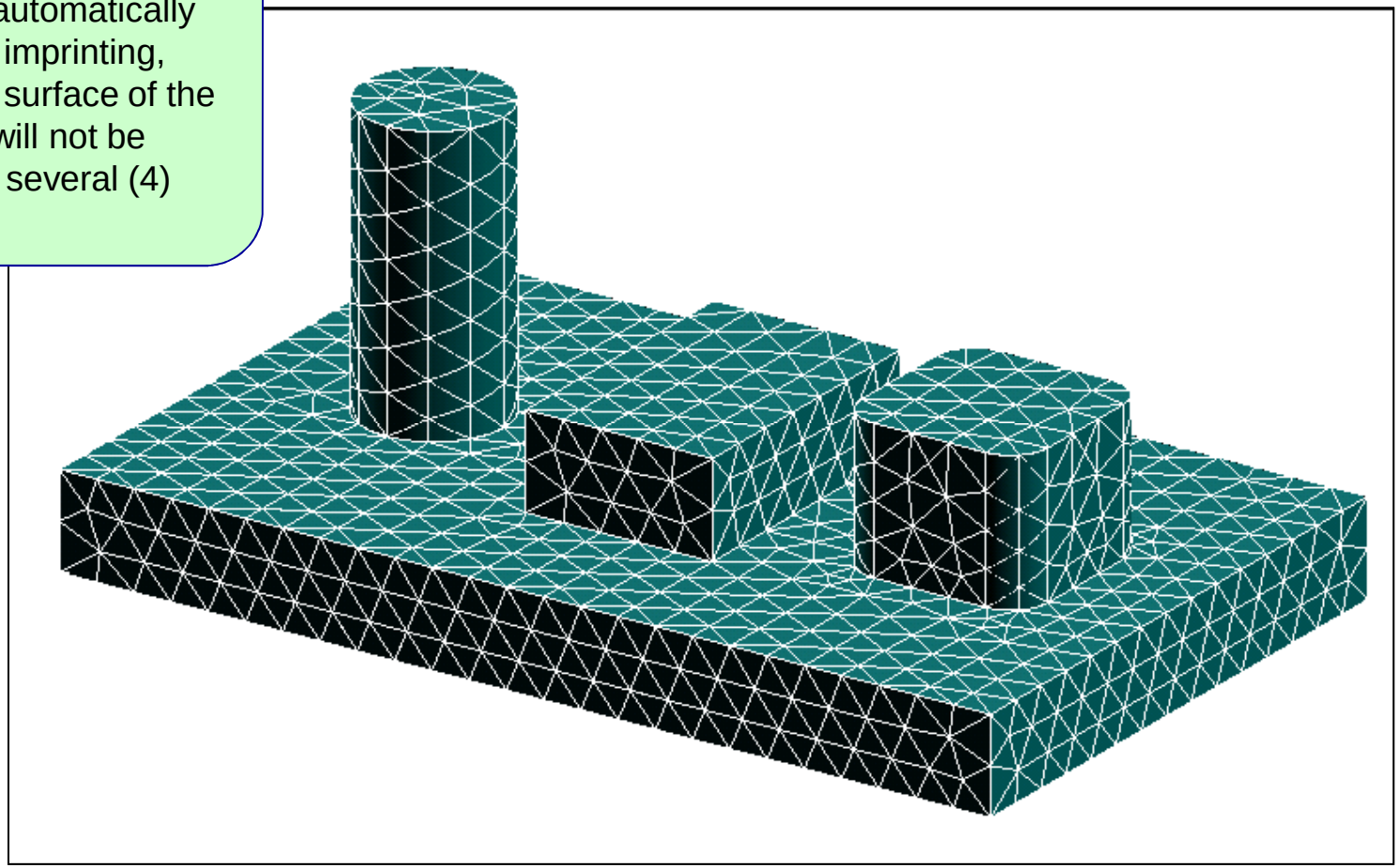




Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 3

After creating the mesh, Patran will automatically remove the imprinting, and the top surface of the base plate will not be divided into several (4) solid faces.



Copyright© 2012 MSC.Software Corporation

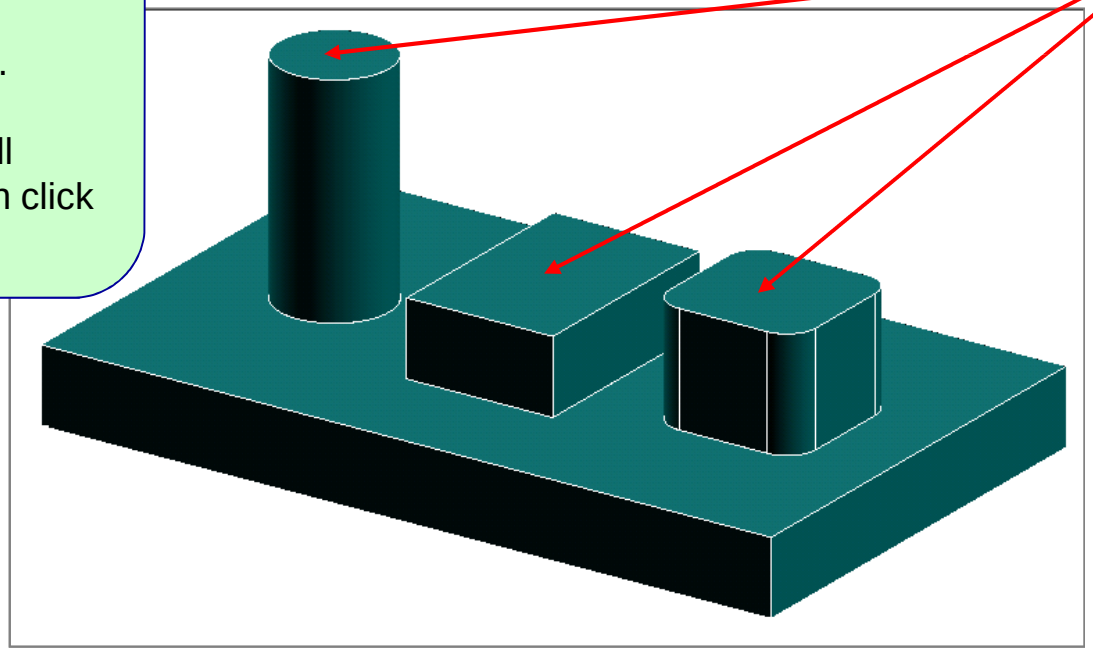
CASE STUDY: ASSEMBLY MESH EX. 3

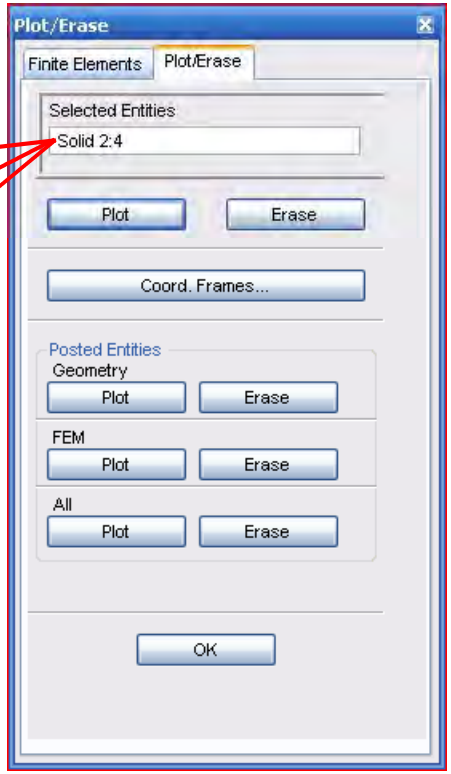
To verify the previous statement, perform the following steps:

Select **Plot/Erase** under the *Display* menu

Erase the FEM.

Select the small solids, and then click **Erase**.





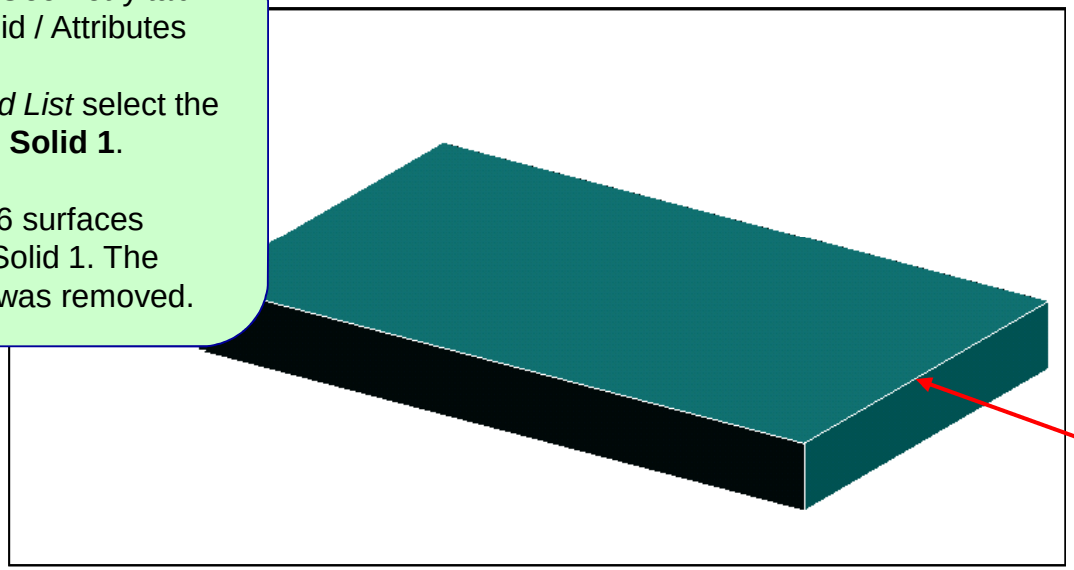
Copyright© 2012 MSC.Software Corporation

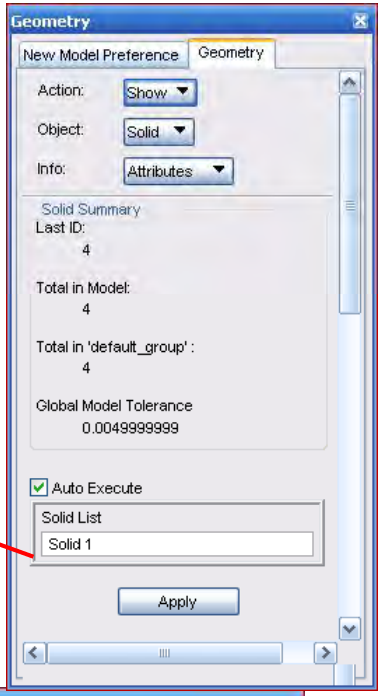
CASE STUDY: ASSEMBLY MESH EX. 3

Under the Geometry tab: Show / Solid / Attributes

Under *Solid List* select the base solid, **Solid 1**.

There are 6 surfaces (faces) in Solid 1. The imprinting was removed.





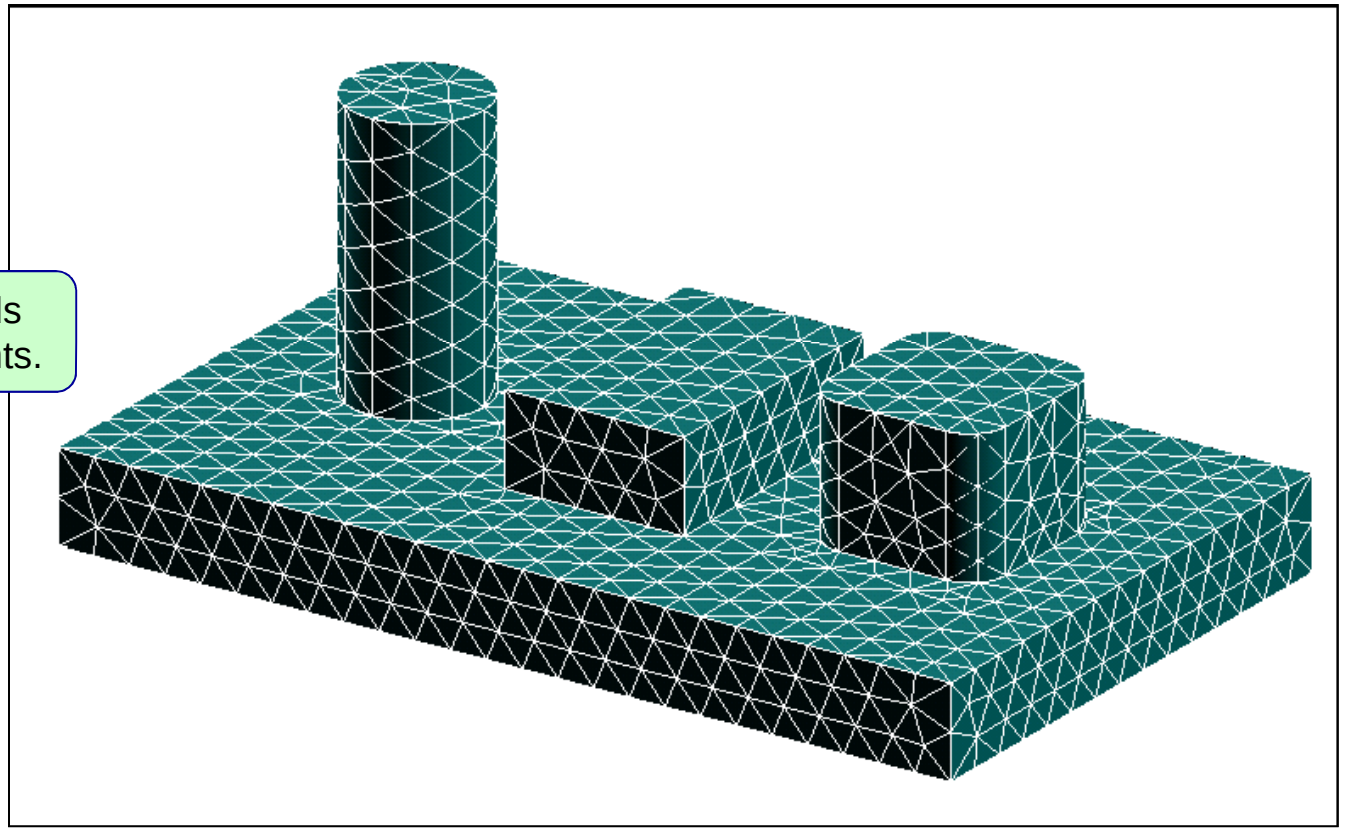
Show Solid Attribute Information

Solid ID	Vertices	Surfaces	Surface Area	Slenderness Ratio	Volume	Type
1	8	6	152.	0.47055751	60.	OrdinaryBody
Summary	--	--	--	--	60.	--

Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 3

Plot the three solids and the tet elements.

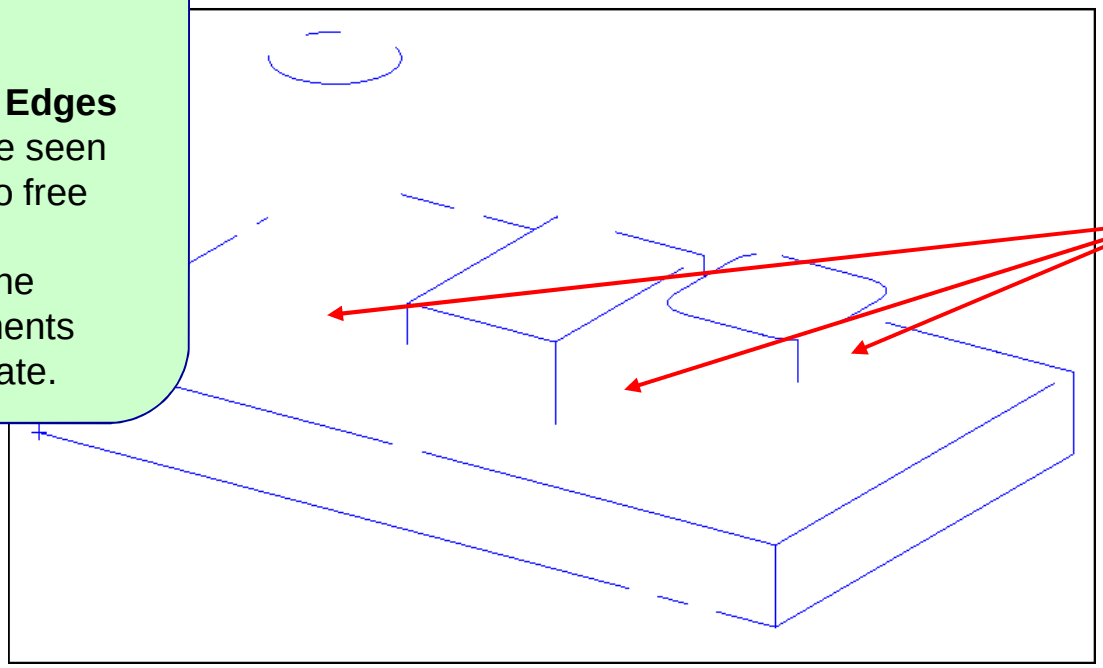


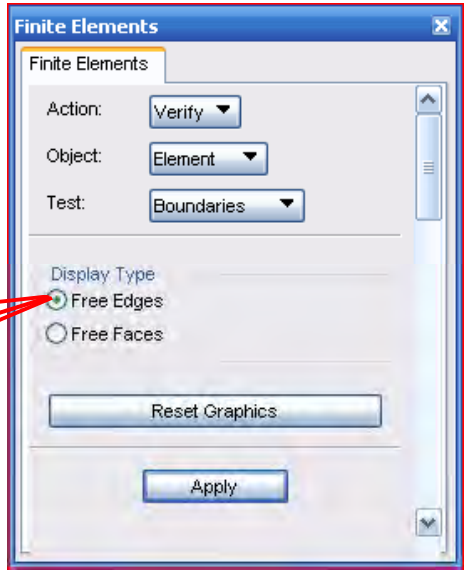
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 3

Find all element free edges.

Select the **Free Edges** toggle. It can be seen that there are no free edges at the intersection of the smaller components and the base plate.

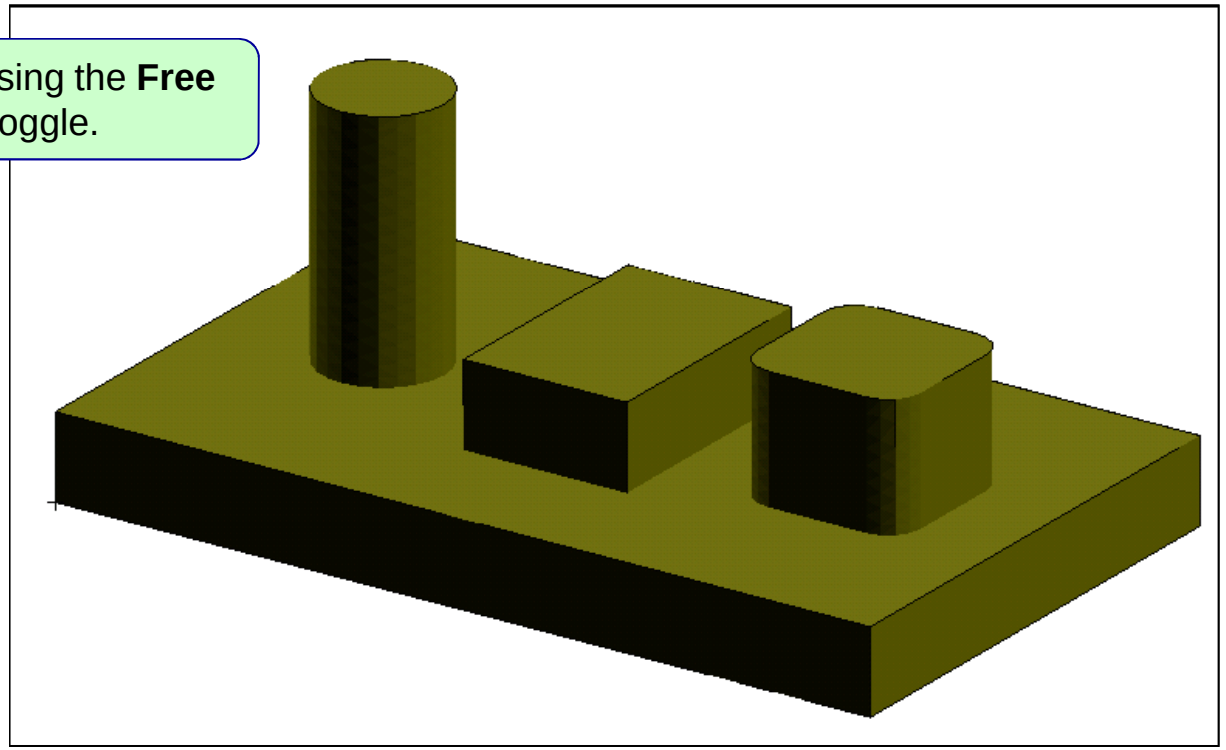


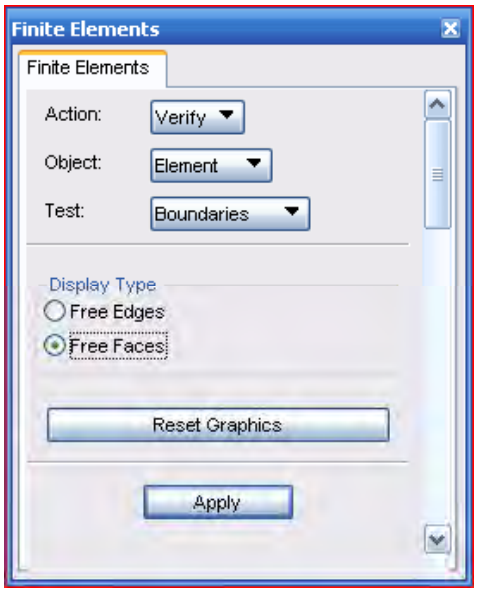


Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 3

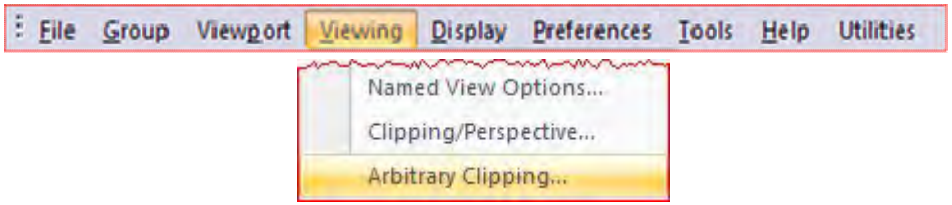
Verify using the **Free Faces** toggle.



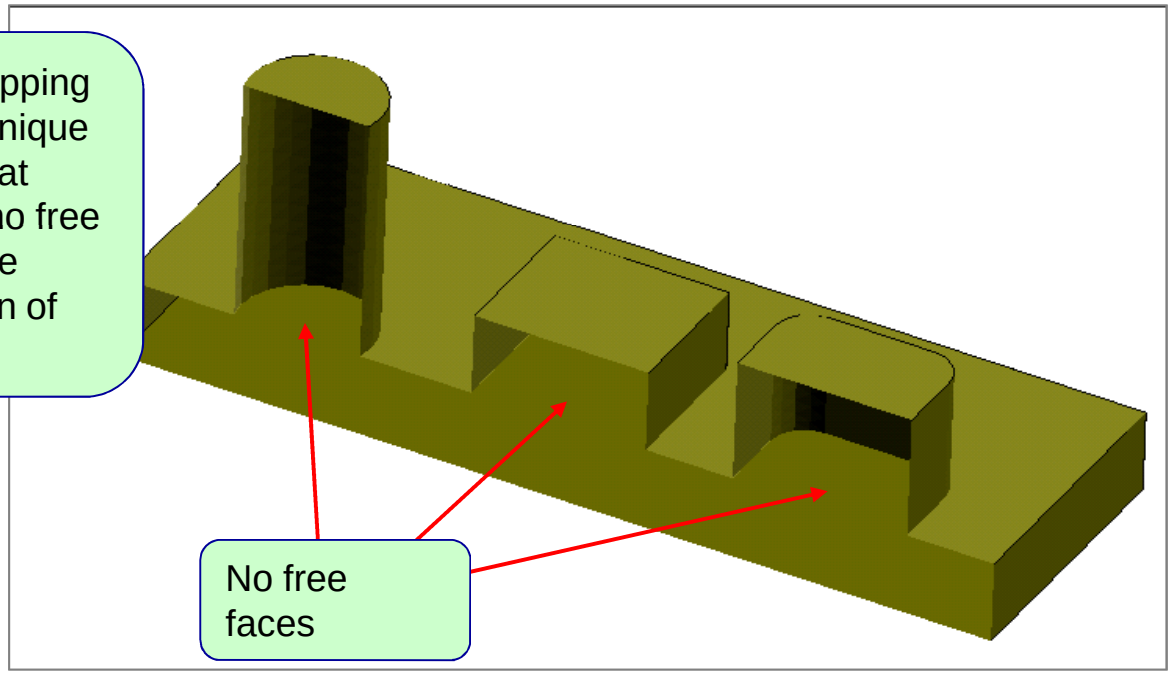


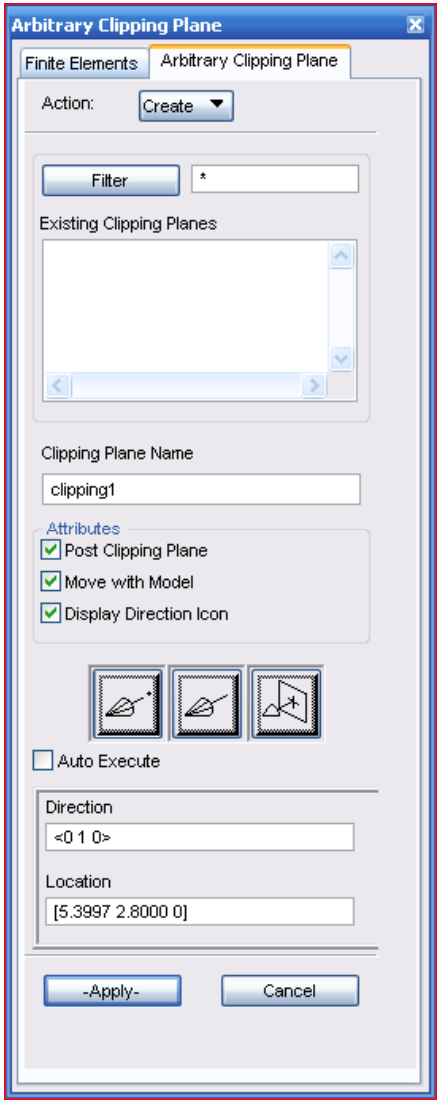
Copyright© 2012 MSC.Software Corporation

CASE STUDY: ASSEMBLY MESH EX. 3



Use the clipping plane technique to verify that there are no free faces at the intersection of the solids.





Copyright© 2012 MSC.Software Corporation

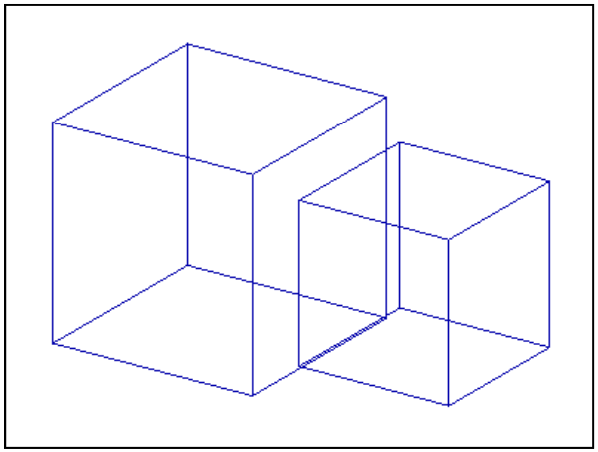
GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

- **There can be gaps and/or intersections between adjacent solids of assemblies.**
- **There are ways of dealing with them:**
  - In Patran
    - Gap
      - TetMesh Parameters, Proximity Tol.
      - Edit geometry
    - Intersection
      - Edit geometry
  - In CAD
    - Edit geometry

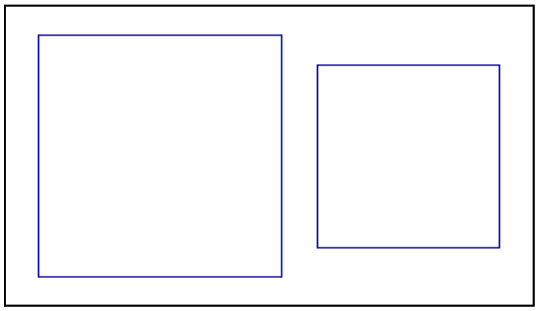
Copyright© 2012 MSC.Software Corporation

GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

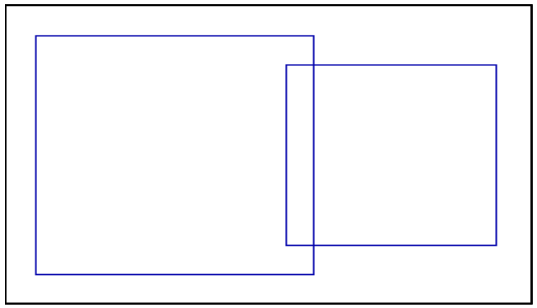
- **Simple example of two parasolid blocks:**
  - Gap
  - Intersection



Two Blocks



Gap

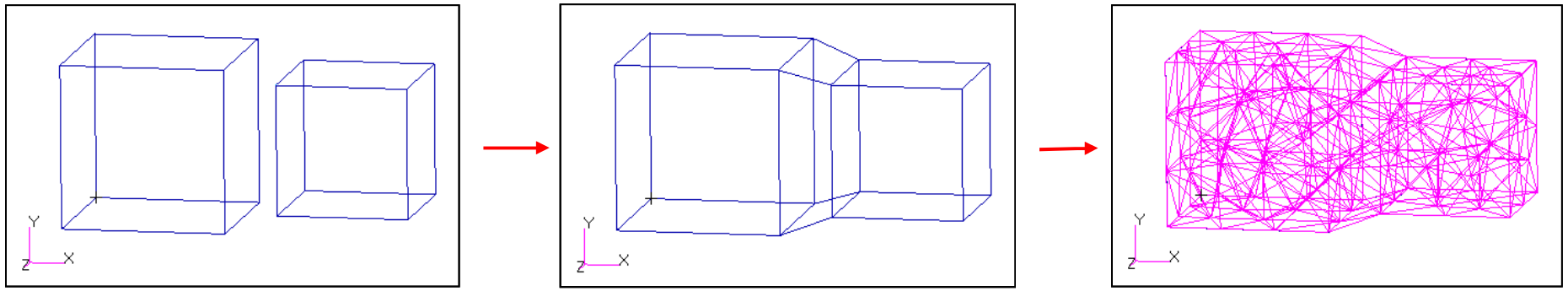


Intersection

Copyright© 2012 MSC.Software Corporation

GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

- **Meshing across a gap for adjacent parasolid solids:**
  - Difference between the two solids at the interface is exaggerated. Such differences should not be created during geometry creation.



Before TetMesh

Gap between adjacent solids

During TetMesh

Corridor temporarily created between adjacent solids

After TetMesh

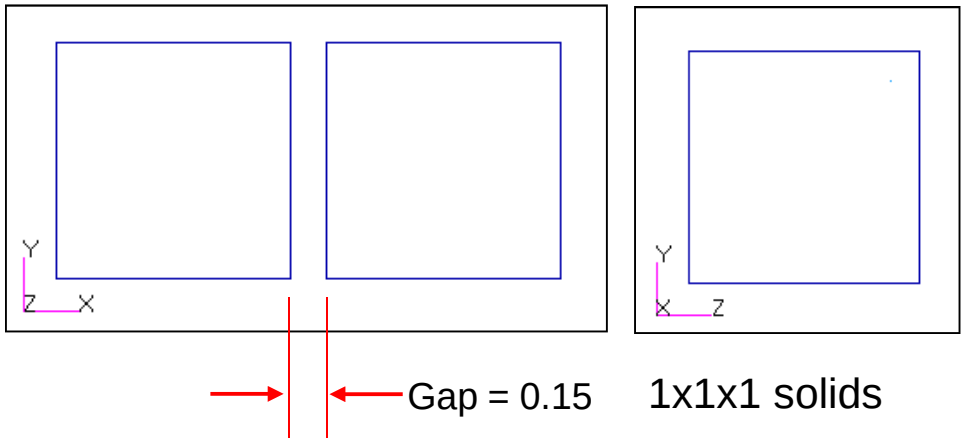
Tets have been created through corridor

(Temporary corridor removed)

Copyright© 2012 MSC.Software Corporation

GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

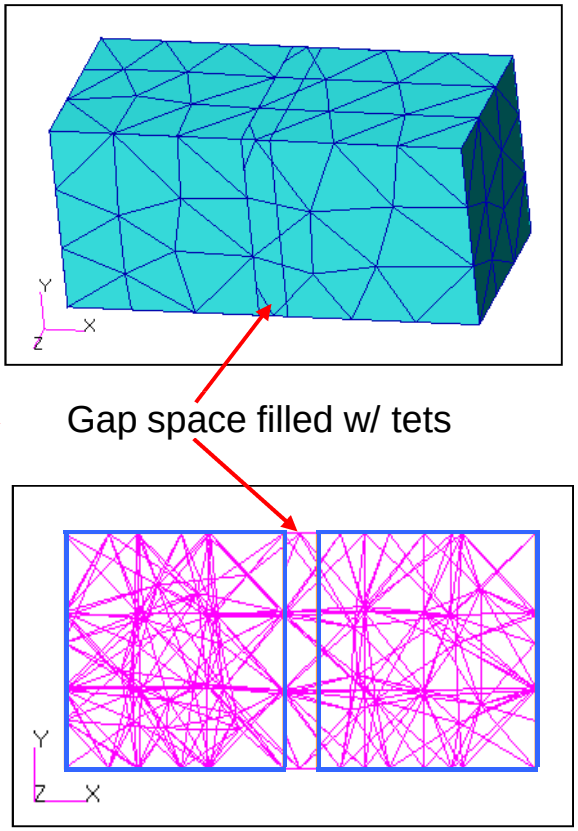
- **Create a TetMesh across a gap**
  - Effect of gap dimension in one direction
  - Use Proximity Tol.
  - Gap is too large; it is not realistic !



TetMesh using Proximity Tol.

Tol. = 0.16

GEL = 0.333



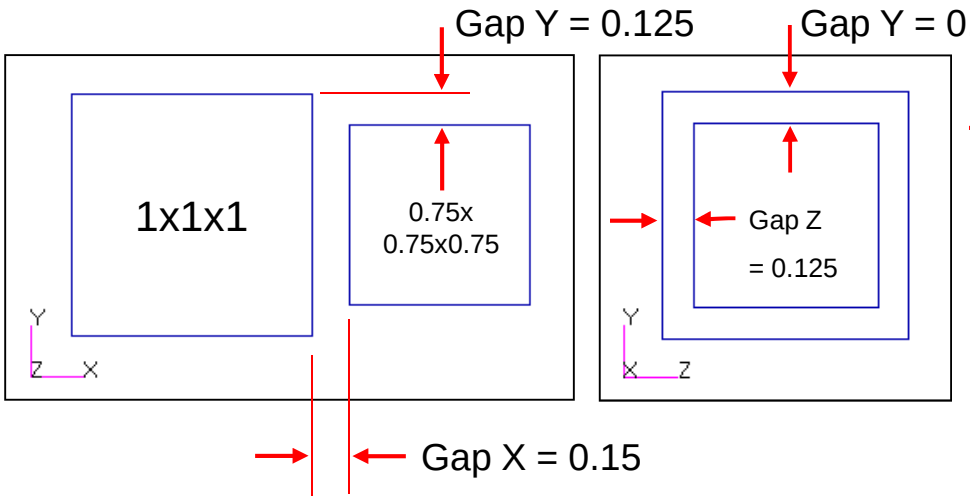
  - The following message from Patran appears

    $# The proximity tolerance specified is a large percentage of the element size meshing success and quality may be degraded.

Copyright© 2012 MSC.Software Corporation

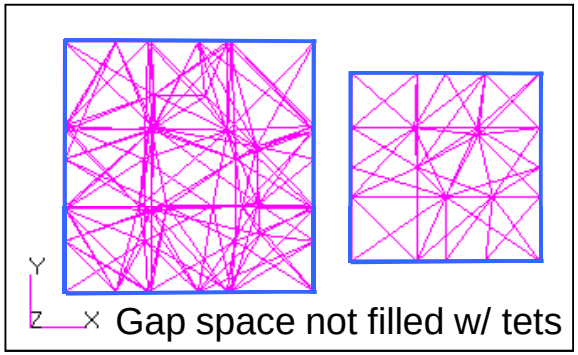
GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

- **Create a TetMesh across a gap**
  - Effect of gap dimension in three directions
  - Using Proximity Tol.



TetMesh using Proximity Tol.

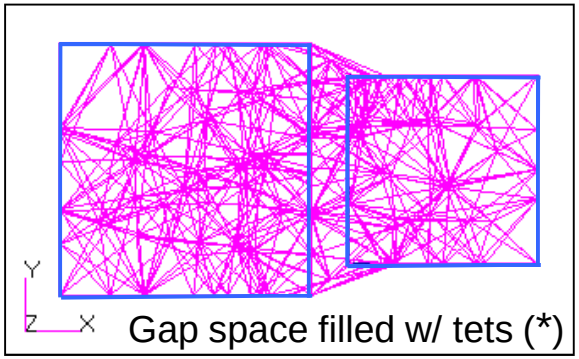
Tol. = 0.19



GEL = 0.333

TetMesh using Proximity Tol.

Tol. = 0.20

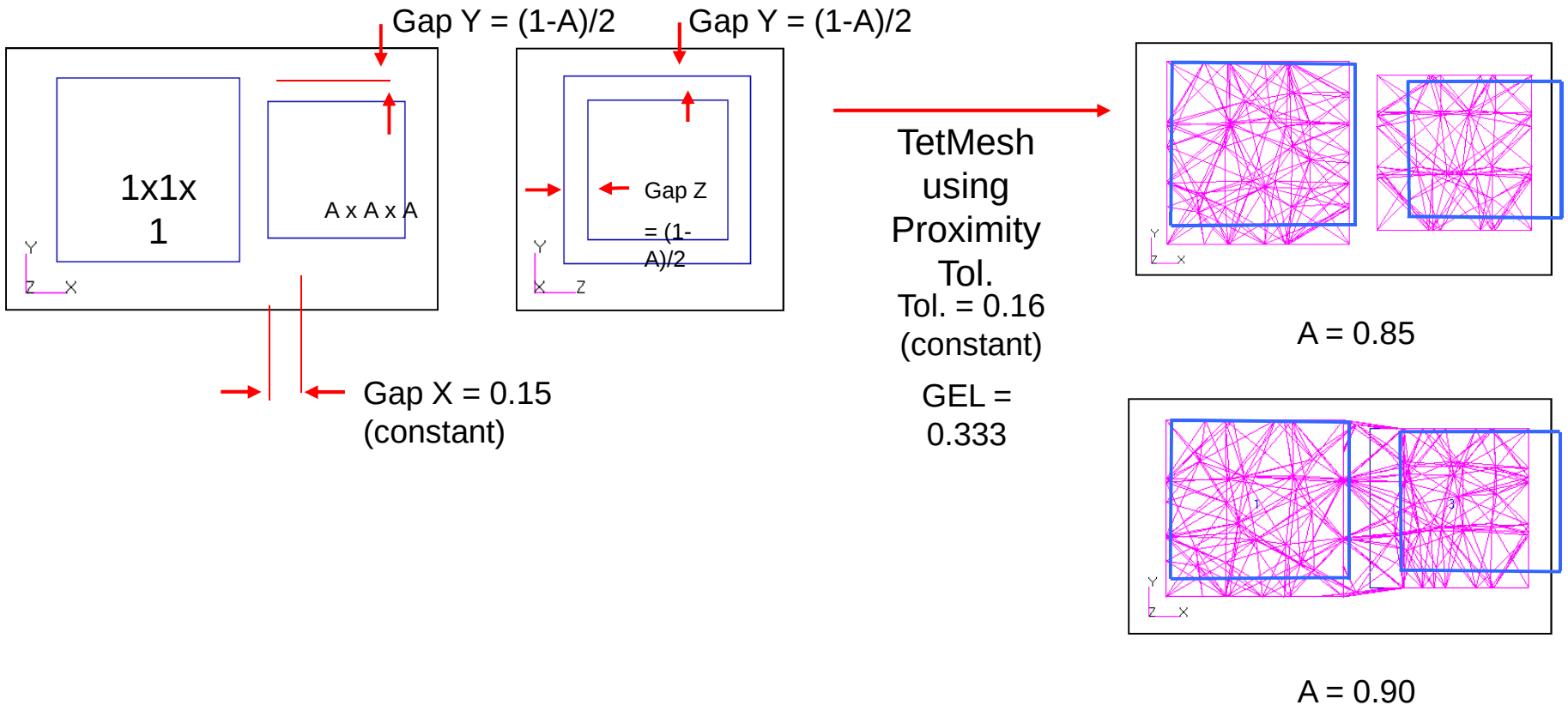


* $# The proximity tolerance specified is a large percentage of the element size. Meshing success and quality may be degraded.

Copyright© 2012 MSC.Software Corporation

GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

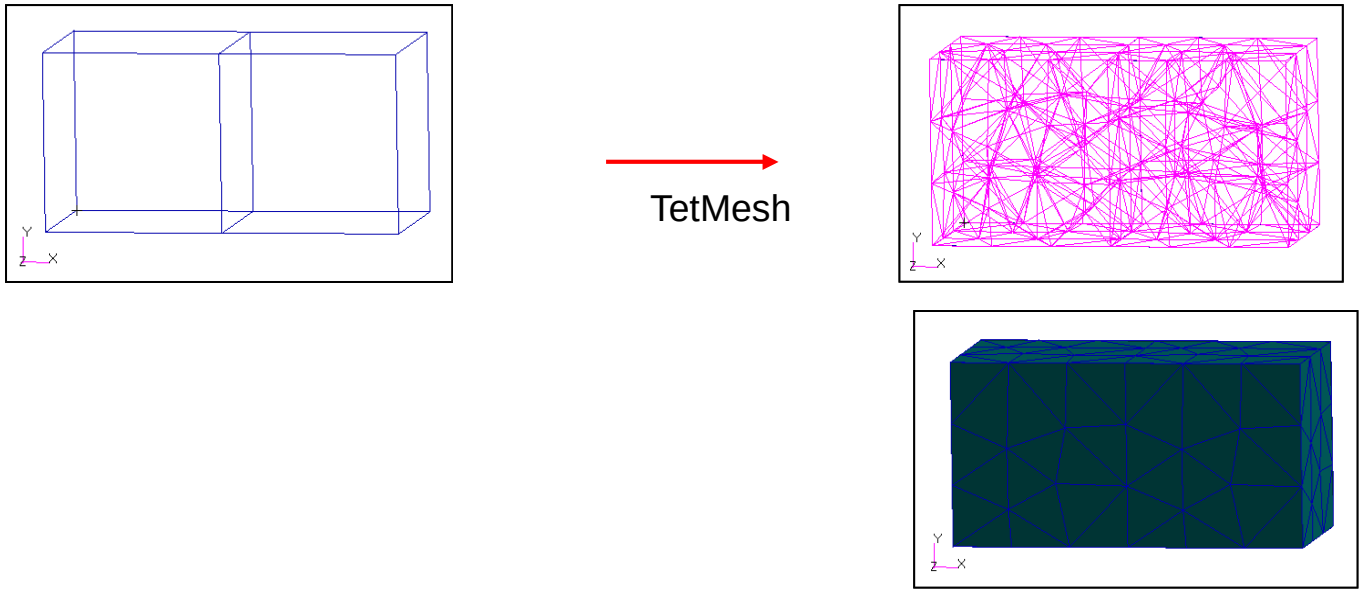
- **Create a TetMesh across a gap**
  - Vary size of second solid, keeping the gap size and proximity tolerance constant



Copyright© 2012 MSC.Software Corporation

GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

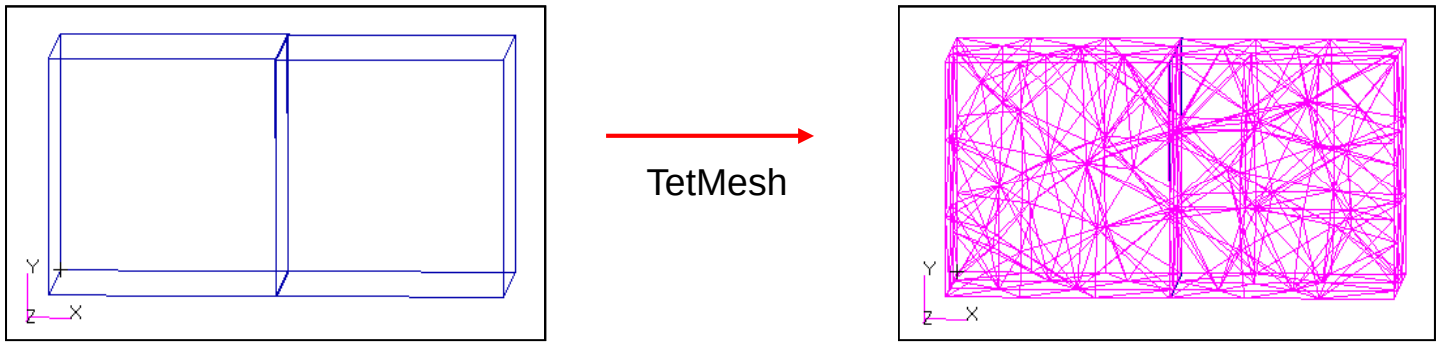
- **Create a TetMesh through an increasing intersection for adjacent parasolid solids**
  - No intersection; solid faces within tolerance of each other
- **Create a TetMesh through an increasing intersection for adjacent parasolid solids**
  - No intersection; solid faces within tolerance of each other



Copyright© 2012 MSC.Software Corporation

GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

- **Create a TetMesh through an intersection**
  - Intersection is small enough for tet mesh to be created, but some nodes are not associated to either solid.



$# Warning :

$# Bodies are intersecting slightly, meshing may result in unassociated nodes

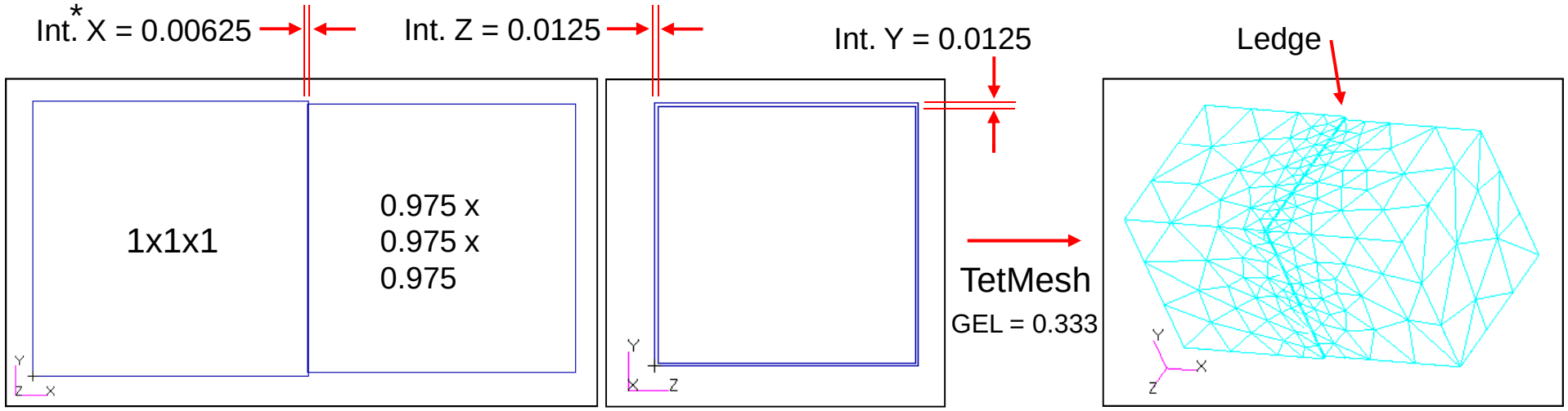
$# Warning reported from application FEM

$# The following node ids have not been associated to the geometry : 1 7 33 34 …

Copyright© 2012 MSC.Software Corporation

GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

- **Create a TetMesh through an intersection**
  - Intersection is small, but > global model tolerance of 0.005.
  - Proximity Tol. not used for intersecting solids.



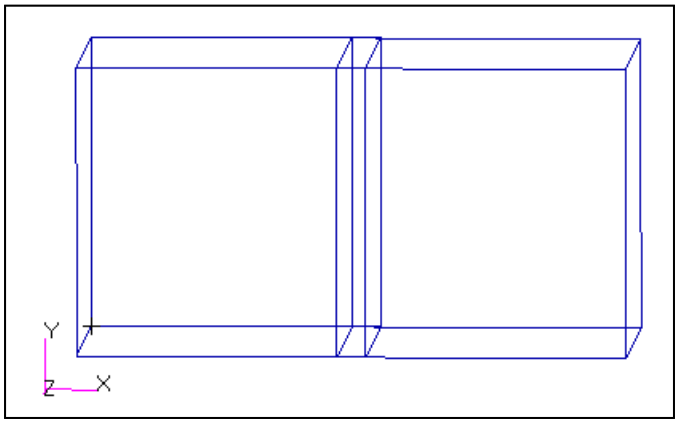
* Note: “Int.” stands for intersection

  - The occurrence of a “ledge” is very undesirable !
  - The capability of the TetMesher to deal with intersecting solids is only useful for solids that intersect slightly (on the order of the tolerance).

Copyright© 2012 MSC.Software Corporation

GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

- **Create a TetMesh through an intersection**
  - Intersection is too large for a tet mesh to be created



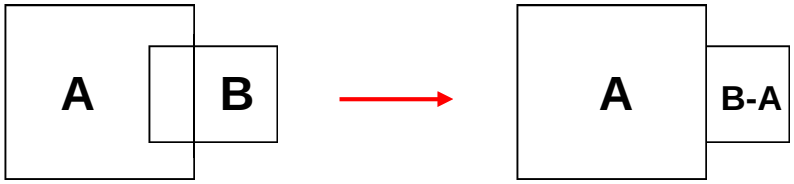
$# Warning :

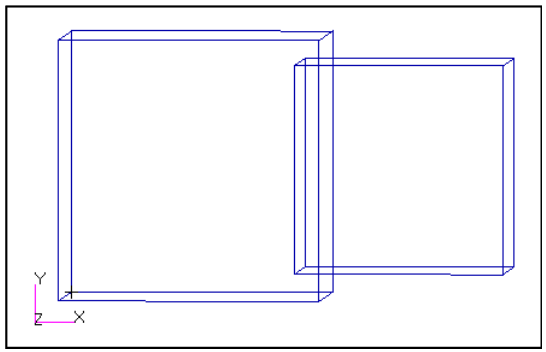
$# Solids intersect, assembly meshing aborted

Copyright© 2012 MSC.Software Corporation

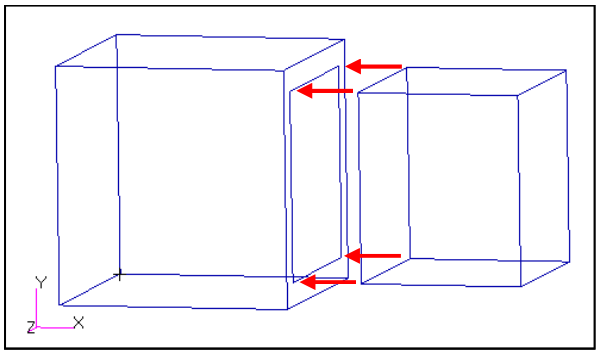
GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

- **Meshing through a large intersection for adjacent parasolid solids**
  - Difference between the two solids at the interface is exaggerated. Such differences should not be created during geometry creation.

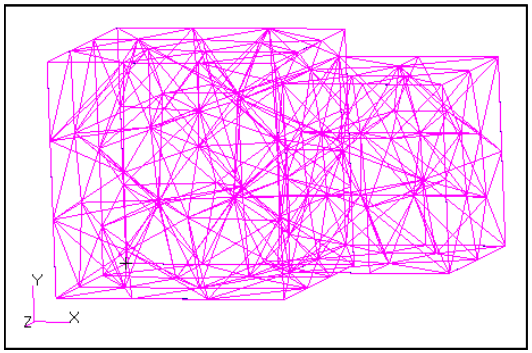




Before TetMesh
Intersection of adjacent solids



Edit Solids
Boolean subtraction, then imprint of adjacent solids

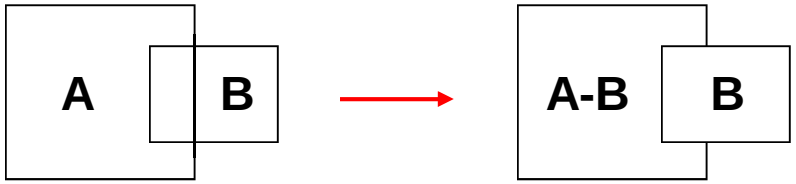


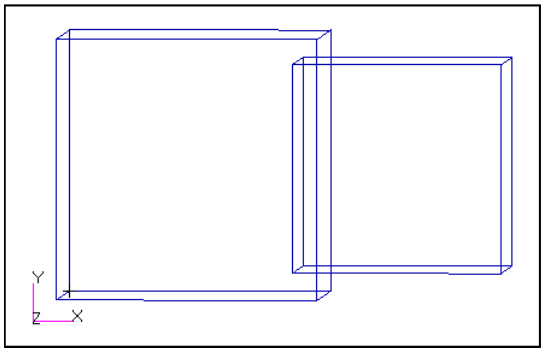
After TetMesh
Tets created along new edges

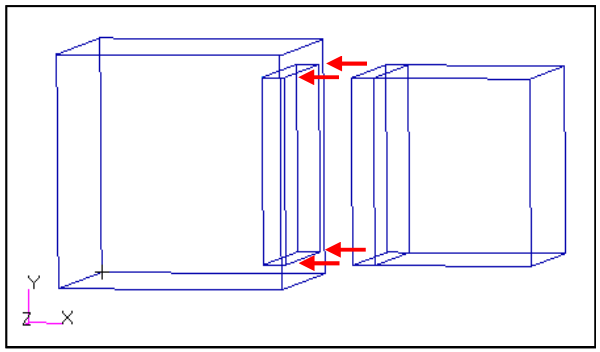
Copyright© 2012 MSC.Software Corporation

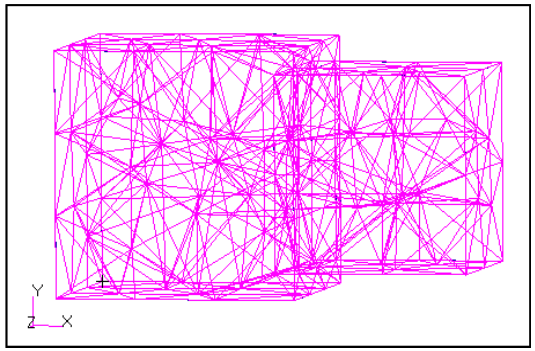
GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

- **Meshing through a large intersection for adjacent parasolid solids**
  - Difference between the two solids at the interface is exaggerated. Such differences should not be created during geometry creation.









Before TetMesh
Intersection of adjacent solids

Edit Solids
Boolean subtraction, then imprint of adjacent solids

After TetMesh
Tets created along new edges

Copyright© 2012 MSC.Software Corporation

GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

- **Guidelines for using the parasolid assembly TetMesher Proximity Tolerance parameter:**
  - While TetMeshing the preceding simple geometry it is observed that
    - At best, the proximity tolerance will have to be set to a value slightly greater than the size of a gap at the interface between adjacent solids. If the gap at the interface is three dimensional, which is likely, the tolerance parameter may have to be set even higher. Bridging gaps with tet elements using TetMesh with a large proximity tolerance, that is, large fraction of the element size, can degrade the quality of the mesh.
    - Large gaps can be TetMeshed using proximity tolerance, but the resulting mesh may be outside the region/domain of where the geometry might be. Thus, it may not be possible to obtain an accurate estimate of stress at some interfaces.
    - If the amount of an intersection > the tolerance, even though a TetMesh may be obtained, there may be a very undesired “ledge” (sliver solid face) with sliver-shaped tet elements. This will result in, at best, a greater mesh density near the interface.

Copyright© 2012 MSC.Software Corporation

GAPS AND INTERSECTIONS IN ASSEMBLY MESHING

- **Guidelines for using the parasolid assembly TetMesher Proximity Tolerance parameter**
  - On TetMeshing the simple geometry, it is observed that
    - If an intersection is too large, the TetMesher will fail to create a tet mesh.
    - If an intersection is slightly large, a tet mesh will be created, but some tet element nodes will not be associated to either solid at the interface.
  - Conclusions
    - The TetMesh parameter Proximity Tol. can be used to enable the creation of a tet mesh that is continuous across a gap.
    - It is best to limit the value of the parameter to the order of the geometry tolerance (global model tolerance in Patran), e.g. 2 x 0.005 for a GMT = 0.005
    - It is best to create the geometric solids, including their position relative to each other, within the selected tolerance, so there will be no need to change the value of Proximity Tol. from that of its default value, the global model tolerance.

Copyright© 2012 MSC.Software Corporation

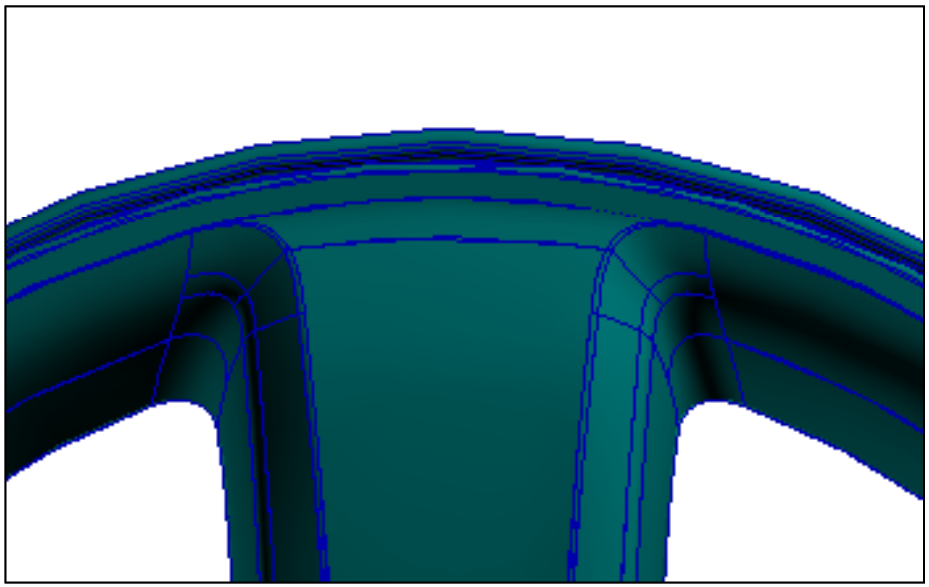
SLIVER/DEGENERATE SOLID FACES FOR PARASOLID ASSEMBLIES

- **The solids of parasolid assemblies can have faces which are degenerate or nearly degenerate.**
- **Solids may be TetMeshable individually, but the meshing may fail when the solids are meshed as an assembly.**
- **An approach to avoid degenerate mesh**
  - TetMesh
  - Determine face(s) for which meshing failed
  - Create surface for face
  - Determine surface / face attributes
  - Mesh surface / face
  - Modify the face using CAD

Copyright© 2012 MSC.Software Corporation

CASE STUDY: TETMESH WITH DEGENERATE SOLID FACE

- **There are three parasolid solids in the assembly**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: TETMESH WITH DEGENERATE SOLID FACE

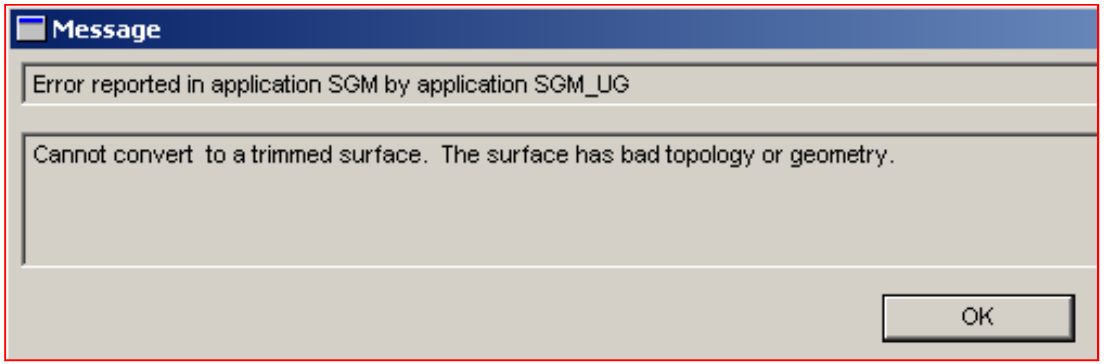
- **TetMesh the assembly**
  - Mesh the three solids simultaneously.
- **Some contents of the session file**

```
$# Completed Meshing the Vertices...
$# Completed Meshing the Edges...
$# Completed Meshing the Edges...
$# Completed Meshing the Edges...
$# Completed Meshing the Edges...
$# Warning :
$#     Unable to mesh Solid 3.313
```

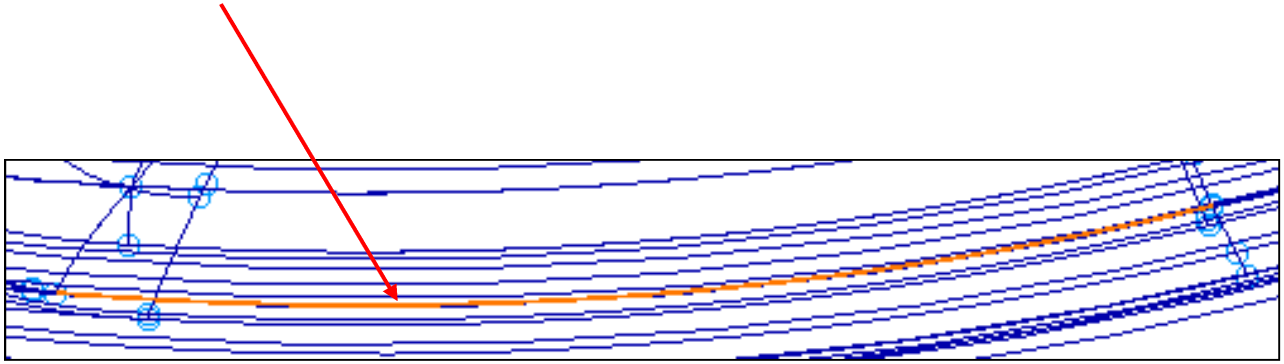
- **Create a surface for the solid face Solid 3.313**
  - Under the *Geometry* tab, set *Action, Object,* and *Method* as **Create, Surface,** and **Extract,** respectively

Copyright© 2012 MSC.Software Corporation

CASE STUDY: TETMESH WITH DEGENERATE SOLID FACE



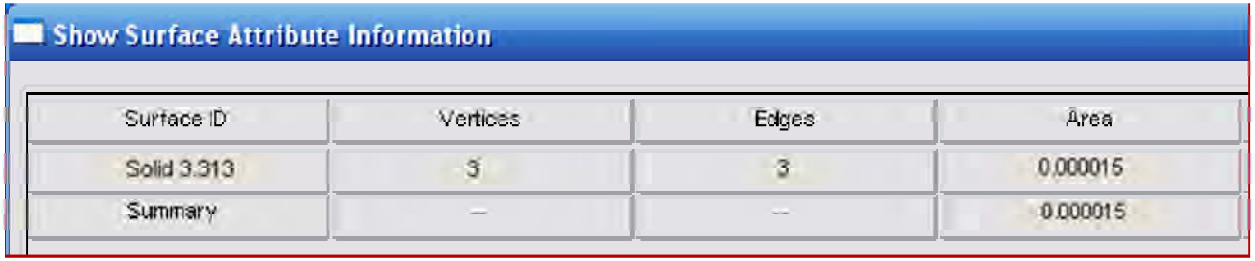
- **The solid face is shown below:**



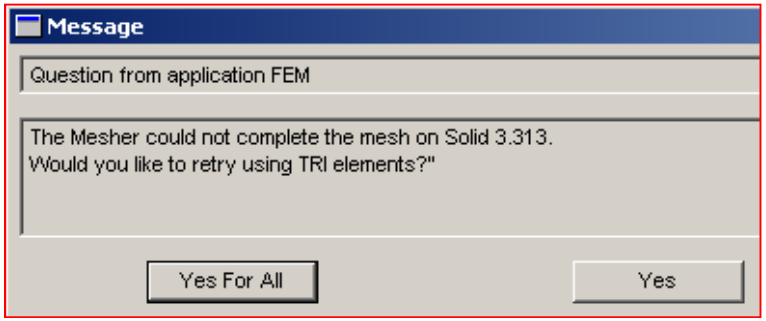
Copyright© 2012 MSC.Software Corporation

CASE STUDY: TETMESH WITH DEGENERATE SOLID FACE

- **The face attributes are**



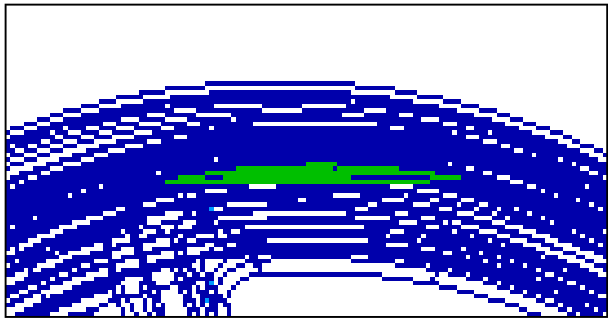
- **Notice the area of the face is small 0.000015**
- **Paver mesh the face**
  - Use the Patran specified value of the global edge length, 6.83046e-005

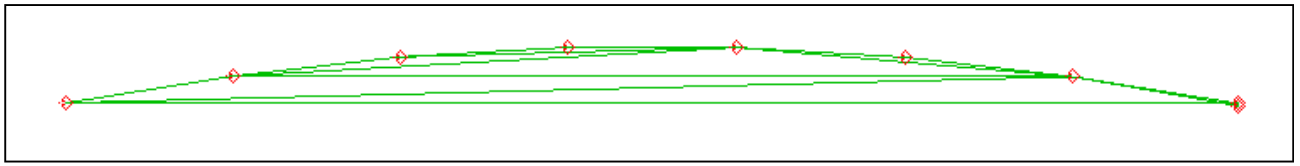


Copyright© 2012 MSC.Software Corporation

CASE STUDY: TETMESH WITH DEGENERATE SOLID FACE

- **The tri meshed face is shown as follows:**





Copyright© 2012 MSC.Software Corporation

CASE STUDY: TETMESH WITH DEGENERATE SOLID FACE

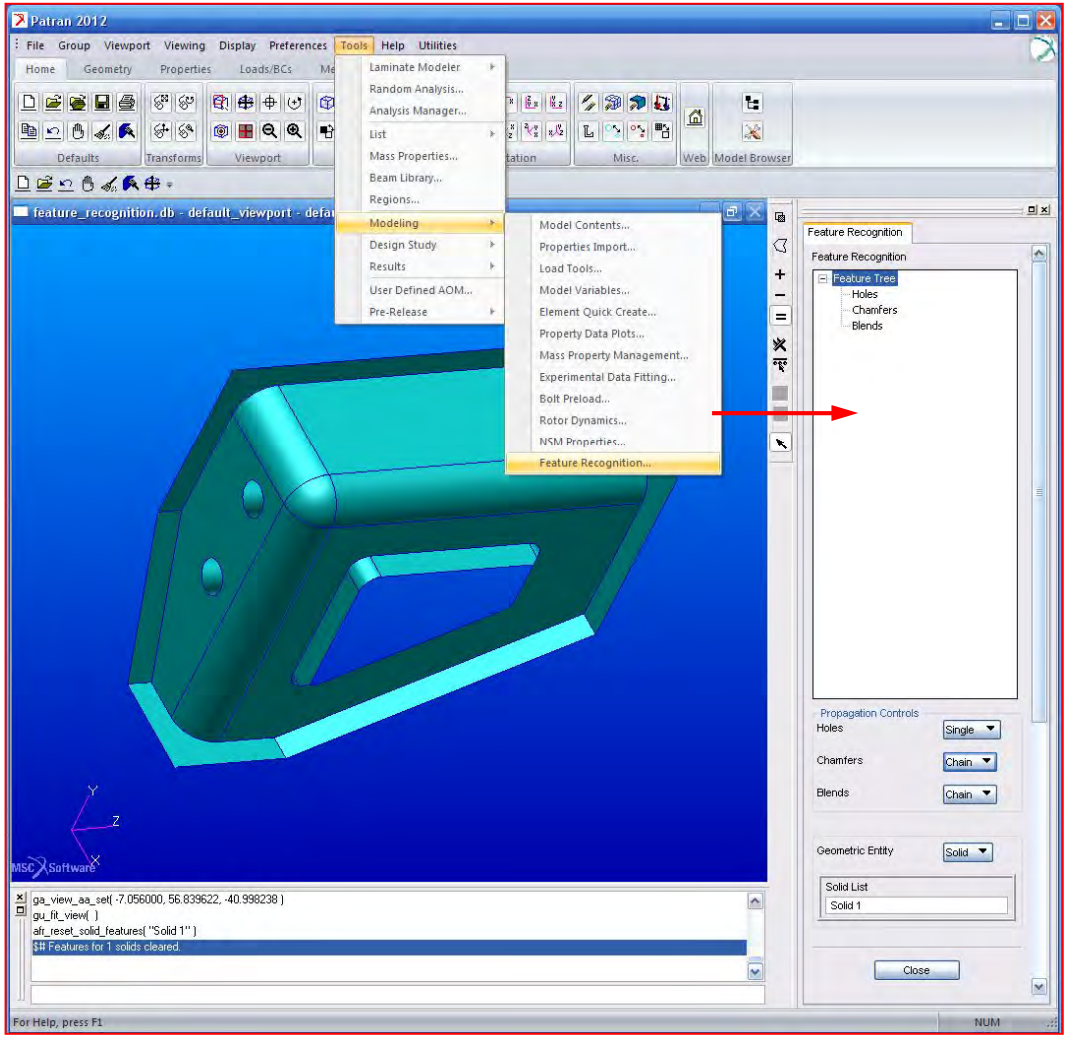
- **The session file created by Paver Quad meshing**

```
fem_create_mesh_surf_4( "Paver", 49680, "Solid 3.313", 4, ["6.83046e-005",  @
"0.1", "0.2", "1.0"], "Quad4", "#", "#", "Coord 0", "Coord 0",  @
fem_create_mesh_surfa_num_nodes, fem_create_mesh_surfa_num_elems,  @
fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )
$# Warning reported from application MSH
$#     Failed to resolve edge crossings.
$# Warning reported from application MSH
$#     Trying to create p-element mesh.
$# Warning reported from application MSH
$#     Zero edge length found in crossing.
$# Warning reported from application MSH
$#     Trying to mesh in parametric space.
$# Question from application FEM
$#     The Mesher could not complete the mesh on Solid 3.313.
$# Would you like to retry using TRI elements?"
$? YES 2008020
$# Warning reported from application MSH
$#     Unknown fatal failure in the paver.
$# Warning reported from application MSH
$#     Trying to create p-element mesh.
$# Warning reported from application MSH
$#     Failed to resolve edge crossings.
$# Warning reported from application MSH
$#     Trying to mesh in parametric space.
$# 9 nodes and 7 elements created for Solid 3.313.
$# === 9 nodes created.  IDs  = 135269:135277.
$# === 7 elements created.  IDs  = 72240:72246.
```


Copyright© 2012 MSC.Software Corporation

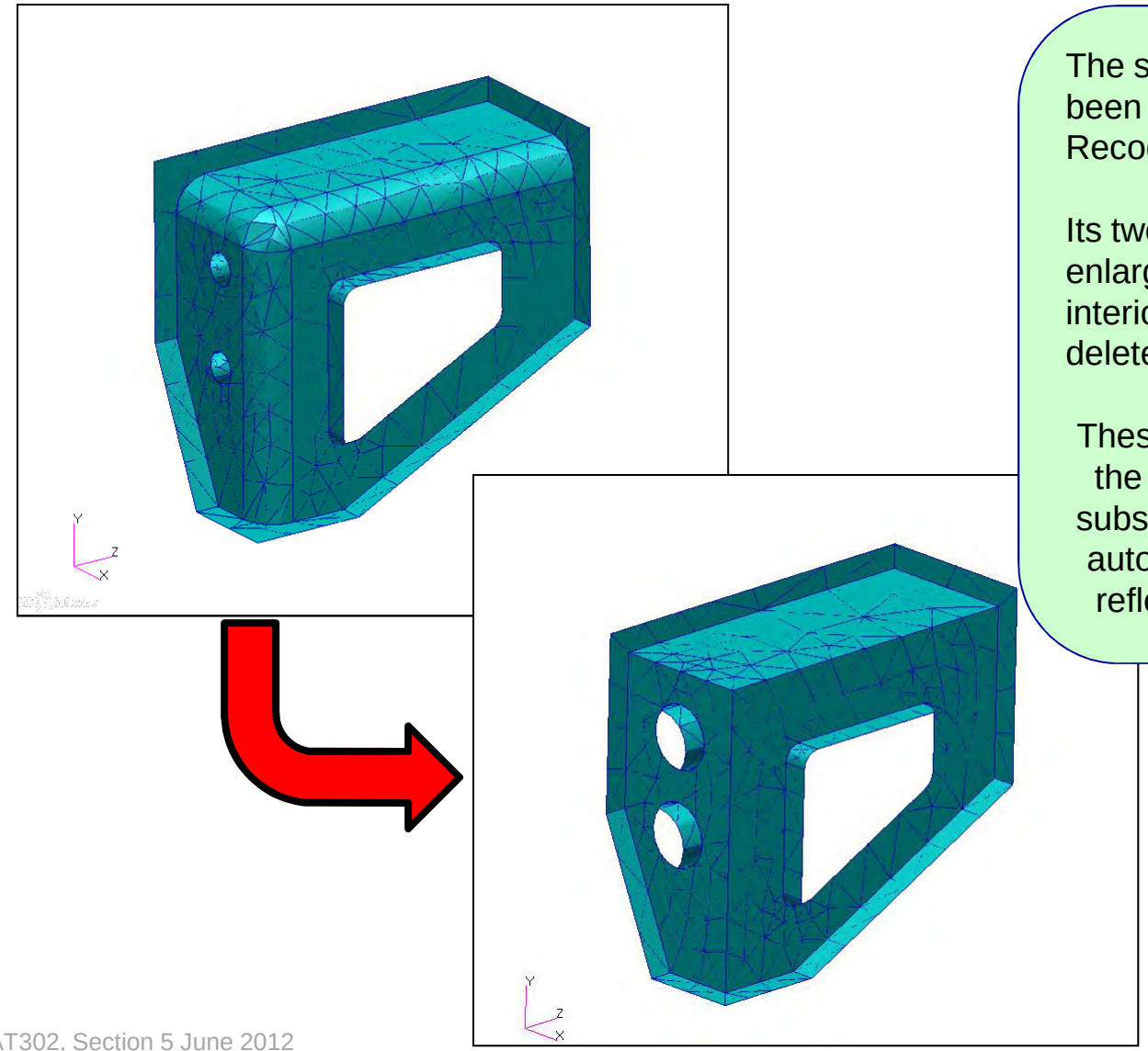
FEATURE RECOGNITION

- **Tools/Modeling/Feature Recognition allows the user to quicky eid or delete solid geometry features such as:**
  - Holes
  - Chamfers
  - Blends
- **The tools will only work on Parasolid-type solids**
- **Changes in the geometry will also be reflected in an auto-update of the mesh**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: FEATURE RECOGNITION



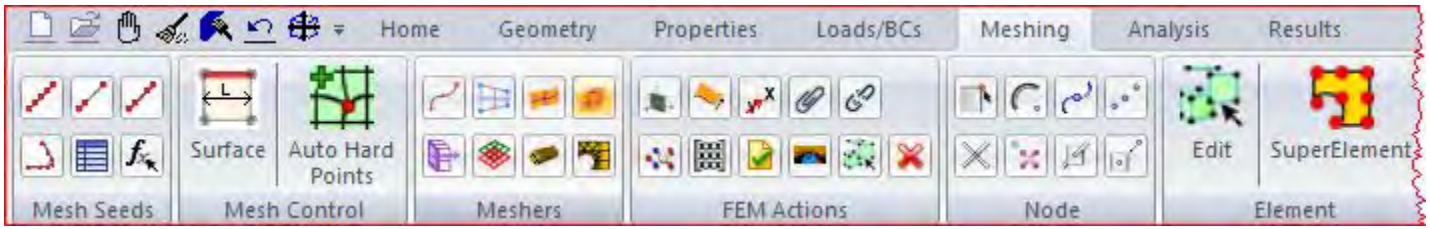
The solid shown on the left has been edited using the Feature Recognition Tool.

Its two round holes has been enlarged and the blend in the interior of the solid has been deleted.

These changes are instituted on the solid; but, as shown in the subsequent figure, the mesh will automatically update as well to reflect the modified geometry.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FEATURE RECOGNITION



Mesh the solid:

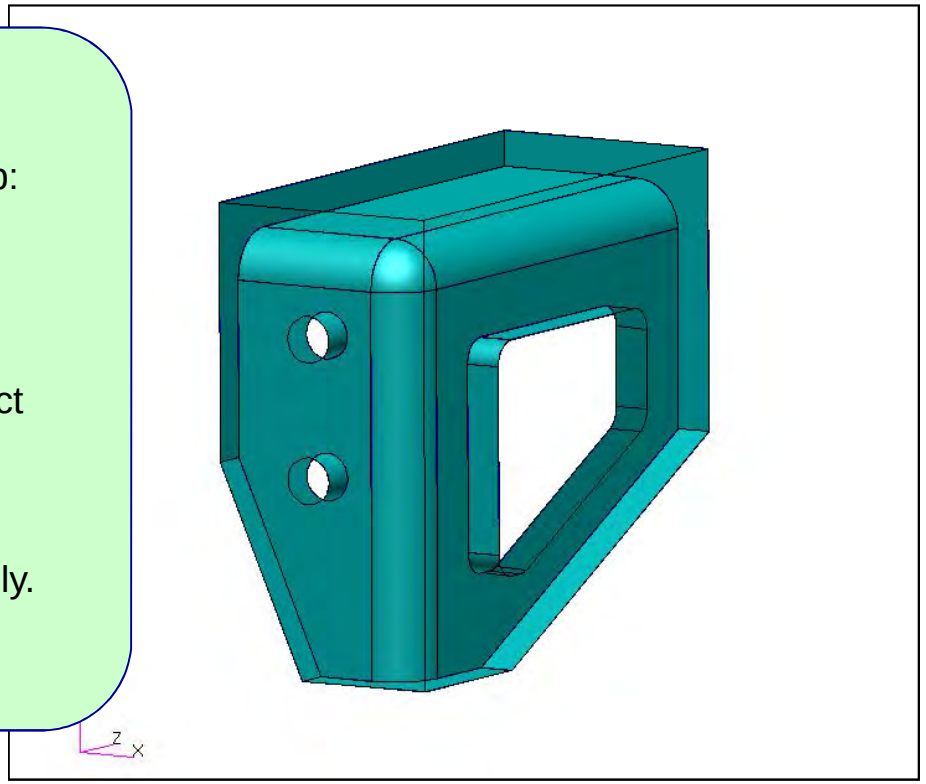
Under the Meshing tab:
Create / Mesh / Solid

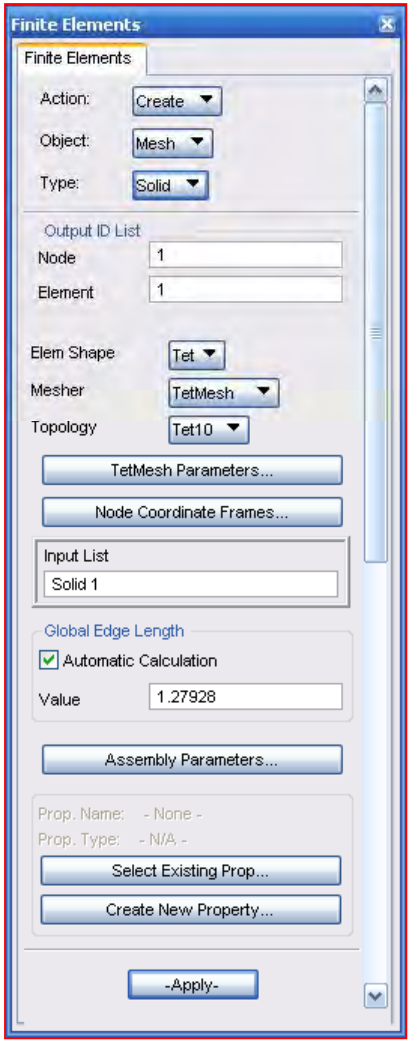
Use Tet10 elements.

Under *Input List*, select Solid 1.

Allow the *GEL* to be calculated automatically.

Click Apply.





Copyright© 2012 MSC.Software Corporation

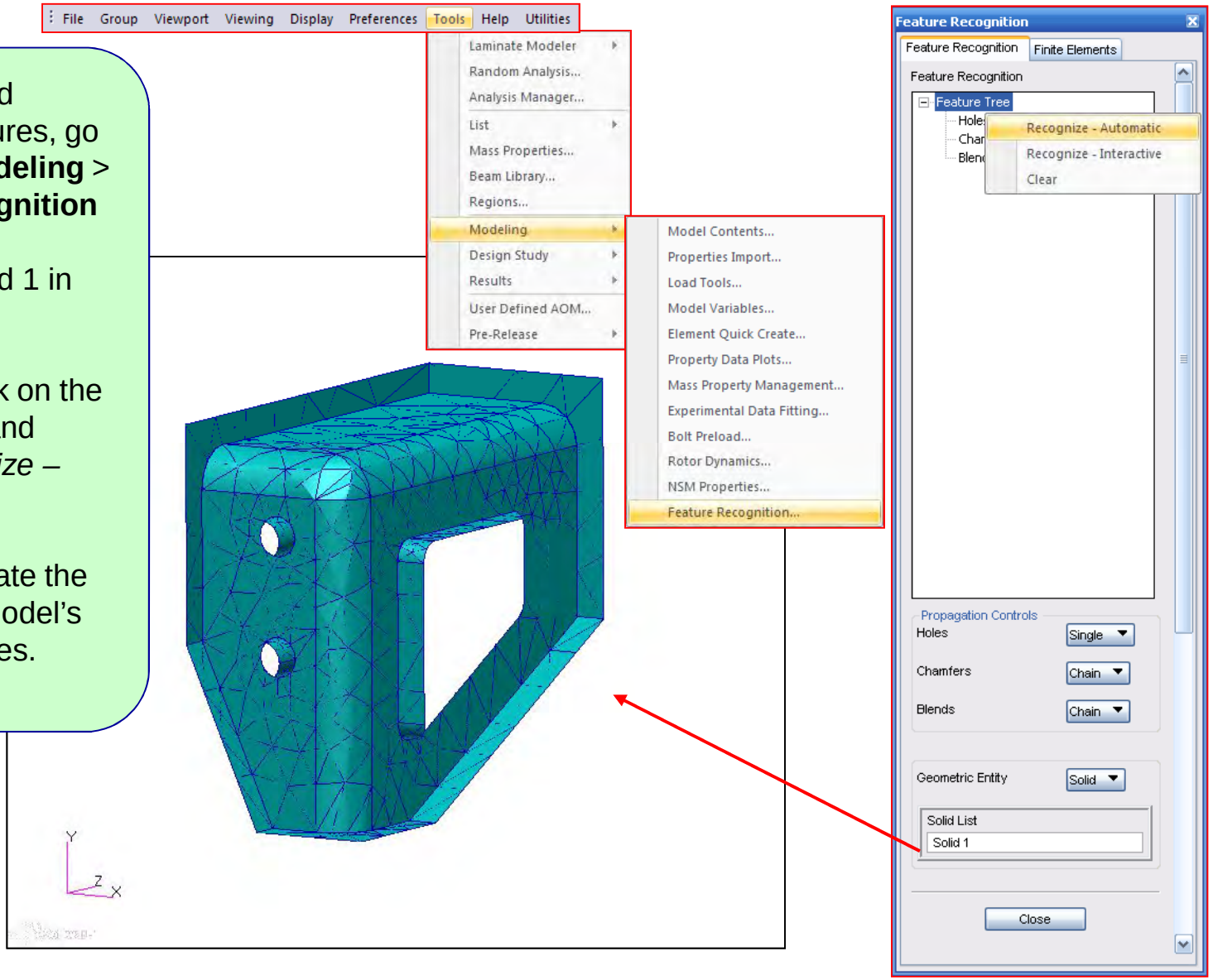
CASE STUDY: FEATURE RECOGNITION

To edit the solid geometry features, go to **Tools** > **Modeling** > **Feature Recognition**

Select the Solid 1 in the *Solid List*.

Then right-click on the *Feature Tree* and select *Recognize – Automatic.*

This will populate the tree with the model's editable features.

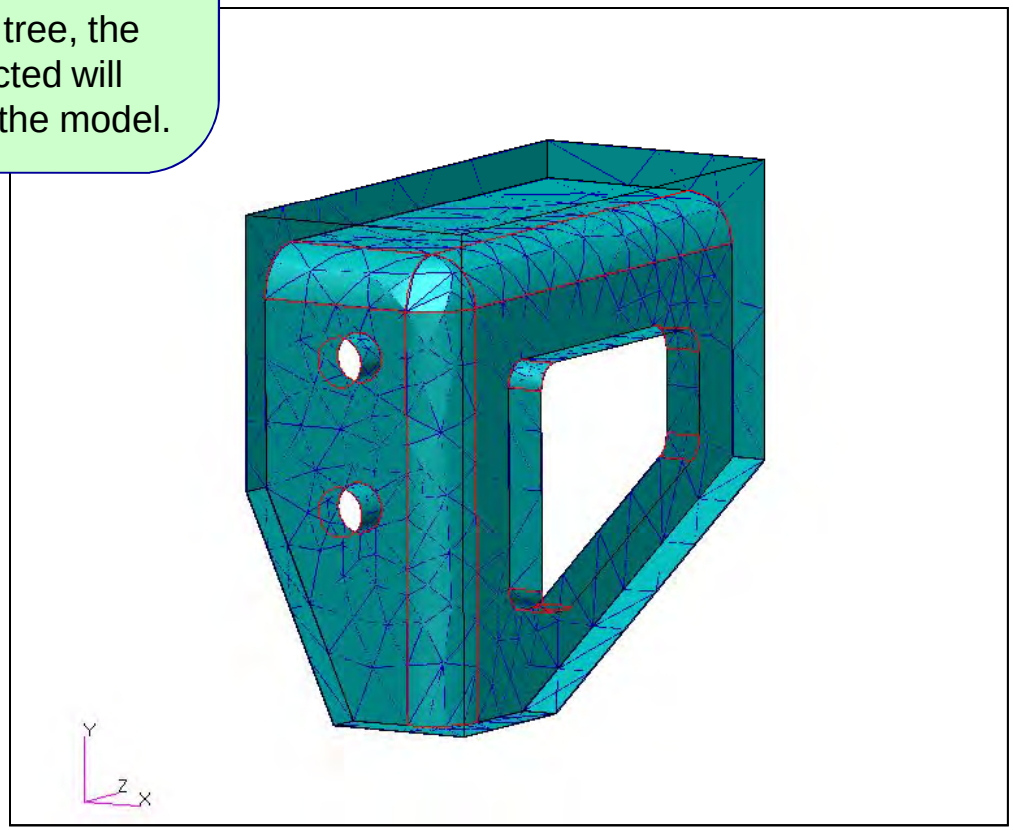


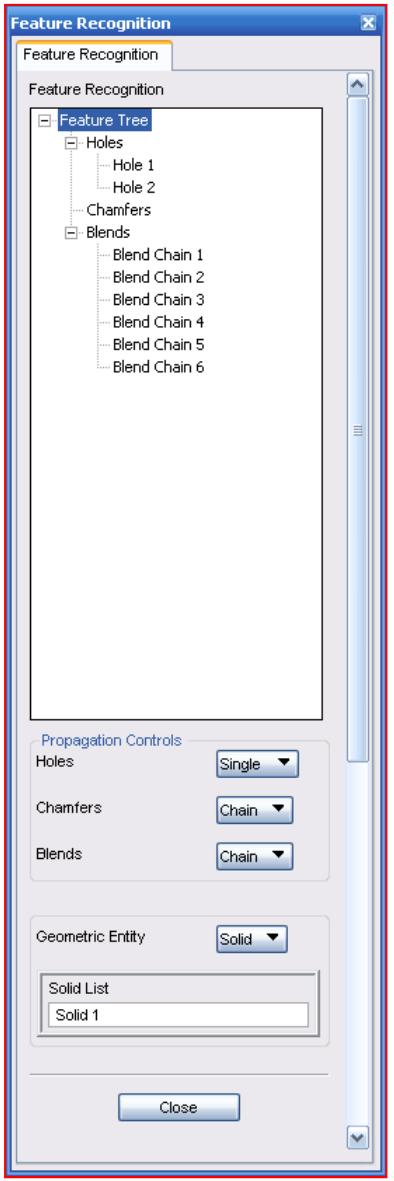
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FEATURE RECOGNITION

The features that are available to be edited or deleted are now listed in the tree.

As you click on each entry on the tree, the feature selected will highlight on the model.





Copyright© 2012 MSC.Software Corporation

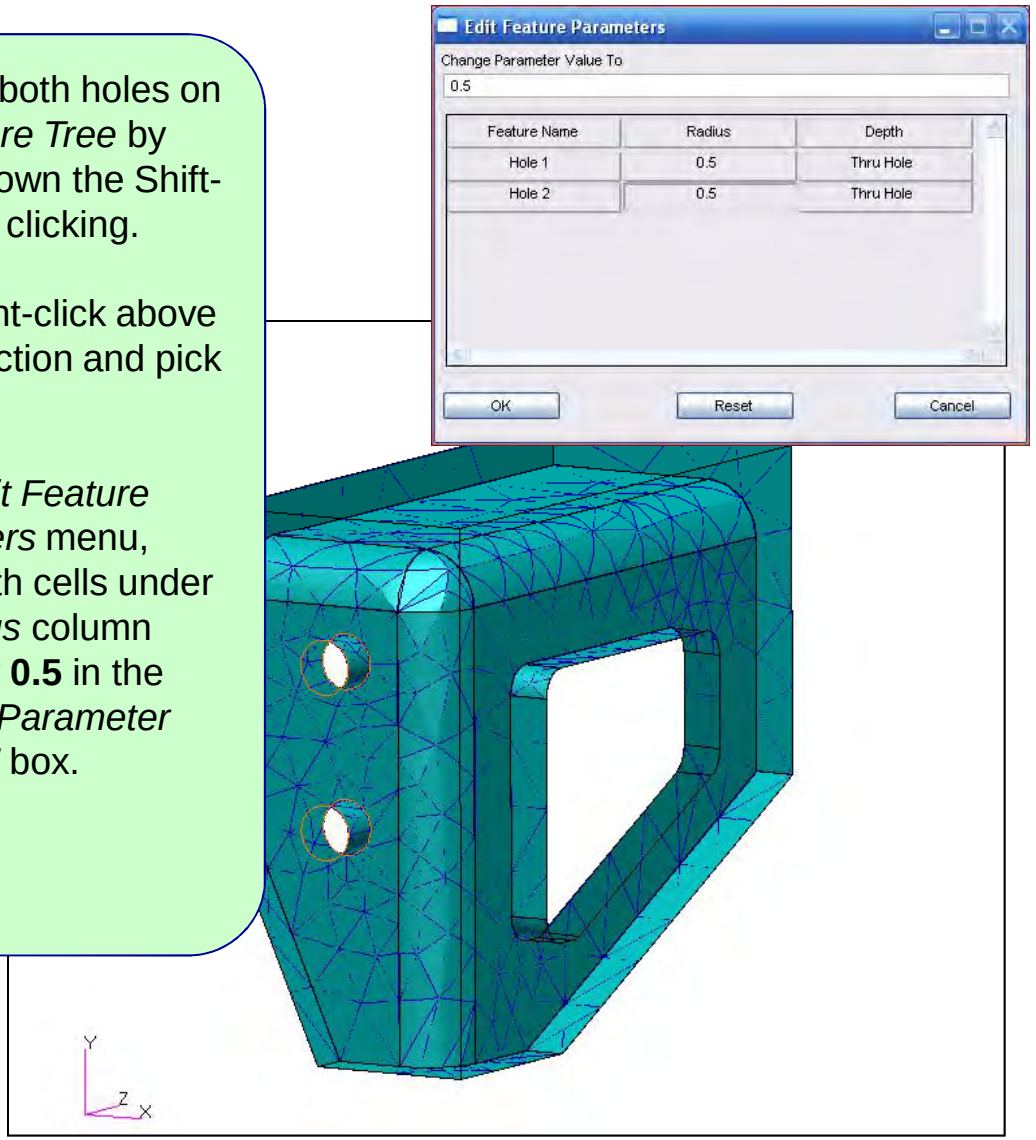
CASE STUDY: FEATURE RECOGNITION

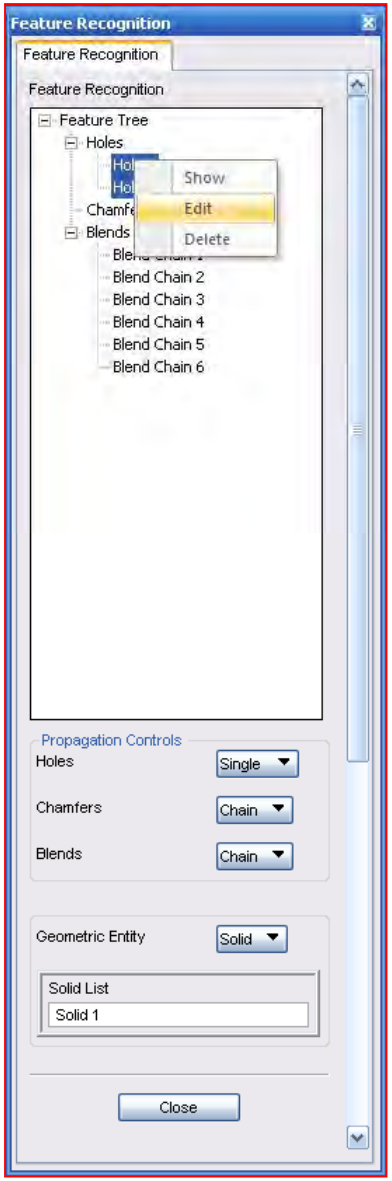
Highlight both holes on the *Feature Tree* by holding down the Shift-key while clicking.

Then, right-click above your selection and pick Edit.

In the *Edit Feature Parameters* menu, select both cells under the *Radius* column and enter **0.5** in the *"Change Parameter Value To"* box.

Click **OK**.



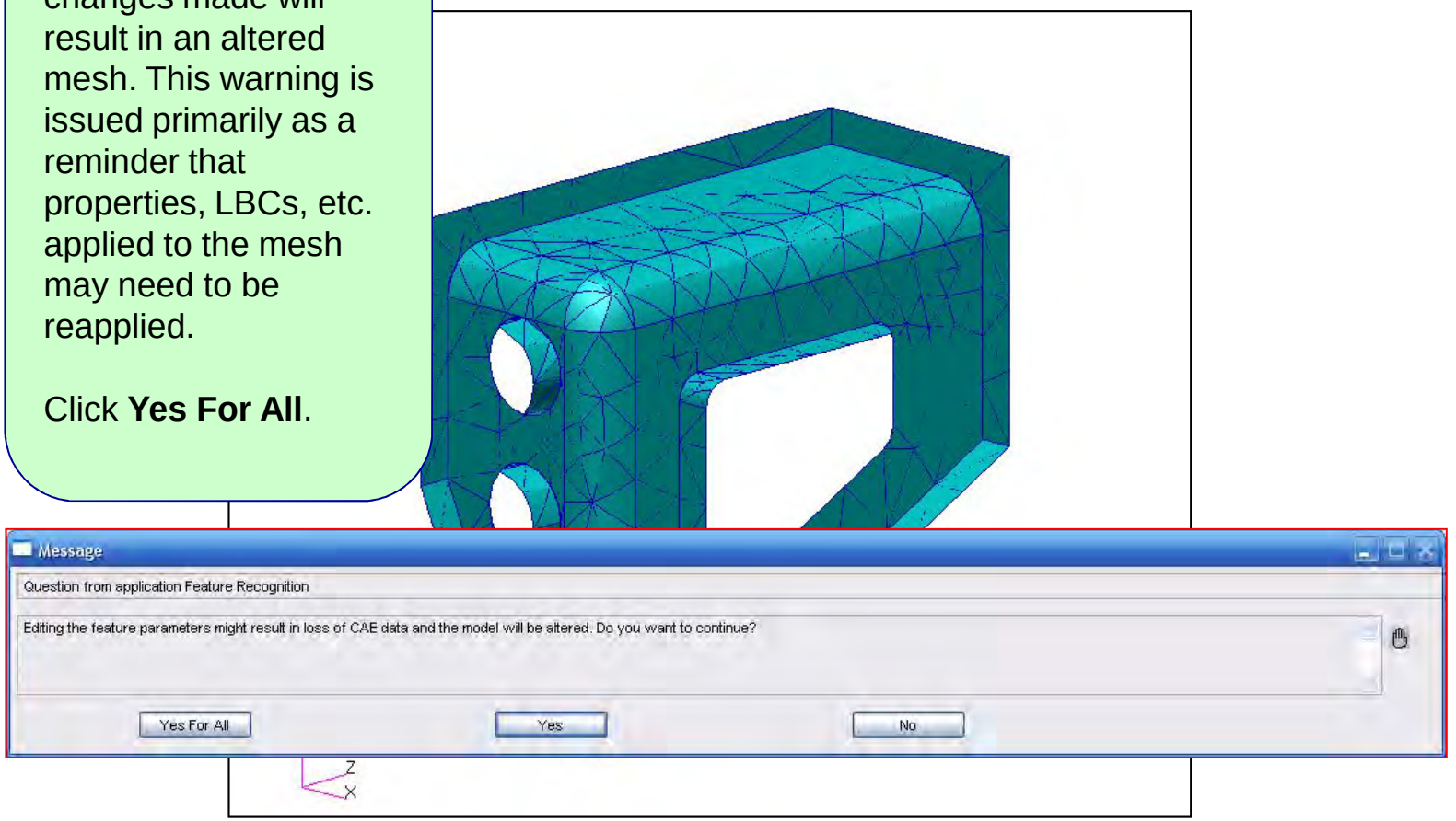


Copyright© 2012 MSC.Software Corporation

CASE STUDY: FEATURE RECOGNITION

Editing the feature will invoke a message informing you the changes made will result in an altered mesh. This warning is issued primarily as a reminder that properties, LBCs, etc. applied to the mesh may need to be reapplied.

Click **Yes For All**.



Copyright© 2012 MSC.Software Corporation

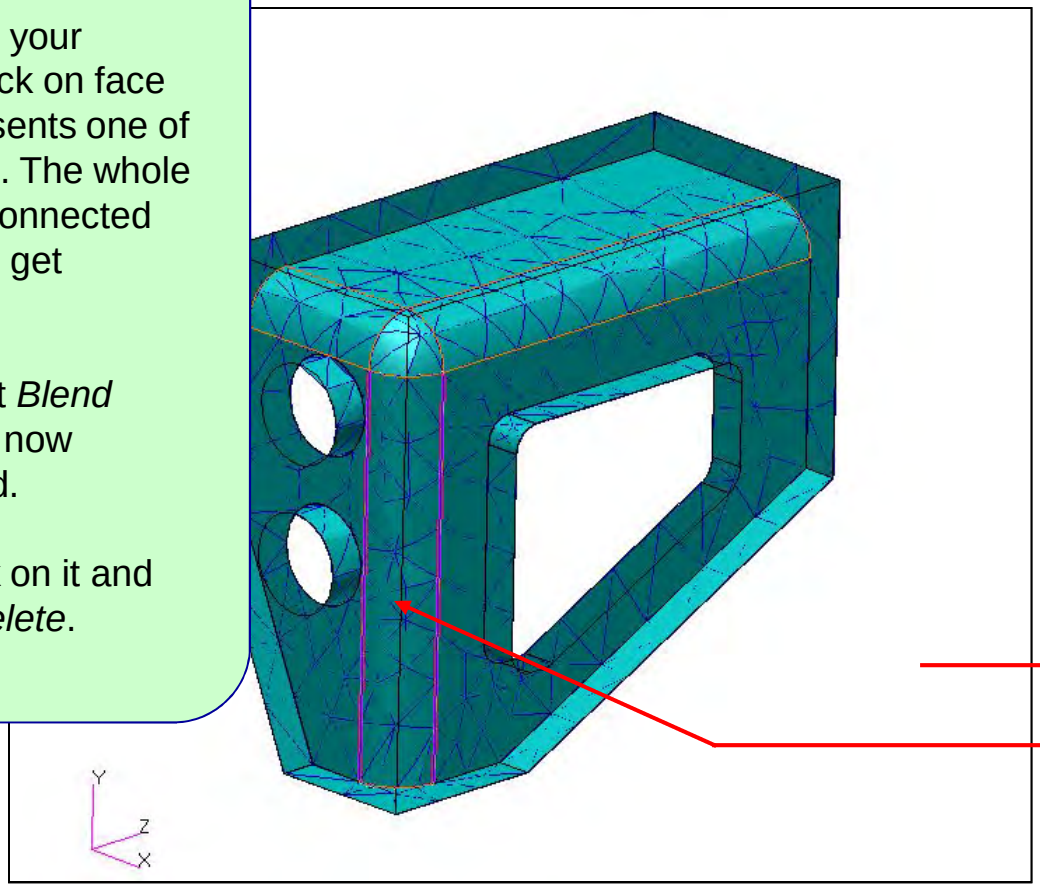
CASE STUDY: FEATURE RECOGNITION

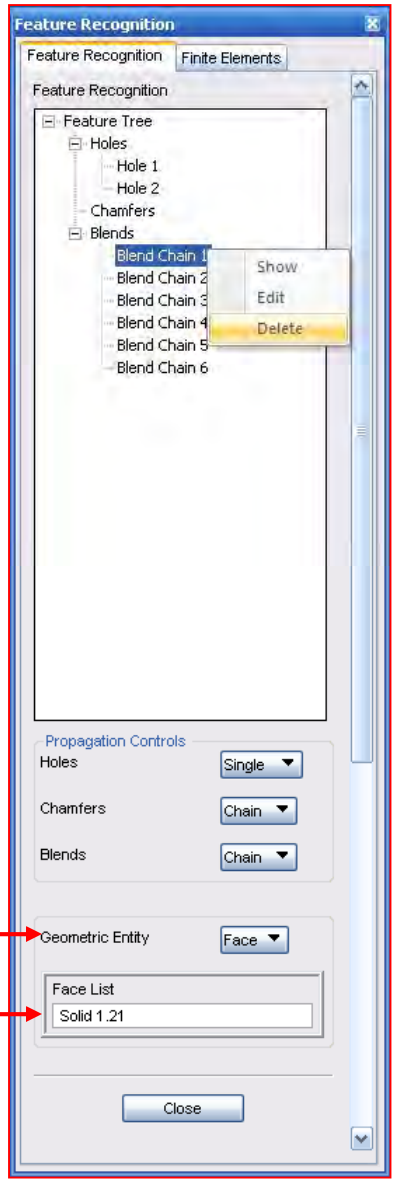
Change the *Geometric Entity* dropdown to Face.

Then, with your mouse, click on face that represents one of the blends. The whole series of connected blends will get selected.

Notice that *Blend Chain 1* is now highlighted.

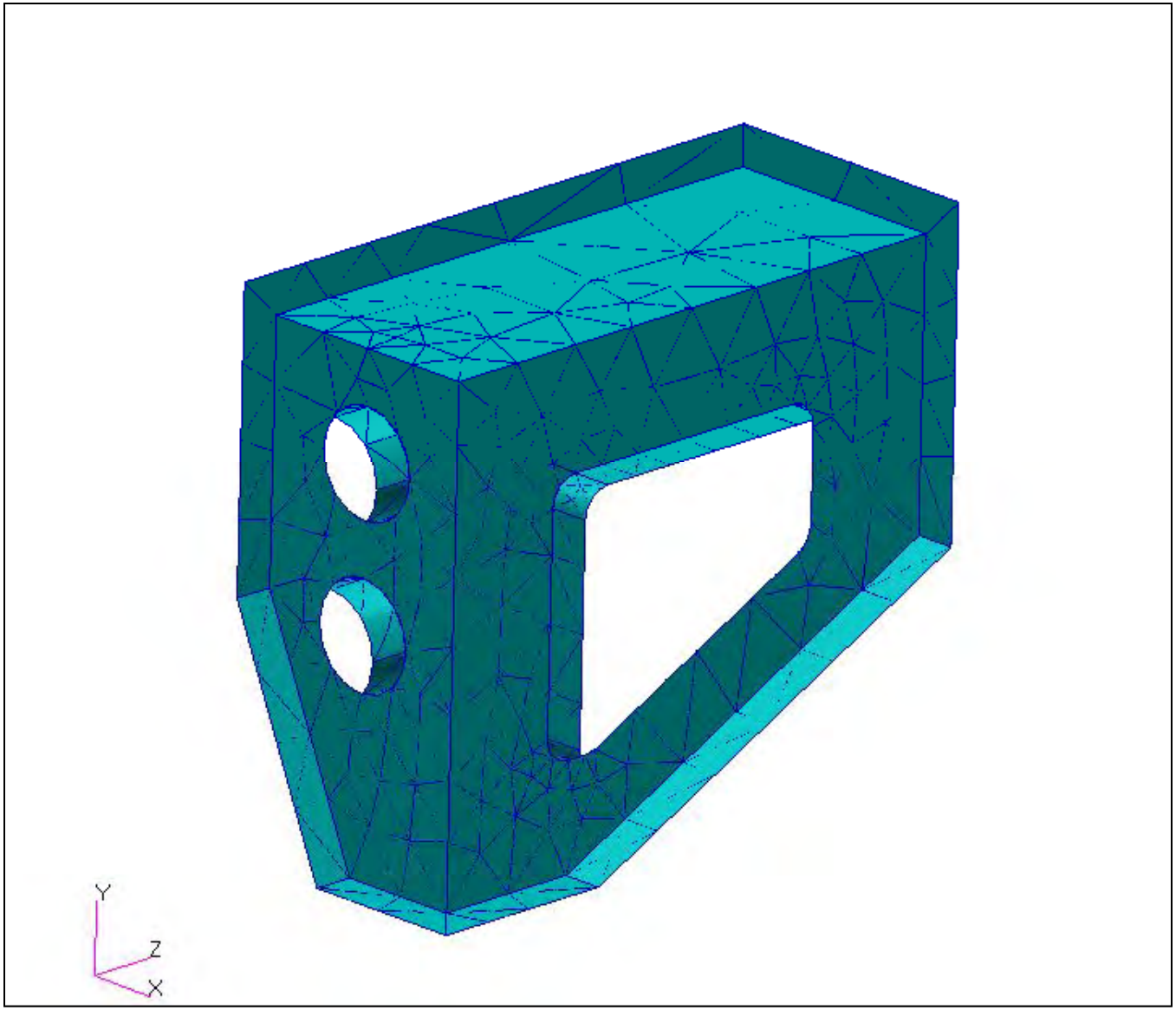
Right-click on it and choose *Delete*.





Copyright© 2012 MSC.Software Corporation

CASE STUDY: FEATURE RECOGNITION



Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

SECTION 6

CUSTOMIZATION: TAILORING THE WORK ENVIRONMENT

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

TOPICS DISCUSSED

- **Customization of the way work is done in Patran can help to minimize the time it takes to complete a task.**
- **There are several areas we will look at:**
  - Files
    - Patran template database, file to specify Patran session parameters, Patran session file, etc.
  - Command-line arguments
  - Environment variables
  - Using the GUI
    - Picking, Toolbar icons
  - Short-cuts
    - Key mapping
  - Changing the finite element solver preference
    - For example, Nastran -> MSC Marc

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

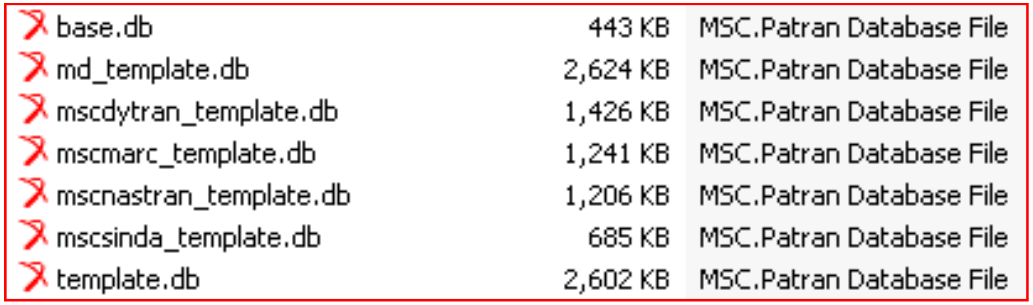
FILES

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

PATRAN TEMPLATE DATABASE

- **Every time a new database is created, a template database is copied to the startup directory and renamed to create the new database.**
- **Default template is template.db, but *any* Patran db can be used as a template.**
- **There are a number of other predefined templates in the Patran installation:**

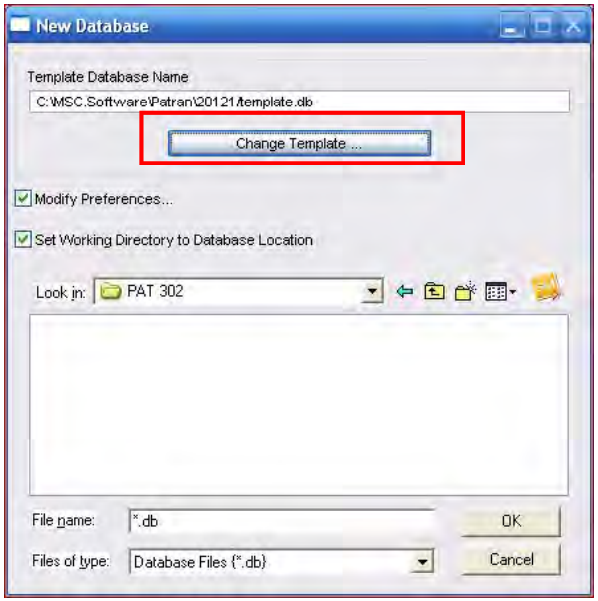


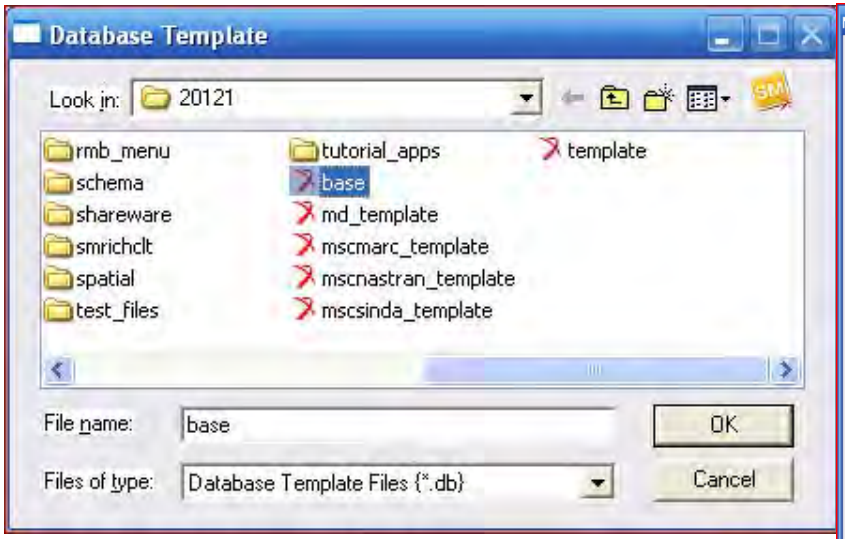
base.db	443 KB	MSC.Patran Database File
md_template.db	2,624 KB	MSC.Patran Database File
mscdytran_template.db	1,426 KB	MSC.Patran Database File
mscmarc_template.db	1,241 KB	MSC.Patran Database File
mscnastran_template.db	1,206 KB	MSC.Patran Database File
mscsinda_template.db	685 KB	MSC.Patran Database File
template.db	2,602 KB	MSC.Patran Database File

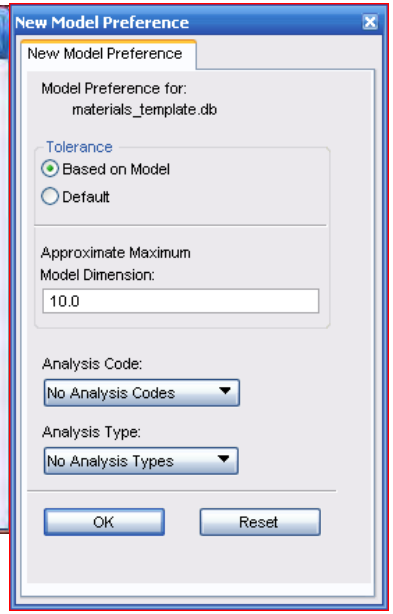
Copyright© 2012 MSC.Software Corporation

PATRAN TEMPLATE DATABASE

- **Example:**
  - base.db does not have any Patran solver preferences loaded in it.
  - Create a new database using **base.db**







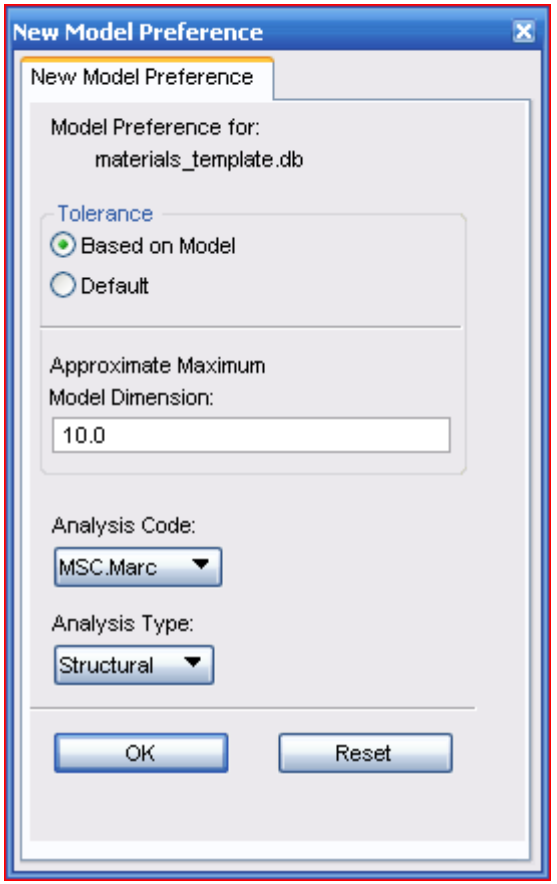
Copyright© 2012 MSC.Software Corporation

PATRAN TEMPLATE DATABASE

- **Load only the preference(s) that will be needed**
  - Load the MSC Marc preference

```
load_mscmarc()
```

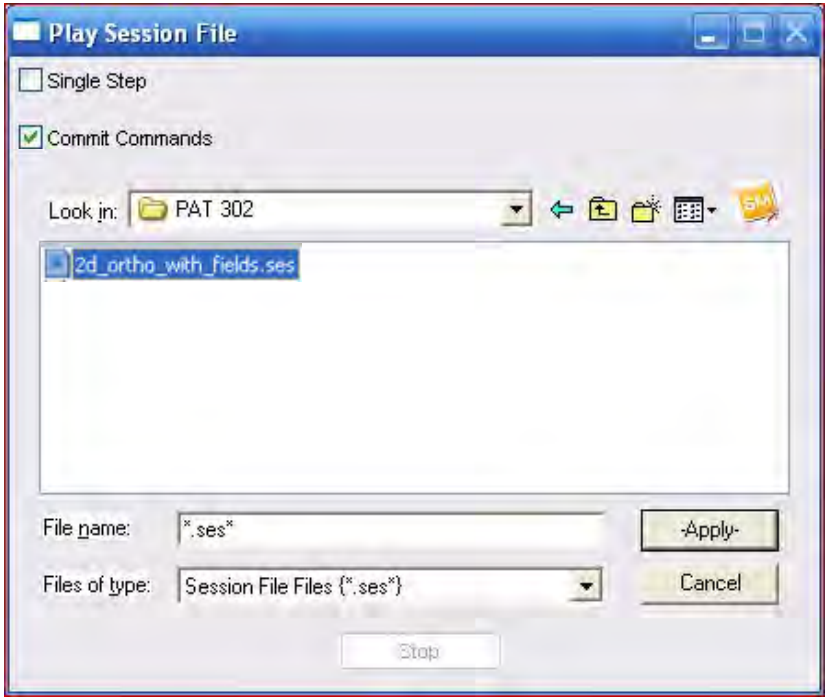
```
load_mscmarc()
$# Loading Analysis Code definitions for MSC.Marc...
$#  ---
$#  --- Checking [ Analysis Type ] data
$#  --- Checking [ Material Category ] data
$#  --- Checking [ Constitutive Model ] data
$#  --- Checking [ Material Option Category ] data
$#  --- Checking [ Material Option ] data
$#  --- Checking [ Material Property Word ] data
$#  ---
$#  --- Checking Page [ 1 ] of [ 1367 ]
$#  --- Checking Page [ 11 ] of [ 1367 ]
...
$#  ---
$#  --- No Errors found.
$#  ---
...
$# Load of Analysis Code definitions for MSC.Marc complete.
```



Copyright© 2012 MSC.Software Corporation

PATRAN TEMPLATE DATABASE

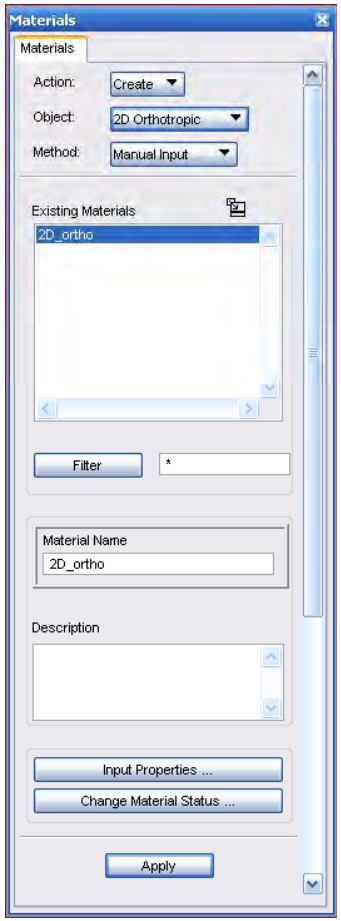
- **Continue by loading any special material properties**
  - Play a previously saved Patran session file that creates a 2D orthotropic material set and corresponding fields.
    - Pull down **File > Session > Play…**

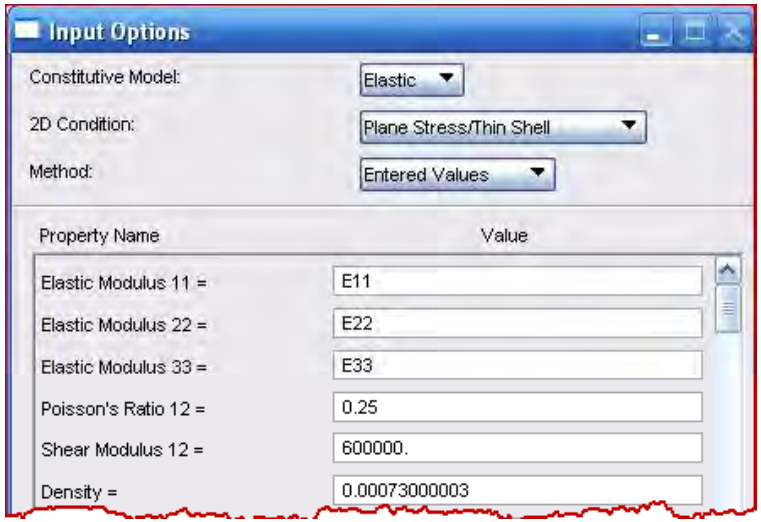


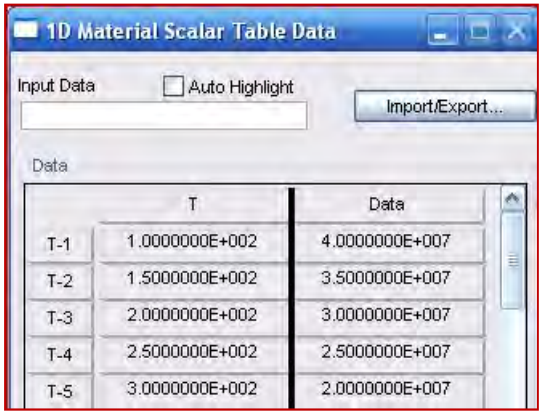
Copyright© 2012 MSC.Software Corporation

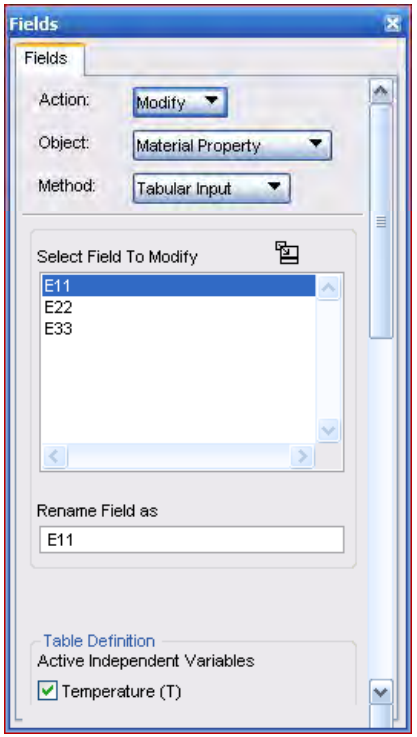
PATRAN TEMPLATE DATABASE

- **Material set and fields created by playing the session file:**









Copyright© 2012 MSC.Software Corporation

PATRAN TEMPLATE DATABASE

- **Quit Patran**
- **Copy the database to the desired location, e.g. the Patran installation directory.**
- **This database can be used in future as a template to create new databases.**

Copyright© 2012 MSC.Software Corporation

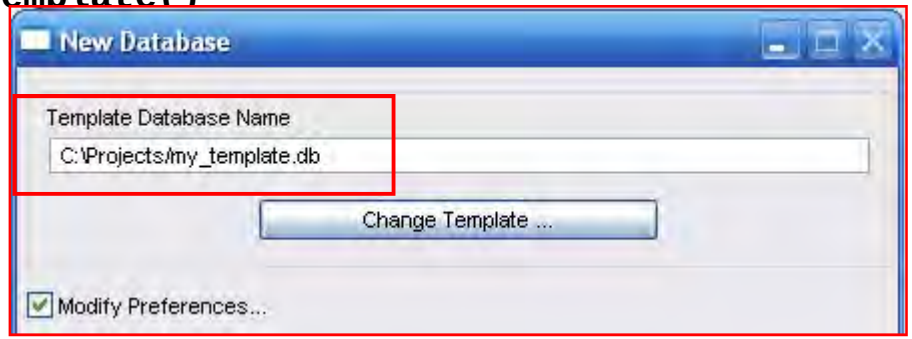
PATRAN TEMPLATE DATABASE

- **You can tell Patran to use a different default template database with the help of some settings.pcl entries**

```
Pref_env_set_string("template_path","c:\Projects")
Pref_env set_string("template_name","my_template.db")
```

- **For this to work properly, it requires the following entry at the bottom of the p3epilog.pcl file:**

```
au_set_default_template()
```



Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

- **Use the Patran settings.pcl file to specify various session parameters.**

- **When Patran launches, it looks for a 'settings.pcl' file in the following order:**
  - Start-In /working directory
  - User's home directory
    - %HOMEDRIVE%%HOMEPATH% on Windows
  - Patran installation directory

- **There are four types of variables, as follows:**
  - pref_env_set_**integer**("<variable name>", <value>)
  - pref_env_set_**real**("<variable name>", <value>)
  - pref_env_set_**logical**("<variable name>", <value>)
  - pref_env_set_**string**("<variable name>", <value>)

Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

- **Sample settings.pcl file:**

```
pref_env_set_integer( "message_warning", 3 )
pref_env_set_integer( "create_dup_geometry", 3 )
pref_env_set_integer( "graphics_colors", 120 )
pref_env_set_string( "graphics_hardware", "YES" )
pref_env_set_string( "graphics_fullcolor", "NO" )
pref_env_set_string( "graphics_refresh", "NO" )
pref_env_set_real( "hc_letter_ht", 0.8 )
pref_env_set_logical( "display_template_preference", TRUE )
pref_env_set_string( "insight_graphics_hardware", "NO" )
pref_env_set_string( "result_capture_filename", "patran.rpt" )
pref_env_set_string( "result_quick_transform", "Default" )
pref_env_set_string( "result_quick_avg_method", "DeriveAverage" )
pref_env_set_logical( "result_dbopen_display", TRUE )
pref_env_set_integer( "result_loadcase_abbreviate", 30 )
pref_env_set_string( "p3team_graphics_hardware", "NO" )
pref_env_set_logical( "elements_mpcs_share_id_space", FALSE )
pref_env_set_logical( "elements_mpcs_warn_if_ids_conflict", FALSE )
pref_env_set_logical( "show_cycle_picking_form", TRUE )
pref_env_set_logical( "patran_metadata_tools", FALSE )
pref_env_set_string( "ABAQUS_VERSION", "V6.X" )
pref_env_set_string( "INTERBASE_TMP_DIR", "C:\DOCUME~1\e1286\LOCALS~1\Temp" )
pref_env_set_logical( "ShowIconHelp", TRUE )
pref_env_set_string( "mouse_map_combination_1", "ROTXY" )
pref_env_set_string( "mouse_map_combination_2", "ZOOM" )
pref_env_set_string( "mouse_map_combination_3", "TRANSXY" )
pref_env_set_string( "mouse_map_combination_4", "ROTZ" )
pref_env_set_logical( "revert_enabled", FALSE )
pref_env_set_integer( "VisibleHistoryItems", 3 )
pref_env_set_string( "NASTRAN_MODFIL_SUFFIX", ".bdf" )
pref_env_set_string( "select_menu_layout", "horizontal" )
```


Copyright© 2012 MSC.Software Corporation

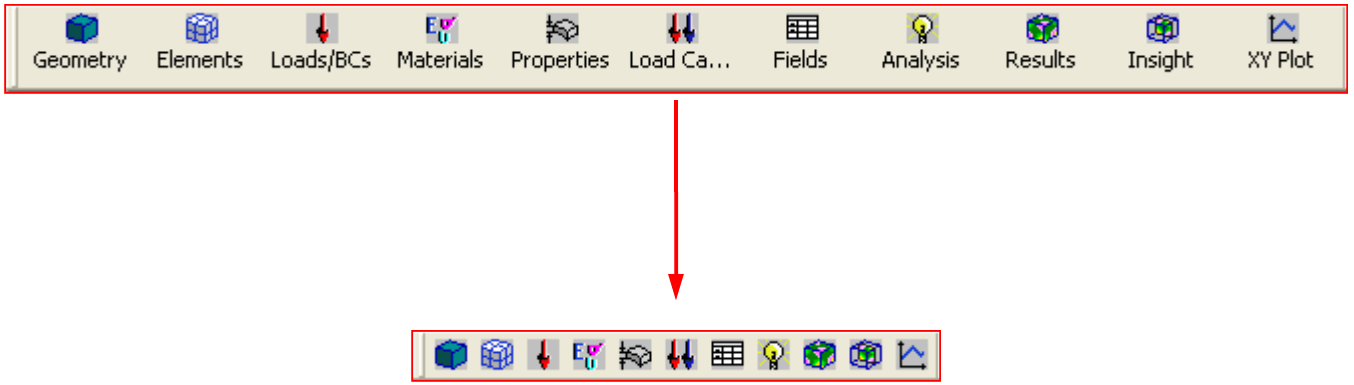
PATRAN SETTINGS.PCL FILE

- **Many entries can be controlled from within the GUI and need not be set manually.**
  - Changes are saved to settings.pcl when quitting Patran.
  - E.g. graphics_hardware (Preferences/Graphics), revert_enabled (Preferences/Global)
- **Some entries only apply to Unix/Linux or to Windows**
  - E.g. Unix/Linux only : ApplSwitchIsPopup
  - E.g. Windows only: HideAppLabels
- **Some entries are obsolete or nearly so**
  - E.g. INTERBASE_TMP_DIR
- **The next few slides present a selection of useful settings**

Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

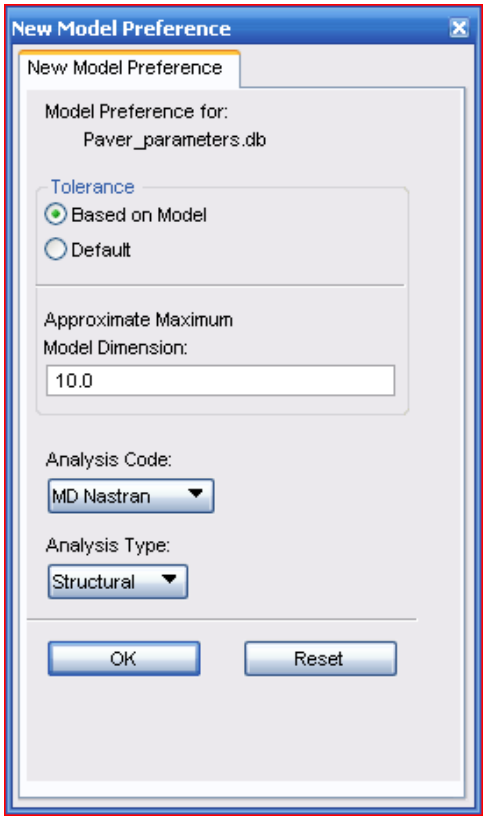
- **pref_env_set_logical(“HideAppLabels”, TRUE)**
  - Hides labels on the application buttons (Windows only)



Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

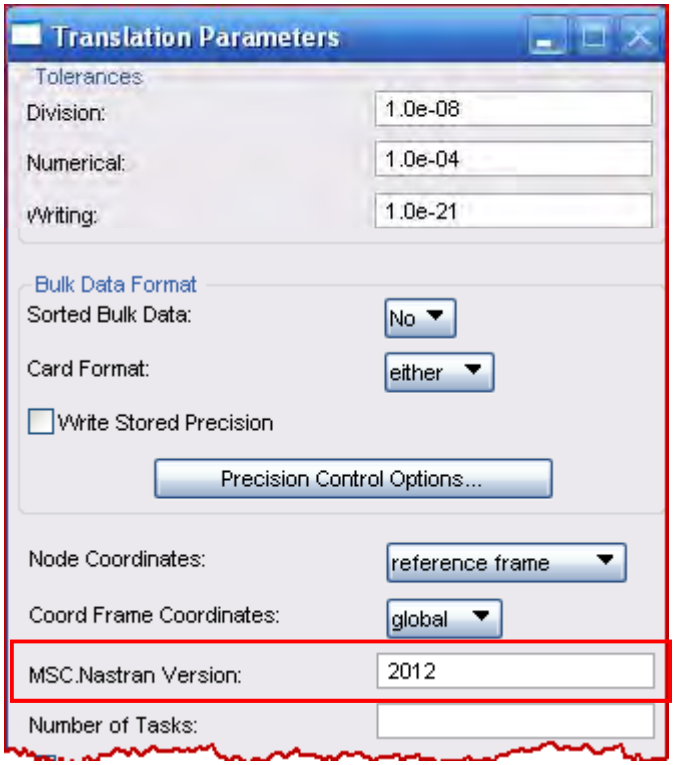
- **pref_env_set_logical(“display_template_preference”, FALSE)**
  - Prevents display of the preference selection form when creating a new database:



Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

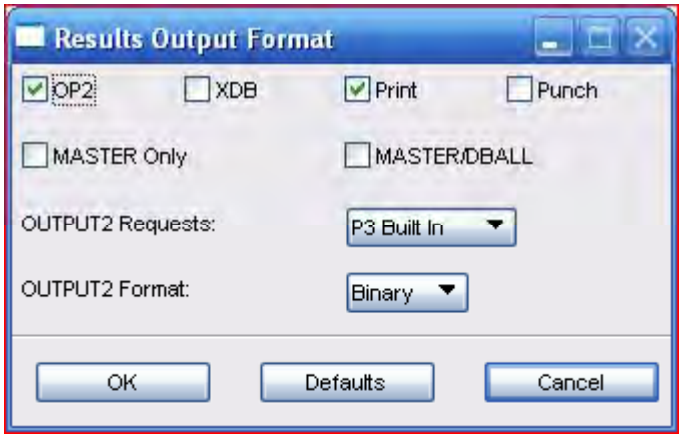
- **pref_env_set_string(“NASTRAN_VERSION”, “2012”)**
  - Sets the default Nastran version in Translation Parameters



Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

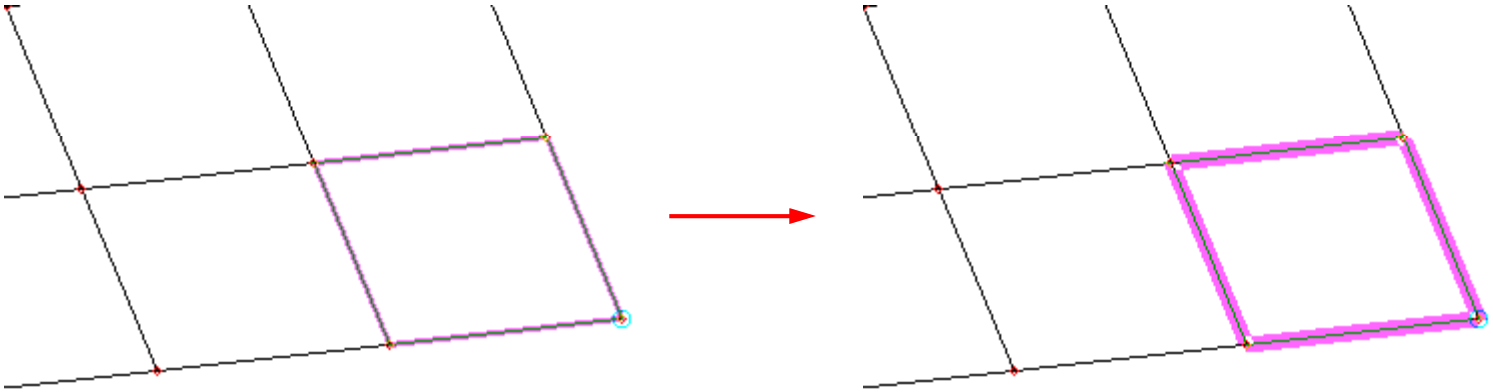
- **pref_env_set_string(“NastranResultsOutput”, “OP2 and Print”)**
  - Sets the default Nastran output files
  - Possible values:
    - “XDB Only”
    - “XDB and Print”
    - “OP2 Only”
    - “OP2 and Print”
    - “Print Only”
    - “None”



Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

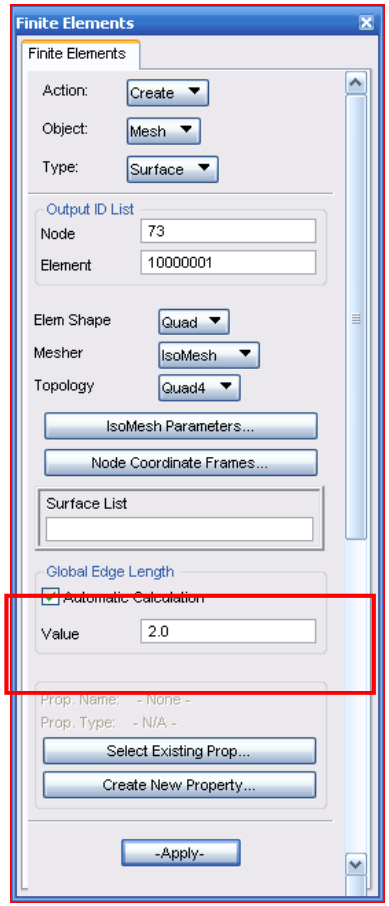
- **pref_env_set_string(“preselection_line_width”, 8)**
  - Sets the width of preselection highlighting graphics. Default is 3.



Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

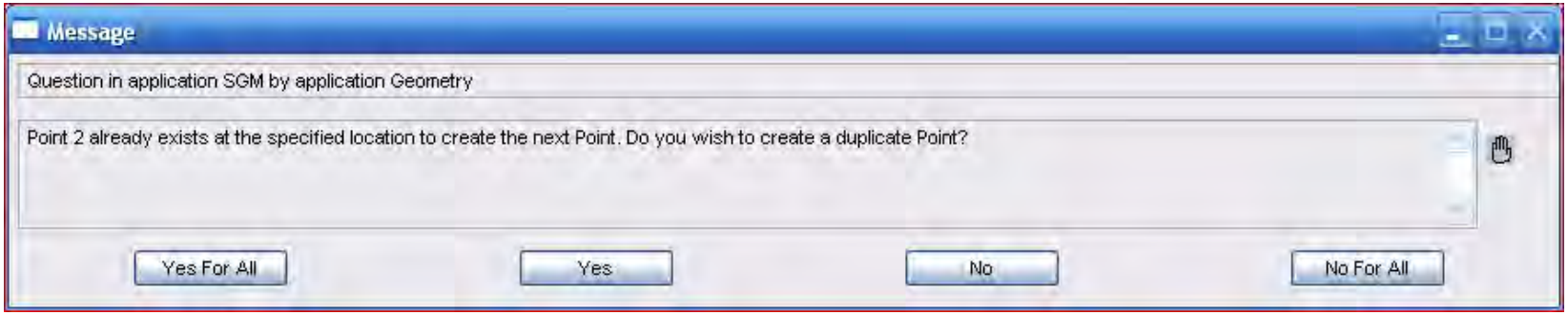
- **pref_env_set_real(“DefaultMeshSize”, 2.0)**
  - Sets the default value of GEL on the meshing forms



Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

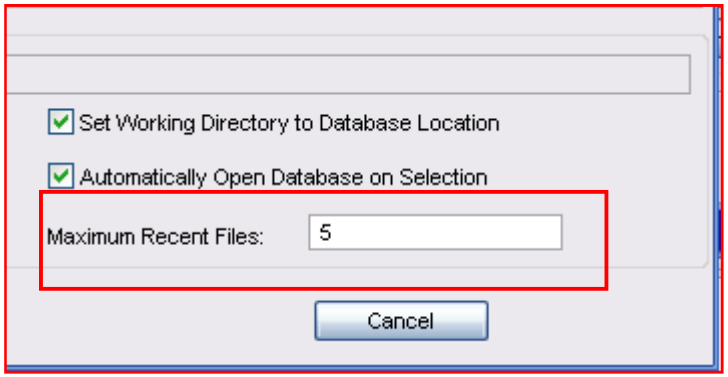
- **pref_env_set_integer(“create_dup_geometry”, 3)**
  - Controls creation of duplicate geometry.
    - 1 creates duplicate geometry automatically
    - 2 never creates duplicate geometry
    - 3 (default) asks user for permission before creation



Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

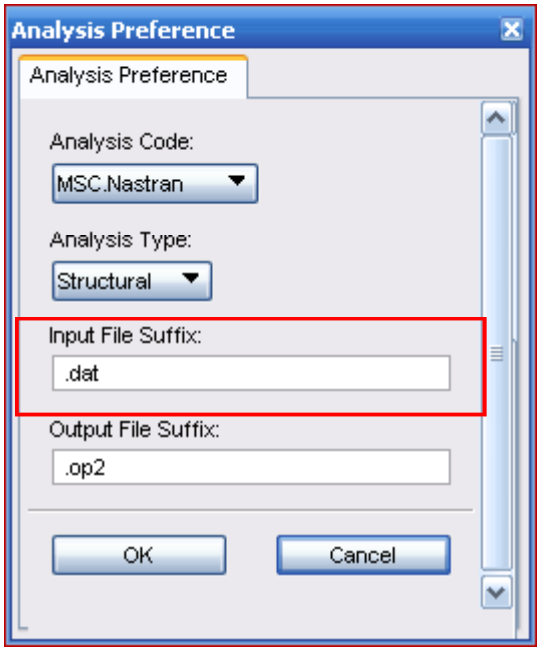
- **pref_env_set_integer(“max_num_recent_files”, 5)**
  - Controls the maximum number of files visible under **File > Open Recent…** (default 10)



Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

- **pref_env_set_string(“NASTRAN_MODFIL_SUFFIX”, “.dat”)**
  - Controls the default extension when writing a Nastran input file



Copyright© 2012 MSC.Software Corporation

PATRAN SETTINGS.PCL FILE

- **Some Unix/Linux-only settings:**

Variable name	Purpose	Type	Possible values
VisibleHistoryItems	Default size of history window in lines.	Integer	1 < number < 80
SaveVisHistoryLineCount	Saves visible history line count when Patran terminates.	Logical	TRUE or FALSE
ApplSwitchHeight	Height of application switches area.	Real	0 < number < 1
ToolbarHeight	Height of toolbar area.	Real	0 < number < 1
entity_picking_cursor	Specifies cursor type (shape) for single picking.	String	holeangle , xhole , +hole, + , x
select_menu_layout	Specify select menu orientation as vertical or horizontal.	String	vertical , horizontal
select_menu_type	Specifies whether the select menu (form) appears automatically, or appears as a popup by using keymapping.	String	form , popup
ApplSwitchIsPopup	No application buttons in main menu, but appear in popup menu if TRUE.	Logical	TRUE or FALSE

Copyright© 2012 MSC.Software Corporation

START-UP FILES

p3prolog.pcl

- This file is one of the first PCL files to be read by Patran during start-up. Since Patran has not yet been initialized when this file is read, this file is only useful for the initialization of global variables, additions to the PCL path, or linking PCL libraries.

p3midilog.pcl

- The p3midilog.pcl file is read after the p3prolog.pcl file and immediately prior to the initialization of the Patran graphical user interface. This file is typically used to augment the PCL path definition prior to the initialization of Patran's toolbar.

p3epilog.pcl

- The p3epilog.pcl file is one of the last files read during the Patran start-up process. Since Patran has been initialized by the time this file is read, this file can include calls to almost any PCL function. Typically, the p3epilog.pcl file can contain statements which add to the PCL path and link PCL libraries.

Copyright© 2012 MSC.Software Corporation

COMMAND-LINE ARGUMENTS

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

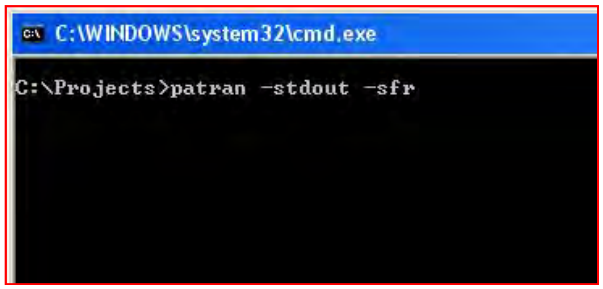
COMMAND-LINE ARGUMENTS

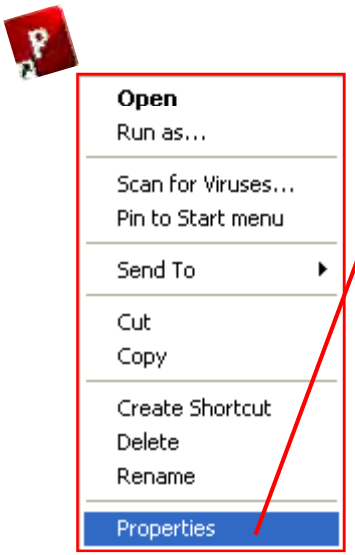
- **Patran has a number of command-line options that can be applied.**
  - Full details in the Installation and Operations Manual
- **Unix – commands issued in the Unix shell**

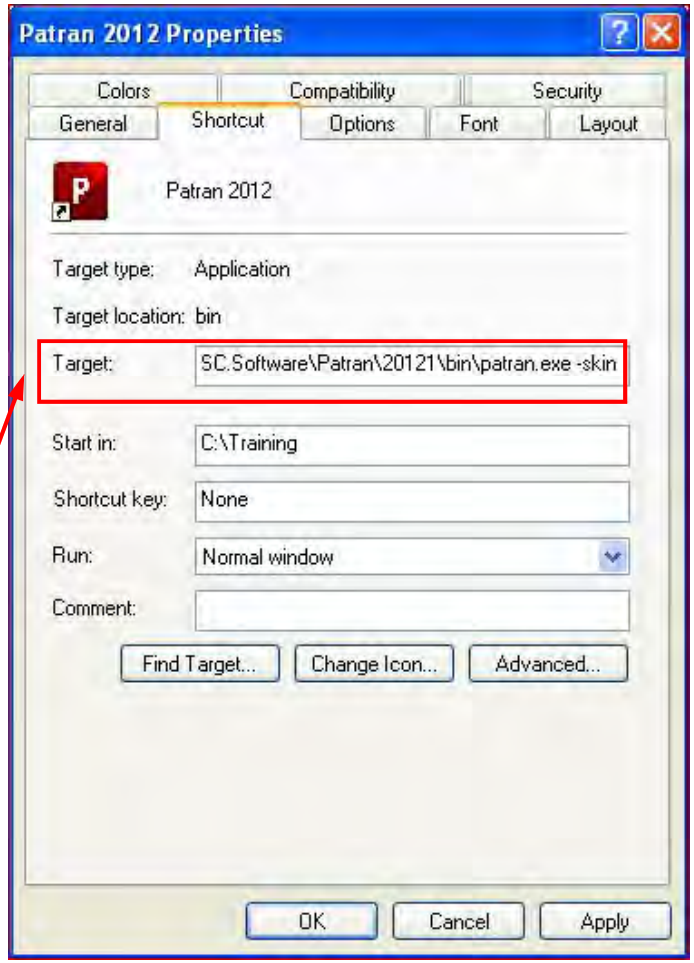
Copyright© 2012 MSC.Software Corporation

COMMAND-LINE ARGUMENTS

- **Windows – entered in the “Target” field of the desktop shortcut**
  - Can also be typed in a DOS window







Copyright© 2012 MSC.Software Corporation

COMMAND-LINE ARGUMENTS

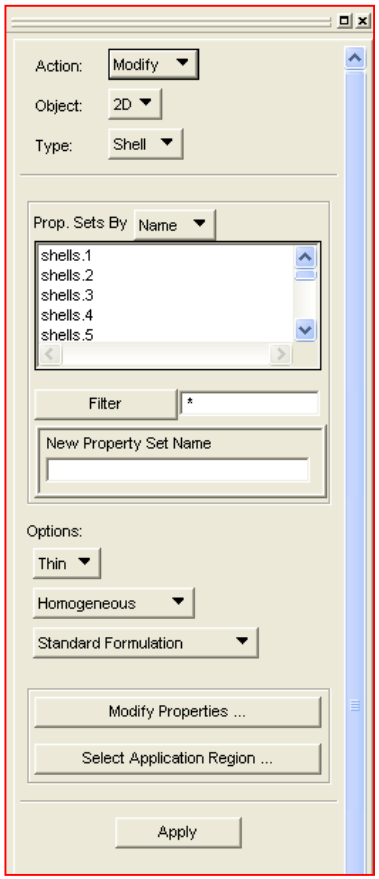
- **A selection of commonly-used arguments:**

-b	Batch mode (no graphics). If entered with no other flags, Patran will run a test of licencing and write permissions, then exit. Usually used with –sfp.
-sfp *file.ses*	Patran will execute the session file called *file.ses* on startup.
-sfr *file.ses*	Patran will record session file entries to a session file called *file.ses* rather than the default patran.ses.xx. Suppresses session file creation if no filename is supplied.
-graphics	Used with –b and –sfp: allows Patran to be run in batch mode but still output graphical image files if the session file requests them.
-stdout	Windows only. Causes a DOS window to remain visible when running Patran - some information messages appear only there. Can additionally specify a log file where the messages will be stored.
-l	Enables large listbox mode for LBCs, load cases, properties, materials and fields.
-EchoEnv	Causes Patran to list all environment variables that it recognizes. Useful for debugging problems.
-hl	Unix only. Sets the number of visible lines in the history window. Default 200, max. 1000. (Windows default is 50 and cannot be changed.)

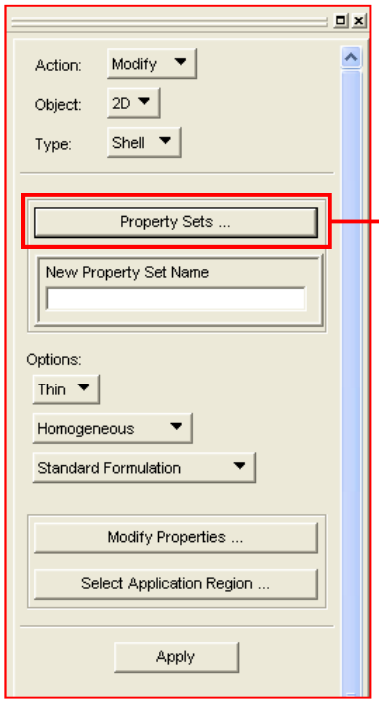
Copyright© 2012 MSC.Software Corporation

COMMAND-LINE ARGUMENTS

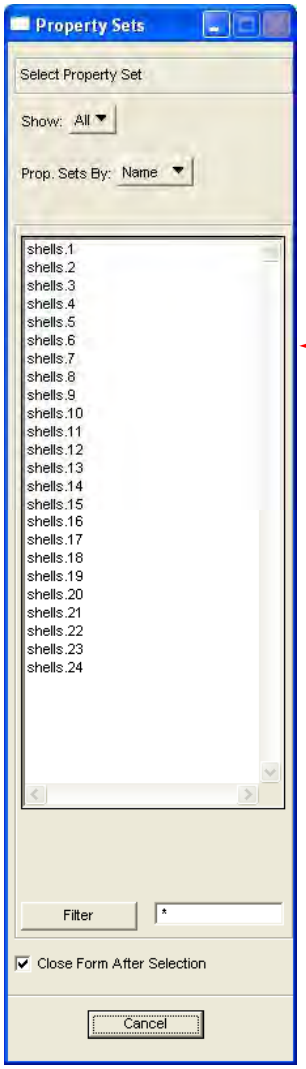
- **Effect of -l argument:**

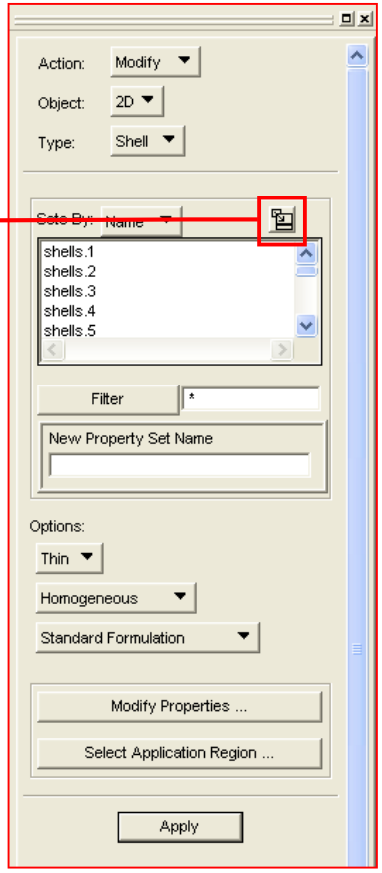


2007 without -l



With -l



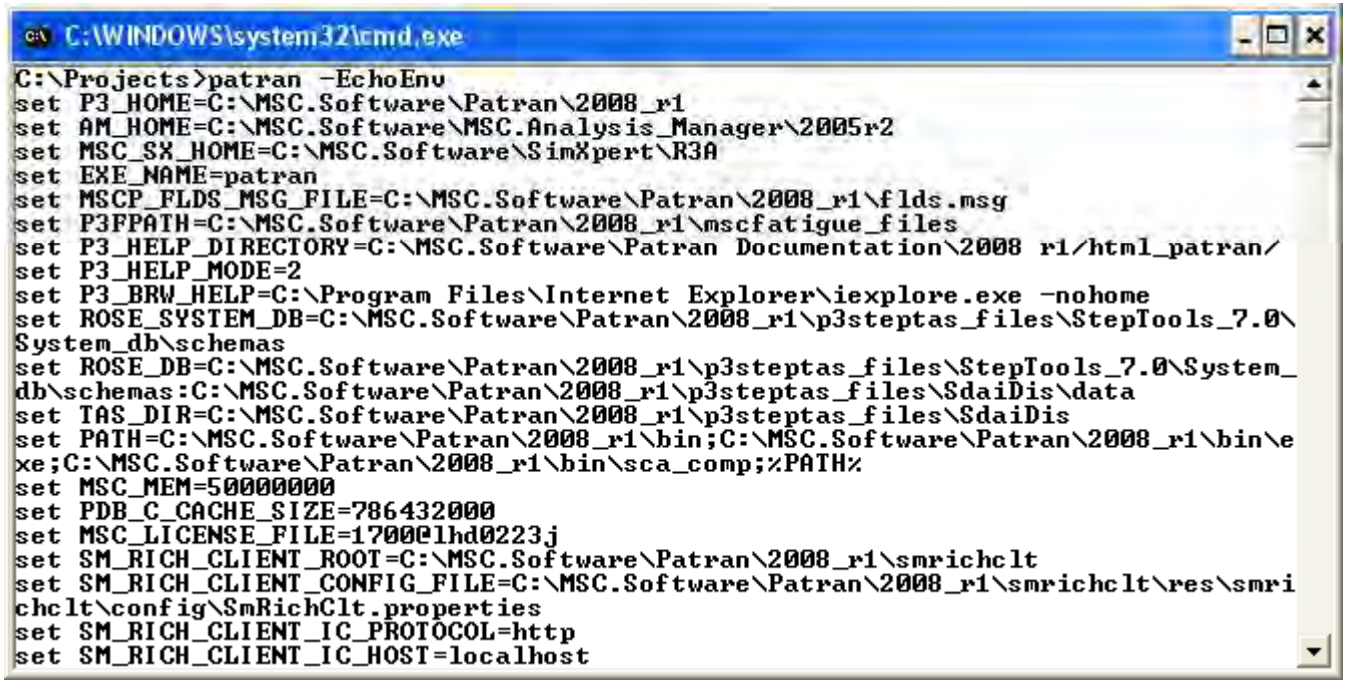


2012 without -l

Copyright© 2012 MSC.Software Corporation

COMMAND-LINE ARGUMENTS

- **Effect of -EchoEnv argument:**



Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

ENVIRONMENT VARIABLES

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

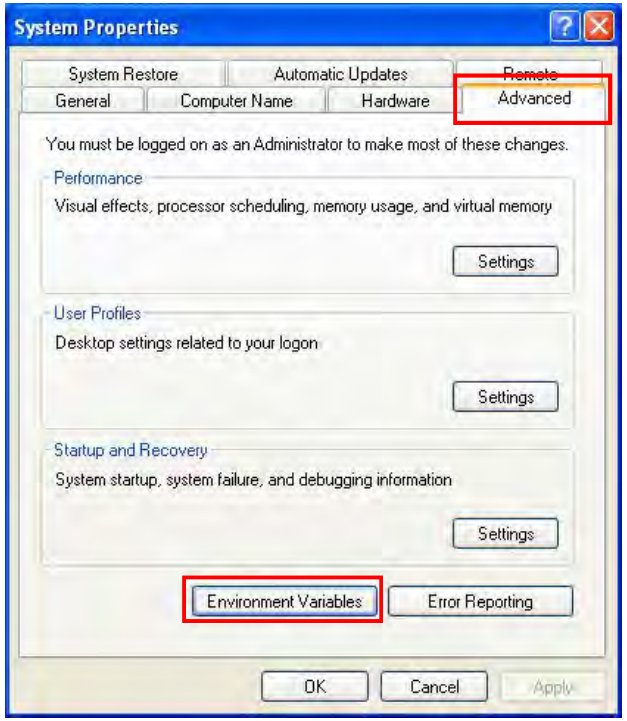
ENVIRONMENT VARIABLES

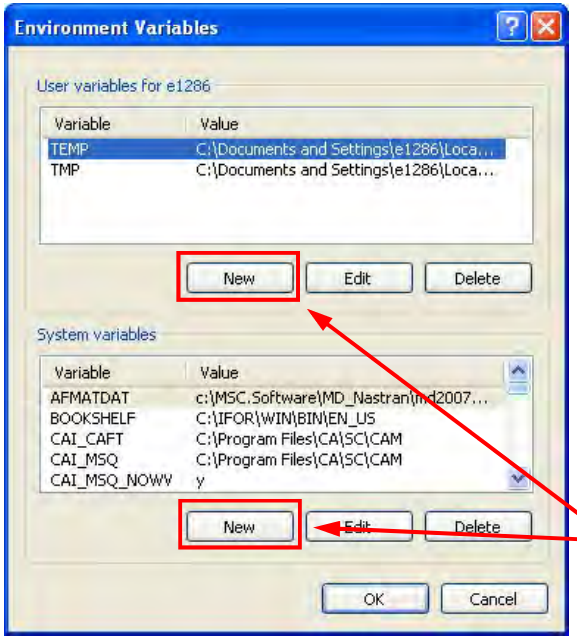
- **A number of environment variables can also be defined to control or modify Patran behaviour**
- **Can be set in the shell or in the site_setup file (site_setup.pm on Windows)**
- **Most (but not all!) are documented in the Installation and Operations Manual**

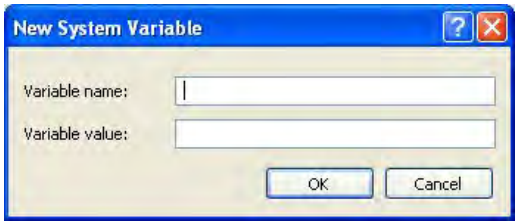
Copyright© 2012 MSC.Software Corporation

ENVIRONMENT VARIABLES

- **Setting environment variables:**
  - Unix – set in the shell. Syntax depends on shell being used.
    - setenv <variable name> <value>
    - export <variable name>=<value>
  - Windows – use Control Panel/System







Can set as system or user variable

Copyright© 2012 MSC.Software Corporation

ENVIRONMENT VARIABLES

- **Setting environment variables (cont.):**
  - Or can set in site_setup.pm file (found in P3_HOME)
  - site_setup on Unix

  - Syntax in the file is:

    - Windows
    - setEnv('<variable name>','<variable value>');

    - e.g. setEnv('MSC_MEM','10000000');

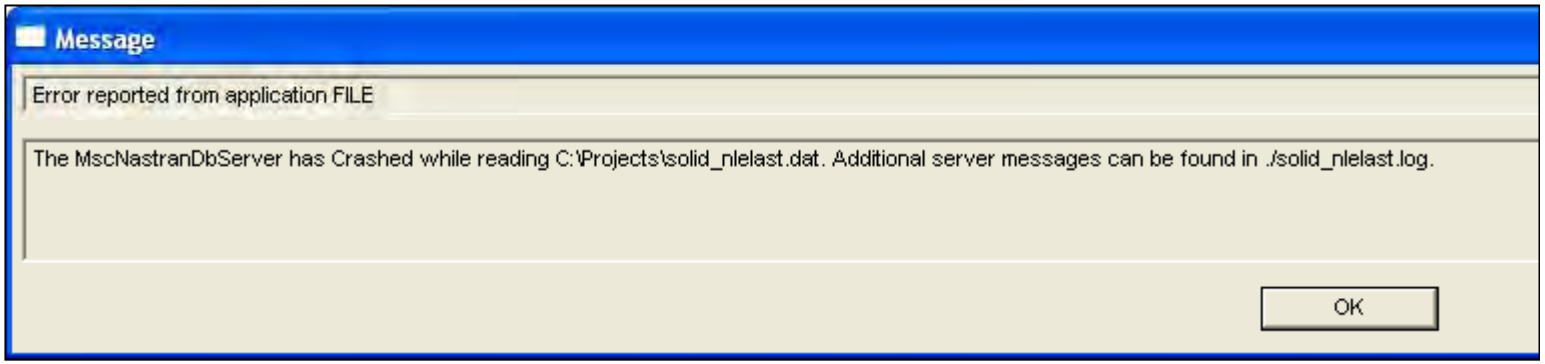
    - Unix
    - setEnv <variable name> <variable value>

    - e.g. setEnv MSC_MEM 10000000

Copyright© 2012 MSC.Software Corporation

ENVIRONMENT VARIABLES

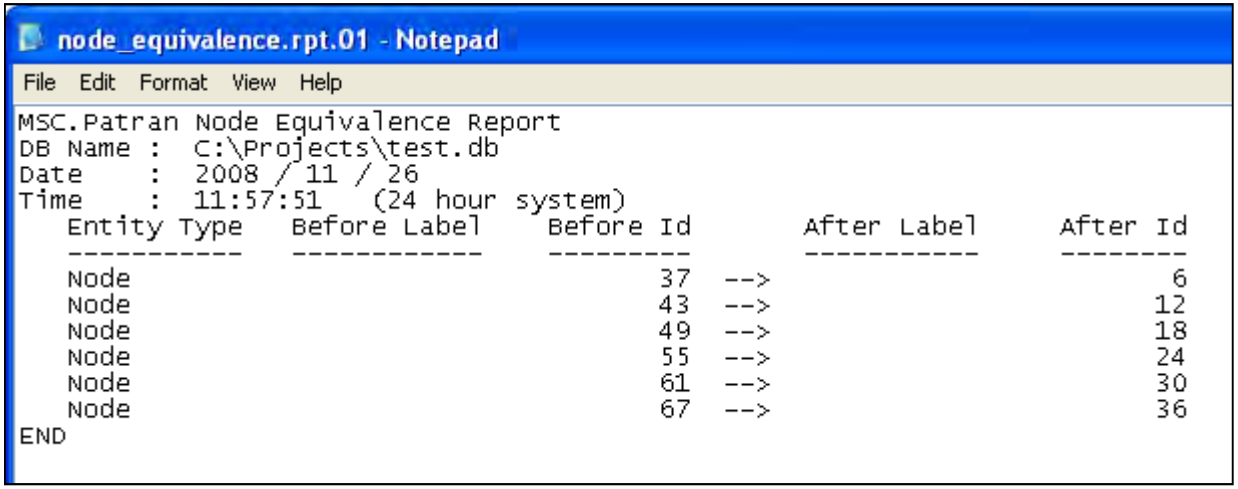
- **MSC_MEM <value in words of memory>**
  - Sets the amount of memory available for importing Nastran decks
  - Can increase default for large models
  - e.g. setEnv('MSC_MEM','10000000') in site_setup assigns 40 Mb



Copyright© 2012 MSC.Software Corporation

ENVIRONMENT VARIABLES

- **WRITE_EQUIVALENCE_REPORT YES**
  - Causes a report to be written to file for each equivalence operation
  - File name is node_equivalence.rpt.nn



Copyright© 2012 MSC.Software Corporation

USING THE GUI

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

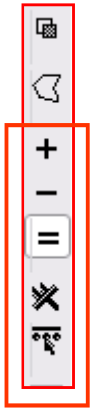
PICKING ENTITIES

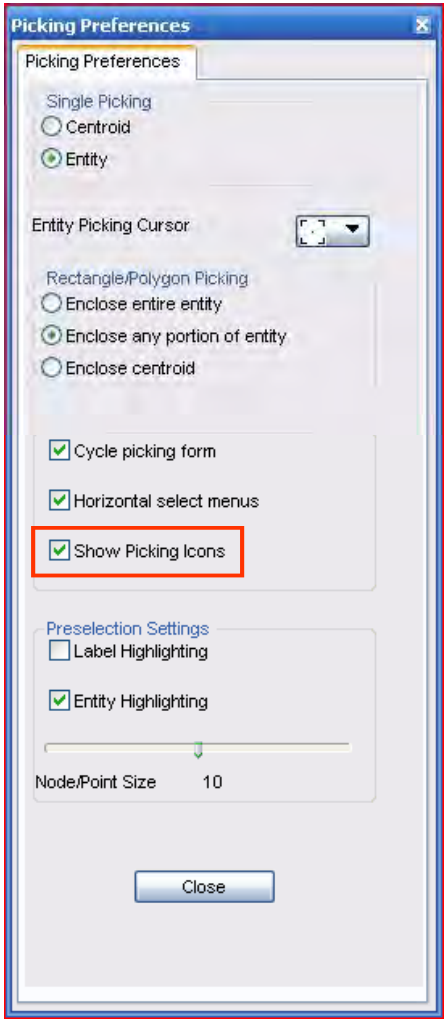
- **Picking can be done using the standard methods:**
  - Put mouse focus in a list box, e.g. Starting Surface List
  - Pick an entity in the viewport, e.g. Solid 1.6
  - Pick again, and the first entity selected is replaced by the current entity selected, unless the shift key is used.
  - After selecting several entities, you may wish to deselect an entity. This can be done by:
    - using the right mouse button (2007r1 and older versions).
    - holding down **Ctrl *and* Shift** and clicking with the right mouse button (2007r2 onwards).

Copyright© 2012 MSC.Software Corporation

PICKING ENTITIES

- "Show Picking Icons" activates additional icons on the select menu, giving extra options for screen selection
  - "+" sets addition mode: new selections are added to the contents of the databox (no need to use Shift key)
  - "-" sets subtraction mode: new selections are removed from the contents of the databox
  - "=" (default) sets replace mode: new selections replace the contents of the databox
  - (Pick Clear) clears the selected/picked entities and updates Viewport(s) only.
  - (Pick All) selects all entities based on the current Picking Filter set to and updates Viewport(s) as well as the current select databox.





Copyright© 2012 MSC.Software Corporation

DATA BOX SYNTAX

Syntax / Format	Description
Point 1 2 3	Refers to points 1, 2, and 3
Point 1:9:2	Points 1 through 9 by 2
Node 1:#	All nodes in the database
Curve 1 2, 3/4	Different forms for delimiters: space, “,” and “/”
Surface 3.1	References an entity associated with a higher order entity (e.g. edge 1 of surface 3; an edge is similar to a curve)
Solid 1:10.2	Combinations of entity ID syntax is possible (face 2 of solids 1 through 10)
[x y z]	Square brackets signifies coordinate specification
[xn28, 1, 2]	Individual coordinates can reference existing entities, such as x = the x coordinate of node 28
[1, zp5, 3] [1, z5, 3]	y = the z coordinate of point 5 When a point is referenced the letter “p” can be dropped
[1, 2, ‘-64.0/20.0‘]	Mathematical operations like division are possible to determine the individual components
<R T Z>	< > signifies a vector definition
{[x1 y1 z1][x2 y2 z2]}	Signifies an axis with first point representing the base and the second determining the direction

Copyright© 2012 MSC.Software Corporation

MANUAL SELECTION, SHORTCUTS

- **Abbreviations**
  - Point: point, pt
  - Curve: curve, crv, c
  - Surface: surface, surf, s
  - Solid: solid, sld, sl
  - Node: node, nde, n
  - Element: element, elem, elm, el,e, quad, bar, tri, tet, wedge, hex

  Coordinate system: coord, cord, cf

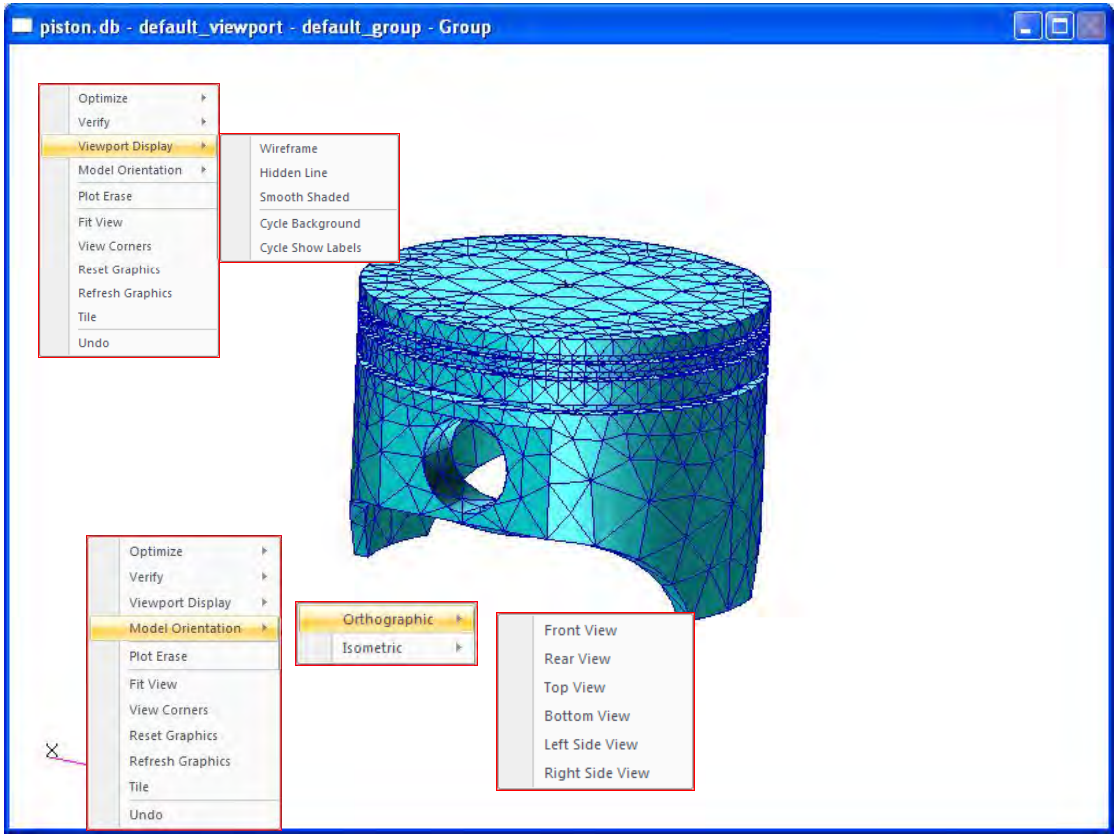
- **Not case sensitive**

Copyright© 2012 MSC.Software Corporation

RIGHT MOUSE BUTTON MENUS

- **Right mouse button can be used in the viewport to pull up menus of common actions such as “Fit View” and model orientations**
- **If an entity is selected, entity-specific options are available**

2007r2 onwards

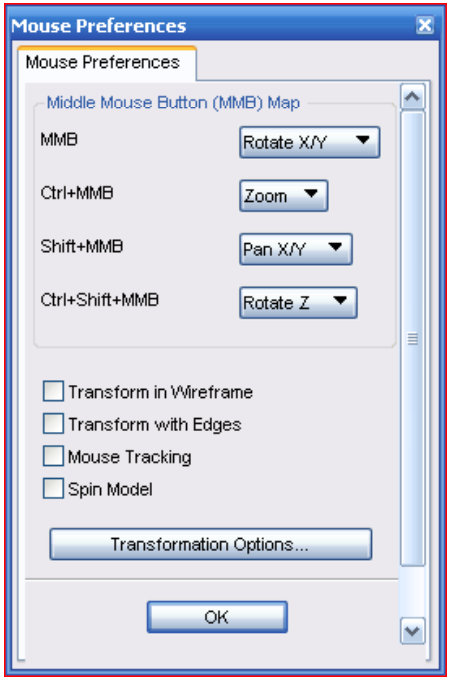


Copyright© 2012 MSC.Software Corporation

MIDDLE MOUSE BUTTON

- In Patran 2007r2 new middle mouse button functionality was introduced
- Can now use Ctrl and Shift to affect middle mouse button action.
- Definitions can be changed under Preferences / Mouse…
- Changes stored in the settings.pcl file:

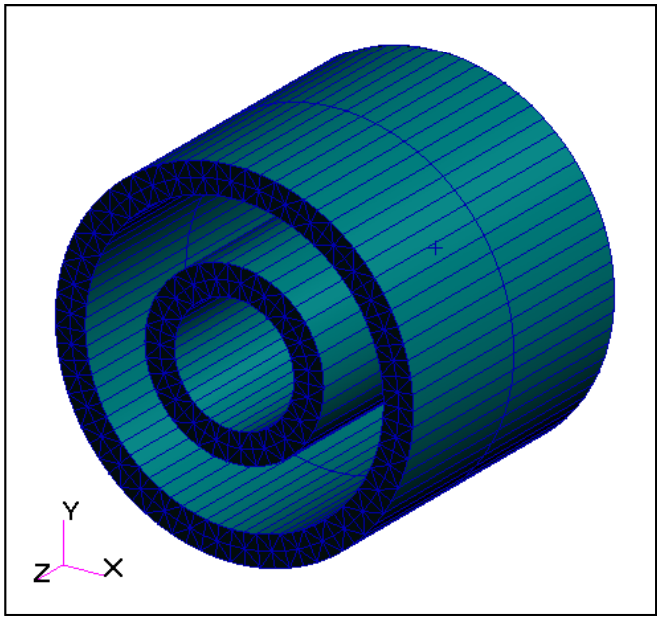
```
pref_env_set_string( "mouse_map_combination_1", "ROTXY" )
pref_env_set_string( "mouse_map_combination_2", "ZOOM" )
pref_env_set_string( "mouse_map_combination_3", "TRANSXY" )
pref_env_set_string( "mouse_map_combination_4", "ROTZ" )
```

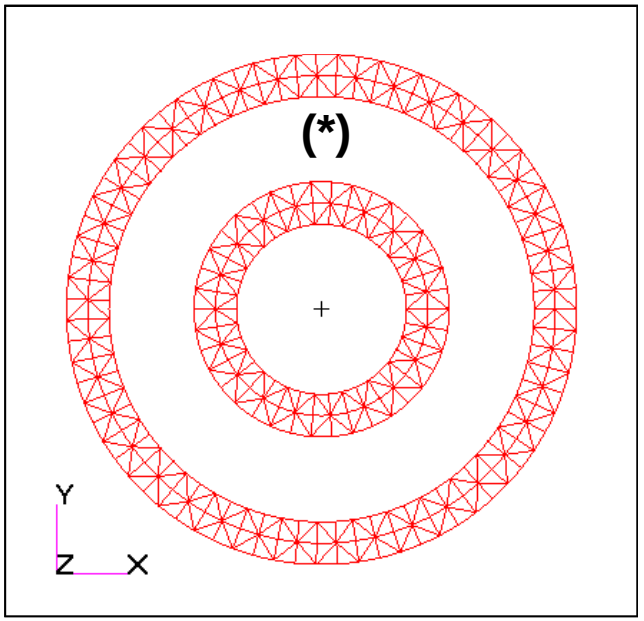


Copyright© 2012 MSC.Software Corporation

PICKING ENTITIES

- **Apply pressure between the two meshes (*).**
- **It is to be normal to the free faces of the elements.**
  - Radial direction
- **There is no geometry.**
- **The challenge is to specify the application region.**

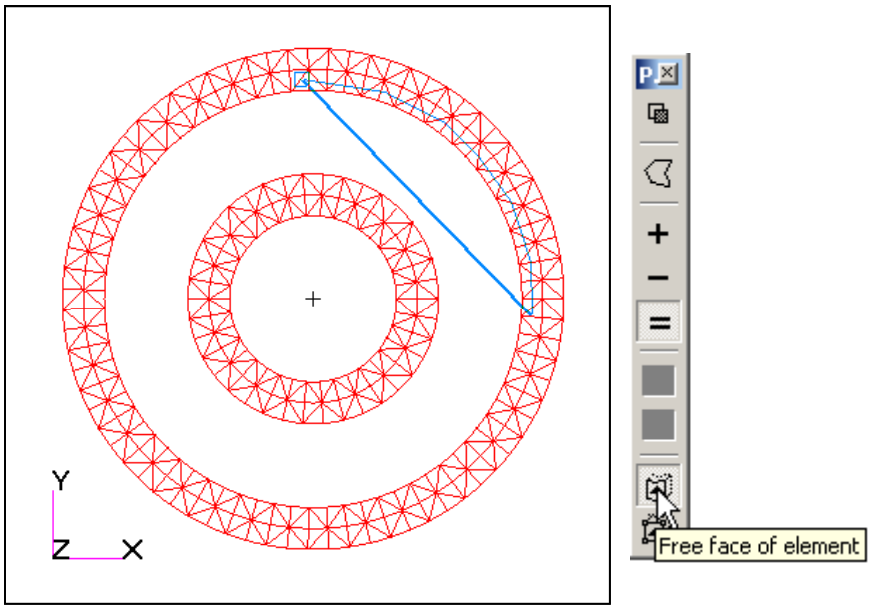


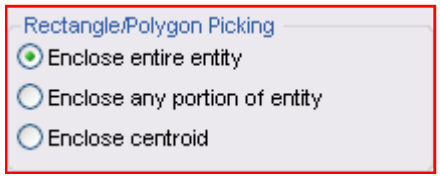


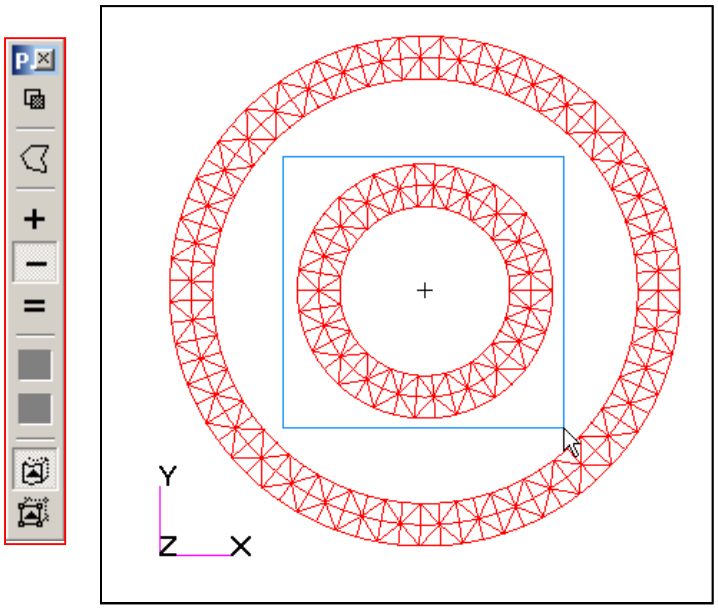
Copyright© 2012 MSC.Software Corporation

PICKING ENTITIES

- **Pick all free faces inside of polygon**





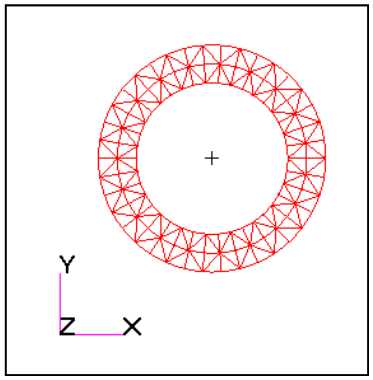


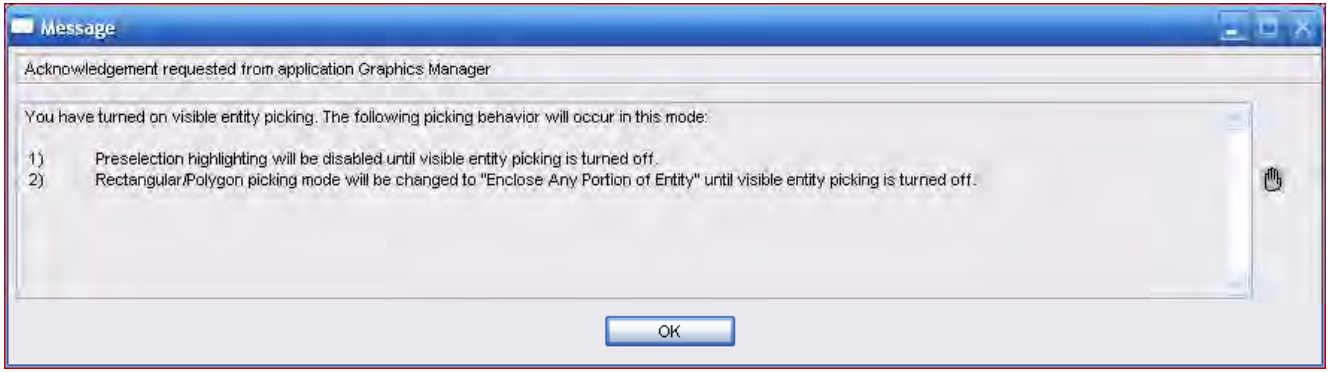
- **Reject all free faces inside the rectangle**

Copyright© 2012 MSC.Software Corporation

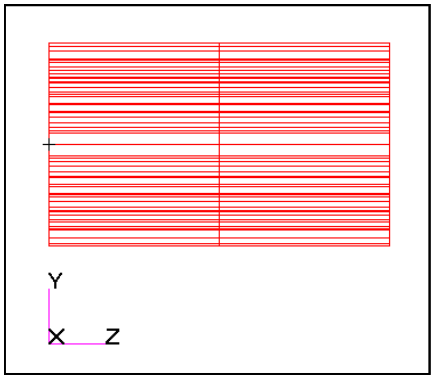
PICKING ENTITIES

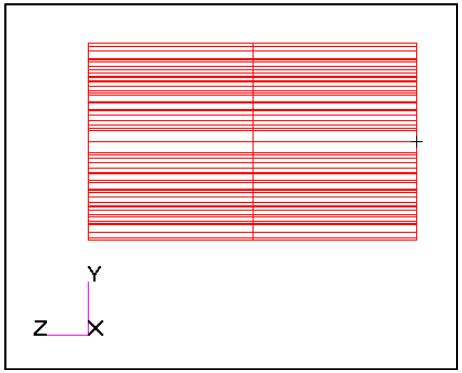
- **Erase all elements outside of inner elements**
- **Select Visible entities only icon**



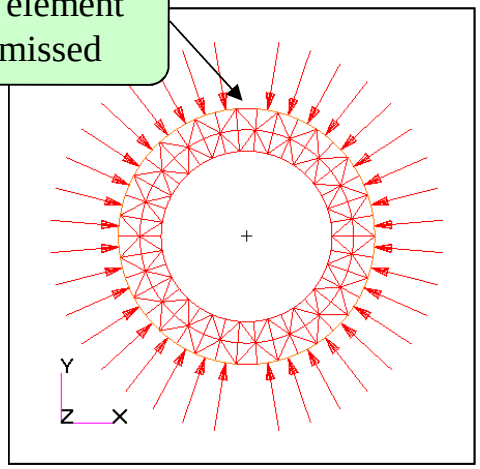


Select all visible element faces





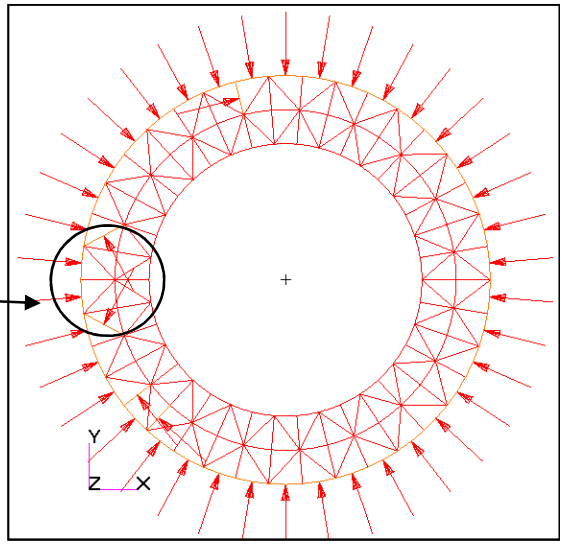
Some element faces missed

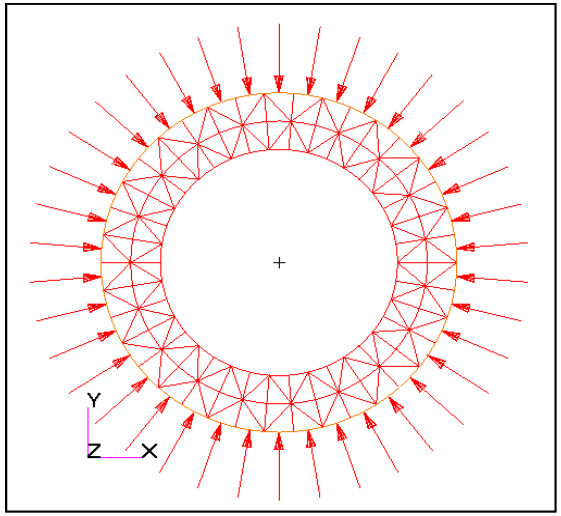


Copyright© 2012 MSC.Software Corporation

PICKING ENTITIES

- Rotate inner mesh to see the X-Y view
- Select more element free faces
- Oops ! Got some interior faces
- Turn off visible only picking. Set to pick any face of element, and any portion of entity.
- Edit the pressure LBC
- Select the area of the unwanted red arrows for the application region, and use the Remove button.
- Finally, have the correct pressure on the inside mesh

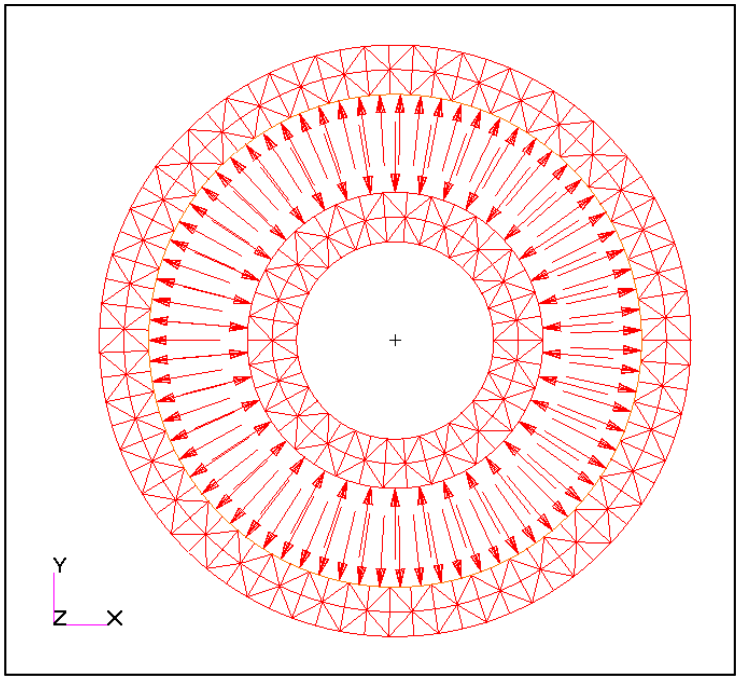


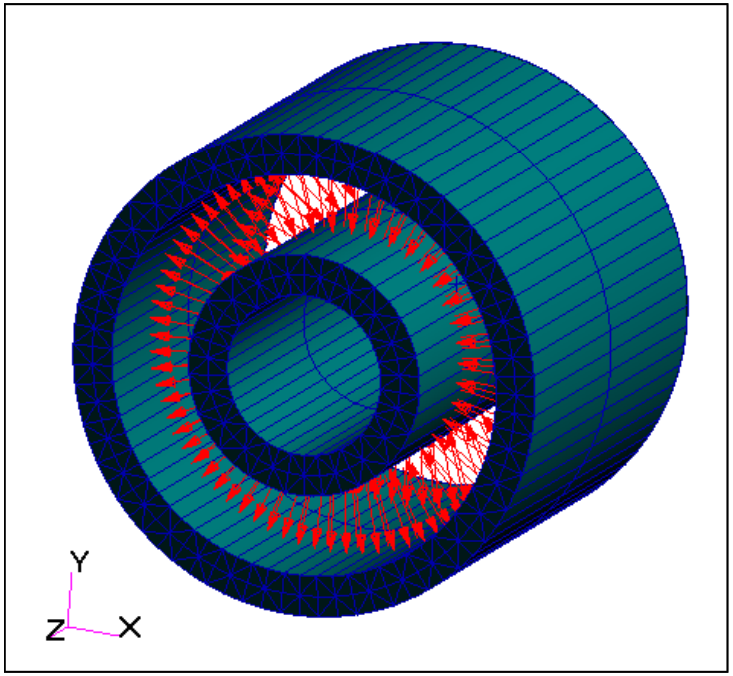


Copyright© 2012 MSC.Software Corporation

PICKING ENTITIES

- **The pressure arrows are shown for both inner and outer mesh**
- **For this example a separate pressure LBC was created for the two meshes**

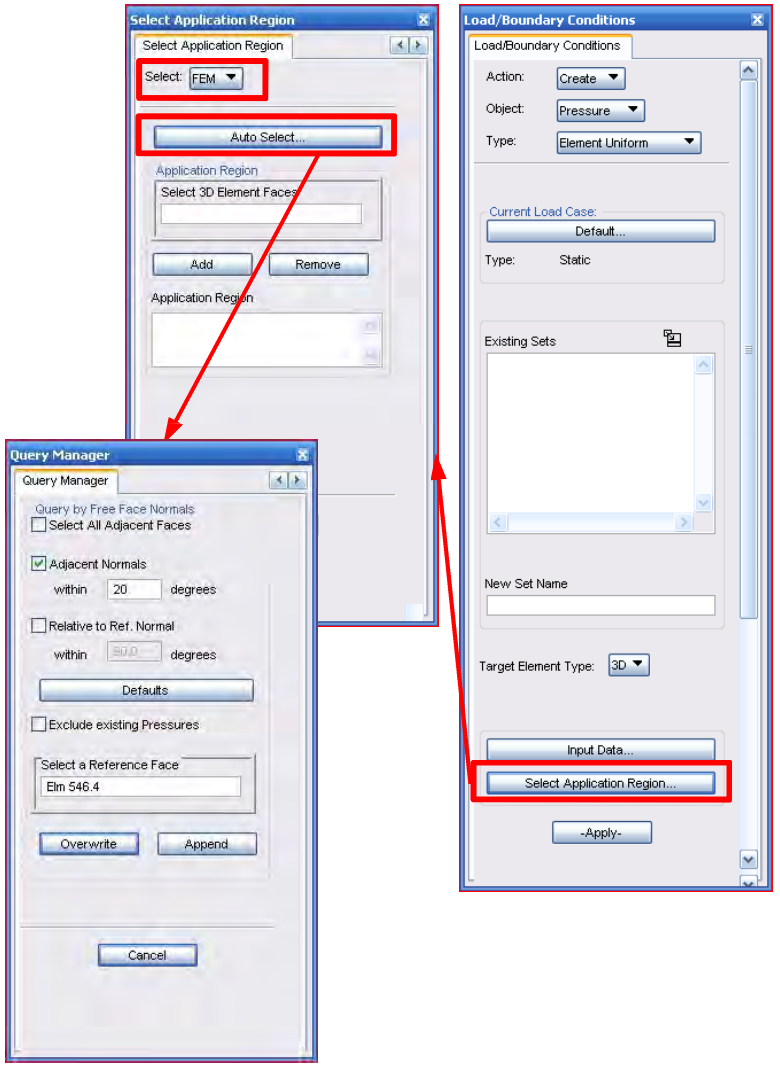




Copyright© 2012 MSC.Software Corporation

PICKING ENTITIES

- **It would have been possible to create a single pressure LBC.**
- **There is at least one other way to specify the application region of this LBC, like using the Query Free Face Selection Tool discussed in the previous section.**



Copyright© 2012 MSC.Software Corporation

TOOLBAR EDITING

- **Contents of the toolbar are controlled by p3toolbar.def**
- **Patran searches for this file in the user's home directory, followed by the Patran installation directory**

```
*START TOOLBAR    =Transforms

  *ICON     = tbrotatexy.bmp
  *CLASS    = uil_toolbar
  *FUNCTION = rotate_xy
  *HELP     = Mouse rotate XY
*LOAD ITEM
.
.
.
*END TOOLBAR
```

*START TOOLBAR and *END TOOLBAR entries on Windows only

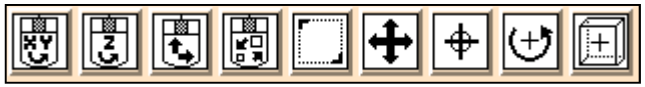
Copyright© 2012 MSC.Software Corporation

TOOLBAR EDITING

Toolbar Icons

For Unix

- Icons are 28x28 pixels
- Black and white only
- Editor is “bitmap”



For Windows

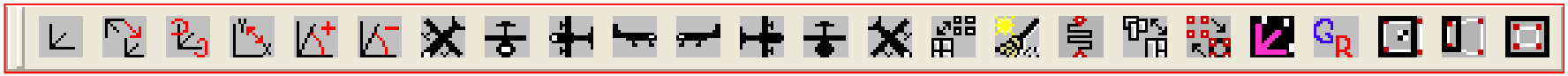
- Icons are 16x16 pixels
- Can be in colour
- Editor using e.g. MS Paint

Copyright© 2012 MSC.Software Corporation

TOOLBAR EDITING

- **Example: adding Utilities icons to the toolbar**
  - Add the contents of file *add_to_p3_toolbar.def* to the p3_toolbar.def file
  - Add entries to p3prolog.pcl for path to icons

```
sys_path( "ADD", “C:\MSC.Software\Patran\20121 // @
  "/shareware/msc/unsupported/utilities/icons" )
```



View tools

Custom view icons

Rectangle /polygon pick options

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

SHORT-CUTS

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

KEYMAPPING

- **Keymapping is easy to do and can make it possible to work faster**
- **Doing it is similar to creating a toolbar icon**
  - Create a PCL function
  - Compile it
  - Load the compiled PCL function using the p3epilog.pcl file
  - Create the keymap in the file Patran.EventMaps
    - .Patran.EventMaps on Unix

Copyright© 2012 MSC.Software Corporation

KEYMAPPING

- **Patran.EventMaps syntax**

```
[modifier]<Key>[key]: [function]
```

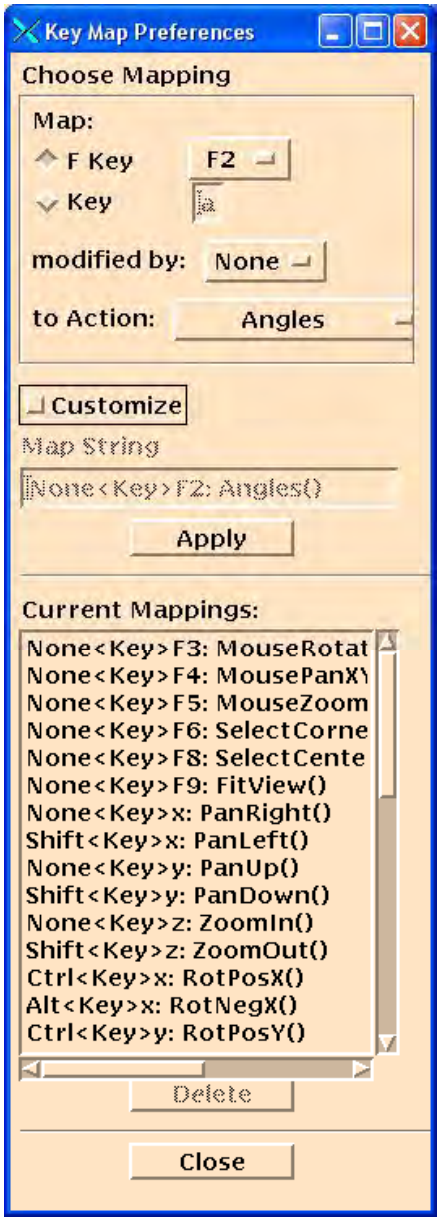
  - Modifiers are:
    - None
    - Shift
    - Ctrl
    - Alt

```
None<Key>F2: MouseRotateXY()
None<Key>F3: MouseRotateZ()
None<Key>F4: MousePanXY()
None<Key>F5: MouseZoom()
None<Key>F6: SelectCorners()
None<Key>F8: SelectCenter()
None<Key>F9: FitView()
None<Key>x: PanRight()
Shift<Key>x: PanLeft()
None<Key>y: PanUp()
Shift<Key>y: PanDown()
None<Key>z: ZoomIn()
Shift<Key>z: ZoomOut()
Ctrl<Key>x: RotPosX()
Alt<Key>x: RotNegX()
Ctrl<Key>y: RotPosY()
Alt<Key>y: RotNegY()
Ctrl<Key>z: RotNegZ()
Alt<Key>z: RotPosZ()
None<Key>l: LabelsOn()
Shift<Key>l: LabelsOff()
Ctrl<Key>s: SpectrumOnOff()
<Key>osfUp:PanUp()
<Key>osfDown:PanDown()
<Key>osfLeft:PanLeft()
<Key>osfRight:PanRight()
None<Key>1: RearView()
None<Key>2: BottomView()
None<Key>3: Iso1View()
None<Key>4: LeftSideView()
None<Key>5: Iso2View()
None<Key>6: RightSideView()
None<Key>7: FrontView()
None<Key>8: TopView()
None<Key>9: Iso3View()
Ctrl<Btn3Down>: PopupGMenu(*selectMenu)
```


Copyright© 2012 MSC.Software Corporation

KEYMAPPING

- **On Unix/Linux there is a GUI for editing the .Patran.EventMaps file**
  - Preferences / Key Map
- **Windows users must edit the file manually**
- **CallPCL command allows for running custom PCL functions**



Copyright© 2012 MSC.Software Corporation

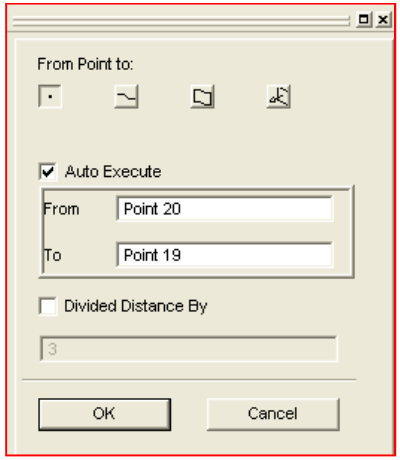
KEYMAPPING

- **Keymapping can be used to activate two “hidden” Patran tools**
- **Add the following entries to the Patran.EventMaps file**

Shift<Key>m: CallPCL(uil_measurement_display)
Shift<Key>c: CallPCL(uil_calculator_display)

  - **TIP:** If you are adding to the Patran.EventMaps file, make sure that you enter a carriage return after the last entry.

- **The keymaps defined call up the measurement and calculator tools, respectively**



Copyright© 2012 MSC.Software Corporation

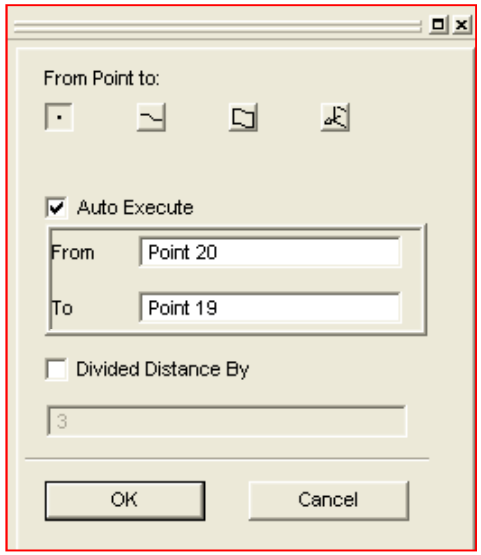
KEYMAPPING

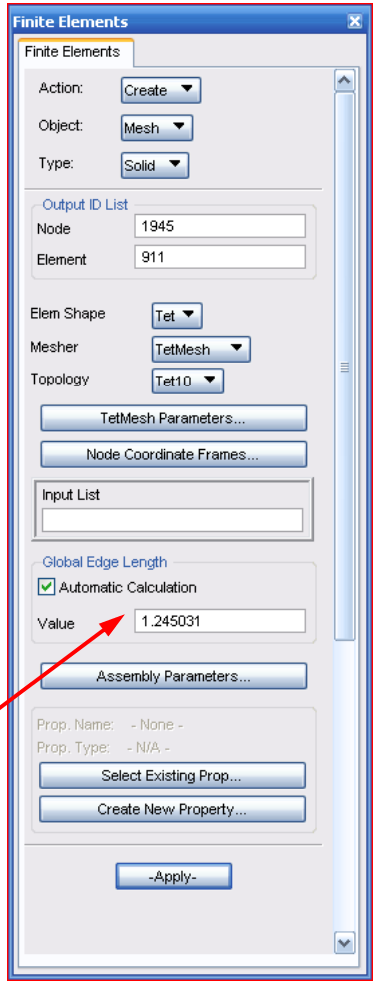
- **Calculator & measurement tools usage**
  - Display the form with a widget that requires measurement and/or calculation
  - Click in the specific databox you wish to fill with data
  - Click in the viewport
  - Hit Shift with M or C
  - Take your measurement / do the calculation
  - The calculator and measurement tools are available from any forms within Patran

Copyright© 2012 MSC.Software Corporation

KEYMAPPING

- **Example: measurement tool**





This value was filled automatically once the measurement was completed

Set GEL to length of this edge

Copyright© 2012 MSC.Software Corporation

KEYMAPPING

- **Some other useful keymaps**

  - Quickly change the current group. Each keypress changes current group to the next in the list of posted groups.

    - Shift<Key>g: CallPCL(au_switch_current_group.main)

  - Change the rectangle/polygon pick option

    - None<Key>q: CallPCL(au_rectpoly_centroid)
    - None<Key>w: CallPCL(au_rectpoly_partial)
    - None<Key>e: CallPCL(au_rectpoly_whole)

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

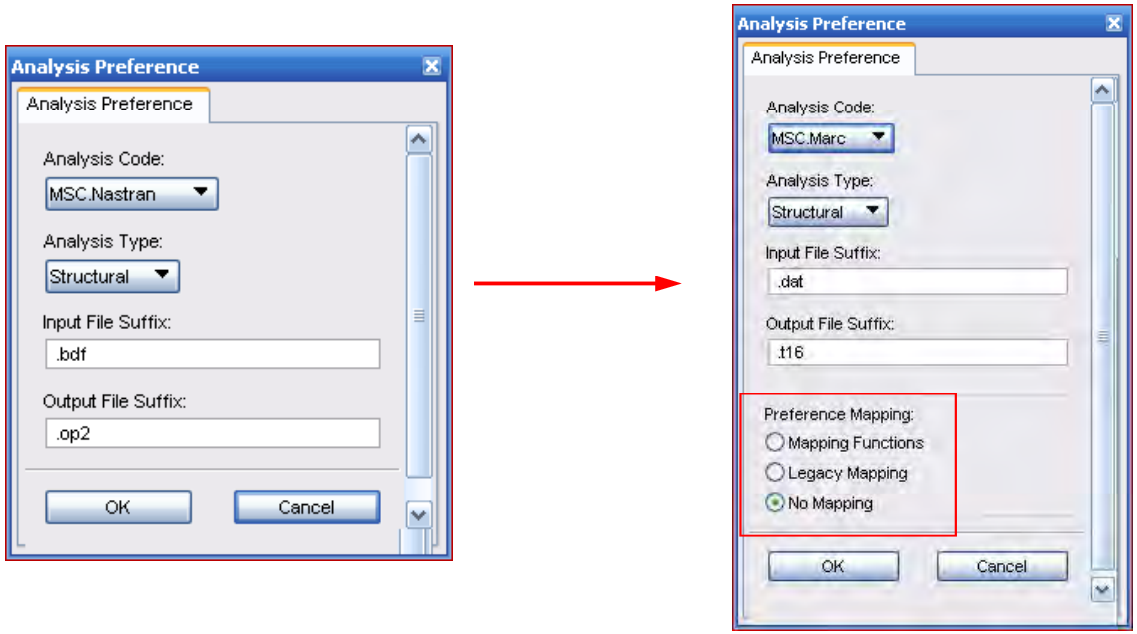
CHANGE SOLVER PREFERENCE

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

CHANGE PREFERENCE

- **If the finite element solver preference in Patran needs to be changed, e.g. Nastran -> MSC Marc, it will be desired to preserve material and element property, load and boundary condition, MPCs, and nonlinear characteristics (e.g. contact) information.**



Copyright© 2012 MSC.Software Corporation

CHANGE PREFERENCE

- **If it is desired to leave the database unchanged because the model is simple, or it is known that both solver preferences support the same parameters**
  - No Mapping
- **If it desired to convert only the simple code-specific information, such as element types and/or basic load and boundary conditions, to another solver preference**
  - Legacy Mapping
  - More complex items, i.e. MPC, material orientation, and nonlinear material property, are not converted
- **If it is desired to convert the complex as well as the simple parts of a model**
  - Mapping Functions

Copyright© 2012 MSC.Software Corporation

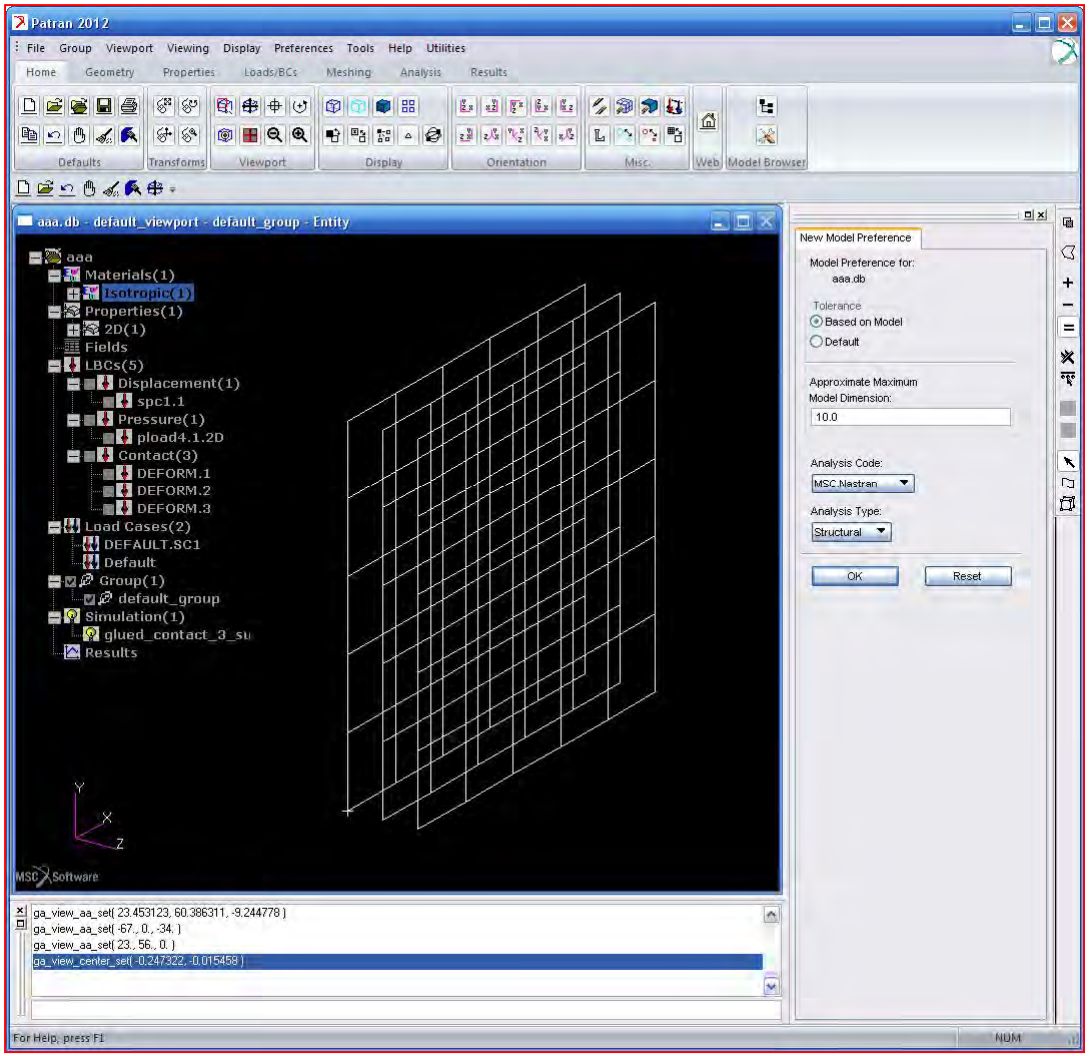
CHANGE PREFERENCE

- **If Mapping Functions is used**
  - Patran uses many mapping tables to convert code-specific information from one solver to another
    - Separate mapping tables are used for material properties and element properties
  - A limited number of mapping tables are included with the MSC.Patran delivery package
    - Nastran -> MSC Marc
  - Can find detailed information on the mapping tables in the Patran online user manual
  - Can access the latest information on the mapping tables at MSC's web site under product updates

Copyright© 2012 MSC.Software Corporation

MODEL BROWSER TREE

- The Model Browser Tree (MBT) allows users to quickly see what is in the current database in a user-friendly tree structure.
- Users can create, copy, paste, modify, or delete those database entities by right clicking on the elements of the tree to quickly navigate to different Patran forms.
- Available on Windows and Linux platforms

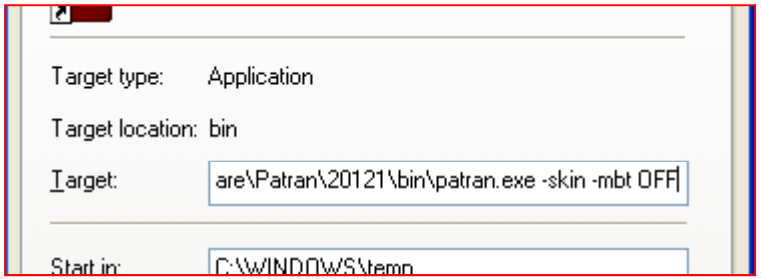


Copyright© 2012 MSC.Software Corporation

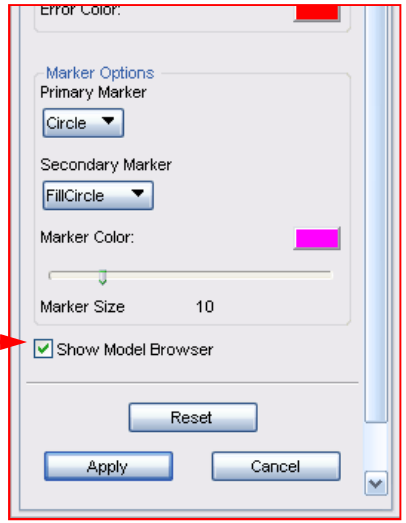
MODEL BROWSER TREE

- **The Model Browser Tree (MBT) is ON by default.**
- **To deactivate, add OFF after the –"mbt" command line argument. For example:**

```
C:\MSC.Software\Patran\20121\bin\patran.exe -skin -mbt OFF
```



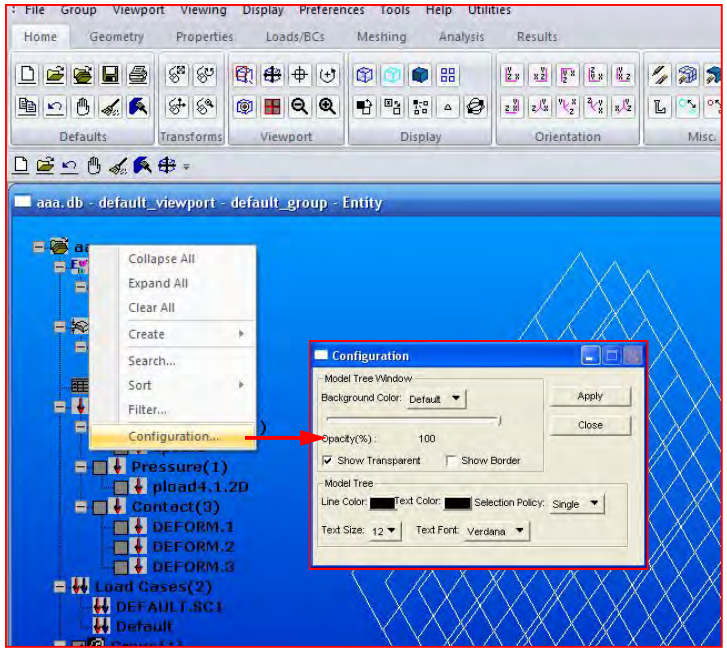
- **To show or hide the MBT, use the button on the Toolbar/Ribbon or the toggle in Preferences/Graphics**



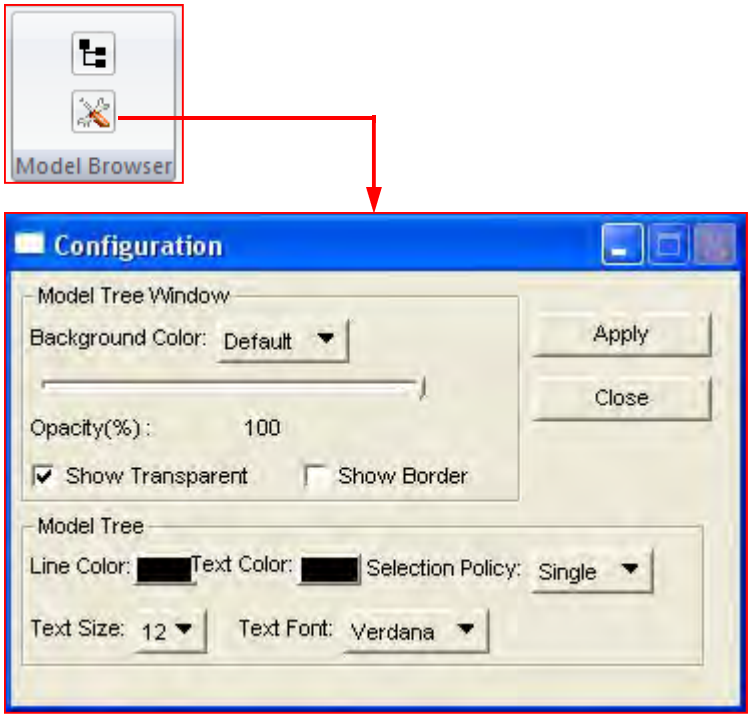
Copyright© 2012 MSC.Software Corporation

MODEL BROWSER TREE

- **To reposition the MBT, click on it with the middle mouse botton and drag to move to desired location**
- **To configure the MBT's appearance, right click on the tree and select "Configuration…" or click on the icon on the Toolbar/Ribbon**



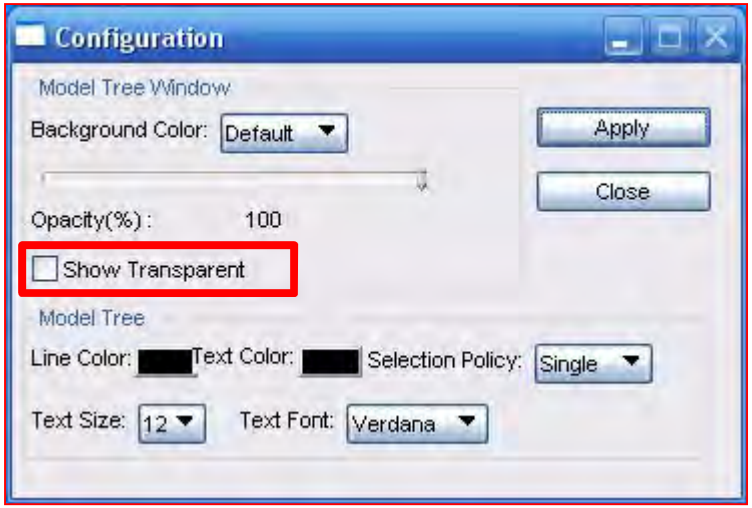
OR

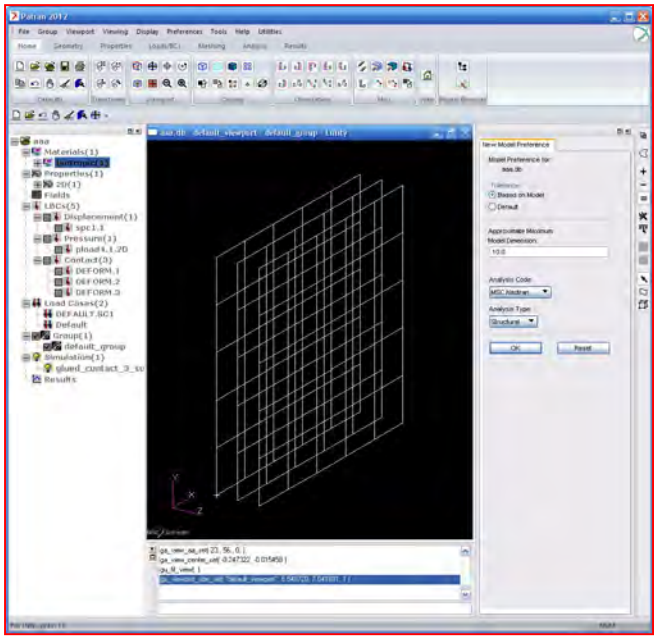


Copyright© 2012 MSC.Software Corporation

MODEL BROWSER TREE

- **The Model Browser Tree is defaulted to be transparent against any background**
- **To show the tree in a dock-able window format, uncheck the "Show Transparent" toggle and click Apply.**

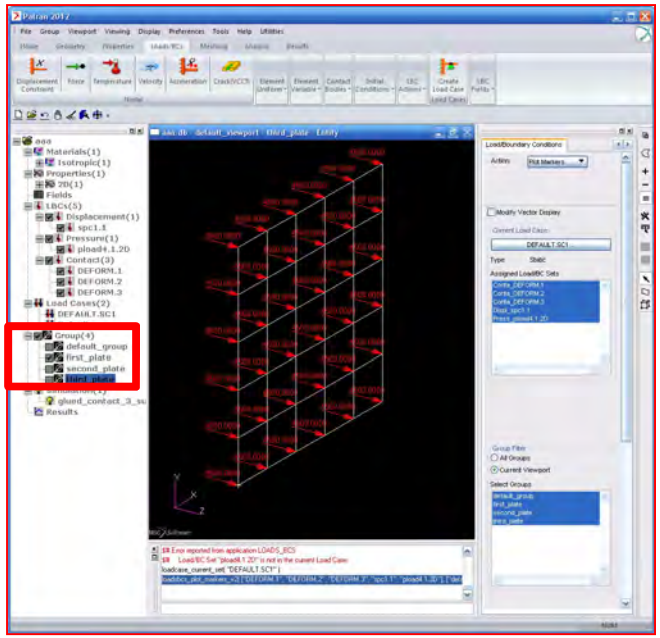


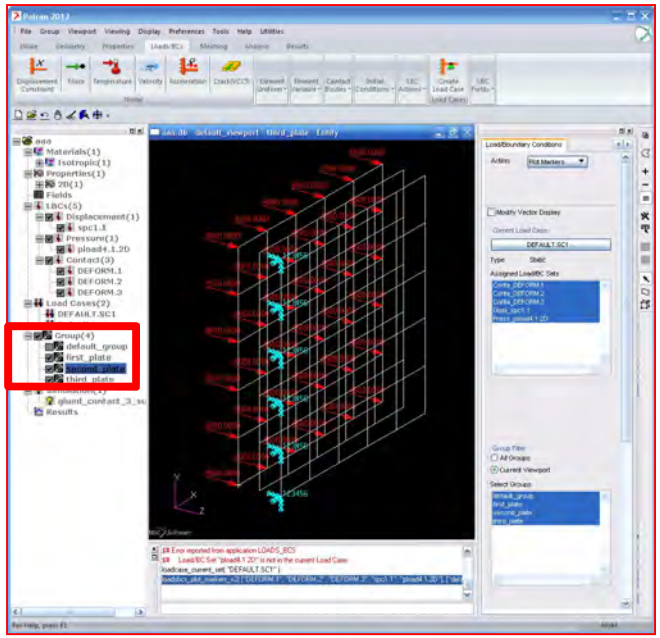


Copyright© 2012 MSC.Software Corporation

MODEL BROWSER TREE

- **Database entities which support post/unpost capabilities can be toggled ON/OFF. This is most useful with Groups, which can be toggled ON/OFF quickly via the Model Browser Tree.**





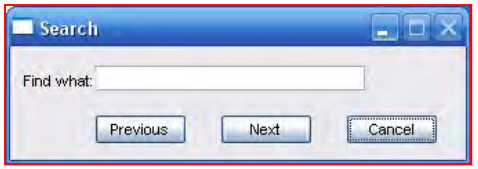
Copyright© 2012 MSC.Software Corporation

MODEL BROWSER TREE

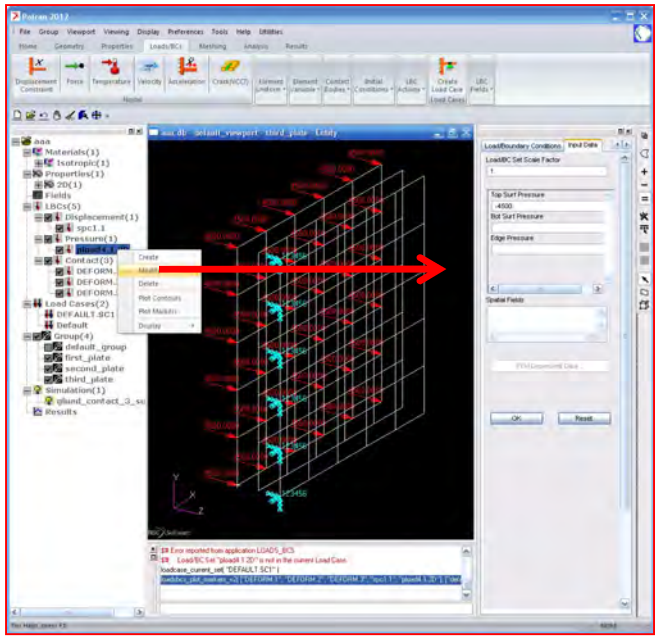
- **Select and highlight by mouse left click or with “shift” and “ctrl” keys for multiple selection.**
- **Vertical and horizontal scrollbars will appear when items go beyond the tree view window.**
- **Drag and drop is supported for certain tree items when creating associativities**

Copyright© 2012 MSC.Software Corporation

MODEL BROWSER TREE



- **Popup menus are context-sensitive**
- **Right-click on a selected tree item to:**
  - Search for, sort, and filter tree entities
  - Create, Modify, or Delete entities from the tree (i.e. materials, properties, LBC's, load cases, fields, groups, simulations (analysis jobs) and results plots).



Copyright© 2012 MSC.Software Corporation

EXERCISES

- Perform Workshop 15 “Editing the Toolbar” in your exercise workbook.
- Perform Workshop 16 “Keymapping” in your exercise workbook.
- Optional: try out some of the other customizations presented in this section.

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

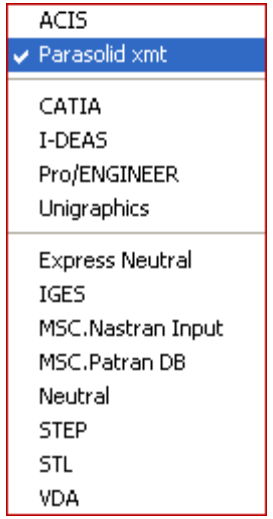
SECTION 7

IMPORT CAD GEOMETRY INTO PATRAN

Copyright© 2012 MSC.Software Corporation

GEOMETRY IMPORT METHODS

- **Following are the different types of geometry that can be imported into Patran:**
  - ACIS or Parasolid xmt kernel (engine) geometry; can come from any source that produces geometry using this kernel.
  - CATIA, I-DEAS, Pro/ENGINEER, or Unigraphics
  - Express Neutral is a standard format used to create an intermediate file that Patran can read,
    e.g. CATIA -> Express Neutral file -> Patran.
  - IGES is an older standard format for transmitting geometry.
  - Neutral supports only Patran parametric cubic geometry.
  - STEP is a modern standard format.
  - STL supports sterolithography data models.



Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

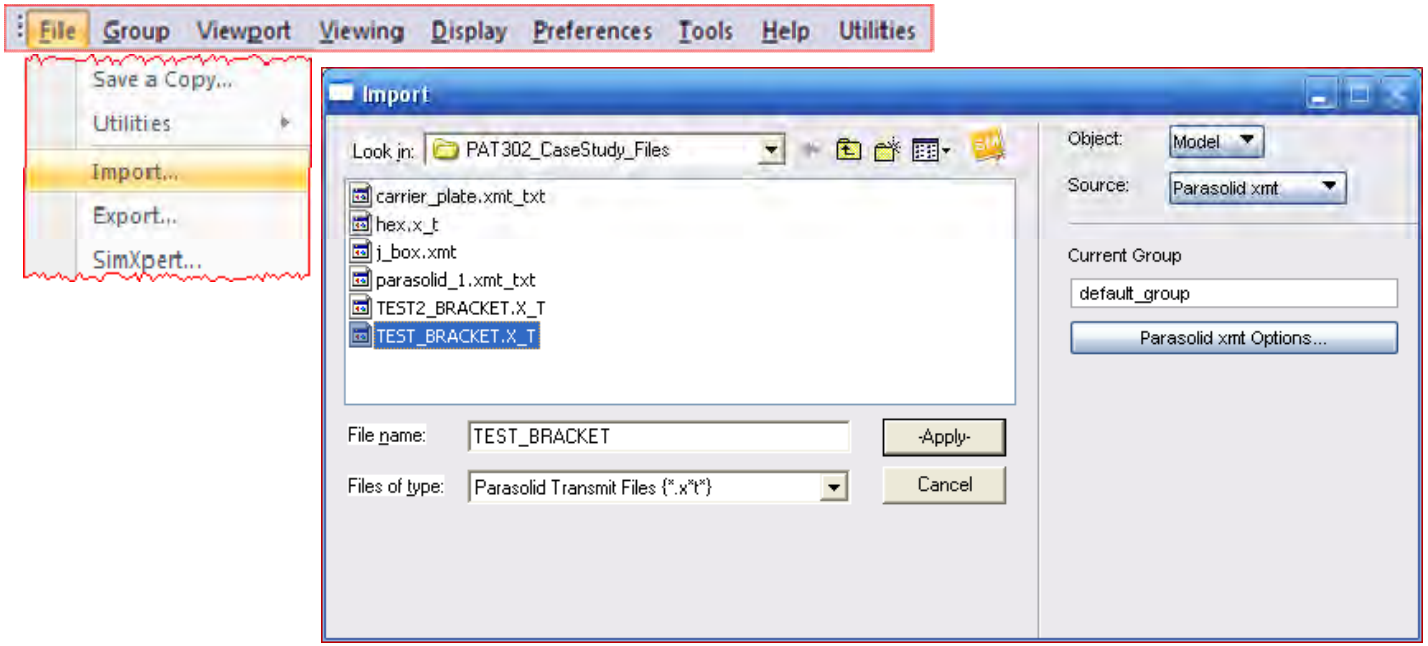
IMPORT PARASOLID XMT FILE

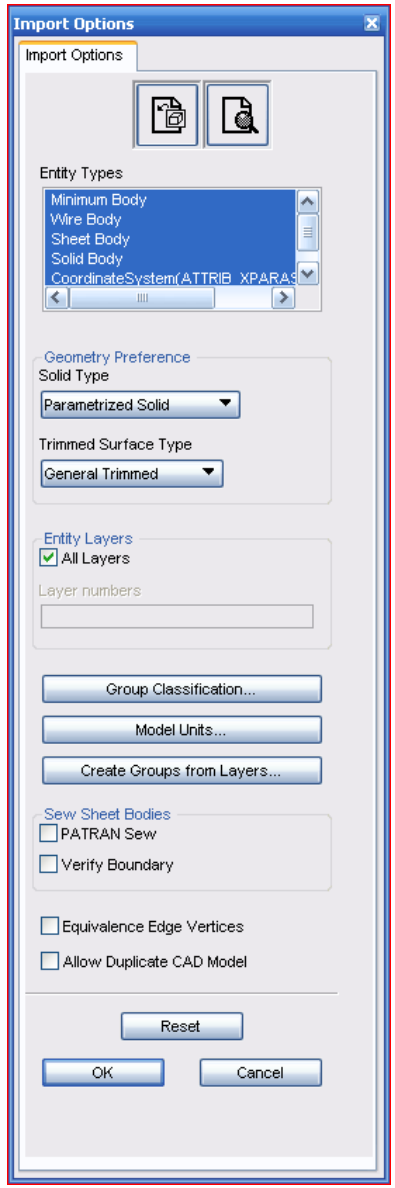
Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

- **Import a Parasolid xmt file, TEST_BRACKET.X_T**
  - Use default Parasolid xmt Options parameter values.

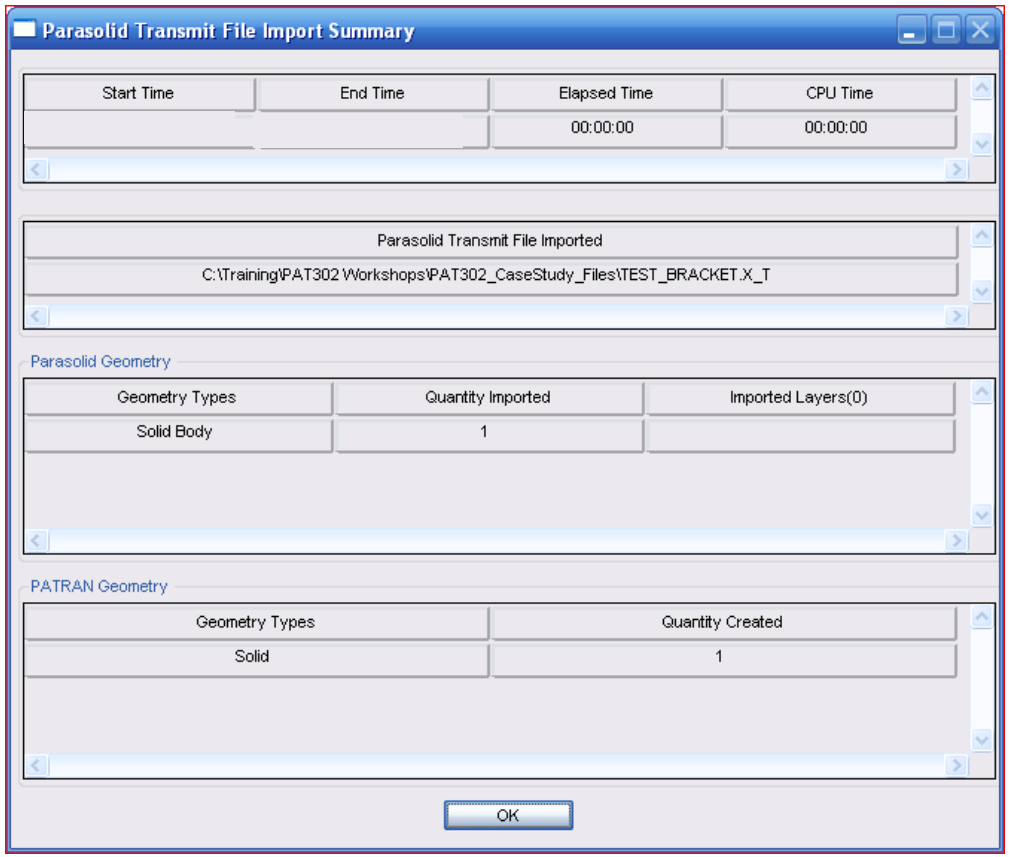




Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

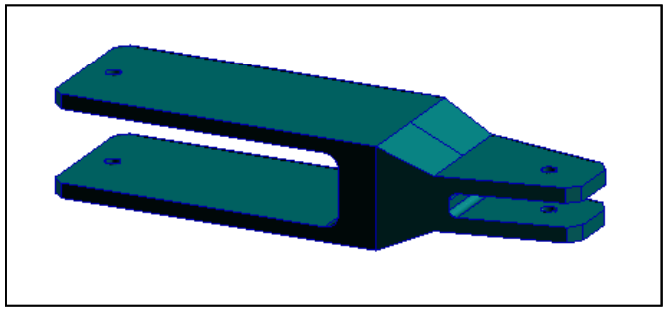
- **The import summary indicates that there was one Solid Body, and one Solid created in Patran.**

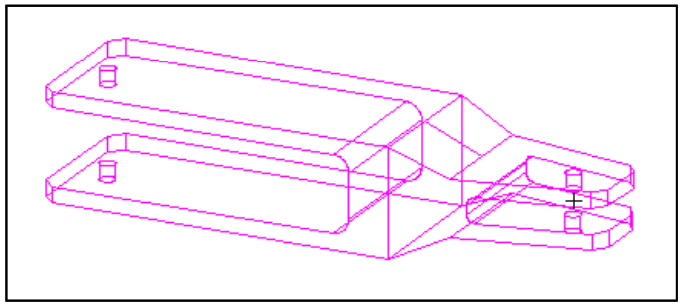


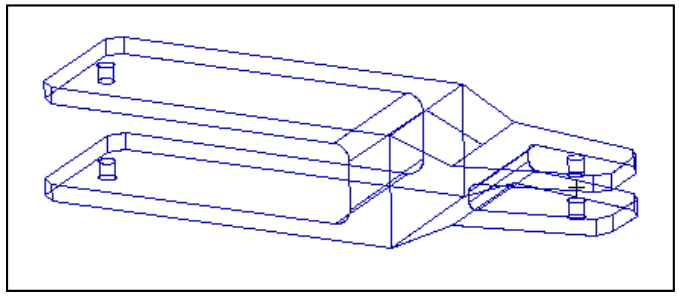
Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

- **This is one solid (B-rep. Parasolid) with 74 vertices and 41 faces.**
- **When the solid is disassembled into surfaces, they are congruent (Geometry: Verify/Surface/Boundary)**
  - $# There are no free surface edges.



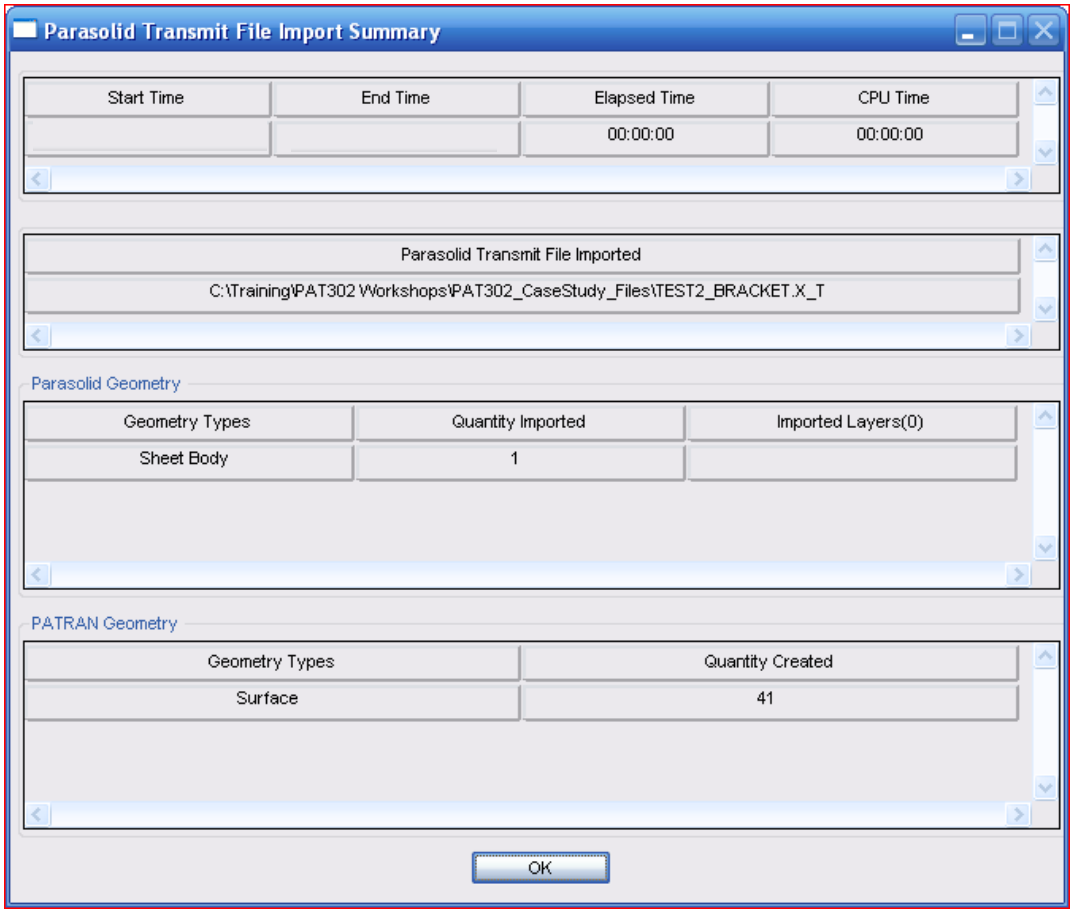




Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

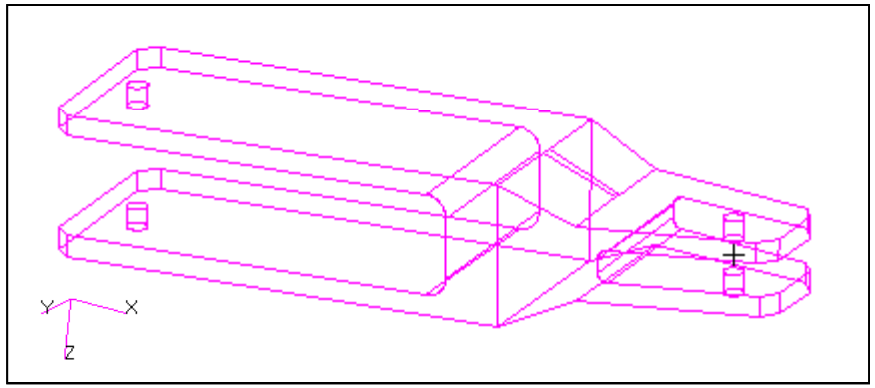
- **Import another Parasolid file, TEST2_BRACKET.X_T; use default Parasolid xmt Options parameter values.**
- **The import summary indicates that one Sheet Body, and 41 Surfaces were created in Patran**
  - Only one sheet body was created because the 41 Parasolid surfaces were “sewed” together to create a single “sheet solid” in CAD.



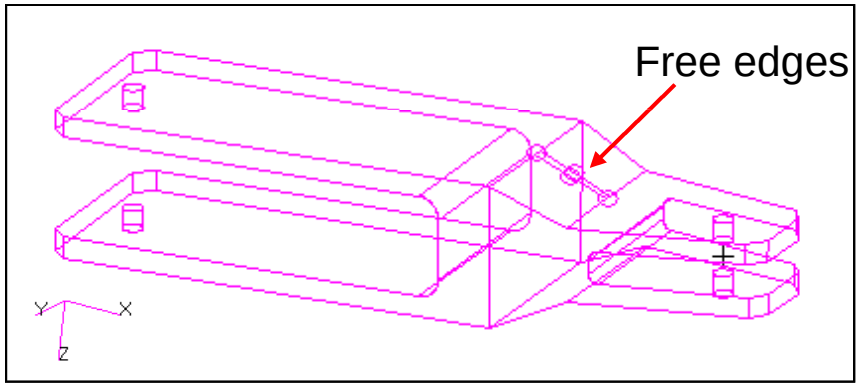
Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

- **Surfaces created from import, and the result of Verify for them**



41 Surfaces From Import

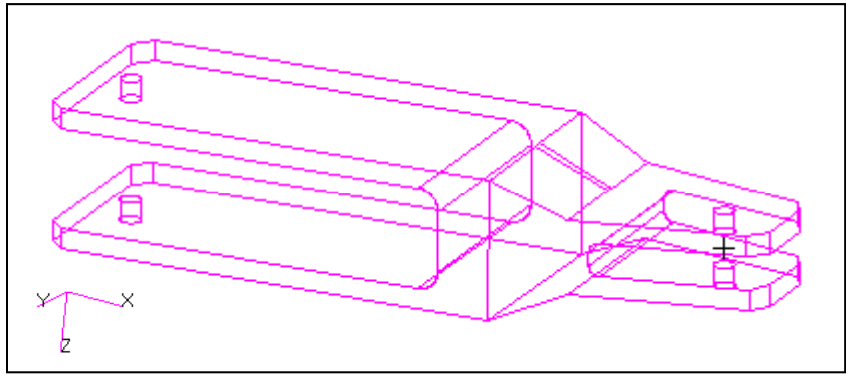


Geometry: Verify/Surface/Boundary

Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

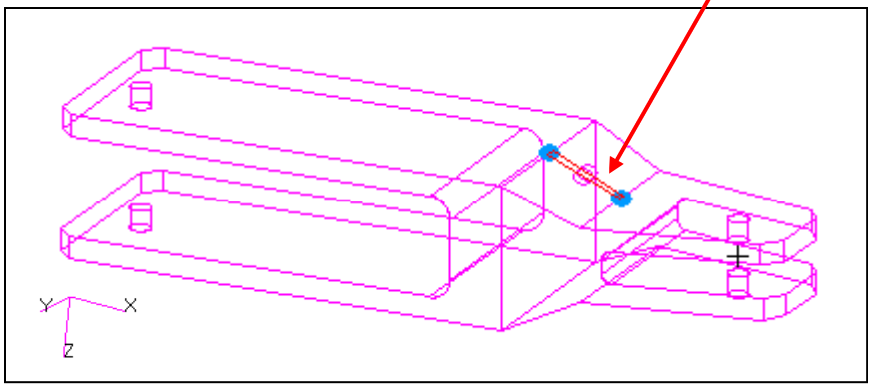
- **Use Geometry: Edit/Surface/Sew for surfaces from import.**



Sew Surfaces – Adds Vertices and/or Equivalences Vertices

Equivalencing geometric vertices is like equivalencing FEM nodes; the remaining vertices are used to define the geometry (e.g. surfaces).

Free and nonmanifold edges present



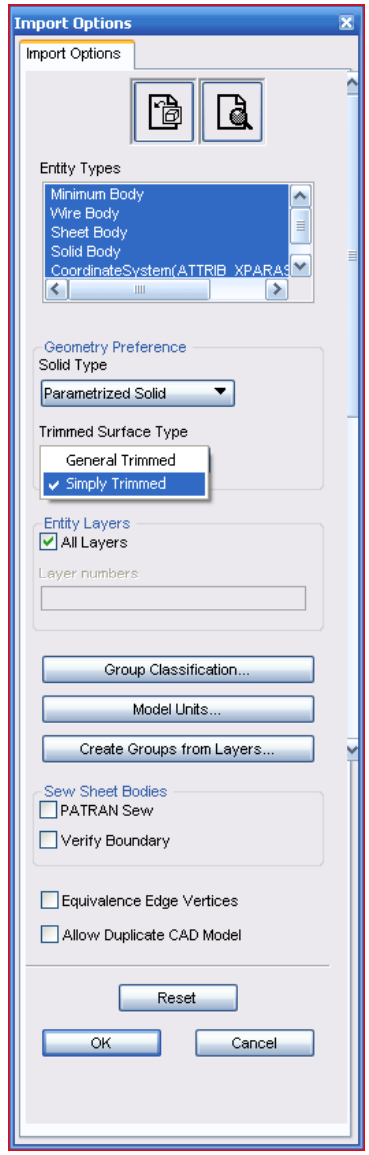
Surfaces Still not Congruent After Sewing

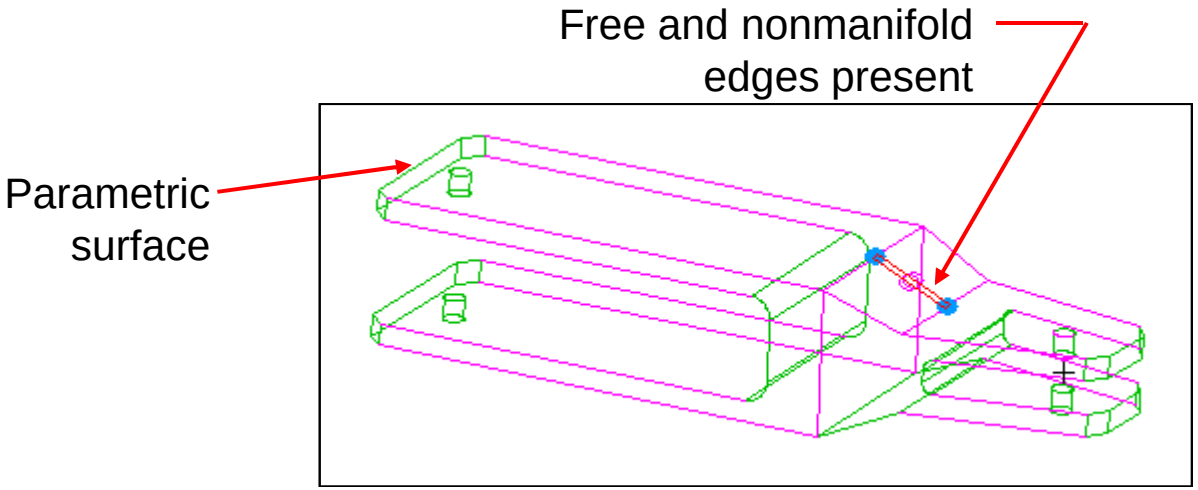
It will be necessary to manually resolve the incongruency issues, e.g. break and delete surfaces.

Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

- **Re-import the Parasolid file TEST2_BRACKET.X_T**
- **This time, under Parasolid xmt Options, select:**
  - Simply Trimmed – when possible, creates parametric surfaces (green)
  - Patran Sew – Geometry: Edit / Surface / Sew
  - Verify Boundary – Geometry: Verify / Surface / Boundary

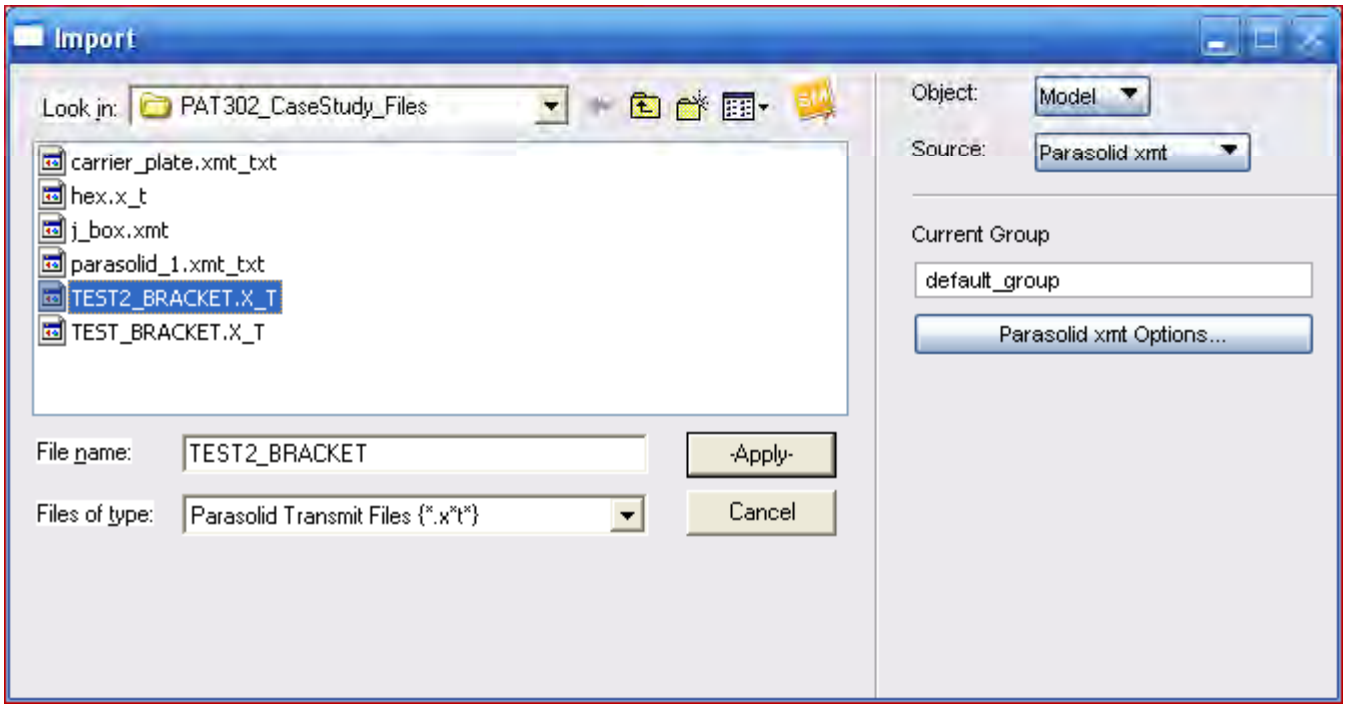


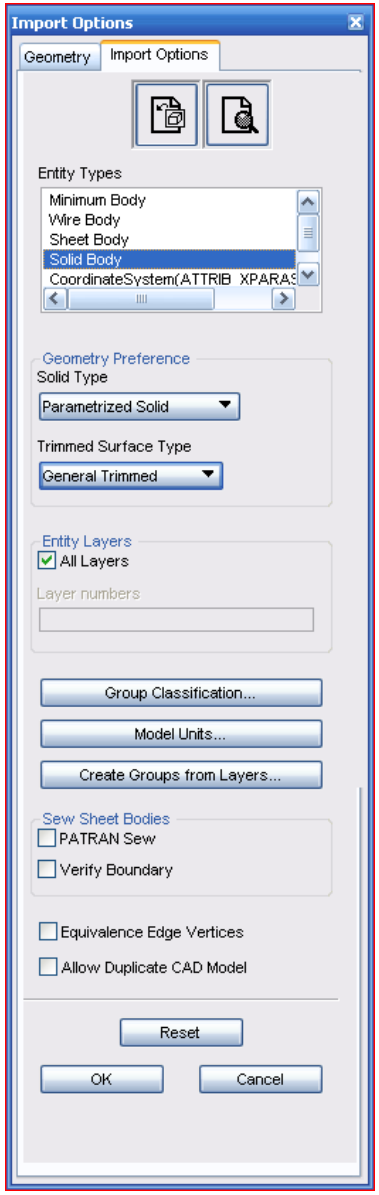


Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

- **Re-import TEST2_BRACKET.X_T**
- **Set Entity Types, under Parasolid xmt Options, to be only Solid Body.**

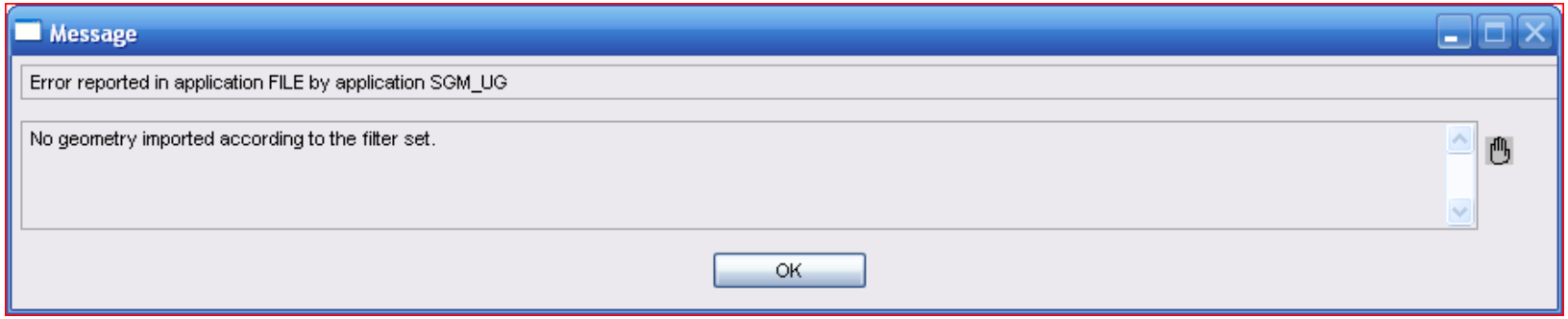




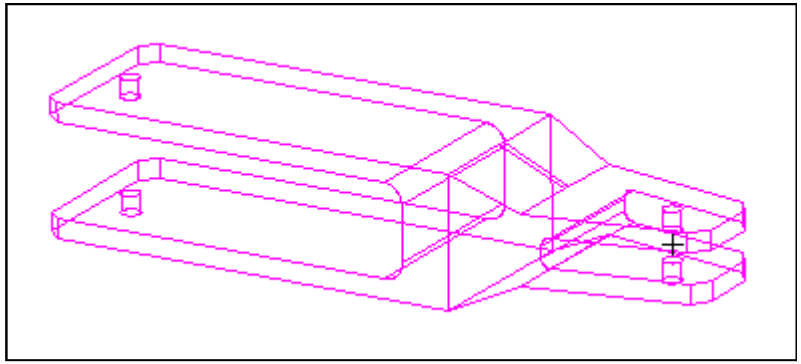
Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

- **The following error message appears:**



- **Change import Entity Types to Sheet Body.**
  - The sheet body is imported, giving 41 surfaces.



Copyright© 2012 MSC.Software Corporation

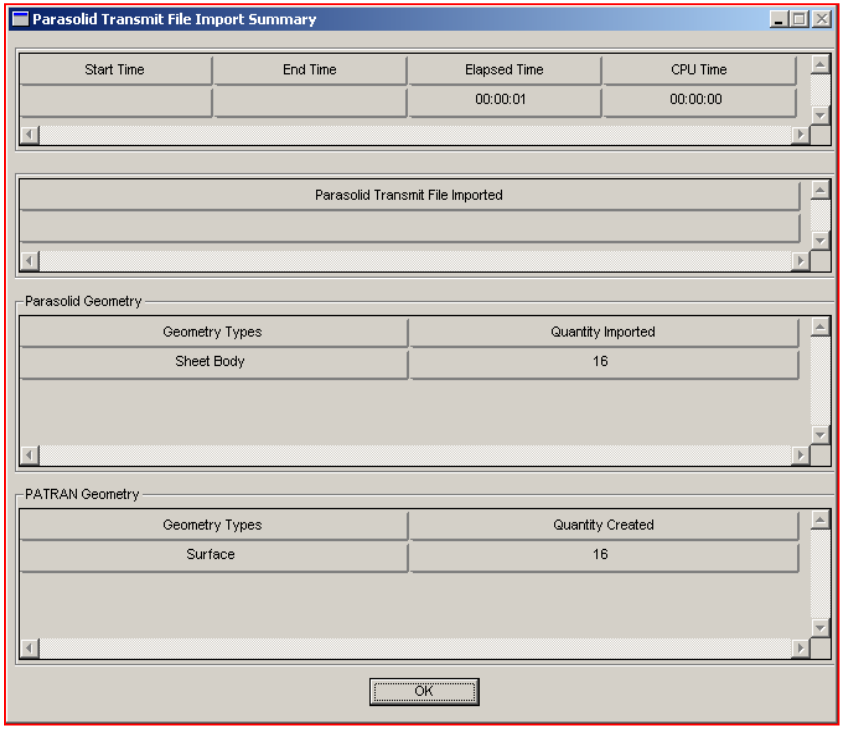
IMPORT PARASOLID XMT FILE

- **Note: Even though file TEST2_BRACKET.X_T is a Parasolid file, the geometry in it is not that of a solid, but a set of surfaces. This is why the import failed when the Entity Types was set to Solid Body, and succeeded when it was set to Sheet Body.**
  - The file was created by a CAD program that recognized the geometry as what is called a “sheet solid”.

Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

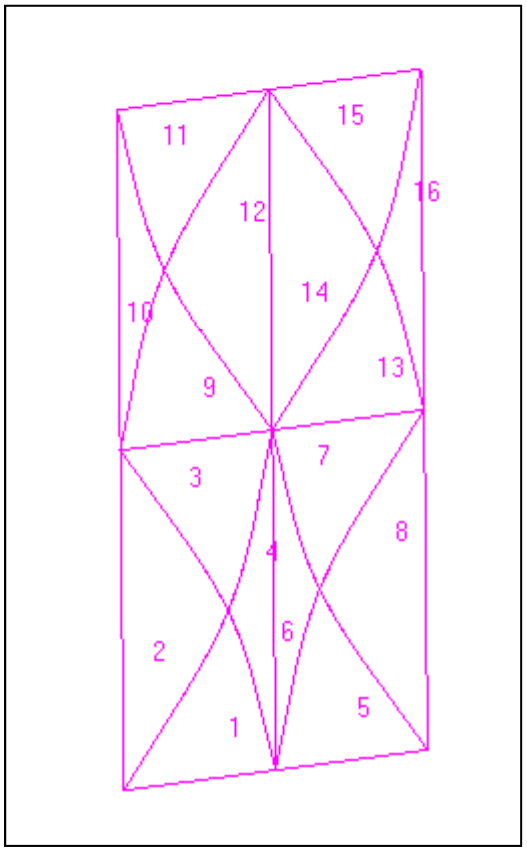
- **Import a set of Parasolid surfaces (sheet bodies) that were created as separate entities.**
- **Parasolid Geometry, 16 sheet bodies**
- **Patran Geometry, 16 surfaces**

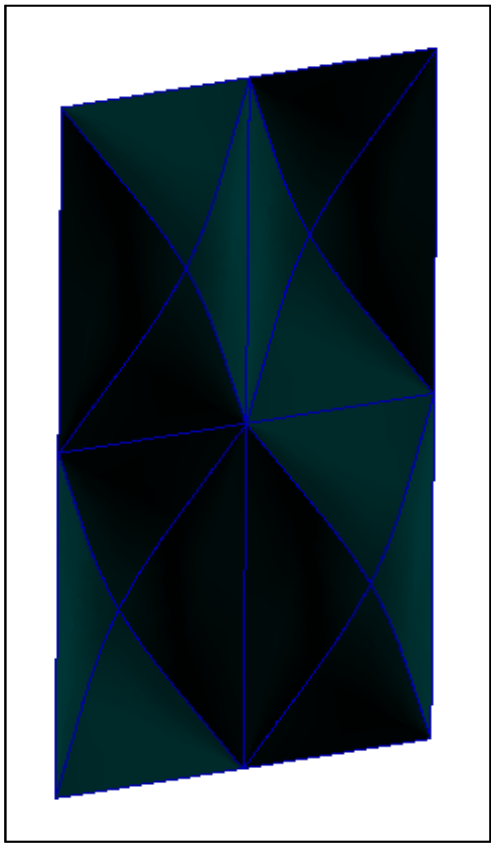


Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

- **The sixteen (curved) surfaces, unshaded and shaded:**





Copyright© 2012 MSC.Software Corporation

IMPORT PARASOLID XMT FILE

- **A Wire Body is:**
  - One step up from a “minimum body”
  - One dimensional, topologically
  - Set of connected edges
    - Open Wire Body has two ends
    - Closed Wire Body has no ends
- **A Wire Body must consist of a single void region.**
- **Every vertex in a Wire Body must be used to define either one or two edges.**
- **A Wire Body in Patran is represented by a Curve.**

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

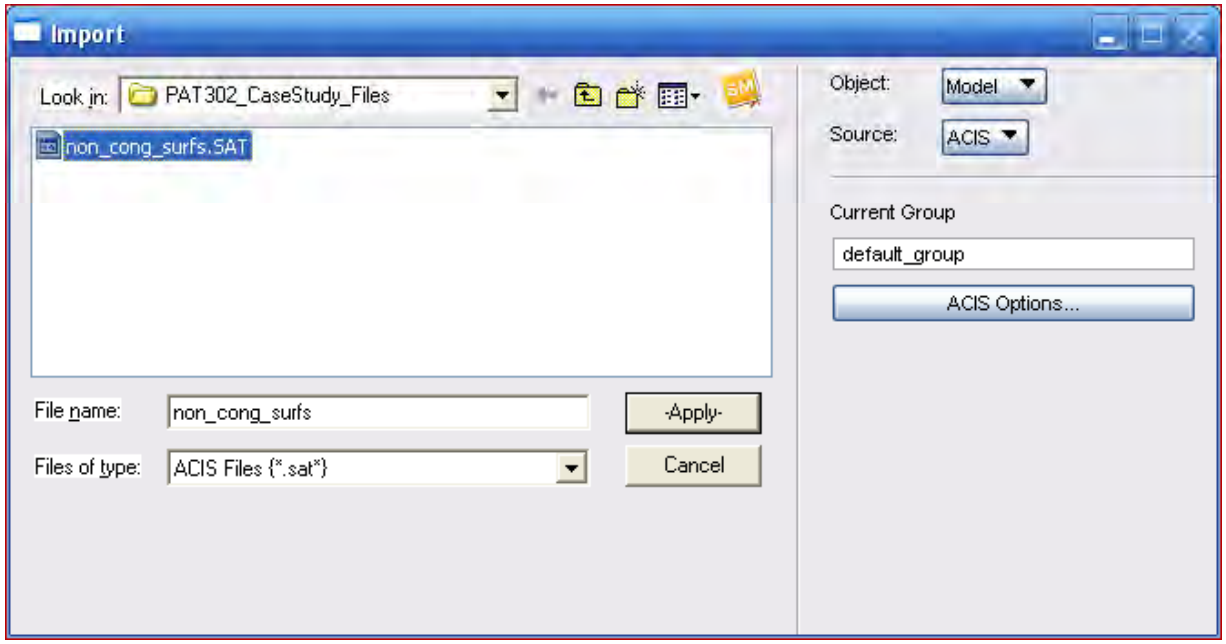
IMPORT ACIS FILE

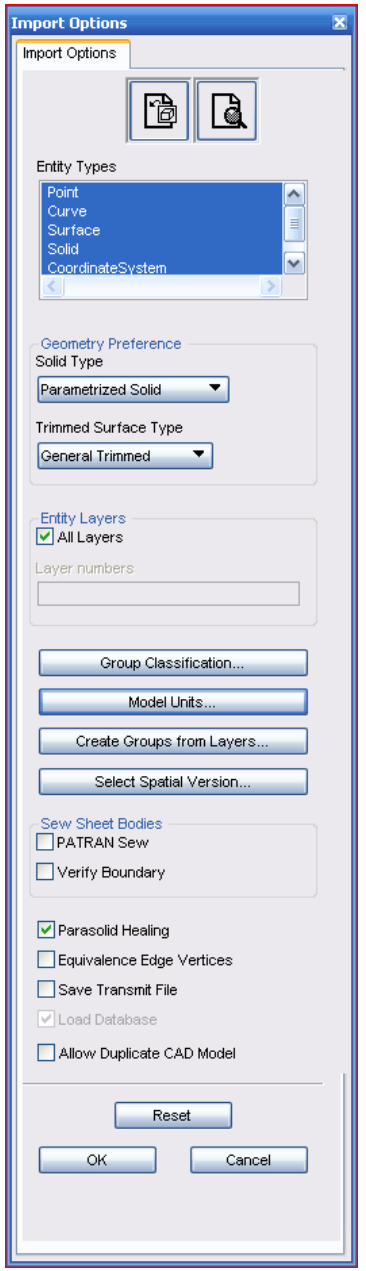
Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

- **Import ACIS file, non_cong_surfs.SAT**
  - Initially, unselect all ACIS Options parameters.
    - This includes using None for the model units.

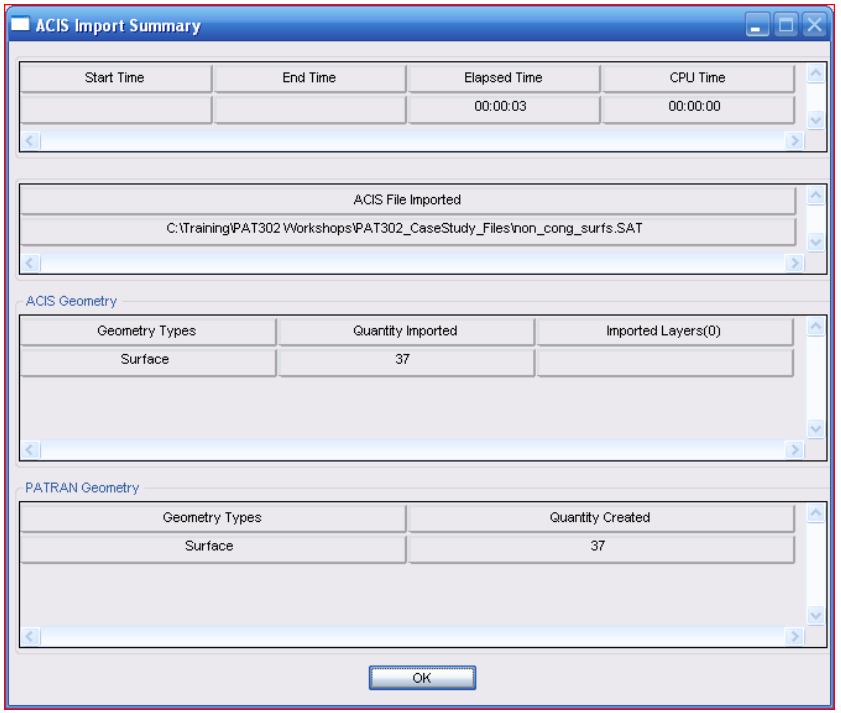




Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

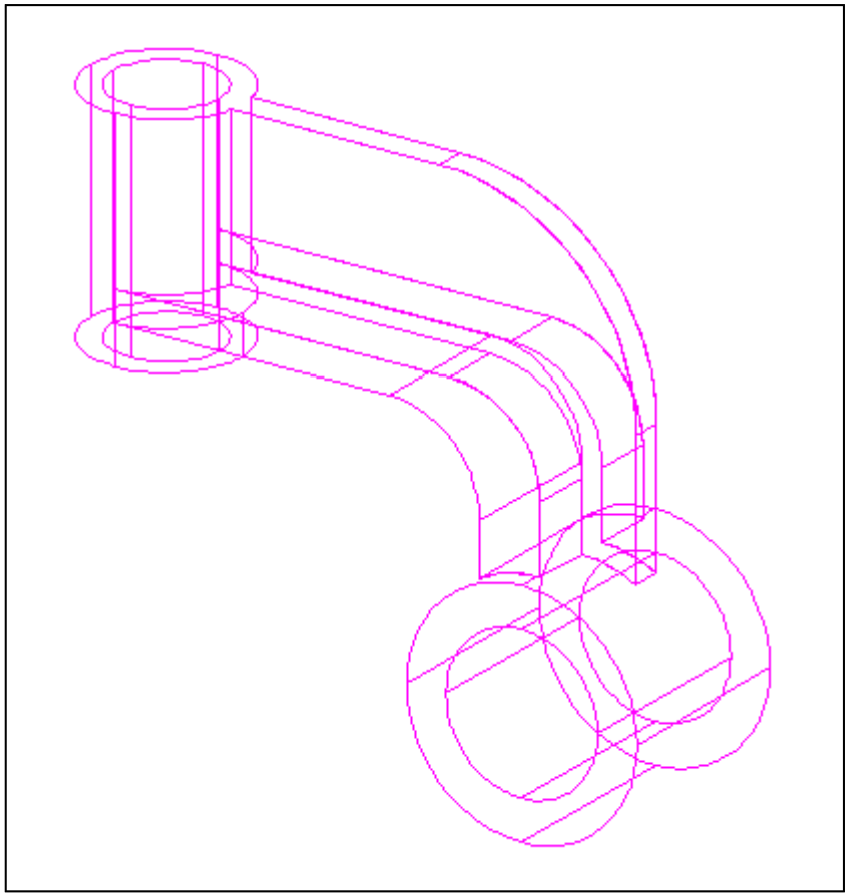
- **Import a set of ACIS surfaces (sheet bodies) that were created as separate entities.**
- **ACIS Geometry, 37 surfaces (sheet bodies)**
- **Patran Geometry, 37 surfaces**



Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

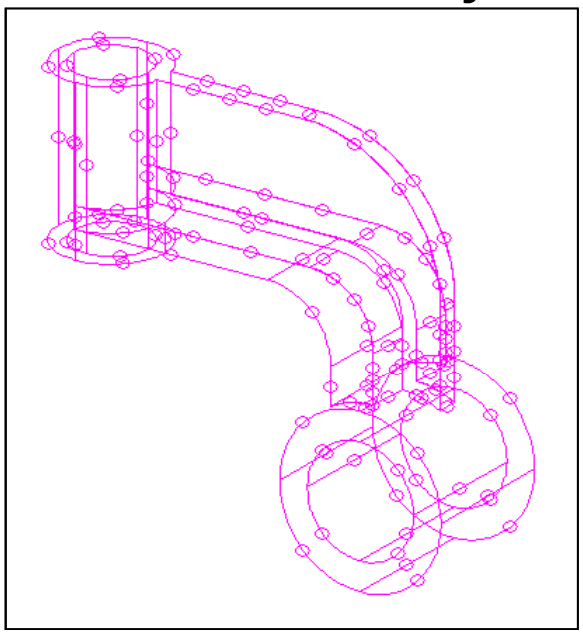
- **The imported trimmed surfaces (37)**

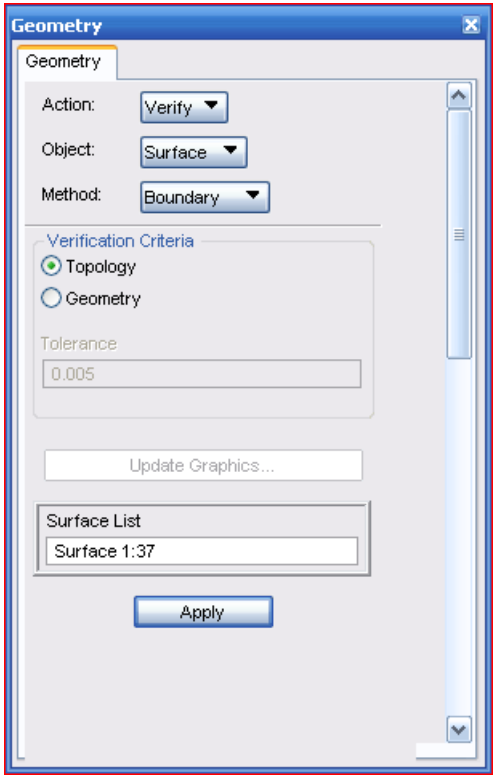


Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

- **Determine the areas of incongruency using Geometry: Verify / Surface / Boundary**



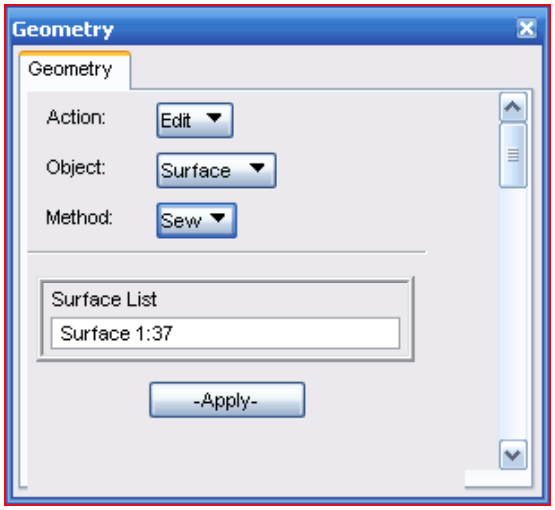


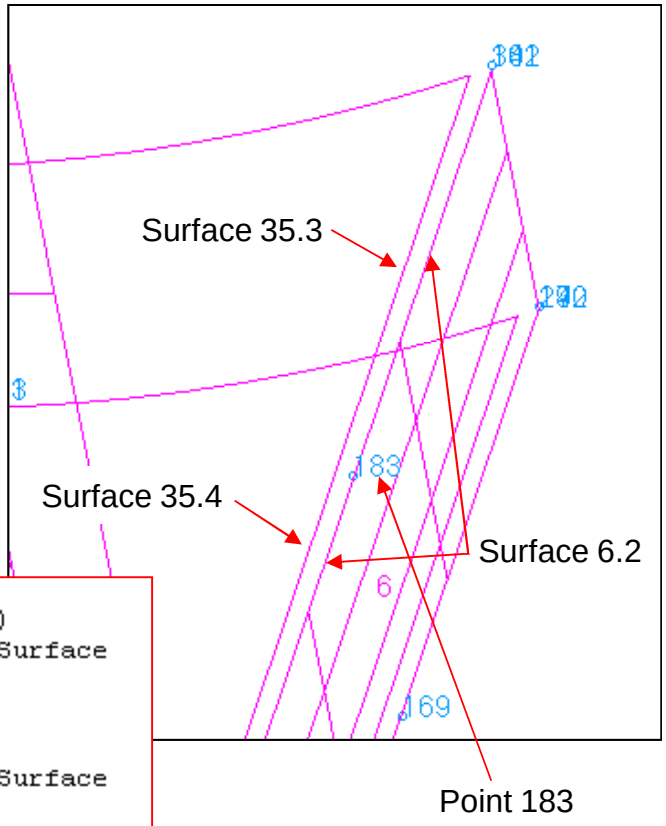

```
sgm_verify_surface_boundary( "Surface 1:37", 0.0088047404, 1 )
$# Warning reported from application SGM
$#     Free edges and/or non-manifold edges exist. Free edges may be due to
$# edges not matching directly or a missing face.  Non-manifold edges are due
$# to more than two surfaces connecting at the same edges. Edges are
$# highlighted and marked with primary markers for free edges and secondary
$# markers for non-manifold edges.
```


Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

- **Repair the incongruent surfaces, using Geometry: Edit/Surface/Sew**

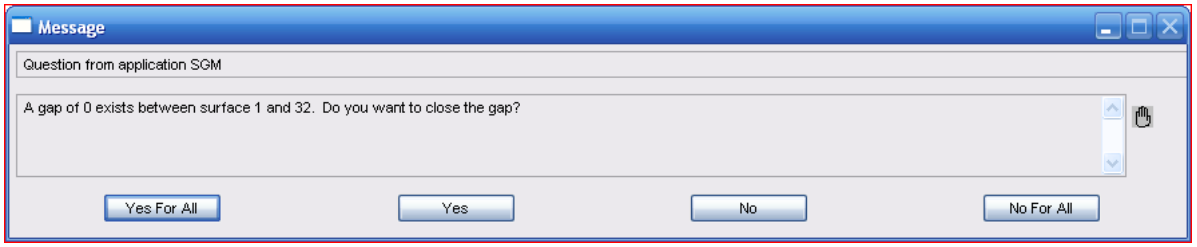


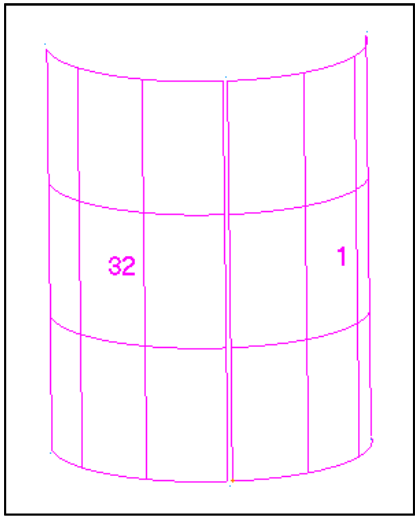



```
sgm_edit_surface_sew( "Surface 1:37", sgm_edit_surface_se_surface_ids )
$# T-Edge Surface 6.2 broken at Point 183 into edges Surface 35.3 and Surface
$# 35.4.
$# T-Edge Surface 6.5 broken at Point 169 into edges Surface 34.11 and
$# Surface 34.1.
$# T-Edge Surface 7.2 broken at Point 194 into edges Surface 36.3 and Surface
$# 36.4.
...
```

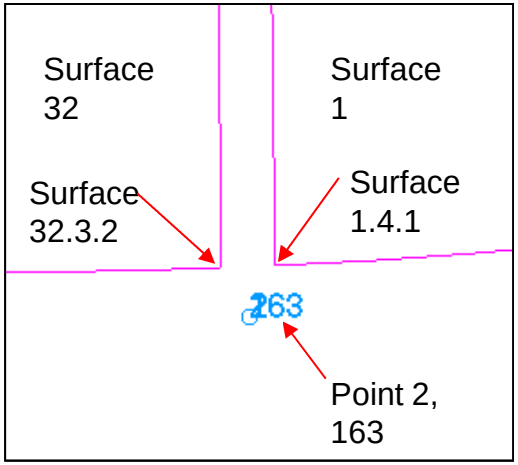

Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE





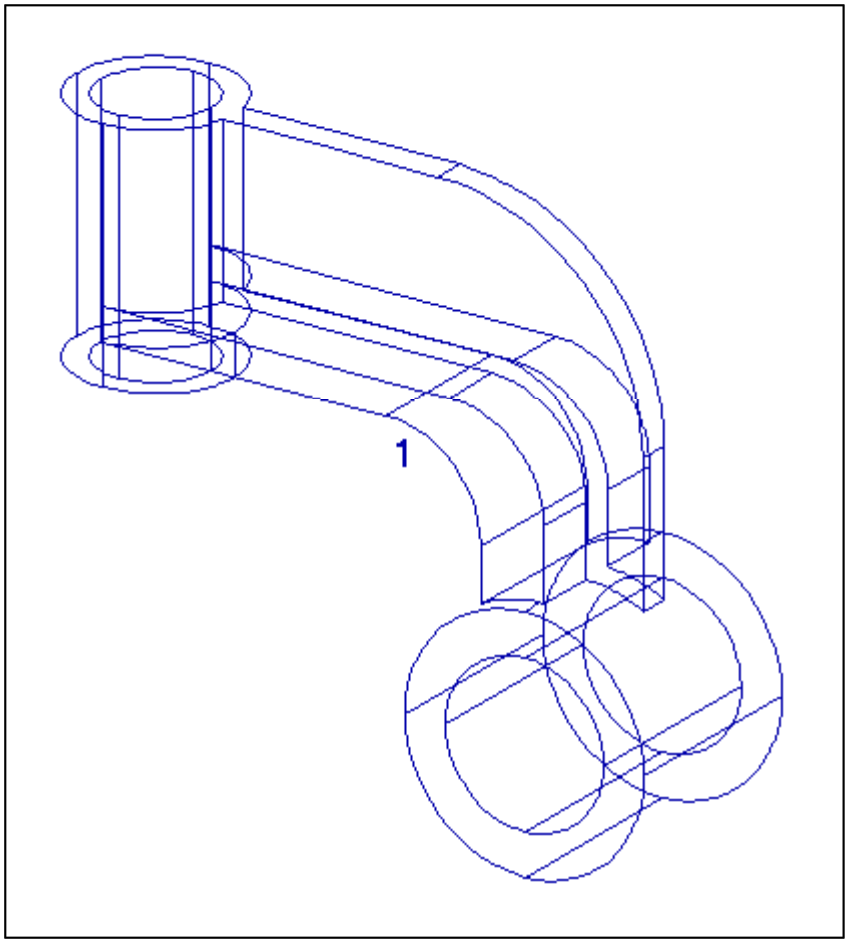

```
$# Question from application SGM
$#     A gap of 0 exists between surface 1 and 32.  Do you want to close the
$# gap?
$? YESFORALL 38001105
$# Gap edges Surface 1.4 and Surface 32.3 merged into edge Surface 1.4.
$# Gap edges Surface 1.3 and Surface 5.5 merged into edge Surface 1.3.
$# Gap edges Surface 1.2 and Surface 31.1 merged into edge Surface 1.2.
...
```



Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

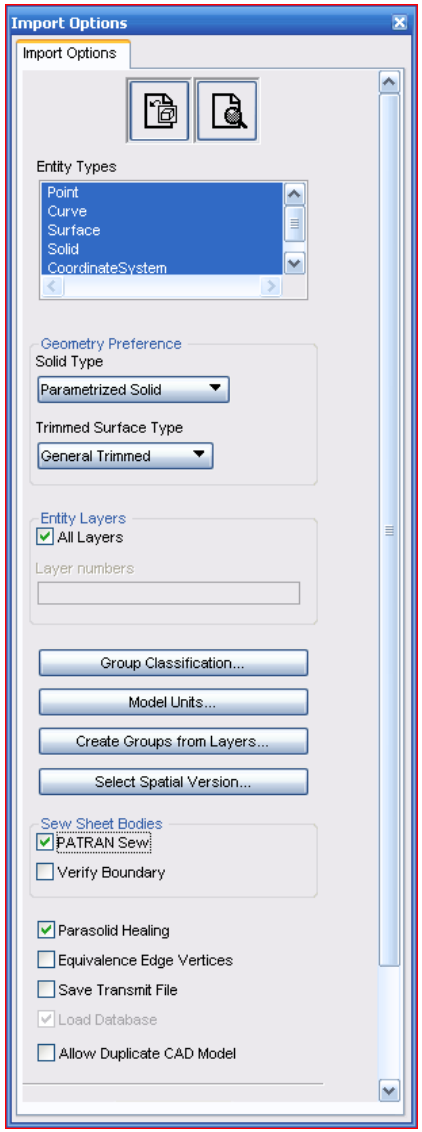
- **Create a B-rep solid**
  Geometry: Create/Solid/B-rep



Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

- **Create a new Patran database and import the ACIS file again, but this time, toggle on the import parameter PATRAN Sew.**



Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

- **The following is from the session file:**

```
$# T-Edge Surface 8.1 broken at Point 185 into edges Surface 35.5 and Surface 35.6.
$# T-Edge Surface 8.4 broken at Point 178 into edges Surface 34.9 and Surface 34.10.
$# T-Edge Surface 19.2 broken at Point 176 into edges Surface 34.7 and Surface 34.8.
$# T-Edge Surface 9.4 broken at Point 214 into edges Surface 37.11 and Surface 37.12.
$# T-Edge Surface 10.3 broken at Point 212 into edges Surface 37.9 and Surface 37.10.
$# T-Edge Surface 12.3 broken at Point 198 into edges Surface 36.7 and Surface 36.8.
$# T-Edge Surface 16.3 broken at Point 196 into edges Surface 36.5 and Surface 36.6.
$# T-Edge Surface 11.3 broken at Point 210 into edges Surface 37.7 and Surface 37.8.
$# T-Edge Surface 19.5 broken at Point 187 into edges Surface 35.7 and Surface 35.8.
$# Gap edges Surface 1.4 and Surface 32.3 merged into edge Surface 1.4.
$# Gap edges Surface 1.3 and Surface 5.5 merged into edge Surface 1.3.
$# Gap edges Surface 1.2 and Surface 31.1 merged into edge Surface 1.2.
$# Gap edges Surface 1.1 and Surface 4.7 merged into edge Surface 1.1.
$# Gap edges Surface 2.4 and Surface 29.3 merged into edge Surface 2.4.
$# Gap edges Surface 2.3 and Surface 19.10 merged into edge Surface 2.3.
$# Gap edges Surface 2.2 and Surface 28.1 merged into edge Surface 2.2.
```

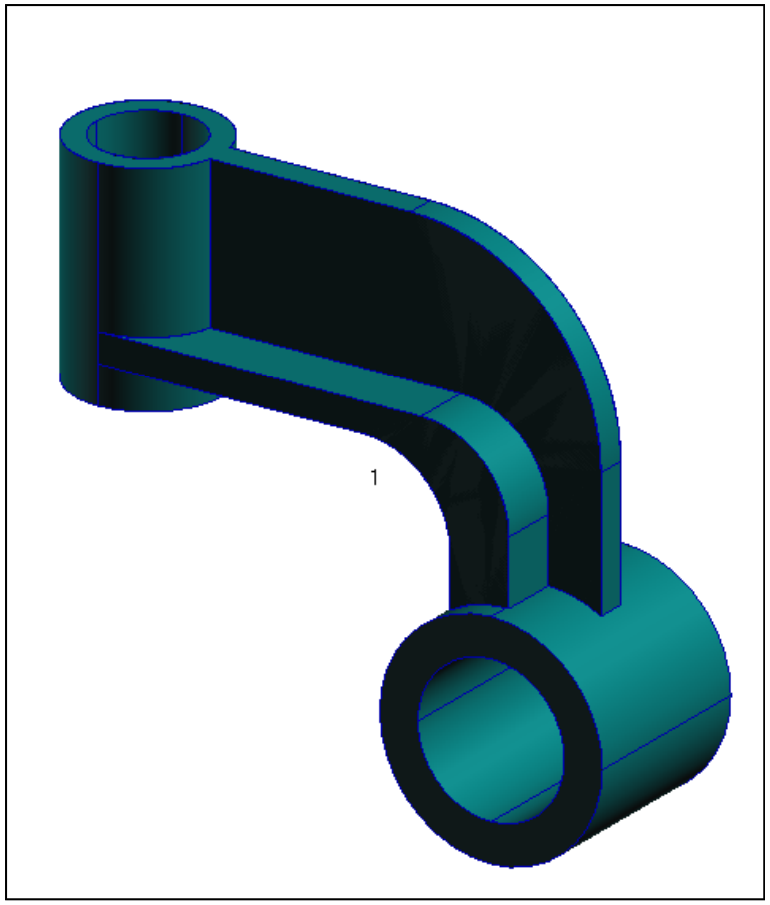
- **Determine the areas of incongruency, using Geometry: Verify/Surface/Boundary**

```
sgm_verify_surface_boundary( "Surface 1:37", 0.0049999999, 1 )
$# There are no free surface edges.
```


Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

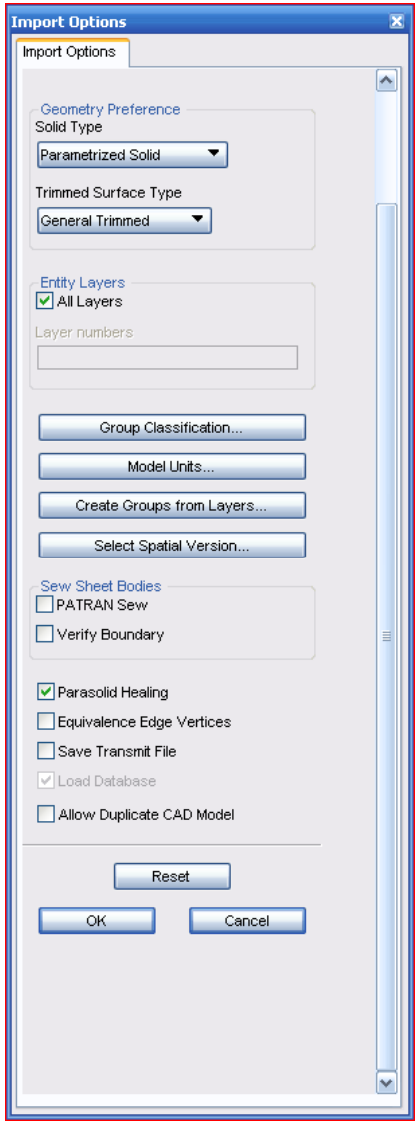
- **The B-rep solid created after editing the surfaces, using Sew:**



Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

- **The import parameters are used for the following purpose**
  - PATRAN Sew – Geometry: Edit/Surface/Sew
  - Verify Boundary – Geometry: Verify/Surface/Boundary
  - Parasolid Healing – repairs created Parasolid
  - Equivalence Edge Vertices – causes topological vertices within tolerance to be equivalenced



Copyright© 2012 MSC.Software Corporation

IMPORT ACIS FILE

- **If there are problems importing an ACIS file, Parasolid Healing may be used for better results.**

Copyright© 2012 MSC.Software Corporation

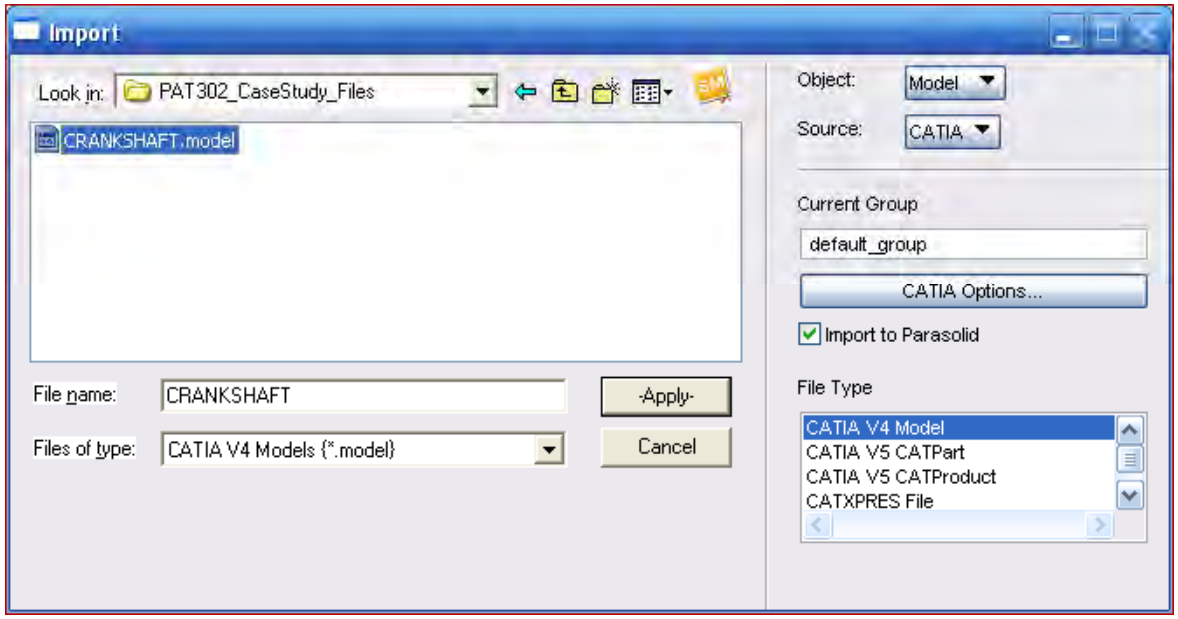
IMPORT CATIA MODEL FILE

Copyright© 2012 MSC.Software Corporation

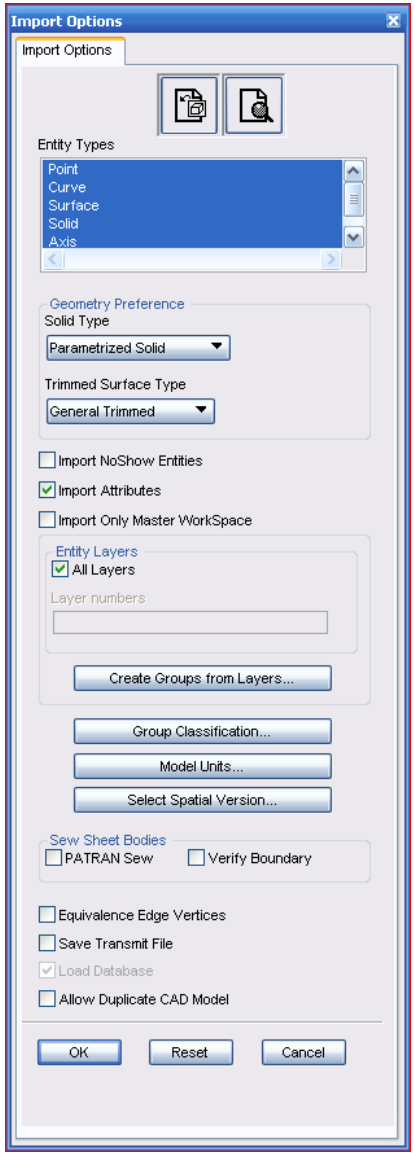
Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

- **Import a CATIA model file using Import to Parasolid.**



- **The CATIA Options are essentially the same as those for Parasolid xmt import.**



Copyright© 2012 MSC.Software Corporation

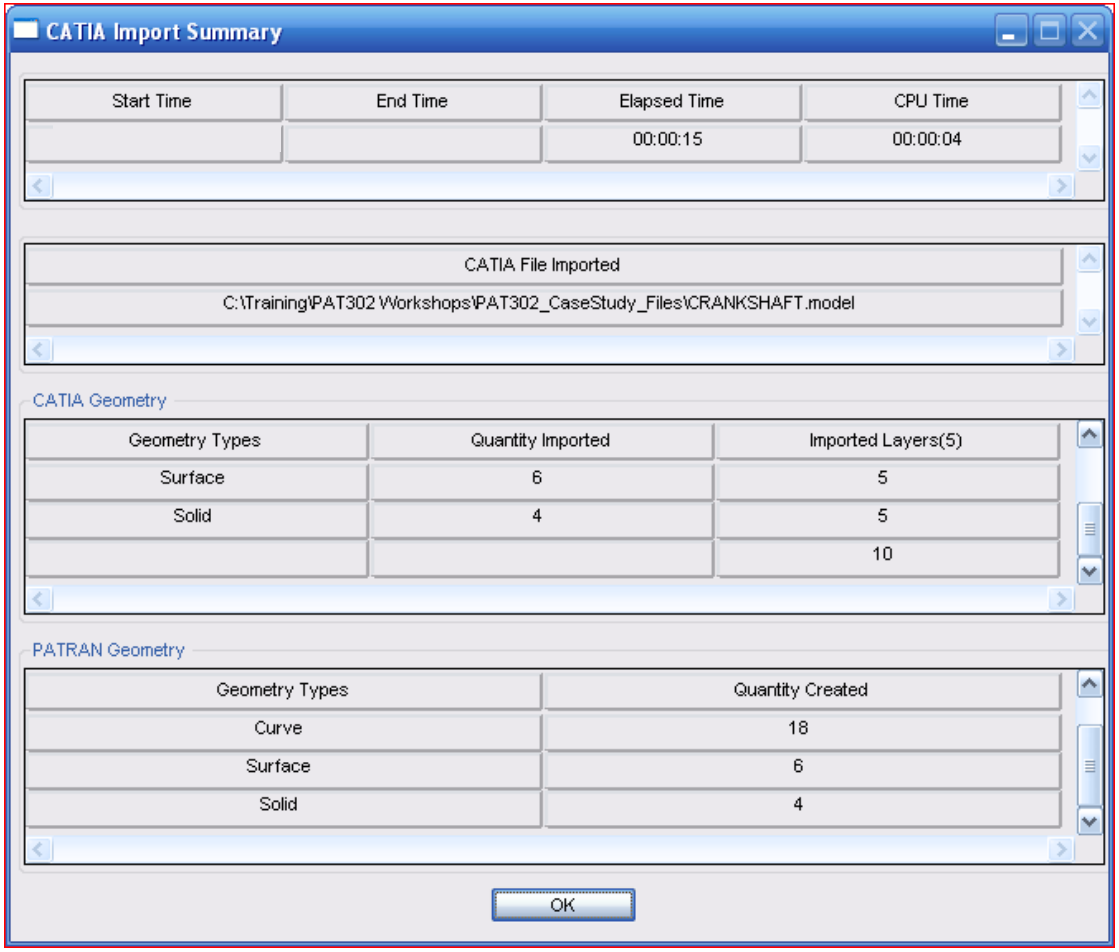
IMPORT CATIA MODEL FILE

- **These are some of the messages written to the session file during import:**

```
p3_express_options_file( "express.opt", FALSE, 3, 1, 52054, TRUE, [39.370079,  @
0., 0., 0., 39.370079, 0., 0., 0., 39.370079, 0., 0., 0.], 0, ["0"], [[-1]], [ @
[-1]], [[-1]], TRUE, [0], TRUE, [0], TRUE, [0], 3, 1, 3 )
utl_process_spawn( "D:\MSC.Software\MD_Patran\2006\bin\xplus_cp.exe ""D:\W" // @
"Ork\In\PAT302\pat302_files\PAT302_CaseStudy_Files_V2005\CRANKSHAFT.model"" // @
"" CRANKSHAFT MD_Patran +sT +aon", TRUE )
INTEGER uil_file_import_cat_catia_count(5)
INTEGER uil_file_import_ca_patran_count(4)
INTEGER uil_file_import_catiap_n_layers
INTEGER uil_file_import_catia_layer_ids(5,256)
REAL uil_file_import_catia_model_tol(2)
p3_ug_xmt_import_v1( "CRANKSHAFT.xmt_txt", "", "express.opt", TRUE, 0,  @
uil_file_import_cat_catia_count, uil_file_import_ca_patran_count,  @
uil_file_import_catiap_n_layers, uil_file_import_catia_layer_ids,  @
uil_file_import_catia_model_tol, TRUE, "", FALSE )
$# Warning reported from application SGM_UG
$#     MD Patran detected a difference between the model unit embedded in the
$# transmit file and the unit as selected in the transmit file import options
$# form.  The model unit of 39.370079, which you supplied, will override the
$# embedded unit of 1000..
$# The Global Model Tolerance has been changed from 0.005 to the newly
$# calculated Global Model Tolerance of 0.0001.
$# Group "CRANKSHAFT_L5" has been created.
$# Adding geometry to group "CRANKSHAFT_L5"...
```


Copyright© 2012 MSC.Software Corporation

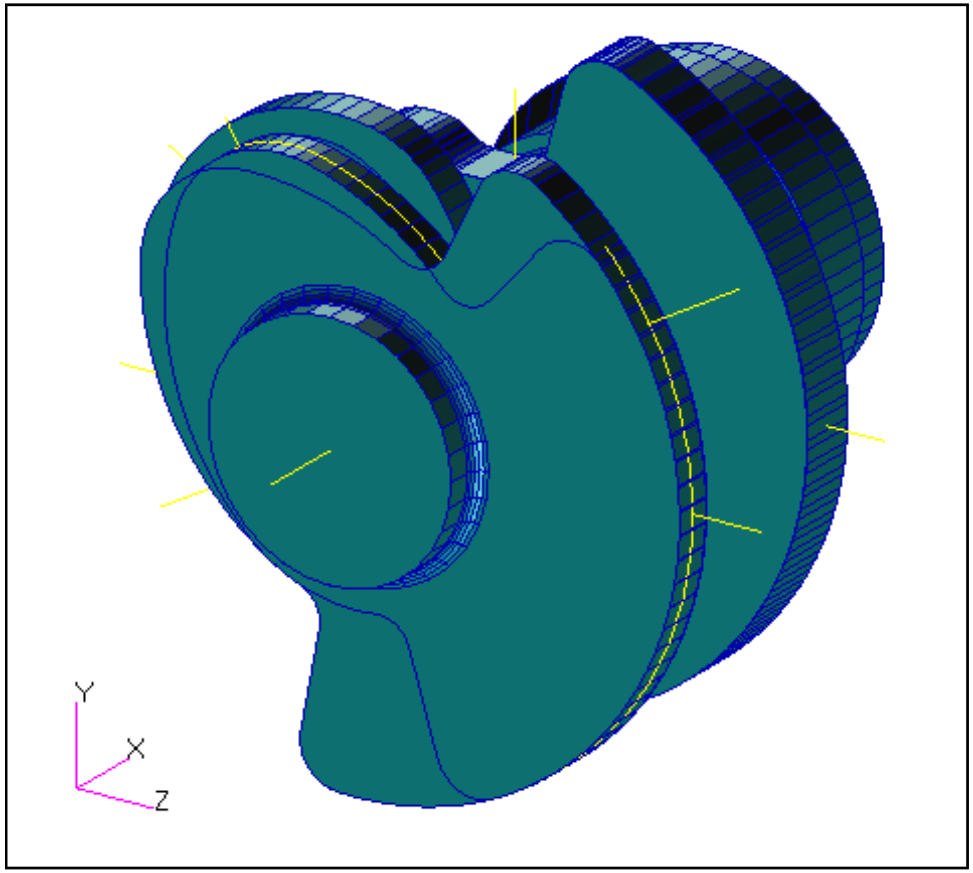
IMPORT CATIA MODEL FILE



Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

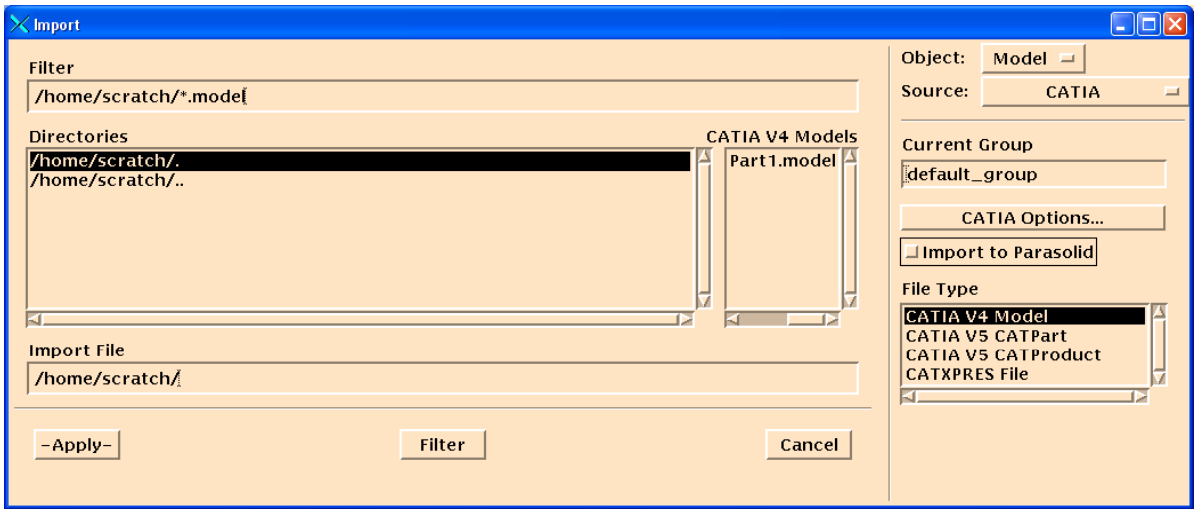
- **Imported curves (Wire Body, yellow) and solids (Solid Body, shaded)**



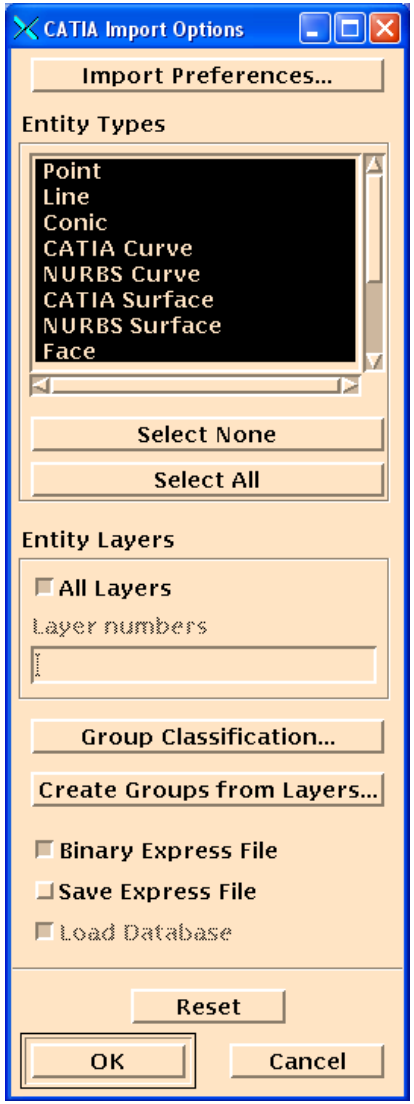
Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

- **Import a CATIA model with flawed features, not using Import to Parasolid (Unix only).**



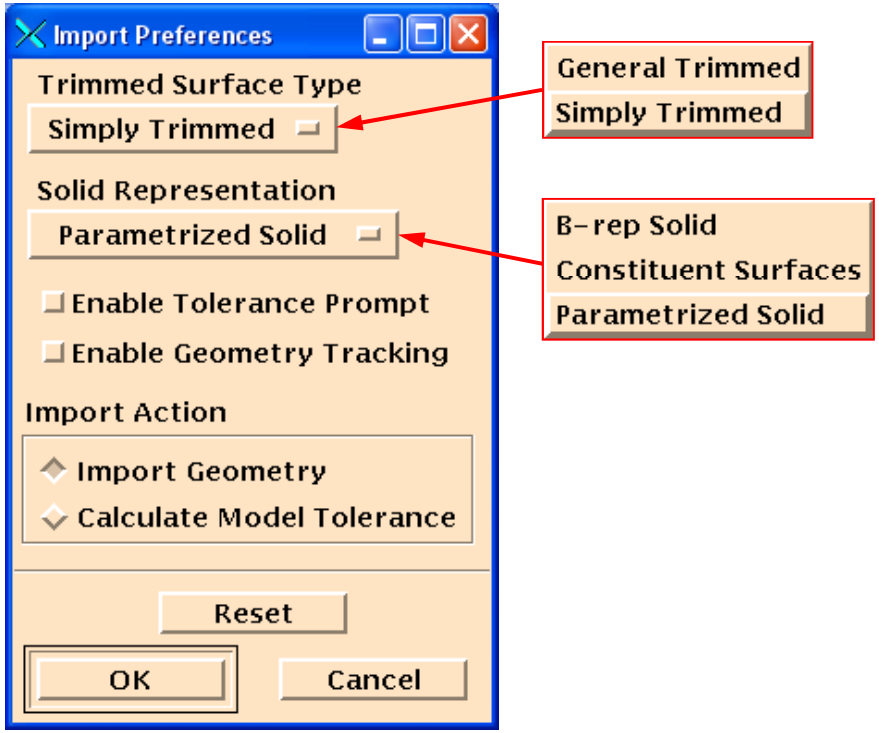
- **The options available under CATIA Options are different.**
  - Some more of these are described on the next slide.



Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

- **Import Preferences**

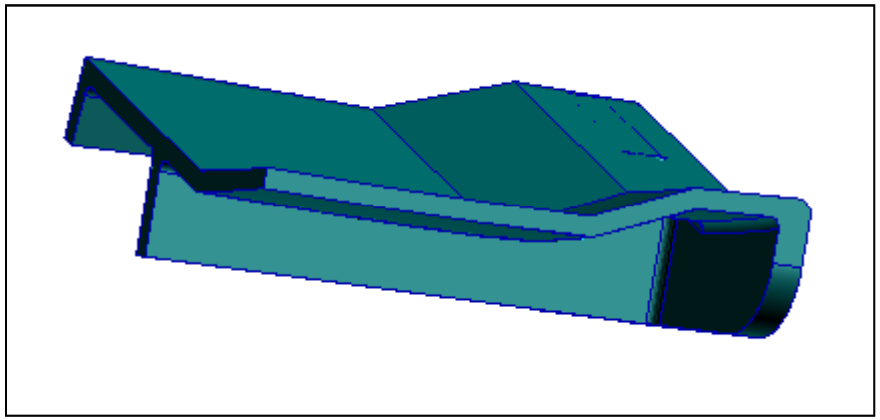


- **Enable Tolerance Prompt**
  - Allows the user to change the import tolerance
- **Enable Geometry Tracking**
  - Causes a file to be created with information about any geometry defects, e.g. problem silver surfaces used to connect regions of the geometry

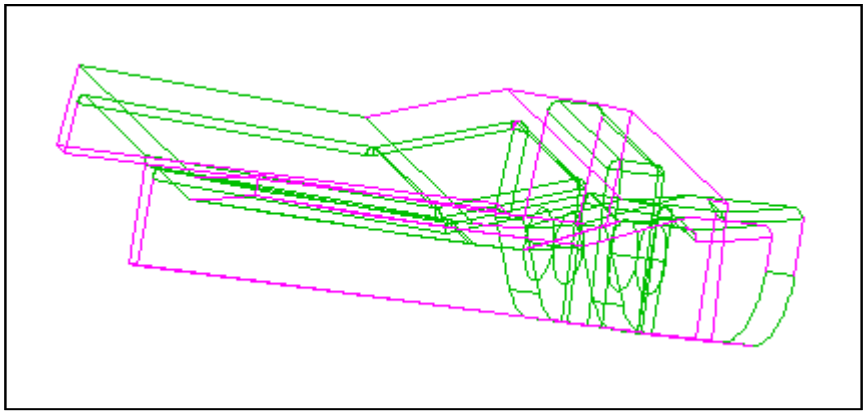
Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

- **Imported CATIA model.**
- **Surfaces obtained instead of solid on import.**



Shaded Surfaces

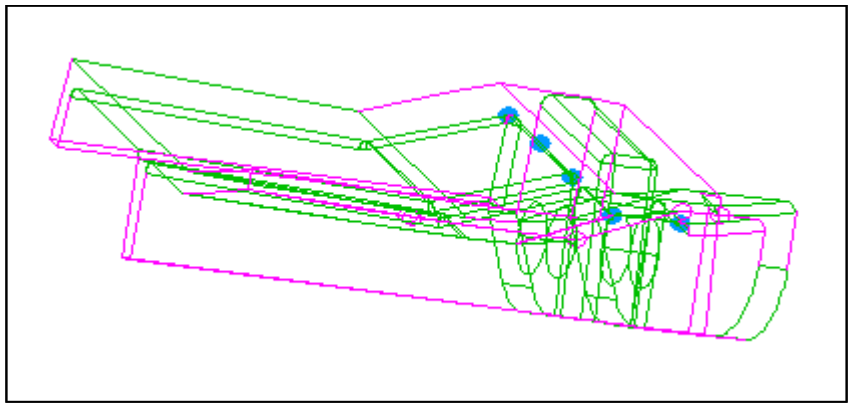


Surfaces

Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

- **Determine why geometry is bad.**
- **Find incongruent and non-manifold surfaces using Geometry: Verify/Surface/Boundary.**
- **Use tracking file to establish specific nature of problems.**



Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

- **Geometry tracking file PATRAN_Geometry.log**

```
                G E O M E T R Y    T R A C K I N G    R E S U L T S

FILE :  C:\Cdev\PAT302\2002\Cstudy\ImportCATIA\BAD_GEOMETRY.model
This file keeps track of some geometry metrics which help the user determine
the validity of the model.  The following metrics are tracked:
1. The length of the shortest curve greater than  1.E-6
   The curve may have a length less than the geometric tolerance.
2. The number of degenerate curves where the length of the curve is less
   than the geometric tolerance (it may be zero length).
3. The number of sliver or gap surfaces.  Gap surfaces are bi-parametric
   surfaces which have two degenerate edges (degenerate curves as defined in
   (2)) opposite of each other.
4. The number of degenerate gap surfaces.  This is a subset of (3) where the
   non-degenerate edges have been equivalenced such that they are represented
   by the same curve.
5. The number of 'topological glue' surfaces.  These surfaces trimmed surfaces
   which have a zero or very small surface area.  They are used by certain
   geometric modellers to create a valid topology model where surfaces come
   together forming 'T' joints
6. The number of B-Rep face surfaces which are automatically edited during
   during import.
7. Tracks errors from external systems.  These are errors which occur during
   operations occuring in systems external to PATRAN, but are passed to
   PATRAN to help explain possible failures during file import.
```

Continued on the next slide

Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

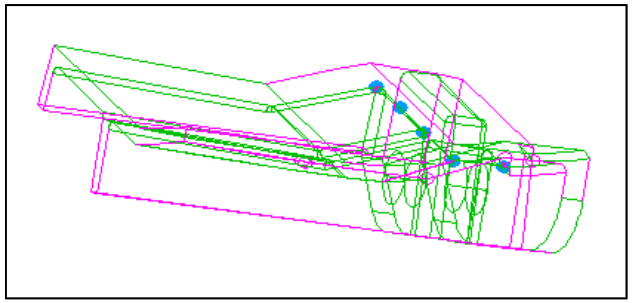
- **Geometry tracking file PATRAN_Geometry.log**

```
ERRORS:

METRICS:
Shortest Curve Length:  1.12055
Number of Degenerate Curves:  13
   Center Coordinates and ID's of the Degenerate Curves:
          [  -628.285 1489.93 -5355.57  ]  :  Surface 6.4
          [  -620.989 1463.23 -5419.72  ]  :  Surface 6.7
          [  -347.175 1505.1 -5420.91  ]  :  Surface 34.1
          [  -519.939 1498.24 -5468.43  ]  :  Surface 36.1
          [  -454.354 1498.24 -5421.  ]  :  Curve 16
          [  -353.526 1498.24 -5421.  ]  :  Surface 37.4
          [  -532.409 1498.07 -5418.94  ]  :  Curve 6
          [  -460.427 1502.73 -5420.92  ]  :  Curve 14
          [  -316.907 1530.78 -4219.94  ]  :  Surface 244.3
          [  -522.026 1497.88 -5468.65  ]  :  Surface 284.5
          [  -620.934 1463. -5420.74  ]  :  Curve 4
          [  -530.498 1498.58 -5418.51  ]  :  Surface 2.3
          [  -628.222 1489.68 -5356.63  ]  :  Curve 13
Number of Gap Surfaces:  2
   Center Coordinates and ID's of the Gap Surfaces:
          [  -531.475 1498.39 -5418.76  ]  :  Surface 2
          [  -458.132 1499.48 -5420.95  ]  :  Surface 4
Number of Degenerate Gap Surfaces:  2
   Center Coordinates and ID's of the Degenerate Gap Surfaces:
          [  -531.475 1498.39 -5418.76  ]  :  Surface 2
          [  -458.132 1499.48 -5420.95  ]  :  Surface 4
Number of 'Topological Glue' Surfaces:  1
   Center Coordinates and ID's of the 'Topological Glue' Surfaces:
          [  -521.019 1498.14 -5468.54  ]  :  Surface 304
Number of B-Rep face surfaces edited during import:  0
External System Errors:  0
```

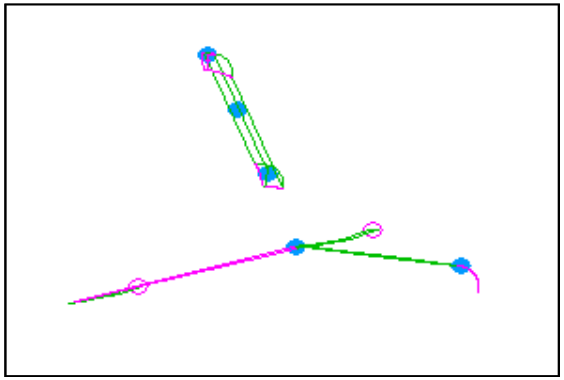

Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

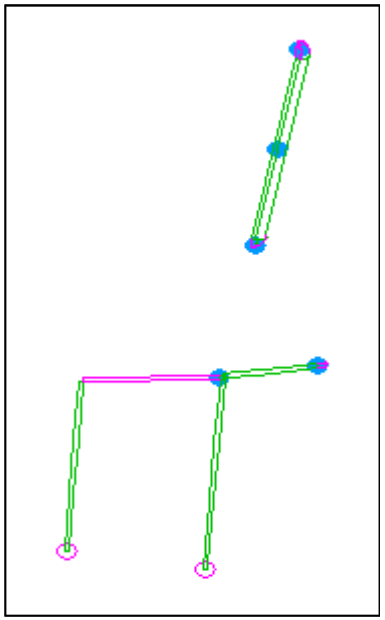
- **Incongruent surfaces**



All Surfaces

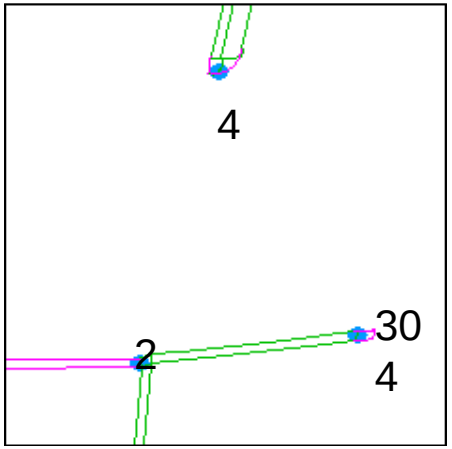


Only Incongruent Surfaces



Top View

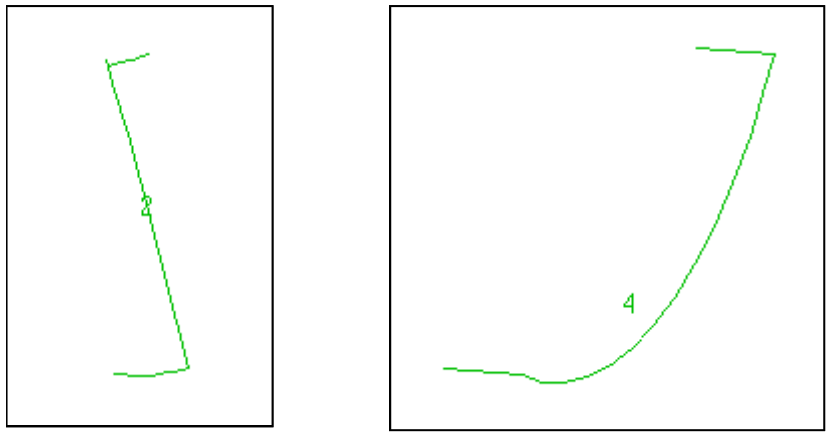
Zoom & Show Surface Labels



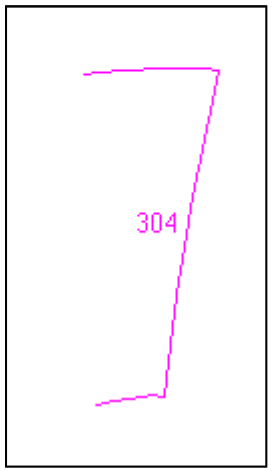
Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

- **Problem surfaces**



Gap and degenerate gap surface Surface 2, 4 has zero area, 4 vertices, and 4 edges each.

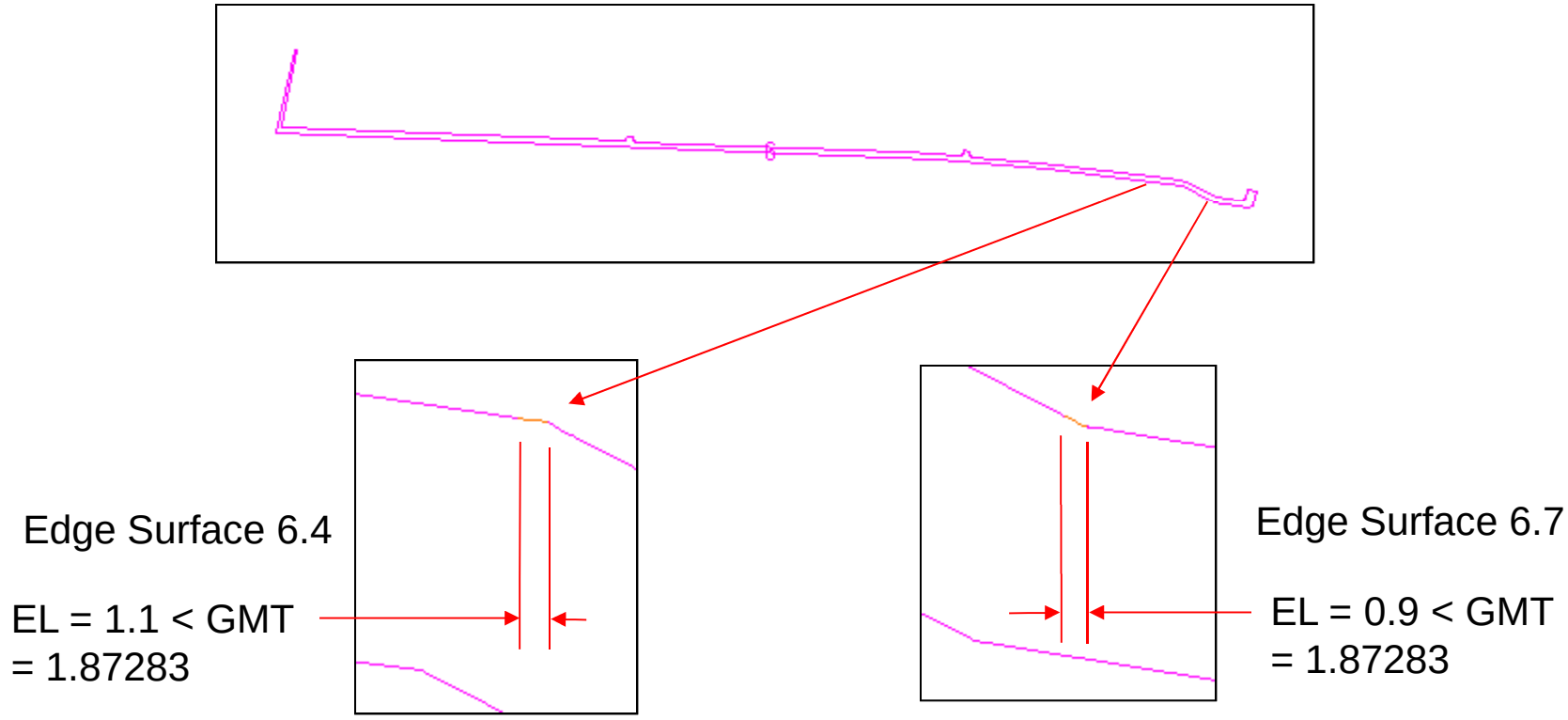


Topological glue surface, Surface 304, has zero area, 2 vertices, and 6 edges.

Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

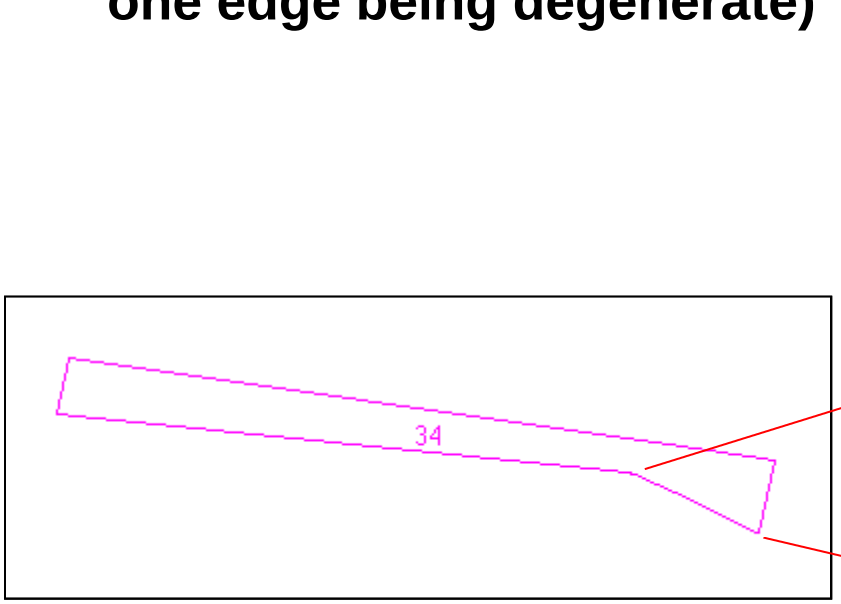
- **Another problem surface, Surface 6 (has 29 vertices and 31 edges, two edges being degenerate)**

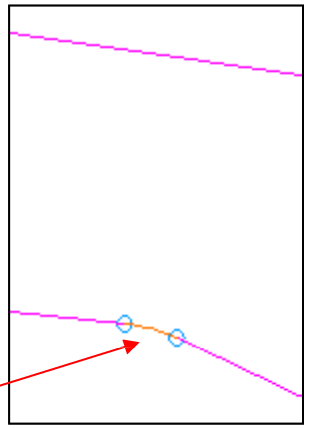


Copyright© 2012 MSC.Software Corporation

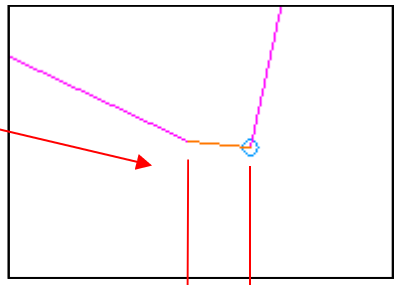
IMPORT CATIA MODEL FILE

- **Another problem surface, Surface 34 (has 6 vertices and 7 edges, one edge being degenerate)**





This edge is not degenerate, but it is short (4.41 mm long) and may negatively influence the meshing.



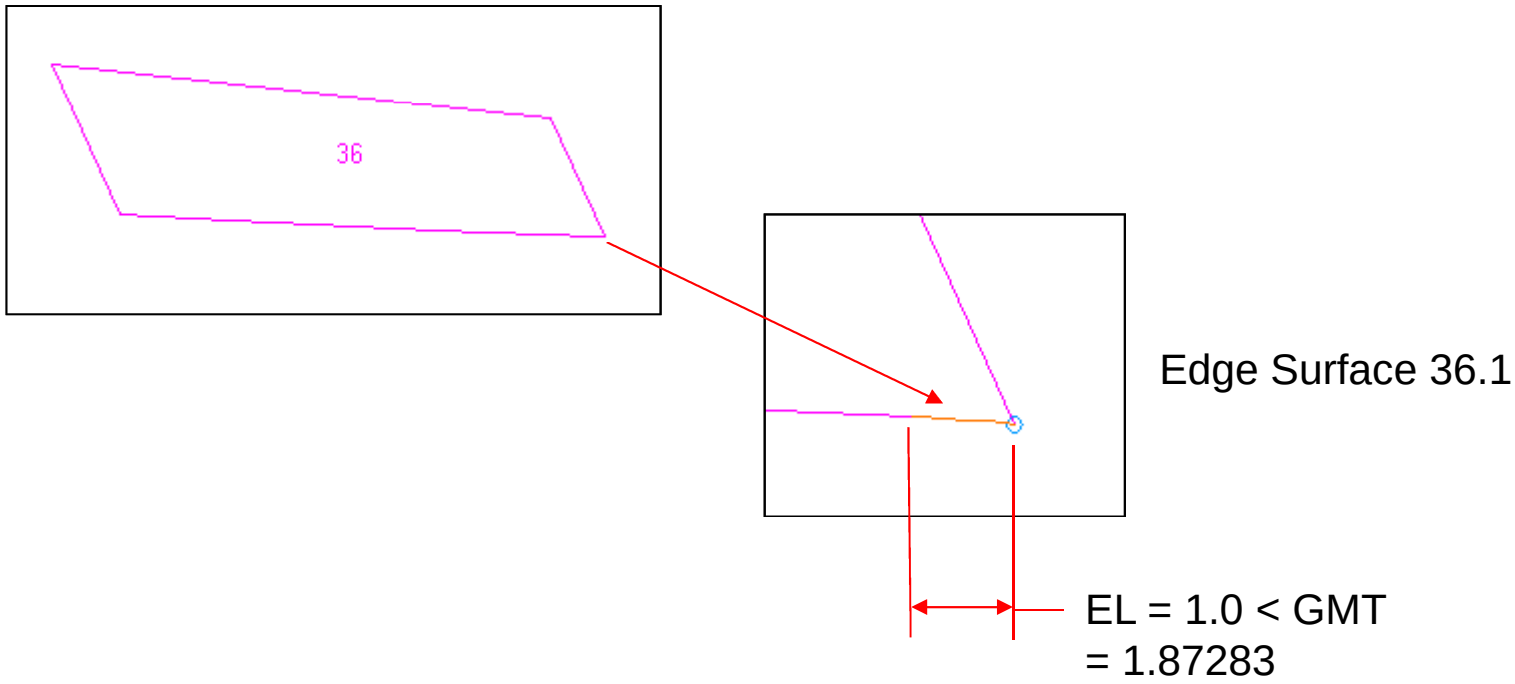
Edge Surface 34.1

EL = 0.9 < GMT = 1.87283

Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

- **Another problem surface, Surface 36 (has 4 vertices and 5 edges, one edge being degenerate)**



Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

- **Repair the geometry (surfaces) by**
  - Reconstructing the surfaces in CATIA, so the degenerate edges are not needed/used, e.g. Surface 6.4.
  - Reconstructing the surfaces in CATIA, so a topological glue surface is not needed to “connect” the surfaces to be able to create a solid in CATIA.
  - Changing the tolerance in Patran; see next slide.

Copyright© 2012 MSC.Software Corporation

IMPORT CATIA MODEL FILE

- **Comments on selecting a GMT for import of a CATIA file**
  - It is important that the GMT not be too large so that surface edges become degenerate upon import (edge vertices equivalenced).
  - On surface meshing, it is important that the GMT not be too small so that close (adjacent) vertices are not equivalenced upon import.
  - Largest Distance Between Adjacent Vertices < GMT < Shortest Edge Length
    - Shortest edge can be found using Geometry: Show/Curve/Length Range
    - Do not have a way to determine the largest distance between all the adjacent vertices.
  - Its best to select the value for the GMT, so use of Geometry: Edit/Surface/Sew can be avoided.
  - Once in a while, it is possible to use the following “trick”
    - CAD (e.g. CATIA) -> Patran fails, but obtain Patran’s estimate of the GMT for the import
    - CAD (e.g. CATIA) -> IGES -> Patran w/ GMT from failed import

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

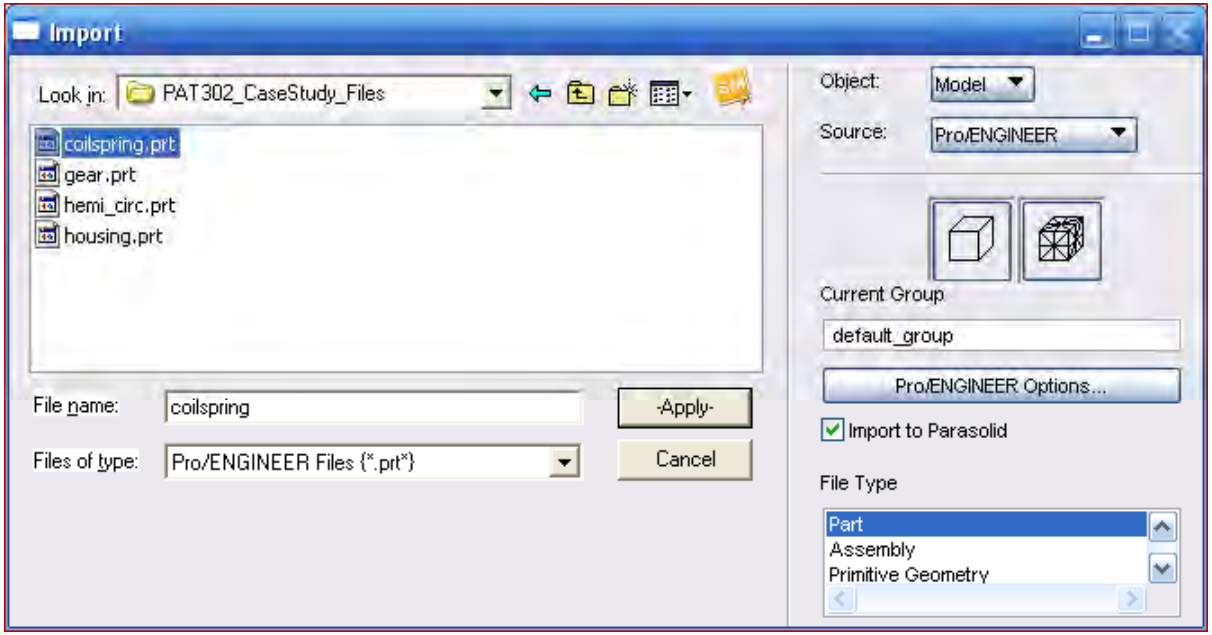
IMPORT PRO/ENGINEER MODEL FILE

Copyright© 2012 MSC.Software Corporation

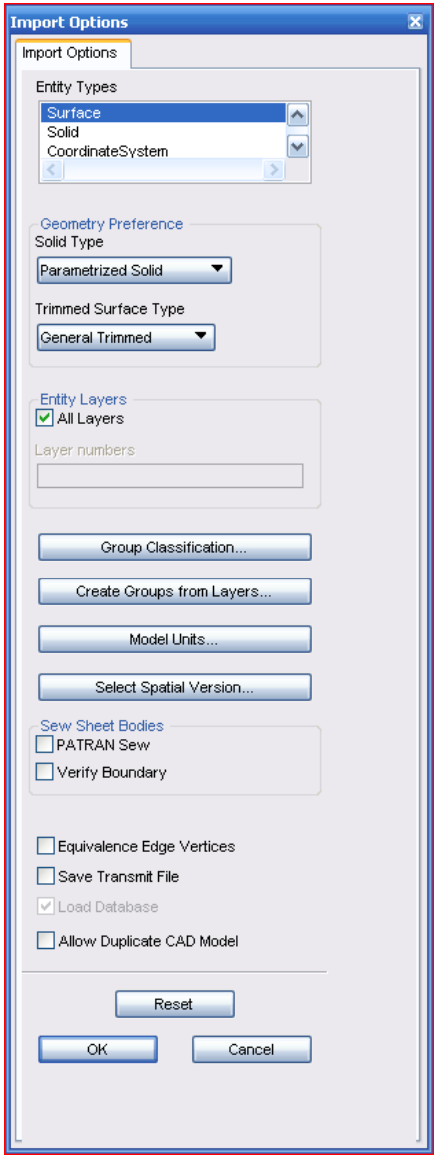
Copyright© 2012 MSC.Software Corporation

IMPORT PRO/ENGINEER MODEL FILE

- Import a Pro/E model file, using Import to Parasolid.



- The Pro/E Options are essentially the same as those for Parasolid xmt import.

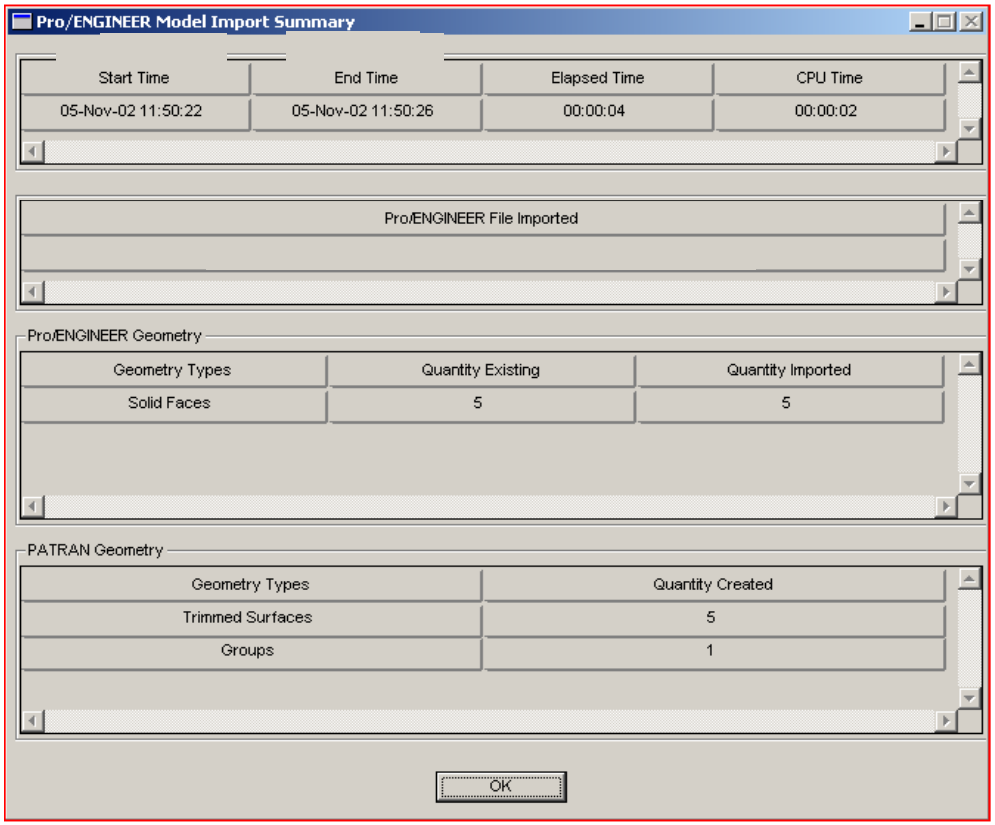


Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

- **Solid Faces (5) were imported.**

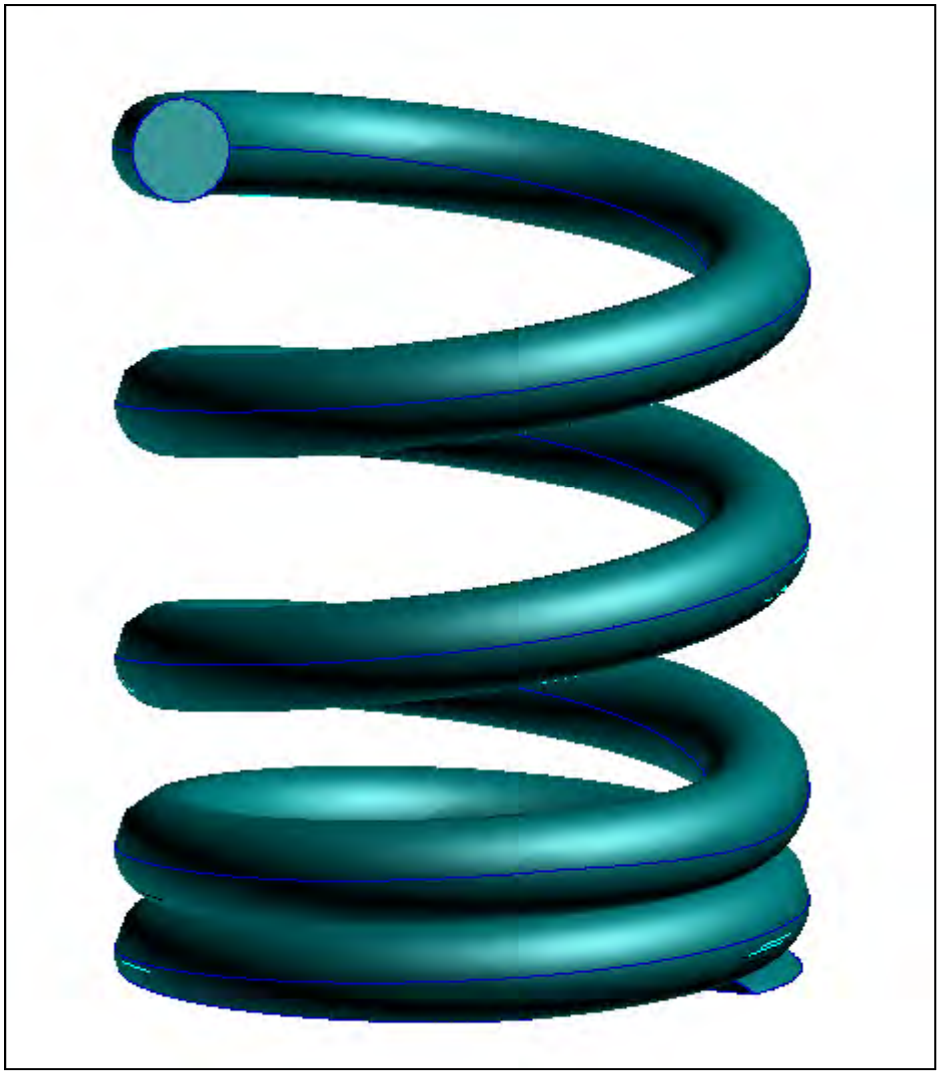
- **Trimmed surfaces (5) were created in Patran.**



Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

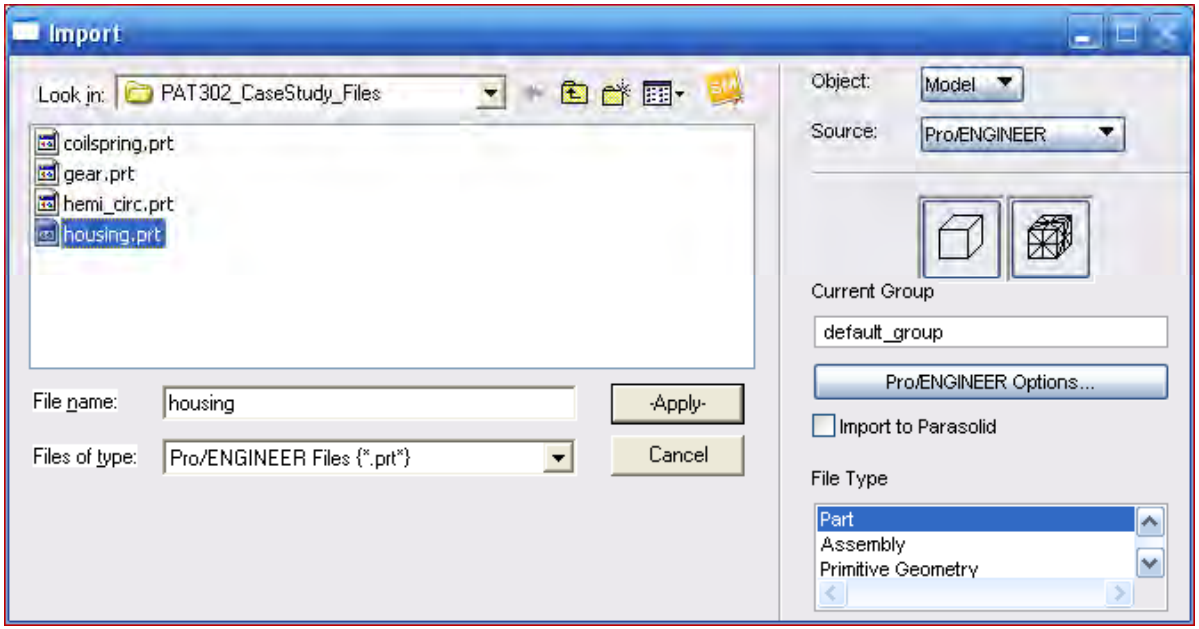
- **Solid created from imported surfaces**

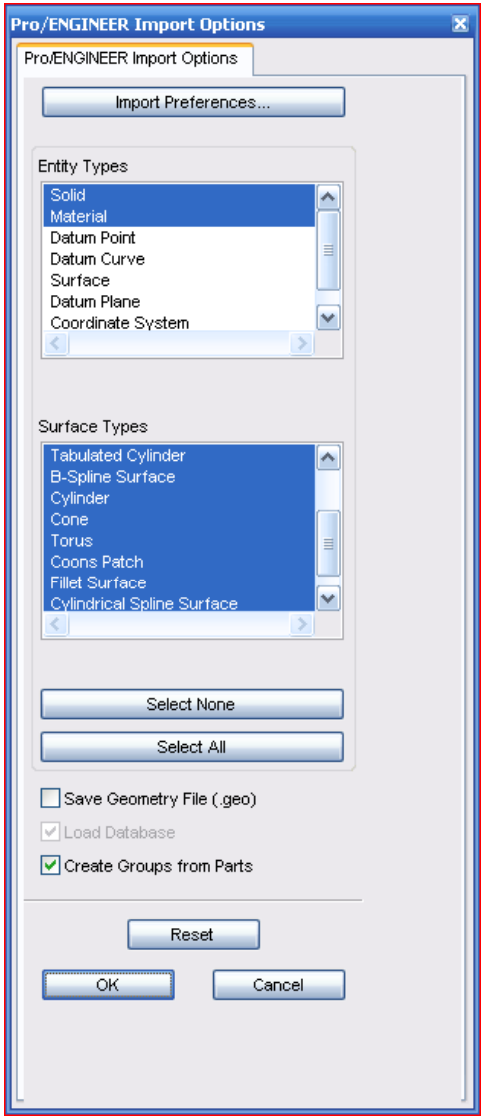


Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

- **Import a Pro/E model file, not using Import to Parasolid.**



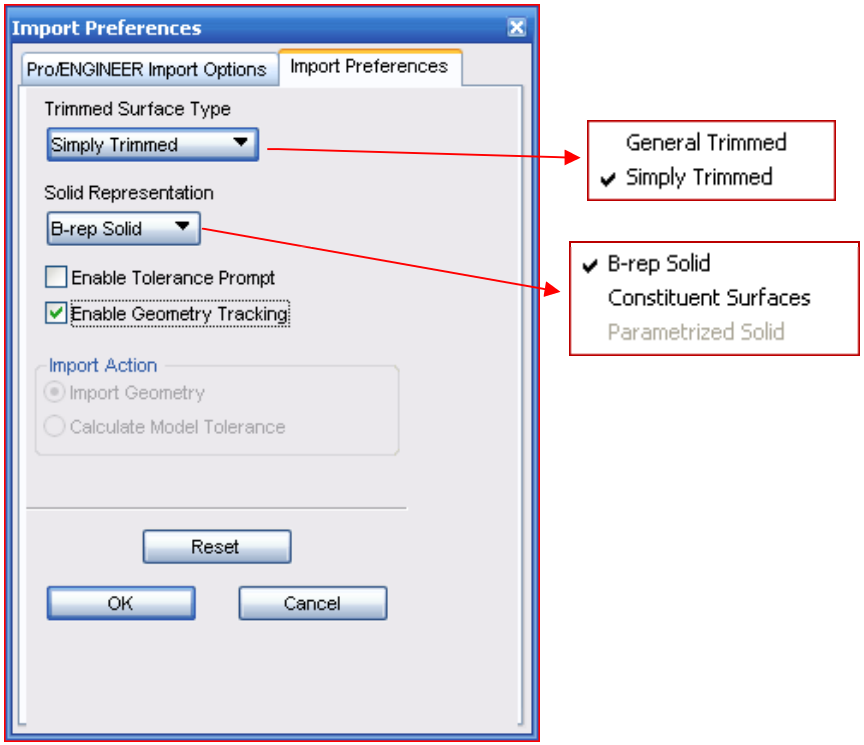


- **The Pro/E Options are different.**

Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

- **Import Preferences**

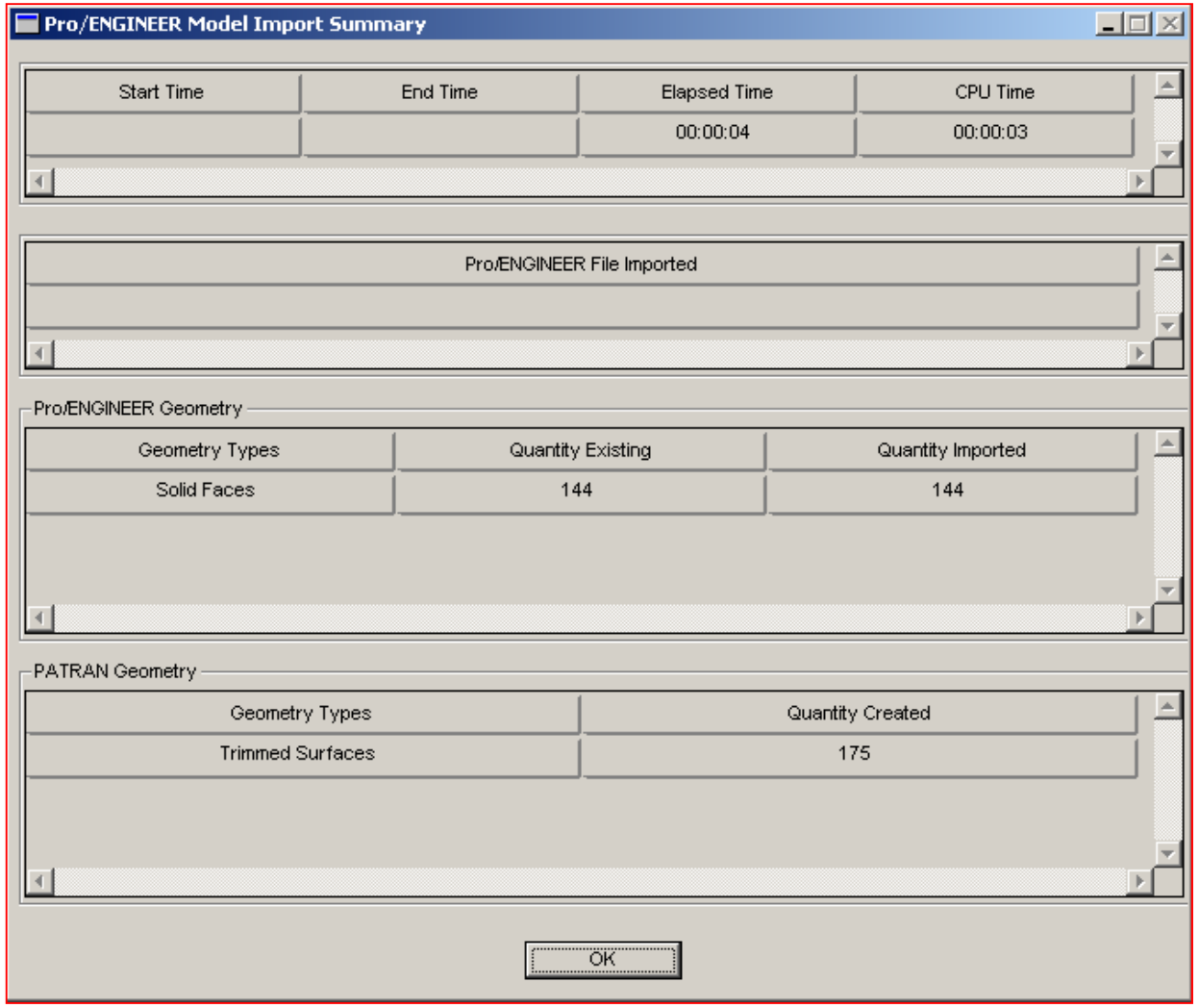


- **Enable Tolerance Prompt**
  - Causes form to appear during import, that allows the user to change the import tolerance
- **Enable Geometry Tracking**
  - Causes a file to be created with information about geometry defects, e.g. parametric surface with two opposing edges that are degenerate, and the other edges are not degenerate
  - This file has the same information as the one for CATIA import

Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

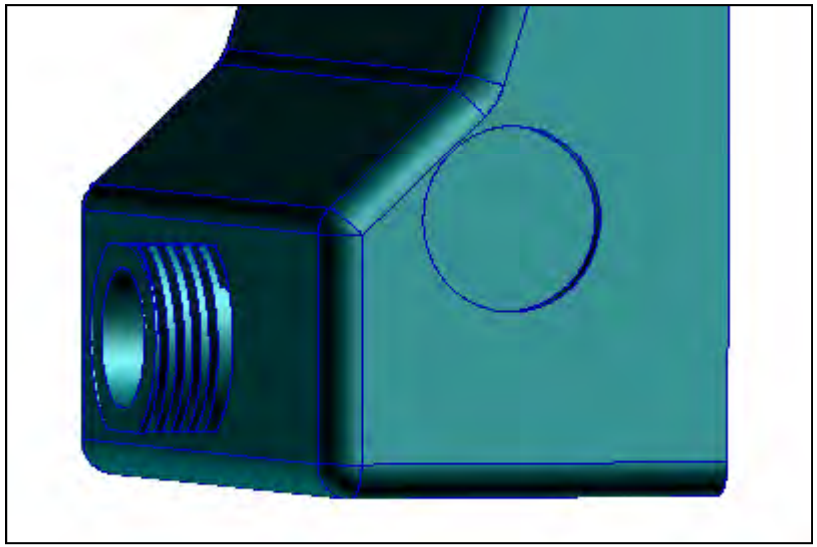
- **Solid Faces (144) were imported.**
- **Trimmed surfaces (175) were created in Patran.**



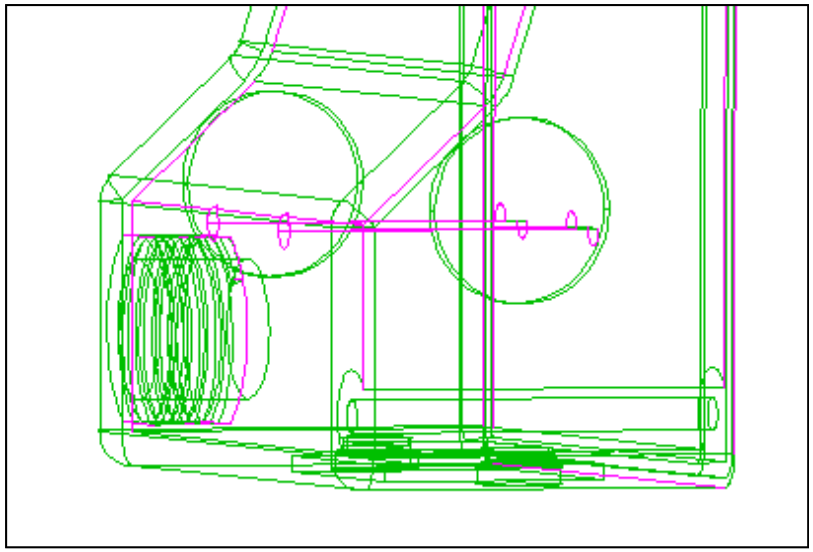
Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

- **Imported Pro/E model**
- **Surfaces obtained instead of solid on import.**



Shaded Surfaces



Surfaces

Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

- **Check to see if it is possible to create a solid from the surfaces:**
  - Geometry: Verify/Surface/Boundary

    sgm_verify_surface_boundary("Surface 1:175", 0.035500001, 1)

    $# There are no free surface edges.

  - Geometry: Verify/Solid/B-rep

    sgm_verify_solid_brep("Surface 1:175", 0.035500001)

    $# Surface 1:175 form a valid B-rep Solid.

  - Geometry: Create/Solid/B-rep

    ge_const_solid_brep("1", "Surface 1:175", TRUE, gm_create_solid_br_created_ids)
    - There is a problem; see the subsequent messages in the next slides.

Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

Geometry: Create/Solid/B-rep (Cont.)

$# Warning reported from application Geometry

$# A solid could not be created in Parasolid. Will attempt to create a non-Parasolid solid.

$# Warning reported from application SGM

$# The body is non-manifold. It contains a face or edge Surface 34 that does not have a manifold connection to the body …

$# Warning reported from application SGM

$# The body is non-manifold. It contains a face or edge Surface 149 that does not have a manifold connection to the body …

$# Error reported from application SGM

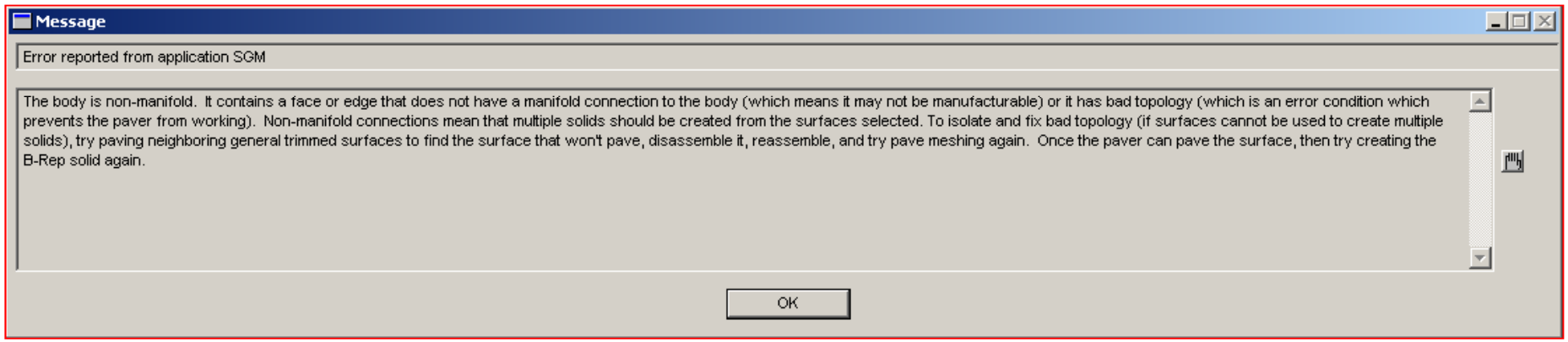
$# The body is non-manifold. It contains a face or edge that does not have a manifold connection to the body (which means it may not be manufacturable) …

$# UNDO: Create Solid B-rep for Surface 1:175

Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

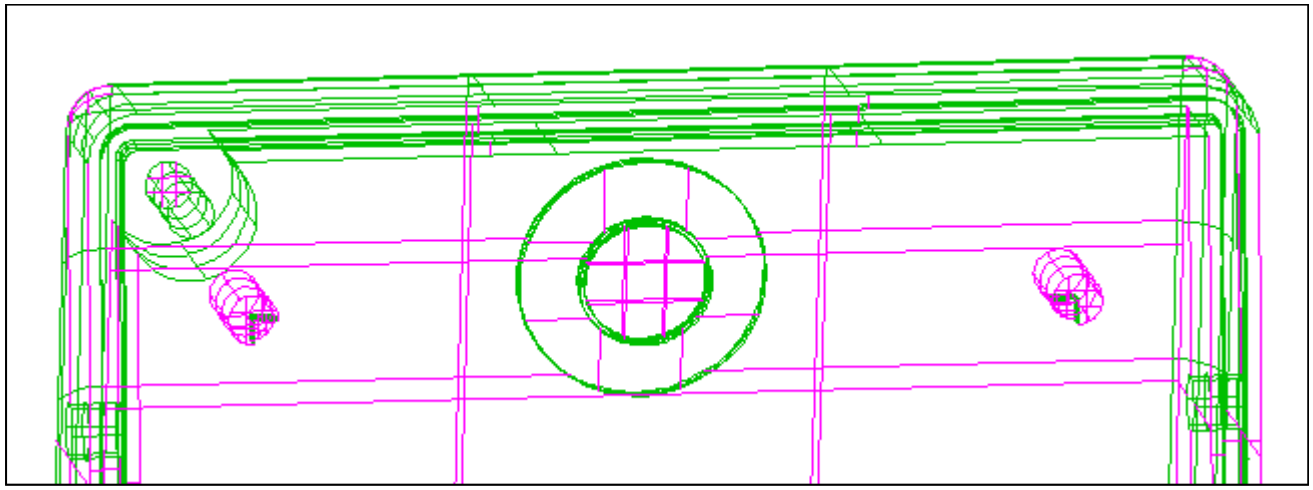
- **Patran displays the following window**



Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

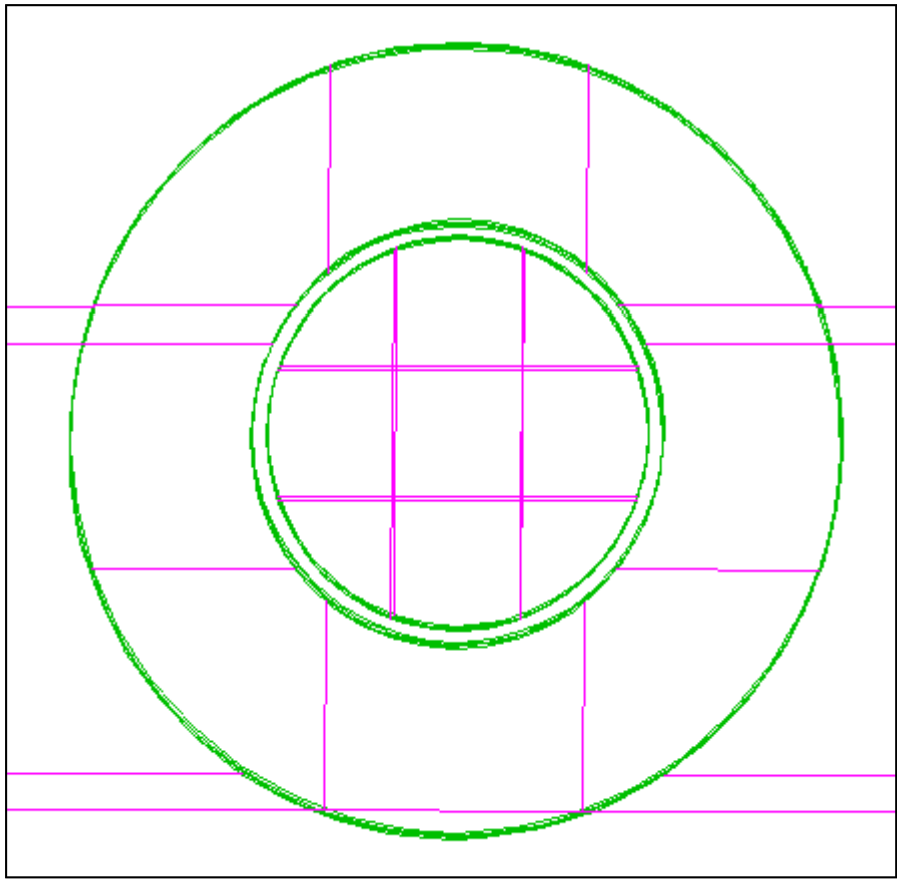
- **In the session file (or history window), surface 34, 149, 174, and 175 are indicated as being not manifold to the body -- “does not have a manifold connection to the body”**
- **Look at these surfaces:**

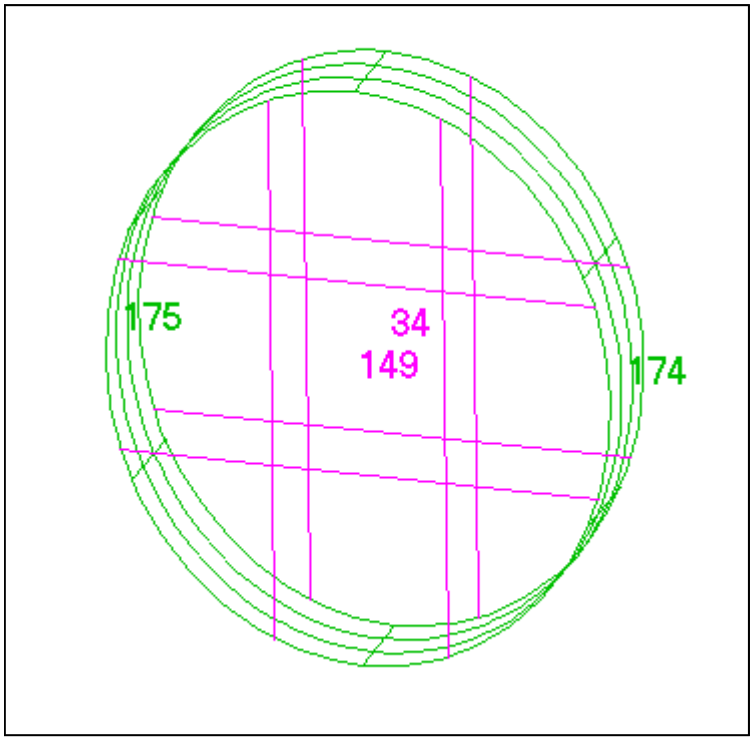


Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

- **Surfaces Surface 34, 149, 174, 175**

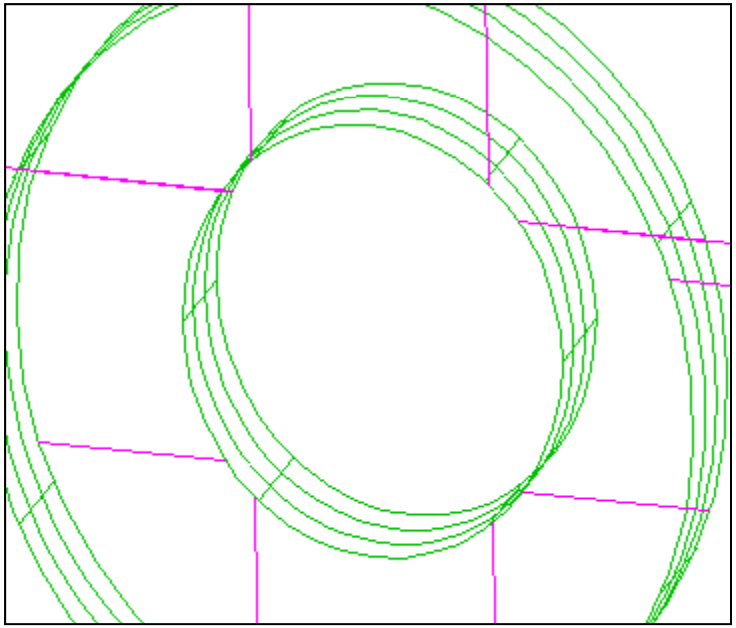


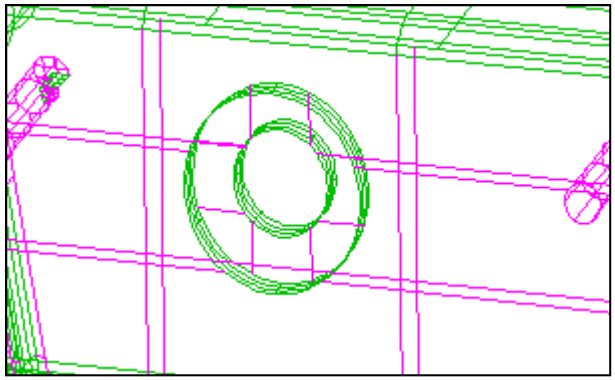


Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

- **Surface 34, 149, 174, 175 are not needed at the hole in the body.**
  - Delete them.

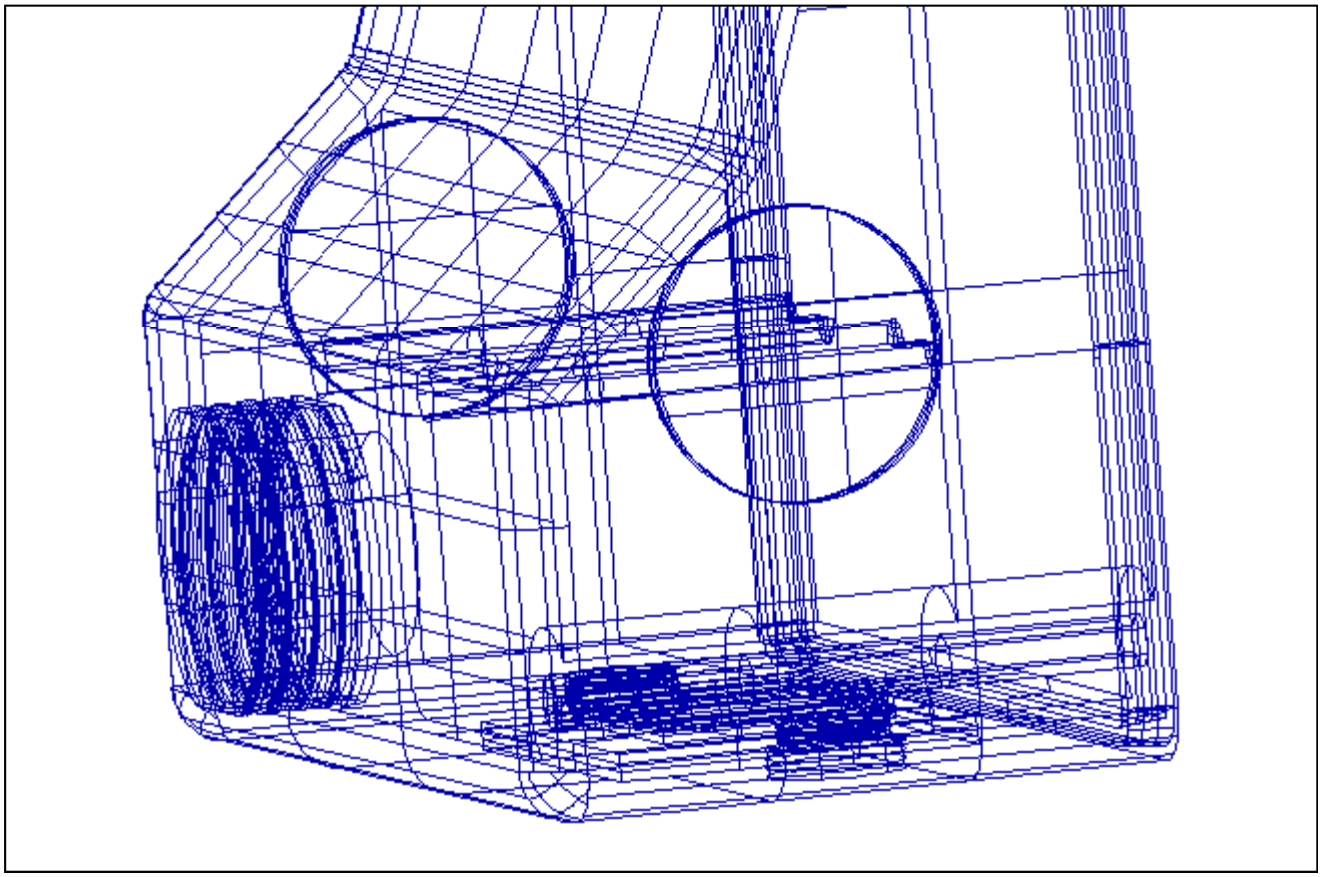




Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

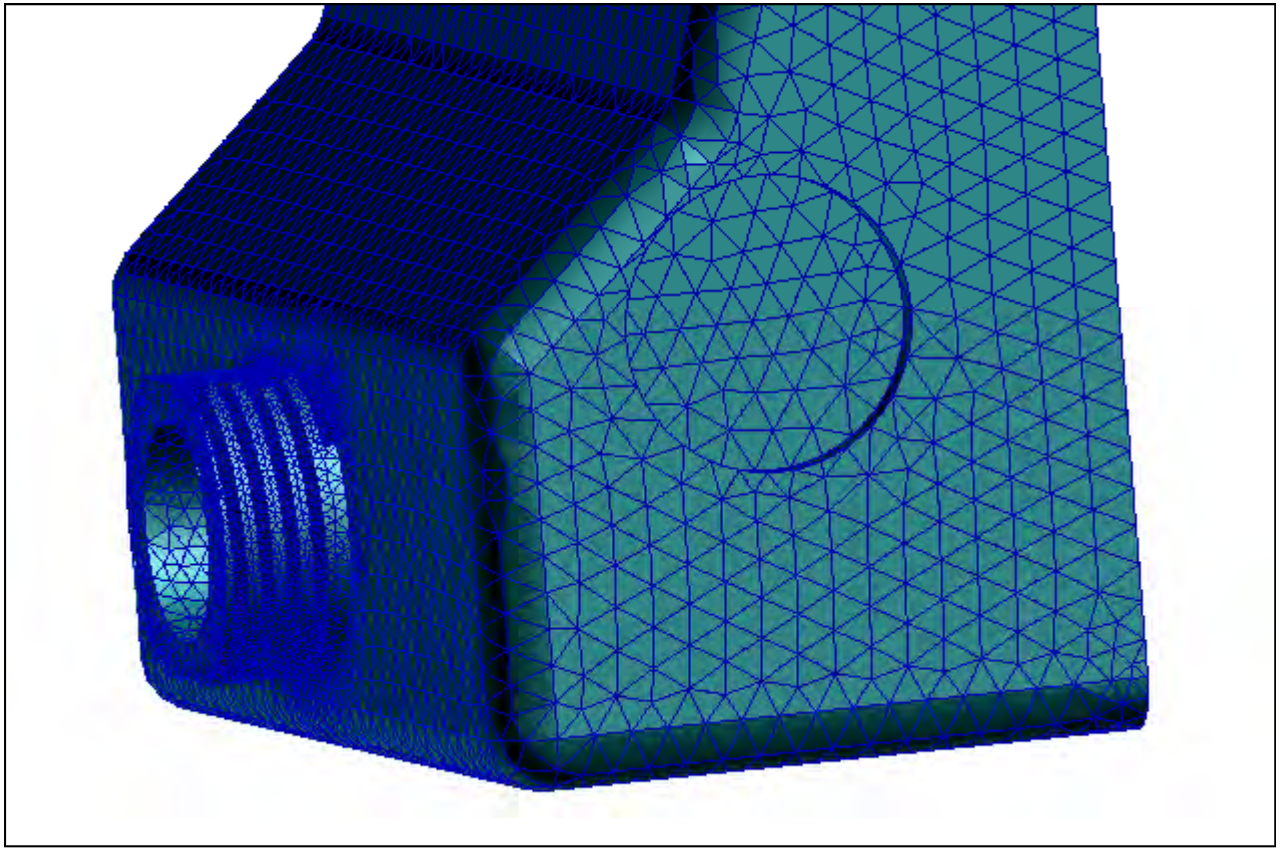
- **Create the B-rep solid using the remaining 171 surfaces**



Copyright© 2012 MSC.Software Corporation

IMPORT PROENGINEER MODEL FILE

- **TetMesh the solid**



Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

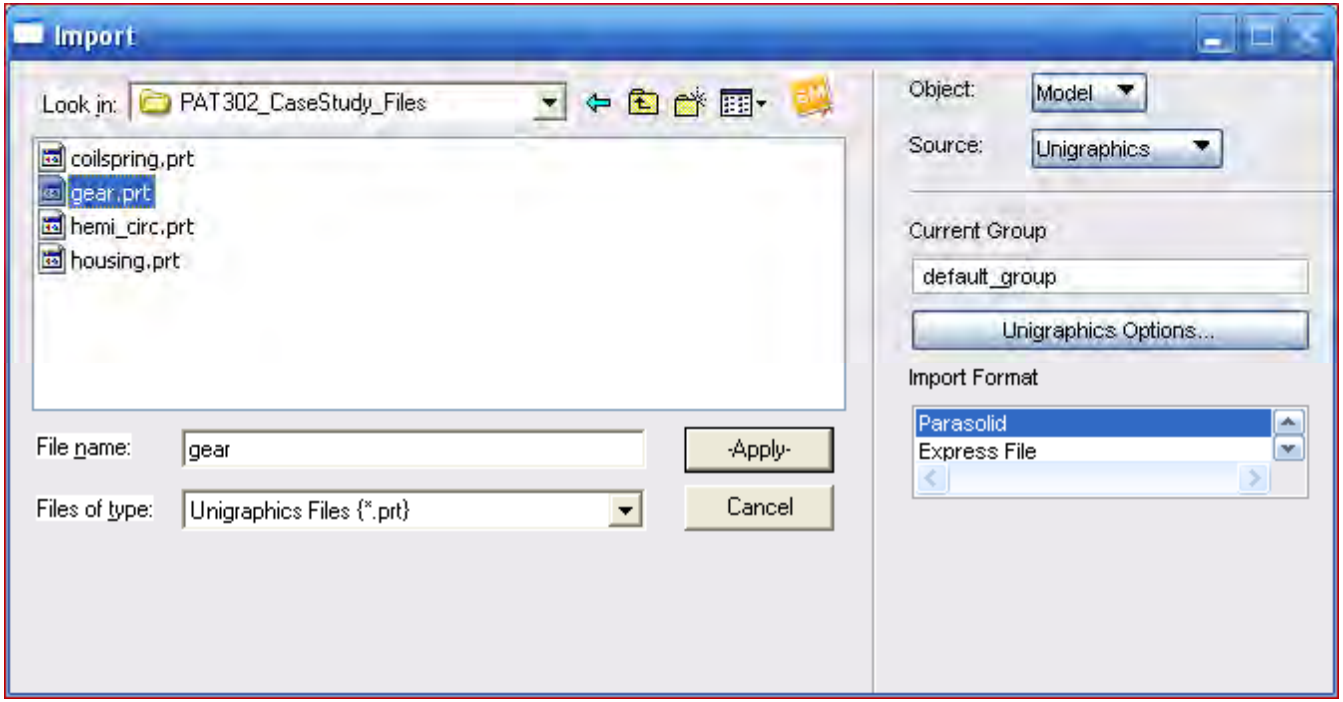
IMPORT UNIGRAPHICS MODEL FILE

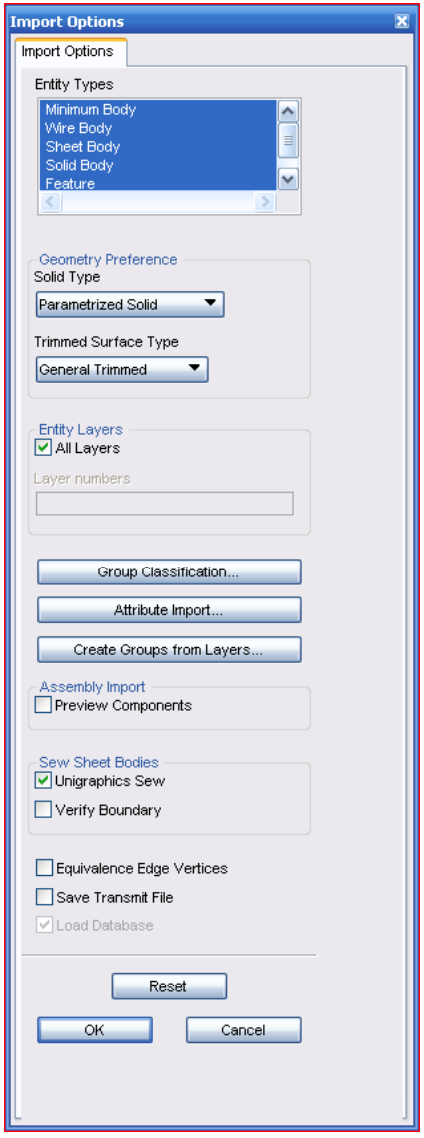
Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **Import a Unigraphics model file.**





- **The Unigraphics Options are similar to those for Parasolid xmt import.**

Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **These are the messages issued to the output window during import:**

```
********************
* P3/UNIGRAPHICS
* 14.0.016
********************

P3-UG server connected ...

   (UGS NX Version: NX V3.0)

 Preloading Unigraphics part: D:\WOrk\In\PAT302\pat302_files\PAT302_CaseStudy_Fi
les_V2005\gear.prt ...
 Unigraphics Database D:\WOrk\In\PAT302\pat302_files\PAT302_CaseStudy_Files_V200
5\gear.prt Preloaded Successfully

 Opening Unigraphics part: E:\TEMP\PAT302\07\07_3_ug_copy\gear.prt_msc_07_3 ...
 Unigraphics Database E:\TEMP\PAT302\07\07_3_ug_copy\gear.prt_msc_07_3 Opened su
ccessfully

 Preprocessing the sheets and curves for integrity and validity...
 Finished preprocessing the sheets and curves...
```


Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **More output information (in session file)**

```
p3_ps_open_ug_v2( "D:\Work\In\PAT302\pat302_files\PAT302_CaseStudy_Files_v" // @
"2005\gear.prt", [1, 1], [0], ["Attributes", "2", "*", "*", "I", "0",  @
"Components", "0"], uil_file_import_paras_num_parts )
$# Warning reported from application SGM_UG
$#     Importing Unigraphics Features will require that the Unigraphics part
$# file be modified to reflect the changes made in MD Patran. Therefore a
$# copy of the part is made to
$# E:\TEMP\PAT302\07\07_3_ug_copy\gear.prt_msc_07_3 prior to import. Please
$# do not remove or modify this copy in any way, as subsequent model updates
$# are dependent on the undisturbed copy. This copy will not work with any
$# other MD Patran databases.
$# Traversing Unigraphics Part File gear.prt_msc_07_3 ...
p3_express_options_file( "express.opt", FALSE, 3, 1, 52054, FALSE, [0., 0.,  @
0., 0., 0., 0., 0., 0., 0., 0., 0., 0.], 0, ["0"], [[-1]], [[-1]], [[-1]],  @
TRUE, [0], TRUE, [0], TRUE, [0], 3, 1, 3 )
INTEGER uil_file_import__total_ug_count(5)
INTEGER uil_file_import__total_p3_count(4)
INTEGER uil_file_import_paraso_n_layers
INTEGER uil_file_import_paras_layer_ids(5,256)
REAL uil_file_import_total_model_tol(2)
p3_ug_xmt_import_v1( ".\gear-ps0001.xmt_txt",  @
"E:\TEMP\PAT302\07\07_3_ug_copy\gear.prt_msc_07_3", "express.opt", TRUE, 1,  @
uil_file_import__total_ug_count, uil_file_import__total_p3_count,  @
uil_file_import_paraso_n_layers, uil_file_import_paras_layer_ids,  @
uil_file_import_total_model_tol, TRUE, "", FALSE )
$# The Global Model Tolerance has been changed from 0.005 to the newly
$# calculated Global Model Tolerance of 0.0001.
$# Group "gear_L1" has been created.
$# Adding geometry to group "gear_L1"...
$# The Global Model Tolerance has been changed from 0.0001 to the newly
$# calculated Global Model Tolerance of 0.314159.
db_put_origin_model_name( "gear E:\TEMP\PAT302\07\07_3_ug_copy\gear.prt_ms" // @
"c_07_3", "D:\Work\In\PAT302\pat302_files\PAT302_CaseStudy_Files_v2005\gea" // @
"r.prt" )
$# Unigraphics Part File Traversal Completed.
pfk_create_ug_import( "E:\TEMP\PAT302\07\07_3_ug_copy\gear.prt_msc_07_3", [1,  @
1], [0], ["Attributes", "2", "*", "*", "I", "0", "Components", "0"], 1,  @
"express.opt", FALSE, 3, 1, 52054, FALSE, [0., 0., 0., 0., 0., 0., 0., 0., 0., @
 0., 0., 0.], 0, ["0"], [[-1]], [[-1]], [[-1]], TRUE, [0], TRUE, [0], TRUE, [ @
0], 3, 1, 3 )
```

(1)

(2)

(1) Create Parasolid transmit file from Parasolid file, gear.prt.

(2) Import Parasolid transmit file, gear-ps0001.xmt_txt.

Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

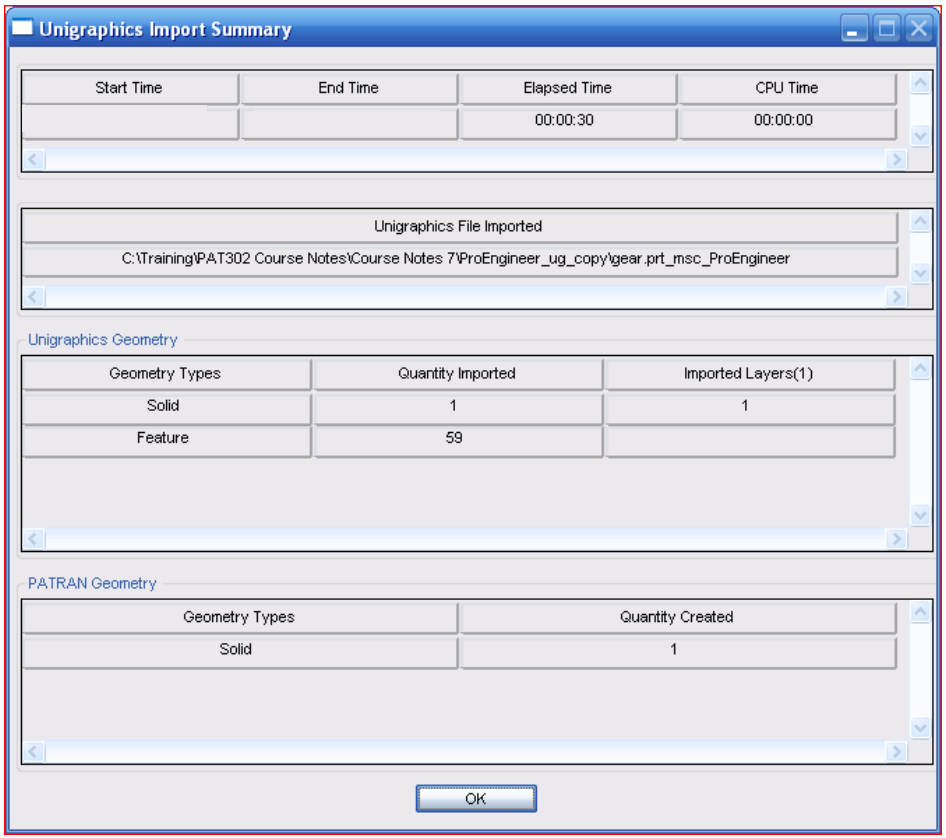
- **Importing a Unigraphics part file is similar to importing a Parasolid xmt (transmit) file.**
  - Part of importing a UG part file involves creating an xmt file, then importing the xmt file; so, many of the comments for the Parasolid xmt file import apply here.
  - In addition to importing the xmt file, UG feature expressions (i.e. fillet radius, edge length) can be imported, and subsequently edited in Patran.

Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **Both Solid (1) and Feature (59) types were imported.**

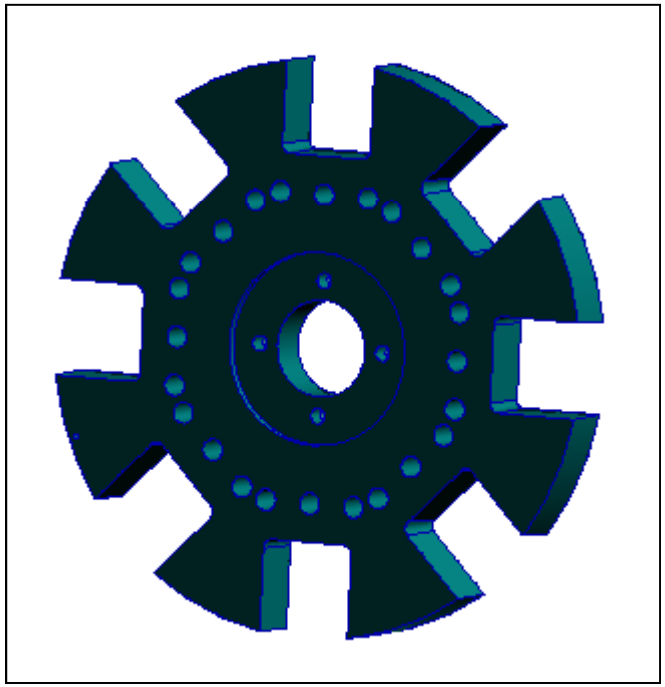
- **Solid (1) was created in Patran**

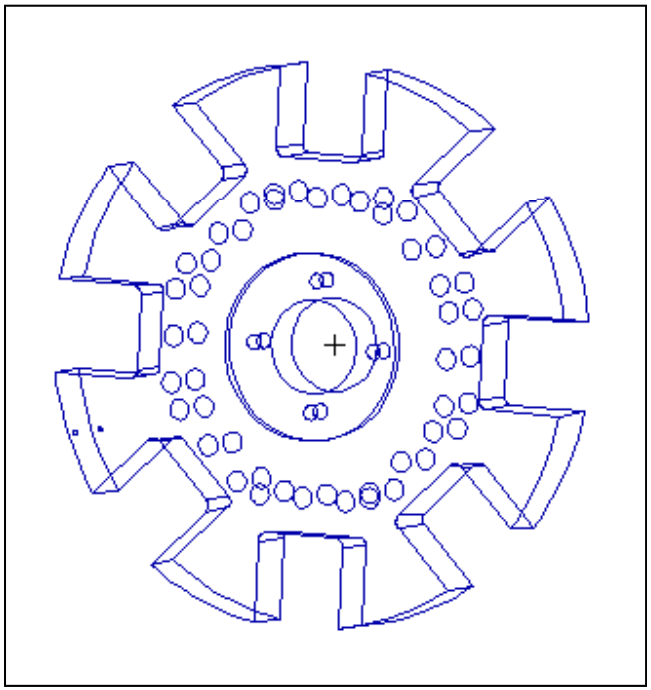


Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **Imported Parasolid solid:**

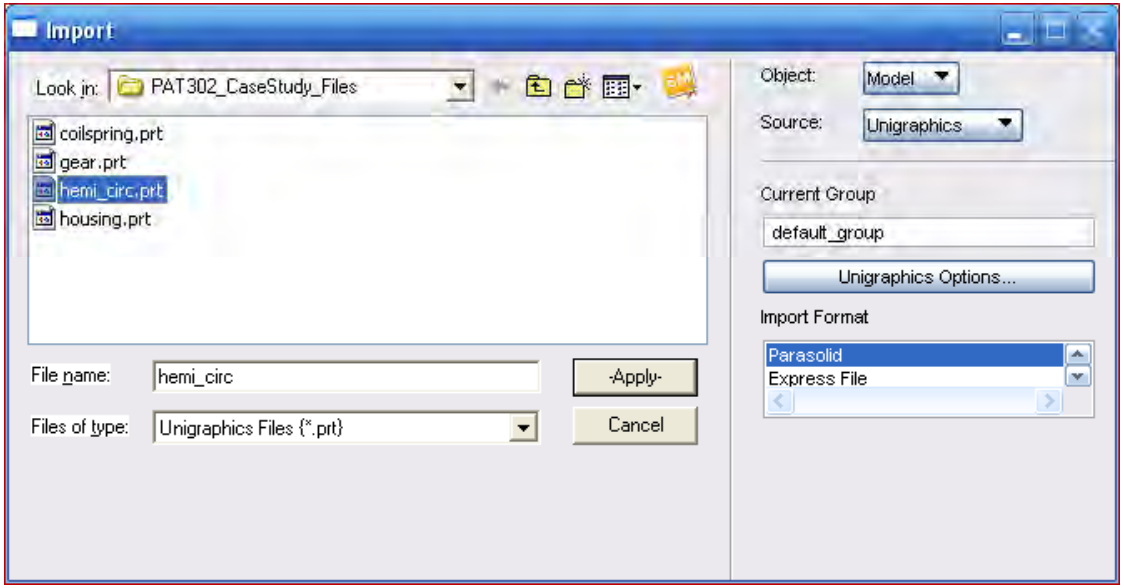


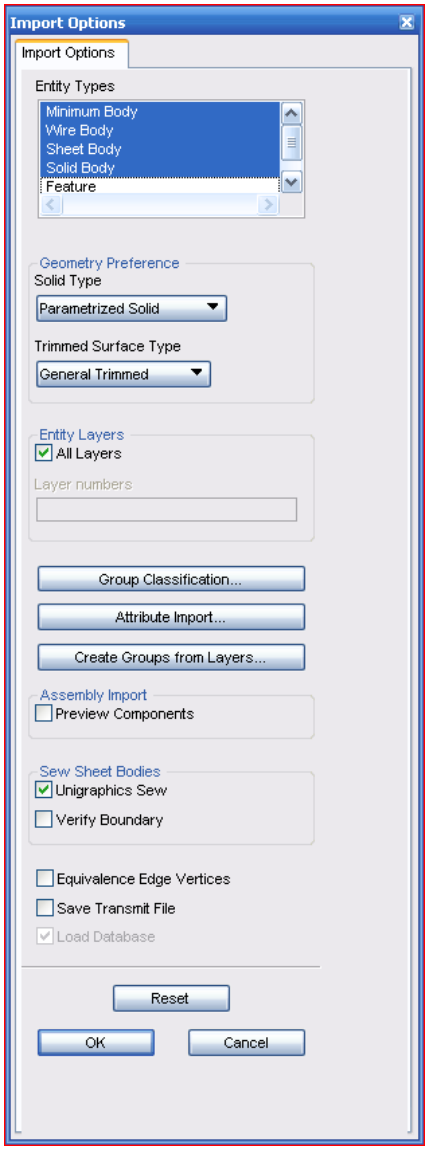


Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **Import a Unigraphics model file with UG attributes.**



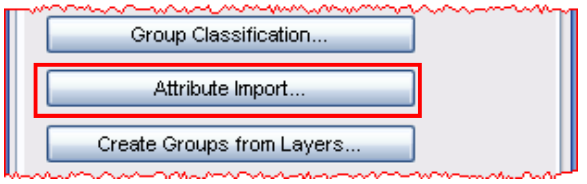


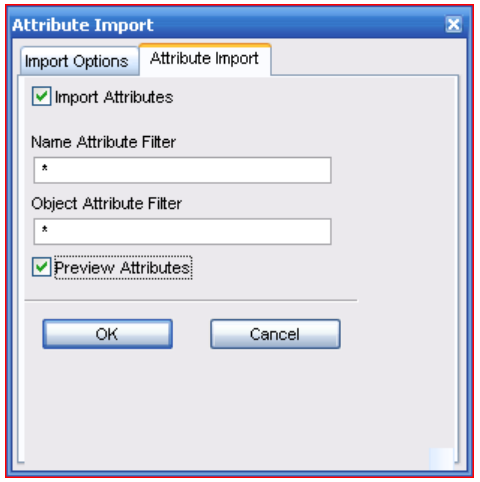
- **Unigraphics Options allows the specification of attribute import.**

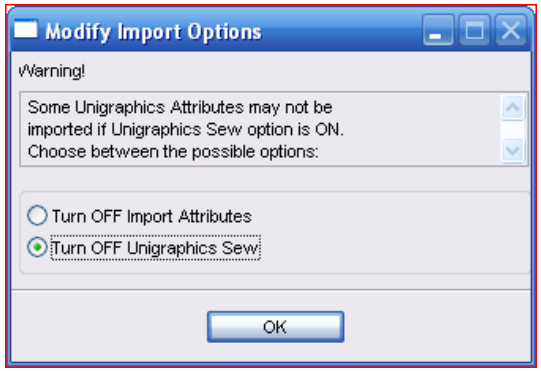
Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **There are Name Attributes and Object Attributes:**
  - Unigraphics handles are imported as Patran attributes.
  - Unigraphics handles correspond to Unigraphics entities.
- **Must select one of two options for importing attributes:**
  - Unselect importing attributes
  - Unselect doing Unigraphics sew







Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **Information in session file:**

(1) Create Parasolid transmit file from Parasolid file, hemi_circ.prt.

(2) List name and object attributes.

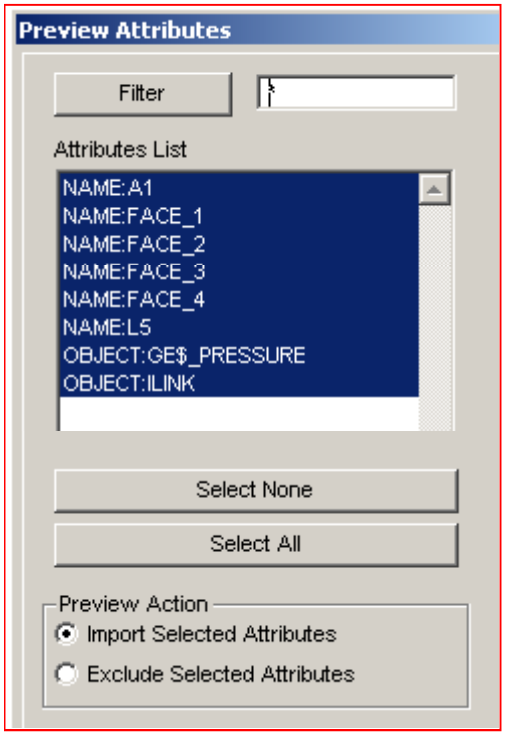
(3) Import Parasolid transmit file, hemi_circ-ps0001.xmt_txt.

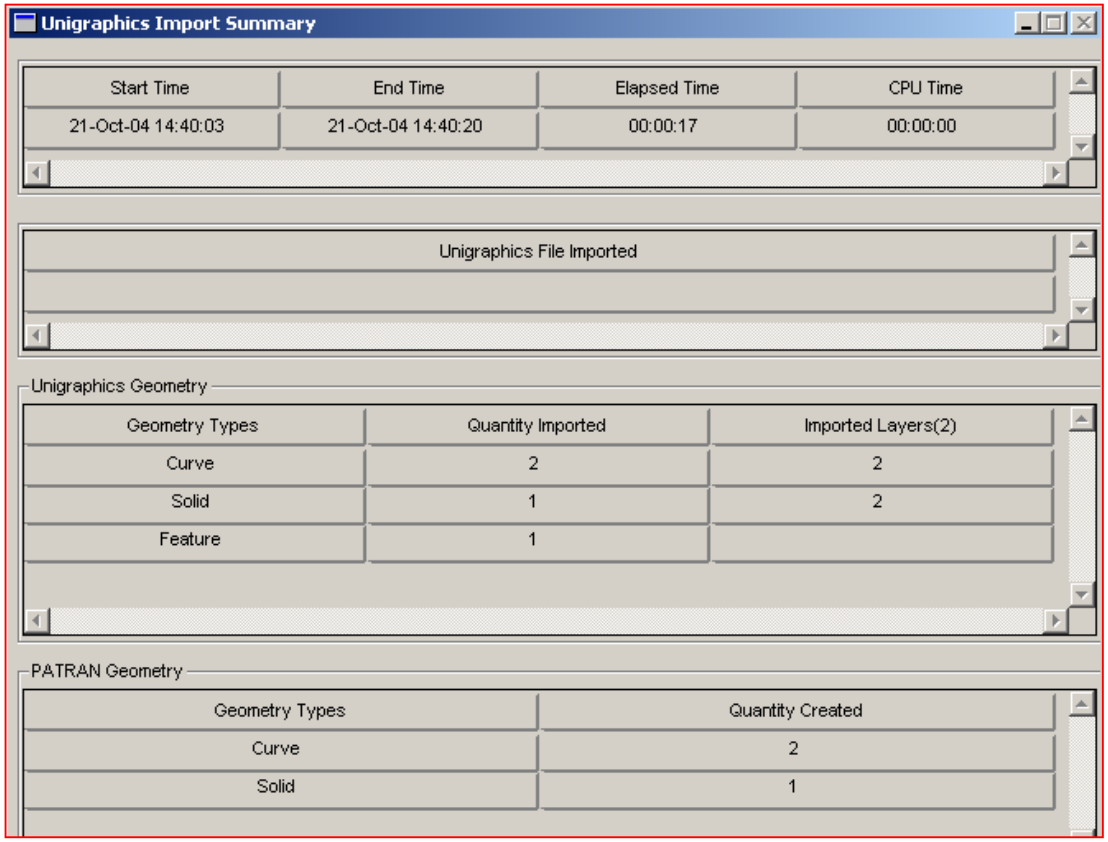
```
p3_ps_open_ug_v2( "C:\Cdev\PAT302\2005\CN\CAD_import\UG\Attributes\" // @
"hemi_circ.prt", [2, 3, 4], [0], ["Attributes", "2", "*", "*", "I", "0",  @
"Components", "0"], uil_file_import_paras_num_parts )  (1)

p3_ug_get_attr_list( "C:\Cdev\PAT302\2005\CN\CAD_import\UG\Attribut" // @
"es\hemi_circ.prt", "*", "*", uil_file_import__num_attributes,  @
uil_file_import_attribute_names )  (2)

p3_ps_open_ug_v2( "C:\Cdev\PAT302\2005\CN\CAD_import\UG\Attributes\" // @
"hemi_circ.prt", [1, 3], [0], ["Attributes", "2", "*", "*", "I", "0",  @
"Components", "0"], uil_file_import_paras_num_parts )

p3_ug_xmt_import_v1( ".\hemi_circ-ps0001.xmt_txt",  @
"C:\Cdev\PAT302\2005\CN\CAD_import\UG\Attributes\v2k5_UG_att3_ug_co" // @
"py\hemi_circ.prt_msc_v2k5_UG_att3", "", TRUE, 0,  @
uil_file_import__total_ug_count, uil_file_import__total_p3_count,  @
uil_file_import_paraso_n_layers, uil_file_import_paras_layer_ids,  @
uil_file_import_total_model_tol, TRUE, "" )  (3)

db_put_origin_model_name( "hemi_circ C:\Cdev\PAT302\2005\CN\CAD_imp" // @
"ort\UG\Attributes\v2k5_UG_att3_ug_copy\hemi_circ.prt_msc_v2k5_UG_att3",  @
"C:\Cdev\PAT302\2005\CN\CAD_import\UG\Attributes\hemi_circ.prt" )

$# Unigraphics Part File Traversal Completed.

pfk_create_ug_import( "C:\Cdev\PAT302\2005\CN\CAD_import\UG\Attribu" // @
"tes\v2k5_UG_att3_ug_copy\hemi_circ.prt_msc_v2k5_UG_att3", [1, 3], [0], [ @
"Attributes", "2", "*", "*", "I", "0", "Components", "0"], 0, "", FALSE, 3, 1, @
 52054, FALSE, [0., 0., 0., 0., 0., 0., 0., 0., 0., 0., 0., 0.], 0, ["0"], [[ @
-1]], [[-1]], [[-1]], TRUE, [0], TRUE, [0], TRUE, [0], 3, 1, 1 )
```


Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **Patran preview attributes and import summary window appears:**

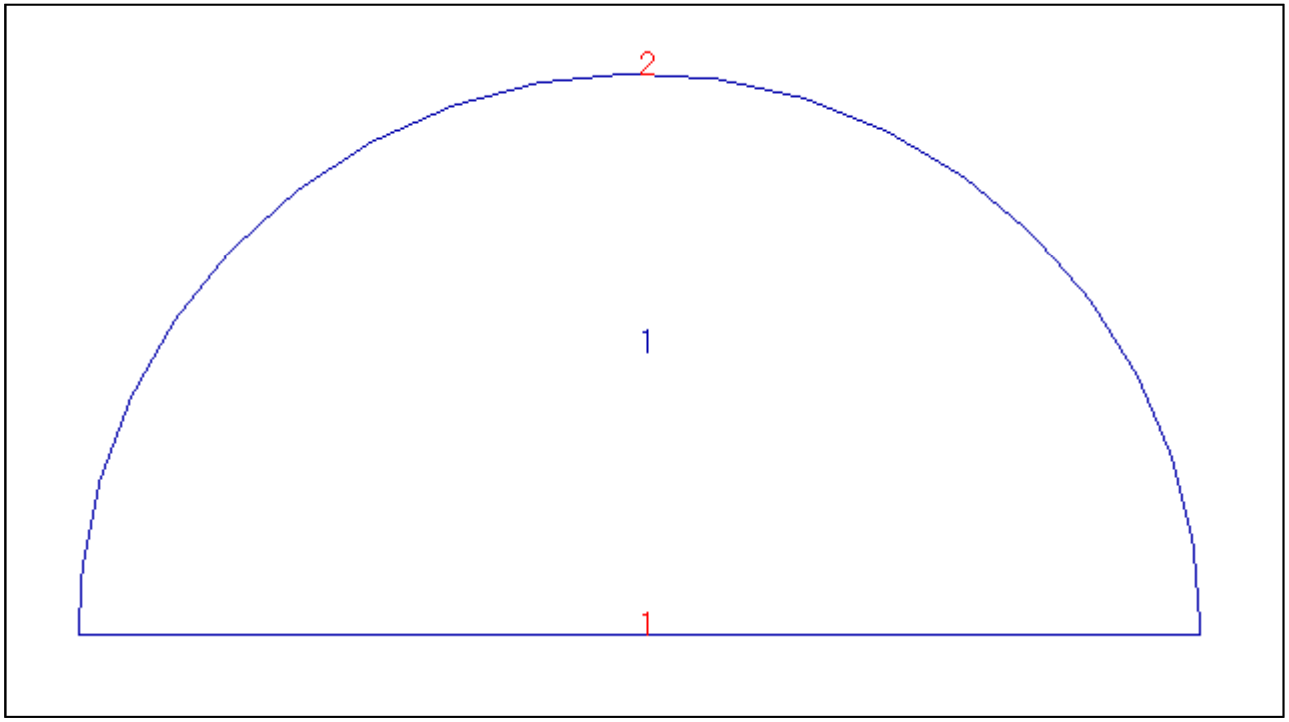




Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **Imported Parasolid solid and Parasolid curves**



Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **Determine which attribute entities are using PCL**

```
FUNCTION test_attribute_import( output_name )

   STRING output_name[]
   INTEGER i, j, status, nc, nsrf, nsol, entity_id(1), min_label, label, objAttCount
   STRING  attr_name[VIRTUAL], str_title[VIRTUAL](VIRTUAL), str_value[VIRTUAL](VIRTUAL)
   INTEGER endLabel, startLabel, fid

   utl_process_spawn( "rm -f " // output_name, TRUE )
   status = fio_openw( output_name, fid )

   status = db_count_solid( nsol )
   status = db_get_min_solid_label( min_label )
   status = db_get_max_solid_label( endLabel )
   label = min_label
   startLabel = label

   fio_writec( fid, "minimum solid label = %d \n", startLabel )
   fio_writec( fid, "maximum solid label = %d \n", endLabel )
   IF( label > 0 ) THEN
      fio_writec( fid, "Number of solids = %d\n",nsol)
      FOR (i = startLabel TO endLabel)
         status = db_get_solid_id( label, entity_id(1) )
         status = p3_cad_get_entity_name( entity_id(1), 4, attr_name )
         IF( status == 0 ) THEN
            fio_writec( fid, "solid %d : Attribute name = %s\tstatus is %d\n",entity_id(1), attr_name, status )
         END IF

         status = p3_cad_get_entity_strings( entity_id(1), 4, objAttCount, str_title, str_value )
         FOR( j = 1 TO objAttCount )
           fio_writec( fid, "     solid %d : string attribute: %s = %s\n",entity_id(1), str_title(j), str_value(j))
         END FOR
         fio_writec( fid,  "\n" )
         label += 1
      END FOR
   END IF
   ...
   ...
END FUNCTION
```


Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **PCL function for retrieving CAD name attribute**

p3_cad_get_entity_name (p3id, p3type, attribute)

Description:

This function will retrieve a string of a CAD name attribute for a MSC.Patran entity. The MSC.Patran identifier and entity type are used to lookup the associated CAD name attribute.

Input:

INTEGER	p3id	The MSC.Patran id of the entity used to look up the associated CAD name attribute.
INTEGER	p3type	The MSC.Patran entity type. Valid entity type values are: 1 for a point, 2 for a curve, 3 for a surface, and 4 for a solid.

Output:

STRING	attribute	The CAD name attribute.
INTEGER	<Return Value>	This function returns a value of 0 if successful.

Error Conditions:

1	Out of memory
10	A corrupt attribute was detected.
255	Invalid string length.
257	Can't reallocate virtual string array.
259	Not a virtual string.
13000211	Virtual memory has been exhausted.
13000212	There is an error interacting with the PERSISTENT_MEMORY relation in the database.
13000213	A database lookup failed to locate the target index key.
<non-zero>	Attribute not found

Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **PCL function for retrieving CAD string attribute**

p3_cad_get_entity_strings (p3id, p3type, attrib_count, attrib_titles, attrib_values)

Description:

This function will retrieve all CAD string attributes for a MSC.Patran entity. The MSC.Patran identifier and entity type are used to look up the associated CAD string attributes.

Input:

INTEGER	p3id	The MSC.Patran id of the entity used to look up the associated CAD name attributes.
INTEGER	p3type	The MSC.Patran entity type. Valid entity type values are: 1 for a point, 2 for a curve, 3 for a surface, and 4 for a solid.

Output:

INTEGER	attrib_count	The number of string attributes associated to the referenced MSC.Patran entity.
STRING[]	attrib_titles	An array of CAD name attribute titles.
STRING[]	attrib_values	An array of CAD name attribute values.
INTEGER	<Return Value>	This function returns a value of 0 if successful.

Error Conditions:

1	Out of memory
10	A corrupt attribute was detected.
255	Invalid string length.
257	Can't reallocate virtual string array.
259	Not a virtual string.
13000211	Virtual memory has been exhausted.
13000212	There is an error interacting with the PERSISTENT_MEMORY relation in the database.
13000213	A database lookup failed to locate the target index key.
<non-zero>	Attribute not found

Copyright© 2012 MSC.Software Corporation

IMPORT UNIGRAPHICS MODEL FILE

- **Compile and execute the PCL function:**

```
!!input test_attribute_import
```

```
!!input test_attribute_import
$# Compiling: test_attribute_import
$#  Compiled: test_attribute_import
```

```
test_attribute_import("t4_p1_1_pclAttDump.txt" )
```

t4_p1_1_pclAttDump.txt

```
minimum solid label = 1
maximum solid label = 1
Number of solids = 1

minimum surface label = 0
maximum surface label = 0
Number of surfaces = 0

minimum curve label = 1
maximum curve label = 2
Number of curves = 2
curve 1 : name attribute = L5 status is 0
curve 1 : string attribute:  ILINK = RMheni_circ_seed.prt R0000003800000001
curve 2 : name attribute = A1 status is 0
curve 2 : string attribute:  ILINK = RMheni_circ_seed.prt R0000003900000001
```


Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

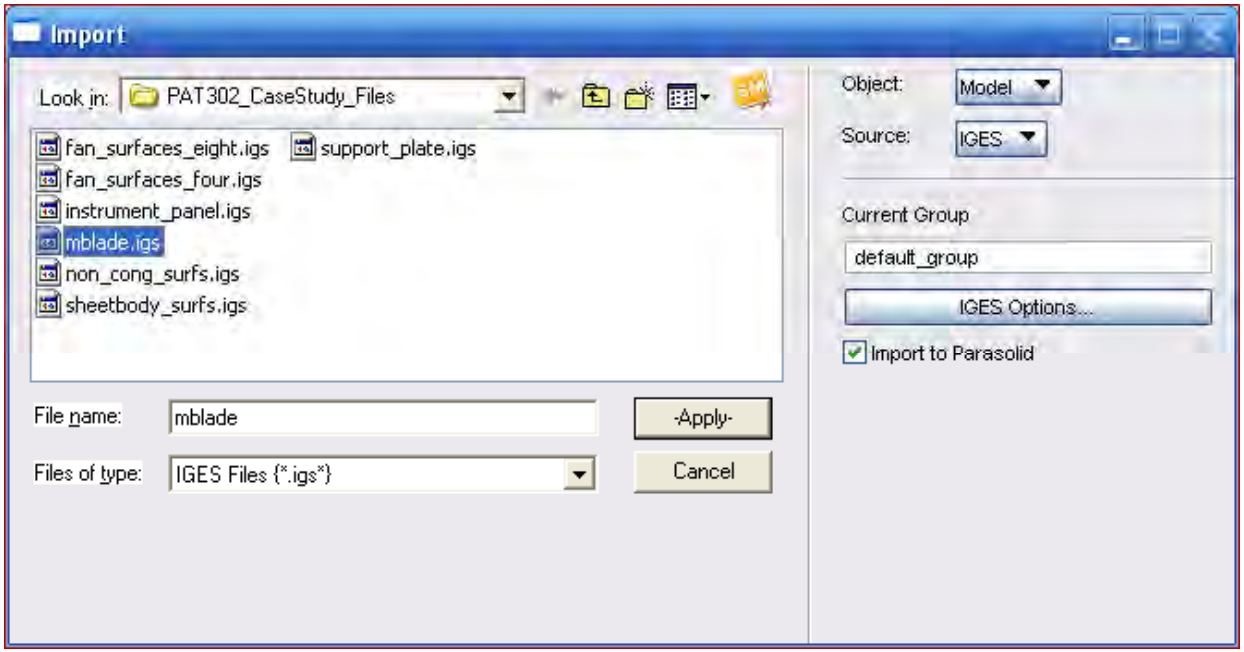
IMPORT IGES MODEL FILE

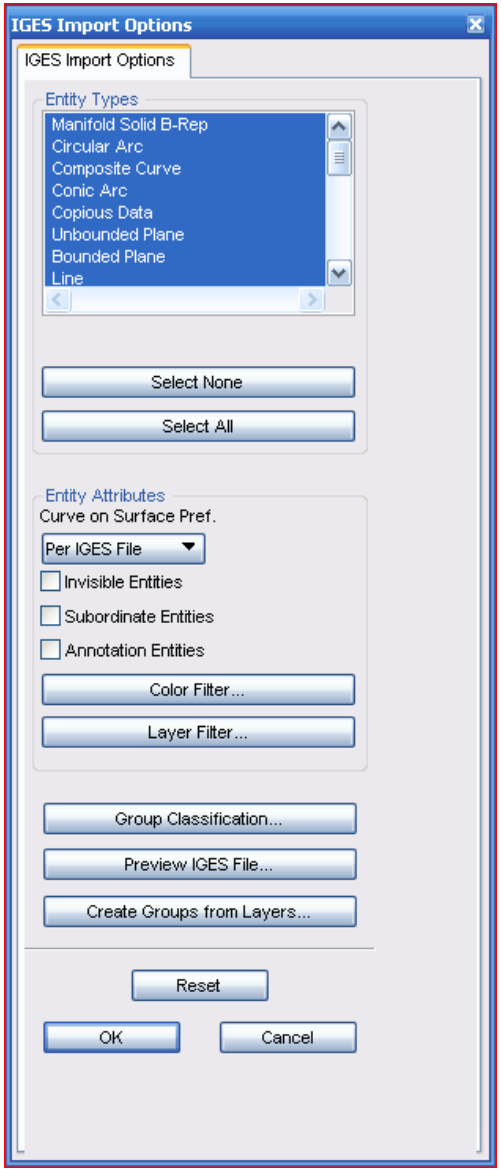
Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

IMPORT IGES MODEL FILE

- **Import an IGES model file, mblade.igs**

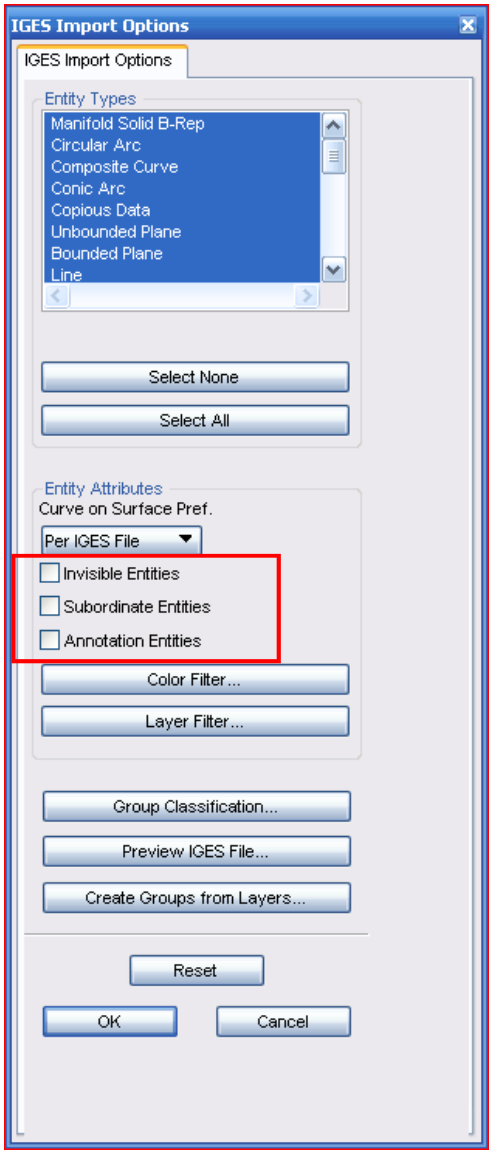




Copyright© 2012 MSC.Software Corporation

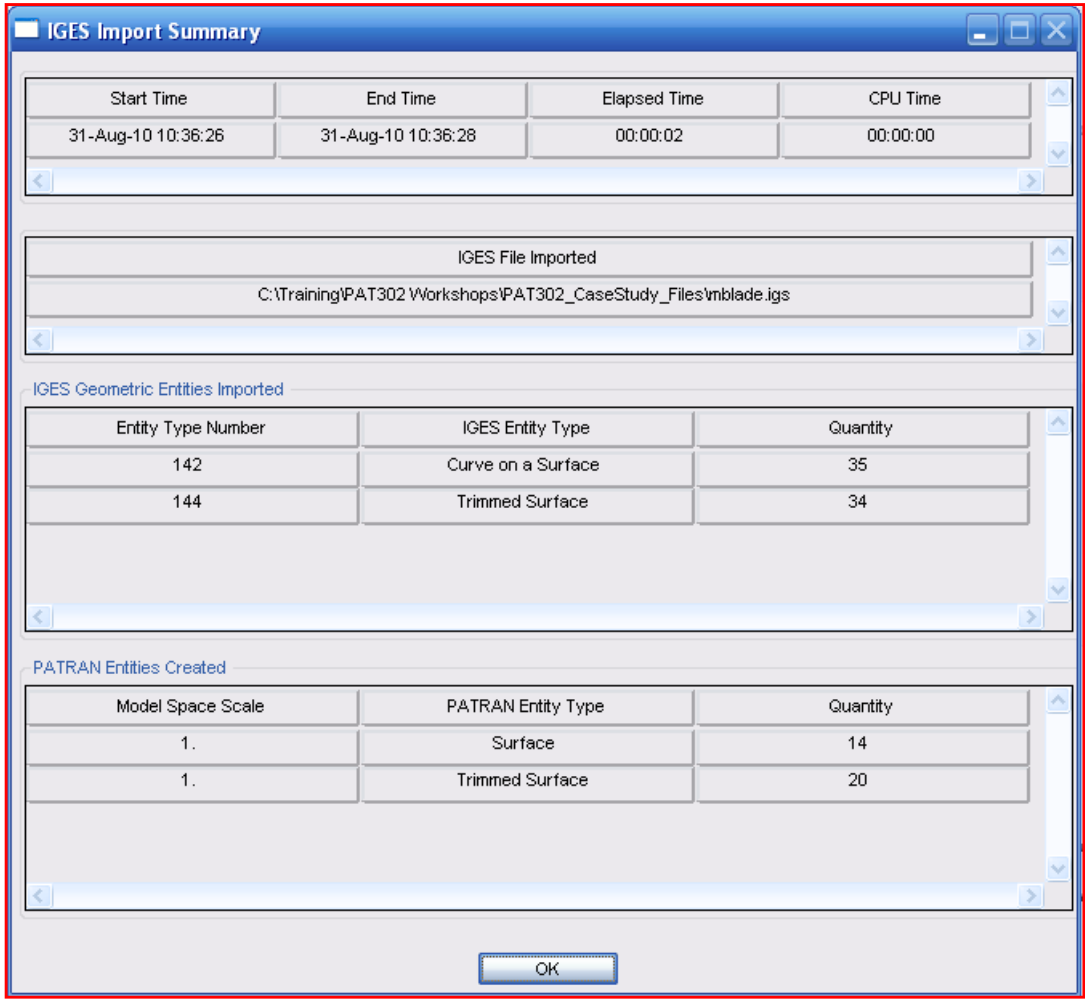
IMPORT IGES MODEL FILE

- **Occasionally it may be necessary to select some of the additional entity classifications**
  - Invisible Entities
  - Subordinate Entities
  - Annotation Entities
- **Not selecting these may prevent the import of the IGES model**
  - Some CAD packages output IGES with entities incorrectly tagged



Copyright© 2012 MSC.Software Corporation

IMPORT IGES MODEL FILE



Copyright© 2012 MSC.Software Corporation

IMPORT IGES MODEL FILE

- **If your IGES part contains surface, it may be best to toggle the "Import to Parasolid" option off**
- **Comments on selecting a GMT for import of an IGES file**
  - It is important that the GMT not be too large so that surface edges become degenerate upon import (edge vertices equivalenced).
  - For surface meshing, it is important that the GMT not be too small so that close (adjacent) vertices are not equivalenced upon import.
  - Largest Distance Between Adjacent Vertices **<** GMT **<** Shortest Edge Length
    - Shortest edge can be found using Geometry: Show/Curve/Length Range
    - No way of determining the largest distance between all the adjacent vertices.
  - Its best to select the value for the GMT so use of Geometry: Edit/Surface/Sew can be avoided.
    - For example model, use 0.001 <= GMT <= 0.01
  - Once in a while, it is possible to use the following "trick"
    - CAD (e.g. Pro/E) -> Patran fails, but obtain Patran's estimate of the GMT for the import.
    - CAD (e.g. Pro/E) -> IGES -> Patran w/ GMT from failed import.

Copyright© 2012 MSC.Software Corporation

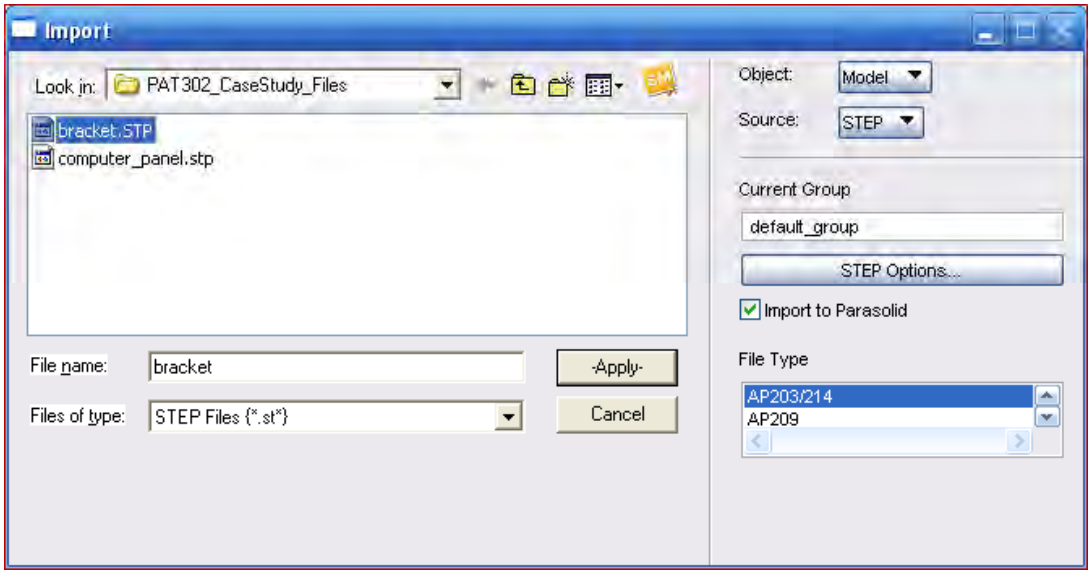
IMPORT STEP MODEL FILE

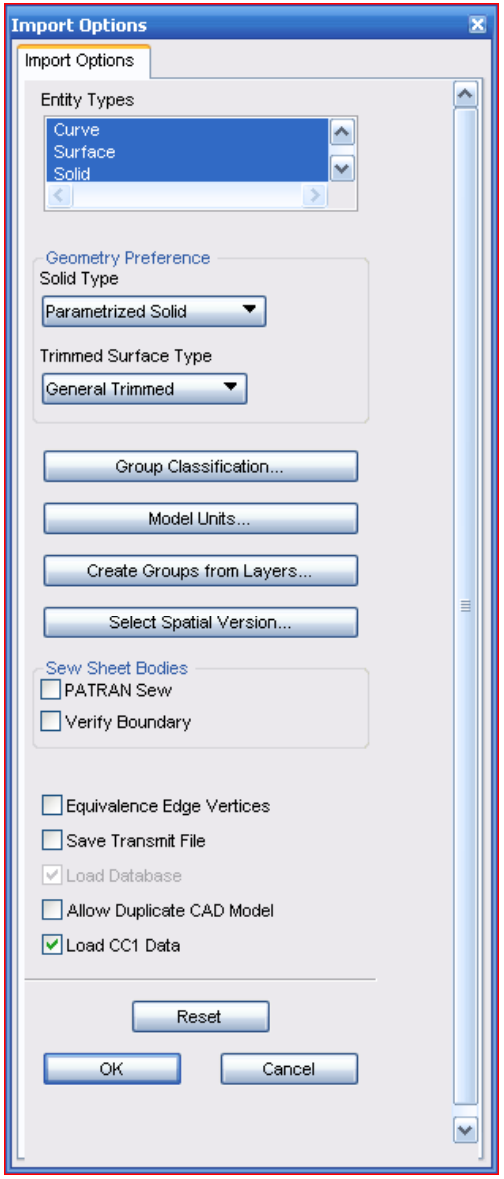
Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

IMPORT STEP MODEL FILE

- **Import a STEP AP203/214 (geometry only) model file using Import to Parasolid.**





- **The STEP Options (AP203/214) are essentially the same as those for the Parasolid xmt import.**

Copyright© 2012 MSC.Software Corporation

IMPORT STEP MODEL FILE

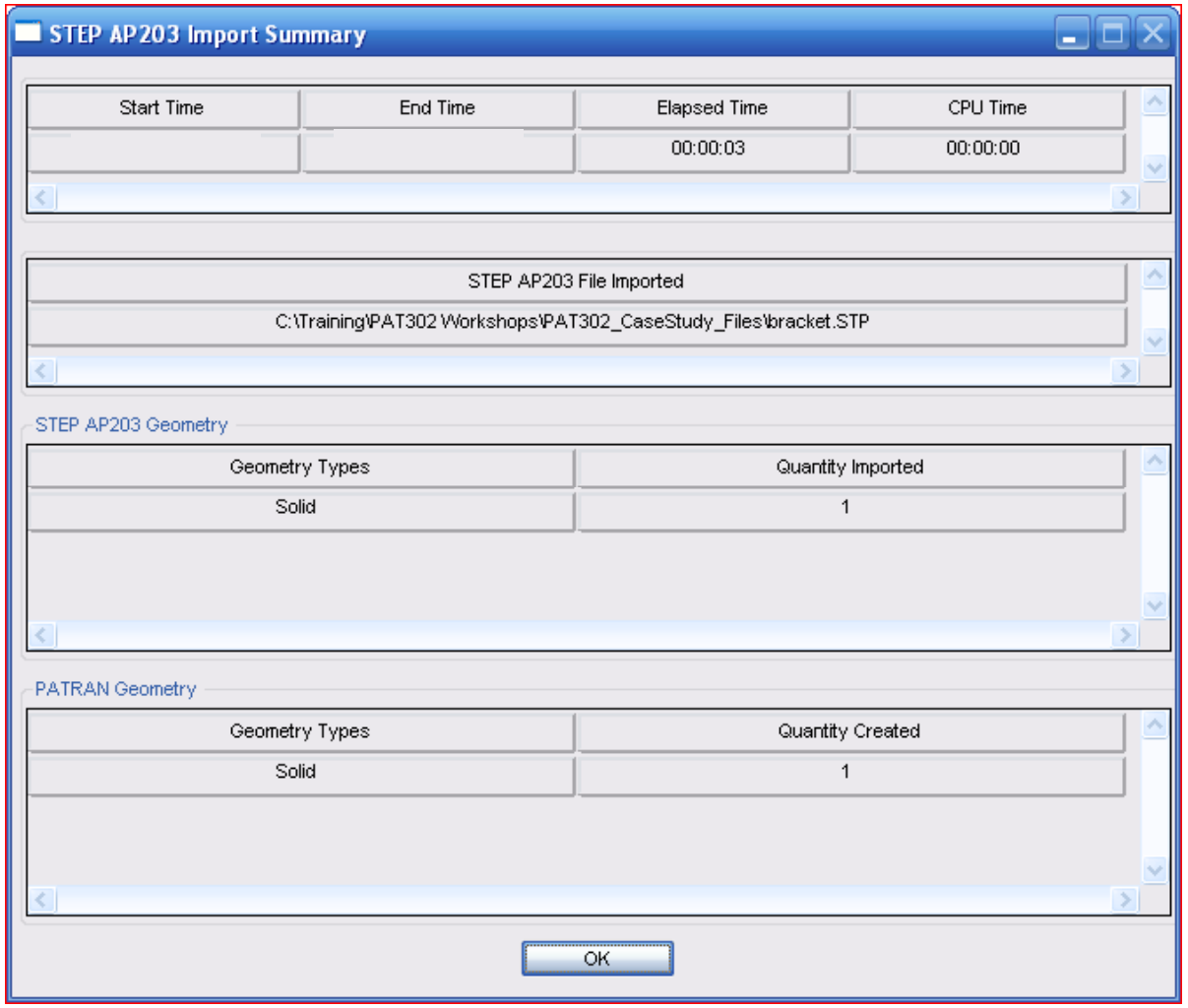
- **The following messages were written to the session file during import:**

```
p3_express_options_file( "express.opt", FALSE, 3, 1, 52054, TRUE, [39.370079,  @
0., 0., 0., 39.370079, 0., 0., 0., 39.370079, 0., 0., 0.], 0, ["0"], [[-1]], [ @
[-1]], [[-1]], TRUE, [0], TRUE, [0], TRUE, [0], 3, 1, 1 )
...
utl_process_spawn( "C:\msc\patran2005\bin\xplus_sp.exe ""C:\Cdev\PA" // @
"T302\2005\CN\CAD_import\STEP\bracket.STP"" bracket MSC.Patran", TRUE )
...
p3_ug_xmt_import( "bracket.xmt_txt", "", "express.opt", TRUE, 0,  @
uil_file_import_s_step203_count, uil_file_import_st_patran_count,  @
uil_file_import_step2_model_tol )
...
uil_file_import_step203para.import_cc1_data(  @
"C:\Cdev\PAT302\2005\CN\CAD_import\STEP\bracket.STP" )
$# Loading dynamic library C:\msc\patran2005\bin\exe\callap203import.dll
```


Copyright© 2012 MSC.Software Corporation

IMPORT STEP MODEL FILE

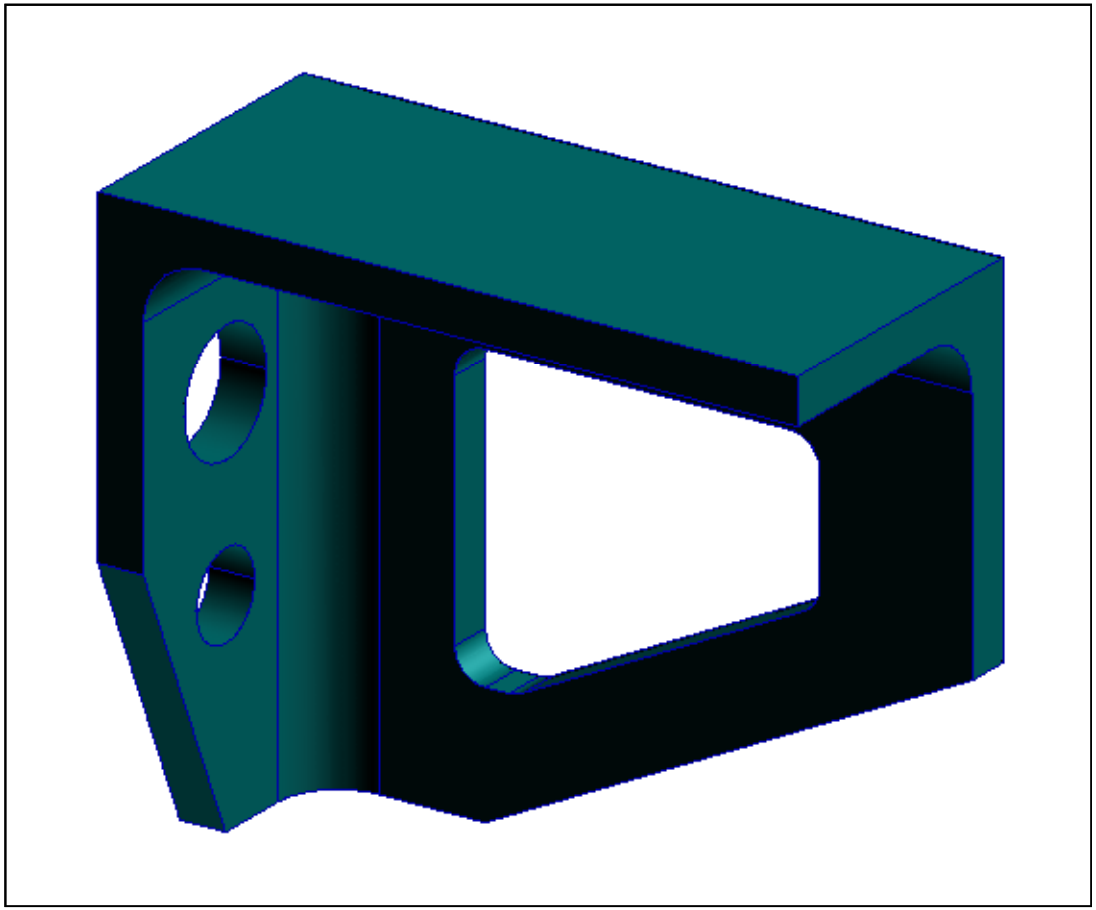
- **Solid (1) was imported.**
- **Solid (1) was created in Patran.**



Copyright© 2012 MSC.Software Corporation

IMPORT STEP MODEL FILE

- **Imported solid:**



Copyright© 2012 MSC.Software Corporation

EXERCISES

- **Perform Workshop 17 “Geometry Import and Repair” in your exercise workbook**
- **Perform Workshop 18 “Editing Imported Geometry with the Feature Recognition Tool” in your exercise workbook**
- **Optional: users of Unigraphics may like to go through Workshops 23 and 24**

Copyright© 2012 MSC.Software Corporation

SECTION 8

FIELDS

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

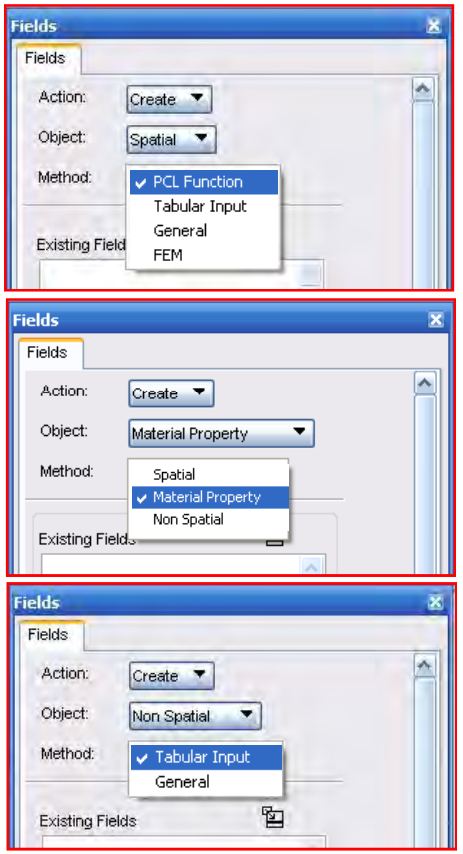
FIELDS

- **Fields (functions) allow the creation and modification of a multitude of data sets. Data fields are used in the following modeling areas:**
  - Loads and boundary conditions
  - Material properties
  - Element properties
- **Field input can either be tabular or continuous functions, with the input being scalar or vector.**
- **Complex (number) scalar fields are also permitted if the Nastran analysis preference is used.**
  - This allows, for example, real/imaginary or magnitude/phase components of a frequency dependent function to be defined.

Copyright© 2012 MSC.Software Corporation

TYPES OF FIELDS

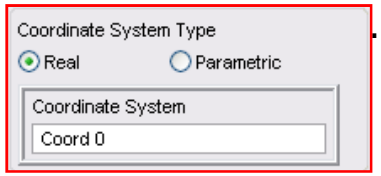
- **There are three basic types of fields with several different input options. They are summarized as follows:**
  - Spatial field
    - These fields describe data based on spatial variation. They can vary over real space, or parametric space for geometry. Spatial fields can be either scalar or vector.
  - Material property field
    - Defines a material property as a function of temperature, strain, strain-rate, time or frequency (the material state variable), or an appropriate combination of these variables.
  - Non-spatial field
    - Defines a scalar field for dynamic analysis applications. Function can vary with time, frequency, temperature, displacement, velocity, or a user-defined variable.



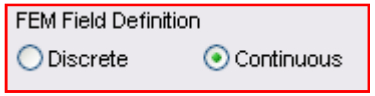
Copyright© 2012 MSC.Software Corporation

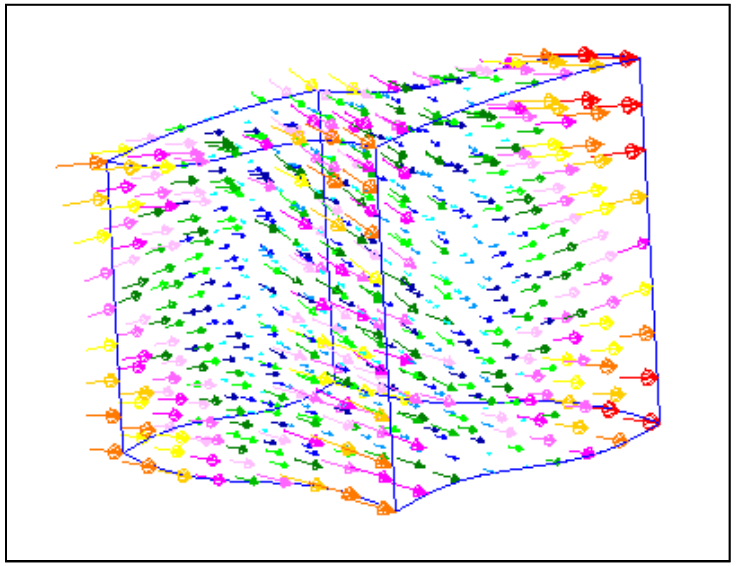
BASICS OF SPATIAL FIELDS

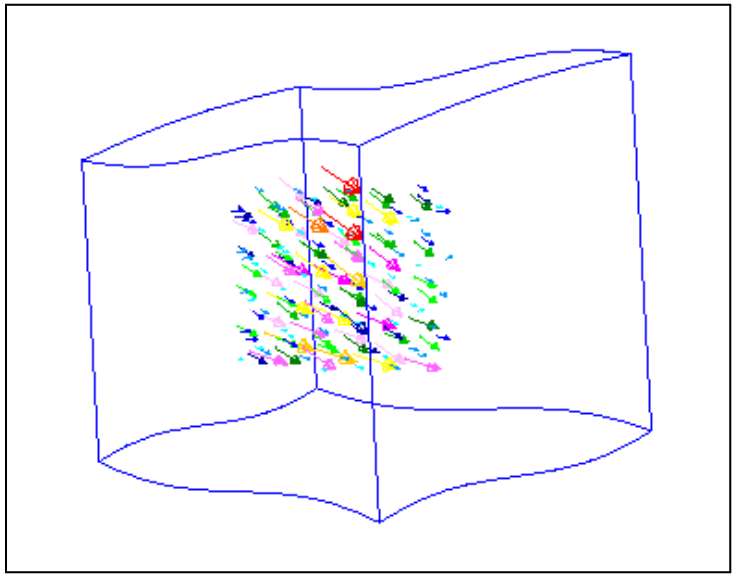
- **Spatial fields have several parts to their definitions.**
  - The first part is the region the field will be applied to
    - The field can be used for the entire or part of the region of the model if the Real, or Discrete or Continuous option is selected, depending on the field type being



OR



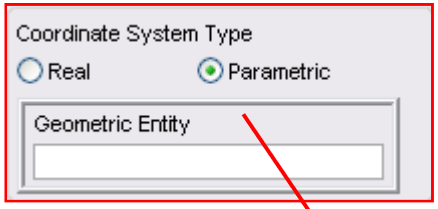




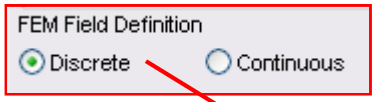
Copyright© 2012 MSC.Software Corporation

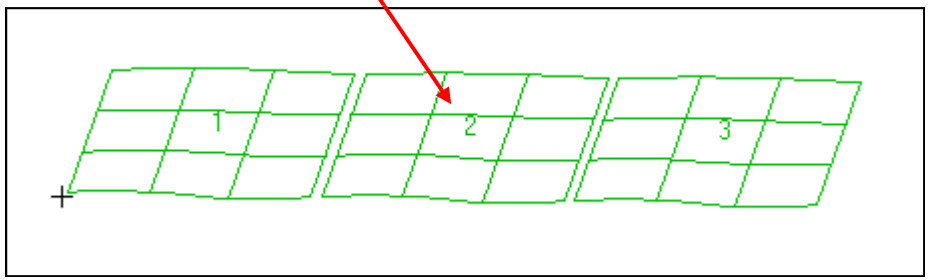
BASICS OF SPATIAL FIELDS

- If the field is being used for the region of a single geometric entity, e.g. surface, or part or all of the finite element entities, e.g. nodes, the Parametric or Discrete options are available.

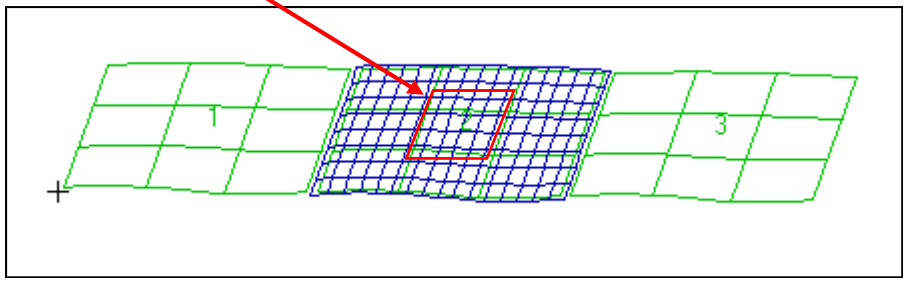


OR





Domain (application region) is over single surface

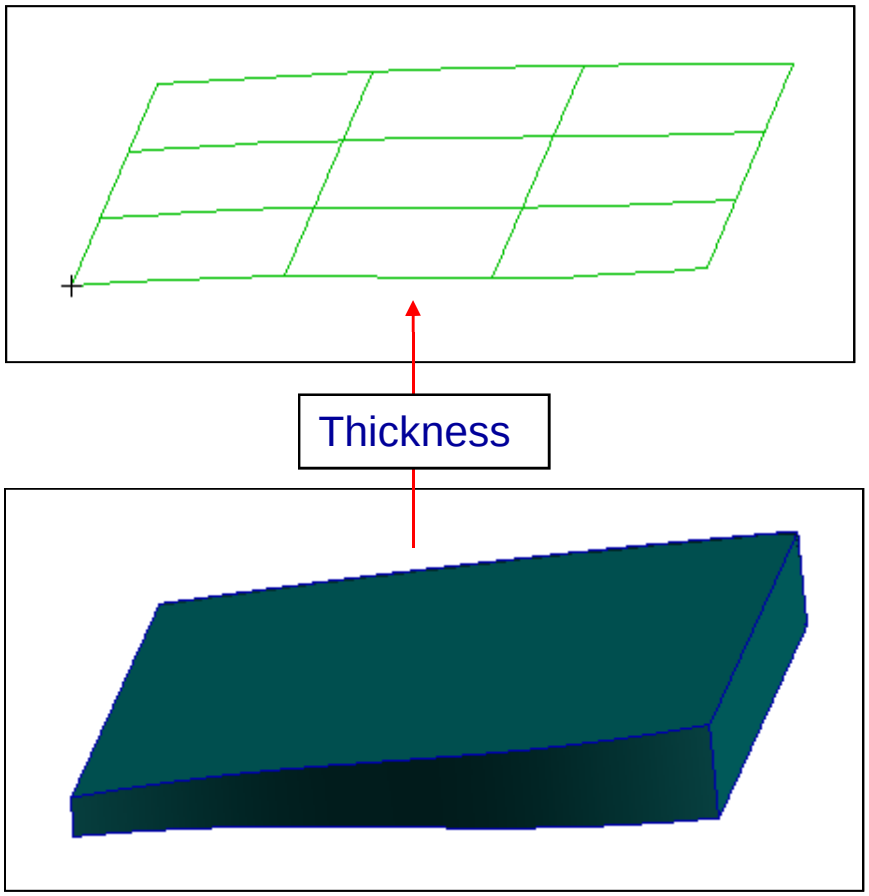


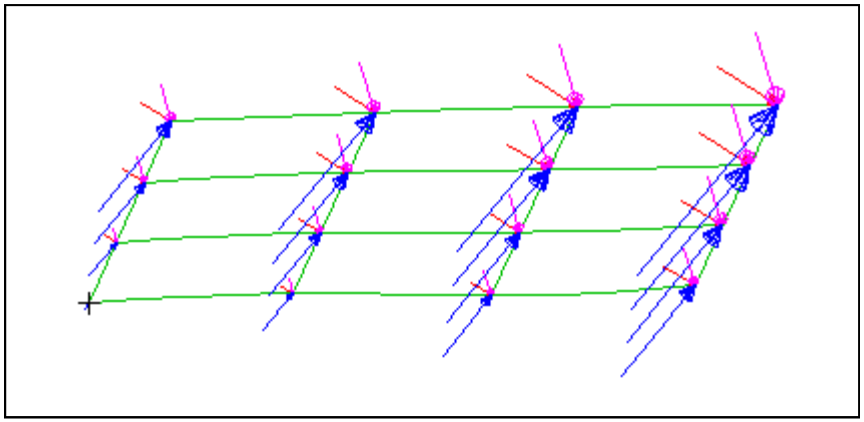
Use subset of all nodes

Copyright© 2012 MSC.Software Corporation

BASICS OF SPATIAL FIELDS

– The second part is the type of field being created, scalar or vector.



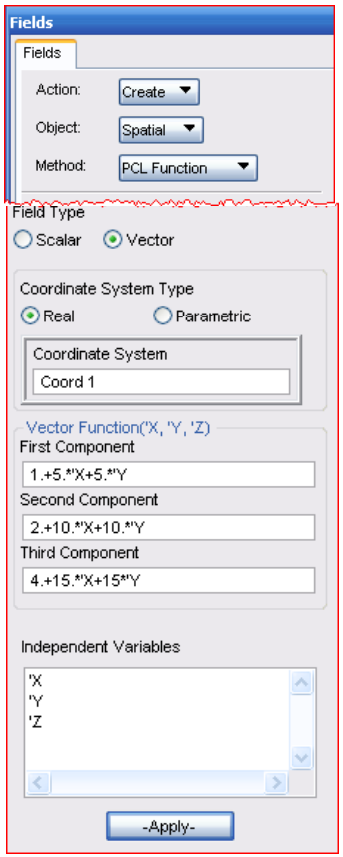


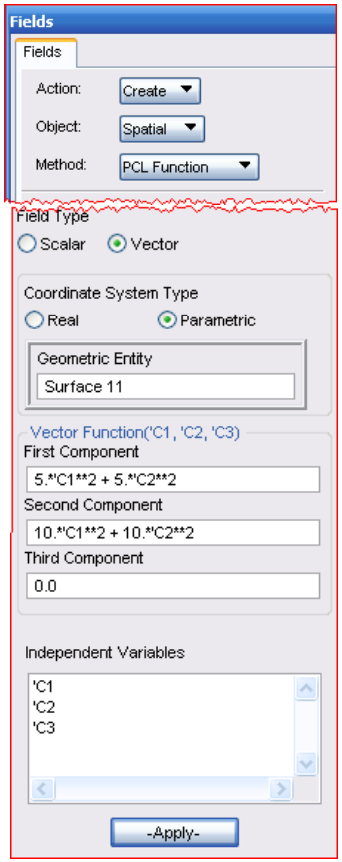
Vector Field

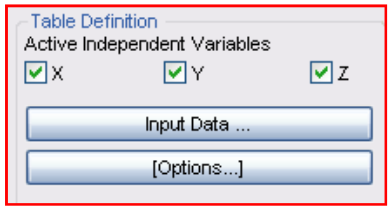
Copyright© 2012 MSC.Software Corporation

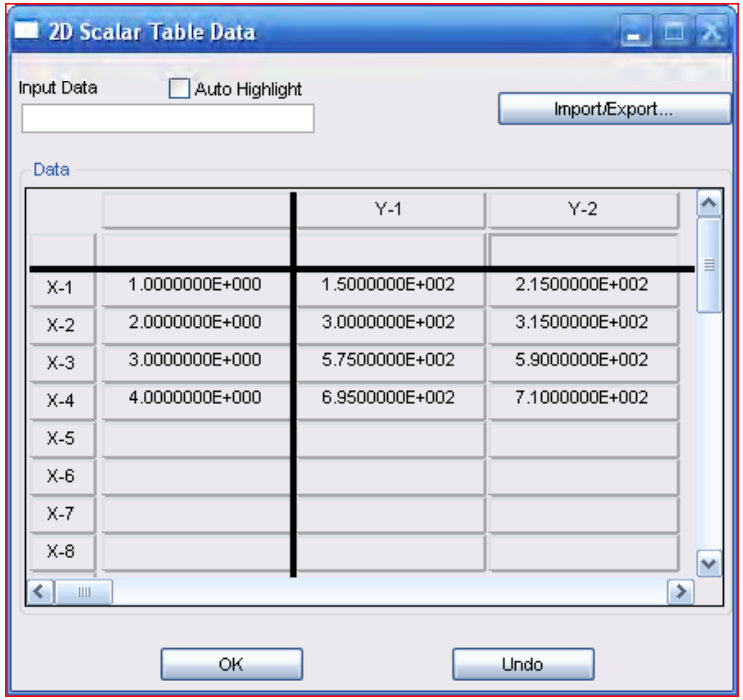
BASICS OF SPATIAL FIELDS

– The third part is the specification of the field data.





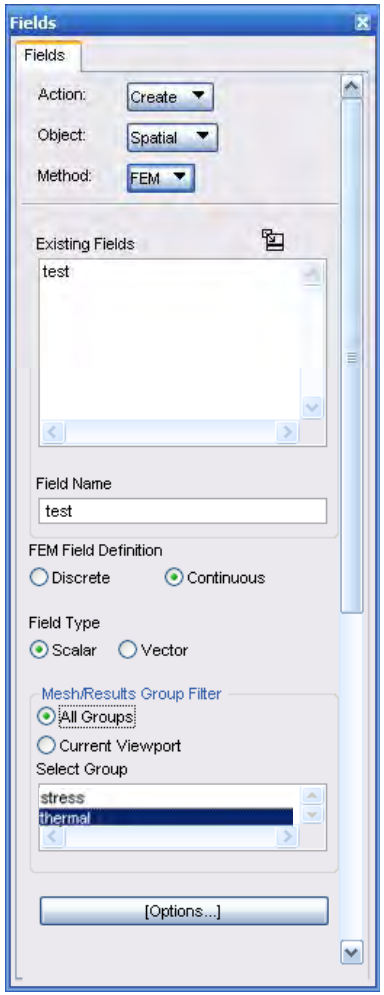


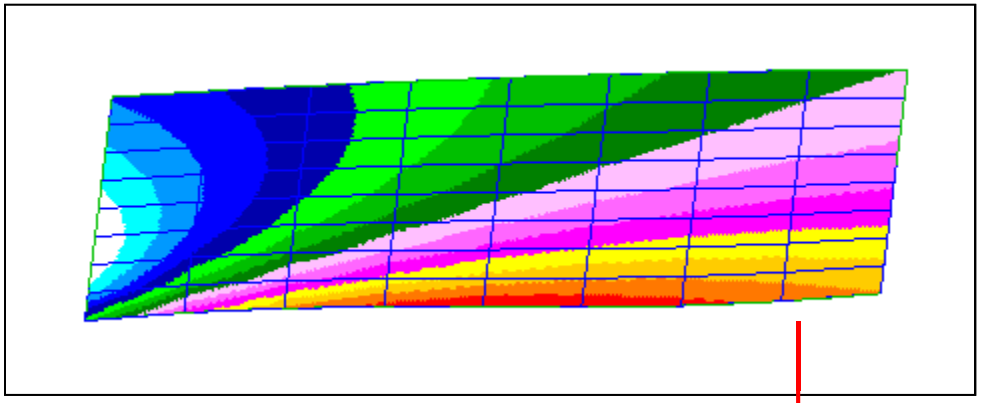


Copyright© 2012 MSC.Software Corporation

BASICS OF SPATIAL FIELDS

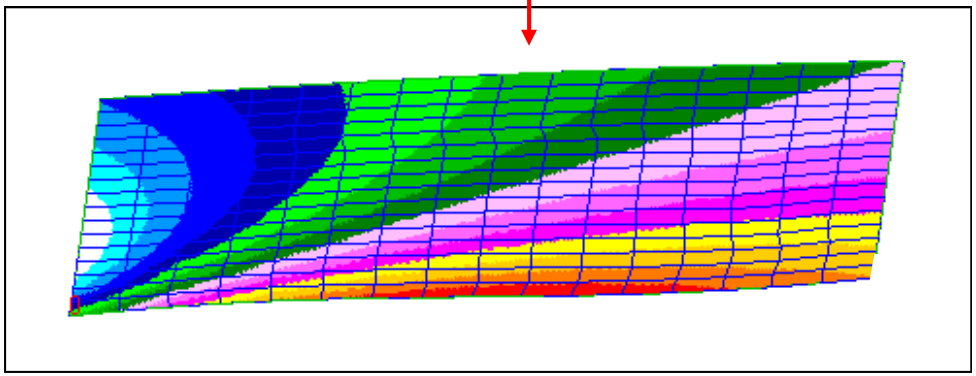
– The third part is the specification of the field data.





Thermal Results

FEM field

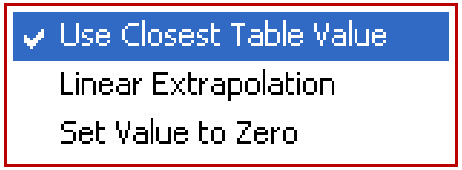


Stress Model LBC From Thermal Results

Copyright© 2012 MSC.Software Corporation

BASICS OF SPATIAL FIELDS

- **Some field types have options that allow specification of the averaging method between or beyond data points.**

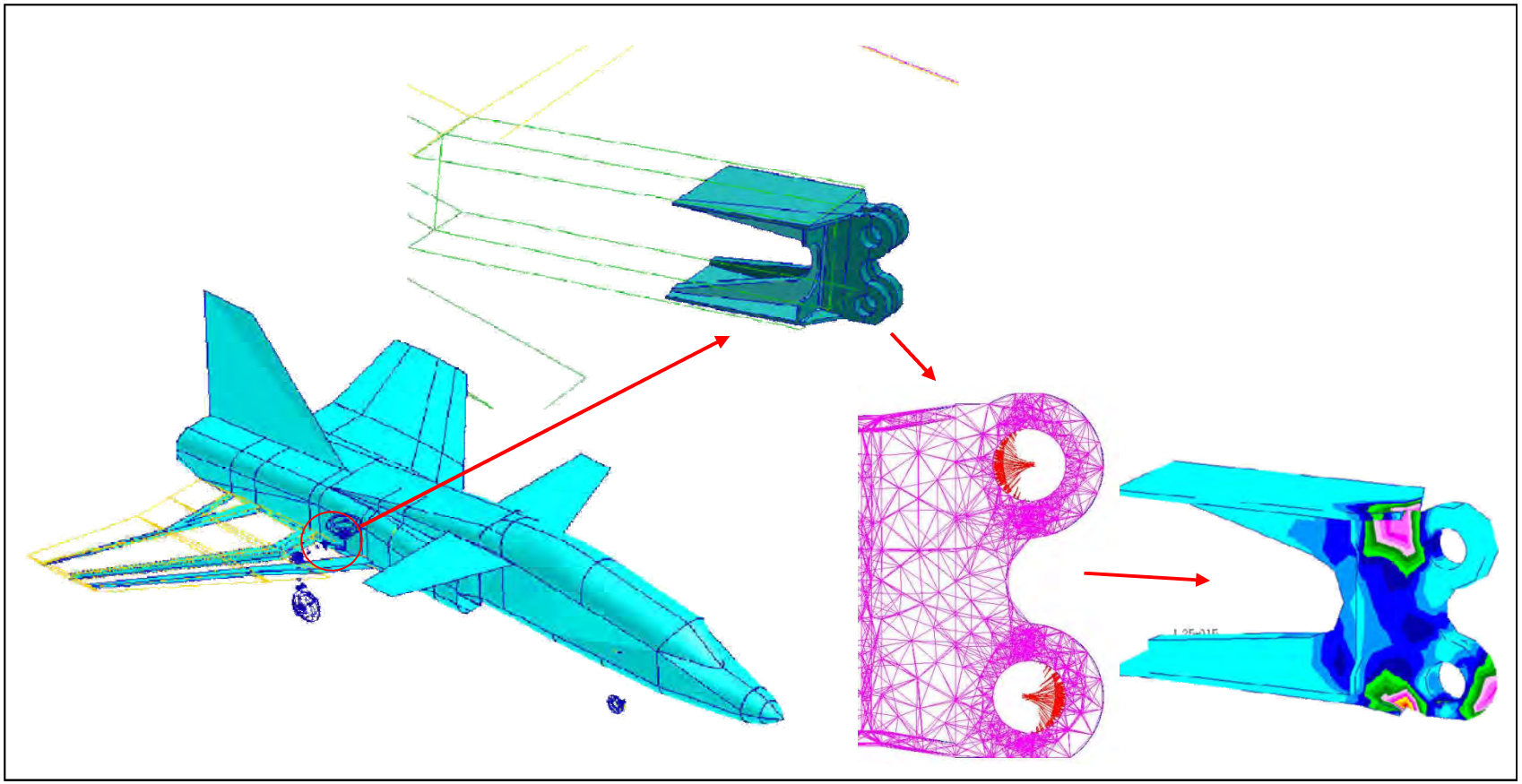


  - Use Closest Table Value – use table value whose independent variable value(s) is closest to that of the interpolation point
  - Linear Extrapolation – extrapolate beyond or interpolate between table entries
  - Set Value to Zero – specify data as zero beyond table entries
- **In some cases, the averaging options will have no effect. Their effect is dependent on how the field is used, and sometimes, on which finite element solver is used.**
- **Because averaging will be done, fields need to be checked for accuracy.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, BOLT LOADING REPRESENTED BY COS2 FIELD

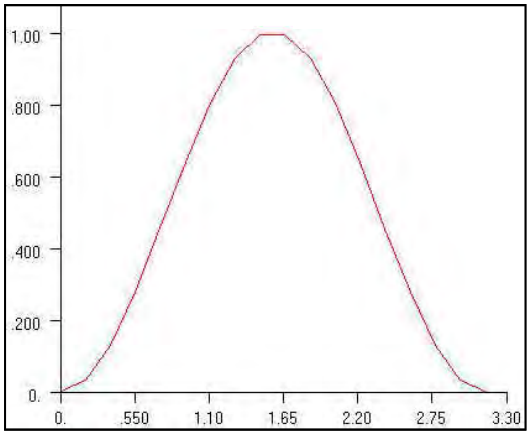
- **Create a real scalar field to represent a cos2 bolt loading.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, GOAL

- **The goal is to conduct a preliminary design study of a wing-to-body 3D fitting. Due to the high loads and close (very similar) dimensions, careful attention to the loading conditions must be considered even for the initial study.**
- **To make the loading condition more realistic, a loading distribution of cosine squared (cos2q) is used.**

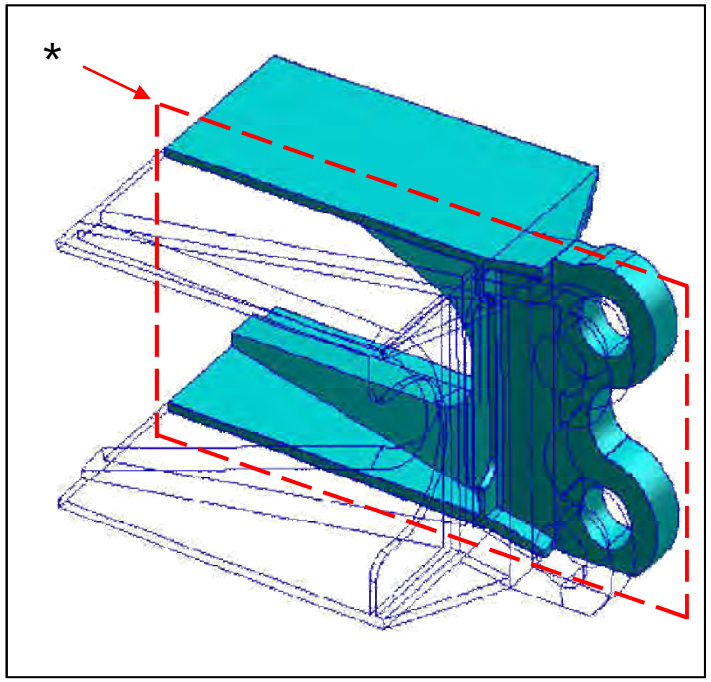


- **To make the pressure loading application region more realistic, the direction in which the load from the pin acts will also be considered.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, APPROACH TO APPLYING LOAD

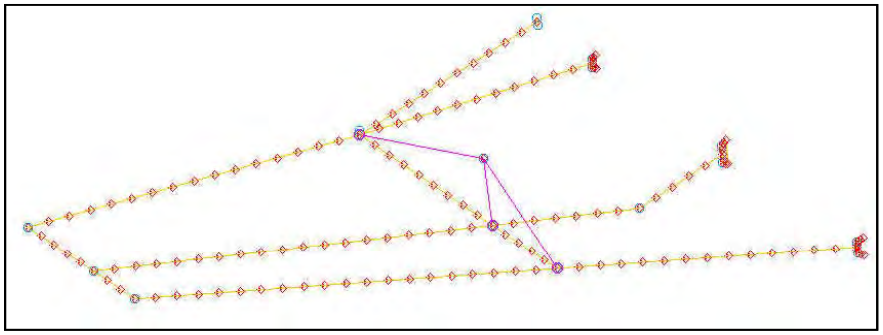
- **Approach to creating the analysis model:**
  - Create a 3D geometry model for the 3D fitting.
    - The model is taken to be symmetric about a vertical plane (*), between the pin hole pairs, that divides the fitting into two pieces:



Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, APPROACH TO APPLYING LOAD

- **Approach to creating the analysis model**
  - The applied loads are obtained from a free-body analysis of a beam element model of the main spars.

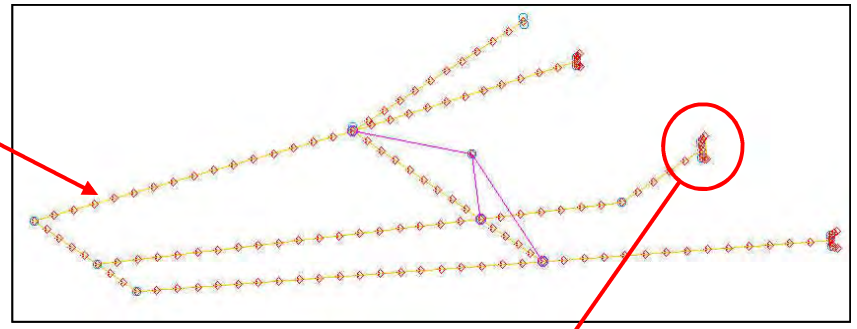


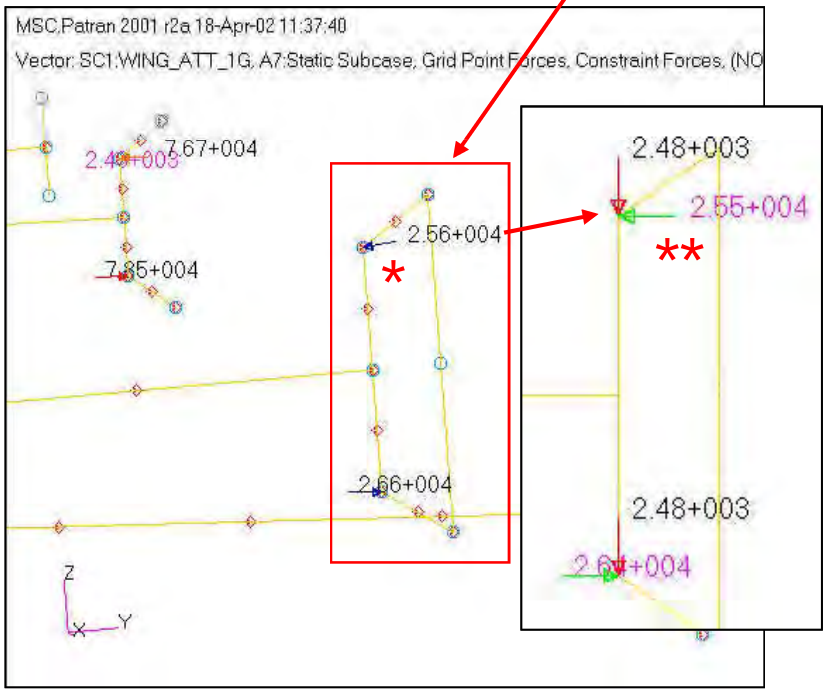
    - The loads for the two halves of the model will be taken to be equal (loads are symmetric about the vertical plane).
  - The applied concentrated forces will have to be represented by pressure.
    - Pressure will be obtained by integrating a pressure field, in the direction of loading, over an application area.
  - The pressure field will be applied to solid geometry faces

Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, CONSTRAINT (REACTION) FORCES

- From the full aircraft model, a beam element representation of the main spares is made and analyzed.
- The free-body forces (*) will be used to determine the pressure loading. (Both halves of the wing-to-body fitting are represented here.)
- The components of these forces (**) will be used to establish the required direction of the pressure loading.

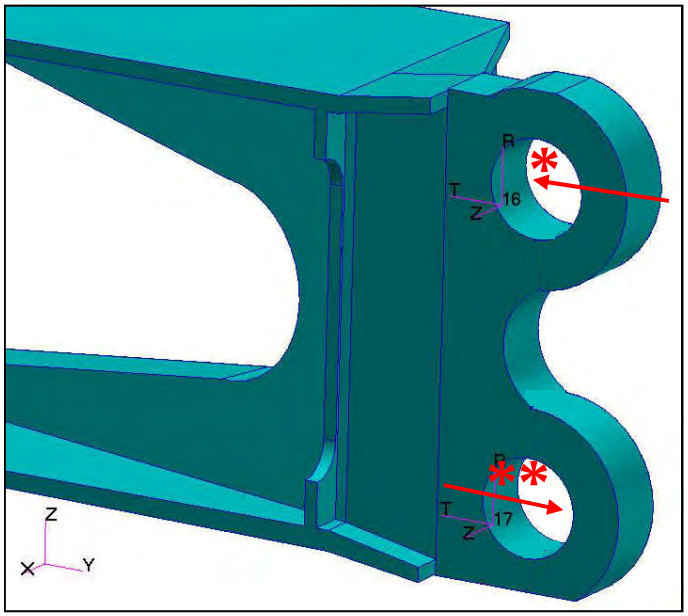




Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, CREATE PRESSURE FROM FORCE

- **The force (both symmetric halves of the fitting are included) at the top (*) and bottom (**) bolts is 25,600 lbf and 26,600 lbf, respectively.**
- **To determine the constant amplitude of the cosine squared function, p0 , it is necessary to integrate the pressure function over the area the pressure is to be applied to; it is the component of the applied pressure function in the direction of the applied force that is to be integrated.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, CREATE PRESSURE FROM FORCE

- **Integrating the component of the pressure function, $p = p_0 \cos^2(\theta + \pi/2)$, in the direction of the applied load, the following is found:**

$$f = \int_0^{\pi} p_0 \cos^2(\theta + \pi/2)\sin(\theta) R t d\theta = \frac{4}{3} p_0 R t$$

where R is the radius of the bolt hole, and t is the thickness of the clevis.

- **Solve for p_0 in terms of the applied concentrated load**

$$p_0 = \frac{3f}{4Rt} = \frac{3f}{2Dt}$$

- **Applying this formula to one of the applied forces, the following is found:**

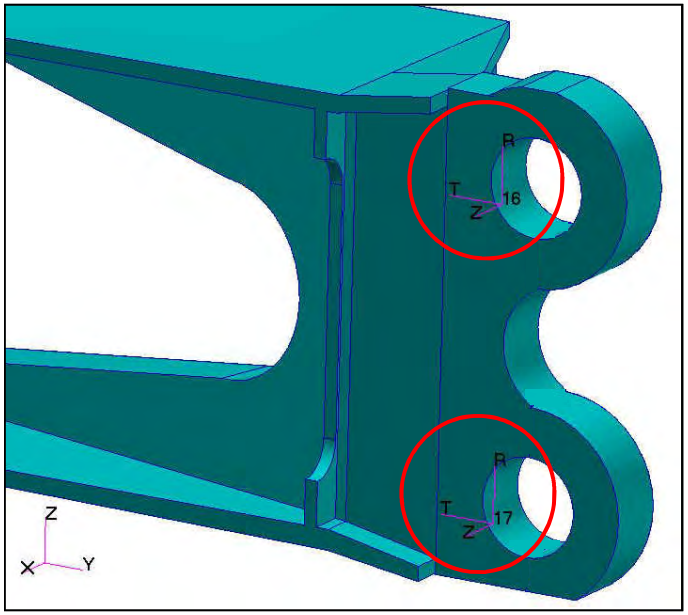
$$p_0 = \frac{3f}{2Dt} = \frac{3 * 25600}{2 * 1.501 * (2 * 0.980)} = 13052.5 \text{psi}$$

where D = 1.501 in and t = 0.980 in.; multiply t by 2 for symmetry.

Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, ESTABLISHING THE LOAD APPLICATION AREA

- **Create coordinate systems at the center of the bolt holes to do the following:**
  - Specify the direction of the loading.
  - Break the geometry to create solid faces the loading will be applied to.
  - Use the coordinate systems to provide variables for the creation of the fields for the loading.
- **On the symmetry model, create a cylindrical coordinate system at the center of each hole.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, ESTABLISHING THE LOAD APPLICATION AREA

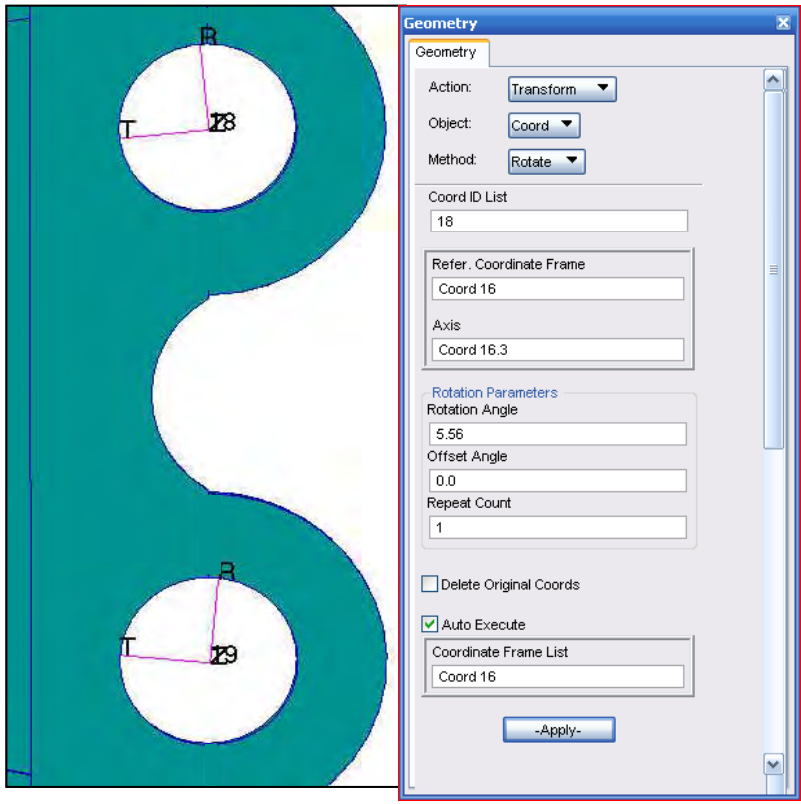
- **Calculate the angle to rotate the coordinate frames. This will be based on the components of the free-body force.**

θ_{top} = Tan-1(-2,480 lbf / -25,500 lbf)

= 5.56 + 180 degrees

θ_{bottom} = Tan-1(-2,480 lbf / 26,400 lbf) = -5.37 degrees

- **The previously created coordinate frame at the center of the holes is used to create the rotated frames.**
  - The original coordinate frames were deleted.

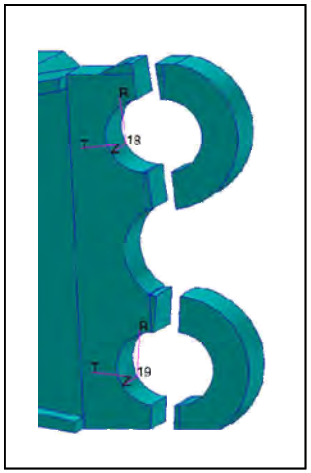


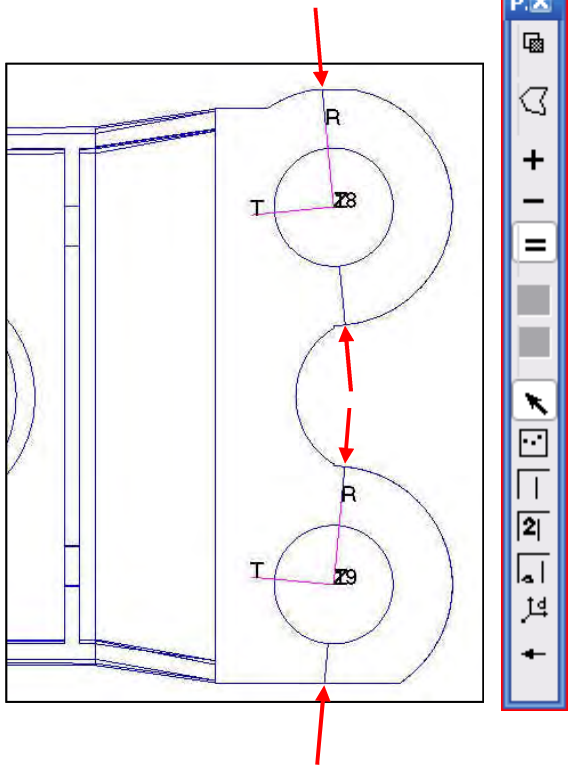
Copyright© 2012 MSC.Software Corporation

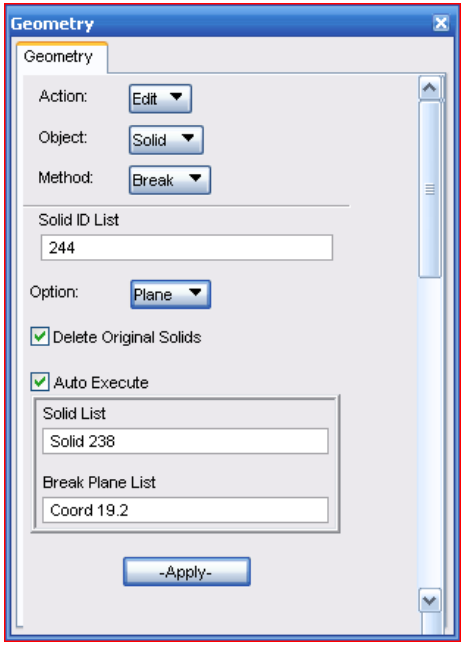
CASE STUDY 1, ESTABLISHING THE LOAD APPLICATION AREA

- **Break the solid using the rotated coordinate frames. This gives the needed solid faces for applying the pressure.**
  - In Geometry: Edit/Solid/Break, Option: Plane, use the second (theta) direction from the picking filter and the hole center coordinate frame to break the solid.

Shown only for clarity, the broken pieces of the solid are displayed after being translated away from the original solid.



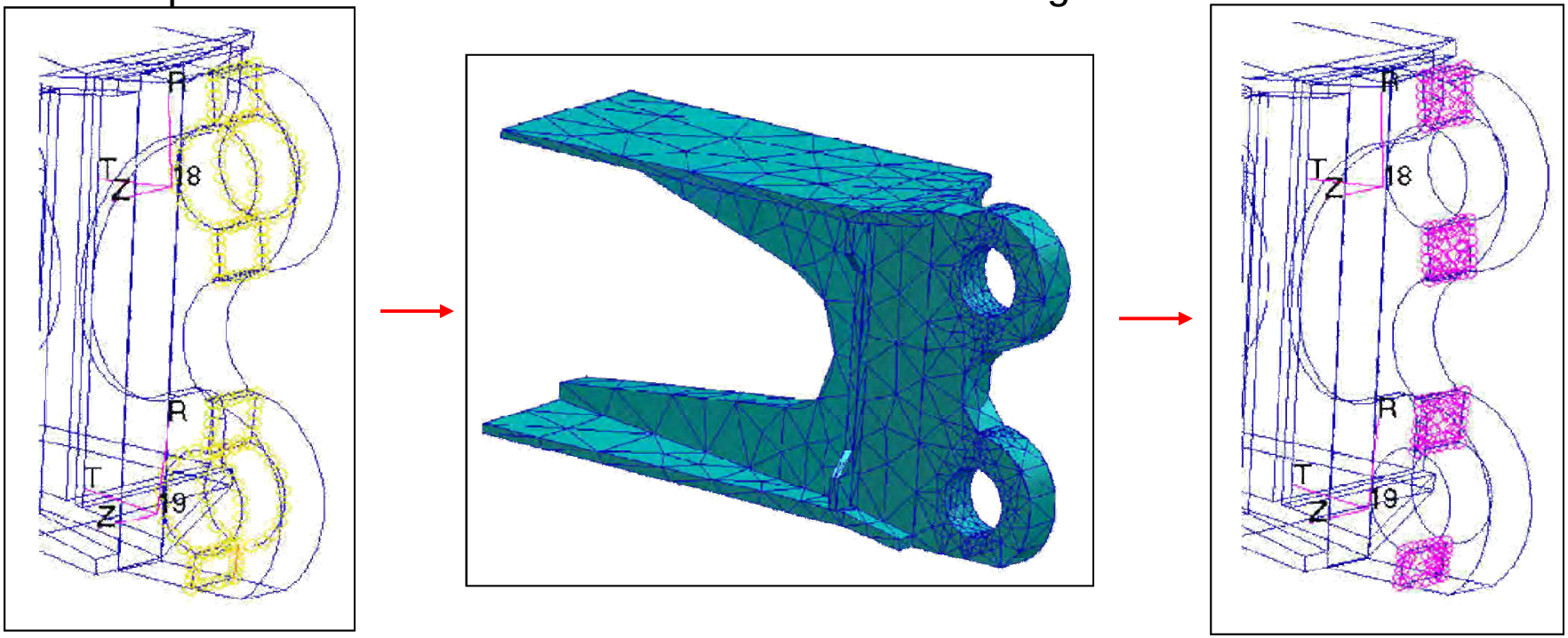




Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, MESH THE SOLIDS

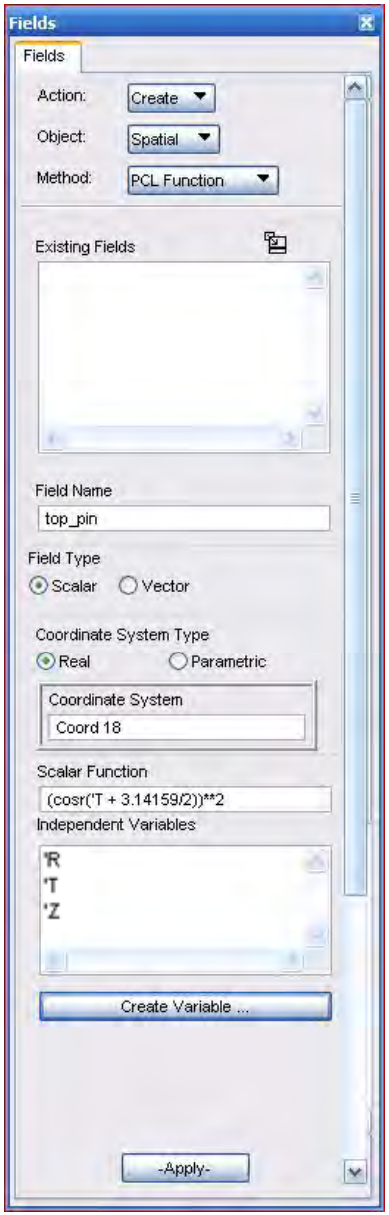
- **TetMesh the model**
  - Use mesh seeds for a finer mesh around the pin holes.
  - Mesh the 3 solids simultaneously.
  - Equivalence the model to connect the tets at the geometric interfaces.



Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, CREATE THE FIELD

- **A separate field and pressure load set needs to be created for each pin hole loading.**
- **Create the field for the top pin hole.**
  - As the field will be used for pressure loading, Field Type must be **Scalar**.
  - Coordinate System Type is set as **Real** and Coordinate System is the cylindrical coordinate system, created by rotating about the center of the hole, **Coord 18**.

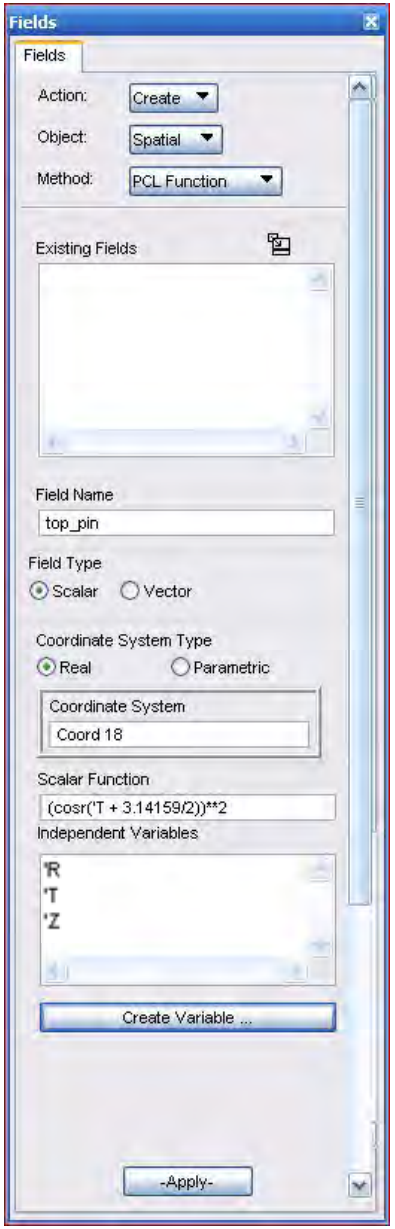


Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, CREATE THE FIELD

- **Scalar Function in the form has 4 key components:**
- **(cosr('T+3.14159/2))**2**

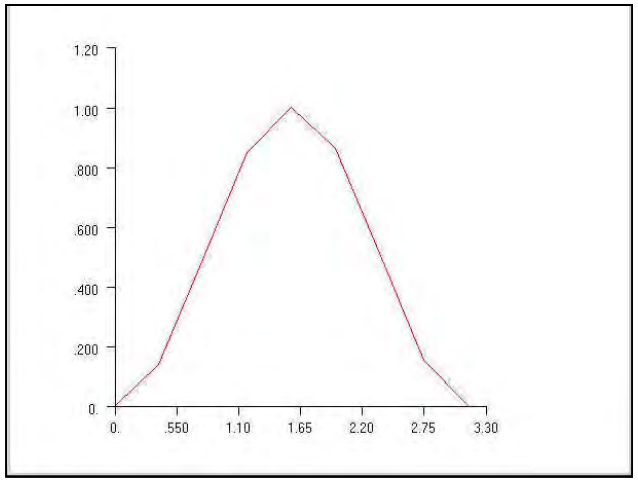
cosr	**This specifies that the argument of cosine is in radians.**
'T	**Taken from the Independent Variables list, this is the spatial variable based on the coordinate system selected. The coordinate systems created in Patran are in radians.**
3.14159/2	**This value is used to rotate the field a quarter turn or 90 degrees.**
****2**	**This is the exponential or squared term of $\cos^2$.**

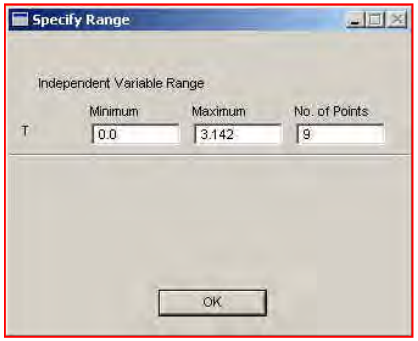


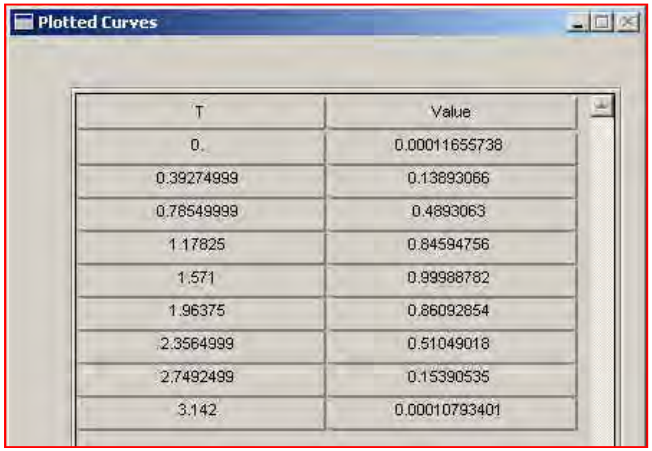
Copyright© 2012 MSC.Software Corporation

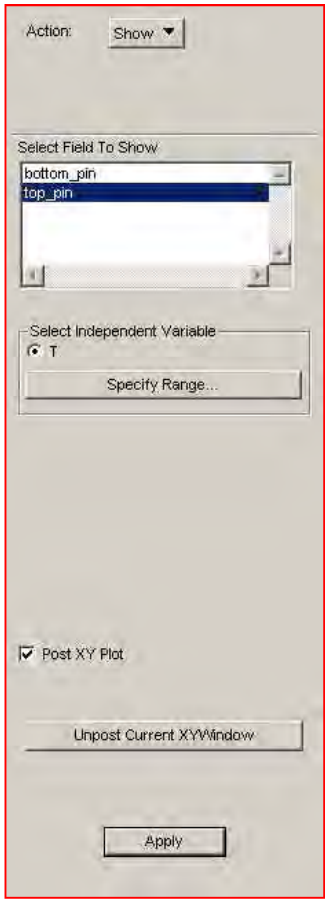
CASE STUDY 1, VERIFY CREATED FIELD

- **Verify the created field.**
- **In Fields: Show, Specify Range gives a range over which the field acts and the number of points for the display.**
  - In this case, only the loaded side of the hole is specified, 0 radians to 3.142 radians with 9 divisions.





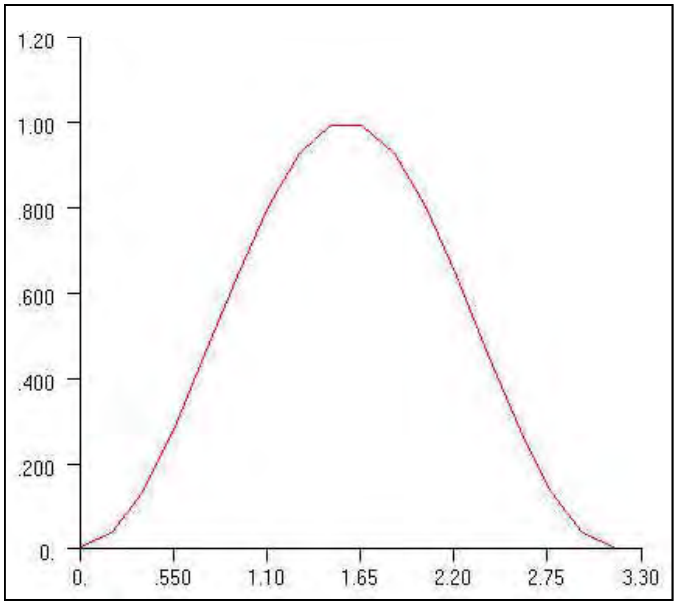


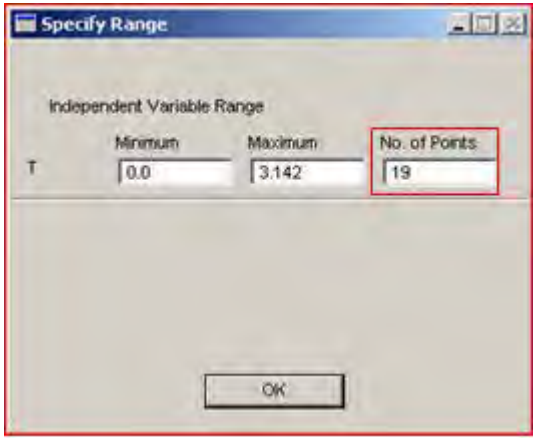


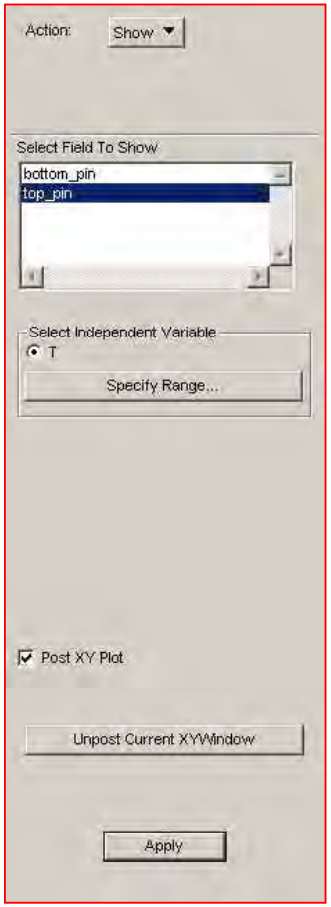
Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, VERIFY CREATED FIELD

- **If the plot or table data does not look correct, increase the number of points.**
  - Sometimes, confusion over degrees or radians can arise, that can be hidden by using too few points.
  - The Plot below shows a much smoother curve than before; the coarser curve could be representative of several superimposed functions.



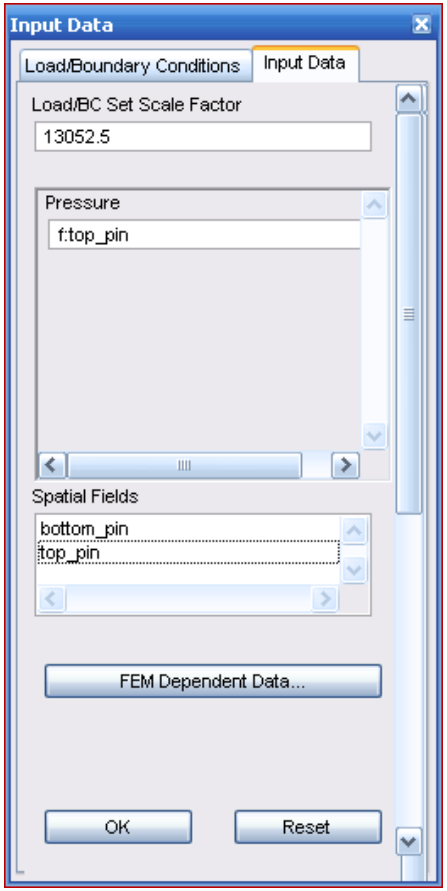


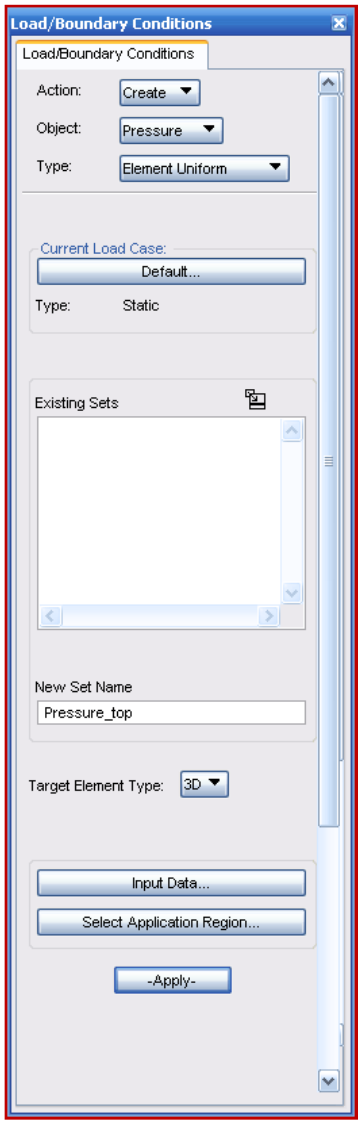


Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, CREATE THE PRESSURE LOAD

- **Create the load for the top pin.**
  - Target Element Type is 3D because solid finite elements are used.
  - In the Input Data form, the Load/BC Set Scale Factor is set to 13,052.5 psi.
  - The Pressure input is field top_pin.

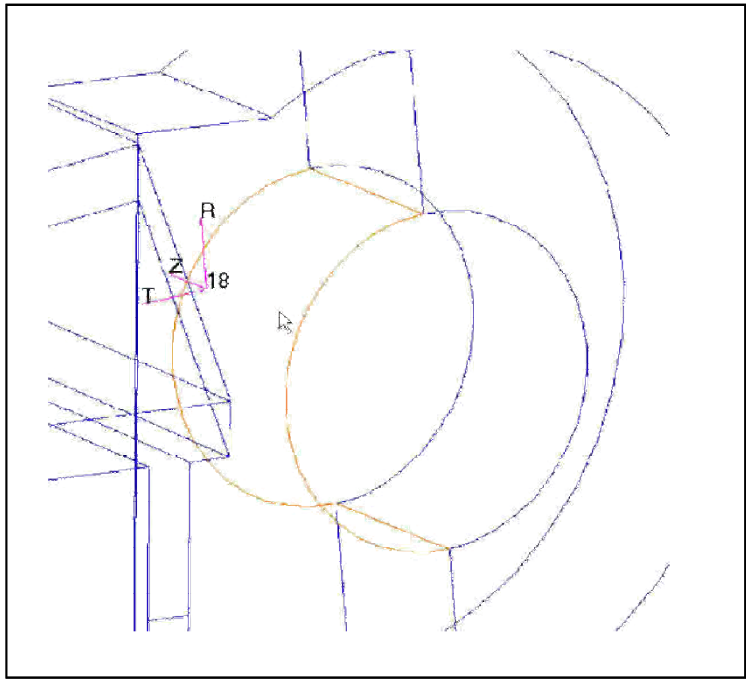


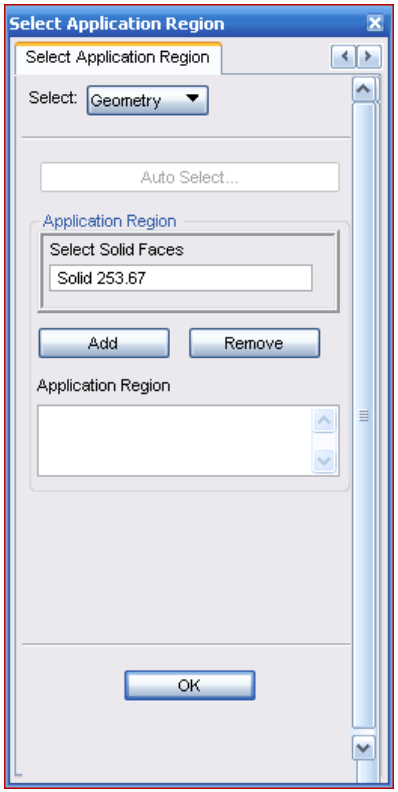


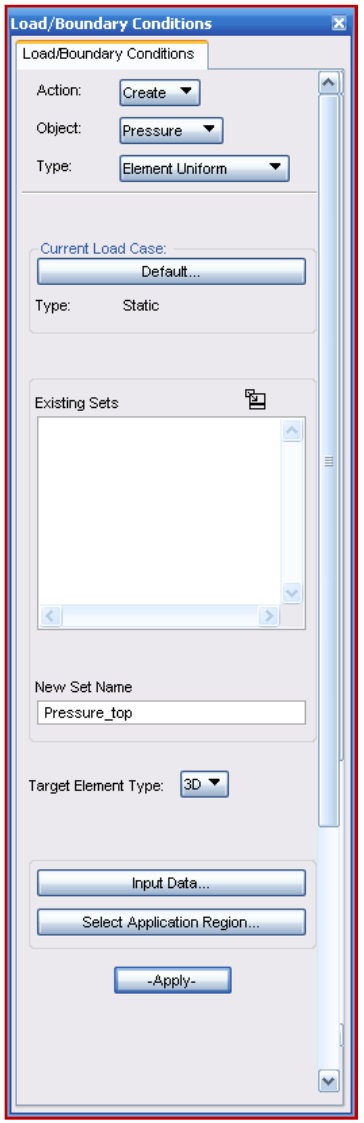
Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, CREATE THE PRESSURE LOAD

- **The application region form is set to Geometry and the solid face representing the pin contact area is selected.**

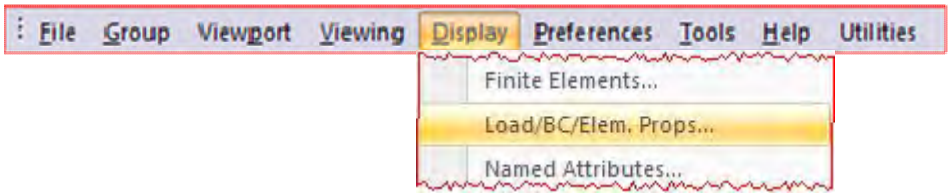




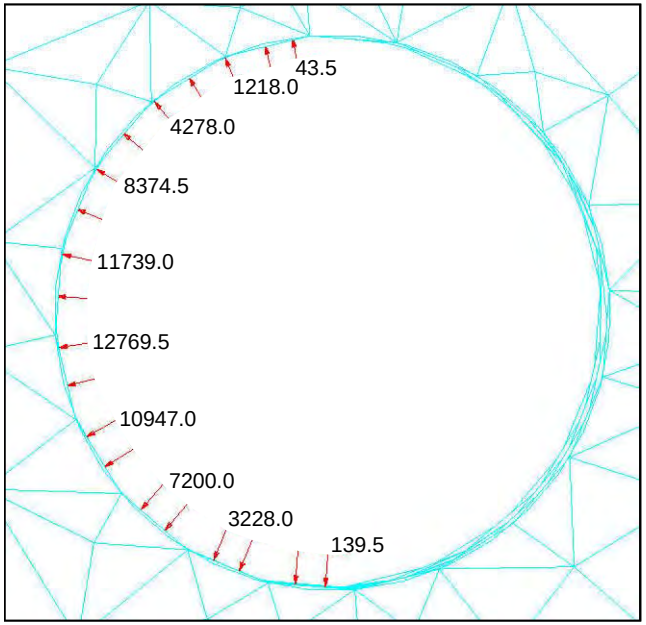


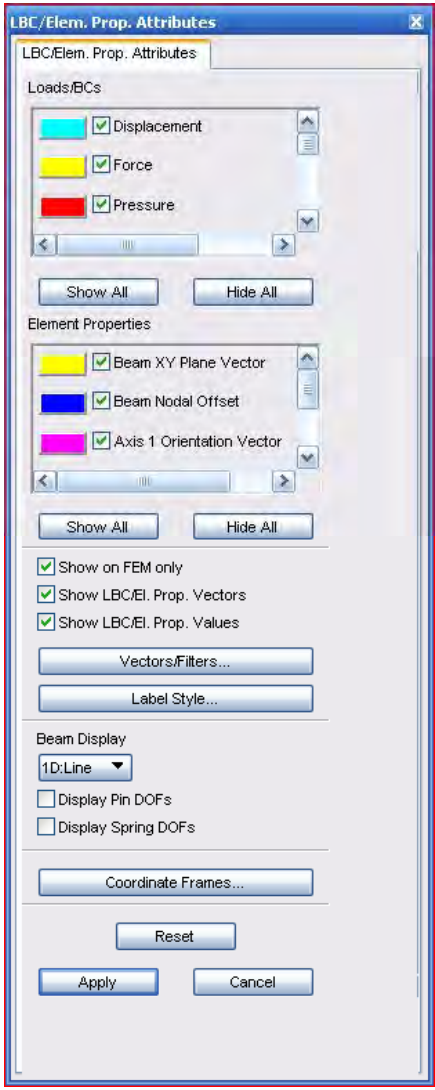
Copyright© 2012 MSC.Software Corporation

CASE STUDY 1, DISPLAYING THE PRESSURE LOAD



- **Display / Load/BC/Elem. Props… is used to view the pressures on the finite elements.**
  - For clarity, only some of the elements and their pressures are shown.
- **Notice that the pressures correctly approach zero at the ends, and are close to the value of 13,052.5 psi near the center.**

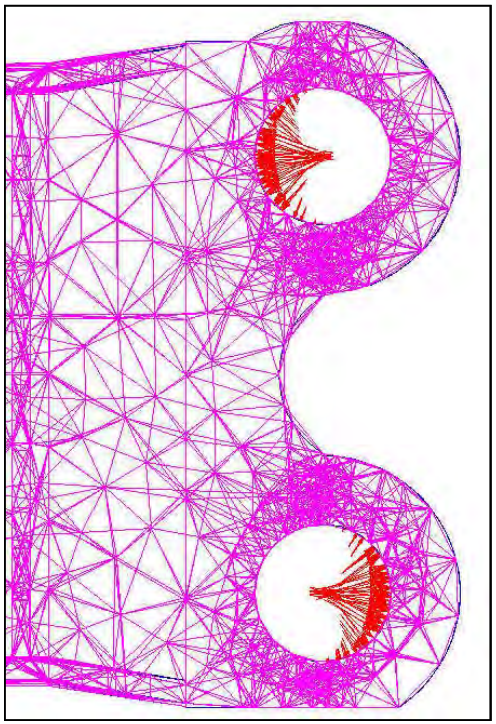


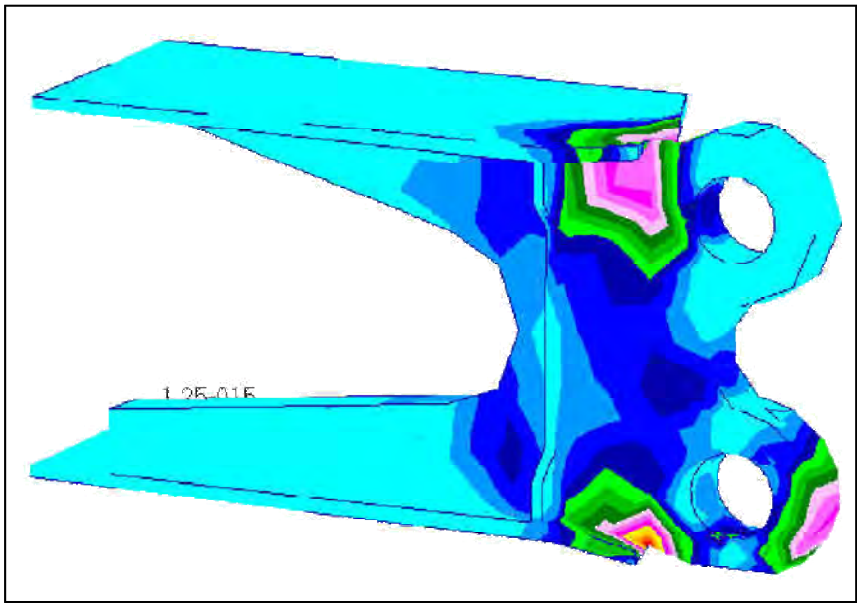


Copyright© 2012 MSC.Software Corporation

CASE STUDY 1: RUNNING THE ANALYSIS

- **The image of the finite element model (to the left) displays the pressure markers at the two holes.**
- **The fringe plot to the right shows stress tensor results:**



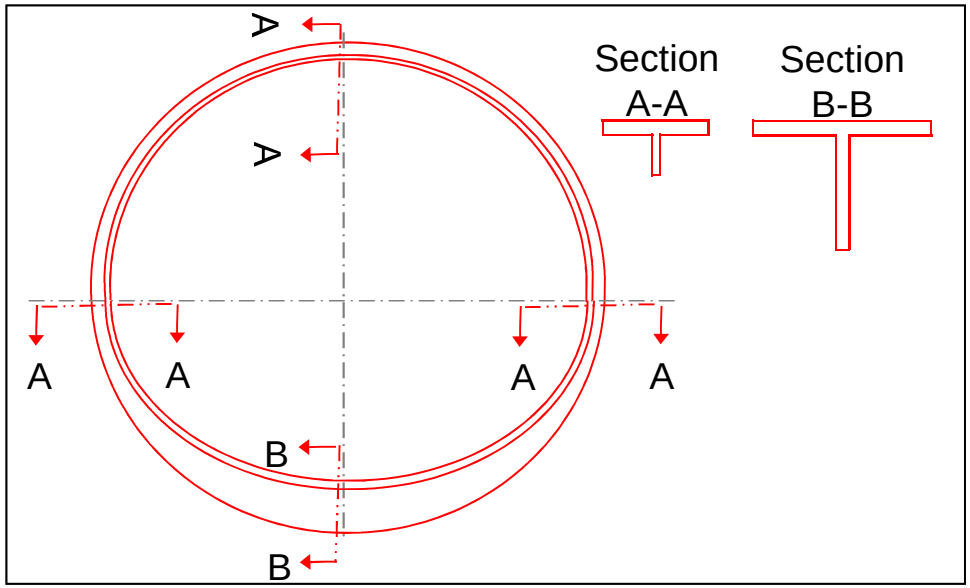


Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

CASE STUDY 2, SPATIAL/PCL FUCNTION FIELDS, PARAMETRIC

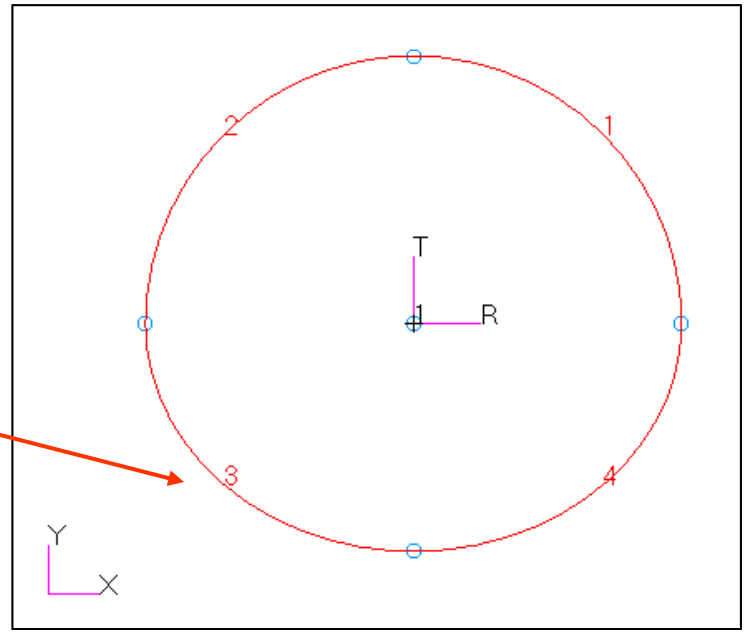
- **Model a submarine stiffening ring that has varying cross-sectional dimensions, using beam elements.**
- **The top of the ring has a constant cross-section (A-A), and the bottom cross-section varies from A-A to B-B to A-A.**
- **Use Fields: Create / Spatial / PCL Function**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 2, CREATE GEOMETRY

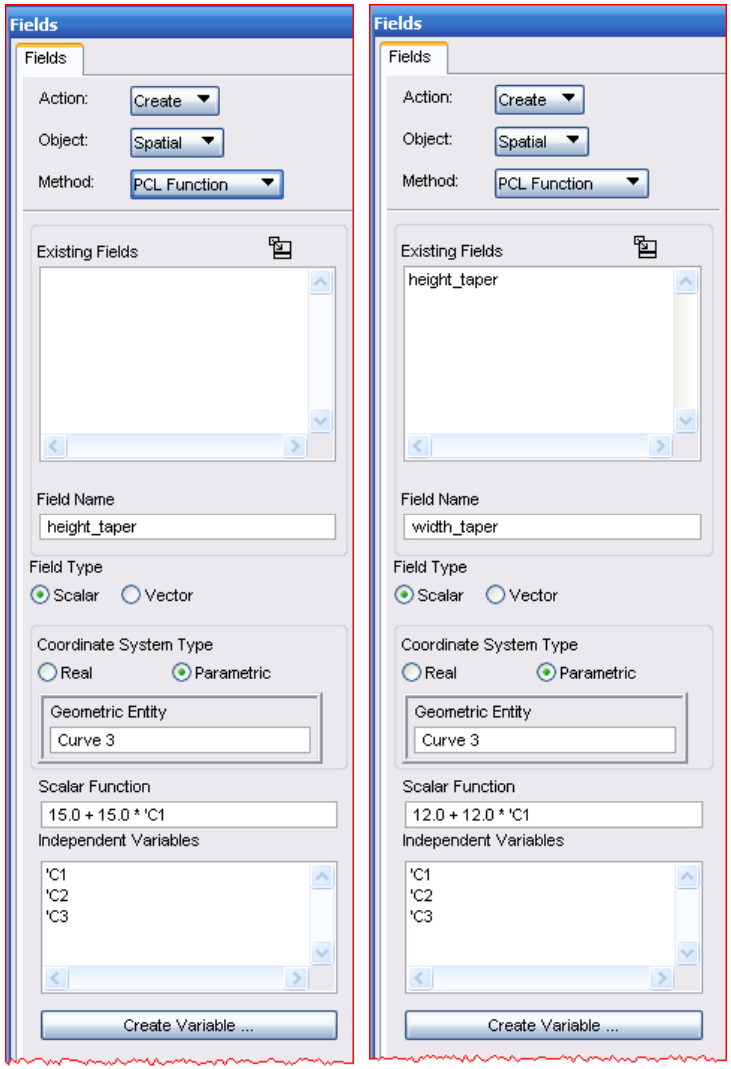
- First, create geometric curves that represent the ring. When meshed with beam elements, they have either constant or varying cross-sectional properties.
- The top curves (Curve 1, 2) are for elements with constant cross-sectional properties, and the bottom curves are for varying properties.
- For this case study, create the model just for Curve 3.



Copyright© 2012 MSC.Software Corporation

CASE STUDY 2, CREATE THE FIELD

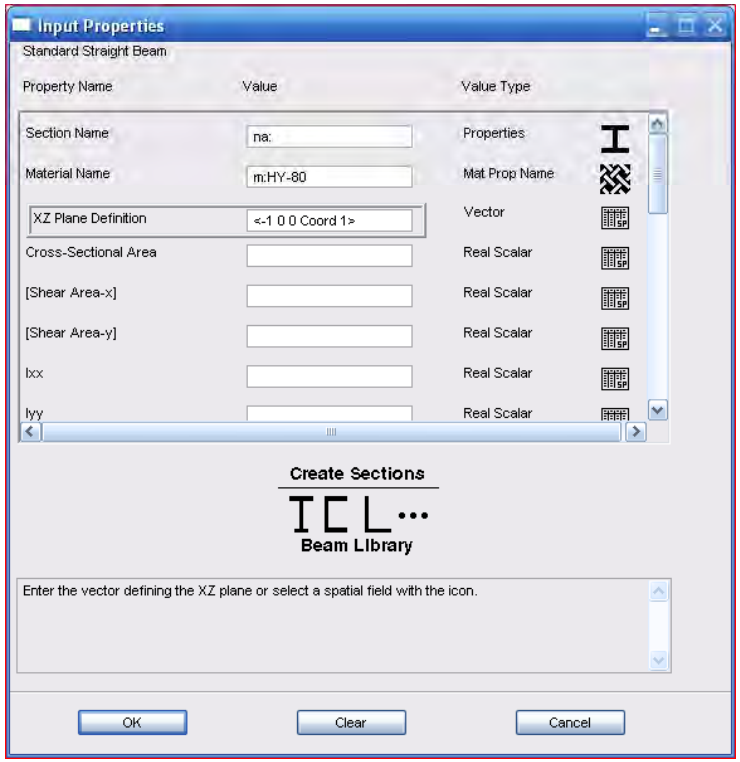
- It is necessary to create two spatially varying fields, one for the height of the cross-section, and the other for the width of the cross-section.
- Use Parametric for Coordinate System Type.

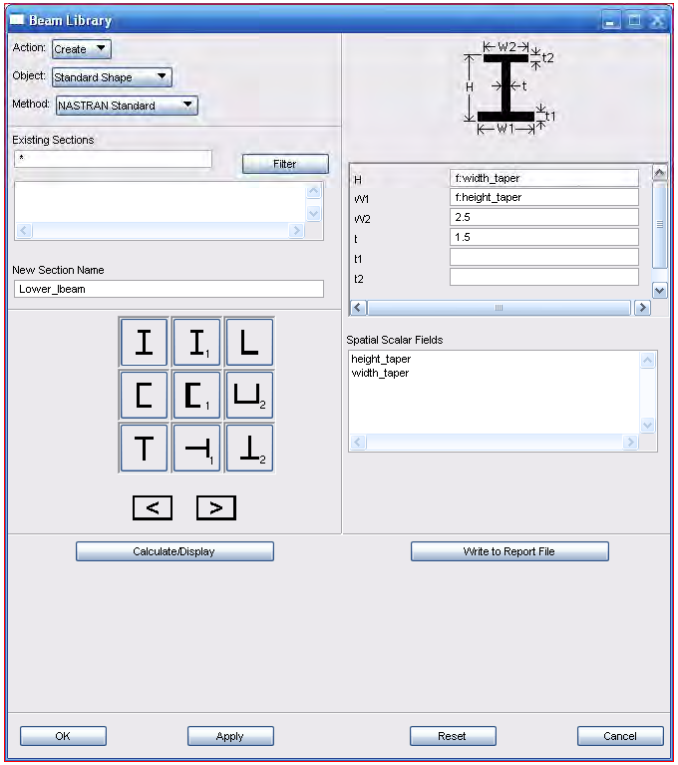


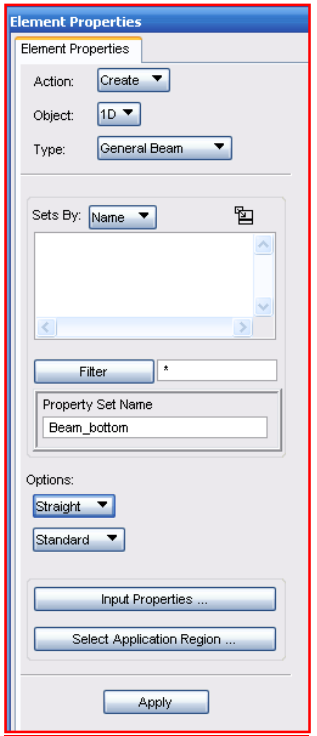
Copyright© 2012 MSC.Software Corporation

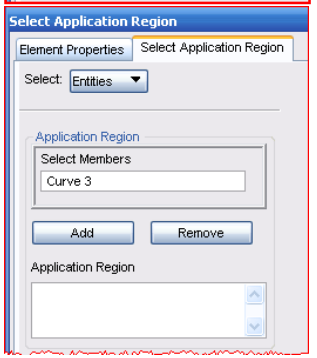
CASE STUDY 2, CREATE PROPERTIES USING THE FIELDS

- **In Properties, the dimensions of the “I” beam are specified using Beam Library.**
- **The appropriate fields (e.g. height_taper) are used from the Spatial Scalar Fields menu.**





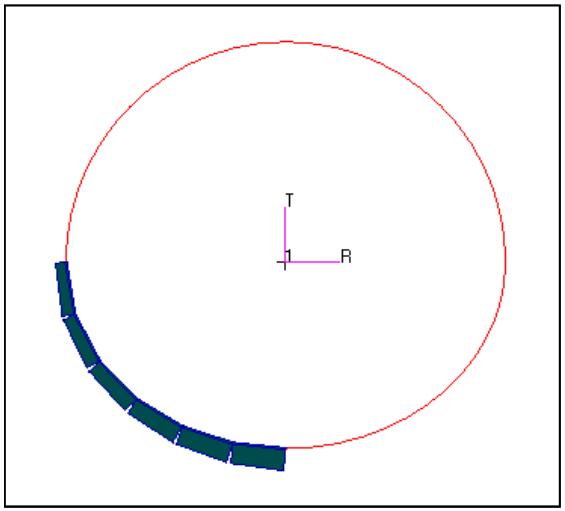


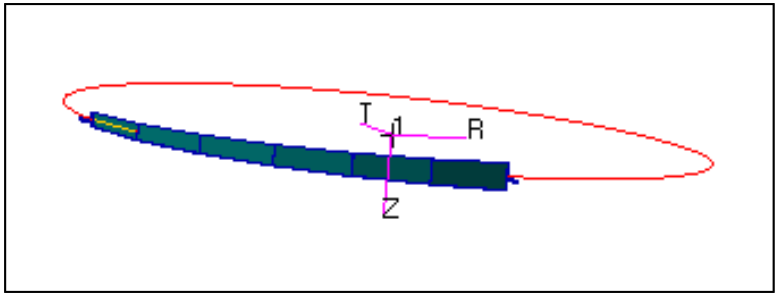


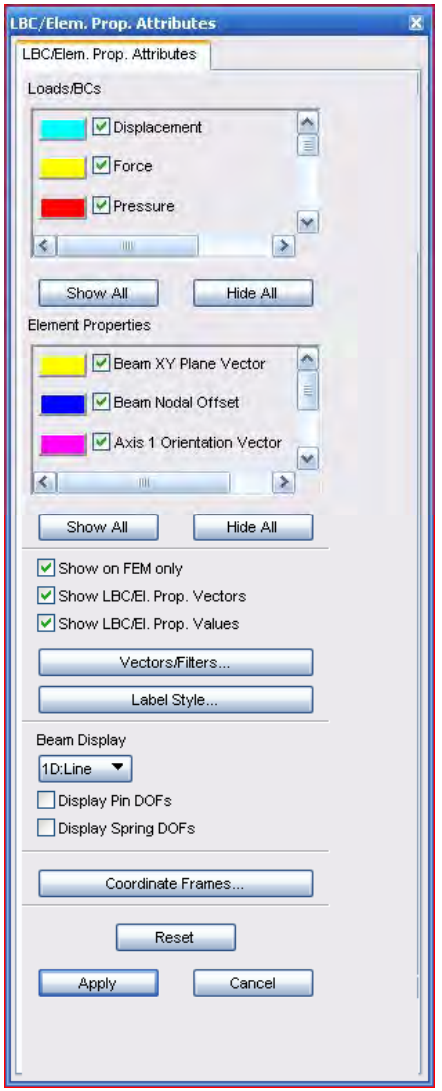
Copyright© 2012 MSC.Software Corporation

CASE STUDY 2, VERIFY THE BEAM CROSS-SECTION PROPERTIES

- **Use Display: Load/BC/Elem. Props, Beam Display to view the finished beam cross-sections in the viewport.**



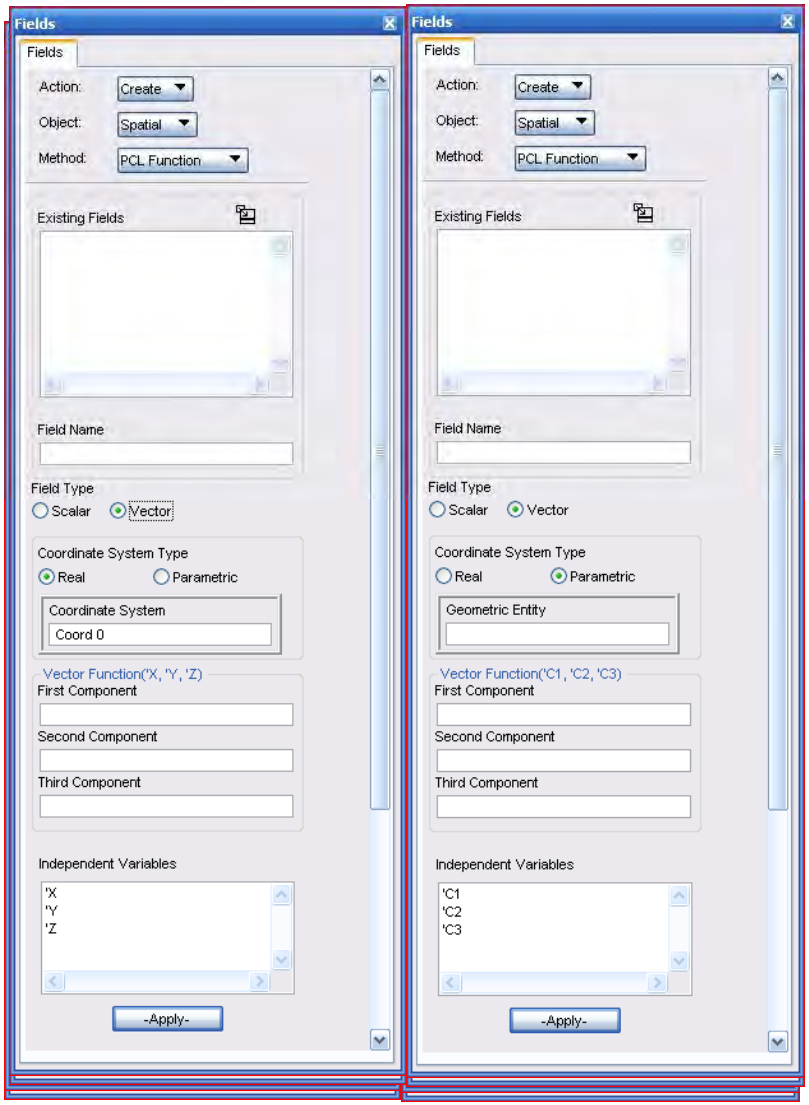




Copyright© 2012 MSC.Software Corporation

FIELDS
SPATIAL/PCL FUNCTION, VECTOR

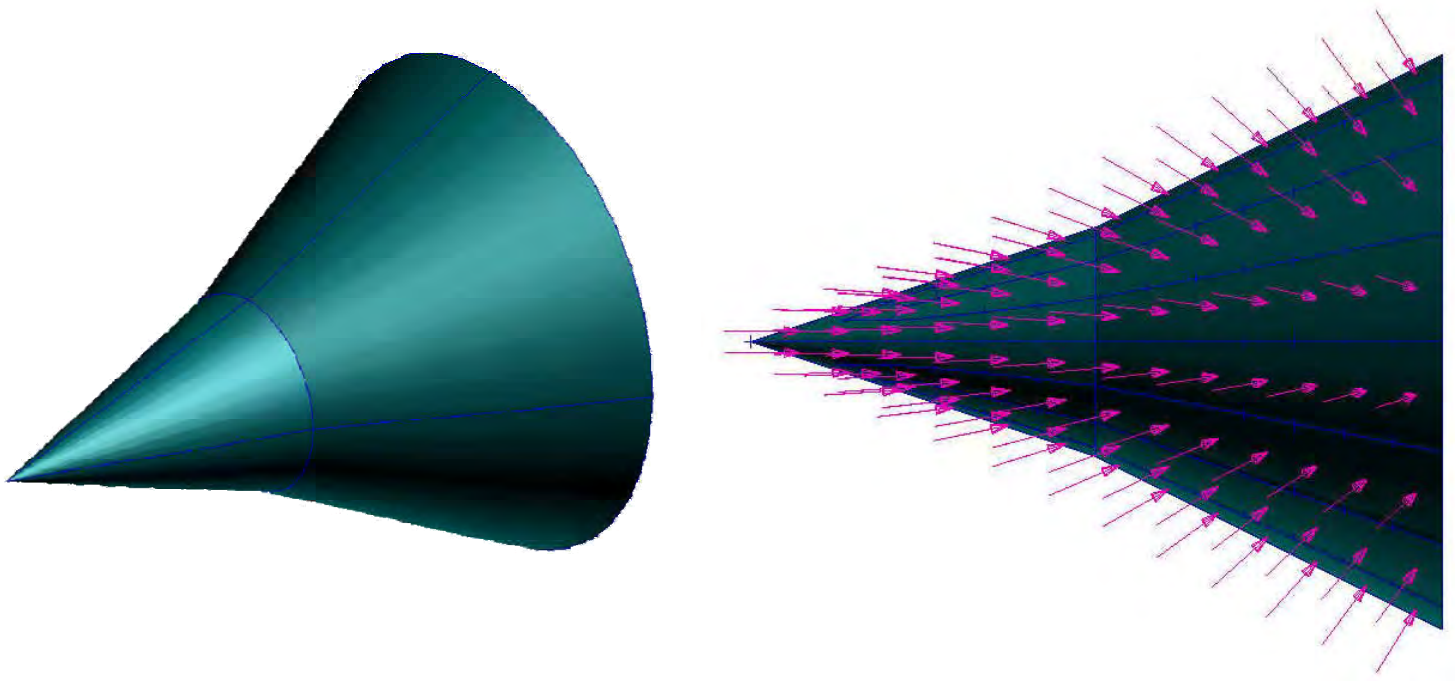
- **Spatial/PCL Function, Vector is similar to Scalar except that individual direction components can be specified:**
  - Using Real for Coordinate System Type will make the independent variables relative to the coordinate system chosen.
  - Using Parametric for Coordinate System Type will make the independent variables relative to the geometric entity parametric coordinate system.



Copyright© 2012 MSC.Software Corporation

CASE STUDY 3, VARYING TRACTION LOAD ON A SPIKE

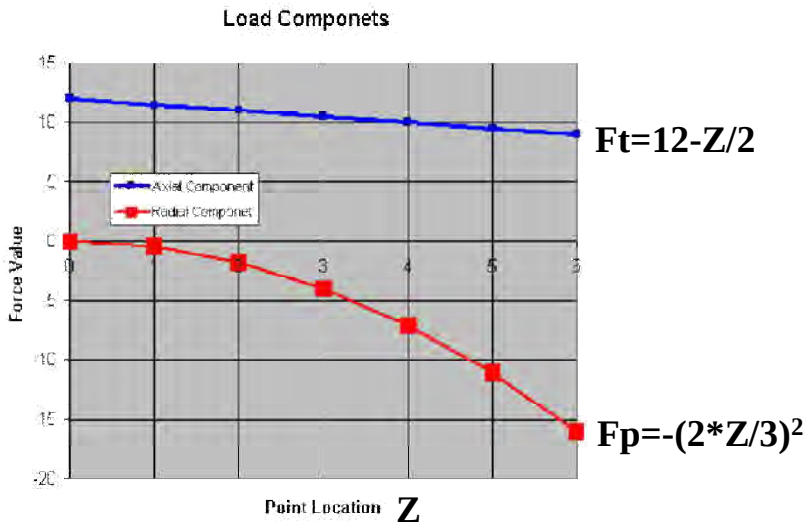
- **Use Spatial/PCL Function to make a vector field, representing a varying traction load and radial-pressure load on a spike.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 3, VARYING LOAD

- **The traction load from tip to base varies linearly from 12 to 9 over the 6 unit length of the spike.**
- **The radial-pressure load starts at 0 and decreases exponentially to -16 at the base.**

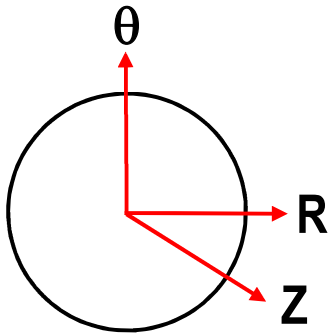


- **The chart and equations describe the individual components for the vectors.**
  - Fp is the radial-pressure and Ft is the traction load.

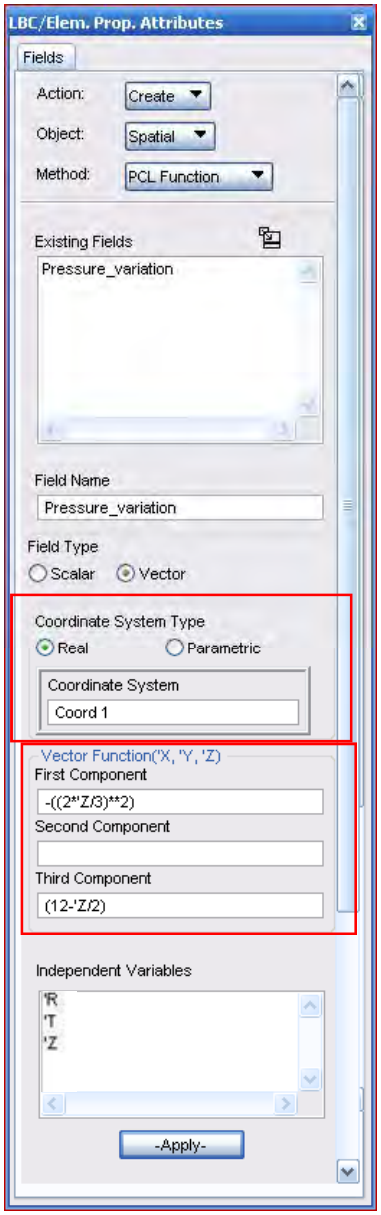
Copyright© 2012 MSC.Software Corporation

CASE STUDY 3, CREATE THE FIELD

- **Spatial/PCL Function, Vector references the cylindrical coordinate system at the tip of the spike, Coord 1.**
  - With reference to a cylindrical coordinate system, the first component will be applied only in the radial direction, R, and the third component will be applied only in the axial direction, Z.



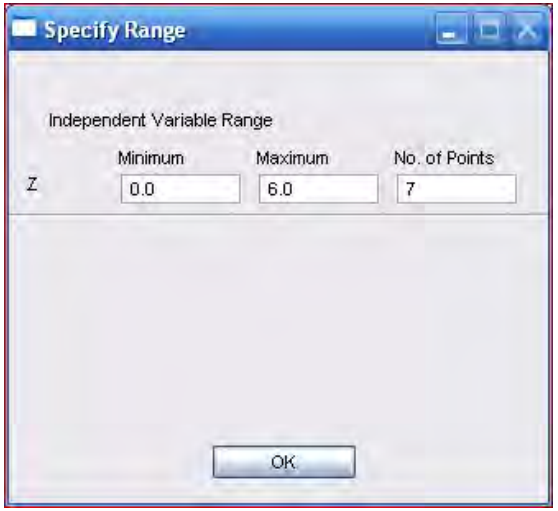
- **In both equations, the distance along the spike from the tip, Z, is the independent variable.**

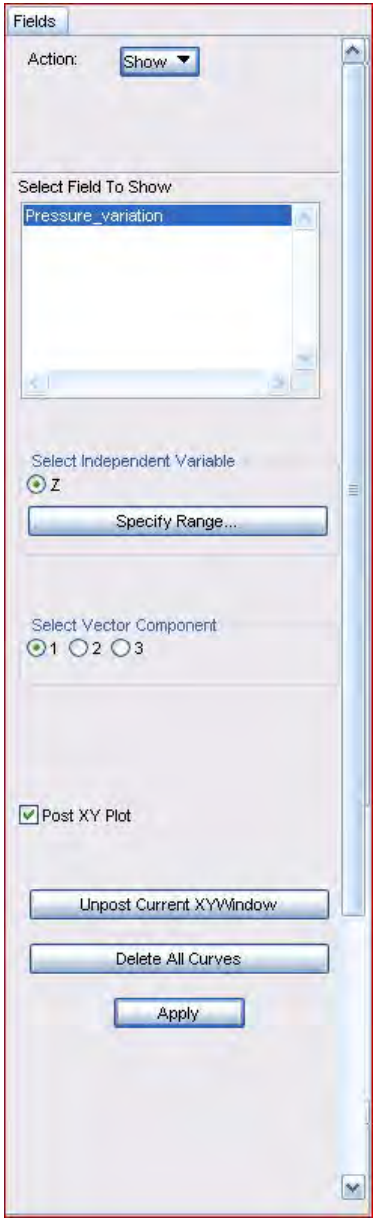


Copyright© 2012 MSC.Software Corporation

CASE STUDY 3, CHECK THE FIELD

- **Use Fields/Show to check the equation.**
  - In the Select Independent Variable, only the Z variable is shown, as it is the only variable used in the equations.
  - Specify the range of the equations as the model length. The range is from 0 to 6 units with 7 divisions.

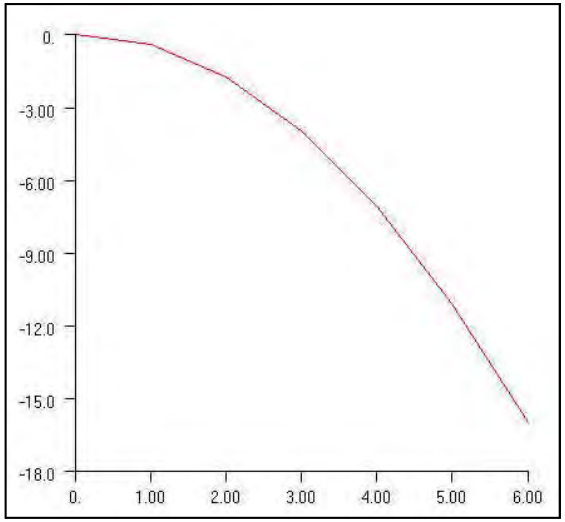




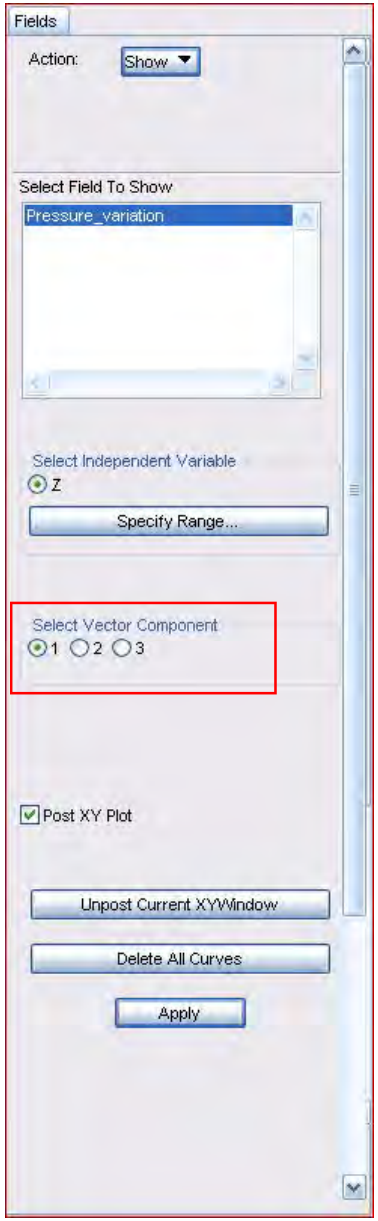
Copyright© 2012 MSC.Software Corporation

CASE STUDY 3, CHECK THE FIELD

- **Only one component of the vector function can be plotted and tabulated at a time.**
- **The first direction, radial, component is shown.**
  - Note the negative values and exponential trend.



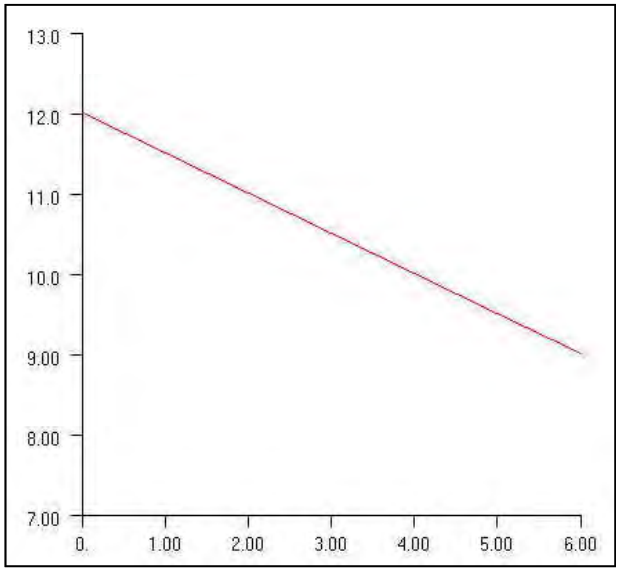
Z	Value
0.	0.
1.	-0.44444448
2.	-1.7777779
3.	-4.
4.	-7.1111116
5.	-11.111111
6.	-16.



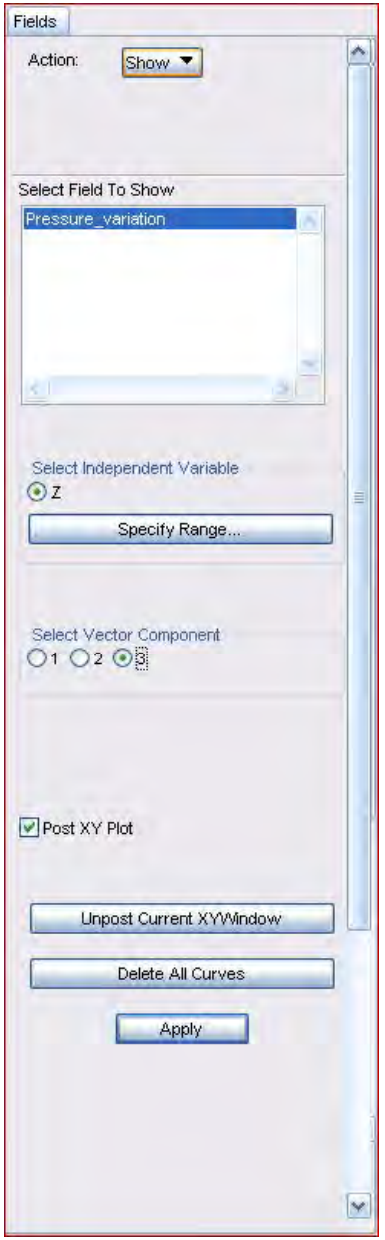
Copyright© 2012 MSC.Software Corporation

CASE STUDY 3, CHECK THE FIELD

- **The third direction, axial, component is now plotted.**
  - Note the positive values and negative linear trend.



Z	Value
0.	12.
1.	11.5
2.	11.
3.	10.5
4.	10.
5.	9.5
6.	9.

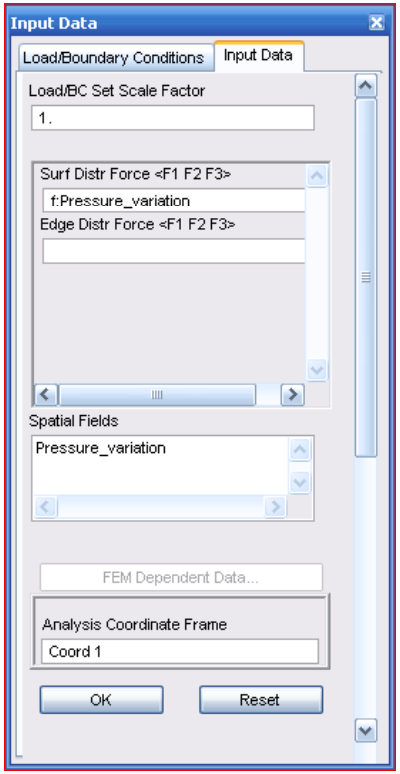


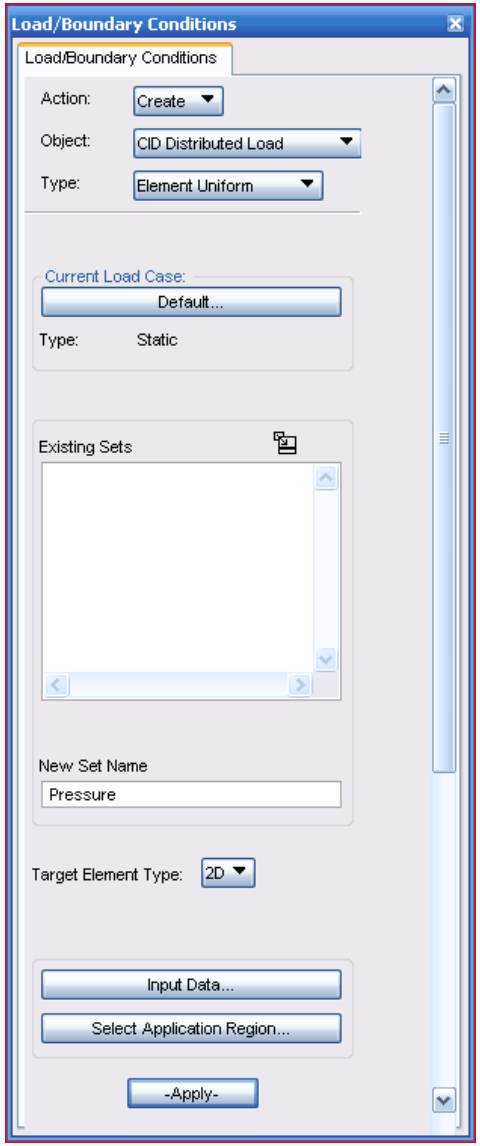
Copyright© 2012 MSC.Software Corporation

CASE STUDY 3, CHECK THE FIELD

- **To load the spike, use CID Distributed Load.**
  - The load is applied to shell elements so the **Target Element Type** is 2D.

- **From the Input Data form, select the spatial field previously created.**
  - No scale factor is needed, so the **Load/BC Set Scale Factor** is left as the default value 1.
- **For the analysis coordinate frame, select the cylindrical coordinate system at the tip of the model.**

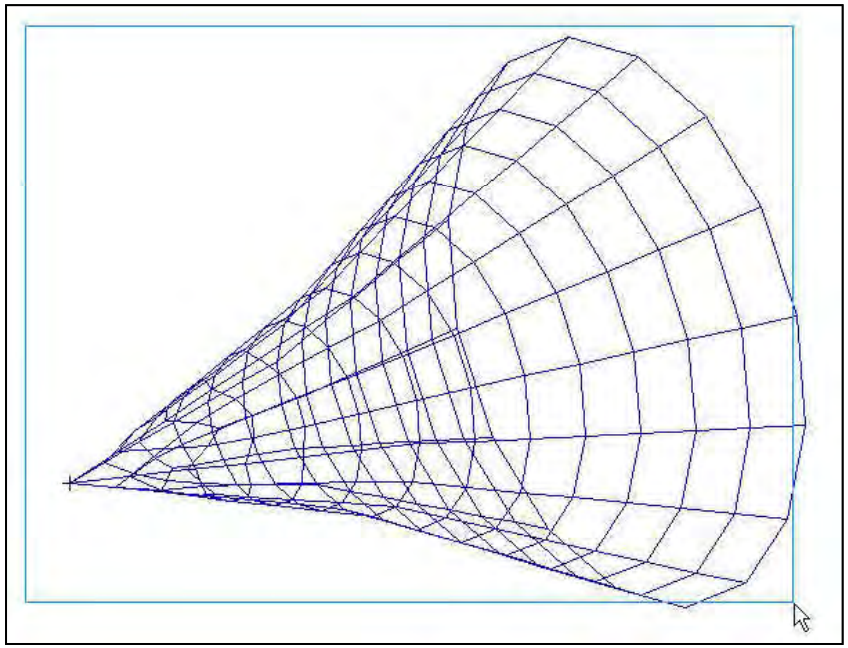


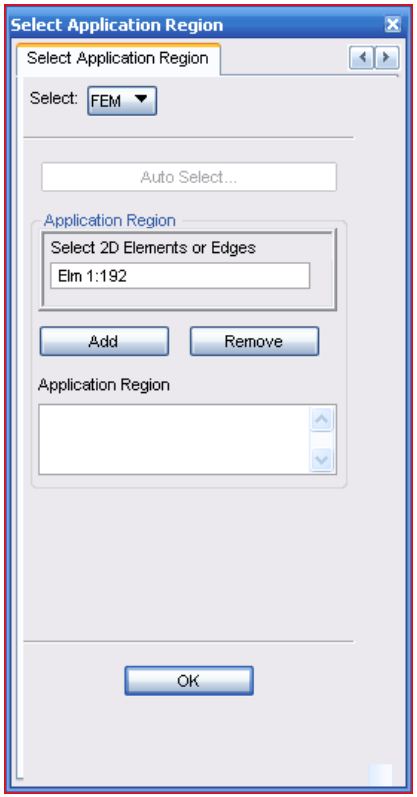


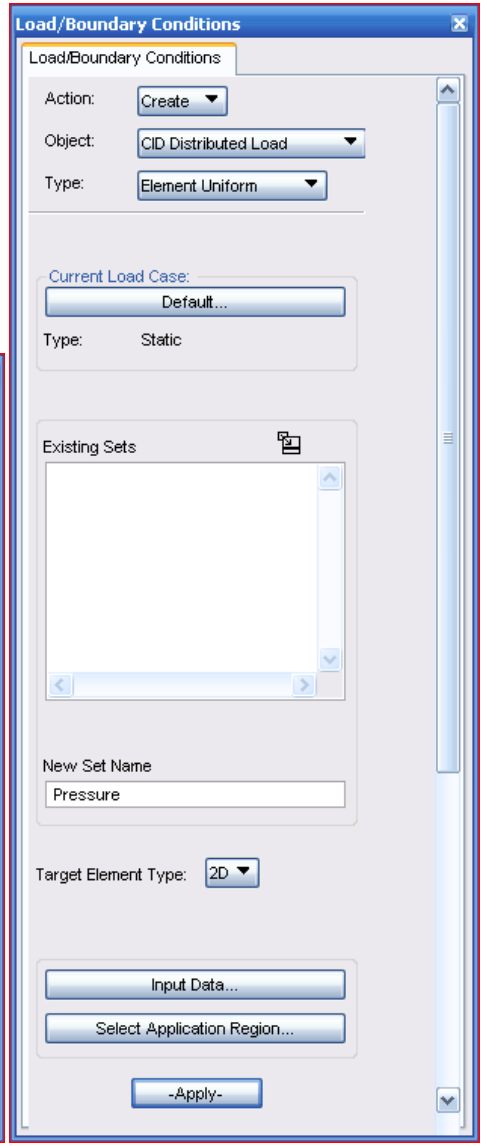
Copyright© 2012 MSC.Software Corporation

CASE STUDY 3, CHECK THE FIELD

- **For the Select Application Region, select all the shell elements used to model the spike**



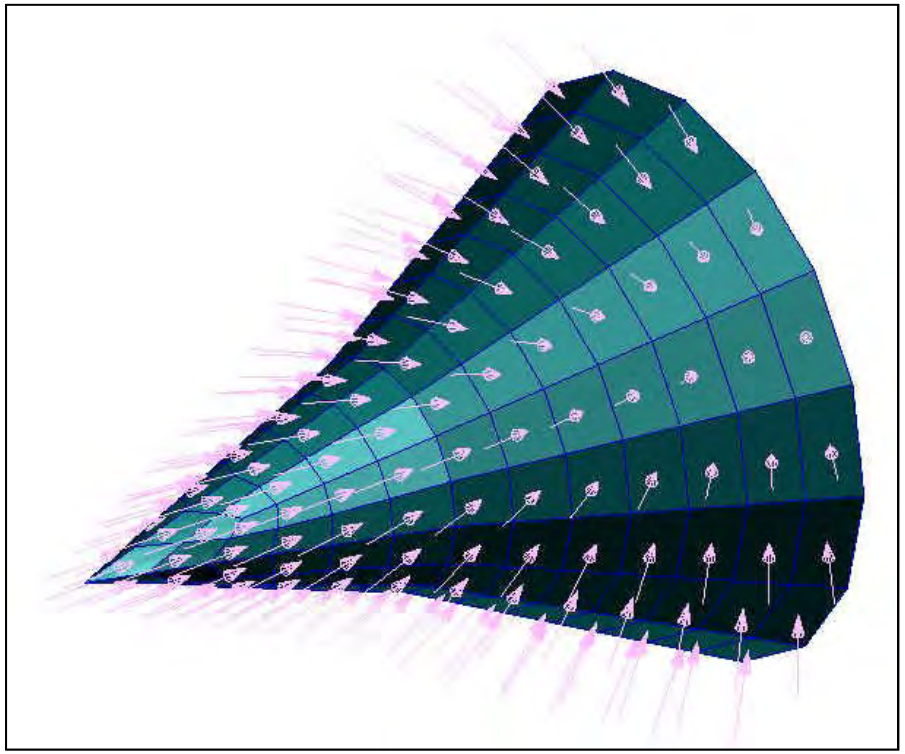


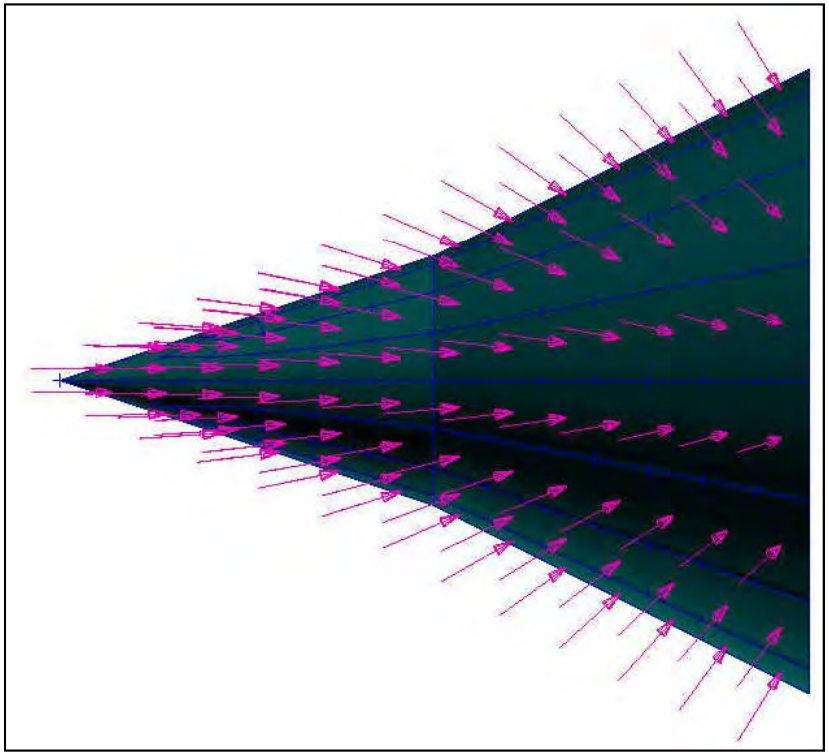


Copyright© 2012 MSC.Software Corporation

CASE STUDY 3, CHECK THE FIELD

- **Display of the varying load on the elements:**



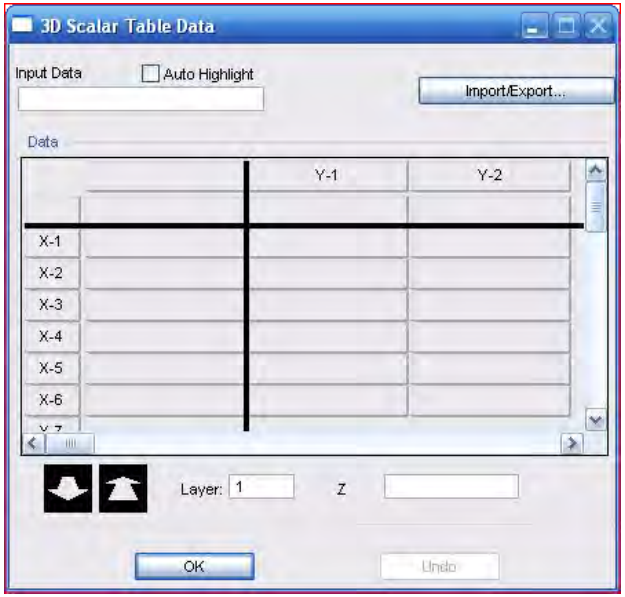


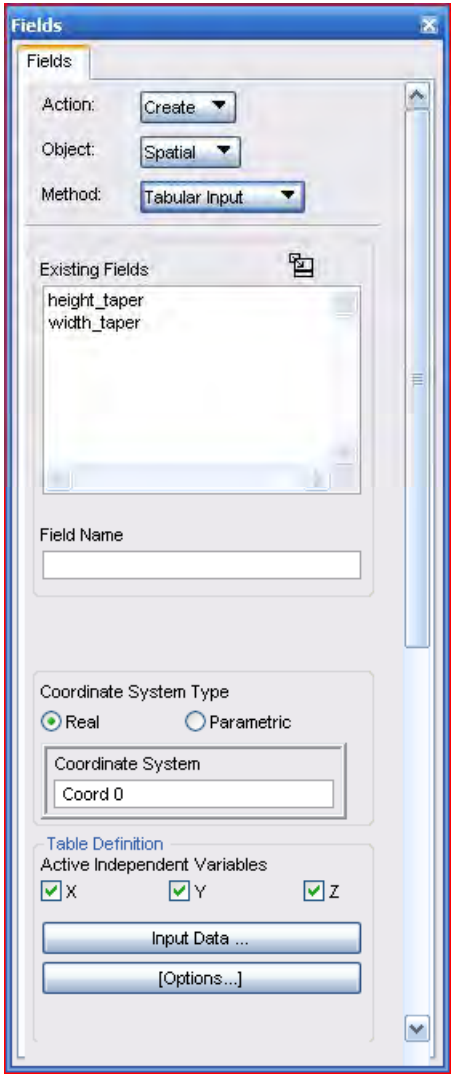
Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

FIELDS, SPATIAL/TABULAR INPUT, REAL

- **Spatial/Tabular Input, Real**
  - Real for Coordinate System Type uses the specified coordinate system.
  - The Independent Variables chosen will affect the layout of table input form.
  - If all 3 directions are chosen, the Z direction will be specified using table layers.
  - Import/Export

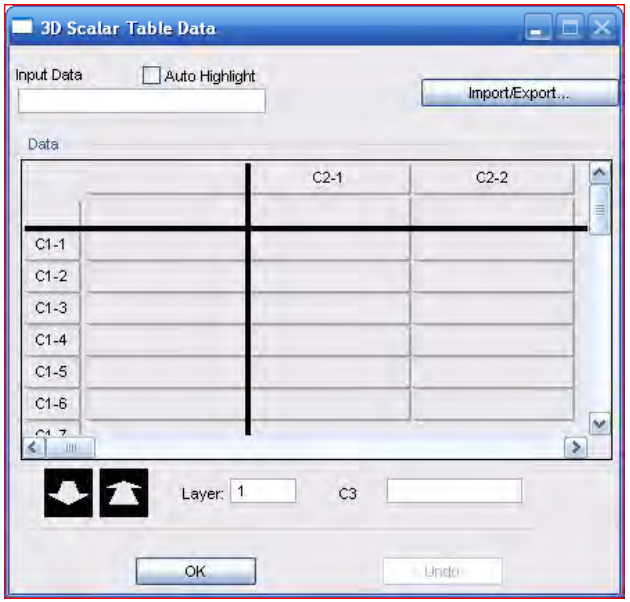


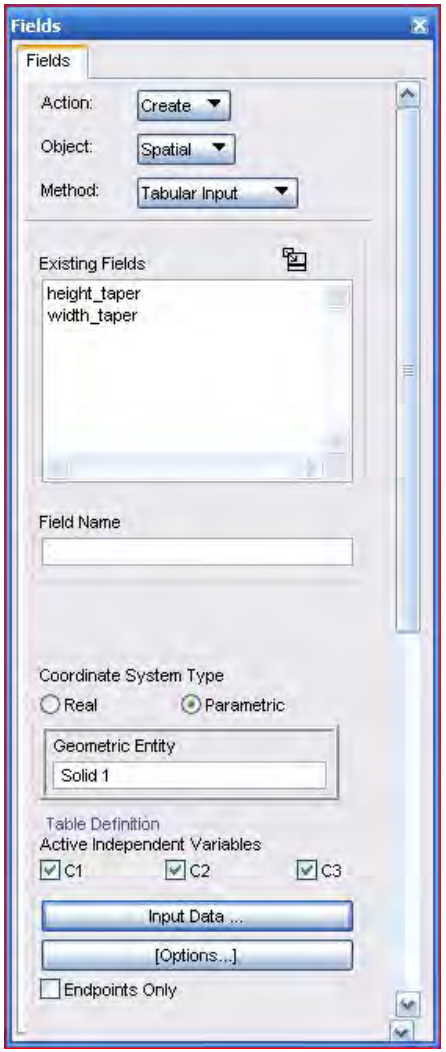


Copyright© 2012 MSC.Software Corporation

FIELDS, SPATIAL/TABULAR INPUT, PARAMETRIC

- **Spatial/Tabular Input, Parametric, Endpoints Only: no**
  - The Parametric selection will make the tabular data relative to the chosen Geometric Entity parametric coordinate system.
  - Input Data will be controlled in the same fashion as the Coordinate System Type set to Real.
  - Import/Export

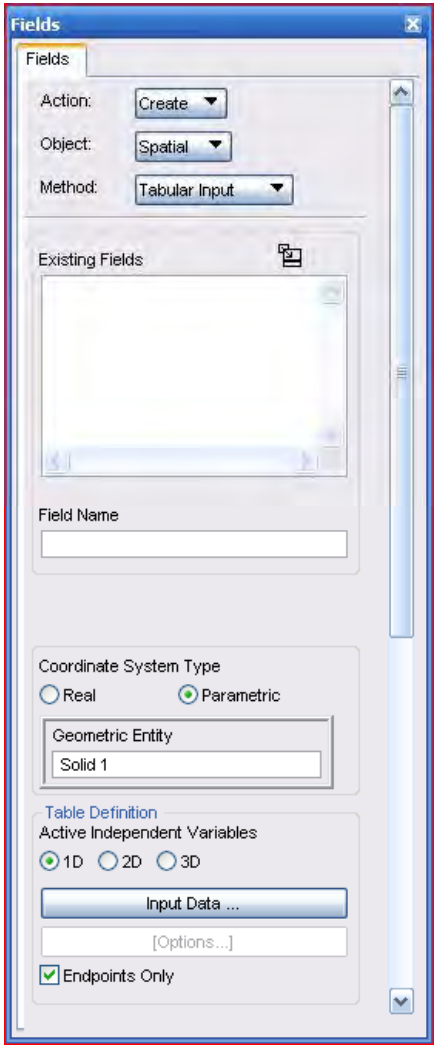




Copyright© 2012 MSC.Software Corporation

FIELDS, SPATIAL/TABULAR INPUT, PARAMETRIC

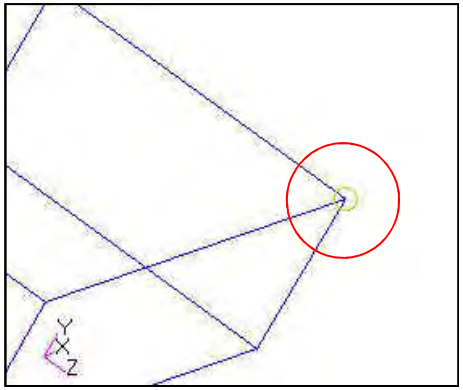
- **Spatial/Tabular Input, Parametric, Endpoints Only: yes**
  - Enabling Endpoints Only (yes) limits the input data to only the corners of the Geometric Entity selected.
  - Linear interpolation across the parametric space will occur between the corners.

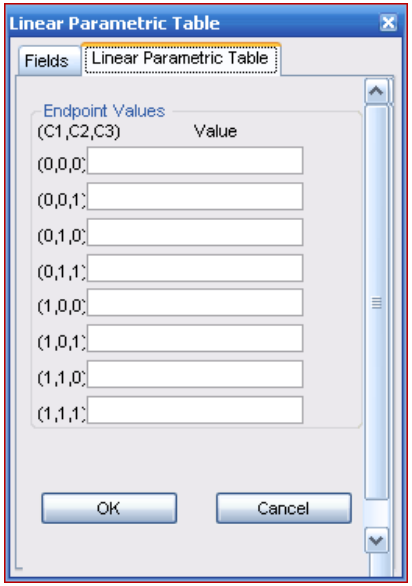


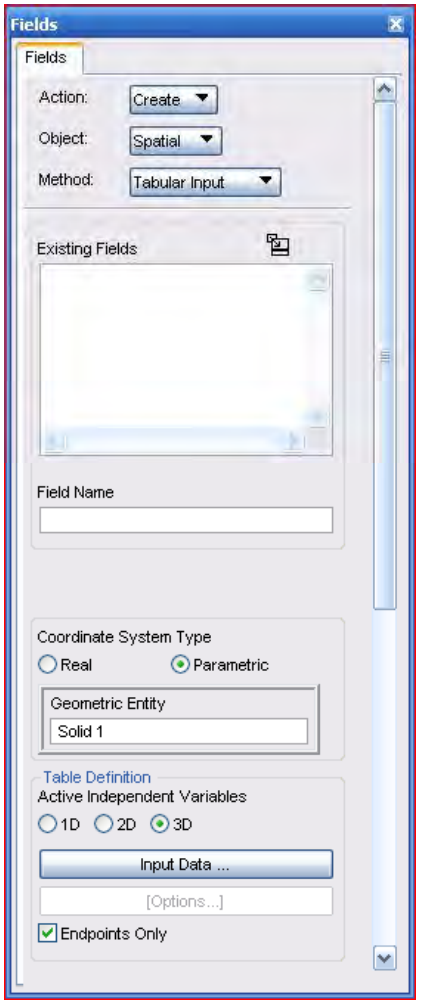
Copyright© 2012 MSC.Software Corporation

FIELDS, SPATIAL/TABULAR INPUT, PARAMETRIC

- **Spatial/Tabular Input, Parametric, Endpoints Only: yes**
- **The selected Active Independent Variables will determine what data can be input.**
  - The Input Data form will reflect the variables selected.
  - Input Data allows the input of data at the specified corners of the Geometric Entity selected.



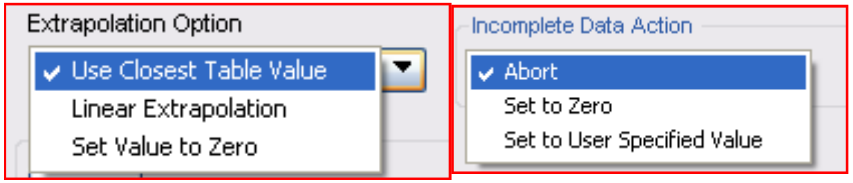


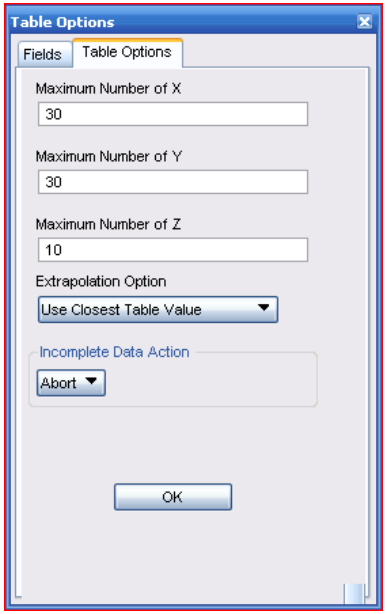


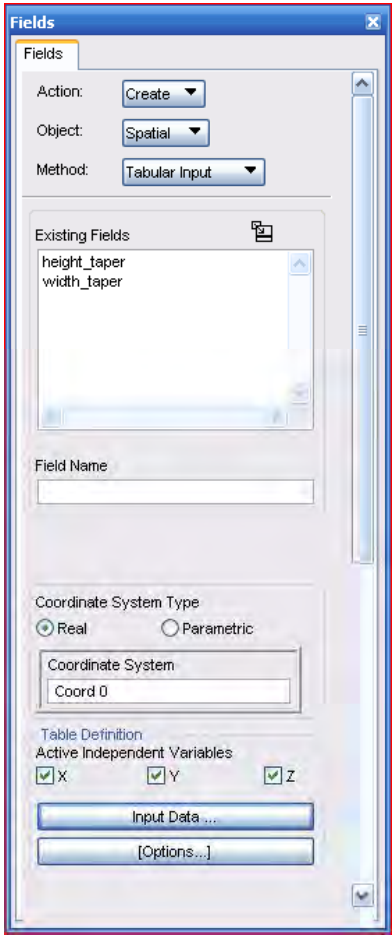
Copyright© 2012 MSC.Software Corporation

FIELDS, SPATIAL/TABULAR INPUT, OPTIONS

- **Spatial/Tabular Input, [Options]**
  - The top portion of the Tabular Input, [Options] controls how many data points for each direction will be available in the tabular input.
  - Extrapolation Option allows the selection of the following:
    - Use Closest Table Value
    - Linear Extrapolation
    - Set Value to Zero
  - Incomplete Data Action allows the selection how not having adequate data will be dealt with





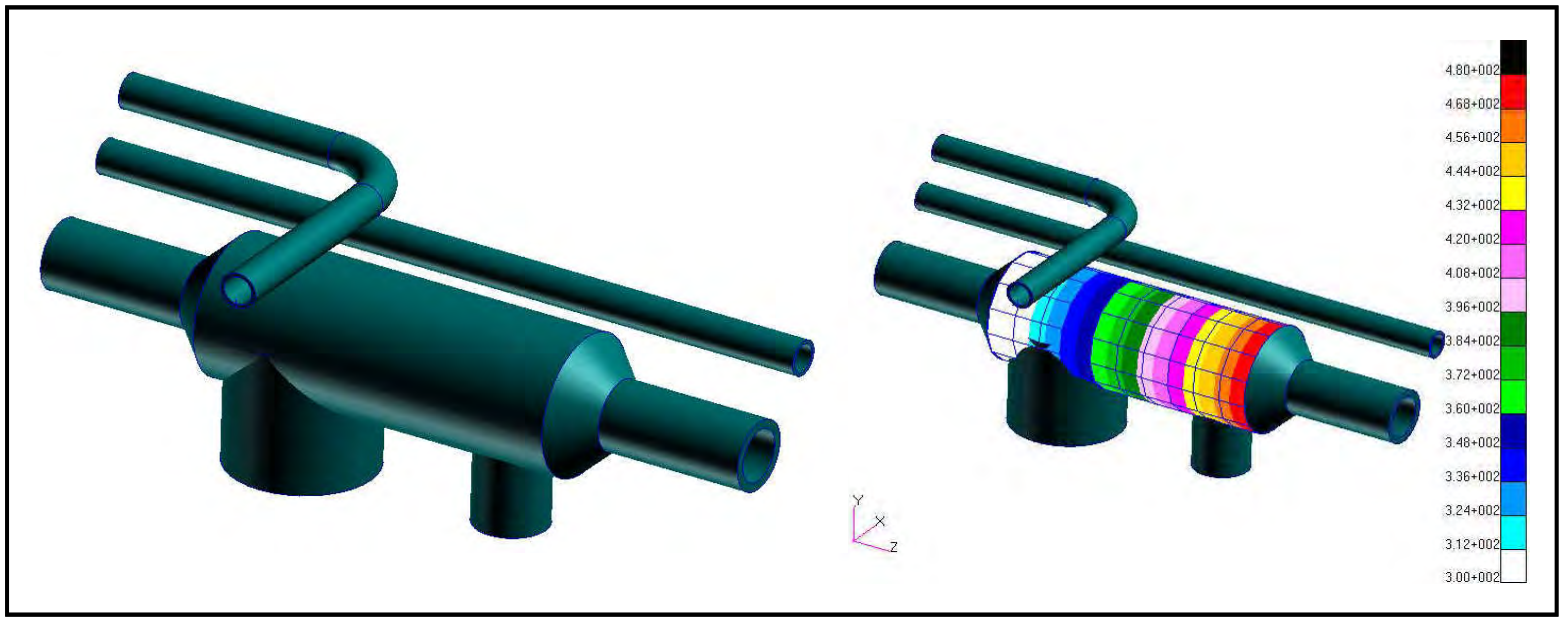


Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, RADIATION TO TUBES

- **A high temperature heat exchanger is radiating to thin tubes that are close to it. From the given data, make a Tabular Field for estimating the temperature distribution of the tubes.**

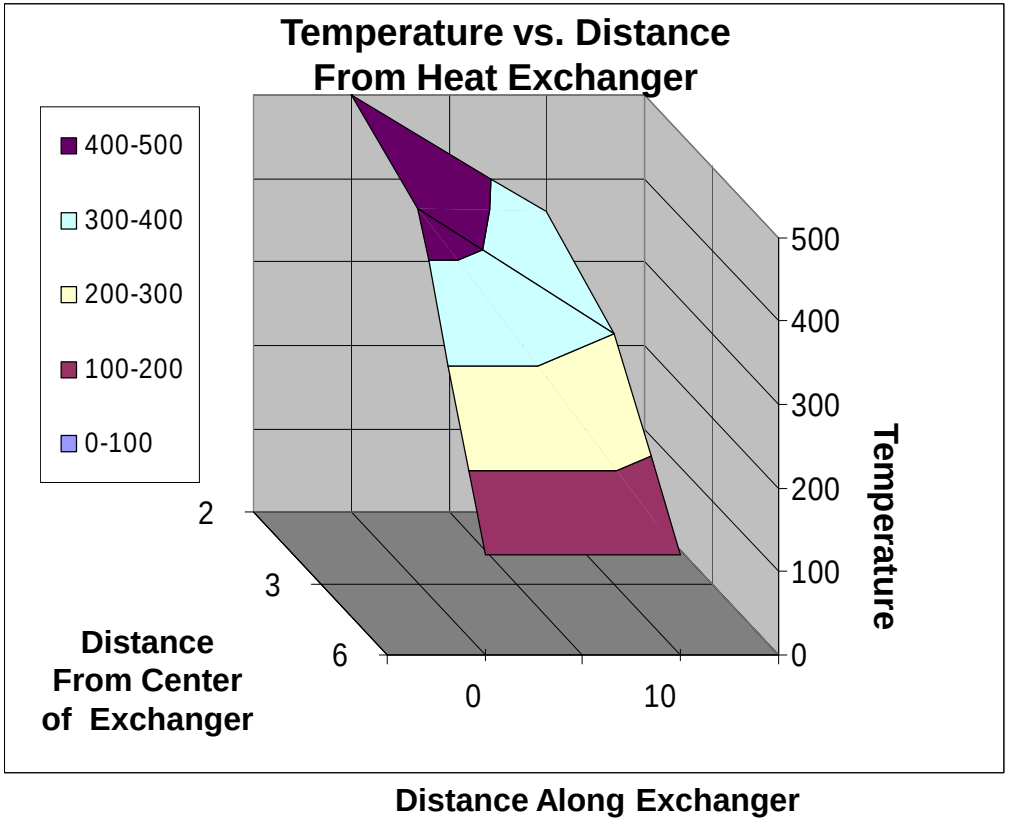


Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, TEMPERATURE VERSUS DISTANCE

- **The effect of the heat exchanger's radiation is approximated by the temperature distribution graph and table below.**
  - Create a Tabular Field from the data using linear interpolation between data points.

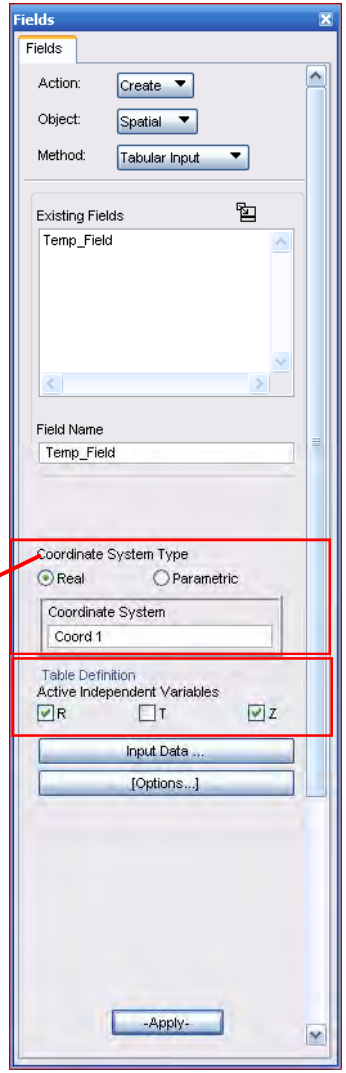
Distance(radial) from center of exchanger	Distance along heat exchanger	
	0	10
2	500 °F	360 °F
3	450 °F	300 °F
6	120 °F	120 °F



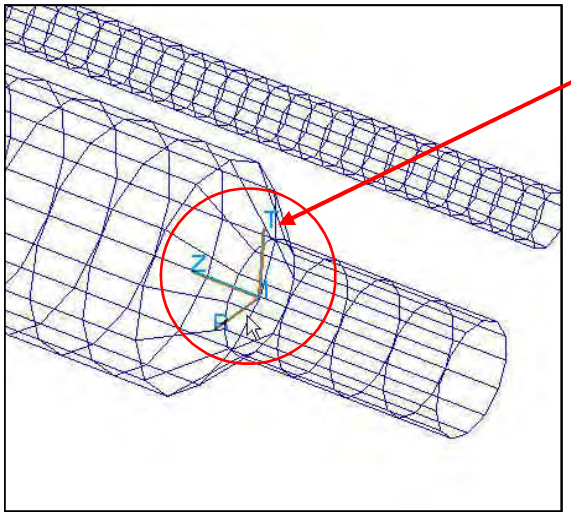
Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, FIELD: SPATIAL/TABULAR INPUT

- **Use Real for the Spatial/Tabular Input Coordinate System Type, and reference the cylindrical coordinate system at the base of the heat exchanger.**
  - By choosing a cylindrical coordinate system, the **Active Independent Variables** change from X Y Z to R T Z.



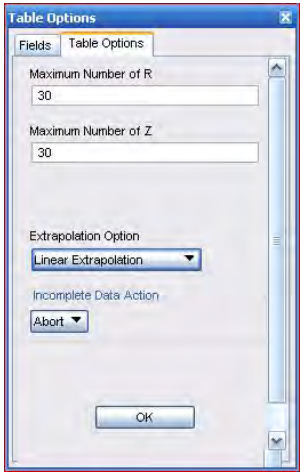
- **From the given data, temperature variations will only vary as a function of radius and distance along the exchanger. For this reason, only R and Z are picked for the Active Independent Variables.**

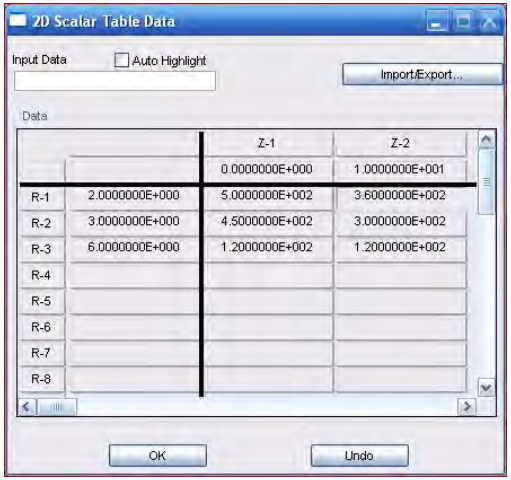


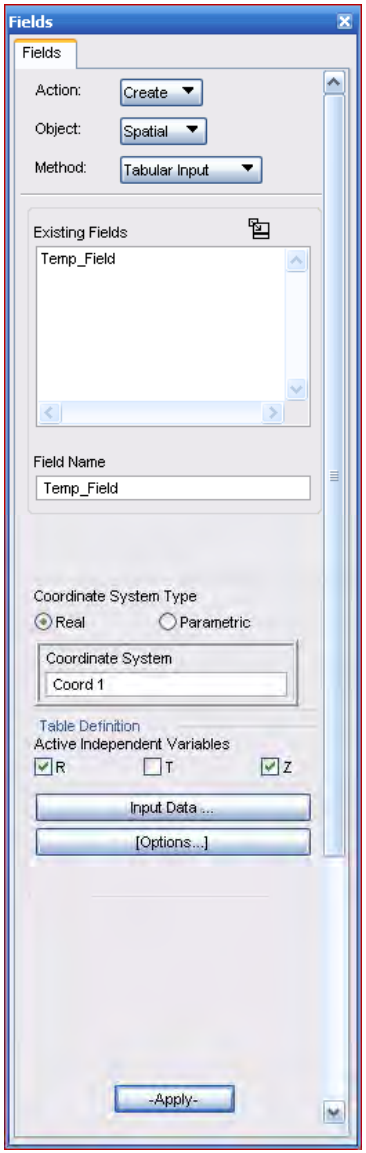
Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, FIELD: SPATIAL/TABULAR INPUT

- **Input Data provides a 2 dimensional table with independent variables, R and Z.**
  - The radii are in the “R” column (rows1,2,3), and the distances along exchanger are in the “Z” row (column 1,2).
- **Under [Options], Extrapolation Option should be set to Linear Extrapolation.**
  - In this case, the option could be ignored, as Nastran will only make a linear extrapolation from one point to another.



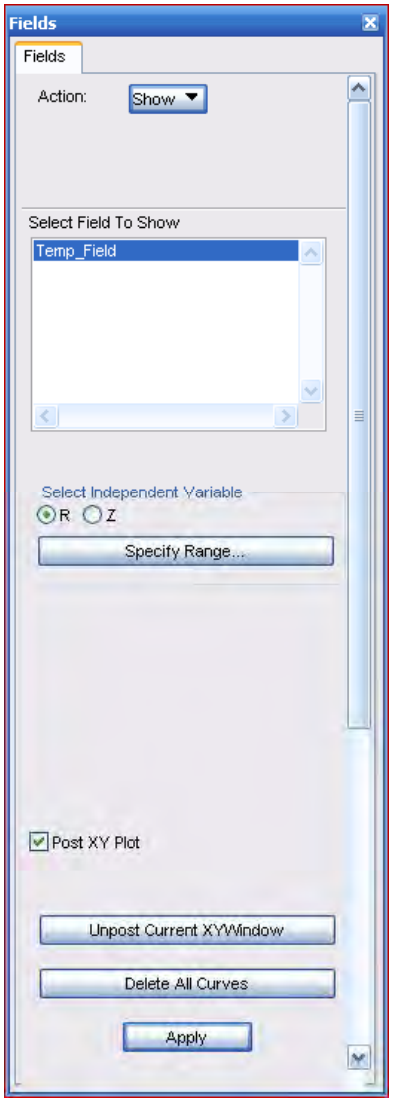


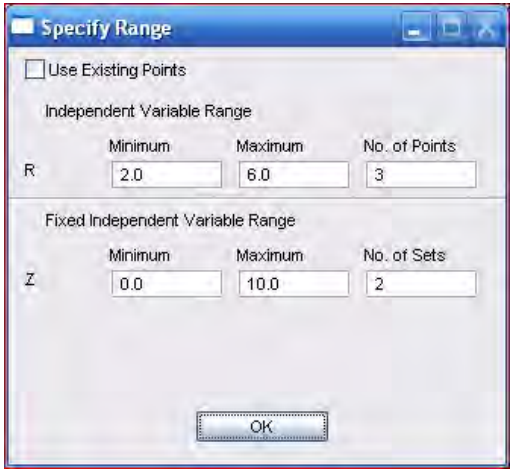


Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, FIELD: SPATIAL/TABULAR INPUT

- **Verify the field by using Show.**
  - Under **Select Independent Variable**, only one direction can be chosen. The chosen variable will become the Independent Variable in the form under **Specify Range**.
  - The **Specify Range** form generally appears filled in with the range and number of points taken from the input. This can, of course, be changed manually, or if no values appear, try checking **Use Existing Points**.

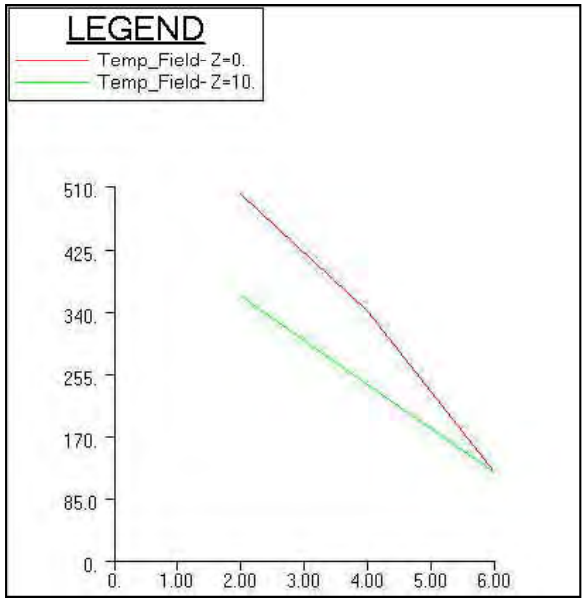


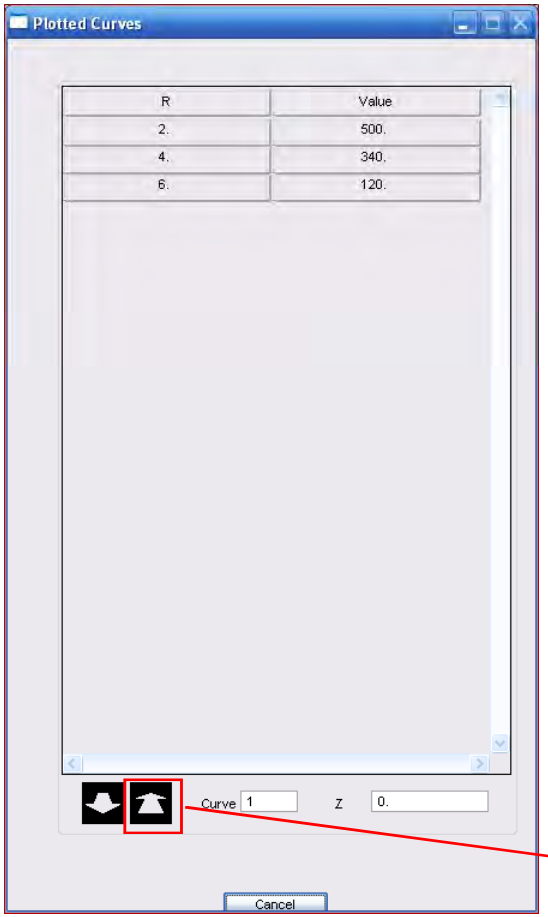


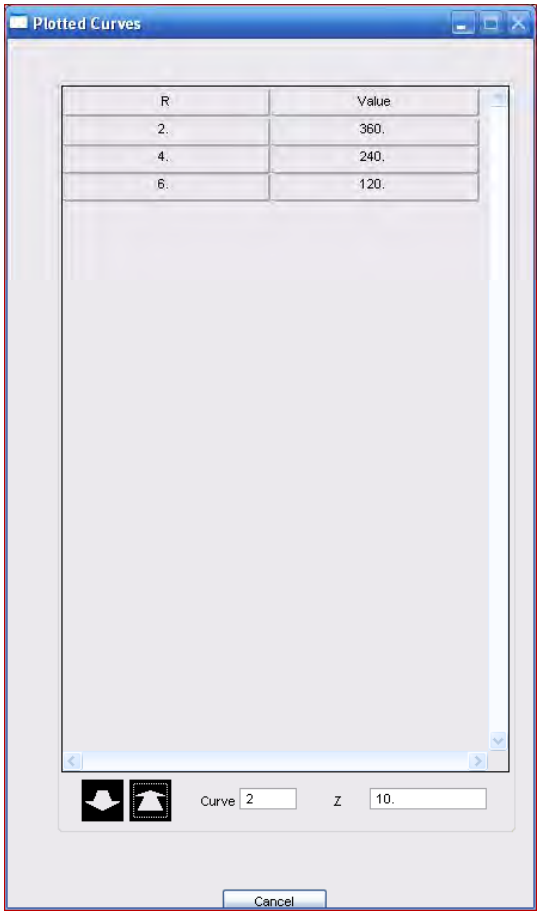
Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, SHOW FIELD

- **The plot shows the temperature gradient in the radial direction. There is a curve for each end of the exchanger, Z = 0 or 10.**



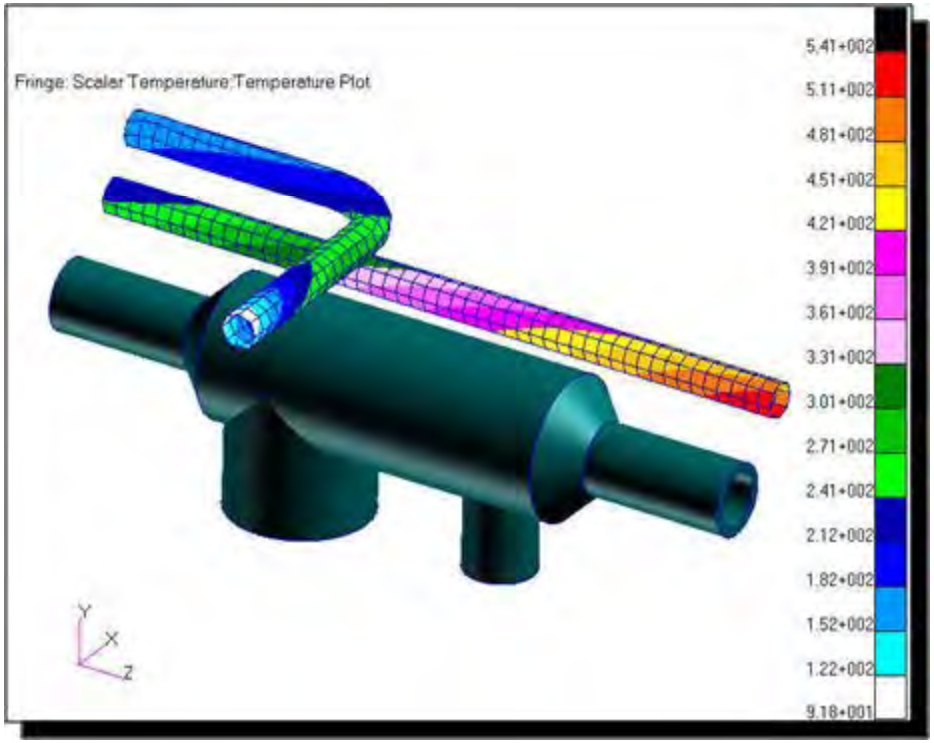




Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, TEMPERATURE LBC USING FIELD

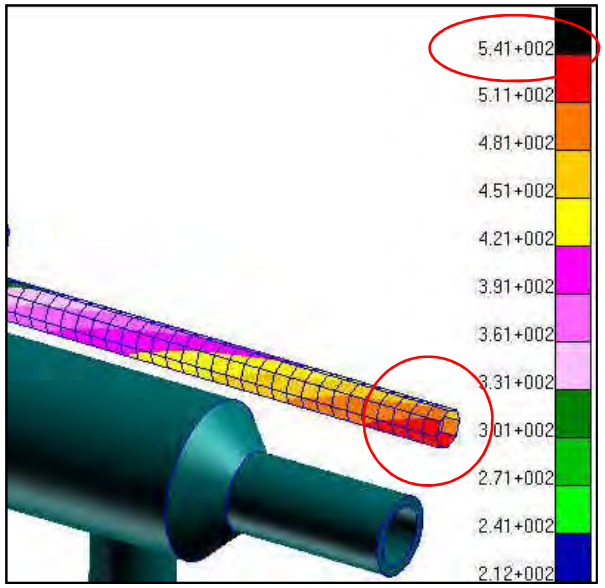
- **Using the field to create a temperature LBC for the thin tubes produces the following temperature distribution**
- **There may be a problem with this temperature distribution**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, TEMPERATURE LBC USING FIELD

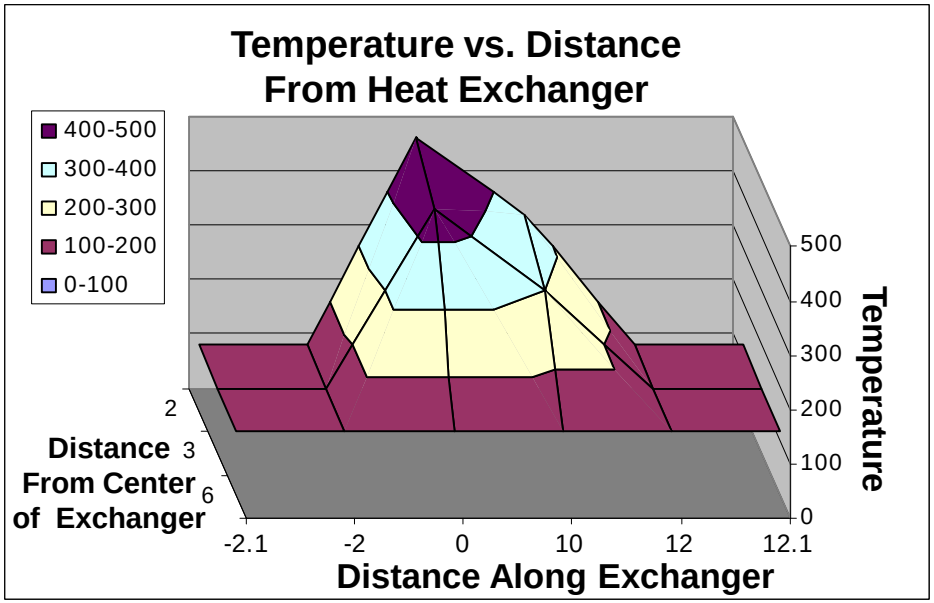
- **The maximum temperature is 541 degrees, which is > maximum temperature specified in the field, 500 degrees.**
- **It may be desired to limit the temperature created using the table.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, ADD DATA TO FIELD DEFINITION

- **To limit the field temperature, a decrease in temperature to the ambient 120 degrees is added to the field.**
  - The extra data points will establish a zero temperature gradient for the remainder of the model.
    - Note that the distance –2.0 or 12.0 establish the drop in temperature, while –2.1 and 12.1 with identical temperature values establish the zero gradient.

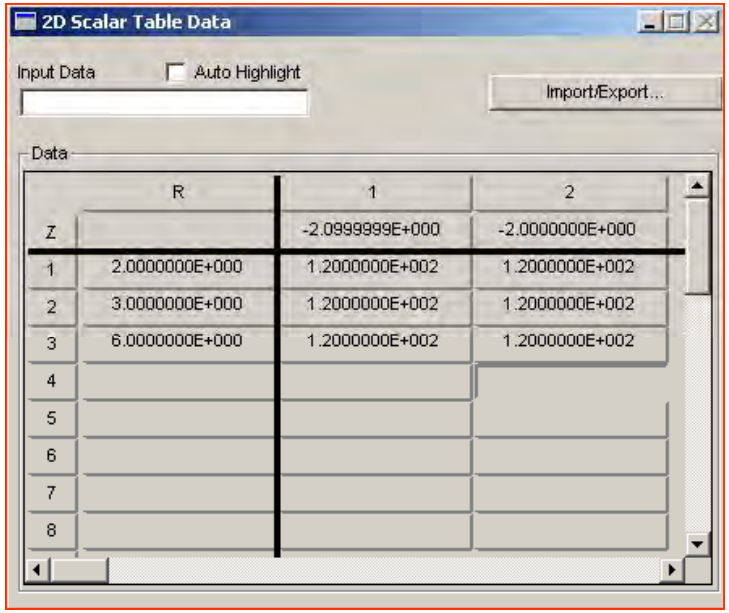


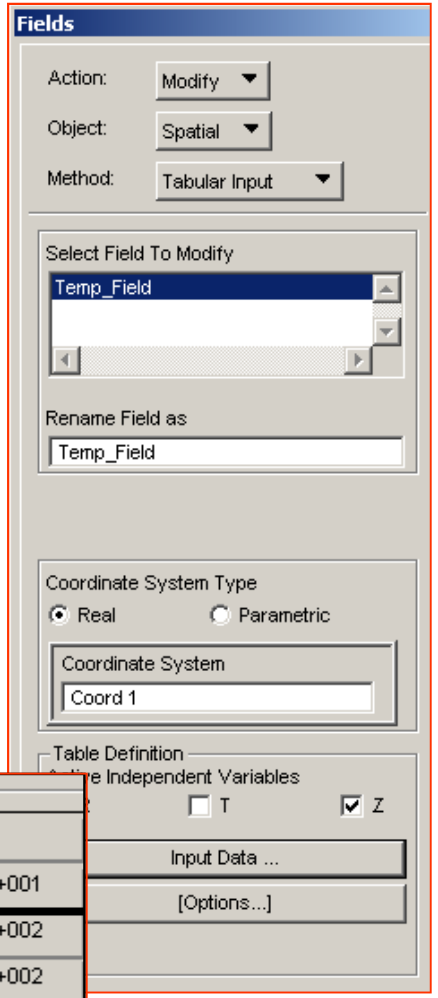
Distance (radial) from center of exchanger	Distance along heat exchanger					
	-2.1	-2.0	0.0	10.0	12.0	12.1
2	120 °F	120 °F	500 °F	360 °F	120 °F	120 °F
3	120 °F	120 °F	450 °F	300 °F	120 °F	120 °F
6	120 °F	120 °F	120 °F	120 °F	120 °F	120 °F

Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, ADD DATA TO FIELD DEFINITION

- **Modify the field, Temp_Field.**
  - Add the additional columns of data from the previous slide.





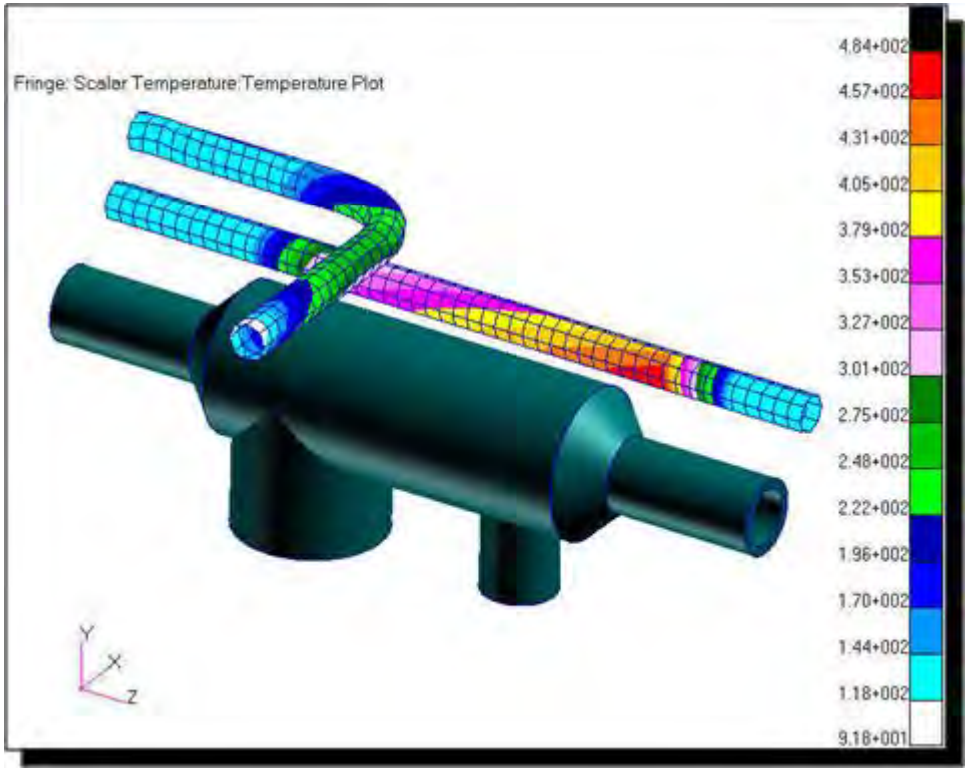
Data

	1	2	3	4	5	6
Z	-2.0999999E+000	-2.0000000E+000	0.0000000E+000	1.0000000E+001	1.2000000E+001	1.2100000E+001
1	1.2000000E+002	1.2000000E+002	5.0000000E+002	3.6000000E+002	1.2000000E+002	1.2000000E+002
2	1.2000000E+002	1.2000000E+002	4.5000000E+002	3.0000000E+002	1.2000000E+002	1.2000000E+002
3	1.2000000E+002	1.2000000E+002	1.2000000E+002	1.2000000E+002	1.2000000E+002	1.2000000E+002

Copyright© 2012 MSC.Software Corporation

CASE STUDY 4, TEMPERATURE LBC USING FIELD MODIFIED

- **The temperature distribution from the modified field, is shown below. The ends of the thin tubes are at or approaching 120 degrees.**

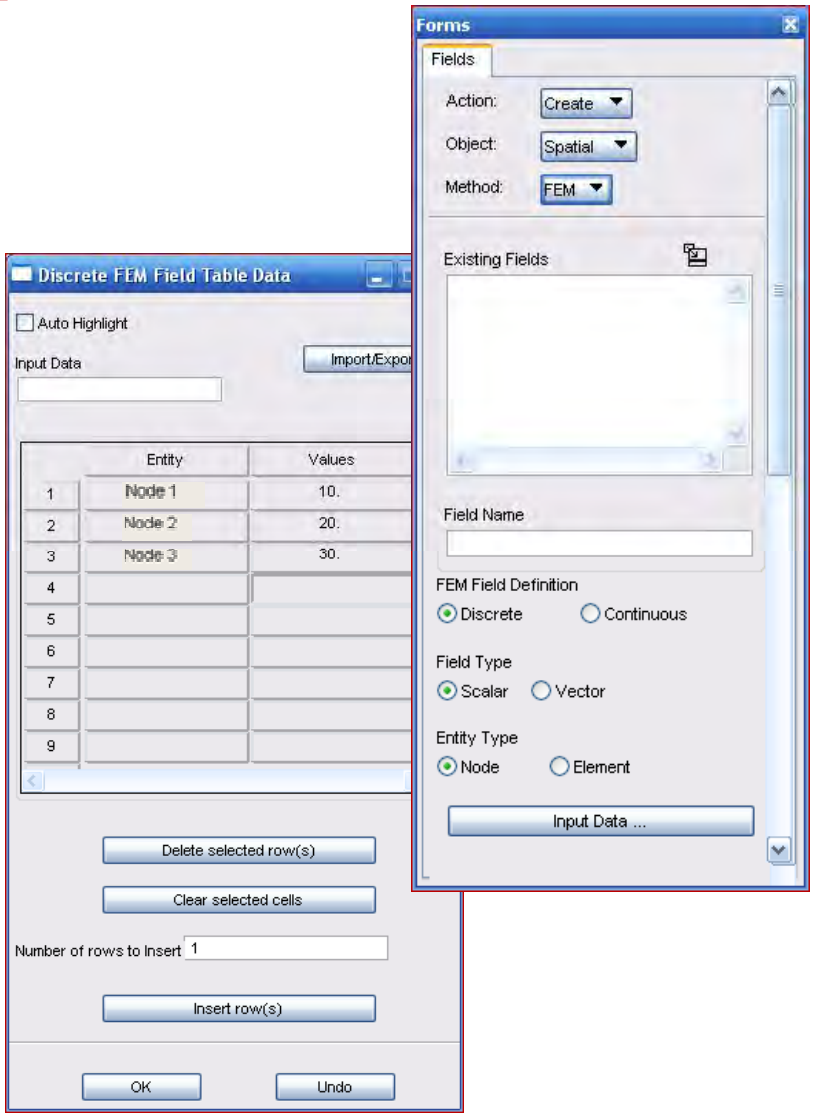


Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

FIELDS, SPATIAL/FEM, DISCRETE

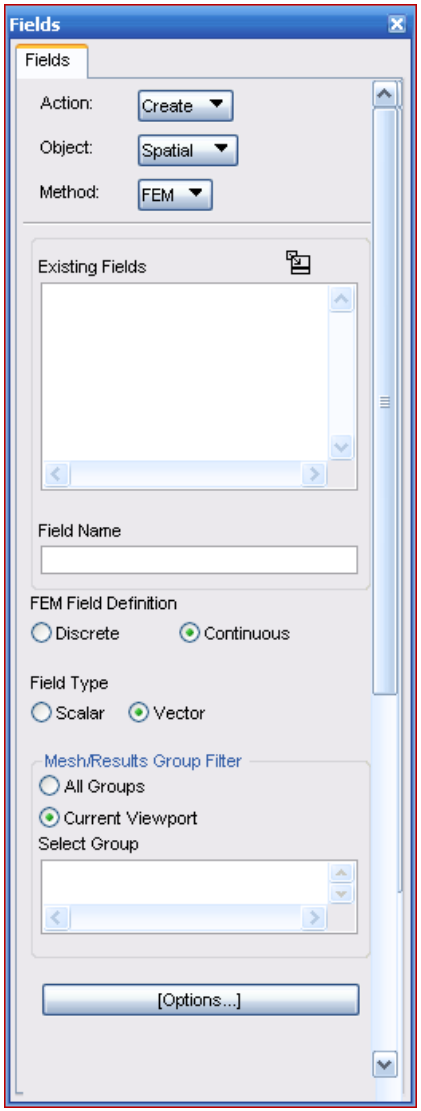
- **Spatial/FEM, Discrete is used to create a field for a part of a finite element model.**
  - Field Type can be either Scalar or Vector.
  - Entity Type can be either on Node or Element.
- **The Input Data form is used to input data (Values) corresponding to either nodes or elements.**
  - If the data is scalar, the value is a real scalar number, e.g. 10.5.
  - If the data is vector, the value is a vector, e.g. <x y z>.



Copyright© 2012 MSC.Software Corporation

FIELDS, SPATIAL/FEM, CONTINUOUS

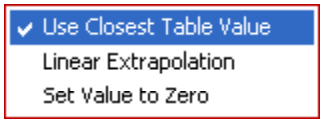
- **This is a useful tool for transferring displayed results (e.g. temperature distribution) to a Load/BC or element property.**
- **Spatial/FEM, Continuous is used to create a field based on an existing (displayed) scalar or vector plot for a finite element model.**
  - The plot must be displayed in a viewport.
- **Mesh/Results Group Filter allows different selections for the field by**
  - individual group
  - multiple groups through selection
  - what is currently in the viewport

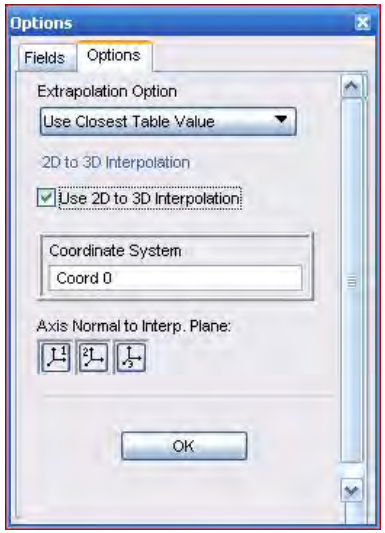


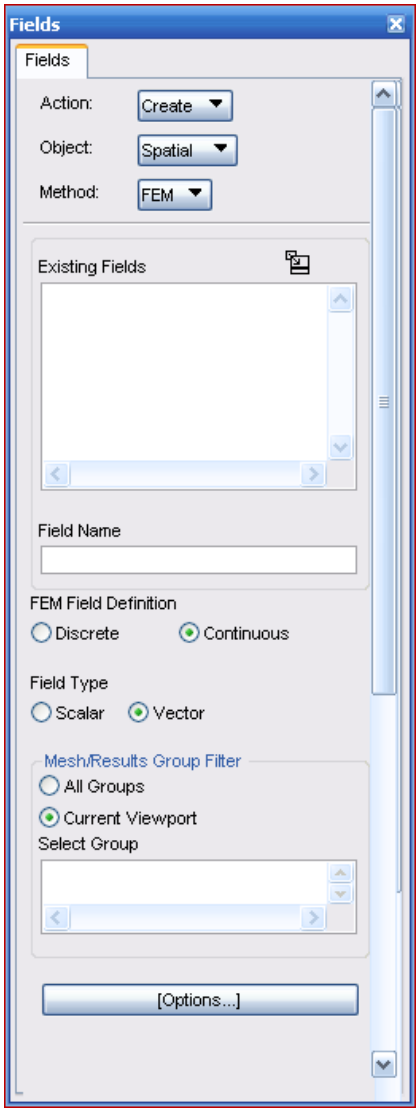
Copyright© 2012 MSC.Software Corporation

FIELDS, SPATIAL/FEM, CONTINUOUS, OPTIONS

- **Extrapolation Option provides the extrapolation methods that the other field types have.**
- **Using 2D to 3D Interpolation will project a 2D field into 3D space, as defined by the Axis Normal to Interp. Plane and the Coordinate System chosen.**
  - This is particularly valuable when the FEM field created will be applied to an entity that does not exactly match the original shells.
  - Additionally, an FEM field created from a 2D shell model could be applied to a 3D solid element model.





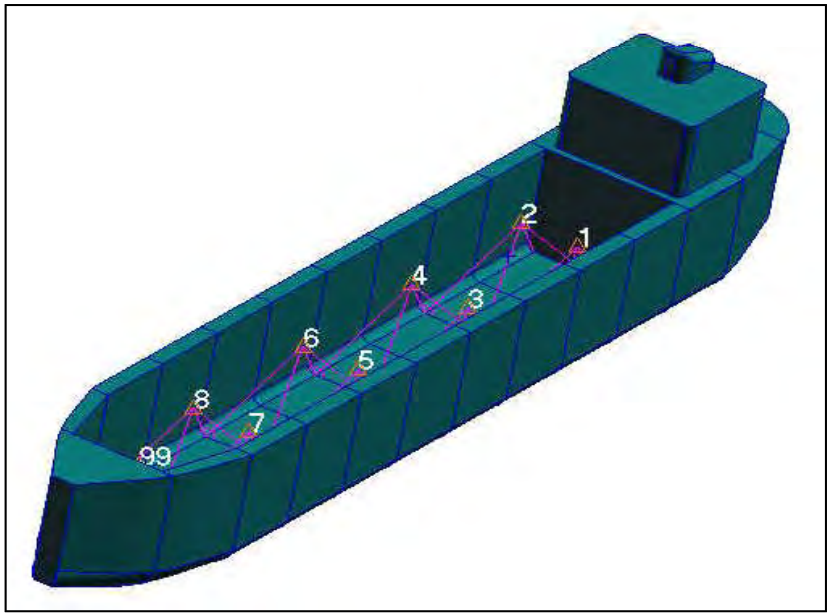


Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

CASE STUDY 5, MODIFICATION OF MASS THROUGH FIELDS

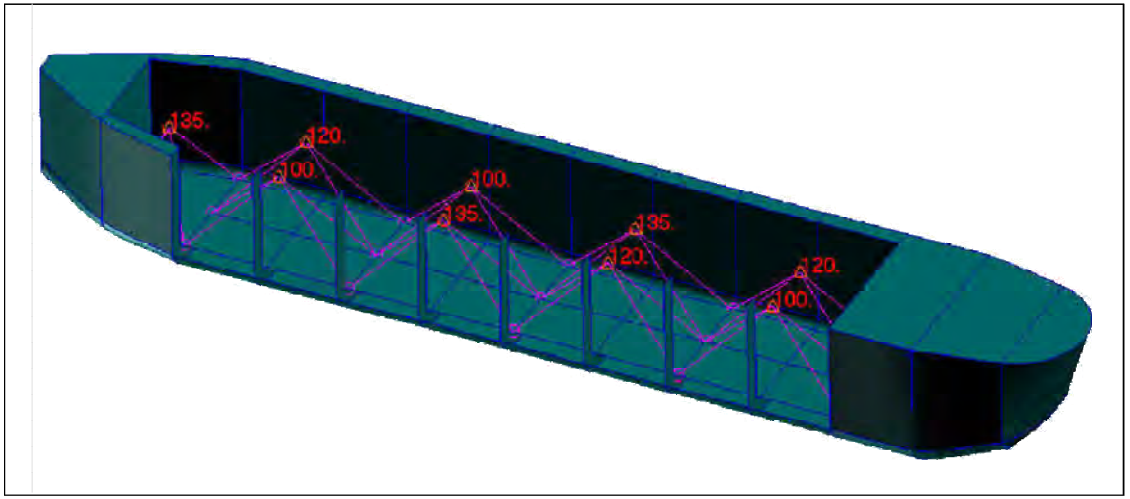
- **A cargo container ship must be analyzed with many different loads and loading/weight distributions.**
- **Use Patran Fields to manage and modify a number of masses that are used to represent the different loads.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 5, DESCRIPTION

- **The different potential loads for a cargo ship are represented by Nastran CONM2s connected to the structure with RBE2s.**



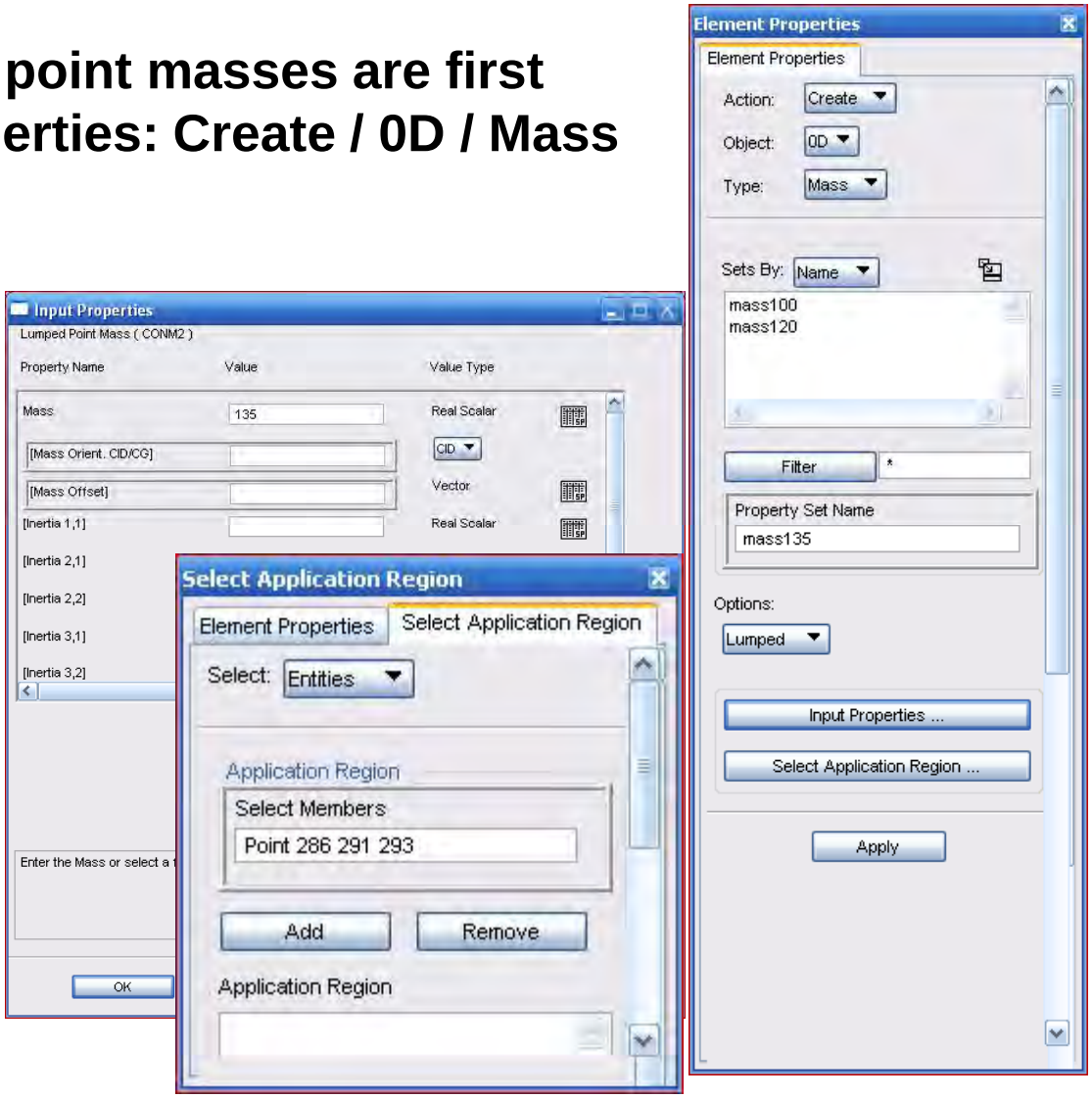
- **Patran has a special feature for importing model files with multiple point masses. The masses are consolidated into a single field used in a single property.**
- **These individual masses, for the field, can then be displayed and modified in a single table.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY 5, CREATING POINT MASSES

- **In this case study, the point masses are first created in Patran Properties: Create / 0D / Mass**

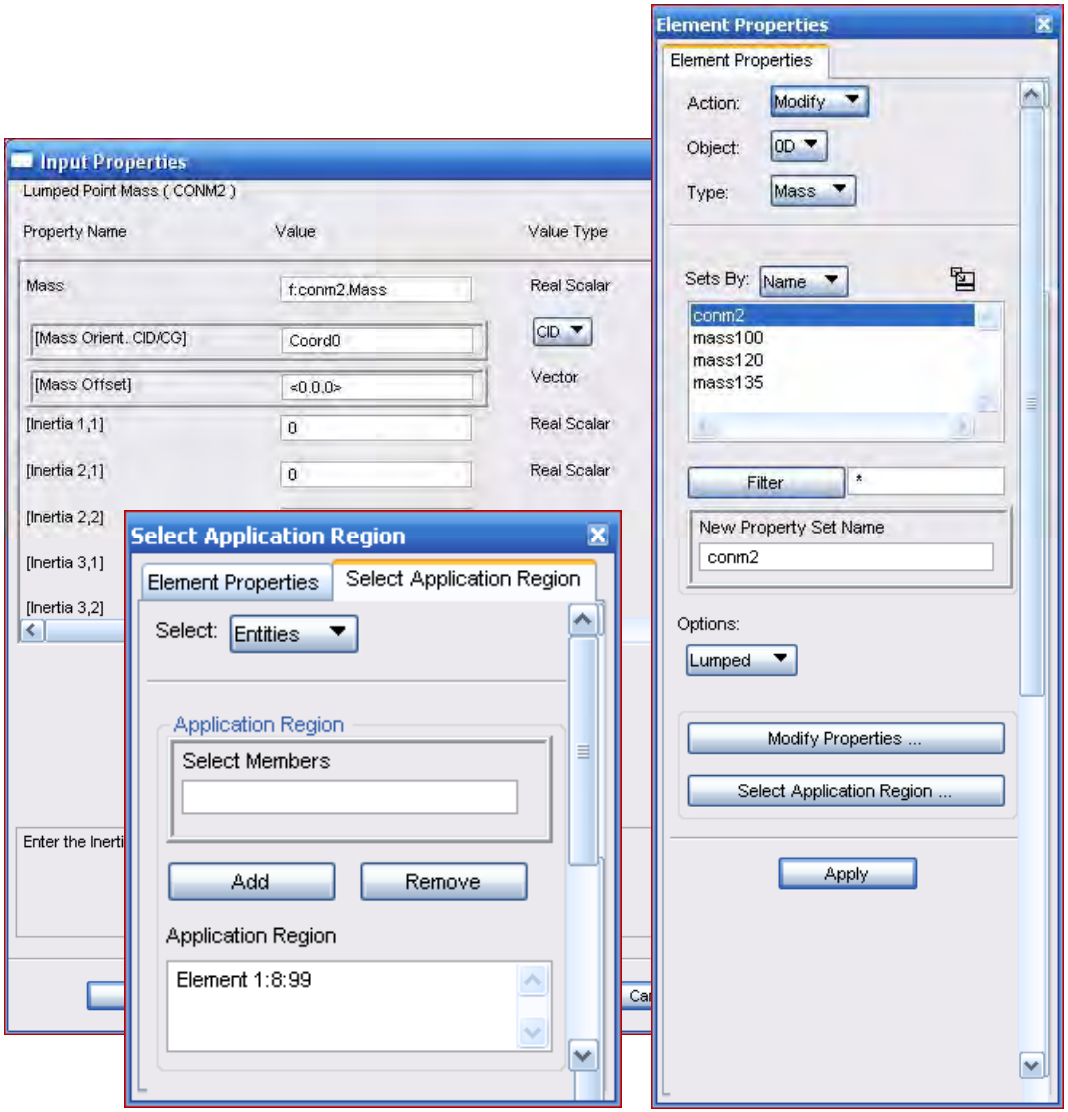
- **Three property sets; mass100, mass120, and mass135; each with its mass, e.g. 135.0 for property set mass135, and corresponding application region of three points (see the figure in the previous slide), are created.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 5, MODIFICATION OF THE MASS FIELD

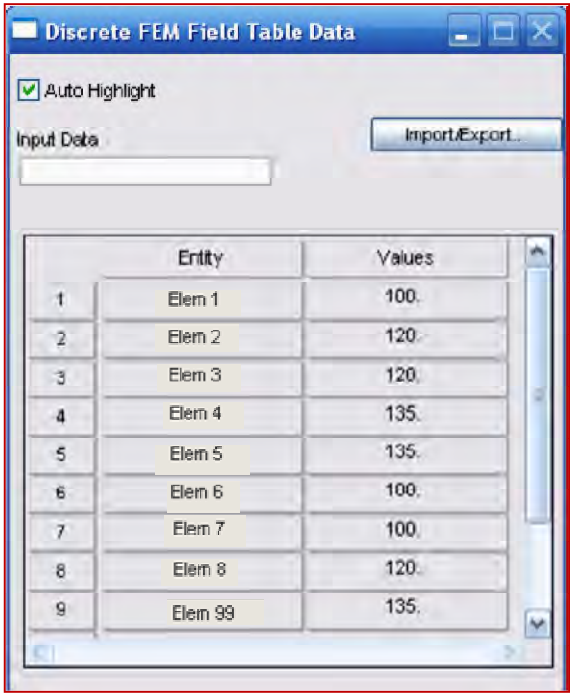
- **Export a solver file (e.g. .DAT), then import the file.**
- **A new mass property set named conm2 is created.**
- **Properties: Modify shows that the new mass set, conm2, references the field f:conm2.Mass.**

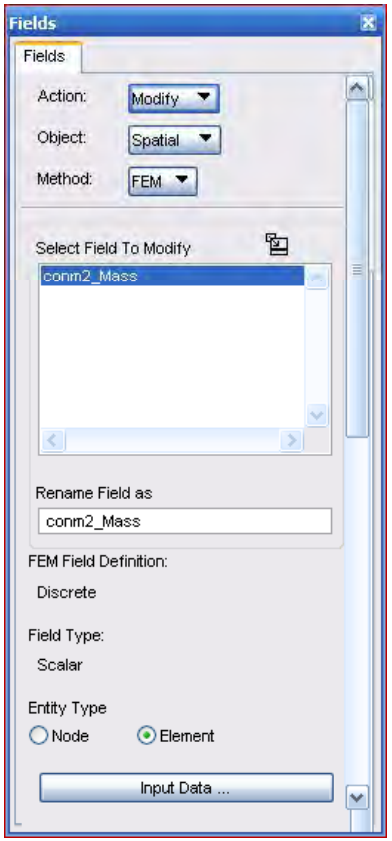


Copyright© 2012 MSC.Software Corporation

CASE STUDY 5, MODIFICATION OF THE MASS FIELD

- **Fields: Modify, and select Spatial. Select the field conm2.Mass.**
- **Select Input Data. The table consists of element IDs and corresponding 0D mass values.**

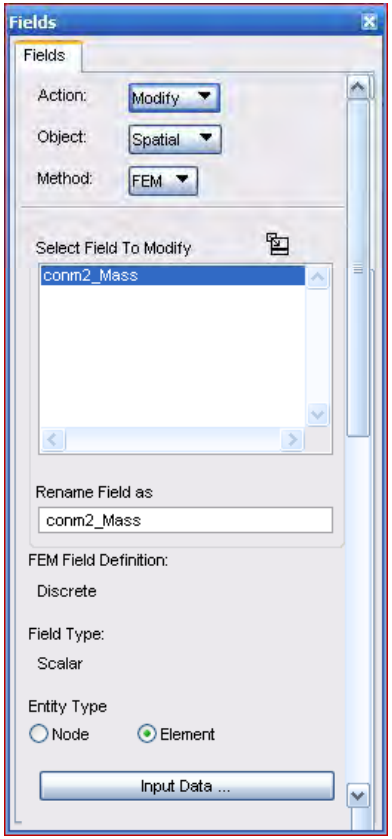


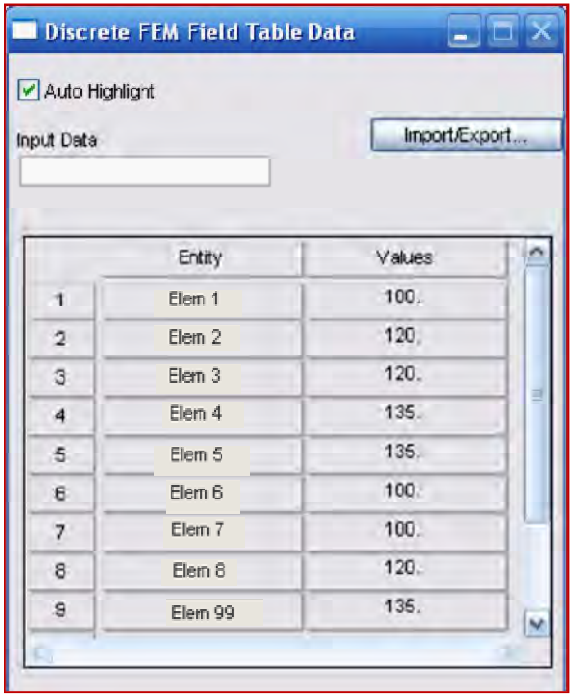


Copyright© 2012 MSC.Software Corporation

CASE STUDY 5, MODIFICATION OF THE MASS FIELD

- **Click on a particular Values cell (mass). The value will be displayed in the Input Data box.**
  - An Entity or Values entry can be changed once selected.
- **After changing the value, press the Enter key to write the value to the cell.**
  - After changing the data, click the Apply button on the main Fields form. This will update the field.

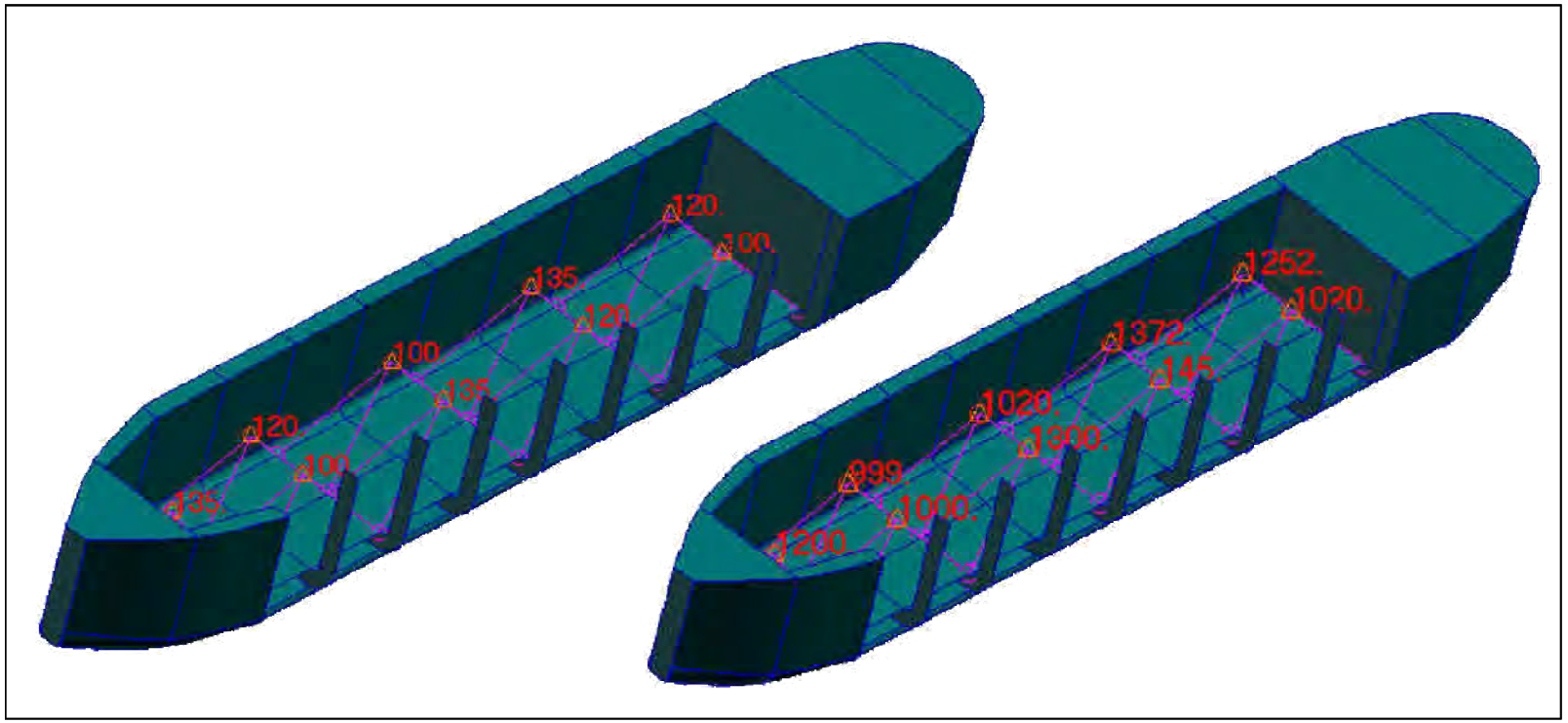




Copyright© 2012 MSC.Software Corporation

CASE STUDY 5, MODIFICATION OF THE MASS FIELD

- **The original mass properties and modified mass properties are shown below:**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 5, ALTERNATE METHOD

- **For this case study, individual properties (e.g. mass135, mass=135, Point 266, 291, 293) were first created, then the fields representing these properties were created as a result of exporting then importing a solver model file (e.g. .DAT). Another option would have been to make the field for the masses directly in Patran. Either approach would work.**

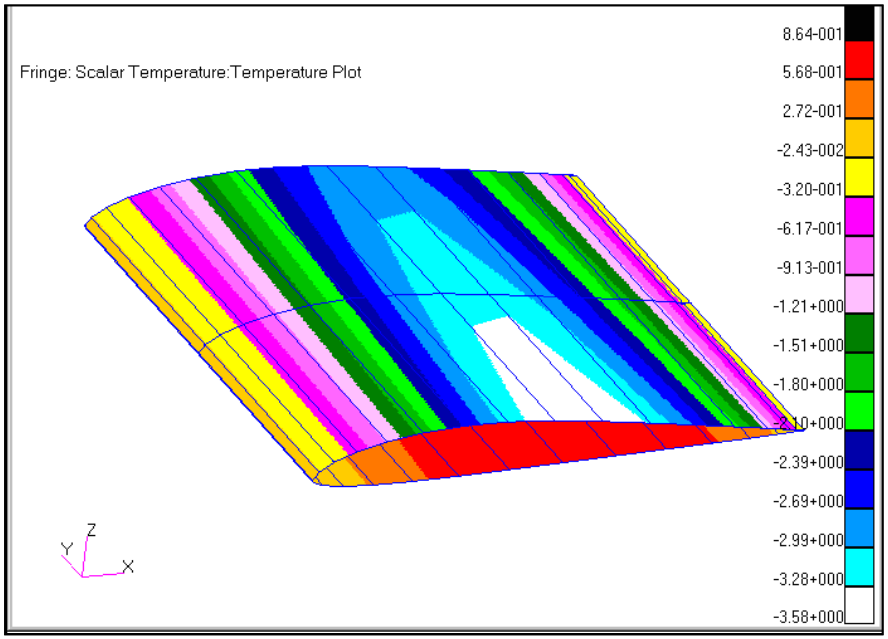
Copyright© 2012 MSC.Software Corporation

CASE STUDY 6

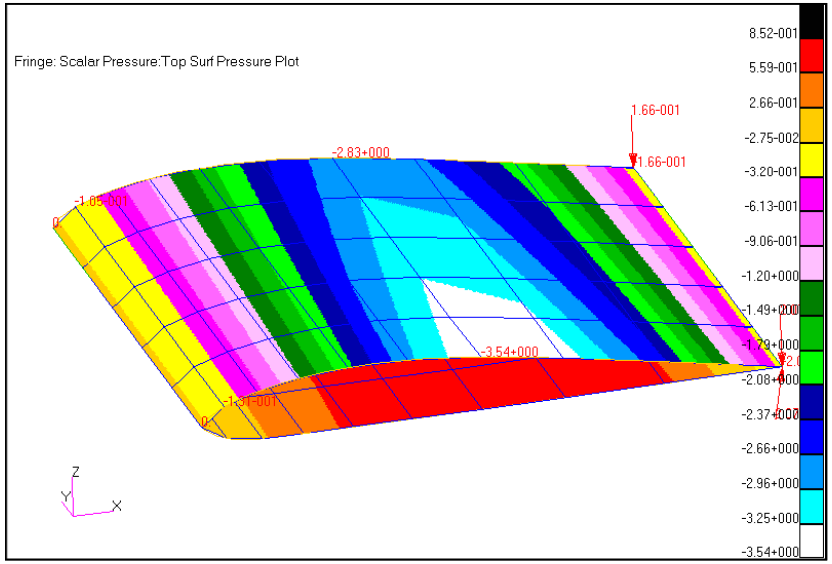
FEM FIELD, CONTINUOUS
CFD PRESSURE TO PATRAN PRESSURE

Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, APPLYING CFD PRESSURE DATA FEM FIELD, CONTINUOUS



Imported CFD pressure data as Patran temperature contour



Patran pressure contour for structural model

Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, FEM FIELD, CONTINUOUS

- **This case study demonstrates the use of FEM Field, Continuous as a way of creating an LBC for a structural model from results for a different type of model (e.g. a CFD model).**
- **This case study demonstrates one way to get CFD pressure data into Patran.**
- **The method shown is for CFD data in the form of point locations and corresponding pressures.**
- **An attempt will be made to explain problems with this approach and discuss alternatives. There are other approaches for getting the data into Patran.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, CFD PRESSURE DATA

- **Given**
  - Table of position (x,y,z) and corresponding pressure data.
  - Model is of an Eppler 205 airfoil consisting of 3 rib type section of data.

- **Example of given data:**

x	y	z	pressure
1.0000	0.0000	0.0000	-0.2072
0.9705	0.0000	0.0043	-0.4144
0.9229	0.0000	0.0120	-1.1037
0.8562	0.0000	0.0220	-1.8829
0.7741	0.0000	0.0342	-2.6467
0.6811	0.0000	0.0478	-3.2535
0.5822	0.0000	0.0615	-3.5787
0.4827	0.0000	0.0734	-3.5436

Copyright© 2012 MSC.Software Corporation

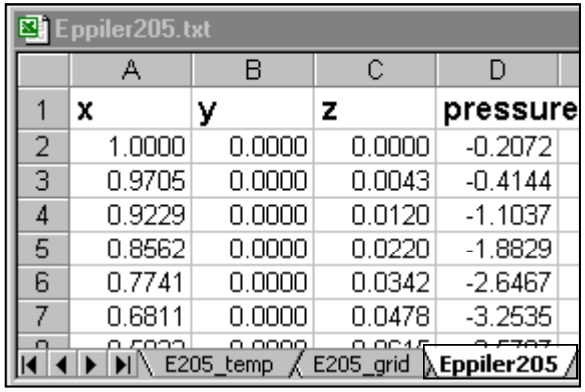
CASE STUDY 6, PATRAN IMPORT METHODS

- **The most practical way to import this form of data is to put it in the form of Nastran input records (bulk data).**
  - This could be accomplished in a number of ways:
    - A script could be created that reads the data, then writes it in the format needed for Nastran (bulk data)
    - The text information could be brought into another program such as Microsoft Excel for manipulation into Nastran input record format.
    - A modification or adjustment to the CFD code could possibly produce Nastran records.
  - Once the data is in the form of an Nastran input file, it can be imported into Patran using File/Import/Nastran Input.
- **Other approaches include the use of**
  - Patran neutral and results files
  - ABAQUS input file, .inp file (text file)
  - ANSYS input file (text file)

Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, CREATE NASTRAN INPUT FILE

- **In this example, Microsoft Excel is used. Below, is a screen snap-shot of the raw data brought in from the text file.**



Eppiler205.txt

	A	B	C	D
1	x	y	z	pressure
2	1.0000	0.0000	0.0000	-0.2072
3	0.9705	0.0000	0.0043	-0.4144
4	0.9229	0.0000	0.0120	-1.1037
5	0.8562	0.0000	0.0220	-1.8829
6	0.7741	0.0000	0.0342	-2.6467
7	0.6811	0.0000	0.0478	-3.2535

E205_temp | E205_grid | Eppiler205

- **The Excel file data is divided into three sheets. The first one (shown) contains the raw data.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, CREATE NASTRAN INPUT FILE

- **Second sheet**
  - Location (x,y,z) data

Eppiler205.txt

	A	B	C	D	E	F
1						
2	GRID	1	0	1.00000	0.00000	0.00000
3	GRID	2	0	0.97049	0.00000	0.00427
4	GRID	3	0	0.92285	0.00000	0.01196
5	GRID	4	0	0.85624	0.00000	0.02199
6	GRID	5	0	0.77412	0.00000	0.03419
7	GRID	6	0	0.68108	0.00000	0.04777
8	GRID	7	0	0.58218	0.00000	0.06147

E205_temp | **E205_grid** | Eppiler205

- **Third sheet**
  - Pressure data

Eppiler205.txt

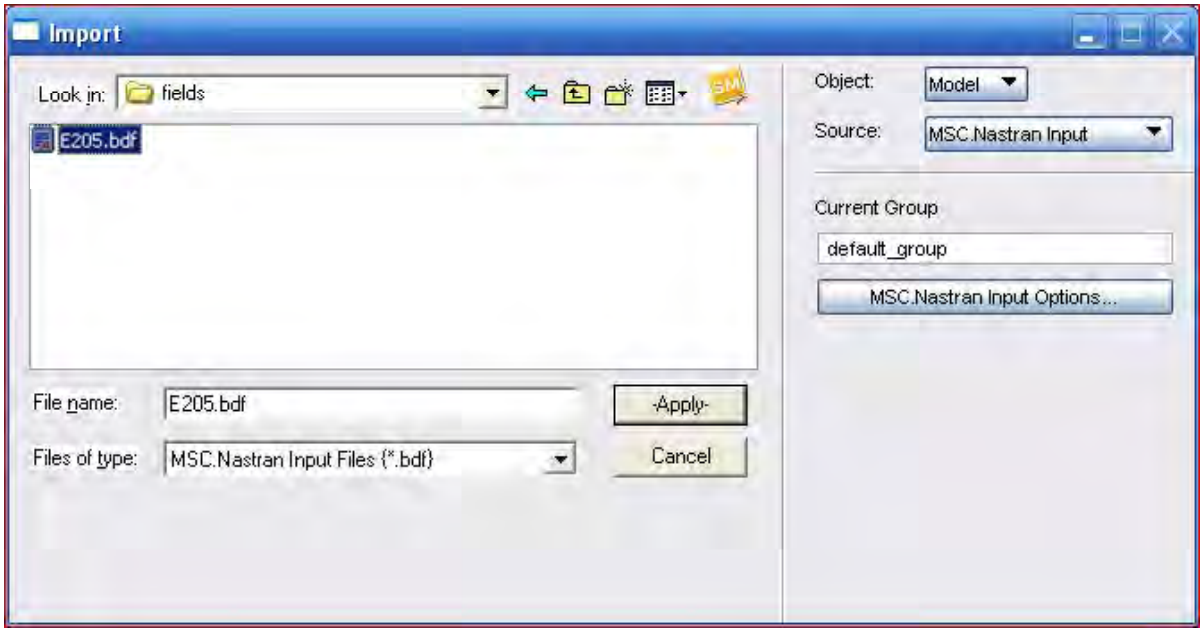
	A	B	C	D	E
1					
2	TEMP	1	1	-0.20720	
3	TEMP	1	2	-0.41440	
4	TEMP	1	3	-1.10373	
5	TEMP	1	4	-1.88287	
6	TEMP	1	5	-2.64672	
7	TEMP	1	6	-3.25352	
8	TEMP	1	7	-3.57866	

E205_temp | E205_grid | Eppiler205

Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, CREATE NASTRAN INPUT FILE

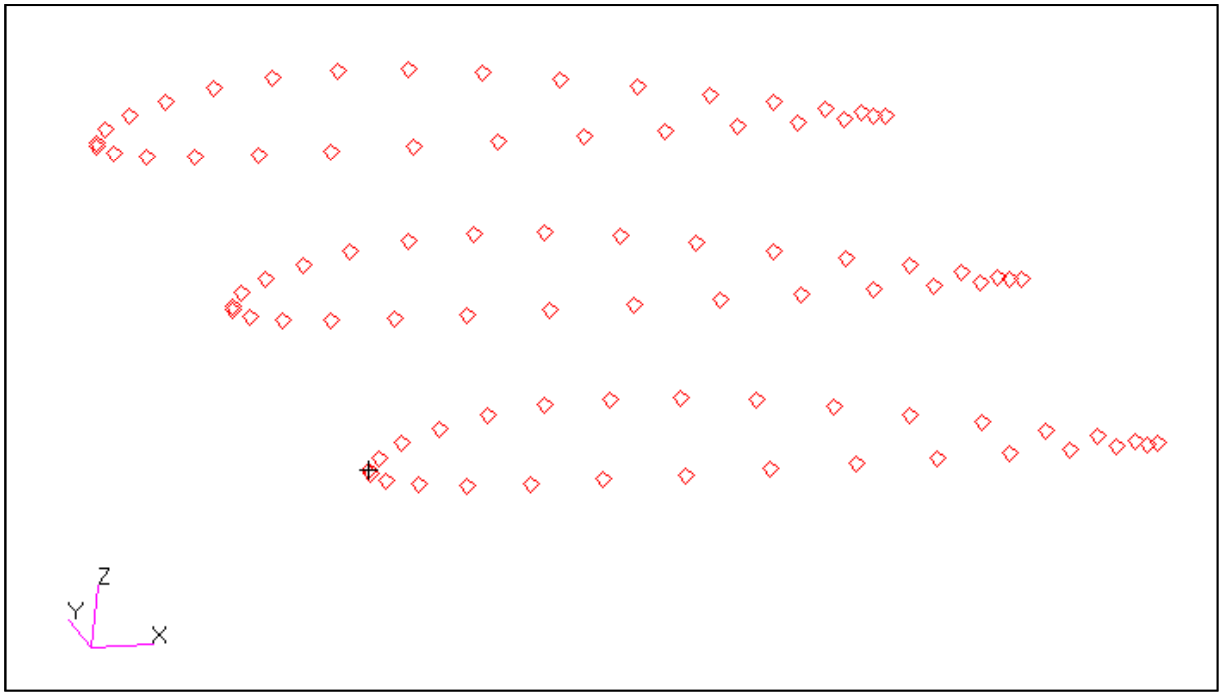
- **Once the data is arranged, write it to a text file(s).**
- **Once the text file(s) is created, it can be imported into Patran.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, VERIFY IMPORTED NASTRAN INPUT FILE DATA

- **Imported Nastran grid points, called Patran nodes.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, VERIFY IMPORTED NASTRAN INPUT FILE DATA

- **Under Loads/BCs, the temperature should be plotted to make sure the data is correct.**
  - If Tempe_temp.1 is not listed under Assigned Load/BC Sets, go to Case Control. Make sure the load is assigned to a load case.

Copyright© 2012 MSC.Software Corporation

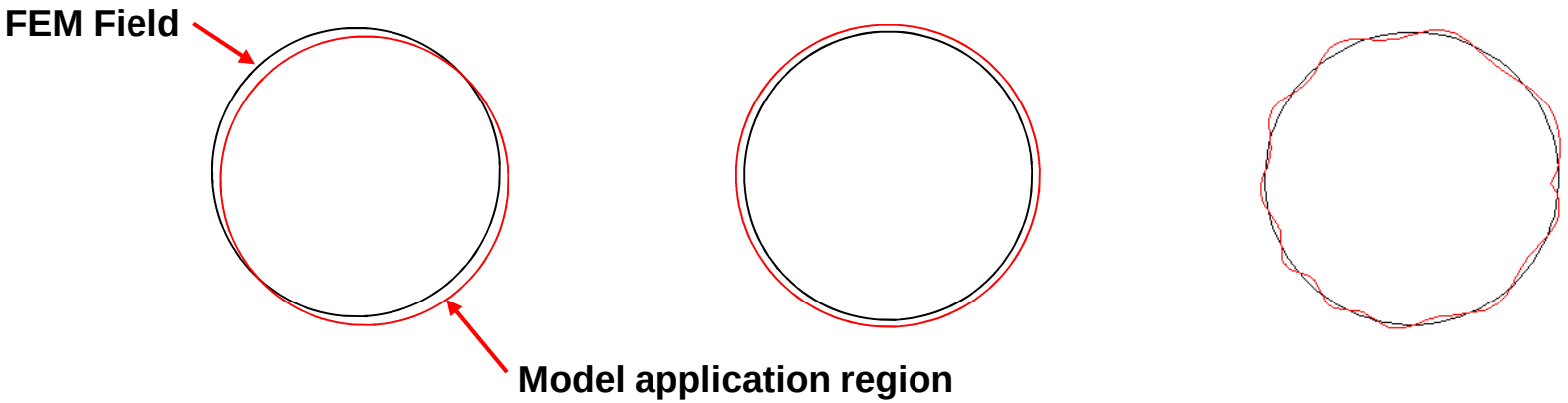
CASE STUDY 6
TEMPERATURE TO PRESSURE IN PATRAN

- **Use the temperature information in Patran to create pressure for a structural model. This is done using a continuous FEM Field. Once the field is created, it can then be used to create a Loads/BCs pressure set.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY 6
GENERAL ISSUES FOR CONTINUOUS FEM FIELDS

- **There are several issues that may have to be dealt with in creating continuous FEM Fields**
  - One issue is that if the domain/region for the creation of the FEM Field is not exactly (within tolerance) where the field is to be applied, then, when the field is used averaging (interpolating/extrapolating) must be performed. The effects of the averaging are not always known. The averaging Options may help to improve the quality of averaging. Sometimes, values obtained by the FEM Field, outside of the domain used for creating it, are of poor quality.



Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, GENERAL ISSUES FOR CONTINUOUS FEM FIELDS

- (Continued) This issue can be partially resolved for 2D models by enabling an Option to extend the field in one of three coordinate directions. Effectively, this maps 2D data to 3D space. This can be an effective method, but limits the field creation to surfaces (of elements) to those that do not overlap in that "one" direction. This can be dealt with by having a group for each surface (of elements). This is discussed herein.
- Another approach is to generate the FEM Field from a domain that is slightly larger than the application region that the field will be used for. This is not always possible for the given model and data, but may be implemented in Patran by slightly scaling-up the original model. A scaling of 1% may be acceptable.
- "Secondary" FEM Field averaging is done if the model the field is being applied to is very small relative to the size of the domain used for creating the FEM Field; as the size of the model approaches the global model tolerance (e.g. 0.005), secondary averaging is done. A possible approach for dealing with this problem is to scale-up the model, even by a factor as large as 10.

Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, GENERAL ISSUES FOR CONTINUOUS FEM FIELDS

- Another issue with creating FEM Fields is that this type of field must be created from a contour/fringe or vector plot that is displayed on the screen. For scalar results (e.g. temperature), only contour plots can be created, and elements are needed to create this type of plot.
  - This is a problem for this CASE STUDY as elements were not available with the CFD data.
  - Some CFD software will output elements (generally, triangular elements). If the software being used does not do this, then 2D elements can be created manually in Patran. Sometimes, 3D elements are necessary.

Copyright© 2012 MSC.Software Corporation

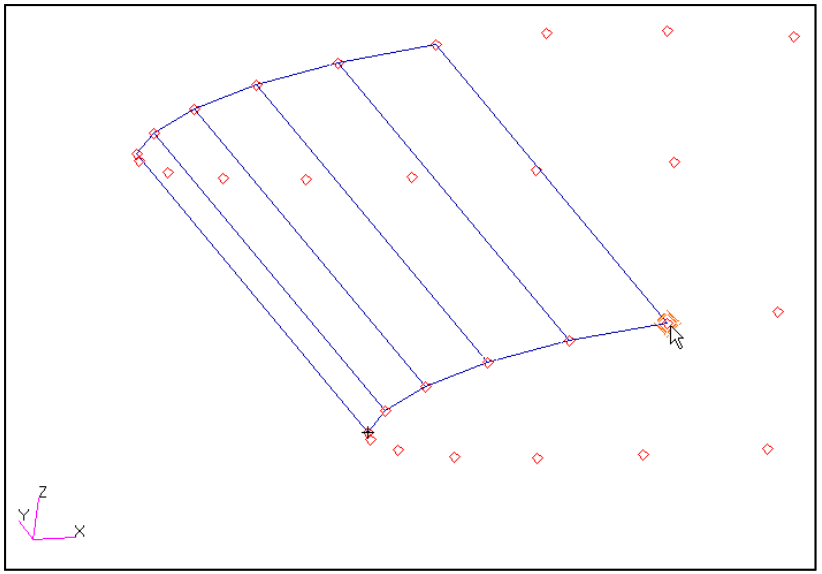
CASE STUDY 6, PROBLEM USING 1D ELEMENTS

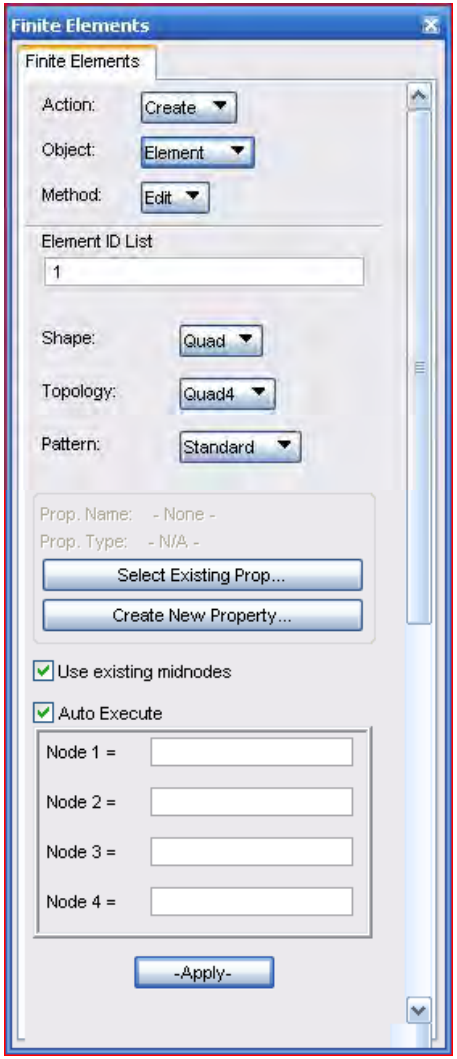
- **Attempts have been made to make an FEM Field with 1D elements that connected all the grids. This FEM Field was then applied to a 2D set of elements. The averaging was poor. The only exception was where the CFD node locations very closely or exactly coincided with the structural mesh nodes that the pressure was to be applied to.**
- **It is not recommended to use FEM Fields created from 1D elements unless the field is applied to 1D elements with the same node locations.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, MANUALLY CREATE 2D ELEMENTS

- **Because there are no 2D elements to create the FEM Field from, it is necessary to create the elements manually.**
  - Use the form for Elements, and select the nodes that are to be at the corners of the Quad4 elements.

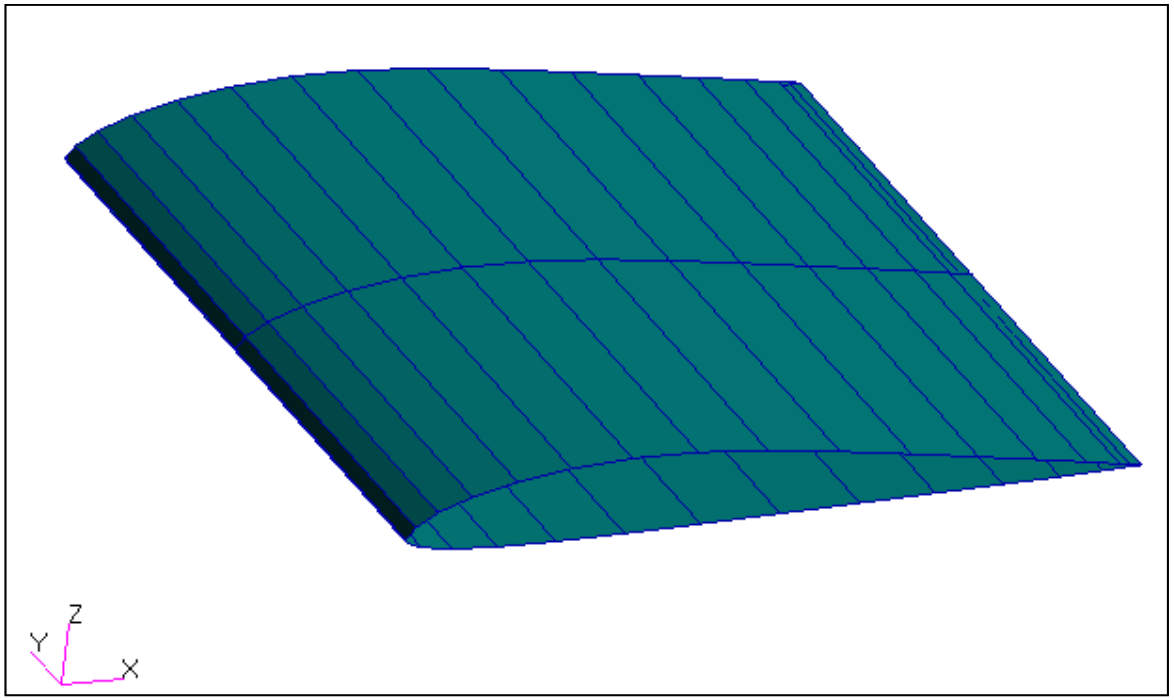




Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, MANUALLY CREATE 2D ELEMENTS

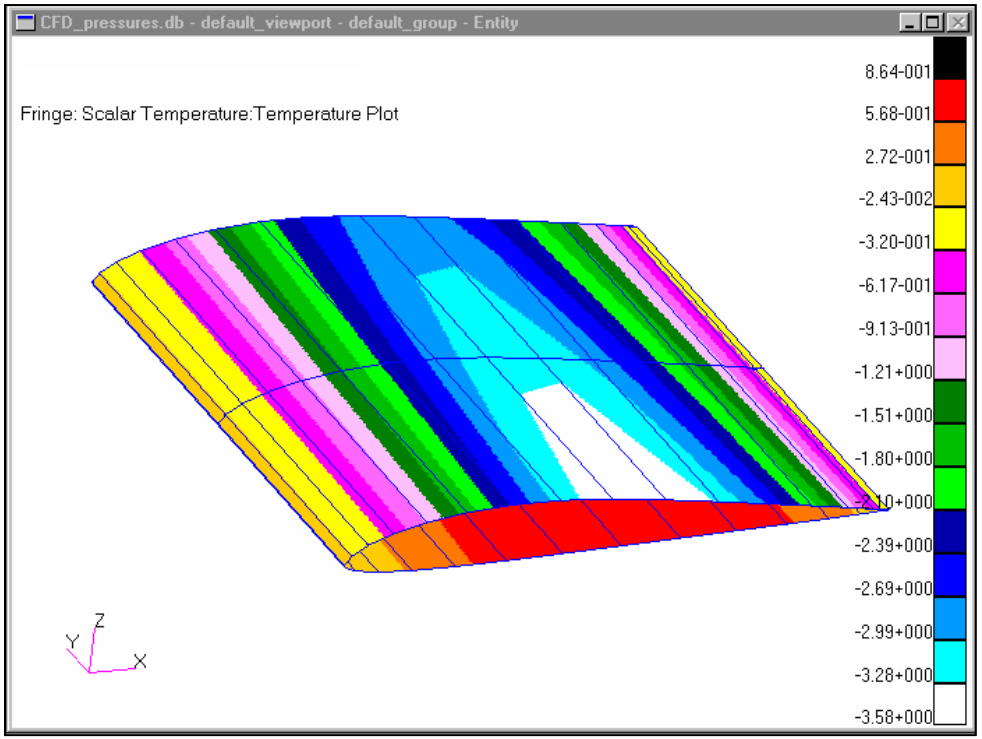
- **Once the 2D elements are created, the model should look like the following. There are no elements on the sides.**

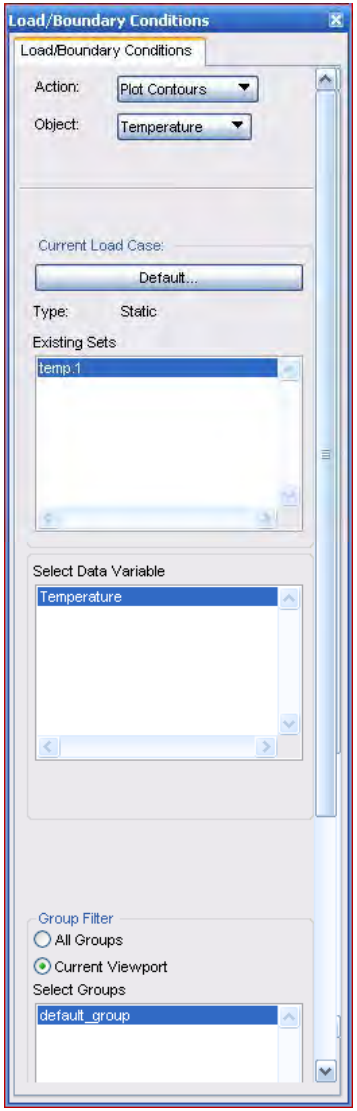


Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, DISPLAYING TEMPERATURE CONTOUR

- **With the 2D elements created, using the imported nodes, the temperature data can be displayed.**

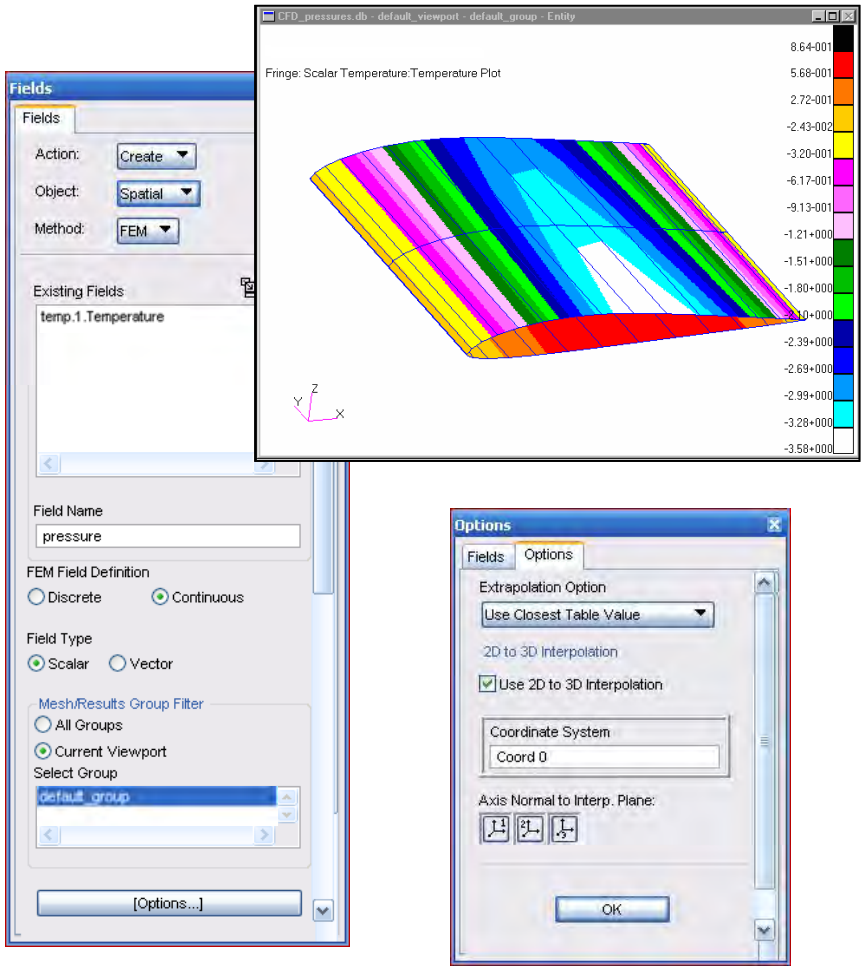




Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, CREATE FEM FIELD, CONTINUOUS

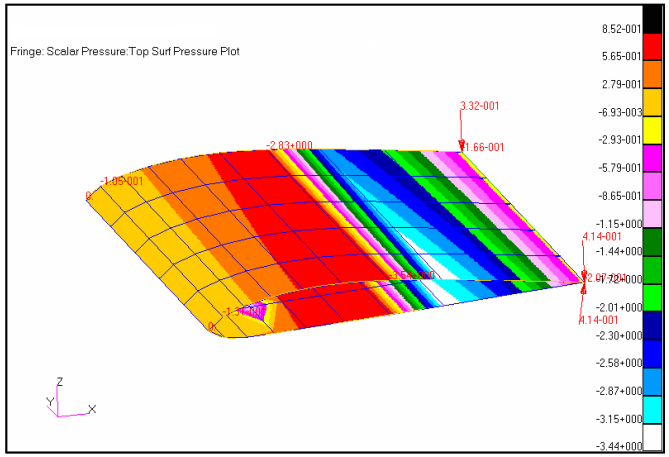
- **Create the FEM Field using the posted temperature fringe.**
- **Fill out the Fields menu as shown, and go to Options**
  - Under Options, check the 2D to 3D interpolation, and pick the direction to project. In this case, it is the Z-direction.
- **Ignore (incorrectly) the warning about creating fields from overlapping surfaces, of 2D elements (as seen looking in the Z-direction).**



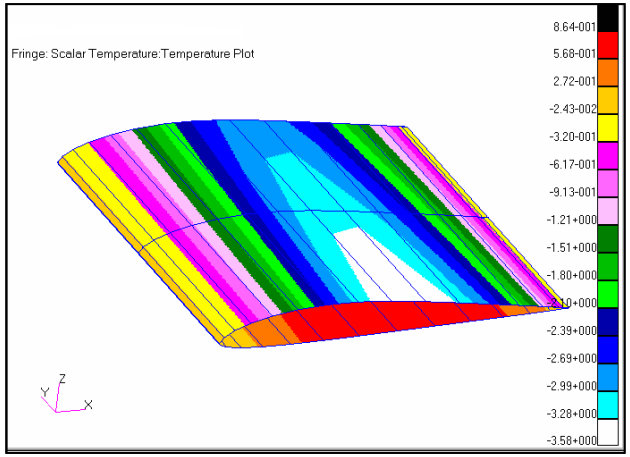
Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, CREATE FEM FIELD, CONTINUOUS

- **Looking ahead, the consequences of ignoring the surface overlap can be seen.**
  - Notice that the pressure distribution for the top and bottom surfaces is similar.This is the first visual clue that something is wrong.



Field, and subsequently, pressure LBC made incorrectly from a temperature contour.

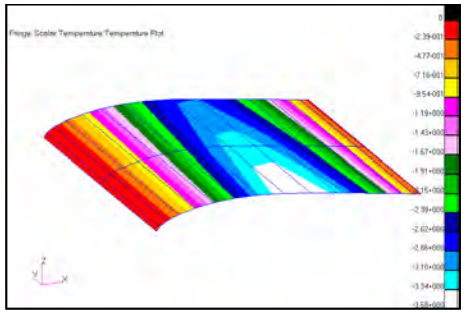


Original Temperature Contour

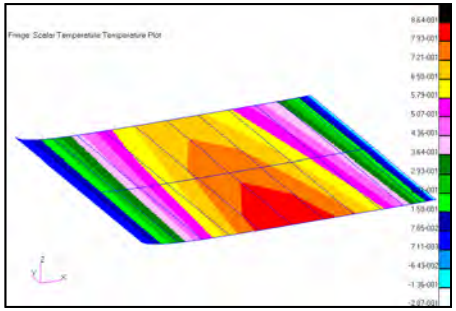
Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, CREATE FEM FIELD, CONTINUOUS

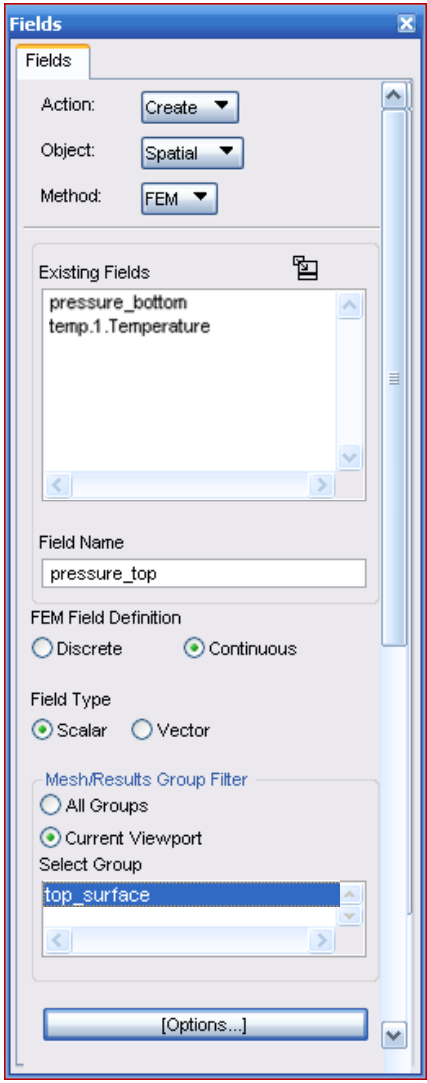
- **To make the FEM Field correctly, simply put the top and bottom surfaces (of elements) in separate groups.**
  - Post only one of the two groups, then create an FEM Field for it. Repeat this for the other group.
  - The form to the right shows that the FEM field, “pressure_bottom”, was created, and “pressure_top” is ready for creation using the top surface of elements.
- **Again, make sure that the 2D to 3D interpolation under Options is enabled.**



Top Surface



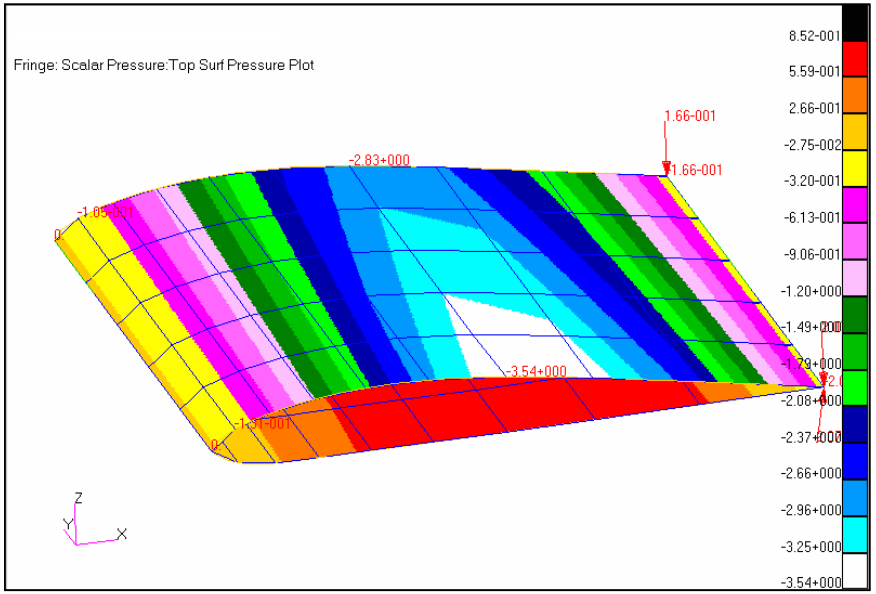
Bottom Surface

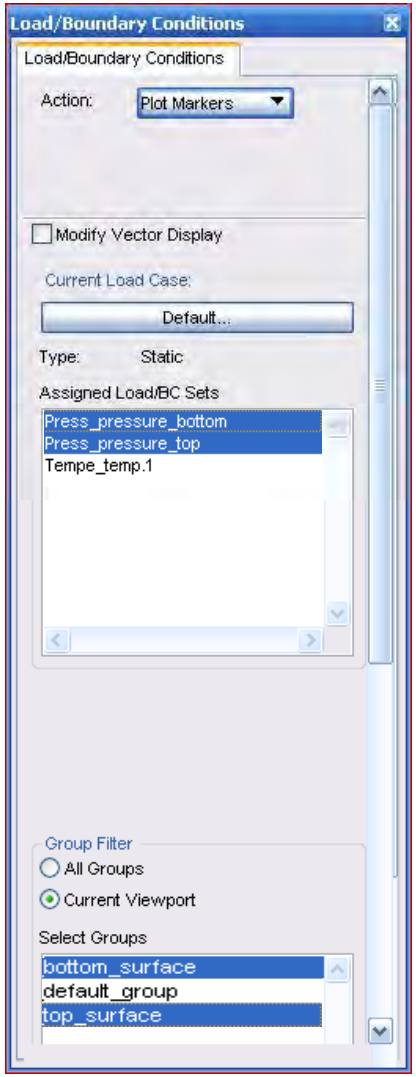


Copyright© 2012 MSC.Software Corporation

CASE STUDY 6, CREATE FEM FIELD, CONTINUOUS

- **Create the pressure LBCs using the two FEM Fields.**
  - Loads/BCs: Create/Pressure/Element Uniform
  - Input Data
    - Select one FEM Field (e.g. pressure_top)
  - Application Region
    - Select the corresponding surface (e.g. top surface (of elements))





Copyright© 2012 MSC.Software Corporation

EXERCISES

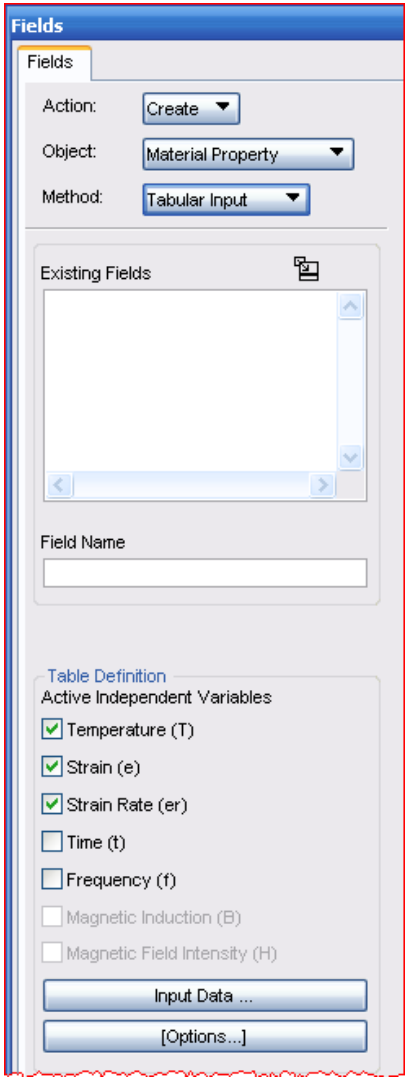
- Perform Workshop 19 “Global/Local Modeling Using FEM Fields” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

FIELDS, MATERIAL PROPERTY/TABULAR INPUT

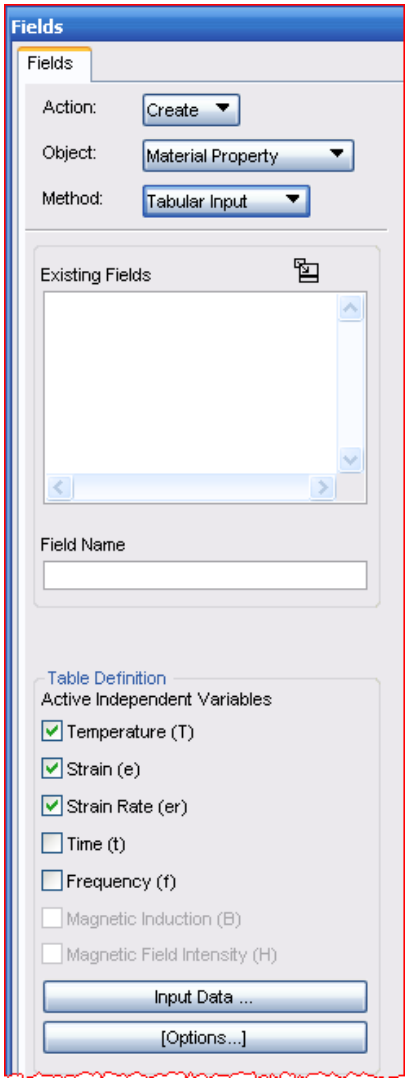
- **This type of field is used to create varying material properties using tabular input.**
- **The independent variable(s) can be chosen to appropriately describe the material behavior.**
  - Any combination of the three variables Temperature, Strain, or Strain Rate can be used for a field. If the variable Time or Frequency is used, only one of them can be used, and none of the other three variables can be used.

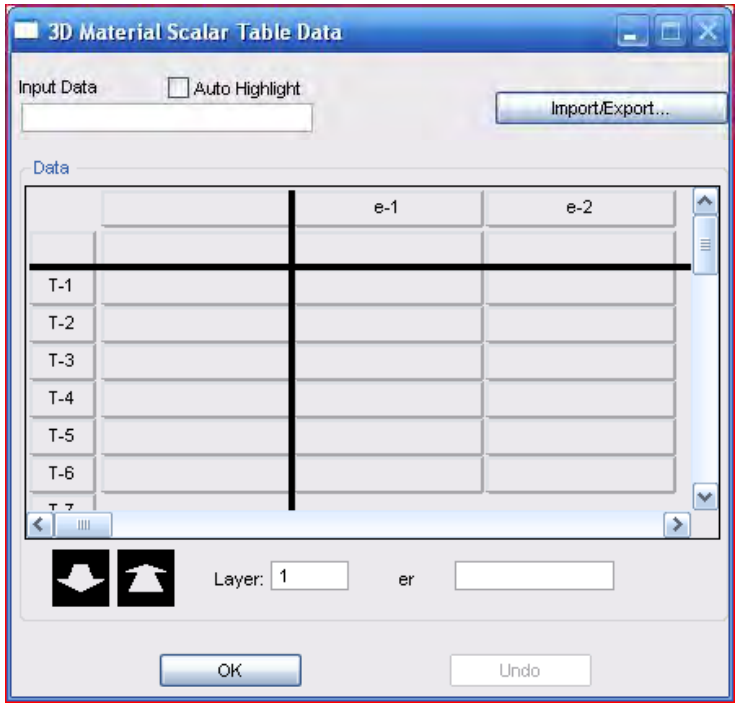


Copyright© 2012 MSC.Software Corporation

FIELDS, MATERIAL PROPERTY/TABULAR INPUT

- **The number of variables selected determines whether a one-, two-, or three-dimensional table for input will be displayed.**

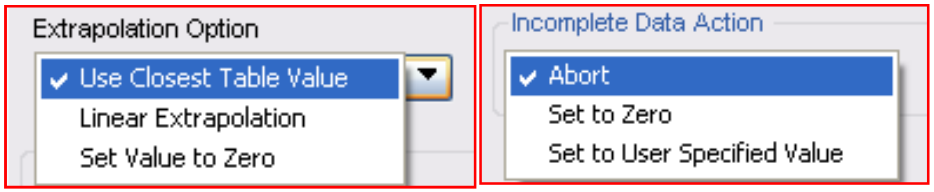


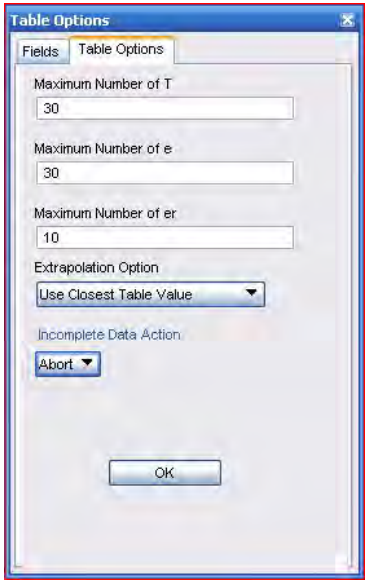


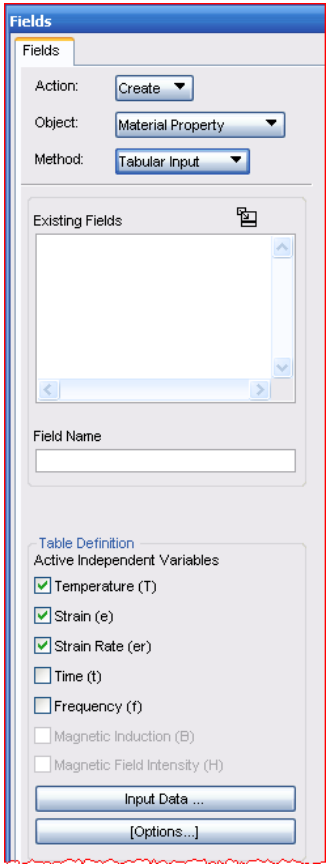
Copyright© 2012 MSC.Software Corporation

FIELDS, MATERIAL PROPERTY/TABULAR INPUT, OPTIONS

- **The top portion of the Tabular Input, [Options] form controls how many data points can be input for each variable (e.g. Temperature).**
- **Extrapolation Option includes the following choices:**
  - Use Closest Table Value
  - Linear Extrapolation
  - Set Value to Zero
- **Incomplete Data Action allows the selection of how not having adequate data will be dealt with**



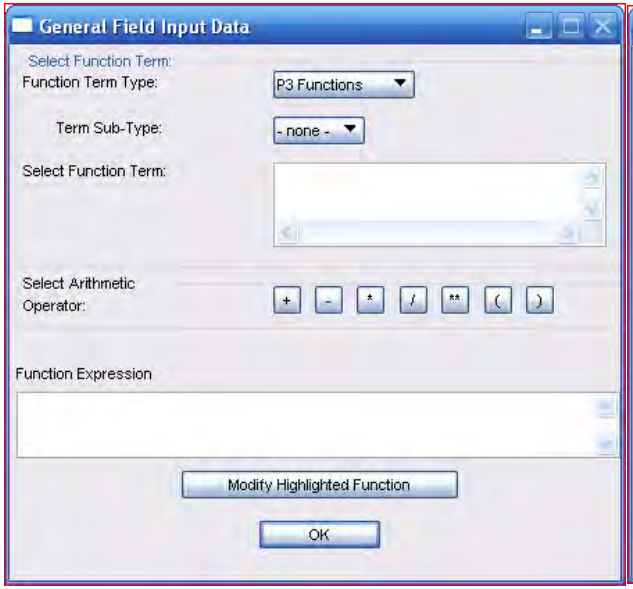


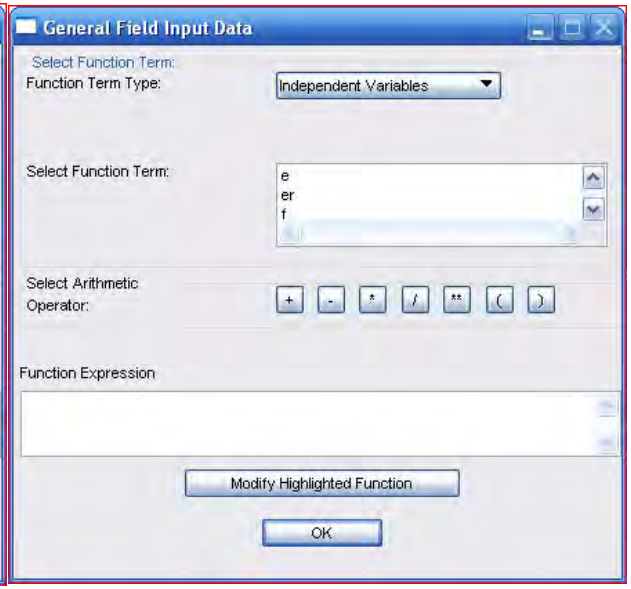


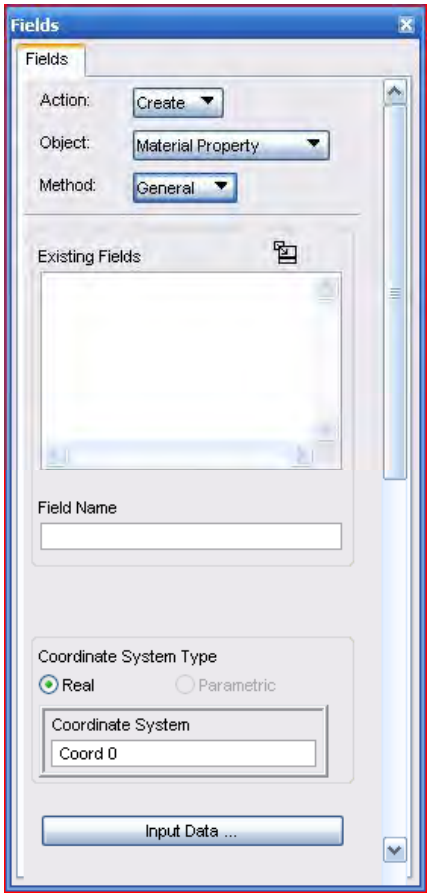
Copyright© 2012 MSC.Software Corporation

FIELDS, MATERIAL PROPERTY/GENERAL

- **Material Properties/General allows the cross reference of any material property to any other property or a user defined equation.**
  - With **Function Term Type** set to **P3 Functions**, an original function will be created.
  - With **Function Term Type** set to **Independent Variables**, predefined functions (e.g. er) can be picked and included in the equation.



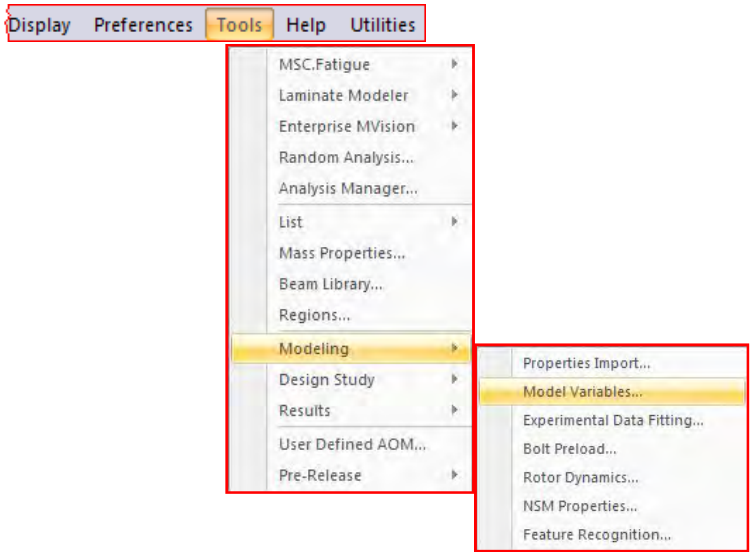


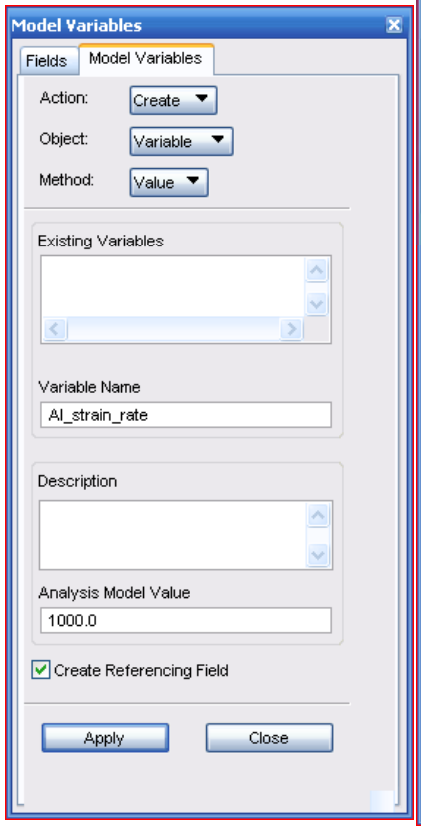


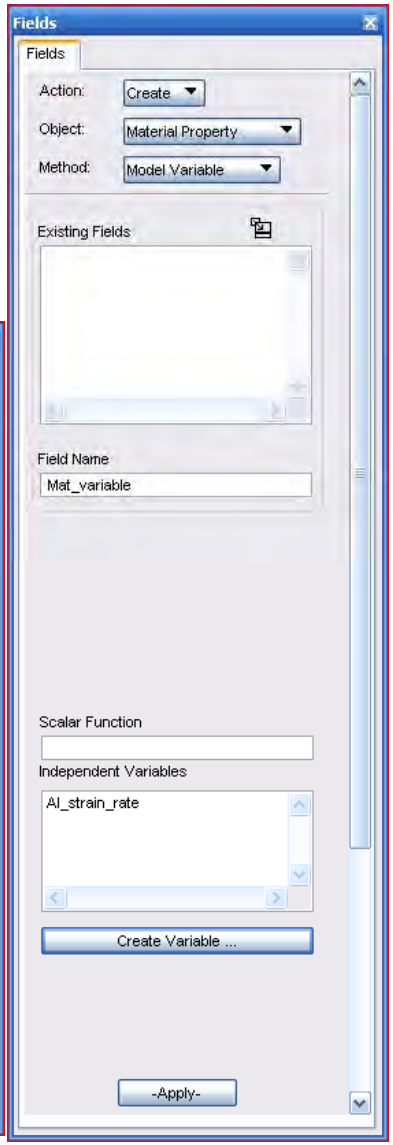
Copyright© 2012 MSC.Software Corporation

FIELDS, MATERIAL PROPERTY/MODEL VARIABLE

- Material Property/Model Variable is intended for creating fields from user defined variables.
- To create, modify, show, or delete the variables for Material Property fields, select Create/Variable
- The variables can also be accessed under Tools: Model Variables from the Patran main menu.



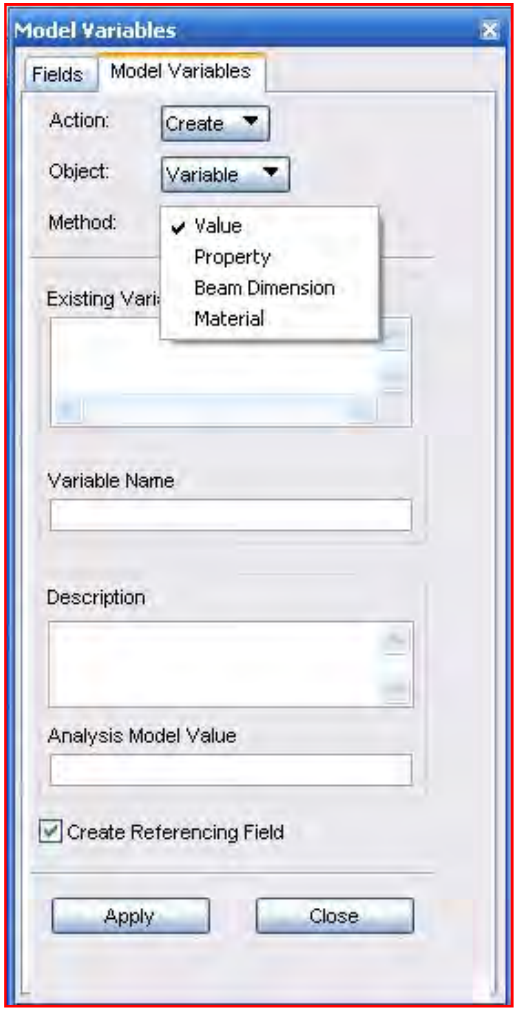




Copyright© 2012 MSC.Software Corporation

FIELDS MODEL VARIABLE

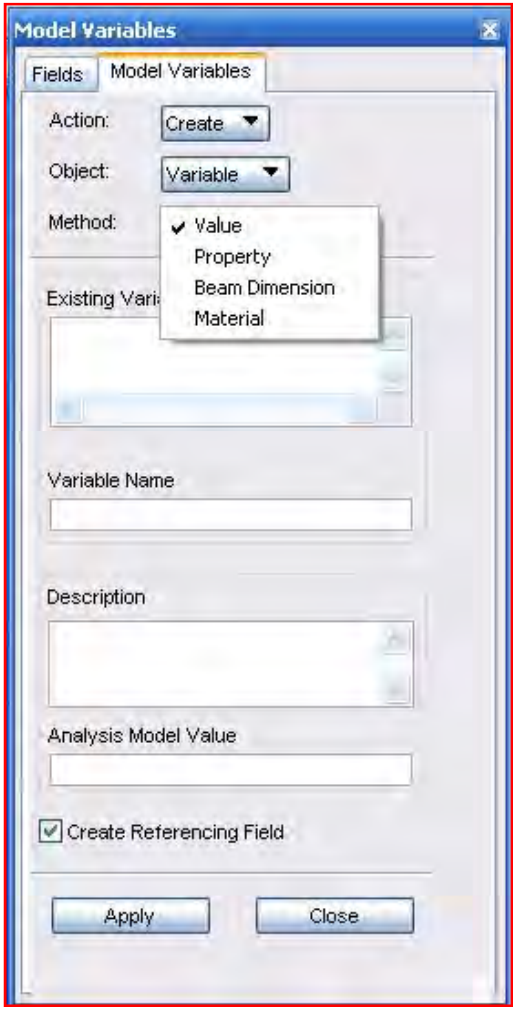
- **Model variables are single value parameters (constants).**
- **The value can be specified either by specifying a numerical value or extracting one from existing Property, Beam Dimension or Material.**
  - If the value is extracted, the original numerical value is replaced with a variable name.
  - Once the variable is created, it can be changed or checked under Modify/Variable or Show/Variable.
- **Once variables are used in LBCs, materials, or element properties, they can be modified; they can be used in parametric studies.**



Copyright© 2012 MSC.Software Corporation

FIELDS MODEL VARIABLE

- **Create/Variable/Value has an additional option to subsequently create a Field.**
  - If **Create Referencing Field** is checked, then, a Field that references the variable will be created with the same name as the variable. The Field is pickable from Load/BCs, Properties and Properties/Beam Library.
    - For the field to appear in Material, the variable has been created under Variable/Material
  - If the **Create Referencing Field** is un-checked, then a field will not be created automatically.
    - A Field created with that variable can still be used in any part of Patran where it is appropriate (e.g. Loads/BCs).

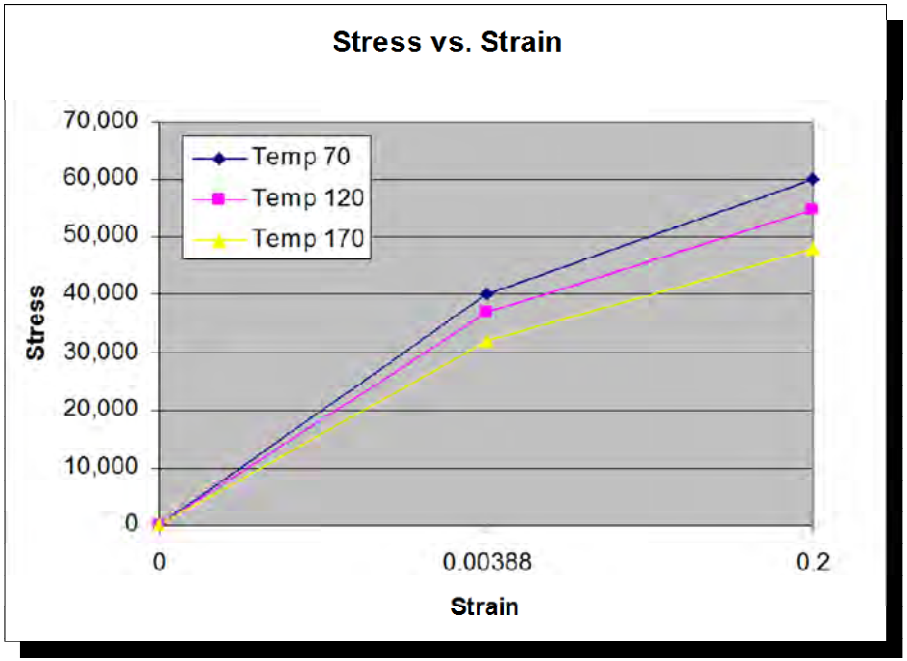


Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

CASE STUDY 7, MATERIAL PROPERTY FIELD

- **Create a single field that describes an Aluminum alloy over a temperature and strain domain.**
- **Use the table data provided**



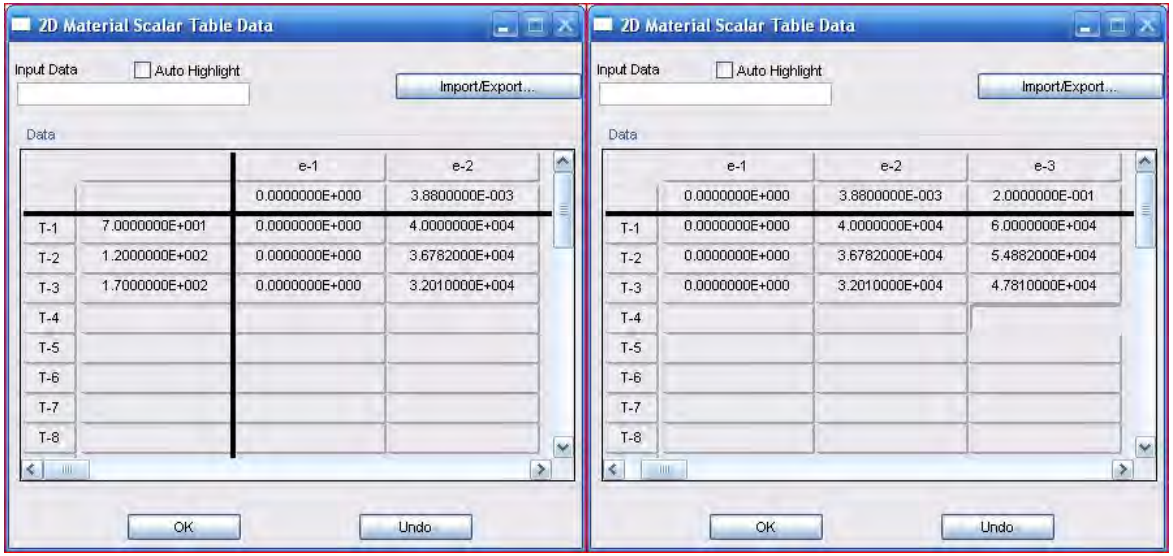
Stress vs. Temperature and Strain

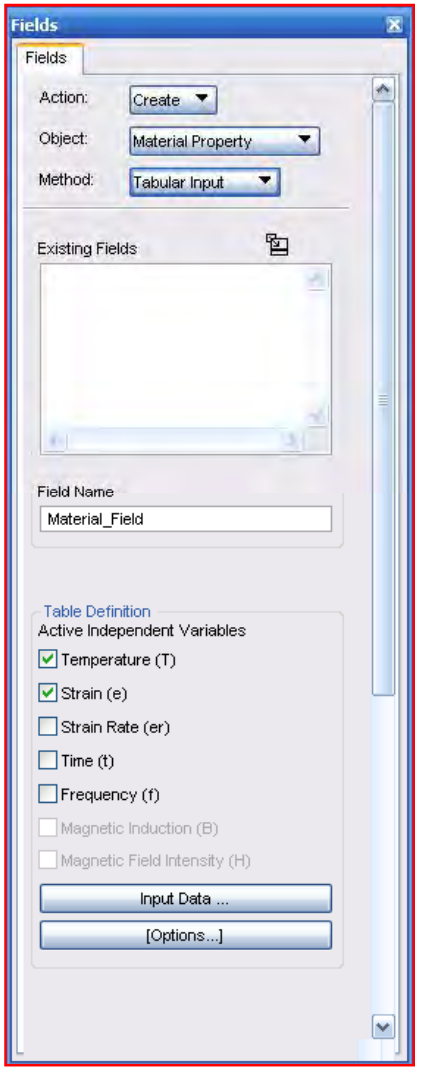
		Strain	
Temp (F)	0	0.00388	0.2
70	0	40,000	60,000
120	0	36,782	54,882
170	0	32,010	47,810

Copyright© 2012 MSC.Software Corporation

CASE STUDY 7, MATERIAL PROPERTY FIELD

- **To create the field, select Temperature and Strain for Active Independent Variables.**
- **Enter the data in the Input Data form**
  - The first column are the temperatures (rows 1,2,3).
  - The first row contains the strains (columns 1,2,3).
  - The table entries are the stresses for the corresponding temperatures and strains.

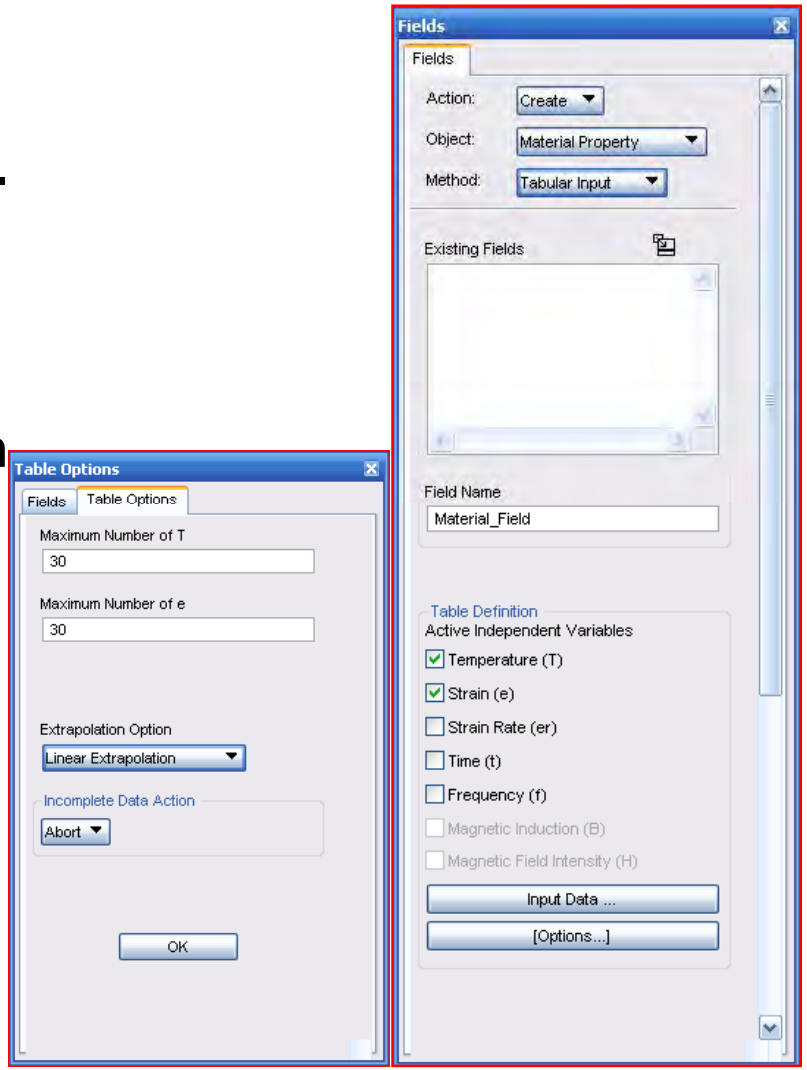




Copyright© 2012 MSC.Software Corporation

CASE STUDY 7, MATERIAL PROPERTY FIELD

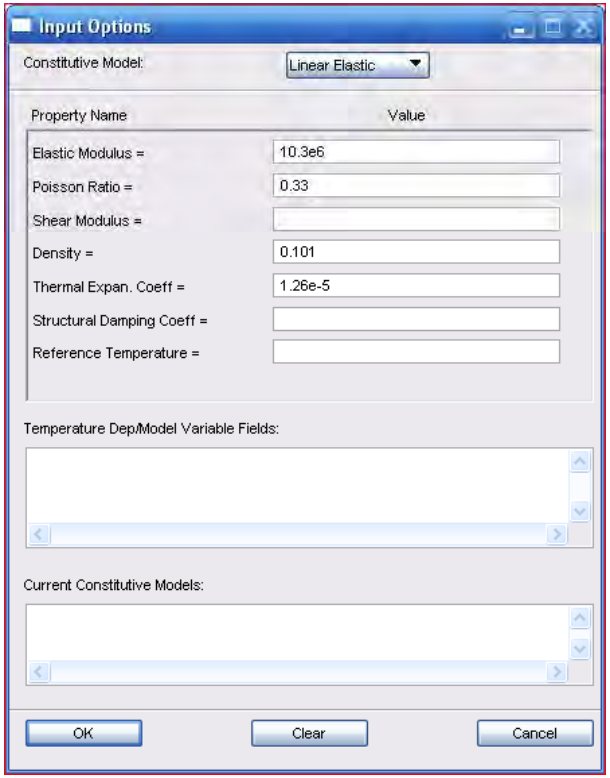
- **In [Options] the Extrapolation Option chosen does not affect the field evaluation for the Nastran preference. Nastran will do a linear interpolation between adjacent (strain,stress) points.**
- **The number of rows and columns can be specified, but for this case, 30 is sufficient.**

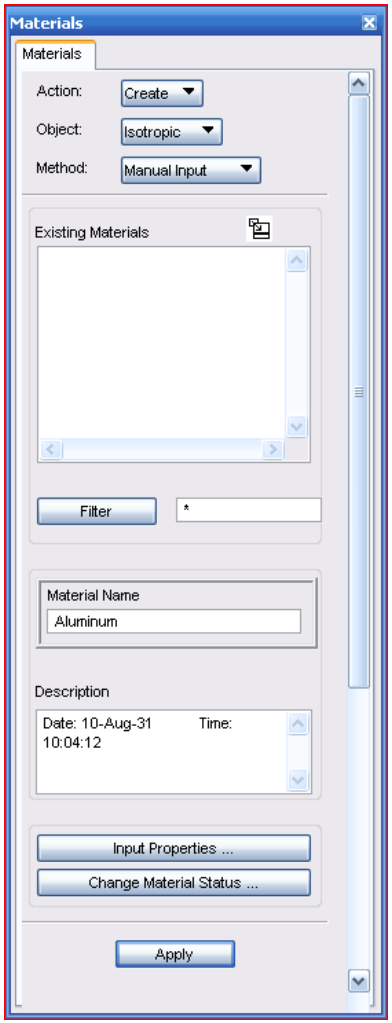


Copyright© 2012 MSC.Software Corporation

CASE STUDY 7, CREATE MATERIAL USING FIELD

- **In Materials, Input Properties there are two steps for making the nonlinear material.**
  - With Constitutive Model set to **Linear Elastic**, the linear properties can be specified.
- **Although not always required, many times the elastic modulus for the linear elastic constitutive model is set to match the first elastic modulus of the data used to create the stress/strain field.**
  - For Nastran, consult the Quick Reference Guide for more details.

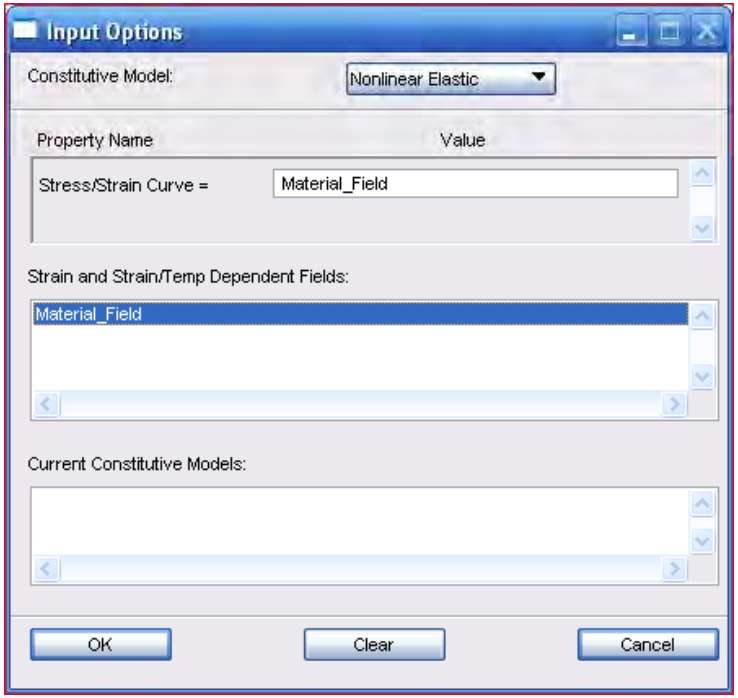


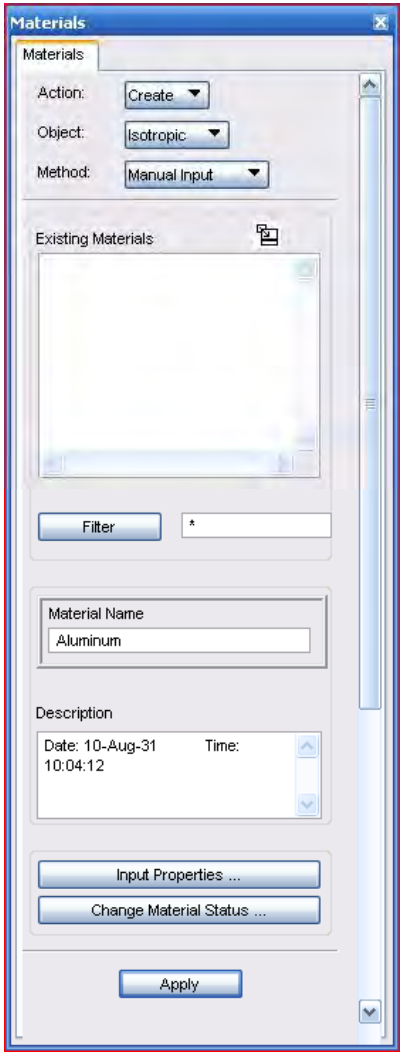


Copyright© 2012 MSC.Software Corporation

CASE STUDY 7, CREATE MATERIAL USING FIELD

- Set the **Constitutive Model** to **Nonlinear Elastic**, and select the previously created field from the **Strain and Strain/TEMP Dependent Fields**, Material_Field for the nonlinear properties of Aluminum.



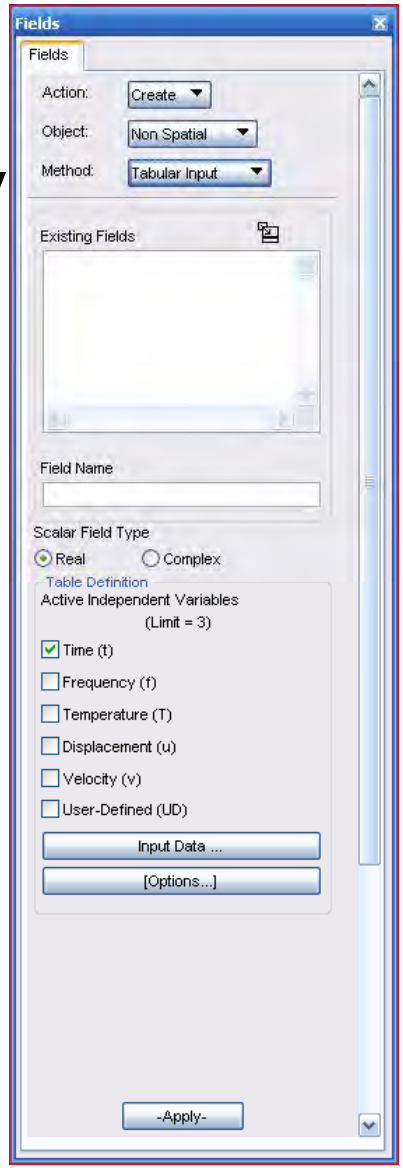


Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

FIELDS, NON SPATIAL/TABULAR INPUT

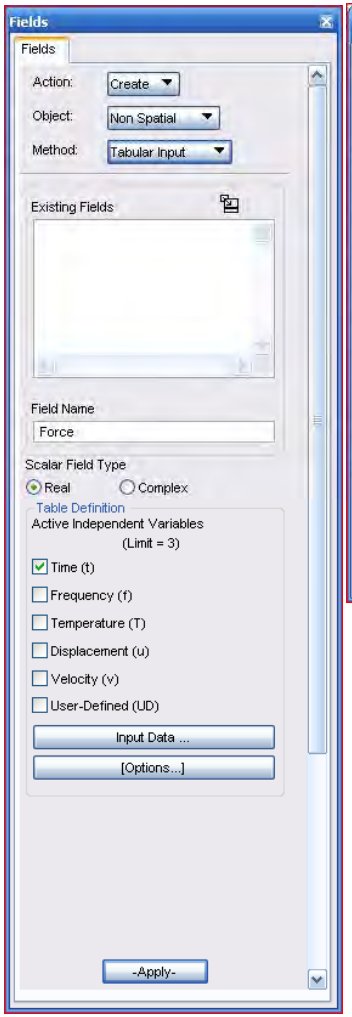
- **Non Spatial, Real fields are used to create time or frequency dependent fields for transient or frequency response analysis.**
- **Non Spatial, Complex fields are used primarily for frequency response analysis.**

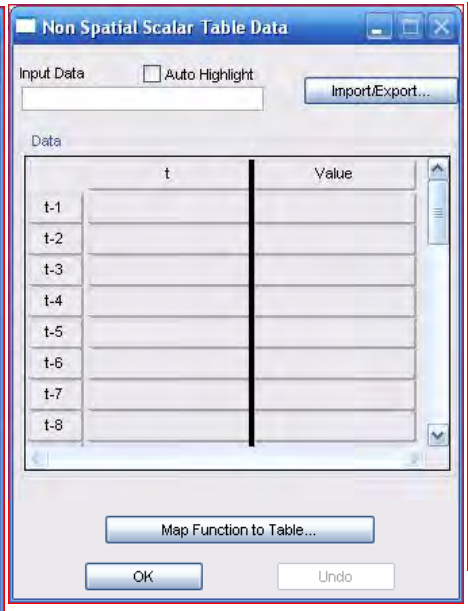


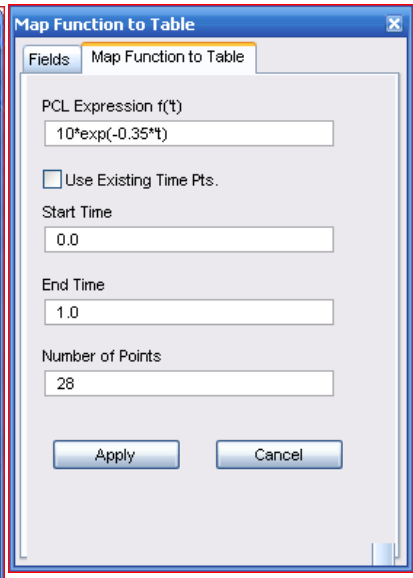
Copyright© 2012 MSC.Software Corporation

FIELDS, NON SPATIAL/TABULAR INPUT

- **Create non-spatial field**
  - Real
  - Time
  - Map Function to Table
  - PCL Expression f(time)



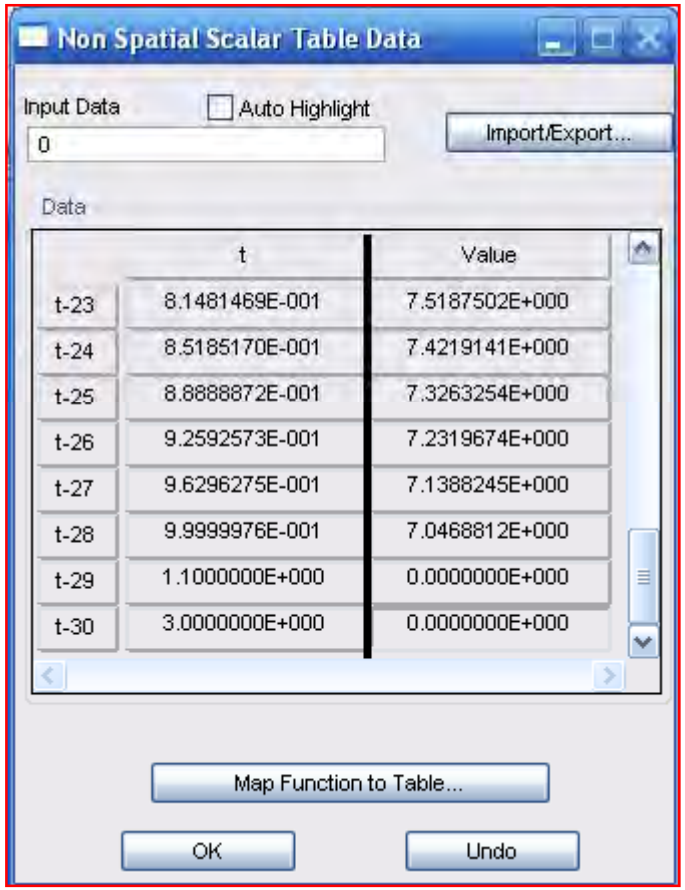




Copyright© 2012 MSC.Software Corporation

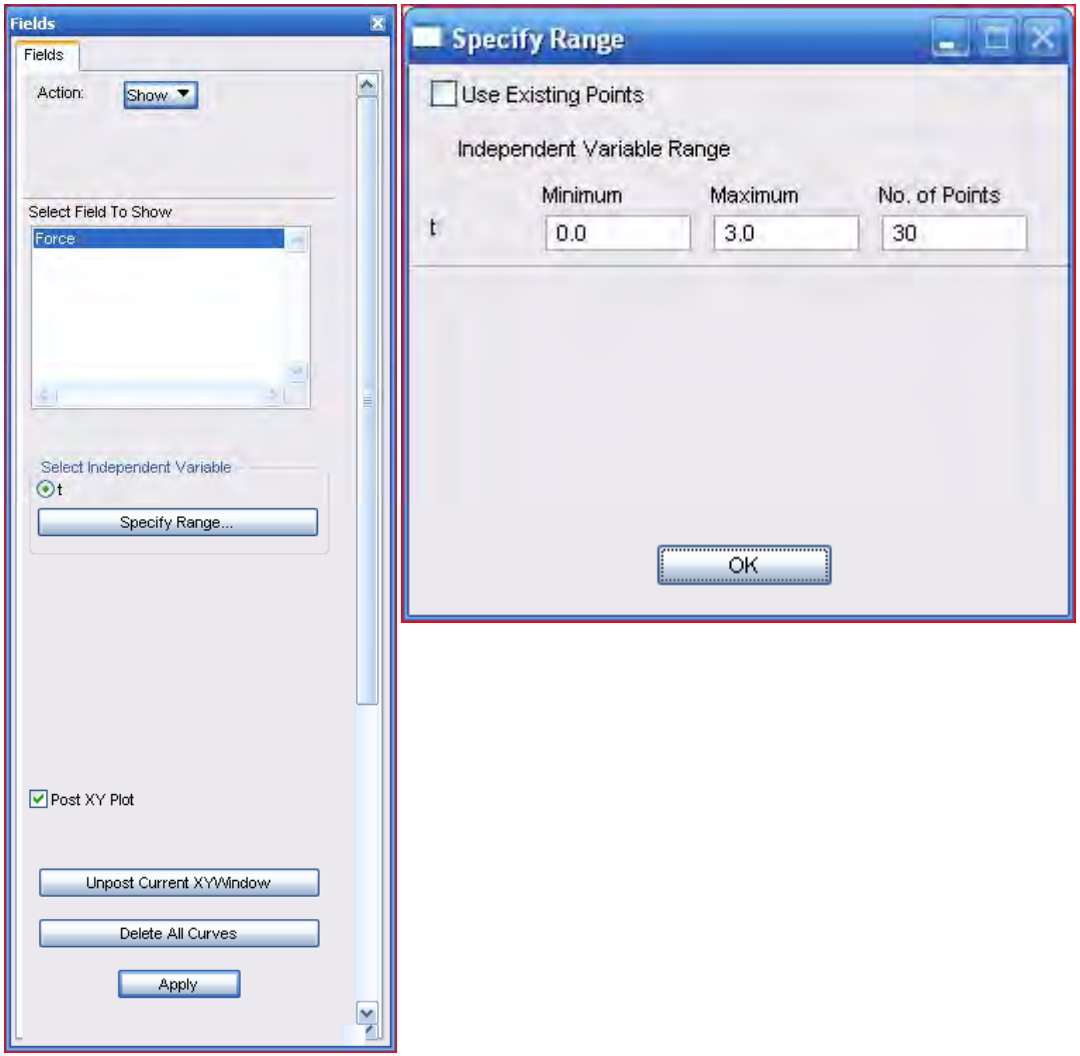
FIELDS, NON SPATIAL/TABULAR INPUT

- **Edit the columns to include two additional points (Time, Value)**



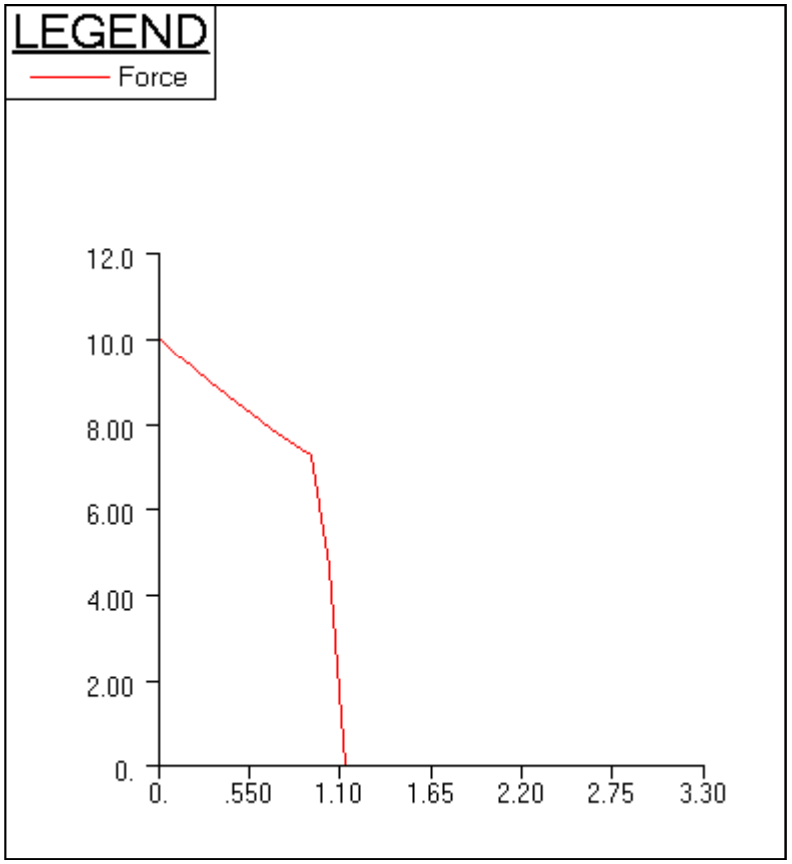
Copyright© 2012 MSC.Software Corporation

FIELDS, NON SPATIAL/TABULAR INPUT



Copyright© 2012 MSC.Software Corporation

FIELDS, NON SPATIAL/TABULAR INPUT



Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

SECTION 9

PCL FOR NON-PROGRAMMERS

Copyright© 2012 MSC.Software Corporation

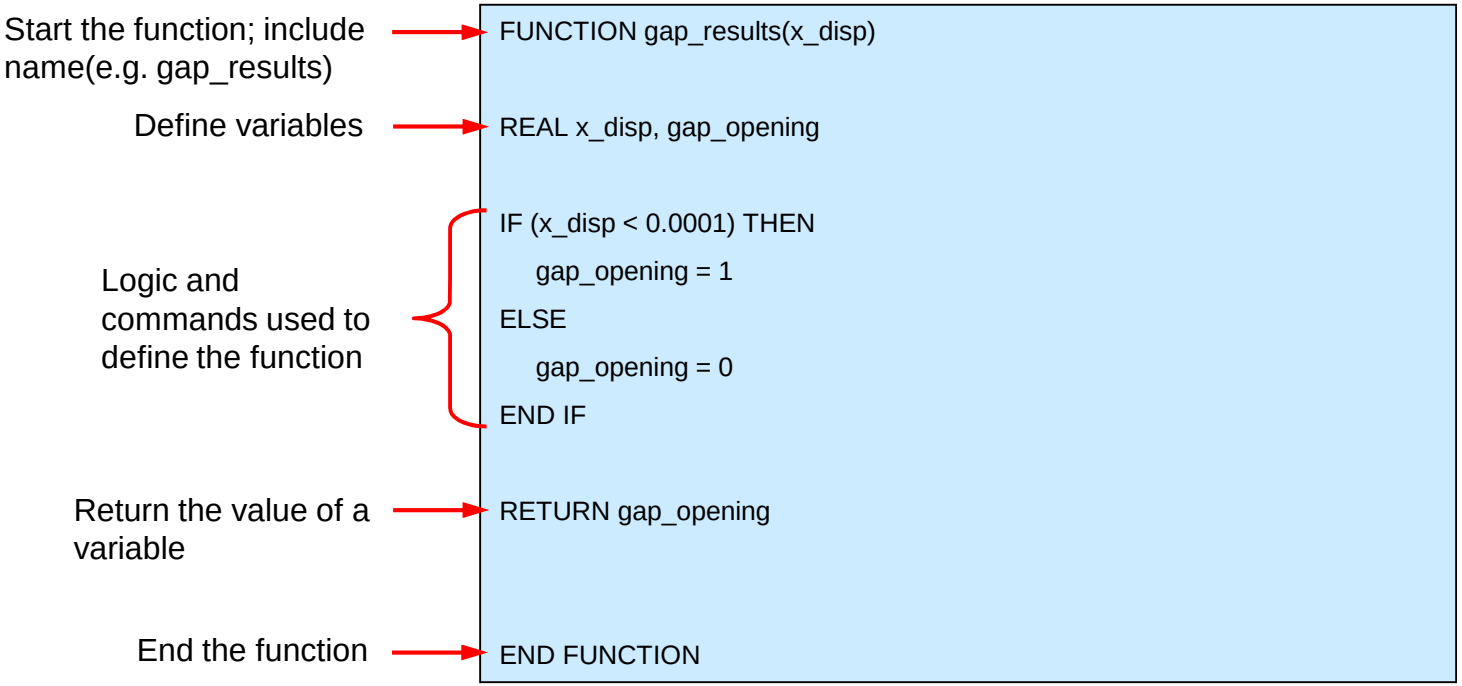
OVERVIEW OF PCL

- **What is PCL ?**
  - PCL is an acronym for Patran Command Language.
  - It is a fully functional programming language, i.e. IF…THEN, FOR…
  - Users can create functions in Patran.
  - Functions can perform multiple tasks, and can eliminate repetitive steps.
- **How to use PCL ?**
  - Create a PCL function.
  - Compile the PCL function in either Patran or in the external PCL compiler.
  - Once the PCL function is compiled, it can be executed in Patran using the command window (line), or by including it in a library and have that loaded during the start of Patran.

Copyright© 2012 MSC.Software Corporation

PCL BASICS

- **Common PCL Function layout:**



Copyright© 2012 MSC.Software Corporation

PCL BASICS

- **Operators commonly used in PCL programming**

Operators						Definitions
+	-	!				Unary Plus or Minus, Logical Not
**						Exponentiation
*	/					Multiplication and Division
+	-					Addition or Subtraction
//						String Concatenation
<	>	<=	>=	==	!=	Relational Operators
||	&&					Logical Or, Logical And
+=	-=	=				Increment, Decrement, Assignment

Copyright© 2012 MSC.Software Corporation

PCL BASICS

- **Steps to create PCL Functions**
  - Declare the function
    - Use the function declaration, FUNCTION function_name (variable1,variable2, …)
  - Define the variables
    - All the variables used in a PCL function have to be defined, i.e. INTEGER, REAL, etc.
    - Can set initial value of the variables.
    - If a variable is not assigned a value, it is automatically set to the default value, which is either null or zero depending on its type (e.g. LOGICAL).
  - Specify the PCL expressions
    - This is where the tasks of the PCL function are defined.
  - Obtain the results
  - Declare end of the function
    - At the end of the PCL function, the END FUNCTION statement must be used to terminate it.

Copyright© 2012 MSC.Software Corporation

VARIABLE DECLARATION

- **PCL function variable types**
  - REAL variable1, variable2, variable3, …
    - This declares the value of the indicated variables as real numbers, which can be from 1.0E-30 to 1.0E+30.
  - INTEGER variable1, variable2, variable3, …
    - Integers can have a value from –2147483648 and 2147483648.
  - LOGICAL logical1, logical2
    - The variables can have the value true, false, yes, or no.
  - STRING string1[80], string2[100]
    - STRING data type is alphanumeric (characters).
    - The variable name needs to have [#] at its end to indicate how many characters the variable can store.
    - The characters have to be surrounded by double quotes, e.g. “abcdef”.

Copyright© 2012 MSC.Software Corporation

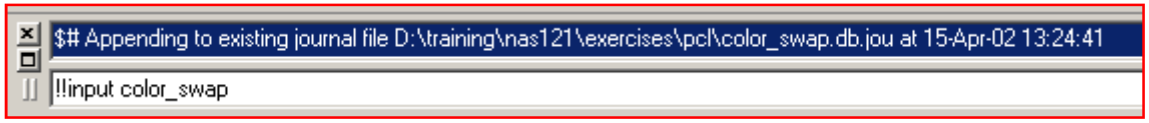
DEFINE THE PCL EXPRESSIONS

- **An expression can be a simple mathematical function, such as**
  - sum = variable1 + variable2 * variable3
- **PCL expressions can be logical statements, such as IF… THEN, or looping functions FOR… or WHILE…**
- **PCL expressions can be written directly in Patran's command line, and executed from there. However, they are generally written in a text file, with subsequent compilation using the file.**
- **In addition, it is possible to use PCL expressions in an Patran session file by "playing" the session file.**

Copyright© 2012 MSC.Software Corporation

COMPILE THE PCL FUNCTION

- **Compile and execute PCL functions directly in Patran:**
  - To compile a function, enter the following command in the command window (line) of Patran:
    - !!input pcl_function_filename.pcl
      - Example:



  - To execute the function in Patran, enter the following command in the command window:
    - pcl_function_filename (variable1, variable2, variable3, …)
      - Example:

Copyright© 2012 MSC.Software Corporation

COMPILE THE PCL FUNCTION

- **Alternatively, it is possible to compile PCL functions in the command prompt (e.g. C:\>) environment.**
  - At the command prompt, execute p3pclcomp.exe to enter the PCL environment.
  - The commands for this environment are same as those in the Patran environment.

Copyright© 2012 MSC.Software Corporation

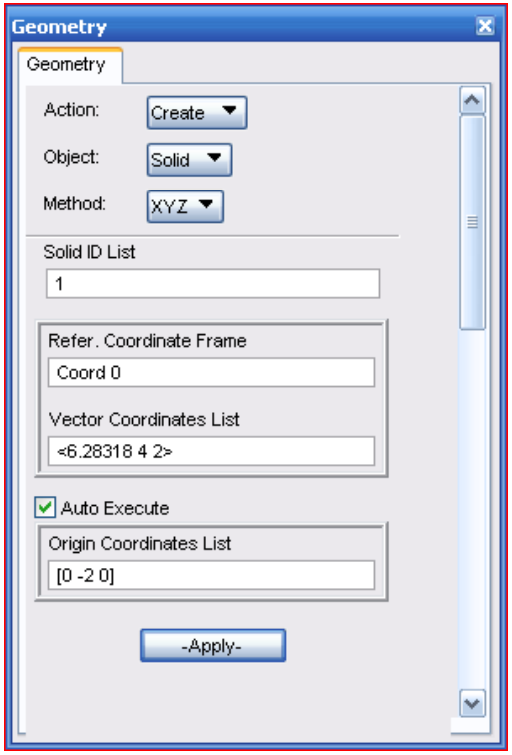
COMPILE THE PCL FUNCTION

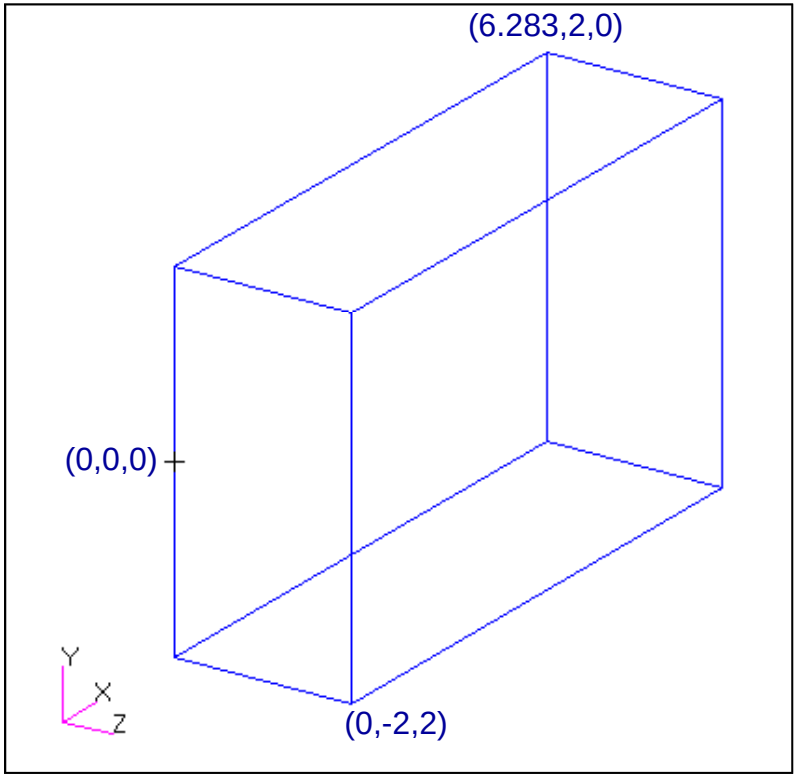
- **It is possible to generate a library of PCL functions and have that library loaded during the start of Patran.**
- **To create a library, use the following command:**
  - !!library create library_name.plb
- **To add a PCL function to a library, use the following command to compile, then assign it:**
  - !!compile pcl_function_filename.pcl library_name.plb
- **To load a library during the start of Patran, the following line needs to be added to the p3epilog.pcl file:**
  - sys_library(“ADD”, “library_name.plb”)

Copyright© 2012 MSC.Software Corporation

CASE STUDY: CHANGE BLACK TO WHITE

- **Create the model that the PCL function will be used to create the pressure load.**

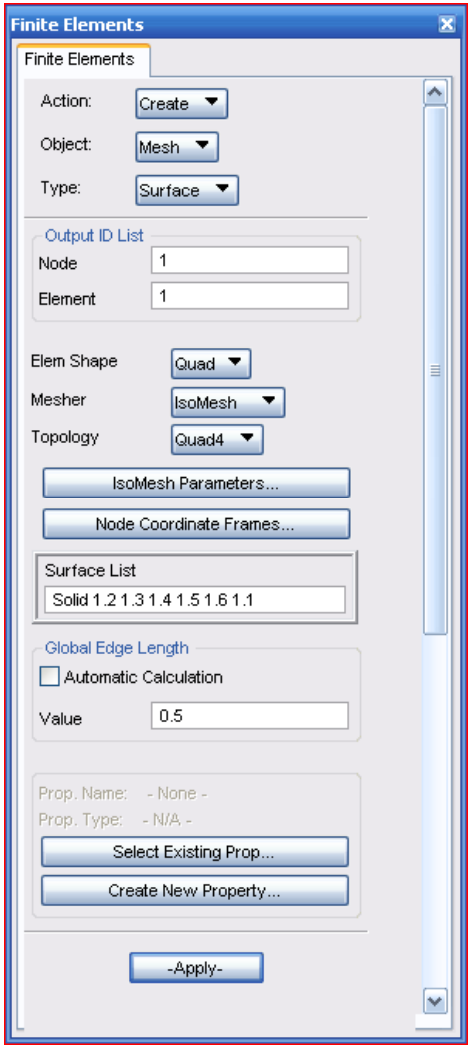


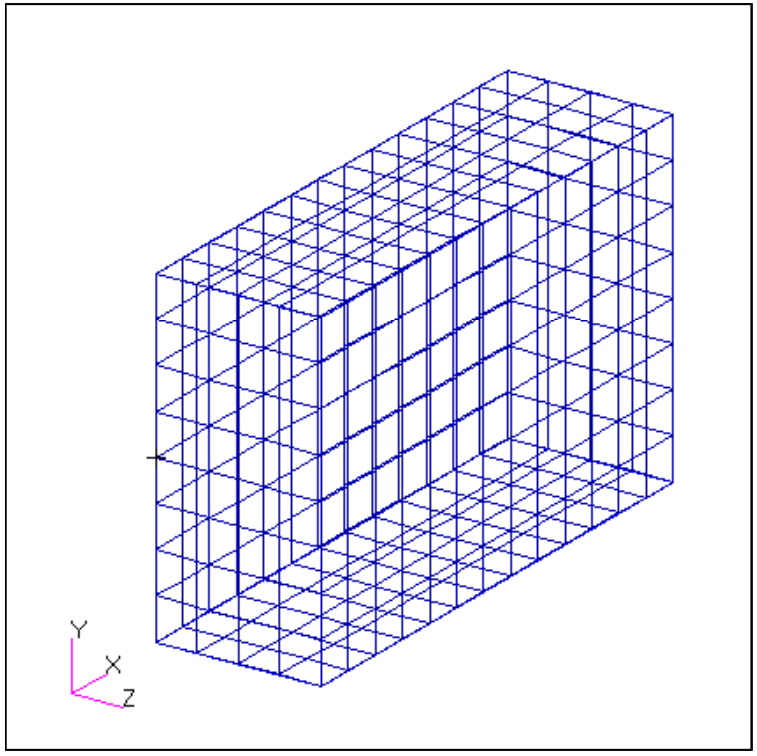


Copyright© 2012 MSC.Software Corporation

CASE STUDY: CHANGE BLACK TO WHITE

- **Create the model that the PCL function will be used to create the pressure load.**





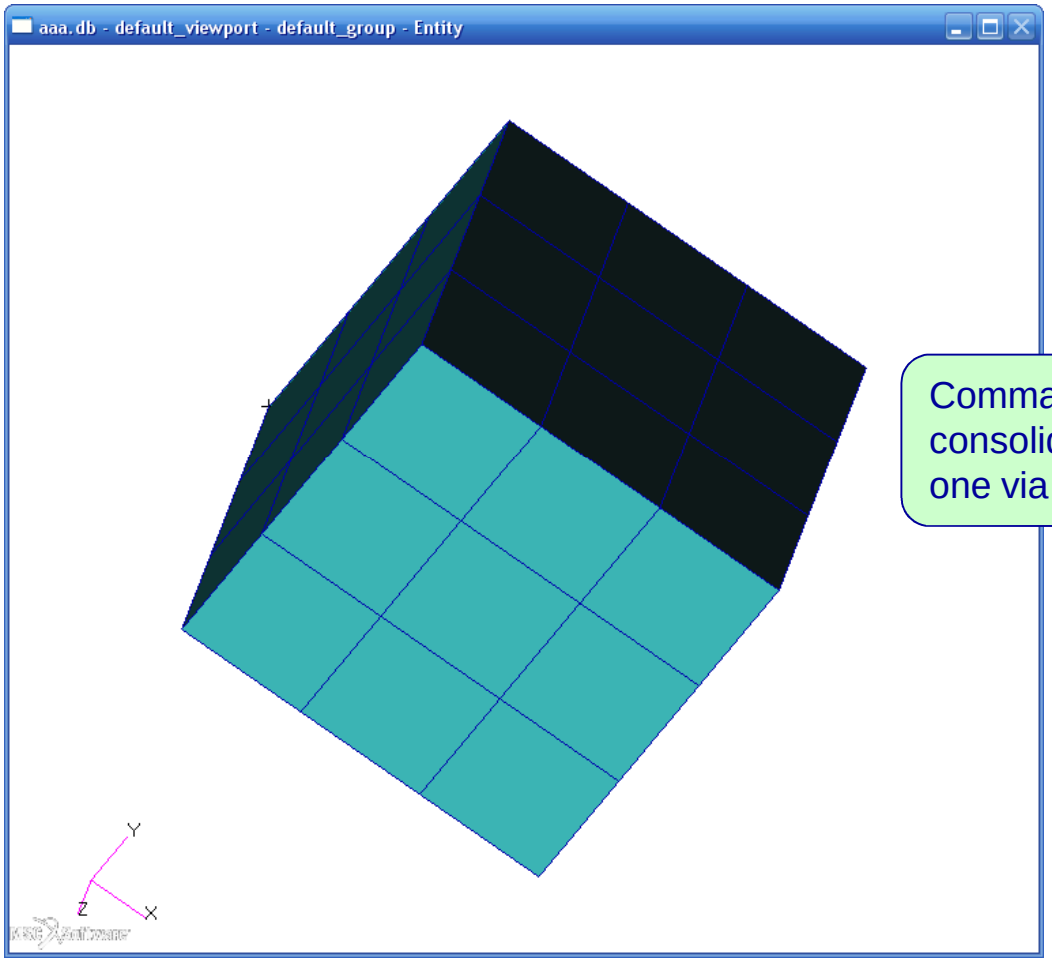
Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW

- **At times, it will be convenient to have a custom PCL Function that invokes a particular view, render style, and display of only the entities you want from one command.**
  - For this example, build a 1x1x1 solid, mesh it with Hex elements.
    - Then orient the view to desired.
    - Show it in Smooth Shaded mode.
    - Then Erase the geometry to show only elements.
- **From these steps, you can create a PCL function and use it to perform the task from one command instead of the multiple steps.**
- **The simplest method to create such a function is to perform the above tasks in a new Patran database, and then extract the function representing the above tasks from the session file.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW



- **Manual steps**
  - Create a new Patran database.
  - Build a 1x1x1 solid
  - Mesh it with Hex elements.

Commands to consolidate as one via PCL

- Then orient the view to desired.
- Show it in Smooth Shaded mode.
- Then Erase the geometry to show only elements.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW

- **Create PCL function from session file**
  - Exit Patran and open the session file
  - Find the PCL commands indicated (*) and copy into a new text file

Command to set the display to Smooth Shaded

Commands to orient the model in the view

Command to erase the geometry

```
STRING ids[VIRTUAL]
asm_const_hpat_xyz("1","<1 1 1>","[0 0 0]","Coord 0",ids)
$# 1 Hpat created: Hpat 1
uil_toolbar.shaded_smooth(  )
ui_exec_function( "mesh_seed_display_mgr", "init" )
INTEGER fem_create_mesh_solid_num_nodes
INTEGER fem_create_mesh_solid_num_elems
STRING fem_create_mesh_s_nodes_created[VIRTUAL]
STRING fem_create_mesh_s_elems_created[VIRTUAL]
fem_create_mesh_sol_5( "Solid 1", "IsoMesh", "Hex8", 1, ["0.399999"], 49152,  @
0, 1, 0, 1, 0., "", "#", "#", "Coord 0", "Coord 0",  @
fem_create_mesh_solid_num_nodes, fem_create_mesh_solid_num_elems,  @
fem_create_mesh_s_nodes_created, fem_create_mesh_s_elems_created )
$# 64 nodes and 27 elements created for Solid 1.
$# === 64 nodes created.  IDs  = 1:64.
$# === 27 elements created.  IDs  = 1:27.
ga_view_zoom_set( 0.77079839 )
ga_view_aa_set( 31.268990, -11.446634, -33.926395 )
gu_fit_view(  )
gm_plot_erase_geometry( 0 )
```


Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW

- **Create PCL function from session file (Cont.)**
  - Add two entries in the text file to create a PCL function
  - Save the file, calling it my_view.pcl, in your working directory

Add this line →

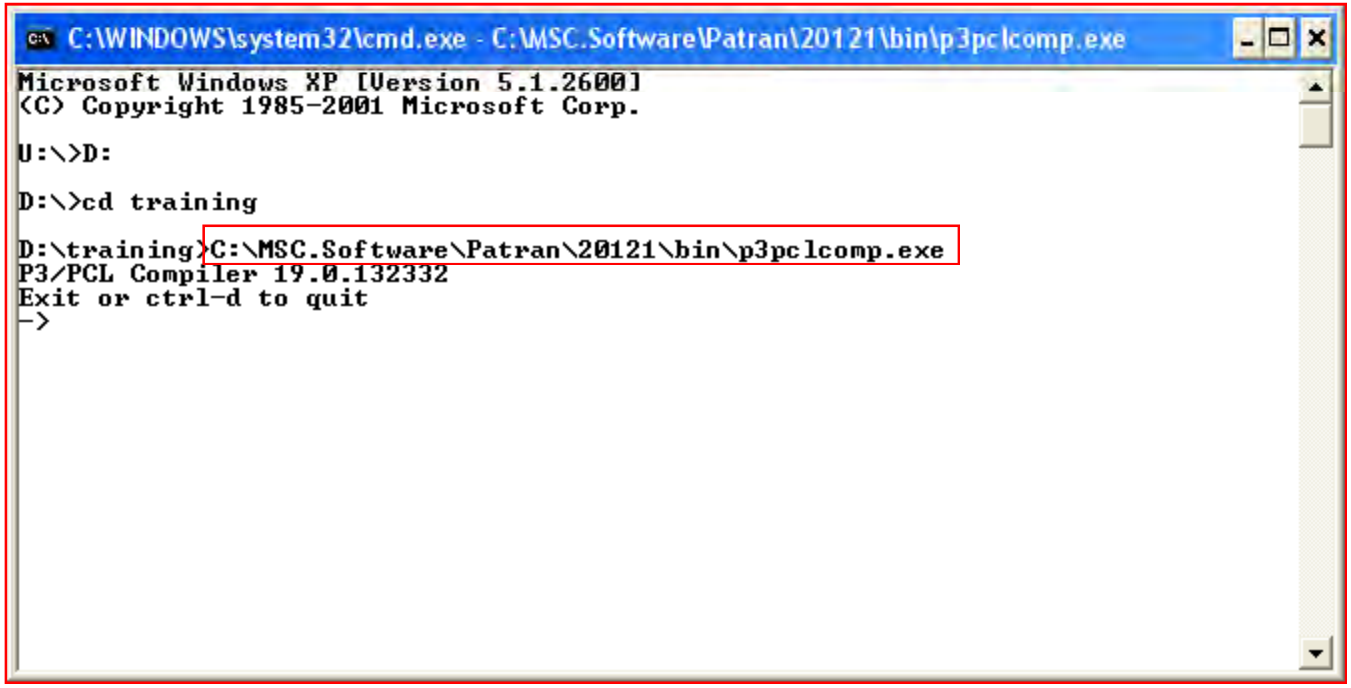
Add this line →

```
FUNCTION my_view()

uil_toolbar.shaded_smooth(  )
ga_view_zoom_set( 0.77079839 )
ga_view_aa_set( 31.268990, -11.446634, -33.926395 )
gu_fit_view(  )
gm_plot_erase_geometry( 0 )

END FUNCTION
```


Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW

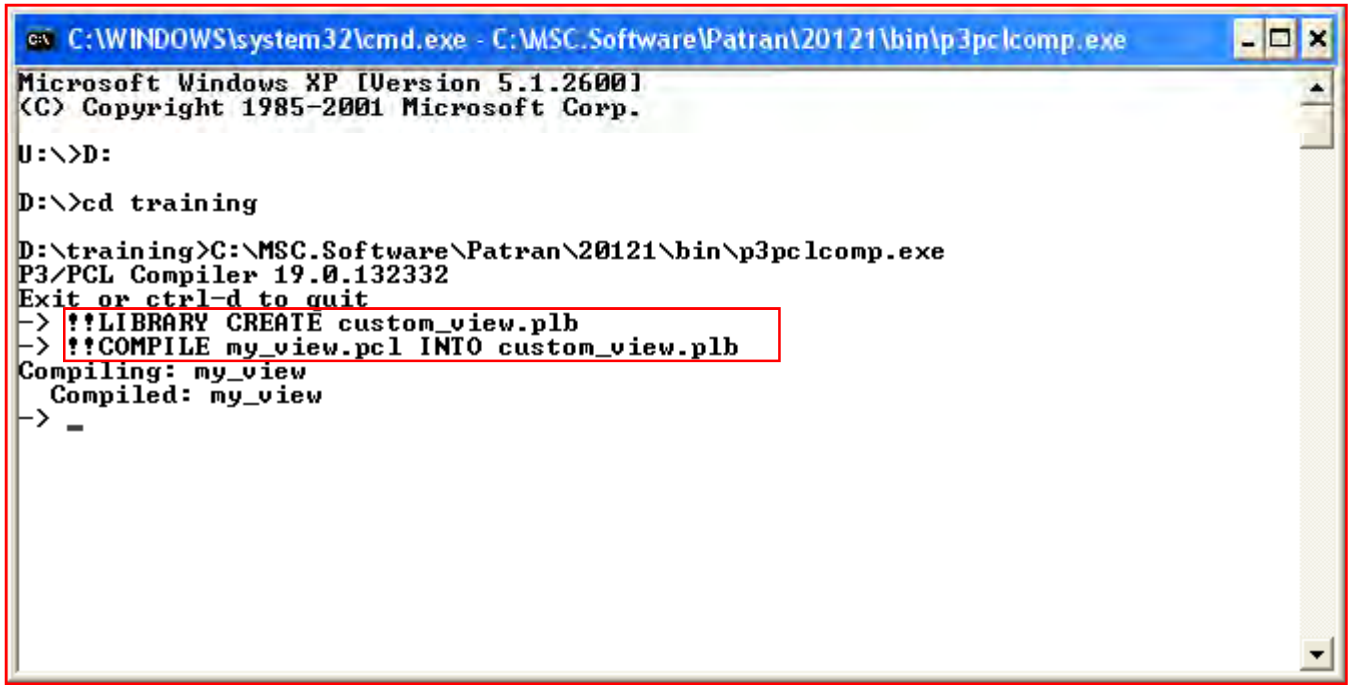
- **To enable the use of this function for any Patran session, compile and load it into a library.**
- **Do these tasks in the PCL compiler environment.**
  - Execute p3pclcomp.exe from a DOS prompt in the working directory that contains the my_view.pcl file



Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW

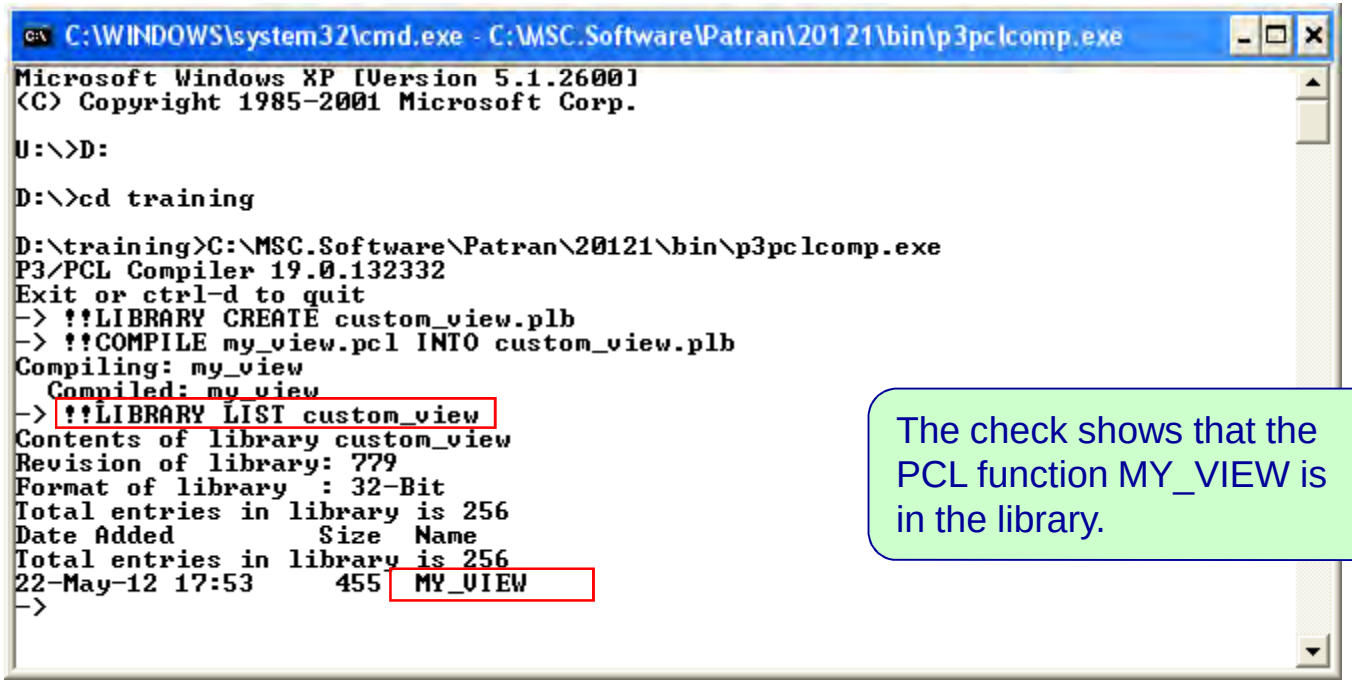
- **Create a library called custom_view.plb**
- **Compile the PCL function my_view.pcl and assign it to the library just created**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW

- **Before exiting the compiler environment, it is always a good idea to determine if the PCL function has been correctly compiled and added to the intended library.**
- **To check the library content, use the following command:**
  - !!library list <library_name>



Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW

- **To have the library loaded during the start of Patran, the p3epilog.pcl file needs to be modified to include the library name.**
- **To add a library, use the command shown in the figure below in p3epilog.pcl.**
- **If you do not have a p3epilog.pcl file to edit, simply create one and save it in the Start-in, Home, or Installation directory.**

Comment →
Command →

```
ui_write( " " )
ui_write( "Processing personal p3epilog.pcl ..." )

/* Load the swap from black to white function */

sys_library("add", "custom_view.plb")

ui_write( " " )
ui_write( "Processing personal p3epilog.pcl complete." )
```


Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW

- **Once Patran is run, it can be determined whether or not the library has been loaded by typing “!!library LIST <library_name>” in the command line.**

- **To execute the custom view function, enter my_view() in the command line.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW

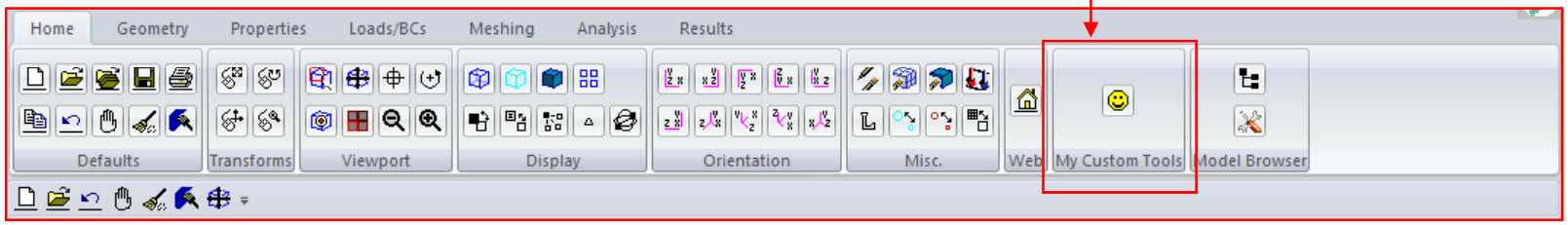
- **Alternatively, the user can execute the PCL function by having an icon in Patran.**
- **This is done using the file $P3_HOME\p3toolbar.def**
- **This file points to**
  - A bitmap file that describes the icon (which you will have to create yourself).
  - The FUNCTION name
- **You can create a separate Toolbar called “My Custom Tools”, which in this example will be adjacent to the Model Browser Toolbar**
- **To create and load an icon into Patran, put the bitmap file for the icon graphics into the directory $P3_HOME\icons.**
  - The dimension of the icon is set at 16 pixels by 16pixels.

```
*START TOOLBAR    = My Custom Tools

  *ICON     = my_view_icon.bmp
  *CLASS    =
  *FUNCTION = my_view
  *HELP     =
*LOAD ITEM

*END TOOLBAR

*START TOOLBAR    = Model Browser

  *ICON     = modeltree.bmp
  *CLASS    =
  *FUNCTION = toggleModelTree
  *HELP     = Show/Hide Model Browser
*LOAD ITEM
```


Copyright© 2012 MSC.Software Corporation

CASE STUDY: MAKING A CUSTOM VIEW

Now the icon is shown in the Patran Toolbar after it has been loaded.



Copyright© 2012 MSC.Software Corporation

CASE STUDY: FUNCTION FOR PRESSURE

- **The purpose of this case study is to create a pressure load for a model which is partially submerged in water.**
- **The pressure will be zero where the model is above the water, and equal to (rho)gh where the model is submerged, where h = distance below the free surface of the water.**
- **A method to create a pressure load such as this is to create a PCL function that will determine the pressure at different locations (elevations), and then create a Spatial Field using this PCL function.**

Copyright© 2012 MSC.Software Corporation

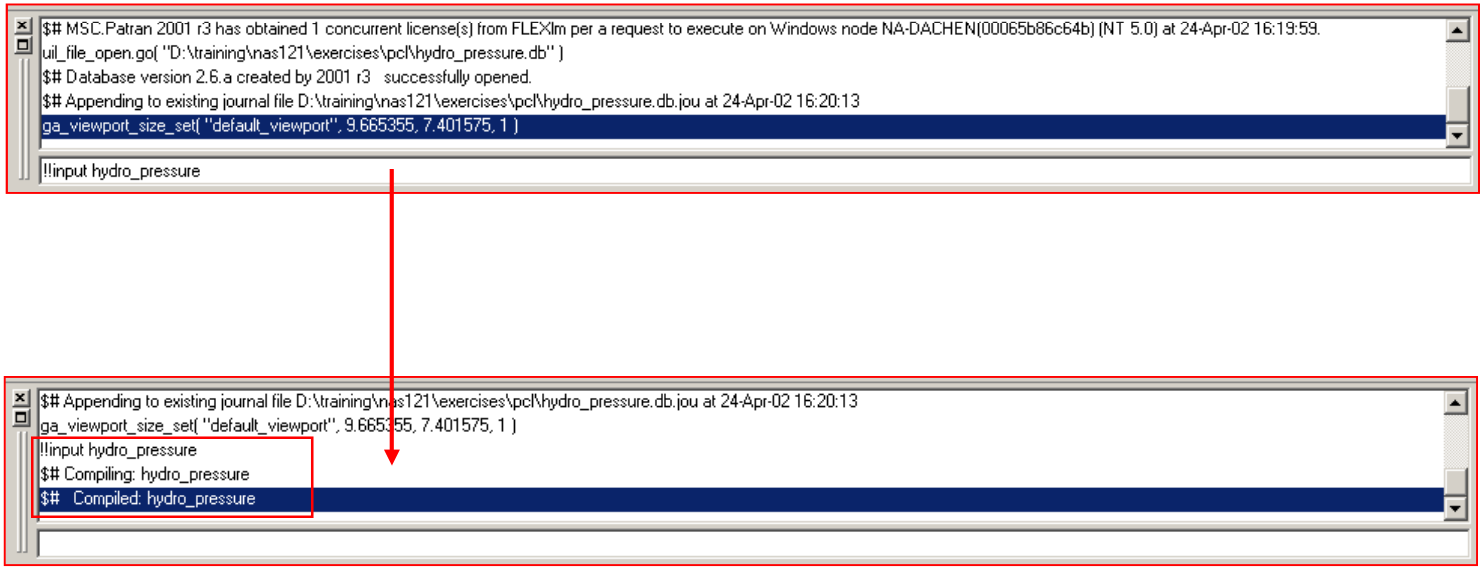
CASE STUDY: FUNCTION FOR PRESSURE

```
function hydro_pressure(x,y,z)

/* declare variables */
real x, y, z, y_wave, y_rel
real p, rho, grav

/* specify the density of the fluid */
rho = 1.0

/* specify the acceleration of gravity */
grav = 9.81

/* function defining the surface of the fluid */
y_wave = sinr(x)

/* y position relative to the wave */
y_rel = y - y_wave

/* if the object is submerge the pressure is calculated */
/* otherwise, pressure = 0                              */
if (y_rel >= 0) then
           p = 0
else
           p = abs(rho*grav*y_rel)
end if


/* return the pressure value to MSC.Patran */
return p

end function
```


Copyright© 2012 MSC.Software Corporation

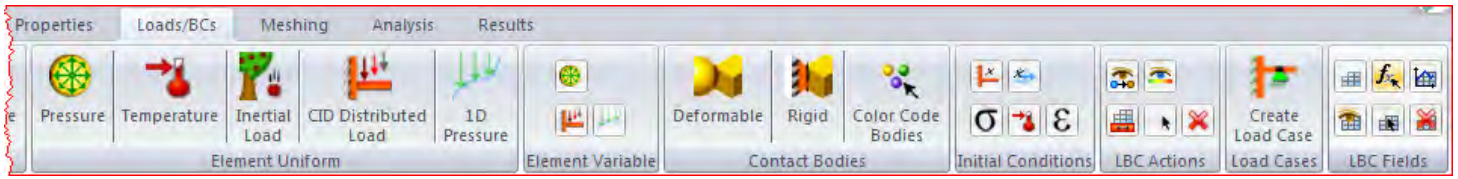
CASE STUDY: FUNCTION FOR PRESSURE

- **Because the parameters in this function may change from time to time, such as the equation representing the wave profile, the best compiling method for this PCL function may be to compile it in the command line in Patran.**

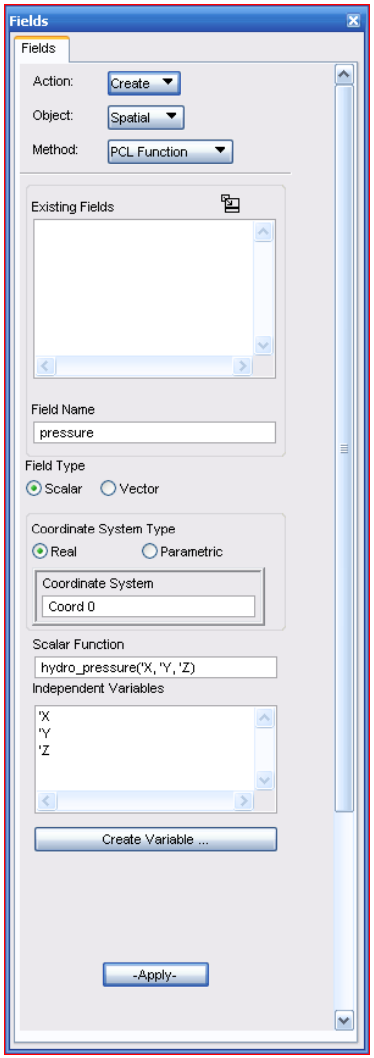


Copyright© 2012 MSC.Software Corporation

CASE STUDY: FUNCTION FOR PRESSURE

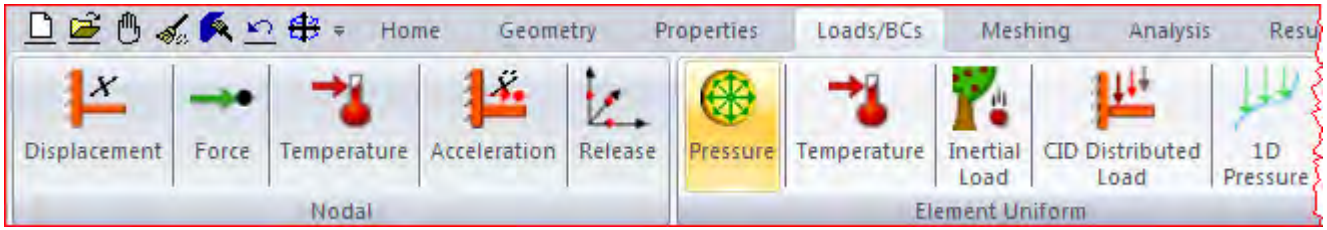


- **Create the Spatial PCL Function field, using the compiled PCL function hydro_pressure.**
  - Enter the function name and the variables in the Scalar Function box.

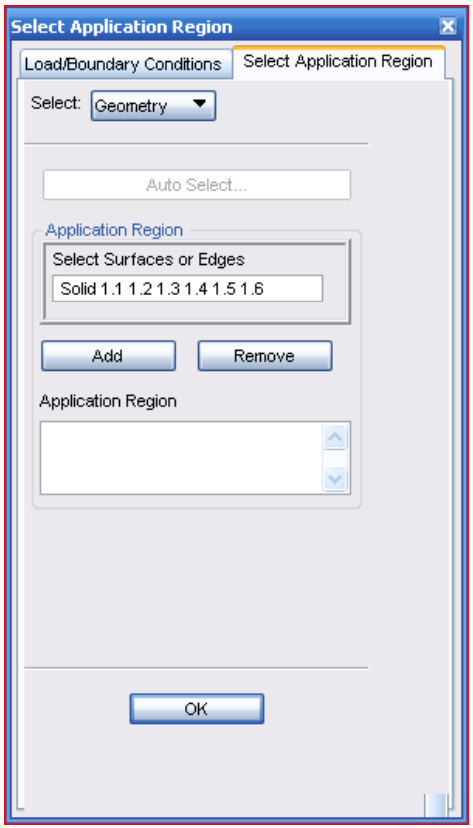


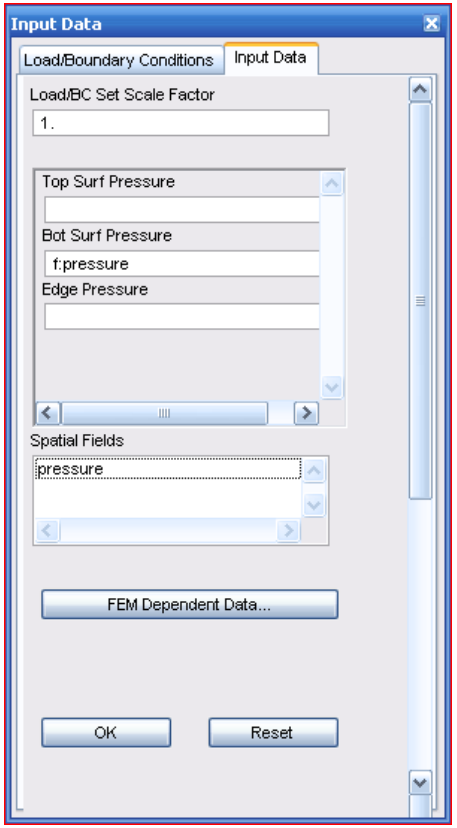
Copyright© 2012 MSC.Software Corporation

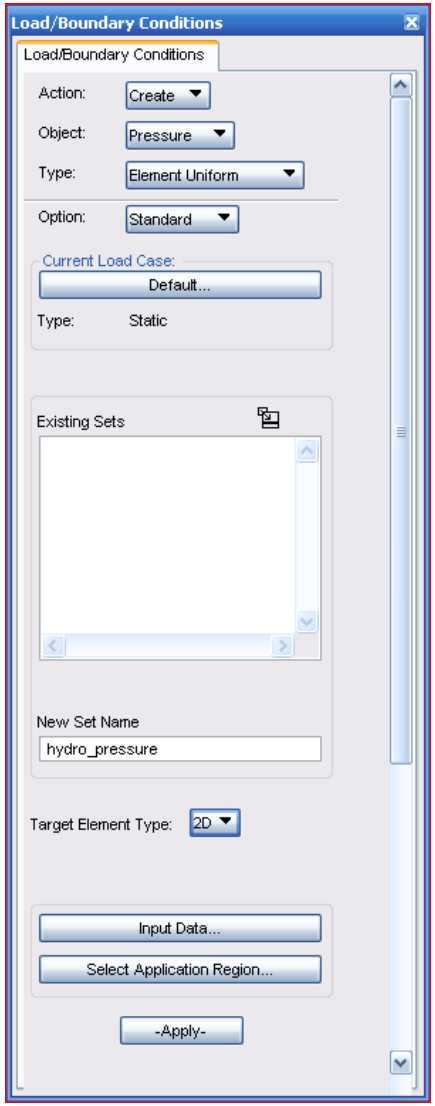
CASE STUDY: FUNCTION FOR PRESSURE



- **Create the pressure load, using the field created from the PCL function.**



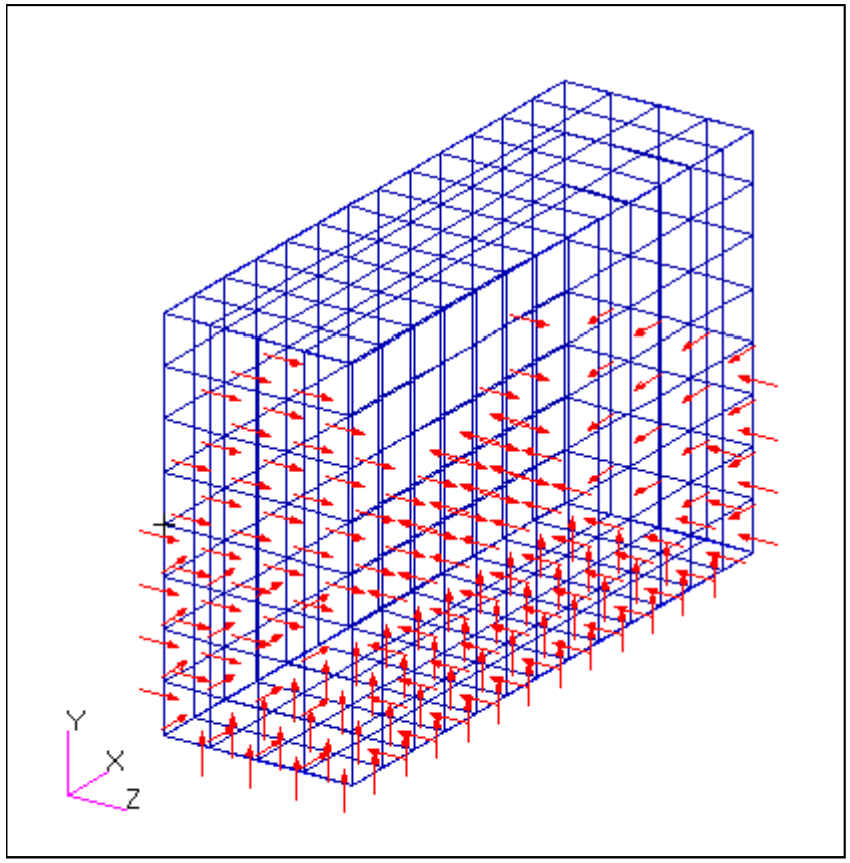


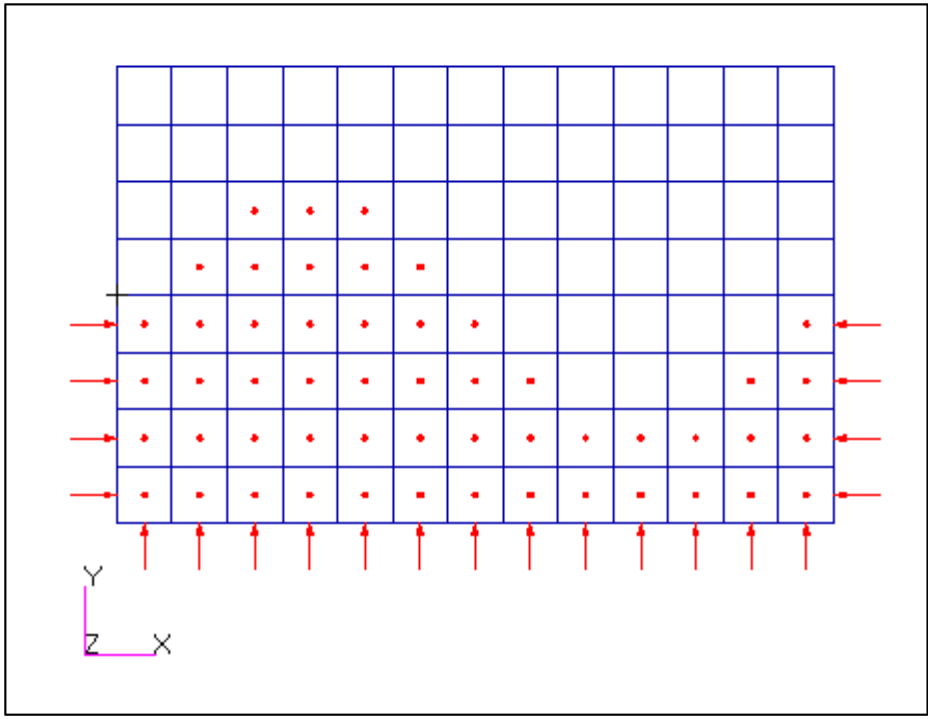


Copyright© 2012 MSC.Software Corporation

CASE STUDY: FUNCTION FOR PRESSURE

The pressure load will look like this on the model.





Copyright© 2012 MSC.Software Corporation

MORE ON VARIABLE DECLARATION

- **Arrays**
  - Available for all data types
    - REAL a(10)
    - INTEGER B(20)
  - Can be assigned lower and upper bound
    - array_name(low:high), where “low” and “high” have integer value, and are for the lower and upper bound, respectively.
    - Default lower bound is 1, not 0.
  - Row “major”, unlike FORTRAN.
    - INTEGER my_array(2,3) = 1,2,3,4,5,6 =>
    - REAL disp(6,200)
    - INTEGER ids(0:2,0:4,0:10)

$$\begin{bmatrix} 1 & 2 & 3 \\ 4 & 5 & 6 \end{bmatrix}$$

Copyright© 2012 MSC.Software Corporation

MORE ON VARIABLE DECLARATION

- **Virtual arrays**
  - Have variable dimensions.
  - The amount of storage space for a virtual array is allocated by a PCL function.
  - The space for a virtual array can be released and allocated for a different virtual array.

```
FUNCTION
…
REAL mydata(VIRTUAL)
sys_allocate_array(mydata,1,100)
…
sys_free_array(mydata)
…
```


Copyright© 2012 MSC.Software Corporation

MORE ON VARIABLE DECLARATION

- **PCL variable accessibility in PCL program:**
  - Specify variable as
    - GLOBAL – available to all functions
    - LOCAL – only available in a single FUNCTION; default
    - STATIC – same as LOCAL, but retains value between FUNCTION calls
    - CLASSWIDE – available to all FUNCTIONs within the same class; retains value for FUNCTION calls

Copyright© 2012 MSC.Software Corporation

MORE ON PCL EXPRESSIONS

- **Branching**
  - PCL is a block structured language. Some of the language elements that control branching and skipping are
    - IF THEN, ELSE
    - SWITCH, CASE
    - BREAK
  - Following is a description of the IF THEN, ELSE statement
    - IF (logical expression) THEN
    - statements
    - ELSE IF (logical expression) THEN
    - statements
    - ELSE
    - statements
    - END IF

Copyright© 2012 MSC.Software Corporation

MORE ON PCL EXPRESSIONS

- **Looping**
  - Repeat execution of a set of statements included in a loop.
  - Loops include initialization, incrementing, execution, and testing.
  - A loop may be nested in another loop.
  - Some PCL block structures are
    - WHILE
    - REPEAT
    - FOR
  - Following is a description of the FOR statement
    - FOR (variable = numeric expression TO Numeric expression [BY numeric expression]) [label]
    - Statements
    - END FOR

Copyright© 2012 MSC.Software Corporation

PCL DIRECTIVES

- **PCL directives begin with !!**
- **Some examples are**
  - !!input *file* – reads input from the file *file*
    - If a PCL function is encountered, it is compiled.
  - !!path [ADD] *directory* – include directory (path) in a set of directories that will be searched to find files for a PCL function.
    - Used to specify the order in which directories are searched for a file.
    - !!path lists the current directories (paths) that will be searched, e.g.

```
!path
$# Current path...
$#   C:\Documents and Settings\mwu
$#   C:\MSC.Software\Patran\20111
$#   C:\MSC.Software\Patran\20111\helpfiles
$#   C:\MSC.Software\Patran\20111\alters
$#   C:\MSC.Software\Patran\20111\icons
$#   C:\MSC.Software\Patran\20111\icons\RibbonIcons
```

  - !!trace – PCL execution verification
    - !!trace calls – indicates the sequence of routines, as they are called; if set to nocalls, the sequence will not be displayed.
    - !!trace exits – indicates the sequence of routines which have been exited; if set to noexits, the sequence will not be displayed.

Copyright© 2012 MSC.Software Corporation

PCL DIRECTIVES

- **Some examples are (Cont.)**
  - !!debug – causes the storing of PCL line contents in a file for future reference
    - Used to determine the executed lines of code.
    - Invoked when compiling PCL routines.
    - Example
      - !!debug on
      - !!input hydro_pressure
      - !!trace lines

```
!!debug on
$# !! DEBUG is ON
!!input hydro_pressure
$# Compiling: hydro_pressure
$#   Compiled: hydro_pressure
!!trace lines
$# !! TRACE level mask is STDOUT NOCALLS NOEXITS LINES NOTIMESTAMP
loadsbcs_create2( "debug_hydro_pressuire", "Pressure", "Element Uniform", "2D", "Static", ["Solid 1.1 1.2 1.3 1.4 1.5 1.6"],
$# Load/BC set "debug_hydro_pressuire" created.
```

```
Output window
[Line Number  365]
[   8:rho = 1.0]
[  11:grav = 9.81]
[  14:y_wave = sinr(x)]
[  17:y_rel = y - y_wave]
[  21:if (y_rel >= 0) then]
[  24:p = abs(rho*grav*y_rel)]
[  29:return p]
[Line Number  366]
```


Copyright© 2012 MSC.Software Corporation

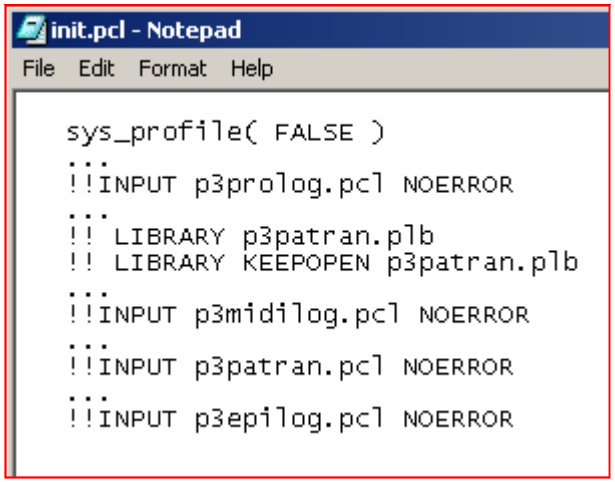
PCL DIRECTIVES

- **They are processed differently from other PCL commands.**
  - They are processed immediately when encountered.
- **Do not embed in (use inside) PCL functions. Instead, use corresponding PCL commands:**
  - sys_input corresponds to !!input
  - sys_library corresponds to !!library
  - sys_path corresponds to !!path

Copyright© 2012 MSC.Software Corporation

STARTING PATRAN, BRIEFLY

- **The files read when Patran is started are listed, together with a brief description of what goes in them**
  - init.pcl
    - Reads (in order) the files that have PCL function calls, including any user PCL calls.



Copyright© 2012 MSC.Software Corporation

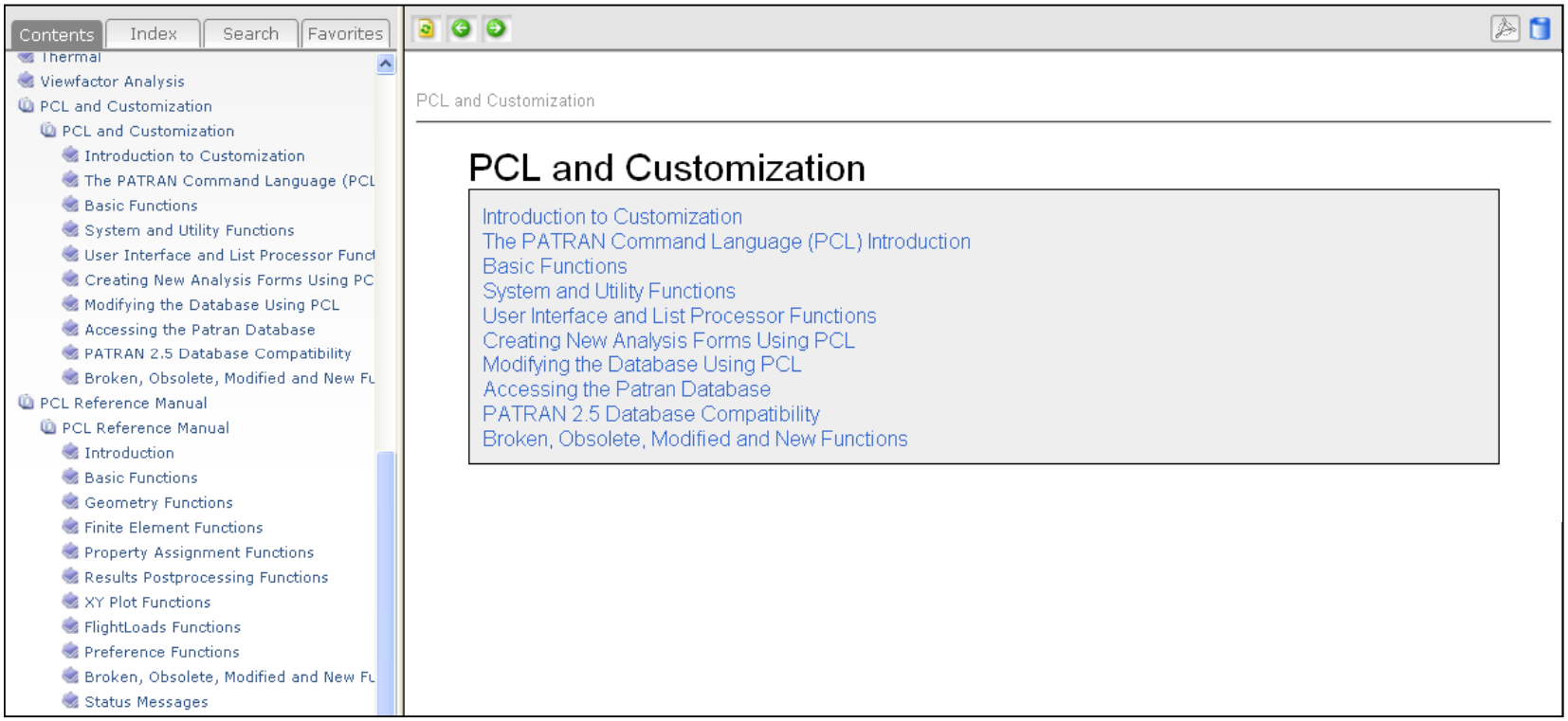
STARTING PATRAN, BRIEFLY

- **The files read when Patran is started (Cont.)**
  - p3prolog.pcl
    - Read prior to loading Patran libraries
      - Example: global real radius=1.0
  - p3midilog.pcl
    - Read after loading Patran libraries
      - Example: uil_main_form_parameters
  - p3patran.pcl
    - Same as p3epilog.pcl; extra file that can be read separate from p3epilog.pcl
  - p3epilog.pcl
    - PCL commands read and executed, and PCL functions compiled
      - Example: !!input sample.pcl
      - !!library sample.plb
- **Entries that can be placed in p3prolog.pcl can be placed instead in p3midilog.pcl or p3epilog.pcl.**

Copyright© 2012 MSC.Software Corporation

WHERE TO GO FOR HELP

- **Select Contents and Index from the Help menu in Patran.**



- **Select PCL from the Help menu in Patran.**
  - Same as Contents and Index

Copyright© 2012 MSC.Software Corporation

EXERCISES

- Perform Workshop 20 “Creating PCL Functions” in your exercise workbook.
- Perform Workshop 21 “PCL via Session File Via Pipe Intersection” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

SECTION 10

RESULTS

Copyright© 2012 MSC.Software Corporation

Copyright© 2012 MSC.Software Corporation

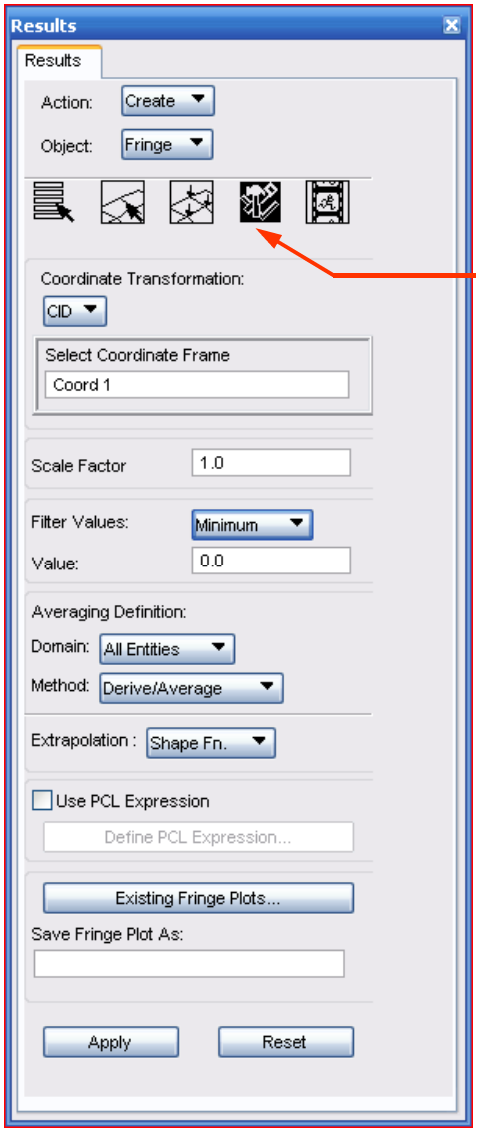
FRINGE PLOTS OF SCALAR RESULTS

- **Interpretation of scalar results (e.g. stress component) using fringe plots:**
  - Results: Create/Fringe
  - The things that must be selected are
    - Result case
    - Result type
    - Layer position
    - Quantity type
    - Target entity
    - Display attributes
    - Plot options
    - Animation options
  - The following case study covers some topics under Plot Options.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

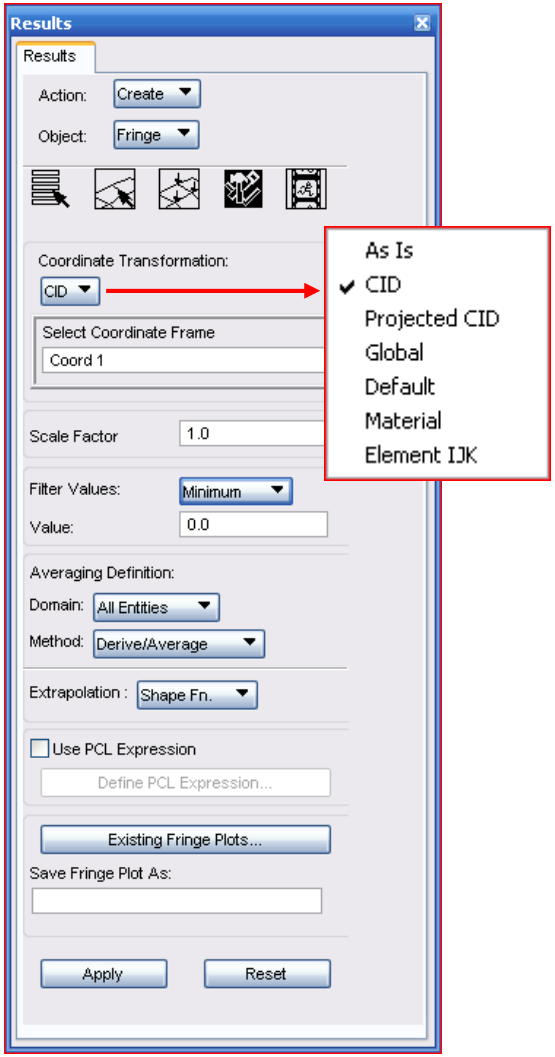
- **The Plot Options Form is used to control the following:**
  - **Coordinate Transformation**, e.g. transform vector components
  - **Scale Factor** – multiply results by factor
  - **Filter Values** – filter results displayed using result values
  - **Averaging Domain and Method** – how element results are combined
  - **Extrapolation method** – how results are combined in an element
  - **Use** a PCL Expression
  - **Re-loading** of an existing plot
  - **Saving** current plot for future use



Plot Options button

Copyright© 2012 MSC.Software Corporation

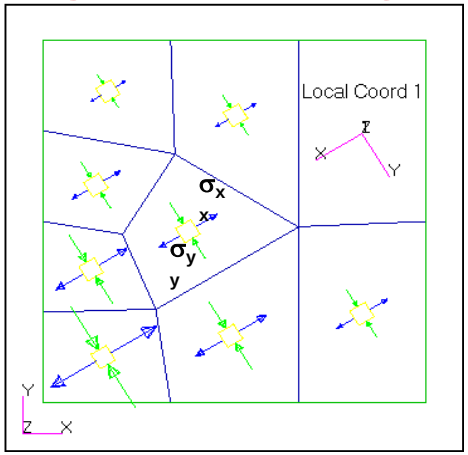
CASE STUDY: FRINGE PLOT OPTIONS



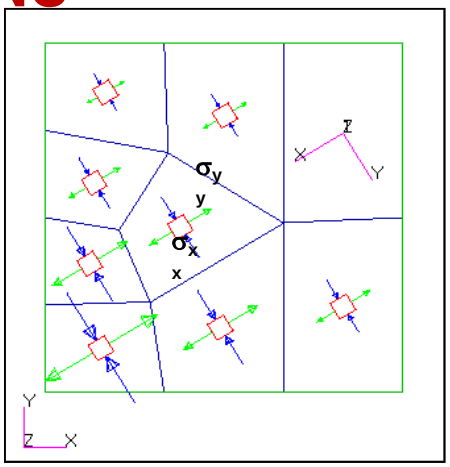
- **Coordinate transformation options**
  - **As Is** –no transformation. Results in solver coordinate system(s).
  - **CID** – Patran local coordinate system, e.g. Coord 3.
  - **Projected CID** – coordinate system projected onto element, e.g. Coord 2.1.
  - **Global** – Patran global coordinate system, Coord 0.
  - **Default** – results are kept in the solver coordinate system(s).
  - **\Material** – element coordinate systems based on a material definition and angle. Only for quad and tri topology.
  - **Element IJK** – Patran defined element coordinate systems. These can be different from solver element coordinate systems.
- **Patran user manual provides detailed information on these transformations.**
  - **IT IS IMPORTANT TO USE THIS MATERIAL**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

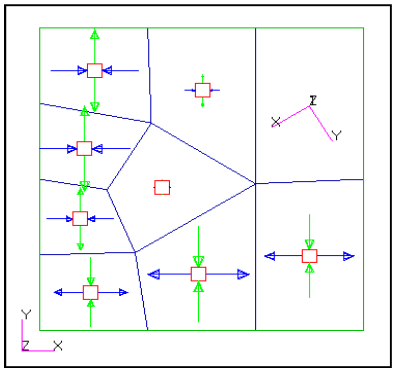


CID, Select Coordinate Frame Coord 1

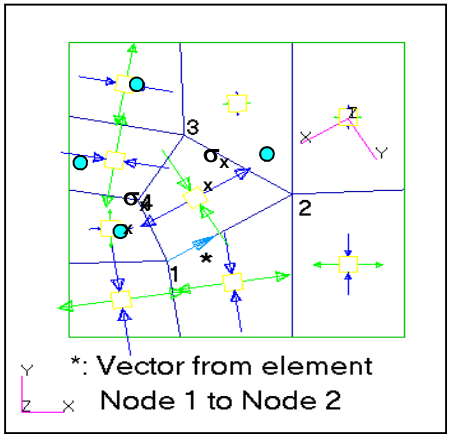


Projected CID, Select CF Axis Coord 1.2, changing axis changes σ_{xx} direction by 90^{o}

Coord 0



Global – uses Patran Coord 0

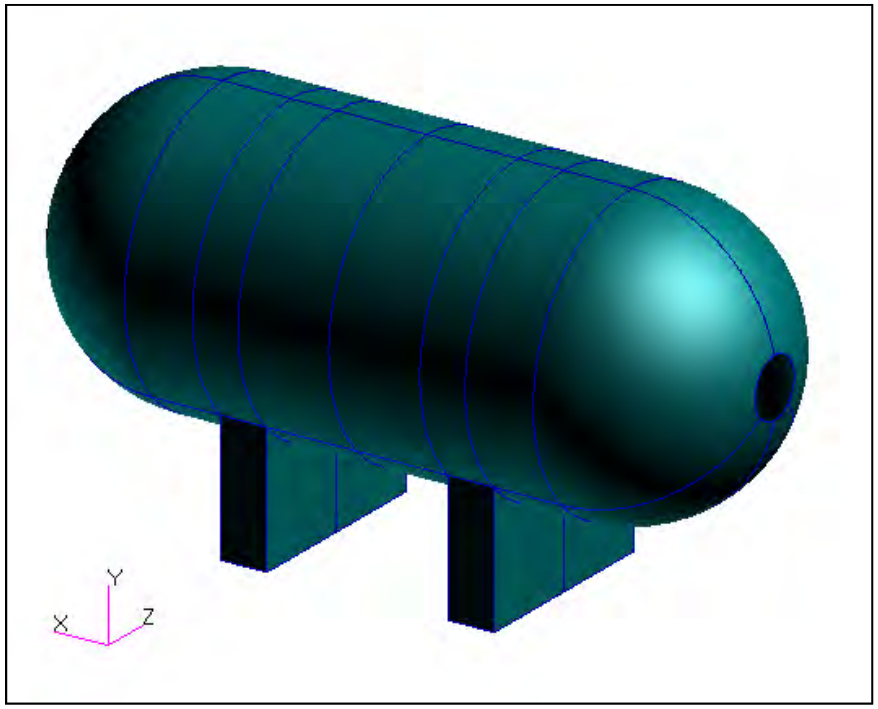


Element IJK – uses first two element nodes for σ_{xx} direction

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

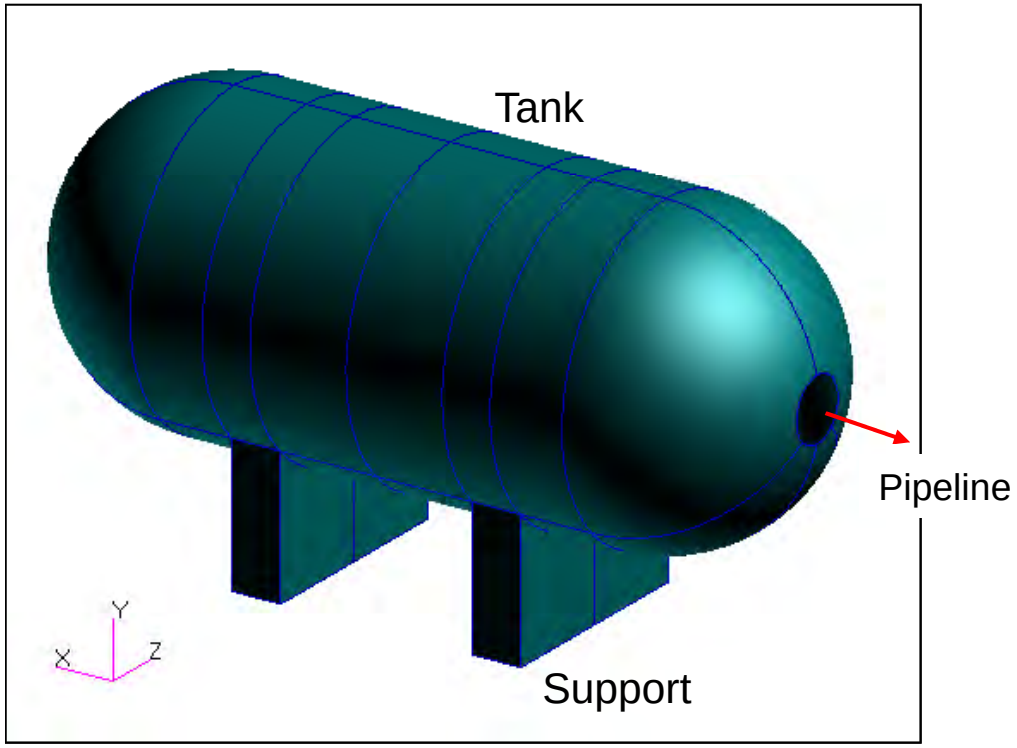
- **A static linear elastic model is used to demonstrate some of the parameters available in Plot Options.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Model**
  - Tank
  - Supports at the bottom of the tank
  - Ends of tank connected to pipeline
  - Internal pressure in tank
  - Base of supports movement
  - Pipe applies torque



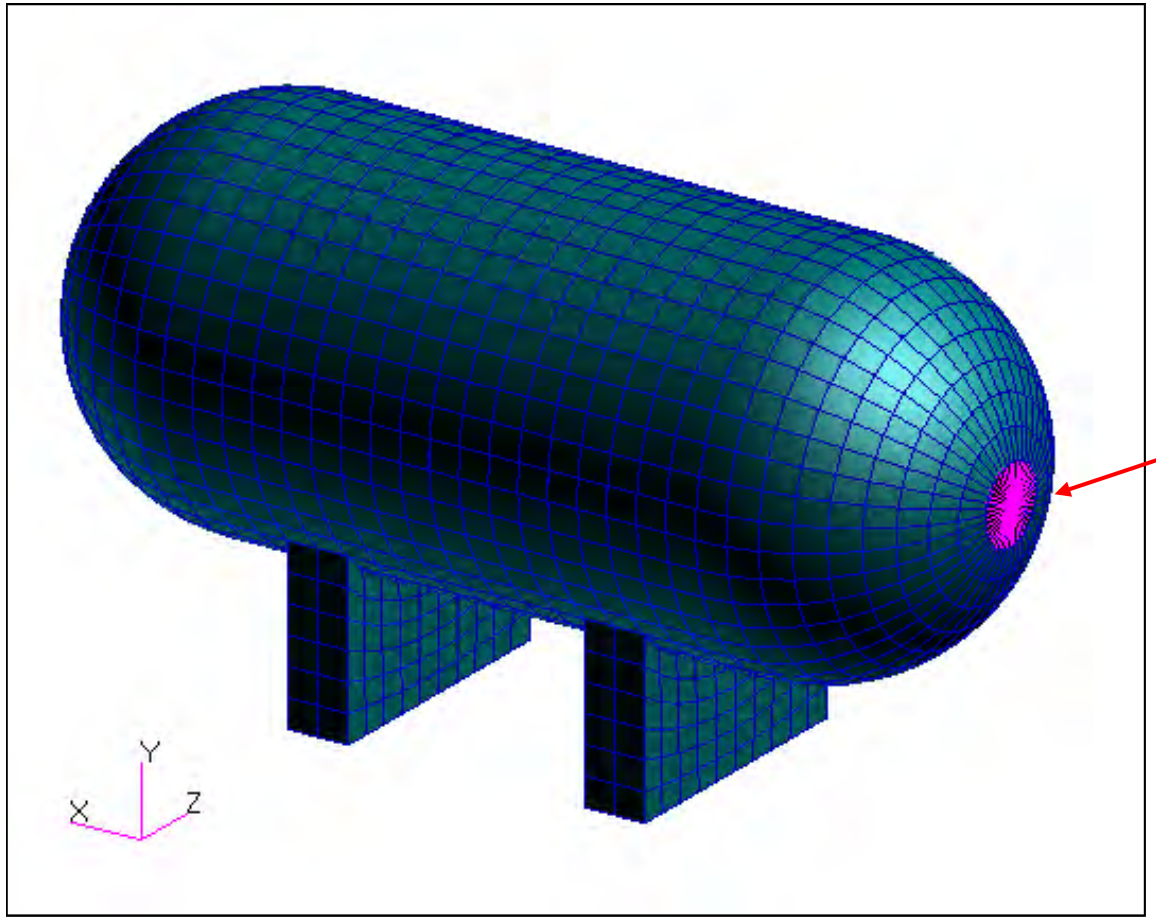
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **IsoMesh surfaces and solids that represent tank and tank supports.**
  - Coarse mesh GEL = 4.5
- **Connect elements.**
- **Create MPC at one end of tank.**
  - For applying torque about tank centerline, and transmitting torque to edge of tank hole.

Copyright© 2012 MSC.Software Corporation

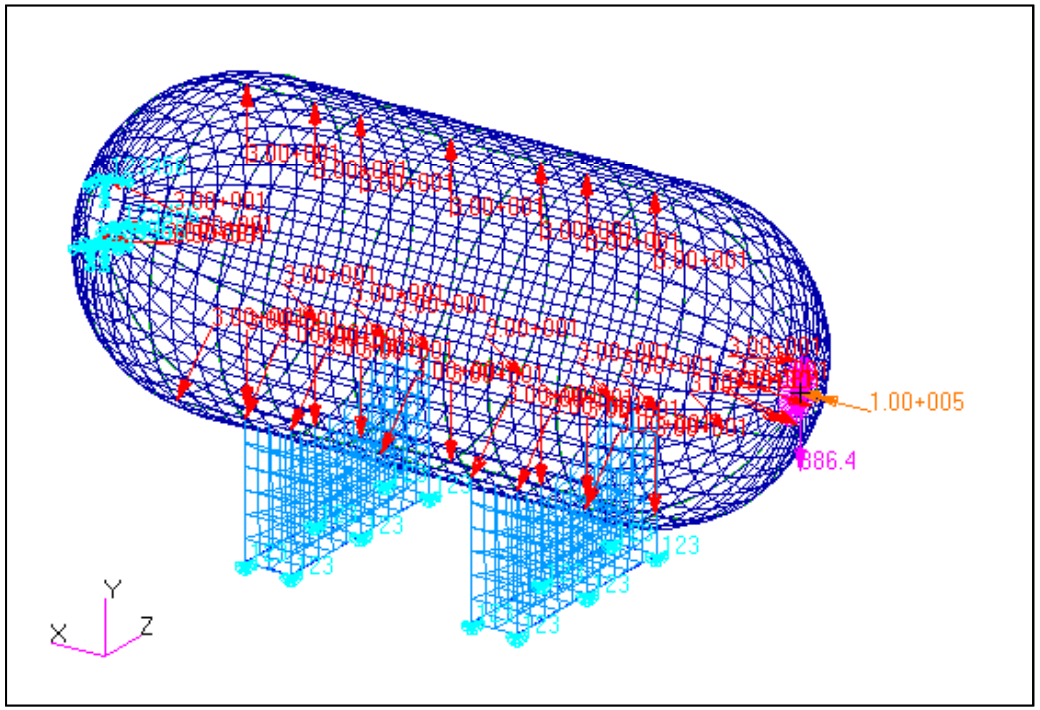
CASE STUDY: FRINGE PLOT OPTIONS



Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Create constraints at the base of supports and other end of the tank.**
- **Create loads**
  - Internal pressure
  - Inertial loading
  - Torque on one end of tank

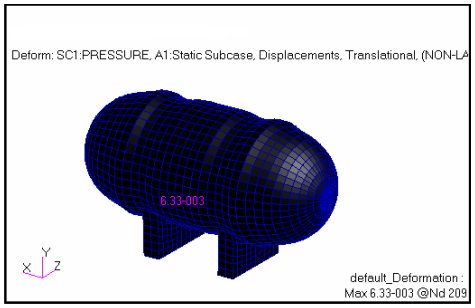


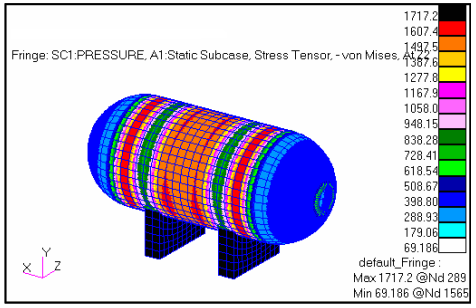
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

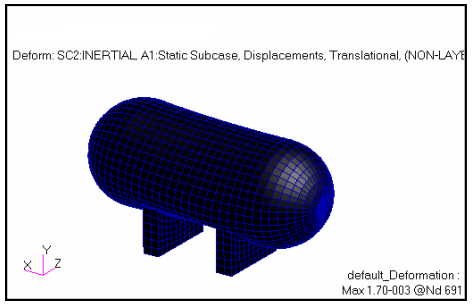
- **Run three separate load cases:**

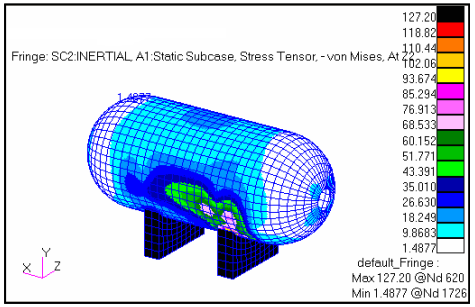
Internal pressure



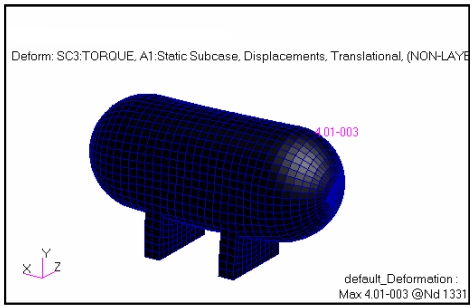


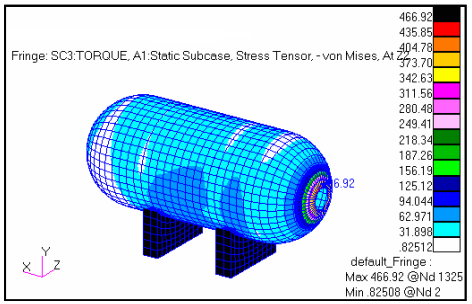
Inertial loading





Torsion on end

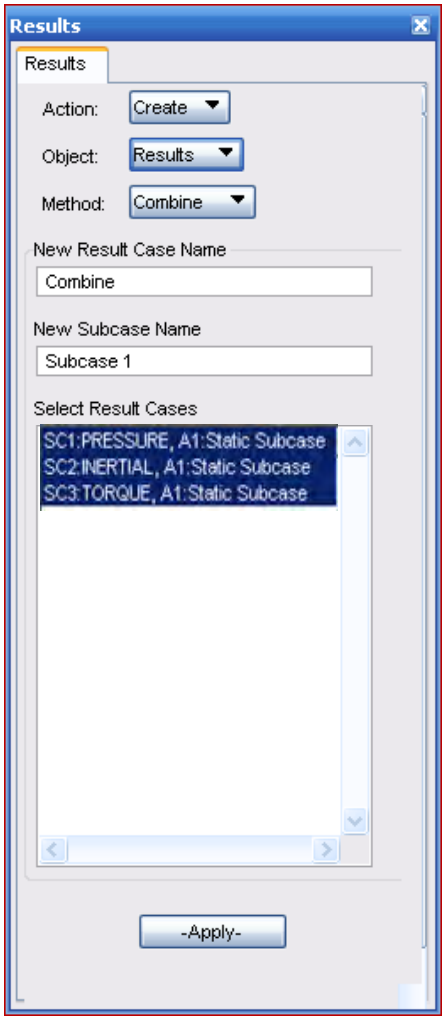


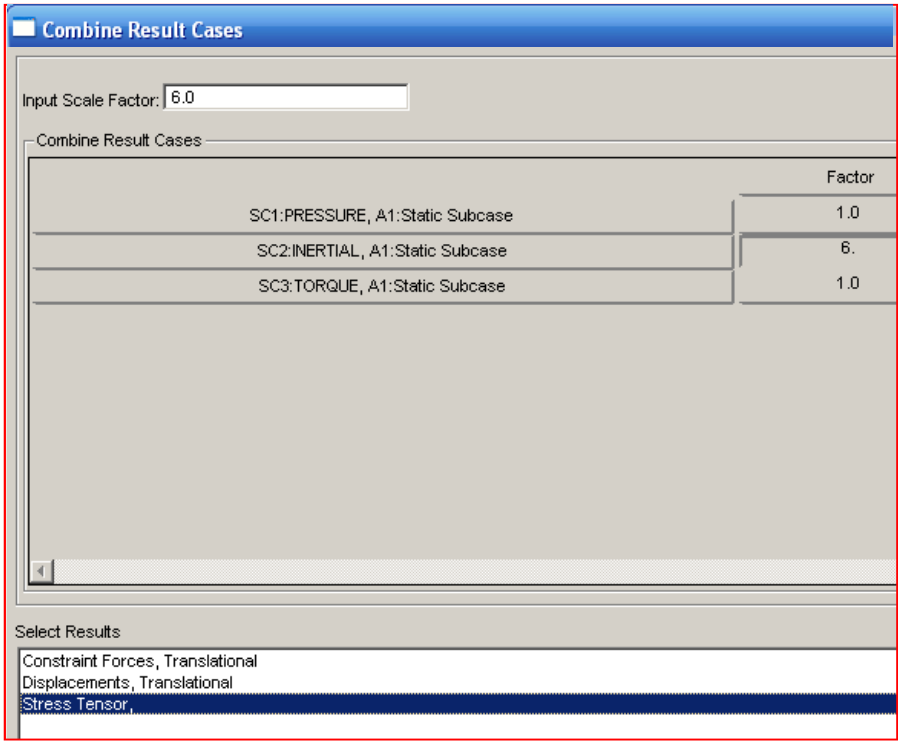


Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Combine the results from the three load cases:**

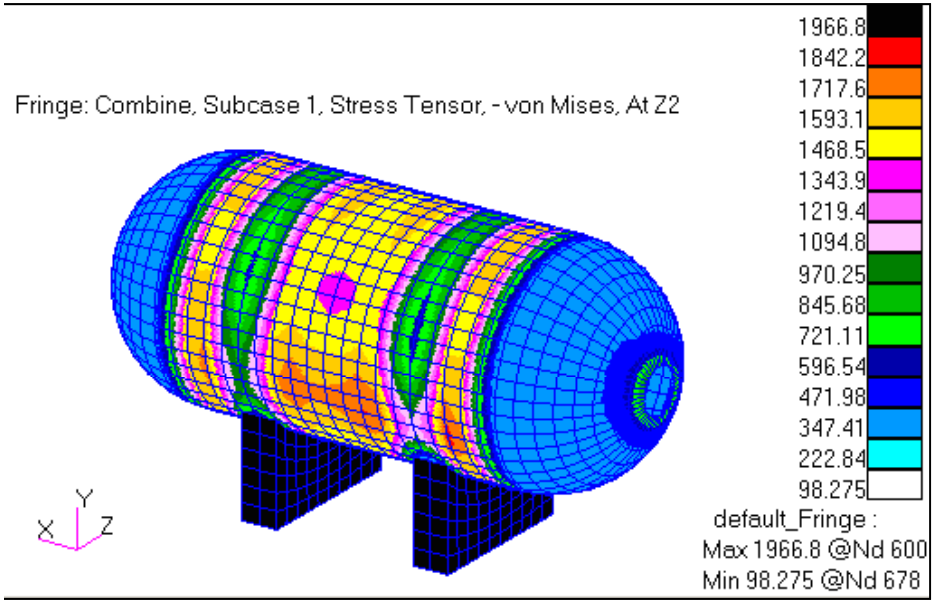


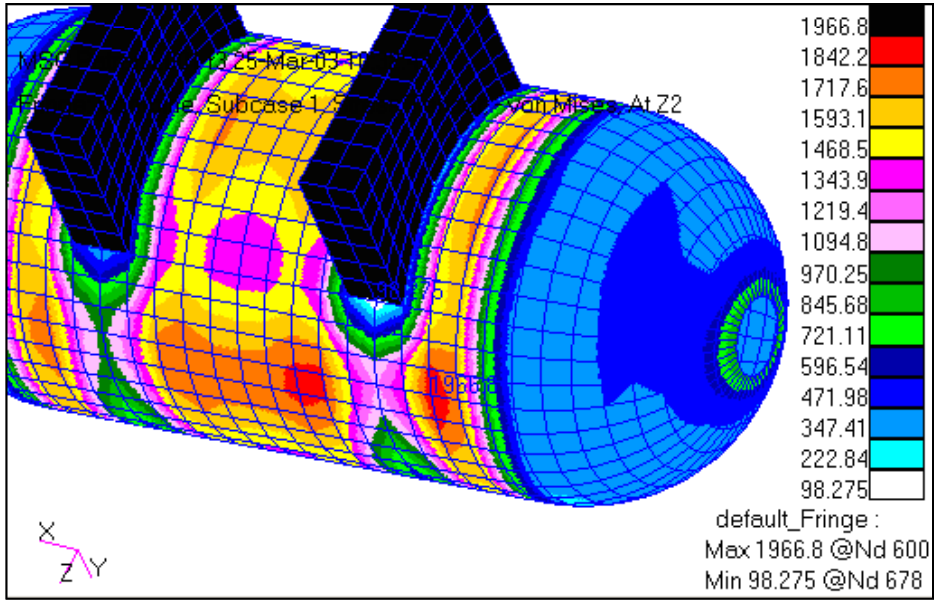


Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Combined stress**

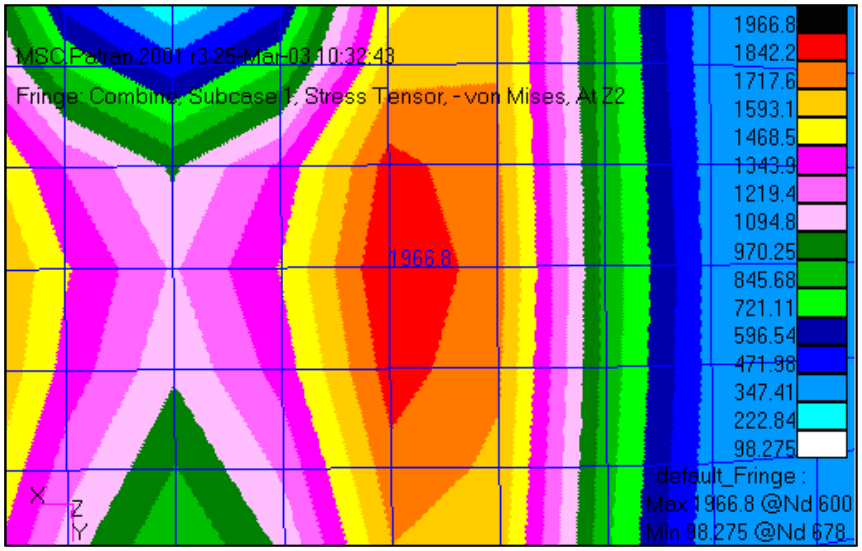




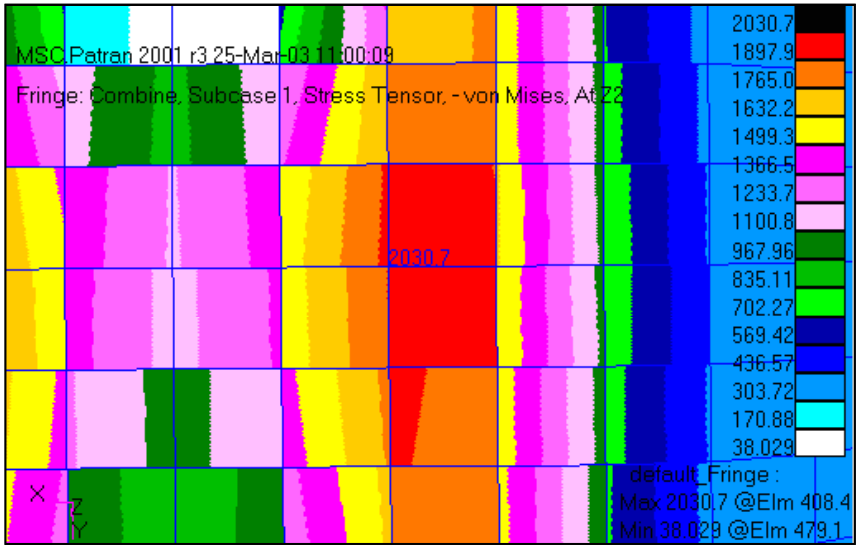
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Compare fringe for Averaging Definition, Domain: All Entities or None:**



All Entities



None

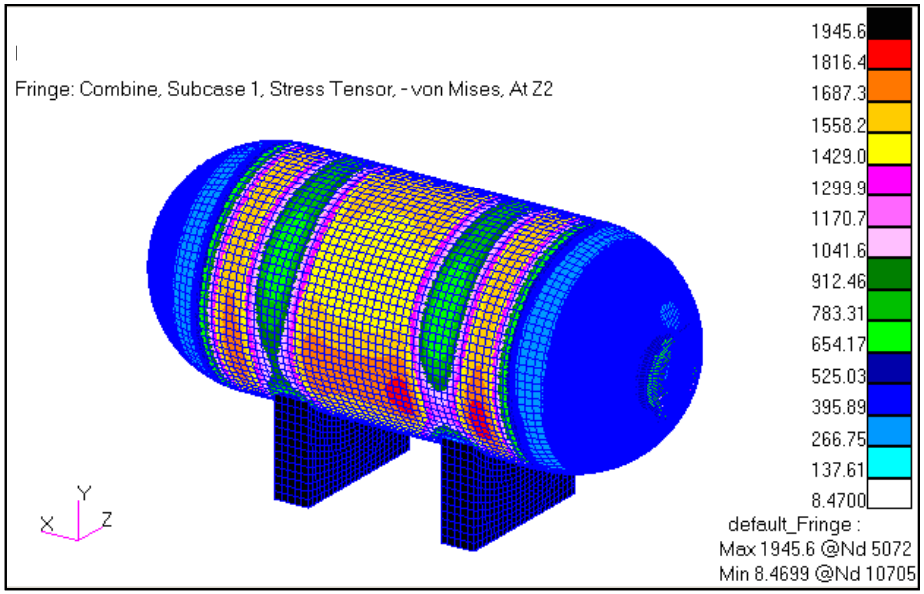
Note: **Both have As Is, Derive/Average, Shape Fn.**

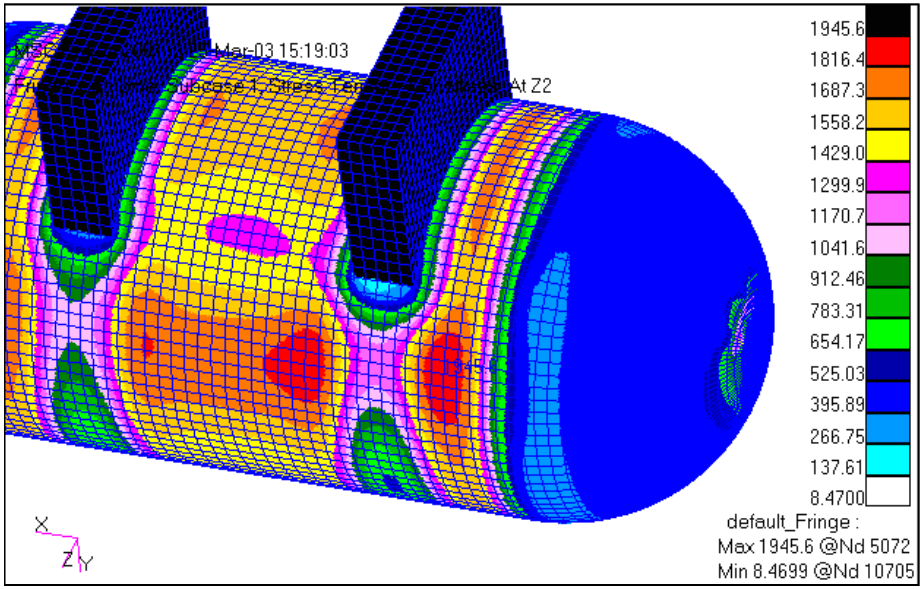
- **Too many color bands for some elements for both averaging methods.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Re-mesh**
  - Finer mesh GEL = 2.0
- **Combine the results as before:**

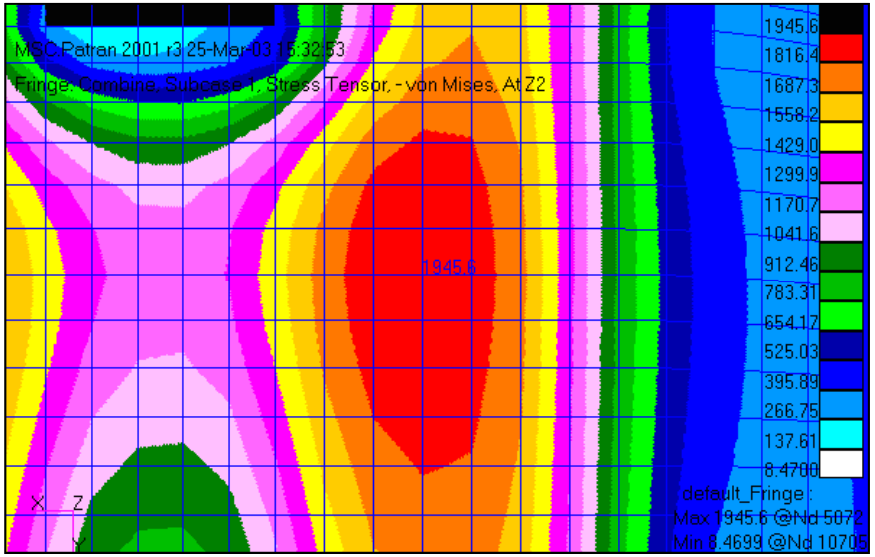




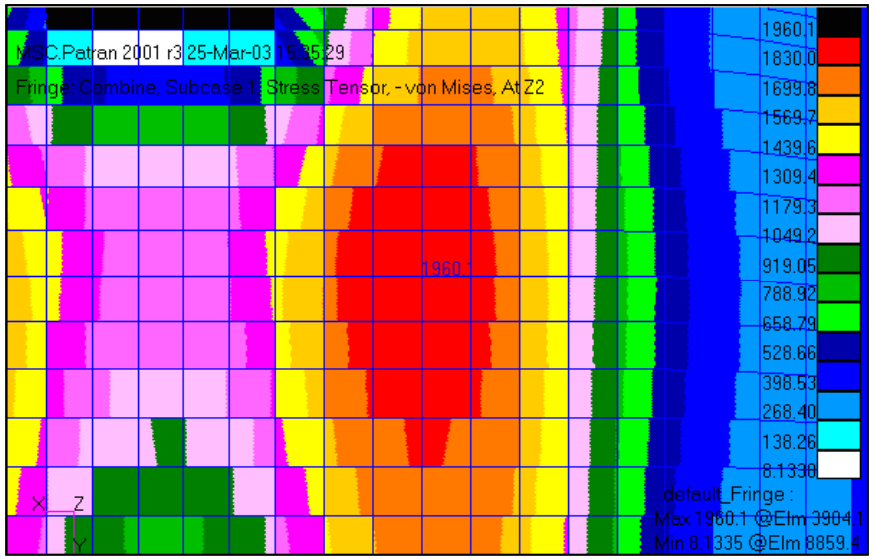
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Compare fringe for finer mesh using Averaging Definition, Domain: All Entities or None:**



All Entities



None

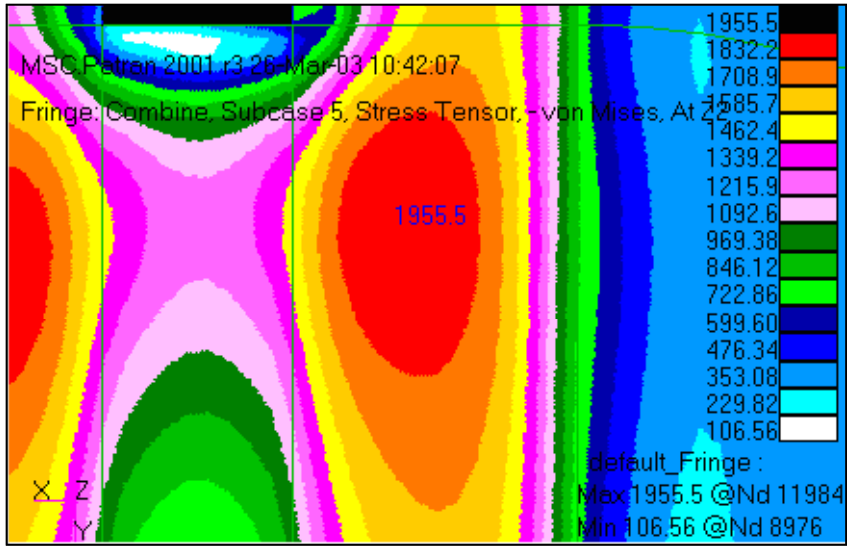
Note: **Both have As Is, Derive/Average, Shape Fn.**

- **Better, but still too many color bands for some elements for both averaging methods.**

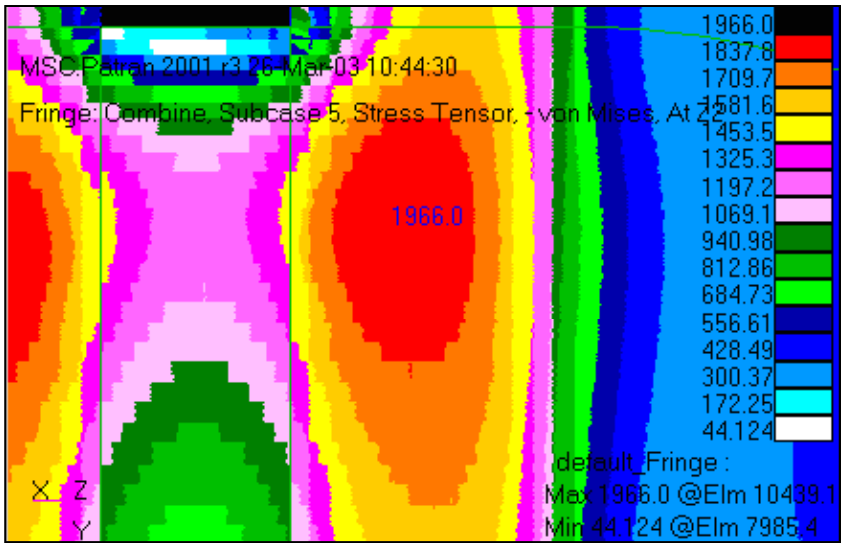
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Compare fringe for finest mesh (GEL = 1.0) using Averaging Definition, Domain: All Entities or None:**



All Entities



None

Note: **Both have As Is, Derive/Average, Shape Fn.**

- **Good; not too many color bands per element. Notice that the two fringes are approaching each other.**

Copyright© 2012 MSC.Software Corporation

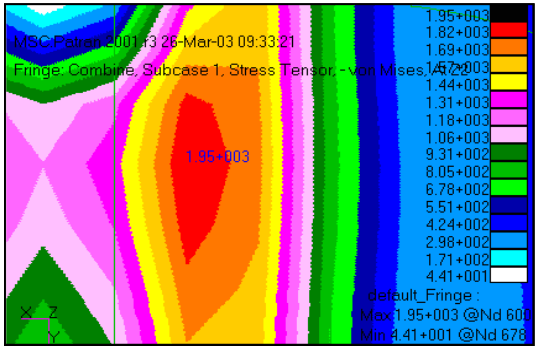
CASE STUDY: FRINGE PLOT OPTIONS

- **Now, look at the effects of changing other Plot Option parameters, and just using All Entities.**
  - Derive/Average to Average/Derive
  - Shape Fn. to Average
  - Shape Fn. to Centroid

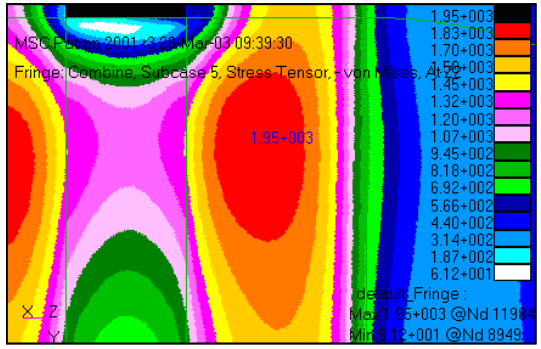
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

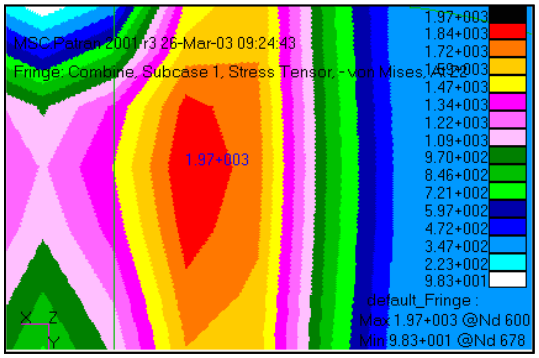
- **Average/Derive (A/D) versus Derive/Average (D/A)**



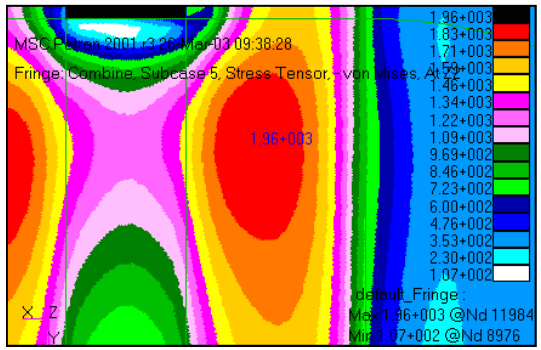
A/D, Coarse Mesh



A/D, Finest Mesh



D/A, Coarse Mesh



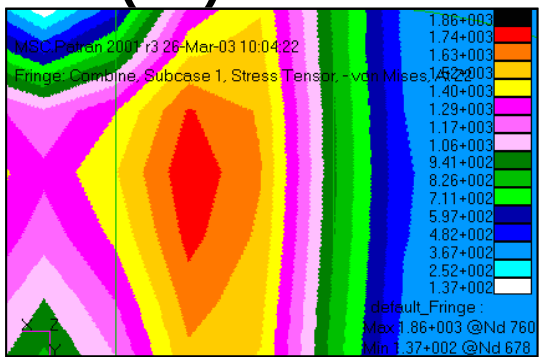
D/A, Finest Mesh

Note: **All have As Is, All Entities, Shape Fn.**

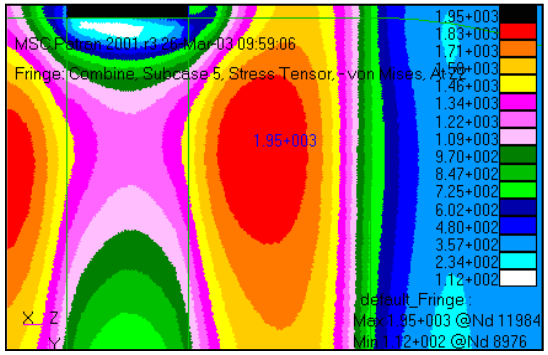
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

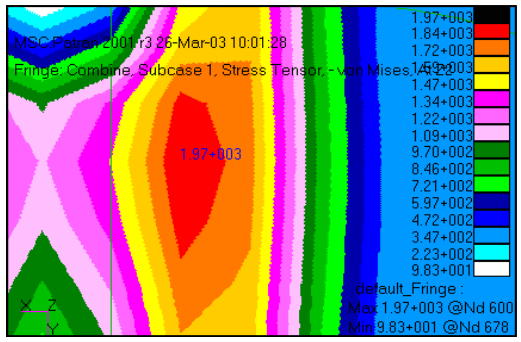
- **Shape Fn. (SF) versus Average (A)**



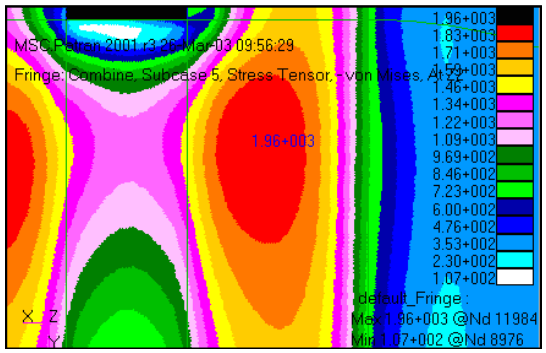
A, Coarse Mesh



A, Finest Mesh



SF, Coarse Mesh



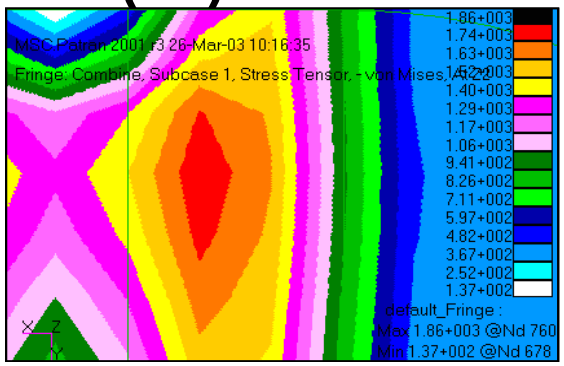
SF, Finest Mesh

Note: All have As Is, All Entities, Derive/Average

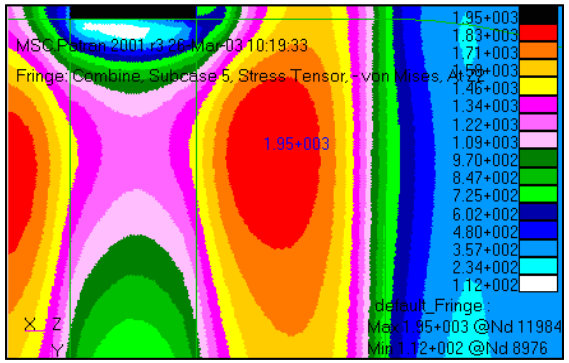
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

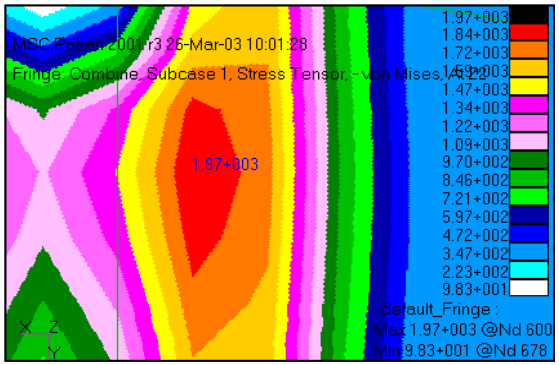
- **Shape Fn. (SF) versus Centroid (C)**



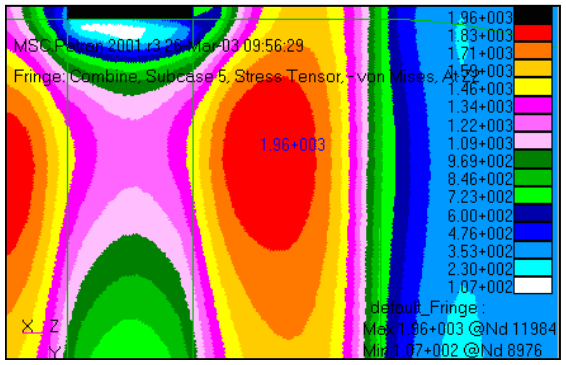
C, Coarse Mesh



C, Finest Mesh



SF, Coarse Mesh



SF, Finest Mesh

Note: All have As Is, All Entities, Derive/Average

Copyright© 2012 MSC.Software Corporation

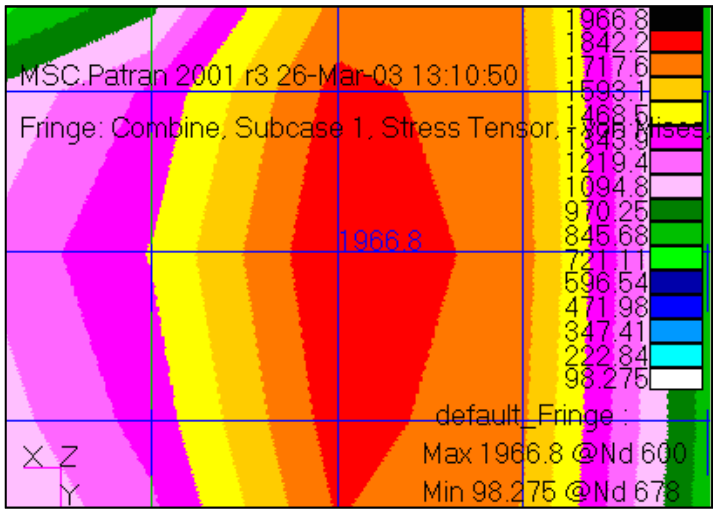
CASE STUDY: FRINGE PLOT OPTIONS

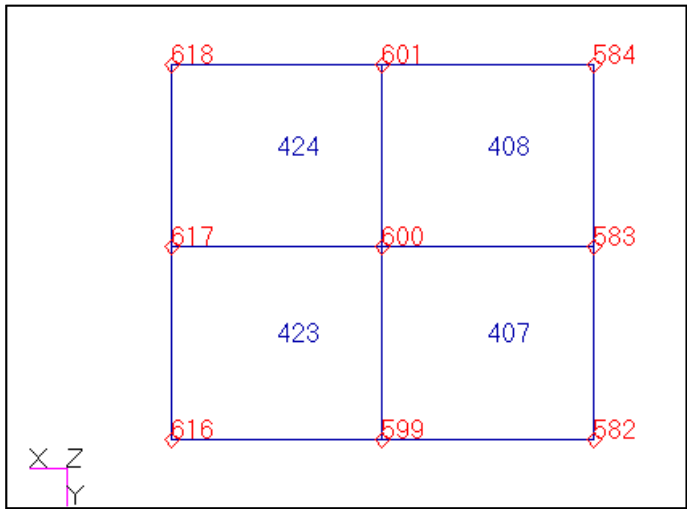
- **Observations**
  - Finer the mesh, lesser the effect of the choice of parameters (e.g. All Entities or None).
  - If the mesh is too coarse in areas of high stress gradient, at high stress levels, using All Entities, then None will result in substantially different fringe plots.
    - This will indicate that the mesh needs to be finer in those areas.
  - For some models using Shape Fn., Average or Centroid can result in at least slightly different fringes.
  - For some models, using Derive/Average or Average/Derive does not affect the fringe, regardless of whether the mesh is coarse or fine.
  - It is important to compare the results displayed in Patran with those created by the solver, e.g. Attach XDB versus .f06.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Look at the case study example for the coarse mesh to see how Patran uses the solver results to calculate its results, e.g. Nastran stress -> Patran stress**
  - Look at the three Nastran .f06 ASCII results files:
    - Pressure (internal), Inertial loading, Torsion (on one end)
  - Look at the node where there is maximum stress, Node 600, and the associated elements, Element 407, 408, 423, 424:





Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Example stresses**
  - Stresses for Element 408 for the three subcases

1 MSC.NASTRAN JOB CREATED ON 20-MAR-03 AT 14:10:38 MARCH 24, 2003 MSC.NASTRAN 2/ 4/02 PAGE 602

PRESSURE

0 SUBCASE 1

S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4) OPTION = BILIN

ELEMENT ID	GRID-ID	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES (ZERO SHEAR) ANGLE	MAJOR	MINOR	VON MISES
0 408	CEN/4	-2.500000E-01	1.676117E+03	4.663313E+02	-6.182010E+00	-0.2928	1.676148E+03	4.662997E+02	1.498439E+03
		2.500000E-01	1.935953E+03	1.262477E+03	-3.102248E+00	-0.2639	1.935967E+03	1.262463E+03	1.702262E+03
	583	-2.500000E-01	1.659227E+03	4.671432E+02	-6.182010E+00	-0.2971	1.659259E+03	4.671111E+02	1.481984E+03
		2.500000E-01	1.917765E+03	1.263238E+03	-3.102248E+00	-0.2716	1.917780E+03	1.263223E+03	1.688499E+03
	584	-2.500000E-01	1.659227E+03	4.655195E+02	-6.182010E+00	-0.2967	1.659259E+03	4.654875E+02	1.482382E+03
		2.500000E-01	1.917765E+03	1.261717E+03	-3.102248E+00	-0.2709	1.917780E+03	1.261702E+03	1.688225E+03
	601	-2.500000E-01	1.693006E+03	4.655195E+02	-6.182010E+00	-0.2885	1.693037E+03	4.654883E+02	1.514914E+03
		2.500000E-01	1.954141E+03	1.261717E+03	-3.102248E+00	-0.2567	1.954155E+03	1.261703E+03	1.716116E+03
	600	-2.500000E-01	1.693006E+03	4.671432E+02	-6.182010E+00	-0.2889	1.693037E+03	4.671120E+02	1.514507E+03
		2.500000E-01	1.954141E+03	1.263238E+03	-3.102248E+00	-0.2573	1.954155E+03	1.263224E+03	1.716369E+03

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Example stresses (Cont.)**
  - Stresses for Element 408 for the three subcases

1 MSC.NASTRAN JOB CREATED ON 20-MAR-03 AT 14:10:38 MARCH 24, 2003 MSC.NASTRAN 2/ 4/02 PAGE 1291
INERTIAL
0 SUBCASE 2

S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4) OPTION = BILIN

ELEMENT ID	GRID-ID	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES (ZERO SHEAR) ANGLE	MAJOR	MINOR	VON MISES
0 408	CEN/4	-2.500000E-01	-3.098058E+01	3.030259E+01	-3.931691E+01	-63.9655	4.950804E+01	-5.018602E+01	8.633825E+01
		2.500000E-01	3.027239E+01	5.444157E+01	-2.645793E+01	-57.2742	7.144408E+01	1.326988E+01	6.582015E+01
	583	-2.500000E-01	-2.675301E+01	3.497486E+01	-3.931691E+01	-64.0660	5.409494E+01	-4.587309E+01	8.667240E+01
		2.500000E-01	2.034735E+01	6.040412E+01	-2.645793E+01	-63.5627	7.355944E+01	7.192030E+00	7.024013E+01
	584	-2.500000E-01	-2.675301E+01	2.563033E+01	-3.931691E+01	-61.8351	4.668083E+01	-4.780351E+01	8.182777E+01
		2.500000E-01	2.034735E+01	4.847903E+01	-2.645793E+01	-58.9982	6.437767E+01	4.448712E+00	6.227261E+01
	601	-2.500000E-01	-3.520815E+01	2.563033E+01	-3.931691E+01	-63.8644	4.492175E+01	-5.449957E+01	8.623446E+01
		2.500000E-01	4.019742E+01	4.847903E+01	-2.645793E+01	-49.4475	7.111823E+01	1.755822E+01	6.416685E+01
	600	-2.500000E-01	-3.520815E+01	3.497486E+01	-3.931691E+01	-65.8749	5.258282E+01	-5.281610E+01	9.127821E+01
		2.500000E-01	4.019742E+01	6.040412E+01	-2.645793E+01	-55.4500	7.862214E+01	2.197940E+01	7.025999E+01

1 MSC.NASTRAN JOB CREATED ON 20-MAR-03 AT 14:10:38 MARCH 24, 2003 MSC.NASTRAN 2/ 4/02 PAGE 1980
TORQUE
0 SUBCASE 3

S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4) OPTION = BILIN

ELEMENT ID	GRID-ID	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY	PRINCIPAL STRESSES (ZERO SHEAR) ANGLE	MAJOR	MINOR	VON MISES
0 408	CEN/4	-2.500000E-01	-2.327065E+01	-3.584448E+00	-7.161915E+01	-48.9128	5.886483E+01	-8.571993E+01	1.259320E+02
		2.500000E-01	1.933862E+01	1.511258E+01	-5.288489E+01	-43.8560	7.015269E+01	-3.570149E+01	9.327674E+01
	583	-2.500000E-01	-1.787524E+01	3.159664E+00	-7.161915E+01	-49.1772	6.502950E+01	-7.974507E+01	1.255942E+02
		2.500000E-01	1.436517E+01	2.310488E+01	-5.288489E+01	-47.3618	7.180015E+01	-3.433010E+01	9.380151E+01
	584	-2.500000E-01	-1.787524E+01	-1.032856E+01	-7.161915E+01	-46.5080	5.761658E+01	-8.582038E+01	1.250179E+02
		2.500000E-01	1.436517E+01	7.120276E+00	-5.288489E+01	-43.0408	6.375153E+01	-4.226609E+01	9.244029E+01
	601	-2.500000E-01	-2.866605E+01	-1.032856E+01	-7.161915E+01	-48.6477	5.270635E+01	-9.170096E+01	1.265711E+02
		2.500000E-01	2.431207E+01	7.120276E+00	-5.288489E+01	-40.3839	6.929510E+01	-3.786275E+01	9.412280E+01
	600	-2.500000E-01	-2.866605E+01	3.159664E+00	-7.161915E+01	-51.2634	6.061247E+01	-8.611886E+01	1.277114E+02
		2.500000E-01	2.431207E+01	2.310488E+01	-5.288489E+01	-44.6730	7.659681E+01	-2.917986E+01	9.462357E+01

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Stresses at Node 600 for the four elements**

PRESSURE

S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)

ELEMENT ID	GRID-ID	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY
407	CEN/4	-2.500000E-01	1.674251E+03	4.685103E+02	-4.019310E+00
		2.500000E-01	1.935774E+03	1.266394E+03	-5.718250E+00
	600	-2.500000E-01	1.687446E+03	4.665695E+02	-4.019310E+00
		2.500000E-01	1.953846E+03	1.263198E+03	-5.718250E+00
408	CEN/4	-2.500000E-01	1.676117E+03	4.663313E+02	-6.182010E+00
		2.500000E-01	1.935953E+03	1.262477E+03	-3.102248E+00
	600	-2.500000E-01	1.693006E+03	4.671432E+02	-6.182010E+00
		2.500000E-01	1.954141E+03	1.263238E+03	-3.102248E+00
423	CEN/4	-2.500000E-01	1.537250E+03	9.216394E+02	-5.154676E+00
		2.500000E-01	1.538958E+03	8.174963E+02	-8.579890E+00
	600	-2.500000E-01	1.823384E+03	9.180715E+02	-5.154676E+00
		2.500000E-01	1.819177E+03	8.125279E+02	-8.579890E+00
424	CEN/4	-2.500000E-01	1.542389E+03	9.149080E+02	-1.025905E+01
		2.500000E-01	1.540857E+03	8.092199E+02	-4.677585E+00
	600	-2.500000E-01	1.827579E+03	9.196488E+02	-1.025905E+01
		2.500000E-01	1.818164E+03	8.130617E+02	-4.677585E+00

INERTIAL

S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)

ELEMENT ID	GRID-ID	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY
407	CEN/4	-2.500000E-01	-2.574122E+01	3.744400E+01	-1.712740E+01
		2.500000E-01	2.652378E+01	5.931532E+01	-2.045111E+01
	600	-2.500000E-01	-2.478440E+01	3.652250E+01	-1.712740E+01
		2.500000E-01	3.528828E+01	5.930370E+01	-2.045111E+01
408	CEN/4	-2.500000E-01	-3.098058E+01	3.030259E+01	-3.931691E+01
		2.500000E-01	3.027239E+01	5.444157E+01	-2.645793E+01
	600	-2.500000E-01	-3.520815E+01	3.497486E+01	-3.931691E+01
		2.500000E-01	4.019742E+01	6.040412E+01	-2.645793E+01
423	CEN/4	-2.500000E-01	-3.289067E+01	2.657005E+01	-1.895104E+01
		2.500000E-01	4.783808E+01	9.499256E+01	-1.675141E+01
	600	-2.500000E-01	-2.804659E+01	2.487978E+01	-1.895104E+01
		2.500000E-01	4.599145E+01	9.846719E+01	-1.675141E+01
424	CEN/4	-2.500000E-01	-4.809116E+01	1.762577E+01	-3.382100E+01
		2.500000E-01	5.065319E+01	9.345893E+01	-2.234502E+01
	600	-2.500000E-01	-3.901119E+01	2.033376E+01	-3.382100E+01
		2.500000E-01	5.190263E+01	9.929759E+01	-2.234502E+01

TORQUE

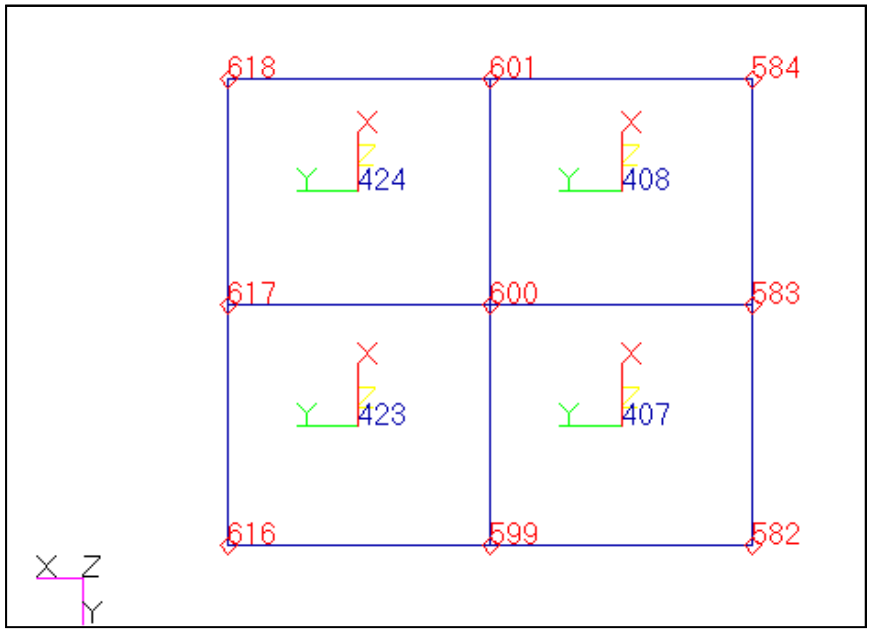
S T R E S S E S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)

ELEMENT ID	GRID-ID	FIBER DISTANCE	STRESSES IN ELEMENT COORD SYSTEM NORMAL-X	NORMAL-Y	SHEAR-XY
407	CEN/4	-2.500000E-01	-2.181223E+01	8.026799E+00	-5.820134E+01
		2.500000E-01	2.084371E+01	2.645805E+01	-5.187624E+01
	600	-2.500000E-01	-2.214953E+01	3.551084E+00	-5.820134E+01
		2.500000E-01	2.703867E+01	2.360251E+01	-5.187624E+01
408	CEN/4	-2.500000E-01	-2.327065E+01	-3.584448E+00	-7.161915E+01
		2.500000E-01	1.933862E+01	1.511258E+01	-5.288489E+01
	600	-2.500000E-01	-2.866605E+01	3.159664E+00	-7.161915E+01
		2.500000E-01	2.431207E+01	2.310488E+01	-5.288489E+01
423	CEN/4	-2.500000E-01	-2.986038E+01	-1.548602E+00	-5.659727E+01
		2.500000E-01	3.449507E+01	4.799138E+01	-4.757421E+01
	600	-2.500000E-01	-2.502215E+01	-4.819981E+00	-5.659727E+01
		2.500000E-01	3.349866E+01	4.475710E+01	-4.757421E+01
424	CEN/4	-2.500000E-01	-3.719154E+01	-1.015439E+01	-6.271164E+01
		2.500000E-01	2.612406E+01	3.209285E+01	-4.805843E+01
	600	-2.500000E-01	-3.063703E+01	-6.977973E+00	-6.271164E+01
		2.500000E-01	2.940615E+01	4.220444E+01	-4.805843E+01

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Element coordinate system for each of the four elements.**
  - Under the Meshing tab: Show/Element/Coord. Sys.
- **This is of use in determining how the element stress components should be combined.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Combine the three load case results for fiber distance + 2.5E-01 (Patran layer Z2)**
  - Pressure + 6.0 * Inertial + Torque

		Stresses in Elm Coord System		
Element ID	**Grid ID**	**Normal x**	**Normal y**	**Shear xy**
407	600	2192.61	1642.62	-180.30
408	600	2219.44	1640.75	-214.74
423	600	2128.62	1448.11	-156.65
424	600	2158.97	1451.06	-186.84

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Calculate the Von Mises stress**

		Stresses in Elm Coord System			Stress
Element ID	**Grid ID**	**Normal x**	**Normal y**	**Shear xy**	**Von Mises**
407	600	2192.61	1642.62	-180.30	2000.7
408	600	2219.44	1640.75	-214.74	2028.8
423	600	2128.62	1448.11	-156.65	1902.7
424	600	2158.97	1451.06	-186.84	1933.9

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Compare von Mises stress from manual calculation with that from Patran:**
  - Average manual von Mises stress at Node 600 = 1966.5
  - Average Patran von Mises stress at Node 600 = 1966.8

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Look at the stresses at the centroid for the four associated elements.**
- **Combine the three load case results for fiber distance + 2.5E-01 (Patran layer Z2).**
  - Pressure + 6.0 * Inertial + Torque

		Stresses in Elm Coord System		
Element ID	**Location**	**Normal x**	**Normal y**	**Shear xy**
407	Centroid	2115.73	1647.75	-180.30
408	Centroid	2136.91	1604.23	-214.74
423	Centroid	1860.50	1435.43	-156.65
424	Centroid	1870.88	1402.07	-186.84

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FRINGE PLOT OPTIONS

- **Compare the average x, y, and xy stress component from manual calculation and Patran:**

		Stresses in Elm Coord System		
Source	**Location**	**Normal x**	**Normal y**	**Shear xy**
Manual	Node 600	1996.0	1522.4	-184.6
Patran	Node 600	1996.0	1522.6	-184.6

Copyright© 2012 MSC.Software Corporation

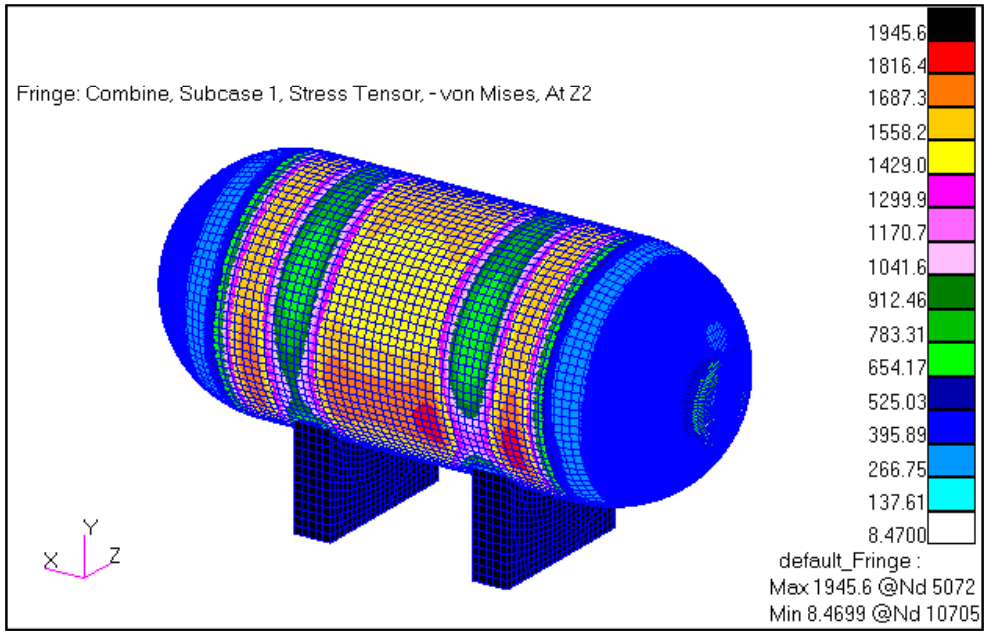
CASE STUDY: FRINGE PLOT OPTIONS

- **It is important to determine how stresses are being calculated by Patran for the options that are being used.**
- **Use the Patran user guide as needed.**

Copyright© 2012 MSC.Software Corporation

SPECTRUMS AND RANGES

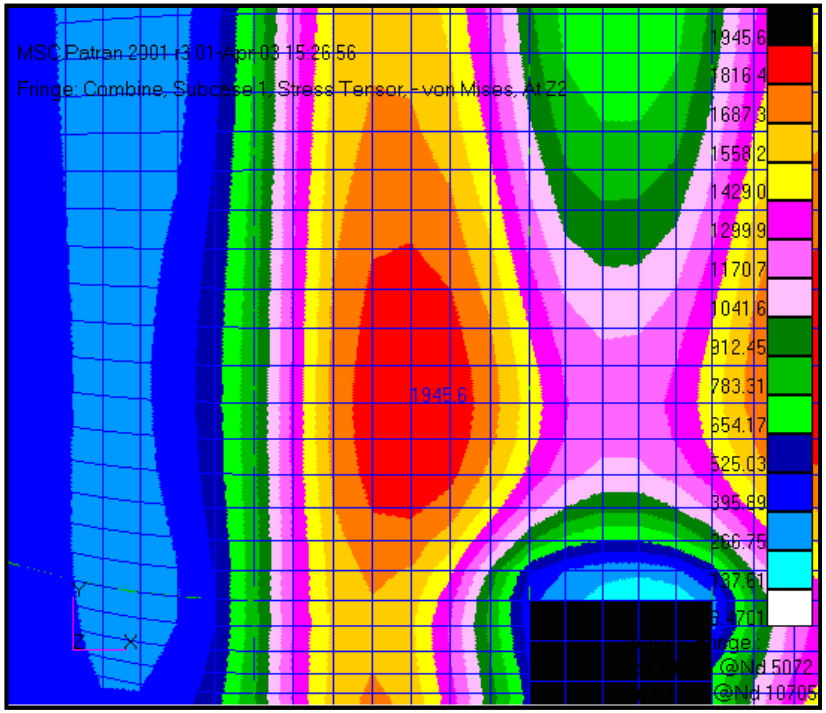
- **It is possible to display a stress fringe in different viewports using the same range.**
- **Fringe colors can be interpolated.**

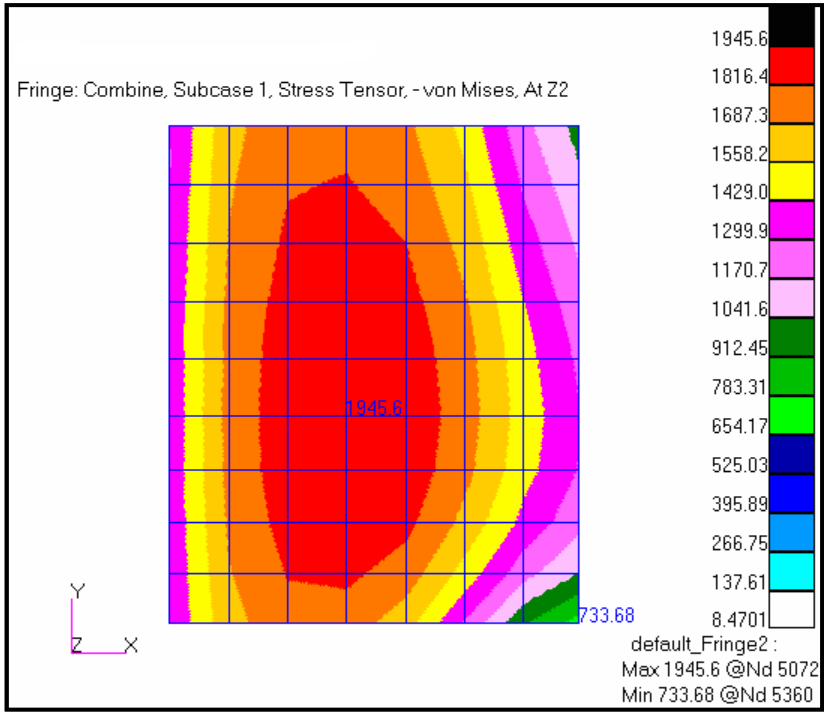


Copyright© 2012 MSC.Software Corporation

SPECTRUMS AND RANGES

- **Two viewports, each with a fringe, both with the same range:**

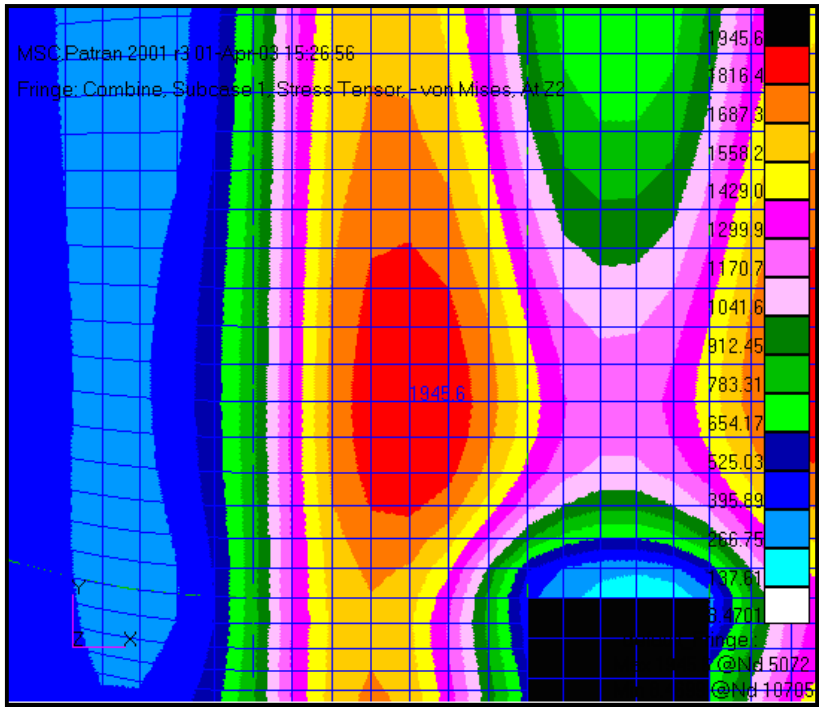


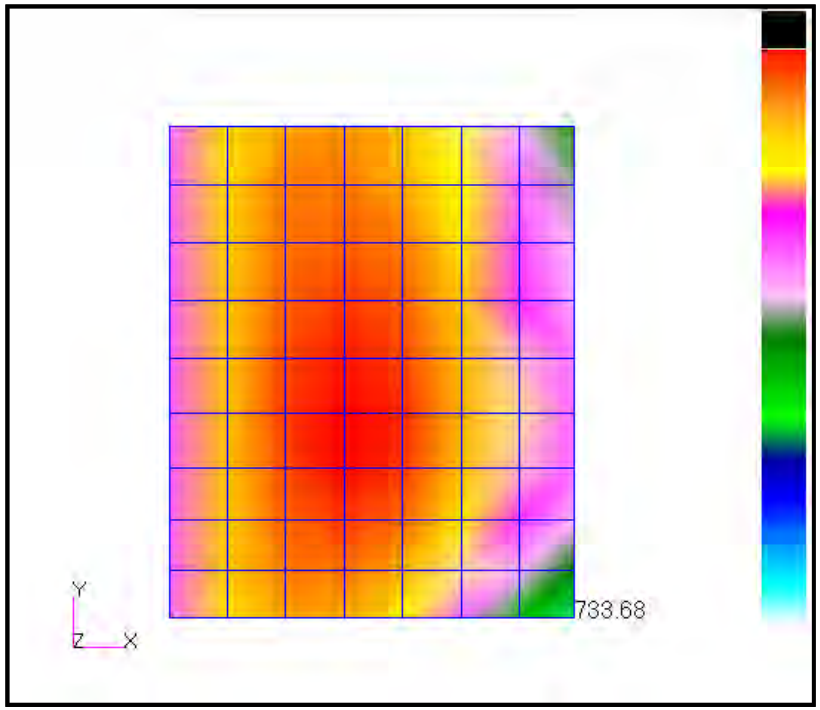


Copyright© 2012 MSC.Software Corporation

SPECTRUMS AND RANGES

- **The same two viewports with the same fringes, but one fringe is parabolically interpolated:**





Copyright© 2012 MSC.Software Corporation

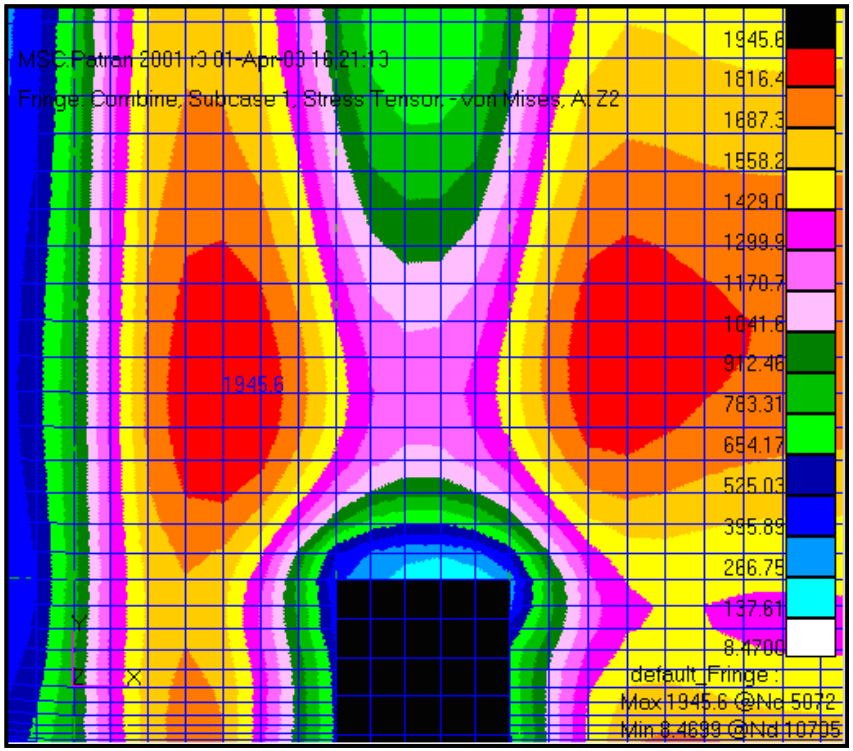
SPECTRUMS AND RANGES

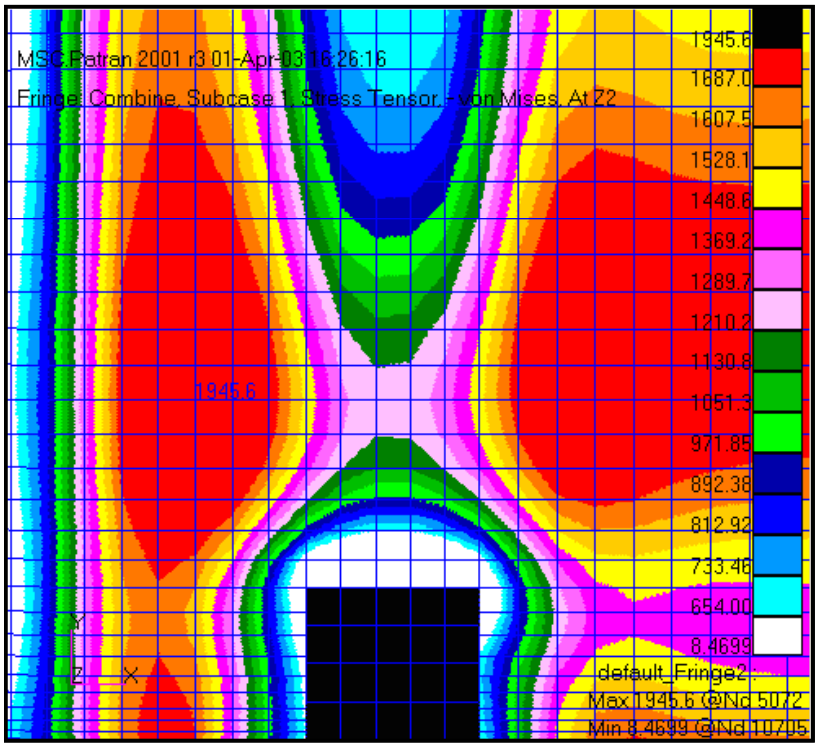
- **A fringe can be interpolated using the GUI in two different ways:**
  - Results: Create/Fringe
    - Display Attributes/Spectrum
  - **Or**
  - Results: Create/Fringe
    - Display Attributes/Style: Continuous
- **It may be necessary to use hardware rendering.**

Copyright© 2012 MSC.Software Corporation

SPECTRUMS AND RANGES

- **Two fringes with the same range, but one with thresholding at both ends of its spectrum:**

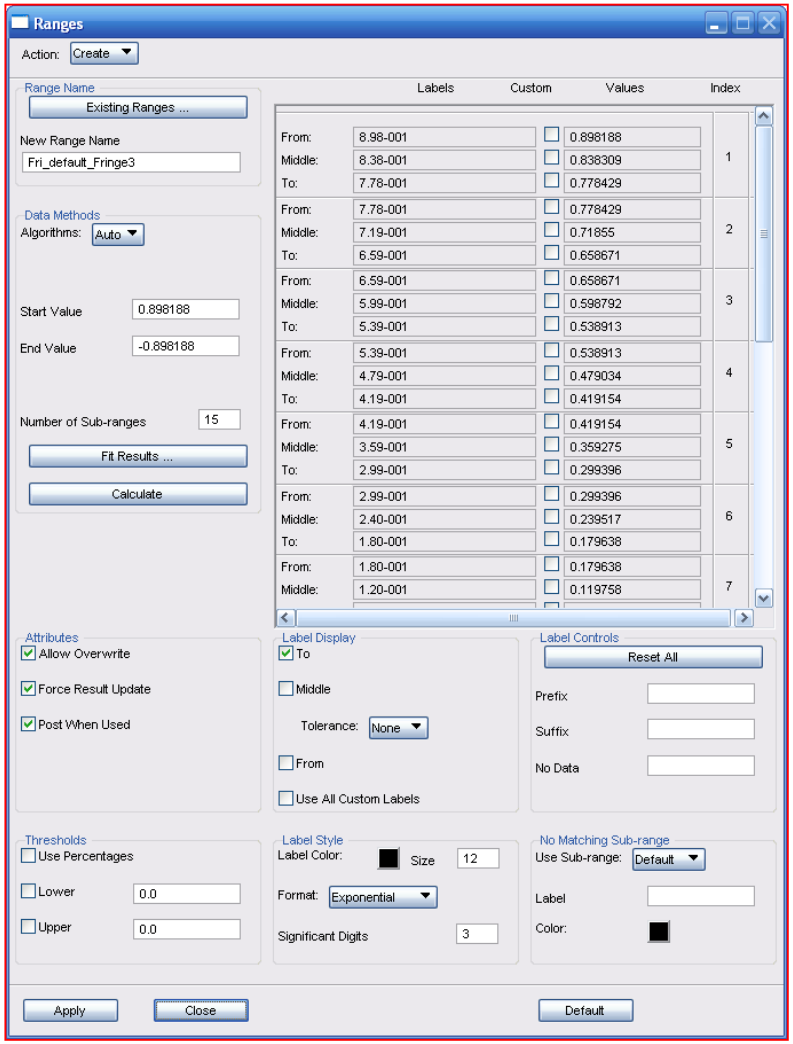




Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

- **To create a new range**
  - Results: Create / Fringe
  - Click on Display Attributes
  - Select Range and click on Define Range
- **Display: Ranges in the Patran main menu**



Copyright© 2012 MSC.Software Corporation

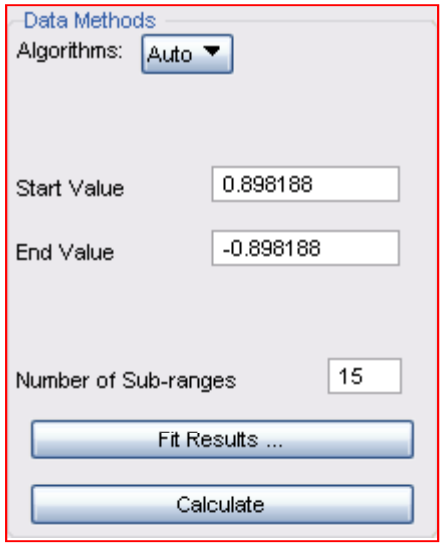
CREATING A RANGE

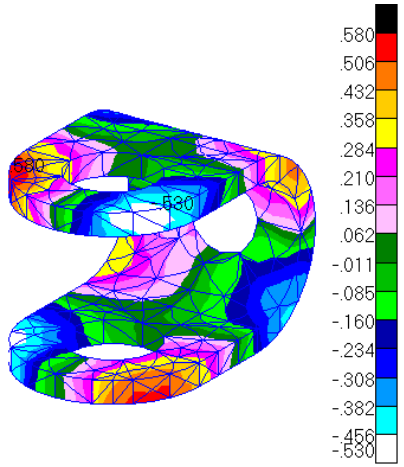
- **Under Ranges several Actions can be performed, e.g. Create, Copy, Modify, etecetra.**
- **Existing Ranges can be used if desired; if not, enter a new name under New Range Name**
- **Calculate Range under Data Methods**
  - The method of calculating the Range is specified under Algorithms

Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

- **Calculate Range under Data Methods (continued)**
  - Enter the Number of Sub-ranges, click Fit Results, then select posted or un-posted plot
  - Click Calculate to calculate the Range, and enter the values into the spreadsheet



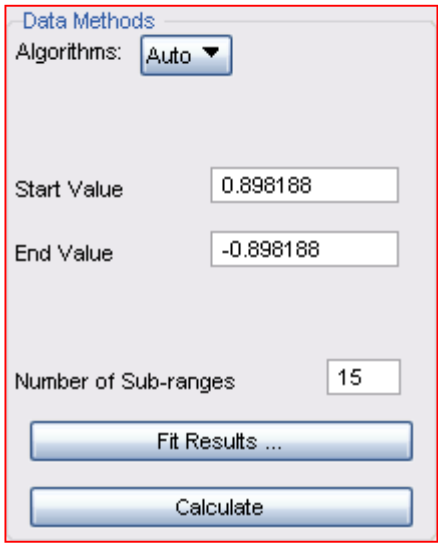


	Labels	Custom	Values	Index
From:	.580		0.580279	1
Middle:	.543		0.543279	
To:	.506		0.50628	
From:	-.456		-0.45571	15
Middle:	-.493		-0.492709	
To:	-.530		-0.529709	

Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

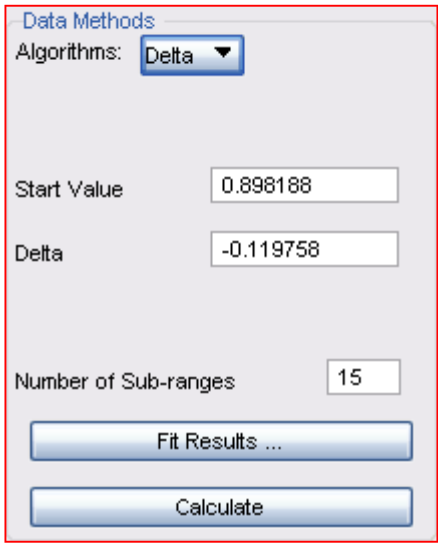
- **ALGORITHMS**
  - Auto
    - Allows the specification of Start Value (upper end of Range) and End Value (lower end of Range)
    - Difference between any two adjacent values is constant
    - Clicking Fit Results button automatically enters the current plot's maximum value in Start Value and the plot's minimum value in End Value
    - If this is all that is to be done make sure to press the Calculate button to calculate the Range values and enter them into the spreadsheet



Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

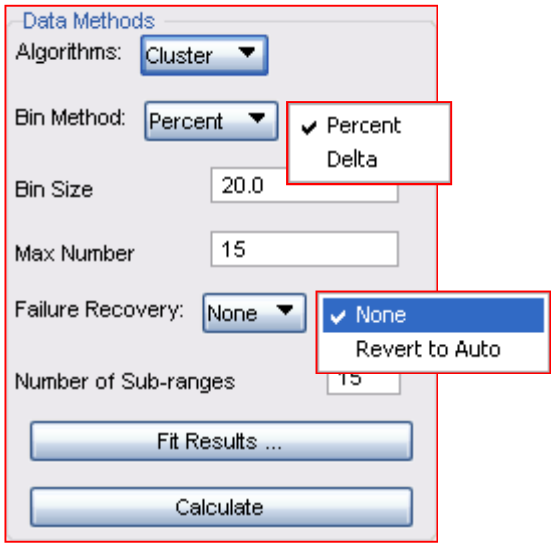
- Delta
  - Specify the Start Value (upper end of Range) and a value to increment or decrement by (Delta) (e.g. start @10000 and increment by –500 to obtain a range of 10000, 9500, 9000, etc.)
  - Clicking Fit Results button automatically enters the current plot's maximum value in Start Value, and the Delta value necessary to define the entire Range
  - If this is all that is to be done make sure to press the Calculate button to calculate the Range values and enter them into the spreadsheet



Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

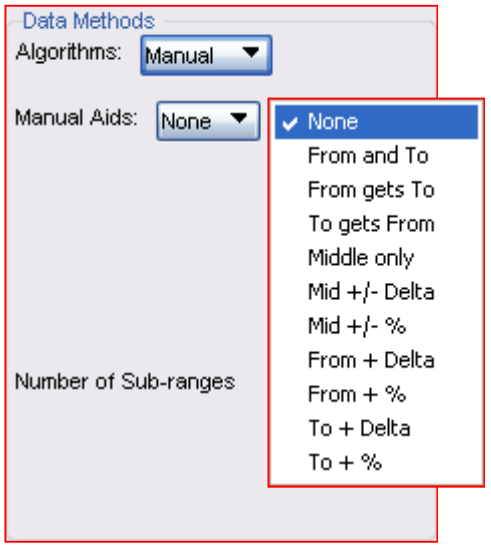
- **Cluster**
  - Data is D1 < D2 < D3 < … < Dn
  - First Sub-range
    - Lower end: has D1
    - Upper end:
      - Has D2 IF D2 – D1 <= Bin Size
      - Does not have D2 IF D2 – D1 > Bin Size
  - Second Sub-range
    - Lower end
      - Has D2 IF D2 – D1 > Bin Size
      - Has D3 IF D2 is upper end of First Sub-range
    - Upper end
      - Has D3 IF
      - D3 not lower end of second Sub-range, AND
      - IF D3 – D2 <= Bin Size
  - Other Sub-ranges …



Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

- Manual
  - Define Sub-ranges by entering data into the cells in the Values column of the Ranges spreadsheet
  - None – data must be entered into all cells
  - From and To – data must be entered into all From and To cells; Middle values are calculated
  - From gets To – data must be entered into all From cells; the To values are set equal to the adjacent Sub-range From values; Middle values are calculated
  - To gets From - data must be entered into all To cells; the From values are set equal to the adjacent Sub-range To values; Middle values are calculated



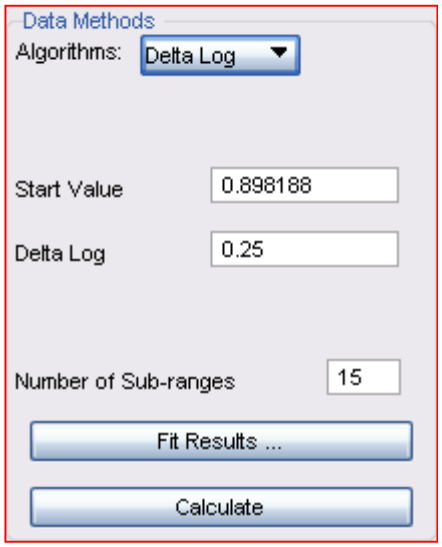
Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

- **Delta Log**
  - The (upper) end value of a Sub-range is equal to its (lower) end value times (x) 10 to the power of Delta Log

$$End\ Sub-range\ n = Start\ Value \cdot \prod_{i=1}^{n} 10^{Delta\ Log}$$

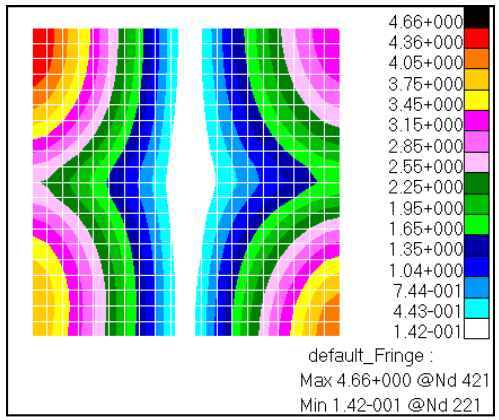
  - Example
    - End of Sub-range 2
    - Start Value = 5.0
    - Delta Log = 0.5
    - End Sub-range 2 = 5.0 x 100.5 x 100.5 = 50.0

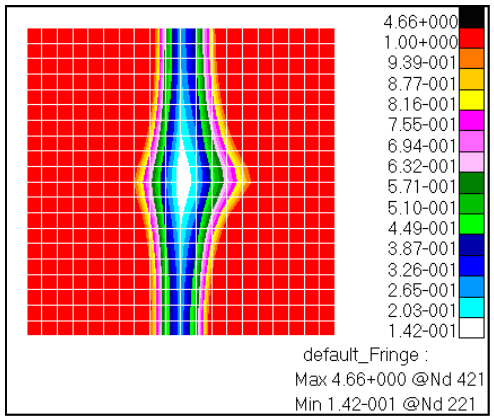


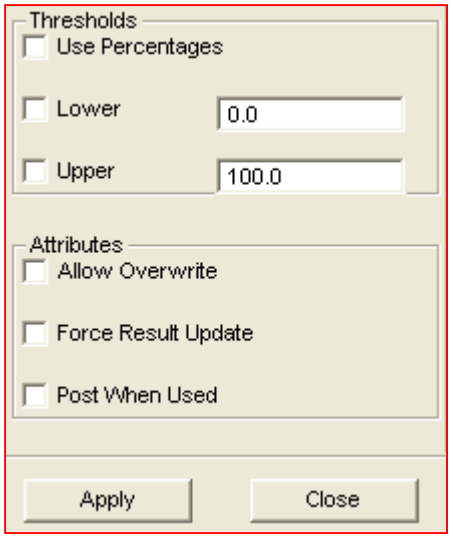
Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

- **Thresholds are used to manually specify the size of the upper and lower Sub-range. The intermediate Sub-ranges are created using the Data Methods Algorithm selected, e.g. Auto.**
- **After defining Thresholds Sub-ranges, click Apply**



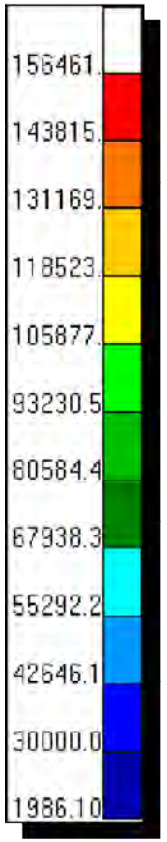




Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

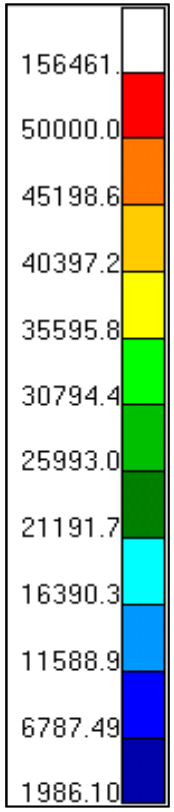
- **Lower**
  - Specify a Range value just above the minimum value of the Range
  - All values above this value will be equally spaced to the value on the top of the Range
  - Example
    - Want to see all values from 30,000psi to the minimum value shown in blue
    - All values between 30,000psi and the maximum value are shown with different spectrum colors



Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

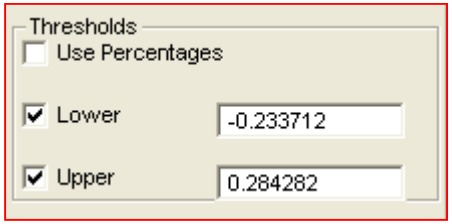
- **Upper**
  - Specify a Range value just below the maximum value of the Range
  - All values below this value will be equally spaced to the value at the bottom of the Range
  - Example
    - Want to see all values from 50,000psi to the maximum value shown in red
    - All values between the minimum value and 50,000psi are shown with different spectrum colors

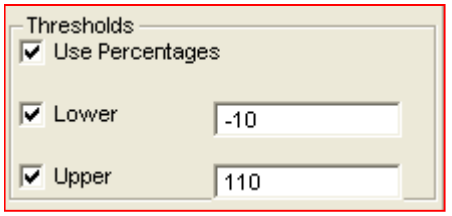


Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

- **Use Percentages**
  - Specify Lower and Upper as result values
    - If a result value is entered then the Lower and Upper sub-range intervals will be defined to have a value of the minimum result value and maximum result value, respectively.
  - Specify Lower and Upper as percent of the result values
    - Negative Percent Values produce lower thresholds below the lower range value. Percent Values greater than 100% produce threshold values above the upper range value. Percent values between 0% and 100% produce threshold values between the lower and upper range values.





Copyright© 2012 MSC.Software Corporation

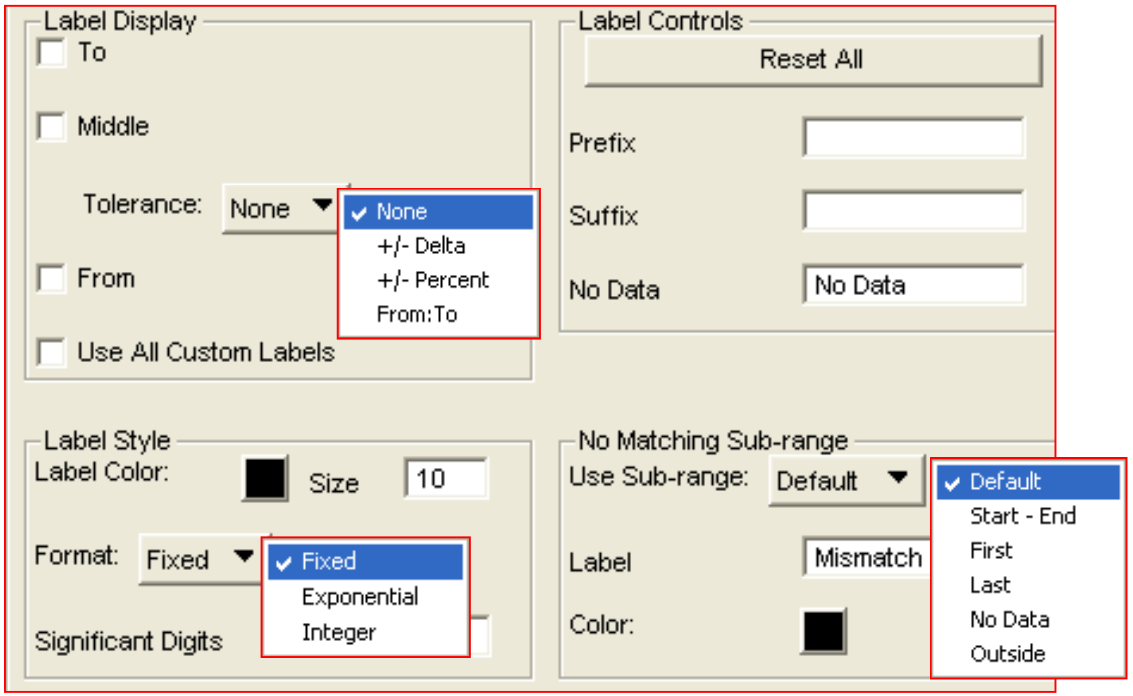
CREATING A RANGE

- **Attributes**
  - Allow Overwrite
    - If unselected this option will not allow the range's definition to be modified when attributes of a tool that uses it are changed. This is to prevent accidental overwriting of the range by an auto-range result plot.
  - Force Result Update
    - When selected it allows the range definition to be modified if the results value associated to a plot that uses the range is changed. It causes the sub-range intervals to be recalculated every time the range is used. It does not work in conjunction with Data Methods/Algorithms Manual.
  - Post When Used
    - This option causes the range to be posted to a viewport if a Results tool that is posted to the same viewport references it.

Copyright© 2012 MSC.Software Corporation

CREATING A RANGE

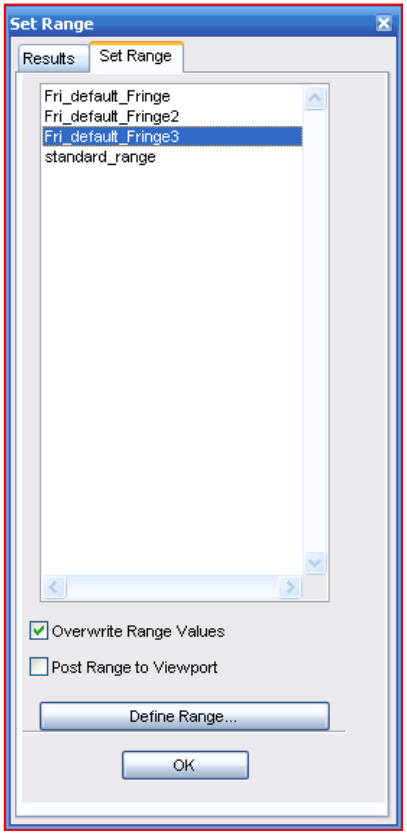
- **LABEL, ETC.**
  - Selections for Label, etc.



Copyright© 2012 MSC.Software Corporation

RANGE CONTROL FORM

- **Appears only when select Range… from any Fringe or Marker plot form using Display Attributes/Range…**
- **Control whether the values for the Range are overwritten for each new plot**
- **Control posting a range to a given viewport**



Copyright© 2012 MSC.Software Corporation

RESULTS PROCESSING SETTINGS

- **The following are some settings.pcl file variables that are applicable to Patran Results processing:**

Variable name	Purpose	Type	Possible values
result_capture_ filename	Sets the default output report filename	String	Desired file name
result_use_title_ variables	Causes title in Display Attributes to be shown, or not	Logical	TRUE or FALSE

Copyright© 2012 MSC.Software Corporation

RESULTS PROCESSING SETTINGS

- **settings.pcl file variables (Cont.)**

Variable name	Purpose	Type	Possible values
result_loadcase_abbreviate	Cause abbreviation if number of result cases exceeds a value	Integer	Number of result cases to cause abbreviation of result case names
result_quick_avg_domain	Sets the default averaging domain for Quick Plot	String	All, Material, Property, Etype, Target, Element
result_quick_avg_method	Sets the default averaging method for Quick Plot	String	DeriveAverage, AverageDerive, Difference, Sum

Copyright© 2012 MSC.Software Corporation

RESULTS PROCESSING SETTINGS

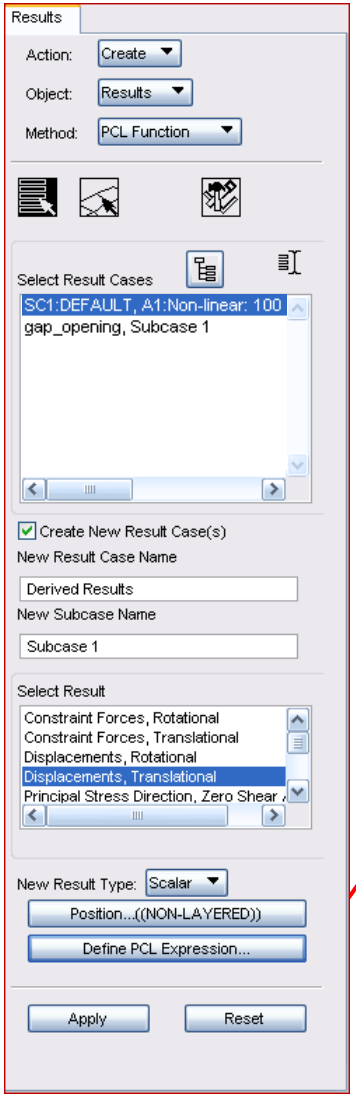
- **settings.pcl file variables (Cont.)**

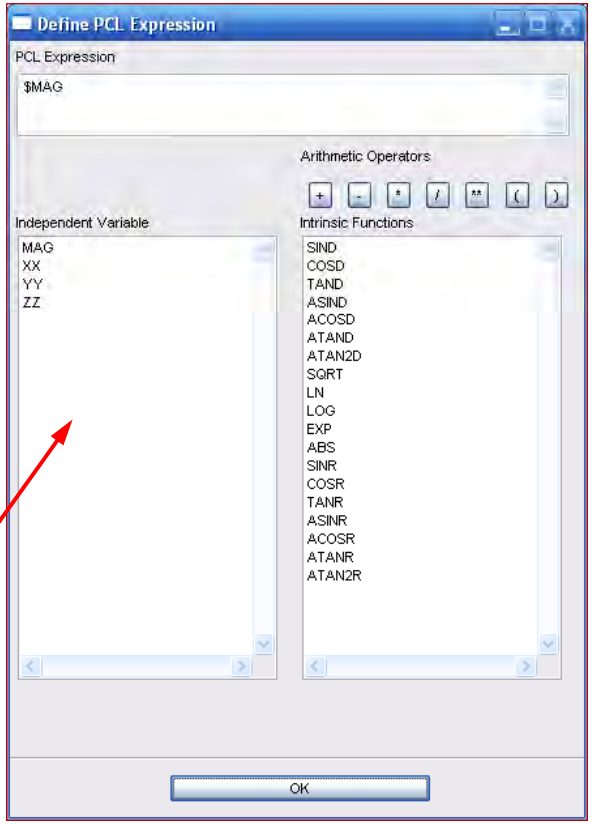
Variable name	Purpose	Type	Possible values
result_quick_ transform	Sets the default coordinate transformation for Quick Plot	String	Default, Global, CID, ProjectedCID, None, Material, ElementIJK
result_quick_ extrap_ method	Sets the extrapolation method for Quick Plot	String	ShapeFunc, Average, Centriod, Max, Min, AsIs

Copyright© 2012 MSC.Software Corporation

PCL PROCESSING OF RESULTS

- **PCL can be used to process results data in Patran.**
  - Results: Create/Results/PCL Function

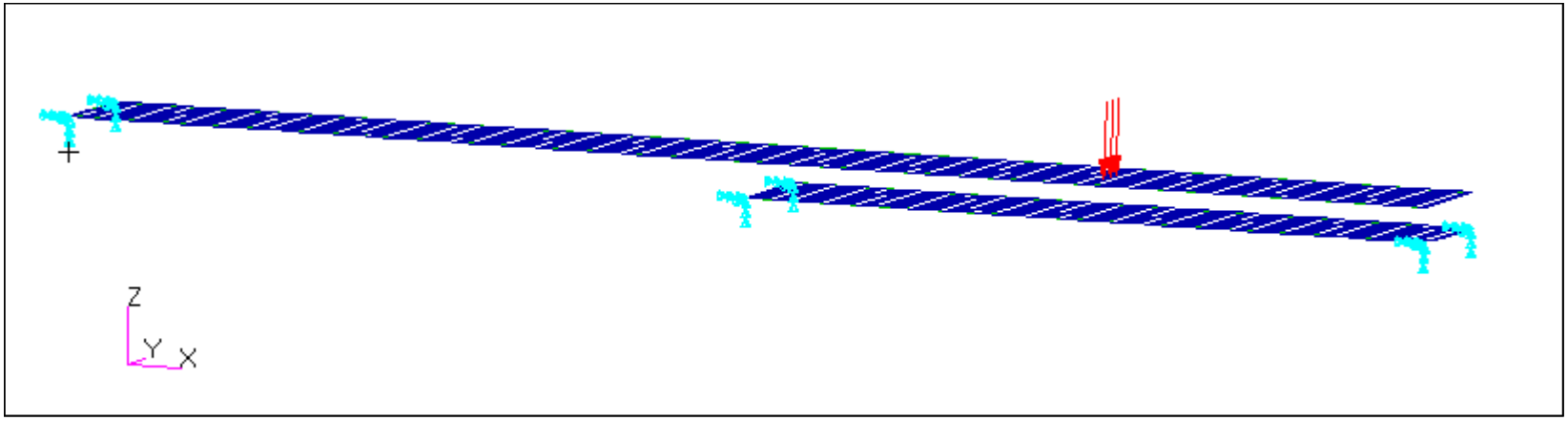




Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

- **A cantilevered plate is loaded with forces at ¾ of its length from its fixed end. The plate is above a fixed surface. As the forces on the plate are increased, the plate contacts the fixed surface.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

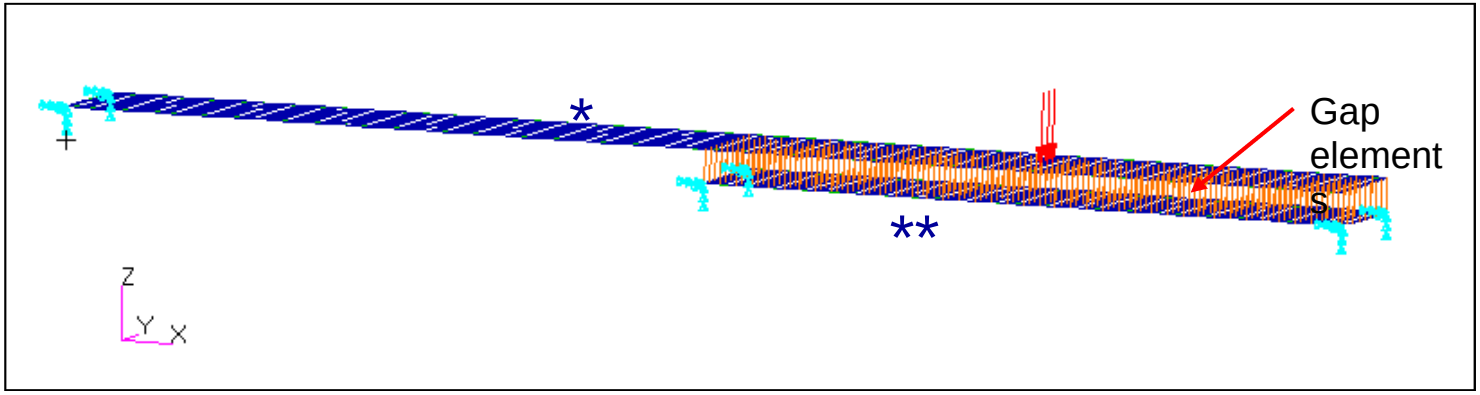
- **Create the PCL function**
  - The goal of this function is to use the gap displacement results read into Patran to determine whether a gap is closed or open.
  - Open a text editor and enter the following commands, and save as a file named, gap_results_function.pcl.

```
FUNCTION gap_results(x_disp)

REAL x_disp, gap_opening

IF (x_disp < 0.2499) THEN
     gap_opening = 1
ELSE
     gap_opening = 0
END IF

RETURN gap_opening

END FUNCTION /* gap_results */
```


Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

- **The model in Patran consists of**
  - Cantilevered plate (*) quad4 mesh
    - Plate is 10” x 1” x 0.1” thick
    - GEL = 0.125”
    - Aluminum: E=10e6 psi, Nu=0.3
    - Constrain one end
  - Quad4 mesh for the fixed surface (**)
    - Surface is 5” x 1”
    - GEL = 0.125”
    - Constrain all nodes



Copyright© 2012 MSC.Software Corporation

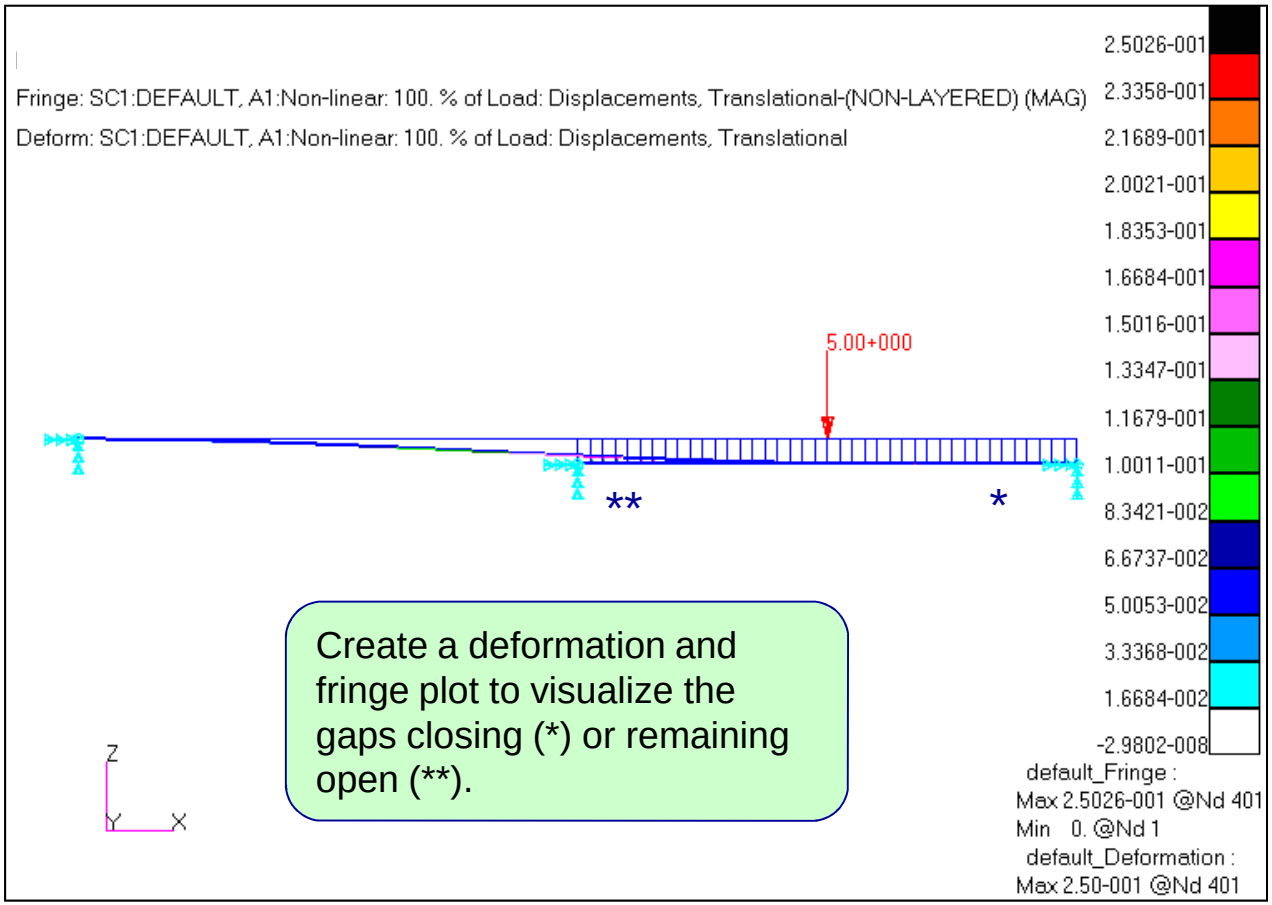
CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

- **The model in Patran consists of (CONT.)**
  - Gap elements connecting ½ of cantilevered plate nodes and fixed surface nodes
    - Initial gap opening is 0.25”
    - Closed gap stiffness is 1000 lbf/in
  - Forces on cantilevered plate
    - 7.5” from fixed end
    - On three nodes nearest to the centerline of plate
    - 5 lbf per node

- **Run the nonlinear static analysis**

Copyright© 2012 MSC.Software Corporation

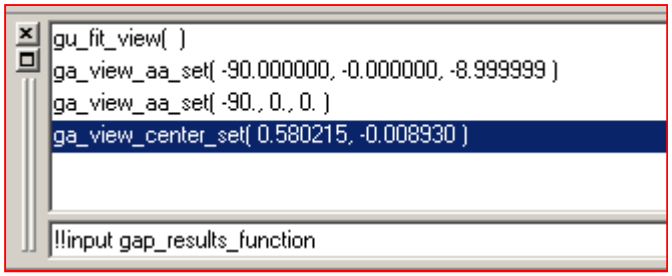
CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

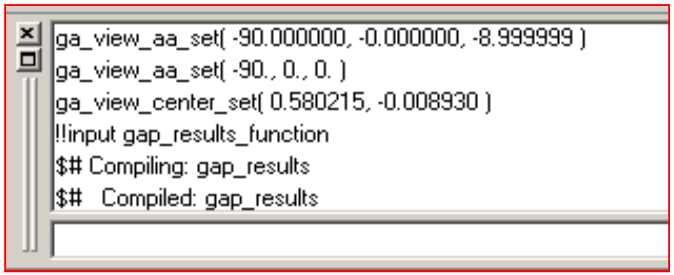


Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

- **Compile the PCL function in Patran:**

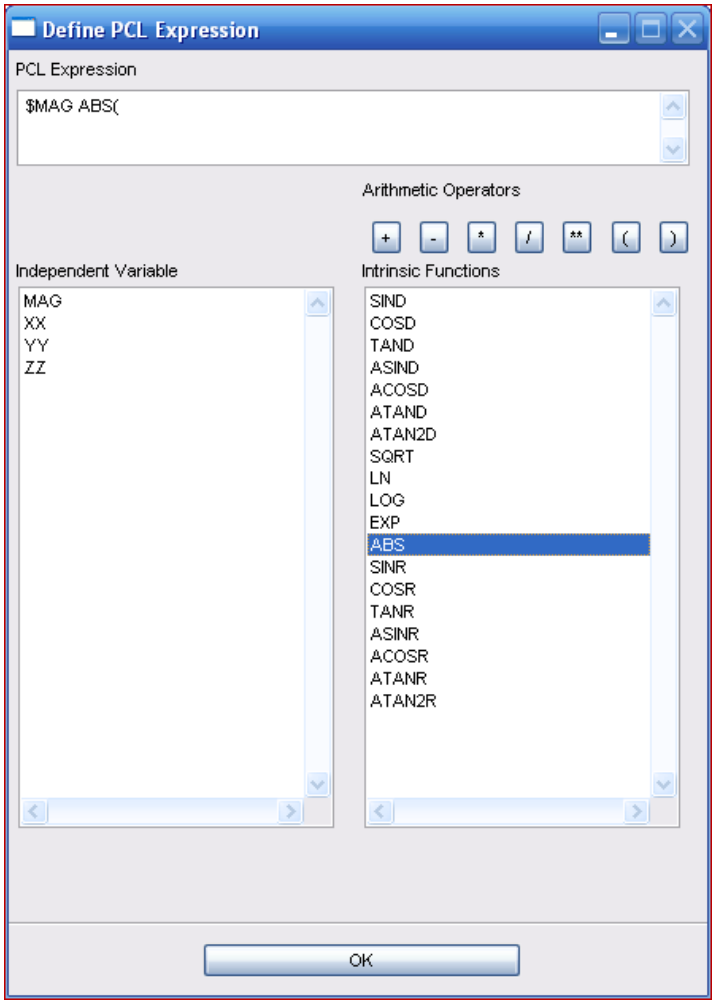


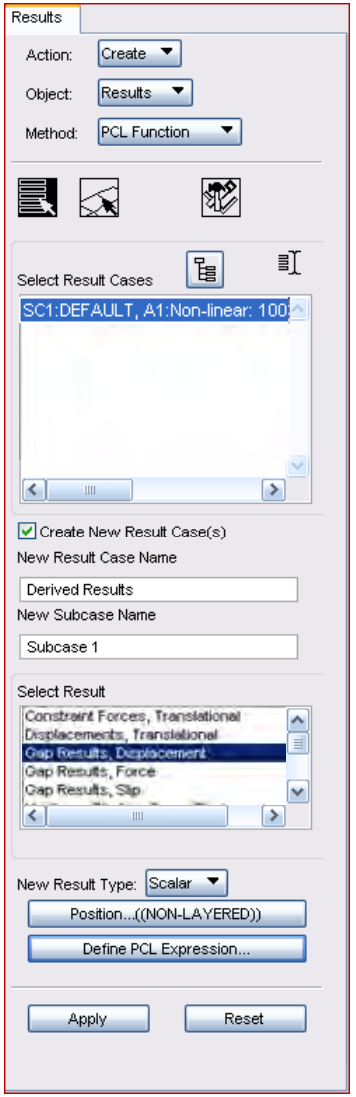


Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

- **Create a result case using the compiled PCL function.**

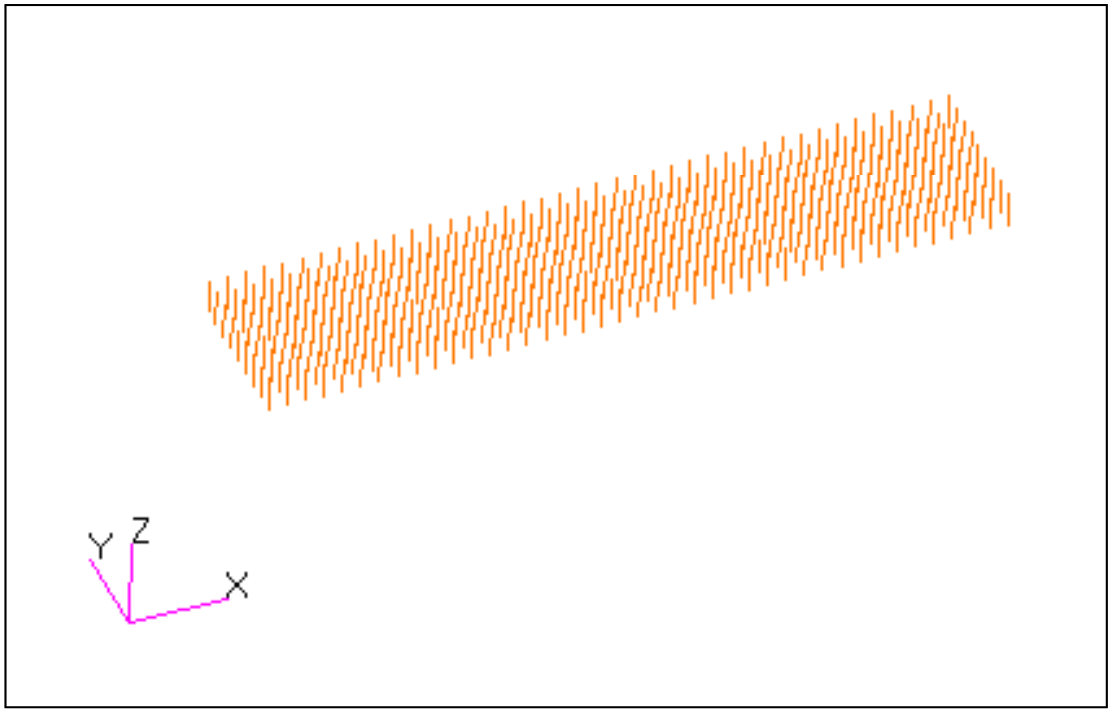


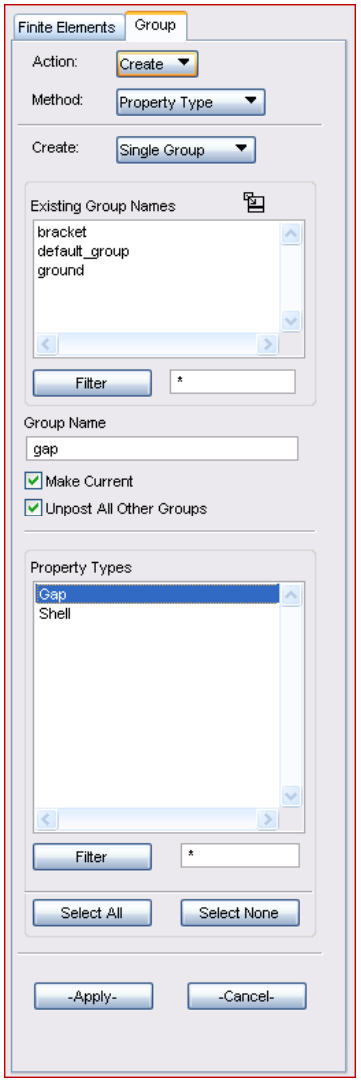


Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

- **Create a group from just the gap elements and associated nodes.**

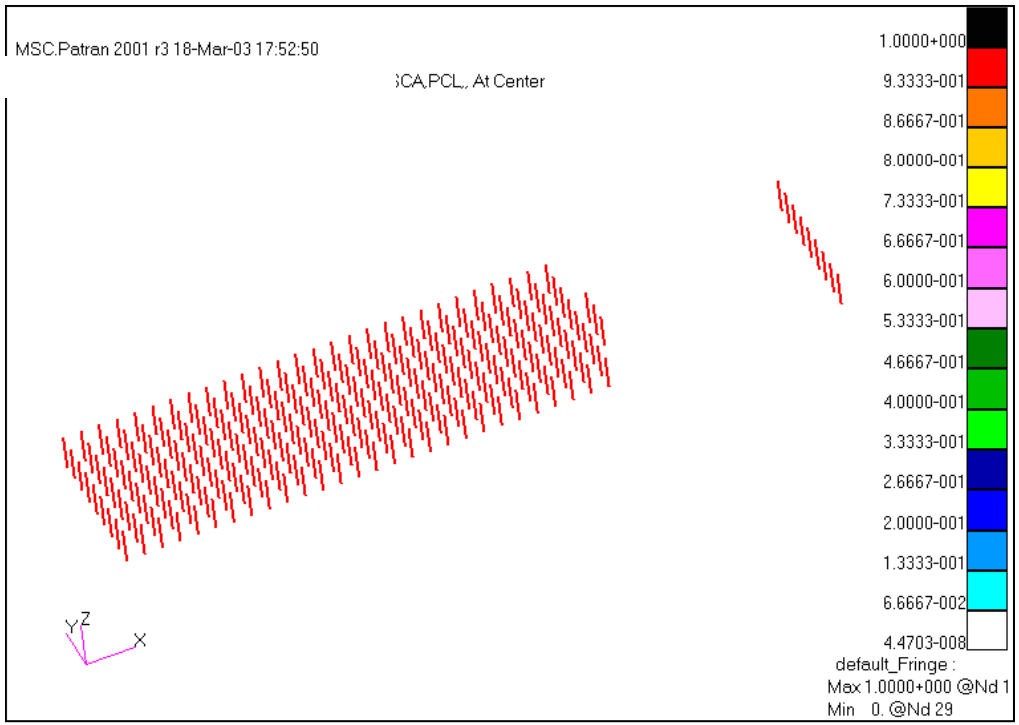


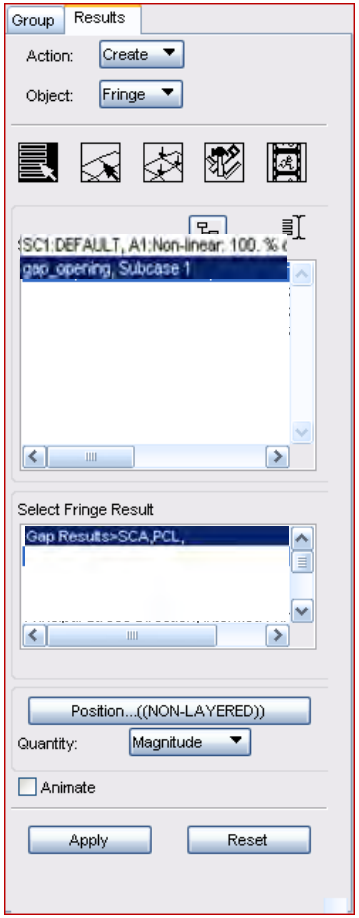


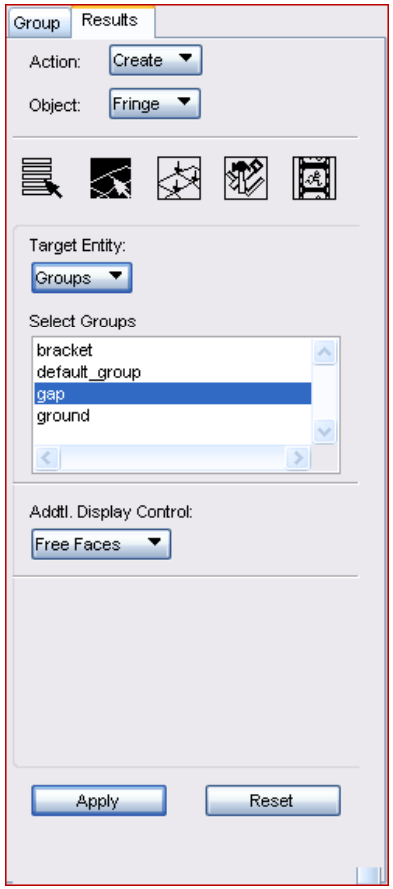
Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

- **Create a fringe plot using the new result case and the group of the gap elements.**



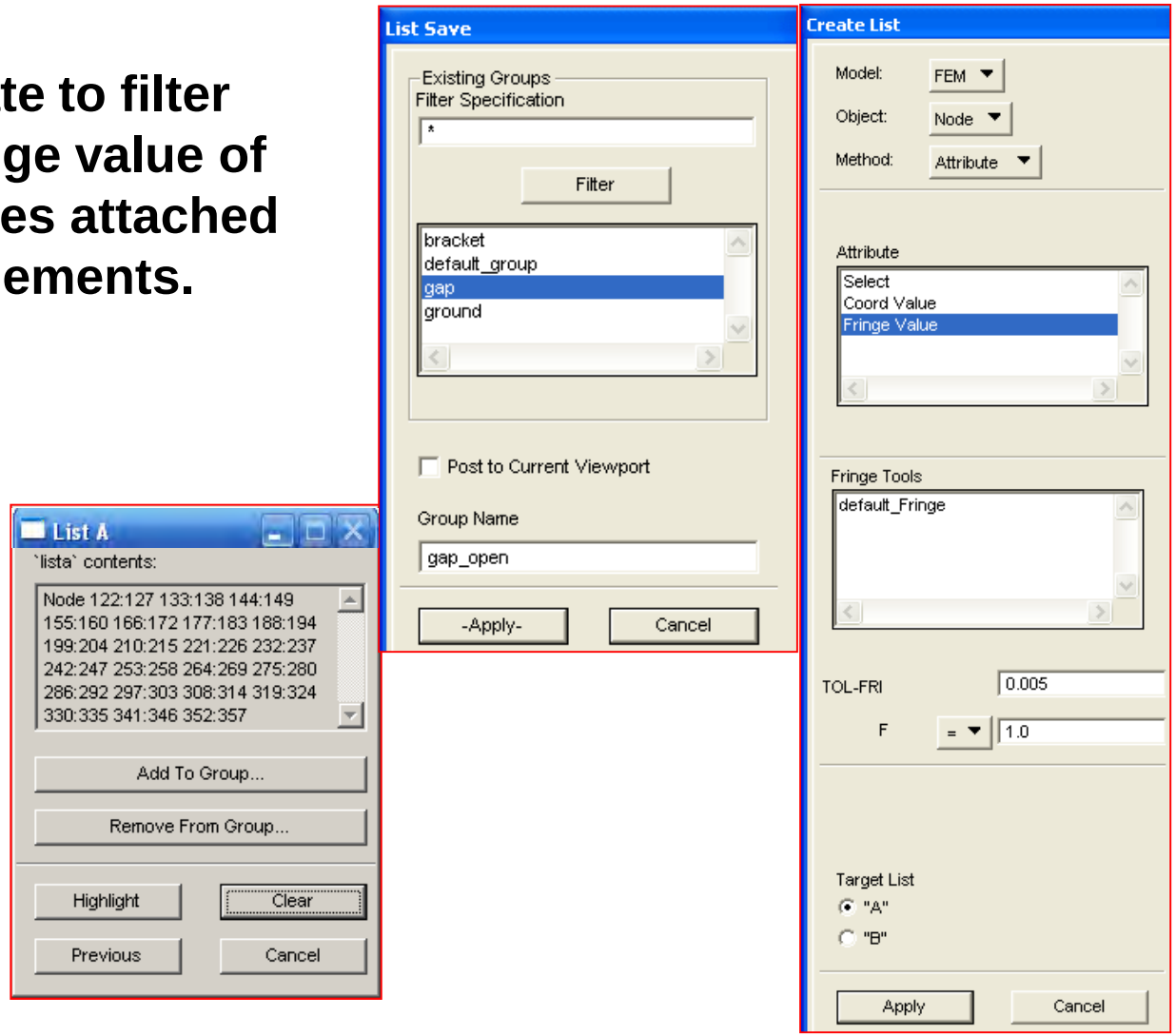




Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

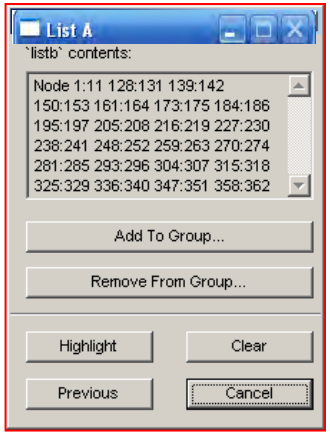
- **Use Tools: List/Create to filter the nodes with a fringe value of 1. These are the nodes attached to the opened gap elements.**

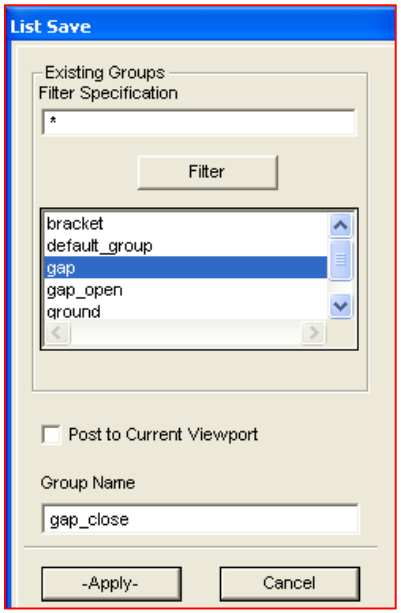


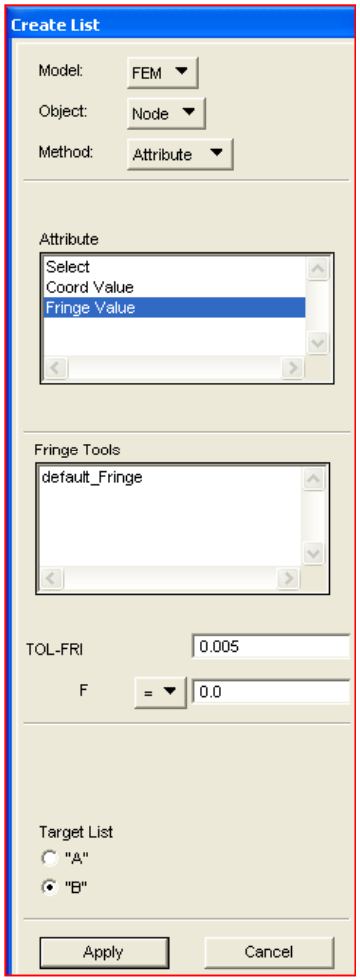
Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

- **Use Tools: List/Create to filter the nodes with a fringe value of 0. These are the nodes attached to the closed gap elements.**



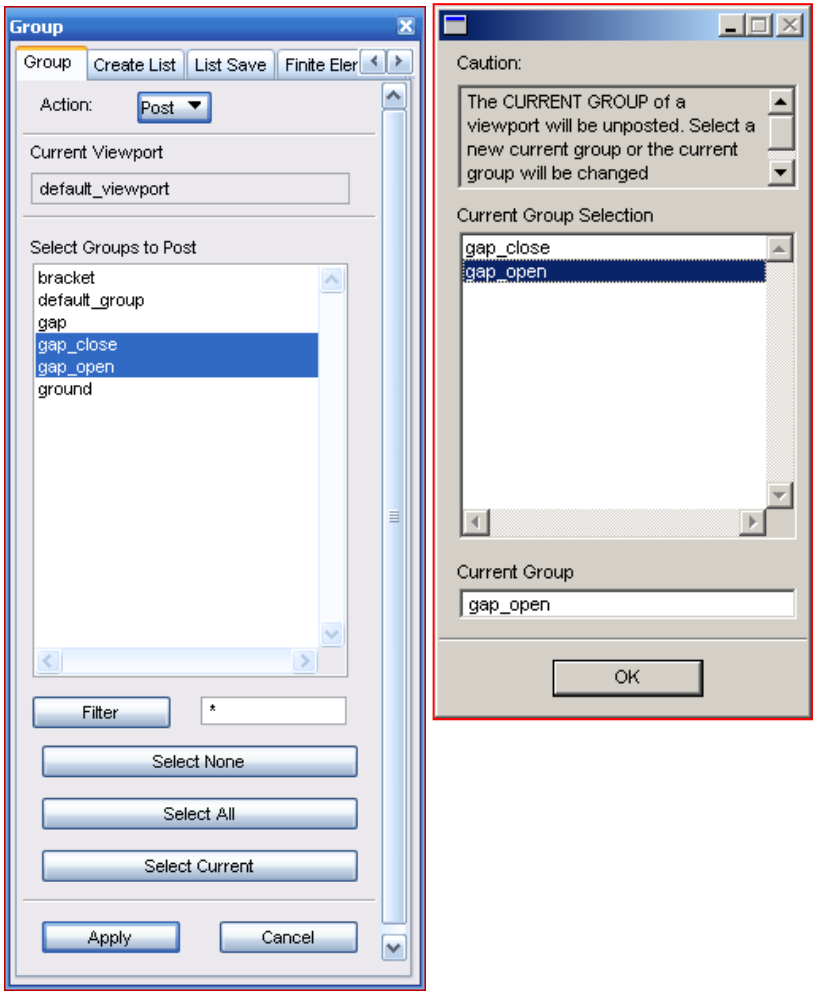




Copyright© 2012 MSC.Software Corporation

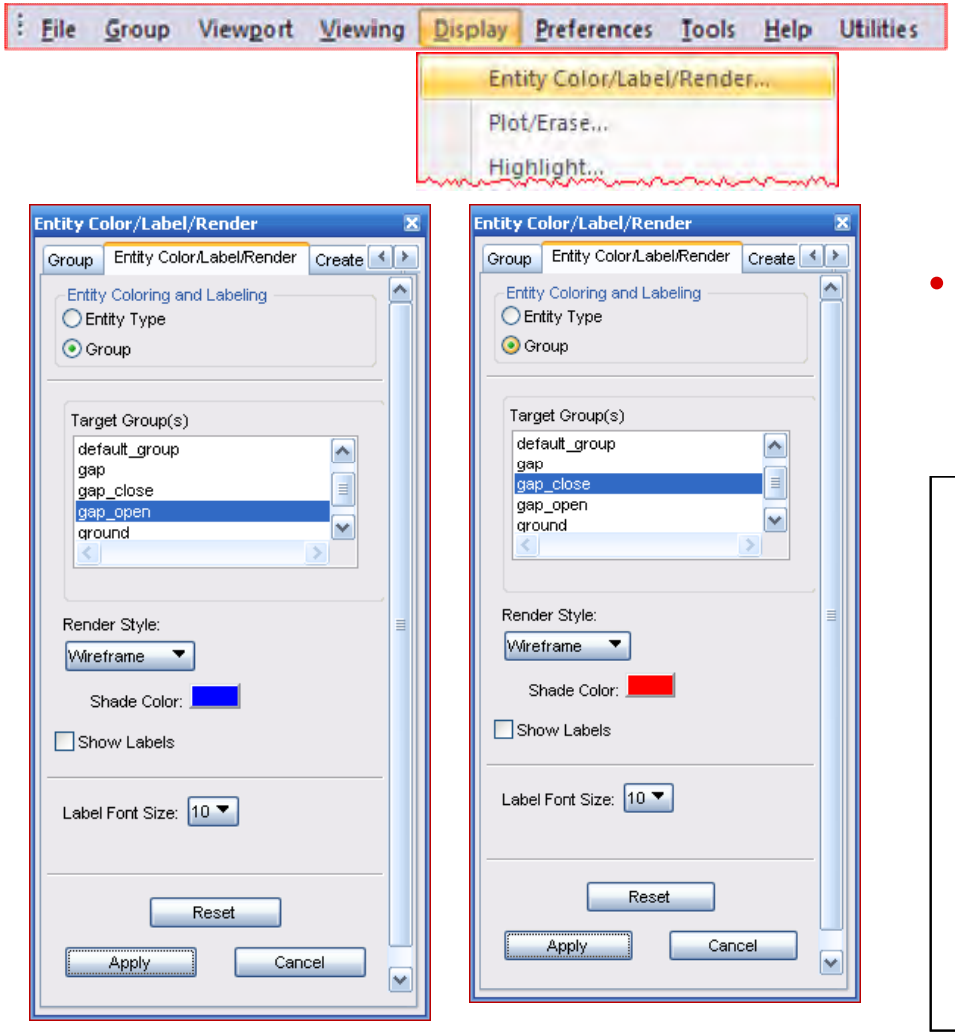
CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

- **Post the two groups just created.**

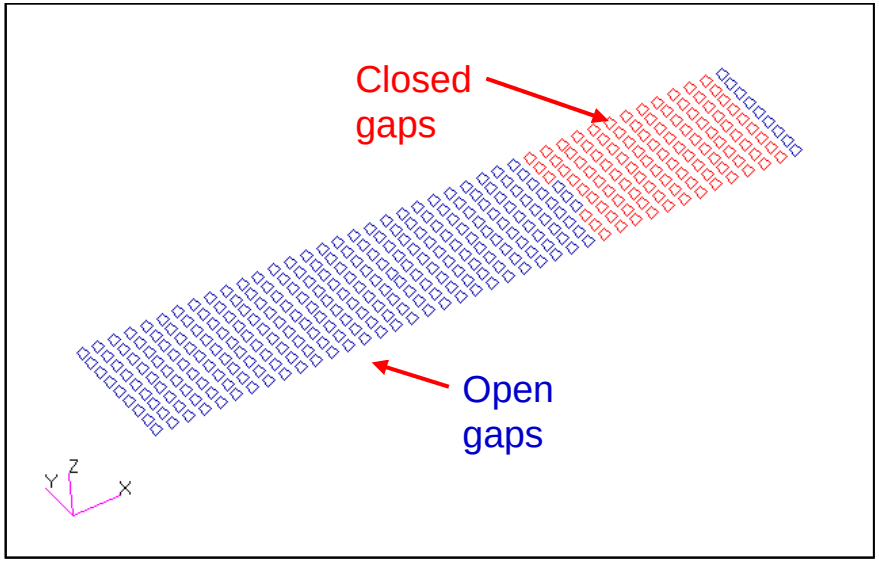


Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS



- **Use group mode to display the gap_open and gap_close groups.**



Copyright© 2012 MSC.Software Corporation

CASE STUDY: VISUALIZE OPEN OR CLOSE OF GAPS

The session file can be used for capturing the functions used to perform the previous 4 steps. This can be used to create a PCL function and hence, the same steps do not need to be performed over and over again.

Copy from session file

```
Function gap_openings()

STRING uil_list_create_current_list[VIRTUAL]
list_create_node_att_fringe( [1., 0.], "Equals", 0.0049999999, "", "lista",  @
uil_list_create_current_list )
list_save_group( "lista", "gap_open", FALSE )
list_create_node_att_fringe( [0., 0.], "Equals", 0.0049999999, "", "listb",  @
uil_list_create_current_list )
list_save_group( "listb", "gap_close", FALSE )
uil_viewport_post_groups.posted_groups( "default_viewport", 2, ["gap_close",  @
"gap_open"] )
ga_group_current_set("gap_open")
$# Warning reported from application Results
$#     The specified target nodes or elements already have associated results
$# data or the specified target is empty or invalid.
sys_poll_option( 2 )
ga_group_color_set( "gap_open", 4 )
coloring_method( 2 )
sys_poll_option( 0 )
ga_group_color_set( "gap_open", 10 )
ga_group_color_set( "gap_open", 4 )

End Function
```


Copyright© 2012 MSC.Software Corporation

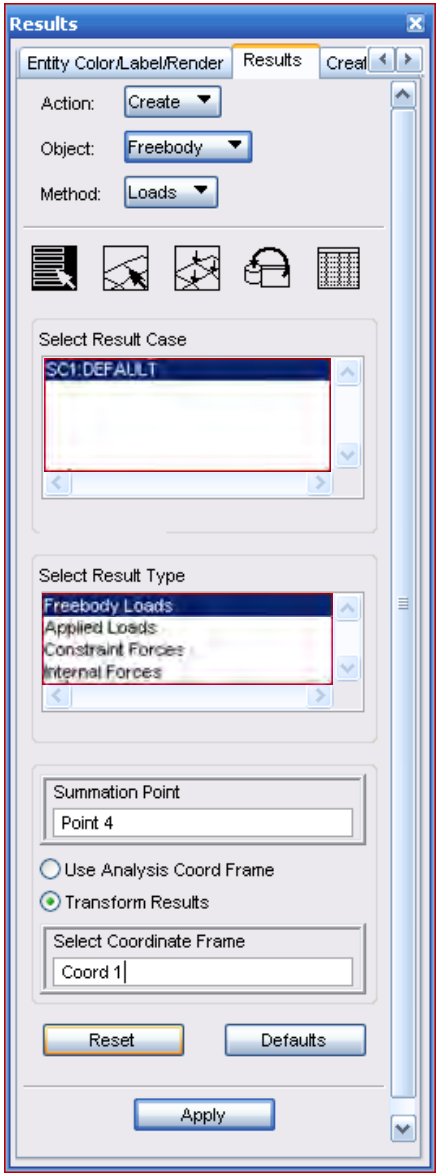
USING FEM FIELD WITH RESULTS

- **An FEM field should *only* be used for results (scalar) that have been averaged, e.g. Results: Create/Fringe, Plot Options/Averaging Definition/Domain: All Entities**
- **Do not create an FEM field for results like Results: Create/Fringe, Plot Options/Averaging Definition/Domain: None**

Copyright© 2012 MSC.Software Corporation

FREEBODY ANALYSIS

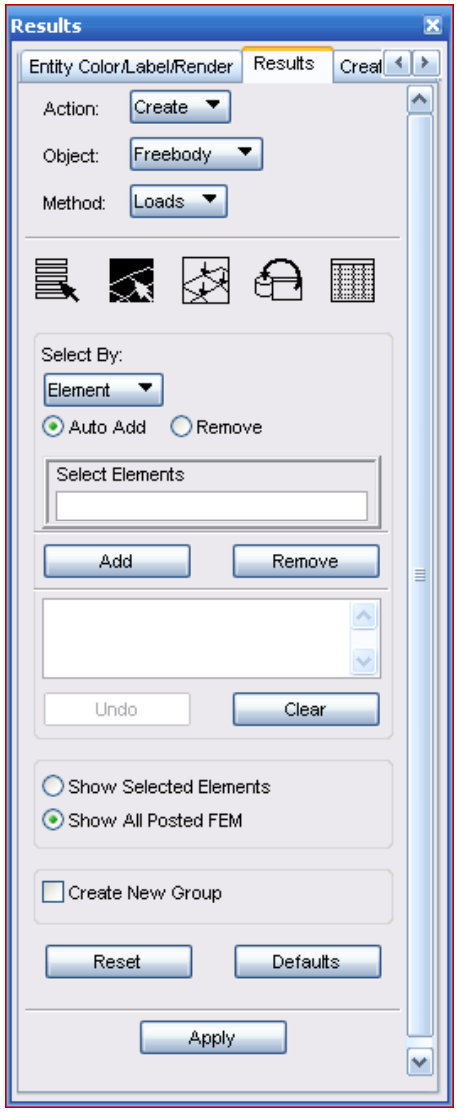
- **The freebody tool is used to create freebody diagrams for loads or displacements.**
- **The tool is accessed under Results as:**
  - Results: Create/Freebody
    - Under it, there are three choices
      - Loads – graphical vector display of freebody loads at all perimeter nodes of the target entities
      - Interface – causes the calculation of the total force and moment acting across a boundary (interface) at a point
      - Displacements – similar to Loads except that displacements are displayed on the perimeter nodes instead of loads
    - Summation Point – forces and moments are summed about the specified point (location, e.g. Point 4)
    - Transform Results – transform results using an Patran local coordinate system, e.g. Coord 1

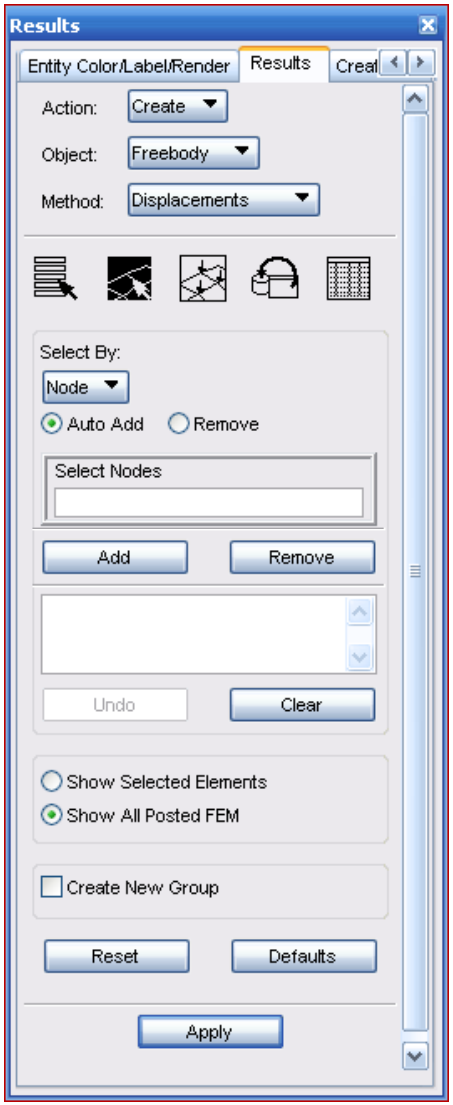


Copyright© 2012 MSC.Software Corporation

FREEBODY ANALYSIS

- **Target entities**
  - Define a freebody by using Select Elements, Select Materials, Select Nodes, etc.
  - Two toggles for specifying what entities are displayed:
    - Show Selected Elements – show only elements that define the freebody.
    - Show All Posted FEM – show all elements, not just freebody elements.

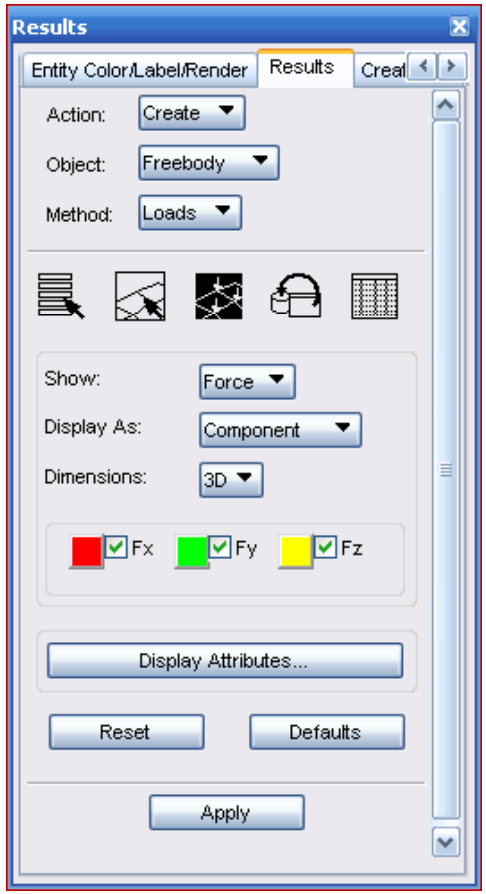


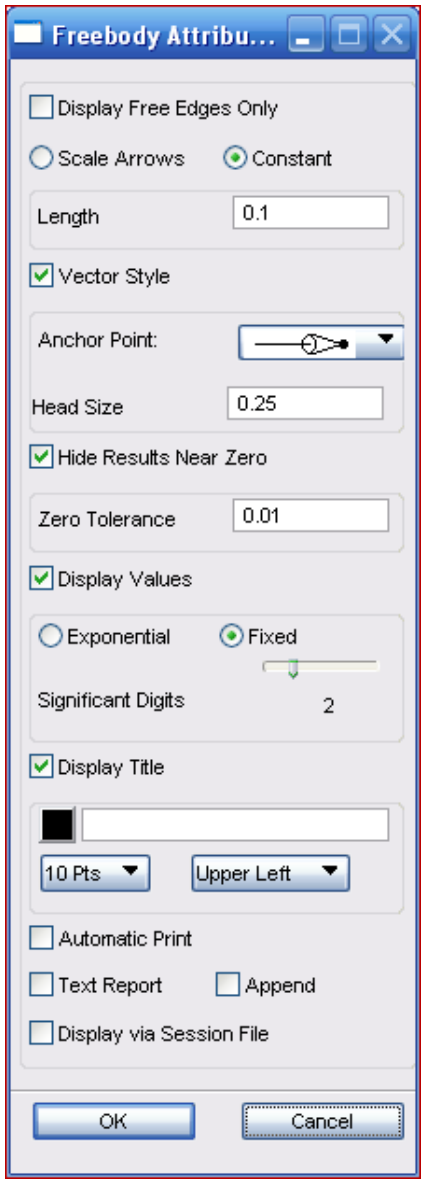


Copyright© 2012 MSC.Software Corporation

FREEBODY ANALYSIS

- **Display attributes**
  - Similar to those for other types of Results

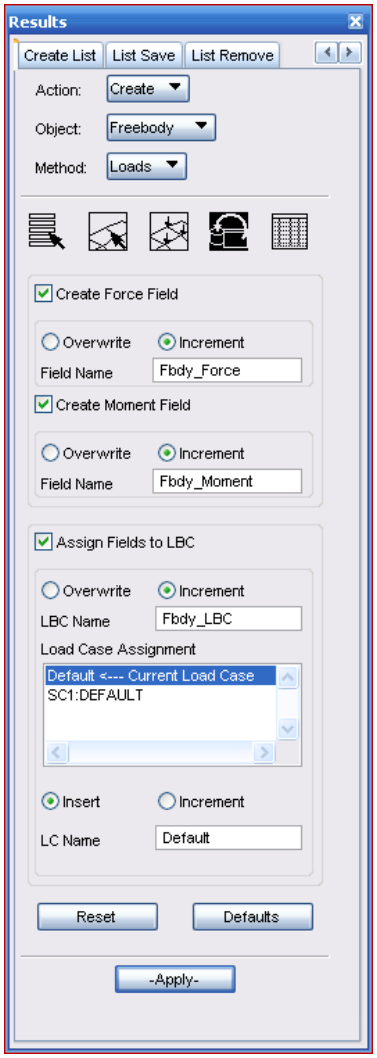


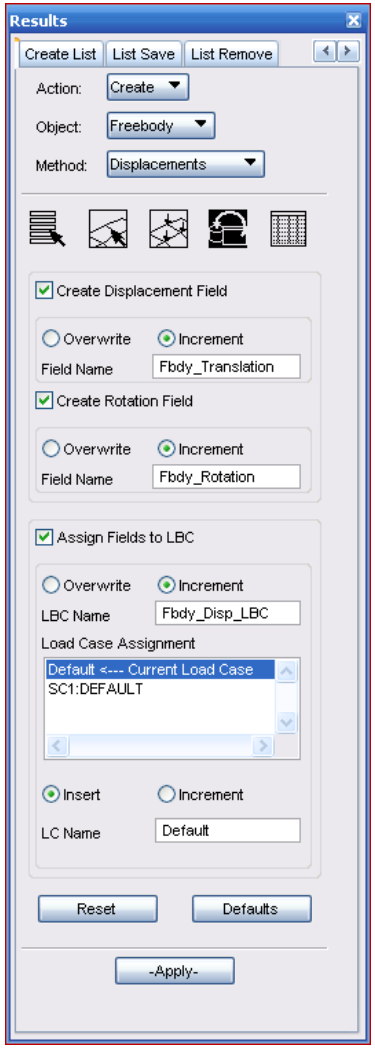


Copyright© 2012 MSC.Software Corporation

FREEBODY ANALYSIS

- **Create a Fields, Loads/BCs, and/or Load Cases from the graphical display:**
  - Create a field
    - Loads (force and/or moment)
    - Displacements (translation and/or rotation)
  - Create a Loads/BCs using a field(s)
  - Assign a Loads/BCs to a load case





Copyright© 2012 MSC.Software Corporation

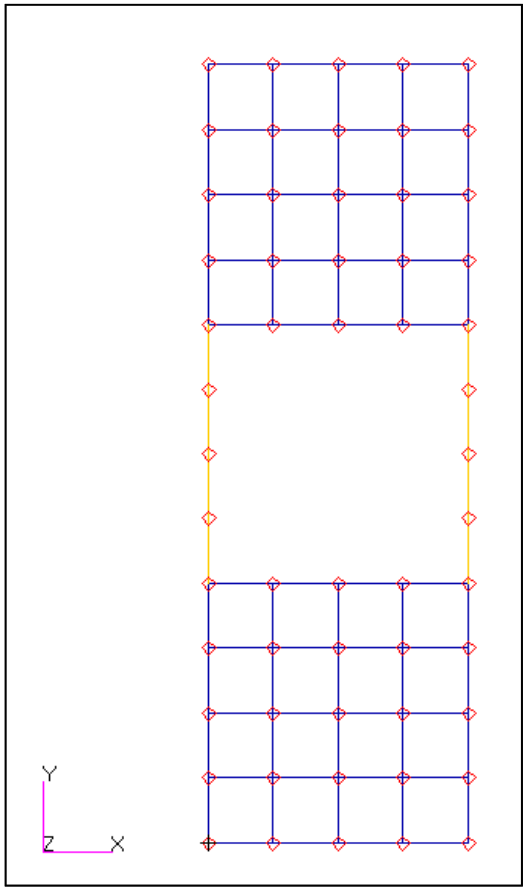
CASE STUDY: FREEBODY OF SIMPLE MODEL

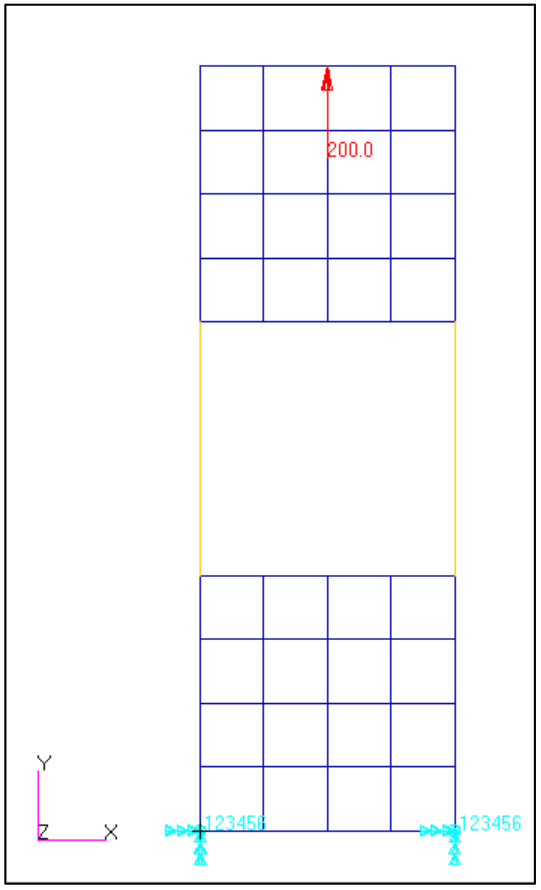
- **Use the freebody tool under Results to demonstrate how to create and use freebody loads or displacements to perform an analysis of a free body (substructure).**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

- **Finite element model with two types of elements:**
  - 2D plate
    - Applied weak stiffness about plate normal
  - 1D bar
- **All elements connected**
- **Bottom plate nodes fixed**
- **Force of 200 lbf applied to top center plate node**

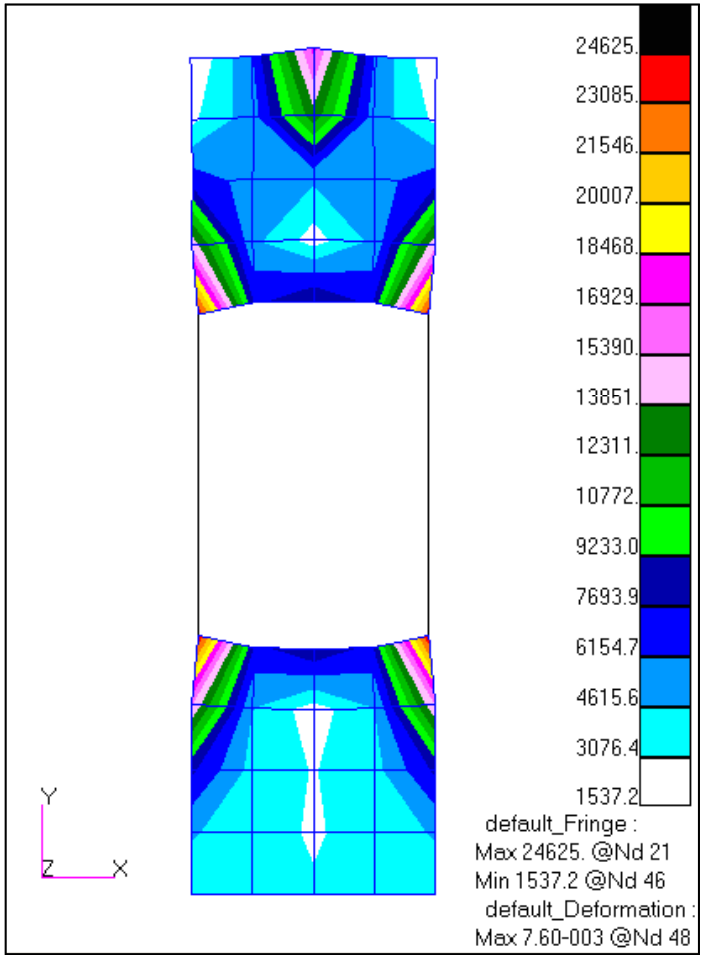




Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

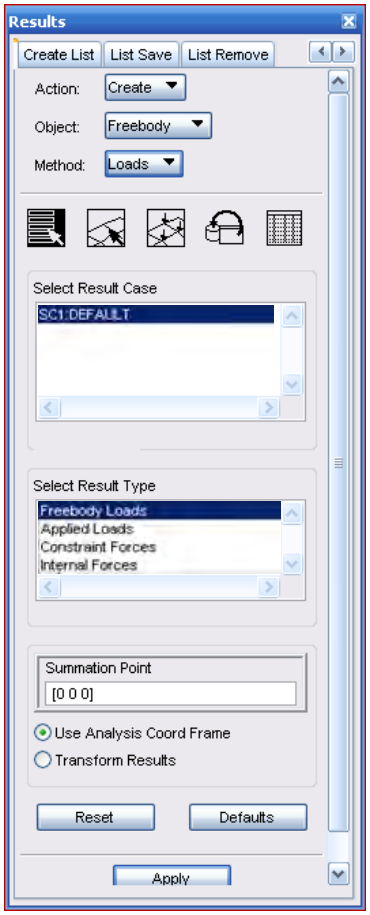
- **Deformation and von Mises stress results for entire model:**

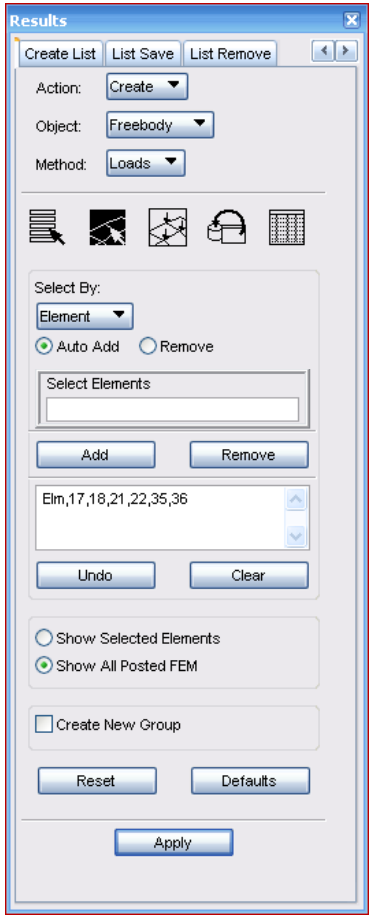


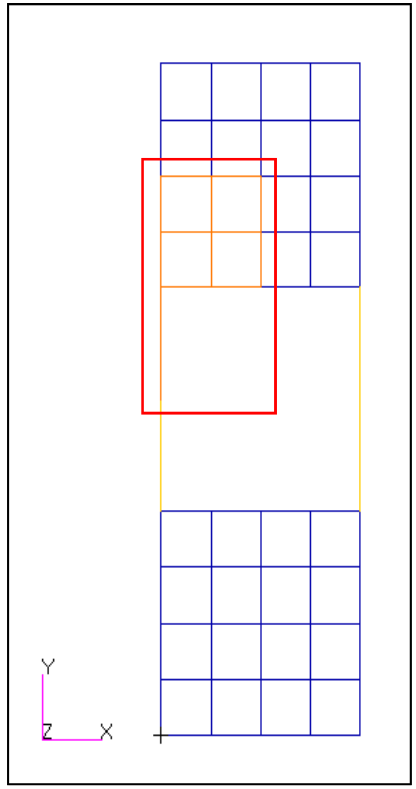
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

- **Select elements for freebody.**
- **Display entire model.**



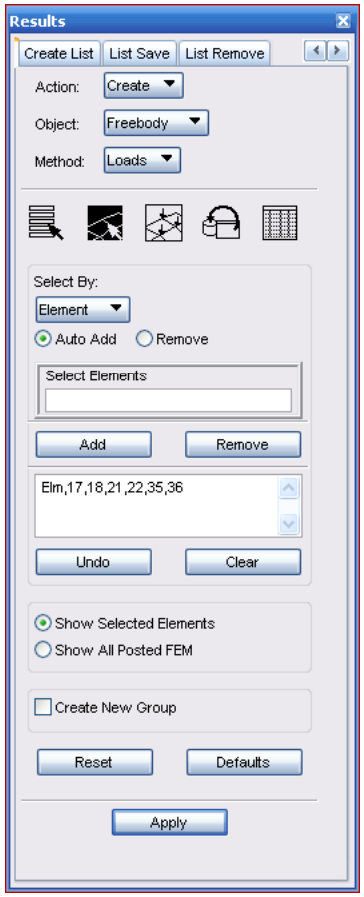


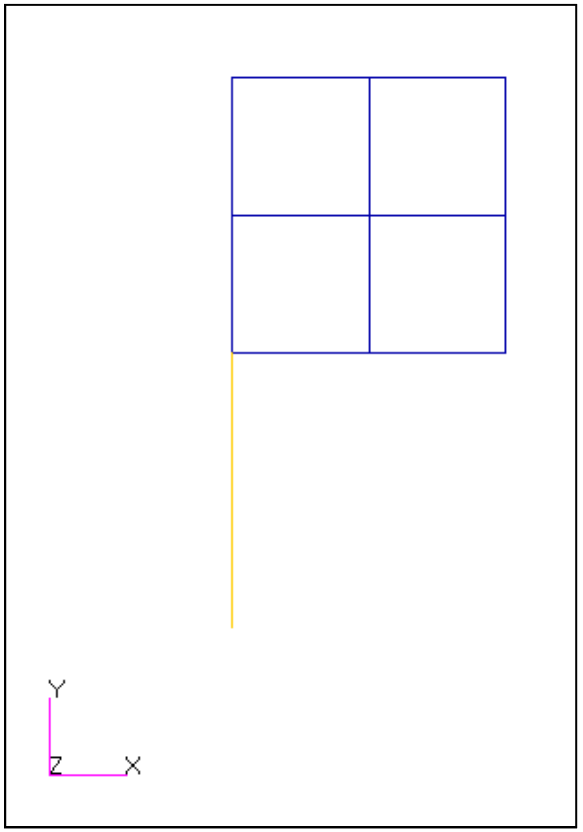


Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

- **Show just the freebody:**

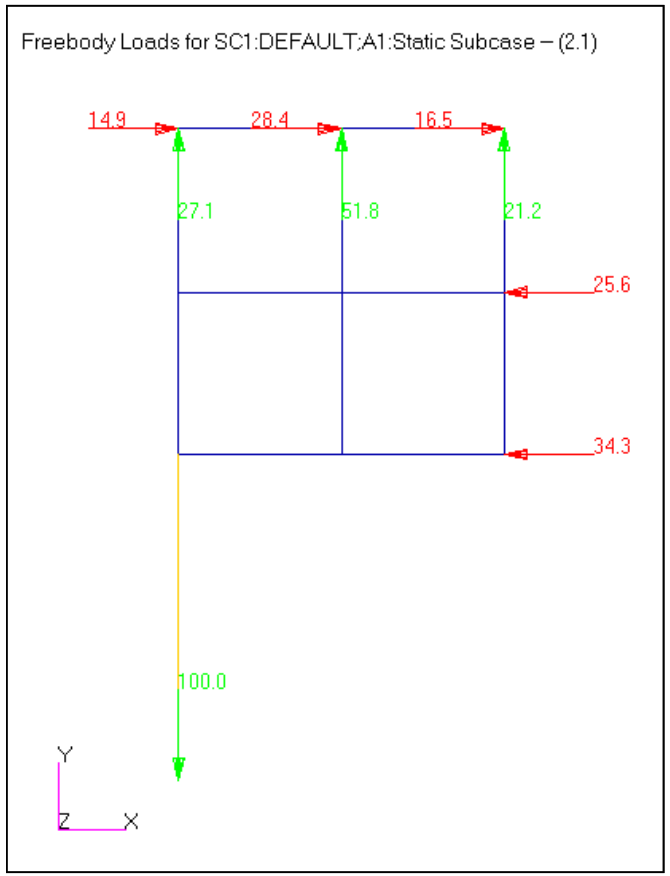




Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

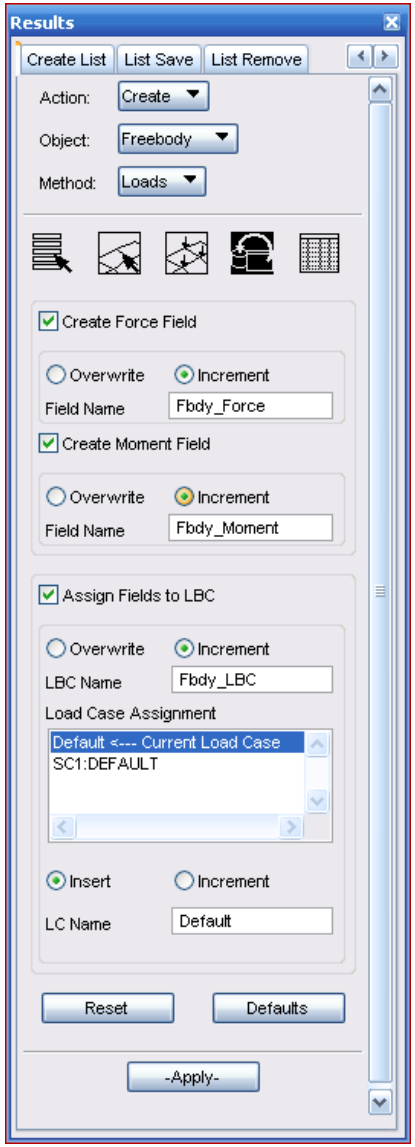
- **Apply**
- **The freebody loads are displayed.**
- **Sum the forces in the X or Y direction:**
  - Sum force X dir = 0.1 ~ 0.0
  - Sum force Y dir = 0.1 ~ 0.0

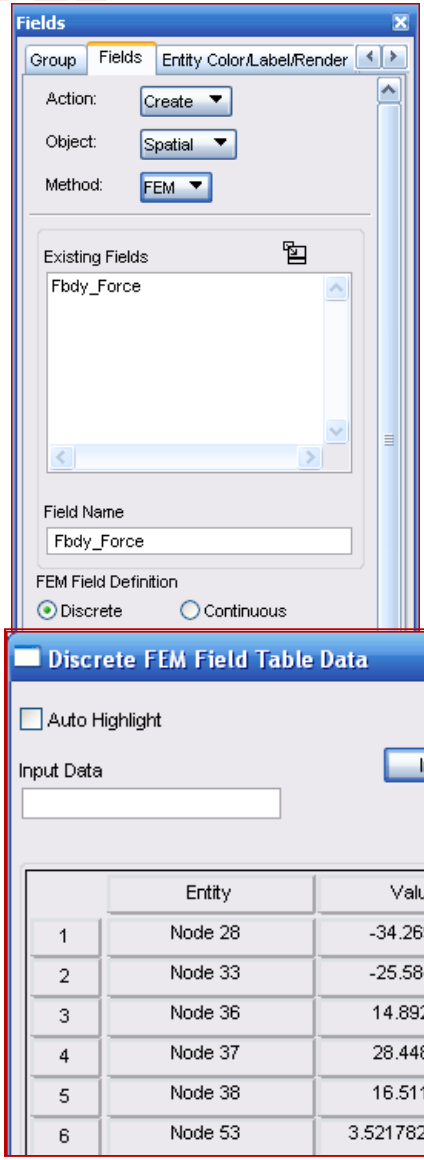


Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

- **Create**
  - Field – discrete, based on force at nodes
    - Fbdy_Force
  - Loads/BCs – based on field
    - Fbdy_LBC
  - LC – includes Loads/BCs
    - freebody
- **Check the field created.**

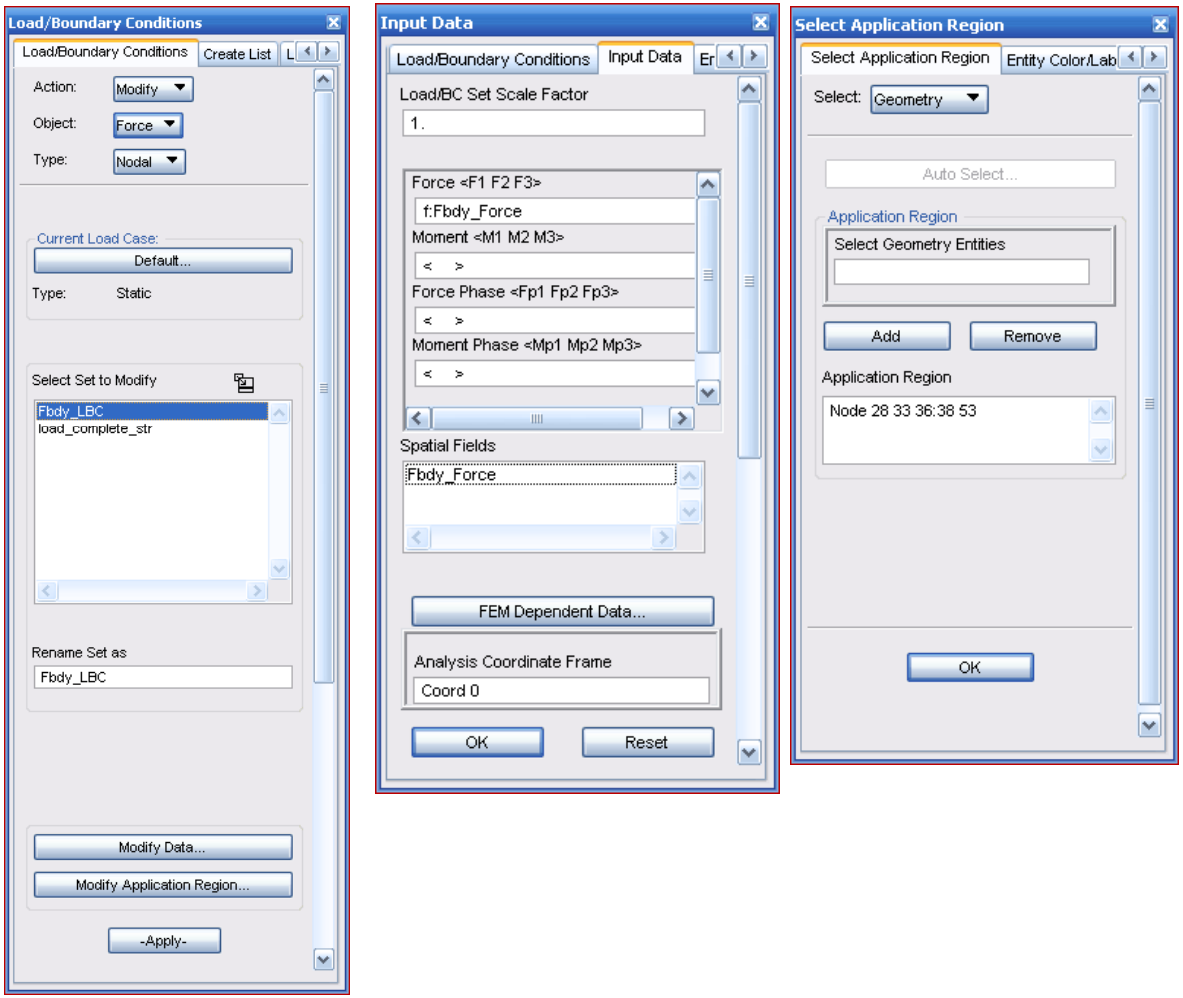




Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

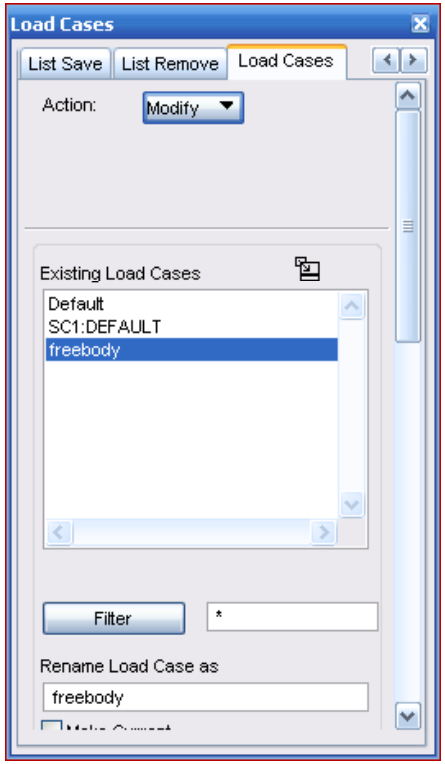
- **Check the Loads/BCs created:**

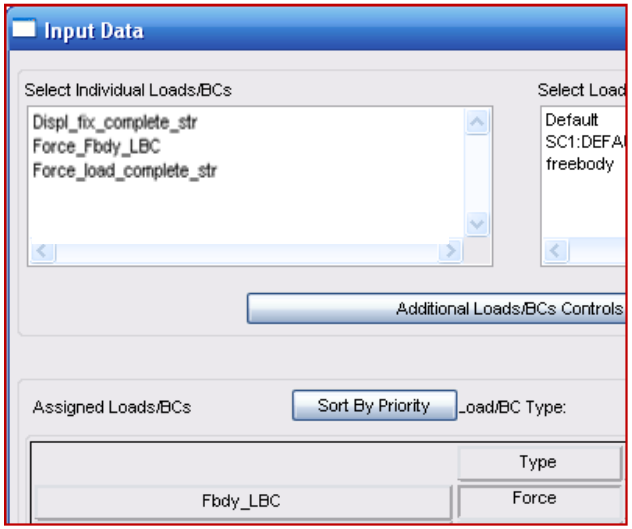


Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

- **Check the load case created:**

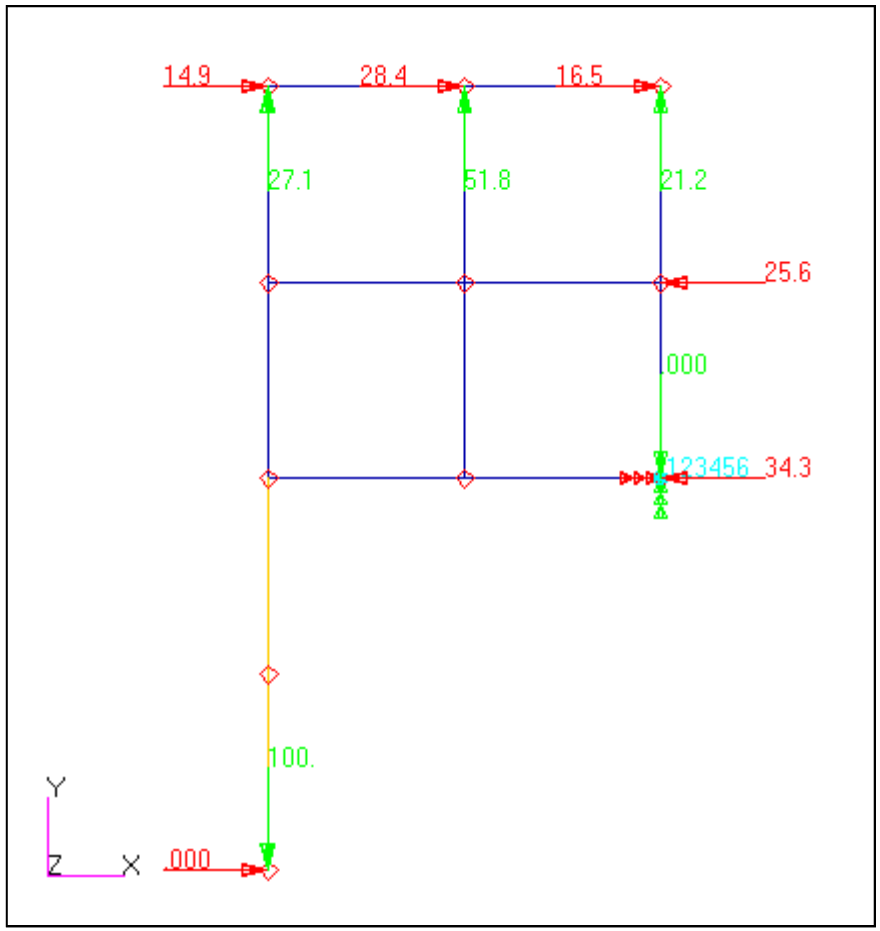




Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

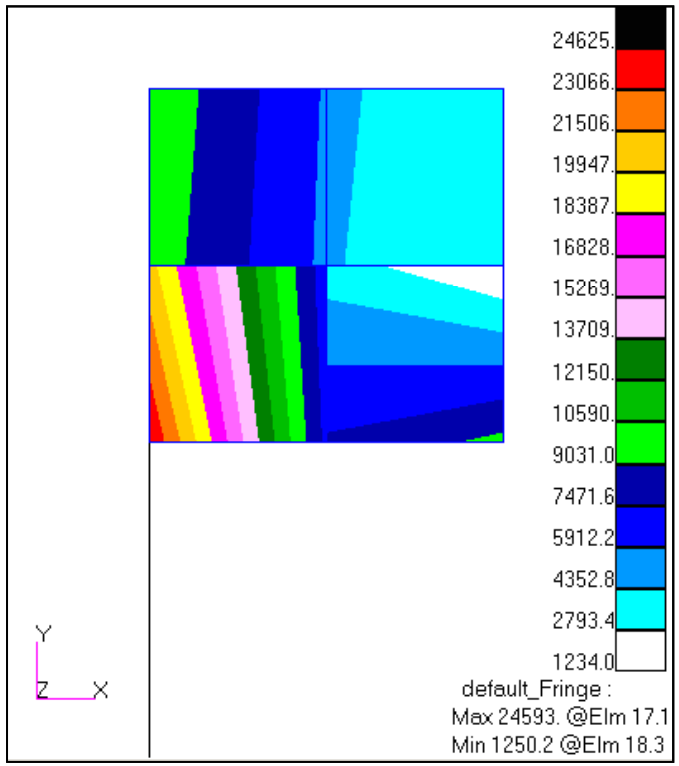
- **Delete all elements and nodes that are not a part of the freebody.**
- **Create a fixed d-of-f LBC at right bottom of freebody.**
- **Display the two Loads/BCs for the freebody model.**
- **Run the linear static analysis for the freebody model.**



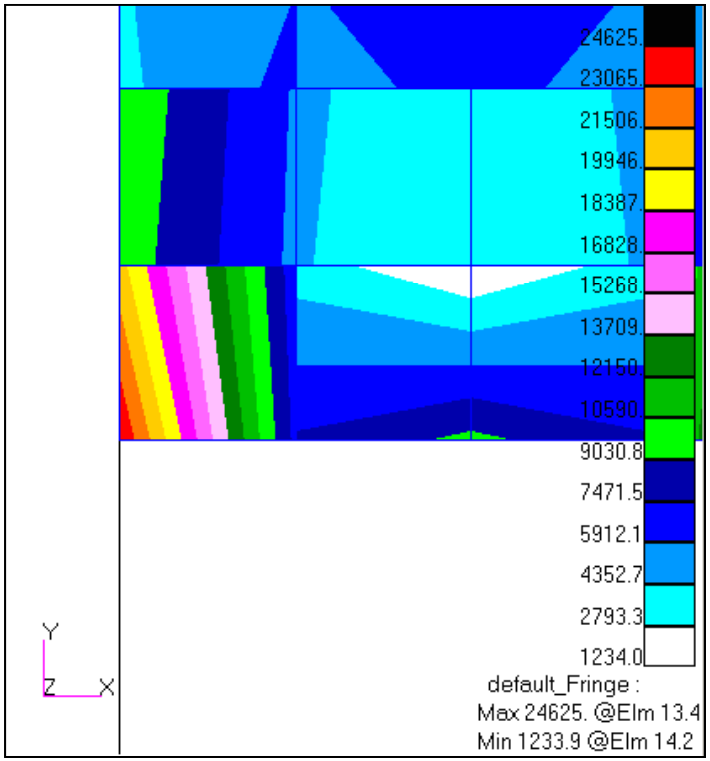
Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

- **Compare the fringe (no averaging at element connections) for the freebody and complete model:**



Freebody Model



Complete Model

Copyright© 2012 MSC.Software Corporation

CASE STUDY: FREEBODY OF SIMPLE MODEL

- **The given procedure can be performed for displacements instead of freebody forces.**
- **Instead of using the freebody tool under results, it is possible to use the continuous spatial FEM field approach.**
  - Display results, e.g. vector results, like displacements
  - Create an FEM field
    - Create/Spatial/FEM
    - Continuous
    - Vector
  - Create a Loads/BCs set using the FEM field, e.g. Create/ Displacement/Nodal.

Copyright© 2012 MSC.Software Corporation

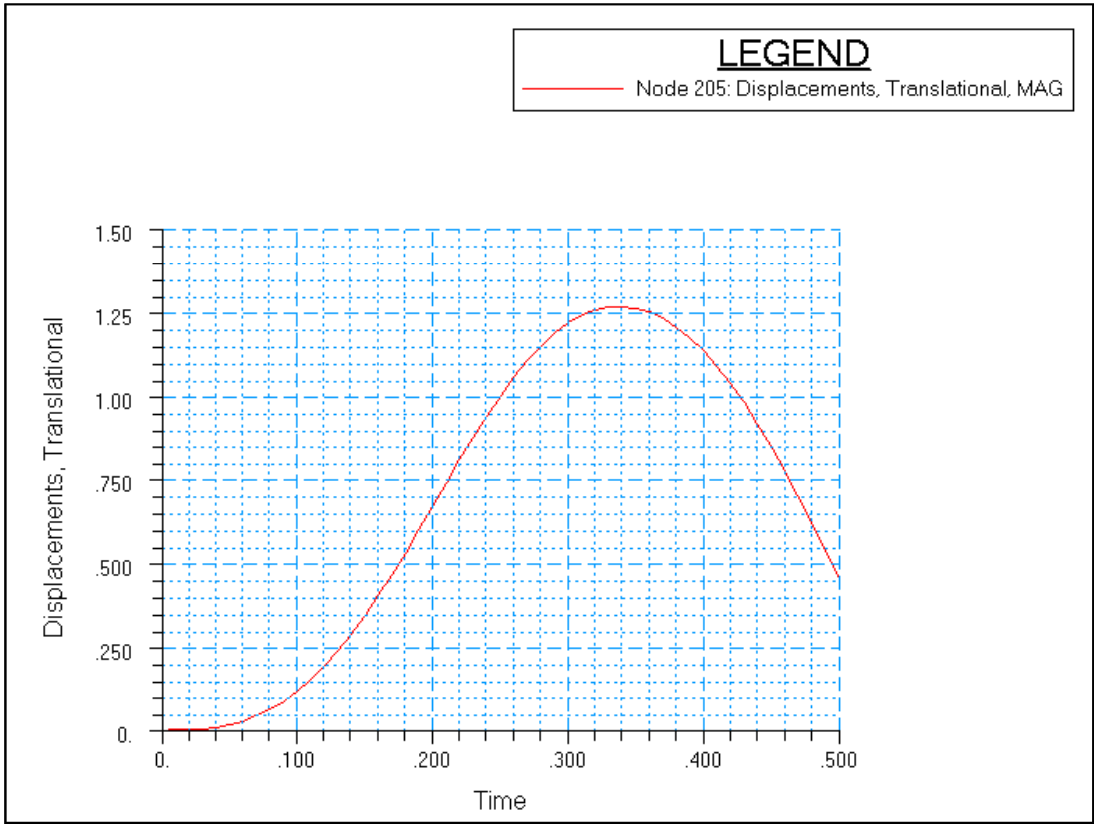
XY PLOT

- **The XY Plot application can be used to modify the Patran XY plot results display, e.g. add tick marks.**
- **It can also be used to read data, such as test data.**
- **The analysis and test data can be displayed in the same XY window.**
- **Following is a simple example for doing this.**

Copyright© 2012 MSC.Software Corporation

CASE STUDY: XY PLOT FOR ANALYSIS VERSUS TEST DATA

- **Transient linear model result shown as XY plot:**

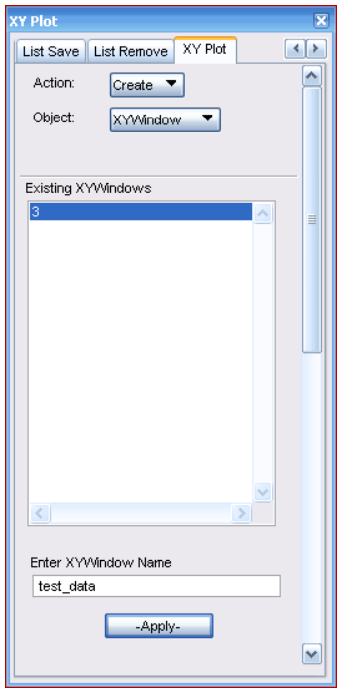


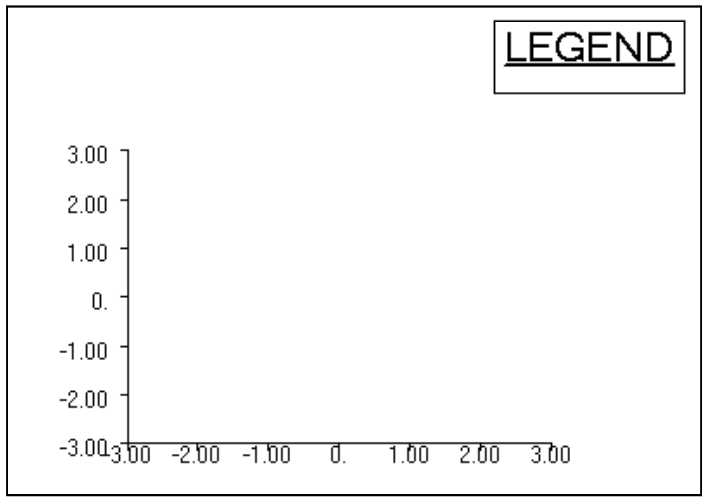
Copyright© 2012 MSC.Software Corporation

CASE STUDY: XY PLOT FOR ANALYSIS VERSUS TEST DATA

- **Read test data into Patran database.**
  - This will create a curve.
- **First, create a new XY window:**

```
XYDATA, test data
0.00 0.000
0.05 0.027
0.10 0.123
0.15 0.370
0.20 0.673
0.25 1.050
0.30 1.227
0.35 1.280
0.40 1.170
0.45 0.830
0.50 0.430
```

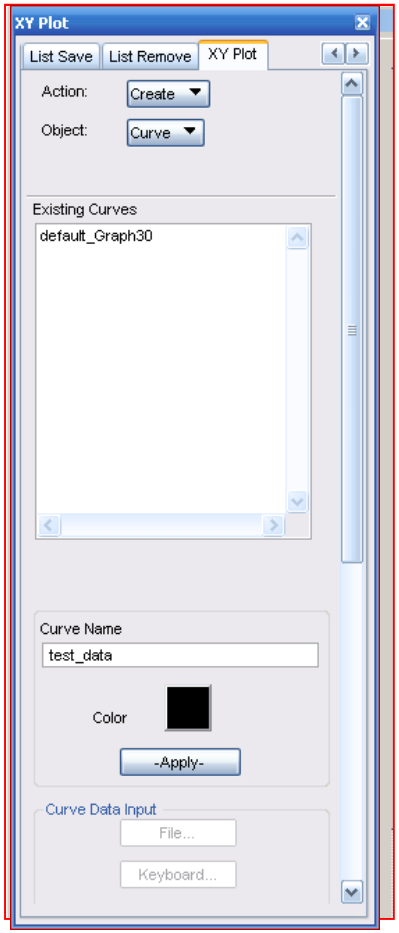


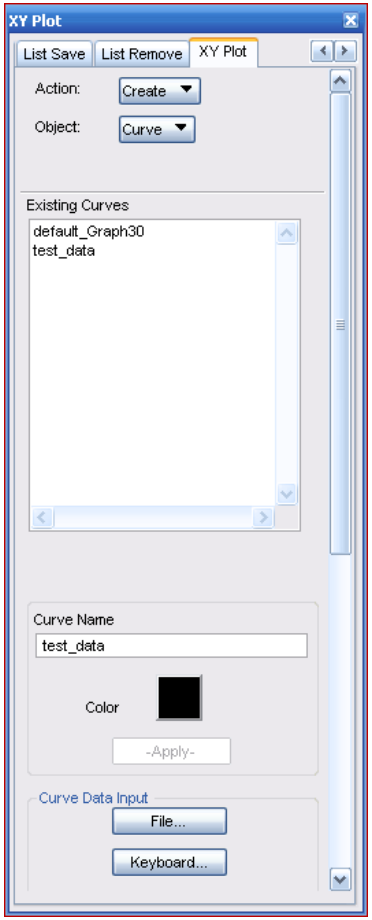


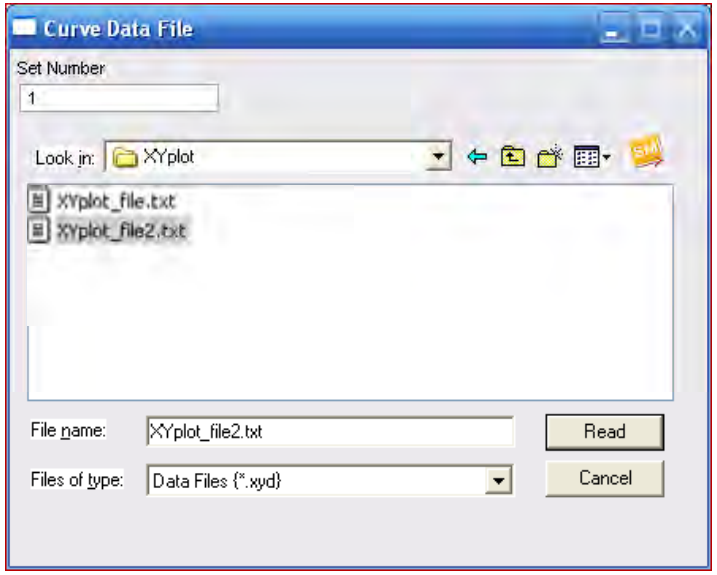
Copyright© 2012 MSC.Software Corporation

CASE STUDY: XY PLOT FOR ANALYSIS VERSUS TEST DATA

- **Read test data:**



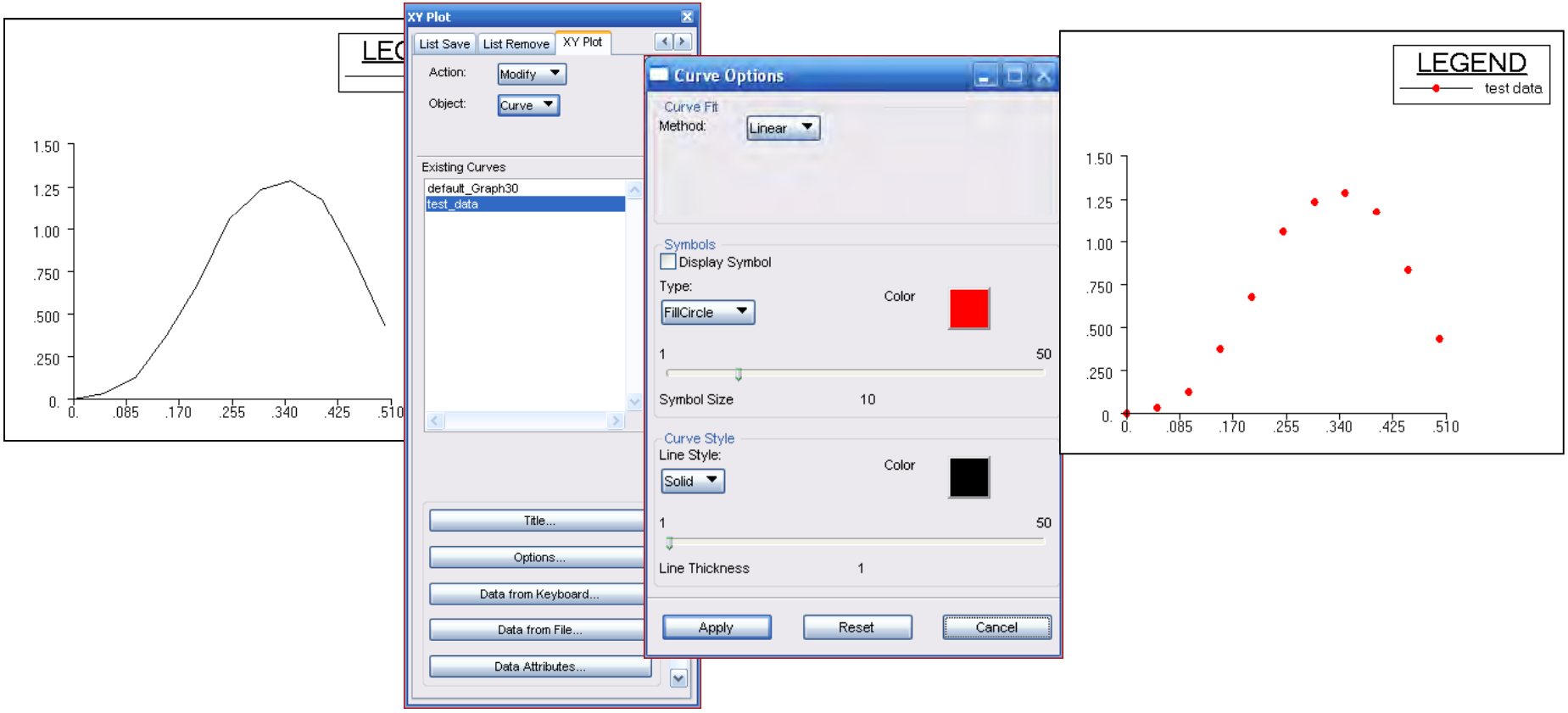




Copyright© 2012 MSC.Software Corporation

CASE STUDY: XY PLOT FOR ANALYSIS VERSUS TEST DATA

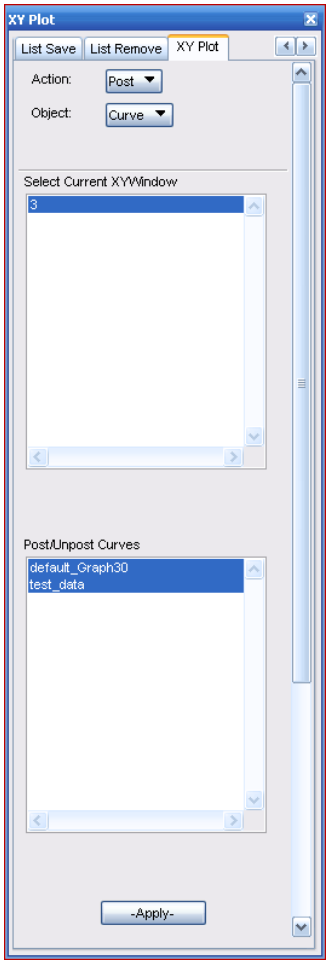
- **Test data displayed as a curve.**
- **Show data as only points.**

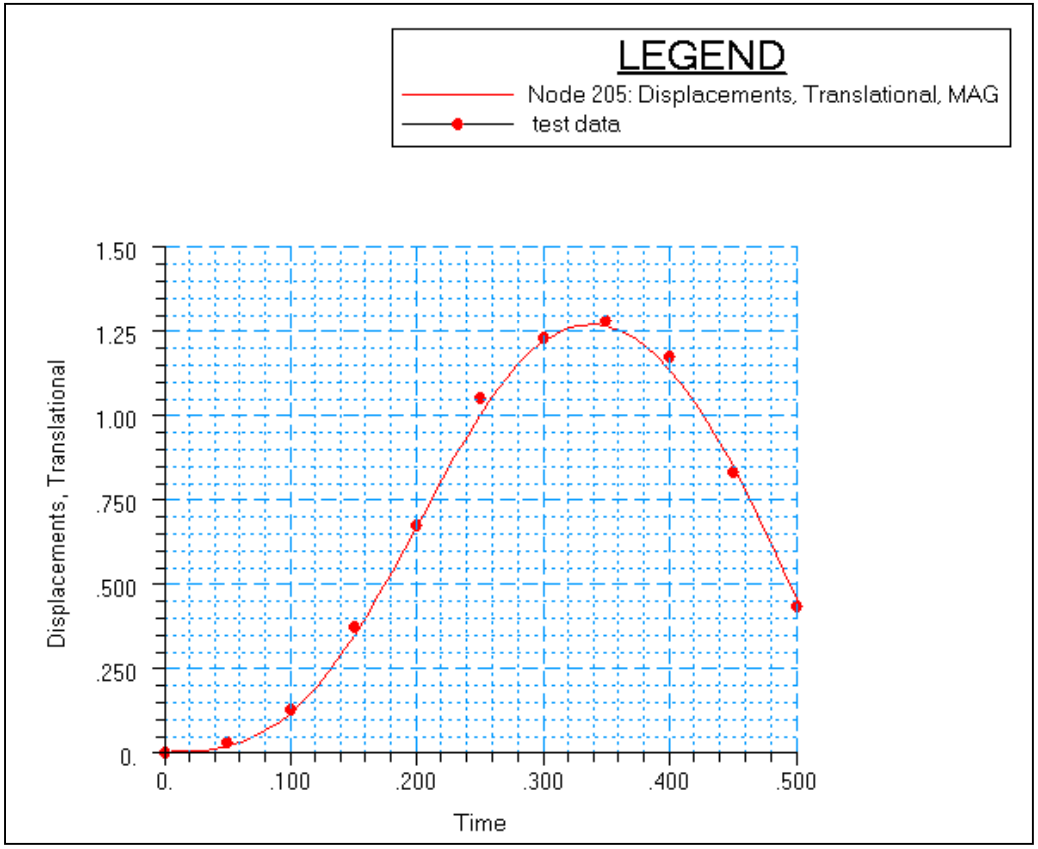


Copyright© 2012 MSC.Software Corporation

CASE STUDY: XY PLOT FOR ANALYSIS VERSUS TEST DATA

- **Create a new XY window, and post both curves to it:**

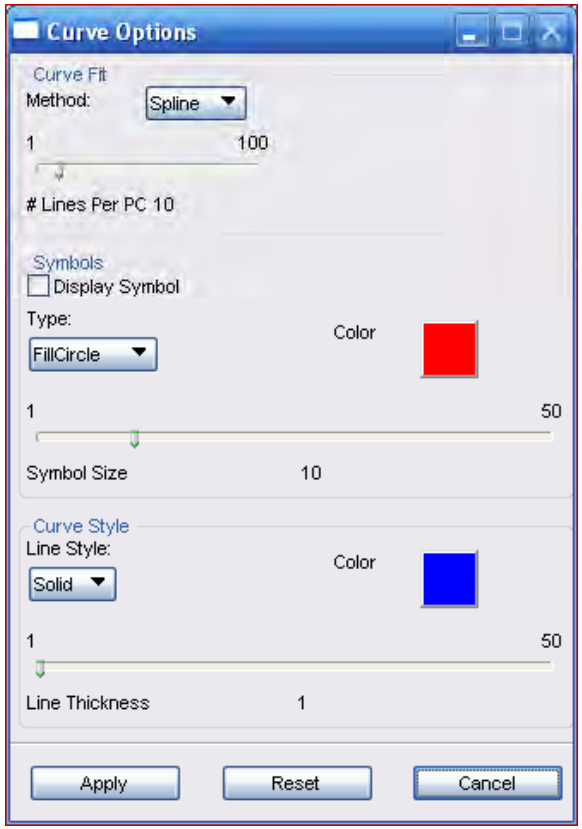


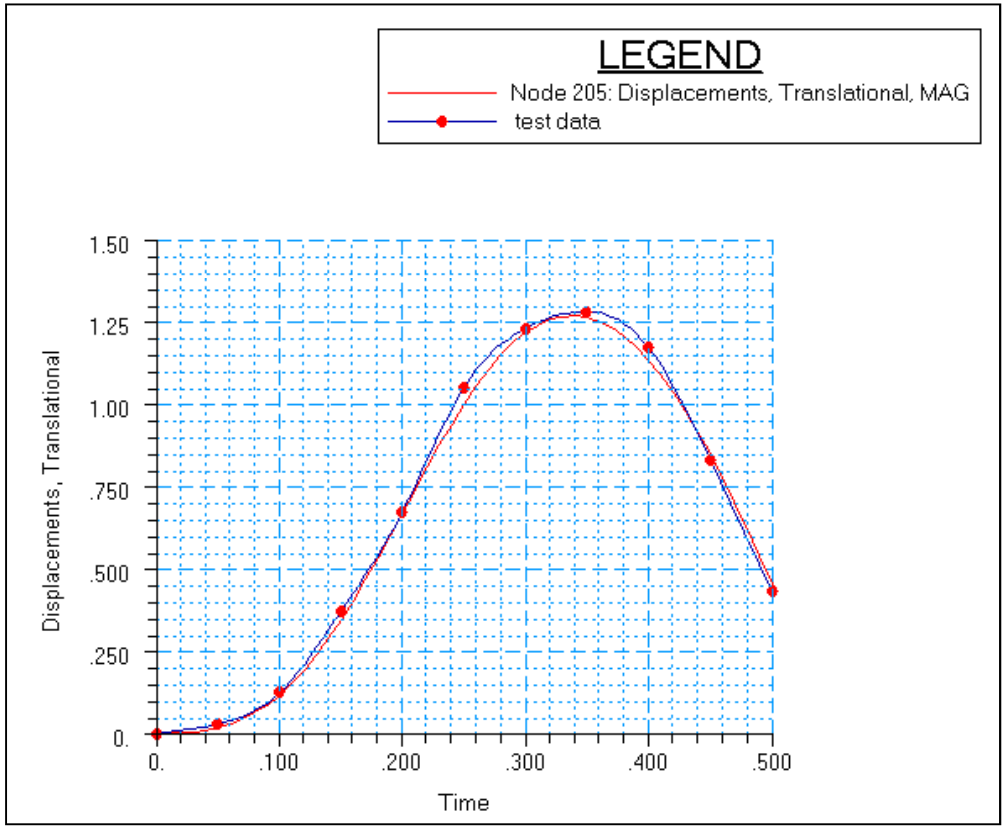


Copyright© 2012 MSC.Software Corporation

CASE STUDY: XY PLOT FOR ANALYSIS VERSUS TEST DATA

- **Fit a curve through the test data points using the spline method:**





Copyright© 2012 MSC.Software Corporation

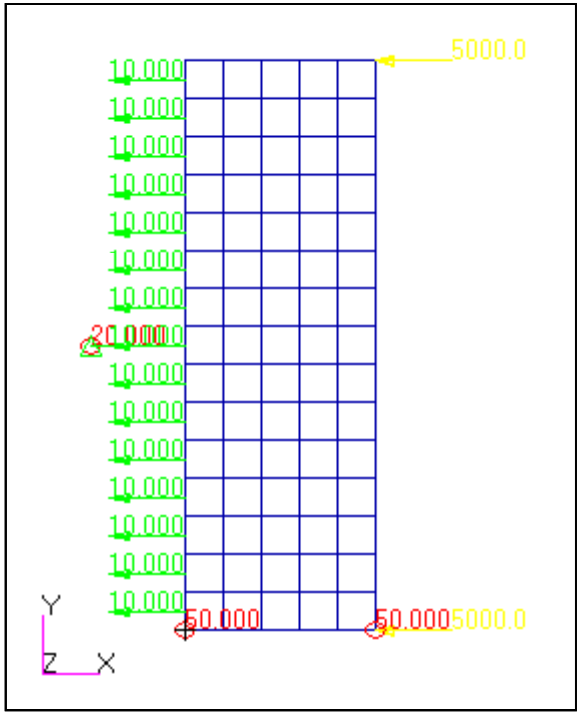
READ RESULTS USING NEUTRAL FILE

- **The object of this material is to briefly describe how to read results into an Patran database from an unsupported finite element code.**
- **The results can then be used in the creation of a Loads/BCs for a different finite element model using a continuous spatial FEM field.**

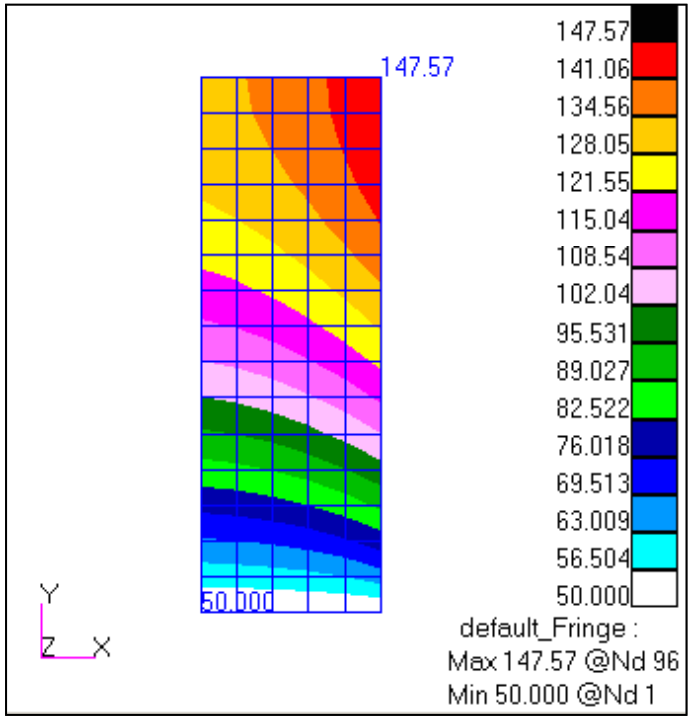
Copyright© 2012 MSC.Software Corporation

CASE STUDY: READ FOREIGN RESULTS

- **Perform a heat transfer analysis using a finite element code that is not supported by Patran:**



Finite Element Model



Results From Analysis

Copyright© 2012 MSC.Software Corporation

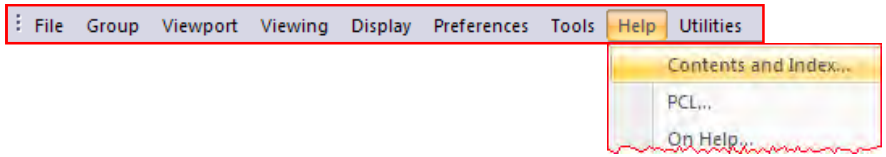
CASE STUDY: READ FOREIGN RESULTS

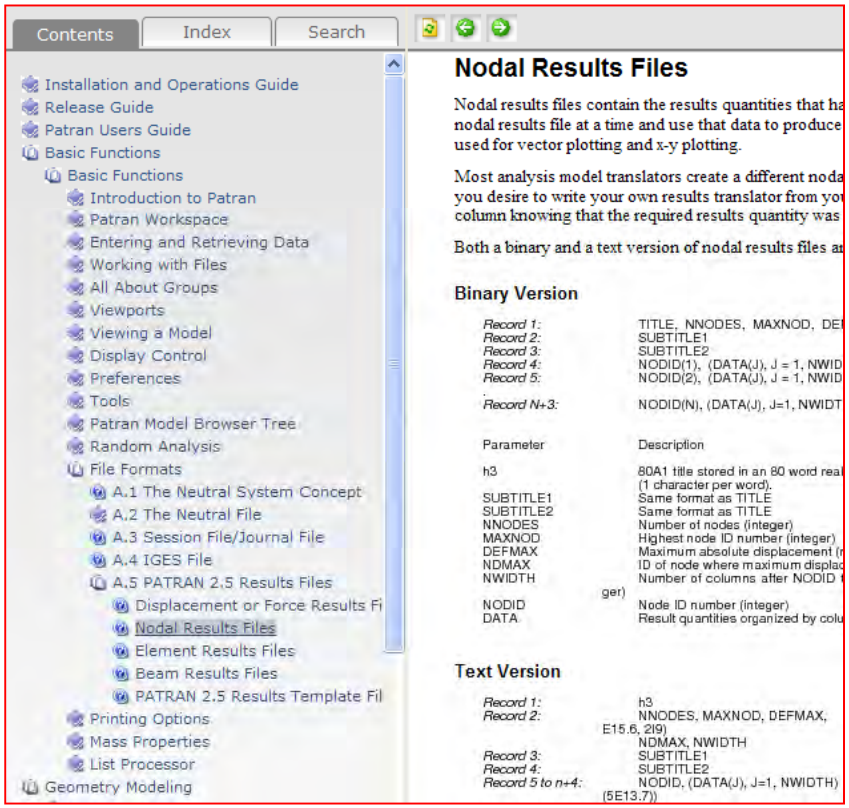
- **A file of temperature at nodes is output from the solver, e.g. output.res.**
- **The data in the file is used to create a file that can be read into Patran.**
  - The format of a file that can be read is given in the Patran Reference Manual

Copyright© 2012 MSC.Software Corporation

CASE STUDY: READ FOREIGN RESULTS

- **Consult the Patran Reference Manual for the format.**





Copyright© 2012 MSC.Software Corporation

CASE STUDY: READ FOREIGN RESULTS

- **The Nodal Results File format shown should be followed to format the data prior to import.**

Nodal Results Files

Nodal results files contain the results quantities that have been calculated at the node points. Up to 200 columns of results quantities can be stored for each node. MSC.Patran will process a single column of a nodal results file at a time and use that data to produce contour plots. MSC.Patran allows nodal results files to be used for element color-coding by averaging the data at the elements. Nodal results files can also be used for vector plotting and x-y plotting.

Most analysis model translators create a different nodal results file for each load case. Each of the MSC supported translator documents contains a table listing the type of results stored in each column. If you desire to write your own results translator from your own in-house analysis program, you must document the results contained in each column of the results file. A user can then process a particular column knowing that the required results quantity was put in that column by the translator.

Both a binary and a text version of nodal results files are supported in MSC.Patran. The format of the file will be automatically determined before reading in the data.

Text Version

Table 0-1 *Record 1:***TITLE(80A1)**
Table 0-2 *Record 2:***NNODES, MAXNOD, DEFMAX,(2I9 E15.6, 2I9)**
Table 0-3 NDMAX, NWIDTH
Table 0-4 *Record 3:***SUBTITLE1(80A1)**
Table 0-5 *Record 4:***SUBTITLE2(80A1)**
Table 0-6 *Record 5 to n+4:***NODID, (DATA(J), J=1, NWIDTH)(I8, (5E13.7))**

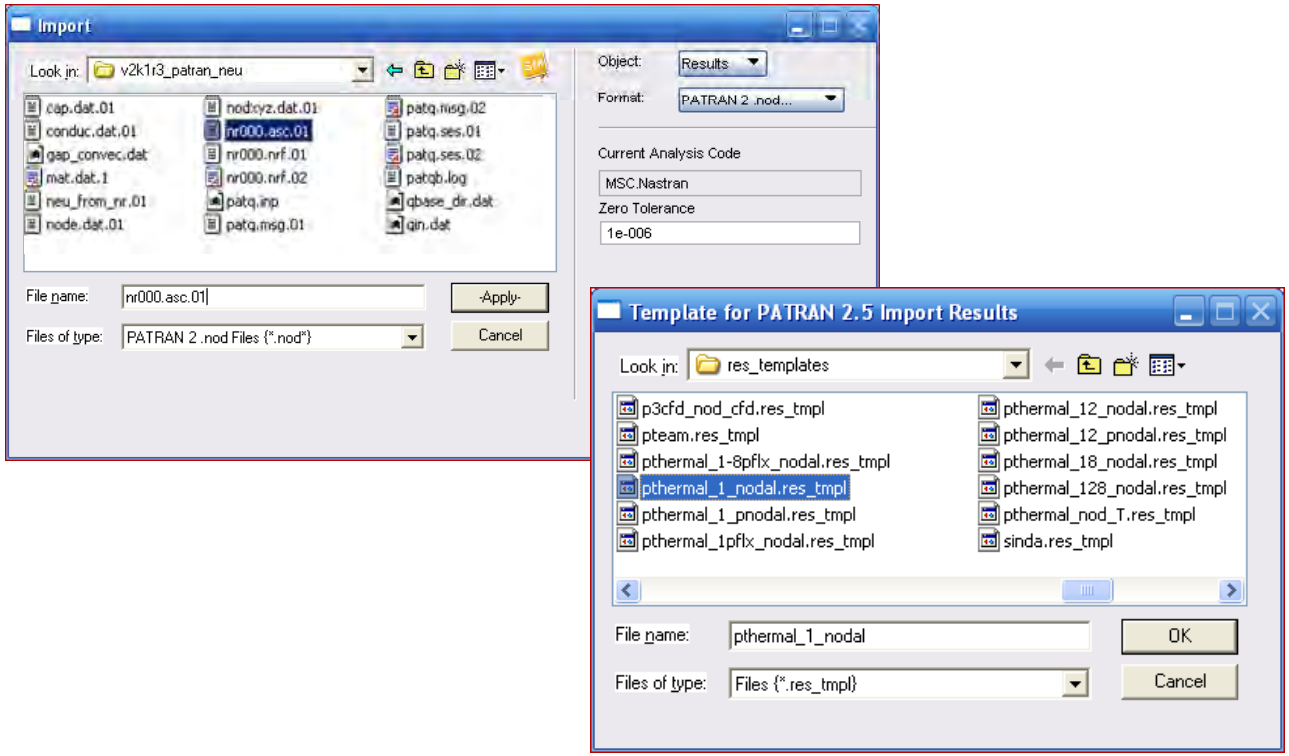
Note: MAXNOD, DEFMAX, NDMAX can be set to zero unless it is a displacement results.

Copyright© 2012 MSC.Software Corporation

CASE STUDY: READ FOREIGN RESULTS

- **The results file is created following the required format.**
- **It is read into a Patran database under File/Import, Object: Results, Format: Patran 2 .nod.**
- **You will also be asked to pick a template file with which to import.**

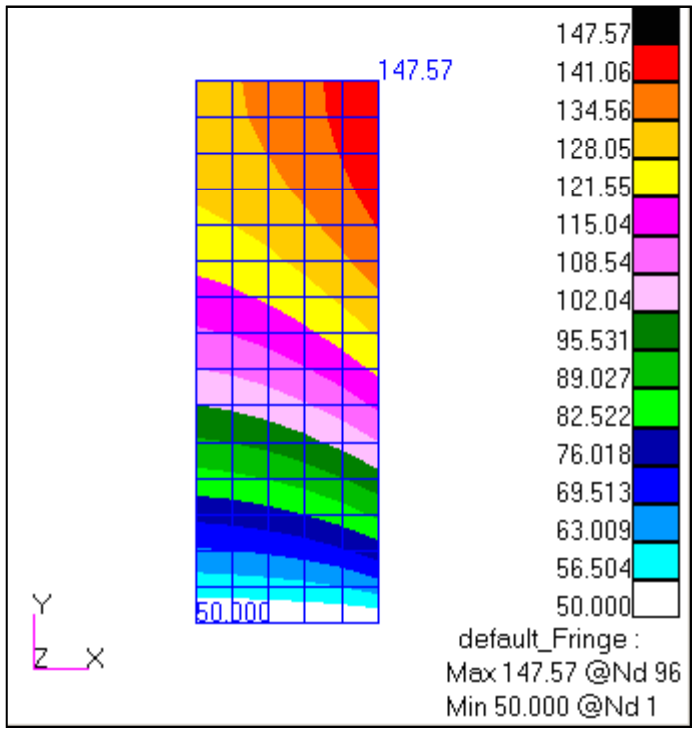
```
TIME:    0.0000000000D+00 SECONDS
       97        0   0.000000E+00
TEMP SCALE: K  ENERGY BALANCE:  5.3
 MSC.PATRAN THERMAL JOB CREATED ON
      10.5000000E+02
      20.5000000E+02
      30.5000000E+02
      40.5000000E+02
      50.5000000E+02
      60.5000000E+02
      70.5953401E+02
      80.5998959E+02
      90.6062062E+02
     100.6161764E+02
     110.6329257E+02
     120.6705315E+02
     130.6908344E+02
     140.6976754E+02
     150.7091112E+02
     160.7267915E+02
     170.7543770E+02
     180.7955293E+02
     190.7825848E+02
     200.7912400E+02
      ...
      ...
```



Copyright© 2012 MSC.Software Corporation

CASE STUDY: READ FOREIGN RESULTS

- **The data read into Patran appears under Results, e.g. Create/Fringe.**

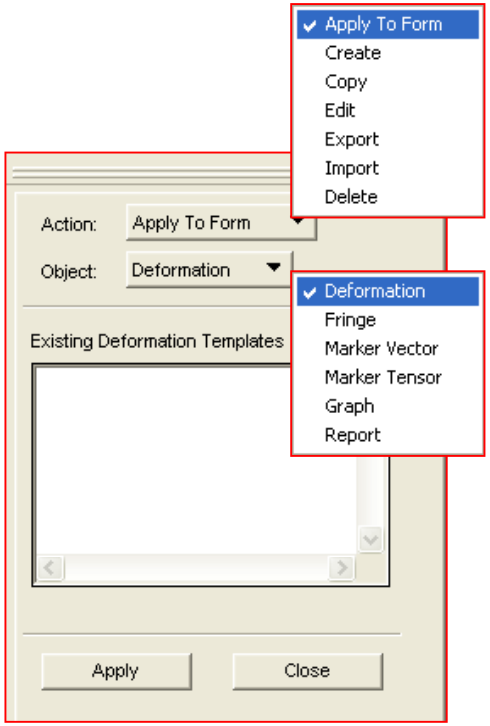


- An FEM field can be created from this fringe plot of imported thermal results.

Copyright© 2012 MSC.Software Corporation

RESULTS TEMPLATES

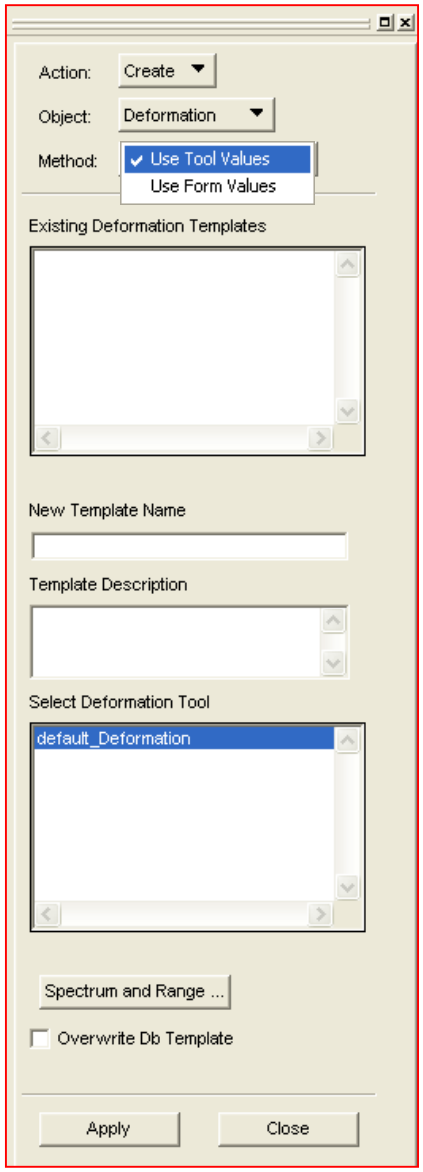
- **Found under Tools pull-down menu**
- **Allows the user to create result templates based on selected options coming from the Display Attributes and Plot Options of Results-Create tools**
  - Fringe, Deformation, etc.
- **Allows creation of consistent plots without having to set Display Attributes & Plot Options every time**
  - Time and productivity saver!
- **Lots of Additional Information on Results Templates can be found in the Patran Help under Basic Functions - Tools**



Copyright© 2012 MSC.Software Corporation

RESULTS TEMPLATES

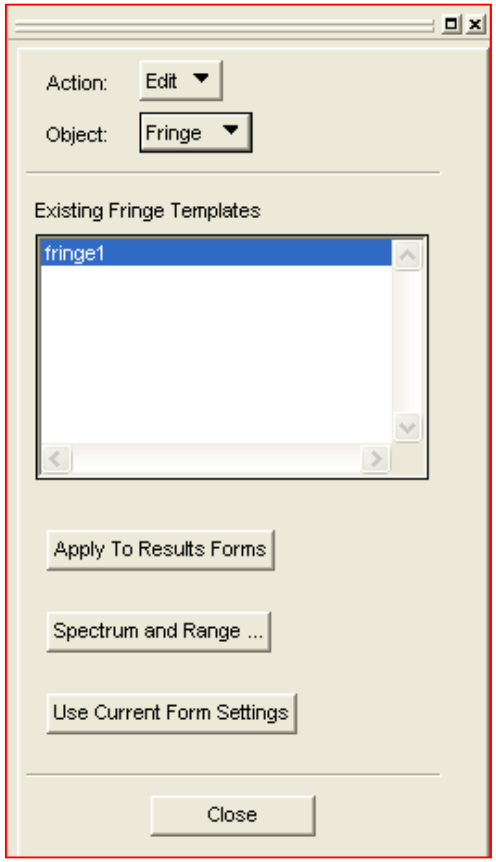
- **Actions**
  - Apply to Form: With the appropriate Results form open, apply a stored Template to the Display Attributes & Plot Options panels
  - Create: Make a new Template
    - Use Tool Values: values from a displayed results plot
    - Use Form Values: current settings on displayed Results form



Copyright© 2012 MSC.Software Corporation

RESULTS TEMPLATES

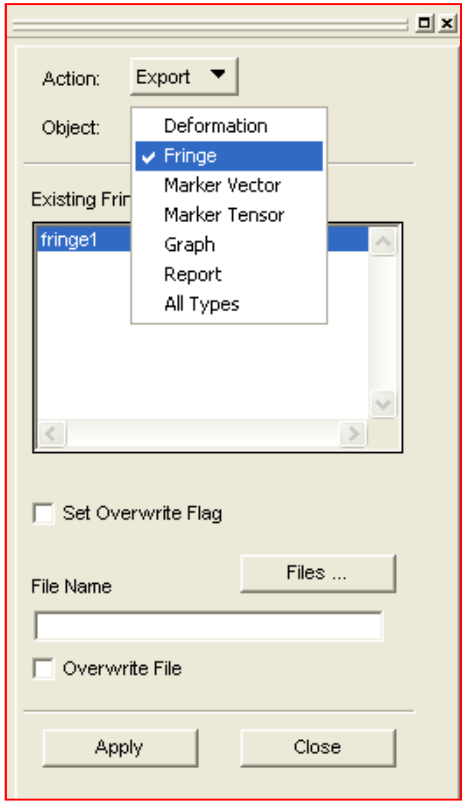
- **Actions (cont.)**
  - Copy: Take the contents of an existing Template and copy into a new Template
    - May want to keep original, but make another with some tweaks to it
  - Edit: Essentially a “Modify” action. Use to update an existing Template, or to update the settings of a displayed Results Display Attributes & Plot Options forms
  - Delete: Delete a Template from the Database



Copyright© 2012 MSC.Software Corporation

RESULTS TEMPLATES

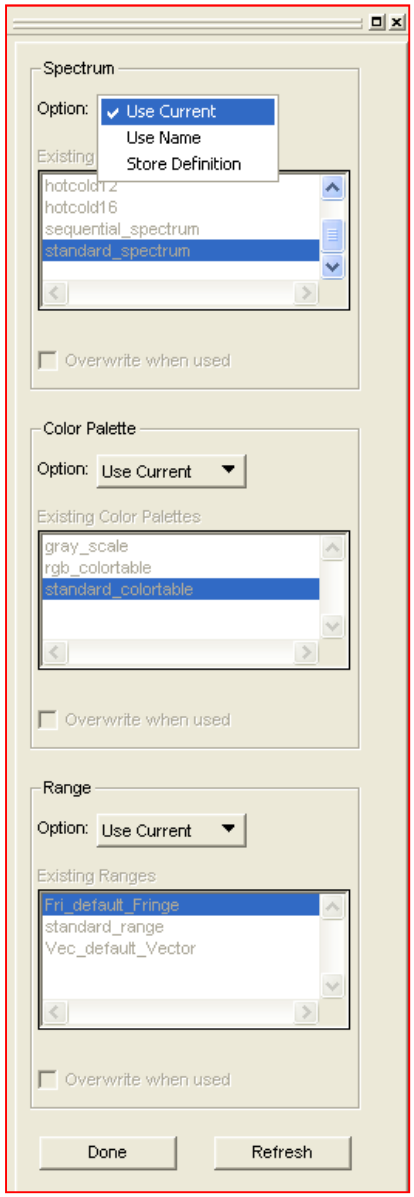
- **Actions (cont.)**
  - Import: Allows you to Import a Patran session file containing definitions for different Results Templates
    - Default file search looking for res_tmpl*.ses*
  - Export: Allows you to Export the Templates you have created inside your Database to a Patran session file
    - Option to Export a certain type of Result, or all Result Template Types
    - Default File syntax is res_tmpl.ses
    - As with all session files, format is ASCII
  - Import/Export allows a team to create one set of Templates and pass along to all members of that team so that all plots created by the group are consistent (same style/format)



Copyright© 2012 MSC.Software Corporation

RESULTS TEMPLATES

- **Using Spectrums & Ranges**
  - When creating a Template, a Range, Color Palette, and/or Spectrum can be selected to be used with it
  - Options include selecting a specific name, using the current default, or storing the contents into a definition name and optionally overwriting its contents, if it already exists



Copyright© 2012 MSC.Software Corporation

RESULTS TEMPLATES

- **Customization**
  - Patran can be configured so that when you create a new database and/or open an existing database, a predefined set of Results Templates will be loaded into the database
    - Don’t need to always go to Results Templates / Import to load your Templates
  - In the case of loading upon opening an existing database, Patran can be configured to replace or keep existing templates with the same name and tool type as those in the predefined set
  - The easiest way to do the initial creation of the templates is via the Patran GUI, even though a session file could be created outside of Patran if the proper format is known

Copyright© 2012 MSC.Software Corporation

RESULTS TEMPLATES

- **Customization (cont.)**
  - Automatic loading of Templates into a NEW database via settings.pcl:
    - **`pref_env_set_logical( "ResTmplAutoLoadNewDb", TRUE )`**
  - Automatic loading of Templates into an existing database via settings.pcl:
    - **`pref_env_set_logical( "ResTmplAutoLoadOpenDb", TRUE )`**
  - The Default value of FALSE disables both of these settings

Copyright© 2012 MSC.Software Corporation

RESULTS TEMPLATES

- **Customization (cont.)**
  - Default name of the Template Initialization File is res_tmpl_init.ses
  - File location is searched for in the order of Current Working Directory, Home Directory, & Patran Installation Directory
  - Two environment variables can be set to change the defaults for the File Name and the Path Search
    - RES_TMPL_DEF_FILE_NAME
    - RES_TMPL_DEF_PATH_NAME
      - If either or both are set, they are searched before the default search settings
      - The Template Path variable may be a relative path

Copyright© 2012 MSC.Software Corporation

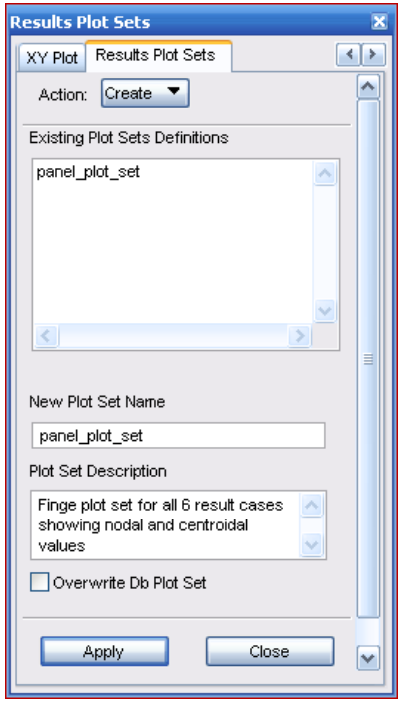
RESULTS TEMPLATES

- **Customization (cont.)**
  - Two other settings.pcl variables can be used:
    - pref_env_set_logical("ResTmplAutoLoadAllFiles", TRUE)
      - Directs MSC.Patran to find all res_tmpl_init.ses files in the search path and load them all into the Database
      - Default value of FALSE allows MSC.Patran to behave as described on the previous slides
    - pref_env_set_string("ResTmplAutoLoadDirOrder", "1,2,3,4")
      - Directs the order for MSC.Patran to search for Initialization files
      - None, Current Working Directory, Home Directory, & Installation Directory are the definitions for 1, 2, 3, & 4, respectively

Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

- Found under Tools / Results/ Plot Sets
- A Plot Set is a catalogue of different result type plots (Deformation, Fringe, etc.), calling out different Result Types, Positions, Display Attributes, and Plot Options
- Uses Result Templates in the catalogue definitions
- Plot Sets can be saved to the database, and can be Exported / Imported for use in other databases
- HUGE time saver when regenerating plots after a rerun of an analysis
- Lots of additional information on Plot Sets can be found in Patran Help under Basic Functions - Tools



Copyright© 2012 MSC.Software Corporation

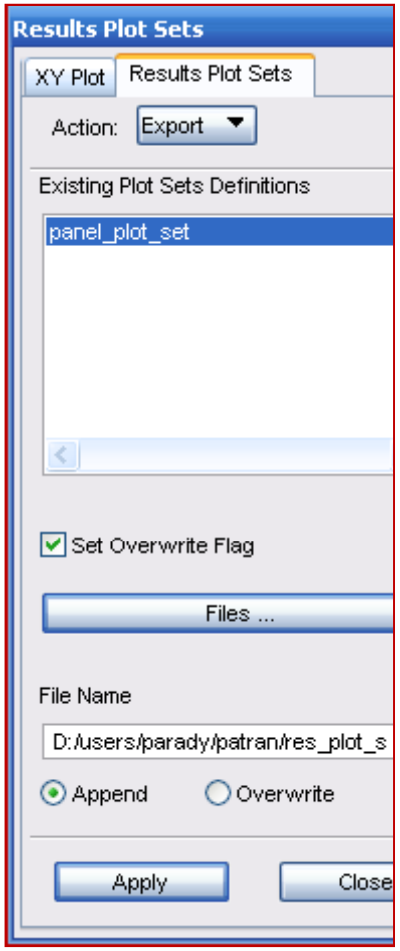
RESULTS PLOT SETS

- **Actions**
  - Create: For every new Plot Set, a name has to be created, and optionally can tag a description and an override tag if the name already exists in the database
  - Copy: Copy an existing Plot Set into another Plot Set
    - Great for making another Plot Set with nearly all the same settings, but with some “tweaks” to the definitions
  - Delete: Delete a Plot Set definition from the database
    - Templates, Results & Settings are not affected

Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

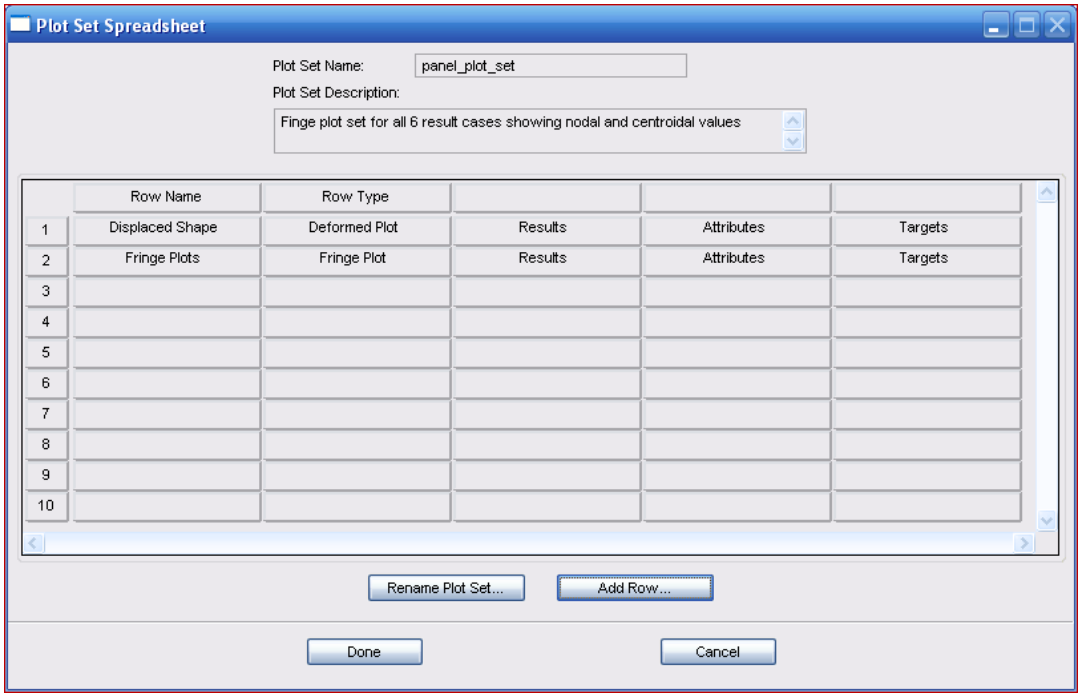
- **Actions (cont.)**
  - Import: Import into a Database an existing Plot Set Definition Session file
    - Default File Search Format is res_plot_set*.ses*
    - Option upon Import to overwrite an existing Plot Set Definition with the same name
  - Export: Create a Patran Session File that contains the definition of the Plot Set(s) created in the existing Database
    - Append/Overwrite File Name allows to build up a single Plot Set file in steps or to overwrite an existing Plot Set file with the new definitions
    - Set Overwrite Flag defines a switch within the session file such that upon import, any Plot Set with the same name will be overwritten



Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

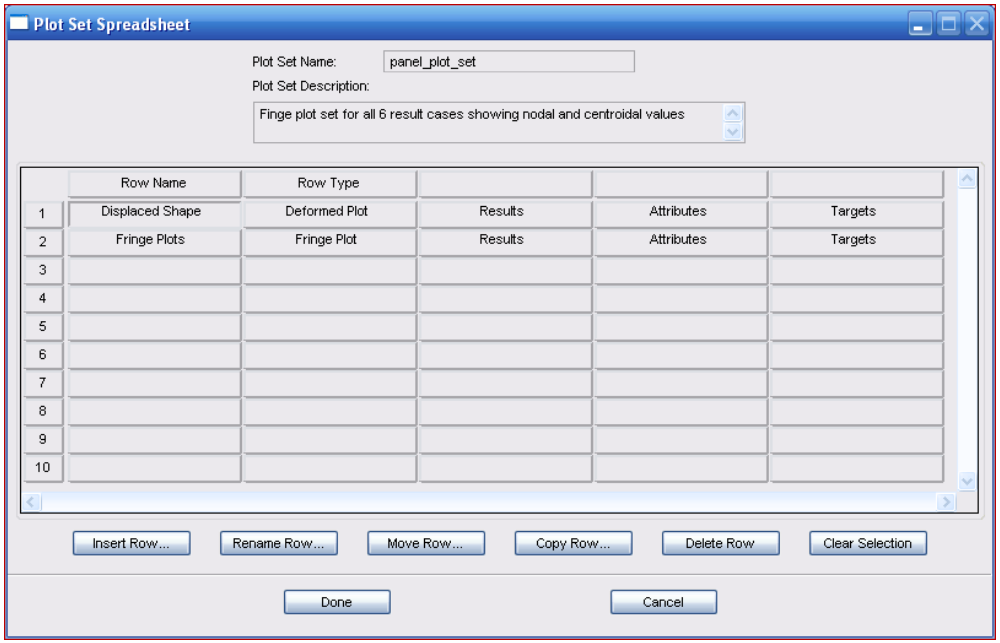
- **Action: Edit**
  - Click on Existing Plot Set Name & Click Edit
  - Editing done via a spreadsheet
    - Each Row represents a Plot Type contribution to the entire Plot Set
    - Renaming of Plot Set can be done via this interface



Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

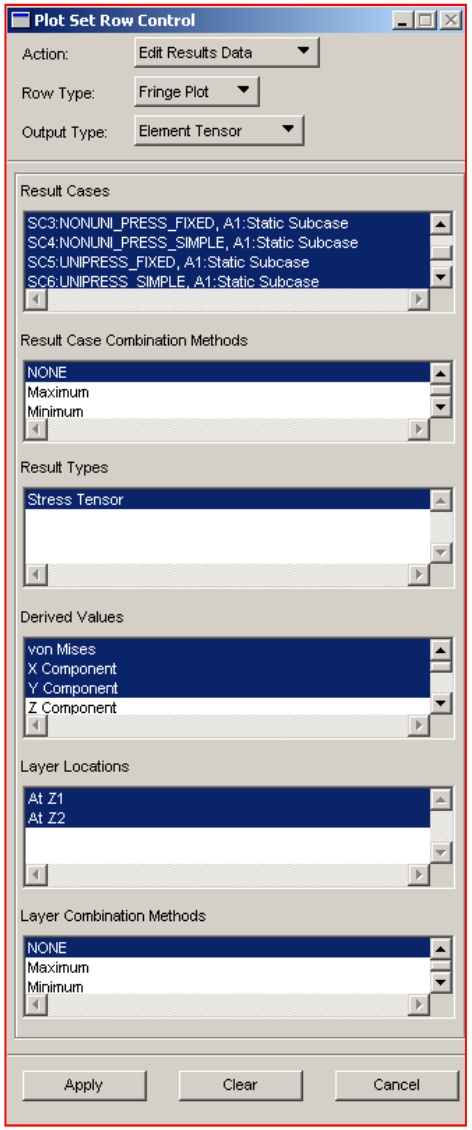
- **Action: Edit (cont.)**
  - Clicking on a Cell under column “Row Name” presents different Row Options (think of it as Row Management)
    - Insert, Rename, Move, Copy, Delete Rows
    - Clear Selection Brings you back to standard edit spreadsheet



Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

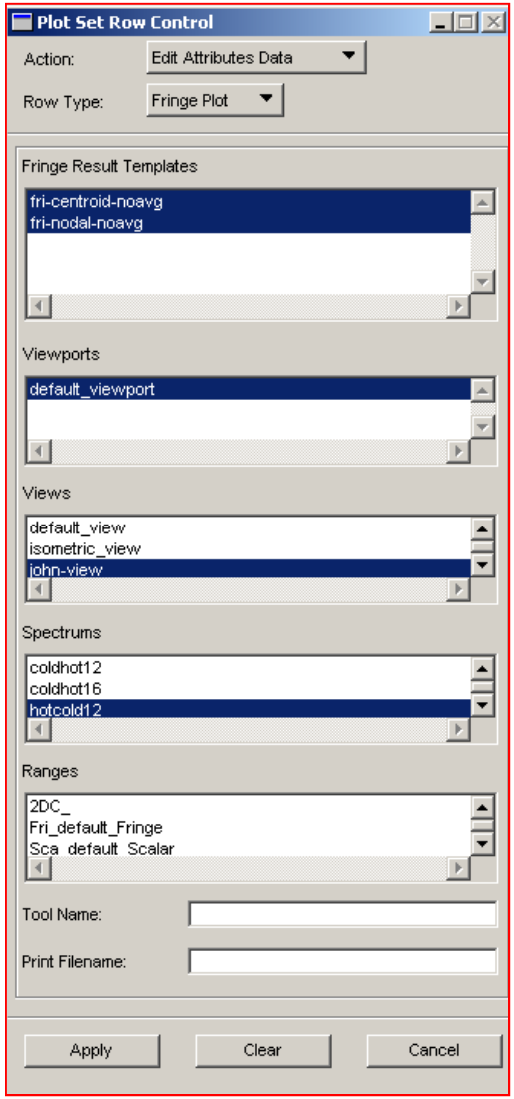
- **Action: Edit (cont.)**
  - Results/Attributes/Targets Provide different selection options for the Row Type being modified
    - Options for a Deformation Row are different from those for a Fringe or Marker Row
  - Sample Results Selection Form is shown
    - Select one or more Result Cases
    - Optionally can select a Combination method
    - Select one or more result types & quantities (derived values) where appropriate
    - Select one or more layer positions and combination methods where appropriate



Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

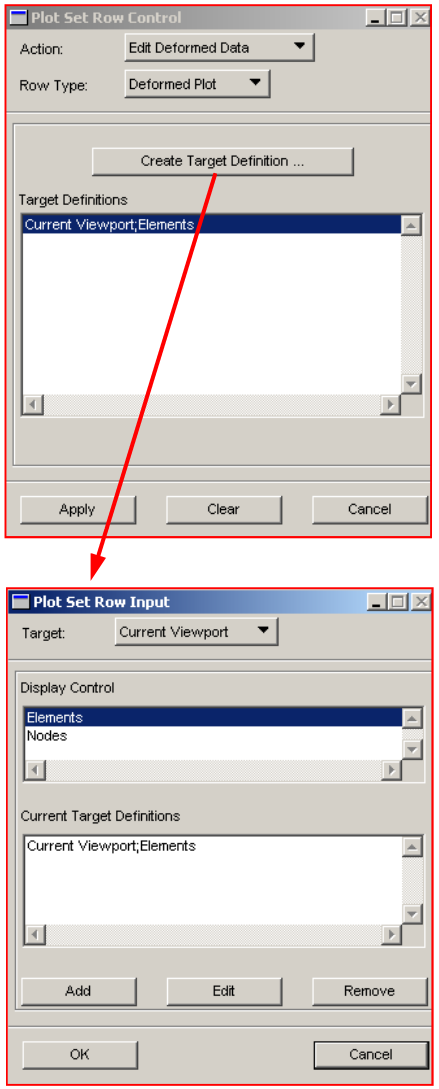
- **Action: Edit (cont.)**
  - Attribute Control
    - Select existing Results Templates
      - Need Results Templates before using Plot Sets
      - Select which Viewport, View (from Viewing/Named View Options pull-down menu), Spectrum, and Range (where appropriate)
    - Optionally can define a Tool Name that can be referenced later on within Patran under Results: Post
    - Optionally can define a filename where this series of plots can be stored
      - This is done when using the Print option in Plot Sets



Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

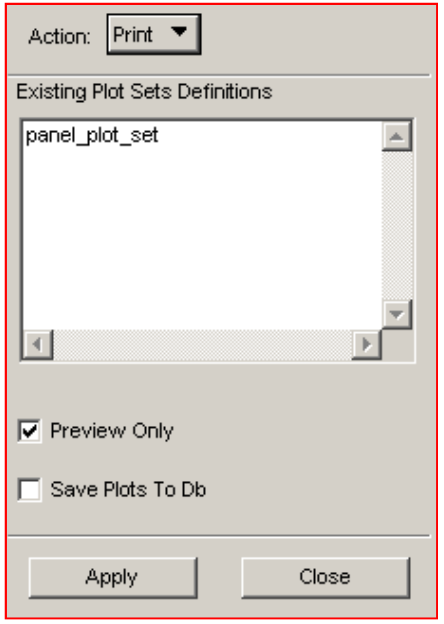
- **Action: Edit (cont.)**
  - Targets are the settings for what types of entities the results will be posted on
  - Click on Targets in the Plot Set row of choice
  - Click on Create Target Definition and select appropriate Display Control to add to Current Target Definitions
  - Targets can be thought of in the same way as Target Entities in the Results form
    - Current Viewport, Groups, Elements, etc.



Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

- **Action: Plot**
  - Once all the editing is done (saved as you go along, no need for Apply), it is time to review the Plot Set
  - Select Plot Set Name
  - Can either preview the plots within Patran, or preview and save
- **Note that “Save Plots to Db” actually does two things, depending on the attributes settings within the Plot Set Definition**
  - Plots generated can be saved to the database as Plot Names to be reviewed later under Results: Post
  - Plots generated can be saved to the directory where the images were defined to be saved (default is directory where Patran was started)

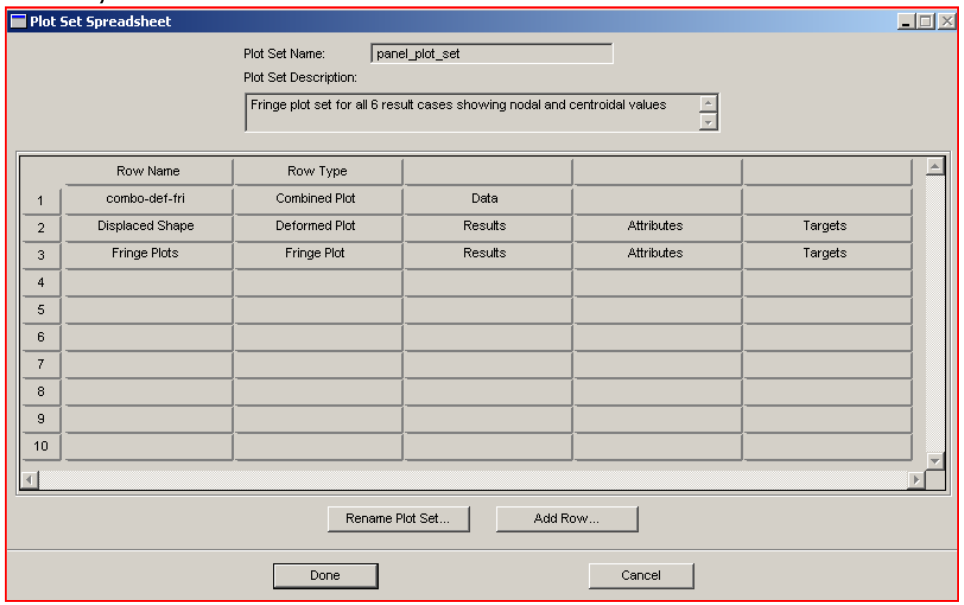


Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

- **Combined Plot Set Rows**
  - Option to Create a Row that will do Combination Plots of the Plot Set Rows immediately following the Combined Plot Set Row
    - For Combined Plots, you must define a Combination Plot Row before the Rows you wish to combine
    - When Plot Sets sees a new Combination Plot Row, the Plot Set Rows after that, until the next Combination Plot Row, will be used in that Combination

- **In this example, the first row is the Combination Plot row, and the next two rows for Deformation & Fringe will be used in the Combination Schema**



Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

- **Combined Plot Set Rows (cont.)**
  - Two types of Combination/Iteration methods to combine data
    - Iterator
      - Takes every possible combination of options within the rows following the combination definition and makes the appropriate number of plots
      - Example: Fringe Plot has 2 titles; Deformation Plot has 2 Subcases, 4 Show As, 2 Targets, 2 Titles, 4 Viewports
      - Resulting number of Plots created is 2 x 2 x 4 x 2 x 2 x 4, or 256 plots
    - Tuple
      - The Tuple iterator will produce individual plots for each possible pairing of the data values needed to describe each plot included in the Plot Set
      - Can make Mistakes here with unmatched pairings
      - Example: Fringe Plot has 2 Loadcases, 4 Targets, 2 Viewports; Deformation Plot has 2 Subcases, 4 Show As, 2 Titles
      - Since the number of pairings between Fringe and Deformation are the same,
      - 2 x 4 x 2, or 16 plots, will be combined with this method

Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

- **Bring it All Together: Example 1**
  - 2 Rows, Deformation & Fringe (no Combinations)
  - Deformation: 6 result cases, no result case combination methods, 1 result type (Displacements-Translational), shown as resultant
    - 1 Template, 1 Viewport, 1 View, 1 Target
    - 6 Plots total created (6 x 1 x 1 x 1 x 1 x 1)
  - Fringe: 6 result cases, no result case combination methods, 3 stress tensor quantities (Von Mises, X & Y Components), 2 Layers (Z1 & Z2), with no layer combination methods
    - 2 Templates, 1 Viewport, 1 View, 1 Spectrum, 1 Range, 1 Target
    - 72 Plots total created (6 x 3 x 2 x 2 x 1 x 1 x 1 x 1 x 1)

Copyright© 2012 MSC.Software Corporation

RESULTS PLOT SETS

- **Bring it All Together: Example 2**
  - Combination Plot Row that includes 2 Plot Set rows of Deformation & Fringe
  - Deformation & Fringe each have 6 result cases and 1 result quantity (Resultant Deformation & Von Mises Stress at Maximum of Z1/Z2 Layers), with 1 of every other Attributes/Targets/Templates
  - Product Iterator Method includes 2 rows and generates 6 x 6 or 36 plots
  - Tuple Method matches up the 6 Result Cases appropriately (since they are equally matched) and creates only 6 plots
    - Need to be very careful in defining Rows and Combination Plot Set Row that all the variations have equal or matching pairings

Copyright© 2012 MSC.Software Corporation

EXERCISES

- Perform Workshop 22 “Postprocessing Using Plot Sets” in your exercise workbook.

Copyright© 2012 MSC.Software Corporation