

MSC.Software Corporation

Europe

MSC.Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone: (49) (89) 43 19 87 0
Fax: (49) (89) 43 61 71 6

Corporate

MSC.Software Corporation
2 MacArthur Place
Santa Ana, CA 92707 USA
Telephone: (800) 345-2078
Fax: (714) 784-4056

Asia Pacific

MSC.Software Japan Ltd.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjuku-Ku
Tokyo 160-0023, JAPAN
Telephone: (81) (3)-6911-1200
Fax: (81) (3)-6911-1201

MD Nastran Thermal Analysis

NAS104 Course Notes



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 © 2007 MSC.Software Corporation. All Rights Reserved. This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of MSC.Software Corporation is prohibited.

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

CONTENTS

<u>Section</u>	<u>Page</u>
1.0 Overview	1-1
Topics day-by-day.....	1-3
Capabilities.....	1-10
2.0 MD Nastran Thermal Analysis	2-1
Overview.....	2-3
Heat Transfer, Modes of.....	2-5
MD Nastran Thermal Preference Table (MD Patran).....	2-6
MD Nastran Finite Elements.....	2-7
MD Nastran Input File Entries.....	2-9
Format of the .DAT File.....	2-38
3.0 Conduction	3-1
Comments on Conduction.....	3-3
Conduction, Theory.....	3-4
Supported Element Topologies.....	3-5
CHEXA Conduction Element.....	3-9
MD Nastran Example Problems.....	3-13
Example 1a, Linear Conduction.....	3-14
Project 1, Steady State Conduction.....	3-17
Constant Conduction.....	3-18
Temperature Dependent Conduction.....	3-33
Workshop 1, Getting Started, Creating a Conduction Model.....	3-43
Conduction Through Anisotropic Materials.....	3-45
Element Coordinate Frames.....	3-47
Material Orientation Defaults.....	3-50

CONTENTS

<u>Section</u>	<u>Page</u>
3.0 Conduction (continued)	3-1
Anisotropic Conduction, MAT5.....	3-52
Project 2, Analysis With Orthotropic Materials.....	3-53
Material Relative to Zero deg Rotation.....	3-55
Material Relative to 45 deg Rotation.....	3-61
Conduction Through MD Nastran SOL 400 3D Composite Elements.....	3-65
Volumetric Heat Generation.....	3-73
3x3 CHEXA Mesh With QVOL Heat Entry.....	3-75
Volumetric Heat Generation (QVOL).....	3-78
Project 3, Volumetric Heating as a Function of Temperature.....	3-79
Workshop 2, Circuit Board and Chips Using Conduction and Heating.....	3-88
4.0 Convection	4-1
Heat Through Surface Element.....	4-3
Boundary Condition Element CHBDYE.....	4-5
Free Convection CONV.....	4-8
Surface Boundary Conditions.....	4-10
QBDY3.....	4-15
Thermal Loads.....	4-17
Free Convection	4-21
Example 1b, Nonlinear Free Convection Along the edge of a Rod.....	4-23
Example 1c, Convection as a Function of Temperature H(Temp).....	4-30
Project 4, Free Convection, H=H(Temp)	4-36
Forms in Free Convection.....	4-53
Free Convection Template.....	4-54
Workshop 3, Free Convection From Printed Circuit Board.....	4-56
Forced Convection	4-57
Forced Convection CONVM.....	4-59
Example 6, Forced Convection Tube Flow.....	4-62
Project 5, Part 1, 1D Flow Networks.....	4-65

CONTENTS

<u>Section</u>	<u>Page</u>
4.0 Convection (continued)	4-1
Project 5, Part 2, 1D Flow Networks, Use Correlation	4-76
Workshop 4, Forced Air Convection From Printed Circuit Board	4-80
Coupled Advection	4-81
Coupled Advection Resistors	4-83
Project 6, Coupled Advection	4-84
Project 6, Part 1, Coupled Advection, Coarse Mesh	4-88
Project 6, Part 2, Coupled Advection, Fine Mesh	4-111
Duct Flow	4-119
Duct Flow Topology	4-121
Project 7, Create Duct Flow	4-124
Project 7, Duct Flow in Patran	4-130
Difference Between Duct Flow and Coupled Advection	4-135
Workshop 5, Axisymmetric Flow in a Pipe	4-136
Workshop 6, Typical Avionics Flow	4-136
Workshop 7, Thermal Contact Resistance	4-136
5.0 Transient Thermal Analysis (SOL 159, 400, 600)	5-1
Transient Thermal Analysis	5-3
Theory and Capabilities	5-4
Time Dependent Temperature	5-6
Time Dependent Thermal Loads	5-8
Transient Thermal Analysis Bulk Data C_p and ρ	5-16
Transient Thermal Analysis Case Control	5-17
.DAT File Pointer Systems Example	5-18
Transient Thermal Analysis Bulk Data DLOAD	5-21
Transient Thermal Analysis Bulk Data TLOAD2	5-22
Transient Thermal Analysis Bulk Data TSTEPNL	5-28
Example 7d, Time Varying Heat Transfer Coefficient	5-32
Project 8, QVOL (Time)	5-42
Workshop 8, Transient Thermal	5-76
Workshop 9, Transient Thermal Analysis of a Cooling Fin	5-76
Workshop 10, Holman Example 4-14, Radiation Heating and Cooling of Ceramics	5-76
Workshop 11, Heating a Block of Ice Cream	5-76

CONTENTS

<u>Section</u>	<u>Page</u>
5.0 Transient Thermal Analysis (SOL 159, 400, 600) (continued)	5-1
Control by Thermostat	5-77
BC Control Through CNTRLND	5-79
Control Mechanism	5-80
Example A, Lumped Capacity Analysis With Thermostat Using QVOL	5-82
Example B, Thermostat Controller Using Radiation or Advection	5-97
Case 1, No Controller	5-99
Case 2, Cooling by Radiation	5-103
Case 3, Heating by Mass Flow Rate	5-108
Classical Heating Thermostat	5-113

CONTENTS

<u>Section</u>	Page
6.0 Radiation	6-1
Radiation Topics	6-3
Radiation Theory	6-4
Steady State Solution Method	6-7
Radiation Functionality	6-9
Ambient Radiation Exchange	6-13
Example 1E, Radiation Boundary Condition	6-17
Steady State Solution Control, NLPARM	6-26
Directional Heat Flux	6-27
Example 4B, Directional Solar Load With Radiation Boundary Condition	6-29
Directional Flux, MD Patran Interface	6-36
Project 9, Directional Heat Loads	6-37
Workshop 12, Radiation to Space	6-49
Workshop 13, Directional Heat Loads	6-49
Enclosure Radiation	6-51
View Factor Definition	6-53
Fundamentals of View Factor Calculations	6-54
View Factor Computation	6-57
Enclosure Radiation, View Factor	6-63
CHBDYG Surface of Cavity	6-65
View Groups Surface Elements	6-66
Gaussian Integration Method	6-69
Characteristics of Enclosure	6-72
Use Hemi-cube Method to Calculate View Factor	6-76
Comparison of View Factor Methods	6-81
Example 5A, Single Cavity Enclosure Radiation With Shadowing	6-83
Enclosures, Closed or Open	6-88
Project 10, Single Cavity Enclosure Radiation	6-90
Different Ambient Temperature	6-107

CONTENTS

<u>Section</u>	Page
6.0 Radiation (continued)	6-1
View Factors in the Punch File.....	6-111
Use the Existing View Factors.....	6-113
Workshop 14, Radiation Enclosures.....	6-114
Workshop 23, Pixel Hemi-cube View Factor Method.....	6-114
Axi-Symmetric View Factor.....	6-115
Axi-symmetric View Factor Example.....	6-117
Example C, Axi-Symmetric View Factor Calculation Cryogenic Heat Shielding.....	6-118
Project 11, Concentric Spheres for Axi-symmetric View Factors.....	6-135
Wavelength Dependent Emissivity.....	6-173
Two Elements With Radiation Exchange.....	6-183



CONTENTS

<u>Section</u>	Page
7.0 Additional Topics	7-1
Additional Topics	7-3
Thermal Stress Analysis	7-5
Workshop 15, Thermal Stress Analysis With Directional Heat Loads	7-18
Workshop 16, Thermal Stress Analysis of a Bi-metallic Plate	7-18
Linear Thermal Stress Analysis Using STRUCTURAL HEAT ANALYSIS (SOL 101)	7-19
Thermal Stress Analysis Using MD Nastran SOL 400	7-33
Thermal Contact	7-39
Restart Transient Thermal Analysis	7-55
Restart From Thermal Analysis to Stress Analysis	7-69
NOLINi for Thermostat Control	7-77
Use of TABLEMi	7-85
Radiation Matrix	7-93
Performance of View Factor Calculation	7-109
CHBDYi Boundary Condition Surface Elements	7-115
Workshop 17, Import IGES File and Auto tet Mesh	7-119
Workshop 18, Create Group and List	7-119
Workshop 19, Nastran Bulk Data Entries	7-119
Workshop 20, Radiation Shield Using View Entry	7-119
Workshop 21, Thermal Stress Form Transient Heat Transfer Run	7-119
8.0 Basic Thermal Analysis Theory	8-1
Brief Introduction to Heat Transfer	8-3
Brief Introduction to Conduction	8-13
Brief Introduction to Convection	8-27
Brief Introduction to Radiation	8-37

CONTENTS

<u>Section</u>	Page
9.0 Nonlinear Solvers	9-1
Nonlinear Solvers.....	9-3
Steady-state Thermal Analysis.....	9-5
Steady-state Solver Steps.....	9-7
Steady-state Convergence.....	9-14
Recommendations for Steady-state.....	9-24
Transient Thermal Analysis.....	9-27
Transient Solver Steps.....	9-30
Transient Convergence.....	9-39
Recommendations for Transient.....	9-49
10.0 Appendix A	10-1
MD Nastran Input Files.....	10-3
Cross References.....	10-8
New MD Nastran Input Files.....	10-9

SECTION 1

OVERVIEW



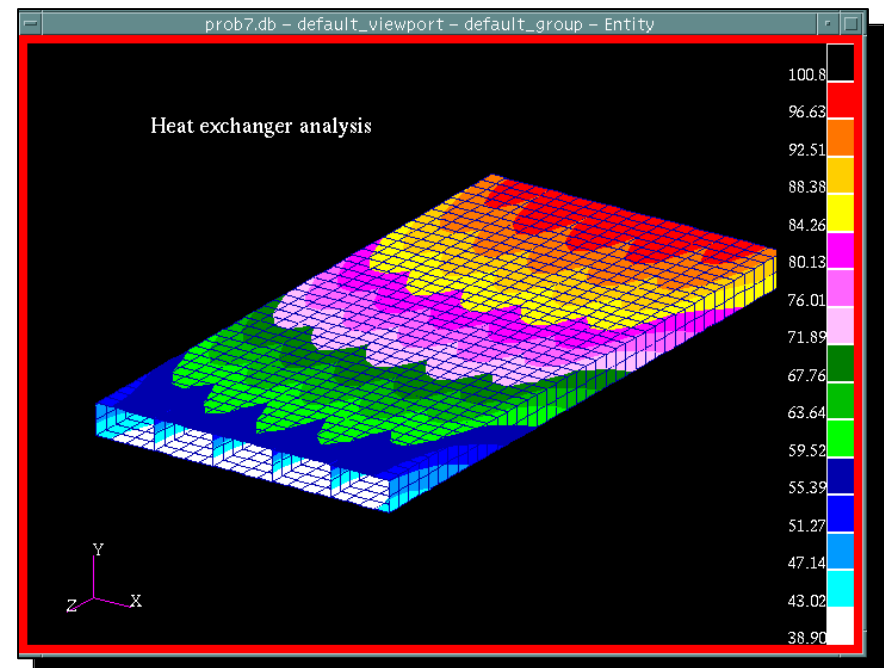
AGENDA

- DAY ONE

- MD NASTRAN Overview - Thermal analysis
- Linear Conduction and Nonlinear Conduction (Projects 1 and 1A)
- Anisotropic Material (Project 2)
- Free Convection (Projects 3)

- DAY TWO

- 1D Flow Network (Project 4)
- Coupled Advection (Project 5)
- Transient Thermal Analysis (Project 6)



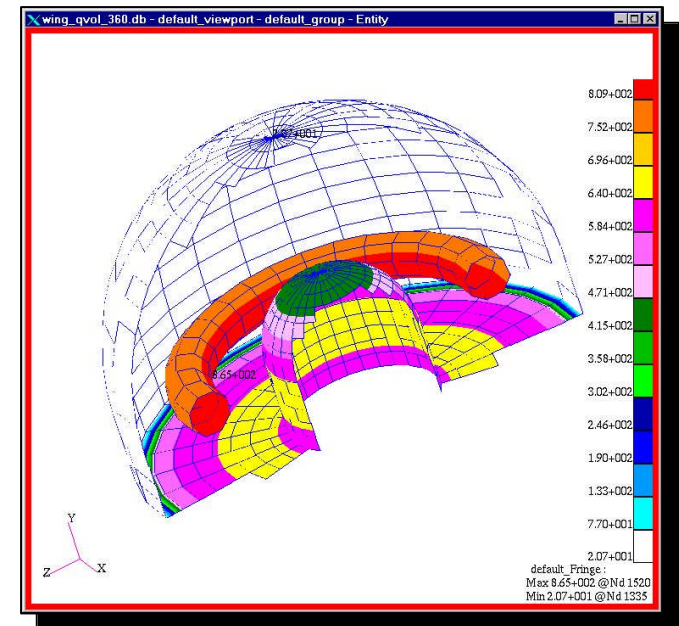
AGENDA

- DAY THREE

- Transient Thermal Analysis (Cont.)
- Thermostat Control
- Directional Heat Load with radiation loss to ambient (Project 7)
- Enclosure Radiation (Projects 8)
- View factor calculation
- Multiple cavities
- Axi-symmetry view factor

- DAY FOUR

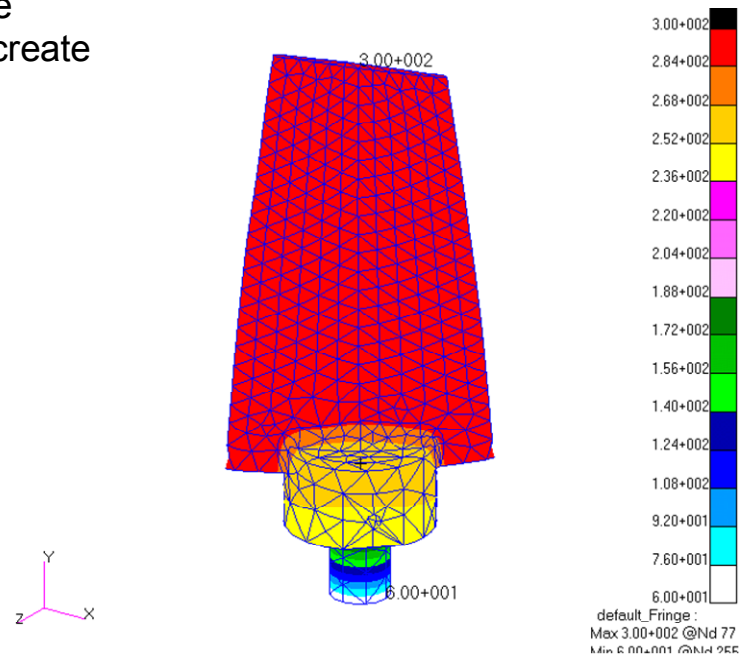
- Thermal stress analysis
- SOL 101 (Thermal and Stress Analysis in one run)
- Restart for Transient Thermal Analysis
- Radiation Matrix



IMPORT SOLID GEOMETRY

NASTRAN thermal result

- In PATRAN, you can use Create – Solid-B-rep to create a solid from the existing surfaces



Nastran Thermal Analysis, June 2007

S1-1

simulating REALITY™

MSC Software

NAS104, Section 1, September 2008

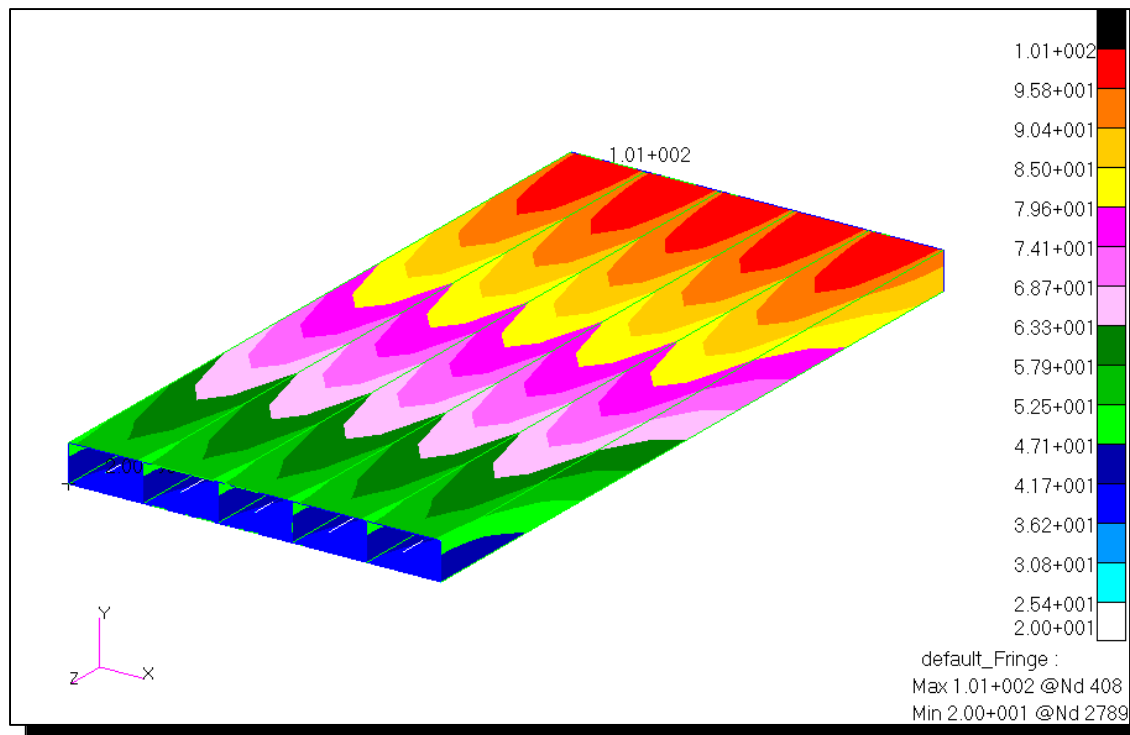
Copyright© 2008 MSC Software Corporation

S1-5

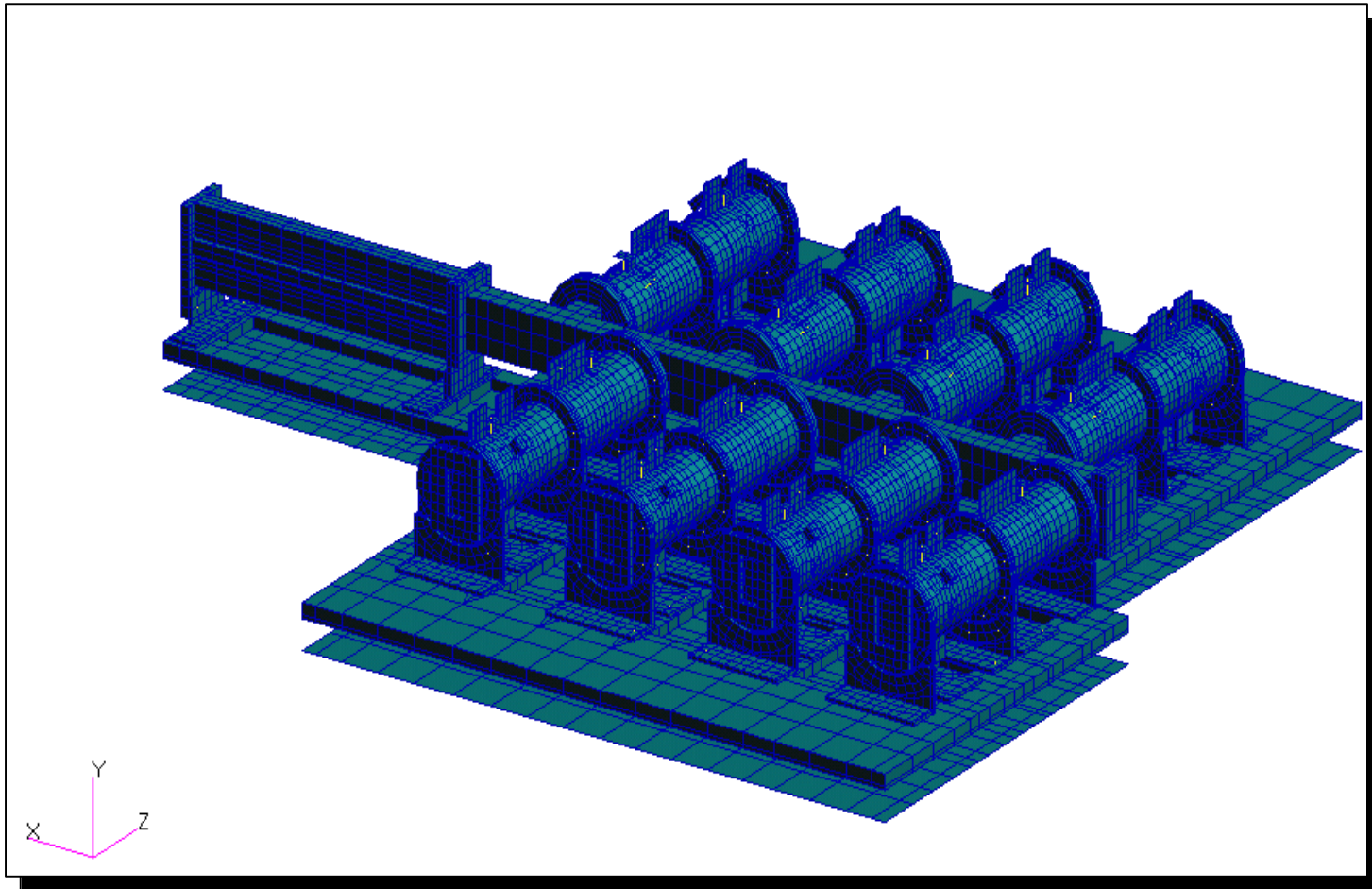
MSC Software

NASTRAN HEAT EXCHANGER MODEL

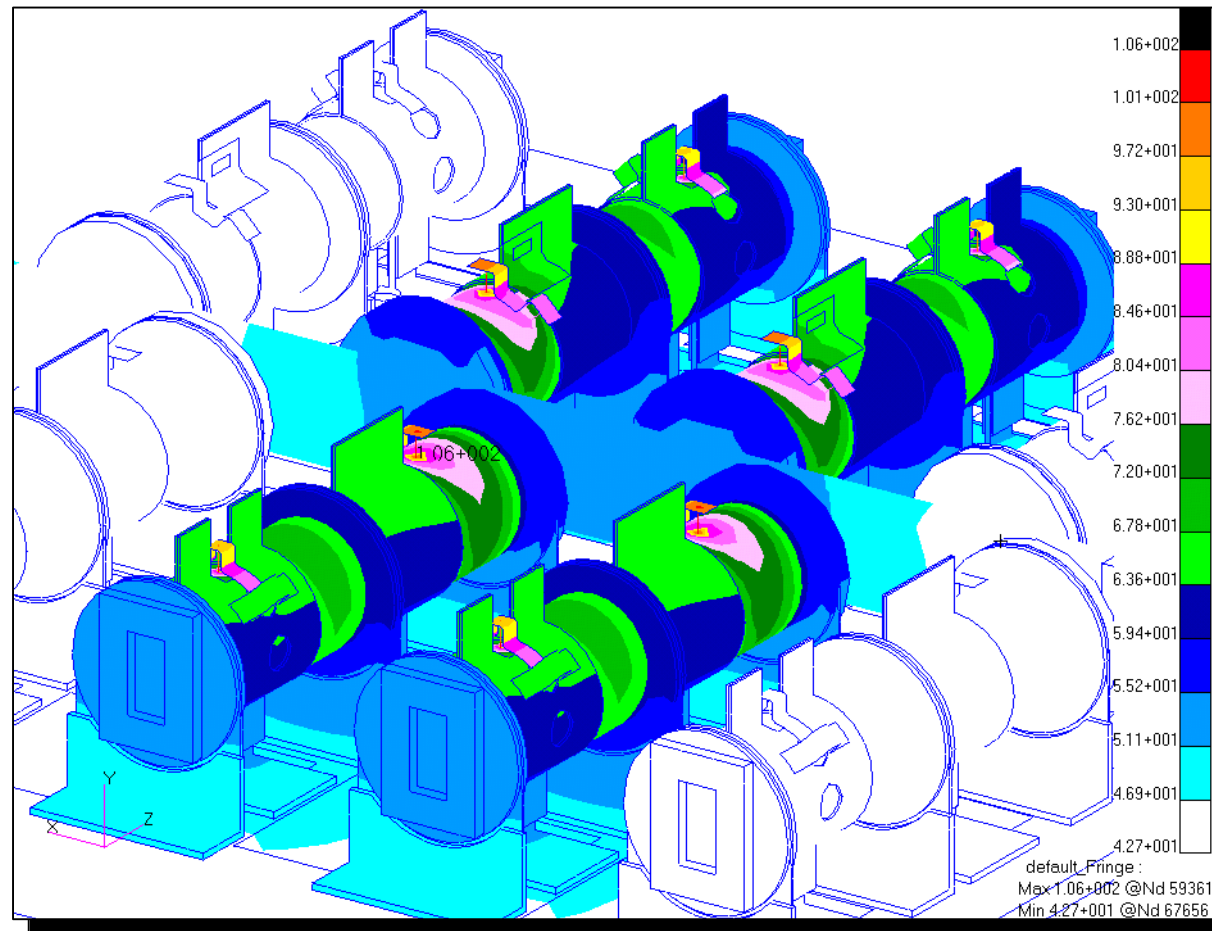
- Heat flux is applied on one side with 20 Watt/in².
- The inlet fluid temp is 20°, and the maximum surface temp is 100.8°.



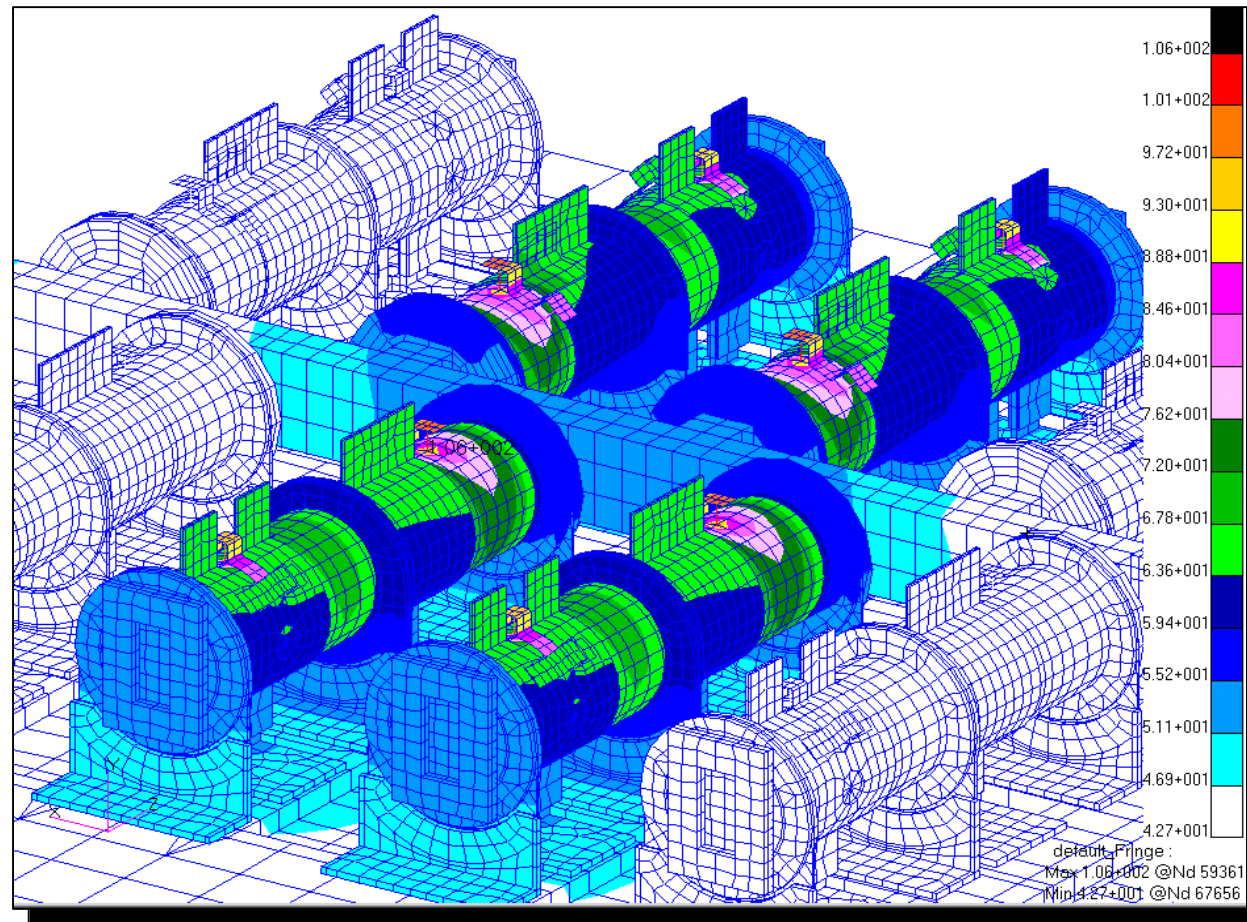
THERMAL RADIATION MODEL



THERMAL CONTOUR



THERMAL CONTOUR (Cont.)



CAPABILITIES

- Solution Methods
 - Steady state
 - Transient
- Heat conduction
 - Temperature-dependent conductivity
 - Temperature-dependent specific heat
 - Anisotropic thermal conductivity
 - Time-dependent internal heat generation
- Free convection boundaries
 - Temperature-dependent heat transfer coefficient
 - Time-dependent heat transfer coefficient
- Forced convection
 - Tube flow field models – $H(Pr, Re)$
 - Temperature-dependent fluid viscosity, density and specific heat
 - Time-dependent mass flow rate
 - Duct flow coupling to rectangular surface elements
- Radiation to space
 - Temperature-dependent emissivity and absorptivity
 - Time-dependent exchange
- Radiation enclosures
 - Temperature-dependent emissivity
 - Diffuse view factor calculations with self and third-body shadowing
 - Spectral selective surfaces, emissivity is a function of wavelength
 - Radiation enclosure control
 - Multiple radiation enclosures

CAPABILITIES (Cont.)

- Applied heat loads
 - Directional heat flux
 - Surface normal heat flux
 - Grid point nodal power
 - Temperature-dependent flux
 - Time-dependent heat flux
 - Volumetric heating for conduction element
- Temperature boundary conditions
 - Specified constant temperatures for steady state and transient
 - Specified time-varying temperatures for transient
- Initial conditions
 - Starting temperatures for nonlinear steady state analyses
 - Starting temperatures for all transient analyses



SECTION 2

MD NASTRAN THERMAL ANALYSIS



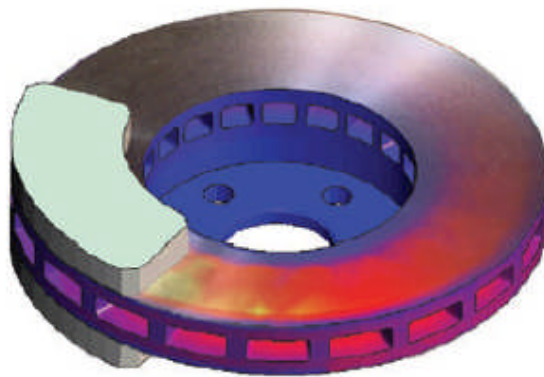


OVERVIEW

- Why a Structural Analyst may have to perform Thermal Analysis.
- Modes of heat transfer available in MD NASTRAN and MD Patran support:
 - Conduction
 - Convection
 - Radiation
 - Steady state analysis
 - Transient analysis
 - Linear versus nonlinear
 - Thermal stress analysis

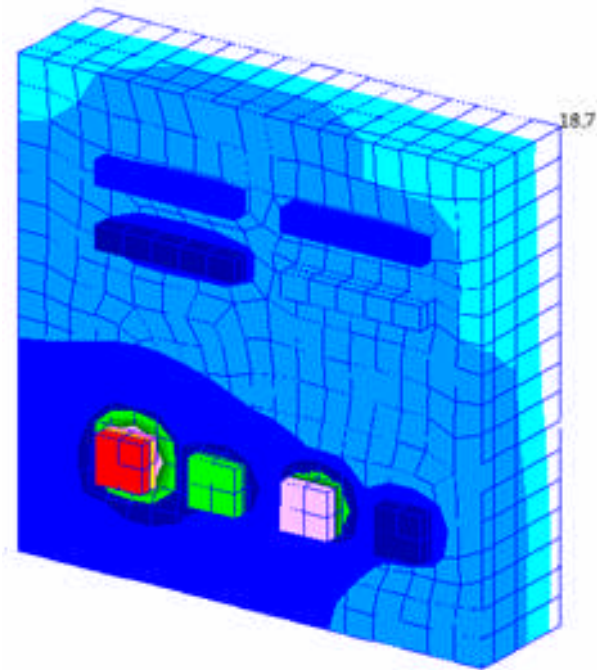
HEAT TRANSFER, YES OR NO

- Should a Heat Transfer analysis be performed ?
 - When the solution for the temperature field in a solid (or fluid) is desired, and the temperature is not influenced by other fields, a heat transfer analysis is appropriate.



Coupled Fields
(No)

VS



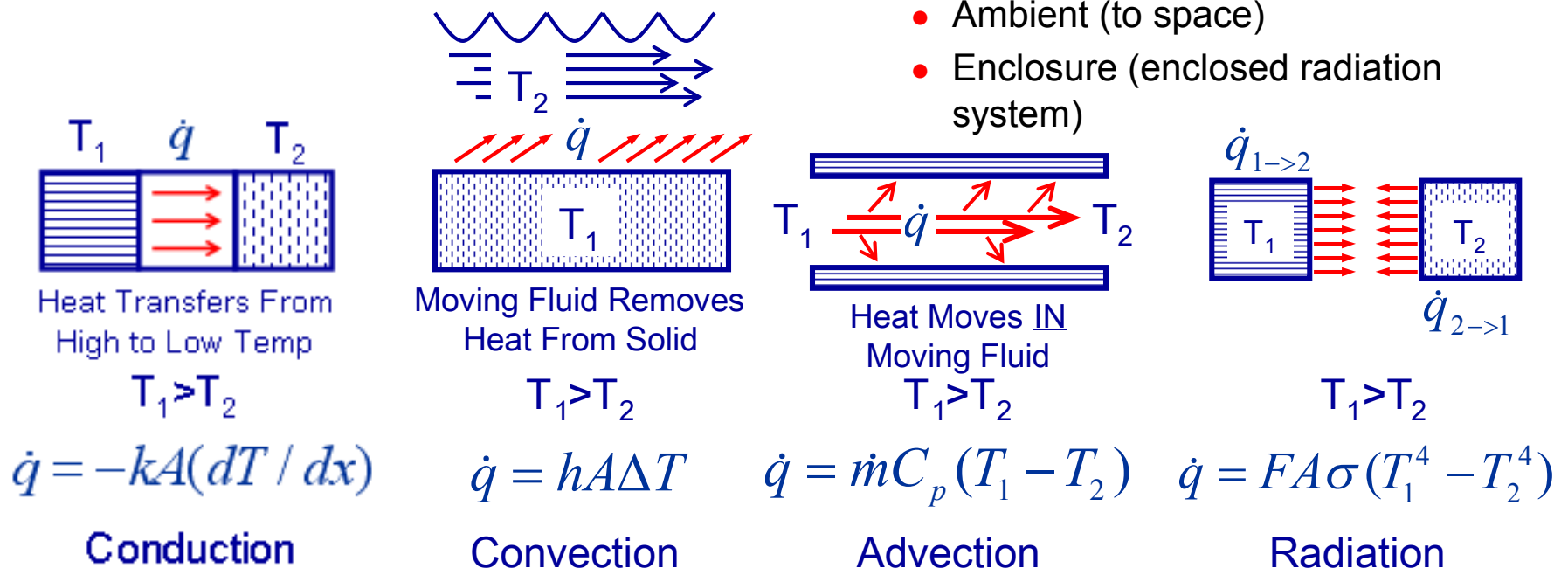
Thermal Field
(Yes)

MODES OF HEAT TRANSFER

- Mechanisms of Heat Transfer

- Conduction (through a material)
- Convection (to a fluid)
 - Natural (free)
 - Forced

- Advection (energy carried in a fluid)
- Radiation (energy transfer through free space (no intervening medium) from one surface to another)
 - Ambient (to space)
 - Enclosure (enclosed radiation system)



MD NASTRAN THERMAL PREFERENCE TABLE (MD PATRAN)

- The following table outlines the options when the MD Patran Preference Analysis Type is set to Thermal:

Object	Type	Option	
Radiation	Nodal Element Uniform	Ambient Space Ambient Nodes Enclosures (View factor)	
Convention	Nodal Element Uniform Element Variable	To ambient Flow Tube to Ambient Coupled Coupled Flow Tube Coupled Advection Duct Flow	Time Function Spatial field
Applied Heat	Nodal Element Uniform Element Variable	Normal Fluxes Directional Fluxes Nodal Source Volumetric Generation Total Heat	Time function Spatial field
Temperature	Nodal	Fixed	Time function Spatial field
Initial Temp	Nodal	Initial	Spatial field

NAS
Copy

MD NASTRAN ELEMENTS

Type	Elements
1D (conduction)	BAR,BEAM,BEND,ROD, TUBE
2D (conduction)	TRIA3,TRIA6,QUAD4, QUAD8
3D (conduction)	HEXA,PENTA,TETRA
Axi-symmetric (conduct)	CTRIAX6
Scalar	CELASi (resistive), CDAMPi (thermal capacitance)
General	DMI (direct matrix input), DMIG (dmi at points), TF (dyn. transfer function)

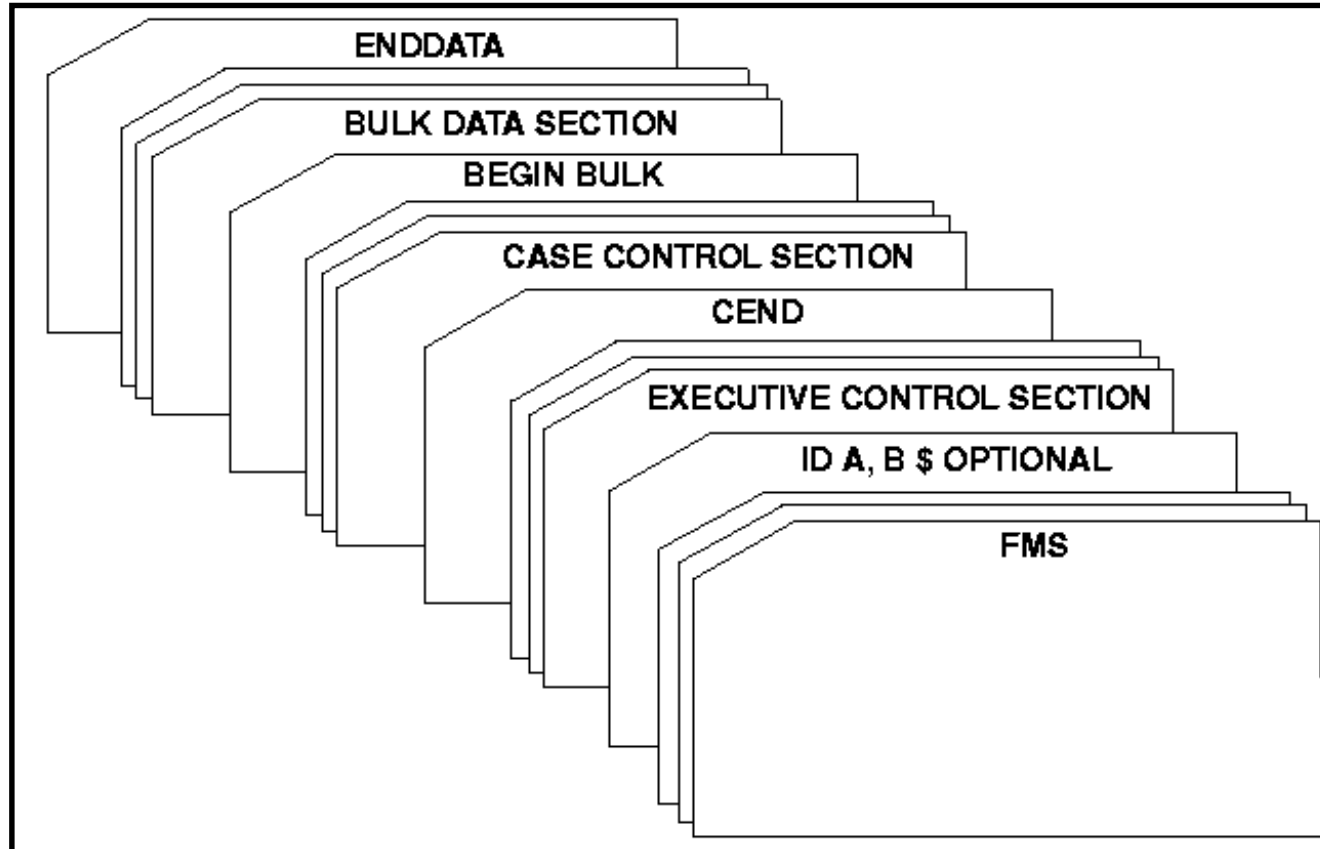


MD NASTRAN INPUT FILE ENTRIES





MD NASTRAN INPUT FILE



MD NASTRAN INPUT FILE (Cont.)

- File Management Section (FMS)
 - Includes the "NASTRAN" statement (optional - determines overall program control for the current run).
 - Allocates files, controls restarts and database operations.
 - The goal of the FMS is to make the operating system invisible to the user.
- Executive Control Section
 - Solution type, time allowed, program modifications and system diagnostics
- Case Control Section
 - Outputs requests and selects certain Bulk Data items such as loadings and constraints to be used.
- Bulk Data Section
 - Structural model definition and solution conditions



MD NASTRAN INPUT FILE (Cont.)

- The Bulk Data section is where you provide the description of the model, and the loading conditions and constraints.
- As mentioned in the previous section, the Executive Control section provides overall control of the solution, and the Case Control section provides control of the individual loadings to be applied and the output requests.



INTRODUCTION TO BULK DATA SECTION

- The Bulk Data Section contains all data necessary for describing a structural model.
- Bulk Data definitions include
 - Geometry
 - User-definable coordinate systems
 - Geometric locations of grid (node) points
 - Constraints
 - Material Properties
 - Element Properties
 - Loads
- The Bulk Data is not required to be input in any set order. It is automatically sorted (alphabetically) at the beginning of the analysis.



FORMAT OF BULK DATA

- The format of the Bulk Data section is based on each line:
 - 80 possible columns
 - Divided into 10 fields
- Every object described in the Bulk Data section is called an “Entry”.
- An Entry may span multiple lines.
- The format of each Entry is pre-defined; they are described in the MD Nastran Quick Reference Guide (QRG), Section 5.
- Only a few basic entries will be described in these notes.
- Not all options will be discussed for each entry. For a full description, check the QRG.



SAMPLE BULK DATA ENTRY

CROD

Rod Element Connection

Defines a tension-compression-torsion element.

Format:

1	2	3	4	5	6	7	8	9	10
CROD	EID	PID	G1	G2					

Example:

CROD	12	13	21	23					

Field	Contents
EID	Element identification number. (Integer > 0)
PID	Property identification number of a PROD entry. (Integer > 0; Default = EID)
G1, G2	Grid point identification numbers of connection points. (Integer > 0; G1 ≠ G2)

Remarks:

1. Element identification numbers should be unique with respect to all other element identification numbers.
2. See [CONROD, 1276](#) for alternative method of rod definition.
3. Only one element may be defined on a single entry.

FORMAT OF BULK DATA ENTRIES

- Data in each field may be:
 - Integer 5
 - Real 1.0E+7
 - BCD (Character String)

} Used for majority of data entries
- Each field of the input has a pre-determined data type.
- You must enter you data correctly.
- An integer number has no decimal point.
- Examples:
 - 1
 - 134
 - 267

FORMAT OF BULK DATA ENTRIES (Cont.)

- Real data has a decimal point and may have an exponent.
- There are several ways to represent real numbers.
- For example, the real value 123.45 might be represented using any of the following:

123.45

1.2345+2

12.345E+01

.12345E3

- All the above values represent the same number.

FORMAT OF BULK DATA ENTRIES (Cont.)

- BCD (or character) data is represented as a text string.
- It must start with a letter (A-Z).
- It may contain numbers in the text (0-9).
- It must be 8 or fewer characters long.
- No special characters or embedded blanks may be used.
- Examples:
 - TEST123
 - X32
 - DUM1



FORMAT OF BULK DATA ENTRIES (Cont.)

- The first field of an entry is its name.
- All subsequent fields provide data as defined for that entry in the QRG.
- If an entry spans more than one line, then a “continuation” is needed.
- Each line of input will use one of three formats:
 - Small Field
 - Free Field
 - Large Field



SMALL FIELD FORMAT

- When using Small Field, each line is divided into 10 fields.
- Each field is 8 columns long.
- This is the format used by most pre-processing programs when writing an input file for MD Nastran.
- Example Small Field Entry

1	2	3	4	5	6	7	8	9	10
← 8 →	← 8 →	← 8 →	← 8 →	← 8 →	← 8 →	← 8 →	← 8 →	← 8 →	← 8 →
GRID	10		7.5	8.6	9.		456		

- When using small field, you must be sure to count the number of columns for each field.
- There is no requirement for numbers to be right- or left-justified (the program handles this for you).

FREE FIELD FORMAT

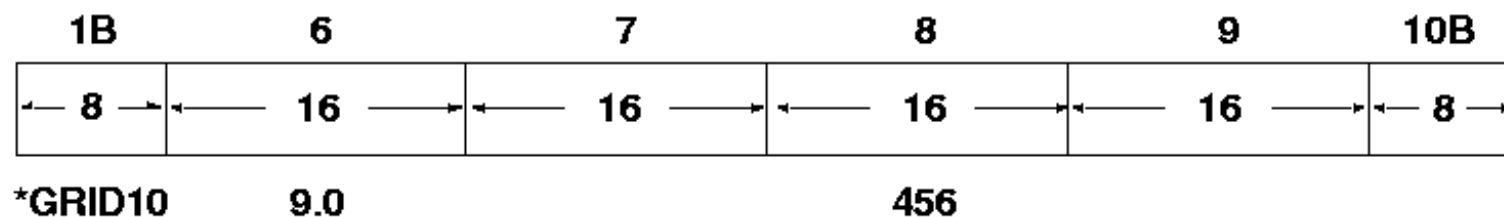
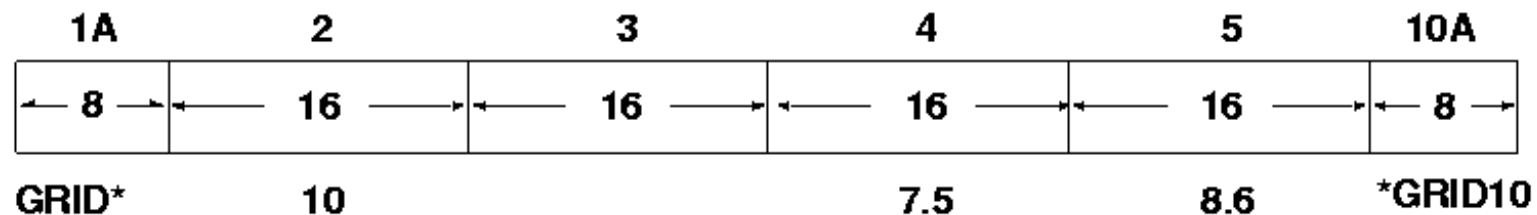
- This is similar to Small Field, but fields are separated by either a comma or a space (using commas is recommended).
- Rules:
 - To skip a field, use two commas in succession.
 - Integers or BCD fields with more than 8 characters will result in a FATAL error.
 - Real numbers with more than 8 characters will be rounded off to 8 characters (therefore, some precision will be lost).
- Example: (Same entry as on the previous slide)
GRID,10,,7.5,8.6,9.,,456

LARGE FIELD FORMAT

- Expands a line in the Bulk Data to 2 lines.
- When used:
 - The first and last fields of each line are 8 columns.
 - The intermediate fields are 16 columns (there are only 4 intermediate fields per line - resulting in each line being 80 columns long).
- Large Field is denoted by adding an asterisk (*) after the name in the first field of an entry and by an asterisk (*) in the first column of the second line of the entry.
- See next slide for the format.

LARGE FIELD FORMAT (Cont.)

- Sample Large Field Format Entry:



GENERAL INPUT FORMAT RULES

- Errors result if data extends beyond its field into another field.
- Fields 1 and 10 must be left justified. Fields 2 through 9 do not have this requirement.
- Input items should not have embedded blanks.

Example: (free field)

G	R	I	D	,	1	0	,		7	.	5	,	8		.	6	,	9	.	0

Since a blank is a valid separator,
MSC/NASTRAN will interpret this as an
integer 8 rather than the intended value,
8.6. Since real numbers are required for
coordinate data, a fatal error will result.

GENERAL INPUT FORMAT RULES (Cont.)

- All Real numbers (including zero) must have a decimal point (this is a common error).
- Many fields have a default value; if these fields are left blank, the default value will be used (see the QRG).



CONTINUATION ENTRIES

- Many input entries require more than one line of input.
- If this is the case, then “continuation” entries must be used.
- Continuation entries may be generated automatically when the entries are in sorted order. The parent entry must be blank in columns 74-80 (field 10), and the continuation entry must be blank in columns 2-8 (field 1). For small field entries, the first column of the continuation entry may be blank or contain a “+” symbol. For large field entries, the first column of the continuation entry must contain a “*” symbol.



CONTINUATION ENTRIES (Cont.)

- Input rules
 - Unless you use automatic generation, a (+) or (*) is required in column 1, field 1 of a continuation entry. The remaining contents in field 1 of a continuation entry must be identical to the entry in field 10 (columns 73 through 80) of the parent entry (or the preceding continuation entry).
 - Any entry in the first column of field 10 on the parent entry is ignored by the continuation entry.
 - Small field and Large field continuation entries may be used together in defining a single data item entry
- An example of this is shown on the next slide.

CONTINUATION ENTRIES (Cont.)

- Parent entry: PBAR,10,20,1.25,,,,,,+PB10
The plus sign is optional. The first column of field 10 on the parent entry is ignored by MSC/NASTRAN.
- Continuation entry: +PB10,2.0,5.0,-2.0,-5.0
The plus sign is required in column 1, field 1 of a continuation entry.

INPUT GENERATION – REPLICATION

- To avoid the time-consuming input of each Bulk Data entry individually, repetitive fields can be generated from a single entry definition. Rules governing this capability are:
 - Duplication of a field from the preceding entry is accomplished by coding the symbol = in the associated field.
 - Duplication of all remaining fields from the preceding entry is accomplished by coding the symbol == in the first of the fields to be repeated.
 - Generation of an incremented value from the previous entry is defined by coding *X or *(X) where X is the real or integer value of the increment (Note: parentheses are optional).
 - Repeated replication is indicated by coding =n or the optional =(n) in field 1, where n is the number of entry images to be generated using the values of the increments on the preceding generation entry.



INPUT GENERATION – REPLICATION (Cont.)

- Generation/replication rules apply to all Bulk Data entries unless denoted otherwise on specific entry definition pages in the QRG.
- Preprocessing programs generally generate a separate entry for each item and do not use replication.
- In this seminar and others, replication is often used as a method to shorten the file so that we can show the complete input, rather than just showing parts of it.



GENERATION OF CONTINUATION ENTRIES

- Continuation fields (fields 1 and 10) may be replicated using the following conventions
 - Only letters of the alphabet and integers may be used. They are coded into a base 36 number. That is, the sequence of numbers is 0, 1, 2,...8, 9, A, B,...Z.
 - The first character in the field 1 or 10 is not incremented.
 - MD Nastran increments continuation fields by +1.
 - The number of characters in an incremented field is not increased. For example, if the field in the first entry is “0”, the field in the 37th entry is also “0” resulting in an illegal duplicate entry. A method to solve this problem is to start a first entry with “00”. This will provide 36 squared unique fields.
 - See Section 3.5.1 of the *MD NASTRAN Handbook for Linear Analysis* for examples of continuation entries in Small field and Large field formats.



EXAMPLE OF REPLICATION

I N P U T B U L K D A T A D E C K E C H O

```

.      1      ..      2      ..      3      ..      4      ..      5      ..      6      ..      7      ..      8      ..      9      ..     10
.
GRID,1,,1.,0.,0.
=,* (1),,* (1.),== $GRID GENERATION CARD
=(3)                $GRID REPLICATION CARD
CBEAM,100,1,1,2,0.,1.,0.
=,*10,=,*1,*1,== $CBEAM GENERATION CARD
=2                  $CBEAM REPLICATION CARD
PBEAM,1,1,24.,72.,32.
,3.,-2.,3.,2.,-3.,2.,-3.,-2. $PBEAM CONTINUATION CARD
,YESA,0.2 $PBEAM CONTINUATION CARD
,=,*0.2 $GENERATION OF PREVIOUS CARD
=3        $REPLICATION OF PREVIOUS CARD
EIGR,10,MGIV,,30.
,MASS      $EIGR CONTINUATION CARD
$
ENDDATA
INPUT BULK DATA CARD COUNT =      15

```

BULK DATA GENERATED BY REPLICATION EXAMPLE

S O R T E D B U L K D A T A E C H O																				
CARD																				
COUNT	1	2	3	4	5	6	7	8	9	10										
1-	CBEAM	100	1	1	2	0.	1.	0.												
2-	CBEAM	110	1	2	3	0.	1.	0.												
3-	CBEAM	120	1	3	4	0.	1.	0.												
4-	CBEAM	130	1	4	5	0.	1.	0.												
5-	EIGR	10	MGIV	30.																+000008
6-	++000008MASS																		+000009	
7-	GRID	1	1.		0.	0.														
8-	GRID	2	2.		0.	0.														
9-	GRID	3	3.		0.	0.														
10-	GRID	4	4.		0.	0.														
11-	GRID	5	5.		0.	0.														
12-	PBEAM	1	1	24.	72.	32.														+000001
13-	++0000013.		-2.	3.	2.	-3.	2.	-3.	-2.											+000002
14-	++000002YESA		0.2																	+000003
15-	++000003YESA		.4																	+000004
16-	++000004YESA		.6																	+000005
17-	++000005YESA		.8																	+000006
18-	++000006YESA		1.0																	+000007
ENDDATA																				
TOTAL COUNT=		19																		

NAS104

Copyright © 2008 MSC Software Corporation

S2-34



MD NASTRAN REPLICATION COMMAND

```
chbdyp,10,10,line,,,1,2,1
```

```
=,*1,=,=,=,=,*1,*1,==
```

```
=7
```

CHBDYP	10	10	LINE	1	2	1
CHBDYP	11	10	LINE	2	3	1
CHBDYP	12	10	LINE	3	4	1
CHBDYP	13	10	LINE	4	5	1
CHBDYP	14	10	LINE	5	6	1
CHBDYP	15	10	LINE	6	7	1
CHBDYP	16	10	LINE	7	8	1
CHBDYP	17	10	LINE	8	9	1
CHBDYP	18	10	LINE	9	10	1

REPLICATE THE CONTINUITY CARDS

```
CHBDYG,EID,,TYPE,IVIEWF,IVIEWB,RADMIDF,RADMIDB
,+
+,G1,G2,G3,G4,G5,G6,G7,G8
CHBDYG,21,,REV,,,,,,+C1
+C1,201,202
CHBDYG,22,,REV,,,,,,+C2
+C2,202,203

CHBDYG,21,,REV,,,,,,+C1
=,*1,=,=,=,=,=,=,*1
=7
+C1,201,202
*1,*1,*1
=7
```

COMMON ERRORS IN ENTRY FORMAT

- The following are recommendations on how to avoid some commonly made errors in the input
 - Failing to leave the proper number of fields blank when defining data values causes either a fatal error or wrong answers. Be sure to leave the proper number of blanks or include the correct number of commas to delimit data fields.
 - Be sure to use the correct format for integer numbers and real numbers. See individual entry format in the MD Nastran Quick Reference Guide for these specifications.
 - Be sure to define all the required fields for the Bulk Data entries.



FORMAT OF THE DAT FILE

NASTRAN statement

(optional)

File Management

(Use for restart purpose)

Executive Control

(required - sets solution type)

CEND

(keyword delimiter)

Case Control

(declares loads/output requests)

BEGIN BULK

(keyword delimiter)

Bulk Data

(specifies model details)

ENDDATA

(keyword delimiter)

FILE MANAGEMENT

- Principal uses
 - Initialization of the MD Nastran database
 - Restarts
 - Controlling scratch space; requires manual input
 - Using FMS commands, scratch can span multiple hard drives for large jobs
 - Assigning file units (e.g. to produce output for Patran); requires manual input



EXECUTIVE CONTROL

- Principal uses
 - Specify MD Nastran solution sequence
 - Specify maximum allowable CPU time
- Example:
 - **SOL 153** (Nonlinear steady-state thermal solution sequence)
 - **CEND** (designates the end of the executive control section, Control END)

CASE CONTROL

- Principal uses
 - Select desired loading and boundary conditions
 - Specify convergence criteria
 - Request output sets (sent F06, XDB and/or 'punch' file)

CASE CONTROL - STEADY STATE EXAMPLE

ANALYSIS = HEAT

(analysis type)

ECHO = NONE

(BULK DATA echo

control)

THERMAL = ALL

(nodal temperatures)

SPCFORCE = ALL

(nodal constraint heats)

FLUX = ALL

(element fluxes/gradients)

SPC = 1

(select constraint set)

LOAD = 1

(select static load)

NASTRAN INPUT TEST DECK

- BEGIN BULK
- \$ Direct Text Input for Bulk Data
- \$ Elements and Element Properties for region : rod

GRID 1 0. 0. 0.

GRID 2 .1 0. 0.

CROD 1 1 1 2

PROD 1 1 .007854

- \$ Description of Material : Date: 03-Jun-07 Time: 15:43:43

MAT4 1 204.

MD NASTRAN FILES

- DAT Input file describing FE model and run parameters
- F06 ASCII results file
- XDB Binary results file
- F04 Run history(disk space used, files assigned, etc.)
- LOG Summary of execution links
- DBALL Main database - contains assembled matrices and solutions
(*required for restart*)
- MASTER Serves as a directory listing of files used during a run
(*required for restart*)



SECTION 3

CONDUCTION





CONDUCTION

- Theory and capabilities
- Supported topologies
- Isotropic materials (MAT4)
 - Constant properties
 - Temperature-varying properties
- Anisotropic materials (MAT5)
- Heat generation

CONDUCTION, THEORY

- Second Law of Thermodynamics
 - Heat flows *from* regions of **high temperature** to regions of **low temperature**.
- Fourier's Law (empirical)
 - $q_k = (-1.0) * k * (\partial T / \partial x)$
- Conservation of Energy

$$\nabla^2 T + \frac{\dot{q}}{k} = \frac{1}{\alpha} \frac{\partial T}{\partial t}$$

SUPPORTED ELEMENT TOPOLOGIES, FINITE ELEMENTS

- These MD Nastran finite elements are isoparametric
- 1D
 - CBAR, CBEAM, CBEND, CROD, CTUBE
 - These elements cannot represent a thermal gradient, through their material, normal to their centerline
- 2D
 - CQUAD4, CQUAD8, CTRIA3, CTRIA6
 - These finite elements cannot represent a thermal gradient, through their material, normal to their mid-plane
- 3D
 - CHEXA, CPENTA, CTETRA
- AXISYMMETRIC
 - CTRIA6

SUPPORTED ELEMENT TOPOLOGIES, SCALAR ELEMENTS

- CELASi (i=1,2,3,4)
- Used to model a one-dimensional conduction element with a specific thermal resistance.
- $R = L/KA$, where CELAS2 used inverse of R
 - $G = 1/R = KA/L$ (Watt/C)
- CDAMPi (i=1,2,3,4)
- Used to model lumped heat capacity (capacitance).

SUPPORTED ELEMENT TOPOLOGIES, MD NASTRAN SOL 400

- MD Nastran SOL 400 has many types of thermal elements

Element	# Nodes	Type (Dim)	Marc El # / Prop	Type (Dim)	Marc El # / Prop	Type (Dim)	Marc El # / Prop	Type (Dim)	Marc El # / Prop
CROD	2	Rod (1D)	36 / PRODN1						
CQUAD4	4	Membrn (2D)	198 / PSHLN1	Planar (2D)	39 / PSHLN2	Shell (3D)	85 / PSHLN1	Planar Comp	177 / PLCOMP
CQUAD8	8	Membrn (2D)	199 / PSHLN1	Planar (2D)	41 / PSHLN2	Shell (3D)	86 / PSHLN1	Planar Comp	179 / PLCOMP
CTRIA3	3	Membrn (2D)	196 / PSHLN1	Planar (2D)	37 / PSHLN2	Shell (3D)	50 / PSHLN1		
CTRIA6	6	Membrn (2D)	197 / PSHLN1	Planar (2D)	131 / PSHLN2				

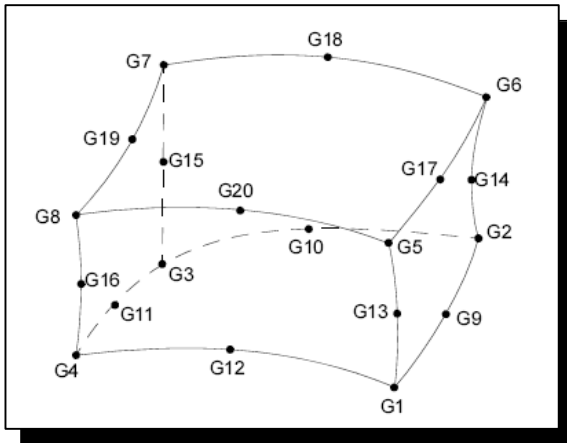
SUPPORTED ELEMENT TOPOLOGIES, MD NASTRAN SOL 400

- MD Nastran SOL 400 has many types of thermal elements

Element	# Nodes	Type	Marc El # / Prop	Type	Marc El # / Prop
CHEXA	8	Solid (3D)	43 / PSLDN1	Solid Comp	175 / PCOMPLS
CHEXA	20	Solid (3D)	44 / PSLDN1	Solid Comp	176 / PCOMPLS
CPENTA	6	Solid (3D)	137 / PSLDN1		
CTETRA	4	Solid (3D)	135 / PSLDN1		
CTETRA	10	Solid (3D)	133 / PSLDN1		

CHEXA CONDUCTION ELEMENT

- MD Nastran CHEXA entry.



CHEXA

Six-Sided Solid Element Connection

Defines the connections of the six-sided solid element with eight to twenty grid points.

Format:

	1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6		
	G7	G8	G9	G10	G11	G12	G13	G14		
	G15	G16	G17	G18	G19	G20				

Example:

CHEXA	71	4	3	4	5	6	7	8		
	9	10	0	0	30	31	53	54		
	55	56	57	58	59	60				

Field	Contents	Type	Default
EID	Element identification number.	(0 < Integer < 100,000,000)	Required
PID	Property identification number of a PSOLID or PLSOLID entry.	Integer > 0	Required
Gi	Grid point identification numbers of connection points.	Integer ≥ 0 or blank	Required

CHEXA CONDUCTION ELEMENT (Cont.)

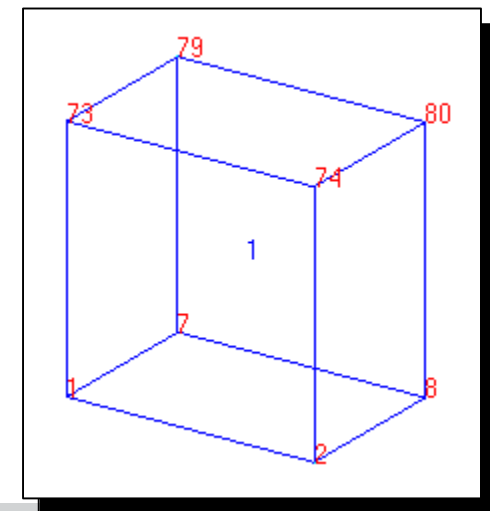
- Element connectivity is defined on the MD Nastran CHEXA entry. For Element 1 with eight grids (nodes):

1	2	3	4	5	6	7	8	9	10
CHEXA	EID	PID	G1	G2	G3	G4	G5	G6	
CHEXA	1	1	1	2	8	7	73	74	
	G7	G8							
	80	79							

```

CHEXA 1 1 1 2 8 7 73 74
      80 79
CHEXA 2 1 2 3 9 8 74 75
      81 80
CHEXA 3 1 3 4 10 9 75 76
      82 81
PSOLID,1,100
MAT4,100,2.3
$ Thermal conductivity is 2.3
    
```

.bdf file extract



ELEMENT PROPERTIES

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		

Example:

PSOLID	2	100	6	TWO	GRID	REDUCED			

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 5., 6., 7., and 9. (Integer, Character, or blank)
STRESS	Location selection for stress output. See Remarks 8. and 9. (Integer, Character, or blank)
ISOP	Integration scheme. See Remarks 5., 6., 7., and 9. (Integer, Character, or blank)
FCTN	Fluid element flag. (Character: "PFLUID" indicates a fluid element, "SMECH" indicates a structural element; Default = "SMECH.")

MATERIAL PROPERTIES

MAT4

Heat Transfer Material Properties, Isotropic

Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.

Format:

1	2	3	4	5	6	7	8	9	10
MAT4	MID	K	CP	ρ	H	μ	HGEN	REFENTH	
	TCH	TDELTA	QLAT						

Example:

MAT4	1	204.	.900	2700.					

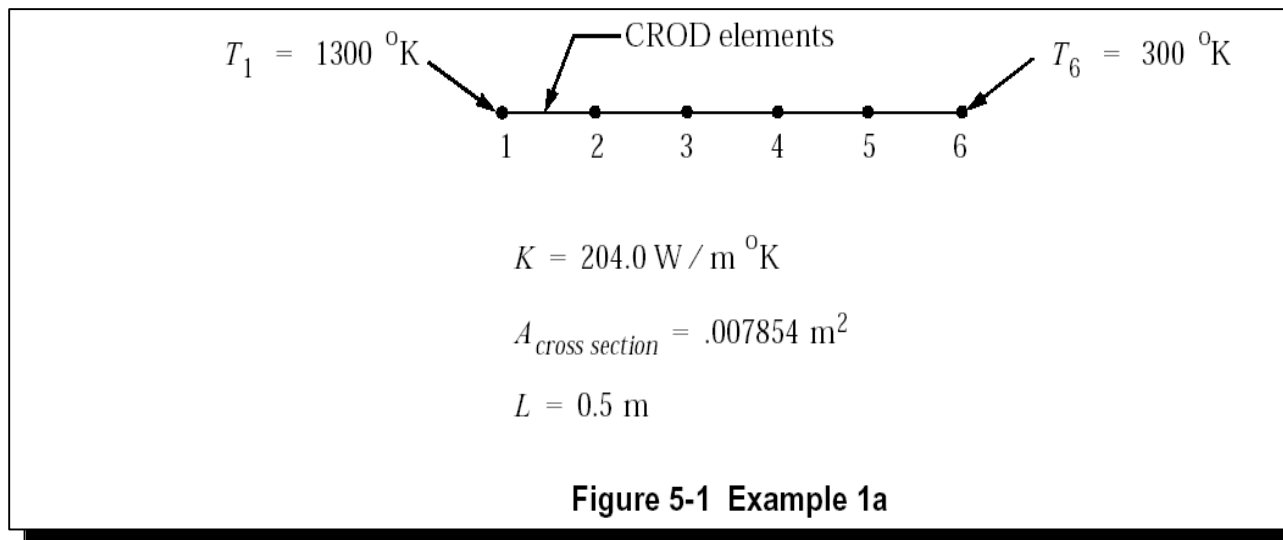
Field	Contents
MID	Material identification number. (Integer > 0)
K	Thermal conductivity. (Blank or Real > 0.0)
CP	Heat capacity per unit mass at constant pressure (specific heat). (Blank or Real $\geq$ 0.0)
ρ	Density. (Real > 0.0; Default = 1.0)
H	Free convection heat transfer coefficient. (Real or blank)
μ	Dynamic viscosity. See Remark 2. (Real > 0.0 or blank)
HGEN	Heat generation capability used with QVOL entries. (Real $\geq$ 0.0; Default = 1.0)
REFENTH	Reference enthalpy. (Real or blank)
TCH	Lower temperature limit at which phase change region is to occur. (Real or blank)
TDELTA	Total temperature change range within which a phase change is to occur. (Real $\geq$ 0.0 or blank)
QLAT	Latent heat of fusion per unit mass associated with the phase change. (Real > 0.0 or blank)

MD NASTRAN EXAMPLE PROBLEMS

- There are 28 example exercises available in the MD Nastran 2002 Thermal Analysis User's Guide. Several of them will be focused on in this course.
1. Example 1a – Linear conduction
 2. Example 1b – Nonlinear free convection
 3. Example 1c – Temperature dependent heat transfer coefficient
 4. Example 1e – Radiation boundary conditions
 5. Example 4b – Directional solar load with radiation boundary condition
 6. Example 6 – Forced convection tube flow
 7. Example 7d – Time varying heat transfer coefficient
 8. Example 5a – Single cavity enclosure radiation with shadowing

EXAMPLE 1A, LINEAR CONDUCTION

- Linear conduction model
 - Five CROD elements, with two nodes each



EXAMPLE 1A, MD NASTRAN INPUT

- SPC = 10 is used to specify temperature boundary conditions.
- TEMPD, 20, 1300.0 is used to specify the initial guess temperature in steady state analysis

```
SOL 153
TIME 10
CEND
TITLE = EXAMPLE 1a
EX1A
ANALYSIS = HEAT
THERMAL = ALL
FLUX = ALL
SPCF = ALL
OLOAD = ALL
SPC = 10
TEMP(INIT) = 20
NLPARM = 100
BEGIN BULK
NLPARM,100
$
GRID,1,,0.0,0.0,0.0
GRID,2,,0.1,0.0,0.0
GRID,3,,0.2,0.0,0.0
GRID,4,,0.3,0.0,0.0
GRID,5,,0.4,0.0,0.0
GRID,6,,0.5,0.0,0.0
$
CROD,1,5,1,2
CROD,2,5,2,3
CROD,3,5,3,4
CROD,4,5,4,5
CROD,5,5,5,6
PROD,5,15,.0078540
MAT4,15,204.0
$
SPC,10,1,,1300.0
SPC,10,6,,300.0
TEMPD,20,1300.0
$
ENDDATA
```

EXAMPLE 1A, MD NASTRAN OUTPUT

LOAD STEP = 1.00000E+00

TEMPERATURE VECTOR

POINT ID.	TYPE	ID	VALUE	ID+1 VALUE	ID+2 VALUE	ID+3 VALUE	ID+4 VALUE	ID+5 VALUE
1	S	1.300000E+03	1.100000E+03	9.000000E+02	7.000000E+02	5.000000E+02	3.000000E+02	

LOAD STEP = 1.00000E+00

FORCES OF SINGLE-POINT CONSTRAINT

POINT ID.	TYPE	ID	VALUE	ID+1 VALUE	ID+2 VALUE	ID+3 VALUE	ID+4 VALUE	ID+5 VALUE
1	S	3.204432E+03	0.0	0.0	0.0	0.0	0.0	-3.204432E+03

LOAD STEP = 1.00000E+00

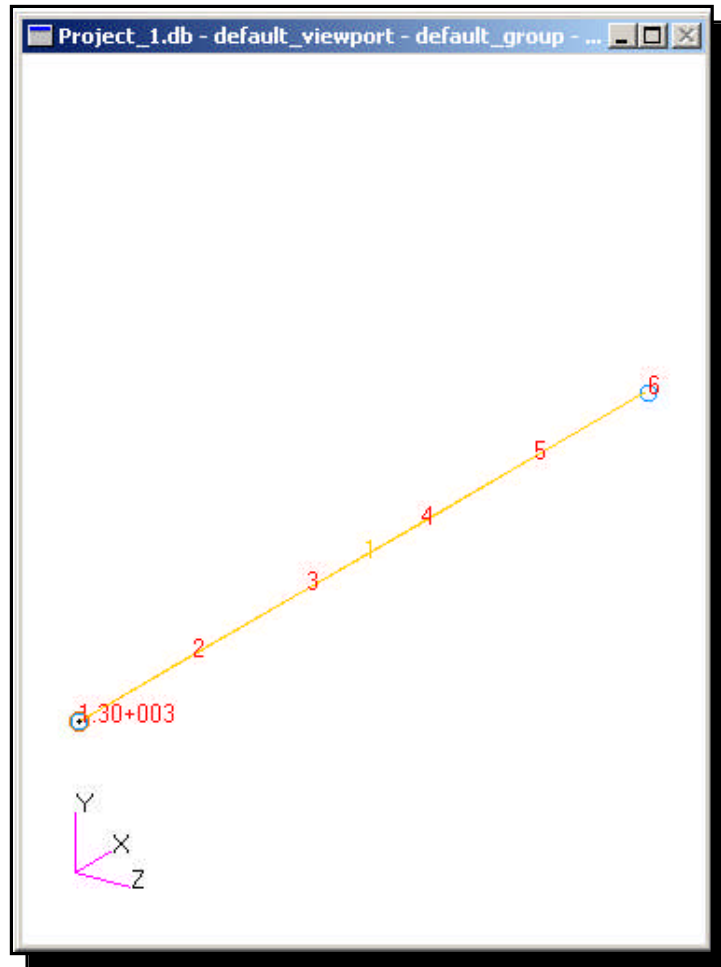
FINITE ELEMENT TEMPERATURE GRADIENTS AND FLUXES

ELEMENT-ID	EL-TYPE	X-GRADIENT	Y-GRADIENT	Z-GRADIENT	X-FLUX	Y-FLUX	Z-FLUX
1	ROD	-2.000000E+03			4.080000E+05		
2	ROD	-2.000000E+03			4.080000E+05		
3	ROD	-2.000000E+03			4.080000E+05		
4	ROD	-2.000000E+03			4.080000E+05		
5	ROD	-2.000000E+03			4.080000E+05		

PROJECT 1, STEADY STATE CONDUCTION

- Part 1, Constant conductivity (this is a repeat of Example 1a)
 - Create and mesh geometry representing an aluminum rod.
 - Create constant conductivity, K , property.
 - Apply fixed temperature BCs.
 - Run Steady State analysis.
 - Post-process temperature results.
- Part 2, Conductivity as a function of temperature
 - Copy the DB file from Part 1 into new directory.
 - Use Fields: Create/Material Property to specify $K = K(T)$.
 - Run Steady State analysis.
 - Post-process temperature results.

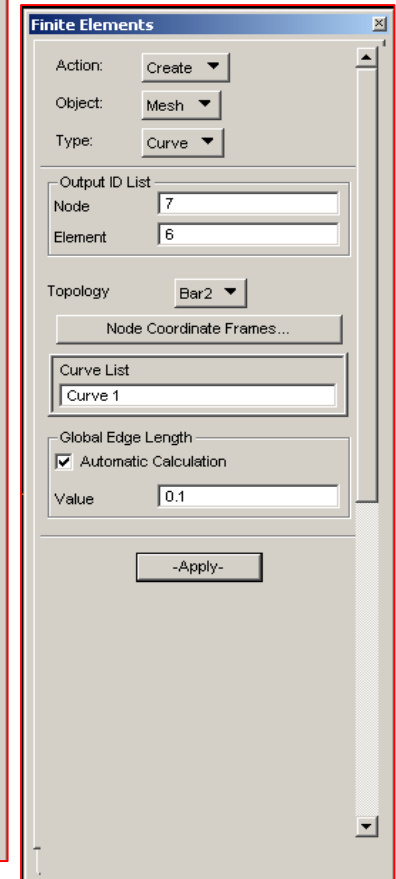
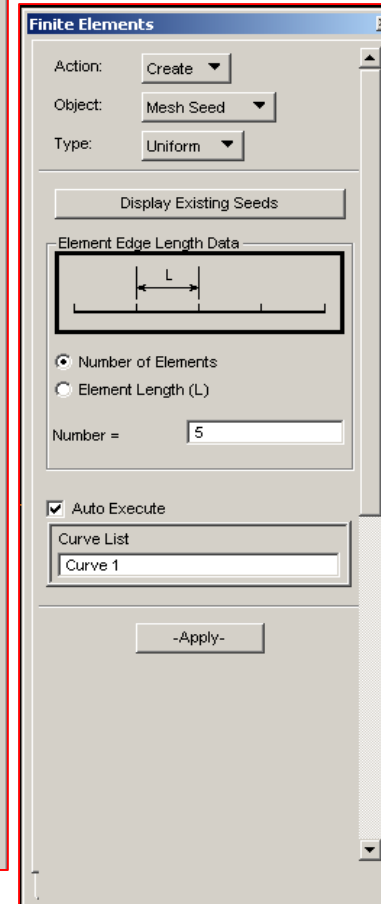
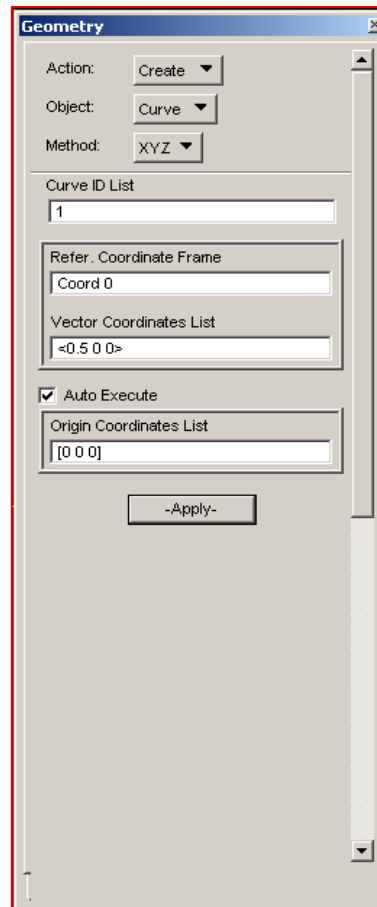
PROJECT 1, PART 1, $K = \text{CONSTANT}$



- This project is aimed at illustrating the steps required to set up a simple steady state conduction problem.
- Throughout, we will take time to discuss the relationship between the Patran Nastran Thermal preference and the MD Nastran files

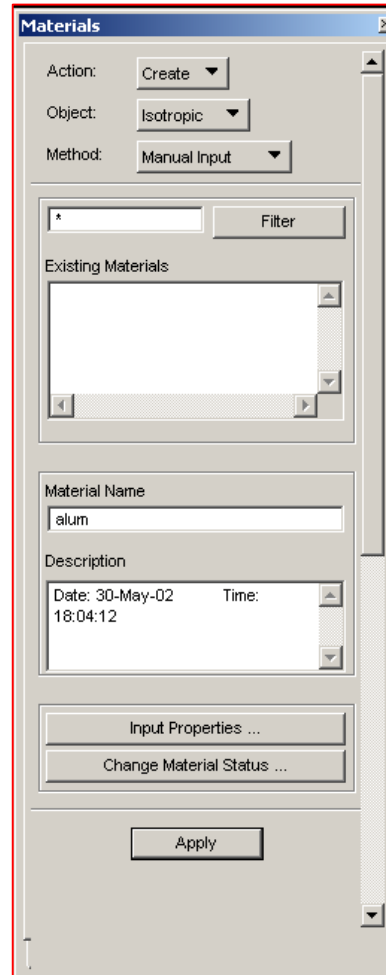
PROJECT 1, PART 1, K = CONSTANT

- Create a geometric curve using Geometry: Create/Curve/XYZ.
- Create mesh seeds (controls element size during creation) on curve.
- Mesh curve, Curve 1
 - Note that because of the existing mesh seeds, the Global Edge Length does not have any effect on the element size.

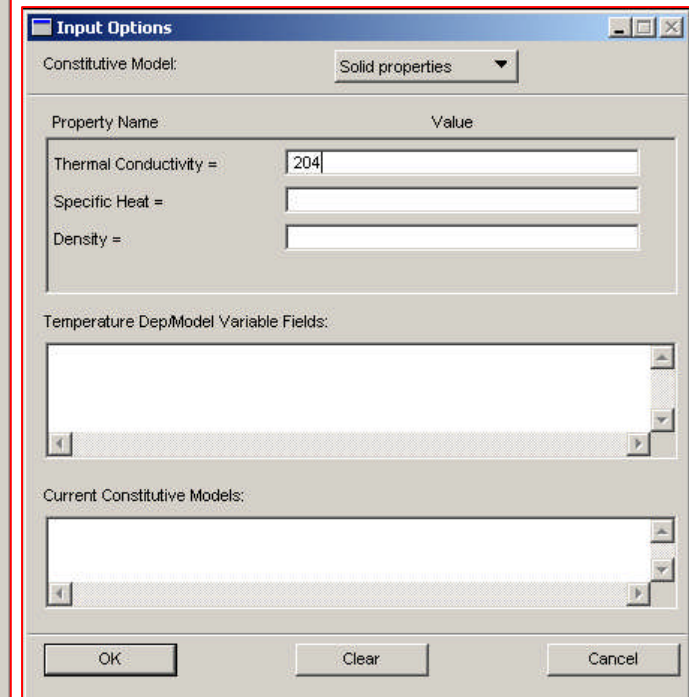


PROJECT 1, PART 1, $K = \text{CONSTANT}$

- Specify constant conduction for material.



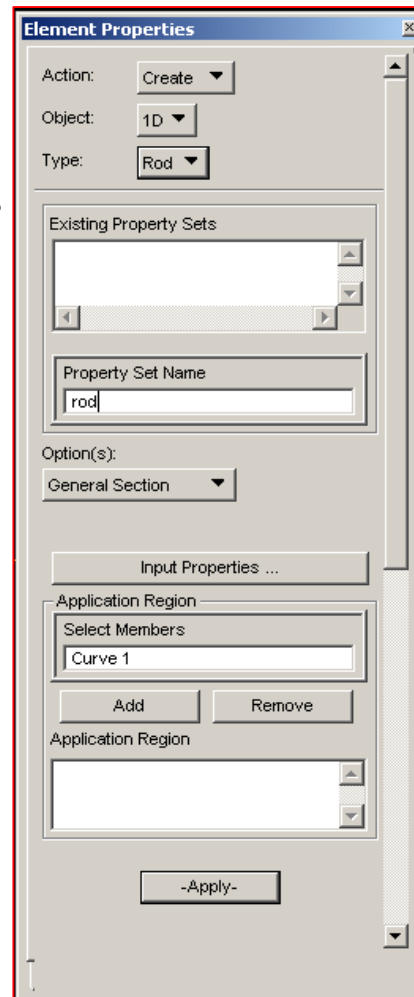
The Materials dialog box is used to create or modify material properties. It features a title bar with a close button. The main area contains several sections: 'Action' with a 'Create' dropdown, 'Object' with an 'Isotropic' dropdown, and 'Method' with a 'Manual Input' dropdown. Below these is a search field with a '*' placeholder and a 'Filter' button. The 'Existing Materials' section contains a list box. The 'Material Name' section has a text field with 'alum' entered. The 'Description' section has a text field with 'Date: 30-May-02 Time: 18:04:12'. At the bottom are buttons for 'Input Properties ...', 'Change Material Status ...', and 'Apply'.



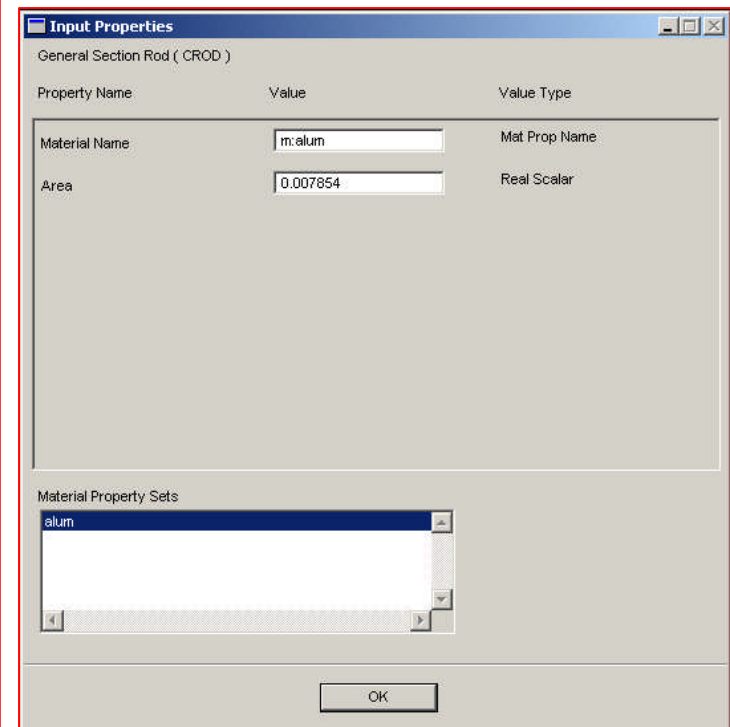
The Input Options dialog box is used to specify the constitutive model and input properties. It has a title bar with standard window controls. The 'Constitutive Model' section has a dropdown menu set to 'Solid properties'. The 'Property Name' and 'Value' section contains three rows: 'Thermal Conductivity =' with a value of '204', 'Specific Heat =' with an empty field, and 'Density =' with an empty field. The 'Temperature Dep./Model Variable Fields' section contains a large empty list box. The 'Current Constitutive Models' section also contains a large empty list box. At the bottom are 'OK', 'Clear', and 'Cancel' buttons.

PROJECT 1, PART 1, K = CONSTANT

- Specify the topologically 1D shapes as 1D rod elements, CROD.
- The cross-sectional area is specified under Input Properties.
- The Application Region is the geometric curve, Curve 1.



The 'Element Properties' dialog box is shown. It has a title bar 'Element Properties' and a close button. The 'Action' dropdown is set to 'Create'. The 'Object' dropdown is set to '1D'. The 'Type' dropdown is set to 'Rod'. Below these is a section for 'Existing Property Sets' with a list box and navigation arrows. A 'Property Set Name' text box contains the text 'rod'. Below that is an 'Option(s):' dropdown set to 'General Section'. There is an 'Input Properties ...' button. The 'Application Region' section has a 'Select Members' text box containing 'Curve 1', and 'Add' and 'Remove' buttons. Below that is an empty 'Application Region' list box. At the bottom is an '-Apply-' button.



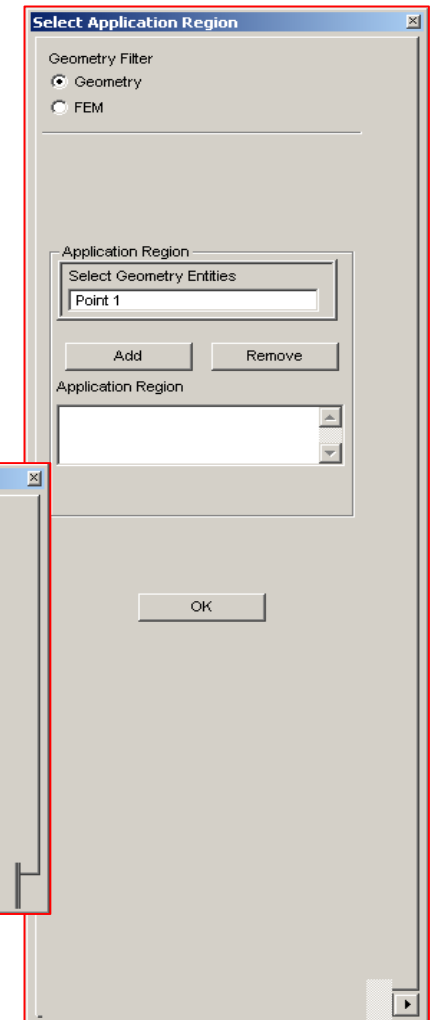
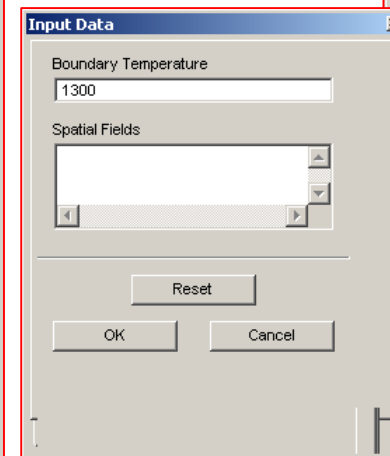
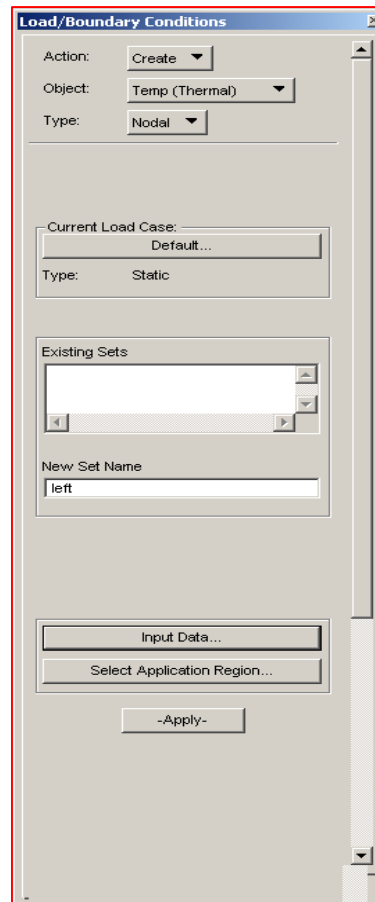
The 'Input Properties' dialog box is shown. It has a title bar 'Input Properties' and standard window controls. The subtitle is 'General Section Rod (CROD)'. It contains a table with the following data:

Property Name	Value	Value Type
Material Name	m:alum	Mat Prop Name
Area	0.007854	Real Scalar

Below the table is a 'Material Property Sets' section with a list box containing 'alum'. At the bottom is an 'OK' button.

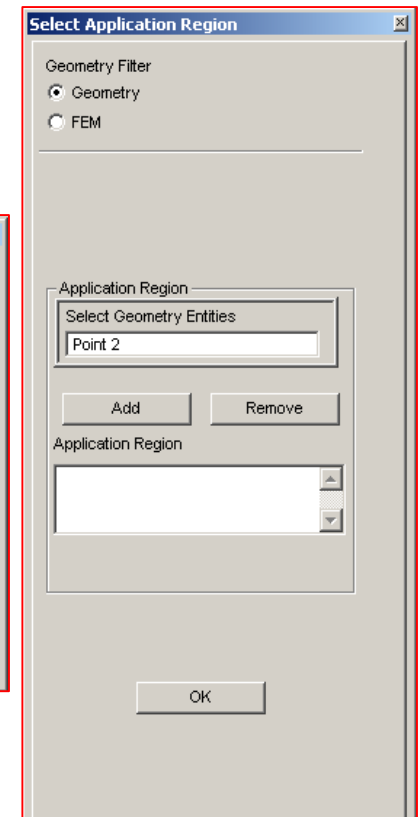
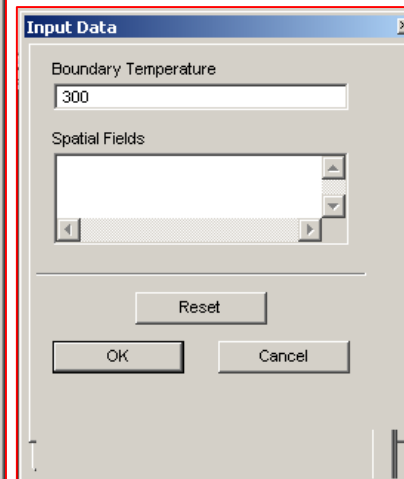
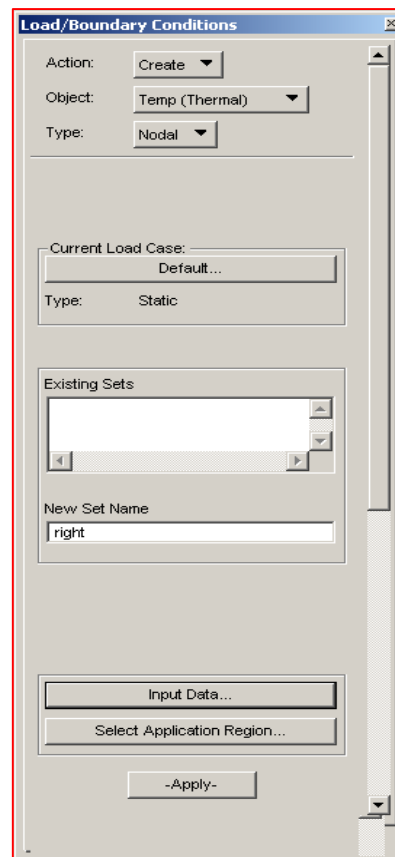
PROJECT 1, PART 1, K = CONSTANT

- Create constant temperature LBC at one end of bar element mesh.
 - Temperature is 1300 degrees
 - Application Region is Point 1



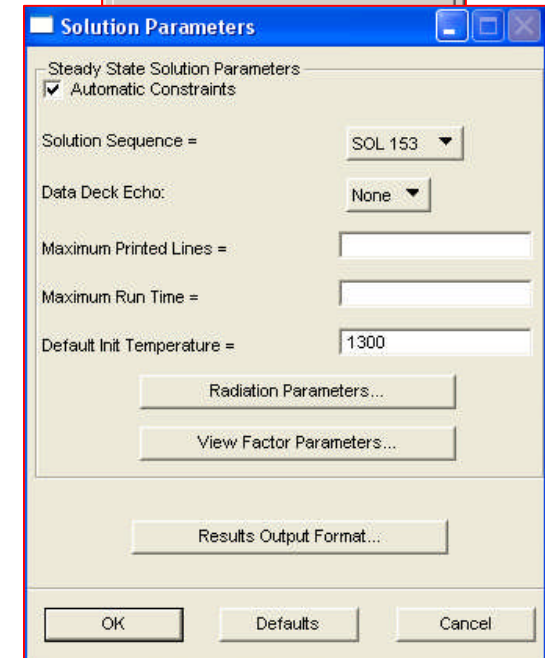
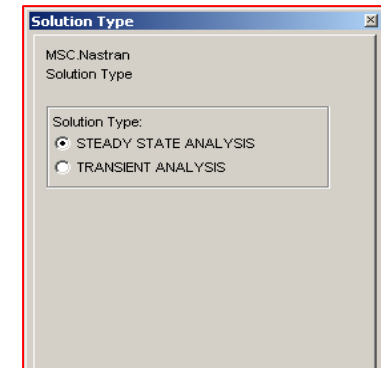
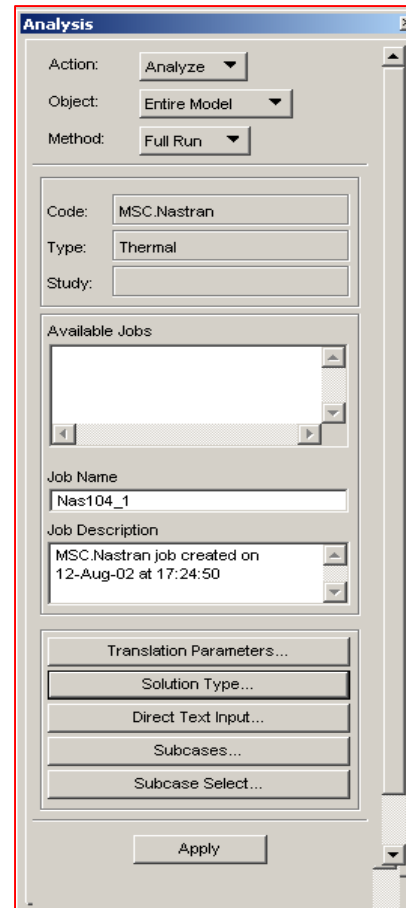
PROJECT 1, PART 1, K = CONSTANT

- Create constant temperature LBC at the other end of bar element mesh.
 - Temperature is 300 degrees
 - Application Region is Point 2



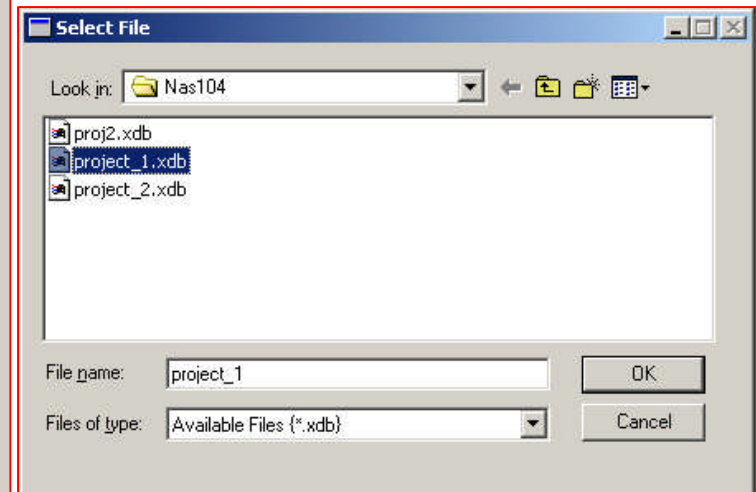
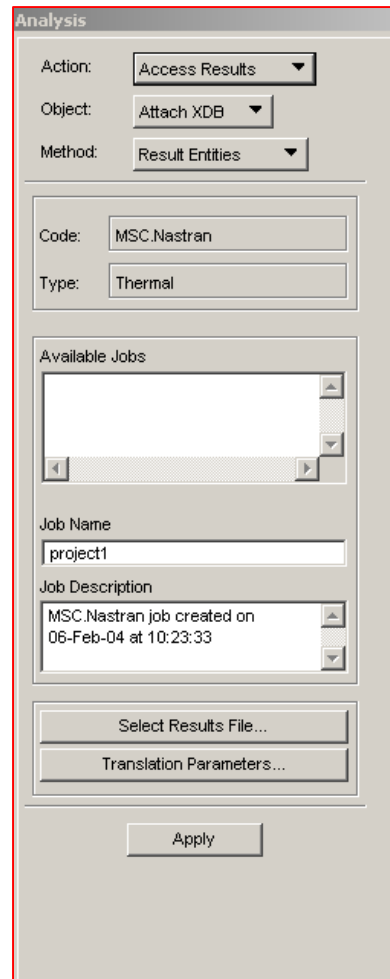
PROJECT 1, PART 1, K = CONSTANT

- Run the steady state thermal analysis using MD Nastran.
 - Under Solution Type, select STEADY STATE ANALYSIS.
 - Under Solution Parameters, make the selections noted.



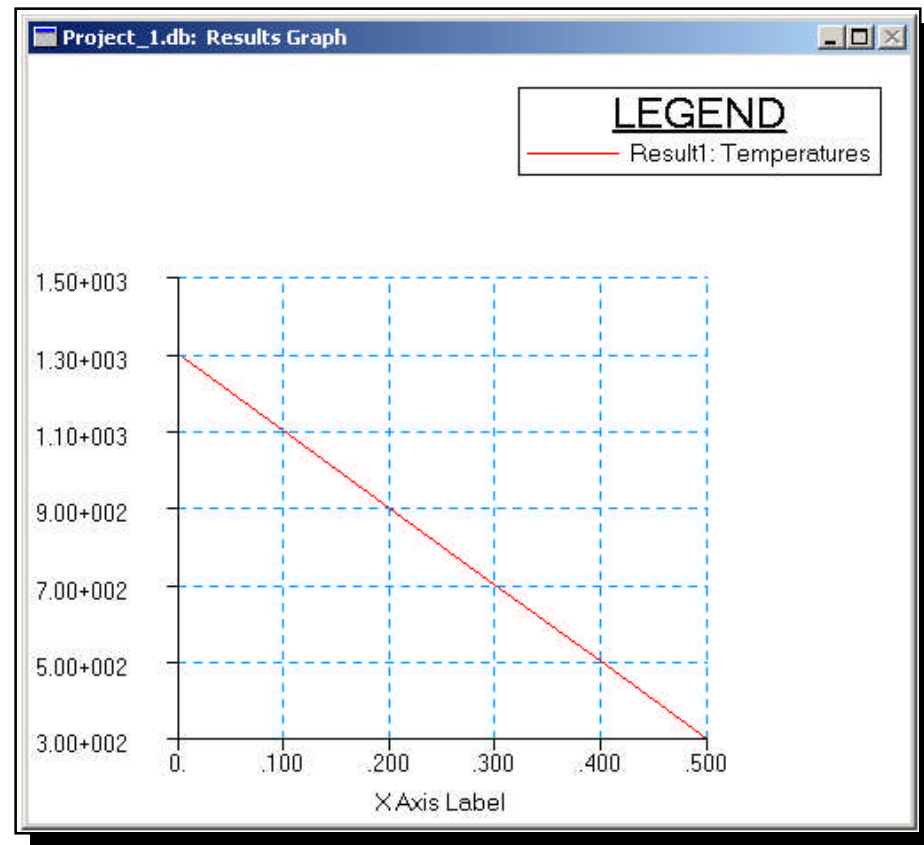
PROJECT 1, PART 1, K = CONSTANT

- Access the MD Nastran results file, project1.xdb.
 - Instead of reading a results file project1.op2, attach the given results file.
 - Keeps the Patran database size smaller.



PROJECT 1, PART 1, $K = \text{CONSTANT}$

- Temperature versus X-direction path results



PROJECT 1, PART 1, K = CONSTANT

- The MD Nastran .DAT input file.
- Take a look at the form and content of the CROD, PROD and MAT4 records.
- Notice how the temperature BC's and initial conditions are specified and called out by Case Control.
- Convergence criteria are set by the NLPARM card.
- Note helpful comments (e.g. the material name).
- Note the presence of 'additional' records (e.g. PARAMs: SIGMA, TABS, AUTOSPC, POST, PATVER).



PROJECT 1, PART 1, K = CONSTANT

Extracted from MD Nastran model .DAT file

```
SOL 153
TIME 600
$ Direct Text Input for Executive Control
CEND
ANALYSIS = HEAT
TITLE = MSC/NASTRAN job created on 01-Jul-98 at 16:18:30
ECHO = SORT
MAXLINES = 999999999
TEMPERATURE(INITIAL) = 1
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 1
  THERMAL(SORT1,PRINT)=ALL
  FLUX(SORT1,PRINT)=ALL
BEGIN BULK
```

PROJECT1.DAT

PROJECT 1, PART 1, K = CONSTANT

Extracted from MD Nastran model .DAT file

```
PARAM      POST      0
PARAM      AUTOSPC YES
PARAM      TABS      0.
PARAM*     SIGMA      1.714-9
NLPARM     1          0          AUTO      5          25          PW          NO          +          A
+          A          .001      1.-7
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : rod
PROD       1          1          .007854
CROD       1          1          1          2
CROD       2          1          2          3
CROD       3          1          3          4
CROD       4          1          4          5
CROD       5          1          5          6
$ Referenced Material Records
$ Material Record : alum
$ Description of Material : Date: 22-May-98
MAT4       1          204.
$ Nodes of the Entire Model
GRID       1          0.          0.          0.
GRID       2          .1          0.          0.
```

PROJECT1.DAT

PROJECT 1, PART 1, K = CONSTANT

Extracted from MD Nastran model .DAT file

```
GRID      3      .2      0.      0.
GRID      4      .3      0.      0.
GRID      5      .4      0.      0.
GRID      6      .5      0.      0.
$ Loads for Load Case : Default
$ Fixed Temperatures of Load Set : left
SPC        1        1        1      1300.
$ Fixed Temperatures of Load Set : right
SPC        1        6        1      300.
$ Initial Temperatures from Temperature Load Sets
TEMP       1        1      1300.    6      300.
$ Default Initial Temperature
TEMPD      1        0.
$ Referenced Coordinate Frames
ENDDATA
```

PROJECT1.DAT

PROJECT 1, PART 1, K = CONSTANT

- The MD Nastran .f06 output file.
- The Executive Control and Case Control are echoed; Bulk Data is suppressed by default.
- NON-LINEAR ITERATION MODULE OUTPUT - shows convergence progress
- TEMPERATURE VECTOR - nodal temperatures
- FORCES OF SINGLE-POINT CONSTRAINT - heats of constraint
- FINITE ELEMENT TEMPERATURE GRADIENTS AND FLUXES – X,Y,Z results in element frame

PROJECT 1, PART 1, K = CONSTANT

Extract from MD Nastran ASCII Results File

```
0
SUBCASE 1
  LOAD STEP = 1.000000E+00

                                T E M P E R A T U R E   V E C T O R

      POINT ID.   TYPE    ID   VALUE      ID+1 VALUE      ID+2 VALUE      ID+3 VALUE      ID+4 VALUE      ID+5 VALUE
        1         S    1.300000E+03  1.100000E+03  9.000000E+02  7.000000E+02  5.000000E+02  3.000000E+02
1  MSC/NASTRAN JOB CREATED ON 01-JUN-07 AT 16:18:30                                JUNE 1, 2007 MSC/NASTRAN
DEFAULT
0
SUBCASE 1
  LOAD STEP = 1.000000E+00

                                F I N I T E   E L E M E N T   T E M P E R A T U R E   G R A D I E N T S   A N D   F L U X E S

      ELEMENT-ID  EL-TYPE      X-GRADIENT      Y-GRADIENT      Z-GRADIENT      X-FLUX      Y-FLUX
Z-FLUX
        1        ROD          -2.000000E+03
        2        ROD          -2.000000E+03
        3        ROD          -2.000000E+03
        4        ROD          -2.000000E+03
        5        ROD          -2.000000E+03
                                4.080000E+05
                                4.080000E+05
                                4.080000E+05
                                4.080000E+05
                                4.080000E+05
                                4.080000E+05
1  MSC/NASTRAN JOB CREATED ON 01-JUN-07 AT 16:18:30                                JUNE 1, 2007 MSC/NASTRAN
```

PROJECT1.f06

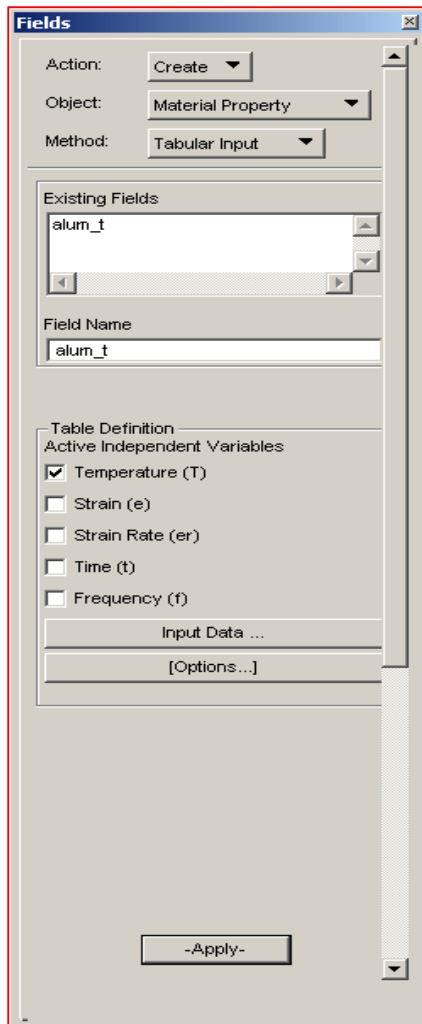


PROJECT 1, PART 2, $K = K(\text{TEMP})$

- Create a new directory named **Project1_Part2**.
- Copy **Project1_Part1.db**, from Part 1, into the new directory, and rename the file as **Project1_Part2.db**.
- Open **Project1_Part2.db** in Patran.
- Create a tabular temperature field for conductivity which peaks at 650 degrees Celsius.
- Modify the material to reference the newly created temperature Field (function).
- Post process to obtain the temperature plot, and examine the new MD Nastran files.

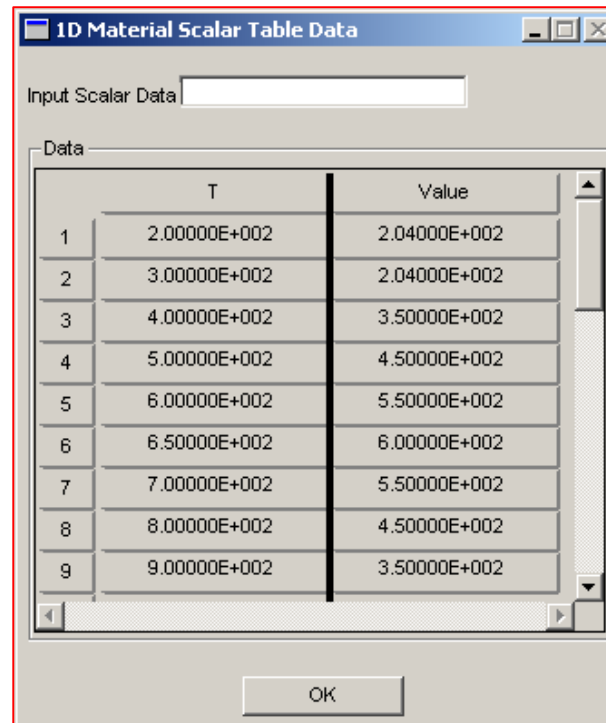
PROJECT 1, PART 2, $K = K(\text{TEMP})$

- Create temperature dependent function for the conduction material property.



The Fields dialog box is shown with the following settings:

- Action: Create
- Object: Material Property
- Method: Tabular Input
- Existing Fields: alum_t
- Field Name: alum_t
- Table Definition: Active Independent Variables
 - ☒ Temperature (T)
 - ☐ Strain (e)
 - ☐ Strain Rate (er)
 - ☐ Time (t)
 - ☐ Frequency (f)
- Buttons: Input Data ..., [Options...], -Apply-



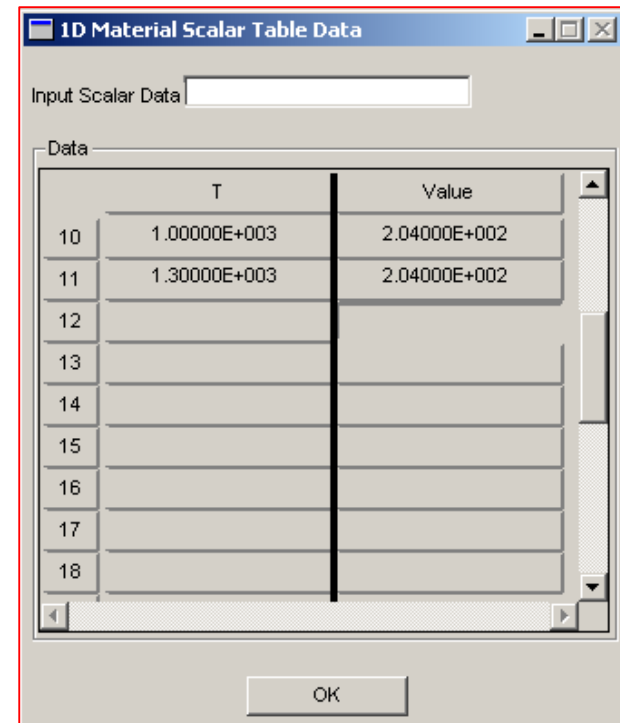
1D Material Scalar Table Data

Input Scalar Data

Data

	T	Value
1	2.00000E+002	2.04000E+002
2	3.00000E+002	2.04000E+002
3	4.00000E+002	3.50000E+002
4	5.00000E+002	4.50000E+002
5	6.00000E+002	5.50000E+002
6	6.50000E+002	6.00000E+002
7	7.00000E+002	5.50000E+002
8	8.00000E+002	4.50000E+002
9	9.00000E+002	3.50000E+002

OK



1D Material Scalar Table Data

Input Scalar Data

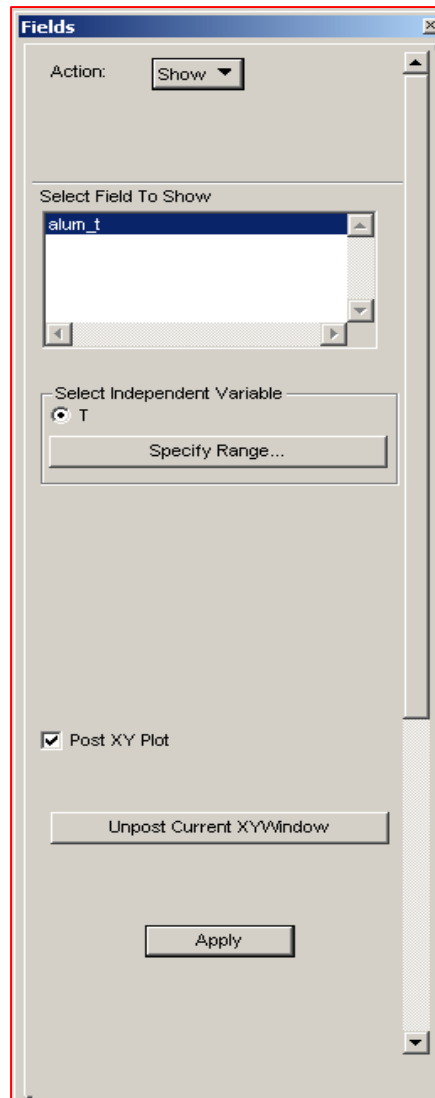
Data

	T	Value
10	1.00000E+003	2.04000E+002
11	1.30000E+003	2.04000E+002
12		
13		
14		
15		
16		
17		
18		

OK

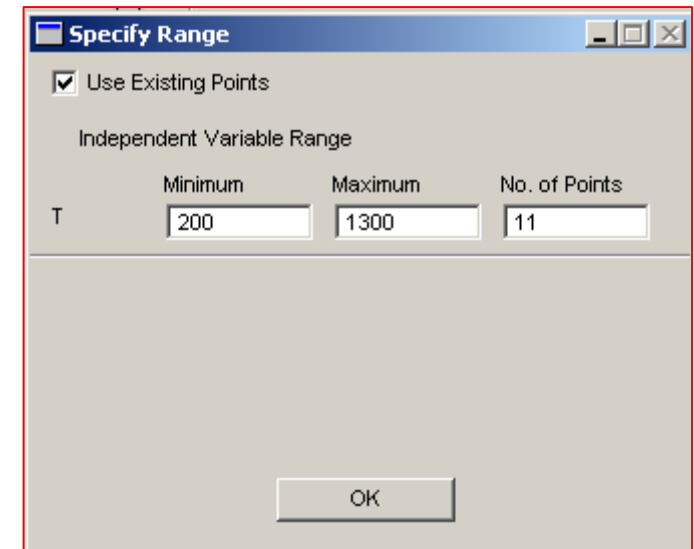
PROJECT 1, PART 2, $K = K(\text{TEMP})$

- Verify that the temperature dependent conductivity function is correct.



The 'Fields' dialog box is shown with the following settings:

- Action: Show
- Select Field To Show: alum_t
- Select Independent Variable: T
- Specify Range... button
- Post XY Plot: ☒
- Unpost Current XYWindow button
- Apply button



The 'Specify Range' dialog box is shown with the following settings:

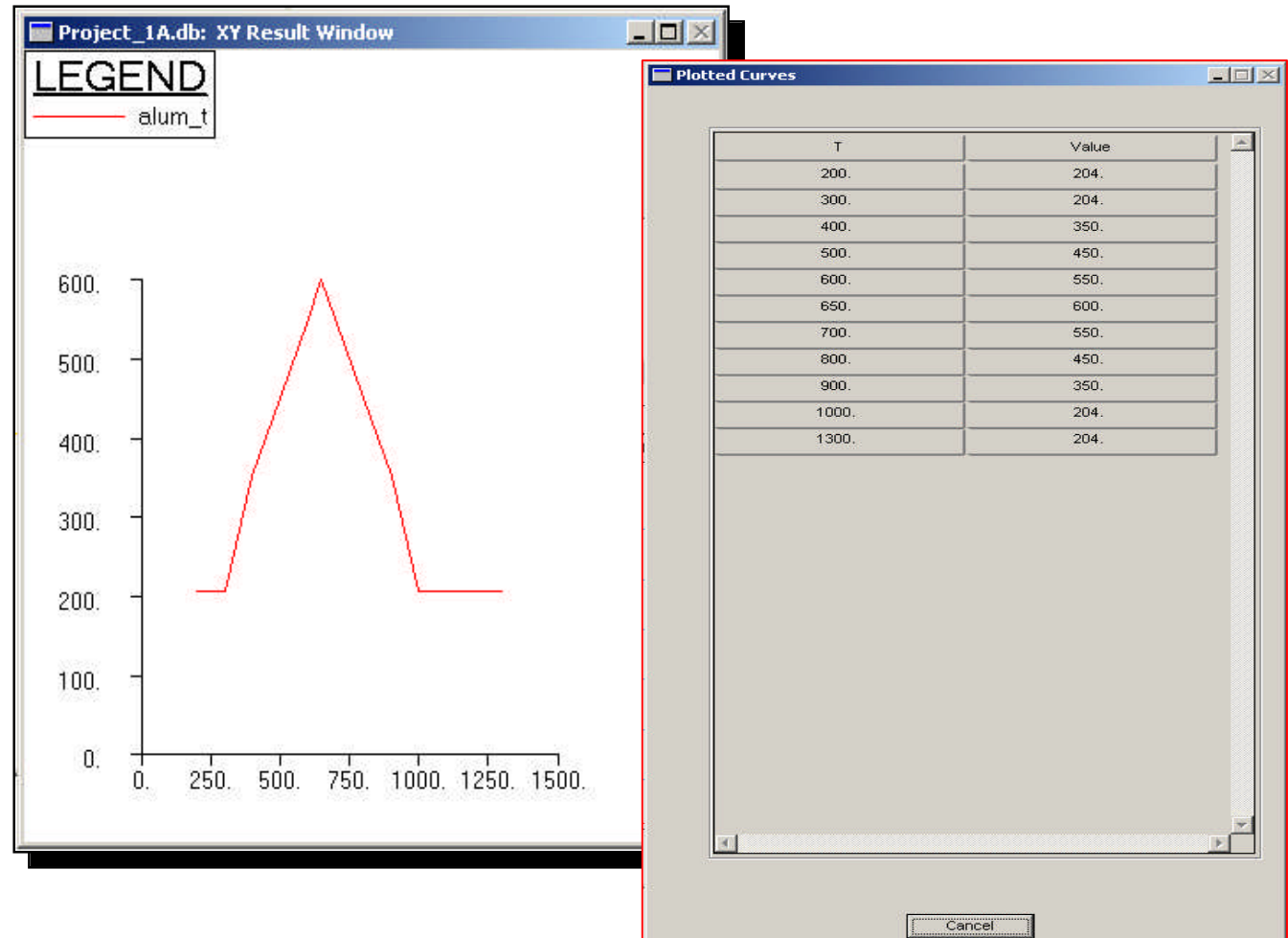
- Use Existing Points: ☒
- Independent Variable Range table:

	Minimum	Maximum	No. of Points
T	200	1300	11

OK button

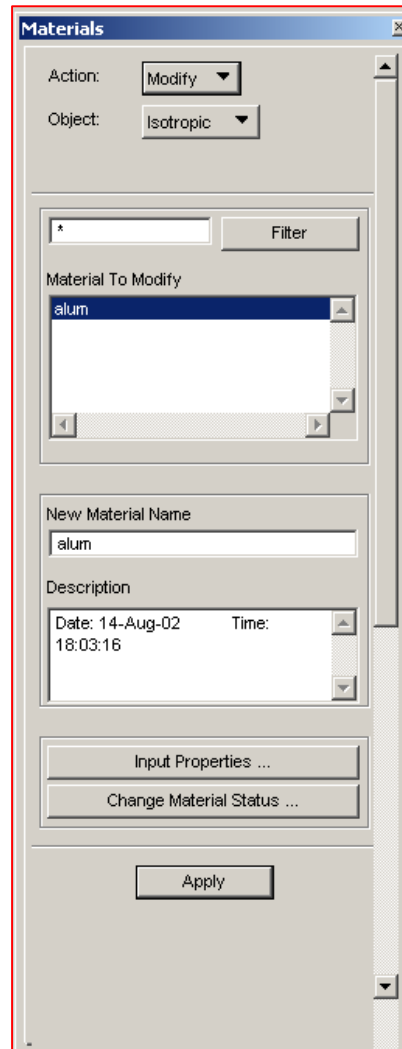
PROJECT 1, PART 2, $K = K(\text{TEMP})$

- Verification continued.

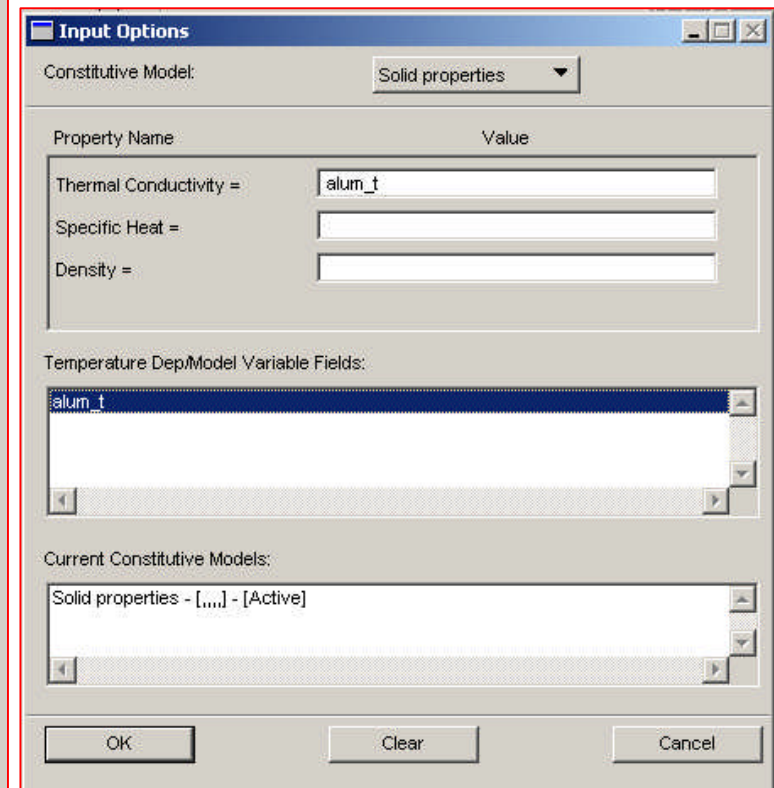


PROJECT 1, PART 2, $K = K(\text{TEMP})$

- Modify the conductivity to include temperature dependent function.



The Materials dialog box is used to modify material properties. It features a search bar with a filter button, a list of materials to modify (currently showing 'alum'), and fields for a new material name and description. The description field shows a date and time stamp. At the bottom, there are buttons for 'Input Properties ...', 'Change Material Status ...', and 'Apply'.



The Input Options dialog box is used to define the constitutive model for a material. It shows the 'Solid properties' model selected. The 'Property Name' and 'Value' columns are used to define material properties. The 'Temperature Dep/Model Variable Fields' section shows 'alum_t' selected. The 'Current Constitutive Models' section shows 'Solid properties - [...] - [Active]'. Buttons for 'OK', 'Clear', and 'Cancel' are at the bottom.

Property Name	Value
Thermal Conductivity =	alum_t
Specific Heat =	
Density =	

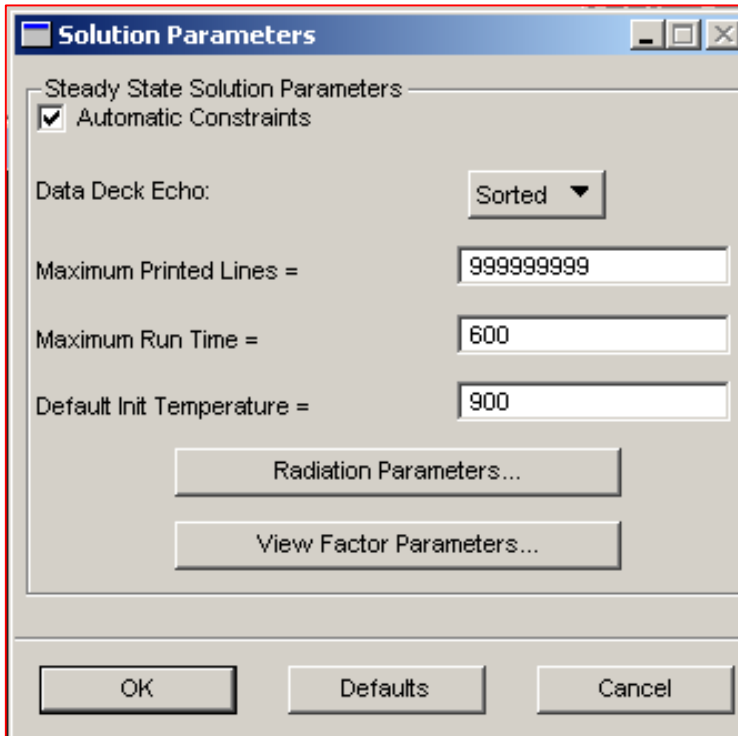
Temperature Dep/Model Variable Fields:

alum_t

Current Constitutive Models:

Solid properties - [...] - [Active]

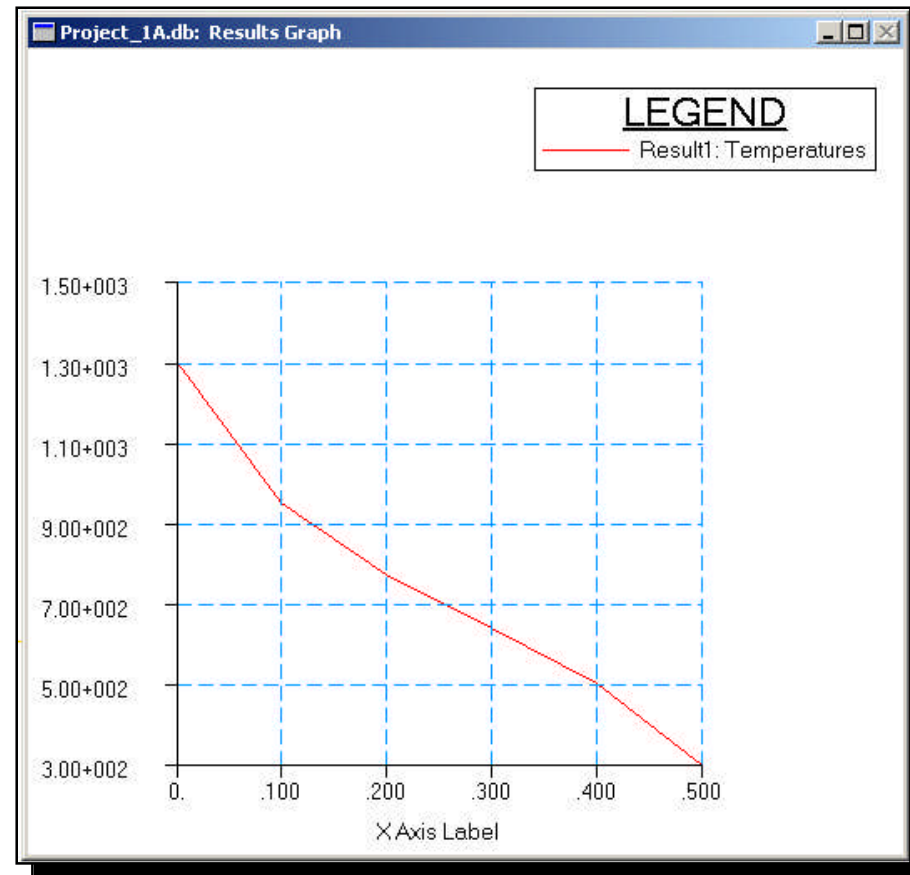
PROJECT 1, PART 2, $K = K(\text{TEMP})$



- Note that the Default initial temperature has been set to 900 degrees instead of 1300 degree, as was done for the previous problem.
 - Note: The initial temperature specification becomes important for convergence in radiation dominant problems.
- It is a good practice to have the initial (guessed) temperature (TEMPD) somewhere between your minimum and maximum temperature boundary conditions.

PROJECT 1, PART 2, $K = K(\text{TEMP})$

- Temperature versus X-direction path results



PROJECT 1, PART 2, K = K(TEMP)

Extract from MD Nastran ACSII Results File

```
0                                N O N - L I N E A R   I T E R A T I O N

      STIFFNESS UPDATE TIME      0.09 SECONDS
      ITERATION TIME             0.00 SECONDS

      - - - CONVERGENCE FACTORS - - -
OITERATION      EUI      EPI      EWI      LAMBDA      DLMAG
1    3.6714E+01    1.2164E+06    6.6263E+05    1.0000E-01    2.0787E+03
2    4.6298E-02    7.7274E+05    5.0114E+04    3.6764E-01    1.1734E+03
3    2.5080E-02    1.4878E+05    5.2448E+03    2.8009E-01    2.1876E+02
4    3.8106E-03    4.3102E+04    3.0188E+02    2.8490E-01    5.5085E+01
5    8.4501E-04    1.3270E+04    2.6202E+01    2.9638E-01    1.8630E+01
6    1.0828E-04    7.8692E+03    3.5682E+00    4.4470E-01    1.1870E+01
7    9.0914E-05    1.1174E+03    1.1230E-01    2.9334E-01    1.4579E+00
8    3.0700E-05    3.5478E+02    2.4425E-02    3.0543E-01    5.5234E-01
9    1.5216E-06    4.4025E+01    2.3781E-04    2.1476E-01    5.7797E-02
10   9.1694E-07    1.2036E+01    3.2268E-05    2.4407E-01    1.8947E-02
11   3.9523E-08    1.4948E+00    2.9203E-07    1.8414E-01    1.9987E-03
12   3.1472E-08    3.9194E-01    3.8085E-08    2.2317E-01    6.2589E-04

0*** USER      INFORMATION MESSAGE 6186,
      *** SOLUTION HAS CONVERGED ***
```

PROJECT 1, PART 2, K = K(TEMP)

Extract from MD Nastran ASCII Results File

LOAD STEP = 1.00000E+00
5 ROD -2.024025E+03 7.108403E+05

TEMPERATURE VECTOR

0

SUBCASE 1

POINT ID.	TYPE	ID	VALUE	ID+1	VALUE	ID+2	VALUE	ID+3	VALUE	ID+4	VALUE	ID+5	VALUE
1	S	1.300000E+03	9.515490E+02	7.691247E+02	6.389279E+02	5.024025E+02	3.000000E+02						
MSC/NASTRAN JOB CREATED ON 01-JUN-07 AT 16:18:30 JUNE 1, 2007 MSC/NASTRAN													
12/11/96 PAGE 10													
DEFAULT													

0

NONLINEAR

SUBCASE 1

LOAD STEP = 1.00000E+00

F I N I T E E L E M E N T T E M P E R A T U R E G R A D I E N T S A N D F L U X E S

ELEMENT-ID	EL-TYPE	X-GRADIENT	Y-GRADIENT	Z-GRADIENT	X-FLUX	Y-FLUX
Z-FLUX						
1	ROD	-3.484510E+03			7.108401E+05	
2	ROD	-1.824242E+03			7.108401E+05	
3	ROD	-1.301969E+03			7.108406E+05	
4	ROD	-1.365253E+03			7.108397E+05	

EXERCISES

- Perform Workshop 1 “Getting Started, Creating a Conduction Model” in the exercise workbook.



CONDUCTION THROUGH ANISOTROPIC MATERIALS



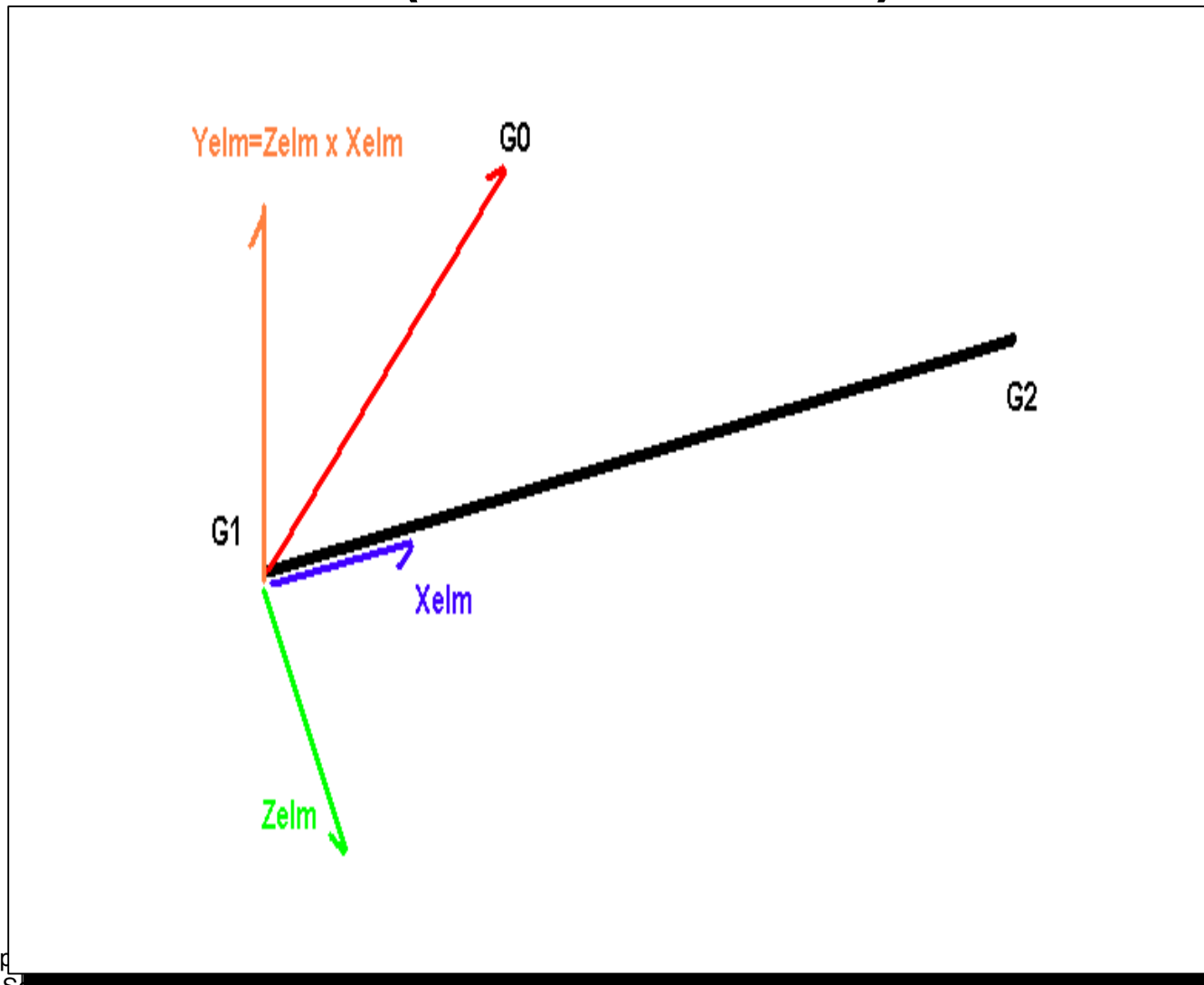


ELEMENT COORDINATE FRAMES PRELUDE TO ANISOTROPY

- Motivation
 - Proper orientation of material conductivity.
 - Proper interpretation of temperature gradient and flux results.
- 6 systems
 - 1D elements
 - 2D elements (CTRIA and CQUAD)
 - 3D elements (CHEXA, CPENTA and CTETRA)

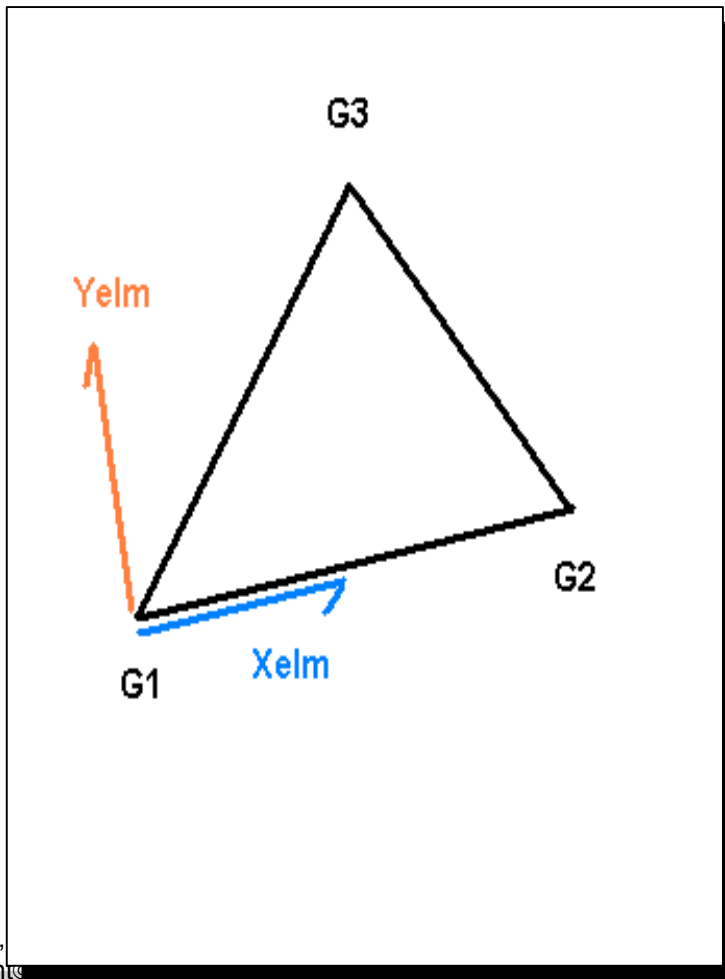


ELEMENT COORDINATE FRAME 1D (NO OFFSETS)

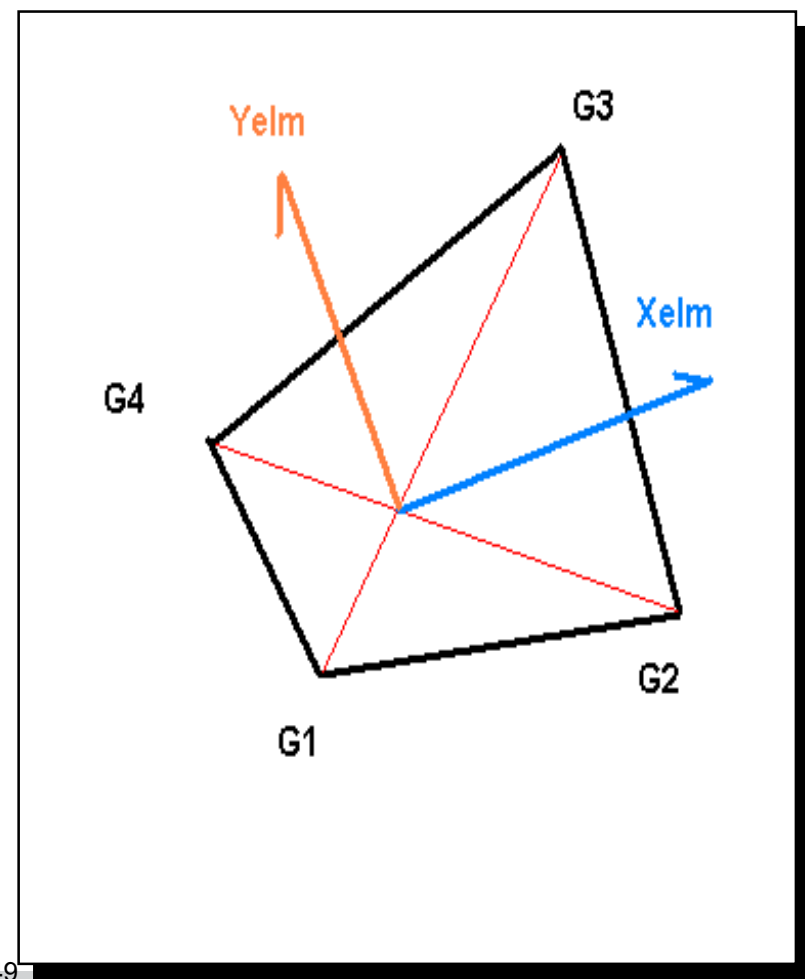


ELEMENT COORDINATE FRAMES 2D AND 3D

CTRIA3



CTETRA



MATERIAL ORIENTATION DEFAULTS

- Bar/beam - always aligned with element coordinate frame.
- Tria - defaults to element coordinate frame (change with rotation or MCID).
- Quad - defaults to element coordinate frame *only* when the elements are rectangular (change with rotation or MCID).
- Solid element orientation defaults to the basic coordinate frame (override with MCID).

MATERIAL ORIENTATION RECOMMENDATION

- For anisotropic materials, make sure that you specify the desired MCID (avoid specifying THETA).

ANISOTROPIC CONDUCTION (MAT5)

- The MAT5 record is used to define materials having direction dependent conductivity.
- The MAT5 record differs from the MAT4 in that there are no fields for specifying H or μ .
- Temperature dependence of the MAT5 properties is specified via the MATT5 (which again references TABLEM2 record).



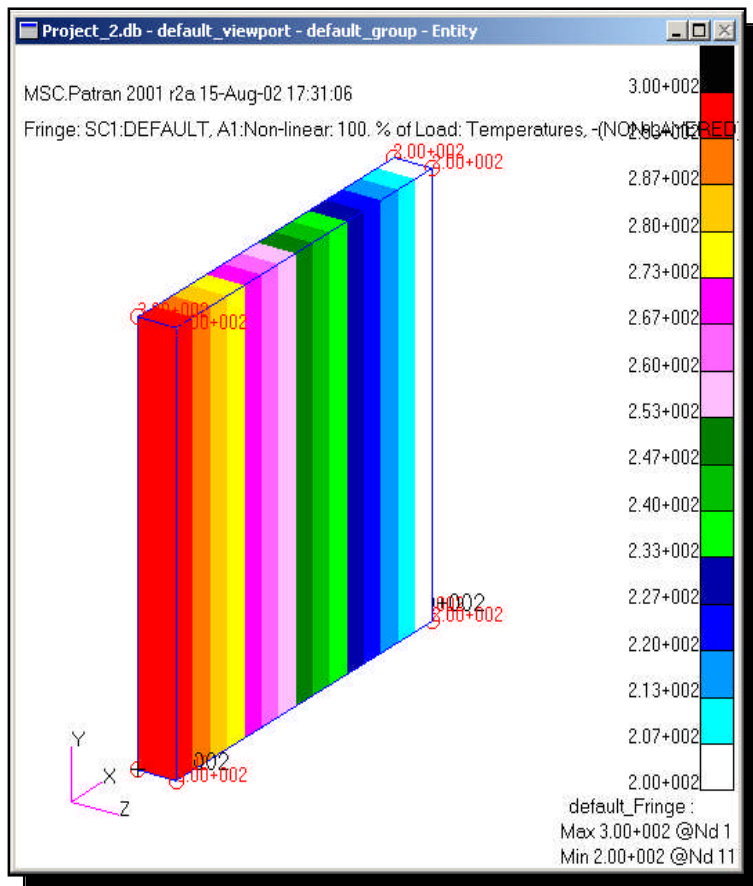
PROJECT 2

ANALYSIS WITH ORTHOTROPIC MATERIALS

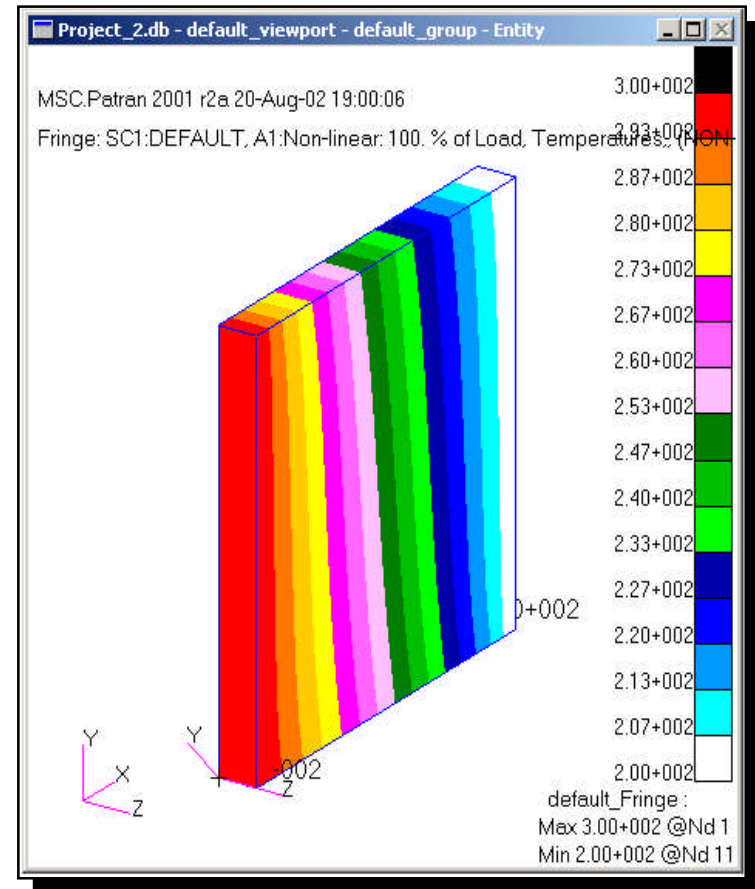
- 10x10x1 plate
- Finite element model using solid elements (CHEXA).
- $K_x=204.0$, $K_y=304.0$, $K_z=404.0$
- Two material orientations
 - Part 1, MD Nastran basic coordinate system, or
 - Part 2, Local coordinate system, rotated 45° about Z-axis (use MCID)
- Left face held at 300°C , and right face at 200°C .
- Steady-state analyses
- Plot fringe results.
- Examine .f06 file.

PROJECT 2

ANALYSIS WITH ORTHOTROPIC MATERIALS



Material Coordinate Directions
Parallel to MD Nastran Basic



Material Coordinate Directions
Parallel to Rotated Coord System

PROJECT 2, PART 1, BASIC COORD ANALYSIS WITH ORTHOTROPIC MATERIALS

- Create 10x10x1 geometric solid.
- IsoMesh the solid with CHEXA solid elements.

Geometry

Action:

Object:

Method:

Solid ID List

Refer. Coordinate Frame

Vector Coordinates List

☒ Auto Execute

Origin Coordinates List

Finite Elements

Action:

Object:

Type:

Output ID List

Node

Element

Elem Shape

Mesher

Topology

Solid List

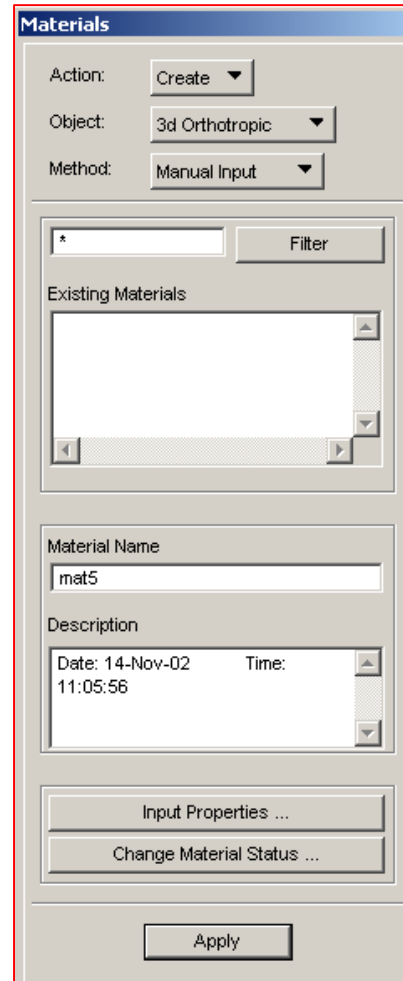
Global Edge Length

☐ Automatic Calculation

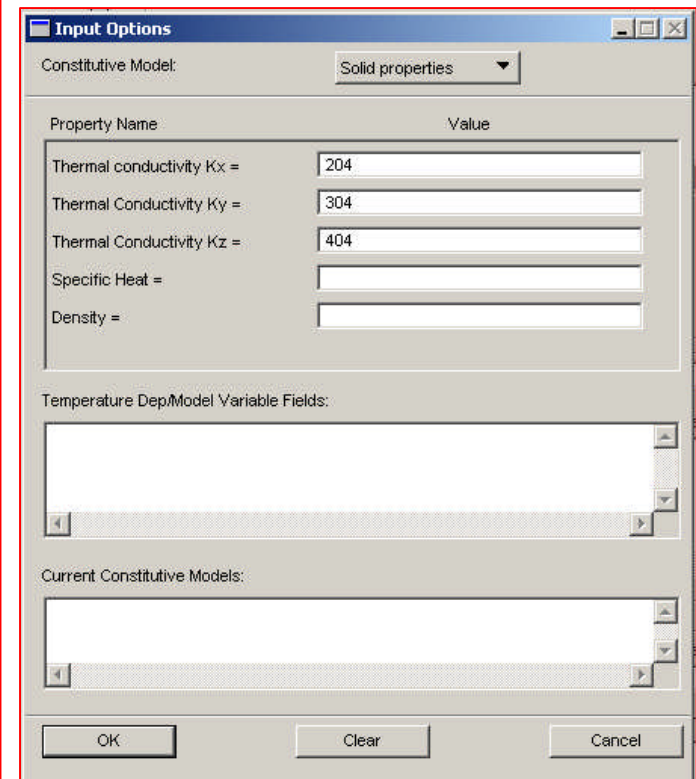
Value

PROJECT 2, PART 1, BASIC COORD ANALYSIS WITH ORTHOTROPIC MATERIALS

- Create 3D orthotropic material.
- The properties, K_x , K_y , K_z , are constant and are to be used for the default direction.



The Materials dialog box is used to create a new material. It features a title bar 'Materials' and a close button. The 'Action' dropdown is set to 'Create', 'Object' is '3d Orthotropic', and 'Method' is 'Manual Input'. There is a search field with a '*' character and a 'Filter' button. Below is a list box for 'Existing Materials'. The 'Material Name' field contains 'mat5', and the 'Description' field shows 'Date: 14-Nov-02' and 'Time: 11:05:56'. At the bottom are buttons for 'Input Properties ...', 'Change Material Status ...', and 'Apply'.



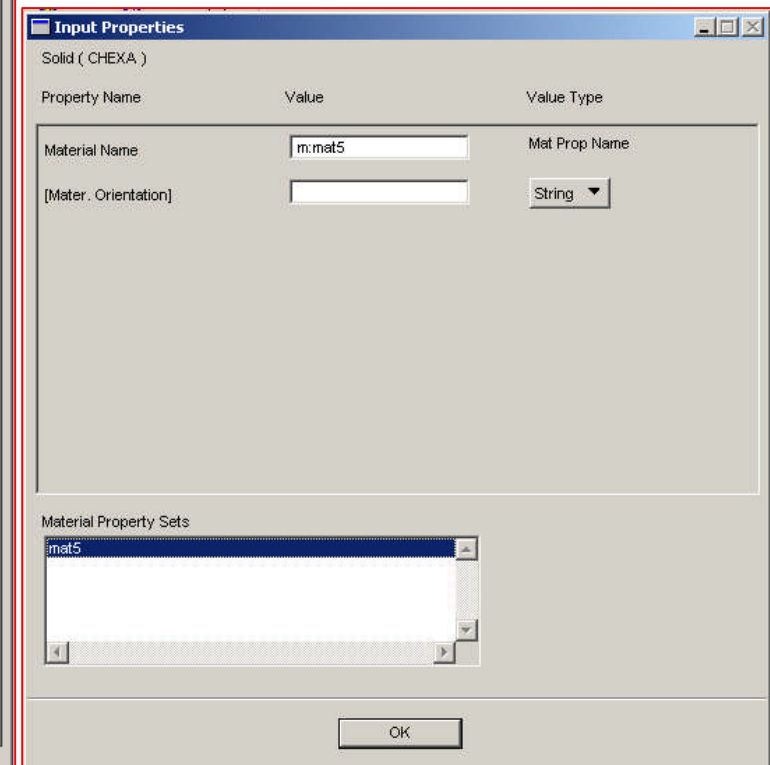
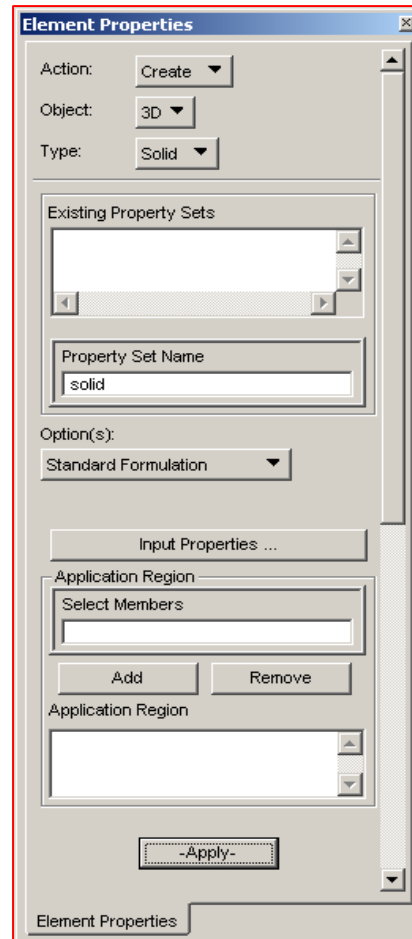
The Input Options dialog box is used to define the constitutive model for the material. It has a title bar 'Input Options' and standard window controls. The 'Constitutive Model' dropdown is set to 'Solid properties'. Below is a table for material properties:

Property Name	Value
Thermal conductivity K_x =	204
Thermal Conductivity K_y =	304
Thermal Conductivity K_z =	404
Specific Heat =	
Density =	

Below the table is a section for 'Temperature Dep/Model Variable Fields' with an empty list box. At the bottom is a section for 'Current Constitutive Models' with another empty list box. The dialog concludes with 'OK', 'Clear', and 'Cancel' buttons.

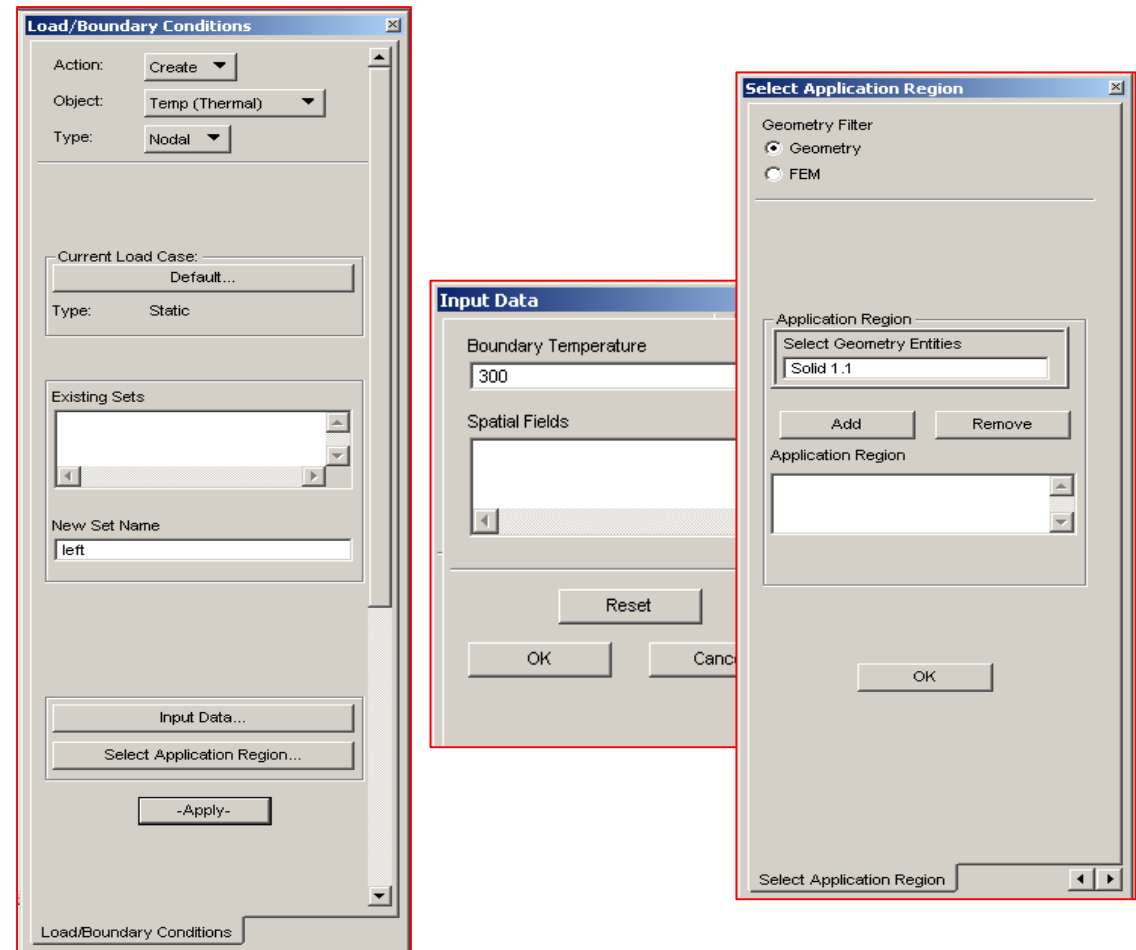
PROJECT 2, PART 1, BASIC COORD ANALYSIS WITH ORTHOTROPIC MATERIALS

- Create 3D solid element properties.
- The default material direction (orientation) is used.



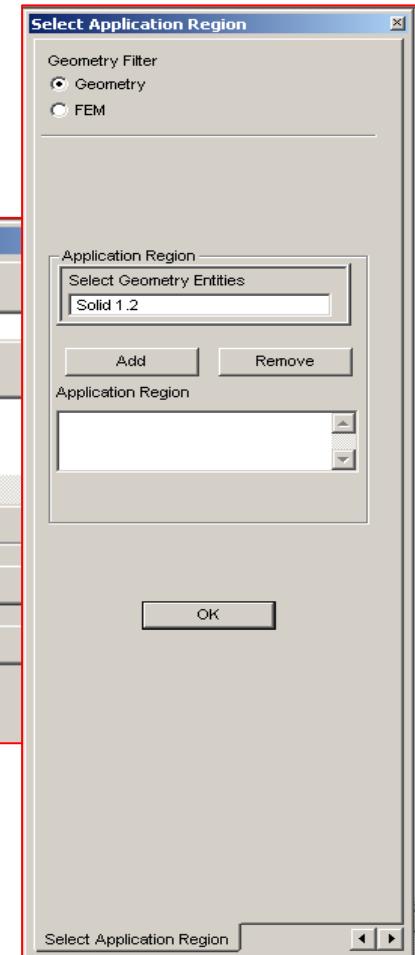
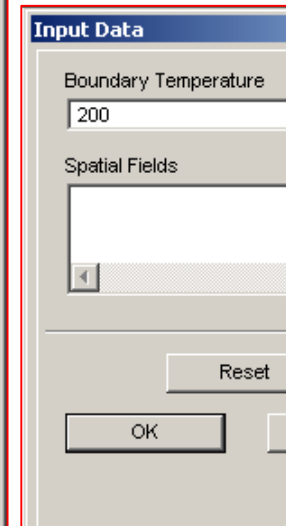
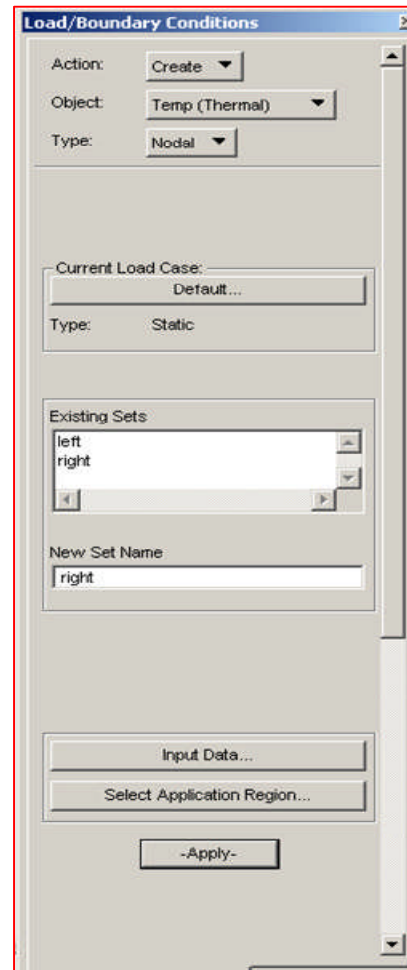
PROJECT 2, PART 1, BASIC COORD ANALYSIS WITH ORTHOTROPIC MATERIALS

- Create a fixed temperature of 300 °C on the left face of the solid, Solid 1.1.



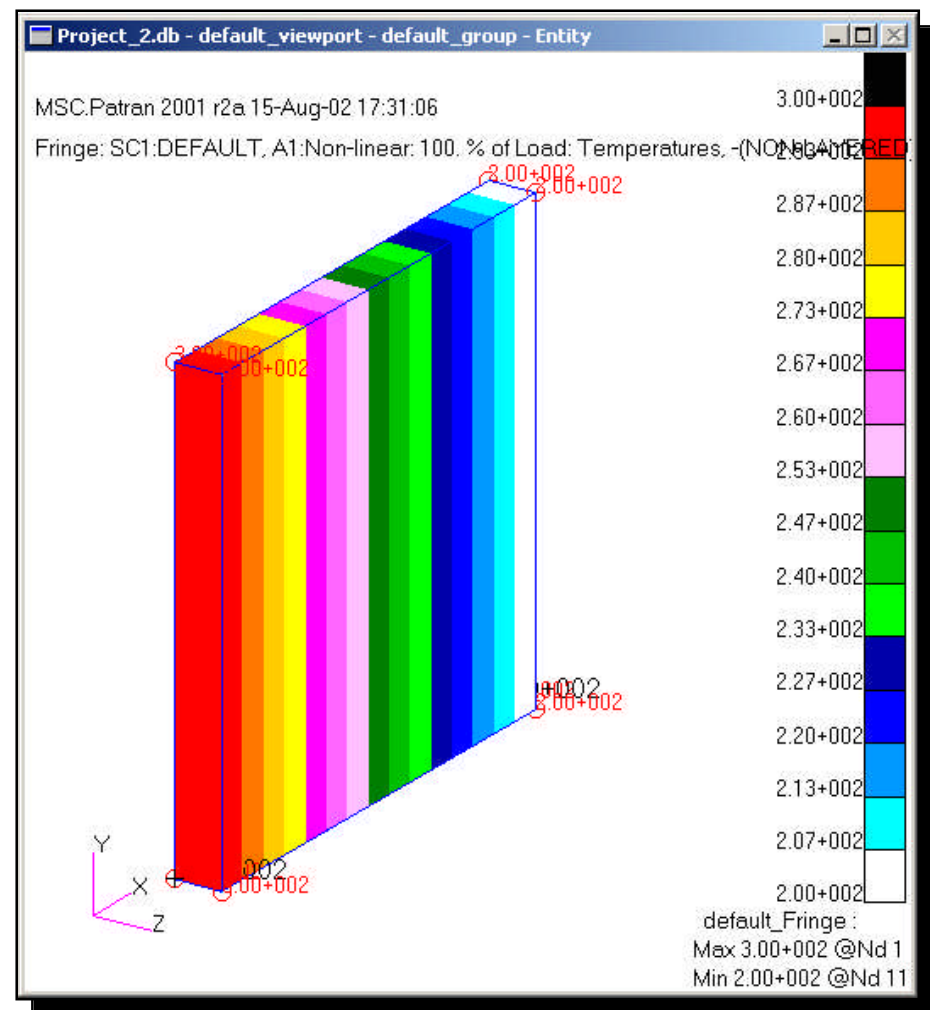
PROJECT 2, PART 1, BASIC COORD ANALYSIS WITH ORTHOTROPIC MATERIALS

- Create a fixed temperature of 200 °C on the right face of the solid, Solid 1.2.



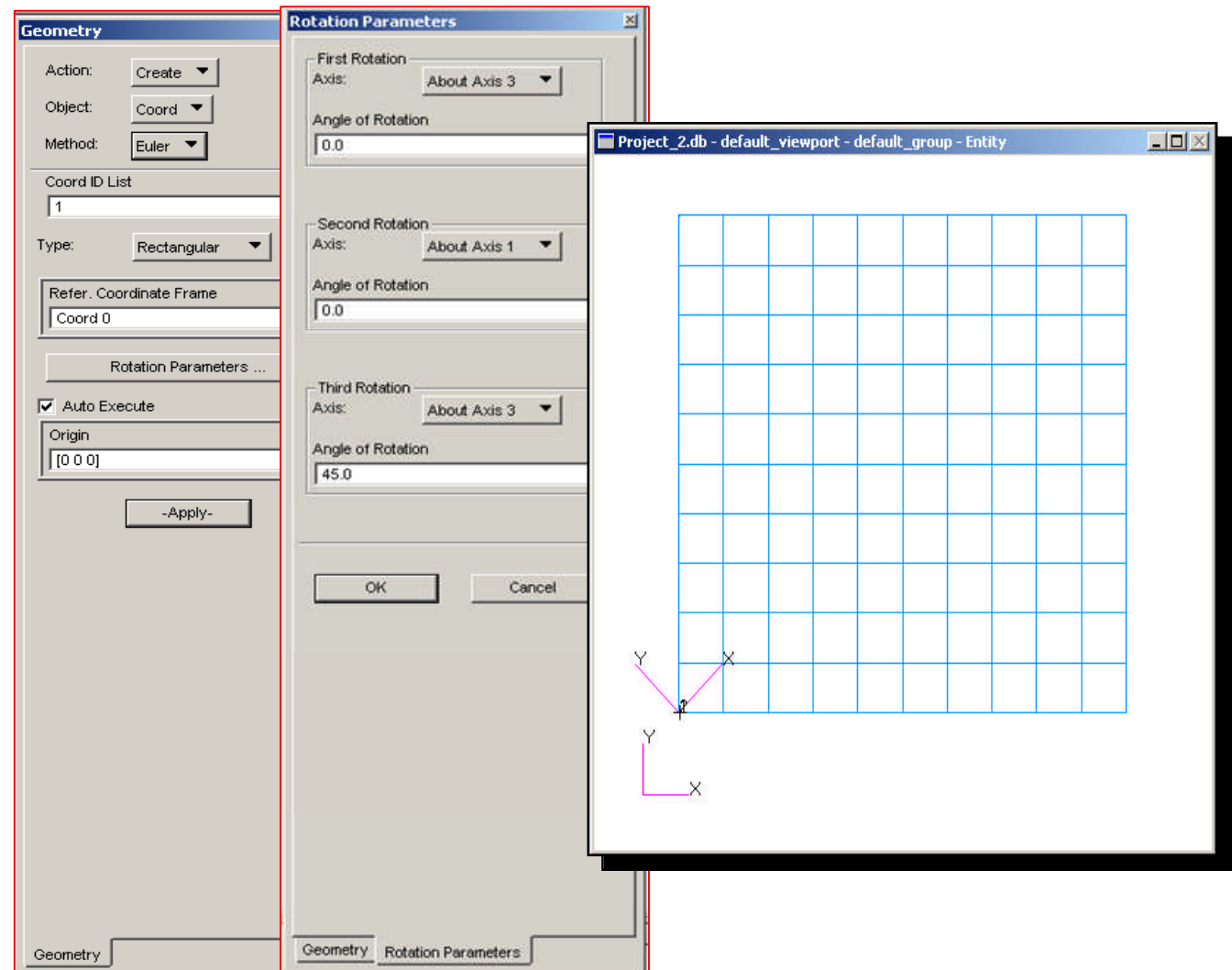
PROJECT 2, PART 1, BASIC COORD ANALYSIS WITH ORTHOTROPIC MATERIALS

- Temperature fringe plot for default material orientation.



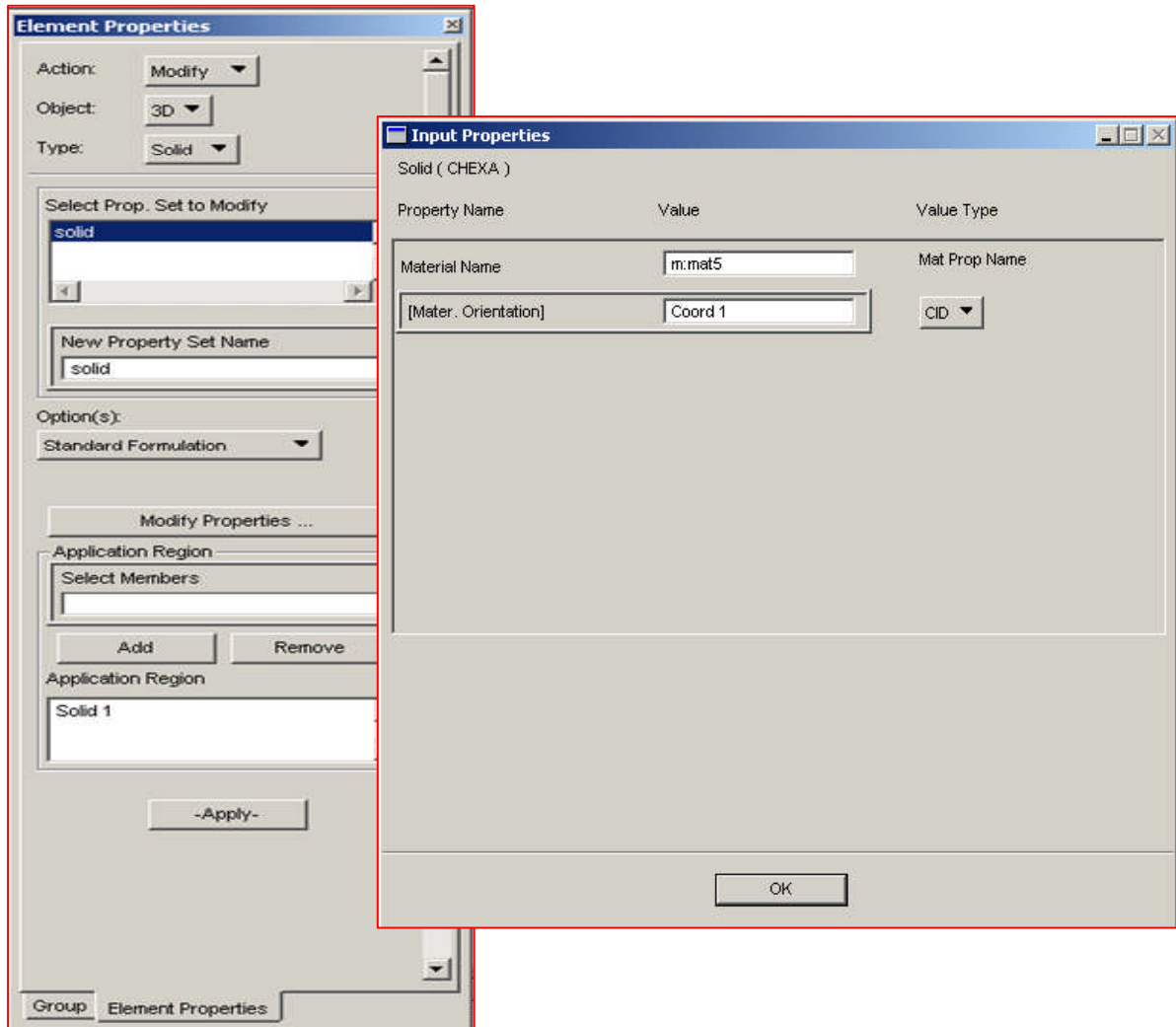
PROJECT 2, PART 2, ROTATED COORD ANALYSIS WITH ORTHOTROPIC MATERIALS

- Create local rotated coordinate frame for the material orientation.



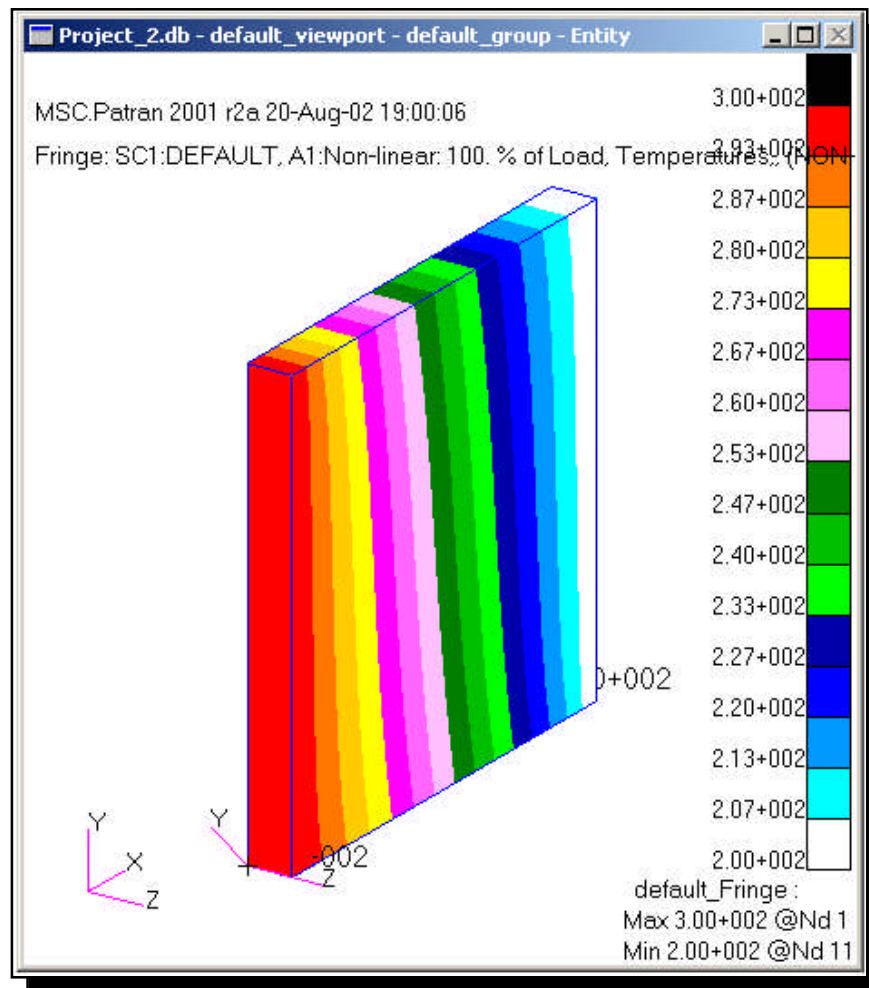
PROJECT 2, PART 2, ROTATED COORD ANALYSIS WITH ORTHOTROPIC MATERIALS

- Modify the 3D solid element property set to include the local rotated coordinate frame, Coord 1.



PROJECT 2, PART 2, ROTATED COORD

ANALYSIS OF ORTHOTROPIC MATERIALS



This picture shows that the temperature contour has been shifted because the material aligns with the material coordinate system that is rotated 45 degrees about the Z-direction.

PROJECT 2, PART 2, ROTATED COORD ANALYSIS OF ORTHOTROPIC MATERIALS

Extract from MD Nastran model .DAT file

```
$PSOLID, PID, MID, CORDM
PSOLID    1      1      1
$CHEXA, EID, PID, G1, G2, G3, G4, G5, G6
CHEXA     1      1      1      2      13      12      122      123      +      B
+         B 134      133
CORD2R    1      0.      0.      0.      0.      0.      1.      +      CX
+         CX .707107 .707107 0.
$MAT5, MID, KXX, KXY, KXZ, KYY, KYZ, KZZ
MAT5      1      204.      304.      404.
```

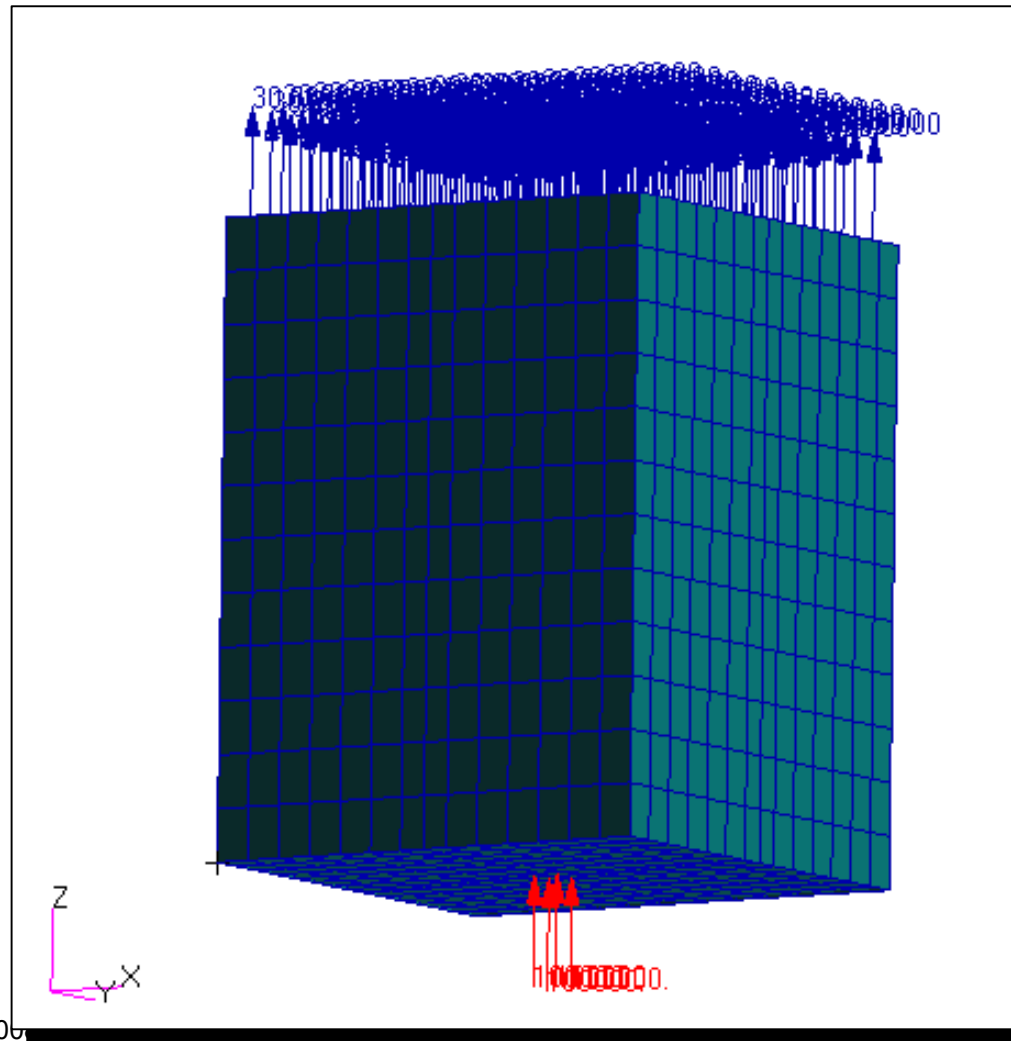
CONDUCTION THROUGH MD NASTRAN SOL 400 3D COMPOSITE ELEMENTS



SOL 400 3D COMPOSITE ELEMENT

- Steady-state analysis
- The model consists of a rectangular shaped box of CHEXA elements, with free convection and heat flux boundary conditions.
- Mesh consists of 2352 CHEXA elements, with 20 nodes per element.
 - The mesh in the X- and Y-Directions is biased towards the center of the mesh
 - The mesh in the Z-Direction is not biased
- The free convection is to an ambient temperature of 0.0 °F.
- The heat flux is on the four element faces at the center of the X-Y plane.

SOL 400 3D COMPOSITE ELEMENT



NAS104, Section 3, September 2006

Copyright© 2008 MSC Software Corporation

S3-68

SOL 400 3D COMPOSITE ELEMENT

```

SOL 400
CEND
ANALYSIS = HSTAT
TEMPERATURE(INITIAL) = 1
SUBCASE 1
    TITLE=This is a default subcase.
    NLPARM = 1
    SPC = 1
    LOAD = 2
BEGIN BULK
$PCOMPLS,PID,DIRECT,CORDM,SB,ANAL
PCOMPLS,1,-1,,,IH
$,C20,BEH20,INT20,BEH20H,INT20H
,C20,,,SLCO,Q
$,IDI,MIDI,TI,THETAi
,101,1,0.001
,102,1,0.001
,103,1,0.001
,104,1,0.001
,105,1,0.001
,106,1,0.001
,107,1,0.001
,108,1,0.001
,109,1,0.001
,110,1,0.001
,111,1,0.001
,112,1,0.001
NLPARM    1      1      1      0      ITER    1      25      UPW      NO
$PSOLID   1      1      1      0
MDLPRM,NLDIAG,10
CHEXA     1      1      23      21      1      3      773      766
           768      771      22      14      2      15      774      767
           769      772      777      770      776      775
...
MAT4      1      1.
    
```

SOL 400 3D COMPOSITE ELEMENT

```

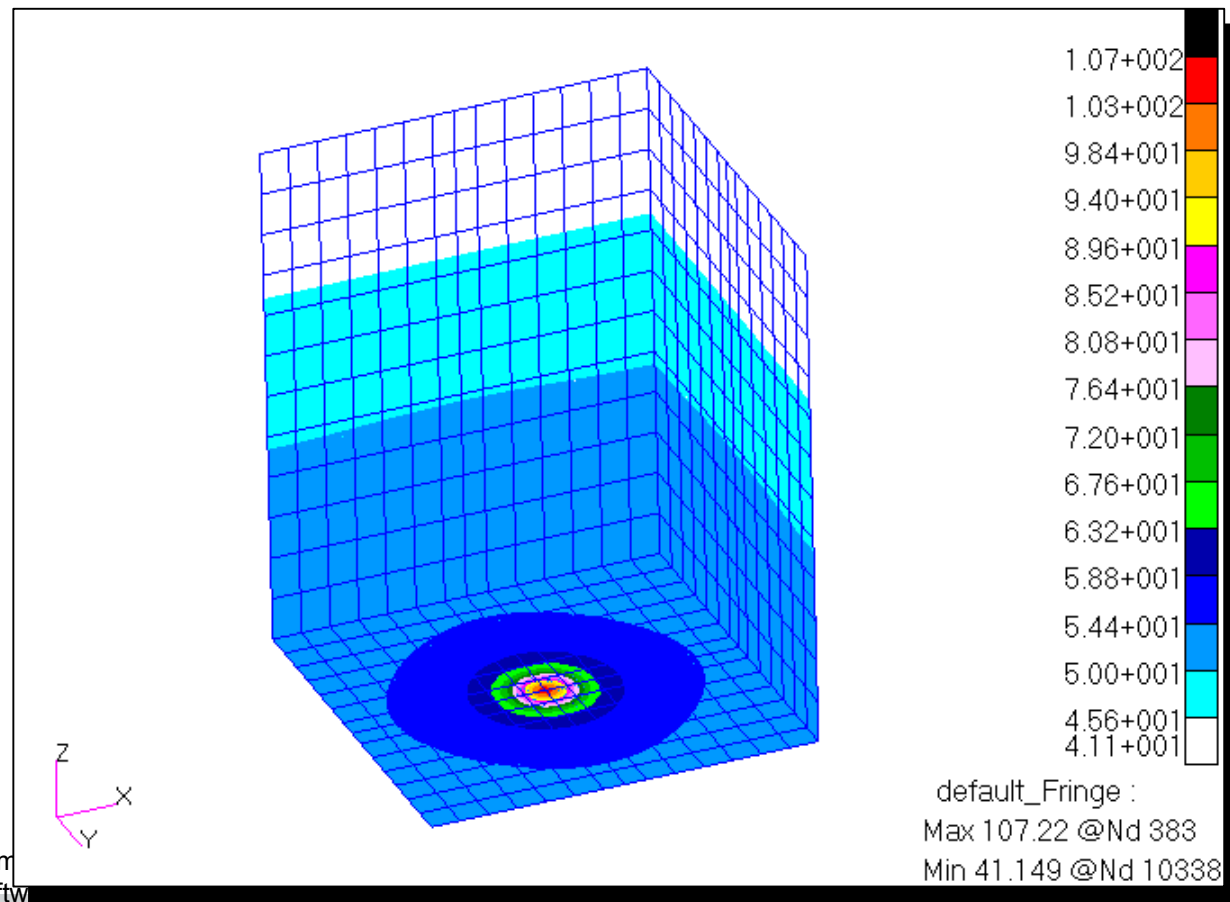
$ Nodes of the Entire Model
GRID      1              0.      0.      0.
...
$ Fixed Temperatures of Load Set : conv_3d
SPC       1      11206      1      0.
$ Normal Heat Flux of Load Set : flux
QBDY3     2      100000.      100001
QBDY3     2      100000.      100002
QBDY3     2      100000.      100003
QBDY3     2      100000.      100004
$ Convection to Ambient of Load Set : conv_3d
PCONV     1      1001      0      0.
CONV      100005      1      0      0      11206
...
$ CHBDYG Surface Elements
CHBDYG     100001      AREA8
          383      370      133      166      379      369      153      382
CHBDYG     100002      AREA8
          385      372      370      383      380      371      379      384
CHBDYG     100003      AREA8
          391      383      166      186      387      382      173      390
CHBDYG     100004      AREA8
          393      385      383      391      388      384      387      392
CHBDYG     100005      AREA8
          10336      10338      10341      10343      10340      10346      10345      10347
...

$ Free Convection Heat Transfer Coefficients
MAT4      1001              30.
$ Scalar Points
SPOINT    11206
ENDDATA bc2d5421

```

SOL 400 3D COMPOSITE ELEMENT

- Results for 3D composite elements
- Temperature gradient in the three directions for the elements



NAS104, Section 3, Septem
Copyright© 2008 MSC.Softw

EXERCISES

- Perform Workshop 22 “MD Nastran SOL 400 3D Composite Element” in the exercise workbook.

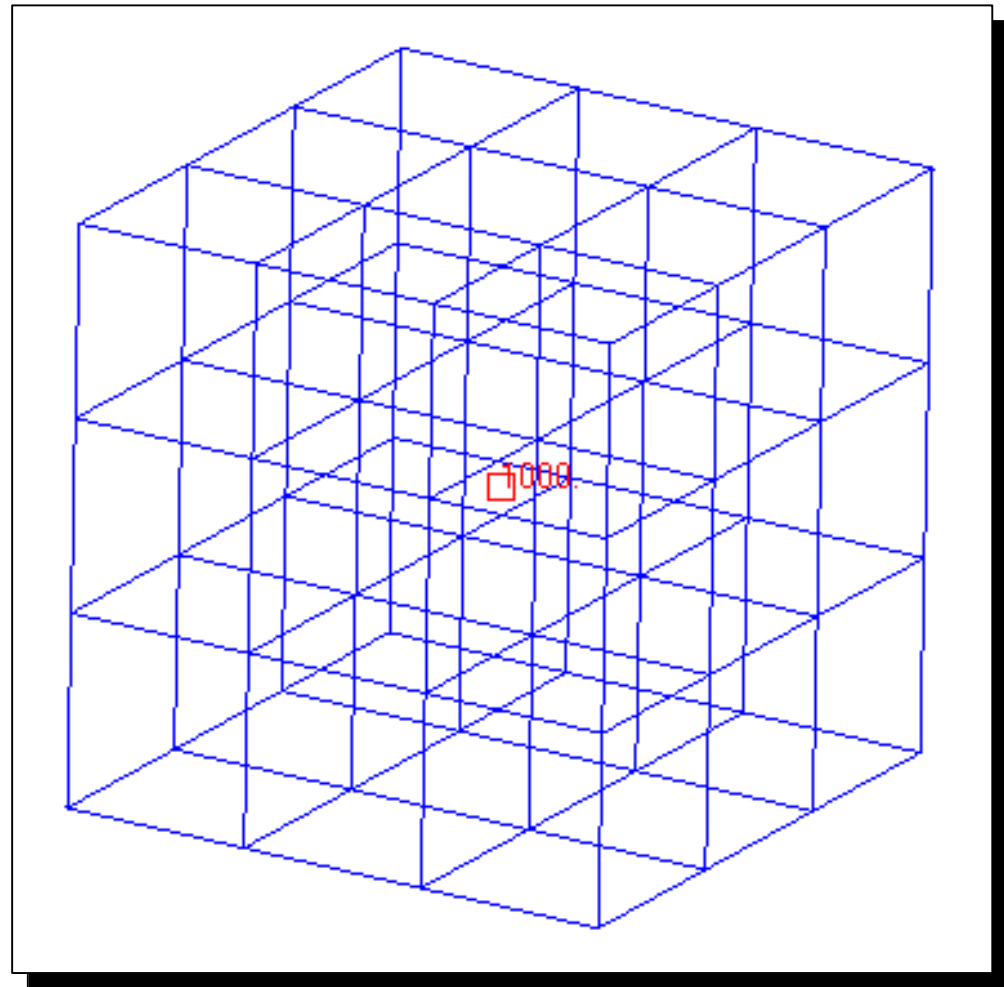
VOLUMETRIC HEAT GENERATION





3 x 3 CHEXA MESH WITH QVOL HEAT

- CHEXA mesh with volumetric heating (QVOL) for center element



QVOL BULK DATA ENTRY

QVOL

Volume Heat Addition

Defines a rate of volumetric heat addition in a conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
QVOL	SID	QVOL	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

Example:

QVOL	5	10.0	101	10	12	11	9		

Field	Contents
SID	Load set identification. (Integer > 0)
QVOL	Power input per unit volume produced by a heat conduction element. (Real)
CNTRLND	Control point used for controlling heat generation. (Integer ≥ 0 ; Default = 0)
EIDi	A list of heat conduction elements. (Integer > 0 or "THRU" or "BY")

MAT4 BULK DATA ENTRY W/ HGEN

MAT4

Heat Transfer Material Properties, Isotropic

Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.

Format:

1	2	3	4	5	6	7	8	9	10
MAT4	MID	K	CP	ρ	H	μ	HGEN	REFENTH	
	TCH	TDELTA	QLAT						

Example:

MAT4	1	204.	.900	2700.					

Field	Contents
MID	Material identification number. (Integer > 0)
K	Thermal conductivity. (Blank or Real > 0.0)
CP	Heat capacity per unit mass at constant pressure (specific heat). (Blank or Real $\geq$ 0.0)
ρ	Density. (Real > 0.0; Default = 1.0)
H	Free convection heat transfer coefficient. (Real or blank)
μ	Dynamic viscosity. See Remark 2. (Real > 0.0 or blank)
HGEN	Heat generation capability used with QVOL entries. (Real $\geq$ 0.0; Default = 1.0)

VOLUMETRIC HEAT GENERATION (QVOL)

- Thermal loads arising from volumetric heat generation are specified by the QVOL entry
 - $Q_{in} = \text{volume}_{ELEMENT} * HGEN * QVOL * u_{CNTRLND}$

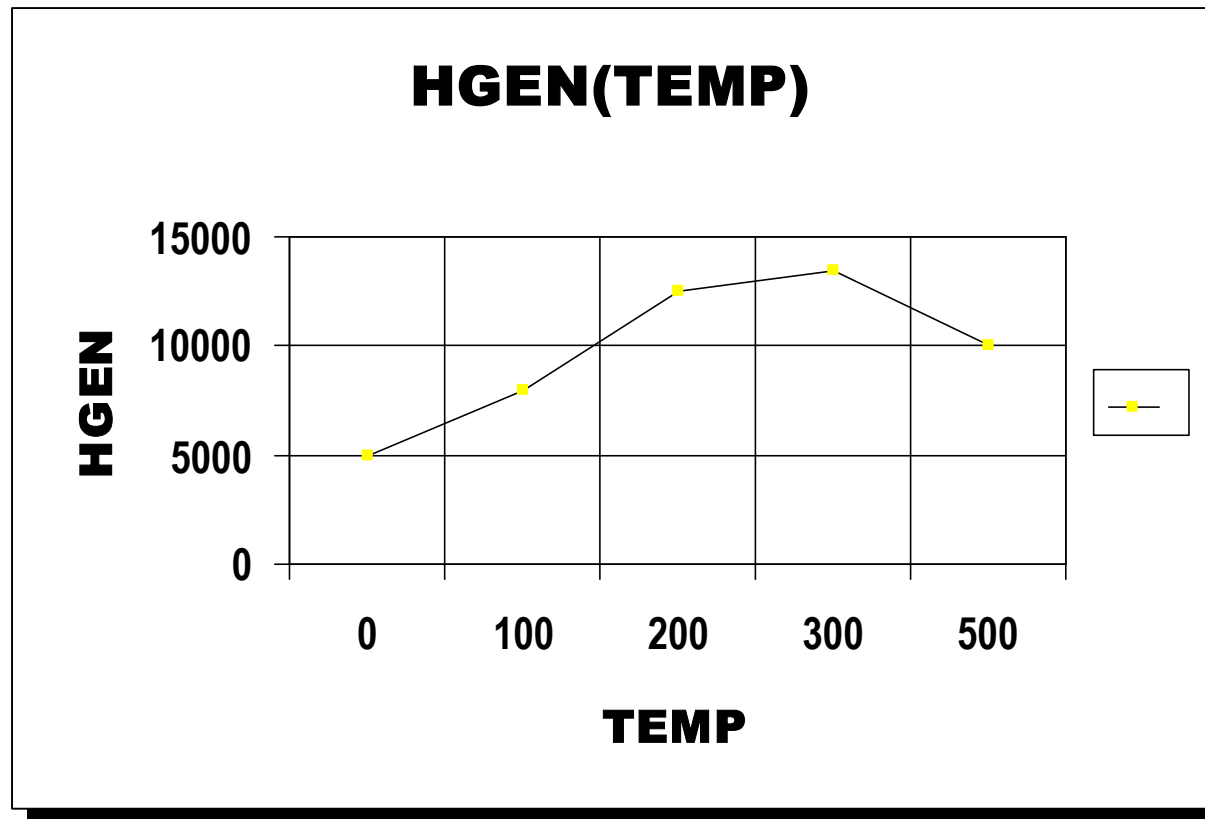
$$\nabla^2 T + \frac{\dot{q}}{k} = \frac{1}{\alpha} \frac{\partial T}{\partial t}$$

$$\dot{q} = HGEN \cdot QVOL$$

- Use the previous example, now using isotropic material. Apply volumetric heating as a function of temperature, and keep the 200 °C on the right face.

PROJECT 3, VOLUMETRIC HEATING AS A FUNCTION OF TEMPERATURE

- HGEN is a function of temperature (TEMP)

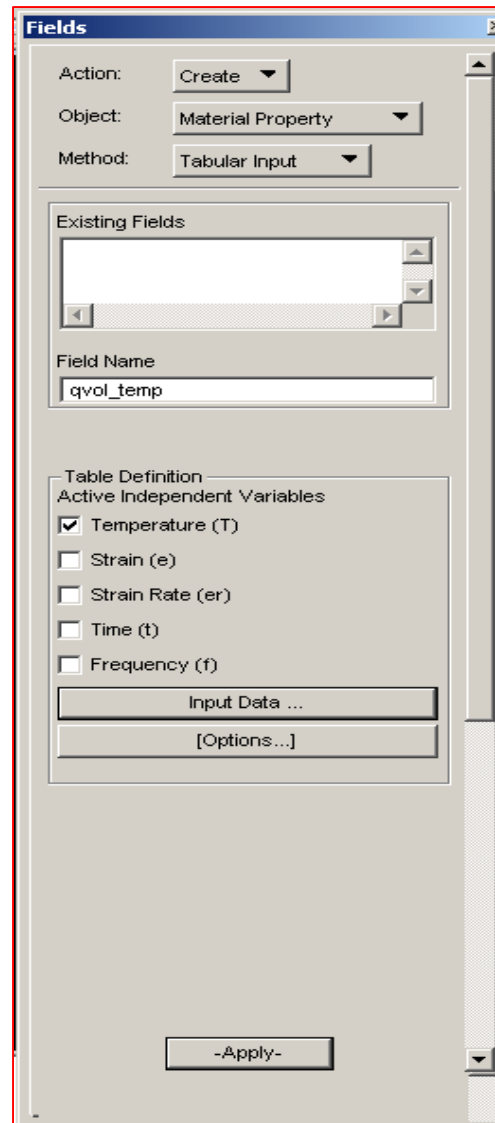


PROJECT 3, Q_{IN} AND QVOL

- The total heat is $Q_{in} = \text{volume}_{ELEMENT} * \text{HGEN}(\text{temperature}) * \text{QVOL}$
 - $\text{volume}_{ELEMENT}$ (volume of element) is calculated by Patran
 - $\text{HGEN}(\text{temperature})$ is calculated by Patran using a material property field, e.g. MAT4, MATT4, and TABLEM1
 - QVOL = constant, e.g. 10.0

PROJECT 3, Patran HGEN(TEMP)

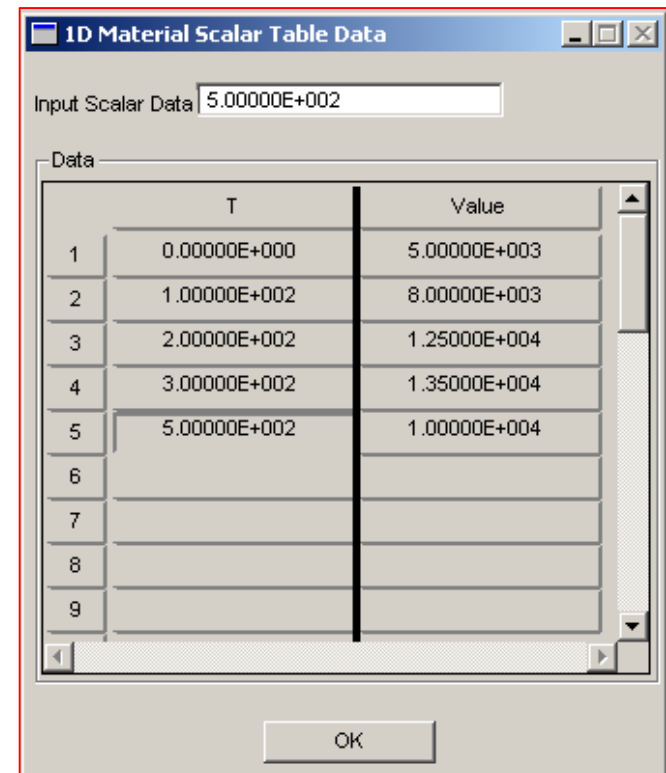
- Create field for material property.
- Input the field data using tabular input.
- The material property, HGEN, is temperature dependent.



The Fields dialog box is used to create a new field. It includes the following sections:

- Action:** Create (dropdown)
- Object:** Material Property (dropdown)
- Method:** Tabular Input (dropdown)
- Existing Fields:** A list box for selecting existing fields.
- Field Name:** qvol_temp (text input)
- Table Definition:**
 - Active Independent Variables:
 - ☒ Temperature (T)
 - ☐ Strain (e)
 - ☐ Strain Rate (er)
 - ☐ Time (t)
 - ☐ Frequency (f)
- Input Data ...** (button)
- [Options...]** (button)

-Apply-



The 1D Material Scalar Table Data dialog box is used to input scalar data. It includes the following sections:

- Input Scalar Data:** 5.00000E+002 (text input)
- Data:** A table with 9 rows and 2 columns: T and Value.

	T	Value
1	0.00000E+000	5.00000E+003
2	1.00000E+002	8.00000E+003
3	2.00000E+002	1.25000E+004
4	3.00000E+002	1.35000E+004
5	5.00000E+002	1.00000E+004
6		
7		
8		
9		

OK

PROJECT 3, INPUT MATERIAL CONDUCTIVITY

Materials

Action:

Object:

Method:

*

Existing Materials

Material Name

Description
Date: 14-Nov-02 Time: 11:05:56

Input Options

Constitutive Model:

Property Name	Value
Thermal Conductivity =	204
Specific Heat =	
Density =	

Temperature Dep/Model Variable Fields:

Current Constitutive Models:

PROJECT 3, HGEN(TEMP) UNDER MAT'L MENU

- Change the constitutive model from *Solid properties* to *Heat generation* using the Input Properties form.
- Select the Field **qvol_temp** listed under Temperature Dep/Model Variable Fields.

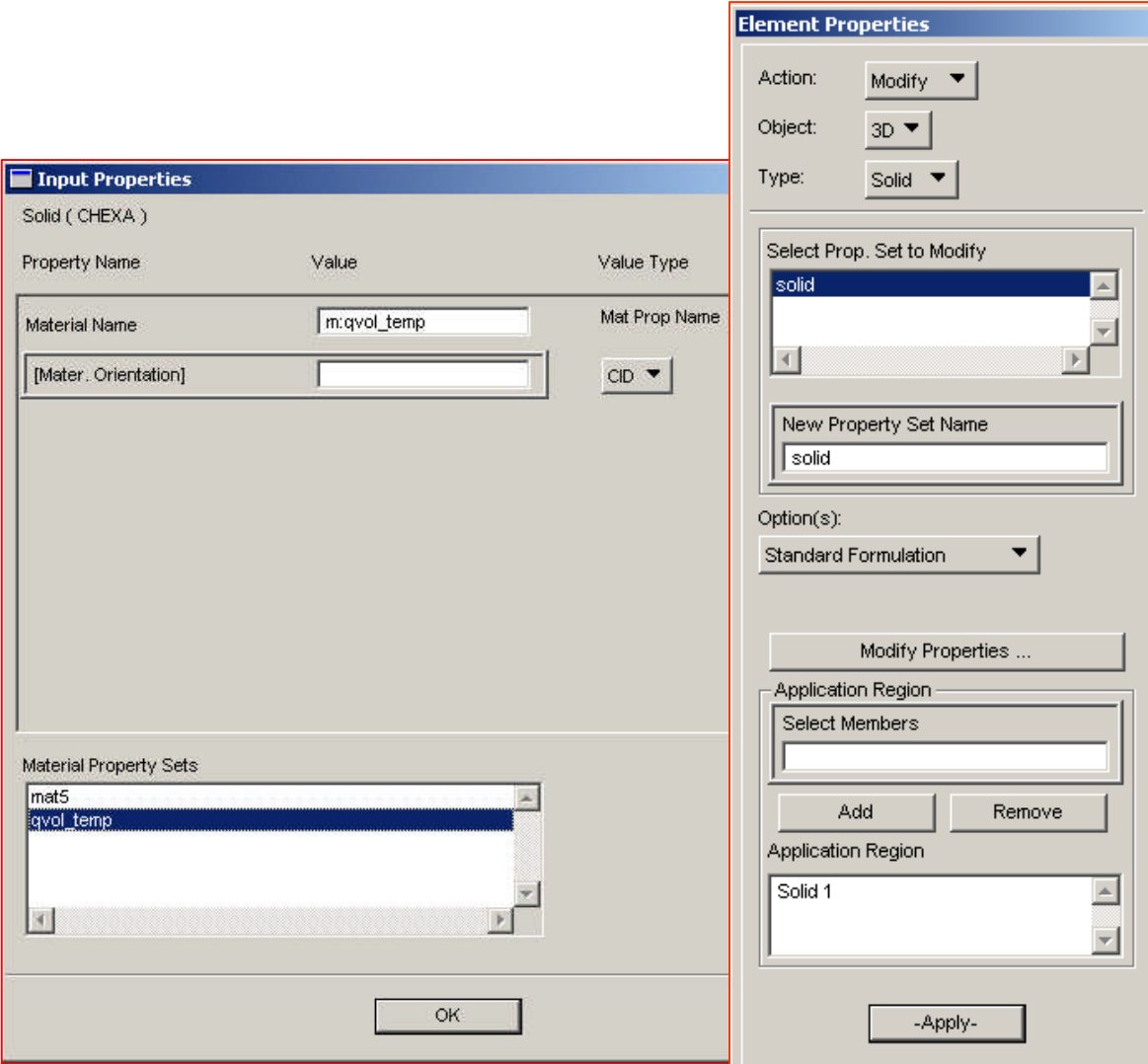
The screenshot shows the 'Input Options' dialog box with the following settings:

- Constitutive Model:** Heat generation
- Property Name:** Heat Generation Multiplier =
- Value:** qvol_temp
- Temperature Dep/Model Variable Fields:** qvol_temp
- Current Constitutive Models:** (Empty list)

Buttons at the bottom: OK, Clear, Cancel.

PROJECT 3, CREATE 3D ELEMENT PROPERTY

- Create properties for the 3D CHEXA elements.
- Use material property qvol_temp.



The image shows two overlapping dialog boxes from the MSC Software interface. The 'Input Properties' dialog is in the foreground, and the 'Element Properties' dialog is in the background.

Input Properties Dialog:

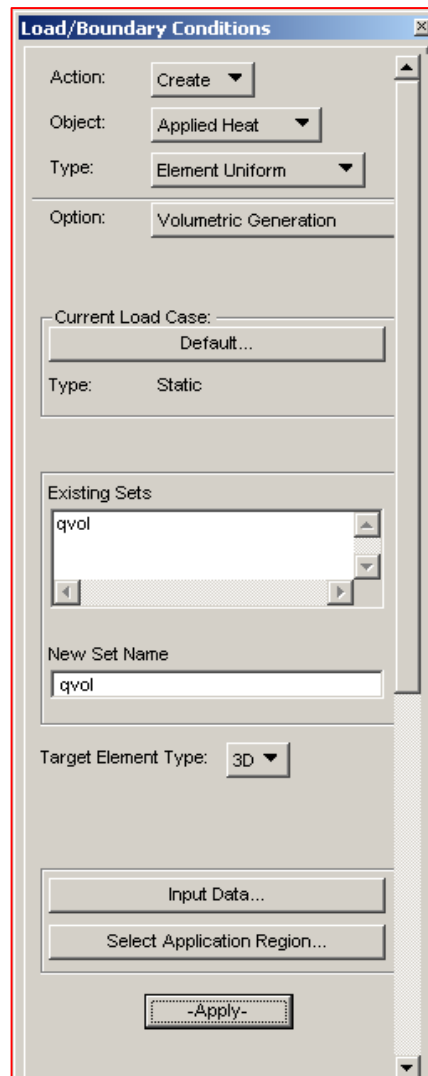
- Title: Solid (CHEXA)
- Property Name: Value Value Type
- Material Name: m:qvol_temp Mat Prop Name
- [Mater. Orientation] CID
- Material Property Sets: mat5, qvol_temp
- OK button

Element Properties Dialog:

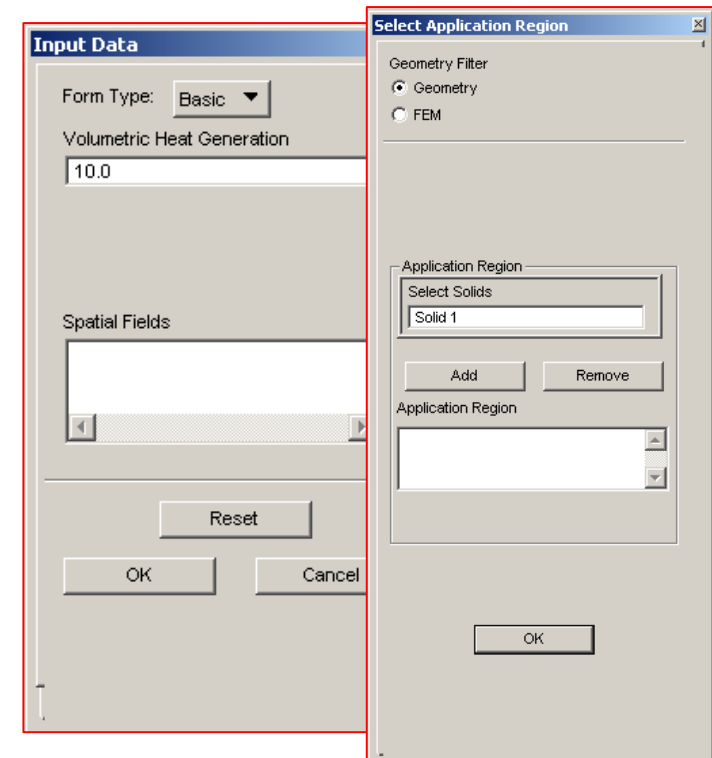
- Action: Modify
- Object: 3D
- Type: Solid
- Select Prop. Set to Modify: solid
- New Property Set Name: solid
- Option(s): Standard Formulation
- Modify Properties ... button
- Application Region: Select Members
- Add Remove buttons
- Application Region: Solid 1
- Apply- button

PROJECT 3, APPLY QVOL LOADING

- Create applied internal heating using Option: Volumetric Generation.
- $QVOL = 10.0$
- Apply the heating to the entire solid model, Solid 1.



Load/Boundary Conditions dialog box. Action: Create. Object: Applied Heat. Type: Element Uniform. Option: Volumetric Generation. Current Load Case: Default... Type: Static. Existing Sets: qvol. New Set Name: qvol. Target Element Type: 3D. Buttons: Input Data..., Select Application Region..., -Apply-.



Input Data dialog box: Form Type: Basic. Volumetric Heat Generation: 10.0. Spatial Fields: (empty). Buttons: Reset, OK, Cancel.

Select Application Region dialog box: Geometry Filter: Geometry (selected). Application Region: Select Solids: Solid 1. Buttons: Add, Remove, OK.

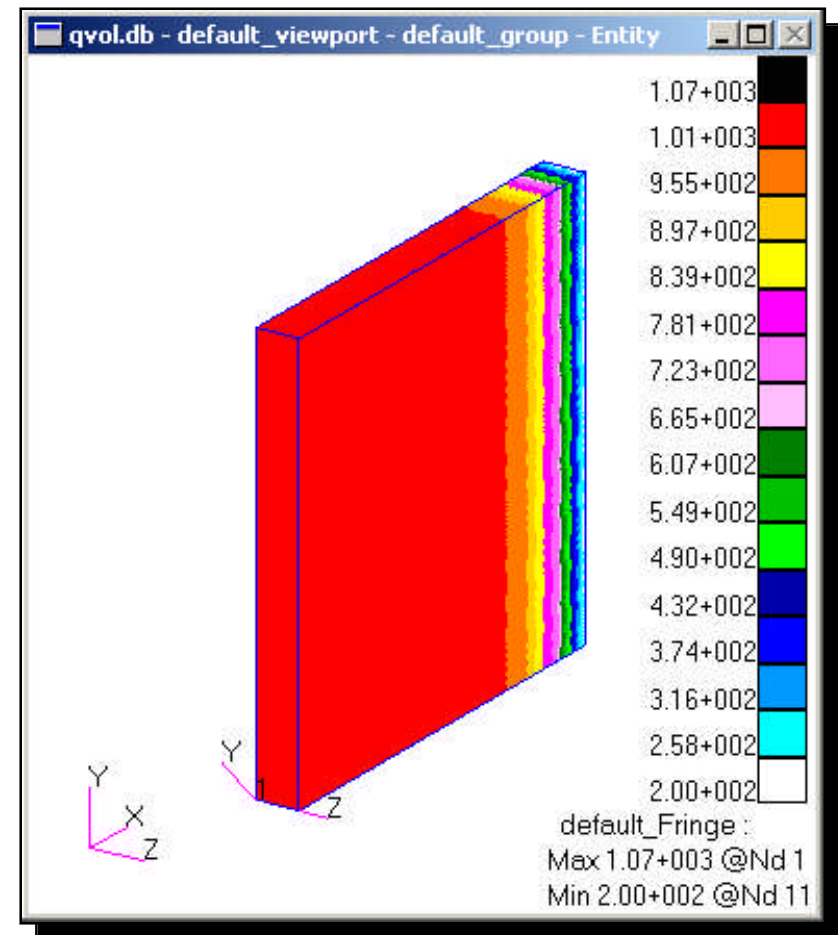
PROJECT 3, NASTRAN BULK DATA (QVOL.BDF)

Extract from MD Nastran model .DAT file

```
$MAT4, MID, K, CP, RHO, H, MU, HGEN
MAT4, 1, 204., , , , , 1.
$MATT4, MID, T(K), T(CP), , T(H), T(MU), T(HGEN)
MATT4, 1, , , , , , 1
$ Material Tables ( No Conversions Needed )
$ Temperature Dependent Material Table : qvol_temp
TABLEM1 1 + CX
+ CX 0. 5000. 100. 8000. 200. 12500. 300. 13500. + CY
+ CY 500. 10000. ENDT
QVOL 2 10. 1
QVOL 2 10. 2
```

PROJECT 3, TEMPERATURE CONTOUR

- Total heat is
 $Q_{IN} = \text{volume}_{ELM} * HGEN(\text{temp}) * QVOL$
 - Create field to be used for HGEN(temp)
 - Create material property
 - Constant K
 - Heat generation, HGEN(temperature)
 - Modify 3D element property, using material property.
 - Create volumetric generation using QVOL = 10.0



EXERCISES

- Perform Workshop 2 “Circuit Board and Chips Using Conduction and Heating” in the exercise workbook.

SECTION 4

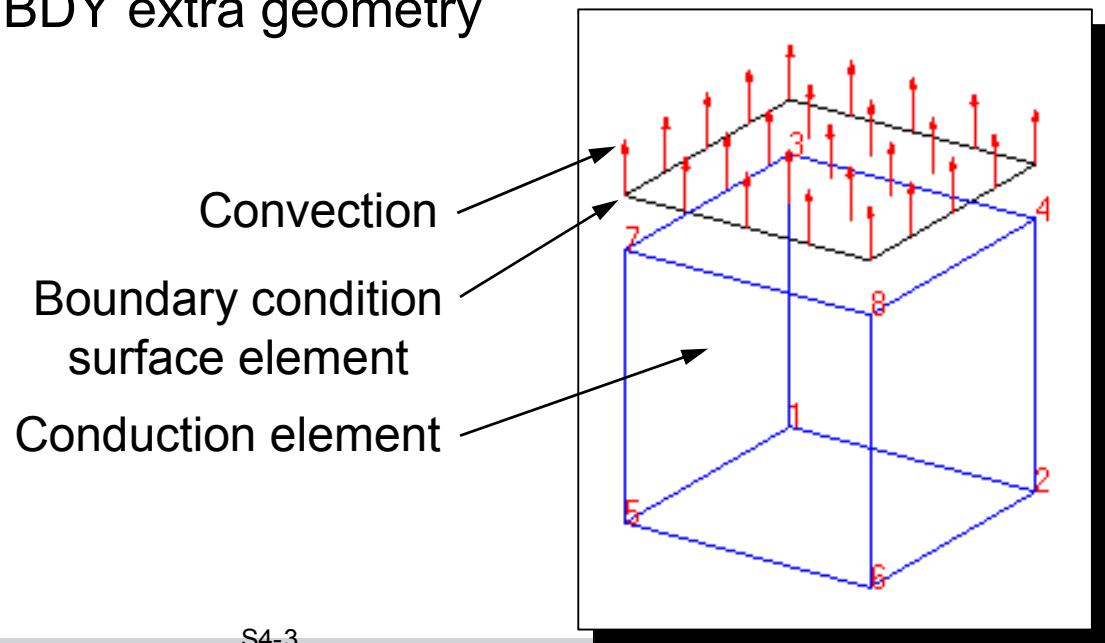
CONVECTION





HEAT THROUGH SURFACE ELEMENT

- Convection to a face of a conduction element (e.g. CHEXA) must be through a boundary condition surface element
- Boundary condition surface element
 - CHBDYE – reference to heat conduction element
 - CHBDYG – uses grid point IDs of conduction element
 - CHBDYP – uses PHBDY extra geometry



BC SURFACE ELEMENTS

Element	CHBDYE	CHBDYG	CHBDYP
	ALL CONDUCT	REV	POINT
		AREA3	LINE
		AREA4	FTUBE
		AREA6	TUBE
		AREA8	ELCYL

BOUNDARY CONDITION ELEMENT CHBDYE

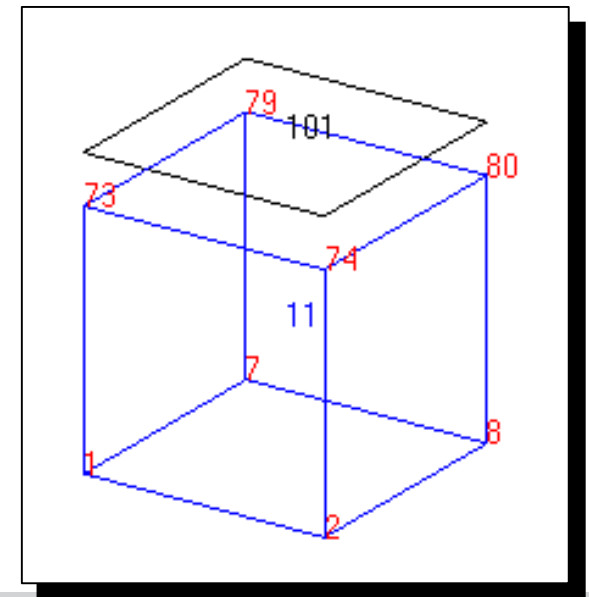
- Defines a boundary condition surface element with reference to a heat conduction element.

1	2	3	4	5	6	7	8	9	10
CHBDYE	EID	EID2	SIDE	IVIEWF	IVIEWB	RADMID F	RADMID B		
CHBDYE	101	1	1						

- EID – unique CHBDYE ID
- EID2 – a heat conduction element ID
- SIDE – a conduction element side ID
- IVIEW – to participate in view factor exchange only
- RADMID – defines emissivity through RADM entry

CHEXA	11	1	1	2	8	7	73	74
	80	79						
CHBDYE, 101, 11, 6								

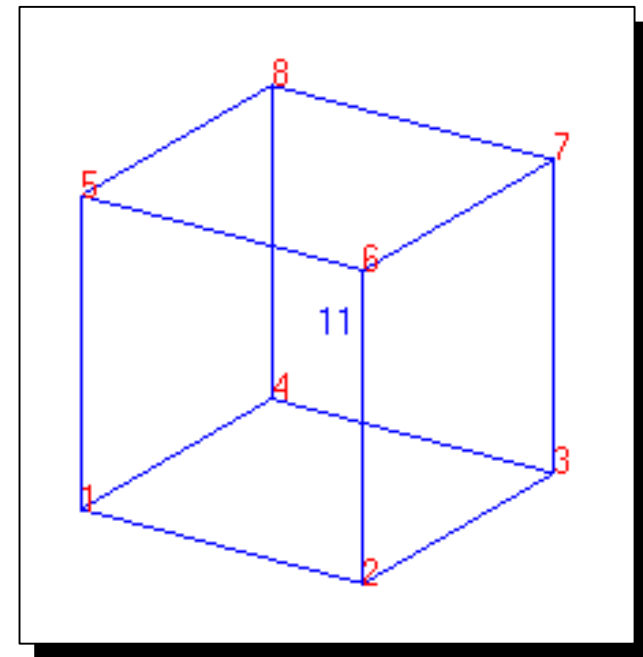
.bdf file extract



CHBDYE, SIDE CONVECTION FOR SOLID ELEMENTS

CHEXA	11	1	1	2	3	4	5	6
	7	8						

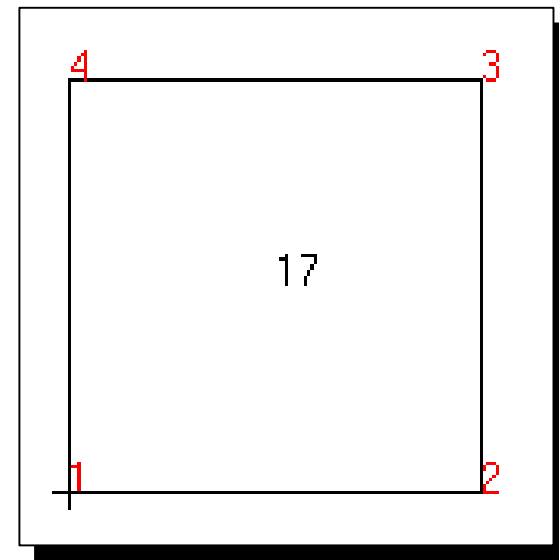
Side	Grid	Grid	Grid	Grid
1	4	3	2	1
2	1	2	6	5
3	2	3	7	6
4	3	4	8	7
5	4	1	5	8
6	5	6	7	8



CHBDYE, SIDE CONVECTION FOR SHELL ELEMENTS

```
CQUAD4    17      1      1      2      3      4
CHBDYE, 201, 17, 1
```

- Side 1 of shell element (top), the first side is the one given by the right hand rule for the grid point connectivities.
- Side 2 through 5 is the line type. The second side is for the edge connecting Grid 1 and Grid 2. The remaining sides are numbered consecutively.



FREE CONVECTION CONV

- Specifies a free convection boundary condition through connection to a boundary condition surface element, CHBDYi entry.

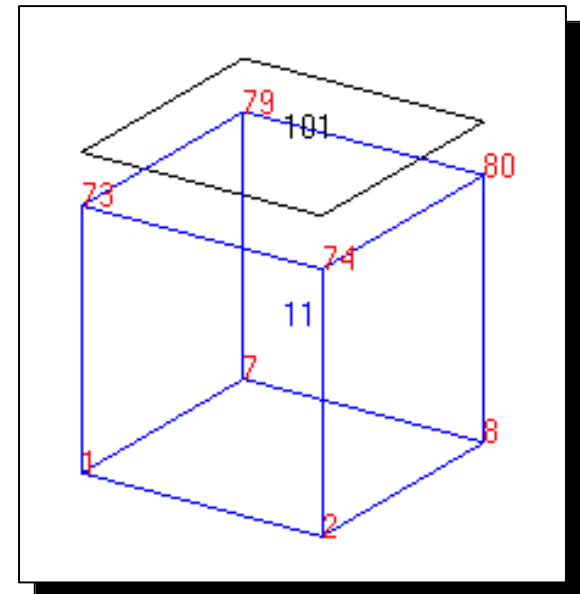
1	2	3	4	5	6	7	8	9	10
CONV	EID	PCONID	FLMND	CNTRLND	TA1	TA2	TA3	TA4	
CONV	101	5			777				

- EID – CHBDYi ID
- PCONID – convection property of PCONV
- CNTRLND – usually blank, unless one wants to define convection as a function of time
- TAi – ambient points used for convection

```

CHEXA      11      1      1      2      8      7      73      74
           80      79
CHBDYE,101,11,6
CONV,101,5,,,777

SPOINT,777
SPC,1,777,1,25.0
TEMP,5,777,25.0
    
```



FREE CONVECTION PCONV

- Specifies a free convection boundary condition PR through connection to a surface element (CHBDYi entry).

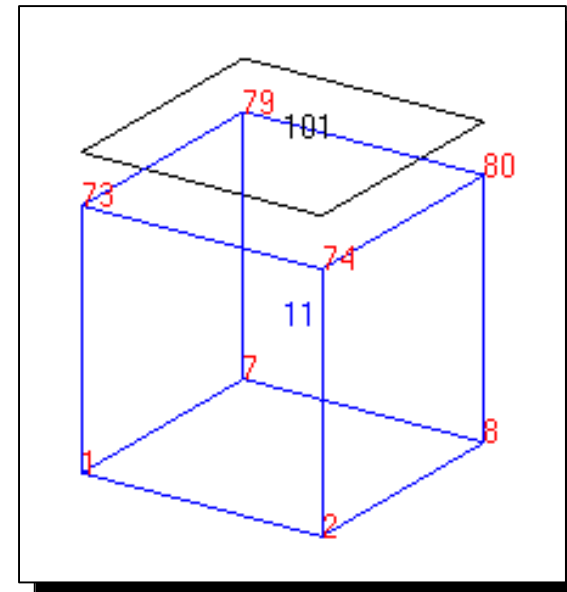
1	2	3	4	5	6	7	8	9	10
PCONV	PCONID	MID	FORM	EXPF					
PCONV	5	9	0	0.0					

- PCONID – convection property ID
- MID – material ID (MAT4 or MAT5)
- FORM – type of formula used for free convection
- EXPF – free convection exponent
- $Q = h \cdot u_{\text{cntrlnd}} \cdot (T - T_{\text{amb}})^{\text{expf}} \cdot (T - T_{\text{amb}})$. If you want to define $Q = hA \cdot (T - T_{\text{amb}})$ set Form = 0 and EXPF = 0.0, where h is defined on the MAT4 entry

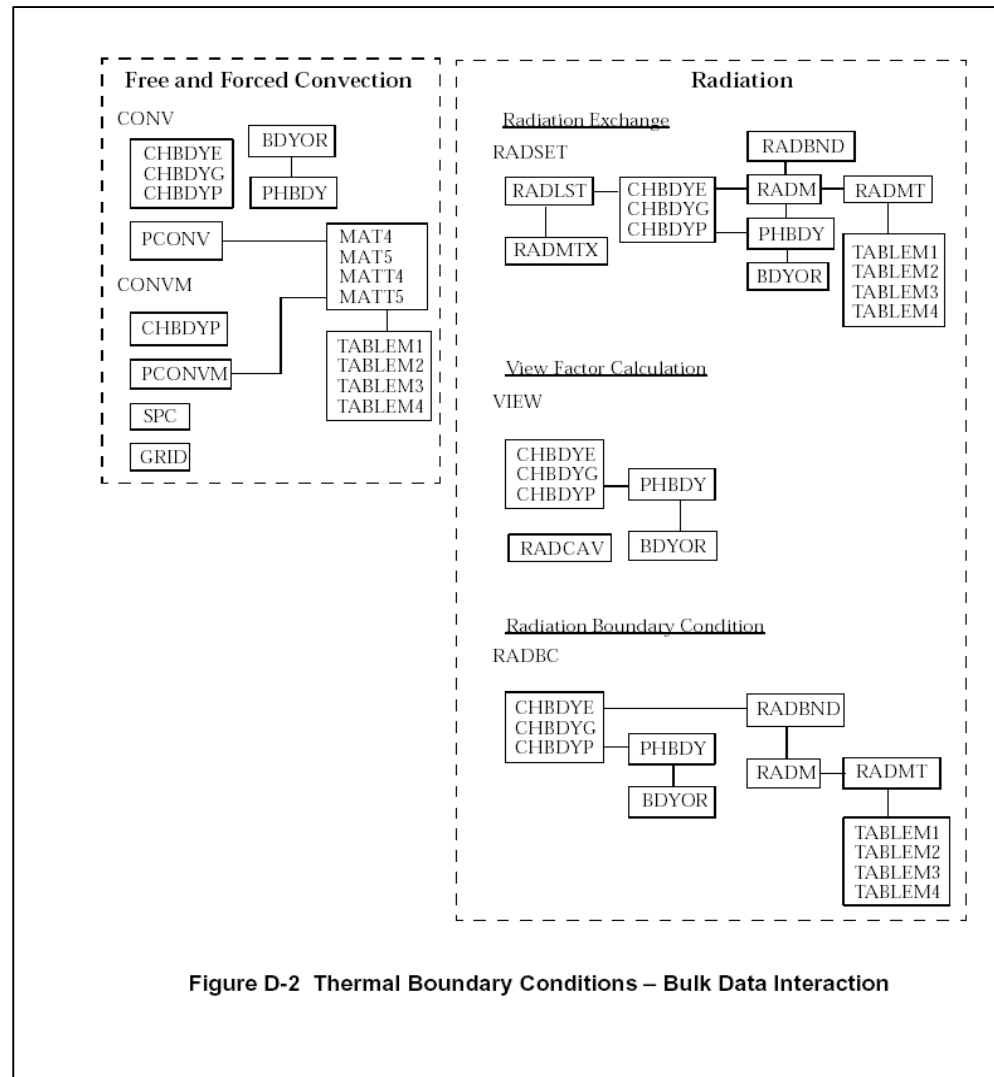
```

CHEXA    11      1      1      2      8      7      73      74
          80      79
CHBDYE,101,11,6
CONV,101,5,,,777
PCONV,5,333,0,0.0
MAT4,333,,,,,0.234
$The 6th field on the MAT4 card defines the free convection
coefficient.
SPOINT,777
SPC,1,777,1,25.0
TEMP,5,777,25.0
    
```

.bdf file extract



SURFACE BOUNDARY CONDITIONS

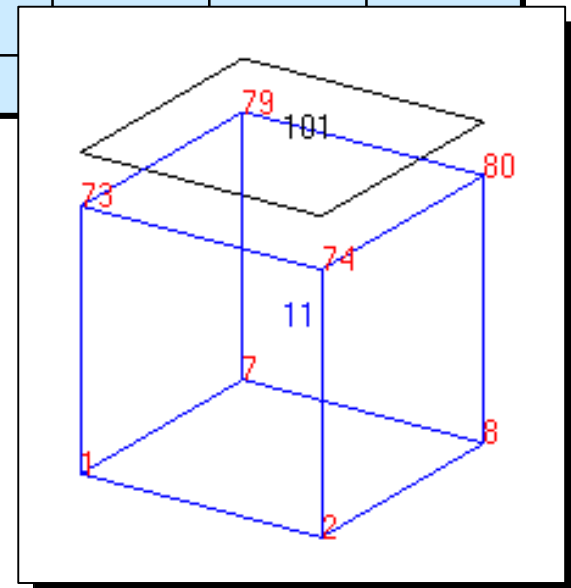


CHBDYG

- Defines a boundary condition surface element without reference to a property entry.

1	2	3	4	5	6	7	8	9	10
CHBDYG	EID		TYPE	IVIEWF	IVIEWB	RADMID F	RADMID B		
	G1	G2	G3	G4	G5	G6	G7	G8	

1	2	3	4	5	6	7	8	9	10
CHBDYG	101		AREA4						
	73	74	80	79					



CHBDYG

CHBDYG

Geometric Surface Element Definition (Grid Form)

Defines a boundary condition surface element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYG	EID		TYPE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		
	G1	G2	G3	G4	G5	G6	G7	G8	

Example:

CHBDYG	2		AREA4	3	3	2	2		
	100	103	102	101					

Field	Contents
EID	Surface element identification number. (Unique (0 < Integer < 100,000,000) among all elemental entries)
TYPE	Surface type. See Remark 3. (Character)
IVIEWF	A VIEW entry identification number for the front face. See Remark 2. for default. (Integer > 0)
IVIEWB	A VIEW entry identification number for the back face. See Remark 2. for default. (Integer > 0)
RADMIDF	RADM identification number for front face of surface element. See Remark 2. for default. (Integer > 0)
RADMIDB	RADM identification number for back face of surface element. See Remark 2. for default. (Integer ≥ 0)
Gi	Grid point IDs of grids bounding the surface. (Integer > 0)

CHBDYP

- Defines a boundary condition surface element with reference to PHBDY. In general, a CHBDYP element is a line type element.
- For example, one can define convection along the edge of rod element.

1	2	3	4	5	6	7	8	9	10
CHBDYP	EID	PID	TYPE	IVIEWF	IVIEWB	G1	G2	G0	
	RADMIDF	RADMIDB	GMID	CE	E1	E2	E3		

1	2	3	4	5	6	7	8	9	10
CHBDYP	201	1	LINE			11	12		
					0.0	1.0	0.0		

CHBDYP

CHBDYP

Geometric Surface Element Definition (Property Form)

Defines a boundary condition surface element with reference to a PHBDY entry.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYP	EID	PID	TYPE	IVIEWF	IVIEWB	G1	G2	G0	
	RADMIDF	RADMIDB	GMID	CE	E1	E2	E3		

Example:

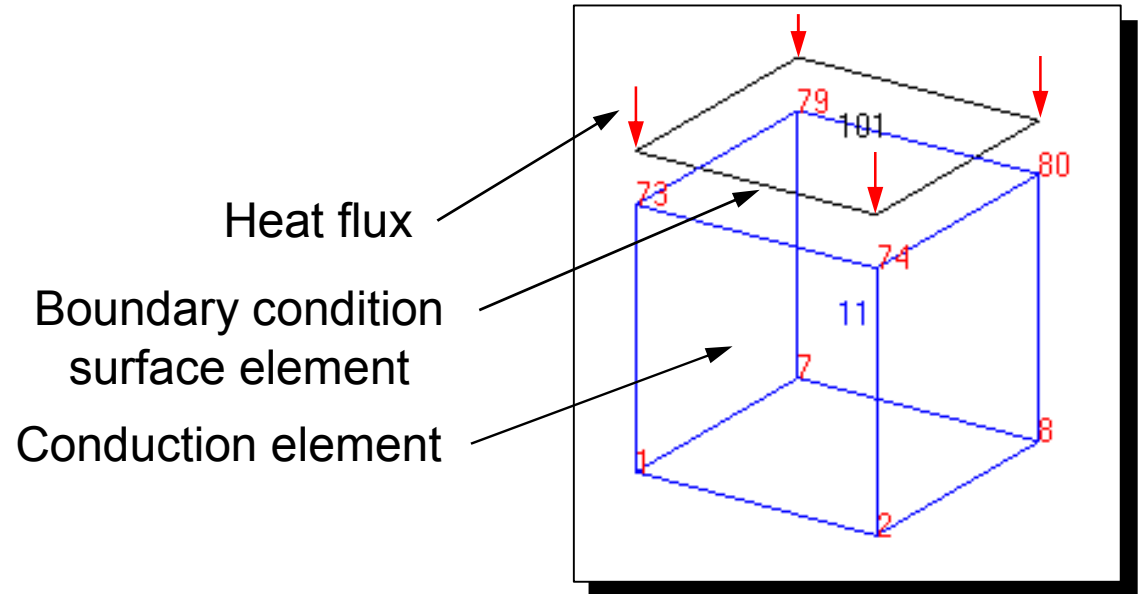
CHBDYP	2	5	POINT	2	2	101		500	
	3	3			0.0	0.0	1.0		

Field	Contents
EID	Surface element identification number. (Unique (0 < Integer < 100,000,000) among all element identification numbers.)
PID	PHBDY property entry identification numbers. (Integer > 0)
TYPE	Surface type. See Remark 3. (Character)
IVIEWF	VIEW entry identification number for the front face. (Integer > 0 or blank)
IVIEWB	VIEW entry identification number for the back face. (Integer > 0 or blank)
G1, G2	Grid point identification numbers of grids bounding the surface. (Integer > 0)
G0	Orientation grid point. (Integer ≥ 0; Default = 0)
RADMIDF	RADM entry identification number for front face. (Integer ≥ 0 or blank)
RADMIDB	RADM entry identification number for back face. (Integer ≥ 0 or blank)
GMID	Grid point identification number of a midside node if it is used with the line type surface element.
CE	Coordinate system for defining orientation vector. (Integer ≥ 0; Default = 0)
Ei	Components of the orientation vector in coordinate system CE. The origin of the orientation vector is grid point G1. (Real or blank)

QBDY3

- Defines a uniform heat flux load for a boundary surface.

1	2	3	4	5	6	7	8	9	10
QBDY3	SID	Q0	CNTRLND	EID1	EID2	EID3	EID4	EID5	
QBDY3	10	200.0		101					



QBDY3 (Cont.)

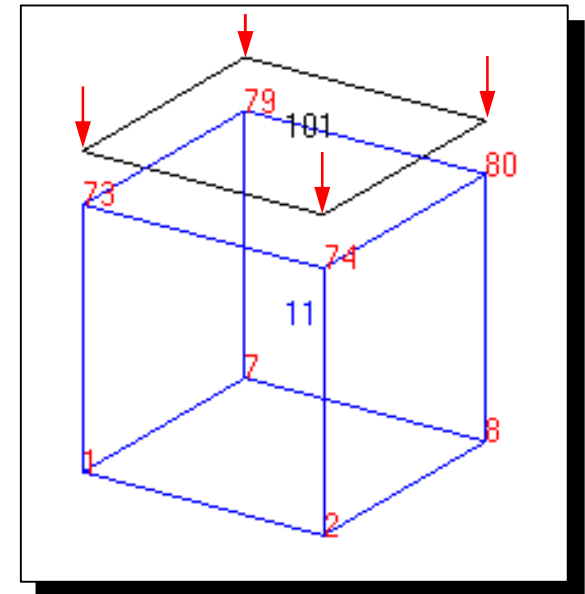
- Defines a uniform heat flux load for a boundary surface.

1	2	3	4	5	6	7	8	9	10
QBDY3	SID	Q0	CNTRLND	EID1	EID2	EID3	EID4	EID5	
QBDY3	10	200.0		101					

- SID – load set ID
- Q0 – thermal heat flux load (power per area)
- CNTRLND – control point for thermal flux load. In general it is not used.
- EIDj – CHBDYi element ID
- Pin = (effective area)*Q0, where the effective area is define on the CHBDYi entry

```

CHEXA      11      1      1      2      8      7      73      74
           80      79
CHBDYE,101,11,6
QBDY3,10,200.0,,101
$ We are applying a heat flux load of 200.0 to the surface that is
on side 6 of the conduction element 11. See the side definition for
the CHBDYE entry. In this case, this is the area bounded by Grid
73,74,80,79.
    
```



THERMAL LOADS

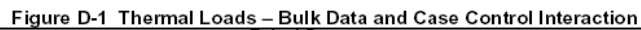
- QVECT – directional heat flux from distance source
- QVOL – volumetric heat load (power /volume)
- QHBDY – heat flux applied to an area defined by grid points (no CHBDYi entries required)
- QBDY1 – heat flux applied to boundary condition surface elements
- QBDY2 – heat flux applied to grid point
- QBDY3 – similar to QBDY1 except with control node
- SLOAD – apply power to a grid point

LOAD SELECTION

- $\text{LOAD} = \text{SID}$, where SID is an integer used in steady state analysis (SOL 153) to request application of the load Bulk Data labeled with the given SID. Only one LOAD command per subcase may be specified in the Case Control Section.
- $\text{DLOAD} = \text{SID}$ is used in transient analysis (SOL 159) to request the application of the dynamic load Bulk Data with the given SID. Only one DLOAD command per subcase may be specified in the Case Control Section.
- For steady state analysis, any number of loads defined in the Bulk Data may be referenced from a single Case Control request by specifying the same SID for all loads of interest.
- For transient analysis, the static load entries are not selected by the Case Control SID, rather they are referenced by the TLOADi entry EXCITEID. The SID required for Case Control selection is the TLOADi entry SID.
- The schematic for these processes is illustrated below.

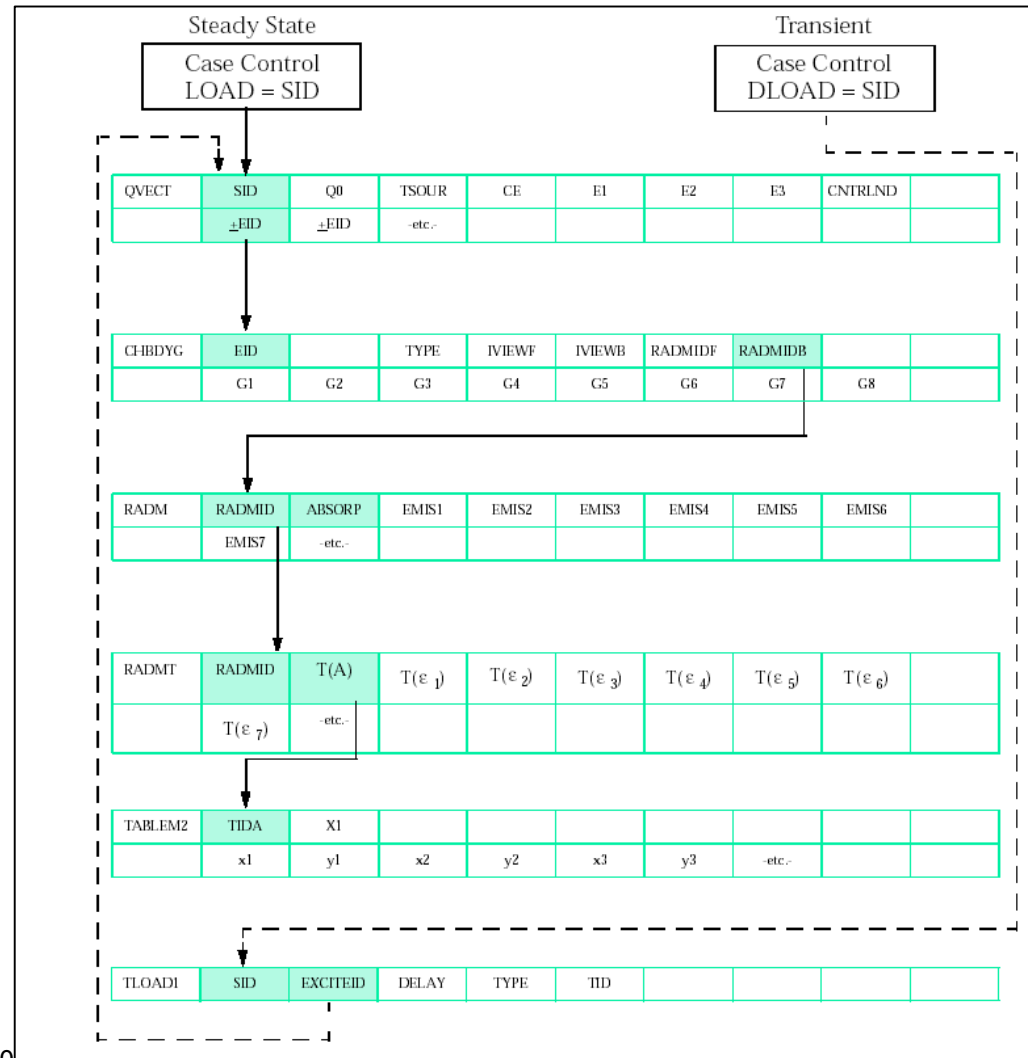


LOAD SET



Copyright© 2008 MSC.Software Corporation

SCHEMATIC FOR LOAD SELECTION

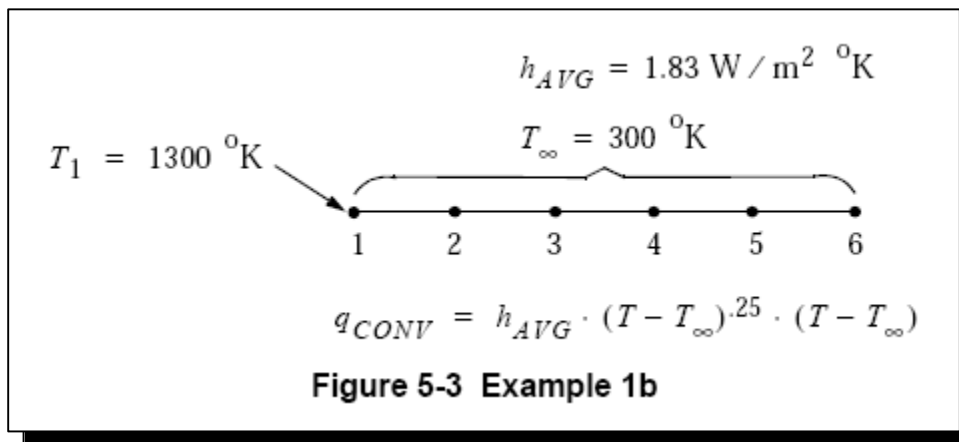


FREE CONVECTION





EXAMPLE 1B, NONLINEAR FREE CONVECTION ALONG THE EDGE OF A ROD



- C_p Specific heat
- ν Kinematic viscosity
- k Thermal conductivity
- Pr Prandtl number
- Gr Grashof number
- Nu Nusselt number
- T_w Wall temperature
- T_{∞} Ambient temperature

EXAMPLE 1B, DERIVATION

d Diameter

β Volume coefficient of expansion
(For an ideal gas $\beta = (1 / T)$ where T is the absolute temperature of the gas)

g Acceleration due to gravity

Assume air properties at 800°K.

$$C_p = 1.098 \text{ KJ} / \text{Kg } ^\circ\text{K}$$

$$\nu = .823 \times 10^{-4} \text{ m}^2 / \text{s}$$

$$k = .058 \text{ W} / \text{m } ^\circ\text{K}$$

$$Pr = .689$$

$$\begin{aligned} Gr &= \frac{g\beta d^3(T_w - T_\infty)}{\nu^2} \\ &= \frac{9.80 \frac{\text{m}}{\text{s}^2} \cdot \frac{1}{800 \text{ } ^\circ\text{K}} \cdot (.10)^3 \text{ m}^3 \cdot 1000 \text{ } ^\circ\text{K}}{(.823 \times 10^{-4})^2 \frac{\text{m}^4}{\text{s}^2}} \\ &= 1.8 \times 10^6 \end{aligned}$$

$$Gr \cdot Pr = 1.25 \times 10^6$$

EXAMPLE 1B, DERIVATION (Cont.)

for

$$10^4 \leq Gr \cdot Pr \leq 10^9 \quad \text{and for horizontal cylinders,}$$

$$\left. \begin{array}{l} \text{COEF} = .53 \\ m = .25 \end{array} \right\} \quad (\text{see J. P. Holman, Heat Transfer})$$

so

$$Nu = \frac{hd}{k} = \text{COEF} \left(\frac{g\beta d^3}{\nu^2} \cdot Pr \cdot (T - T_\infty) \right)^m$$

or

$$h = \frac{.53 (.058)}{.10} \left\{ \frac{9.80 (1/800) (.10)^3 (.689)}{(.823 \times 10^{-4})^2} \right\}^{.25} (T - T_\infty)^{.25}$$

Note: The equation for h is nonlinear.

therefore

$$h \cong 1.83 (T - T_\infty)^{.25} \text{ W/m}^2 \text{ } ^\circ\text{K}$$

EXAMPLE 1B, NONLINEAR FREE CONVECTION INPUT FILE

```
ID MSC, EX1B  $
$ V68 D_CHU CC327 18-NOV-1993
$ID MSC-NASTRAN V68
SOL 153
TIME 10
CEND
TITLE = EXAMPLE 1b
ANALYSIS = HEAT
THERMAL = ALL
FLUX = ALL
SPCF = ALL
OLOAD = ALL
SPC = 10
TEMP(INIT) = 20
NLPARM = 100
BEGIN BULK
NLPARM,100
$
GRID,1,,0.0,0.0,0.0
GRID,2,,0.1,0.0,0.0
GRID,3,,0.2,0.0,0.0
GRID,4,,0.3,0.0,0.0
GRID,5,,0.4,0.0,0.0
GRID,6,,0.5,0.0,0.0
GRID,99,,99.0,99.0,99.0
```

```
$
CROD,1,5,1,2
CROD,2,5,2,3
CROD,3,5,3,4
CROD,4,5,4,5
CROD,5,5,5,6
PROD,5,15,.0078540
MAT4,15,204.0,,1.83
$
CHBDYP,10,25,LINE,,,1,2,,+CHP10
+CHP10,,,,,0.0,1.0,0.0
CHBDYP,20,25,LINE,,,2,3,,+CHP20
+CHP20,,,,,0.0,1.0,0.0
CHBDYP,30,25,LINE,,,3,4,,+CHP30
+CHP30,,,,,0.0,1.0,0.0
CHBDYP,40,25,LINE,,,4,5,,+CHP40
+CHP40,,,,,0.0,1.0,0.0
CHBDYP,50,25,LINE,,,5,6,,+CHP50
+CHP50,,,,,0.0,1.0,0.0
```


EXAMPLE 1B, NONLINEAR FREE CONVECTION INPUT FILE (Cont.)

```
PHBDY,25,.3141593
$
CONV,10,35,,,99,99
CONV,20,35,,,99,99
CONV,30,35,,,99,99
CONV,40,35,,,99,99
CONV,50,35,,,99,99
PCONV,35,15,0,0.25
$
SPC,10,1,,1300.0
SPC,10,99,,300.0
TEMPD,20,1300.0
$
ENDDATA
```

- This is Example 1b from the MD Nastran 2002 Thermal Analysis User's Guide.
- The effective width for the convection area is equal to $\text{Pi} \times \text{Diam} = 3.14159 \times 0.1 = 0.3141593$

EXAMPLE 1B, OUTPUT

NON-LINEAR ITERATION MODULE OUTPUT

STIFFNESS UPDATE TIME 0.02 SECONDS
ITERATION TIME 0.00 SECONDS

SUBCASE 1
LOAD FACTOR 1.0000000

--- CONVERGENCE FACTORS ---

--- LINE SEARCH DATA ---

ITERATION	EUI	EPI	EWI	LAMBDA	DLMAG	FACTOR	E-FIRST	E-FINAL	NQNV	NLS	ENIC	NDV	MDV
1	1.1262E-13	9.7786E-02	1.1007E-16	1.0000E-01	2.9097E+02	1.0000E+00	0.0000E+00	0.0000E+00	0	0		0	1
2	1.6730E+01	1.9848E-02	3.6841E-03	1.5149E-01	6.5130E+01	1.0000E+00	-2.4629E-01	-2.4629E-01	0	0		0	1
3	2.7737E-02	1.3951E-04	3.4216E-06	7.9258E-02	4.3181E-01	1.0000E+00	4.7255E-03	4.7255E-03	1	0	0	0	1
4	2.9687E-05	3.3073E-06	4.8656E-10	5.1482E-02	1.0361E-02	1.0000E+00	-6.0189E-03	-6.0189E-03	2	0	-1	0	1

*** USER INFORMATION MESSAGE 6186 (NCONVG)

*** SOLUTION HAS CONVERGED ***

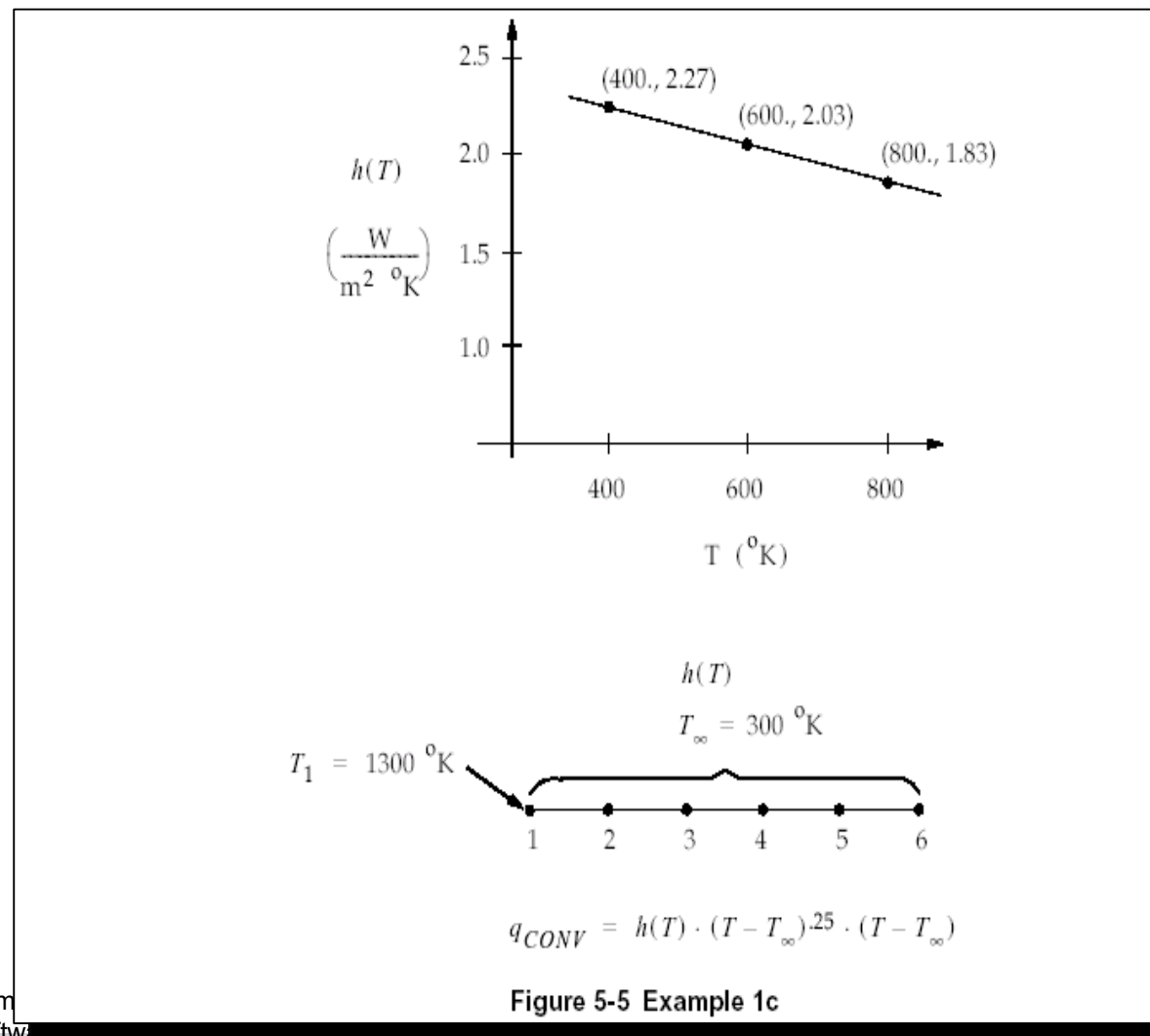
EXAMPLE 1B, OUTPUT (Cont.)

```
0                                SPCFORCE RESULTANT
SUBCASE/
DAREA ID      T1      T2      T3      R1      R2      R3
0      1  -1.9271113E-02
1  EXAMPLE 1B      EX1B      JANUARY 26, 2004 MD NASTRAN 8/22/03 PAGE  9

0
LOAD STEP = 1.00000E+00
                                TEMPERATURE VECTOR

POINT ID.  TYPE  ID  VALUE  ID+1 VALUE  ID+2 VALUE  ID+3 VALUE  ID+4 VALUE  ID+5 VALUE
      1    S    1  1.300000E+03  1.225588E+03  1.169515E+03  1.130402E+03  1.107303E+03  1.099665E+03
     99    S    3  3.000000E+02
```

EXAMPLE 1C, CONVECTION AS A FUNCTION OF TEMPERATURE $h(T)$



EXAMPLE 1C, INPUT FILE

```
MAT4,15,204.0,,,1.0
MATT4,15,,,40
TABLEM2,40,0.0,,,,,,+TBM
+TBM,400.0,2.27,600.0,2.03,800.0,1.83,ENDT
$
CHBDYP,10,25,LINE,,,1,2,,+CHP10
+CHP10,,,,,0.0,1.0,0.0
CHBDYP,20,25,LINE,,,2,3,,+CHP20
+CHP20,,,,,0.0,1.0,0.0
CHBDYP,30,25,LINE,,,3,4,,+CHP30
+CHP30,,,,,0.0,1.0,0.0
CHBDYP,40,25,LINE,,,4,5,,+CHP40
+CHP40,,,,,0.0,1.0,0.0
CHBDYP,50,25,LINE,,,5,6,,+CHP50
+CHP50,,,,,0.0,1.0,0.0
PHBDY,25,.3141593
$
CONV,10,35,,,99,99
CONV,20,35,,,99,99
CONV,30,35,,,99,99
CONV,40,35,,,99,99
CONV,50,35,,,99,99
PCONV,35,15,0,0.25
$
SPC,10,1,,1300.0
SPC,10,99,,300.0
TEMPD,20,1300.0
$
ENDDATA
```



EXAMPLE 1C, OUTPUT

0

SPCFORCE RESULTANT

SUBCASE/

DAREA ID

T1

T2

T3

R1

R2

R3

0 1 -1.6784741E-02

1 EXAMPLE 1C

EX1C

JANUARY 26, 2004 MD NASTRAN 8/22/03

PAGE 9

0

LOAD STEP = 1.00000E+00

TEMPERATURE VECTOR

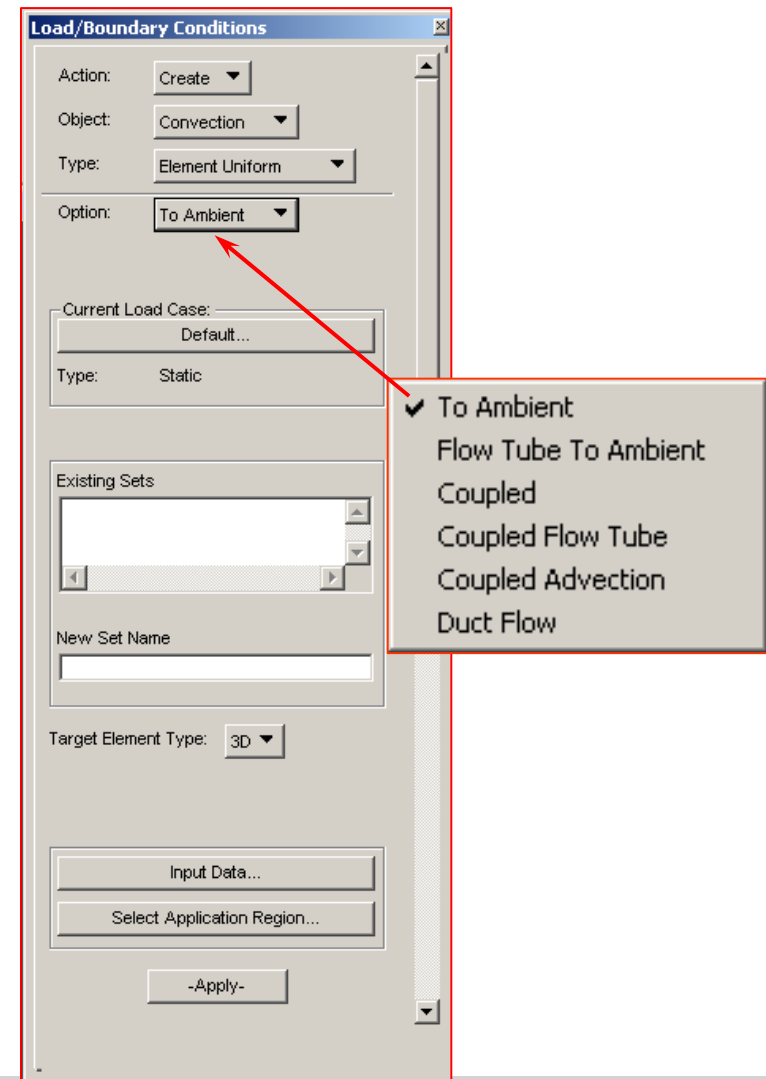
POINT ID. TYPE ID VALUE ID+1 VALUE ID+2 VALUE ID+3 VALUE ID+4 VALUE ID+5 VALUE

1 S 1.300000E+03 1.223213E+03 1.165064E+03 1.124369E+03 1.100287E+03 1.092315E+03

99 S 3.000000E+02

FREE CONVECTION VIA MD PATRAN PREFERENCE

- Free Convection, Loads/BCs: Create/Convection, Option
 - To Ambient – heat transfer between surface of a body and surrounding media at known temperature
- Coupled
 - Similar to To Ambient, except there are two application regions
 - Target Element Type (Application Region 1): 1D, 2D, 3D
 - Region 2 (Application Region 2): Nodal, 1D, 2D, 3D
 - The nodal temperatures for Region 2, as well as Region 1, can be determined by the MD Nastran solver



FREE CONVECTION VIA MD PATRAN

PREFERENCE (Cont.)

- There are two formula types, and if the convection coefficient is a function of temperature then you can select the method for the film temperature. The default is Average Temp.

The screenshot shows the 'Input Data' dialog box with the following settings:

- Form Type:** Advanced
- Convection Coefficient:** f:Field_H_spatial
- Ambient Temperature:** 300
- Formula Type Option:**
 - ☒ $q = h(T_s - T_a) \cdot \text{EXPF} \cdot (T_s - T_a)$
 - ☐ $q = h(T_s \cdot \text{EXPF} - T_a \cdot \text{EXPF})$
- Convection Exponent (EXPF):** (empty field)
- Reference Temperature Option:**
 - ☒ Average Temp $(T_s + T_a)/2$
 - ☐ Surface Temp (T_s)
 - ☐ Ambient Temp (T_a)
 - ☐ Film Temperature
- * Temperature Function:** f:Field_H_temp_dep
- * Time Function:** f:Field_H_time_dep
- * Time Function:** f:Field_Tamb_time_dep
- Spatial Fields:** Field_H_spatial
- Temperature Dependent Fields:** Field_H_temp_dep
- Time Dependent Fields:** Field_H_time_dep, Field_Tamb_time_dep

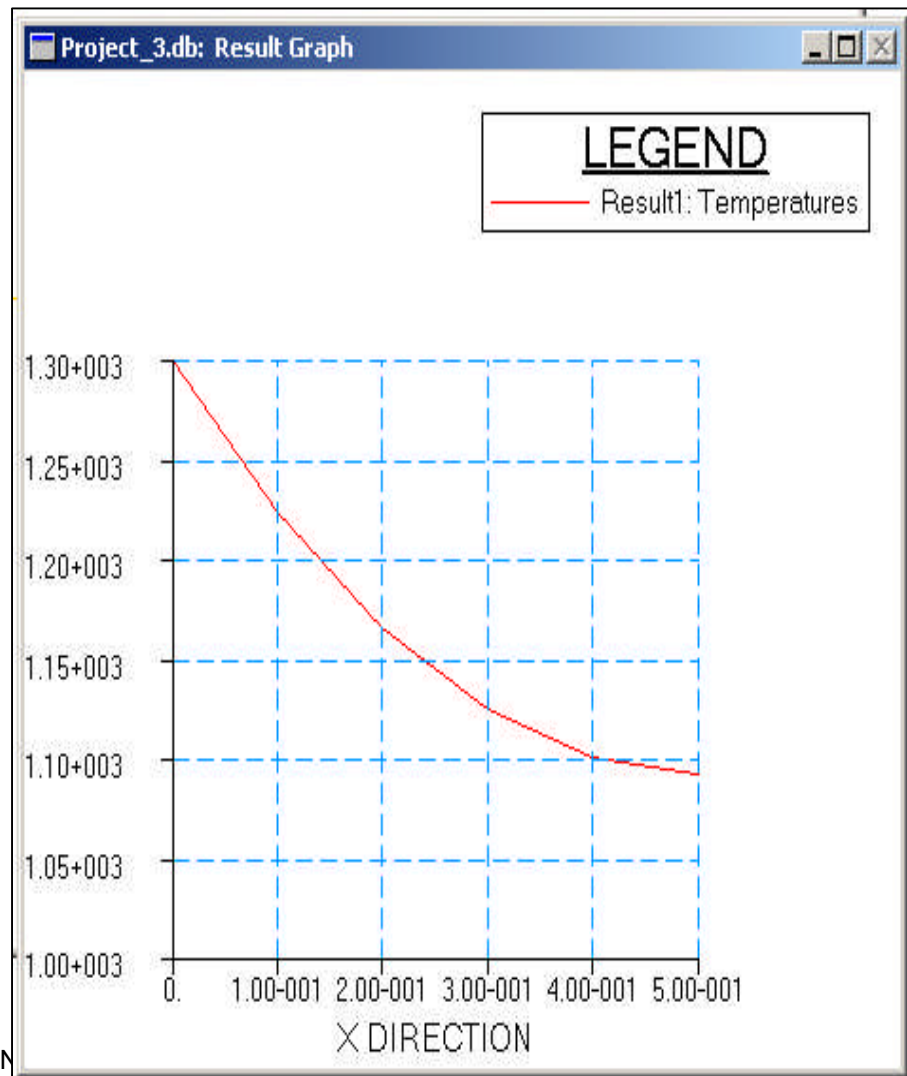
Buttons at the bottom: OK, Reset, Cancel.

FREE CONVECTION, VARIABLE H AND TAMB

- Variable convection coefficient and ambient temperature
 - The user may specify H as a function of temperature and/or time.
 - The user may specify Tamb as a function of time.
- Use Fields to specify the variation
 - Fields: Create/Spatial or Non Spatial
 - Spatial is (X,Y,Z) dependent
 - Non Spatial is time or temperature dependent



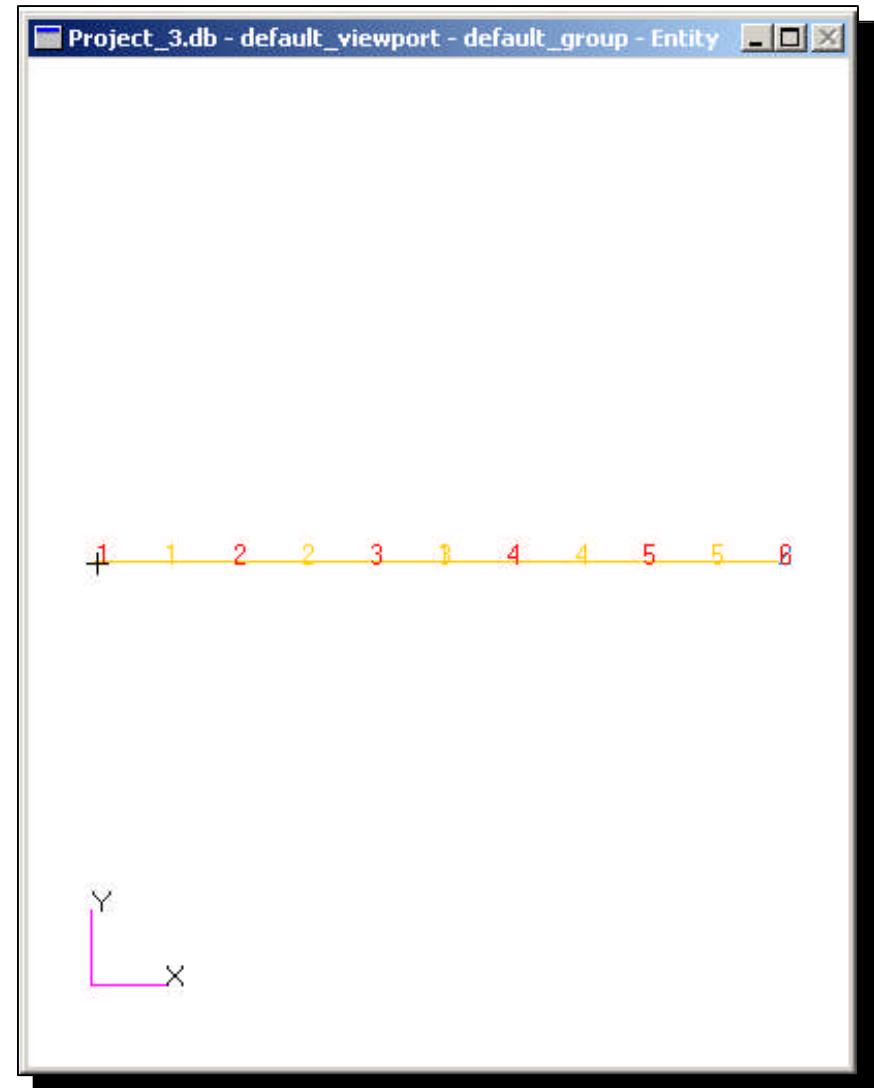
PROJECT 4, FREE CONVECTION, $H=H(TEMP)$



- Based on Exercise 1c in the MSC Nastran Thermal Analysis User's Guide.
- One end of the aluminum rod is held at 1300.0 °K.
- The rod loses heat to a fluid fixed at 300.0 °K.
- Perform a steady-state analysis and review the .DAT and .f06 files.

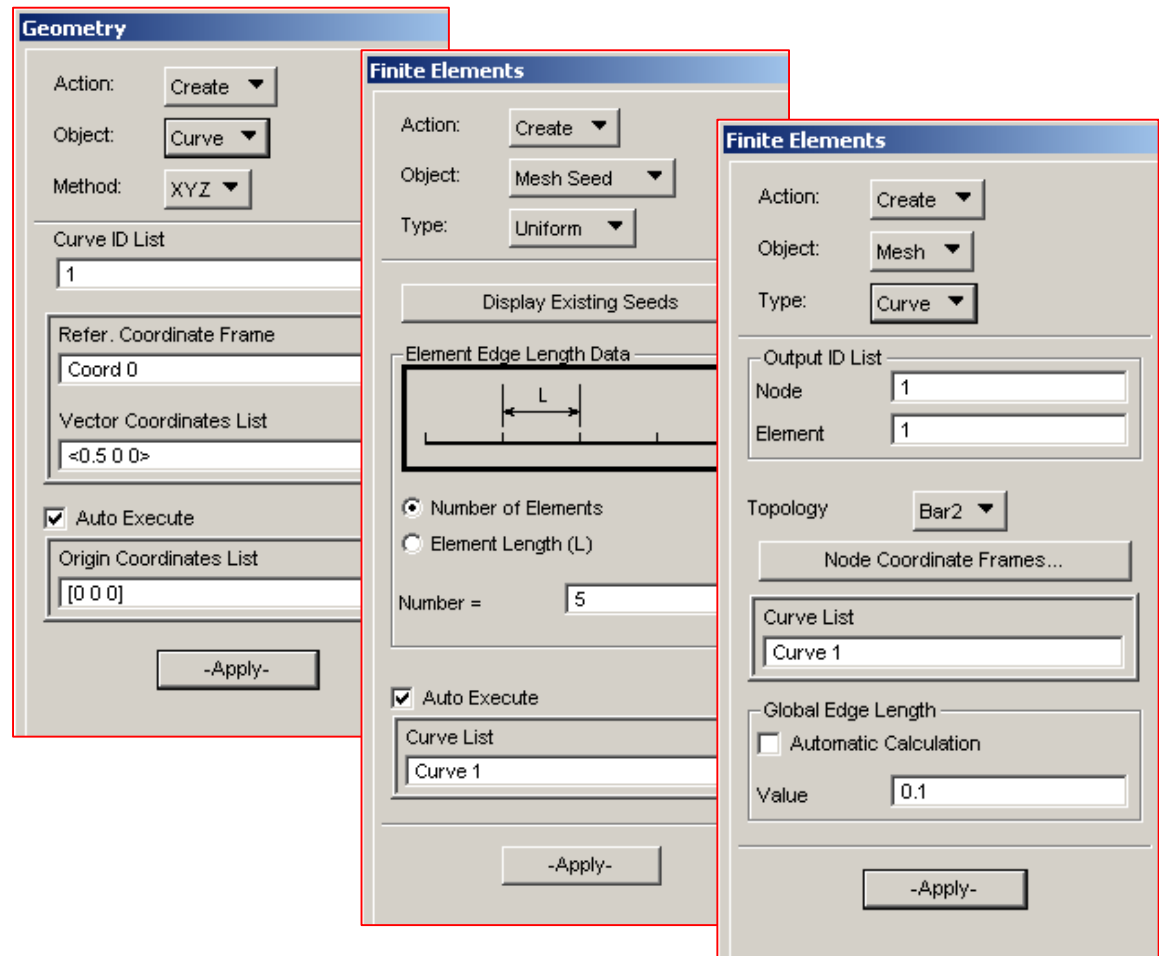
PROJECT 4, FREE CONVECTION, $H=H(TEMP)$

- In the MD Nastran 2002 Thermal Analysis User's Guide, the diameter and length of the aluminum rod were specified as 0.1m and 0.5m, respectively.



PROJECT 4, FREE CONVECTION, $H=H(TEMP)$

- Create the finite element mesh.
 - Create a curve 0.5 units long.
 - Create 5 element mesh seeds on the curve.
 - Mesh the curve with the IsoMesh.
 - The value of the Global Edge Length does not affect the meshing.



PROJECT 4, FREE CONVECTION, $H=H(TEMP)$

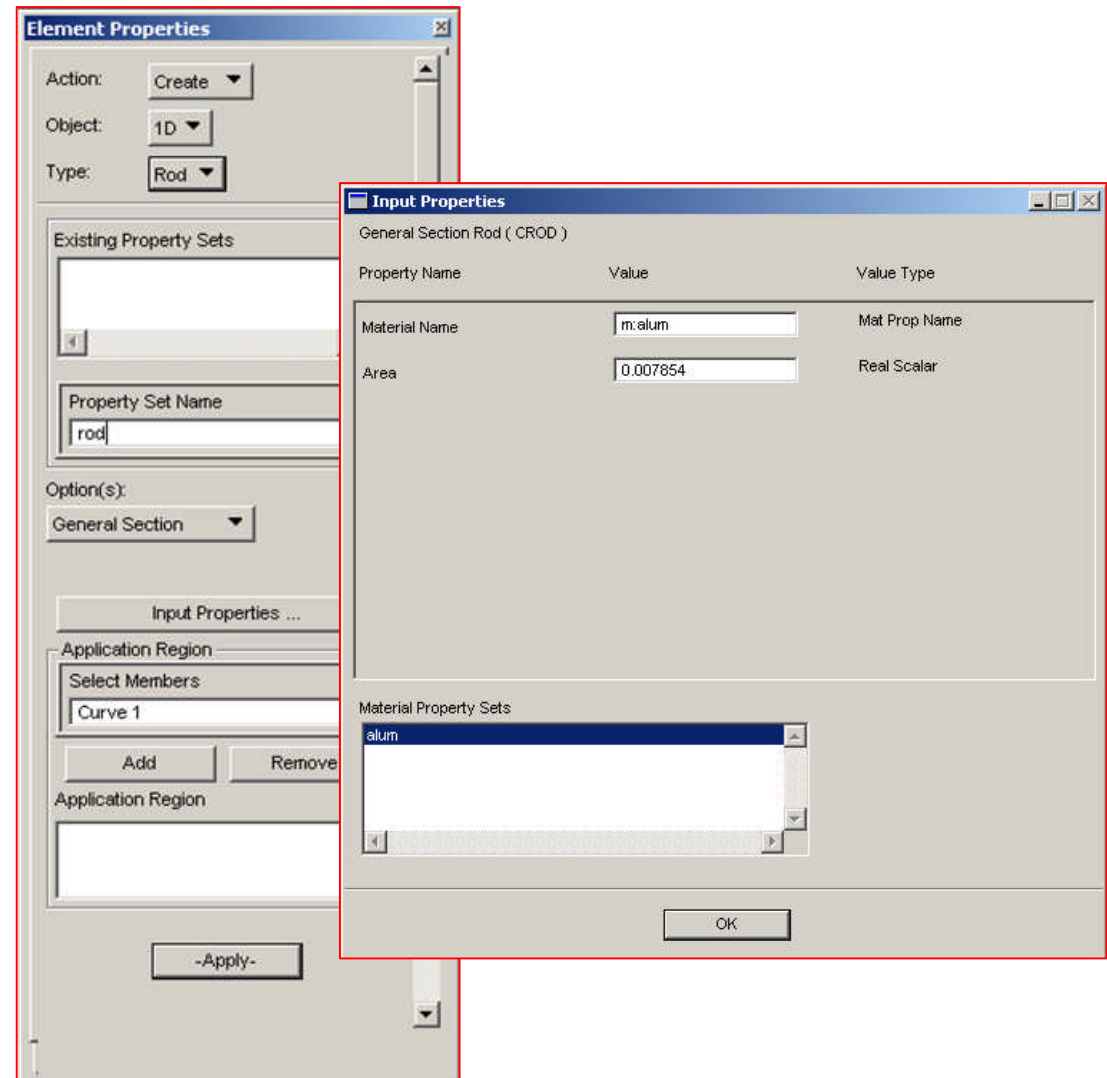
- Create the material properties.
 - Constant thermal conductivity.

The image shows two overlapping dialog boxes from a software application. The 'Materials' dialog box on the left has a title bar 'Materials' and contains the following fields: 'Action:' with a dropdown set to 'Create', 'Object:' with a dropdown set to 'Isotropic', and 'Method:' with a dropdown set to 'Manual Input'. Below these is a search field with an asterisk and a 'Filter' button. An 'Existing Materials' list is empty. The 'Material Name' field contains 'alum'. The 'Description' field shows 'Date: 14-Nov-02' and 'Time: 16:50:10'. At the bottom are buttons for 'Input Properties ...', 'Change Material Status ...', and 'Apply'. The 'Input Options' dialog box on the right has a title bar 'Input Options' and contains: 'Constitutive Model:' with a dropdown set to 'Solid properties'. Below is a table with two columns, 'Property Name' and 'Value'. The table has three rows: 'Thermal Conductivity =' with the value '204', 'Specific Heat =' with an empty field, and 'Density =' with an empty field. Below the table is a section for 'Temperature Dep/Model Variable Fields:' with an empty list box. At the bottom are buttons for 'OK', 'Clear', and 'Cancel'.

Property Name	Value
Thermal Conductivity =	204
Specific Heat =	
Density =	

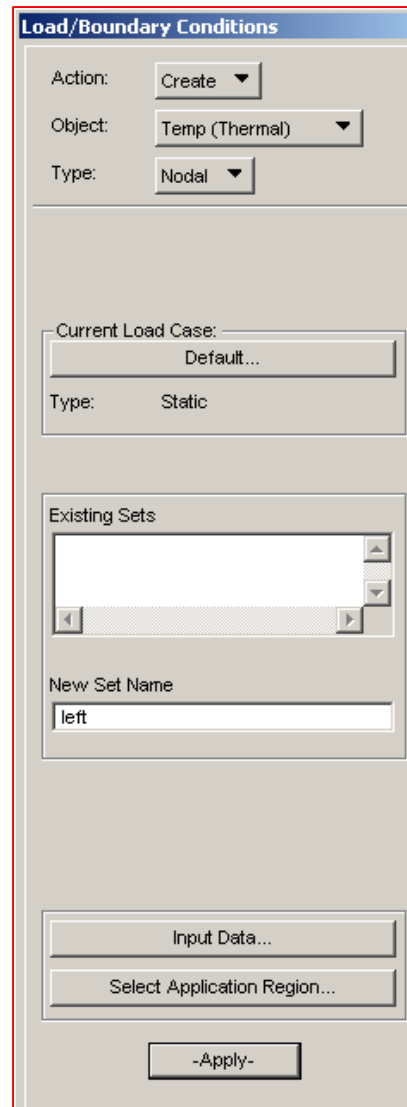
PROJECT 4, FREE CONVECTION, $H=H(TEMP)$

- Create 1D finite element properties.
 - 1D Rod
 - Constant cross-section area
 - The application region is the curve, Curve 1.



PROJECT 4, FREE CONVECTION, $H=H(TEMP)$

- Create a fixed temperature BC at one end of the rod.
 - Specify 1300 °K.
 - Apply at Point 1.



Load/Boundary Conditions

Action:

Object:

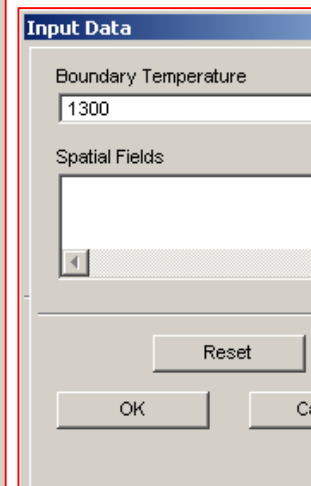
Type:

Current Load Case:

Type: Static

Existing Sets

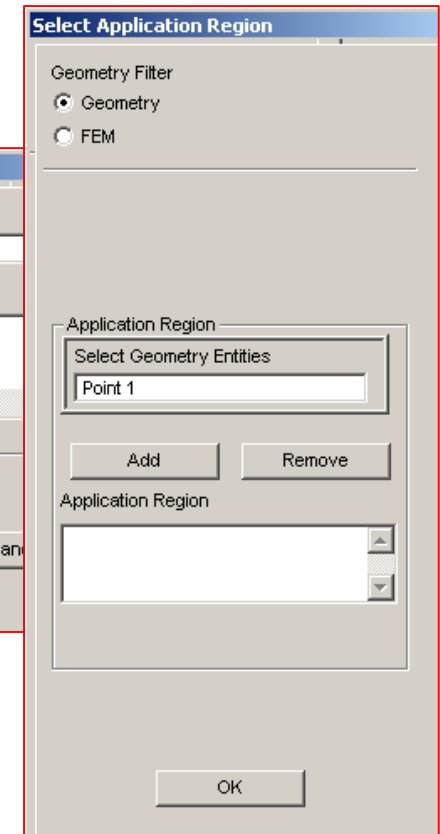
New Set Name:



Input Data

Boundary Temperature:

Spatial Fields



Select Application Region

Geometry Filter

☒ Geometry

☐ FEM

Application Region

Select Geometry Entities

Application Region

PROJECT 4, FREE CONVECTION, $h=h(T)$

- Create a temperature varying convection coefficient, $h = h(T)$.
 - Specify data using tabular input.
 - Set the function to vary only with temperature.

The image shows two overlapping dialog boxes from MSC Software. The 'Fields' dialog box is in the background, and the '1D Material Scalar Table Data' dialog box is in the foreground.

Fields Dialog Box:

- Action: Create
- Object: Material Property
- Method: Tabular Input
- Existing Fields: (Empty list)
- Field Name: h_temp
- Table Definition - Active Independent Variables:
 - ☒ Temperature (T)
 - ☐ Strain (e)
 - ☐ Strain Rate (er)
 - ☐ Time (t)
 - ☐ Frequency (f)
- Buttons: Input Data ..., [Options...]

1D Material Scalar Table Data Dialog Box:

- Input Scalar Data: (Empty text box)
- Data Table:

	T	Value
1	4.00000E+002	2.27000E+000
2	6.00000E+002	2.03000E+000
3	8.00000E+002	1.83000E+000
4		
5		
6		
7		
8		
9		

OK

PROJECT 4, FREE CONVECTION, $h=h(\text{TEMP})$

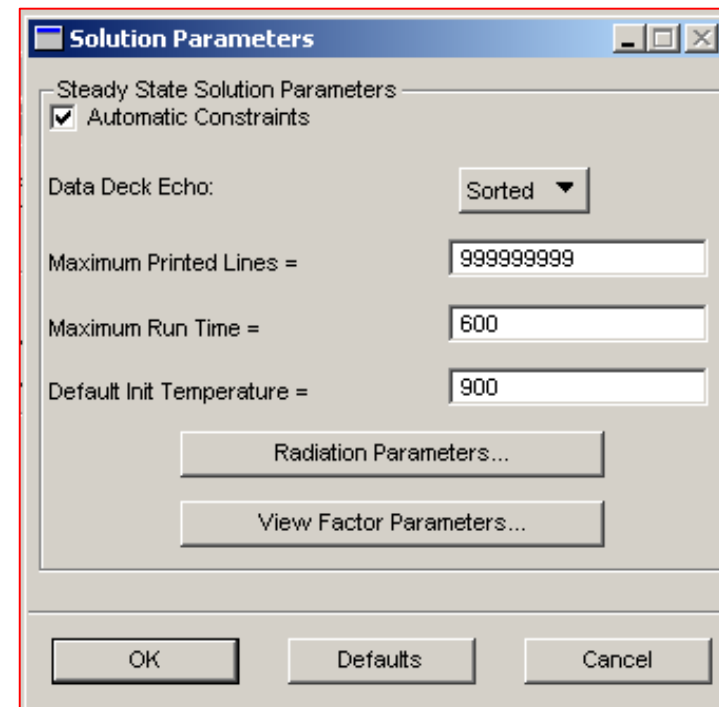
- Create temperature varying convection.
 - Element Uniform instead of Nodal
 - Use To Ambient
 - 1D
 - Scale factor for the convection coefficient of 1.0
 - Ambient temp of 300.0
 - Exponent of 0.25
 - Use $h = h(T)$ Field
 - Apply to Curve 1

The screenshot displays three overlapping dialog boxes from the MSC Software Nastran 2D/3D interface:

- Load/Boundary Conditions**:
 - Action: Create
 - Object: Convection
 - Type: Element Uniform
 - Option: To Ambient
 - Current Load Case: Default...
 - Type: Static
 - Existing Sets: (empty list)
 - New Set Name: conv
 - Target Element Type: 1D
 - Buttons: Input Data..., Select Application Region..., -Apply-
- Input Data**:
 - Form Type: Advanced
 - Convection Coefficient: 1.0
 - Ambient Temperature: 300.0
 - Formula Type Option:
 - ☒ $q=h(T_s-T_a)**EXPF*(T_s-T_a)$
 - ☐ $q=h(T_s**EXPF-T_a**EXPF)$
 - Convection Exponent (EXP): 0.25
 - Spatial Fields: (empty list)
 - Temperature: h_temp
 - Reference Temperature Option:
 - ☒ Average Temp $(T_s+T_a)/2$
 - ☐ Surface
 - ☐ Ambient
 - ☐ Film Temp
 - Buttons: OK, Reset
- Select Application Region**:
 - Geometry Filter:
 - ☒ Geometry
 - ☐ FEM
 - Application Region:
 - Select Curves: Curve 1
 - Buttons: Add, Remove
 - Buttons: OK

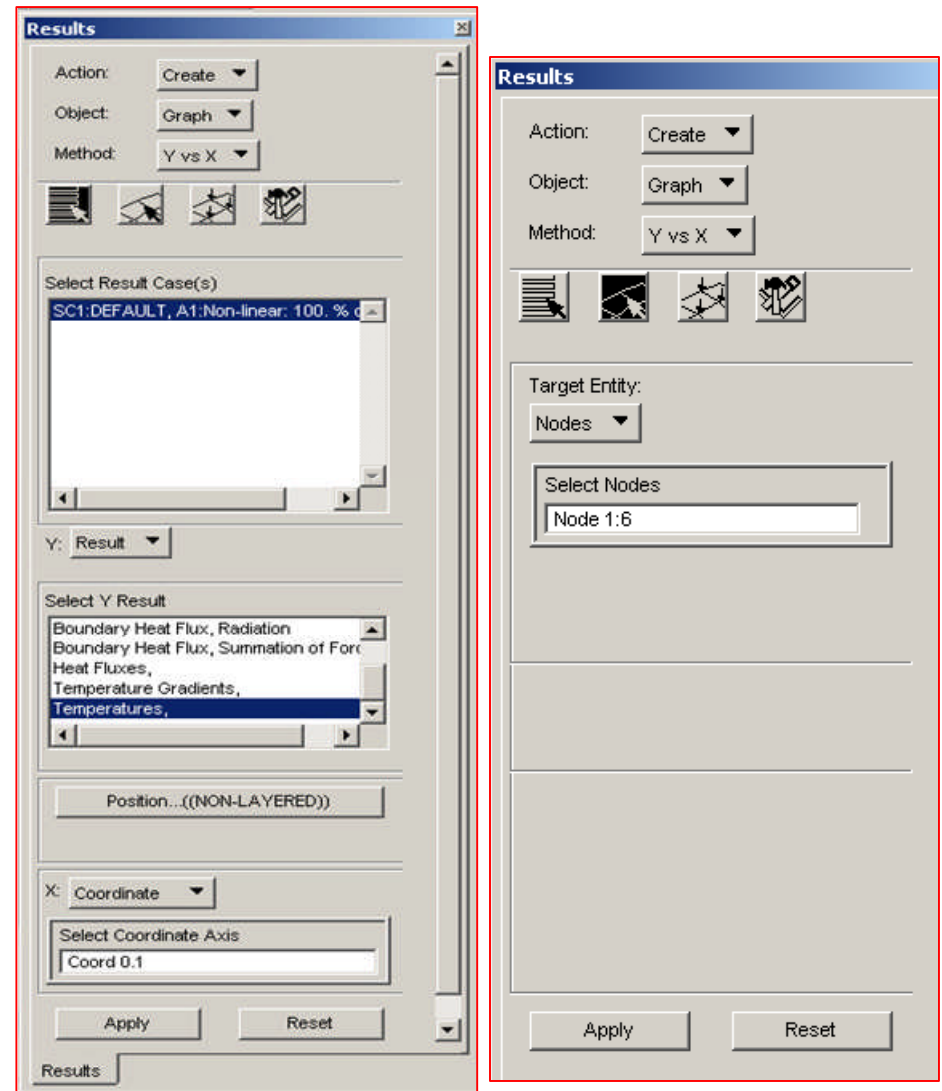
PROJECT 4, FREE CONVECTION, $H=H(\text{TEMP})$

- Run steady-state simulation.
 - Default initial temperature of 900 °K.
 - No radiation for this problem.
- Read the results.

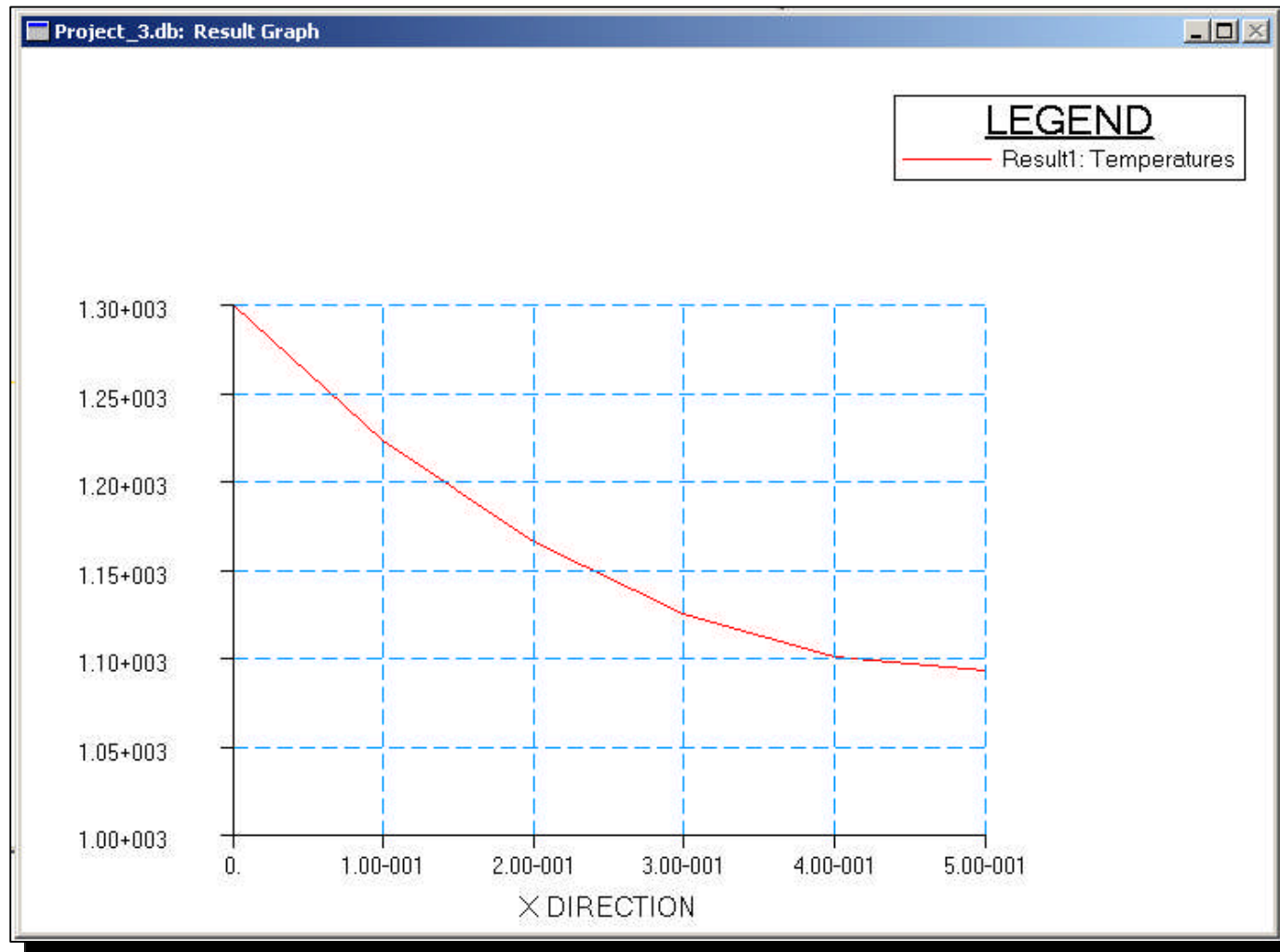


PROJECT 4, FREE CONVECTION, $H=H(TEMP)$

- Create a plot of temperature versus the path defined by Nodes 1:6.



PROJECT 4, FREE CONVECTION, $H=H(TEMP)$



NAS104, Section 4, September 2008

Copyright© 2008 MSC Software Corporation

S4-46

PROJECT 4, FREE CONVECTION, $H=H(TEMP)$

Extracted from MD Nastran model .DAT file

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on July      06, 1998 at
$ 11:12:53.
ASSIGN OUTPUT2 = 'project3.op2', UNIT = 12
$ Direct Text Input for File Management Section
$ Steady State Analysis, Database
SOL 153
TIME 600
$ Direct Text Input for Executive Control
CEND
ANALYSIS = HEAT
TITLE = MSC/NASTRAN job created on 06-Jul-98 at 11:12:05
ECHO = SORT
MAXLINES = 999999999
TEMPERATURE(INITIAL) = 1
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 1
  THERMAL (SORT1, PRINT)=ALL
  FLUX (SORT1, PRINT)=ALL
```



PROJECT 4, FREE CONVECTION, $H=H(TEMP)$

Extracted from MD Nastran model .DAT file

```
BEGIN BULK
NLPARM 1 0 AUTO 5 25 PW NO + A
+ A .001 1.-7
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : rod
PROD 1 1 .007854
CROD 1 1 1 2
CROD 2 1 2 3
CROD 3 1 3 4
CROD 4 1 4 5
CROD 5 1 5 6
$ Referenced Material Records
$ Material Record : alum
$ Description of Material : Date: 06-Jul-98 Time: 10:53:38
MAT4 1 204.
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
GRID 2 .1 0. 0.
GRID 3 .2 0. 0.
GRID 4 .3 0. 0.
GRID 5 .4 0. 0.
GRID 6 .5 0. 0.
```

PROJECT 4, FREE CONVECTION, $H=H(TEMP)$

```

$ Fixed Temperatures of Load Set : left
SPC      1      1      1      1300.
$ Fixed Temperatures of Load Set : conv
SPC      1      7      1      300.
$ Convection to Ambient of Load Set : conv
PCONV    1      1001    0      .25
CONV     1      1      0      0      7
CONV     2      1      0      0      7
CONV     3      1      0      0      7
CONV     4      1      0      0      7
CONV     5      1      0      0      7
$ Initial Temperatures from Temperature Load Sets
TEMP     1      1      1300.    7      300.
$ Default Initial Temperature
TEMPD    1      900.
$ CHBDYP Surface Elements and Element Properties
CHBDYP   1      1      LINE      1      2      +      B
+      B      .57735 .57735 .57735
CHBDYP   2      1      LINE      2      3      +      C
+      C      .57735 .57735 .57735
CHBDYP   3      1      LINE      3      4      +      D
+      D      .57735 .57735 .57735
CHBDYP   4      1      LINE      4      5      +      E
+      E      .57735 .57735 .57735
CHBDYP   5      1      LINE      5      6      +      F
+      F      .57735 .57735 .57735
PHBDY    1      .31416
    
```

Extracted from MD Nastran model .DAT file

PROJECT 4, FREE CONVECTION, $H=H(TEMP)$

Extracted from MD Nastran model .DAT file

```
$ Free Convection Heat Transfer Coefficients
MAT4      1001                      1.
MATT4     1001                      1
$ Material Tables ( No Conversions Needed )
$ Temperature Dependent Material Table : h_temp
TABLEM1   1                        +      G
+      G 400.      2.27      600.      2.03      800.      1.83      ENDT
$ Scalar Points
SPOINT    7
$ Referenced Coordinate Frames
ENDDATA
```


PROJECT 4, FREE CONVECTION, H=H(TEMP)

Extract from MD Nastran ASCII Results File

NON - LINEAR ITERATION MODULE OUTPUT

STIFFNESS UPDATE TIME 0.09 SECONDS
SUBCASE 1
ITERATION TIME 0.01 SECONDS
LOAD FACTOR 1.0000000

- - - CONVERGENCE FACTORS - - -

- - - LINE SEARCH

DATA - - -

ITERATION	EUI	EPI	EWI	LAMBDA	DLMAG	FACTOR	E-FIRST
E-FINAL	NQNV	NLS	ENIC	NDV	MDV		
1	2.0720E+01	3.9058E-03	8.6650E-04	1.0000E-01	1.2258E+01	1.0000E+00	8.5258E-03
8.5258E-03	0	0	0	1			
2	2.6654E-03	1.0407E-04	4.8540E-07	6.3323E-02	3.2779E-01	1.0000E+00	1.6088E-02
1.6088E-02	1	0	0	1			
3	1.3388E-05	9.0986E-07	5.0656E-11	3.6033E-02	2.8021E-03	1.0000E+00	3.5347E-03
3.5347E-03	2	0	-2	0	1		
0*** USER INFORMATION MESSAGE 6186,							
*** SOLUTION HAS CONVERGED ***							

SUBID 1 LOOPID 1 LOAD STEP 1.000 LOAD FACTOR 1.00000000

PROJECT 4, FREE CONVECTION, H=H(TEMP)

Extract from MD Nastran ASCII Results File

1 SUBCASE

LOAD STEP = 1.00000E+00

T E M P E R A T U R E V E C T O R

POINT ID.	TYPE	ID	VALUE	ID+1	VALUE	ID+2	VALUE	ID+3	VALUE	ID+4	VALUE	ID+5	VALUE
1	S	1.300000E+03	1.223212E+03	1.165063E+03	1.124368E+03	1.100286E+03	1.092315E+03						
7	S	3.000000E+02											

MSC/NASTRAN JOB CREATED ON 06-JUL-98 AT 11:12:05 JULY 6, 1998 MSC/NASTRAN 12/11/96

PAGE 11

DEFAULT

1 SUBCASE

LOAD STEP = 1.00000E+00

H E A T F L O W I N T O H B D Y E L E M E N T S (CHBDY)

ELEMENT-ID	APPLIED-LOAD	FREE-CONVECTION	FORCED-CONVECTION	RADIATION	TOTAL
1	0.000000E+00	-3.110860E+02	0.000000E+00	0.000000E+00	-3.110860E+02
2	0.000000E+00	-2.892282E+02	0.000000E+00	0.000000E+00	-2.892282E+02
3	0.000000E+00	-2.729203E+02	0.000000E+00	0.000000E+00	-2.729203E+02
4	0.000000E+00	-2.621096E+02	0.000000E+00	0.000000E+00	-2.621096E+02
5	0.000000E+00	-2.567264E+02	0.000000E+00	0.000000E+00	-2.567264E+02

FORMS IN FREE CONVECTION

Free Convection Application

Free convection heat transfer is available through the CONV Bulk Data entry. In MSC.Nastran, free convection is governed by relationships of the following forms:

- $q = H \cdot (T - TAMB)^{\exp f}(T - TAMB)$
- $q = H \cdot u_{CNTRLND}(T - TAMB)^{\exp f}(T - TAMB)$
- $q = H(T^{\exp f} - TAMB^{\exp f})$
- $q = H \cdot u_{CNTRLND}(T^{\exp f} - TAMB^{\exp f})$

where

H = free convection heat transfer coefficient

T = surface temperature

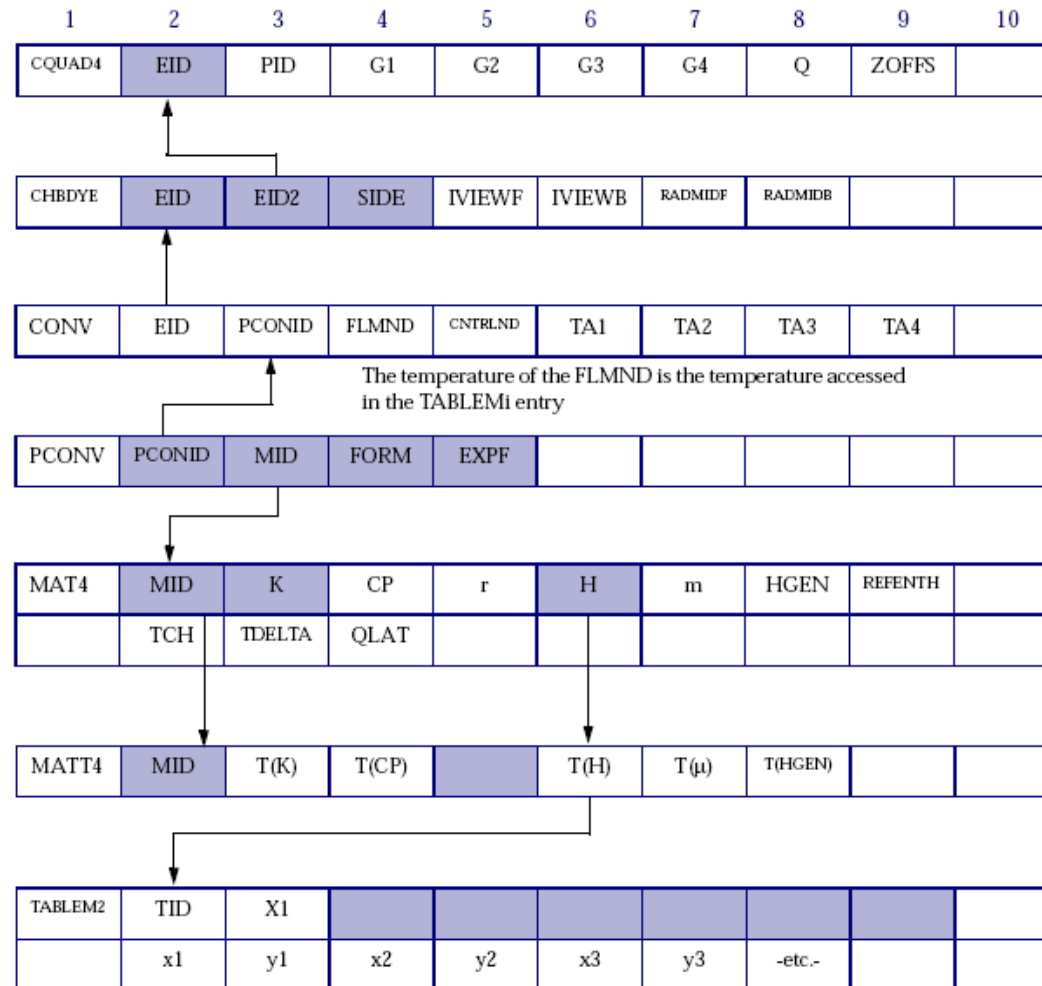
$TAMB$ = ambient temperature

$u_{CNTRLND}$ = value of the control node (dimensionless)

- The $u_{CNTRLND}$ in general is used in transient thermal analysis to provide a convection coefficient as a function of time.

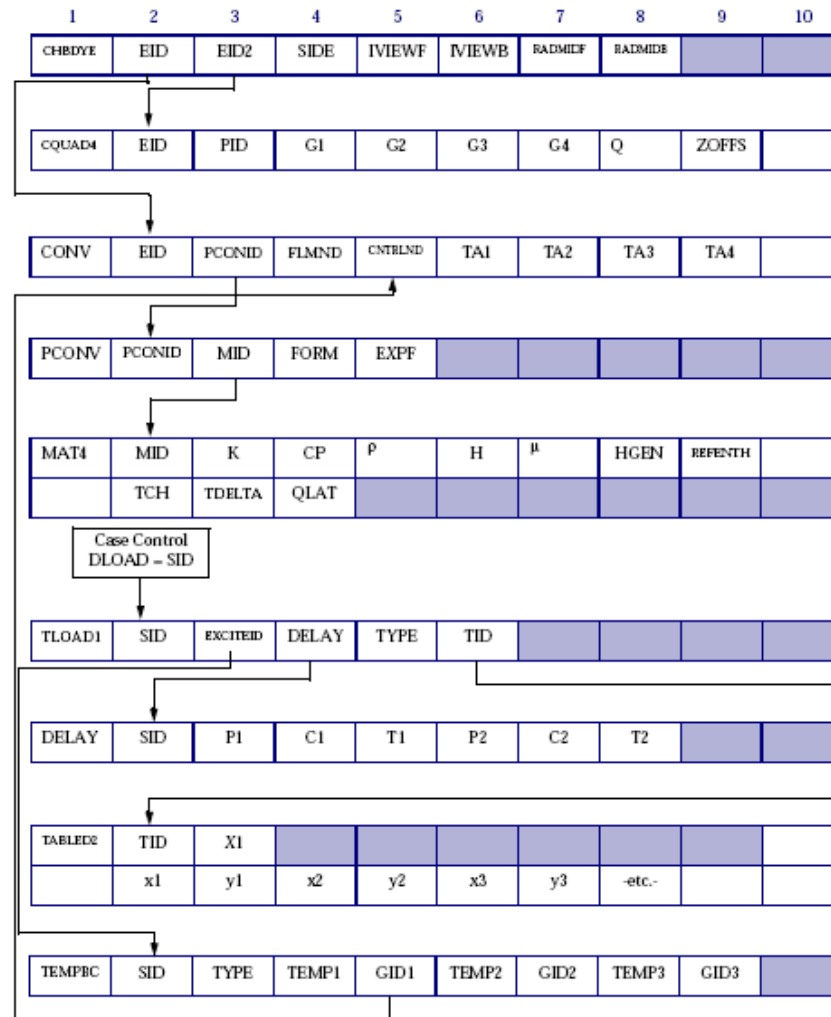
FREE CONVECTION TEMPLATE

Free Convection - Temperature-Dependent Heat Transfer Coefficient



FREE CONVECTION TEMPLATE

Free Convection - Time-Dependent Heat Transfer Coefficient



EXERCISES

- Perform Workshop 3 “Free Convection From Printed Circuit Board” in the exercise workbook.

FORCED CONVECTION



FORCED CONVECTION CONVM

CONVM

Heat Boundary Element Forced Convection Entry

Specifies a forced convection boundary condition for heat transfer analysis through connection to a surface element (CHBDYi entry).

Format:

1	2	3	4	5	6	7	8	9	10
CONVM	EID	PCONID	FLMND	CNTMDOT	TA1	TA2			

Example:

CONVM	101	1	201	301	20	21			

Field	Contents
EID	CHBDYP element identification number. (Integer > 0)
PCONID	Convection property identification number of a PCONVM entry. (Integer > 0)
FLMND	Point used for fluid film temperature. (Integer ≥ 0 ; Default = 0)
CNTMDOT	Control point used for controlling mass flow. (Integer > 0)
TA1, TA2	Ambient points used for convection. (Integer > 0 for TA1 and Integer ≥ 0 for TA2; Default for TA2 is TA1.)

FORCED CONVECTION PCONVM

PCONVM

Forced Convection Property Definition

Specifies the forced convection boundary condition properties of a boundary condition surface element used for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
PCONVM	PCONID	MID	FORM	FLAG	COEF	EXPR	EXPPI	EXPPO	

Example:

PCONVM	3	2	1	1	.023	0.80	0.40	0.30	

Field	Contents
PCONID	Convection property identification number. (Integer > 0)
MID	Material property identification number. (Integer > 0)
FORM	Type of formula used for convection. (Integer = 0, 1, 10, 11, 20, or 21; Default = 0)
FLAG	Flag for mass flow convection. (Integer = 0 or 1; Default = 0)
COEF	Constant coefficient used for forced convection. (Real > 0.0)
EXPR	Reynolds number convection exponent. (Real $\geq$ 0.0; Default = 0.0)
EXPPI	Prandtl number convection exponent for heat transfer into the working fluid. (Real $\geq$ 0.0; Default = 0.0)
EXPPO	Prandtl number convection exponent for heat transfer out of the working fluid. (Real $\geq$ 0.0; Default = 0.0)

FORCED CONVECTION PCONVM (Cont.)

4. FORM controls the type of formula used in determination of the forced convection film coefficient h . There are two cases:
- If FORM = 0, 10, or 20 then $h = \text{coef} \cdot \text{Re}^{\text{EXPR}} \cdot \text{Pr}^{\text{EXPP}}$.
 - If FORM = 1, 11, or 21 then the above h is multiplied by k and divided by the average hydraulic diameter.
 - FORM also specifies the reference temperature used in calculating material properties for the fluid if FLMND = 0.
 - If FORM = 0 or 1, the reference temperature is the average of element grid point temperatures (average) and the ambient point temperature (average).
 - If FORM = 10 or 11, the reference temperature is the surface temperature (average of element grid point temperatures).
 - If FORM = 20 or 21, the reference temperature is the ambient temperature (average of ambient point temperature).
5. In the above expression, EXPP is EXPPI or EXPPO, respectively, for heat flowing into or out of the working fluid. This determination is performed internally.

EXAMPLE 6, FORCED CONVECTION TUBE FLOW

- Example 6, Forced Convection Tube Flow, Constant Property Flow
- Demonstrated Principles
 - Forced Convection Fluid Elements
 - Control Node for Mass Flow Rate
 - Relationships for Tube Flows
 - Film Nodes for Forced Convection
 - Constant Heat Transfer Coefficient

Working Fluid = Water:

$$K = .065 \text{ W/m } ^\circ\text{C}$$

$$C_p = 4200. \text{ J/kg } ^\circ\text{C}$$

$$\rho = 1000. \text{ kg/m}^3$$

$$\mu = .0 \times 10^{-3} \text{ kg/m sec}$$

$$h = 200. \text{ W/m}^2 ^\circ\text{C} = \text{constant}$$

$$\dot{m} = 0.1 \text{ kg/sec}$$

$$\text{DIA} = .05 \text{ m}$$

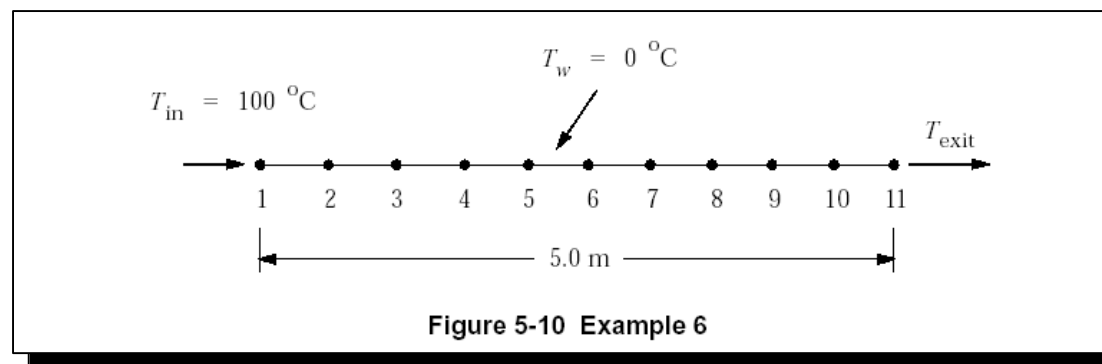


Figure 5-10 Example 6

EXAMPLE 6, MD NASTRAN INPUT

```
SOL 153
TIME 10
CEND
TITLE = EXAMPLE 6                                EX6
ANALYSIS = HEAT
THERMAL = ALL
FLUX = ALL
SPCF = ALL
OLOAD = ALL
SPC = 10
TEMP(INIT) = 20
NLPARM = 100
BEGIN BULK
NLPARM,100
$
GRID,1,,0.0,0.0,0.0
GRID,2,,0.5,0.0,0.0
GRID,3,,1.0,0.0,0.0
GRID,4,,1.5,0.0,0.0
GRID,5,,2.0,0.0,0.0
GRID,6,,2.5,0.0,0.0
GRID,7,,3.0,0.0,0.0
GRID,8,,3.5,0.0,0.0
GRID,9,,4.0,0.0,0.0
GRID,10,,4.5,0.0,0.0
GRID,11,,5.0,0.0,0.0
GRID,50,,50.0,50.0,50.0
GRID,99,,99.0,99.0,99.0
```

```
CHBDYP,10,25,FTUBE,,,1,2
CHBDYP,20,25,FTUBE,,,2,3
CHBDYP,30,25,FTUBE,,,3,4
CHBDYP,40,25,FTUBE,,,4,5
CHBDYP,50,25,FTUBE,,,5,6
CHBDYP,60,25,FTUBE,,,6,7
CHBDYP,70,25,FTUBE,,,7,8
CHBDYP,80,25,FTUBE,,,8,9
CHBDYP,90,25,FTUBE,,,9,10
CHBDYP,100,25,FTUBE,,,10,11
PHBDY,25,,0.05,0.05
$
CONVM,10,95,,50,99
CONVM,20,95,,50,99
CONVM,30,95,,50,99
CONVM,40,95,,50,99
CONVM,50,95,,50,99
CONVM,60,95,,50,99
CONVM,70,95,,50,99
CONVM,80,95,,50,99
CONVM,90,95,,50,99
CONVM,100,95,,50,99
$
PCONVM,95,15,0,1,200.0,0.0,0.0,0.0
MAT4,15,0.65,4200.0,1000.0,,1.0E-03
$
SPC,10,1,,100.0
SPC,10,99,,0.0
SPC,10,50,,0.1
$
TEMP,20,1,100.0
TEMP,20,99,0.0
TEMP,20,50,0.1
TEMPD,20,100.0
$
ENDDATA
```

EXAMPLE 6, MD NASTRAN OUTPUT

```

0
LOAD STEP = 1.00000E+00

                                T E M P E R A T U R E   V E C T O R

POINT ID.  TYPE      ID  VALUE    ID+1 VALUE    ID+2 VALUE    ID+3 VALUE    ID+4 VALUE    ID+5 VALUE
    1      S      1.000000E+02  9.639484E+01  9.291965E+01  8.956976E+01  8.634062E+01  8.322791E+01
    7      S      8.022740E+01  7.733508E+01  7.454703E+01  7.185950E+01  6.926884E+01
   50      S      1.000000E-01
   99      S      0.0

1  EXAMPLE 6                      EX6                      JANUARY 12, 2004  MD NASTRAN  9/23/03  PAGE
   10

0
LOAD STEP = 1.00000E+00

                                L O A D   V E C T O R

POINT ID.  TYPE      ID  VALUE    ID+1 VALUE    ID+2 VALUE    ID+3 VALUE    ID+4 VALUE    ID+5 VALUE
    7      S      0.0          0.0          0.0          0.0          0.0
   50      S      0.0
   99      S      0.0

1  EXAMPLE 6                      EX6                      JANUARY 12, 2004  MD NASTRAN  9/23/03  PAGE
   11

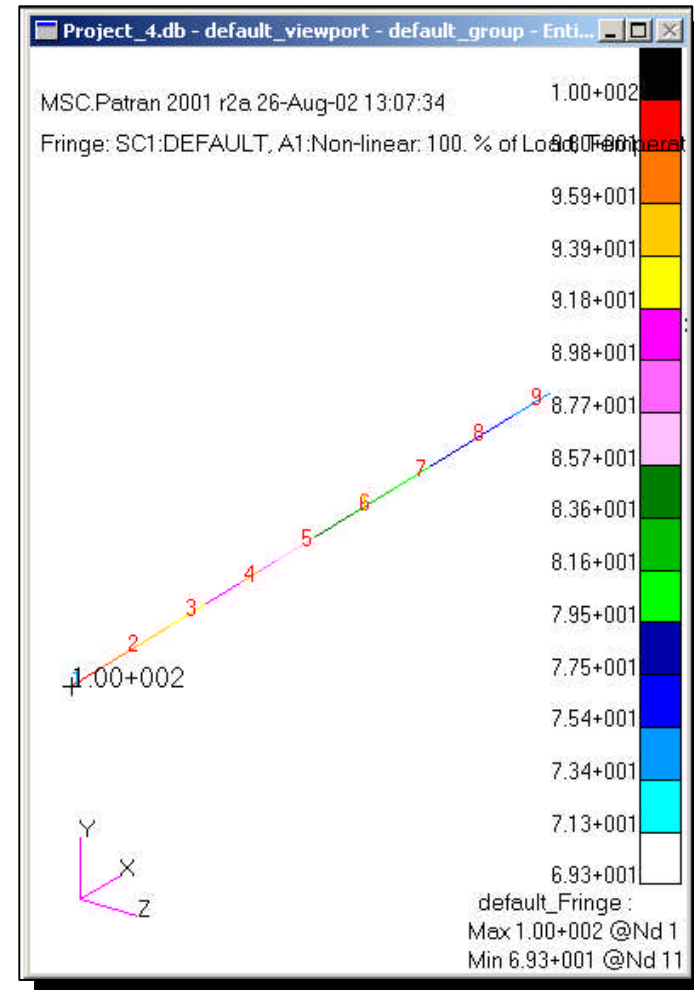
0
LOAD STEP = 1.00000E+00

                                F O R C E S   O F   S I N G L E - P O I N T   C O N S T R A I N T

POINT ID.  TYPE      ID  VALUE    ID+1 VALUE    ID+2 VALUE    ID+3 VALUE    ID+4 VALUE    ID+5 VALUE
    7      S      0.0          0.0          0.0          0.0          0.0
   50      S      0.0
   99      S     -1.338981E+04
    
```

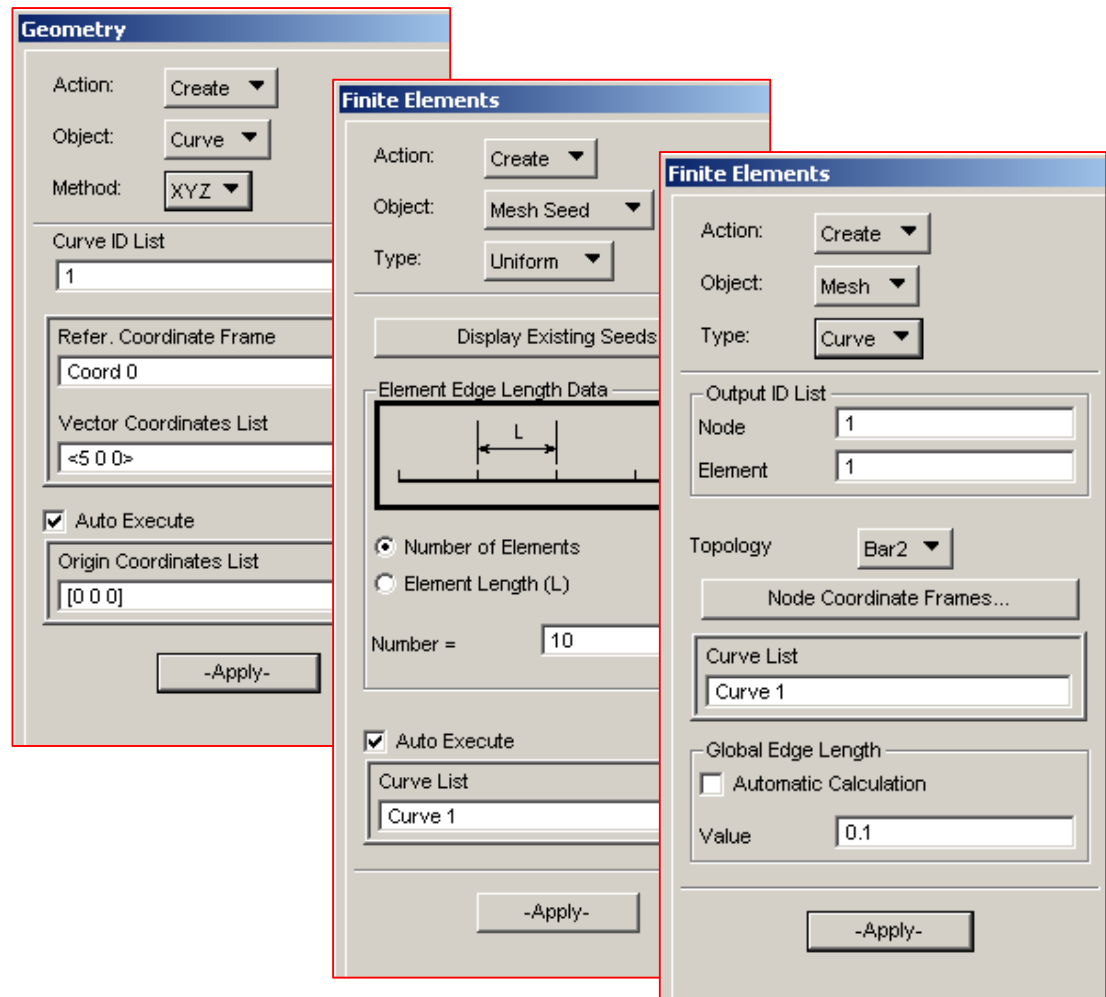
PROJECT 5, PART 1, 1D FLOW NETWORKS

- Based on Example 6 in the MD Nastran Thermal Analysis User's Guide.
- Water flowing at 0.1kg/sec enters a 0.05 m diameter tube.
- The inlet is fixed at 100.0 C.
- Heat is convected away from the flow to a wall temperature of 0.0 C.
- The convection coefficient is constant at 200 Watts/(m²*C).
- Run a SS analysis to determine the outlet temperature.
- Illustrates use of CHBDYP (FTUBE).



PROJECT 5, PART 1, 1D FLOW NETWORKS

- Create a curve 5m long in the X-direction.
- Mesh seed for 10 elements on Curve 1.
- IsoMesh Curve 1
 - The mesh seeds control how many 1D bar elements are created on the curve.



PROJECT 5, PART 1, 1D FLOW NETWORKS

- Specify the properties for the fluid.
- Necessary to specify
 - Thermal Conductivity, K
 - Specific Heat, C_p
 - Density, ρ
 - Dynamic Viscosity, μ

The image shows two overlapping software dialog boxes from MSC Software. The 'Materials' dialog box on the left has a title bar 'Materials' and contains the following fields: 'Action:' with a dropdown set to 'Create', 'Object:' with a dropdown set to 'Isotropic', and 'Method:' with a dropdown set to 'Manual Input'. Below these is a search bar with an asterisk and a 'Filter' button. The 'Existing Materials' section is empty. The 'Material Name' field contains 'fluid', and the 'Description' field contains 'Date: 23-Nov-02 Time: 22:46:15'. At the bottom are buttons for 'Input Properties ...', 'Change Material Status ...', and 'Apply'. The 'Input Options' dialog box on the right has a title bar 'Input Options' and a 'Constitutive Model:' dropdown set to 'Fluid properties'. It contains a table with the following data:

Property Name	Value
Thermal Conductivity =	0.65
Specific Heat =	4200.
Density =	1000.
Dynamic Viscosity =	0.001

Below the table is a 'Temperature Dep/Model Variable Fields:' section with an empty list box. The 'Current Constitutive Models:' section contains a list box with 'Fluid properties - [...] - [Active]'. At the bottom are 'OK', 'Clear', and 'Cancel' buttons.

PROJECT 5, PART 1, 1D FLOW NETWORKS

- Define 10 flow tubes on Curve 1.
 - Type: Flow Tube
 - Material property set, fluid
 - Hydraulic diameter = $4 \cdot A/P = 4 \cdot \pi \cdot R^2 / (2 \cdot \pi \cdot R) = 2 \cdot R = D$

The image shows two overlapping dialog boxes from the MSC Software interface. The 'Element Properties' dialog is in the background, and the 'Input Properties' dialog is in the foreground.

Element Properties Dialog:

- Action: Create
- Object: 1D
- Type: Flow Tube
- Existing Property Sets: (empty list)
- Property Set Name: ftube
- Input Properties ... button
- Application Region: Select Members (Curve 1)
- Add and Remove buttons
- Application Region: (empty list)
- Apply- button

Input Properties Dialog:

Flow Tube (CHBDYP)

Property Name	Value	Value Type
Material Name	m:fluid	Mat Prop Name
Hydraulic Diam. at Nod	0.05	Real Scalar
[Hydraulic Diam. at Nod]		Real Scalar

Field Definitions: (empty list)

OK button

PROJECT 5, PART 1, 1D FLOW NETWORKS

- Specify inlet temperature of fluid in tube.
 - 100.0 C
 - Point 1 (at one end of the tube)

The image shows three overlapping software dialog boxes from MSC Software:

- Load/Boundary Conditions**:
 - Action: Create
 - Object: Temp (Thermal)
 - Type: Nodal
 - Current Load Case: Default...
 - Type: Static
 - Existing Sets: (empty list)
 - New Set Name: fluid_inlet
 - Buttons: Input Data..., Select Application Region..., -Apply-
- Input Data**:
 - Boundary Temperature: 100
 - Spatial Fields: (empty list)
 - Buttons: Reset, OK, Cancel
- Select Application Region**:
 - Geometry Filter: ☒ Geometry, ☐ FEM
 - Application Region: Select Geometry Entities
 - Point 1
 - Buttons: Add, Remove
 - Application Region: (empty list)
 - Button: OK

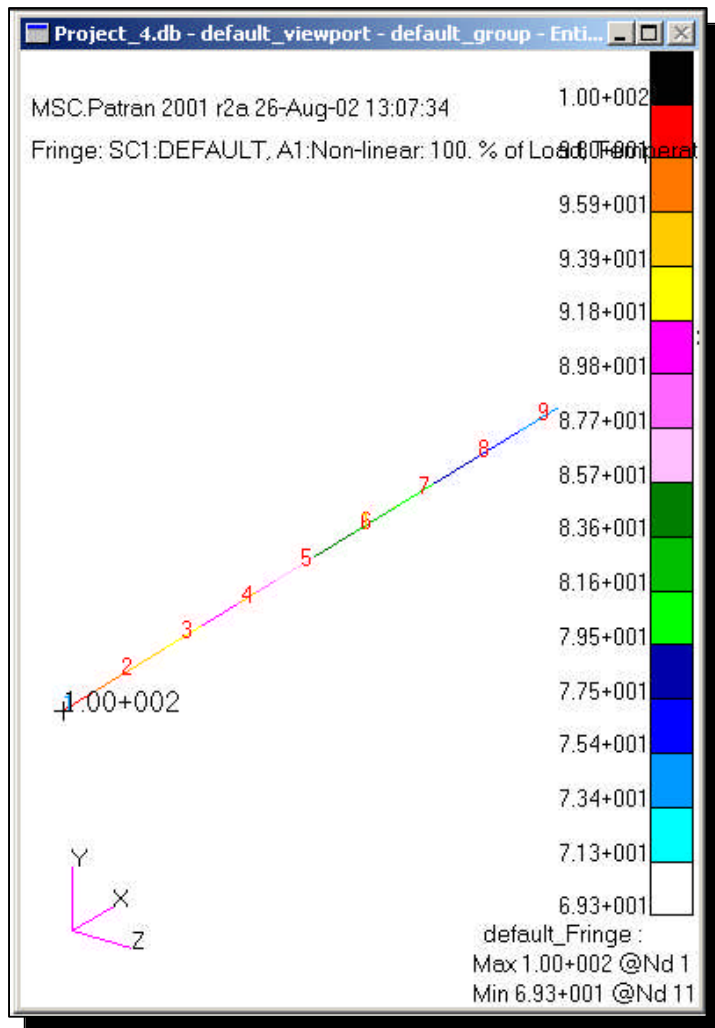
PROJECT 5, PART 1, 1D FLOW NETWORKS

- Define forced convection from flow in tube to ambient temperature.
 - Option: Flow Tube To Ambient
 - Target Element Type: 1D
 - Only 1D because flow tube
 - $\dot{m} = 0.1 \text{ kg/sec}$
 - $T_{\text{amb}} = 0 \text{ C}$
 - $h = 200 \text{ Watts/(m}^2\text{C)}$
 - All flow tubes on Curve 1

The image shows three overlapping software dialog boxes from MSC Software:

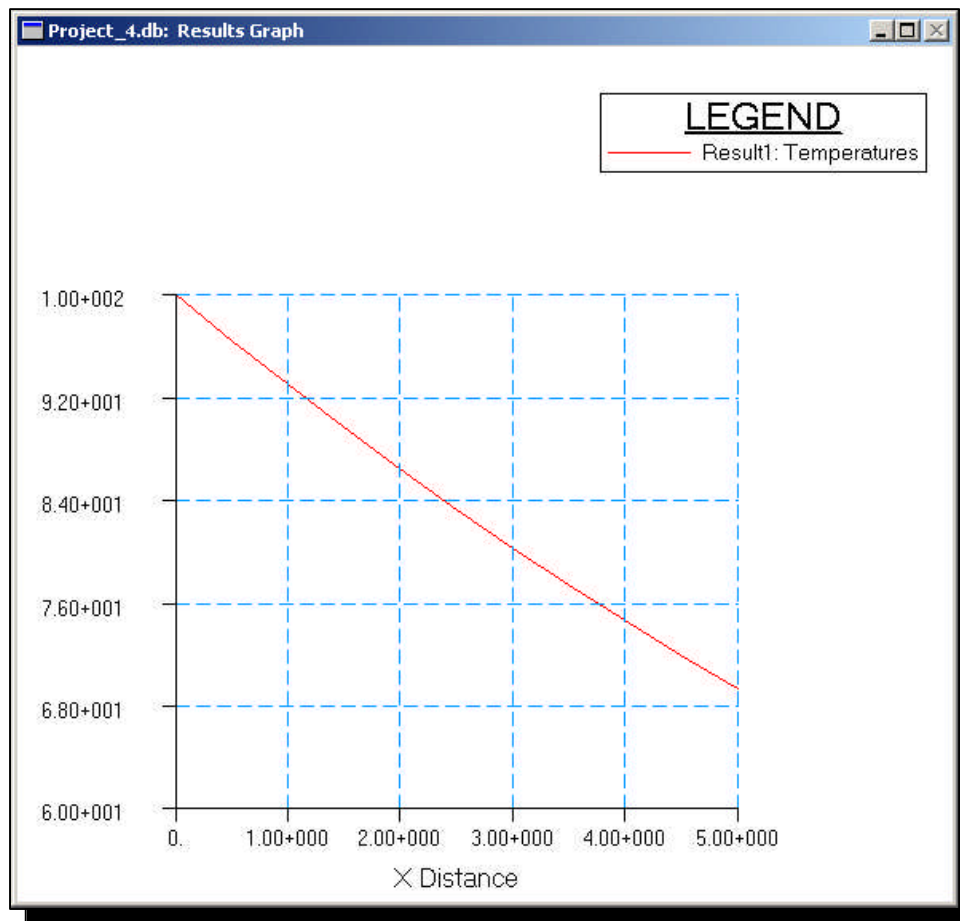
- Load/Boundary Conditions**:
 - Action: Create
 - Object: Convection
 - Type: Element Uniform
 - Option: Flow Tube To Ambient
 - Current Load Case: Default...
 - Type: Static
 - Existing Sets: (empty list)
 - New Set Name: forced
 - Target Element Type: 1D
 - Buttons: Input Data..., Select Application Region..., -Apply-
- Input Data**:
 - Form Type: Basic
 - Mass Flow Rate: 0.1
 - Ambient Temperature: 0
 - Heat Transfer Coefficient: 200
 - Buttons: Reset, OK
- Select Application Region**:
 - Geometry Filter: ☒ Geometry, ☐ FEM
 - Application Region: Select Curves
 - Curve 1
 - Buttons: Add, Remove
 - Application Region: (empty list)
 - Buttons: OK

PROJECT 5, PART 1, 1D FLOW NETWORKS



- Note that the inlet temperature is 100°C, and the fluid temperature decreases as the fluid travels down the duct.

PROJECT 5, PART 1, 1D FLOW NETWORKS



- The fluid temperature at $X = 0$ (inlet) is 100°C , and the fluid temperature has decreased to 69.6°C at $X = 5$ (tube exit).

PROJECT 5, PART 1, 1D FLOW NETWORKS

Extracted from MD Nastran model .DAT file

```

GRID      1      0.      0.      0.
GRID      2      .5      0.      0.
...
GRID     11      5.      0.      0.
$
CHBDYP    1      1      FTUBE      1      2
CHBDYP    2      1      FTUBE      2      3
...
CHBDYP   10      1      FTUBE     10     11
PHBDY     1      .05      .05
$ Scalar Points
SPOINT   12      THRU    13
$Tube Flow Convection to Ambient of Load Set : forced
CONVM     1      1      0      13     12
CONVM     2      1      0      13     12
...
CONVM    10      1      0      13     12
PCONVM    1      1      0      1      200.    0.    0.    0.
MAT4      1      .65    4200.   1000.    .001
    
```

PROJECT 5, PART 1, 1D FLOW NETWORKS

Extracted from MD Nastran model .DAT file

```
$ Loads for Load Case : Default
$ Fixed Temperatures of Load Set : Fluid_inlet
SPC      1      1      1      100.
$ Fixed Temperatures of Load Set : forced
$ Note that the SPOINT 13 represent the Mass flow rate
$ The SPOINT 12 represents the ambient temperature at
$ Zero degree
SPC      1      12      1      0.
SPC      1      13      1      .1
```


PROJECT 5, PART 1, 1D FLOW NETWORKS

Extract from MD Nastran ASCII Results File

0

SUBCASE 1

LOAD STEP = 1.00000E+00

TEMPERATURE VECTOR

POINT ID.	TYPE	ID	VALUE	ID+1	VALUE	ID+2	VALUE	ID+3	VALUE	ID+4	VALUE	ID+5	VALUE
1	S	1.000000E+02	9.639484E+01	9.291965E+01	8.956976E+01	8.634062E+01	8.322791E+01						
7	S	8.022740E+01	7.733508E+01	7.454703E+01	7.185950E+01	6.926884E+01	0.0						
13	S	1.000000E-01											

0

SUBCASE 1

LOAD STEP = 1.00000E+00

HEAT FLOW INTO HBDY ELEMENTS (CHBDY)

ELEMENT-ID	APPLIED-LOAD	FREE-CONVECTION	FORCED-CONVECTION	RADIATION	TOTAL
1	0.000000E+00	0.000000E+00	-1.514167E+03	0.000000E+00	-1.514167E+03
2	0.000000E+00	0.000000E+00	-1.459578E+03	0.000000E+00	-1.459578E+03
3	0.000000E+00	0.000000E+00	-1.406958E+03	0.000000E+00	-1.406958E+03
4	0.000000E+00	0.000000E+00	-1.356235E+03	0.000000E+00	-1.356235E+03
5	0.000000E+00	0.000000E+00	-1.307341E+03	0.000000E+00	-1.307341E+03
6	0.000000E+00	0.000000E+00	-1.260209E+03	0.000000E+00	-1.260209E+03
7	0.000000E+00	0.000000E+00	-1.214777E+03	0.000000E+00	-1.214777E+03
8	0.000000E+00	0.000000E+00	-1.170982E+03	0.000000E+00	-1.170982E+03
9	0.000000E+00	0.000000E+00	-1.128766E+03	0.000000E+00	-1.128766E+03
10	0.000000E+00	0.000000E+00	-1.088072E+03	0.000000E+00	

PROJECT 5, PART 2, 1D FLOW NETWORKS

USE CORRELATION

- As seen for Project 5, Part 1, PCONVM
PCONVM,1,1,0,1,200.,0.,0.,0.
FORM = 0 => $h = \text{coef} * \text{Re}^{**} \text{Expr} * \text{Pr}^{**} \text{Expp}$
coef = 200.
Expr = 0.
Expp = 0.
=> $h = \text{coef} = 200.$
- Now, use correlation
- For turbulent flow, $\text{Nu} = 0.023 * \text{Re}^{**} 0.8 * \text{Pr}^{**} 0.3$, where
 $\text{Nu} = h * d / k$. The following is found
 $h = (k/d) * 0.023 * \text{Re}^{**} 0.8 * \text{Pr}^{**} 0.3$ (FORM = 1)

PROJECT 5, PART 2, 1D FLOW NETWORKS

USE CORRELATION (Cont.)

- Entries for the PCONVM record
PCONVM,PCONID,MID,FORM,FLAG,COEF,EXPR,
EXPPI, EXPPO
- Change the current PCONVM record to a comment by adding a “\$”
\$PCONVM,1,1,0,1,200.,0.,0.,0.
- Create a new PCONVM record for (advanced) correlation
PCONVM,1,1,1,1,0.023,0.8,0.3,0.3
- Do this using MD Patran

PROJECT 5, PART 2, 1D FLOW NETWORKS

MODIFY CONVECTION WITH CORRELATION

Load/Boundary Conditions

Action:

Object:

Type:

Option:

Current Load Case:

Type: Static

Select Set to Modify

Rename Set as

Target Element Type:

Input Data

Form Type:

Mass Flow Rate

Ambient Temperature

Heat Transfer Coefficient

Formula Type Option
☐ $h = \text{coef} * \text{Re}^{**} \text{Expr} * \text{Pr}^{**} \text{Expp}$
☒ $h = k / d * \text{coef} * \text{Re}^{**} \text{Expr} * \text{Pr}^{**}$

Reference Temperature Option
☒ Average Temp $(T_s + T_a) / 2$
☐ Surface Temp (T_s)
☐ Ambient Temp (T_a)
☐ Film Temperature

Reynolds Exponent

Prandtl Exponent, Heat In

Prandtl Exponent, Heat Out

PROJECT 5, PART 2, 1D FLOW NETWORKS

MODIFY CONVECTION WITH CORRELATION

Extract from MD Nastran ASCII Results File.

```

0 SUBCASE 1
LOAD STEP = 1.00000E+00

      T E M P E R A T U R E   V E C T O R

POINT ID.   TYPE      ID   VALUE      ID+1 VALUE      ID+2 VALUE      ID+3 VALUE      ID+4 VALUE      ID+5 VALUE
    1         S      1.000000E+02  9.506394E+01  9.037153E+01  8.591073E+01  8.167013E+01  7.763884E+01
    7         S      7.380654E+01  7.016341E+01  6.670010E+01  6.340774E+01  6.027790E+01  0.0
   13         S      1.000000E-01
1 MSC/NASTRAN JOB CREATED ON 06-JUL-98 AT 15:33:40          JULY 6, 1998 MSC/NASTRAN 12/11/96 PAGE
11
DEFAULT
0 SUBCASE 1
LOAD STEP = 1.00000E+00

      H E A T   F L O W   I N T O   H B D Y   E L E M E N T S   (CHBDY)

ELEMENT-ID   APPLIED-LOAD   FREE-CONVECTION   FORCED-CONVECTION   RADIATION   TOTAL
    1      0.000000E+00      0.000000E+00      -2.073146E+03      0.000000E+00      -2.073146E+03
    2      0.000000E+00      0.000000E+00      -1.970814E+03      0.000000E+00      -1.970814E+03
    3      0.000000E+00      0.000000E+00      -1.873533E+03      0.000000E+00      -1.873533E+03
    4      0.000000E+00      0.000000E+00      -1.781055E+03      0.000000E+00      -1.781055E+03
    5      0.000000E+00      0.000000E+00      -1.693141E+03      0.000000E+00      -1.693141E+03
    6      0.000000E+00      0.000000E+00      -1.609566E+03      0.000000E+00      -1.609566E+03
    7      0.000000E+00      0.000000E+00      -1.530117E+03      0.000000E+00      -1.530117E+03
    8      0.000000E+00      0.000000E+00      -1.454589E+03      0.000000E+00      -1.454589E+03
    9      0.000000E+00      0.000000E+00      -1.382790E+03      0.000000E+00      -1.382790E+03
   10      0.000000E+00      0.000000E+00      -1.314535E+03      0.000000E+00      -1.314535E+03
  
```

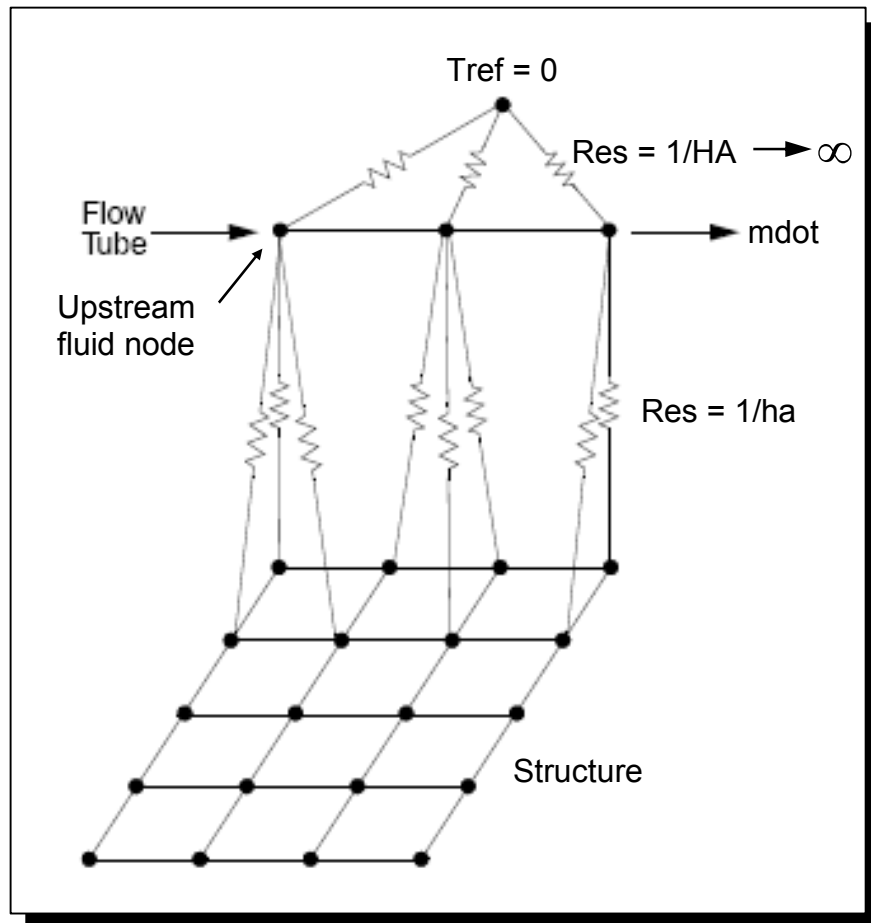
EXERCISES

- Perform Workshop 4 “Forced Air Convection From Printed Circuit Board” in the exercise workbook.

COUPLED ADVECTION

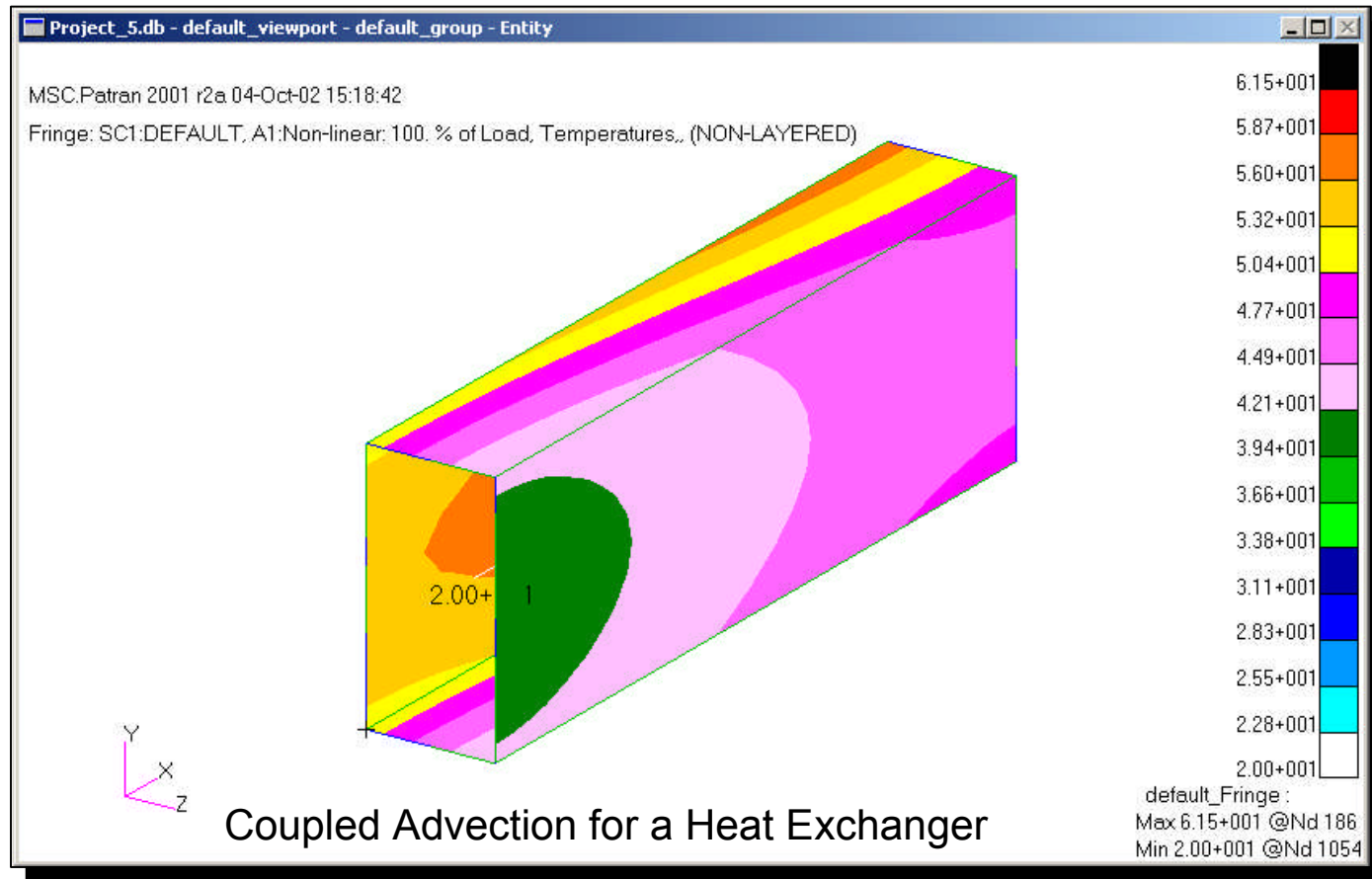


COUPLED ADVECTION



- Connection between an advection stream and a structural surface
- The forced convection tube relationships “disabled” by setting H (on PCONVM entry) to $1e-20$
- The convection between the flow stream and the structure surface is found from $\dot{q} = h \cdot a \cdot (T_{\text{flow stream}} - T_{\text{structure}})$
 - The coefficient “ h ” must be supplied by the user
 - The area “ a ” of a structural element is calculated by Patran
 - Application regions are used to associate flow tube and structure nodes. Patran uses a closest approach.

PROJECT 6, COUPLED ADVECTION



PROJECT 6, COUPLED ADVECTION

- Aluminum heat sink with a dimension of 1 inch height (Y-direction), 0.5 inch wide (Z-direction), and 3 inches in length (X-direction).
- The channel section is 0.8 inch by 0.4 inch.
- The fluid is air, and the inlet temperature is 20°C.

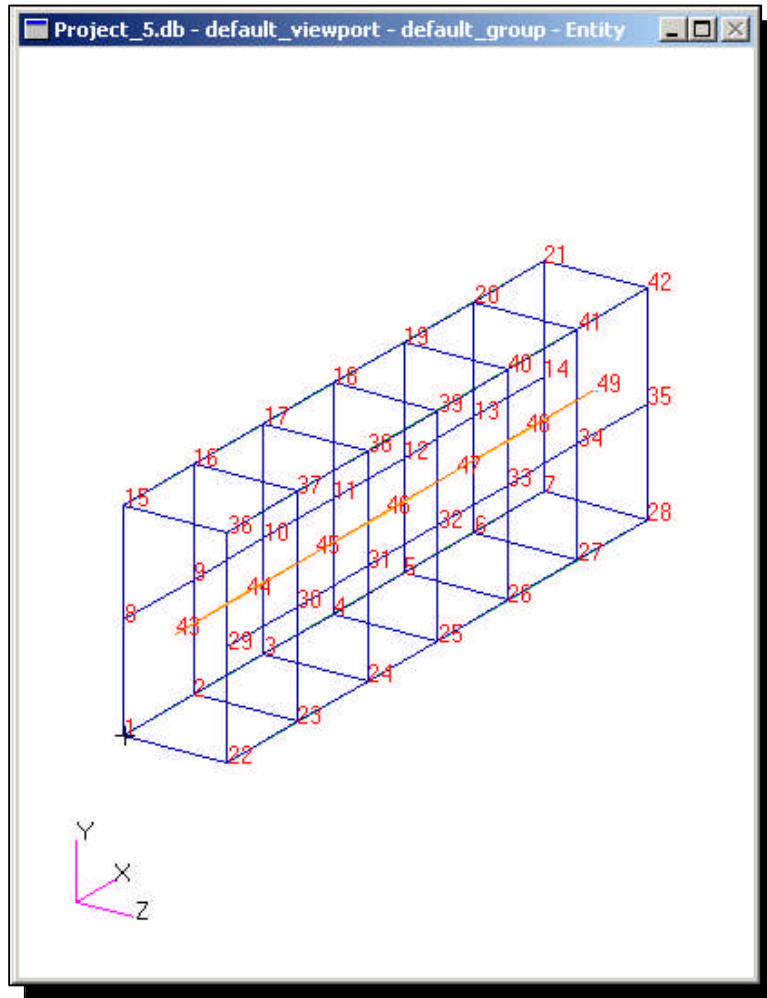
PROJECT 6, COUPLED ADVECTION

- The inlet mass flow rate is 0.008333 lbm/sec.
- The heat flux is imposed on one face (1 inch by 3 inch) with a value of 20 Watts/inch**2.
- The convection coefficient is assumed to be constant at 0.3 Watt/inch**2*C.

PROJECT 6, COUPLED ADVECTION

- Objective, determine the exit fluid temperature, and the maximum heat sink temperature.
- First model is with mesh element size of 0.5 inch.

PROJECT 6, PART 1, COUPLED ADVECTION



- Connect fluid element (1D bar from Node 43 to Node 44) to conduction element (2D CQUAD4 using Node 1,2,23, 22).
- Convect off surface element that has Node 1,2,23,22 using convection coefficient.

PROJECT 6, PART 1, CHBDYP (FTUBE) AND CONVM

- The following is used to model the flow stream (flow tube).
- \$CHBDYP,EID,PID,TYPE,,,G1,G2
- CHBDYP,100037,1,FTUBE,,,43,44
- \$PHBDY,PID,AF,D1,D2
- PHBDY,1,,0.5333,0.5333
- \$CONVM,EID,PCONID,FLMND,CNTMDOT,TA1
- CONVM,100037,1,0,51,50
- \$PCONVM,PCONID,MID,FORM,FLAG,COEF
- PCONVM,1,2,0,1,1.0E-20

PROJECT 6, PART 1, CHBDYP (FTUBE) AND CONVM (Cont.)

- The following is used to model the flow stream (flow tube).
- SPOINT 51 is used to specify the mass flow rate.
- SPOINT 50 is the ambient point that each fluid element is connected to.
- Very small convective coefficient ($\text{COEF} = 1.0\text{E-}20$), and thus the thermal resistance ($1/\text{HA}$) is very large. The reference temperature is arbitrarily set at zero degree.

PROJECT 6, PART 1, CHBDYG (AREA4) AND CONV

- The following is used to model the convection connection between the flow stream and the structure.
- \$CHBDYG,EID,,TYPE,,,,,+
- \$+,G1,G2,G3,G4
- CHBDYG,100001,,AREA4,,,,,+
- +,1,2,9,8
- \$CONV,EID,PCONID,FLMND,CNTRLND,TA1,TA2,...
- CONV,100001,1,0,0,43,44,44,43
- \$PCONV,PCONID,MID,FORM,EXPF
- PCONV,1,1001,0,0.
- \$MAT4,MID,,,H
- MAT4,1001,,,,0.3

PROJECT 6, PART 1, COUPLED ADVECTION

Create first side of channel

Geometry

Action:

Object:

Method:

Surface ID List

Refer. Coordinate Frame

Vector Coordinates List

☒ Auto Execute

Origin Coordinates List

Create other side
of channel

Geometry

Action:

Object:

Method:

Surface ID List

Type of Transformation
☒ Cartesian in Refer. CF
☐ Curvilinear in Refer. CF

Refer. Coordinate Frame

Translation Vector

Translation Parameters
Repeat Count

☐ Delete Original Surfaces

☒ Auto Execute

Surface List

PROJECT 6, PART 1, COUPLED ADVECTION

Create bottom and top
of channel

Geometry

Action:

Object:

Method:

Surface ID List

☐ Manifold

Manifold Surface

☒ Auto Execute

Surface Vertex 1 List

Surface Vertex 2 List

Surface Vertex 3 List

Surface Vertex 4 List

Geometry

Action:

Object:

Method:

Surface ID List

☐ Manifold

Manifold Surface

☒ Auto Execute

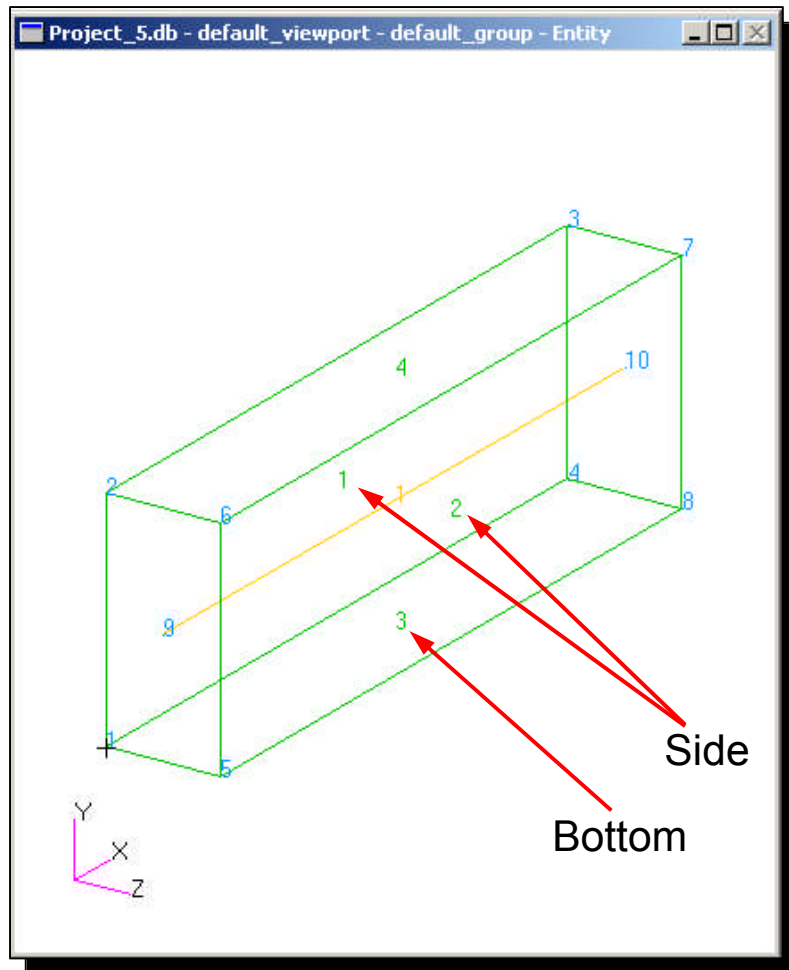
Surface Vertex 1 List

Surface Vertex 2 List

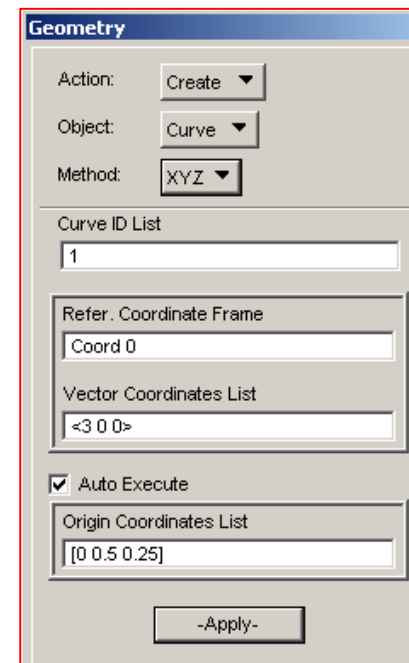
Surface Vertex 3 List

Surface Vertex 4 List

PROJECT 6, PART 1, COUPLED ADVECTION



- After generating four surfaces (Surface 1:4), create a curve at the center of the channel (Curve 1) where the fluid elements will be created.
 - CHBDYP, FTUBE element



PROJECT 6, PART 1, COUPLED ADVECTION

- Create the 2D elements that will be used for in-plane conduction and out-of-plane convection.
- IsoMesh the four surfaces with a coarse mesh.
 - Quad4 topology
 - Surface List: Surface 1:4
 - GEL = 0.5
- Connect the adjacent Quad4 elements.

Finite Elements

Action:

Object:

Type:

Output ID List

Node

Element

Elem Shape

Mesher

Topology

Surface List

Global Edge Length

☐ Automatic Calculation

Value

Finite Elements

Action:

Object:

Method:

Nodes to be excluded

Equivalencing Tolerance

PROJECT 6, PART 1, COUPLED ADVECTION

- Create the fluid elements at the center of the channel.
 - CHBDY(FTUBE)
- IsoMesh the curve at the center of the channel.
 - Bar2 topology
 - Curve List: Curve 1
 - GEL = 0.5

Finite Elements

Action: Create
Object: Mesh
Type: Curve

Output ID List
Node: 43
Element: 37

Topology: Bar2
Node Coordinate Frames...

Curve List
Curve 1

Global Edge Length
☐ Automatic Calculation
Value: 0.5

-Apply-

PROJECT 6, PART 1, COUPLED ADVECTION

- Create the material property set for the heat sink (aluminum duct).
 - $K = 4.0$

The image shows two overlapping dialog boxes from the MSC Software interface. The 'Materials' dialog box is in the background, and the 'Input Options' dialog box is in the foreground.

Materials Dialog Box:

- Action:** Create (dropdown)
- Object:** Isotropic (dropdown)
- Method:** Manual Input
- Existing Materials:** (empty list box)
- Material Name:** alum
- Description:** Date: 24-Nov-02 Time: 21:45:08
- Buttons:** Input Properties ..., Change Material Status ..., Apply

Input Options Dialog Box:

- Constitutive Model:** Solid properties (dropdown)
- Property Name:** Thermal Conductivity =, Specific Heat =, Density =
- Value:** 4 (for Thermal Conductivity)
- Temperature Dep/Model Variable Fields:** (empty list box)
- Current Constitutive Models:** (empty list box)
- Buttons:** OK, Clear, Cancel

PROJECT 6, PART 1, COUPLED ADVECTION

- Create the material property set for the air flowing in the channel.
 - K
 - Cp
 - Rho
 - Mu

Input Options

Constitutive Model: Fluid properties

Property Name	Value
Thermal Conductivity =	6.66e-4
Specific Heat =	456.2
Density =	5.01e-5
Dynamic Viscosity =	1.03e-6

Temperature Dep/Model Variable Fields:

Current Constitutive Models:

OK Clear Cancel

PROJECT 6, PART 1, COUPLED ADVECTION

- Create 2D element properties for the sides of the channel.
 - Aluminum
 - Thickness = 0.05
 - Surface 1, 2
- Create 2D element properties for the bottom and top of the channel.
 - Aluminum
 - Thickness = 0.10
 - Surface 3, 4

The image shows two overlapping dialog boxes from the MSC Software interface. The 'Element Properties' dialog is in the background, and the 'Input Properties' dialog is in the foreground.

Element Properties Dialog:

- Action: Create
- Object: 2D
- Type: Shell
- Existing Property Sets: (Empty list)
- Property Set Name: side
- Option(s): Standard Formulation
- Input Properties: (Button)
- Application Region: Select Members (Empty list)
- Add: (Button)
- Application Region: Surface 1 2
- Apply- (Button)

Input Properties Dialog:

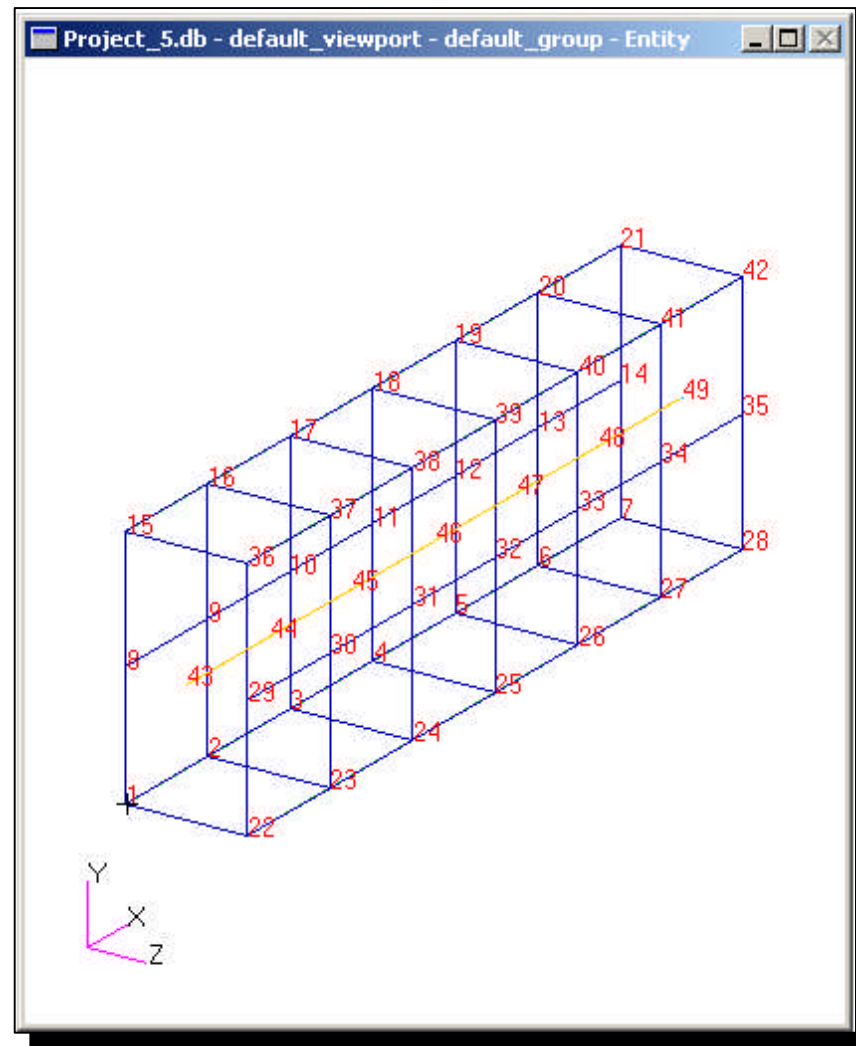
Stan. Homogeneous Plate(CQUAD4)

Property Name	Value	Value Type
Material Name	m.alum	Mat Prop Name
[Material Orientation]		CID
Thickness	0.05	Real Scalar

Field Definitions: (Empty list)

OK (Button)

PROJECT 6, PART 1, COUPLED ADVECTION



NAS104, Section 4, September 2008

Copyright© 2008 MSC.Software Corporation

S4-100

PROJECT 6, PART 1, COUPLED ADVECTION

- Create fixed temperature of 20 C at inlet to channel.
 - Apply at Point 9 on one end of Curve 1.

The image shows two overlapping dialog boxes from the MSC Software interface. The 'Load/Boundary Conditions' dialog box is in the background, and the 'Select Application Region' dialog box is in the foreground.

Load/Boundary Conditions Dialog:

- Action:** Create
- Object:** Temp (Thermal)
- Type:** Nodal
- Current Load Case:** Default...
- Type:** Static
- Existing Sets:** (Empty list)
- New Set Name:** inlet
- Buttons:** Input Data..., Select Application Region..., -Apply-

Select Application Region Dialog:

- Geometry Filter:** ☒ Geometry, ☐ FEM
- Application Region:** Select Geometry Entities, Point 9
- Buttons:** Add, Remove, OK

PROJECT 6, PART 1, COUPLED ADVECTION

- Create coupled advection between 2D shell elements and 1D flow tube elements.
 - Convection Coefficient
 - Mass flow rate

The screenshot displays two overlapping dialog boxes from the MSC Software interface. The 'Load/Boundary Conditions' dialog is in the background, and the 'Input Data' dialog is in the foreground.

Load/Boundary Conditions Dialog:

- Action: Create
- Object: Convection
- Type: Element Uniform
- Option: Coupled Advection
- Current Load Case: Default...
- Type: Static
- Existing Sets: (Empty list)
- New Set Name: advection
- Target Element Type: 2D
- Region 2: 1D
- Buttons: Input Data..., Select Application Region..., -Apply-

Input Data Dialog:

- Surface Option: Top
- Form Type: Basic
- Top Surf Convection Coef: 0.3
- Mass Flow Rate: 0.008333
- * Temperature Function: (Empty text field)
- Buttons: OK, Reset, Cancel

PROJECT 6, PART 1, COUPLED ADVECTION

- There are two regions that have to be specified.
 - Application Region (Target Element Type: 2D)
 - 2D conduction elements
 - Companion Region (Region 2: 1D)
 - 1D Flow Tube elements

The dialog box is titled "Select Application Region". It has a "Geometry Filter" section with radio buttons for "Geometry" (selected) and "FEM". Below this is an "Order:" dropdown menu set to "Selection". The "Application Region" section contains a "Select Surfaces or Edges" list with "Surface 1:4" selected, and "Add" and "Remove" buttons. Below this is an "Application Region" list box. The "Active List" checkbox is checked. The "Companion Region" section contains an empty list box and an unchecked "Active List" checkbox. An "OK" button is at the bottom.

The dialog box is titled "Select Application Region". It has a "Geometry Filter" section with radio buttons for "Geometry" (selected) and "FEM". Below this is an "Order:" dropdown menu set to "Selection". The "Application Region" section contains a "Select Curves" list with "Curve 1" selected, and "Add" and "Remove" buttons. Below this is an "Application Region" list box containing "Surface 1:4". The "Active List" checkbox is unchecked. The "Companion Region" section contains an empty list box and a checked "Active List" checkbox. An "OK" button is at the bottom.

PROJECT 6, PART 1, COUPLED ADVECTION

- Apply a constant heat flux to one of the side surfaces.
 - Heat Flux = 20 Watts/in**2
 - Apply to Surface 1

The image shows two overlapping dialog boxes from the MSC Software interface. The 'Load/Boundary Conditions' dialog is in the background, and the 'Select Application Region' dialog is in the foreground.

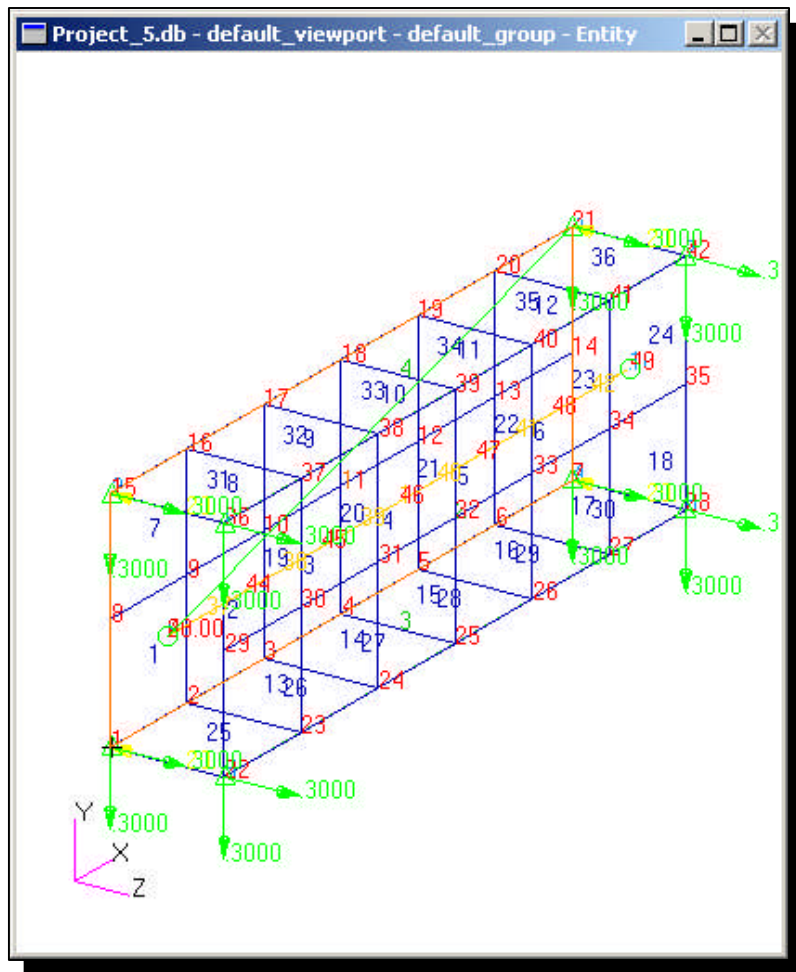
Load/Boundary Conditions Dialog:

- Action:** Create
- Object:** Applied Heat
- Type:** Element Uniform
- Option:** Normal Fluxes
- Current Load Case:** Default...
- Type:** Static
- Existing Sets:** (Empty list)
- New Set Name:** flux
- Target Element Type:** 2D
- Buttons:** Input Data..., Select Application Region..., -Apply-

Select Application Region Dialog:

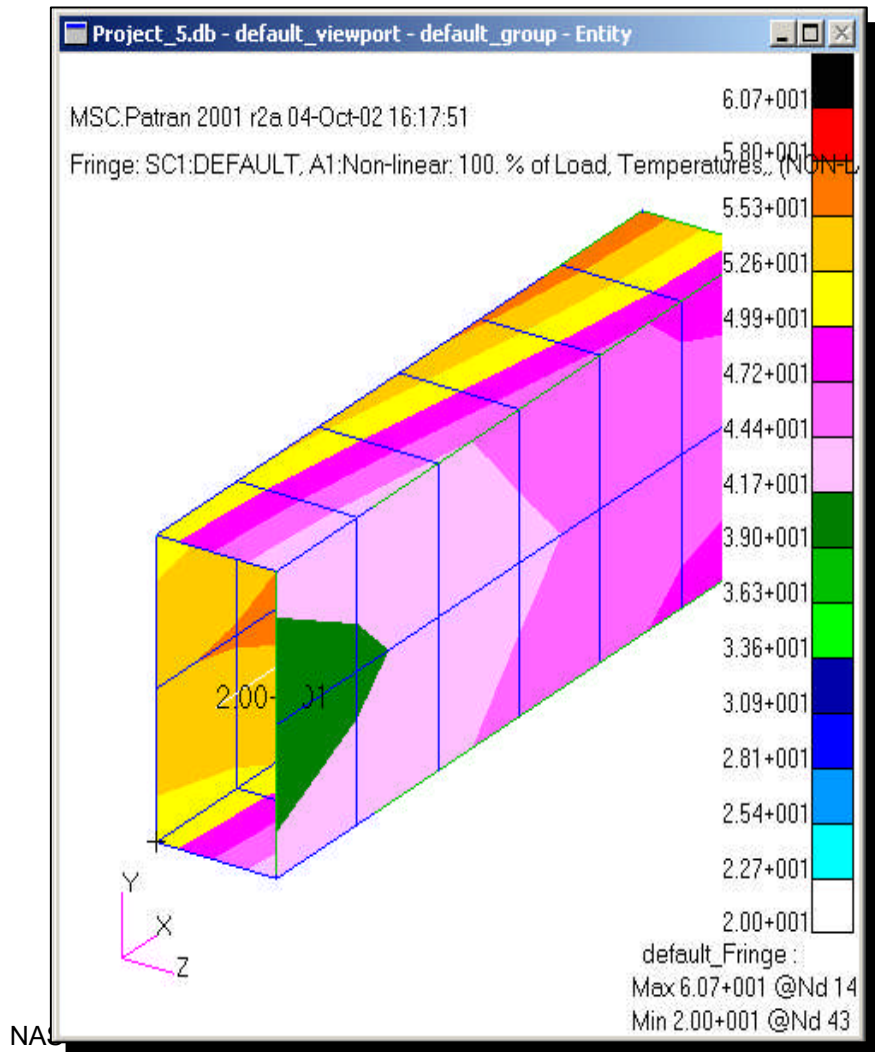
- Geometry Filter:** ☒ Geometry, ☐ FEM
- Application Region:** Select Surfaces or Edges, Surface 1
- Buttons:** Add, Remove, OK

PROJECT 6, PART 1, COUPLED ADVECTION



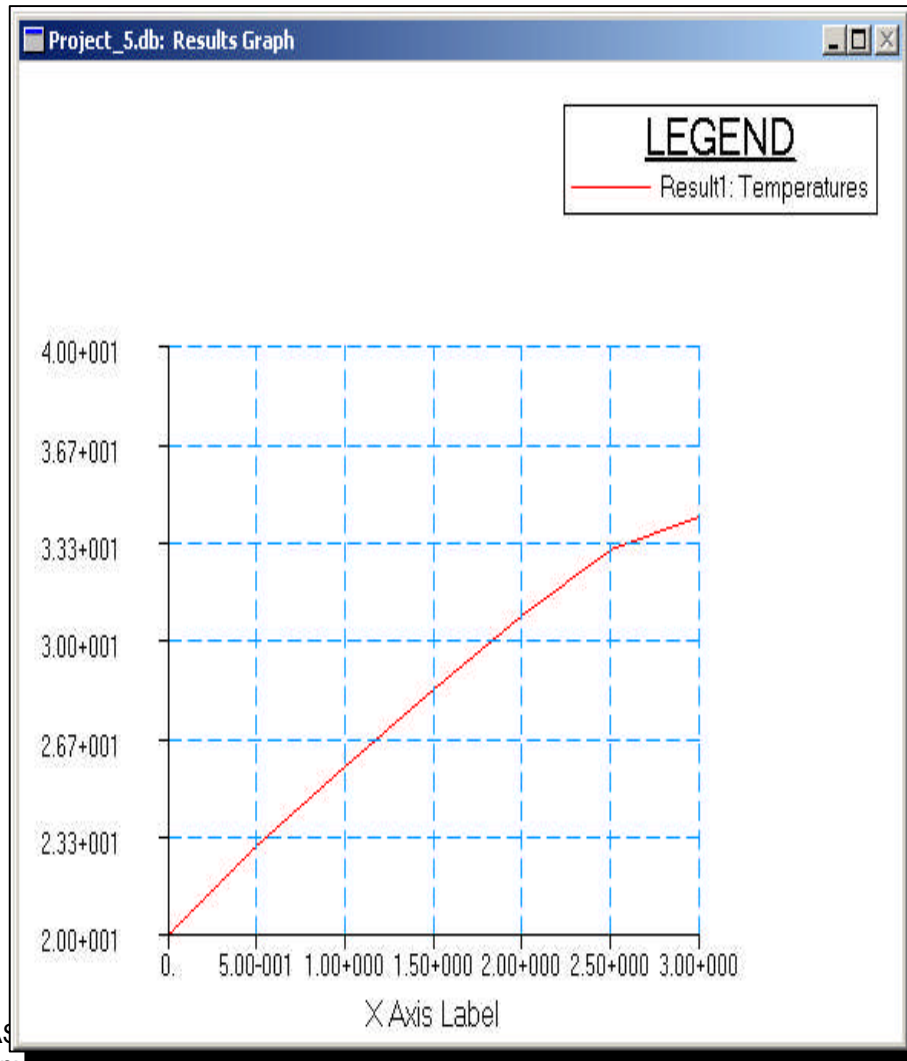
- The picture shows the markers for coupled advection as green arrows, and for the flux as yellow arrows.

PROJECT 6, PART 1, COUPLED ADVECTION



- MD Nastran thermal results
- Temperature fringe
- The temperature contour is not smooth because there were not many conduction elements and not many corresponding flow tube elements.

PROJECT 6, PART 1, COUPLED ADVECTION



- The fluid temperature rises from 20 degrees (at the inlet) to 34 degrees (at the outlet) because of the heat load.

PROJECT 6, PART 1, COUPLED ADVECTION

NASTRAN INPUT FILE -- project5.bdf

```

SOL 153
TIME 600
$ Direct Text Input for Executive Control
CEND
ANALYSIS = HEAT
TITLE = MSC/NASTRAN job created on 06-Jul-98 at 18:21:56
ECHO = SORT
MAXLINES = 999999999
TEMPERATURE(INITIAL) = 1
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 1
  LOAD = 2
  THERMAL(SORT1,PRINT)=ALL
  FLUX(SORT1,PRINT)=ALL
  SPCF=ALL
BEGIN BULK
PARAM POST 0
PARAM AUTOSPC YES
NLPARM 1 0 AUTO 5 25 PW NO + A
+ A .001 1.-7
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : heatsink
PSHELL 1 1 .1
CQUAD4 1 1 1 2 9 8
CQUAD4 2 1 2 3 10 9
CQUAD4 3 1 3 4 11 10
CQUAD4 4 1 4 5 12 11
CQUAD4 5 1 5 6 13 12
CQUAD4 6 1 6 7 14 13
CQUAD4 7 1 8 9 16 15
CQUAD4 8 1 9 10 17 16
CQUAD4 9 1 10 11 18 17
CQUAD4 10 1 11 12 19 18

```

```

CQUAD4 11 1 12 13 20 19
CQUAD4 12 1 13 14 21 20
CQUAD4 13 1 22 23 30 29
CQUAD4 14 1 23 24 31 30
CQUAD4 15 1 24 25 32 31
CQUAD4 16 1 25 26 33 32
CQUAD4 17 1 26 27 34 33
CQUAD4 18 1 27 28 35 34
CQUAD4 19 1 29 30 37 36
CQUAD4 20 1 30 31 38 37
CQUAD4 21 1 31 32 39 38
CQUAD4 22 1 32 33 40 39
CQUAD4 23 1 33 34 41 40
CQUAD4 24 1 34 35 42 41
$ Elements and Element Properties for region : side
PSHELL 3 1 .05
CQUAD4 25 3 22 1 2 23
CQUAD4 26 3 23 2 3 24
CQUAD4 27 3 24 3 4 25
CQUAD4 28 3 25 4 5 26
CQUAD4 29 3 26 5 6 27
CQUAD4 30 3 27 6 7 28
CQUAD4 31 3 36 15 16 37
CQUAD4 32 3 37 16 17 38
CQUAD4 33 3 38 17 18 39
CQUAD4 34 3 39 18 19 40
CQUAD4 35 3 40 19 20 41
CQUAD4 36 3 41 20 21 42
MAT4 1 4.
$ Material Record : air
$ Description of Material : Date: 06-Jul-98 Time: 17:57:13
MAT4 2 6.66-4 456.200 5.01-5 1.03-6
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
GRID 2 .5 0. 0.
GRID 3 1. 0. 0.
GRID 4 1.5 0. 0.
GRID 5 2. 0. 0.
GRID 6 2.5 0. 0.
GRID 7 3. 0. 0.
GRID 8 0. .5 0.
GRID 9 .5 .5 0.
GRID 10 1. .5 0.

```

PROJECT 6, PART 1, COUPLED ADVECTION

NASTRAN INPUT FILE -- project5.bdf

```

GRID 11      1.5 .5 0.
GRID 12      2. .5 0.
GRID 13      2.5 .5 0.
GRID 14      3. .5 0.
GRID 15      0. 1. 0.
GRID 16      .5 1. 0.
GRID 17      1. 1. 0.
GRID 18      1.5 1. 0.
GRID 19      2. 1. 0.
GRID 20      2.5 1. 0.
GRID 21      3. 1. 0.
GRID 22      0. 0. .5
GRID 23      .5 0. .5
GRID 24      1. 0. .5
GRID 25      1.5 0. .5
GRID 26      2. 0. .5
GRID 27      2.5 0. .5
GRID 28      3. 0. .5
GRID 29      0. .5 .5
GRID 30      .5 .5 .5
GRID 31      1. .5 .5
GRID 32      1.5 .5 .5
GRID 33      2. .5 .5
GRID 34      2.5 .5 .5
GRID 35      3. .5 .5
GRID 36      0. 1. .5
GRID 37      .5 1. .5
GRID 38      1. 1. .5
GRID 39      1.5 1. .5
GRID 40      2. 1. .5
GRID 41      2.5 1. .5
GRID 42      3. 1. .5
GRID 43      0. .5 .25
GRID 44      .5 .5 .25
GRID 45      1. .5 .25
GRID 46      1.5 .5 .25
GRID 47      2. .5 .25
GRID 48      2.5 .5 .25
GRID 49      3. .5 .25

```

\$ Loads for Load Case : Default

\$ Fixed Temperatures of Load Set : inlet

SPC 1 43 1 20.

\$ Fixed Temperatures of Load Set : advection

SPC 1 50 1 0.

SPC 1 51 1 .008333

\$ Normal Heat Flux of Load Set : flux

QBDY3 2 20. 1

QBDY3 2 20. 2

QBDY3 2 20. 3

QBDY3 2 20. 4

QBDY3 2 20. 5

QBDY3 2 20. 6

QBDY3 2 20. 7

QBDY3 2 20. 8

QBDY3 2 20. 9

QBDY3 2 20. 10

QBDY3 2 20. 11

QBDY3 2 20. 12

\$ Coupled Advection of Load Set : advection

PCONV 1 1001 0 0.

CONV 1 1 0 0 43 44 44 43

CONV 2 1 0 0 44 45 45 44

CONV 3 1 0 0 45 46 46 45

CONV 4 1 0 0 46 47 47 46

CONV 5 1 0 0 47 48 48 47

CONV 6 1 0 0 48 49 49 48

CONV 7 1 0 0 43 44 44 43

CONV 8 1 0 0 44 45 45 44

CONV 9 1 0 0 45 46 46 45

CONV 10 1 0 0 46 47 47 46

CONV 11 1 0 0 47 48 48 47

CONV 12 1 0 0 48 49 49 48

CONV 13 1 0 0 43 44 44 43

CONV 14 1 0 0 44 45 45 44

CONV 15 1 0 0 45 46 46 45

CONV 16 1 0 0 46 47 47 46

CONV 17 1 0 0 47 48 48 47

CONV 18 1 0 0 48 49 49 48

CONV 19 1 0 0 43 44 44 43

CONV 20 1 0 0 44 45 45 44

CONV 21 1 0 0 45 46 46 45

CONV 22 1 0 0 46 47 47 46

CONV 23 1 0 0 47 48 48 47

CONV 24 1 0 0 48 49 49 48

CONV 25 1 0 0 43 43 44 44

CONV 26 1 0 0 44 44 45 45

CONV 27 1 0 0 45 45 46 46

CONV 28 1 0 0 46 46 47 47

CONV 29 1 0 0 47 47 48 48

CONV 30 1 0 0 48 48 49 49

CONV 31 1 0 0 43 43 44 44

CONV 32 1 0 0 44 44 45 45

CONV 33 1 0 0 45 45 46 46

CONV 34 1 0 0 46 46 47 47

CONV 35 1 0 0 47 47 48 48

CONV 36 1 0 0 48 48 49 49

PROJECT 6, PART 1, COUPLED ADVECTION

NASTRAN INPUT FILE -- project5.bdf

```
$ Coupled Advection of Load Set : advection
PCONVM* 1      2      0      1      *   B
*   B 1.-20    0.      0.      0.
CONVM 37 1 0 51 50
CONVM 38 1 0 51 50
CONVM 39 1 0 51 50
CONVM 40 1 0 51 50
CONVM 41 1 0 51 50
CONVM 42 1 0 51 50
$ Initial Temperatures from Temperature Load Sets
TEMP 1 43 20. 50 0. 51 .008333
$ Default Initial Temperature
TEMPD 1 70.
$ CHBDYG Surface Elements
CHBDYG 1      AREA4      +   C
+   C 1 2 9 8
CHBDYG 2      AREA4      +   D
+   D 2 3 10 9
CHBDYG 3      AREA4      +   E
+   E 3 4 11 10
CHBDYG 4      AREA4      +   F
+   F 4 5 12 11
CHBDYG 5      AREA4      +   G
+   G 5 6 13 12
CHBDYG 6      AREA4      +   H
+   H 6 7 14 13
CHBDYG 7      AREA4      +   I
+   I 8 9 16 15
CHBDYG 8      AREA4      +   J
+   J 9 10 17 16
CHBDYG 9      AREA4      +   K
+   K 10 11 18 17
CHBDYG 10     AREA4      +   L
+   L 11 12 19 18
CHBDYG 11     AREA4      +   M
+   M 12 13 20 19
CHBDYG 12     AREA4      +   N
+   N 13 14 21 20
CHBDYG 13     AREA4      +   O
+   O 22 23 30 29
```

```
CHBDYG 14     AREA4      +   P
+   P 23 24 31 30
CHBDYG 15     AREA4      +   Q
+   Q 24 25 32 31
CHBDYG 16     AREA4      +   R
+   R 25 26 33 32
CHBDYG 17     AREA4      +   S
+   S 26 27 34 33
CHBDYG 18     AREA4      +   T
+   T 27 28 35 34
CHBDYG 19     AREA4      +   U
+   U 29 30 37 36
CHBDYG 20     AREA4      +   V
+   V 30 31 38 37
CHBDYG 21     AREA4      +   W
+   W 31 32 39 38
CHBDYG 22     AREA4      +   X
+   X 32 33 40 39
CHBDYG 23     AREA4      +   Y
+   Y 33 34 41 40
CHBDYG 24     AREA4      +   Z
+   Z 34 35 42 41
CHBDYG 25     AREA4      +   AA
+   AA 22 1 2 23
CHBDYG 26     AREA4      +   AB
+   AB 23 2 3 24
CHBDYG 27     AREA4      +   AC
+   AC 24 3 4 25
CHBDYG 28     AREA4      +   AD
+   AD 25 4 5 26
CHBDYG 29     AREA4      +   AE
+   AE 26 5 6 27
CHBDYG 30     AREA4      +   AF
+   AF 27 6 7 28
CHBDYG 31     AREA4      +   AG
+   AG 36 15 16 37
```

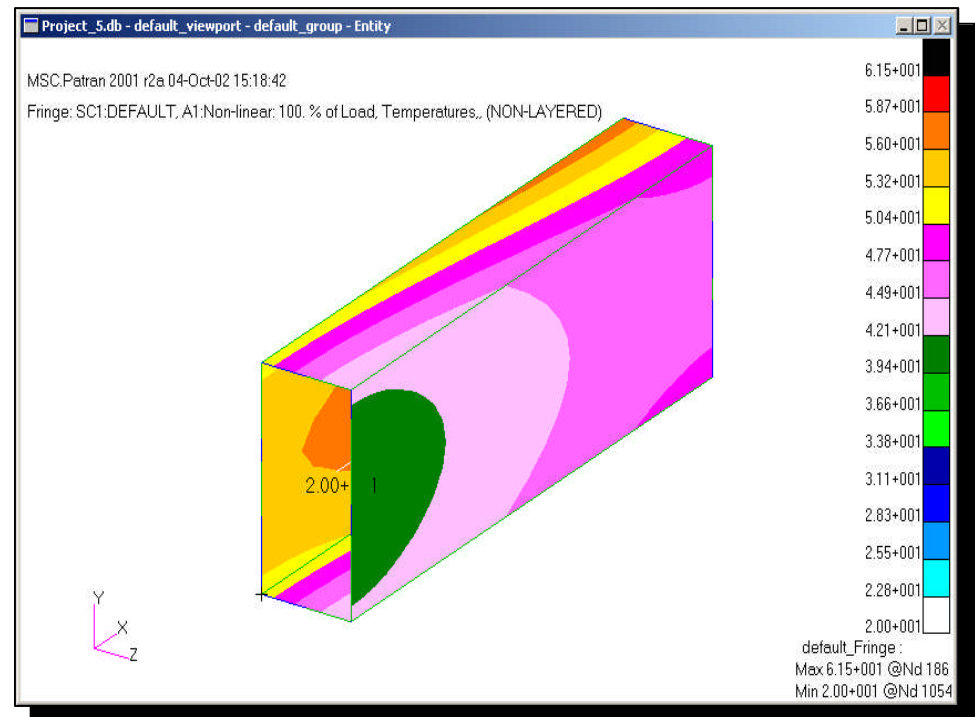
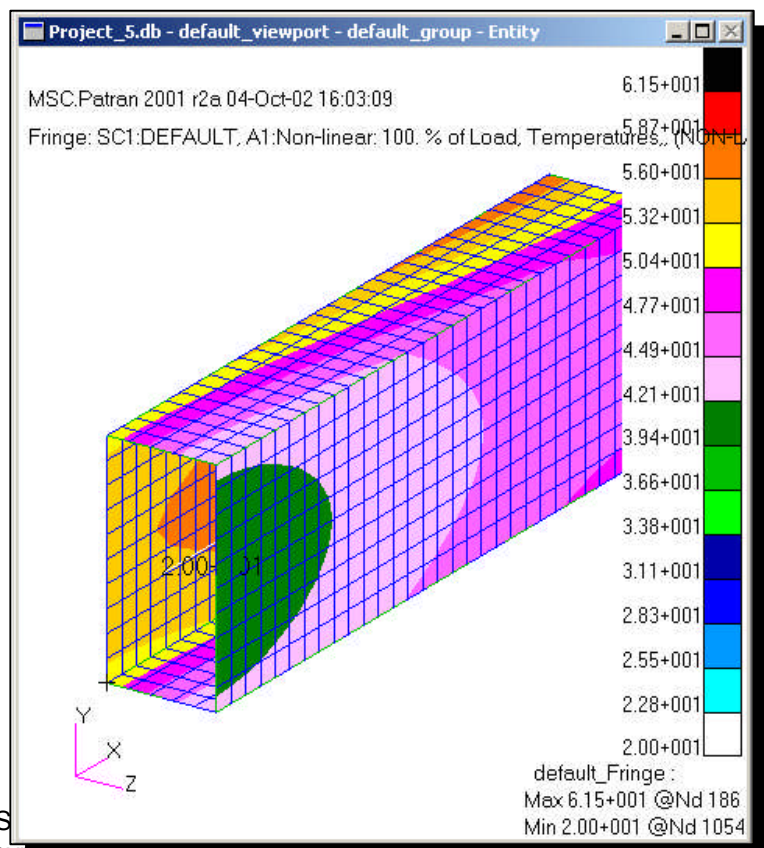
```
CHBDYG 32     AREA4      +   AH
+   AH 37 16 17 38
CHBDYG 33     AREA4      +   AI
+   AI 38 17 18 39
CHBDYG 34     AREA4      +   AJ
+   AJ 39 18 19 40
CHBDYG 35     AREA4      +   AK
+   AK 40 19 20 41
CHBDYG 36     AREA4      +   AL
+   AL 41 20 21 42
$ CHBDYP Surface Elements and Element Properties
CHBDYP 37 1 FTUBE 43 44 +   AM
+   AM .57735 .57735 .57735
CHBDYP 38 1 FTUBE 44 45 +   AN
+   AN .57735 .57735 .57735
CHBDYP 39 1 FTUBE 45 46 +   AO
+   AO .57735 .57735 .57735
CHBDYP 40 1 FTUBE 46 47 +   AP
+   AP .57735 .57735 .57735
CHBDYP 41 1 FTUBE 47 48 +   AQ
+   AQ .57735 .57735 .57735
CHBDYP 42 1 FTUBE 48 49 +   AR
+   AR .57735 .57735 .57735
PHBDY 1 .5333 .5333
$ Free Convection Heat Transfer Coefficients
MAT4 1001 .3
$ Scalar Points
SPOINT 50 THRU 51
$ Referenced Coordinate Frames
ENDDATA
```

PROJECT 6, PART 2, COUPLED ADVECTION

- Redo this problem with a smaller mesh element size of 0.1 inch.
 - 1/5 of previous element size
- Compare the results for the previous thermal analysis (coarse mesh) with those for this analysis (fine mesh).

PROJECT 6, PART 2, COUPLED ADVECTION

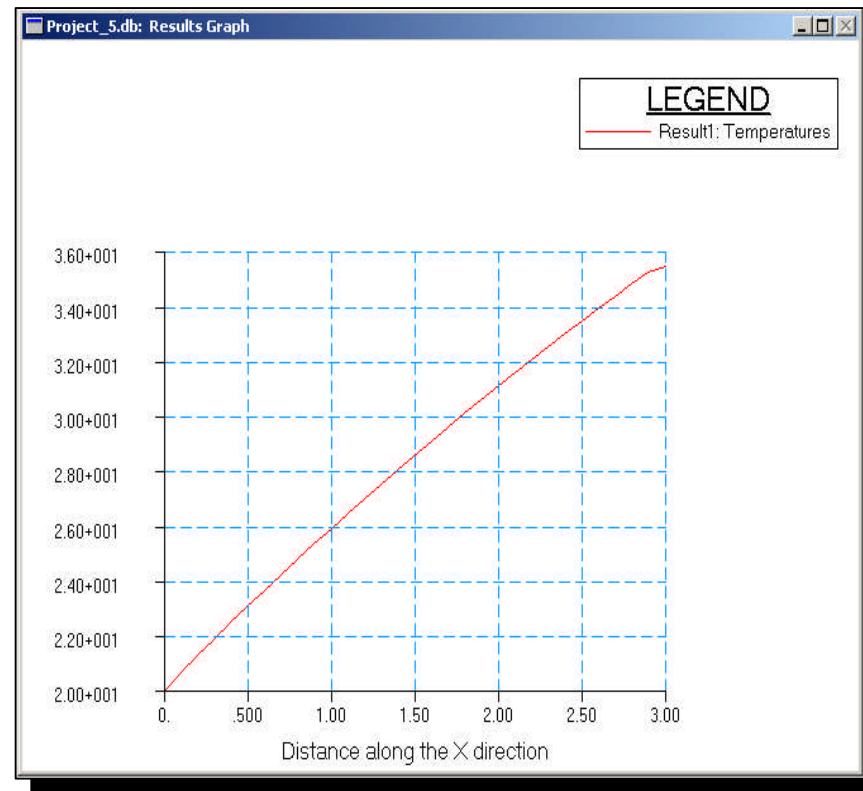
- The temperature contour is for the finer mesh model.



S4-112

PROJECT 6, PART 2, COUPLED ADVECTION

- For the finer mesh model the fluid temperature rises from 20 degrees (at the inlet) to 35.5 degrees (at the outlet).



PROJECT 6, PART 2, COUPLED ADVECTION

Extract from MD Nastran ASCII Results File

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

      T E M P E R A T U R E   V E C T O R

POINT ID.  TYPE      ID  VALUE      ID+1 VALUE      ID+2 VALUE      ID+3 VALUE      ID+4 VALUE      ID+5 VALUE
    1      S      5.153986E+01  5.210973E+01  5.326322E+01  5.463775E+01  5.595658E+01  5.696083E+01
    7      S      5.735125E+01  5.504722E+01  5.559027E+01  5.671677E+01  5.806875E+01  5.936684E+01
   13      S      6.035074E+01  6.073112E+01  5.153986E+01  5.210973E+01  5.326322E+01  5.463775E+01
   19      S      5.595658E+01  5.696083E+01  5.735125E+01  4.214294E+01  4.271282E+01  4.386631E+01
   25      S      4.524083E+01  4.655967E+01  4.756392E+01  4.795434E+01  4.074147E+01  4.128452E+01
   31      S      4.241101E+01  4.376300E+01  4.506110E+01  4.604499E+01  4.642537E+01  4.214294E+01
   37      S      4.271282E+01  4.386631E+01  4.524083E+01  4.655967E+01  4.756392E+01  4.795434E+01
   43      S      2.000000E+01  2.293754E+01  2.568529E+01  2.828691E+01  3.075201E+01  3.306177E+01
   49      S      3.417375E+01  0.0          8.333000E-03
```


PROJECT 6, PART 2, COUPLED ADVECTION

- It is seen the fluid exit temperature is 35.457 degrees

Extract from MD Nastran ASCII Results File

0	1055	S	2.000000E+01	2.063807E+01	2.126295E+01	2.187582E+01	2.247770E+01	2.306953E+01
	1061	S	2.365211E+01	2.422616E+01	2.479229E+01	2.535103E+01	2.590285E+01	2.644812E+01
	1067	S	2.698717E+01	2.752025E+01	2.804756E+01	2.856925E+01	2.908540E+01	2.959606E+01
	1073	S	3.010121E+01	3.060081E+01	3.109475E+01	3.158287E+01	3.206497E+01	3.254082E+01
	1079	S	3.301011E+01	3.347248E+01	3.392756E+01	3.437487E+01	3.481390E+01	3.524408E+01
	1085	S	3.545688E+01	0.0	8.333000E-03			

PROJECT 6, PART 2, COUPLED ADVECTION

- Energy balance

- $\Delta T = Q / (\dot{M} \cdot C_p)$

Where Q = total heat

$\dot{M}$ = mass flow rate

C_p = specific heat for the fluid

- $T_{\text{exit}} = T_{\text{in}} + Q / (\dot{M} \cdot C_p) = 20 + (20) \cdot (3) / (0.008333 \cdot 456.2) = 35.78$

PROJECT 6, PART 2, COUPLED ADVECTION

- Energy loss (transfer) by convection

Extract from MD Nastran ASCII Results File

LOAD STEP = 1.000000E+00

H E A T F L O W I N T O H B D Y E L E M E N T S (CHBDY)

ELEMENT-ID	APPLIED-LOAD	FREE-CONVECTION	FORCED-CONVECTION	RADIATION	TOTAL
1	5.000000E+00	-2.407725E+00	0.000000E+00	0.000000E+00	2.592275E+00
2	5.000000E+00	-2.258144E+00	0.000000E+00	0.000000E+00	2.741856E+00
3	5.000000E+00	-2.151414E+00	0.000000E+00	0.000000E+00	2.848586E+00



DUCT FLOW

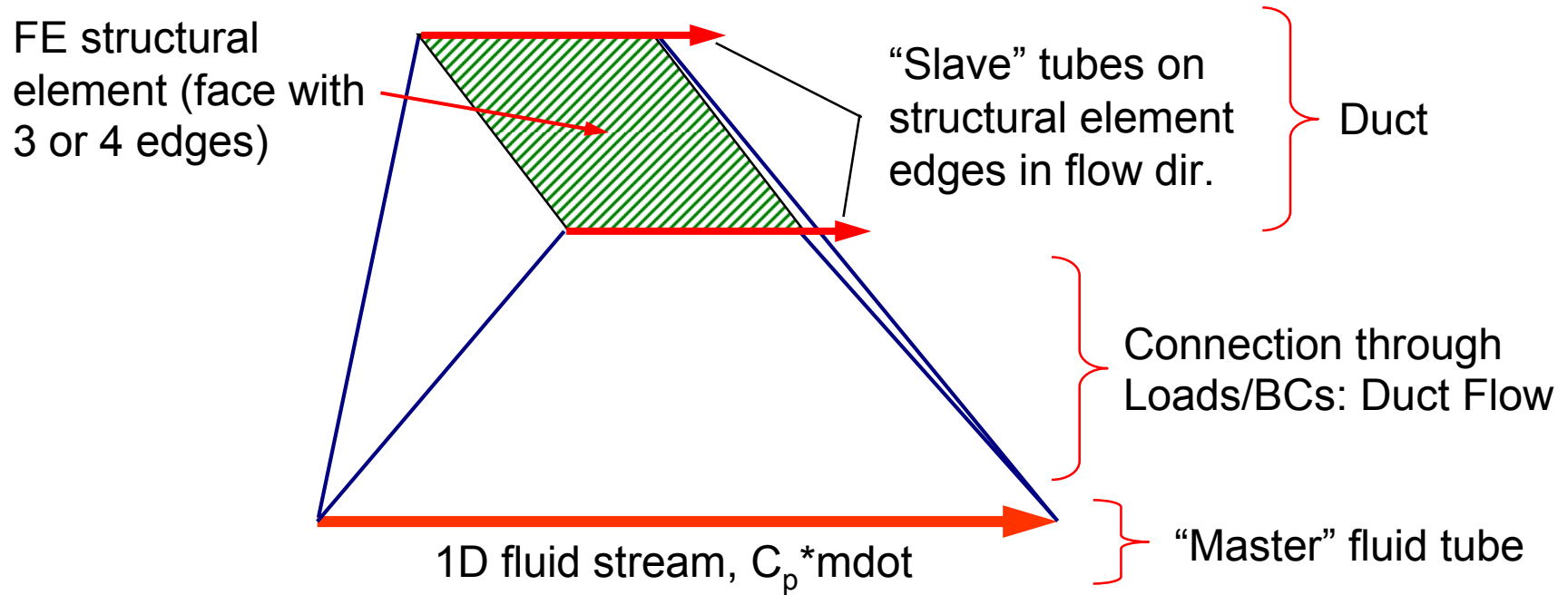




DUCT FLOW

- MD Nastran Duct Flow provides a means of coupling advective flows to FE model geometry. This is done using flow tubes. There are flow tubes for both the fluid stream and FE structure.
- The flow tubes have both forced convection and advection capability. It is necessary to disable certain flow tube capability.
- For simple convective formulations the ambient environment must be known, where as for Duct Flow the fluid temperatures are determined by the MD Nastran solver at run time.

SCHEMATIC FOR DUCT FLOW



The structural elements must be 2D or 3D

PURPOSE OF DUCT FLOW FLUID TUBE

- “Master” fluid tube carries the bulk fluid energy, $C_p \cdot \dot{m} \cdot \Delta T$ (Watt/°K).
- “Master” fluid tube is represented by the CONVM and PHBDYP entry, and is selected in MD Patran using a geometric curve or 1D Flow Tube elements.
- “Slave” fluid tube provides the convection area, and the calculation of h .
- “Slave” fluid tube (in flow direction) is represented by the CONVM and PHBDYP on FE (structure) edges, and is selected in MD Patran using geometric surfaces or solid faces, or 2D or 3D solid element faces.

PROJECT 7

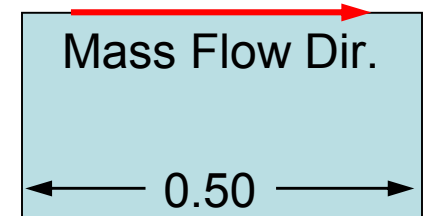
CREATE DUCT FLOW

- “Master” fluid tube only provides advection, so eliminate the forced convection part contributed by the “master” tube
- “Slave” fluid tubes only provide forced convection, so eliminate the advection part contributed by the “slave” tubes
- Use the same hydraulic diameter for both the “master” and “slave” tubes because it is desired to use the same Reynold’s number for the empirical relationship
- Follow the steps MD Patran uses to create a Duct Flow model

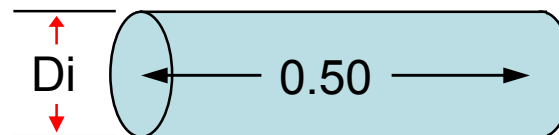
PROJECT 7, ELIMINATE ADVECTION FOR SLAVE TUBES

- From convection area determine the diameter of equivalent flow tube for duct
- The convection area is equal to the area of an element
 - The length of an element, in the mass flow direction, is 3 inches/6 = 0.5 inch
 - $\frac{1}{2}$ of the conduction element width (normal to the flow direction) is used as the flow tube circumference, $0.50/2 = 0.25$

Circumference = 0.25



- Diameter is found from $\pi * D_i * L = 0.25 * 0.50$
 - $D_i = 0.079577$



PROJECT 7, ELIMINATE ADVECTION FOR SLAVE TUBES (Cont.)

- Use the value of this diameter, D_i , for the Flow Tube coefficient term
$$Nu = 0.023 \cdot Re^{0.8} \cdot Pr^{0.4}$$
- The diameter that was written to the PHBDY bulk data record for Project 6, Part 1 is 0.5333 inch.
- Because of fixed geometry, cannot change the PHBDY diameters D_1 and D_2 , thus in order to obtain the correct surface area it is necessary to modify the coefficient.
- A new coefficient is obtained by starting by multiplying the original coefficient by the ratio D_i/D_p , where D_i = diameter of the cylinder, and D_p = diameter that was written to the PHBDY bulk data record for Project 6, Part 1 for the “slave” tube.
- For $D_i = 0.079577$ and $D_p = 0.5333$
$$coef\_1 = 0.023 \cdot (D_i/D_p) = 0.023 \cdot (0.079577/0.5333) = 0.003432.$$

PROJECT 7, ELIMINATE ADVECTION FOR SLAVE TUBES (Cont.)

- Remove the advection part contributed by the slave tubes.
- Eliminate of the advection (energy transport) from the slave flow tubes by dividing C_p (specific heat) by a factor of 1000000. In order to match the exact Prandtl number, it is necessary to divide the thermal conductivity by a factor of 1000000. This means that the coefficient must be multiplied by a factor of 1000000.
- Consequently, it is seen that $\text{coef\_2} = 1000000 * 0.003432 = 3432$
- There are actual fin areas that were not modeled. Assuming the total fin surface area is 3 times the outside surface area, multiply the coefficient by a factor of 3.

$$\text{coef} = \text{coef\_3} = 3432 * 3 = 10296.0$$

- It is seen that this number is exactly the coefficient that MD Nastran writes to the PCONVM bulk data record for the slave tubes
PCONVM, 3, 2, 1, 1, 10296.0, 0.8, 0.4, 0.4

PROJECT 7, ELIMINATE CONVECTION FOR MASTER TUBE

- Eliminate the force convection component from the “master” fluid tube.
- Can divide the coefficient 0.023 by a factor of 1000000
=> 2.3E-8, or use 1.0E-20
- This can be seen on the following bulk data entry
PCONVM, 1, 1, 0, 1, 1.0E-20, 0., 0., 0.

PROJECT 7, HYDRAULIC DIAMETER AND REYNOLD'S NUMBER

- Use the same hydraulic diameter for both the “master” and “slave” tubes because it is desired to use the same Reynold's number for the empirical relationship.

$$Re = 4 \cdot \dot{m} / (\pi \cdot d \cdot \mu)$$

$$Re = (4 \cdot 8.333E-3) / (3.14159 \cdot 0.5333 \cdot 1.03E-6) = 1.9326E4$$

Thus, this is a turbulent flow

$$Pr = C_p \cdot \mu / k = 456.2 \cdot 1.03E-6 / 6.66E-4 = 0.7055$$

$$Nu = 0.023 \cdot Re^{0.8} \cdot Pr^{0.4} = 53.71$$

$$Nu = h \cdot d / k$$

$$h = Nu \cdot k / d = 53.71 \cdot 6.66E-4 / 0.5333 = 0.067 \text{ Watt}/(\text{inch}^2 \cdot ^\circ\text{C})$$

PROJECT 7, DUCT FLOW IN PATRAN

Load/Boundary Conditions

Action:

Object:

Type:

Option:

Current Load Case:

Type: Static

Existing Sets

New Set Name

Target Element Type:

Region 2:

Input Data

Mass Flow Rate

Heat Transfer Coefficient

Extended Surface Multiplier

Formula Type Option
☐ $h = \text{coef} * \text{Re}^{**} \text{Expr} * \text{Pr}^{**} \text{Expp}$
☒ $h = k/d * \text{coef} * \text{Re}^{**} \text{Expr} * \text{Pr}^{**}$

Reference Temperature Option
☒ Average Temp $(T_s + T_a)/2$
☐ Surface Temp (T_s)
☐ Ambient Temp (T_a)
☐ Film Temperature

Reynolds Exponent

Prandtl Exponent, Heat In

Prandtl Exponent, Heat Out

Select Application Region

Geometry Filter
☒ Geometry
☐ FEM

Order:

Application Region

Application Region

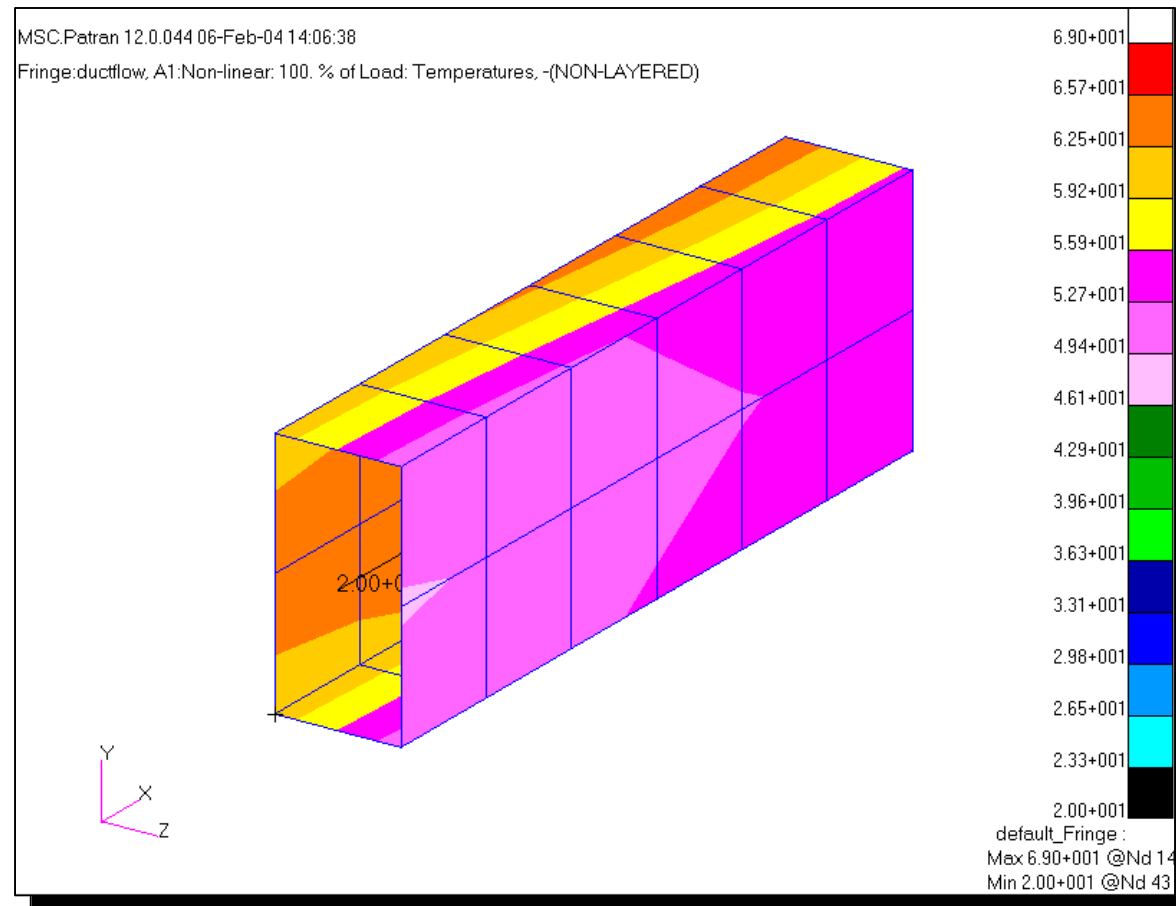
☐ Active List

Companion Region

☒ Active List

PROJECT 7, DUCT FLOW RESULTS

NASTRAN INPUT FILE : p5_duct.bdf



NAS104, Section 4, September 2008

Copyright© 2008 MSC.Software Corporation

S4-131



PROJECT 7, INPUT FILE DUCT FLOW

```
$ Duct Flow of Load Set : ductflow
PCONVM 1 2 0 1 1.-20 0. 0. 0.
CONVM 100013 1 0 51 50
CONVM 100014 1 0 51 50
CONVM 100015 1 0 51 50
CONVM 100016 1 0 51 50
CONVM 100017 1 0 51 50
CONVM 100018 1 0 51 50
PCONVM 2 100002 1 1 10296.8 .3 .3
CONVM 100013 2 0 51 1 2
CONVM 100013 2 0 51 8 9
CONVM 100014 2 0 51 2 3
CONVM 100014 2 0 51 9 10
CONVM 100015 2 0 51 3 4
CONVM 100015 2 0 51 10 11
CONVM 100016 2 0 51 4 5
CONVM 100016 2 0 51 11 12
CONVM 100017 2 0 51 5 6
CONVM 100017 2 0 51 12 13
CONVM 100018 2 0 51 6 7
CONVM 100018 2 0 51 13 14
CONVM 100013 2 0 51 8 9
CONVM 100013 2 0 51 15 16
CONVM 100014 2 0 51 9 10
CONVM 100014 2 0 51 16 17
CONVM 100015 2 0 51 10 11
CONVM 100015 2 0 51 17 18
CONVM 100016 2 0 51 11 12
CONVM 100016 2 0 51 18 19
CONVM 100017 2 0 51 12 13
CONVM 100017 2 0 51 19 20
CONVM 100018 2 0 51 13 14
CONVM 100018 2 0 51 20 21
CONVM 100013 2 0 51 22 23
CONVM 100013 2 0 51 29 30
CONVM 100014 2 0 51 23 24
CONVM 100014 2 0 51 30 31
CONVM 100015 2 0 51 24 25
CONVM 100015 2 0 51 31 32
CONVM 100016 2 0 51 25 26
CONVM 100016 2 0 51 32 33
```

```
CONVM 100017 2 0 51 26 27
CONVM 100017 2 0 51 33 34
CONVM 100018 2 0 51 27 28
CONVM 100018 2 0 51 34 35
CONVM 100013 2 0 51 29 30
CONVM 100013 2 0 51 36 37
CONVM 100014 2 0 51 30 31
CONVM 100014 2 0 51 37 38
CONVM 100015 2 0 51 31 32
CONVM 100015 2 0 51 38 39
CONVM 100016 2 0 51 32 33
CONVM 100016 2 0 51 39 40
CONVM 100017 2 0 51 33 34
CONVM 100017 2 0 51 40 41
CONVM 100018 2 0 51 34 35
CONVM 100018 2 0 51 41 42
CONVM 100013 2 0 51 22 23
CONVM 100013 2 0 51 1 2
CONVM 100014 2 0 51 23 24
CONVM 100014 2 0 51 2 3
CONVM 100015 2 0 51 24 25
CONVM 100015 2 0 51 3 4
CONVM 100016 2 0 51 25 26
CONVM 100016 2 0 51 4 5
CONVM 100017 2 0 51 26 27
CONVM 100017 2 0 51 5 6
CONVM 100018 2 0 51 27 28
CONVM 100018 2 0 51 6 7
CONVM 100013 2 0 51 36 37
CONVM 100013 2 0 51 15 16
CONVM 100014 2 0 51 37 38
CONVM 100014 2 0 51 16 17
CONVM 100015 2 0 51 38 39
CONVM 100015 2 0 51 17 18
CONVM 100016 2 0 51 39 40
CONVM 100016 2 0 51 18 19
CONVM 100017 2 0 51 40 41
CONVM 100017 2 0 51 19 20
CONVM 100018 2 0 51 41 42
CONVM 100018 2 0 51 20 21
$ Initial Temperatures from Temperature Load Sets
TEMP* 1 43 20. 50
* 0. 51 .008333
$ Default Initial Temperature
TEMPD 1 90.
```

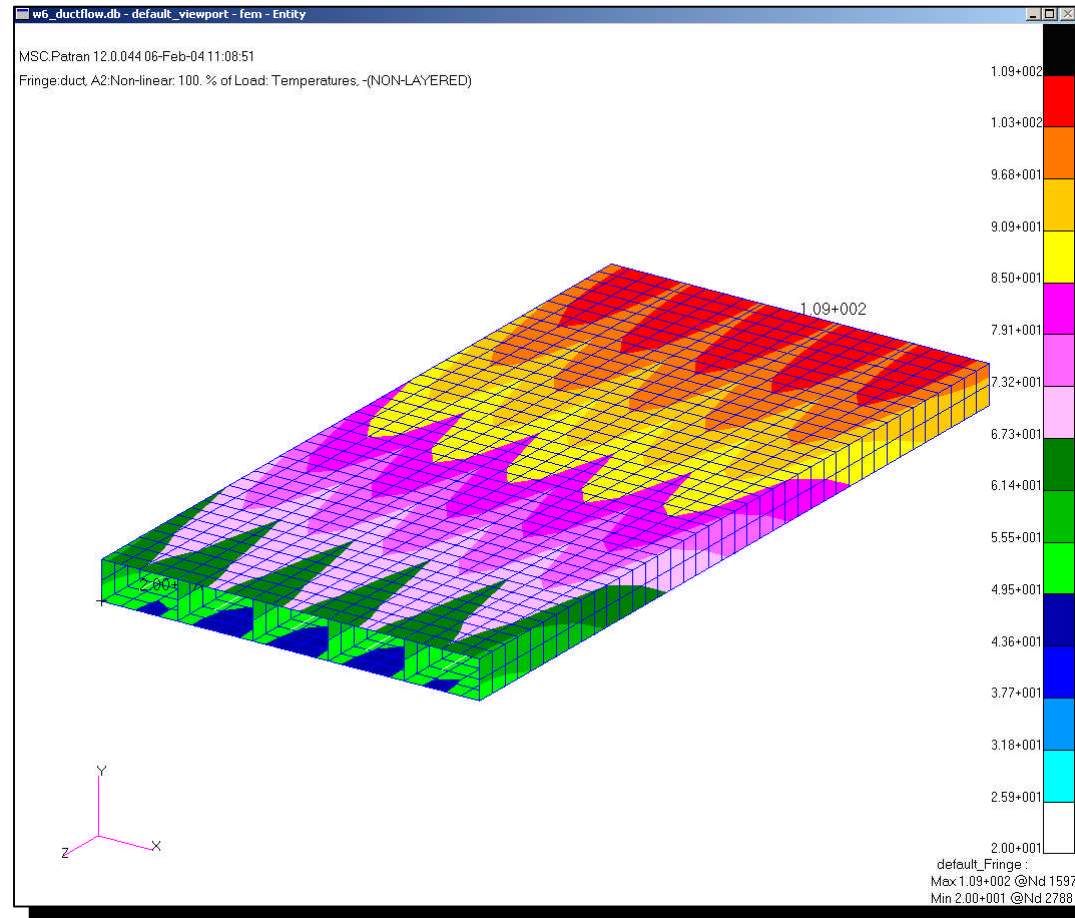
PROJECT 7, INPUT FILE DUCT FLOW (Cont.)

```
$ CHBDYG Surface Elements
CHBDYG 100001 AREA4
  1  2  9  8
CHBDYG 100002 AREA4
  2  3 10  9
CHBDYG 100003 AREA4
  3  4 11 10
CHBDYG 100004 AREA4
  4  5 12 11
CHBDYG 100005 AREA4
  5  6 13 12
CHBDYG 100006 AREA4
  6  7 14 13
CHBDYG 100007 AREA4
  8  9 16 15
CHBDYG 100008 AREA4
  9 10 17 16
CHBDYG 100009 AREA4
 10 11 18 17
CHBDYG 100010 AREA4
 11 12 19 18
CHBDYG 100011 AREA4
 12 13 20 19
CHBDYG 100012 AREA4
 13 14 21 20
```

```
$ CHBDYP Surface Elements and Element Properties
CHBDYP 100013 1 FTUBE 43 44
               .57735 .57735 .57735
CHBDYP 100014 1 FTUBE 44 45
               .57735 .57735 .57735
CHBDYP 100015 1 FTUBE 45 46
               .57735 .57735 .57735
CHBDYP 100016 1 FTUBE 46 47
               .57735 .57735 .57735
CHBDYP 100017 1 FTUBE 47 48
               .57735 .57735 .57735
CHBDYP 100018 1 FTUBE 48 49
               .57735 .57735 .57735
PHBDY 1 .5333 .5333
$ Corrected Material Properties of Flow Tubes
MAT4 100002 6.66-10 4.562-4 5.01-5 1.03-6
$ Scalar Points
SPOINT 50 THRU 51
$ Referenced Coordinate Frames
ENDDATA 0bc93788
```

PROJECT 7, DUCT FLOW RESULTS

NASTRAN INPUT FILE : w6_ductflow.bdf



DIFFERENCE BETWEEN DUCT FLOW AND COUPLED ADVECTION

- The main difference between Duct Flow and Coupled Advection, is that for Duct Flow the user uses the equation for h , e.g. $h = \text{coef} * \text{Re}^{**} \text{Expr} * \text{Pr}^{**} \text{Expp}$, where as for Coupled Advection the user must determine the value of h to use.
- Another difference is how the convection is applied – the application regions used
 - Duct Flow: Target Element Type (1D), Region 2 (2D, 3D)
 - Coupled Advection: Target Element Type (1D, 2D, 3D), Region 2 (1D)
- Also, for Duct Flow it is possible to specify a scale factor for multiplying the convective surface area calculated by Patran for the elements. The product is the convection area.



EXERCISES

- Perform Workshop 5 “Axisymmetric Flow in a Pipe” in the exercise workbook.
- Perform Workshop 6 “Typical Avionics Flow” in the exercise workbook.
- Perform Workshop 7 “Thermal Contact Resistance” in the exercise workbook.

SECTION 5

TRANSIENT THERMAL ANALYSIS





TRANSIENT THERMAL ANALYSIS

- Theory and capabilities
- System characteristics that cause transient behavior
 - Time dependent boundary temperature
 - Time dependent thermal storage
 - Time dependent thermal loads
 - Time dependent heat transfer coefficient
 - Time dependent ambient temperature
 - Time dependence for the flow field mass flow rate
 - Time dependence for radiation to space
 - Time dependence for radiation in enclosures
- Analysis of time dependent thermal problems

THEORY AND CAPABILITIES

- Sum all the heat flow rates into Node i

$$\sum_{j=1}^n \dot{q}_{ij} = \rho v C_p \dot{T}_i$$

- where

$\dot{q}_{ij}$ is the heat flow rate into Node i for thermal Resistor j

ρ is the density of the mass at Node i

v is the volume of the mass at Node i

C_p is the specific heat of the mass at Node i

$\dot{T}_i$ is the temperature rate of the mass at Node i

- This is the starting point for creating the transient thermal equation. The MD Nastran equation is based on FE.

THEORY AND CAPABILITIES (Cont.)

Transient Analysis

Basic Equations. The general equation solved in transient analysis has the form

$$[B]\{\dot{u}\} + [K]\{u\} + \{\mathfrak{R}\}\{u + T_{abs}\}^4 = \{P\} + \{N\} \quad \text{Eq. 4-19}$$

To take phase change into consideration, the heat diffusion equation is converted into

$$\{\dot{H}\} + [K]\{u\} + \{\mathfrak{R}\}\{u + T_{abs}\}^4 = \{P\} + \{N\} \quad \text{Eq. 4-20}$$

Note: In “ ” on page 69, H represents enthalpy, not the convection heat transfer coefficient.

where:

The equilibrium equation is solved by Newmark's method with adaptive time stepping.

$[B]$ = a heat capacity matrix

$[K]$ = a heat conduction matrix

$\{\mathfrak{R}\}$ = a radiation exchange matrix

$\{P\}$ = a vector of applied heat loads that are constant or functions of time, but not functions of temperature

$\{N\}$ = a vector of nonlinear heat loads that depend on temperature

$\{H\}$ = an enthalpy vector

$\{\dot{H}\}$ = $\{dH / dt\}$

$\{u\}$ = a vector of grid point temperature

$\{\dot{u}\}$ = $\{du / dt\}$

T_{abs} = the absolute temperature scale adjustment required for radiation heat transfer exchange or radiation boundary conditions when all other temperatures and units are specified in deg-F or deg-C.

TIME DEPENDENT TEMPERATURE

TEMPBC

Grid Point Temperatures

Defines the temperature boundary conditions for heat transfer analysis. Applies to steady-state and transient conditions.

Format:

1	2	3	4	5	6	7	8	9	10
TEMPBC	SID	TYPE	TEMP1	GID1	TEMP2	GID2	TEMP3	GID3	

Example:

TEMPBC	10	STAT	100.0	1	100.0	2	100.0	3	

Alternate Format and Example:

TEMPBC	SID	TYPE	TEMP1	GID1	"THRU"	GID2	"BY"	INC	
TEMPBC	20	STAT	100.0	4	THRU	50	BY	2	

Field	Contents
SID	Temperature set identification number. (Integer > 0)
TYPE	Type of temperature boundary condition. See Remarks. (Character; Default = "STAT"); STAT - Constant temperature boundary condition TRAN - Time-varying temperature boundary condition
TEMPi	Temperature (Real)
GIDi	Grid point identification number. (Integer>0 or "THRU" or "BY")
INC	Grid point number increment. (Integer)

TIME DEPENDENT TEMPERATURE (Cont.)

- Specify time varying ambient temperature for free convection

1	2	3	4	5	6	7	8	9	10
TEMPBC	SID	TYPE	TEMP1	GID1	TEMP2	GID2	TEMP3	GID3	
TEMPBC	300	TRAN	500.0	99					

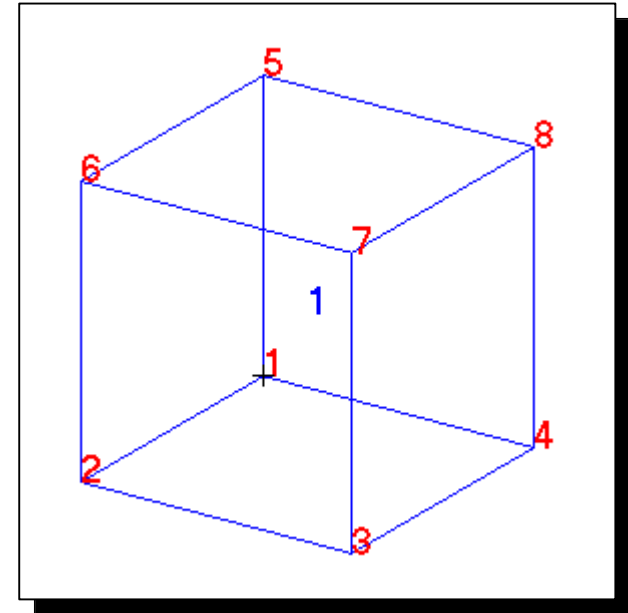
- SID – temperature set ID
- TYPE – STAT, constant temperature boundary condition
TRAN, time varying temperature boundary condition
- TEMPi – temperature factor for grid point GIDi
- GIDi – grid point ID i

```

CHEXA      1      1      1      2      3      4      5      6
           7      8

CHBDYE,10,1,6
CONV,10,5,,,99

TLOAD1,200,300,,,400
TABLED1,400,,,,,,,,+TBD1
+TBD1,0.0,0.0,1000.0,1.0,2000.0,1.0,3000,0.0,+TBD2
+TBD2,4000.0,0.0,ENDT
TEMPBC,300,TRAN,500.0,99
TEMP,20,99,0.0
TEMPD,20,0.0
    
```



NAS104, Session 3, September 2008

Copyright© 2008 MSC Software Corporation

S5-7

TIME DEPENDENT THERMAL LOADS

- QVECT – directional heat flux from distance source
- QVOL – volumetric heat load (power /volume)
- QHBDY – heat flux applied to an area defined by grid points (no CHBDYi entries required)
- QBDY1 – heat flux applied to boundary condition surface elements
- QBDY2 – heat flux applied to grid point
- QBDY3 – similar to QBDY1 except with control node
- SLOAD – power apply to a grid point

TIME DEPENDENT THERMAL LOADS (Cont.)

- Heat flux for transient loading, TLOADi, and time varying $u_{CNTRLND}$

QBDY3

Boundary Heat Flux Load for a Surface

Defines a uniform heat flux load for a boundary surface.

Format:

1	2	3	4	5	6	7	8	9	10
QBDY3	SID	Q0	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

Example:

QBDY3	2	20.0	10	1	THRU	50	BY	2	

Field	Contents
SID	Load set identification number. (Integer > 0)
Q0	Thermal heat flux load, or load multiplier. Q0 is positive for heat flow into a surface. (Real)
CNTRLND	Control point for thermal flux load. (Integer ≥ 0; Default = 0)
EIDi	CHBDYj element identification numbers. (Integer ≠ 0 or "THRU" or "BY")

TIME DEPENDENT THERMAL LOADS (Cont.)

- Heat flux for transient loading, TLOADi, and time varying u_{CNTRLND}

Remarks:

1. QBDY3 entries must be selected in Case Control (LOAD = SID) to be used in steady state. The total power into a surface is given by the equation:
 - if $\text{CNTRLND} \leq 0$ then $P_{in} = (\text{Effective area}) \cdot Q0$
 - if $\text{CNTRLND} > 0$ then $P_{in} = (\text{Effective area}) \cdot Q0 \cdot u_{\text{CNTRLND}}$where u_{CNTRLND} is the temperature of the control point and is used as a load multiplier.
2. In transient analysis, SID is referenced by a TLOADi Bulk Data entry through the EXCITEID specification. A function of time $F(t - \tau)$ defined on the TLOADi multiplies the general load, with τ specifying time delay. The load set identifier on the TLOADi entry must be selected in Case Control (DLOAD = SID) for use in transient analysis. If multiple types of transient loads exist, they must be combined by the DLOAD Bulk Data entry.
3. The CNTRLND multiplier cannot be used with any higher-order elements.
4. When using "THRU" or "BY", all intermediate CHBDYE, CHBDYG, or CHBDYP elements must exist.

TIME DEPENDENT THERMAL LOADS (Cont.)

- Specify time varying ambient temperature for free convection

1	2	3	4	5	6	7	8	9	10
QBDY3	SID	Q0	CNTRLND	EID1	EID2	EID3	EID4	EID5	
QBDY3	500	50000.0		10	THRU	60	BY	10	

- SID – load set ID
- Q0 – thermal heat flux load, or load multiplier
- EID_i – CHBDY_j element ID *i*

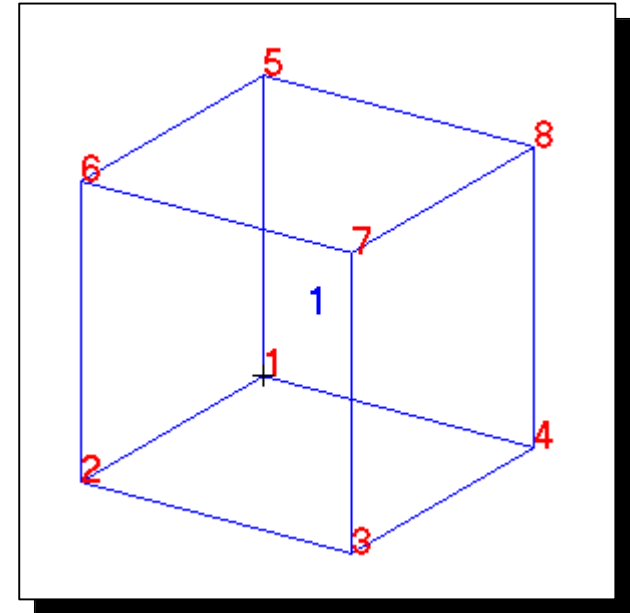
```

CHEXA      1      1      1      2      3      4      5      6
           7      8

CHBDYE,10,1,6
CONV,10,5,,50,99

DLOAD,200,1.0,1.0,300
TLOAD1,300,500,,,700
TABLED1,700,,,,,,,,+TBD700
+TBD700,0.0,1.0,1000.0,1.0,ENDT
QBDY3,500,50000.0,,10,THRU,60,BY,10

SPC,10,99,,0.0
TEMP,20,99,0.0
TEMPD,20,0.0
    
```



TIME DEPENDENT THERMAL LOADS (Cont.)

- $\text{LOAD} = \text{SID}$, where SID is an integer used in steady state analysis (SOL 153) to request application of the load Bulk Data labeled with the given SID. Only one LOAD command per subcase may be specified in the Case Control Section.
- $\text{DLOAD} = \text{SID}$ is used in transient analysis (SOL 159) to request the application of the dynamic load Bulk Data with the given SID. Only one DLOAD command per subcase may be specified in the Case Control Section.
- For steady state analysis, any number of loads defined in the Bulk Data may be referenced from a single Case Control request by specifying the same SID for all loads of interest.
- For transient analysis, the static load entries are not selected by the Case Control SID, rather they are referenced by the TLOADi entry EXCITEID. The SID required for Case Control selection is the TLOADi entry SID.
- The schematic for these processes is illustrated below.



TIME DEPENDENT THERMAL LOADS (Cont.)

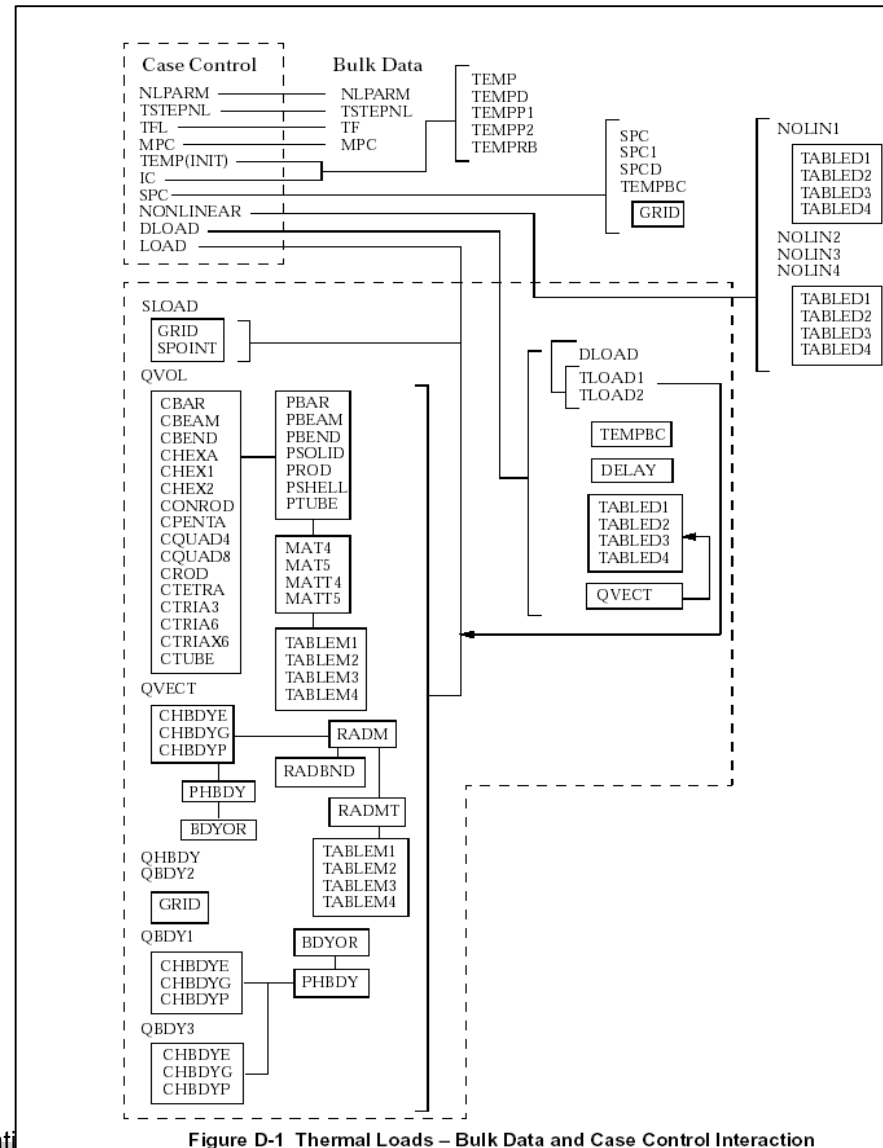
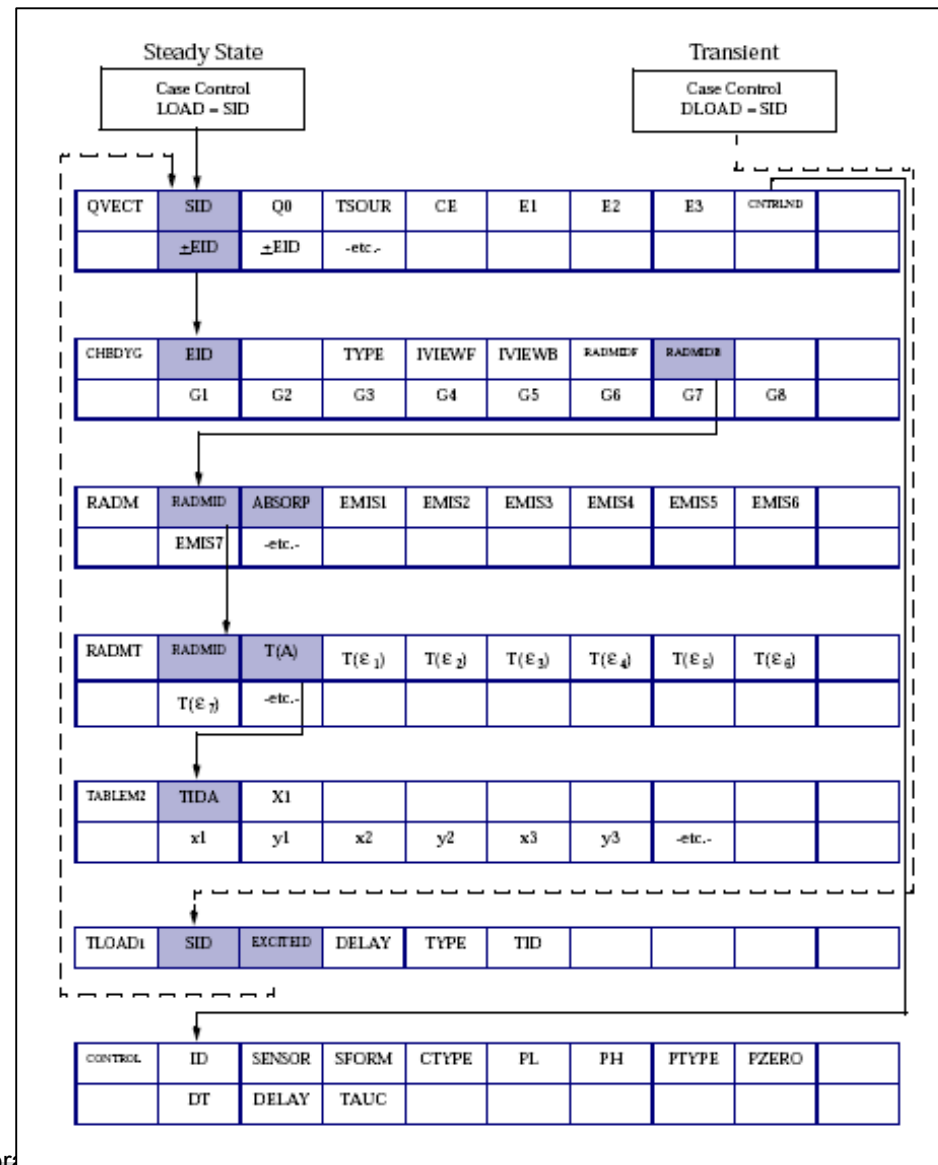


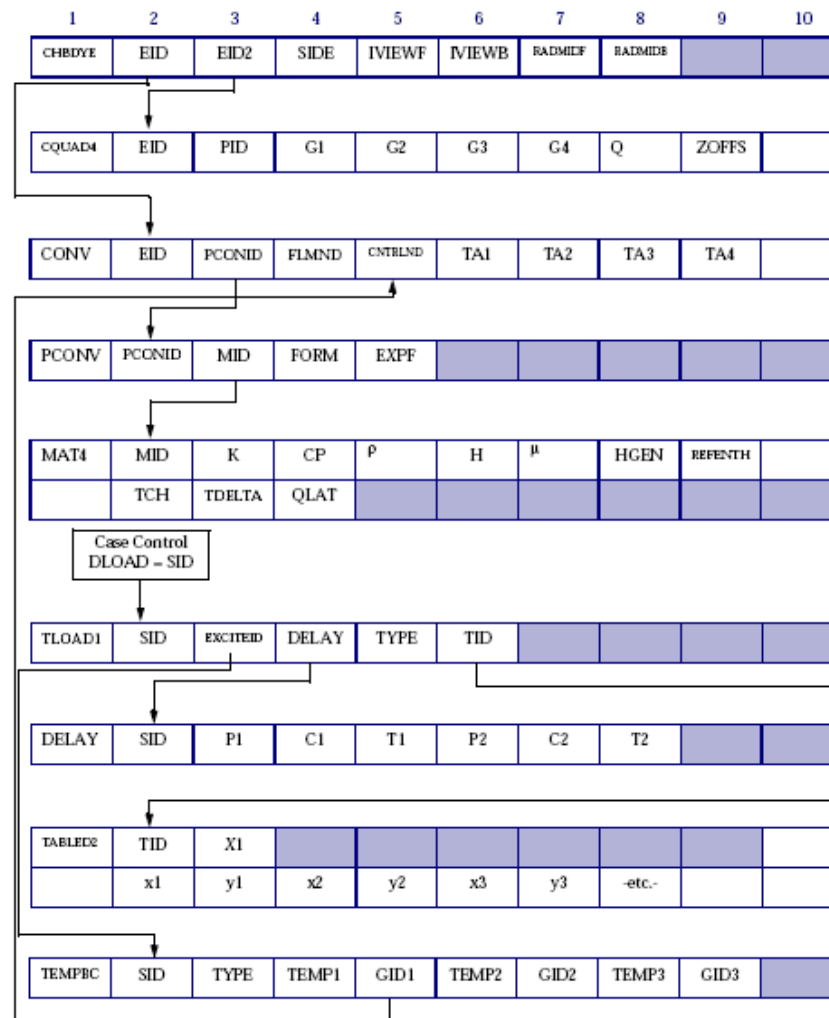
Figure D-1 Thermal Loads – Bulk Data and Case Control Interaction

TIME DEPENDENT THERMAL LOADS (Cont.)



TIME DEPENDENT HEAT TRAN COEFFICIENT

Free Convection - Time-Dependent Heat Transfer Coefficient



TRANSIENT THERMAL ANALYSIS

BULK DATA CP AND ρ

- For transient analysis need to include the specific heat and density in the MAT4 entry
 - MAT4, MID, K, CP, ρ
 - MAT4, 12, 204.0, 896.0, 2707.0

TRANSIENT THERMAL ANALYSIS

CASE CONTROL

- **ANALYSIS = HEAT** analysis type for subcase
- **ECHO = NONE** BULK DATA echo control
- **THERMAL = ALL** output nodal temperatures
- **SPCFORCE = ALL** output nodal constraint heats
- **FLUX = ALL** output element fluxes/gradients
- **SPC = 1** specify the temperature set
- **DLOAD = 1** specify dynamic load set
- **TSTEPNL = 1** specify time step adjustment
- **IC = 1** specify initial temperature set

.DAT FILE POINTER SYSTEMS EXAMPLE

```
...  
CEND  
ANALYSIS = HEAT  
...  
IC = 2  
SPC = 1  
DLOAD = 1  
TSTEPNL = 1  
...
```

```
BEGIN BULK  
...  
TSTEPNL, 1, 200, 1., 1,  
    ADAPT, 10, , UPW, ...  
SPOINT, 100001  
SPC, 1, 100001, 1, 1000.  
SPOINT, 100002  
...
```


.DAT FILE POINTER SYSTEMS EXAMPLE (Cont.)

```
TLOAD1,101,101,,,1  
$ h_of_time  
TABLED2,1,0.,...,ENDT  
TEMPBC,101,TRAN,1.,100002  
...  
DLOAD,1,1.,1.,101  
...
```

```
...  
TEMP,2,100001,1000.  
TEMP,2,100002,13.5  
...  
ENDDATA d0b69853
```

HOW TLOAD1 POINTS TO TABLED1

- \$TLOAD1,SID,EXCITEID,DELAY,TYPE,TID
- TLOAD1,10,21,,,121
- \$TABLED1,TID,,,,,,,,+,
- \$+,x1,y1,x2,y2,x3,y3,ENDT
- TABLED1,121,,,,,,,,+,
- +,0.0,25.0,10.0,35.0,100.0,40.0,ENDT
- \$QVOL,SID,QVOL,CNTRLND,EID1
- QVOL,21,1.0,,301
- Volumetric heating is for conduction element ID 301 with time dependent loading shown by the table ID 121.

TRANSIENT THERMAL ANALYSIS

BULK DATA DLOAD

- The DLOAD record in the case control section points to a DLOAD record in the bulk data section
 - DLOAD = 120
 - BEGIN BULK
 - \$DLOAD, SID, S, S1, L1, S2, L2, etc.
 - DLOAD, 120, 1.0, 1.0, 10, 1.0, 20

TRANSIENT THERMAL ANALYSIS

BULK DATA TLOAD2

- One form of MD Nastran transient loading

TLOAD2

Transient Response Dynamic Excitation, Form 2

Defines a time-dependent dynamic excitation or enforced motion of the form

$$\{P(t)\} = \begin{cases} 0 & , \quad t < (T1 + \tau) \text{ or } t > (T2 + \tau) \\ \{A\} \tilde{t}^B e^{C\tilde{t}} \cos(2\pi F\tilde{t} + P) & , \quad (T1 + \tau) \leq t \leq (T2 + \tau) \end{cases}$$

for use in a transient response problem, where $\tilde{t} = t - T1 - \tau$

Format:

1	2	3	4	5	6	7	8	9	10
TLOAD2	SID	EXCITEID	DELAYI/ DELAYR	TYPE	T1	T2	F	P	
	C	B	US0	VS0					

TRANSIENT THERMAL ANALYSIS

BULK DATA TLOAD2 (Cont.)

- One form of MD Nastran transient loading

Field	Contents
SID	Set identification number. (Integer > 0)
EXCITEID	Identification number of DAREA or SPCD entry set or a thermal load set (in heat transfer analysis) that defines {A}. See Remarks 2. and 3. (Integer > 0)
DELAYI	Identification number of DELAY Bulk Data entry that defines time delay τ . See Remark 5. (Integer > 0 or blank)
DELAYR	Value of time delay τ that will be used for all degrees-of-freedom that are excited by this dynamic load entry. See Remark 5. (Real or blank)
TYPE	Defines the type of the dynamic excitation. See Remarks 2. and 3. (Integer, character or blank; Default = 0)
T1	Time constant. (Real ≥ 0.0)
T2	Time constant. (Real; T2 > T1)
F	Frequency in cycles per unit time. (Real ≥ 0.0 ; Default = 0.0)
P	Phase angle in degrees. (Real; Default = 0.0)
C	Exponential coefficient. (Real; Default = 0.0)
B	Growth coefficient. (Real; Default = 0.0)

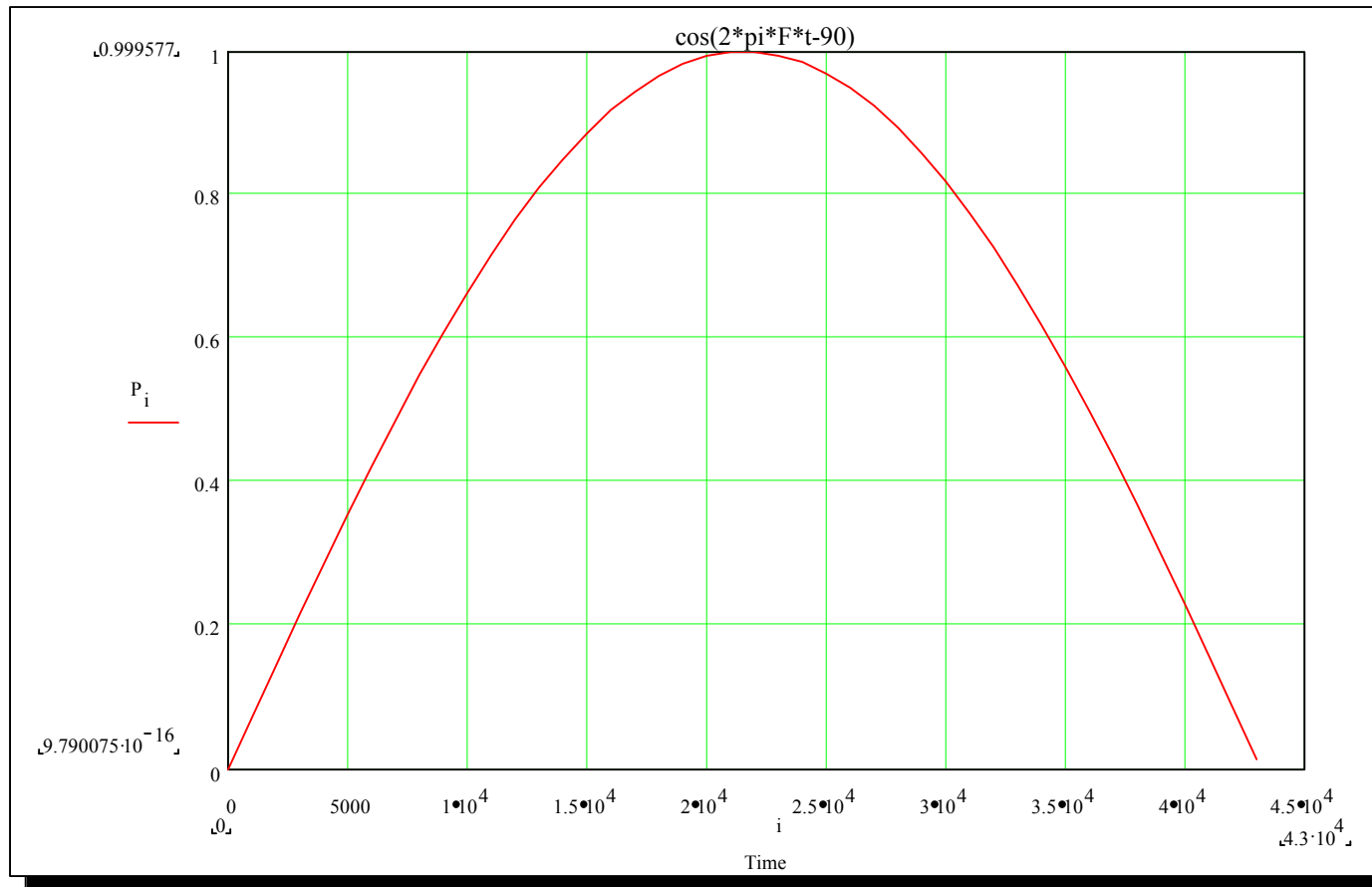
TRANSIENT THERMAL ANALYSIS

BULK DATA TLOAD2, STEP FUNCTION

- Define a step function from time = 0.0 to 10.0
\$TLOAD2, SID, EXCITEID, DELAY, TYPE, T1, T2, F, P
TLOAD2, 10, 120,,, 0.0, 10.0
QVOL, 120, 200.0,, 1, 2
- Define a step function from 20.0 to 25.0
TLOAD2, 10, 120,,, 20.0, 25.0
QBDY3, 120, 1.0E4,, 1, THRU, 4

TRANSIENT THERMAL ANALYSIS

BULK DATA TLOAD2, SIN(ΩT)



NAS104, Section 5, September 2008

Copyright© 2008 MSC Software Corporation

S5-25



TRANSIENT THERMAL ANALYSIS

BULK DATA TLOAD2, SIN(Ω T) (Cont.)

- See Example 9a in the MD Nastran 2002 Thermal Analysis User's Guide.
- TLOAD2, 300, 1000,,, 0.0, 43200.0, 1.157E-5, -90.0
 - where
 - T1 (time constant) = 0.0
 - T2 (time constant) = 43200.0
 - F (frequency) = 1.157E-5
 - P (phase angle) = -90 degrees

TRANSIENT THERMAL ANALYSIS

DO NOT USE AUTOSPC

- PARAM,AUTOSPC,YES is not allowed for transient thermal analysis. AUTOSPC is a Case Control Section command.

AUTOSPC

Constrains Stiffness Singularities via m-sets or s-sets

Requests that stiffness singularities and near singularities be automatically constrained via single or multipoint constraints.

Format:

$$\text{AUTOSPC} \left[\left(\begin{array}{c} \text{[RESIDUAL]} \\ \text{[PRINT]} \\ \text{[NOPUNCH]} \end{array} \right), \left(\begin{array}{c} \text{[NOPUNCH]} \\ \text{[PUNCH]} \end{array} \right), [\text{SID} = n], [\text{EPS} = r1], [\text{EPSSING} = r2], \right. \\ \left. \left(\begin{array}{c} \text{[SPC]} \\ \text{[MPC]} \end{array} \right), \left(\begin{array}{c} \text{[ZERO]} \\ \text{[NOZERO]} \end{array} \right) \right] = \left\{ \begin{array}{c} \text{YES} \\ \text{NO} \end{array} \right\}$$

Examples:

AUTOSPC=YES

AUTOSPC (PRINT, PUNCH, SID=100, EPS=1.E-6, MPC)=YES

NAS104,

Copyright© 2008 MSC Software Corporation

S5-27



TRANSIENT THERMAL ANALYSIS

BULK DATA TSTEPNL

- The TSTEPNL Bulk Data entry used to specify parametric control for nonlinear transient thermal analysis

TSTEPNL

Parameters for Nonlinear Transient Analysis

Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. TSTEPNL is intended for SOLs 129, 159, 400, 600 and 700.

Format:

1	2	3	4	5	6	7	8	9	10
TSTEPNL	ID	NDT	DT	NO	METHOD	KSTEP	MAXITER	CONV	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS		
	MAXBIS	ADJUST	MSTEP	RB	MAXR	UTOL	RTOLB	MINITER	

Example:

TSTEPNL	250	100	.01	1	ADAPT	2	10	PW	
	1.E-2	1.E-3	1.E-6	2	10	2	.02		
	5	5	0	0.75	16.0	0.1	20.		

TRANSIENT THERMAL ANALYSIS

BULK DATA TSTEPNL (Cont.)

- TSTEPNL Bulk Data parametric control

Field	Contents
ID	Identification number. (Integer > 0)
NDT	Number of time steps of value DT. See Remark 2. (Integer ≥ 4)
DT	Time increment. See Remark 2. (Real > 0.0)
NO	Time step interval for output. Every NO-th step will be saved for output. See Remark 3. (Integer $\neq 0$; Default = 1)
METHOD	Method for stiffness matrix update and the transient time integration schemes. For SOLs 129 and 159, only METHOD=ADAPT is allowed and it is also the default. For SOL 400, all options are allowed. The default is AUTO with non-contact analysis and FNT with contact analysis. See Remark 4. (Character: "AUTO", "ITER", "ADAPT", "SEMI", "FNT", or "PFNT")
KSTEP	KSTEP is the number of converged bisection solutions between stiffness updates for ADAPT method. (Default = 2). It is the number of iterations before stiffness updates for ITER method (Default = 10). See Remark 18. and 19. (Integer > 0).
MAXITER	Limit on number of iterations for each time step. See Remarks 5., 6., and 18. (Integer $\neq 0$; Default = 10 for ADAPT method and 25 for non-ADAPT methods)
CONV	Flags to select convergence criteria. See Remark 7. and 21. (Character: Default = "PW" for SOLs 129 and 159, "UPW" for SOL 400 with non-contact analysis, and "PV" for SOL 400 with contact analysis.)

TRANSIENT THERMAL ANALYSIS

BULK DATA TSTEPNL (Cont.)

- TSTEPNL Bulk Data parametric control

Field	Contents
EPSU	Error tolerance for displacement (U) criterion. See Remark 20. (Real $\neq$ 0.0; Default = 1.0E-2 for all methods except PFTN. For PFTN, Default = -1.0E-2)
EPSP	Error tolerance for load (P) criterion. (Real > 0.0; Default = 1.0E-3 for SOLs 129 and 159, 1.0E-2 for SOL 400)
EPSW	Error tolerance for work (W) or energy criterion. See Remark 20. (Real > 0.0; Default = 1.0E-6 for SOLs 129 and 159, 1.0E-2 for SOL 400 and non-PFNT methods, -1.0E-2 for SOL 400 and PFNT method)
MAXDIV	Limit on the number of diverging conditions for a time step before the solution is assumed to diverge. See Remark 8. (Integer $\neq$ 0; Default = 2)
MAXQN	Maximum number of quasi-Newton correction vectors to be saved on the database. See Remark 9. (Integer $\geq$ 0; Default = 10 for all methods except PFNT. For PFNT, Default = 0)
MAXLS	Maximum number of line searches allowed per iteration. See Remark 9. (Integer $\geq$ 0; Default = 2 for all methods except PFNT. For PFNT, Default = 0)
FSTRESS	Fraction of effective stress ($\bar{\sigma}$) used to limit the subincrement size in the material routines. See Remark 10. (0.0 < Real < 1.0; Default = 0.2)

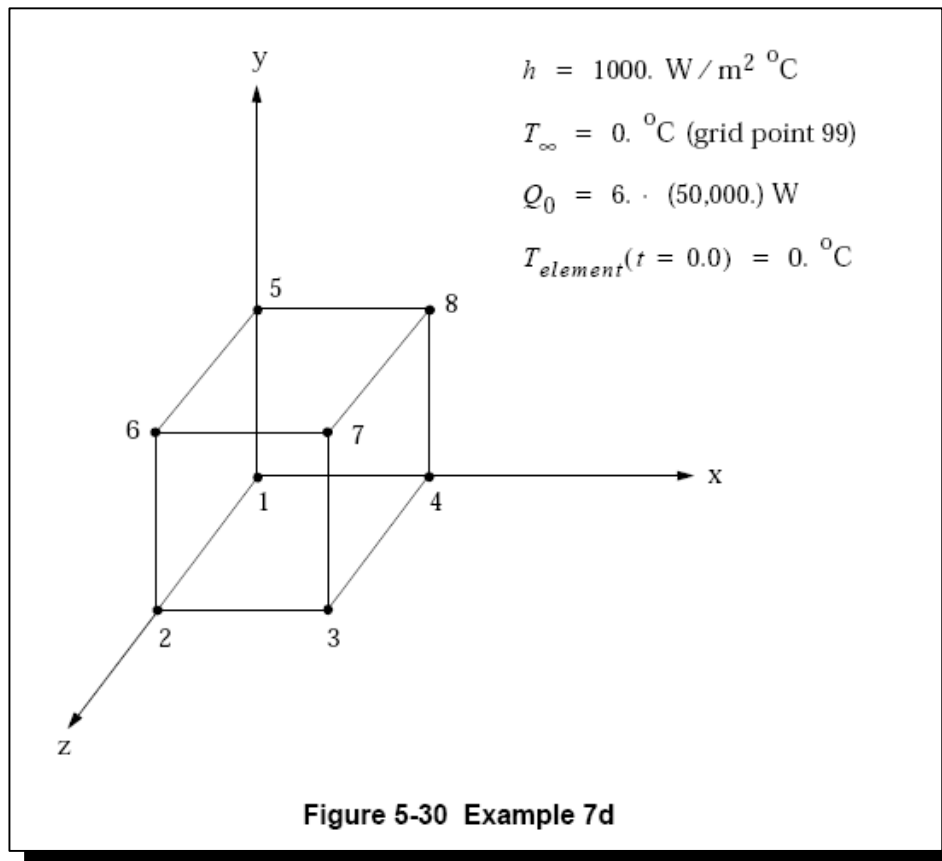
TRANSIENT THERMAL ANALYSIS

BULK DATA TSTEPNL (Cont.)

- TSTEPNL Bulk Data parametric control

Field	Contents
MAXBIS	Maximum number of bisections allowed for each time step. See Remark 11. and 12. ($-9 \leq \text{Integer} \leq 9$; Default = 5 except for MAXITER < 0 and SOL 400. For MAXITER < 0 and SOL 400, Default = 0)
ADJUST	Time step skip factor for automatic time step adjustment. See Remark 13. (Integer ≥ 0 ; Default = 5)
MSTEP	Number of steps to obtain the dominant period response. See Remark 14. ($10 \leq \text{Integer} \leq 200$; Default = variable between 20 and 40 for SOL 129 and between 10 and 20 for SOL 400.)
RB	Define bounds for maintaining the same time step for the stepping function during the adaptive process. See Remark 14. ($0.1 \leq \text{Real} \leq 1.0$; Default = 0.6)
MAXR	Maximum ratio for the adjusted incremental time relative to DT allowed for time step adjustment. See Remark 15. (Real ≥ 1.0 ; Default = 32.0)
UTOL	Tolerance on displacement or temperature increment below which a special provision is made for numerical stability. See Remark 16. ($0.001 < \text{Real} \leq 1.0$; Default = 0.1)
RTOLB	Maximum value of incremental rotation (in degrees) allowed per iteration to activate bisection. See Remark 17. (Real > 2.0 ; Default = 20.0)
MINITER	Minimum number of iterations for a load increment, SOL 400 only. (Default = 1 except for the contact analysis. For contact analysis, Default = 2)

EXAMPLE 7d, TIME VARYING HEAT TRANSFER COEFFICIENT



- Control node is to be used to produce a free convective heat transfer coefficient that varies with time.
- $H(\text{time}) = 1000 \cdot u_{\text{CNTRLND}} = 1000 \cdot \text{TABLED1}$
- Surface heat flux (50,000 Watts) on six faces with QBDY3.
- Initially, the structure is at zero degree at time = 0 sec.

EXAMPLE 7d, $H(\text{TIME}) = 1000 * \text{TABLED1}$

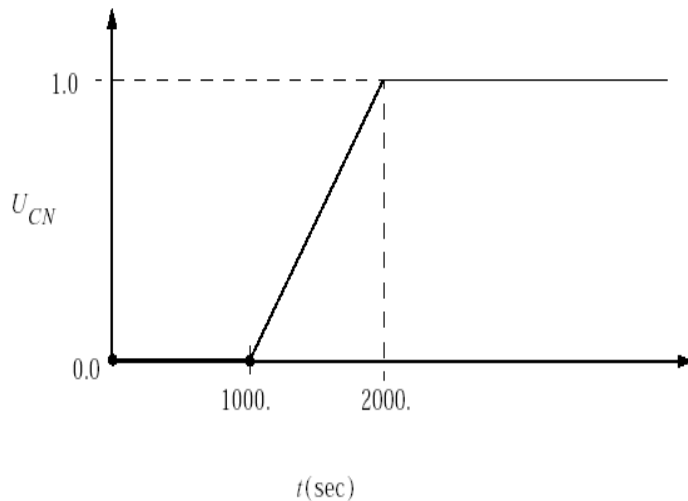


Figure 5-22 Control Node (Grid Point 50) for Free Convection Boundary Condition

```
SOL 159
TIME 10
CEND
TITLE = EXAMPLE 7D
EX7D
ANALYSIS = HEAT
THERMAL = ALL
FLUX = ALL
SPCF = ALL
OLOAD = ALL
SPC = 10
IC = 20
TSTEPNL = 100
DLOAD = 200
OUTPUT (XYPLOT)
XTITLE = TIME, SECONDS
YTITLE = GRID 1 TEMPERATURE DEGREES CELSIUS
TCURVE = GRID 1 TEMPERATURE VS. TIME
XYPLOT TEMP/1 (T1)
XTITLE = TIME, SECONDS
YTITLE = GRID 50 TEMPERATURE DEGREES CELSIUS
TCURVE = GRID 50 TEMPERATURE VS. TIME
XYPLOT TEMP/50 (T1)
BEGIN BULK
TSTEPNL,100,490,10.0,1
```

EXAMPLE 7d, MD NASTRAN INPUT FILE

```
GRID,1,,0.0,0.0,0.0
GRID,2,,0.0,0.0,1.0
GRID,3,,1.0,0.0,1.0
GRID,4,,1.0,0.0,0.0
GRID,5,,0.0,1.0,0.0
GRID,6,,0.0,1.0,1.0
GRID,7,,1.0,1.0,1.0
GRID,8,,1.0,1.0,0.0
GRID,50,,50.0,50.0,50.0
GRID,99,,99.0,99.0,99.0
$
CHEXA,1,5,1,2,3,4,5,6,+CHX1
+CHX1,7,8
PSOLID,5,15
MAT4,15,204.0,896.0,2707.0,1000.0
$
CHBDYE,10,1,1
CHBDYE,20,1,2
CHBDYE,30,1,3
CHBDYE,40,1,4
CHBDYE,50,1,5
CHBDYE,60,1,6
$
CONV,10,35,,50,99
CONV,20,35,,50,99
CONV,30,35,,50,99
CONV,40,35,,50,99
CONV,50,35,,50,99
CONV,60,35,,50,99
PCONV,35,15,0,0.0
```

```
DLOAD,200,1.0,1.0,300,1.0,400
$
TLOAD1,300,500,,,700
TABLED1,700,,,,,,,,+TBD700
+TBD700,0.0,1.0,1000.0,1.0,ENDT
QBDY3,500,50000.0,,10,THRU,60,BY,10
$
TLOAD1,400,600,,,800
TEMPBC,600,TRAN,1.0,50
TABLED1,800,,,,,,,,+TBD800
+TBD800,0.0,0.0,1000.0,0.0,2000.0,1.0,5000.0,1.0,+TBD801
+TBD801,ENDT
SPC,10,99,,0.0
TEMP,20,99,0.0
TEMPPD,20,0.0
$
ENDDATA
```


EXAMPLE 7d, CONTROL NODE, H(TIME)

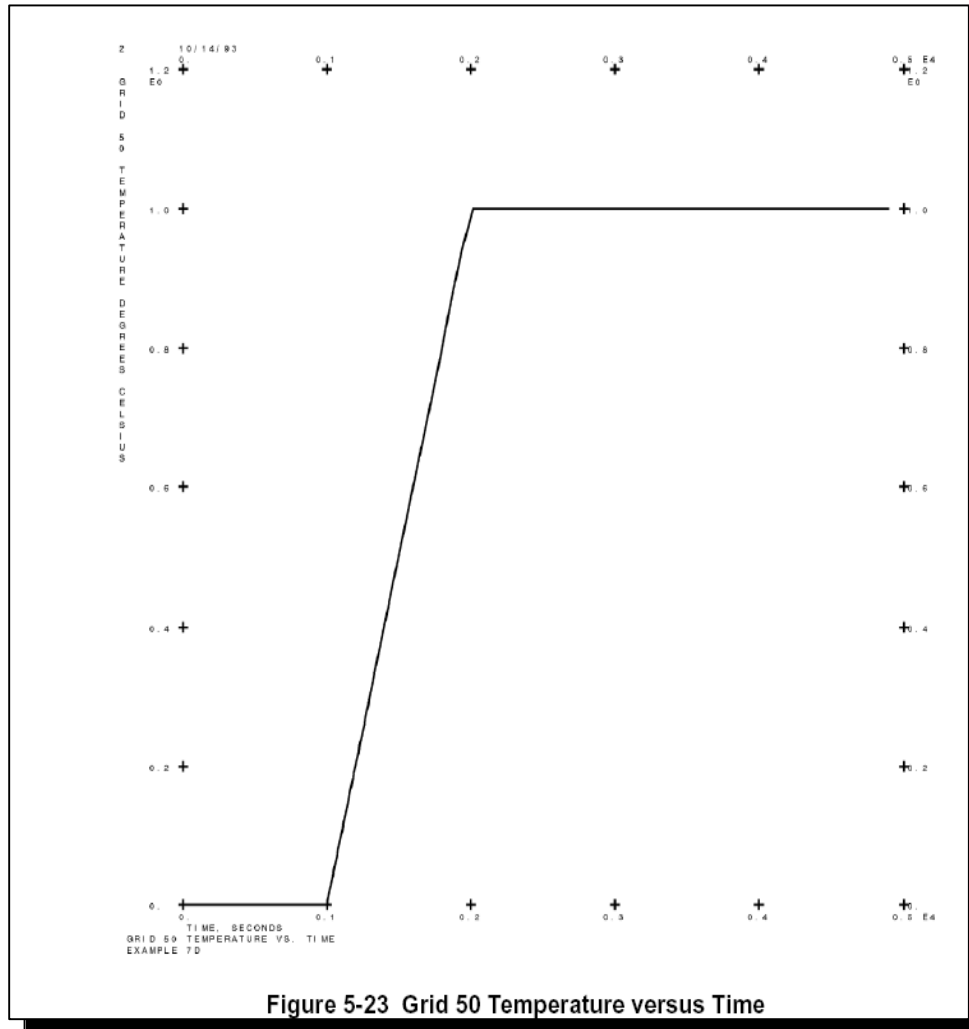
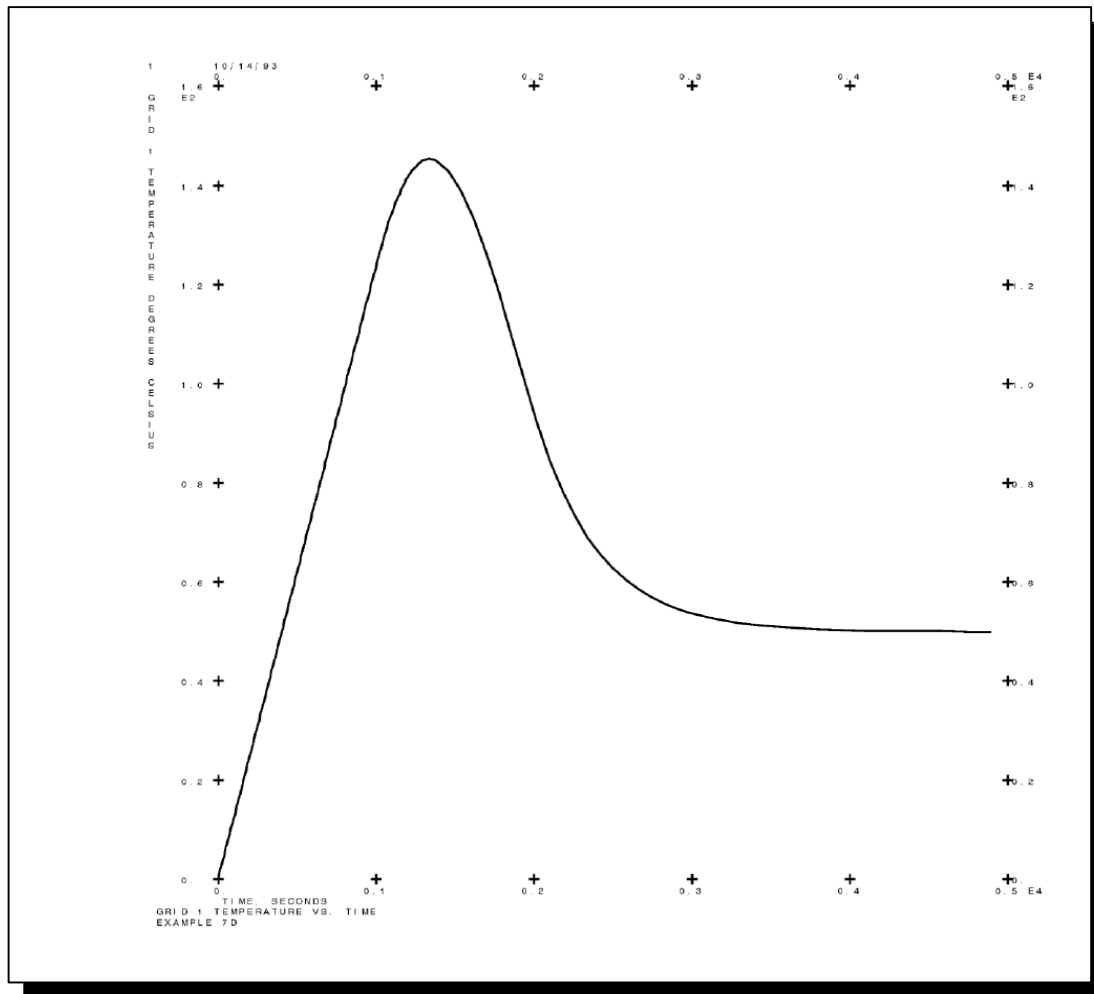


Figure 5-23 Grid 50 Temperature versus Time

- Using the MD Nastran output (PLOT) capability, the time history for the control node (Node 50), used to calculate the convection coefficient as function of time, can be plotted.

EXAMPLE 7d, TEMPERATURE RESPONSE



- Temperature response for Grid 1

EXAMPLE 7d, MD NASTRAN SOL 400

- For MD Nastran SOL 400 transient analysis, the nodal temperature that needs to be constrained as a function of time is done using Bulk Data entries that are different than those used for MD Nastran SOL 159

```

DLOAD,200,1.0,1.0,300,1.0,400
$
TLOAD1,300,500,,,700
TABLED1,700,,,,,,+TBD700
+TBD700,0.0,1.0,1000.0,1.0,ENDT
QBDY3,500,50000.0,,10,THRU,60,BY,10
$
TLOAD1,400,600,,,800
TEMPBC,600,TRAN,1.0,50
TABLED1,800,,,,,,+TBD800
+TBD800,0.0,0.0,1000.0,0.0,2000.0,1.0,5000.0,1.0,+TBD801
+TBD801,ENDT
SPC,10,99,,0.0
TEMP,20,99,0.0
TEMPPD,20,0.0
$
ENDDATA
    
```

SOL 159

```

DLOAD,200,1.0,1.0,300,1.0,400,1.0,1000
$
TLOAD1,300,500,,,700
TABLED1,700,,,,,,+TBD700
+TBD700,0.0,1.0,1000.0,1.0,ENDT
QBDY3,500,50000.0,,10,THRU,60,BY,10
$
TLOAD1,400,600,,,1,800
$TEMPBC,600,TRAN,1.0,50
SPC1,10,,50
SPCD,600,50,,,1.0
TABLED1,800,,,,,,+TBD800
+TBD800,0.0,0.0,1000.0,0.0,2000.0,1.0,5000.0,1.0,+TBD801
+TBD801,ENDT
$SPC,10,99,,0.0
TLOAD1,1000,1100,,,1,1200
SPC1,10,,99
SPCD,1100,99,,,0.0
TABLED1,1200,,,,,,+TBD1200
+TBD1200,0.0,1.0,1000.0,1.0,ENDT
TEMP,20,99,0.0
TEMPPD,20,0.0
$
ENDDATA
    
```

SOL 400

EXAMPLE 7d, MD NASTRAN SOL 400

- When there are both constant (SPC Bulk Data entry for SOL 153 and 159) and transient (TEMPBC Bulk Data entry for SOL 159) temperature constraints they must be represented using SPC1 and SPCD Bulk Data entries.
- The reason for using SPC1 and SPCD entries is MD Nastran SOL 400 has a coupled analysis capability (e.g. coupled thermal and structural capability). The structural capability uses SPC1 and SPCD Bulk Data entries to enforce displacement. The thermal capability needs to be represented the same way.

EXAMPLE 7d, MD NASTRAN SOL 400

SPC1

Single-Point Constraint, Alternate Form

Defines a set of single-point constraints.

Format:

1	2	3	4	5	6	7	8	9	10
SPC1	SID	C	G1	G2	G3	G4	G5	G6	
	G7	G8	G9	-etc.-					

Example:

SPC1	3	2	1	3	10	9	6	5	
	2	8							

Alternate Format and Example:

SPC1	SID	C	G1	"THRU"	G2				
SPC1	313	12456	6	THRU	32				

Field	Contents
SID	Identification number of single-point constraint set. (Integer > 0)
C	Component numbers. (Any unique combination of the Integers 1 through 6 with no embedded blanks for grid points. This number must be Integer 0 or blank for scalar points.)
Gi	Grid or scalar point identification numbers. (Integer > 0 or "THRU"; For "THRU" option, $G1 < G2$.)

EXAMPLE 7d, MD NASTRAN SOL 400

SPCD

Enforced Motion Value

Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.

Format:

1	2	3	4	5	6	7	8	9	10
SPCD	SID	G1	C1	D1	G2	C2	D2		

Example:

SPCD	100	32	436	-2.6	5		2.9		

Field	Contents
SID	Set identification number of the SPCD entry. (Integer > 0)
Gi	Grid or scalar point identification number. (Integer > 0)
Ci	Component numbers. ($0 \leq \text{Integer} \leq 6$; up to six unique Integers may be placed in the field with no embedded blanks, a blank or 0 is treated the same as 1.)
Di	Value of enforced motion for components Gi at grid Ci. (Real)

EXAMPLE 7d, MD NASTRAN SOL 400

- To access MD Nastran SOL 400, the needed Executive and Case Control statements are

...

SOL 400

...

CEND

...

ANALYSIS = HTRAN

...

BULKDATA

PROJECT 8, QVOL (TIME)

- This project comes from Example 7c in the MD Nastran 2002 Thermal Analysis User's Guide.
- The objective is to apply time varying volumetric heating to a single 1x1x1 CHEXA element.
- HGEN = 10.0
- Also, convect to ambient temperature at 0 °C with a constant h of 100.0 W/(m² °C).
- Initially the solid is at 0 °C.

PROJECT 8, QVOL (TIME)

- Create a 1x1x1 geometric solid.
- IsoMesh the solid.
 - Hex8 topology
 - GEL = 1.0

Geometry

Action:

Object:

Method:

Solid ID List

Refer. Coordinate Frame

Vector Coordinates List

☒ Auto Execute

Origin Coordinates List

Finite Elements

Action:

Object:

Type:

Output ID List
Node
Element

Elem Shape

Mesher

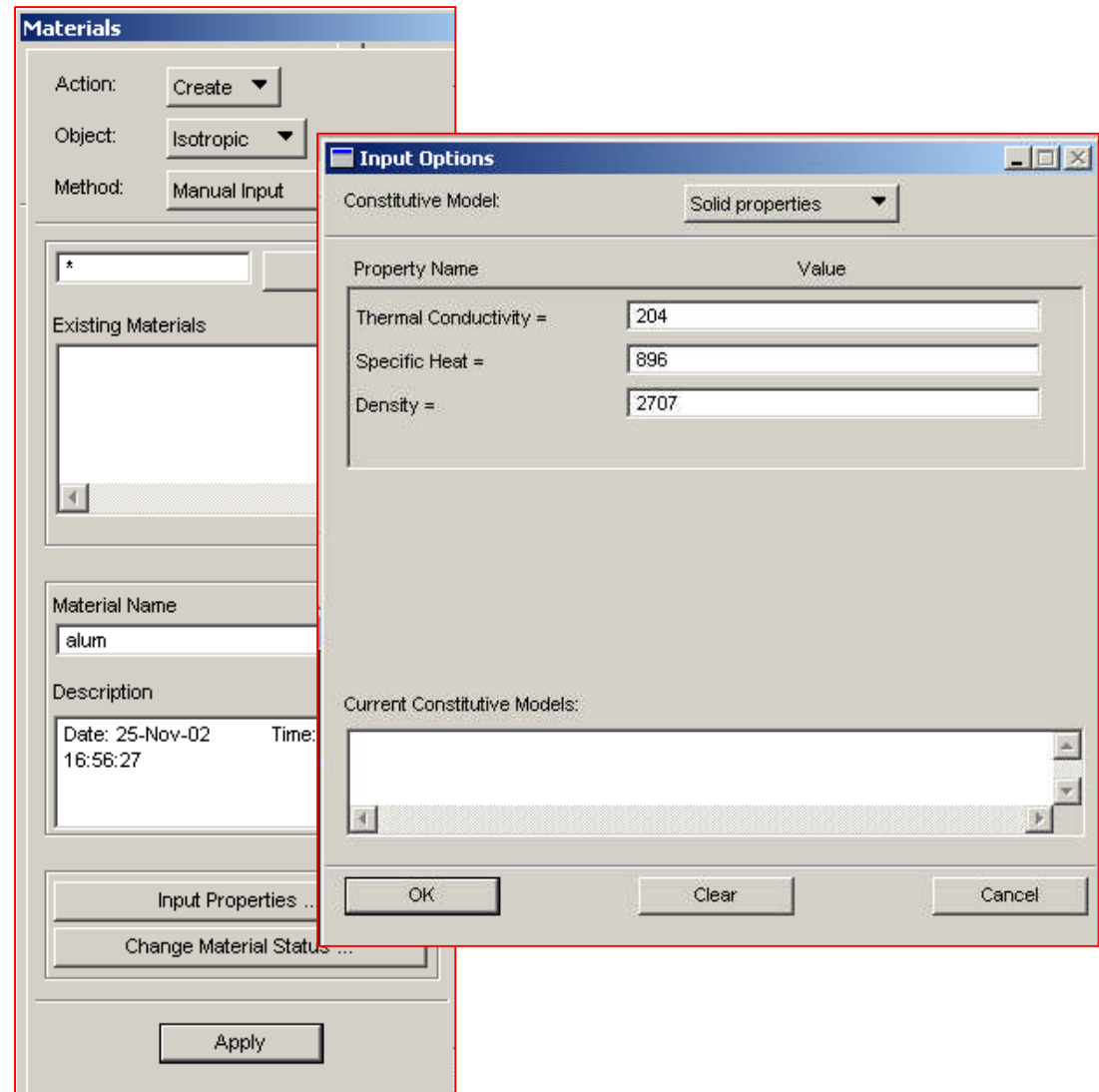
Topology

Solid List

Global Edge Length
☐ Automatic Calculation
Value

PROJECT 8, QVOL (TIME)

- Create a material property set for aluminum.
 - K
 - Cp
 - Rho



The screenshot shows the 'Materials' dialog box with the 'Create' action selected. The 'Object' is set to 'Isotropic' and the 'Method' is 'Manual Input'. The 'Material Name' is 'alum' and the 'Description' is 'Date: 25-Nov-02 Time: 16:56:27'. The 'Input Properties' button is highlighted. The 'Input Options' dialog box is open, showing the 'Constitutive Model' as 'Solid properties'. The 'Property Name' and 'Value' table is as follows:

Property Name	Value
Thermal Conductivity =	204
Specific Heat =	896
Density =	2707

The 'Current Constitutive Models' list is empty. The 'OK', 'Clear', and 'Cancel' buttons are visible at the bottom of the 'Input Options' dialog.

PROJECT 8, QVOL (TIME)

- Change the Constitutive Model to “Heat generation”.
- 10.0 is the factor by which the heat generation is multiplied.

The screenshot shows the 'Input Options' dialog box with the following settings:

- Constitutive Model:** Heat generation
- Property Name:** Heat Generation Multiplier =
- Value:** 10.0
- Temperature Dep/Model Variable Fields:** (Empty list)
- Current Constitutive Models:** Solid properties - [....] - [Active]

Buttons at the bottom: OK, Clear, Cancel.

PROJECT 8, QVOL (TIME)

- Create 3D element property set.
 - Aluminum
 - Solid 1

The image shows two overlapping dialog boxes from the MSC Software QVOL (TIME) application. The 'Element Properties' dialog is on the left, and the 'Input Properties' dialog is on the right.

Element Properties Dialog:

- Action: Create
- Object: 3D
- Type: Solid
- Existing Property Sets: (Empty list)
- Property Set Name: solid
- Option(s): Standard Formulation
- Input Properties ... (Button)
- Application Region: Select Members (Solid 1)
- Add (Button)
- Remove (Button)
- Application Region: (Empty list)
- Apply- (Button)

Input Properties Dialog:

Solid (CHEXA)

Property Name	Value	Value Type
Material Name	m:alum	Mat Prop Name
[Mater. Orientation]		String

Material Property Sets

- alum

OK (Button)

PROJECT 8, QVOL (TIME)

- To create a time dependent thermal load (e.g. applied heating), it is necessary to first create a time dependent load case. When this is done, the Loads/BCs form in MD Patran will have the necessary entry lists for the transient inputs.

Load Cases

Action: Create

* Filter

Existing Load Cases

Default

Load Case Name

tran

☒ Make Current

Load Case Type:

Time Dependent

Description

Assign/Prioritize Loads/BCs

Load Case Scale Factor

1.0

-Apply-

PROJECT 8, QVOL (TIME)

- Specify QVOL as a function of time.
 - Create a Non Spatial field named qvol_time.

Non Spatial Scalar Table Data

Input Scalar Data:

Data

	Time(t)	Value
1	0.00000E+000	0.00000E+000
2	1.00000E+003	1.00000E+000
3	2.00000E+003	0.00000E+000
4	3.00000E+003	0.00000E+000
5		
6		
7		
8		

Map Function to Table...

OK

PROJECT 8, QVOL (TIME)

- The QVOL function is equal to $10000.0 * \text{qvol\_time}$ (non spatial time dependent field).

The image shows three overlapping software dialog boxes from MSC Software:

- Load/Boundary Conditions**:
 - Action: Create
 - Object: Applied Heat
 - Type: Element Uniform
 - Option: Volumetric Generation
 - Current Load Case: tran...
 - Type: Time Depend
 - Existing Sets: (empty list)
 - New Set Name: qvol
 - Target Element Type: 3D
 - Buttons: Input Data..., Select Application Region..., -Apply-
- Input Data**:
 - Form Type: Basic
 - Volumetric Heat Generation: 10000.
 - * Time Function: f.qvol_time
 - Spatial Fields: (empty list)
 - Time Dependent: qvol_time
 - Buttons: OK, Reset
- Select Application Region**:
 - Geometry Filter: ☒ Geometry, ☐ FEM
 - Application Region: Select Solids (Solid 1)
 - Buttons: Add, Remove
 - Application Region: (empty list)
 - Button: OK

PROJECT 8, QVOL (TIME)

- Create convection to ambient temperature.
 - Constant convection coefficient $h = 100$
 - Ambient temperature is 0 C.
 - No time function is used.

The screenshot displays the MSC Software QVOL (TIME) interface. The 'Load/Boundary Conditions' dialog box is open, showing the following settings:

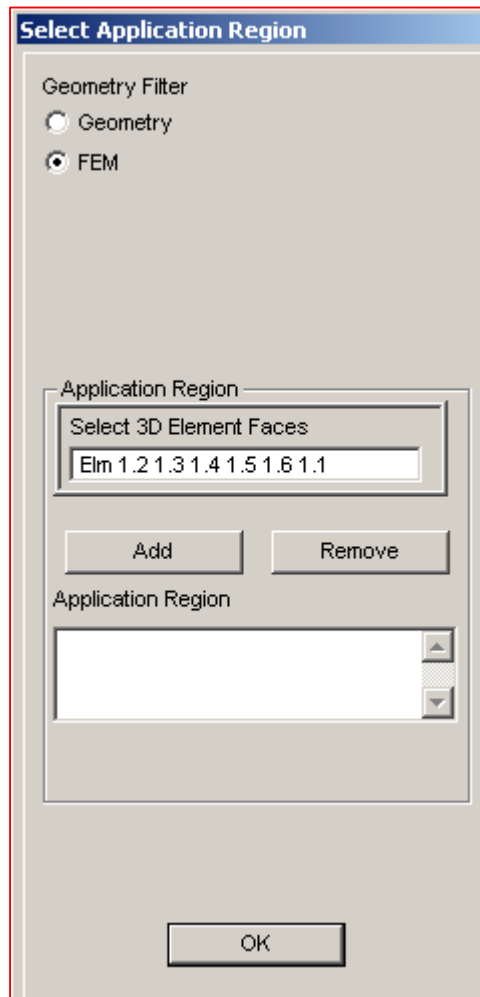
- Action: Create
- Object: Convection
- Type: Element Uniform
- Option: To Ambient
- Current Load Case: tran...
- Type: Time Dependent
- Existing Sets: (empty list)
- New Set Name: conv
- Target Element Type: 3D

The 'Input Data' dialog box is also open, showing the following settings:

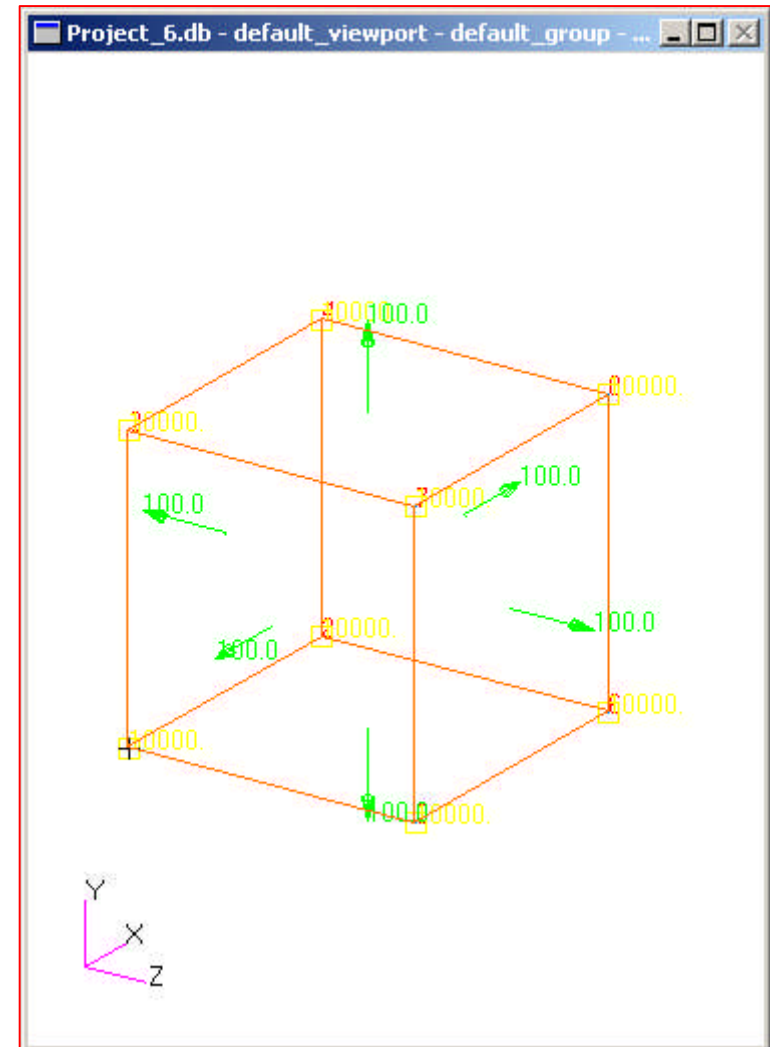
- Form Type: Basic
- Convection Coefficient: 100
- Ambient Temperature: 0
- * Temperature Function: (empty)
- * Time Function: (empty)
- Spatial Fields: (empty)
- Temperature Dependent Fields: (empty)
- Time Dependent Fields: qvol_time

Buttons at the bottom of the 'Input Data' dialog include OK, Reset, and Cancel. The 'Load/Boundary Conditions' dialog has an 'Input Data...' button and a 'Select Application P...' button.

PROJECT 8, QVOL (TIME)



- Pick all the faces at one time.



PROJECT 8, QVOL (TIME)

Analysis

Action:

Object:

Method:

Code:

Type:

Study:

Available Jobs

Job Name

Job Description

Solution Type

MSC.Nastran
Solution Type

Solution Type:

☐ STEADY STATE ANALYSIS

☒ TRANSIENT ANALYSIS

Solution Sequence: 159

Solution Parameters

Transient Solution Parameters

☐ Print Out Singularities

Data Deck Echo:

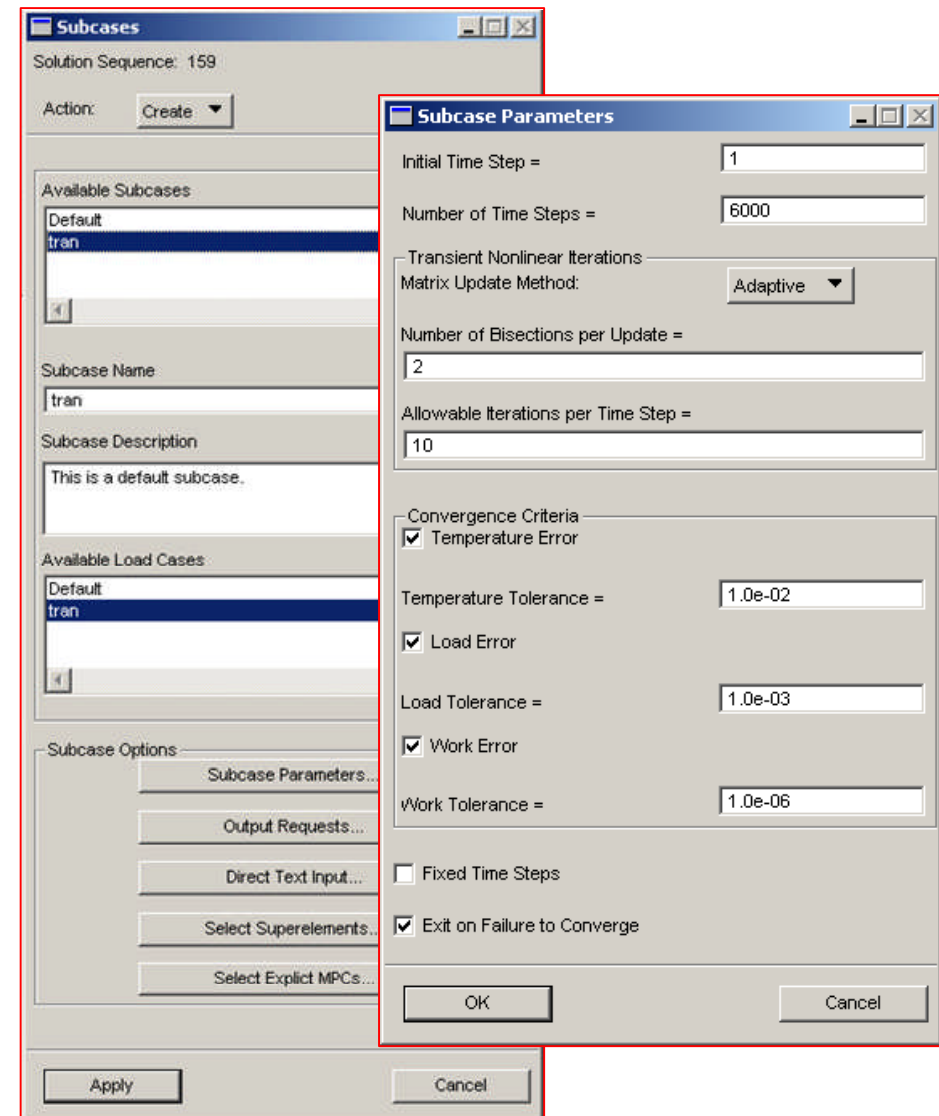
Maximum Printed Lines =

Maximum Run Time =

Default Init Temperature =

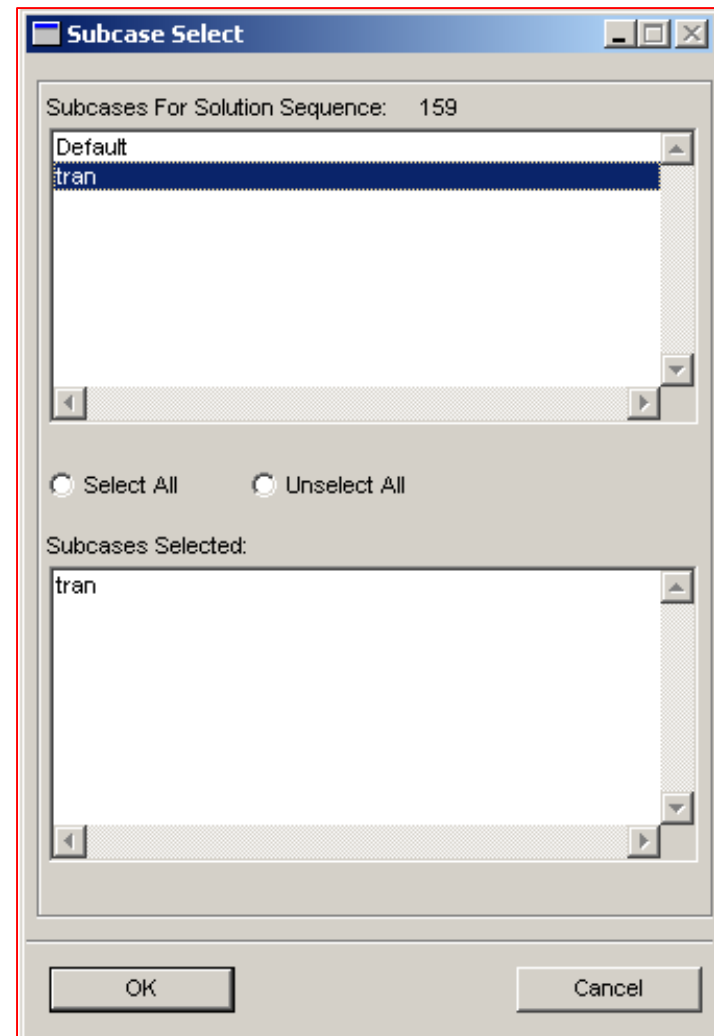
PROJECT 8, QVOL (TIME)

- Create a subcase under Subcases.
 - A subcase is automatically created for each load case that is created. They both have the same name.
 - It may be necessary to change some of the subcase settings, e.g. change some of the settings under Subcase Parameters.
 - It is possible to create a new subcase using the available load cases. When this is done, specify the values for the various subcase parameters.



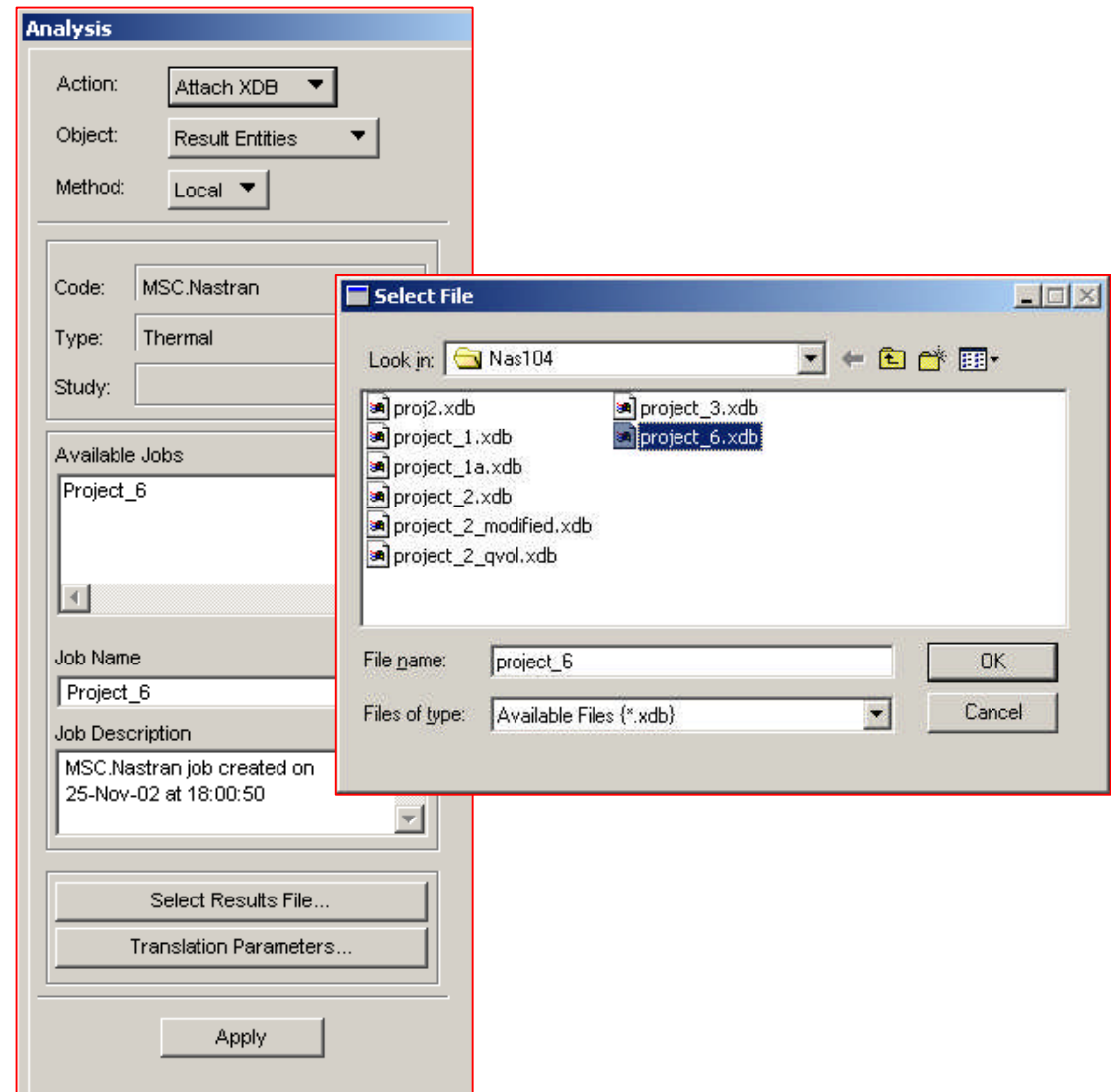
PROJECT 8, QVOL (TIME)

- Under Subcase Select, select the subcase named “tran” for the transient analysis. If necessary, deselect the subcase named “Default”.



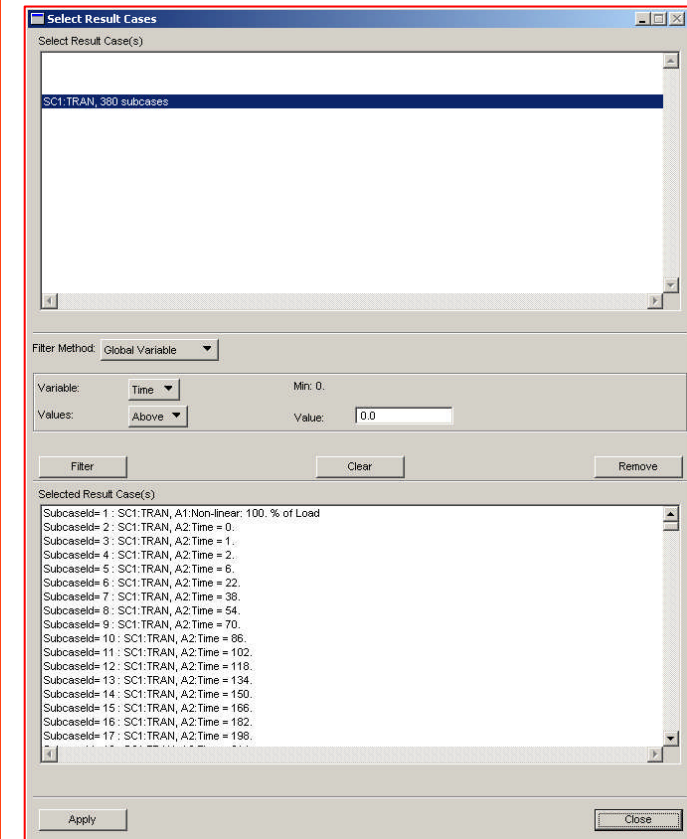
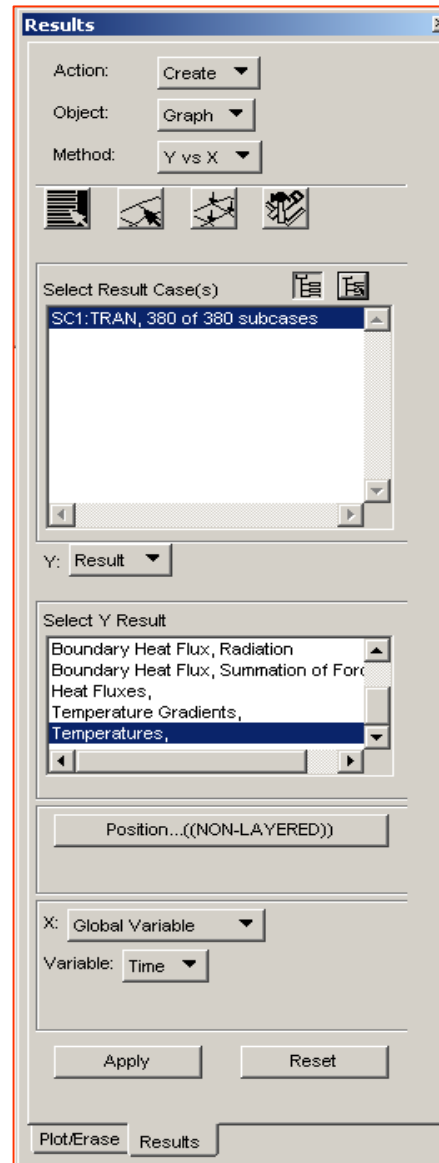
PROJECT 8, QVOL (TIME)

- Access the transient thermal analysis results by “attaching” the .xdb file to the MD Patran database.



PROJECT 8, QVOL (TIME)

- Create a plot of temperature versus time at a node(s).
- After selecting the result case (SC1:TRAN), result type (Temperatures), X: Global Variable, Variable: Time, select the second icon at the top of the Results form, Target Entities.



NAS104, Section 5, September 2008

Copyright© 2008 MSC Software Corporation

PROJECT 8, QVOL (TIME)

Results

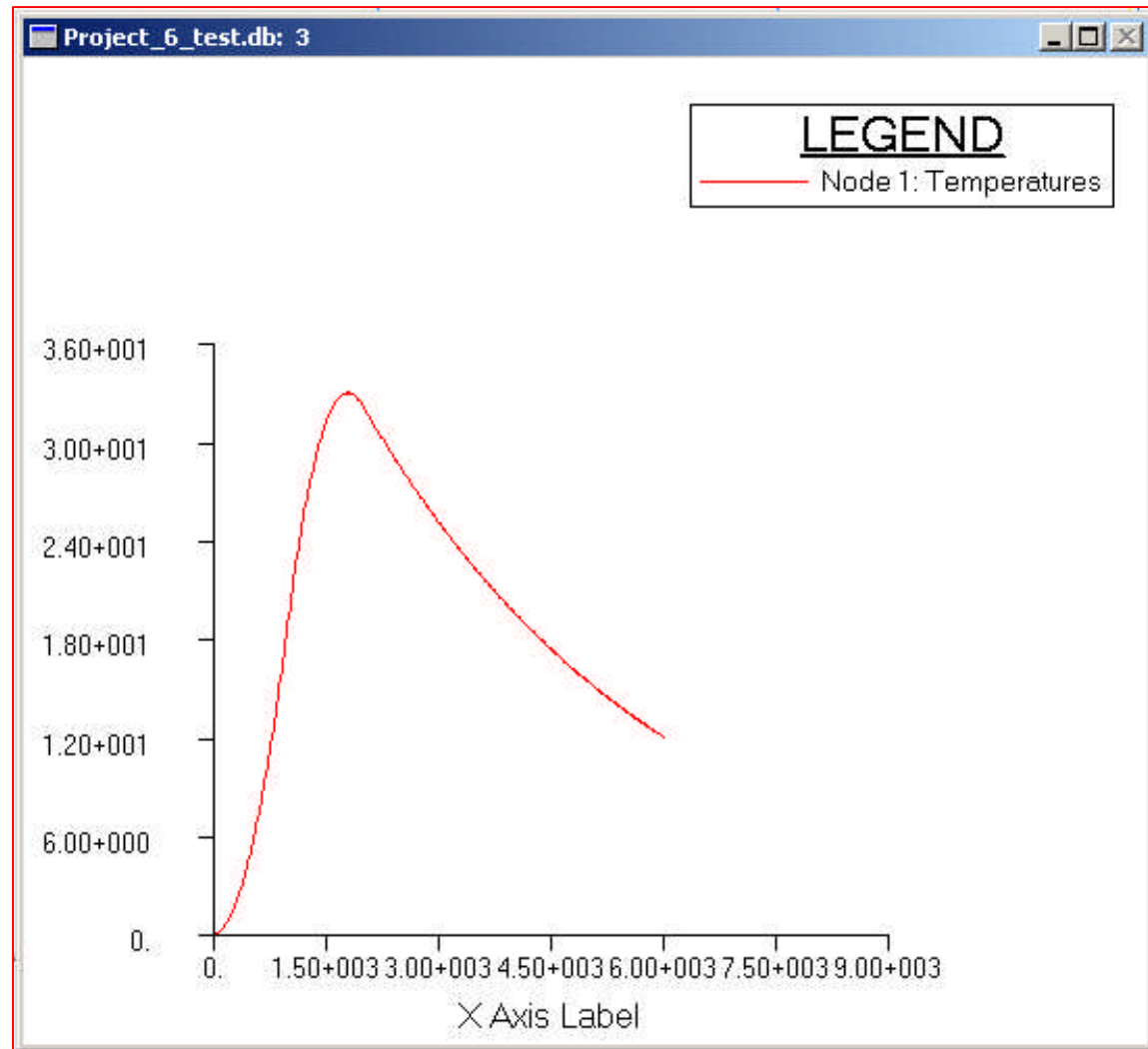
Action:

Object:

Method:

Target Entity:

Select Nodes



PROJECT 8, QVOL (TIME)

Extracted from MD Nastran model .DAT file

```
SOL 159
TIME 600
CEND
ANALYSIS = HEAT
TITLE = MSC/NASTRAN job created on 08-Jul-98 at
16:41:04
ECHO = SORT
MAXLINES = 999999999
SPC = 1
IC = 1
SUBCASE 1
$ Subcase name : tran
  SUBTITLE=tran
  TSTEPNL = 1
  DLOAD = 2
  THERMAL (SORT2, PRINT) =ALL
  FLUX (SORT2, PRINT) =ALL
```


PROJECT 8, QVOL (TIME)

Extracted from MD Nastran model .DAT file

```
PSOLID    1      1      0
CHEXA     1      1      1      2      4      3      5      6      +      B
+         B      8      7
$ Referenced Material Records
$ Material Record : alum
$ Description of Material : Date: 08-Jul-98      Time: 16:15:34
MAT4      1      204.    896.    2707.      10.
$ Nodes of the Entire Model
GRID      1      0.      0.      0.
GRID      2      1.      0.      0.
GRID      3      0.      1.      0.
GRID      4      1.      1.      0.
GRID      5      0.      0.      1.
GRID      6      1.      0.      1.
GRID      7      0.      1.      1.
GRID      8      1.      1.      1.
```

PROJECT 8, QVOL (TIME)

Extracted from MD Nastran model .DAT file

```
$ Loads for Load Case : tran
TLOAD1  4      3      1
DLOAD   2      1.    1.    4
$ Fixed Temperatures of Load Set : conv
SPC      1      9      1    0.
$ Volumetric Heat Generation of Load Set : qvol
QVOL     3      10000.    1
$ Scalar Points
SPOINT   9
```

```
$ Dynamic Load Table : qvol_time
TABLED1  1
+      C 0.    0.    1000.  1.    2000.  0.    3000.  0.    +      C
+      D ENDT
```

PROJECT 8, QVOL (TIME)

Extracted from MD Nastran model .DAT file

\$ Initial Temperatures from Temperature Load Sets

TEMP 1 9 0.

\$ Default Initial Temperature

TEMPD 1 0.

\$ CHBDYG Surface Elements

CHBDYG	1		AREA4			+	E
+	E	1	2	6	5		
CHBDYG	2		AREA4			+	F
+	F	3	7	8	4		
CHBDYG	3		AREA4			+	G
+	G	1	3	4	2		
CHBDYG	4		AREA4			+	H
+	H	2	4	8	6		
CHBDYG	5		AREA4			+	I
+	I	6	8	7	5		
CHBDYG	6		AREA4			+	J
+	J	5	7	3	1		

PROJECT 8, QVOL (TIME)

Extracted from MD Nastran model .DAT file

```
$ Convection to Ambient of Load Set : conv
PCONV    1      1001    0      0.
CONV     1        1      0      0      9
CONV     2        1      0      0      9
CONV     3        1      0      0      9
CONV     4        1      0      0      9
CONV     5        1      0      0      9
CONV     6        1      0      0      9
```

```
$ Free Convection Heat Transfer Coefficients
MAT4      1001                                100.
```

PROJECT 8, QVOL (TIME)

- Bulk Data entry TSTEPNL used to control the nonlinear solver

Extracted from MD Nastran model .DAT file

```
TSTEPNL ,1, 6000, 1., 1 , ADAPT, 2 , -10, UPW, +  
+ , .01, .001 , 1.-6
```

- The value for the recommended initial time step is

$$\Delta t|_{initial} = \frac{1}{n} (\Delta x)^2 (\rho C_p / \kappa)$$

- where Δx = minimum element length
 κ = minimum conductivity for the model
For the value of n use 10

PROJECT 8, QVOL (TIME)

- For this model (a single 1x1x1 CHEXA element) this formula gives

- Aluminum

- $\kappa = 204.0 \text{ W/(m}^* \text{ }^{\circ}\text{C)}$
- $C_p = 896.0 \text{ J/(kg}^* \text{ }^{\circ}\text{C)}$
- $\rho = 2707.0 \text{ kg/m}^3$

- $\Delta x = 1.0 \text{ m}$

- The estimate of the initial time step is found from

$$\Delta t |_{initial} = \frac{1}{n} (\Delta x)^2 (\rho C_p / \kappa)$$

- The initial time step from this formula is 1189.0 sec. This value is >> 1.0 sec.

- Lets see what happens to the time step size as the nonlinear solver adjusts it

PROJECT 8, QVOL (TIME)

- The initial time step is 1.0 sec, but the adaptive time step strategy will increase the time step, as indicated in the ADJUST column

Extract from MD Nastran ASCII Results File

0 NON - LINEAR ITERATION MODULE OUTPUT														
STIFFNESS UPDATE TIME										SUBCASE 1				
ITERATION TIME														
TIME	ITER	DISP.	LOAD	ERROR	FACTORS	CONVERGENCE	FACTORS	LINE SEARCH DATA	ITR MAT NO.					
					WORK	LAMBDA(I)	DL MAG	E-FIRST	E-FINAL	NONV	NLS	DIV	DIV	BIS ADJUST
1.000000E+00	2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	1.9984E-15	1.000	0.000E+00	0.000E+00	0	0	0	1 1.000
2.000000E+00	2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	6.6613E-15	1.000	0.000E+00	0.000E+00	0	0	0	1 1.000
*** USER INFORMATION MESSAGE 6204 *** 0.010 SECONDS REQUIRED TO DECOMPOSE MATRIX.														
6.000000E+00	2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	1.0658E-14	1.000	0.000E+00	0.000E+00	0	0	0	1 4.000
*** USER INFORMATION MESSAGE 6204 *** 0.000 SECONDS REQUIRED TO DECOMPOSE MATRIX.														
2.200000E+01	2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	1.1013E-13	1.000	0.000E+00	0.000E+00	0	0	0	1 16.000
3.800000E+01	2	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	0.0000E+00	1.5632E-13	1.000	0.000E+00	0.000E+00	0	0	0	1 16.000

PROJECT 8, QVOL (TIME)

Extract from MD Nastran ASCII Results File

```
0
SUBCASE 1
  POINT-ID = 1
  T E M P E R A T U R E   V E C T O R

    TIME          TYPE          VALUE
  0.0             S             0.0
  1.000000E+00    S             2.082017E-05
  2.000000E+00    S             8.285914E-05
  6.000000E+00    S             7.454431E-04
  2.200000E+01    S             1.001259E-02
  3.800000E+01    S             2.977672E-02
  5.400000E+01    S             5.999637E-02
  7.000000E+01    S             1.006302E-01
  8.600000E+01    S             1.516372E-01
  1.020000E+02    S             2.129763E-01
  1.180000E+02    S             2.846066E-01
  1.340000E+02    S             3.664876E-01
  1.500000E+02    S             4.585788E-01
```

- It is seen that the first two time steps are 1.0 sec, and that the next two time steps are 4.0 sec and 16.0 sec

PROJECT 8, QVOL (TIME)

- If a fixed time step is desired, then adaptive time stepping can be deactivated by setting ADJUST = 0 for the TSTEPNL entry
 - TSTEPNL, 1, 6000, 1.0, 1 , ADAPT, 2 , -10, UPW, +
+, 0.01, 0.001, 1.0-6, , , , , +
+, , 0 (ADJUST value goes here)

PROJECT 8, QVOL (TIME)

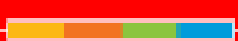
- Deactivate the automatic adjustment (adaptivity)

TSTEPNL	1	6000	10.0	1	ADAPT	UPW	++000001
							++000002
		0					

Extracted from MD Nastran
model .DAT file

```
0
SUBCASE 1
  POINT-ID = 3
  T E M P E R A T U R E   V E C T O R

    TIME          TYPE          VALUE
    0.0           S            0.0
    1.000000E+01  S            2.061824E-03
    2.000000E+01  S            8.236363E-03
    3.000000E+01  S            1.851345E-02
    4.000000E+01  S            3.288296E-02
    5.000000E+01  S            5.133478E-02
    6.000000E+01  S            7.385881E-02
    7.000000E+01  S            1.004450E-01
    8.000000E+01  S            1.310833E-01
    9.000000E+01  S            1.657638E-01
    1.000000E+02  S            2.044763E-01
    1.100000E+02  S            2.472110E-01
```



PROJECT 8, QVOL (TIME)

- Use the SET command in the Case Control Section for output of temperatures
 - CEND
 - .
 - SET 77 = 3,6,7
 - THERMAL = 77
 - .
 - BEGIN BULK
 - .
 - Define GRIDs 3, 6, 7
 - .
 - ENDDATA
 - In this case, output the temperature at the GRID points listed in SET 77 (GRID 3, 6, 7).

PROJECT 8, QVOL (TIME)

- Create XY plots of temperature
 - OUTPUT(XYPLOT)
 - XGRID=YES
 - YGRID=YES
 - XTITLE= TIME
 - YTITLE= TEMP
 - XYPLOT TEMP/3(T1)
- Read the material on creating an XY plot in the MD Nastran Quick Reference Guide and MD Nastran 2002 Thermal Analysis User's Guide for more details

PROJECT 8, QVOL (TIME)

- Specify at what times output is desired
 - CEND
 - ...
 - SET 99 = 0.0,100.0,200.0,500.0
 - OTIME = 99
 - ...
 - BEGIN BULK

PROJECT 8, QVOL (TIME)

- Use OTIME to specify when (at what times) output is written

Extract from MD Nastran ASCII Results File

```
0
SUBCASE 1
  POINT-ID =          3
  T E M P E R A T U R E   V E C T O R

      TIME           TYPE           VALUE
0.0              S             0.0
1.000000E+02      S             2.046509E-01
2.000000E+02      S             8.115298E-01
5.000000E+02      S             4.948450E+00
```

PROJECT 8, QVOL (TIME)

- What is the difference between SORT1 and SORT2 output ?
- SORT1 output

Extract from MD Nastran ASCII Results File

```
0
SUBCASE 1
  TIME = 5.000000E+02
      T E M P E R A T U R E   V E C T O R

POINT ID.    TYPE      ID    VALUE      ID+1 VALUE      ID+2 VALUE      ID+3 VALUE      ID+4 VALUE      ID+5 VALUE
    1         S      4.948450E+00  4.948450E+00  4.948450E+00  4.948450E+00  4.948450E+00  4.948450E+00
    7         S      4.948450E+00  4.948450E+00  0.0
```

- SORT1 output is used to display the results from a thermal analysis, for a specified time for all specified grids or elements
- SORT2 output
 - SORT2 output is used to display the results from a thermal analysis, for a specified grid or element for all specified times

PROJECT 8, QVOL (TIME)

- Specify output using SORT2
 - SORT2 format is requested using
THERMAL (SORT2,PRINT,PUNCH) = ALL
THERMAL (SORT2,PRINT,PUNCH) = 77
FLUX (SORT2,PRINT) = ALL
- Note, if OUTPUT (XYPLOT) is in the Case Control Section, it must be commented out, e.g. \$ added in the front of OUTPUT, to obtain SORT1 output. This is because, by default, XYPLOT uses SORT2 output.

PROJECT 8, QVOL (TIME)

- Results in SORT1 format in punch file.

Extract from MD Nastran ASCII Punch File

\$TITLE	=	MSC/NASTRAN JOB	49	
\$SUBTITLE	=	TRAN	50	
\$LABEL	=		51	
\$DISPLACEMENTS			52	
\$REAL OUTPUT			53	
\$SUBCASE ID	=	1	54	
\$TIME	=	0.5000000E+03	55	
TEMP*	4	1	4.948450E+00	56
TEMP*	4	2	4.948450E+00	57
TEMP*	4	3	4.948450E+00	58
TEMP*	4	4	4.948450E+00	59
TEMP*	4	5	4.948450E+00	60
TEMP*	4	6	4.948450E+00	61
TEMP*	4	7	4.948450E+00	62
TEMP*	4	8	4.948450E+00	63
TEMP*	4	9	0.000000E+00	64

EXERCISES

- Perform Workshop 8 “Transient Thermal” in the exercise workbook.
- Perform Workshop 9 “Transient Thermal Analysis of a Cooling fin” in the exercise workbook.
- Perform Workshop 10 “Holman example 4-14, Radiation Heating and Cooling of Ceramics” in the exercise workbook.
- Perform Workshop 11 “Heating a Block of Ice Cream” in the exercise workbook.

CONTROL BY THERMOSTAT



BC CONTROL THROUGH CNTRLND

1	2	3	4	5	6	7	8	9	10
QBDY3	SID	Q0	CNTRLND	EID1	EID2	EID3	EID4	EID5	
QBDY3	101	0.04	777	1001	THRU	1004			

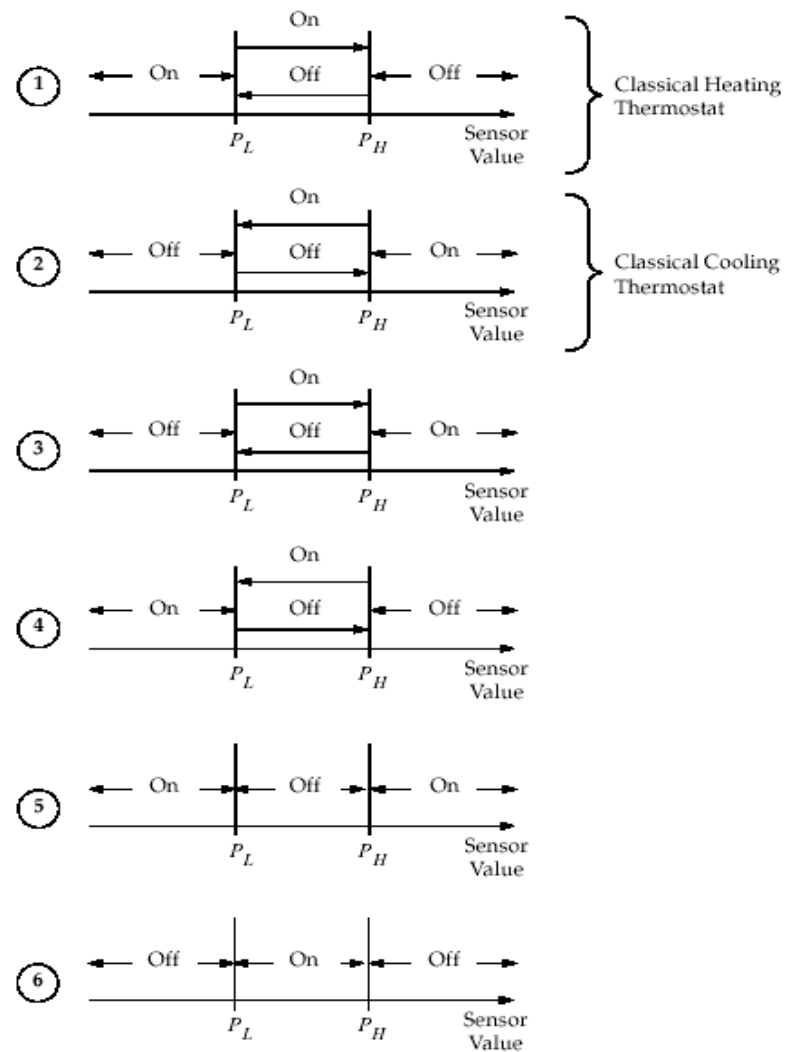
- The CONTRLT entry (thermal control mechanism) is referenced by the CNTRLND entry identified on any of the **QVECT**, **QVOL**, **QBDY3**, **CONV**, **CONVM**, and **RADBC** Bulk Data entries.
- Note, that thermostat can only be used in an MD Nastran transient thermal analysis (e.g. SOL 159).

CONTROL MECHANISM, CONTRLT

1	2	3	4	5	6	7	8	9	10
CONTRLT	ID	SENSOR	SFORM	CTYPE	PL	PH	PTYPE	PZERO	
CONTRLT	777	57	T	TSTAT	65.0	75.0	1	0.0	

- ID – control node ID
- Sensor – grid or scalar point ID of the sensor
- SFORM – sensor output form (only T)
- CTYPE – control type (only TSTAT)
- PL, PH – lower and upper limit value for desired temperature of the thermostat (real values)
- PTYPE – process type (Form 1 to 6); see next page
- PZERO – initial controller value (only 0.0 or 1.0)
- See MD Nastran Quick Reference Guide for PTYPE form

PTYPE 1 to 6



NAS104, Section 5, September 2008

Copyright© 2008 MSC Software Corporation

S5-81

EXAMPLE A, LUMPED CAPACITY ANALYSIS WITH THERMOSTAT USING QVOL

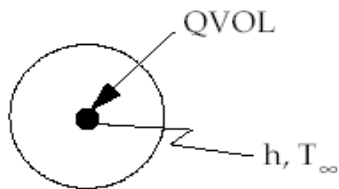


Figure 10-4 Lumped Heat Capacitance Model

- Lumped heat capacitance with thermostat control of a volumetric heat source (test1a_fix.dat).
- A general assumption for the lumped heat capacitance model is that the Biot number, $Bn = h(\text{volume}/\text{area})/k$, is < 0.1 . If this is the case, a lumped capacity type of analysis assumes a uniform temperature distribution in the solid, and surface convection is large relative to the internal conduction resistance. Furthermore, a simple first order differential equation can be used to arrive at the analytical solution.

EXAMPLE A, ANALYTICAL SOLUTION

- It is assumed that the temperature varies with time, but not spatially (x,y,z) in the body (mass). An energy balance yields

$$\rho v C_p \dot{T} = \dot{q}_s - \dot{q}_c$$

- where

$$\begin{aligned} \rho v C_p &= 6000 & \dot{q}_s &= 12000 & \dot{q}_c &= h A (T - T_\infty) = h A T \\ hA &= 100 & T_\infty &= 0 & B &= hA/(\rho v C_p) = 1/60 & T(0) &= 0 \end{aligned}$$

- Start at time of zero to solve this equation, $T(0) = 0$

$$\dot{T} + B T = 2$$

$$T(t) = 120(1 - e^{-Bt})$$

EXAMPLE A, ANALYTICAL SOLUTION

- Determine the time when the temperature reaches 100 °C for the first time.

$$100 = 120(1 - e^{-Bt_1}) \quad \Rightarrow \quad total_1 = t_1 = 107.51 \text{ sec}$$

EXAMPLE A, ANALYTICAL SOLUTION

- Now, that the temperature is at 100 °C, turn the heater off.

$$\rho v C_p \dot{T} = -\dot{q}_c$$

- Start at time of zero to solve this equation, $T(0) = 100$

$$\dot{T} + B T = 0$$

$$T(t) = 100 e^{-Bt}$$

- Determine the time when the temperature has dropped to 80 °C for the first time.

$$80 = 100 e^{-Bt_2} \Rightarrow t_2 = 13.39 \text{ sec}$$

- The total time is

$$total_2 = t_1 + t_2 = 107.51 + 13.39 = 120.90 \text{ sec}$$

EXAMPLE A, ANALYTICAL SOLUTION

- Now, that the temperature has dropped to 80 °C, turn the heater on.

$$\rho v C_p \dot{T} = \dot{q}_s - \dot{q}_c$$

- Start at time of zero to solve this equation, $T(0) = 80$

$$\dot{T} + B T = 2$$

$$T(t) = C_1 + C_2 e^{-Bt}$$

- Because there are two constants in the solution, it is necessary to have another time condition. This condition is chosen at time of infinity (steady state condition).

$$0 = \dot{q}_s - \dot{q}_c \quad h A T(\infty) = 12000 \quad T(\infty) = 120$$

$$T(t) = 120 - 40e^{-Bt}$$

EXAMPLE A, ANALYTICAL SOLUTION

- Determine the time when the temperature reaches 100 °C for the second time.

$$100 = 120 - 40e^{-Bt_3} \quad \Rightarrow \quad t_3 = 41.59 \text{ sec}$$

- The total time is

$$total_3 = t_1 + t_2 + t_3 = 107.51 + 13.39 + 41.59 = 162.49 \text{ sec}$$

EXAMPLE A, QVOL BULK DATA ENTRY

QVOL

Volume Heat Addition

Defines a rate of volumetric heat addition in a conduction element.

Format:

1	2	3	4	5	6	7	8	9	10
QVOL	SID	QVOL	CNTRLND	EID1	EID2	EID3	EID4	EID5	
	EID6	etc.							

Example:

QVOL	5	10.0	101	10	12	11	9		

Field	Contents
SID	Load set identification. (Integer > 0)
QVOL	Power input per unit volume produced by a heat conduction element. (Real)
CNTRLND	Control point used for controlling heat generation. (Integer ≥ 0; Default = 0)
EIDi	A list of heat conduction elements. (Integer > 0 or "THRU" or "BY")

Unlike the QBDY3 who's thermal heat flux load is applied to CHBDYi surface elements, the QVOL power input per unit volume is applied to conduction elements. This is a volmetric heat load (power in/volume).

EXAMPLE A, CONTRLT BULK DATA ENTRY

CONTRLT

Thermal Control Element for Heat Transfer Analysis

Defines the control mechanism for QVECT, QVOL, QBCY3, in heat transfer analysis (SOL 159).

Format:

1	2	3	4	5	6	7	8	9	10
CONTRLT	ID	Sensor	SFORM	CTYPE	Pl	Ph	PTYPE	PZERO	
	DT	Delay	TAUc	TA8					

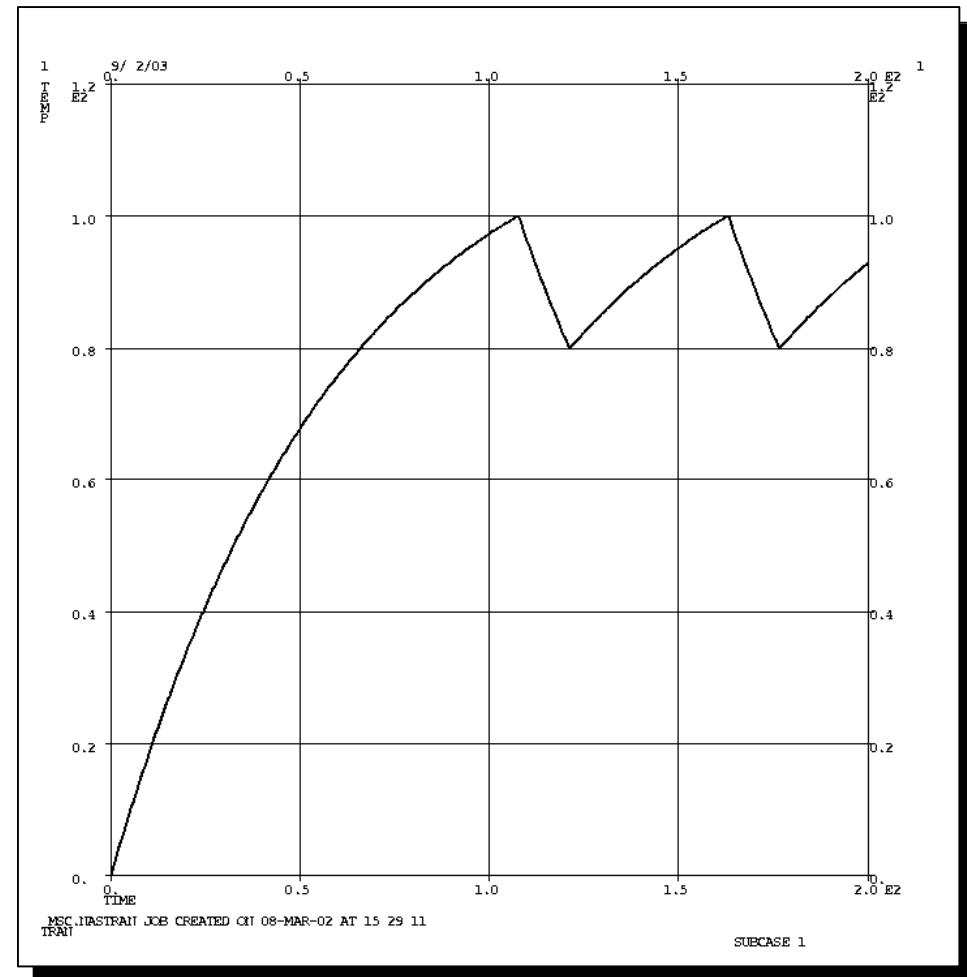
Example:

CONTRLT	100	20	3	68.	73.	1	0.		

Field	Contents
ID	Control node ID as well as CONTROLT ID. See Remark 1. (Integer > 0; no Default)
Sensor	Grid or scalar point ID of the sensor. See Remark 2. (Integer > 0; no Default)
SFORM	Sensor output form. See Remark 3. (Character, T; Default = T)
CTYPE	Control type. See Remark 4. (Character, TSTAT for thermostat; Default = TSTAT)
Pl, Ph	Lower and upper limit value for desired temperature in the thermostat. See Remark 5. (Real; no Default)
PTYPE	Process type. See Remark 5. (Integer value 1 through 6; no Default)
PZERO	Initial controller value. See Remark 4. (0. < Real < 1.; Default = 0)
DT	Monitoring time interval, or sampling period. (Real > 0.; Default = 0)
Delay	Time delay after the switch is triggered or time for delayed control action in PID control. (Real < 0.; Default = 0)
TAUc	Decay time constant for actuator response. (Real > 0.; Default = 0)

EXAMPLE A, LUMPED CAPACITY MODEL

- In order to create a finite element model of a lumped system, a single CHEXA element that has the same thermal mass and convection coefficient, as previously shown in Figure 10-4, is used.
- Initially, the element is at zero °C, and is heated by the internal heat generation with 12000 Watts.
- Furthermore, free convection occurs on the six faces on the CHEXA element.
- The heater is designed to maintain the temperature between 80 and 100 °C. When the temperature reaches 100 °C, the heater is turned off, and when the temperature drops to 80 °C, the heater is turned back on.
- The CONTRLT Bulk Data entry allows the user to control the temperature to vary within two bounds.



EXAMPLE A, MD NASTRAN BULK DATA INPUT Test1a_fix.dat

```
SOL 159
CEND
ANALYSIS = HEAT
TITLE = MD Nastran job created on 08-Mar-02 at 15:29:11
ECHO = SORT
MAXLINES = 999999999
SPC = 1
IC = 1
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : tran
  SUBTITLE=tran
  TSTEPNL = 1
  DLOAD = 2
  THERMAL (SORT2, PRINT) =ALL
  FLUX (SORT2, PRINT) =ALL
  OLOAD (SORT2, PRINT) =ALL
  SPCFORCES (SORT2, PRINT) =ALL
OUTPUT (XYPLOT)
XGRID=YES
YGRID=YES
XTITLE=time
YTITLE=temp
XYPLOT TEMP/2 (T1)
BEGIN BULK
CONTRLT, 99, 1, T, TSTAT, 80.0, 100.0, 1, 1.0, +
+, 0.0, 0.0, 0.0, 0.0
PARAM   PRGPST NO
TSTEPNL, 1, 2000, 0.10, 5, ADAPT, 2, -10, UPW
```

EXAMPLE A, MD NASTRAN BULK DATA

INPUT Test1a_fix.dat

```
PSOLID 1 1 0
$ Pset: "solid" will be imported as: "psolid.1"
CHEXA 1 1 1 2 4 3 5 6
      8 7
$ Referenced Material Records
$ Material Record : alum
$ Description of Material : Date: 08-Mar-02 Time: 15:24:54
MAT4 1 204. 60. 100.
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
GRID 2 1. 0. 0.
GRID 3 0. 1. 0.
GRID 4 1. 1. 0.
GRID 5 0. 0. 1.
GRID 6 1. 0. 1.
GRID 7 0. 1. 1.
GRID 8 1. 1. 1.
$ Loads for Load Case : tran
TLOAD1 4 3 1
DLOAD 2 1. 1. 4
$ Fixed Temperatures of Load Set : fixed temperature
SPC 1 9 1 0.
```

EXAMPLE A, MD NASTRAN BULK DATA

INPUT Test1a_fix.dat

```

$ Volumetric Heat Generation of Load Set : qvol
$QVOL, SID, QVOL, CNTRLND, EID1, EID2, etc.
QVOL,3,12000.,99,1
$ Convection to Ambient of Load Set : convection
PCONV 1 1001 0 0.
$CONV, EID, PCONID, FLMND, CNTRLND, TA1, TA2,
CONV 100001 1 0 0 9
CONV 100002 1 0 0 9
CONV 100003 1 0 0 9
CONV 100004 1 0 0 9
CONV 100005 1 0 0 9
CONV 100006 1 0 0 9
$ Time Dependent Nodal Temperatures of Load Set :
convection
$ Referenced Dynamic Load Tables
$ Constant Load Table
TABLED1 1
0. 1. 1000. 1. ENDT
$ Initial Temperatures from Temperature Load Sets
TEMP 1 9 0.
$ Default Initial Temperature
TEMPD 1 0.

```

```

$ CHBDYG Surface Elements
CHBDYG 100001 AREA4
5 7 3 1
CHBDYG 100002 AREA4
2 4 8 6
CHBDYG 100003 AREA4
1 2 6 5
CHBDYG 100004 AREA4
3 7 8 4
CHBDYG 100005 AREA4
1 3 4 2
CHBDYG 100006 AREA4
6 8 7 5
$ Free Convection Heat Transfer
Coefficients
MAT4 1001 16.6667
$ Scalar Points
SPOINT 9
$ Referenced Coordinate Frames
ENDDATA 259d5d50

```

EXAMPLE A, MD NASTRAN OUTPUT TEMPERATURE

POINT-ID = 1

TEMPERATURE VECTOR

TIME	TYPE	VALUE
1.000000E+02	S	9.731579E+01
1.005000E+02	S	9.750403E+01
1.010000E+02	S	9.769072E+01
1.015000E+02	S	9.787585E+01
1.020000E+02	S	9.805945E+01
1.025000E+02	S	9.824152E+01
1.030000E+02	S	9.842209E+01
1.035000E+02	S	9.860116E+01
1.040000E+02	S	9.877873E+01
1.045000E+02	S	9.895484E+01
1.050000E+02	S	9.912949E+01
1.055000E+02	S	9.930268E+01
1.060000E+02	S	9.947444E+01
1.065000E+02	S	9.964477E+01
1.070000E+02	S	9.981369E+01
1.075000E+02	S	9.998121E+01
1.080000E+02	S	9.944842E+01

EXAMPLE A, MD NASTRAN OUTPUT TEMPERATURE

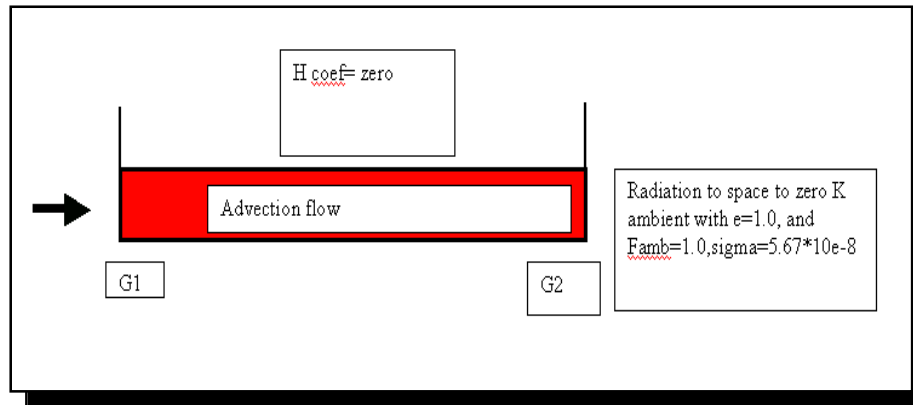
1.115000E+02	S	9.381325E+01
1.120000E+02	S	9.303472E+01
1.125000E+02	S	9.226266E+01
1.130000E+02	S	9.149701E+01
1.135000E+02	S	9.073770E+01
1.140000E+02	S	8.998470E+01
1.145000E+02	S	8.923795E+01
1.150000E+02	S	8.849740E+01
1.155000E+02	S	8.776299E+01
1.160000E+02	S	8.703467E+01
1.165000E+02	S	8.631240E+01
1.170000E+02	S	8.559612E+01
1.175000E+02	S	8.488579E+01
1.180000E+02	S	8.418135E+01
1.185000E+02	S	8.348276E+01
1.190000E+02	S	8.278996E+01
1.210000E_02	S	8.000000E+01
1.620000E+02	S	9.968965E+01
1.625000E+02	S	9.985819E+01
1.630000E+02	S	1.000253E+02
1.635000E+02	S	9.929352E+01

EXAMPLE A, MD NASTRAN RESULTS VS ANALYTICAL

	Time of thermostat on or off	
Temperature (°C)	Analytical (Seconds)	MD Nastran (Seconds)
100	107.51	107.50
80	120.90	121.00
100	162.49	163.00

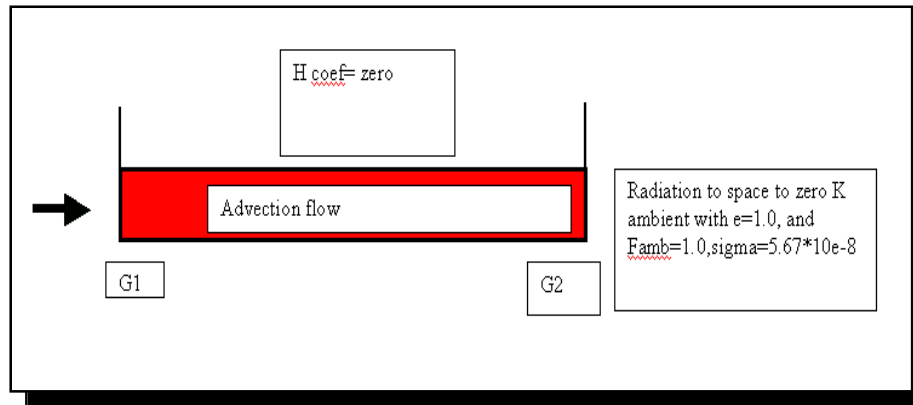
- MD NASTRAN results are in excellent agreement with the theory.

EXAMPLE B, THERMOSTAT CONTROLLER USING RADIATION OR ADVECTION



- The designer wants to control the sensor temperature to be within a band from 400 °C to 500 °C, so that the electronics will function properly.
- Heat is injected into the system through advection. Heat is radiated to space.
- Case 1
- No temperature control is used for this case.
- This case is performed to aid the designing of the system (it must be achievable).

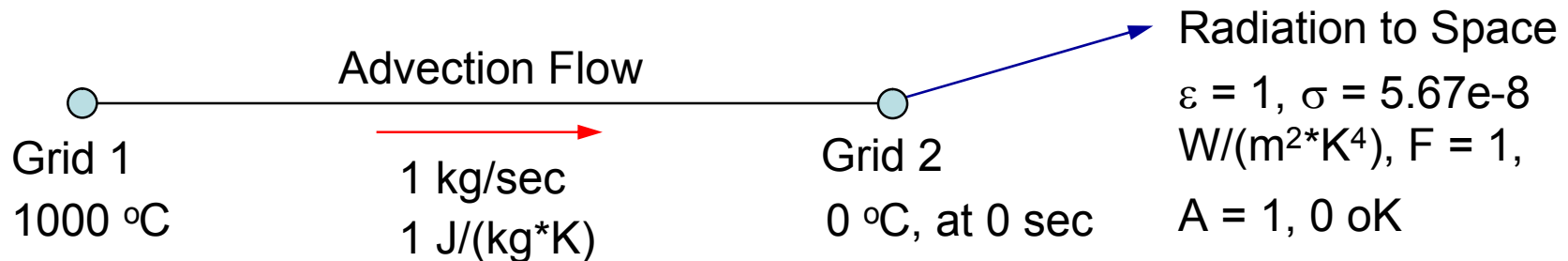
EXAMPLE B, THERMOSTAT CONTROLLER USING RADIATION OR ADVECTION



- Case 2
- Radiation to ambient (space) is used, by turning it on or off, to control the sensor temperature.
- Case 3
- Change the desired temperature band to 250 °C to 300 °C.
- Use advection (mass flow) to control the sensor temperature.

EXAMPLE B, CASE 1, NO CONTROLLER

- The simple forced convection and radiation system model



- Energy balance

$$\dot{m} C_p (T_1 - T_2) = \varepsilon \sigma F A (T_2^4 - T_\infty^4)$$

- Solve this equation for T_2

$$T_2 = 329.74 \text{ } ^\circ\text{C}$$

- Now, solve this problem using MD Nastran

EXAMPLE B, CASE 1, NO CONTROLLER

convm_control1.dat

CONVM

$K = 0.01 \text{ W/(m}^2\text{K)}$

$\dot{M} = 1.0 \text{ kg/sec}$

$C_p = 1.0 \text{ J/(Kg}^{\circ}\text{K)}$

RADBC

$\epsilon = 1.0$

$F_{amb} = 1.0$

$\sigma = 5.67\text{e-}8 \text{ W/(m}^2\text{K}^4)$

Let $T(\text{Grid 2}, t = 0) = 0$

$T(\text{Grid1}) = 1000$

SOL 159

\$ Direct Text Input for Executive Control

CEND

ANALYSIS = HEAT

TITLE = MD Nastran job created on 15-Aug-03 at 11:45:10

ECHO = NONE

SPC = 1

IC = 1

\$ Direct Text Input for Global Case Control Data

SUBCASE 1

\$ Subcase name : tran

SUBTITLE=tran

TSTEPNL = 1

THERMAL(SORT2,PRINT)=ALL

FLUX(SORT2,PRINT)=ALL

OLOAD(SORT2,PRINT)=ALL

SPCFORCES(SORT2,PRINT)=ALL

\$ Direct Text Input for this Subcase

BEGIN BULK

PARAM POST 0

PARAM* SIGMA 5.6696-8

PARAM PRGPST NO

TSTEPNL 1 200 .5 2 ADAPT 2 -10 U

EXAMPLE B, CASE 1, NO CONTROLLER

convm_control1.dat (Cont.)

```

MAT4 1 .01 1. 50.
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
GRID 2 1. 0. 0.
$ Loads for Load Case : tran
$ Fixed Temperatures of Load Set : nodal_radbc
$SPC, SID, G1, C1, D1
SPC 1 3 1 0.
$ Fixed Temperatures of Load Set : flowtube
SPC 1 4 1 0.
SPC 1 5 1 1.
$ Fixed Temperatures of Load Set : inlet_temp
SPC 1 1 1 1000.
$ Tube Flow Convection to Ambient of Load Set : flowtube
$PCONVM, PCONID, MID, FORM, FLAG
PCONVM 1 1 0 1 0. 0. 0. 0.
$CONVM, EID, PCONID, FLMND, CNTMDOT, TA1
CONVM 100001 1 0 5 4
$ Radiation Boundary Conditions of Load Set : nodal_radbc
$RADBC, NODAMB, FAMB, CNTRLND, EID1
RADBC 3 1. 0 100002

```

```

$TEMP, SID, G1, T1, G2, T2, G3, T3
TEMP 1 1 1000. 3 0. 4 0.
TEMP 1 5 1.
$ Default Initial Temperature
TEMPD 1 0.
$ CHBDYP Surface Elements and Element Properties
$CHBDYP, EID, PID, TYPE,, G1, G2
CHBDYP 100001 1 FTUBE 1 2
$, RADMIDF,,, E1, E2, E3
.57735 .57735 .57735
CHBDYP 100002 2 POINT 2
1 .57735 .57735 .57735
$PHBDY, PID, AF, D1, D2
PHBDY 1 1.12838 1.12838
PHBDY 2 1.
$ Radiation Material Properties
RADM 1 1. 1.
$ Scalar Points
SPOINT 3 THRU 5
$ Referenced Coordinate Frames
ENDDATA d93a9e2d

```

EXAMPLE B, MD NASTRAN OUTPUT

- The solution approaches the value that was previously determined, 329.74 °C.
- There is no control of the temperature.

0			SUBCASE 1		
POINT-ID = 2			TEMPERATURE VECTOR		
TIME	TYPE	VALUE			
0.0	S 0.0				
1.000000E+00	S	5.822008E+01			
2.000000E+00	S	1.128242E+02			
3.000000E+00	S	1.631460E+02			
4.000000E+00	S	2.077425E+02			
5.000000E+00	S	2.449214E+02			
6.000000E+00	S	2.736627E+02			
7.000000E+00	S	2.942279E+02			
8.000000E+00	S	3.079800E+02			
9.000000E+00	S	3.167094E+02			
1.000000E+01	S	3.220526E+02			
1.100000E+01	S	3.252703E+02			
1.200000E+01	S	3.271600E+02			
1.300000E+01	S	3.282572E+02			
1.400000E+01	S	3.288901E+02			
1.500000E+01	S	3.292537E+02			
1.600000E+01	S	3.294621E+02			
1.700000E+01	S	3.295814E+02			
1.800000E+01	S	3.296497E+02			
1.900000E+01	S	3.296887E+02			
2.000000E+01	S	3.297110E+02			
2.100000E+01	S	3.297237E+02			
2.200000E+01	S	3.297310E+02			
2.300000E+01	S	3.297352E+02			
2.400000E+01	S	3.297376E+02			
2.500000E+01	S	3.297389E+02			
2.600000E+01	S	3.297397E+02			
2.700000E+01	S	3.297401E+02			
2.800000E+01	S	3.297404E+02			
2.900000E+01	S	3.297405E+02			
3.000000E+01	S	3.297406E+02			
3.100000E+01	S	3.297407E+02			

EXAMPLE B, CASE 2, COOLING BY RAD

- Case 2
- Initially, there will be no radiation to space (RADBC), so the temperature of Grid 2 will increase due to the advection energy from the fluid. When the temperature of Grid 2 reached 500 degrees, the radiation to space will begin, causing the temperature at Grid 2 to decrease (towards 329.74 °C). When the temperature of Grid 2 becomes less than 400 degrees, the radiation to space will be stopped.
- A cooling thermostat type of cycle will be used. Initially the controller will be set to OFF.

EXAMPLE B, CASE 2, COOLING BY RAD

- In this example, the sensor grid point is Grid 2, PL is equal to 400.0, and the PH is equal to 500.0, and the classical cooling thermostat is used (PTYPE = 2).
- \$CONTRLT, ID, Sensor, SFORM, CTYPE, PL, PH, PTYPE
- CONTRLT, 777, 2,T, TSTAT, 400.0, 500.0, 2, 0.0, +
+, 0.0, 0.0, 0.0, 0.0
- The radiation to space is turned ON or OFF based on the sensor temperature for Grid 2.
-
- RADBC,3,1.0,777,100002
-
- NASTRAN input file is convm_control2_cool1.dat

EXAMPLE B, CASE 2, COOLING BY RADIATION convm_control2_cool1.dat

```
SOL 159
$ Direct Text Input for Executive Control
CEND
ANALYSIS = HEAT
TITLE = MD Nastran job created on 15-Aug-03 at 11:45:10
ECHO = NONE
SPC = 1
IC = 1
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : tran
  SUBTITLE=tran
  TSTEPNL = 1
  THERMAL(SORT2,PRINT)=ALL
  FLUX(SORT2,PRINT)=ALL
  OLOAD(SORT2,PRINT)=ALL
  SPCFORCES(SORT2,PRINT)=ALL
$ Direct Text Input for this Subcase
  OUTPUT(XYLOT)
  XGRID=YES
  YGRID=YES
  XTITLE=TIME IN SECONDS
  YTITLE= TEMP FOR THE SENSOR
  CURVELINESYMB=6
  XYLOT TEMP/2(T1)
```

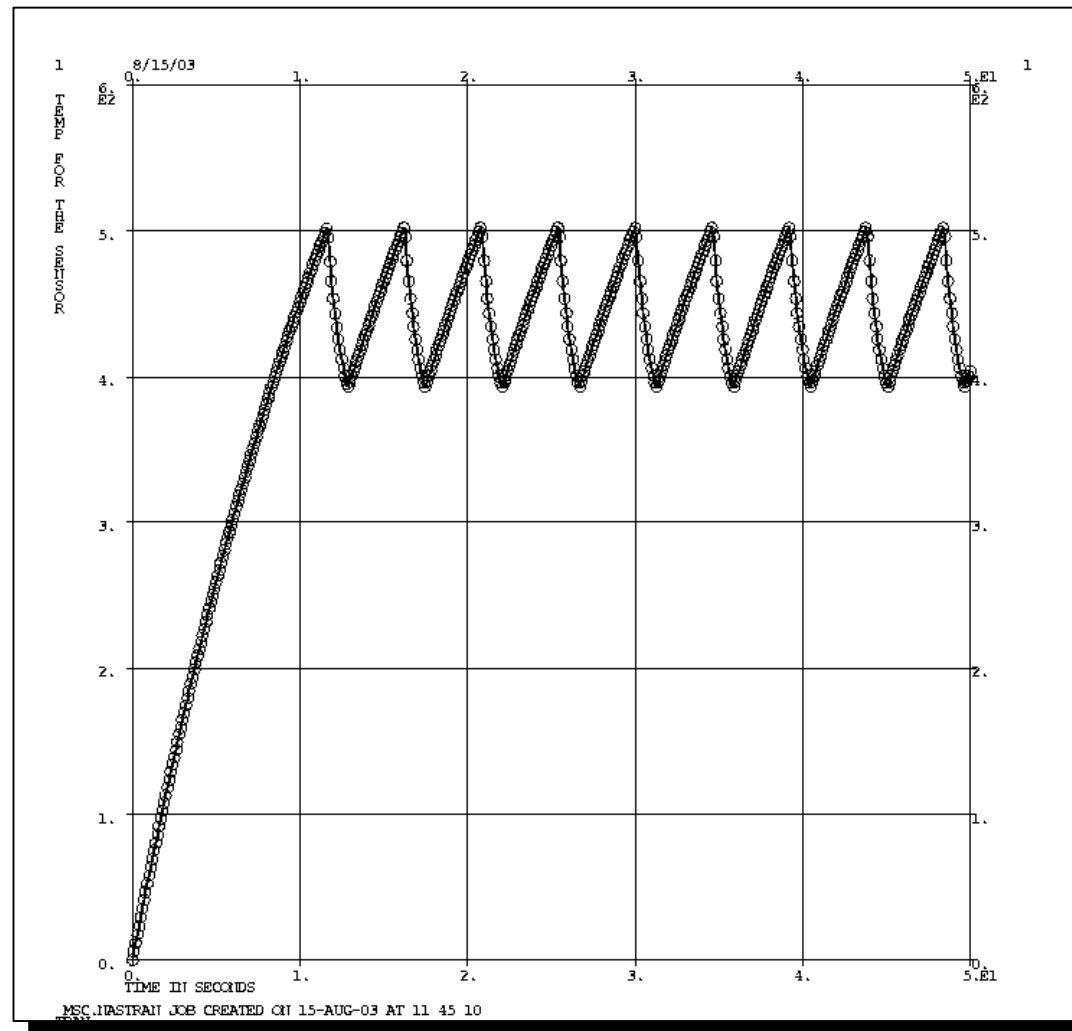
```
BEGIN BULK
$CONTRLT, ID, Sensor, SFORM, CTYPE, PL, PH, PTYPE, PZERO
CONTRLT, 777, 2,T, TSTAT, 400.0, 500.0, 2, 0.0 +
+, 0.0, 0.0, 0.0
PARAM POST 0
PARAM* SIGMA 5.6696-8
PARAM PRGPST NO
TSTEPNL 1 500 .1 1 ADAPT 2 -10 U
.01
MAT4 1 .01 1. 50.
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
GRID 2 1. 0. 0.
$ Loads for Load Case : tran
$ Fixed Temperatures of Load Set : nodal_radbc
SPC 1 3 1 0.
$ Fixed Temperatures of Load Set : flowtube
SPC 1 4 1 0.
```

EXAMPLE B, CASE 2, COOLING BY RADIATION convm_control2_cool1.dat

```
SPC 1 5 1 1.  
$ Fixed Temperatures of Load Set : inlet_temp  
SPC 1 1 1 1000.  
$ Tube Flow Convection to Ambient of Load Set : flowtube  
PCONVM 1 1 0 1 0. 0. 0. 0.  
CONVM 100001 1 0 5 4  
$ Radiation Boundary Conditions of Load Set : nodal_radbc  
$RADBC, NODAMB, FAMB, CNTRLND, EID1  
$RADBC 3 1. 0 100002  
RADBC, 3, 1.0, 777, 100002  
$ Initial Temperatures from Temperature Load Sets  
TEMP 1 1 1000. 3 0. 4 0.  
TEMP 1 5 1.  
$ Default Initial Temperature  
TEMPD 1 0.
```

```
$ CHBDYP Surface Elements and Element Properties  
CHBDYP 100001 1 FTUBE 1 2  
                  .57735 .57735 .57735  
CHBDYP 100002 2 POINT 2  
                  .57735 .57735 .57735  
PHBDY 1 1.12838 1.12838  
PHBDY 2 1.  
$ Radiation Material Properties  
RADM 1 1. 1.  
$ Scalar Points  
SPOINT 3 THRU 5  
$ Referenced Coordinate Frames  
ENDDATA d93a9e2d
```


EXAMPLE B, MD NASTRAN OUTPUT



NAS104, Section 5, September 2008

Copyright© 2008 MSC Software Corporation

S5-107

EXAMPLE B, CASE 3, HEATING BY MASS FLOW RATE

- The designer has the option to control the mass flow rate (turn it on or off) based on the sensor temperature.
- For this case, the temperature will be controlled to vary between 250 and 300 °C.
- For this case, there will be a constant radiation to space load (it will always be on).

EXAMPLE B, CASE 3, HEATING BY MASS FLOW RATE convm_control3b.dat

```
SOL 159
$ Direct Text Input for Executive Control
CEND
ANALYSIS = HEAT
TITLE = MD Nastran job created on 15-Aug-03 at 11:45:10
ECHO = NONE
SPC = 1
IC = 1
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : tran
  SUBTITLE=tran
  TSTEPNL = 1
  THERMAL(SORT2,PRINT)=ALL
  FLUX(SORT2,PRINT)=ALL
  OLOAD(SORT2,PRINT)=ALL
  SPCFORCES(SORT2,PRINT)=ALL
$ Direct Text Input for this Subcase
  OUTPUT(XYLOT)
  XGRID=YES
  YGRID=YES
  XTITLE=TIME IN SECONDS
  YTITLE= TEMP FOR THE SENSOR
  CURVELINESYMB=6
  XYPLOT TEMP/2(T1)
```

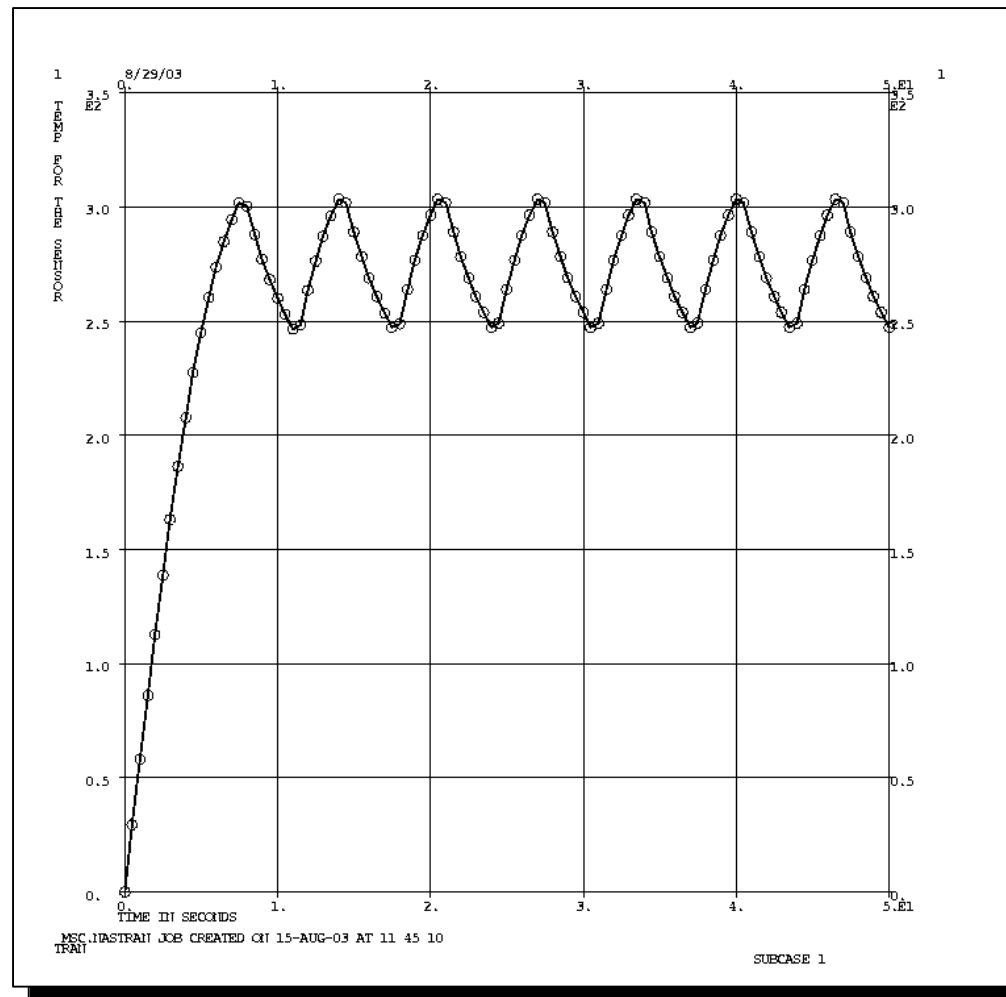
```
BEGIN BULK
$mdot = U(CNTMDOT)*Factor (=1.0)
$CONTRLT entry used to control temperature band
$Type 5f
$CONTRLT, ID, Sensor, SFORM, CTYPE, PL, PH, PTYPE, PZERO
$CONTRLT, 777, 2,T, TSTAT, 400.0, 500.0, 2, 0.0 +
$, 0.0, 0.0, 0.0
CONTRLT, 5, 2,T, TSTAT, 250.0, 300.0, 1, 1.0, +
+, 0.0, 0.0, 0.0
PARAM POST 0
PARAM* SIGMA 5.6696-8
PARAM PRGPST NO
TSTEPNL, 1, 100, 0.5, 1, ADAPT, 2, -10, U, 0.01
MAT4 1 .01 1. 50.
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
GRID 2 1. 0. 0.
$ Loads for Load Case : tran
$ Fixed Temperatures of Load Set : nodal_radbc
SPC 1 3 1 0.
$ Fixed Temperatures of Load Set : flowtube
SPC 1 4 1 0.
SPC 1 5 1 1.
$ Fixed Temperatures of Load Set : inlet_temp
SPC 1 1 1 1000.
```

EXAMPLE B, CASE 3, HEATING BY MASS FLOW RATE convm_control3b.dat (Cont.)

```
PCONVM 1 1 0 1 0. 0. 0. 0.
CONVM, 100001, 1,, 5, 4
RADBC 3 1. 0 100002
$ Time Dependent Nodal Temperatures of Load Set : nodal_radbc
$ Time Dependent Nodal Temperatures of Load Set : flowtube
$ Initial Temperatures from Temperature Load Sets
TEMP 1 1 1000. 3 0. 4 0.
TEMP 1 5 1.
$ Default Initial Temperature
TEMPD 1 0.
$ CHBDYP Surface Elements and Element Properties
CHBDYP 100001 1 FTUBE 1 2
.57735 .57735 .57735
CHBDYP 100002 2 POINT 2
1 .57735 .57735 .57735
PHBDY 1 1.12838 1.12838
PHBDY 2 1.
$ Radiation Material Properties
RADM 1 1. 1.
$ Scalar Points
SPOINT 3 THRU 5
$ Referenced Coordinate Frames
ENDDATA d93a9e2d
```

- The mass flow rate for control is defined as follows:
$$\dot{m} = U(\text{CNTMDOT}) * \text{Factor}$$
 - where
 $U(\text{CNTMDOT})$ is the mass flow rate corresponding to the control point CNTMDOT of the CONMV entry
Factor is the scale factor for Field 8 of the CONVM entry (QRG hidden field); default value is 1.0
- The temperature band control is performed using the CONTRLT entry. The node used for the control is the ID for the CONTRLT entry.

EXAMPLE B, SENSOR TEMPERATURE

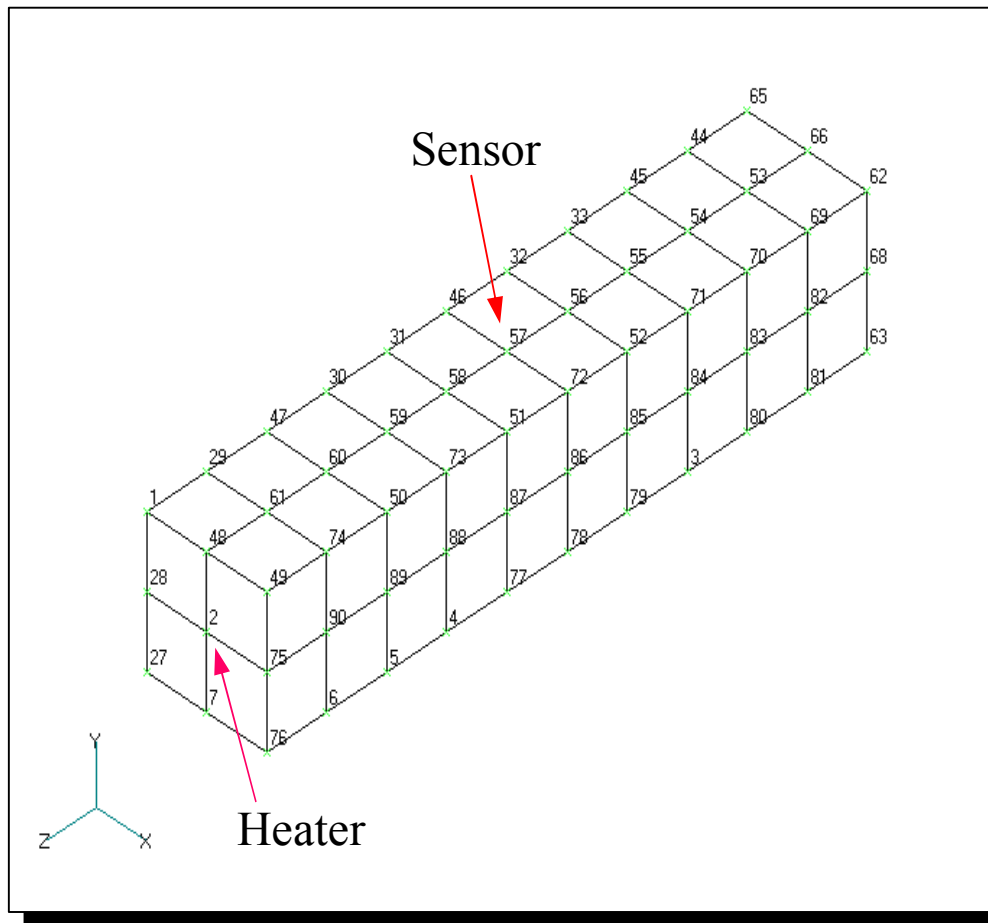




CLASSICAL HEATING THERMOSTAT

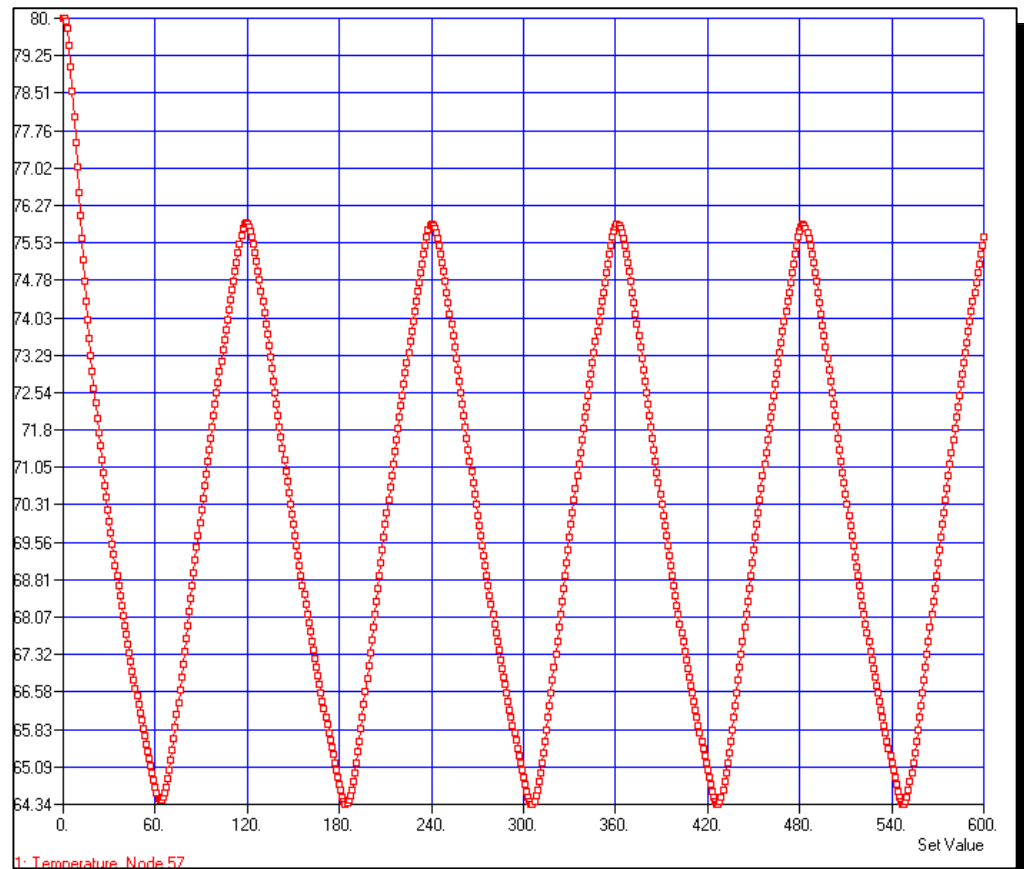


THERMOSTAT CONTROL BY HEATING

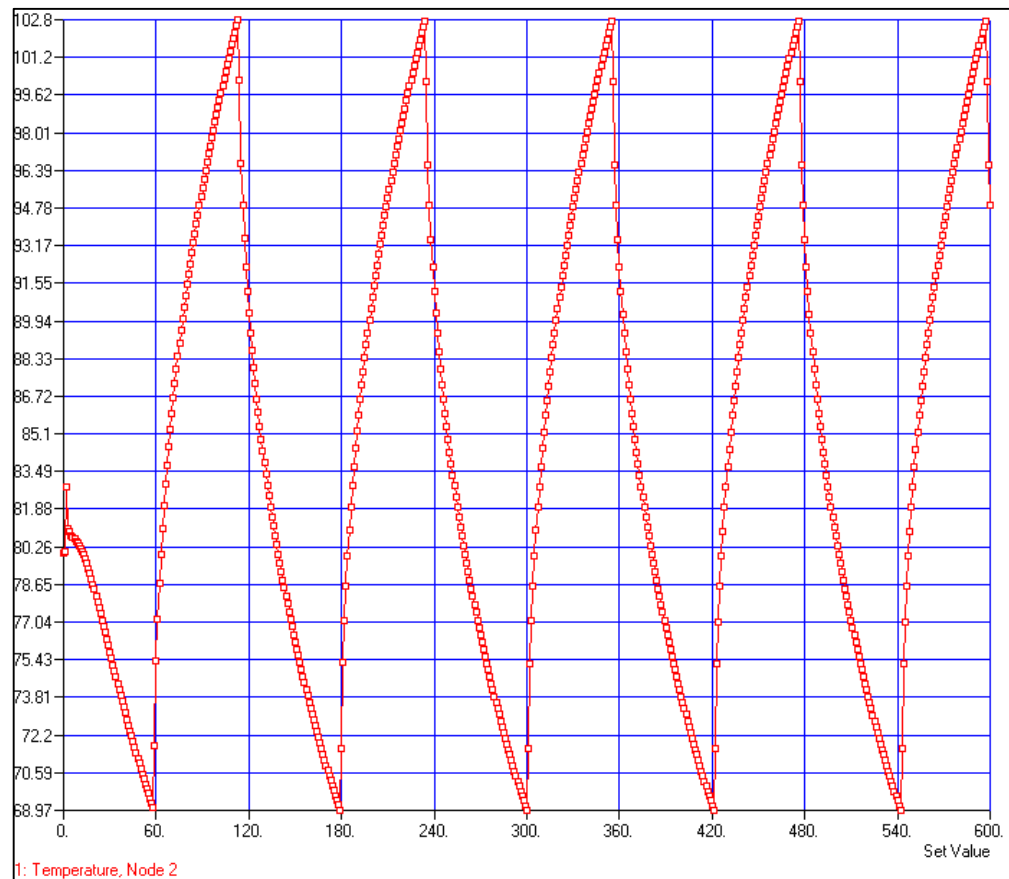


- The heater is on the left face of the model, and the sensor node is Node 57 which is at the top center of the model. The sensor monitors the temperature, as it varies from about 65° to 75° . When the temperature drops below 65° , the heater is turned on. When the temperature exceeds 75° , the heater is turned off.

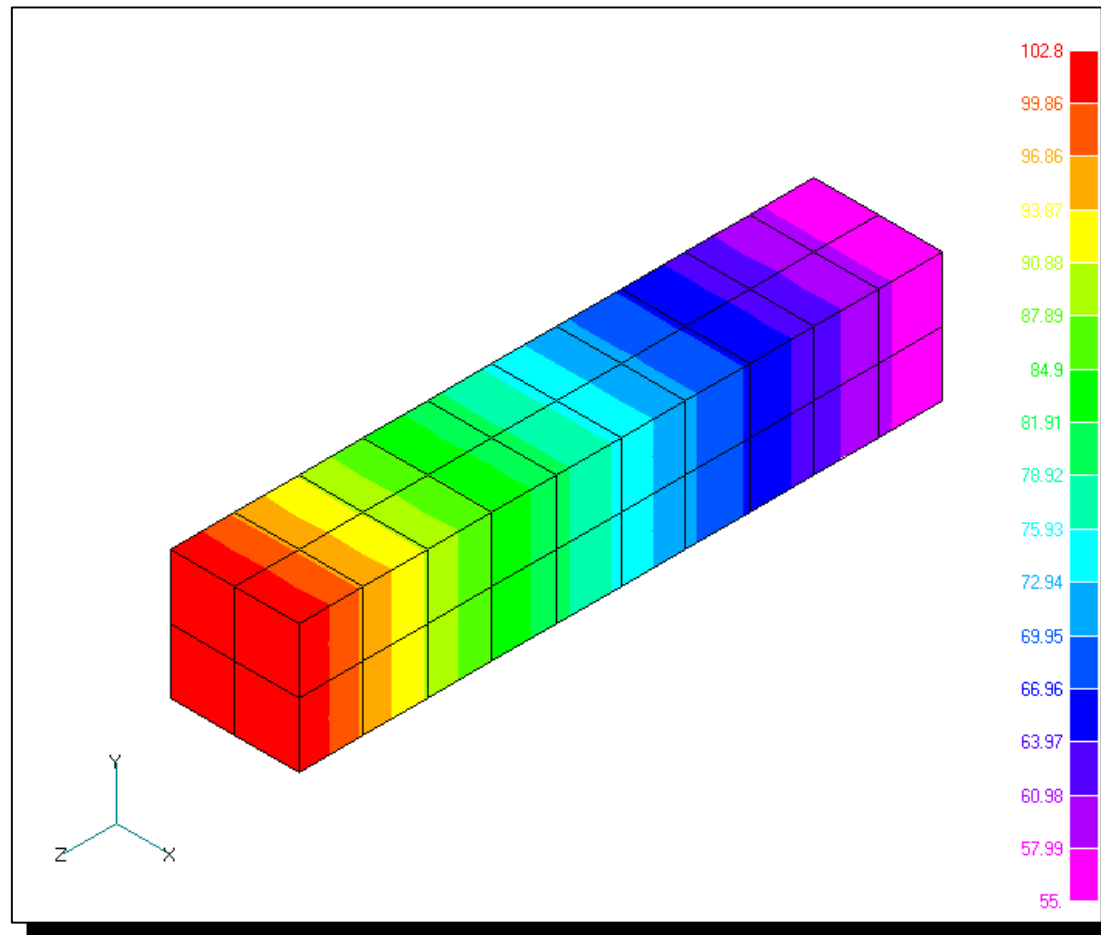
SENSOR TEMPERATURE



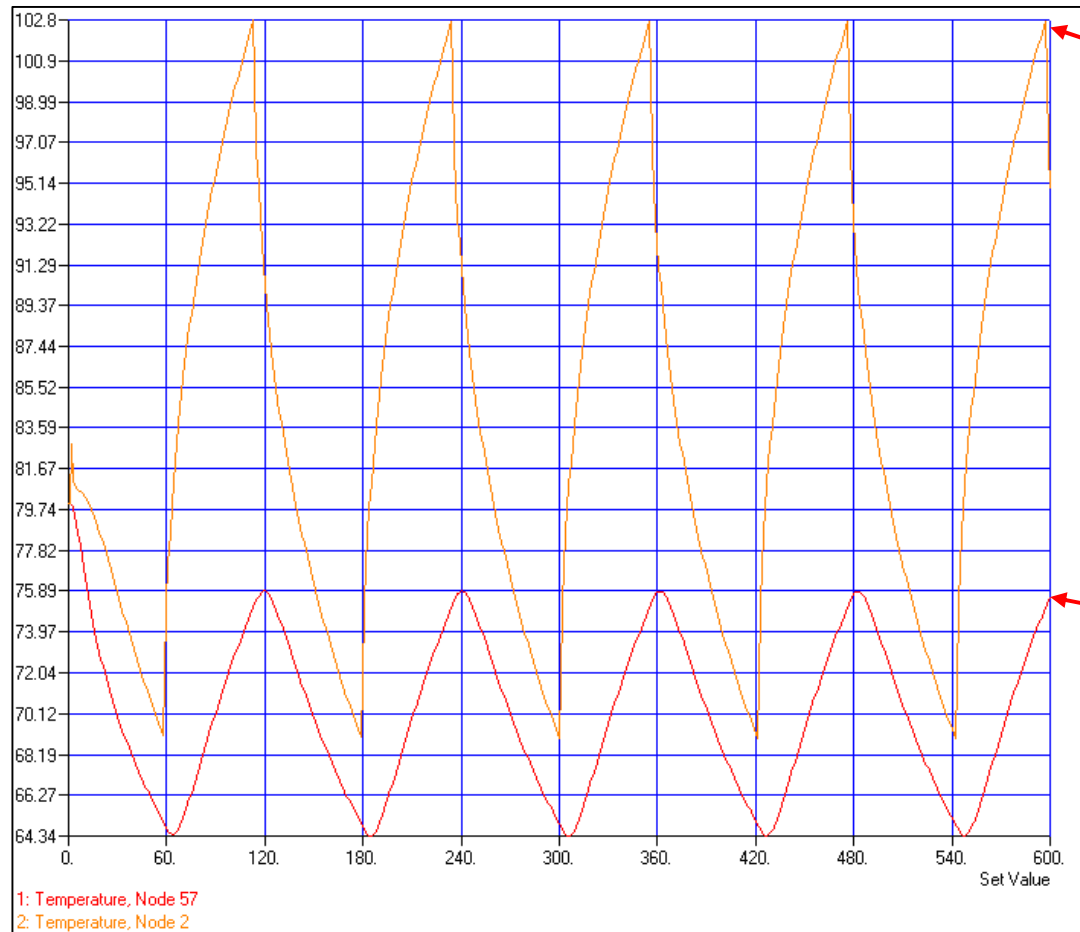
TEMPERATURE OF THE HEATER



TEMPERATURE CONTOUR



OVERLAY CURVES



The heater turns on and off on the basis of the sensor temperature.

The sensor monitors the temperature. The temperature varies from about 65° to 75°.



SECTION 6

RADIATION





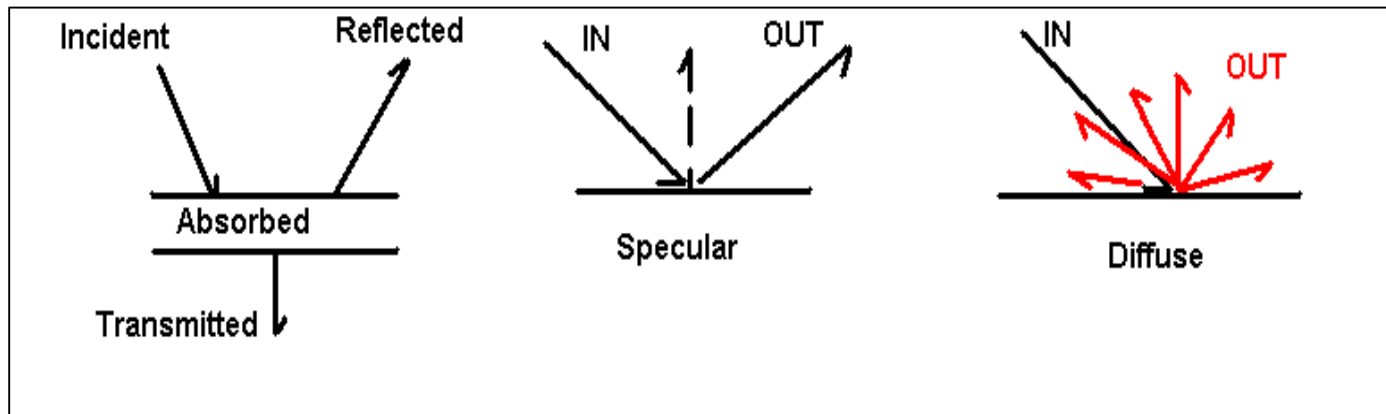
RADIATION TOPICS

- Theory
- Capabilities
- Ambient exchange
- Applied directional fluxes
- Radiation enclosures

RADIATION THEORY

- Thermal radiation: radiation at wavelengths (λ) in the range $0.1 < \lambda < 100 \mu\text{m}$
 - This range includes all visible light ($0.35 < \lambda < 0.75 \mu\text{m}$), and portions of the ultraviolet (short wavelength) and infrared (long wavelength) regions.
- Blackbody -- body that absorbs all radiation for all wavelengths

RADIATION THEORY



- Absorptivity - fraction of irradiation absorbed
- Reflectivity - fraction of irradiation reflected
- Emissivity - E/E_b (ratio of total hemishperical power actually emitted to that emitted by a blackbody)
- Absorptivity + Reflectivity + Transmissivity = 1.0

RADIATION THEORY

- In general, emissivity (ε) is a function of both, T and λ .
- Graybody -- a surface whose emissivity is not a function of λ
- For a graybody, Kirchoff's Identity states that $\varepsilon = \alpha = f(T)$

STEADY STATE SOLUTION METHOD

Steady State Analysis

Basic Equations. The steady state heat balance equation is

$$[K]\{u\} + [\mathfrak{R}]\{u + T_{abs}\}^4 = \{P\} + \{N\} \quad \text{Eq. 4-7}$$

where:

$[K]$ = a heat conduction matrix

$[\mathfrak{R}]$ = a radiation exchange matrix

$\{P\}$ = a vector of applied heat loads that are independent of temperature

$\{N\}$ = a vector of nonlinear heat loads that are temperature dependent

$\{u\}$ = a vector of grid point temperatures

T_{abs} = the absolute temperature scale adjustment required for radiation heat transfer exchange or radiation boundary conditions when all other temperatures and units are specified in deg-F or deg-C.



RADIATION FUNCTIONALITY





RADIATION CAPABILITIES

- Ambient exchange
 - Temperature dependent emissivity
 - Temperature dependent absorptivity
 - Time dependent view factor
- Applied fluxes
 - Temperature and time varying directional heat fluxes
- Radiation enclosures
 - Temperature dependent emissivity (absorptivity equal to emissivity)
 - Diffuse surface view factor calculations with self and third-body shadowing.

RADIATION CAPABILITIES

- Radiation enclosures
 - Adaptive view factor calculations with error estimation
 - Net view factors
 - Radiation matrix control
 - Radiation enclosure control
 - Multiple enclosures

AMBIENT RADIATION EXCHANGE





RADIATION, PRELIMINARY NOTE

- 2 PARAM entries are required for *all* radiation analyses
 - SIGMA provides the value of the Stefan-Boltzmann constant
 - PARAM,SIGMA,value
 - TABS specifies the offset required to convert degrees Fahrenheit to degrees Rankine (459.67), or degrees Celsius to degrees Kelvin (273.15)
 - PARAM,TABS,value



TABS AND SIGMA VIA PATRAN INTERFACE

- The value for TABS and SIGMA can be provided using the MD Patran interface for MD Nastran radiation problems.

The screenshot displays the MD Patran interface with several overlapping dialog boxes. The 'Analysis' dialog box is the background, showing 'Action: Analyze', 'Object: Entire Model', and 'Method: Full Run'. The 'Solution Type' dialog box is overlaid on it, showing 'MSC.Nastran Solution Type' and 'Solution Type: STEADY STATE ANALYSIS'. The 'Solution Parameters' dialog box is overlaid on the 'Solution Type' dialog, showing 'Steady State Solution Parameters' and 'Automatic Constraints' checked. The 'Radiation Parameters' dialog box is overlaid on the 'Solution Parameters' dialog, showing 'Absolute Temperature Scale: 0.0 Degree Kelvin/Rankine' and 'Stefan-Boltzmann Constant: 1.7140E-9 BTU/HR/FT2/R4'. The 'Radiation Parameters' dialog also includes 'TABS Override: 0.0' and 'SIGMA Override: 1.714e-09'.

Analysis

Action: Analyze
Object: Entire Model
Method: Full Run

Code: MSC.Nastran
Type: Thermal

Available Jobs

Job Name

Job Description (TITLE)

MSC.Nastran job created
18-Jul-08 at 11:45:34

SUBTITLE

LABEL

Translation Parameters...
Solution Type...

Solution Type

MSC.Nastran
Solution Type

Solution Type:
☒ STEADY STATE ANALYSIS
☐ TRANSIENT ANALYSIS
☐ STRUCTURAL HEAT ANALYSIS

Solution Parameters

Steady State Solution Parameters
☒ Automatic Constraints

Solution Sequence = SOL 153
Data Deck Echo: None
Maximum Printed Lines =
Maximum Run Time =
Default Init Temperature = 0.0

Radiation Parameters...
View Factor Parameters...
Results Output Format...

Radiation Parameters

Absolute Temperature Scale:
0.0 Degree Kelvin/Rankine
Stefan-Boltzmann Constant:
1.7140E-9 BTU/HR/FT2/R4

TABS Override:
0.0
SIGMA Override:
1.714e-09

OK Defaults Cancel

EXAMPLE 1E, RAD BOUNDARY CONDITION

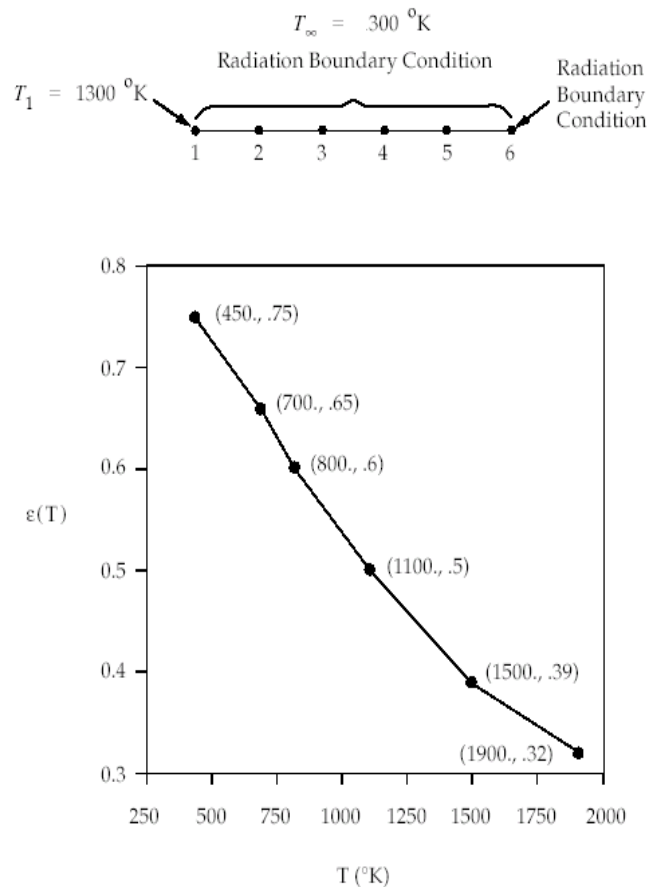
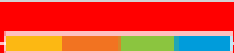


Figure 5-8 Example 1e – Emissivity as a Function of Temperature

- In this example, there is radiation loss along the entire length of the rod, and additionally, there is radiation loss at the end point (Node 6).
- The view factor is assumed to be 1.0.
- Emissivity is a function of temperature as shown in the image.



EXAMPLE 1E, RADBC

RADBC

Space Radiation Specification

Specifies an CHBDYi element face for application of radiation boundary conditions.

Format:

1	2	3	4	5	6	7	8	9	10
RADBC	NODAMB	FAMB	CNTRLND	EID1	EID2	EID3	-etc.-		

Example:

RADBC	5	1.0	101	10					

Field	Contents
NODAMB	Ambient point for radiation exchange. (Integer > 0)
FAMB	Radiation view factor between the face and the ambient point. (Real ≥ 0.0)
CNTRLND	Control point for radiation boundary condition. (Integer ≥ 0 ; Default = 0)
EIDi	CHBDYi element identification number. (Integer $\neq 0$ or "THRU" or "BY")

EXAMPLE 1E, RADIATION TO SPACE

```
ID MSC, EX1E      $
$ V68 D_CHU CC327 18-NOV-1993
$ID MSC-NASTRAN V68
SOL 153
TIME 10
CEND
TITLE = EXAMPLE 1e
EX1E
ANALYSIS = HEAT
THERMAL = ALL
FLUX = ALL
SPCF = ALL
OLOAD = ALL
SPC = 10
TEMP(INIT) = 20
NLPARM = 100
BEGIN BULK
NLPARM,100
PARAM,SIGMA,5.67E-8
PARAM,TABS,0.0
```

```
$
GRID,1,,0.0,0.0,0.0
GRID,2,,0.1,0.0,0.0
GRID,3,,0.2,0.0,0.0
GRID,4,,0.3,0.0,0.0
GRID,5,,0.4,0.0,0.0
GRID,6,,0.5,0.0,0.0
GRID,99,,99.0,99.0,99.0
$
CROD,1,5,1,2
CROD,2,5,2,3
CROD,3,5,3,4
CROD,4,5,4,5
CROD,5,5,5,6
PROD,5,15,.0078540
```



EXAMPLE 1E, RADIATION TO SPACE

```
MAT4,15,1.0
MATT4,15,40
TABLEM2,40,0.0,,,,,,,,+TBM1
+TBM1,173.16,215.0,273.16,202.0,373.16,206.0,473.16,215.0,+TBM2
+TBM2,573.16,228.0,673.16,249.0,ENDT
$
CHBDYP,10,25,LINE,,,1,2,,+CHP10
+CHP10,45,,,0.0,1.0,0.0
CHBDYP,20,25,LINE,,,2,3,,+CHP20
+CHP20,45,,,0.0,1.0,0.0
CHBDYP,30,25,LINE,,,3,4,,+CHP30
+CHP30,45,,,0.0,1.0,0.0
CHBDYP,40,25,LINE,,,4,5,,+CHP40
+CHP40,45,,,0.0,1.0,0.0
CHBDYP,50,25,LINE,,,5,6,,+CHP50
+CHP50,45,,,0.0,1.0,0.0
CHBDYP,60,26,POINT,,,6,,,+CHP60
+CHP60,45,,,1.0,0.0,0.0
PHBDY,25,.3141593
PHBDY,26,.0078540
```

```
RADBC,99,1.0,,10,THRU,60,BY,10
RADM,45,1.0,1.0
RADMT,45,41,41
TABLEM2,41,0.0,,,,,,,,+TBM3
+TBM3,450.0,0.75,700.0,0.65,800.0,0.60,1100.0,0.50,+TBM4
+TBM4,1500.0,0.39,1900.0,0.32,ENDT
$
SPC,10,1,,1300.0
SPC,10,99,,300.0
TEMPD,20,1300.0
$
ENDDATA
```

EXAMPLE 1E, OUTPUT

LOAD STEP = 1.00000E+00

TEMPERATURE VECTOR

POINT ID.	TYPE	ID	VALUE	ID+1	VALUE	ID+2	VALUE	ID+3	VALUE	ID+4	VALUE	ID+5	VALUE
1	S		1.300000E+03		1.144522E+03		1.031890E+03		9.541739E+02		9.066078E+02		8.866313E+02
99	S		3.000000E+02										

1 EXAMPLE 1E EX1E JANUARY 26, 2004 MD NASTRAN 1/23/04 PAGE 10

0

LOAD STEP = 1.00000E+00

LOAD VECTOR

POINT ID.	TYPE	ID	VALUE	ID+1	VALUE	ID+2	VALUE	ID+3	VALUE	ID+4	VALUE	ID+5	VALUE
99	S		0.0										

1 EXAMPLE 1E EX1E JANUARY 26, 2004 MD NASTRAN 1/23/04 PAGE 11

0

LOAD STEP = 1.00000E+00

FORCES OF SINGLE-POINT CONSTRAINT

POINT ID.	TYPE	ID	VALUE	ID+1	VALUE	ID+2	VALUE	ID+3	VALUE	ID+4	VALUE	ID+5	VALUE
1	S		5.631669E+03		0.0		0.0		0.0		0.0		0.0
99	S		-5.631539E+03										

MD NASTRAN has improved the
radiation convergence for coarse mesh

EXAMPLE 1E, REFINE MESH

- Refine the mesh with 10 elements using MD Nastran duplicating commands.

```
$ NASTRAN ex1e_10.dat
GRID,1,,0.0,0.0,0.0
=,*1,=,*0.05,==
=9
GRID,99,,99.0,99.0,99.0
$
CROD,1,5,1,2
=,*1,=,*1,*1
=8
PROD,5,15,.0078540
MAT4,15,1.0
MATT4,15,40
TABLEM2,40,0.0,,,,,,,,+TBM1
+TBM1,173.16,215.0,273.16,202.0,373.16,206.0,473.
16,215.0,+TBM2
+TBM2,573.16,228.0,673.16,249.0,ENDT
$
CHBDYP,10,25,LINE,,,1,2,,+CHP10
=,*1,=,=,=,*1,*1,=,*1
=8
+CHP10,45,,,,0.0,1.0,0.0
*1,==
=8
CHBDYP,60,26,POINT,,,11,,,+CHP60
+CHP60,45,,,,1.0,0.0,0.0
```

```
PHBDY,25,.3141593
PHBDY,26,.0078540
$
RADBC,99,1.0,,10,THRU,19,BY,1
RADBC,99,1.0,,60
RADM,45,1.0,1.0
RADMT,45,41,41
TABLEM2,41,0.0,,,,,,,,+TBM3
+TBM3,450.0,0.75,700.0,0.65,800.0,0.60,1100.0,0.5
0,+TBM4
+TBM4,1500.0,0.39,1900.0,0.32,ENDT
$
SPC,10,1,,1300.0
SPC,10,99,,300.0
TEMPD,20,1300.0
$
ENDDATA
```

EXAMPLE 1E, ECHO MD NASTRAN INPUT

0	SORTED BULK DATA ECHO																				
ENTRY																					
COUNT	.	1	..	2	..	3	..	4	..	5	..	6	..	7	..	8	..	9	..	10	.
1-	CHBDYP	10		25		LINE								1		2					+CHP10
2-	+CHP10	45								0.0		1.0		0.0							
3-	CHBDYP	11		25		LINE								2		3					+CHP11
4-	+CHP11	45								0.0		1.0		0.0							
5-	CHBDYP	12		25		LINE								3		4					+CHP12
6-	+CHP12	45								0.0		1.0		0.0							
7-	CHBDYP	13		25		LINE								4		5					+CHP13
8-	+CHP13	45								0.0		1.0		0.0							
9-	CHBDYP	14		25		LINE								5		6					+CHP14
10-	+CHP14	45								0.0		1.0		0.0							
11-	CHBDYP	15		25		LINE								6		7					+CHP15
12-	+CHP15	45								0.0		1.0		0.0							
13-	CHBDYP	16		25		LINE								7		8					+CHP16
14-	+CHP16	45								0.0		1.0		0.0							
15-	CHBDYP	17		25		LINE								8		9					+CHP17
16-	+CHP17	45								0.0		1.0		0.0							
17-	CHBDYP	18		25		LINE								9		10					+CHP18
18-	+CHP18	45								0.0		1.0		0.0							
19-	CHBDYP	19		25		LINE								10		11					+CHP19
20-	+CHP19	45								0.0		1.0		0.0							
21-	CHBDYP	60		26		POINT										11					+CHP60
22-	+CHP60	45										1.0		0.0		0.0					

EXAMPLE 1E, ECHO MD NASTRAN INPUT

0	SORTED BULK DATA EC					
HO						
23-	CROD	1	5	1	2	
24-	CROD	2	5	2	3	
25-	CROD	3	5	3	4	
26-	CROD	4	5	4	5	
27-	CROD	5	5	5	6	
28-	CROD	6	5	6	7	
29-	CROD	7	5	7	8	
30-	CROD	8	5	8	9	
31-	CROD	9	5	9	10	
32-	CROD	10	5	10	11	
33-	GRID	1		0.0	0.0	0.0
34-	GRID	2		.05	0.0	0.0
35-	GRID	3		.1	0.0	0.0
36-	GRID	4		.15	0.0	0.0
37-	GRID	5		.2	0.0	0.0
38-	GRID	6		.25	0.0	0.0
39-	GRID	7		.3	0.0	0.0
40-	GRID	8		.35	0.0	0.0
41-	GRID	9		.4	0.0	0.0
42-	GRID	10		.45	0.0	0.0
43-	GRID	11		.5	0.0	0.0
44-	GRID	99		99.0	99.0	99.0

EXAMPLE 1E, REFINE MESH TEMP RESULT

The end point has a temp equal to 884.14

```
0
LOAD STEP = 1.00000E+00
      TEMPERATURE VECTOR

POINT ID.  TYPE      ID  VALUE      ID+1 VALUE      ID+2 VALUE      ID+3 VALUE      ID+4 VALUE      ID+5 VALUE
      1      S      1.300000E+03      1.215472E+03      1.143219E+03      1.081756E+03      1.029942E+03      9.868965E+02
      7      S      9.519230E+02      9.244869E+02      9.042006E+02      8.908038E+02      8.841412E+02
     99      S      3.000000E+02
1  EXAMPLE 1E              EX1E      JANUARY 27, 2004 MD NASTRAN 8/22/03 PAGE 10
```

```
0
LOAD STEP = 1.00000E+00
      LOAD VECTOR

POINT ID.  TYPE      ID  VALUE      ID+1 VALUE      ID+2 VALUE      ID+3 VALUE      ID+4 VALUE      ID+5 VALUE
      7      S      0.0          0.0          0.0          0.0          0.0
     99      S      0.0
1  EXAMPLE 1E              EX1E      JANUARY 27, 2004 MD NASTRAN 8/22/03 PAGE 11
```

```
0
LOAD STEP = 1.00000E+00
      FORCES OF SINGLE-POINT CONSTRAINT

POINT ID.  TYPE      ID  VALUE      ID+1 VALUE      ID+2 VALUE      ID+3 VALUE      ID+4 VALUE      ID+5 VALUE
      1      S      5.515212E+03      0.0          0.0          0.0          0.0          0.0
      7      S      0.0          0.0          0.0          0.0          0.0
     99      S      -5.515519E+03
```

STEADY STATE SOLUTION CONTROL

NLPARM

Parameters for Nonlinear Static Analysis Control

Defines a set of parameters for nonlinear static analysis iteration strategy.

Format:

1	2	3	4	5	6	7	8	9	10
NLPARM	ID	NINC	DT	KMETHOD	KSTEP	MAXITER	CONV	INTOUT	
	EPSU	EPSP	EPSW	MAXDIV	MAXQN	MAXLS	FSTRESS	LSTOL	
	MAXBIS				MAXR		RTOLB	MINITER	

Example:

NLPARM	15	5		ITER					

Field	Contents
ID	Identification number. (Integer > 0)
NINC	Number of increments. See Remark 16. (Integer > 0; Default = 10)
DT	Incremental time interval for creep analysis. See Remark 3. (Real ≥ 0.0 ; Default = 0.0 for no creep.)
KMETHOD	Method for controlling stiffness updates. See Remark 4. (Character = "AUTO", "ITER", "SEMI", "FNT", or "PFNT"; Default = "AUTO" for SOL 106, "AUTO" for SOL 400 with non-contact analysis, and "FNT" for SOL 400 with contact analysis.)

DIRECTIONAL HEAT FLUX



EXAMPLE 4B, DIRECTIONAL SOLAR LOAD WITH RADIATION BOUNDARY CONDITION

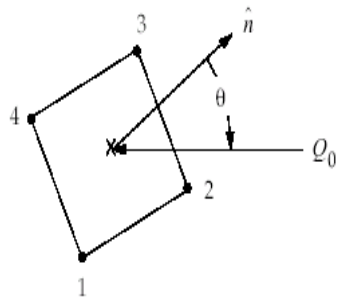


Figure 5-4 Example 4b

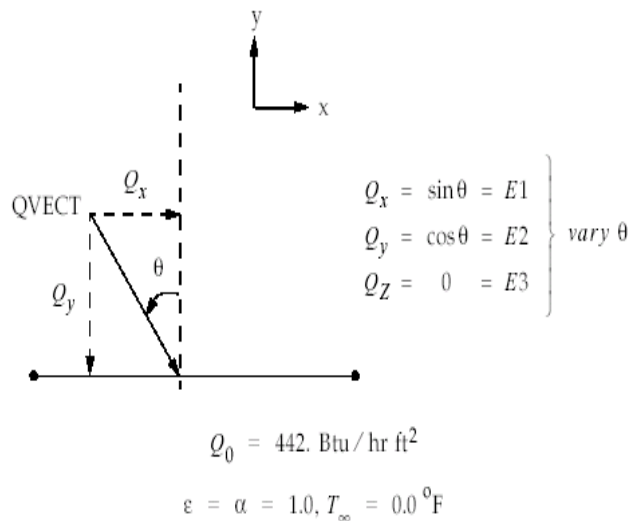


Table 5-1 Equilibrium Temperature versus Angle of Incident Radiation

θ (deg)	$T_{\text{plate}} (^{\circ}\text{F})$	Direction Cosines	
		E1	E2
0	282.0	0.0	-1.0
10	279.6	0.173648	-0.984808
20	272.2	0.342020	-0.939693
30	259.8	0.5	-0.866025
40	241.8	0.642788	-0.766044
50	217.6	0.766044	-0.642788
60	185.8	0.866025	-0.5
70	144.1	0.939693	-0.342020
80	87.2	0.984808	-0.173648
90	0.0	1.0	0.0

Note: $\theta = 80^{\circ}$ case is illustrated in the input file listing.

EXAMPLE 4B, QVECT

- Defines thermal vector flux from a distance source into the face of one or more CHBDYi elements

1	2	3	4	5	6	7	8	9	10
QVECT	SID	Q0	TSOUR	CE	E1 or TID1	E2 or TID2	E3 or TID3	CNTRLND	
	EID1	EID2	-etc-						

1	2	3	4	5	6	7	8	9	10
QVECT	200	442.0			1.0	0.0	0.0		
	10	11	12	13					

- Q0 – Magnitude of thermal flux
- TSOUR – Temperature of the radiant source; only needed if emissivity is a function of wavelength
- CE – Coordinate system of thermal flux
- Ei - Vector components (direction cosines)
- TIDi – TABLEDj ID, the direction cosines can be function of time
- CNTRLND – control point
- EIDk – CHBDYi ID

EXAMPLE 4B, QVECT

For use in steady-state analysis, the load set is selected in the Case Control Section (LOAD = SID). The total power into an element is given by:

- If CNTRLND = 0 then, $P_{in} = -\alpha A(\vec{e} \cdot \vec{n}) \cdot Q0$.
- If CNTRLND > 0 then, $P_{in} = -\alpha A(\vec{e} \cdot \vec{n}) \cdot Q0 \cdot u_{CNTRLND}$.

where

α = face absorptivity (supplied from a RADM statement).

A = face area as determined from a CHBDYi surface element.

$\vec{e}$ = vector of direction cosines E1, E2, E3.

$\vec{n}$ = face normal vector. See CHBDYi entries.

$\vec{e} \cdot \vec{n}$ = 0 if the vector product is positive, (i.e., the flux is coming from behind the face).

$u_{cntrlnd}$ = temperature value of the control point used as a load multiplier.

EXAMPLE 4B, DIRECTIONAL HEAT FLUX

- QBDY1 describes a uniform applied heat flux on surface elements
 - Not directional
- QVECT defines radiative vector flux from a distant source
 - Steady state, $P_{\text{out}} = \text{Area} * \alpha * (\underline{e} \cdot \underline{n}) * Q_{\text{mag}}$
 - Transient, $P_{\text{out}} = \text{Area} * \alpha * (\underline{e} \cdot \underline{n}) * Q_{\text{mag}} * f(\tau)$
where, $\underline{e}$ and $\underline{n}$ are the flux and element normal directions, respectively

EXAMPLE 4B, MD NASTRAN INPUT

```
ID MSC, EX4B      $
$ V68 D CHU CC327 18-NOV-1993
$ID MSC_NASTRAN V68
SOL 153
TIME 10
CEND
TITLE = EXAMPLE 4b
EX4B
ANALYSIS = HEAT
THERMAL = ALL
FLUX = ALL
SPCF = ALL
OLOAD = ALL
SPC = 10
TEMP(INIT) = 20
NLPARM = 100
LOAD = 200
BEGIN BULK
PARAM,TABS,459.67
PARAM,SIGMA,.1714E-8
NLPARM,100
$
```

```
GRID,1,,0.0,0.0,0.0
GRID,2,,0.0,0.0,1.0
GRID,3,,1.0,0.0,1.0
GRID,4,,1.0,0.0,0.0
GRID,99,,99.0,99.0,99.0
$
CQUAD4,1,5,1,2,3,4
PSHELL,5,15,0.1
MAT4,15,204.0
RADM,45,1.0,1.0
$
CHBDYG,10,,AREA4,,,45,,,+CHG10
+CHG10,1,2,3,4
QVECT,200,442.0,,, .984808,-.173648,0.0,,+QVCT1
+QVCT1,10
RADBC,99,1.0,,10
$
SPC,10,99,,0.0
TEMPD,20,0.0
$
ENDDATA
```

EXAMPLE 4B, OUTPUT

LOAD STEP = 1.00000E+00

TEMPERATURE VECTOR

POINT ID.	TYPE	ID VALUE	ID+1 VALUE	ID+2 VALUE	ID+3 VALUE	ID+4 VALUE	ID+5 VALUE
1	S	8.717700E+01	8.717700E+01	8.717700E+01	8.717700E+01		
99	S	0.0					

1 EXAMPLE 4B EX4B JANUARY 26, 2004 MD NASTRAN 8/22/03 PAGE 9

LOAD STEP = 1.00000E+00

LOAD VECTOR

POINT ID.	TYPE	ID VALUE	ID+1 VALUE	ID+2 VALUE	ID+3 VALUE	ID+4 VALUE	ID+5 VALUE
1	S	1.918810E+01	1.918810E+01	1.918810E+01	1.918810E+01		
99	S	0.0					

1 EXAMPLE 4B EX4B JANUARY 26, 2004 MD NASTRAN 8/22/03 PAGE 10

LOAD STEP = 1.00000E+00

FORCES OF SINGLE-POINT CONSTRAINT

POINT ID.	TYPE	ID VALUE	ID+1 VALUE	ID+2 VALUE	ID+3 VALUE	ID+4 VALUE	ID+5 VALUE
1	S	0.0	0.0	0.0	0.0		
99	S	-7.675240E+01					

1 EXAMPLE 4B EX4B JANUARY 26, 2004 MD NASTRAN 8/22/03 PAGE 11

1 EXAMPLE 4B EX4B JANUARY 26, 2004 MD NASTRAN 8/22/03 PAGE 12

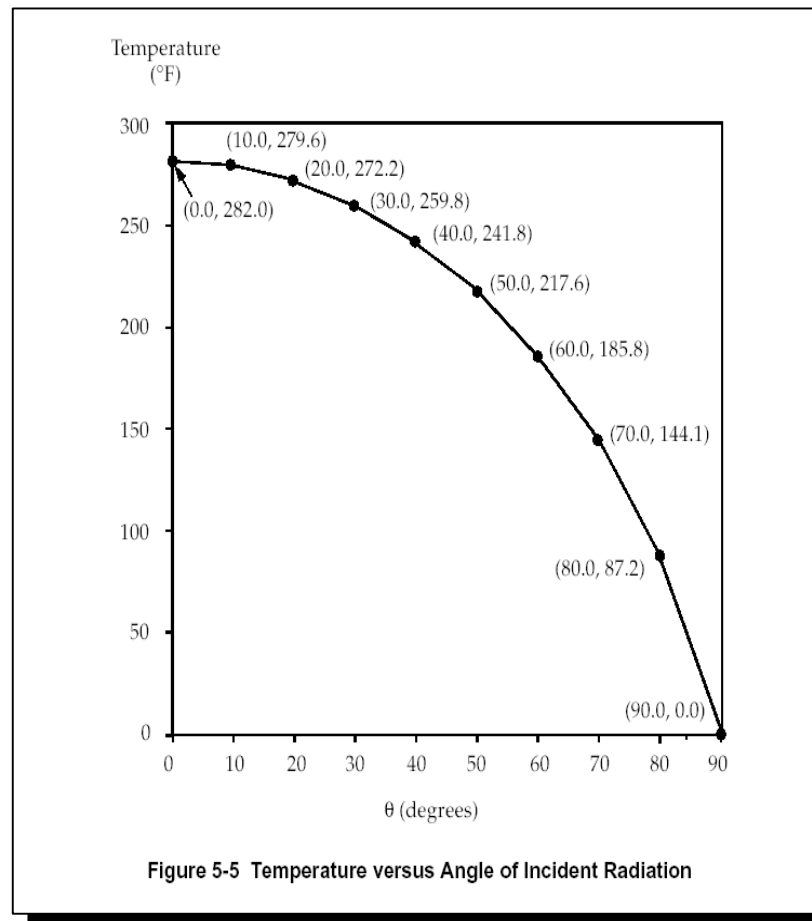
LOAD STEP = 1.00000E+00

HEAT FLOW INTO HBDY ELEMENTS (CHBDY)

ELEMENT-ID	APPLIED-LOAD	FREE-CONVECTION	FORCED-CONVECTION	RADIATION	TOTAL
10	7.675242E+01	0.000000E+00	0.000000E+00	-7.675240E+01	2.288818E-05

EXAMPLE 4B, TEMP VERSUS ANGLE

- Element temperature versus angle of incident radiation



DIRECTIONAL FLUX, MD PATRAN INTERFACE

- Loads/BCs: Create/Applied Heat/Elemental Uniform, Option: Directional Fluxes

The screenshot shows the 'Input Data' dialog box in MD Patran. The 'Form Type' is set to 'Basic'. The 'Thermal Vec Type' is set to 'Constant'. The 'Absorptivity' is set to 1. The 'Heat Flux' is set to 442. The 'Incident Thermal Vector' is set to <0,-1,0>. The 'Temperature Function' is set to f:absorp_temperature. The 'Time Function' is set to f:heat_flux_time. The 'DirCos e1(t) of Thermal Vec' is empty. The 'DirCos e2(t) of Thermal Vec' is empty. The 'DirCos e3(t) of Thermal Vec' is empty. The 'Time Dependent Fields' list contains heat_flux_time. The 'Spatial Fields' list is empty. The dialog has OK, Reset, and Cancel buttons.

- Absorptivity may be a function of Temperature.
- The Flux magnitude may be a function of time.
- The Temperature of the radiant source is used to determine wavelength distribution of source radiation (assumed as a blackbody).

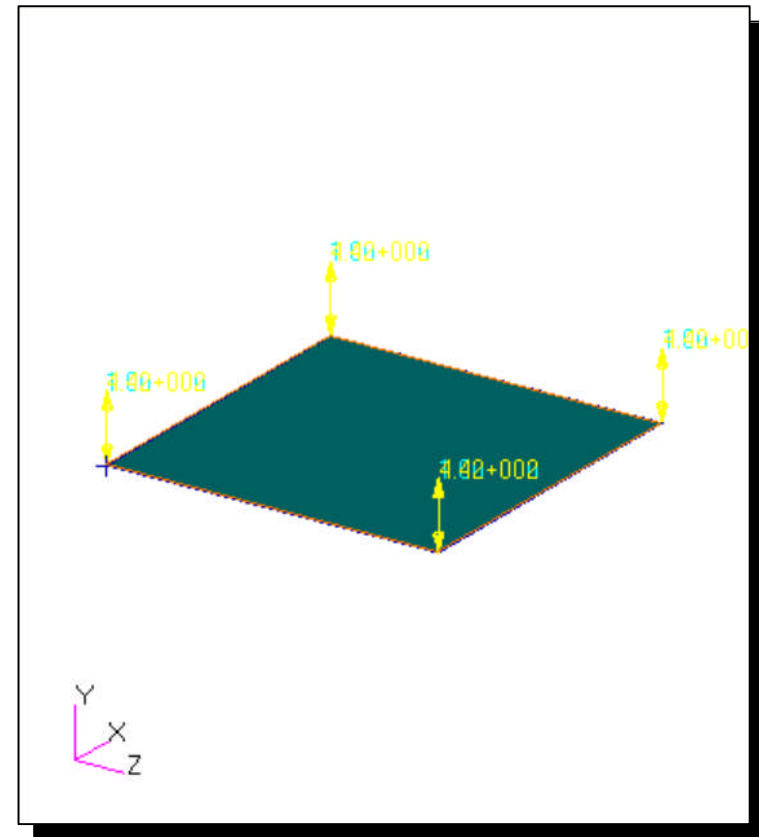
Note: MD Patran MD Nastran Thermal preference currently does not support wavebands.

PROJECT 9, DIRECTIONAL HEAT LOADS

- 1x1x0.1 plate
 - $K_x = 204.0 \text{ Btu/hr/(ft}^2\text{R)}$ for Aluminum
 - FE model using 1 plate element (CQUAD4)
 - Applied heat flux of $442.0 \text{ Btu/hr/ft}^2$ for 3 surface orientations (0.0 and 60.0 degrees)
 - Radiation to surroundings at $0.0^\circ\text{Fahrenheit}$
 - Run SS analyses to determine equilibrium temperatures
 - Post process
 - Examine .f06 ACSii results file
-
- Problem based on MD Nastran 2002 Thermal Analysis User's Guide, Example 4b

PROJECT 9, DIRECTIONAL HEAT LOADS

- Objective is to model directional solar heat flux loads, and with the plate at radiative equilibrium.
- Note, the image displays the directional heat load in the - Y direction.



PROJECT 9, DIRECTIONAL HEAT LOADS

The screenshot displays two panels from the MSC Software interface. The **Geometry** panel on the left is configured with Action: Create, Object: Surface, and Method: XYZ. The Surface ID List contains the value 1. The Refer. Coordinate Frame is set to Coord 0, and the Vector Coordinates List contains the value <1 0 1>. The Auto Execute checkbox is checked, and the Origin Coordinates List contains the value [0 0 0]. The Finite Elements panel on the right is configured with Action: Create, Object: Mesh, and Type: Surface. The Output ID List shows Node 1 and Element 1. The Elem Shape is set to Quad, the Mesher is IsoMesh, and the Topology is Quad4. The Surface List contains the value Surface 1. The Global Edge Length section has the Automatic Calculation checkbox unchecked and the Value set to 1.0.

Geometry

Action: Create
Object: Surface
Method: XYZ

Surface ID List
1

Refer. Coordinate Frame
Coord 0

Vector Coordinates List
<1 0 1>

☒ Auto Execute

Origin Coordinates List
[0 0 0]

-Apply-

Finite Elements

Action: Create
Object: Mesh
Type: Surface

Output ID List
Node 1
Element 1

Elem Shape: Quad
Mesher: IsoMesh
Topology: Quad4

IsoMesh Parameters...
Node Coordinate Frames...

Surface List
Surface 1

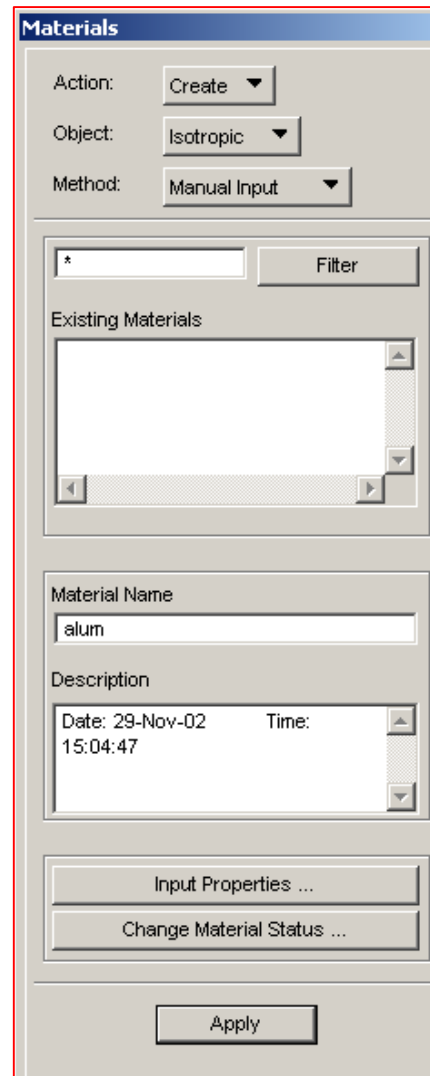
Global Edge Length
☐ Automatic Calculation
Value: 1.0

-Apply-

- Note, the surface lies in the X-Z Plane, and the mesh density (one shell element) corresponds to Global Edge Length equal to 1.0.

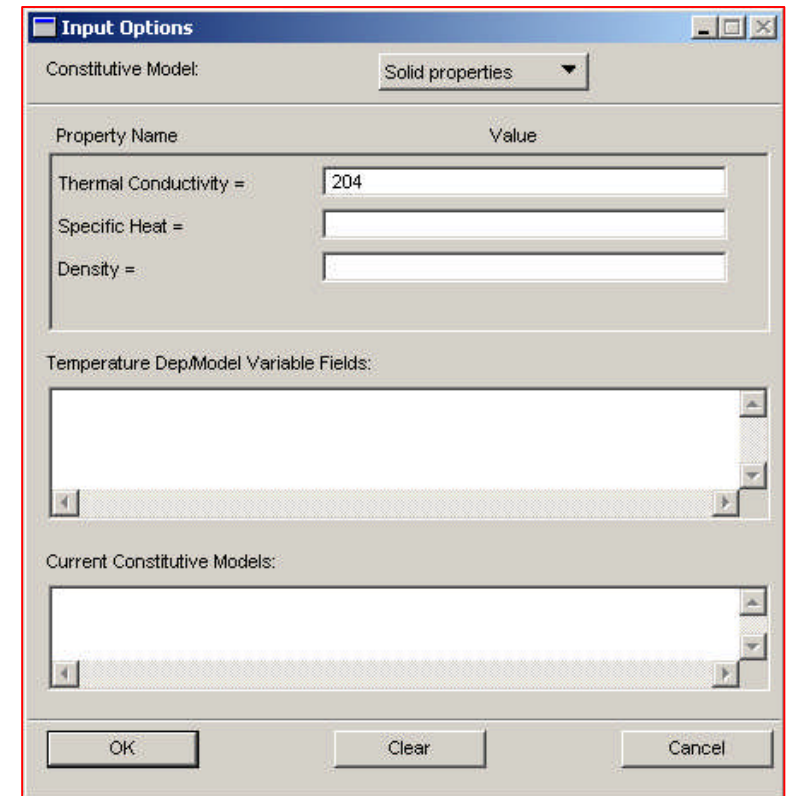
PROJECT 9, DIRECTIONAL HEAT LOADS

- Create thermal properties for solid material, aluminum.



The Materials dialog box is used to create or modify material properties. It includes a title bar 'Materials' and several sections:

- Action:** A dropdown menu set to 'Create'.
- Object:** A dropdown menu set to 'Isotropic'.
- Method:** A dropdown menu set to 'Manual Input'.
- Filter:** A text input field with an asterisk and a 'Filter' button.
- Existing Materials:** A list box showing existing materials.
- Material Name:** A text input field containing 'alum'.
- Description:** A text input field containing 'Date: 29-Nov-02 Time: 15:04:47'.
- Buttons:** 'Input Properties ...', 'Change Material Status ...', and 'Apply'.

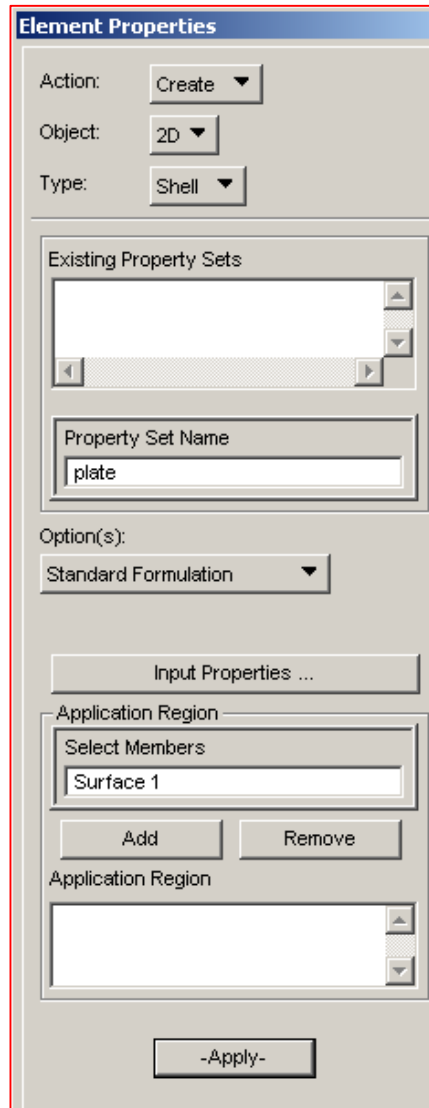


The Input Options dialog box is used to specify the constitutive model and its properties. It includes a title bar 'Input Options' and several sections:

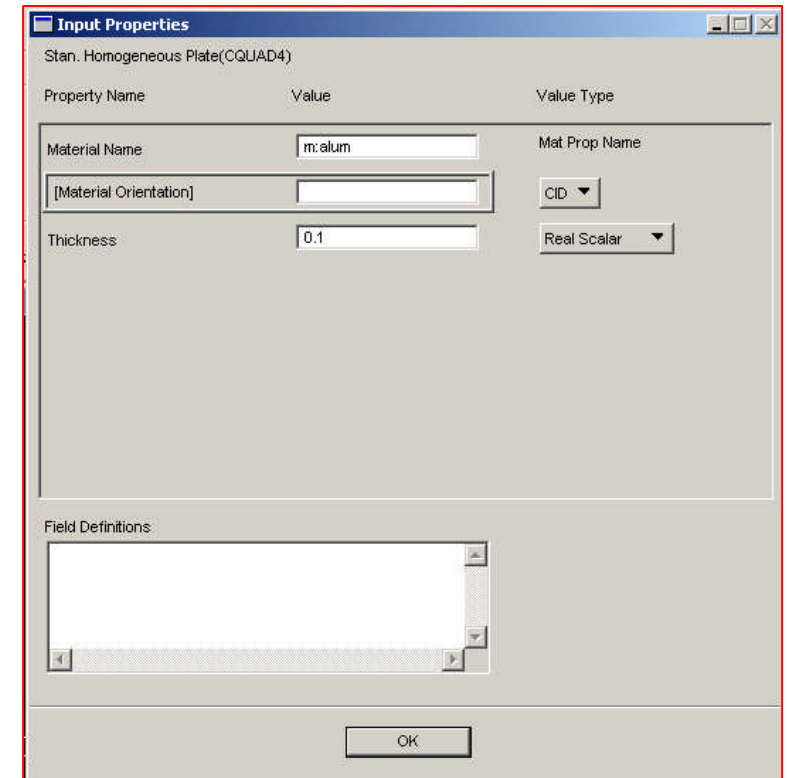
- Constitutive Model:** A dropdown menu set to 'Solid properties'.
- Property Name:** A text input field.
- Value:** A text input field.
- Thermal Conductivity =:** A text input field containing '204'.
- Specific Heat =:** A text input field.
- Density =:** A text input field.
- Temperature Dep/Model Variable Fields:** A list box.
- Current Constitutive Models:** A list box.
- Buttons:** 'OK', 'Clear', and 'Cancel'.

PROJECT 9, DIRECTIONAL HEAT LOADS

- Create element properties for the single shell element.
 - Material: alum
 - Thickness = 0.1



The 'Element Properties' dialog box is used to define the properties of a new element. It includes fields for Action (Create), Object (2D), and Type (Shell). There is a list of Existing Property Sets and a field for Property Set Name (plate). The Option(s) dropdown is set to Standard Formulation. There is a button for Input Properties ... and a section for Application Region with a Select Members list (Surface 1) and Add/Remove buttons. At the bottom, there is an -Apply- button.



The 'Input Properties' dialog box is used to define the input properties for a standard homogeneous plate (CQUAD4). It includes a table with Property Name, Value, and Value Type. The table has three rows: Material Name (m:alum), [Material Orientation] (empty), and Thickness (0.1). The Value Type for Thickness is Real Scalar. There is a button for Field Definitions and an OK button.

Property Name	Value	Value Type
Material Name	m:alum	Mat Prop Name
[Material Orientation]		CID
Thickness	0.1	Real Scalar

PROJECT 9, DIRECTIONAL HEAT LOADS

- Create radiation LBC for radiation to ambient temperature.
 - Emissivity = 1.0
 - Absorptivity = 1.0
 - Ambient temperature = 0.0°F
 - View factor = 1.0
 - AR: Surface 1; has associated single shell element

The screenshot displays three overlapping dialog boxes from the MSC Software interface:

- Load/Boundary Conditions**:
 - Action: Create
 - Object: Radiation
 - Type: Element Uniform
 - Option: Ambient Space
 - Current Load Case: Default...
 - Type: Static
 - Existing Sets: (empty list)
 - New Set Name: radbc
 - Target Element Type: 2D
 - Buttons: Input Data..., Select Application Region..., -Apply-
- Input Data**:
 - Surface Option: Top
 - Top Surf Emissivity: 1
 - Top Surf Absorptivity: 1
 - Ambient Temperature: 0
 - View Factor: 1
 - Spatial Fields: (empty list)
 - Buttons: OK, Reset
- Select Application Region**:
 - Geometry Filter: ☒ Geometry, ☐ FEM
 - Application Region: Select Surfaces or Edges, Surface 1
 - Buttons: Add, Remove
 - Application Region: (empty list)
 - Button: OK

PROJECT 9, DIRECTIONAL HEAT LOADS

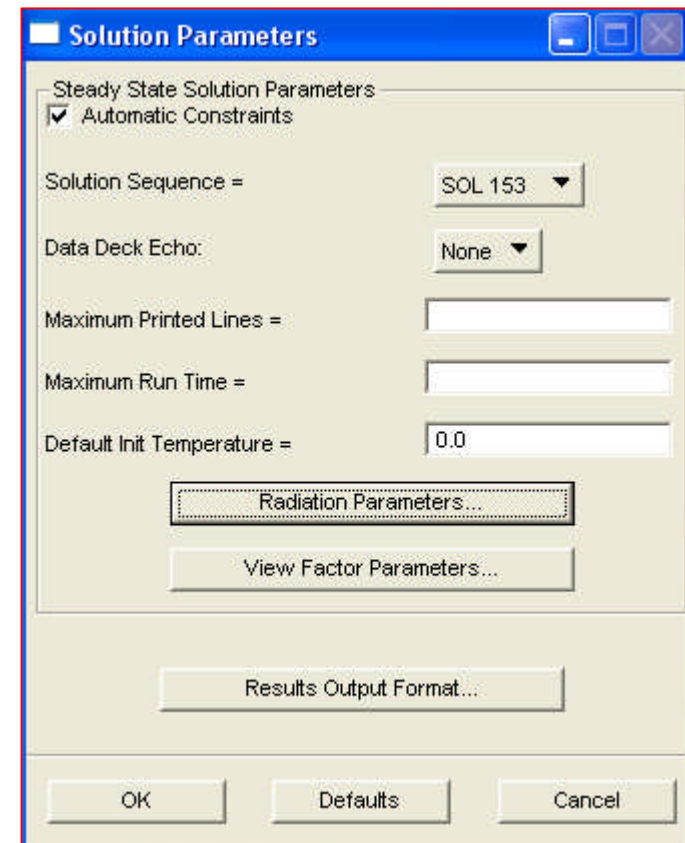
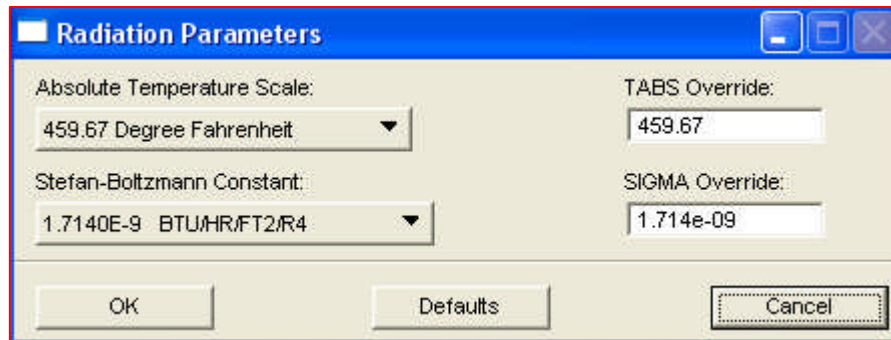
- Create LBC applied heat with specified direction.
 - Absorptivity = 1.0
 - Heat flux = 442.0 Btu/hr/ft**2
 - Vector = <0,-1,0>
 - AR: Surface 1

The image shows three overlapping software dialog boxes from MSC Software:

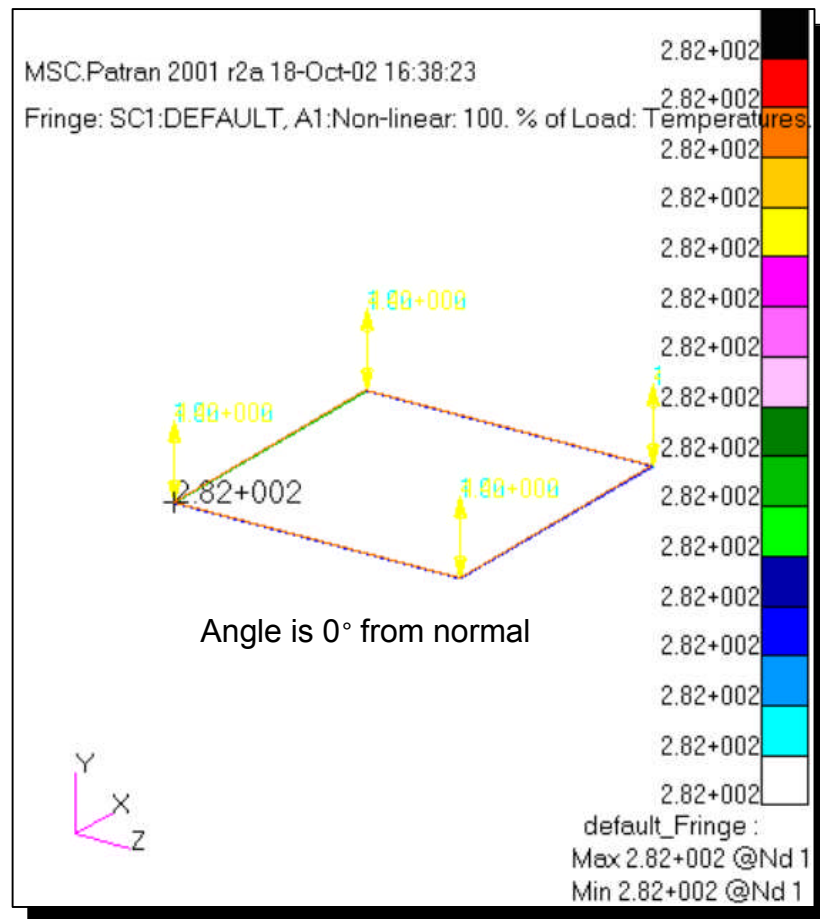
- Load/Boundary Conditions**:
 - Action: Create
 - Object: Applied Heat
 - Type: Element Uniform
 - Option: Directional Fluxes
 - Current Load Case: Default...
 - Type: Static
 - Existing Sets: (empty list)
 - New Set Name: qvect
 - Target Element Type: 2D
 - Buttons: Input Data..., Select Application Region..., -Apply-
- Input Data**:
 - Surface Option: Top
 - Top Surf Absorptivity: 1
 - Top Surf Heat Flux: 442
 - Incident Thermal Vector: <0, -1, 0>
 - Form Type: (empty)
 - * Temperature: (empty)
 - Buttons: OK, Reset
- Select Application Region**:
 - Geometry Filter: ☒ Geometry, ☐ FEM
 - Application Region: Select Surfaces or Edges, Surface 1
 - Buttons: Add, Remove
 - Application Region: (empty list)
 - Button: OK

PROJECT 9, DIRECTIONAL HEAT LOADS

- Run analysis
 - Analysis: Analyze/Entire Model/
Full Run
- Solution Type
 - Steady State Analysis
 - Solution Parameters
 - Radiation Parameters



PROJECT 9, DIRECTIONAL HEAT LOADS



- Flux direction is zero degrees from normal to plate; flux direction vector is $\langle 0, -1, 0 \rangle$.
- The temperature is constant at 282°F. This result agrees with that for Example 4b in the MSC. Nastran 2002 Thermal Analysis User's Guide.

PROJECT 9, DIRECTIONAL HEAT LOADS

Extract from MD Nastran ASCII Results File

```

0
SUBCASE 1
  LOAD STEP = 1.00000E+00

                                T E M P E R A T U R E   V E C T O R

POINT ID.    TYPE      ID  VALUE      ID+1 VALUE      ID+2 VALUE      ID+3 VALUE      ID+4 VALUE      ID+5 VALUE
   1         S        2.819614E+02  2.819614E+02  2.819614E+02  2.819614E+02  0.0
1  MD NASTRAN JOB CREATED ON 08-JUL-98 AT 10:04:05              JULY  8, 1998  MD NASTRAN
12/11/96  PAGE      10
  DEFAULT
0
SUBCASE 1
  LOAD STEP = 1.00000E+00

                                H E A T   F L O W   I N T O   H B D Y   E L E M E N T S   (CHBDY)

ELEMENT-ID    APPLIED-LOAD    FREE-CONVECTION    FORCED-CONVECTION    RADIATION    TOTAL
   1          4.420000E+02      0.000000E+00      0.000000E+00      -4.419938E+02  6.195068E-03
SUBCASE 1
    
```

- It is seen that the temperature is 281.96°F.
- The output from each CHBDYG element reflect the direction heat load of 442 Btu/hr and the emitted radiation -441.99 to space at 0°F.

PROJECT 9, DIRECTIONAL HEAT LOADS

Extracted from MD Nastran model .DAT file

```

PSHELL 1 1 .1
CQUAD4 1 1 1 2 4 3
MAT4 1 204.
$ Nodes of the Entire Model
GRID 1 0. 0. 0.
GRID 2 0. 0. 1.
GRID 3 1. 0. 0.
GRID 4 1. 0. 1.
$ Loads for Load Case : Default
$ Fixed Temperatures of Load Set : radbc
SPC 1 5 1 0.
$ Directional Heat Flux of Load Set : qvect
QVECT 2 442. 0. -1. 0. 0 + B
+ B 1
$ Radiation Boundary Conditions of Load Set : radbc
RADBC 5 1. 0 1
$ Initial Temperatures from Temperature Load Sets
TEMP 1 5 0.
$ Default Initial Temperature
TEMPD 1 100.
$ CHBDYG Surface Elements
CHBDYG 1 AREA4 1 + C
+ C 1 2 4 3
$ Radiation Material Properties
RADM 1 1. 1.
$ Scalar Points
SPOINT 5
$ Referenced Coordinate Frames
ENDDATA
    
```

PROJECT 9, DIRECTIONAL HEAT LOADS

- QVECT specification in Bulk Data

```
$QVECT, SID, Q0, TSOUR, CE, E1, E2, E3, CNTRLND,+
$,EID1,EID2, etc.
QVECT,2,442.,,0.,-1.,0.,+
+,1
```

Obtained from MD Nastran model .DAT file

1	2	3	4	5	6	7	8	9	10
QVECT	SID	Q0	TSOUR	CE	E1 or TID1	E2 or TID2	E3 or TID3	CNTRLND	
	EID1	EID2	-etc-						

- Change the angle from zero degree from normal (vector $\langle 0, -1, 0 \rangle$) to 60° from normal.
- The direction cosines for 60° are $(0.866025, -0.5, 0.0)$ for $(E1, E2, E3)$.
- Can then modify the QVECT bulk data entry.
- QVECT,2,442.,,0.866025,-0.5,0.0

EXERCISES

- Perform Workshop 12 “Radiation to Space” in the exercise workbook.
- Perform Workshop 13 “Directional Heat Loads” in the exercise workbook.



ENCLOSURE RADIATION (VIEW FACTOR CALCULATIONS)

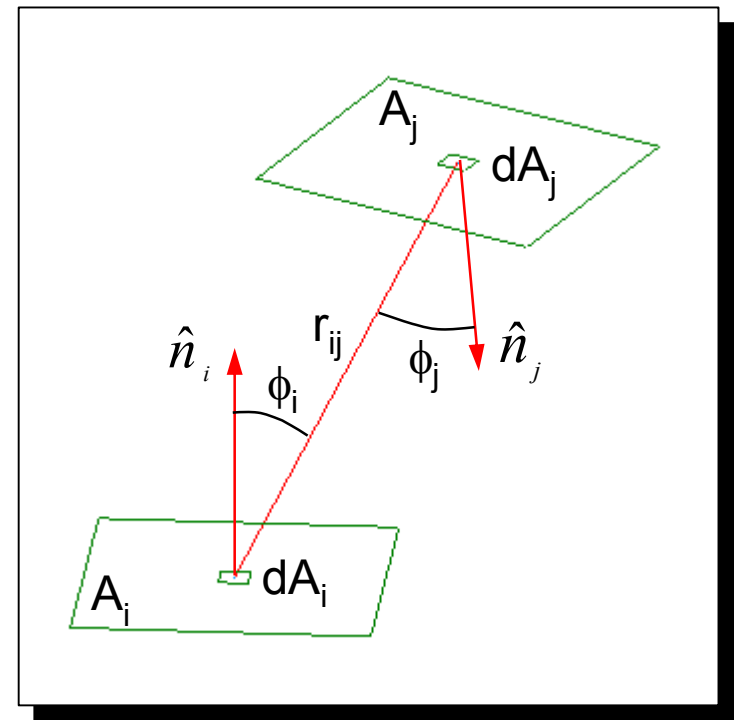




VIEW FACTOR DEFINITION

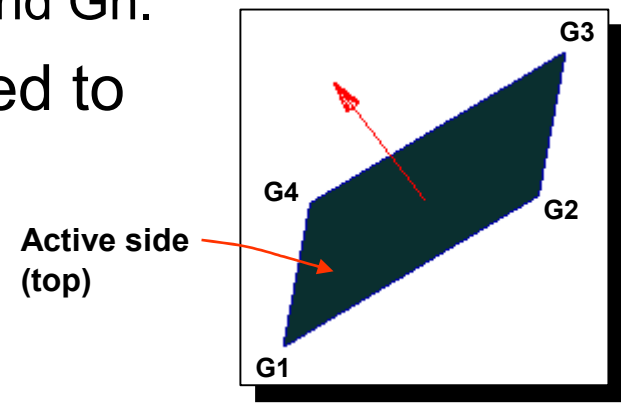
- Integral to be evaluated to determine the view factor, F_{ij} , which is the fraction of total radiant energy leaving A_i which is intercepted by A_j .

$$F_{ij} = \frac{1}{A_i} \iint_{A_i A_j} \frac{\cos \phi_i \cos \phi_j}{\pi r_{ij}^2} dA_i dA_j$$



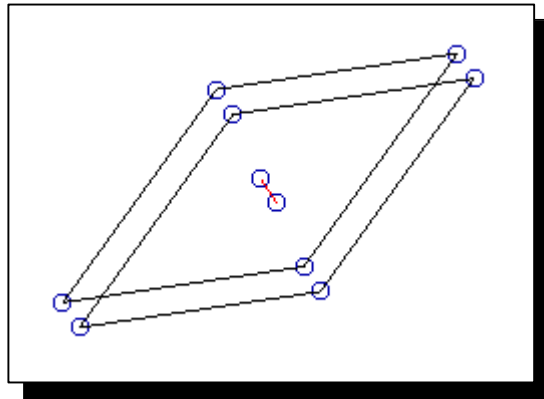
FUNDAMENTALS OF VF CALCULATIONS

- View factors can only be determined for pairs of CHBDYi surface elements
- Many times it is necessary to have a unique CHBDYi surface element on both sides of a conduction surface element, particularly when third-body shadowing occurs
- Active side of surface element
 - The active side of a CHBDYi surface element is defined by the direction of the element normal. The normal is defined by the right-hand rule when going from Grid G1 to Grid Gn.
- Only active surface elements are used to calculate view factors

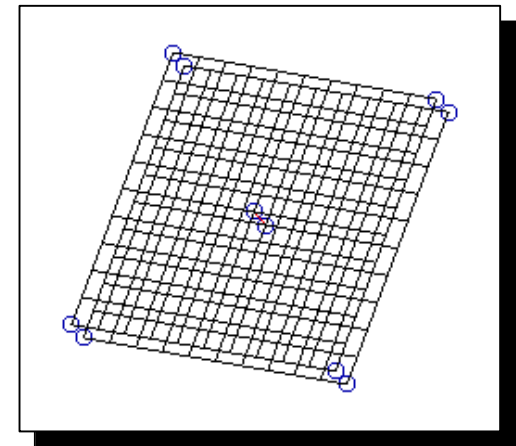


FUNDAMENTALS OF VF CALCULATIONS

- The quality of the calculated view factors is highly dependent on the quality of the surface element mesh
- When the distance between any two elements is $\ll$ element size, there can be inaccuracies in the calculated view factors



- However, too many elements (that are small) will result in a computationally intensive problem

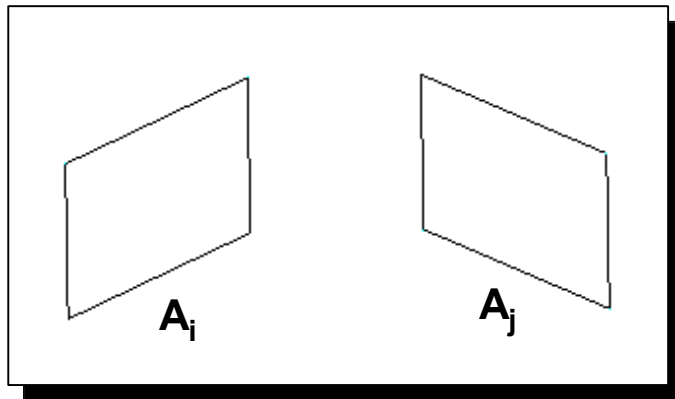


FUNDAMENTALS OF VF CALCULATIONS

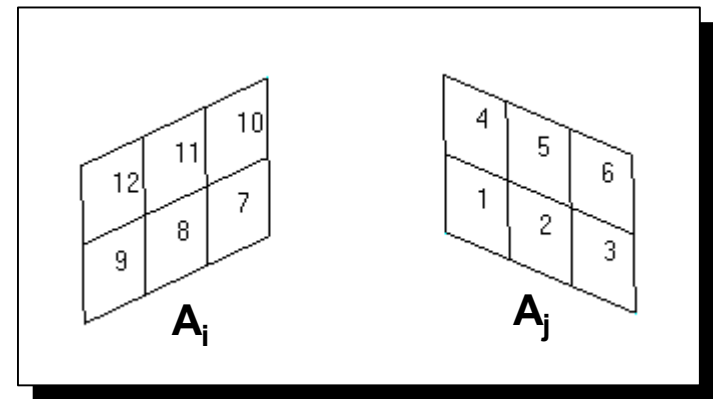
- There can be multiple radiation cavities (enclosures). The surface element groups are arranged by unique cavity ID. No surface element may be in more than one cavity.

VIEW FACTOR COMPUTATION

- VIEW module, and RADCAV and VIEW Bulk Data entry
- Two methods
 - First method, Finite Difference – discretizes CHBDYi surface elements into a number of finite sized sub-elements, and treats the integrals as double summations over all the finite sized sub-elements on surfaces A_i and A_j . This method is referred to as the finite difference method, and is an extension of “view factor algebra”.



CHBDYi Elements



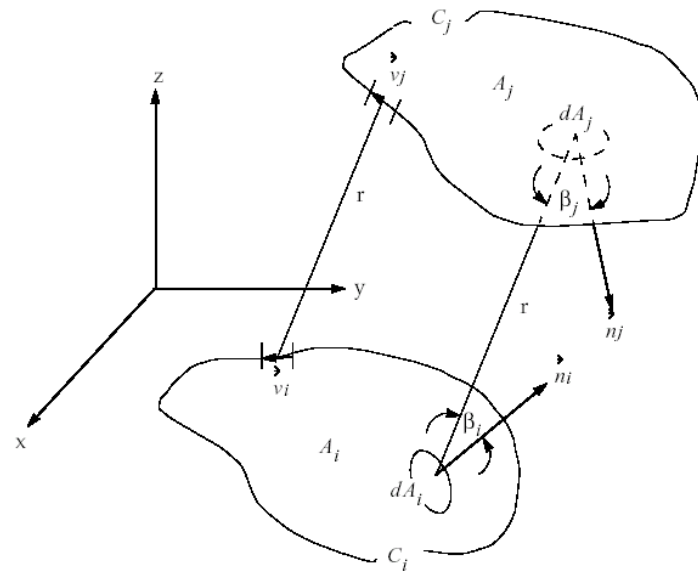
Sub-elements

VIEW FACTOR COMPUTATION (Cont.)

- VIEW module, and RADCAV and VIEW Bulk Data entry
- Two methods
 - Second method, Contour Integration – transforms the area integrals into contour integrals, and divides the perimeter into finite line segments. A similar double contour summation is then performed around surfaces A_i and A_j .

$$F_{ij} = \frac{1}{A_i} \oint_{A_i} \oint_{A_j} \left(\frac{\cos \beta_i \cos \beta_j}{\pi r_{ij}^2} \right) dA_i dA_j \quad (\text{dimensionless}) \quad \text{Eq. E-1}$$

where F_{ij} is defined as the fraction of the radiant emission leaving surface i which arrives at surface j .



A_i and A_j are diffuse emitters and reflectors
 A_i and A_j are black.
 A_i and A_j are isothermal.

VIEW FACTOR COMPUTATION (Cont.)

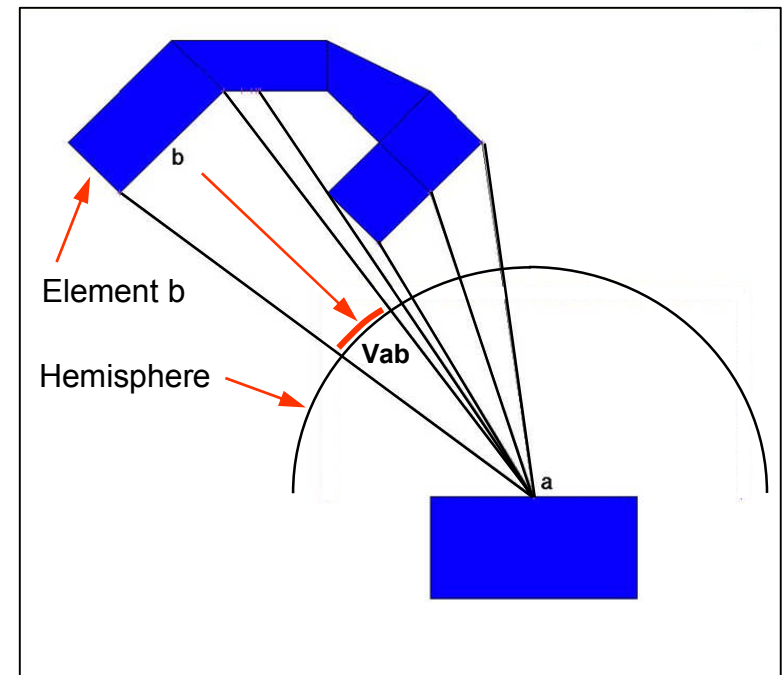
- VIEW module, and RADCAV and VIEW Bulk Data entry
- Two methods
 - In general, area integration (Finite Difference) is faster than contour integration (Contour Integration), but does not provide as accurate an answer.

VIEW FACTOR COMPUTATION (Cont.)

- VIEW3D module and VIEW3D Bulk Data entry
- One method
 - Gaussian Integration
 - Used for general 3D and axi-symmetric geometry

VIEW FACTOR COMPUTATION (Cont.)

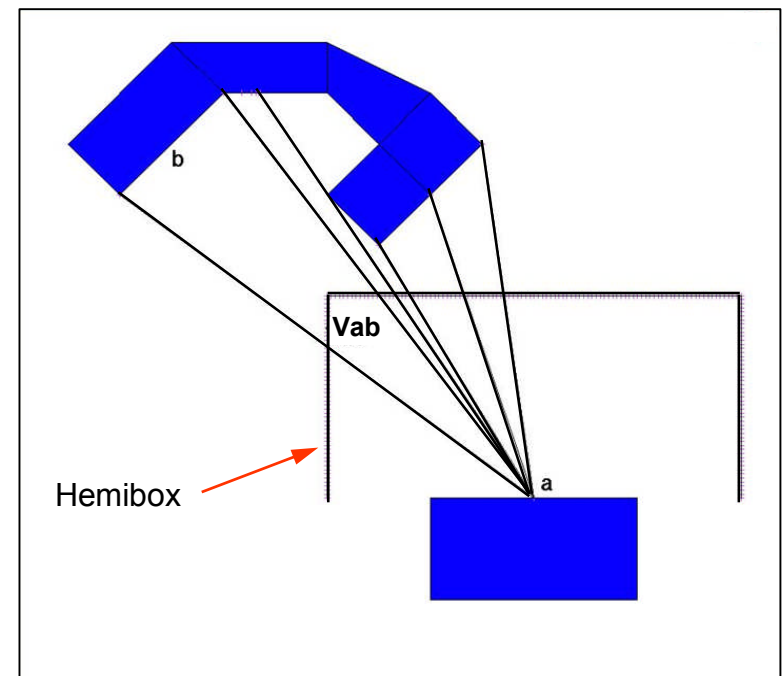
- “Pixel based modified Hemi-cube” method
 - For enclosures radiation is being emitted and received in multiple directions
 - This lead to the concept of a hemisphere, where an element is projected onto the it
 - Good for 2D and axi-symmetric, but problematic for 3D
 - Overcome problem for 3D by dividing hemisphere into pixels, and “count” them. Better, but still some problem for 3D.



Project Elem b onto hemisphere
Repeat, but first divide hemi-
sphere into pixels

VIEW FACTOR COMPUTATION (Cont.)

- “Pixel based modified Hemi-cube” method
 - Replace the hemisphere with a hemicube
 - Better, but still problems at edges and 3D corners
 - Change dimensions of the cube to reduce leakage out of the sides of it
 - Now, transition to a hemibox, then to a hemiplane
 - The final method obtained is called “pixel based modified Hemi-cube”. Good for 2D, 3D, and axi-symmetric.



Replace hemisphere with hemicube, then transition to hemibox, then to hemiplane

ENCLOSURE RADIATION, VIEW FACTOR

- Enclosure radiation, view factors to be calculated during MD Nastran run

CHBDYG	EID		TYPE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		
	G1	G2	G3	G4	G5	G6	G7	G8	

RADCAV	ICAVITY	ELEAMB	SHADOW	SCALE	PRTPOCH	NFECI	RMAX	NCOMP	
	SET11	SET12	SET21	SET22	SET31	SET32	Etc.		

VIEW	IVIEW	ICAVITY	SHADE	NB	NG	DISLIN			

VIEW3D	ICAVITY	GITB	GIPS	CIER	ETOL	ZTOL	WTOL	RADCHK	

ENCLOSURE RADIATION, VIEW FACTOR

- CHBDYG identifies the radiation surface geometry and material.
- VIEW groups the surface elements into their respective radiation cavities and provides control information for using the finite difference method when determining view factors (VIEW module). The VIEW Bulk Data entry also specifies that view factors are to be calculated.
- RADCAV identifies the characteristics of each radiation cavity and provides control information for using the finite difference or contour integration method when determining view factors (VIEW module).
- VIEW3D provides the control quantities involved in using the Gaussian integration adaptive view factor module (VIEW3D module).

CHBDYG SURFACE OF CAVITY

CHBDYG

Geometric Surface Element Definition (Grid Form)

Defines a boundary condition surface element without reference to a property entry.

Format:

1	2	3	4	5	6	7	8	9	10
CHBDYG	EID		TYPE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		
	G1	G2	G3	G4	G5	G6	G7	G8	

Example:

CHBDYG	2		AREA4	3	3	2	2		
	100	103	102	101					

Field

Contents

EID	Surface element identification number. (Unique (0 < Integer < 100,000,000) among all elemental entries)
TYPE	Surface type. See Remark 3. (Character)
IVIEWF	A VIEW entry identification number for the front face. See Remark 2. for default. (Integer > 0)
IVIEWB	A VIEW entry identification number for the back face. See Remark 2. for default. (Integer > 0)
RADMIDF	RADM identification number for front face of surface element. See Remark 2. for default. (Integer > 0)
RADMIDB	RADM identification number for back face of surface element. See Remark 2. for default. (Integer ≥ 0)
Gi	Grid point IDs of grids bounding the surface. (Integer > 0)

VIEW GROUPS SURFACE ELEMENTS

VIEW

View Factor Definition

Defines radiation cavity and shadowing for radiation view factor calculations.

Format:

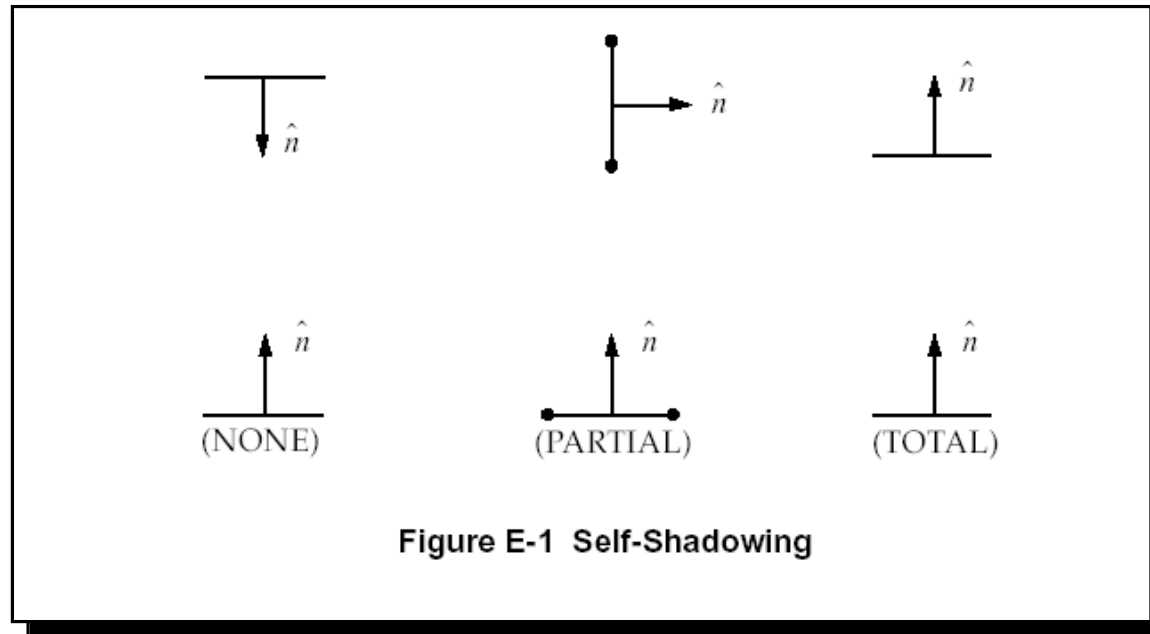
1	2	3	4	5	6	7	8	9	10
VIEW	IVIEW	ICAVITY	SHADE	NB	NG	DISLIN			

Example:

VIEW	1	1	BOTH	2	3	0.25			

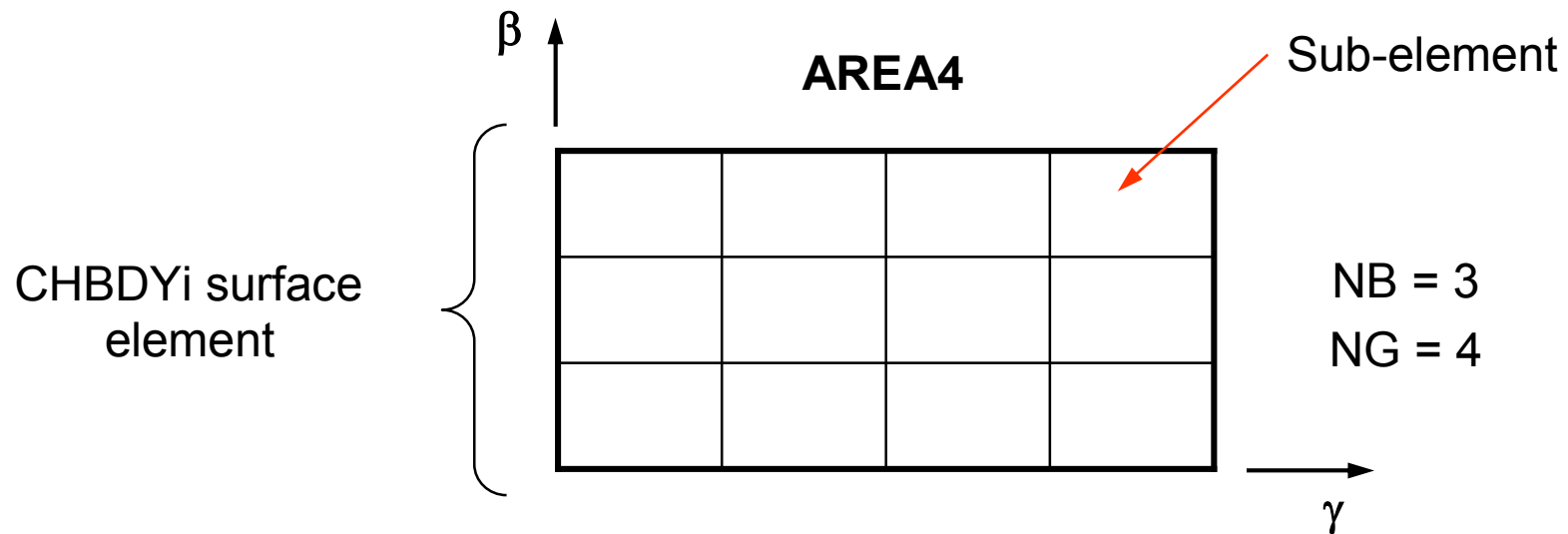
Field	Contents
IVIEW	Identification number. (Integer > 0)
ICAVITY	Cavity identification number for grouping the radiant exchange faces of CHBDYi elements. (Integer > 0)
SHADE	Shadowing flag for the face of CHBDYi element. (Character, Default = "BOTH") NONE means the face can neither shade nor be shaded by other faces. KSHD means the face can shade other faces. KBSHD means the face can be shaded by other faces. BOTH means the face can both shade and be shaded by other faces. (Default)
NB	Subelement mesh size in the beta direction. (Integer > 0; Default = 1)
NG	Subelement mesh size in the gamma direction. (Integer > 0; Default = 1)
DISLIN	The displacement of a surface perpendicular to the surface. See Figure 8-203 . (Real; Default = 0.0)

SELF-SHADOWING IN VIEW FACTOR CALCULATIONS



VIEW GROUPS SURFACE ELEMENTS (Cont.)

- VIEW module, VIEW and RADCAV Bulk Data entry, Finite Difference or Contour Integration method
- Parameters NB and NG are used to specify the number of sub-elements in two orthogonal directions



GAUSSIAN INTEGRATION METHOD

VIEW3D

View Factor Definition - Gaussian Integration Method

Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.

Format:

1	2	3	4	5	6	7	8	9	10
VIEW3D	ICAVITY	GITB	GIPS	CIER	ETOL	ZTOL	WTOL	RADCHK	

Example:

VIEW3D	1	2	2	4		1.0E-6			

Field	Contents
ICAVITY	Radiant cavity identification number on RADCAV entry. (Integer > 0)
GITB	Gaussian integration order to be implemented in calculating net effective view factors in the presence of third-body shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default = 4)
GIPS	Gaussian integration order to be implemented in calculating net effective view factors in the presence of self-shadowing. (Integer 2, 3, 4, 5, 6 or 10; Default = 4)
CIER	Discretization level used in the semi-analytic contour integration method. ($1 \leq \text{Integer} \leq 20$; Default = 4)
ETOL	Error estimate above which a corrected view factor is calculated using the semi-analytic contour integration method. (Real ≥ 0.0 ; Default = 0.1)
ZTOL	Assumed level of calculation below which the numbers are considered to be zero. (Real ≥ 0.0 ; Default = 1.E-10)
WTOL	Assumed degree of warpage above which the actual value of F_{ii} will be calculated. ($0.0 \leq \text{Real} \leq 1.0$; Default = 0.01)

GAUSSIAN INTEGRATION METHOD (Cont.)

- VIEW3D module, VIEW3D Bulk Data entry, Gaussian Integration method
- Is semi-adaptive, causing, upon detection of excessive error or shadowing, automatic invoking of semi-analytic contour integration or higher order Gaussian integration to reduce to the error
- For general 3D geometry, this method is superior for both accuracy and speed to those methods available with the VIEW module
- The Gaussian Integration method is designed for use with general 3D and axi-symmetric geometry. The view factors for planar geometry can only be calculated using VIEW module methods, not VIEW3D Gaussian Integration.



GAUSSIAN INTEGRATION METHOD (Cont.)

- No surface sub-division is available for the Gaussian Integration method. Therefore, create a good finite element mesh. Accuracy can be substantially controlled by using integration order.
- When there are a large number of surfaces in an enclosure, ≥ 1000 , it may be good to reduce the values of GITB, GIPS, and CIER to 2.

CHARACTERISTICS OF ENCLOSURE

RADCAV

Radiation Cavity Identification

Identifies the characteristics of each radiant enclosure.

Format:

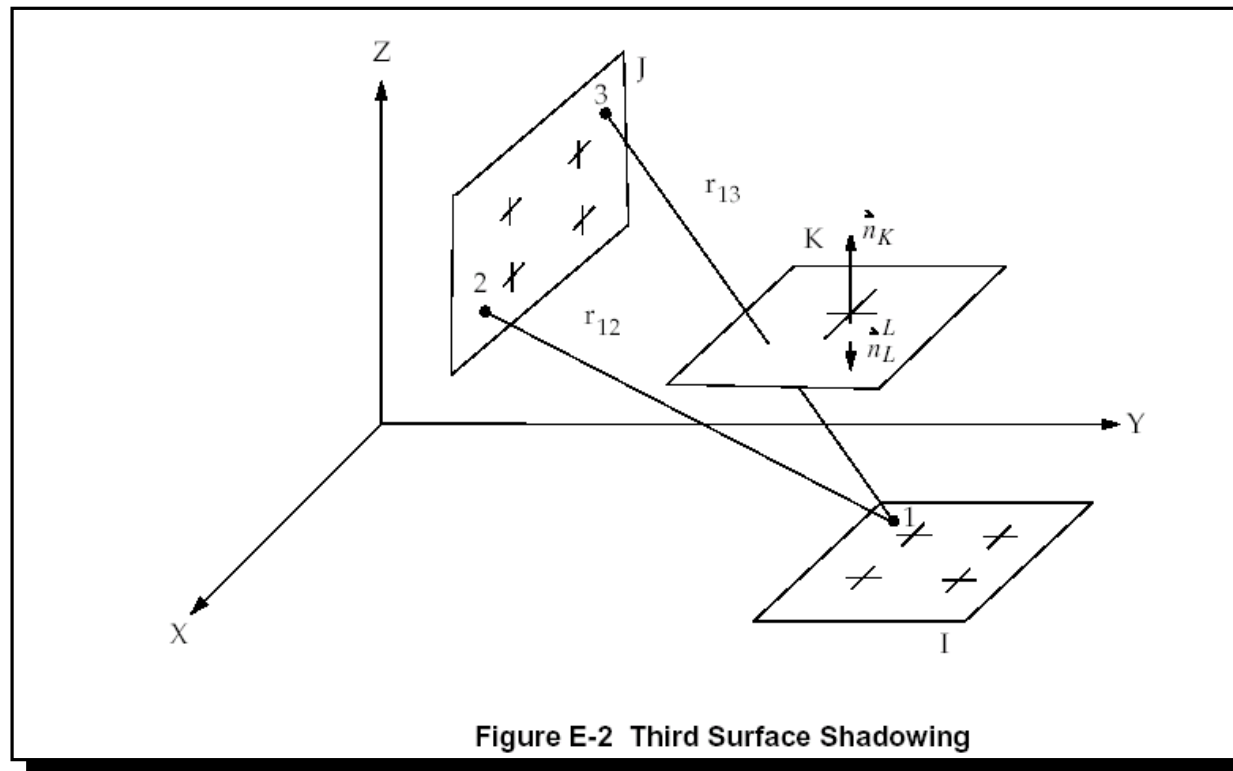
1	2	3	4	5	6	7	8	9	10
RADCAV	ICAVITY	ELEAMB	SHADOW	SCALE	PRTPCH	NFECI	RMAX	NCOMP	
	SET11	SET12	SET21	SET22	SET31	SET32	etc.		

Example:

RADCAV	1	1					.99		
	3	5	4	5	7	5			

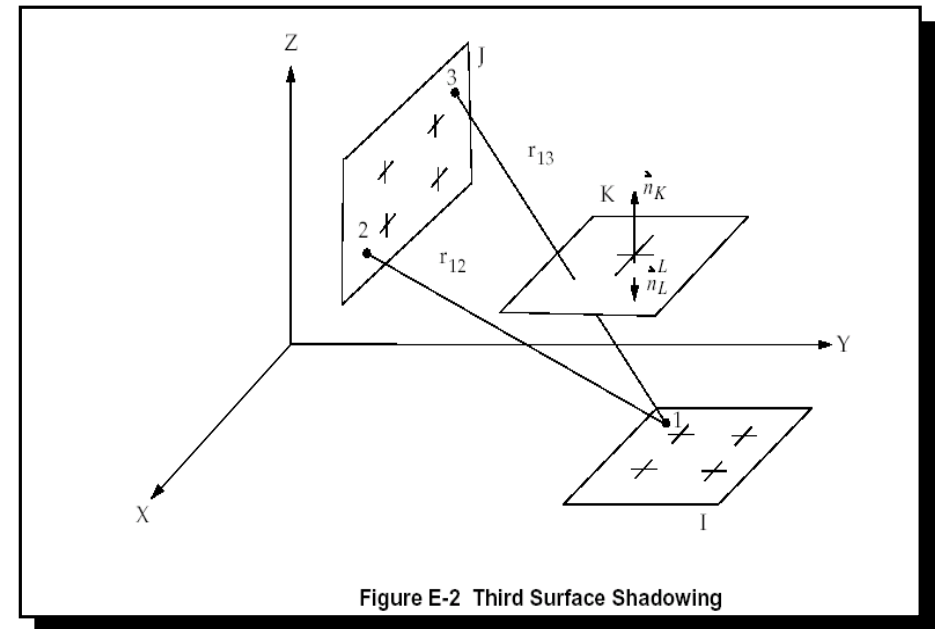
Field	Contents
ICAVITY	Unique cavity identification number associated with enclosure radiation. (Integer > 0)
ELEAMB	CHBDYi surface element identification number for radiation if the view factors add up to less than 1.0. (Unique Integer > 0 among all CHBDYi elements or blank.)
SHADOW	Flag to control third body shading calculation during view factor calculation for each identified cavity. (Character = "YES" or "NO"; Default = "YES")
SCALE	View factor that the enclosure sum will be set to if a view factor is greater than 1.0. ($0.0 \leq \text{Real} \leq 1.0$; Default = 0.0)
PRTPCH	Facilitates the blocking of view factor printing and punching onto RADLST and RADMTX entries. (Integer = 0, 1, 2, 3, 4, or 5; Default = blank):

THIRD SURFACE SHADOWING



SHADOWING CONSIDERATION

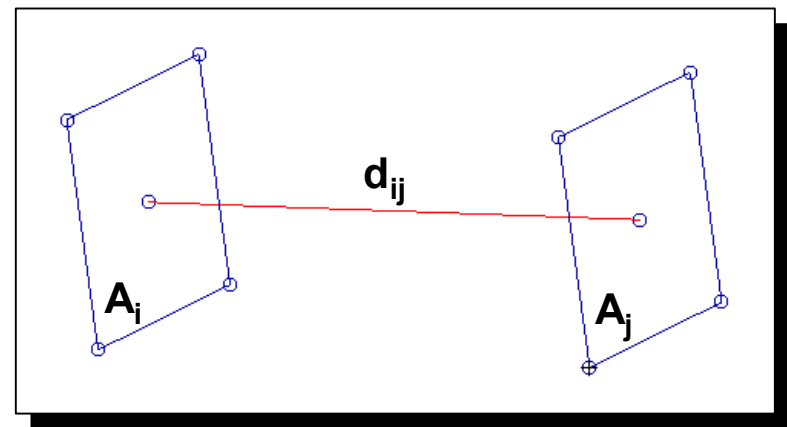
- Shadowing calculations use considerable resources while calculating view factors. If the geometry is such that no shadowing can occur, it is recommended to not use that calculation capability. Use the SHADOW field of the RADCAV entry to do this. Specifying NO will cause no shadowing calculations to be performed.
- If the geometry is very complex, then one should set the SHADE flag of the VIEW entry to be BOTH. This means that MD NASTRAN will then be used to determine which surfaces cause others to be shaded.
- If a limited subset of surfaces in a problem is involved in shadowing, the most efficient calculations will be performed if the SHADE field of the VIEW Bulk Data entry is appropriately specified, e.g. KSHD.



CHARACTERISTICS OF ENCLOSURE (Cont.)

- RADCAV Bulk Data entry
- NFECI entry
 - NFECI = FD causes the finite difference method to be used to calculate the view factors (only VIEW module used)
 - NFECI = CONT causes the contour integration method to be used to calculate the view factors (only VIEW module used)
 - NFECI = (blank) or 0 causes either the finite difference or contour integration method to be used, depending on the value of the RMAX entry
 - Contour integration is used for all pairs of surface elements, A_i and A_j , for which

$$\frac{A_j}{d_{ij}^2} > RMAX$$



USE HEMI-CUBE METHOD TO CALC VF

- This method can be used for all SOL types
- To use this method include an entry in the Bulk Data
 - NLMOPTS, HEMICUBE

NLMOPTS (SOL 400)

Nonlinear Material Options

Specifies nonlinear material optional schemes.

Format:

1	2	3	4	5	6	7	8	9	10
NLMOPTS	"CREEP"	valc1	valc2	valc3	valc4				
	"ASSM"	vala							
	"TSHEAR"	vals							
	"LRGSTRN"	valle							
	HEMICUBE	Value	NPIXEL		CUTOFF	FRACTION	FACCNT	FACTOL	

Examples:

NLMOPTS	CREEP	2							
NLMOPTS	HEMICUBE	1	500		0	0.01			

USE HEMI-CUBE METHOD TO CALC VF (Cont.)

- To use this method include an entry in the Bulk Data
 - NLMOPTS, HEMICUBE

HEMICUBE	Keyword to select the view factor calculation method in MD Nastran.
Field 3	VALUE=0 - Use Nastran finite difference, contour integration, or Gaussian integration method (Default).
Field 3	VALUE=1- (HEMI) Use pixel based modified hemicube method.
Field 4	NPIXEL=500 - Enter the number of pixels (Default = 500).
Field 6	CUTOFF=0 - Enter the fraction of the maximum view factor that is to be used as a cutoff. View factors calculated below this cutoff are ignored (Default = 0).
Field 7	FRACTION - Enter the fraction of the maximum view factors that is to be treated implicitly. View factors values smaller than this cutoff are treated explicitly (Fraction = 1.0E-2).
Field 8	Facnt = set 1 to activate explicit treatment of reflection matrix. Default = 0)
Field 9	Factol = tolerance to be used on the above iteration on the Poljak equations.

USE HEMI-CUBE METHOD TO CALC VF (Cont.)

- The time to calculate the view factors using the Hemi-cube method can be reduced using the explicit treatment of the reflection matrix in the RMG2 module.
- The 8th field of the NLMOPTS entry can be set to 1 to use the explicit treatment of the reflection matrix.
- Because the reflection matrix is never calculated and factorized, the calculation of the radiation exchange matrix requires significantly less effort.
- The Hemi-cube method is less accurate (it is approximate) than the Contour Integration method, but using it for very large radiation enclosure problems will produce accurate enough results, and will greatly reduce the time necessary to calculate the view factors.

RADIATION MATERIAL PROPERTIES

RADM

Radiation Boundary Material Property

Defines the radiation properties of a boundary element for heat transfer analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RADM	RADMID	ABSORP	EMIS1	EMIS2	EMIS3	EMIS4	EMIS5	EMIS6	
	EMIS7	-etc.-							

Example:

RADM	11		.45	.33	.29	.20	.17	.13	

Field	Contents
RADMID	Material identification number. (Integer > 0)
ABSORP	Surface absorptivity or the temperature function curve multiplier if ABSORP is variable. See Remark 2. ($0.0 \leq \text{Real} \leq 1.0$)
EMISi	Surface emissivity at wavelength LAMBDAi or the temperature function curve multiplier if EMISi is variable (See the RADBNB entry.) ($0.0 \leq \text{Real} \leq 1.0$)

COLLECTION OF CAVITIES FOR AN ANALYSIS

RADSET

Identifies a Set of Radiation Cavities

Specifies which radiation cavities are to be included for radiation enclosure analysis.

Format:

1	2	3	4	5	6	7	8	9	10
RADSET	ICAVITY1	ICAVITY2	ICAVITY3	ICAVITY4	ICAVITY5	ICAVITY6	ICAVITY7	ICAVITY8	
	ICAVITY9	-etc.-							

Example:

RADSET	1	2	3	4					

Field	Contents
ICAVITYi	Unique identification number for a cavity to be considered for enclosure radiation analysis. (Integer > 0)

Remark:

1. For multiple radiation cavities, RADSET specifies which cavities are to be included in the analysis.

COMPARISON OF VF METHODS

- 2D geometry

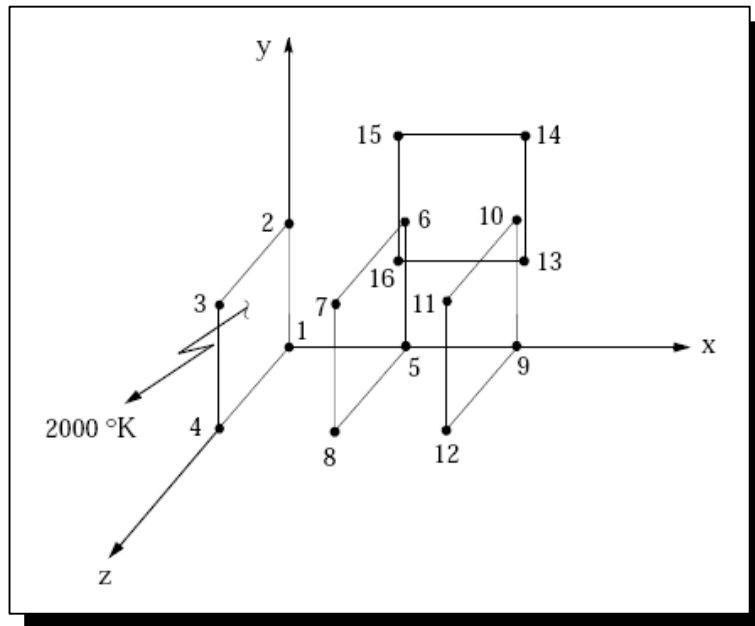
Method (Module)	Accuracy	Speed
Finite Difference (VIEW)	Less accurate than Contour Integration	Faster than Contour Integration
Contour Integration (VIEW)	Most accurate of all methods	Slowest of all methods
Gaussian Integration (VIEW3D)	N/A for 2D	N/A for 2D
Hemi-cube	Less than Contour Integration, but ok for large problems	Fastest for large problems

COMPARISON OF VF METHODS (Cont.)

- 3D geometry

Method (Module)	Accuracy	Speed
Finite Difference (VIEW)	Less accurate than Contour Integration	Faster than Contour Integration
Contour Integration (VIEW)	More accurate than Finite Difference	Slowest of all methods
Gaussian Integration (VIEW3D)	More accurate than VIEW module methods	Faster than VIEW module methods
Hemi-cube	Less than Contour Integration, but ok for large problems	Fastest for large problems

EXAMPLE 5A, SINGLE CAVITY ENCLOSURE RADIATION WITH SHADOWING



The statements essential to the radiation solution process are described as follows:

RADSET	Requests which cavities are to be included as radiation enclosures for the thermal analysis.
RADLST / RADMTX	Provides the view factors required for generation of the radiation matrix. Since they are not provided by the user in this example, they are determined by use of the view module in the course of the analysis.
RADCAV	Provides various global controls used for the calculation of view factors within the identified cavity.
VIEW	Provides the connection between a surface element and its assigned cavity and requests that view factors be calculated among those surface elements assigned to the same cavity.
VIEW3D	Requests that the view factors be calculated using the adaptive gaussian integration view factor routine as opposed to the default finite difference calculation.
CHBDYi	Describes the surface elements used in the enclosure, and associates them with the VIEW and RADM Bulk Data entries.
RADM	Provides the radiative surface properties (emissivity), in this case a constant value of 1.0.

EXAMPLE 5A, MD NASTRAN INPUT

```
SOL 153
TIME 10
CEND
TITLE = EXAMPLE 5a
EX5A
ANALYSIS = HEAT
THERMAL = ALL
FLUX = ALL
SPCF = ALL
OLOAD = ALL
SPC = 10
TEMP(INIT) = 20
NLPARM = 100
BEGIN BULK
PARAM,TABS,0.0
PARAM,SIGMA,5.67E-08
NLPARM,100
$
GRID,1,,0.0,0.0,0.0
GRID,2,,0.0,1.0,0.0
GRID,3,,0.0,1.0,1.0
GRID,4,,0.0,0.0,1.0
GRID,5,,1.0,0.0,0.0
GRID,6,,1.0,1.0,0.0
GRID,7,,1.0,1.0,1.0
GRID,8,,1.0,0.0,1.0
GRID,9,,2.0,0.0,0.0
GRID,10,,2.0,1.0,0.0
GRID,11,,2.0,1.0,1.0
GRID,12,,2.0,0.0,1.0
GRID,13,,1.5,0.0,-1.0
GRID,14,,1.5,1.0,-1.0
GRID,15,,0.5,1.0,-1.0
GRID,16,,0.5,0.0,-1.0
```

```
CQUAD4,1,5,1,2,3,4
CQUAD4,2,5,5,6,7,8
CQUAD4,3,5,9,12,11,10
CQUAD4,4,5,13,14,15,16
PSHELL,5,15,0.1
MAT4,15,204.0
$
CHBDYG,10,,AREA4,55,,45,,,+CHG10
+CHG10,1,2,3,4
CHBDYG,20,,AREA4,56,,45,,,+CHG20
+CHG20,5,6,7,8
CHBDYG,21,,AREA4,56,,45,,,+CHG21
+CHG21,5,8,7,6
CHBDYG,30,,AREA4,55,,45,,,+CHG30
+CHG30,9,12,11,10
CHBDYG,40,,AREA4,57,,45,,,+CHG40
+CHG40,13,14,15,16
$
RADM,45,1.0,1.0
RADSET,65
RADCAV,65,,YES
VIEW,55,65,KBSHD
VIEW,56,65,KSHD
VIEW,57,65,NONE
VIEW3D,65,,,,,,3
$
SPC,10,1,,2000.0,2,,2000.0
SPC,10,3,,2000.0,4,,2000.0
TEMPD,20,2000.0
$
ENDDATA
```

EXAMPLE 5A, MD NASTRAN VF OUTPUT

*** VIEW FACTOR MODULE *** OUTPUT DATA *** CAVITY ID = 65 ***

SURF-I	SURF-J	AREA-I	AI*FIJ	C* PARTIAL FIJ	ERROR	SHADING	ERROR	SCALE
10 -	21	1.0000E+00	1.97750E-01	1.97750E-01	2.5529E-01	NO	YES	
10	30	1.0000E+00	6.84135E-02	6.84135E-02	7.3895E-02	NO	NO	
10	40	1.0000E+00	4.08547E-02	4.08547E-02	6.6278E-02	NO	NO	
10	30	1.0000E+00	0.00000E+00	0.00000E+00	0/256			
10	21	1.0000E+00	1.99944E-01	1.99944E-01				
10 -SUM OF			2.40799E-01	2.40799E-01				
20 -	30	1.0000E+00	1.97750E-01	1.97750E-01	2.5529E-01	NO	YES	
20	40	1.0000E+00	1.31841E-02	1.31841E-02	2.0133E-02	YES	NO	
20	40	1.0000E+00	1.16713E-02	1.16713E-02				
20	30	1.0000E+00	1.99944E-01	1.99944E-01				
20 -SUM OF			2.11616E-01	2.11616E-01				
21 -	40	1.0000E+00	1.31841E-02	1.31841E-02	2.0133E-02	YES	NO	
21	40	1.0000E+00	1.16713E-02	1.16713E-02				
21 -SUM OF			2.11616E-01	2.11616E-01				
30 -	40	1.0000E+00	4.08547E-02	4.08547E-02	6.6278E-02	NO	NO	
30 -SUM OF			2.40799E-01	2.40799E-01				
40 -SUM OF			1.05052E-01	1.05052E-01				

EXAMPLE 5A, TEMPERATURE OUTPUT

LOAD STEP = 1.00000E+00

TEMPERATURE VECTOR

POINT ID.	TYPE	ID VALUE	ID+1 VALUE	ID+2 VALUE	ID+3 VALUE	ID+4 VALUE	ID+5 VALUE
1	S	2.000000E+03	2.000000E+03	2.000000E+03	2.000000E+03	1.132229E+03	1.132229E+03
7	S	1.132229E+03	1.132229E+03	7.732048E+02	7.732048E+02	7.732048E+02	7.732048E+02
13	S	9.168311E+02	9.168311E+02	9.168311E+02	9.168311E+02		

0

LOAD STEP = 1.00000E+00

LOAD VECTOR

POINT ID.	TYPE	ID VALUE	ID+1 VALUE	ID+2 VALUE	ID+3 VALUE	ID+4 VALUE	ID+5 VALUE
13	S	0.0	0.0	0.0	0.0		

0

LOAD STEP = 1.00000E+00

FORCES OF SINGLE-POINT CONSTRAINT

POINT ID.	TYPE	ID VALUE	ID+1 VALUE	ID+2 VALUE	ID+3 VALUE	ID+4 VALUE	ID+5 VALUE
1	S	2.217105E+05	2.217105E+05	2.217105E+05	2.217105E+05	0.0	0.0
13	S	0.0	0.0	0.0	0.0		

LOAD STEP = 1.00000E+00

HEAT FLOW INTO HBDY ELEMENTS (CHBDY)

ELEMENT-ID	APPLIED-LOAD	FREE-CONVECTION	FORCED-CONVECTION	RADIATION	TOTAL
10	0.000000E+00	0.000000E+00	0.000000E+00	-8.868420E+05	-8.868420E+05
20	0.000000E+00	0.000000E+00	0.000000E+00	-8.865104E+04	-8.865104E+04
21	0.000000E+00	0.000000E+00	0.000000E+00	8.865102E+04	8.865102E+04
30	0.000000E+00	0.000000E+00	0.000000E+00	4.393639E-03	4.393639E-03
40	0.000000E+00	0.000000E+00	0.000000E+00	6.051011E-03	6.051011E-03

EXAMPLE 5A, RADIATION PUNCH FILE

\$RADMTX, ICAVITY, INDEX, $F_{i,j}$, $F_{i+1,j}$, $F_{i+2,j}$, ...

RADMTX 65 1 0.0 0.0 .199944 0.0 .040855

RADMTX 65 2 0.0 0.0 .199944 .011671

RADMTX 65 3 0.0 0.0 .011671

RADMTX 65 4 0.0 .040855

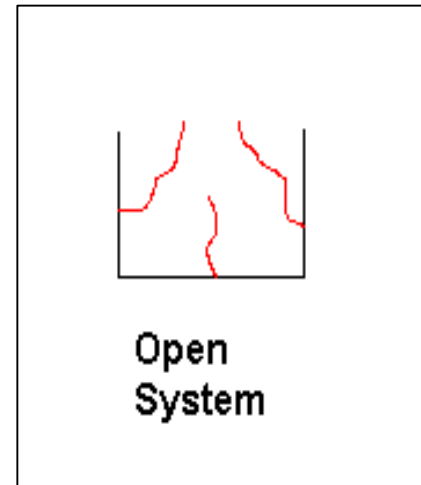
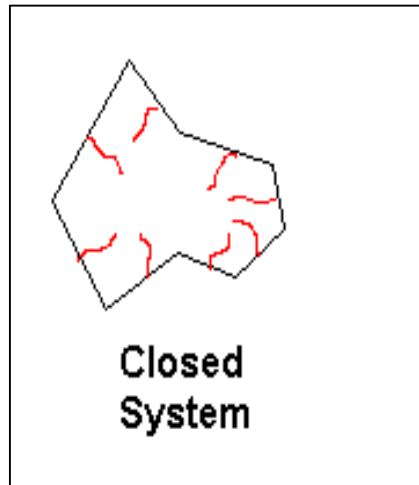
RADMTX 65 5 0.0

\$RADLST, ICAVITY, MTXTYP, EID1, EID2, EID3, ...

RADLST 65 1 10 20 21 30 40

- Compare the RADMTX data with that for the .f06 file.
- The data for one RADMTX entry corresponds to a specified column of the radiation exchange matrix.

ENCLOSURES

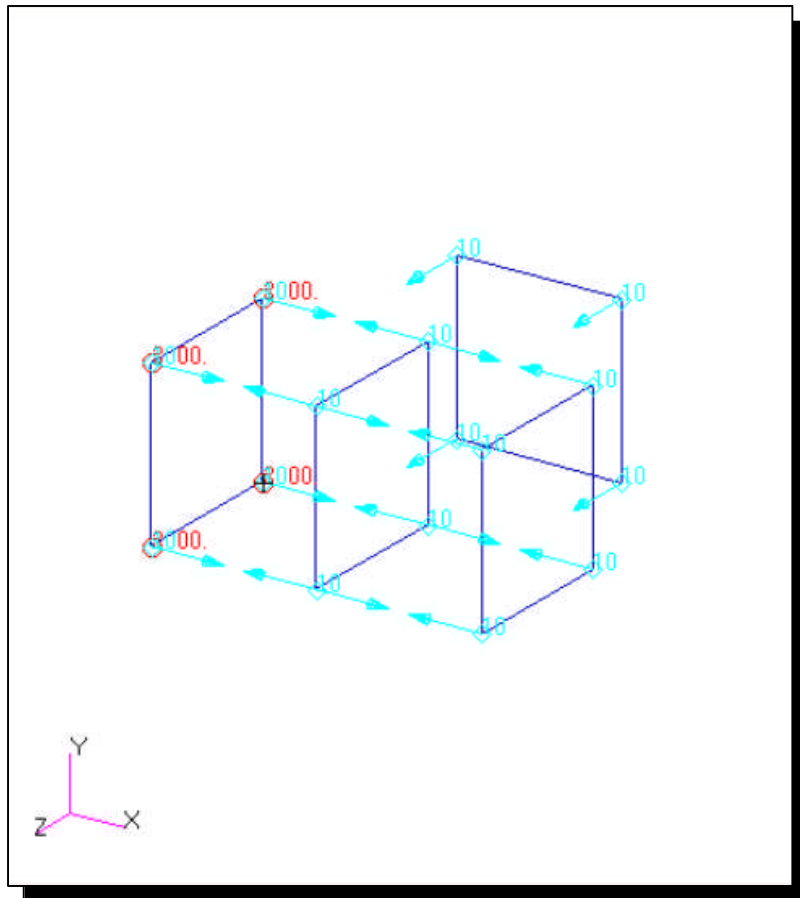


- For closed enclosures, the sum of the viewfactors is 1.0.
- In general, open enclosures lose energy that does not “see” another surface.
- MD Nastran provides ambient elements to account for losses to space for open enclosures.

BULK DATA, ENCLOSURE RADIATION

- Can use multiple enclosures in MD Nastran to divide a model into a set of smaller parts, hence reducing the number of view factor calculations.

PROJECT 10, SINGLE CAVITY ENCLOSURE RADIATION



- Three 1x1x0.1m plates are disposed along the basic X-axis at 1m intervals; another plate is off to one side.
 - Each plate is represented by one CQUAD4 element.
- The left-most plate is held at 2000.0°K, and the ambient space temperature is at 0°K.
- The material for all CQUAD4s is aluminum ($k = 204.0 \text{ W/m/K}$).
- Perform a SS analysis to determine the equilibrium temperatures of the plates.
- Review the .DAT, .F06, and .PCH files.
- Based on Exercise 5a in the MSC. Nastran 2002 Thermal Analysis User's Guide.

PROJECT 10, SINGLE CAVITY ENCLOSURE

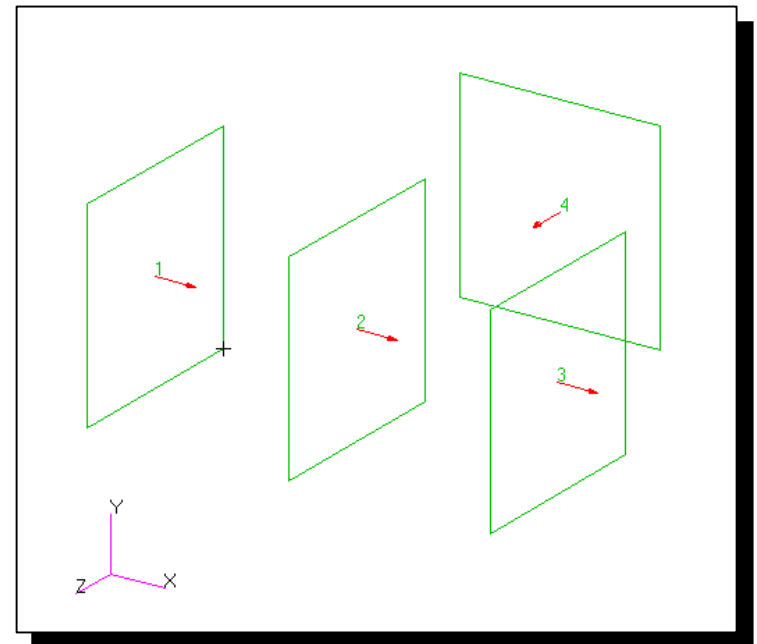
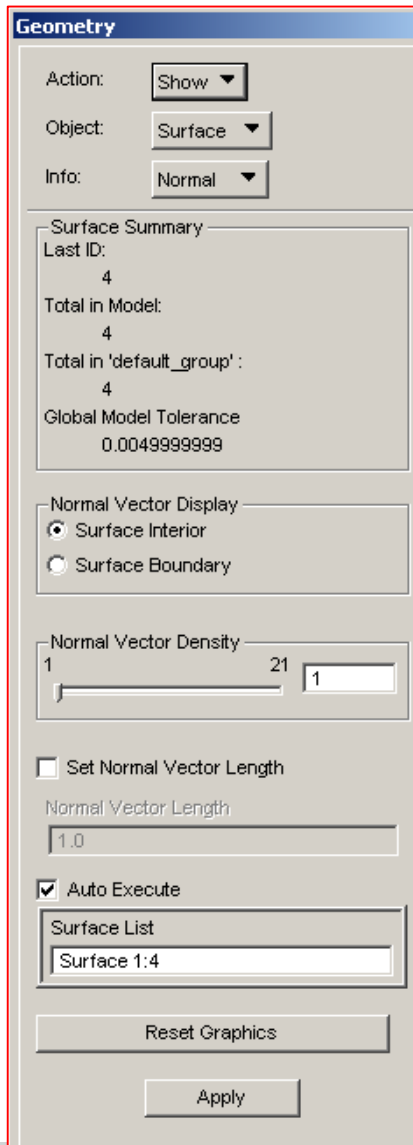
- Create first surface. It is in the YZ plane.
- Copy the first surface created twice by transforming it using the vector $\langle 1 \ 0 \ 0 \rangle$.
- Create the final surface, Surface 4, parallel to the XY plane.

The image displays three screenshots of the 'Geometry' dialog box, illustrating the steps to create Surface 4:

- Left Screenshot:** The 'Action' is set to 'Create', 'Object' is 'Surface', and 'Method' is 'XYZ'. The 'Surface ID List' contains '1'. The 'Refer. Coordinate Frame' is 'Coord 0'. The 'Vector Coordinates List' is $\langle 0 \ 1 \ 1 \rangle$. The 'Auto Execute' checkbox is checked. The 'Origin Coordinates List' is $[0 \ 0 \ 0]$. The '-Apply-' button is at the bottom.
- Middle Screenshot:** The 'Action' is set to 'Transform', 'Object' is 'Surface', and 'Method' is 'Translate'. The 'Surface ID List' contains '2'. The 'Type of Transformation' is 'Cartesian in Refer. CF'. The 'Refer. Coordinate Frame' is 'Coord 0'. The 'Translation Vector' is $\langle 1 \ 0 \ 0 \rangle$. The 'Translation Parameters Repeat Count' is '2'. The 'Delete Original Surfaces' checkbox is unchecked. The 'Auto Execute' checkbox is checked. The 'Surface List' contains 'Surface 1'. The '-Apply-' button is at the bottom.
- Right Screenshot:** The 'Action' is set to 'Create', 'Object' is 'Surface', and 'Method' is 'XYZ'. The 'Surface ID List' contains '4'. The 'Refer. Coordinate Frame' is 'Coord 0'. The 'Vector Coordinates List' is $\langle 1 \ 1 \ 0 \rangle$. The 'Auto Execute' checkbox is checked. The 'Origin Coordinates List' is $[0.5 \ 0 \ -1]$. The '-Apply-' button is at the bottom.

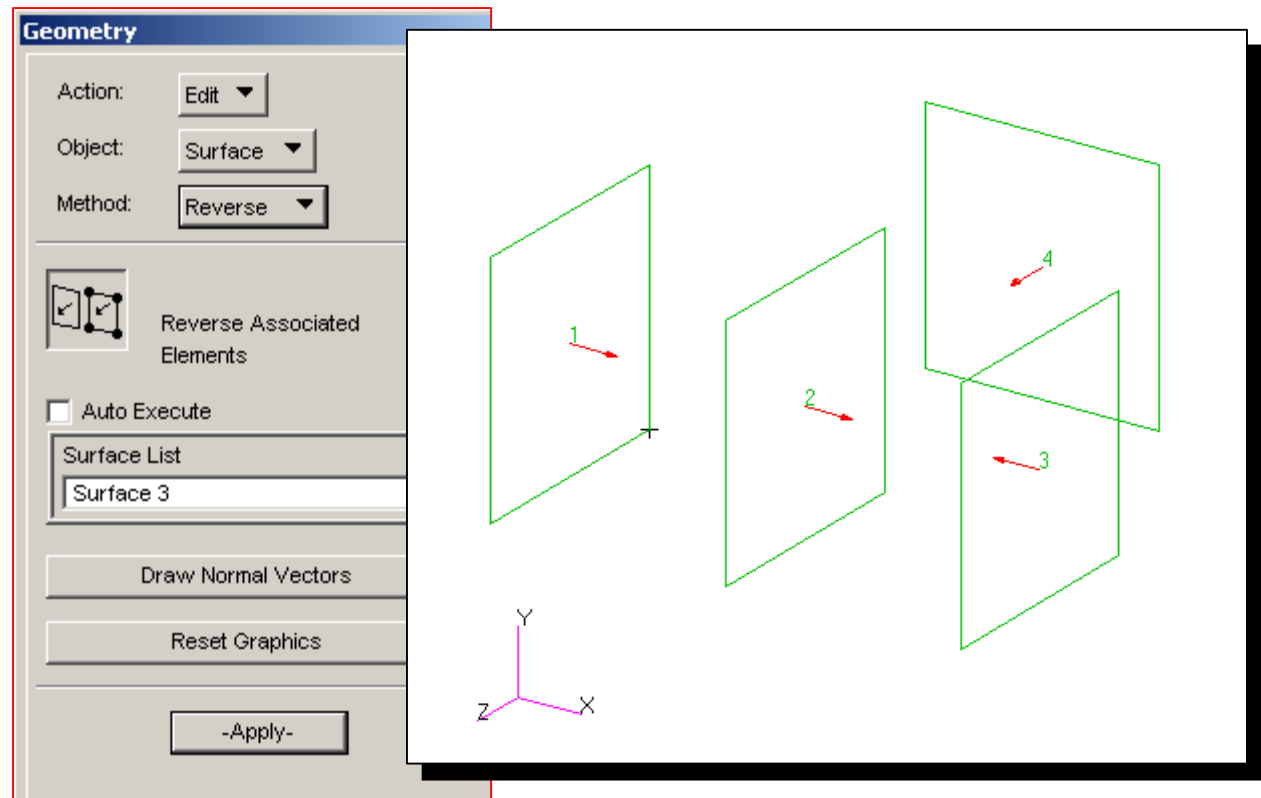
PROJECT 10, SINGLE CAVITY ENCLOSURE

- Display the normal for the surfaces.
- The normal for Surface 1:3 is in the positive X-direction.
- The normal for Surface 4 is in the positive Z-direction.



PROJECT 10, SINGLE CAVITY ENCLOSURE

- Change the direction of the normal for the surface to the right of the surfaces, Surface 3.



PROJECT 10, SINGLE CAVITY ENCLOSURE

- IsoMesh all surfaces with one CQUAD4 element per surface.
 - Surface List: Surface 1:4
 - GEL: 1.0

The screenshot shows the 'Finite Elements' dialog box with the following settings:

- Action: Create
- Object: Mesh
- Type: Surface
- Output ID List:
 - Node: 1
 - Element: 1
- Elem Shape: Quad
- Mesher: IsoMesh
- Topology: Quad4
- Buttons: IsoMesh Parameters..., Node Coordinate Frames...
- Surface List: Surface 1:4
- Global Edge Length:
 - ☐ Automatic Calculation
 - Value: 1.0
- Button: -Apply-

PROJECT 10, SINGLE CAVITY ENCLOSURE

- Create isotropic material properties for the solid plates.
 - Aluminum
 - $K = 204.0$ Watts/m/K

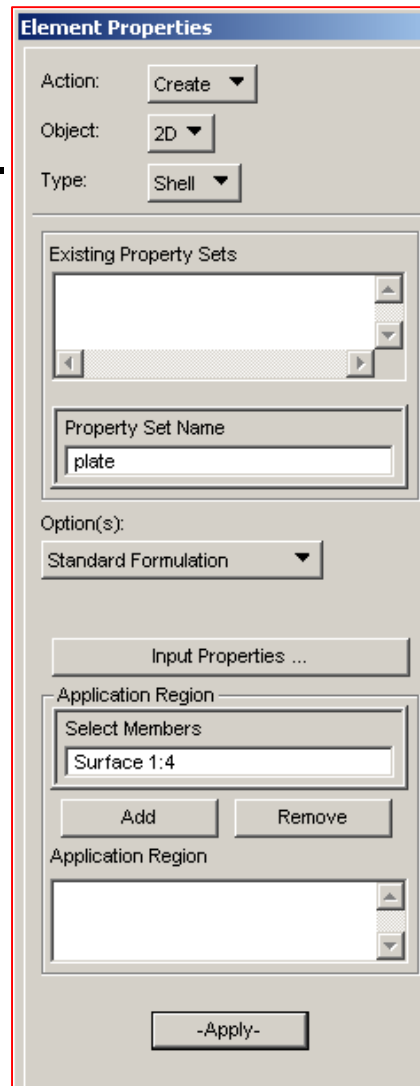
The image shows two overlapping software windows from MSC Software. The background window is titled 'Materials' and has a tabbed interface. The 'Create' tab is active, showing fields for 'Action' (set to 'Create'), 'Object' (set to 'Isotropic'), and 'Method' (set to 'Manual Input'). Below these are fields for 'Material Name' (containing 'alum') and 'Description' (containing 'Date: 29-Nov-02 Time: 15:04:47'). At the bottom are buttons for 'Input Properties ...', 'Change Material Status ...', and 'Apply'. The foreground window is titled 'Input Options' and is also a tabbed interface. The 'Solid properties' tab is active, showing a table for material properties:

Property Name	Value
Thermal Conductivity =	204
Specific Heat =	
Density =	

Below the table are sections for 'Temperature Dep/Model Variable Fields' and 'Current Constitutive Models', both with empty list boxes. At the bottom of the 'Input Options' window are buttons for 'OK', 'Clear', and 'Cancel'.

PROJECT 10, SINGLE CAVITY ENCLOSURE

- Create 2D shell element properties.
 - Aluminum
 - Thickness = 0.1 m
 - AR: Surface 1:4



Element Properties

Action:

Object:

Type:

Existing Property Sets

Property Set Name:

Option(s):

Input Properties ...

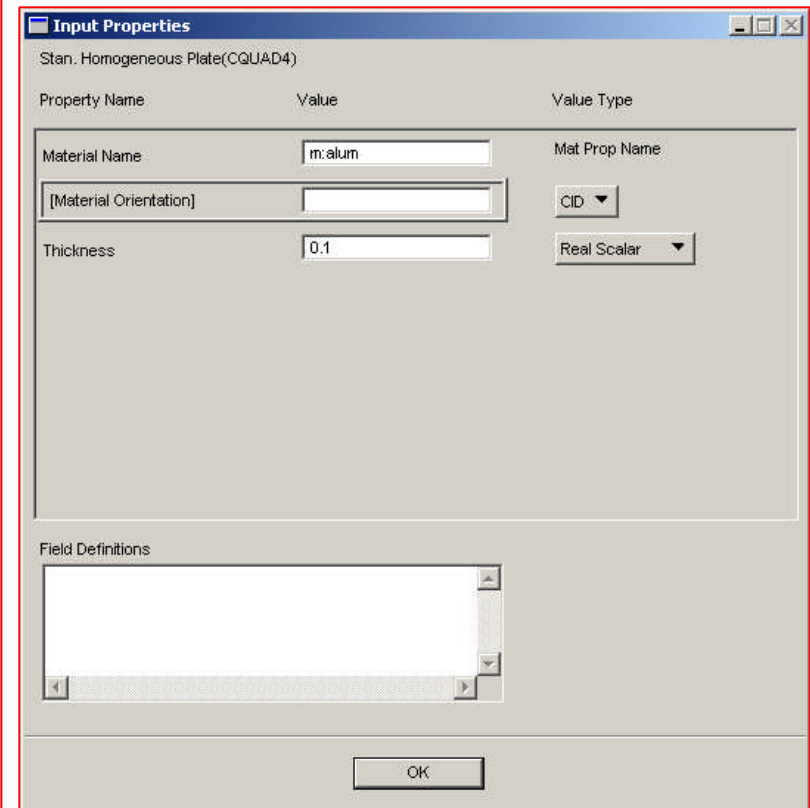
Application Region

Select Members

Add Remove

Application Region

-Apply-



Input Properties

Stan. Homogeneous Plate(CQUAD4)

Property Name	Value	Value Type
Material Name	m:alum	Mat Prop Name
[Material Orientation]		CID CID
Thickness	0.1	Real Scalar Real Scalar

Field Definitions

OK

PROJECT 10, SINGLE CAVITY ENCLOSURE

- Apply constant temperature to the left plate, Surface 1.
 - 2000°K
 - AR: Surface 1

The image shows three overlapping software dialog boxes from a finite element analysis (FEA) application. The 'Load/Boundary Conditions' dialog is the largest, with 'Action' set to 'Create', 'Object' to 'Temp (Thermal)', and 'Type' to 'Nodal'. It has a 'Current Load Case' dropdown set to 'Default...', a 'Type' dropdown set to 'Static', and an 'Existing Sets' list. Below this is a 'New Set Name' field containing 'tempbc'. At the bottom are buttons for 'Input Data...', 'Select Application Region...', and '-Apply-'. The 'Input Data' dialog is smaller and centered over the first, with 'Boundary Temperature' set to '2000' and an empty 'Spatial Fields' list. It has 'Reset', 'OK', and 'Cancel' buttons. The 'Select Application Region' dialog is on the right, with 'Geometry Filter' set to 'Geometry' and 'FEM' unselected. It has an 'Application Region' section with a 'Select Geometry Entities' dropdown set to 'Surface 1', 'Add' and 'Remove' buttons, and an 'OK' button at the bottom.

PROJECT 10, SINGLE CAVITY ENCLOSURE

- Create radiation LBC using an enclosure.
 - Surface Option: Top
 - Surface can be shaded, yes
 - Enclosure ID: 10
 - End surfaces, Surface 1,3,4

The screenshot displays three overlapping dialog boxes from the MSC Software interface:

- Load/Boundary Conditions**:
 - Action: Create
 - Object: Radiation
 - Type: Element Uniform
 - Option: Enclosures
 - Current Load Case: Default...
 - Type: Static
 - Existing Sets: (empty list)
 - New Set Name: can_be_shaded_top
 - Target Element Type: 2D
 - Buttons: Input Data..., Select Application Region..., -Apply-
- Input Data**:
 - Surface Option: Top
 - Top Surf Emissivity: 1
 - Enclosure ID: 10
 - * Temperature Function: (empty)
 - ☐ Surface Can Shade
 - ☒ Surface Can Be Shaded
 - ☒ Third Body Shading
 - ☐ Complete
 - Buttons: OK, Reset
- Select Application Region**:
 - Geometry Filter: ☒ Geometry, ☐ FEM
 - Application Region: Select Surfaces or Edges
 - Selected: Surface 1 3 4
 - Buttons: Add, Remove
 - Application Region: (empty list)
 - Button: OK

PROJECT 10, SINGLE CAVITY ENCLOSURE

- Create another radiation LBC using an enclosure.
 - Surface Option: Top
 - Surface can shade, yes
 - Enclosure ID: 10
 - Middle surface, Surface 2

The screenshot displays three overlapping dialog boxes from the MSC Software interface:

- Load/Boundary Conditions**:
 - Action: Create
 - Object: Radiation
 - Type: Element Uniform
 - Option: Enclosures
 - Current Load Case: Default...
 - Type: Static
 - Existing Sets: can_be_shaded_top
 - New Set Name: can_shade_top
 - Target Element Type: 2D
 - Buttons: Input Data..., Select Application Region..., -Apply-
- Input Data**:
 - Surface Option: Top
 - Top Surf Emissivity: 1
 - ☒ Surface Can Shade
 - ☐ Surface Can Be Shaded
 - Enclosure ID: 10
 - * Temperature Function: (empty)
 - ☒ Third Body
 - ☐ Complete
 - Buttons: OK, Reset
- Select Application Region**:
 - Geometry Filter: ☒ Geometry, ☐ FEM
 - Application Region: Select Surfaces or Edges, Surface 2
 - Buttons: Add, Remove
 - Application Region: (empty list box)
 - Button: OK

PROJECT 10, SINGLE CAVITY ENCLOSURE

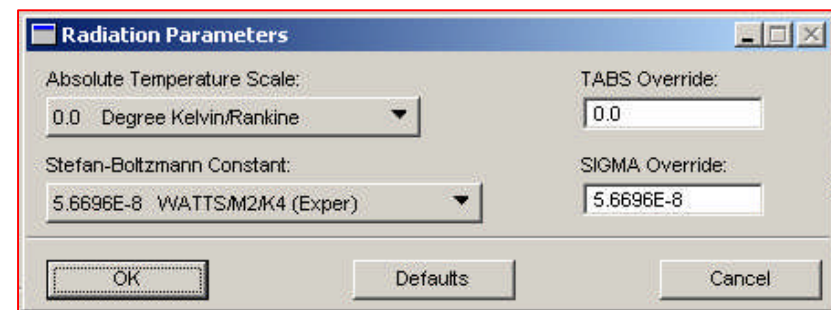
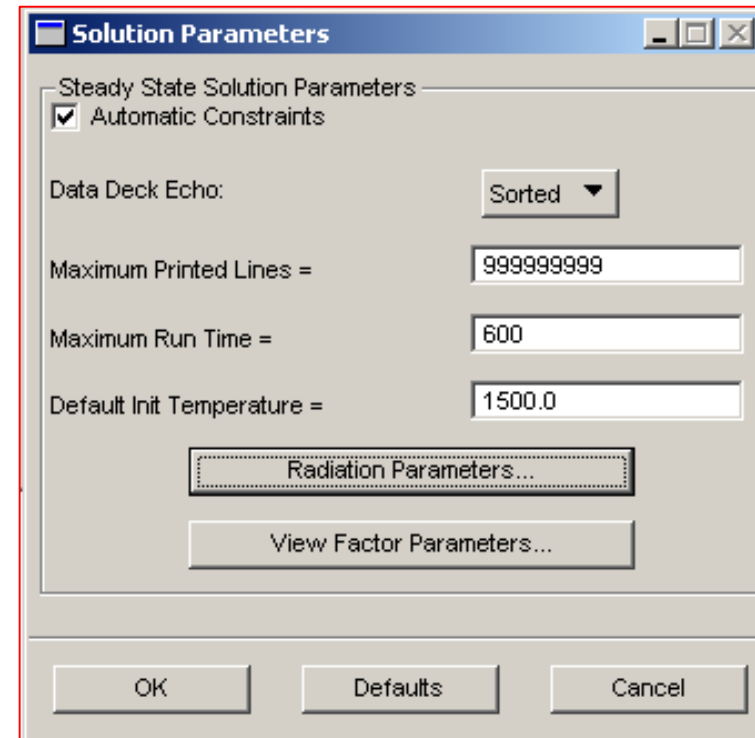
- Create another radiation LBC using an enclosure.
 - Surface Option: Bottom
 - Surface can shade, yes
 - Enclosure ID: 10
 - Middle surface, Surface 2

The screenshot displays three overlapping dialog boxes from the MSC Software interface:

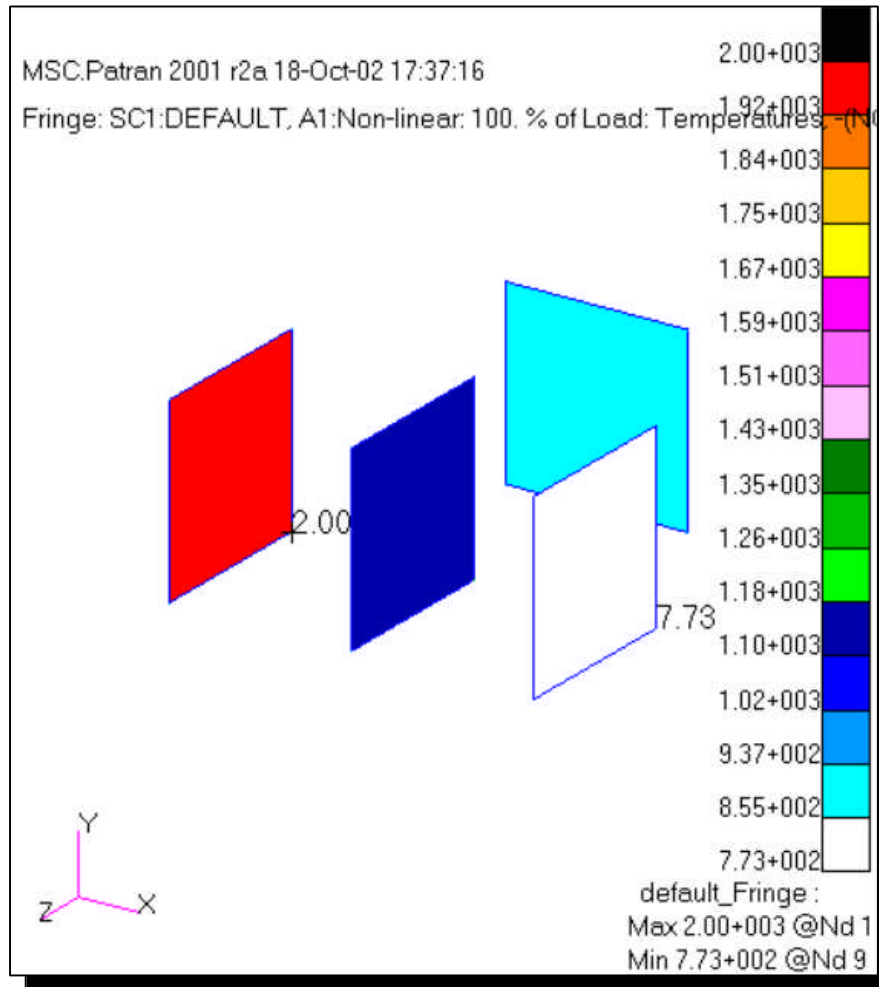
- Load/Boundary Conditions**:
 - Action: Create
 - Object: Radiation
 - Type: Element Uniform
 - Option: Enclosures
 - Current Load Case: Default...
 - Type: Static
 - Existing Sets: can_be_shaded_top, can_shade_top
 - New Set Name: can_shade_bottom
 - Target Element Type: 2D
 - Buttons: Input Data..., Select Application Region..., -Apply-
- Input Data**:
 - Surface Option: Bottom
 - Enclosure ID: 10
 - Bottom Surf Emissivity: 1
 - * Temperature Function: (empty)
 - ☒ Surface Can Shade
 - ☐ Surface Can Be Shaded
 - ☒ Third
 - ☐ Comp
 - Spatial Fields: (empty)
 - Temperature: (empty)
 - Buttons: OK, Reset
- Select Application Region**:
 - Geometry Filter:
 - ☒ Geometry
 - ☐ FEM
 - Application Region:
 - Select Surfaces or Edges: Surface 2
 - Buttons: Add, Remove
 - Application Region: (empty)
 - Button: OK

PROJECT 10, SINGLE CAVITY ENCLOSURE

- Run analysis Analysis:
Analyze/Entire Model/Full
Run
- Solution Type
 - Steady State
 - Solution Parameters
 - Radiation Parameters



PROJECT 10, SINGLE CAVITY ENCLOSURE



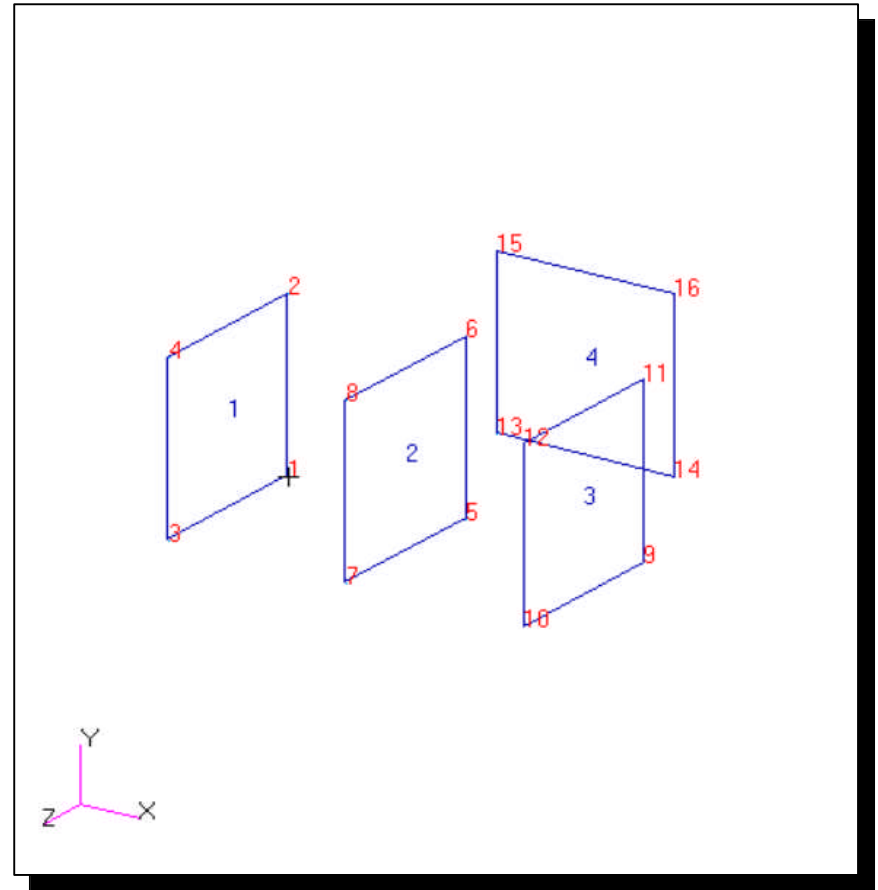
- The hottest plate is to the left at 2000°K . The coldest plate is at the right at 773°K .

PROJECT 10, SINGLE CAVITY ENCLOSURE

Extracted from MD Nastran model .DAT file

```
$ Radiation Enclosures of Load Set : top
RADCAV  10          0.      0
VIEW3D*  10          4          4          4          *      B
*      B .1          1.-10      .01          3
VIEW      1          10      KBSHD
$ Radiation Enclosures of Load Set : middle_top
VIEW      2          10      KSHD
$ Radiation Enclosures of Load Set : middle_bottom
VIEW      3          10      KSHD
RADSET  10
```

PROJECT 10, SINGLE CAVITY ENCLOSURE



CHBDYG	1		AREA4	1		1		+	C
+	C	1	2	4	3				
CHBDYG	2		AREA4	1		1		+	D
+	D	9	10	12	11				
CHBDYG	3		AREA4	1		1		+	E
+	E	13	14	16	15				
CHBDYG	4		AREA4	2	3	1	1	+	F
+	F	5	6	8	7				
\$ Radiation Material Properties									
RADM	1			1.					

Extracted from MD Nastran model .DAT file

PROJECT 10, SINGLE CAVITY ENCLOSURE

Extract from MD Nastran ASCII Results File

0

```

*** VIEW FACTOR MODULE *** OUTPUT DATA *** CAVITY ID =      10 ***

```

SURF-I	SURF-J	AREA-I	ELEMENT TO ELEMENT AI*FIJ	VIEW FACTORS FIJ	C* ERROR	PARTIAL SHADING	ERROR	SCALE
1 -	2	1.0000E+00	6.84135E-02	6.84135E-02	7.3895E-02	NO	NO	
1	3	1.0000E+00	4.08547E-02	4.08547E-02	6.6278E-02	NO	NO	
1	-4	1.0000E+00	1.97750E-01	1.97750E-01	2.5529E-01	NO	YES	
1	2	1.0000E+00	0.00000E+00	0.00000E+00	0/256			
1	-4	1.0000E+00	1.99944E-01	1.99944E-01				
1	-SUM OF		2.40799E-01	2.40799E-01				
2 -	3	1.0000E+00	4.08547E-02	4.08547E-02	6.6278E-02	NO	NO	
2	4	1.0000E+00	1.97750E-01	1.97750E-01	2.5529E-01	NO	YES	
2	4	1.0000E+00	1.99944E-01	1.99944E-01				
2	-SUM OF		2.40799E-01	2.40799E-01				
3 -	4	1.0000E+00	1.31841E-02	1.31841E-02	2.0133E-02	YES	NO	
3	4	1.0000E+00	1.16713E-02	1.16713E-02				
3	-4	1.0000E+00	1.31841E-02	1.31841E-02	2.0133E-02	YES	NO	
3	-4	1.0000E+00	1.16713E-02	1.16713E-02				
3	-SUM OF		1.05052E-01	1.05052E-01				
4	-SUM OF		2.11616E-01	2.11616E-01				
-4	-SUM OF		2.11616E-01	2.11616E-01				

PROJECT 10, SINGLE CAVITY ENCLOSURE

Extract from MD Nastran ASCII Results File

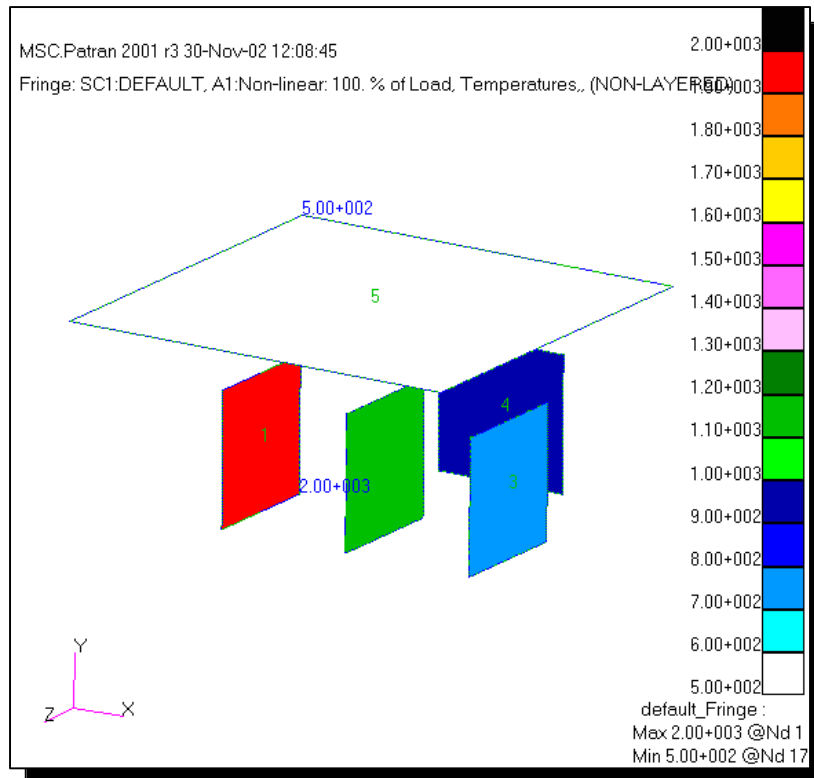
```
0
SUBCASE 1
  LOAD STEP = 1.00000E+00
      T E M P E R A T U R E   V E C T O R

      POINT ID.    TYPE      ID    VALUE      ID+1 VALUE      ID+2 VALUE      ID+3 VALUE      ID+4 VALUE      ID+5 VALUE
          1         S      2.000000E+03  2.000000E+03  2.000000E+03  2.000000E+03  1.132229E+03  1.132229E+03
          7         S      1.132229E+03  1.132229E+03  7.732046E+02  7.732046E+02  7.732046E+02  7.732046E+02
         13         S      9.168311E+02  9.168311E+02  9.168311E+02  9.168311E+02
```


DIFFERENT AMBIENT TEMPERATURE

- The previous problem is for radiation loss to space at an absolute zero degree temperature.
- If the ambient temperature was not at zero degree, but at some other temperature (e.g. 500°K), how is it modeled using MD Nastran?
- Use an ambient element.

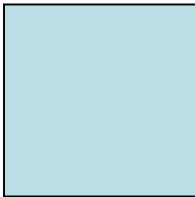
ENCLOSURES WITH AMBIENT ELEMENT



- To use ambient elements, it is necessary to add boundary condition surface elements (e.g. CHBDYG) to the model, including corresponding fictitious material properties (e.g. RADM).
- The GRIDs of the ambient elements (e.g. G1,..., G4) must be constrained at the desired ambient temperature (e.g. SPC).
- This is based on Exercise 5b in the MD Nastran 2002 Thermal Analysis User's Guide.

ADDING AMBIENT ELEMENT

- Define a boundary condition surface element (e.g. CHBDYG, EID ,, AREA4), and, if desired, a corresponding conduction finite element (e.g. CQUAD4) with corresponding properties (e.g. PSHELL and MAT4), to represent the ambient temperature.



For this problem, fix the surface at 500°K using the SPC bulk data entry.

CHBDYG and CQUAD4

- It is not necessary to define the conduction element, only the boundary condition surface element. The conduction element is used in Patran so there is a graphical representation for the ambient element.
- The size of the ambient element should be about two orders of magnitude times that of the size of the largest model element.

AMBIENT ELEMENT IN MD NASTRAN

- \$RADCAV,ICAVITY,ELEAMB
RADCAV,10,99
- \$CHBDYG,EID,,TYPE,IVIEWF,IVIEWB,RADMIDF,,,+
\$+,G1,G2,G3,G4
CHBDYG,99,,AREA4,,,1,,,+
+,17,18,19,20
- \$SPC,SID,G1,C1,D1,G2,C2,D2
SPC,1,17,,500.0,18,,500.0
SPC,1,19,,500.0,20,,500.0
- CQUAD4,5,5,17,18,19,20
PSHELL,5,15,0.1
MAT4,15,204.0

VIEW FACTORS IN THE PUNCH FILE

- A punch file is created automatically when the view factors (VFs) are calculated, and the VFs are saved in the file.
- View factor calculation can be very time consuming, thus if the VFs do not change, it may best to use the ones previously calculated. This can be done by using the VFs in the punch file.



VIEW FACTORS IN THE PUNCH FILE

Extract from MD Nastran punch file

```
$RADMTX, ICAVITY, INDEX, Fij, Fi+1j, Fi+2j, ...  
RADMTX      10      1  0.0      0.0  .040855      0.0  .199944  
RADMTX      10      2  0.0      .040855 .199944      0.0  
RADMTX      10      3  0.0      .011671 .011671  
RADMTX      10      4  0.0      0.0  
RADMTX      10      5  0.0  
$RADLST, ICAVITY, MIXTYP, EID1, EID2, ...  
RADLST      10      1      1      2      3      4      -4
```

USE THE EXISTING VIEW FACTORS

- MD Nastran checks to see if there is an existing view factor data punch file before it performs view factor calculations.
- Use the existing view factors that are in a punch file, by using the Bulk Data statement INCLUDE. Using this causes the contents of the external punch file (e.g. view factors) to be included in the Bulk Data portion of the MD Nastran input file.

- The format to be used is

BEGIN BULK

...

INCLUDE 'rad.pch'

...

EXERCISES

- Perform Workshop 14 “Radiation Enclosures” in the exercise workbook.
- Perform Workshop 23 “Pixel Hemi-cube View Factor Method” in the exercise workbook.

AXI-SYMMETRY VIEW FACTOR

- Axi-symmetric view factor example
 - MD Nastran Bulk Data input file
 - File is shield_c4a.bdf
 - Cavity radiation leakage (heat transfer) to space
 - Modify the type of radiation exchange matrix used for the cavities to eliminate cavity radiation leakage to space
- Concentric spheres axi-symmetric view factor analysis using MD Patran

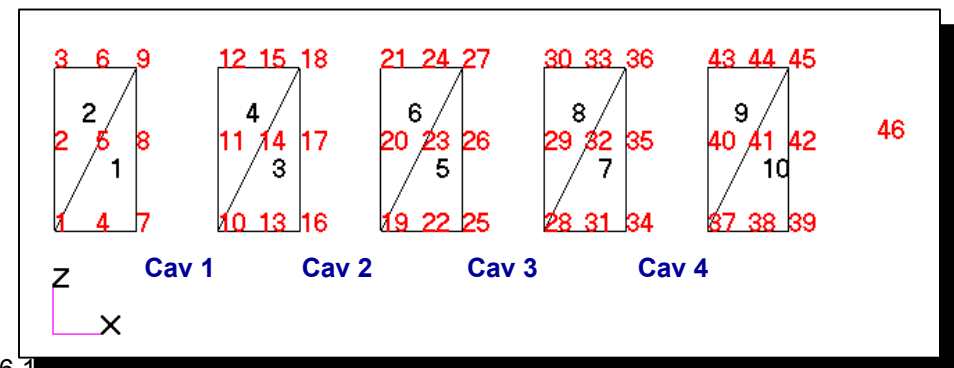
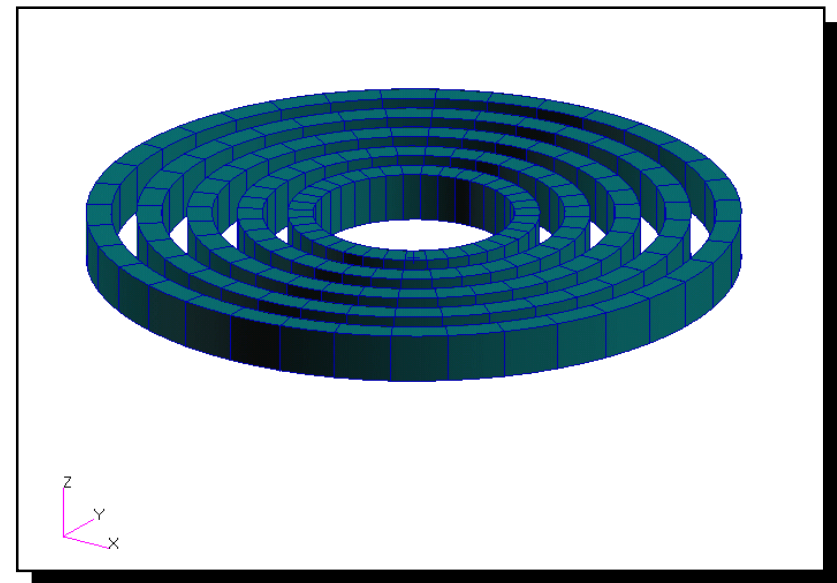


AXI-SYMMETRIC VIEW FACTOR EXAMPLE

EXAMPLE C, AXI-SYMMETRIC VIEW FACTOR CALCULATION CRYOGENIC HEAT SHIELDING

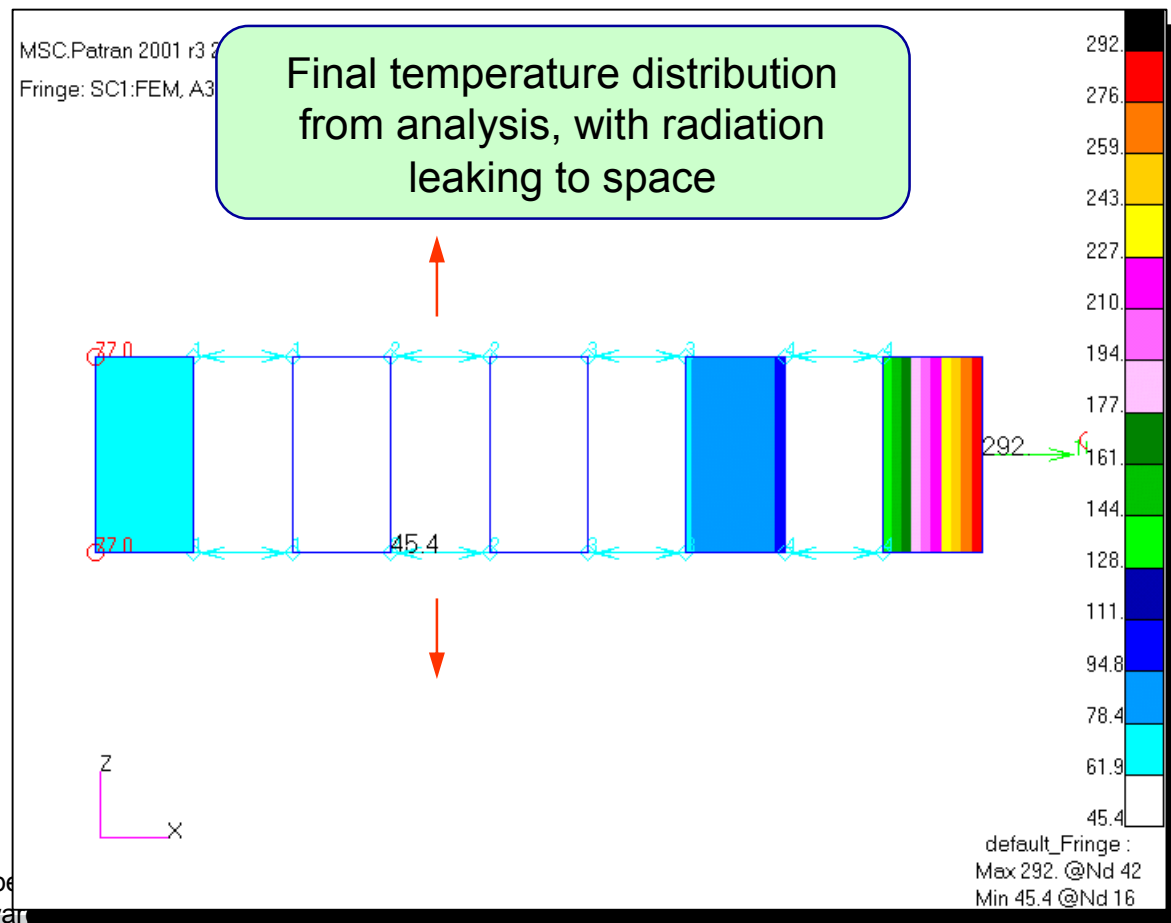
- The temperature at the left outside edge (boundary) is 77°K.
- There are four cavities, each bounded by a pair of free axi-sym element edges. The surface emissivity = 0.5
- These cavities are surrounded by space at zero degree K.
- The right outside edge convects to ambient (Node 46) at 293°K. The convection coefficient, h , is 10.0 Watts/(m²*°K).
- The thermal conductivity of the material is 0.01 Watt/(m*°K).

Cryogenic heat shielding

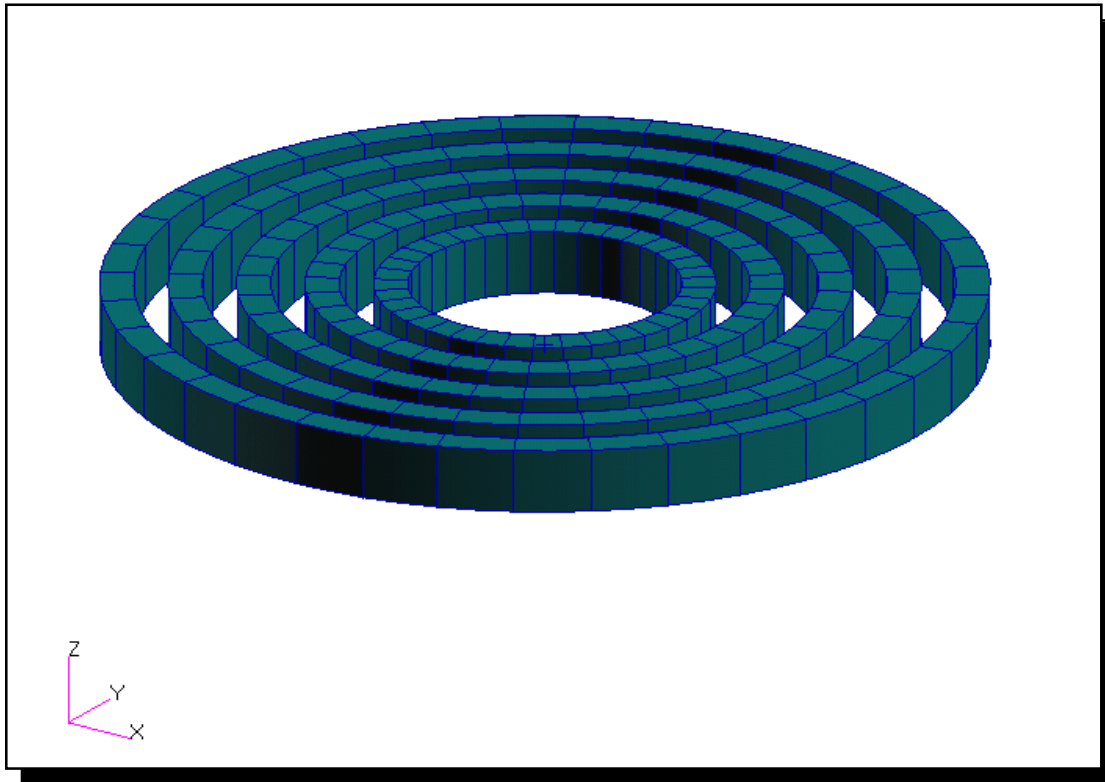


EXAMPLE C, AXI-SYMMETRIC VIEW FACTOR CALCULATION, LEAK TO SPACE

- Axi-symmetric view factor calculation with some radiation leaking (heat transfer) to space because the cavities are open.



EXAMPLE C, A FULL 3D MODEL



If one runs a full 3D model then there are many more view factors to compute. Thus, it will take more run time.

Cavities are open, so there is radiation leakage to space.

EXAMPLE C, AXI-SYMMETRIC VIEW FACTOR CALCULATION NASTRAN BULK DATA INPUT

```
$NASTRAN test deck: shield_c4a.bdf
SOL 153
TIME 600
$ Direct Text Input for Executive Control
CEND
ANALYSIS = HEAT
TITLE = MD Nastran job created on 13-Jun-02 at 17:42:33
ECHO = NONE
MAXLINES = 999999999
TEMPERATURE(INITIAL) = 1
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : fem
  SUBTITLE=fem
  NLPARM = 1
  SPC = 1
  THERMAL(SORT1,PRINT)=ALL
  FLUX(SORT1,PRINT)=ALL
  OLOAD(SORT1,PRINT)=ALL
  SPCFORCES(SORT1,PRINT)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
PARAM      POST      0
PARAM      AUTOSPC YES
PARAM*     SIGMA      5.6696-8
NLPARM     1          0          AUTO      5          25          PW          NO
          .001      1.-7
$ Direct Text Input for Bulk Data
```

EXAMPLE C, AXI-SYMMETRIC VIEW FACTOR CALCULATION NASTRAN BULK DATA INPUT

```
$ Elements and Element Properties for region : axi
CTRIAX6 1      1      9      5      1      4      7      8
CTRIAX6 2      1      1      5      9      6      3      2
CTRIAX6 3      1     18     14     10     13     16     17
CTRIAX6 4      1     10     14     18     15     12     11
CTRIAX6 5      1     27     23     19     22     25     26
CTRIAX6 6      1     19     23     27     24     21     20
CTRIAX6 7      1     36     32     28     31     34     35
CTRIAX6 8      1     28     32     36     33     30     29
CTRIAX6 9      1     45     44     43     40     37     41
CTRIAX6 10     1     37     38     39     42     45     41
$ Referenced Material Records
$ Material Record : mat4
$ Description of Material : Date: 13-Jun-02      Time: 17:37:22
MAT4      1      .01
$ The Grids are not listed in this file
```


EXAMPLE C, AXI-SYMMETRIC VIEW FACTOR CALCULATION NASTRAN BULK DATA INPUT

```

$ Loads for Load Case : fem
$ Radiation Enclosures of Load Set : c1
RADCAV 1 0.
VIEW3D 1 4 4 4 .1 1.-10 .01 3
VIEW 1 1 BOTH
$ Radiation Enclosures of Load Set : c2
RADCAV 2 0.
VIEW3D 2 4 4 4 .1 1.-10 .01 3
VIEW 2 2 BOTH
$ Radiation Enclosures of Load Set : c3
RADCAV 3 0.
VIEW3D 3 4 4 4 .1 1.-10 .01 3
VIEW 3 3 BOTH
$ Radiation Enclosures of Load Set : c4
RADCAV 4 0.
VIEW3D 4 4 4 4 .1 1.-10 .01 3
VIEW 4 4 BOTH
RADSET 1 2 3 4
$ Fixed Temperatures of Load Set : temp
SPC 1 1 1 77. 2 1 77.
SPC 1 3 1 77.
$ Fixed Temperatures of Load Set : ambient
SPC 1 46 1 293.
$ Coupled Convection of Load Set : conv
PCONV 1 1001 0 0.
CONV 100009 1 0 0 46
$ Initial Temperatures from Temperature Load Sets
TEMP 1 1 77. 2 77. 3 77.
TEMP 1 46 293.
$ Default Initial Temperature
TEMPD 1 200.
    
```

- Note that no new MD Nastran bulk data entries are added to be able to calculate the axi-symmetric view factors.
- The shading flags (SHADE) set on the VIEW entry should be set to BOTH, because with an axi-symmetric view configuration it is difficult for the user to decide which part of a surface is shaded and which is not.
- MD Nastran uses that all lost radiation goes to space at absolute zero.

EXAMPLE C, CHBDYG,REV BULK DATA ENTRY

1	2	3	4	5	6	7	8	9	10
CHBDYG	EID		TYPE	IVIEWF	IVIEWB	RADMIDF	RADMIDB		
	G1	G2	G3						

1	2	3	4	5	6	7	8	9	10
CHBDYG	100001		REV	1		1			
	7	9	8						

EXAMPLE C, SPECIFY NASTRAN BULK DATA ENTRIES TO PERFORM VIEW FACT CALCS

```
$RADCAV, ICAVITY
RADCAV 1
$RADSET, ICAVITY1, ICAVITY2, ICAVITY3, ICAVITY4
RADSET 1 2 3 4
$CHBDYG, EID, , TYPE, IVIEWF, , RADMIDF
CHBDYG 100001 REV 1 1
7 9 8
$VIEW, IVIEW, ICAVITY, SHADE
VIEW 1 1 BOTH
$VIEW3D, ICAVITY
VIEW3D 1
$RADM, RADMID, ABSORP, EMIS1
RADM 1 0.5
```

Calculate view factors for four different cavities

For each CHBDYG entry, an IVIEW (F or B) entry identification number can be used if it is desired to calculate a view factor for the corresponding edge. Also, the RADMID (F or B) refers to an RADM entry that specifies the emissivity for the edge.

The shading flag should be set to BOTH. If it is, MD Nastran will sort-out how the surfaces/edges are shaded and/or shade.

VIEW3D is tied to VIEW entry through the ICAVITY identifier. The VIEW3D is the new default view factor module since MSC/NASTRAN version 68.

EXAMPLE C, AXI-SYM VIEW FACT CALCULATION MD NASTRAN INPUT

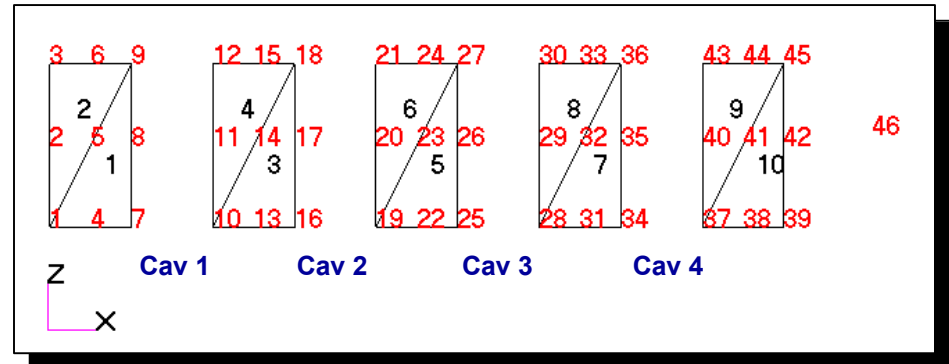
```

$ CHBDYG Surface Elements
CHBDYG 100001      REV 1      1
      7      9      8
CHBDYG 100002      REV 1      1
      12     10     11
CHBDYG 100003      REV 2      1
      16     18     17
CHBDYG 100004      REV 2      1
      21     19     20
CHBDYG 100005      REV 3      1
      25     27     26
CHBDYG 100006      REV 3      1
      30     28     29
CHBDYG 100007      REV 4      1
      34     36     35
CHBDYG 100008      REV 4      1
      43     37     40
CHBDYG 100009      REV
      39     45     42

$ Free Convection Heat Transfer Coefficients
MAT4 1001 10.

$ Radiation Material Properties
RADM 1 .5

ENDDATA 0bd76e0d
  
```



Note that the grid connectivities of an CHBDYG,EID,,REV element should be **clockwise** about the center of a cavity in the axi-symmetric view factor analysis.

For example, Cavity 1 is defined by CHBDYG 100001 and 100002. The corner grids for these two boundary condition surface elements are Grid 7, 9, and Grid 12, 10, respectively.

EXAMPLE C, AXI-SYMMETRIC VIEW FACTOR OUTPUT

*** VIEW FACTOR MODULE *** OUTPUT DATA *** CAVITY ID = 1 ***

ELEMENT TO ELEMENT VIEW FACTORS					C*	PARTIAL		
SURF-I	SURF-J	AREA-I	AI*FIJ	FIJ		ERROR	SHADING	SCALE
100001 -	100001	3.9254E+00	0.00000E+00	0.00000E+00			NO	
100001	100002	3.9254E+00	2.53224E+00	6.45097E-01			NO	
100001	-SUM OF		2.53224E+00	6.45097E-01				
100002 -	100002	4.7104E+00	2.38964E-01	5.07307E-02			NO	
100002	-SUM OF		2.77120E+00	5.88311E-01				

*** VIEW FACTOR MODULE *** OUTPUT DATA *** CAVITY ID = 2 ***

ELEMENT TO ELEMENT VIEW FACTORS					C*	PARTIAL		
SURF-I	SURF-J	AREA-I	AI*FIJ	FIJ		ERROR	SHADING	SCALE
100003 -	100003	5.4955E+00	0.00000E+00	0.00000E+00			NO	
100003	100004	5.4955E+00	3.51699E+00	6.39975E-01			NO	
100003	-SUM OF		3.51699E+00	6.39975E-01				
100004 -	100004	6.2806E+00	2.10152E-01	3.34607E-02			NO	
100004	-SUM OF		2.10152E-01	3.34607E-02				

EXAMPLE C, AXI-SYMMETRIC VIEW FACTOR OUTPUT

*** VIEW FACTOR MODULE *** OUTPUT DATA *** CAVITY ID =					3 ***			
ELEMENT TO ELEMENT VIEW FACTORS					C*	PARTIAL		
SURF-I	SURF-J	AREA-I	AI*FIJ	FIJ		ERROR	SHADING	SCALE
100005 -	100005	7.0657E+00	0.00000E+00	0.00000E+00			NO	
100005	100006	7.0657E+00	4.49774E+00	6.36563E-01			NO	
100005 -SUM OF			4.49774E+00	6.36563E-01				
100006 -	100006	7.8507E+00	1.92210E-01	2.44831E-02			NO	
100006 -SUM OF			1.92210E-01	2.44831E-02				
1 MD NASTRAN JOB CREATED ON 13-JUN-02 AT 17:42:33						MAY 19, 2003 MD		
NASTRAN 12/ 4/02 PAGE 7								
0								
*** VIEW FACTOR MODULE *** OUTPUT DATA *** CAVITY ID =					4 ***			
ELEMENT TO ELEMENT VIEW FACTORS					C*	PARTIAL		
SURF-I	SURF-J	AREA-I	AI*FIJ	FIJ		ERROR	SHADING	SCALE
100007 -	100007	8.6358E+00	0.00000E+00	0.00000E+00			NO	
100007	100008	8.6358E+00	5.47620E+00	6.34127E-01			NO	
100007 -SUM OF			5.47620E+00	6.34127E-01				
100008 -	100008	9.4209E+00	1.72790E-01	1.83412E-02			NO	
100008 -SUM OF			1.72790E-01	1.83412E-02				

EXAMPLE C, CLOSE THE FOUR CAVITIES

- If it is desired to solve the radiation problem with the four cavities closed (no leakage to space), during the first simulation create a punch file. It will have the radiation exchange factors (area*view factor) (RADLST and RADMTX entries).
- In the punch file, change from a non-conservative to a conservative radiation matrix.
- Perform a second simulation, using the modified punch file.
- Doing this is the same as adding ambient elements. (One ambient element per cavity.)

EXAMPLE C, CONSERVATIVE RAD MATRIX

- Change radiation matrix, $[R]$, so it is conservative
- So far in this exercise, radiation was lost to space at zero degree K ; this was because $[R]$ was non-conservative. The radiation punch file can be included from the first analysis, then the value of MTXTYP changed from 1 to 4 for the RADLST entry, so that the radiation matrix is conservative. This means that there will not be any radiation lost to space at zero degree K .
- \$RADLST,ICAVITY,MTXTYP,EID1,EID2,EID3,etc.
- MTXTYP type
 - 1: Symmetric view factor matrix $[F]$, and non-conservative radiation matrix $[R]$
 - 4: Symmetric view factor matrix $[F]$, and conservative radiation matrix $[R]$

EXAMPLE C, CHANGING THE PUNCH FILE CONTENTS

The original punch file contains:

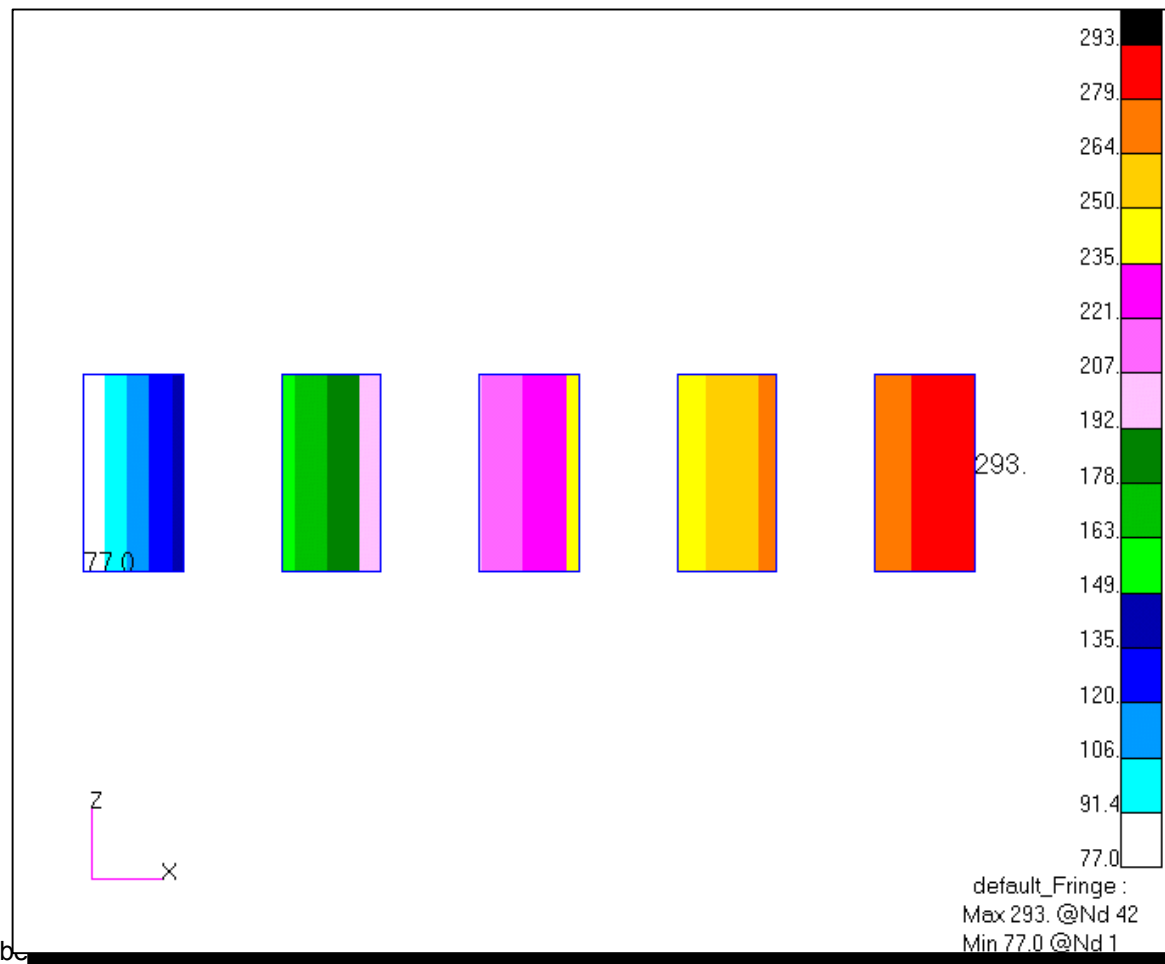
RADMTX	1	1	0.0	2.53224
RADMTX	1	2	.238964	
RADLST	1	1	100001	100002
RADMTX	2	1	0.0	3.51699
RADMTX	2	2	.210152	
RADLST	2	1	100003	100004
RADMTX	3	1	0.0	4.49774
RADMTX	3	2	.19221	
RADLST	3	1	100005	100006
RADMTX	4	1	0.0	5.4762
RADMTX	4	2	.17279	
RADLST	4	1	100007	100008

Change Field 3 of RADLST from 1 to 4:

RADMTX	1	1	0.0	2.53224
RADMTX	1	2	.238964	
RADLST	1	4	100001	100002
RADMTX	2	1	0.0	3.51699
RADMTX	2	2	.210152	
RADLST	2	4	100003	100004
RADMTX	3	1	0.0	4.49774
RADMTX	3	2	.19221	
RADLST	3	4	100005	100006
RADMTX	4	1	0.0	5.4762
RADMTX	4	2	.17279	
RADLST	4	4	100007	100008

EXAMPLE C, CONSERVATIVE RAD MATRIX RESULTS

- Temperature results for the conservative radiation matrix.



EXAMPLE C, ENERGY BALANCE FOR THE RADIATION SURFACES

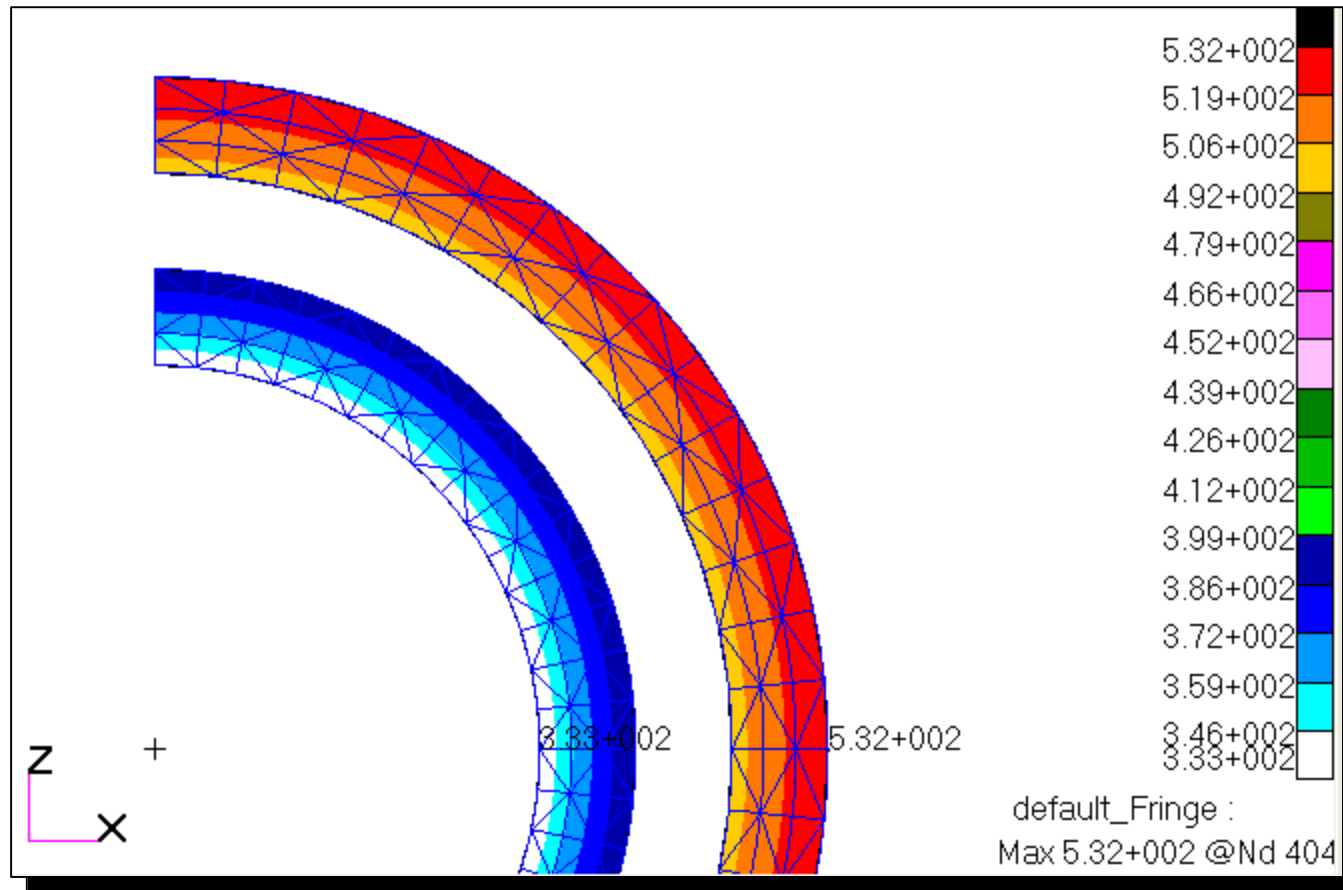
- For each cavity, e.g. Element ID 100003 and 100004, it can be seen that the heat flow out of one face is equal to the negative of that for the opposing face. There is no radiation leakage to space.

0							
SUBCASE 1							
LOAD STEP = 1.00000E+00							
FORCES OF SINGLE-POINT CONSTRAINT							
POINT ID.	TYPE	ID	VALUE	ID+1	VALUE	ID+2	VALUE
1	S	-1.494710E+00	-5.958359E+00	-1.490657E+00	0.0	0.0	0.0
43	S	0.0	0.0	0.0	8.943727E+00		
LOAD STEP = 1.00000E+00							
HEAT FLOW INTO HBDY ELEMENTS (CHBDY)							
ELEMENT-ID	APPLIED-LOAD	FREE-CONVECTION	FORCED-CONVECTION	RADIATION	TOTAL		
100001	0.000000E+00	0.000000E+00	0.000000E+00	8.943602E+00	8.943602E+00		
100002	0.000000E+00	0.000000E+00	0.000000E+00	-8.943602E+00	-8.943602E+00		
100003	0.000000E+00	0.000000E+00	0.000000E+00	8.943707E+00	8.943707E+00		
100004	0.000000E+00	0.000000E+00	0.000000E+00	-8.943707E+00	-8.943707E+00		
100005	0.000000E+00	0.000000E+00	0.000000E+00	8.944277E+00	8.944277E+00		
100006	0.000000E+00	0.000000E+00	0.000000E+00	-8.944277E+00	-8.944277E+00		
100007	0.000000E+00	0.000000E+00	0.000000E+00	8.943882E+00	8.943882E+00		
100008	0.000000E+00	0.000000E+00	0.000000E+00	-8.943882E+00	-8.943882E+00		
100009	0.000000E+00	8.942629E+00	0.000000E+00	0.000000E+00	8.942629E+00		

EXAMPLE C, AUTO USING CONSERVATIVE

- For the current version of MD Nastran, including
 - nastran system (87) = 3as the first line/record (nastran command section) in the input .bdf file, will cause the thermal analysis to be performed without any radiation lost to space. This will eliminate the need to perform a second simulation (using an edited punch file).

PROJECT 11, CONCENTRIC SPHERES FOR AXI-SYMMETRIC VIEW FACTORS



NAS104, Section 6, September 2008

Copyright© 2008 MSC Software Corporation

S6-135

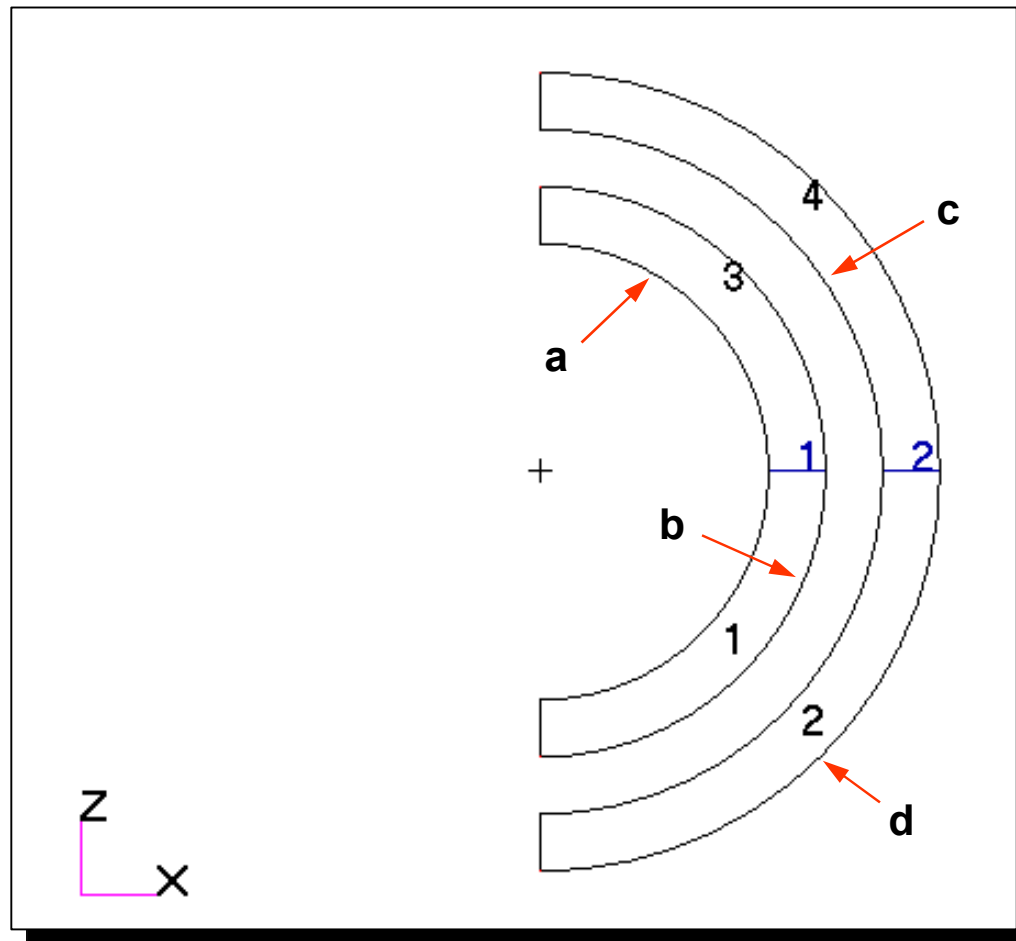
PROJECT 11, RAD BETWEEN SPHERES

- Radiation Between Concentric Spheres

- A typical radiation heat-exchange problem between gray bodies is solved here in order to show the MD Nastran axi-symmetric view factor capability.
- In MD Nastran, the radiation heat exchange is based on the computation of the viewfactor depending solely upon the geometric shape of the radiating boundaries. The geometry of the model is shown in the figure on the next page. There are two concentric spherical bodies and eight surface edges that represent four spherical surfaces. Surfaces 1 and 3 define the inner (first) body, and Surfaces 2 and 4 define the outer (second) spherical gray body. Inner surface (a) has a radius of r_a , next surface (b) radius of r_b , next surface (c) radius of r_c , and outer surface (d) radius of r_d . Temperatures on inner (a) and outer (b) surfaces are known. Radiative heat transfer takes place between the two middle surfaces (b and c).
- $r_a = 8$, $r_b = 10$, $r_c = 12$, and $r_d = 14$

PROJECT 11, TWO CONCENTRIC SPHERER BODIES

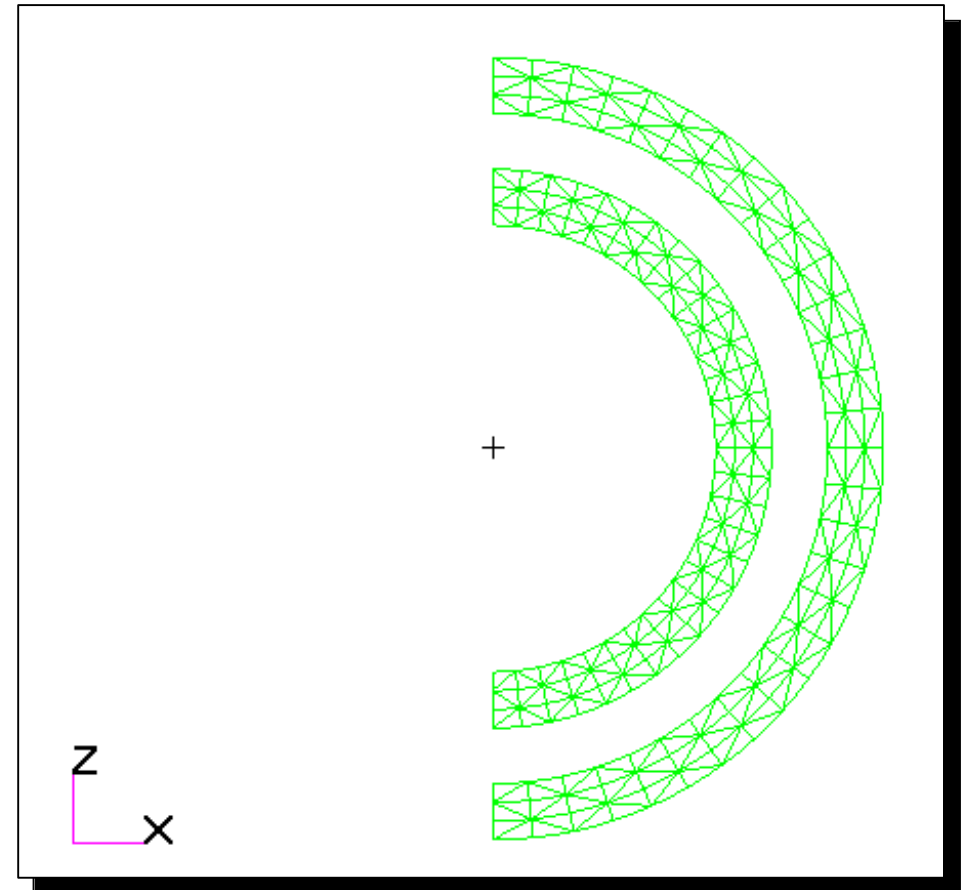
- Four surfaces that represent the two concentric spherical bodies



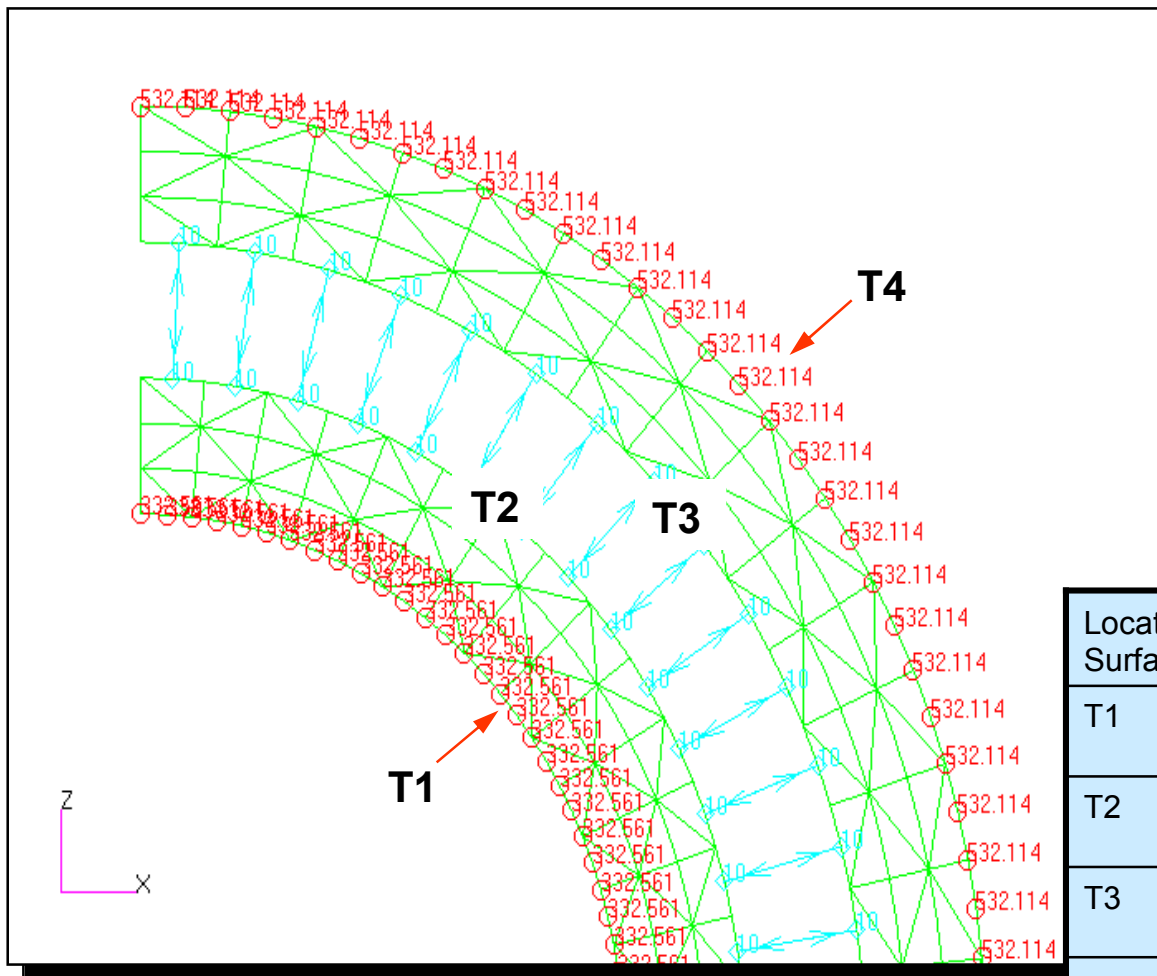
PROJECT 11, CONCENTRIC SPHERES

- This model is made of 360 CTRIAX6 elements.
- The thermal conductivity is equal to $1.0\text{E-}4 \text{ Watt}/(\text{mm}^* \text{ } ^\circ\text{C})$
- Emissivity value is

Temp(C)	Emissivity
300	0.3
500	0.5



PROJECT 11, CONCENTRIC SPHERES

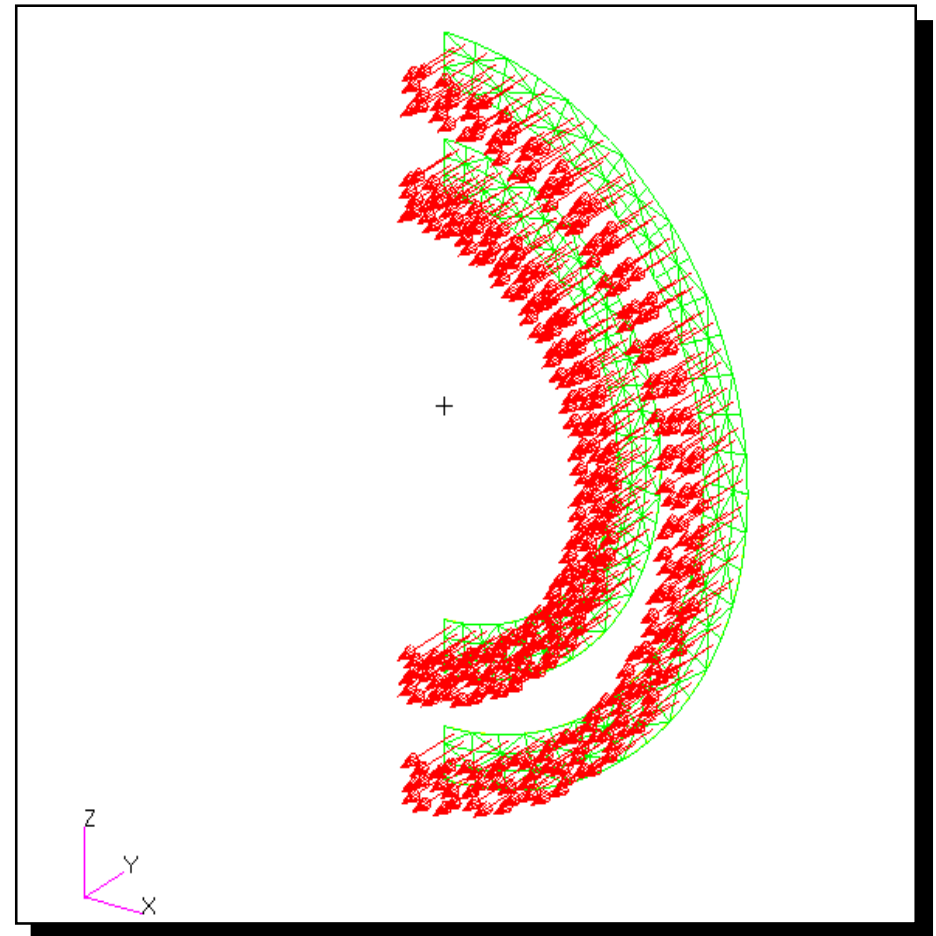


- Inner edge is held at 332.561 °C, and the outer edge is at 532.114 °C. The view factor calculation for the region between the inner edge and outer edge, is indicated by the blue arrows.

Location on Surface	Analytical Temperature (°C)	MD Nastran Temperature (°C)
T1	332.561	332.561
T2	400.00	399.18
T3	500.00	499.49
T4	532.114	532.114

PROJECT 11, AXI-SYMMETRIC ORIENTATIONS

- If using MD Patran to create the surfaces, make sure that the model is created in the X-Z plane. Also, it is required to have the surface normal vectors point out of the screen (oriented in the negative Y-direction).
- Use Geometry: Edit/Surface/Reverse to display the surface normals, and if necessary, reverse their direction.



PROJECT 11, CREATE POINTS FOR CURVES

- Create four points, to be used to create two straight curves
- The coordinates for them are
 - [8 0 0], [10 0 0], [12 0 0], and [14 0 0]

Geometry

Action:

Object:

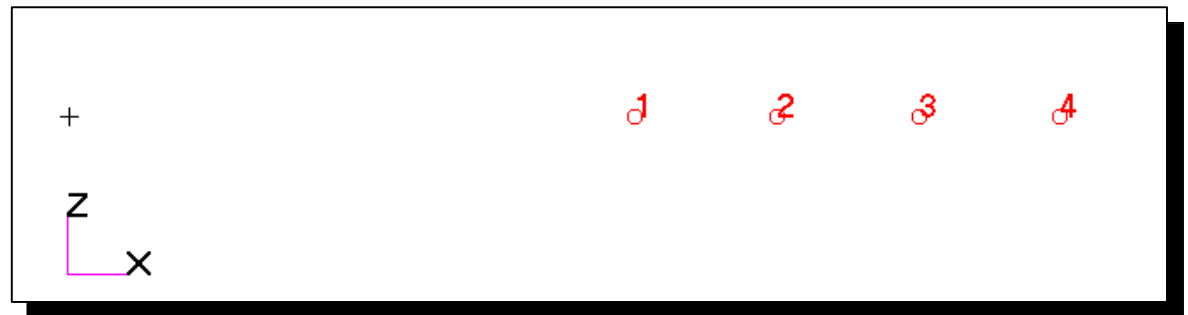
Method:

Point ID List

Refer. Coordinate Frame

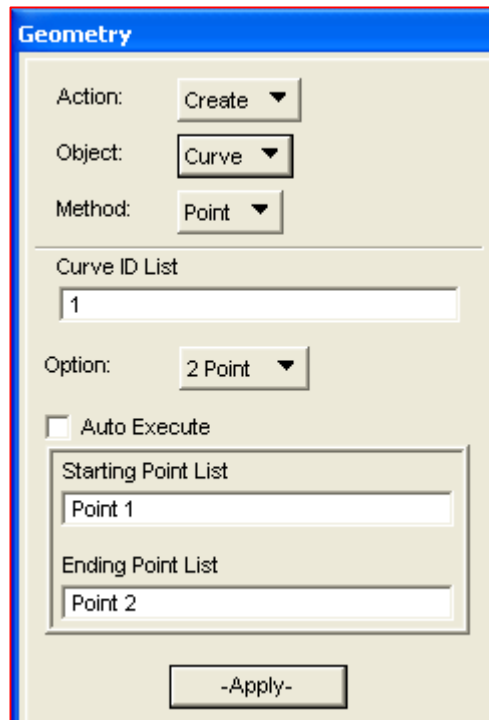
☒ Auto Execute

Point Coordinates List

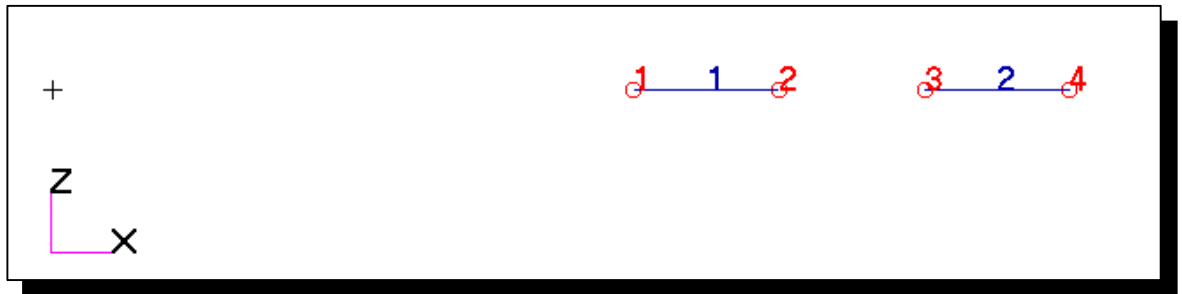


PROJECT 11, CREATE CURVES

- Create two straight curves using the four points
- Create Curve 1 using Points 1, 2
- Create Curve 2 using Points 3, 4

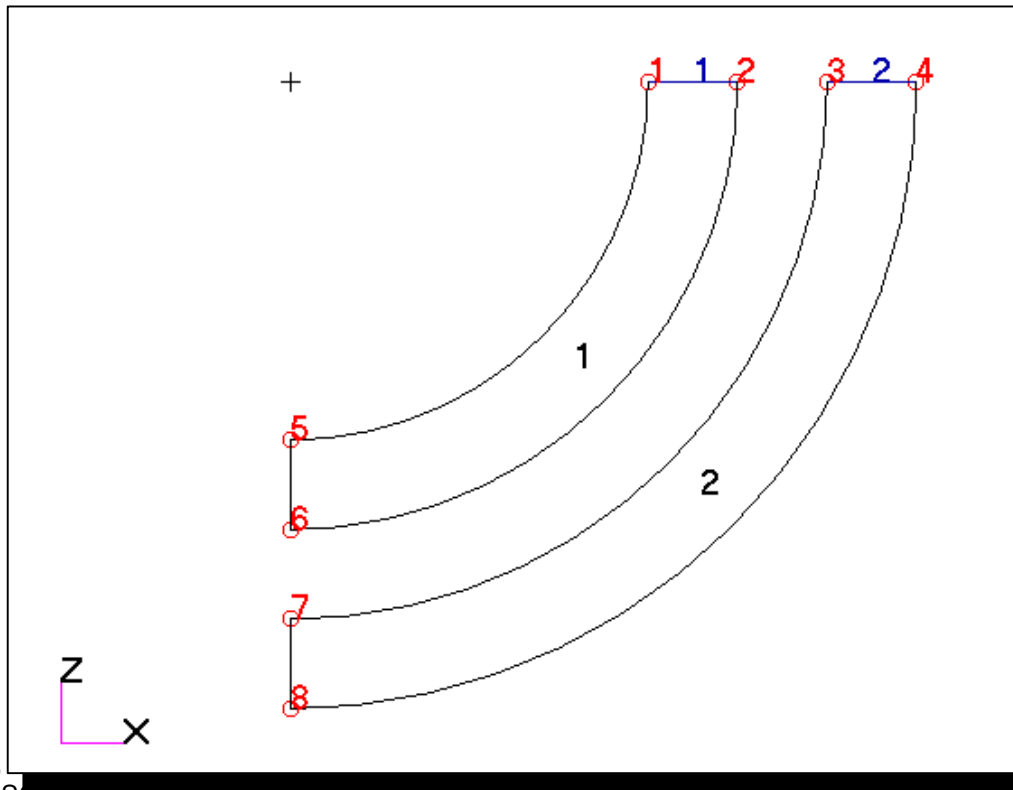


The screenshot shows the 'Geometry' software interface. The 'Action' dropdown is set to 'Create', 'Object' is 'Curve', and 'Method' is 'Point'. The 'Curve ID List' contains the number '1'. The 'Option' dropdown is set to '2 Point'. There is an unchecked checkbox for 'Auto Execute'. Below this, the 'Starting Point List' contains 'Point 1' and the 'Ending Point List' contains 'Point 2'. An '-Apply-' button is at the bottom.



PROJECT 11, CREATE TWO SURFACES

- Create two surfaces by rotating the two straight curves
- Create Surface 1 by rotating Curve 1
- Create Surface 2 by rotating Curve 2



Geometry

Action:

Object:

Method:

Surface ID List

Refer. Coordinate Frame

Axis

Sweep Parameters
Total Angle

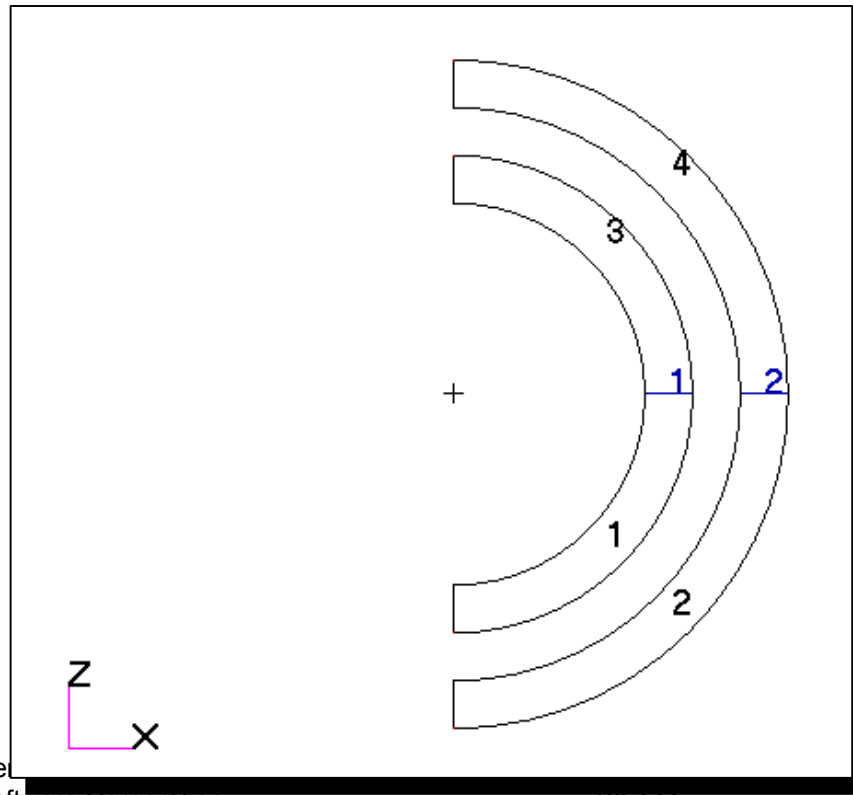
Offset Angle

☐ Auto Execute

Curve List

PROJECT 11, CREATE TWO SURFACES

- Create two surfaces by rotating the two straight curves
- Create Surface 3 by rotating Curve 1
- Create Surface 4 by rotating Curve 2



Geometry

Action:

Object:

Method:

Surface ID List

Refer. Coordinate Frame

Axis

Sweep Parameters
Total Angle

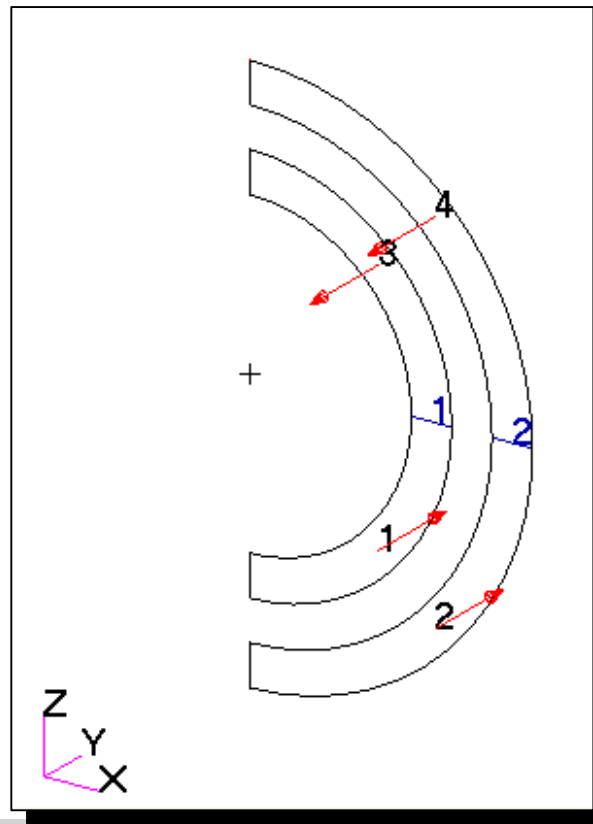
Offset Angle

☐ Auto Execute

Curve List

PROJECT 11, SHOW SURFACE NORMALS

- Display surface normals. Subsequently, reverse the element normals for the elements obtained by IsoMesh the surfaces.



Geometry

Action:

Object:

Info:

Surface Summary

Last ID: 4

Total in Model: 4

Total in 'default_group': 4

Global Model Tolerance: 0.0049999999

Normal Vector Display

☒ Surface Interior

☐ Surface Boundary

Normal Vector Density

1 21

☒ Set Normal Vector Length

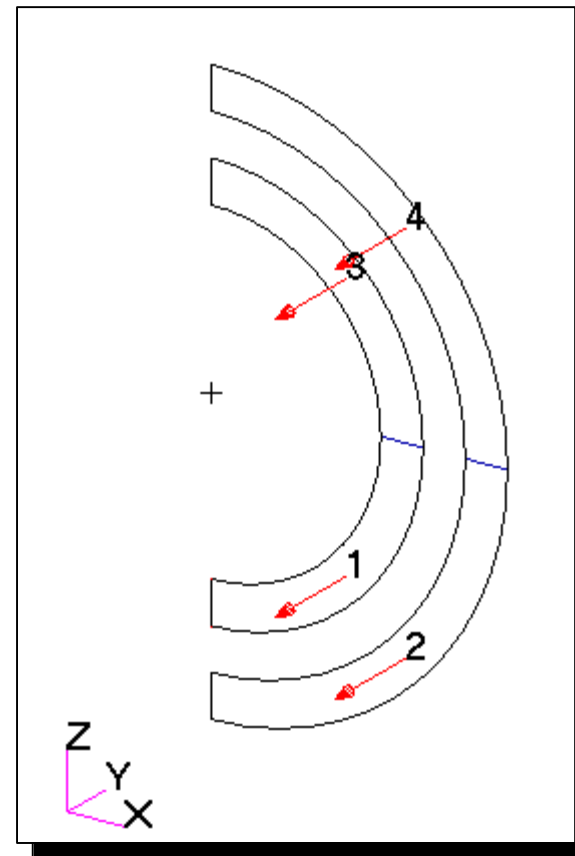
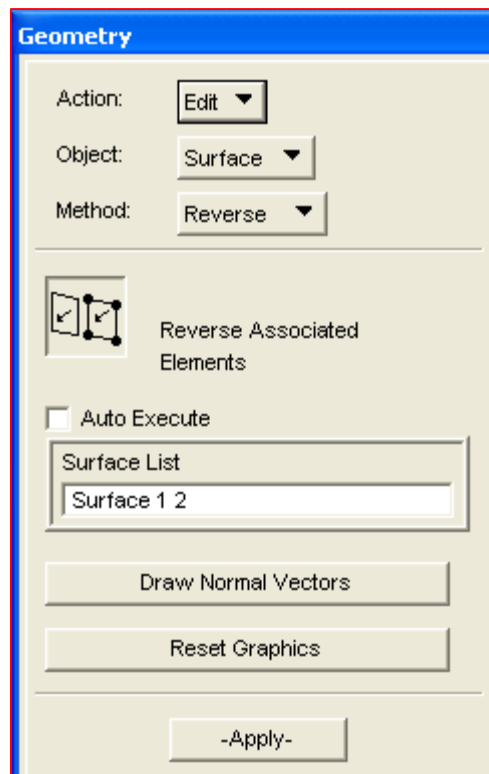
Normal Vector Length:

☐ Auto Execute

Surface List

PROJECT 11, REVERSE SURFACE NORMALS

- Orient all surface normals so they point in the $-Y$ -direction



PROJECT 11, CREATE MESH SEED

- Create mesh seeds uniformly for subsequent IsoMeshing

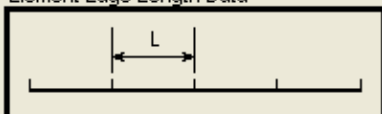
Finite Elements

Action:

Object:

Type:

Element Edge Length Data



☒ Number of Elements
☐ Element Length (L)

Number =

☐ Auto Execute

Curve List

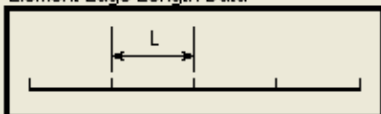
Finite Elements

Action:

Object:

Type:

Element Edge Length Data

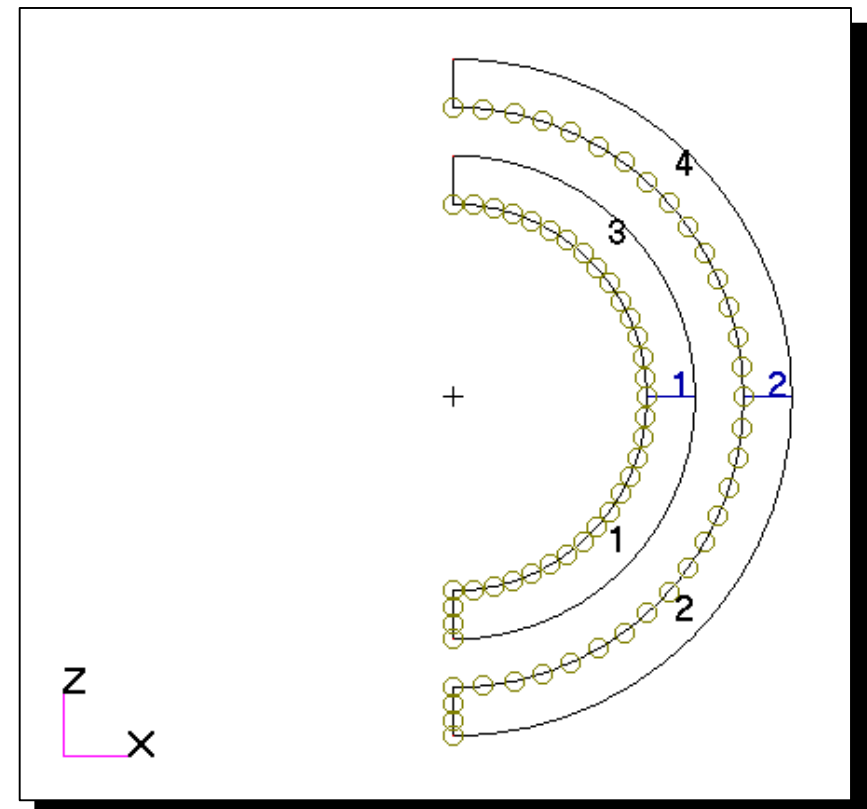


☒ Number of Elements
☐ Element Length (L)

Number =

☐ Auto Execute

Curve List



PROJECT 11, CREATE CTRIAX6 ELEMENTS

- IsoMesh
- TRIAX6
- All surfaces
- Use default value for GEL

Finite Elements

Action:

Object:

Type:

Output ID List

Node

Element

Elem Shape

Mesher

Topology

Surface List

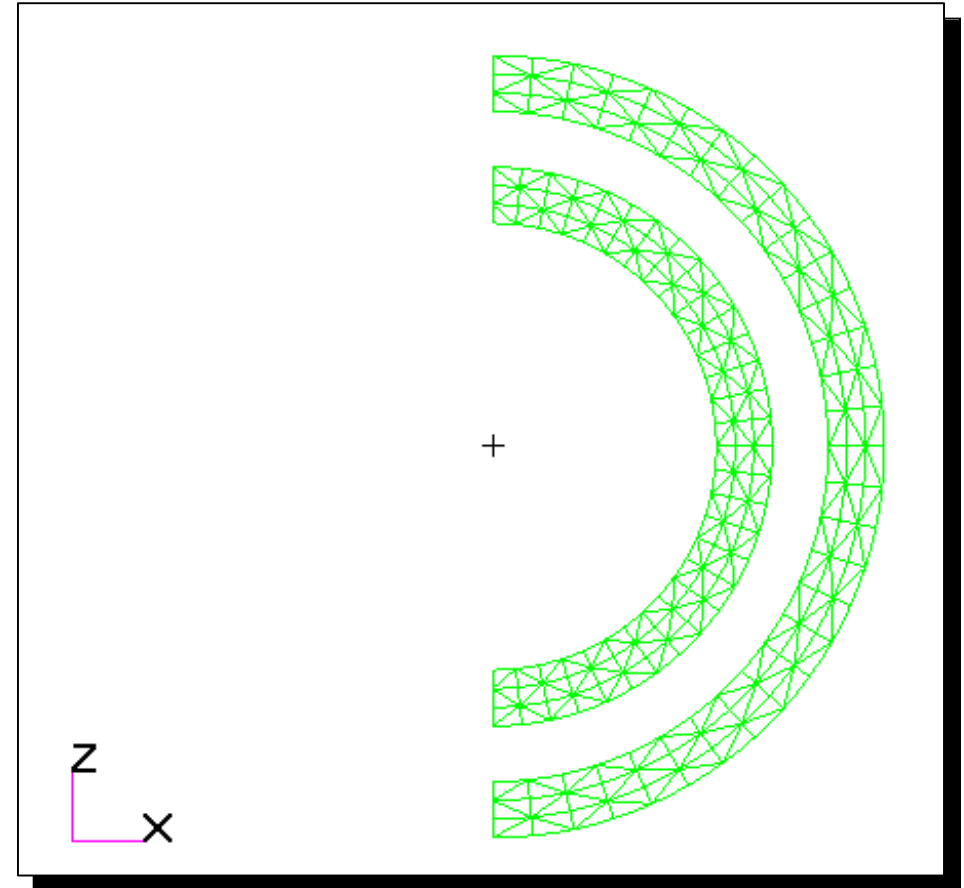
Global Edge Length

☒ Automatic Calculation

Value

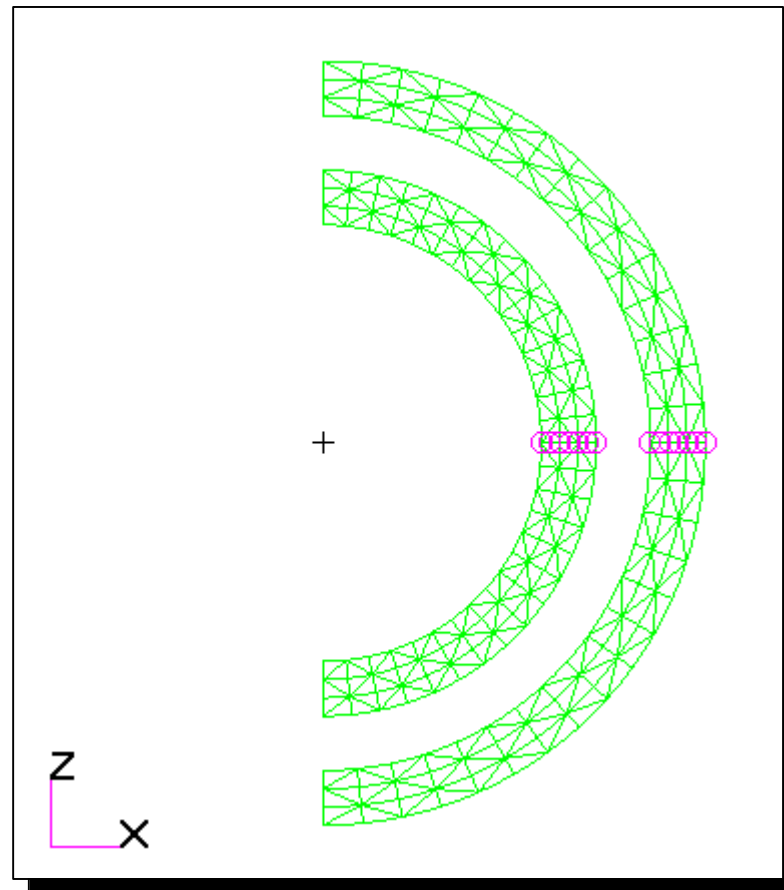
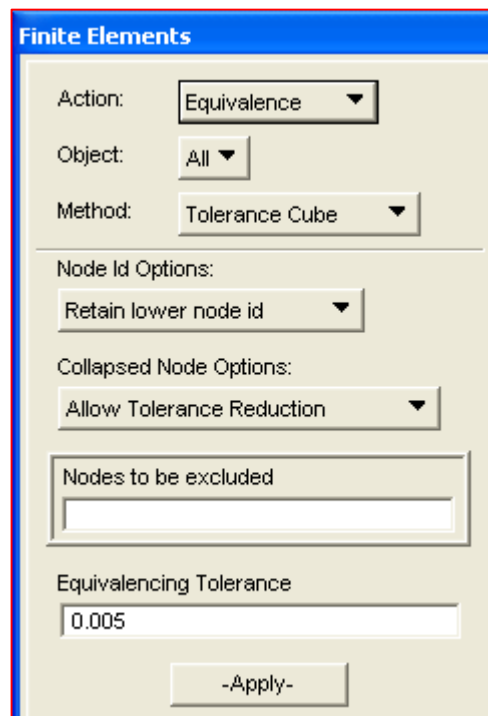
Prop. Name: - None -

Prop. Type: - N/A -



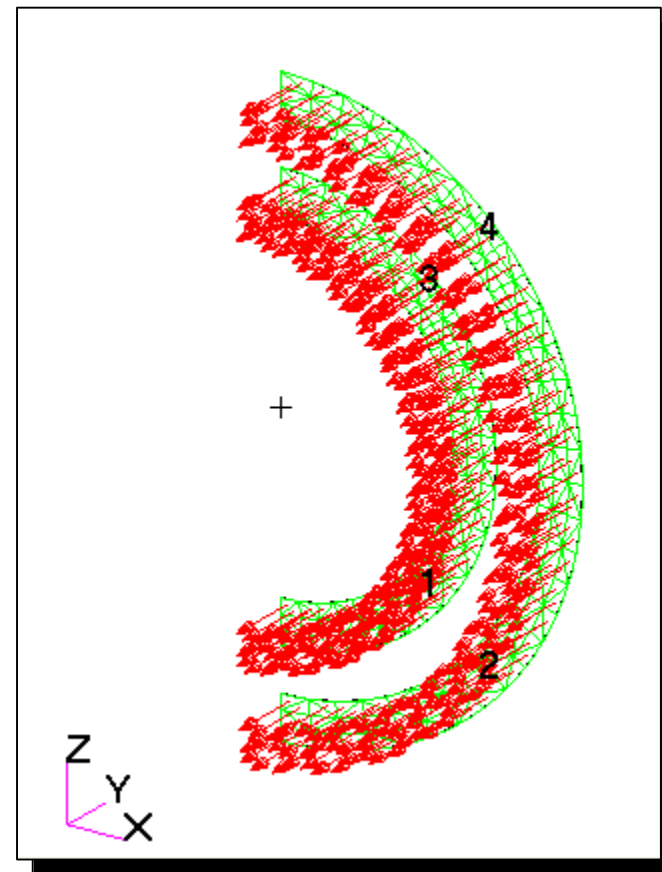
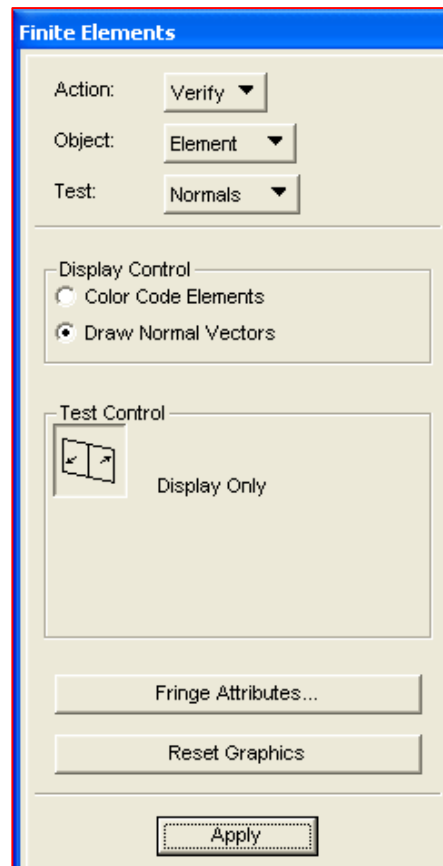
PROJECT 11, EQUIVALENCE THE NODES

- Connect the elements at the geometric interfaces



PROJECT 11, CHECK THE ELEMENT NORMALS

- The element normals point in the $-Y$ -direction, as they should



PROJECT 11, MATERIAL PROPERTY

Materials

Action:

Object:

Method:

Existing Materials

Filter

Material Name

Description

Date: 23-Jul-08 Time: 12:42:41

Input Options

Constitutive Model:

Property Name	Value
Thermal Conductivity =	1e-4
Specific Heat =	
Density =	

Temperature Dep/Model Variable Fields:

Current Constitutive Models:

PROJECT 11, ELEMENT PROPERTY

- Create 2D axi-symmetric solid element properties

The screenshot shows the **Element Properties** dialog box in MSC Nastran. The **Action** is set to **Create**, the **Object** is **2D**, and the **Type** is **Axisym Solid**. The **Sets By** dropdown is set to **Name**. The **Property Set Name** is **axi_view**. The **Input Properties** sub-dialog is open, showing a table for **Axisym Solid (CTRIA6)**.

Property Name	Value	Value Type
[Material Orientation]		CID
Material Name	m:mat4	Mat Prop Name

The **Select Application Region** sub-dialog is also open, showing **Entities** selected. The **Application Region** is **Surface 1:4**. The **OK** button is visible at the bottom of the sub-dialog.

Buttons at the bottom of the main dialog include **Apply**, **Close**, **Input Properties ...**, and **Select Application Region ...**.

S6-152

PROJECT 11, INNER EDGE TEMPERATURE BC

Load/Boundary Conditions

Action:

Object:

Type:

Current Load Case:

Type:

Existing Sets

New Set Name

Input Data

Boundary Temperature

Spatial Fields

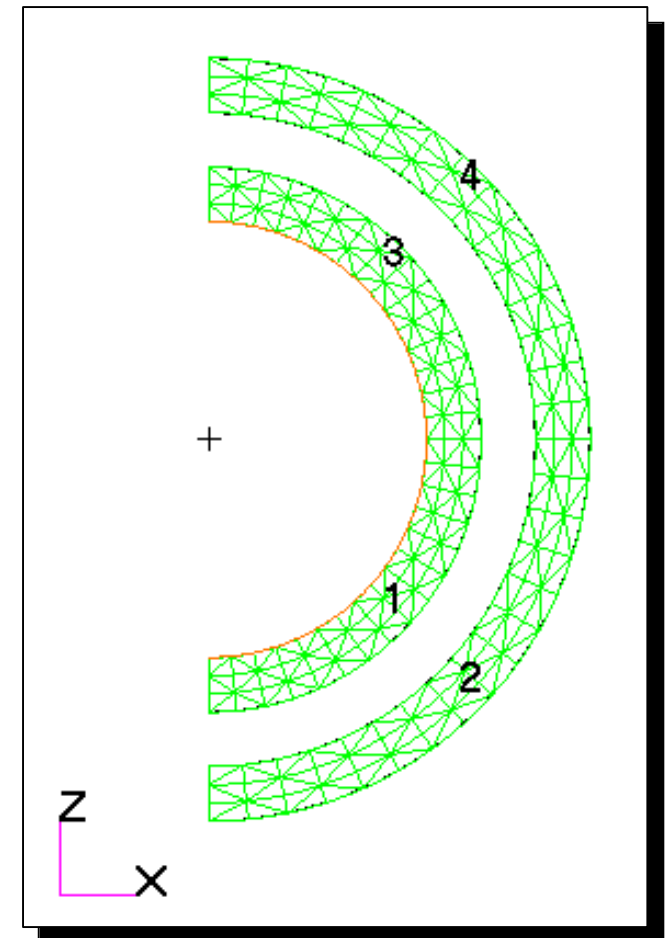
Select Application Region

Select:

Application Region

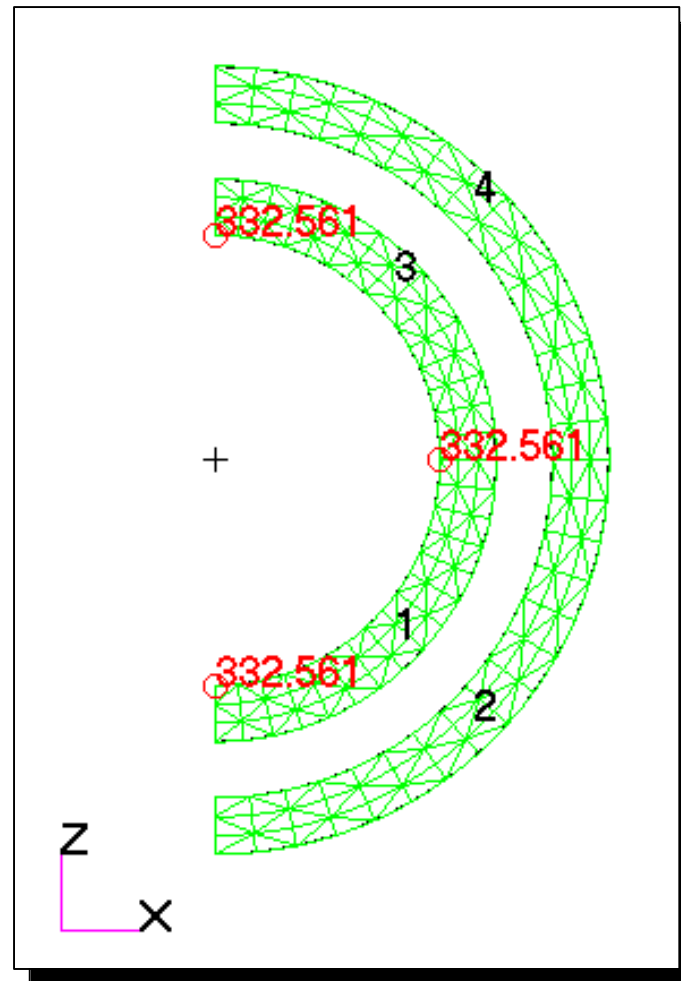
Select Geometry Entities

Application Region



PROJECT 11, INNER EDGE TEMPERATURE BC

- A temperature of 332.561 °C is assigned to the two inner edges



PROJECT 11, OUTER EDGE TEMP BC

Load/Boundary Conditions

Action:

Object:

Type:

Current Load Case:

Type: Static

Existing Sets

inside

New Set Name

outside

Input Data

Boundary Temperature

532.114

Spatial Fields

Select Application Region

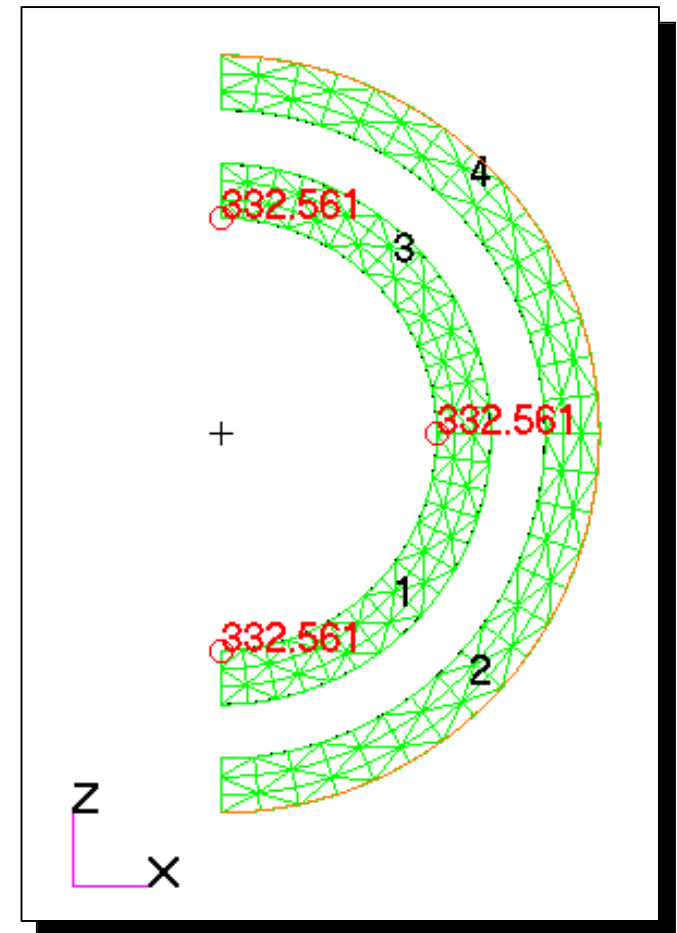
Select:

Application Region

Select Geometry Entities

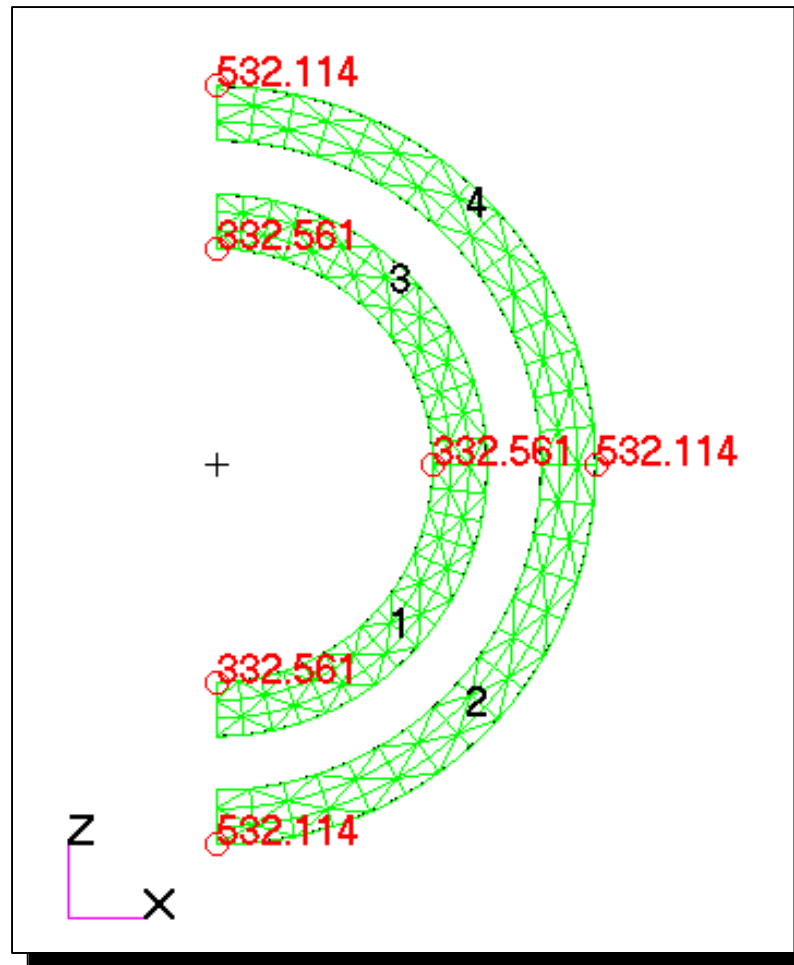
Application Region

Surface 2.2 4.3



PROJECT 11, ALL TEMPERATURE BCs

- The temperature BCs for the inner and outer edges



PROJECT 11, CREATE FIELD FOR EMISSIVITY

Fields

Action: **Create**

Object: **Material Property**

Method: **Tabular Input**

Existing Fields

Field Name
emiss_temp

Table Definition
Active Independent Variables

- ☒ Temperature (T)
- ☐ Strain (e)
- ☐ Strain Rate (er)
- ☐ Time (t)
- ☐ Frequency (f)
- ☐ Magnetic Induction (B)
- ☐ Magnetic Field Intensity (H)

Input Data ...

[Options...]

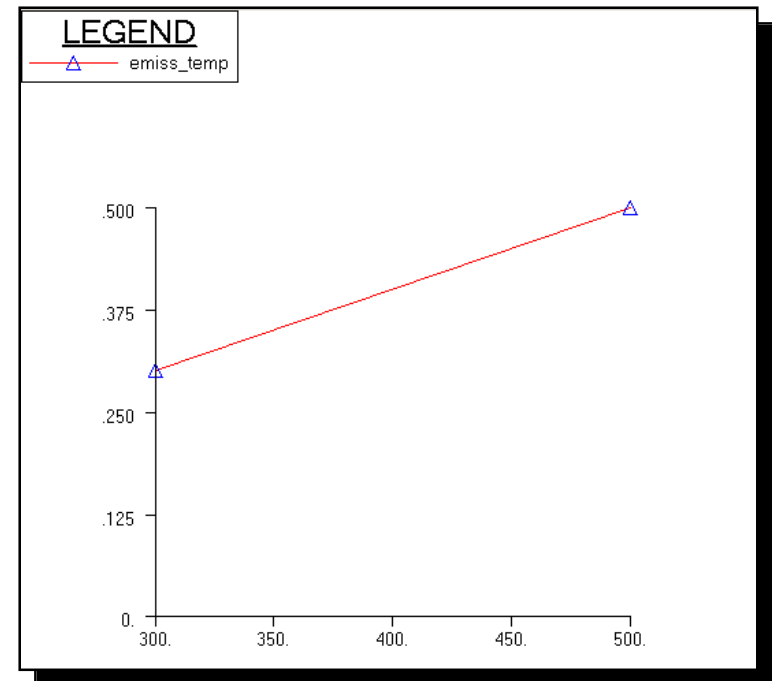
1D Material Scalar Table Data

Input Data ☐ Auto Highlight **Import/Export...**

Data

	T	Data
T-1	3.0000000E+002	3.0000001E-001
T-2	5.0000000E+002	5.0000000E-001
T-3		
T-4		
T-5		
T-6		
T-7		
T-8		
T-9		

OK Undo



PROJECT 11, RADIATION VIEW FACTORS

Load/Boundary Conditions

Action:

Object:

Type:

Option:

Current Load Case:

Type:

Existing Sets

New Set Name

Target Element Type:

Input Data

Surface Option:

Enclosure ID:

Edge Emissivity:

* Temperature Function:

☒ Surface Can Shade

☒ Third Body Shading

☒ Surface Can Be Shaded

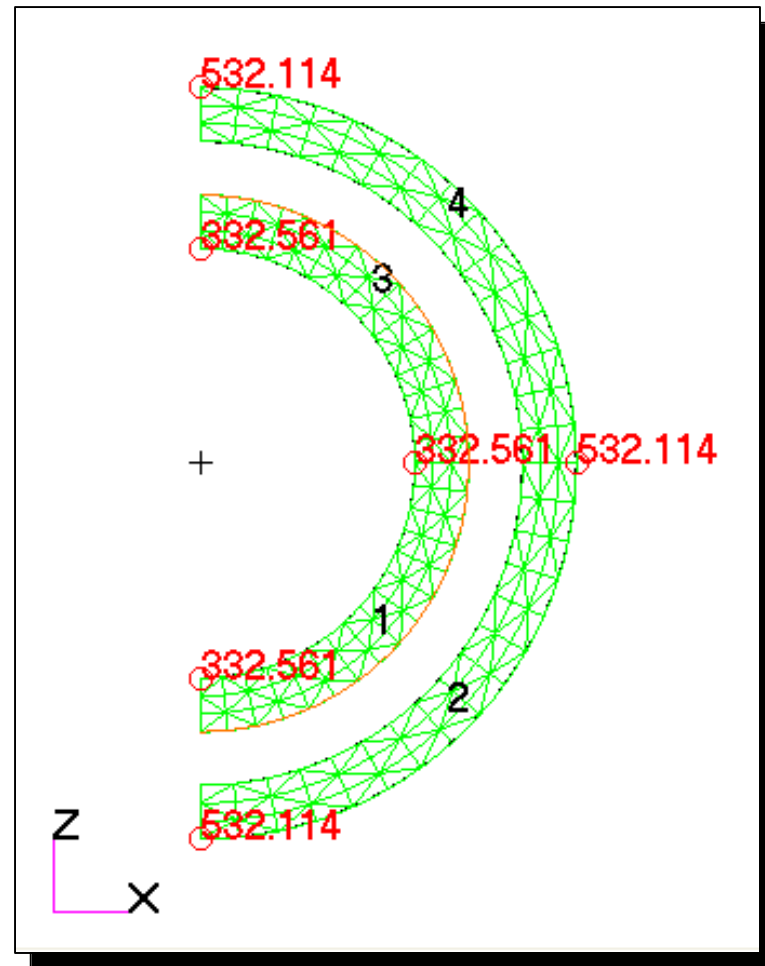
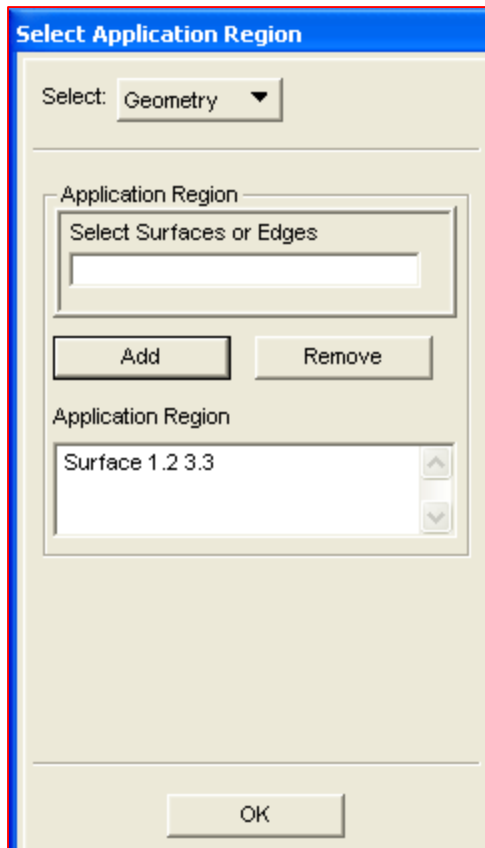
☐ Complete Enclosure

Spatial Fields:

Temperature Dependent Fields:

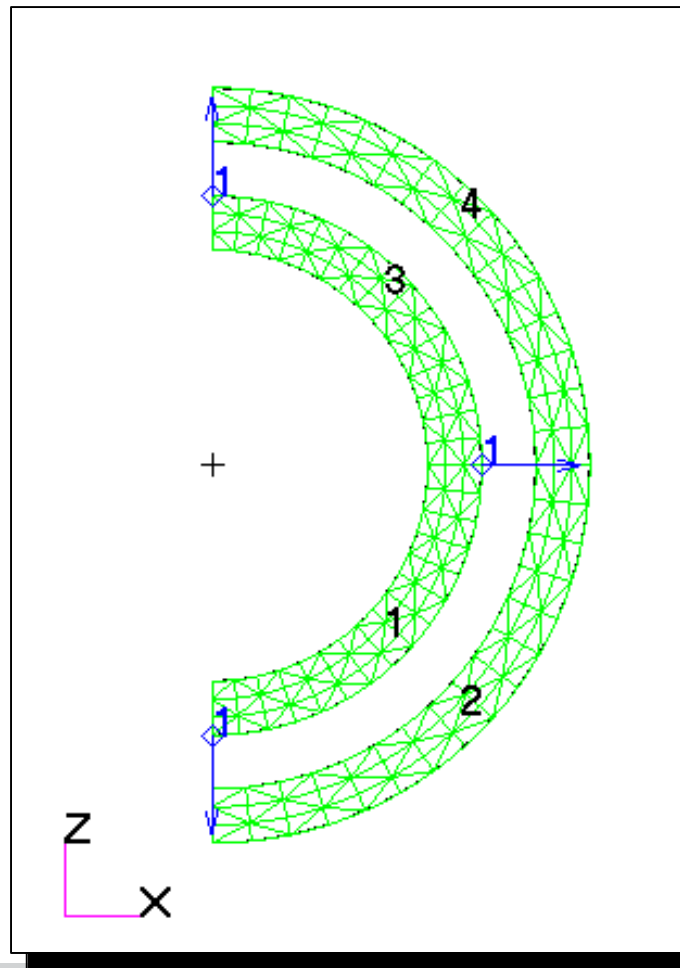
For axi-symmetric problems always check both flags to let the MD Nastran program sort-out the shadings.

PROJECT 11, RADIATION VIEW FACTORS



PROJECT 11, INNER VIEW FACTOR MARKERS

- Note that the view orientations markers point from inside to outside.



PROJECT 11, RADIATION VIEW FACTORS

Load/Boundary Conditions

Action:

Object:

Type:

Option:

Current Load Case:

Type:

Existing Sets 

view_inside

New Set Name

Target Element Type:

Input Data   

Surface Option:

Enclosure ID

Edge Emissivity

* Temperature Function

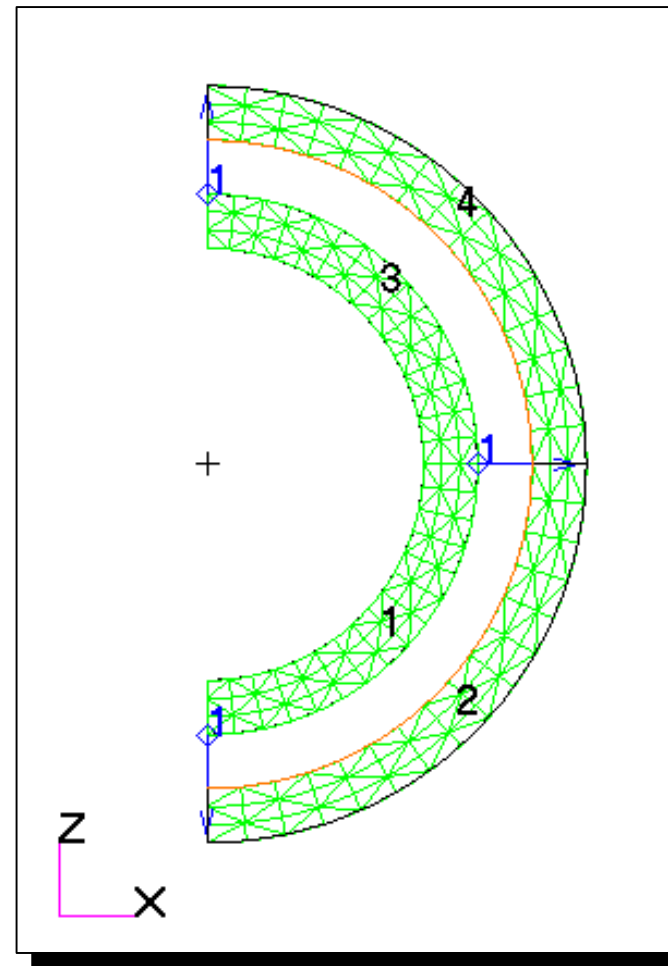
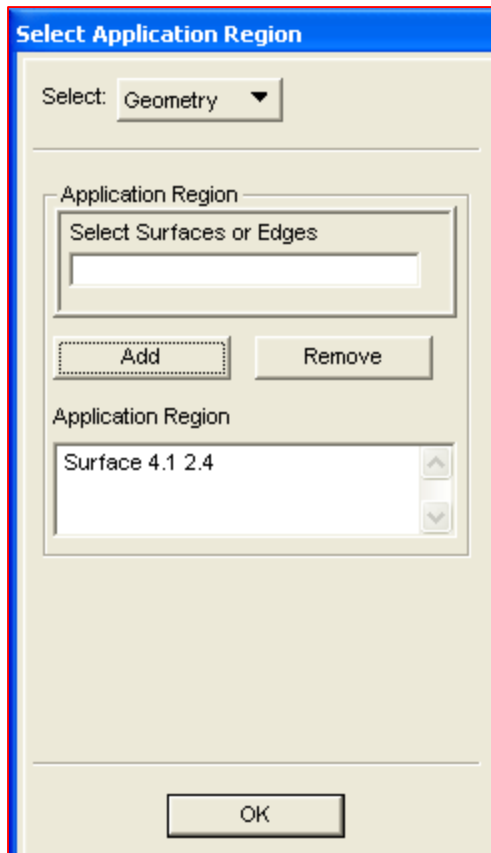
☒ Surface Can Shade ☒ Third Body Shading

☒ Surface Can Be Shaded ☐ Complete Enclosure

Spatial Fields

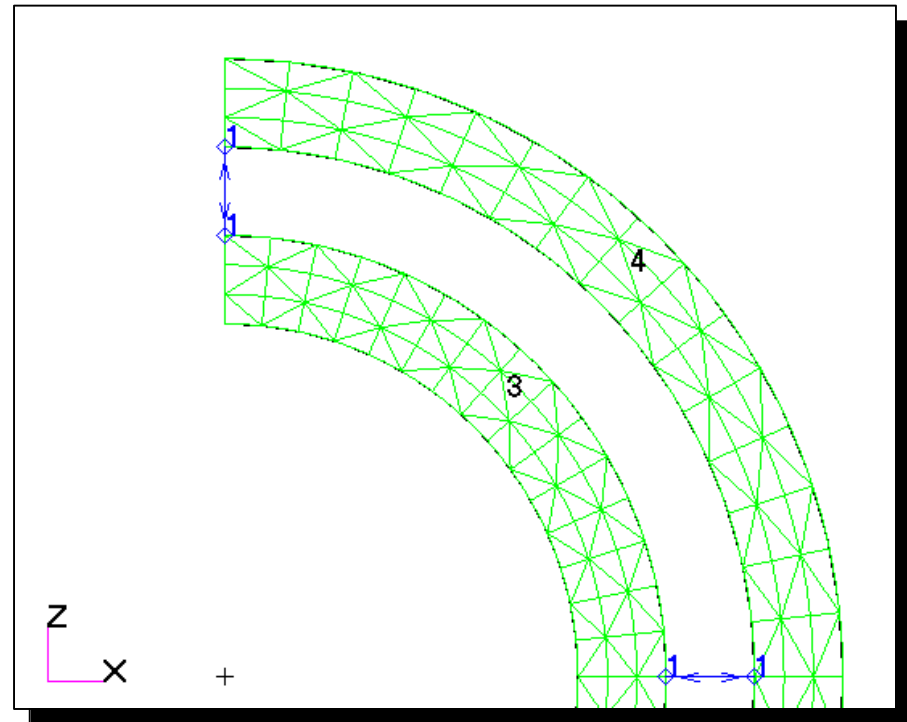
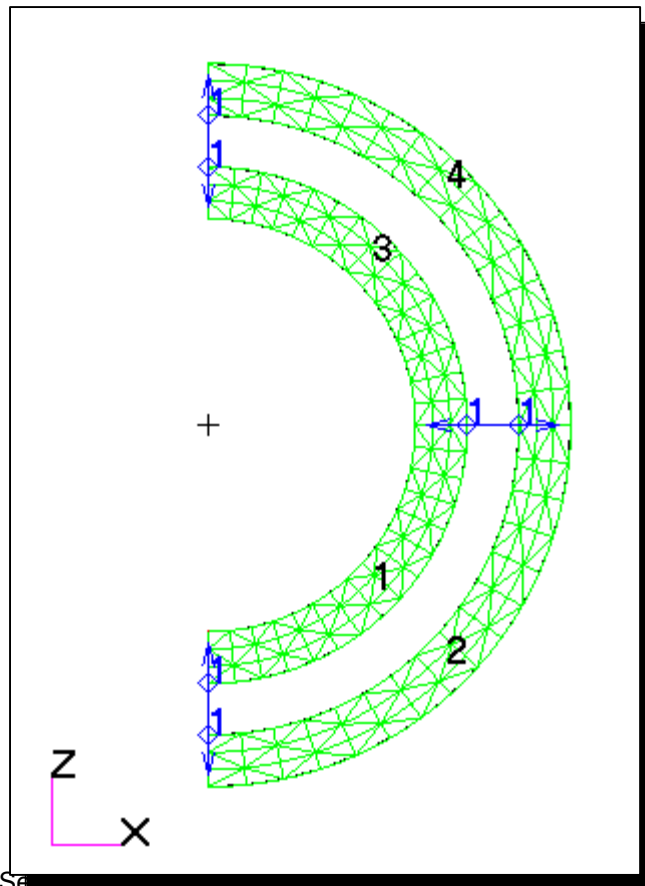
Temperature Dependent Fields

PROJECT 11, SELECT ELEMENTS



PROJECT 11, OUTER VIEW FACTOR MARKER

- Note that the outer view orientations markers point from outside to inside.



PROJECT 11, $X \geq 0$?

Finite Elements

Action:

Object:

Info:

Node Summary

Last ID: 868

Total in Model: 854

Total Unreferenced: 0

Total in 'default_group': 854

☐ Write to Report

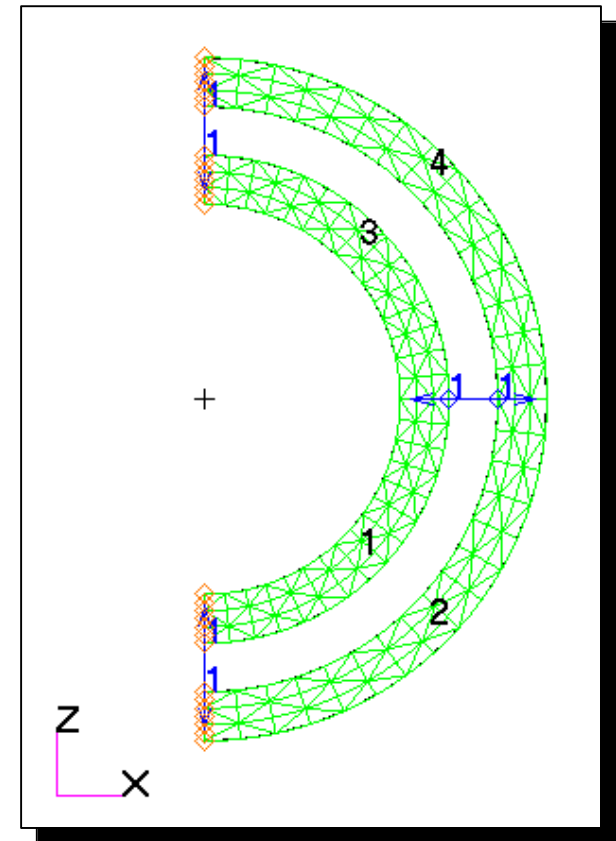
Coordinate Frame

☐ Auto Execute

Node List

Node 31:434:31 645:651 862

In MD Nastran axi-symmetric analysis, the model grid points must be in the X-Z plane of the basic coordinate system. The X (= R) coordinate must be ≥ 0 to zero. It cannot be a negative number, otherwise it will result in a fatal message. The axis of rotation is the Z-axis



☐ Show Node Location Information

Node ID	Coord 1 Value	Coord 2 Value	Coord 3 Value	Reference CID
31	-3.4969111E-007	0.	-8.	(Global) Rectangular
62	-3.6426158E-007	0.	-8.333333	(Global) Rectangular
93	-3.7883206E-007	0.	-8.666667	(Global) Rectangular
124	-3.934025E-007	0.	-9.	(Global) Rectangular

NAS
Copy

S6-164

PROJECT 11, ANALYZE/TRANSLATION

- Change the value of the tolerance Writing from $1.0e-21$ to $1.0e-06$
- The zero value is set to $1.0e-06$, anything less than this will be written as zero (0.0).
- The value of the tolerance is changed to force the X-coordinates at the axis of rotation and the Y-coordinates to be zero.

The image shows two side-by-side dialog boxes from the MSC.Nastran software. The left dialog, titled 'Analysis', has 'Action' set to 'Analyze', 'Object' to 'Entire Model', and 'Method' to 'Full Run'. It also shows 'Code' as 'MSC.Nastran' and 'Type' as 'Thermal'. Below these are fields for 'Job Name' (rad_between_concentric_spheres) and 'Job Description (TITLE)' (MSC.Nastran job created on 23-Jul-08 at 15:19:02). The right dialog, titled 'Translation Parameters', shows 'Tolerances' with 'Division' at $1.0e-08$, 'Numerical' at $1.0e-04$, and 'Writing' at $1.0e-06$. It also includes options for 'Bulk Data Format', 'Card Format', and checkboxes for 'Write Properties on Element Entries', 'Write Continuation Markers', 'Write Global Ply IDs', and 'Convert CBARs to CBEAMS'. The 'Geometry Check' is set to 'Inform'. At the bottom of the right dialog are buttons for 'OK', 'Defaults', and 'Cancel'.

NAS104, Section 6, September 2008

Copyright© 2008 MSC.Software Corporation

PROJECT 11, ANALYZE/TRANSLATION

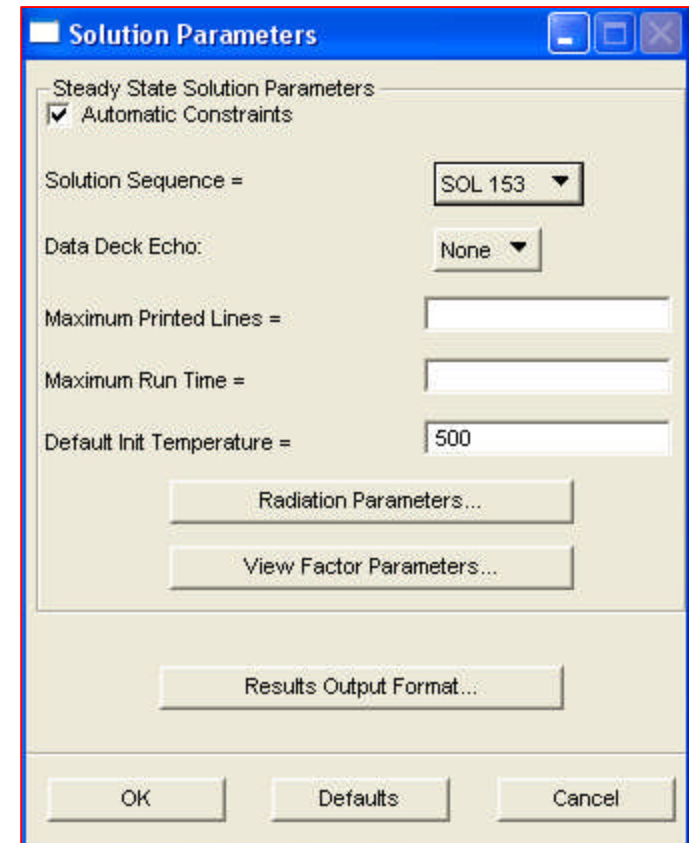
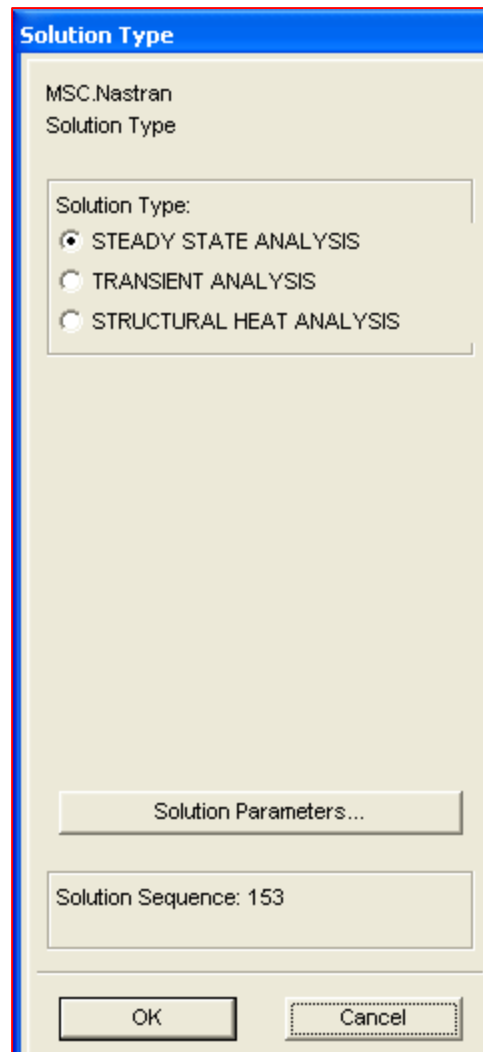
- Instead of changing the value of the tolerance Writing, it is possible to use the utility Utilities/FEM-Nodes/Modify Node Coordinates, and force the appropriate coordinates to have a value of zero

The screenshot shows the 'Translation Parameters' dialog box with the following settings:

- Tolerances:**
 - Division: 1.0e-08
 - Numerical: 1.0e-04
 - Writing: 1.0e-06
- Bulk Data Format:**
 - Sorted Bulk Data: No
 - Card Format: either
 - ☐ Write Stored Precision
 - Button: Precision Control Options...
- Node Coordinates:**
 - reference frame (dropdown)
 - Coord Frame Coordinates: global (dropdown)
 - MSC.Nastran Version: 2008.
 - Number of Tasks: (empty field)
- Checkboxes:**
 - ☐ Write Properties on Element Entries
 - ☐ Write Continuation Markers
 - ☒ Write Global Ply IDs
 - ☐ Convert CBARs to CBEAMs
- Geometry Check:**
 - Inform (dropdown)
 - ☐ Use Iterative Solver
- Buttons:**
 - Ext. Superelement Spec...
 - Numbering Options...
 - Bulk Data Include File...
 - OK, Defaults, Cancel

PROJECT 11, ANALYZE/SOLUTION TYPE

- Use 500 for the initial temperature
- Click Radiation Parameters...



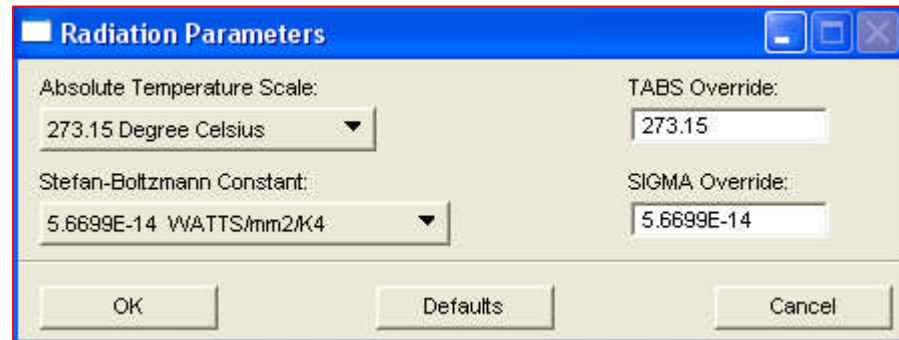
PROJECT 11, ANALYZE/SOLUTION TYPE

- The Stefan-Boltzmann constant (SIGMA Override value) will be written as zero to the MD Nastran input file, because of specifying the Write value as 1.0e-06.

- PARAM SIGMA 0.

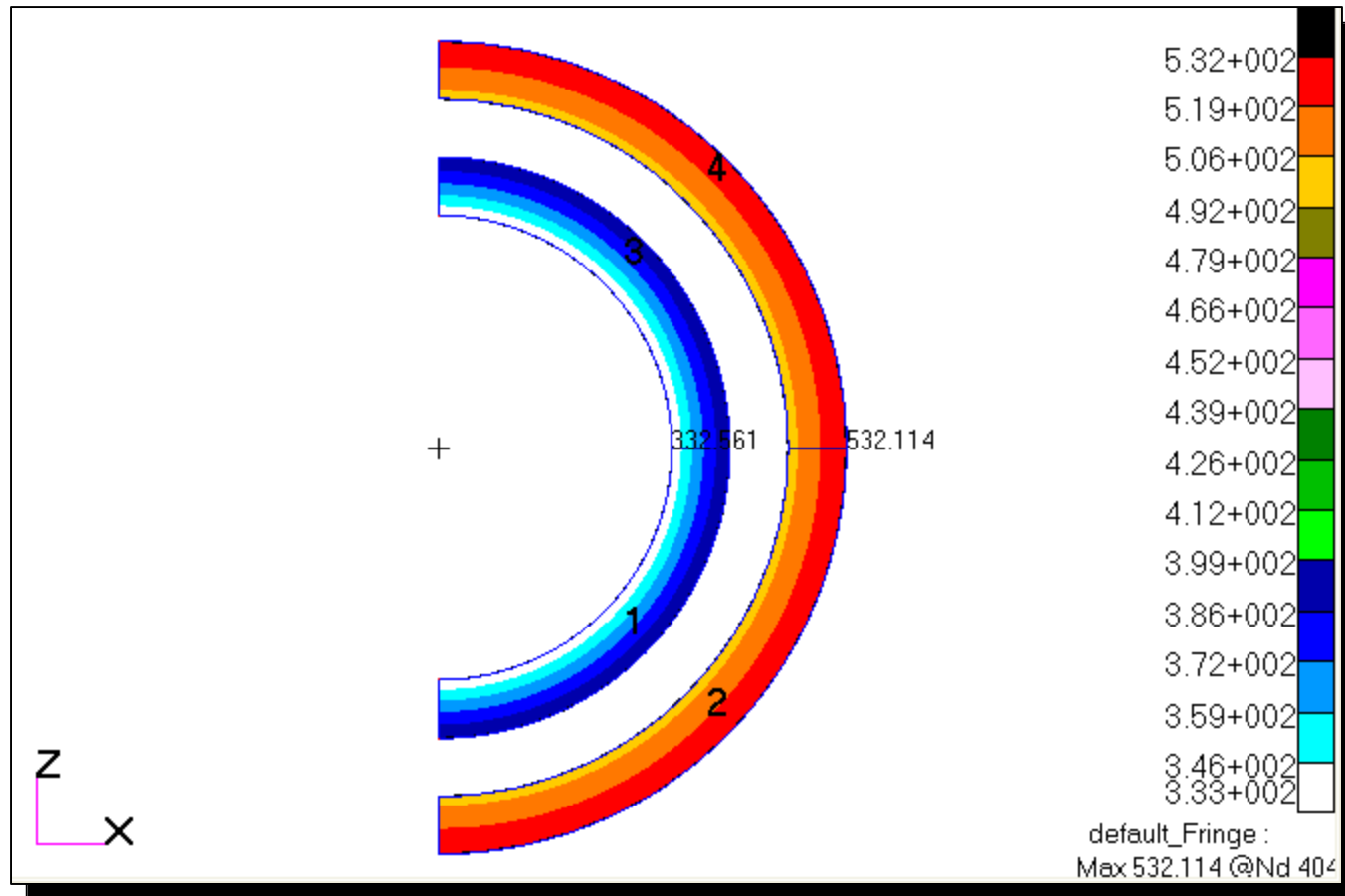
- Replace this with

- PARAM,SIGMA,5.67e-14

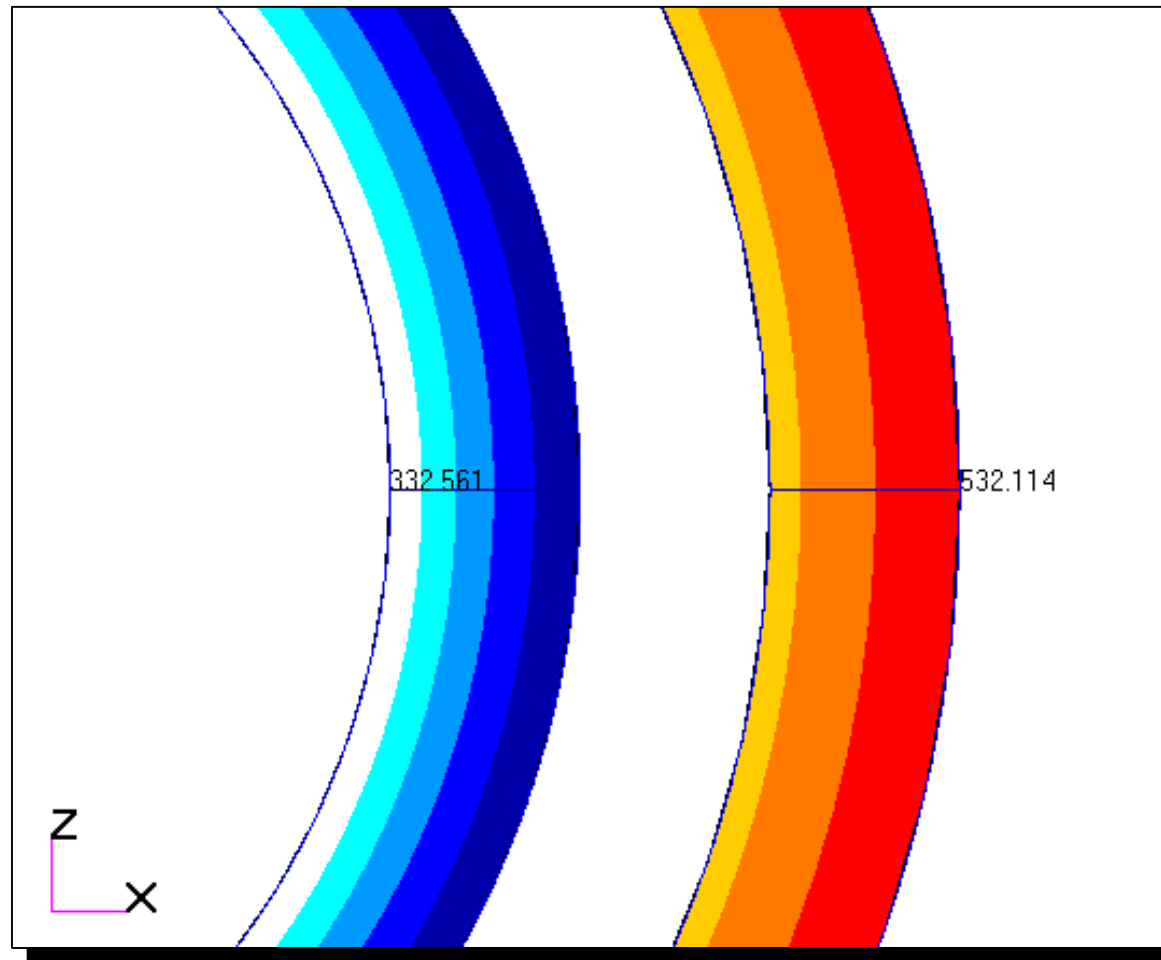


- Also, NLPARM, ID, NINC, DT, KMETHOD, KSTEP, MAXITER, CONV, INTOUT, +, EPSU, EPSP, EPSW, ...
 - The default value for EPSW is 1.0e-07
 - The value of EPSW is written out as zero because of the value of Write
 - Edit the Bulk Data in the MD Nastran input file to specify the correct value of EPSW

PROJECT 11, TEMPERATURE RESULTS



PROJECT 11, TEMPERATURE RESULTS

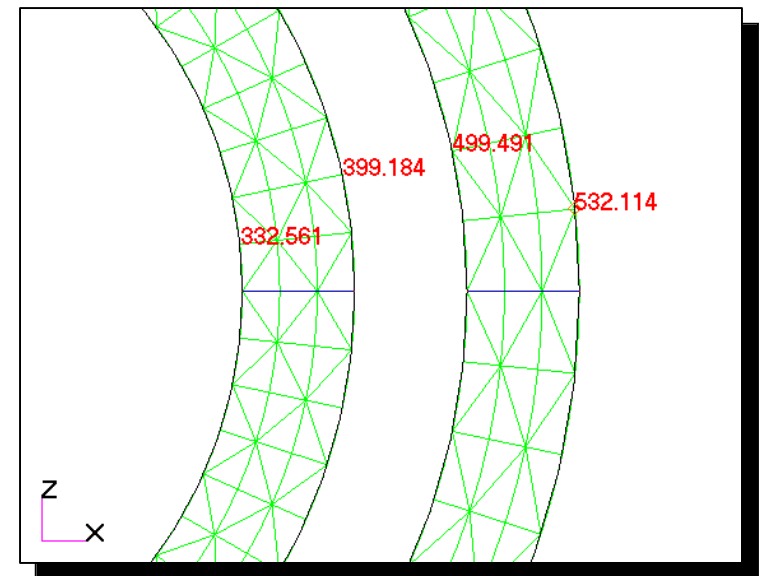
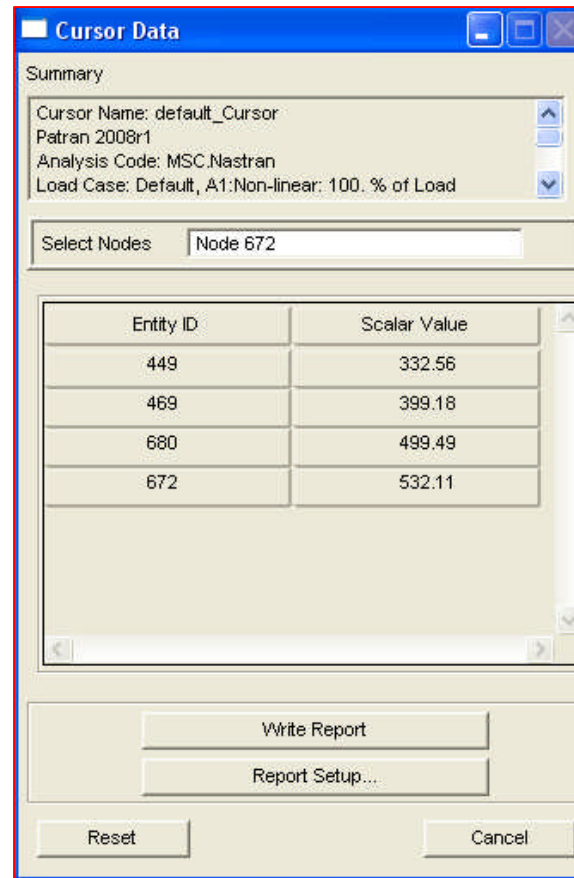
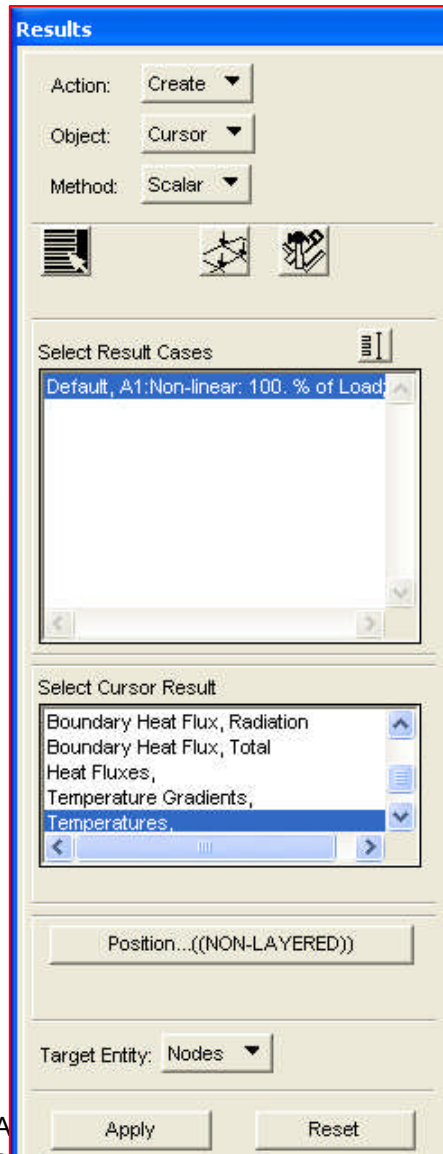


NAS104, Section 6, September 2008

Copyright© 2008 MSC Software Corporation

S6-170

PROJECT 11, CREATE- CURSOR- SCALAR



PROJECT 11, SUPPORTING FILES

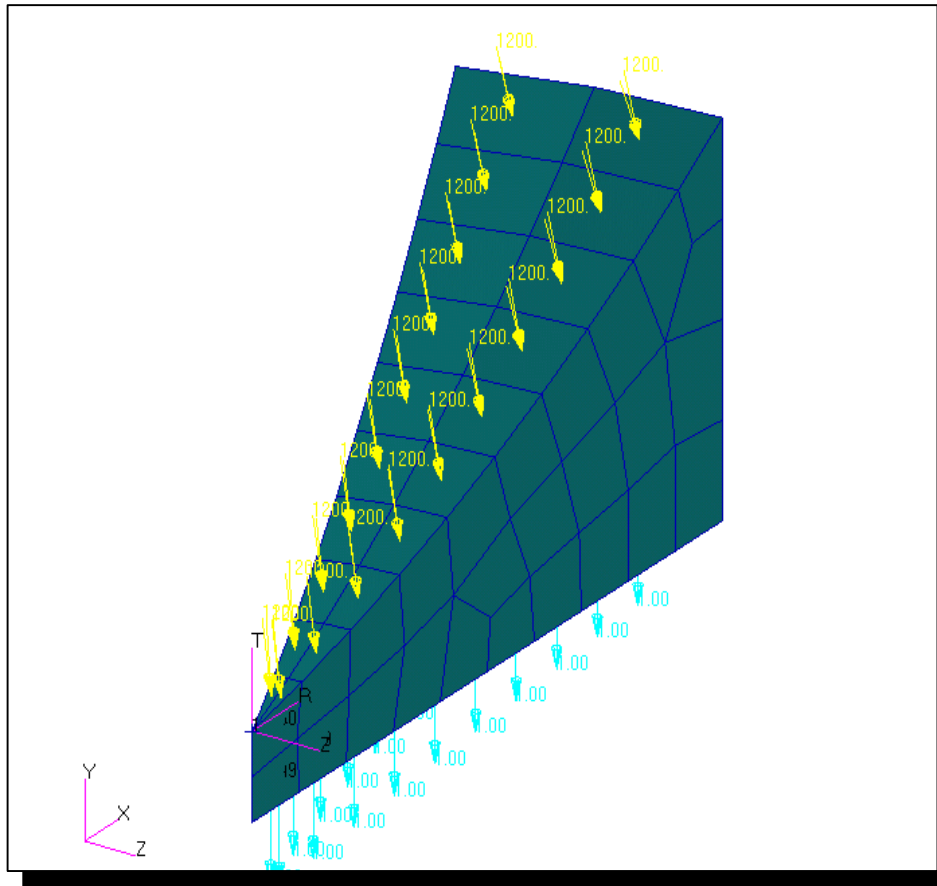
- There are several files that were created by doing this work
 - rad_between_concentric_spheres.bdf
 - rad_between_concentric_spheres.db
 - rad_between_concentric_spheres.xdb

WAVELENGTH DEPENDENT EMISSIVITY



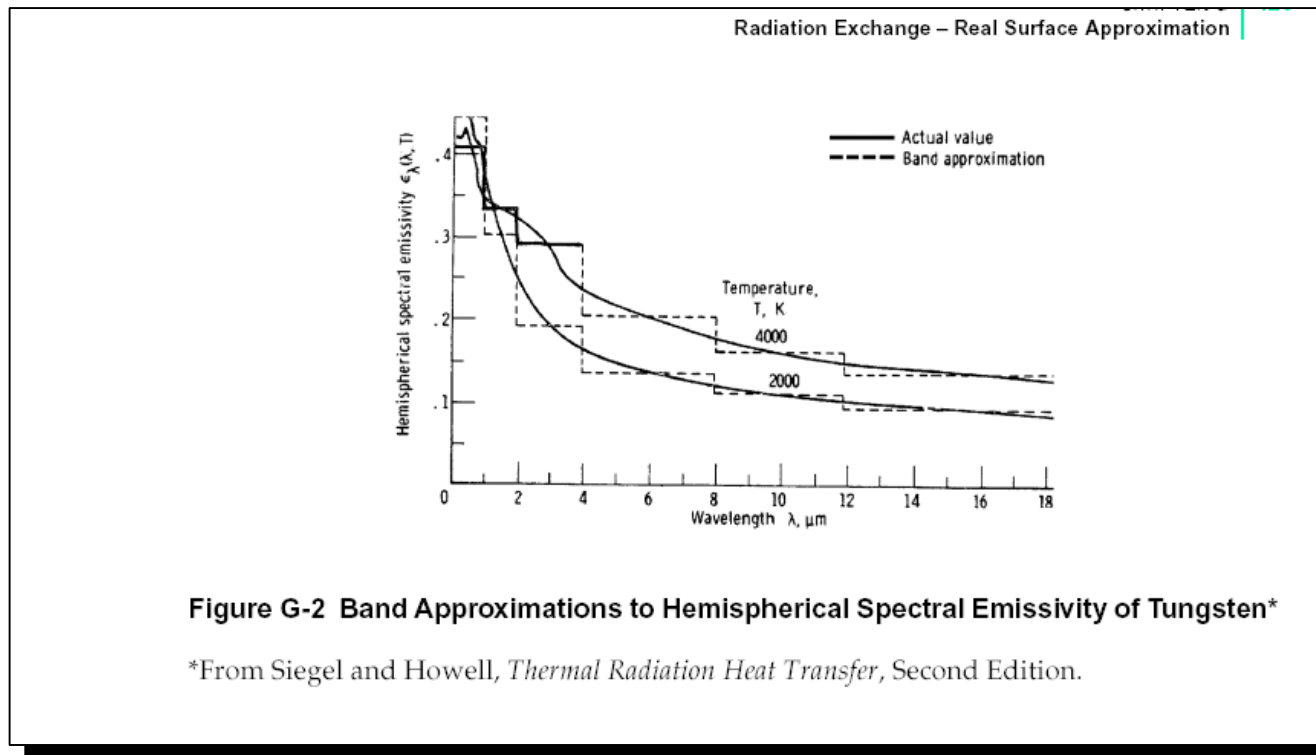


WAVELENGTH DEPENDENT EMISSIVITY

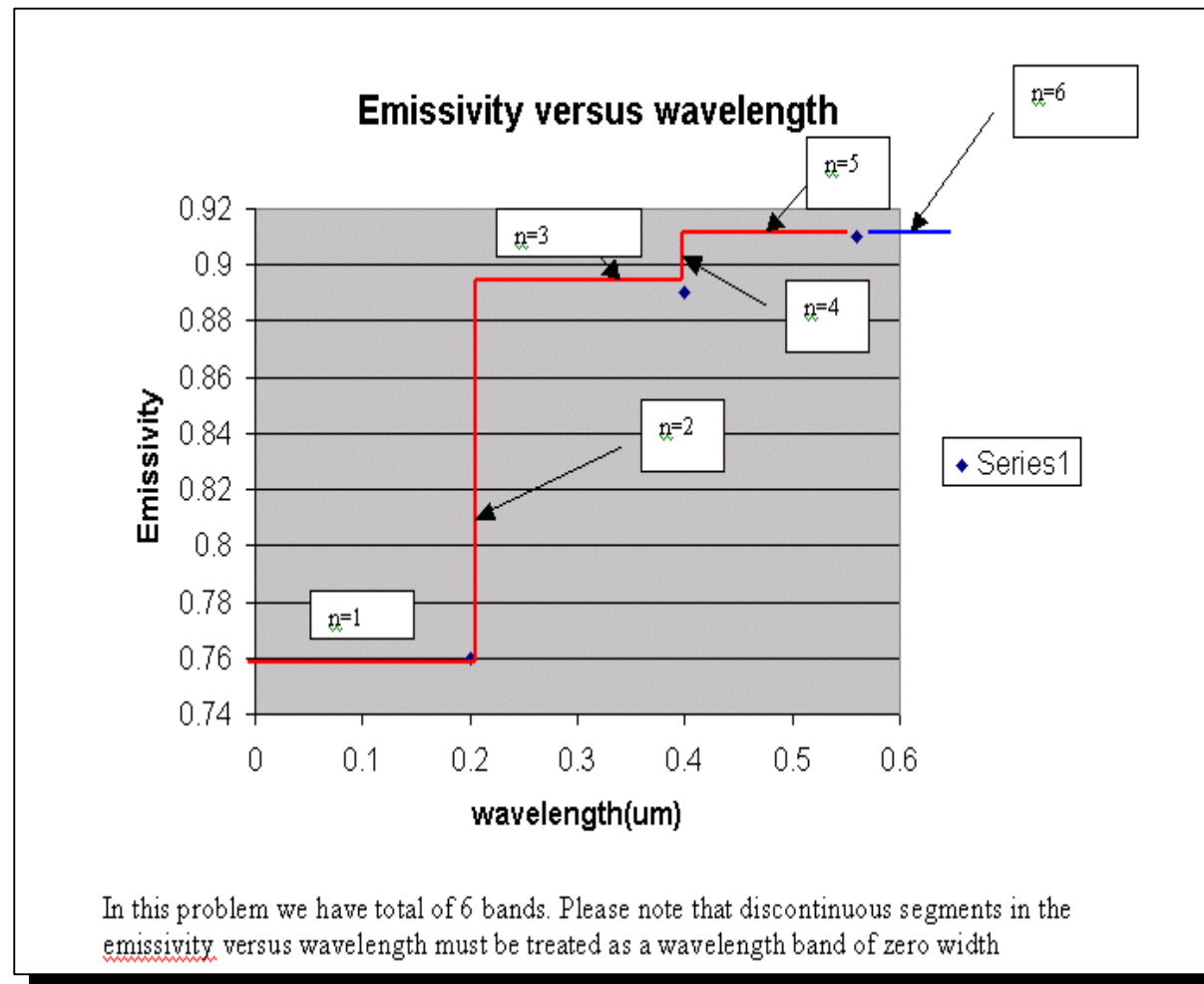


- Model with wavelength dependent emissivity
- Boundary conditions
 - A directional heat flux vector, with heat flux magnitude of 1200 Watts/m², is in the negative Y-direction.
 - On the bottom of the plate, energy is radiated to space at 20°C, with a view factor equal to 1.

WAVELENGTH DEPENDENT EMISSIVITY



WAVELENGTH DEPENDENT EMISSIVITY



RADBND BULK DATA ENTRY

RADBND

Radiation Wavelength Band Definition

Specifies Planck's second radiation constant and the wavelength breakpoints used for radiation exchange problems.

Format:

1	2	3	4	5	6	7	8	9	10
RADBND	NUMBER	PLANCK2	LAMBDA1	LAMBDA2	LAMBDA3	LAMBDA4	LAMBDA5	LAMBDA6	
	LAMBDA7	etc.							

Example:

RADBND	6	14388.0	1.0	2.0	4.0	8.0	12.0		

Field	Contents
NUMBER	Number of radiation wave bands. See Remarks. (Integer > 1)
PLANCK2	Planck's second radiation constant. See Remarks. (Real > 0.0)
LAMBDAi	Highest wavelength of the i-th wave band. See Remarks. (Real $\geq$ 0.0)

RADBND

\$ Radiation Material Properties

RADBND, 6, 14388.0 , 0.2, 0.2, 0.4, 0.4, 0.56

RADM, 1, 1.0, 0.76, 0.76, 0.89, 0.89, 0.91

QVECT WITH RADBND

MD Nastran input file: glass_solid.bdf

This is the input file without RADBND, and TSOUR of the QVECT entry

The directional heating in MD Nastran Thermal analysis is modeled with the QVECT bulk data entry. The radiation to space is modeled with RADBC bulk data entry.

```
$QVECT,SID,Q0,TSOUR,CE,E1,E2,E3,CNTRLND,+  
+,EID1,EID2,etc
```

```
QVECT, 2, 1200.0, , , 0., -1., 0., 0,+  
+, 100001
```

As one can see from the above entry, the heat flux value is 1200, and the incoming directional cosines is equal to (0,-1,0) along the negative Y-direction. Note that there is a CE field on Field 5 on the QVECT entry where the coordinate system can be specified. In this case, the default rectangular coordinate system is being used. The element ID 100001 is the CHBDYG element ID.

Another thing here is the TSOUR entry, this is defined as the temperature of the radiant source. This entry is only used if the RADBND bulk data entry is being used, where the absorptivity or emissivity is a function of the wavelength.

Currently in MD Patran version 2001, the RADBND entry is not supported, and therefore the user must input the RADBND manually in the MD Nastran input file. Additionally, the TSOUR entry is not supported. The user must edit the input file to add the TSOUR value of 5780 °K (effective surface temperature of the sun).

```
QVECT,2 ,1200.0, 5780.0, , 0., -1., 0., 0,+  
+, 100001
```

MD Nastran input file: glass_solid_band1_c_bug1.dat

This is the input file including the RADBND entry, and TSOUR of the QVECT entry.

MD NASTRAN INPUT FILE

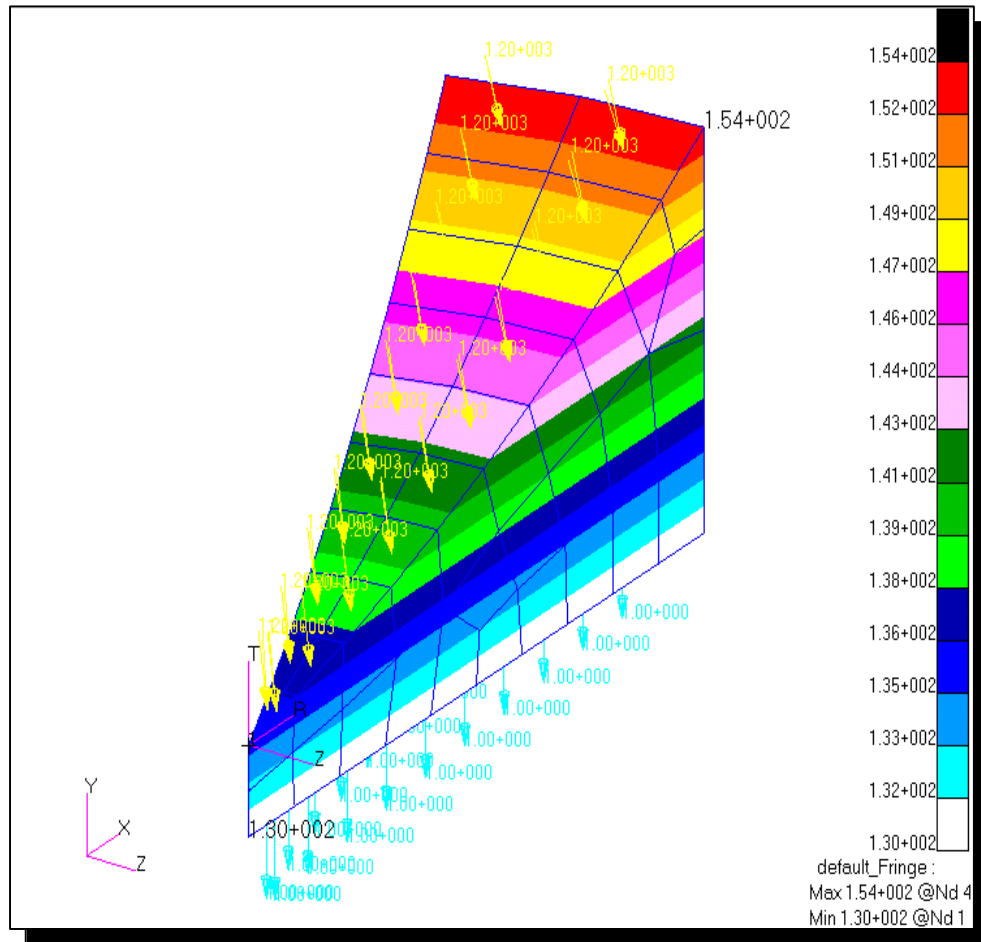
```

QVECT 2 1200. 5780.0 0. -1. 0. 0
100001
QVECT 2 1200. 5780.0 0. -1. 0. 0
100002
QVECT 2 1200. 5780.0 0. -1. 0. 0
100003
QVECT 2 1200. 5780.0 0. -1. 0. 0
100004
QVECT 2 1200. 5780.0 0. -1. 0. 0
100005
QVECT 2 1200. 5780.0 0. -1. 0. 0
100006
QVECT 2 1200. 5780.0 0. -1. 0. 0
100007
QVECT 2 1200. 5780.0 0. -1. 0. 0
100008
QVECT 2 1200. 5780.0 0. -1. 0. 0
100009
QVECT 2 1200. 5780.0 0. -1. 0. 0
100010
QVECT 2 1200. 5780.0 0. -1. 0. 0
100011
QVECT 2 1200. 5780.0 0. -1. 0. 0
100012
QVECT 2 1200. 5780.0 0. -1. 0. 0
100013
QVECT 2 1200. 5780.0 0. -1. 0. 0
100014
QVECT 2 1200. 5780.0 0. -1. 0. 0
100015
QVECT 2 1200. 5780.0 0. -1. 0. 0
100016
QVECT 2 1200. 5780.0 0. -1. 0. 0
100017
QVECT 2 1200. 5780.0 0. -1. 0. 0
100018
    
```

```

$ Radiation Boundary Conditions of Load Set : radbc
RADBC 115 1. 0 100021
RADBC 115 1. 0 100022
RADBC 115 1. 0 100023
RADBC 115 1. 0 100024
RADBC 115 1. 0 100025
RADBC 115 1. 0 100026
RADBC 115 1. 0 100027
RADBC 115 1. 0 100028
RADBC 115 1. 0 100029
RADBC 115 1. 0 100030
RADBC 115 1. 0 100031
RADBC 115 1. 0 100032
RADBC 115 1. 0 100033
RADBC 115 1. 0 100034
RADBC 115 1. 0 100035
RADBC 115 1. 0 100036
$ Radiation Material Properties
RADBNB, 6, 14388.0, 0.2, 0.2, 0.4, 0.4, 0.56
RADM, 1, 1.0, 0.76, 0.76, 0.89, 0.89, 0.91
$RADM 1 1.
$RADMT 1 1
RADM, 2, 1.0, 1.0, 1.0, 1.0, 1.0, 1.0
$RADM 2 1. 1.
$ Material Tables ( No Conversions Needed )
$ Temperature Dependent Material Table : alpha_temp
TABLEM1 1
0. .56 500. .67 1000. .78 2000. .78
ENDT
SPC 1 115 1 20.
    
```

OUTPUT



MODEL SUMMARY

NUMBER OF GRID POINTS = 114
NUMBER OF SPOINT ENTRIES = 1

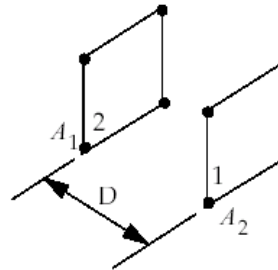
NUMBER OF CHBDYG ELEMENTS = 40
NUMBER OF CHEXA ELEMENTS = 48
NUMBER OF CPENTA ELEMENTS = 4
NUMBER OF RADBC ELEMENTS = 20

TWO ELEMENTS WITH RADIATION EXCHANGE



TWO ELEMENTS WITH RADIATION EXCHANGE

F.5 Two Element Example for Radiant Exchange



$$[R]_e = \sigma[A\varepsilon - A\alpha(A - F(I - \alpha))^{-1}F\varepsilon] \quad \text{Eq. F-29}$$

$$[F] = \begin{bmatrix} \backslash & \\ & A \\ & & \backslash \end{bmatrix} [f] \quad \text{Eq. F-30}$$

$$\begin{aligned} (A - F(I - \alpha))^{-1} &= \left[\begin{bmatrix} A_1 & 0 \\ 0 & A_2 \end{bmatrix} - \begin{bmatrix} 0 & A_1 f_{12} \\ A_2 f_{21} & 0 \end{bmatrix} \begin{bmatrix} 1 - \varepsilon_1 & 0 \\ 0 & 1 - \varepsilon_2 \end{bmatrix} \right]^{-1} \\ &= \begin{bmatrix} A_1 & -A_1 f_{12}(1 - \varepsilon_2) \\ -A_2 f_{21}(1 - \varepsilon_1) & A_2 \end{bmatrix}^{-1} \end{aligned} \quad \text{Eq. F-31}$$

TWO ELEMENTS WITH RADIATION EXCHANGE (Cont.)

For ease of illustration (and manipulation) let surfaces 1 and 2 be black bodies, then

$$\varepsilon_1 = \varepsilon_2 = 1.0$$

then

$$(A - F(I - \alpha))^{-1} = \begin{bmatrix} \frac{1}{A_1} & 0 \\ 0 & \frac{1}{A_2} \end{bmatrix} \quad \text{Eq. F-32}$$

$$[R]_e = \sigma \left[\begin{bmatrix} A_1 & 0 \\ 0 & A_2 \end{bmatrix} - \begin{bmatrix} A_1 & 0 \\ 0 & A_2 \end{bmatrix} \begin{bmatrix} \frac{1}{A_1} & 0 \\ 0 & \frac{1}{A_2} \end{bmatrix} \begin{bmatrix} 0 & A_1 f_{12} \\ A_2 f_{21} & 0 \end{bmatrix} \right] \quad \text{Eq. F-33}$$

$$[R]_e = \sigma \begin{bmatrix} A_1 & -A_1 f_{12} \\ -A_2 f_{21} & A_2 \end{bmatrix} \quad \text{Eq. F-34}$$

TWO ELEMENTS WITH RADIATION EXCHANGE (Cont.)

$$\{Q\}_e = -\sigma \begin{bmatrix} A_1 & -A_1 F_{12} \\ -A_2 F_{21} & A_2 \end{bmatrix}_e \begin{Bmatrix} u_1^4 \\ u_2^4 \end{Bmatrix}_e \quad \text{Eq. F-35}$$

To further define a specific problem, let $A_1 = A_2 = D = 1.0$, $T_1 = 1000$ and $T_2 = 0.0$. For this geometry then, $f_{12} = f_{21} \cong .20$. The resulting heat flows are:

$$\begin{aligned} Q_1 &= -\sigma(1000^4 - .2(0)) = -\sigma(1000)^4 \\ Q_2 &= -\sigma(-.2(0) - .2(1000)^4) = .2\sigma(1000)^4 \end{aligned} \quad \text{Eq. F-36}$$

Note: Since the exchange matrix $[R]_e$ is not conservative, we recognize that the MSC.Nastran default condition assumes a third exchange surface representing a loss to space. Therefore, the loss to space is:

$$\begin{aligned} Q_1 + Q_2 + Q_3 &= 0.0 \\ Q_3 &= .8\sigma(1000)^4 \end{aligned} \quad \text{Eq. F-37}$$



SECTION 7

ADDITIONAL TOPICS





ADDITIONAL TOPICS

- Thermal stress analysis for nonlinear or linear thermal systems, but only for linear structural systems
- Linear thermal stress analysis using STRUCTURAL HEAT ANALYSIS (SOL 101)
- Restart transient thermal analysis
- Restart from thermal analysis to stress analysis
- Use of NOLINi entries for thermostat control
- Use of TABLEMi entries
- Radiation matrix
- Performance increase of view factor calculation
- CHBDYi boundary condition surface elements



THERMAL STRESS ANALYSIS





THERMAL STRESS ANALYSIS

- This procedure is used for a nonlinear or linear heat transfer analysis
- First, perform a heat transfer analysis using MD Nastran, run1
- The heat transfer analysis is used to calculate the temperature distribution throughout a structure.
- Output all the nodal temperatures to a punch file using the Case Control command
 - THERMAL(PUNCH)=ALL

THERMAL STRESS ANALYSIS (Cont.)

- Second, perform a stress/displacement analysis using MD Nastran, run2.
- Use the temperature entries in the punch file from the first run, run1, for the thermal load of this run.
- The Case Control command `TEMPERATURE(LOAD)=n` is used to select the Bulk Data `TEMP` set loading data from run1.
- Provide thermal expansion coefficient for the `MAT1` entry.

THERMAL STRESS ANALYSIS (Cont.)

- The Bulk Data TEMP entries output to the punch file for the first run, run1, are in double field format for increased accuracy.
- THERMAL(SORT1,PUNCH)=ALL
 - SORT1: output is a tabular listing of GRID point temperatures for each load or time point
 - PUNCH: causes a punch file to be created
 - ALL: outputs temperatures for all GRID points
 - In transient analysis, all GRID point temperatures at all time points are output in SORT2 format by default.
- Control time points that are output with the Case Control command OTIME.

THERMAL STRESS, HEAT TRANSFER, RUN1

Extracted from MD Nastran model .DAT file

```
ID stress2,msc
SOL 153
TIME 10
CEND
TITLE=SEMI-INFINITE RECTANGULAR FIN
TEMP(init)=23
LABEL=heat transfer analysis
ANALYSIS=HEAT
NLPARM=300
SPC=10
LOAD=33
THERMAL (PUNCH) =ALL
FLUX=ALL
```

THERMAL STRESS, HEAT TRANSFER, RUN1

Extracted from MD Nastran model .DAT file

```
BEGIN BULK
$ Ambient temperature is represented by spoint 100
SPOINT      100
$Convection from top and bottom edges
CHBDYP,10,10,LINE,,,1,2,1
=,*1,=,=,=,*1,*1,==
=7
CHBDYP,19,10,LINE,,,11,12,1
=,*1,=,=,=,*1,*1,==
=7
CONV,10,20,,,100,100
=,*1,==
=16
```

THERMAL STRESS, HEAT TRANSFER, RUN1

Extracted from MD Nastran model .DAT file

```
CQUAD4,1,10,1,2,12,11
=,*1,=,*1,*1,*1,*1
=7
GRID,1,,10.0
=,*1,=,*10.0,==
=8
GRID,11,,10.0,10.0
=,*1,=,*10.0,==
=8
MAT4    10      1.0                      1.0
$ The effective edge for the plate is 1.0
PHBDY   10      1.0
PCONV   20      40      0      0.0
$Convective coefficient as a function Temp
MAT4     40                      0.0005
MATT4    40                      45
TABLEM1 45
++0000010.0      1.0      10.0      3.0      100.0      3.5      200.0      4.0      +000001
++000002300.     4.0      ENDT      +000002
```

THERMAL STRESS, HEAT TRANSFER, RUN1

Extracted from MD Nastran model .DAT file

```
$
NLPARM 300          AUTO
PSHELL 10 10 1.0
$ The heat flux along the edge between grid 1 and 11
QHBDY 33 LINE .90 1.0 1 11
$ Volumetric heating
QVOL 33 .01 1 THRU 9
$The ambient temp is fixed at 100 degree
SPC 10 100 1 0.0
TEMP 23 100 0.0
$ Initial guess temperature
TEMPD 23 100.0
ENDDATA
```

THERMAL STRESS, PUNCH FILE RUN1.PCH

\$TITLE	=SEMI-INFINITE RECTANGULAR FIN	1
\$SUBTITLE=		2
\$LABEL	=HEAT TRANSFER ANALYSIS	3
\$DISPLACEMENTS		4
\$REAL OUTPUT		5
\$SUBCASE ID =	1	6
TEMP*	1 1 8.692107E+01	7
TEMP*	1 2 7.880087E+01	8
TEMP*	1 3 7.217424E+01	9
TEMP*	1 4 6.681807E+01	10
TEMP*	1 5 6.255383E+01	11
TEMP*	1 6 5.924054E+01	12
TEMP*	1 7 5.676937E+01	13
TEMP*	1 8 5.505955E+01	14
TEMP*	1 9 5.405537E+01	15
TEMP*	1 10 5.372424E+01	16
TEMP*	1 11 8.692107E+01	17
TEMP*	1 12 7.880087E+01	18
TEMP*	1 13 7.217424E+01	19
TEMP*	1 14 6.681807E+01	20
TEMP*	1 15 6.255383E+01	21
TEMP*	1 16 5.924054E+01	22
TEMP*	1 17 5.676937E+01	23
TEMP*	1 18 5.505955E+01	24
TEMP*	1 19 5.405537E+01	25
TEMP*	1 20 5.372424E+01	26
TEMP*	1 100 0.000000E+00	27

Extracted from MD Nastran punch file run1.pch

THERMAL STRESS, STRESS/DISPL, RUN2

Extracted from MD Nastran model .DAT file

```
ID stress2,msc
SOL 101
TIME 10
CEND
TITLE=SEMI-INFINITE RECTANGULAR FIN
LABEL=stress analysis
SPC=20
TEMP (LOAD)=1
DISP=ALL
STRESS=ALL
BEGIN BULK
INCLUDE 'run1.pch'
MAT1      10      10.0+6      0.33      6.5-6      60.0
SPC1      20      12      1      11      10      20
SPC1      20      0      100
GRDSET                                           3456
```

Add all the previous Bulk Data entries from file run1.dat.

THERMAL STRESS, STRESS/DISPL, RUN2

Extracted from MD Nastran ASCII Results File

```
0      STRESS ANALYSIS

      D I S P L A C E M E N T   V E C T O R
      POINT ID.      TYPE      T1      T2
      1      G      0.0      0.0
      2      G      1.440945E-03  -8.765265E-04
      3      G      2.116879E-03  -3.866043E-04
      4      G      2.472052E-03  -2.789631E-04
      5      G      2.497912E-03  -1.243974E-04
      6      G      2.281249E-03  -2.133548E-05
      7      G      1.876258E-03   6.258629E-05
      8      G      1.332701E-03   1.064134E-04
      9      G      7.127683E-04   1.866366E-04
     10      G      0.0      0.0
```

THERMAL STRESS, STRESS/DISPL, RUN2

Extracted from MD Nastran ASCII Results File

STRESSES IN QUADRILATERAL ELEMENTS (QUAD4)									
ELEMENT		FIBRE	STRESSES IN ELEMENT COORD SYSTEM				PRINCIPAL STRESSES (ZERO SHEAR)		
ID.		DISTANCE	NORMAL-X	NORMAL-Y	SHEAR-XY	ANGLE	MAJOR	MINOR	VON MISES
0	1	-5.000000E-01	-2.762115E+02	-7.005862E+02	2.273737E-13	0.0000	-2.762115E+02	-7.005862E+02	6.112315E+02
		5.000000E-01	-2.762115E+02	-7.005862E+02	2.273737E-13	0.0000	-2.762115E+02	-7.005862E+02	6.112315E+02
0	2	-5.000000E-01	-2.762115E+02	1.652899E+02	1.136868E-13	90.0000	1.652899E+02	-2.762115E+02	3.863528E+02
		5.000000E-01	-2.762115E+02	1.652899E+02	1.136868E-13	90.0000	1.652899E+02	-2.762115E+02	3.863528E+02

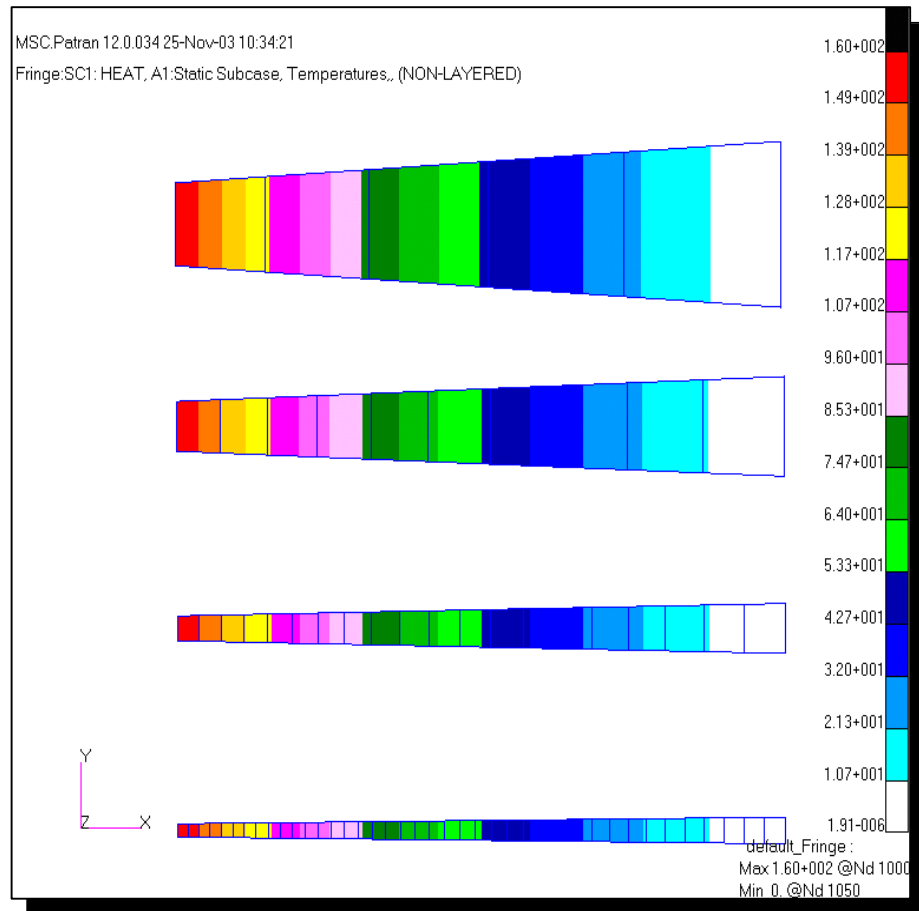
EXERCISES

- Perform Workshop 15 “Thermal Stress Analysis with Directional Heat Loads” in the exercise workbook.
- Perform Workshop 16 “Thermal Stress Analysis of a Bi-metallic Plate” in the exercise workbook.

LINEAR THERMAL STRESS ANALYSIS USING STRUCTURAL HEAT ANALYSIS (SOL 101)

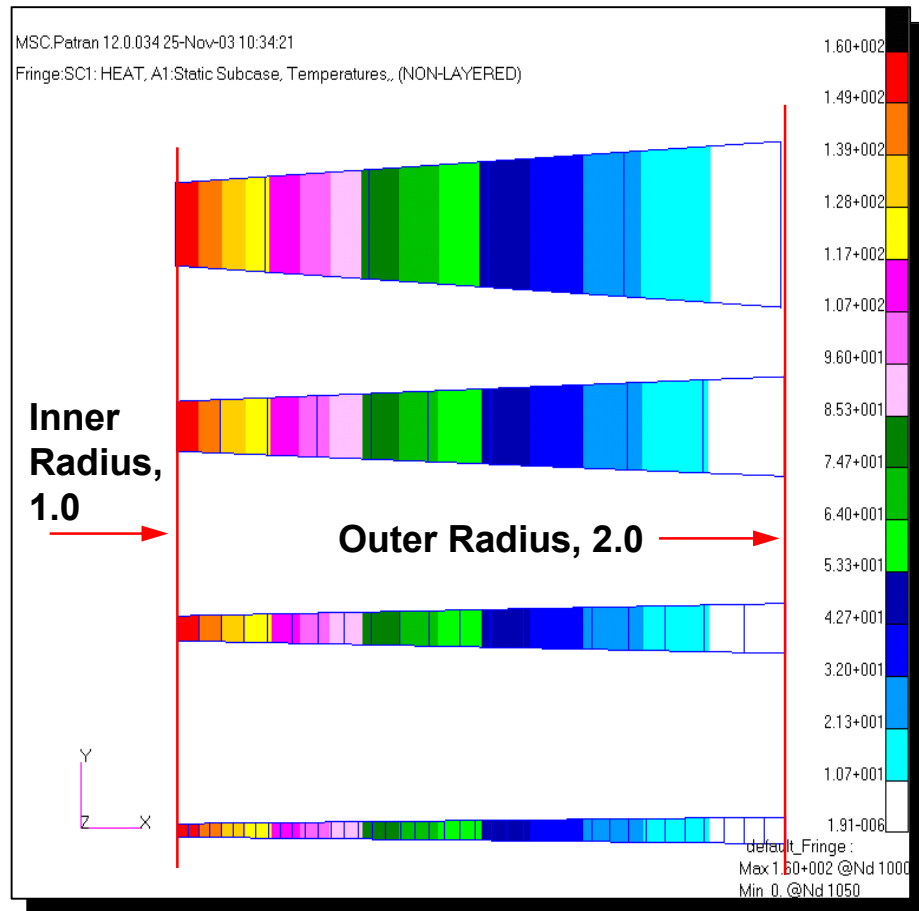


THERMAL STRESS ANALYSIS USING SOL 101



- If the thermal analysis is a linear analysis (without radiation and temperature dependent material properties), SOL 101 can be used in place of the default SOL 153 to obtain the heat transfer temperature result, and execute a thermal stress analysis, based on the heat transfer temperature result, in a single run.

THERMAL STRESS, ANALYTICAL SOLUTION



- Following is an example of a heat transfer analysis, followed by a structural analysis, both using SOL 101.
- There is a closed form solution to check the accuracy of the MD Nastran solution.
- The example is for a spherical shell with constant inner and outer temperature. The inner radius is 1.0 and the outer radius is 2.0. The temperature distribution is solved for in the first subcase. This solution is nontrivial, and varies as $T(r) = C1 + C2/r$, where $C1$ and $C2$ are constants.
- The displacements and stresses are solved for in the second subcase. Both radial and hoop stresses are solved for. The radial displacement varies as $u(r) = D1*r + D2 + D3/r**2$, where $D1$, $D2$, and $D3$ are constants.

ANALYTICAL SOLUTION

THEORY FOR TEMPERATURES, DISPLACEMENTS AND STRESSES IN SPHERICAL SHELL

TEMPERATURE SPECIFIED AT INNER AND OUTER SURFACES
11/25/03

C PROBLEM PARAMETERS

RA = INNER RADIUS
RB = OUTER RADIUS
EM = ELASTIC MODULUS
NU = POISSONS RATIO
AL = COEFFICIENT OF EXPANSION
T0 = REFERENCE TEMPERATURE
TA = INNER TEMPERATURE
TB = OUTER TEMPERATURE
THERMAL CONDUCTIVITY (ANY CONSTANT VALUE)

C DERIVED CONSTANTS

$T1 = (RB \cdot TB - RA \cdot TA) / (RB - RA)$
 $T2 = RB \cdot RA \cdot (TB - TA) / (RB - RA)$
 $C1 = (RB \cdot RB + RB \cdot RA + RA \cdot RA) / (RB + RA)$
 $C2 = (RB \cdot RB \cdot RA \cdot RA) / (RB + RA)$
 $S0 = -EM \cdot AL \cdot RB \cdot RA \cdot (TB - TA) / ((1.0 - NU) \cdot C1 \cdot (RB - RA))$
 $U0 = S0 \cdot (1.0 - 2.0 \cdot NU) / EM + AL \cdot ((RB \cdot TB - RA \cdot TA) / (RB - RA) - T0)$
 $U1 = .5Q0 \cdot S0 \cdot (1.0 + NU) \cdot C1 / EM$
 $U2 = -.5Q0 \cdot S0 \cdot (1.0 + NU) \cdot C2 / EM$

C RESULTS

R = RADIUS (RA < R < RB)

TEMP(R) = T1 - T2/R

U-RAD(R) = U0 * R + U1 + U2/R**2

S-RAD(R) = S0 * (1.0 - C1/R + C2/R**3)

S-AZI(R) = S0 * (1.0 - .50 * C1/R - .50 * C2/R**3)

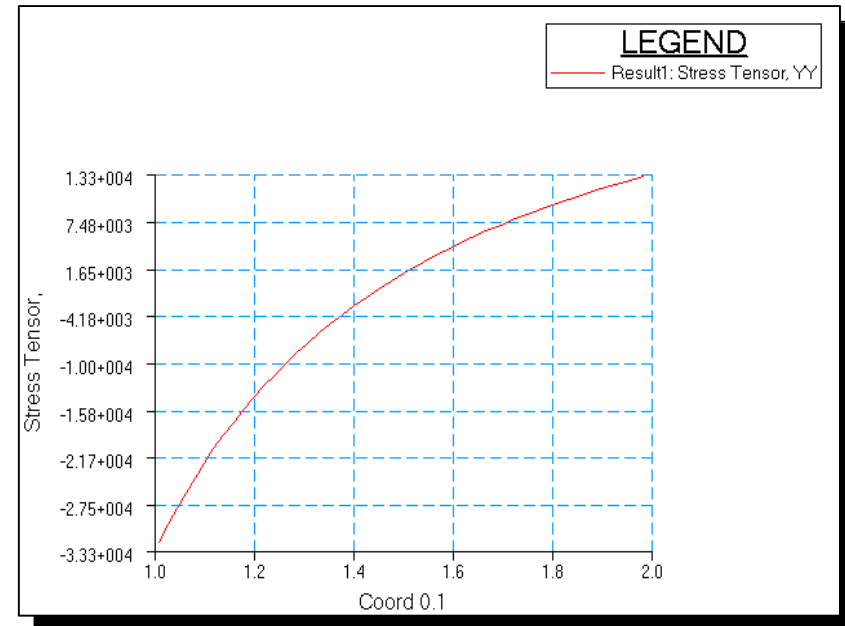
C THEORY FOR TEMPERATURES, DISPLACEMENTS AND STRESSES IN SPHERICAL SHELL

C TEMPERATURE SPECIFIED AT INNER AND OUTER SURFACES, GEOMETRIC INTERVALS

RA = 1.0 ! INNER RADIUS
RB = 2.0 ! OUTER RADIUS
EM = 30.e6 ! ELASTIC MODULUS
NU = .33 ! POISSON'S RATIO
AL = 6.5e-6 ! COEFFICIENT OF EXPANSION
T0 = 60.0 ! REFERENCE TEMPERATURE
TA = 160.0 ! INNER TEMPERATURE
TB = 0.0 ! OUTER TEMPERATURE

ANALYTICAL SOLUTION

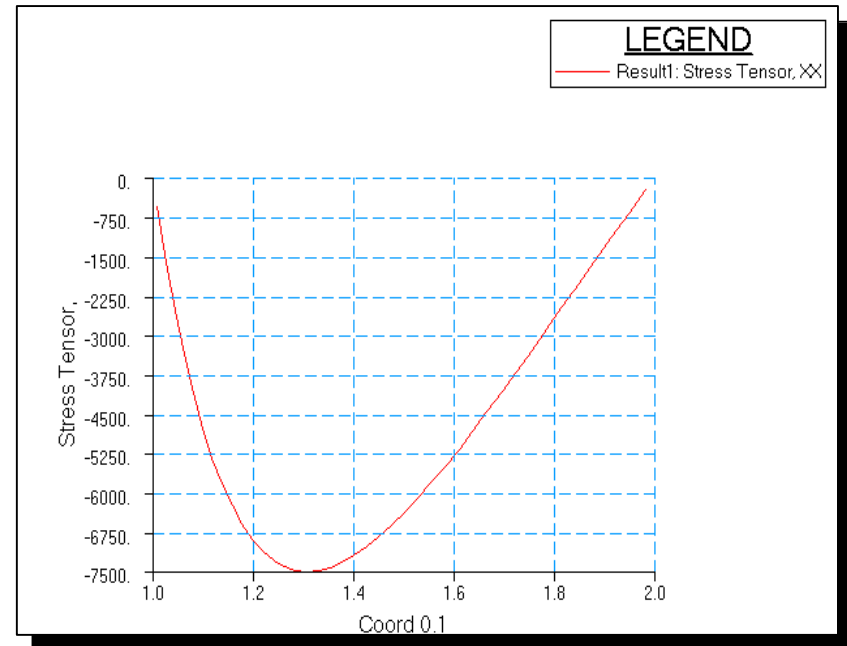
RAD	TEMPER	DISPL-R	SIGMA-R	SIGMA-T
1.00000	1.600000E+02	-9.285714E-05	-1.475960E-27	-3.326226E+04
1.01750	1.544963E+02	-6.973528E-05	-1.097199E+03	-3.111184E+04
1.03530	1.490892E+02	-4.829193E-05	-2.084742E+03	-2.904434E+04
1.05340	1.437782E+02	-2.848913E-05	-2.968972E+03	-2.705652E+04
1.07180	1.385632E+02	-1.028866E-05	-3.755987E+03	-2.514519E+04
1.09050	1.334434E+02	6.347715E-06	-4.451642E+03	-2.330727E+04
1.10960	1.283922E+02	2.153311E-05	-5.064511E+03	-2.153072E+04
1.12900	1.234367E+02	3.521299E-05	-5.596091E+03	-1.982265E+04
1.14870	1.185758E+02	4.742630E-05	-6.051612E+03	-1.818015E+04
1.16880	1.137851E+02	5.826144E-05	-6.437791E+03	-1.659275E+04
1.18920	1.090885E+02	6.769123E-05	-6.756938E+03	-1.506625E+04
1.21000	1.044628E+02	7.578937E-05	-7.014721E+03	-1.359109E+04
1.23110	9.993014E+01	8.254442E-05	-7.213825E+03	-1.217233E+04
1.25270	9.544823E+01	8.803857E-05	-7.359528E+03	-1.079504E+04
1.27460	9.105916E+01	9.223967E-05	-7.453745E+03	-9.470515E+03
1.29680	8.676126E+01	9.518530E-05	-7.500329E+03	-8.196343E+03
1.31950	8.251610E+01	9.692182E-05	-7.502733E+03	-6.959611E+03
1.34260	7.834351E+01	9.745362E-05	-7.463589E+03	-5.764772E+03
1.36600	7.426061E+01	9.680694E-05	-7.386138E+03	-4.615191E+03
1.38990	7.023239E+01	9.499734E-05	-7.272427E+03	-3.499653E+03
1.41420	6.627634E+01	9.204659E-05	-7.125317E+03	-2.421820E+03
1.43890	6.239211E+01	8.797625E-05	-6.947299E+03	-1.380343E+03
1.46410	5.856431E+01	8.278530E-05	-6.739877E+03	-3.699925E+02
1.48970	5.480835E+01	7.651017E-05	-6.506012E+03	6.062257E+02
1.51570	5.112357E+01	6.917275E-05	-6.247878E+03	1.549594E+03
1.54220	4.749579E+01	6.076125E-05	-5.966446E+03	2.464727E+03
1.56920	4.392557E+01	5.128724E-05	-5.663496E+03	3.352345E+03
1.59660	4.042591E+01	4.080270E-05	-5.341946E+03	4.210129E+03
1.62450	3.698369E+01	2.928647E-05	-5.002337E+03	5.042163E+03
1.65290	3.359913E+01	1.675124E-05	-4.646237E+03	5.849172E+03
1.68180	3.027233E+01	3.210085E-06	-4.275152E+03	6.631877E+03
1.71120	2.700327E+01	-1.132367E-05	-3.890515E+03	7.391000E+03
1.74110	2.379186E+01	-2.683653E-05	-3.493696E+03	8.127257E+03
1.77150	2.063788E+01	-4.331485E-05	-3.085994E+03	8.841355E+03
1.80250	1.753121E+01	-6.080244E-05	-2.667287E+03	9.536181E+03
1.83400	1.448201E+01	-7.923224E-05	-2.240087E+03	1.021004E+04
1.86610	1.148063E+01	-9.865176E-05	-1.804148E+03	1.086560E+04
1.89870	8.536367E+00	-1.189894E-04	-1.361876E+03	1.150138E+04
1.93190	5.640043E+00	-1.402963E-04	-9.129129E+02	1.211986E+04
1.96560	2.800163E+00	-1.624970E-04	-4.595504E+02	1.271971E+04
2.0	9.860761E-31	-1.857143E-04	-5.534852E-28	1.330490E+04



Sigma-T stress from NASTRAN run

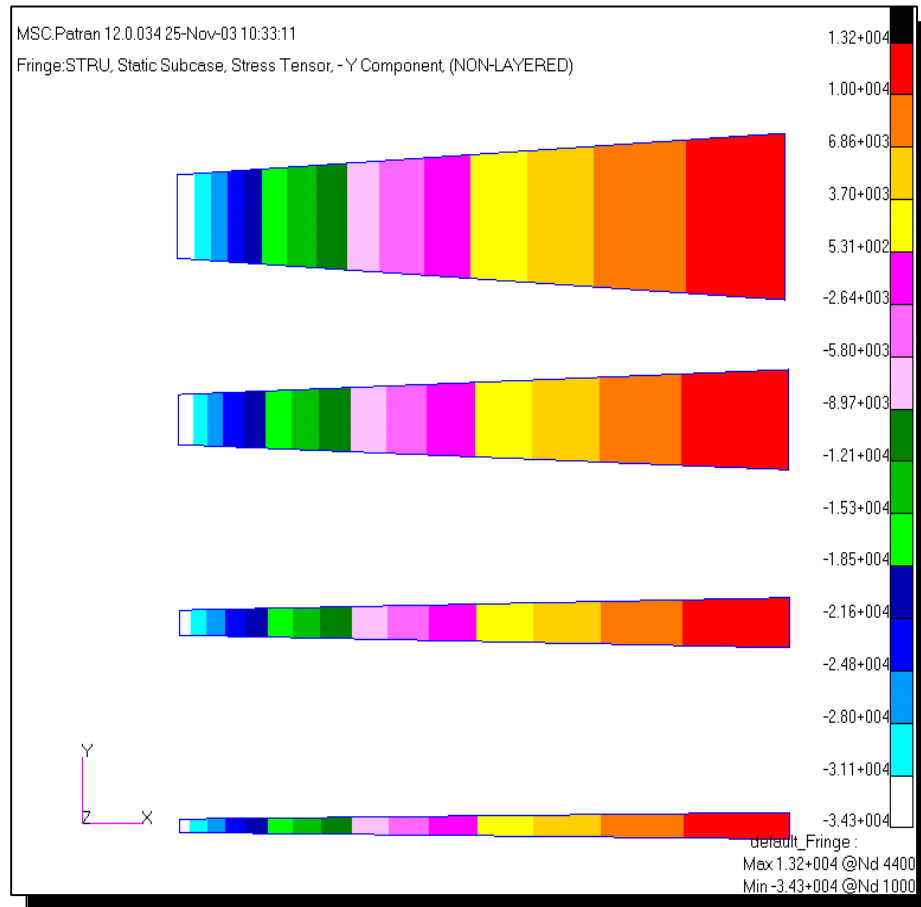
ANALYTICAL SOLUTION

RAD	TEMPER	DISPL-R	SIGMA-R	SIGMA-T
1.00000	1.600000E+02	-9.285714E-05	-1.475960E-27	-3.326226E+04
1.01750	1.544963E+02	-6.973528E-05	-1.097199E+03	-3.111184E+04
1.03530	1.490892E+02	-4.829193E-05	-2.084742E+03	-2.904434E+04
1.05340	1.437782E+02	-2.848913E-05	-2.968972E+03	-2.705652E+04
1.07180	1.385632E+02	-1.028866E-05	-3.755987E+03	-2.514519E+04
1.09050	1.334434E+02	6.347715E-06	-4.451642E+03	-2.330727E+04
1.10960	1.283922E+02	2.153311E-05	-5.064511E+03	-2.153072E+04
1.12900	1.234367E+02	3.521299E-05	-5.596091E+03	-1.982265E+04
1.14870	1.185758E+02	4.742630E-05	-6.051612E+03	-1.818015E+04
1.16880	1.137851E+02	5.826144E-05	-6.437791E+03	-1.659275E+04
1.18920	1.090885E+02	6.769123E-05	-6.756938E+03	-1.506625E+04
1.21000	1.044628E+02	7.578937E-05	-7.014721E+03	-1.359109E+04
1.23110	9.993014E+01	8.254442E-05	-7.213825E+03	-1.217233E+04
1.25270	9.544823E+01	8.803857E-05	-7.359528E+03	-1.079504E+04
1.27460	9.105916E+01	9.223967E-05	-7.453745E+03	-9.470515E+03
1.29680	8.676126E+01	9.518530E-05	-7.500329E+03	-8.196343E+03
1.31950	8.251610E+01	9.692182E-05	-7.502733E+03	-6.959611E+03
1.34260	7.834351E+01	9.745362E-05	-7.463589E+03	-5.764772E+03
1.36600	7.426061E+01	9.680694E-05	-7.386138E+03	-4.615191E+03
1.38990	7.023239E+01	9.499734E-05	-7.272427E+03	-3.499653E+03
1.41420	6.627634E+01	9.204659E-05	-7.125317E+03	-2.421820E+03
1.43890	6.239211E+01	8.797625E-05	-6.947299E+03	-1.380343E+03
1.46410	5.856431E+01	8.278530E-05	-6.739877E+03	-3.699925E+02
1.48970	5.480835E+01	7.651017E-05	-6.506012E+03	6.062257E+02
1.51570	5.112357E+01	6.917275E-05	-6.247878E+03	1.549594E+03
1.54220	4.749579E+01	6.076125E-05	-5.966446E+03	2.464727E+03
1.56920	4.392557E+01	5.128724E-05	-5.663496E+03	3.352345E+03
1.59660	4.042591E+01	4.080270E-05	-5.341946E+03	4.210129E+03
1.62450	3.698369E+01	2.928647E-05	-5.002337E+03	5.042163E+03
1.65290	3.359913E+01	1.675124E-05	-4.646237E+03	5.849172E+03
1.68180	3.027233E+01	3.210085E-06	-4.275152E+03	6.631877E+03
1.71120	2.700327E+01	-1.132367E-05	-3.890515E+03	7.391000E+03
1.74110	2.379186E+01	-2.683653E-05	-3.493696E+03	8.127257E+03
1.77150	2.063788E+01	-4.331485E-05	-3.085994E+03	8.841355E+03
1.80250	1.753121E+01	-6.080244E-05	-2.667287E+03	9.536181E+03
1.83400	1.448201E+01	-7.923224E-05	-2.240087E+03	1.021004E+04
1.86610	1.148063E+01	-9.865176E-05	-1.804148E+03	1.086560E+04
1.89870	8.536367E+00	-1.189894E-04	-1.361876E+03	1.150138E+04
1.93190	5.640043E+00	-1.402963E-04	-9.129129E+02	1.211986E+04
1.96560	2.800163E+00	-1.624970E-04	-4.595504E+02	1.271971E+04
2.0	9.860761E-31	-1.857143E-04	-5.534852E-28	1.330490E+04



Sigma-R stress from NASTRAN run

THERMAL STRESS, SINGLE SOL 101



- To solve this problem it is necessary to use
 - SUBCASE 1 for linear heat transfer analysis
 - Define the thermal loads, temperature boundary conditions, and output requests
 - SUBCASE 2 for structural analysis
 - Define thermal loads (from heat transfer analysis), structural boundary conditions, and output requests
 - BEGIN BULK
 - PARAM, HEATSTAT, YES

THERMAL STRESS, SINGLE SOL 101

- Use PARAM, HEATSTAT, YES to enable this solution

HEATSTAT

Default = NO

In SOL 101, if PARAM,HEATSTAT,YES is entered, then temperatures are computed in a linear steady state heat transfer and then applied as thermal loads in a subsequent thermal stress analysis. Two subcases are required. The first defines the temperature loads, boundary conditions, and output requests for the heat transfer analysis and the second subcase defines the thermal loads, boundary conditions, and output requests for the thermal stress analysis. Thermal loads in the second subcase are requested through the command

TEMP(LOAD) = Heat Transfer Subcase ID

If this default is not acceptable, then in heat transfer subcase add the Case Control word TSTRU=SID and in structures subcase here

TEMP(LOAD) = SID

See the Case Control command, [TSTRU, 500](#). PARAM,NESET is no longer used. HEATSTAT not supported for p-elements.

SOL 101 INPUT FILE: t_sph_expand2.dat

```

SOL      101
CEND
TITLE = TEMPERATURES AND STRESSES IN A SPHERE
ECHO=NONE
SUBCASE 1
  SUBTITLE   = HEAT TRANSFER ANALYSIS
  SPC        = 1
  THERMAL    = ALL
SUBCASE 2
  SUBTITLE   = STRUCTURAL ANALYSIS
  TEMP (LOAD) = 1
  DISPLACEMENT = ALL
  STRESS     = ALL
  GPSTRESS   = ALL
  STRFIELD   = ALL
  
```

```

BEGIN BULK
INCLUDE 'sph_model.txt'
MAT1    100    30.+6      .33      6.5-6    60.
MAT4    100    .6
PARAM, HEATSTAT, YES
PARAM   POST    -1
PSOLID  100    100
SPC     1      1000      1      160.      1001      1      160.
SPC     1      1002      1      160.      1003      1      160.
SPC     1      2000      1      160.      2001      1      160.
SPC     1      2002      1      160.      2003      1      160.
SPC     1      3000      1      160.      3001      1      160.
SPC     1      3002      1      160.      3003      1      160.
SPC     1      4000      1      160.      4001      1      160.
SPC     1      4002      1      160.      4003      1      160.
SPC1    1      1      1050    THRU    1053
SPC1    1      1      2100    THRU    2103
SPC1    1      1      3200    THRU    3203
SPC1    1      1      4400    THRU    4403
ENDDATA
  
```

SOL 101 INPUT FILE: sph_model.txt

- Following is a portion of the model nodes

GRID	1000	1	1.	5.	0.0	1	23456
GRID	1001	1	1.	5.	90.	1	23456
GRID	1002	1	1.	5.	270.	1	23456
GRID	1003	1	1.	5.	180.	1	23456
GRID	1010	1	1.1487	5.	0.0	1	23456
GRID	1011	1	1.1487	5.	90.	1	23456
GRID	1012	1	1.1487	5.	270.	1	23456
GRID	1013	1	1.1487	5.	180.	1	23456
GRID	1020	1	1.3195	5.	0.0	1	23456
GRID	1021	1	1.3195	5.	90.	1	23456
GRID	1022	1	1.3195	5.	270.	1	23456
GRID	1023	1	1.3195	5.	180.	1	23456
GRID	1030	1	1.5157	5.	0.0	1	23456
GRID	1031	1	1.5157	5.	90.	1	23456
GRID	1032	1	1.5157	5.	270.	1	23456
GRID	1033	1	1.5157	5.	180.	1	23456
GRID	1040	1	1.7411	5.	0.0	1	23456
GRID	1041	1	1.7411	5.	90.	1	23456
GRID	1042	1	1.7411	5.	270.	1	23456
GRID	1043	1	1.7411	5.	180.	1	23456
GRID	1050	1	2.	5.	0.0	1	23456
GRID	1051	1	2.	5.	90.	1	23456
GRID	1052	1	2.	5.	270.	1	23456
GRID	1053	1	2.	5.	180.	1	23456

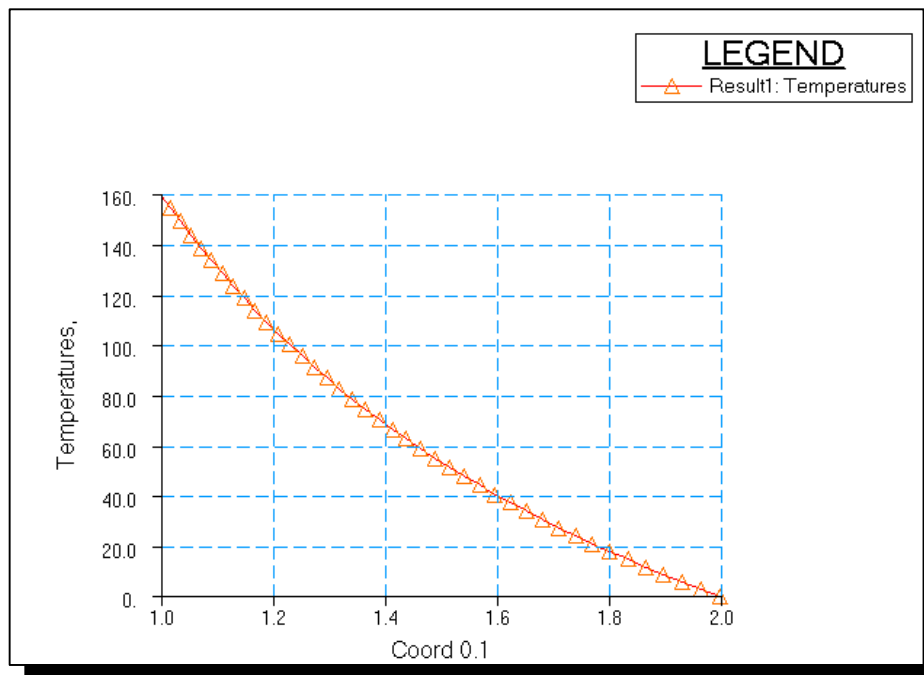
SOL 101 INPUT FILE: sph_model.txt

- Following is a portion of the model elements

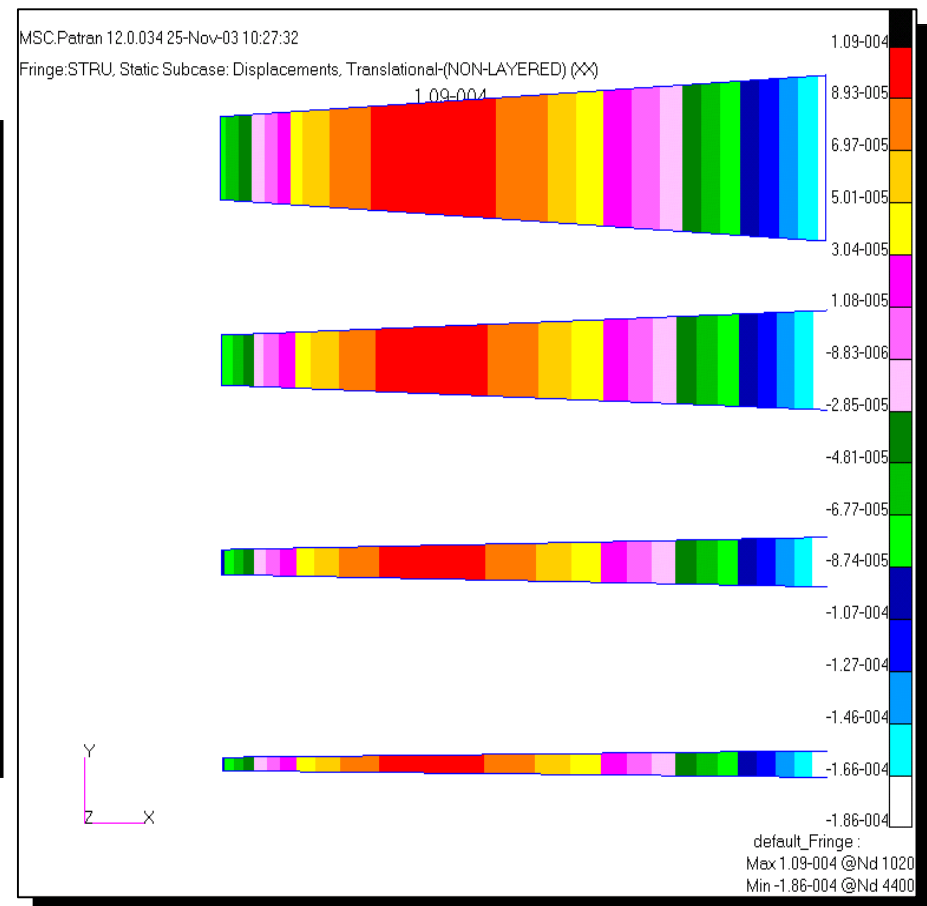
CHEXA	101	100	1000	1001	1003	1002	1010	1011	+M00000
+M00000	1013	1012							
CHEXA	102	100	1010	1011	1013	1012	1020	1021	+M00001
+M00001	1023	1022							
CHEXA	103	100	1020	1021	1023	1022	1030	1031	+M00002
+M00002	1033	1032							
CHEXA	104	100	1030	1031	1033	1032	1040	1041	+M00003
+M00003	1043	1042							
CHEXA	105	100	1040	1041	1043	1042	1050	1051	+M00004
+M00004	1053	1052							
CHEXA	201	100	2000	2001	2003	2002	2010	2011	+M00005
+M00005	2013	2012							
CHEXA	202	100	2010	2011	2013	2012	2020	2021	+M00006
+M00006	2023	2022							
CHEXA	203	100	2020	2021	2023	2022	2030	2031	+M00007
+M00007	2033	2032							
CHEXA	204	100	2030	2031	2033	2032	2040	2041	+M00008
+M00008	2043	2042							

THERMAL STRESS, SINGLE SOL 101

- Results from structural analysis



Temperature vs Coord 0.1



Displacement Fringe



THERMAL STRESS ANALYSIS USING MD NASTRAN SOL 400





MD SOL 400 THERMAL STRESS ANALYSIS

- This procedure is used for a nonlinear SS heat transfer and nonlinear static structural analysis

NL SS thermal => NL static structural

- The analysis is performed using only one subcase, and two steps
 - STEP 1 (is for nonlinear steady-state heat transfer analysis)
 - ANALYSIS = HSTAT
 - Define thermal LBCs
 - Specify NLPARM for thermal analysis entry
 - Specify TSTRU = n
 - STEP 2 (is for nonlinear static structural analysis)
 - ANALYSIS = NLSTAT
 - Define structural LBCs
 - Specify NLPARM for structural analysis entry
 - Specify TEMP(LOAD) = n

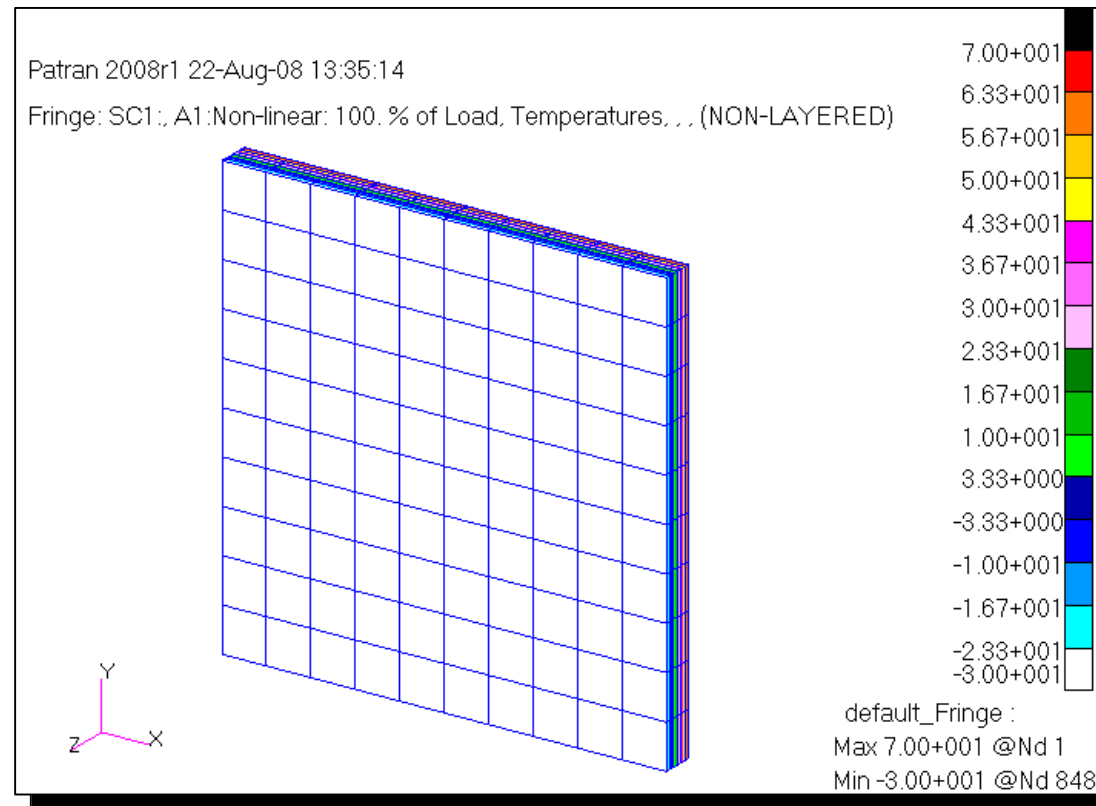
MD SOL 400 THERMAL STRESS ANALYSIS

- The resulting computed temperatures from STEP 1 are automatically mapped internally in MD Nastran as the temperature load for STEP 2.
- The Case Control command TSTRU = n, in STEP 1, will prepare for the temperature results to be used for STEP 2 with TEMP(LOAD) = n.
- TEMP(INIT) = m references the initial stress free temperature.
- There is no need for the punch file or required Patran mapping as Field for the temperature load using SOL 400.

```
SOL 400
CEND
TITLE = STRUCTURE RUN WITH THERMAL LOAD
ECHO = NONE
TEMP(INIT) = 1
SUBCASE 1
STEP 1
  ANALYSIS = HSTAT
  TITLE=This is a default subcase.
  SPC = 1
  THERMAL(SORT1,PRINT,punch)=ALL
  FLUX(SORT1,PRINT)=ALL
  NLPARM = 9
  TSTRU = 5
STEP 2
  ANALYSIS = NLSTAT
  SPC = 2
  TEMP(LOAD) = 5
  DISP =ALL
  STRESS=ALL
  NLPARM = 11
BEGIN BULK
  TEMPD,1,-30.0
  NLPARM,9,1,,FNT,1,,PW
  NLPARM,11,1,,FNT
```

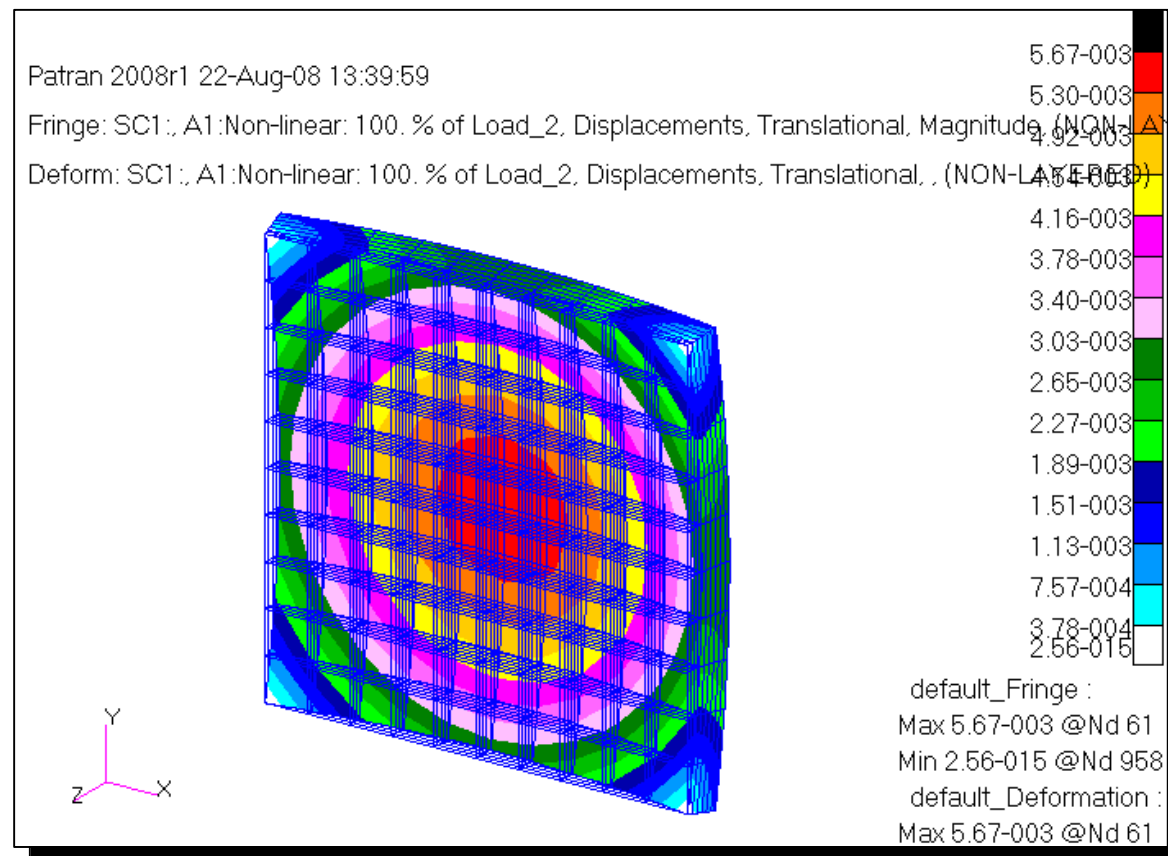
MD SOL 400 THERMAL STRESS ANALYSIS

- MD Nastran SOL 400 example file, ws16_chain.dat
- STEP 1, ANALYSIS = HSTAT



MD SOL 400 THERMAL STRESS ANALYSIS

- STEP 2, ANALYSIS = NLSTAT



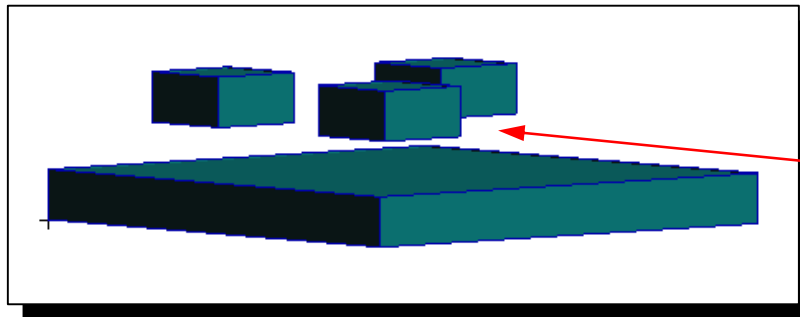
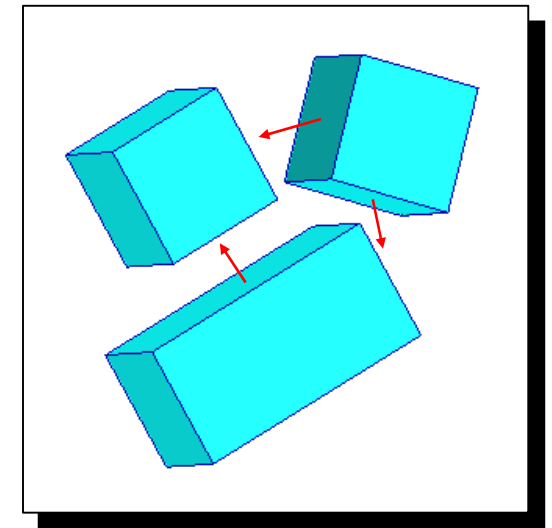
THERMAL CONTACT





THERMAL CONTACT

- Contact between bodies allows transfer of thermal energy.
 - Conduction
 - Convection
 - Radiation
- An example of this is three chips in contact with a PWB.

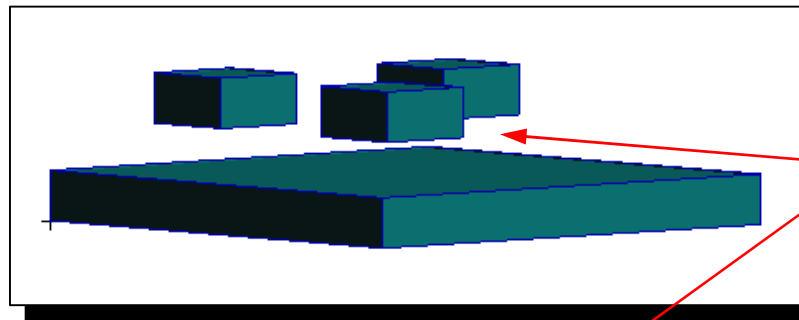


Layer of epoxy

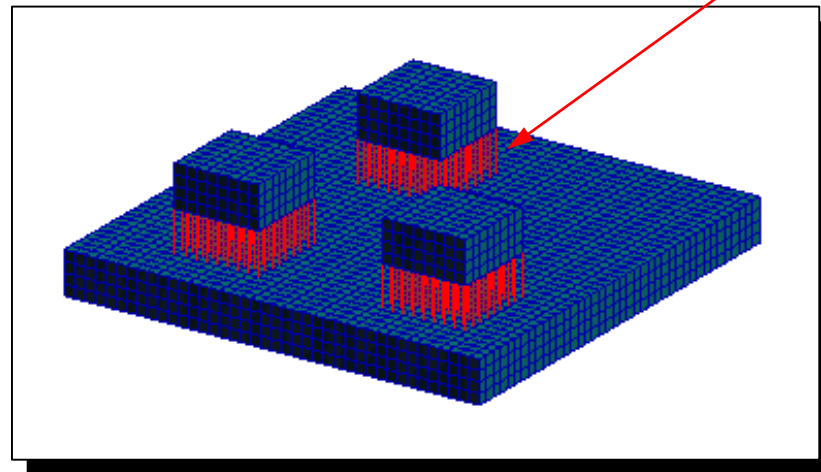
- Thermal energy may be transmitted by a thin layer of epoxy or solder.

THERMAL CONTACT

- The layer of bonding material can be modeled using
 - Solid finite elements, or
 - Convection



Solid elements
or convection

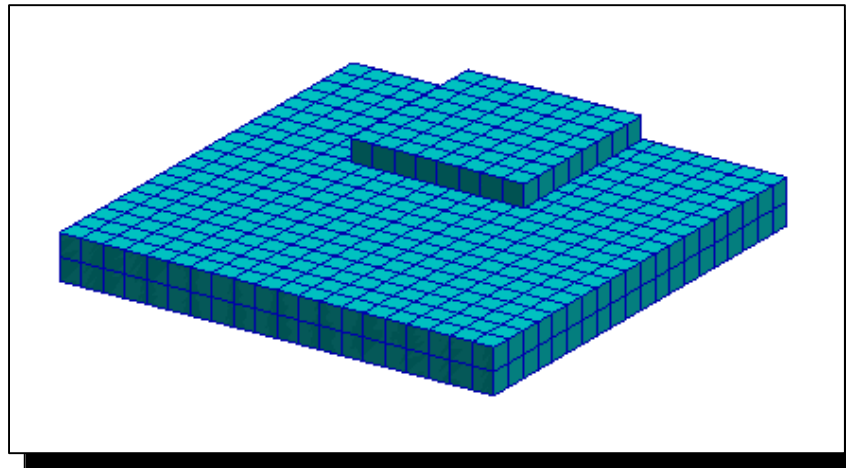


THERMAL CONTACT

- This can be done either manually, or a contact table can be used. This topic covers using a contact table, Analysis: Analyze/Subcases.../Subcase Parameters.../Contact Table....

THERMAL CONTACT

- A simple example with one chip and a PWB is used to demonstrate how to use a contact table.
- First, create the geometry, and mesh it.



- Second, create two contact bodies using Loads/BCs:
Create/Contact/Element Uniform, Option: Deformable Body

THERMAL CONTACT

- New Set Name: chip
- Input Data...
 - Use the default data
- Select Application Region...
 - Select Solid 2
- Apply
- Repeat this for the pwb, using Solid 1

Select Application Region

Select: Geometry ▾

Application Region

Select Solids

Add Remove

Master Region

Solid 2

OK

Load/Boundary Conditions

Action: Create ▾

Object: Contact ▾

Type: Element Uniform ▾

Option: Deformable Body ▾

Current Load Case: Default...

Type: Static

Existing Sets

New Set Name: chip

Target Element Type: 3D ▾

Input Data...

Select Application Region...

-Apply-

THERMAL CONTACT

- Analysis
 - Solution Type
 - SOL 153
 - Subcases
 - Subcase Parameters
 - Contact Table

Solution Type

MSC.Nastran
Solution Type

Solution Type:

☒ STEADY STATE ANALYSIS

☐ TRANSIENT ANALYSIS

☐ STRUCTURAL HEAT ANALYSIS

Subcase Parameters

Static Nonlinear Iterations
Number of Load Increments =
10

Matrix Update Method: Automatic

Number of Iterations per Update =
5

Allowable Iterations per Increment =
25

Convergence Criteria

☐ Temperature Error

Temperature Tolerance = 1.0e-03

☐ Load Error

Load Tolerance = 1.0e-03

☐ Work Error

Work Tolerance = 1.0e-07

Contact Table...

Subcases

Solution Sequence: 153

Action: Create

Available Subcases

Default

Subcase Name

Available Load Cases

Default

Subcase Options

Subcase Parameters...

Analysis

Action: Analyze

Object: Entire Model

Method: Full Run

Code: MSC.Nastran

Type: Thermal

Available Jobs

Job Name
pwb_with_chip

Job Description (TITLE)
MSC.Nastran job created on
30-Jul-08 at 16:27:23

SUBTITLE

LABEL

Translation Parameters...

Solution Type...

Direct Text Input...

Select Superelements...

Subcases...

THERMAL CONTACT

- Contact table.
- The convection and radiation (not used for this example) data is input under Contact Table Properties.

Contact Table

Parameters Defining Contact Between Bodies

Global Contact Detection: Default (by body #)

☐ View Table Only

Touch All Glue All Deactivate All Import/Export Select Existing

Contact Matrix

	Body Type	Release	1	2
1-chip	Deformable	N		T
2-pwb	Deformable	N	T	

Specific Parameters for Body Pairs

Touching Body: chip, pwb

Touched Body: chip, pwb

Force Removal: ☒ Immediate ☐ Gradual

Contact Detection: ☐ Automatic ☐ 1st->2nd ☐ Double Sided ☐ 2nd->1st

Contact Table Properties:

Distance Tolerance: 0.600000 Field: 0.

Bias Factor: 0. Field: 0.

Near Contact Dist. Tol.: 3. Field: 0.

Analysis Properties: Thermal

Heat Transfer Coeff.: 1.2

Near Contact Heat Trf Coeff.: 0.

Distance Dep. Conductivity: 0.

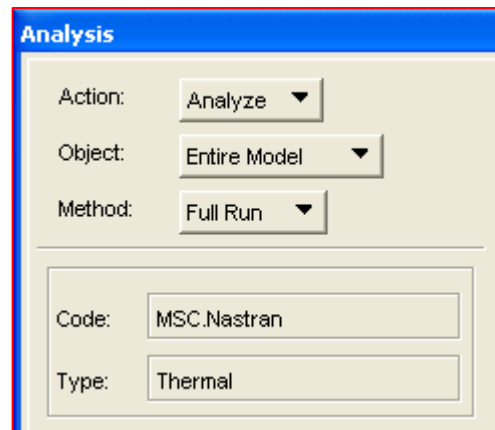
Natural Convection Exp.: 0.

Surface Emissivity: 0.

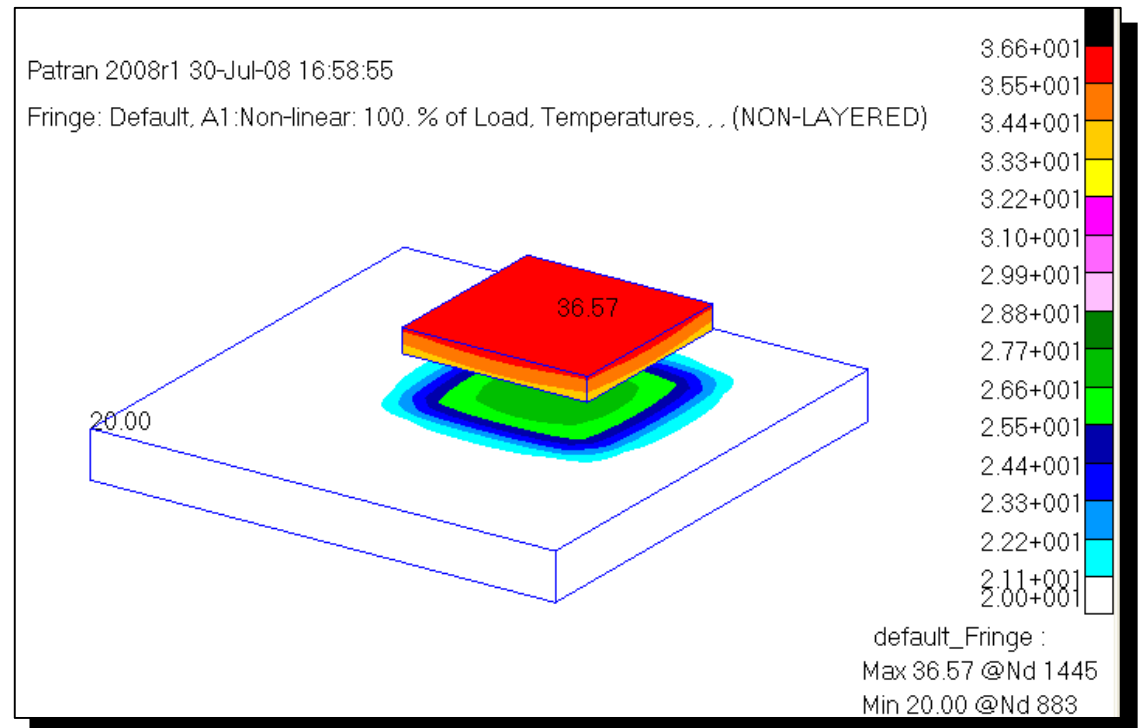
Distance Dep. Conv. Coeff.: 0.

THERMAL CONTACT

- Perform the analysis

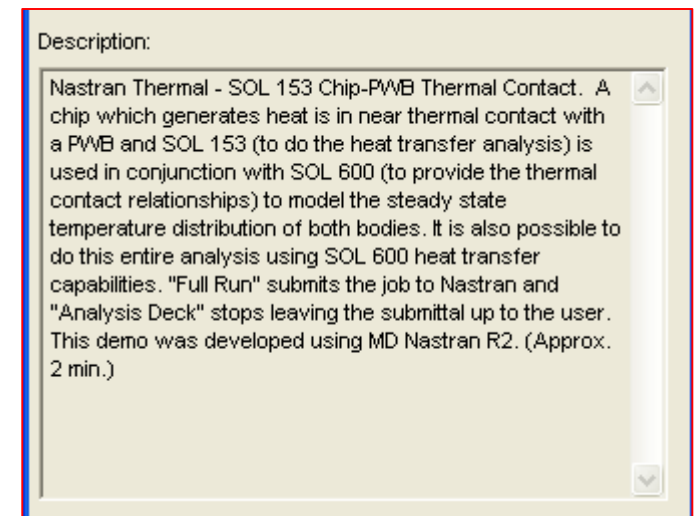
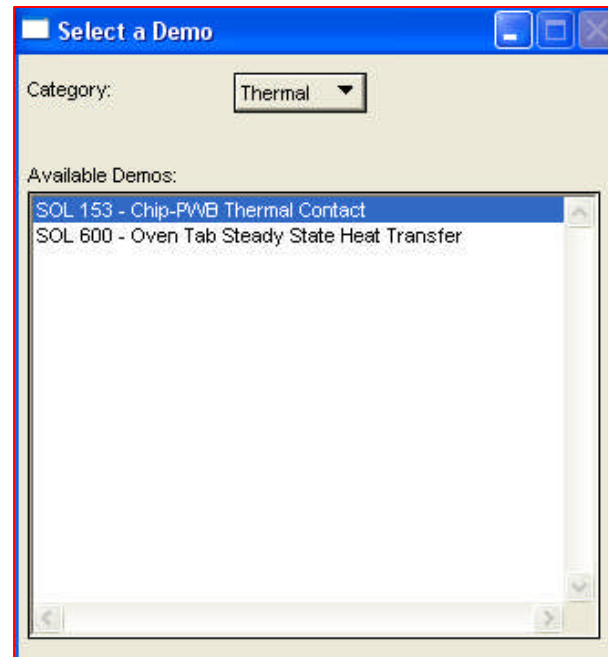
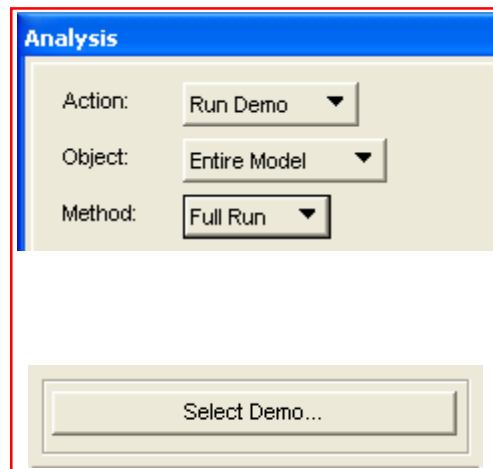


- Display the temperature results



THERMAL CONTACT

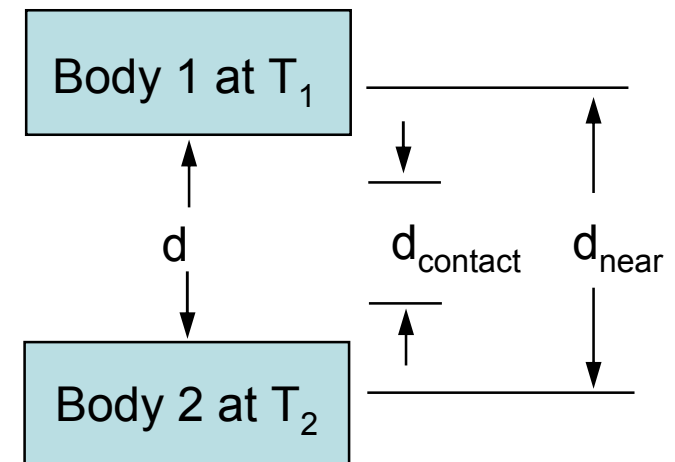
- This simple problem is one from the set of Patran Demo problems



- Or, play session file thermal_contact.ses

THERMAL CONTACT ALGORITHM

- There are three separation conditions for thermal contact
 - First condition $d < d_{contact}$
 - Conduction (only) between the two bodies
- Second condition $d_{contact} < d < d_{near}$
 - A small gap exists between the two bodies
 - Convection, natural convection, and radiation
- Third condition $d_{near} < d$
 - Heat flux to the environment
 - Convection and radiation



CONTACT TABLE PARAMETERS

- Select general parameters
 - Body Type – deformable or rigid body
 - Deformable – finite element w/ properties and thermal analysis
 - Rigid body – constant temp, or finite elements and thermal analysis
 - Release – yes or no; specify no if to keep the bodies in contact
 - Cell (i,j) – the i,j cell of the matrix; values are T (touch), G (glued), or blank (no contact). Rows, Touching Body. Columns, Touched Body.
- Body Pairs
 - Touching Body and Touched Body – used to see what bodies an active cell references; also, used to see what the Contact Table Properties values are for a contact pair.

CONTACT TABLE PARAMETERS

- Contact Table Properties
 - Distance Tolerance – if the distance between the bodies is less than this, they are considered as contacting. This is d_{contact} on a previous page.
 - Bias Factor – bias (+ or -) the Distance Tolerance domain.
 - Near Contact Dist. Tol – if the distance between the bodies is greater than this, the heat flux is considered as being to the environment. This is d_{near} on a previous page.

CONTACT TABLE PARAMETERS

- Analysis Properties

- Heat Transfer Coeff – this is h for convection
- Near Contact Heat Trf Coeff – is the convection coefficient for $d_{\text{contact}} < d < d_{\text{near}}$.
- Distance Dep. Conductivity -- separation distance dependent heat transfer conductivity for $d_{\text{contact}} < d < d_{\text{near}}$. Its expression is the coefficient times $(1 - d/d_{\text{near}})$ times the contact temperature difference.
- Natural Convection Exp – the exponent used for natural convection
- Surface Emissivity – the emissivity of the radiating surfaces
- Distance Dep. Conv. Coeff – separation distance dependent heat transfer coefficient for $d_{\text{contact}} < d < d_{\text{near}}$. Its expression is the coefficient times the ratio d/d_{near} times the contact temperature difference.



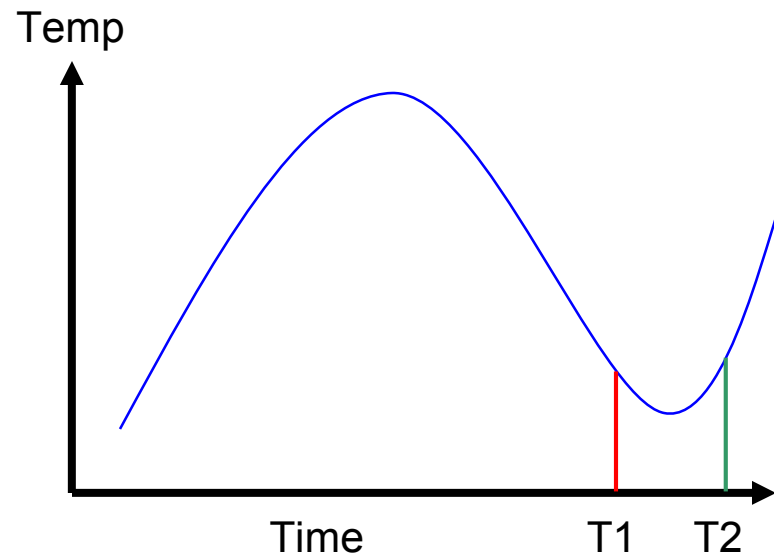
RESTART TRANSIENT THERMAL ANALYSIS





RESTART TRANSIENT THERMAL ANALYSIS

- Restarting causes the transient thermal analysis to restart/continue at a specific time.



It is possible to restart the job at time T1 to change the loading or use a smaller time step for the next load case. Solve the thermal problem for the time interval $T1 \leq \text{Time} \leq T2$.

RESTART TRANSIENT THERMAL ANALYSIS

- For the initial run (used to be called “coldstart” run)
 - Save the .MASTER and .DBALL files by adding SCRATCH = NO to the MD Nastran command line
 - NASTRAN stat1 SCRATCH = NO
 - SOL 159
- For performing a restart run, use the following commands in the FMS and Case Control section
 - RESTART VERSION = LAST, KEEP
 - ASSIGN MASTER = ‘stat1.MASTER’
 - SOL 159
 - CEND
 - PARAM, STIME, T1
 - PARAM, LOOPID, n
 - ...
 - BEGIN BULK
 - ...



RESTART TRANSIENT THERMAL ANALYSIS

The nastran Command

MD Nastran is executed with a command called nastran. (Your system manager may assign a different name to the command.) The nastran command permits the specification of keywords used to request options affecting MD Nastran job execution. The format of the nastran command is:

```
nastran input_data_file [keyword1 = value1 keyword2 = value2 ...]
```

where input_data_file is the name of the file containing the input data, and keyword_i = value_i is one or more optional keyword assignment argument(s). For example, to run a job using the data file example1.dat, enter the following command:

```
nastran example1
```

scratch={yes|no|mini} Default=no

Deletes the database files at the end of the run. If the database files are not required, “scratch=yes” can be used to remove them, thereby preventing cluttering of the directory with unwanted files. If “mini” is specified, a reduced size database (that can only be used for data recovery restarts) will be created. See [Database Concepts](#) (p. 513) in the *MSC Nastran Reference Manual* for further details on the “mini” database. If scratch=post is specified, a reduced size database intended for use by Patran or the toolkit will be created. Scratch=post also performs the actions of NASTRAN INDEX=19.

Example: nastran example scratch=yes

All database files created by the run are deleted at the end of the job in the same way as the FMS statement INIT MASTER(S).

RESTART TRANSIENT THERMAL ANALYSIS

RESTART Reuses Database From a Previous Run

Requests that data stored in a previous run be used in the current run.

Format:

RESTART $\left[\text{PROJECT} = \text{'project'} \text{ VERSION} = \left[\begin{array}{c} \text{version-ID} \\ \text{LAST} \end{array} \right] \left[\begin{array}{c} \text{KEEP} \\ \text{NOKEEP} \end{array} \right] \text{ LOGICAL} = \text{dbname} \right]$

Examples:

1. RESTART VERSION=7

Version number 7 will be retrieved for this run (version 8). At the end of the run, version 7 will be deleted.

2. PROJ='FENDER'
RESTART

The last version under project-ID FENDER will be used in the current run.

3. ASSIGN RUN1='run1.MASTER'
RESTART LOGICAL=RUN1

The run1.MASTER and its associated database will be used (read only) for restart purposes.

RESTART TRANSIENT THERMAL ANALYSIS

ASSIGN

Assigns Physical File

Assigns physical file names or other properties to DBset members or special FORTRAN files that are used by other FMS statements or DMAP modules. Also, assigns physical name and/or other properties to modal neutral files (.mnf) for MD Nastran/ADAMS interface.

Format 1: Assign a DBset member name

```
ASSIGN log-name= $\begin{bmatrix} = \text{'filename1'} \\ = * \\ = '**' \end{bmatrix}$  [TEMP] [DELETE] [SYS='sys-spec']
```

Examples:

1. Assign the DBALL DBset:

```
ASSIGN DB1='filename of member DB1'  
INIT DBALL LOGI=(DB1)
```

RESTART TRANSIENT THERMAL ANALYSIS

STIME

Default = 0.0 (SOLs 109, 112, 129 and 159 only)

In restarts from previous transient analysis runs, the user provides $STIME = t_N$ where t_N is the last time step of the subcase to be continued with a new or changed subcase in the new run. Thus, the loading and printout will start from t_N as though the original run contained the new subcase data.

In SOLs 109 and 112 restarts from previous SOLs 109 and 112 runs, STIME is used to specify the proper starting time of the restart run. If STIME exceeds the last output time of the previous run, the starting time is assumed to be the last output time. Otherwise, the starting time is assumed to be the output time of the previous run (not necessarily the last output time) that is closest to STIME. In other words, the starting time of the restart run need not be the last output time of the previous run, but may be any time earlier than that. The program informs the user that it is a restart run and indicates the starting time (determined as above) that is used for the restart run.

In SOLs 109 and 112 restarts, the user must ensure that the model and the constraints as well as the subcase setup in the restart run are the same as those in the previous run. The user may, however, specify different TSTEP and DLOAD requests in Case Control and also different TSTEP and dynamic loading entries in the Bulk Data compared to the previous run. The loading and the results output from the restart run will start from the new starting time.

Note: That the responsibility of ensuring that the model and the constraints as well as the subcase setup in the restart run are the same as those in the previous run is left to the user; the program does not check for this condition. If this condition is not met, the program may terminate the execution with a fatal error or give erroneous results.

RESTART TRANSIENT THERMAL ANALYSIS

LOOPID

Default = 0

LOOPID defines the desired loop number for initial conditions in a restart of SOLs 106, 129, 153, and 159. By default in SOLs 106 and 153 the restart proceeds from the last loop ID of the subcase defined by SUBCASID or SUBID. In SOLs 106, and 153 PARAM, SUBID or SUBCASID may also be specified.

RESTART TRANSIENT THERMAL ANALYSIS

- Initial run (“coldstart” run)
 - SOL 159

Extracted from MD Nastran model .DAT file

```
NASTRAN sol159 SCR = NO
ID stress2,msc
SOL 159
TIME 10
CEND
TITLE=Use the include punch file for stress calculation
IC=23
SUBCASE 1
  LABEL=heat transfer analysis
  ANALYSIS=HEAT
  TSTEPNL=400
  SPC=10
  DLOAD=35
  THERMAL (PUNCH,PRINT, SORT1)=ALL
  FLUX=ALL
BEGIN BULK
PARAM,CURVPLOT,1
INCLUDE 'run1.bulk'
ENDDATA
```

RESTART TRANSIENT THERMAL ANALYSIS

- Restart run
 - SOL 159

Extracted from MD Nastran model .DAT file

```
$The following is the restart run that request the analysis starting at time equals
$18.0 and from loopid equals to 1
$*****
RESTART version=1 keep
ASSIGN MASTER='sol159.MASTER'
ID dcc,msc
SOL 159
TIME 10
CEND
TITLE= Using a RESTART to continue the transient run from previous converged time step
IC=23
$ Select the last stime from the previous transient run
PARAM,STIME,18.0
PARAM,LOOPID,1
THERMAL(PUNCH,PRINT)=ALL
FLUX=ALL
ANALYSIS=HEAT
SPC=10
DLOAD=35
SUBCASE 1
    LABEL=coldstart
    TSTEPNL=400
SUBCASE 2
    LABEL=restarting from time equal to 18.0
    TSTEPNL=500
BEGIN BULK
TSTEPNL      500          30          2.0          1          ADAPT
ENDDATA
```

RESTART TRANSIENT THERMAL ANALYSIS

- Initial run output

Extracted from MD Nastran ASCII Results File

```

0                                NON - LINEAR  I T E R A T I O N   M O D U L E   O U T P U T

      STIFFNESS UPDATE TIME      0.10 SECONDS                                SUBCASE      1
      ITERATION TIME            0.00 SECONDS

      - - - - - ERROR FACTORS - - - CONVERGENCE FACTORS - - - LINE SEARCH DATA - - - ITR MAT NO.
TIME  ITER  DISP.  LOAD      WORK      LAMBDA(I)  DLMAG    FACTOR  E-FIRST  E-FINAL  NQNV  NLS  DIV  DIV  BIS  ADJUST

1.000000E+00  1  1.0000E+00  2.1549E-04 -7.8470E-07 9.9900E-01  3.1246E-04  1.000 -7.325E-05 -7.325E-05  0  0  0  1  1.000
2.000000E+00  1  1.0000E+00  2.0770E-04 -6.9002E-07 9.9900E-01  3.0074E-04  1.000 -8.707E-05 -8.707E-05  0  0  0  1  1.000
*** USER INFORMATION MESSAGE 6204 ***          0.000 SECONDS REQUIRED TO DECOMPOSE MATRIX.
4.000000E+00  2  1.1926E-10  3.4152E-08 -2.1482E-14 8.6665E-05  4.9383E-08  1.000 -3.922E-05 -3.922E-05  1  0  0  1  2.000
6.000000E+00  1  1.0000E+00  3.3255E-05 -4.7085E-08 9.9900E-01  4.9571E-05  1.000 -5.211E-06 -5.211E-06  1  0  0  1  2.000
*** USER INFORMATION MESSAGE 6204 ***          0.000 SECONDS REQUIRED TO DECOMPOSE MATRIX.
1.000000E+01  2  1.1672E-10  1.6149E-08  2.3121E-14 2.5601E-05  2.4818E-08  1.000  7.046E-06  7.046E-06  1  0  0  1  4.000
1.400000E+01  1  1.0000E+00  4.8324E-05 -9.9568E-08 9.9900E-01  7.5365E-05  1.000 -8.385E-06 -8.385E-06  1  0  0  1  4.000
1.800000E+01  1  1.0000E+00  8.8488E-05 -2.5555E-07 9.9900E-01  1.3741E-04  1.000 -2.477E-05 -2.477E-05  1  0  0  1
0*** USER INFORMATION MESSAGE 4550,
*** NEW STIFFNESS MATRIX IS REQUIRED ***

^^^ LAST ADJUSTED DELTA T: OLDDT=      4.000000E+00
^^^ USER INFORMATION MESSAGE 9003 (NLTRAN)  CURRENT VALUE OF STIME IS      1.800000E+01
^^^ USER INFORMATION MESSAGE 9005 (NLTRAN) -  THE SOLUTION FOR LOOPID=      1 IS SAVED FOR RESTART.
    
```

RESTART TRANSIENT THERMAL ANALYSIS

- Restart run output

Extracted from MD Nastran ASCII Results File

0 NON - LINEAR ITERATION MODULE OUTPUT															
STIFFNESS UPDATE TIME			0.12 SECONDS			SUBCASE			2						
ITERATION TIME			0.00 SECONDS												

RESTART TRANSIENT THERMAL ANALYSIS

- Output for both initial and restart run

```
POINT-ID =          1

  T E M P E R A T U R E    V E C T O R

      TIME                TYPE          VALUE
0.0                      S          1.000000E+02
1.000000E+00             S          1.014250E+02
2.000000E+00             S          1.026051E+02
4.000000E+00             S          1.044845E+02
6.000000E+00             S          1.059412E+02
1.000000E+01             S          1.082583E+02
1.400000E+01             S          1.101837E+02
1.800000E+01            S          1.119725E+02
2.000000E+01            S          1.128452E+02
2.200000E+01             S          1.137137E+02
3.000000E+01             S          1.172198E+02
3.800000E+01             S          1.208462E+02
4.600000E+01             S          1.246572E+02
5.400000E+01             S          1.282932E+02
6.200000E+01             S          1.312252E+02
7.000000E+01             S          1.335425E+02
7.800000E+01             S          1.355272E+02
```

Extracted from MD Nastran
ASCII Results File

RESTART FROM THERMAL ANALYSIS TO STRESS ANALYSIS



RESTART FOR THERMAL STRESS ANALYSIS

- Instead of using a punch file to transition from heat transfer to stress/displacement analysis, it is possible to use restart for a heat transfer analysis.

RESTART FOR THERMAL STRESS ANALYSIS

- Initial run (coldstart run)
 - SOL 153
 - Stress1.dat

Extracted from MD Nastran model .DAT file

```
ID stress2,msc
SOL 153
TIME 10
CEND
TITLE=SEMI-INFINITE RECTANGULAR FIN
TEMP(init)=23
SUBCASE 1
  LABEL=heat transfer analysis
  ANALYSIS=HEAT
  NLPARM=300
  SPC=10
  LOAD=33
  THERMAL=ALL
  FLUX=ALL
BEGIN BULK
INCLUDE 'run1.bulk'
```

RESTART FOR THERMAL STRESS ANALYSIS

- Restart run
 - SOL 153
 - Stress1_r.dat

```
RESTART VERSION=1 KEEP  
ASSIGN MASTER='stress1.MASTER'
```

```
ID stress2,msc
```

```
SOL 153
```

```
TIME 10
```

```
APP COUPLED
```

```
CEND
```

```
TITLE=SEMI-INFINITE RECTANGULAR FIN
```

```
TEMPERATURE (INITIAL)=23
```

```
$ Look at the output from the previous thermal run and obtain the loopid.
```

```
PARAM,LOOPID,1
```

```
PARAM,SUBID,2
```

```
SUBCASE 1
```

```
    LABEL=heat transfer analysis
```

```
    ANALYSIS=HEAT
```

```
    NLPARM=300
```

```
    SPC=10
```

```
    LOAD=33
```

```
    THERMAL=ALL
```

```
    FLUX=ALL
```

```
SUBCASE 2
```

```
    LABEL=thermal stress analysis
```

```
    ANALYSIS=STRUCTURE
```

```
    TEMP (LOAD)=777
```

```
    NLPARM=300
```

```
    SPC=20
```

```
    DISP=ALL
```

```
    STRESS=ALL
```

```
    ELFORCE=ALL
```

Extracted from MD Nastran model .DAT file

Add the second subcase for stress/displacement analysis.

RESTART FOR THERMAL STRESS ANALYSIS

- Add only the structure material properties and the structural constraints. The automatic restart logic will simply add the records from the database.

Extracted from MD Nastran model .DAT file

BEGIN	BULK						
mat1	10	10.0+6		0.33		6.5-6	60.0
spc1	20	12	1	11	10	20	
spc1	20	0	100				
grdset							3456
ENDDATA							

RESTART FOR THERMAL STRESS ANALYSIS

- MD Nastran results
 - Stress1_r.f06

Extracted from MD Nastran ASCII Results File

```
0    THERMAL STRESS ANALYSIS
SUBCASE 2
LOAD STEP = 2.000000E+00

DISPLACEMENT VECTOR

POINT ID.  TYPE  T1    T2    T3    R1    R2    R3
1          G    0.0    0.0    0.0    0.0    0.0    0.0    0.0
2          G  1.082767E-03  1.285110E-03  0.0    0.0    0.0    0.0
3          G  1.941993E-03  1.254213E-03  0.0    0.0    0.0    0.0
4          G  2.349874E-03  1.486964E-03  0.0    0.0    0.0    0.0
5          G  2.460323E-03  1.613035E-03  0.0    0.0    0.0    0.0
6          G  2.318849E-03  1.716098E-03  0.0    0.0    0.0    0.0
7          G  1.998447E-03  1.828509E-03  0.0    0.0    0.0    0.0
8          G  1.507600E-03  1.747243E-03  0.0    0.0    0.0    0.0
9          G  1.070955E-03  2.348248E-03  0.0    0.0    0.0    0.0
```

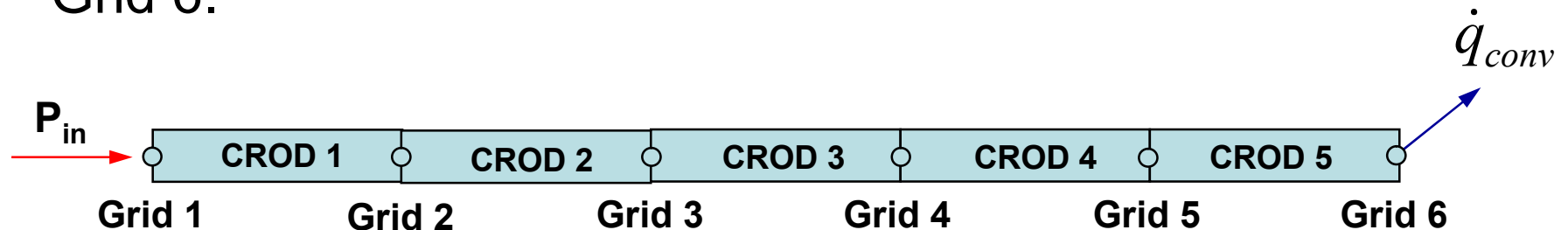


NOLINi FOR THERMOSTAT CONTROL



NOLIN3 FOR THERMOSTAT CONTROL

- The temperature at the end of the rod element structure at Grid 6 will be controlled using a NOLIN3 function and an MPC.
- There is a convective loss to the ambient environment at Grid 6.



- When the temperature at Grid 6 drops below $< 100\text{ }^{\circ}\text{C}$, heating occurs at the opposite end, Grid 1, using P_{in} .
- When the temperature at Grid 6 is $> 100\text{ }^{\circ}\text{C}$, the heating (using P_{in}) is stopped.

NOLIN3 FOR THERMOSTAT CONTROL

- P_{in} is defined as

$$P_{in} = \begin{cases} 50000 & , T_{50} > 0 \\ 0 & , T_{50} \leq 0 \end{cases}$$

- where T_{50} is defined as

$$T_{50} = 100 - T_6$$

- The NOLIN3 function is defined as

$$P_i = \begin{cases} S \cdot (X_j)^A & , X_j > 0 \\ 0 & , X_j \leq 0 \end{cases}$$

NOLIN3 FOR THERMOSTAT CONTROL

- For this case, Grid i = 1 and Grid j = 50
- \$NOLIN3, SID, GI, CI, S, GJ, CJ, A
NOLIN3, 300, 1, 1, 50000.0, 50, 1, 0.0
- See Example 10 in the MD Nastran 2002 Thermal Analysis User's Guide.

NOLIN1 FOR POWER INPUT

- $P_i(t) = S \cdot T(X_j(t))$
- This shows that the power into Grid i depends on the temperature at Grid j through a table T (TABLEDk)
- In this case, Grid i=1 and Grid j=6

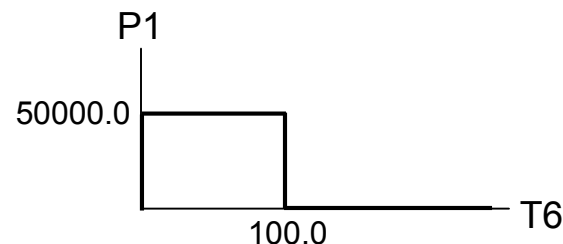
```
$NOLIN1,SID,GI,CI,S,GJ,CJ,TABLE
```

```
NOLIN1,300,1,1,1.0,6,1,123
```

```
TABLED1,123,,,,,,,,+,
```

```
+,0.0,5.0E4,100.0,5.0E4,100.5,0.0,300.0,0.0,+,
```

```
+,ENDT
```



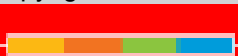
MPC RELATIONSHIP

- $A1*T1 + A2*T2 + ... + An*Tn = 0$
- $T99 = (T1+T2+T3+T4)/4$
where T99 is a dependent node based on the above relationship
 $4.0*T99-T1-T2-T3-T4=0$
\$MPC,SID,G1,C1,A1,G2,C2,A2,,+
\$+,,G3,C3,A3,G4,C4,A4,,+
MPC,10,99,1,4.0,1,1,-1.0,,+
+,,2,1,-1.0,3,1,-1.0,,+
+,,4,1,-1.0

MPC EXAMPLE

- Example 1d in the MD Nastran Thermal User's Guide.
 - $T_{\text{film node}} = (T1 + T2 + T_{\text{amb}}) / 3$

USE OF TABLEMi



TABLEM2

- $y = z * y_T(x - X1)$
- $x - X1$ is input to the table, TABLEM2.
- $X1$ is the shift value for the table.
- z is the scale multiplier from the MAT4 bulk data card.
- y is returned, using the above shown algorithm.
- Suppose, the temperature used in the analysis has unit degree F, and the table argument value is in degree R, then the $X1$ value = -459.67.

TABLEM2 (Cont.)

- Analysis

	TABLE*	X1
degree F	degree R	-459.67
degree C	degree K	-273.15
degree R	degree F	+459.67
degree K	degree C	+273.15

* Table argument

TABLEM3

- $y = z * y_T((x-X1)/X2)$
 - $(x-X1)/X2$ is input to the table, TABLEM3.
 - X1 and X2 are the table parameters.
 - z is the scale multiplier from the MAT4 bulk data entry.
 - y is returned, using the above shown algorithm.
-
- Take the analysis temperature to be unit degree F.
 - The data in TABLEM3 is of unit Watt/m/C; it is needed to obtain data for unit Btu/hr/ft/F.
 - Note, 1 Watt/m/C = 0.5778 Btu/hr/ft/F.

TABLEM3 (Cont.)

- $T_F = 1.8 * T_C + 32$
 $T_C = (T_F - 32) / 1.8$
- $y = z * y_T((x - X1) / X2)$
Set $X1 = 32.0$ and $X2 = 1.8$
- TABLEM3,50,32.0,1.8,,,,,,+
+,100.0,1.0,250.0,2.0,500.0,2.5,1000.0,2.67,+
+,ENDT
- For example, a table argument value of 100°C corresponds to the analysis temperature 212°F.

TABLEM3 (Cont.)

- Now, scale the y (dependent variable) value of the table by 0.5778, where $1 \text{ Watt/m/C} = 0.5778 \text{ Btu/hr/ft/F}$.
- For constant thermal conductivity, K
- \$MAT4, MID, K
- MAT4, 15, 204.0
- $204.0 \text{ Watt/m/C} * 0.5778 = 117.87 \text{ Btu/hr/ft/F}$
- MAT4, 15, 117.87
- For table dependent thermal conductivity, T(K)
- MAT4, 15, 0.5778
- \$MATT4, MID, T(K)
- MATT4, 15, 50
- TABLEM3, 50, 32.0, 1.8,,,,,,+
- $y = z * y_T((x-X1)/X2)$ has units of Btu/hr/ft/F



RADIATION MATRIX



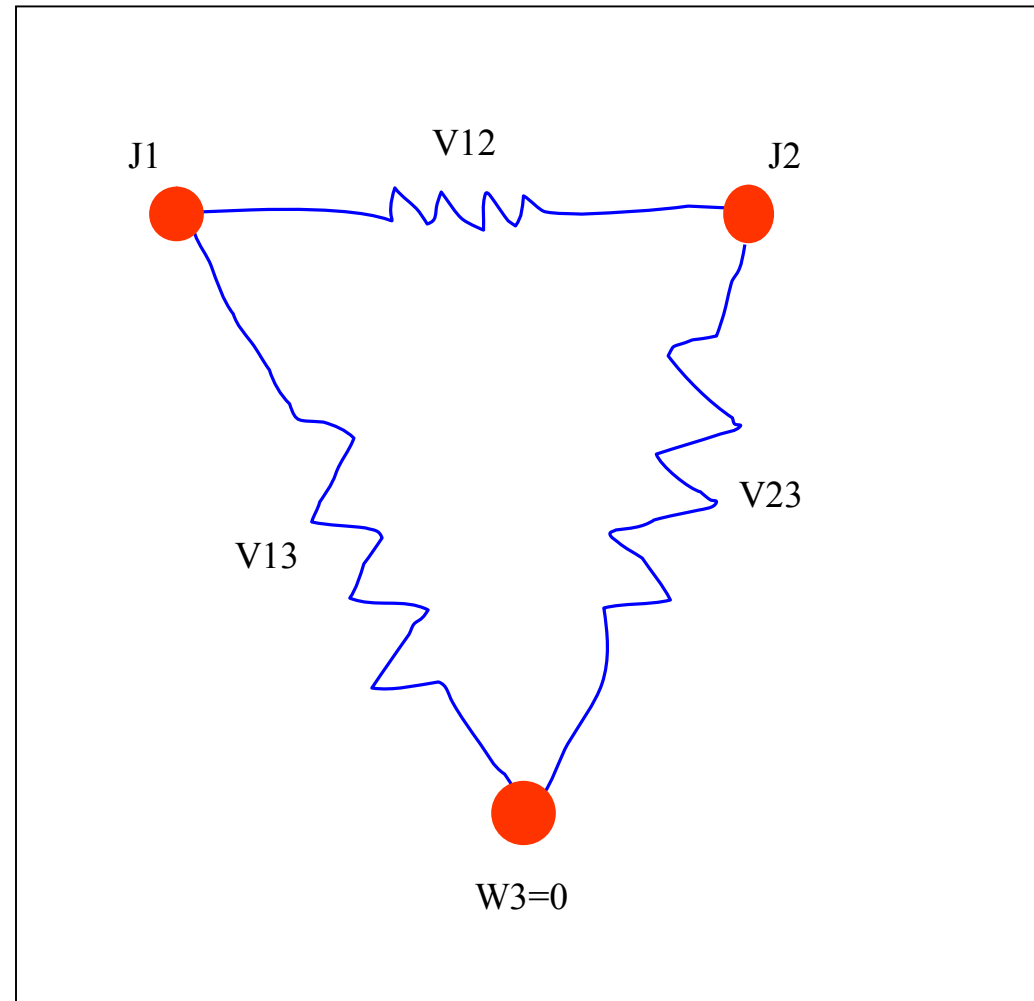


RESISTOR NETWORK

$$V_{12} = 1/A_{1F12}$$

$$V_{13} = 1/A_{1F13}$$

$$V_{23} = 1/A_{2F23}$$



RESISTOR NETWORK

- For each node, set the sum of the currents into it equal to zero. This yields the following equations.
- There are three nodes involved in the exchange.

$$\left(\frac{-1}{V_{13}} - \frac{1}{V_{12}}\right)J_1 + \frac{1}{V_{12}}J_2 = 0$$

$$\left(\frac{1}{V_{12}}\right)J_1 - \left(\frac{1}{V_{23}} + \frac{1}{V_{12}}\right)J_2 = 0$$

$$K_1 = \frac{-1}{V_{13}} - \frac{1}{V_{12}}$$

$$K_2 = \frac{1}{V_{12}}$$

$$K_3 = \frac{1}{V_{12}}$$

$$K_4 = \frac{1}{V_{23}} + \frac{1}{V_{12}}$$

$$K_1 = -1$$

$$K_2 = 0.717343$$

$$K_3 = 0.717343$$

$$K_4 = 1$$

SOLVE THE NETWORK EQUATIONS

$$T1 = 200$$

$$J1 = \sigma T1^4 = 90.719$$

$$J2 = 0.717343 J1 = 65.077$$

$$T2 = \left(\frac{J2}{\sigma} \right)^{0.25} = 184.061$$

CONCENTRIC SPHERES

- Example 9-1 : Thermal Radiation Heat transfer:
 - A spherical vacuum bottle consists of two silvered, concentric glass spheres, the inner one being 6 inches in diameter and the evacuated gap between the spheres being 0.25 inch. The emissivity of the silver coating is 0.02. If hot coffee at 200°F is in the bottle, and the outside temperature is 70°F, what is the radiative heat leakage out of the bottle?
 - The surface area of the sphere is $4 \cdot \pi \cdot R^2$
 - The inner radius is $6 \text{ in} / 2 = 3 \text{ in}$
 - And the outer radius is equal to $6.5 \text{ in} / 2 = 3.25 \text{ in}$
 - The inner surface area is equal to $4 \cdot \pi \cdot 3^2 = 113.0973$

NASTRAN INPUT FILE : two_cup.dat

```
$Model concentric sphere with two nodes
$ Length in Inches
SOL 153
CEND
ANALYSIS = HEAT
TITLE = MD Nastran job created on 05-Dec-03 at
13:33:05
ECHO = NONE
TEMPERATURE (INITIAL) = 1
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
    SUBTITLE=Default
    NLPARM = 1
    SPC = 1
    THERMAL (SORT1, PRINT)=ALL
    FLUX (SORT1, PRINT)=ALL
    OLOAD (SORT1, PRINT)=ALL
    SPCFORCES (SORT1, PRINT)=ALL
BEGIN BULK
Param      autospc yes
param,tabs,460.0
param,sigma,1.19e-11
```

```
nlparm,1
grid,101,,0.0,0.0
grid,102,,1.0,0.0
spoint,777
chbdyp,1,1,point,10,,101
,1,,,1.0,0.0,0.0
chbdyp,2,2,point,10,,102
,1,,,1.0,0.0,0.0
phbdy,1,113.0973
phbdy,2,132.7323
spc,1,101,1,200.0
spc,1,102,1,70.0
temp,1,101,200.0
temp,1,102,70.0
tempd,1,190.0
radm,1,,0.02
radset,1
radcav,1
view,10,1
view3d,10
radlst,1,1,1,2
$f12=1.0,f21=0.85207,f22=0.147929
radmtx,10,1,0.0,113.0973
radmtx,10,2,19.6350
enddata
```

NASTRAN RESULTS

LOAD STEP = 1.00000E+00

FORCES OF SINGLE-POINT CONSTRAINT

POINT ID.	TYPE	ID	VALUE	ID+1	VALUE	ID+2	VALUE	ID+3	VALUE	ID+4	VALUE	ID+5	VALUE
101	S		1.625897E+00	-1.625894E+00									
777	S		0.0										

1 MD NASTRAN JOB CREATED ON 05-DEC-03 AT 13:33:05 DECEMBER 15, 2003 MD NASTRAN 8/22/03 PAGE
10

0
1 MD NASTRAN JOB CREATED ON 05-DEC-03 AT 13:33:05 DECEMBER 15, 2003 MD NASTRAN 8/22/03 PAGE
11

DEFAULT

0 SUBCASE 1

LOAD STEP = 1.00000E+00

HEAT FLOW INTO HBDY ELEMENTS (CHBDY)

ELEMENT-ID	APPLIED-LOAD	FREE-CONVECTION	FORCED-CONVECTION	RADIATION	TOTAL
1	0.000000E+00	0.000000E+00	0.000000E+00	-1.625897E+00	-1.625897E+00
2	0.000000E+00	0.000000E+00	0.000000E+00	1.625894E+00	1.625894E+00

MATHCAD FILE: siegel_sphere_rad1.mcd

The R matrix is always SYMMETRIC

Model of concentric sphere in Siegel exampe 9-1

$$I := \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

$$E_{\alpha} := \begin{bmatrix} 0.02 & 0. \\ 0. & 0.02 \end{bmatrix} \quad E_{\varepsilon} := \begin{bmatrix} 0.02 & 0.0 \\ 0.0 & 0.02 \end{bmatrix}$$

$$F := \begin{bmatrix} 0 & 113.0973 \\ 113.0973 & 19.6350 \end{bmatrix}$$

$$A := \begin{bmatrix} 113.0973 & 0 \\ 0 & 132.7323 \end{bmatrix}$$

$$B := F \cdot (I - E_{\alpha})$$

$$B = \begin{bmatrix} 0 & 110.835 \\ 110.835 & 19.242 \end{bmatrix}$$

MATHCAD FILE: siegel_sphere_rad1.mcd

- This equation produces $Q=1.626$, which matches the NASTRAN results.

$$\sigma := 1.19 \cdot 10^{-11}$$

$$T := A \cdot E_{\varepsilon} - \left[A \cdot E_{\alpha} \left[A - F \cdot (I - E_{\alpha}) \right]^{-1} \right] \cdot F \cdot E_{\varepsilon}$$

$$T = \begin{bmatrix} 1.233 & -1.233 \\ -1.233 & 1.233 \end{bmatrix}$$

$$R := \sigma \cdot T$$

$$R = \begin{bmatrix} 1.467 \cdot 10^{-11} & -1.467 \cdot 10^{-11} \\ -1.467 \cdot 10^{-11} & 1.467 \cdot 10^{-11} \end{bmatrix}$$

$$u := \begin{bmatrix} 660^4 \\ 530^4 \end{bmatrix}$$

$$Q := R \cdot u$$

$$Q = \begin{bmatrix} 1.626 \\ -1.626 \end{bmatrix}$$

NASTRAN INPUT FILE: two_cup_type2.dat

```
$Using import R matrix
$Model concentric sphere with two nodes
$ Length in Inches
SOL 153
DIAG 8
COMPILE nlstatic list
ALTER 'IF ( NORADMAT>=0 )' (1,-1)$
MATPRN RGG
CEND
ANALYSIS = HEAT
TITLE = MD Nastran job created on 05-Dec-03 at
13:33:05
ECHO = NONE
TEMPERATURE (INITIAL) = 1
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  NLPARM = 1
  SPC = 1
  THERMAL (SORT1,PRINT)=ALL
  FLUX (SORT1,PRINT)=ALL
  OLOAD (SORT1,PRINT)=ALL
  SPCFORCES (SORT1,PRINT)=ALL
BEGIN BULK
```

```
Param      autospc yes
param,tabs,460.0
$param,sigma,1.19e-11
nlparm    1      0      auto      5      25
pw        no
          .001      1.-7
grid,101,,0.0,0.0
grid,102,,1.0,0.0
spoint,777
chbdyp,1,1,point,10,,101
,1,,,1.0,0.0,0.0
chbdyp,2,2,point,10,,102
,1,,,1.0,0.0,0.0
phbdy,1,113.0973
phbdy,2,132.7323
spc,1,101,1,200.0
spc,1,102,1,70.0
temp,1,101,200.0
temp,1,102,70.0
temprd,1,190.0
radm,1,,0.02
radset,1
radcav,1
view,10,1
view3d,10
param,sigma,1.0
radlst,1,2,1,2
$f12=1.0,f21=0.85207,f22=0.147929
radmtx*,10,1,1.4669e-11,-1.4669e-11
radmtx*,10,2,1.4669e-11
enddata
```

NASTRAN OUTPUT

```
0
0      MATRIX RGG      (GINO NAME 101 ) IS A DB  PREC      3 COLUMN X      3 ROW SYMMETRC MATRIX.
0COLUMN      1      ROWS      1 THRU      2      -----
      ROW
      1)      -1.4669D-11  1.4669D-11
0COLUMN      2      ROWS      1 THRU      2      -----
      ROW
      1)      1.4669D-11 -1.4669D-11
0COLUMNS      3 THRU      3 ARE NULL.
0THE NUMBER OF NON-ZERO TERMS IN THE DENSEST COLUMN =      2
0THE DENSITY OF THIS MATRIX IS  44.44 PERCENT.
```

NASTRAN OUTPUT

0

SUBCASE 1

LOAD STEP = 1.00000E+00

FORCES OF SINGLE-POINT CONSTRAINT

POINT ID.	TYPE	ID	VALUE	ID+1	VALUE	ID+2	VALUE	ID+3	VALUE	ID+4	VALUE	ID+5	VALUE
101	S		-1.625949E+00	1	1.625949E+00								
777	S		0.0										

1 MD NASTRAN JOB CREATED ON 05-DEC-03 AT 13:33:05 DECEMBER 17, 2003 MD NASTRAN 8/22/03 PAGE 50

0

1 MD NASTRAN JOB CREATED ON 05-DEC-03 AT 13:33:05 DECEMBER 17, 2003 MD NASTRAN 8/22/03 PAGE 51

DEFAULT

0

SUBCASE 1

LOAD STEP = 1.00000E+00

HEAT FLOW INTO HBDY ELEMENTS (CHBDY)

ELEMENT-ID	APPLIED-LOAD	FREE-CONVECTION	FORCED-CONVECTION	RADIATION	TOTAL
1	0.000000E+00	0.000000E+00	0.000000E+00	1.625949E+00	1.625949E+00
2	0.000000E+00	0.000000E+00	0.000000E+00	-1.625949E+00	-1.625949E+00

NASTRAN INPUT FILE FOR SPECULAR SURFACE

```

param      autospc yes
param,tabs,460.0
param,sigma,1.19e-11
lnparm    1      0      auto    5      25
pw        no
          .001    1.-7
grid,101,,0.0,0.0
grid,102,,1.0,0.0
spoint,777
chbdyp,1,1,point,10,,101
,1,,,1.0,0.0,0.0
chbdyp,2,2,point,10,,102
,1,,,1.0,0.0,0.0
phbdy,1,113.0973
phbdy,2,113.0973
spc,1,101,1,200.0
spc,1,102,1,70.0
temp,1,101,200.0
temp,1,102,70.0
tempd,1,190.0
radm,1,,0.02
radset,1
radcav,1
view,10,1
view3d,10
radlst,1,1,1,2
$f12=1.0,f21=0.85207,f22=0.147929
radmtx,10,1,0.0,113.0973
radmtx,10,2,0.0
enddata
    
```

```

*** USER WARNING MESSAGE 6427 (VIEWD)
    VIEW FACTORS WILL NOT BE COMPUTED FOR RADIATION
    CAVITY ID=      1 BECAUSE A RADIATION MATRIX IS
    ALREADY DEFINED.

    USER INFORMATION:  THE RADIATION MATRIX IS
    DEFINED ON RADLST OR RADMTX BULK DATA ENTRIES.
    1    MD NASTRAN JOB CREATED ON 05-DEC-03 AT 13:33:05
    DECEMBER 17, 2003 MD NASTRAN 8/22/03 PAGE      4

0
0    MATRIX RGG      (GINO NAME 101 ) IS A DB  PREC
3 COLUMN X          3 ROW SYMMETRIC MATRIX.
0COLUMN      1      ROWS      1 THRU      2      -----
-----
    ROW
    1)      1.3595D-11 -1.3595D-11
0COLUMN      2      ROWS      1 THRU      2      -----
-----
    ROW
    1)      -1.3595D-11 1.3595D-11
0COLUMNS      3 THRU      3 ARE NULL.
0THE NUMBER OF NON-ZERO TERMS IN THE DENSEST COLUMN =
2
0THE DENSITY OF THIS MATRIX IS 44.44 PERCENT.
    
```

NASTRAN SPECULAR RESULTS

0

SUBCASE 1

LOAD STEP = 1.00000E+00

FORCES OF SINGLE-POINT CONSTRAINT

POINT ID.	TYPE	ID	VALUE	ID+1	VALUE	ID+2	VALUE	ID+3	VALUE	ID+4	VALUE	ID+5	VALUE
101	S	1	1.506852E+00	-1	506852E+00								
777	S	0	0.0										

1 MD NASTRAN JOB CREATED ON 05-DEC-03 AT 13:33:05 DECEMBER 17, 2003 MD NASTRAN 8/22/03 PAGE 11

0

1 MD NASTRAN JOB CREATED ON 05-DEC-03 AT 13:33:05 DECEMBER 17, 2003 MD NASTRAN 8/22/03 PAGE 12

DEFAULT

0

SUBCASE 1

LOAD STEP = 1.00000E+00

HEAT FLOW INTO HBDY ELEMENTS (CHBDY)

ELEMENT-ID	APPLIED-LOAD	FREE-CONVECTION	FORCED-CONVECTION	RADIATION	TOTAL
1	0.000000E+00	0.000000E+00	0.000000E+00	-1.506852E+00	-1.506852E+00
2	0.000000E+00	0.000000E+00	0.000000E+00	1.506852E+00	1.506852E+00



PERFORMANCE OF VIEW FACTOR CALCULATION



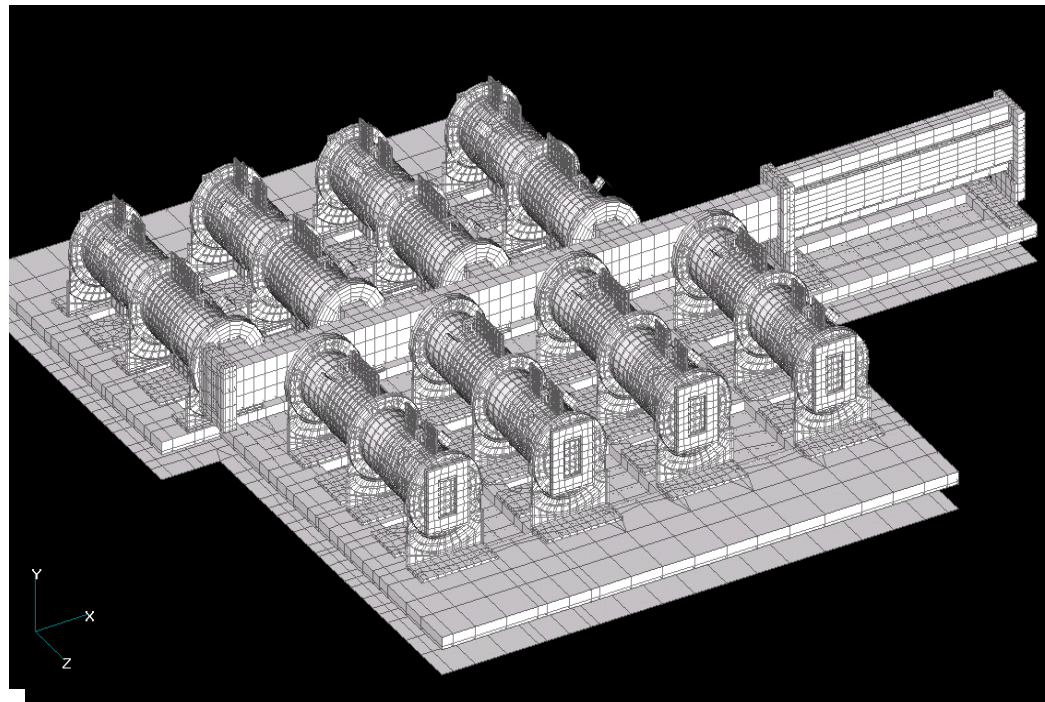


IMPROVE VIEW FACTOR PERFORMANCE

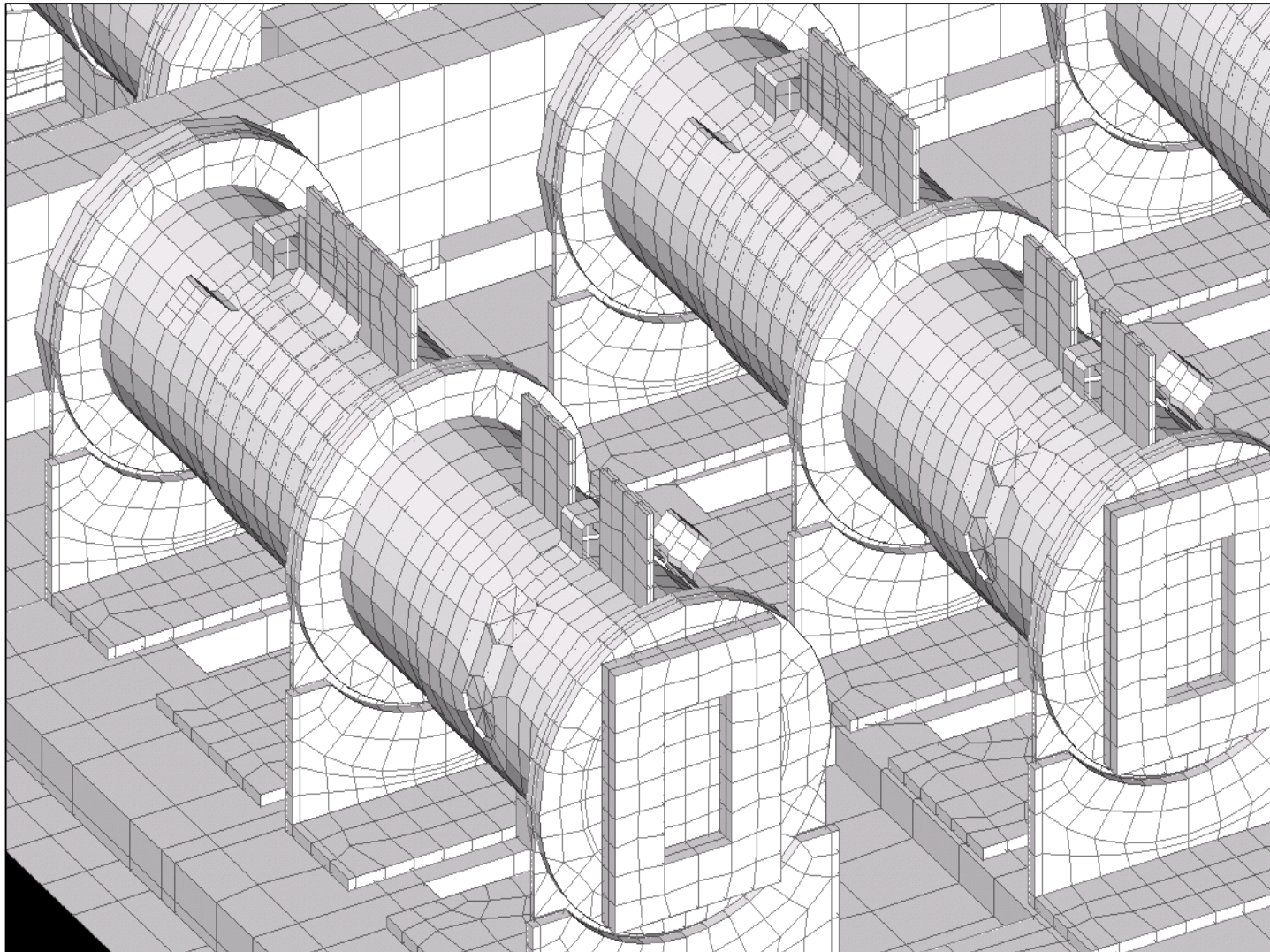
- The following problem has about 40000 radiation elements, and MD Nastran version 2001 took over 26734 CPU sec to perform the analysis.
- MD Nastran version 2004 took only 2425 CPU sec to perform the analysis. The performance increased by a factor of 11.

41183 RADIATION ELEMENTS

- This is a part from a Satellite. The heat source was imposed on the top of each cylinder, and there was a view factor exchange involving the entire model.



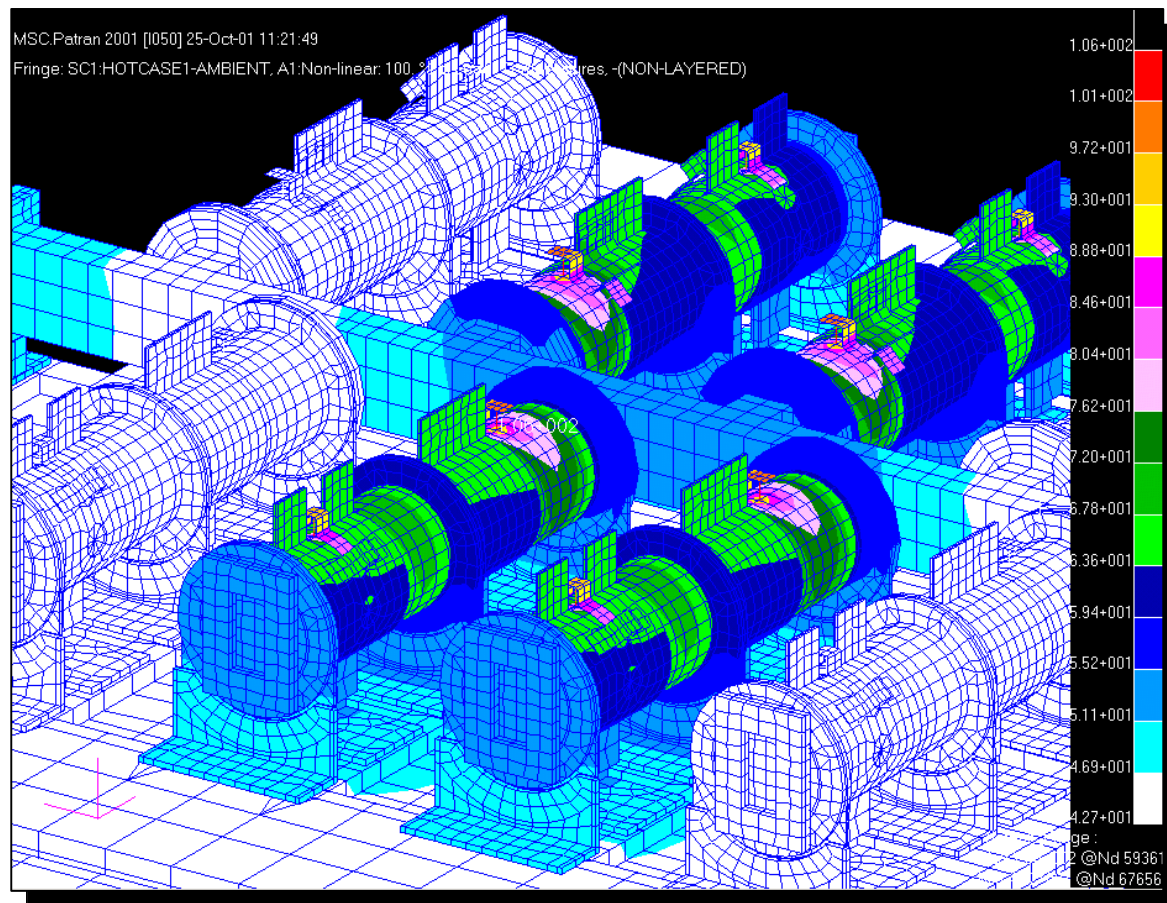
ZOOM IN ON THE MODEL



NAS104, Section
Copyright© 2008 MSC Software Corporation

S7-113

TEMPERATURE CONTOUR



NAS104, Section 7, September 2008

Copyright© 2008 MSC.Software Corporation

S7-114

CHBDYi BOUNDARY CONDITION SURFACE ELEMENTS





CHBDYi SURFACE ELEMENTS

- The CHBDYi boundary condition surface elements are the mechanism for applying heat (heat flux, convection, radiation) to conduction finite elements
- The CHBDYi surface elements were created as a means of more easily determining the LBC area. The LBC area could have been determined using the faces of conduction elements, but using CHBDYi elements was a better choice.
- If, for example, the conduction element is 2D isoparametric (heat conduction only in the plane of the element), and the external heat load (to the conduction element) is a heat flux, the energy from the heat flux load will be absorbed by the 2D element. The energy is uniformly distributed to each of the element's nodes.

CHBDYi SURFACE ELEMENTS

- The heat flow within the 2D element will still be in the plane of the element
- There is no conduction through the CHBDYi elements

EXERCISES

- Perform Workshop 17 “Import IGES file and Auto tet Mesh” in the exercise workbook.
- Perform Workshop 18 “Create Group and List” in the exercise workbook.
- Perform Workshop 19 “Nastran Bulk Data Entries” in the exercise workbook.
- Perform Workshop 20 “Radiation Shield Using View Entry” in the exercise workbook.
- Perform Workshop 21 “Thermal Stress From Transient Heat Transfer Run” in the exercise workbook.



SECTION 8

BASIC THERMAL ANALYSIS THEORY





BRIEF INTRODUCTION TO HEAT TRANSFER





TYPES OF THERMAL SYSTEMS

- Thermal problems can be categorized as
 - Steady-state
 - Steady-state analyses are concerned with state point solutions with constant/fixed boundary conditions
 - Transient
 - Transient analyses are characterized by solution evolution over time, and in addition to energy exchange with the environment, involves thermal energy storage
 - Linear
 - Properties; i.e. conductivity, convection coefficient; they do not change with
 - Temperature
 - The boundary conditions do not involve
 - Radiation



TYPES OF THERMAL SYSTEMS (Cont.)

- Thermal problems can be categorized as (continued)
 - Nonlinear
 - Temperature dependent material properties.
 - Conduction
 - Convection
 - Natural
 - Forced
 - Radiation boundary conditions.
 - Ambient
 - Enclosure
 - Steady-state or transient solution.
 - All nonlinear analyses necessarily involve solution iteration, error estimation, and convergence criteria.



HEAT TRANSFER EQUATION

- Thermal equilibrium between heat sources, energy flow density and temperature rate is expressed by the Energy Conservation Law, which may be written as

$$\rho C_p \frac{\partial T}{\partial t} + \{\nabla\}^T \{\sigma\} = \dot{q}$$

- where
 - $\{\nabla\}$ = gradient vector operator
 - $\{\sigma\}$ = rate of energy density flow

- Rate of energy density flow has a diffusion and a convection part

$$\{\sigma\} = -[\Lambda]\{\nabla\}T + \rho C_p T \{v\}$$

where $[\Lambda]$ is the conductivity matrix.

Assume that the continuum is incompressible and that there is no spatial variation of ρ and C_p . Then, the conservation law is

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \{v\}^T \{\nabla\}T - \{\nabla\}^T [\Lambda] \{\nabla\}T = \dot{q}$$

Without convection this equation can be reduced to

$$\rho C_p \frac{\partial T}{\partial t} - \lambda \{\nabla\}^T \{\nabla\}T = \dot{q}$$

HEAT TRANSFER LOADS & BCs

- Specify temperature for set of nodes
 - Spatial varying
 - Time varying

$$T = T(x, y, z; t)$$

The screenshot shows the 'Load/Boundary Conditions' dialog box with the 'Input Data' sub-dialog open. The 'Load/Boundary Conditions' dialog has the following fields: Action: Create, Object: Temp (Thermal), Type: Nodal, Current Load Case: Thermal Transient..., and Type: Time Dependent. The 'Input Data' sub-dialog has the following fields: Boundary Temperature: f:field_spatial_varying, * Time Function: f:field_time_varying, Spatial Fields: field_spatial_varying, and Time Dependent Fields: field_time_varying. The 'Existing Sets' list is empty. The 'New Set Name' field contains 'wall_temp'. The 'OK', 'Reset', and 'Cancel' buttons are at the bottom.

HEAT TRANSFER LOADS & BCs (Cont.)

- Specify applied heat to element faces

- Spatial varying
- Time varying

$$q_s = q_s(x, y, z; t)$$

- Normal flux
- Directional flux
- Nodal source
- Volumetric generation
- Total heat

- Application region
 - 1D: edge
 - 2D: edge, face
 - 3D: face

The image shows two overlapping dialog boxes from the MSC Software interface. The background dialog is titled 'Load/Boundary Conditions' and has the following settings: Action: Create, Object: Applied Heat, Type: Element Uniform, Option: Normal Fluxes, Current Load Case: Thermal Transient..., and Type: Time Dependent. The foreground dialog is titled 'Input Data' and has the following settings: Form Type: Basic, Heat Flux: f:field_spatial_varying, * Time Function: f:field_time_varying, Spatial Fields: field_spatial_varying, Time Dependent Fields: field_time_varying, and Target Element Type: 3D. The 'Input Data' dialog also has buttons for OK, Reset, and Cancel.

HEAT TRANSFER LOADS & BCs (Cont.)

- Specify convection heat transfer to the environment

$$-k_f \frac{\partial T}{\partial n} \Big|_{n=0} = h_C (T_S - T_\infty)$$

- To ambient
- Flow tube to ambient
- Coupled
- Coupled flow tube
- Coupled advection
- Duct flow
- h is film coefficient
- Application region is the film

Load/Boundary Conditions

Action:

Object:

Type:

Option:

Current Load Case:

Input Data

Form Type:

Convection Coefficient:

Ambient Temperature:

* Temperature Function:

* Time Function:

* Time Function:

Spatial Fields:

Temperature Dependent Fields:

Time Dependent Fields:

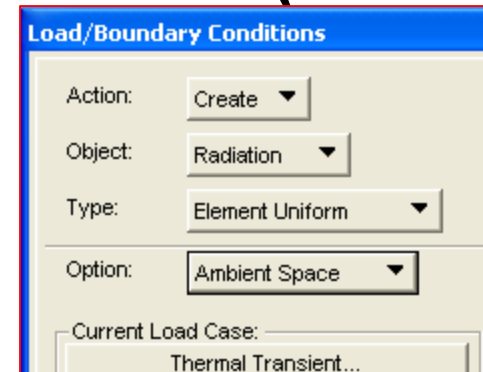
OK Reset Cancel

HEAT TRANSFER LOADS & BCs (Cont.)

- Specify radiation heat transfer

$$q_S = \varepsilon \sigma F_G A (T_S^4 - T_\infty^4)$$

- Ambient space
- Ambient nodes
- Enclosure
- σ is the Stefan-Boltzmann constant
- ε is the emissivity



Load/Boundary Conditions

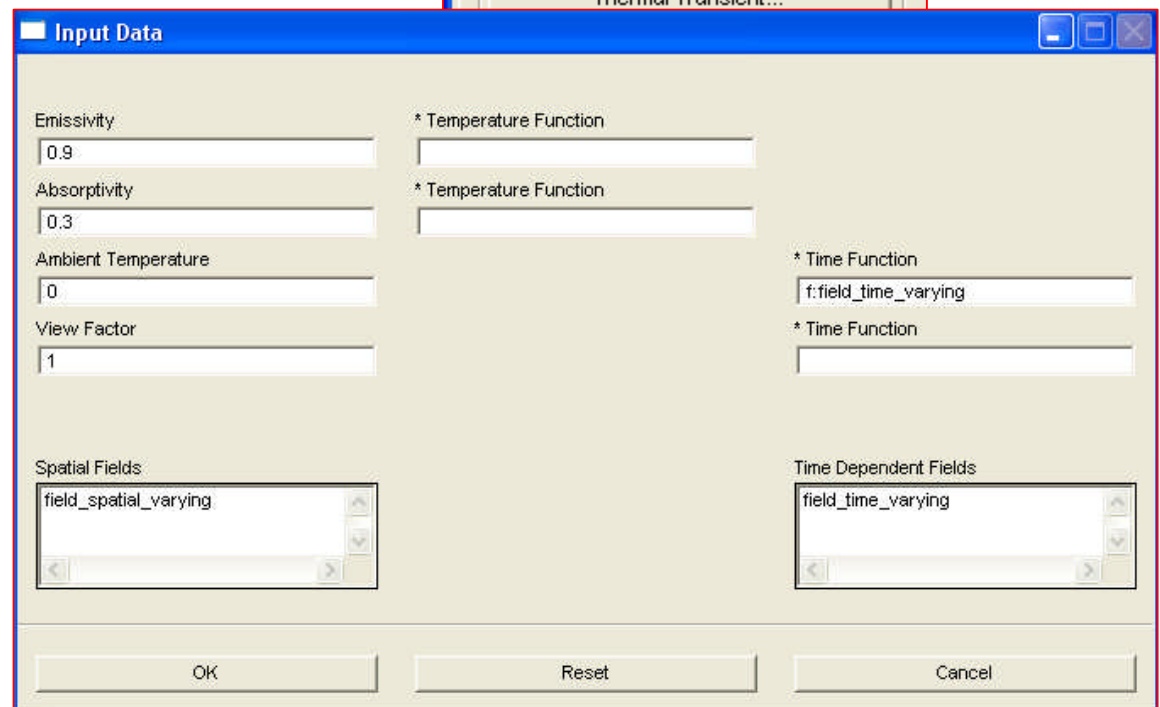
Action: Create

Object: Radiation

Type: Element Uniform

Option: Ambient Space

Current Load Case: Thermal Transient...



Input Data

Emissivity: 0.9

Absorptivity: 0.3

Ambient Temperature: 0

View Factor: 1

* Temperature Function

* Temperature Function

* Time Function: f:field_time_varying

* Time Function

Spatial Fields: field_spatial_varying

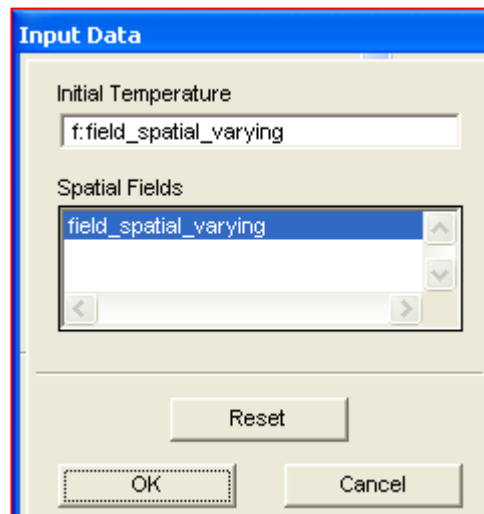
Time Dependent Fields: field_time_varying

OK Reset Cancel

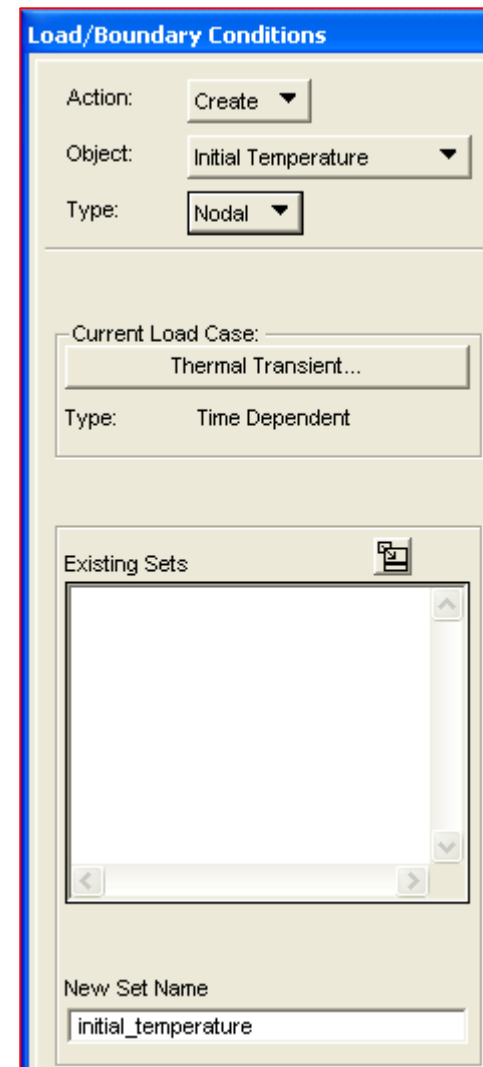
HEAT TRANSFER INITIAL CONDITIONS

- Specify the initial conditions
- Transient analysis only

$$T(x, y, z; 0) = T_{initial}(x, y, z)$$



The 'Input Data' dialog box is used to define the initial temperature. It features a blue title bar and a light beige background. The 'Initial Temperature' section contains a text field with the value 'f:field_spatial_varying'. Below this, the 'Spatial Fields' section has a list box containing 'field_spatial_varying'. At the bottom, there are 'Reset', 'OK', and 'Cancel' buttons.



The 'Load/Boundary Conditions' dialog box is used to create and manage load cases. It has a blue title bar and a light beige background. The 'Action' dropdown is set to 'Create', the 'Object' dropdown is set to 'Initial Temperature', and the 'Type' dropdown is set to 'Nodal'. The 'Current Load Case' section shows 'Thermal Transient...'. The 'Type' is set to 'Time Dependent'. The 'Existing Sets' section is empty. The 'New Set Name' field contains 'initial_temperature'.

BRIEF INTRODUCTION TO CONDUCTION





TOPICS ON CONDUCTION

- Laws affecting conduction
- Supported element topologies
 - 1D, 2D, 3D, Axisymmetric
- Assemble element conduction matrices to form system matrix
- Create elements
- Material for conduction
 - Isotropic (K)
 - Orthotropic (Kxx, Kyy, Kzz)
 - Anisotropic (Kxx, Kxy, Kxz, Kyy, Kyz, Kzz)
 - Temperature independent or dependent
- Constraints and loads that can be applied to conduction elements
 - Temperature (constant, spatial dependent, or time dependent)
 - Heat flux to a surface (normal flux)
 - Heat generation



LAWS AFFECTING CONDUCTION

- Second law of thermodynamics
 - Heat only flows **from** regions of **high temperature** **to** regions of **low temperature**.
- Fourier's Law (empirical)
 - One dimension $\dot{q} = -kA(dT / dx)$
 - General $\{\dot{q}\} = -k \left\{ \frac{\partial T}{\partial x_i} \right\}$
- First law of thermodynamics - conservation of energy

$$\nabla^2 T + \frac{\dot{q}}{k} = \frac{1}{\alpha} \frac{\partial T}{\partial t}$$

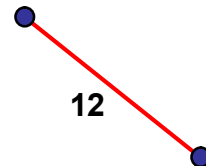
SUPPORTED ELEMENT TOPOLOGIES

- Element types
- The MD Nastran SOL 153 and 159 elements are isoparametric

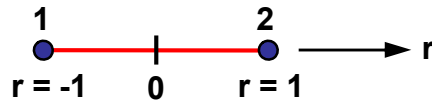
1D	2D	3D	Axisym
CBAR	CQUAD4	CHEXA	CTRIAX6
CBEAM	CQUAD8	CPENTA	
CBEND	CTRIA3	CTETRA	
CONROD	CTRIA6		
CROD			
CTUBE			

SUPPORTED ELEMENT TOPOLOGIES (Cont.)

- 1D element



- Isoparametric element with 1 dimension, (r)



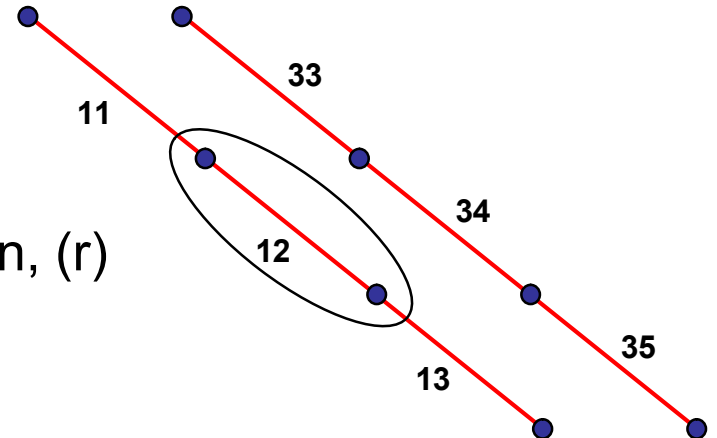
- Interpolation within element

$$T = \sum_{i=1}^2 h_i(r) T_i$$

$$h_1 = (1-r)/2, h_2 = (1+r)/2$$

- Element conduction matrix

$$[k]_e^{(j)} = \int_{V^{(j)}} [B]^{(j)T} [k]^{(j)} [B]^{(j)} dV^{(j)}$$



SUPPORTED ELEMENT TOPOLOGIES (Cont.)

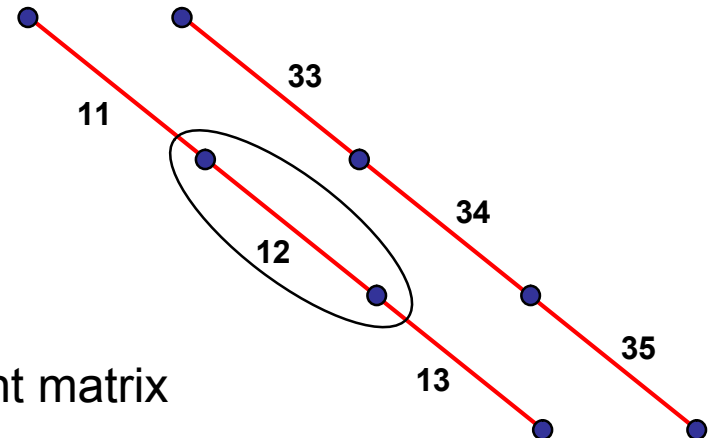
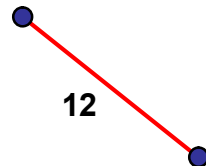
- 1D element

- where

- $[k]_e^{(j)}$ = element j conduction matrix
- $[B]^{(j)}$ = element j temperature gradient matrix
- $[k]^{(j)}$ = element j thermal conductivities
- The integration is over the volume of element j

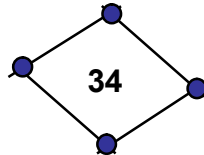
- An example is

$$[k]_e^{(j)} = \frac{Ak}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}^{(j)}$$

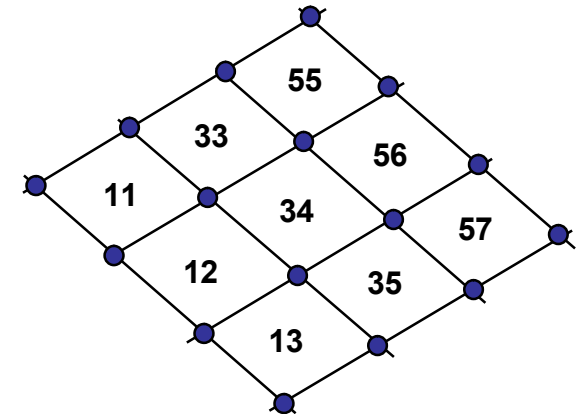
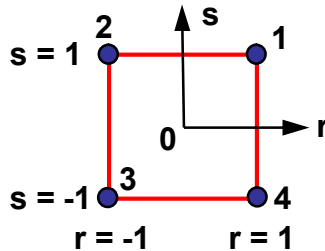


SUPPORTED ELEMENT TOPOLOGIES (Cont.)

- 2D element



- Isoparametric element with 2 dimensions, (r,s)



- Interpolation within element

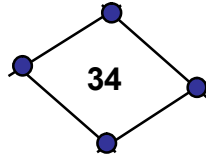
$$T = \sum_{i=1}^4 h_i(r, s) T_i$$

$$h_1 = (1+r)(1+s)/4, \quad h_2 = (1-r)(1+s)/4$$

$$h_3 = (1-r)(1-s)/4, \quad h_4 = (1+r)(1-s)/4$$

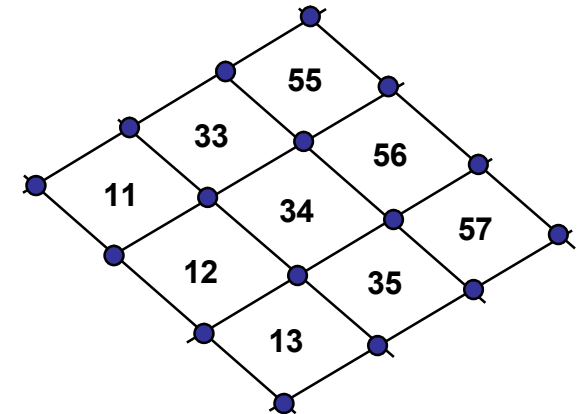
SUPPORTED ELEMENT TOPOLOGIES (Cont.)

- 2D element



- Element conduction matrix

$$[k]_e^{(j)} = \int_{V^{(j)}} [B]^{(j)T} [k]^{(j)} [B]^{(j)} dV^{(j)}$$



SUPPORTED ELEMENT TOPOLOGIES (Cont.)

- 3D element
 - Isoparametric element with 3 dimensions, (r,s,t)
 - Interpolation within element

$$T = \sum_{i=1}^8 h_i(r, s, t) T_i$$

- Element conduction matrix

$$[k]_e^{(j)} = \int_{V^{(j)}} [B]^{(j)T} [k]^{(j)} [B]^{(j)} dV^{(j)}$$

SUPPORTED ELEMENT TOPOLOGIES (Cont.)

- Recommendations
 - Use linear elements, unless have substantial curvature and desire to minimize the number of elements
 - If doing thermal analysis (calculate temperatures) to structural analysis (stress analysis) mapping (loads are temperatures from thermal analysis), it is best to use the same type of element that is to be used for the structural analysis, e.g. Tet10, and not Tet4.
 - Loads and boundary conditions do not affect which type of element should be used
- Special Elements
 - This category is for elements that are not finite elements, but of course they can be used in modeling a thermal process. There are two types of special elements. They are
 - CELASi (scalar spring) -- simple resistive component

SUPPORTED ELEMENT TOPOLOGIES (Cont.)

- Special Elements
 - CELASi (scalar spring)
 - Used to model a one-dimensional conduction element with a thermal resistance.
 - $R = L/kA$, where CELAS2 used inverse of R
 - $G = 1/R = kA/L$ (Watt/°C, type of unit)
 - DMI or DMIG (direct matrix input) -- complex component
 - DMI
 - For defining matrix data blocks.

$$[name] = [X_{ij}]$$

- where the elements X_{ij} may be real or complex (real and imaginary part). The matrix is defined by a single header entry and one or more column entries. Only one header is required. A column entry is required for each column with nonzero elements.
- For more information see the MD Nastran Quick Reference Guide.



ASSEMBLE ELEMENT MATRICES

- Assemble all element conduction and other system matrices

- Derive element conduction matrices

$$[k]_e^{(j)} = \int_{V^{(j)}} [B]^{(j)T} [k]^{(j)} [B]^{(j)} dV^{(j)}$$

- Sum the element conduction matrices

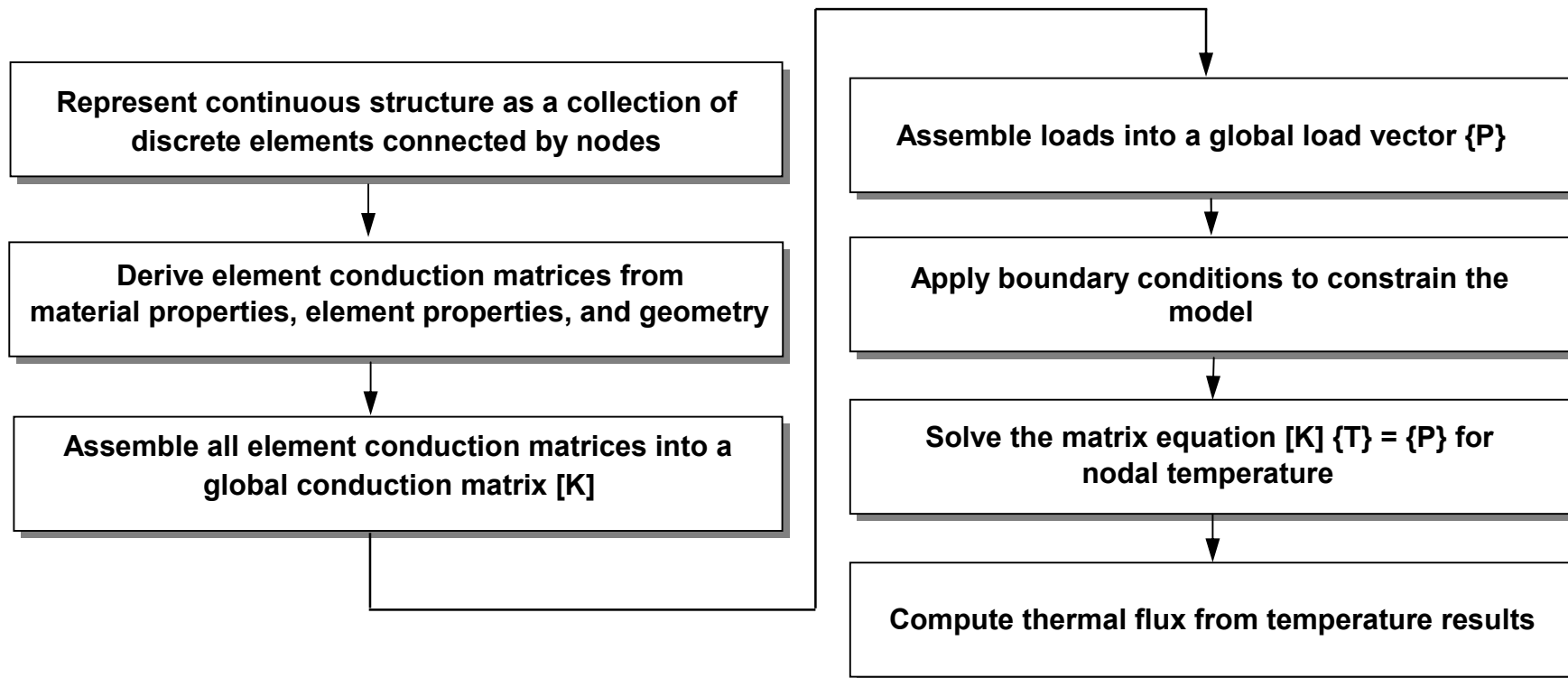
$$[K] = \sum_j [k]_e^{(j)}$$

- Create other necessary system matrices, and create the conduction system matrix equation

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

ASSEMBLE ELEMENT MATRICES (Cont.)

- Summary of the steady-state finite element method



BRIEF INTRODUCTION TO CONVECTION





HEAT TRANSFER VIA CONVECTION

- The transfer of energy (heat) between a solid boundary and a fluid takes place by a combination of conduction and mass transport
- The fluid motion can be induced by two different processes.
 - The fluid may be moved as a result of density differences due to a temperature variation within the fluid. This type of heat transfer is called **free (or natural) convection**.
 - The fluid motion may be caused by an external source, such as a pump or blower. This type of heat transfer is called **forced convection**.
- A simple representation of convection heat transfer is given by

$$q_{\text{surface to fluid}} = h_c A (T_s - T_\infty)$$

- where
 - $q_{\text{surface to fluid}}$ = rate of heat transfer by convection
 - h_c = convective heat transfer coefficient, or average unit thermal convective conductance



HEAT TRANSFER VIA CONVECTION (Cont.)

- where
 - A = heat transfer area
 - T_s = surface temperature
 - T_∞ = fluid temperature
- The convective heat transfer coefficient is actually a complicated function
 - Fluid flow
 - Thermal properties of the fluid medium
 - Geometry of the thermal system
 - Its numerical value is in general not uniform over a surface
 - Depends on the location where the fluid temperature T_∞ is specified
- Convective heat transfer is modeled with SimXpert using empirically derived functions

HEAT TRANSFER VIA CONVECTION (Cont.)

- Free convection

$$Nu = \frac{h_c L}{k_f} \longrightarrow h_c = Nu \frac{k_f}{L}$$

$$q_{\text{surface to fluid}} = Nu \cdot k_f \frac{A}{L} (T_s - T_\infty)$$

$$Nu = f(Gr) g(Pr)$$

- where

- Nu = Nusselt number (ratio of the temperature gradient in the fluid at the surface to a reference temperature gradient)
- Gr = Grashof number (ratio of buoyant to viscous forces)
- Pr = Prandtl number
- The Reynolds number (Re) is not needed for free convection

HEAT TRANSFER VIA CONVECTION (Cont.)

- Free convection

- To Ambient $q = h (T_s - T_a) ** EXPF * (T_s - T_a)$

$$q = h (T_s ** EXPF - T_a ** EXPF)$$

Element Uniform, Target Element Type: 1D, 2D, 3D
1D, 2D, or 3D structure -> ambient temperature

- Coupled $q = h (T_s - T_a) ** EXPF * (T_s - T_a)$

$$q = h (T_s ** EXPF - T_a ** EXPF)$$

Element Uniform, Target Element Type: 1D, 2D, 3D
Region 2: Nodal, 1D, 2D, 3D
1D, 2D, or 3D structure -> Nodal, 1D, 2D, or 3D structure

HEAT TRANSFER VIA CONVECTION (Cont.)

- Forced convection

$$Nu = \frac{h_c L}{k_f} \longrightarrow h_c = Nu \frac{k_f}{L}$$

$$q_{surface\ to\ fluid} = Nu \cdot k_f \frac{A}{L} (T_s - T_\infty)$$

$$Nu = f(Re) g(Pr)$$

- where

- Nu = Nusselt number (ratio of the temperature gradient in the fluid at the surface to a reference temperature gradient)
- Re = Reynolds number (ratio of inertia to viscous forces)
- Pr = Prandtl number

HEAT TRANSFER VIA CONVECTION (Cont.)

- Forced convection

- Flow Tube To Ambient $h = coef * Re^{**} Expr * Pr^{**} Expp$

$$h = k / d * coef * Re^{**} Expr * Pr^{**} Expp$$

Element Uniform, Target Element Type: 1D

1D Flow Tube -> ambient temperature

- Coupled Flow Tube $h = coef * Re^{**} Expr * Pr^{**} Expp$

$$h = k / d * coef * Re^{**} Expr * Pr^{**} Expp$$

Element Uniform, Target Element Type: 1D

Region 2: Nodal, 1D, 2D, 3D

1D Flow Tube -> Nodal, 1D, 2D, or 3D structure

HEAT TRANSFER VIA CONVECTION (Cont.)

- Forced convection

- Coupled Advection

$$q = h (T_s - T_a) ** EXPF * (T_s - T_a)$$

$$q = h (T_s ** EXPF - T_a ** EXPF)$$

Element Uniform, Target Element Type: 1D, 2D, 3D

Region 2: 1D

1D, 2D, or 3D structure -> 1D Flow Tube

- Duct Flow

$$h = coef * Re ** Expr * Pr ** Expp$$

$$h = k / d * coef * Re ** Expr * Pr ** Expp$$

Element Uniform, Target Element Type: 1D

Region 2: 2D, 3D

1D Flow Tube, w/o convection -> 2D or 3D structure, w/o advection



BRIEF INTRODUCTION TO RADIATION





RADIATION CONCEPTS

- Thermal radiation
 - Radiation at wavelengths (λ) in the domain $0.1 \mu\text{m} < \lambda < 100 \mu\text{m}$
 - This domain includes all visible light ($0.35 \mu\text{m} < \lambda < 0.75 \mu\text{m}$), portions of the ultraviolet light sub-domain (short wavelength), and infrared light sub-domain (long wavelength)
- Total emissive power, E
 - The total amount of radiation emitted by a body per unit area and unit time
- Black body
 - Ideal radiator. Emits at any specified temperature the maximum possible amount of thermal radiation at all wavelengths
 - Absorbs all radiation for all wavelengths



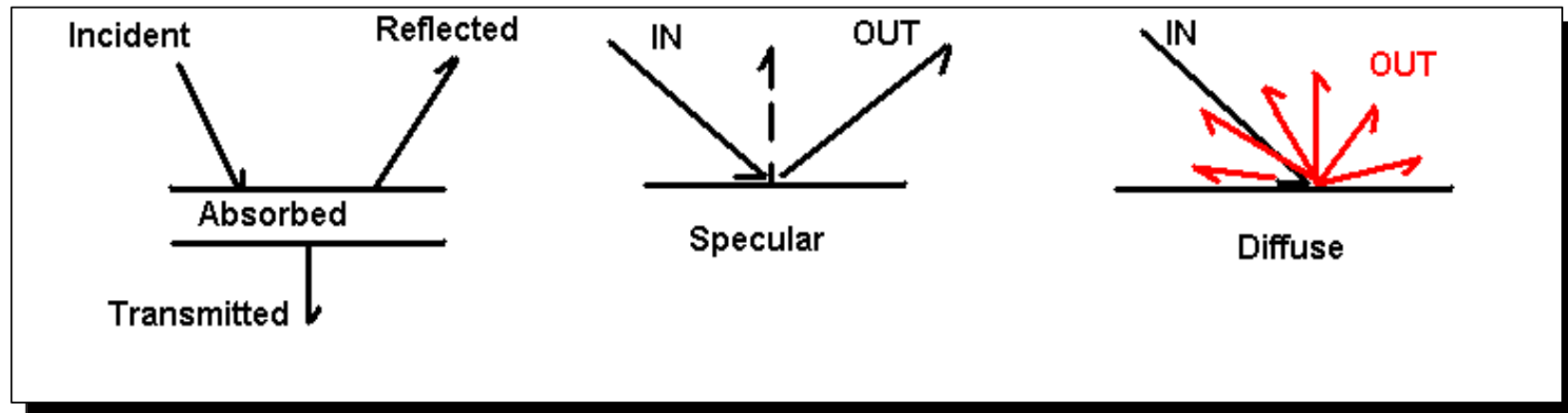
RADIATION CONCEPTS (Cont.)

- Grey body
 - Not black body, because does not absorb all incident thermal radiation
 - A grey body emits a constant fraction of what a black body emits, at all wavelengths, for both bodies at the same temperature
 - Real bodies (grey bodies) emit radiation at a rate lower than black bodies
 - If the absorptivity is > 0.9 , in general, it is acceptable to neglect reflection off of thermally radiating surfaces
 - However, there are numerous problems involving surfaces of low absorptivity and emissivity



RADIATION CONCEPTS (Cont.)

- Absorption, reflection, and transmission



- When radiation impinges on a body it is partially absorbed, partially reflected, and partially transmitted.

$$\alpha + \rho + \tau = 1$$

- α = absorptivity
- ρ = reflectivity
- τ = transmissivity

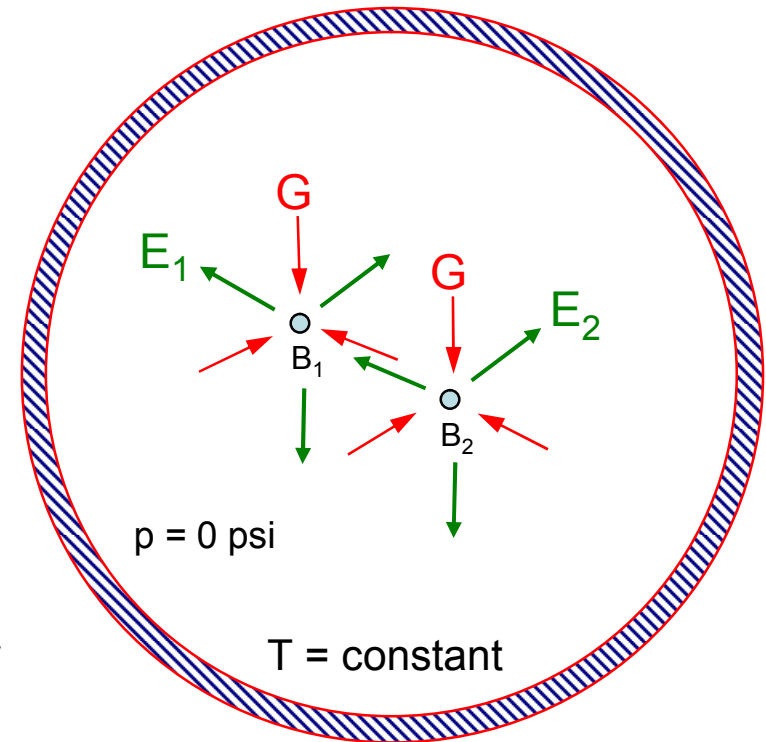
RADIATION CONCEPTS (Cont.)

- Kirchhoff's law
 - Thermal equilibrium
 - Body B_i
 - Surface area of B_i is A_i
 - Absorptivity, α_i
 - Total emissive power, E_i , of B_i

$$A_1 G \alpha_1 = A_1 E_1 \quad A_2 G \alpha_2 = A_2 E_2$$

$$G = \frac{E_1}{\alpha_1} = \frac{E_2}{\alpha_2} = \frac{E}{\alpha}$$

- At thermal equilibrium the ratio of E and α is the same for all bodies



RADIATION CONCEPTS (Cont.)

- Kirchhoff's law
 - Emissivity (in general it is a function of T and λ)

$$\varepsilon = \frac{E}{E_b}$$

- Grey body at thermal equilibrium

$$\alpha = \varepsilon$$

- Black body at thermal equilibrium

$$\alpha_b = \varepsilon_b = 1$$

RADIATION CONCEPTS (Cont.)

- Planck's law

$$E_{b\lambda} = (8.0\pi hc\lambda^{-5}) / (e^{(hc)/(\lambda kT)} - 1)$$

- where

- h = Planck's constant
 - k = Boltzmann's constant

- Stefan-Boltzmann equation

$$E_b = \int_0^{\infty} E_{b\lambda} d\lambda = \sigma T^4$$

- where

- σ = Stefan-Boltzmann constant

RADIATION CONCEPTS (Cont.)

- Net rate of radiant heat transfer between a surface and its surroundings

$$q_S = \sigma A (T_{surface}^4 - T_{ambient}^4)$$

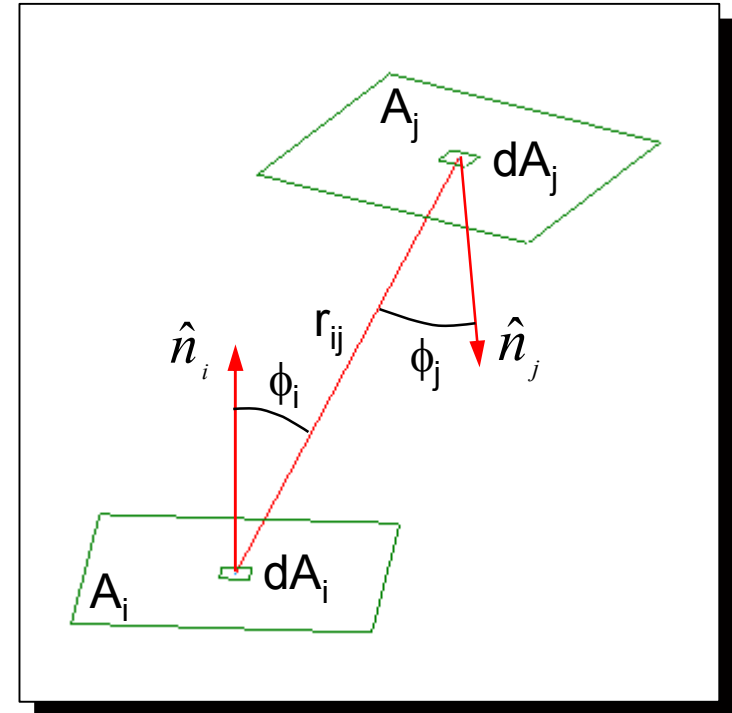
- Net rate of radiant heat flow between two grey bodies (surfaces)

$$q_S = \varepsilon \sigma F_G A (T_S^4 - T_\infty^4)$$

- where
 - F_G is the geometric view factor

RADIATION CONCEPTS (Cont.)

- Net rate of heat flow by radiation between two grey bodies (surfaces) A_i and A_j



$$q_{i \rightarrow j} = \varepsilon \sigma (T_i^4 - T_j^4) \iint_{A_i A_j} \frac{\cos \phi_i \cos \phi_j}{\pi r_{ij}^2} dA_i dA_j$$

RADIATION CONCEPTS (Cont.)

- Viewfactor

$$F_{ij} = \frac{1}{A_i} \iint_{A_i A_j} \frac{\cos \phi_i \cos \phi_j}{\pi r_{ij}^2} dA_i dA_j$$

- Reciprocity theorem $A_i F_{ij} = A_j F_{ji}$

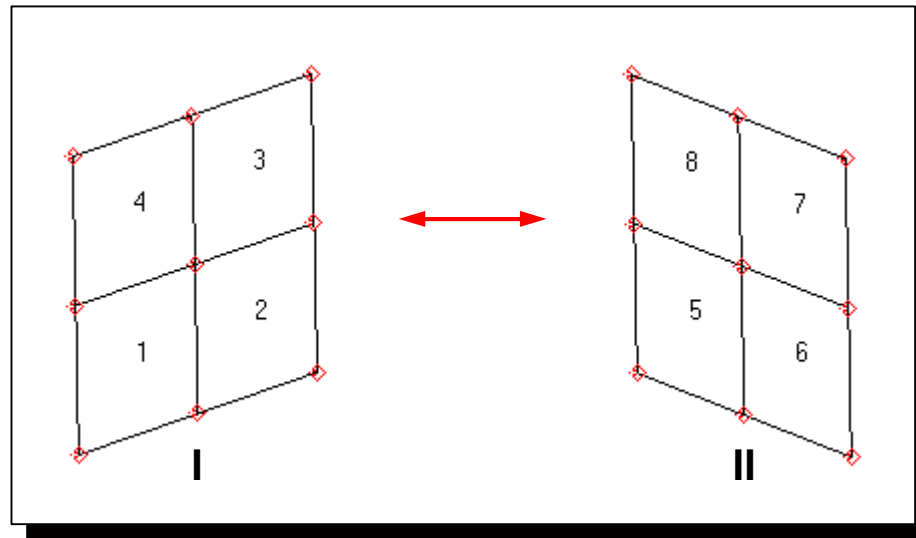
- In general $F_{ij} \neq F_{ji}$

- Net radiation from A_i to other surfaces in enclosure $q_{i,net} = \sum_{k=1, k \neq i}^n A_i F_{ik} (E_{bi} - E_{bk})$

- Sum of F_{ik} for enclosure $\sum_{k=1, k \neq i}^n F_{ik} = 1$

FINITE DIFFERENCE CALC OF VF

- Example of two CHBDYi elements (I and II), each with four sub-elements (I: 1 thru 4, or II: 5 thru 8)



- View factor from sub-element k, of CHBDYi element I, to CHBDYi element II

$$F_{kII} = F_{k5} + F_{k6} + F_{k7} + F_{k8}, k = 1, \dots, 4$$

FINITE DIFFERENCE CALC OF VF (Cont.)

- From definition of the view factor

$$A_{II}F_{II,I} = A_1F_{1II} + A_2F_{2II} + A_3F_{3II} + A_4F_{4II}$$

- From reciprocity

$$A_I F_{I,II} = A_{II} F_{II,I} \Rightarrow F_{I,II} = A_{II} F_{II,I} / A_I$$

- Combining the above equations yields

$$\begin{aligned} F_{I,II} = & [A_1(F_{1,5} + F_{1,6} + F_{1,7} + F_{1,8}) + \\ & A_2(F_{2,5} + F_{2,6} + F_{2,7} + F_{2,8}) + \\ & A_3(F_{3,5} + F_{3,6} + F_{3,7} + F_{3,8}) + \\ & A_4(F_{4,5} + F_{4,6} + F_{4,7} + F_{4,8})] / A_I \end{aligned}$$

FINITE DIFFERENCE CALC OF VF (Cont.)

- So, the view factor from I to II is approximated by

$$F_{I,II} \cong \frac{1}{A_I} \sum_{i=1}^4 \sum_{j=5}^8 \frac{\cos \phi_i \cos \phi_j}{\pi r_{ij}} A_i A_j$$

- This is an approximation because it is assumed that the two angles, ϕ_i and ϕ_j , and the distance r_{ij} are constant for all pairs of sub-elements (sub-element pair (Elm i, Elm j), where $i = 1, \dots, 4$; and $j = 5, \dots, 8$)

SECTION 9

NONLINEAR SOLVERS





NONLINEAR SOLVERS

- Steady-state, transient, linear, or nonlinear thermal systems can be modeled using MD Nastran
- The MD Nastran solvers used are for nonlinear system response
 - Nonlinear solvers are used to solve linear problems
- Following are some brief comments about the steady-state (SOL 153) and transient (SOL 159), nonlinear solvers
 - How they work
 - Convergence
 - Recommendations on solving problems





STEADY-STATE THERMAL ANALYSIS





STEADY-STATE SOLVER STEPS

- The familiar conduction heat transfer equation is

$$\nabla^2 T + \frac{\dot{q}}{k} = \frac{1}{\alpha} \frac{\partial T}{\partial t}$$

- A transition is made from this equation (strong form) to a variational formulation (weak form). The matrix equation corresponding to the variational formulation is

$$[M]\{\dot{T}\} + [K]\{T\} = \{F\}$$

- The MD Nastran Thermal steady-state equation derived from this equation is

$$[K]\{T\} + [\mathcal{R}]\{T + T_{abs}\}^4 = \{P\} + \{N\}$$

STEADY-STATE SOLVER STEPS (Cont.)

- This is a nonlinear matrix equation. It is solved using the Newton-Raphson iteration scheme. A residual load vector function is defined as the **difference** between the applied thermal load vector and the thermal load vector due to element temperature.

$$\{R\} = (\{P\} + \{N\}) - ([K]\{T\} + [\mathcal{R}]\{T + T_{abs}\}^4)$$

- The residual load vector function is equal to zero for the solution temperature vector. The task is to determine the solution temperatures. The residual load vector function is approximated by its first-order Taylor series expansion about the temperature vector from the i-th iteration.

$$\{R(\{T\})\} \cong \{R(\{T\}^i)\} + \left[\frac{\partial \{R\}}{\partial \{T\}} \right]^i (\{T\} - \{T\}^i)$$

STEADY-STATE SOLVER STEPS (Cont.)

- The Taylor series expansion is evaluated at the temperatures for the (i + 1)-th iteration, and set equal to zero.

$$\{R(\{T\}^{i+1})\} \cong \{R(\{T\}^i)\} + \left[\frac{\partial \{R\}}{\partial \{T\}} \right]^i (\{T\}^{i+1} - \{T\}^i) = \{0\}$$

- From this the following equation is arrived at.

$$[K_T]^i \{\Delta T\}^i = \{R\}^i$$

- where

$$\{\Delta T\}^i = \{T\}^{i+1} - \{T\}^i$$

- Note

$$\{T\}^{i+1} = \{T\}^i + \{\Delta T\}^i$$

STEADY-STATE SOLVER STEPS (Cont.)

- where

$$[K_T]^i \cong \left([K]^i + 4[\mathcal{R}]^i \{T^i + T_{abs}\}^3 \right) - \left[\frac{\partial \{N\}}{\partial \{T\}} \right]^i$$

- This is an iterative process, not linear. Start with $i = 0$. Thus, the first equation to be solved is

$$[K_T]^0 \{\Delta T\}^0 = \{R\}^0$$

- From the solution to this equation the next temperature vector is found from

$$\{T\}^1 = \{T\}^0 + \{\Delta T\}^0$$

STEADY-STATE SOLVER STEPS (Cont.)

- Next, use $i = 1$. The next equation to be solved is

$$[K_T]^1 \{\Delta T\}^1 = \{R\}^1$$

- From the solution to this equation the next temperature vector is found from

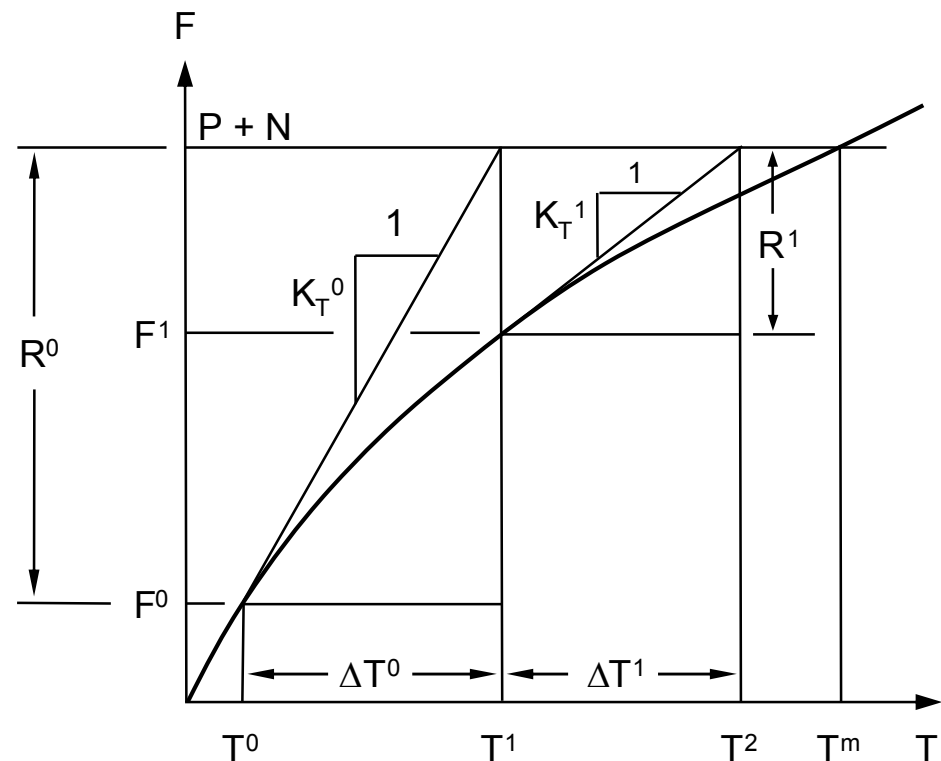
$$\{T\}^2 = \{T\}^1 + \{\Delta T\}^1$$

- This process is repeated until a converged solution is obtained, $\{T\}^m$.

STEADY-STATE SOLVER STEPS (Cont.)

- Graphical representation of the solution process
 - Expression (F) for the thermal load vector due to element temperature

$$F \Rightarrow [K]\{T\} + [\mathcal{R}]\{T + T_{abs}\}^4$$



STEADY-STATE SOLVER STEPS (Cont.)

- Since matrix decomposition is time consuming, MD Nastran does not update the left-hand side matrix at each iteration. The tangential matrix is updated only when the solution fails to converge or the iteration efficiency can be improved. However, the residual vector is updated at each iteration.
- In concert with the Newton-Raphson method, the following options are provided to improve the efficiency of the iteration.
 - Tangential matrix update strategy
 - Line search method
 - Bisection of loads
 - Quasi-Newton (BFGS) updates

STEADY-STATE CONVERGENCE

- Convergence of nonlinear steady-state solution process
 - The form, whose entries are used to specify control of the nonlinear steady-state solution process, is repeated here for convenience. MD Nastran parameter (alpha) names are shown in the MD Patran GUI.

Subcase Parameters

Static Nonlinear Iterations
Number of Load Increments =
NINC

Matrix Update Method:
Automatic
Semi-Automatic
Controlled Iters.

Number of Iterations per Update =
KSTEP

Allowable Iterations per Increment =
MAXITER

Convergence Criteria
☒ Temperature Error
Temperature Tolerance =
EPSU

☒ Load Error
Load Tolerance =
EPSP

☒ Work Error
Work Tolerance =
EPSW

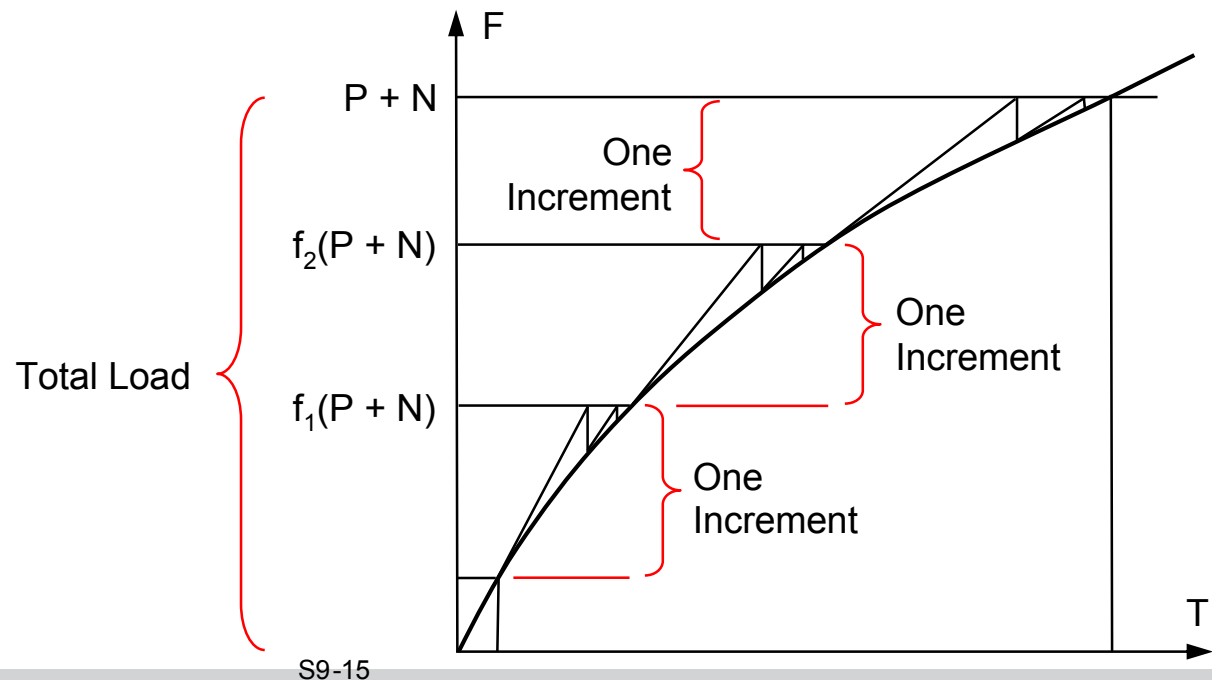
Contact Table...

Solvers / Options...

OK Cancel

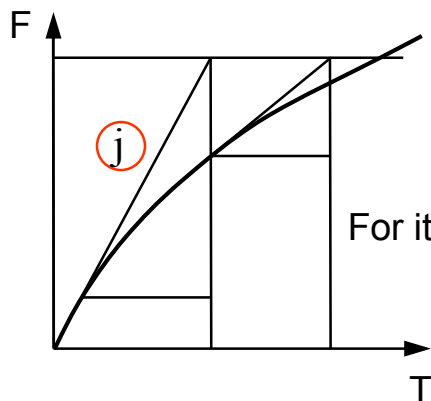
STEADY-STATE CONVERGENCE (Cont.)

- Convergence of nonlinear steady-state solution process
 - Iteration control
 - The incremental and iterative solution processes are controlled by the parameters specified under Static Nonlinear Iterations.
 - For each subcase, load, and SPC temperature, changes are processed incrementally with the number of subdivisions defined by the value of NINC.



STEADY-STATE CONVERGENCE (Cont.)

- Convergence of nonlinear steady-state solution process
 - Iteration control
 - The incremental and iterative solution processes are controlled by the parameters specified under Static Nonlinear Iterations.
 - The Matrix Update Method field is used to specify the tangential matrix update strategy. Three separate options may be selected.
 - Automatic: the program automatically selects the most efficient strategy based on convergence rates. At each iteration, the number of iterations required to converge, as well as the computing time with and without matrix update, are estimated. The tangential matrix is updated if **(a)** the estimated number of iterations to converge exceeds MAXITER, **(b)** the estimated time required for convergence with current matrix exceeds the estimated time to converge with matrix update, or **(c)** the solution diverges. The tangential matrix is also updated on convergence if KSTEP is less than the number of steps required for convergence with the current matrix.



For iteration
"j"

iters to
converge
estimated

Automatic

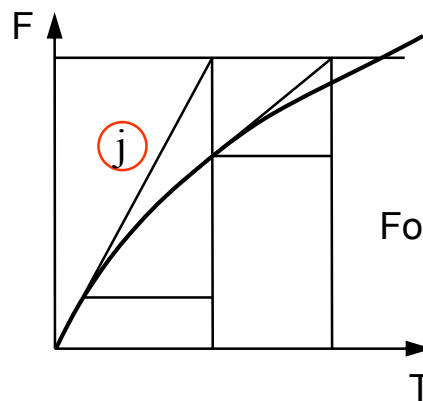
$[K_T]^j$

updated if

- a) Est # iters to converge > MAXITER,
- b) Est time to converge with current > Est time to converge with update, or
- c) Solution process diverges

STEADY-STATE CONVERGENCE (Cont.)

- Convergence of nonlinear steady-state solution process
 - Iteration control
 - The Matrix Update Method field is used to specify the tangential matrix update strategy. Three separate options may be selected.
 - Semi-Automatic: this option is identical to the Automatic option, except that the program updates the tangential matrix after the first iteration.
 - Controlled Iters: the program updates the tangential matrix at every KSTEPth iteration or at convergence if $KSTEP \leq MAXITER$. However, the tangential matrix is never updated if $KSTEP > MAXITER$. Note that the Newton-Raphson method is obtained if $KSTEP = 1$, and the modified Newton-Raphson method is selected by setting $KSTEP = MAXITER$.



For iteration
"j"



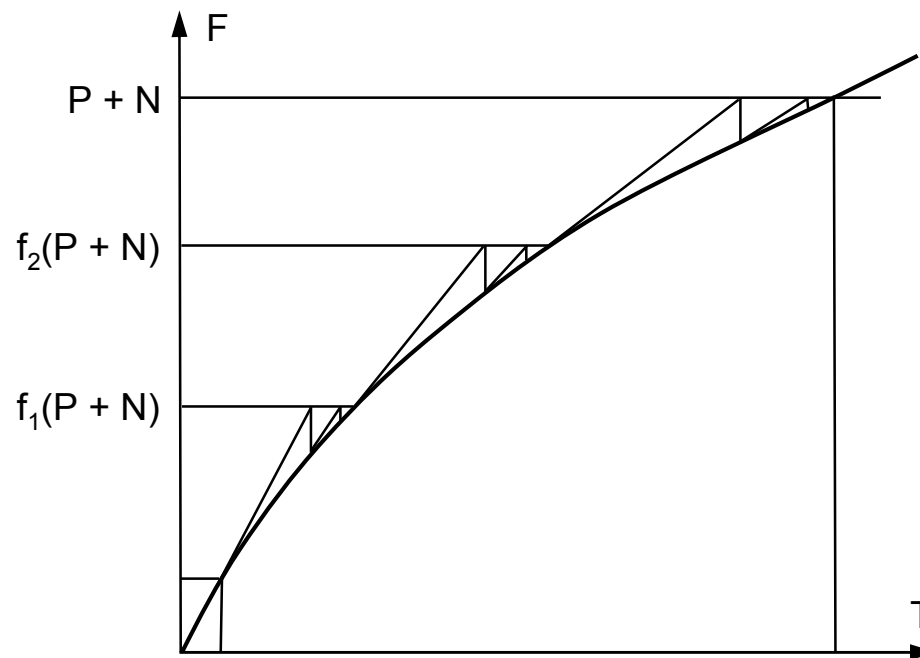
$$[K_T]^j$$

updated every KSTEPth iteration or at
convergence if $KSTEP \leq MAXITER$

Controlled Iters

STEADY-STATE CONVERGENCE (Cont.)

- Convergence of nonlinear steady-state solution process
 - Iteration control
 - The number of iterations for a **load increment** is limited to MAXITER. If the solution does not converge in MAXITER iterations, the current load increment is bisected and the analysis for that increment is repeated.



STEADY-STATE CONVERGENCE (Cont.)

- Convergence of nonlinear steady-state solution process
 - Iteration control
 - The divergence rate is defined as the ratio of energy error before and at the end of the current iteration, $i+1$
 - The divergence parameter for the i -th iteration, $NDIV_i$, is incremented as follows

$$NDIV_{i+1} = NDIV_i + 1, \text{ for less severe error}$$

$$NDIV_{i+1} = NDIV_i + 2, \text{ for more severe error}$$

- This computation is performed for all of the iterations of the current load increment
- The solution for the current load increment is taken as diverged when $NDIV_{\text{final}} \geq \text{MAXDIV}$

STEADY-STATE CONVERGENCE (Cont.)

- Convergence of nonlinear steady-state solution process
 - Iteration output
 - For each iteration, the related output is printed under the following headings:

Parameters	Description
ITERATIONS	Iteration i
EUI	Relative error for temperature
EPI	Relative error for load
EWI	Relative error for energy
LAMBDA	Rate of convergence
DLMAG	Absolute norm of the residual vector
FACTOR	Final value for the line search parameters
E-FIRST	Divergence rate, initial error before line search
E-FINAL	Error at the end of the line search
NQNV	Number of quasi-Newton vectors appended
NLS	Number of line searches performed during the iteration
ENIC	Expected number of iterations for convergence
NDV	Number of occurrences of probable divergence during the iteration
MDV	Number of occurrences of bisection conditions during the iteration

STEADY-STATE CONVERGENCE (Cont.)

- Convergence of nonlinear steady-state solution process
 - Convergence criteria
 - The convergence criteria are characterized by the dimensionless error functions and the convergence tolerances. To ensure accuracy and efficiency, multiple criteria with errors measured about temperatures, loads, and energy are provided.
 - Temperature error function

- Since the error in temperatures is not known, a contraction factor q is introduced to formulate the temperature error function, which is defined as

$$q = \frac{\|T^{i+1} - T^i\|}{\|T^i - T^{i-1}\|} = \frac{\|\Delta T^i\|}{\|\Delta T^{i-1}\|}$$

- The final form of the temperature error function is obtained by introducing a weighted normalization. The result is

$$E_U = \frac{q}{1-q} \frac{\{|\omega|\}^T \cdot \{|\Delta T|\}}{\{|\omega|\}^T \cdot \{|\Delta T|\}}$$

- where the weighting function $\{\omega\}$ is defined as the square root of the diagonal terms of the tangent matrix.

STEADY-STATE CONVERGENCE (Cont.)

- Convergence of nonlinear steady-state solution process

- Convergence criteria

- Load error function

- The load error function is defined as

$$E_P = \frac{\{ |R| \}^T \cdot \{ |T| \}}{\{ |P'| \}^T \cdot \{ |T| \}}$$

- where

$$\{ |P'| \} = \{ |P_{ld}| \} + \{ |\Delta P_{ld}| \}$$

- where $\{P_{ld}\}$ is the applied thermal load at the previous load step, and $\{\Delta P_{ld}\}$ is the incremental load.

- Energy error function

- The energy (or work) error function is defined as

$$E_W = \frac{\{ |R| \}^T \cdot \{ |\Delta T| \}}{\{ |P'| \}^T \cdot \{ |T| \}}$$

STEADY-STATE CONVERGENCE (Cont.)

- Convergence of nonlinear steady-state solution process
 - Convergence criteria
 - Using the error functions
 - At every iteration, error functions are evaluated and the results printed in the convergence table under the headings EUI, EPI, and EWI. The convergence test is performed by comparing the value of the error functions with the convergence tolerances, e.g.
 - $E_U < EPSU$?
 - $E_P < EPSP$?
 - $E_W < EPSW$?
 - where the value of EPSU, EPSP, and EPSW are tolerances specified in the MD Nastran Subcase Parameters form. The solution has converged if these tests are satisfied. Note that only those criteria selected by the user (specified in the Convergence Criteria part of the form) are used to check for convergence. The tolerances should not be too restrictive so that many more iterations are performed than need to be, or too un-restrictive so that there is poor accuracy. It is recommended that the default values be used until better values are found through iteration experience.

RECOMMENDATIONS FOR STEADY-STATE

- Recommendations
 - The following are recommendations, designed to aid the user.
 - Initial temperature estimate
 - For highly nonlinear problems, the iterative solution is sensitive to the initial temperature guess. It is recommended to overshoot (i.e., make a high initial guess) the estimated temperature vector in a radiation-dominated problem.
 - Incremental load
 - Incremental loading reduces the imbalance of the equilibrium equation caused by applied loads. The single-point constraints (temperature specified by SPC in the Bulk Data) and the applied loads (specified by QHBDY, QBDYi, VECFLUX, and QVOL) can be incremented. If the solution takes more iterations than the default values of the maximum number of iterations allowed for convergence (MAXITER), the increment size should be decreased. For linear problems, no incremental load steps are required.
 - Convergence criteria
 - At the beginning stages of a new analysis, it is recommended that the defaults be used for all options. However, all the Error options (Temperature, Load, and Work) may be selected to improve the efficiency of convergence. For problems with poor convergence, the tolerances EPSU, EPSP, and EPSW can be increased within the limits of reasonable accuracy.

RECOMMENDATIONS FOR STEADY-STATE

- Recommendations
 - The following are recommendations, designed to aid the user.
 - Convergence criteria
 - For radiation problems that do not converge, and have an error code SFM 4551, use the full Newton Raphson method so the stiffness matrix will be updated every iteration. This is done using NLPARM, ID, NINC, DT, KMETHOD, ..., where KMETHOD = FNT.
 - SOL153
 - The best method determined through experience is to use the “full” Newton-Raphson method. This involves updating the conduction tangent matrix every iteration. To do this use Matrix Update Method: Controlled Iters, Number of Iterations per Update (KSTEP) = 1. Because the conduction tangent matrix is calculated for each iteration it is necessary to have only one increment, NINC = 1.
 - For problems that are not highly nonlinear it is acceptable to use Matrix Update Method: Automatic.





TRANSIENT THERMAL ANALYSIS





TRANSIENT THERMAL ANALYSIS

- Transient analysis
 - Models are similar to those for steady-state analysis, except must also have
 - Specific heat (heat capacity per unit mass)
 - Density
 - As needed
 - Lumped thermal capacitance
 - Dynamic transfer function
 - The equation for the variation of temperature with time, as shown on the following page, will have to be solved as for steady-state analysis. Also, integrating the set of equations will require specifying parameters, e.g. integration interval/step.

TRANSIENT SOLVER STEPS

- The familiar conduction heat transfer equation is

$$\nabla^2 T + \frac{\dot{q}}{k} = \frac{1}{\alpha} \frac{\partial T}{\partial t}$$

- A transition is made from this equation (strong form) to a variational formulation (weak form). The matrix equation corresponding to the variational formulation is

$$[M]\{\dot{T}\} + [K]\{T\} = \{F\}$$

- The MD Nastran Thermal transient equation derived from this equation is

$$[B]\{\dot{T}\} + [K]\{T\} + [\mathcal{R}]\{T + T_{abs}\}^4 = \{P\} + \{N\}$$

TRANSIENT SOLVER STEPS (Cont.)

- This is a nonlinear matrix equation. It is solved using the Newton-Raphson iteration scheme. A residual load vector function is defined as the **difference** between the applied thermal load vector and the thermal load vector due to element temperature and temperature rate.

$$\{R(t)\} = (\{P\} + \{N\}) - ([B]\{\dot{T}\} + [K]\{T\} + [\mathcal{R}]\{T + T_{abs}\}^4)$$

- The residual load vector function, at a given time, is equal to zero for the solution temperature vector at that time. The task is to determine the solution temperatures and their rates. The residual load vector function is approximated by its first-order Taylor series expansion about the temperature vector from the i-th iteration, for the n+1 time point (the solution is known for time point n).

$$\{R_{n+1}\} \cong \{R_{n+1}\}^i + \left[\frac{\partial \{R\}}{\partial \{T\}} \right]_{n+1}^i (\{T_{n+1}\} - \{T_{n+1}\}^i)$$

TRANSIENT SOLVER STEPS (Cont.)

- The Taylor series expansion is evaluated at the temperatures for the $(i + 1)$ -th iteration, and set equal to zero.

$$\{R_{n+1}\}^{i+1} \cong \{R_{n+1}\}^i + \left[\frac{\partial \{R\}}{\partial \{T\}} \right]_{n+1}^i (\{T_{n+1}\}^{i+1} - \{T_{n+1}\}^i) = \{0\}$$

- From this the following equation is arrived at.

$$[\tilde{K}_T]_{n+1}^i \{\Delta T_{n+1}\}^i = \{R_{n+1}\}^i$$

- where

$$\{\Delta T_{n+1}\}^i = \{T_{n+1}\}^{i+1} - \{T_{n+1}\}^i$$

- Note

$$\{T_{n+1}\}^{i+1} = \{T_{n+1}\}^i + \{\Delta T_{n+1}\}^i$$

TRANSIENT SOLVER STEPS (Cont.)

- where

$$[\tilde{K}_T]_{n+1}^i = \left[\frac{\partial(-\{R\})}{\partial\{T\}} \right]_{n+1}^i = \frac{1}{\theta\Delta t} [B]_{n+1}^i + [K_T]_{n+1}^i \cong \frac{1}{\theta\Delta t} [B]_n + [K_T]_n$$

- and, where the approximation for the time derivative is given by

$$\{\dot{T}_{n+1}\}^i \cong \frac{1}{\theta\Delta t} (\{T_{n+1}\}^i - \{T_n\}) + (1 - 1/\theta) \{\dot{T}_n\}$$

- Another approximation used is

$$[K_T]_n \cong \left([K]_n + 4[\mathcal{R}]_n \{T_n + T_{abs}\}^3 \right) - \left[\frac{\partial\{N\}}{\partial\{T\}} \right]_n$$

TRANSIENT SOLVER STEPS (Cont.)

- The residual load vector function for the i-th iteration is given by

$$\{R_{n+1}\}^i = (\{P_{n+1}\} + \{N_{n+1}\}^i) - ([B]_{n+1}^i \{\dot{T}_{n+1}\}^i + [K]_{n+1}^i \{T_{n+1}\}^i + [\mathcal{R}]_{n+1}^i \{T_{n+1} + T_{abs}\}^{i4})$$

- This process is the same as for the steady-state analysis, this is an iterative process. Start with $i = 0$. Thus, the first equation to be solved is

$$[\tilde{K}_T]_{n+1}^0 \{\Delta T_{n+1}\}^0 = \{R_{n+1}\}^0$$

- From the solution to this equation the next temperature vector is found from

$$\{T_{n+1}\}^1 = \{T_{n+1}\}^0 + \{\Delta T_{n+1}\}^0$$

TRANSIENT SOLVER STEPS (Cont.)

- Next, use $i = 1$. The next equation to be solved is

$$[\tilde{K}_T]_{n+1}^1 \{\Delta T_{n+1}\}^1 = \{R_{n+1}\}^1$$

- From the solution to this equation the next temperature vector is found from

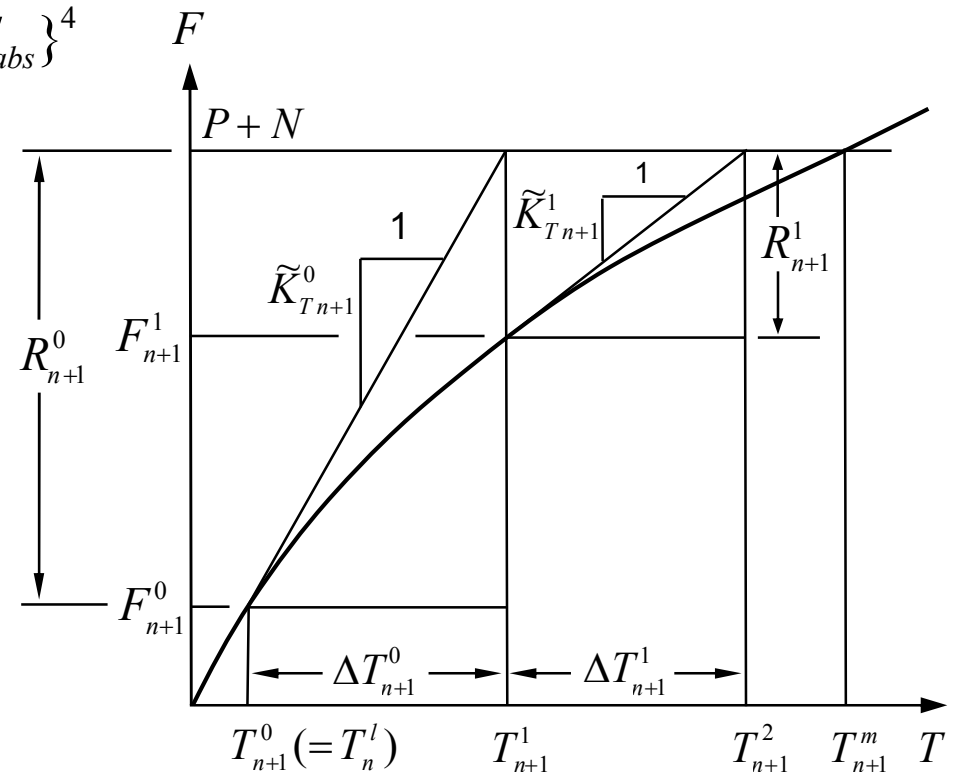
$$\{T_{n+1}\}^2 = \{T_{n+1}\}^1 + \{\Delta T_{n+1}\}^1$$

- This process is repeated until a converged solution is obtained, $\{T_{n+1}\}^m$.
- The process is repeated for the next time point, $n+2$ ($t_{n+2} = t_{n+1} + \Delta t_{n+1}$). Set $\{T_{n+2}\}^0 = \{T_{n+1}\}^m$.

TRANSIENT SOLVER STEPS (Cont.)

- Graphical representation of the solution process
 - Expression (F) for the thermal load vector due to element temperature and temperature rate

$$F \Rightarrow [B] \{\dot{T}\} + [K] \{T\} + [\mathcal{R}] \{T + T_{abs}\}^4$$



TRANSIENT SOLVER STEPS (Cont.)

- Since matrix decomposition is time consuming, MD Nastran does not update the left-hand side matrix at each iteration. The tangential matrix is updated only when the solution fails to converge or the iteration efficiency can be improved. However, the residual vector is updated at each iteration.
- In concert with the Newton-Raphson method, the following options are provided to improve the efficiency of the iteration process.
 - Tangential matrix update strategy
 - Line search method
 - Bisection of loads
 - Quasi-Newton (BFGS) updates



FORMULATION STEP COMMENTS

- These options are specified under Analysis in MD Nastran. In general, if the solution process diverges, a line search algorithm, a bisection of loads method, or the quasi-Newton update method are implemented in an effort to improve the solution obtained. If the solution still fails to converge using all the above methods, the tangential stiffness matrix is updated, and the iteration is resumed. The user may refer to the MD Nastran Handbook for Nonlinear Analysis for a detailed description of the above mentioned algorithms.



TRANSIENT CONVERGENCE

- Convergence of nonlinear transient solution process
 - The form, whos entries are used to specify control of the nonlinear transient solution process, is repeated here for convenience. MD Nastran parameter (alpha) names are shown in the MD Patran GUI

The screenshot shows the 'Subcase Parameters' dialog box with the following settings:

- Initial Time Step = DT
- Number of Time Steps = NDT
- Transient Nonlinear Iterations
 - Matrix Update Method: Adaptive
 - Number of Bisections per Update = KSTEP
 - Allowable Iterations per Time Step = MAXITER
- Convergence Criteria
 - ☒ Temperature Error
 - Temperature Tolerance = EPSU
 - ☒ Load Error
 - Load Tolerance = EPSP
 - ☒ Work Error
 - Work Tolerance = EPSW
- ☐ Fixed Time Steps
- ☒ Exit on Failure to Converge
- Contact Table... (button)
- OK (button)
- Cancel (button)

TRANSIENT CONVERGENCE (Cont.)

- Convergence of nonlinear transient solution process
 - Automatic time stepping
 - An example of adjusting the time step
 - MD Nastran estimates an optimal time step size, and the step size evolves based on the convergence condition. The time step is doubled ($\Delta t_{n+1} = 2\Delta t_n$) as $\{\Delta T_n\}$ ($= \{T_n\} - \{T_{n-1}\}$) becomes small.

$$\frac{\|\{\dot{T}_n\}\|}{\|\{\dot{T}\}\|_{\max}} < UTOL \text{ (default = 0.1)}$$

- where $\|\{\dot{T}\}\|_{\max}$ is the maximum value of the norms computed from the previous time steps, and UTOL is a tolerance on the temperature increment specified in MD Nastran.
- If the temperature increment exceeds the tolerance, a proper time step size can be predicted from the following calculation, where ω_n is the inverse of the characteristic time.

$$\omega_n = \frac{\{\Delta T_n\}^T [K_{T_n}] \{\Delta T_n\}}{\{\Delta T_n\}^T \{\Delta H_n\}}$$

TRANSIENT CONVERGENCE (Cont.)

- Convergence of nonlinear transient solution process
 - Automatic time stepping
 - The next time step is adjusted using

$$\Delta t_{n+1} = f(r_n) \Delta t_n$$

- where $f(r)$ is a scaling factor defined as follows

$$f(r_n) = 0.25, r_n < 0.5 \cdot RB$$

$$f(r_n) = 0.5, 0.5 \cdot RB \leq r_n < RB$$

$$f(r_n) = 1.0, RB \leq r_n < 2.0$$

$$f(r_n) = 2.0, 2.0 \leq r_n < 3.0 / RB$$

$$f(r_n) = 4.0, r_n \geq 3.0 / RB$$

- where

$$r_n = \frac{1}{MSTEP} \left(\frac{2\pi}{\omega_n} \right) \left(\frac{1}{\Delta t_n} \right)$$

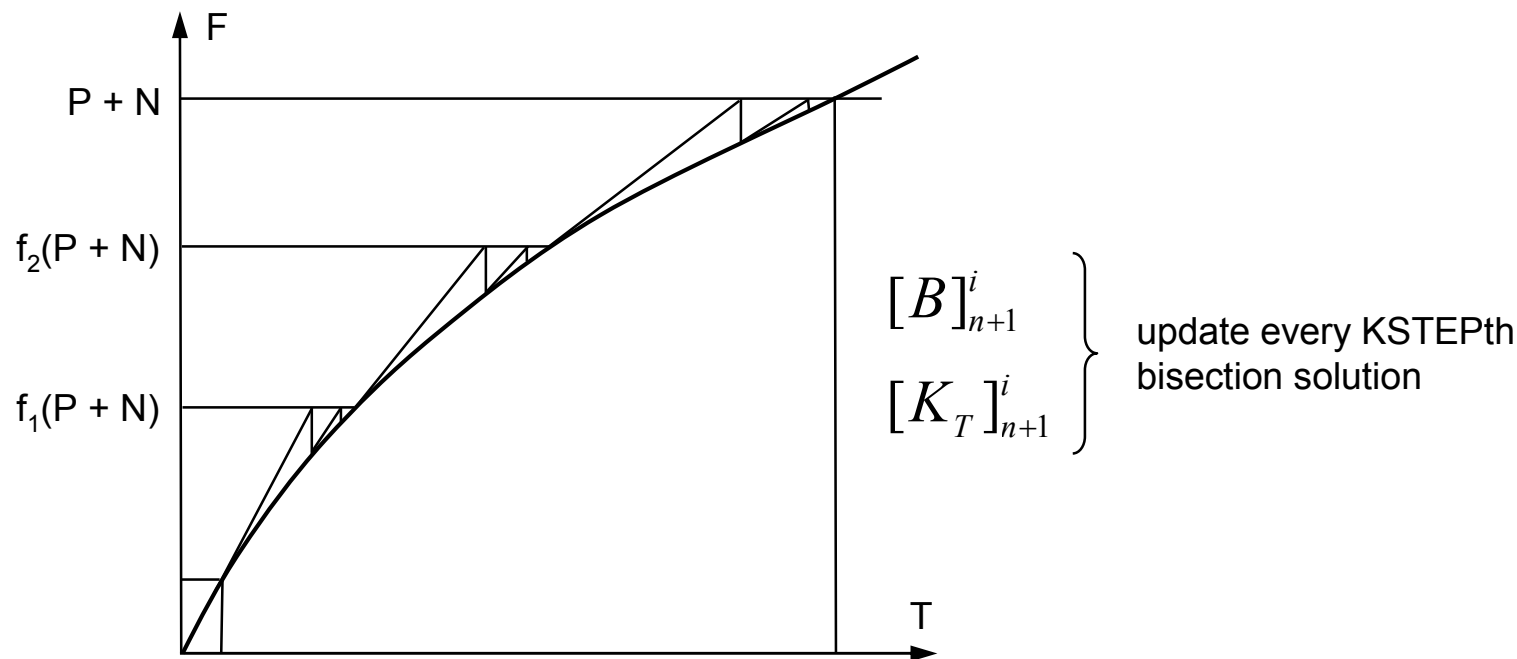
TRANSIENT CONVERGENCE (Cont.)

- Convergence of nonlinear transient solution process
 - Integration and iteration control
 - The initial time increment and the number of time steps are specified by DT and NDT, respectively. So, the duration of the solution of the problem is approximately $NDT \cdot DT$. This time is approximate because the time increment may be adjusted (increased or decreased) during the solution of the problem, so that the number of time steps may not be equal to NDT. However, the total solution time is close to $NDT \cdot DT$.

$$Total\ time \cong NDT \cdot DT$$

TRANSIENT CONVERGENCE (Cont.)

- Convergence of nonlinear transient solution process
 - Integration and iteration control
 - During the bisection process (time step is bisected) the heat capacitance matrix and tangential conduction matrix are updated every KSTEPth converged bisection solution.



TRANSIENT CONVERGENCE (Cont.)

- Convergence of nonlinear transient solution process
 - Integration and iteration control
 - The number of iterations for a **time step** is limited to MAXITER. If MAXITER is specified as negative, the analysis is terminated on the second divergence condition during the same time step or when the solution diverges for five consecutive time steps. If MAXITER is specified as positive, the program computes the best solution and continues the analysis until divergence occurs again. If the solution does not converge in MAXITER iterations, the process is considered divergent. Either bisection or matrix update is activated when the process diverges.

Number of iterations for
time step $\leq$ MAXITER

Process diverges $\Rightarrow$ Bisection of time step or matrix update

TRANSIENT CONVERGENCE (Cont.)

- Convergence of nonlinear transient solution process
 - Integration and iteration control
 - The convergence criteria are defined under Convergence Criteria. Two types of things must be specified
 - Error: Temperature, Load, Work
 - Tolerance: Temperature (EPSU), Load (EPSP), Work (EPSW)
 - The requested criteria (combination of Temperature Error (U), Load Error (P), and Work Error (W)) are satisfied upon convergence. Note that at least two iterations are required to check the temperature convergence criterion.
 - If the bisection option is used, the time step is bisected upon divergence. Otherwise, the left-hand side matrices are updated, and the computation for the current time step is repeated.
 - The BFGS update and the line search process are performed in the same way as in steady state analysis.

TRANSIENT CONVERGENCE (Cont.)

- Convergence of nonlinear transient solution process
 - Integration and iteration control
 - The number of bisections for a load increment is limited to $|MAXBIS|$. Different actions are taken when the solution diverges, depending on the sign of $MAXBIS$. If $MAXBIS$ is positive and the solution does not converge after $MAXBIS$ bisections, the best solution is computed and the analysis is continued to the next time step. If $MAXBIS$ is negative and the solution does not converge in $|MAXBIS|$ bisections, the analysis is terminated.
 - Iteration output
 - At each iteration or time step, the related output data are printed

TRANSIENT CONVERGENCE (Cont.)

- Convergence of nonlinear transient solution process
 - Integration and Iteration control
 - The divergence rate is defined as the ratio of energy error before and at the end of the current iteration, $i+1$
 - The divergence parameter for the i -th iteration, $NDIV_i$, is incremented as follows

$$NDIV_{i+1} = NDIV_i + 1, \text{ for less severe error}$$

$$NDIV_{i+1} = NDIV_i + 2, \text{ for more severe error}$$

- This computation is performed for all of the iterations of the current time step
- The solution for the current time step is taken as diverged when $NDIV_{\text{final}} \geq \text{MAXDIV}$

TRANSIENT CONVERGENCE (Cont.)

- Convergence of nonlinear transient solution process

Parameters	Description
TIME	Cumulative time for the duration of the analysis.
ITER	Iteration count for each time step.
DISP	Relative error in terms of temperatures
LOAD	Relative error in terms of loads
WORK	Relative error in terms of work
LAMDBA(I)	Rate of convergence
DLMAG	Absolute norm of the residual vector
FACTOR	Final value of the line search parameter.
E-FIRST	Divergence rate, initial error before line search.
E-FINAL	Error at the end of line search.
NQNV	Number of quasi-Newton vectors appended.
NLS	Number of line searches performed during the iteration.
ITR DIV	Number of occurrences of divergence detected during the adaptive iteration.
MAT DIV	Number of occurrences of bisection conditions during the iteration.
NO. BIS	Number of bisections executed for the current time interval.
ADJUST	Ratio of time step adjustment relative to DT.

RECOMMENDATIONS FOR TRANSIENT

- Convergence of nonlinear transient solution process
 - Recommendations
 - The following are recommendations designed to aid the user.

- Time step size

- To avoid inaccurate results or results from an unstable analysis, a proper initial time step associated with the spatial mesh size is suggested. The selection criterion is

$$\Delta t|_{initial} = \frac{1}{n} (\Delta x)^2 \frac{\rho C_p}{k}$$

- where $\Delta t|_{initial}$ is the initial time step, n is the modification number of the time scale, Δx is the mesh size (dimension of the smallest element), ρ is the material density, C_p is the specific heat at constant pressure, and k is the thermal conductivity. A suggested value for n is 10. For highly nonlinear problems a small initial time step size is recommended.
- Numerical stability
 - Numerical stability is controlled by the damping parameter η . For linear problems $\eta = 0$ (no numerical damping) is adequate, but for nonlinear problems a larger value of η may be advisable. Increasing the value of η improves numerical stability; however, the solution accuracy can be reduced. The recommended range of values is from 0.0 to 0.1 (the default value is 0.01). The numerical damping is specified using PARAM, NDAMP, (value of η). This is a Bulk Data PARAM entry.



RECOMMENDATIONS FOR TRANSIENT

- Convergence of nonlinear transient solution process
 - Recommendations
 - The following are recommendations designed to aid the user.
 - Initial temperatures and boundary temperatures
 - The specification of initial temperatures and boundary condition temperatures should be consistent. For a given point, the initial temperature should be equal to the boundary condition temperature at $t = 0$.
 - Convergence criteria
 - At the beginning stages of a new analysis, it is recommended that the defaults be used for all options. However, all three Error options (UPW) may be selected to improve the efficiency of convergence. For nonlinear problems with time-varying boundary conditions, the U Error option must be selected, because the large conductance (internally generated) affects the calculations of the PW Error functions. For problems with poor convergence, the tolerances EPSU, EPSP, and EPSW can be increased within the limits of reasonable accuracy.
 - When there is a time varying temperature boundary condition, e.g. time varying ambient temperature, set the convergence criteria to just “U”. An example of this can be seen for Example 7b of the exercises (Chapter 5) in the MD Nastran 2002 Thermal Analysis User’s Guide.

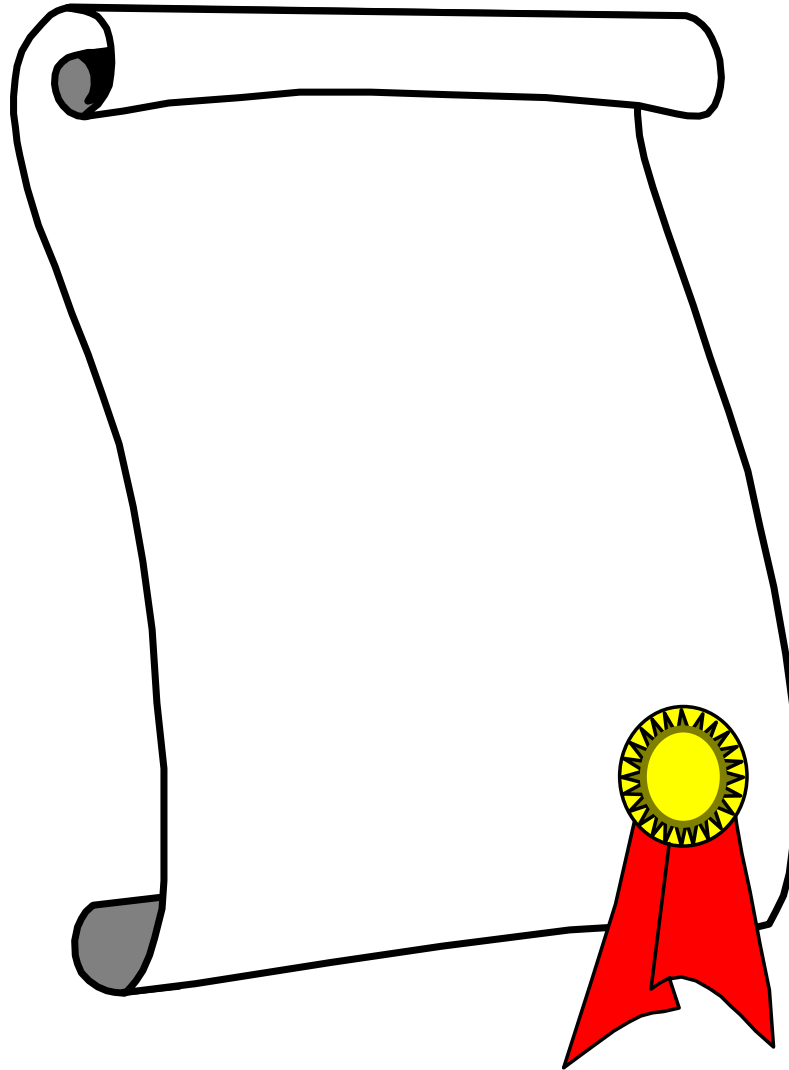


RECOMMENDATIONS FOR TRANSIENT

- Convergence of nonlinear transient solution process
 - Recommendations
 - The following are recommendations designed to aid the user.
 - Fixed time step
 - If a fixed time step is desired, the adaptive time stepping can be deactivated by clicking the checkbox for Fixed Time Steps.



APPENDIX A





MD NASTRAN INPUT FILES

- project1.bdf - linear S.S. conduction
- project1a.bdf - nonlinear S.S. conduction
- project2.bdf - anisotropic material w/ zero deg rot
- project2a.bdf - anisotropic material w/ 45 deg rot
- project3.bdf - convection as a function temperature
- project4.bdf - 1D flow network
- project5.bdf - Coupled Advection
- project5a.bdf - Coupled Advection with refine mesh

MD NASTRAN INPUT FILES (Cont.)

- project6.bdf - transient thermal analysis
- project7.bdf - directional heat load and radiation to space
- project8.bdf - single cavity enclosure radiation

MD NASTRAN INPUT FILES (Cont.)

- project6_adapt.dat - constant time step
- project6_otime.dat - using OTIME in Case Control to output temperatures at a specific time
- project6_sort1.dat - output transient temperature in SORT1 output

MD NASTRAN INPUT FILES (Cont.)

- run1.dat - thermal analysis
- run2.dat - thermal stress run using the punch output from run1
- sol159.dat - Coldstart of transient thermal analysis
- sol159r.dat - Restarting using the database from sol159

MD NASTRAN INPUT FILES (Cont.)

- stress1.dat - thermal analysis
- stress1_r.dat - Restart from the thermal analysis into nonlinear static thermal stress analysis using the database from stress1



CROSS REFERENCES

- MD Nastran 2002 Thermal Analysis User's Guide
- Example 1a - Project 1
- Example 1c - Project 4
- Example 6 - Project 5
- Example 7c - Project 8
- Example 4b - Project 9
- Example 5a - Project 10



NEW MD NASTRAN INPUT FILES

- Thermostat test
- convm_control1.dat
- convm_control2_cool1.dat
- convm_control3b.dat
- axisymmetric view factor
- shield_c4a.bdf
- nas_sphere_geom1_tablem2.bdf
- waveband problem
- glass_solid_6.bdf



NEW MD NASTRAN INPUT FILES (Cont.)

- SOL101 thermal stress run
- t_sph_expand2.dat
- sph_model.txt (includes file for thermal stress)