

Composite Material Analysis Using MSC Nastran

NAS113 Course Notes

June 2014

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 © 2014 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, MD 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	Introduction to Composites in MSC Nastran
	Applications
	Aerospace
	Motorsport
	Automotive
	Marine
	Energy
	Others
	Composite Terminology
	Constituent Materials
	Composite Development Process Diagram
	Laminate Tools from MSC
	SimCompanion
2	Overview of Classical Lamination Theory
	Ply Definition
	Tape Plies
	MAT8 Bulk Data Entry
	Patran 2D Orthotropic
	Composite Material – Stacking Plies
	PCOMP Bulk Data Entry
	Building Composites in Patran
	CQUAD4 Bulk Data Entry

CONTENTS

Section		Page
2	Overview of Classical Lamination Theory (Continued)	
	Patran Composite Properties	2-14
	Exercise – WS1 “Making a Composite Model”	2-21
	Ply Material Constants	2-23
	Rotation to Material Coordinate System	2-25
	Composite Properties	2-26
	MSC Nastran Composite Theory	2-28
	Equivalent PSHELL/MAT2	2-30
	MSC Nastran Lamination Matrices	2-31
	Patran Lamination Theory	2-32
	Patran Composite Pshell	2-34
	Types of Lay-Ups	2-36
	Effective Moduli	2-46
	Elastic Properties Plots	2-47
	Patran Material Symmetry Options	2-49
	Patran Property Symmetry Options	2-51
	Exercise – WS2 “Forward Swept Wing”	2-53
3	Advanced Composite Elements - Part 1 Composite Beam	
	Types of Composite Elements	3-2
	When Can I Use a Composite Beam Element	3-3
	Composite Beam Using VAM	3-4

CONTENTS

Section	Page
3	Advanced Composite Elements – Part 1 Composite Beam (Continued)
	Example 3-7
	Composite Beam Using VAM – Guidelines and Limitations 3-21
4	Advanced Composites Elements – Part 2 Solid Composite
	Types of Composite Elements 4-2
	Solid Composites 4-3
	General Anisotropic/Orthotropic Solid 4-4
	Material Coordinate System 4-6
	Anisotropic Material 4-7
	Anisotropic Material – MAT9 4-10
	Orthotropic Material 4-11
	Solid with Layered Information 4-14
	Solid – Layered Composite 4-17
	Solid Shell 4-18
	Defining Layered Solid or Solid Shell Composite Property 4-19
	Material Orientation 4-22
	Solid: Transverse Shear 4-25
	Example problem 4-26
	Three Different Run Techniques 4-37
	Shell Model 4-38
	Solid Model 4-43

CONTENTS

Section		Page
4	Advanced Composite Elements – Part 2 Solid Composite (Continued)	
	Layered Solid	4-46
	Solid Shell	4-51
	Exercise – WS3 “Layup of Solid composite Shell”	4-55
5	Composite Post-Processing and Failure Theories	
	Supported Solutions in MSC Nastran	5-2
	MSC Nastran Calculation of Ply Results	5-3
	Requesting Ply Stresses and Strains in MSC Nastran	5-6
	Viewing Ply Stresses and Strains in MSC Nastran	5-8
	Requesting Ply Stresses and Strain in Patran	5-9
	Plotting Stresses and Strains in Patran	5-12
	Ply and Laminate Failure Theorms	5-13
	Requesting Ply Failure Indices in MSC Nastran	5-15
	MAT8 Ply Allowables	5-16
	Fiber Direction Tension Allowable	5-18
	Fiber Compresssion Allowable	5-22
	Matrix Direction Tension Allowable	5-24
	Matrix Direction Compression Allowable	5-26
	Shear Allowable	5-28
	PCOMP Failure Fields	5-30
	Failure Theorems	5-31

CONTENTS

Section		Page
5	Composite Post-Processing and Failure Theories (Continued)	
	Failure Under Combined Loading	5-35
	Patran Ply Allowables Input	5-47
	Patran Composite Properties Form	5-49
	Patran Failure Index Request	5-51
	MSC Nastran Failure Index Output	5-52
	Patran Max Failure Index Plotting	5-54
	Strength Ratios	5-56
	PARAM,NOCOMPS	5-61
	PARAM, NOCOMPS in Patran	5-63
	Composite Plate Offset Z0	5-66
	CQUAD4 and CTRIA3 Corner Thicknesses	5-68
	General MSC Nastran Composite Interface Information	5-71
	Do Not Use QUAD8 or TRIA6 for Composites	5-72
	CQUAD4	5-73
	Global Plies	5-74
	Exercise – WS4 “Ply Direction Tailoring for Strength”	5-86
	PCOMP LAM Options	5-87

CONTENTS

Section

Page

6	Advanced Failure Theory and Prediction	
	3 Advanced Composite Failure Methods (PFA, VCCT, CZM)	6-2
	1. Progressive composite failure (PFA)	6-3
	Failure Mechanisms of Uni-Directional Lamina	6-4
	Composite Failure Criteria	6-6
	MSC Nastran Input Data Format	6-8
	Definitions:Strength Coefficients	6-9
	Types of Failure Criterion (Maximum, Hill, Hoffmann, Tsai-Wu, Hashin, Puck)	6-10
	User-Defined Criteria	6-33
	Progressive Failure	6-34
	Progressive Failure - Example	6-39
	Exercise -- WS 5 "Progressive Failure Analysis"	6-41
	2. Virtual Crack Closure Technique (VCCT)	6-43
	VCCT Basic Information	6-44
	VCCT in 3D	6-47
	VCCT – Crack Types	6-48
	Modes of Crack Extension	6-49
	VCCT – Definition in MSC Nastran	6-50
	VCCT – Basic (No Crack Propagation)	6-51
	VCCT – Crack Propagation	6-52
	VCCT – Crack Growth by Release Glued Contact	6-53

CONTENTS

Section		Page
6	Advanced Failure Theory and Prediction (Continued)	
	VCCT – Examples	6-55
	Exercise – WS 6 “VCCT – Double Cantilever Beam”	6-59
	3. Cohesive Zone Modeling (CZM)	6-63
	Cohesive Zone Modeling – Basic Information	6-64
	Cohesive Zone Modeling – Elements	6-65
	Cohesive Zone Modeling – Material	6-68
	Cohesive Zone Modeling – Viscous Damping	6-76
	Cohesive Zone Modeling – Material Definition	6-77
	Cohesive Zone Modeling – Results	6-78
	CZM Examples	6-79
	Exercise – WS 7 “CZM – Lap-Shear Joint”	6-84
7	Interlaminar Shear	
	[G3] Stiffness Matrix	7-2
	Transverse Shear Stiffness Theory	7-3
	Transverse Shear Stress	7-4
	Transverse Shear Stress Example Calculation	7-8
	Transvers Shear Modulus Example Calculation	7-16
	Patran Transverse Shear Properties	7-19
	Patran Transverse Shear Stress	7-20
	Exercise – WS8 “Transverse Shear Stress and Stiffness”	7-21

CONTENTS

Section	Page	
8	Nonlinear Composite Analysis	
	Linear vs. Nonlinear Analysis	8-2
	FEM Quantities in Linear Analysis	8-3
	Linear Analysis Consequences	8-5
	Nonlinear Analysis	8-6
	Sources of Nonlinearities	8-8
	Contact and Constraint Changes	8-9
	Geometric Nonlinearity	8-10
	Material Nonlinearity	8-14
	Case Control Setup Between Linear and Nonlinear Solution	8-20
	SOL 400 Input File Example	8-21
	Offset Modeling	8-23
	Element Offsets for Beams and Shells	8-25
	Example	8-26
9	Buckling Analysis	
	Linear Buckling (SOL 105)	9-2
	Linear Theory	9-3
	Solution of the Eigenvalue Problem	9-4
	Rules for SOL 105 Buckling Analysis	9-5
	Data Entries for Linear Buckling	9-6
	EIGRL Entry	9-9

CONTENTS

Section	Page
9 Buckling Analysis (Continued)	
Composite Plate Linear Buckling Example	9-10
Exercise – WS9 “Linear Buckling”	9-13
Nonlinear Buckling	9-14
Instability Phenomena	9-15
Linear vs. Nonlinear	9-18
Data Entries for Nonlinear Analysis	9-20
NLSTEP Entry	9-21
Buckling and Element Offsets	9-28
Post Buckled Strength	9-31
Exercise – WS10 “Post Buckled Strength”	9-37
Honeycomb Sandwich Shear Buckling	9-38
10 Optimization of Composites	
Definition of MSC Nastran Optimization	10-2
Objective – DESOBJ	10-5
Responses – DRESP1	10-6
Design Variables – DESVAR	10-7
Discrete Design Variables - DDVAL	10-8
Property Variables – DVPREL1	10-10
Constraints – DCONSTR	10-12
MSC Nastran Optimization Input File	10-14
Patran Interface	10-16

CONTENTS

Section	Page
10 Optimization of Composites (Cont.)	
Modification Needed for Some Optimization Entries	10-29
MSC Nastran Optimization Output	10-30
Patran Optimization Output	10-36
Exercise – WS11 “Ply Direction Tailoring Strength Using the Optimizer”	10-40
Exercise – WS12 “Ply Direction Tailoring for Stiffness Using the Optimizer”	10-41
Case Study – Fixed Total Thickness	10-42
11 Practical Usage Guidelines	
Strength Ratio Output for Laminated Composites	11-2
Stress Calculation	11-8
Coordinate System for Solid Composite	11-9
Nonlinear Analysis	11-10
Plotting Tips for Nonlinear Analysis	11-11
Tips for Glue Contacts	11-12
Use Optimization	11-14
12 Laminate Modeler	
Patran Composites : Laminate Builder Tool	12-6
Patran Composites: Failure Indices	12-7
Why Laminate Modeler?	12-8
What is a Laminate Modeler?	12-9
Getting Started with Laminate Modeler	12-10
Creating LM_Materials	12-13

CONTENTS

Section	Page
12 Laminate Modeler (Continued)	
Creating LM_Plies	12-14
Verifying/Showing LM_Plies	12-17
Creating LM_Layups	12-18
Verifying/Showing LM_Layups	12-20
Verifying/Showing Laminate	12-23
Composite Data Management	12-27
Support for Solid Elements	12-29
Interface to Fibersim	12-30
Post Processing Support	12-31
Creating Laminate Results	12-32
Why Use MSC Laminate Modeler?	12-34
Appendix	
A Composite Theory	

SECTION 1

INTRODUCTION TO COMPOSITES IN MSC NASTRAN

OVERVIEW

- **The objective of the seminar is to familiarize attendees with definitions of composite theories in MSC Nastran which include:**
 - Composite user interface
 - Composite definitions
 - Lamination theory
 - Composite failure
 - Ply tailoring
 - Interlaminar shear
 - Solid Composite
 - Advanced Failure Theories
 - Introducing the usage of optimization as related to composite
 - Buckling and nonlinear analysis
- **Patran composite user interface is also discussed and shown.**

WHAT IS A COMPOSITE MATERIAL?

- **A composite material is a combination of two or more materials that has superior performance than the individual constituents**
- **A composite is usually a strong material (the “reinforcement”) surrounded by another material (the “matrix”)**
 - The reinforcement provides strength and stiffness
 - The matrix holds the reinforcement in place and transfers load to the reinforcement
- **A composite that uses fibers as the reinforcement is a “fiber reinforced composite” (FRC) or sometimes “fiber reinforced plastic” (FRP)**
- **The reinforcement can be randomly distributed or it can be oriented in specific directions.**
- **If it is arranged in layers, its called a “laminated composite” and a each layer is called a “ply” or a “lamina”.**

APPLICATIONS

- **Composites are used in many different industries**
 - Aerospace
 - Motor Sports
 - Auto
 - Marine
 - Energy
 - Sporting Goods
 - Others

Aerospace

- Early composite aircraft



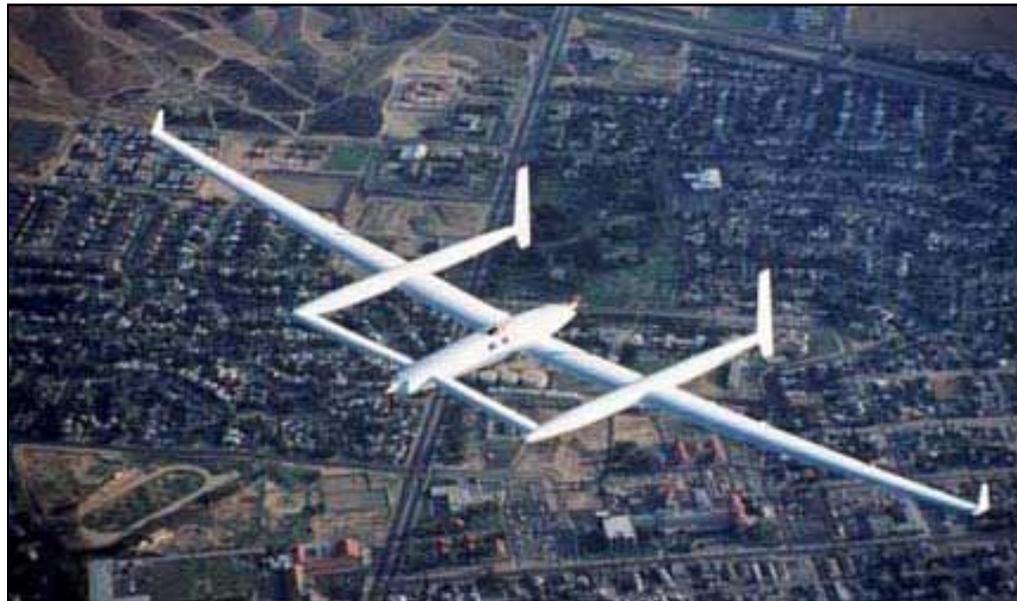
AEROSPACE

- **F-111 horizontal stabilizer was the first flightworthy boron composite component (1964).**



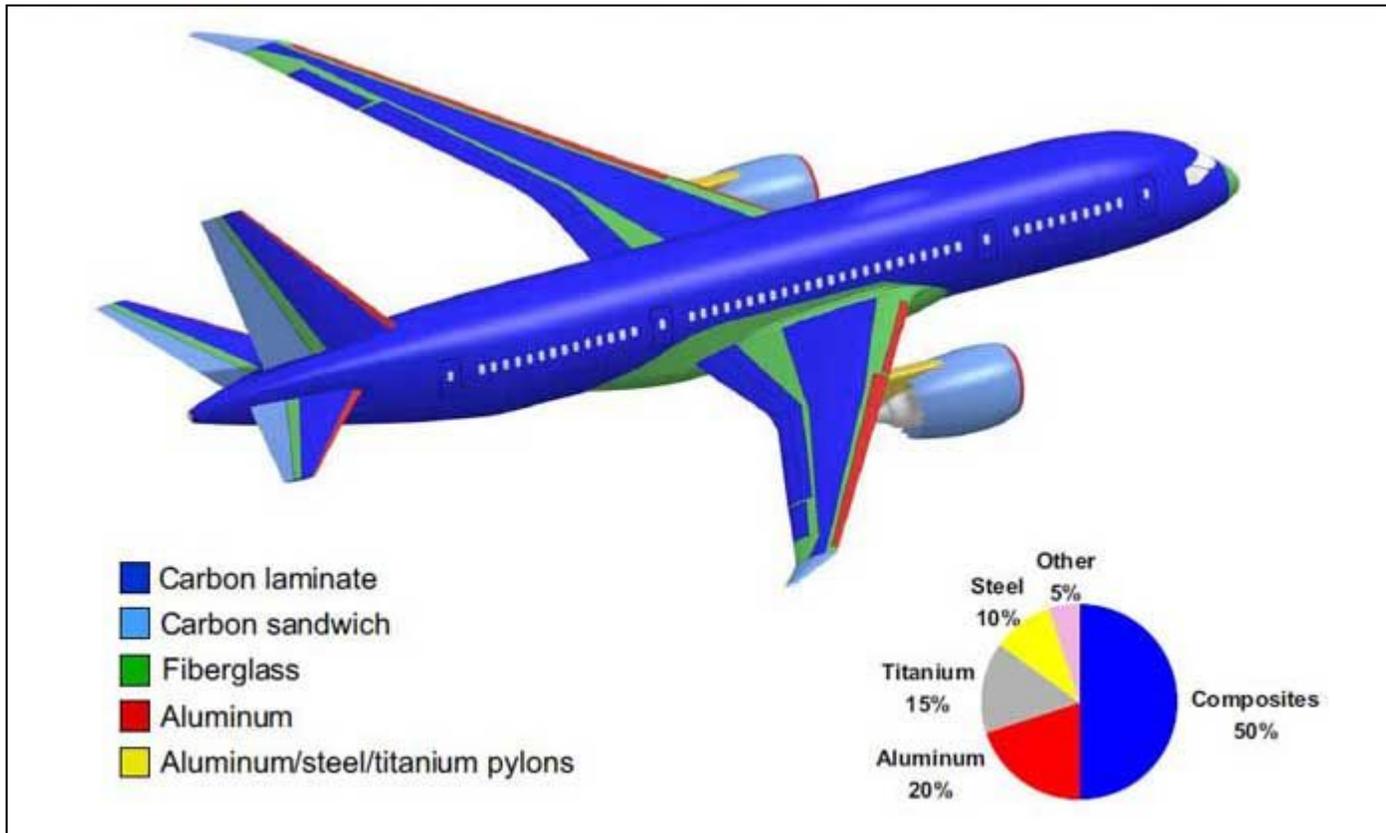
AEROSPACE

- **Voyager Aircraft (1986)**
 - Constructed entirely of composite material. The airframe (without fuel) weighed only 9% of the take off weight.
 - The first non-stop flight around the world took nine days to complete.



AEROSPACE

- Boeing 787 “dreamliner” structure is 50% composite, 20% aluminum



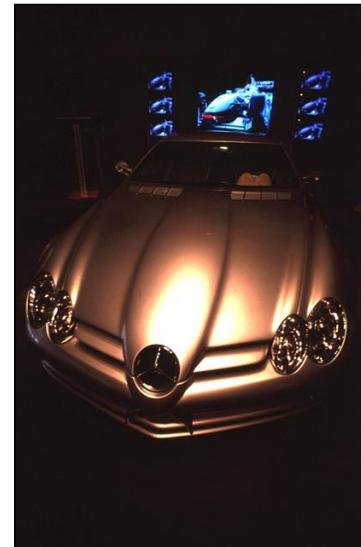
MOTORSPORT

- **Most race cars are largely composite**
- **Desirable for many of the same reasons as in aerospace**
 - Light
 - Stiff
- **Also:**
 - Easily molded into complex contours without expensive tooling
 - Absorbs energy in a crash better than metal



AUTOMOTIVE

- **McLaren F1**
 - Super car
 - Cost \$750K
- **Mercedes-Benz SLR**
 - Sports car
 - Carbon fibre bodywork & monocoque
 - Price c. \$250K
- **Now being introduced to mainstream production cars**
 - Hoods
 - Body panels
 - Suspension components
 - Engine mounts
 - Etc.



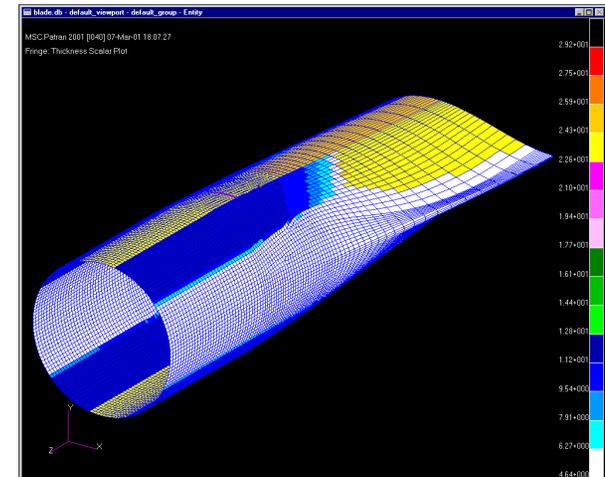
MARINE

- **Composites provide:**
 - Stiffness for rigid sails
 - Reduced weight
 - Corrosion resistance
 - Improved strength and fatigue
 - Reduced radar return



ENERGY

- **Wind Turbine Blade**
- **Up to 80m long**
- **Composites provide:**
 - Stiffness for such a long slender part
 - Fatigue and corrosion resistance for long life with reduced maintenance



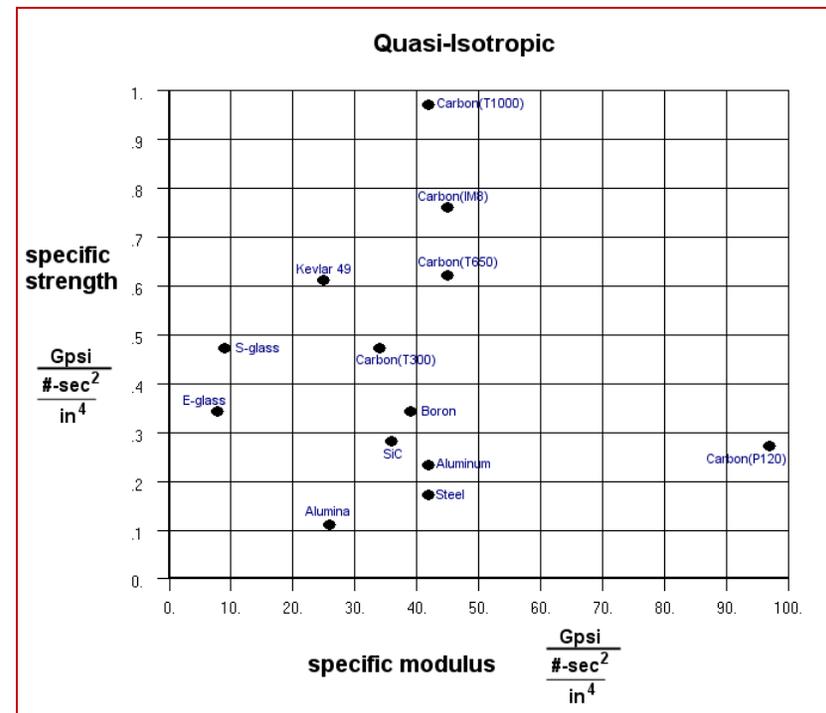
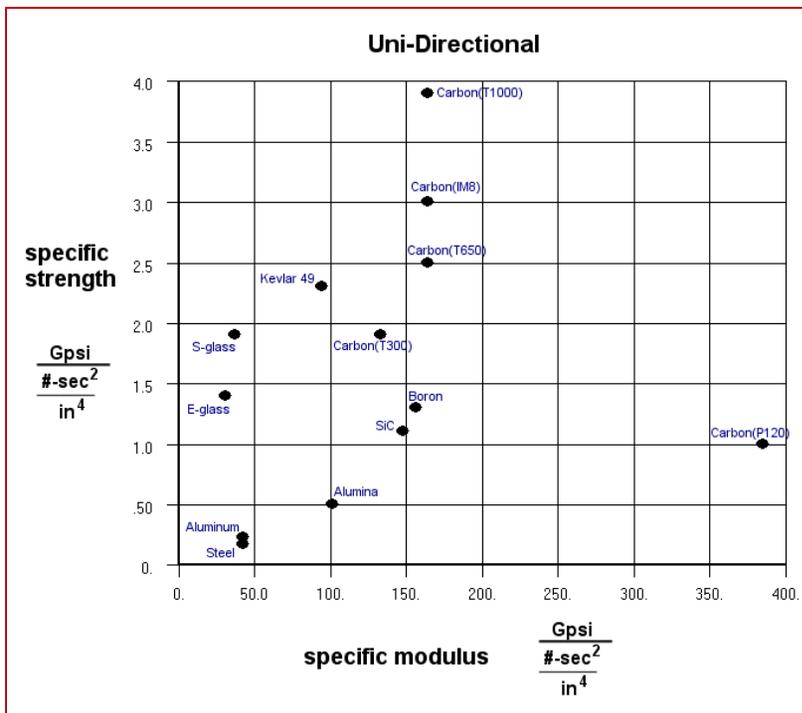
OTHERS

- Helmets
- Tennis rackets
- Bicycle frames and wheels
- Prosthesis limbs
- Golf clubs
- Kayaks

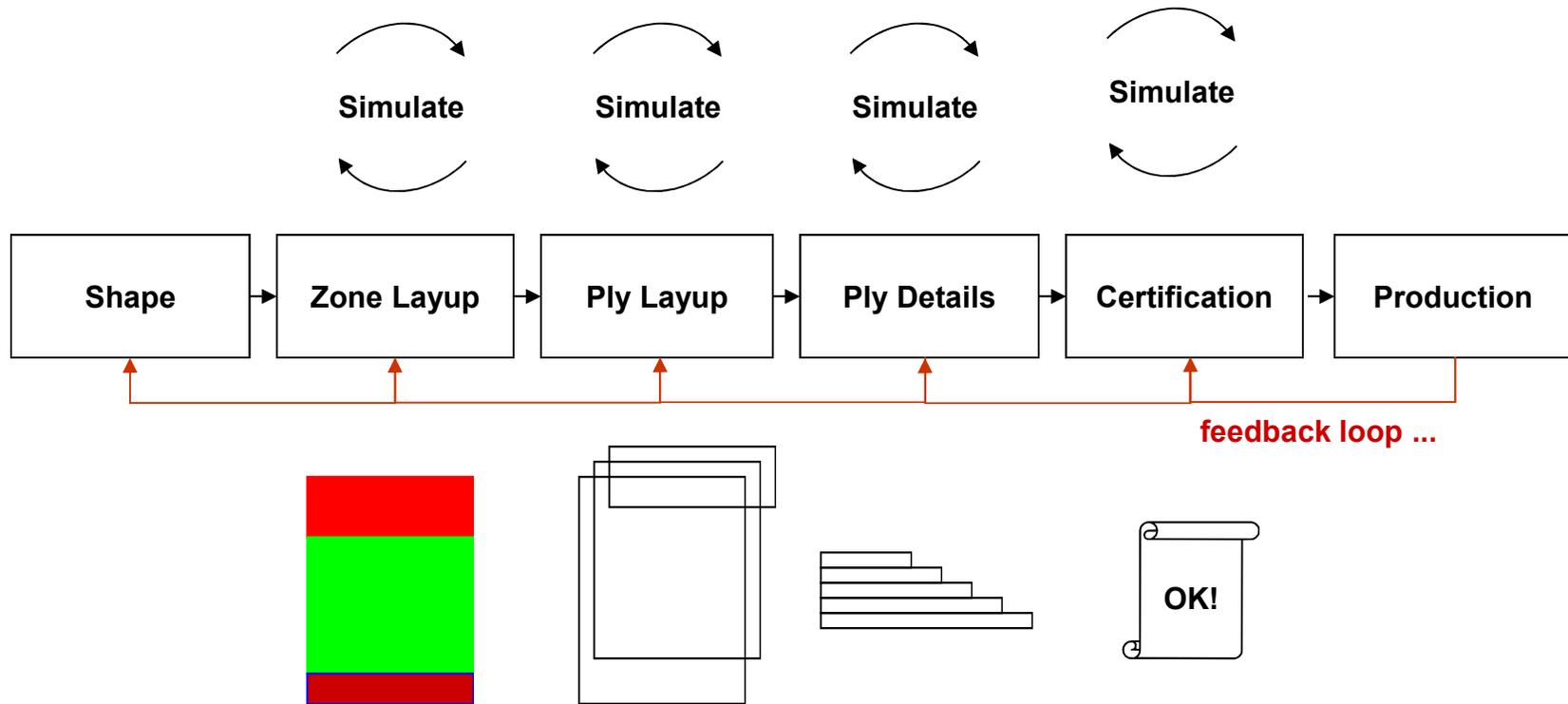


MATERIAL COMPARISON

- These plots compare the specific strengths and moduli of different materials
- Specific means divided by density
- Many applications require light weight as well as strength and stiffness



COMPOSITES DEVELOPMENT PROCESS



TOOLS

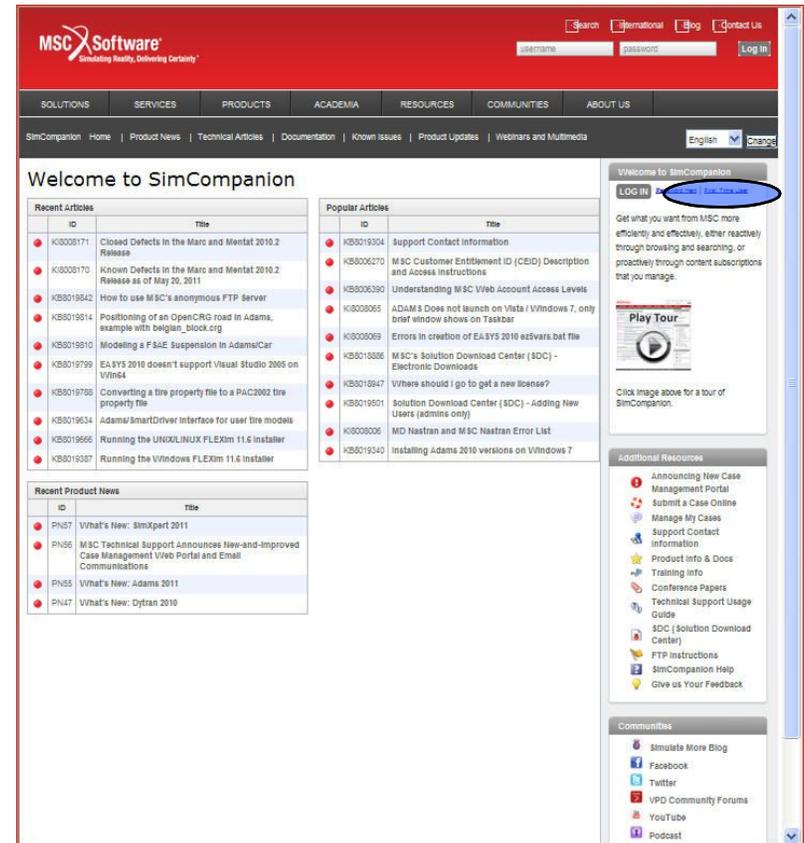
- **Composites development is a multi-faceted process where design, analysis, and manufacture must be considered simultaneously.**
- **The aim of simultaneous engineering is to use various analysis results to drive the optimization of a structure and its manufacturing process.**
- **MSC Software offers a range of CAE products which can enhance the concurrent engineering of composites:**
 - Material engineering
 - Composite part analysis
 - Material data management

MSC PRODUCTS FOR COMPOSITES

- **In addition to MSC Nastran, MSC has other tools for engineers who are designing with composite materials**
- **These include:**
 - Laminate Modeler – a graphical tool for defining composite materials and evaluating results
 - Digimat – a specialized analysis tool for predicting and optimizing the performance of a composite
 - Material Center – a web-enabled database system for storing, controlling, and distributing material data
 - Marc – A nonlinear finite element program that shares many features with MSC Nastran
- **All of these interface with MSC Nastran to share data as part of an integrated composite material design and analysis process**

SIMCOMPANION

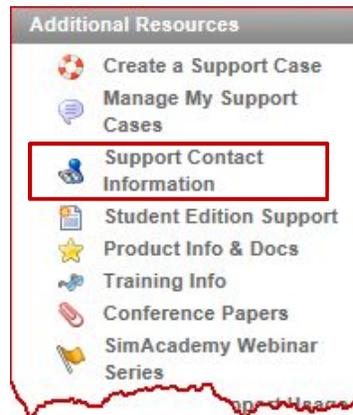
- One stop for full online support
- Find answers to your questions
- Search across ALL content
- Subscribe to email notification
- Single sign-on to ALL content
- Access to other support resources
 - Case Management Portal
 - Discussion Forums
 - Training Information



<http://simcompanion.mscsoftware.com>

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	supporte.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 89 431 967 277
Denmark	+45 61 22 32 00
Finland	0800-9-14709
France	+33 5 34 60 44 80

SIMCOMPANION

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

Additional Resources

- [Create a Support Case](#)
- [Manage My Support Cases](#)
- [Support Contact Information](#)
- [Student Edition Support](#)
- [Product Info & Docs](#)
- [Training Info](#)
- [Conference Papers](#)
- [SimAcademy Webinar Series](#)
- [Technical Support Usage Guide](#)
- [SDC \(Solution Download Center\)](#)
- [FTP Instructions](#)
- [SimCompanion Help](#)
- [Give us Your Feedback](#)

Docs

Product Information and Documentation

Docs ID: DOC9275
Status: Published
Published date: 09/25/2009
Updated: 07/03/2013

Description

Please click on desired MSC Product icon, to find the summary of Product Information and Documentation for current and prior versions, such as:

- What's New
- Release Guides
- Hardware & Software Requirements
- Set Up Guides (Installation, Licensing, and Configuration)
- Other product-specific content...

CAE Tools

MSC Nastran Product Information & Documentation

Back to all Docs | Email Page | Printer Friendly | Recommend Change | Subscribe | Manage Document

Docs ID: DOC9282
Status: Published
Published date: 09/27/2009
Updated: 03/25/2014
Reported In: MSC Nastran - MSC Nastran Docs

Description

MSC Nastran Product Information & Documentation

	Version 2013.1.1	Version 2013.1	Version 2013	Version 2012.2	Version 2012	Version 2010	Version 2008 r1	Version 2007 r1	Version 2005
What's New									
Release Guide									
Hardware & Software Requirements									
Set Up Guides (Installation, Licensing, & Configuration)									
User's Guides									
Getting Started with MSC Nastran									
Linear Static Analysis									
Dynamic Analysis									
Quick Reference Guide									
Design Sensitivity and Optimization									
Implicit Nonlinear									
Explicit Nonlinear									
DMAP Programmer's Guide									
Demonstration Problems									

SIMCOMPANION

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

Communities

-  Simulate More Blog
-  Facebook
-  Twitter
-  VPD Community Forums
-  YouTube
-  Podcast



My Account | Search | International | Blog | Contact Us

Log Out

Products
Industry Solutions
Services
Academic
Training & Support
Partners
Corporate
Store

Search Simcompanion My Community Home | FAQ | Getting Started

SimDesigner | SimManager | SimXpert

Actran | Adams | Dytran | Easy5 | Fatigue | Marc | Nastran | Patran | Universities & Student Edition

Main Index | Search | My Profile | My Subscriptions | Who's Online | User List | Mark all read

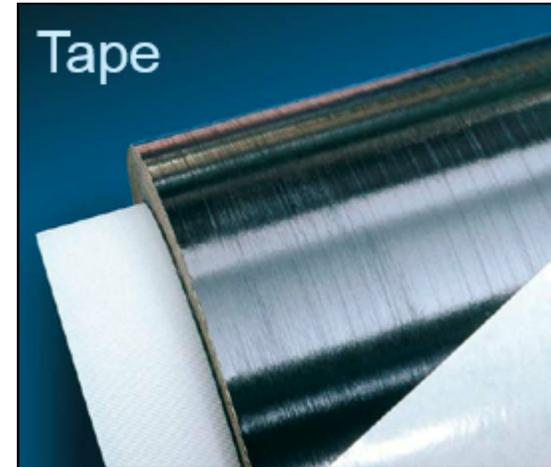
General News	Threads	Posts	Last post
<ul style="list-style-type: none">  MSC Software Corporate News News articles and press releases relating to MSC Software.  Virtual Product Development Events Worldwide events discussing the application of Virtual Product Development such as User Conferences, Trade Shows, Seminars and Webinars.  Virtual Product Development Stories Practical stories describing the application of Virtual Product Development tools in industry. 	4 (4 new)	4 (4 new)	<ul style="list-style-type: none">  MSC Software Corporate New... (twebb) - 02/03/05 04:11 AM  2009 MSC Software VPD Conf... (mtown) - 02/06/09 06:31 AM  NASTRAN + ADAMS "Marriage" (pmcna) - 06/07/04 10:34 AM
Core Product News	Threads	Posts	Last post
<ul style="list-style-type: none">  News for the Actran Community Product Announcements, Product Alerts and Technical Tips relating to Actran.  News for the Adams Community Product Announcements, Product Alerts and Technical Tips relating to your Adams products.  News for the Dytran Community Product Announcements, Product Alerts and Technical Tips relating to your Dytran products.  News for the Easy5 Community Product Announcements, Product Alerts and Technical Tips relating to your Easy5 products.  News for the Fatigue Community Product Announcements, Product Alerts and Technical Tips relating to Fatigue products.  News for the Marc Community Product Announcements, Product Alerts and Technical Tips relating to your Marc products.  News for the Nastran Community Product Announcements, Product Alerts and Technical Tips relating to your Nastran products.  News for the Patran Community Product Announcements, Product Alerts and Technical Tips relating to your Patran products. 	2 (2 new)	2 (2 new)	<ul style="list-style-type: none">  MSC Actran MasterKey licen... (jjanevic) - 06/16/04 02:46 PM  MSC launches Adams 2007 r1 (Helen) - 05/07/07 12:15 PM  Dytran 2007 r1 released (Helen) - 05/09/07 11:14 AM  Webinar - Simulation of Co... (Tim_Madsen) - 08/05/05 10:28 AM  Webinar - Simulation of Co... (Tim_Madsen) - 08/04/05 07:11 PM  Marc 2007 r1 is launched! (Helen) - 07/18/07 04:56 PM  MSC Nastran 2007 r1 releas... (Helen) - 05/09/07 10:48 AM  Patran 2008 r2a Patch Updat... (doug_gagne) - 09/29/09 10:01 AM
SimDesigner Product News	Threads	Posts	Last post
<ul style="list-style-type: none">  News for the MSC SimDesigner Community Product Announcements, Product Alerts, and Technical Tips relating to your MSC SimDesigner products. 	21 (22 new)	23 (23 new)	<ul style="list-style-type: none">  SimDesigner Forums Reorgan... (DavidI) - 06/29/06 03:44 PM

SECTION 2

OVERVIEW OF CLASSICAL LAMINATION THEORY

PLY DEFINITION

- Typically, a ply is a flat group of fibers embedded in a matrix.
- The matrix is usually an isotropic material that holds the fibers together.
- In a ply called **Tape**, the fibers are unidirectional.
- In a ply called **Cloth**, the fibers are woven at 0 and 90 degree directions.

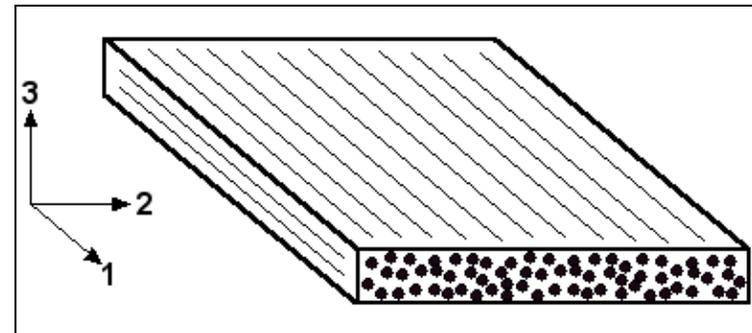


TAPE PLIES

- **Fiber:**
 - Unidirectional
 - Direction is the 1 axis of the ply coordinate system

- **Matrix:**
 - Glue that holds fibers together
 - Matrix direction is the 2 axis
 - 90 degrees to the 1 axis

- **Material properties are:**
 - 2D orthotropic material in Patran
 - MAT8 in MSC Nastran



MAT8 BULK DATA ENTRY

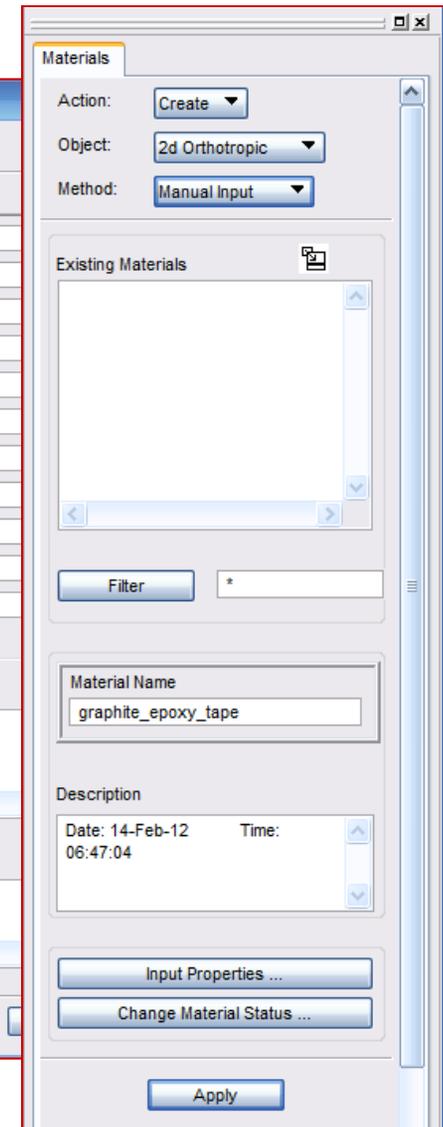
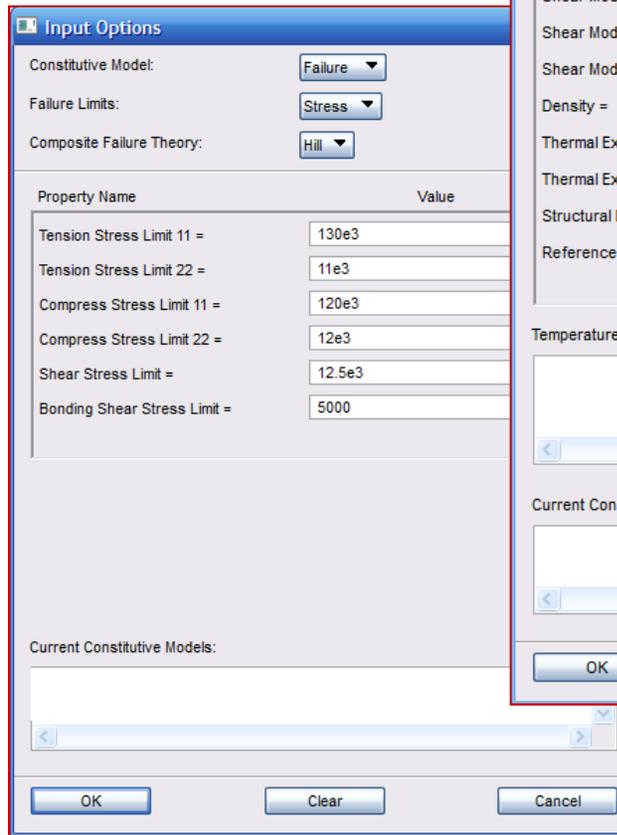
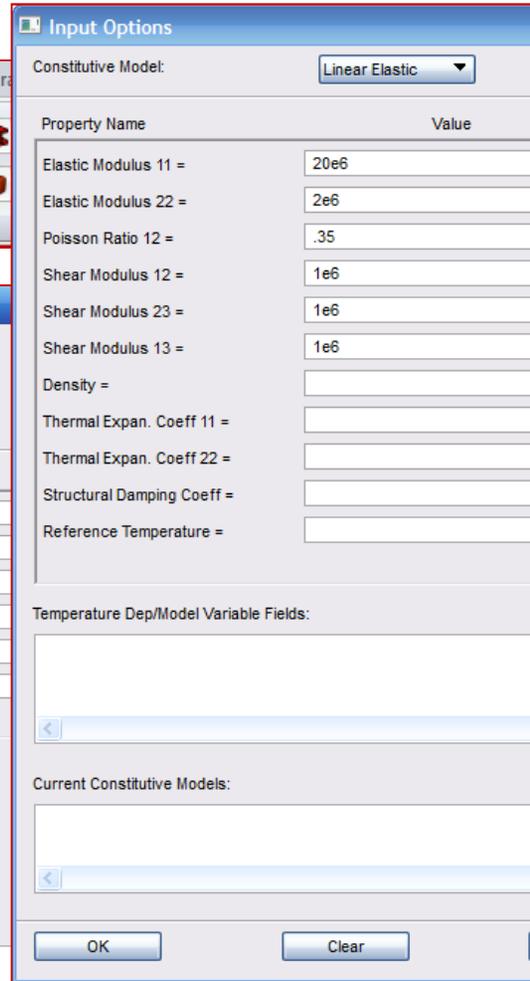
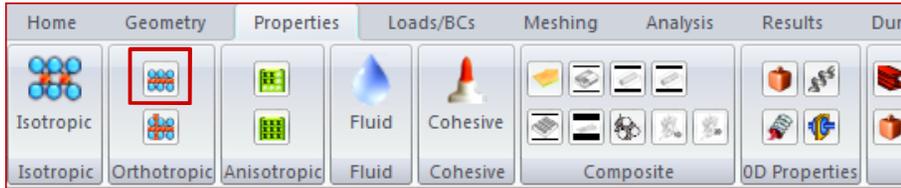
- **Defines the ply orthotropic properties**
 - Elastic properties are E1, E2, NU12, G12, G1Z, G2Z.
 - Allowables are Xt, Xc, Yt, Yc, S.
 - Use STRN=1.0 if allowables are in units of strain.
 - F12 is used for the Tsai-Wu failure theorem.
 - Thermal coefficients of expansion are A1 and A2.
 - The MAT8 TREF reference temperature is not used since it is overridden by the PCOMP TREF.
 - Density is RHO.
 - The MAT8 GE structural damping is not used since it is overridden by the PCOMP GE field.
- **The example below is typical for a graphite/epoxy tape:**

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
MAT8	1	20.+6	2.+6	0.35	1.6+6	1.6+6	1.6+6	1.3-4	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
	-2.3-7	4.5-6		1.3+5	1.2+5	1.1 +4	1.2+4	1.25+4	
	GE	F12	STRN						

.bdf file extract

```
mat8, 1, 20.+6, 2.+6, 0.35, 1.0+6, 1.0+6, 1.0+6, 1.3-4,+
+, -2.3-7, 4.5-6,, 1.3+5, 1.2+5, 1.1+4, 1.2+4, 1.25+4
```

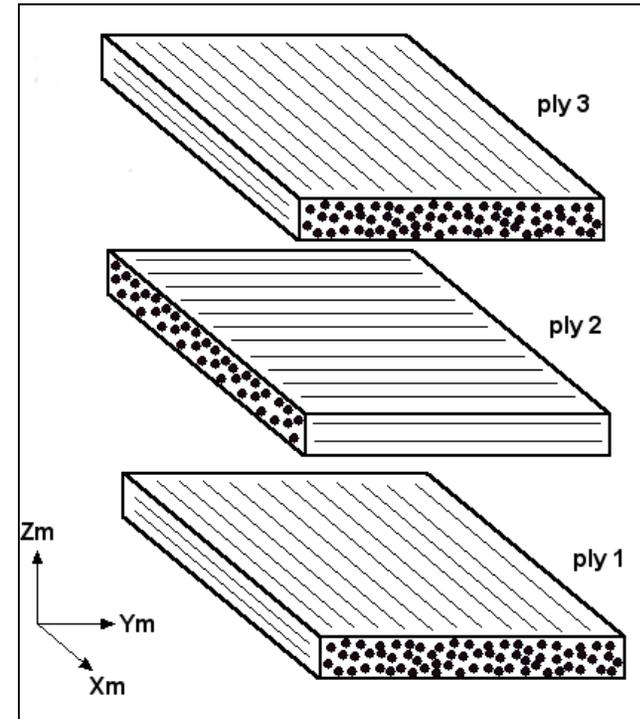
PATRAN 2D ORTHOTROPIC



Properties: Orthotropic
 Create/ 2d Orthotropic/ Manual Input
 Material Name
 Input Properties
 Linear Elastic
 Apply
 Input Properties
 Failure
 Apply
 Note that Linear Elastic and Failure properties must be input separately with an Apply between and after.

COMPOSITE MATERIAL

- Stack of plies
- Each ply has a different direction, material, and thickness
- Composite properties are calculated in the material coordinate system (X_m , Y_m , Z_m)



SYMMETRIC LAY-UPS

- Symmetric lay-ups have a mirror image of the lay-up above the centerline as below the centerline.

ply #	mat id	thickness	angle
3	1	0.0054	0.0
2	1	0.0054	90.
1	1	0.0054	0.0

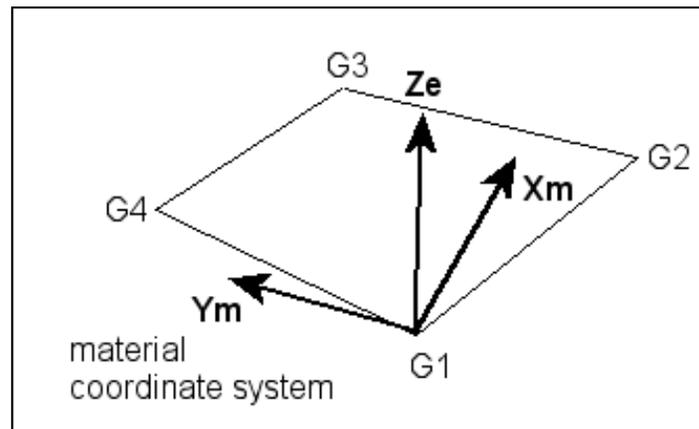
0/90/0

ply #	mat id	thickness	angle
8	1	0.0054	0.0
7	1	0.0054	45.
6	1	0.0054	-45.
5	1	0.0054	90.
4	1	0.0054	90.
3	1	0.0054	-45.
2	1	0.0054	45.
1	1	0.0054	0.0

(0/45/-45/90)_{sym}

COMPOSITE MATERIAL

- Z_m is the same as the element Z axis (Z_e)
 - Right hand rule of grid ordering--G1,G2,G3,G4
- X_m is in the direction of the 0 degree ply
- Positive angles are defined by right hand rule around Z_m



PCOMP BULK DATA ENTRY

- Defines the composite layout

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
PCOMP	1			5000.0	HILL	0.0			
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2	
	1	0.0054	0.0	YES	1	0.0054	45.0	YES	
	MID3	T3	THETA3	SOUT3	ect.				
	1	0.0054	90.0						

- Z0 is composite offset
 - Use default = -(composite thickness)/2
- NSM is nonstructural mass
- SB is allowable interlaminar shear stress
 - Put as Bonding Shear Stress in Patran 2D
 - Orthotropic Material (page 2-6)
 - **Required for failure indices**
- FT is the ply failure theorem
 - **Required for failure indices**
- TREF is reference temperature
 - Overrides TREFs on MAT8 plies
- GE is element damping
 - Overrides GE on MAT8 plies
- LAM is layup options
- MIDi is ply material ID
 - MAT8 ID
- Ti is ply thickness
- THETAi is ply angle
- SOUTi is data recovery option

PCOMP BULK DATA ENTRY

- The composite example below is an 8 ply layup, symmetric about its centerline, with an equal number of plies in each of the 0, +45, 90 degree directions.

ply #	mat id	thickness	angle
8	1	0.0054	0.0
7	1	0.0054	45.
6	1	0.0054	-45.
5	1	0.0054	90.
4	1	0.0054	90.
3	1	0.0054	-45.
2	1	0.0054	45.
1	1	0.0054	0.0

Ze
↑

----- centerline

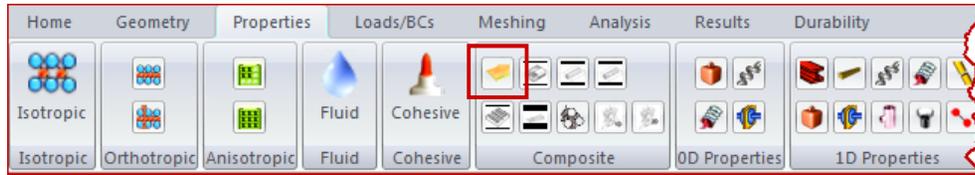
.bdf file extract

```
PCOMP, 1,,, 5000., HILL
, 1, .0054, 0., YES
, 1, .0054, 45., YES
, 1, .0054, -45., YES
, 1, .0054, 90., YES
, 1, .0054, 90., YES
, 1, .0054, -45., YES
, 1, .0054, 45., YES
, 1, .0054, 0., YES
```

Equivalent input

```
PCOMP, 1,,, 5000., HILL,,SYM
, 1, .0054, 0., YES
, , , 45.,
, , , -45.,
, , , 90.,
```

PATRAN COMPOSITE



Properties: Composite

Create/ Composite/
Laminate

To create a ply, click on a
ply material in Existing
Materials. Repeat for
each of the plies

Thickness for all layers:
0.0054 <return>

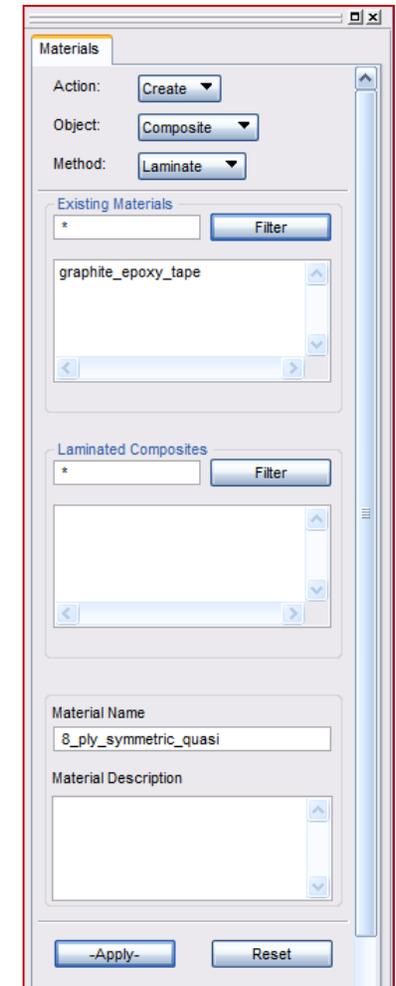
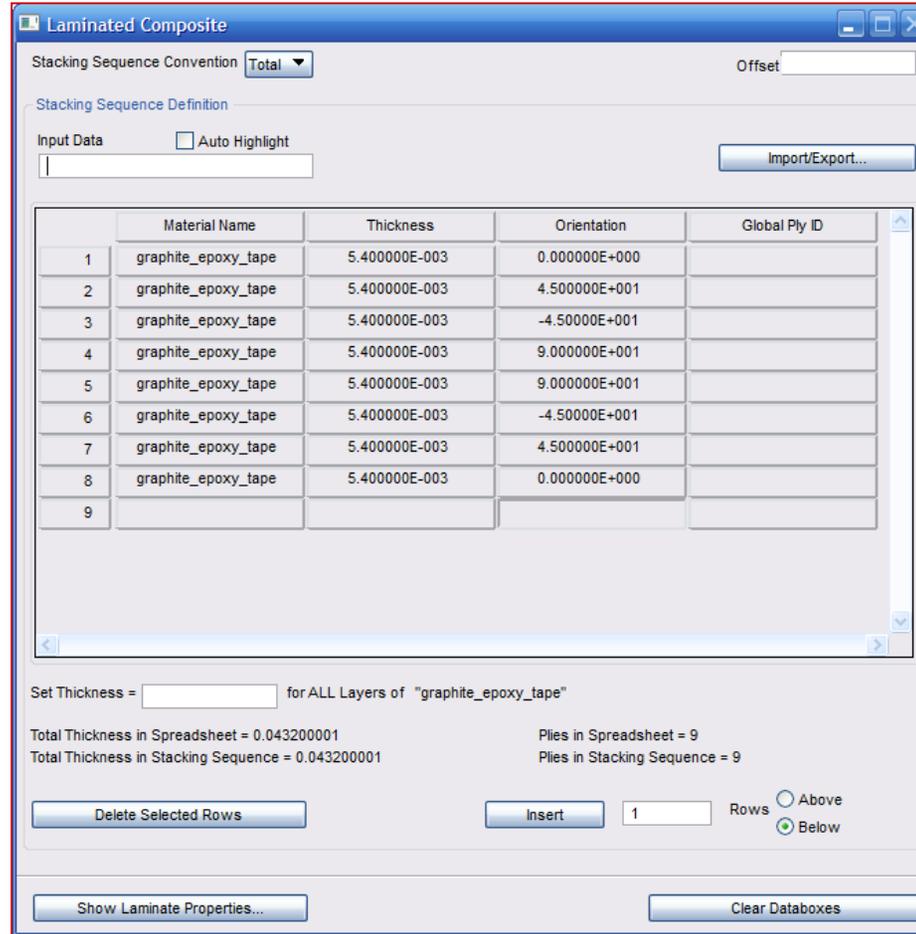
Click on first cell in
Orientation column

Text Entry Mode =
Overwrite

Orientations: 0 45 -45 90
90 -45 45 0

Load Text Into
Spreadsheet

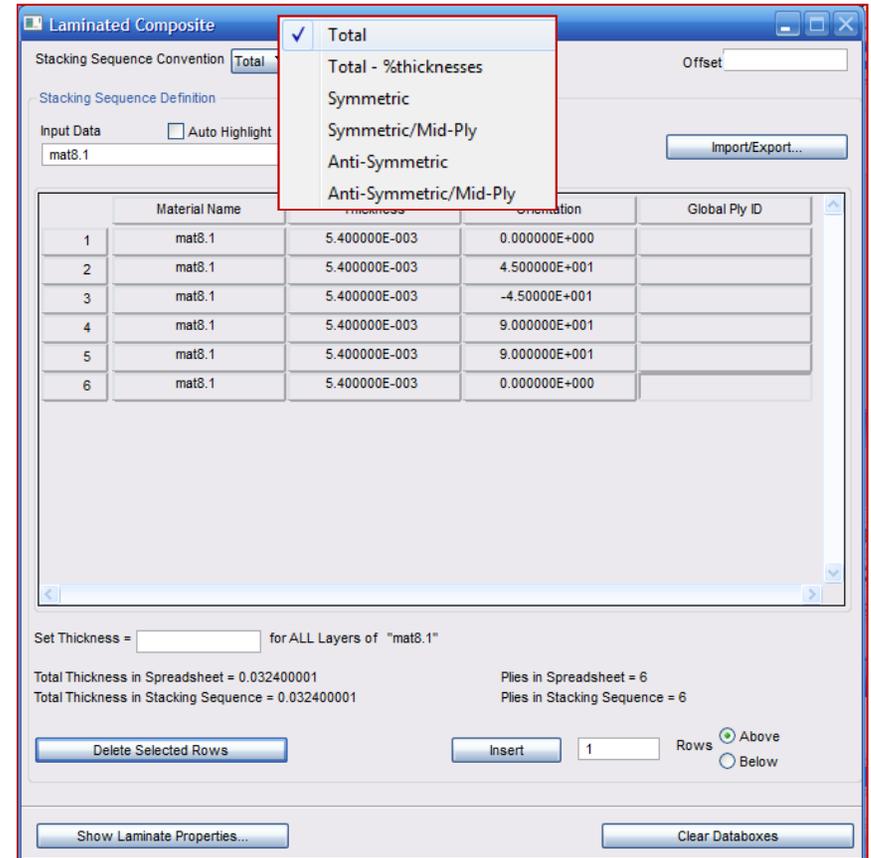
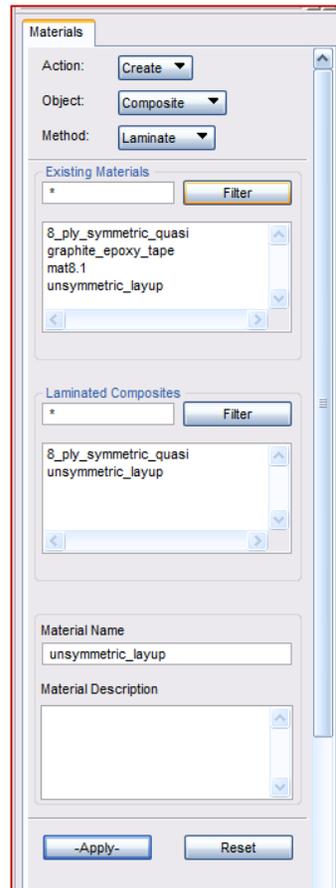
Apply



PATRAN MATERIAL SYMMETRY OPTIONS



Materials:
 Create/ Composite
 Select composite material in Laminated Composite section
 Pull down Stacking Sequence Convention
 Select symmetry type



PATRAN MATERIAL SYMMETRY TYPES

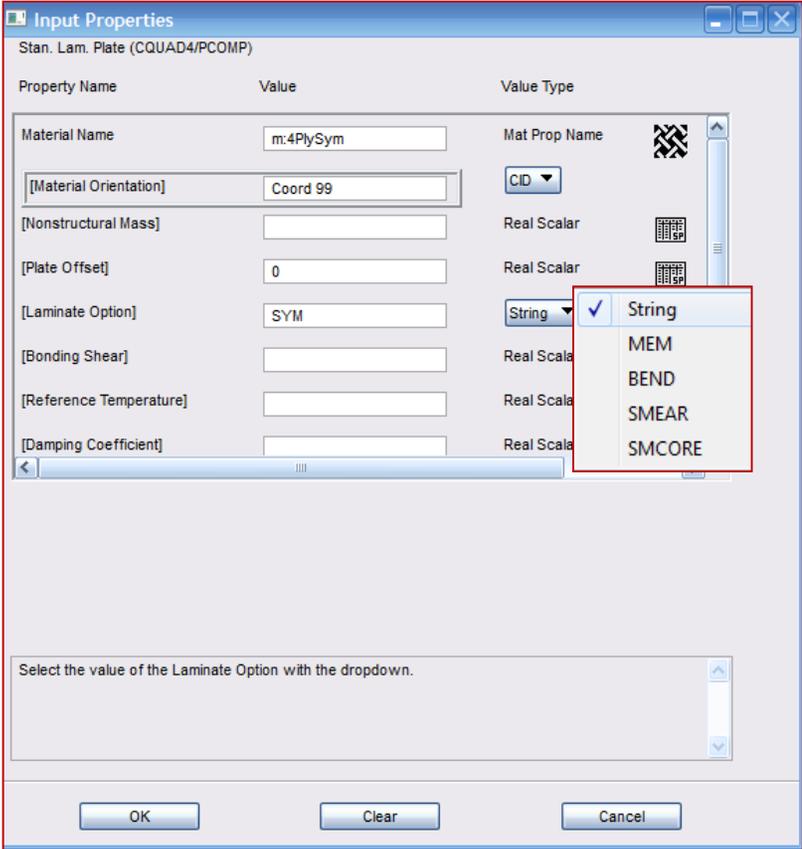
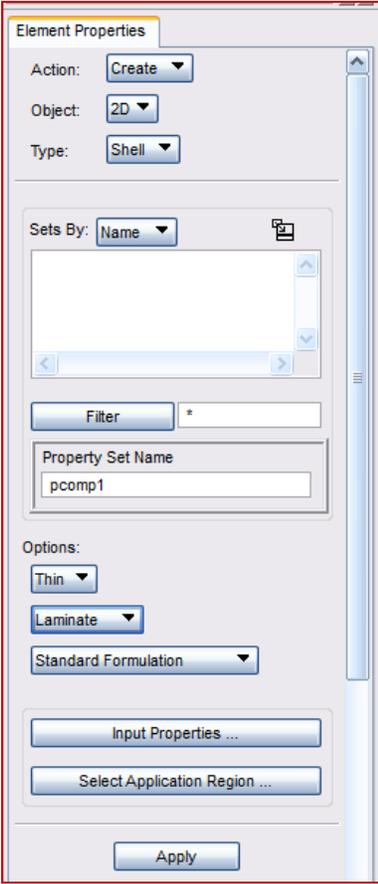
- **If Symmetry is selected, then the symmetric plies are written to the PCOMP and the SYM option on the LAM field is used.**
- **If any of:**
 - Symmetric/Mid-Ply
 - Anti-Symmetric
 - Anti-Symmetric/Mid-Ply

are selected, then the expanded symmetric layup is written to the PCOMP without using a LAM option.

PATRAN PROPERTY SYMMETRY OPTIONS



- Properties:
- Create/ 2D/ Shell
- Select Property Set Name
- Option: Laminate
- Input Properties
- Select "SYM" from Laminate Option menu
- OK
- Apply

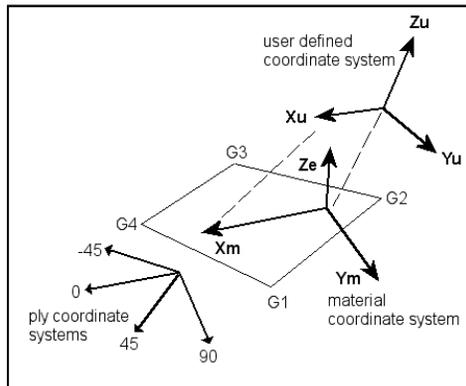


CQUAD4/CTRIA3 BULK DATA ENTRY

- Defines the composite plate in conjunction with PCOMP.

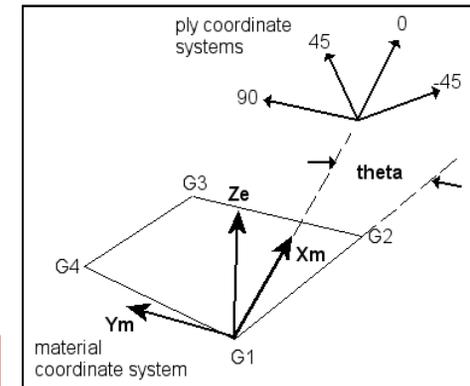
1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	

- **Material coordinate system can be defined in two ways:**
 - MCID – (integer) - ID of a user defined coordinate system whose X-axis is projected onto the element to define the X-axis of the material coordinate system.
 - THETA – (real) - an angle between the G1-G2 vector of the element and the X-axis of the material coordinate system. The positive sense of this angle is the right hand rule direction around the Z-axis of the element.



CQUAD4, 1, 1, 1, 2, 5, 4, 99

CQUAD4, 1, 1, 1, 2, 5, 4, 25.0



PATRAN COMPOSITE PROPERTIES

Properties: 2D shell
Property Set Name
Option: Laminate
Input Properties
 Click on the Mat Prop Name icon  to select the material
 Click on coord. sys. for projection to material coord. sys.
OK
Select elements
Apply

The screenshot shows the PATRAN software interface with the following components:

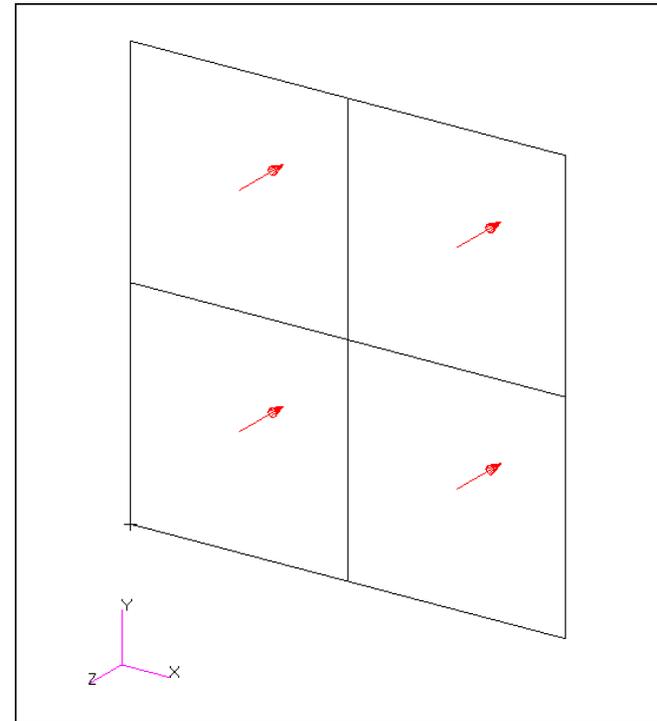
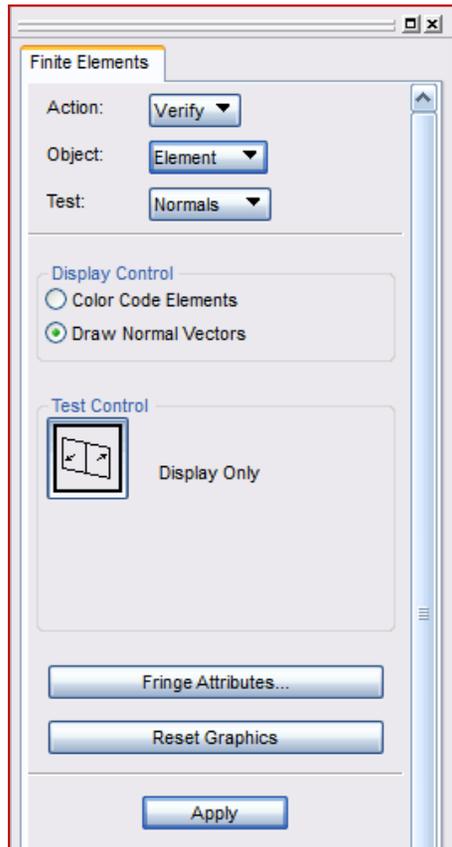
- Top Menu Bar:** Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, Results, Durability.
- Properties Toolbar:** Isotropic, Orthotropic, Anisotropic, Fluid, Cohesive, Composite, 0D Properties, 1D Properties, 2D Properties.
- Input Properties Dialog:**

Property Name	Value	Value Type
Material Name	m:8_ply_symmetric_quasi	Mat Prop Name
[Material Orientation]	Coord 99	CD
[Nonstructural Mass]		Real Scalar
[Plate Offset]		Real Scalar
[Laminate Option]		String
[Bonding Shear]		Real Scalar
[Reference Temperature]		Real
[Damping Coefficient]		Real
- Select Material Dialog:** Shows a list of existing materials, with "8_ply_symmetric_quasi graphite_epoxy_tape" selected.
- Element Properties Dialog:**
 - Action: Create
 - Object: 2D
 - Type: Shell
 - Sets By: Name
 - Property Set Name: composite1
 - Options: Thin, Laminate, Standard Formulation
- Diagram:** A 3D coordinate system (X, Y, Z) is shown with a rectangular element divided into four quadrants labeled 1, 2, 3, and 4. Nodes are numbered 1 through 9.

PATRAN MATERIAL COORD. Z-AXIS



Elements:
Verify/ Element/
Normals
Draw Normal Vectors
Apply



PATRAN MATERIAL COORD. X-AXIS

The screenshot displays the PATRAN software interface. The top menu bar includes Home, Geometry, Properties, Loads/BCs, Meshing, Analysis, Results, and Durability. The Properties panel is open, showing the 'Element Properties' section with an 'Action' dropdown set to 'Show'. Below this, the 'Select Property' section lists 'Material Orientation' as the selected property. The 'Type' is set to 'Coord Id', and the 'Display Method' is 'Vector Plot'. The 'Select Groups' section shows 'default_group' as the selected group. A green callout box on the left contains the text: 'Properties: Show', 'Material Orientation', and 'Apply'. To the right, a 3D visualization shows a rectangular element with four magenta arrows pointing upwards, each labeled '1.000'. The axes are labeled X, Y, and Z, and the text 'Xm axis' is positioned above the element.

MSC NASTRAN INPUT FILE

- Example of input with 8 layers on the PCOMP entry.

```

SOL 101
CEND
TITLE = Composite Workshop Chapter 2 - Sample Composite
Input
  SPC = 1
  LOAD = 1
  DISP = ALL
  STRESS =ALL
$
BEGIN BULK
PARAM, POST, -1
$
MAT8, 1, 2.+7, 2.+6, .35, 1.+6, 1.+6, 1.+6
,,,130000., 120000., 11000., 12000., 12500.
PCOMP, 1,,, 5000., HILL
, 1, .0054, 0., YES
, 1, .0054, 45., YES
, 1, .0054, -45., YES
, 1, .0054, 90., YES
, 1, .0054, 90., YES
, 1, .0054, -45., YES
, 1, .0054, 45., YES
, 1, .0054, 0., YES
$
CQUAD4 1 1 1 2 5 4 99
CQUAD4 2 1 2 3 6 5 99

```

```

GRID 1 0. 0. 0.
GRID 2 0. .5 0.
GRID 3 0. 1. 0.
GRID 4 .5 0. 0.
GRID 5 .5 .5 0.
GRID 6 .5 1. 0.
GRID 7 1. 0. 0.
GRID 8 1. .5 0.
GRID 9 1. 1. 0.
$
SPC1,1,1235,1
SPC1,1,135,2,3
$
FORCE 1 3 500. 0. 1. 0.
FORCE 1 6 500. 0. 1. 0.
FORCE 1 6 500. 0. 1. 0.
FORCE 1 9 500. 0. 1. 0.
FORCE 1 7 250. 1. 0. 0.
FORCE 1 8 250. 1. 0. 0.
$
CORD2R, 99,, 0., 0., 0., 0., 0., 1.
, 0., 1., 0.
ENDDATA

```

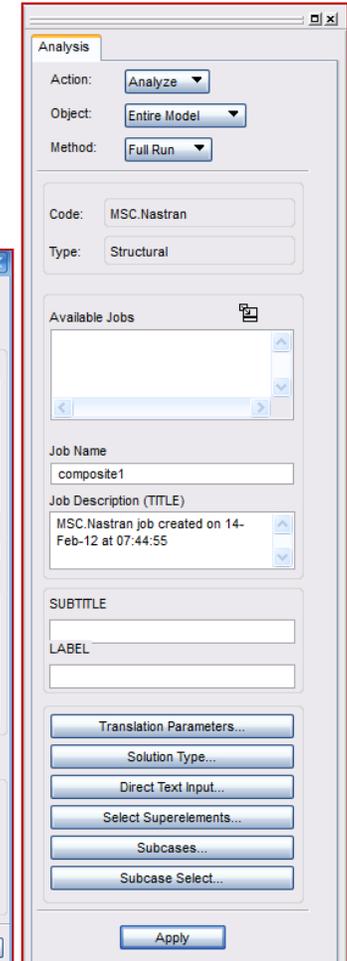
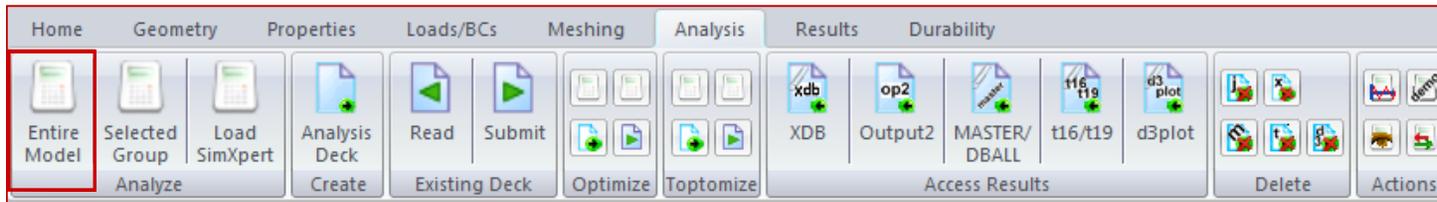
MSC NASTRAN PLY STRESS OUTPUT

- Printed in the f06 file if STRESS=ALL or STRAIN=ALL Case Control Commands are used.

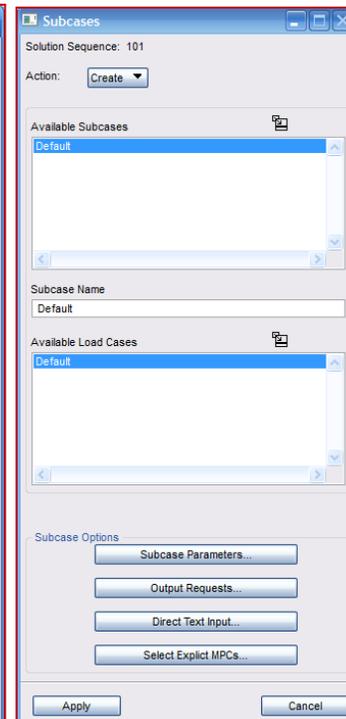
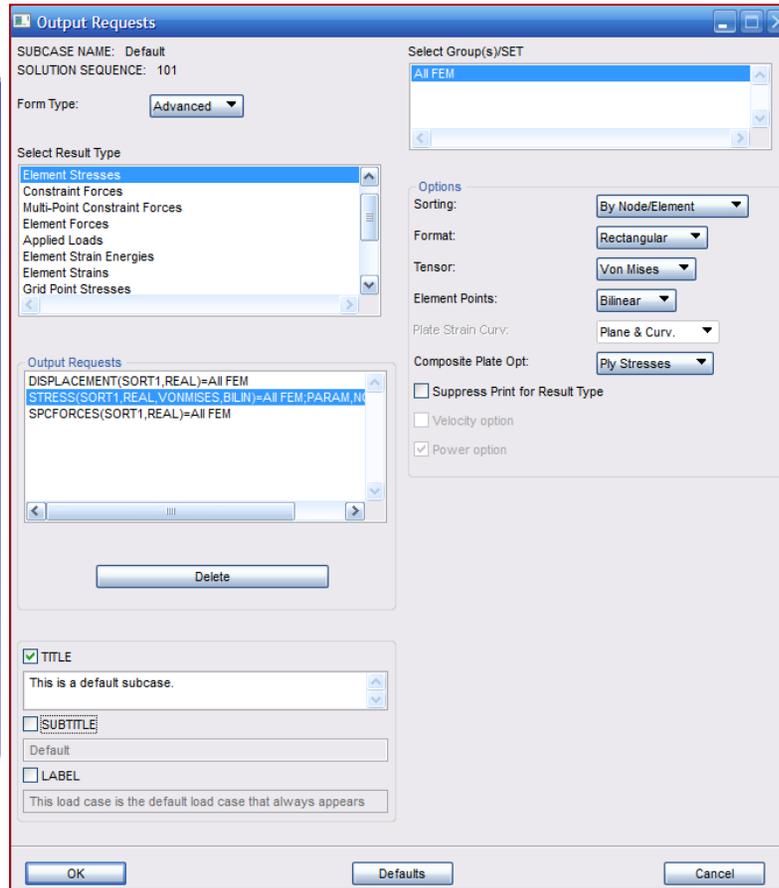
STRESSES IN LAYERED COMPOSITE ELEMENTS (QUAD4)											
ELEMENT ID	PLY ID	STRESSES IN FIBER AND MATRIX DIRECTIONS			INTER-LAMINAR STRESSES		PRINCIPAL STRESSES (ZERO SHEAR)			MAX SHEAR	
		NORMAL-1	NORMAL-2	SHEAR-12	SHEAR XZ-MAT	SHEAR YZ-MAT	ANGLE	MAJOR	MINOR		
1	1	2.55820E+05	2.81603E+04	2.73019E+04	0.0	0.0	6.74	2.59049E+05	2.49319E+04	1.17058E+05	
1	2	4.96222E+05	1.19674E+04	-2.69492E+03	0.0	0.0	-0.32	4.96237E+05	1.19524E+04	2.42142E+05	
1	3	-3.72387E+04	4.79000E+04	2.69492E+03	0.0	0.0	88.19	4.79852E+04	-3.73239E+04	4.26546E+04	
1	4	2.03163E+05	3.17071E+04	-2.73019E+04	0.0	0.0	-8.83	2.07406E+05	2.74647E+04	8.99705E+04	
1	5	2.03163E+05	3.17071E+04	-2.73019E+04	0.0	0.0	-8.83	2.07406E+05	2.74647E+04	8.99705E+04	
1	6	-3.72387E+04	4.79000E+04	2.69492E+03	0.0	0.0	88.19	4.79852E+04	-3.73239E+04	4.26546E+04	
1	7	4.96222E+05	1.19674E+04	-2.69492E+03	0.0	0.0	-0.32	4.96237E+05	1.19524E+04	2.42142E+05	
1	8	2.55820E+05	2.81603E+04	2.73019E+04	0.0	0.0	6.74	2.59049E+05	2.49319E+04	1.17058E+05	
2	1	2.20297E+05	-1.59550E+04	9.95088E+03	0.0	0.0	2.41	2.20715E+05	-1.63734E+04	1.18544E+05	
2	2	9.15727E+04	-7.28449E+03	-2.31267E+04	0.0	0.0	-12.54	9.67154E+04	-1.24272E+04	5.45713E+04	
2	3	-1.02861E+05	5.81209E+03	2.31267E+04	0.0	0.0	78.47	1.05290E+04	-1.07578E+05	5.90535E+04	
2	4	-2.31585E+05	1.44826E+04	-9.95088E+03	0.0	0.0	-87.69	1.48844E+04	-2.31987E+05	1.23436E+05	
2	5	-2.31585E+05	1.44826E+04	-9.95088E+03	0.0	0.0	-87.69	1.48844E+04	-2.31987E+05	1.23436E+05	
2	6	-1.02861E+05	5.81209E+03	2.31267E+04	0.0	0.0	78.47	1.05290E+04	-1.07578E+05	5.90535E+04	
2	7	9.15727E+04	-7.28449E+03	-2.31267E+04	0.0	0.0	-12.54	9.67154E+04	-1.24272E+04	5.45713E+04	
2	8	2.20297E+05	-1.59550E+04	9.95088E+03	0.0	0.0	2.41	2.20715E+05	-1.63734E+04	1.18544E+05	
3	1	-5.90459E+04	1.03837E+04	8.14704E+03	0.0	0.0	83.40	1.13269E+04	-5.99891E+04	3.56580E+04	
3	2	1.11984E+05	-1.13646E+03	9.35916E+03	0.0	0.0	4.70	1.12753E+05	-1.90558E+03	5.73294E+04	
3	3	-4.72039E+04	9.58604E+03	-9.35916E+03	0.0	0.0	-80.88	1.10887E+04	-4.87066E+04	2.98976E+04	
3	4	1.23826E+05	-1.93411E+03	-8.14704E+03	0.0	0.0	-3.69	1.24352E+05	-2.45970E+03	6.34056E+04	
3	5	1.23826E+05	-1.93411E+03	-8.14704E+03	0.0	0.0	-3.69	1.24352E+05	-2.45970E+03	6.34056E+04	
3	6	-4.72039E+04	9.58604E+03	-9.35916E+03	0.0	0.0	-80.88	1.10887E+04	-4.87066E+04	2.98976E+04	
3	7	1.11984E+05	-1.13646E+03	9.35916E+03	0.0	0.0	4.70	1.12753E+05	-1.90558E+03	5.73294E+04	
3	8	-5.90459E+04	1.03837E+04	8.14704E+03	0.0	0.0	83.40	1.13269E+04	-5.99891E+04	3.56580E+04	
4	1	8.79761E+04	9.55942E+01	1.42040E+04	0.0	0.0	8.96	9.02149E+04	-2.14316E+03	4.61790E+04	
4	2	1.69212E+05	-5.37626E+03	-5.88892E+03	0.0	0.0	-1.93	1.69411E+05	-5.57467E+03	8.74926E+04	

.f06 file extract

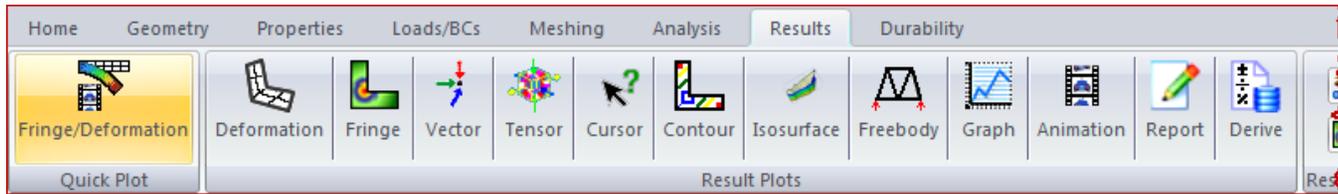
PATRAN PLY OUTPUT REQUEST



Analysis:
 Analyze/ Entire Model/ Full Run
 Translation Parameters/ XDB as Data Output
 Subcases/ Create
 Output Requests/ Advanced/ Element Stress
 Ply Stresses
 OK
 Apply



PATRAN PLY STRESS RESULTS



- Analysis: Access Results
- Read XDB
- Results: Create/Quick Plot
- Result/ Stress Tensor
- Position/ Layer 1
- Quantity/ X Component
- Select Deformation
- Result: Displacements, Translational
- Apply

Results

Action: Create

Object: Quick Plot

Select Result Cases

Default, A1.Static Subcase;-THIS IS A...

Select Fringe Result

Stress Invariants, Major Principal
Stress Invariants, Maximum Shear
Stress Invariants, Minor Principal
Stress Tensor

Position...(Layer 6)

Quantity: X Component

Select Deformation Result

Constraint Forces, Rotational
Constraint Forces, Translational
Displacements, Rotational
Displacements, Translational

Animate

Apply

- von Mises
- X Component
- Y Component
- Z Component
- XY Component
- YZ Component
- ZX Component
- Max Principal
- Mid Principal
- Min Principal
- Hydrostatic
- 1st Invariant
- 2nd Invariant
- 3rd Invariant
- Tresca
- Max Shear
- Octahedral
- Max Principal 2D
- Min Principal 2D
- Tresca 2D
- Max Shear 2D

Results Select...

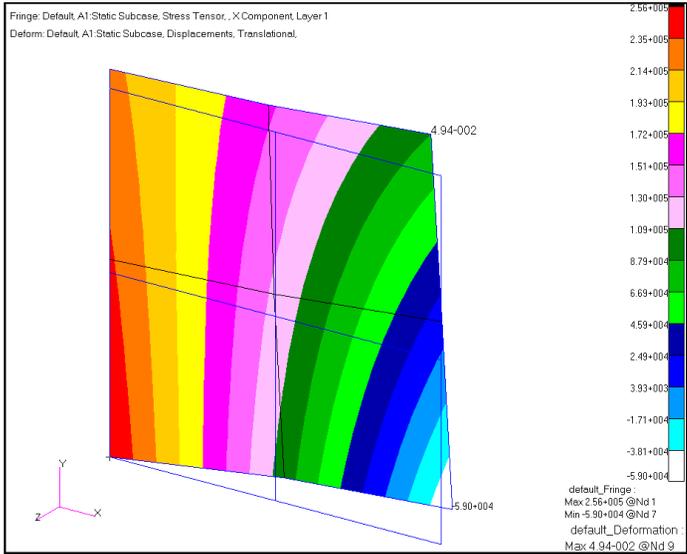
Positions

- Layer 1
- Layer 2
- Layer 3
- Layer 4
- Layer 5
- Layer 6
- Layer 7
- Layer 8

Filter

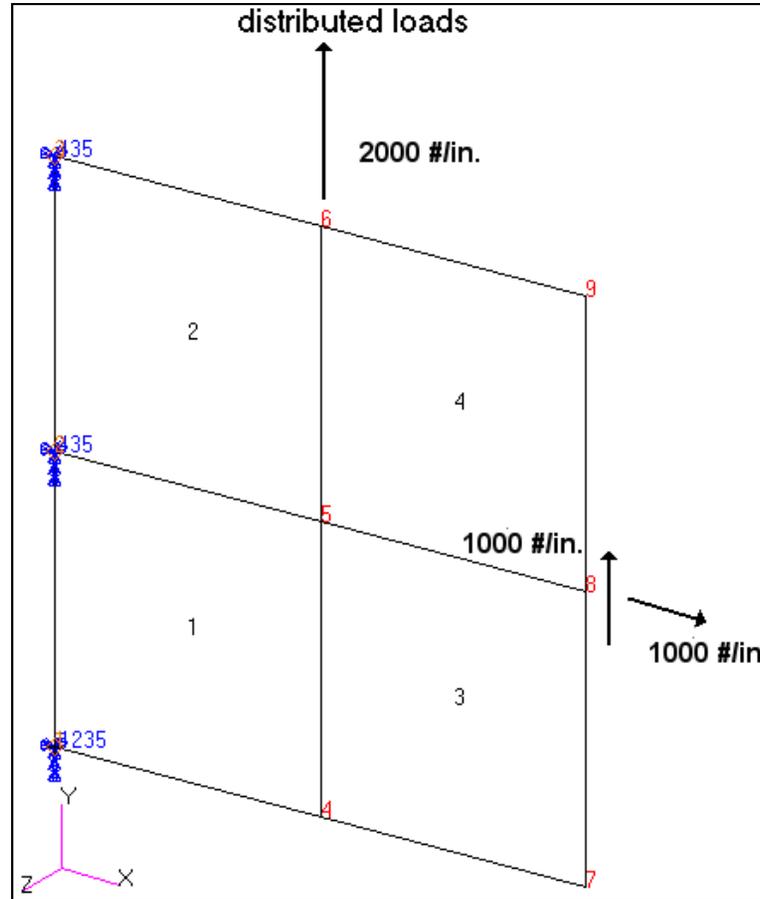
Option: Maximum

Close



EXERCISE

- Perform Workshop 1 “Making a Composite Model”



ASSUMPTIONS

- **Perfect bonding between plies**
 - Zero thickness
 - No slip

- **Plane stress for composite**
 - $\sigma_z=0$ throughout composite
 - $\tau_{xz}=\tau_{yz}=0$ at the top and bottom surface

- **Linear strain distribution through thickness**

PLY MATERIAL CONSTANTS

- 2D orthotropic in ply coordinate system (1, 2, 3)
- Stress/strain relations in principal ply material directions:

$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{Bmatrix} = \begin{bmatrix} \frac{E_1}{1-\nu_{12}\nu_{21}} & \frac{E_1\nu_{21}}{1-\nu_{12}\nu_{21}} & 0 \\ \frac{E_2\nu_{12}}{1-\nu_{12}\nu_{21}} & \frac{E_2}{1-\nu_{12}\nu_{21}} & 0 \\ 0 & 0 & G_{12} \end{bmatrix} \begin{Bmatrix} \epsilon_1 \\ \epsilon_2 \\ \gamma_{12} \end{Bmatrix}$$

- **Four independent constants:**
 - E_1 , E_2 , ν_{12} (or ν_{21}), and G_{12}
 - where the relationship between E_1 , E_2 , ν_{12} and ν_{21} is:

$$\frac{\nu_{12}}{E_1} = \frac{\nu_{21}}{E_2}$$

PLY MATERIAL CONSTANTS

- or:
$$\begin{Bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_6 \end{Bmatrix} = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix} \begin{Bmatrix} \epsilon_1 \\ \epsilon_2 \\ \epsilon_6 \end{Bmatrix}$$

– and:

$$\{\sigma\} = [Q]\{\epsilon\}$$

- The ply (1, 2, 3) system is rotated from the material coordinate system (X_m, Y_m, Z_m) by the ply angle θ
- The positive sense of θ is according to the right hand rule around the Z-axis of the element coordinate system (Z_e)

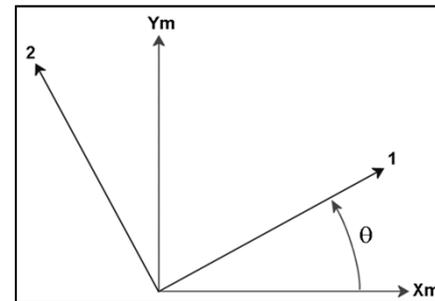
ROTATION TO MATERIAL COORDINATE SYSTEM

- Rotate the ply properties $[Q]$ from the ply coordinate system (1, 2, 3) to the material coordinate system (X_m, Y_m, Z_m) to make $[\bar{Q}]$

- Using the equation: $[\bar{Q}] = [U]^T [Q] [U]$

where: $[U] = \begin{bmatrix} \cos^2\theta & \sin^2\theta & 2\sin\theta\cos\theta \\ \sin^2\theta & \cos^2\theta & -2\sin\theta\cos\theta \\ -\sin\theta\cos\theta & \sin\theta\cos\theta & \cos^2\theta - \sin^2\theta \end{bmatrix}$

- Positive sense of θ shown in figure:



COMPOSITE PROPERTIES

- For each lay up of plies, the $[\bar{Q}]$ of each ply is integrated to create the symmetric composite property matrices A, B, and D in the following form:

$$\begin{Bmatrix} F_x \\ F_y \\ F_s \\ M_x \\ M_y \\ M_t \end{Bmatrix} = \begin{bmatrix} A_{xx} & A_{xy} & A_{xs} & B_{xx} & B_{xy} & B_{xt} \\ A_{yx} & A_{yy} & A_{ys} & B_{yx} & B_{yy} & B_{yt} \\ A_{sx} & A_{sy} & A_{ss} & B_{sx} & B_{sy} & B_{st} \\ B_{xx} & B_{xy} & B_{xs} & D_{xx} & D_{xy} & D_{xt} \\ B_{yx} & B_{yy} & B_{ys} & D_{yx} & D_{yy} & D_{yt} \\ B_{tx} & B_{ty} & B_{ts} & D_{tx} & D_{ty} & D_{tt} \end{bmatrix} \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \\ \chi_x \\ \chi_y \\ \chi_{xy} \end{Bmatrix}$$

where:

F_i = membrane forces

s = shear

M_i = bending moments

t = twist

ϵ_i = membrane strain

xy = shear

χ_i = bending curvature

xy = twist

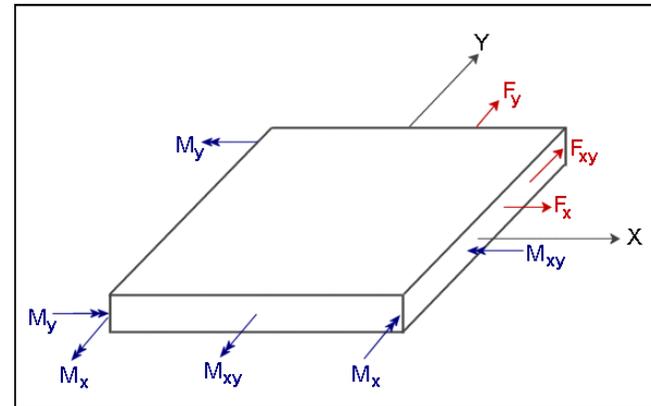
COMPOSITE PROPERTIES

- Positive sense for lamination theory forces and moments:
- A, B, and D are calculated as follows:

$$A_{ij} = \sum_{k=1}^n (\bar{Q}_{ij})_k t_k$$

$$B_{ij} = \sum_{k=1}^n (\bar{Q}_{ij})_k t_k \bar{z}_k$$

$$D_{ij} = \sum_{k=1}^n (\bar{Q}_{ij})_k \left(t_k \bar{z}_k^2 + \frac{t_k^3}{12} \right)$$



where:

n = number of plies

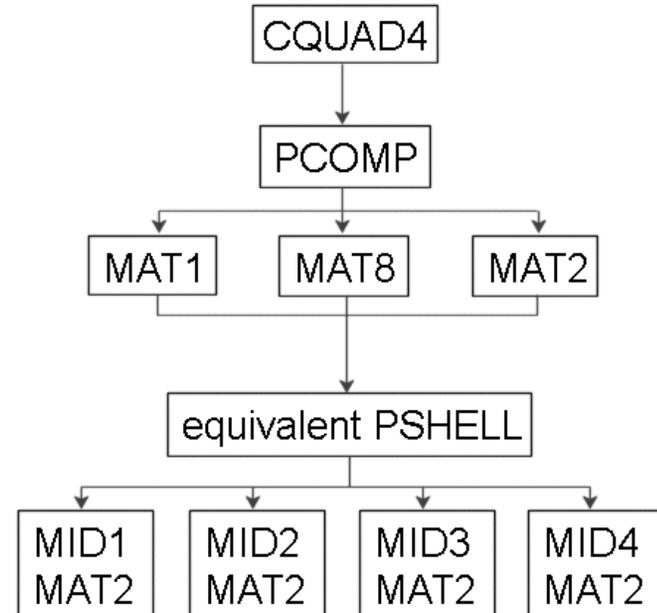
k = ply number

t_k = ply k thickness

\bar{z}_k = distance from composite center to ply k center

MSC NASTRAN COMPOSITE THEORY

- **PCOMP is a composite preprocessor**
 - Generates equivalent PSHELL/MAT2 that are used internally
- **PSHELL/MAT2 are viewed with:**
 - To f06:
 - NASTRAN SYSTEM (361)=1 or
 - NASTRAN PRTPCOMP=1
 - ECHO=PUNCH writes to the .pch file



MSC NASTRAN COMPOSITE THEORY

```
*** USER INFORMATION MESSAGE 4379 (IFP6CD)
    THE USER SUPPLIED PCOMP BULK DATA CARDS ARE REPLACED BY THE FOLLOWING PSHELL AND MAT2 CARDS.
    WARNING, MAT2 RECORDS WITH MID GREATER THAN 400000000 USE A SPECIAL FORMAT FOR PCOMPS.
    REFER TO REMARK 13 OF THE MAT2 DESCRIPTION IN THE MSC.NASTRAN QUICK REFERENCE GUIDE.
PSHELL      1      100000001  3.2400E-02  200000001  1.0000E+00  300000001  1.0000E+00  0.0000E+00
            -1.6200E-02  1.6200E-02  400000001
MAT2        100000001  9.7318E+06  2.1133E+06  6.4575E-11  9.7318E+06  1.6562E-09  2.4046E+06  0.0000E+00
            0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
            0
MAT2        200000001  1.5574E+07  1.3330E+06  5.0620E+05  5.4501E+06  5.0620E+05  1.6243E+06  0.0000E+00
            0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
            0
MAT2        300000001  8.8986E+05  0.0000E+00  0.0000E+00  6.7546E+05  0.0000E+00  0.0000E+00  0.0000E+00
            0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
            0
MAT2        400000001  2.7210E+05  2.3410E+05  1.2655E+05  -7.4030E+05  1.2655E+05  2.3410E+05  0.0000E+00
            0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00  0.0000E+00
            0
```

.f06 file excerpt

EQUIVALENT PSHELL/MAT2

- Generated MID1 numbering is PID+100,000,000
- Generated MID2 numbering is PID+200,000,000 and so on.

1	2	3	4	5	6	7	8	9	10
PSHELL	PID	MID1	T	MID2	12/T**3	MID3	TS/T	NSM	
PSHELL	1	100000001	3.2400E-02	200000001	1.0000E+00	300000001	1.0000E+00	0.0000E+00	
	Z1	Z2	MID4						
	0.0000E+00	0.0000E+00	400000001						

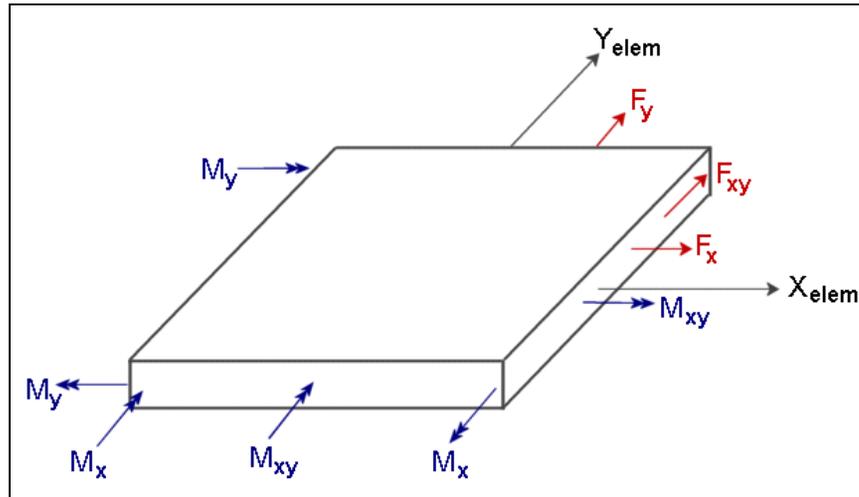
1	2	3	4	5	6	7	8	9	10
MAT2	MID	G11	G12	G13	G22	G23	G33	RHO	
MAT2	100000001	9.7318E+06	2.1133E+06	6.4575E-11	9.7318E+06	1.6562E-09	2.4046E+06	0.0000E+00	
	A1	A2	A3	TREF	GE	ST	SC	SS	
	0.0000E+00								
	MCSID								
	0								

- Upper triangle only, since it is symmetric.

$$\begin{bmatrix} G_{11} & G_{12} & G_{13} \\ & G_{22} & G_{23} \\ [sym] & & G_{33} \end{bmatrix}$$

MSC NASTRAN LAMINATION MATRICES

- Properties of MID1 are proportional to the A matrix
- Properties of MID2 are proportional to the D matrix
- Properties of MID4 are proportional to the B matrix
- Positive sense for MSC Nastran element forces and moments:



$$[MID1] = \frac{[A]}{T}$$

$$[MID2] = \frac{12}{T^3} [D]$$

$$[MID4] = \frac{[B]}{-T^2}$$

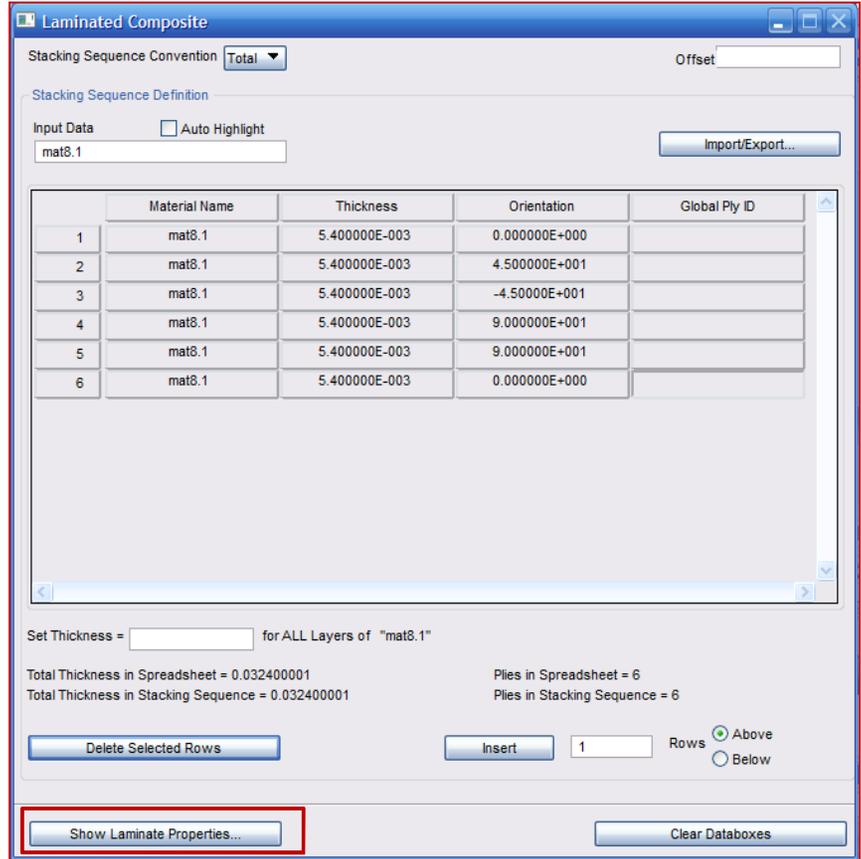
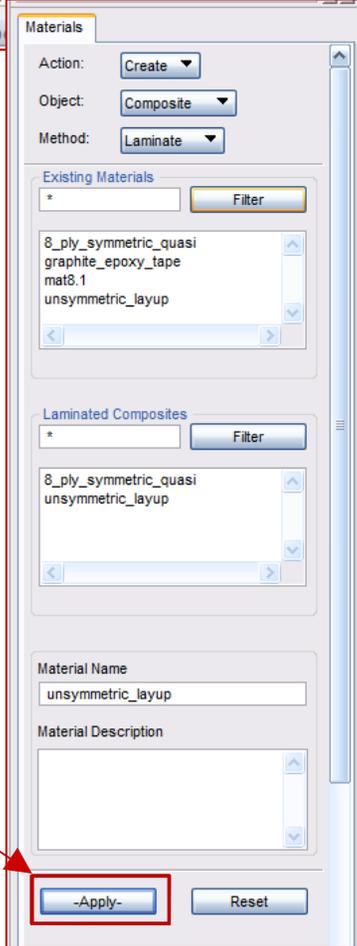
where:

T = laminate thickness

PATRAN LAMINATION THEORY



Materials:
 Create/ Composite
 Select composite material
 in Laminated Composite
 section
 Show Laminate Properties
 A,B, and D Matrices
 E's, ν 's, G's and Q_{ij} 's
 Note: If creating a new
 material, click Apply first



PATRAN LAMINATION THEORY



Composite Material Properties

Membrane, Bending, and Coupling Matrices

	Membrane			Bending		
	2.92E+005	1.22E+005	-1.6E-004	-1.8E+003	1.53E-005	-3.0E-006
Membrane	1.22E+005	4.88E+005	-1.2E-002	1.53E-005	1.86E+003	-7.8E-005
	-1.6E-004	-1.2E-002	1.34E+005	-3.0E-006	-7.8E-005	-2.2E-005
	-1.8E+003	1.53E-005	-3.0E-006	6.24E+001	1.63E+001	5.74E+000
Bending	1.53E-005	1.86E+003	-7.8E-005	1.63E+001	6.43E+001	5.74E+000
	-3.0E-006	-7.8E-005	-2.2E-005	5.74E+000	5.74E+000	1.82E+001

High Precision Value

Composite Property Display Options

A, B, and D Matrices 3D Flexibility Matrix Thermal: Kij, Ni, and Mi

3D Elasticity Matrix E's, NU's, G's, and Qij's CTE's, CME's and Others

Cancel

Composite Material Properties

Engineering Constants and 2D Elasticity Matrix

	E11,22,33	NU12,23,13	G12,23,31	Q	
	6.05E+006	2.49E-001	3.11E+006	6.75E+006	2.82E+006
	1.01E+007	0.00E+000	0.00E+000	2.82E+006	1.13E+007
	0.00E+000	0.00E+000	0.00E+000	-3.7E-003	-2.8E-001
				3.11E+006	

High Precision Value

Composite Property Display Options

A, B, and D Matrices 3D Flexibility Matrix Thermal: Kij, Ni, and Mi

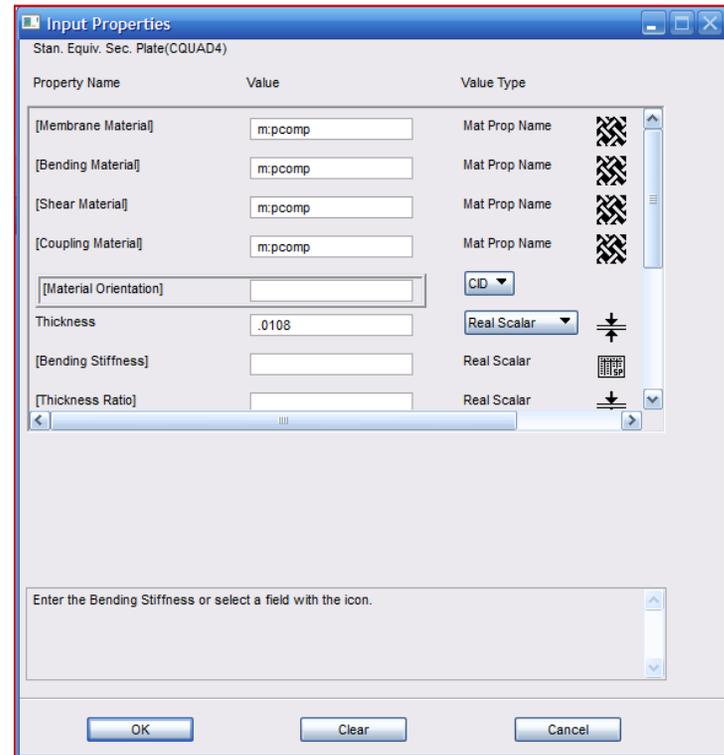
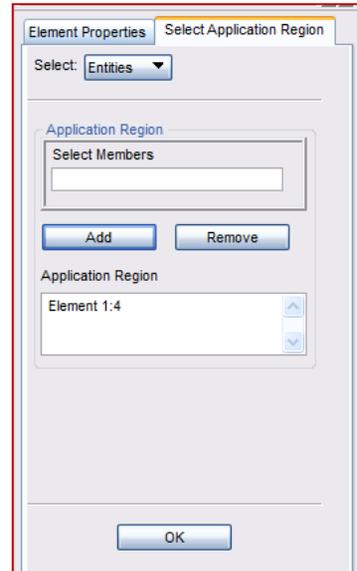
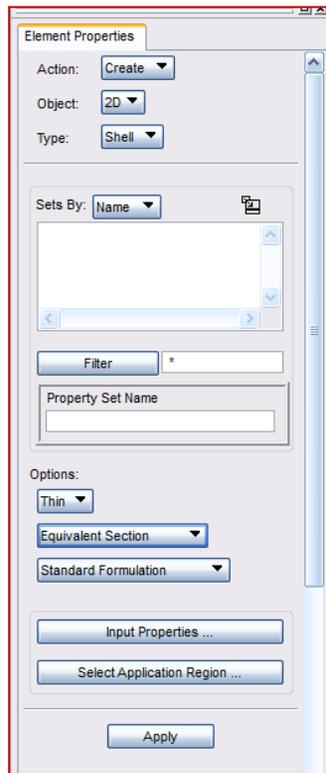
3D Elasticity Matrix E's, NU's, G's, and Qij's CTE's, CME's and Others

Cancel

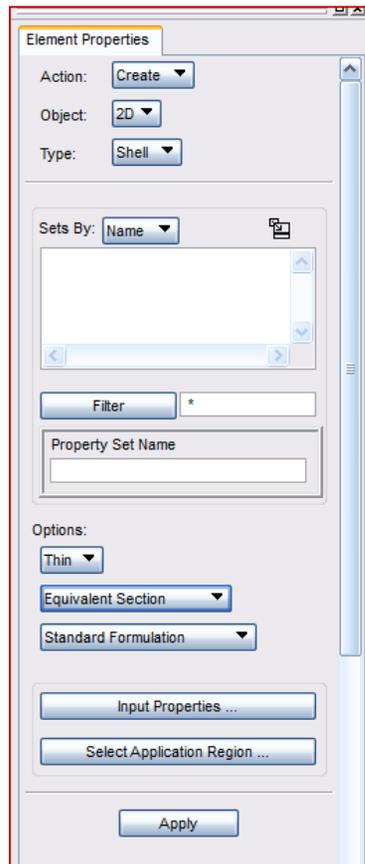
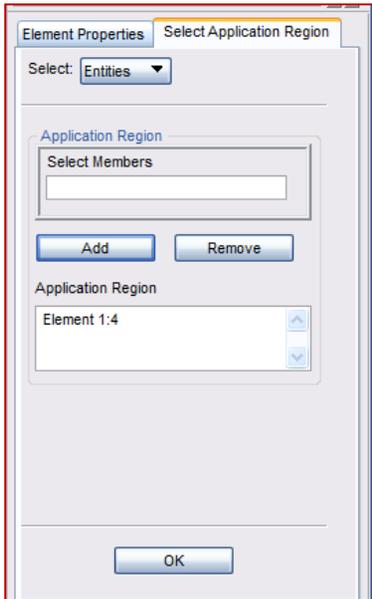
PATRAN COMPOSITE PSHELL



Properties:
Create/ 2D/Shell
Equivalent
Section
Input Properties:
Pick the
composite
property for
Membrane,
Bending, Shear
and Coupling



PATRAN COMPOSITE PSHELL



```

PSHELL 2 2 .0432 3 4
5
$ Bending material properties
MAT2 3 1.454+7 2.420+6 854214. 4.299+6 854214. 2.711+6 0.
0. 0. 0. 0.
$ Shear material properties
MAT2 4 97.7525 16.2625 28.8844 0.
0. 0. 0. 0.
$ Coupling material properties
MAT2 5 0. -.01226 0. -.01635 0. -.004088 0.
0. 0. 0. 0.
$ Membrane material properties
MAT2 2 9.029+6 2.815+6 9.029+6 3.106+6 0.
0. 0. 0. 0.
    
```

SYMMETRIC LAY-UPS

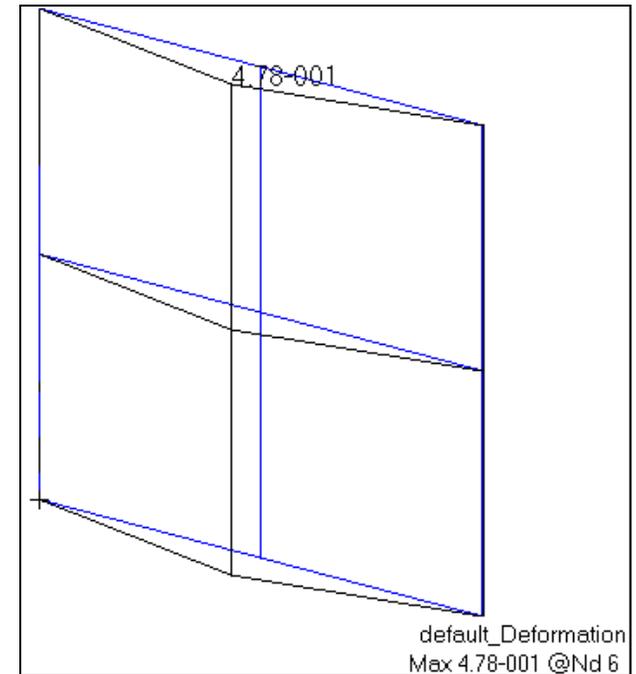
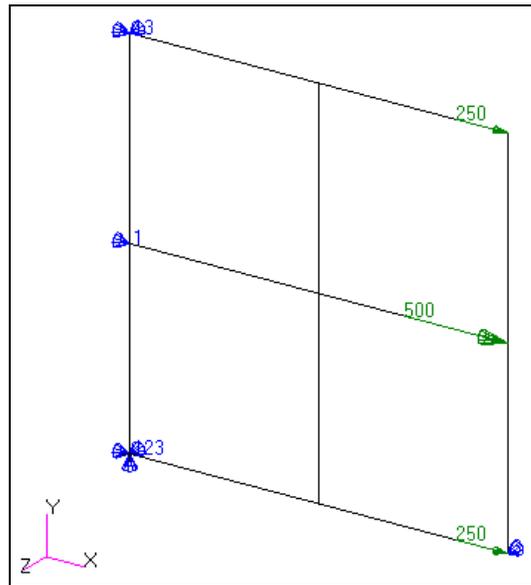
- In a symmetric lay-up, the [B] matrix has zero terms, so an equivalent material for MID4 is not created:

PSHELL	1	100000001	4.3200E-02	200000001	1.0000E+00	300000001	1.0000E+00	0.0000E+00
	-2.1600E-02	2.1600E-02	0					
MAT2	100000001	9.0295E+06	2.8156E+06	4.8431E-11	9.0295E+06	1.2421E-09	3.1069E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	200000001	1.4550E+07	2.4206E+06	8.5421E+05	4.2993E+06	8.5421E+05	2.7119E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	300000001	8.9476E+05	0.0000E+00	0.0000E+00	7.1010E+05	0.0000E+00	0.0000E+00	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

NONSYMMETRIC LAY-UPS

- A 0/90 lay-up is nonsymmetric

2	1	0.0054	90.
1	1	0.0054	0.0
ply #	mat id	thickness	angle

NONSYMMETRIC LAY-UPS

- In an nonsymmetric lay-up, the [B] matrix has non-zero terms, so an equivalent MID4 material is created:

PSHELL	1	100000001	1.0800E-02	200000001	1.0000E+00	300000001	1.0000E+00	0.0000E+00
	-5.4000E-03	5.4000E-03	400000001					
MAT2	100000001	1.1136E+07	7.0868E+05	9.6863E-11	1.1136E+07	2.4842E-09	1.0000E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	200000001	1.1136E+07	7.0868E+05	9.6863E-11	1.1136E+07	2.4842E-09	1.0000E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	300000001	7.4875E+05	0.0000E+00	0.0000E+00	7.4875E+05	0.0000E+00	0.0000E+00	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	400000001	2.2779E+06	0.0000E+00	-2.4216E-11	-2.2779E+06	-6.2106E-10	0.0000E+00	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							

BALANCED LAY-UPS

- In a balanced lay-up, if there is an angled ply, there is also a ply at the negative of that angle.
- This (0/45/-45/90) sym is balanced because, for every +45 degree ply, there is a -45 degree ply.

The diagram illustrates a balanced lay-up cross-section. A vertical arrow labeled Z_e points upwards from the centerline. The centerline is indicated by a dashed line. The lay-up consists of eight plies, numbered 1 to 8 from bottom to top. The properties of each ply are listed in the table below:

ply #	mat id	thickness	angle
8	1	0.0054	0.0
7	1	0.0054	45.
6	1	0.0054	-45.
5	1	0.0054	90.
4	1	0.0054	90.
3	1	0.0054	-45.
2	1	0.0054	45.
1	1	0.0054	0.0

BALANCED LAY-UPS

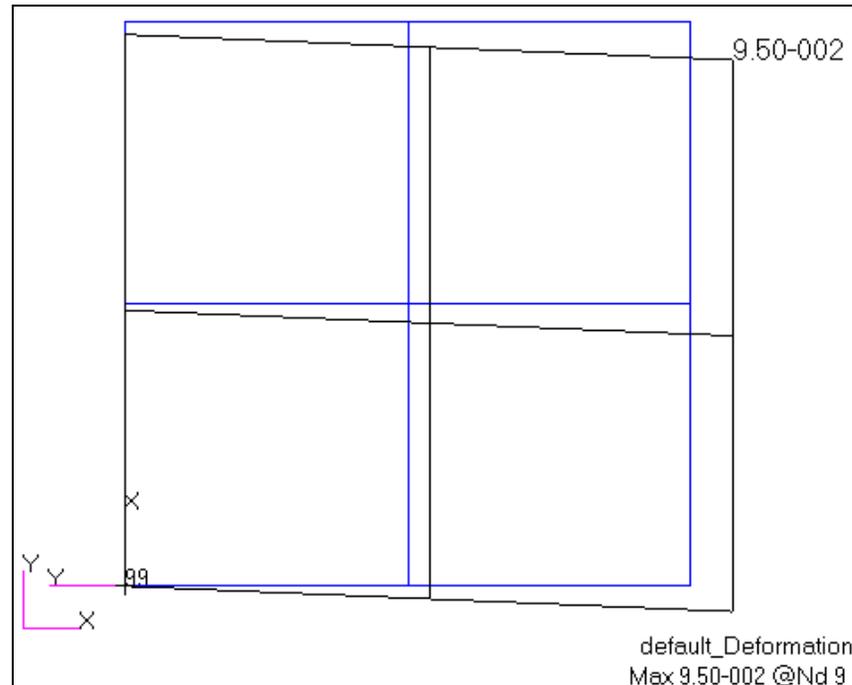
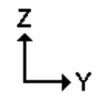
- In a balanced lay-up, the stretching/shearing coupling terms in the [A] matrix are zero:

PSHELL	1	100000001	4.3200E-02	200000001	1.0000E+00	300000001	1.0000E+00	0.0000E+00
	-2.1600E-02	2.1600E-02	0					
MAT2	100000001	9.0295E+06	2.8156E+06	4.8431E-11	9.0295E+06	1.2421E-09	3.1069E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	200000001	1.4550E+07	2.4206E+06	8.5421E+05	4.2993E+06	8.5421E+05	2.7119E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	300000001	8.9476E+05	0.0000E+00	0.0000E+00	7.1010E+05	0.0000E+00	0.0000E+00	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

UNBALANCED LAY-UPS

- A (45) lay-up is unbalanced because there are no -45 degree plies.

1	1	0.0054	45.
ply #	mat id	thickness	angle



UNBALANCED LAY-UPS

- In an unbalanced lay-up, the A_{13} and A_{23} terms are non-zero.

PSHELL	1	100000001	5.4000E-03	200000001	1.0000E+00	300000001	1.0000E+00	0.0000E+00
	-2.7000E-03	2.7000E-03	400000001					
MAT2	100000001	6.9226E+06	4.9226E+06	4.5558E+06	6.9226E+06	4.5558E+06	5.2139E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	200000001	6.9226E+06	4.9226E+06	4.5558E+06	6.9226E+06	4.5558E+06	5.2139E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	300000001	8.3333E+05	0.0000E+00	0.0000E+00	8.3333E+05	0.0000E+00	0.0000E+00	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	400000001	0.0000E+00						
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							

QUASI-ISOTROPIC LAY-UPS

- Quasi-isotropic lay-ups have an equal number of plies in the 0, 45, -45, and 90 degree directions.
- This $(0/45/-45/90)_{\text{sym}}$ laminate is quasi-isotropic, because 25% of the plies are in each of the 0, 45, -45, and 90 degree directions.

ply#	mat id	thickness	angle
8	1	0.0054	0.0
7	1	0.0054	45.
6	1	0.0054	-45.
5	1	0.0054	90.
4	1	0.0054	90.
3	1	0.0054	-45.
2	1	0.0054	45.
1	1	0.0054	0.0

centerline

QUASI-ISOTROPIC LAY-UPS

- In a quasi-isotropic lay-up, the 11 and 22 terms in the [A] matrix are equal:

PSHELL	1	100000001	4.3200E-02	200000001	1.0000E+00	300000001	1.0000E+00	0.0000E+00
	-2.1600E-02	2.1600E-02	0					
MAT2	100000001	9.0295E+06	2.8156E+06	4.8431E-11	9.0295E+06	1.2421E-09	3.1069E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	200000001	1.4550E+07	2.4206E+06	8.5421E+05	4.2993E+06	8.5421E+05	2.7119E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	300000001	8.9476E+05	0.0000E+00	0.0000E+00	7.1010E+05	0.0000E+00	0.0000E+00	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

EFFECTIVE MODULI

- For symmetric lay-ups:

$$E_x = (A_{xx}A_{yy} - A_{xy}^2) / (A_{yy}T)$$

$$E_y = (A_{xx}A_{yy} - A_{xy}^2) / (A_{xx}T)$$

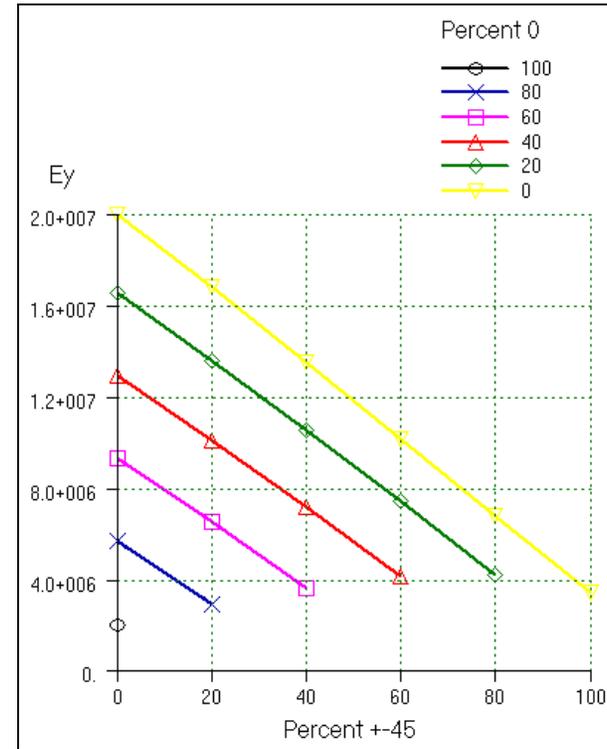
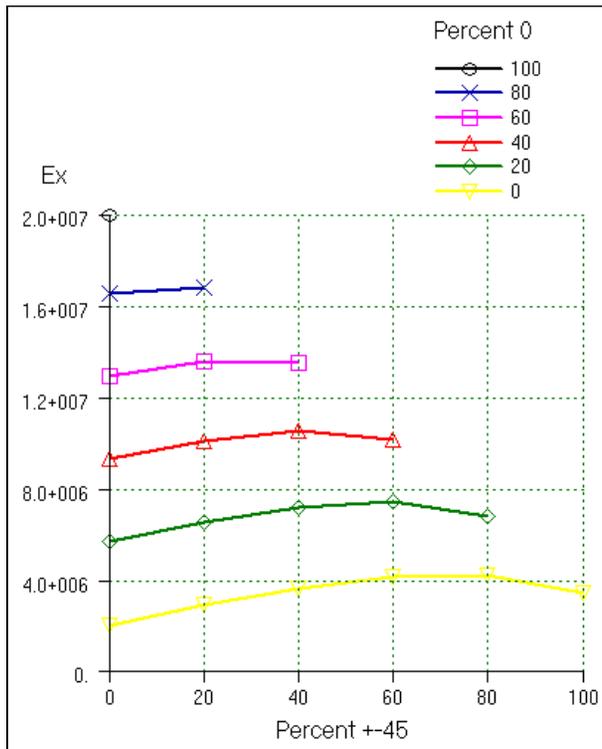
$$G_{xy} = A_{ss} / T$$

$$\nu_{xy} = A_{xy} / A_{yy}$$

where: T = total thickness

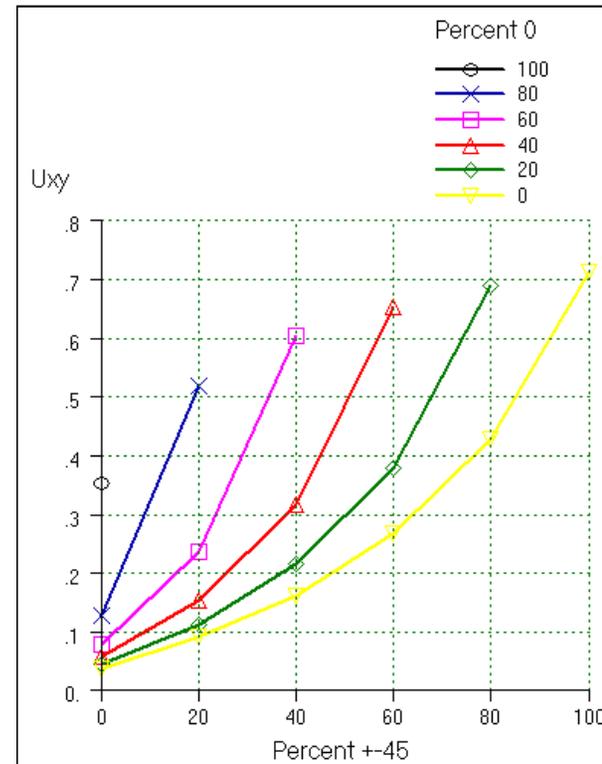
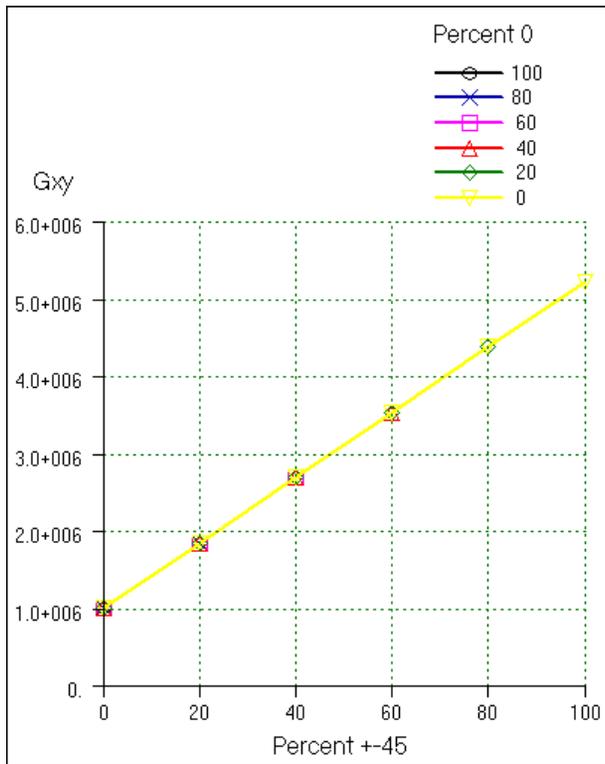
ELASTIC PROPERTIES PLOTS

- Show the effect of layup changes on E_x , E_y , G_{xy} , U_{xy}
- Below are E_x and E_y variation with %0, % ± 45 degree plies



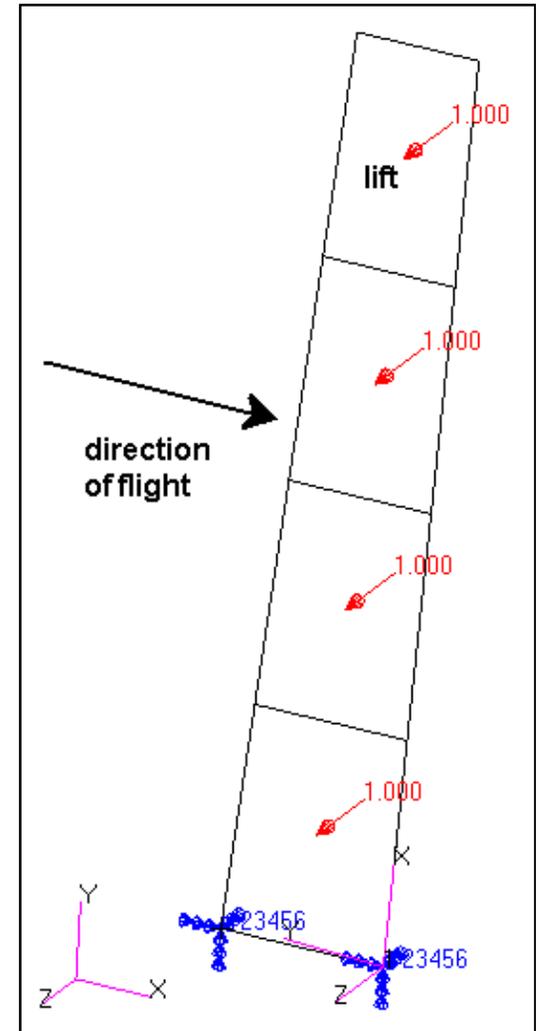
ELASTIC PROPERTIES PLOTS

- Below are G_{xy} and U_{xy} variation with %0, %+-45 degree plies



EXERCISE

- Perform Workshop 2 “Forward Swept Wing”.



SECTION 3

ADVANCED COMPOSITE ELEMENTS

Part 1: COMPOSITE BEAM

TYPE OF COMPOSITE ELEMENTS

- **Shell elements were discussed in the previous section**
 - Most popular and most commonly used, for composite structures
- **Additional advanced composite elements are available:**
 - Beam elements
 - Solid elements
- **The Composite Beam is discussed in this section**
- **Solid Composite and Solid Shells will be discussed in the next section**

WHEN CAN I USE A COMPOSITE BEAM ELEMENT?

- **Similar to other FEA analyses, if one of the dimensions of the component is much larger than the other 2 dimensions, the composite beam element may be a candidate for this type of analysis.**
- **Possible Applications:**
 - Stiffeners—plane and car
 - Sail boat keel beam
 - Composite rotor blade
 - Wind turbine blade
 - etc.
- **The above examples can also be modeled using shells or solid elements, but will require many more elements to represent the same behavior.**

COMPOSITE BEAM USING VAM

- **Based on VAM (Variational Asymptotic Method) formulation**
 - Theoretical background – originated from work done by Prof. Carlos Cesnik from the University of Michigan
 - Analysis of slender composite structures
 - Dimensional reduction of general non-homogeneous anisotropic beam of arbitrary cross-sections
- **Assumption – small local strains**
- **Can also be used for modeling isotropic beam**
- **Done using the existing arbitrary PBMSECT (Property Beam cross SECTION) entry**
 - PBRSECT (similar, for CBARs) does not support referencing a PCOMP for laminate composite layups

COMPOSITE BEAM USING VAM (CONT.)

- **Steps for defining the cross section**

- Uses a 3 point CBEAM3 element

- Similar to CBEAM, except for an additional mid point
- Mid point must lie on a straight line connecting the end points
 - In Patran, mesh curves with BAR2 elements, and convert to BAR3

- Define the cross-sectional profile

- 3 different options are available for defining the cross section

- OP -- Open Profile

- » Using a series of points specified on the SET1 entry

- GS -- General Section

- » The cross section is usually defined by using a separate bulk data section

- CP -- Closed Profile

- » This is good for closed “tube-like” cross sections

- Must use the Timoshenko option for the arbitrary beam solution algorithm:
(PARAM,ARBMSTYP,TIMOSHEN)

- While for MSC Nastran 2013, Timoshenko is the default method for VAM (formerly this was VKI), it is still required when using a Core or Layer (which the next examples will be doing).

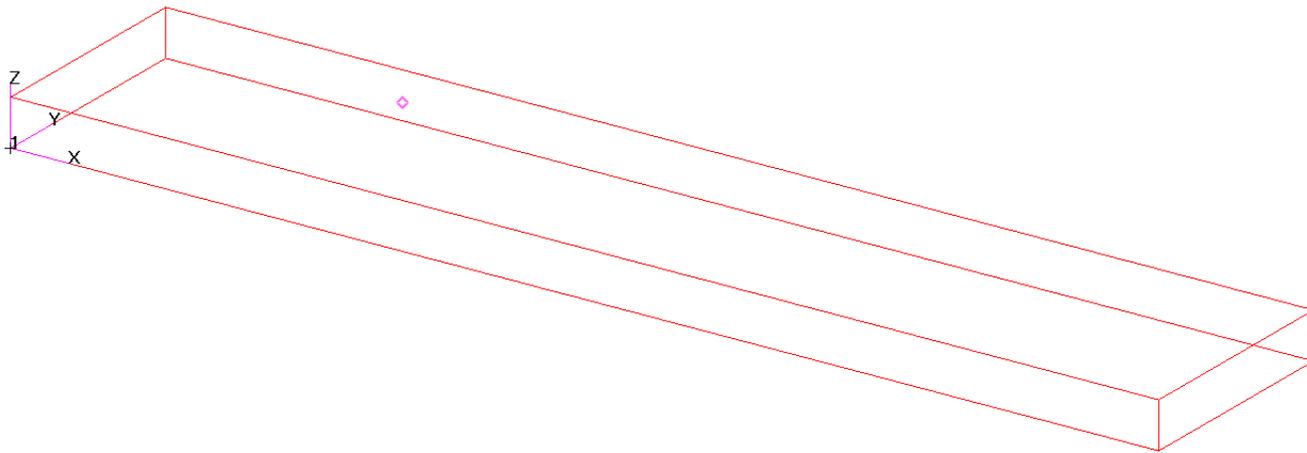
COMPOSITE BEAM USING VAM (CONT.)

- **Steps for defining the cross section (cont.)**

- Define the cross sectional property using the “core=” option – which points to the appropriate PCOMP entry
 - In its simplest form, similar to the PCOMP definition for a shell element
 - Optionally, can also define layers at the top and bottom, but easiest to define with just the core option (and have the referenced PCOMP contain all layers)
 - **New Keywords**
 - Core(id)=[PCID,PT=(pid1,pid2)]; specifies the CORE part of composite lay-up.
 - » PCID – the ID of PCOMP/PCOMPG (similar to the shell elements)
 - » PT=(pid1,pid2) – the starting point ID (pid1), and ending point ID (pid2) of the line segment
 - » Core can also be shortened as C
- Best illustrated by an example

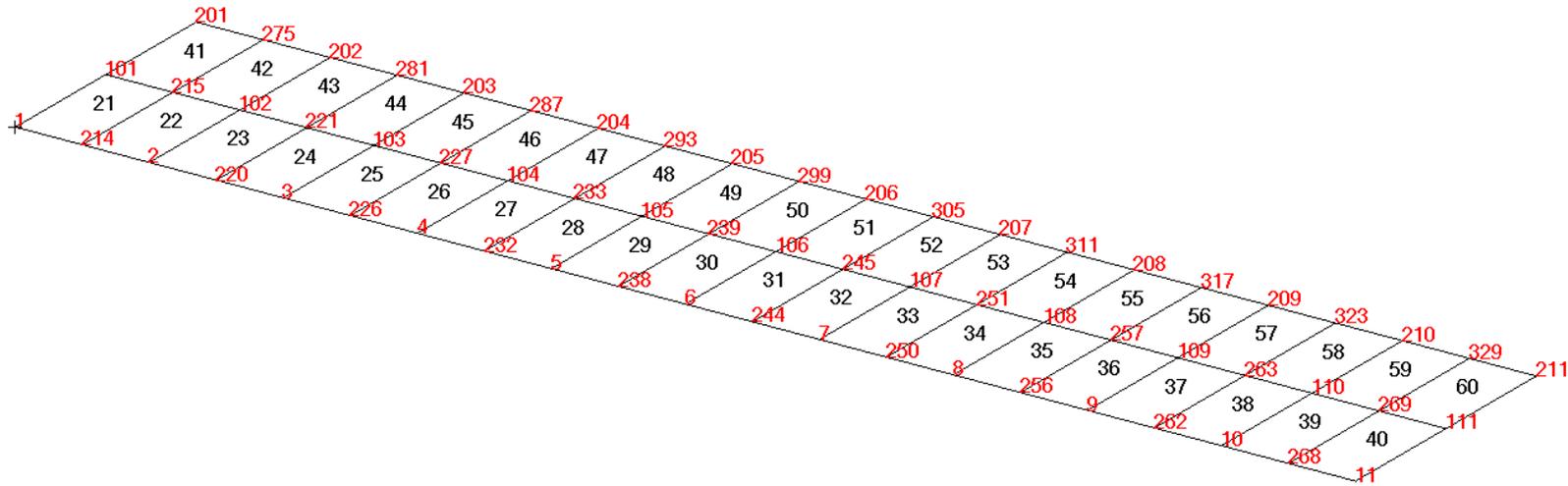
EXAMPLE 1: Demonstrate and Compare OP approach

- The following cantilever beam will be modeled using:
 - Composite shell elements (similar to what was covered in previous section)
 - Composite beam elements (Open Profile)
 - Dimensions of both models ($L = 25.0$, $W = 5.0$, $T = 1.0$)
 - Fixed left end, 2 loading subcases (axial and bending)



SHELL MODEL

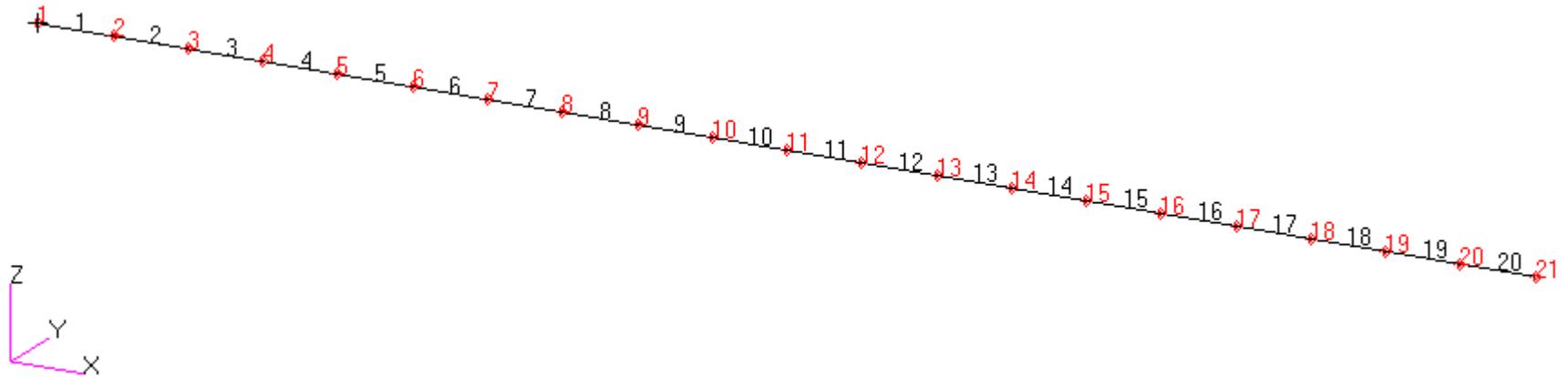
- Shell Model Connectivity



- Note the left elements (21, 41), where stresses will be highest
- Note one of the far right nodes (111), where displacement will be highest
- These will be referenced in the output SETs for the composite shell model

BEAM MODEL

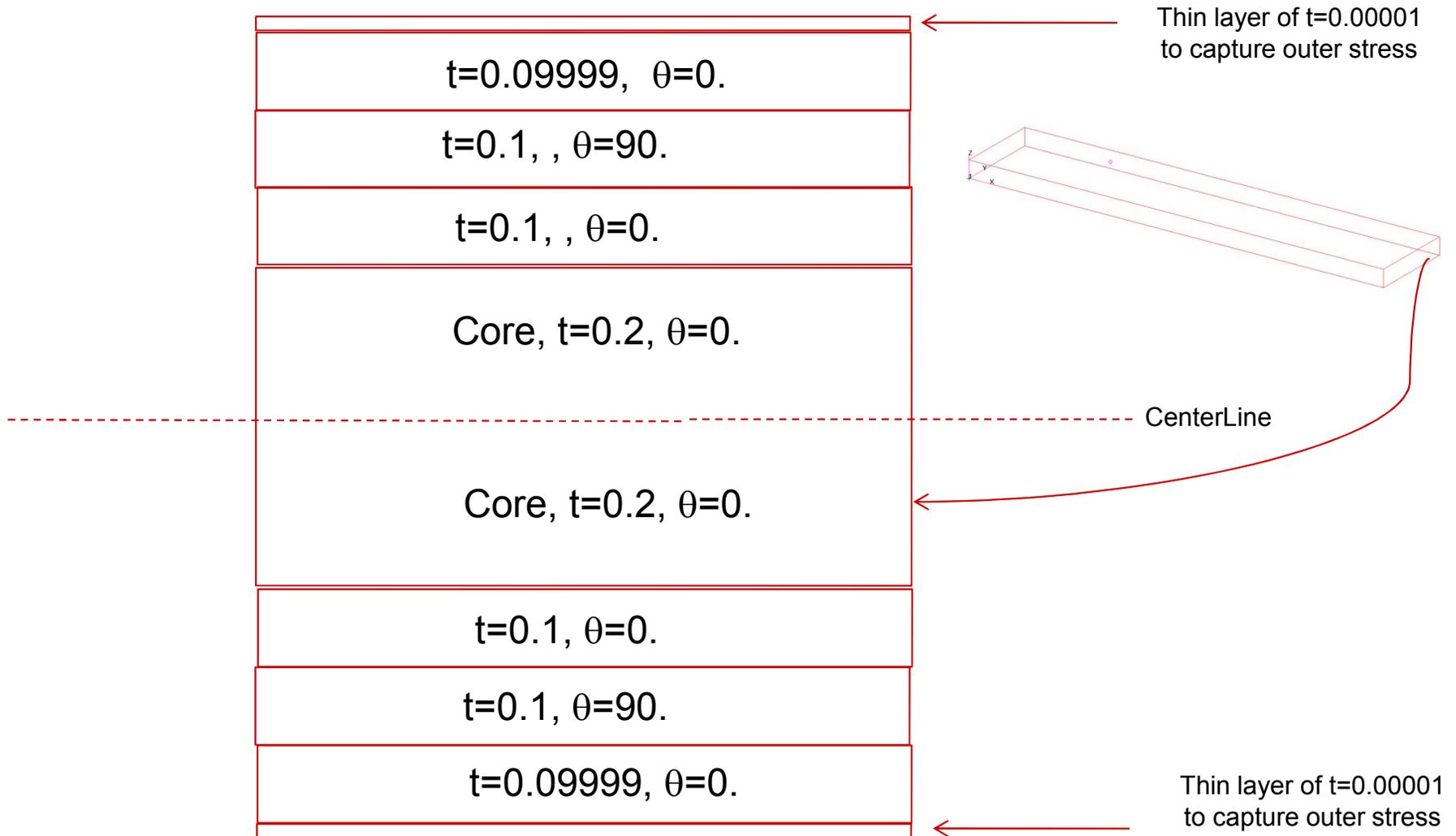
- **Beam Model Connectivity**



- Note the left element (1), where stress will be highest
- Note the far right node (21), where displacement will be highest
- These will be referenced in the output SETs for the composite beam model

COMPOSITE CROSS SECTION (Represented by Both Elements)

- Composite Cross-Section



SHELL MODEL INPUT

Abridged Input (shell1.dat)

```

SOL 101
$
CEND
  TITLE=Standard linear shell element
  SPC = 2
set 100 = 111
set 200 = 21, 41
  disp = 100
  stress = 200
$
SUBCASE 1
  subtitle = axial load
  load = 1
$
SUBCASE 2
  subtitle = bending load
  load = 2
$
BEGIN BULK
.

```

```

PCOMP 1
1 .00001 0. YES 1 .09999 0. YES
1 .1 90. YES 1 .1 0. YES
1 .4 0. YES 1 .1 0. YES
1 .1 90. YES 1 .09999 0. YES
1 .00001 0. YES
$
MAT8 1 100000. 5000. .4 3000. 3000. 2000. 1.-4
$
CQUAD4 21 1 214 215 101
.
GRID 1 0. 0. 0.
GRID 2 2.5 0. 0.
.
$
SPC1 2 123456 1 101 201
...
$
ENDDATA

```

BEAM MODEL INPUT (OP Option)

Abridged Input (beam-op.dat)

```

SOL 101
CEND
$ Direct Text Input for Global Case Control Data
TITLE = Beam Composite, Open Profile
set 100 = 21
set 200 = 1
$
disp = 100
stress = 200
$
SUBCASE 1
$ Subcase name : ax_load1
  SUBTITLE= axial load
  SPC = 2
  LOAD = 2
SUBCASE 2
$ Subcase name : bnd_load2
  SUBTITLE= bending load
  SPC = 2
  LOAD = 4
BEGIN BULK
  
```

```

PARAM,ARBMfem,YES $ generate bdf of cross sect
PARAM,ARBMPS,YES $ generate postscript of cross sect
param,arbmstyp,timoshen $ use Timoshenko solution
$
CBEAM3 1 1 1 2 101 0. 0. 1.
.
$
PBMSECT 1 1 op
.
$
SET1 101 1001 1002 1003 1004 1005 1006
$
POINT 1001 0. 0.
POINT 1002 1.0 0.
POINT 1003 2.0 0.
POINT 1004 3.0 0.
POINT 1005 4.0 0.
POINT 1006 5.0 0.
$
  
```

OUTer (centerline) Profile

outp=101,c=100

Core:
Points to PCOMP 100

BEAM MODEL INPUT (OP Option)

Abridged Input (beam-op.dat) cont.

```

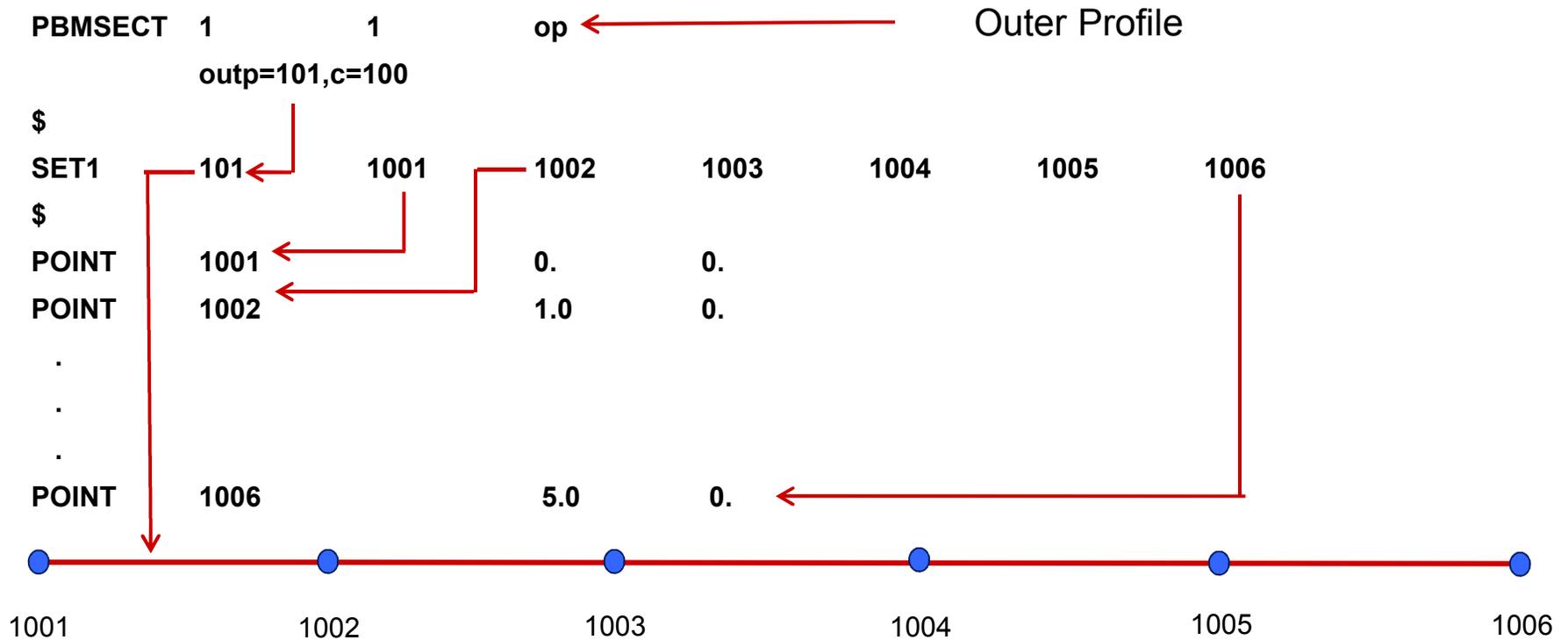
$
PCOMP 100
1 .00001 0. YES 1 .09999 0. YES
1 .1 90. YES 1 .1 0. YES
1 .2 0. YES 1 .2 0. YES
1 .1 0. YES 1 .1 90. YES
1 .09999 0. YES 1 .00001 0. YES
$
MAT8 1 100000. 5000. .4 3000. 3000. 2000. 1-4
$
GRID 1 0. 0. 0.
.
SPCADD 2 5
LOAD 2 1. 1. 1
SPC1 5 123456 1
LOAD 4 1. 1. 3
FORCE 1 21 0 400. 1. 0. 0.
FORCE 3 21 0 3. 0. 0. -1.
$
ENDDATA
  
```

Annotations in the image:

- A red arrow points from the value '100' in the PCOMP header to the first '1' in the first data row.
- A purple arrow points from the circled '1' in the fifth data row to the first '1' in the MAT8 header.
- The '1' in the fifth data row is circled in purple.
- The '1' in the MAT8 header is circled in purple.

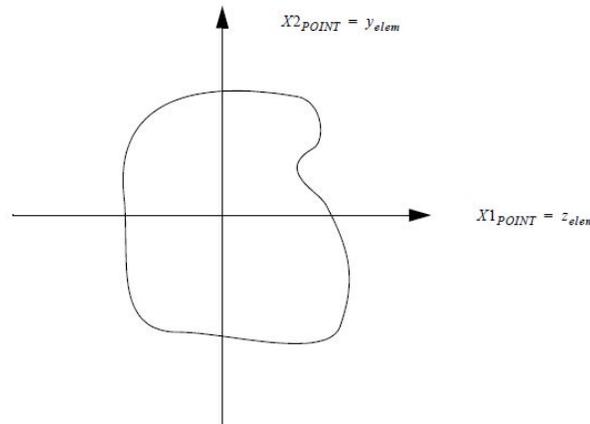
CROSS SECTIONAL PROFILE FOR THE BEAM MODEL

- Cross section profile as described by the combination of the **OUTP/SET1** and points



CROSS SECTIONAL PROFILE FOR THE BEAM MODEL

- Only the POINT entry ID should be listed under SET1 or SET3 entries which, in turn, are referenced by OUTP, INP and BRP.
 - In addition, the POINT entry for defining an arbitrary beam cross section must have the CP and X3 fields left blank.
 - X1 and X2 effectively describe the Z_{elem} and Y_{elem} , respectively



SET1	101	1001	1002	1003	1004	1005	1006
\$							
POINT	1001	C	0.	0.	X		
POINT	1002	P	1.0	0.	3		
POINT	1003		2.0	0.			
POINT	1004		3.0	0.			
POINT	1005		4.0	0.			
POINT	1006		5.0	0.			

CROSS SECTIONAL PROPERTY FOR THE BEAM MODEL

- **Cross section properties as described by PCOMP/MAT8**
 - Description of PCOMP is exactly the same as those used by the QUAD4/TRIA3 elements

PBMSECT 1 1 op
 outp=101,c=100

\$
 \$

PCOMP	100					0.	0.	
1	.00001	0.	YES	1	.09999	0.	YES	
1	.1	90.	YES	1	.1	0.	YES	
1	.2	0.	YES	1	.2	0.	YES	
1	.1	0.	YES	1	.1	90.	YES	
1	.09999	0.	YES	1	.00001	0.	YES	

\$
 \$

MAT8	1	100000.	5000.	.4	3000.	3000.	2000.	1-4

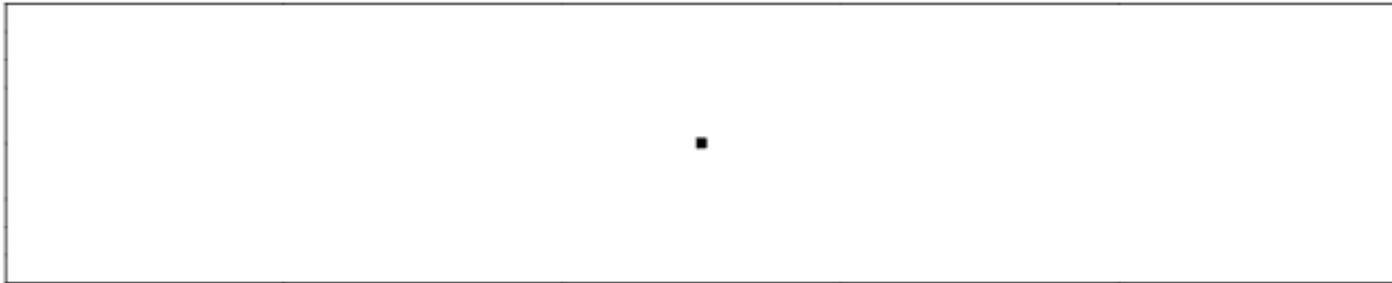
\$

CROSS SECTION PLOTTING FOR THE BEAM MODEL

- **There are 2 parameters for plotting the beam sections**
 - param,arbmps,yes
 - Above parameter creates a postscript file (.ps) that plots the outline of the cross section.
 - This can be viewed with any postscript viewer (e.g., ghostview)
 - param,arbmfm,yes
 - Above parameter creates a .bdf file of the cross section that can be plotted in a pre-processor
 - The .bdf filename uses the following convention
 - jobname_bmyy_zz.bdf
 - yy ID of PBMSECT
 - zz station ID for the PBMSECT, '01' for end A
 - This model is oriented in the YZ plane (effectively, the element YZ coordinate frame)

CROSS SECTION PLOTTING FOR THE BEAM MODEL (CONT.)

- Outline postscript plot of profile (generated by “param,arbmps,yes”)
 - This can be viewed by any postscript viewer (e.g., ghostview)



CROSS SECTION PLOTTING FOR THE BEAM MODEL (CONT.)

- Plot of cross section .bdf file generated by “param,arbmfm, yes”
 - This should be done to ensure the cross section is properly defined on the PBMSECT entry

10066	10050	10052	10040	10040	10030	10028	10020	10029	10010	10010
10064	10049	10050	10039	10039	10029	10027	10019	10018	10009	10009
10063	10048	10049	10038	10038	10028	10032	10018	10021	10008	10008
10062	10047	10048	10037	10044	10027	10031	10017	10017	10007	10007
	10046		10036		10026		10016		10006	
10061		10047		10043		10030		10016		10006
	10045		10035		10025		10015		10005	
10060		10055		10037		10026		10020		10005
10059	10044	10046	10034	10042	10024	10029	10014	10015	10004	10004
10058	10043	10045	10033	10036	10023	10025	10013	10014	10003	10003
10057	10042	10058	10032	10038	10022	10028	10012	10013	10002	10002
	10041		10031		10021		10011		10001	10001



DISPLACEMENTS OUTPUT COMPARISON

- **Output from composite beam elements**
 - Remember, Subcase 1 is axial (T1) loading, Subcase 2 is bending (T3) loading

```

0
SUBCASE 1
DISPLACEMENT VECTOR
POINT ID.  TYPE  T1  T2  T3  R1  R2  R3
21  G  2.455140E-02  0.0  0.0  0.0  0.0  0.0
1 MSC.NASTRAN JOB CREATED ON 22-MAR-12 AT 10:42:57 MARCH 27, 2012 MSC.NASTRAN 11/22/11 PAGE 14
LOAD2
0
SUBCASE 2
DISPLACEMENT VECTOR
POINT ID.  TYPE  T1  T2  T3  R1  R2  R3
21  G  0.0  0.0  -5.244019E-01  0.0  3.109019E-02  0.0
    
```

- **Output from composite shell elements**

```

0
SUBCASE 1
DISPLACEMENT VECTOR
POINT ID.  TYPE  T1  T2  T3  R1  R2  R3
111  G  2.454317E-02  -3.278874E-10  0.0  0.0  0.0  -6.812093E-12
1 STANDARD LINEAR SHELL ELEMENT MARCH 27, 2012 MSC.NASTRAN 11/22/11 PAGE 14
BENDING LOAD
0
SUBCASE 2
DISPLACEMENT VECTOR
POINT ID.  TYPE  T1  T2  T3  R1  R2  R3
111  G  0.0  0.0  -5.240583E-01  1.113469E-10  3.096623E-02  0.0
    
```

STRESS OUTPUT COMPARISON

- Output from composite beam elements

```

0
                                SUBCASE 1
                                STRESSES IN BEAM3 ELEMENTS      ( C B E A M 3 )
                                - STRESSES IN LOCAL COORDINATE SYSTEM -
ELEMENT-ID  GRID/ STRESS
GAUSS COMPONENT  SXC          SXD          SXE          SXF          S-MAX          S-MIN          M.S.-T  M.S.-C
0           1
           1   SX  9.883417E+01  9.883417E+01  9.883417E+01  9.883417E+01  9.883417E+01  9.883417E+01
           SY  2.456698E-17  2.456698E-17  2.456698E-17  2.456698E-17  2.456698E-17  2.456698E-17
0
                                SUBCASE 2
0           1
           1   SX -1.251937E+02 -1.251937E+02  1.251937E+02  1.251937E+02  1.251937E+02 -1.251937E+02
           SY -3.348695E-01 -3.348695E-01 -3.348695E-01 -3.348695E-01 -3.348695E-01 -3.348695E-01
    
```

- Output from composite shell elements

```

0
                                SUBCASE 1
                                STRESSES IN LAYERED COMPOSITE ELEMENTS ( QUAD 4 )
                                STRESSES IN FIBER AND MATRIX DIRECTIONS INTER-LAMINAR STRESSES PRINCIPAL STRESSES (ZERO SHEAR) MAX
ELEMENT  PLY ID NORMAL-1 NORMAL-2 SHEAR-12 SHEAR XZ-MAT SHEAR YZ-MAT ANGLE MAJOR MINOR SHEAR
0       21  1  9.87856E+01  1.76206E+00  2.51553E-01  0.0  0.0  0.15  9.87863E+01  1.76141E+00  4.85124E+01
0       21  2  9.87856E+01  1.76206E+00  2.51553E-01  0.0  0.0  0.15  9.87863E+01  1.76141E+00  4.85124E+01
0
                                SUBCASE 2
                                STRESSES IN LAYERED COMPOSITE ELEMENTS ( QUAD 4 )
                                STRESSES IN FIBER AND MATRIX DIRECTIONS INTER-LAMINAR STRESSES PRINCIPAL STRESSES (ZERO SHEAR) MAX
ELEMENT  PLY ID NORMAL-1 NORMAL-2 SHEAR-12 SHEAR XZ-MAT SHEAR YZ-MAT ANGLE MAJOR MINOR SHEAR
0       21  1 -1.22103E+02 -2.28048E+00 -2.06255E-01 -5.00830E-05  3.29401E-07 -89.90 -2.28012E+00 -1.22103E+02  5.99115E+01
0       21  2 -1.09892E+02 -2.05243E+00 -1.85629E-01 -4.50751E-01  2.96463E-03 -89.90 -2.05211E+00 -1.09893E+02  5.39203E+01
    
```

Example 2 - BEAM MODEL (GS Option)

- In the previous example, we use the OP option.
- We will rerun the same model, using the GS option for defining the cross section
 - For the GS option, the cross section must be defined
 - This is frequently done through a separate bulk data section
 - When done in this manner, the “outm=” on the PBMSECT is used to refer to the bulk section ID:

```
BEGIN BULK Arbmodel xxx
```

where xxx is the ID of the separate bulk data section, and will be referenced by the PBSECT entry

- This section is placed at the end of the main bulk data section, just before the ENDDATA delimiter
- For this example, one PCOMP entry is needed for each layer (in the separate BULK section, externally referenced through an INCLUDE entry).

Example 2 - BEAM MODEL INPUT (GS Option)

Abridged Input (beam-gs.dat)

```
SOL 101
CEND
$ Direct Text Input for Global Case Control Data
TITLE = Beam Composite, General Section
set 100 = 21
set 200 = 1
$
disp = 100
stress = 200
$
SUBCASE 1
$ Subcase name : ax_load1
  SUBTITLE = axial load
  SPC = 2
  LOAD = 2
SUBCASE 2
$ Subcase name : bnd_load2
  SUBTITLE = bending load
  SPC = 2
  LOAD = 4
BEGIN BULK
```

```
$
PARAM,ARBMPS,YES
param,arbmstyp,timoshen
$
CBEAM3 1 1 1 2 101 0. 0. 1.
.
$
PBMSECT 1 1 gs ← GS option
          outm = 1000
$
MAT8 1 10000. 5000. .4 3000. 3000. 2000. .0001
.
.
Begin Bulk Arbmodel 1000 ← ID of additional bulk data section
$
include 'beam-gs-cs.bdf' ← Input file containing the cross section
$
ENDDATA
```

Note: A dummy material reference is needed in the main bulk data section, but is not used

Example 2 - BEAM MODEL INPUT (GS Option)

Abridged listing of beam-gs-cs.bdf

```

$
$           NUMBER OF GRID =      66           $
$           NUMBER OF ELEMNT=      50           $
$
CQUAD4     10001   10001   10001   10012   10013   10002
CQUAD4     10002   10002   10002   10013   10014   10003
.
CQUAD4     10050   10010   10051   10065   10066   10052
$
pcomp 10001  1  0.00001  0.  yes
pcomp 10002  1  0.09999  0.  yes
pcomp 10003  1  0.1      90. yes
pcomp 10004  1  0.1      0.  yes
pcomp 10005  1  0.2      0.  yes
pcomp 10006  1  0.2      0.  yes
pcomp 10007  1  0.1      0.  yes
pcomp 10008  1  0.1      0.  yes
pcomp 10009  1  0.09999  0.  yes
pcomp 10010  1  0.00001  0.  yes
$
MAT8      1      100000. 5000.  .4      3000.  3000.  2000.  1.-4
$
GRID 10001.....
.
.
GRID 10066.....
$

```

10066	10050	10052	10040	10040	10030	10028	10020	10029	10010	10010
10064	10049	10050	10039	10039	10029	10027	10019	10018	10009	10009
10063	10048	10049	10038	10038	10028	10032	10018	10021	10008	10008
10062	10047	10048	10037	10044	10027	10031	10017	10017	10007	10007
10061	10046	10047	10036	10043	10026	10030	10016	10016	10006	10006
10060	10045	10046	10035	10037	10025	10026	10015	10020	10005	10005
10059	10044	10046	10034	10042	10024	10029	10014	10015	10004	10004
10058	10043	10045	10033	10036	10023	10025	10013	10014	10003	10003
10057	10042	10045	10032	10034	10022	10025	10012	10012	10002	10002
10056	10041	10053	10031	10034	10021	10028	10011	10013	10001	10001

Example 2 - BEAM MODEL OUTPUT (GS Option)

- Note that the results are identical to those from using the OP Option

```

SUBCASE 1
DISPLACEMENT VECTOR
POINT ID. TYPE T1 T2 T3 R1 R2 R3
21 G 2.455140E-02 0.0 0.0 0.0 0.0 0.0
MSC.NASTRAN JOB CREATED ON 22-MAR-12 AT 10:42:57 APRIL 6, 2012 MSC.NASTRAN 11/22/11 PAGE 19
LOAD2

SUBCASE 2
DISPLACEMENT VECTOR
POINT ID. TYPE T1 T2 T3 R1 R2 R3
21 G 0.0 0.0 -5.244019E-01 0.0 3.109019E-02 0.0

SUBCASE 1
STRESSES IN BEAM 3 ELEMENTS (CBEAM3)
ELEMENT-ID GRID/ STRESS - STRESSES IN LOCAL COORDINATE SYSTEM -
1 GAUSS COMPONENT SXC SXD SXE SXF S-MAX S-MIN M.S.-T M.S.-C
1 1 SX 9.883417E+01 9.883417E+01 9.883417E+01 9.883417E+01 9.883417E+01 9.883417E+01
SY 2.456698E-17 2.456698E-17 2.456698E-17 2.456698E-17 2.456698E-17 2.456698E-17
SZ -8.169344E-22 8.169353E-22 -8.169365E-22 8.169353E-22 8.169353E-22 -8.169365E-22

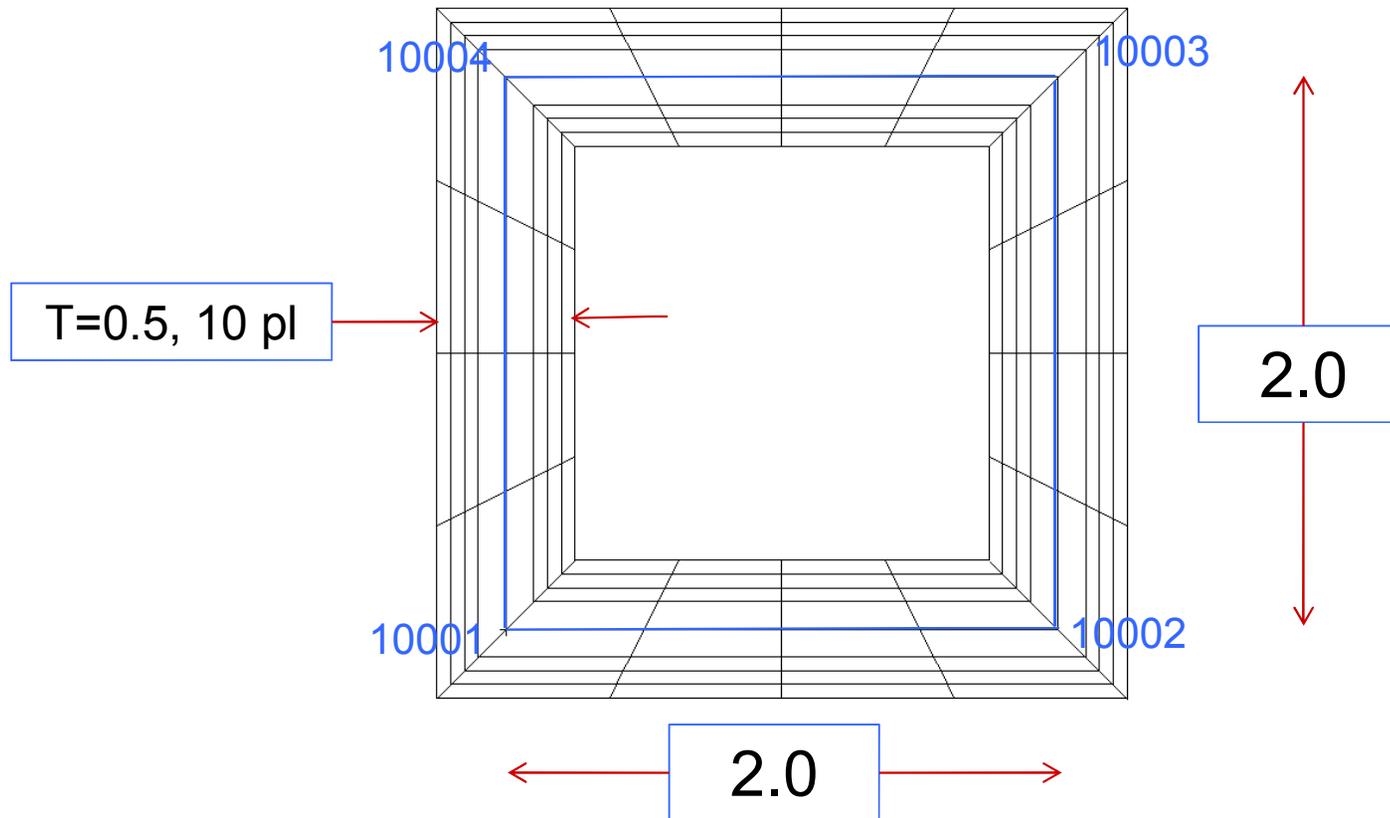
SUBCASE 2
STRESSES IN BEAM 3 ELEMENTS (CBEAM3)
ELEMENT-ID GRID/ STRESS - STRESSES IN LOCAL COORDINATE SYSTEM -
1 GAUSS COMPONENT SXC SXD SXE SXF S-MAX S-MIN M.S.-T M.S.-C
1 1 SX -1.251937E+02 -1.251937E+02 1.251937E+02 1.251937E+02 1.251937E+02 -1.251937E+02
SY -3.348695E-01 -3.348695E-01 -3.348695E-01 -3.348695E-01 -3.348695E-01 -3.348695E-01
SZ 6.529511E-11 -2.271073E-22 1.305902E-10 2.271071E-22 1.305902E-10 -2.271073E-22
    
```

– From OP results:

- Displ: T1@21, SC1 = 2.455140E-02 ; T3@21, SC2 = -5.244019E-01
- Stress: XC@1, SC1 = 9.883417E+01 ; SC2 = -1.251937E+02

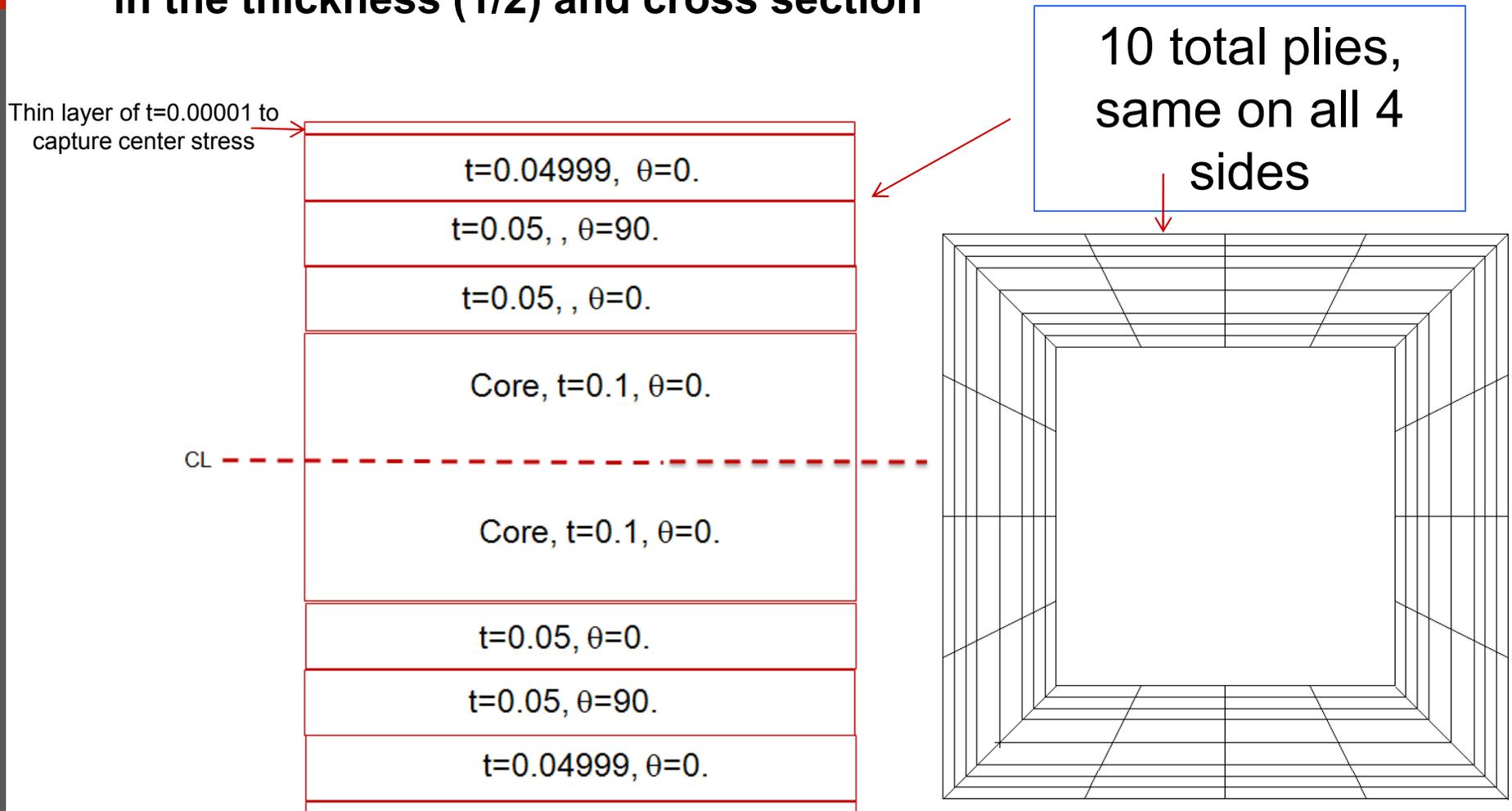
Example 3 - BEAM MODEL (CP – Option)

- Previously, we have looked at using the OP (example 1) and GS (example 2) options.
- We will use the CP (Closed Profile) option for the “rectangular” box section shown below



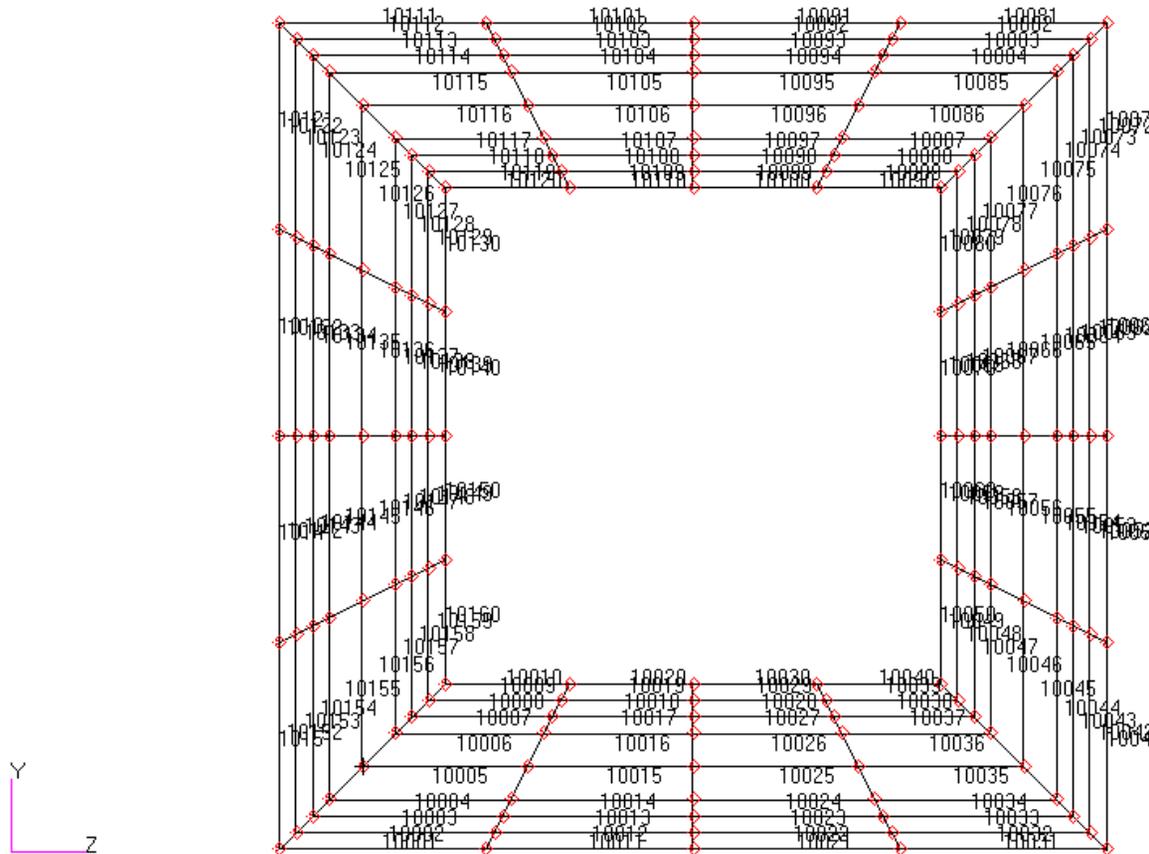
Example 3 - BEAM MODEL (CP – Option)

- Same material as in previous examples, except for the differences in the thickness (1/2) and cross section



Example 3 - BEAM MODEL (CP – Option)

- Beam box section cross-section



Example 3 - BEAM MODEL (CP – Option)

- Abridged Input (beam-cp.dat)

```

CBEAM3  1      1      1      2      101  0.    0.    1.
.
CBEAM3  20     1      20     21     120  0.    0.    1.
$
PCOMP   200
      1      .00001  0.    YES  1      .04999  0.    YES
      1      .05    90.   YES  1      .05    0.    YES
      1      .1     0.    YES  1      .1     0.    YES
      1      .05    0.    YES  1      .05    90    YES
      1      .04999  0.    YES  1      .00001  0.    YES
PBMSECT 1      1      cp ←
      outp=101,c=200 ←
$
SET1  101  1001  1002  1003  1004
$
POINT 1001  0.    0.
POINT 1002  2.0  0.
POINT 1003  2.0  2.
POINT 1004  0.0  2.
$
MAT8   1      100000. 5000.  .4    3000.  3000.  2000.  1.-4
  
```

Closed Profile

CenterLine Path

Example 3 - BEAM MODEL (CP – Option)

- Result Output

```

0
                                                                SUBCASE 1
                                D I S P L A C E M E N T   V E C T O R
POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    21      G      3.067287E-02  0.0      0.0      0.0      0.0      0.0
1 MSC.NASTRAN JOB CREATED ON 22-MAR-12 AT 10:42:57          APRIL  8, 2012 MSC.NASTRAN 11/22/11 PAGE  13
0                                                                SUBCASE 2

                                D I S P L A C E M E N T   V E C T O R
POINT ID.  TYPE      T1      T2      T3      R1      R2      R3
    21      G      0.0      0.0      -7.995695E-02  0.0      4.084843E-03  0.0
0                                                                SUBCASE 1

                                S T R E S S E S   I N   B E A M   3   E L E M E N T S           ( C B E A M   3 )
                                - STRESSES IN LOCAL COORDINATE SYSTEM -
ELEMENT-ID  GRID/ STRESS  SXC      SXD      SXE      SXF      S-MAX      S-MIN      M.S.-T  M.S.-C
    1
    1      SX      1.234230E+02  1.234230E+02  1.234230E+02  1.234230E+02  1.234230E+02  1.234230E+02
    SY      4.397556E-18  4.397556E-18 -4.397556E-18 -4.397556E-18  4.397556E-18 -4.397556E-18
    SZ      -4.397556E-18  4.397556E-18  4.397556E-18 -4.397556E-18  4.397556E-18 -4.397556E-18
    2      SX      1.234230E+02  1.234230E+02  1.234230E+02  1.234230E+02  1.234230E+02  1.234230E+02
    SY      4.397556E-18  4.397556E-18 -4.397556E-18 -4.397556E-18  4.397556E-18 -4.397556E-18
    SZ      -4.397556E-18  4.397556E-18  4.397556E-18 -4.397556E-18  4.397556E-18 -4.397556E-18
0                                                                SUBCASE 2

                                S T R E S S E S   I N   B E A M   3   E L E M E N T S           ( C B E A M   3 )
                                - STRESSES IN LOCAL COORDINATE SYSTEM -
ELEMENT-ID  GRID/ STRESS  SXC      SXD      SXE      SXF      S-MAX      S-MIN      M.S.-T  M.S.-C
    1
    1      SX      -4.114951E+01 -4.114951E+01  4.114951E+01  4.114951E+01  4.114951E+01 -4.114951E+01
    SY      -5.340491E-01 -5.340491E-01 -5.340491E-01 -5.340491E-01 -5.340491E-01 -5.340491E-01
    SZ      3.387451E-01 -3.387451E-01  3.387451E-01 -3.387451E-01  3.387451E-01 -3.387451E-01

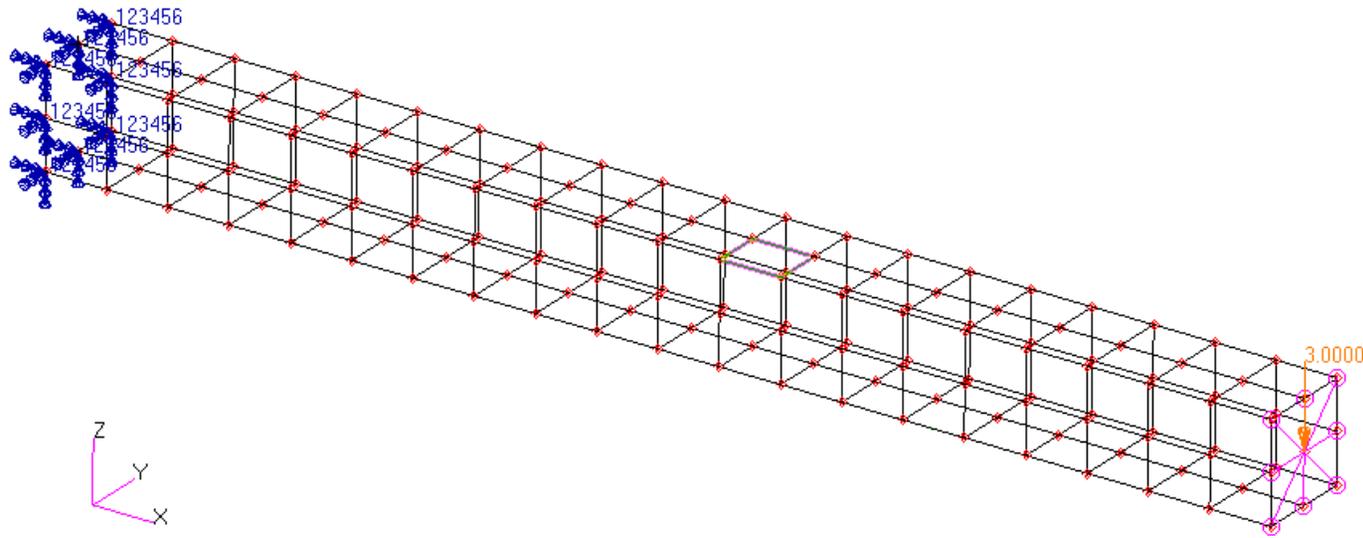
```

@bottom
(compression)

@top
(tension)

Example 3 - BEAM MODEL (CP – Option)

- As a comparison, here is a look at the rectangular cross section, using the same PCOMP specification, as a shell element model.



PCOMP	1							
1	1.-5	0.	YES	1	.04999	0.	YES	
1	.05	90.	YES	1	.05	0.	YES	
1	.1	0.	YES	1	.1	0.	YES	
1	.05	0.	YES	1	.05	90.	YES	
1	.04999	0.	YES	1	1.-5	0.	YES	

Example 3 - BEAM MODEL (CP – Option)

- Looking at the displacements
 - Beam model

```
DISPLACEMENT VECTOR
POINT ID.  TYPE  T1  T2  T3  R1  R2  R3
      21    G  3.067287E-02  0.0  0.0  0.0  0.0  0.0
MSC.NASTRAN JOB CREATED ON 22-MAR-12 AT 10:42:57          APRIL  8, 2012 MSC.NASTRAN 11/22/11  PAG
                                                    SUBCASE 2
```

```
DISPLACEMENT VECTOR
POINT ID.  TYPE  T1  T2  T3  R1  R2  R3
      21    G  0.0  0.0  -7.995695E-02  0.0  4.084843E-03  0.0
                                                    SUBCASE 1
```

- Shell model

```
MAXIMUM DISPLACEMENTS
SUBCASE/
DAREA ID  T1  T2  T3  R1  R2  R3
      1  3.0678160E-02  1.1945469E-04  1.1945514E-04  3.7133735E-10  7.8330260E-05  7.8330260E-05
      2  4.2703361E-03  2.6171650E-05  8.4238790E-02  2.0941197E-05  4.7577247E-03  5.7267448E-06
```

Example 3 - BEAM MODEL (CP – Option)

- Looking at the stresses
 - Beam model

```

                S T R E S S E S   I N   B E A M 3   E L E M E N T S
    ELEMENT-ID  GRID/ STRESS          - STRESSES IN LOCAL COORDINATE SYSTEM -
                GAUSS COMPONENT  SXC          SXD          SXE          SXF          S-M
    1
    1           1   SX           1.234230E+02  1.234230E+02  1.234230E+02  1.234230E+02  1.234
                SY           4.397556E-18  4.397556E-18 -4.397556E-18 -4.397556E-18  4.397
    S T R E S S E S   I N   B E A M 3   E L E M E N T S
    IT-ID       GRID/ STRESS          - STRESSES IN LOCAL COORDINATE SYSTEM
                GAUSS COMPONENT  SXC          SXD          SXE          SXF
    1
    1           1   SX           -4.114951E+01 -4.114951E+01  4.114951E+01  4.114951E+01
    
```

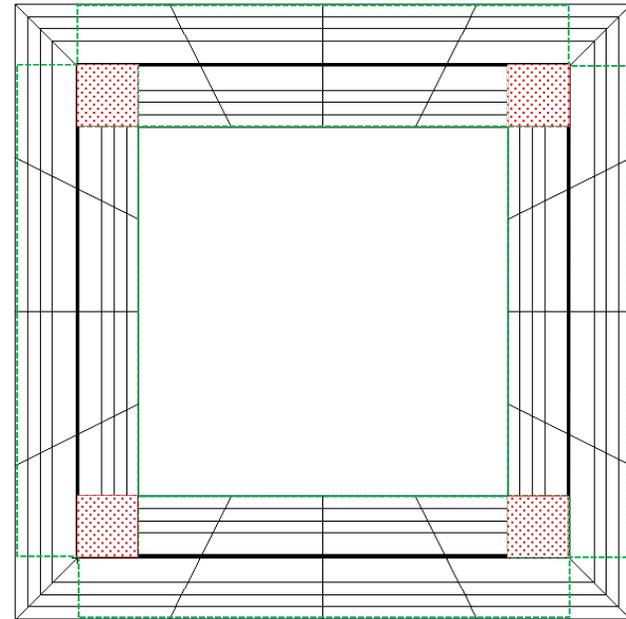
- Shell model (at Ply 10, furthest out)

```

    S T R E S S E S   I N   L A Y E R E D   C O M P O S I T E   E L E M E N T S   ( Q U A D 4 )
    ELEMENT  PLY  STRESSES IN FIBER AND MATRIX DIRECTIONS  INTER-LAMINAR STRESSES  PRINCIPAL STRESSES (ZERO
    ID       ID  NORMAL-1    NORMAL-2    SHEAR-12    SHEAR XZ-MAT  SHEAR YZ-MAT  ANGLE  MAJOR  MI
    1       1   1.22888E+02  2.16428E+00 -1.30101E-01 -1.99930E-05 -5.86824E-07 -0.06  1.22888E+02  2.16
    1       1   1.22947E+02  2.16547E+00 -1.30204E-01 -8.99701E-02 -2.64076E-03 -0.06  1.22948E+02  2.16
    1       2   -3.45319E+00  6.04092E+00  1.30409E-01 -9.34690E-02 -4.37193E-02  89.21  6.04271E+00 -3.45
    1       10  1.24081E+02  2.18815E+00 -1.32160E-01 -2.84448E-17 -1.49812E-17 -0.06  1.24081E+02  2.18
    S T R E S S E S   I N   L A Y E R E D   C O M P O S I T E   E L E M E N T S   ( Q U A D 4 )
    ELEMENT  PLY  STRESSES IN FIBER AND MATRIX DIRECTIONS  INTER-LAMINAR STRESSES  PRINCIPAL STRESSES (ZERO
    ID       ID  NORMAL-1    NORMAL-2    SHEAR-12    SHEAR XZ-MAT  SHEAR YZ-MAT  ANGLE  MAJOR  MI
    1       10  4.79424E+01  8.92162E-01  9.15990E-02 -8.72713E-17 -2.24074E-17  0.11  4.79425E+01  8.91
    ELEMENT  PLY  STRESSES IN FIBER AND MATRIX DIRECTIONS  INTER-LAMINAR STRESSES  PRINCIPAL STRESSES (ZERO
    ID       ID  NORMAL-1    NORMAL-2    SHEAR-12    SHEAR XZ-MAT  SHEAR YZ-MAT  ANGLE  MAJOR  MI
    81      10  -4.79424E+01 -8.92162E-01 -9.15990E-02  8.72713E-17  2.24074E-17 -89.89 -8.91984E-01 -4.79
    
```

Example 3 - BEAM MODEL (CP – Option)

- Note that the axial stresses and displacements are nearly identical
- The bending stresses/displacements are fairly close (around 5%), but the shell representation seems a little less stiff (greater displacement, and has a slightly higher stress).
 - If you overlay an assumption of the shell mesh extrapolated normal out to $\pm 1/2T$ against the mesh output by PARAM,ARBMFEM (reflecting internal approach):
 - An assumed overlay results in the same total area (hence axial identical), but the red regions are “double-covered”, while the outermost corners are missed (which results in a slightly reduced I_y and I_z).
 - This is a fairly “thick” set of plies relative to the dimensions of the cross section. A thinner layup would have smaller error.



COMPOSITE BEAM USING VAM

- **Guideline and Limitations**

- Third point of CBEAM3 is ignored if it points to a PBMSECT ID.
- Midside node must lie on a straight line between the 2 end points
- Center line of a profile, defined by SET1 or SET3, must fall between plies
 - (e.g., if there is a core at the center, it must be broken down into 2 cores)
- TEMPRB (1D temperature field) is not supported
- NSM is not supported (for composites through PBMSECT)
- SOL 200 (optimization) does not support composite beam elements
- Limited pre-processing support
 - Can design the beam elements (in Patran), but cannot apply a Laminate Composite material to them.
 - Need to reference a placeholder property, and manually create the PBMSECT in the input file using the same ID (and remove the dummy PBEAM afterwards)
 - Importing the BDF (into Patran) will not bring in the elements, which reference the PBMSECT
 - Nodes are imported, as are the PCOMP entries



SECTION 4

ADVANCED COMPOSITE ELEMENTS – Part 2

Solid Composite

TYPE OF COMPOSITE ELEMENTS

- **Shells and beam elements were discussed in the previous sections**
- **There are cases where neither shells nor beams satisfy the need**
 - For example, if you are interested in peeling stresses, etc.
- **Solid composite will be discussed in this section**

SOLID COMPOSITES

- **Solid Composites may be needed in cases where**
 - Laminate is thick and the material is multi-directional
 - Transverse shear are critical
 - Composite core is soft as compared to the face sheets
 - Peeling stresses are important

- **There are 3 types of solid composites**
 - General anisotropic/orthotropic
 - Layered Solid
 - Solid Shell

GENERAL ANISOTROPIC/ORTHOTROPIC SOLID

- The properties are defined by the PSOLID entry

PSOLID Properties of Solid Elements

Defines the properties of solid elements (CHEXA, CPENTA, and CTETRA entries).

Format:

1	2	3	4	5	6	7	8	9	10
PSOLID	PID	MID	CORDM	IN	STRESS	ISOP	FCTN	COROT	

Example:

PSOLID	2	100	6	TWO	GRID	REDUCED			
--------	---	-----	---	-----	------	---------	--	--	--

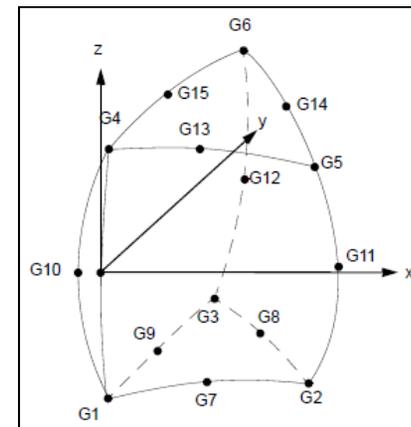
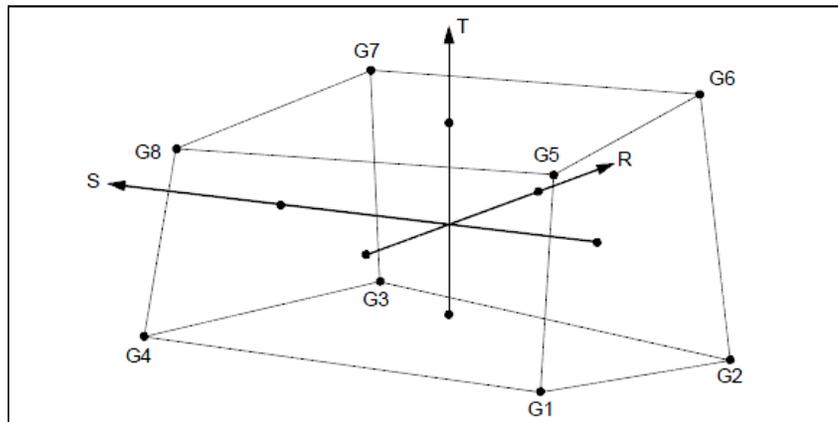
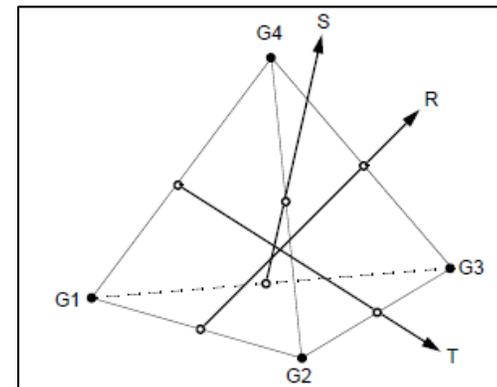
GENERAL ANISOTROPIC/ORTHOTROPIC SOLID

- The properties are defined by the PSOLID entry (continued)

Field	Contents
PID	Property identification number. (Integer > 0)
MID	Identification number of a MAT1, MAT4, MAT5, MAT9, or MAT10 entry. (Integer > 0)
CORDM	Identification number of the material coordinate system. See Remarks 3. and 4. (Integer; Default = 0, which is the basic coordinate system; see Remark 3.)
IN	Integration network. See Remarks 6., 7., 8., and 10. (Integer; Character, or blank)
STRESS	Location selection for stress output. See Remarks 9. and 10. (Integer; Character, or blank)
ISOP	Integration scheme. See Remarks 6., 7., 8., and 10. (Integer; Character, or blank)
FCTN	Fluid element flag. (Character: "FFLUID" indicates a fluid element with frequency dependent rigid absorber properties, "PFLUID" indicates a fluid element, "SMECH" indicates a structural element; Default = "SMECH.")
COROT	Corotational request. SOL 700 only. (Integer; Default = 0): 0 Do not rotate 1 Force local coordinate system to rotate with element

MATERIAL COORDINATE SYSTEM

- You can request solid element stress output in 3 different coordinate systems (on the CORDM field)
 - Basic coordinate system (CORDM=0, default for linear analysis)
 - Material coordinate system (CORDM > 0)
 - Element coordinate system (CORDM = -1)
- Element coordinate system
 - Based on element shape
 - Based on element connectivity



ANISOTROPIC MATERIAL

- **The anisotropic material is defined by the MAT9 entry**
- **For anisotropic material, you can have up to 21 material constants**
- **When using MAT9, it is advisable to define a material coordinate system on the PSOLID entry (can be basic)**
- **The MAT9 is supported by CHEXA, CTETRA, and CPENTA elements**

ANISOTROPIC MATERIAL

- The 3-dimensional anisotropic material is defined by

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{Bmatrix} = [G] \begin{Bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{Bmatrix} - (T - T_{REF}) \begin{Bmatrix} A_1 \\ A_2 \\ A_3 \\ A_4 \\ A_5 \\ A_6 \end{Bmatrix}$$

where

$$[G] = \begin{bmatrix} G11 & & & & & \\ G12 & G22 & & & & \\ G13 & G23 & G33 & & & \\ G14 & G24 & G34 & G44 & & \\ G15 & G25 & G35 & G45 & G55 & \\ G16 & G26 & G36 & G46 & G56 & G66 \end{bmatrix}$$

symmetric

ANISOTROPIC MATERIAL

- The MAT9 format is as follows:

Format:

1	2	3	4	5	6	7	8	9	10
MAT9	MID	G11	G12	G13	G14	G15	G16	G22	
	G23	G24	G25	G26	G33	G34	G35	G36	
	G44	G45	G46	G55	G56	G66	RHO	A1	
	A2	A3	A4	A5	A6	TREF	GE		

Field	Contents
MID	Material identification number. (Integer > 0)
Gij	Elements of the 6 x 6 symmetric material property matrix in the material coordinate system. (Real)
RHO	Mass density. (Real)
Ai	Thermal expansion coefficient. (Real)
TREF	Reference temperature for the calculation thermal loads, or a temperature-dependent thermal expansion coefficient. See Remark 7. (Real or blank)
GE	Structural element damping coefficient. See Remarks 6. and 8. (Real)

ANISOTROPIC MATERIAL – MAT9

Home Geometry Properties Loads/BCs Meshing Analysis Results Durability

Isotropic Orthotropic Anisotropic Fluid Cohesive Composite

Input Options

Constitutive Model: Linear Elastic

Property Name	Value
Stiffness 11 =	
Stiffness 12 =	
Stiffness 13 =	
Stiffness 14 =	
Stiffness 15 =	
Stiffness 16 =	
Stiffness 22 =	
Stiffness 23 =	
Stiffness 24 =	
Stiffness 25 =	
Stiffness 26 =	
Stiffness 33 =	
Stiffness 34 =	
Stiffness 35 =	
Stiffness 36 =	
Stiffness 44 =	
Stiffness 45 =	
Stiffness 46 =	
Stiffness 55 =	
Stiffness 56 =	
Stiffness 66 =	
Density =	
Thermal Expan. Coeff 11 =	
Thermal Expan. Coeff 22 =	
Thermal Expan. Coeff 33 =	
Thermal Expan. Coeff 12 =	
Thermal Expan. Coeff 31 =	
Thermal Expan. Coeff 23 =	
Structural Damping Coeff =	
Reference Temperature =	

G matrix

Materials

Action: Create

Object: 3d Anisotropic

Method: Manual Input

Existing Materials

Filter *

Material Name

Description

Input Properties ...

Change Material Status ...

Apply

ORTHOTROPIC MATERIAL

- When the anisotropic material has 3 planes of symmetry, the anisotropic material becomes orthotropic
- The number of material constants reduced from 21 to 9
- Many practical designs are based on orthotropic material rather than general anisotropic material
- The [G] matrix simplifies to

$$G_{11} = \left(\frac{1 - \nu_{yz} \nu_{zy}}{E_y E_z \Delta} \right)$$

$$G_{12} = \left(\frac{\nu_{yx} + \nu_{zx} \nu_{yz}}{E_y E_z \Delta} \right)$$

$$G_{13} = \left(\frac{\nu_{zx} + \nu_{yx} \nu_{zy}}{E_y E_z \Delta} \right)$$

$$G_{22} = \left(\frac{1 - \nu_{xz} \nu_{zx}}{E_x E_z \Delta} \right)$$

$$G_{23} = \left(\frac{\nu_{zy} + \nu_{xy} \nu_{zx}}{E_x E_z \Delta} \right)$$

$$G_{33} = \left(\frac{1 - \nu_{xy} \nu_{yx}}{E_x E_y \Delta} \right)$$

ORTHOTROPIC MATERIAL

where:

$$\begin{aligned} \nu_{ij} &= \text{Poisson's ratios where } \nu_{ij}/E_i = \nu_{ji}/E_j \\ E_x, E_y, E_z &= \text{Young's modulus in the x-, y- and z-directions} \\ G_{xy}, G_{yz}, G_{zx} &= \text{shear moduli} \\ \Delta &= \frac{1 - \nu_{xy}\nu_{yx} - \nu_{yz}\nu_{zy} - \nu_{zx}\nu_{xz} - 2\nu_{yx}\nu_{zy}\nu_{xz}}{E_x E_y E_z} \end{aligned}$$

$$\begin{aligned} |\nu_{yx}| &< \sqrt{\frac{E_y}{E_x}} & |\nu_{zy}| &< \sqrt{\frac{E_z}{E_y}} & |\nu_{xz}| &< \sqrt{\frac{E_x}{E_z}} \\ |\nu_{xy}| &< \sqrt{\frac{E_x}{E_y}} & |\nu_{yz}| &< \sqrt{\frac{E_y}{E_z}} & |\nu_{zx}| &< \sqrt{\frac{E_z}{E_x}} \end{aligned}$$

$$G_{44} = G_{xy}$$

$$G_{55} = G_{yz}$$

$$G_{66} = G_{zx}$$

$$G_{14} = G_{15} = G_{16} = 0.0$$

$$G_{24} = G_{25} = G_{26} = 0.0$$

$$G_{34} = G_{35} = G_{36} = 0.0$$

Shear/Extension
coupling terms

$$G_{45} = G_{46} = G_{56} = 0.0$$

Shear-shear coupling

ORTHOTROPIC MATERIAL

- The orthotropic material can be represented by the MAT9 by retaining the following terms:
 - G_{11} , G_{12} , G_{13} , G_{22} , G_{23} , G_{33} , G_{44} , G_{55} , G_{66}
- If running SOL 400, you can alternatively use the MATORT entry (we will get to this shortly)

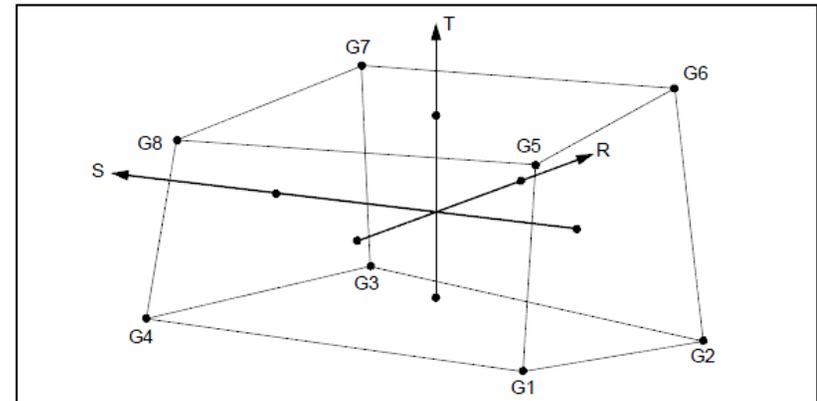
SOLID WITH LAYERED INFORMATION

- **There are cases where it is also necessary to define only a single element to represent the full thickness of the material, but where individual layer definition is also desired in a thick laminate**
 - → A layered solid instead of shell can be used
 - Inter-ply peeling stresses can still be captured with this definition approach
- **Input set up is similar to PCOMP for a shell element**
 - Each layer is assumed to be placed parallel to a pair of opposite element faces, so that the “thickness” direction is from one of the element faces to its opposite one
 - Ply numbering proceeds from bottom to top, in the thickness direction for the element
- **There are 2 types of solid available for defining layered information**
 - Solid Layered composite (often referred to as regular solid composites)
 - Solid Shell

DEFINING LAYERED SOLID OR SOLID SHELL COMPOSITE PROPERTY

- Only CHEX elements are supported
- Integration schemes defined by INT8 and INT20 fields on PCOMPLS
 - The layered solid uses L (linear) or Q (quadratic) integration
 - The solid shell uses ASTN (assumed enhanced strain formulation)
- The DIRECT field (default=+1) defines the direction of the ply and whether the thicknesses are defined in terms of
 - % of thickness +1, +2, or +3
 - Actual thickness -1, -2, or -3

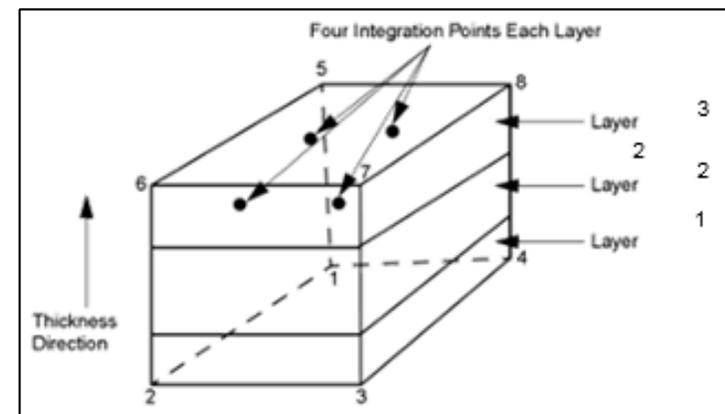
Layer orientation			
DIRECT	Normal to Layer Plane	Layers run parallel from face (ply numbering starts here)	to face (ends here)
1	Element T direction	G1-G2-G3-G4	G5-G6-G7-G8
2	Element R direction	G1-G4-G8-G5	G2-G3-G7-G6
3	Element S direction	G1-G2-G6-G5	G4-G3-G7-G8



- For INTi=ASTN, only Direct= +1 or -1 is allowed

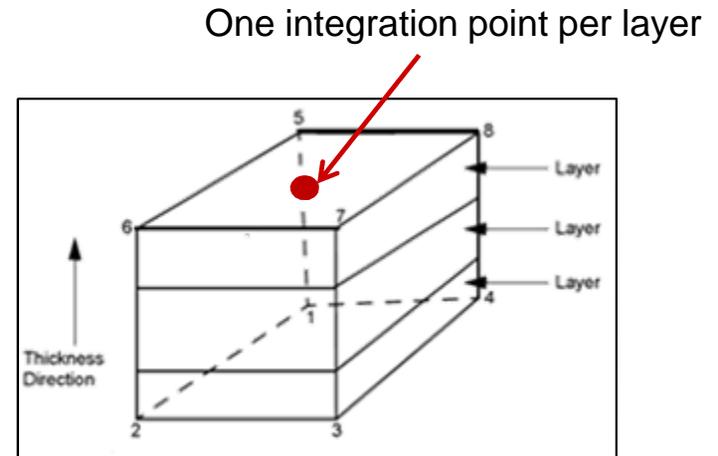
LAYERED SOLID COMPOSITE

- Defined by INTi on PCOMPLS: INT8=L or INT20=Q
- The layered solid element is based on a solid continuum element formulation
 - It has appropriate interpolation functions derived that take account of the fact that it is now a layered element rather than homogeneous
 - It is not just a collection of plane stress shells through the thickness
- 3D ply materials are required (e.g. MAT1, MAT9, or MATORT) to specify the transverse shear stiffness
- A minimum of 2 layers is recommended for each element to avoid rank deficiency (element mechanisms)
 - Each layer has 4 integration points
 - Each layer may have different material
 - Maximum of 510 layers per element



SOLID SHELL COMPOSITE

- **Defined using INTi on PCOMPLS**
 - INT8 or INT20 = ASTN
(assumed strain integration)
- **Each layer has 1 integration point**
- **Maximum of 2040 layers allowed per element**
- **Unlike layered solids, layers in a solid shell must be specified in the orientation DIRECT = 1 (between faces 1-2-3-4 and 5-6-7-8)**
 - This requires care in generating the mesh
- **➔ For structure with significant bending load, the solid shell formulation is preferred over layered solid**



DEFINING LAYERED SOLID OR SOLID SHELL COMPOSITE PROPERTY

- PCOMPLS can only be referenced by CHEXA elements
- PCOMPLS format:

Format:

1	2	3	4	5	6	7	8	9	10
PCOMPLS	PID	DIRECT	CORDM	SB	ANAL				
	"C8"	BEH8	INT8	BEH8H	INT8H	←			
	"C20"	BEH20	INT20	BEH20H	INT20H	←			
	ID1	MID1	T1	THETA1					
	ID2	MID2	T2	THETA2					

8-noded elements

20-noded elements

Example:

PCOMPLS	782	1							
	1001	171	.3	12.3					
	100	175	.7	77.7					

DEFINING LAYERED SOLID OR SOLID SHELL COMPOSITE PROPERTY

- **PCOMPLS Format: (cont.)**

Field	Contents
PID	Property identification number. (Integer > 0)
DIRECT	The layer direction for BEHi=COMPS or AXCOMP. See Remark 5. for direction definition. A positive value implies that the composite layer inputs is a fractional percent of the total element thickness in the ply direction and is recommended. A negative value implies that the composite layer input is the actual thickness of that ply. (Integer ± 1 , ± 2 , or ± 3 ; Default +1)
CORDM	Identification number of the material coordinate system. See Remark 10. (Integer; Default = 0, which is basic)
SB	Allowable shear stress of the bonding material (allowable interlaminar shear stress). (Real ≥ 0.0)
ANAL	Analysis type. ANAL='IS' - Implicit structural elements are being referred to. ANAL='IH' - Implicit heat analysis elements are being referred to. ANAL='ISH' - Implicit structural and heat elements are being referred to. (Character Default ISH)
C8	Keyword indicating that two items following apply to elements with eight corner grids. (Character)
C20	Keyword indicating that two items following apply to elements with eight corner grids and twelve mid-side grids. (Character)

DEFINING LAYERED SOLID OR SOLID SHELL COMPOSITE PROPERTY

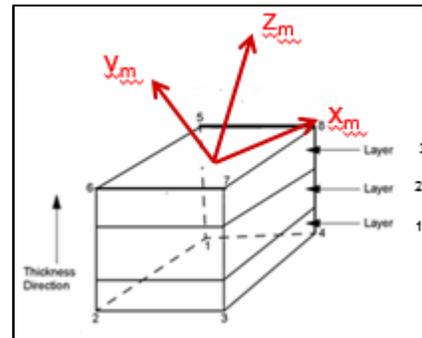
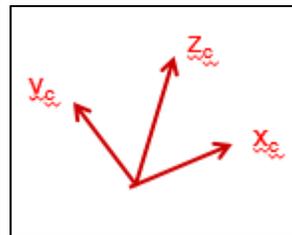
- **PCOMPLS Format: (cont.)**

Field	Contents
BEHi	Element structural behavior. See Remarks 4. and 7. (Character default: SLCOMP for BEH8 and BEH20)
INTi	Integration scheme. See Remark 9. (Character default: L for INT8, Q for INT20)
BEHiH	Element heat behavior. See Remarks 4. and 8. (Character Default: SLCOMP for BEH8H and BEH20H)
INTiH	Integration scheme. See Remark 9. (Character Default: L for INT8H, Q for INT20H)
IDi	Global Ply ID. Must be unique with respect to other plies in this entry. See Remark 2. (Integer > 0)
MIDi	Material ID for the ply. See Remark 3. (Integer > 0)
Ti	Either fractional percent of the total element thickness or actual thickness of that ply depending on \pm value of DIRECT. See Remarks 5. and 6.. (Real > 0.0)
THETAi	Orientation angle of the ply in the plane of the plies. Measured relative to the projection z-axis defined by CORDM on the plane defined by DIRECT. See Remark 1. (Real; Default = 0.0)

- **MID can point to MAT1, MAT9, MATORT, and MATHE among common structural materials. Additional options are in the QRG.**

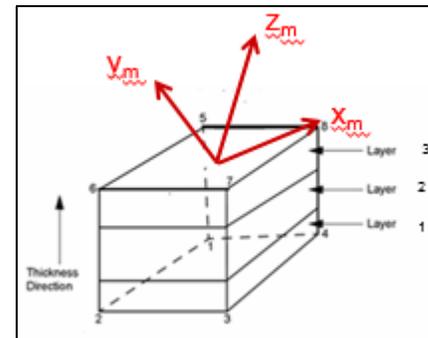
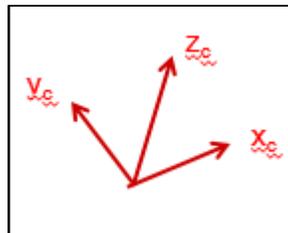
MATERIAL ORIENTATION

- The CORDM entry is interpreted differently depending on whether layered solid or solid shell is used
- For Layered Solid Composites (INTi=L or Q): Coord is used directly
 - The coordinate system defines on the CORDM directly defines the local coordinate system. There is no axis projection.
 - If a rotation angle is defined, the positive θ direction follows the right hand rule about the local z-axis



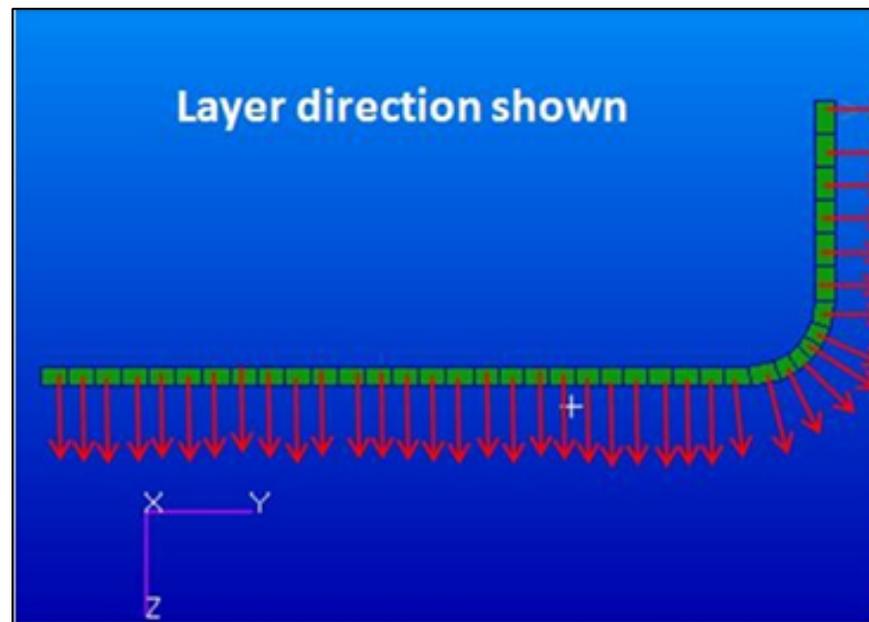
MATERIAL ORIENTATION

- **For Solid Shells (INTi = ASTN): Axis projection is used**
 - The x-axis defined on the CORDM projects onto the element face normal to the element T-direction to create the local x-axis.
 - The local z-axis is in the thickness direction, defined between the centers of the opposing element faces
 - The local y-axis is backed out as the cross product of these two directions (local-z \times local-x)
 - For ply orientation angles, the positive θ direction follows the right hand rule about the local z-axis
 - Taken together, this is very similar to the way MCID and ply angle are defined for the CQUAD4



MATERIAL ORIENTATION

- The element coordinate system (CORDM=-1) is sometimes useful for certain configuration
- The use of material coordinate system will require multiple material coordinate systems for the following structure



SOLID: TRANSVERSE SHEAR AND PEEL STRESS

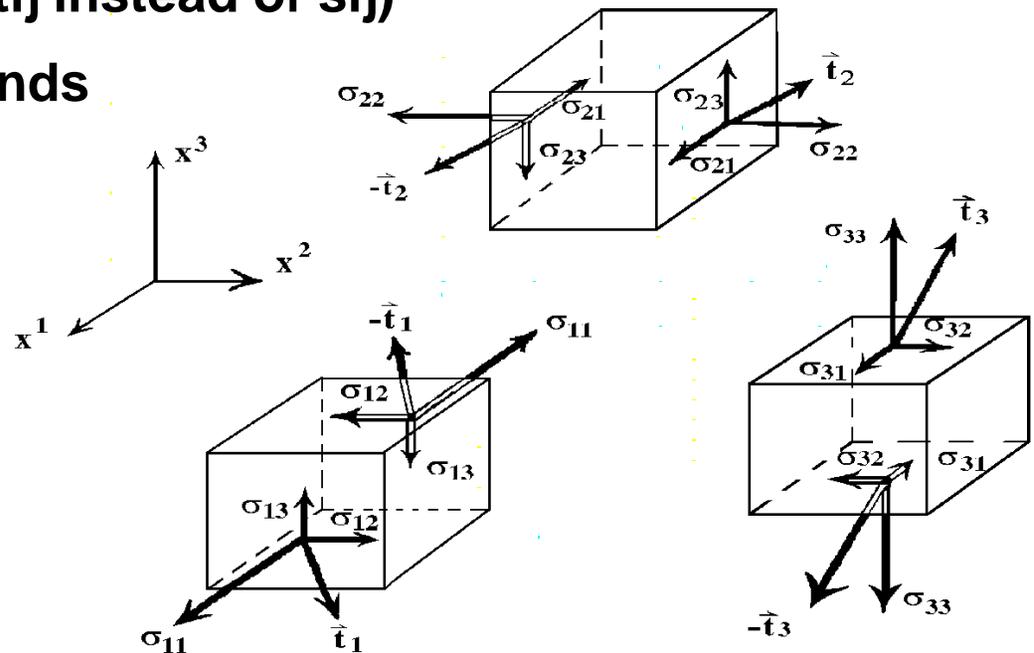
- Interlaminar shear is also known as transverse shear
 - Takes into account “shear deformation” effects
- These out-of-plane stresses can be significant at discontinuities such as holes or free edges
- The non-zero s_{23} and s_{31} tend to slide the plies over each other, thus causing them to delaminate (shear components are commonly referred to as t_{ij} instead of s_{ij})
- The normal stress, s_{33} tends to peel the plies apart

For Equilibrium:

$$\sigma_{12} = \sigma_{21}$$

$$\sigma_{23} = \sigma_{32}$$

$$\sigma_{31} = \sigma_{13}$$

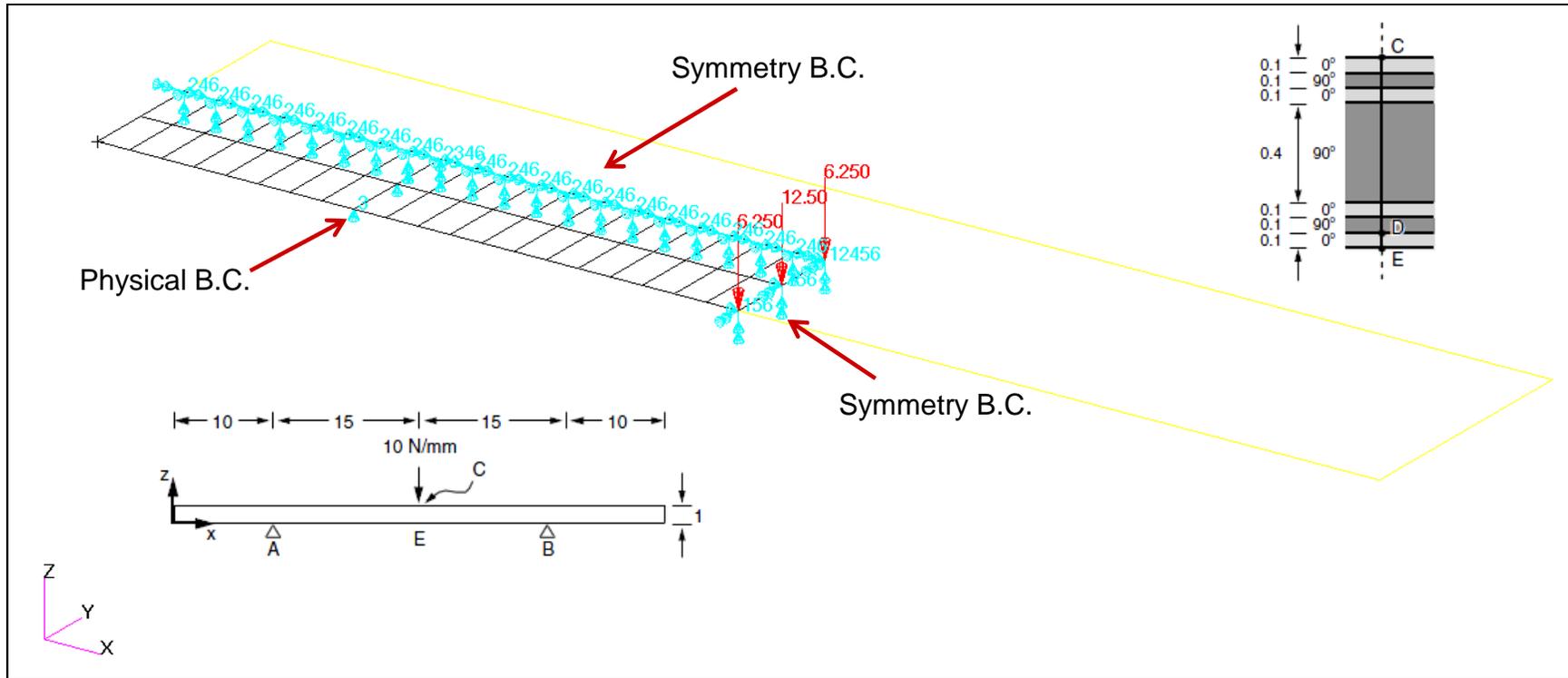


SOLID: TRANSVERSE SHEAR AND PEEL STRESS

- The interlaminar shear and normal stresses are calculated by averaging the stresses in the stacked two layers
- The stresses are transformed into a component tangent to the interface and a component normal to the interface
- The number of output interlaminar shear/normal stresses will always be one fewer than the number of layers...e.g. 5 layers will only have 4 interlaminar shears available, since there are only 4 ply interfaces.

EXAMPLE PROBLEM

- Full Model dimension is 50x10x1 mm
- Use symmetry and model quarter model
 - BC shown for QUAD4 model for clarity
 - BC is similar for HEXA model



EXAMPLE PROBLEM (CONT.)

- **Material properties**

Material Constant	Value
E1	100000. MPa
E2	5000. MPa
E3	5000. MPa
nu12	0.4
nu23	0.3
nu31	0.02
G12	3000. MPa
G13	2000. Mpa
G23	2000. Mpa
Density, ρ	1.E-4

3D ORTHOTROPIC

- We have previously discussed 2D orthotropic (PCOMP/MAT8)
- For 3D orthotropic with PCOMPLS, the MATORT can be used

MATORT	MID	E1	E2	E3	N12	NU23	NU31	RHO	
	G12	G23	G31	A1	A2	A3	TREF	GE	

- For this problem, the MATORT will look something like

MATORT	1	100000.	5000.	5000.	0.4	0.3	.02	1.E-4	
	3000.	2000.	2000.						

ORTHOTROPIC MATERIAL - MATORT

- Define 3D Orthotropic Properties

The screenshot shows the software's 'Properties' tab with 'Orthotropic' selected. The 'Input Options' dialog box is open, showing the 'Constitutive Model' set to 'Linear Elastic'. The 'Property Name' and 'Value' table is as follows:

Property Name	Value
Elastic Modulus 11 =	100e3
Elastic Modulus 22 =	5e3
Elastic Modulus 33 =	5e3
Poisson Ratio 12 =	0.4
Poisson Ratio 23 =	0.3
Poisson Ratio 31 =	0.02
Shear Modulus 12 =	3e3
Shear Modulus 23 =	2e3
Shear Modulus 31 =	2e3
Density =	1e-4
Thermal Expan. Coeff 11 =	
Thermal Expan. Coeff 22 =	
Thermal Expan. Coeff 33 =	
Structural Damping Coeff =	
Reference Temperature =	

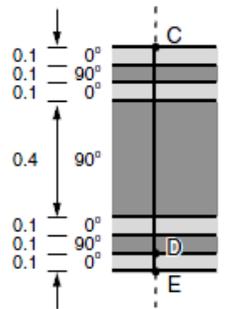
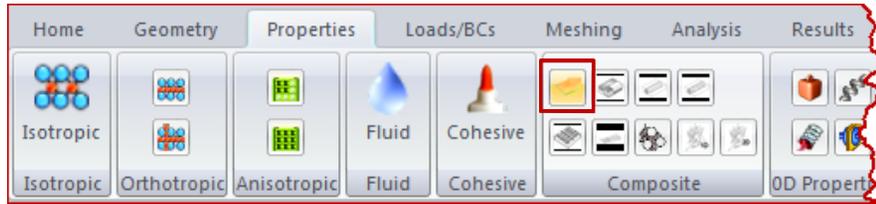
The material is orthotropic, with the following properties:

$$\begin{array}{lll}
 E_1 = 100GPa & \nu_{12} = 0.4 & G_{12} = 3GPa \\
 E_2 = 5GPa & \nu_{23} = 0.3 & G_{13} = 2GPa \\
 E_3 = 5GPa & \nu_{31} = 0.02 & G_{23} = 2GPa
 \end{array}$$

The 'Materials' panel shows the following configuration:

- Action: Create
- Object: 3d Orthotropic
- Method: Manual Input
- Existing Materials: (empty list)
- Filter: *
- Material Name: ortho1
- Description: (empty text area)
- Buttons: Input Properties ..., Change Material Status ..., Apply

CREATE SOLID LAYUP



all dimensions in mm

Laminated Composite

Stacking Sequence Convention: Total - %thicknesses Offset: _____

Stacking Sequence Definition

Input Data Auto Highlight Import/Export...

	Material Name	% Thickness	Orientation	Global Ply ID
1	ortho1	1.000000E-3	0.000000E+0	
2	ortho1	9.999000E+0	0.000000E+0	
3	ortho1	1.000000E+1	9.000000E+1	
4	ortho1	1.000000E+1	0.000000E+0	
5	ortho1	4.000000E+1	9.000000E+1	
6	ortho1	1.000000E+1	0.000000E+0	
7	ortho1	1.000000E+1	9.000000E+1	
8	ortho1	9.999000E+0	0.000000E+0	
9	ortho1	1.000000E-3	0.000000E+0	

Set Thickness = _____ for ALL Layers of "ortho1"

% Thickness of Stacking Sequence = 100. Plies in Stacking Sequence = 9

Delete Selected Rows Insert Rows Above Below

Show Laminate Properties... Clear Databoxes

Materials

Action: Create Object: Composite Method: Laminate

Existing Materials

* Filter

lam1
ortho1

Laminated Composites

* Filter

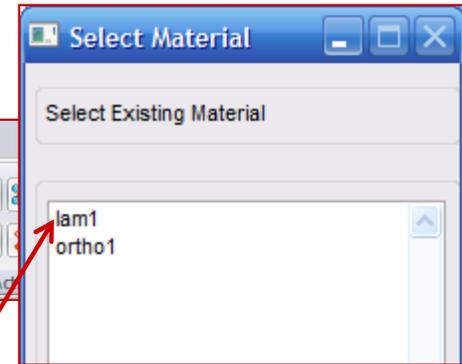
lam1

Material Name: lam1

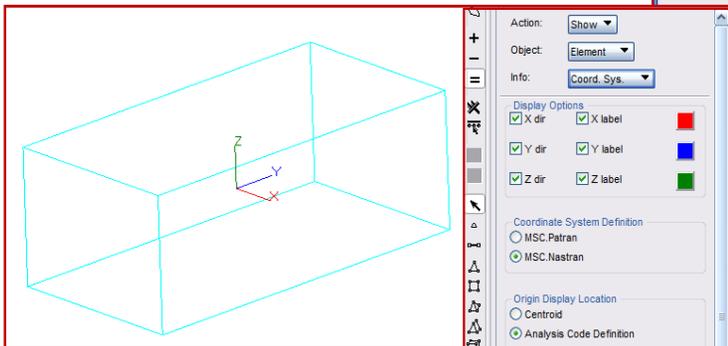
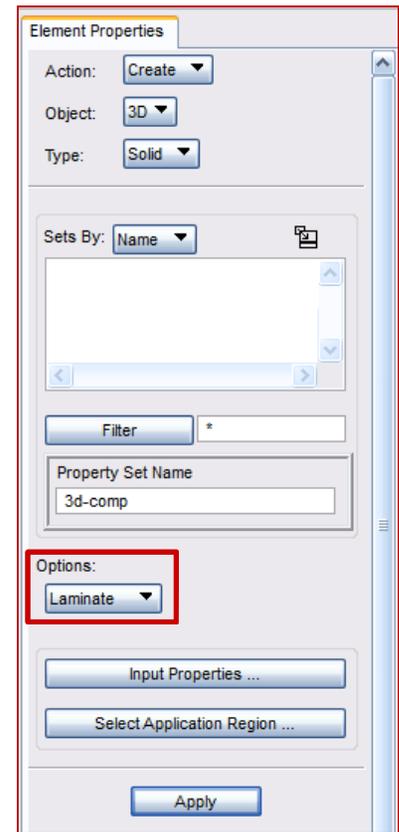
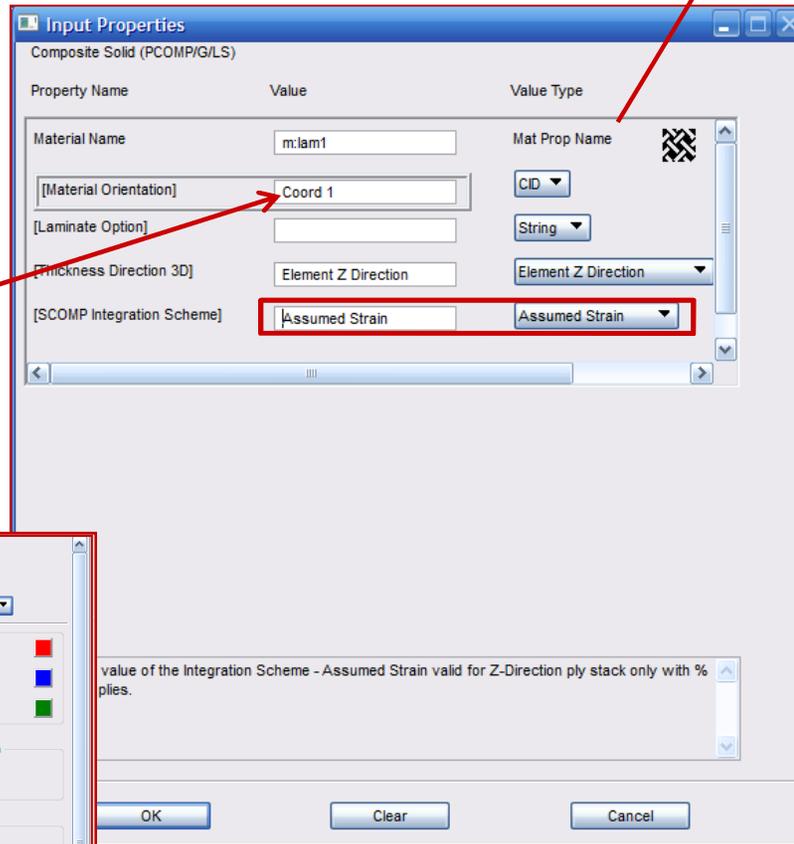
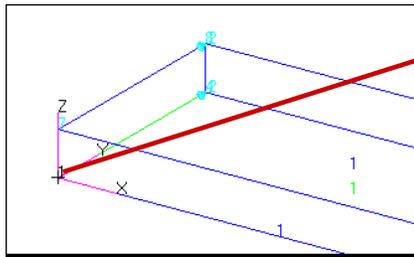
Material Description:

-Apply- Reset

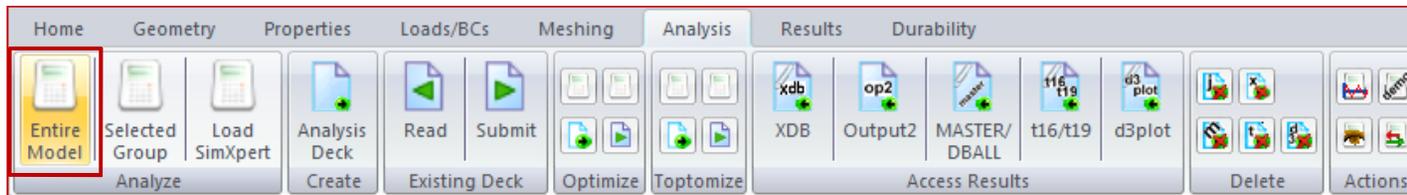
CREATE 3D SOLID SHELL- PCOMPLS



- **PCOMPLS**
 - 3D Composite
 - Laminate
 - Orientation
 - Assumed Strain



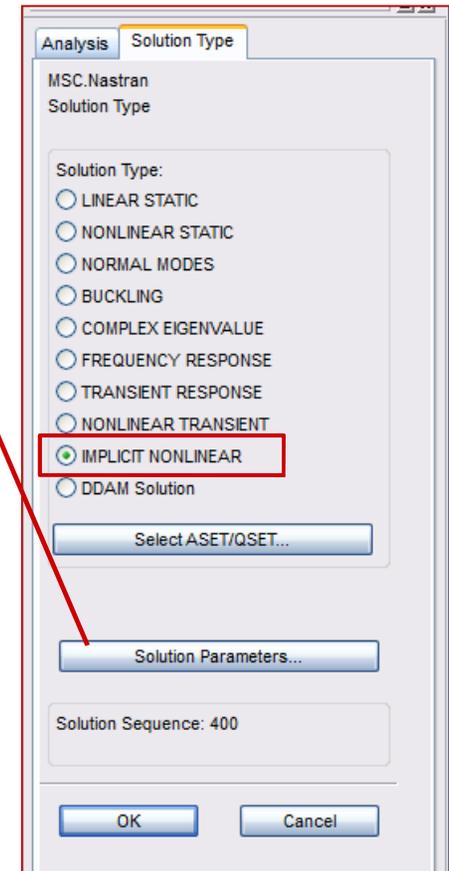
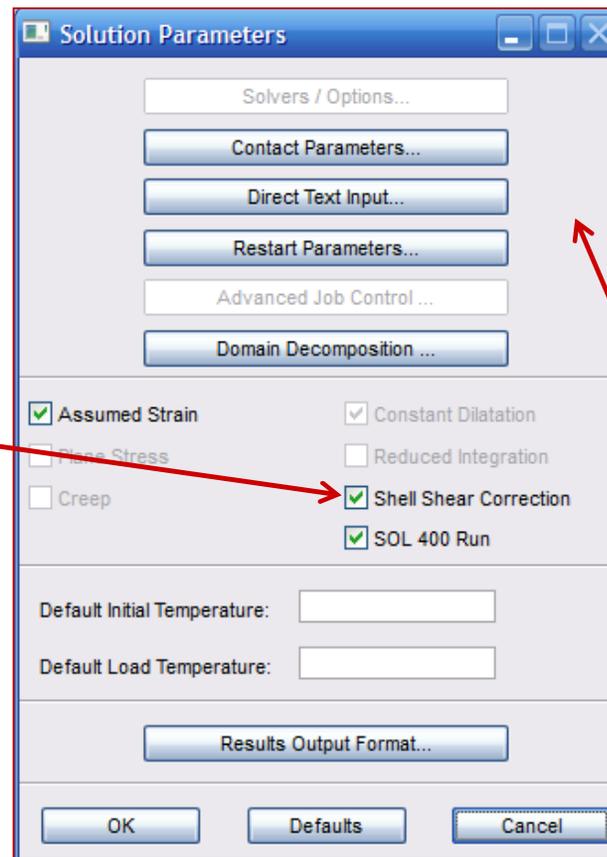
JOB SETUP



- **Analysis Setup**
 - Solution Type
 - Implicit Nonlinear
 - Solution Parameters

NLMOPTS, TSHEAR, TSHEAR

This is needed to obtain correct parabolic shear distribution through the thickness of the shell

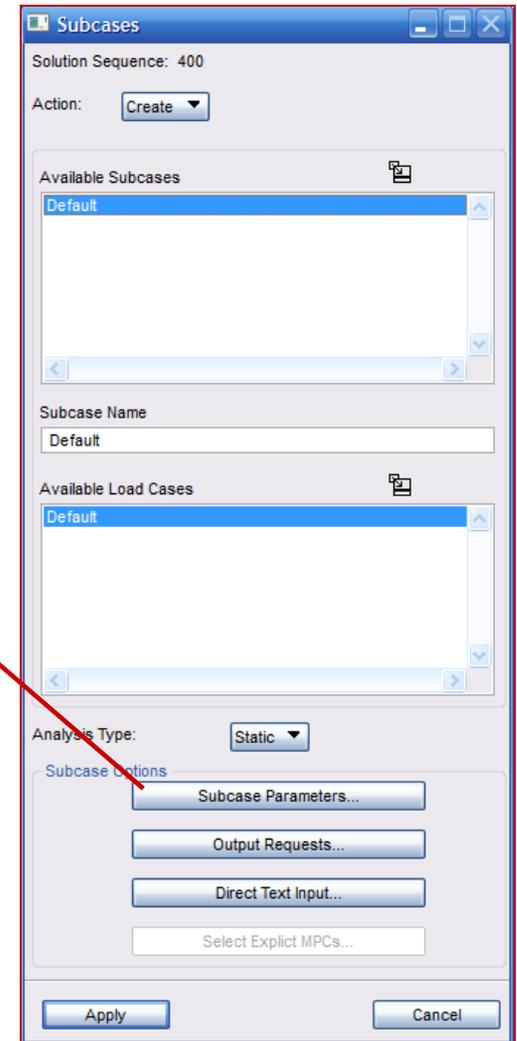
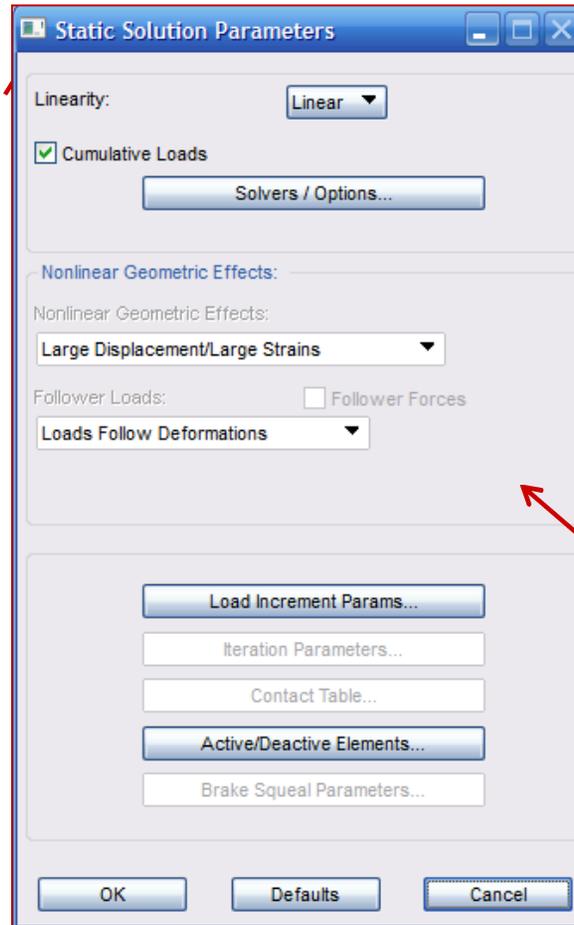


JOB SETUP

- Subcase Parameters
 - Set “Linear”

```
SUBCASE 1
STEP 1
TITLE=This is a default subcase.
ANALYSIS = LNSTATICS
SPC = 2
LOAD = 2
DISPLACEMENT(SORT1, REAL)=ALL
SPCFORCES(SORT1, REAL)=ALL
STRESS(SORT1, REAL, VONMISES, BILIN)=ALL
```

Generates



JOB SETUP

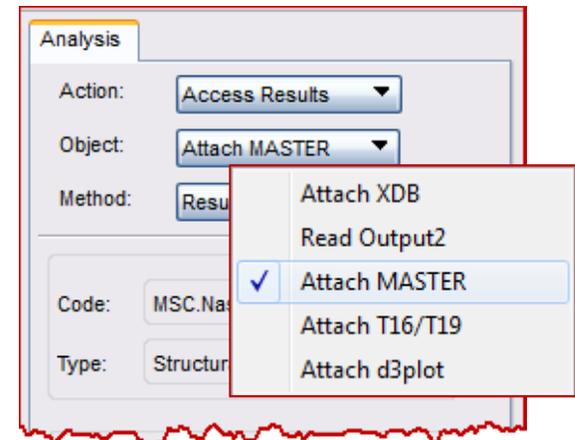
- **Getting the Results:**

- Not available via xdb, need MASTER/DBALL
- Run with SCR=POST
 - 316=19 is a “data recovery dball only” – not as big as a restart
 - Similar to XDB indexed data, but modern and go-forward
 - SCR=POST is equiv to SYSTEM(316)=19

Creating the Job



Accessing the Results



INDEX (316)

Indexes and/or saves a minimum set of data blocks to the database needed to for postprocessing in Patran or the toolkit. This cell must be used with scratch=no on the nastran command. This cell has several options which are set by adding the following values:

Value:	Action:
1	Index IFP data blocks.
2	Index OFP data blocks.
4	Save above data blocks to the MASTER dbset.
16	Save above data blocks to the DBALL dbset.

For example, INDEX=7 will index the IFP and OFP data blocks and save them to the MASTER dbset. (Scratch=post is equivalent to scratch=no and INDEX=19.)

```

$ NASTRAN input file created by the Patran 2008r1 (MD Enabled) input
$ file translator on August 09, 2008 at 13:24:00.
$ Direct Text Input for Nastran System Cell Section
NASTRAN SYSTEM(316)=19
$ Replace the 19 in the above line with 7
$ to get a restart DBALL instead of what is currently written
$ Direct Text Input for File Management Section
$ Implicit Nonlinear Analysis
SOL 400
$ Direct Text Input for Executive Control
CEND
    
```

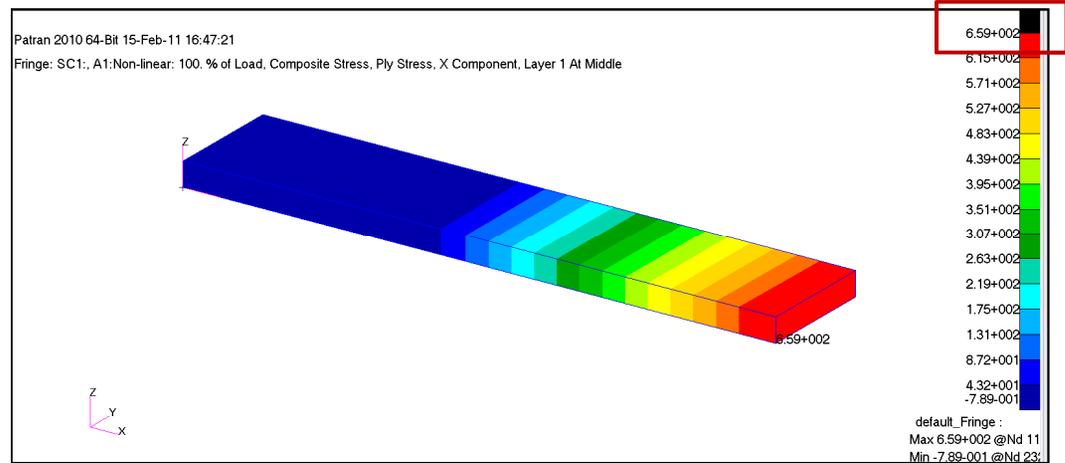
RESULTS

- Results:
 - 2.5 x 2.5 mesh
 - Stresses at center of layer

LOAD STEP = 1.00000E+00

DISPLACEMENT VECTOR

POINT ID.	TYPE	T1	T2	T3	R1
1	G	-5.095006E-02	-1.928312E-06	1.017965E+00	0.0
11	G	0.0	1.563930E-03	-1.063035E+00	0.0



0 LOAD STEP = 1.00000E+00 SUBCASE 1 STEP 1

STRESSES AND STRAINS FOR LAYERED COMPOSITE ELEMENTS

ELEMENT ID	PLY ID	INTEG. POINT ID	S11	S22	S33	S12	S23	S31
80	1	1	6.589E+02	1.191E+01	7.981E-01	3.208E-01	-4.668E-02	-5.014E+00
			6.598E-03	-3.022E-04	-3.190E-03	1.069E-04	-3.241E-05	-2.830E-03
	2	1	5.931E+02	1.066E+01	5.052E-01	3.109E-01	-4.668E-02	-5.014E+00
			5.886E-03	-2.716E-04	-2.911E-03	1.036E-04	-3.241E-05	-2.830E-03
	3	1	-1.501E+01	2.088E+01	-5.794E+00	-2.912E-01	5.014E+00	-4.668E-02
			-2.104E-04	4.583E-03	-2.351E-03	-9.707E-05	2.830E-03	-3.241E-05
	4	1	3.299E+02	5.652E+00	-6.663E-01	2.715E-01	-4.668E-02	-5.014E+00
			3.279E-03	-1.492E-04	-1.792E-03	9.050E-05	-3.241E-05	-2.830E-03
	5	1	-6.959E-01	-5.604E-01	-2.150E+00	-2.222E-01	5.014E+00	-4.668E-02
			3.881E-06	1.967E-05	-3.935E-04	-7.407E-05	2.830E-03	-3.241E-05
	6	1	-3.281E+02	-6.857E+00	-3.595E+00	1.729E-01	-4.668E-02	-5.014E+00
			-3.240E-03	1.569E-04	1.005E-03	5.764E-05	-3.241E-05	-2.830E-03
	7	1	1.362E+01	-2.200E+01	1.495E+00	-1.532E-01	5.014E+00	-4.668E-02
			2.182E-04	-4.543E-03	1.564E-03	-5.106E-05	2.830E-03	-3.241E-05
	8	1	-5.914E+02	-1.186E+01	-4.767E+00	1.335E-01	-4.668E-02	-5.014E+00
			-5.847E-03	2.794E-04	2.124E-03	4.449E-05	-3.241E-05	-2.830E-03
	9	1	-6.572E+02	-1.311E+01	-5.060E+00	1.236E-01	-4.668E-02	-5.014E+00
			-6.499E-03	3.100E-04	2.403E-03	4.121E-05	-3.241E-05	-2.830E-03

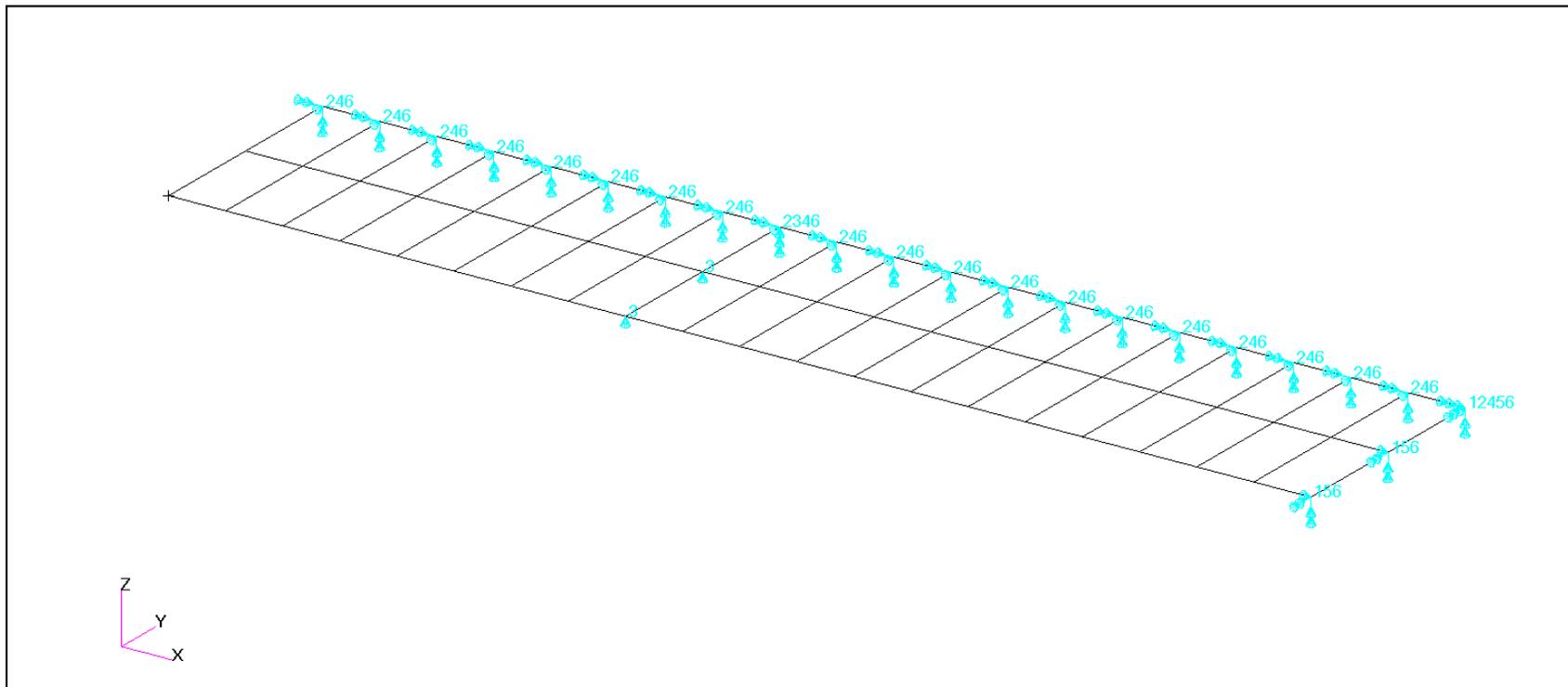
Note that stresses are obtained at the center of the element (for solid shell) & center of the thickness for that layer

THREE DIFFERENT RUN TECHNIQUES

- **Run model using 3 different ways**
 - 2D Shell
 - Solid
 - Layered solid
 - Solid shell

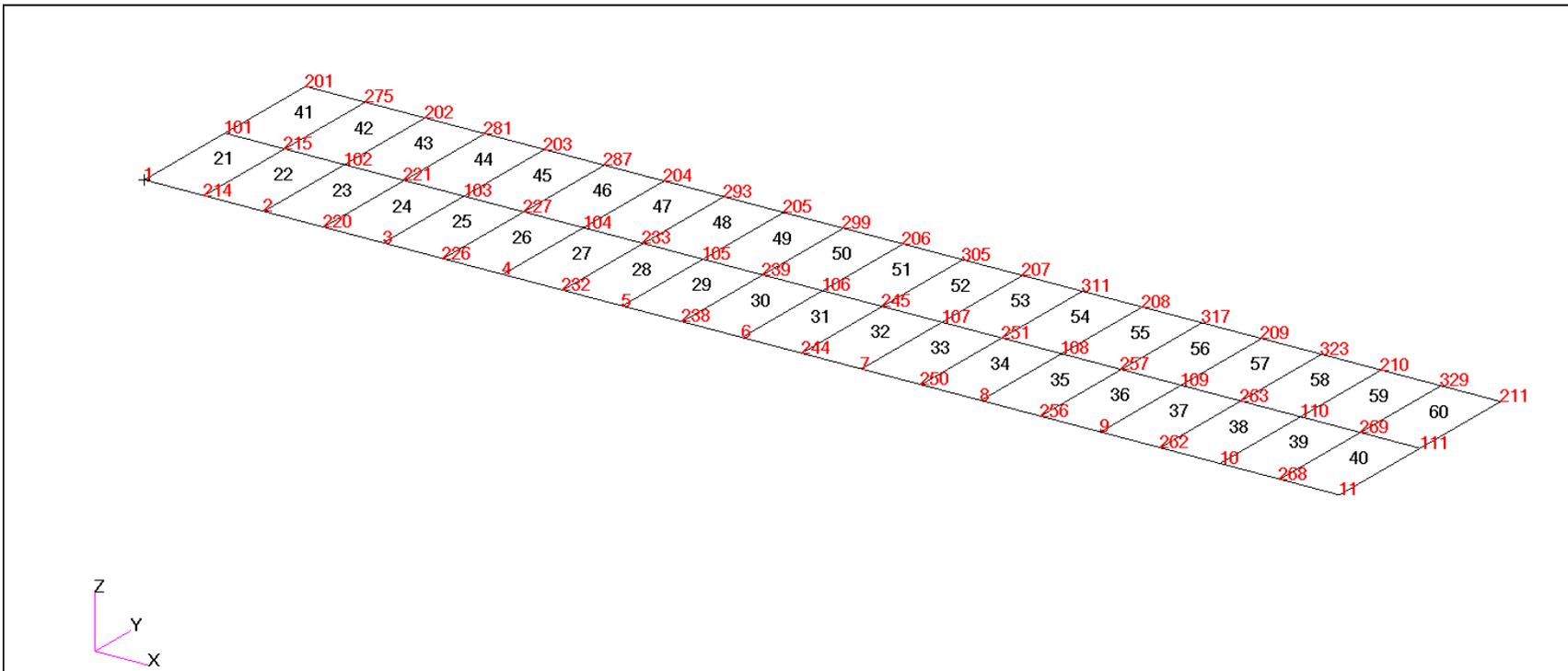
SHELL MODEL

- Boundary Condition (1/4 model)



SHELL MODEL

- Shell Model (1/4 model due to symmetry)



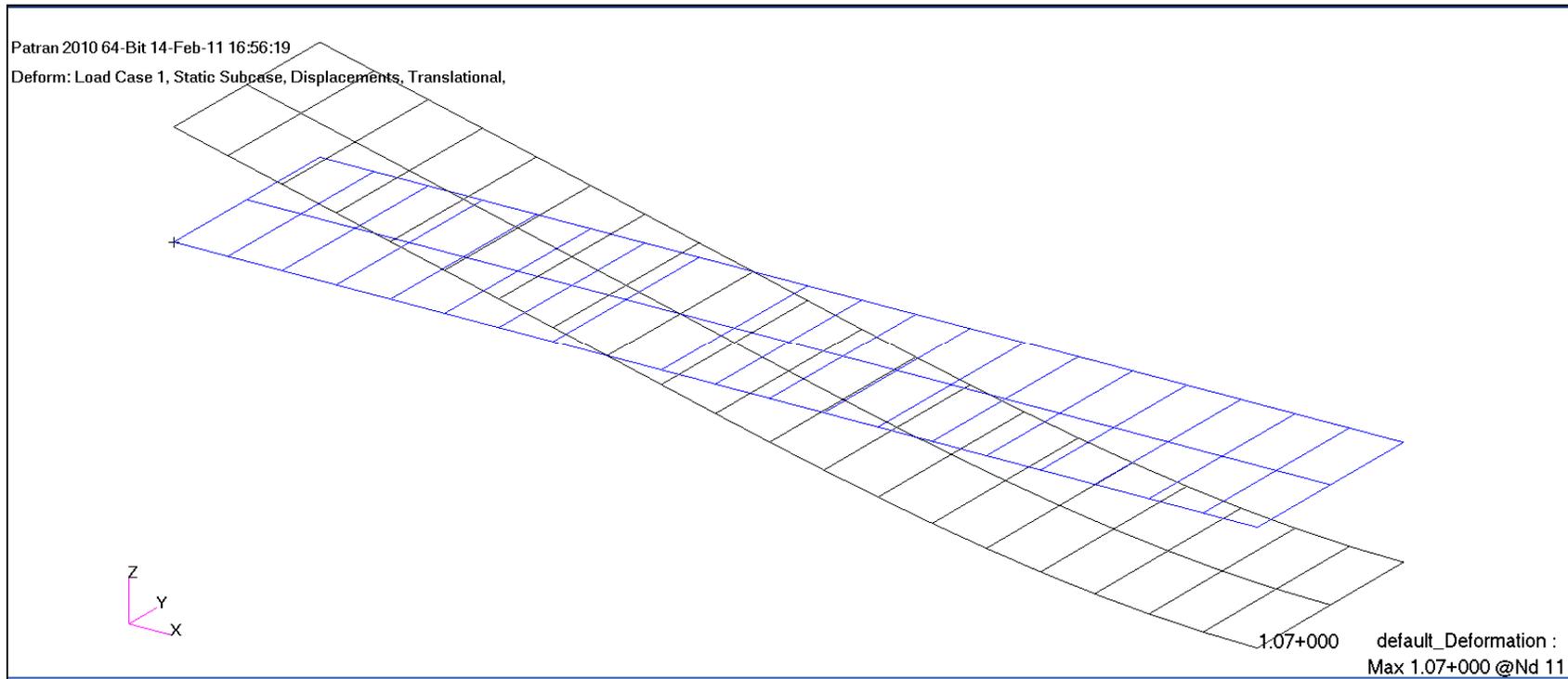
SHELL MODEL

- Abridged Input File

```
SOL 101
$
CEND
.
.
$
BEGIN BULK
PARAM POST 0
PARAM PRTMAXIM YES
PCOMP 1 0. 0.
      1 .00001 0. YES
      1 .09999 0. YES
      1 .1 90. YES
      1 .1 0. YES
      1 .4 0. YES
      1 .1 0. YES
      1 .1 90. YES
      1 .09999 0. YES
      1 .00001 0. YES
$
$
$ Pset: "2D_1" will be imported as: "pshell.1"
CQUAD4 21 1 1 214 215 101
.
.
$ Referenced Material Records
$ Description of Material : Composite Material Created by MIFRDR
MAT8 1 100000. 5000. .4 3000. 3000. 2000. 1.-4
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
.
.
$
ENDDATA
```

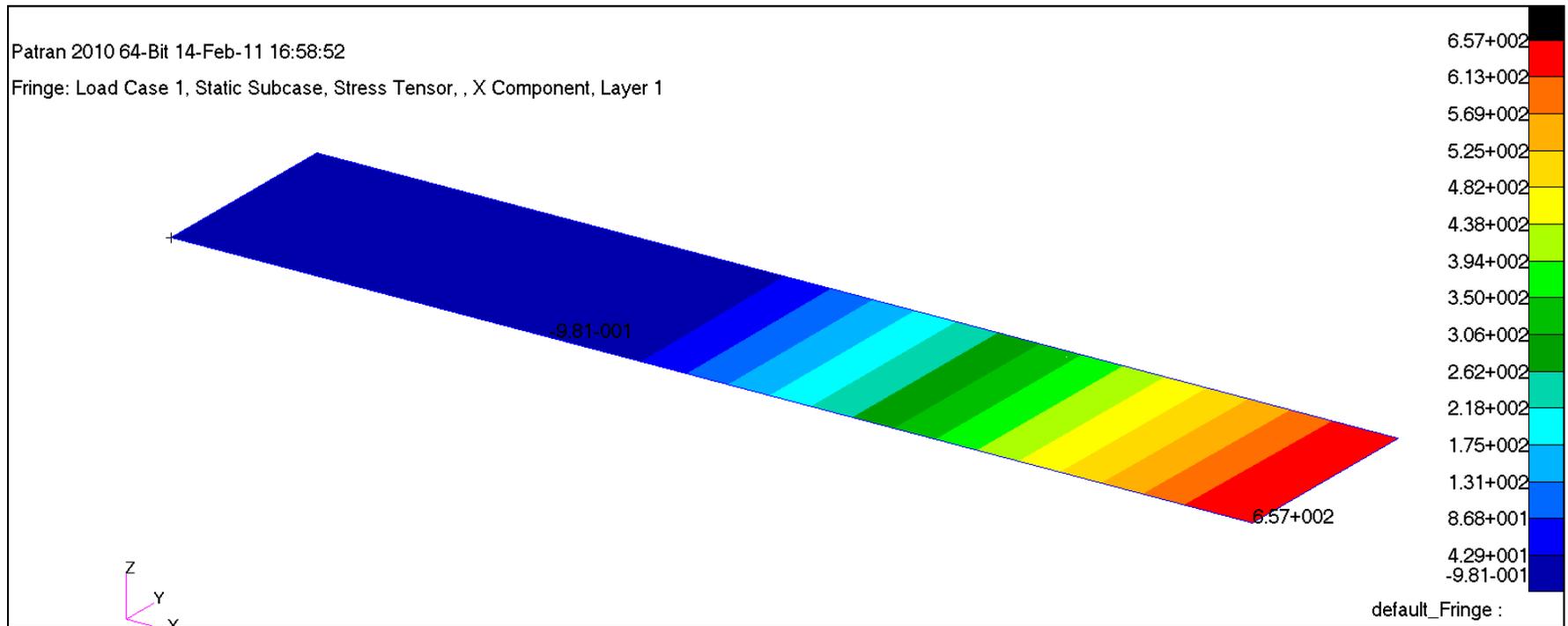
SHELL MODEL

- Deflection Plot



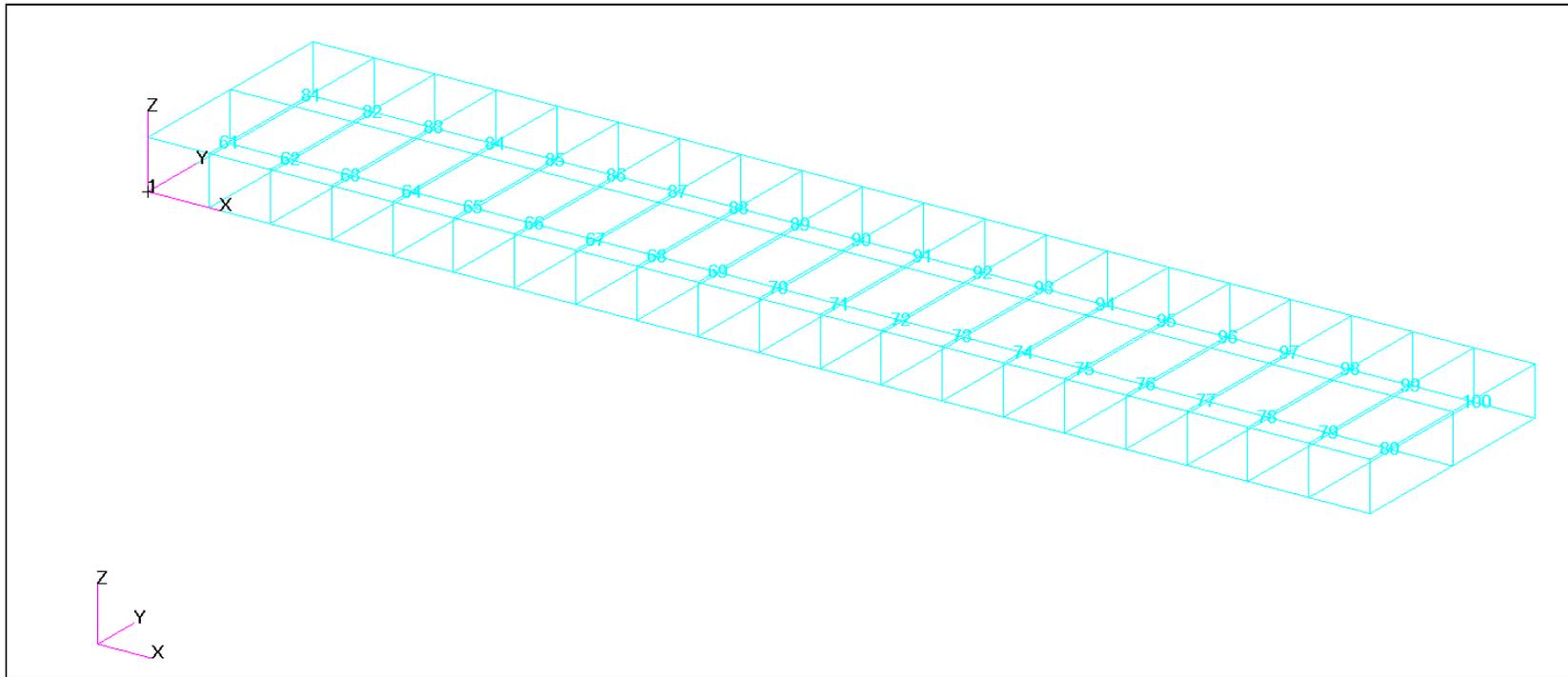
SHELL MODEL

- Stress (σ_x)



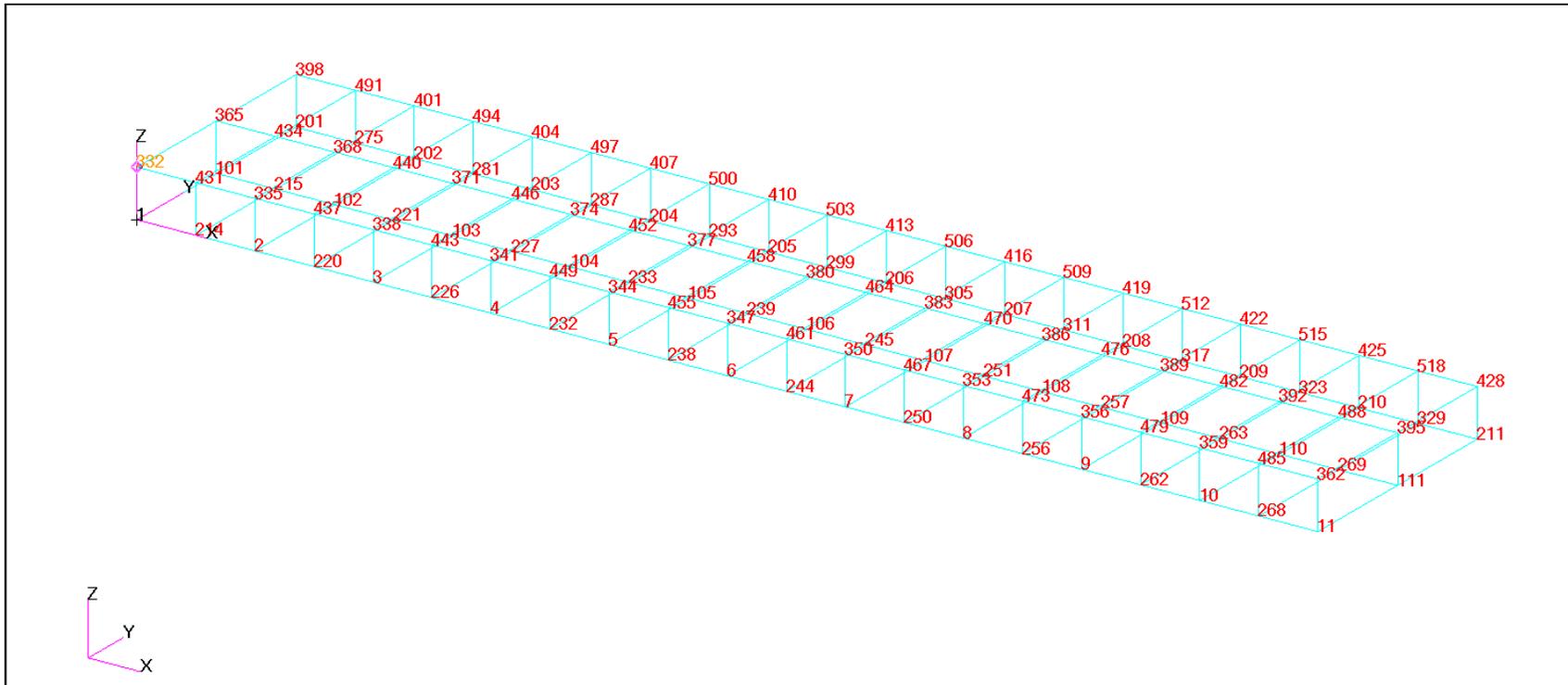
SOLID MODEL

- Solid Model (Elem ID)



SOLID MODEL

- Solid Model (Grid IDs)



LAYERED SOLID

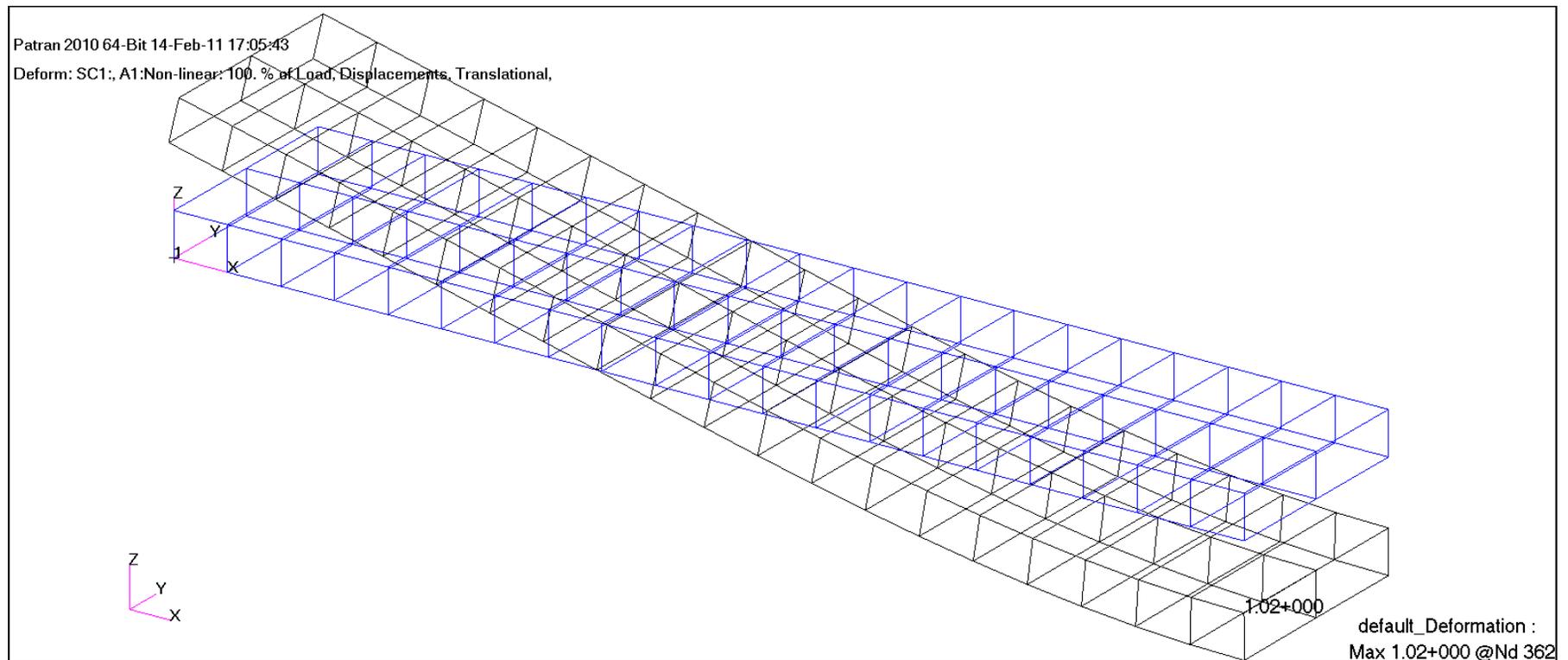
- Abridged Input for Layered Solid

```

SOL 400
$
$ Direct Text Input for Executive Control
CEND
TITLE = ug06 - 3-D Laminated Strip under 3 point bending
$ Direct Text Input for Global Case Control Data
SUBCASE 1
  TITLE=Linear composite brick element
  ANALYSIS=LNSTATICS
  NLPARM = 1
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
NLMOPTS TSHEAR TSHEAR
PARAM PRITMAXIM YES
NLPARM,1,1,,ITER,1,1,UP,NO
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : 3D_1
PCOMPLS 1 -1 1
  C8 SLCOMP L
  1 1 .001 0.
  2 1 .099 0.
  3 1 .1 90.
  4 1 .1 0.
  5 1 .4 90.
  6 1 .1 0.
  7 1 .1 90.
  8 1 .099 0.
  9 1 .001 0.
CORD2R 1 0. 0. 0. 0. 0. 1.
  1. 0. 0.
$
CHEXA 61 1 1 214 215 101 332 431
  434 365
.
MATORT 1 100000. 5000. 5000. .4 .3 .02 1.-4
  3000. 2000. 2000.
-1
GRID 1 0. 0. 0.
.
LOAD 2 1. 1. 1 1. 4
FORCE 1 362 0 6.25 0. 0. -1.
FORCE 1 428 0 6.25 0. 0. -1.
FORCE 4 395 0 12.5 0. 0. -1.
ENDDATA
  
```

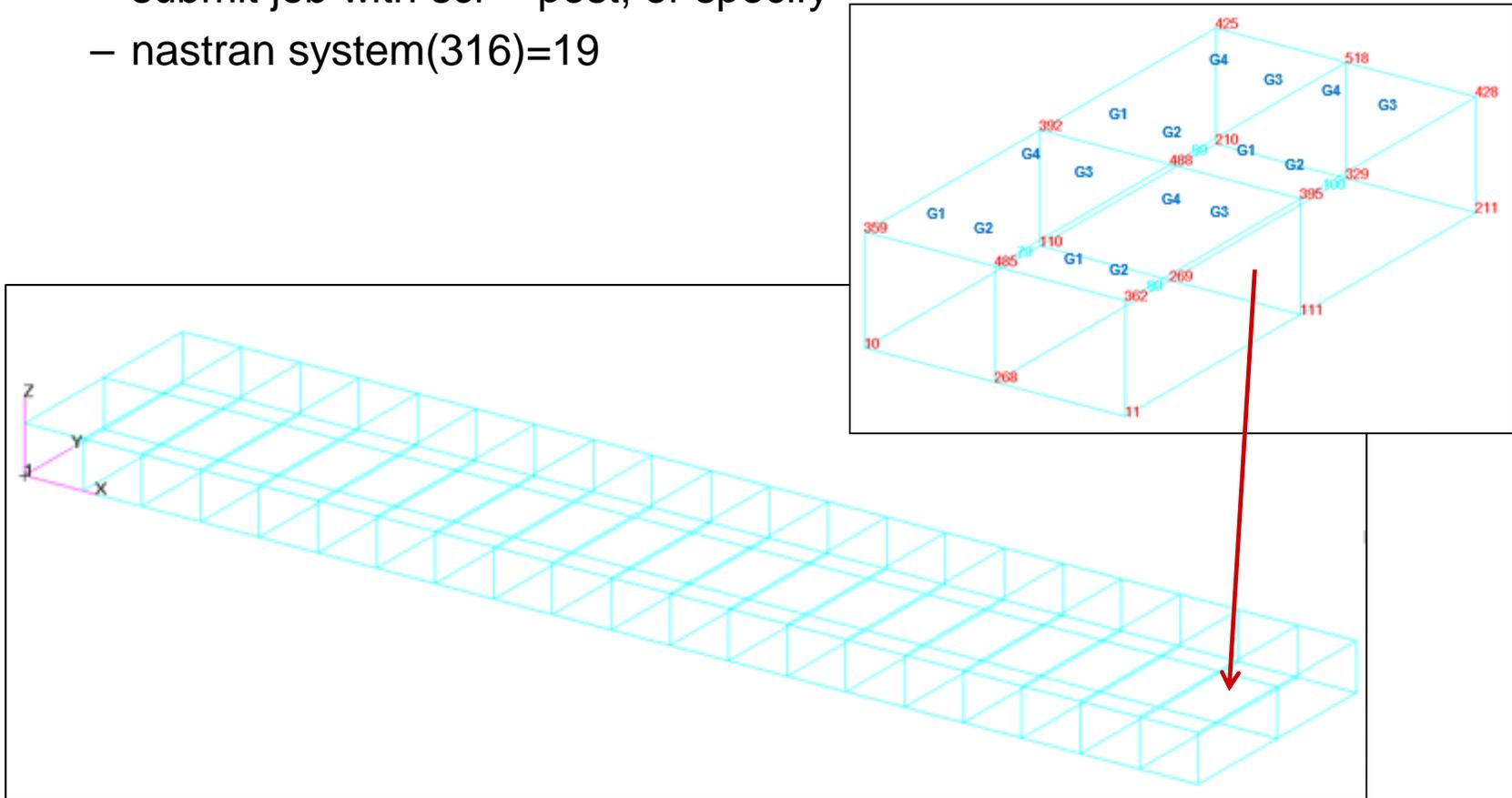
LAYERED SOLID

- Deflected shape



LAYERED SOLID

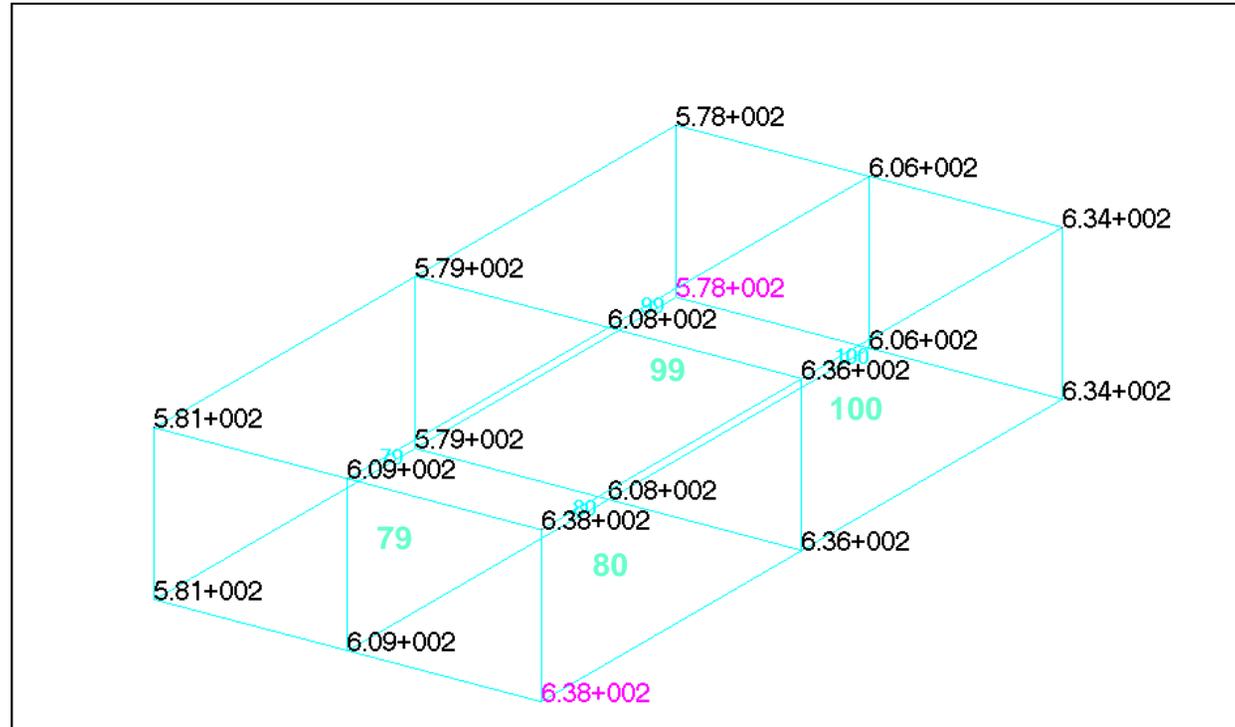
- Stress (σ_x)
- For solid shell, the stresses are obtained from the .master
 - submit job with scr = post, or specify
 - nastran system(316)=19



LAYERED SOLID

- σ_x at ply# 1

Patran



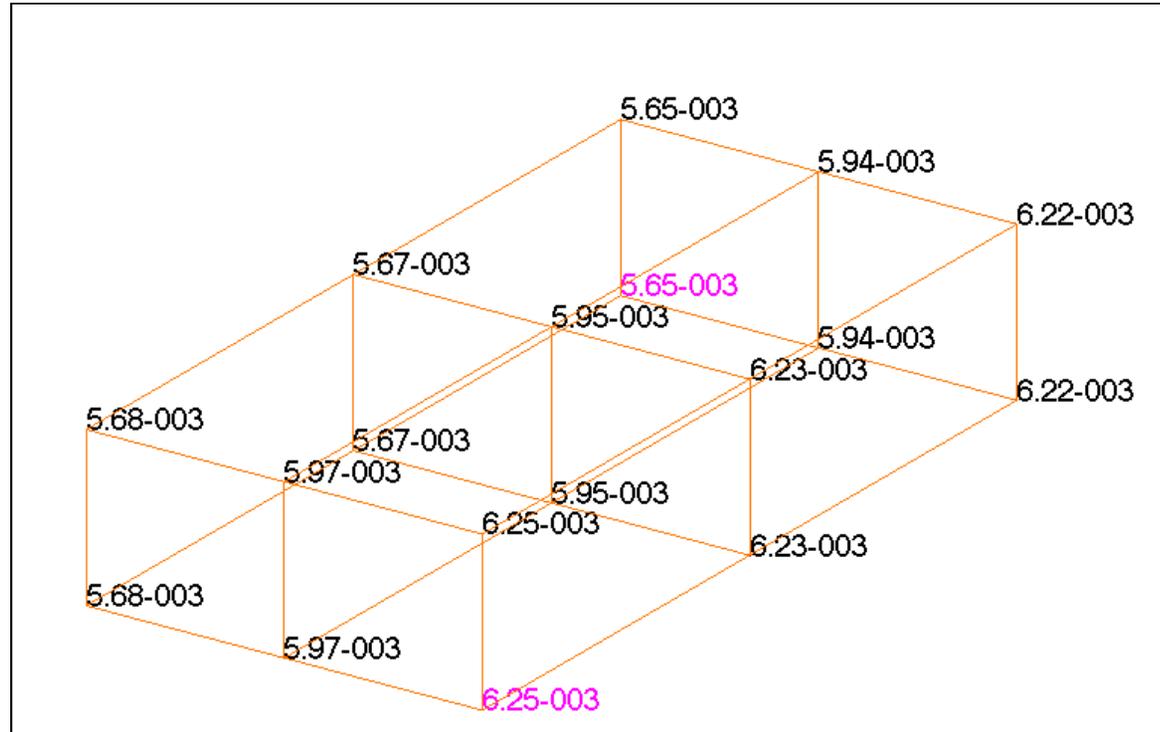
MSC Nastran F06

ELEM ID	PLY ID	INT PT 1	INT PT 2	INT PT 3	INT PT 4
79	1	581.8	582.2	579.4	579.9
80	1	640.2	637.9	637.1	634.8
99	1	578.2	578.7	577	577.5
100	1	635.7	633.3	634.5	632.1

LAYERED SOLID

- ϵ_x at ply # 1

Patran



MSC Nastran F06

ELEM ID	PLY ID	INT PT 1	INT PT 2	INT PT 3	INT PT 4
79	1	5.69E-03	5.69E-03	5.67E-03	5.67E-03
80	1	6.27E-03	6.27E-03	6.24E-03	6.24E-03
99	1	5.66E-03	5.66E-03	5.65E-03	5.65E-03
100	1	6.22E-03	6.22E-03	6.21E-03	6.21E-03

SOLID SHELL

- Abridged Input for Solid Shell

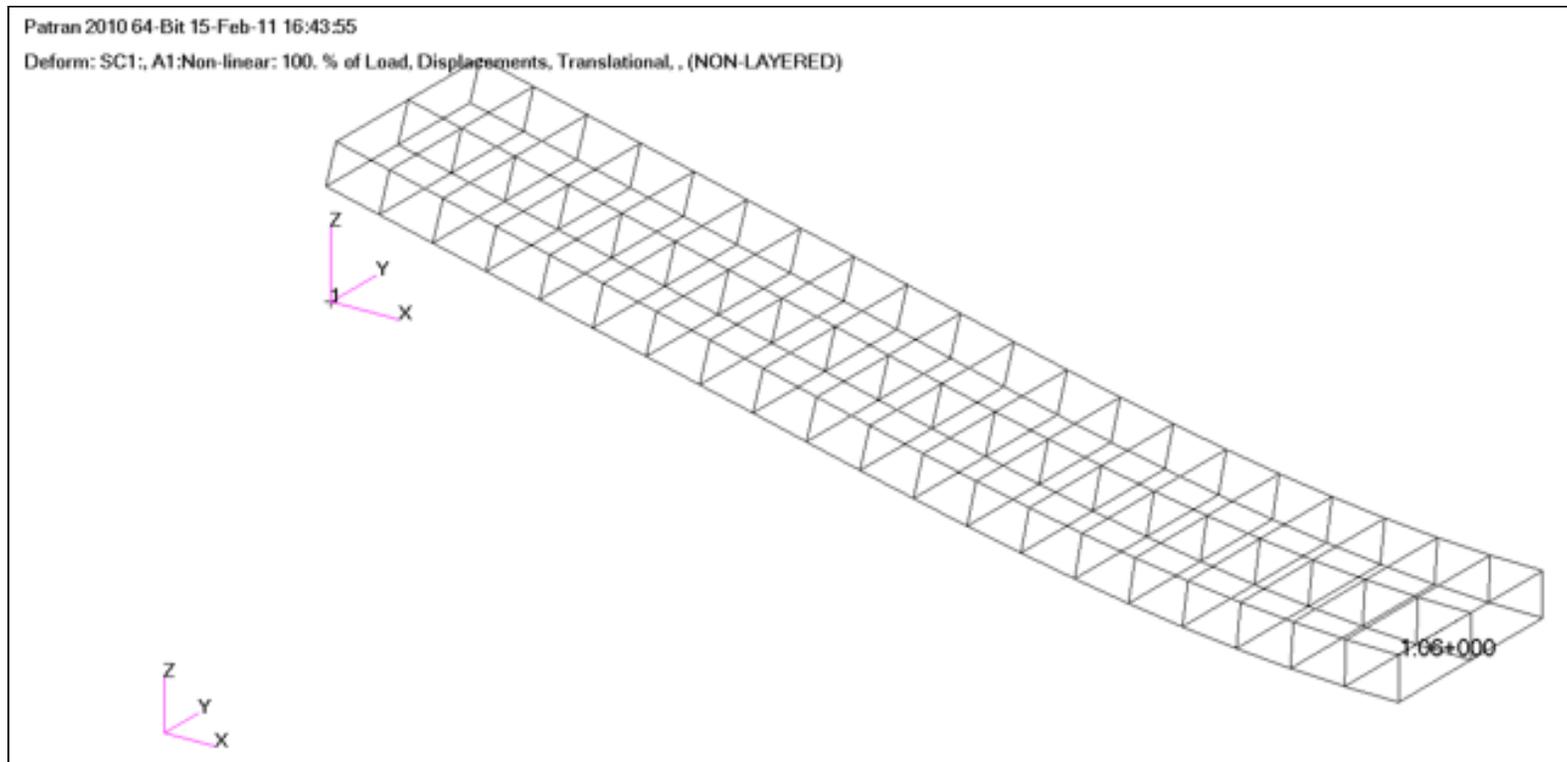
```

$
SOL 400
$
CEND
$
SUBCASE 1
  TITLE=Solid shell element
  ANALYSIS=LNSTATICS
  NLPARM = 1
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
$
BEGIN BULK
param,post,0
NLMOPTS,TSHEAR,TSHEAR
NLPARM,1,1,,ITER,1,1,UP,NO
$ Elements and Element Properties for region : 3D_1
PCOMPLS 1 -1 1
  C8      SLCOMP  ASTN
  1      1      .00001  0.
  2      1      .09999  0.
  3      1      .1      90.
  4      1      .1      0.
  5      1      .4      90.
  6      1      .1      0.
  7      1      .1      90.
  8      1      .09999  0.
  9      1      .00001  0.
MATORT  1      100000. 5000. 5000. .4 .3 .02 1.-4
        3000. 2000. 2000.
$
CORD2R  1      0. 0. 0. 0. 0. 0. 1.
        1. 0. 0.
$
CHEXA  61      1      1      214 215 101 332 431
        434 365
.
$
GRID  105      10. 2.5 0.
.
LOAD  2      1. 1. 1 1. 4
$
.
FORCE  1      362 0 6.25 0. 0. -1.
FORCE  1      428 0 6.25 0. 0. -1.
FORCE  4      395 0 12.5 0. 0. -1.
ENDDATA

```

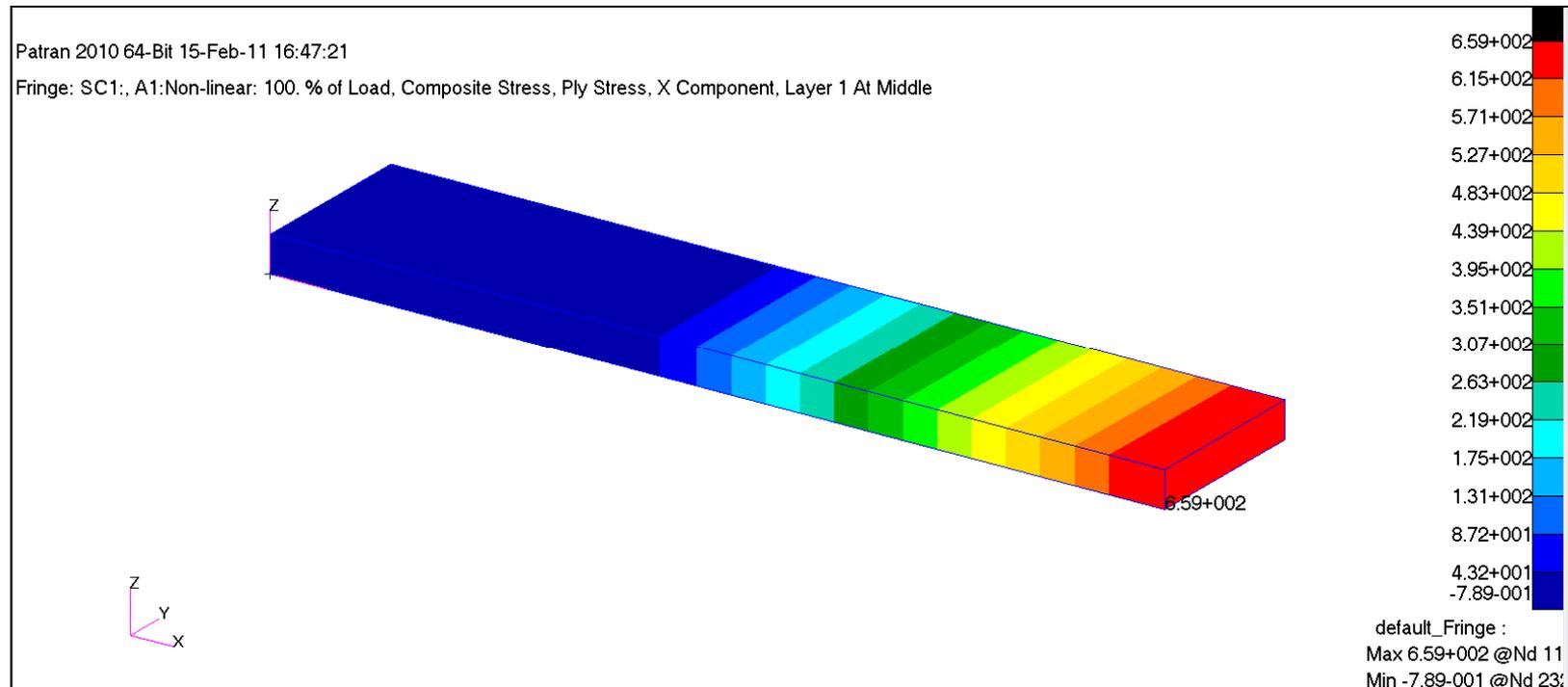
SOLID SHELL

- Deflected Shape



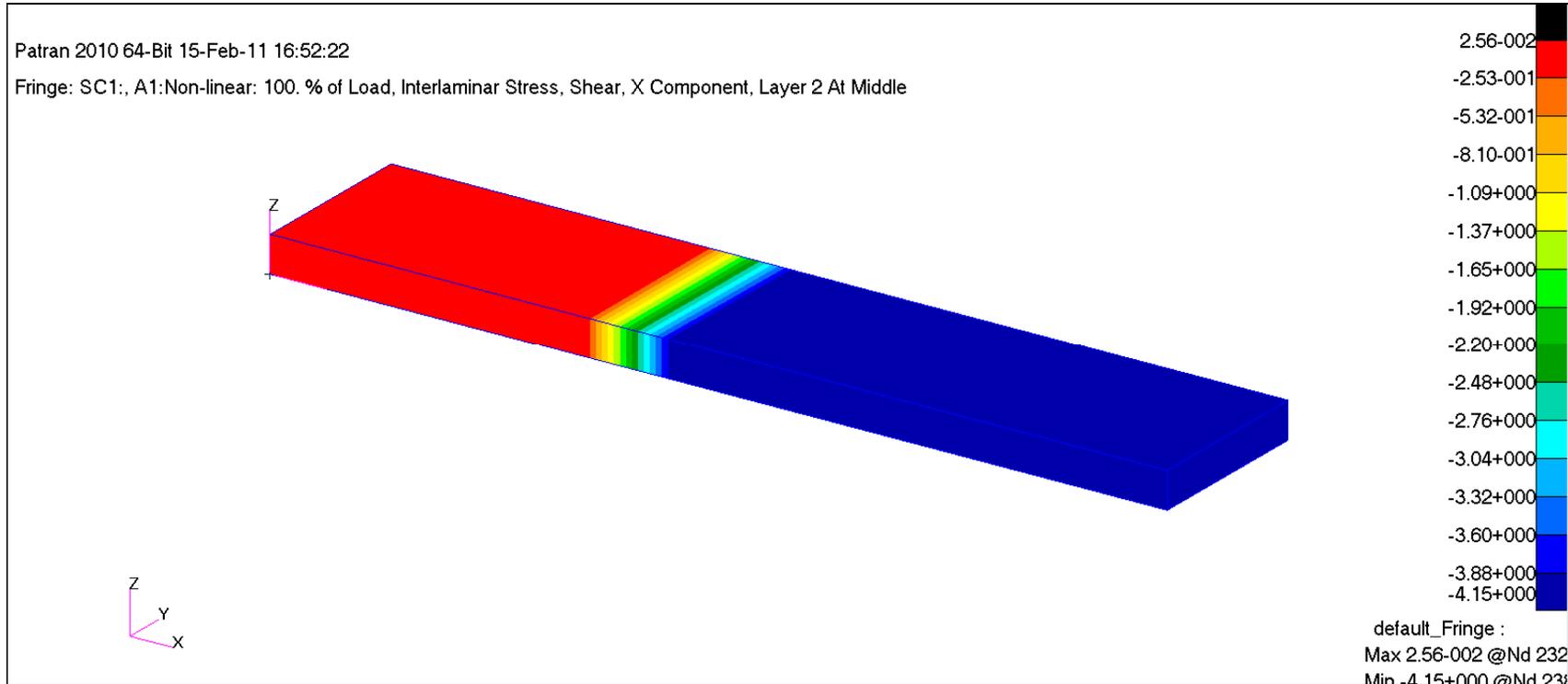
SOLID SHELL

- Stresses (σ_x)
- For solid shell, the stresses are obtained from the .master
 - submit job with scr = post, or specify
 - nastran system(316)=19



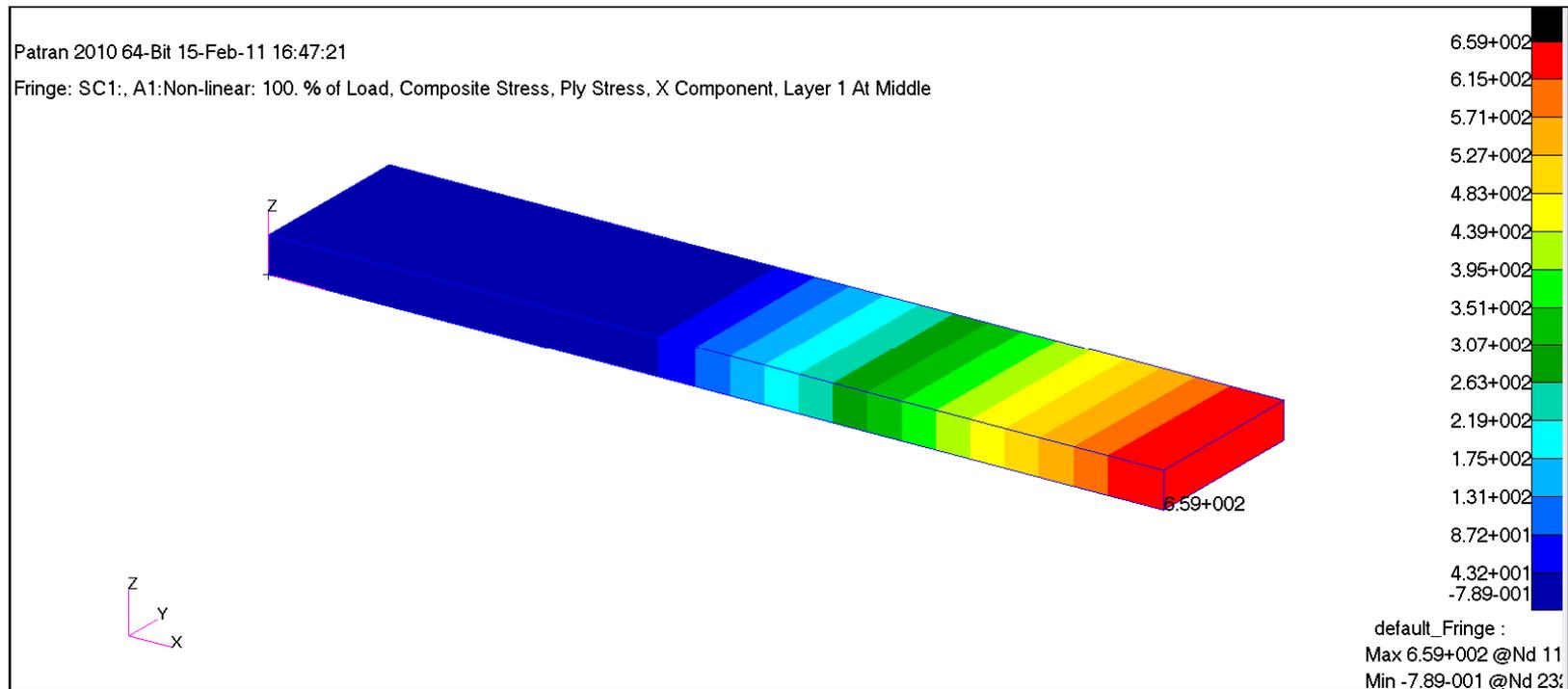
SOLID SHELL

- Interlaminar Stresses
- For solid shell, the stresses are obtained from the .master
 - submit job with scr = post, or specify
 - nastran system(316)=19



EXERCISE

- Perform workshop 3 “Layup Solid Composite Shell”





SECTION 5

COMPOSITE POST PROCESSING & FAILURE THEORIES

SUPPORTED SOLUTIONS IN MSC NASTRAN

- **Composite elastic properties are supported in all solutions sequences**
- **Composite ply data recovery is supported in the following solutions**
 - SOL 101 – Linear Statics
 - SOL 103 – Normal Modes
 - SOL 105 – Linear Buckling
 - SOL 106 – Nonlinear Statics
 - Large displacement
 - Temperature dependent materials
 - SOL 600 – Nonlinear Statics and Dynamics
 - Strain dependent materials
 - SOL 400
- **Basic laminated plate theory assumptions hold when interpreting the results:**
 - Linear strain distribution through the thickness from membrane and bending loads
 - Perfectly bonded plies where the bond thickness is zero and no slippage between plies

MSC NASTRAN CALCULATION OF PLY RESULTS

- Element strains are transformed to ply strains in the element coordinate system:

$$\{\boldsymbol{\varepsilon}_{pxy}\} = \{\boldsymbol{\varepsilon}_e\} + z\{\mathbf{K}_e\}$$

where:

$\{\boldsymbol{\varepsilon}_{pxy}\}$ = ply strains in element coordinate system

$\{\boldsymbol{\varepsilon}_e\}$ = element strains

z = distance from the plate reference plane to the center of the ply

$\{\mathbf{K}_e\}$ = element curvatures



MSC NASTRAN CALCULATION OF PLY RESULTS

- The ply strains need to be rotated to the ply coordinate system (1,2)

$$\{\epsilon_{p12}\} = [T]\{\epsilon_{pxy}\}$$

where: $\{\epsilon_{p12}\}$ = ply strains in ply coordinate system
 $\{\epsilon_{pxy}\}$ = ply strains in the element coordinate system

$$[T] = \begin{bmatrix} \cos^2 \theta & \sin^2 \theta & 2 \sin \theta \cos \theta \\ \sin^2 \theta & \cos^2 \theta & -2 \sin \theta \cos \theta \\ -\sin \theta \cos \theta & \sin \theta \cos \theta & \cos^2 \theta - \sin^2 \theta \end{bmatrix}$$

θ = angle from element coordinate system to ply coordinate system

MSC NASTRAN CALCULATION OF PLY RESULTS

- The ply strains can be changed to ply stresses using Hooke's Law and the ply material constitutive matrix.

$$\{\sigma_p\} = [Q]\{\varepsilon_p\}$$

where: $\{\sigma_p\}$ = ply stresses
 $\{\varepsilon_p\}$ = ply strains

$$[Q] = \begin{bmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{bmatrix}$$

See pgs. 3-4 and 3-5

- These ply stresses or strains are used in the composite failure theories

REQUESTING PLY STRESSES AND STRAINS IN MSC NASTRAN

- For ply stresses, use **STRESS** case control command.
- For ply strains, use **STRAIN** case control command.

```
SOL 101
CEND
TITLE = Composite Workshop Chapter 2 - Sample
Composite Input
    SPC = 1
    LOAD = 1
    DISP = ALL
    STRESS =ALL
$
BEGIN BULK
.
```

.dat file excerpt

REQUESTING PLY STRESSES AND STRAINS IN MSC NASTRAN

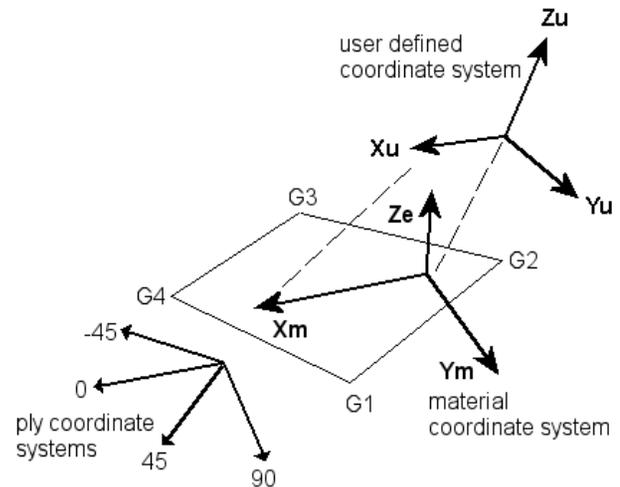
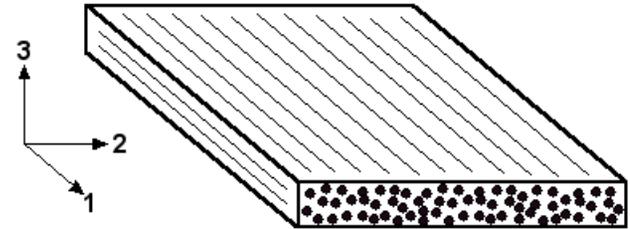
- The previous case control produces the following ply stresses:

STRESSES IN LAYERED COMPOSITE ELEMENTS (QUAD4)											
ELEMENT ID	PLY ID	STRESSES IN FIBRE AND MATRIX DIRECTIONS			INTER-LAMINAR STRESSES		PRINCIPAL STRESSES (ZERO SHEAR)			MAX SHEAR	
		NORMAL-1	NORMAL-2	SHEAR-12	SHEAR XZ-MAT	SHEAR YZ-MAT	ANGLE	MAJOR	MINOR		
0	1	1	-2.32938E+04	2.66174E+04	1.78380E+04	0.0	0.0	72.22	3.23372E+04	-2.90136E+04	3.06754E+04
0	1	2	3.00912E+05	4.77975E+03	1.53469E+04	0.0	0.0	2.96	3.01705E+05	3.98654E+03	1.48859E+05
0	1	3	-4.76318E+04	2.82568E+04	-1.53469E+04	0.0	0.0	-78.99	3.12429E+04	-5.06179E+04	4.09304E+04
0	1	4	2.76574E+05	6.41909E+03	-1.78380E+04	0.0	0.0	-3.76	2.77747E+05	5.24636E+03	1.36250E+05

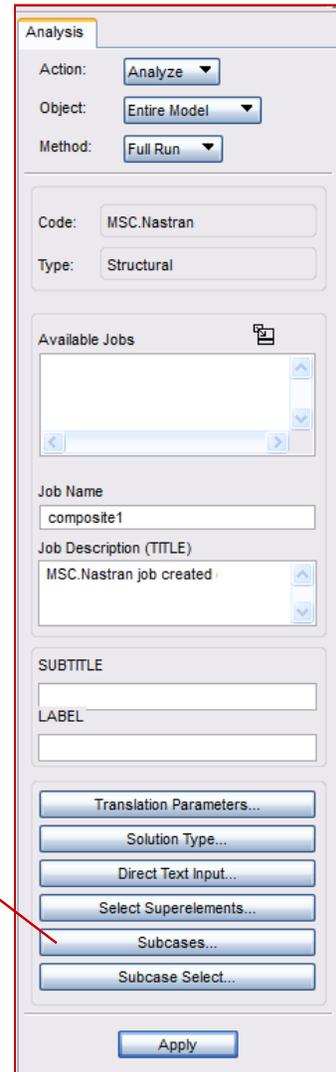
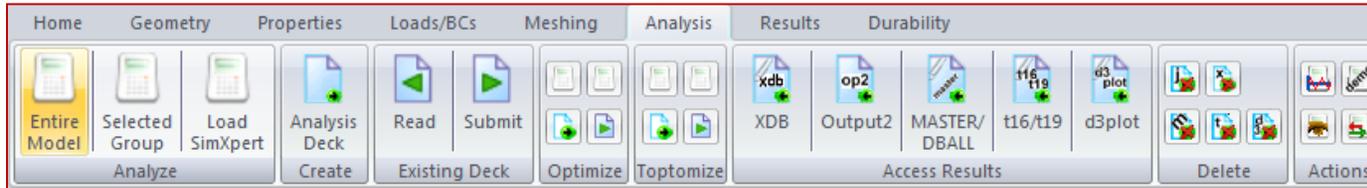
.f06 file excerpt

VIEWING PLY STRESSES AND STRAINS IN MSC NASTRAN

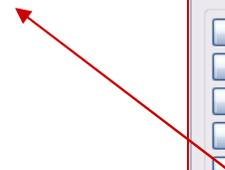
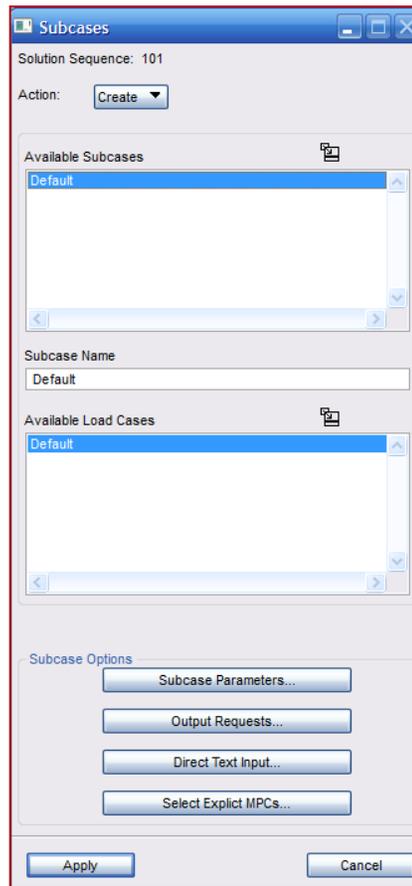
- **In-plane stresses or strains are in the ply 123 coordinate system**
 - NORMAL-1 or σ_1
 - NORMAL-2 or σ_2
 - SHEAR-12 or τ_{12}
- **Interlaminar shear stresses or strains are in the material XYZ coordinate system**
 - SHEAR XZ-MAT or τ_{xz}
 - SHEAR YZ-MAT or τ_{yz}
 - Calculated in between the plies
 - Ply 1 output is between ply 1 and ply 2
 - Last ply output is on outer surface
 - Value should be numeric zero
- **The other outputs are present only for historical plate stress or strain output and are not usually interesting for composites.**



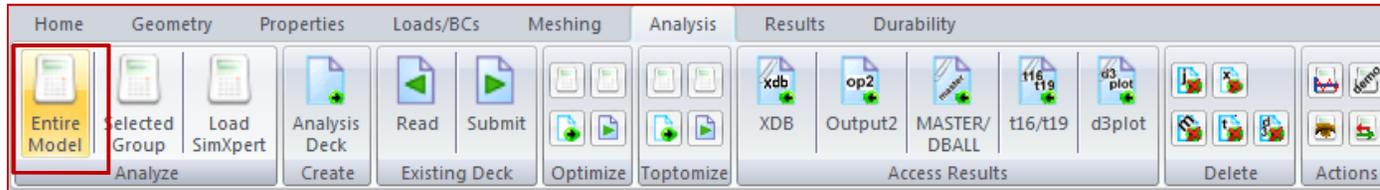
REQUESTING PLY STRESSES AND STRAINS IN PATRAN



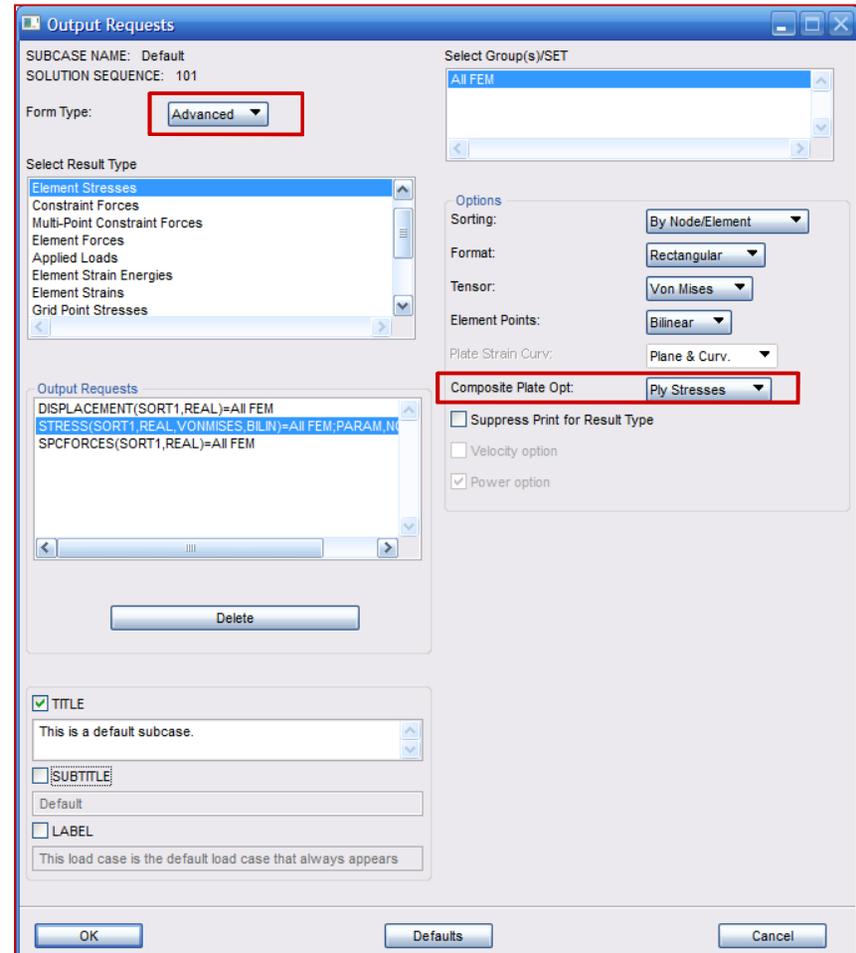
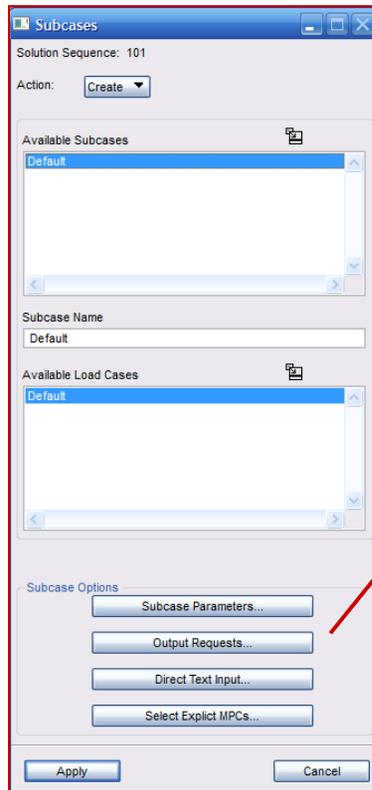
Analysis:
Analyze/ Entire Model/ Full Run
Subcases



REQUESTING PLY STRESSES AND STRAINS IN PATRAN



Output Requests/ Advanced/

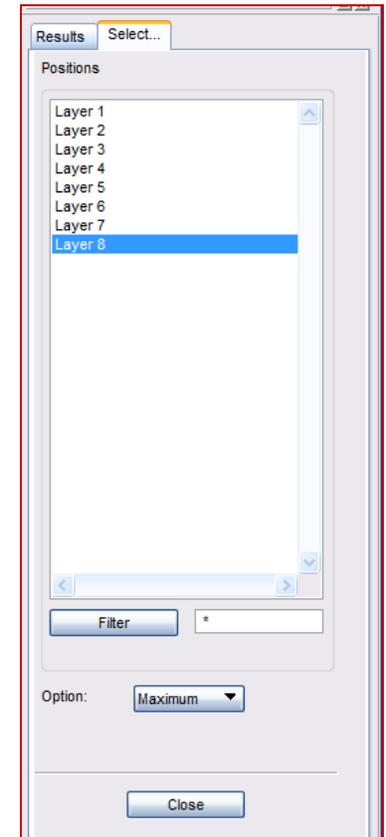


REQUESTING PLY STRESSES AND STRAINS IN PATRAN

- **For ply stresses**
 - Select STRESS and Ply Stresses
- **For ply strains**
 - Select STRAIN and Center

PLOTTING PLY STRESSES AND STRAINS IN PATRAN

- See Section 2 on how to display ply stresses or strains
- In Quantity
 - X Component = Normal 1
 - Y Component = Normal 2
 - Z Component is not used
 - XY Component = Shear 12
 - XZ Component = SHEAR XZ-MAT
 - YZ Component = SHEAR YZ-MAT
- Each Layer ID corresponds to a ply ID in the Position popup
- The highest stress in any ply of the selected Quantity will be plotted by selecting all the layers, and Option=Maximum.



PLY FAILURE THEOREMS

- **MSC Nastran failure theorems are calculated at the ply level.**
- **If composite failure is assumed with any ply failure**
 - “first ply failure” technique is being used.
 - This is a conservative approach suitable for limit loads.
- **For residual strength or ultimate load analysis**
 - If the ply failed in the fiber direction, the failed ply may be removed and the analysis run again.
 - If failure is in the matrix direction, higher ultimate load allowables should be used in that direction.
- **Failure indices (FI) greater than 1.0 indicate a failed ply.**
 - However, if the failure index is not a linear function (Hill, Hoffman, Tsai-Wu), then its value cannot be used as a linear measure of residual strength.
- **Strength ratios (SR) less than 1.0 indicate a failed ply.**
 - SR can be used as a linear indication of the amount of residual strength.
- **Ply failure analysis is useful**
 - to easily calculate failures for any layup.
 - To decrease composite failure testing if a different composite material system is used.

LAMINATE FAILURE THEOREMS

- **Laminate failure analysis uses the composite element output instead of ply output.**
- **Element load failure theorems**
 - Set PARAM,NOCOMPS,-1 which will disable ply stress or strain output
 - Either set PARAM,OMID,YES which prints or punches element loads, “equivalent stresses”, and strains in the material coordinate system (not supported by .op2 or .xdb)
 - Or use equivalent PSHELL/MAT2 combinations from ECHO=PUNCH case control command and
 - MCSID coordinate system on the MAT2s
 - PARAM,CURV,1 post-processing
- **These element outputs are then combined in user supplied theorems in an external computer program or in a postprocessor with user supplied formula capabilities such as Patran’s Create Results utility.**
- **Laminate failure theory is more accurate since each actual layup is tested.**
 - However, each layup used must be tested.
 - Changing material systems requires retesting.
- **Can be used in all solution sequences.**

REQUESTING PLY FAILURE INDICES IN MSC NASTRAN

- **Allowables must be entered on the MAT8 bulk data entry**
 - $X_t, X_c, Y_t, Y_c, S, F_{12}$
- **S_b and FT must be entered on the PCOMP bulk data entry**
 - S_b is interlaminar stress allowable
 - FT is the failure theorem to be used
- **For stress based failure theorems, the STRESS case control must be used.**
- **For strain based failure theorems, both, STRESS and STRAIN case control must be used.**

MAT8 PLY ALLOWABLES

- **Defines the inputs to failure theorems for combined loading failure envelope.**
 - X_t is the ply allowable stress or strain in the ply 1 direction for tension (for tape, usually in the fiber direction).
 - X_c is the ply allowable stress or strain in the ply 1 direction for compression (for tape, usually in the fiber direction).
 - Y_t is the ply allowable stress or strain in the ply 2 direction for tension.
 - Y_c is the ply allowable stress or strain in the ply 2 direction for compression.

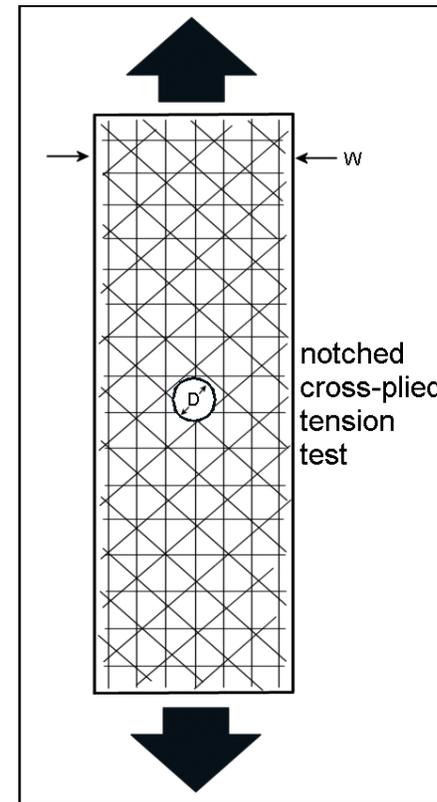
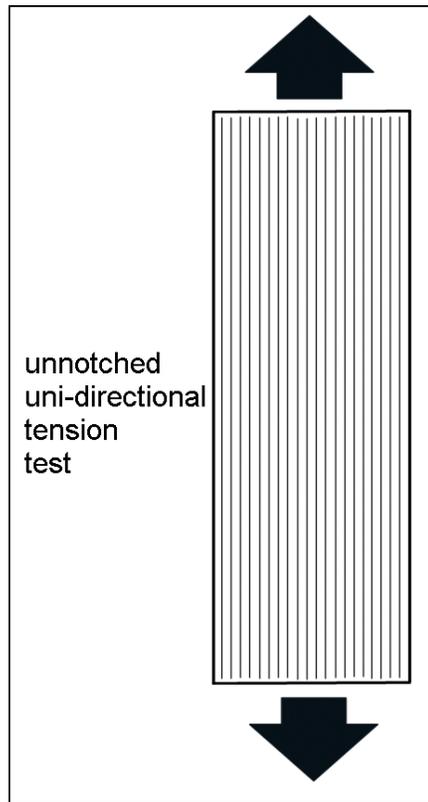
MAT8 PLY ALLOWABLES

- S is the ply allowable stress or strain in the shear 12 direction.
- Use STRN=1.0 if allowables are in units of strain (for Max Strain theorem only).
- F12 is used for the Tsai-Wu failure theorem.

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
MAT8	1	20.+6	2.+6	0.35	1.0+6	1.0+6	1.0+6	1.3-4	
	A1	A2	TREF	Xt	Xc	Yt	Yc	S	
	-2.3-7	4.5-6	0.0	1.3+5	1.2+5	1.1+4	1.2+4	1.25+4	
	GE	F12	STRN						

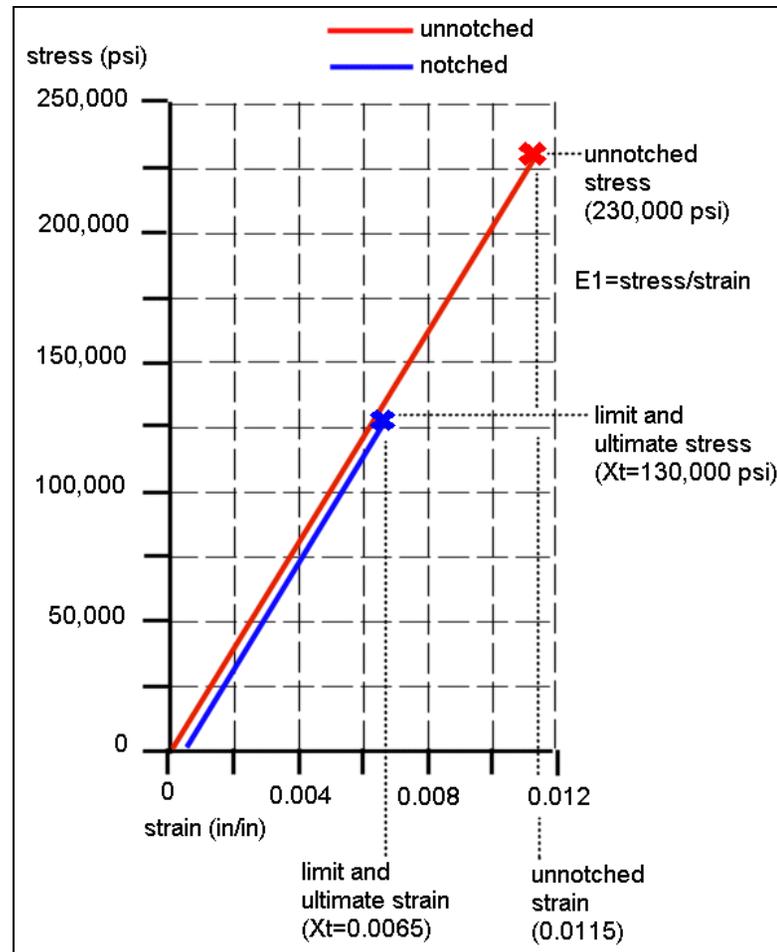
FIBER DIRECTION TENSION ALLOWABLE - X_t

- For tape, X_t comes from the unidirectional fiber tension test and/or a notched cross-ply laminate
- Failure mode is fiber breakage



FIBER DIRECTION TENSION ALLOWABLE – Xt

- Response is generally linear



FIBER DIRECTION TENSION ALLOWABLE – X_t

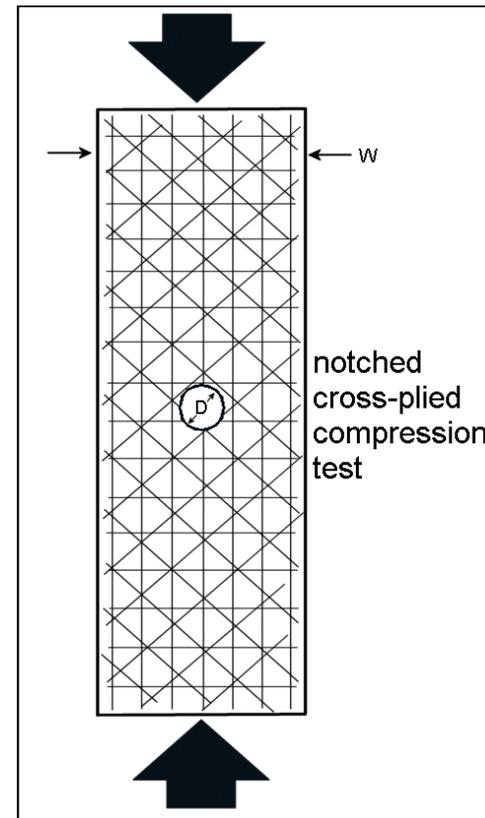
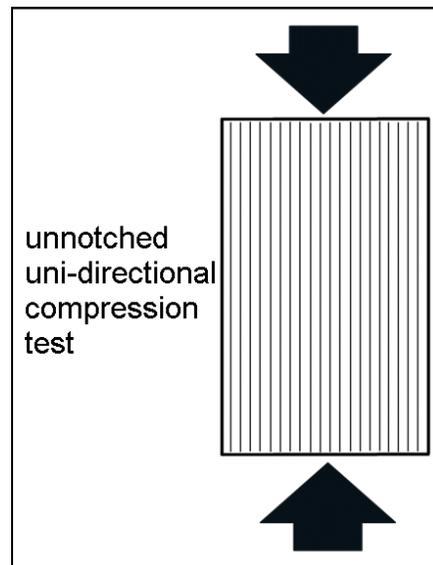
- **Notched results account for:**
 - Edge effects are theoretical singular stress fields due to
 - material property discontinuities between plies
 - geometric discontinuities
 - Undetectable manufacturing flaws
 - Maximum allowed void volume
 - Maximum allowed delamination size
 - Undetectable damage (tool drop)
- **Notching is minimized by design criteria**
 - Fastener spacing and distance from edges
 - Minimum ply angle percentages
 - Ply positioning rules such as even distribution of angled plies, symmetry and balance
- **The W/D ratio is selected based on tests of the above irregularities.**
- **An unnotched FEA model using notched allowables predicts failure on the notched test.**

FIBER DIRECTION TENSION ALLOWABLE – X_t

- **If the notched allowable is dependent on layup, you will need different allowables for each PCOMP.**
- **Create a detailed FEA model of the notched test specimen**
 - Use unnotched allowables
 - Match test failure
 - Either use different element sizes
 - Or find the “characteristic” radius
 - This detailed model can be used for detailed analysis
 - Unusual loads or lay-ups not tested
 - Bolted joints with/without bypass loads
- **Notched allowables are used for limit load and ultimate load X_t because notching in this direction affects ultimate strength.**

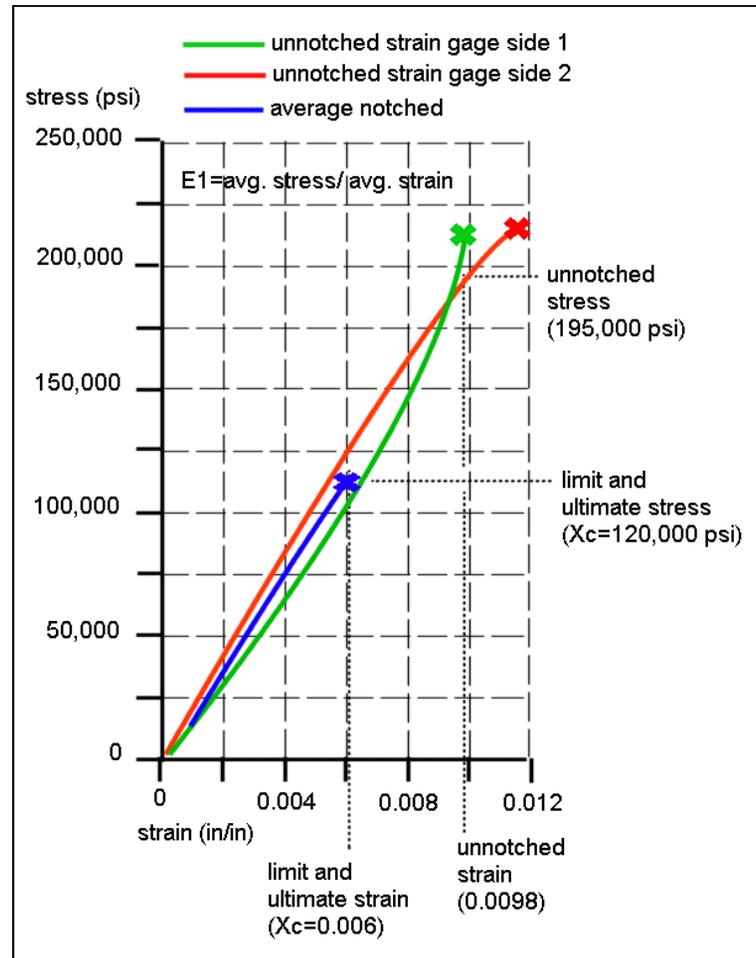
FIBER DIRECTION COMPRESSION ALLOWABLE - X_c

- For tape, X_c comes from the unidirectional fiber compression test and/or a notched cross-ply laminate
- Failure mode is fiber micro buckling.



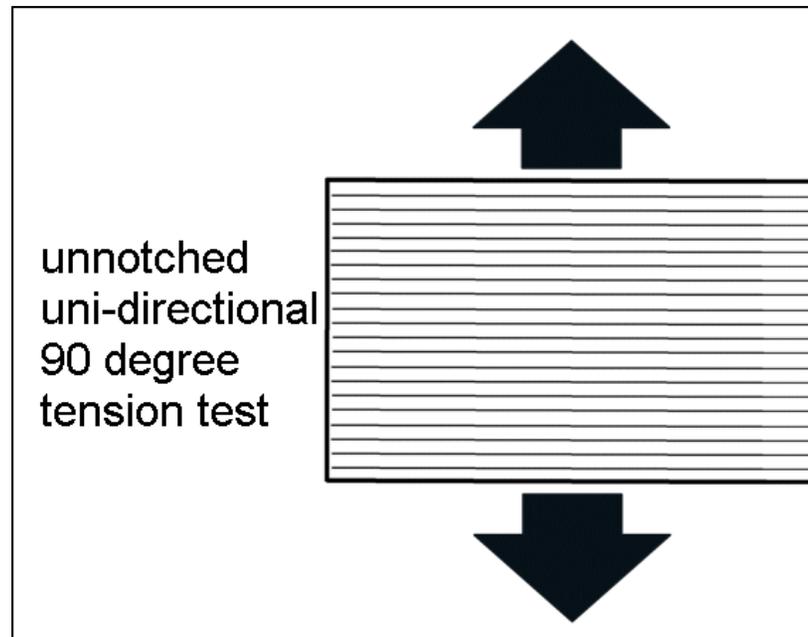
FIBER DIRECTION COMPRESSION ALLOWABLE – Xc

- Worst environment is “hot-wet”



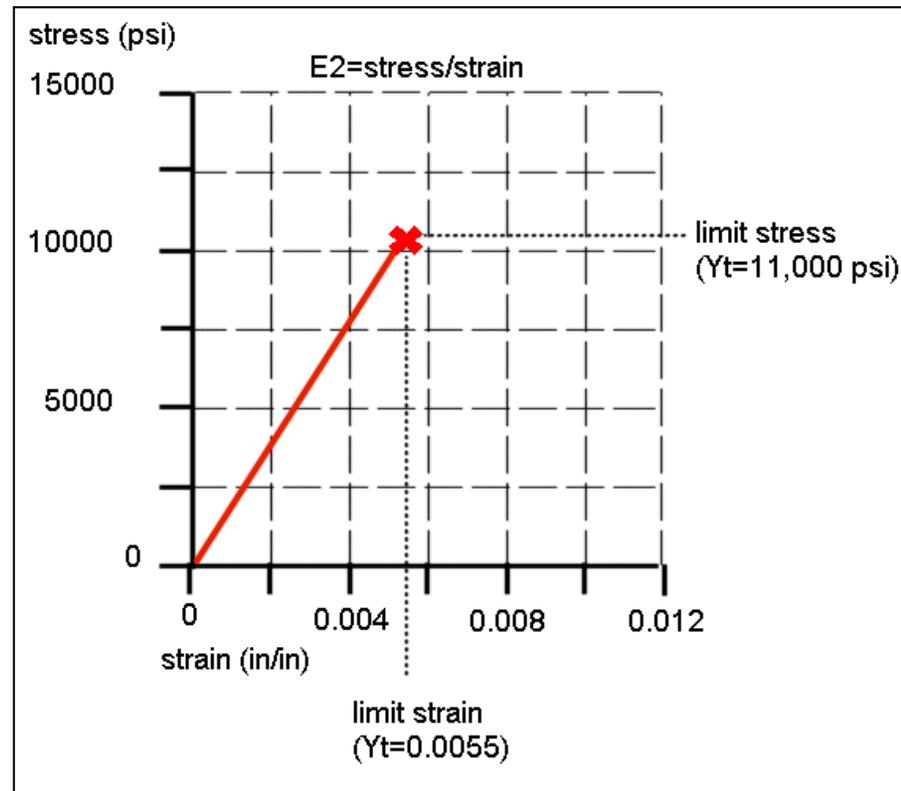
MATRIX DIRECTION TENSION ALLOWABLE - Y_t

- For tape, Y_t comes from the unidirectional 90 degree fiber tension test
- Failure mode is matrix micro cracking.



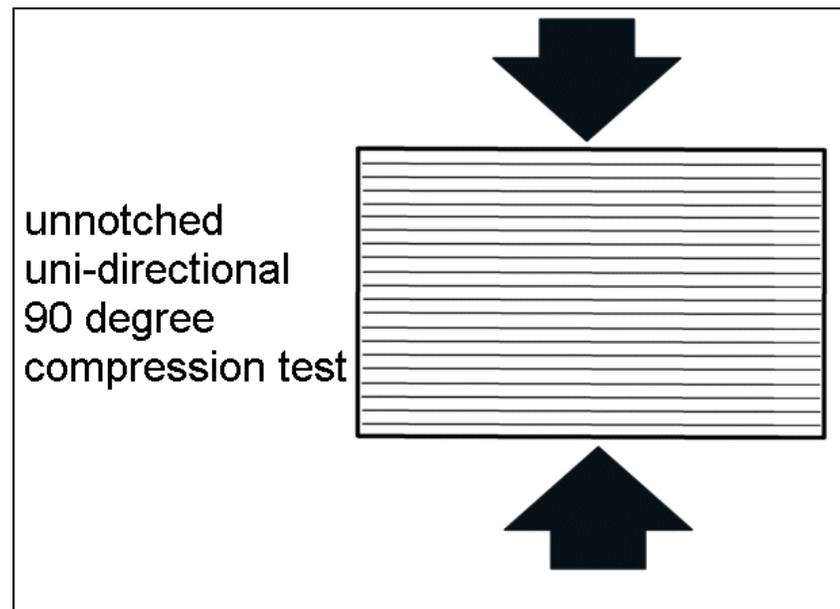
MATRIX DIRECTION TENSION ALLOWABLE – Y_t

- Worst environment is “cold”
- Y_t is a limit load allowable
- Residual strength may be available for ultimate loads



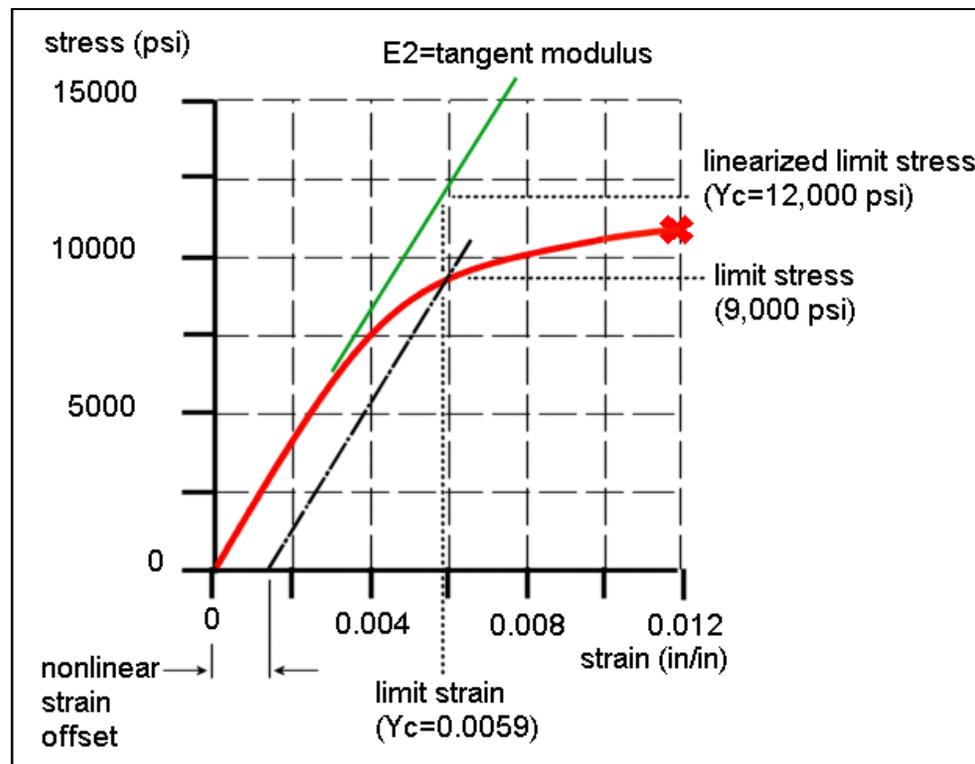
MATRIX DIRECTION COMPRESSION ALLOWABLE – Y_c

- For tape, Y_c comes from the unidirectional 90 degree fiber compression test
- Failure mode is matrix damage from nonlinear hysteresis cycling.
 - Worst environment is “hot-wet”.
 - Failure never occurs, so the limit allowable is a percent offset from linear.



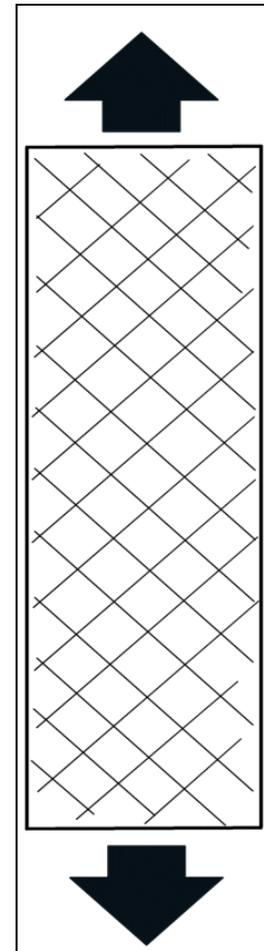
MATRIX DIRECTION COMPRESSION ALLOWABLE – Y_c

- Y_c is a limit load allowable because it is a fatigue limit.
 - Additional residual strength may be available for ultimate loads.
- If $E2$ =tangent modulus, then linearized stress must be used for Y_c .

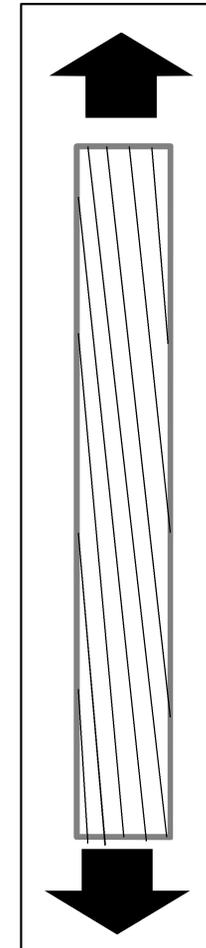


SHEAR ALLOWABLE - S

- **S comes from the +/-45 degree tension test or a 10 degree off-axis test**
- **Failure mode is matrix damage from nonlinear hysteresis cycling.**
 - Worst environment is “hot-wet”.



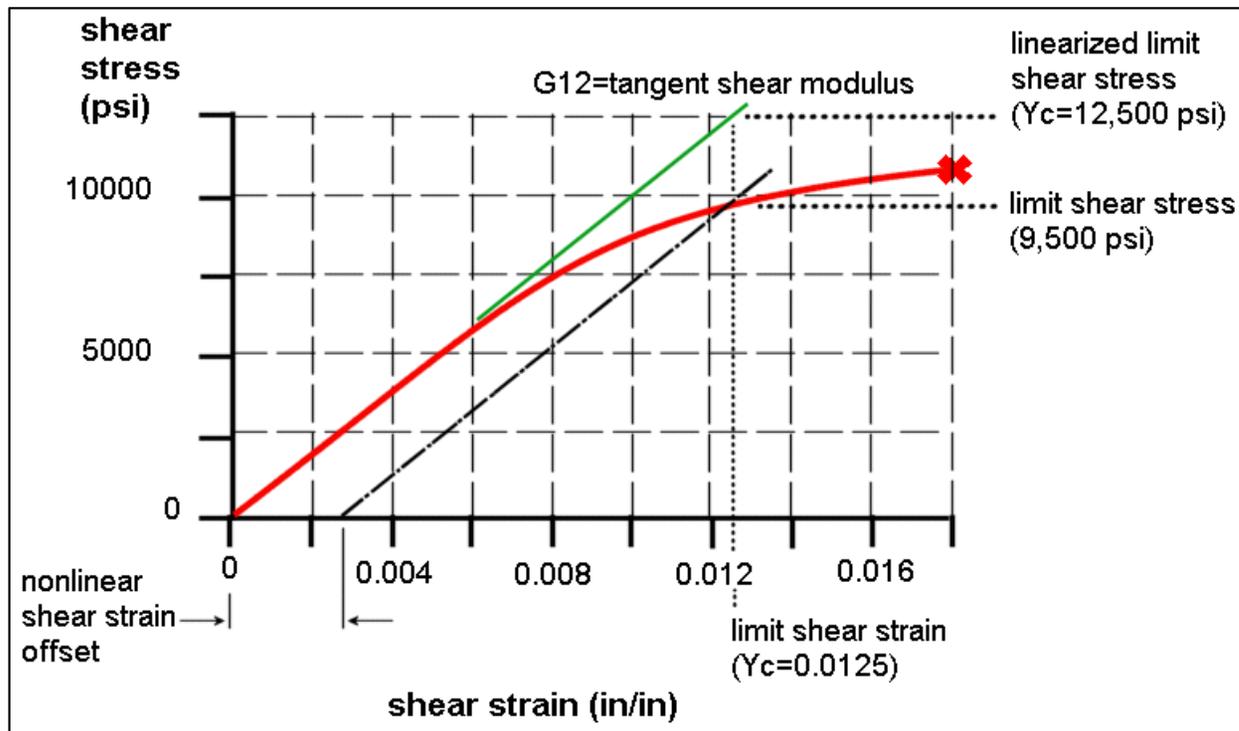
±45 degree coupon



10 degree off-axis coupon

SHEAR ALLOWABLE – S

- **S is a limit load allowable since it is a fatigue issue.**
 - Failure never occurs, so the limit allowable is a percent offset from linear.
 - Additional residual strength may be available for ultimate loads.
- **If G_{12} =tangent modulus, then linearized stress must be used for S.**



PCOMP FAILURE FIELDS

- S_b is the interlaminar shear stress allowable.
 - It is required, if FT is selected.
- FT selects the failure theorem to be used:
 - HILL
 - HOFF - Hoffman
 - TSAI – Tsai-Wu
 - STRN – max strain or stress

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
PCOMP	1			5000.0	HILL			SYM	
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2	
	1	0.0054	0.0	YES	1	0.0054	45.0	YES	
	MID3	T3	THETA3	SOUT3	MID4	T4	THETA4	SOUT4	
	1	0.0054	-45.	YES	1	0.0054	90.0	YES	

HILL FAILURE THEOREM

where:

$$\frac{\sigma_1^2}{X^2} + \frac{\sigma_2^2}{Y^2} - \frac{\sigma_1\sigma_2}{X^2} + \frac{\tau_{12}^2}{S^2}$$

$\sigma_1, \sigma_2, \tau_{12}$ = are ply stresses

$X = X_T$ if σ_1 is tensile

$X = X_C$ if σ_1 is compressive

$Y = Y_T$ if σ_2 is tensile

$Y = Y_C$ if σ_2 is compressive

- For the $\sigma_1 \sigma_2$ term, $X = X_T$ if σ_1 and σ_2 are of the same sign.
- otherwise $X = X_C$
- See pg 4-6 and 4-7 for definitions of $\sigma_1, \sigma_2, \tau_{12}$

HOFFMAN FAILURE THEOREM

- Same as Hill, with two additional terms
- Terms account for differing strengths in tension and compression

$$\frac{\sigma_1^2}{X_T X_C} + \frac{\sigma_2^2}{Y_T Y_C} - \frac{\sigma_1 \sigma_2}{X_T X_C} + \frac{\tau_{12}^2}{S^2} + \sigma_1 \left[\frac{1}{X_T} - \frac{1}{X_C} \right] + \sigma_2 \left[\frac{1}{Y_T} - \frac{1}{Y_C} \right]$$

TSAI-WU FAILURE THEOREM

$$\frac{\sigma_1^2}{X_T X_C} + \frac{\sigma_2^2}{Y_T Y_C} + 2F_{12}\sigma_1\sigma_2 + \frac{\tau_{12}^2}{S^2} + \sigma_1\left[\frac{1}{X_T} - \frac{1}{X_C}\right] + \sigma_2\left[\frac{1}{Y_T} - \frac{1}{Y_C}\right]$$

- **Same as Hoffman with a variable F_{12} times the $\sigma_1\sigma_2$ term**
 - Use the F12 field on the MAT8 to input this term
- **Setting these failure theorems to 1.0 cause them to represent ellipsoids in $\sigma_1, \sigma_2, \tau_{12}$ space.**
- **No failure occurs inside the ellipsoid, that is, the failure index is less than 1.0**
- **For the ellipsoid to have finite strength in all directions:**

$$F_{12}^2 < \frac{1}{X_T X_C Y_T Y_C}$$

- **Typical values for F_{12} are 0.5 or -0.5 times:**

$$F_{12} = \left(\frac{-1}{2(X_T X_C)} \right) \sqrt{\frac{1}{X_T X_C Y_T Y_C}} \text{ will give you the same as the Hoffman theorem.}$$

MAX STRAIN AND MAX STRESS FAILURE THEOREMS

$$Fl_1 = \frac{\epsilon_1}{X}, Fl_2 = \frac{\epsilon_2}{Y}, Fl_{12} = \frac{\gamma_{12}}{S}$$

- **Max Strain:**

- Three failure indices, one for each direction

- Uses strain allowables

- Tension or compression allowables are used based on the sign of the strain

- → Only the maximum of the three values is reported

$\epsilon_1, \epsilon_2, \gamma_{12}$ are ply strains from the MSC Nastran analysis.

X, Y, S are ply strain allowables from the MAT8 bulk data entry (set STRN term on MAT8 to 1.0).

- **Max Stress:**

- Same as Max Strain, but strain times modulus are divided by stress allowables.

- Use STRN as FT on the PCOMP, but enter allowables as stress values.

$$Fl_1 = \frac{e_1 E_1}{X}, Fl_2 = \frac{e_2 E_2}{Y}, Fl_{12} = \frac{\gamma_{12} G_{12}}{S}$$

BONDING OR INTERLAMINAR FAILURE THEOREMS

- In element coordinate system, τ_{xz} , τ_{yz} are ply stresses from the MSC Nastran analysis (see pg 4-8).
- S_b is ply bonding stress allowable S_b from the PCOMP bulk data entry.

$$Fl_{bxz} = \frac{\tau_{xz}}{S_b}, Fl_{byz} = \frac{\tau_{yz}}{S_b}$$

FAILURE UNDER COMBINED LOADING

- Setting the failure theorems to 1.0 cause them to represent ellipsoids in $\sigma_1, \sigma_2, \tau_{12}$ space.
- No failure occurs inside the ellipsoid, that is, the failure index is less than 1.0
- Plotting the loads that cause failure indices of 1.0 for a particular composite for varying amounts of $\sigma_1, \sigma_2, \tau_{12}$ create a set of envelope curves that, if you are inside the curve, failure does not occur.
- Plots are X load (F_x , running load in X direction (pounds/inch)) versus Y load (F_y) with varying amount of shear load (F_{xy}), where:
 - Percent shear = $100 * F_{xy} / (F_x + F_y + F_{xy})$

FAILURE UNDER COMBINED LOADING

- The properties used in all of the envelope (or carpet) plots are for typical graphite/epoxy tape:
 - $E1=20e6$
 - $E2=2e6$
 - $G12=1e6$
 - $NU12=0.35$
 - $Tply=0.0054$
 - $Xt=0.0110$ in/in =230,000 psi
 - $Xc=0.0098$ in/in =195,000 psi
 - $Yt=0.0065$ in/in =13,000 psi
 - $Yc=0.0160$ in/in =32,000 psi
 - $S=0.0120$ in/in =12,000 psi

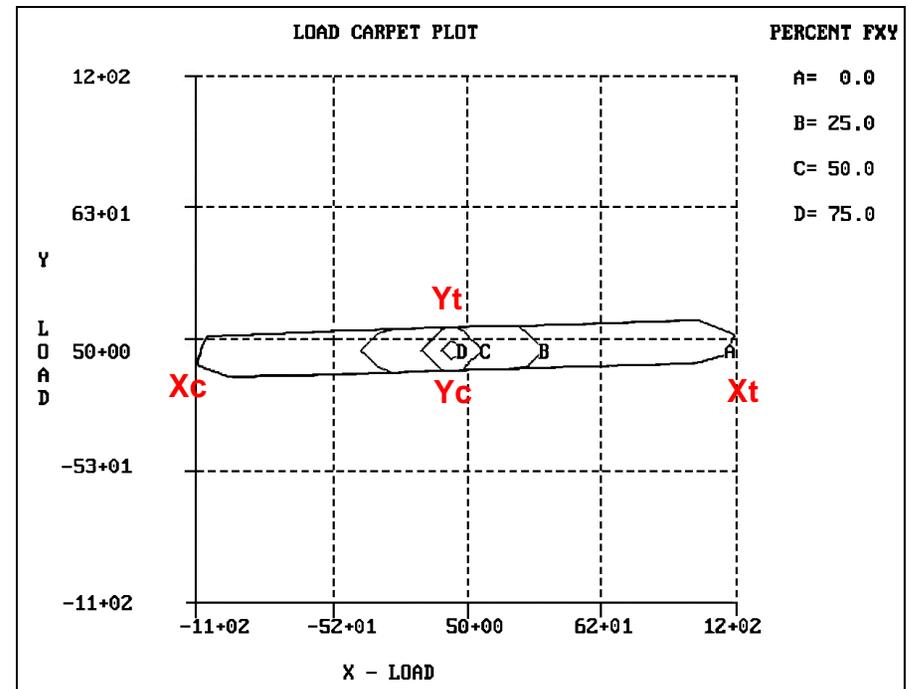
FAILURE UNDER COMBINED LOADING

- Since the Max Strain theory does not combine loads, the envelope is discontinuous when the failure mode changes.

$$F_{I_1} = \frac{\epsilon_1}{X}, F_{I_2} = \frac{\epsilon_2}{Y}, F_{I_{12}} = \frac{\gamma_{12}}{S}$$

- Uni-axial test points are shown as allowable obtained:
 - X_t, X_c, Y_t, Y_c
 - S is at the center of the D ring
- Since this is a one ply laminate, it shows the failure envelope of a ply which is strong in the X direction and weak in the Y direction.

Max strain failure theorem
1 ply graphite/epoxy tape



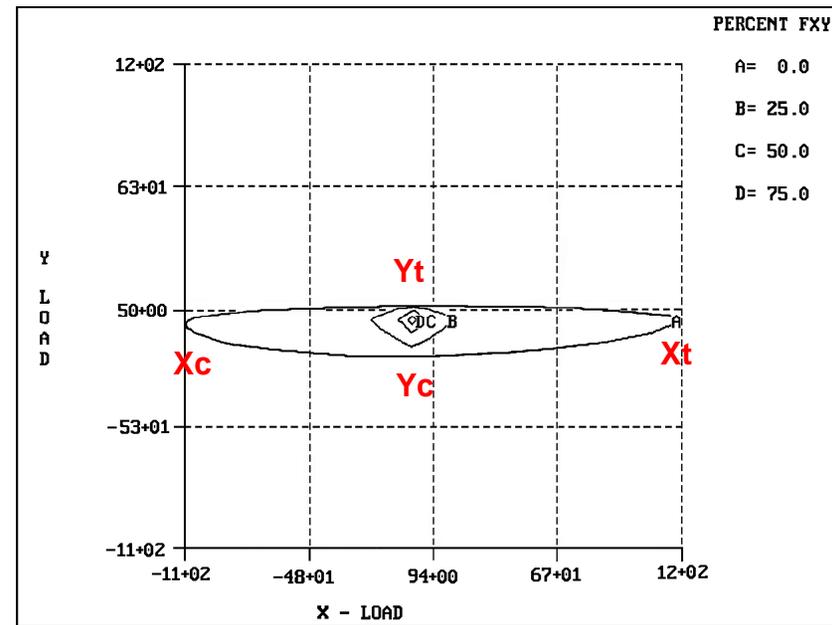
FAILURE UNDER COMBINED LOADING

- The Hill failure theorem is smooth due to its combination of loads.

$$\frac{\sigma_1^2}{X^2} + \frac{\sigma_2^2}{Y^2} - \frac{\sigma_1\sigma_2}{X^2} + \frac{\tau_{12}^2}{S^2}$$

- The allowable points are the same between theorems since they come from the same test results.
- The Hill has a smaller envelope than the max theorems around the X_t and X_c point when some Y load is combined.

Hill failure theorem
1 ply graphite/epoxy tape



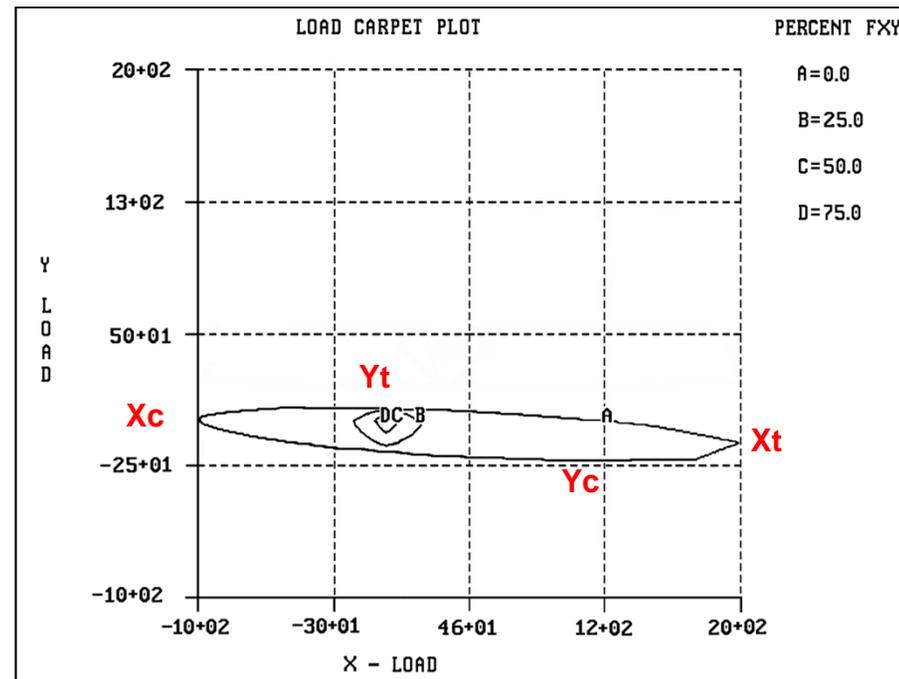
FAILURE UNDER COMBINED LOADING

$$\frac{\sigma_1^2}{X_T X_C} + \frac{\sigma_2^2}{Y_T Y_C} + 2F_{12} \sigma_1 \sigma_2 + \frac{\tau_{12}^2}{S^2} + \sigma_1 \left[\frac{1}{X_T} - \frac{1}{X_C} \right] + \sigma_2 \left[\frac{1}{Y_T} - \frac{1}{Y_C} \right]$$

$$F_{12} = 0.5 \sqrt{\left[\frac{1}{X_T} - \frac{1}{X_C} \right]^2 \left[\frac{1}{Y_T} - \frac{1}{Y_C} \right]^2}$$

- **F12 = 0.5 * (root quantity) causes the combination of compression Y load to the X load around X_t to allow much larger X loads.**

Tsai-Wu failure theorem
1 ply graphite/epoxy tape

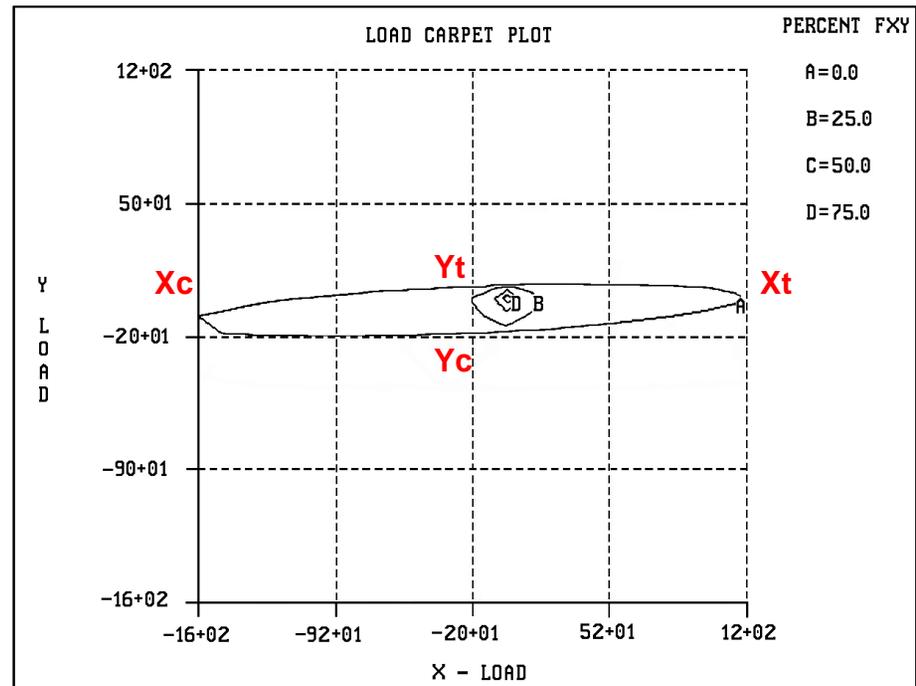


FAILURE UNDER COMBINED LOADING

$$F_{12} = -0.5 \sqrt{\left[\frac{1}{X_T} - \frac{1}{X_C}\right]^2 + \left[\frac{1}{Y_T} - \frac{1}{Y_C}\right]^2}$$

- **F12 = -0.5 * (root quantity) causes the combination of compression Y load to the X load around X_c to allow much larger X loads.**
- **→ The effect of F12 is large**

Tsai-Wu failure theorem
1 ply graphite/epoxy tape

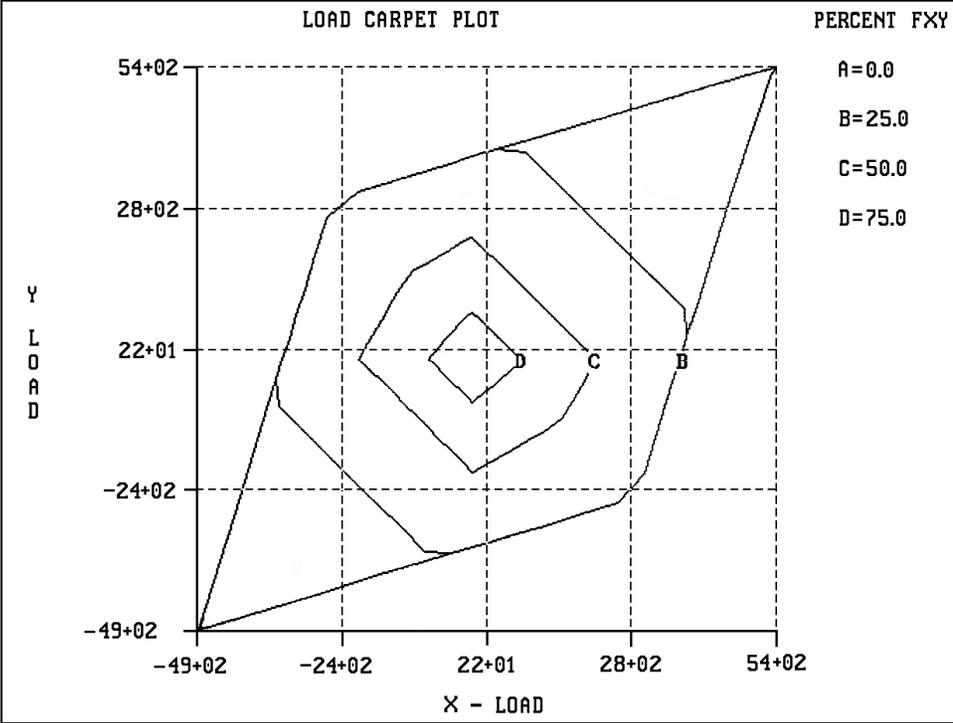


FAILURE UNDER COMBINED LOADING

$$F_{I_1} = \frac{\epsilon_1}{X}, F_{I_2} = \frac{\epsilon_2}{Y}, F_{I_{12}} = \frac{\gamma_{12}}{S}$$

- The tension-tension (5400 #/in) and compression-compression (-4900 #/in) are the strongest combination of loads.
- The areas of highest combinations are narrow.

Max strain failure theorem
8 ply graphite/epoxy tape (0/45/-45/90)sym



FAILURE UNDER COMBINED LOADING

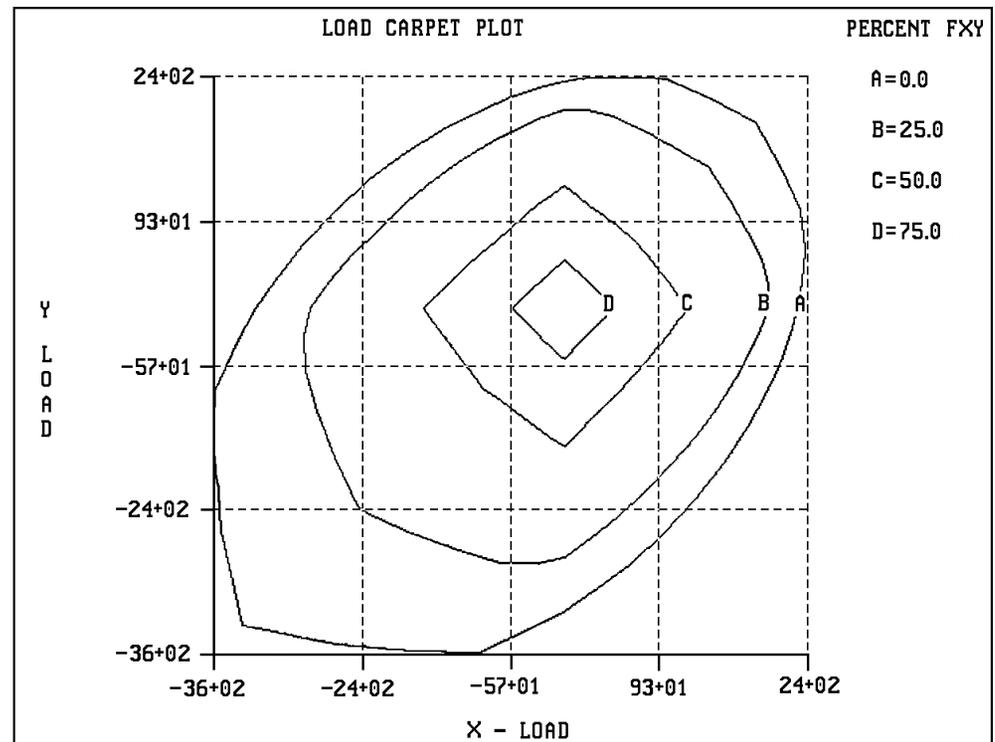
$$\frac{\sigma_1^2}{X_T X_C} + \frac{\sigma_2^2}{Y_T Y_C} + 2F_{12} \sigma_1 \sigma_2 + \frac{\tau_{12}^2}{S^2} + \sigma_1 \left[\frac{1}{X_T} - \frac{1}{X_C} \right] + \sigma_2 \left[\frac{1}{Y_T} - \frac{1}{Y_C} \right]$$

$$F_{12} = 0.5 \sqrt{\left[\frac{1}{X_T} - \frac{1}{X_C} \right]^2 \left[\frac{1}{Y_T} - \frac{1}{Y_C} \right]^2}$$

- Using 0.5 to calculate F12 causes the tension-tension and compression-compression load combinations to be much more rounded than the max strain theory.
- Allowable loads are much less, 2400 #/in and -3600 #/in, respectively.

Tsai-Wu failure theorem

8 ply graphite/epoxy tape (0/45/-45/90)sym



FAILURE UNDER COMBINED LOADING

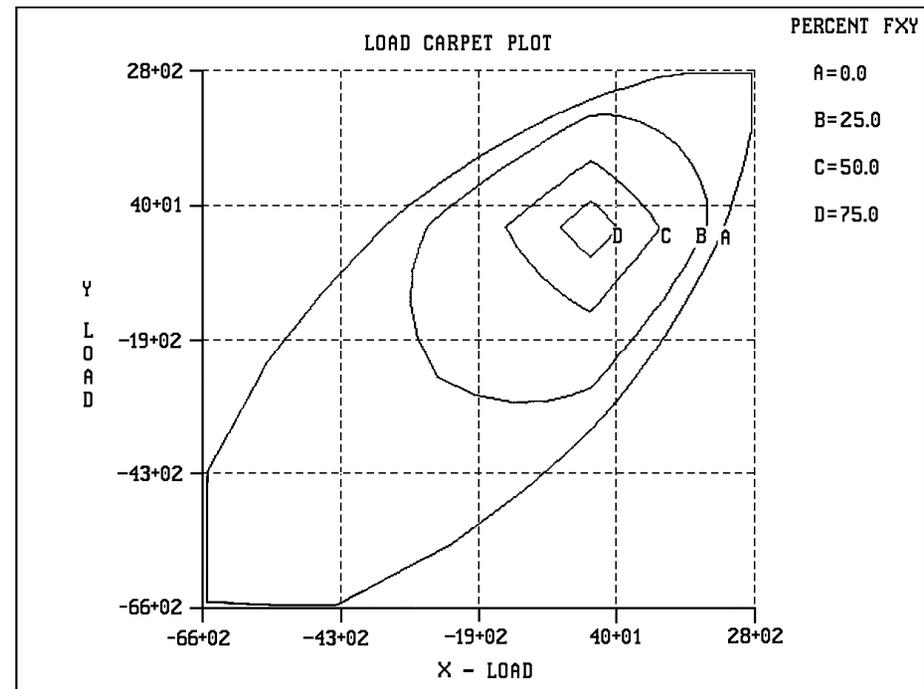
$$\frac{\sigma_1^2}{X_T X_C} + \frac{\sigma_2^2}{Y_T Y_C} + 2F_{12} \sigma_1 \sigma_2 + \frac{\tau_{12}^2}{S^2} + \sigma_1 \left[\frac{1}{X_T} - \frac{1}{X_C} \right] + \sigma_2 \left[\frac{1}{Y_T} - \frac{1}{Y_C} \right]$$

$$F_{12} = -0.5 \sqrt{\left[\frac{1}{X_T} - \frac{1}{X_C} \right]^2 \left[\frac{1}{Y_T} - \frac{1}{Y_C} \right]^2}$$

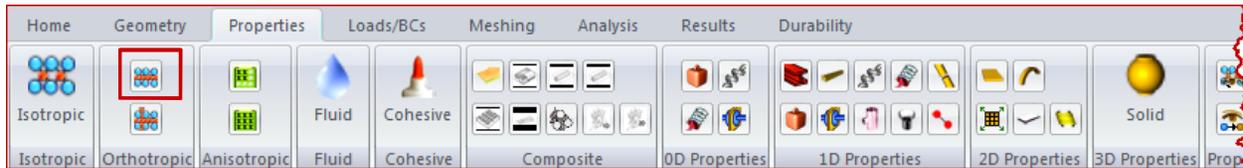
- Using -0.5 to calculate F12 causes the tension-tension to increase somewhat (2800 #/in) and the compression-compression to increase greatly (-6600 #/in).
- As it did for plies, the F12 factor greatly affects the failure envelope for general laminates.

Tsai-Wu failure theorem

8 ply graphite/epoxy tape (0/45/-45/90)sym



PATRAN PLY ALLOWABLES INPUT



Select Composite Failure Theory

Select Failure Limits units (Stress or Strain)

Enter :

Tension Stress Limit 11 (X_t)

Tension Stress Limit 22 (Y_t)

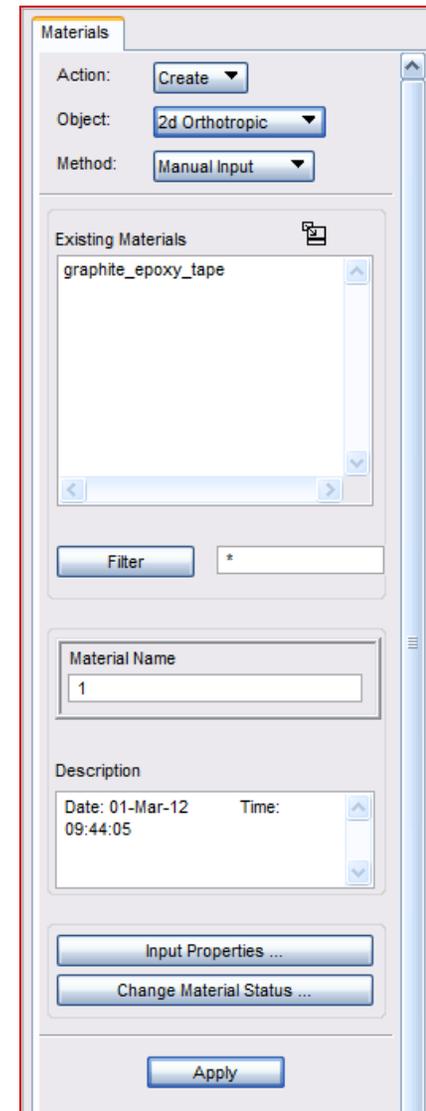
Compress Stress Limit 11 (X_c)

Compress Stress Limit 22 (Y_c)

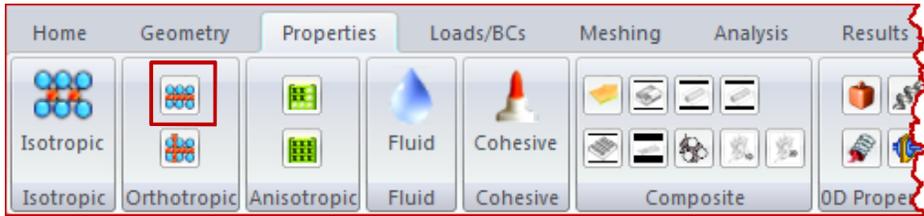
Shear Stress Limit (S)

Interaction Term (F12) if Tsai-Wu theorem is used.

Don't enter Bonding Shear Stress Limit (S_b)– this gets defined with the shell properties if you select the 'laminate' option.

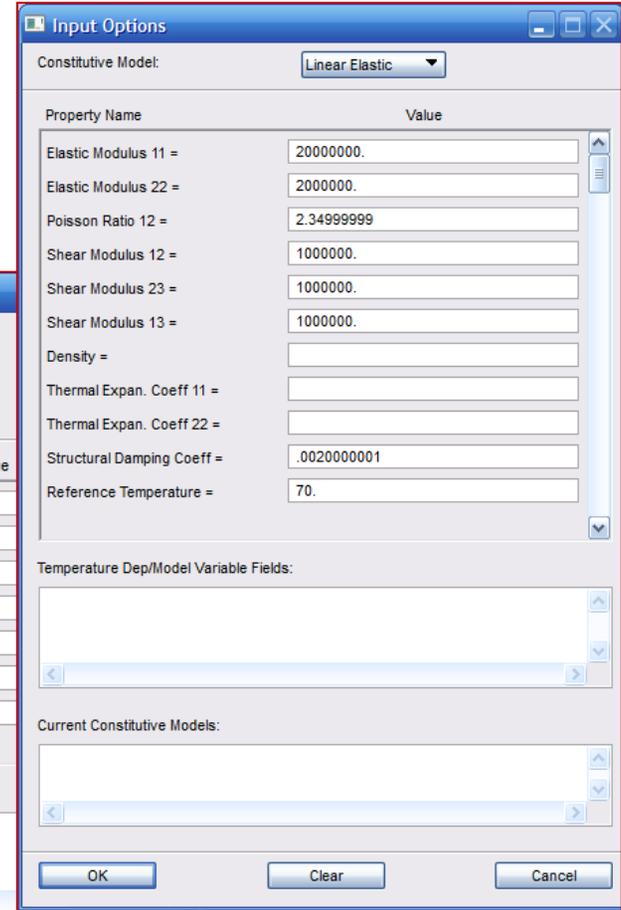
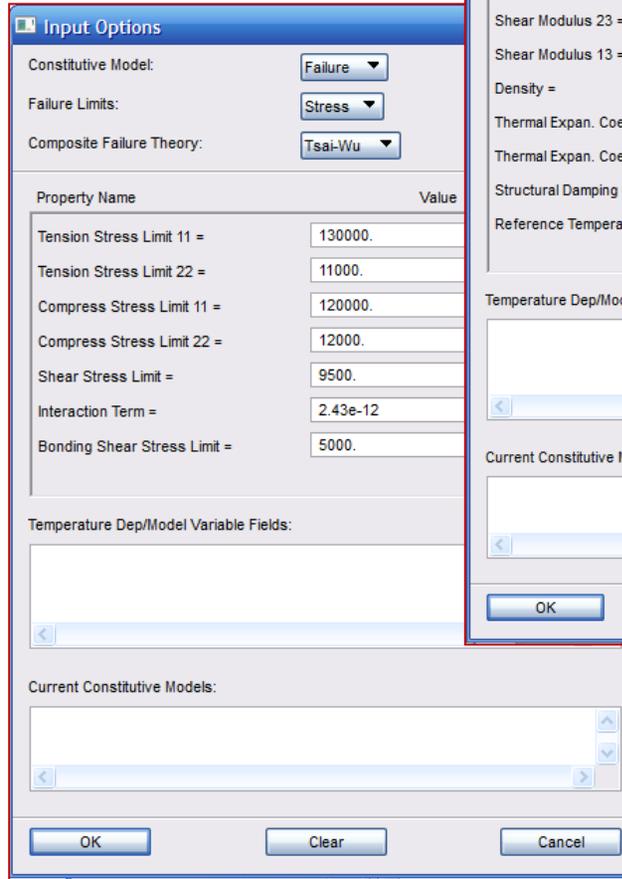


PATRAN PLY ALLOWABLES INPUT

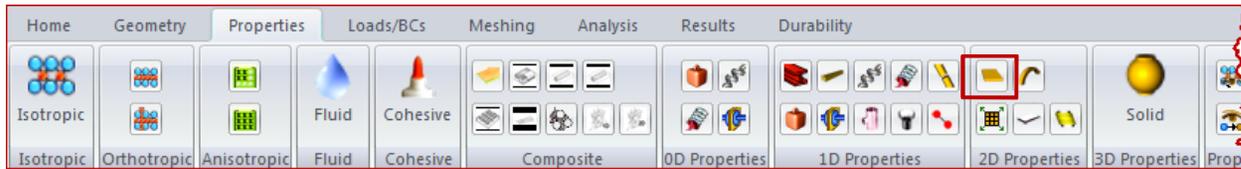


Select Composite Failure Theory
Select Failure Limits units (Stress or Strain)
Enter :

- Tension Stress Limit 11 (Xt)**
- Tension Stress Limit 22 (Yt)**
- Compress Stress Limit 11 (Xc)**
- Compress Stress Limit 22 (Yc)**
- Shear Stress Limit (S)**
- Interaction Term (F12) if Tsai-Wu theorem is used.**



PATRAN COMPOSITE PROPERTIES FORM



If the Symmetric option is used in Materials/ Create/ Composite, then the SYM option will be written to the PCOMP entry.

Z0 on PCOMP is input as Plate Offset. Leave blank for default of $-T/2$.

If you want the reference temperature, damping coefficient, or bonding stress allowable defined, do it here.

Element Properties

Action: Create

Object: 2D

Type: Shell

Sets By: Name

Filter: *

Property Set Name

Options:

Thin

Laminate

Standard Formulation

Input Properties ...

Select Application Region ...

Apply

Input Properties

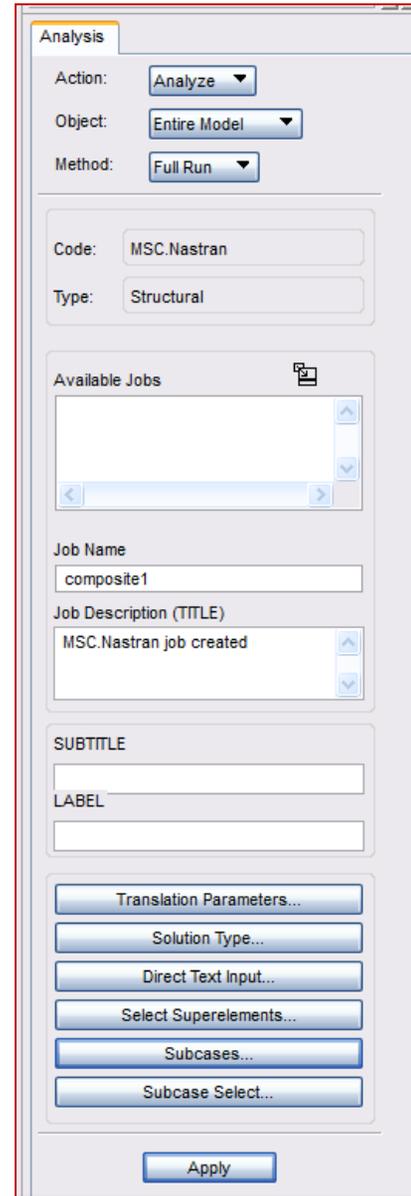
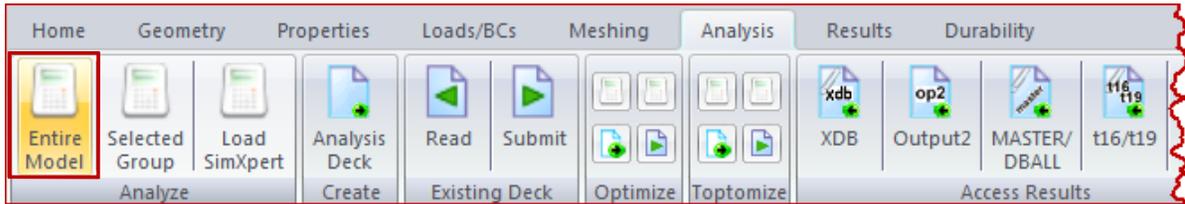
Stan. Lam. Plate (CQUAD4/PCOMP)

Property Name	Value	Value Type
Material Name		Mat Prop Name
[Material Orientation]		CID
[Nonstructural Mass]		Real Scalar
[Plate Offset]		Real Scalar
[Laminate Option]		String
[Bonding Shear]		Real Scalar
[Reference Temperature]		Real Scalar
[Damping Coefficient]		Real Scalar

Enter the Material Name or select a material with the icon.

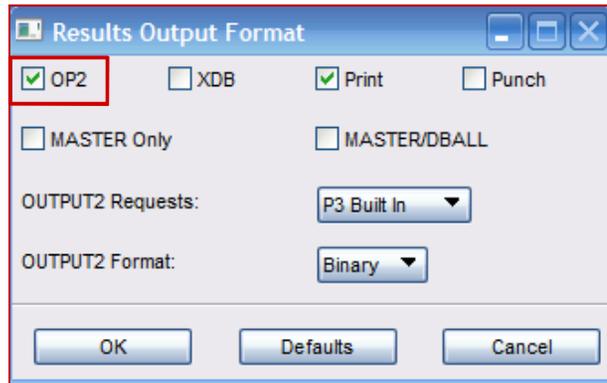
OK Clear Cancel

PATRAN FAILURE INDEX REQUEST



The .op2 output option must be used to view failure indices in Patran.

Failure indices are automatically output when ply stresses or strains are requested and ply failure allowables and failure theorems are input



MSC NASTRAN FAILURE INDEX OUTPUT

- **Printed in the f06 file if allowables on PCOMP and MAT8 bulk data entries are present**
- **Output is given for each element and for each ply**
 - Max of plies is printed for the element
 - If $FI > 1.0$ (i.e. failure has occurred) then a flag of *** is printed for that element
- **FP are in-plane failure indices**
 - For Max Stress or Strain failure theorems, the highest of the three calculated values (1, 2, 12) is printed along with the direction.
- **FB are interlaminar or bonding failure indices**
 - Calculated in between the plies
 - Ply 1 output is between ply 1 and ply 2
 - Last ply output is on outer surface
 - Value should be numeric zero

MSC NASTRAN FAILURE INDEX OUTPUT

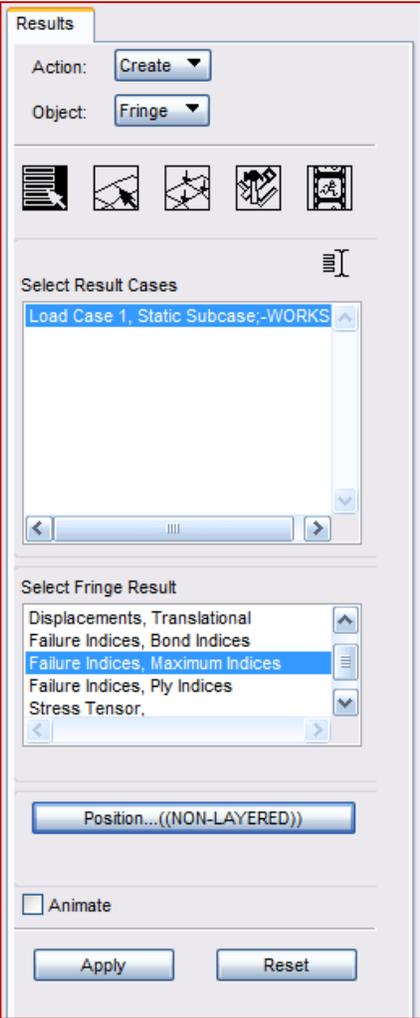
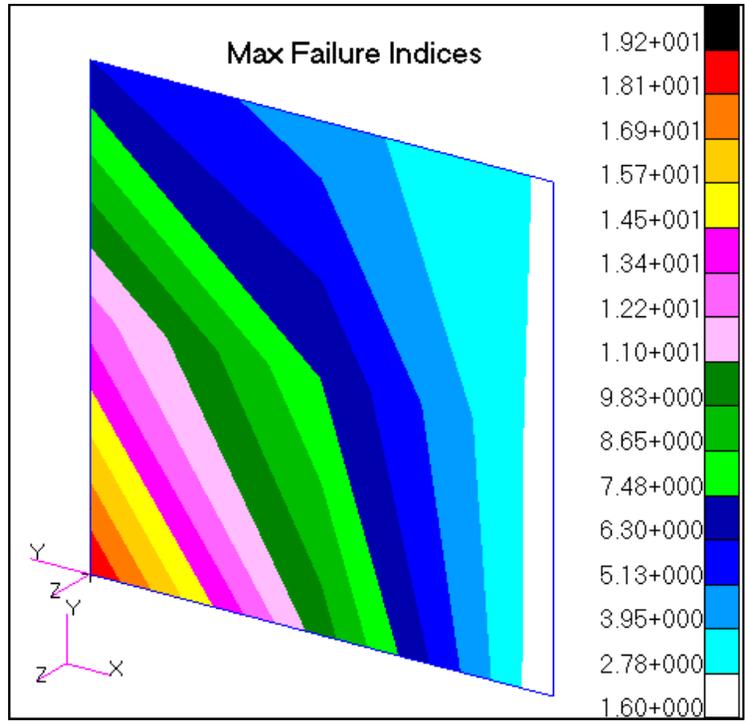
FAILURE INDICES FOR LAYERED COMPOSITE ELEMENTS (QUAD4)						
ELEMENT ID	FAILURE THEORY	PLY ID	FP=FAILURE INDEX FOR PLY (DIRECT STRESSES/STRAINS)	FB=FAILURE INDEX FOR BONDING (INTER-LAMINAR STRESSES)	FAILURE INDEX FOR ELEMENT MAX OF FP,FB FOR ALL PLYS	FLAG
1	HILL	1	14.7704			
		2	15.4489		0.0000	
		3	19.2287		0.0000	
		4	15.1403		0.0000	
		5	15.1403		0.0000	
		6	19.2287		0.0000	
		7	15.4489		0.0000	
		8	14.7704		0.0000	
					19.2287	***

.f06 file extract

PATRAN MAX FAILURE INDEX PLOTTING

- First, plot the maximum failure index from any ply or failure theorem.
- This shows which element is closest to failure.
- Look in the f06 file to see where failure is, ply or bond.

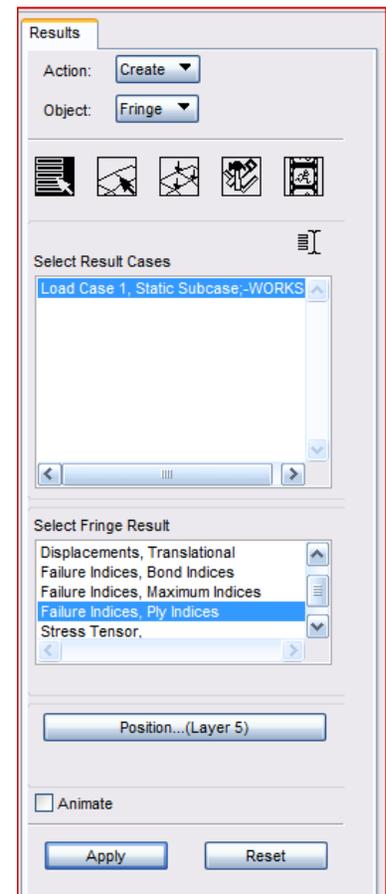
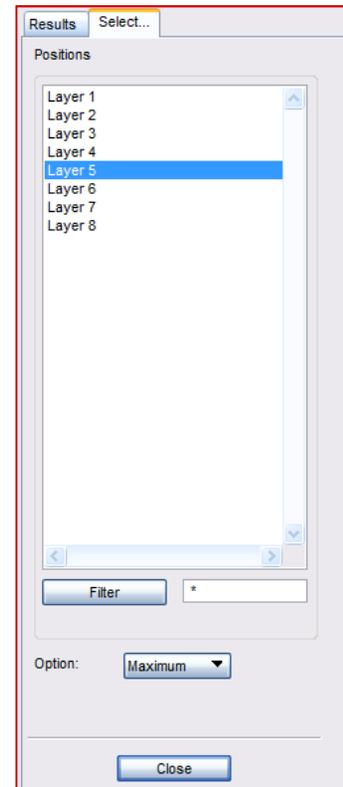
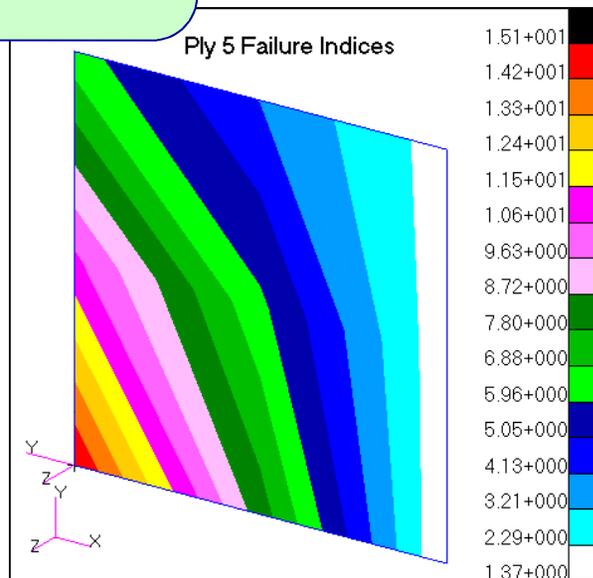
Attach op2 file
 Results
 Create/ Fringe
 Select Result Case/
 Load Case 1
 Select Fringe Result/
 Failure Indices,
 Maximum
 Apply



PATRAN PLY FAILURE INDEX PLOTTING

- More information can be obtained by plotting ply and bond failure indices.
- Max strain and stress theorems have 3 failure indices per ply.
 - Directions 1, 2, and 12

Select Fringe Result/
Failure Indices, Ply Indices
Position/ Layer 5
Apply



STRENGTH RATIOS

- **SR = Allowable loads / actual loads**
- **Use PARAM,SRCOMPS,YES to obtain both strength ratios and failure indices.**
- **Strength Ratio (SR) is a better indicator of remaining strength than failure indices**
 - Shows how far to failure
 - Similar to factor of safety
 - Failure indices do not show this since they are usually nonlinear functions.
 - Exceptions to this are max strain and stress theorems and interlaminar bonding failure
 - At small loads, failure indices can be negative.
- **SRs are supported as responses in SOL 200 Optimization analysis only for SOL 101.**
- **They are not available for display in Patran without DMAP**

CALCULATION OF STRENGTH RATIOS

- Using TSAI-WU failure criteria as an example:

$$FI = \frac{\sigma_1^2}{X_T X_C} + \frac{\sigma_2^2}{Y_T Y_C} + 2F_{12} \sigma_1 \sigma_2 + \frac{\tau_{12}^2}{S^2} + \sigma_1 \left[\frac{1}{X_T} - \frac{1}{X_C} \right] + \sigma_2 \left[\frac{1}{Y_T} - \frac{1}{Y_C} \right]$$

- Substituting the (actual stresses) with (SR * actual stress) and setting FI=1.0, the equation becomes

$$0.0 = \left[\frac{\sigma_1^2}{X_T X_C} + \frac{\sigma_2^2}{Y_T Y_C} + 2F_{12} \sigma_1 \sigma_2 + \frac{\tau_{12}^2}{S^2} \right] SR^2 + \left[\sigma_1 \left[\frac{1}{X_T} - \frac{1}{X_C} \right] + \sigma_2 \left[\frac{1}{Y_T} - \frac{1}{Y_C} \right] \right] SR - 1.0$$

- Now solve for roots (SRs) for the quadratic equation
- Loads can now be linearly scaled based on the SR to yield FI=1.0

MSC NASTRAN STRENGTH RATIO OUTPUT

- Note that **SR=0.2280** compares to an equivalent **FI=19.2287** (pg 4-42)
- This indicates that the load is too high by a factor of:
 - $1/SR = 1/0.2280 = 4.39$
- Divide the applied loads by 4.39 (or multiply them by 0.228) to find allowable load

STRENGTH RATIOS FOR LAYERED COMPOSITE ELEMENTS (QUAD4)						
ELEMENT ID	FAILURE THEORY	PLY ID	SRP-STRENGTH RATIO FOR PLY (DIRECT STRESSES/STRAINS)	SRB-STRENGTH RATIO FOR BONDING (INTER-LAMINAR STRESSES)	STRENGTH RATIO FOR ELEMENT MIN OF SRP,SRB FOR ALL PLIES	FLAG
1	HILL	1	2.601980E-01			
		2	2.544198E-01			
		3	2.280472E-01			
		4	2.569999E-01			
		5	2.569999E-01			
		6	2.280472E-01			
		7	2.544198E-01			
		8	2.601980E-01			
					2.280472E-01	***

.f06 file excerpt

PATRAN STRENGTH RATIO POST-PROCESSING

- Currently Patran cannot post-process Strength Ratio data.
- The DMAP alter below will replace Failure Index data with the Strength Ratio data (note that the label is still failure index)

```
SOL 101
$
$
compile sedrcvr $ v2012.1
alter 'output2.*oes1c,oeffit,oesrt' (1,-1) $
copy oesrt/oeffitz/always/-1 $
type parm,,i,n,reclpoef $
reclpoef=1 $
do while ( reclpoef>=0 ) $
paraml oeffitz/oeffitzz/'tabrepi'/s,n,reclpoef/2//25 $
reclpoef = reclpoef + 2 $
paraml oeffit/'dti'/s,n,reclpoef/2//s,n,junkit $
equivx oeffitzz/oeffitz/always $
enddo $ reclpoef
equivx oeffitzz/oeffit/always $
endalter

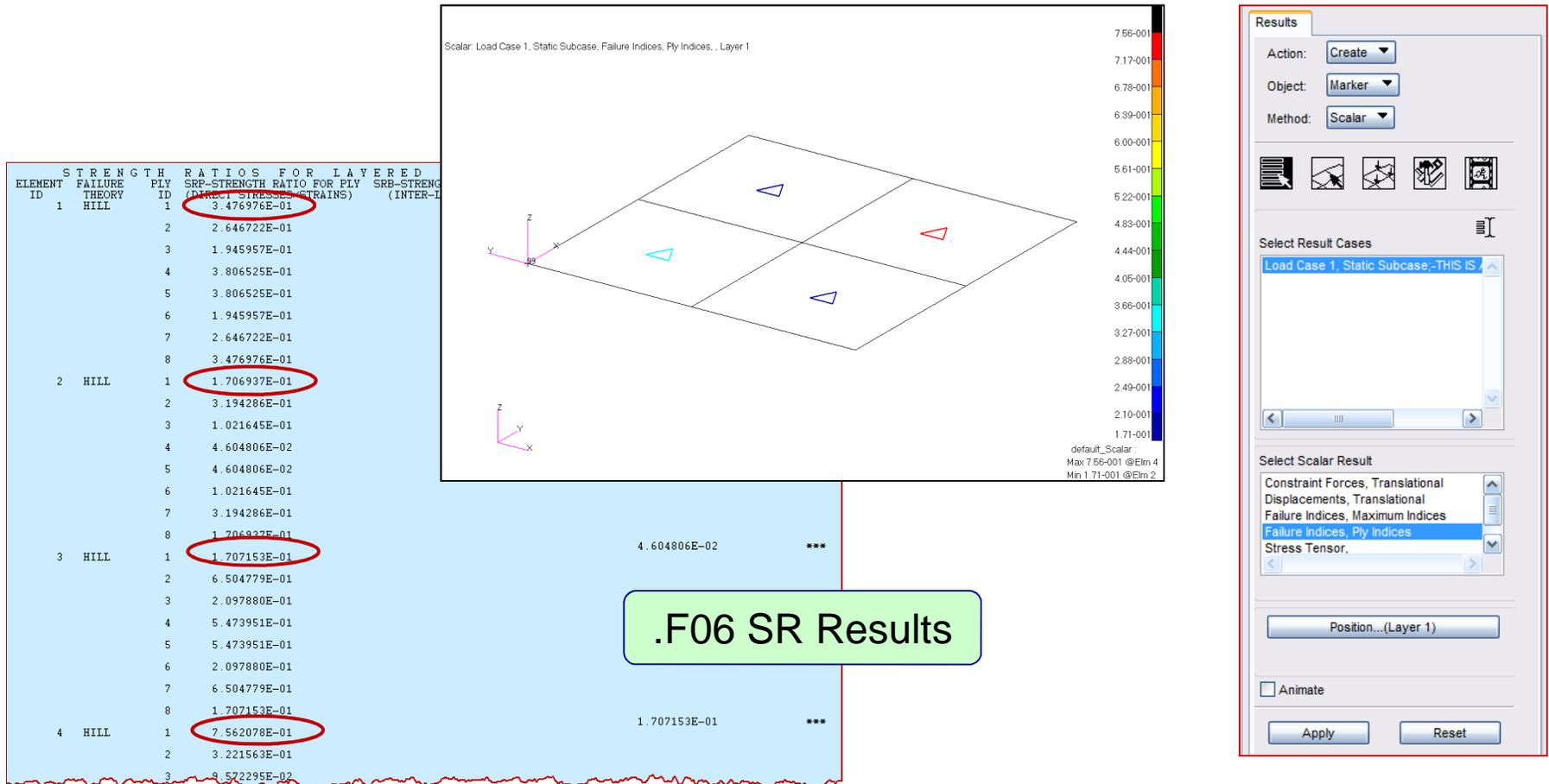
CEND
TITLE = 4 ELEMENTS - QUAD4 COMPOSITE
STRESS = ALL
SPC = 1
```

.DMAP alter

PATRAN STRENGTH RATIO POST-PROCESSING

- **Patran and F06 Results**

- Note Patran Marker Strength Ratio Results are still labeled Failure Indices using the DMAP method.



.F06 SR Results

PARAM, NOCOMPS

- =1, ply output (default)
- =0, element output and ply output
- =-1, element output
- **Element output types**
 - Element strains
 - Equivalent stresses
 - Assumes homogeneous material
 - Values are incorrect because composites are non-homogeneous
- **If element strains are selected**
 - The principle strain direction will indicate the direction of maximum strain.
 - May need more fibers in that direction.
- **If element stresses are selected**
 - Element forces in the material coordinate system may be “backed out”.
- **See Laminate Failure Theorems for further use of PARAM,NOCOMPS.**

PARAM, NOCOMPS

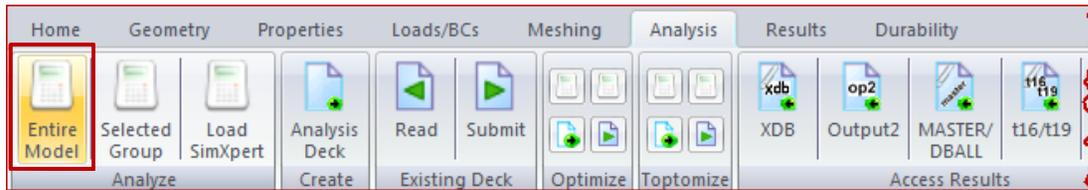
- When PARAM,NOCOMPS,-1 bulk data parameter is used along a STRAIN=ALL case control command, the following output is produced:

.f06 file extract

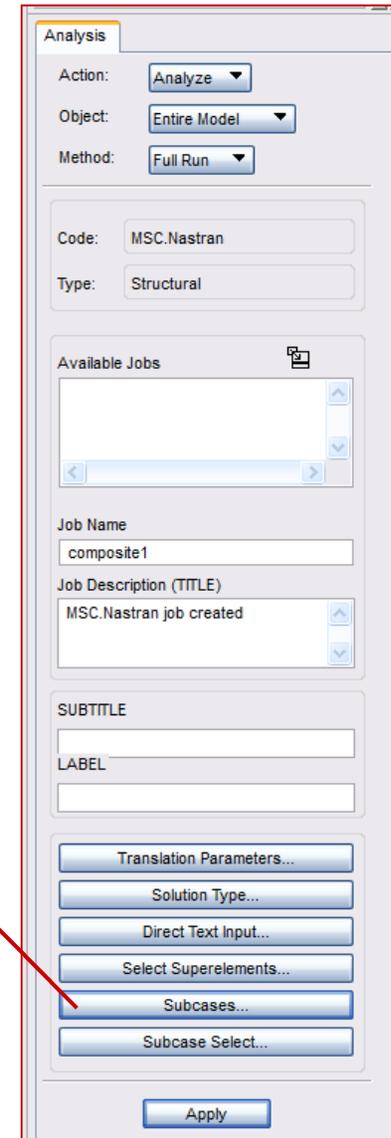
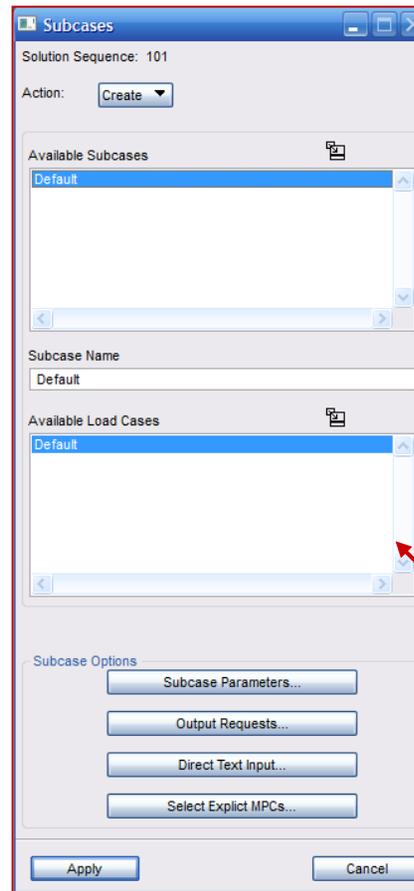
S T R A I N S I N Q U A D R I L A T E R A L E L E M E N T S (Q U A D 4)									
ELEMENT	STRAIN		STRAINS IN ELEMENT COORD SYSTEM			PRINCIPAL STRAINS (ZERO SHEAR)			
ID.	CURVATURE		NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
0	1	0.0	1.229821E-02	9.603290E-03	2.730186E-02	42.1813	2.466802E-02	-2.766525E-03	1.744082E-02
		-1.000000E+00	0.0	0.0	0.0	0.0	0.0	0.0	0.0
0	2	0.0	1.129405E-02	-1.183270E-02	9.950880E-03	11.6405	1.231902E-02	-1.285768E-02	1.453689E-02
		-1.000000E+00	0.0	0.0	0.0	0.0	0.0	0.0	0.0
0	3	0.0	-3.134008E-03	6.225147E-03	8.147041E-03	69.4804	7.749762E-03	-4.658625E-03	7.237704E-03
		-1.000000E+00	0.0	0.0	0.0	0.0	0.0	0.0	0.0
0	4	0.0	4.397135E-03	-1.491786E-03	1.420404E-02	33.7407	9.140880E-03	-6.235531E-03	8.930243E-03
		-1.000000E+00	0.0	0.0	0.0	0.0	0.0	0.0	0.0

- When the principal strains are plotted in Patran with markers, the direction of maximum strains are shown.

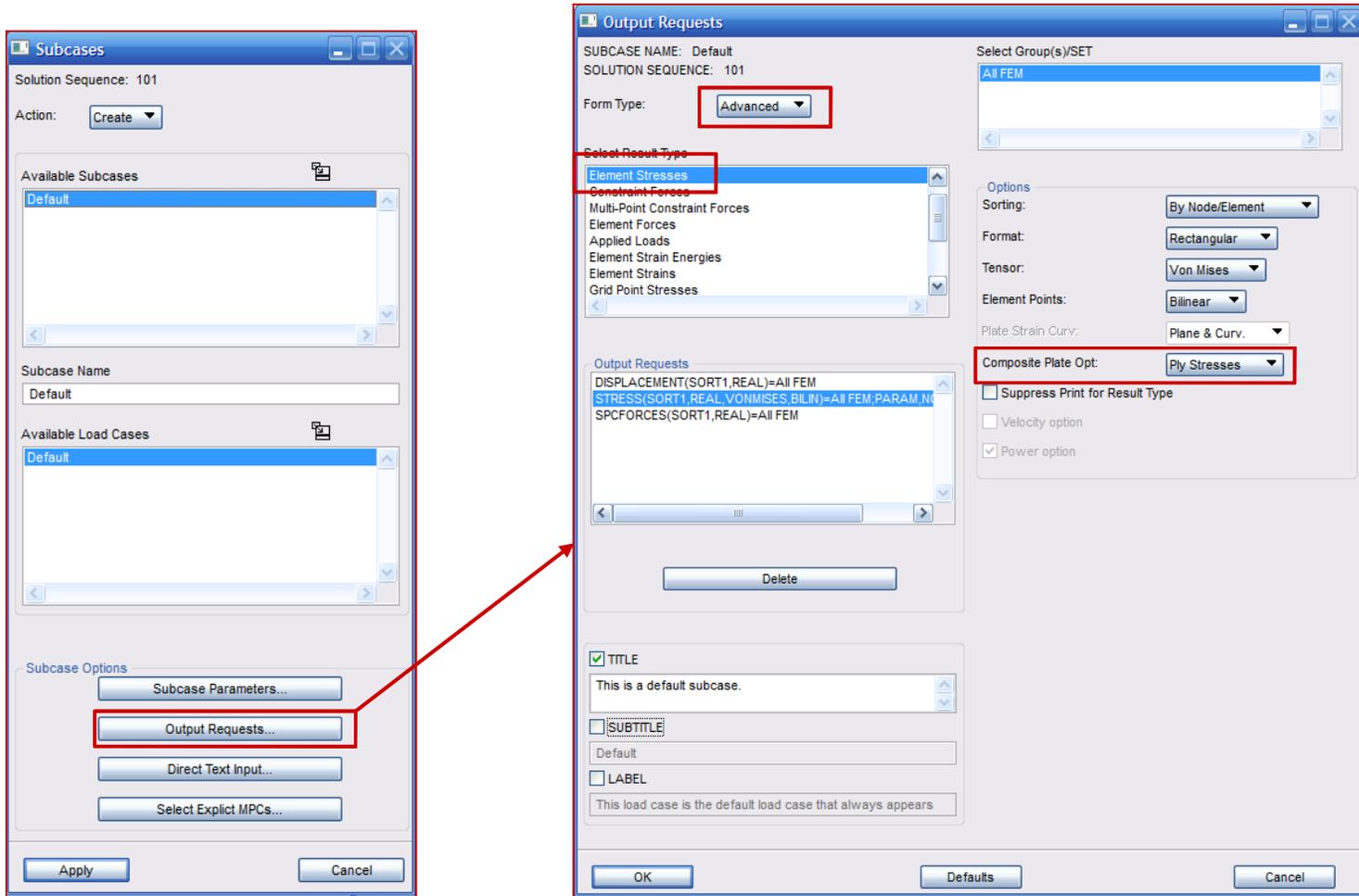
PARAM, NOCOMPS IN PATRAN



Analysis:
Subcases



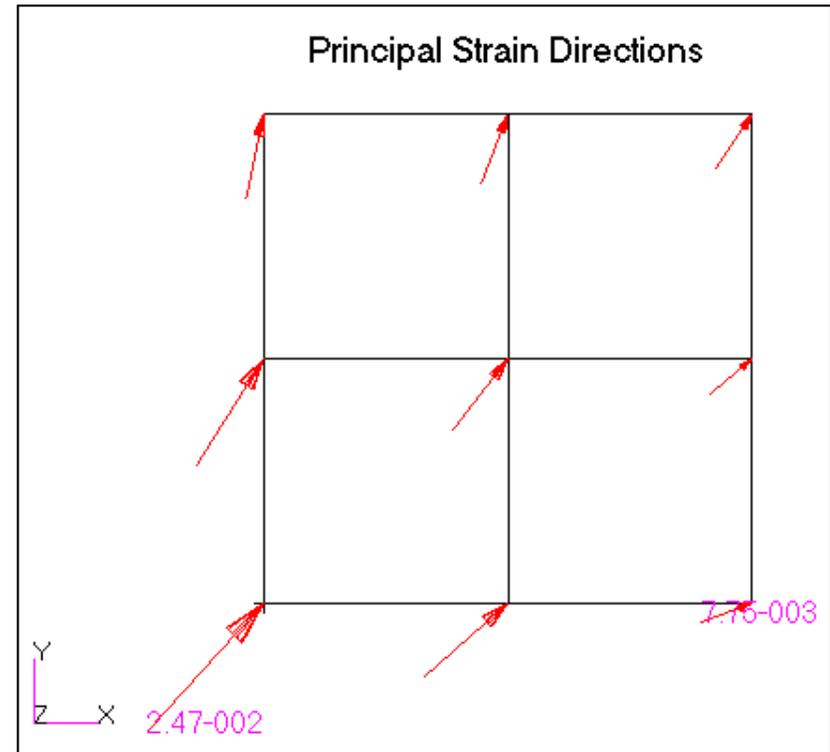
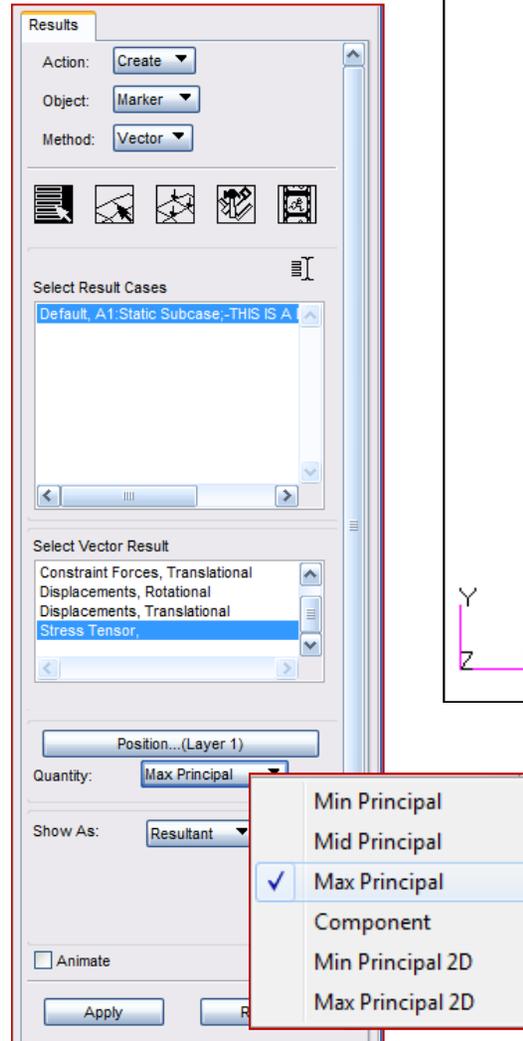
PARAM, NOCOMPS IN PATRAN



Output Requests/ Advanced
For Composite Plate Opt, change “Ply Stresses” to “Element Stresses”

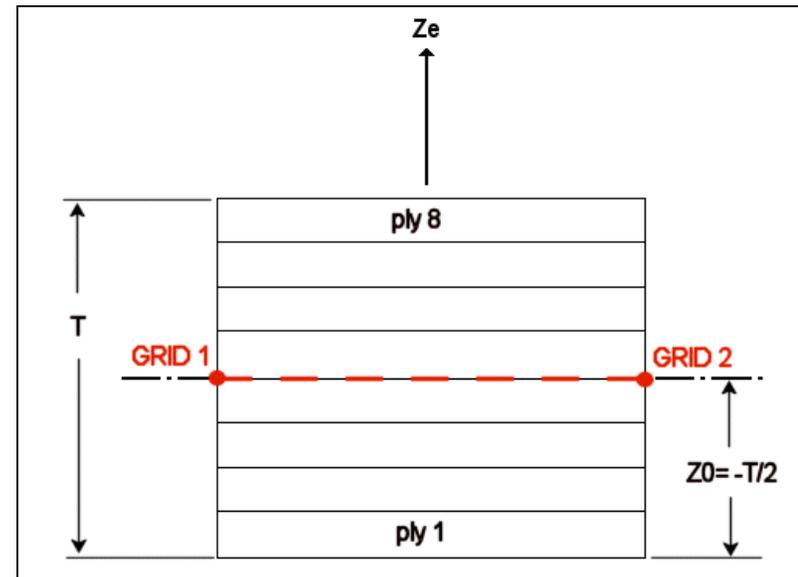
PARAM, NOCOMPS IN PATRAN

Results:
Create/ Marker Result/
"Strain Tensor" Quantity/
"Max Principal"
Apply



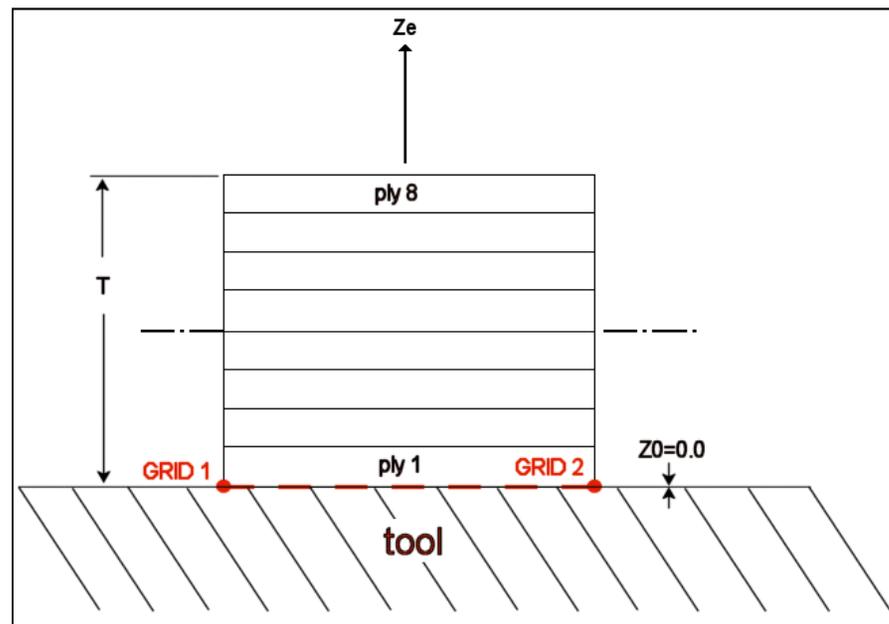
COMPOSITE PLATE OFFSET Z0

- **Z0 is the distance from the GRID locations to the bottom of the composite plate.**
 - field 3 on the PCOMP bulk data entry
- **This is normally half the thickness of the plate in direction opposite to the element's Z axis direction.**
 - $Z0 = -T/2$
 - Default value
- **Plate offsets can also be set on the ZOFFS field of the QUAD4 bulk data entry.**
- **ZOFFS and Z0 are additive.**
- **Entered in Patran as Plate Offset on Property Menu.**



COMPOSITE PLATE OFFSET Z0

- If your grid coordinates are on the surface of the composite manufacturing tool, use $Z0=0.0$ or $Z0=-T$, depending on the direction of the element normal.
- This results in a correctly offset element representing the correct location of the composite plate.



CQUAD4 AND CTRIA3 CORNER THICKNESSES

- **Ti are thicknesses at grids Gi**
 - if TFLAG is blank or 0, then Ti are actual user-specified thicknesses
 - If TFLAG is equal to 1, then Ti are fractions of the sum of the ply thicknesses on the PCOMP bulk data entry

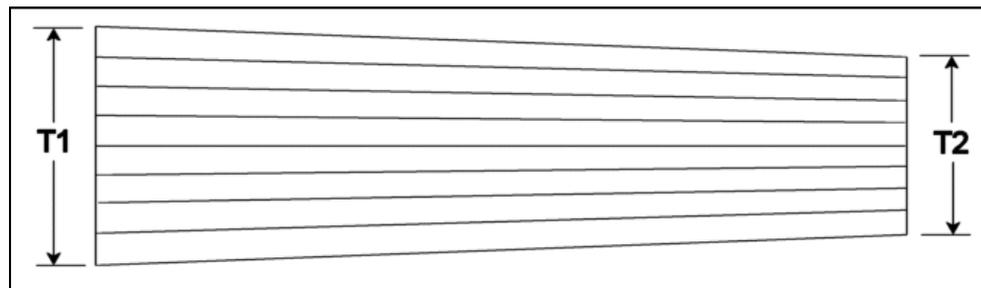
1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	MCID	ZOFFS	
CQUAD4	1	1	1	2	5	4	99		
		TFLAG	T1	T2	T3	T4			
			0.05	0.04	0.04	0.05			

CQUAD4 AND CTRIA3 CORNER THICKNESSES

- The effect on the layup of specifying TFLAG=0 and a different T_i on the CQUAD4 and CTRIA3 than the PCOMP total thickness is to multiply each ply thickness by the factor:

$$T_{\text{ply}} = T_i / T$$

where T is the original laminate thickness (sum of plies) from the PCOMP bulk data entry.



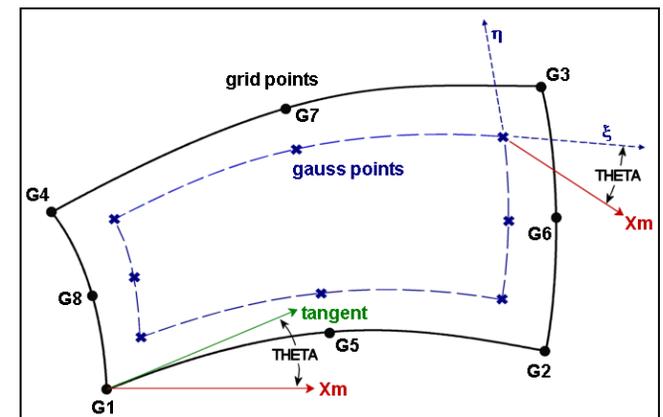
- Since plates are constant strain in the normal membrane directions, the effect of corner thicknesses tends to be the same as if the average of the corner thicknesses are used.
- There are differences for membrane shear strain and for bending because of linear moments and linear shear strains.

GENERAL MSC NASTRAN COMPOSITE INTERFACE INFORMATION

- **On the PCOMP bulk data entry:**
 - If the failure theorem is specified, FT, then the interlaminar shear stress allowable, Sb, is required.
 - Reference temperature, TREF, and material damping, GE, are used for all plies. Similar values on the MATi bulk data entries specified for each ply, MIDi, are ignored.
 - Ply MIDs must refer to either MAT1, MAT2, or MAT8 Bulk Data entries
 - Ply field replication:
 - The default for MID2, ..., MIDn is the last defined MIDi.
 - The same logic applies to Ti.
 - At least one of the four values (MIDi, Ti, THETAi, SOUTi) must be present for a ply to exist.
 - The minimum number of plies is one.
- **On the MAT8 bulk data entry:**
 - If G1Z and G2Z values are specified as zero or blank, then there is zero shear flexibility.

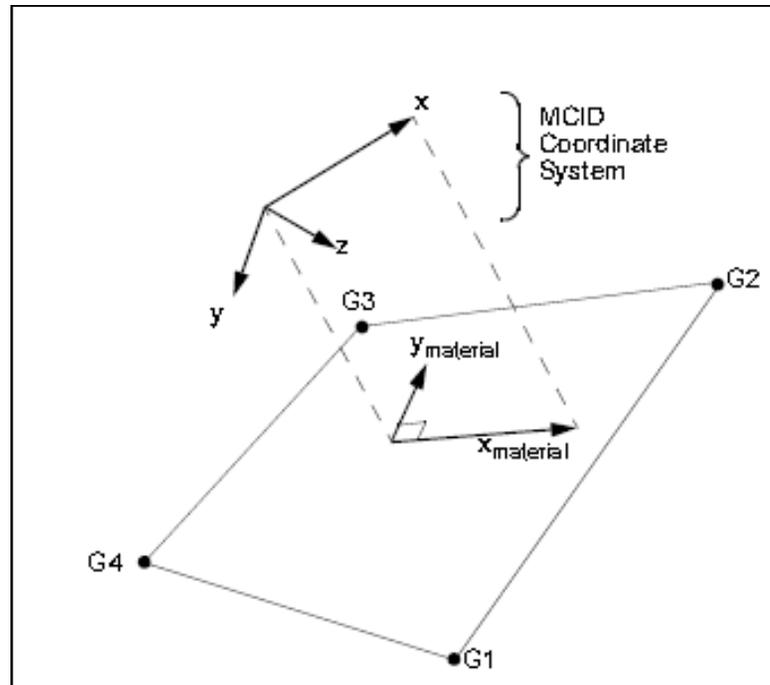
DO NOT USE QUAD8 OR TRIA6 FOR COMPOSITES

- Shell elements with mid-side nodes cannot be used with non-isotropic materials such as composites.
- Reasons for this:
 - The material coordinate system is element shape dependent.
 - The direction of the X axis of the material coordinate system (X_m) is found by projecting the MCID onto the element at grid G1. This is then used to calculate the angle THETA, between the edge tangent and X_m
 - Interior to the element, the direction of X_m is calculated at each gauss point by rotating THETA degrees from the x parametric axis.
 - For higher order elements, the X_m direction changes quadratically across the element based on edge curvature, which is not handled by the process for generating the element stiffness matrix.



DO NOT USE QUAD8 OR TRIA6 FOR COMPOSITES

- A QUAD4 element defines an isoparametric membrane-bending or plane strain quadrilateral plate element.
- All of the four edges of the element are straight, the material orientation is consistent across the element, and a stiffness matrix can be generated.

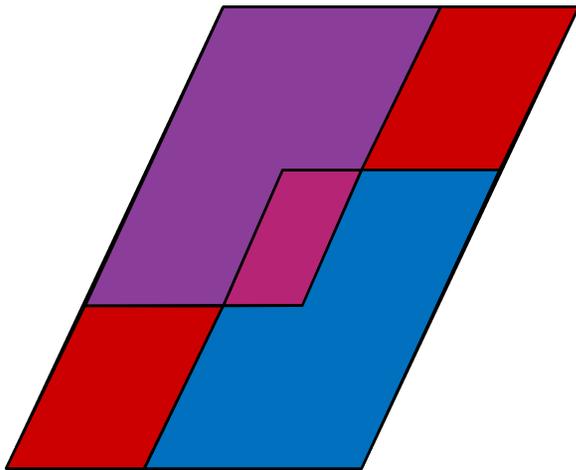


GLOBAL COMPOSITE PLY TRACKING

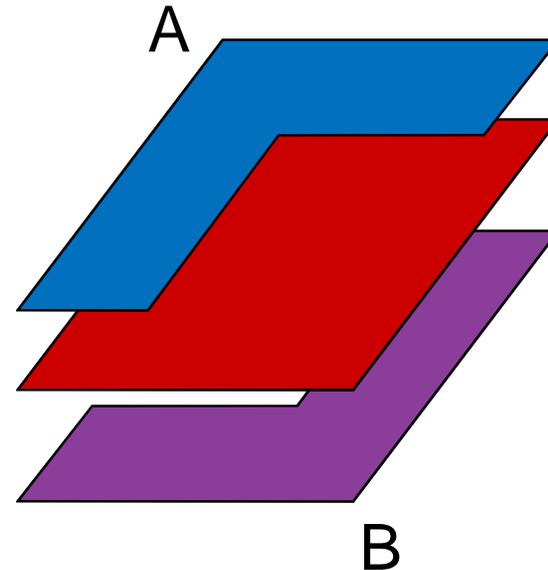
- **Idealization of large composite panel requires each ply within a panel be accurately modeled**
- **Adjacent element**
 - may contain different number of plies
 - May contain plies that are not continuous in terms of ply numbering scheme
- **Zone description traditionally used by analysts**
 - e.g., PCOMP
- **Ply description needed by manufacturers**

GLOBAL COMPOSITE PLY TRACKING

Zones—typical analysis



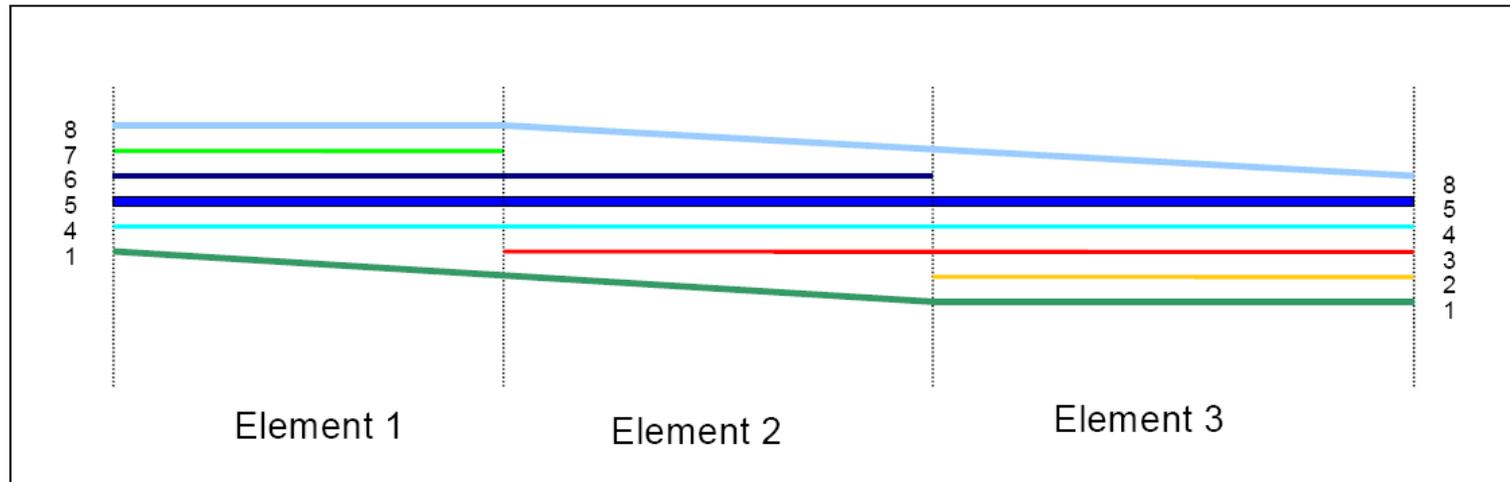
Plies—typical manufacturing



- **Note that the red ply is on the bottom (layer 1) in corner A and the top (layer 2) in corner B. This makes it laborious to manually identify and consistently post process results for each ply.**

GLOBAL COMPOSITE PLY TRACKING

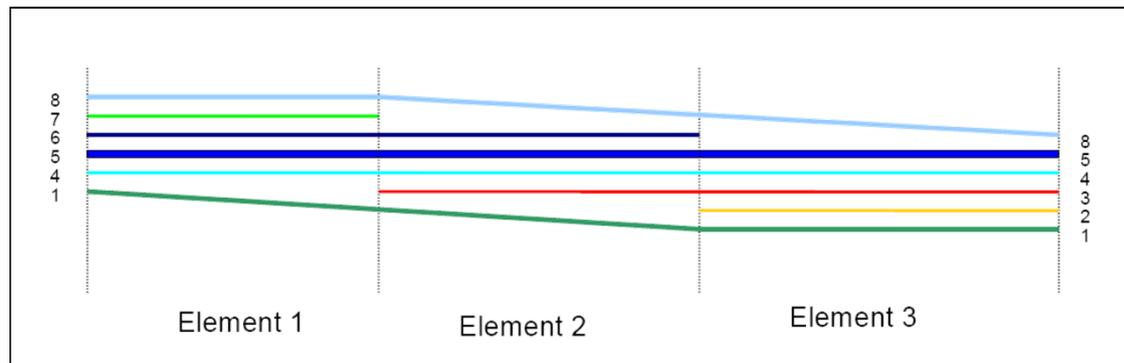
- Global ply ID (PCOMPG) addresses this issue
- Can also sort global ply ID for a given element set—GPRSORT Case Control command
- Example: The three elements modeled below have plies 1, 4, 5, and 8 in common. Plies 2, 3, 6, and 7 are in some of the elements but not others.



GLOBAL COMPOSITE PLY TRACKING

EXAMPLE (cont.):

- One can consider it as internal ID (PCOMP) and external ID (PCOMPG)
- This allows the display of ply results across elements with different property entries



INTERNAL PLY ID	GLOBAL PLY ID		
	Element 1	Element 2	Element 3
1	1	1	1
2	4	3	2
3	5	4	3
4	6	5	4
5	7	6	5
6	8	8	8

GLOBAL PLY ID - PCOMPG

- In the example below, element 1 references a property with 8 plies, and element 3 a property with 7 plies because ply # 5 drops off
- Displaying results for ply #7 on elements 1 and 3 would erroneously combine 45.0 degree ply results with 0.0 degree ply results
- If global ply ids are defined as below, then displaying global ply id 18 results would correctly combine 0.0 degree ply results.

element 1					element 3				
ply#	global plyid	matid	thickness	angle	ply#	global plyid	matid	thickness	angle
8	18	1	0.0054	0.0	7	18	1	0.0054	0.0
7	17	1	0.0054	45.0	6	17	1	0.0054	45.0
6	16	1	0.0054	-45.0	5	16	1	0.0054	-45.0
5	15	1	0.0054	90.0	4	15	1	0.0054	90.0
4	14	1	0.0054	90.0	3	13	1	0.0054	-45.0
3	13	1	0.0054	-45.0	2	12	1	0.0054	45.0
2	12	1	0.0054	45.0	1	11	1	0.0054	0.0
1	11	1	0.0054	0.0					

GLOBAL PLY ID – PCOMPG

- Same as PCOMP but
 - One ply per continuation line
 - Global ply ID added (GPLYIDn)
- Global ply ids define physical ply definitions for post processing.
- All ply output refers to global ply id instead of ply number.
- Used where ply drop-off occurs.
- Not available for optimization (SOL 200)

.bdf file extract

```
PCOMPG, 1,,, 5000., HILL
, 11, 1, .0054, 0., YES
, 12, 1, .0054, 45., YES
, 13, 1, .0054, -45., YES
, 15, 1, .0054, 90., YES
, 16, 1, .0054, -45., YES
, 17, 1, .0054, 45., YES
, 18, 1, .0054, 0., YES
```

1	2	3	4	5	6	7	8	9	10
PCOMPG	PID	Z0	NSM	SB	FT	TREF	GE	LAM	
PCOMPG	1			5000.0	HILL	0.0			
	GPLYID1	MID1	T1	THETA1	SOUT1				
	11	1	0.0054	0.0	YES				
	GPLYID2	MID2	T2	THETA2	SOUT2				
	13	1	0.0054	90.0					

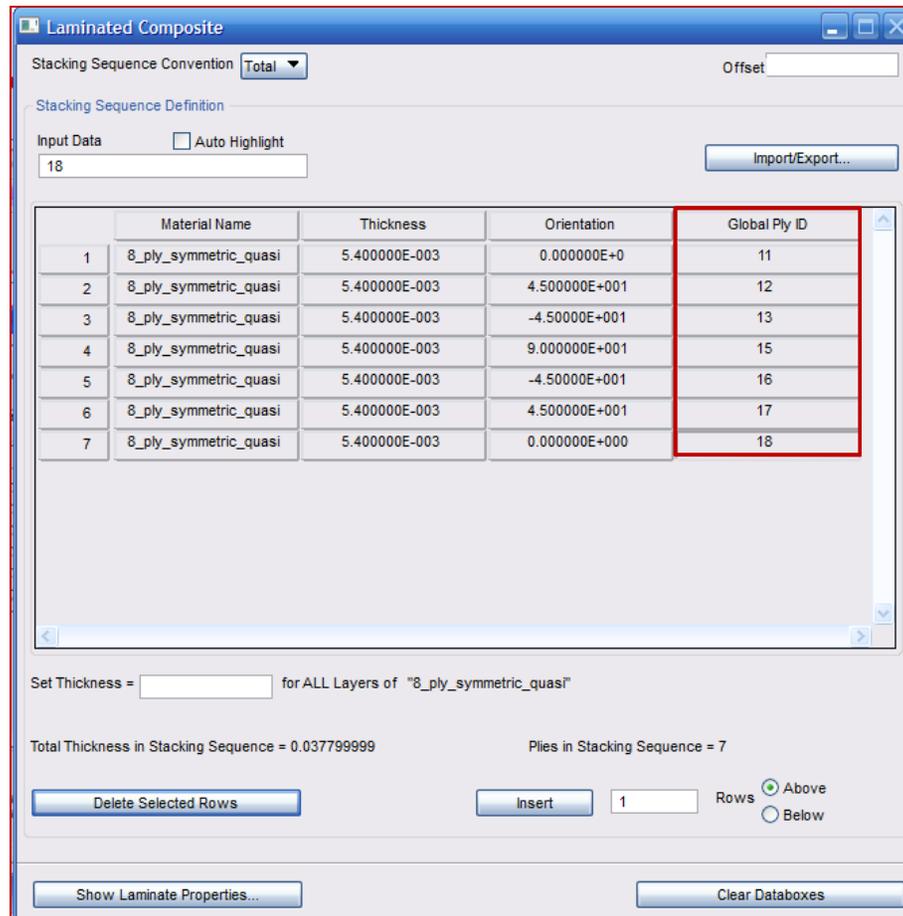
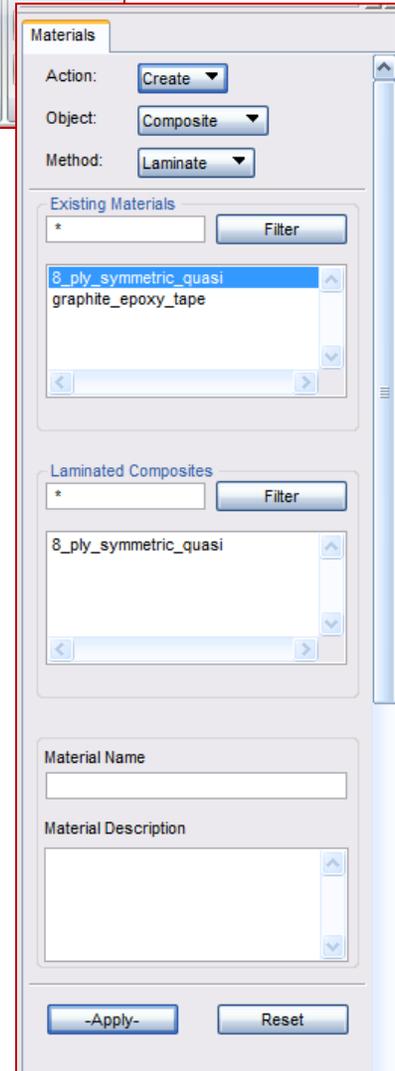
GLOBAL PLY ID – PCOMPG

- **GPRSORT case control command sorts to global ply id, not element**
 - GPRSORT=n, where n is Case Control SET command containing an element list or =ALL
 - Sorted by global ply ID

FAILURE INDICES FOR LAYERED COMPOSITE ELEMENTS						
GLOBAL PLY ID	FAILURE THEORY	ELEMENT ID	FP=FAILURE INDEX FOR PLY (DIRECT STRESSES/STRAINS)	FB=FAILURE INDEX FOR BONDING (INTER-LAMINAR STRESSES)	MAX OF FP,FB FOR ALL ELEMENTS REFERENCED BY GLOBAL PLY	FLAG
11	HILL	1	14.6234			
	HILL	2	5.5536	0.0000		
	HILL	3	2.8625	0.0000		
	HILL	4	1.8783	0.0000		
12	HILL	1	15.2880		14.6234	***
	HILL	2	4.3704	0.0000		
	HILL	3	2.1801	0.0000		
	HILL	4	2.3087	0.0000		
13	HILL	1	19.0313		15.2880	***
	HILL	2	4.5190	0.0000		
	HILL	3	2.4729	0.0000		
	HILL	4	2.7306	0.0000		

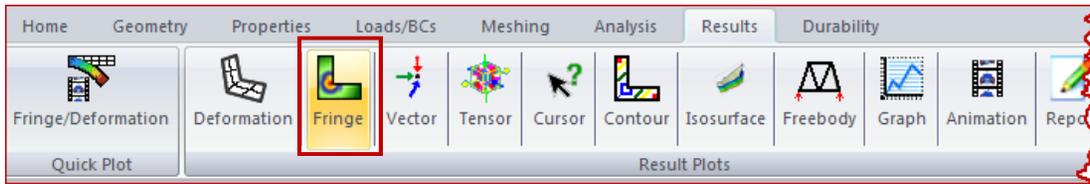
.f06 file excerpt

PATRAN - GLOBAL PLY ID

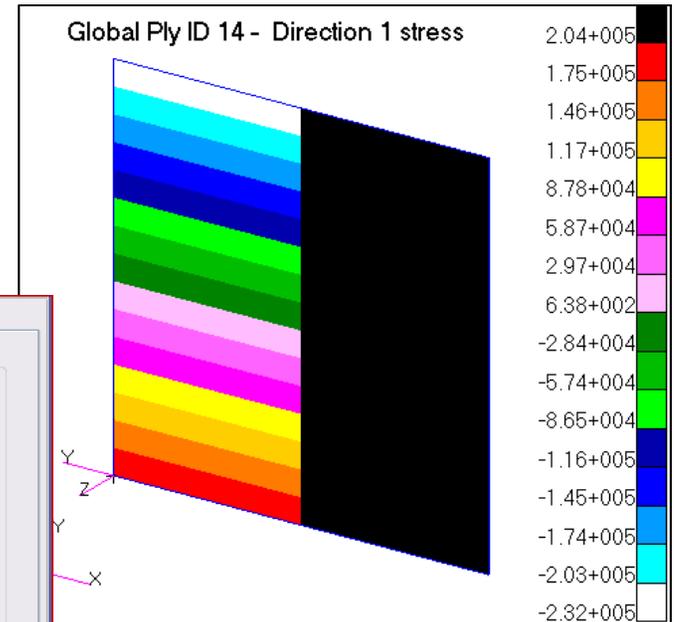
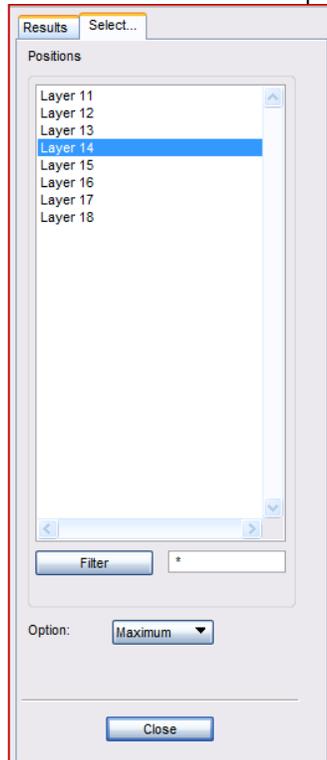
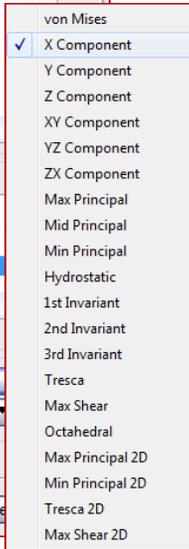
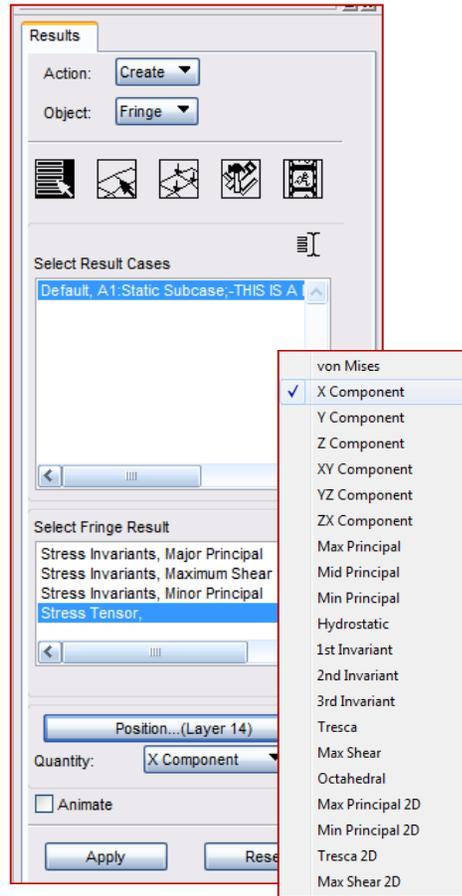


Materials:
 Modify/ Composite/
 Laminate
 Click on first row of
 Global Ply ID
 column
 Enter "11 12 13 15
 16 17 18" for
 Global Ply IDs
 Select Load Text
 Into Spreadsheet
 Apply

PATRAN - GLOBAL PLY ID



Results:
 Create/ Fringe
 Result/
 "Stress Tensor"
 Position/
 "Layer 14"
 Quantity/
 "X Component"
 Apply

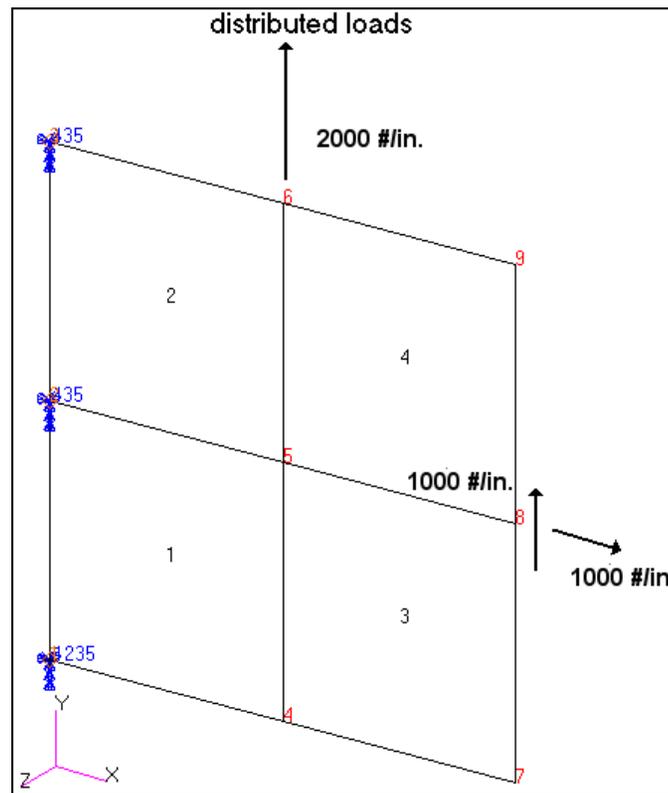


PATRAN - GLOBAL PLY ID

- **In the previous fringe plot**
 - Since global ply id 14 is only in elements 1 and 2 due to ply drop-off, Elements 3 and 4 have no information.
- **Note that whenever PCOMPG is used in a MSC Nastran run that Patran will have global ply ids for position instead of ply #s.**
 - You should specify a global ply id for all layers.
 - Do not mix global ply ids and ply #s (PCOMPs and PCOMPGs).

EXERCISE

- Perform WS4 “Ply Direction Tailoring for Strength” in your exercise workbook



PCOMP(G) LAM OPTIONS

- **Special Laminate options**
 - SYM (symmetric), PCOMP only
 - MEM (membrane only)
 - BEND (bending only)
 - SMEAR (smear properties)
 - SMCORE (smear face sheets with core)

PCOMP(G) LAM OPTIONS

- **SYM (symmetric) (PCOMP only)**
 - Makes the lay-up symmetric about the outer edge of the last layer
 - MSC Nastran automatically doubles the number of plies
- **MEM (membrane only, MID1)**
 - No bending or shear terms
 - Similar to CSHEAR in that it is not self supporting
 - Must have other elements with it to support bending loads
 - Not suggested for general use
- **BEND (bending only, MID2)**
 - No membrane or shear terms
 - Similar to PSHELL if only MID2 is used
 - Must have other elements with it to support bending loads
 - Not suggested for general use

PCOMP(G) LAM OPTIONS

- **SMEAR - smeared properties**
 - Used for preliminary design
 - All layers must be defined
 - Stacking sequence ignored
 - Assumes MID1 = MID2
 - MID3 and MID4 are not used
 - No transverse shear flexibility available
 - No membrane-bending coupling available
 - This option is used so that the optimization results are not affected by the position of the plies in the lay-up.
 - It assumes that either:
 - Many layers are used.
 - Bending stiffness difference between actual layup and smeared layup is insignificant.
 - Ply responses not available, only homogeneous results are available

PCOMP(G) LAM OPTIONS

- **SMCORE - smeared face sheets with core**
 - $t_{\text{face}} = t_1 + t_2 + \dots + t_{n-1}$ (total thickness of face sheets)
 - $t_{\text{core}} = t_n$ (last layer, total core thickness)
 - All plies must be specified
 - Stiffness matrix calculation based on
 - Half the face sheet thicknesses placed above core
 - Half the face sheet thicknesses placed below core
 - Allows capturing effects due to core offset
 - Stacking sequence ignored
 - Ply responses not available, only homogeneous results are available

PCOMP(G) LAM OPTIONS

- **Special LAM options must be used with care and with full understanding**
- **Primary use for SMEAR and SMCORE options are for quick preliminary design**

SECTION 6

ADVANCED FAILURE THEORY & PREDICTION

ADVANCED COMPOSITE FAILURE METHODS

- **In this section, we will discuss 3 advanced composite failure methods**
 - Progressive Failure Analysis (PFA)
 - Virtual Crack Closure Technique (VCCT)
 - Cohesive Zone Modeling (CZM)

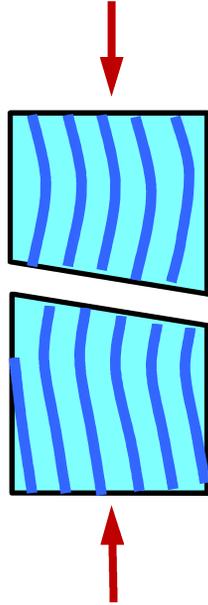
PROGRESSIVE COMPOSITE FAILURE

FAILURE MECHANISMS OF UNI-DIRECTIONAL LAMINA



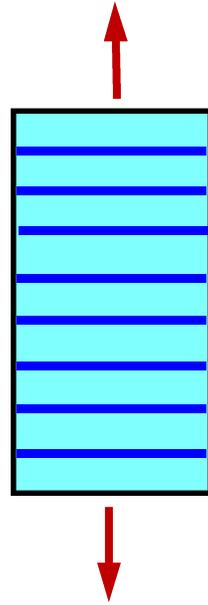
Longitudinal Tension

Fiber Failure



Longitudinal Compression

Fiber Failure



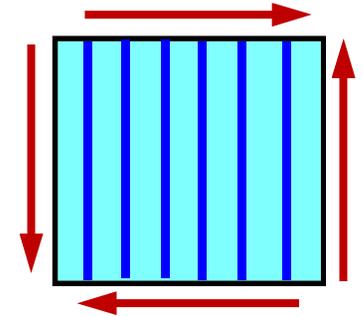
Transverse Tension

Matrix Failure



Transverse Compression

Matrix Failure



In-Plane Shear

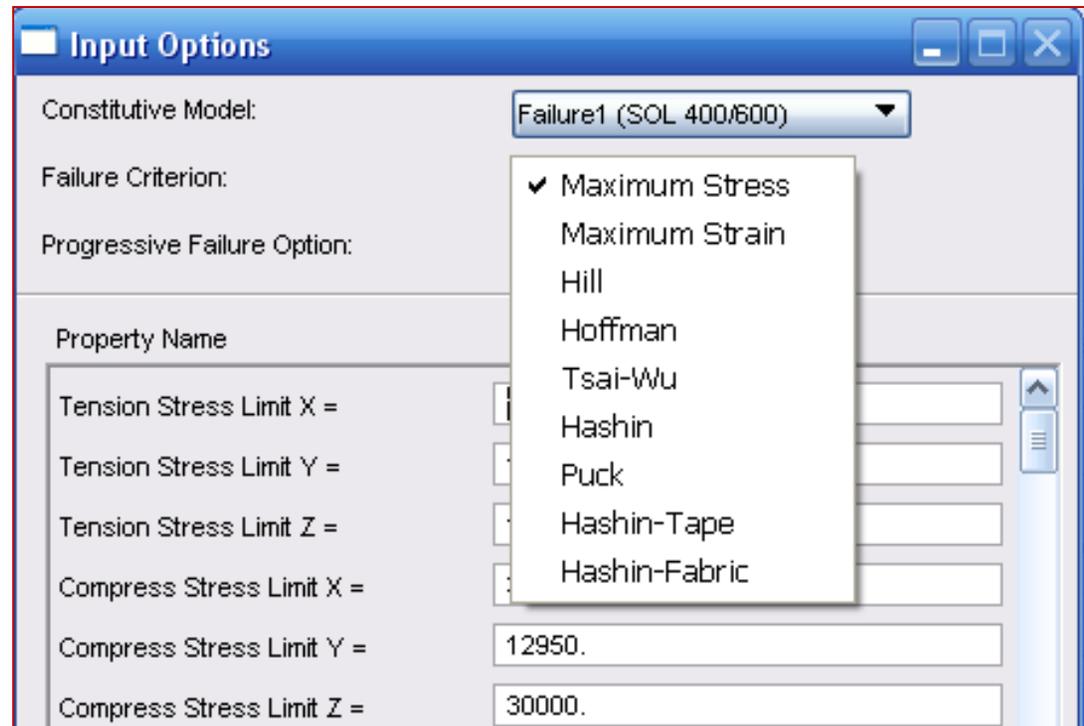
Fiber-Matrix Interface Failure

COMPOSITE ANALYSIS

- **Traditional Composite Analysis**
 - Linear analysis
 - Compute ply failure index and strength ratio
 - Compute bonding failure index and strength ratio
- **Incentives for Going Beyond Linear Analysis**
 - Evaluate the ultimate capacity of a composite structure by progressively failing the plies
 - Simulate delamination/de-bonding
 - Simulate crack propagation

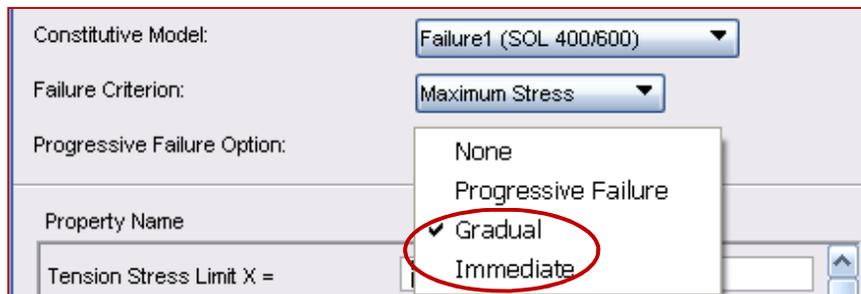
COMPOSITE FAILURE CRITERIA

- **Composite Failure**
 - Most of the criteria are semi-empirical in nature
- **Composite Failure on Layer Basis**
 - Maximum Stress
 - Maximum Strain
 - Hill
 - Hoffman
 - Tsai-Wu
 - Hashin
 - Puck
 - Hashin-Tape
 - Hashin-Fabric
 - User defined (UFAIL)



PROGRESSIVE FAILURE ANALYSIS (PFA)

- **Progressive Composite Failure**
 - Flagged through the MATF bulk data (ITYPE = 2 or 3)
 - Up to three failure criteria can be selected
 - Only the primary failure criterion is used for PFA
 - The other two are only used to calculate failure indices
 - The behavior up to the failure point is linear elastic
 - Upon failure...
 - When failure index is larger than one, degrade material moduli
 - Selective degradation – if matrix fails, do not change fiber properties
 - Stiffness drops gradually or immediately



MSC NASTRAN INPUT DATA FORMAT

MATF	MID	ITYPE							
	"CRI"	Criterion	V ₁ ¹	V ₂ ¹	V ₃ ¹	V ₄ ¹	V ₅ ¹	V ₆ ¹	1st
	V ₇ ¹	V ₈ ¹	V ₉ ¹	Find ¹	V ₁₀ ¹	V ₁₁ ¹	V ₁₂ ¹	W ₁ ¹	
	W ₂ ¹	W ₃ ¹	W ₄ ¹	W ₅ ¹	W ₆ ¹	W ₇ ¹	W ₈ ¹	W ₉ ¹	
	"PF"	A1	A2	A3	A4	A5			
	"CRI"	Criterion	V ₁ ²	V ₂ ²	V ₃ ²	V ₄ ²	V ₅ ²	V ₆ ²	2nd
	V ₇ ²	V ₈ ²	V ₉ ²	Find ²	V ₁₀ ²	V ₁₁ ²	V ₁₂ ²	W ₁ ²	
	W ₂ ²	W ₃ ²	W ₄ ²	W ₅ ²	W ₆ ²	W ₇ ²	W ₈ ²	W ₉ ²	
	"CRI"	Criterion	V ₁ ³	V ₂ ³	V ₃ ³	V ₄ ³	V ₅ ³	V ₆ ³	3rd
	V ₇ ³	V ₈ ³	V ₉ ³	Find ³	V ₁₀ ³	V ₁₁ ³	V ₁₂ ³	W ₁ ³	
	W ₂ ³	W ₃ ³	W ₄ ³	W ₅ ³	W ₆ ³	W ₇ ³	W ₈ ³	W ₉ ³	

- **ITYPE:** 0 – No PFA; 2 – Gradual Selective; 3 – Immediate Selective
- **Criterion:** 1 – Max. Stress; 2 – Max. Strain; 3 – Hill; 4 – Hoffman; 5 – Tsai-Wu; 7 – Hashin; 8 – Puck; 10 – Hashin-Tape; 11 – Hashin-Fabric; 13 – User Subroutine
- **PF:** Numerical factors controlling stiffness reduction

DEFINITIONS: STRENGTH COEFFICIENTS

Various coefficients are commonly used to describe strengths:

V_1	Tensile strength along the x-axis
V_2	Compressive strength along the x-axis
V_3	Tensile strength along the y-axis
V_4	Compressive strength along the y-axis
V_5	Tensile strength along the z-axis
V_6	Compressive strength along the z-axis
V_7	Shear strength in the XY plane
V_8	Shear strength in the YZ plane
V_9	Shear strength in the ZX plane
V_{10}	Interactive strength XY
V_{11}	Interactive strength YZ
V_{12}	Interactive strength ZX

F_{IND} Failure index scale factor

W_1	Tensile strain along the x-axis
W_2	Compressive strain along the x-axis
W_3	Tensile strain along the y-axis
W_4	Compressive strain along the y-axis
W_5	Tensile strain along the z-axis
W_6	Compressive strain along the z-axis
W_7	Shear strain in the XY plane
W_8	Shear strain in the YZ plane
W_9	Shear strain in the ZX plane

MAXIMUM STRESS FAILURE CRITERION

- **Directly compares individual stress components and specified critical stresses**
- **Six failure indices (FI) are calculated**
 - The strength ratios (SR) are the reciprocals of the corresponding failure indices
- **Ignores interaction between loads in different directions. No direct strength coupling exists between normal strength and shear strength, and hence the criterion needs to be applied with great care for stress states other than uniaxial**

MAXIMUM STRESS FAILURE CRITERION

$$\begin{aligned}
 &1. \left\{ \begin{array}{ll} \left(\frac{\sigma_1}{X_t} \right) & \text{if } \sigma_1 > 0 \\ \left(\frac{\sigma_1}{X_c} \right) & \text{if } \sigma_1 < 0 \end{array} \right. \\
 &2. \left\{ \begin{array}{ll} \left(\frac{\sigma_2}{Y_t} \right) & \text{if } \sigma_2 > 0 \\ \left(\frac{\sigma_2}{Y_c} \right) & \text{if } \sigma_2 < 0 \end{array} \right. \\
 &3. \left\{ \begin{array}{ll} \left(\frac{\sigma_3}{Z_t} \right) & \text{if } \sigma_3 > 0 \\ \left(\frac{\sigma_3}{Z_c} \right) & \text{if } \sigma_3 < 0 \end{array} \right. \\
 &4. \left(\frac{|\sigma_{12}|}{S_{12}} \right) \\
 &5. \left(\frac{|\sigma_{23}|}{S_{23}} \right) \\
 &6. \left(\frac{|\sigma_{31}|}{S_{31}} \right)
 \end{aligned}$$

Patran Form

Tension Stress Limit X =	<input type="text" value="V<sub>1</sub>"/>
Tension Stress Limit Y =	<input type="text" value="V<sub>3</sub>"/>
Tension Stress Limit Z =	<input type="text" value="V<sub>5</sub>"/>
Compress Stress Limit X =	<input type="text" value="V<sub>2</sub>"/>
Compress Stress Limit Y =	<input type="text" value="V<sub>4</sub>"/>
Compress Stress Limit Z =	<input type="text" value="V<sub>6</sub>"/>
Shear Stress Limit XY =	<input type="text" value="V<sub>7</sub>"/>
Shear Stress Limit YZ =	<input type="text" value="V<sub>8</sub>"/>
Shear Stress Limit ZX =	<input type="text" value="V<sub>9</sub>"/>

X_t, X_c are the maximum allowable stresses in the 1-direction in tension and compression. (V_1, V_2)
 Y_t, Y_c are maximum allowable stresses in the 2-direction in tension and compression. (V_3, V_4)
 Z_t, Z_c are maximum allowed stresses in the 3-direction in tension and compression. (V_5, V_6)
 S_{12} maximum allowable in-plane shear stress. (V_7)
 S_{23} maximum allowable 23 shear stress. (V_8)
 S_{31} maximum allowable 31 shear stress. (V_9)

MAXIMUM STRAIN FAILURE CRITERION

- **Analogous to stress criterion, but uses strain components in direct comparison to strain allowables**
- **Six failure indices are calculated**
 - The strength ratios are the reciprocals of the corresponding failure indices
- **No direct strength coupling exists between normal strain and shear strain**
- **Failure strains in the different loading directions vary less than failure stresses do, which can make this criterion more stable**

MAXIMUM STRAIN FAILURE CRITERION

Patran Form

$$1. \begin{cases} \left(\frac{\epsilon_1}{\epsilon_{1t}} \right) & \text{if } \epsilon_1 > 0 \\ \left(-\frac{\epsilon_1}{\epsilon_{1c}} \right) & \text{if } \epsilon_1 < 0 \end{cases}$$

$$2. \begin{cases} \left(\frac{\epsilon_2}{\epsilon_{2t}} \right) & \text{if } \epsilon_2 > 0 \\ \left(-\frac{\epsilon_2}{\epsilon_{2c}} \right) & \text{if } \epsilon_2 < 0 \end{cases}$$

$$3. \begin{cases} \left(\frac{\epsilon_3}{\epsilon_{3t}} \right) & \text{if } \epsilon_3 > 0 \\ \left(-\frac{\epsilon_3}{\epsilon_{3c}} \right) & \text{if } \epsilon_3 < 0 \end{cases}$$

$$4. \left(\frac{\gamma_{12}}{\xi_{12}} \right) \quad \epsilon_{1t}, \epsilon_{1c}$$

are the maximum allowable strains in the 1 direction in tension and compression. (W_1, W_2)

$\epsilon_{2t}, \epsilon_{2c}$ are the maximum allowable strains in the 2 direction in tension and compression. (W_3, W_4)

$$5. \left(\frac{\gamma_{23}}{\xi_{23}} \right) \quad \epsilon_{3t}, \epsilon_{3c}$$

are the maximum allowable strains in the 3 direction in tension and compression. (W_5, W_6)

ξ_{12} is the maximum allowable shear strain in the 12 plane. (W_7)

$$6. \left(\frac{\gamma_{31}}{\xi_{31}} \right) \quad \xi_{23}$$

is the maximum allowable shear strain in the 23 plane. (W_8)

ξ_{31} is the maximum allowable shear strain in the 31 plane. (W_9)

Tension Strain Limit X =	<input type="text" value="W1"/>
Tension Strain Limit Y =	<input type="text" value="W3"/>
Tension Strain Limit Z =	<input type="text" value="W5"/>
Compress Strain Limit X =	<input type="text" value="W2"/>
Compress Strain Limit Y =	<input type="text" value="W4"/>
Compress Strain Limit Z =	<input type="text" value="W6"/>
Shear Strain Limit XY =	<input type="text" value="W7"/>
Shear Strain Limit YZ =	<input type="text" value="W8"/>
Shear Strain Limit ZX =	<input type="text" value="W9"/>

HILL FAILURE CRITERION

- Tensile and compressive failure stress allowables are identical
- Orthotropic materials only– no anisotropic materials (MAT9)
- One failure index (FI) value is calculated
 - The strength ratio (SR) is:

$$SR = 1 / \sqrt{FI}$$

- Assumes that a superposed hydrostatic stress has no influence on failure

HILL FAILURE CRITERION

$$FI = \left\{ \left[\frac{\sigma_1^2}{X^2} + \frac{\sigma_2^2}{Y^2} + \frac{\sigma_3^2}{Z^2} - \left(\left[\frac{1}{X^2} + \frac{1}{Y^2} - \frac{1}{Z^2} \right] \sigma_1 \sigma_2 - \left[\frac{1}{X^2} + \frac{1}{Z^2} - \frac{1}{Y^2} \right] \sigma_1 \sigma_3 - \left[\frac{1}{Y^2} + \frac{1}{Z^2} - \frac{1}{X^2} \right] \sigma_2 \sigma_3 + \frac{\sigma_{12}^2}{S_{12}^2} + \frac{\sigma_{13}^2}{S_{13}^2} + \frac{\sigma_{23}^2}{S_{23}^2} \right) \right\} / F$$

- X maximum allowable stress in the 1 direction (V₁)
- Y maximum allowable stress in the 2 direction (V₃)
- Z maximum allowable stress in the 3 direction (V₅)
- S₁₂ maximum allowable in-plane shear stress. (V₇)
- S₂₃ maximum allowable 23 shear stress. (V₈)
- S₃₁ maximum allowable 31 shear stress. (V₉)
- F failure index scale factor (F_{IND})

Tension Stress Limit X =	<input type="text" value="V<sub>1</sub>"/>
Tension Stress Limit Y =	<input type="text" value="V<sub>3</sub>"/>
Tension Stress Limit Z =	<input type="text" value="V<sub>5</sub>"/>
Compress Stress Limit X =	<input type="text"/>
Compress Stress Limit Y =	<input type="text"/>
Compress Stress Limit Z =	<input type="text"/>
Shear Stress Limit XY =	<input type="text" value="V<sub>7</sub>"/>
Shear Stress Limit YZ =	<input type="text" value="V<sub>8</sub>"/>
Shear Stress Limit ZX =	<input type="text" value="V<sub>9</sub>"/>
Failure Index =	<input type="text" value="F<sub>IND</sub>"/>

Patran Form

HOFFMANN FAILURE CRITERION

- The Hoffmann criterion is essentially Hill criterion modified to allow unequal maximum allowable stresses in tension and compression

$$FI = \left[\begin{aligned} &C_1(\sigma_2 - \sigma_3)^2 + C_2(\sigma_3 - \sigma_1)^2 + C_3(\sigma_1 - \sigma_2)^2 \\ &+ C_4\sigma_1 + C_5\sigma_2 + C_6\sigma_3 \\ &+ C_7\sigma_{23}^2 + C_8\sigma_{13}^2 + C_9\sigma_{12}^2 \end{aligned} \right] / F$$

X_t, X_c are the maximum allowable stresses in the 1-direction in tension and compression. (V_1, V_2)

Y_t, Y_c are maximum allowable stresses in the 2-direction in tension and compression. (V_3, V_4)

Z_t, Z_c are maximum allowed stresses in the 3-direction in tension and compression. (V_5, V_6)

S_{12} maximum allowable in-plane shear stress. (V_7)

S_{23} maximum allowable 23 shear stress. (V_8)

S_{31} maximum allowable 31 shear stress. (V_9)

F failure index scale factor (F_{IND})

$$C_1 = \frac{1}{2} \left(\frac{1}{Z_t Z_c} + \frac{1}{Y_t Y_c} - \frac{1}{X_t X_c} \right)$$

$$C_2 = \frac{1}{2} \left(\frac{1}{X_t X_c} + \frac{1}{Z_t Z_c} - \frac{1}{Y_t Y_c} \right)$$

$$C_3 = \frac{1}{2} \left(\frac{1}{X_t X_c} + \frac{1}{Y_t Y_c} - \frac{1}{Z_t Z_c} \right)$$

$$C_4 = \frac{1}{X_t} - \frac{1}{X_c}$$

$$C_5 = \frac{1}{Y_t} - \frac{1}{Y_c}$$

$$C_6 = \frac{1}{Z_t} - \frac{1}{Z_c}$$

$$C_7 = \frac{1}{S_{23}^2}$$

$$C_8 = \frac{1}{S_{13}^2}$$

$$C_9 = \frac{1}{S_{12}^2}$$

TSAI-WU FAILURE CRITERION

- Tsai-Wu is a tensor polynomial failure criterion
- One failure index is calculated
- Accounts for different tensile and compressive strengths by incorporating linear terms
- As for the 2D Tsai-Wu theory, uses interaction constants F_{12} , F_{23} and F_{31} that can be used to “tune” the response
 - For the failure surface to be closed, the interaction constants are constrained by the below limitations:

$$F_{12}^2 < \frac{1}{X_t X_c} \bullet \frac{1}{Y_t Y_c} \quad F_{23}^2 < \frac{1}{Y_t Y_c} \bullet \frac{1}{Z_t Z_c} \quad F_{31}^2 < \frac{1}{X_t X_c} \bullet \frac{1}{Z_t Z_c}$$

TSAI-WU FAILURE CRITERION

$$FI = \left[\left(\frac{1}{X_t} - \frac{1}{X_c} \right) \sigma_1 + \left(\frac{1}{Y_t} - \frac{1}{Y_c} \right) \sigma_2 + \left(\frac{1}{Z_t} - \frac{1}{Z_c} \right) \sigma_3 + \frac{\sigma_1^2}{X_t X_c} + \frac{\sigma_2^2}{Y_t Y_c} + \frac{\sigma_3^2}{Z_t Z_c} + \frac{\tau_{12}^2}{S_{12}^2} + \frac{\tau_{23}^2}{S_{23}^2} + \frac{\tau_{13}^2}{S_{13}^2} + 2F_{12} \sigma_1 \sigma_2 + 2F_{23} \sigma_2 \sigma_3 + 2F_{13} \sigma_1 \sigma_3 \right] / F$$

X_t, X_c are the maximum allowable stresses in the 1-direction in tension and compression. (V_1, V_2)

Y_t, Y_c are maximum allowable stresses in the 2-direction in tension and compression. (V_3, V_4)

Z_t, Z_c are maximum allowed stresses in the 3-direction in tension and compression. (V_5, V_6)

S_{12} maximum allowable in-plane shear stress. (V_7)

S_{23} maximum allowable 23 shear stress. (V_8)

S_{31} maximum allowable 31 shear stress. (V_9)

F_{12} Interactive strength constant for the 12 plane (V_{10})

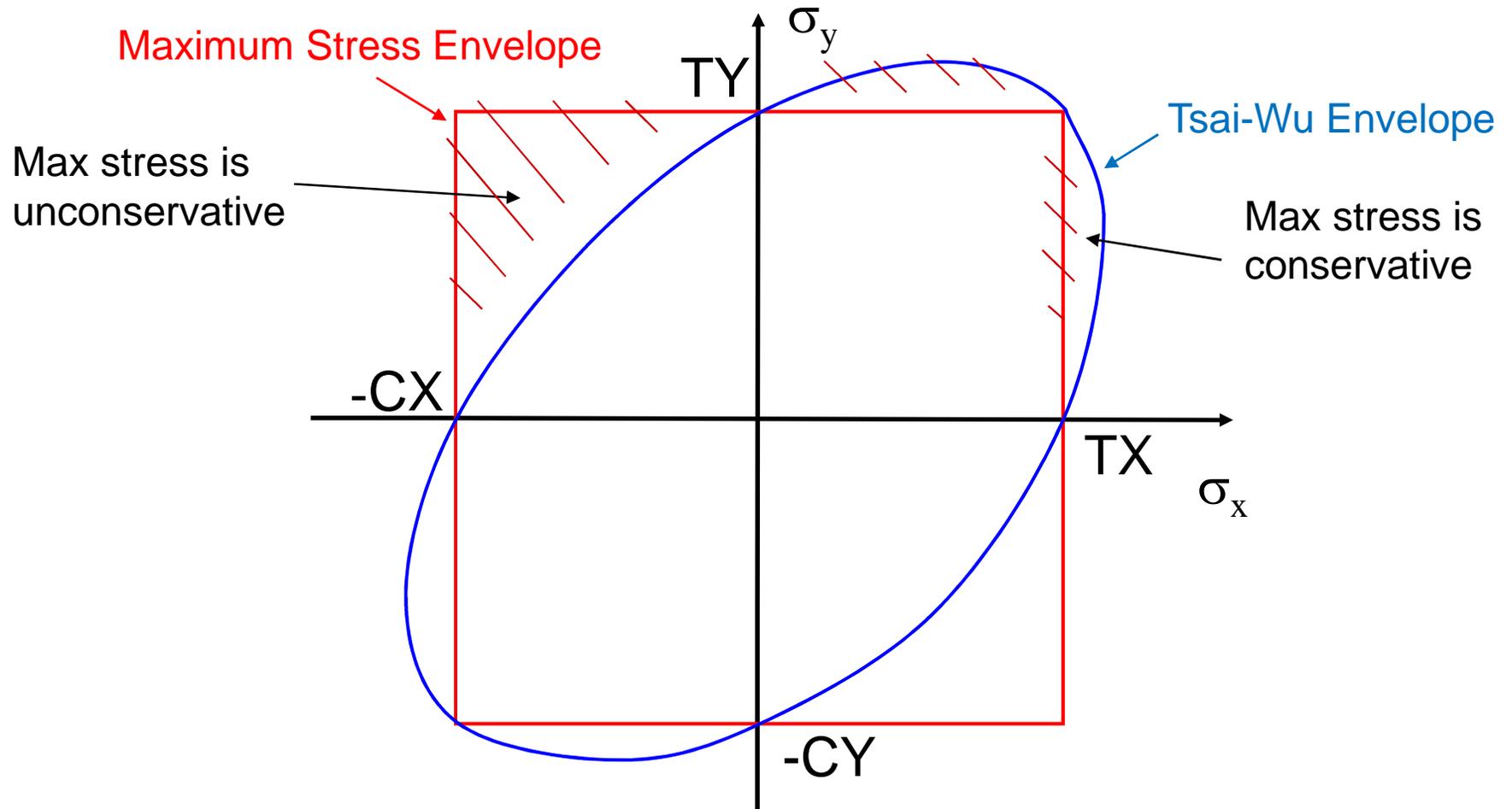
F_{23} Interactive strength constant for the 23 plane (V_{11})

F_{13} Interactive strength constant for the 31 plane (V_{12})

F failure index scale factor (F_{IND})

Tension Stress Limit X =	<input type="text" value="V<sub>1</sub>"/>
Tension Stress Limit Y =	<input type="text" value="V<sub>3</sub>"/>
Tension Stress Limit Z =	<input type="text" value="V<sub>5</sub>"/>
Compress Stress Limit X =	<input type="text" value="V<sub>2</sub>"/>
Compress Stress Limit Y =	<input type="text" value="V<sub>4</sub>"/>
Compress Stress Limit Z =	<input type="text" value="V<sub>6</sub>"/>
Shear Stress Limit XY =	<input type="text" value="V<sub>7</sub>"/>
Shear Stress Limit YZ =	<input type="text" value="V<sub>8</sub>"/>
Shear Stress Limit ZX =	<input type="text" value="V<sub>9</sub>"/>
Shear Stress Bond (SB) =	<input type="text"/>
Failure Index =	<input type="text" value="F<sub>IND</sub>"/>
Interactive Strength XY =	<input type="text" value="V<sub>10</sub>"/>
Interactive Strength YZ =	<input type="text" value="V<sub>11</sub>"/>
Interactive Strength ZX =	<input type="text" value="V<sub>12</sub>"/>

COMPARISON BETWEEN MAXIMUM STRESS & TSAI-WU



HASHIN FAILURE CRITERION

- **Distinguishes between fiber failure and matrix failure**
- **Four failure indices are calculated:**
 - **Fiber tension**
 - **Fiber compression**
 - **Matrix tension**
 - **Matrix compression**
- **Simple criterion with fewer input parameters**

HASHIN FAILURE CRITERION

- Fiber failure**

1st failure index:
$$\left(\frac{\sigma_{11}}{X_t}\right)^2 + \frac{1}{S^2}(\sigma_{12}^2 + \sigma_{13}^2)$$
 Tension fiber mode,
 Compressive fiber mode, $\sigma_1 < 0$

2nd failure index:
$$\frac{|\sigma_{11}|}{X_c}$$

- Matrix failure**

3rd failure index:
$$\left[\frac{1}{Y_t}(\sigma_2 + \sigma_3)^2 + \frac{1}{S_{23}^2}(\sigma_{23}^2 - \sigma_2\sigma_3) + \frac{1}{S_{12}^2}(\sigma_{12}^2 + \sigma_{13}^2)\right]$$
 Tensile matrix mode, $\sigma_2 + \sigma_3 > 0$
 Compressive matrix mode, $\sigma_2 + \sigma_3 < 0$

4th failure index:
$$\left[\frac{1}{Y_c}\left(\left(\frac{Y_c}{2S_{23}}\right)^2 - 1\right)(\sigma_2 + \sigma_3) + \frac{1}{4S_{23}^2}(\sigma_2 + \sigma_3)^2 + \frac{1}{S_{23}^2}(\sigma_{23}^2 - \sigma_2\sigma_3) + \frac{1}{S_{12}^2}(\sigma_{12}^2 + \sigma_{13}^2)\right]$$

HASHIN FAILURE CRITERION

- **Material properties:**

- X_t – maximum fiber tension (V_1)
- X_c – maximum fiber compression (V_2)
- Y_t – maximum matrix tension (V_3)
- Y_c – maximum matrix compression (V_4)
- S_{12} – layer shear strength (V_{10})
- S_{23} – transverse shear strength (V_{11})

The screenshot shows the 'Input Options' dialog box for the Hashin failure criterion. The 'Constitutive Model' is set to 'Failure1 (SOL 400/600)', the 'Failure Criterion' is 'Hashin', and the 'Progressive Failure Option' is 'Progressive Failure'. Below these are several input fields for material properties:

Property Name	Value
Max Fiber Tension =	<input type="text"/>
Max Fiber Compression =	<input type="text"/>
Max Matrix Tension =	<input type="text"/>
Max Matrix Compression =	<input type="text"/>
Layer Shear Strength =	<input type="text"/>
Transverse Shear Strength =	<input type="text"/>
Shear Stress Bond (SB) =	<input type="text"/>

Below the table are two empty text boxes for 'Temperature Dep/Model Variable Fields' and 'Current Constitutive Models'. At the bottom are 'OK', 'Clear', and 'Cancel' buttons.

PUCK FAILURE CRITERION

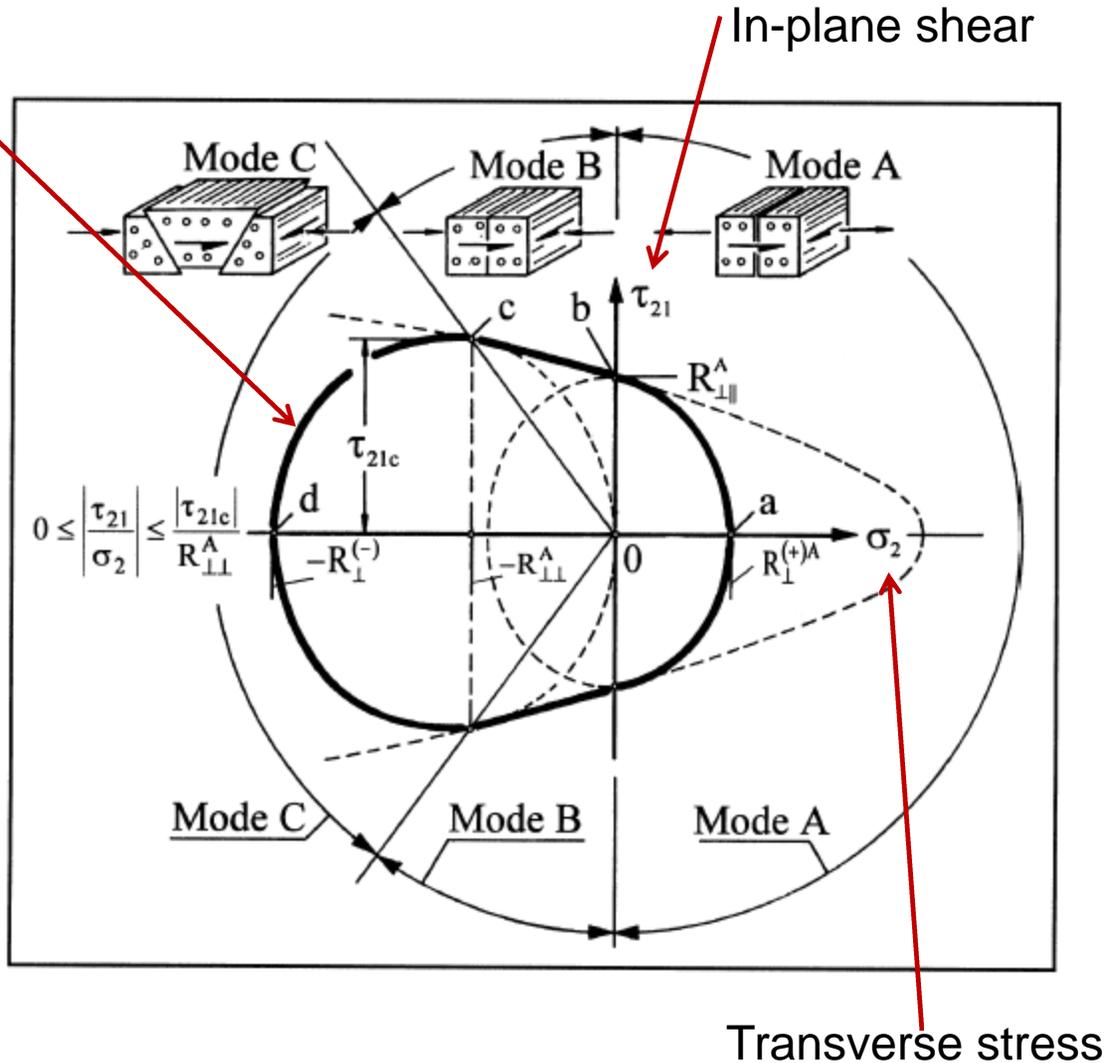
- **Distinguishes between fiber failure and matrix failure**
- **Five failure indices are calculated:**
 - Fiber tension
 - Fiber compression
 - Matrix, Mode A: Tension
 - Matrix, Mode B: Compression
 - Matrix, Mode C: Compressive wedge
- **Currently one of the more popular criteria**

PUCK FAILURE CRITERION

Failure envelope

- **Three modes of matrix failure**

- Mode A for transverse tension
- Mode B for high shear/low transverse compression
- Mode C for higher transverse compression with or without shear
- Mode C is the most dangerous due to “wec effect”, which can lead catastrophic failure.



From Puck, Shurmann: Failure Analysis of FRP Laminates by means of Phenomenological Models, Composites Science and Technology, 58 (1998), p. 1052.

PUCK FAILURE CRITERION

- Fiber failure**

1st failure index:

Tensile fiber mode, $\sigma_1 > 0$

$$\frac{\sigma_1}{X_t}$$

2nd failure index:

Compressive fiber mode, $\sigma_1 < 0$

$$\frac{|\sigma_1|}{X_c}$$

- Matrix failure**

3rd failure index:

Mode A, $\sigma_2 > 0$, $\theta_{fp} = 0$

$$\left[\sqrt{\left(\frac{\sigma_{12}}{S_{12}}\right)^2 + \left(1 - p_{12t} \frac{Y_t}{S_{12}}\right)^2 \left(\frac{\sigma_2}{Y_t}\right)^2} + p_{12t} \frac{\sigma_2}{S_{12}} \right]$$

Mode B, $\sigma_2 < 0$ and $0 \leq \left| \frac{\sigma_2}{\sigma_{12}} \right| \leq \frac{R^A}{\sigma_{21c}}$, $\theta_{fp} = 0$

4th failure index:

$$\left[\frac{1}{S_{12}} \left(\sqrt{\sigma_{12}^2 + (p_{12c} \sigma_2)^2} + p_{12c} \sigma_2 \right) \right]$$

Mode C, $\sigma_2 < 0$ and $0 \leq \left| \frac{\sigma_{12}}{\sigma_2} \right| \leq \frac{\sigma_{21c}}{R^A}$.

5th failure index:

$$\left[\left(\left(\frac{\sigma_{12}}{2(1 + p_{23c} S_{12})} \right)^2 + \left(\frac{\sigma_2}{Y_c} \right)^2 \right) \frac{Y_c}{|\sigma_2|} \right]$$

PUCK FAILURE CRITERION

- **Material properties:**

- X_t – maximum fiber tension (V_1)
- X_c – maximum fiber compression (V_2)
- Y_t – maximum matrix tension (V_3)
- Y_c – maximum matrix compression (V_4)
- S_{12} – layer shear strength (V_{10})
- p12c, p12t, p23c, p23t – slopes of the failure envelope (W_1, W_2, W_3, W_4)
 - p12c and p23c are dependent for plane stress, enter only one of them
 - p12t defaults to p12c and p23t to p23c
 - Typical values (carbon/epoxy): p12c=p12t=0.35; p23c=p23t=0.27

Max Fiber Tension =	<input type="text" value="V<sub>1</sub>"/>
Max Fiber Compression =	<input type="text" value="V<sub>2</sub>"/>
Max Matrix Tension =	<input type="text" value="V<sub>3</sub>"/>
Max Matrix Compression =	<input type="text" value="V<sub>4</sub>"/>
Layer Shear Strength =	<input type="text" value="V<sub>10</sub>"/>
Slope P12C of Fracture Envelope	<input type="text" value="W<sub>1</sub>"/>
Slope P12T of Fracture Envelope	<input type="text" value="W<sub>2</sub>"/>
Slope P23C of Fracture Envelope	<input type="text" value="W<sub>3</sub>"/>
Slope P23T of Fracture Envelope	<input type="text" value="W<sub>4</sub>"/>

HASHIN-TAPE FAILURE CRITERION

- **Special form for composites manufactured from tape preform**
- **Orientations:**
 - 1-direction is in the tape fiber direction
 - 2-direction is perpendicular to the fiber in the plane of the tape
 - 3-direction is through the thickness

HASHIN-TAPE FAILURE CRITERION

- Fiber failure**

Tensile fiber mode, $\sigma_1 > 0$

1st failure index:
$$\left[\left(\frac{\sigma_1}{X_t} \right)^2 + \left(\frac{\sigma_{12}}{S_{12}} \right)^2 + \left(\frac{\sigma_{13}}{S_{13}} \right)^2 \right]$$

Compressive fiber mode, $\sigma_1 < 0$

2nd failure index:
$$\left[\left(\frac{\sigma_1}{X_c} \right)^2 + \left(\frac{\sigma_{12}}{S_{12}} \right)^2 + \left(\frac{\sigma_{13}}{S_{13}} \right)^2 \right]$$

- Matrix failure**

Tensile matrix mode, $\sigma_2 + \sigma_3 > 0$

3rd failure index:
$$\left[\left(\frac{\sigma_2 + \sigma_3}{Y_t} \right)^2 - \frac{\sigma_{22}\sigma_{33}}{S_{23}^2} + \left(\frac{\sigma_{12}}{S_{12}} \right)^2 + \left(\frac{\sigma_{13}}{S_{13}} \right)^2 + \left(\frac{\sigma_{23}}{S_{23}} \right)^2 \right]$$

Compressive matrix mode, $\sigma_2 + \sigma_3 < 0$

4th failure index:
$$\left[\left(\left(\frac{Y_c}{2S_{23}} \right)^2 - 1 \right) \left(\frac{\sigma_2 + \sigma_3}{Y_c} \right) + \left(\frac{\sigma_2 + \sigma_3}{2S_{23}} \right)^2 - \frac{\sigma_2\sigma_3}{S_{23}^2} + \left(\frac{\sigma_{12}}{S_{12}} \right)^2 + \left(\frac{\sigma_{13}}{S_{13}} \right)^2 + \left(\frac{\sigma_{23}}{S_{23}} \right)^2 + \nu_6 \left(\frac{\sigma_1}{S_1} \right)^2 \right]$$

HASHIN-TAPE FAILURE CRITERION

- **Material properties:**

- X_t – maximum tape fiber tension (V_1)
- X_c – maximum tape fiber compression (V_2)
- Y_t – maximum tape cross-fiber tension (V_3)
- Y_c – maximum tape cross-fiber compression (V_4)
- S_{12} – layer shear strength (V_{10})
- S_{23} – Y-Z transverse shear strength (V_{11})
- S_{31} – Z-X transverse shear strength (V_{12})
- S_1 – maximum fiber stress for matrix compression (V_5)
- V_6 – contribution factor for S_1

HASHIN-FABRIC FAILURE CRITERION

- **Special form for composites manufactured from fabric preform**
- **Orientations:**
 - 1-direction is in the first fiber direction (warp)
 - 2-direction is in the second fiber direction (weave)
 - 3-direction is through the thickness

HASHIN-FABRIC FAILURE CRITERION

- Fiber failure**

1st failure index:

Tensile fiber 1 mode, $\sigma_1 > 0$

$$\left[\left(\frac{\sigma_1}{X_t} \right)^2 + \left(\frac{\sigma_{12}}{S_{12}} \right)^2 + \left(\frac{\sigma_{13}}{S_{13}} \right)^2 \right]$$

Compressive fiber 1 mode, $\sigma_1 < 0$

2nd failure index:

$$\left[\left(\frac{\sigma_1}{X_c} \right)^2 + \left(\frac{\sigma_{12}}{S_{12}} \right)^2 + \left(\frac{\sigma_{13}}{S_{13}} \right)^2 \right]$$

Tensile fiber 2 mode, $\sigma_2 > 0$

3rd failure index:

$$\left[\left(\frac{\sigma_2}{Y_t} \right)^2 + \left(\frac{\sigma_{12}}{S_{12}} \right)^2 + \left(\frac{\sigma_{13}}{S_{13}} \right)^2 \right]$$

Compressive fiber 2 mode, $\sigma_2 < 0$

4th failure index:

$$\left[\left(\frac{\sigma_2}{Y_c} \right)^2 + \left(\frac{\sigma_{12}}{S_{12}} \right)^2 + \left(\frac{\sigma_{13}}{S_{13}} \right)^2 \right]$$

- Matrix failure**

5th failure index:

Tensile matrix mode, $\sigma_3 > 0$

$$\left[\left(\frac{\sigma_3}{Z_t} \right)^2 + \left(\frac{\sigma_{12}}{S_{12}} \right)^2 + \left(\frac{\sigma_{13}}{S_{13}} \right)^2 + \left(\frac{\sigma_{23}}{S_{23}} \right)^2 \right]$$

Compressive matrix mode, $\sigma_3 < 0$

6th failure index:

$$\left[\left(\frac{\sigma_3}{Z_c} \right)^2 + \left(\frac{\sigma_{12}}{S_{12}} \right)^2 + \left(\frac{\sigma_{13}}{S_{13}} \right)^2 + \left(\frac{\sigma_{23}}{S_{23}} \right)^2 \right]$$

HASHIN-FABRIC FAILURE CRITERION

- **Material properties:**

- X_t – maximum first fiber tension (V_1)
- X_c – maximum first fiber compression (V_2)
- Y_t – maximum second fiber tension (V_3)
- Y_c – maximum second fiber compression (V_4)
- Z_t – maximum thickness tension (V_5)
- Z_c – maximum thickness compression (V_6)
- S_{12} – layer shear strength (V_{10})
- S_{23} – Y-Z transverse shear strength (V_{11})
- S_{31} – Z-X transverse shear strength (V_{12})

USER-DEFINED CRITERIA

- MSC Nastran allows the user to define a custom failure criterion
- The user writes a custom user subroutine (UPROGFAIL), written in Fortran for the analysis
- The model could for example include micro-structural failure models or multiple sub-criteria models, each with a particular mechanism of failure

```
      subroutine ext_uprogfail(nelem, nint, kcus, matus, stress, strain,  
      & icrit, fi, redfac0, redfac, ideact, dt, dtcl, time,  
      & timeinc, nstats, ncomp, isunit, idata, rdata, cdata,  
      & len_idata, len_rdata, len_cdata, error_code)  
c.... user subroutine to calculate the current stiffness reduction  
c.... factors for progressive failure  
c....  
c.... the routine is called when failure occurs  
c....  
c.... nelem   = user element number  
c.... nint    = integration point number  
c.... kcus(1) = layer number  
c.... kcus(2) = internal layer number  
c.... matus(1) = user material id  
c.... matus(2) = internal material id  
c.... stress  = current total stresses in preferred system  
c....          in full tensor format: (s11, s22, s33, s12, s23, s32)  
c.... strain  = current total strains in preferred system  
c....          in full tensor format: (e11, e22, e33, e12, e23, e32)  
c.... icrit   = current failure criterion:  
c....          1 - maximum stress  
c....          2 - maximum strain  
c....          3 - tsai-wu  
c....          4 - hoffman  
c....          5 - hill  
c....          6 - not used  
c....          7 - user  
c....          8 - hashin  
c....          9 - hashin fabric  
c....          10 - not used  
c....          11 - not used  
c....          12 - hashin tape  
c....          13 - puck  
c.... fi      = array of current failure indices  
c.... redfac0 = array of current reduction factors
```

PROGRESSIVE FAILURE

- **How failure affects different material moduli – selective degradation**
 - Maximum stress or Maximum strain criteria
 - Each modulus is reduced separately for each respective failure mode
 - For example, if failure in the 2-direction is found, only E_2 is degraded.
 - Criteria with a single failure index (Hill, Hoffman, Tsai-Wu)
 - All moduli are reduced with the same factor when $FI > 1$
 - Hashin, Puck and Hashin-tape criteria
 - E_1 and E_3 are reduced if fiber failure is found
 - E_2 , G_{12} , G_{23} and G_{31} are reduced if matrix failure is found
 - Hashin-fabric criterion
 - E_1 , E_2 and E_3 are reduced separately for each respective failure mode
 - G_{12} , G_{23} and G_{31} are reduced based on the worst of the failure modes

PROGRESSIVE FAILURE

- **Immediate stiffness degradation (ITYPE = 3)**
 - When failure occurs, set the respective material moduli to a fraction of the original value (residual stiffness factor), by default 1%
- **Gradual stiffness degradation (ITYPE = 2)**
 - Only decrease the moduli so that the largest failure index is equal to one
 - The stiffness is not reduced to less than 1% (can be user modified)
- **Option for element deactivation upon failure**
 - This option is not supported in SOL400 yet

PROGRESSIVE FAILURE

- What are those factors for stiffness reduction?

Constitutive Model:	Failure1 (SOL 400/600) ▼
Failure Criterion:	Hashin ▼
Progressive Failure Option:	Gradual ▼
Property Name	Value
Max Fiber Tension =	<input type="text"/>
Max Fiber Compression =	<input type="text"/>
Max Matrix Tension =	<input type="text"/>
Max Matrix Compression =	<input type="text"/>
Layer Shear Strength =	<input type="text"/>
Transverse Shear Strength =	<input type="text"/>
Shear Stress Bond (SB) =	<input type="text"/>
Residual Stiffness Factor =	<input type="text" value="A<sub>1</sub>"/>
Matrix Compression Factor =	<input type="text" value="A<sub>2</sub>"/>
Shear Stiffness Factor =	<input type="text" value="A<sub>3</sub>"/>
E33 Fiber Failure Factor =	<input type="text" value="A<sub>4</sub>"/>
Shear Fiber Failure Factor =	<input type="text" value="A<sub>5</sub>"/>

PROGRESSIVE FAILURE

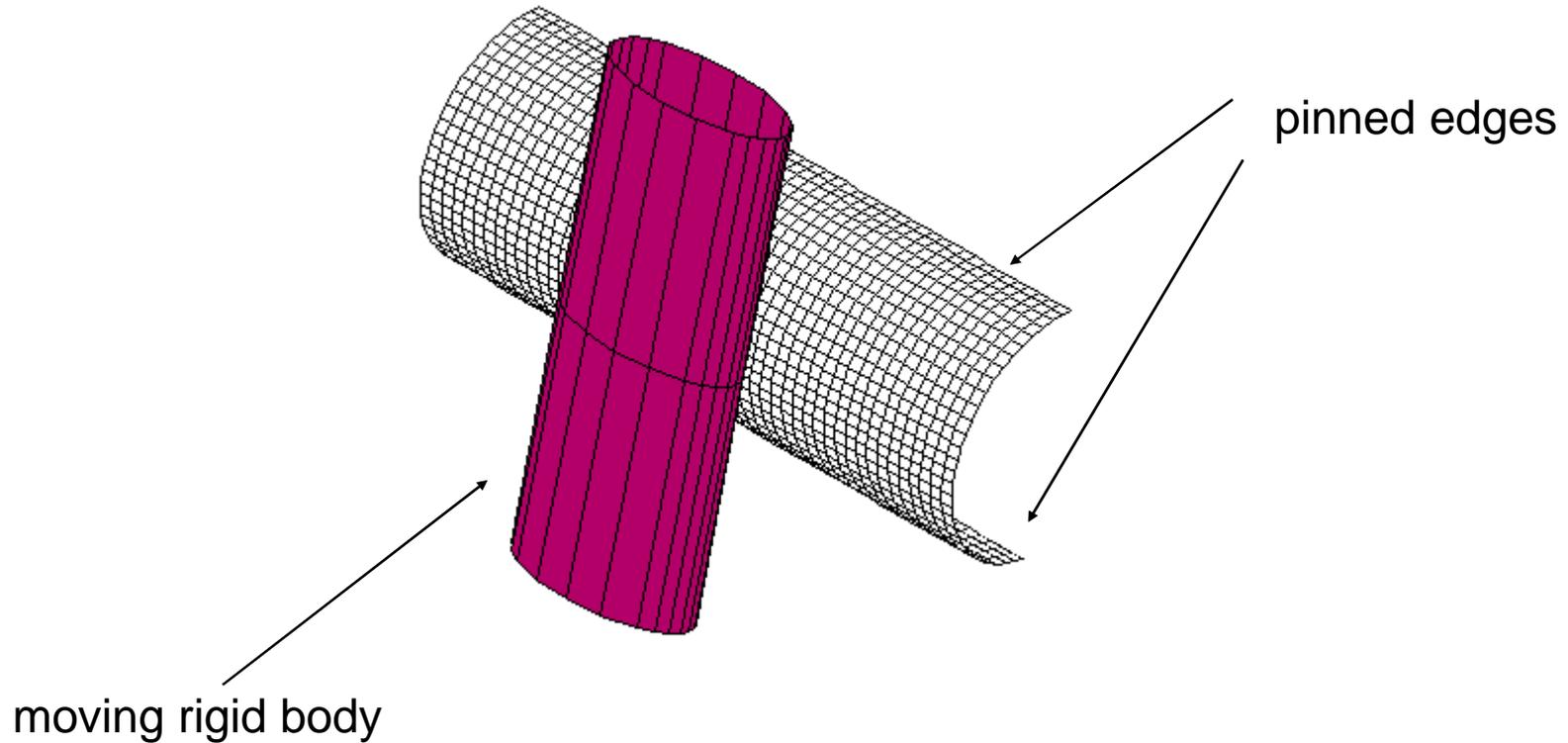
- **RESIDUAL STIFFNESS FACTOR (A_1)**
 - For the “immediate” option this is the fraction of the original modulus that is used when failure occurs
 - For the “gradual” option this is the smallest fraction of the modulus that will be used
 - For all failure criteria
- **MATRIX COMPRESSION FACTOR (A_2)**
 - By default, a failure in the matrix reduces the shears and E_2 by the same factor. For some materials, E_2 reduces less (or not at all) in compression. Setting this factor to 1.0 leads to no E_2 reduction for matrix compression failure. It can take values between 0 and 1.0.
 - For Hashin, Puck and Hashin-tape criteria
- **SHEAR STIFFNESS FACTOR (A_3)**
 - Some materials show less reduction of the shear moduli compared with E_2 . Setting this factor to 0.5 has the effect that it reduces with half the factor used for E_2 . A factor of 1.0 leads to no shear stiffness reduction.
 - For Hashin, Puck and Hashin-tape criteria

PROGRESSIVE FAILURE

- **E_3 REDUCTION FROM FIBER FAILURE (A_4)**
 - By default, E_3 only reduces due to fiber failure. With this factor you can specify that it also depends on matrix failure. Set to 1.0 to only depend on matrix failure and to less than 1.0 to depend on both.
 - For Hashin, Puck and Hashin-tape criteria
- **SHEAR REDUCTION FROM FIBER FAILURE (A_5)**
 - This factor allows the shear moduli to decrease due to matrix failure. The worst shear reduction from fiber and matrix is used. Using a factor less than 1.0 leads to less shear reduction compared with E_1 reduction.
 - For Hashin, Puck and Hashin-tape criteria

PROGRESSIVE FAILURE – EXAMPLE

- Rigid elliptical cylinder hitting composite shell

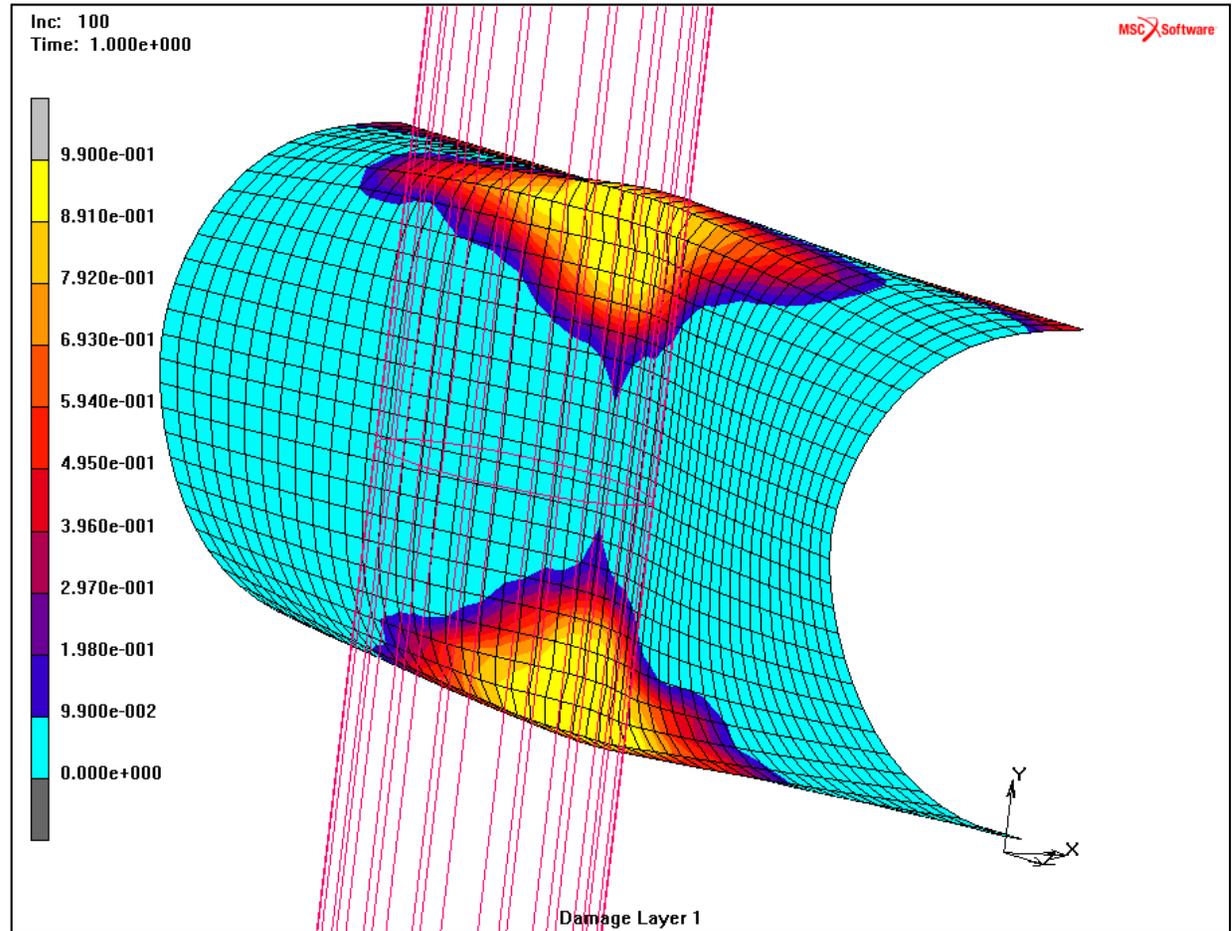


Composite with 5 layers

[0 45 90 -45 0]

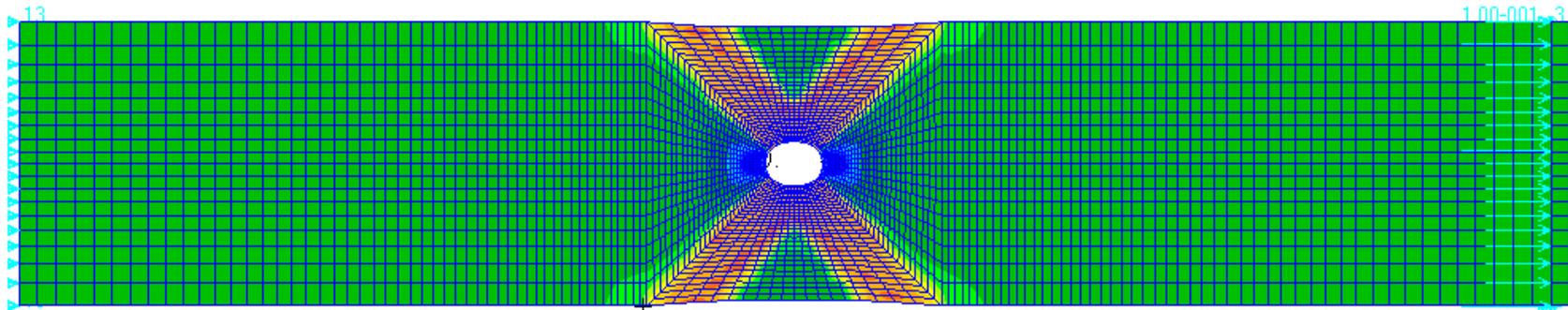
PROGRESSIVE FAILURE – EXAMPLE

- Damage at outer layer (layer 1)
- Damage value indicates the degree of material stiffness degradation
- Change fringe plot element extrapolation to average so damage does not show up as > 1
- Yellow area – fully damaged (damage value=0.99 for residual stiffness factor 0.01)



PROGRESSIVE FAILURE ANALYSIS – WORKSHOP 5

- Composite plate under tension load
- Puck failure criterion
- Gradual selective stiffness degradation
- Go to **WS5** in your workshop booklet



PROGRESSIVE FAILURE ANALYSIS – WORKSHOP 5

Input Options (Left)

Constitutive Model: Linear Elastic

Property Name	Value
Elastic Modulus 11 =	17000000.
Elastic Modulus 22 =	1400000.
Poisson Ratio 12 =	0.34
Shear Modulus 12 =	450000.
Shear Modulus 23 =	450000.
Shear Modulus 13 =	450000.
Density =	0.060000021
Thermal Expan. Coeff 11 =	
Thermal Expan. Coeff 22 =	
Structural Damping Coeff =	
Reference Temperature =	

Input Options (Right)

Constitutive Model: Failure1 (SOL 400/600)

Failure Criterion: Puck

Progressive Failure Option: Gradual

Property Name	Value
Max Fiber Tension =	200000.
Max Fiber Compression =	100000.
Max Matrix Tension =	15000.
Max Matrix Compression =	30000.
Layer Shear Strength =	15000.
Slope P12C of Fracture Envelope	0.34999999
Slope P12T of Fracture Envelope	0.34999999
Slope P23C of Fracture Envelope	0.27000001
Slope P23T of Fracture Envelope	0.27000001
Shear Stress Bond (SB) =	
Residual Stiffness Factor =	
Matrix Compression Factor =	
Shear Stiffness Factor =	
E33 Fiber Failure Factor =	
Shear Fiber Failure Factor =	
Deactivate Fiber Tension =	
Deactivate Fiber Compression =	
Deactivate Matrix Tension =	
Deactivate Matrix Compression =	

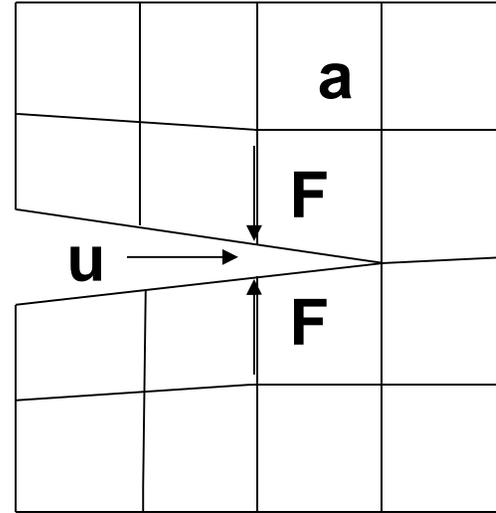
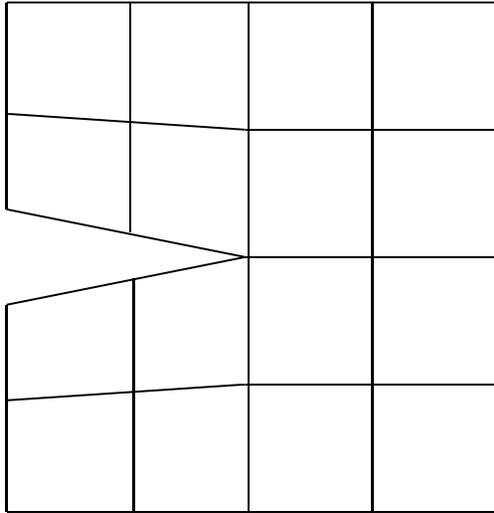
PCOMP	1							
	1	.01	45.	YES	1	.01	-45.	YES
	1	.01	45.	YES	1	.01	-45.	YES
	1	.01	90.	YES	1	.01	45.	YES
	1	.01	-45.	YES	1	.01	0.	YES
	1	.01	-45.	YES	1	.01	45.	YES
	1	.01	90.	YES	1	.01	-45.	YES
	1	.01	45.	YES	1	.01	-45.	YES
	1	.01	45.	YES				
\$								
MAT8	1	1.7+7	1.4+6	.34	450000.	450000.	450000.	
\$								
MATF	1	2						
	CRI	8	200000.	100000.	15000.	30000.		
					15000.			.35
		.35	.27	.27				

VCCT AND CRACK PROPAGATION

VCCT – VIRTUAL CRACK CLOSURE TECHNIQUE

- **In fracture mechanics, a crack starts to grow when**
 - $G > G_c$
where G is the energy release rate
 G_c is the crack growth resistance (also called fracture toughness)
- **VCCT is a simple but general method to evaluate energy release rate of a crack**
- **Simple and robust implementation – only involves:**
 - Force needed to keep the crack closed, F
 - Crack opening displacement, u
 - Geometry around the crack tip, a

VCCT

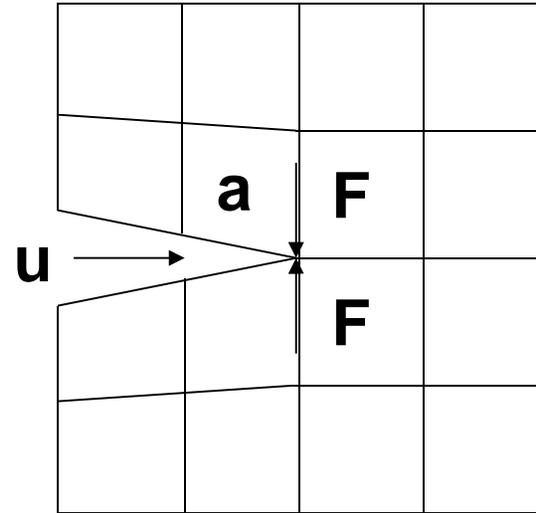
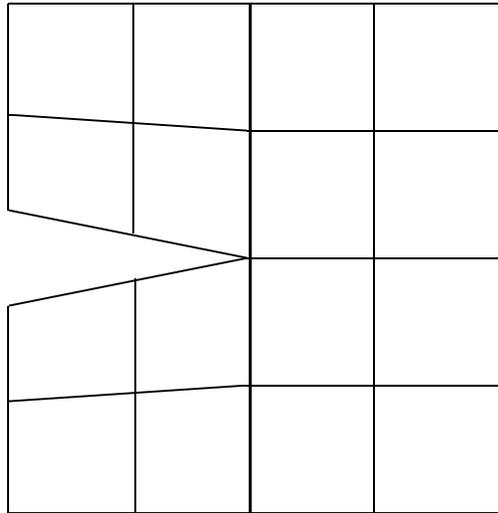


- Consider a crack growing a length: a
- The crack is initially closed
- Energy release rate:

$$G = Fu/2a$$

F : force needed to close the crack (or keep it closed)
 u : crack opening displacement

VCCT



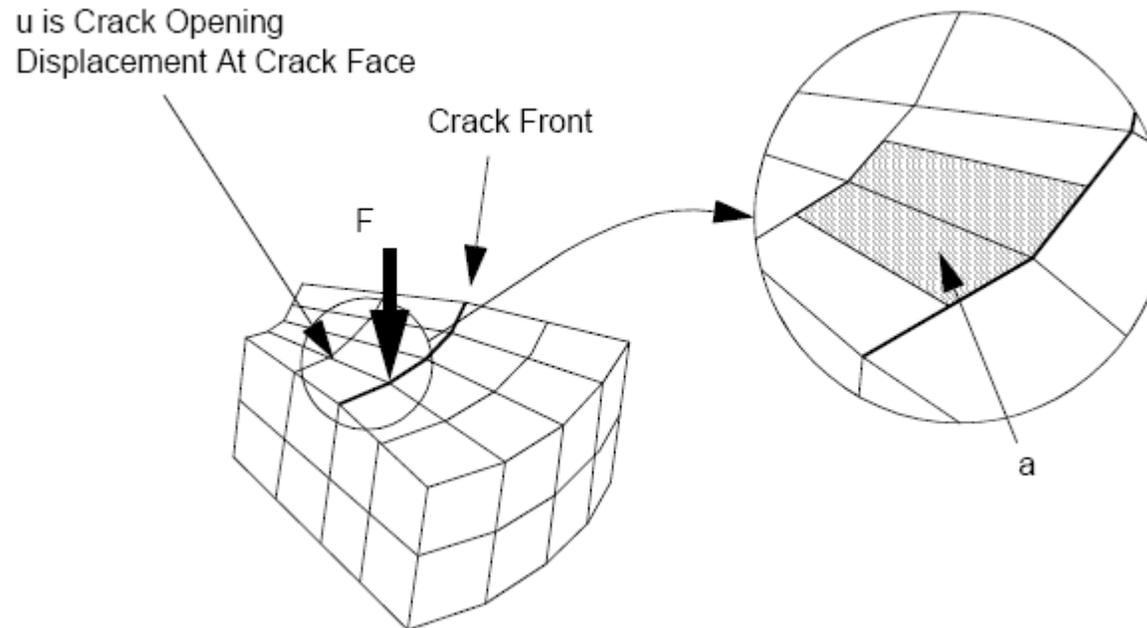
- **FEM approximation: Use consistent nodal force at tip and crack opening at first crack segment**
- **Energy release rate:**

$$G = Fu/2a$$

- **This avoids two analyses (with closed crack to get the force and with open crack to get the crack opening)**
- **For higher order elements also use forces and displacements at mid-side nodes**

VCCT IN 3D

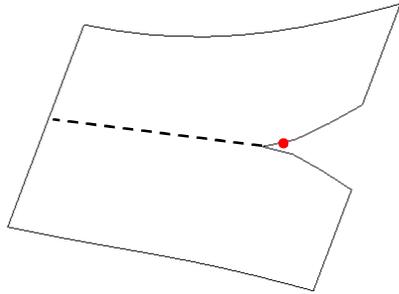
- 3D solids



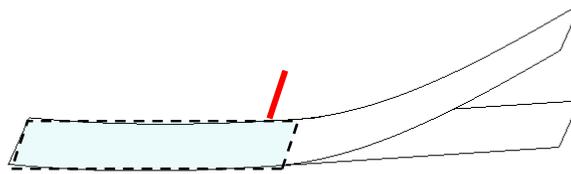
- Energy release rate G is calculated for each node along the crack front
- Forces and displacements similar to 2D
- Area (a) from crack faces of elements around tip node

VCCT – CRACK TYPES

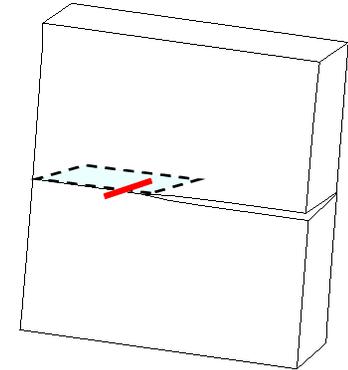
- Supported crack types are shown below



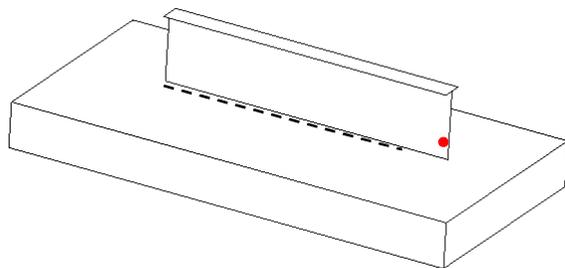
line crack – 2D or shell



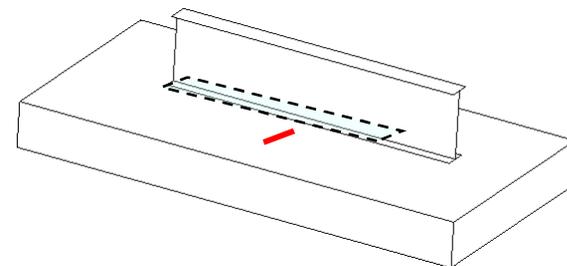
face crack – shell to shell



face crack – 3D solid



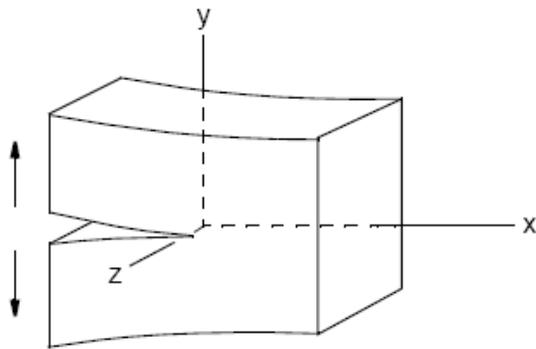
line crack – shell edge to solid or shell



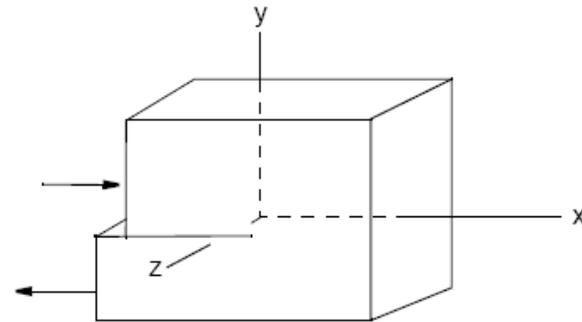
face crack – shell to solid

MODES OF CRACK EXTENSION

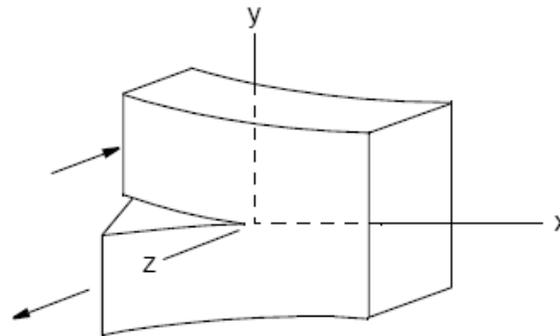
- All three modes of crack extension are supported
- Energy release rates G_I , G_{II} and G_{III} corresponding to each mode



(a) Mode I: Opening



(b) Mode II: Sliding



(c) Mode III: Tearing

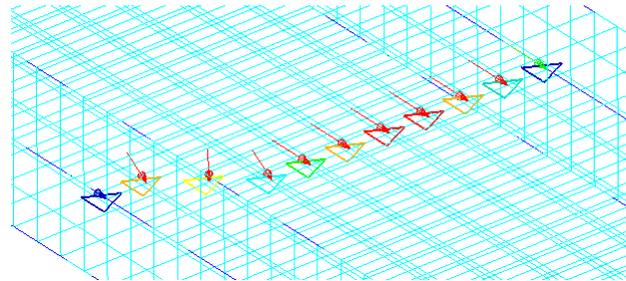
VCCT – DEFINITION IN MSC NASTRAN

VCCT	ID	IDCR	ITYPE	IGROW	INCM	METHOD			
	CGI	GC					GC-II	GC-III	
	TABCGI	TABGC					TABGC-II	TABGC-III	
	G1	G2	G3	G4	G5	etc.			

- **ID** – Identification of a matching Case Control VCCT entry
- **IDCR** – Identification of the crack
- **ITYPE** – Crack growth type: 0 (no crack growth) or 2 (direct growth)
- **IGROW** – Crack growth method: 2 (release glued contact)
- **INCM** – Crack growth increment control for fatigue (not used)
- **METHOD** – Crack growth direction method: 1 (maximum hoop stress direction)
- **CGI** – Crack growth increment (not used for release glued contact method)
- **GC, GC-II, GC-III** – Crack growth resistance (fracture toughness)
- **TABCGI** – Table for crack growth increment
- **TABGC, TABGC-II, TABGC-III** – Tables for crack growth resistance
- **G_i** – Grids along the crack front

VCCT – BASIC (NO CRACK PROPAGATION)

- **ITYPE = 0**
- **User only needs to define the crack front**
- **Basic VCCT definition gives you**
 - Calculation of the energy release rate G and the respective modes G_I , G_{II} and G_{III}
 - Estimated crack growth direction



- Results are saved for post-processing and printed to the f06 file
- Results output format in the f06 file:

CRACK TIP		VCCT CRACK			RESULTS			
CRACK ID	GRID ID	TOTAL	ENERGY RELEASE RATE MODE I	ENERGY RELEASE RATE MODE II	ENERGY RELEASE RATE MODE III	ESTIMATED CRACK GROWTH DIRECTION X	ESTIMATED CRACK GROWTH DIRECTION Y	ESTIMATED CRACK GROWTH DIRECTION Z

VCCT – CRACK PROPAGATION

- **Crack growth mode**
 - Direct Growth (ITYPE=2)
 - Fatigue (not supported in SOL400 yet)
- **Crack growth method**
 - By releasing glued contact
 - Along element edges (not supported in SOL400 yet)
 - With global remeshing (not supported in SOL400 yet)
- **Crack growth criteria**
 - Total energy release rate $G > G_C$ (enter G_C only)
 - Individual modes $G_I > G_{C-I}$ or $G_{II} > G_{C-II}$ or $G_{III} > G_{C-III}$ (enter $G_C = G_{C-I}$, G_{C-II} and G_{C-III})
 - Mixed modes (not supported in SOL400 yet)
- **Crack growth direction method**
 - Maximum hoop stress direction
 - Along pure mode (not supported in SOL400 yet)
 - User defined direction (not supported in SOL400 yet)

VCCT – CRACK GROWTH BY RELEASE GLUED CONTACT

- **Connect meshes with glued contact**
 - Create contact bodies
 - Set glue contact in contact table
- **Meshes of contact bodies do not need to match up**
 - For non-matching meshes, crack front nodes must be on the touching (slave) body
- **User only defines initial crack**
 - Using UNGLUE option, or
 - Not include the crack area in contact bodies
- **Solver automatically finds new crack nodes and releases glued contact**
- **Released nodes can contact (touching) again, but will not glue**

VCCT – CRACK GROWTH BY RELEASE GLUED CONTACT

- **User defines crack growth resistance**
 - G_c for total energy release rate, or
 - G_{c-I} , G_{c-II} and G_{c-III} for individual modes
- **Crack grows when $G > G_c$ OR $G_I > G_{c-I}$ or $G_{II} > G_{c-II}$ or $G_{III} > G_{c-III}$**
- **G_c can be a function of time or temperature defined by a table**
- **Growth direction is the maximum hoop stress direction**
- **Growth increment is the edge length of the released element**
- **Solver checks for growth when increment is converged and writes crack growth detection status in the f06 file**

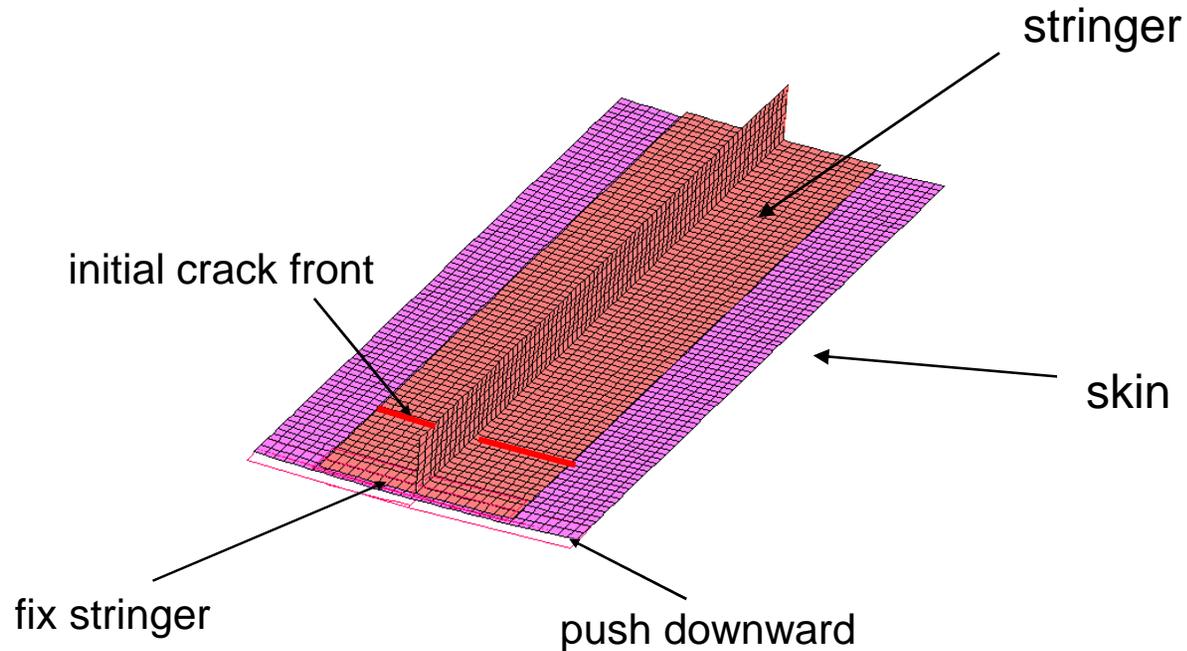
```
no crack growth detected for crack: crack_20
largest energy release rate 1.1925E+00
  crack growth resistance 1.2000E+00
  crack growth failure index 9.9373E-01
    at crack tip node 10360
```

or

```
crack growth detected for crack: crack_20
  crack tip node 8182
current energy release rate 1.4082E+00
  crack growth resistance 1.2000E+00
  crack growth failure index 1.1735E+00
```

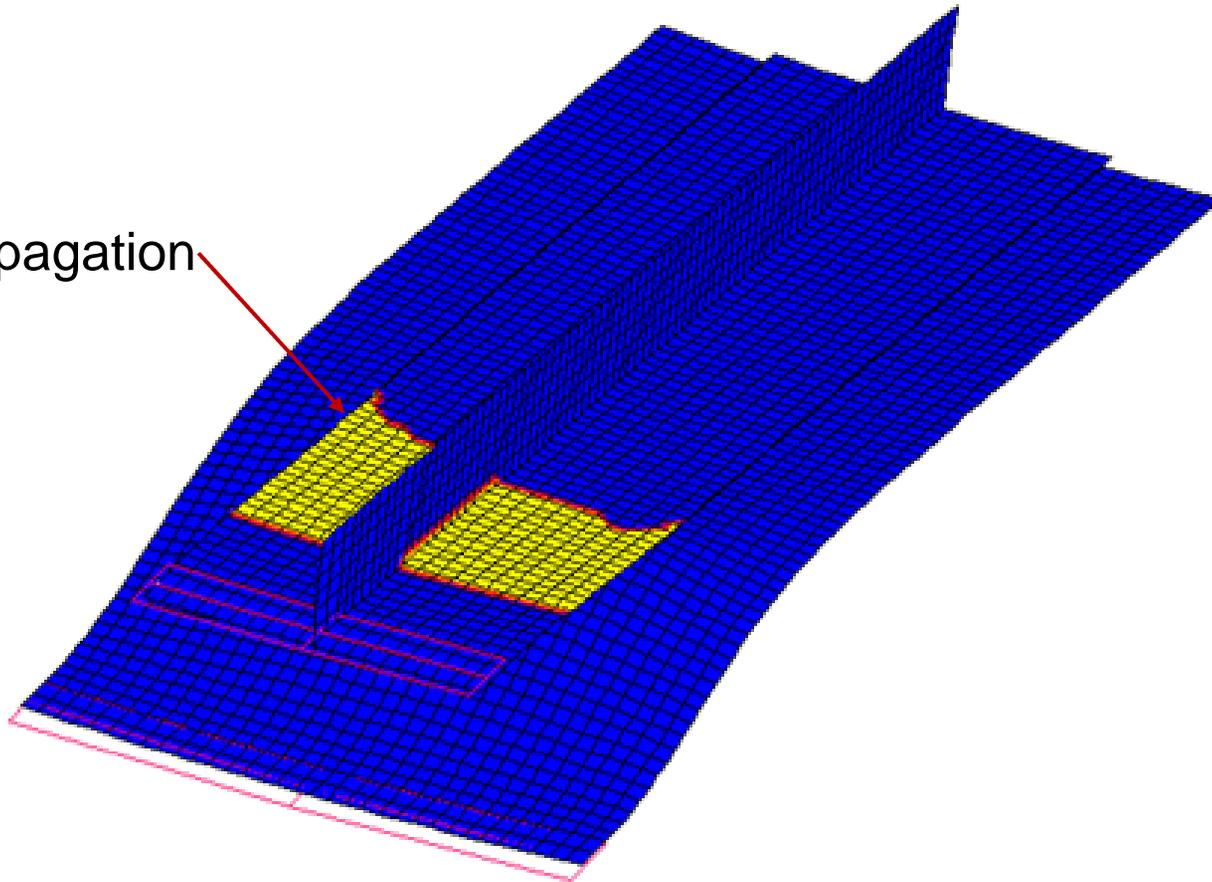
VCCT EXAMPLE: SKIN-STRINGER DELAMINATION

- Wagner/Balzani, Computers & Structures 2008
- Stringer glued to skin
- Initial crack created by UNGLUE option



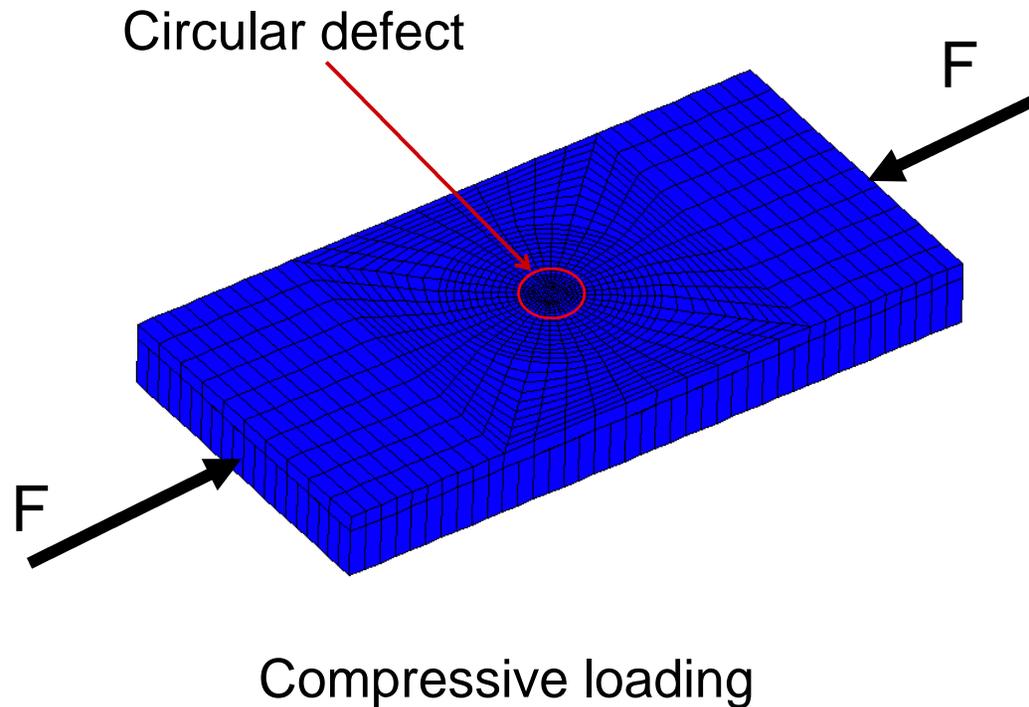
VCCT EXAMPLE: SKIN – STRINGER DELAMINATION

Crack Propagation

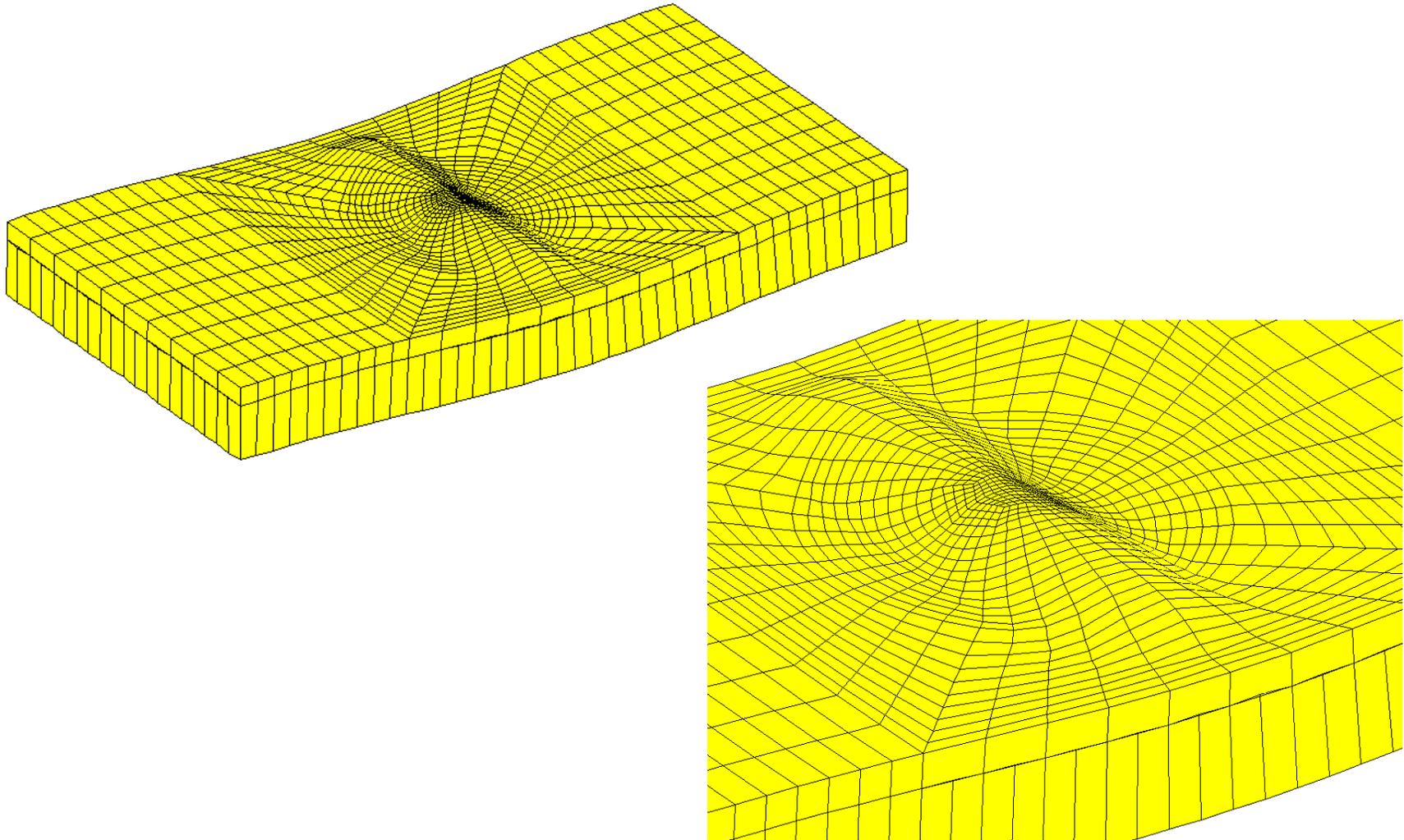


VCCT EXAMPLE – BUCKLING DELAMINATION

- 4-ply Composite modeled with 2 layers of solid elements
- Defect between 3rd and 4th ply
- Glue parts together, except at defect

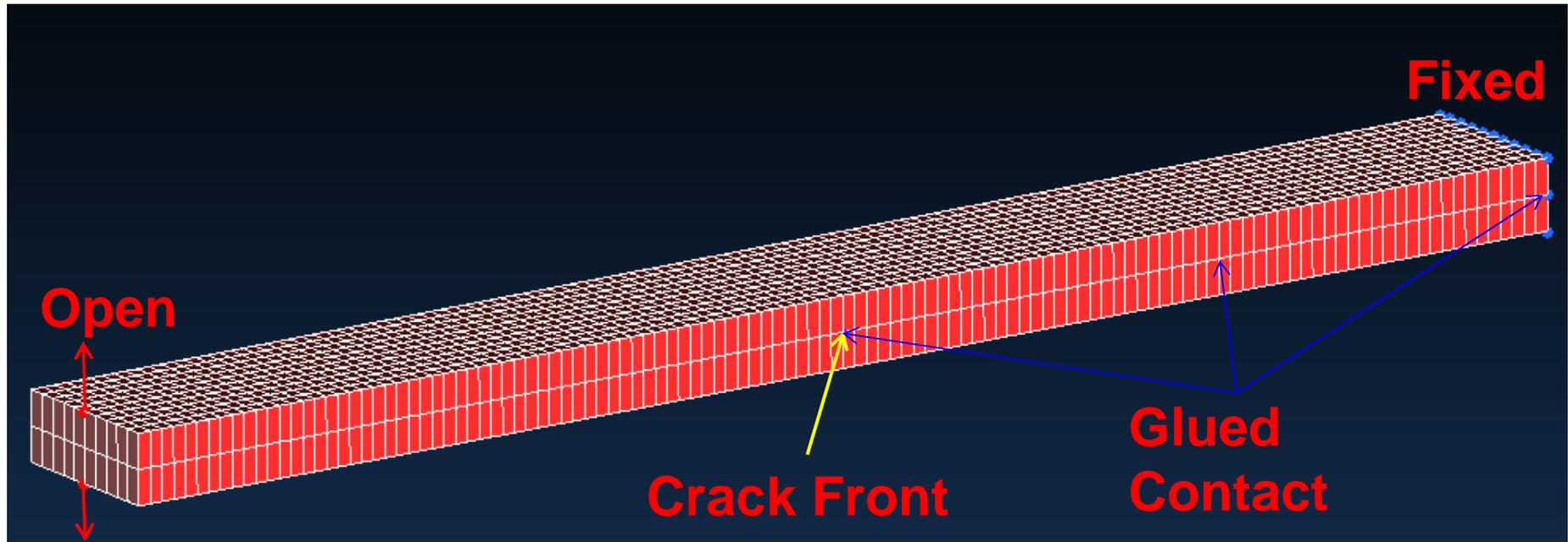


VCCT EXAMPLE – BUCKLING DELAMINATION



VCCT – WORKSHOP 6

Double Cantilever Beam

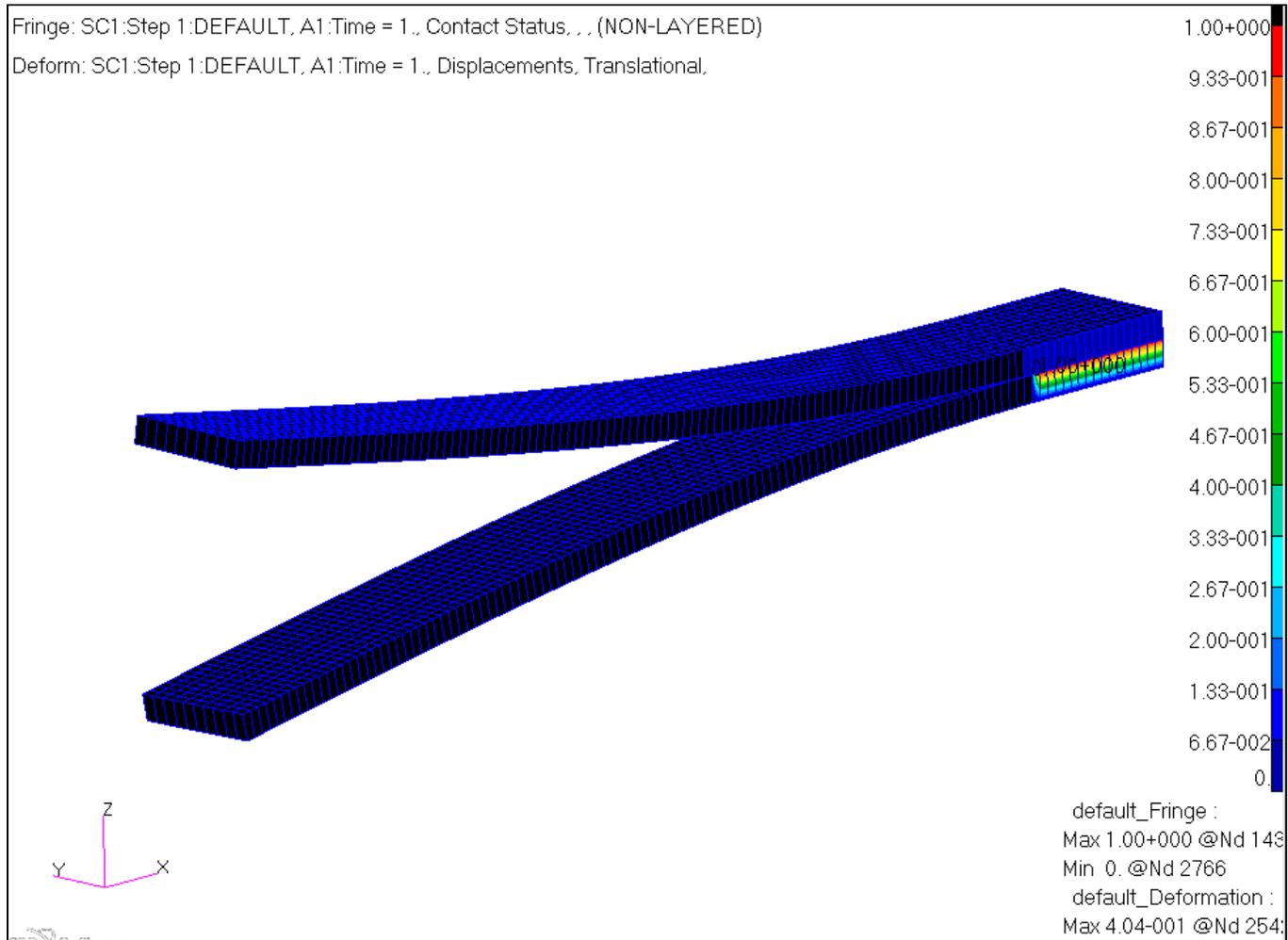


VCCT WORKSHOP 6 – INPUT DECK

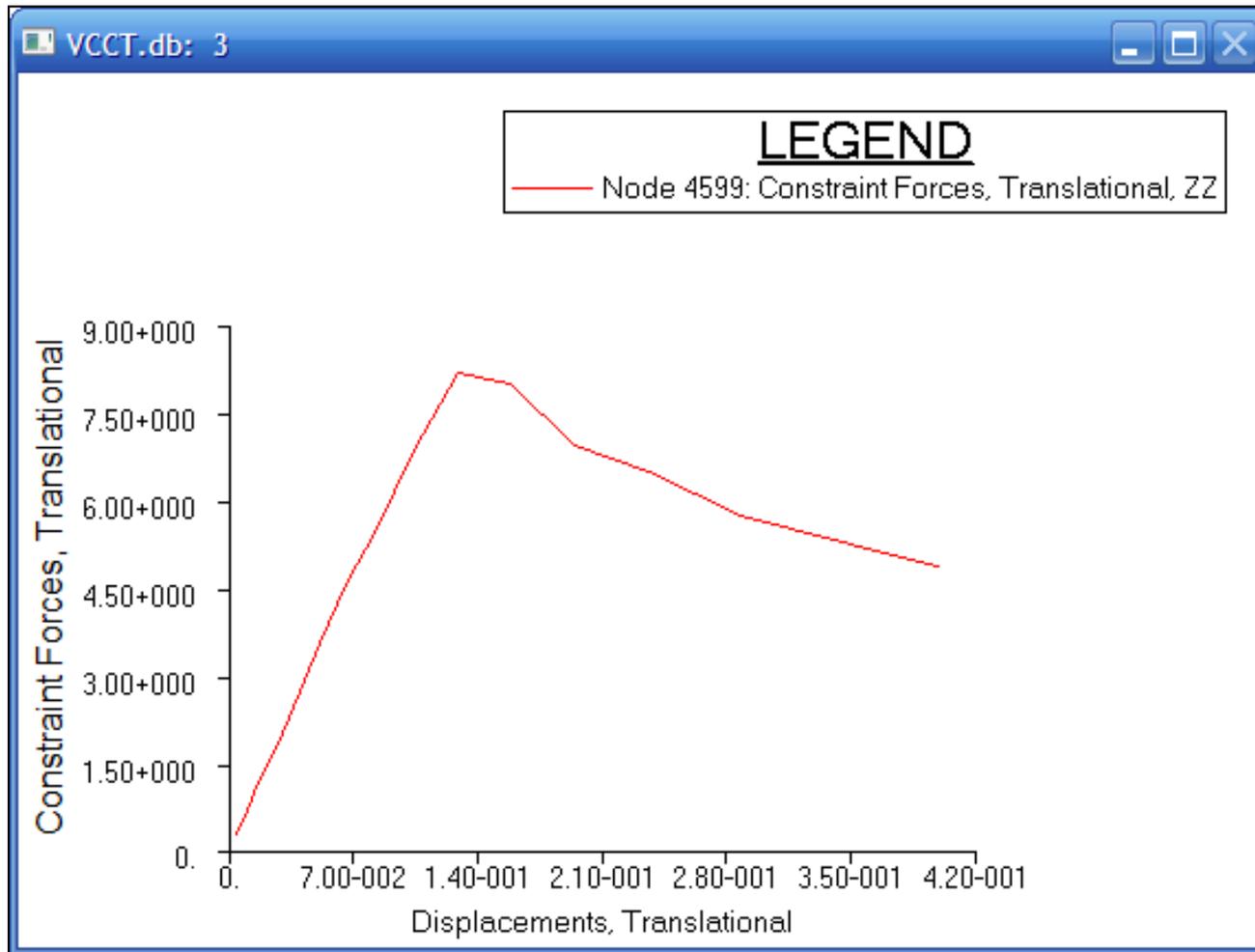
```

SOL 400
CEND
SUBCASE 1
  SPC = 92
  LOAD = 22
  VCCT = 1
  DISPLACEMENT(SORT1, PRINT, REAL)=ALL
  STRESS(SORT1, PRINT, REAL, VONMISES, CORNER)=ALL
  SPCFORCES(SORT1, PRINT, REAL)=ALL
  BOUTPUT(SORT1, PRINT)=ALL
  BCONTACT = 1
  ANALYSIS = NLSTAT
  NLSTEP = 1
BEGIN BULK
PARAM  LGDISP      1
BCPARA  0 NLGLUE  1
$
VCCT      1      20      2      2      1      1      0.0      0.0+
+         0.0      1.2      0      0      0      0      0      0+
+         0      0      0      0      0      0      0      0+
+         6991    7354    7717    8080    8443    8806    9169    9532+
+         9895    10258   10621
$
BCTABLE  1
+       SLAVE      1      0      0      0.0      1
+
+       MASTERS    2
$
MATORT   2  1.8E+7 1600000. 1600000. 0.28 0.4 0.025 0.057+
+       900000. 500000. 900000.
+       1      1
PCOMPLS  2      -1
+       C8  SLCOMP  ASTN
+       1      2      0.04  45.
+       2      2      0.04 -45.
+       3      2      0.04  0.0
+       4      2      0.04 -45.
+       5      2      0.04  45.
  
```

VCCT WORKSHOP 6 – CRACK PROPAGATION



VCCT WORKSHOP 6 – LOAD-DEFLECTION CURVE



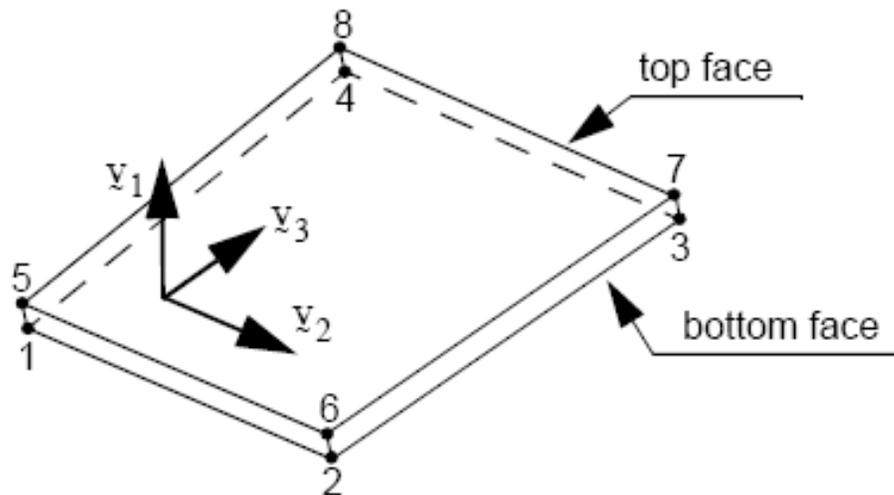
COHESIVE ZONE MODELING

COHESIVE ZONE MODELING (CZM)

- ***Delamination*** is known to be one of the predominant failure mechanisms in laminated composites and may results in a serious loss of the structural integrity
- ***Cohesive Zone Modeling (CZM)*** is a technique to simulate the onset and progress of delamination
- Applications of CZM are not limited to laminated composites, but include e.g. adhesively-bonded joints and discrete cracking of concrete
- **The implementation of CZM in MSC Nastran is based on:**
 - Library of special elements (so-called *interface* elements)
 - Material model to characterize the cohesive interface behavior

COHESIVE ZONE MODELING - ELEMENTS

- The constitutive behavior of these elements is expressed in terms of tractions versus relative displacements between the top and bottom edges(2D)/faces(3D) of the elements



$$v_n = u_1^{\text{top}} - u_1^{\text{bottom}}$$

$$v_s = u_2^{\text{top}} - u_2^{\text{bottom}}$$

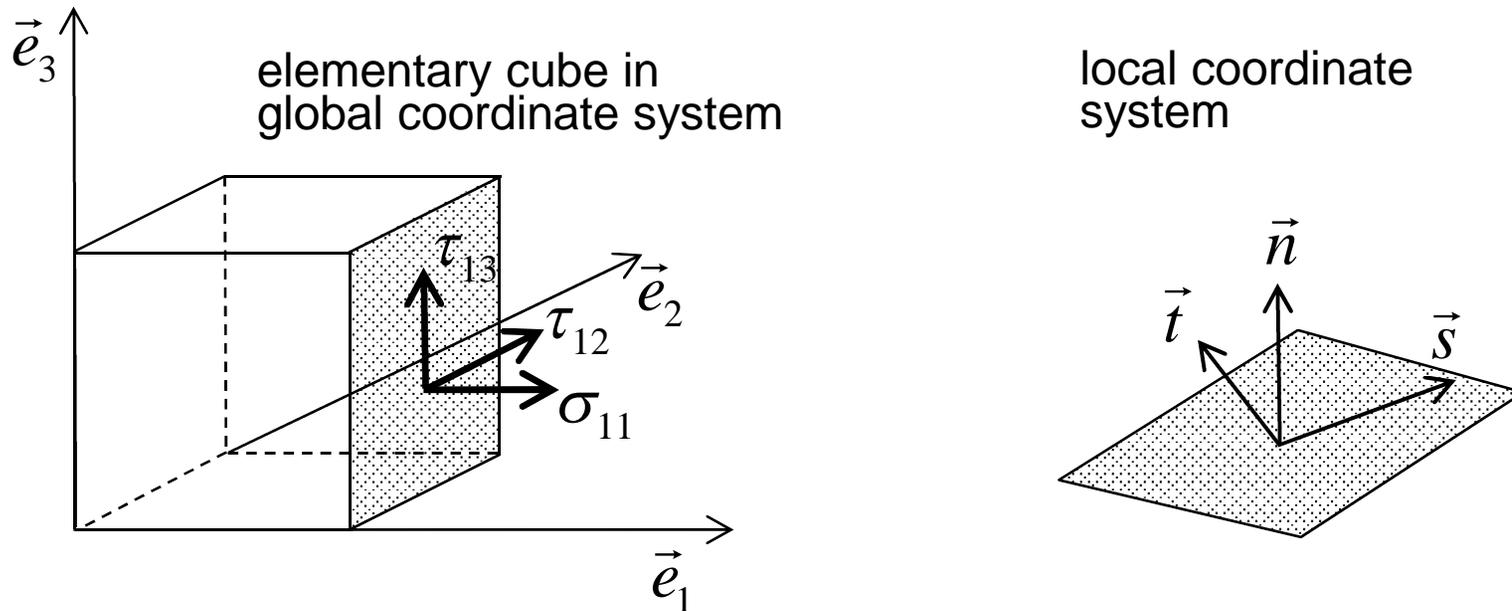
$$v_t = u_3^{\text{top}} - u_3^{\text{bottom}}$$

$$v = \sqrt{v_n^2 + v_s^2 + v_t^2}$$

v is the effective opening displacement

COHESIVE ZONE MODELING – ELEMENTS

- The concept of interface elements:

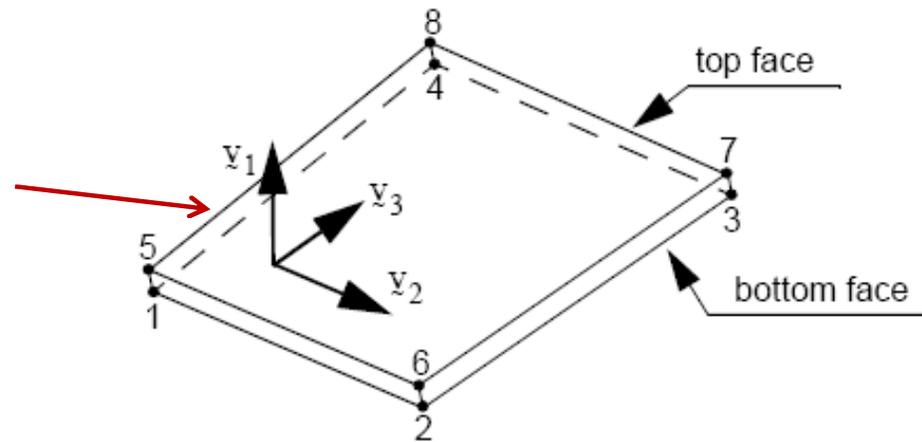


- For a 3D interface element, the non-zero stress components to be included are the tractions $[\sigma_{nn}, \tau_{ns}, \tau_{tn}]$, which are *work-conjugate* to the relative displacements $[vn, vs, vt]$ across the cohesive surface

COHESIVE ZONE MODELING – ELEMENTS

- **Available interface elements:**
 - 4-node and 8-node planar (**CIFQUAD**)
 - 4-node and 8-node axi-symmetric (**CIFQDX**)
 - 8-node and 20-node 3D (**CIFHEX**)
 - 6-node and 15-node 3D (**CIFPENT**)
- **The elements are allowed to be infinitesimally thin; in which case the top and the bottom faces may coincide**
- **It is important to orient the thickness direction (defined by element connectivity) correctly because of the special constitutive behavior of these elements**

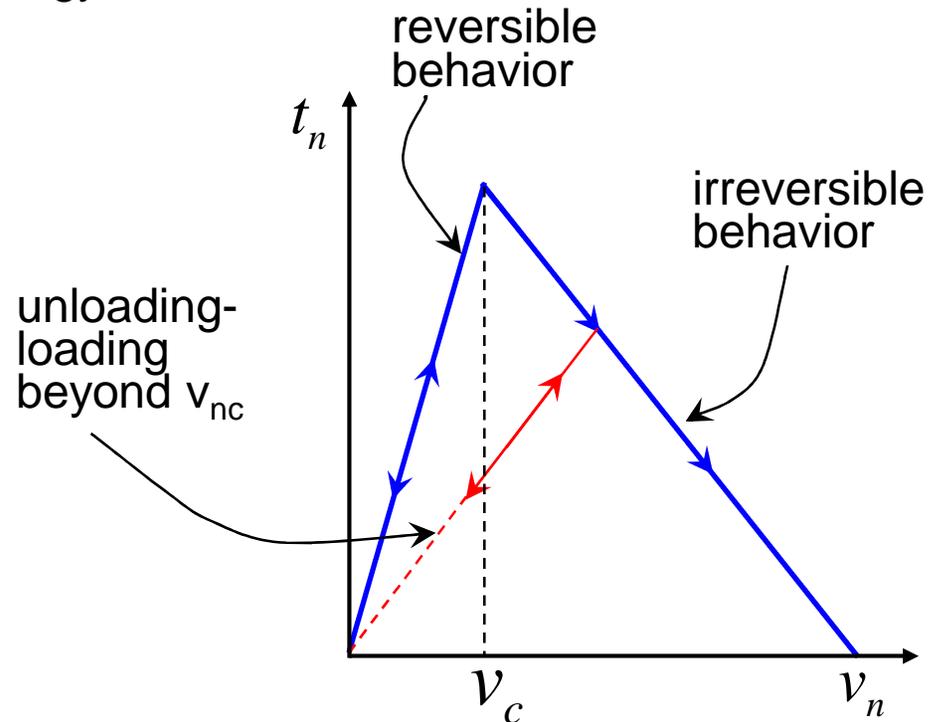
Thickness direction:
Face 1-2-3-4 to Face 5-6-7-8



COHESIVE ZONE MODELING - MATERIAL

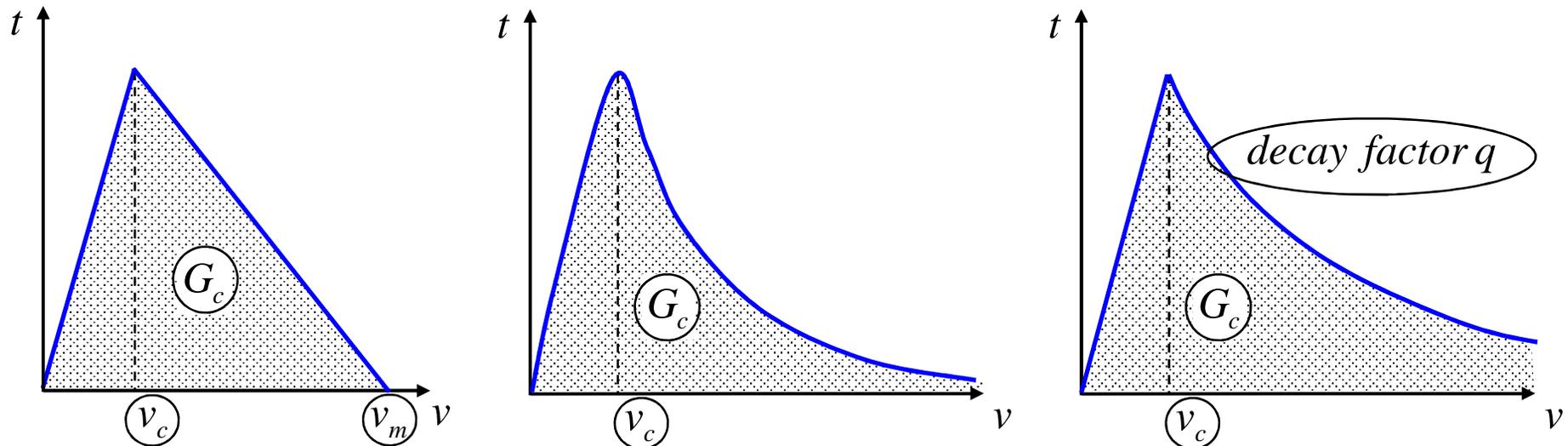
- **Material behavior:**

- Initially reversible behavior
- Irreversible behavior as soon as a critical relative displacement has been reached
- Area below the traction-relative displacement curve is called the *cohesive energy* G_c
- Note: G_c units = Energy/Area



COHESIVE ZONE MODELING – MATERIAL

- **Available material models**
 - Bilinear (3 input quantities: G_c V_c V_m)
 - Exponential (2 input quantities: G_c V_c)
 - Linear-Exponential (3 input quantities: G_c V_c q)
- **Input quantities**
 - G_c is the cohesive energy
 - V_c is the critical effective opening displacement
 - V_m is the maximum effective opening displacement
 - q is the exponential decay factor



COHESIVE ZONE MODELING – MATERIAL

- Basic equations:

$t = \frac{2G_c v}{v_m v_c} \text{ if } 0 \leq v \leq v_c$ $t = \frac{2G_c}{v_m} \left(\frac{v_m - v}{v_m - v_c} \right) \text{ if } v_c < v \leq v_m$ $t = 0 \text{ if } v > v_m$	$\left. \vphantom{\begin{matrix} t = \frac{2G_c v}{v_m v_c} \\ t = \frac{2G_c}{v_m} \left(\frac{v_m - v}{v_m - v_c} \right) \\ t = 0 \end{matrix}} \right\} \text{ (bilinear)}$	$G_c = \frac{t_c v_m}{2}$	
$t = G_c \frac{v}{v_c} e^{-v/v_c}$		 (exponential)	$G_c = e t_c v_c$
$t = \frac{2qG_c}{v_c(q+2)} \frac{v}{v_c} \text{ if } 0 \leq v \leq v_c$ $t = \frac{2qG_c}{v_c(q+2)} e^{q(1-v/v_c)} \text{ if } v > v_c$		$\left. \vphantom{\begin{matrix} t = \frac{2qG_c}{v_c(q+2)} \frac{v}{v_c} \\ t = \frac{2qG_c}{v_c(q+2)} e^{q(1-v/v_c)} \end{matrix}} \right\} \text{ (linear-exponential)}$	$G_c = \frac{t_c v_c (q+2)}{2q}$

COHESIVE ZONE MODELING – MATERIAL

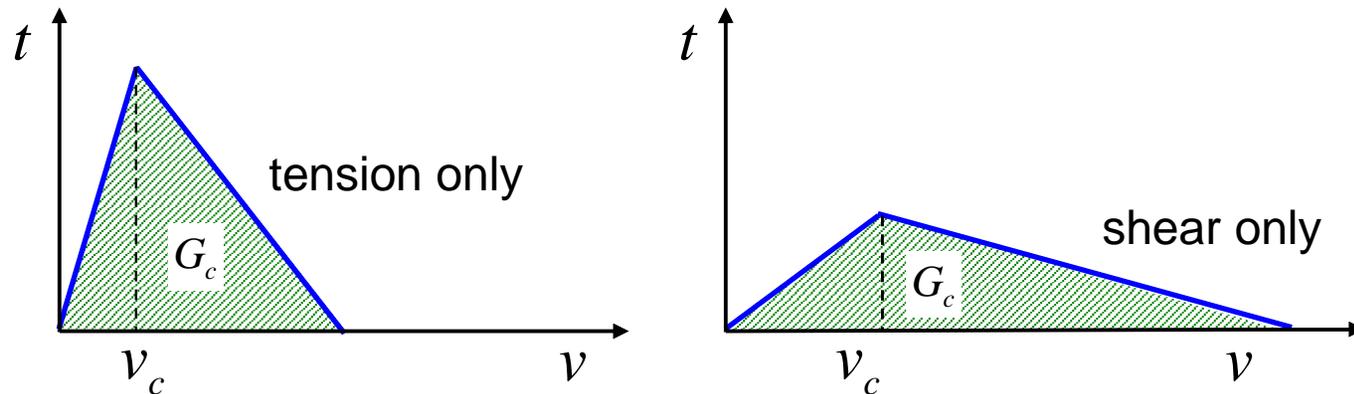
- Transfer of uniaxial data into a multi-axial case is done via an equivalent relative opening displacement ν and an equivalent effective traction t
- The basic equations assume that the maximum traction in tension and shear is the same and the cohesive energy in tension and shear is the same
- The following parameters are introduced to reflect the material difference between tension and shear, and between tension and compression
 - Shear-normal stress ratio β_1 which defines the ratio between the maximum traction in shear and in tension
 - Shear-normal energy ratio β_2 which defines the ratio between the cohesive energy in shear and in tension
 - Stiffening factor in compression F which scales up the material stiffness in compression

COHESIVE ZONE MODELING – MATERIAL

- Shear-normal stress ratio β_1

$$v = \sqrt{v_n^2 + \beta_1^2 v_s^2 + \beta_1^2 v_t^2}$$

Example: $\beta_1 = 0.5$



- **Limitation:**
 - The cohesive energy in tension and shear mode is still the same

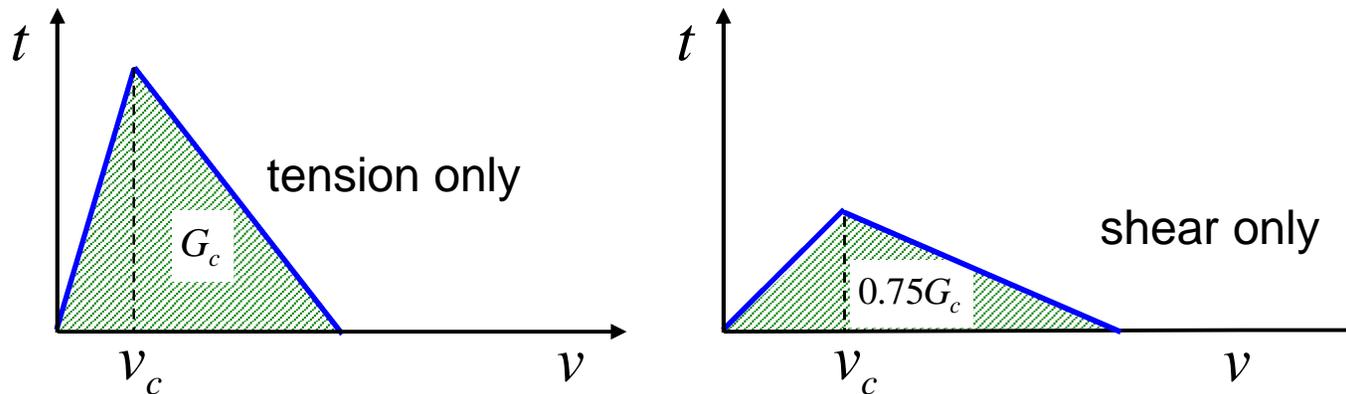
COHESIVE ZONE MODELING – MATERIAL

- Shear-normal energy ratio β_2

$$v = \sqrt{v_n^2 + \beta_1^2 v_s^2 + \beta_1^2 v_t^2}$$

$$G_c^{shear\ only} = \beta_2 G_c^{tension\ only}$$

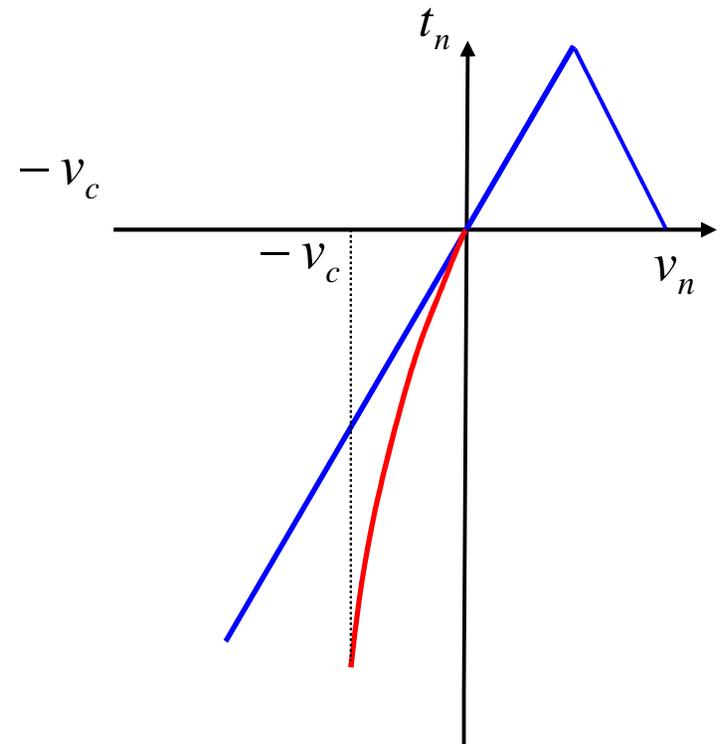
Example: $\beta_1 = 0.5$; $\beta_2 = 0.75$



COHESIVE ZONE MODELING – MATERIAL

- **Stiffening factor in compression F**
 - Compression will not cause failure of the interface layer but may cause penetration
 - Penetration can be reduced by increasing the stiffness factor in compression F
 - The stiffness at an opening displacement of equals F times the stiffness at a zero opening displacement

$$\left[\frac{\partial t_n}{\partial v_n} \right]_{-v_c} = F \left[\frac{\partial t_n}{\partial v_n} \right]_0$$



COHESIVE ZONE MODELING – MATERIAL

- Sometimes cohesive materials are defined by the cohesive energy and the maximum traction (t_c), instead of the critical or maximum opening displacement. The relationships to convert such data are:

- exponential model: $v_c = \frac{G_c}{et_c}$;

- linear model: $v_m = \frac{2G_c}{t_c}$;

- linear-exponential model: $v_c = \frac{2qG_c}{t_c(q+2)}$

- For the linear model, user still needs the critical opening displacement

COHESIVE ZONE MODELING – VISCOUS DAMPING

- Upon the onset of delamination the FE analysis may become instable
- User can activate some viscous (numerical) damping to improve convergence
- Viscous damping adds an extra viscous traction to the regular traction

$$t_v = \min\left(\frac{\zeta t_c \dot{v}}{\dot{v}_r}, t_c\right)$$

in which ζ is the viscous energy factor, \dot{v} is the equivalent relative displacement rate and \dot{v}_r is the reference value of the equivalent relative displacement rate (which can be user-defined or calculated by the solver)

COHESIVE ZONE MODELING – MATERIAL DEFINITION

MCOHE	MID	MODEL		TID					
	COHE	CRTOD	MAXOD	SNSR	EXP	VED	RRRD	SFC	
	SNER								

MID Identification number of a MCOHE entry. (Integer > 0)

MODEL (Integer > 0; Default = 1)

1 Bilinear model

2 Exponential model

3 Linear-exponential model

TID Table identifier for a combination of TABLES1/TABLEST for cohesive energy vs temperature. (Integer ≥ 0 ; Default = 0)

COHE G_c Cohesive energy. (Real ≥ 0.0)

CRTOD V_c Critical opening distance. (Real ≥ 0.0)

MAXOD V_m Maximum opening displacement (bilinear model only). (Real ≥ 0.0)

SNSR β_1 Shear Normal Stress Ratio. (Real > 0.0; Default = 1.0)

EXP q Exponential decay factor (linear-exponential model only). (Real > 0.0; Default = 1.0)

VED ζ Factor for viscous energy dissipation. (Real ≥ 0.0 ; Default = 0.0)

RRRD \dot{v}_r Reference rate of relative displacement. Used only if $VED \neq 0.0$. A value of 0.0 implies that the reference rate will be automatically calculated. (Real ≥ 0.0 ; Default = 0.0)

SFC F Stiffening factor in compression. (Real ≥ 0.0 ; Default = 1.0)

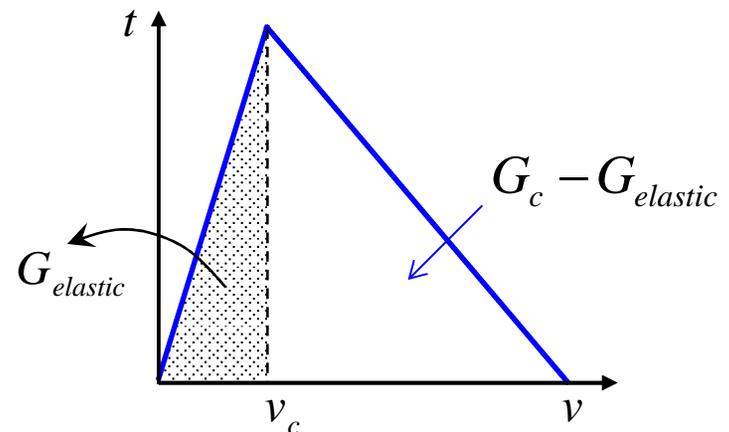
SNER β_2 Shear Normal Energy Ratio. (Real > 0.0; Default = 1.0)

COHESIVE ZONE MODELING - RESULTS

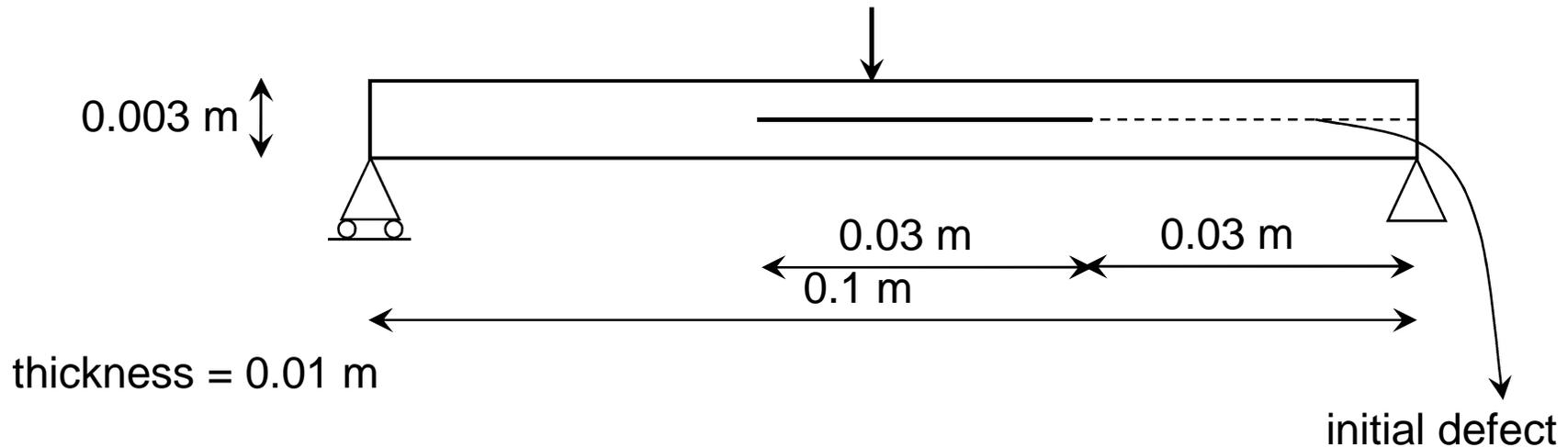
- Stress and strain in the cohesive elements can be requested for output
- A damage parameter can be used for post processing, indicating the amount of irreversible cohesive energy:

$$D = \frac{G - G_{elastic}}{G_c - G_{elastic}} ; 0 \leq D \leq 1 ; v_e \geq v_c$$

- If the maximum damage in all the integration points has been reached, the element can be automatically deactivated (not supported in SOL400 yet)



CZM EXAMPLE – END NOTCH FLEXURE (ENF)



Isotropic material:

$$E = 150 \text{ Gpa}$$

$$\nu = 0.25$$

Interface material (exponential):

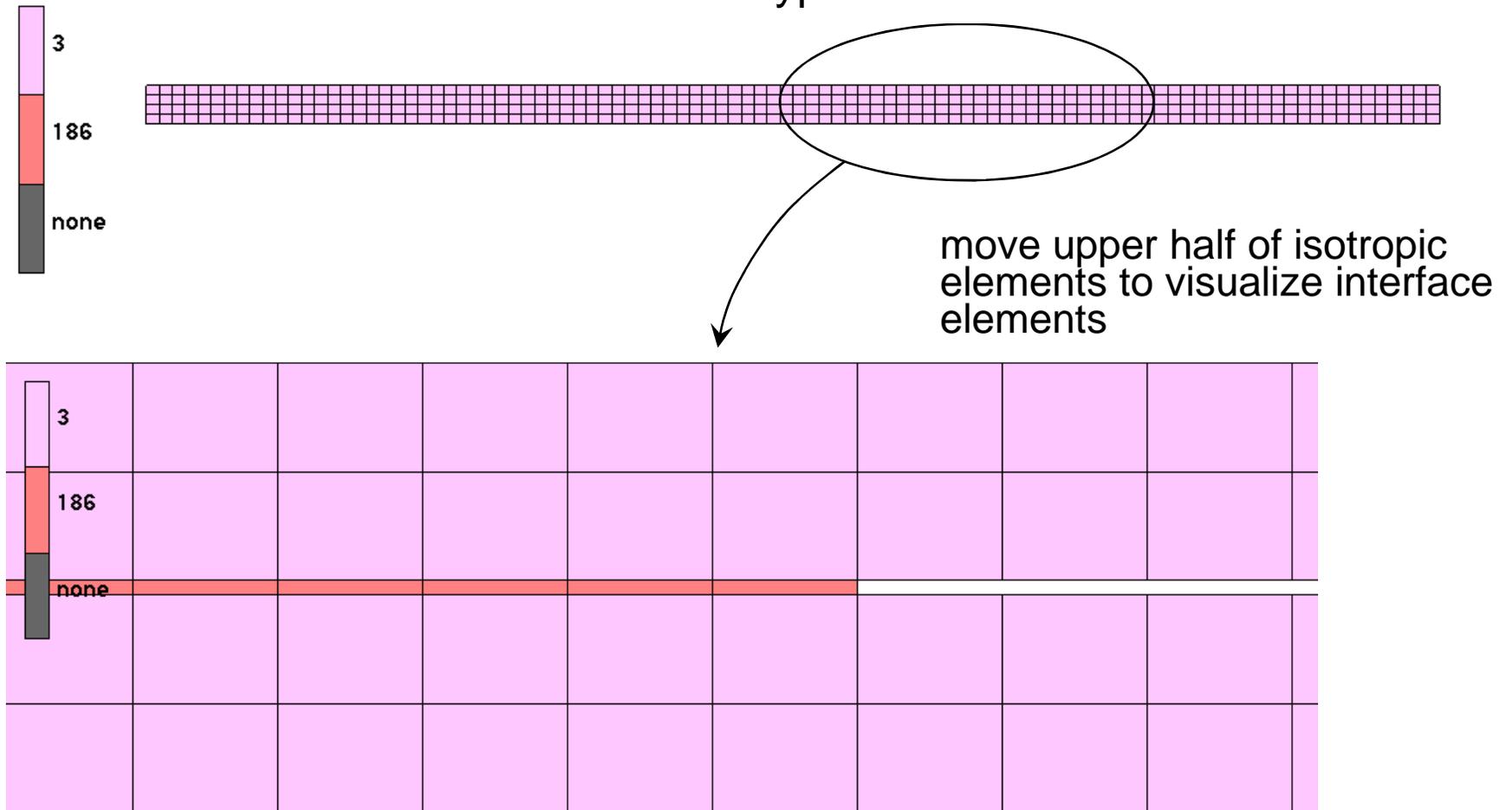
$$G_c = 1450 \text{ N / m}$$

$$t_c = 116.62 \text{ Mpa}$$

$$\beta_{1,2} = 1$$

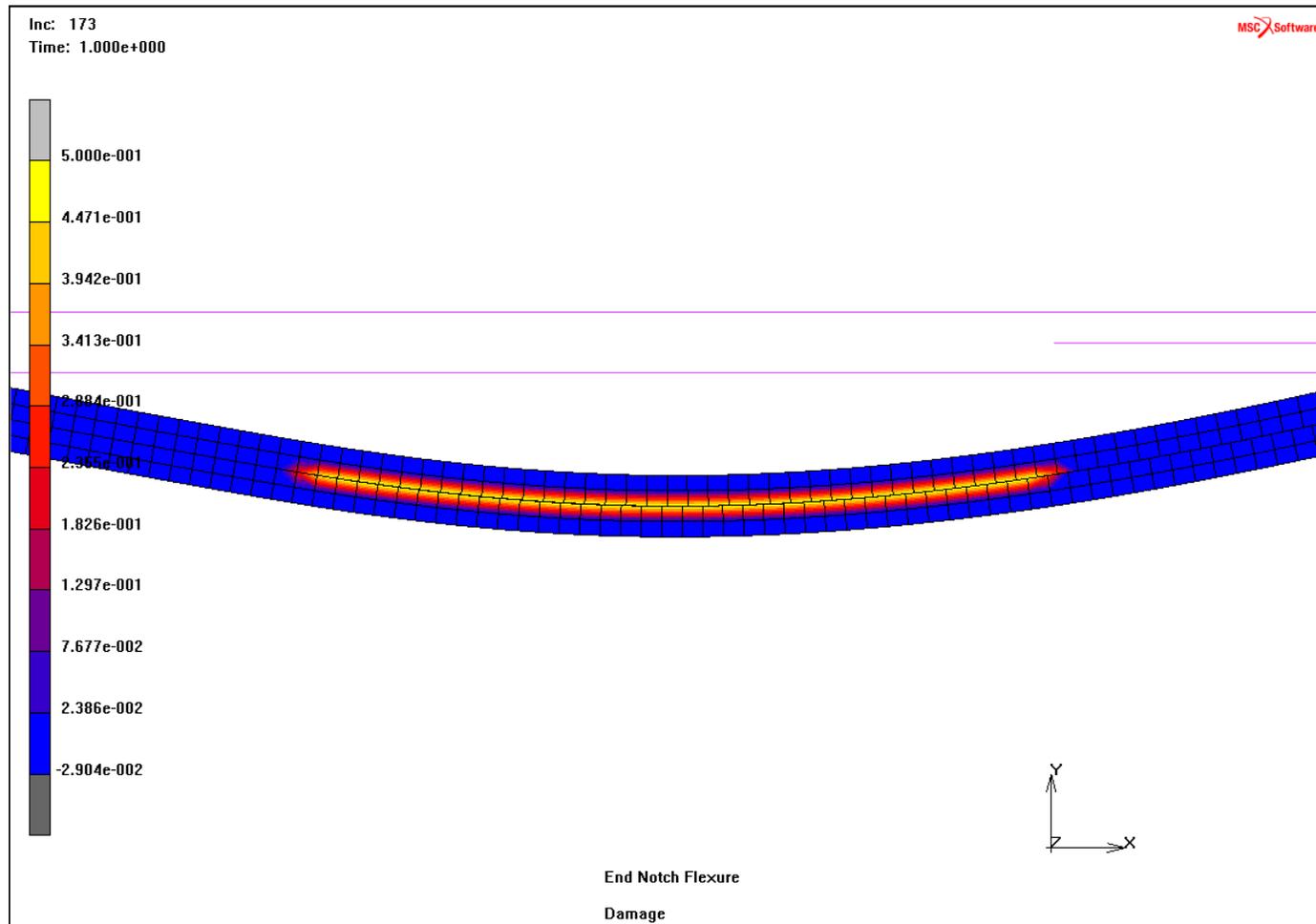
CZM EXAMPLE – END NOTCH FLEXURE (ENF)

FE model and element types used

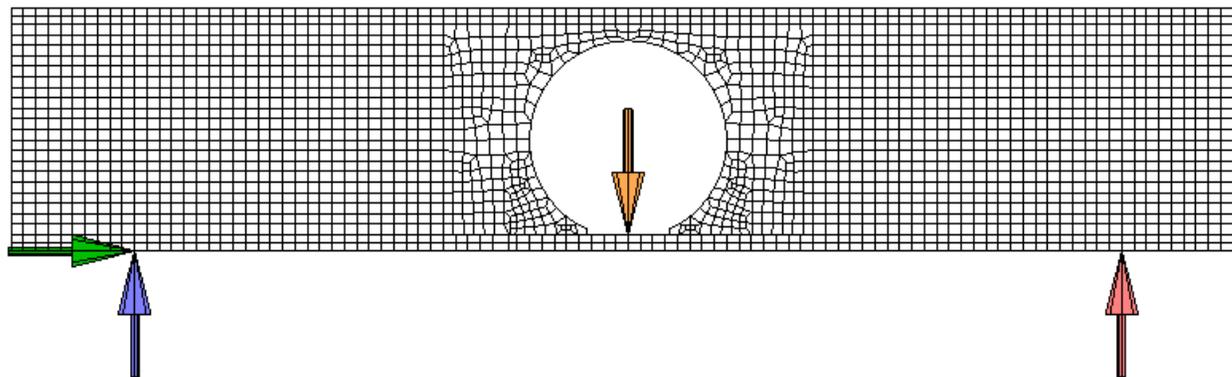
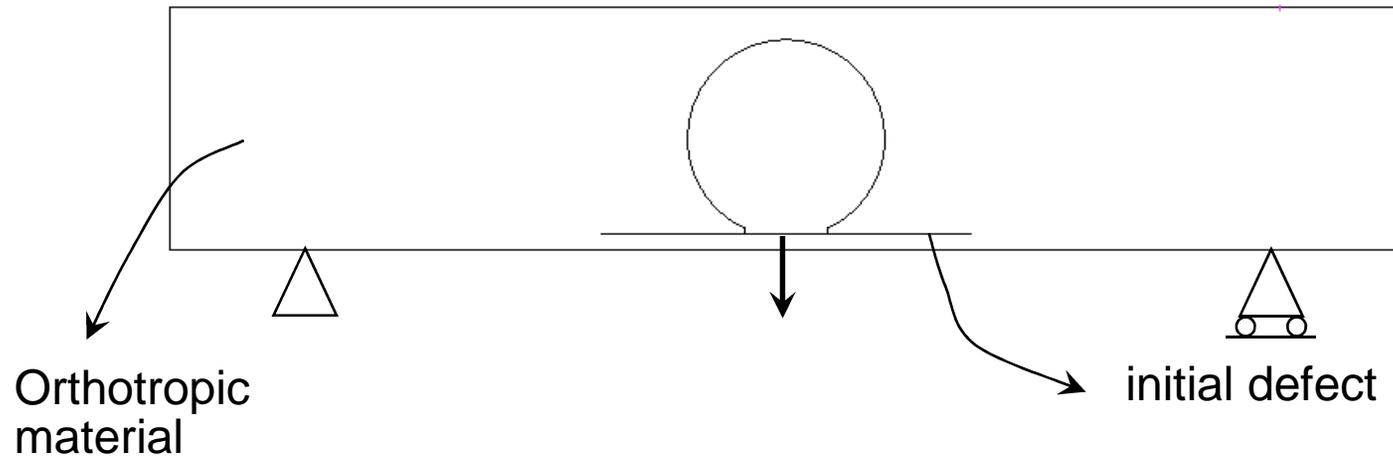


CZM EXAMPLE – END NOTCH FLEXURE (ENF)

Damage Evolution at Maximum Load Level

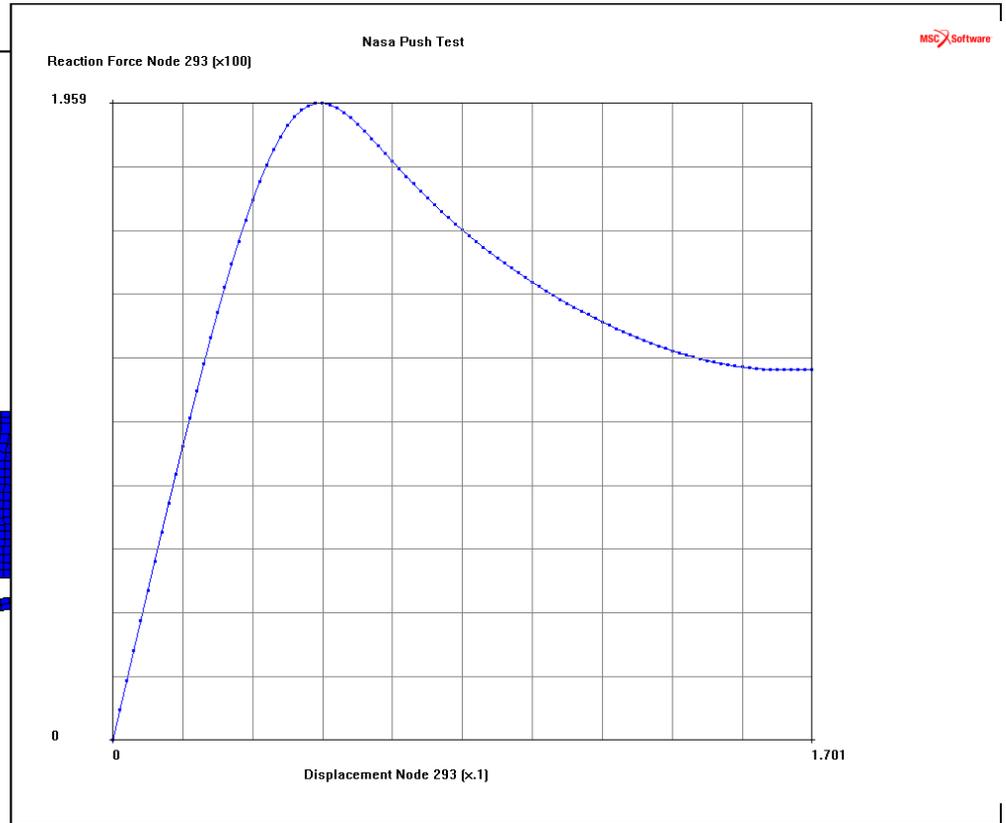
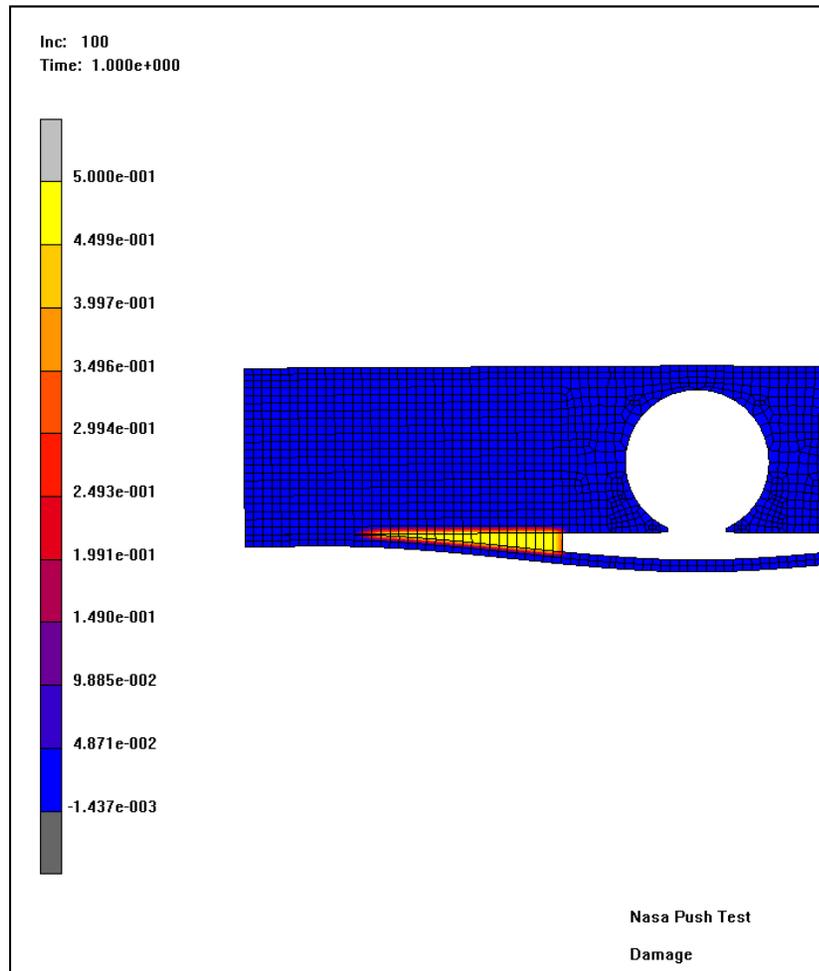


CZM EXAMPLE – NASA PUSH TEST



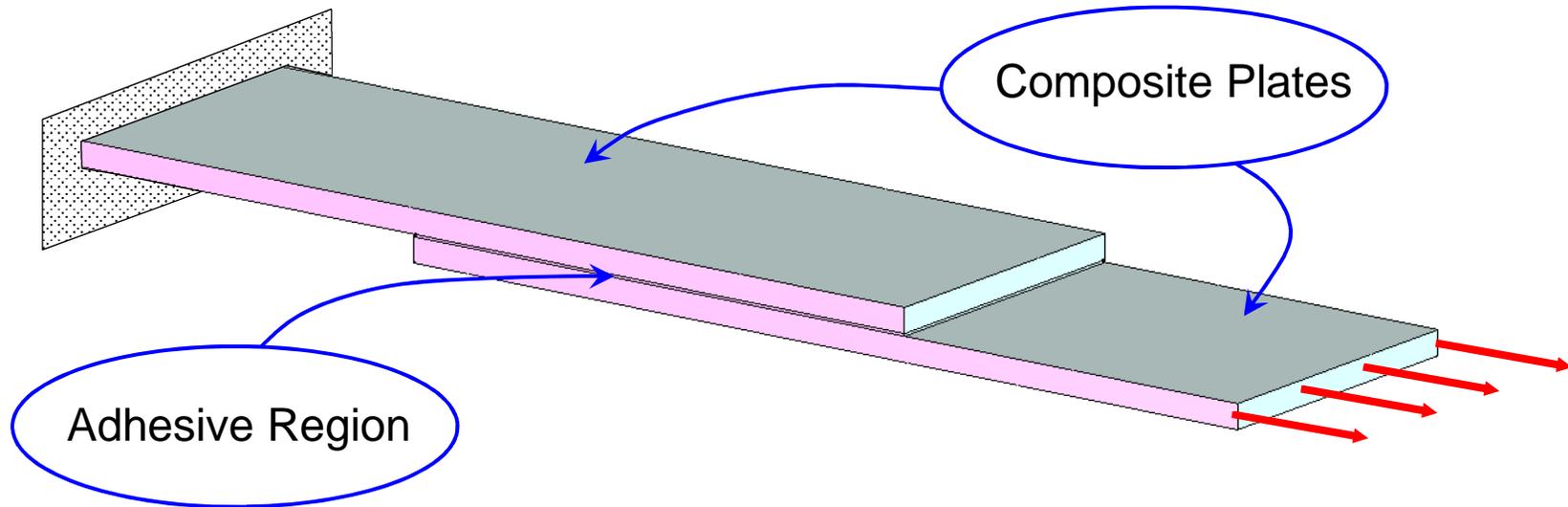
CZM EXAMPLE – NASA PUSH TEST

Results (Default Control Settings)



CZM – WORKSHOP 7

- Go to WS7



- **Interface material (exponential)**

- $G_c = 2.0 \text{ in-lb/in}^2$
- $V_c = 0.0002$
- $\beta_{1,2} = 1$

CZM WORKSHOP 7- INPUT FILE

```

SOL 400
CEND
ECHO = SORT
$! Case Control Section
SUBCASE 1
$! Subcase name : DefaultLoadCase
$LBCSET SUBCASE1          DefaultLbcSet
  SPC = 1
  LOAD = 1
  DISPLACEMENT(SORT1,PRINT,REAL)=ALL
  STRESS(SORT1,PRINT,REAL,VONMISES,CORNER)=ALL
  SPCFORCES(SORT1,PRINT,REAL)=ALL
  BCONTACT = 1
  ANALYSIS = NLSTAT
  NLSTEP = 1
BEGIN BULK
$
MCOHE          2          2          +
+              2.  0.0002          Cohesive
$
PCOHE          2          2          0          Interfac
$
CIFHEX        601          1771          1772          1393          1392          1365          1366          1773          1770+
+              602          1772          2          1394          1393          1366          1367          1775          1773+
CIFHEX        603          1772          1774          1395          1394          1367          1368          1777          1775+
+              604          1774          1776          1396          1395          1368          1369          1779          1777+
CIFHEX
+

```

Cohesive Material

Element Property

Cohesive Interface Elements

MCOHE

PCOHE

CIFHEX
+
CIFHEX
+
CIFHEX
+
CIFHEX
+

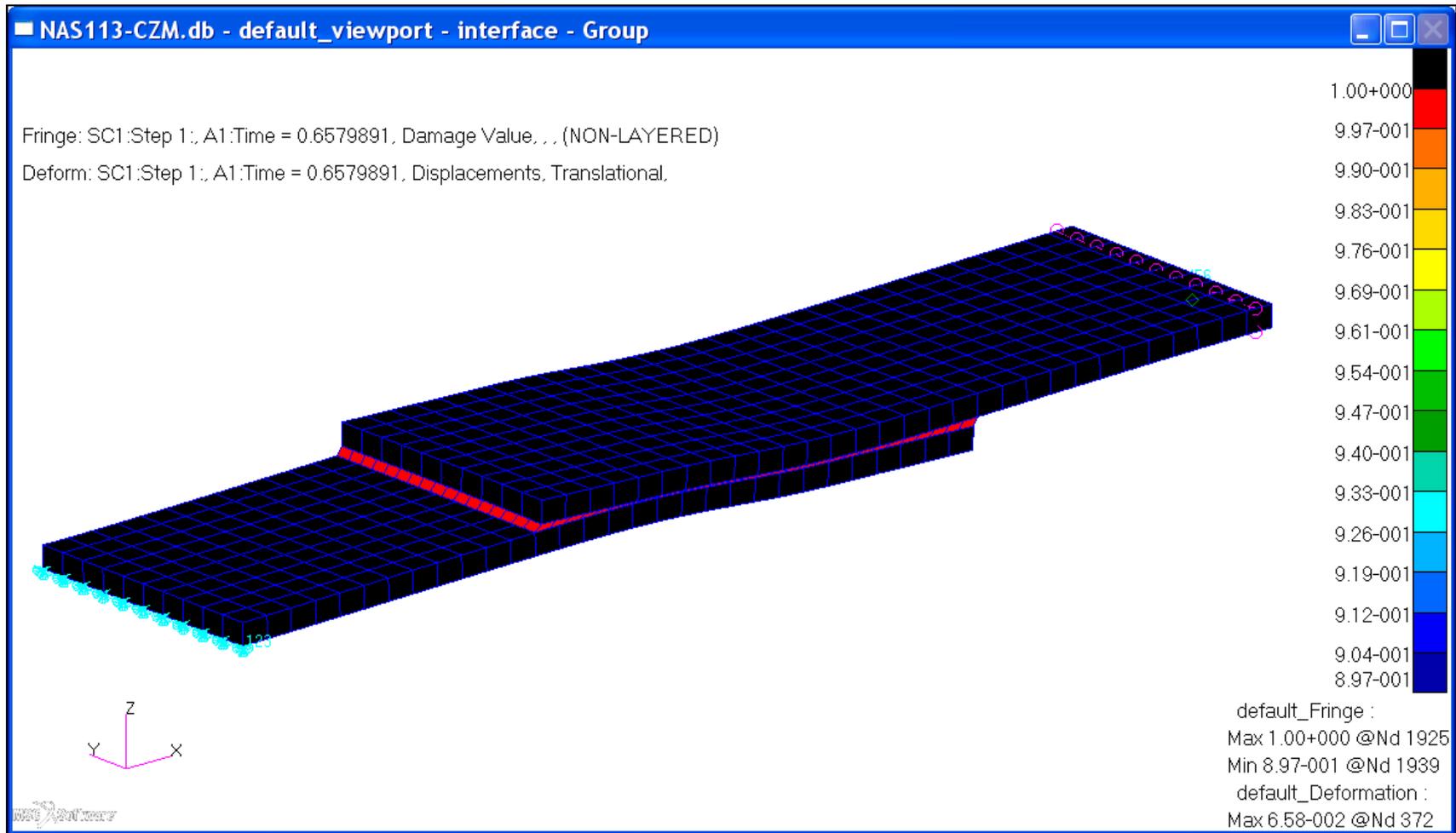
2 2
2. 0.0002

2 2 0

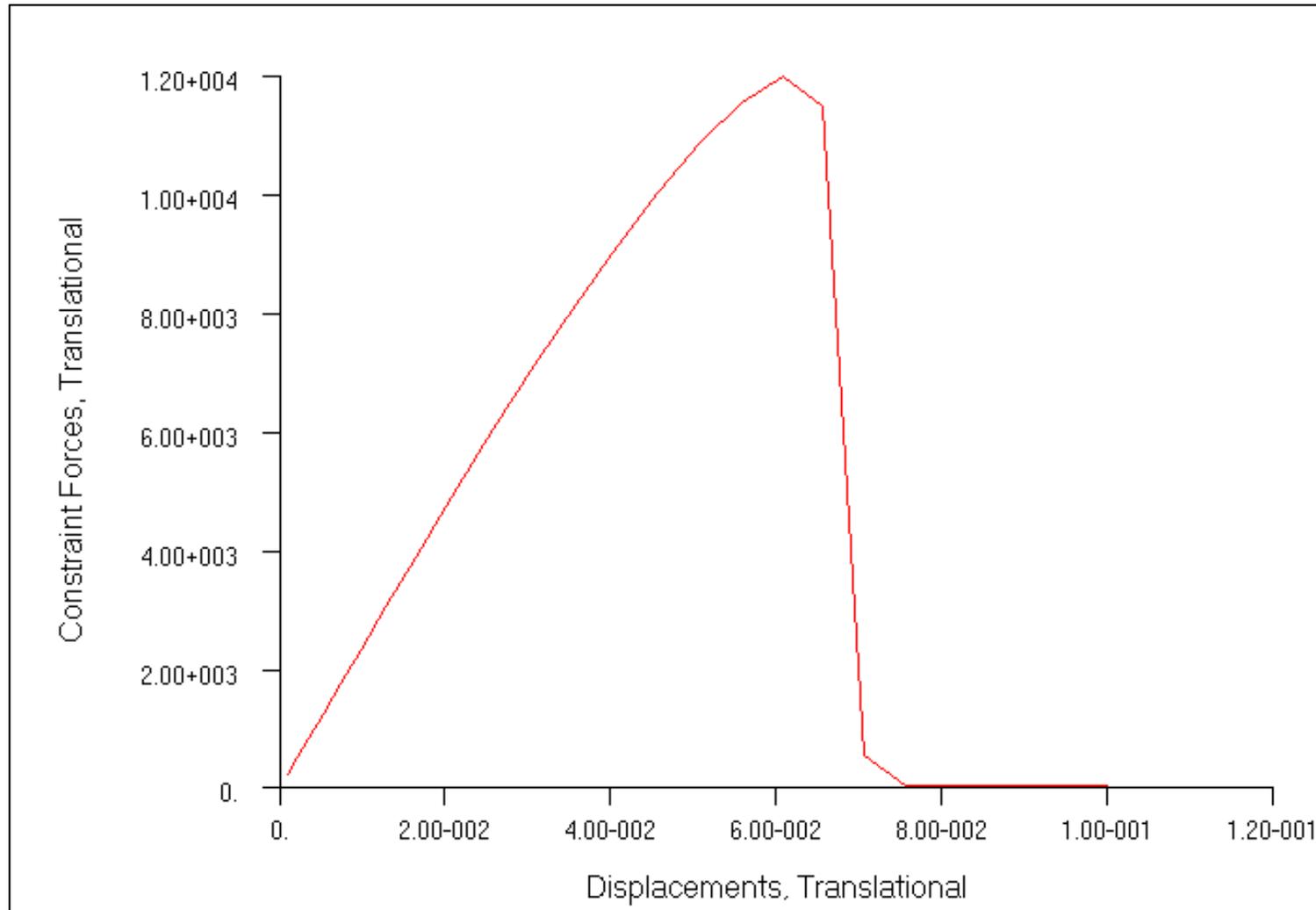
601 1771 1772 1393 1392 1365 1366 1773 1770+
602 1772 2 1394 1393 1366 1367 1775 1773+
603 1772 1774 1395 1394 1367 1368 1777 1775+
604 1774 1776 1396 1395 1368 1369 1779 1777+

+ Cohesive
Interfac

CZM WORKSHOP 7 – INTERFACE DAMAGE



CZM WORKSHOP 7 – LOAD-DEFLECTION CURVE





SECTION 7

INTERLAMINAR SHEAR

[G3] STIFFNESS MATRIX

- **Plate elements have a shear stiffness which can be defined as:**

where:

$$\begin{Bmatrix} V_x \\ V_y \end{Bmatrix} = [G_3] \begin{Bmatrix} \gamma_{xz} \\ \gamma_{yz} \end{Bmatrix}$$

- V_x and V_y are shear forces in the X and Y axis of the material coordinate system
 - $[G_3]$ is MID3 on PSHELL bulk data entry (see Appendix pgs. 7, 12, and 20 for details)
 - γ_{xz} and γ_{yz} are shear strains
- **The effective transverse shear stiffness matrix for composite plate elements uses elementary beam theory.**
 - **Loads are assumed to be transverse and not twist**
 - **If twist loads are significant, some error in the stiffness may be present.**

TRANSVERSE SHEAR STIFFNESS THEORY

- The transverse shear modulus for the X direction of the material coordinate system is:

$$\frac{1}{G_x} = \frac{T}{V_x^2} \sum_{i=1}^n \frac{(\tau_{xz})_i^2}{(G_x)_i} t_i \quad \text{Eq. 6-1}$$

(See [MSC Nastran Reference Manual](#), Section 13)

– Where:

- G_x is G1Z on the MAT8 bulk data entry, rotated to the X axis of the material coordinate system.
- V_x is transverse shear force in the direction of the X axis of the material coordinate system.
- T is total composite thickness.
- τ_{xz} is transverse shear stress in the direction of the X axis of the material coordinate system.
- “i” is ply number and “n” is number of plies
- t_i is the thickness of ply i

1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G1Z	G2Z	RHO	
MAT8	1	20.+6	2.+6	0.35	1.6+6	1.6+6	1.6+6	1.3-4	

TRANSVERSE SHEAR STRESS

- In a region of constant E_x , we can integrate to yield the following expression (the prediction of transverse shear stress in a composite):

$$\tau_{xzi} = C_i + \frac{V_x}{(\bar{E}I)_x} \left(\bar{z}_x z - \frac{z^2}{2} \right) E_{xi}$$

Eq. 6-2

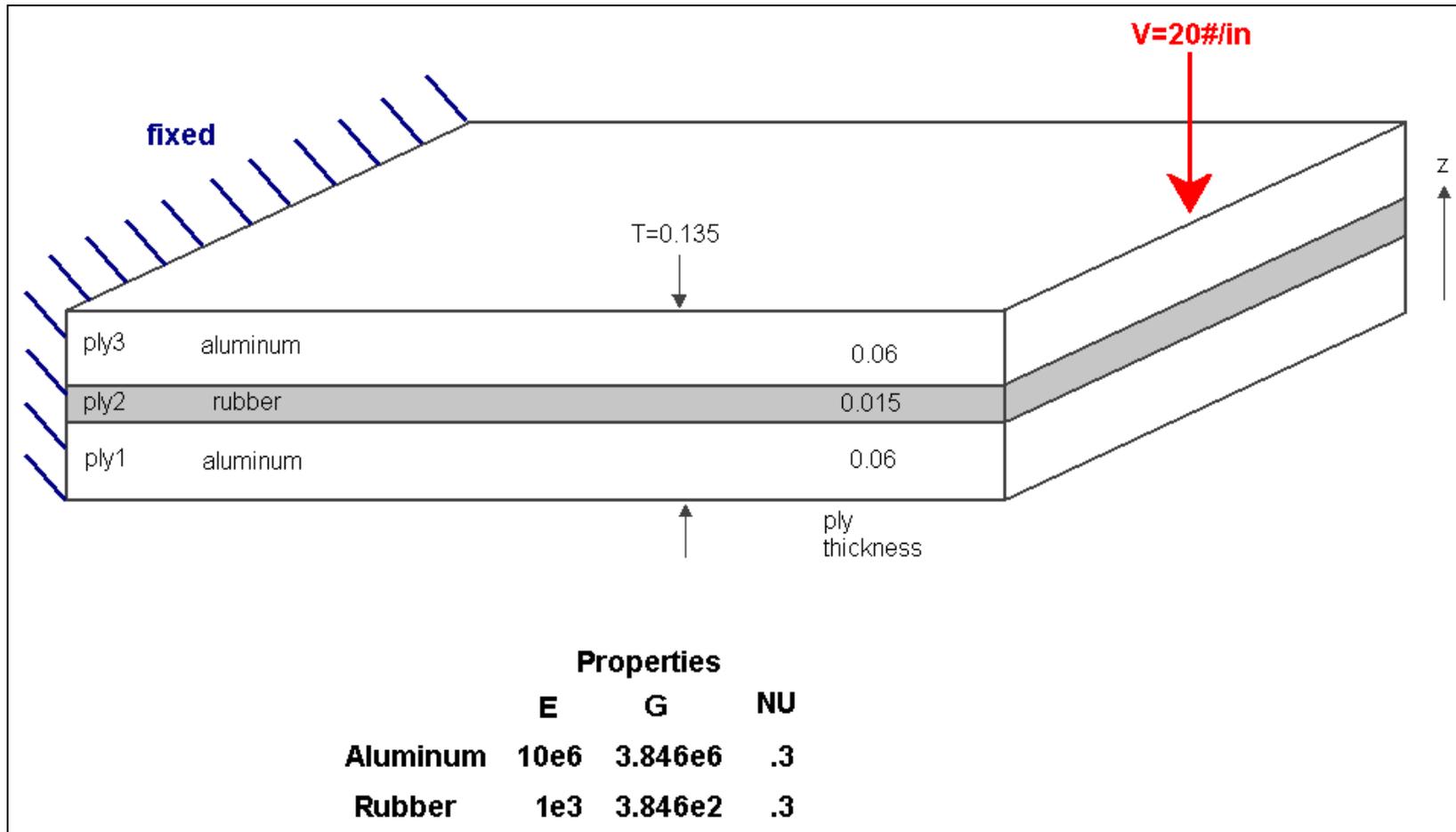
(See [MSC Nastran Reference Manual](#), Section 13)

- **Where:**
 - C_i is a constant of integration
 - V_x is the transverse shear force
 - $(\bar{E}I)_x$ (the average bending stiffness) is $1/D_{11}^*$ where D_{11}^* is the 11 term from the inverse of the ABD 6x6 matrix (see Appendix, pg. 7-10, 20, and 23)
 - z is the stress distance from composite center
 - \bar{z}_x is the location of the neutral axis
 - E_{xi} is Young's Modulus of ply i in the X axis of the material coordinate system

TRANSVERSE SHEAR STRESS

- The transverse shear stresses versus thicknesses are inserted into Eq. 6-1 to calculate the overall laminate transverse shear modulus.
- See Section [MSC Nastran Reference Manual](#), Section 13 for more theoretical details.
- Additionally, see Appendix A of this course for detailed equations and derivations.

EXAMPLE OF TRANSVERSE SHEAR MODULUS CALCULATION



TRANSVERSE SHEAR EXAMPLE MSC NASTRAN INPUT FILE

.dat file

- Symmetry is used in this file.
- In order to demonstrate a good transverse shear distribution, each .06 aluminum layer will be split into 3 plies (of .02")
- The transverse shear load, V_x , will be represented as 4 lbf

```
init master(s)
nastran prtpcomp=1
SOL 101
TIME 5
CEND
TITLE = MSC NASTRAN transverse shear example
DISP = ALL
STRESS = ALL
STRAIN = ALL
FORCE = ALL
spc=1
load=1
BEGIN BULK
PARAM POST -1
$
GRID 1 0 0.0 0.0 0.0 0
...
$
CQUAD4 1 1 1 2 3 4
$
pcomp,1,,,,,sym,+
, 1, 0.02, 0.0,yes
, 1, 0.02, 0.0,yes
, 1, 0.02, 0.0,yes
, 2, 0.0075, 0.0,yes
$
mat1,1,10.e6,3.846e6
mat1,2, 1.e3,3.846e2
$
FORCE 1 2 0 2. -1.
FORCE 1 3 0 2. -1.
$
SPC1 1 1 123456 0.0
SPC1 1 4 123456 0.0
$
ENDDATA
```

TRANSVERSE SHEAR STRESS EXAMPLE CALCULATION

- At the bottom surface of the first layer, t_{xz} is zero, so equation 6-2 can be used to calculate C1 (integration constant for ply 1):

$$C_1 = -\frac{V_x}{(\bar{E}I)_x} \left(\bar{z}_x z_0 - \frac{z_0^2}{2} \right) E_{x1}$$

Eq. 6-3

(Equation 13-28 in the [MSC Nastran Reference Manual](#))

- **Since:**

– $\bar{z}_x = 0.0$ because of symmetry

– $\bar{E}I = 1/4.884e^{-4}$ for this example. This term is not available from MSC Nastran. The ABD 6x6 matrix must be inverted manually.

– $z_0 = -T/2 = -0.0675$ for bottom surface

– $E_{xi} = 10e6$ for ply 1

- **Then:**
$$C_1 = -\frac{4.0}{4.884e^{-4}} \left(-\frac{(-0.0675)^2}{2} \right) 10e^6 = 44.51$$
 Eq. 6-3

TRANSVERSE SHEAR STRESS EXAMPLE CALCULATION

- **Plugging C1 back into equation 6-2 gives the transverse shear stress between layer 1 and 2:**

$$\tau_{xz1} = C_1 + \frac{V_x}{(\bar{E}I)_x} \left(\bar{z}_x z_1 - \frac{z_1^2}{2} \right) E_{x1} \quad \text{Eq. 6-4}$$

- **Where:**

- $z_1 = -0.0675 + 0.02 = -0.0475$ at the location between ply 1 and 2
- $E_{xi} = 10e6$ for ply 1

- **So:**

$$\tau_{xz1} = 44.51 + \frac{4.0}{4.884e^{-4}} \left(-\frac{(-0.0475)^2}{2} \right) 10e^6 = 22.47 \text{ psi} \quad \text{Eq. 6-5}$$

TRANSVERSE SHEAR STRESS EXAMPLE CALCULATION

- Normally, at the interface between two plies with different E's, (since the stress is the same) the E_{x2} is plugged into equation 6-4 and solved for C_2 :

$$C_2 = \tau_{xz1} - \frac{V_x}{(\bar{E}I)_x} \left(\bar{z}_x z_1 - \frac{z_1^2}{2} \right) E_{x2} \quad \text{Eq. 6-6}$$

- That, in turn, is then used in equation 6-4 for the calculation of τ_{xz} between ply 1 and 2.
- This procedure is repeated to calculate τ_{xz} between every ply, until the top surface is reached (where $\tau_{xz}=0$ once again)

TRANSVERSE SHEAR STRESS EXAMPLE CALCULATION

- However, since the first 3 plies are the same material and orientation (and thus the same E_{xi}), $C1=C2=C3$, so in this example the calculation simplifies to:

$$\tau_{xz2} = 44.51 + \frac{4.0}{4.884e^{-4}} \left(-\frac{(-0.0275)^2}{2} \right) 10e^6 = 37.12 \text{ psi} \quad \text{Eq. 6-7}$$

$$\tau_{xz3} = 44.51 + \frac{4.0}{4.884e^{-4}} \left(-\frac{(-0.0075)^2}{2} \right) 10e^6 = 43.96 \text{ psi} \quad \text{Eq. 6-8}$$

TRANSVERSE SHEAR STRESS EXAMPLE CALCULATION

- As mentioned previously, at the interface between two plies with different E's, since the shear stress is the same, the E_{x4} is plugged into equation 6-8 and solved for C_4 :

$$C_4 = \tau_{xz3} - \frac{V_x}{(\bar{EI})_x} \left(\bar{z}_x z_3 - \frac{z_3^2}{2} \right) E_{x4} \quad \text{Eq. 6-9}$$

- Where:**
 - $E_{x4} = 1e3$ for ply 4

- So:**
$$C_4 = 43.96 - \frac{4.0}{4.884e^{-4}} \left(-\frac{(-0.0075)^2}{2} \right) 1e^3 = 43.96 \quad \text{Eq. 6-10}$$

- Then:**
$$\tau_{xz4} = 43.96 + \frac{4.0}{4.884e^{-4}} \left(-\frac{(0.0)^2}{2} \right) 10e^3 = 43.96 \text{ psi} \quad \text{Eq. 6-11}$$

TRANSVERSE SHEAR STRESS EXAMPLE CALCULATION

- **At the center ply, since $Z_4=0$, then $\tau_{xz} = C_4$**
- **The calculation of C_4 normally would have raised the stress at the center.**
 - However, since the E of the center ply is so low ($1e3$), coupled with the small thickness, the term to the right of τ_{xz} works out to around $5.49e-5$.
 - In turn, because C_4 did not change from C_3 , and at the middle the z term is zero, the stress at τ_{xz4} is the same as ply 3 (τ_{xz3}).
- **If all the plies had the same E , then C_4 would have been higher as would the maximum stress, and the entire distribution would have looked parabolic (as typically in a rectangular beam of a constant material).**

TRANSVERSE SHEAR STRESS EXAMPLE CALCULATION

- The top half of the layup has stresses symmetric to the bottom half.
- Compare the hand calculated transverse shear stresses to those calculated by MSC NASTRAN
 - These are listed in the SHEAR XZ-MAT column, on the following slide.
 - The values correlating to the hand-calculated values are boxed in blue.

Hand Calcs:

- Txz1 = 22.47 psi
- Txz2 = 37.12 psi
- Txz3 = 43.96 psi
- Txz4 = 43.96 psi

MSC Nastran:

```
INTER-LAMINAR  
SHEAR XZ-MAT SHE  
-2.24664E+01 -1.8  
-3.71184E+01 -2.9  
-4.39560E+01 -3.5  
-4.39561E+01 -3.5  
-4.39560E+01 -3.5  
-3.71184E+01 -2.9
```

TRANSVERSE SHEAR STRESS EXAMPLE CALCULATION

.f06 file excerpt

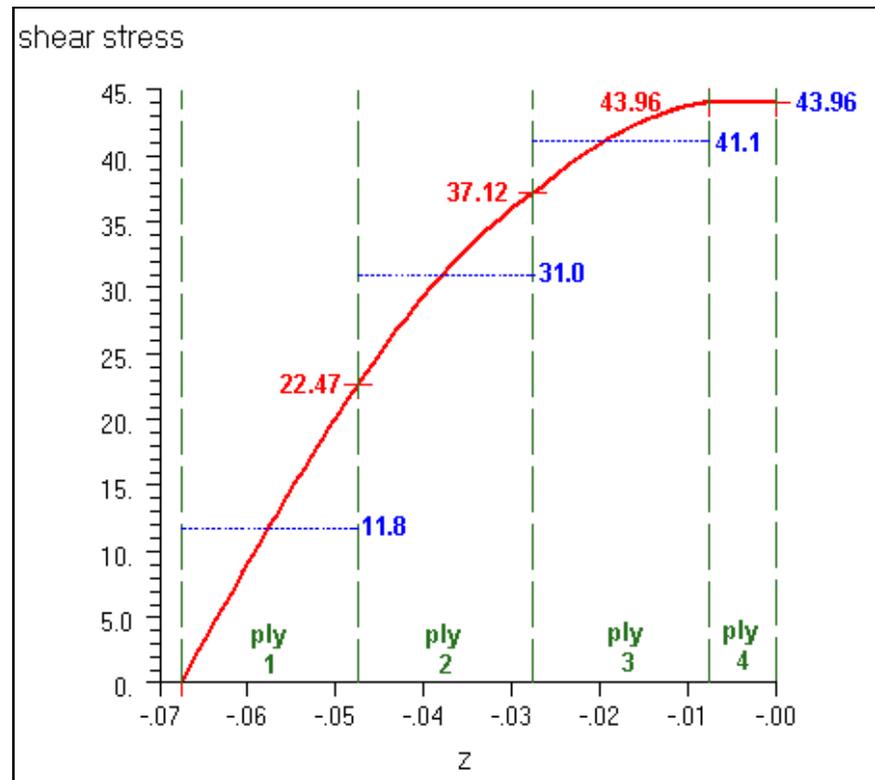
STRESSES IN LAYERED COMPOSITE ELEMENTS (QUAD4)											
ELEMENT ID	PLY ID	STRESSES IN FIBER AND MATRIX DIRECTIONS			INTER-LAMINAR STRESSES			PRINCIPAL STRESSES (ZERO SHEAR)			MAX
ID	ID	NORMAL-1	NORMAL-2	SHEAR-12	SHEAR XZ-MAT	SHEAR YZ-MAT	ANGLE	MAJOR	MINOR	SHEAR	
0	1	1	-5.61661E+02	-4.21318E+01	-5.99415E-14	-2.24664E+01	-1.81225E-15	-90.00	-4.21318E+01	-5.61661E+02	2.59764E+02
0	1	2	-3.66300E+02	-2.74773E+01	-3.90923E-14	-3.71184E+01	-2.99415E-15	-90.00	-2.74773E+01	-3.66300E+02	1.69412E+02
0	1	3	-1.70940E+02	-1.28227E+01	-1.82431E-14	-4.39560E+01	-3.54570E-15	-90.00	-1.28227E+01	-1.70940E+02	7.90587E+01
0	1	4	-3.66300E-03	-2.74773E-04	-3.90922E-19	-4.39561E+01	-3.54570E-15	-90.00	-2.74773E-04	-3.66300E-03	1.69411E-03
0	1	5	3.66301E-03	2.74773E-04	3.90923E-19	-4.39560E+01	-3.54570E-15	0.00	3.66301E-03	2.74773E-04	1.69412E-03
0	1	6	1.70940E+02	1.28227E+01	1.82431E-14	-3.71184E+01	-2.99415E-15	0.00	1.70940E+02	1.28227E+01	7.90587E+01
0	1	7	3.66300E+02	2.74773E+01	3.90923E-14	-2.24664E+01	-1.81225E-15	0.00	3.66300E+02	2.74773E+01	1.69412E+02
0	1	8	5.61661E+02	4.21318E+01	5.99415E-14	-3.65402E-13	-2.94750E-29	0.00	5.61661E+02	4.21318E+01	2.59764E+02

- The transverse shear stresses are output in the material coordinate system.
- Since there are n+1 transverse shear stress locations, and only n ply outputs, the first zero stress lower surface is not printed.
 - Ply 1's transverse shear stresses are between ply 1 and 2.
 - Ply 2's transverse shear stresses are between plies 2 and 3, etc.
- The near zero transverse shear stress (on the order of e-13 above), at the upper surface (listed on Ply 8) indicates correct calculation.

TRANSVERSE SHEAR MODULUS EXAMPLE CALCULATION

- Equation 6-1 is used so shear distribution through the thickness is needed.
- This curve is plotted from data from the MSC NASTRAN transverse shear stress output in the .f06 file in red at the ply boundaries.
- The blue are the estimated averages through the ply thickness.

Patran cannot do this plot so the Patran neutral plotter was used and then edited with a paint program.



TRANSVERSE SHEAR MODULUS EXAMPLE CALCULATION

- Using equation 6-1:
$$\frac{1}{G_x} = \frac{T}{V_x^2} \sum_{i=1}^n \frac{(\tau_{xz})_i^2}{(G_x)_i} t_i$$

$$\frac{1}{G_x} = \frac{0.135}{4^2} \left[0.02 \left(\frac{22.47^2 + 37.12^2 + 43.96^2}{3.846e^6} \right) + 0.0075 \left(\frac{43.96^2}{3.846e^2} \right) \right] \times 2 \quad \text{Eq. 6-12}$$

$$G_x = 1.572e^3$$

- Which compares very well with the G11 term of MID3 shown on the next page.

TRANSVERSE SHEAR MODULUS EXAMPLE CALCULATION

MID3
(transverse shear)

PSHELL	1	100000001	1.3500E-01	200000001	1.0000E+00	300000001	1.0000E+00	0.0000E+00
	-6.7500E-02	6.7500E-02	0					
MAT2	100000001	9.7685E+06	2.9310E+06	0.0000E+00	9.7685E+06	0.0000E+00	3.4187E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	200000001	1.0974E+07	3.2929E+06	0.0000E+00	1.0974E+07	0.0000E+00	3.8407E+06	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00
	0							
MAT2	300000001	1.5722E+03	0.0000E+00	0.0000E+00	1.5722E+03	0.0000E+00	0.0000E+00	0.0000E+00
	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00

.f06 file excerpt

$$G_x = 1.572e^3$$

- **G_x is closer to the rubber value because equation 6-1 closely resembles a series spring equation.**
 - Softest spring dominates: $1/K = 1/K_1 + 1/K_2 + \dots$

PATRAN TRANSVERSE SHEAR PROPERTIES

The screenshot shows the PATRAN software interface with the 'Properties' menu highlighted. The 'Laminated Composite' dialog box is open, showing a table with two rows of material data:

Material Name	Thickness	Orientation
1 graphite_epoxy_tape	5.400000E-003	0.000000E+0
2 graphite_epoxy_tape	5.400000E-003	4.500000E+001

Below the table, it shows 'Total Thickness in Spreadsheet = 0.0162' and 'Plies in Spreadsheet = 4'. There are buttons for 'Delete Selected Rows', 'Insert', and 'Show Laminated Properties...'. The 'Insert' button is set to '1' Rows, with 'Above' selected.

The screenshot shows the 'Materials' dialog box. The 'Action' is 'Modify', 'Object' is 'Composite', and 'Method' is 'Laminate'. Under 'Existing Materials', '8_ply_symmetric_quasi' and 'graphite_epoxy_tape' are listed. Under 'Laminated Comp. To Modify', '8_ply_symmetric_quasi' is selected. There are 'Filter' buttons for both sections. At the bottom, there are '-Apply-' and 'Reset' buttons.

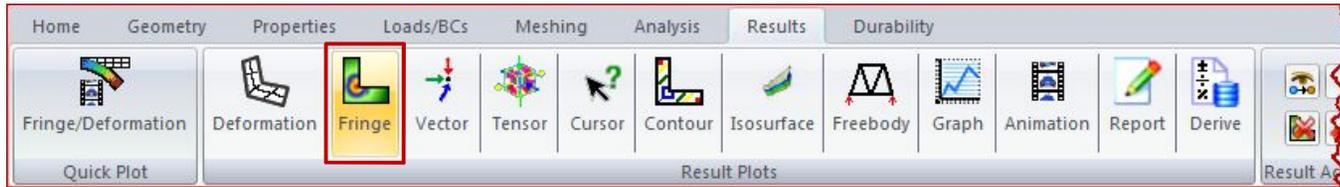
Materials/Modify/ Composite
 Laminated Comp. To Modify
 "pcomp.3"
 Show Laminate Properties
 E's, NU's, G's, and Qij's
 G12,23,31 show calculated 3D
 shear terms
 Also seen in 3D Elasticity Matrix
 and 3D Flexibility Matrix.
 Note that the 23 and 31 terms do
 not agree with the analysis.

The screenshot shows the 'Composite Material Properties' dialog box. It displays the 'Engineering Constants and 2D Elasticity Matrix' table:

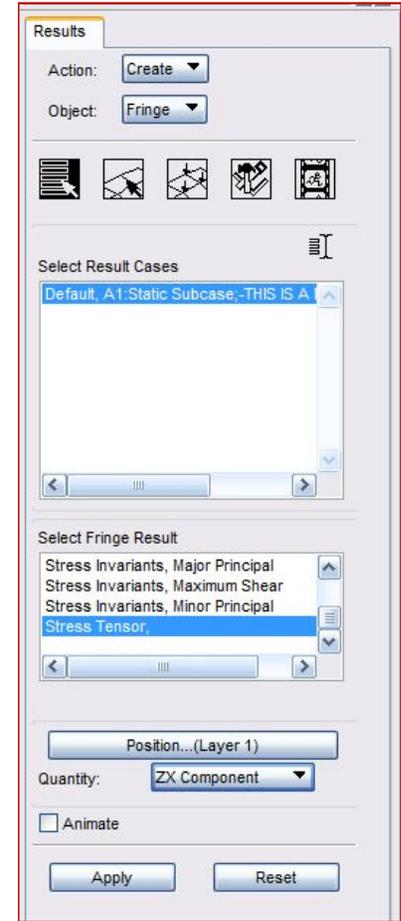
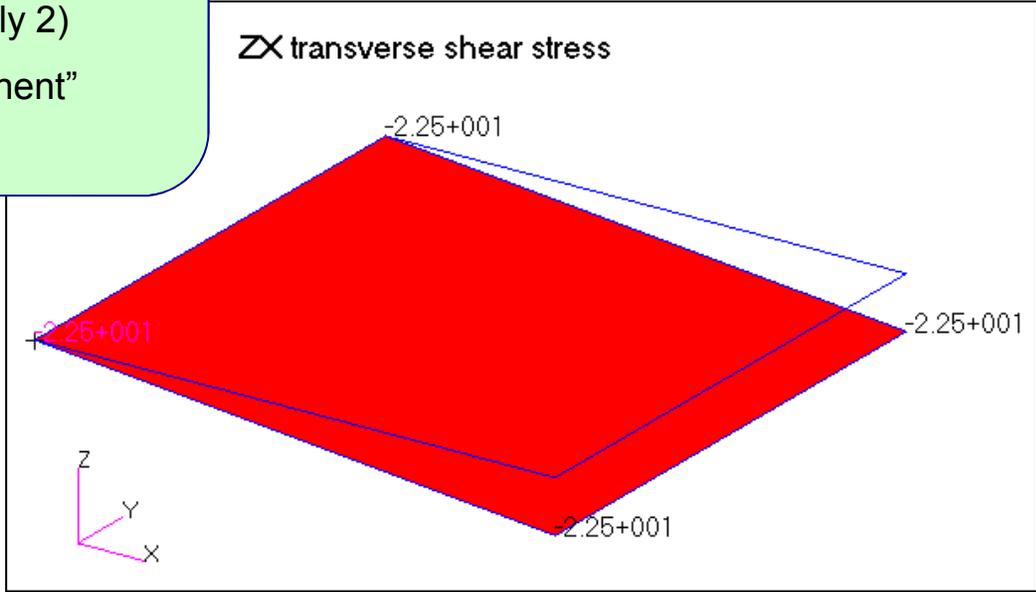
E11,22,33	NU12,23,13	G12,23,31	Q
0.00E+000	0.00E+000	1.00E-004	0.00E+000
0.00E+000	0.00E+000	0.00E+000	0.00E+000
0.00E+000	0.00E+000	0.00E+000	1.00E-004

At the bottom, there are 'Composite Property Display Options' with radio buttons for 'A, B, and D Matrices', '3D Flexibility Matrix', 'Thermal: Kij, Ni, and Mi', '3D Elasticity Matrix', 'E's, NU's, G's, and Qij's', and 'CTE's, CME's and Others'. The 'E's, NU's, G's, and Qij's' option is selected. There is a 'Cancel' button at the bottom.

PATRAN TRANSVERSE SHEAR STRESS

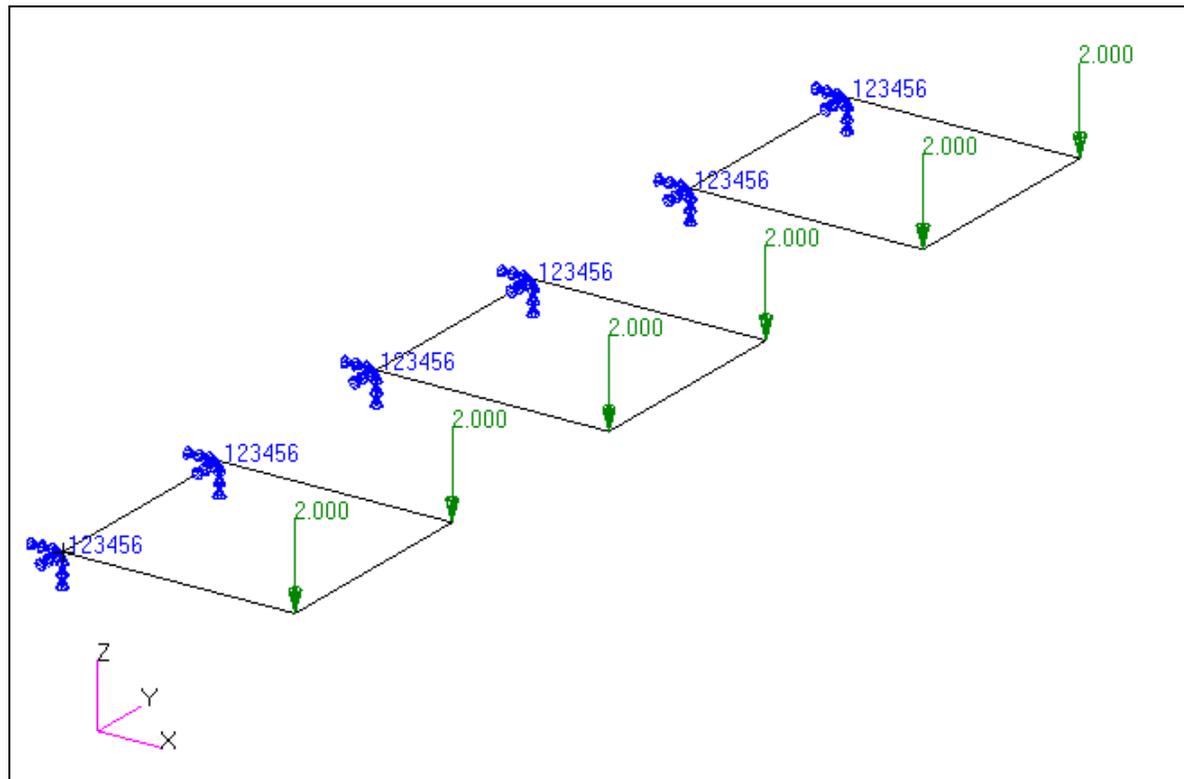


Results/Create/Fringe
Stress Tensor
Position "Layer1" for ply 1
(between ply 1 and ply 2)
Quantity "ZX Component"
Apply



EXERCISE

- Perform Workshop 8 “Transverse Shear Stress and Stiffness”



SECTION 8

NONLINEAR COMPOSITE ANALYSIS

LINEAR VS. NONLINEAR ANALYSIS

FEM QUANTITIES IN LINEAR ANALYSIS

- Kinematics

$$\mathbf{u}_e = \mathbf{T}_{eg} \cdot \mathbf{u}_g$$

Element Displacement Vector **Displacement Transformation Matrix** **Global Displacement Vector**

- Compatibility

$$\boldsymbol{\varepsilon} = \mathbf{B} \cdot \mathbf{u}_e$$

Element Strains **Strain Displacement Matrix** **Element Displacement Vector**

- Constitutive Law

$$\boldsymbol{\sigma} = \mathbf{D} \cdot \boldsymbol{\varepsilon}$$

Element Stresses **Stress-Strain Relationship** **Element Strains**

FEM QUANTITIES IN LINEAR ANALYSIS

- Equilibrium

$$\mathbf{P} = \sum \mathbf{T}_{eg}^T \cdot \mathbf{F}_e$$

External Load Vector Force Transformation Matrix Element Forces

- Constraints

$$\mathbf{u}_g = \alpha$$

- The transformation matrices do not change
- Force equals displacement transformation
- The constraints (SPC, MPC) do not change

LINEAR ANALYSIS CONSEQUENCES

- Solving a Linear System of Equations

$$\mathbf{K} \cdot \mathbf{u} = \mathbf{P}$$

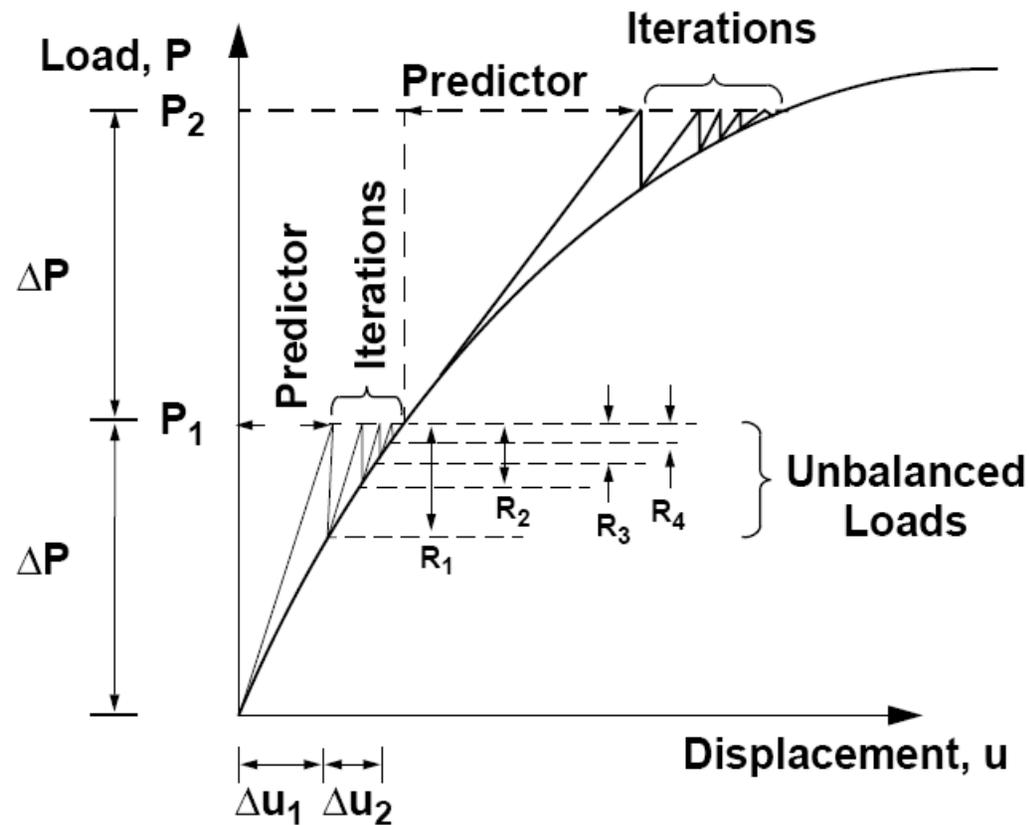
- In Linear Analysis it follows that:
 - Loads are independent of deformation
 - Displacements are directly proportional to the loads
 - Results for different loads can be superimposed

NONLINEAR ANALYSIS

- **In Nonlinear Analysis, upon deformation the following changes occur:**
 - Geometric nonlinearity $T_{(disp)}, \alpha$
 - Follower forces $T_{(force)}$
 - Large strain B
 - Material nonlinearity D
 - Contact α
- **It follows that the system of equations gets nonlinear, the load sequence is unique and the results must not be superimposed**

NONLINEAR ANALYSIS

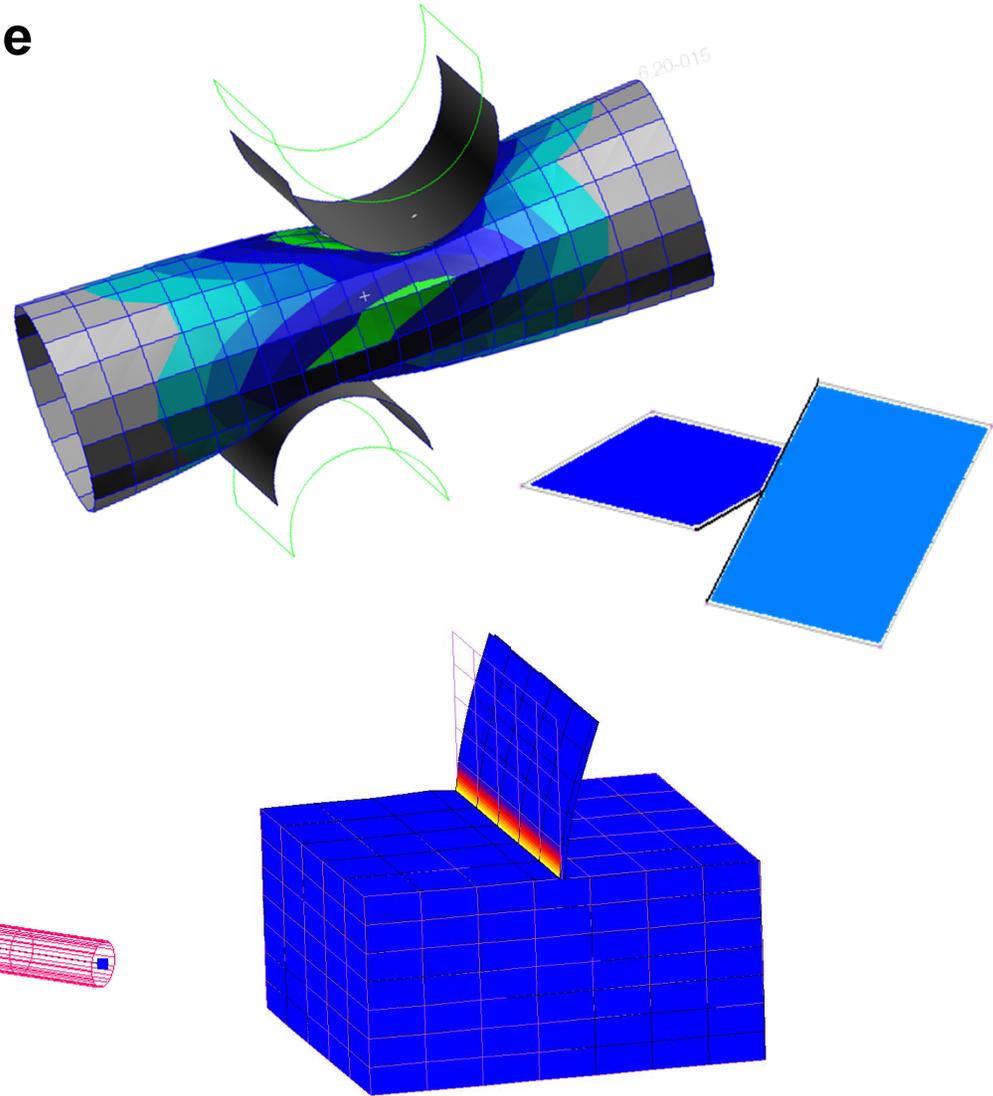
- Since the system of equations has become nonlinear an iteration strategy is needed



SOURCES OF NONLINEARITIES

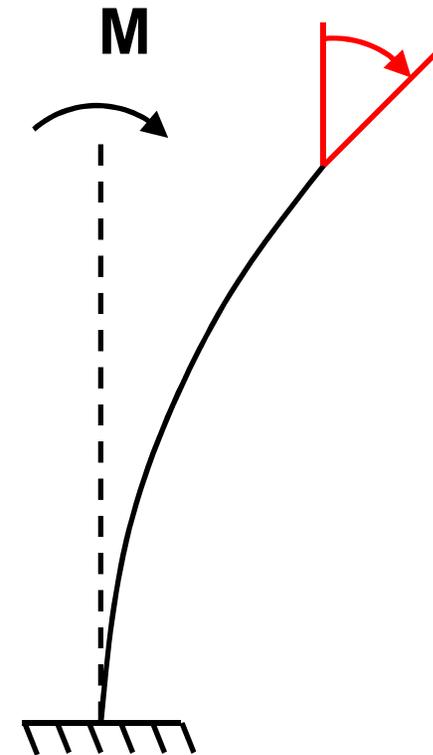
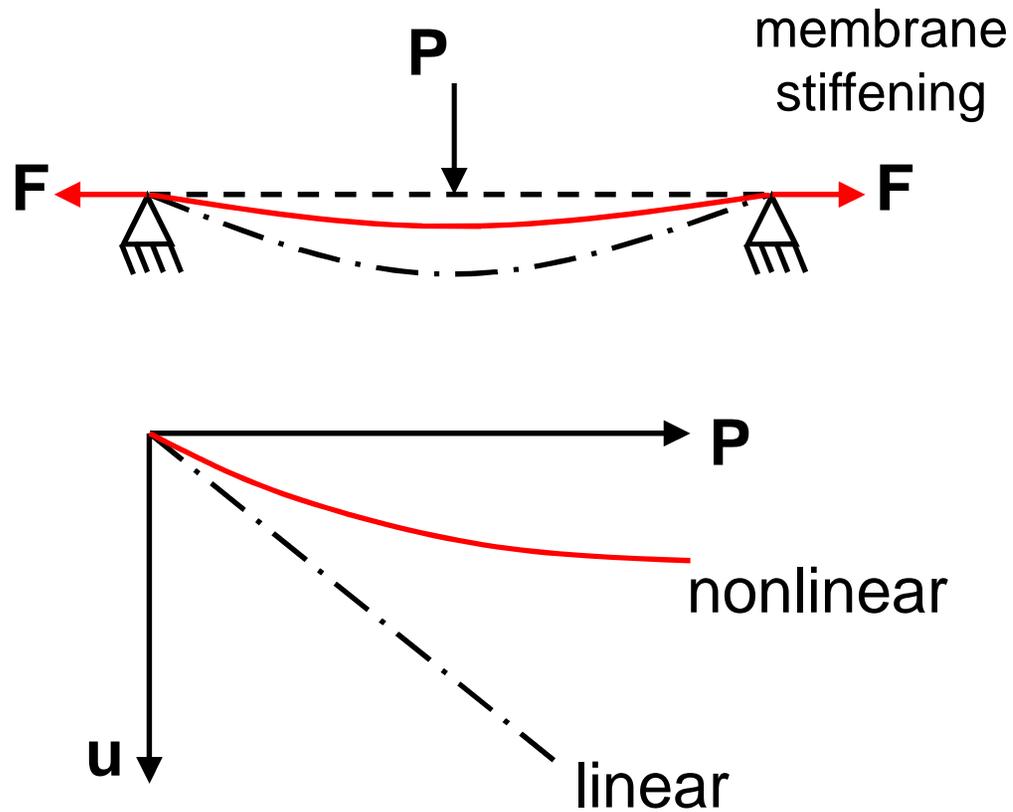
CONTACT

- **Deformable to Deformable**
 - Surface to Surface
 - Edge to Surface
 - Edge to Edge
 - Beam to Beam
- **Rigid to Deformable**



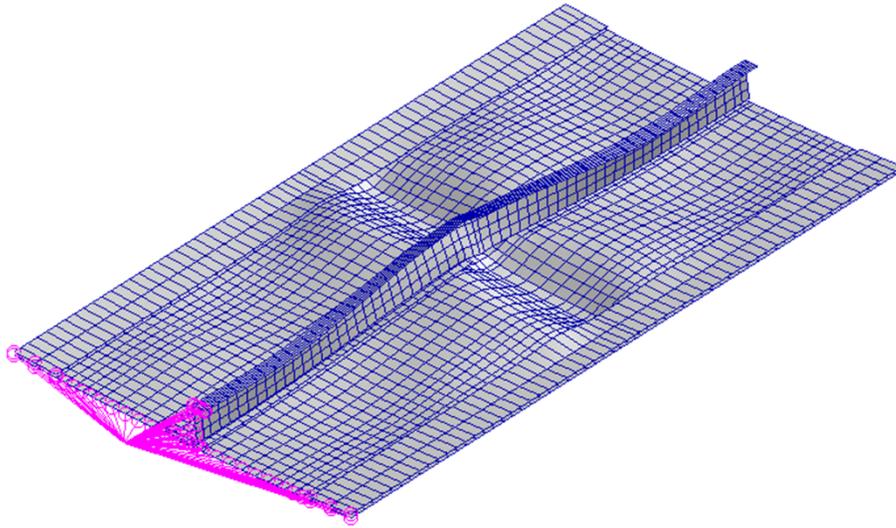
GEOMETRIC NONLINEARITY

- Large displacements and rotations, but small strains

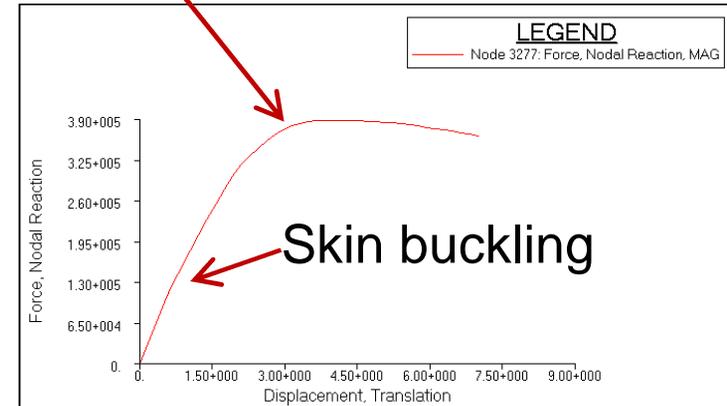


GEOMETRIC NONLINEARITY

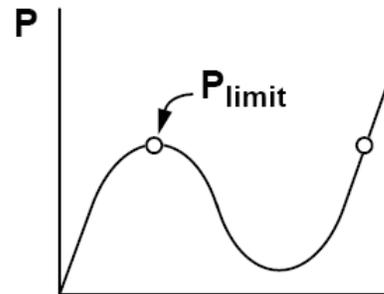
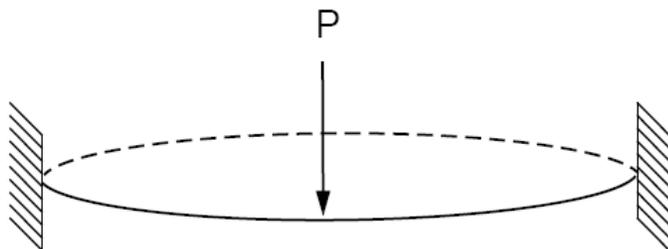
- Buckling



Panel failure



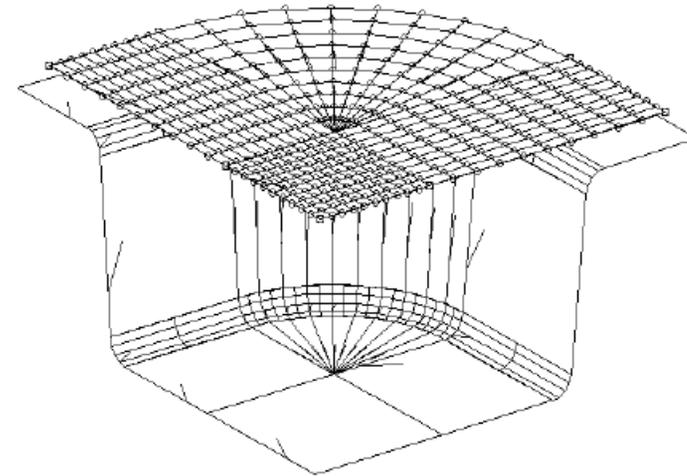
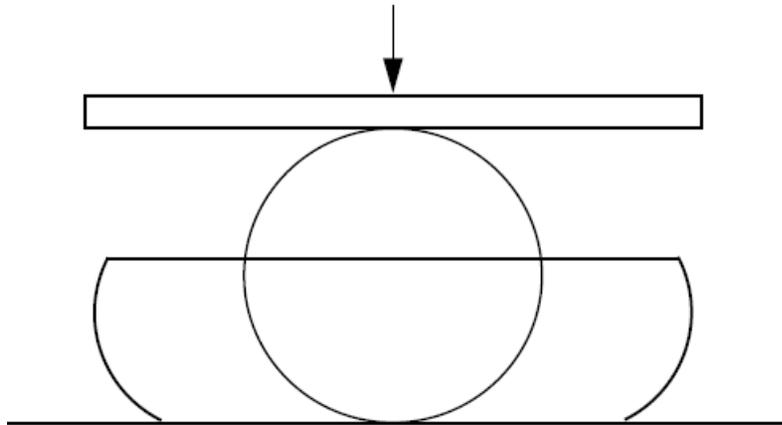
- Snap-Through



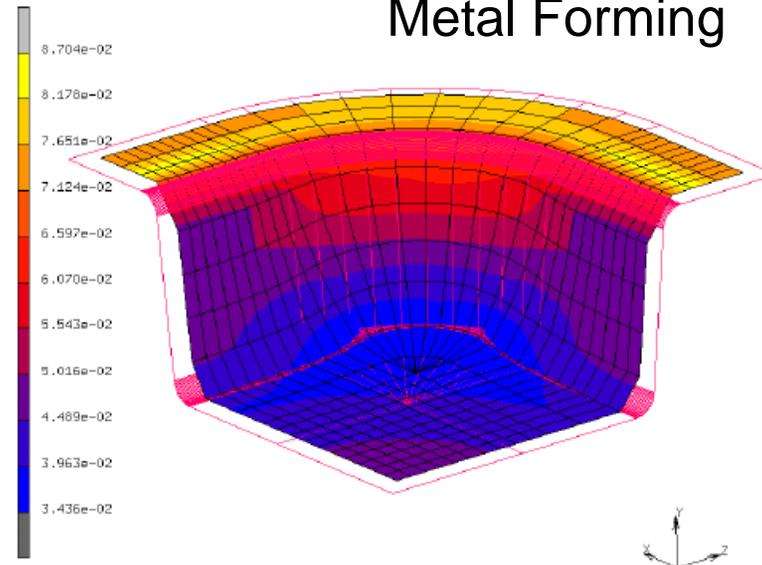
GEOMETRIC NONLINEARITY

- Large displacement and rotation and large strains

Rubber Bearing

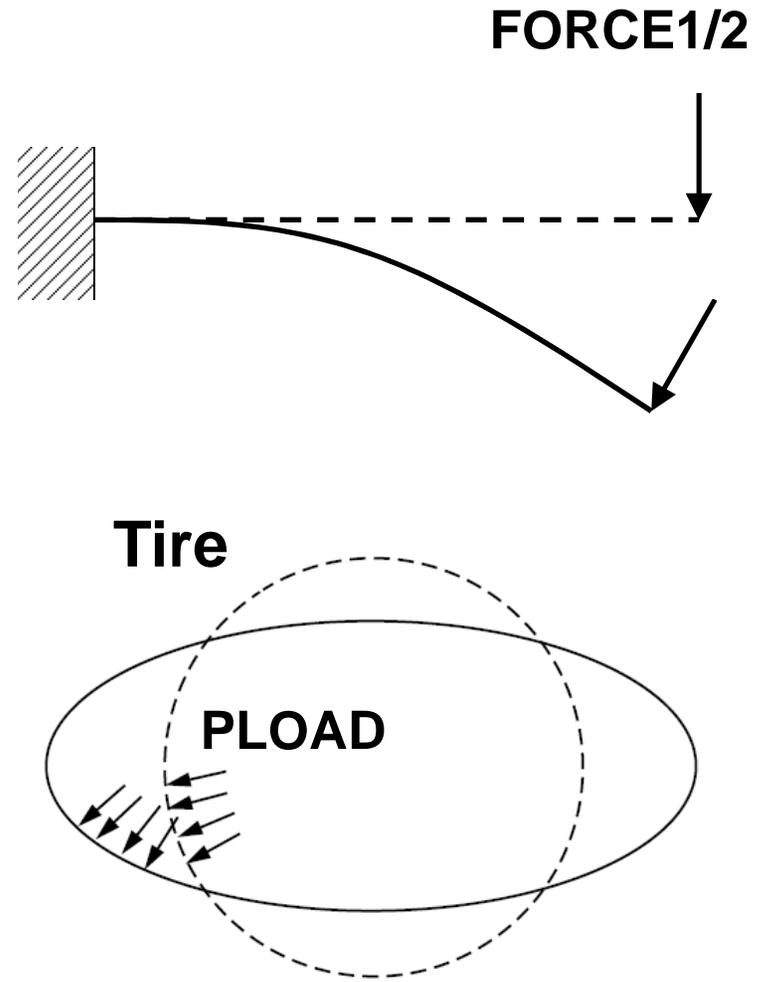
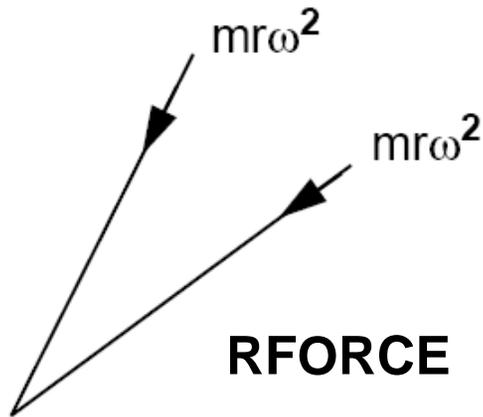


Metal Forming



GEOMETRIC NONLINEARITY

- Follower Forces

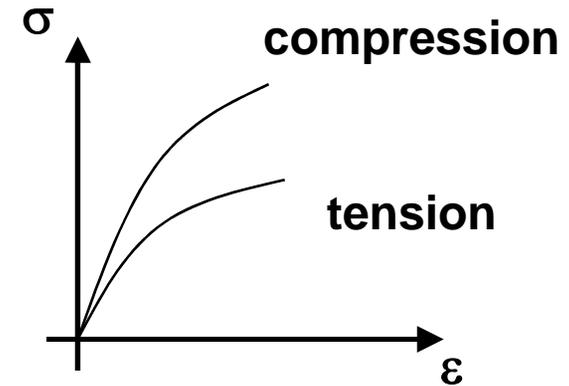


Temperature Loads

MATERIAL NONLINEARITY

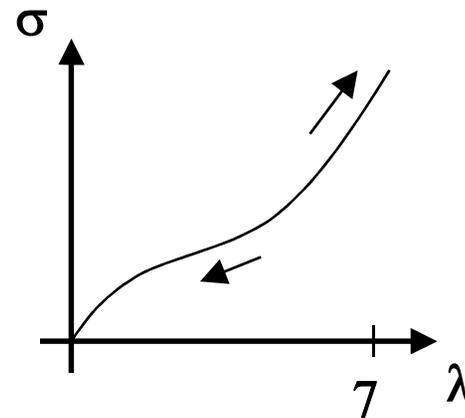
- **Nonlinear Elastic**

- small strains
- different curves for tension and compression
- after unloading structure is un-deformed



- **Hyperelastic**

- normally large strain
- Poisson's ratio close to 0.5
- mainly rubber



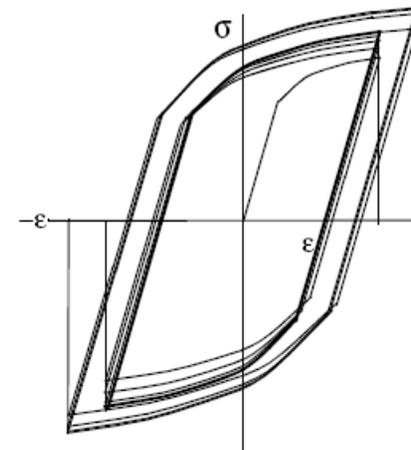
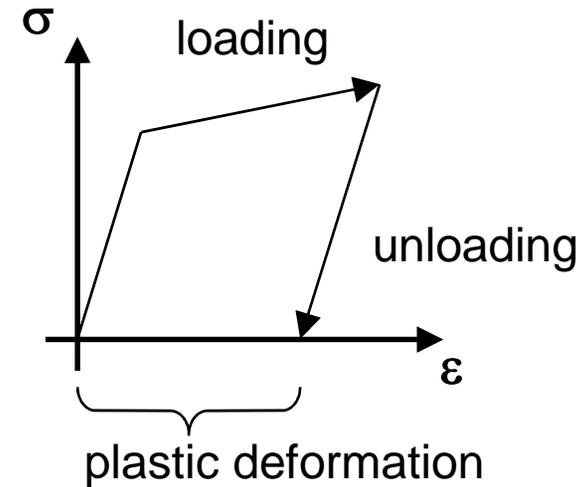
MATERIAL NONLINEARITY

- **Elastic-Plastic**

- small and large strain
- isotropic, anisotropic, pressure dependent
- initial stress and plastic strain

- **Cyclic Plasticity**

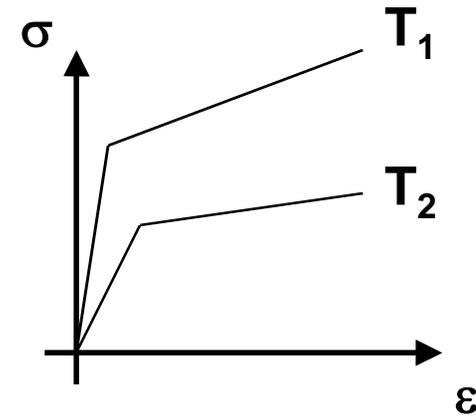
- yield stress changes with the number of cycles
- based on the work of Chaboche



MATERIAL NONLINEARITY

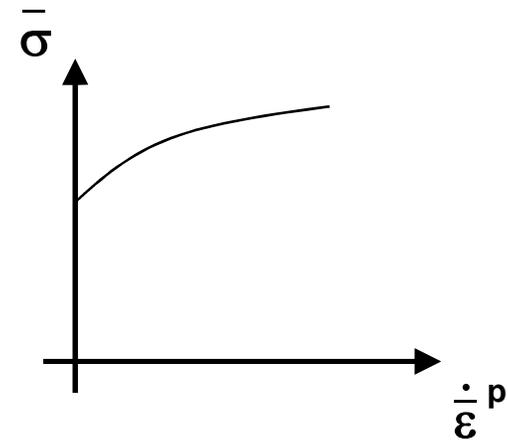
- **Temperature Dependent**

- stress–strain curve depends on temperature
- applies to each type of material nonlinearity



- **Rate Dependent**

- yield stress depends on equivalent plastic strain rate
- applies to elastic-plastic material



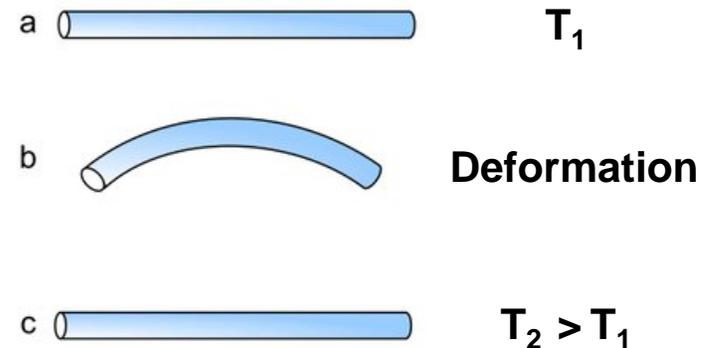
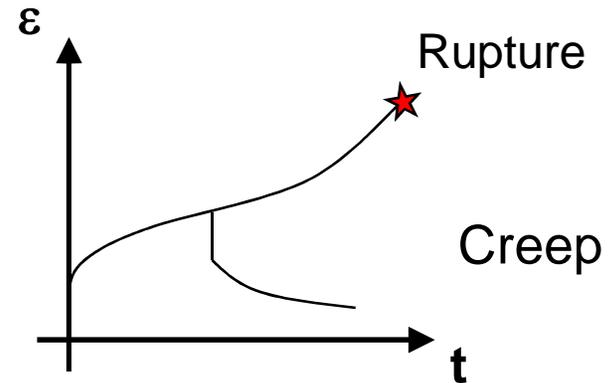
MATERIAL NONLINEARITY

- **Time Dependent**

- Material properties change with time
- Creep and relaxation
- Visco-Elasticity and Visco-Plasticity

- **Shape Memory**

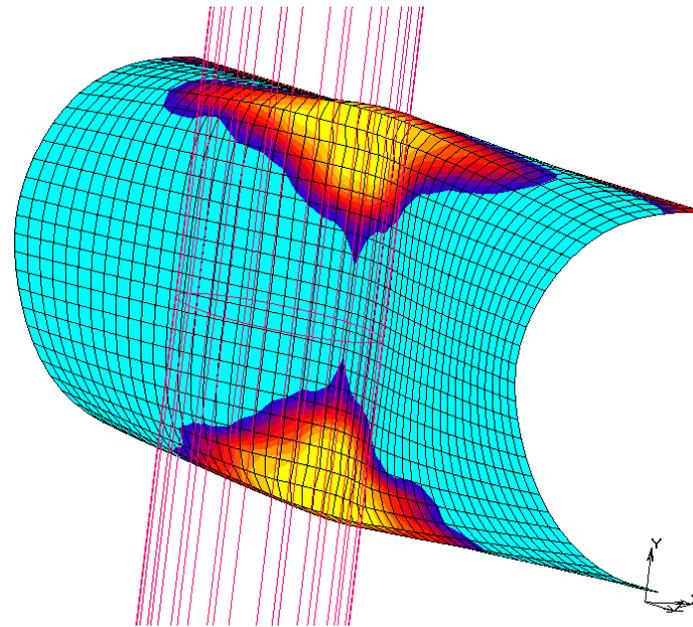
- Material properties depend on crystal structure (Martensite & Austenite)
- Phase changes due to temperature and stresses



MATERIAL NONLINEARITY

- **Progressive failure of composites is a nonlinear phenomenon**
- **Composite Failure Criteria**
 - Maximum Stress
 - Maximum Strain
 - Hill
 - Hoffman
 - Tsai-Wu
 - Hashin
 - Puck
 - Hashin-Tape
 - Hashin-Fabric
 - User defined (UFAIL)

Yellow: means outer ply, fully damaged



Rigid elliptical cylinder hitting composite shell

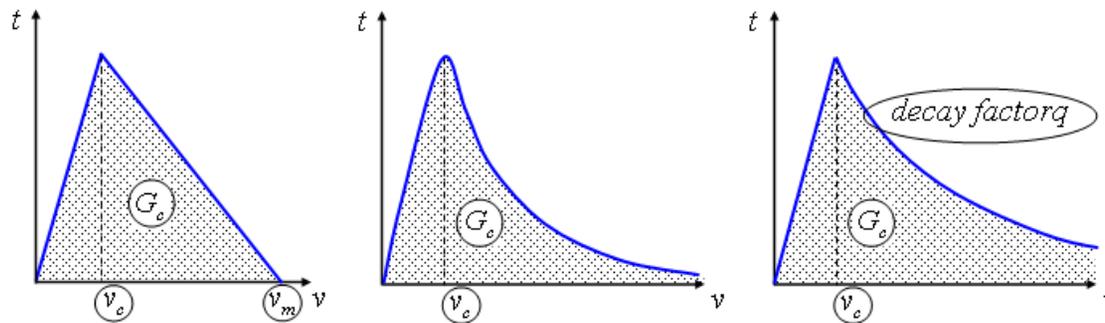
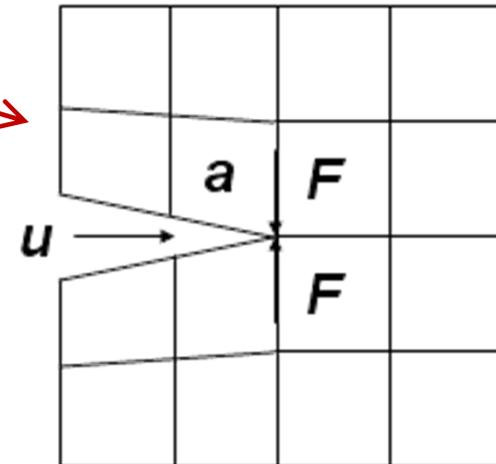
MATERIAL NONLINEARITY

- **Fracture**

- Application: Delamination in Composite Analysis

- Methods:

- Virtual Crack Closure Technique (VCCT)
- Cohesive Zone Modeling



CASE CONTROL SETUP BETWEEN LINEAR & NONLINEAR SOLUTION

- **The SUBCASE setup is different between SOL 101 and the old SOL 106 nonlinear solution**
 - SOL 101
 - Each subcase is a separate loading
 - SOL 106 (nonlinear static)
 - The 2nd subcase is a continuation of the 1st subcase and the 3rd subcase is the continuation of the 2nd subcase, etc.
 - SOL 129 (nonlinear transient)
 - The 2nd subcase is a continuation of the 1st subcase and the 3rd subcase is the continuation of the 2nd subcase, etc.
- **In SOL 400, the usage of subcase is similar to SOL 101 (by default)**
 - STEPs are used to simulate continuation of loading within a subcase
 - Both nonlinear static and nonlinear transient can be included in the same subcase by using the ANALYSIS Case Control command

SOL 400 INPUT

- **Nonlinear Iteration Strategy**
 - NLPARM - Nonlinear Parameters for Statics
 - TSTEPNL - Nonlinear Parameters for Transient
 - NLSTEP - New Nonlinear Parameters
- **Geometric Nonlinear Analysis**
 - param, lgdisp, 1
- **Material Nonlinear Analysis**
 - MATS1, for elastic-plastic Material

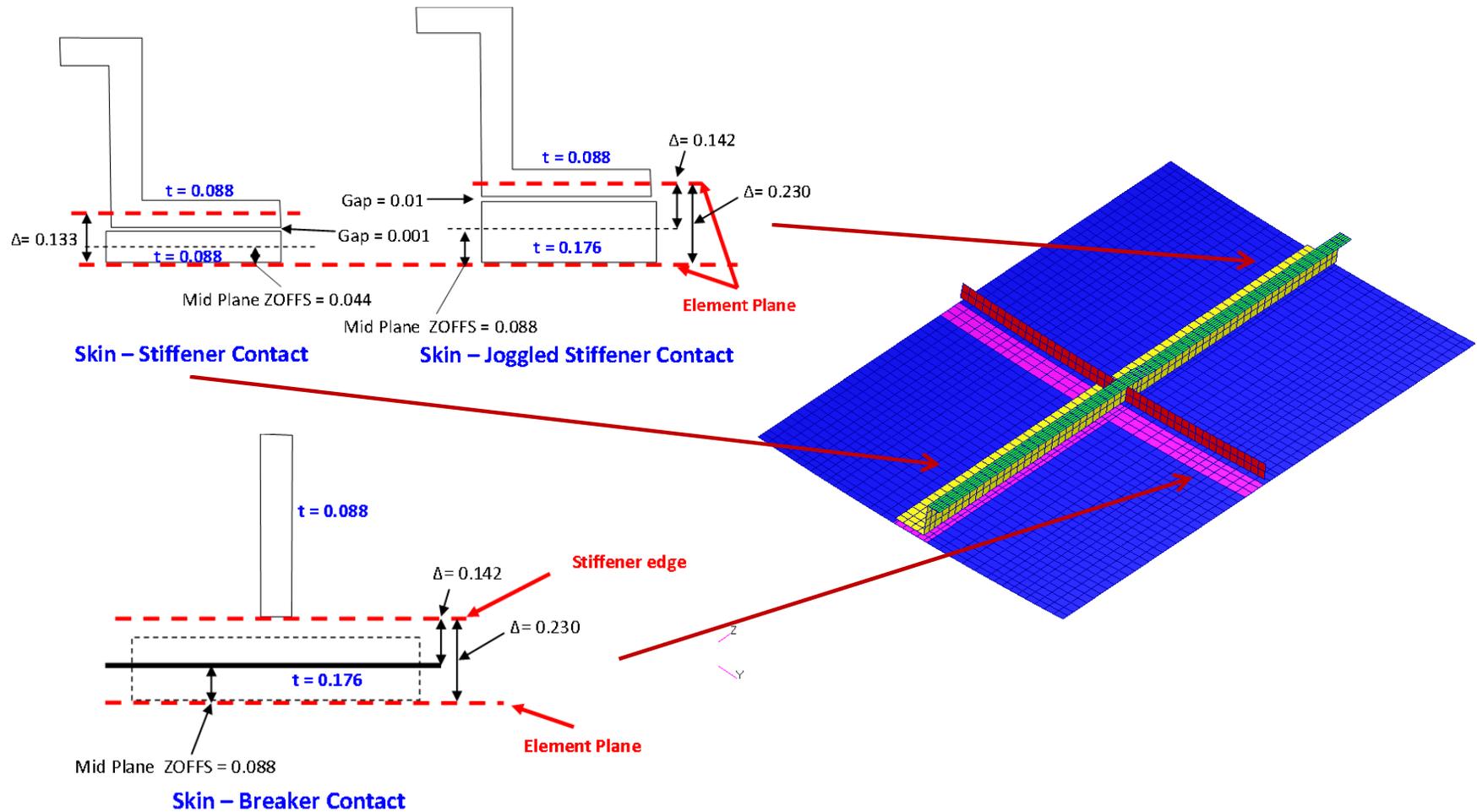
SOL 400 INPUT FILE EXAMPLE

```
SOL 400
DIAG 8
CEND
TITLE = THIS IS A DEMO INPUT EXAMPLE
SUBCASE 10
  STEP 1
    LOAD = 1
    NLPARM = 110
  STEP 2
    ANALYSIS = NLTRAN
    DLOAD = 3
    TSTEPNL = 130
BEGIN BULK
PARAM, LGDISP, 1
$. . . . . 2 . . . . . 3 . . . . . 4 . . . . . 5 . . . . . 6 . . . . . 7 . . . . . 8 . . . . . 9 . . . . . 0
NLPARM 110      25          ITER      1          15          P          NO
+          0.05        -3
MAT1      1      210000.    0.3      7.85-9  1.2-6
MATS1     1          PLASTIC 1000.    1          1          240.
.
```

OFFSET MODELING

- **Many times in typical aircraft model design, the following are located at different locations:**
 - Center line of skin
 - Neutral axis of stiffener
 - Location of the grids—typically at the Outer Mold Line (OML)
- **The above effects can be modeled with offset**
 - Offset available in SOL 400

TYPICAL OFFSET



ELEMENT OFFSETS FOR BEAMS AND SHELLS

- **MDLPRM,OFFDEF,type**

- **ELMOFF**

- Offset normal to shell element, rotates with the element (default)

- **NODIF**

- Same as LROFF but no differential stiffness (KDIFF) effects

- **NOTHRM**

- Same as LROFF but thermal load effects off

- **NODT**

- NODIF + NOTHRM

- **ELMZ**

- Same as LROFF but offset normal to shell element

- **NOMASS**

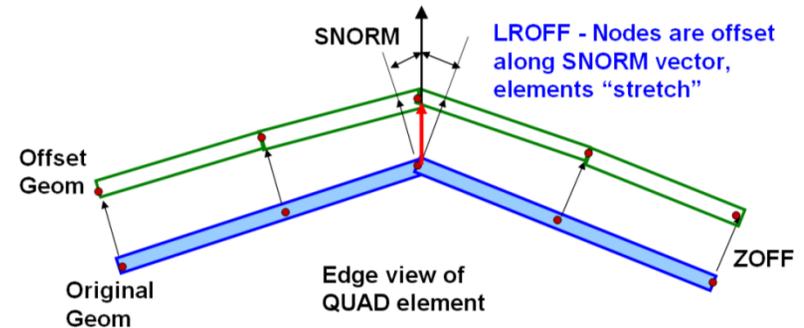
- LROFF but no mass effects

- **NDMTZ**

- ELMZ but no thermal or mass effects

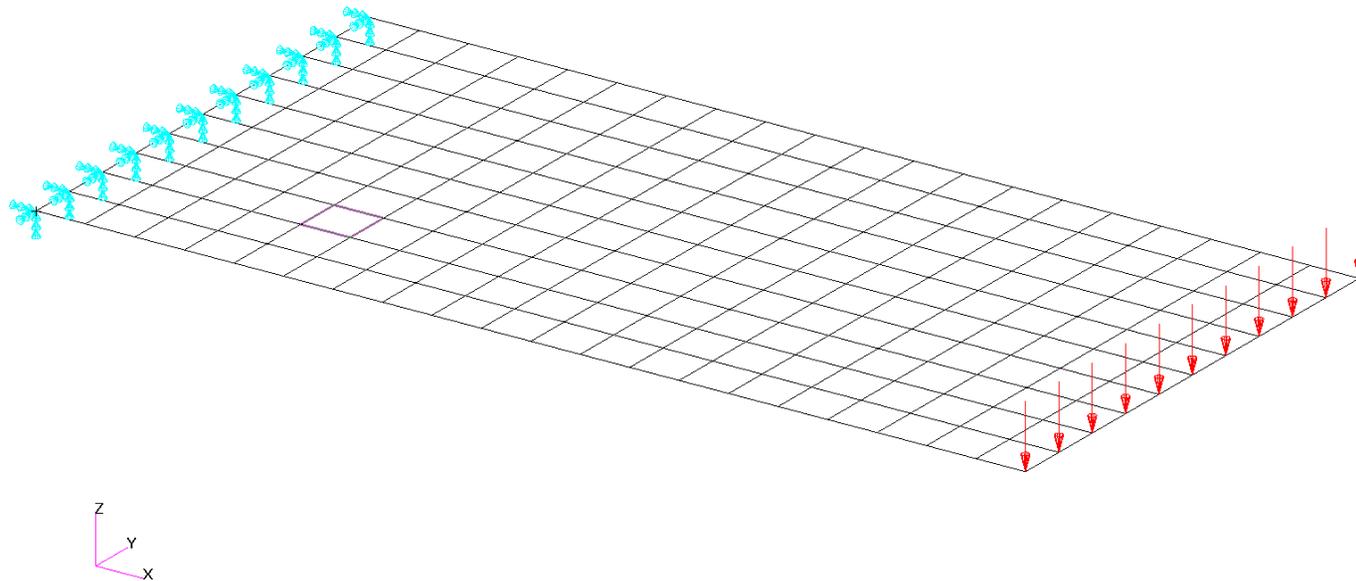
- **LROFF**

- Large rotation offsets based on normal at nodes (SNORM)
- SNORM vector for shells, thermal load effects, mass moment of inertia
- Not supported for solutions 106 and 129



EXAMPLE

- Cantilever plate, 20 in. x 10 in.
- The model is analyzed using:
 - SOL 101
 - SOL 400 with small displacement
 - SOL 400 with large displacement



SOL 101 INPUT

```
SOL 101
CEND
$
SPC = 2
DISPLACEMENT(PLOT)=ALL
SUBCASE 1
  TITLE = 16 pound load
  LOAD = 2
$
BEGIN BULK
$
PARAM      POST      0
PARAM      PRTMAXIM YES
$
PCOMP      1
           1      .0055  45.      YES      1      .0055  -45.      YES
           1      .0055   0.      YES      1      .0055   90.      YES
           1      .0055  45.      YES      1      .0055  -45.      YES
           1      .0055   0.      YES      1      .0055   90.      YES
$
MAT8      1      2.06+7  1.49+6   .27      1.04+6  1.04+6  1.04+6  .057
$ Pset: "shells" will be imported as: "pshell.1"
CQUAD4    1      1      1      2      23      22
.
.
.
```

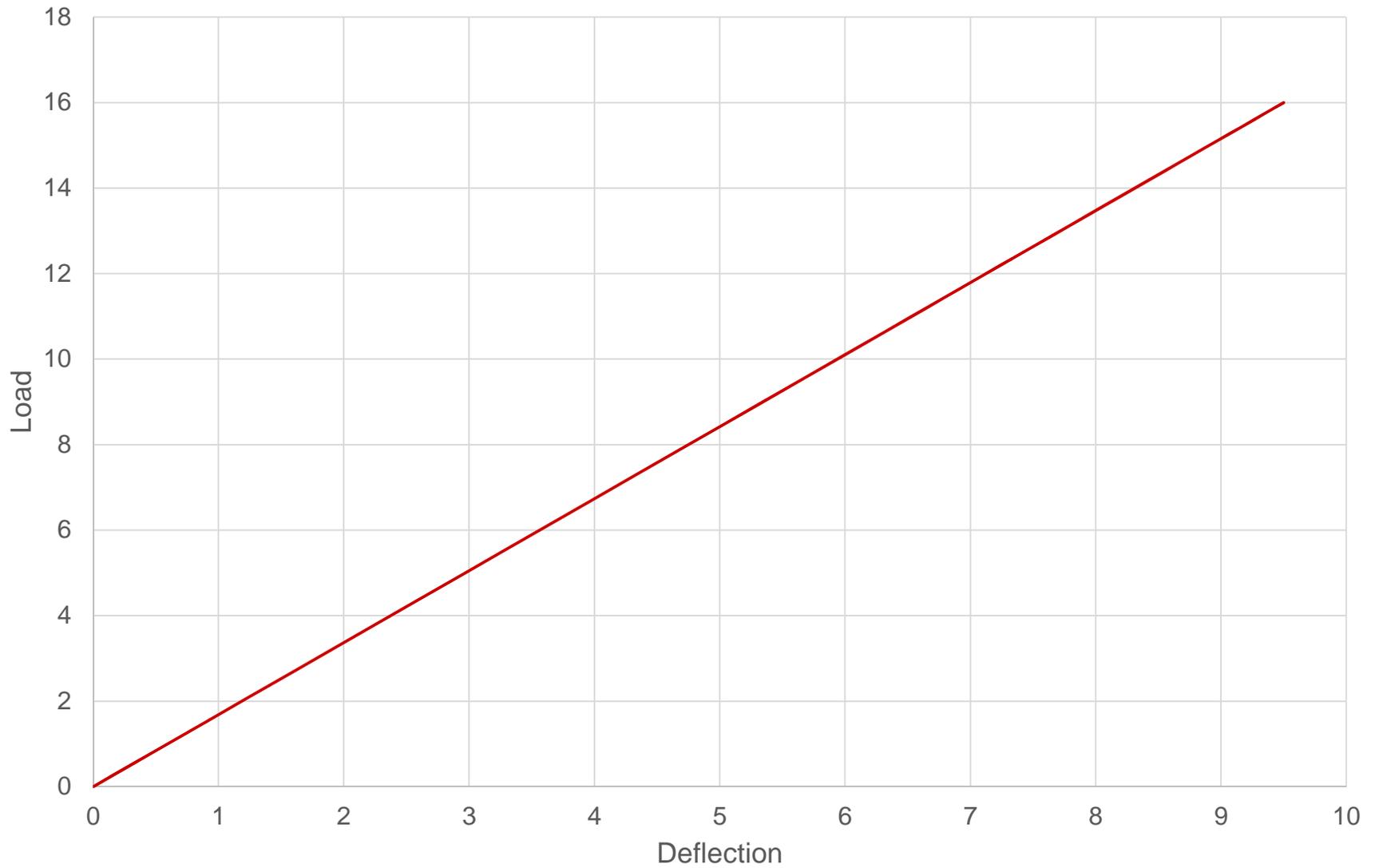
SOL 400 INPUT – SMALL DISPLACEMENT

```
SOL 400
CEND
SPC = 2
DISPLACEMENT(PLOT)=ALL
SUBCASE 1
  STEP 1
  TITLE = 16 pound load
  ANALYSIS = NLSTATIC
  LOAD = 5
  NLSTEP = 1
BEGIN BULK
$ Enable differential stiffness for offset elements
MDLPRM,OFFDEF,LROFF
PARAM   POST   0
PARAM   PRMAXIM YES
$
$ Disable large displacement
PARAM   LGDISP  -1
NLSTEP  1
      ADAPT   .05
NLMOPTS LRGSTRN 1
      ASSM   ASSUMED
$
PCOMP   1
      1      .0055  45.      YES   1      .0055  -45.   YES
      1      .0055   0.      YES   1      .0055   90.   YES
      1      .0055  45.      YES   1      .0055  -45.   YES
      1      .0055   0.      YES   1      .0055   90.   YES
MAT8    1      2.06+7  1.49+6  .27    1.04+6  1.04+6  1.04+6  .057
$ Pset: "shells" will be imported as: "pshell.1"
CQUAD4  201    1      232    233    254    253
.
.
```

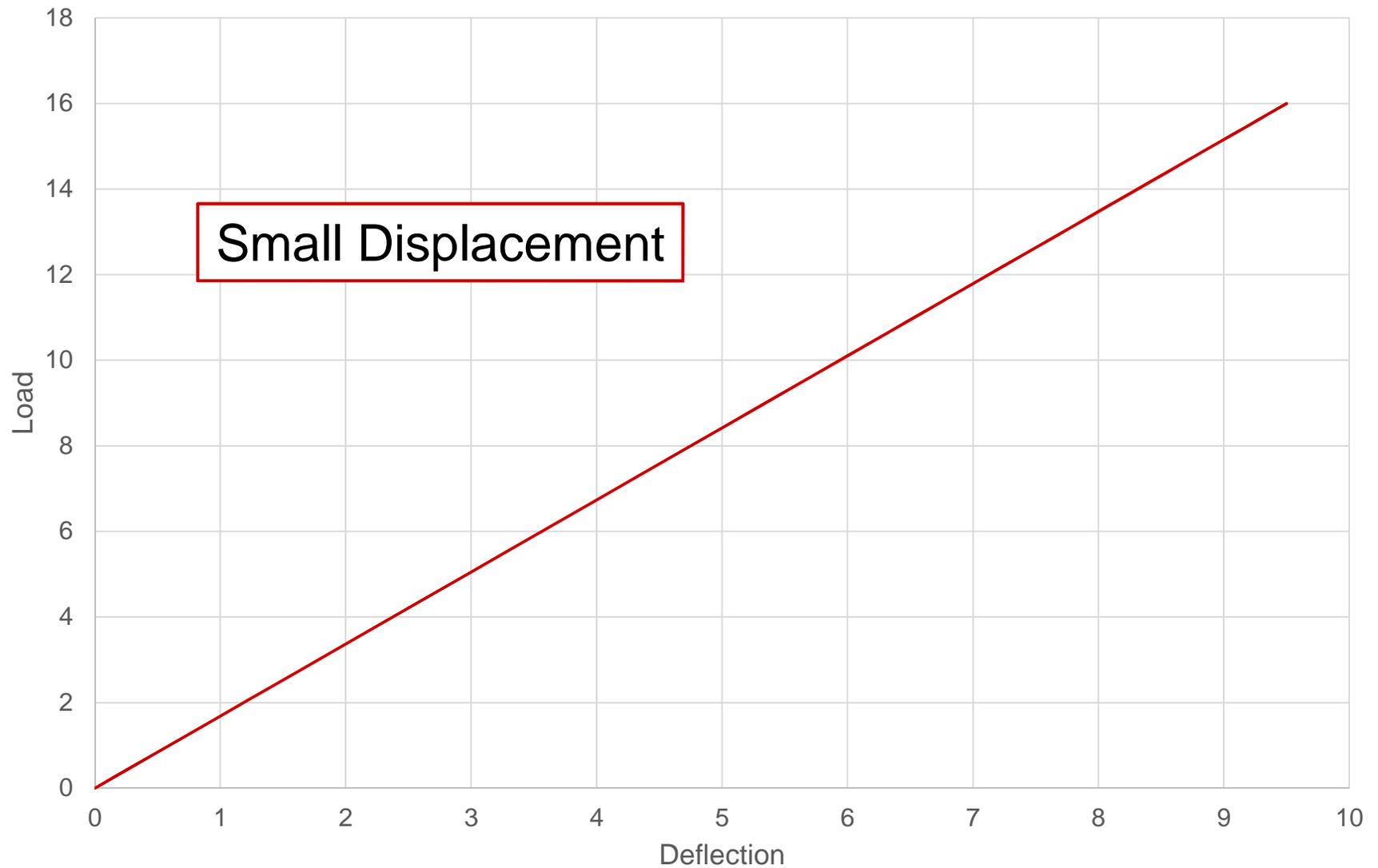
SOL 400 INPUT – LARGE DISPLACEMENT

```
SOL 400
CEND
SPC = 2
DISPLACEMENT(PLOT)=ALL
SUBCASE 1
  STEP 1
  TITLE = 16 pound load
  ANALYSIS = NLSTATIC
  LOAD = 5
  NLSTEP = 1
BEGIN BULK
$ Enable differential stiffness for offset elements
MDLPRM,OFFDEF,LROFF
PARAM POST 0
PARAM PRTMAXIM YES
$
$ Enable large displacement
PARAM LGDISP 1
NLSTEP 1
  ADAPT .05
NLMOPTS LRGSTRN 1
  ASSM ASSUMED
$
PCOMP 1 SYM
  1 .0055 45. YES 1 .0055 -45. YES
  1 .0055 0. YES 1 .0055 90. YES
  1 .0055 45. YES 1 .0055 -45. YES
  1 .0055 0. YES 1 .0055 90. YES
MAT8 1 2.06+7 1.49+6 .27 1.04+6 1.04+6 1.04+6 .057
$ Pset: "shells" will be imported as: "pshell.1"
CQUAD4 201 1 232 233 254 253
.
.
```

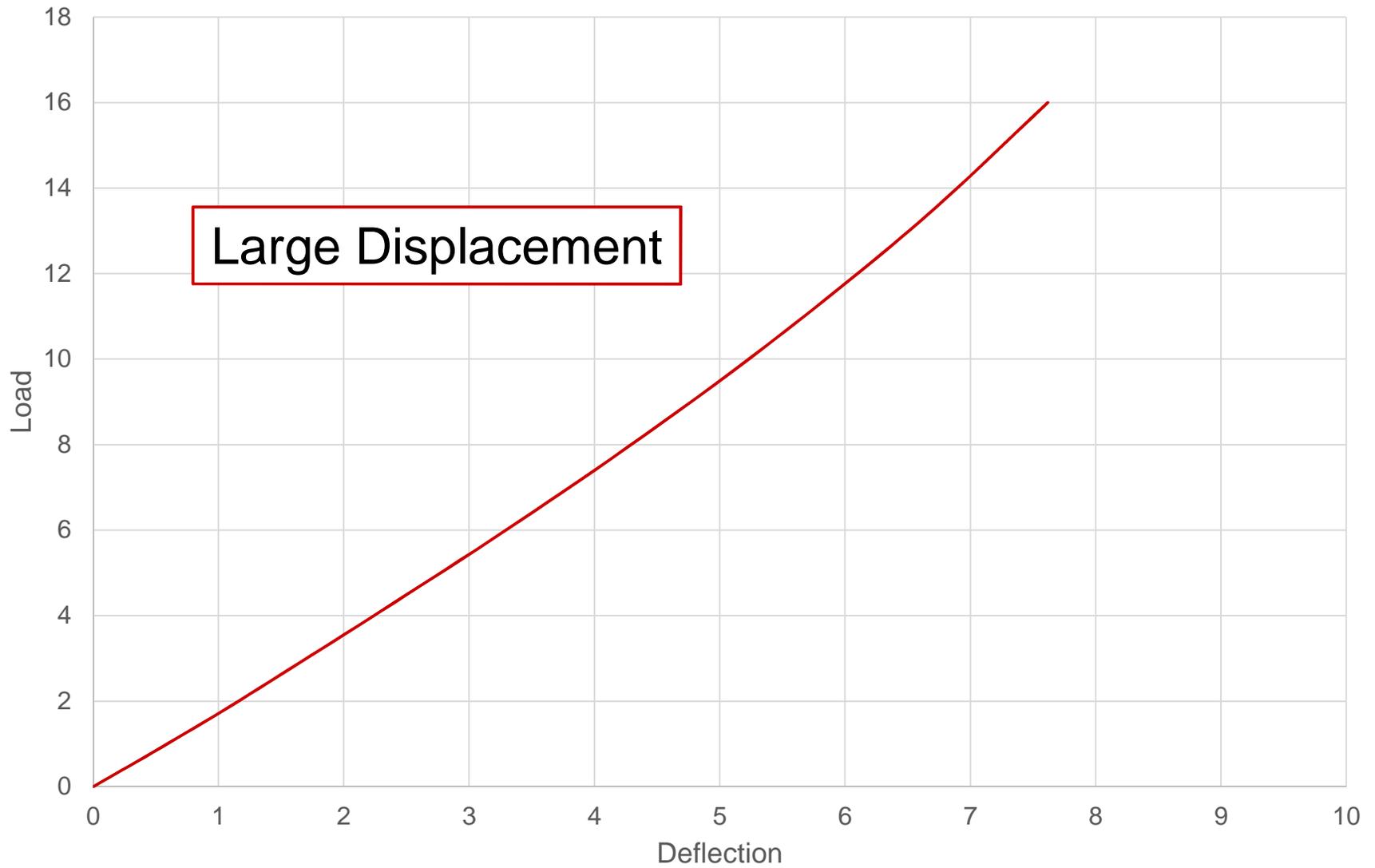
SOL 101 – LOAD-DISPLACEMENT CURVE



SOL 400 – LOAD-DISPLACEMENT CURVE



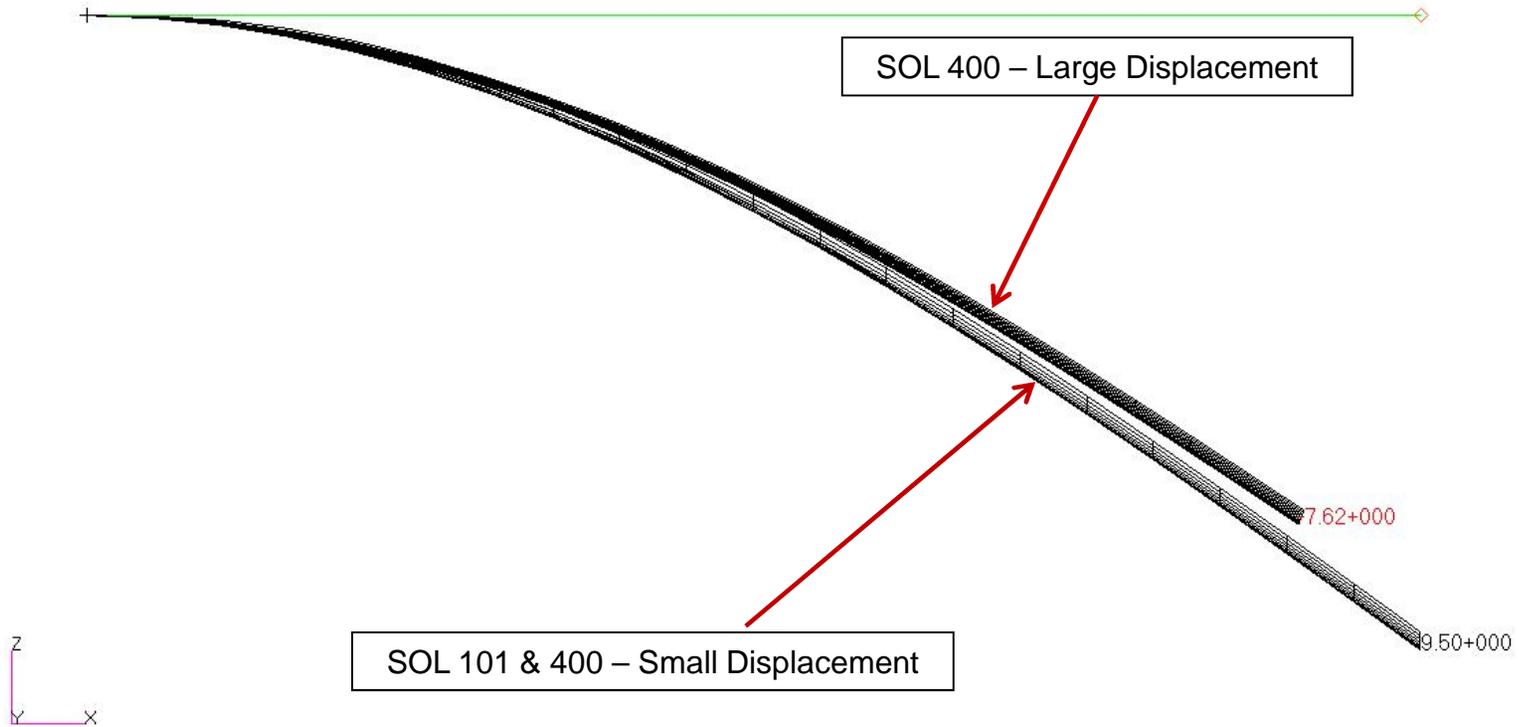
SOL 400 – LOAD-DISPLACEMENT CURVE



DEFORMATION COMPARISON

Patran 2013 64-Bit 07-Jul-14 10:25:37

Cursor: SC1.: A2:Non-linear: 100. % of Load, Displacements, Translational, Z Component, (NON-LAYERED)



default_Deformation :
Max 9.50+000 @Nd 42







SECTION 9

BUCKLING ANALYSIS

LINEAR BUCKLING (SOL 105)

- The equilibrium equations for a structure subjected to a constant force system take the following form:

$$[K] \{ u \} = \{ P \}$$

- There is a differential stiffness, $[K_D]$, which is a stiffness that results from including the higher-order terms of the strain-displacement relations. These relations are assumed to be independent of the displacements of the structure, associated with an arbitrary intensity of load.

LINEAR THEORY

- Let λ be an arbitrary scalar multiplier for another “intensity” of load.

$$([K] + \lambda[K_D])\{u^*\} = \{\lambda P\}$$

- By perturbing the structure slightly at a variety of load intensities, there are load intensities that can be found that possess unstable equilibrium positions. This leads to the associated eigenvalue problem for buckling.

$$([K] + \lambda[K_D])\{\delta u^*\} = 0$$

SOLUTION OF THE EIGENVALUE PROBLEM

$$[K + \lambda K_D] \{ \Phi \} = 0$$

- The solution is nontrivial (different from zero) only for specific values of $\lambda = \lambda_i$ for $i = 1, 2, 3, \dots, n$
- That makes the matrix $[K + \lambda K_D]$ singular
- To each eigenvalue λ_i , there is a corresponding distinct eigenvector $\{ \phi_i \}$.
- $\{ \phi_i \}$ can be scaled by any constant multiplier and still be a solution to Equation 1.
- The components of $\{ \phi_i \}$ are real numbers.

RULES FOR SOL 105 BUCKLING ANALYSIS

- **SOL 105 may be applicable for structures with slight material imperfections.**
- **Or, slightly non-centric loadings**
 - (i.e. load does not align with centroid, producing a small degree of bending).
- **Must use engineering judgment**
- **Generally, applicable to slender or thin elements under compressive loading**

DATA ENTRIES FOR LINEAR BUCKLING

- **Executive Control Section**

SOL 105

- **Case Control Section**

SUBCASE 1

LOAD = M

Defines static loading condition
(LOAD, TEMP, DEFORM)

SUBCASE 2

METHOD = N

Selects eigenvalue extraction method

STATSUB = i

Selects static subcase to use for buckling
solution (defaults to first subcase)

DATA ENTRIES FOR LINEAR BUCKLING (CONT.)

- **The Case Control must contain at least two subcases.**
- **Normally, the first subcase is the static solution under loading.**
- **METHOD must appear in a separate subcase to select an EIGB or EIGRL entry from the Bulk Data for the buckling solution.**
- **Output requests may be placed in any selected subcases.**

DATA ENTRIES FOR LINEAR BUCKLING (CONT.)

- **Bulk Data Section**
- **Static loading condition required for subcase 1**
- **For subcase 2, the METHOD case control command points to the:**
 - EIGB - Eigenvalue extraction (for buckling) data entry
 - or
 - EIGRL - Eigenvalue extraction data (vibration or buckling) with the Lanczos method

EIGRL ENTRY

- EIGRL Entry - recommended eigenvalue solution method
- Defines data needed to perform real eigenvalue or buckling analysis with the Lanczos Method.

1	2	3	4	5	6	7	8	9	10
EIGRL	SID	V1	V2	ND	MSGLVL	MAXSET	SHFSCCL	NORM	
EIGRL	5			1					

Field

Contents

SID

Set identification number (unique integer > 0)

V1, V2

Vibration analysis: Frequency range of interest

Buckling analysis: I range of interest ($V1 < V2$, real). If all modes below a frequency are desired, set V2 to the desired frequency and leave V1 blank. It is not recommended to put 0.0 for V1, it is more efficient to use a small negative number or to leave it blank.

ND

Number of roots desired (integer > 0 or blank)

MSGLVL

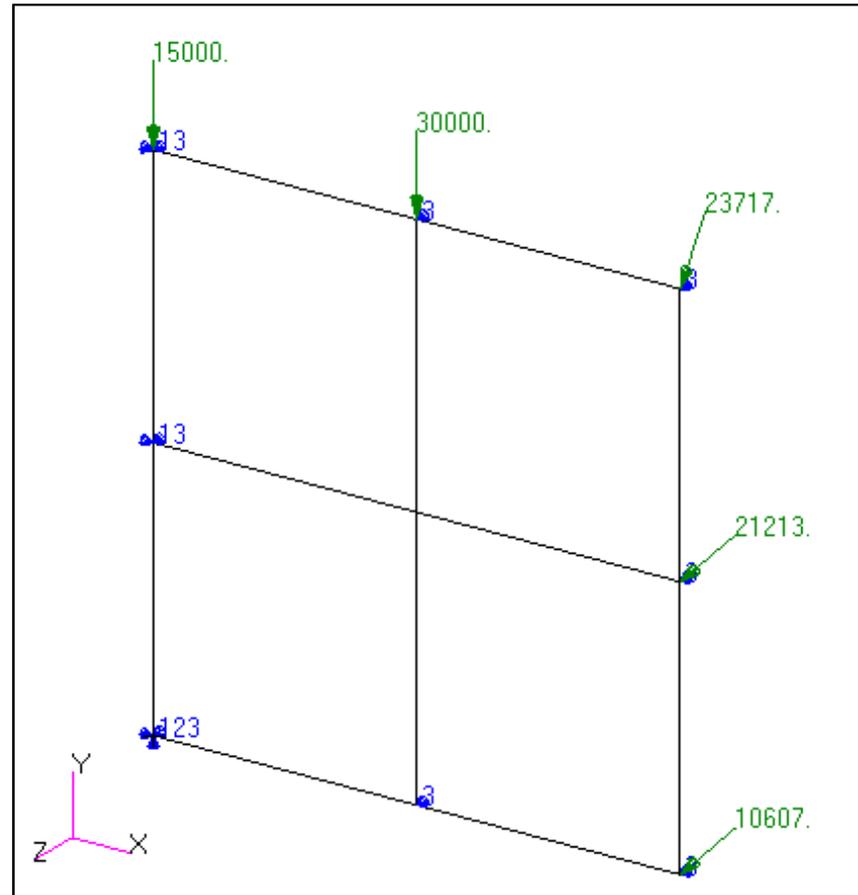
Diagnostic level (integer 0 through 3 or blank)

MAXSET

Number of vectors in block (integer 1 through 15 or blank)

COMPOSITE PLATE LINEAR BUCKLING EXAMPLE

- This corresponds to Prob9.dat
- The question is whether the lay-up optimized for strength model (prob4.dat) will buckle first.
- The model is changed to be 30'' by 30'' (30x as large).
- It is fixed in the Z direction all around the edges (pinned).
- Loads are also increased by 30x in the LOAD bulk data entry.
- Finally, appropriate eigenvalue solution entries are made to the file.



COMPOSITE PLATE LINEAR BUCKLING EXAMPLE (CONT.)

```
nastran prtpcomp=1
init master(s)
SOL 105
CEND
TITLE = WS 09 - Linear Buckling
  SPC = 1
  DISP = ALL
subcase 1
  STRESS =ALL
  load = 2
subcase 2
  statsub=1
  method=10
$
BEGIN BULK
PARAM, POST, -1
$
eigr1,10,,,1
$
PCOMP, 1,,, 5000., HILL,,,sym
, 1, .0054, 0., YES
, 1, .0054, 0., YES
, 1, .0054, 45., YES
, 1, .0054, -45., YES
, 1, .0054, 90., YES
, 1, .0054, 90., YES
, 1, .0054, 90., YES
MAT8, 1, 2.+7, 2.+6, .35, 1.+6, 1.+6, 1.+6
,,,,,130000., 120000., 11000., 12000., 12500.
```

```
CQUAD4, 1, 1, 1, 2, 5, 4, 99
CQUAD4, 2, 1, 2, 3, 6, 5, 99
CQUAD4, 3, 1, 4, 5, 8, 7, 99
CQUAD4, 4, 1, 5, 6, 9, 8, 99
GRID, 1,, 0., 0., 0.
GRID, 2,, 0., 15., 0.
GRID, 3,, 0., 30., 0.
GRID, 4,, 15., 0., 0.
GRID, 5,, 15., 15., 0.
GRID, 6,, 15., 30., 0.
GRID, 7,, 30., 0., 0.
GRID, 8,, 30., 15., 0.
GRID, 9,, 30., 30., 0.
$
SPC1,1, 1, 1,2,3
SPC1,1, 2, 1
spc1,1, 3, 1,2,3,4,6,7,8,9
$
load,2,-1.0,30.0,1
$
FORCE, 1, 3,, 500., 0., 1., 0.
FORCE, 1, 6,, 500., 0., 1., 0.
FORCE, 1, 6,, 500., 0., 1., 0.
FORCE, 1, 9,, 500., 0., 1., 0.
FORCE, 1, 7,, 250., 1., 0., 0.
FORCE, 1, 8,, 250., 1., 0., 0.
FORCE, 1, 8,, 250., 1., 0., 0.
FORCE, 1, 9,, 250., 1., 0., 0.
FORCE, 1, 7,, 250., 0., 1., 0.
FORCE, 1, 8,, 250., 0., 1., 0.
FORCE, 1, 8,, 250., 0., 1., 0.
FORCE, 1, 9,, 250., 0., 1., 0.
$
CORD2R, 99,, 0., 0., 0., 0., 0., 1.
, 0., 1., 0.
ENDDATA
```

.dat file

COMPOSITE PLATE LINEAR BUCKLING EXAMPLE (CONT.)

- The linear buckling load is the eigenvalue times the load applied in subcase 1.
- The panel will fail in buckling at 0.0218 times the initially solved (prob 4) strength failure load.

0

SUBCASE 2

E I G E N V A L U E A N A L Y S I S S U M M A R Y (R E A D M O D U L E)

.f06 file extract

BLOCK SIZE USED 2

NUMBER OF DECOMPOSITIONS 3

NUMBER OF ROOTS FOUND 1

NUMBER OF SOLVES REQUIRED 13

0

SUBCASE 2

MODE NO.	EXTRACTION ORDER	EIGENVALUE	REAL RADIANS	EIGENVALUES CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	2.183382E-02	1.477627E-01	2.351717E-02	5.573152E+03	1.216832E+02

0

EIGENVALUE = 2.183382E-02

SUBCASE 2

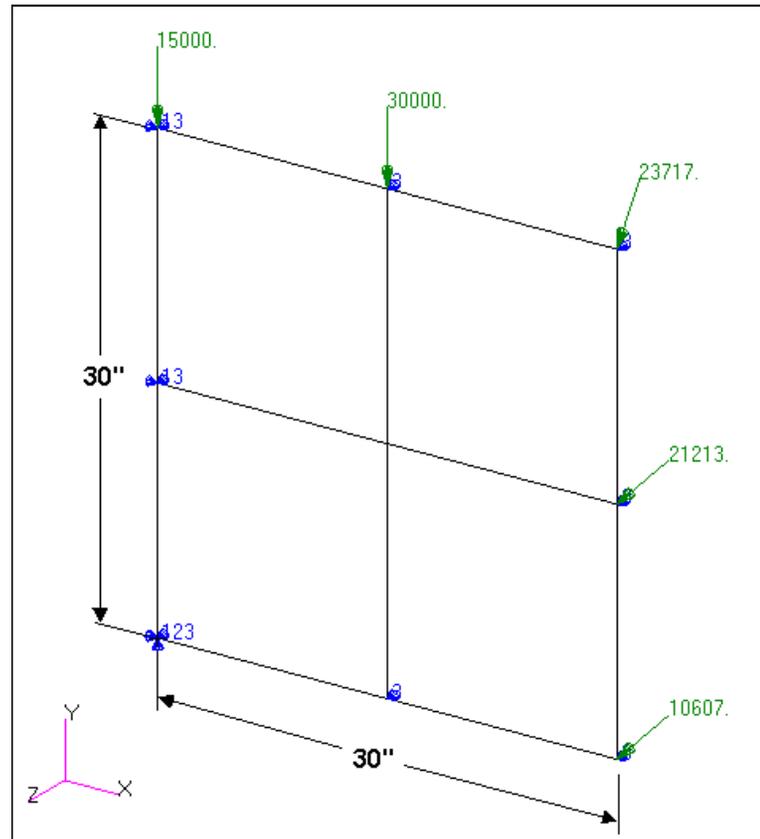
R E A L E I G E N V E C T O R N O . 1

POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	0.0	0.0	-6.876530E-03	-2.357480E-02	2.101358E-19
2	G	0.0	-1.271667E-19	0.0	-1.233789E-02	-1.075896E-01	-1.047022E-18
3	G	0.0	-8.296756E-19	0.0	2.975350E-02	6.746294E-02	-2.500754E-19
4	G	1.290573E-18	2.016648E-18	0.0	1.127643E-01	4.758961E-02	1.274480E-20
5	G	1.217512E-18	1.538059E-18	1.000000E+00	2.400048E-16	-2.899484E-16	-1.645825E-19
6	G	-2.184760E-18	1.119534E-18	0.0	-1.127643E-01	-4.758961E-02	-3.042073E-19
7	G	2.464627E-18	4.698268E-18	0.0	-2.975350E-02	-6.746294E-02	4.723079E-19
8	G	-6.141753E-19	4.684834E-18	0.0	1.233789E-02	1.075896E-01	-3.517192E-19
9	G	-5.646527E-19	2.912704E-18	0.0	6.876530E-03	2.357480E-02	1.092971E-18

Buckled shape

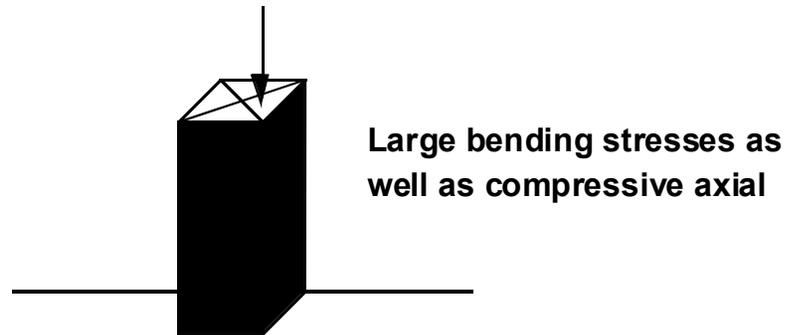
EXERCISE

- Perform Workshop 9 “Linear Buckling” in your workshop booklet.

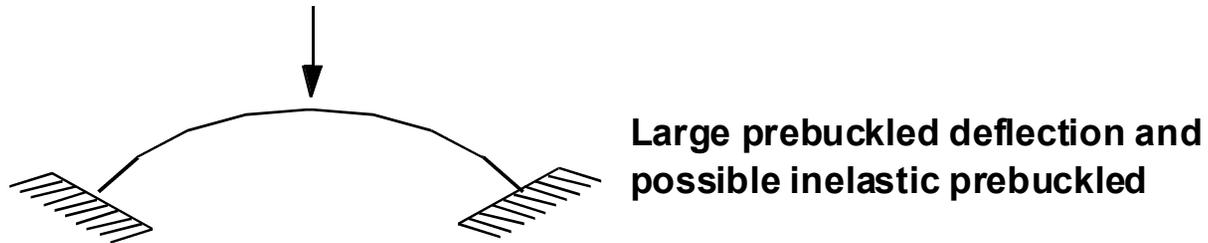


NONLINEAR BUCKLING

- **Highly Eccentrically Loaded Column**



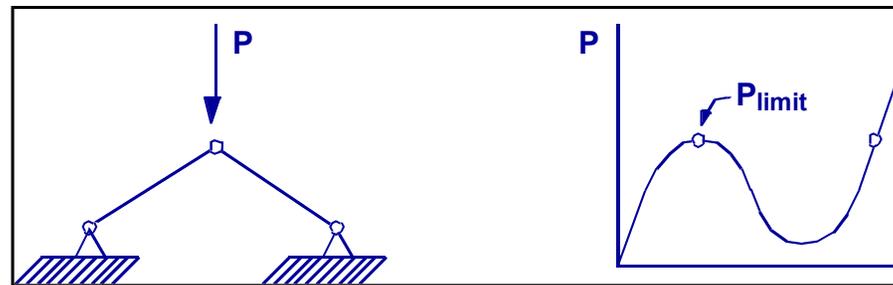
- **Snap-Through of Thin Shell (like the Bottom of an Oil Can)**



INSTABILITY PHENOMENA

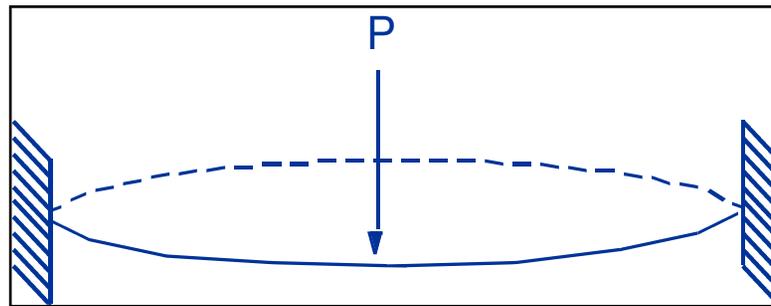
- **Type 1 is snap-through**

- The loss of stability occurs at a stationary point (relative maximum) in the load-deflection space.
- The critical load is termed a limit point. For loads beyond the limit point, the structure “snaps-through” and assumes a completely different displaced configuration.



INSTABILITY PHENOMENA (CONT.)

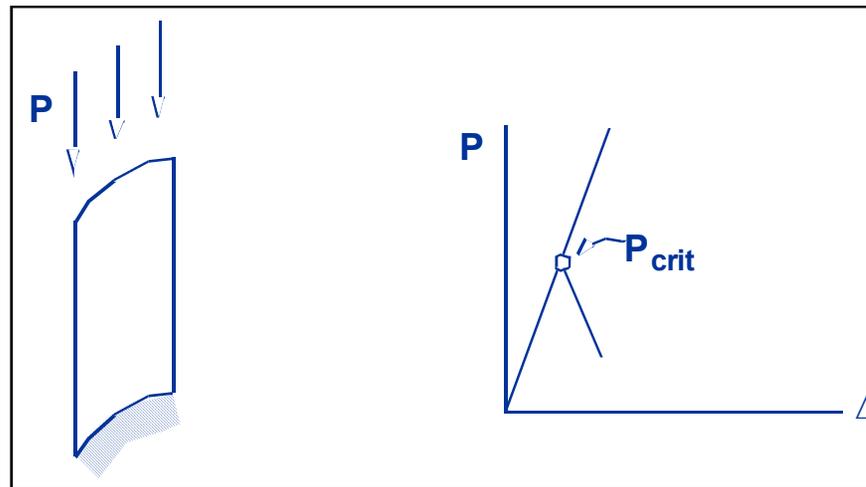
- Arc-length increments are good for snap-through problems.



Snap-Through of Shallow Shells

INSTABILITY PHENOMENA (CONT.)

- **Type 2 is bifurcation buckling:**
 - The loss of stability occurs when two or more equilibrium paths intersect in the load-deflection space.
 - The point of intersection is termed a bifurcation point.
 - For loads beyond the bifurcation point, the structure buckles.



- **Arc length increments may not pick a bifurcation buckling point.**

LINEAR VERSUS NONLINEAR

- **Linear Buckling**

$$[K + \lambda K_D]\{\phi\} = 0$$

- Kinematic relationship is linear
- Constitutive relationship is linear
- Equilibrium is satisfied in perturbed configuration
- Geometric stiffness is assumed proportional to the load
- Use SOL 105

- **Nonlinear Buckling**

with $[K_n + \lambda \Delta K]\{\phi\} = \{0\}$

$$\underbrace{\Delta K}_{\substack{\uparrow \\ \text{Incremental Stiffness}}} = K_n - K_{n-1} \quad \leftarrow \text{Actual Tangent Nonlinear}$$

LINEAR VERSUS NONLINEAR (CONT.)

- **Kinematic relationship is nonlinear.**
- **Constitutive relationship may be nonlinear.**
- **Geometric stiffness is assumed proportional to displacement increment.**
- **Equilibrium is satisfied in perturbed configuration.**
- **Older approach**
 - Use SOL 106 followed by evaluation using PARAM, BUCKLE
 - Buckling evaluation still is linear
 - SOL 106 is less robust in handling convergence for very large strain problems (>5%)
- **Updated approach**
 - Use SOL 400 to capture structural response after initial panel buckling
 - More robust for large structural displacements

DATA ENTRIES FOR NONLINEAR ANALYSIS

- **Executive Control Section**

SOL 400

- **Case Control Section**

NLSTEP=n

Reference to nonlinear parameters

- **Bulk Data Section**

NLSTEP, n

PARAM,LGDISP,1

MDLPRM,OFFDEF,LROFF

Nonlinear time step options data entry

Make elements nonlinear

Enable Large Rotation Offsets; Required for offset beams/plates to get correct differential stiffness (not supported in older NL Nastran)

NLSTEP ENTRY

- **Select nonlinear options**

1	2	3	4	5	6	7	8	9	10
NLSTEP	ID	TOTTIME							
	“GENERAL”	MAXITER	MINITER	MAXBIS	CREEP				

<u>Field</u>	<u>Contents</u>
ID	NLSTEP case control identification number (unique integer > 0)
TOTTIME	Total amount of time for the load case (Real, default=1.0)
“GENERAL”	Keyword for general analysis parameters
MAXITER	Max number of iterations per increment (Integer, default=10)
MINITER	Min number of iterations per increment (Integer, default=1)
MAXBIS	Maximum number of bisections for the step (Integer, default=10)
CREEP	Use creep for the current load case? (Integer, 1=yes, 0=no)

(continued...)

NLSTEP: KEYWORDS

1	2	3	4	5	6	7	8	9	10
	"MECH"	CONV	EPSU	EPSP	EPSW	KMETHOD	KSTEP	MRCONV	
		MAXQN	MAXLS	LSTOL	FSTRESS				

Field

Contents

"MECH"

Keyword for mechanical analysis parameters

CONV

Flags to select convergence criteria (U,P,W,V,N,A or any combination; default=PV)

U = displacement, P = load, W = work

V = vector component method, N = length method, A = auto switch

EPSU

Error tolerance for displacement criterion, U (Real, default=0.1)

EPSP

Error tolerance for load criterion, P (Real, default=0.1)

EPSW

Error tolerance for work criterion, W (Real, default=0.1)

KMETHOD

Method for controlling stiffness updates, PFNT or ITER (default=PFNT)

KSTEP

Number of iterations before stiffness update for KMETHOD=ITER (Integer, default=10)

MRCONV

Flag: should rotations/moments be included in convergence test for CONV='UPV' or 'UPN' (Integer, default=3)

(continued...)

NLSTEP: FIXED STEPPING

1	2	3	4	5	6	7	8	9	10
	"FIXED"	NINC	NO						

- **Method 1: FIXED (choice of FIXED, ARCLN, ADAPT)**

<u>Field</u>	<u>Contents</u>
"FIXED"	Keyword to flag fixed stepping
NINC	Number of increments for fixed stepping (Integer>0, default=50)
NO	Interval for output, every NO th increment saved for output (Integer ≥ 0, default = 1)

(continued...)

NLSTEP: ARC LENGTH LOAD STEPPING

1	2	3	4	5	6	7	8	9	10
	"ARCLN"	TYPE	DTINITFA	MINALR	MAXALR	SCALEA	NDESIRA	NSMAXA	

- Method 2: ARCLN (choice of FIXED, ARCLN, ADAPT)**

<u>Field</u>	<u>Contents</u>
"ARCLN"	Keyword to flag arc length load stepping
TYPE	Constraint type (CRIS, RIKS, MRIKS, default = CRIS)
DTINITFA	Initial time step as a fraction of TOTTIME (Real, default=0.1)
MINALR	Minimum arc length adjustment ratio between increments (0. < Real < 1., default=.25)
MAXALR	Maximum arc length adjustment ratio between increments (1. < Real, default=4.)
SCALEA	Scale factor for controlling the loading contribution in the arc length constraint (Real > 0., default = 0.)
NDESIRA	Desired number of iterations for convergence (Int > 0, default = 4)
NSMAXA	Max num increments in current case (Int, default=1000)

(continued...)

NLSTEP: ADAPTIVE STEPPING

1	2	3	4	5	6	7	8	9	10
	"ADAPT"	DTINITF	DTMINF	DTMAXF	NDESIR	SFACT	INTOUT	NSMAX	
		IDAMP	DAMP	CRITID	IPHYS	LIMTAR	RSMALL	RBIG	
		ADJUST	MSTEP	RB	UTOL				

- **Method 3: ADAPT (choice of FIXED, ARCLN, ADAPT)**

<u>Field</u>	<u>Contents</u>
"ADAPT"	Keyword to flag adaptive load stepping
DTINITF	Initial time step as a fraction of TOTTIME (Real, default=0.01)
DTMINF	Minimum time step as a fraction of TOTTIME (Real, default=1e-5)
DTMAXF	Maximum time step as a fraction of TOTTIME (Real, default=0.5)
NDESIR	Number of desired iterations per increment (Integer, default=4)
SFACT	Factor for increasing time step (Real, default=1.2)
INTOUT	Output flag (Integer ≥ -1 , default = 0) -1 = last increment only, 0 = every inc output, $n > 0$ = every n^{th} inc
NSMAX	Maximum number of increments in load case (Integer, default=99999)
IDAMP	Flag for activating artificial damping (Integer, default=6) 0 = no damping 4 = time step control using damping 5 = damping based time step control, no damping added 6 = add damping when minimum time step is reached

(continued...)

NLSTEP: ADAPTIVE STEPPING (CONT.)

- Additional 'ADAPT' fields...

<u>Field</u>	<u>Contents</u>
DAMP	Damping ratio (Real, default = 2.e-4)
CRITTID	ID of bulk data TABSCTL entry defining user criteria (Integer, default=0)
IPHYS	Flag for addition of automatic physical criteria (Integer, default=2) 2 = Do not add criteria, stop if user criteria not satisfied -2 = Do not add criteria, continue if user criteria not satisfied 1 = Add criteria, stop if user criteria not satisfied -1 = Add criteria, continue if user criteria not satisfied
LIMTAR	0 = treat user criteria from CRITTID as limits, 1 = treat user criteria as targets (integer, default = 0)
RSMALL	Smallest ratio between time step changes due to user criteria (Real, dflt=.1)
RBIG	Largest ratio between time step changes due to user criteria (Real, dflt=10.)
ADJUST	Time step skip factor for auto time step adjust (dynamics, Integer, default=0)
MSTEP	Number of steps to obtain dominant period response (dynamics, $10 \leq \text{Integer} \leq 200$, default = 10)
RB	Bounds for maintaining the same step for stepping fcn ($.1 \leq \text{Real} \leq 1$, dflt=.6)
UTOL	Tolerance on displacement ($.0001 \leq \text{Real} \leq 1.$, default=1.)

NLSTEP: EXAMPLE

NLSTEP	1								
	GENERAL	25							
	MECH	PV				PFNT			
	FIXED	40	1						

- **Sample for mechanical stepping**
 - Step ID 1, corresponding with callout in subcase (NLSTEP=)
 - Maximum iterations per increment = 25
 - Mechanical loading, check convergence using load with vector comparison (PV)
 - Pure Full Newton stiffness matrix update
 - Fixed load stepping with 40 even increments, all increment results written to output

BUCKLING AND ELEMENT OFFSETS

- **Prior to MSC Nastran 2010, beam and plate element offsets have the following limitations:**
 - Differential stiffness not supported, so they could not be used in solutions dependent on inclusion of that attribute (e.g. SOL 105)
 - Offset effect not included in the mass matrix
 - Offset effect not included in computation of thermal, pressure, or gravity loads
 - For curved shell problems, offset was defined in element Z direction, not shell normal direction, resulting in element gaps
 - The transformation was linear and could not be used in nonlinear analysis (SOL 106, SOL 400)

BUCKLING AND ELEMENT OFFSETS (CONT.)

- **These issues are addressed with the addition of the OFFDEF model parameter**
 - Differential stiffness: Offsets are now applicable in SOLs 103, 105, 400
 - Mass: Offsets included in mass matrix generation
 - Loads: Offset effect included in load generation
 - Geometry compatibility: Offsets are in the element normal direction, and no gaps will occur as long as adjacent element angle is less than SNORM
 - Nonlinear effects: Transformations can be used in nonlinear analysis (SOL 400)

BUCKLING AND ELEMENT OFFSETS

- “MDLPRM, OFFDEF, LROFF” in bulk data enables this support
- Offsets must be applied via the ZOFFS parameter on plate elements

1	2	3	4	5	6	7	8	9	10
CQUAD4	EID	PID	G1	G2	G3	G4	THETA or MCID	ZOFFS	

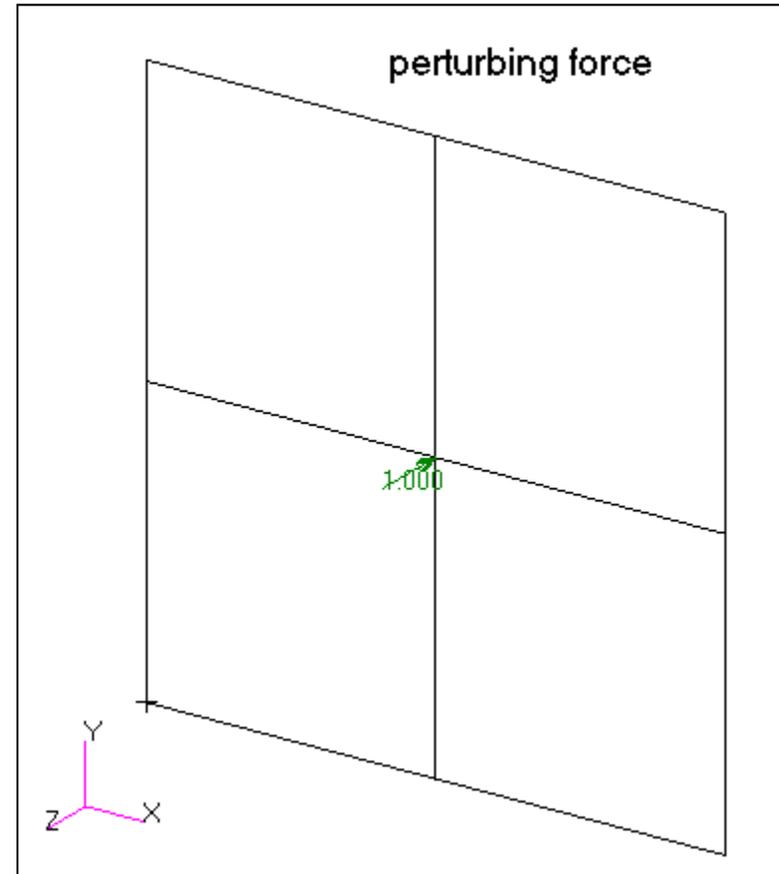
- The Z0 value on PCOMP entries **should *NOT*** be used to defined this offset– does not result in correct inclusion

1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SB	FT	TREF	GE	LAM	

- Detailed documentation is in the MSC Nastran 2010 Release Guide, and any subsequent MSC Nastran Quick Reference Guide.

POST BUCKLED STRENGTH

- If a model is perfectly flat, you must perturb it out of plane to trigger buckling.
- Real structures always have small imperfections in structure or load that will initiate buckling.
- For an older version of Prob 9 (formerly 8b), we increased a load factor to 5 to reach a post-buckled loading regime.



POST BUCKLED STRENGTH (CONT.)

```

SOL 400
CEND
TITLE = Chapter 6, Section 3 - Nonlinear Buckling -
transverse load
  SPC = 1
  DISP = ALL
  STRESS =ALL
subcase 1
  load = 2
  nlstep=20
$
BEGIN BULK
PARAM, POST, -1
param,lgdisp,1
$
nlstep,20
,general,25,
,mech, PV,,,,PFNT,
,fixed,20,1,
$
PCOMP, 1,,,, 5000., HILL,,,,sym
, 1, .0054, 0., YES
, 1, .0054, 0., YES
, 1, .0054, 45., YES
MAT8, 1, 2.+7, 2.+6, .35, 1.+6, 1.+6, 1.+6
,,,,130000., 120000., 11000., 12000., 12500.
$
CQUAD4 1 1 1 2 5 4 99
CQUAD4 2 1 2 3 6 5 99

```

.dat file

```

GRID, 1,, 0., 0., 0.
GRID, 2,, 0., 15., 0.
GRID, 3,, 0., 30., 0.
GRID, 4,, 15., 0., 0.
GRID, 5,, 15., 15., 0.
GRID, 6,, 15., 30., 0.
GRID, 7,, 30., 0., 0.
GRID, 8,, 30., 15., 0.
GRID, 9,, 30., 30., 0.
$
SPC1,1,123,1
SPC1,1,13,2,3
spc1,1,3,4,6,7,8,9
$
load,2,-1.0,5.,1,1.,3
FORCE, 1, 3,, 500., 0., 1., 0.
FORCE, 1, 6,, 500., 0., 1., 0.
FORCE, 1, 6,, 500., 0., 1., 0.
FORCE, 1, 9,, 500., 0., 1., 0.
FORCE, 1, 7,, 250., 1., 0., 0.
FORCE, 1, 8,, 250., 1., 0., 0.
FORCE, 1, 8,, 250., 1., 0., 0.
FORCE, 1, 9,, 250., 1., 0., 0.
FORCE, 1, 7,, 250., 0., 1., 0.
FORCE, 1, 8,, 250., 0., 1., 0.
FORCE, 1, 8,, 250., 0., 1., 0.
FORCE, 1, 9,, 250., 0., 1., 0.
$
force,3,5,,1.0, 0.,0.,1.
$
CORD2R, 99,, 0., 0., 0., 0., 0., 0., 1.
, 0., 1., 0.
ENDDATA

```

Adds transverse load and increases factor for others

Transverse load

POST BUCKLED STRENGTH (CONT.)

- Instabilities start to occur at around 20% of the total load step, as indicated by the iteration occurring at that level

.sts file

```
information summary of job: ./prob8b
version: MSC Nastran 2012
date: Feb 10, 2011; Day Time: 10:43:46
```

subcase /step #	inc #	cycl #	sepa #	cut #	cycl #	split #	separ #	cut #	rmesh #	time step of	total time of	wall time	cpu time	max displ.
---of the inc--- -----of the analysis-----										the inc	the job			
1	0	0	0	0	0	0	0	0	0	0.0000E+00	0.0000E+00	2.00	0.55	0.0000E+00
1	1	1	0	0	1	0	0	0	0	5.0000E-02	5.0000E-02	2.00	0.56	-3.3051E-03
1	2	1	0	0	2	0	0	0	0	5.0000E-02	1.0000E-01	2.00	0.58	-6.6110E-03
1	3	1	0	0	3	0	0	0	0	5.0000E-02	1.5000E-01	2.00	0.59	-9.9177E-03
1	4	3	0	0	6	0	0	0	0	5.0000E-02	2.0000E-01	2.00	0.64	3.1663E-01
1	5	1	0	0	7	0	0	0	0	5.0000E-02	2.5000E-01	2.00	0.66	2.4893E-01
1	6	1	0	0	8	0	0	0	0	5.0000E-02	3.0000E-01	2.00	0.67	2.3601E-01
1	7	1	0	0	9	0	0	0	0	5.0000E-02	3.5000E-01	2.00	0.69	2.6995E-01

.f06 file

%2.00000E-01	4	1	1.00E+00	9.92E-01	1.00E+00	1.000	0	1	0	6.20E+02	2.946E+02	2.19E-02	6.012E-01	0	0	0	3	4	
%2.00000E-01	4	2	3.19E-01	1.07E-01	3.28E+00	0.108	0	1	0	4.00E+01	1.871E+02	1.75E-02	4.544E-01	5	3	1.00	0	4	5
%2.00000E-01	4	3	4.28E-01	9.39E-02	1.91E+00	0.183	0	1	0	3.26E+01	1.612E+02	1.26E-02	3.166E-01	5	3	1.00	0	5	6

POST BUCKLED STRENGTH (CONT.)

- In that problem, the **FIXED** load stepping method worked through the instabilities.
- Other more unstable problems might need another load stepping method (such as the arc length stepping, invoked with the **ARCLN** keyword).

POST BUCKLED STRENGTH (CONT.)

- The buckled shape is now as expected:

0

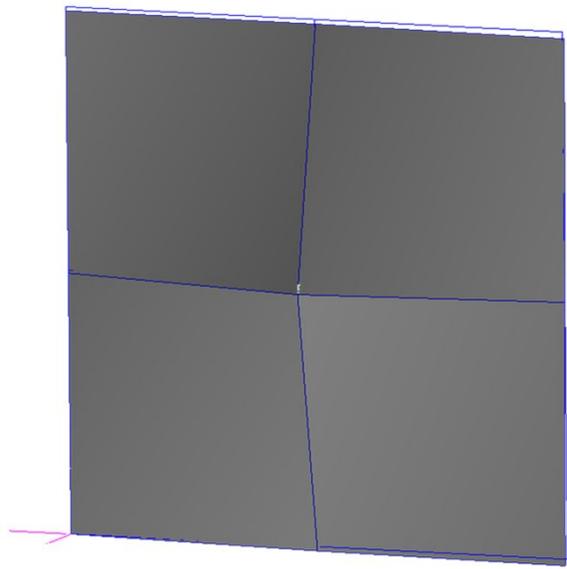
LOAD STEP = 1.00000E+00

SUBCASE 1 STEP 1

D I S P L A C E M E N T V E C T O R

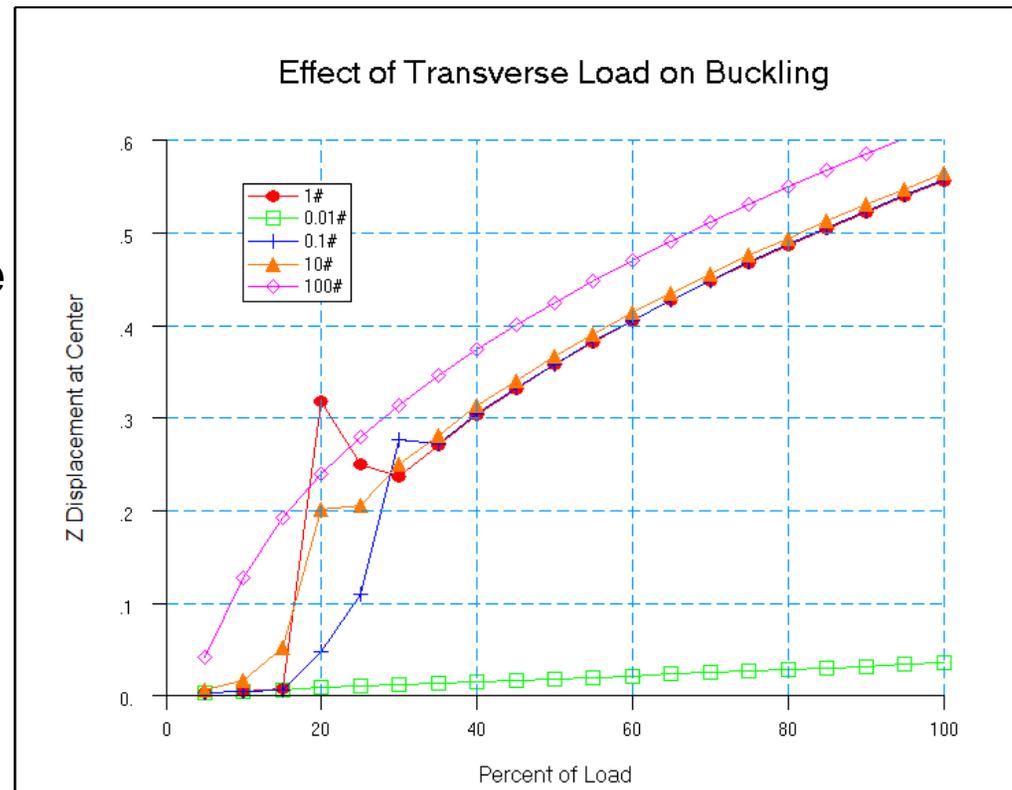
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3
1	G	0.0	0.0	0.0	-3.762029E-03	-1.307052E-02	-1.649161E-03
2	G	0.0	-2.927535E-02	0.0	-6.815525E-03	-5.959568E-02	-3.777423E-04
3	G	0.0	-4.154552E-02	0.0	1.642756E-02	3.743147E-02	-4.453932E-04
4	G	-2.025824E-02	-4.474784E-02	0.0	6.249331E-02	2.624197E-02	-1.731947E-03
5	G	4.974080E-03	-3.828866E-02	5.540228E-01	-4.317936E-05	3.555542E-05	-1.246921E-03
6	G	1.341288E-02	-4.908463E-02	0.0	-6.245833E-02	-2.635791E-02	-5.208944E-04
7	G	-2.568503E-02	-6.870695E-02	0.0	-1.654424E-02	-3.726827E-02	-1.775448E-03
8	G	-2.127939E-03	-7.060813E-02	0.0	6.902822E-03	5.956282E-02	-1.929648E-03
9	G	2.078687E-02	-7.124989E-02	0.0	3.805350E-03	1.305792E-02	-1.657953E-03

.f06 file extract



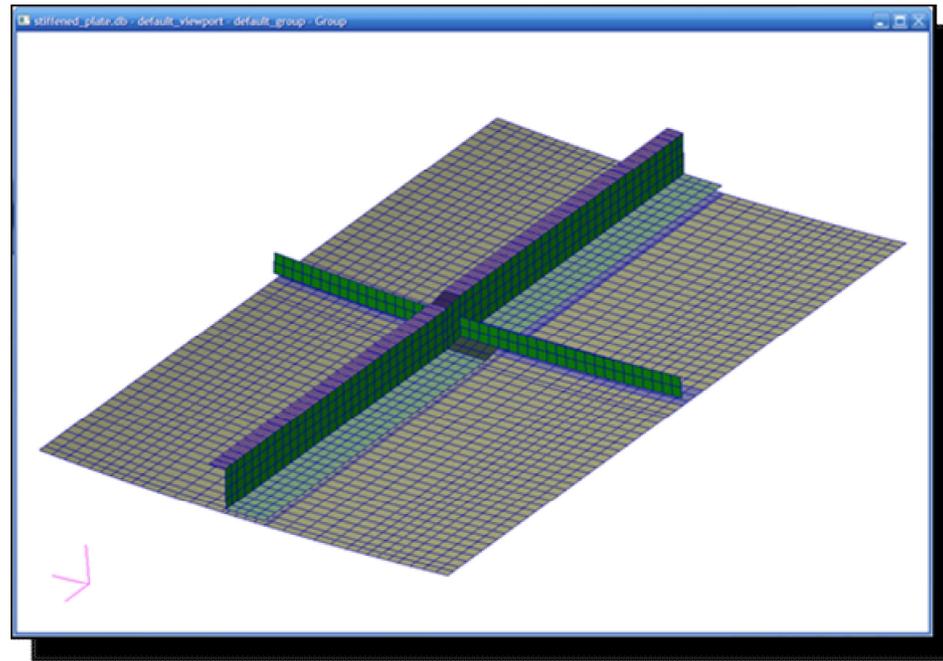
POST BUCKLED STRENGTH (CONT.)

- Note that onset of instability and the post-buckled response is a function of the perturbing load.
- The amount of perturbing force used is a measure of manufacturing tolerances (from test).
- The perturbing force shape is determined by test or analysis.



CONTACT REVIEW, FOR WORKSHOP 10

- As we look at Workshop 10, in addition to the “Large Rotation Offset” parameter (MDLPRM,OFFDEF,LROFF), Glued Contact will be used to connect stiffeners to a skin.



- As such, it would be good to explain some of the parameters involved in setting this up.

CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- **Whether you are using the older “Contact Table” approach, or the newer (Requires Patran 2013, and MSC Nastran 2013.1) “Contact Pair” approach, the following parameters are still used for both:**
 - ISEARCH Defines body detection order
 - ERROR Determines the “detection distance” for contact bodies
 - BIAS Influences “inside” vs “outside” proportions for detection tolerance
 - IGLUE Influences glued contact behavior parameters
 - COPTS Additional options for contact detection and interaction

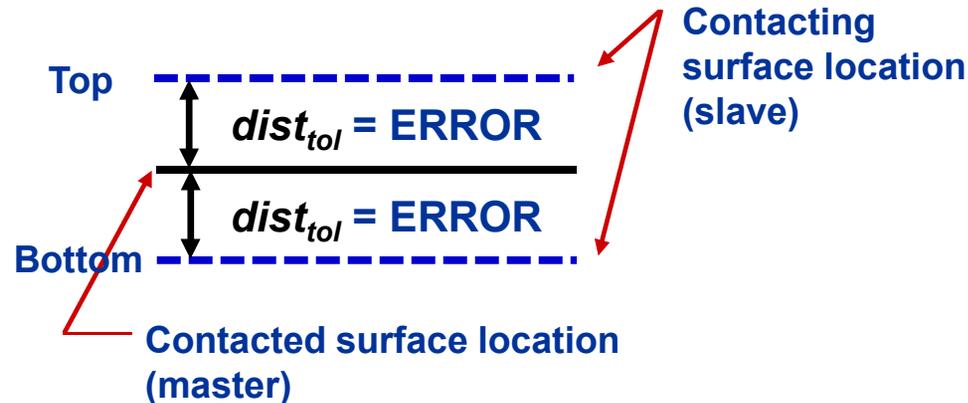
CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- **ISEARCH – controls the manner in which the search takes place.**
 - Prior to MSC Nastran 2013.1, slave and master are defined in the **BCTABLE**, along with the parameters
 - As of MSC Nastran 2013.1, slave and master bodies are defined in the **BCONNECT**, and the geometric properties are defined in a **BCONPRG**
 - This is the format Patran 2013 will default to using
- **ISEARCH=0** (default) → Double-sided search
 - First the lower-ID body is checked against the higher-ID body for contact. If contact is found, contact constraints are created.
 - Next the higher-ID body is checked against the lower-ID body and additional contact constraints are created without conflicting with the existing constraints.
- **ISEARCH=1** → Single-sided search
 - Search order is from slave to master
- **ISEARCH=2** → Automatic
 - Search order is from the body with smaller average element edge size to body with larger average element edge size. The search is single sided.
- The selection should generally be made for ISEARCH such that the finer mesh is slave, and the coarser is the master.
- For the case of perpendicular elements (edge to face), searching should be towards the edge (edge is the slave, face is the master)

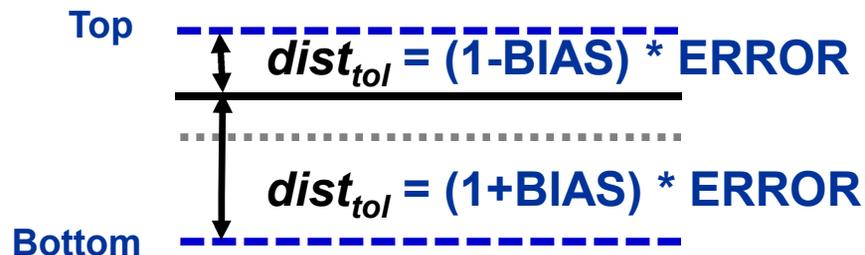
CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- **ERROR and BIAS – controls contact detection distances**

- ERROR – The distance between the two surfaces at which contact is detected. Further details on this (specific to WS10) will be shown later.



- BIAS– A shift in the error zone to one side or the other. We will use 0. (recommended for glued contact)



CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- **IGLUE**– Controls how gaps/overlaps are addressed, and whether moment transfer occurs or just displacements.
 - IGLUE=0 indicates no gluing.

Gluing Options	Transfer deflections only	Transfer deflections and moments (for shell-shell contact)
Displacements of the contact nodes are tied in case of deformable-deformable contact once the node comes in contact. The relative tangential motion of a contact node is zero in case of deformable-rigid contact. *	IGLUE = 1	IGLUE = 3
Insure that there is no relative tangential and normal displacement when a node comes into contact. An existing initial gap or overlap between the node and the contacted body will not be removed, as the node will not be projected onto the contacted body. To maintain an initial gap, ERROR should be set to a value slightly larger than the physical gap.	IGLUE = 2	IGLUE = 4

* **This option is recommended when there is no gap or overlap between contact surfaces or initial stress free contact is specified (i.e. nodes get automatically, physically moved together).**

- In this case, since we wish to maintain the initial gap, IGLUE should be 2 or 4. To handle our edge-face contact later, we will also choose to transfer moments (IGLUE=4)

CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- **COPTx– Controls general contact options.**

- In the case of Workshop 10, it will be used to turn off consideration of element thicknesses in the search for contact.

- COPTS (slave) and COPTM (master) are “packed numbers”, which designate how the surfaces may contact.

- The “packed” refers to a single integer which is a sum of multiple options, each scaled by a power of 10. Specifically:

$$\text{COPTx} = A + 10*B + 1000*C$$

- Where A governs solid element behavior, B governs 2D (either deformable shell, or rigid surface) behavior, and C governs 1D (beam/beam, or edge/beam) behavior.

- In the case of Workshop 10, in one case we will choose to use Option 6 for B:

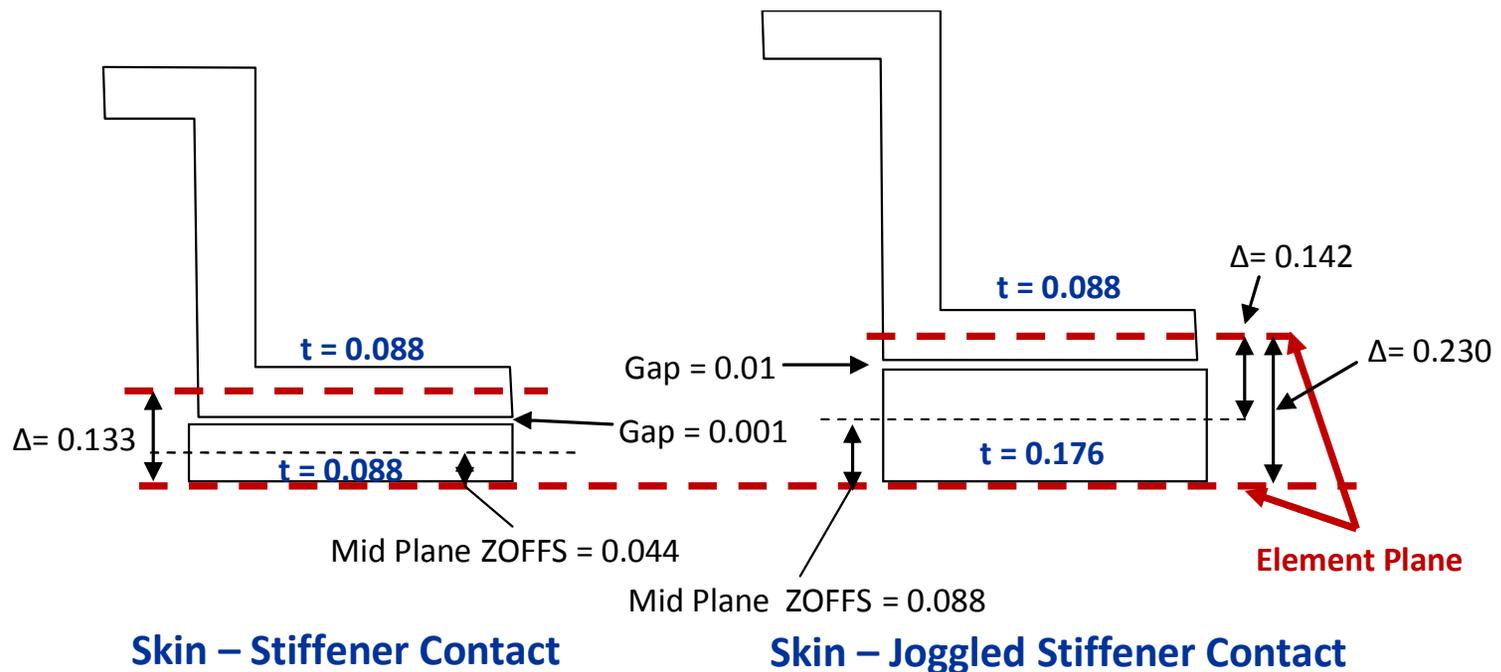
- 6: both top and bottom faces will be in the contact description, shell thickness will be ignored.

- The defaults for A(=1) and C(=1) will be left. The resulting “packed” number will be:

$$\text{COPTx} = 1*1 + 10*6 + 1000*1 = \mathbf{1061}$$

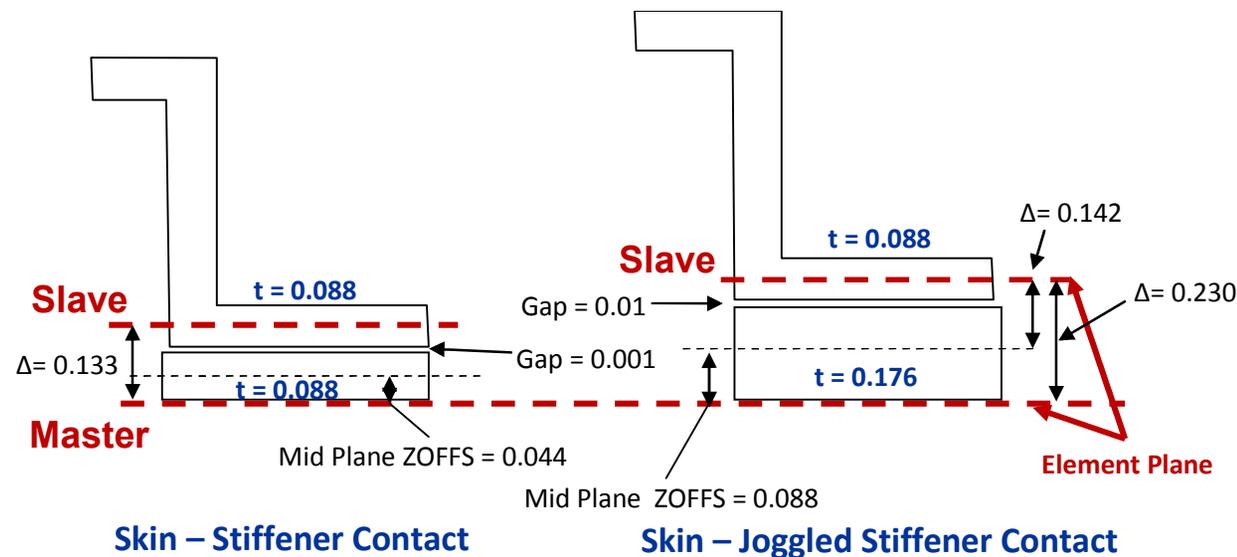
CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- Looking at the contact detection parameters, it is always a good idea to sketch out the bodies, both actual element positions as well as applied offsets and expanded thicknesses.
 - For Workshop 10, the “skin” elements (bottom, in the image below) were modeled at their Outer Mold Line.
 - The stiffener elements were all modeled at their geometric center/midsurface



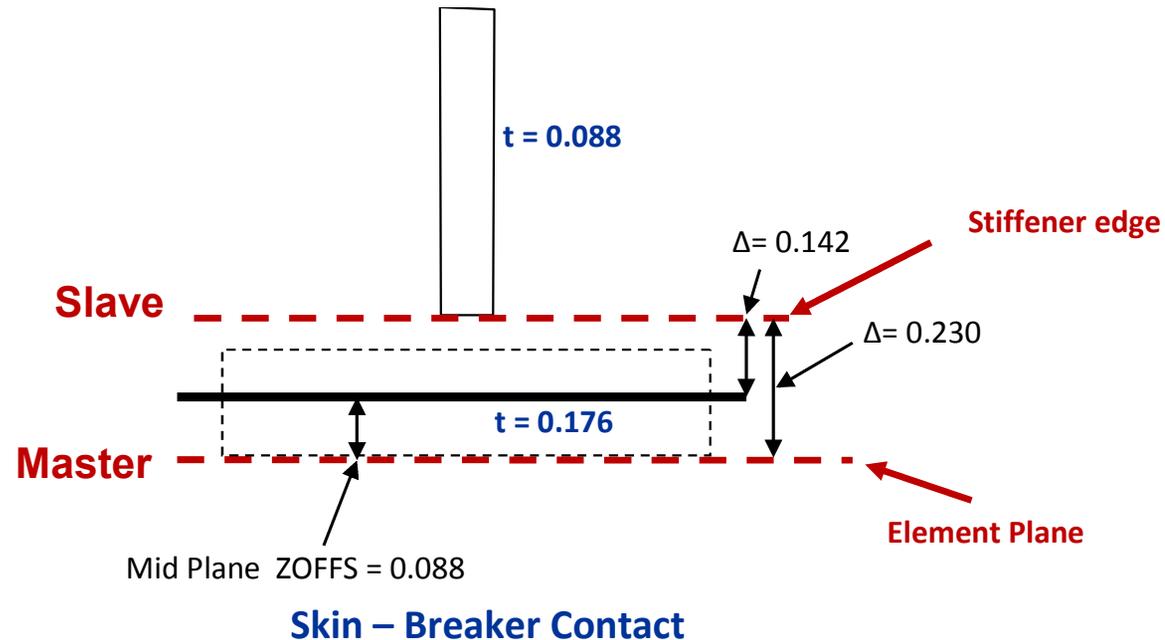
CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- **Contact: Skin to Stiffeners (Regular to Stiffener, and Thicker to Joggled Stiffener)**
 - Note that the **ZOFFS** attribute on CQUAD4 elements is used to offset them by half their thickness (.044 and .088 in this case)
 - Glued contact, in these 2 interactions, will account for element thickness when searching for elements to glue; with some small gaps seen between the calculated element edges (.001, and .01) an **ERROR** parameter closes the remaining gap (ERROR = 0.02, with BIAS=0, easily covers this range)
 - Contact searching is directed from the skin (Master) to the stiffener flanges (Slave)



CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- Contact: Skin to Breaker (edge contact)



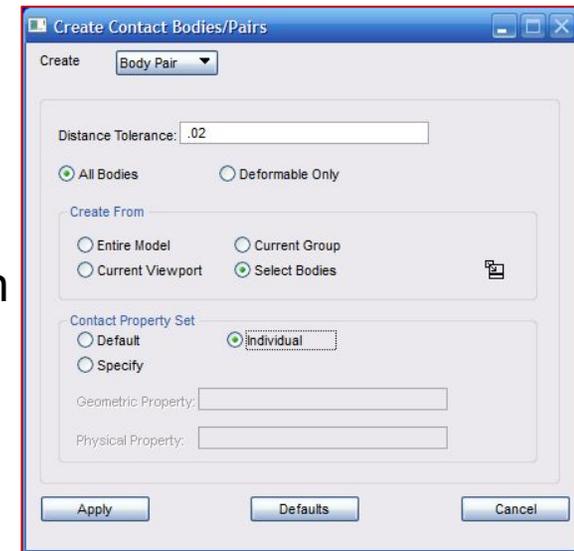
- The Ignore Thickness attribute on **COPTx** will be set to 'True' (10*6), because the contacting elements are perpendicular to each other (and we will specify the actual detection distance).
- For perpendicular elements (such as the Breaker/Skin interaction here), the thickness effect is turned off, and a larger **ERROR** value is used. Element offset is still included in the search. ($\Delta = .142$; ERROR = 0.18)
- Contact searching is directed from the skin (Master) to the breaker elements (Slave)

CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- Using the Contact Pair approach, the Loads/BCs tab has a “Create Body Pair” ribbon icon to conveniently launch the main body pair form.

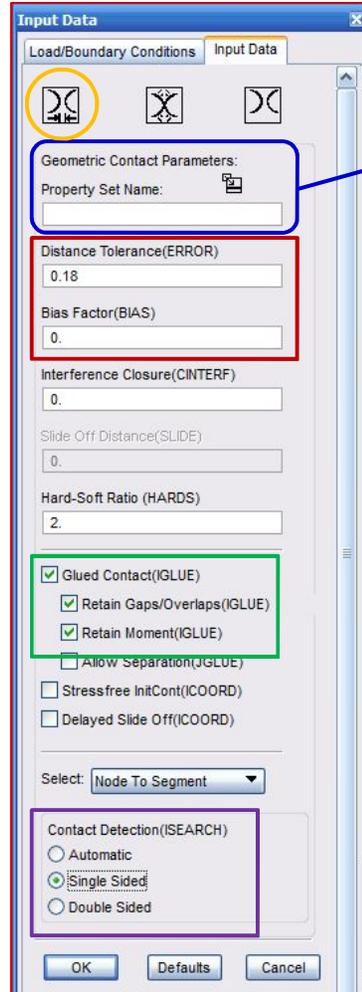
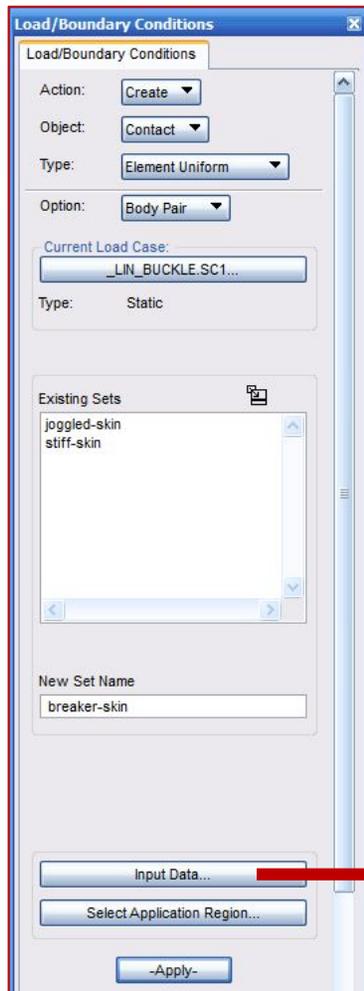


- Additionally, Tools → Modeling → Contact Bodies/Pairs... also offers a method of making the contact pairs.
 - However, since this still requires specific selection of the bodies, and only the ERROR parameter as an initial input, the LBCs form is the better approach for new contact.
 - This works well for an initial, simple contact with a single set of parameters. Or, when contact parameters have already been saved, and you simply wish to create a new body pair that references one of the same property sets.



CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- The main Loads/BCs form for Body Pairs offers all the options discussed.



Previously saved contact property sets

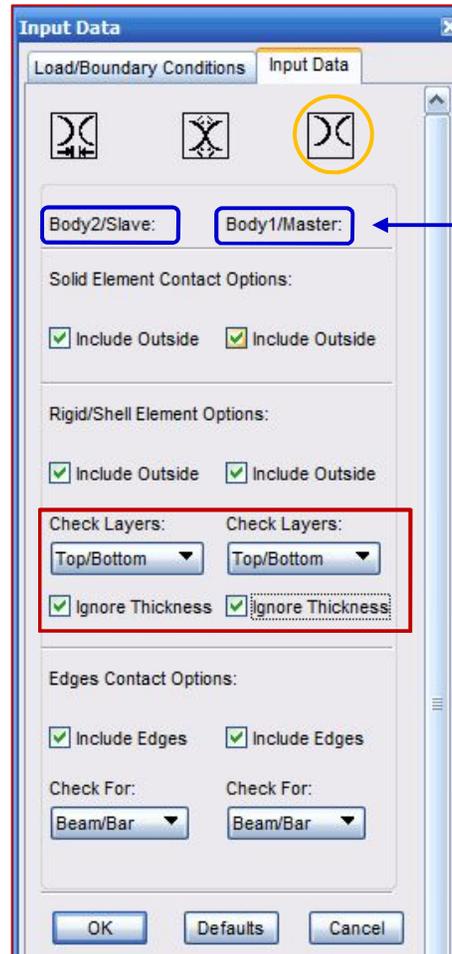
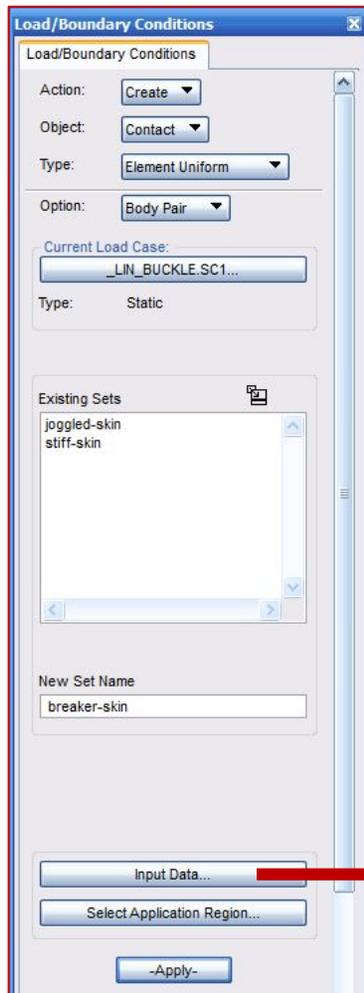
ERROR and BIAS

IGLUE

ISEARCH

CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- Most are visible in the first (Geometric) set of options. The COPTx parameters are available in the third (Contact Options) set.

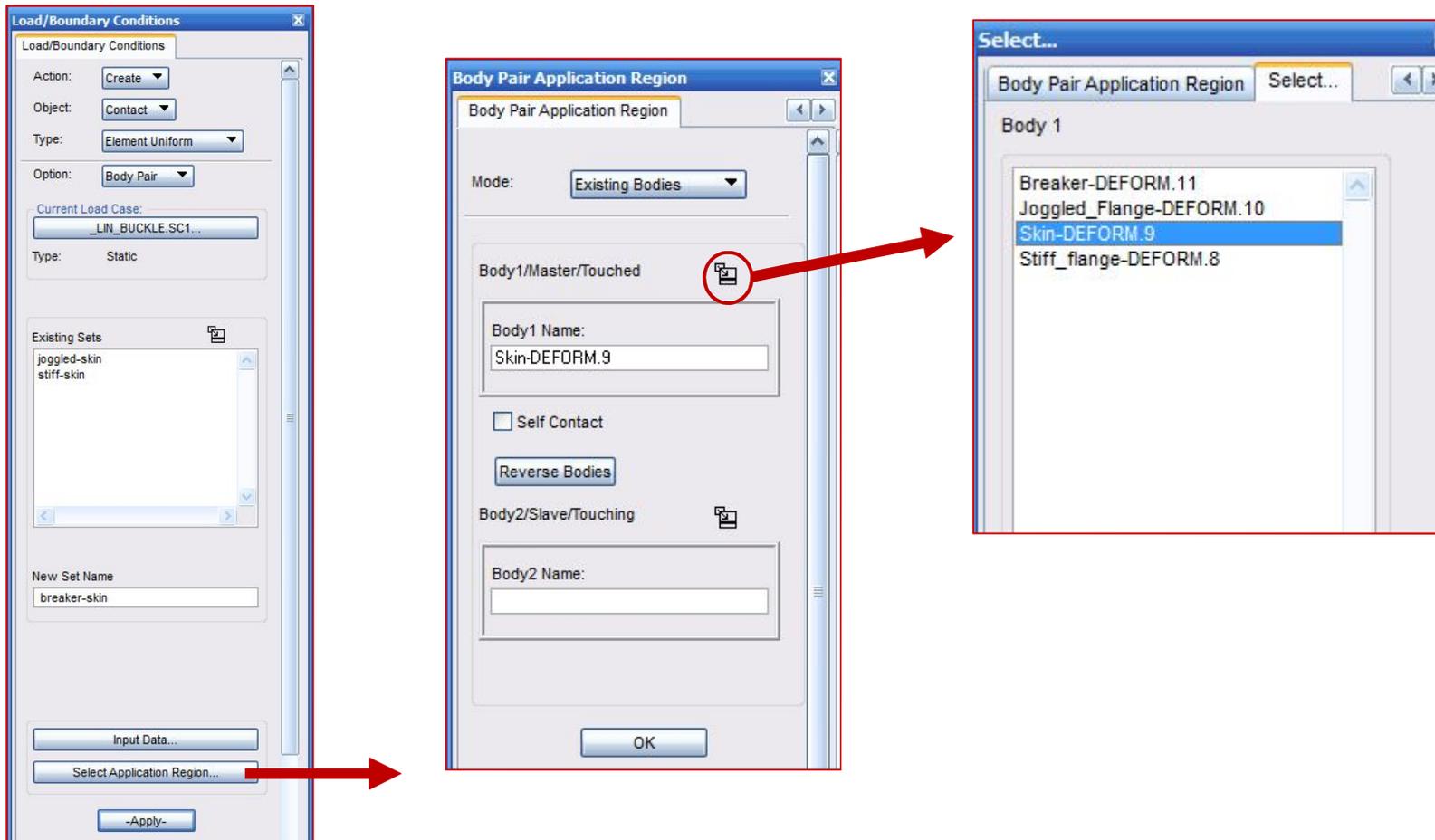


Slave/Master
indication for
columns

COPTx options
for shell contact

CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- Under the Application Region, the Slave and Master bodies are specified. These can be picked from the list of bodies by clicking on the little icon next to the description.



CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- Here is a representative example of the glued contacts for WS10, using the Contact Pair approach.
- Note that, in this case, Body 3 (skin) is common to all the pairings (written in the BCONECT entry).
- For each BCONECT, there is a corresponding PCONPRG with the various parameters we defined
- The Case Control will reference the BCTABL1 entries, which refer to the individual contact pairings included.

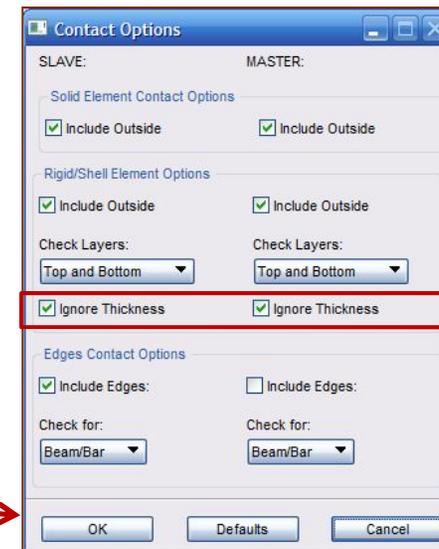
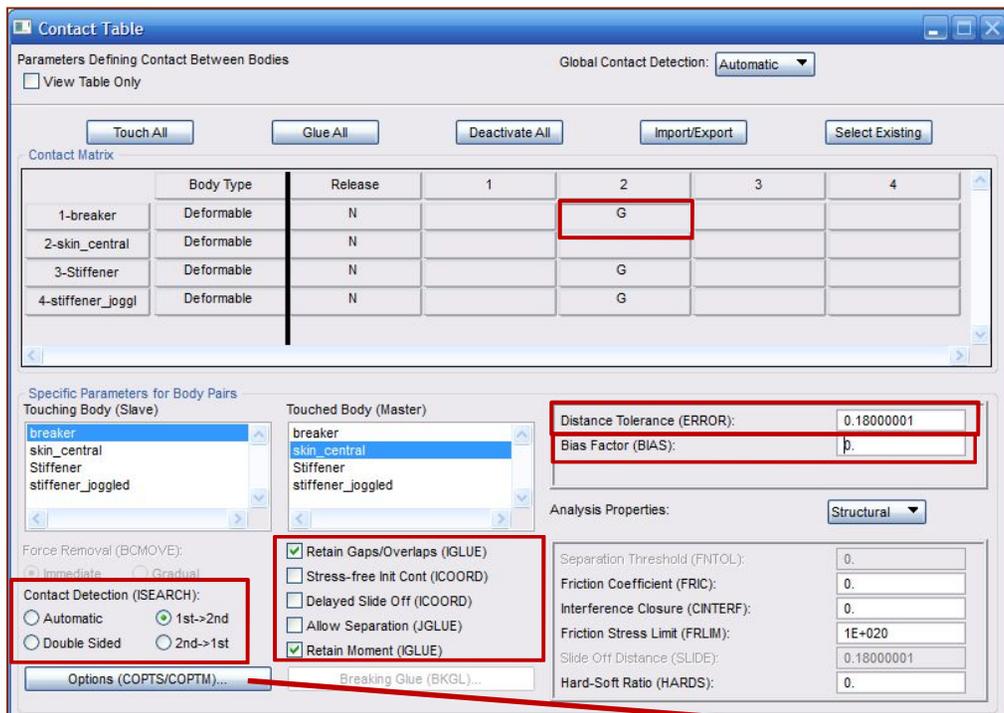
```

$ Body 1 = Breaker
$ Body 2 = Stiffener flange
$ Body 3 = Skin
$ Body 4 = Joggled stiffener flange
BCTABL1 0      8010      8011      8012
$
BCONECT 8010  3008      4      3
BCONECT 8011  3008      2      3
BCONECT 8012  3010      1      3
BCONPRG 3008  BIAS 0.      ERROR .02      IGLUE 4
BCONPRG 3010  BIAS 0.      COPTM 1061      COPTS 1061
          ERROR .18      IGLUE 4      ISEARCH 1
BCTABL1 1      8010      8011      8012
.
.
$ Material Record : AS4_3501-6
MAT8 1 2.06+7 1.49+6 .27 1.04+6 1.04+6 1.04+6 .057
      -5.-7 1.5-5 72. 331000. 208900. 8300. 33100. 10300.
$ Composite Property Reference Material: Skin_base
PCOMP 1 72. SYM
      1 .0055 45. YES 1 .0055 -45. YES
      1 .0055 0. YES 1 .0055 90. YES
      1 .0055 45. YES 1 .0055 -45. YES
      1 .0055 0. YES 1 .0055 90. YES
$
CQUAD4 249 1 3 379 384 6 90. .044
CQUAD4 250 1 379 380 385 384 90. .044
CQUAD4 251 1 380 381 386 385 90. .044
.
.
  
```

Slave/Master pairings

CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- Using the Contact Table in Patran (only option available prior to 2013, through the Analysis form), all of the options discussed are available by selecting any given “cell” in the table
 - Terms on the diagonal are self contact. Terms on opposite sides usually are mirrored (1,2 is the same as 2,1), unless one of the single-sided contact ISEARCH approaches is used



CONTACT REVIEW, FOR WORKSHOP 10 (CONT.)

- Here is a representative example of the various glued contacts for WS10, via the BCTABLE (Contact Table) approach.
 - Note that, even in the this BCTABLE, contacts are by pairs (reflecting cells in the spreadsheet in Patran)
 - Note the body search order. The Skin (body 2) is the MASTER in all pairings.
 - The offsets are applied on the element (CQUAD4) entries themselves, not on the various PCOMP entries.
 - The ERROR, IGLUE, and COPTx are defined per pair.

```

$ Body 1 = Stiffener flange
$ Body 2 = Skin
$ Body 3 = Joggled stiffener flange
$ Body 4 = Breaker
BCTABLE 1
  SLAVE 4 .18
    1 0 0
    FBSH
  MASTERS 2
  SLAVE 1 .02
    1 0 0
    FBSH
  MASTERS 2
  SLAVE 3 .02
    1 0 0
    FBSH
  MASTERS 2
    FBSH

```

ERROR

IGLUE

COPTx

IGLUE = 4:
Retain gaps, transfer moments

```

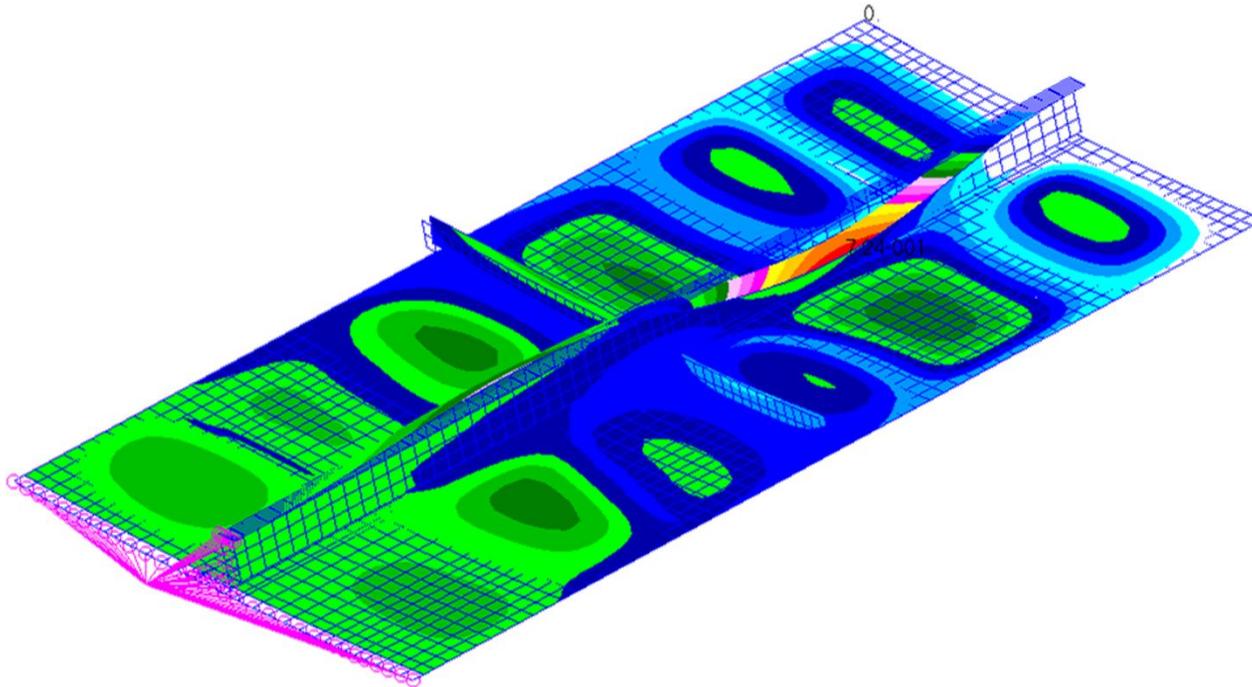
$ Material Record : AS4_3501-6
MAT8 1 2.06+7 1.49+6 .27 1.04+6 1.04+6 1.04+6 .057
      -5.-7 1.5-5 72. 331000. 208900. 8300. 33100. 10300.
$ Composite Property Record created from material record : Skin_base
PCOMP 1 72. SYM
       1 .0055 45. YES 1 .0055 -45. YES
       1 .0055 0. YES 1 .0055 90. YES
       1 .0055 45. YES 1 .0055 -45. YES
       1 .0055 0. YES 1 .0055 90. YES
$
CQUAD4 249 1 3 379 384 6 90. .044
CQUAD4 250 1 379 380 385 384 90. .044
CQUAD4 251 1 380 381 386 385 90. .044
CQUAD4 252 1 381 382 387 386 90. .044
CQUAD4 253 1 382 383 388 387 90. .044

```

Element offsets ZOFFS

EXERCISE

- Perform Workshop 10 “Post Buckled Strength” in your workshop booklet.



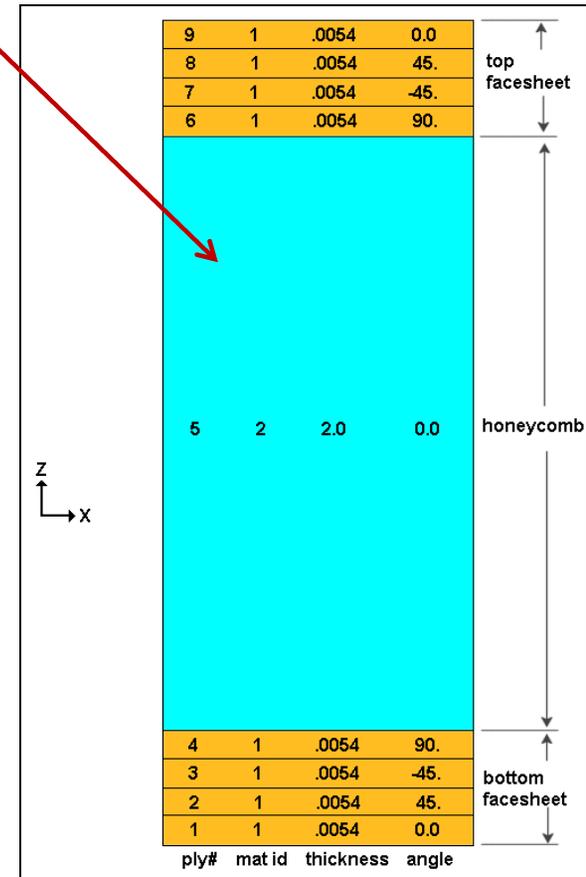
HONEYCOMB SANDWICH SHEAR BUCKLING

- **A low buckling mode is obtained in plates with low transverse shear stiffness.**
- **Whether it is real or not must be determined by test.**
- **Plates theory assumes that the transverse shear deflection is small in comparison to bending deflections.**
- **Typical composites with an organic matrix (graphite/epoxy) have transverse shear stiffness 10 times less than isotropic materials with the same E.**
- **Typical honeycomb sandwich composites have transverse stiffness 100 times less than isotropic materials with the same E.**
 - This may violate the plate theory assumptions.
 - It also tends to make the shear buckling mode dominant.

HONEYCOMB SANDWICH SHEAR BUCKLING (CONT.)

- Honeycomb is modeled as a thick layer
- With no stiffness except for G13 and G23

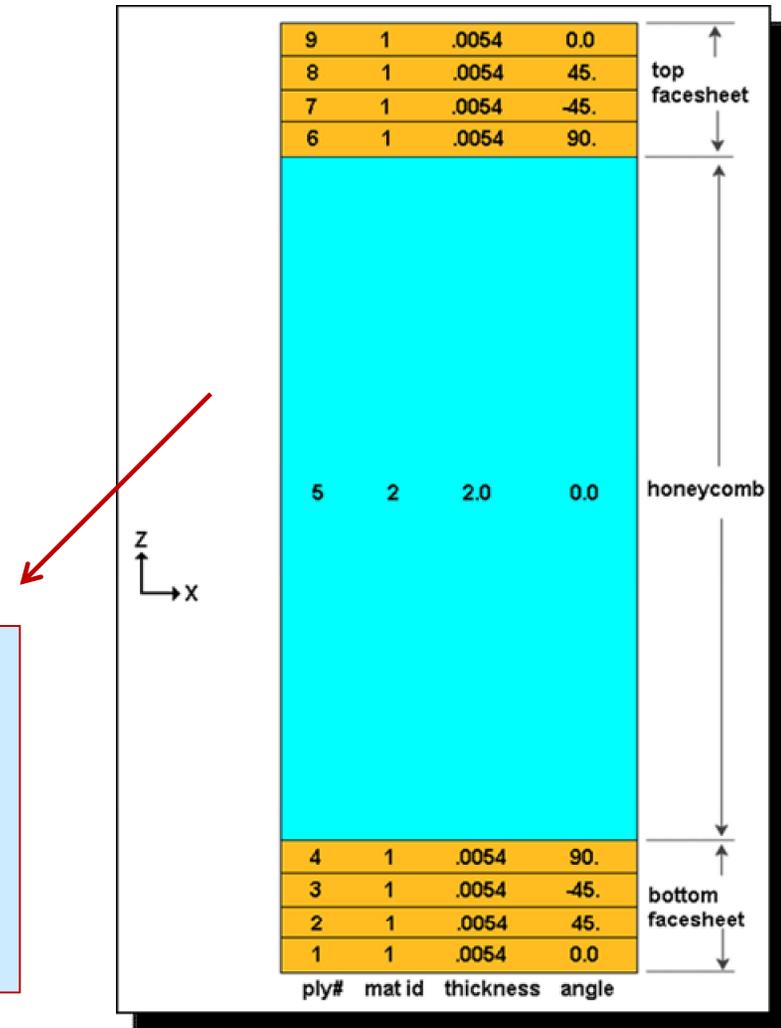
	MATID 1 gr/ep	MATID 2 h/c
E11	20e6	1e3
E22	2e6	1e3
U12	.35	.3
G12	1e6	1e2
G13	1e6	1.0e5
G23	1e6	0.5e5



HONEYCOMB SANDWICH SHEAR BUCKLING (CONT.)

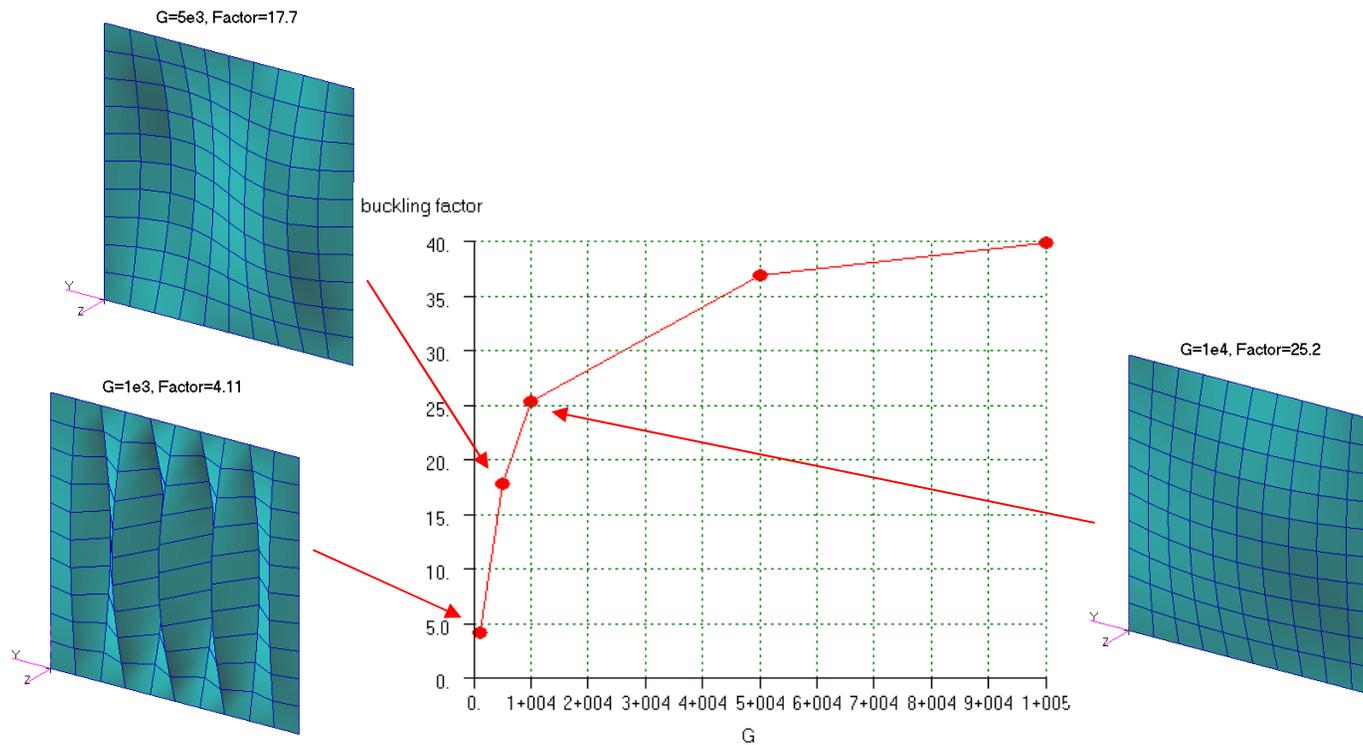
- A honeycomb sandwich with ply face sheets and honeycomb core is modeled as shown below:

```
PCOMP,1,,,,,,,,sym
,1,.0054,0.0,YES
,1,.0054,45.,YES
,1,.0054,45.,YES
,1,.0054,90.0,YES
,1,.0054,-45.,YES
,2,1.0,0.0,YES
MAT8,1,2.e7,2.e6,.35,1.e6,1.e6,1.e6
MAT8,2,1.e3,1.e3,.3,1.e3,1.e5,.5e5
```



HONEYCOMB SANDWICH SHEAR BUCKLING (CONT.)

- Load is uniaxial, simply supported boundaries
- Shear buckling occurs at lower G values



SECTION 10

OPTIMIZATION OF COMPOSITES

DEFINITION OF MSC NASTRAN OPTIMIZATION

- **Automated modifications of the analysis model to achieve a desired objective while satisfying specified design requirements.**
- **For instance, minimizing structural weight through reduced number of plies while not having any plies fail due to applied loads.**
- **...or tailoring structural and thermal deflections by changing ply angles**
- **...or avoiding natural frequencies of the structure**
- **...or increasing the buckling load**

DEFINITION OF MSC NASTRAN OPTIMIZATION

- **Typical Optimization Statement**
 - Objective function
 - Most important goal that you try to accomplish (e.g., minimize the weight)
 - Design Variables
 - Areas that you can modify (e.g., thickness of a plate)
 - Satisfy design constraints
 - Performance and manufacturing criteria (e.g., the minimum fatigue life)

DEFINITION OF MSC NASTRAN OPTIMIZATION

- **The design objective and design constraints operate on design responses**
- **Design responses are derived from design variables and analysis results such as:**
 - Weight
 - Stress
 - Deflections
 - Failure theorems
 - Natural frequencies
 - Combinations of the above
 - etc.

DESIGN RESPONSES

- **A design response is an analytical result.**
- **They are defined with either a DRESP1, DRESP2, or DRESP3 entry.**
- **The DRESP1 is the most basic:**
 - It's simply a result quantity from MSC Nastran: stress, fatigue life, acoustic power, etc.
 - There are sixty possible quantities
 - See Table 8-15 in the QRG from the DRESP1 bulk data entry for a complete list of available quantities
- **The DRESP2 allows you to define a response as an equation using DRESP1 quantities, design variables, tables of constants, etc.**
- **The DRESP3 is like the DRESP2, but the calculation can be done in an external routine, rather than just an equation**

DESIGN RESPONSES

- The DRESP1 requires a unique id number and it can be given a label
- The other values, RTYPE, PTYPE, etc. define which quantity the response represents
- The specific values for these are defined in Table 8-15 of the [QRG](#).

1	2	3	4	5	6	7	8	9	10
DRESP1	ID	LABEL	RTYPE	PTYPE	REGION	ATTA	ATTB	ATT1	
DRESP1	10	weight	WEIGHT						
	ATT2	etc.							

dresp1,10,wtmin,weight

.bdf file extract

OBJECTIVE - DESOBJ

- **The design objective is defined in Case Control**
- **The syntax is:**
 - DESOBJ(MIN)=n
 - where n is the ID of a design response (DRESP1, DRESP2, or DRESP3)
- **You can also specify that the response be maximized:**
 - DESOBJ(MAX)=n
- **Minimized is the default**

DESIGN VARIABLES - DESVAR

- Design variables are model properties that can change.
- Controlled by the DESVAR bulk data entry

1	2	3	4	5	6	7	8	9	10
DESVAR	ID	LABEL	XINIT	XLB	XUB	DELXV	DDVAL		
DESVAR	1	PLY0	1.0	.02	300.		22		

```
desvar,1,ply0,1.0,.02,300.,,22
```

.bdf file extract

- **The connection of a design variable to a property is done with a:**
 - DVPREL1 or 2 for an element property on a property bulk data entry.
 - DVMREL1 or 2 for a material property.
 - DVCREL1 or 2 for an element property on a element bulk data entry.
 - DVGRID for a grid location.

DISCRETE DESIGN VARIABLES - DDVAL

- Some design variables can only be particular values
- For composites, it can be used as an integer multiples of ply thickness

1	2	3	4	5	6	7	8	9	10
DDVAL	ID	DDVAL1	DDVAL2	DDVAL3	DDVAL4	DDVAL5	DDVAL6	DDVAL7	
DDVAL	22	1.0	THRU	200.	INC	1.0			

```
ddval,22  
, 1.0, thru, 300., by, 1.0
```

.bdf file extract

- The DDVAL field is used with composites for discrete thicknesses to represent multiple plies.

WHY DISCRETE DESIGN VARIABLES?

- **Optimization algorithm:**
 - Optimizer is gradient based
 - A design variable can take any value (e.g., 1.232891)
- **Real World:**
 - Material frequently comes in standard gages or sizes (e.g., 1.20, 1.25, 1.30, etc.)
- **Optimization is first done with continuous design variables.**
- **Then a discrete analysis is performed afterwards.**
- **Discrete variable = After Continuum converges**
 - DOE (default)
 - Conservative Discrete Design
 - Round up
 - Round off

PROPERTY VARIABLES - DVPREL1

- The DVPREL1 and DVPREL2 link the element property to the design variable.
- When the design variable changes, so does the element property.

1	2	3	4	5	6	7	8	9	10
DVPREL1	ID	TYPE	PID	PNAME/FID	PMIN	PMAX	CO		
DVPREL1	31	PCOMP	1	13	0.0001	2.0			

	DVID1	COEF1	DVID2	COEF2	DVID3	COEF3	etc.		
	1	0.0054							

PROPERTY VARIABLES - DVPREL1

- **DVIDi are the design variable names and are related as follows:**

$$Prop = C0 + \sum_i COEFi \times DVIDi$$

```
dvprel1,31,pcomp,1,13,.0001,2.0  
,1,0.0054
```

.bdf file excerpt

- **Which multiplies design variable 1 by 0.0054 to get the ply thickness of PCOMP 1 field 13.**

CONSTRAINTS - DCONSTR

- **The constraints are selected in the Case Control with the following commands:**
 - DESSUB=n for subcase constraints
 - DESGLB=n for global constraints
 - where n is the ID of a DCONSTR bulk data entry
- **The constraints set limits to responses**
 - Such as maximum stress or displacements
 - Or the range of an eigenvalue

1	2	3	4	5	6	7	8	9	10
DCONSTR	DCID	RID	LALLOW/LID	UALLOW/UID	LOWFQ	HIGHFQ			
DCONSTR	20	41		1.0					

CONSTRAINTS – DCONSTR

- **DCONSTR 20 has an upper limit of 1.0 on the failure index.**
- **Defined on the DRESP1 number 41 on the elements of PCOMP 1, ply 1, item code 5 (see QRG for item codes description)**

```
dconstr,20,41,,1.0  
dresp1,41,failure,cfailure,pcomp,,5,1,1
```

.bdf file excerpt

MSC NASTRAN OPTIMIZATION INPUT FILE

Example .dat file

```

nastran prtpcomp=1
init master(s)
sol 200
cend
title = optimization
subcase = 1
analysis=statics
desobj = 10
dessub = 20
disp =all
stress=all
spcforces=all
spc=1
load=1
begin bulk
$
param,post,-1
$
PCOMP, 1,,, 5000., HILL,,, SYM,
, 1, .0054, 0., YES
, 1, .0054, 45., YES
$
MAT8, 1, 2.+7, 2.+6, .35, 1.+6, 1.+6, 1.+6, 0.051
,,,130000., 120000., 11000., 12000., 12500.
$
CQUAD4 1 1 1 2 5 4 99
CORD2R, 99,, 0., 0., 0., 0., 0., 1.
, 0., 1., 0.
$
GRID 1 0. 0. 0.
GRID 2 0. .5 0.
$
SPC1,1,1235,1
$
FORCE 1 3 500. 0. 1. 0.

```

```

FORCE 1 6 500. 0. 1. 0.
FORCE 1 6 500. 0. 1. 0.
FORCE 1 9 500. 0. 1. 0.
FORCE 1 7 250. 1. 0. 0.
$
$ optimization model
$
param,naspvt,1
param,despch,1
doptprm,desmax,30,p1,1,p2,15,conv1,1.e-4,+
+,conv2,1.e-6,convdv,1.e-4,convpr,1.e-4
$
drespl,10,wtmin,weight
$
dconstr,20,41,,1.0
dconstr,20,42,,1.0
$
drespl,41,failure,cfailure,pcomp,,5,1,1
drespl,42,failure,cfailure,pcomp,,5,2,1
$
desvar,1,ply0 ,1.0,.02,300.,.5,22
desvar,2,ply45 ,1.0,.02,300.,.5,22
$
ddval,22
, 1.0, thru, 300., by, 1.0
$
dvpres1,31,pcomp,1,13,.0001,2.0
,1,0.0054
dvpres1,32,pcomp,1,17,.0001,2.0
,2,0.0054
$
enddata

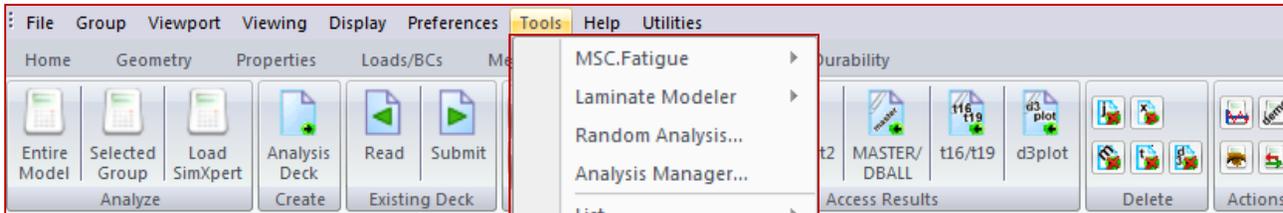
```

MSC NASTRAN OPTIMIZATION INPUT FILE

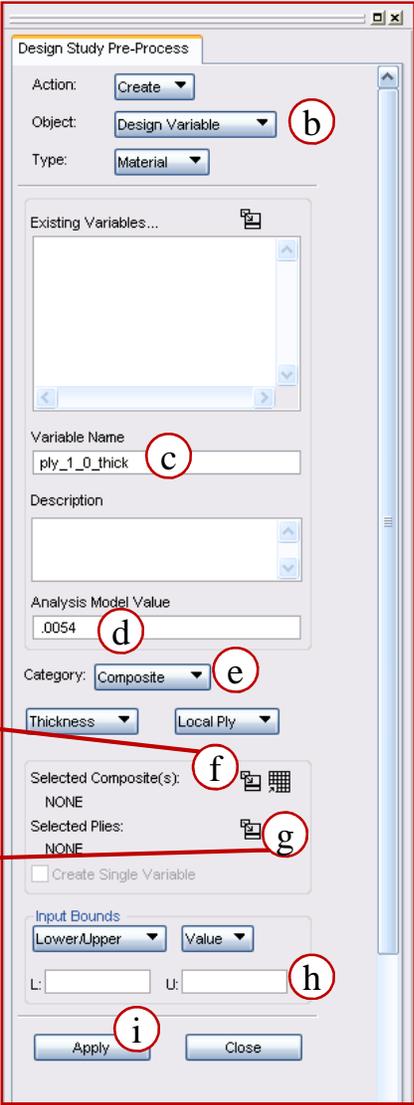
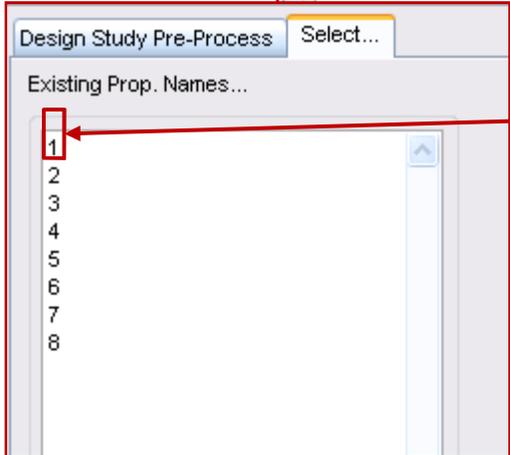
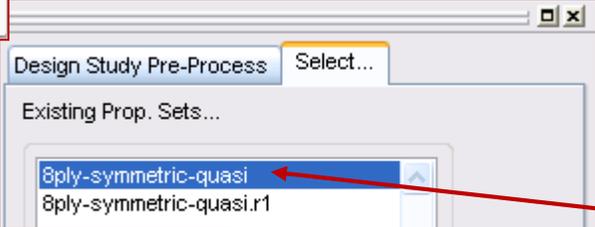
- Each ply is a design variable that will vary to find the minimum weight, while not exceeding a failure index of 1.0.
- Note that the SYM option on the PCOMP's LAM field is used to ensure that the lay-up stays symmetric while the ply thickness design variables change.
- The thickness design variables are set to units of plies instead of length by using the multiplier on the DVPREL1 entry.

PATRAN INTERFACE

PATRAN DESIGN VARIABLES

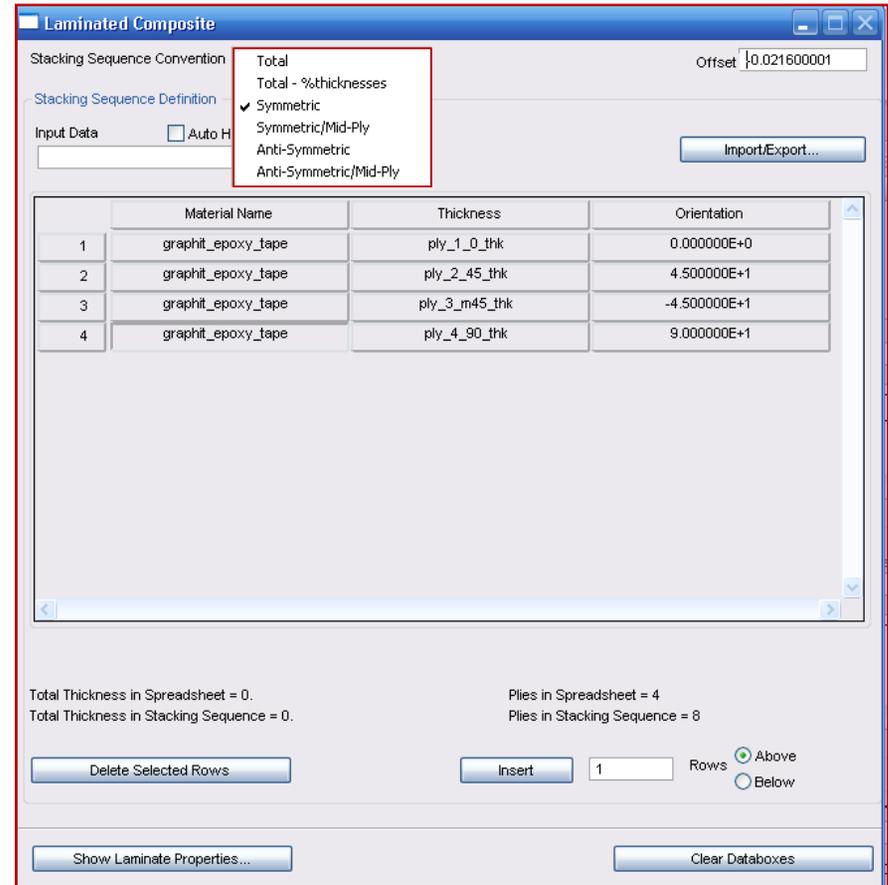


- a) Tools/Design Study/ Pre-Process
 - b) Create/Design Variable/ Material
 - c) Enter Variable Name "ply1_0_thick"
 - d) Enter Analysis Model Value "0.0054"
 - e) Category: Composite
 - f) Selected Composites(s) Select Material Sets 8Ply-symmetric-quasi
 - g) Select Material Properties Select ply 1
 - h) Define upper and lower bounds
 - i) Apply
- Repeat steps f-i for each additional ply (note that for symmetric layup, only need 4 layers)

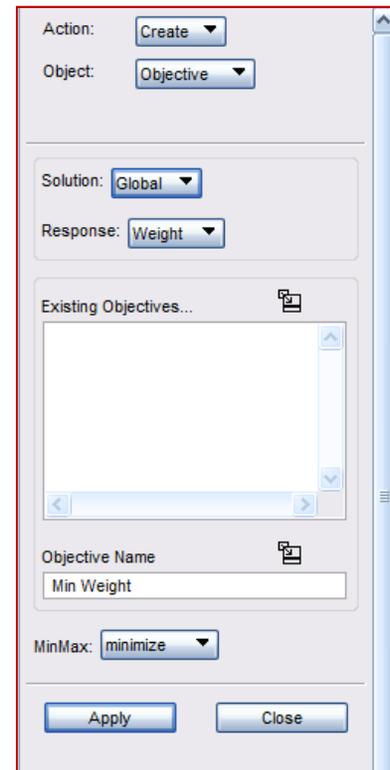
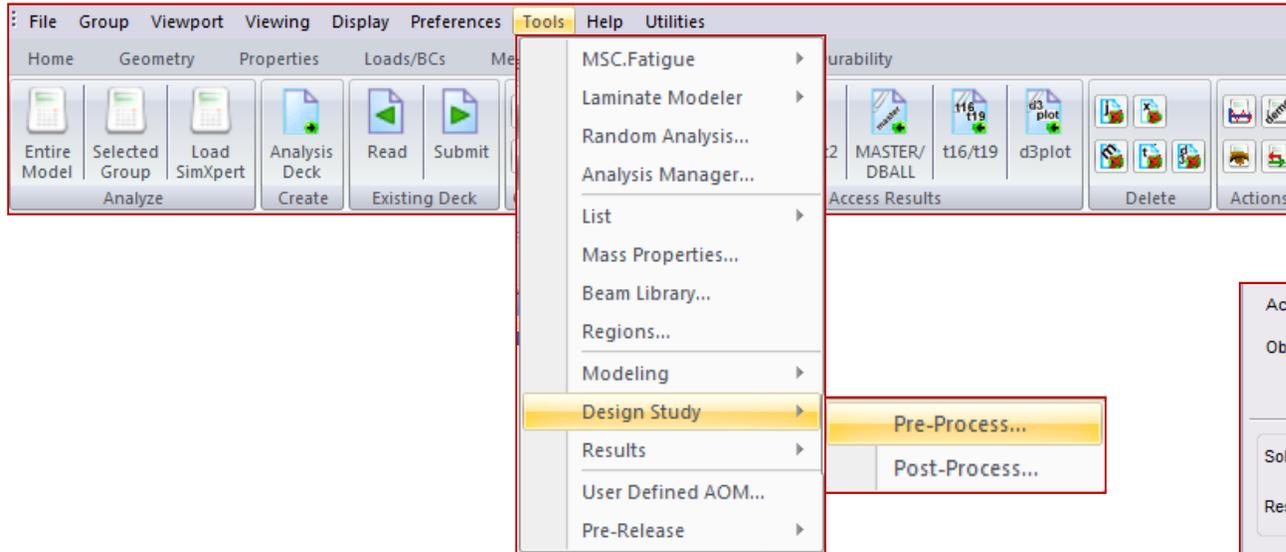


PATRAN SYMMETRY OPTIONS FOR OPTIMIZATION

- In order to get the SYM option on the PCOMP's LAM field, use the Symmetric option in the Materials/Create/Composite spreadsheet.
- This will write the symmetric ply lay-up to the PCOMP and use the SYM option in the LAM field.
- However, if any of the other Stacking Sequence Conventions in the material lay-up spreadsheet are used, then the full lay-up is written and the PCOMP SYM will not be used.

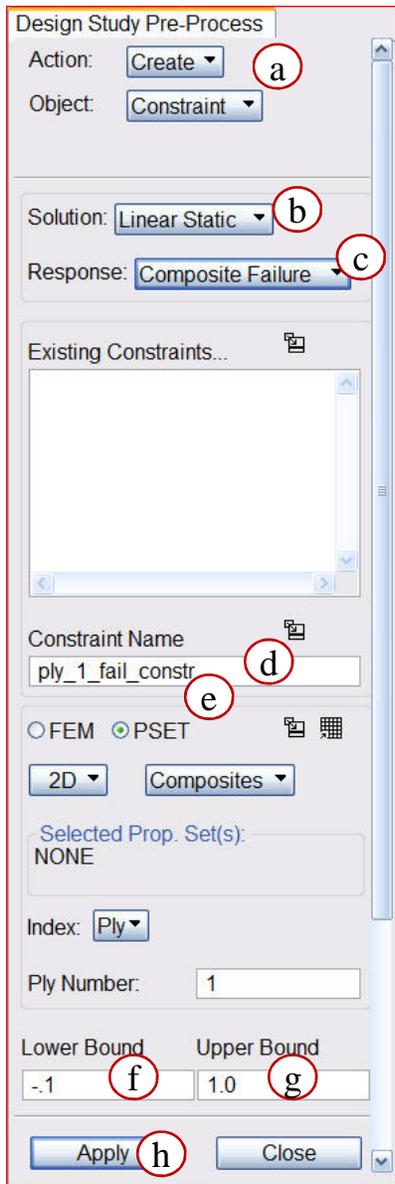


PATRAN DESIGN OBJECTIVE



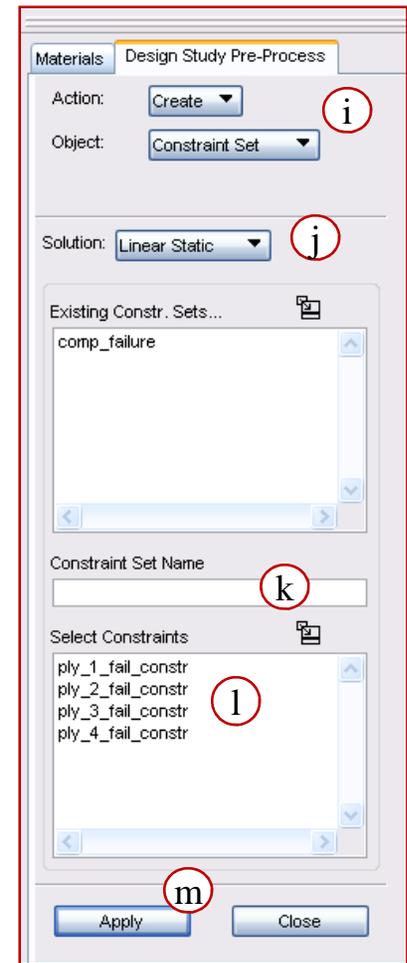
Create/Objective
Solution: Global
Response: Weight
Enter Objective Name:
Min Weight
Apply

PATRAN DESIGN CONSTRAINT AND CONSTRAINT SET

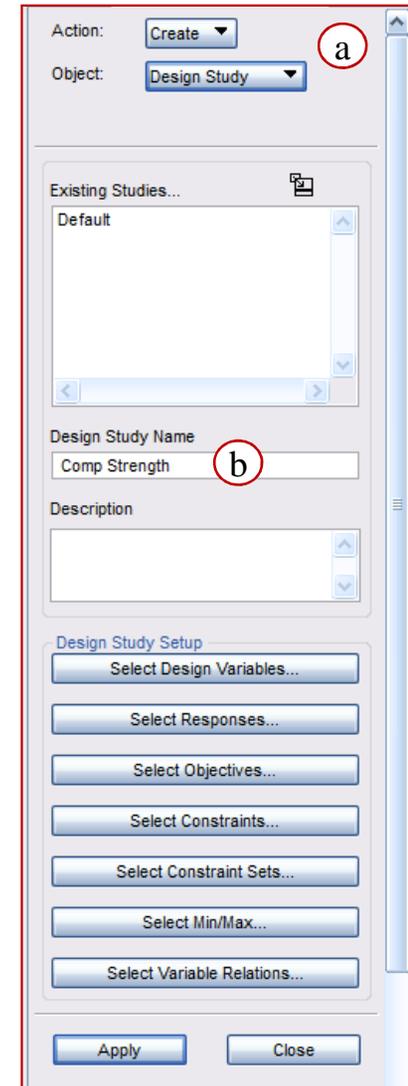
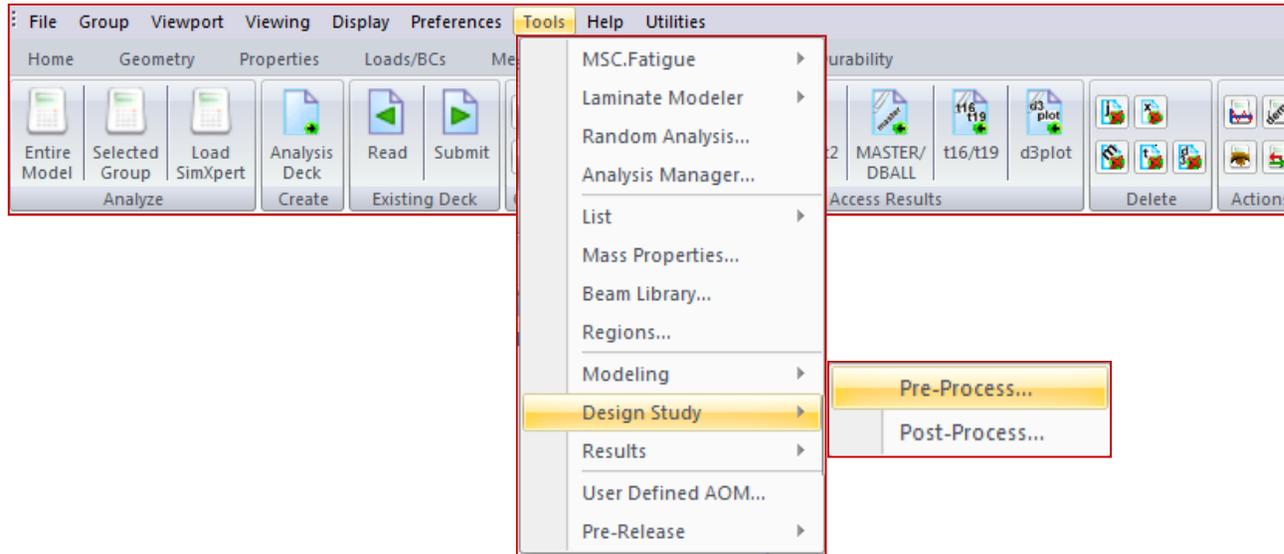


- a) Create/Constraint
 - b) Solution: Linear Static
 - c) Response: Comp. Fail
 - d) Enter Constraint Name: "ply_1_fail_constr"
 - e) Select Property Set comp
 - f) Lower Bound: "-1"
 - g) Upper Bound: "1.0"
 - h) Apply
- Repeat steps e-h for each ply

- i) Create/Constraint Set
- j) Solution: Linear Static
- k) Constraint Set Name: "Comp Fail"
- l) Constraints to be included: select all
- m) Apply



PATRAN DESIGN STUDY



- a) Create/Design Study
- b) *Design Study Name: Comp Strength*

DESIGN STUDY – SELECT VARIABLES

a) Click **Select Design Variables**

b) For each row (design variable):

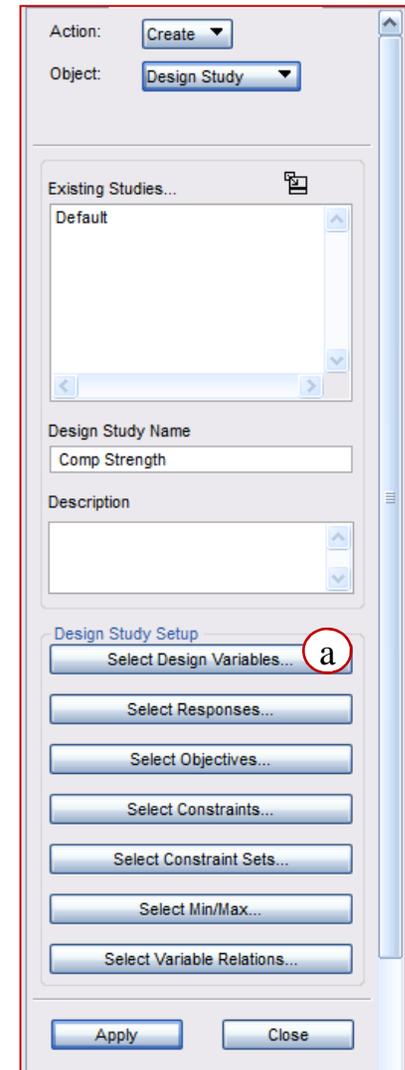
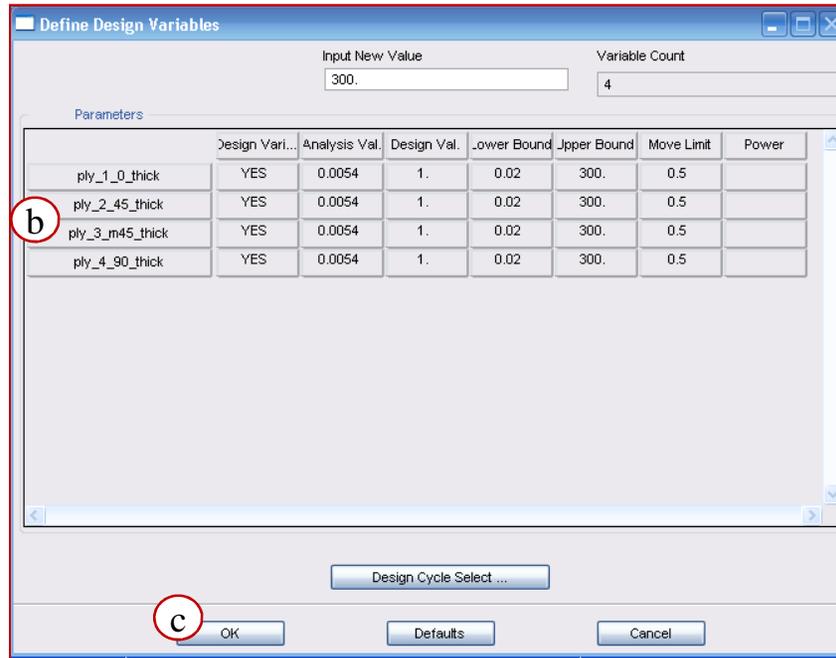
Click in the *Design Val.* cell, enter **1** in the *Input New Value* data box, then press <enter>.

Similarly enter:

0.02 for *Lower Bound*

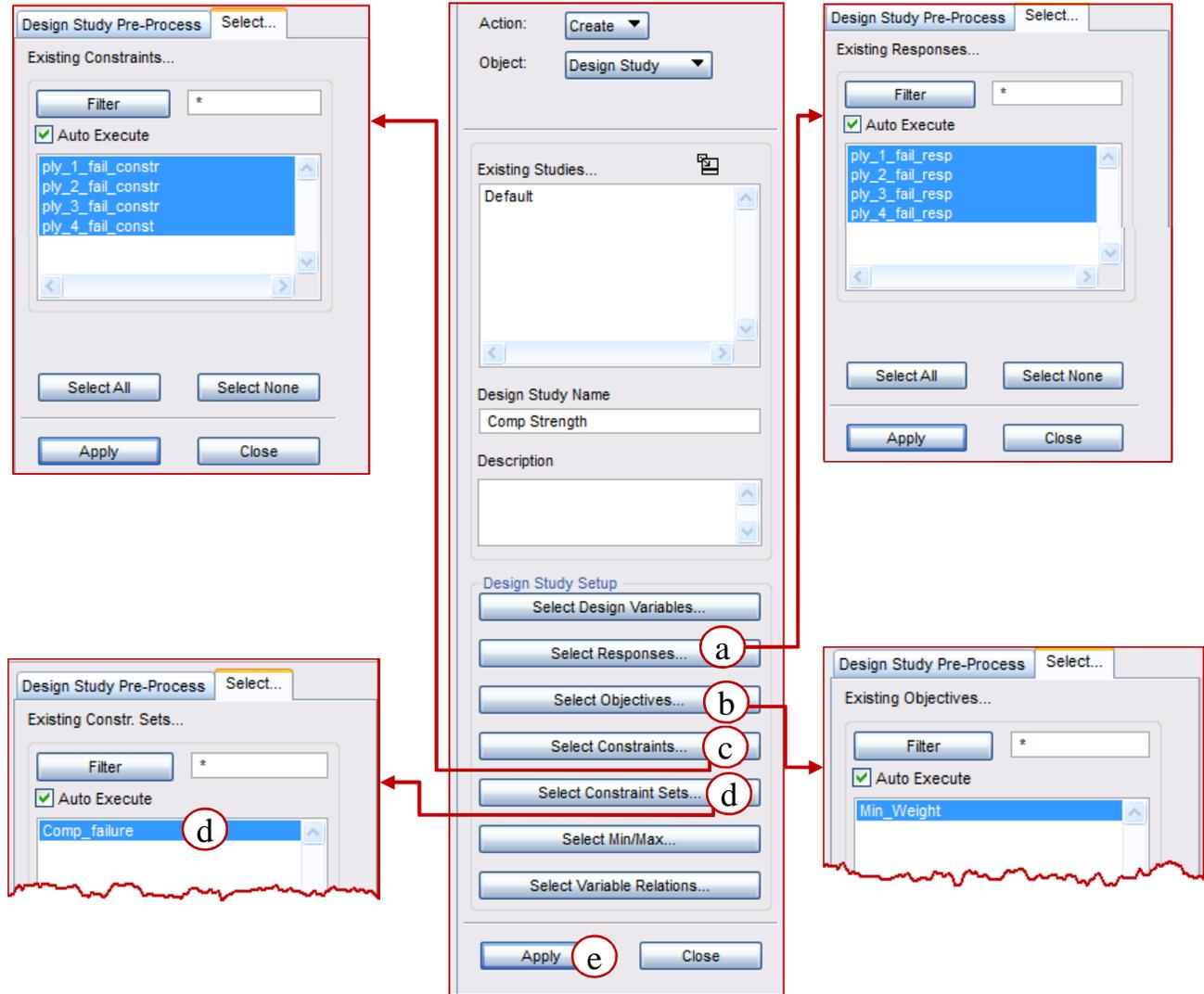
300 for *Upper Bound*

c) **OK**

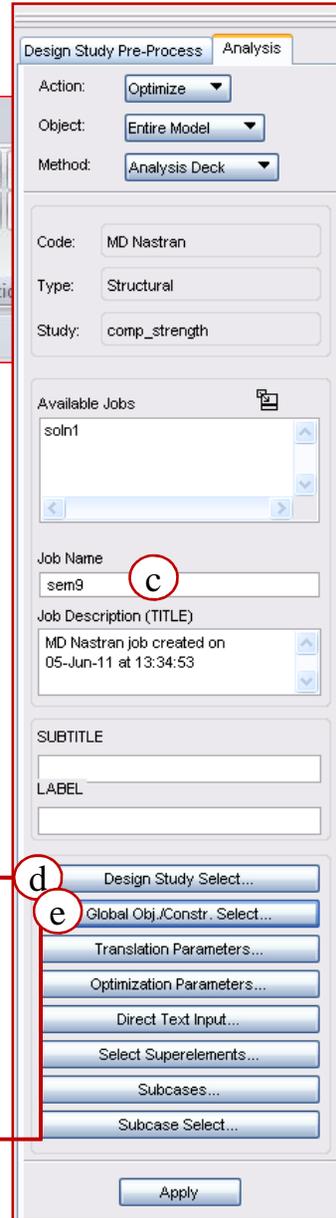
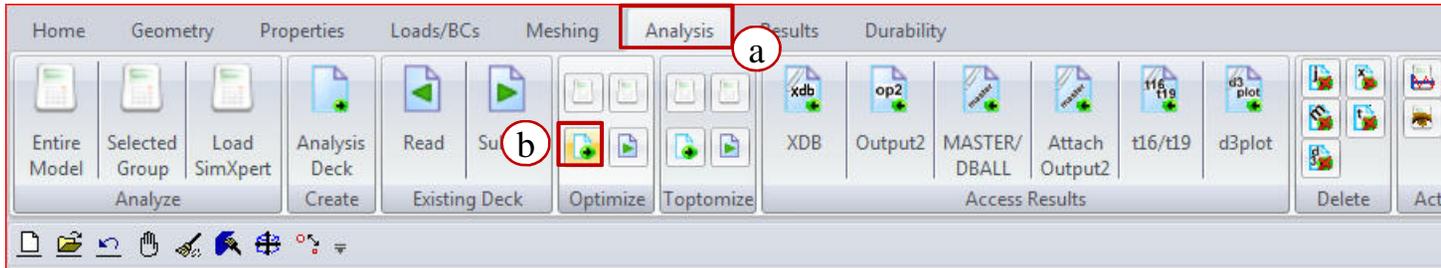


DESIGN STUDY – SELECT RESPONSES, OBJECTIVE, AND CONSTRAINTS

- a) Select Responses:
 Select All
 Close
- b) Select Objective:
 Click on **Min_Weight**
 Close
- c) Select Constraints:
 Select All
 Close
- d) Select Constraints Sets:
 Click on **Comp_failure**
 Close
- e) Apply



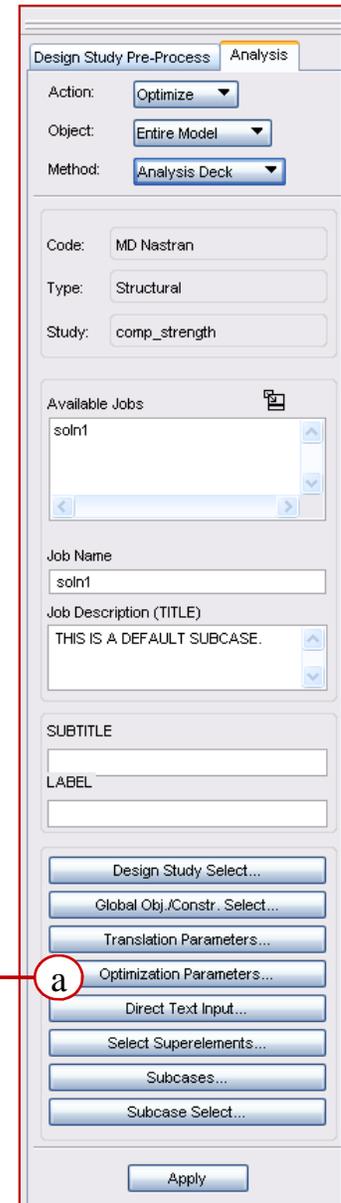
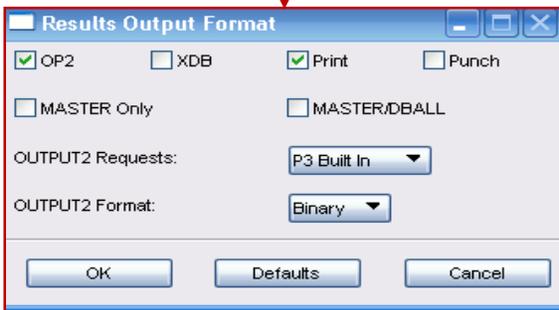
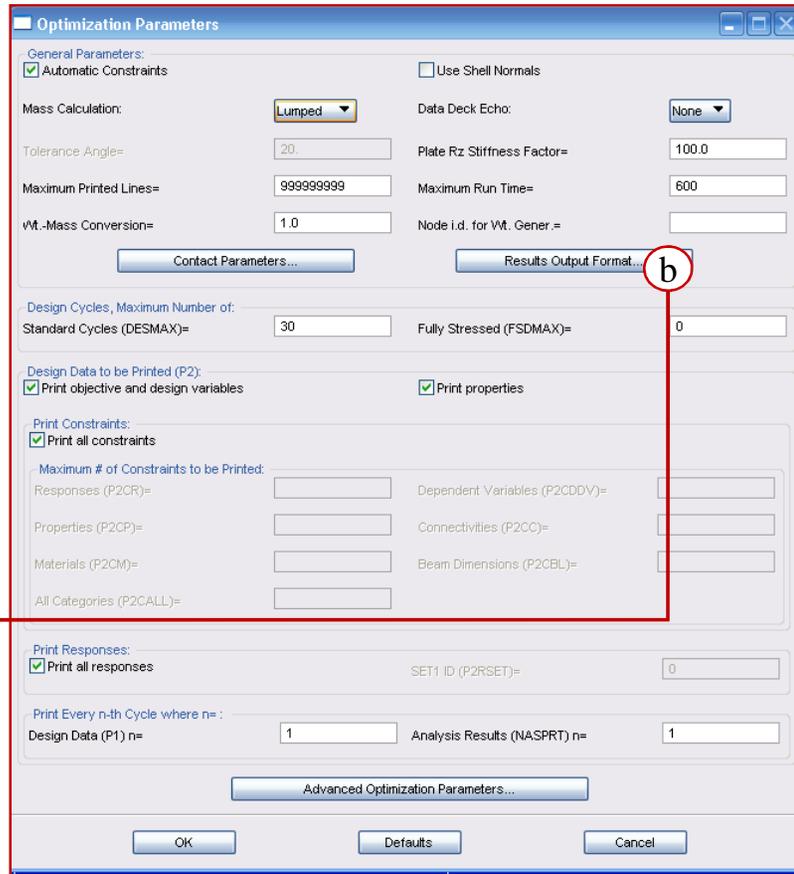
PATRAN – ANALYSIS OPTIMIZE MENUS



- a) Analysis
- b) Create Deck
- c) Job Name **sem9**
- d) Design Study Select
Select **Comp_Strength**
- e) Global Obj/Constr Select.
Min_Weight
- f) OK

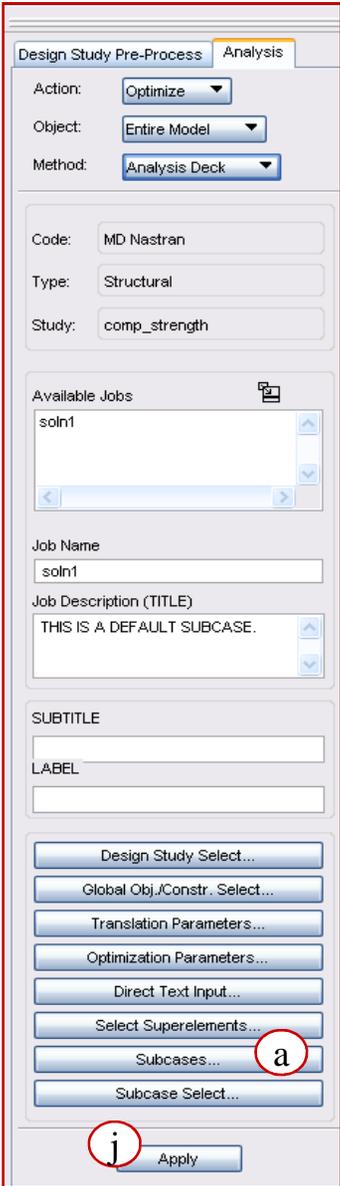
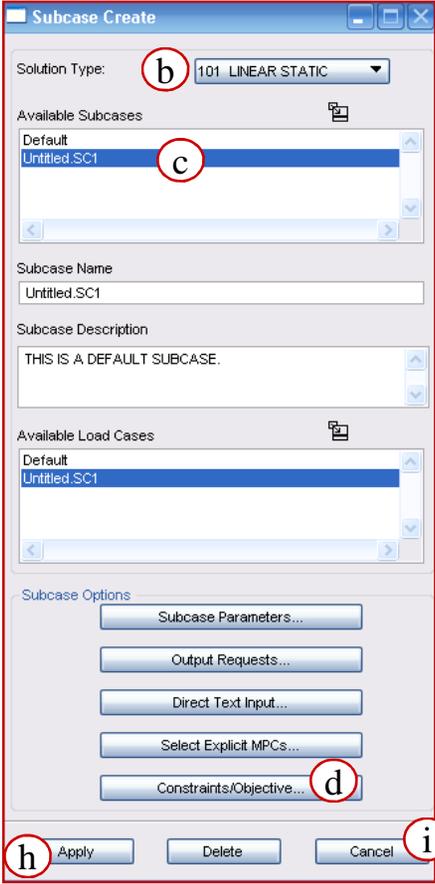
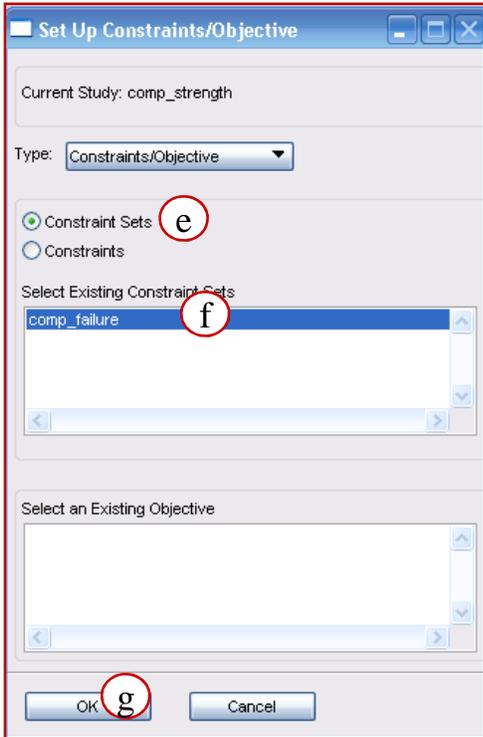
PATRAN – ANALYSIS OPTIMIZE MENUS

- a) Optimization Parameters
 - Enter **30** for *DESMAX*
 - Check **Print properties**
 - Check **Print all constraints**
 - Check **Print all responses**
 - Enter **1** for *P1*
 - Enter **1** *NASPRT*
- b) Click on **Results Output Format...**
- Check **OP2**



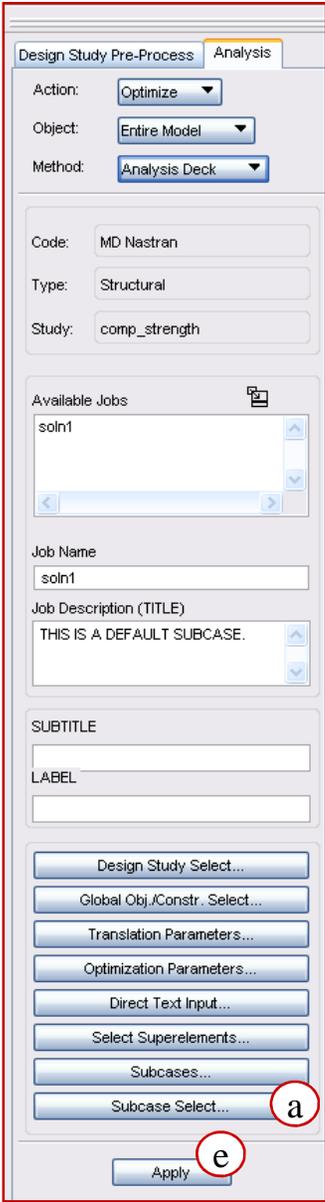
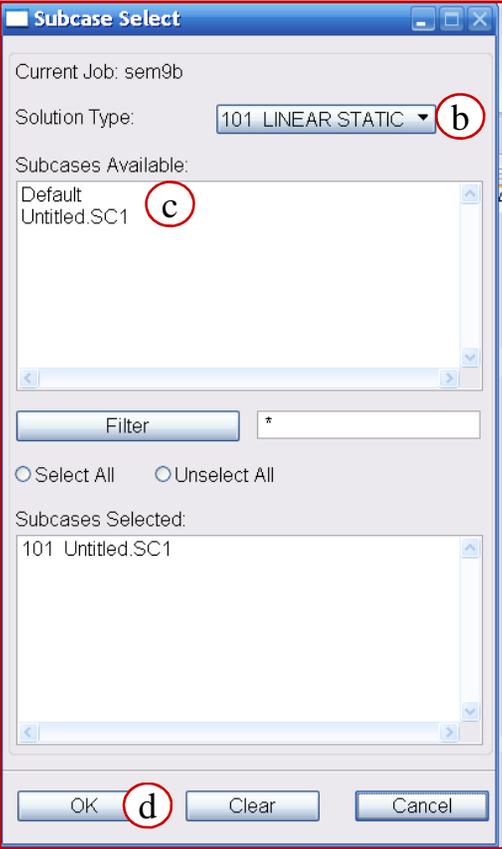
PATRAN – ANALYSIS OPTIMIZE MENUS

- a) Subcases
- b) Solution Type: Linear Static
- c) Select Untitled.SC1
- d) Select Constraints/Objective
- e) Select Constraint Sets
- f) Select **comp_failure**
- g) OK
- h) Apply
- i) Cancel
- j) Apply



PATRAN – ANALYSIS OPTIMIZE MENUS

- a) Subcase Select
- b) Solution Type: Linear Static
- c) Select the desired Subcase
- d) OK
- e) Apply



MODIFICATION NEEDED

- **Due to some Patran limitations, the following entries may need to be modified**
 - The DVPREL1 is not adjusted properly based on the modified DESVAR
 - The PCOMP Z0 field needs to be removed from any properties that are being optimized unless that is also used as a design variable
 - The CQUAD4 Zoff field should be left blank for any quad elements that are being optimized
 - Discrete design variables need to be added if needed
 - Any more complex design variable relations and responses will need to be created if needed, i.e. – DRESP2 and DVPREL2

MSC NASTRAN OPTIMIZATION OUTPUT

- Design variable history is at the end of the f06 printout.
- There were 9 variable design cycles and one discrete.
- The optimized layup is 6 0.0, 14 +45.0, 2 -45.0, 4 90.0 degree plies.
- Compare this to what you have done by hand in workshop # 4

.f06 file excerpt

```

0
                                DESIGN VARIABLE HISTORY
-----
INTERNAL | EXTERNAL | LABEL | INITIAL | 1 | 2 | 3 | 4 | 5 |
DV. ID. | DV. ID. |       |          |   |   |   |   |   |
-----
    1 |      1 | PLY0  | 1.0000E+00 | 1.5000E+00 | 2.2500E+00 | 3.3750E+00 | 4.2851E+00 | 3.5034E+00 |
    2 |      2 | PLY45 | 1.0000E+00 | 1.5000E+00 | 2.2500E+00 | 3.3750E+00 | 5.0625E+00 | 6.4457E+00 |
    3 |      3 | PLYM45 | 1.0000E+00 | 1.5000E+00 | 2.2500E+00 | 3.3750E+00 | 2.3878E+00 | 1.1939E+00 |
    4 |      4 | PLY90 | 1.0000E+00 | 1.5000E+00 | 2.2500E+00 | 3.3750E+00 | 3.4663E+00 | 2.5645E+00 |
-----
INTERNAL | EXTERNAL | LABEL |          | 6 | 7 | 8 | 9 | 9D | 10 |
DV. ID. | DV. ID. |       |          |   |   |   |   |   |   |
-----
    1 |      1 | PLY0  | 2.6599E+00 | 2.3221E+00 | 2.1683E+00 | 2.0115E+00 | 3.0000E+00 |
    2 |      2 | PLY45 | 6.6370E+00 | 6.6490E+00 | 6.7103E+00 | 6.7822E+00 | 7.0000E+00 |
    3 |      3 | PLYM45 | 1.0002E+00 | 1.0000E+00 | 1.0000E+00 | 1.0000E+00 | 1.0000E+00 |
    4 |      4 | PLY90 | 1.9655E+00 | 2.1633E+00 | 2.2166E+00 | 2.2935E+00 | 2.0000E+00 |
  
```

MSC NASTRAN OPTIMIZATION OUTPUT

- The objective and constraint history are shown in the f06 file.
- The weight went up from 2.2e-3 to 7.16e-3 due to failed constraints (failure indices larger than 1.0).
- A positive constraint value is a failed constraint so the maximum constraint went down from 18.2 to -0.08.

OBJECTIVE AND MAXIMUM CONSTRAINT HISTORY				
CYCLE NUMBER	OBJECTIVE FROM APPROXIMATE OPTIMIZATION	OBJECTIVE FROM EXACT ANALYSIS	FRACTIONAL ERROR OF APPROXIMATION	MAXIMUM VALUE OF CONSTRAINT
INITIAL		2.203200E-03		1.822872E+01
1	3.305141E-03	3.304800E-03	1.031421E-04	7.546099E+00
2	4.953912E-03	4.957200E-03	-6.633785E-04	2.798266E+00
3	7.434667E-03	7.435801E-03	-1.524274E-04	6.881182E-01
4	8.374540E-03	8.373053E-03	1.776320E-04	-1.577562E-02
5	7.551713E-03	7.550069E-03	2.177795E-04	-1.262392E-01
6	6.754433E-03	6.754201E-03	3.426514E-05	1.424181E-02
7	6.683824E-03	6.683603E-03	3.302462E-05	-3.322542E-03
8	6.662208E-03	6.662011E-03	2.963676E-05	-8.243322E-04
9	6.657893E-03	6.657666E-03	3.413249E-05	-4.804730E-04
9D	7.160546E-03	7.160400E-03	2.042032E-05	-8.000833E-02

MSC NASTRAN OPTIMIZATION OUTPUT

- The convergence logic for the last variable design cycle is also shown in the f06 file.
- Various messages about the discrete design cycle are shown towards the end of the f06 file.

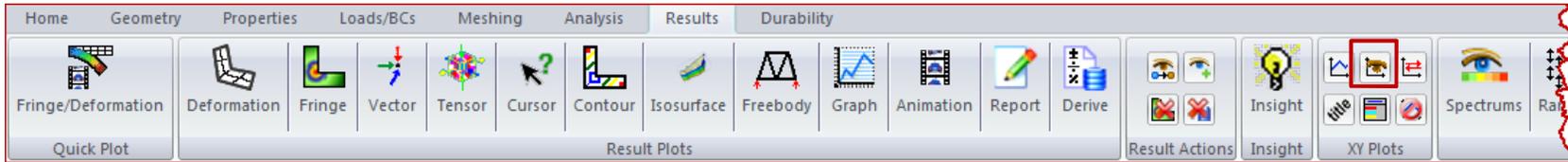
```
***** NORMAL CONVERGENCE CRITERIA SATISFIED ***** (HARD CONVERGENCE DECISION LOGIC)
*****
CONVERGENCE ACHIEVED BASED ON THE FOLLOWING CRITERIA
(HARD CONVERGENCE DECISION LOGIC)

RELATIVE CHANGE IN OBJECTIVE      0.0000E+00  MUST BE LESS THAN  1.0000E-04
OR  ABSOLUTE CHANGE IN OBJECTIVE   0.0000E+00  MUST BE LESS THAN  1.0000E-06
      --- AND ---
      MAXIMUM CONSTRAINT VALUE      2.4827E-03  MUST BE LESS THAN  5.0000E-03
      (CONVERGENCE TO A FEASIBLE DESIGN)
      --- OR ---
      MAXIMUM OF RELATIVE PROP. CHANGES  0.0000E+00  MUST BE LESS THAN  1.0000E-04
AND  MAXIMUM OF RELATIVE D.V. CHANGES  0.0000E+00  MUST BE LESS THAN  1.0000E-04
      (CONVERGENCE TO A BEST COMPROMISE INFEASIBLE DESIGN)
*****
.
.
***** A SOFT FEASIBLE DISCRETE SOLUTION FOUND (SOFT FEASIBILITY DISCRETE SOLUTION CHECK LOGIC) *****
      MAXIMUM CONSTRAINT VALUE      : -6.3885E-02  MUST BE LESS THAN  5.0000E-03
*****
.
.
***** A HARD FEASIBLE DISCRETE SOLUTION FOUND (HARD FEASIBILITY DISCRETE SOLUTION CHECK LOGIC) *****
      MAXIMUM CONSTRAINT VALUE      : -8.0009E-02  MUST BE LESS THAN  5.0000E-03
*****
.
.
*** USER INFORMATION MESSAGE 6464 (DOM12E)
      RUN TERMINATED DUE TO HARD CONVERGENCE TO AN OPTIMUM AT CYCLE NUMBER =      9.
      AND HARD FEASIBLE DISCRETE DESIGN OBTAINED
```

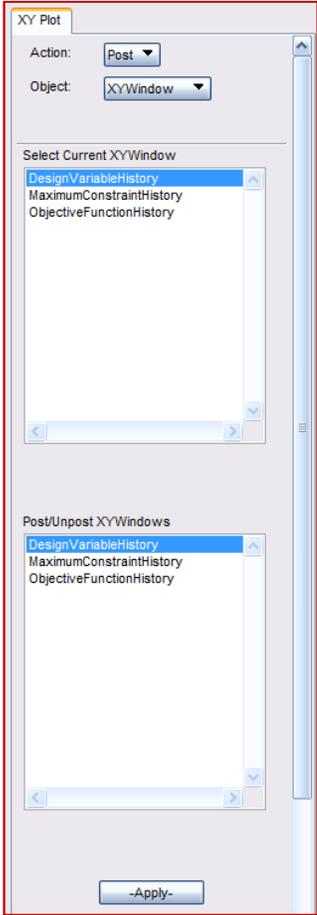
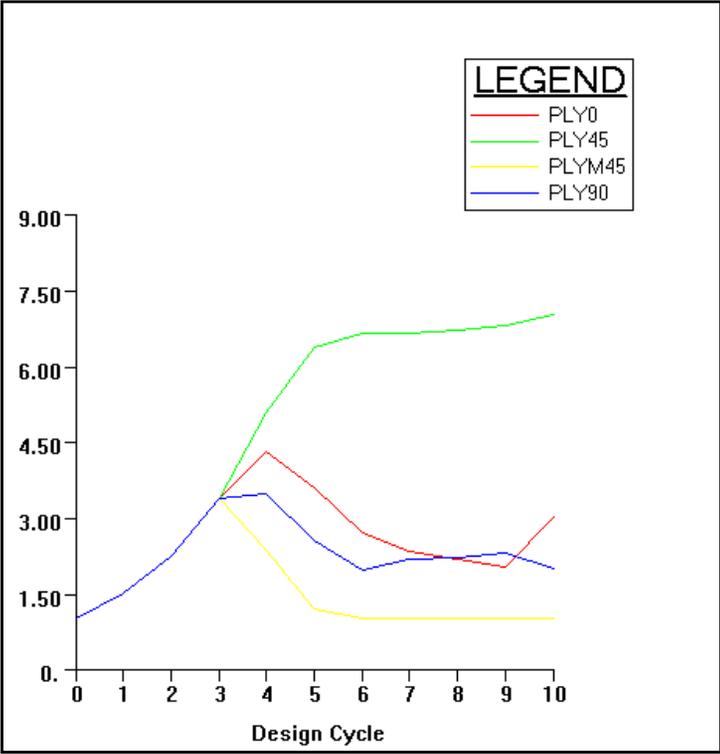
PLOTTING OPTIMIZATION OUTPUT IN PATRAN

- **If the op2 result file is used (param,post,-1) for optimization post-processing in Patran, then the following plots are available:**
 - XY plots
 - Design variable value versus design cycle
 - Objective value versus design cycle
 - Constraint value versus design cycle
 - Fringe plot
 - Failure Indices
 - To request failure index fringe results for all design cycles add PARAM,NASPRT,1.
 - Otherwise, only the final design results are available.

PLOTTING OPTIMIZATION OUTPUT IN PATRAN

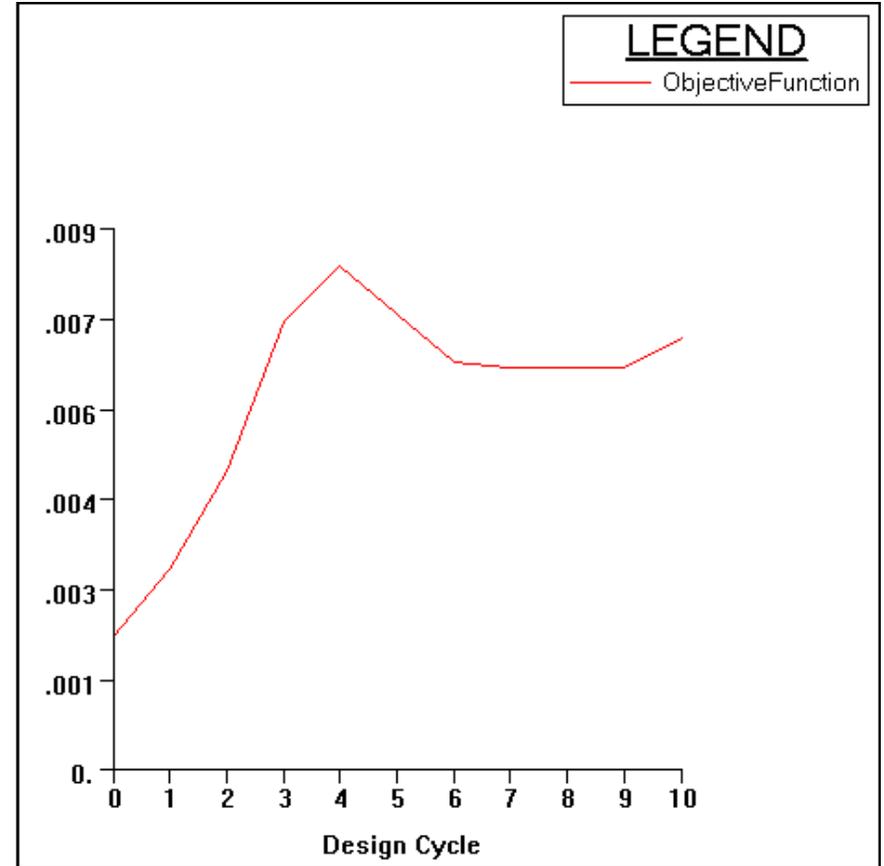
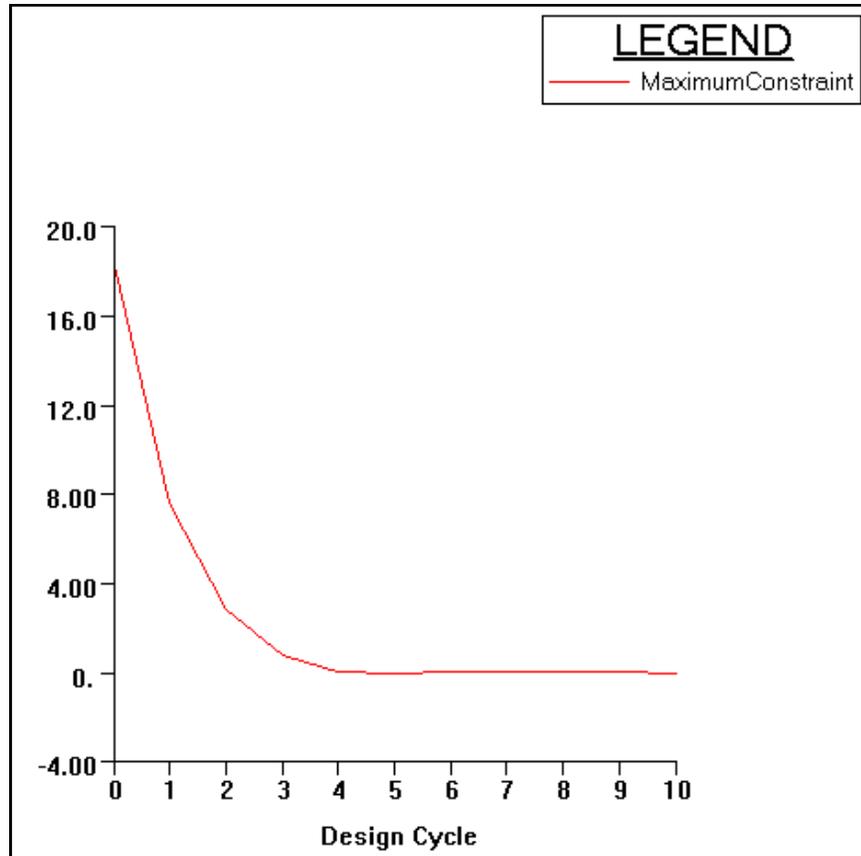


- XY Plot
- Post/XY Window
- Select Current XYWindow
- Design Variable History**
- Post/Unpost XY Windows
- Design Variable History**

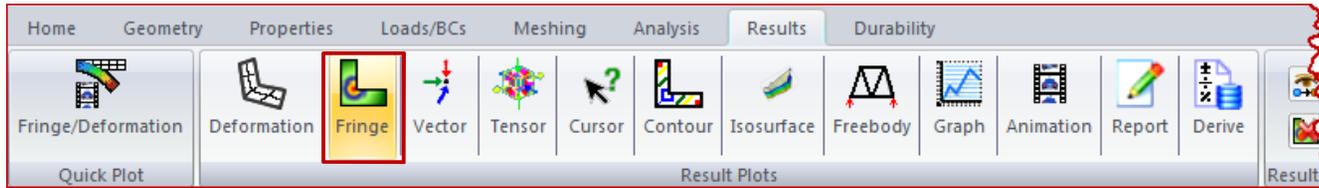


PLOTTING OPTIMIZATION OUTPUT IN PATRAN

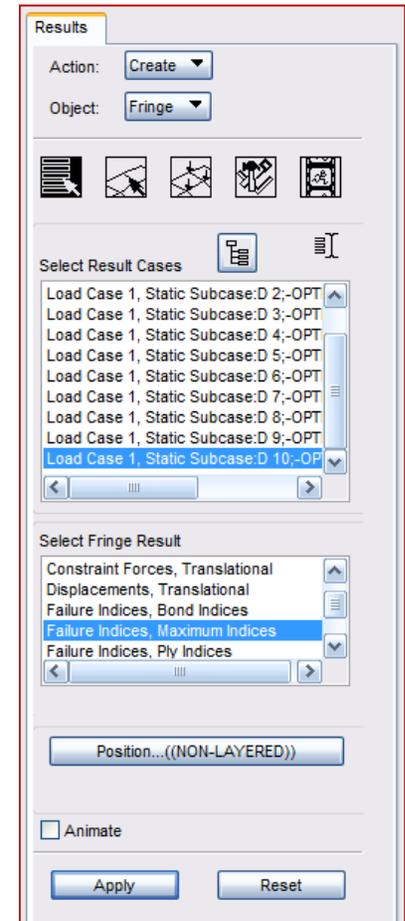
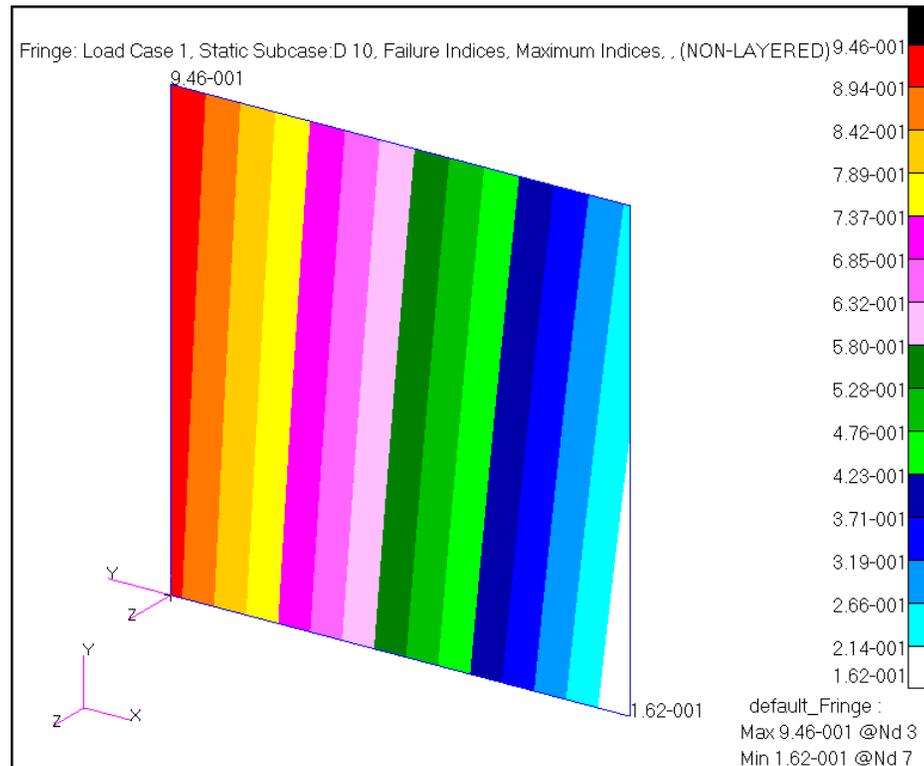
Repeat for **Maximum Constraint History** and **Objective Function History**



PLOTTING OPTIMIZATION OUTPUT IN PATRAN

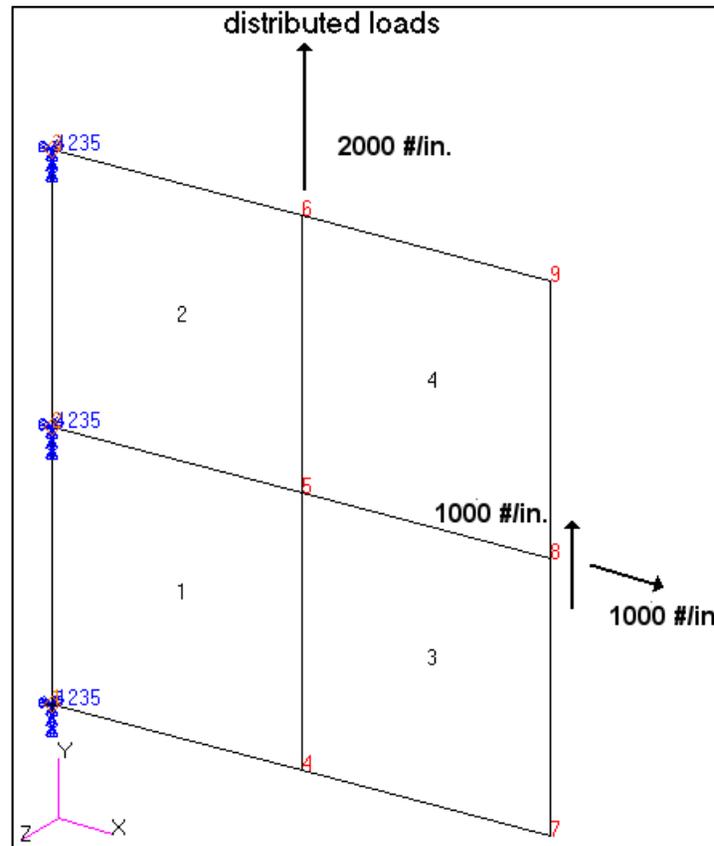


Results:
 Create/ Fringe
 Select Result Case
 "StaticSubcaseD10"
 Select Fringe
 Result "Maximum
 Indices"
 Apply



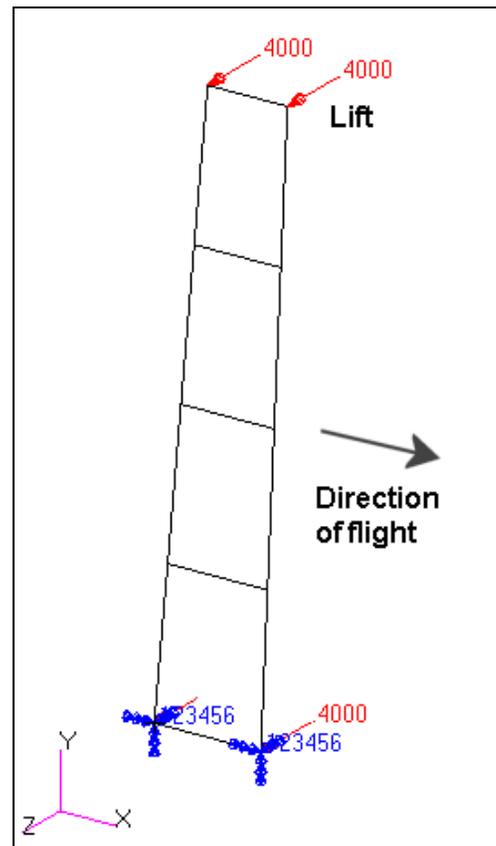
EXERCISE

- Perform WS11 “Ply Direction Tailoring Strength Using the Optimizer”.



EXERCISE

- Perform WS12 “Ply Direction Tailoring for Stiffness Using the Optimizer ”.



CASE STUDY

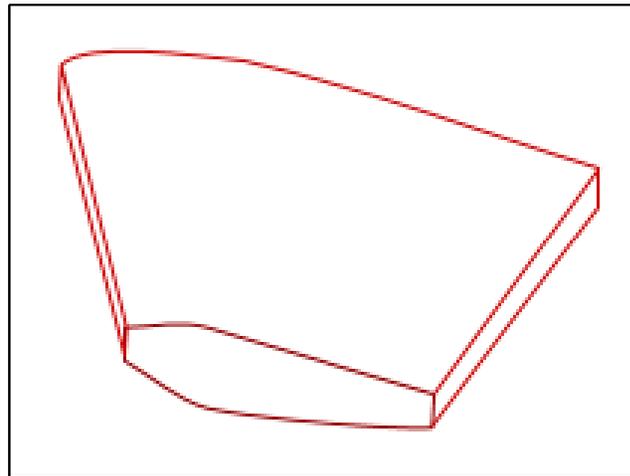
FIXED TOTAL THICKNESS

DESCRIPTION

- **A preliminary analysis of a thin composite wing model.**
- **The wing is a 3D solid varying in thickness both along the cord and the wingspan.**
- **The structure is aluminum for the upper and lower skins over a foam core structure with separate leading and trailing edges bonded to it.**
- **The Outer Mold Line (OML) of the wing is fixed so all of the structure represented by the FEM must fit inside the CAD volume**

INITIAL MODEL

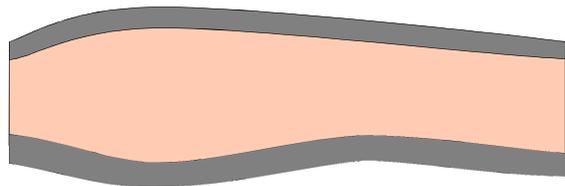
- There are many considerations depending on the maturity of the design.
- The first task is to assess a feasible modeling method given the analysis to be performed and develop an initial working structure.



FEM APPROACH

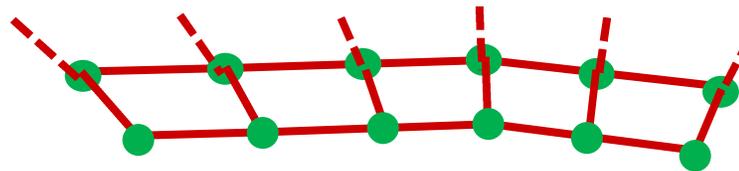
- **All Shell Method**

- With this approach the skin and core for the entire wing can be represented as all shell FEM and PCOMP.
- This setup is difficult. Given the variable thickness of the wing each element could have its own core thickness to achieve the OML.
- A programming method could be applied to automate this thickness definition from the CAD data, but there's a simpler approach that actually will give a higher fidelity result.



Composite skin over honeycomb core

Wing Cross Sections



All CQUAD FEM with PCOMP



Unique 3 ply PCOMP for every element

FEM APPROACH

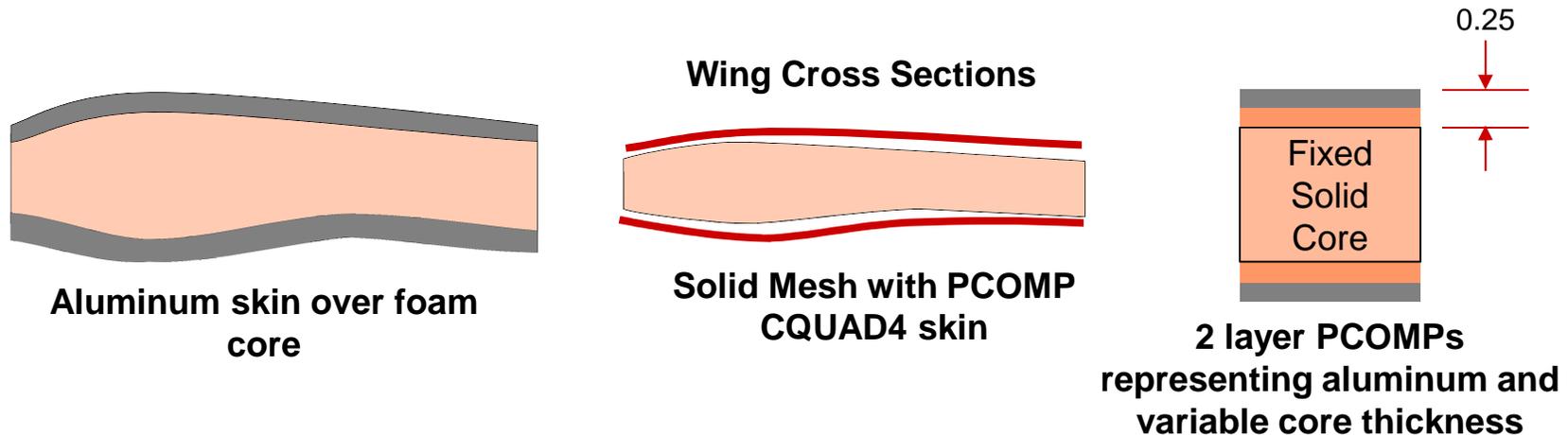
- **Skinned Solid Method**

- In this method the core (or most of the core) will be solid elements and the skin will be shell elements on the top and bottom of the solids.
- The problem is that if the skin becomes thicker, the core needs to become thinner to maintain the aerodynamic shape of the wing
- There are two (at least) possible approaches here:
 1. Move the grid points of the solid core inward in response to the skin thickness design variables. This can be done with the DVGRID card.
 2. Put some of the core thickness into the skin shell elements as an extra layer. This layer can then be decreased in thickness to compensate for any increase in skin thickness.
- For a simple preliminary design, we'll use option 2.
- The solid core CAD must be shrunk or scaled down some amount from the OML to make room for the skins but this is fairly easy to do on either the CAD or Pre-Processor. An offset of $t/2$ is used to get the composite between the core and OML.

FEM APPROACH

- **Skinned Solid Method (Cont.)**

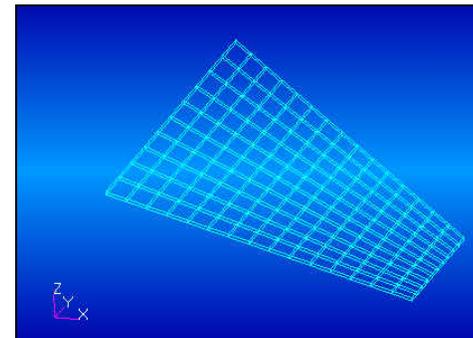
- Given the scaled down solid represents the cord and span variable thickness the shell or 2 ply PCOMP total thickness can be constant to start the optimization.
- The key optimization method to take advantage of is the sum of the combination of composite and core in the skin must stay constant.
- The one disadvantage to this method is to get the total core thickness for a location will come from the solid element plus the top and bottom optimized PCOMP.



COMPOSITE OPTIMIZATION TO HOLD OML (OUTER MOLD LINE)

- Upper and lower skin thicknesses are variables
- The total thickness of upper skin and some core should be 0.25
- The total thickness of the lower skin and some core should be 0.25
- Define a DVPREL1 for each that defines the thickness of the core layer as $0.25 + \text{skin-thick} * (-1.)$ (highlighted in red below)

```
$ ...DESIGN VARIABLE DEFINITION
$ Al_core_T2
DESVAR 10 Upper .125 101
DESVAR 20 Lower .125 101
$ Composite Constraint - round to 0.01 thickness
DDVAL 101 .01 thru .49 by .01
$ ...DEFINITION OF DESIGN VARIABLE TO ANALYSIS MODEL PARAMETER RELATIONS
$ The T1 is always core and T2 is aluminum or the strong stuff
DVPREL1 1 PCOMP 1 T2
10 1.
DVPREL1 2 PCOMP 1 T1 0.25
10 -1.
DVPREL1 3 PCOMP 2 T2
20 1.
DVPREL1 4 PCOMP 2 T1 0.25
20 -1.
```



SECTION 11

PRACTICAL USAGE GUIDELINES

STRENGTH RATIO OUTPUT FOR LAMINATED COMPOSITES

- **Failure Index (FI)**
 - A commonly used failure indicator in composite
 - Indicates whether a structure has failed.
 - Does not generally tell you how much of a safety margin you have
 - $FI=0.9$ indicates no failure, but does not imply an 11% margin
- **Most composite failure criteria are nonlinear**
 - They cannot be used for scaling load factor
 - Use strength ratio as they are linear
- **Strength Ratio (SR)**
 - Better indicator, similar to margin of safety
 - Is equal to Allowable stress / Actual stress

STRENGTH RATIO OUTPUT FOR LAMINATED COMPOSITES

- Challenge is most failure indices are nonlinear
- Use TSAI-WU failure criterion as an example:

$$FI = F11\sigma_1^2 + F22\sigma_2^2 + F12(\sigma_1\sigma_2) + F66\sigma_{12}^2 + F1\sigma_1 + F2\sigma_2$$

where F_{ij} are stress allowables, recast in a more convenient form

- Substitute the (actual stress) with (SR * actual stress) and set $FI=1.0$ and equation becomes

$$[F11\sigma_1^2 + F22\sigma_2^2 + F12(\sigma_1\sigma_2) + F66\sigma_{12}^2] * SR^2 + [F1\sigma_1 + F2\sigma_2] * SR - 1 = 0$$

STRENGTH RATIO OUTPUT FOR LAMINATED COMPOSITES

- **Solve for roots (SRs) for the quadratic equation**
- **Loads can now be scaled based on the SR to yield FI=1.0**
- **Several methods yield somewhat simpler relationships**
 - Maximum strain (stress)
 - $SR = 1 / (FI)$
 - Transverse shear stress
 - $SR = 1 / (FI)$
 - Hill Failure Criteria
 - $SR = 1 / \sqrt{FI}$
- **To request SR output, use the following parameter**
 - PARAM,SRCOMPS,YES
- **Supports plotting of failure indices with op2 (param,post,-1)**

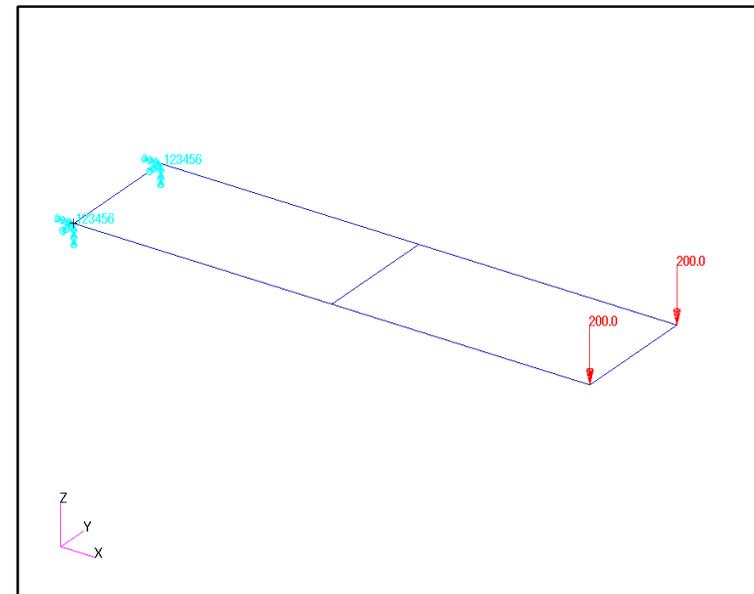
STRENGTH RATIO OUTPUT FOR LAMINATED COMPOSITES

- **Example:**

- Simple plate model subjected to vertical loads. Want to find out load level to yield FI=1.0

```

$
SOL 101
CEND
TITLE = 2 PLATE - QUAD4 COMPOSITE
SUBTITLE =
$
STRESS = ALL
SPC = 1
$
subcase 1
label =
LOAD = 1
$
.
BEGIN BULK
$
PARAM      SRCOMPS YES
GRID       1          0.    0.    0.
.
.
$
CQUAD4    10         1     1     2     5     4
CQUAD4    20         1     2     3     6     5
$
PCOMP     1          .1    0.    1.0+4  TSAI
+         1          .1    0.    YES   1     .1    0.    YES  +
$
MAT8      1          3.+7   7.5+5 .25   3.75+5
+         1          1.5+5  1.+5   6.+3   1.7+4   .1
$
FORCE     1          3          200.   0.    0.    -1.
FORCE     1          6          200.   0.    0.    -1.
$
SPC1      1          123456  1     4
$
ENDDATA
    
```



STRENGTH RATIO OUTPUT FOR LAMINATED COMPOSITES

- **Example:**
 - Based on FI, one would increase load by 23.396%
 - Based on SR, one would increase load by 13.319%

SUBCASE 1						
FAILURE INDICES FOR LAYERED COMPOSITE ELEMENTS (QUAD 4)						
ELEMENT ID	FAILURE THEORY	PLY ID	FP=FAILURE INDEX FOR PLY (DIRECT STRESSES/STRAINS)	FB=FAILURE INDEX FOR BONDING (INTER-LAMINAR STRESSES)	FAILURE INDEX FOR ELEMENT MAX OF FP, FB FOR ALL PLYS	FLAG
10	TSAI-WU	1	0.8104			
		2	0.2711	0.0600	0.8104	
20	TSAI-WU	1	0.1702			
		2	-0.0500	0.0600	0.1702	

SUBCASE 1						
STRENGTH RATIOS FOR LAYERED COMPOSITE ELEMENTS (QUAD 4)						
ELEMENT ID	FAILURE THEORY	PLY ID	SRP-STRENGTH RATIO FOR PLY (DIRECT STRESSES/STRAINS)	SRB-STRENGTH RATIO FOR BONDING (INTER-LAMINAR STRESSES)	STRENGTH RATIO FOR ELEMENT MIN OF SRP, SRB FOR ALL PLYS	FLAG
10	TSAI-WU	1	1.133186E+00			
		2	1.631857E+00	1.666667E+01	1.133186E+00	
20	TSAI-WU	1	3.264916E+00			
		2	5.097456E+00	1.666667E+01	3.264916E+00	

STRENGTH RATIO OUTPUT FOR LAMINATED COMPOSITES

- **Example:**

- 2 new subcases were run

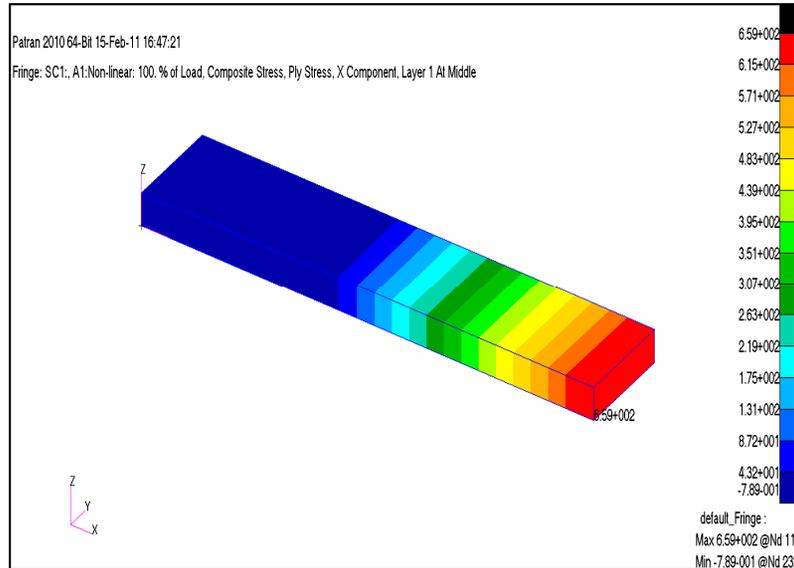
- Increasing load by 23.396%--the structure would have failed with FI=1.15
- Increasing load by 13.319%--"optimum" design with FI=1.0

```

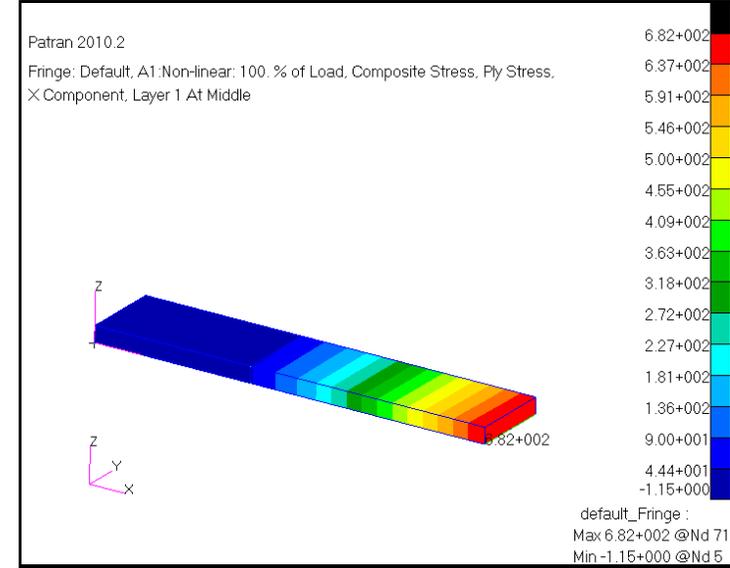
0      INCREASE LOAD BY 23.396%
                                           SUBCASE 2
      FAILURE INDICES FOR LAYERED COMPOSITE ELEMENTS (QUAD4)
ELEMENT FAILURE PLY  FP=FAILURE INDEX FOR PLY  FB=FAILURE INDEX FOR BONDING  FAILURE INDEX FOR ELEMENT  FLAG
  ID    THEORY  ID  (DIRECT STRESSES/STRAINS)  (INTER-LAMINAR STRESSES)  MAX OF FP,FB FOR ALL PLYS
    10   TSAI-WU   1      1.1562                        0.0740                      1.1562      ***
                                           0.4906
    20   TSAI-WU   1      0.2274                        0.0740                      0.2274
                                           2      -0.0444
0      INCREASE LOAD BY 13.319 % BASE
                                           SUBCASE 3
      FAILURE INDICES FOR LAYERED COMPOSITE ELEMENTS (QUAD4)
ELEMENT FAILURE PLY  FP=FAILURE INDEX FOR PLY  FB=FAILURE INDEX FOR BONDING  FAILURE INDEX FOR ELEMENT  FLAG
  ID    THEORY  ID  (DIRECT STRESSES/STRAINS)  (INTER-LAMINAR STRESSES)  MAX OF FP,FB FOR ALL PLYS
    10   TSAI-WU   1      1.0000                        0.0680                      1.0000      ***
                                           2      0.3888
    20   TSAI-WU   1      0.2019                        0.0680                      0.2019
                                           2      -0.0476
  
```

STRESS CALCULATION, SOLIDS

- **Stresses are obtained at the center of the element and at the center of the thickness for solid shell**
 - May want to add thin layer at top and bottom
 - May want to bias mesh (see workshop 3)



Stresses at thin layer 1
with even mesh

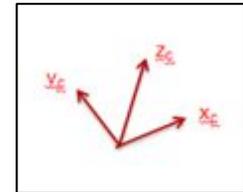


Stresses at thin layer 1
with biased mesh

COORDINATE SYSTEM FOR SOLID COMPOSITE

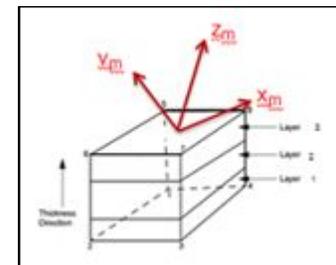
- **Stresses are calculated at the 4 Gauss points when INTi=L or Q (solid composite):**

- The coordinate system referenced by the CORDM field explicitly defines the local coordinate system. There is no projection in this case
- Positive θ direction is defined using the right hand rule about the local z-axis



- **Stresses are calculated at the center when INTi=ASTN (solid shell)**

- The x-axis referenced by the CORDM is projected onto the element to create the local x-axis
- The element z-axis points from the element Face1 towards the opposite face
- Only DIRECT=+1 or -1 is allowed. +1 indicates the local z-axis follows the element positive z direction.
- The local y-axis is obtained by the cross product (local-z x local-x)
- Positive θ direction is defined using the right hand rule about the local z-axis
- This is very similar to the way MCID is defined for the CQUAD4

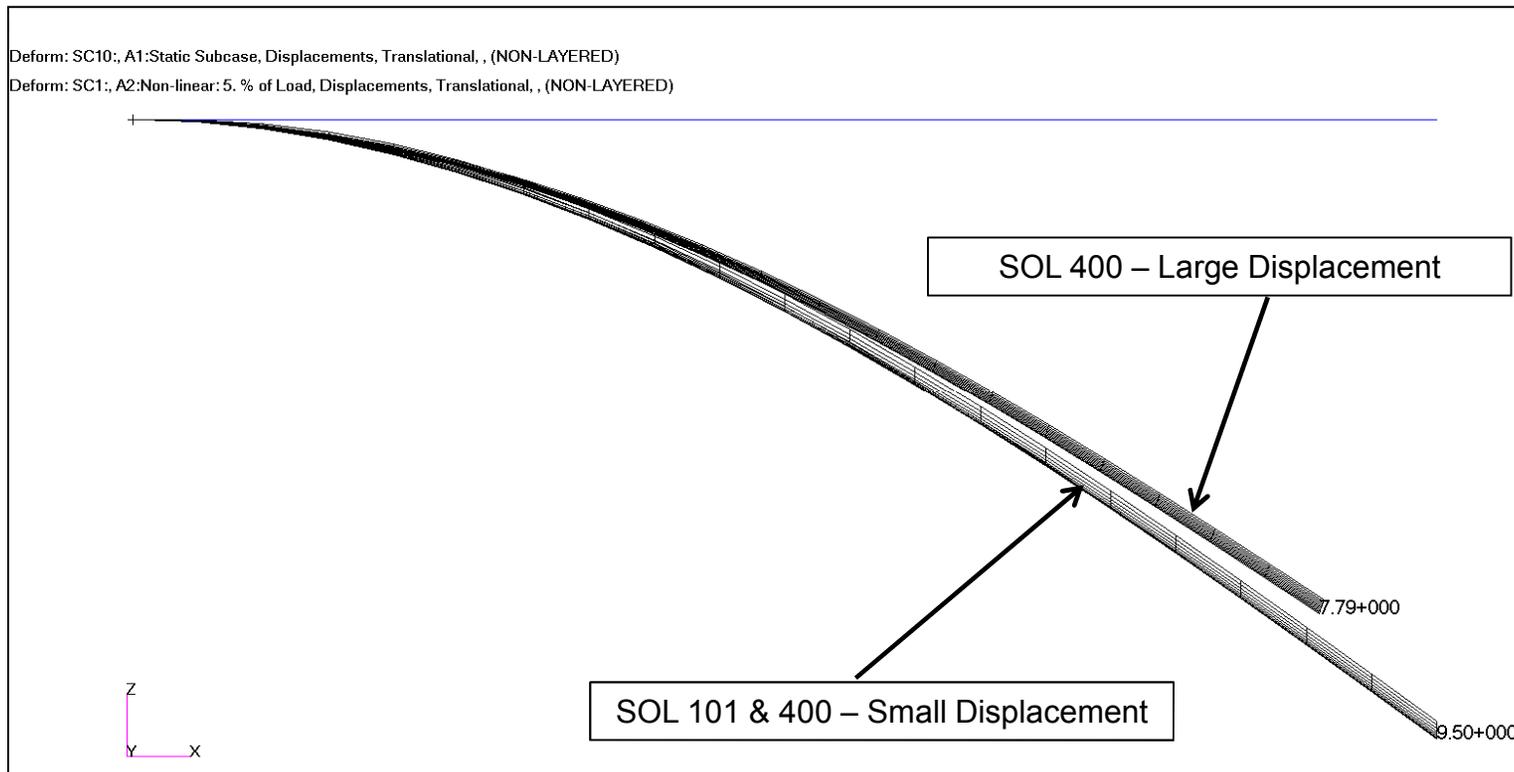


NONLINEAR ANALYSIS

- **Arc length can be a more robust stepping approach for highly unstable structures**
 - Cannot be used with contact.
 - If necessary, this could be circumvented by doing a run with fixed stepping and adding the Case Control command
NLOPRM MPCPCH=BEGN
 - The above command generates equivalent MPCs
 - These MPCs can then be imported and rerun in place of contact.
- **Default stepping for NLSTEP is more likely to converge than NLPARM**
 - Default convergence tolerances lower for NLSTEP than NLPARM

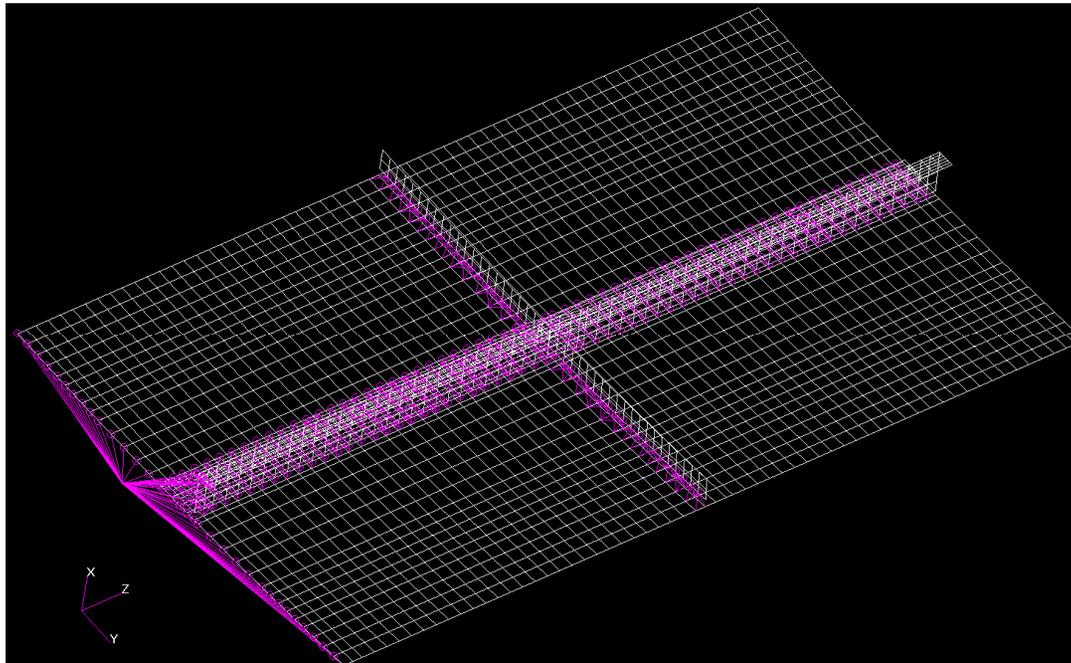
PLOTTING TIPS FOR NONLINEAR ANALYSIS

- **When performing nonlinear analysis, it's best to plot the displacements using true scale**
 - Especially true for large displacement analysis
 - Contact analysis



TIPS FOR GLUED CONTACT

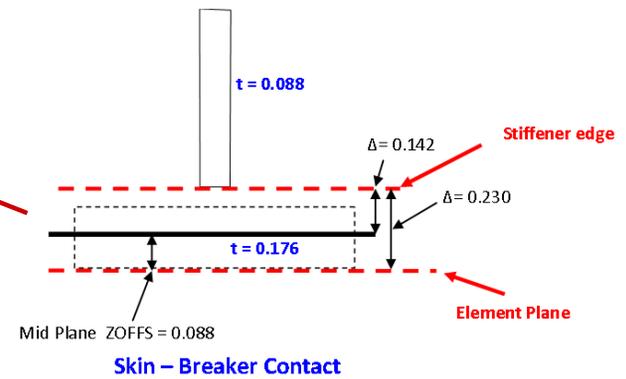
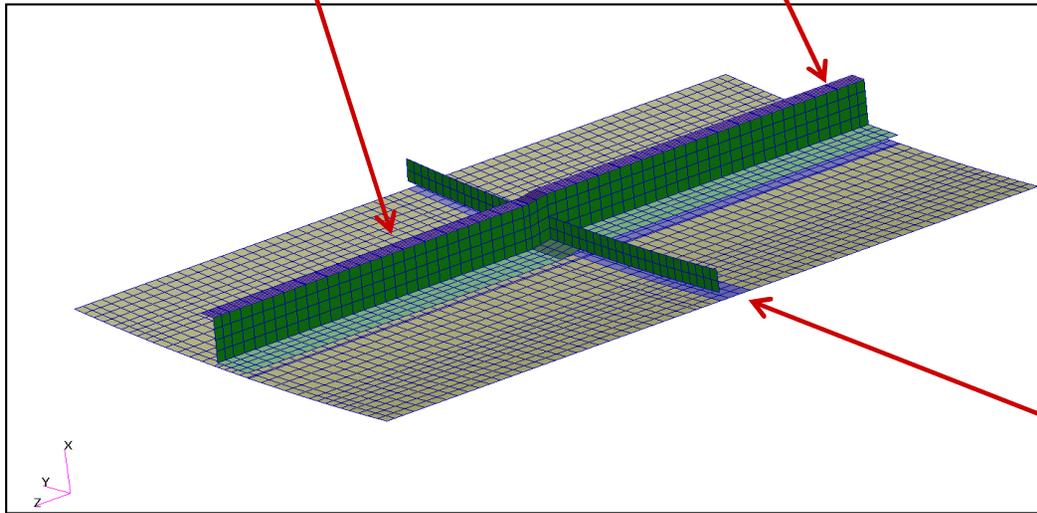
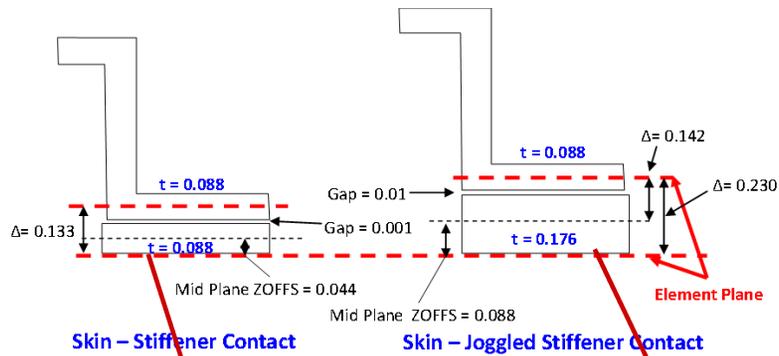
- **Verify proper glued contacts by writing out appropriate MPCs**
 - Use the following Case Control command
 - NLOPRM MPCPCH=BEGN
 - Plot the MPCs for verification



- **When dealing with edge/face contact, make sure to turn off thickness consideration and search from edge to face.**

TIPS FOR GLUED CONTACT

- Sketch cross sections to calculate and evaluate proper gap sizes



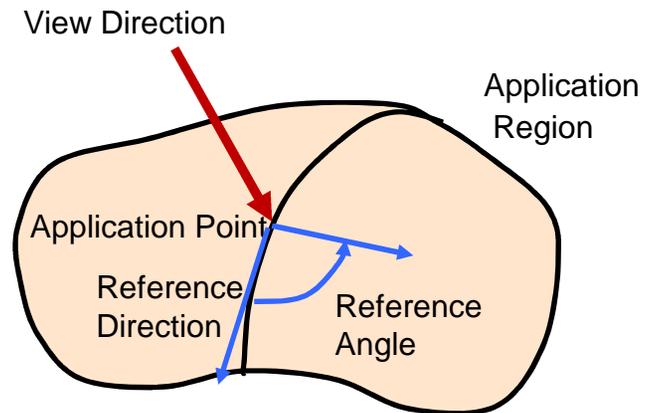
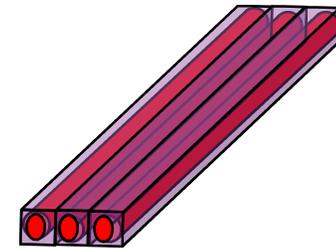
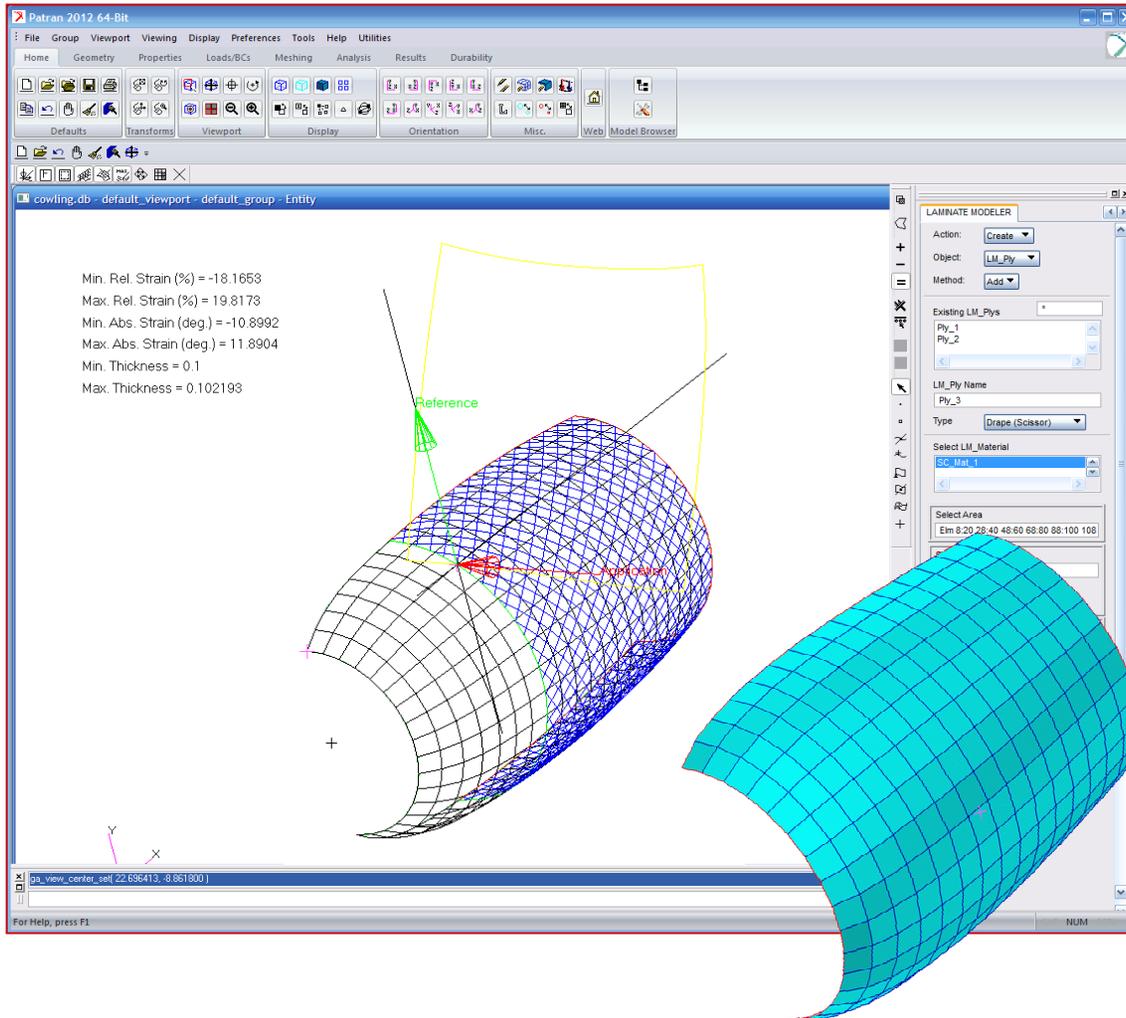
USE OPTIMIZATION

- **Optimization is a powerful and quick tool to come up with a good feasible design**
- **Typical Optimization Statement**
 - Objective function
 - Most important goal that you try to accomplish (e.g. minimize the weight)
 - Design Variables
 - Areas that you can modify (e.g. thickness of a plate)
 - Satisfy design constraints
 - Limits on specified values (e.g. stresses must be below certain values)
- **We do this already—brute force method**

SECTION 12

LAMINATE MODELER

COMPOSITE MATERIALS WITH PATRAN & LAMINATE MODELER

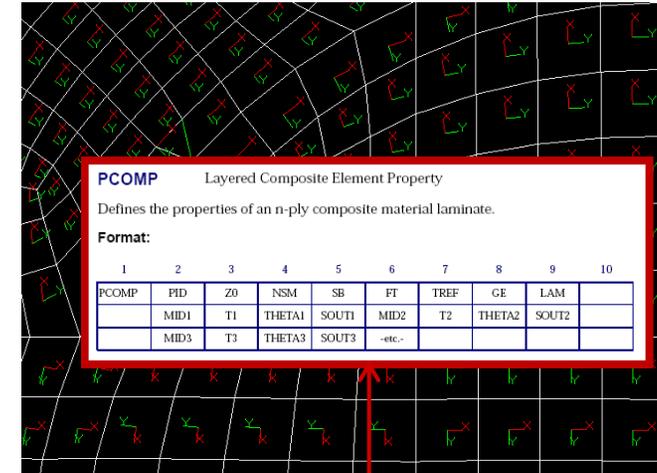


OVERVIEW

- **Modeling Composite Materials With Patran**
 - Patran Materials Application
 - When only a few stack definitions are needed and can be rapidly generated for simple geometric parts
 - Augmented with the Laminate Builder Tool
 - Why Use Patran Laminate Modeler?
 - Useful for large composite parts with many zones of complex curvature
 - Draping algorithms
 - Composite data management

PATRAN COMPOSITES

- Doesn't Patran support composites?
 - Yes. Basic input and verification capabilities
 - Essentially patterned after MSC Nastran
 - Works well for simple structures with limited ply drop-off



PCOMP Layered Composite Element Property

Defines the properties of an n-ply composite material laminate.

Format:

	1	2	3	4	5	6	7	8	9	10
PCOMP	PID	Z0	NSM	SE	FT	TREF	GE	LAM		
	MID1	T1	THETA1	SOUT1	MID2	T2	THETA2	SOUT2		
	MID3	T3	THETA3	SOUT3	-etc.-					

Input Options

Constitutive Model: Linear Elastic

Property Name	Value
Elastic Modulus 11 =	10000000
Elastic Modulus 22 =	2000000
Poisson Ratio 12 =	.25
Shear Modulus 12 =	500000
Shear Modulus 23 =	500000
Shear Modulus 13 =	500000
Density =	
Thermal Expan. Coeff 11 =	
Thermal Expan. Coeff 22 =	
Structural Damping Coeff =	
Reference Temperature =	

Temperature Dep/Model Variable Fields:

Materials

Action: Create
Object: 2d Orthotropic
Method: Manual Input

Existing Materials: my_ply_matl

Filter: *

Material Name: my_ply_matl

Description: Date: 07-Mar-12 18:14:08 Time:

Materials

Action: Create
Object: Composite
Method: Laminated

Existing Materials: my_ply_matl

Layup1: my_ply_matl

Laminated Composites: layup1

Material Name: layup1

Material Description:

-Apply-

Show Laminate Properties

Laminated Composite

Stacking Sequence Convention: Total

Stacking Sequence Definition

Input Data: Auto Highlight

Import/Export...

	Material Name	Thickness	Orientation	Global Ply ID
1	my_ply_matl	1.000000E-3	0.000000E+0	
2	my_ply_matl	1.000000E-3	-5.280000E+1	
3	my_ply_matl	1.000000E-3	-6.249000E+1	

Set Thickness =

Total Thickness in Stacking Sequence =

Delete Selected Row

Show Laminate Properties

Composite Material Properties

Membrane, Bending, and Coupling Matrices

		Membrane	Bending			
	4.22E+004	2.55E+004	8.89E+003	5.49E+000	3.50E+000	4.79E+000
Membrane	2.55E+004	4.40E+004	1.29E+003	3.50E+000	-1.2E+001	-1.0E-001
	8.89E+003	1.29E+003	9.84E+003	4.79E+000	-1.0E-001	3.50E+000
	5.49E+000	3.50E+000	4.79E+000	3.19E+002	1.71E+002	5.53E-003
Bending	3.50E+000	-1.2E+001	-1.0E-001	1.71E+002	3.69E+002	-5.3E-007
	4.79E+000	-1.0E-001	3.50E+000	5.53E-003	-5.3E-007	5.32E-003

High Precision Value

Composite Property Display Options

A, B, and D Matrices 3D Flexibility Matrix Thermal K_{ij}, N_i, and M_i

3D Elasticity Matrix E's, N_U's, G's, and Q_{ij}'s CTE's, CME's and Others

Cancel

MAT8 Shell Element Orthotropic Material Property Definition

Defines the material property for an orthotropic material for isoparametric shell elements.

Format:

	1	2	3	4	5	6	7	8	9	10
MAT8	MID	E1	E2	NU12	G12	G13	G23	RHO		
	A1	A2	TREF	Xt	Xc	Yt	Yc	S		
	GE	F12	STRN							

PATRAN COMPOSITES: LAMINATE BUILDER TOOL

File Group Viewport Viewing Display Preferences Tools Help Utilities

Utilities

- 1-Element
- FEM-General
- Loads/BCs
- Materials
- Properties
- Fjelds

Material Session File Library ...
 Laminate Builder Tool ...
 Rename Materials ...
 Renumber Materials ...

Laminate Builder v5.0

Action: Add
 Object: Layer

Stack: Total
 Plys: 6 / 6 Thickness: 3.0000E-2 / 3.0000E-2

	Orientation	Thickness	Material
1	0.00	.005	m55j_tape
2	60.00	.005	m55j_tape
3	-60.00	.005	m55j_tape
4	-60.00	.005	m55j_tape
5	60.00	.005	m55j_tape
6	0.00	.005	m55j_tape

Thickness: .005

Apply Cancel Load ... Save ... Clear Stats ... Calc ...

Laminate Calcs v1.1

Show: Laminate Stiffness

Normalize

- No Calculation
- Laminate Stiffness
- Laminate Compliance
- Effective Properties
- Ply Stiffness
- Ply Compliance
- Ply CTE
- Ply Stresses
- Ply Strains
- Ply Principal Stresses
- Ply Principal Strains
- Max Stress
- Max Strain
- Tsai-Hill
- Tsai-Wu

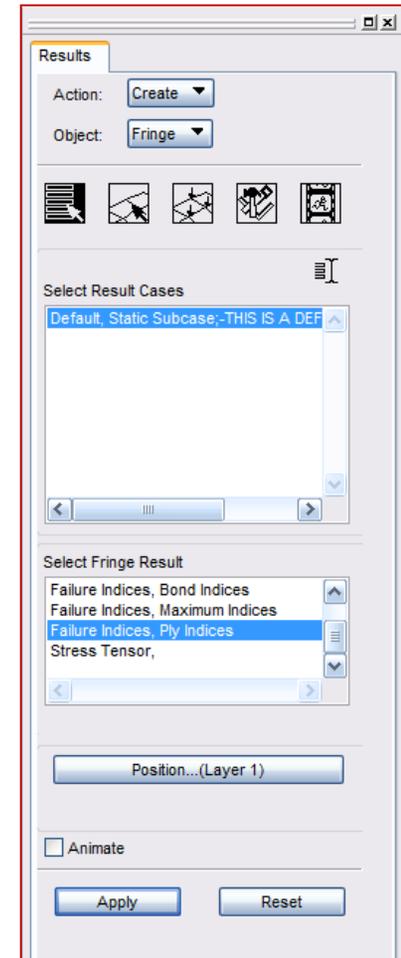
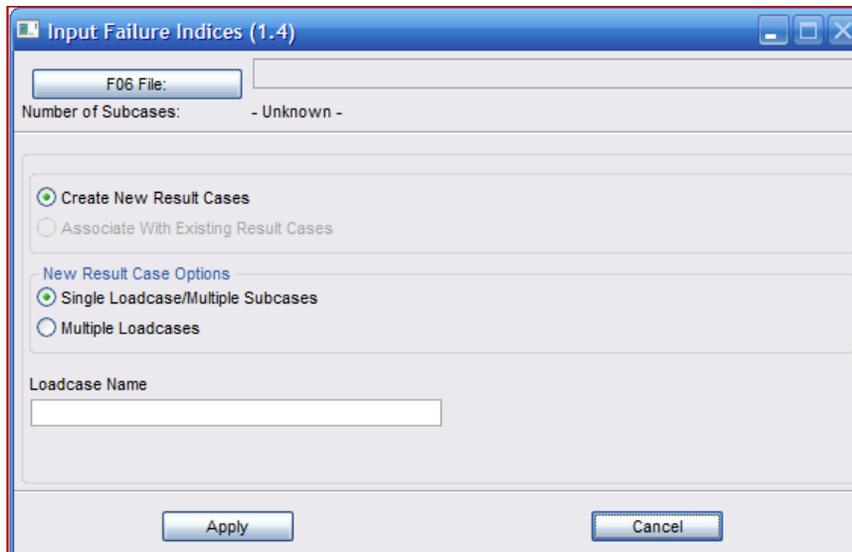
Membrane Stiffness, [A]			Coupling Stiffness, [B]		
E+5	5.621202E+4	0.000000E+0	-6.103516E-5	3.051758E-5	-1.907349E-6
E+4	1.412563E+5	0.000000E+0	3.051758E-5	7.629395E-5	0.000000E+0
E+0	0.000000E+0	4.252216E+4	-1.907349E-6	0.000000E+0	3.767014E-5

Bending Stiffness, [D]		
1.736677E+1	2.506645E+0	2.848839E-1
2.506645E+0	7.240189E+0	1.469091E+0
2.848839E-1	1.469091E+0	1.479905E+0

Close Select ... Plot ... Max/Min ... Dump ...

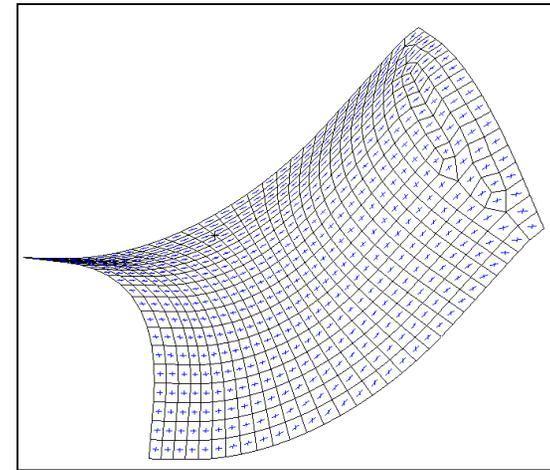
PATRAN COMPOSITES: FAILURE INDICES

- Support for post processing the composite failure indices prior to MSC Nastran 2004 required reading them in from the .f06
 - Utilities > Results > Read MSC.N Failure Indices
- Starting with MSC Nastran 2004 failure indices are supported via the .op2 results file
- A change in allowables or failure criteria etc... requires rerunning the MSC Nastran job

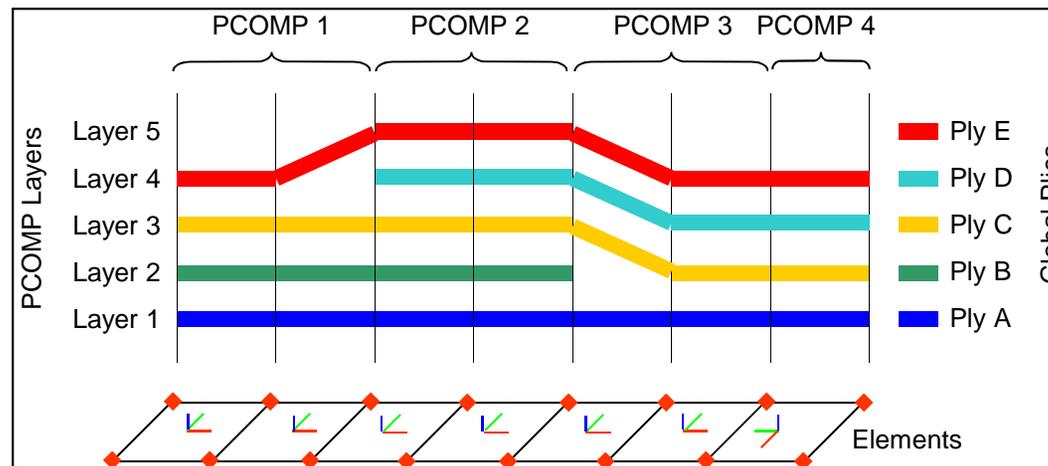


WHY LAMINATE MODELER?

- **Real structures are not flat plates ...**
 - Complex curvature requires draping
 - Manufacturing: Flat pattern development
 - Analysis: Accurate orientation information
 - Complex layups require ply management
 - Countless plies
 - Ply drop-off
 - Element CID misalignment
 - *Results in countless PCOMPs and changes become ... costly ...*

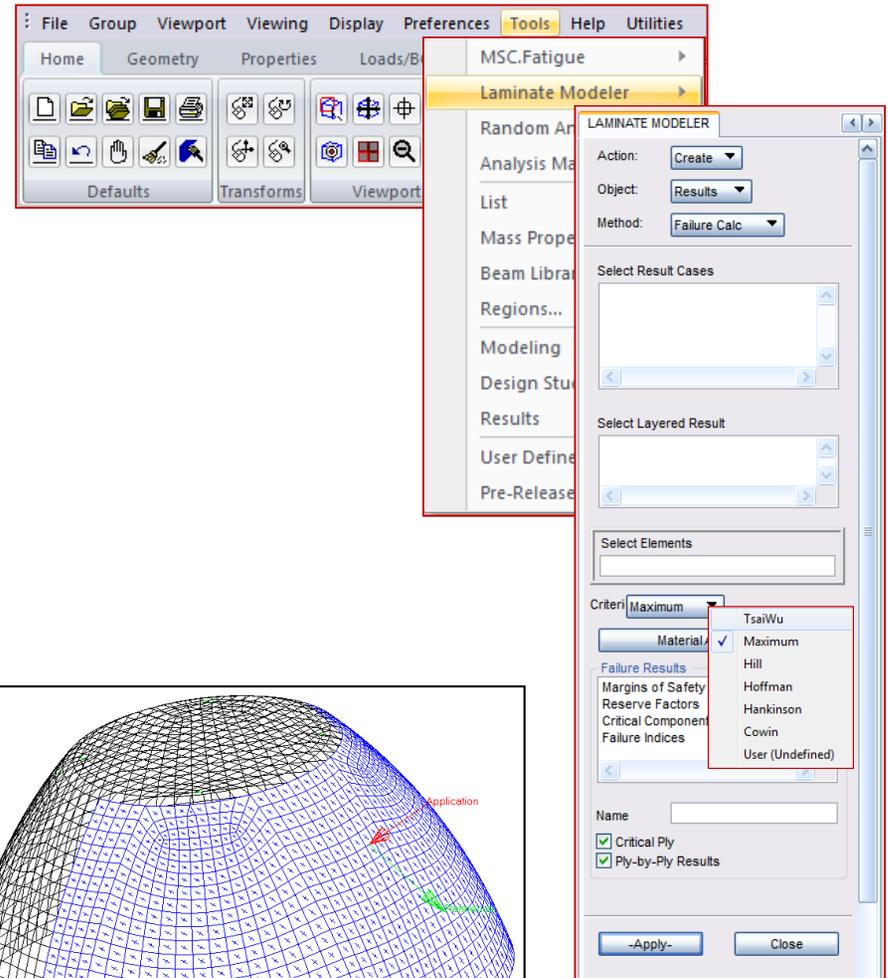


Fiber orientations



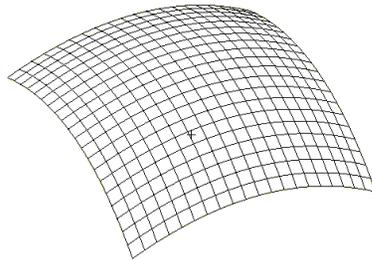
WHAT IS LAMINATE MODELER?

- **Patran vertical application**
 - An add on to core Patran
- **Aids in the design, analysis, and manufacture of laminated composite structures**
- **Process simulation**
 - Ply draping
 - Interface to FiberSIM
- **Composite data management**
- **Verification and visualization tools**
- **Post-processing tools**



GETTING STARTED WITH LAMINATE MODELER

- **Patran tasks prior to starting a Laminate Modeler Session**
 - Create a mesh which defines the surface for which the layup is to be defined



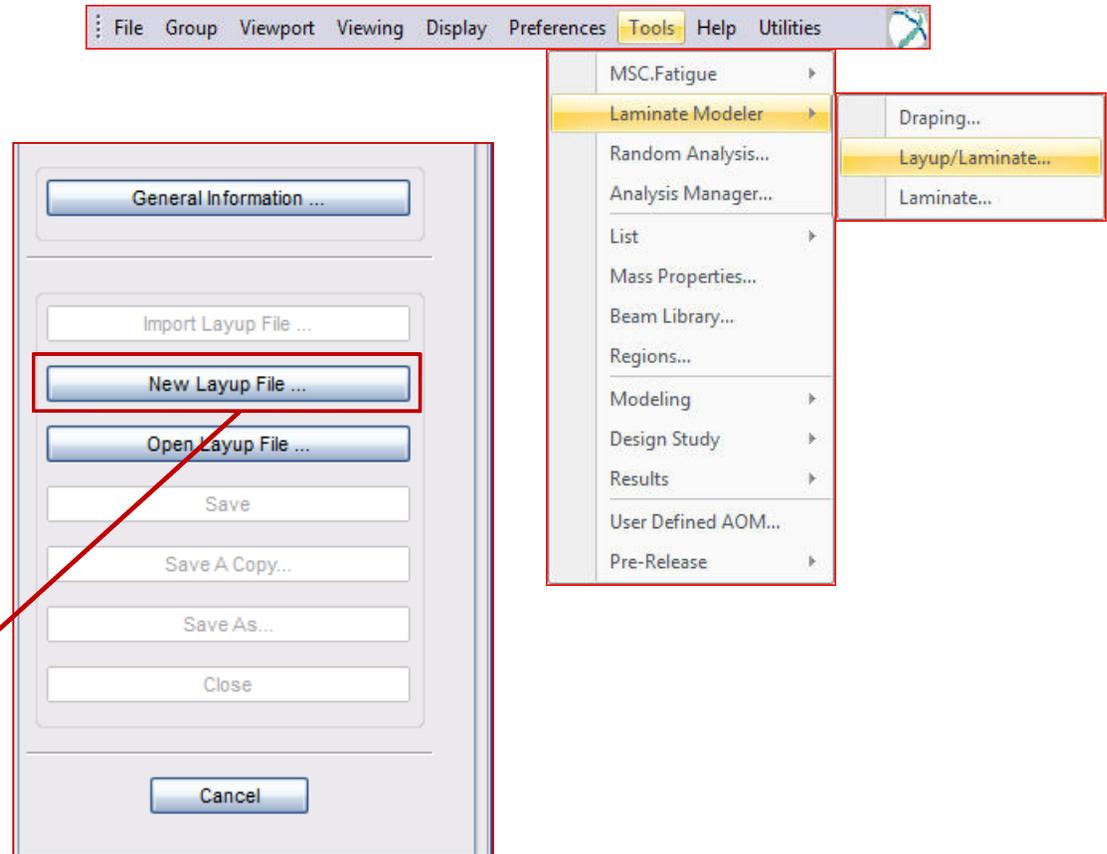
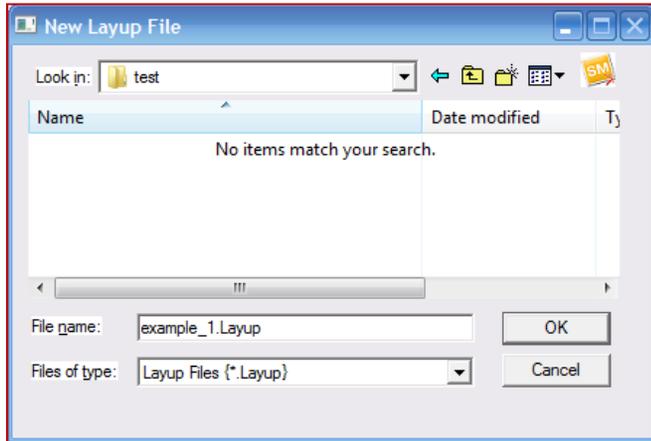
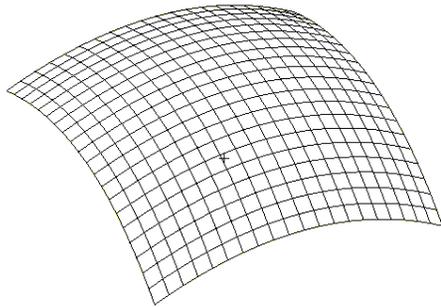
- Verify the mesh, element normals etc...
- Create basic material properties under Patran Materials (ie 2D orthotropic)
- Define any failure allowables if a failure analysis is desired
- Additional material data will also be entered in the Laminate Modeler session

GETTING STARTED WITH LAMINATE MODELER

- **Building Laminates with Laminate Modeler**
 - Creating **LM_Materials** specifying thickness and manufacturing data
 - Creating plies (**LM_Ply**) which has a geometric description as well as a fiber orientation varying across the surface of the ply
 - Creating Laminates (**LM_Layup**) by selecting LM_Plies previously created rather than selecting a material and an orientation like in core Patran
 - Modeling/Analyzing the laminate in its manufactured “as is” state
 - Verify laminates by showing exploded views, cross sections etc..
 - Automatically create multiple composite property entries to account for fiber orientations changing across the surface of the laminate
 - Relieves the analysts from a bookkeeping nightmare
 - Create Results by performing failure analysis inside of Laminate Modeler rather than importing the failure indices from the solver

GETTING STARTED WITH LAMINATE MODELER

- Once the shell model exists Laminate Modeler can be initialized from the Tools menu in Patran



CREATING LM_MATERIALS

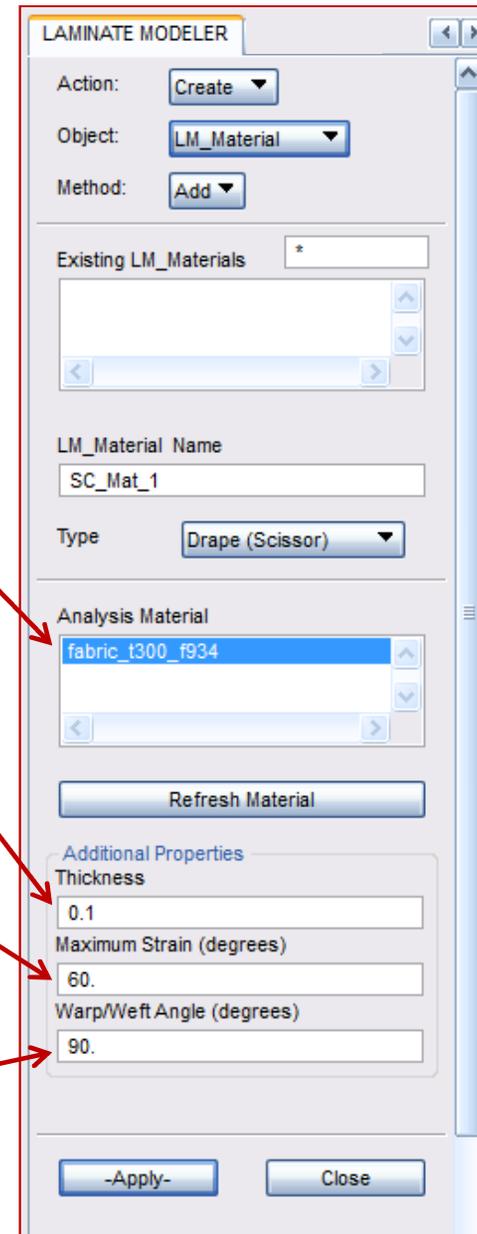
- An LM_Material is physically analogous to the raw material of the ply as it exists on the roll

Existing ply material data created in Patran

Ply thickness

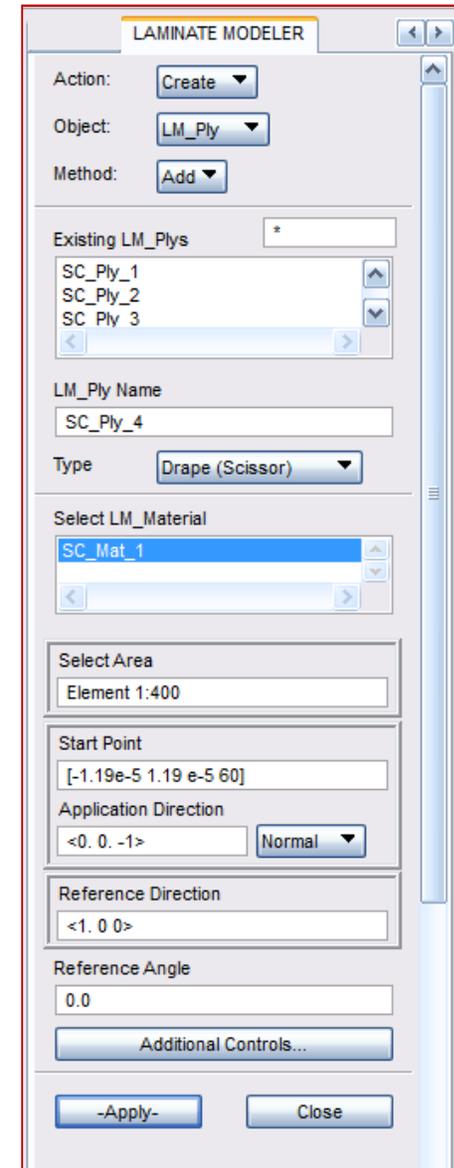
Maximum allowable in-plane shear strain before the material locks (i.e. Material can no longer conform to the surface by shearing)

Initial undeformed angle between the warp and weft yarns in a fabric

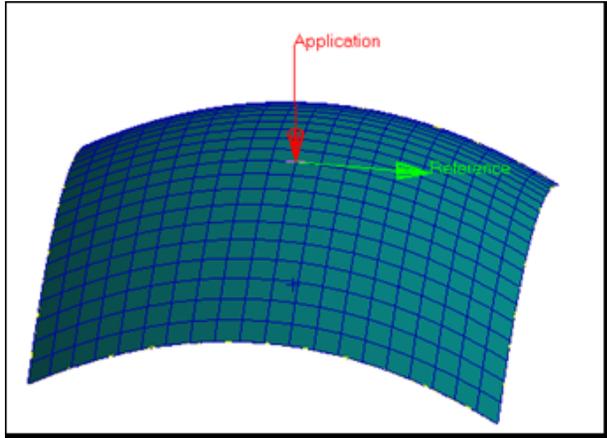


CREATING LM_PLIES

- A ply in Laminate Modeler is an area of LM_Material characterized by the area that it covers and the surface to which it is applied
 - Allows for easy manipulation of complex data as a single entity
 - Physical analogy to a cut-to-size piece of material ready to assembled into the mold.



CREATING LM_PLIES



Start Point: Starting point for the draping simulation process (i.e. The point at which the ply is first attached to the mould surface)

Application Direction: Specifies the surface normal of the undeformed flat ply

LAMINATE MODELER

Action: Create

Object: LM_Ply

Method: Add

Existing LM_Plys: SC_Ply_1, SC_Ply_2, SC_Ply_3

LM_Ply Name: SC_Ply_4

Type: Drape (Scissor)

Select LM_Material: SC_Mat_1

Select Area: Element 1:400

Start Point: [-1.19e-5 1.19 e-5 60]

Application Direction: <0. 0. -1> Normal

Reference Direction: <1. 0 0>

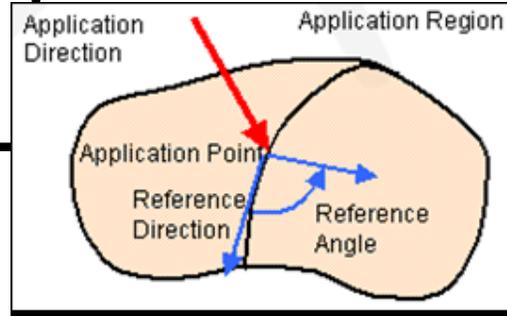
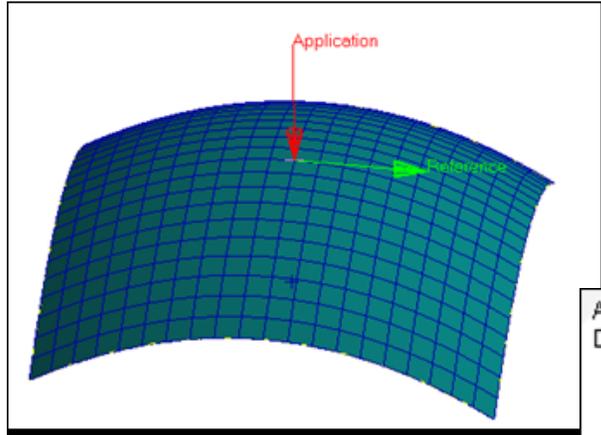
Reference Angle: 0.0

Buttons: -Apply-, Close

Previously defined LM_Material

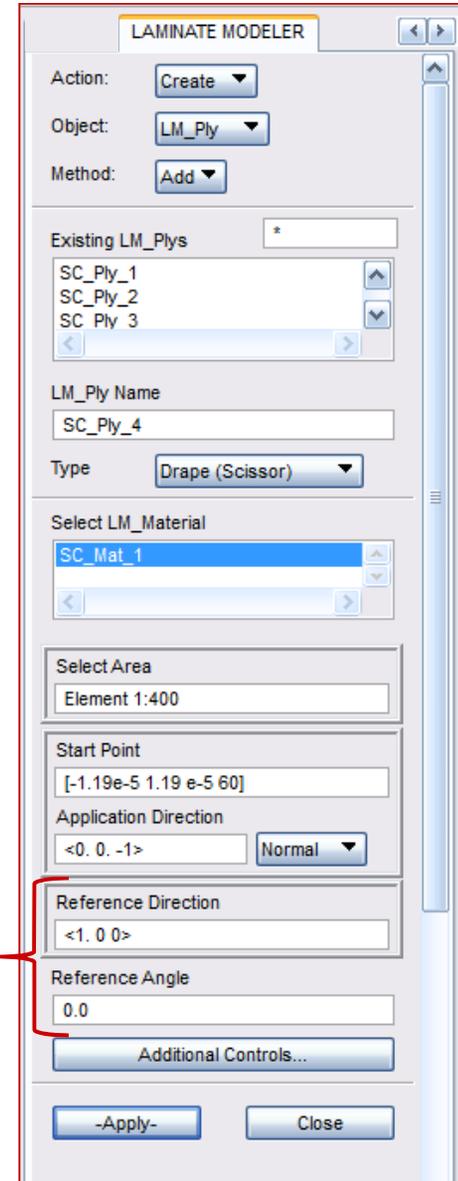
Surfaces or elements defining the extent of the ply

CREATING LM_PLIES

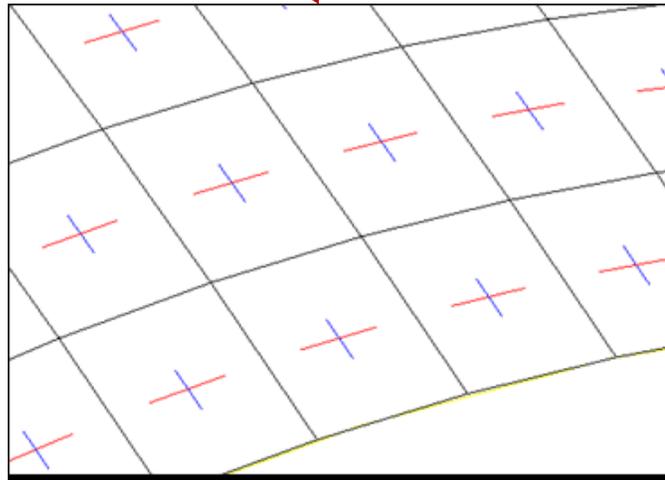
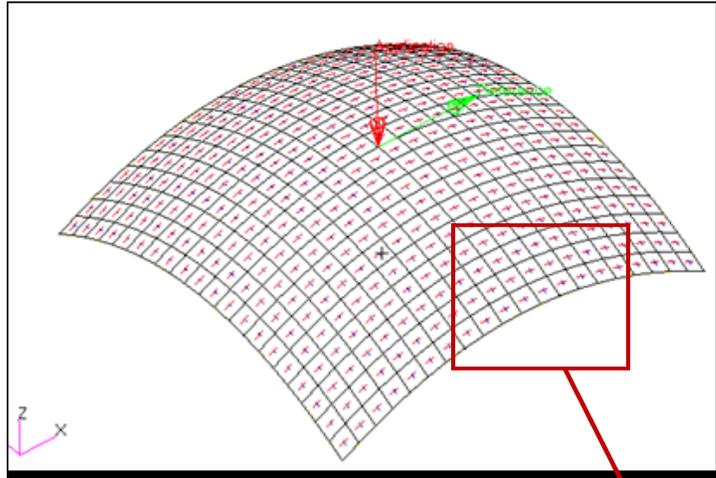


Reference Direction: Vector projected down to the surface which defines the fiber or warp direction.

Reference Angle: Angle of the ply fiber relative to the Reference Direction (counterclockwise when viewed from the Application Direction).



VERIFYING/SHOWING LM_PLIES



LAMINATE MODELER

Action: Show
Object: LM_Ply
Method: Graphics

Auto Execute

Existing LM_Plys
SC_Ply_1
SC_Ply_2
SC_Ply_3

Type: Drape (Scissor)

LM_Material: SC_Mat_1
Analysis Material: fabric_t300_f934
Selected Area: Element 321:328
Thickness: 0.1
Start Point: [-4.39096e-006 13. 0.]
Application Direction: <0.574 -0.815 -0.0755>
Reference Direction: <11.1879 10.5927 14.0269>
Reference Angle: 0.

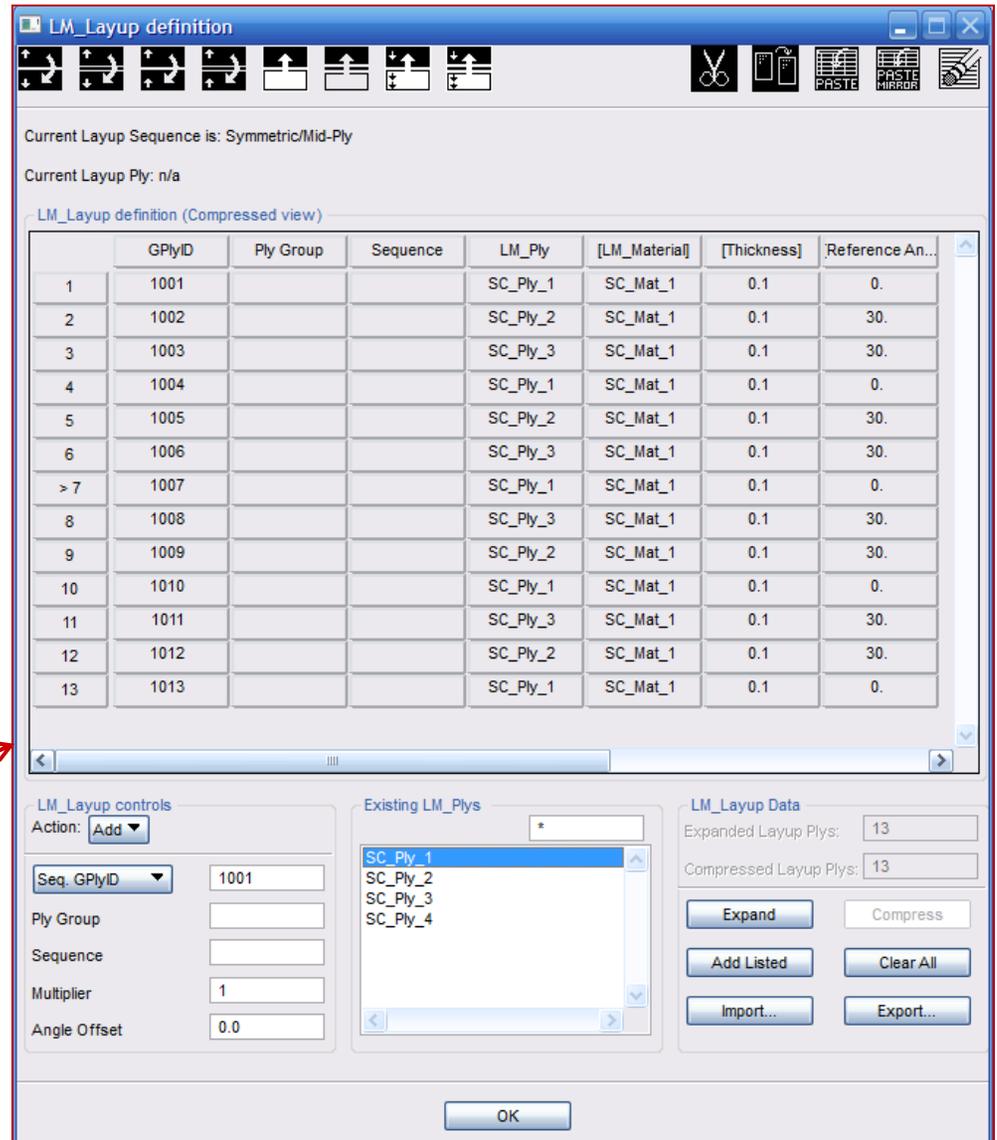
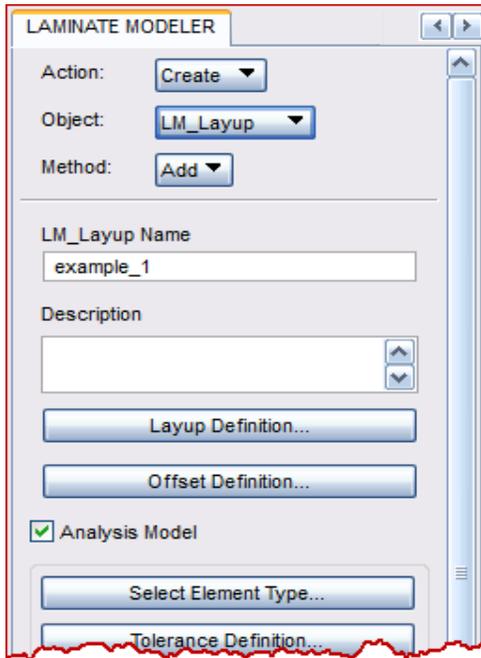
Display Options
 Material
 Application Direction
 Reference Direction
 Angles
 Boundary
 Draped Pattern
 Flat Pattern

Reset Graphics

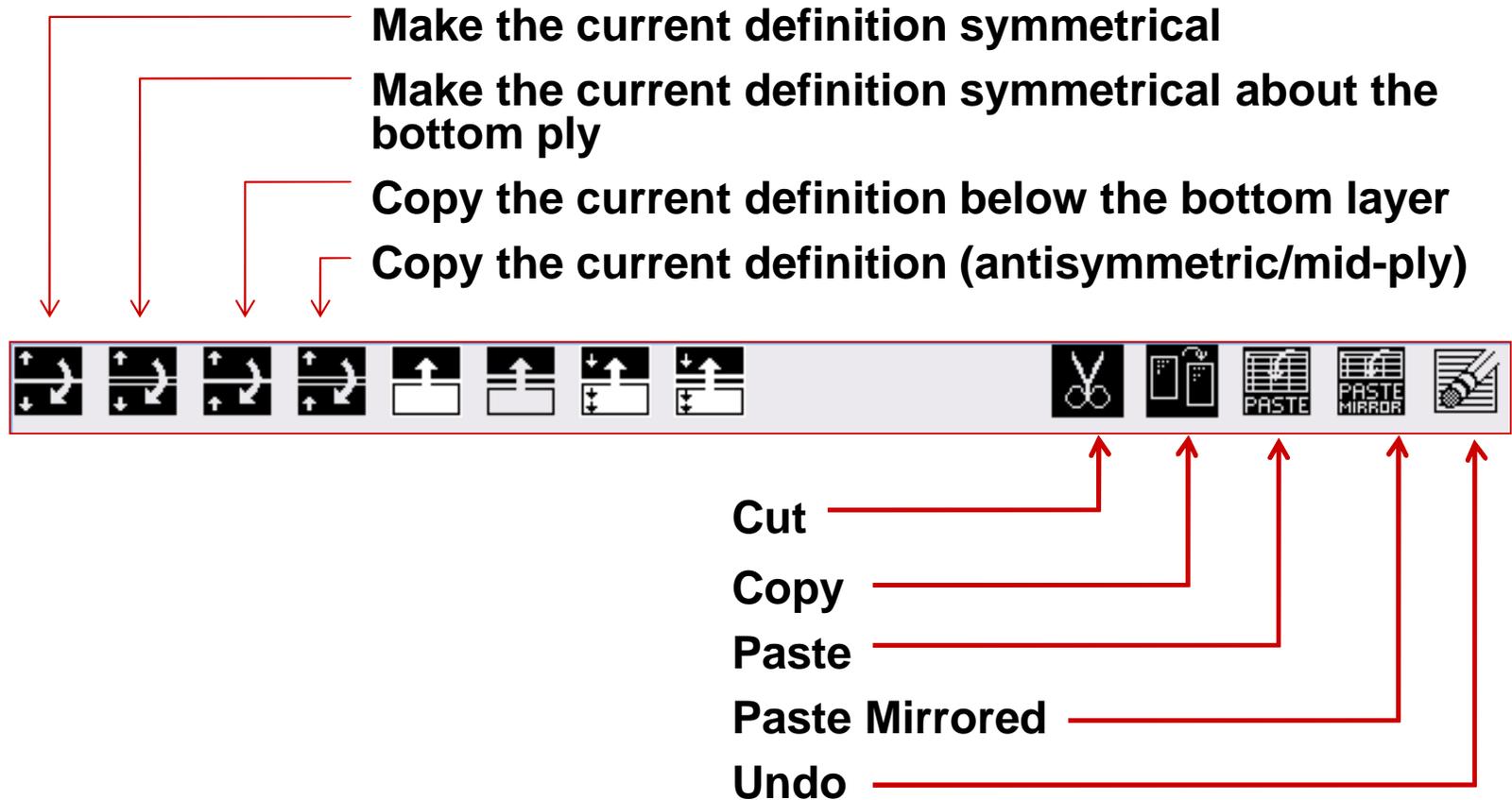
-Apply- Close

CREATING LM_LAYUPS

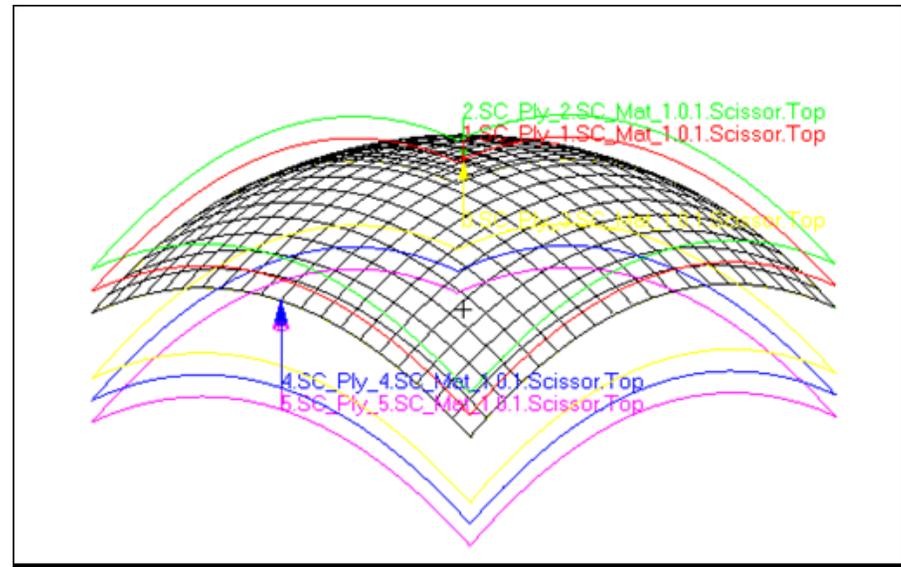
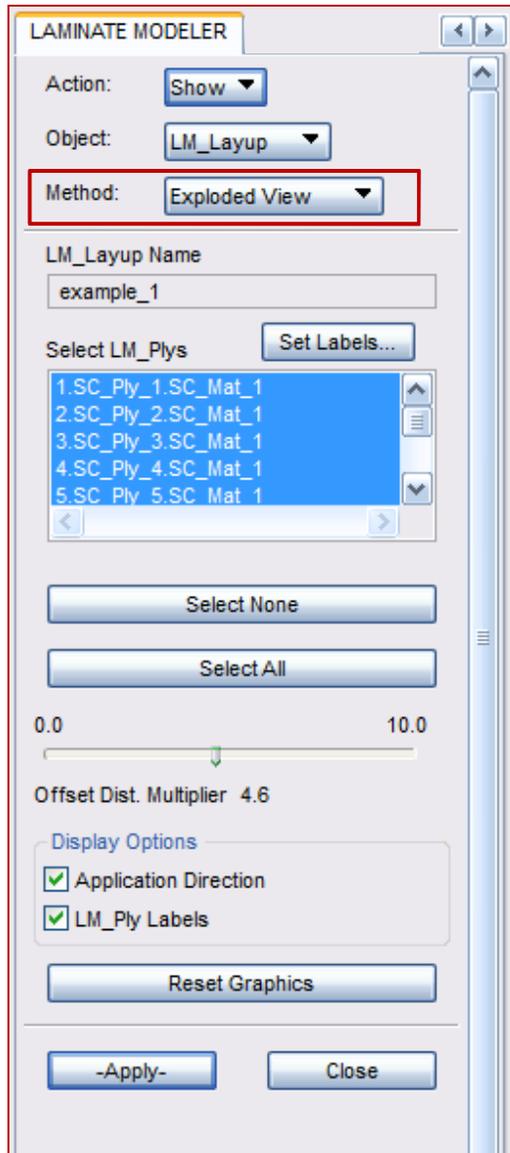
- The difference between this lay-up builder tool and the tools in Patran is that here we have already assigned thickness, material, and directions to the ply.



CREATING LM_LAYUPS

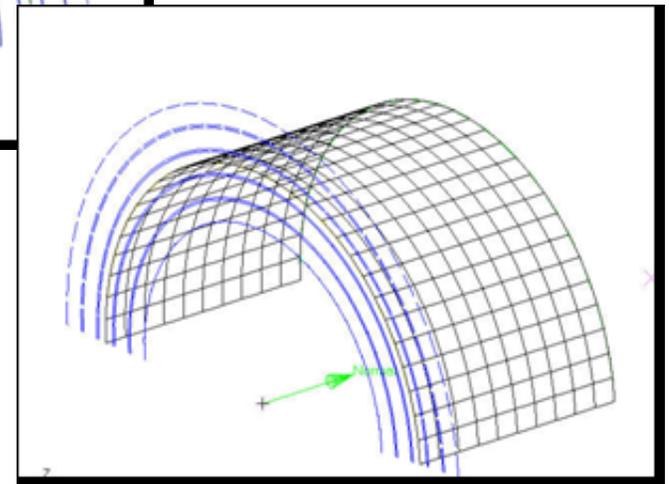
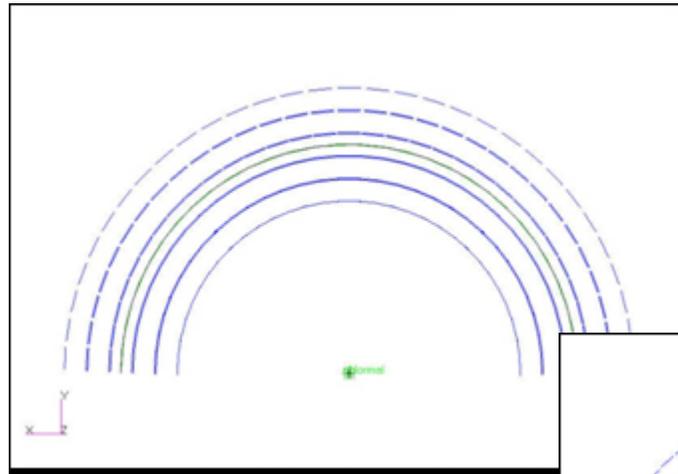
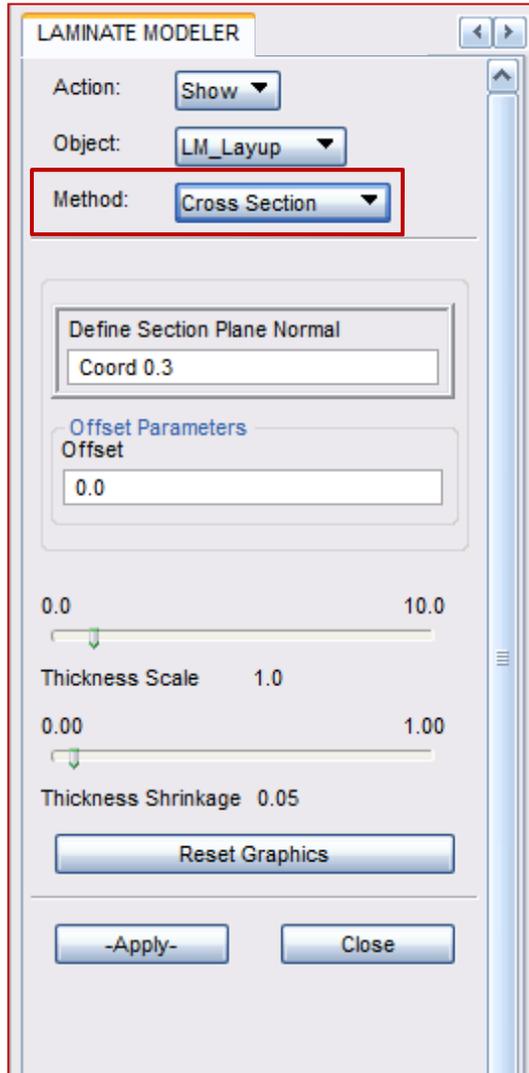


VERIFYING/SHOWING LM_LAYUPS



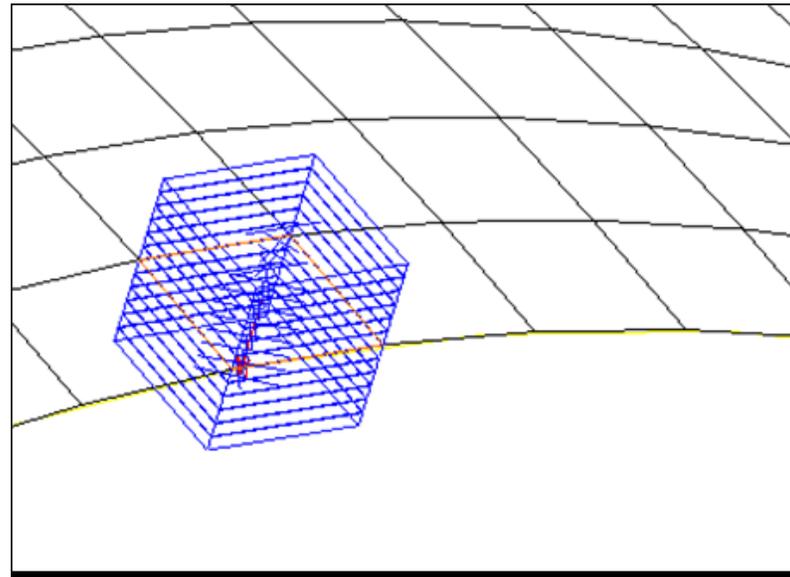
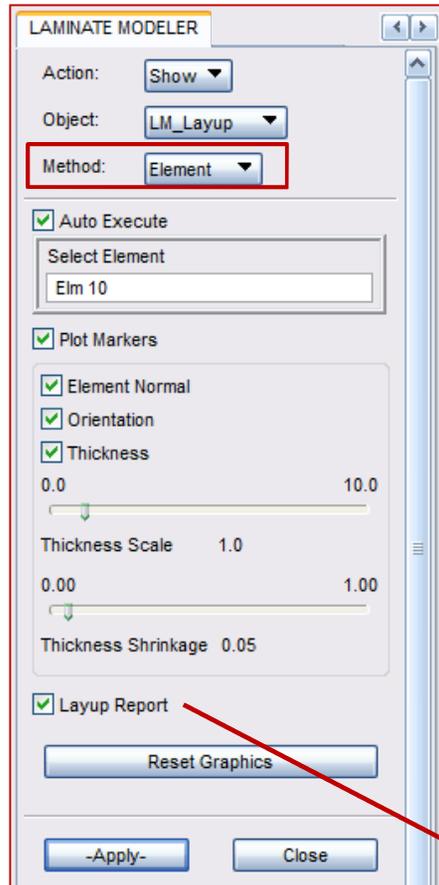
Layup Exploded Views

VERIFYING/SHOWING LM_LAYUPS



Layup Cross Section Views

VERIFYING/SHOWING LM_LAYUPS



Layup Views and Reports for Individual Elements

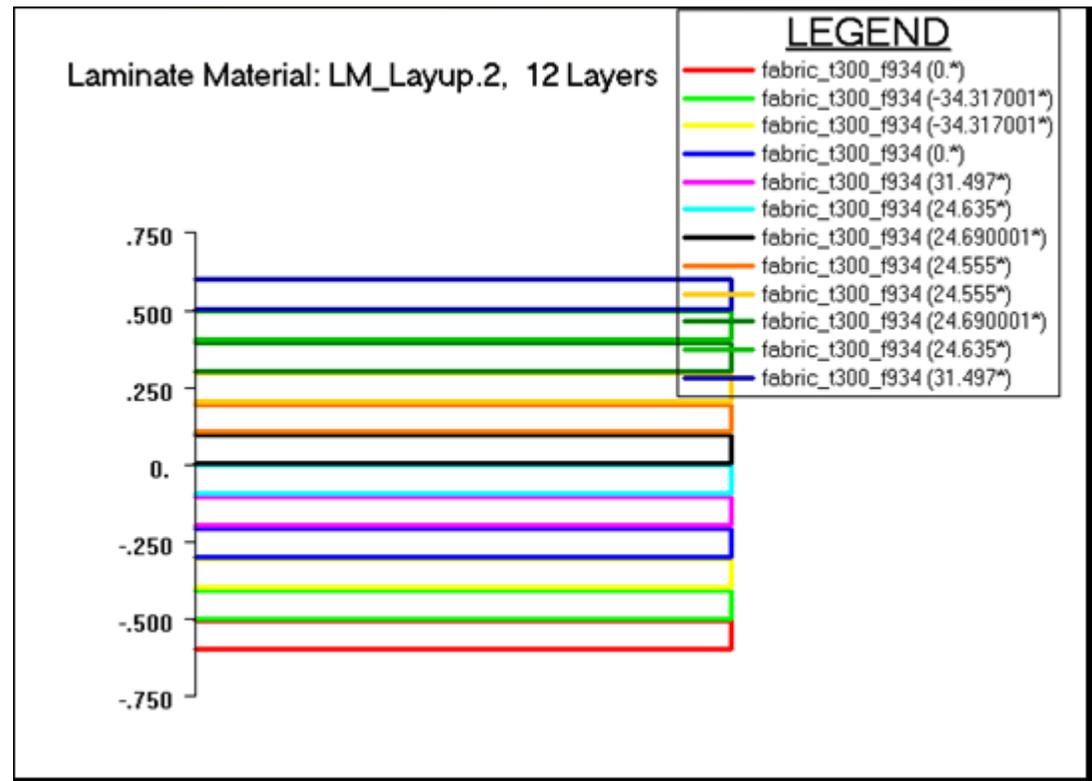
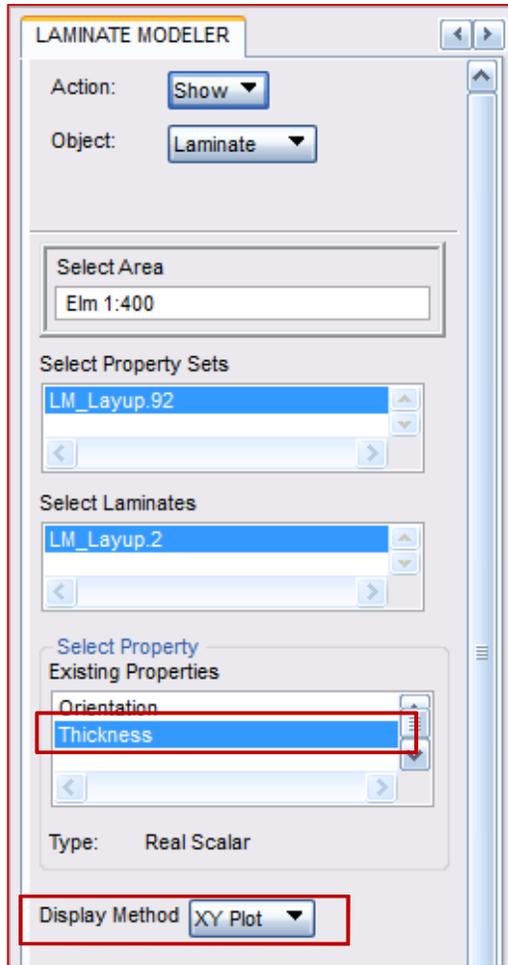
Element Layup Report

	GPlyDs	LM_Ply	LM_Material	Analysis Material	Thickness	Reference Angle	Type	Instance
Bottom	1001	SC_Ply_1	SC_Mat_1	fabric_t300_f934	0.1	0.	Scissor	1
2	1002	SC_Ply_2	SC_Mat_1	fabric_t300_f934	0.1	30.	Scissor	1
3	1003	SC_Ply_3	SC_Mat_1	fabric_t300_f934	0.1	30.	Scissor	1
4	1004	SC_Ply_4	SC_Mat_1	fabric_t300_f934	0.1	0.	Scissor	1

Cancel

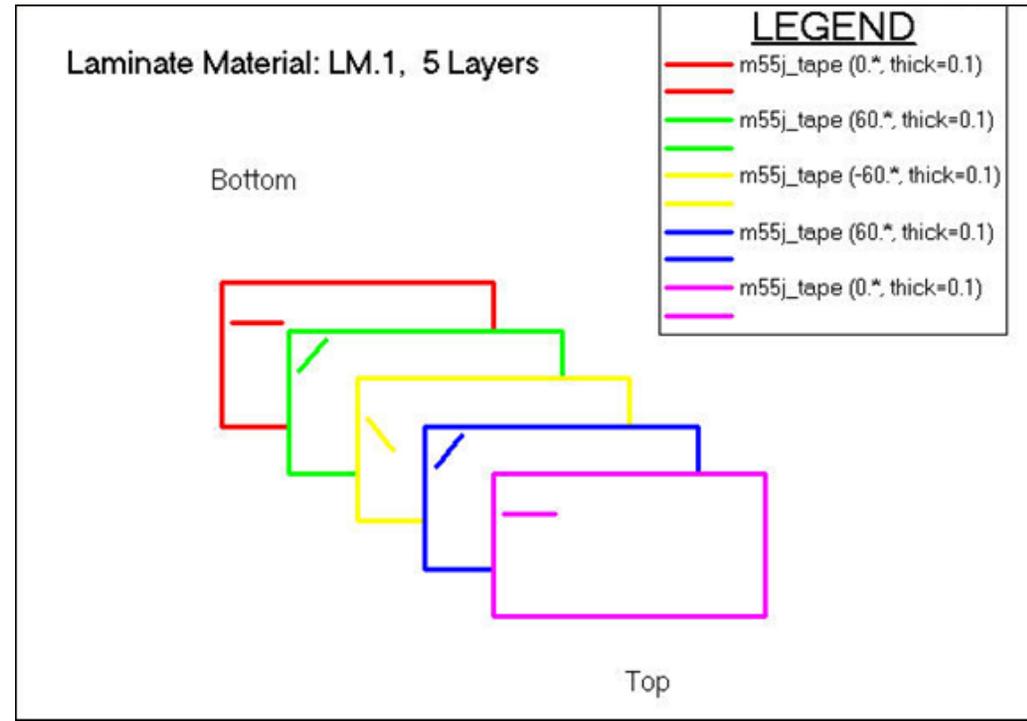
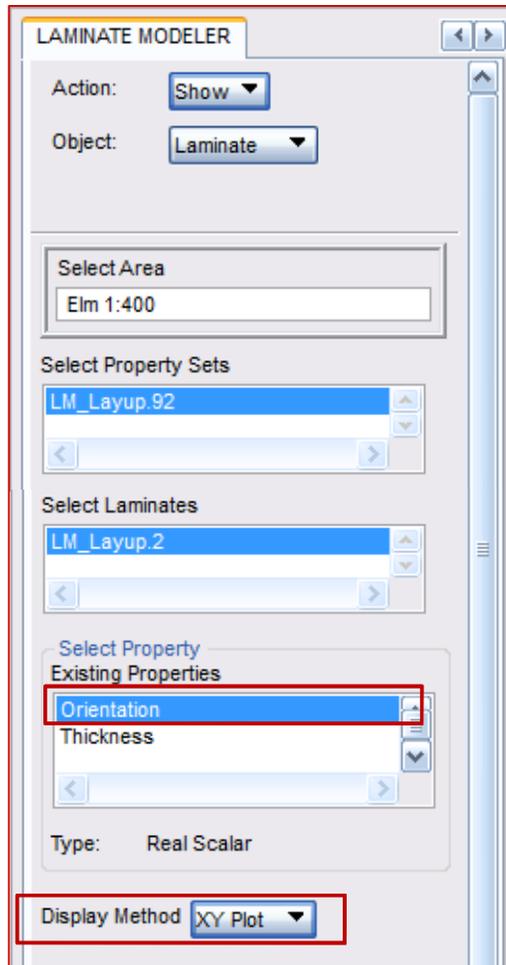
VERIFYING / SHOWING LAMINATE

- Show laminate thickness (xy plots)



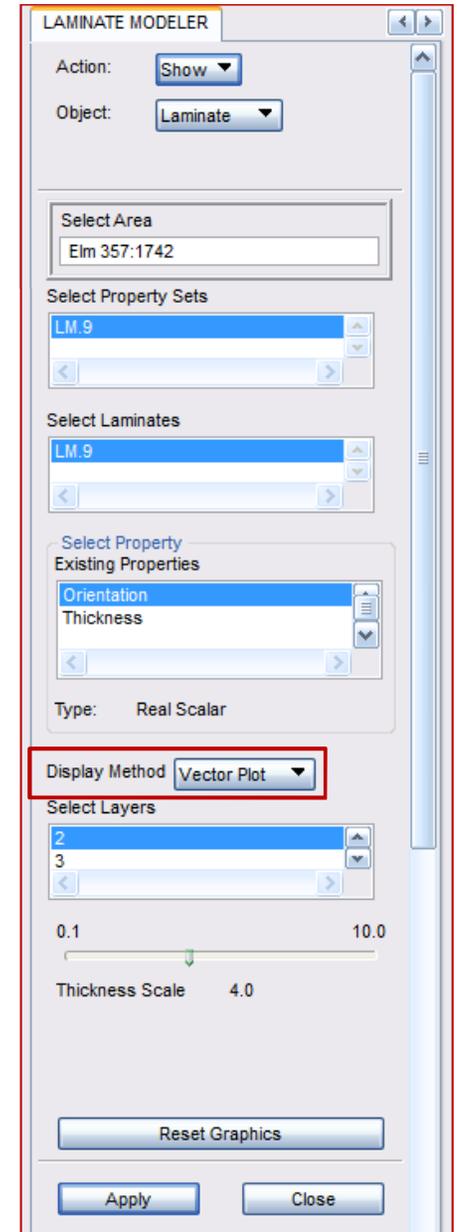
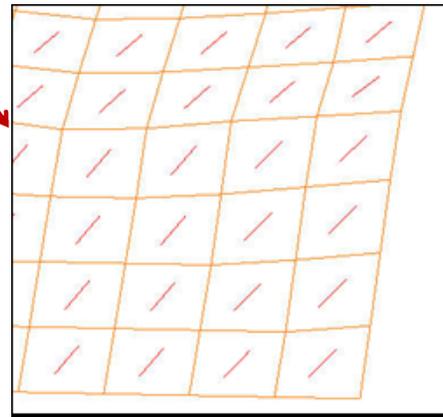
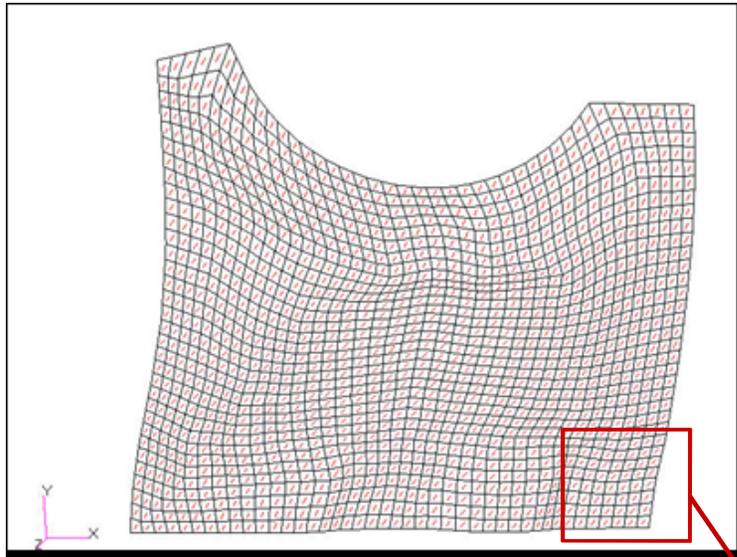
VERIFYING/SHOWING LAMINATE

- Show laminate orientations (xy plots)



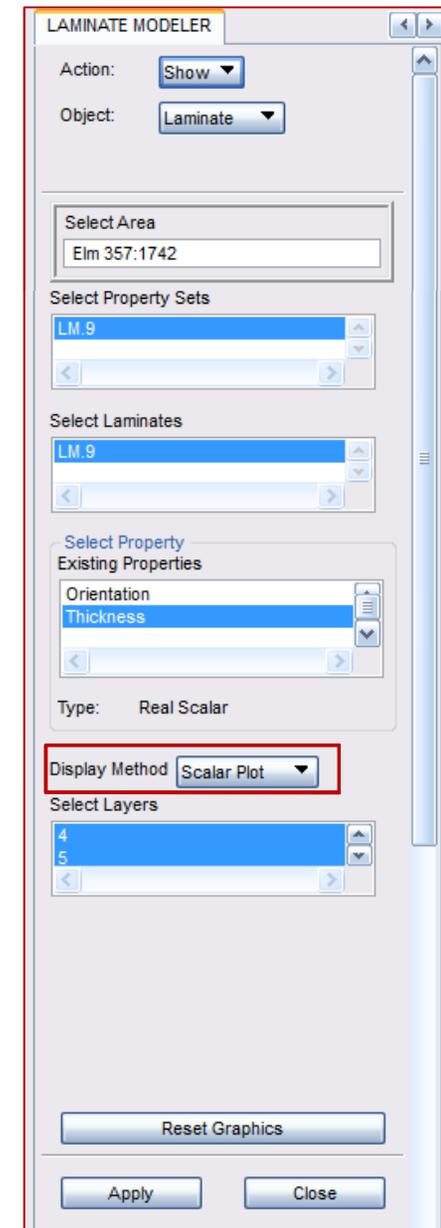
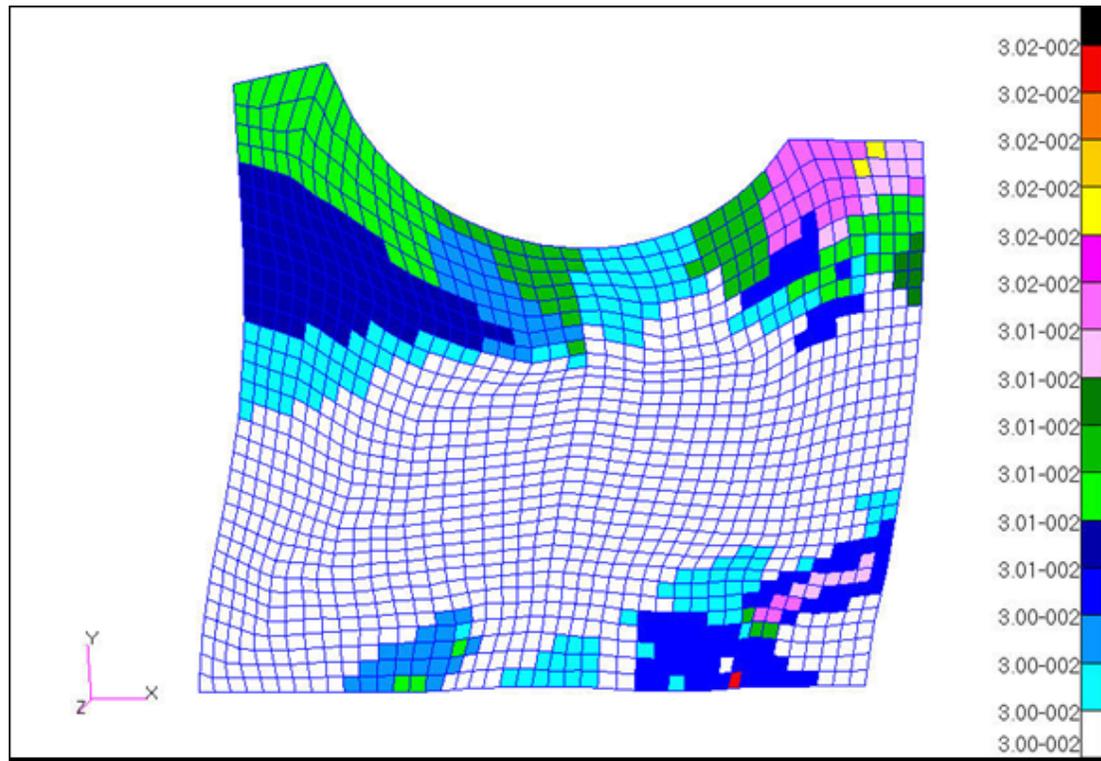
VERIFYING/SHOWING LAMINATE

- Show laminate orientations (Vector Plots)



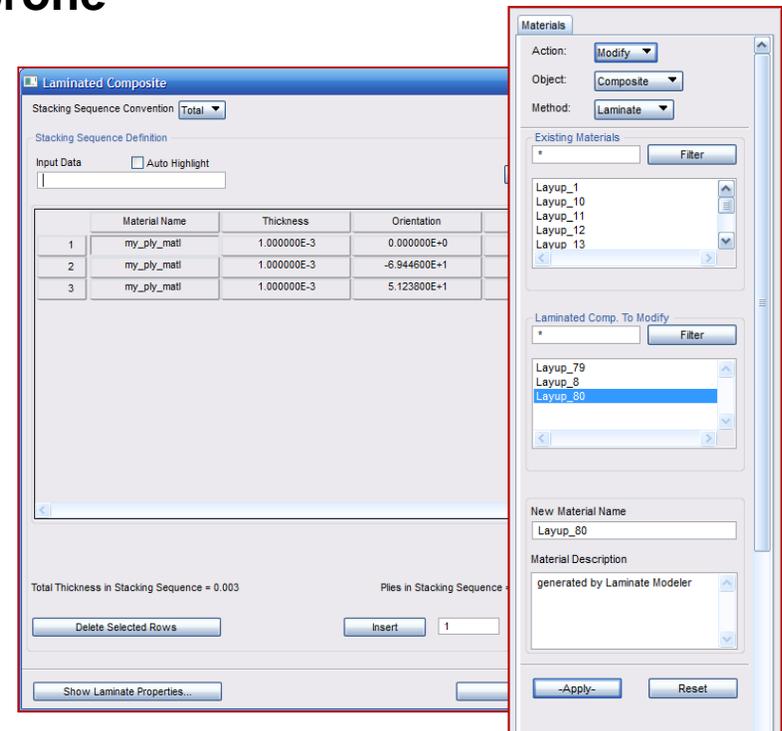
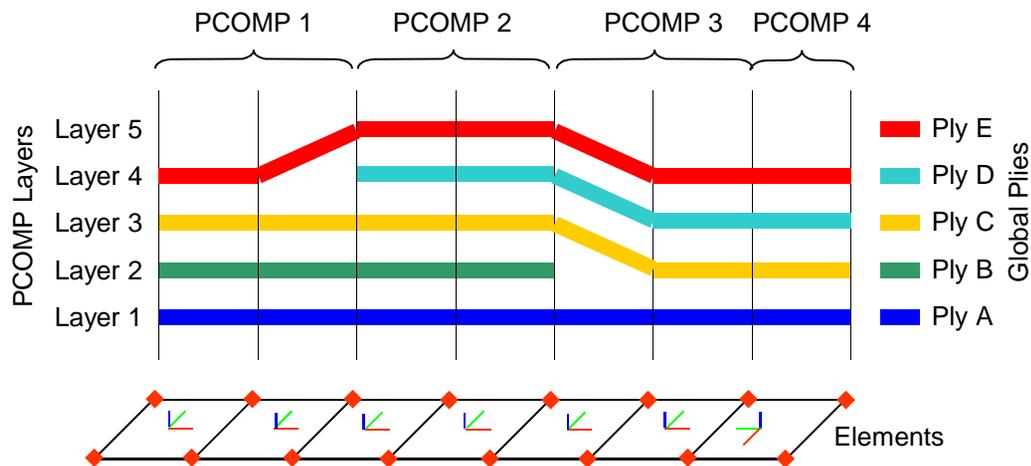
VERIFYING/SHOWING LAMINATE

- Show laminate thickness variations (Scalar Plot)



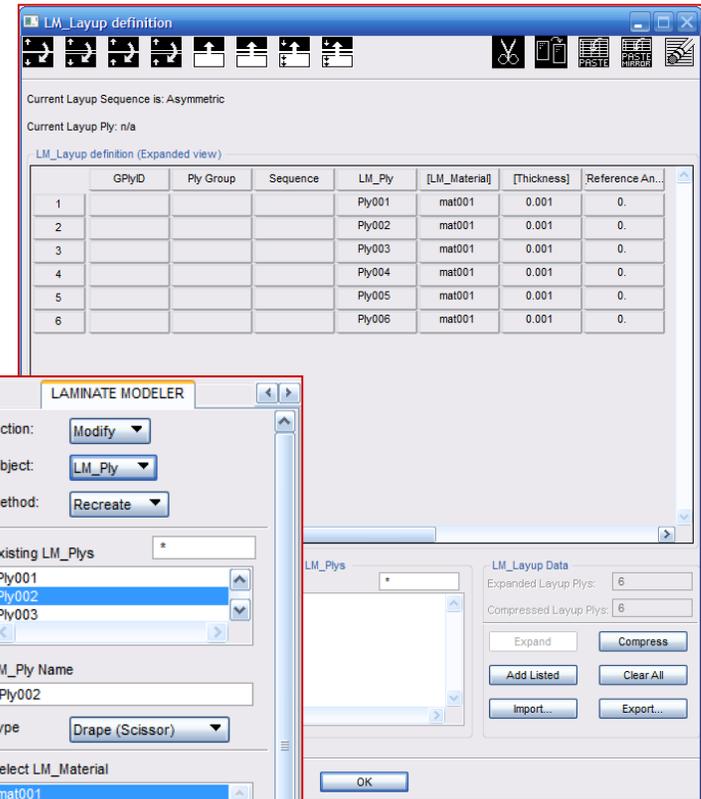
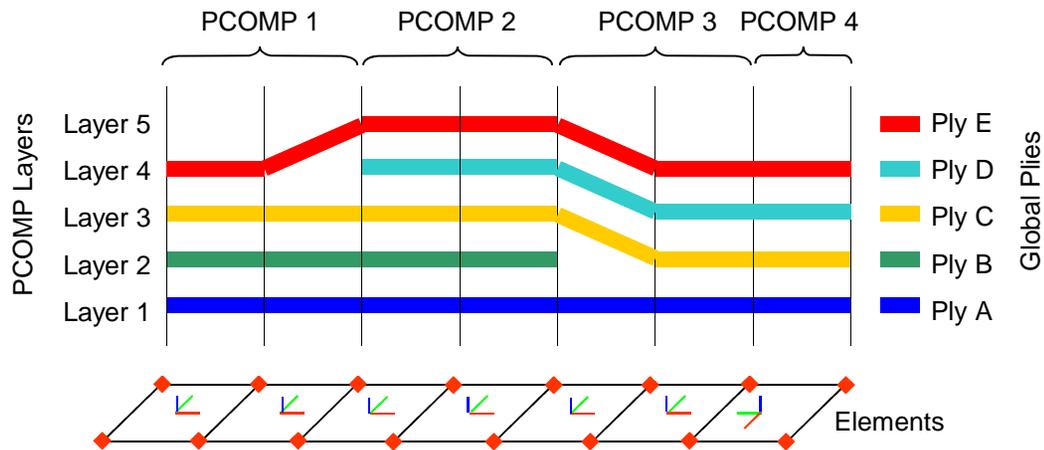
COMPOSITE DATA MANAGEMENT

- Layup modification via the traditional method is inefficient, error prone, and *costly*
- MSC Nastran also supports global ply
- Removing or changing Ply D requires changes to 4 PCOMPs
- Ply D's layer position depends on the PCOMP
- Ply D's orientation values depend on the PCOMP
- Changes are tedious and error prone



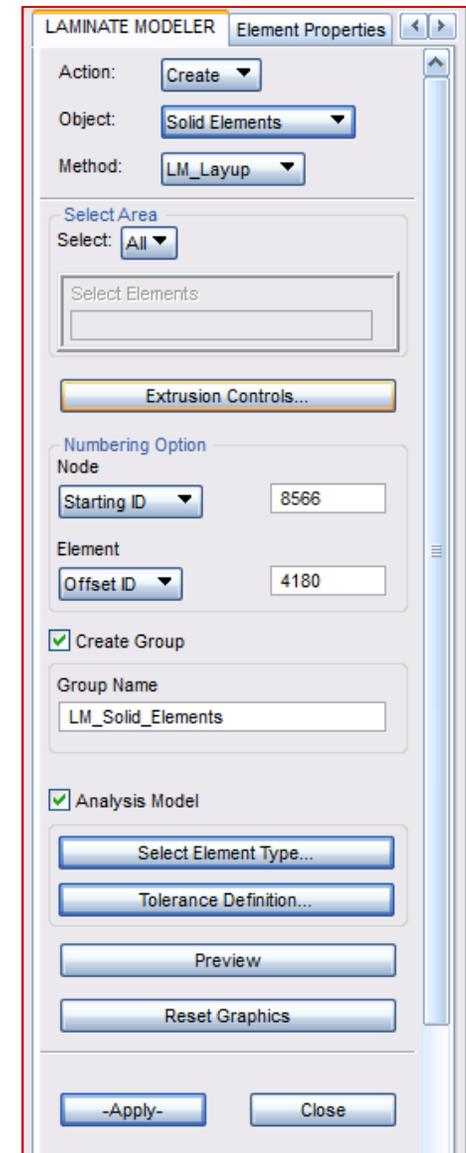
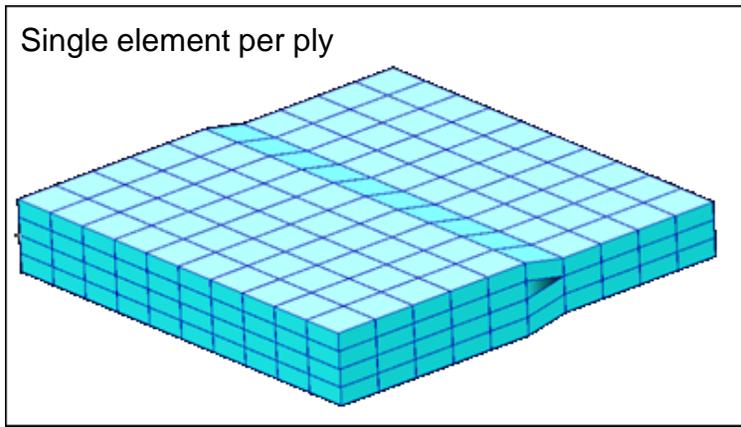
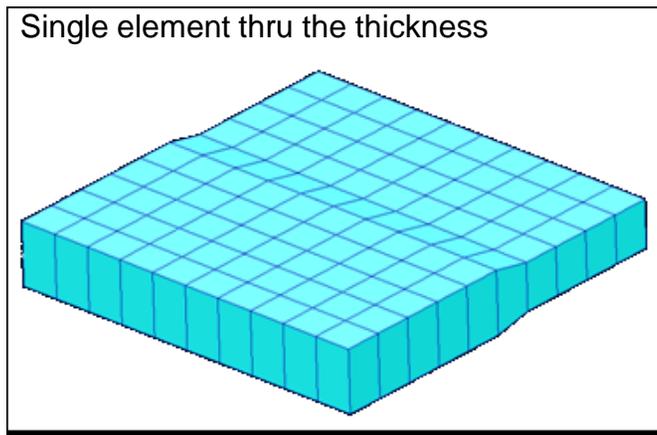
COMPOSITE DATA MANAGEMENT

- Layup modification in Laminate Modeler is efficient and accurate
- Removing or changing Ply D does not require editing PCOMPs
- Ply D is treated as an object
- Modification or deletion of Ply D automatically updates PCOMPs



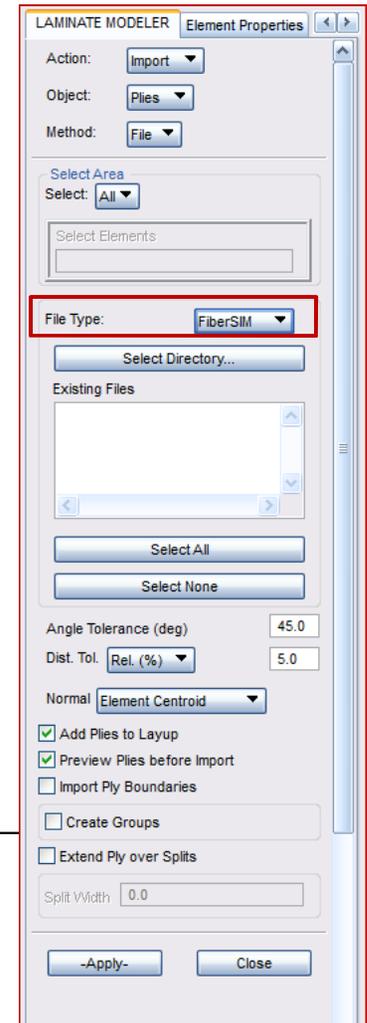
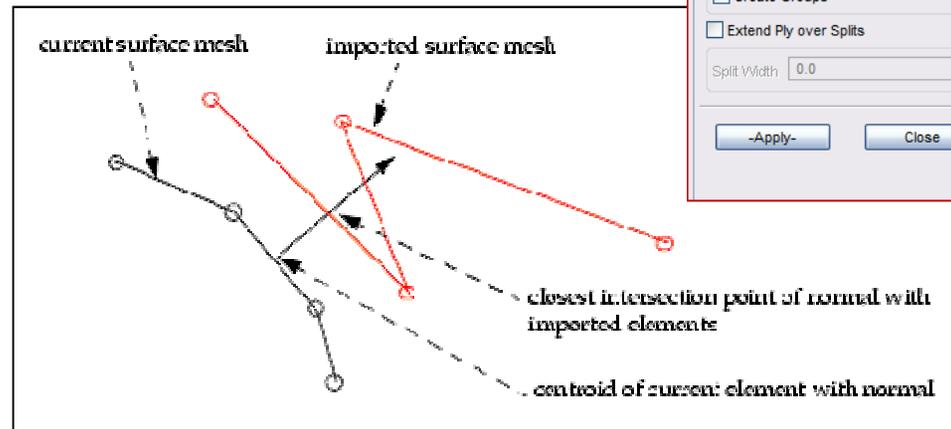
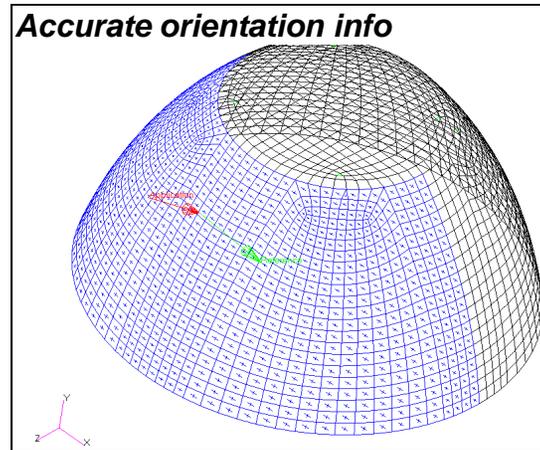
SUPPORT FOR SOLID ELEMENTS

- From 2D to 3D in a single step
- Automatically create 3D model from 2D mesh
 - Allows control over number of elements through thickness
 - Single element
 - Or per ply
- Maps as either
 - 3D Orthotropic
 - Laminate



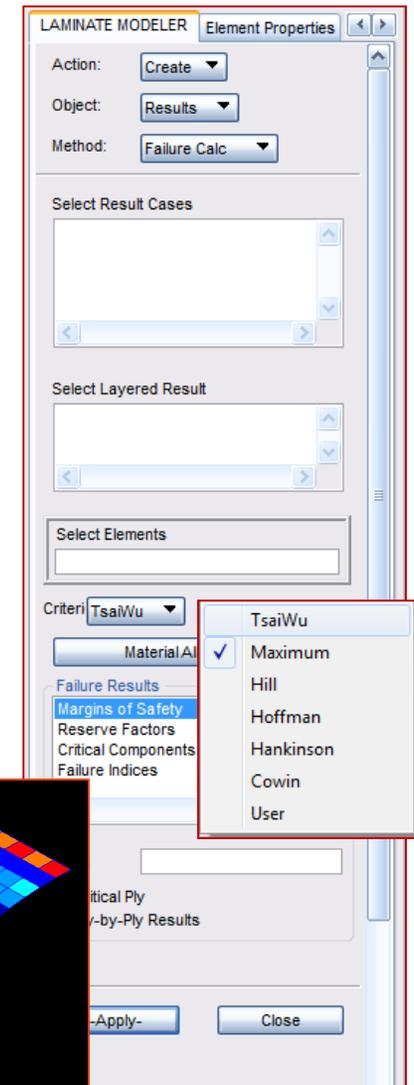
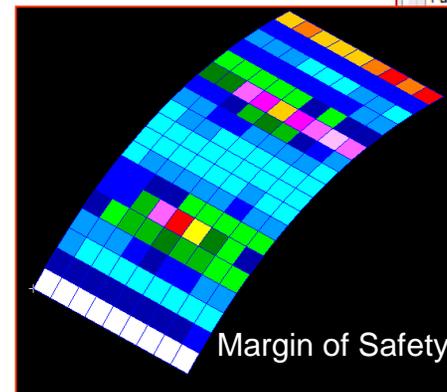
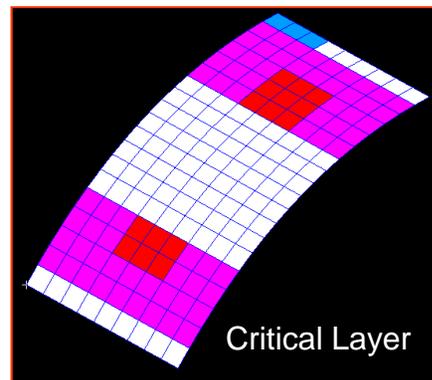
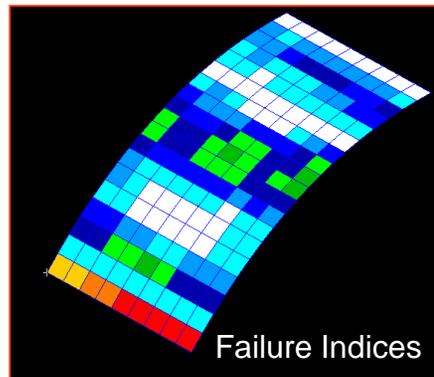
INTERFACE TO FIBERSIM

- **Seamless access to accurate layup data from FiberSIM in CAD**
- **A few details:**
 - FiberSIM generates a tria3 mesh for each ply from warp and fill lines resulting from its draping simulations
 - A layup file is generated for each ply (*.fml), together with a directory file listing all plies in a layup (*.fmd)
 - Laminate Modeler imports these layup files and maps each ply onto the current analysis mesh to generate an equivalent layup for the current mesh
 - Plies are added to the layup in the order in which they are imported (controlled by *.fmd file)



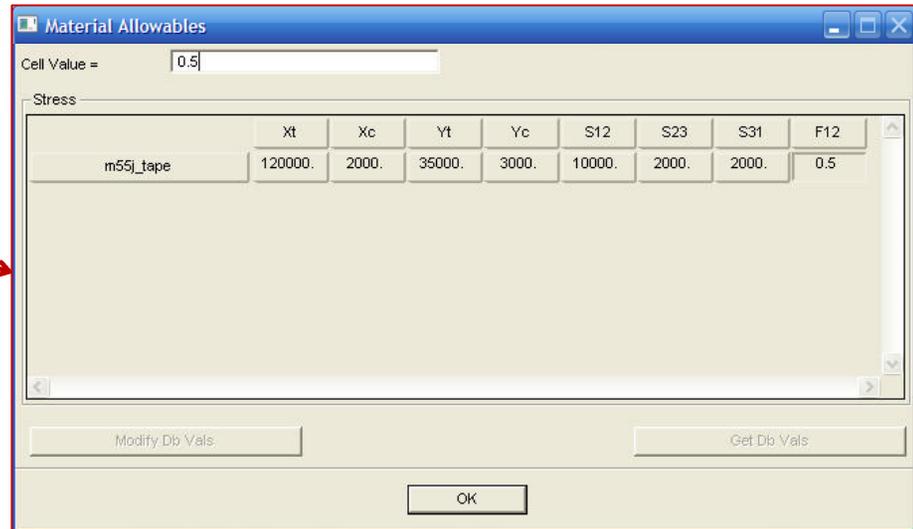
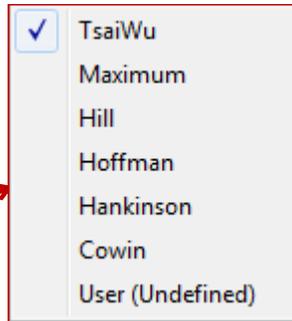
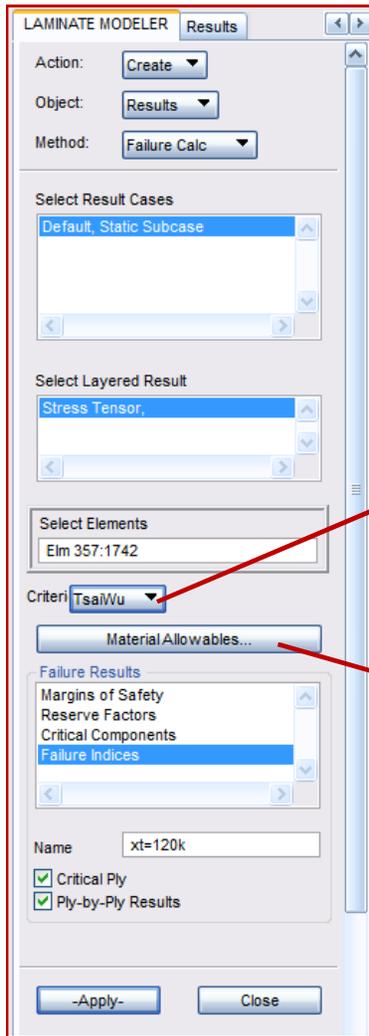
POST-PROCESSING SUPPORT

- Ply-based results rather than traditional MSC Nastran element layer-based results
- On-the-fly failure calculations
- Failure theories
 - Maximum
 - Tsai-Wu
 - Hill
 - Hoffman
 - Hankinson
 - Cowin
 - User-defined
- Failure data
 - Margin of safety
 - Critical component
 - Critical ply
 - Failure indices
 - Ply-by-ply results

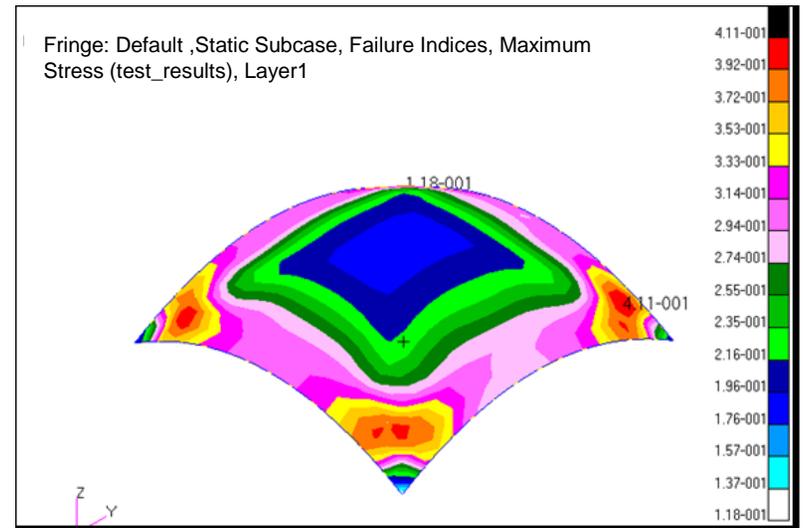
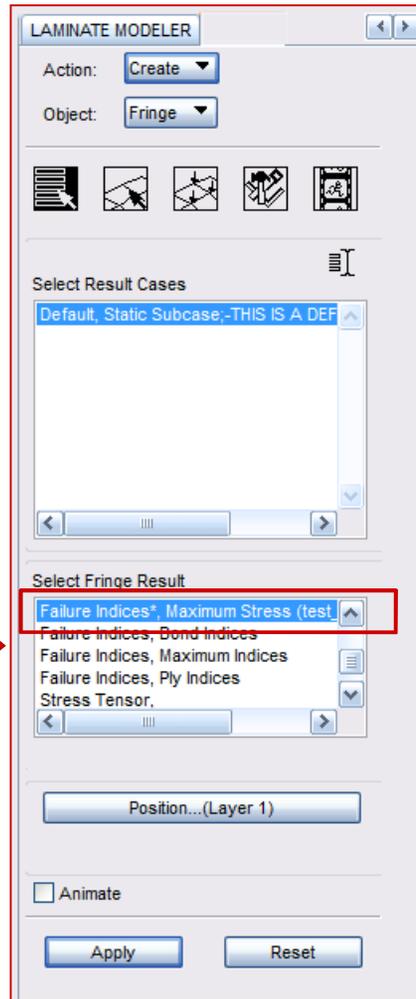
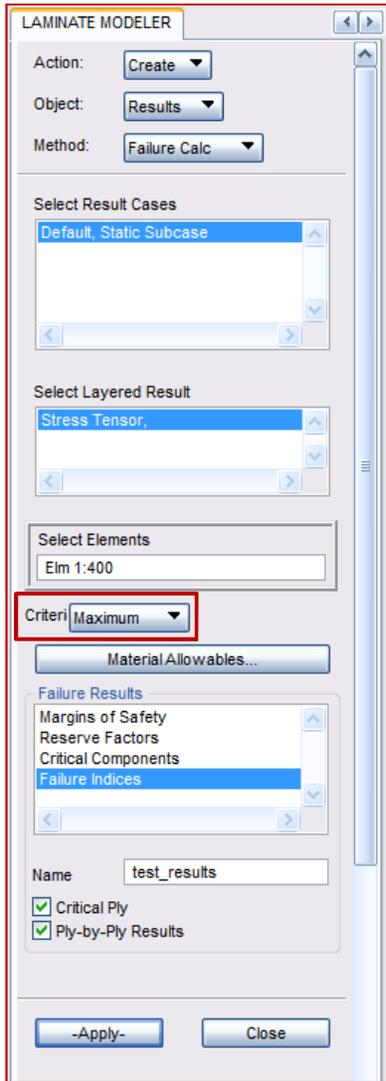


CREATING LAMINATE RESULTS

- New results can be calculated instantly without re-running the MSC Nastran job



CREATING LAMINATE RESULTS



WHY USE LAMINATE MODELER?

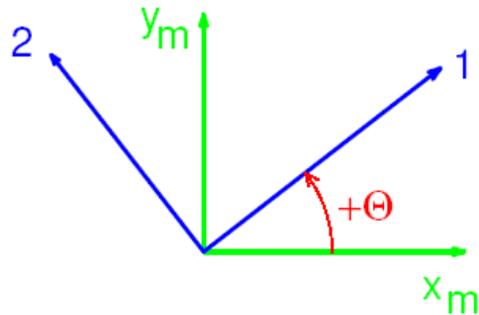
- **Ply draping**
 - Process simulation, accurate layup data
- **Interface to FiberSIM**
 - Seamless access to accurate layup data from CAD
- **Verification and visualization tools**
 - Improve model and results accuracy
- **Composite data management**
 - Efficient and accurate layup changes
- **Post-processing support**
 - Accurate assessment from meaningful results
- **Support for solid elements**
 - From 2D to 3D in a single step
- **Enables efficient and accurate composite analysis**
- **Saves time, saves \$\$\$, more confidence in your design and analysis ... better products**
- **Laminate Modeler (PAT325)**



APPENDIX A

COMPOSITE THEORY FOR PLATE ELEMENTS

Composite Equations used in MSC.Nastran



- 1 - Fiber Direction
- 2 - Matrix Direction

x_m, y_m - Material Coordinate System

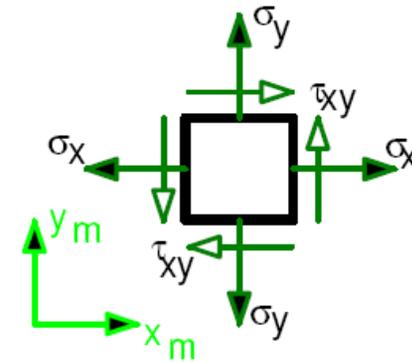
θ - angle between x_m and 1 direction

Stress Transformation $\sigma' = U_{\sigma} \cdot \sigma$

$$\begin{pmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{pmatrix} = \begin{pmatrix} \cos(\theta)^2 & \sin(\theta)^2 & 2 \cdot \sin(\theta) \cdot \cos(\theta) \\ \sin(\theta)^2 & \cos(\theta)^2 & -2 \cdot \sin(\theta) \cdot \cos(\theta) \\ -\sin(\theta) \cdot \cos(\theta) & \sin(\theta) \cdot \cos(\theta) & \cos(\theta)^2 - \sin(\theta)^2 \end{pmatrix} \begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix}$$

Inverse Stress Transformation $\sigma = U_{\sigma}^{-1} \cdot \sigma' = U_{\sigma'} \cdot \sigma'$

$$\begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix} = \begin{pmatrix} \cos(\theta)^2 & \sin(\theta)^2 & -2 \cdot \sin(\theta) \cdot \cos(\theta) \\ \sin(\theta)^2 & \cos(\theta)^2 & 2 \cdot \sin(\theta) \cdot \cos(\theta) \\ \sin(\theta) \cdot \cos(\theta) & -\sin(\theta) \cdot \cos(\theta) & \cos(\theta)^2 - \sin(\theta)^2 \end{pmatrix} \begin{pmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{pmatrix}$$



$$\sigma' = \begin{pmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{pmatrix} \quad \sigma = \begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix}$$

Strain Transformation $\epsilon' = U_{\epsilon} \cdot \epsilon$

$$\begin{pmatrix} \epsilon_1 \\ \epsilon_2 \\ \gamma_{12} \end{pmatrix} = \begin{pmatrix} \cos(\theta)^2 & \sin(\theta)^2 & \sin(\theta) \cdot \cos(\theta) \\ \sin(\theta)^2 & \cos(\theta)^2 & -\sin(\theta) \cdot \cos(\theta) \\ -2 \sin(\theta) \cdot \cos(\theta) & 2 \sin(\theta) \cdot \cos(\theta) & \cos(\theta)^2 - \sin(\theta)^2 \end{pmatrix} \begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{pmatrix}$$

$$\epsilon' = \begin{pmatrix} \epsilon_1 \\ \epsilon_2 \\ \gamma_{12} \end{pmatrix} \quad \epsilon = \begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{pmatrix}$$

Inverse Strain Transformation $\epsilon = U_{\epsilon}^{-1} \cdot \epsilon' = U_{\epsilon'} \cdot \epsilon'$

$$\begin{pmatrix} \epsilon_x \\ \epsilon_y \\ \gamma_{xy} \end{pmatrix} = \begin{pmatrix} \cos(\theta)^2 & \sin(\theta)^2 & -\sin(\theta) \cdot \cos(\theta) \\ \sin(\theta)^2 & \cos(\theta)^2 & \sin(\theta) \cdot \cos(\theta) \\ 2 \sin(\theta) \cdot \cos(\theta) & -2 \sin(\theta) \cdot \cos(\theta) & \cos(\theta)^2 - \sin(\theta)^2 \end{pmatrix} \begin{pmatrix} \epsilon_1 \\ \epsilon_2 \\ \gamma_{12} \end{pmatrix}$$

Relations between Stress and Strain Transformation Matrices

$$U_{\epsilon} = \begin{pmatrix} \cos(\theta)^2 & \sin(\theta)^2 & \sin(\theta) \cdot \cos(\theta) \\ \sin(\theta)^2 & \cos(\theta)^2 & -\sin(\theta) \cdot \cos(\theta) \\ -2 \sin(\theta) \cdot \cos(\theta) & 2 \sin(\theta) \cdot \cos(\theta) & \cos(\theta)^2 - \sin(\theta)^2 \end{pmatrix} \quad U_{\epsilon'} = U_{\sigma}^T$$

$$U_{\sigma} = \begin{pmatrix} \cos(\theta)^2 & \sin(\theta)^2 & 2 \cdot \sin(\theta) \cdot \cos(\theta) \\ \sin(\theta)^2 & \cos(\theta)^2 & -2 \sin(\theta) \cdot \cos(\theta) \\ -\sin(\theta) \cdot \cos(\theta) & \sin(\theta) \cdot \cos(\theta) & \cos(\theta)^2 - \sin(\theta)^2 \end{pmatrix} \quad U_{\epsilon} = U_{\sigma'}^T$$

$$U_{\sigma'} = \begin{pmatrix} \cos(\theta)^2 & \sin(\theta)^2 & -2 \cdot \sin(\theta) \cdot \cos(\theta) \\ \sin(\theta)^2 & \cos(\theta)^2 & 2 \cdot \sin(\theta) \cdot \cos(\theta) \\ \sin(\theta) \cdot \cos(\theta) & -\sin(\theta) \cdot \cos(\theta) & \cos(\theta)^2 - \sin(\theta)^2 \end{pmatrix}$$

Shell Element Orthotropic Material in 1-2 Coordinates (MAT8)

$$\begin{pmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{pmatrix} = \begin{pmatrix} \frac{1}{E_1} & \frac{\nu_{21}}{E_2} & 0 \\ \frac{\nu_{12}}{E_1} & \frac{1}{E_2} & 0 \\ 0 & 0 & \frac{1}{G_{12}} \end{pmatrix} \begin{pmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{pmatrix} \quad \nu_{12} \cdot E_2 = \nu_{21} \cdot E_1 \quad (\text{Symmetry Condition!})$$

Symbolic Inversion

$$\begin{pmatrix} \frac{1}{E_1} & \frac{\nu_{21}}{E_2} & 0 \\ \frac{\nu_{12}}{E_1} & \frac{1}{E_2} & 0 \\ 0 & 0 & \frac{1}{G_{12}} \end{pmatrix}^{-1} \quad \text{expand} \rightarrow \begin{pmatrix} \frac{-1}{-1 + \nu_{12} \cdot \nu_{21}} \cdot E_1 & \frac{-\nu_{21}}{-1 + \nu_{12} \cdot \nu_{21}} \cdot E_1 & 0 \\ \frac{-\nu_{12}}{-1 + \nu_{12} \cdot \nu_{21}} \cdot E_2 & \frac{-1}{-1 + \nu_{12} \cdot \nu_{21}} \cdot E_2 & 0 \\ 0 & 0 & G_{12} \end{pmatrix}$$

$$\begin{pmatrix} \sigma_1 \\ \sigma_2 \\ \tau_{12} \end{pmatrix} = \begin{pmatrix} \frac{E_1}{1 - \nu_{12} \cdot \nu_{21}} & \frac{E_1 \cdot \nu_{21}}{1 - \nu_{12} \cdot \nu_{21}} & 0 \\ \frac{E_2 \cdot \nu_{12}}{1 - \nu_{12} \cdot \nu_{21}} & \frac{E_2}{1 - \nu_{12} \cdot \nu_{21}} & 0 \\ 0 & 0 & G_{12} \end{pmatrix} \begin{pmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{pmatrix} = \begin{pmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{pmatrix} \begin{pmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \gamma_{12} \end{pmatrix}$$

$$\sigma' = Q \cdot \varepsilon'$$

Q-Matrix in the 1-2 (fiber-matrix) coordinate system

$$Q = \begin{pmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{pmatrix}$$

Transformation of Q-Matrix in the Material Coordinate System

$$Q^m = U_\epsilon^T \cdot Q \cdot U_\epsilon \quad Q^m = \begin{pmatrix} Q_{11}^m & Q_{12}^m & Q_{16}^m \\ Q_{12}^m & Q_{22}^m & Q_{26}^m \\ Q_{16}^m & Q_{26}^m & Q_{66}^m \end{pmatrix}$$

Abbreviations to simplify symbolic manipulations

$$c = \cos(\theta) \quad s = \sin(\theta)$$

$$U_\epsilon = \begin{pmatrix} c^2 & s^2 & s \cdot c \\ s^2 & c^2 & -s \cdot c \\ -2 \cdot s \cdot c & 2 \cdot s \cdot c & c^2 - s^2 \end{pmatrix}$$

$$\begin{pmatrix} c^2 & s^2 & s \cdot c \\ s^2 & c^2 & -s \cdot c \\ -2 \cdot s \cdot c & 2 \cdot s \cdot c & c^2 - s^2 \end{pmatrix}^T \cdot \begin{pmatrix} Q_{11} & Q_{12} & 0 \\ Q_{12} & Q_{22} & 0 \\ 0 & 0 & Q_{66} \end{pmatrix} \cdot \begin{pmatrix} c^2 & s^2 & s \cdot c \\ s^2 & c^2 & -s \cdot c \\ -2 \cdot s \cdot c & 2 \cdot s \cdot c & c^2 - s^2 \end{pmatrix} \left| \begin{array}{l} \text{simplify} \\ \text{collect, c, s} \end{array} \right.$$

$$c^4 \cdot Q_{11} + (2 \cdot Q_{12} + 4 \cdot Q_{66}) \cdot s^2 \cdot c^2 + s^4 \cdot Q_{22} - [c^4 \cdot Q_{11} + 2(Q_{12} + 2 \cdot Q_{66}) \cdot s^2 \cdot c^2 + s^4 \cdot Q_{22}] \text{ simplify } \rightarrow 0$$

$$c^4 \cdot Q_{12} + (Q_{11} + Q_{22} - 4 \cdot Q_{66}) \cdot s^2 \cdot c^2 + s^4 \cdot Q_{12} - [(Q_{11} + Q_{22} - 4 \cdot Q_{66}) \cdot s^2 \cdot c^2 + Q_{12} \cdot (s^4 + c^4)] \text{ simplify } \rightarrow 0$$

$$c^4 \cdot Q_{22} + (2 \cdot Q_{12} + 4 \cdot Q_{66}) \cdot s^2 \cdot c^2 + s^4 \cdot Q_{11} - [s^4 \cdot Q_{11} + 2(Q_{12} + 2 \cdot Q_{66}) \cdot s^2 \cdot c^2 + c^4 \cdot Q_{22}] \text{ simplify } \rightarrow 0$$

$$(Q_{11} - 2 \cdot Q_{66} - Q_{12}) \cdot s \cdot c^3 + (2 \cdot Q_{66} + Q_{12} - Q_{22}) \cdot s^3 \cdot c - [(Q_{11} - Q_{12} - 2 \cdot Q_{66}) \cdot s \cdot c^3 + (Q_{12} - Q_{22} + 2 \cdot Q_{66}) \cdot s^3 \cdot c] \text{ simplify } \rightarrow 0$$

$$(2 \cdot Q_{66} + Q_{12} - Q_{22}) \cdot s \cdot c^3 + (Q_{11} - 2 \cdot Q_{66} - Q_{12}) \cdot s^3 \cdot c - [(Q_{11} - Q_{12} - 2 \cdot Q_{66}) \cdot s^3 \cdot c + (Q_{12} - Q_{22} + 2 \cdot Q_{66}) \cdot s \cdot c^3] \text{ simplify } \rightarrow 0$$

$$Q_{66} \cdot c^4 + (Q_{11} + Q_{22} - 2 \cdot Q_{66} - 2 \cdot Q_{12}) \cdot s^2 \cdot c^2 + Q_{66} \cdot s^4 - [(Q_{11} + Q_{22} - 2 \cdot Q_{12} - 2 \cdot Q_{66}) \cdot s^2 \cdot c^2 + Q_{66} \cdot (s^4 + c^4)] \text{ simplify } \rightarrow 0$$

$$Q_{11}^m = c^4 \cdot Q_{11} + 2(Q_{12} + 2 \cdot Q_{66}) \cdot s^2 \cdot c^2 + s^4 \cdot Q_{22}$$

$$Q_{12}^m = (Q_{11} + Q_{22} - 4 \cdot Q_{66}) \cdot s^2 \cdot c^2 + Q_{12} \cdot (s^4 + c^4)$$

$$Q_{22}^m = [s^4 \cdot Q_{11} + 2(Q_{12} + 2 \cdot Q_{66}) \cdot s^2 \cdot c^2 + c^4 \cdot Q_{22}]$$

$$Q_{16}^m = (Q_{11} - Q_{12} - 2 \cdot Q_{66}) \cdot s \cdot c^3 + (Q_{12} - Q_{22} + 2 \cdot Q_{66}) \cdot s^3 \cdot c$$

$$Q_{26}^m = (Q_{11} - Q_{12} - 2 \cdot Q_{66}) \cdot s^3 \cdot c + (Q_{12} - Q_{22} + 2 \cdot Q_{66}) \cdot s \cdot c^3$$

$$Q_{66}^m = (Q_{11} + Q_{22} - 2 \cdot Q_{12} - 2 \cdot Q_{66}) \cdot s^2 \cdot c^2 + Q_{66} \cdot (s^4 + c^4)$$

Compare with equation (2.80) of

Jones, Robert M.
Mechanics of Composite Materials
1975, Hemisphere Publishing Corporation

Resultant Laminate Forces and Moments

Membrane

$$F = \begin{pmatrix} F_x \\ F_y \\ F_{xy} \end{pmatrix}$$

$$\epsilon^o = \begin{pmatrix} \epsilon^o_x \\ \epsilon^o_y \\ \gamma^o_{xy} \end{pmatrix}$$

Bending

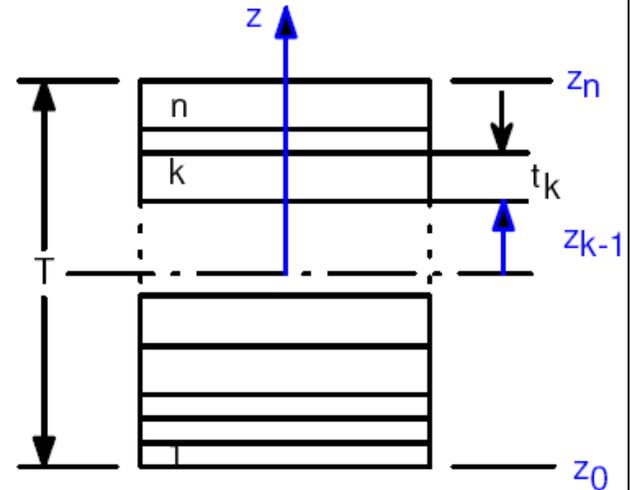
$$M = \begin{pmatrix} M_x \\ M_y \\ M_{xy} \end{pmatrix}$$

$$\chi = \begin{pmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{pmatrix}$$

Transverse Shear

$$Q = \begin{pmatrix} Q_x \\ Q_y \end{pmatrix}$$

$$\gamma = \begin{pmatrix} \gamma_{xz} \\ \gamma_{yz} \end{pmatrix}$$



Classical Lamination Theory (CLT)

$$\begin{pmatrix} F \\ M \end{pmatrix} = \begin{pmatrix} A & B \\ B & D \end{pmatrix} \begin{pmatrix} \epsilon^o \\ \chi \end{pmatrix}$$

MSC.Nastran

$$\begin{pmatrix} F \\ M \\ Q \end{pmatrix} = \begin{pmatrix} T \cdot G_1 & T^2 \cdot G_4 & 0 \\ T^2 \cdot G_4 & I \cdot G_2 & 0 \\ 0 & 0 & T_s \cdot G_3 \end{pmatrix} \begin{pmatrix} \epsilon^o \\ \chi \\ \gamma \end{pmatrix}$$

$$T = \sum_{k=1}^n t_k \quad I = \frac{T^3}{12} \quad T_s = T$$

$$k = 1 \dots n$$

$$z_0 = -\frac{T}{2}$$

$$z_k = z_{k-1} + t_k$$

$$\mu_k = \frac{1}{2} \cdot (z_{k-1} + z_k)$$

Strain Distribution over the Laminate Thickness

$$\begin{pmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{pmatrix} = \begin{pmatrix} \varepsilon^{\circ}_x \\ \varepsilon^{\circ}_y \\ \gamma^{\circ}_{xy} \end{pmatrix} + z \cdot \begin{pmatrix} \chi_x \\ \chi_y \\ \chi_{xy} \end{pmatrix} \quad \varepsilon = \varepsilon^{\circ} + z \cdot \chi \quad \sigma = Q^m \cdot \varepsilon = Q^m(\varepsilon^{\circ} + z \cdot \chi)$$

Forces

$$\begin{pmatrix} F_x \\ F_y \\ F_{xy} \end{pmatrix} = \int_{z_0}^{z_n} \begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix} dz = \int_{z_0}^{z_n} Q^m(\varepsilon^{\circ} + z \cdot \chi) dz = \left(\int_{z_0}^{z_n} Q^m dz \right) \cdot \varepsilon^{\circ} + \left(\int_{z_0}^{z_n} Q^m \cdot z dz \right) \cdot \chi = A \cdot \varepsilon^{\circ} + B \cdot \chi$$

$$A = \int_{z_0}^{z_n} Q(z)^m dz = \sum_{k=1}^n \left(Q_k^m \cdot \int_{z_{k-1}}^{z_k} 1 dz \right) = \sum_{k=1}^n \left[Q_k^m \cdot (z_k - z_{k-1}) \right] = \sum_{k=1}^n \left(Q_k^m \cdot t_k \right)$$

$$B = \int_{z_0}^{z_n} Q(z)^m \cdot z dz = \sum_{k=1}^n \left(Q_k^m \cdot \int_{z_{k-1}}^{z_k} z dz \right) = \sum_{k=1}^n \left[Q_k^m \cdot \frac{1}{2} \cdot [(z_k)^2 - (z_{k-1})^2] \right] = \sum_{k=1}^n \left(Q_k^m \cdot t_k \cdot \mu_k \right)$$

Moments

$$\begin{pmatrix} M_x \\ M_y \\ M_{xy} \end{pmatrix} = \int_{z_0}^{z_n} \begin{pmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{pmatrix} \cdot z dz = \int_{z_0}^{z_n} Q^m(\varepsilon^{\circ} + z \cdot \chi) \cdot z dz = \left(\int_{z_0}^{z_n} Q^m \cdot z dz \right) \cdot \varepsilon^{\circ} + \left(\int_{z_0}^{z_n} Q^m \cdot z^2 dz \right) \cdot \chi = B \cdot \varepsilon^{\circ} + D \cdot \chi$$

$$D = \int_{z_0}^{z_n} Q(z)^m \cdot z^2 dz = \sum_{k=1}^n \left(Q_k^m \cdot \int_{z_{k-1}}^{z_k} z^2 dz \right) = \sum_{k=1}^n \left[Q_k^m \cdot \frac{1}{3} \cdot \left[(z_k)^3 - (z_{k-1})^3 \right] \right] = \sum_{k=1}^n \left[Q_k^m \cdot t_k \cdot \left[(\mu_k)^2 + \frac{1}{12} \cdot (t_k)^2 \right] \right]$$

$$\frac{1}{3} \left[\left(\mu_k + \frac{t_k}{2} \right)^3 - \left(\mu_k - \frac{t_k}{2} \right)^3 \right] \text{ simplify } \rightarrow (\mu_k)^2 \cdot t_k + \frac{1}{12} \cdot (t_k)^3$$

MSC.Nastran equivalent 2D anisotropic Material (MAT2)

$$G = \begin{pmatrix} G_{11} & G_{12} & G_{13} \\ G_{12} & G_{22} & G_{23} \\ G_{13} & G_{23} & G_{33} \end{pmatrix}$$

$$G_1 = \frac{1}{T} \cdot A = \frac{1}{T} \cdot \sum_{k=1}^n \left(Q_k^m \cdot t_k \right)$$

$$G_2 = \frac{1}{I} \cdot D = \frac{1}{I} \cdot \sum_{k=1}^n \left[Q_k^m \cdot t_k \cdot \left[(\mu_k)^2 + \frac{1}{12} \cdot (t_k)^2 \right] \right]$$

$$G_4 = -\frac{1}{T^2} \cdot B = -\frac{1}{T^2} \cdot \sum_{k=1}^n \left(Q_k^m \cdot t_k \cdot \mu_k \right)$$

Numerical Example: 2-ply Laminate

Material: Graphite-epoxy

$$E_1 := 20 \cdot 10^6$$

$$E_2 := .5 \cdot 10^6$$

$$\nu_{12} := .25$$

$$G_{12} := .25 \cdot 10^6$$

$$\nu_{21} := \frac{E_2}{E_1} \cdot \nu_{12}$$

$$\nu_{21} = 0.00625$$

$$t_1 := 0.02$$

$$t_2 := 0.02$$

$$\theta_1 := 0$$

$$\theta_2 := \frac{\pi}{2}$$

$$Q := \begin{pmatrix} \frac{E_1}{1 - \nu_{12} \cdot \nu_{21}} & \frac{E_1 \cdot \nu_{21}}{1 - \nu_{12} \cdot \nu_{21}} & 0 \\ \frac{E_2 \cdot \nu_{12}}{1 - \nu_{12} \cdot \nu_{21}} & \frac{E_2}{1 - \nu_{12} \cdot \nu_{21}} & 0 \\ 0 & 0 & G_{12} \end{pmatrix}$$

$$Q = \begin{pmatrix} 2.00313E+007 & 1.25196E+005 & 0.00000E+000 \\ 1.25196E+005 & 5.00782E+005 & 0.00000E+000 \\ 0.00000E+000 & 0.00000E+000 & 2.50000E+005 \end{pmatrix}$$

$$U_\varepsilon(\theta) := \begin{pmatrix} \cos(\theta)^2 & \sin(\theta)^2 & \sin(\theta) \cdot \cos(\theta) \\ \sin(\theta)^2 & \cos(\theta)^2 & -\sin(\theta) \cdot \cos(\theta) \\ -2 \sin(\theta) \cdot \cos(\theta) & 2 \sin(\theta) \cdot \cos(\theta) & \cos(\theta)^2 - \sin(\theta)^2 \end{pmatrix}$$

$$U_\varepsilon(\theta_1) = \begin{pmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{pmatrix}$$

$$U_\varepsilon(\theta_2) = \begin{pmatrix} 0 & 1 & 0 \\ 1 & 0 & 0 \\ 0 & 0 & -1 \end{pmatrix}$$

$$\overset{\text{ww}}{T} := t_1 + t_2 \quad z_0 := \frac{-T}{2} \quad z_1 := z_0 + t_1 \quad \mu_1 := \frac{1}{2} \cdot (z_0 + z_1) \quad z_2 := z_1 + t_2 \quad \mu_2 := \frac{1}{2} \cdot (z_1 + z_2) \quad I := \frac{T^3}{12}$$

$$T = 0.04 \quad z_0 = -0.02 \quad z_1 = 0 \quad \mu_1 = -0.01 \quad z_2 = 0.02 \quad \mu_2 = 0.01 \quad I = 5.333 \times 10^{-6}$$

$$\overset{\text{ww}}{A} := U_\varepsilon(\theta_1)^T \cdot Q \cdot U_\varepsilon(\theta_1) \cdot t_1 + U_\varepsilon(\theta_2)^T \cdot Q \cdot U_\varepsilon(\theta_2) \cdot t_2 \quad B := U_\varepsilon(\theta_1)^T \cdot Q \cdot U_\varepsilon(\theta_1) \cdot t_1 \cdot \mu_1 + U_\varepsilon(\theta_2)^T \cdot Q \cdot U_\varepsilon(\theta_2) \cdot t_2 \cdot \mu_2$$

$$D := U_\varepsilon(\theta_1)^T \cdot Q \cdot U_\varepsilon(\theta_1) \cdot t_1 \cdot \left(\mu_1^2 + \frac{1}{12} \cdot t_1^2 \right) + U_\varepsilon(\theta_2)^T \cdot Q \cdot U_\varepsilon(\theta_2) \cdot t_2 \cdot \left(\mu_2^2 + \frac{1}{12} \cdot t_2^2 \right)$$

$$G_1 := \frac{1}{T} \cdot A$$

$$G_2 := \frac{1}{I} \cdot D$$

$$G_4 := -\frac{1}{T^2} \cdot B$$

$$G_1 = \begin{pmatrix} 1.0266E+007 & 1.2520E+005 & 3.8089E-012 \\ 1.2520E+005 & 1.0266E+007 & 5.9412E-010 \\ 3.8089E-012 & 5.9412E-010 & 2.5000E+005 \end{pmatrix}$$

$$G_2 = \begin{pmatrix} 1.0266E+007 & 1.2520E+005 & 3.8089E-012 \\ 1.2520E+005 & 1.0266E+007 & 5.9412E-010 \\ 3.8089E-012 & 5.9412E-010 & 2.5000E+005 \end{pmatrix}$$

$$G_4 = \begin{pmatrix} 2.4413E+006 & 0.0000E+000 & -9.5223E-013 \\ 0.0000E+000 & -2.4413E+006 & -1.4853E-010 \\ -9.5223E-013 & -1.4853E-010 & 0.0000E+000 \end{pmatrix}$$

MSC.Nastran

```

PSHELL      100    100000100    4.0000E-02    200000100    1.0000E+00    300000100    1.0000E+00    0.0000E+00
            -2.0000E-02    2.0000E-02    400000100
MAT2         100000100    1.0266E+07    1.2520E+05    1.7622E-11    1.0266E+07    2.7486E-09    2.5000E+05    0.0000E+00
            0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
            0
MAT2         200000100    1.0266E+07    1.2520E+05    1.7622E-11    1.0266E+07    2.7486E-09    2.5000E+05    0.0000E+00
            0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
            0
MAT2         300000100    1.3457E+05    2.7617E-11    0.0000E+00    1.3457E+05    0.0000E+00    0.0000E+00    0.0000E+00
            0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
            0
MAT2         400000100    2.4413E+06    0.0000E+00   -4.4054E-12   -2.4413E+06   -6.8716E-10    0.0000E+00    0.0000E+00
            0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00    0.0000E+00
            0

```

Verification of the G3 matrix (MAT2,300000100) will be shown later!

Transverse Shear Stiffness Coefficients

Transverse Shear Stiffness defined with MAT8

$$\begin{pmatrix} \tau_{1z} \\ \tau_{2z} \end{pmatrix} = \begin{pmatrix} G_{1z} & 0 \\ 0 & G_{2z} \end{pmatrix} \cdot \begin{pmatrix} \gamma_{1z} \\ \gamma_{2z} \end{pmatrix} = G_3^{12} \cdot \begin{pmatrix} \gamma_{1z} \\ \gamma_{2z} \end{pmatrix}$$

Transformation of the Transverse Shear Moduli Matrix in the Material Coordinate System

$$G_3^{xy} = W^T \cdot G_3^{12} \cdot W$$

$$W = \begin{pmatrix} \cos(\theta) & \sin(\theta) \\ -\sin(\theta) & \cos(\theta) \end{pmatrix}$$

Attention: To be consistent with

MSC.nastran V2004 Reference Manual, chapter 4.5

the letter V instead of Q is used for the transverse shear force

Transverse Shear Strain Energy

$$U = \frac{1}{2} \cdot \frac{V^2}{G^m \cdot T} = \frac{1}{2} \int \frac{\tau(z)^2}{G(z)} dz$$

V resultant transverse shear force

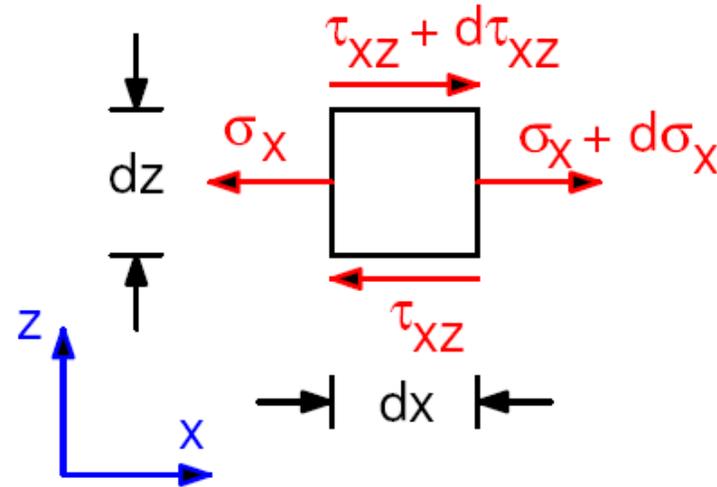
G^m mean value for transverse shear modulus

Specialization for the x-direction

$$\frac{1}{G_x^m} = \frac{T}{V_x^2} \sum_{i=1}^n \int_{z_{i-1}}^{z_i} \frac{(\tau_{xz}(z) \langle \dot{\psi} \rangle)^2}{G_{xz} \langle \dot{\psi} \rangle} dz$$

Equilibrium in x-direction

$$\frac{\partial}{\partial x} \sigma_x + \frac{\partial}{\partial z} \tau_{xz} = 0$$



In Elementary Beam Theory Bending Moment M_x and Transverse Shear Force V_x are related by

$$V_x + \frac{\partial}{\partial x} M_x = 0$$

Bending Stress σ_x

$$\sigma_x = \frac{E_x (\zeta_x - z)}{EI_x^a} M_x$$

ζ_x z coordinate of neutral axis

EI_x^a average bending stiffness

Interlaminar Shear Stresses

$$\frac{\partial}{\partial z} \tau_{xz} = \frac{\partial}{\partial x} \sigma_x = -\frac{E_x (\zeta_x - z)}{EI_x^a} \cdot \frac{\partial}{\partial x} M_x = \frac{E_x (\zeta_x - z)}{EI_x^a} \cdot V_x$$

$$\int \frac{E_x (\zeta_x - z)}{EI_x^a} \cdot V_x dz \rightarrow \frac{E_x}{EI_x^a} \cdot V_x \left(\zeta_x \cdot z - \frac{1}{2} \cdot z^2 \right)$$

$$\tau_{xz}(z) = \frac{E_x}{EI_x^a} \cdot V_x \left(\zeta_x \cdot z - \frac{1}{2} \cdot z^2 \right) + C \quad C \text{ Integration Constant}$$

For the arbitrary ply i the last equation may be written

$$\tau_{xz}(z)^{\langle i \rangle} = \frac{V_x}{EI_x^a} \cdot \left(\zeta_x \cdot z - \frac{1}{2} \cdot z^2 \right) \cdot E_x^{\langle i \rangle} + C^{\langle i \rangle} \quad z_{i-1} \leq z \leq z_i$$

Evaluation of Integration Constants

Ply 1

$$\tau_{xz}(z_0)^{\langle 1 \rangle} = 0 \quad \text{Bottom Surface is free of stresses!}$$

$$C^{\langle 1 \rangle} = -\frac{V_x}{EI_x^a} \cdot \left(\zeta_x \cdot z_0 - \frac{1}{2} \cdot z_0^2 \right) \cdot E_x^{\langle 1 \rangle}$$

$$\tau_{xz}(z)^{(1)} = \frac{V_x}{EI_x^a} \left[\zeta_x \cdot (z - z_0) - \frac{1}{2} \cdot (z^2 - z_0^2) \right] \cdot E_x^{(1)}$$

$$\tau_{xz}(z_1)^{(1)} = \frac{V_x}{EI_x^a} \left[\zeta_x \cdot (z_1 - z_0) - \frac{1}{2} \cdot (z_1^2 - z_0^2) \right] \cdot E_x^{(1)} = \frac{V_x}{EI_x^a} \cdot \left(\zeta_x - \frac{\mu_1}{2} \right) \cdot E_x^{(1)} \cdot t_1$$

Ply 2

$$\tau_{xz}(z_1)^{(1)} = \tau_{xz}(z_1)^{(2)} \quad \text{Continuity of interlaminar shear stress!}$$

$$C^{(2)} = \tau_{xz}(z_1)^{(1)} - \frac{V_x}{EI_x^a} \cdot \left(\zeta_x \cdot z_1 - \frac{1}{2} \cdot z_1^2 \right) \cdot E_x^{(2)}$$

$$\tau_{xz}(z)^{(2)} = \tau_{xz}(z_1)^{(1)} + \frac{V_x}{EI_x^a} \cdot \left[\zeta_x \cdot (z - z_1) - \frac{1}{2} \cdot (z^2 - z_1^2) \right] \cdot E_x^{(2)}$$

$$\tau_{xz}(z_2)^{(2)} = \tau_{xz}(z_1)^{(1)} + \frac{V_x}{EI_x^a} \cdot \left[\zeta_x \cdot (z_2 - z_1) - \frac{1}{2} \cdot (z_2^2 - z_1^2) \right] \cdot E_x^{(2)} = \frac{V_x}{EI_x^a} \cdot (\zeta_x - \mu_1) \cdot E_x^{(1)} \cdot t_1 + \frac{V_x}{EI_x^a} \cdot (\zeta_x - \mu_2) \cdot E_x^{(2)} \cdot t_2$$

Ply i

$$\tau_{xz}(z_{i-1})^{(i-1)} = \tau_{xz}(z_{i-1})^{(i)} \quad \text{Continuity of interlaminar shear stress!}$$

$$\tau_{xz}(z)^{(i)} = \tau_{xz}(z_{i-1})^{(i-1)} + \frac{V_x}{EI_x^a} \cdot \left[\zeta_x \cdot (z - z_{i-1}) - \frac{1}{2} \cdot [z^2 - (z_{i-1})^2] \right] \cdot E_x^{(i)}$$

$$\tau_{xz}(z_i)^{(i)} = \frac{V_x}{EI_x^a} \cdot \sum_{j=1}^i \left[(\zeta_x - \mu_j) \cdot E_x^{(j)} \cdot t_j \right]$$

Ply n

$$\tau_{xz}(z_n)^{\langle n \rangle} = 0$$

Top Surface is free of stresses!

$$\frac{V_x}{EI_x^a} \cdot \sum_{j=1}^n [(\zeta_x - \mu_j) \cdot E_x^{\langle j \rangle} \cdot t_j] = 0$$

$$\zeta_x = \frac{\sum_{j=1}^n (E_x^{\langle j \rangle} \cdot t_j \cdot \mu_j)}{\sum_{j=1}^n (E_x^{\langle j \rangle} \cdot t_j)}$$

With the abbreviation

$$f_x^{\langle i \rangle} = \frac{1}{E_x^{\langle i \rangle}} \cdot \sum_{j=1}^{i-1} [(\zeta_x - \mu_j) \cdot E_x^{\langle j \rangle} \cdot t_j]$$

$$f_x^{\langle 1 \rangle} = 0$$

the shear stress for ply i may be written

$$\tau_{xz}(z)^{\langle i \rangle} = \frac{V_x}{EI_x^a} \cdot \left[f_x^{\langle i \rangle} + \zeta_x (z - z_{i-1}) - \frac{1}{2} \cdot [z^2 - (z_{i-1})^2] \right] \cdot E_x^{\langle i \rangle}$$

Evaluation of G_x^m

$$\frac{1}{G_x^m} = \frac{T}{V_x^2} \cdot \sum_{i=1}^n \int_{z_{i-1}}^{z_i} \frac{(\tau_{xz}(z)^{\langle i \rangle})^2}{G_{xz}^{\langle i \rangle}} dz = \frac{T}{(EI_x^a)^2} \cdot \sum_{i=1}^n \int_{z_{i-1}}^{z_i} \frac{\left[\left[f_x^{\langle i \rangle} + \zeta_x (z - z_{i-1}) - \frac{1}{2} \cdot [z^2 - (z_{i-1})^2] \right] \cdot E_x^{\langle i \rangle} \right]^2}{G_{xz}^{\langle i \rangle}} dz$$

$$\frac{1}{G_x^m} = \frac{T}{(EI_x^a)^2} \sum_{i=1}^n \left[\frac{(E_x \langle \dot{\psi} \rangle)^2}{G_{xz}} \int_{z_{i-1}}^{z_i} \left[f_x \langle \dot{\psi} \rangle + \zeta_x (z - z_{i-1}) - \frac{1}{2} [z^2 - (z_{i-1})^2] \right]^2 dz \right] = \frac{T}{(EI_x^a)^2} \sum_{i=1}^n \left(\frac{1}{G_{xz}} \cdot R_x \langle \dot{\psi} \rangle \right)$$

$$\left[f_x \langle \dot{\psi} \rangle + \zeta_x (z - z_{i-1}) - \frac{1}{2} [z^2 - (z_{i-1})^2] \right]^2 \quad \left| \begin{array}{l} \text{expand} \\ \text{collect, z} \end{array} \right. \blacksquare$$

$$\frac{1}{4} \cdot z^4 - \zeta_x z^3 + \left[\frac{-1}{2} (z_{i-1})^2 + \zeta_x z_{i-1} - f_x \langle \dot{\psi} \rangle + \zeta_x^2 \right] z^2 + \left[\zeta_x (z_{i-1})^2 + 2 \cdot f_x \langle \dot{\psi} \rangle \cdot \zeta_x - 2 \cdot \zeta_x^2 \cdot z_{i-1} \right] z \dots$$

$$+ \left(f_x \langle \dot{\psi} \rangle \right)^2 - \zeta_x (z_{i-1})^3 + \zeta_x^2 (z_{i-1})^2 - 2 \cdot f_x \langle \dot{\psi} \rangle \cdot \zeta_x z_{i-1} + f_x \langle \dot{\psi} \rangle \cdot (z_{i-1})^2 + \frac{1}{4} (z_{i-1})^4$$

$$\int z^4 dz \rightarrow \frac{1}{5} z^5 \quad \int z^3 dz \rightarrow \frac{1}{4} z^4 \quad \int z^2 dz \rightarrow \frac{1}{3} z^3 \quad \int z dz \rightarrow \frac{1}{2} z^2 \quad \int 1 dz \rightarrow z$$

$$\frac{(z_i)^5 - (z_{i-1})^5}{z_i - z_{i-1}} \text{ simplify } \rightarrow (z_i)^4 + z_{i-1} (z_i)^3 + (z_{i-1})^2 (z_i)^2 + (z_{i-1})^3 z_i + (z_{i-1})^4$$

$$\frac{(z_i)^4 - (z_{i-1})^4}{z_i - z_{i-1}} \text{ simplify } \rightarrow (z_i)^3 + z_{i-1} (z_i)^2 + (z_{i-1})^2 z_i + (z_{i-1})^3$$

$$\frac{(z_i)^3 - (z_{i-1})^3}{z_i - z_{i-1}} \text{ simplify } \rightarrow (z_i)^2 + z_{i-1} z_i + (z_{i-1})^2 \quad z_i - z_{i-1} = t_i$$

$$\frac{(z_i)^2 - (z_{i-1})^2}{z_i - z_{i-1}} \text{ simplify } \rightarrow z_i + z_{i-1}$$

$$R_x^{(\hat{\psi})} = \left(E_x^{(\hat{\psi})} \right)^2 \cdot t_i \left[\begin{aligned} & \frac{1}{20} \left[(z_i)^4 + z_{i-1} \cdot (z_i)^3 + (z_{i-1})^2 \cdot (z_i)^2 + (z_{i-1})^3 \cdot z_i + (z_{i-1})^4 \right] - \frac{1}{4} \cdot \zeta_x \left[(z_i)^3 + z_{i-1} \cdot (z_i)^2 + (z_{i-1})^2 \cdot z_i + (z_{i-1})^3 \right] \dots \\ & + \frac{1}{3} \left[-f_x^{(\hat{\psi})} - \frac{1}{2} \cdot (z_{i-1})^2 + \zeta_x \cdot z_{i-1} + \zeta_x^2 \right] \left[(z_i)^2 + z_{i-1} \cdot z_i + (z_{i-1})^2 \right] + \frac{1}{2} \left[\zeta_x \cdot (z_{i-1})^2 + 2 \cdot f_x^{(\hat{\psi})} \cdot \zeta_x - 2 \cdot \zeta_x^2 \cdot z_{i-1} \right] \cdot (z_i + z_{i-1}) \dots \\ & + (-2) \cdot f_x^{(\hat{\psi})} \cdot \zeta_x \cdot z_{i-1} - \zeta_x \cdot (z_{i-1})^3 + f_x^{(\hat{\psi})} \cdot (z_{i-1})^2 + \frac{1}{4} \cdot (z_{i-1})^4 + \left(f_x^{(\hat{\psi})} \right)^2 + \zeta_x^2 \cdot (z_{i-1})^2 \end{aligned} \right]$$

$$\begin{aligned} & \frac{1}{20} \left[(z_i)^4 + z_{i-1} \cdot (z_i)^3 + (z_{i-1})^2 \cdot (z_i)^2 + (z_{i-1})^3 \cdot z_i + (z_{i-1})^4 \right] - \frac{1}{4} \cdot \zeta_x \left[(z_i)^3 + z_{i-1} \cdot (z_i)^2 + (z_{i-1})^2 \cdot z_i + (z_{i-1})^3 \right] \dots \\ & + \frac{1}{3} \left[-f_x^{(\hat{\psi})} - \frac{1}{2} \cdot (z_{i-1})^2 + \zeta_x \cdot z_{i-1} + \zeta_x^2 \right] \left[(z_i)^2 + z_{i-1} \cdot z_i + (z_{i-1})^2 \right] + \frac{1}{2} \left[\zeta_x \cdot (z_{i-1})^2 + 2 \cdot f_x^{(\hat{\psi})} \cdot \zeta_x - 2 \cdot \zeta_x^2 \cdot z_{i-1} \right] \cdot (z_i + z_{i-1}) \dots \\ & + (-2) \cdot f_x^{(\hat{\psi})} \cdot \zeta_x \cdot z_{i-1} - \zeta_x \cdot (z_{i-1})^3 + f_x^{(\hat{\psi})} \cdot (z_{i-1})^2 + \frac{1}{4} \cdot (z_{i-1})^4 + \left(f_x^{(\hat{\psi})} \right)^2 + \zeta_x^2 \cdot (z_{i-1})^2 \end{aligned} \quad \left. \begin{array}{l} \text{substitute, } z_i = z_{i-1} + t_i \\ \text{expand} \\ \text{collect, } f_x^{(\hat{\psi})}, \zeta_x \end{array} \right\}$$

$$\left(f_x^{(\hat{\psi})} \right)^2 + \left[t_i \cdot \zeta_x - z_{i-1} \cdot t_i - \frac{1}{3} \cdot (t_i)^2 \right] \cdot f_x^{(\hat{\psi})} + \frac{1}{3} \cdot \zeta_x^2 \cdot (t_i)^2 + \left[\frac{-2}{3} \cdot z_{i-1} \cdot (t_i)^2 - \frac{1}{4} \cdot (t_i)^3 \right] \cdot \zeta_x + \frac{1}{20} \cdot (t_i)^4 + \frac{1}{4} \cdot z_{i-1} \cdot (t_i)^3 + \frac{1}{3} \cdot (z_{i-1})^2 \cdot (t_i)^2$$

$$\left(f_x^{(\hat{\psi})} \right)^2 + \left[t_i \cdot \zeta_x - z_{i-1} \cdot t_i - \frac{1}{3} \cdot (t_i)^2 \right] \cdot f_x^{(\hat{\psi})} - \left[f_x^{(\hat{\psi})} + \left[t_i \cdot (\zeta_x - z_{i-1}) - \frac{1}{3} \cdot (t_i)^2 \right] \right] \cdot f_x^{(\hat{\psi})} \text{ simplify } \rightarrow 0$$

$$\frac{1}{3} \cdot \zeta_x^2 \cdot (t_i)^2 + \left[\frac{-2}{3} \cdot z_{i-1} \cdot (t_i)^2 - \frac{1}{4} \cdot (t_i)^3 \right] \cdot \zeta_x - \left[\frac{1}{3} \cdot (\zeta_x - 2 \cdot z_{i-1}) - \frac{1}{4} \cdot t_i \right] \cdot \zeta_x \cdot (t_i)^2 \text{ simplify } \rightarrow 0$$

$$\frac{1}{20} \cdot (t_i)^4 + \frac{1}{4} \cdot z_{i-1} \cdot (t_i)^3 + \frac{1}{3} \cdot (z_{i-1})^2 \cdot (t_i)^2 - \left[\frac{1}{3} \cdot (z_{i-1})^2 + \frac{1}{4} \cdot z_{i-1} \cdot t_i + \frac{1}{20} \cdot (t_i)^2 \right] \cdot (t_i)^2 \text{ simplify } \rightarrow 0$$

$$R_x^{(i)} = \left(E_x^{(i)}\right)^2 \cdot t_i \left[f_x^{(i)} + \left[t_i \cdot (\zeta_x - z_{i-1}) - \frac{1}{3} \cdot (t_i)^2 \right] \cdot f_x^{(i)} + \left[\frac{1}{3} \cdot (\zeta_x - 2 \cdot z_{i-1}) - \frac{1}{4} \cdot t_i \right] \cdot \zeta_x \cdot (t_i)^2 + \left[\frac{1}{3} \cdot (z_{i-1})^2 + \frac{1}{4} \cdot z_{i-1} \cdot t_i + \frac{1}{20} \cdot (t_i)^2 \right] \cdot (t_i)^2 \right]$$

Compare: MSC.Nastran V2004 Reference Manual, Chapter 4.5, Equation 4-52

Verification for a single layer element

$$E_x^{(1)} = E \quad G_{xz}^{(1)} = G \quad EI_x^a = \frac{E \cdot T^3}{12} = T \quad z_0 = -\frac{T}{2} \cdot 0 \quad f_x^{(1)} = 0 := T$$

$$\left[\frac{1}{3} \cdot (z_0)^2 + \frac{1}{4} \cdot z_0 \cdot t_1 + \frac{1}{20} \cdot (t_1)^2 \right] \cdot (t_1)^2 \left| \begin{array}{l} \text{substitute, } z_0 = -\frac{T}{2} \\ \text{substitute, } t_1 = T \end{array} \right. \rightarrow \frac{1}{120} \cdot T^4 \quad R_x^{(1)} = \frac{E^2 \cdot T^5}{120}$$

$$\frac{1}{G_x^m} = \frac{T}{\left(EI_x^a\right)^2} \cdot \frac{R_x^{(1)}}{G_{xz}^{(1)}} \quad \frac{T}{\left(\frac{E \cdot T^3}{12}\right)^2} \cdot \frac{\frac{E^2 \cdot T^5}{120}}{G} \text{ simplify } \rightarrow \frac{6}{5 \cdot G} \quad G_x^m = \frac{5}{6} \cdot G \quad \frac{5}{6} = 0.833333 \quad \text{Shear factor for solid plate!}$$

$$\frac{1}{G_x^m} = \frac{T}{\left(EI_x^a\right)^2} \cdot \sum_{i=1}^n \left(\frac{1}{G_{xz}^{(i)}} \cdot R_x^{(i)} \right) \quad G_x^m = \left[\frac{T}{\left(EI_x^a\right)^2} \cdot \sum_{i=1}^n \left(\frac{1}{G_{xz}^{(i)}} \cdot R_x^{(i)} \right) \right]^{-1}$$

A similar derivation can be done for the y-direction. In MSC.Nastran the result is generalized to:

$$G_{kl}^m = \left[\frac{T}{(EI_{kk}^a)^2} \sum_{i=1}^n \left(\frac{1}{G_{kl}^i} R_k^i \right) \right]^{-1}$$

$$k = x, y$$

$$l = x, y$$

To evaluate the average bending stiffness EI_{kk}^a the CLT equations must be specialized for pure Bending ($N = 0$):

$$M = (D - B \cdot A^{-1} \cdot B) \cdot \chi = D^* \cdot \chi$$

$$EI_{xx}^a = 1,1 \text{ term of } D^*$$

The D^* matrix has to be built with Poisson's Ratio $\nu_{12} = 0$

$$EI_{yy}^a = 2,2 \text{ term of } D^*$$

Attention: The matrix D^* is not documented in the MSC.Nastran Reference Manual or the Composites Seminar Notes!

In general

$$G_{xy}^m \neq G_{yx}^m$$

and therefore an average will be used to get the final result:

$$G_3 = \begin{bmatrix} G_{xx}^m & \frac{1}{2}(G_{xy}^m + G_{yx}^m) \\ \frac{1}{2}(G_{xy}^m + G_{yx}^m) & G_{yy}^m \end{bmatrix}$$

Continuation of Numerical Example: 2-ply Laminate

$$G_{1z} = 0.25 \cdot 10^6 \quad G_{2z} = .1 \cdot 10^6 \quad G_{ts} := \begin{pmatrix} G_{1z} & 0 \\ 0 & G_{2z} \end{pmatrix} \quad \underline{\underline{W}}(\theta) := \begin{pmatrix} \cos(\theta) & \sin(\theta) \\ -\sin(\theta) & \cos(\theta) \end{pmatrix}$$

Transformation to Material Coordinates

$$z_0 = -0.02 \quad z_1 = 0 \quad z_2 = 0.02 \quad T = 0.04$$

Ply 1
$$\underline{W}(\theta_1)^T \cdot G_{ts} \cdot \underline{W}(\theta_1) = \begin{pmatrix} 2.5 \times 10^5 & 0 \\ 0 & 1 \times 10^5 \end{pmatrix}$$

$$t_1 = 0.02 \quad \mu_1 = -0.01$$

$$G_{xx}^{(1)} = 2.5 \cdot 10^5 \quad G_{yy}^{(1)} = 1 \cdot 10^5 \quad G_{xy}^{(1)} = 0$$

$$E_x^{(1)} = 20 \cdot 10^6 \quad E_y^{(1)} = .5 \cdot 10^6$$

Ply 2
$$\underline{W}(\theta_2)^T \cdot G_{ts} \cdot \underline{W}(\theta_2) = \begin{pmatrix} 1 \times 10^5 & 9.185 \times 10^{-12} \\ 9.185 \times 10^{-12} & 2.5 \times 10^5 \end{pmatrix}$$

$$t_2 = 0.02 \quad \mu_2 = 0.01$$

$$G_{xx}^{(2)} = 1 \cdot 10^5 \quad G_{yy}^{(2)} = 2.5 \cdot 10^5 \quad G_{xy}^{(2)} = 0$$

$$E_x^{(2)} = .5 \cdot 10^6 \quad E_y^{(2)} = 20 \cdot 10^6$$

$$\zeta_x = \frac{\sum_{j=1}^n (E_x^{(j)} \cdot t_j \cdot \mu_j)}{\sum_{j=1}^n (E_x^{(j)} \cdot t_j)} = \frac{20 \cdot 10^6 \cdot 0.02 \cdot (-0.01) + .5 \cdot 10^6 \cdot 0.02 \cdot 0.01}{20 \cdot 10^6 \cdot 0.02 + .5 \cdot 10^6 \cdot 0.02} = -9.512195 \times 10^{-3}$$

$$\zeta_y = \frac{\sum_{j=1}^n (E_y^{(j)} \cdot t_j \cdot \mu_j)}{\sum_{j=1}^n (E_y^{(j)} \cdot t_j)} = \frac{.5 \cdot 10^6 \cdot 0.02 \cdot (-0.01) + 20 \cdot 10^6 \cdot 0.02 \cdot 0.01}{.5 \cdot 10^6 \cdot 0.02 + 20 \cdot 10^6 \cdot 0.02} = 9.512195 \times 10^{-3}$$

$$f_x^{(i)} = \frac{1}{E_x^{(i)}} \cdot \sum_{j=1}^{i-1} [(\zeta_x - \mu_j) \cdot E_x^{(j)} \cdot t_j] \quad f_x^{(1)} = 0 \quad f_x^{(2)} = \frac{1}{.5 \cdot 10^6} \cdot [-9.512195 \cdot 10^{-3} - (-0.01)] \cdot 20 \cdot 10^6 \cdot .02 = 3.90244 \times 10^{-4}$$

$$f_y^{(i)} = \frac{1}{E_y^{(i)}} \cdot \sum_{j=1}^{i-1} [(\zeta_y - \mu_j) \cdot E_y^{(j)} \cdot t_j] \quad f_y^{(1)} = 0 \quad f_y^{(2)} = \frac{1}{20 \cdot 10^6} \cdot [9.512195 \cdot 10^{-3} - (-0.01)] \cdot .5 \cdot 10^6 \cdot .02 = 9.756098 \times 10^{-6}$$

$$R_x^{(i)} = (E_x^{(i)})^2 \cdot t_i \cdot \left[f_x^{(i)} + t_i \left(\zeta_x - z_{i-1} \right) - \frac{1}{3} \cdot (t_i)^2 \right] \cdot f_x^{(i)} + \left[\frac{1}{3} \cdot (\zeta_x - 2 \cdot z_{i-1}) - \frac{1}{4} \cdot t_i \right] \cdot \zeta_x \cdot (t_i)^2 + \left[\frac{1}{3} \cdot (z_{i-1})^2 + \frac{1}{4} \cdot z_{i-1} \cdot t_i + \frac{1}{20} \cdot (t_i)^2 \right] \cdot (t_i)^2$$

$$R_x^{(1)} = (20 \cdot 10^6)^2 \cdot 0.02 \cdot \left[\left[\frac{1}{3} \cdot [-9.512195 \times 10^{-3} - 2 \cdot (-0.02)] - \frac{1}{4} \cdot 0.02 \right] \cdot (-9.512195 \times 10^{-3}) \cdot 0.02^2 + \left[\frac{1}{3} \cdot (-0.02)^2 + \frac{1}{4} \cdot (-0.02) \cdot 0.02 + \frac{1}{20} \cdot 0.02^2 \right] \cdot 0.02^2 \right]$$

$$R_x^{(1)} = 1.352211 \times 10^4$$

$$R_x^{(2)} = \left[(.5 \cdot 10^6)^2 \cdot 0.02 \cdot \left[\left[3.90244 \cdot 10^{-4} + 0.02 \cdot (-9.512195 \cdot 10^{-3} - 0.0) - \frac{1}{3} \cdot 0.02^2 \right] \cdot 3.90244 \cdot 10^{-4} \dots \right. \right. \\ \left. \left. + \left[\frac{1}{3} \cdot (-9.512195 \cdot 10^{-3} - 2 \cdot 0.0) - \frac{1}{4} \cdot 0.02 \right] \cdot (-9.512195 \cdot 10^{-3}) \cdot 0.02^2 + \left(\frac{1}{3} \cdot 0.02^2 + \frac{1}{4} \cdot 0 \cdot 0.02 + \frac{1}{20} \cdot 0.02^2 \right) \cdot 0.02^2 \right] \right]$$

$$R_x^{(2)} = 3.255247 \times 10^2$$

$$R_y^{(1)} = (E_y^{(1)})^2 \cdot t_i \left[\left[f_y^{(1)} + t_i (\zeta_y - z_{i-1}) - \frac{1}{3} \cdot (t_i)^2 \right] \cdot f_y^{(1)} + \left[\frac{1}{3} \cdot (\zeta_y - 2 \cdot z_{i-1}) - \frac{1}{4} \cdot t_i \right] \cdot \zeta_y \cdot (t_i)^2 + \left[\frac{1}{3} \cdot (z_{i-1})^2 + \frac{1}{4} \cdot z_{i-1} \cdot t_i + \frac{1}{20} \cdot (t_i)^2 \right] \cdot (t_i)^2 \right]$$

$$R_y^{(1)} = (5 \cdot 10^6)^2 \cdot 0.02 \cdot \left[\left[\frac{1}{3} \left[9.512195 \times 10^{-3} - 2 \cdot (-0.02) \right] - \frac{1}{4} \cdot 0.02 \right] \cdot 9.512195 \times 10^{-3} \cdot 0.02^2 + \left[\frac{1}{3} \cdot (-0.02)^2 + \frac{1}{4} \cdot (-0.02) \cdot 0.02 + \frac{1}{20} \cdot 0.02^2 \right] \cdot 0.02^2 \right]$$

$$R_y^{(1)} = 3.255245 \times 10^2$$

$$R_y^{(2)} = (20 \cdot 10^6)^2 \cdot 0.02 \cdot \left[\left[9.756098 \times 10^{-6} + 0.02 \cdot (9.512195 \times 10^{-3} - 0.0) - \frac{1}{3} \cdot 0.02^2 \right] \cdot 9.756098 \times 10^{-6} \dots \right. \\ \left. + \left[\frac{1}{3} \left(9.512195 \times 10^{-3} - 2 \cdot 0.0 \right) - \frac{1}{4} \cdot 0.02 \right] \cdot 9.512195 \times 10^{-3} \cdot 0.02^2 + \left(\frac{1}{3} \cdot 0.0^2 + \frac{1}{4} \cdot 0.0 \cdot 0.02 + \frac{1}{20} \cdot 0.02^2 \right) \cdot 0.02^2 \right]$$

$$R_y^{(2)} = 1.352211 \times 10^4$$

Evaluation of the CLT matrices for $v_{12} = 0$:

$$Q_0 = \begin{pmatrix} E_1 & 0 & 0 \\ 0 & E_2 & 0 \\ 0 & 0 & G_{12} \end{pmatrix} \quad Q_0 = \begin{pmatrix} 2 \times 10^7 & 0 & 0 \\ 0 & 5 \times 10^5 & 0 \\ 0 & 0 & 2.5 \times 10^5 \end{pmatrix}$$

$$A_0 := U_\varepsilon(\theta_1)^T \cdot Q_0 \cdot U_\varepsilon(\theta_1) \cdot t_1 + U_\varepsilon(\theta_2)^T \cdot Q_0 \cdot U_\varepsilon(\theta_2) \cdot t_2 \quad B_0 := U_\varepsilon(\theta_1)^T \cdot Q_0 \cdot U_\varepsilon(\theta_1) \cdot t_1 \cdot \mu_1 + U_\varepsilon(\theta_2)^T \cdot Q_0 \cdot U_\varepsilon(\theta_2) \cdot t_2 \cdot \mu_2$$

$$D_0 := U_\varepsilon(\theta_1)^T \cdot Q_0 \cdot U_\varepsilon(\theta_1) \cdot t_1 \cdot \left(\mu_1^2 + \frac{1}{12} \cdot t_1^2 \right) + U_\varepsilon(\theta_2)^T \cdot Q_0 \cdot U_\varepsilon(\theta_2) \cdot t_2 \cdot \left(\mu_2^2 + \frac{1}{12} \cdot t_2^2 \right)$$

$$D^* := D_0 - B_0 \cdot A_0^{-1} \cdot B_0 \quad D^* = \begin{pmatrix} 1.756911 \times 10^1 & 0 \times 10^0 & 0 \times 10^0 \\ 0 \times 10^0 & 1.756911 \times 10^1 & 0 \times 10^0 \\ 0 \times 10^0 & 0 \times 10^0 & 1.333333 \times 10^0 \end{pmatrix}$$

$$G_{kl}^m = \left[\frac{T}{(EI_{kk}^a)^2} \sum_{i=1}^n \left(\frac{1}{G_{kl}^{\langle \psi \rangle}} \cdot R_k^{\langle \psi \rangle} \right) \right]^{-1}$$

$$EI_{xx}^a = 1,1 \text{ term of } D^* \qquad EI_{xx}^a = 1.756911 \times 10^1$$

$$G_{xx}^m = \frac{1}{\frac{0.04}{(1.756911 \times 10^1)^2} \left(\frac{1.352211 \cdot 10^4}{2.5 \cdot 10^5} + \frac{3.255247 \times 10^2}{1.0 \cdot 10^5} \right)} = 1.345718 \times 10^5$$

$$EI_{yy}^a = 2,2 \text{ term of } D^* \qquad EI_{yy}^a = 1.756911 \times 10^1$$

$$G_{yy}^m = \frac{1}{\frac{0.04}{(1.756911 \times 10^1)^2} \left(\frac{3.255245 \times 10^2}{1.0 \cdot 10^5} + \frac{1.352211 \times 10^4}{2.5 \cdot 10^5} \right)} = 1.345718 \times 10^5$$

MSC.Nastran

```

PSHELL      100      100000100      4.0000E-02      200000100      1.0000E+00      300000100      1.0000E+00      0.0000E+00
            -2.0000E-02      2.0000E-02      400000100

MAT2        300000100      1.3457E+05      2.7617E-11      0.0000E+00      1.3457E+05      0.0000E+00      0.0000E+00      0.0000E+00
            0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00      0.0000E+00
            0
    
```