

ANSYS Thermal Analysis Guide

ANSYS Release 10.0

002184
August 2005

ANSYS, Inc. and
ANSYS Europe,
Ltd. are UL
registered ISO
9001:2000
Companies.

ANSYS Thermal Analysis Guide

ANSYS Release 10.0

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
<http://www.ansys.com>
(T) 724-746-3304
(F) 724-514-9494

Copyright and Trademark Information

© 2005 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, CFX, AUTODYN, and any and all ANSYS, Inc. product and service names are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. Icem CFD is a trademark licensed by ANSYS, Inc. All other trademarks or registered trademarks are the property of their respective owners.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001:2000 Companies.

U.S. GOVERNMENT RIGHTS

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the online documentation in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. The ANSYS third-party software information is also available via download from the Customer Portal on the ANSYS web page. If you are unable to access the third-party legal notices, please contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

| | |
|---|------|
| 1. Analyzing Thermal Phenomena | 1-1 |
| 1.1. How ANSYS Treats Thermal Modeling | 1-1 |
| 1.1.1. Convection | 1-1 |
| 1.1.2. Radiation | 1-1 |
| 1.1.3. Special Effects | 1-2 |
| 1.2. Types of Thermal Analysis | 1-2 |
| 1.3. Coupled-Field Analyses | 1-2 |
| 1.4. About GUI Paths and Command Syntax | 1-2 |
| 2. Steady-State Thermal Analysis | 2-1 |
| 2.1. Available Elements for Thermal Analysis | 2-1 |
| 2.2. Commands Used in Thermal Analyses | 2-4 |
| 2.3. Tasks in a Thermal Analysis | 2-4 |
| 2.4. Building the Model | 2-4 |
| 2.4.1. Creating Model Geometry | 2-4 |
| 2.5. Applying Loads and Obtaining the Solution | 2-5 |
| 2.5.1. Defining the Analysis Type | 2-5 |
| 2.5.2. Applying Loads | 2-5 |
| 2.5.2.1. Constant Temperatures (TEMP) | 2-5 |
| 2.5.2.2. Heat Flow Rate (HEAT) | 2-6 |
| 2.5.2.3. Convections (CONV) | 2-6 |
| 2.5.2.4. Heat Fluxes (HFLUX) | 2-6 |
| 2.5.2.5. Heat Generation Rates (HGEN) | 2-6 |
| 2.5.3. Using Table and Function Boundary Conditions | 2-7 |
| 2.5.4. Specifying Load Step Options | 2-8 |
| 2.5.5. General Options | 2-9 |
| 2.5.6. Nonlinear Options | 2-10 |
| 2.5.6.1. Tracking Convergence Graphically | 2-11 |
| 2.5.7. Output Controls | 2-11 |
| 2.5.8. Defining Analysis Options | 2-12 |
| 2.5.9. Saving the Model | 2-13 |
| 2.5.10. Solving the Model | 2-13 |
| 2.6. Reviewing Analysis Results | 2-13 |
| 2.6.1. Primary data | 2-13 |
| 2.6.2. Derived data | 2-13 |
| 2.6.3. Reading In Results | 2-14 |
| 2.6.4. Reviewing Results | 2-14 |
| 2.7. Example of a Steady-State Thermal Analysis (Command or Batch Method) | 2-15 |
| 2.7.1. The Example Described | 2-15 |
| 2.7.2. The Analysis Approach | 2-16 |
| 2.7.3. Commands for Building and Solving the Model | 2-17 |
| 2.8. Doing a Steady-State Thermal Analysis (GUI Method) | 2-18 |
| 2.9. Doing a Thermal Analysis Using Tabular Boundary Conditions | 2-26 |
| 2.9.1. Running the Sample Problem via Commands | 2-26 |
| 2.9.2. Running the Sample Problem Interactively | 2-27 |
| 2.10. Where to Find Other Examples of Thermal Analysis | 2-30 |
| 3. Transient Thermal Analysis | 3-1 |
| 3.1. Elements and Commands Used in Transient Thermal Analysis | 3-2 |
| 3.2. Tasks in a Transient Thermal Analysis | 3-2 |
| 3.3. Building the Model | 3-2 |
| 3.4. Applying Loads and Obtaining a Solution | 3-2 |

| | |
|--|------------|
| 3.4.1. Defining the Analysis Type | 3-2 |
| 3.4.2. Establishing Initial Conditions for Your Analysis | 3-3 |
| 3.4.2.1. Specifying a Uniform Temperature | 3-3 |
| 3.4.2.2. Specifying a Non-Uniform Starting Temperature | 3-3 |
| 3.4.3. Specifying Load Step Options | 3-4 |
| 3.4.3.1. Defining Time-stepping Strategy | 3-4 |
| 3.4.3.2. General Options | 3-6 |
| 3.4.4. Nonlinear Options | 3-8 |
| 3.4.5. Output Controls | 3-10 |
| 3.5. Saving the Model | 3-11 |
| 3.5.1. Solving the Model | 3-11 |
| 3.6. Reviewing Analysis Results | 3-11 |
| 3.6.1. How to Review Results | 3-12 |
| 3.6.2. Reviewing Results with the General Postprocessor | 3-12 |
| 3.6.3. Reviewing Results with the Time History Postprocessor | 3-12 |
| 3.7. Reviewing Results as Graphics or Tables | 3-13 |
| 3.7.1. Reviewing Contour Displays | 3-13 |
| 3.7.2. Reviewing Vector Displays | 3-13 |
| 3.7.3. Reviewing Table Listings | 3-13 |
| 3.8. Phase Change | 3-13 |
| 3.9. Example of a Transient Thermal Analysis | 3-14 |
| 3.9.1. The Example Described | 3-14 |
| 3.9.2. Example Material Property Values | 3-15 |
| 3.9.3. Example of a Transient Thermal Analysis (GUI Method) | 3-16 |
| 3.9.4. Commands for Building and Solving the Model | 3-16 |
| 3.10. Where to Find Other Examples of Transient Thermal Analysis | 3-17 |
| 4. Radiation | 4-1 |
| 4.1. Analyzing Radiation Problems | 4-1 |
| 4.2. Definitions | 4-1 |
| 4.3. Using LINK31, the Radiation Link Element | 4-2 |
| 4.4. Using the Surface Effect Elements | 4-2 |
| 4.5. Using the AUX12 Radiation Matrix Method | 4-3 |
| 4.5.1. Procedure | 4-3 |
| 4.5.1.1. Defining the Radiating Surfaces | 4-3 |
| 4.5.1.2. Generating the AUX12 Radiation Matrix | 4-5 |
| 4.5.1.3. Using the AUX12 Radiation Matrix in the Thermal Analysis | 4-6 |
| 4.5.2. Recommendations for Using Space Nodes | 4-7 |
| 4.5.2.1. Considerations for the Non-hidden Method | 4-7 |
| 4.5.2.2. Considerations for the Hidden Method | 4-7 |
| 4.5.3. General Guidelines for the AUX12 Radiation Matrix Method | 4-8 |
| 4.6. Using the Radiosity Solver Method | 4-9 |
| 4.6.1. Procedure | 4-9 |
| 4.6.1.1. Defining the Radiating Surfaces | 4-9 |
| 4.6.1.2. Defining Solution Options | 4-10 |
| 4.6.1.3. Defining View Factor Options | 4-11 |
| 4.6.1.4. Calculating and Querying View Factors | 4-12 |
| 4.6.1.5. Defining Load Options | 4-12 |
| 4.6.2. Further Options for Static Analysis | 4-13 |
| 4.7. Advanced Radiosity Options | 4-13 |
| 4.8. Example of a 2-D Radiation Analysis Using the Radiosity Method (Command Method) | 4-17 |
| 4.8.1. The Example Described | 4-17 |
| 4.8.2. Commands for Building and Solving the Model | 4-18 |

| | |
|--|---------|
| 4.9. Example of a 2-D Radiation Analysis Using the Radiosity Method with Decimation and Symmetry (Command Method) | 4-18 |
| 4.9.1. The Example Described | 4-18 |
| 4.9.2. Commands for Building and Solving the Model | 4-19 |
| Index | Index-1 |

List of Figures

| | |
|---|------|
| 2.1. Convergence Norms | 2-11 |
| 2.2. Contour Results Plot | 2-14 |
| 2.3. Vector Display | 2-15 |
| 2.4. Pipe-Tank Junction Model | 2-16 |
| 3.1. Examples of Load vs. Time Curves | 3-1 |
| 3.2. Sample Enthalpy vs. Temperature Curve | 3-14 |
| 4.1. Radiating Surfaces for 3-D and 2-D Models | 4-3 |
| 4.2. Superimposing Elements on Radiating Surfaces | 4-4 |
| 4.3. Orienting the Superimposed Elements | 4-5 |
| 4.4. Decimation | 4-14 |
| 4.5. Planar Reflection | 4-15 |
| 4.6. Cyclic Repetition (Two Repetitions Shown) | 4-15 |
| 4.7. Multiple RSYMM Commands | 4-16 |
| 4.8. Annulus | 4-17 |
| 4.9. Problem Geometry | 4-19 |

List of Tables

| | |
|--|------|
| 2.1. 2-D Solid Elements | 2-2 |
| 2.2. 3-D Solid Elements | 2-2 |
| 2.3. Radiation Link Elements | 2-2 |
| 2.4. Conducting Bar Elements | 2-2 |
| 2.5. Convection Link Elements | 2-2 |
| 2.6. Shell Elements | 2-2 |
| 2.7. Coupled-Field Elements | 2-2 |
| 2.8. Specialty Elements | 2-3 |
| 2.9. Thermal Analysis Load Types | 2-6 |
| 2.10. Load Commands for a Thermal Analysis | 2-7 |
| 2.11. Boundary Condition Type and Corresponding Primary Variable | 2-7 |
| 2.12. Specifying Load Step Options | 2-8 |
| 2.13. Material Properties for the Sample Analysis | 2-16 |

Chapter 1: Analyzing Thermal Phenomena

A *thermal analysis* calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities of interest are:

- The temperature distributions
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes.

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate *thermal stresses* (that is, stresses caused by thermal expansions or contractions).

The following thermal analysis topics are available:

- 1.1. How ANSYS Treats Thermal Modeling
- 1.2. Types of Thermal Analysis
- 1.3. Coupled-Field Analyses
- 1.4. About GUI Paths and Command Syntax

1.1. How ANSYS Treats Thermal Modeling

Only the ANSYS Multiphysics, ANSYS Mechanical, ANSYS Professional, and ANSYS FLOTRAN programs support thermal analyses.

The basis for thermal analysis in ANSYS is a heat balance equation obtained from the principle of conservation of energy. (For details, consult the *ANSYS, Inc. Theory Reference*.) The finite element solution you perform via ANSYS calculates nodal temperatures, then uses the nodal temperatures to obtain other thermal quantities.

The ANSYS program handles all three primary modes of heat transfer: conduction, convection, and radiation.

1.1.1. Convection

You specify convection as a surface load on conducting solid elements or shell elements. You specify the convection film coefficient and the bulk fluid temperature at a surface; ANSYS then calculates the appropriate heat transfer across that surface. If the film coefficient depends upon temperature, you specify a table of temperatures along with the corresponding values of film coefficient at each temperature.

For use in finite element models with conducting bar elements (which do not allow a convection surface load), or in cases where the bulk fluid temperature is not known in advance, ANSYS offers a convection element named LINK34. In addition, you can use the FLOTRAN CFD elements to simulate details of the convection process, such as fluid velocities, local values of film coefficient and heat flux, and temperature distributions in both fluid and solid regions.

1.1.2. Radiation

ANSYS can solve radiation problems, which are nonlinear, in four ways:

- By using the radiation link element, LINK31

- By using surface effect elements with the radiation option (SURF151 in 2-D modeling or SURF152 in 3-D modeling)
- By generating a radiation matrix in AUX12 and using it as a superelement in a thermal analysis.
- By using the Radiosity Solver method.

For detailed information on these methods, see Chapter 4, “Radiation”.

1.1.3. Special Effects

In addition to the three modes of heat transfer, you can account for special effects such as change of phase (melting or freezing) and internal heat generation (due to Joule heating, for example). For instance, you can use the thermal mass element MASS71 to specify temperature-dependent heat generation rates.

1.2. Types of Thermal Analysis

ANSYS supports two types of thermal analysis:

1. *A steady-state thermal analysis* determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.
2. *A transient thermal analysis* determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

1.3. Coupled-Field Analyses

Some types of coupled-field analyses, such as thermal-structural and magnetic-thermal analyses, can represent thermal effects coupled with other phenomena. A coupled-field analysis can use matrix-coupled ANSYS elements, or sequential load-vector coupling between separate simulations of each phenomenon. For more information on coupled-field analysis, see the *ANSYS Coupled-Field Analysis Guide*.

1.4. About GUI Paths and Command Syntax

Throughout this document, you will see references to ANSYS commands and their equivalent GUI paths. Such references use only the command name, because you do not always need to specify all of a command's arguments, and specific combinations of command arguments perform different functions. For complete syntax descriptions of ANSYS commands, consult the *ANSYS Commands Reference*.

The GUI paths shown are as complete as possible. In many cases, choosing the GUI path as shown will perform the function you want. In other cases, choosing the GUI path given in this document takes you to a menu or dialog box; from there, you must choose additional options that are appropriate for the specific task being performed.

For all types of analyses described in this guide, specify the material you will be simulating using an intuitive material model interface. This interface uses a hierarchical tree structure of material categories, which is intended to assist you in choosing the appropriate model for your analysis. See Section 1.1.4.4: Material Model Interface in the *ANSYS Basic Analysis Guide* for details on the material model interface.

Chapter 2: Steady-State Thermal Analysis

The ANSYS Multiphysics, ANSYS Mechanical, ANSYS FLOTRAN, and ANSYS Professional products support steady-state thermal analysis. A steady-state thermal analysis calculates the effects of steady thermal loads on a system or component. Engineer/analysts often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis, performed after all transient effects have diminished.

You can use steady-state thermal analysis to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time. Such loads include the following:

- Convections
- Radiation
- Heat flow rates
- Heat fluxes (heat flow per unit area)
- Heat generation rates (heat flow per unit volume)
- Constant temperature boundaries

A steady-state thermal analysis may be either linear, with constant material properties; or nonlinear, with material properties that depend on temperature. The thermal properties of most material do vary with temperature, so the analysis usually is nonlinear. Including radiation effects also makes the analysis nonlinear.

The following steady-state thermal analysis topics are available:

- 2.1. Available Elements for Thermal Analysis
- 2.2. Commands Used in Thermal Analyses
- 2.3. Tasks in a Thermal Analysis
- 2.4. Building the Model
- 2.5. Applying Loads and Obtaining the Solution
- 2.6. Reviewing Analysis Results
- 2.7. Example of a Steady-State Thermal Analysis (Command or Batch Method)
- 2.8. Doing a Steady-State Thermal Analysis (GUI Method)
- 2.9. Doing a Thermal Analysis Using Tabular Boundary Conditions
- 2.10. Where to Find Other Examples of Thermal Analysis

2.1. Available Elements for Thermal Analysis

The ANSYS and ANSYS Professional programs include about 40 elements (described below) to help you perform steady-state thermal analyses.

For detailed information about the elements, consult the *ANSYS Elements Reference*. That manual organizes element descriptions in numeric order, starting with element LINK1.

Element names are shown in uppercase. All elements apply to both steady-state and transient thermal analyses. SOLID70 also can compensate for mass transport heat flow from a constant velocity field.

Table 2.1 2-D Solid Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|---------|---------|-------------------------|----------------------------|
| PLANE35 | 2-D | Triangle, 6-node | Temperature (at each node) |
| PLANE55 | 2-D | Quadrilateral, 4-node | Temperature (at each node) |
| PLANE75 | 2-D | Harmonic, 4-node | Temperature (at each node) |
| PLANE77 | 2-D | Quadrilateral, 8-node | Temperature (at each node) |
| PLANE78 | 2-D | Harmonic, 8-node | Temperature (at each node) |

Table 2.2 3-D Solid Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|---------|---------|-------------------------|----------------------------|
| SOLID70 | 3-D | Brick, 8-node | Temperature (at each node) |
| SOLID87 | 3-D | Tetrahedron, 10-node | Temperature (at each node) |
| SOLID90 | 3-D | Brick, 20-node | Temperature (at each node) |

Table 2.3 Radiation Link Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|---------|------------|-------------------------|----------------------------|
| LINK31 | 2-D or 3-D | Line, 2-node | Temperature (at each node) |

Table 2.4 Conducting Bar Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|---------|---------|-------------------------|----------------------------|
| LINK32 | 2-D | Line, 2-node | Temperature (at each node) |
| LINK33 | 3-D | Line, 2-node | Temperature (at each node) |

Table 2.5 Convection Link Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|---------|---------|-------------------------|----------------------------|
| LINK34 | 3-D | Line, 2-node | Temperature (at each node) |

Table 2.6 Shell Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|----------|---------|-------------------------|--------------------------------------|
| SHELL57 | 3-D | Quadrilateral, 4-node | Temperature (at each node) |
| SHELL131 | 3-D | Quadrilateral, 4-node | Multiple temperatures (at each node) |
| SHELL132 | 3-D | Quadrilateral, 8-node | Multiple temperatures (at each node) |

Table 2.7 Coupled-Field Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|----------|---------|---------------------------------|---|
| PLANE13 | 2-D | Thermal-stress, 4-node | Temperature, structural displacement, electric potential, magnetic vector potential |
| FLUID116 | 3-D | Thermal-fluid, 2-node or 4-node | Temperature, pressure |

| Element | Dimens. | Shape or Characteristic | DOFs |
|----------|---------|--|--|
| SOLID5 | 3-D | Thermal-stress and thermal-electric, 8-node | Temperature, structural displacement, electric potential, and magnetic scalar potential |
| SOLID98 | 3-D | Thermal-stress and thermal-electric, 10-node | Temperature, structural displacement, electric potential, magnetic vector potential |
| PLANE67 | 2-D | Thermal-electric, 4-node | Temperature, electric potential |
| LINK68 | 3-D | Thermal-electric, 2-node | Temperature, electric potential |
| SOLID69 | 3-D | Thermal-electric, 8-node | Temperature, electric potential |
| SHELL157 | 3-D | Thermal-electric, 4-node | Temperature, electric potential |
| TARGE169 | 2-D | Target segment element | Temperature, structural displacement |
| TARGE170 | 3-D | Target segment element | Temperature, structural displacement |
| CONTA171 | 2-D | Surface-to-surface contact element, 2-node | Temperature, structural displacement |
| CONTA172 | 2-D | Surface-to-surface contact element, 3-node | Temperature, structural displacement |
| CONTA173 | 3-D | Surface-to-surface contact element, 4-node | Temperature, structural displacement |
| CONTA174 | 3-D | Surface-to-surface contact element, 8-node | Temperature, structural displacement |
| CONTA175 | 2-D/3-D | Node-to-surface contact element, 1 node | Temperature, structural displacement, electric potential, vector magnetic potential, scalar magnetic potential (KEYOPT-dependent). You cannot couple magnetic potential with any other DOFs. |

Table 2.8 Specialty Elements

| Element | Dimens. | Shape or Characteristic | DOFs |
|----------|------------------|---|--|
| MASS71 | 1-D, 2-D, or 3-D | Mass, one-node | Temperature |
| COMBIN37 | 1-D | Control element, 4-node | Temperature, structural displacement, rotation, pressure |
| SURF151 | 2-D | Surface effect element, 2-node to 4-node | Temperature |
| SURF152 | 3-D | Surface effect element, 4-node to 9-node | Temperature |
| MATRIX50 | [1] | Matrix or radiation matrix element, no fixed geometry | [1] |
| INFIN9 | 2-D | Infinite boundary, 2-node | Temperature, magnetic vector potential |
| INFIN47 | 3-D | Infinite boundary, 4-node | Temperature, magnetic vector potential |
| COMBIN14 | 1-D, 2-D, or 3-D | Combination element, 2-node | Temperature, structural displacement, rotation, pressure |
| COMBIN39 | 1-D | Combination element, 2-node | Temperature, structural displacement, rotation, pressure |
| COMBIN40 | 1-D | Combination element, 2-node | Temperature, structural displacement, rotation, pressure |

1. As determined from the element types included in this superelement.

2.2. Commands Used in Thermal Analyses

Section 2.7: Example of a Steady-State Thermal Analysis (Command or Batch Method) and Section 2.8: Doing a Steady-State Thermal Analysis (GUI Method) show you how to perform an example steady-state thermal analysis via command and via GUI, respectively.

For detailed, alphabetized descriptions of the ANSYS commands, see the *ANSYS Commands Reference*.

2.3. Tasks in a Thermal Analysis

The procedure for doing a thermal analysis involves three main tasks:

- Build the model.
- Apply loads and obtain the solution.
- Review the results.

The next few topics discuss what you must do to perform these steps. First, the text presents a general description of the tasks required to complete each step. An example follows, based on an actual steady-state thermal analysis of a pipe junction. The example walks you through doing the analysis by choosing items from ANSYS GUI menus, then shows you how to perform the same analysis using ANSYS commands.

2.4. Building the Model

To build the model, you specify the jobname and a title for your analysis. Then, you use the ANSYS preprocessor (PREP7) to define the element types, element real constants, material properties, and the model geometry. (These tasks are common to most analyses. The *ANSYS Modeling and Meshing Guide* explains them in detail.)

For a thermal analysis, you also need to keep these points in mind:

- To specify element types, you use either of the following:
Command(s): ET
GUI: Main Menu > Preprocessor > Element Type > Add/Edit/Delete
- To define constant material properties, use either of the following:
Command(s): MP
GUI: Main Menu > Preprocessor > Material Props > Material Models > Thermal
- To define temperature-dependent properties, you first need to define a table of temperatures. Then, define corresponding material property values. To define the temperatures table, use either of the following:
Command(s): MPTEMP or MPTGEN, and to define corresponding material property values, use **MPDATA**.
GUI: Main Menu > Preprocessor > Material Props > Material Models > Thermal

Use the same GUI menu choices or the same commands to define temperature-dependent film coefficients (HF) for convection.

Caution: If you specify temperature-dependent film coefficients (HF) in polynomial form, you should specify a temperature table before you define other materials having constant properties.

2.4.1. Creating Model Geometry

There is no single procedure for building model geometry; the tasks you must perform to create it vary greatly, depending on the size and shape of the structure you wish to model. Therefore, the next few paragraphs provide

only a generic overview of the tasks typically required to build model geometry. For more detailed information about modeling and meshing procedures and techniques, see the *ANSYS Modeling and Meshing Guide*.

The first step in creating geometry is to build a solid model of the item you are analyzing. You can use either predefined geometric shapes such as circles and rectangles (known within ANSYS as *primitives*), or you can manually define nodes and elements for your model. The 2-D primitives are called *areas*, and 3-D primitives are called *volumes*.

Model dimensions are based on a global coordinate system. By default, the global coordinate system is Cartesian, with X, Y, and Z axes; however, you can choose a different coordinate system if you wish. Modeling also uses a working plane - a movable reference plane used to locate and orient modeling entities. You can turn on the working plane grid to serve as a "drawing tablet" for your model.

You can tie together, or sculpt, the modeling entities you create via Boolean operations. For example, you can add two areas together to create a new, single area that includes all parts of the original areas. Similarly, you can overlay an area with a second area, then subtract the second area from the first; doing so creates a new, single area with the overlapping portion of area 2 removed from area 1.

Once you finish building your solid model, you use meshing to "fill" the model with nodes and elements. For more information about meshing, see the *ANSYS Modeling and Meshing Guide*.

2.5. Applying Loads and Obtaining the Solution

You must define the analysis type and options, apply loads to the model, specify load step options, and initiate the finite element solution.

2.5.1. Defining the Analysis Type

During this phase of the analysis, you must first define the analysis type:

- In the GUI, choose menu path **Main Menu Solution > Analysis Type > New Analysis > Steady-state (static)**.
- If this is a new analysis, issue the command **ANTYPE,STATIC,NEW**.
- If you want to restart a previous analysis (for example, to specify additional loads), issue the command **ANTYPE,STATIC,REST**. You can restart an analysis only if the files **Jobname.ESAV** and **Jobname.DB** from the previous run are available.

2.5.2. Applying Loads

You can apply loads either on the solid model (keypoints, lines, and areas) or on the finite element model (nodes and elements). You can specify loads using the conventional method of applying a single load individually to the appropriate entity, or you can apply complex boundary conditions as tabular boundary conditions (see Section 2.5.14: Applying Loads Using TABLE Type Array Parameters in the *ANSYS Basic Analysis Guide*) or as function boundary conditions (see Section 2.5.15: Applying Loads Using Function Boundary Conditions).

You can specify five types of thermal loads:

2.5.2.1. Constant Temperatures (TEMP)

These are DOF constraints usually specified at model boundaries to impose a known, fixed temperature. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels TBOT, TE2, TE3, . . . , TTOP instead of TEMP when defining DOF constraints.

2.5.2.2. Heat Flow Rate (HEAT)

These are concentrated nodal loads. Use them mainly in line-element models (conducting bars, convection links, etc.) where you cannot specify convections and heat fluxes. A positive value of heat flow rate indicates heat flowing into the node (that is, the element gains heat). If both TEMP and HEAT are specified at a node, the temperature constraint prevails. For SHELL131 and SHELL132 elements with KEYOPT(3) = 0 or 1, use the labels HBOT, HE2, HE3, . . . , HTOP instead of HEAT when defining nodal loads.

Note — If you use nodal heat flow rate for solid elements, you should refine the mesh around the point where you apply the heat flow rate as a load, especially if the elements containing the node where the load is applied have widely different thermal conductivities. Otherwise, you may get a non-physical range of temperature. Whenever possible, use the alternative option of using the heat generation rate load or the heat flux rate load. These options are more accurate, even for a reasonably coarse mesh.

2.5.2.3. Convections (CONV)

Convections are surface loads applied on exterior surfaces of the model to account for heat lost to (or gained from) a surrounding fluid medium. They are available only for solids and shells. In line-element models, you can specify convections through the convection link element (LINK34).

2.5.2.4. Heat Fluxes (HFLUX)

Heat fluxes are also surface loads. Use them when the amount of heat transfer across a surface (heat flow rate per area) is known, or is calculated through a FLOTRAN CFD analysis. A positive value of heat flux indicates heat flowing into the element. Heat flux is used only with solids and shells. An element face may have either CONV or HFLUX (but not both) specified as a surface load. If you specify both on the same element face, ANSYS uses what was specified *last*.

2.5.2.5. Heat Generation Rates (HGEN)

You apply heat generation rates as "body loads" to represent heat generated within an element, for example by a chemical reaction or an electric current. Heat generation rates have units of heat flow rate per unit volume.

Table 2.9: "Thermal Analysis Load Types" below summarizes the types of thermal analysis loads.

Table 2.9 Thermal Analysis Load Types

| Load Type | Category | Cmd Family | GUI Path |
|---|---------------|------------|---|
| Temperature (TEMP, TBOT, TE2, TE3, . . . , TTOP) | Constraints | D | Main Menu > Solution > Define Loads > Apply > Thermal > Temperature |
| Heat Flow Rate (HEAT, HBOT, HE2, HE3, . . . , HTOP) | Forces | F | Main Menu > Solution > Define Loads > Apply > Thermal > Heat Flow |
| Convection (CONV), Heat Flux (HFLUX) | Surface Loads | SF | Main Menu > Solution > Define Loads > Apply > Thermal > Convection Main Menu > Solution > Define Loads > Apply > Thermal > Heat Flux |
| Heat Generation Rate (HGEN) | Body Loads | BF | Main Menu > Solution > Define Loads > Apply > Thermal > Heat Generat |

Table 2.10: “Load Commands for a Thermal Analysis” lists all the commands you can use to apply, remove, operate on, or list loads in a thermal analysis.

Table 2.10 Load Commands for a Thermal Analysis

| Load Type | Solid or FE Model | Entity | Apply | Delete | List | Operate | Settings |
|--------------------------|-------------------|-----------|------------|----------------|----------------|----------------|---|
| Temperature | Solid Model | Keypoints | DK | DKDELE | DKLIST | DTRAN | - |
| " | Finite Element | Nodes | D | DDELE | DLIST | DSCALE | DCUM, TUNIF |
| Heat Flow Rate | Solid Model | Keypoints | FK | FKDELE | FKLIST | FTRAN | - |
| " | Finite Element | Nodes | F | FDELE | FLIST | FSCALE | FCUM |
| Convection, Heat Flux | Solid Model | Lines | SFL | SFLDELE | SFLLIST | SFTRAN | SFGRAD |
| " | Solid Model | Areas | SFA | SFADELE | SFALIST | SFTRAN | SFGRAD |
| " | Finite Element | Nodes | SF | SFDELE | SFLIST | SFSCALE | SFGRAD, SFCUM |
| " | Finite Element | Elements | SFE | SFEDELE | SFELIST | SFSCALE | SFBEAM, SFCUM, SFFUN, SFGRAD |
| Heat Generation Rate | Solid Model | Keypoints | BFK | BFKDELE | BFKLIST | BFTRAN | - |
| " | Solid Model | Lines | BFL | BFLDELE | BFLLIST | BFTRAN | - |
| " | Solid Model | Areas | BFA | BFADELE | BFALIST | BFTRAN | - |
| " | Solid Model | Volumes | BFV | BFVDELE | BFVLIST | BFTRAN | - |
| " | Finite Element | Nodes | BF | BFDELE | BFLIST | BFSCALE | BFCUM |
| " | " | Elements | BFE | BFEDELE | BFELIST | BFSCALE | BFCUM |

You access all loading operations (except List; see below) through a series of cascading menus. From the Solution Menu, you choose the operation (apply, delete, etc.), then the load type (temperature, etc.), and finally the object to which you are applying the load (keypoint, node, etc.).

For example, to apply a temperature load to a keypoint, follow this GUI path:

GUI:

Main Menu> Solution> Define Loads> Apply> Thermal> Temperature> On Keypoints

2.5.3. Using Table and Function Boundary Conditions

In addition to the general rules for applying tabular boundary conditions, some details are information is specific to thermal analyses. This information is explained in this section. For detailed information on defining table array parameters (both interactively and via command), see the *ANSYS APDL Programmer's Guide*.

There are no restrictions on element types.

Table 2.11: “Boundary Condition Type and Corresponding Primary Variable” lists the primary variables that can be used with each type of boundary condition in a thermal analysis.

Table 2.11 Boundary Condition Type and Corresponding Primary Variable

| Thermal Boundary Condition | Cmd. Family | Primary Variable |
|----------------------------|-------------|---------------------|
| Fixed Temperature | D | TIME, X, Y, Z |
| Heat Flow | F | TIME, X, Y, Z, TEMP |

| Thermal Boundary Condition | Cmd. Family | Primary Variable |
|---|-------------|-------------------------------|
| Film Coefficient (Convection) | SF | TIME, X, Y, Z, TEMP, VELOCITY |
| Bulk Temperature (Convections) | SF | TIME, X, Y, Z |
| Heat Flux | SF | TIME, X, Y, Z, TEMP |
| Heat Generation | BF | TIME, X, Y, Z, TEMP |
| Fluid Element (FLUID116) Boundary Condition | | |
| Flow Rate | SFE | TIME |
| Pressure | D | TIME, X, Y, Z |

An example of how to run a steady-state thermal analysis using tabular boundary conditions is described in Section 2.9: Doing a Thermal Analysis Using Tabular Boundary Conditions.

For more flexibility defining arbitrary heat transfer coefficients, use function boundary conditions. For detailed information on defining functions and applying them as loads, see Section 2.5.15: Applying Loads Using Function Boundary Conditions in the *ANSYS Basic Analysis Guide*. Additional primary variables that are available using functions are listed below.

- Tsurf (TS) (element surface temperature for SURF151 or SURF152 elements)
- Density (material property DENS)
- Specific heat (material property C)
- Thermal conductivity (material property KXX)
- Thermal conductivity (material property KYY)
- Thermal conductivity (material property KZZ)
- Viscosity (material property VISC)
- Emissivity (material property EMIS)

2.5.4. Specifying Load Step Options

For a thermal analysis, you can specify general options, nonlinear options, and output controls.

Table 2.12 Specifying Load Step Options

| Option | Command | GUI Path |
|------------------------------------|---------|--|
| General Options | | |
| Time | TIME | Main Menu> Solution> Load Step Opts> Time/Frequenc> Time-Time Step |
| Number of Time Steps | NSUBST | Main Menu> Solution> Load Step Opts> Time/Frequenc> Time and Substps |
| Time Step Size | DELTIM | Main Menu> Solution> Load Step Opts> Time/Frequenc> Time-Time Step |
| Stepped or Ramped Loads | KBC | Main Menu> Solution> Load Step Opts> Time/Frequenc> Time-Time Step |
| Nonlinear Options | | |
| Max. No. of Equilibrium Iterations | NEQIT | Main Menu> Solution> Load Step Opts> Nonlinear> Equilibrium Iter |

| Option | Command | GUI Path |
|----------------------------------|---------------|---|
| Automatic Time Stepping | AUTOTS | Main Menu> Solution> Load Step Opts> Time/Frequenc> Time-Time Step |
| Convergence Tolerances | CNVTOL | Main Menu> Solution> Load Step Opts> Nonlinear> Convergence Crit |
| Solution Termination Options | NCNV | Main Menu> Solution> Load Step Opts> Nonlinear> Criteria to Stop |
| Line Search Option | LNSRCH | Main Menu> Solution> Load Step Opts> Nonlinear> Line Search |
| Predictor-Corrector Option | PRED | Main Menu> Solution> Load Step Opts> Nonlinear> Predictor |
| Output Control Options | | |
| Printed Output | OUTPR | Main Menu> Solution> Load Step Opts> Output Ctrl> Solu Printout |
| Database and Results File Output | OUTRES | Main Menu> Solution> Load Step Opts> Output Ctrl> DB/Results File |
| Extrapolation of Results | ERESX | Main Menu> Solution> Load Step Opts> Output Ctrl> Integration Pt |

2.5.5. General Options

General options include the following:

- *The TIME option.*

This option specifies time at the end of the load step. Although time has no physical meaning in a steady-state analysis, it provides a convenient way to refer to load steps and substeps.

The default time value is 1.0 for the first load step and 1.0 plus the previous time for subsequent load steps.

- *The number of substeps per load step, or the time step size.*

A nonlinear analysis requires multiple substeps within each load step. By default, the program uses one substep per load step.

- *Stepped or ramped loads.*

If you apply stepped loads, the load value remains constant for the entire load step.

If you ramp loads (the default), the load values increment linearly at each substep of the load step.

- *Monitor Results in Real Time*

The **NLHIST** command allows you to monitor results of interest in real time during a solution. Before starting the solution, you can request nodal data such as temperatures or heat flows. You can also request element nodal data such as thermal gradients and fluxes at specific elements to be graphed. The result data are written to a file named **Jobname.nlh**. Nodal results and contact results are monitored at every converged substep while element nodal data are written as specified via the **OUTRES** setting. You can also track results during batch runs. To execute, either:

- Access the ANSYS Launcher and select **File Tracking** from the **Tools** menu.

Or...

- Type **nlhist100** in the command line.

Use the supplied file browser to navigate to your **Jobname.nlh** file, and click on it to invoke the tracking utility. You can use this utility to read the file at any time, even after the solution is complete.

To use this option, use either of these methods:

Command(s): NLHIST

GUI: Main Menu > Solution > Results Tracking

2.5.6. Nonlinear Options

Specify nonlinear load step options if nonlinearities are present. Nonlinear options include the following:

- *Number of equilibrium iterations.*

This option specifies the maximum allowable number of equilibrium iterations per substep. The default value of 25 should be enough for most nonlinear thermal analyses.

- *Automatic time stepping.*

For nonlinear problems, automatic time stepping determines the amount of load increment between substeps, to maintain solution stability and accuracy.

- *Convergence tolerances.*

ANSYS considers a nonlinear solution to be converged whenever specified convergence criteria are met. Convergence checking may be based on temperatures, heat flow rates, or both. You specify a typical value for the desired item (*VALUE* field in the **CNVTOL** command) and a tolerance about the typical value (*TOLER* field). The convergence criterion is then given by $VALUE \times TOLER$. For instance, if you specify 500 as the typical value of temperature and 0.001 as the tolerance, the convergence criterion for temperature is 0.5 degrees.

For temperatures, ANSYS compares the change in nodal temperatures between successive equilibrium iterations ($\Delta T = T_i - T_{i-1}$) to the convergence criterion. Using the above example, the solution is converged when the temperature difference *at every node* from one iteration to the next is less than 0.5 degrees.

For heat flow rates, ANSYS compares the out-of-balance load vector to the convergence criterion. The out-of-balance load vector represents the difference between the applied heat flows and the internal (calculated) heat flows.

- *Termination settings for unconverged solutions.*

If ANSYS cannot converge the solution within the specified number of equilibrium iterations, ANSYS either stops the solution or moves on to the next load step, depending on what you specify as the stopping criteria.

- *Line search.*

This option enables ANSYS to perform a line search with the Newton-Raphson method.

- *Predictor-corrector option.*

This option activates the predictor-corrector option for the degree of freedom solution at the first equilibrium iteration of each substep.

2.5.6.1. Tracking Convergence Graphically

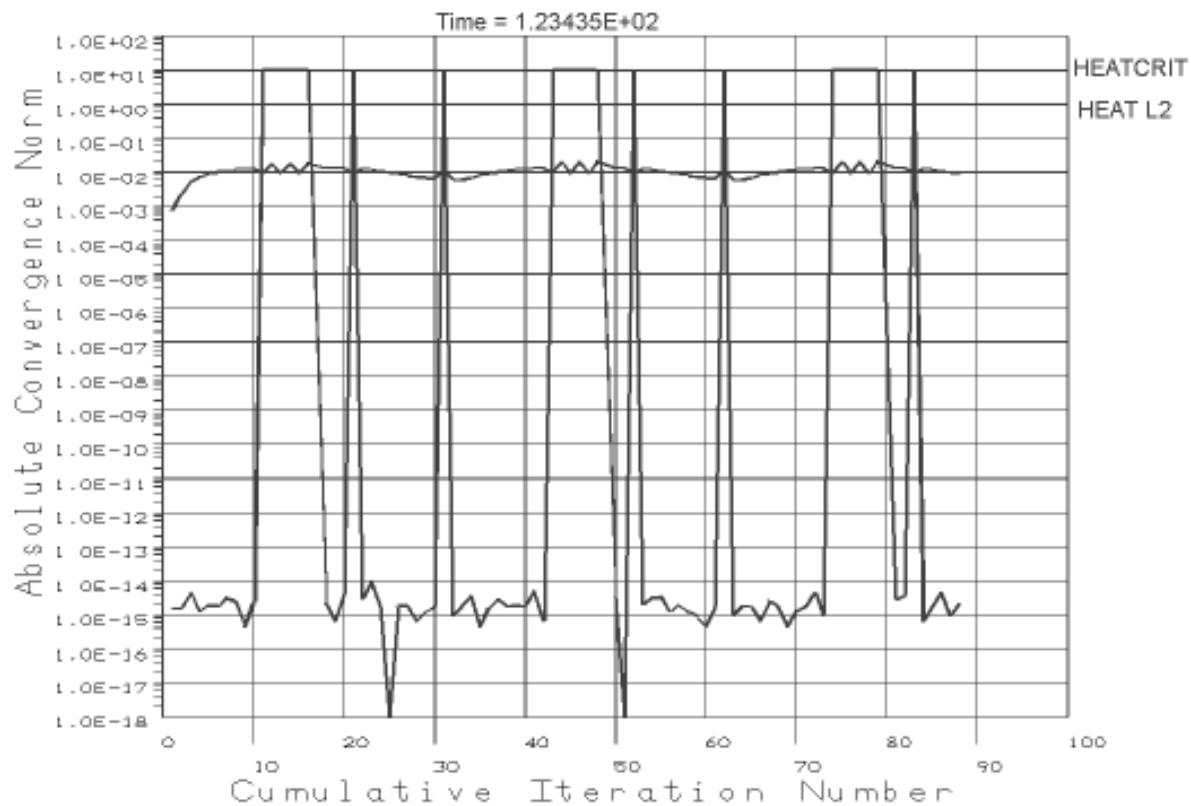
As a nonlinear thermal analysis proceeds, ANSYS computes convergence norms with corresponding convergence criteria each equilibrium iteration. Available in both batch and interactive sessions, the Graphical Solution Tracking (GST) feature displays the computed convergence norms and criteria while the solution is in process. By default, GST is ON for interactive sessions and OFF for batch runs. To turn GST on or off, use either of the following:

Command(s): /GST

GUI: Main Menu> Solution> Load Step Opts> Output Ctrls> Grph Solu Track

Figure 2.1: “Convergence Norms” below shows a typical GST display.

Figure 2.1 Convergence Norms



Displayed by the Graphical Solution Tracking (GST) Feature

2.5.7. Output Controls

The third class of load step options enables you to control output. The options are as follows:

- *Control printed output.*

This option enables you to include any results data in the printed output file (**Jobname.OUT**).

- *Control database and results file output*

This option controls what data ANSYS writes to the results file (**Jobname.RTH**).

- *Extrapolate results.*

Use this option to review element integration point results by copying them to the nodes instead of extrapolating them. (Extrapolation is the default.)

2.5.8. Defining Analysis Options

Next, you define the analysis options. Possible options include:

- *The Newton-Raphson option* (used only in nonlinear analyses). This option specifies how often the tangent matrix is updated during solution. You can specify one of these values:
 - Program-chosen (default; recommended for thermal analysis)
 - Full
 - Modified
 - Initial conductivity

Note — For single-field nonlinear thermal analysis, ANSYS will always use the full Newton-Raphson algorithm.

To use this option, or to turn Newton-Raphson adaptive descent on or off (valid only for the full Newton-Raphson option), use either of these methods:

Command(s): NROPT

GUI: Main Menu > Solution > Analysis Type > Analysis Options

- *Selecting an equation solver*. You can specify any of these values:
 - Sparse solver (default for static and full transient analyses)
 - Frontal solver
 - Jacobi Conjugate Gradient (JCG) solver
 - JCG out-of-memory solver
 - Incomplete Cholesky Conjugate Gradient (ICCG) solver
 - Preconditioned Conjugate Gradient solver (PCG)
 - PCG out-of-memory solver
 - Algebraic Multigrid (AMG) solver
 - Iterative (automatic solver selection option)

Note — The AMG solver is part of Parallel Performance for ANSYS, which is a separately-licensed product. See Chapter 15, “Improving ANSYS Performance and Parallel Performance for ANSYS” in the *ANSYS Advanced Analysis Techniques Guide* for more information about using the AMG solver.

To select an equation solver, use either of the following:

Command(s): EQSLV

GUI: Main Menu > Solution > Analysis Type > Analysis Options

Note — You can use the Iterative (Fast Solution) option for any thermal element except superelements (i.e., as created by AUX12 for radiation analysis). It is not recommended for heat transfer problems involving phase change (use either the sparse or frontal solver for these cases). This option suppresses the creation of the **Jobname.EMAT** and **Jobname.EROT** files.

- *Specifying a temperature offset.* This is the difference in degrees between absolute zero and the zero of the temperature system being used. The offset temperature is included internally in the calculations of pertinent elements (such as elements with radiation effects or creep capabilities). It allows you to input temperatures in degrees Centigrade (instead of Kelvin) or degrees Fahrenheit (instead of Rankine), and then postprocess temperatures in like fashion. For more information, see Chapter 4, "Radiation".

To specify the offset temperature, use either of the following:

Command(s): TOFFST

GUI: Main Menu> Solution> Analysis Type> Analysis Options

2.5.9. Saving the Model

After you have specified the load step and analysis options, you should save a backup copy of the database to prevent your model from being lost if your computer system should fail. Should you ever need to retrieve your model, do so via either of the following:

Command(s): RESUME

GUI: Utility Menu> File> Resume Jobname.db

Utility Menu> File> Resume from

2.5.10. Solving the Model

To start the solution, use either of the following:

Command(s): SOLVE

GUI: Main Menu> Solution> Solve> Current LS

2.6. Reviewing Analysis Results

ANSYS writes the results from a thermal analysis to the thermal results file, **Jobname.RTH**. Results contain the following data:

2.6.1. Primary data

- Nodal temperatures (TEMP, TBOT, TE2, TE3, . . . TTOP)

2.6.2. Derived data

- Nodal and element thermal fluxes (TFX, TFY, TFZ, TFSUM)
- Nodal and element thermal gradients (TGX, TGY, TGZ, TGSUM)
- Element heat flow rates
- Nodal reaction heat flow rates
- ...etc.

You can review these results using the general postprocessor, POST1 (The GUI menu path is **Main Menu> General Postproc.**) Some typical postprocessing operations for a thermal analysis are described below. For a complete description of all postprocessing functions, see the *ANSYS Basic Analysis Guide*.

Note — To review results in the general postprocessor, the ANSYS database must contain the same model for which the solution was calculated. (If necessary, use the resume operation or issue the **RESUME** command to retrieve the model.) In addition, the results file, **Jobname.RTH**, must be available.

2.6.3. Reading In Results

After you enter POST1, read in results for the desired load step and substep. To do so, use either of the following:

Command(s): SET

GUI: Main Menu > General Postproc > Read Results > By Load Step

You can choose the load step to be read by number, or you can request that the first load step be read, the last load step, the next load step, etc. If you are using the GUI, a dialog box presents you with options for choosing the load step to be read.

The *TIME* field enables you to identify the results data by time. If you specify a time value for which no results are available, ANSYS performs linear interpolation to calculate the results at that time.

2.6.4. Reviewing Results

Once you have read results into memory, you can use the ANSYS graphics displays and tables to review them. To display your results, use the following menu paths. Equivalent commands are shown in parentheses.

For contour displays:

Command(s): PLESOL , PLETAB, PLNSOL

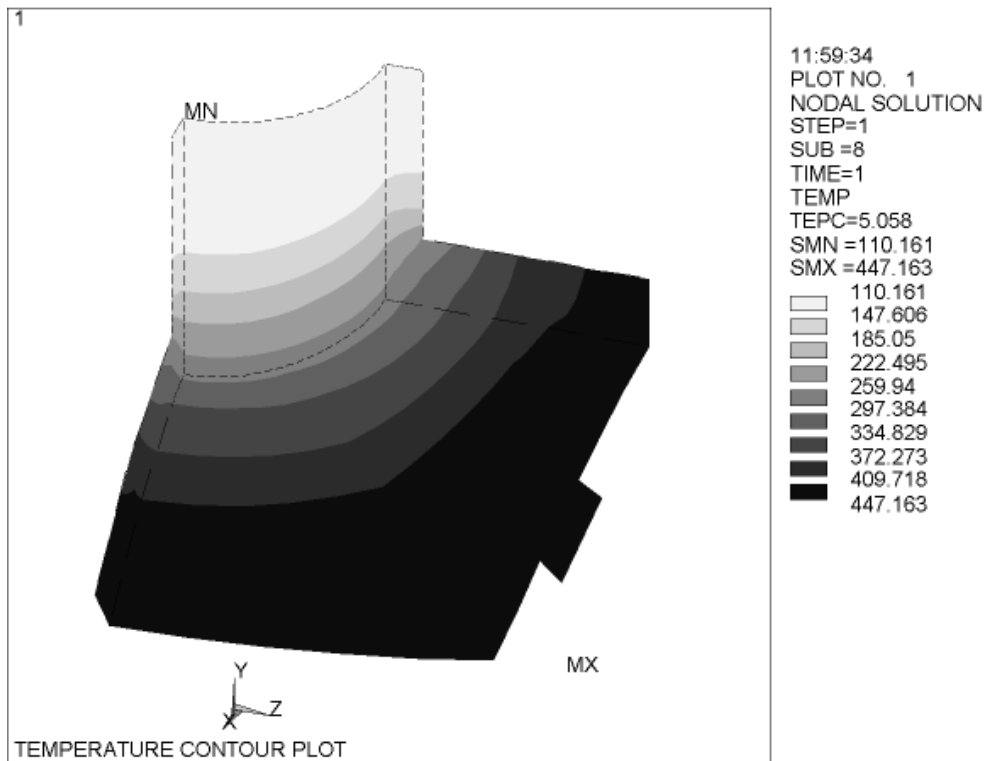
GUI: Main Menu > General Postproc > Plot Results > Contour Plot > Element Solu

Main Menu > General Postproc > Plot Results > Contour Plot > Elem Table

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

Figure 2.2: “Contour Results Plot” shows you an example of a contour display:

Figure 2.2 Contour Results Plot



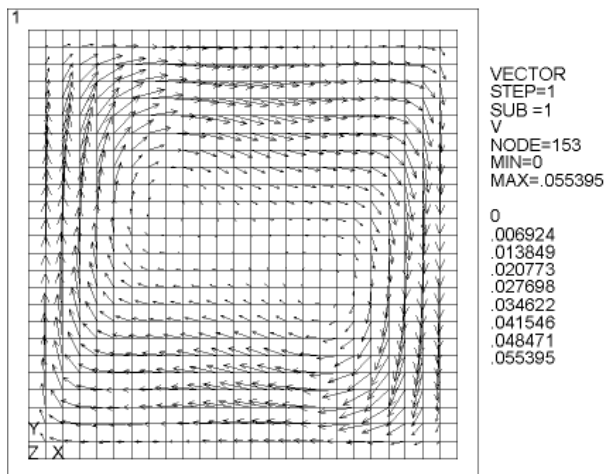
For vector displays:

Command(s): PLVECT

GUI: Main Menu> General Postproc> Plot Results> Vector Plot> Pre-defined or Userdefined

Figure 2.3: “Vector Display” shows you an example of a vector display:

Figure 2.3 Vector Display



For table listings:

Command(s): PRESOL, PRNSOL, PRRSOL

GUI: Main Menu> General Postproc> List Results> Element Solution

Main Menu> General Postproc> List Results> Nodal Solution

Main Menu> General Postproc> List Results> Reaction Solu

When you choose one of the GUI paths or issue one of the commands shown above, the ANSYS program displays the results in a text window (not shown here).

2.7. Example of a Steady-State Thermal Analysis (Command or Batch Method)

This section describes how to do a steady-state thermal stress analysis of a pipe intersection by issuing a sequence of ANSYS commands, either while running ANSYS in batch mode or by issuing the commands manually during an interactive ANSYS session. Section 2.8: Doing a Steady-State Thermal Analysis (GUI Method) explains how to perform the same example analysis by choosing options from the ANSYS menus.

2.7.1. The Example Described

In this example, a cylindrical tank is penetrated radially by a small pipe at a point on its axis remote from the ends of the tank. The inside of the tank is exposed to a fluid of 450°F (232°C). The pipe experiences a steady flow of 100°F (38°C) fluid, and the two flow regimes are isolated from each other by a thin tube. The film coefficient in the tank is a steady 250 Btu/hr-ft²-°F (1420 watts/m²-°K). The film coefficient in the pipe varies with the metal temperature and is given in the material property table below.

The purpose of the example is to determine the temperature distribution at the pipe-tank junction.

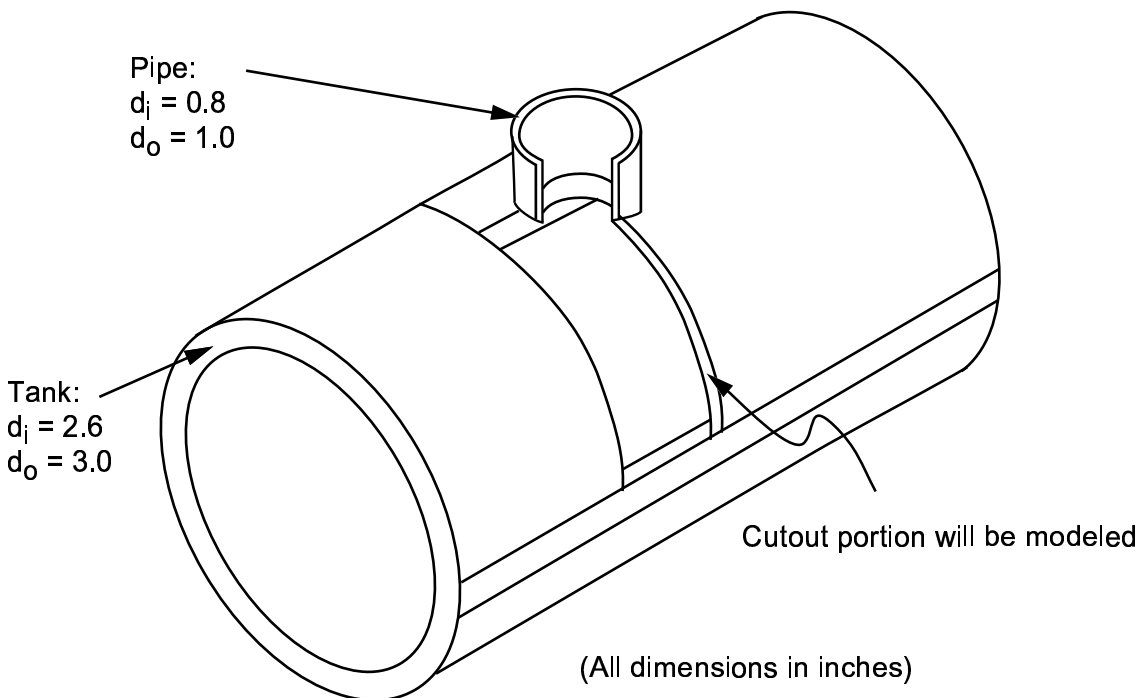
Note — The example analysis presented here is only one of many possible thermal analyses. Not all thermal analyses follow exactly the same steps or perform those steps in the same sequence. The properties of the material or materials being analyzed and the conditions surrounding those materials determine which steps a specific analysis needs to include.

Material properties are as follows:

Table 2.13 Material Properties for the Sample Analysis

| | | | | | | |
|------------------|-------|-------|-------|-------|-------|------------------------------|
| Temperature | 70 | 200 | 300 | 400 | 500 | (°F) |
| Density | 0.285 | 0.285 | 0.285 | 0.285 | 0.285 | (lb/in ³) |
| Conductivity | 8.35 | 8.90 | 9.35 | 9.80 | 10.23 | (Btu/hr-ft-°F) |
| Specific Heat | 0.113 | 0.117 | 0.119 | 0.122 | 0.125 | (Btu/lb-°F) |
| Film Coefficient | 426 | 405 | 352 | 275 | 221 | (Btu/hr-ft ² -°F) |

Figure 2.4 Pipe-Tank Junction Model



2.7.2. The Analysis Approach

The model in this example uses quarter-symmetry to represent the pipe-tank junction. The tank is assumed to be long enough for its remote end to be held at a constant temperature of 450°F. A similar assumption is made at the $Y=0$ plane of the tank.

Building the model involves defining two cylinder primitives and a Boolean overlap operation. A mapped (all-brick) mesh is used. The meshing operation produces warnings for a few distorted elements, but you can ignore the warnings because the cited elements are remote from the region of interest (the junction of the pipe and tank).

Because the analysis uses temperature-dependent material properties, the solution requires multiple substeps (50 in this case). Automatic time stepping also is used. After you solve the model, a temperature contour plot and a vector plot of thermal flux enables you to review the results.

2.7.3. Commands for Building and Solving the Model

The following sequence of commands builds and solves the finite element model. Text preceded by an exclamation mark (!) is comment text.

```

/PREP7
/TITLE,Steady-state thermal analysis of pipe junction
/UNITS,BIN                ! Use British system of units (inches)
/SHOW,                    ! Specify graphics driver for interactive run
!
ET,1,90                   ! Define 20-node, 3-D thermal solid element
MP,DENS,1,.285            ! Density = .285 lbf/in^3
MPTEMP,,70,200,300,400,500 ! Create temperature table
MPDATA,KXX,1,,8.35/12,8.90/12,9.35/12,9.80/12,10.23/12
                          ! Define conductivity values
MPDATA,C,1,,.113,.117,.119,.122,.125
                          ! Define specific heat values
MPDATA,HF,2,,426/144,405/144,352/144,275/144,221/144
                          ! Define film coefficients

! Define parameters for model generation
RI1=1.3                   ! Inside radius of cylindrical tank
RO1=1.5                   ! Outside radius
Z1=2                      ! Length
RI2=.4                    ! Inside radius of pipe
RO2=.5                    ! Outside pipe radius
Z2=2                      ! Pipe length
!
CYLIND,RI1,RO1,,Z1,,90   ! 90 degree cylindrical volume for tank
WPROTA,0,-90             ! Rotate working plane to pipe axis
CYLIND,RI2,RO2,,Z2,-90  ! 90 degree cylindrical volume for pipe
WPSTYL,DEFA             ! Return working plane to default setting
BOPT,NUMB,OFF           ! Turn off Boolean numbering warning
VOVLAP,1,2              ! Overlap the two cylinders
/PNUM,VOLU,1            ! Turn volume numbers on
/VIEW,,-3,-1,1
/TYPE,,4
/TITLE,Volumes used in building pipe/tank junction
VPLLOT
VDELE,3,4,,1           ! Trim off excess volumes

! Meshing
ASEL,,LOC,Z,Z1          ! Select area at remote Z edge of tank
ASEL,A,LOC,Y,0          ! Select area at remote Y edge of tank
CM,AREMOTE,AREA        ! Create area component called AREMOTE
/PNUM,AREA,1
/PNUM,LINE,1
/TITLE,Lines showing the portion being modeled
APLOT
/NOERASE
LPLOT                   ! Overlay line plot on area plot
/ERASE
ACCAT,ALL               ! Concatenate areas and lines
                          ! at remote tank edges

LCCAT,12,7
LCCAT,10,5
LESIZE,20,,.4          ! 4 divisions through pipe thickness
LESIZE,40,,.6          ! 6 divisions along pipe length
LESIZE,6,,.4           ! 4 divisions through tank thickness
ALLSEL                  ! Restore full set of entities
ESIZE,.4                ! Set default element size
MSHAPE,0,3D            ! Choose mapped brick mesh
MSHKEY,1
SAVE                    ! Save database before meshing
VMESH,ALL               ! Generate nodes and elements within volumes
/PNUM,DEFA
/TITLE,Elements in portion being modeled
EPLOT
FINISH
!
/COM, *** Obtain solution ***

```

```

!
/SOLU
ANTYPE,STATIC           ! Steady-state analysis type
NROPT,AUTO              ! Program-chosen Newton-Raphson option
TUNIF,450               ! Uniform starting temperature at all nodes
CSYS,1
NSEL,S,LOC,X,RI1       ! Nodes on inner tank surface
SF,ALL,CONV,250/144,450 ! Convection load at all nodes
CMSEL,,AREMOTE         ! Select AREMOTE component
NSLA,,1                ! Nodes belonging to AREMOTE
D,ALL,TEMP,450         ! Temperature constraints at those nodes
WPROTA,0,-90           ! Rotate working plane to pipe axis
CSWPLA,11,1           ! Define local cylindrical c.s at working plane
NSEL,S,LOC,X,RI2       ! Nodes on inner pipe surface
SF,ALL,CONV,-2,100     ! Temperature-dep. convection load at those nodes
ALLSEL
/PBC,TEMP,,1          ! Temperature b.c. symbols on
/PSF,CONV,,2          ! Convection symbols on
/TITLE,Boundary conditions
NPLOT
WPSTYL,DEFA
CSYS,0
AUTOTS,ON              ! Automatic time stepping
NSUBST,50              ! Number of substeps
KBC,0                  ! Ramped loading (default)
OUTPR,NSOL,LAST       ! Optional command for solution printout
SOLVE
FINISH
!
/COM, *** Review results ***
!
/POST1
/EDGE,,1               ! Edge display
/PLOPTS,INFO,ON       ! Legend column on
/PLOPTS,LEG1,OFF      ! Legend header off
/WINDOW,1,SQUARE      ! Redefine window size
/TITLE,Temperature contours at pipe/tank junction
PLNSOL,TEMP           ! Plot temperature contours
CSYS,11
NSEL,,LOC,X,RO2       ! Nodes and elements at outer radius of pipe
ESLN
NSLE
/SHOW,,1              ! Vector mode
/TITLE,Thermal flux vectors at pipe/tank junction
PLVECT,TF             ! Plot thermal flux vectors
FINISH
/EXIT,ALL

```

2.8. Doing a Steady-State Thermal Analysis (GUI Method)

This section describes how to use the menus on the ANSYS GUI to perform the same steady-state thermal analysis described in Section 2.7: Example of a Steady-State Thermal Analysis (Command or Batch Method). In this version of the sample analysis, instead of issuing commands, you select options from the GUI menus.

Step 1: Give the Analysis a Title

After you have started the ANSYS program and have entered the GUI, you need to begin the analysis by assigning a title to it. To do so, perform these tasks:

1. Choose **Utility Menu > File > Change Title**. The Change Title dialog box appears.
2. Enter the text **Steady-state thermal analysis of pipe junction**.
3. Click on **OK**.

Step 2: Set Measurement Units

You need to specify units of measurement for the analysis. For this pipe junction example, measurements use the British system of units (based on inches). To specify this, type the command **/UNITS,BIN** in the ANSYS Input window and press ENTER.

Step 3: Define the Element Type

The example analysis uses a thermal solid element. To define it, do the following:

1. Choose **Main Menu > Preprocessor > Element Type > Add/Edit/Delete**. The Element Types dialog box appears.
2. Click on Add. The Library of Element Types dialog box appears.
3. In the list on the left, scroll down and pick (highlight) "Thermal Solid." In the list on the right, pick "Brick20node 90."
4. Click on OK.
5. Click on Close to close the Element Types dialog box.

Step 4: Define Material Properties

To define material properties for the analysis, perform these steps:

1. Choose **Main Menu > Preprocessor > Material Props > Material Models**. The Define Material Model Behavior dialog box appears.
2. In the Material Models Available window, double-click on the following options: Thermal, Density. A dialog box appears.
3. Enter .285 for DENS (Density), and click on OK. Material Model Number 1 appears in the Material Models Defined window on the left.
4. In the Material Models Available window, double-click on the following options: Conductivity, Isotropic. A dialog box appears.
5. Click on the Add Temperature button four times. Four columns are added.
6. In the T1 through T5 fields, enter the following temperature values: 70, 200, 300, 400, and 500. Select the row of temperatures by dragging the cursor across the text fields. Then copy the temperatures by pressing Ctrl-c.
7. In the KXX (Thermal Conductivity) fields, enter the following values, in order, for each of the temperatures, then click on OK. Note that to keep the units consistent, each of the given values of KXX must be divided by 12. You can just input the fractions and have ANSYS perform the calculations.

```
8.35/12
8.90/12
9.35/12
9.80/12
10.23/12
```

8. In the Material Models Available window, double-click on Specific Heat. A dialog box appears.
9. Click on the Add Temperature button four times. Four columns are added.
10. With the cursor positioned in the T1 field, paste the five temperatures by pressing Ctrl-v.
11. In the C (Specific Heat) fields, enter the following values, in order, for each of the temperatures, then click on OK.

.113
 .117
 .119
 .122
 .125

12. Choose menu path **Material> New Model**, then enter 2 for the new Material ID. Click on OK. Material Model Number 2 appears in the Material Models Defined window on the left.
13. In the Material Models Available window, double-click on Convection or Film Coef. A dialog box appears.
14. Click on the Add Temperature button four times. Four columns are added.
15. With the cursor positioned in the T1 field, paste the five temperatures by pressing Ctrl-v.
16. In the HF (Film Coefficient) fields, enter the following values, in order, for each of the temperatures. To keep the units consistent, each value of HF must be divided by 144. As in step 7, you can input the data as fractions and let ANSYS perform the calculations.

426/144
 405/144
 352/144
 275/144
 221/144

17. Click on the Graph button to view a graph of Film Coefficients vs. temperature, then click on OK.
18. Choose menu path **Material> Exit** to remove the Define Material Model Behavior dialog box.
19. Click on SAVE_DB on the ANSYS Toolbar.

Step 5: Define Parameters for Modeling

1. Choose **Utility Menu> Parameters> Scalar Parameters**. The Scalar Parameters window appears.
2. In the window's Selection field, enter the values shown below. (Do not enter the text in parentheses.) Press ENTER after typing in each value. If you make a mistake, simply retype the line containing the error.

RI1=1.3 (Inside radius of the cylindrical tank)
 RO1=1.5 (Outside radius of the tank)
 Z1=2 (Length of the tank)
 RI2=.4 (Inside radius of the pipe)
 RO2=.5 (Outside radius of the pipe)
 Z2=2 (Length of the pipe)

3. Click on Close to close the window.

Step 6: Create the Tank and Pipe Geometry

1. Choose **Main Menu> Preprocessor> Modeling> Create> Volumes> Cylinder> By Dimensions**. The Create Cylinder by Dimensions dialog box appears.
2. Set the "Outer radius" field to RO1, the "Optional inner radius" field to RI1, the "Z coordinates" fields to 0 and Z1 respectively, and the "Ending angle" field to 90.
3. Click on OK.
4. Choose **Utility Menu> WorkPlane> Offset WP by Increments**. The Offset WP dialog box appears.
5. Set the "XY, YZ, ZX Angles" field to 0,-90.
6. Click on OK.
7. Choose **Main Menu> Preprocessor> Modeling> Create> Volumes> Cylinder> By Dimensions**. The Create Cylinder by Dimensions dialog box appears.

8. Set the "Outer radius" field to RO2, the "Optional inner radius" field to RI2, the "Z coordinates" fields to 0 and Z2 respectively. Set the "Starting angle" field to -90 and the "Ending Angle" to 0.
9. Click on OK.
10. Choose **Utility Menu > WorkPlane > Align WP with > Global Cartesian**.

Step 7: Overlap the Cylinders

1. Choose **Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Volumes**. The Overlap Volumes picking menu appears.
2. Click on Pick All.

Step 8: Review the Resulting Model

Before you continue with the analysis, quickly review your model. To do so, follow these steps:

1. Choose **Utility Menu > PlotCtrls > Numbering**. The Plot Numbering Controls dialog box appears.
2. Click the Volume numbers radio button to On, then click on OK.
3. Choose **Utility Menu > PlotCtrls > View Settings > Viewing Direction**. A dialog box appears.
4. Set the "Coords of view point" fields to (-3,-1,1), then click on OK.
5. Review the resulting model.
6. Click on SAVE_DB on the ANSYS Toolbar.

Step 9: Trim Off Excess Volumes

In this step, delete the overlapping edges of the tank and the lower portion of the pipe.

1. Choose **Main Menu > Preprocessor > Modeling > Delete > Volume and Below**. The Delete Volume and Below picking menu appears.
2. In the picking menu, type 3,4 and press the ENTER key. Then click on OK in the Delete Volume and Below picking menu.

Step 10: Create Component AREMOTE

In this step, you select the areas at the remote Y and Z edges of the tank and save them as a component called AREMOTE. To do so, perform these tasks:

1. Choose **Utility Menu > Select > Entities**. The Select Entities dialog box appears.
2. In the top drop down menu, select Areas. In the second drop down menu, select By Location. Click on the Z Coordinates radio button.
3. Set the "Min,Max" field to Z1.
4. Click on Apply.
5. Click on the Y Coordinates and Also Sele radio buttons.
6. Set the "Min,Max" field to 0.
7. Click on OK.
8. Choose **Utility Menu > Select > Comp/Assembly > Create Component**. The Create Component dialog box appears.

9. Set the "Component name" field to AREMOTE. In the "Component is made of" menu, select Areas.
10. Click on OK.

Step 11: Overlay Lines on Top of Areas

Do the following:

1. Choose **Utility Menu> PlotCtrls> Numbering**. The Plot Numbering Controls dialog box appears.
2. Click the Area and Line number radio boxes to On and click on OK.
3. Choose **Utility Menu> Plot> Areas**.
4. Choose **Utility Menu> PlotCtrls> Erase Options**.
5. Set "Erase between Plots" radio button to Off.
6. Choose **Utility Menu> Plot> Lines**.
7. Choose **Utility Menu> PlotCtrls> Erase Options**.
8. Set "Erase between Plots" radio button to On.

Step 12: Concatenate Areas and Lines

In this step, you concatenate areas and lines at the remote edges of the tank for mapped meshing. To do so, follow these steps:

1. Choose **Main Menu> Preprocessor> Meshing> Mesh> Volumes> Mapped> Concatenate> Areas**. The Concatenate Areas picking menu appears.
2. Click on Pick All.
3. Choose **Main Menu> Preprocessor> Meshing> Mesh> Volumes> Mapped> Concatenate> Lines**. A picking menu appears.
4. Pick (click on) lines 12 and 7 (or enter in the picker).
5. Click on Apply.
6. Pick lines 10 and 5 (or enter in picker).
7. Click on OK.

Step 13: Set Meshing Density Along Lines

1. Choose **Main Menu> Preprocessor> Meshing> Size Cntrls> ManualSize>Lines> Picked Lines**. The Element Size on Picked Lines picking menu appears.
2. Pick lines 6 and 20 (or enter in the picker) .
3. Click on OK. The Element Sizes on Picked Lines dialog box appears.
4. Set the "No. of element divisions" field to 4.
5. Click on OK.
6. Choose **Main Menu> Preprocessor> Meshing> Size Cntrls> ManualSize> Lines> Picked Lines**. A picking menu appears.
7. Pick line 40 (or enter in the picker).
8. Click on OK. The Element Sizes on Picked Lines dialog box appears.

9. Set the "No. of element divisions" field to 6.
10. Click on OK.

Step 14: Mesh the Model

In this sequence of steps, you set the global element size, set mapped meshing, then mesh the volumes.

1. Choose **Utility Menu> Select> Everything**.
2. Choose **Main Menu> Preprocessor> Meshing> Size Cntrls> ManualSize> Global> Size**. The Global Element Sizes dialog box appears.
3. Set the "Element edge length" field to 0.4 and click on OK.
4. Choose **Main Menu> Preprocessor> Meshing> Mesher Opts**. The Mesher Options dialog box appears.
5. Set the Mesher Type radio button to Mapped and click on OK. The Set Element Shape dialog box appears.
6. In the 2-D shape key drop down menu, select Quad and click on OK.
7. Click on the SAVE_DB button on the Toolbar.
8. Choose **Main Menu> Preprocessor> Meshing> Mesh> Volumes> Mapped> 4 to 6 sided**. The Mesh Volumes picking menu appears. Click on Pick All. In the Graphics window, ANSYS builds the meshed model. If a shape testing warning message appears, review it and click Close.

Step 15: Turn Off Numbering and Display Elements

1. Choose **Utility Menu> PlotCtrls> Numbering**. The Plot Numbering Controls dialog box appears.
2. Set the Line, Area, and Volume numbering radio buttons to Off.
3. Click on OK.

Step 16: Define the Solution Type and Options

In this step, you tell ANSYS that you want a steady-state solution that uses a program-chosen Newton-Raphson option.

1. Choose **Main Menu> Solution> Analysis Type> New Analysis**. The New Analysis dialog box appears.
2. Click on OK to choose the default analysis type (Steady-state).
3. Choose **Main Menu> Solution> Analysis Type> Analysis Options**. The Static or Steady-State dialog box appears.
4. Click on OK to accept the default ("Program-chosen") for "Newton-Raphson option."

Step 17: Set Uniform Starting Temperature

In a thermal analysis, set a starting temperature.

1. Choose **Main Menu> Solution> Define Loads> Apply> Thermal> Temperature> Uniform Temp**. A dialog box appears.
2. Enter 450 for "Uniform temperature." Click on OK.

Step 18: Apply Convection Loads

This step applies convection loads to the nodes on the inner surface of the tank.

1. Choose **Utility Menu> WorkPlane> Change Active CS to> Global Cylindrical**.
2. Choose **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
3. Select Nodes and By Location, and click on the X Coordinates and From Full radio buttons.
4. Set the "Min,Max" field to R11 and click on OK.
5. Choose **Main Menu> Solution> Define Loads> Apply> Thermal> Convection> On Nodes**. The Apply CONV on Nodes picking menu appears.
6. Click on Pick All. The Apply CONV on Nodes dialog box appears.
7. Set the "Film coefficient" field to 250/144.
8. Set the "Bulk temperature" field to 450.
9. Click on OK.

Step 19: Apply Temperature Constraints to AREMOTE Component

1. Choose **Utility Menu> Select> Comp/Assembly> Select Comp/Assembly**. A dialog box appears.
2. Click on OK to select component AREMOTE.
3. Choose **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
4. Select Nodes and Attached To, and click on the Areas,All radio button. Click on OK.
5. Choose **Main Menu> Solution> Define Loads> Apply> Thermal> Temperature> On Nodes**. The Apply TEMP on Nodes picking menu appears.
6. Click on Pick All. A dialog box appears.
7. Set the "Load TEMP value" field to 450.
8. Click on OK.
9. Click on SAVE_DB on the ANSYS Toolbar.

Step 20: Apply Temperature-Dependent Convection

In this step, apply a temperature-dependent convection load on the inner surface of the pipe.

1. Choose **Utility Menu> WorkPlane> Offset WP by Increments**. A dialog box appears.
2. Set the "XY,YZ,ZX Angles" field to 0,-90, then click on OK.
3. Choose **Utility Menu> WorkPlane> Local Coordinate Systems> Create Local CS> At WP Origin**. The Create Local CS at WP Origin dialog box appears.
4. On the "Type of coordinate system" menu, select "Cylindrical 1" and click on OK.
5. Choose **Utility Menu> Select> Entities**. The Select Entities dialog box appears.
6. Select Nodes, and By Location, and click on the X Coordinates radio button.
7. Set the "Min,Max" field to R12.
8. Click on OK.
9. Choose **Main Menu> Solution> Define Loads> Apply> Thermal> Convection> On Nodes**. The Apply CONV on Nodes picking menu appears.
10. Click on Pick All. A dialog box appears.
11. Set the "Film coefficient" field to -2.

12. Set the "Bulk temperature" field to 100.
13. Click on OK.
14. Choose **Utility Menu> Select> Everything**.
15. Choose **Utility Menu> PlotCtrls> Symbols**. The Symbols dialog box appears.
16. On the "Show pres and convect as" menu, select Arrows, then click on OK.
17. Choose **Utility Menu> Plot> Nodes**. The display in the Graphics Window changes to show you a plot of nodes.

Step 21: Reset the Working Plane and Coordinates

1. To reset the working plane and default Cartesian coordinate system, choose **Utility Menu> WorkPlane> Change Active CS to> Global Cartesian**.
2. Choose **Utility Menu> WorkPlane> Align WP With> Global Cartesian**.

Step 22: Set Load Step Options

For this example analysis, you need to specify 50 substeps with automatic time stepping.

1. Choose **Main Menu> Solution> Load Step Options> Time/Frequenc> Time and Substps**. The Time and Substep Options dialog box appears.
2. Set the "Number of substeps" field to 50.
3. Set "Automatic time stepping" radio button to On.
4. Click on OK.

Step 23: Solve the Model

1. Choose **Main Menu> Solution> Solve> Current LS**. The ANSYS program displays a summary of the solution options in a /STAT command window.
2. Review the summary.
3. Choose Close to close the /STAT command window.
4. Click on OK in the Solve Current Load Step dialog box.
5. Click Yes in the Verify message window.
6. The solution runs. When the Solution is done! window appears, click on Close.

Step 24: Review the Nodal Temperature Results

1. Choose **Utility Menu> PlotCtrls> Style> Edge Options**. The Edge Options dialog box appears.
2. Set the "Element outlines" field to "Edge only" for contour plots and click on OK.
3. Choose **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**. The Contour Nodal Solution Data dialog box appears.
4. For "Item to be contoured," pick "DOF solution" from the list on the left, then pick "Temperature TEMP" from the list on the right.
5. Click on OK. The Graphics window displays a contour plot of the temperature results.

Step 25: Plot Thermal Flux Vectors

In this step, you plot the thermal flux vectors at the intersection of the pipe and tank.

1. Choose **Utility Menu > WorkPlane > Change Active CS to > Specified Coord Sys**. A dialog box appears.
2. Set the "Coordinate system number" field to 11.
3. Click on OK.
4. Choose **Utility Menu > Select > Entities**. The Select Entities dialog box appears.
5. Select Nodes and By Location, and click the X Coordinates radio button.
6. Set the "Min,Max" field to RO2.
7. Click on Apply.
8. Select Elements and Attached To, and click the Nodes radio button.
9. Click on Apply.
10. Select Nodes and Attached To, then click on OK.
11. Choose **Main Menu > General Postproc > Plot Results > Vector Plot > Predefined**. A dialog box appears.
12. For "Vector item to be plotted," choose "Flux & gradient" from the list on the left and choose "Thermal flux TF" from the list on the right.
13. Click on OK. The Graphics Window displays a plot of thermal flux vectors.

Step 26: Exit from ANSYS

To leave the ANSYS program, click on the QUIT button in the Toolbar. Choose an exit option and click on OK.

2.9. Doing a Thermal Analysis Using Tabular Boundary Conditions

This section describes how to perform a simple thermal analysis, using a 1-D table to apply loads. This problem is shown twice, once done via commands, and then done interactively using the GUI.

2.9.1. Running the Sample Problem via Commands

Text preceded by an exclamation mark (!) is comment text.

```

/batch,list
/show
/title, Demonstration of position-varying film coefficient using Tabular BC's.
/com
/com * -----
/com * Table Support of boundary conditions
/com *
/com * Boundary Condition Type   Primary Variables   Independent Parameters
/com * -----
/com * Convection:Film Coefficient   X                   -
/com *
/com * Problem description
/com *
/com * A static Heat Transfer problem. A 2 x 1 rectangular plate is
/com * subjected to temperature constraint at one of its end, while the
/com * remaining perimeter of the plate is subjected to a convection boundary
/com * condition. The film coefficient is a function of X-position and is described
/com * by a parametric table 'cnvtab'.
/com **
*dim,cnvtab,table,5,,,x ! table definition.
cnvtab(1,0) = 0.0,0.50,1.0,1.50,2.0      ! Variable name, Var1 = 'X'
cnvtab(1,1) = 20.0,30.0,50.0,80.0,120.0

```

```

/prep7
esize,0.5
et,1,55
rect,0,2,0,1
amesh,1
MP,KXX,,1.0
MP,DENS,,10.0
MP,C,,100.0
lsel,s,loc,x,0
dl,all,,temp,100
alls
lsel,u,loc,x,0
nsl1,s,1
sf,all,conv,%cnvtab%,20
alls
/psf,conv,hcoef,2           ! show convection bc.
/pnum,tabn,on             ! show table names
nplot
fini
/solu
anty,static
kbc,1
nsubst,1
time,60
tunif,50
outres,all,all
solve
finish
/post1
set,last
sflist,all                 ! Numerical values of convection bc's
/pnum,tabn,off           ! turn off table name
/psf,conv,hcoef,2       ! show convection bc.
/pnum,sval,1            ! show numerical values of table bc's
eplot! convection at t=60 sec.
plns,temp
fini

```

2.9.2. Running the Sample Problem Interactively

The same problem is shown here using interactive menu selections on the GUI.

Step 1: Define a 1-D table

1. Choose **Utility Menu > Parameters > Array Parameters > Define/Edit**. The Array Parameters dialog box appears. Click Add...
2. The Add New Array Parameter dialog box appears. Type cnvtab in the "Parameter name" field.
3. Select "Table" for Parameter type.
4. Enter 5,1,1 as I,J,K values
5. Enter X as row variable.
6. Click OK.
7. In the Array Parameters dialog box, make sure cnvtab is highlighted and click Edit. The Table Array:CNVTAB=f(X) table editor dialog box appears. (See Section 3.10.3: TABLE Type Array Parameters in the *ANSYS APDL Programmer's Guide* for details about table arrays.)
8. Two columns appear in the table editor dialog box. The first column is column 0; the second column is column 1. Column 0 contains six boxes. Do not do anything in the first (top) box. In the five other boxes, type 0.0, 0.5, 1.0, 1.5, and 2.0. These are row index values.
9. Column 1 also contains six boxes. You do not have to enter anything in the blue (top) box, because this table is one-dimensional. In the other five boxes, type 20, 30, 50, 80, and 120.

10. Choose **File> Apply/Quit**.
11. Close the Array Parameters dialog box.

Step 2: Define your element type and material properties

1. Choose **Main Menu> Preprocessor> Element Type> Add/Edit/Delete**. The Element Types dialog box appears. Click Add.
2. The Library of Element Types dialog box appears. Select Thermal Solid from the list on the left, and select Quad 4node 55 from the list on the right.
3. Click OK.
4. Close the Element Types dialog box.
5. Choose **Main Menu> Preprocessor> Material Props> Material Models**. The Define Material Model Behavior dialog box appears.
6. In the Material Models Available window, double-click on the following options: Thermal, Density. A dialog box appears.
7. Enter 10.0 for DENS (density). Click on OK. Material Model Number 1 appears in the Material Models Defined window on the left.
8. In the Material Models Available window, double-click on the following options: Conductivity, Isotropic. A dialog box appears.
9. Enter 1.0 for KXX (Thermal conductivity). Click on OK.
10. In the Material Models Available window, double-click on Specific Heat. A dialog box appears.
11. Enter 100.0 for C (Specific Heat). Click on OK.
12. Choose menu path **Material> Exit** to remove the Define Material Model Behavior dialog box.

Step 3: Build and mesh your model

1. Choose **Main Menu> Preprocessor> Modeling> Create> Areas> Rectangle> By Dimensions**. The Create Rectangle by Dimensions dialog box appears.
2. Enter 0, 2 for X1,X2 coordinates.
3. Enter 0, 1 for Y1, Y2 coordinates.
4. Click OK. A rectangular area appears on the screen.
5. Choose **Main Menu> Preprocessor> Meshing> MeshTool**.
6. Under the Size Controls section of the Mesh Tool, click Globl,Set. The Global Element Sizes dialog box appears.
7. Set the "Element endge length" field to 0.5 and click on OK.
8. In the Mesh area of the Mesh Tool, choose Areas and Map and verify that Quad and 3/4 sided are selected.
9. Click on MESH. The Mesh Areas picking menu appears.
10. Click on Pick All. The mesh appears in the Graphics window.
11. Close the MeshTool dialog box.
12. Click on SAVE_DB on the ANSYS Toolbar.

Step 4: Apply Tabular Boundary Conditions

1. Choose **Utility Menu> Plot> Lines**.
2. Choose **Main Menu> Solution> Define Loads> Apply> Thermal> Temperature> On Lines**. The Apply TEMP on Lines picking menu appears.
3. In the Graphics window, select the vertical line at $x=0$ (on the far left of the model). Click OK.
4. The Apply TEMP on lines dialog box appears.
5. Enter 100 for VALUE. Click OK.
6. Choose **Main Menu> Solution> Define Loads> Apply> Thermal> Convection> On Lines**. The Apply CONV on Lines picking menu appears.
7. In the Graphics window, select all lines except the line at $x = 0$.
8. Click OK. The Apply CONV on lines dialog box appears.
9. In the drop-down selection box for "Apply Film Coef on lines," select "Existing table."
10. Remove any value in the VALI field.
11. Enter 20 in the "VAL2I Bulk temperature" field. Click OK.
12. A second Apply CONV on lines dialog box appears. Verify that the selection box for "Existing table" shows CNVTAB. Click OK. The ANSYS Graphics Window displays arrows on all lines except the line at $x = 0$.
13. Choose **Main Menu> Solution> Define Loads> Apply> Thermal> Temperature> Uniform Temp**. The Uniform Temperature dialog box appears.
14. Enter 50 as the uniform temperature. Click OK.

Step 5: Show the applied loads to verify

1. Choose **Utility Menu> PlotCtrls> Symbols**. The Symbols dialog box appears.
2. Select "Convect FilmCoef" in the "Surface Load Symbols" drop down selection box.
3. Select "Arrows" in the "Show pres and convect as" drop down selection box.
4. Click OK.
5. Choose **Utility Menu> PlotCtrls> Numbering**. The Plot Numbering Controls dialog box appears.
6. Click Table Names on. Click OK. The table name CNVTAB appears on the arrows on the right side of the Graphics window.
7. Click on SAVE_DB on the ANSYS Toolbar.

Step 6: Set Analysis Options and Solve

1. Choose **Main Menu> Solution> Analysis Type> New Analysis**. The New Analysis dialog box appears.
2. Verify that "Steady-State" is selected and click OK.
3. Choose **Main Menu> Solution> Load Step Opts> Time/Frequenc> Time and Substps**. The Time and Substep Options dialog box appears.
4. Enter 60 as "Time at end of load step."
5. Enter 1 as "Number of substeps."
6. Choose Stepped. Click OK.

7. Choose **Main Menu> Solution> Load Step Opts> OutputCtrls> DB/Results File**. The Controls for Database and Results File Writing dialog appears. Verify that the "Item to be controlled" field shows "All items."
8. Select "Every substep" for "File write frequency" field. Click OK.
9. Choose **Main Menu> Solution> Solve> Current LS**. Review the **/STATUS** Command dialog box. If OK, click Close.
10. In the Solve Current Load Step dialog box, click OK to begin the solve. When the solution is done, click Close in the "Solution is done!" information box.

Step 7: Postprocess

1. Choose **Main Menu> General Postproc> Read Results> Last Set**.
2. Choose **Utility Menu> List> Loads> Surface Loads> On All Nodes**. The **SFLIST** Command dialog box appears. Review the results and click Close.
3. Choose **Utility Menu> PlotCtrls> Numbering**. The Plot Numbering Controls dialog box appears.
4. Click Table Names display off.
5. Click Numeric contour values on. Click OK.
6. Choose **Utility Menu> PlotCtrls> Symbols**. The Symbols dialog box appears.
7. In the "Surface Load Symbols" drop down selection box, select "Convect FilmCoef."
8. In the "Show pres and convect as" drop down selection box, select "Arrows." Click OK.
9. Choose **Utility Menu> Plot> Elements**. Observe the numbers over the arrows on the model.
10. Choose **Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu**. The Contour Nodal Solution Data dialog box appears.
11. Verify that DOF Solution is selected in the list on the left, and Temperature is selected in the list on the right. Click OK. Observe the resulting display.

Step 8: Finish

1. You are now finished with this sample problem. Click QUIT in the ANSYS Toolbar. Choose a save option and click OK.

2.10. Where to Find Other Examples of Thermal Analysis

Several ANSYS publications, particularly the *ANSYS Verification Manual* and the *Heat Transfer Training Manual*, describe additional examples of steady-state and other types of thermal analyses.

Attending the Heat Transfer seminar may benefit you if your work includes analyzing the thermal response of structures and components such as internal combustion engines, pressure vessels, heat exchangers and furnaces, etc. For more information about this seminar, contact your local ANSYS Support Distributor or telephone the ANSYS Training Registrar at (724) 514-2882.

The *ANSYS Verification Manual* consists of test cases demonstrating the analysis capabilities of the ANSYS program. While these test cases demonstrate solutions to realistic thermal analysis problems, the *ANSYS Verification Manual* does not present them as step-by-step examples with lengthy data input instructions and printouts. However, you should be able to understand each problem by reviewing the finite element model and input data with accompanying comments.

Here is a list of sample thermal test cases (steady-state, transient, and so on) that the *ANSYS Verification Manual* includes:

- VM3 - Thermal Loaded Support Structure
- VM23 - Thermal-structural Contact of Two Bodies
- VM27 - Thermal Expansion to Close a Gap
- VM32 - Thermal Stresses in a Long Cylinder
- VM58 - Centerline Temperature of a Heat Generating Wire
- VM64 - Thermal Expansion to Close a Gap at a Rigid Surface
- VM92 - Insulated Wall Temperature
- VM93 - Temperature-dependent Conductivity
- VM94 - Heat-generating Plate
- VM95 - Heat Transfer From a Cooling Spine
- VM96 - Temperature Distribution in a Short Solid Cylinder
- VM97 - Temperature Distribution Along a Straight Fin
- VM98 - Temperature Distribution Along a Tapered Fin
- VM99 - Temperature Distribution in a Trapezoidal Fin
- VM100 - Heat Conductivity Across a Chimney Section
- VM101 - Temperature Distribution in a Short Solid Cylinder
- VM102 - Cylinder with Temperature Dependent Conductivity
- VM103 - Thin Plate with a Central Heat Source
- VM104 - Liquid-solid Phase Change
- VM105 - Heat-generation Coil with Temperature Dependent Conductivity
- VM106 - Radiant Energy Emission
- VM107 - Thermocouple Radiation
- VM108 - Temperature Gradient Across a Solid Cylinder
- VM109 - Temperature Response of a Suddenly-cooled Wire
- VM110 - Transient Temperature Distribution in a Slab
- VM111 - Cooling of a Spherical Body
- VM112 - Cooling of a Spherical Body
- VM113 - Transient Temperature Distribution in an Orthotropic Metal Bar
- VM114 - Temperature Response to a Linearly Rising Surface Temperature
- VM115 - Thermal Response of a Heat-generating Slab
- VM116 - Heat-conducting Plate with Sudden Cooling
- VM118 - Centerline Temperature of a Heat Generating Wire
- VM119 - Centerline Temperature of an Electrical Wire
- VM121 - Laminar Flow through a Pipe with Uniform Heat Flux
- VM122 - Pressure Drop in a Turbulent Flowing Fluid
- VM123 - Laminar Flow in a Piping System
- VM124 - Discharge of Water from a Reservoir
- VM125 - Radiation Heat Transfer Between Concentric Cylinders
- VM126 - Heat Transferred to a Flowing Fluid
- VM147 - Gray-body Radiation Within a Frustrum of a Cone
- VM159 - Temperature Controlled Heater
- VM160 - Solid Cylinder with Harmonic Temperature Load
- VM161 - Heat Flow from an Insulated Pipe
- VM162 - Cooling of a Circular Fin of Rectangular Profile
- VM164 - Drying of a Thick Wooden Slab
- VM192 - Cooling of a Billet by Radiation
- VM193 - Adaptive Analysis of 2-D Heat Transfer with Convection

Chapter 3: Transient Thermal Analysis

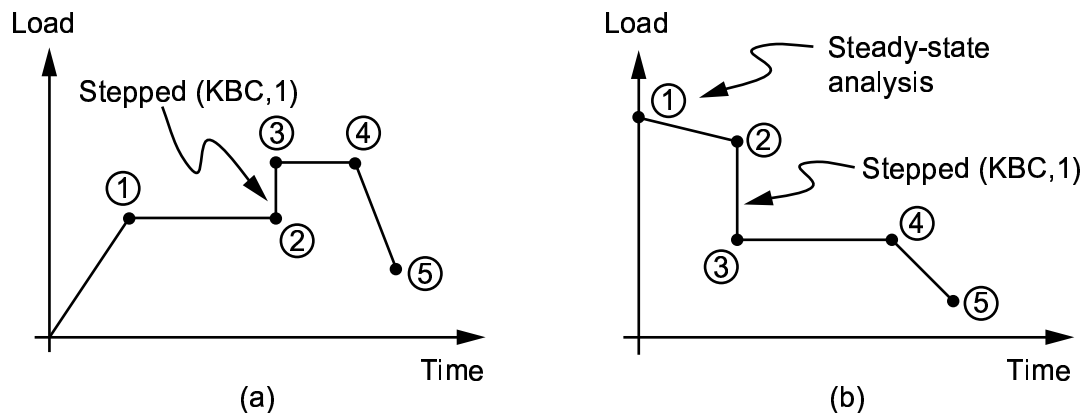
The ANSYS Multiphysics, ANSYS Mechanical, ANSYS Professional, and ANSYS FLOTRAN products support transient thermal analysis. Transient thermal analysis determines temperatures and other thermal quantities that vary over time. Engineers commonly use temperatures that a transient thermal analysis calculates as input to structural analyses for thermal stress evaluations. Many heat transfer applications - heat treatment problems, nozzles, engine blocks, piping systems, pressure vessels, etc. - involve transient thermal analyses.

A transient thermal analysis follows basically the same procedures as a steady-state thermal analysis. The main difference is that most applied loads in a transient analysis are functions of time. To specify time-dependent loads, you can either use the Function Tool to define an equation or function describing the curve and then apply the function as a boundary condition, or you can divide the load-versus-time curve into load steps.

If you use the Function Tool, see Section 2.5.15: Applying Loads Using Function Boundary Conditions in the *ANSYS Basic Analysis Guide* for detailed instructions.

If you use individual load steps, each "corner" on the load-time curve can be one load step, as shown in the following sketches.

Figure 3.1 Examples of Load vs. Time Curves



For each load step, you need to specify both load values and time values, along with other load step options such as stepped or ramped loads automatic time stepping, etc. You then write each load step to a file and solve all load steps together. To get a better understanding of how load and time stepping work, see the example casting analysis scenario in this chapter.

The following transient thermal analysis topics are available:

- 3.1. Elements and Commands Used in Transient Thermal Analysis
- 3.2. Tasks in a Transient Thermal Analysis
- 3.3. Building the Model
- 3.4. Applying Loads and Obtaining a Solution
- 3.5. Saving the Model
- 3.6. Reviewing Analysis Results
- 3.7. Reviewing Results as Graphics or Tables
- 3.8. Phase Change
- 3.9. Example of a Transient Thermal Analysis
- 3.10. Where to Find Other Examples of Transient Thermal Analysis

3.1. Elements and Commands Used in Transient Thermal Analysis

Transient thermal analyses use the same elements as steady-state thermal analyses. See Chapter 2, “Steady-State Thermal Analysis”, for brief descriptions of these elements.

For detailed, alphabetized descriptions of ANSYS commands, see the *ANSYS Commands Reference*.

3.2. Tasks in a Transient Thermal Analysis

The procedure for doing a transient thermal analysis has three main tasks:

- Build the model.
- Apply loads and obtain the solution.
- Review the results.

The rest of this chapter explains each task in the transient thermal analysis process. Because not every transient analysis includes exactly the same tasks, the text both provides general descriptions of the tasks and relates them to example analyses. The examples walk you through doing an analysis via ANSYS commands, then show you how to do the same analysis by choosing items from the ANSYS GUI menus.

3.3. Building the Model

To build the model, you start by specifying the jobname and a title for your analysis. If you are running ANSYS interactively and using its GUI, you also set preferences for the options you want to display. Then, you use the ANSYS preprocessor (PREP7) to do these tasks:

1. Define the element types.
2. If necessary, define element real constants.
3. Define material properties.
4. Define the model geometry.
5. Mesh the model.

These tasks are common to all analyses. The *ANSYS Modeling and Meshing Guide* explains them in detail.

3.4. Applying Loads and Obtaining a Solution

In a transient analysis, the first steps in applying transient loads are to define the analysis type and then establish initial conditions for your analysis.

3.4.1. Defining the Analysis Type

To specify the analysis type, do either of the following:

- In the ANSYS GUI, choose menu path **Main Menu> Solution> Analysis Type> New Analysis> Transient**.
- If this is a new analysis, issue the command **ANTYPE,TRANSIENT,NEW**.

If you want to restart a previous analysis (for example, to specify additional loads), issue the command **ANTYPE,TRANSIENT,REST**. You can restart an analysis only if the files **Jobname.ESAV** and **Jobname.DB** from the previous run are available.

3.4.2. Establishing Initial Conditions for Your Analysis

To establish the initial conditions, you may need to obtain a steady-state solution, or you simply may need to specify a uniform starting temperature at all nodes.

3.4.2.1. Specifying a Uniform Temperature

If you know that the model starts at ambient temperature, specify that temperature at all nodes. To do so, use either of the following:

Command(s): TUNIF

GUI: Main Menu > Preprocessor > Loads > Define Loads > Settings > Uniform Temp

The value you specify via the Uniform Temp dialog box or the **TUNIF** command defaults to the reference temperature, which in turn defaults to zero. (You specify the reference temperature using either item below:

Command(s): TREF

GUI: Main Menu > Preprocessor > Loads > Define Loads > Settings > Reference Temp

Note — Specifying a uniform starting temperature is not the same as applying a temperature DOF constraint (which you do using either item below):

Command(s): D

GUI: Main Menu > Preprocessor > Loads > Define Loads > Apply > Thermal > Temperature > On Nodes

The uniform starting temperature is the temperature in effect at the beginning of an analysis, while a temperature DOF constraint forces a node to have the specified temperature *until it is deleted*. (To delete the temperature, you would choose one of the following:

Command(s): DDELETE

GUI: Main Menu > Preprocessor > Loads > Define Loads > Delete > Thermal > Temperature > On Nodes

3.4.2.2. Specifying a Non-Uniform Starting Temperature

In a transient thermal analysis (but *not* in a steady-state thermal analysis), you can specify one or more non-uniform starting temperatures at a node or a group of nodes. To do so, use either of the following:

Command(s): IC

GUI: Main Menu > Preprocessor > Loads > Define Loads > Apply > Initial Condit'n > Define

You can also apply a non-uniform starting temperature to one or more nodes and at the same time have all other nodes use a uniform starting temperature. You simply specify the uniform temperature before applying the non-uniform temperature to selected nodes.

To display a list of the nodes using a non-uniform starting temperature, choose either of the following:

Command(s): ICLIST

GUI: Main Menu > Preprocessor > Loads > Define Loads > Apply > Initial Condit'n > List Picked

If the initial temperature distribution is not uniform and is not known, you will need to do a steady-state thermal analysis to establish the initial conditions. To do so, perform these steps:

- Specify the appropriate steady-state loads (such as imposed temperatures, convection surfaces, etc.).
- Specify **TIMINT,OFF,THERM** (**Main Menu > Preprocessor > Loads > Load Step Opts > Time/ Frequenc > Time-Time Integration**) to turn off transient effects.
- Use the **TIME** command (**Main Menu > Preprocessor > Loads > Load Step Opts > Time/ Frequenc > Time-Time Step**) to define a value of time. Typically, the time value is extremely small (e.g. 1E-6 seconds).

- Specify ramped or stepped loading using the **KBC** command (**Main Menu**> **Preprocessor**> **Loads**> **Load Step Opts**> **Time/ Frequenc**> **Time-Time Step**). If ramped loading is defined, the effect of the resulting temperature gradients with respect to time should be considered.
- Write the load data to a load step file using the **LSWRITE** command (**Main Menu**> **Preprocessor**> **Loads**> **Load Step Opts**> **Write LS File**).

For the second load step, remember to delete any imposed temperatures unless you know that those nodes will maintain the same temperatures throughout the transient analysis. Also, remember to issue **TIMINT,ON,THERM** in the second load step to turn on transient effects. For more information, see the descriptions of the **D**, **DDELE**, **LSWRITE**, **SF**, **TIME**, and **TIMINT** commands in the *ANSYS Commands Reference*.

3.4.3. Specifying Load Step Options

For a thermal analysis, you can specify general options, nonlinear options, and output controls.

3.4.3.1. Defining Time-stepping Strategy

You can manage your transient problem either by defining multiple load steps (for stepped or ramped boundary conditions) or by using a single load step and tabular boundary conditions (for arbitrary time-varying conditions) with an array parameter to define your time points. However, you can only apply the table method to heat transfer (only) elements, thermal electric elements, thermal surface effect elements, fluid elements, or some combination of these types.

To use the load step method, follow this procedure:

1. Specify the time at the end of the load step using one of these methods:
Command(s): TIME
GUI: Main Menu> **Preprocessor**> **Loads**> **Load Step Opts**> **Time/ Frequenc**> **Time-Time Step**
2. Specify whether your loads are stepped or ramped. Use either of the following:
Command(s): KBC
GUI: Main Menu> **Preprocessor**> **Loads**> **Load Step Opts**> **Time/ Frequenc**> **Time-Time Step**
3. Specify the load values at the end of the load step. (This requires various commands or menu paths, as described in Table 2.9: “Thermal Analysis Load Types” and Table 2.10: “Load Commands for a Thermal Analysis” in this document.)
4. Write information to a load step file using one of these methods:
Command(s): LSWRITE
GUI: Main Menu> **Preprocessor**> **Loads**> **Load Step Opts**> **Write LS File**
5. Repeat steps 1 through 4 for the next load step, then the next, and so on until you have finished writing all load step data to the file.

If you will delete any loads (except temperature constraints), set them to zero over a small time interval instead of deleting them.

To use table parameters, follow this procedure:

1. Define your loading profile (i.e., load vs. time) using **TABLE** type array parameters as described in Section 2.5.14: Applying Loads Using **TABLE** Type Array Parameters in the *ANSYS Basic Analysis Guide*.
2. Specify automatic time stepping on (**AUTOTS,ON**). Specify either time step size (**DELTIM**) or number of substeps (**NSUBST**).

- Specify the time step reset option. You can choose to not reset the time stepping during the solution, to reset the time based on an already-defined array of time values (keytimes), or to reset the time based on a new array of keytimes.

Command(s): TSRES

GUI: Main Menu> Solution> Load Step Opts> Time/Frequenc> Time-Time Step

Main Menu> Solution> Load Step Opts> Time/Frequenc> Time and Sub Steps

If you select new while working interactively, you will be asked to fill in the $n \times 1$ array of keytimes at this time. If you are working in batch mode, you must define the array before issuing **TSRES**, which resets the time step to the initial value as specified on **DELTIM** or **NSUBST**.

If you use an array of time values ($FREQ = \%array\%$ on the **OUTRES** command) in conjunction with a time step reset array (**TSRES** command), you need to ensure that any $FREQ$ array time values exceed the nearest **TSRES** array value by the initial time step increment specified with **DELTIM**, $DTIME$ or **NSUBST**, $NSBSTP$. For example, if you have a $FREQ$ array with the values 1.5, 2, 10, 14.1, and 15, and a **TSRES** array with the values 1, 2, 10, 14, and 16 (where the time stepping would restart at those values), and you specify an initial time step increment of $DTIME = .2$, the program will stop. In this example, the requested $FREQ$ array value of 14.1 does not exist, because the **TSRES** value specified that the time step be reset at 14 and increment at an interval of .2; therefore, the first available time for the $FREQ$ array would be 14.2.

Note — **TSRES** is used only with **AUTOTS,ON**. If constant time stepping is used (**AUTOTS,OFF**), **TSRES** is ignored.

Command(s): *DIM

GUI: Utility Menu> Parameters> Array Parameters> Define/Edit

When you create a keytime array, the time values in the array must be in ascending order and must not exceed the time at the end of the load step as defined on the **TIME** command.

During solution, the time step size will be reset at the keytimes identified in the array. Time step sizes are reset based on initial time step size [**DELTIM**, $DTIME$] or number of substep [**NSUBST**, $NSBSTP$] settings.

- Specify when the information is to be written to the results file using an $n \times 1$ array type parameter, just as you did with the keytime array. You can use the same keytime array that you used to reset time stepping, or you can use a different array. If working interactively, you can create the array at this time or use an existing array. If working in batch mode, you must define the array before issuing **OUTRES**.

Command(s): OUTRES

GUI: Main Menu> Solution> Load Step Opts> Ouput Ctrl's> DB/Results File

Note — You can use the **TSRES** command and time stepping strategy *only* if using the following heat transfer (only) elements, thermal electric elements, thermal surface effect elements, fluid element FLUID116, or some combination of these types:

| | | |
|----------------------|----------|-----------------------|
| LINK31 | SOLID70 | SURF152 |
| LINK32 | MASS71 | SHELL157 ¹ |
| LINK33 | PLANE75 | TARGE169 |
| PLANE35 | PLANE77 | TARGE170 |
| MATRIX50 | SOLID87 | CONTA171 |
| PLANE55 | SOLID90 | CONTA172 |
| SHELL57 | FLUID116 | CONTA173 |
| PLANE67 ¹ | SHELL131 | CONTA174 |
| LINK68 ¹ | SHELL132 | |
| SOLID69 ¹ | SURF151 | |

3.4.3.2. General Options

General options include the following:

- *Solution control option*

This option turns solution control heuristic ON/OFF for thermal analysis. With this option turned ON, you normally specify the number of substeps (**NSUBST**) or the time step size (**DELTIM**), and the time at the end of the load step (**TIME**). The remainder of the solution control commands then default to their optimal values for the particular thermal problem. See the **SOLCONTROL** command in the *ANSYS Commands Reference* for more details.

To turn solution control ON or OFF, use either of the following:

Command(s): SOLCONTROL

GUI: Main Menu > Solution > Analysis Type > Sol'n Controls

- *The time option*

This option specifies time at the end of the load step.

The default time value is 1.0 for the first load step. For subsequent load steps, the default is 1.0 plus the time specified for the previous load step.

To specify time, use either of the following:

Command(s): TIME

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Time/ Frequnc > Time and Sub-stps

Main Menu > Preprocessor > Loads > Load Step Opts > Time/ Frequnc > Time-Time Step

- *Number of substeps per load step, or the time step size*

A nonlinear analysis requires multiple substeps within each load step. By default, the program uses one substep per load step.

In regions of severe thermal gradients during a transient (e.g., surfaces of quenched bodies), there is a relationship between the largest element size in the direction of the heat flow and the smallest time step

¹Thermal DOF only

size that will yield good results. Using more elements for the same time step size will normally give better results, but using more substeps for the same mesh will often give worse results. When using automatic time stepping and elements with midside nodes (quadratic elements), ANSYS recommends that you control the maximum time step size by the description of the loading input and define the minimum time step size (or maximum element size) based on the following relationship:

$$ITS = \Delta^2 / 4 \alpha$$

The Δ value is the conducting length of an element (along the direction of heat flow) in the expected highest temperature gradient. The α value is the thermal diffusivity, given by $k/\rho C$. The k value is the thermal conductivity, ρ is the mass density, and C is the specific heat.

If the above relationship ($ITS = \Delta^2 / 4 \alpha$) is violated when using elements with midside nodes, ANSYS typically computes unwanted oscillations and temperatures outside of the physically possible range. When using elements without midside nodes, the unwanted oscillations are unlikely to occur, and the above recommendation for the minimum time step can be considered somewhat conservative.

Caution: *Avoid using extremely small time steps, especially when establishing initial conditions. Very small numbers can cause calculation errors in ANSYS. For instance, on a problem time scale of unity, time steps smaller than 1E-10 can cause numerical errors.*

To set the number or size of time steps, use either of the following:

Command(s): NSUBST, DELTIM

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Time/ Freq > Freq and Sub-stps or Time and Substps

Main Menu > Preprocessor > Loads > Load Step Opts > Time/ Freq > Time-Time Step

If you apply stepped loads, the load value remains constant for the entire load step. If you ramp loads (the default), the load values increment linearly at each substep (time step) of the load step.

To step or ramp loads, use either of the following:

Command(s): KBC

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Time/ Freq > Time and Sub-stps

Main Menu > Preprocessor > Loads > Load Step Opts > Time/ Freq > Time-Time Step

Main Menu > Preprocessor > Loads > Load Step Opts > Time/ Freq > Freq and Substps

- *Monitor Results in Real Time*

The **NLHIST** command allows you to monitor results of interest in real time during a solution. Before starting the solution, you can request nodal data such as temperatures or heat flows. You can also request element nodal data such as thermal gradients and fluxes at specific elements to be graphed. The result data are written to a file named **Jobname.nlh**. Nodal results and contact results are monitored at every converged substep while element nodal data are written as specified via the **OUTRES** setting. You can also track results during batch runs. To execute, either:

- Access the ANSYS Launcher and select **File Tracking** from the **Tools** menu.

Or...

- Type **nlhist100** in the command line.

Use the supplied file browser to navigate to your **Jobname.nlh** file, and click on it to invoke the tracking utility. You can use this utility to read the file at any time, even after the solution is complete.

To use this option, use either of these methods:

Command(s): NLHIST

GUI: Main Menu> Solution> Results Tracking

3.4.4. Nonlinear Options

For single-field nonlinear thermal analysis, ANSYS allows a choice of three solution options. The *Full* option corresponds to the default full Newton-Raphson algorithm. The *Quasi* option corresponds to only selective reforming of the thermal matrix during solution of the nonlinear thermal problem. The matrix is only reformed if the nonlinear material properties changed by a significant amount (user-controlled). This option performs no equilibrium iterations between time steps. Material properties are evaluated at the temperatures at the beginning of the load step. The *Linear* option forms only one thermal matrix at the first time step of a load step. This option should only be used to obtain a quick approximate solution.

These options in ANSYS can be selected by the **THOPT** command. The *Quasi* and *Linear* solution options perform direct assembly of the thermal matrix and only the ICCG and JCG solvers support solutions under this option. You can choose either of these solvers using the **EQSLV** command.

For the *Quasi* solution option, you have to also specify the material property change tolerance use for matrix reformation. The reform tolerance defaults to .05, corresponding to a 5% change in material properties. The *Quasi* option sets up a single fast material table, with equal temperature points between a maximum and a minimum temperature for evaluation of temperature-dependent material properties. Using this option you have to also specify the number of points (defaults to 64) and the minimum and maximum temperature (defaults to the minimum and maximum temperature defined by the **MPTEMP** command) for the fast material table. All other nonlinear load options are valid with the **THOPT** command.

Command(s): THOPT

GUI: Main Menu> Preprocessor> Loads> Analysis Type> Analysis Options

Specify nonlinear load step options only if nonlinearities are present. Nonlinear options include the following:

- *Number of equilibrium iterations*

This option specifies the maximum allowable number of equilibrium iterations per substep. With **SOLCONTROL,ON**, this command defaults to between 15 and 26 iterations, depending upon the physics of the problem.

To specify the number of equilibrium iterations, use either of the following:

Command(s): NEQIT

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Nonlinear> Equilibrium Iter

- *Automatic Time Stepping*

Also called *time step optimization* in a transient analysis, automatic time stepping allows ANSYS to determine the size of load increments between substeps. It also increases or decreases the time step size during solution, depending on how the model responds. In a transient thermal analysis, the response checked is the thermal eigenvalue. For the **THOPT,Quasi** option, the time step size is also adjusted based on property change during solution. If the eigenvalue is small, a larger time step is used and vice versa. Other things considered in determining the next time step are the number of equilibrium iterations used for the previous time step, and changes in the status of nonlinear elements.

For most problems, you should turn on automatic time stepping and set upper and lower limits for the integration time step. The limits, set via the **NSUBST** command or **DELTIM** command, or the menu path shown below, help to control how much the time step varies.

GUI:

Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time-Time Step

To specify automatic time stepping, use either of the following:

Command(s): AUTOTS

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time and Substps

Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time-Time Step

To change the default values used for automatic time stepping, use either of the following:

Command(s): TINTP

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time Integration

- *Time integration effects*

These load step options determine whether the analysis includes transient effects such as structural inertia and thermal capacitance.

Note — The ANSYS program assumes time integration effects to be on in a transient analysis (unless they were turned off to establish initial conditions). If time integration effects are turned off, ANSYS calculates a steady-state solution.

To specify time integration effects, use either of the following:

Command(s): TIMINT

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time Integration

- *Transient integration parameters*

These parameters control the nature of your time integration scheme and specify the criteria for automatic time stepping. Consult the *ANSYS, Inc. Theory Reference* for details.

To minimize inaccuracies in a solution, you can set the transient integration parameter (the *THETA* value) to 1.0.

To specify transient integration parameters, use either of the following:

Command(s): TINTP

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time Integration

- *Convergence tolerances*

The ANSYS program considers a nonlinear solution to be converged whenever specified convergence criteria are met. Convergence checking may be based on temperatures, heat flow rates, or both. You specify a typical value for the desired item (*VALUE* field on the **CNVTOL** command) and a tolerance about the typical value (*TOLER* field). The convergence criterion is then given by *VALUE* × *TOLER*. For instance, if you specify 500 as the typical value of temperature and 0.001 as the tolerance, the convergence criterion for temperature is 0.5 degrees.

For temperatures, ANSYS compares the change in nodal temperatures between successive equilibrium iterations ($\Delta T = T_i - T_{i-1}$) to the convergence criterion. Using the above example, the solution is converged when the temperature difference at every node from one iteration to the next is less than 0.5 degrees.

For heat flow rates, ANSYS compares the out-of-balance load vector to the convergence criterion. The out-of-balance load vector represents the difference between the applied heat flows and the internal (calculated) heat flows.

To specify convergence tolerances, use either of the following:

Command(s): CNVTOL

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Nonlinear > Convergence Crit

As nonlinear thermal analysis proceeds, ANSYS computes convergence norms with corresponding convergence criteria each equilibrium iteration. Available in both batch and interactive sessions, the Graphical Solution Tracking (GST) feature displays the computed convergence norms and criteria while the solution is in process. By default, GST is ON for interactive sessions and OFF for batch runs. To turn GST on or off, use either of the following:

Command(s): /GST

GUI: Main Menu > Solution > Load Step Opts > OutputCtrls > Grph Solu Track

- *Termination settings for unconverged solutions*

If the ANSYS program cannot converge the solution within the specified number of equilibrium iterations, ANSYS either stops the solution or moves on to the next load step, depending on what you specify as the stopping criteria.

To halt an unconverged solution, use either of the following:

Command(s): NCV

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Nonlinear > Criteria to Stop

- *Line search*

The line search option allows ANSYS to perform a line search with the Newton-Raphson method. To use the line search option, use either of the following:

Command(s): LNSRCH

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Nonlinear > Line Search

- *Predictor-corrector option*

This option activates the predictor-corrector option for the degree of freedom solution at the first equilibrium iteration of each substep.

To use the predictor option, use either of the following:

Command(s): PRED

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Nonlinear > Predictor

3.4.5. Output Controls

This class of load step options enables you to control output. Output controls options are as follows:

- *Control printed output*

This option enables you to include any results data in the printed output file (**Jobname.OUT**). To control printed output, use either of the following:

Command(s): OUTPR

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > OutputCtrls > Solu Printout

- *Control database and results file output*

This option controls what data goes to the results file (**Jobname.RTH**). To control database and results file output, use either of the following:

Command(s): OUTRES

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > OutputCtrls > DB/Results File

- *Extrapolate results*

This option allows you to review element integration point results by copying them to the nodes instead of extrapolating them. (Extrapolation is the default.) To extrapolate results, use either of the following:

Command(s): ERESX

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> OutputCtrls> Integration Pt

3.5. Saving the Model

After you have specified the load step options and analysis options, save your database contents to a backup file. To do so, choose one of the methods shown below:

Command(s): SAVE

GUI: Utility Menu> File> Save As

Utility Menu> File> Save Jobname.db

Backing up your database prevents your model from being lost should your computer system fail. If you need to retrieve your model, choose either of the following:

Command(s): RESUME

GUI: Utility Menu> File> Resume Jobname.db

Utility Menu> File> Resume from

3.5.1. Solving the Model

To start the solution, choose either of the following:

Command(s): LSSOLVE

GUI: Main Menu> Solution> Solve> From LS Files

If you prefer, you can create and solve multiple load steps using array parameters or using the multiple solve method. For information about these methods, see the *ANSYS Basic Analysis Guide*.

To finish your solution and exit from the SOLUTION processor, choose either of the following:

Command(s): FINISH

GUI: Main Menu> Finish

3.6. Reviewing Analysis Results

ANSYS writes the results from a transient thermal analysis to the thermal results file, **Jobname.RTH**. Results contain the following data (all of which are functions of time):

- **Primary data**
 - Nodal temperatures (TEMP)
- **Derived data**
 - Nodal and element thermal fluxes (TFX, TFY, TFZ, TFSUM)
 - Nodal and element thermal gradients (TGX, TGY, TGZ, TGSUM)
 - Element heat flow rates
 - Nodal reaction heat flow rates
 - ...etc.

3.6.1. How to Review Results

You can review these results using either of the following:

- The general postprocessor, POST1. (**Main Menu> General Postproc.**) POST1 enables you to review results at one time step over the entire model or a selected part of the model.
- The time history postprocessor, POST26. (**Main Menu>TimeHist Postproc.**) POST26 lets you review results at specific points in the model over all time steps. Other POST26 capabilities include graph plots of results of data versus time or frequency, arithmetic calculations, and complex algebra.

The next few paragraphs describe some typical postprocessing operations for a transient thermal analysis. For a complete description of all postprocessing functions, see the *ANSYS Basic Analysis Guide*.

Note — To review results in either postprocessor, the ANSYS database must contain the same model for which the solution was calculated. (If necessary, retrieve the model.) In addition, the results file, **Job-name.RTH**, must be available.

3.6.2. Reviewing Results with the General Postprocessor

After you enter POST1, read in results at the desired time point. To do so, use either of the following:

Command(s): SET

GUI: Main Menu> General Postproc> Read Results> By Time/Freq

If you specify a time value for which no results are available, the ANSYS program performs linear interpolation to calculate the results at that time. ANSYS uses the *last* time point if you specify a time that is beyond the time span of the transient.

You also can have ANSYS read results by their load step and substep numbers. To do so, use the following menu path instead of the one shown above: **Main Menu> General Postproc> Read Results> By Load Step**.

Caution: *For a nonlinear analysis, linear interpolation of results data between time points can cause some loss of temporal accuracy. Therefore, take care to specify a time value for which a solution is available.*

3.6.3. Reviewing Results with the Time History Postprocessor

The time history postprocessor, POST26, works with tables of result items versus time, known as *variables*. ANSYS assigns each variable a reference number, with variable number 1 reserved for time.

If you are reviewing your analysis results using POST26, begin by defining the variables.

- To define variables for primary data, use either method below:
Command(s): NSOL
GUI: Main Menu> TimeHist Postproc> Define Variables
- To define variables for derived data, use the following command or GUI path:
Command(s): ESOL
GUI: Main Menu> TimeHist Postproc> Define Variables
- To define variables for reaction data, use either method below:
Command(s): RFORCE
GUI: Main Menu> TimeHist Postproc> Define Variables

Once your variables are defined, you can convert them to a graph, issue **PLVAR (Main Menu> TimeHist Postproc> Graph Variables)**. Choosing this command or menu path also gives you a listing of the variables.

To list only the extreme variable values, use either of the following:

Command(s): EXTREM

GUI: Main Menu> TimeHist Postproc> List Extremes

By reviewing the time-history results at strategic points throughout the model, you can identify the critical time points for further postprocessing with POST1.

POST26 offers many other functions including performing arithmetic operations among variables, moving variables into array parameters, and moving array parameters into variables. For details, see *ANSYS Basic Analysis Guide*.

3.7. Reviewing Results as Graphics or Tables

Once you have read results in, you can use ANSYS graphics displays and tables to review them. To display your results, use the menu paths shown below. Equivalent commands are shown in parentheses.

For examples of contour and vector displays, see either Chapter 2, “Steady-State Thermal Analysis” in this manual or various chapters in the *ANSYS Basic Analysis Guide*.

3.7.1. Reviewing Contour Displays

Command(s): PLESOL

GUI: Main Menu> General Postproc> Plot Results> Contour Plot> Element Solu

Command(s): PLETAB

GUI: Main Menu> General Postproc> Plot Results> Contour Plot> Elem Table

Command(s): PLNSOL

GUI: Main Menu> General Postproc> Plot Results> Contour Plot> Nodal Solu

3.7.2. Reviewing Vector Displays

Command(s): PLVECT

GUI: Main Menu> General Postproc> Plot Results> Vector Plot> Pre-defined or User-defined

3.7.3. Reviewing Table Listings

Command(s): PRESOL

GUI: Main Menu> General Postproc> List Results> Element Solution

Command(s): PRNSOL

GUI: Main Menu> General Postproc> List Results> Nodal Solution

Command(s): PRRSOL

GUI: Main Menu> General Postproc> List Results> Reaction Solu

3.8. Phase Change

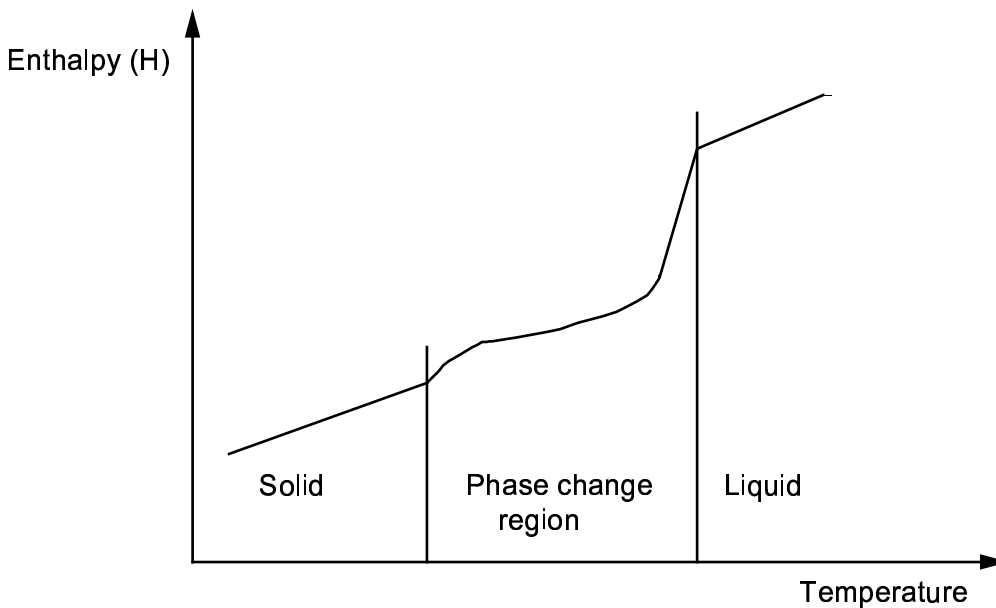
One of the ANSYS program's most powerful features for thermal analysis is its ability to analyze phase change problems, such as a melting or solidifying process. Some of the applications for phase change analysis include:

- The casting of metals, to determine such characteristics as the temperature distribution at different points during the phase change, length of time for the phase change to occur, thermal efficiency of the mold, etc.
- Production of alloys, where chemical differences instead of physical differences cause the phase change.
- Heat treatment problems.

To analyze a phase change problem, you perform a nonlinear transient thermal analysis. The only differences between linear and nonlinear transient analyses are that, in nonlinear analyses:

- You need to account for the latent heat; that is, heat energy that the system stores or releases during a phase change. To account for latent heat, define the enthalpy of the material as a function of temperature (see below):

Figure 3.2 Sample Enthalpy vs. Temperature Curve



Enthalpy, which has units of heat/volume, is the integral of density times specific heat with respect to temperature:

$$H = \int \rho c(T) dT$$

- In nonlinear analysis, you must specify a small enough integration time step for the solution. Also, turn on automatic time stepping so that the program can adjust the time step before, during, and after the phase change.
- Use lower-order thermal elements, such as PLANE55 or SOLID70. If you have to use higher-order elements, choose the diagonalized specific heat matrix option using the appropriate element KEYOPT. (This is the default for most lower-order elements.)
- When specifying transient integration parameters, set *THETA* to 1, so that the Euler backward difference scheme is used for the transient time integration. (*THETA* defaults to 0.5.)
- You may find the line search option helpful in phase change analyses. To exercise the line search option, use either of the following:

Command(s): LNSRCH

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Nonlinear > Line Search

3.9. Example of a Transient Thermal Analysis

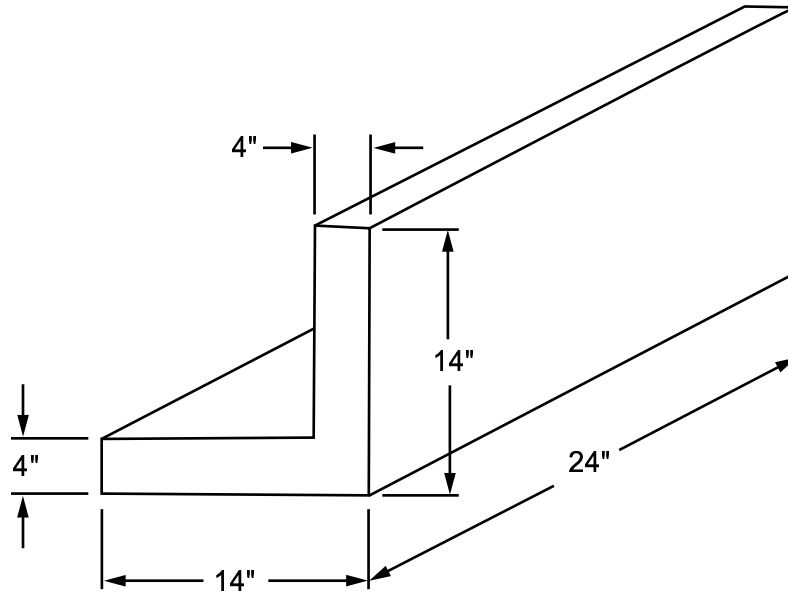
This section presents an example of a transient thermal analysis.

3.9.1. The Example Described

The example analysis this chapter describes is a transient heat transfer analysis of a casting process.

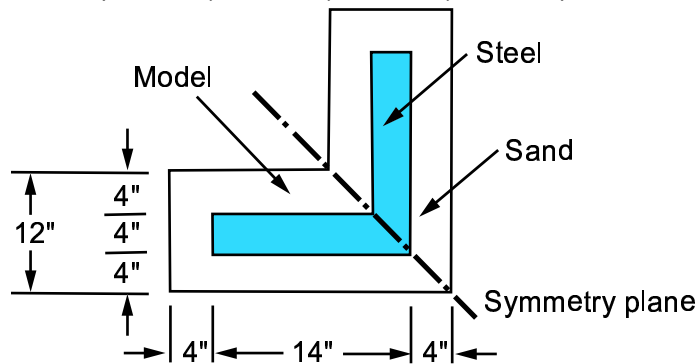
Note — A pictorial version of this example appears in the *Thermal Tutorial*.

This example tracks the temperature distribution in the steel casting and the mold during a three-hour solidification process. The casting is made in an L-shaped sand mold with four-inch thick walls. Conduction occurs between the steel and the sand mold, and convection occurs between the sand mold and the ambient air.



The example performs a 2-D analysis of a slice that is one unit thick. Half symmetry is used to reduce the size of the model. The lower half is the portion modeled.

To analyze the entire thickness of the model, use PLANE55 with KEYOPT(3) = 3 and specify the THK real constant. In this case, the temperature results will not be any different than modeling a one-unit thickness, but the heat flow results (**PRRSOL**, **PRRFOR**, **PRNSOL**, **PRESOL**) will be different.



3.9.2. Example Material Property Values

Sand and steel, the materials used in the sample analysis of the casting, have these properties:

| Item | U.S. Customary Measurement Units |
|---------------------------------------|----------------------------------|
| <i>Material properties for sand:</i> | |
| Conductivity (KXX) | 0.025 Btu/(hr-in-F) |
| Density (DENS) | 0.054 lb/in ³ |
| Specific heat (C) | 0.28 Btu/(lb- °F) |
| <i>Material properties for steel:</i> | |
| <i>Conductivity (KXX):</i> | |
| at 0 °F | 1.44 Btu/(hr-in- °F) |

| Item | U.S. Customary Measurement Units |
|-------------------------------|-------------------------------------|
| at 2643 °F | 1.54 |
| at 2750 °F | 1.22 |
| at 2875 °F | 1.22 |
| <i>Enthalpy (ENTH):</i> | |
| at 0 °F | 0.0 Btu/in ³ |
| at 2643 °F | 128.1 |
| at 2750 °F | 163.8 |
| at 2875 °F | 174.2 |
| <i>Initial conditions:</i> | |
| Temperature of steel | 2875 °F |
| Temperature of sand | 80 °F |
| <i>Convection properties:</i> | |
| Film coefficient | 0.014 Btu/(hr-in ² - °F) |
| Ambient temperature | 80 °F |

Material properties for the sand are constant. The steel casting has temperature-dependent thermal conductivity and enthalpy.

The solution method for this example uses automatic time stepping to determine the proper time step increments needed to converge the phase change nonlinearity. The transition from molten to solid steel uses smaller time steps.

3.9.3. Example of a Transient Thermal Analysis (GUI Method)

The example casting solidification analysis is included in the *Thermal Tutorial*.

3.9.4. Commands for Building and Solving the Model

The following sequence of ANSYS commands builds and solves the casting model. Comments (text preceded by the exclamation mark or ! character) explain what functions the commands perform.

```

/TITLE,CASTING SOLIDIFICATION !Give the analysis a title
/PREP7
K,1,0,0,0
K,2,22,0,0
K,3,10,12,0
K,4,0,12,0
/TRIAD,OFF !Turn triad symbol off
/REPLOT
!
A,1,2,3,4 !Connect keypoints to define mold area
SAVE
RECTNG,4,22,4,8 !Create a primitive rectangle
APLOT !Display areas
AOVLAP,1,2 !Overlap the areas
ADELE,3,,1 !Delete area 3
SAVE
!
MP,DENS,1,0.054 !Define sand properties
MP,KXX,1,0.025
MP,C,1,0.28
!
MPTEMP,1,0,2643,2750,2875,, !Define steel properties
MPDATA,KXX,2,1,1.44,1.54,1.22,1.22,,
MPDATA,ENTH,2,1,0,128.1,163.8,174.2

```

```

MPLOT,KXX,2,,,,,      !Plot steel conductivity
MPLOT,ENTH,2,,,,,    !Plot steel enthalpy
SAVE
!
ET,1,PLANE55          !Use element PLANE55
!
SAVE
SMRT,5                !Specify smart element sizing level 5
MSHAPE,0,2D           !Mesh with quadrilateral-shaped elements
MSHKEY,0              !Specify free meshing
AMESH,5               !Mesh mold area, area 5
!
TYPE,1                !Set element type attribute pointer to 1
MAT,2                 !Set element material attribute pointer to 2
REAL                  !Set element real const set attribute pointer
ESYS,0                !Set the element coord sys attribute pointer
AMESH,4               !Mesh casting area, area 4
!
SAVE
SFL,1,CONV,0.014,,80,, !Apply film coefficient and bulk temperature
SFL,3,CONV,0.014,,80,,
SFL,4,CONV,0.014,,80,,
SAVE
FINISH
/SOLU
!
ANTYPE,4              !Specify transient analysis
SOLCONTROL,ON,0      !Activate optimized nonlinear solu defaults
!
APLOT
ASEL,S,,,4            !Select casting area, area 4
NSLA,S,1              !Select nodes associated with casting area
NPLOT                 !Display casting area nodes
IC,ALL,TEMP,2875      !Apply initial condition of 2875F on casting
NSEL,INVE             !Select nodes of steel area, area 5
/REPLOT               !Display mold area nodes
IC,ALL,TEMP,80        !Apply initial condition of 80F on mold
ALLSEL,ALL            !Select all entities
SAVE
!
TIME,3                !Set time at end of load step
AUTOTS,-1             !Program chosen automatic time stepping
DELTIM,0.01,0.001,0.25,1 !Specify time step sizes
KBC,1                 !Specify stepped loading
!
OUTRES,ALL,ALL        !Write to file at every step
SAVE
/STAT,SOLU            !Display solution options
/REPLOT               !Display all nodes
APLOT                 !Display areas
SOLVE
FINISH
!
/POST26               !Time-history postprocessor
EPLOT                 !Display elements
cntr_pt=node(16,6,0) !Define postprocessing variable
NSOL,2,cntr_pt,TEMP,,center !Specify nodal data to be stored
PLVAR,2               !Display nodal temperature versus time
FINISH
/EOF

```

3.10. Where to Find Other Examples of Transient Thermal Analysis

Several ANSYS publications, particularly the *ANSYS Verification Manual* and the *Heat Transfer Training Manual*, describe additional examples of transient and other types of thermal analyses.

Attending the Heat Transfer seminar may benefit you if you analyze the thermal response of structures and components such as internal combustion engines, pressure vessels, heat exchangers and furnaces, etc. For more

information about this seminar, contact your local ANSYS Support Distributor or telephone the ANSYS Training Registrar at (724) 514-2882.

The *ANSYS Verification Manual* consists of test case analyses demonstrating the analysis capabilities of the ANSYS program. While these test cases demonstrate solutions to realistic thermal analysis problems, the *ANSYS Verification Manual* does not present them as step-by-step examples with lengthy data input instructions and printouts. However, most ANSYS users who have at least limited finite element experience should be able to fill in the missing details by reviewing each test case's finite element model and input data with accompanying comments.

The *ANSYS Verification Manual* contains a variety of transient thermal analysis test cases:

- VM28 - Transient Heat Transfer in an Infinite Slab
- VM94 - Heat Generating Plate
- VM104 - Liquid-Solid Phase Change
- VM109 - Temperature Response of a Suddenly Cooled Wire
- VM110 - Transient Temperature Distribution in a Slab
- VM111 - Cooling of a Spherical Body
- VM112 - Cooling of a Spherical Body
- VM113 - Transient Temperature Distribution in an Orthotropic Metal Bar
- VM114 - Temperature Response to a Linearly Rising Surface
- VM115 - Thermal Response of a Heat Generating Slab
- VM116 - Heat Conducting Plate with Sudden Cooling
- VM159 - Temperature Controlled Heater
- VM192 - Cooling of a Billet by Radiation

Chapter 4: Radiation

Radiation is the transfer of energy via electromagnetic waves. The waves travel at the speed of light, and energy transfer requires no medium. Thermal radiation is just a small band on the electromagnetic spectrum. Because the heat flow that radiation causes varies with the fourth power of the body's absolute temperature, radiation analyses are highly nonlinear.

The following radiation topics are available:

- 4.1. Analyzing Radiation Problems
- 4.2. Definitions
- 4.3. Using LINK31, the Radiation Link Element
- 4.4. Using the Surface Effect Elements
- 4.5. Using the AUX12 Radiation Matrix Method
- 4.6. Using the Radiosity Solver Method
- 4.7. Advanced Radiosity Options
- 4.8. Example of a 2-D Radiation Analysis Using the Radiosity Method (Command Method)
- 4.9. Example of a 2-D Radiation Analysis Using the Radiosity Method with Decimation and Symmetry (Command Method)

4.1. Analyzing Radiation Problems

The ANSYS program provides four methods for radiation analysis, each meant for a different situation:

- You can use LINK31, the radiation link element, for simple problems involving radiation between two points or several pairs of points.
- You can use the surface effect elements, SURF151 and SURF152 for radiation between a surface and a point.
- You can use the AUX12 Radiation Matrix method for more generalized radiation problems involving two or more surfaces. (Only the ANSYS Multiphysics, ANSYS Mechanical, and ANSYS Professional programs offer Radiation Matrix Generator.)
- You can also use the Radiosity Solver method for more generalized radiation problems in 3-D/2-D involving two or more surfaces. This method is supported by all 3-D/2-D elements having a temperature degree of freedom. (Only the ANSYS Multiphysics, ANSYS Mechanical, and ANSYS Professional programs offer Radiosity Solver.)

You can use the four radiation analysis methods for either transient or steady-state thermal analyses. Radiation is a nonlinear phenomenon, so you will need an iterative solution to reach a converged solution.

4.2. Definitions

The following definitions apply to terms used in radiation analysis.

- *Enclosure*: An open or closed enclosure in a radiation problem is a set of surfaces radiating to each other. In ANSYS, you can have many enclosures, with surfaces radiating to each other. ANSYS uses the definition of an enclosure to calculate view factors amongst surfaces belonging to an enclosure. Each open enclosure can have its own space temperature or space node which radiates to the ambient temperature.
- *Radiating Surfaces*: An open or closed enclosure can consist of many surfaces radiating to each other. Each radiating surface has an emissivity and a direction of radiation assigned to it. The Emissivity for a surface can be a function of temperature.

- *View Factors*: To compute radiation exchange between any two surfaces, you calculate the fraction of the radiation leaving surface i which is intercepted by surface j . This fraction is known as the view factor, form factor, or shape factor. In ANSYS, you can calculate view factors using the hidden/non-hidden method for 2-D and 3-D problems or the Hemicube method for 3-D problems.
- *Emissivity*: Emissivity is a surface radiative property defined as the ratio of the radiation emitted by the surface to the radiation emitted by a black body at the same temperature. ANSYS restricts radiation exchange between surfaces to gray-diffuse surfaces. The word grey signifies that emissivity and absorptivity of the surface do not depend on wavelength (either can depend on temperature). The word diffuse signifies that emissivity and absorptivity do not depend on direction. For a gray diffuse surface, emissivity = absorptivity; emissivity + reflectivity = 1. Note that a black body surface has a unit emissivity.
- *Stefan-Boltzmann Constant*: Stefan-Boltzmann constant provides the proportionality constant between the radiative heat flux and the forth power of temperature in the radiation model. The units for the constant depends on the absolute temperature units used in the ANSYS model.
- *Temperature Offset*: The unit of temperature plays an important role in radiation analysis. You can perform radiation calculations in absolute temperature units. If the model is defined in terms of degrees Fahrenheit or degrees Centigrade, you must specify a temperature offset. The temperature offset is 460° for the Fahrenheit system and 273° for the Centigrade system.
- *Space Temperature*: For an open enclosure problem, ANSYS requires specification of a space temperature for energy balance to the ambient. Each enclosure can have its own space temperature.
- *Space Node*: For an open enclosure problem, if the ambient is another body in the model, you can use the temperature of a space node to represent the free-space ambient temperature
- *Radiosity Solver*: The Radiosity Solver method accounts for the heat exchange between radiating bodies by solving for the outgoing radiative flux for each surface, when the surface temperatures for all surfaces are known. The surface fluxes provide boundary conditions to the finite element model for the conduction process analysis in ANSYS. When new surface temperatures are computed, due to either a new time step or iteration cycle, new surface flux conditions are found by repeating the process. The surface temperatures used in the computation must be uniform over each surface facet to satisfy the conditions of the radiation model.

4.3. Using LINK31, the Radiation Link Element

LINK31 is a 2-node, nonlinear line element that calculates the heat flow caused by radiation between two points. The element requires you to specify, in the form of real constants:

- An effective radiating surface area
- Form factor
- Emissivity
- The Stefan-Boltzmann constant.

Limit your use of the LINK31 analysis method to simple cases where you know, or can calculate easily by hand, the radiation form factors.

4.4. Using the Surface Effect Elements

A convenient way to model radiation between a surface and a point is to use surface effect elements superimposed on surfaces that emit or receive radiation. ANSYS provides such elements: SURF151 for 2-D models and SURF152 for 3-D models. The element option KEYOPT(9) activates radiation for these elements. The form factor can be specified as a real constant (defaults to 1) using KEYOPT(9) = 1, or you can calculate a cosine effect (using KEYOPT(9) = 2 or 3) from the basic element orientation and the extra node location.

4.5. Using the AUX12 Radiation Matrix Method

Offered in the ANSYS Multiphysics, ANSYS Mechanical, and ANSYS Professional programs only, this method works for generalized radiation problems involving two or more surfaces receiving and emitting radiation. The method involves generating a matrix of form factors (view factors) between radiating surfaces and using the matrix as a superelement in the thermal analysis. You also can include hidden or partially hidden surfaces, as well as a "space node" that can absorb radiation energy.

The following AUX12 topics are available:

- 4.5.1. Procedure
- 4.5.2. Recommendations for Using Space Nodes
- 4.5.3. General Guidelines for the AUX12 Radiation Matrix Method

4.5.1. Procedure

The AUX12 Radiation Matrix method consists of three steps:

1. Define the radiating surfaces.
2. Generate the radiation matrix.
3. Use the radiation matrix in the thermal analysis.

4.5.1.1. Defining the Radiating Surfaces

To define the radiating surfaces, you create a superimposed mesh of LINK32 elements in 2-D models and SHELL57 elements in 3-D models. To do so, perform the following tasks:

1. Build the thermal model in the preprocessor (PREP7). Radiating surfaces do not support symmetry conditions, therefore models involving radiating surfaces cannot take advantage of geometric symmetry and must therefore be modeled completely (except for 2-D axisymmetric cases). The radiating surfaces usually are *faces* of a 3-D model and *edges* of a 2-D model, as shown below:

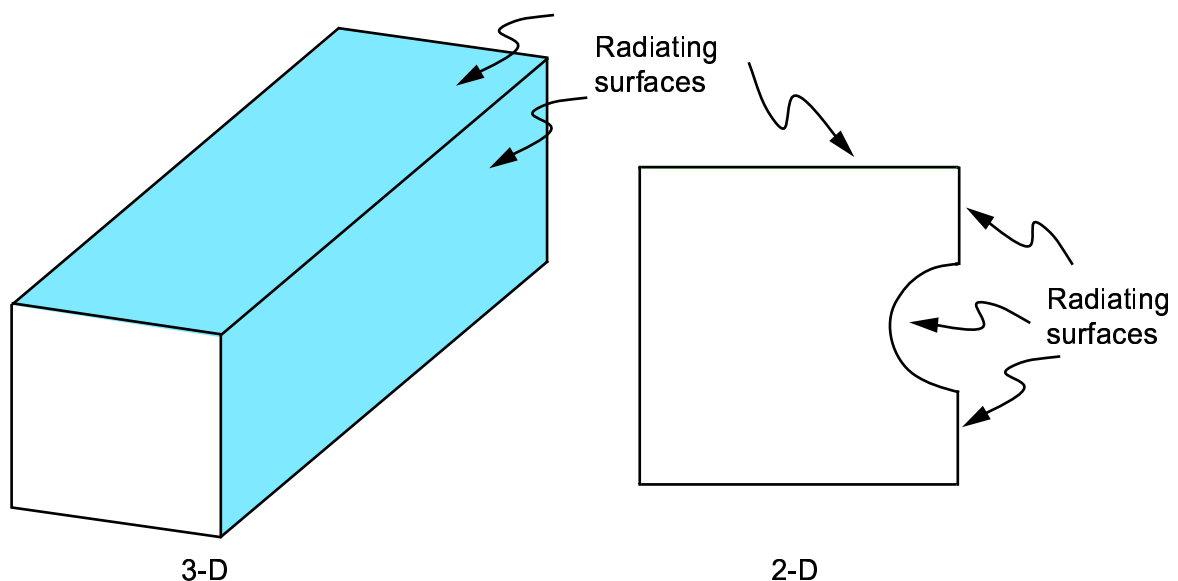


Figure 4.1 Radiating Surfaces for 3-D and 2-D Models

- Superimpose the radiating surfaces with a mesh of SHELL57 elements in 3-D models or LINK32 elements in 2-D models, as shown in the graphic below. The best way to do this is to first create a subset of the surface nodes, and then generate the surface elements using one of the following:

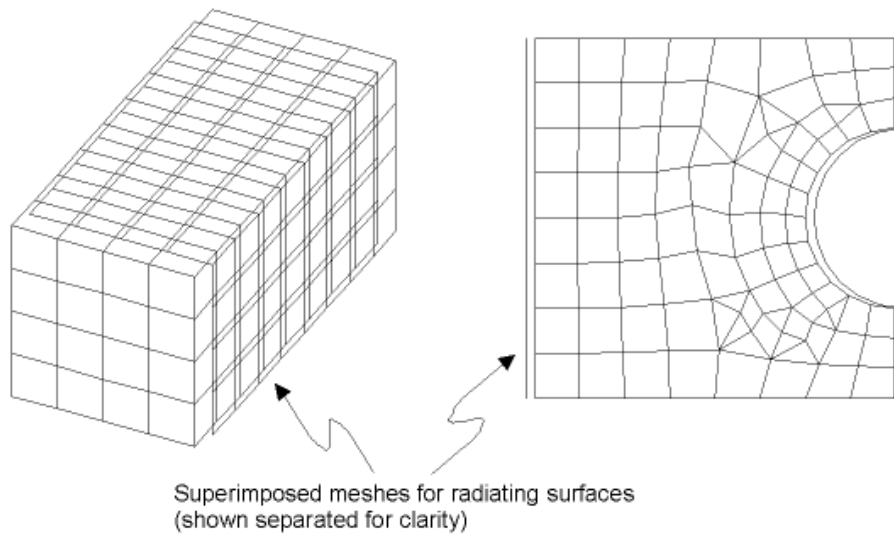
Command(s): ESURF

GUI: Main Menu > Preprocessor > Modeling > Create > Elements > Surf/Contact > Surf Effect > General Surface > Extra Node

Main Menu > Preprocessor > Modeling > Create > Elements > Surf/Contact > Surf Effect > General Surface > No extra Node

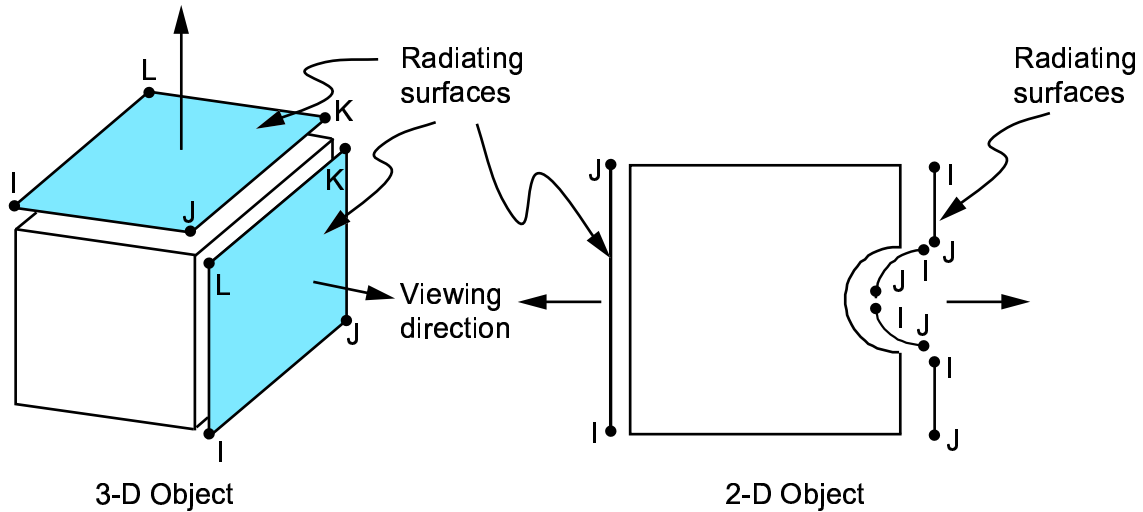
Make sure to first activate the proper element type for the surface elements. Also, if the surfaces are to have different emissivities, assign different material reference numbers before creating the elements.

Figure 4.2 Superimposing Elements on Radiating Surfaces



Caution: *Radiating surface mesh of SHELL57 or LINK32 elements must match (node for node) the underlying solid element mesh. If it does not match, the resulting thermal solution will be incorrect.*

The orientation of the superimposed elements is important. The AUX12 Radiation Matrix Generator assumes that the "viewing" direction (that is, the direction of radiation) is along $+Z_e$ for SHELL57 elements and along $+Y_e$ for LINK32 elements (where e denotes the outward normal direction of the element coordinate system). Therefore, you must mesh the superimposed elements so that the radiation occurs from (or to) the proper face. The order in which the element's nodes are defined controls the element orientation, as shown below:

Figure 4.3 Orienting the Superimposed Elements

3. Define a space node, which simply is a node that absorbs radiant energy not received by other surfaces in the model. Location of this node is not important. An open system usually requires a space node. However, you should not specify a space node for a closed system.

4.5.1.2. Generating the AUX12 Radiation Matrix

Calculating the radiation matrix requires the following inputs:

- Nodes and elements that make up the radiating surfaces
- Model dimensionality (2-D or 3-D)
- Emissivity and Stefan-Boltzmann constant
- The method used to calculate the form factors (hidden or visible)
- A space node, if desired.

To generate the matrix, perform these steps:

1. Enter AUX12 using one of these methods:
Command(s): /AUX12
GUI: Main Menu> Radiation Opt
2. Select the nodes and elements that make up the radiating surfaces. An easy way to do this is to select elements by type and then select all attached nodes. To perform these tasks, use the GUI path **Utility Menu> Select> Entities** or the commands **ESEL,S,TYPE** and **NSLE**. If you have defined a space node, remember to select it.
3. Specify whether this is a 2-D model or a 3-D model, using either of the following:
Command(s): GEOM
GUI: Main Menu> Radiation Opt> Matrix Method> Other Settings

The AUX12 Radiation Matrix Generator uses different algorithms to calculate the form factors for 2-D and 3-D models respectively. It assumes a 3-D model by default. The 2-D models may be either planar ($NDIV$ value = 0), or axisymmetric ($NDIV$ value > 0), with planar as the default. Axisymmetric models are expanded internally to a 3-D model, with $NDIV$ representing the number of axisymmetric sections. For example, setting $NDIV$ to 10 indicates ten sections, each spanning 36 degrees.

4. Define the emissivity using either method shown below. Emissivity defaults to unity (1.0).
Command(s): EMIS
GUI: Main Menu > Radiation Opt > Matrix Method > Emissivities
5. Define the Stefan-Boltzmann constant using either method shown below. The Stefan-Boltzmann constant defaults to $0.119\text{E-}10$ Btu/hr-in²-R⁴. (In S.I. Units, the constant has the value $5.67\text{E-}8$ W/m²-K⁴.)
Command(s): STEF
GUI: Main Menu > Radiation Opt > Matrix Method > Other Settings
6. Specify how to calculate form factors, using either of the following:
Command(s): VTYPE
GUI: Main Menu > Radiation Opt > Matrix Method > Write Matrix

You can choose between the hidden and non-hidden methods:

- The non-hidden method calculates the form factors from every element to every other element regardless of any blocking elements.
 - The hidden method (default) first uses a hidden-line algorithm to determine which elements are “visible” to every other element. (A “target” element is visible to a “viewing” element if their normals point toward each other and there are no blocking elements.) Then, form factors are calculated as follows:
 - Each radiating or “viewing” element is enclosed with a unit hemisphere (or a semicircle in 2-D).
 - All target or “receiving” elements are projected onto the hemisphere or semicircle.
 - To calculate the form factor, a predetermined number of rays are projected from the viewing element to the hemisphere or semicircle. Thus, the form factor is the ratio of the number of rays incident on the projected surface to the number of rays emitted by the viewing element. In general, accuracy of the form factors increases with the number of rays. You can increase the number of rays via the NZONE field on the **VTYPE** command or the **Write Matrix** menu option; both indicate the number of radial sampling zones.
7. If necessary, designate the space node using either of the methods shown below:
Command(s): SPACE
GUI: Main Menu > Radiation Opt > Matrix Method > Other Settings
 8. Use either the **WRITE** command or the **Write Matrix** menu option to write the radiation matrix to the file **Jobname.SUB**. If you want to write more than one radiation matrix, use a separate filename for each matrix. To print your matrices, issue the command **MPRINT,1** before issuing the **WRITE** command.
 9. Reselect all nodes and elements using either of the following:
Command(s): ALLSEL
GUI: Utility Menu > Select > Everything

You now have the radiation matrix written as a superelement on a file.

4.5.1.3. Using the AUX12 Radiation Matrix in the Thermal Analysis

After writing the radiation matrix, re-enter the ANSYS preprocessor (PREP7) and read the matrix in as a superelement. To do so, perform these steps:

1. Re-enter the preprocessor using one of these methods:
Command(s): /PREP7

GUI: Main Menu > Preprocessor Specify MATRIX50 (the superelement) as one of the element types.

2. Switch the element type pointer to the superelement using either of the following:
Command(s): TYPE
GUI: Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes
 3. Read in the superelement matrix using one of these methods:
Command(s): SE
GUI: Main Menu > Preprocessor > Modeling > Create > Elements > Superelements > From .SUB File
 4. Either unselect or delete the superimposed mesh of SHELL57 or LINK32 elements, using either of the following:
Command(s): EDELE
GUI: Main Menu > Preprocessor > Modeling > Delete > Elements
- (The thermal analysis does not require these elements.)
5. Exit from the preprocessor and enter the SOLUTION processor.
 6. Assign the known boundary condition to the space node using either of the following:
Command(s): D, F
GUI: Main Menu > Solution > Define Loads > Apply > option

This boundary typically is a temperature (such as ambient temperature), but also may be a heat flow. The boundary condition value should reflect the actual environmental conditions being modeled.

7. Proceed with the thermal analysis as explained in the other chapters of this manual.

4.5.2. Recommendations for Using Space Nodes

Although modeling radiation does not always require a space node, the decision to or not to use one can affect the accuracy of your thermal analysis results. Keep the following recommendations about space node usage in mind as you build your model.

4.5.2.1. Considerations for the Non-hidden Method

The non-hidden method of form factor calculation usually is accurate enough for any system without requiring special attention to space nodes. Generally, you *should not* specify a space node for a closed system, but you *should* specify one for an open system. Only one situation requires special attention: when modeling an open system which includes gray body radiation (emissivity is less than 1), you *must* use a space node to ensure accurate results.

4.5.2.2. Considerations for the Hidden Method

For the hidden method of form factor calculation, the accuracy of the form factor calculations within AUX12 can affect the accuracy of the radiation calculated to the space node. Because inaccuracies in the calculations accumulate at the space node, the relative error in the space node form factor can be exaggerated in a closed or nearly closed system.

When using the hidden method, you may need to increase the number of rays used in the form factor calculation and to refine the mesh in order to make the form factors more accurate. If this is not possible, consider the following tips when defining the space node:

- For a closed system in which all radiating surfaces form an enclosure and do not radiate to space, do not use a space node.

- If the nature of the problem makes it acceptable to simulate radiation between the radiating surfaces only (ignoring radiation to space), then do not specify a space node. This approach is valid only when modeling black body radiation (where emissivity = 1).
- For a nearly closed system, if you must account for radiation to space, then mesh the opening and constrain the temperature of the nodes in the opening to the temperature of space. The form factor to space will then be calculated explicitly and more accurately.
- For an open system where there are significant losses to space, you can use a space node (with a specified boundary condition) to capture the lost radiation with acceptable accuracy using moderate mesh refinement and a moderate number of rays.

4.5.3. General Guidelines for the AUX12 Radiation Matrix Method

Below are some general guidelines for using the AUX12 Radiation Matrix Generator approach to radiation analysis:

- The non-hidden method should be used if and only if all the radiating surfaces see each other fully. If the non-hidden method is used for cases where *any* blocking effect exists, there will be significant inaccuracies in view factor calculations, and the subsequent thermal analysis results can be physically inaccurate, or the problem might not even converge.
- The hidden method requires significantly longer computer time than the non-hidden method. Therefore, use it only if blocking surfaces are present or if surfaces cannot be grouped.
- In some cases, you may be able to group radiating surfaces so that each group is isolated completely from the other groups in terms of radiation heat transfer. In such cases, you can save significant time by creating a separate radiation matrix for each group using the non-hidden method. (This is true so long as no blocking effects exist within a group.) You can do this by selecting the desired group of radiating surfaces before writing the matrix.
- For the hidden method, increasing the number of rays usually produces more accurate form factors.
- For both hidden and non-hidden methods, the finer the mesh of the radiating surface elements, the more accurate are the form factors. However, when hidden method is used, it is particularly important to have a fairly refined mesh in order to obtain the same level of accuracy in view factor computation as the non-hidden method. Event though increasing the number of rays used (controlled by *NZONE* argument of the **VTYPE** command) helps in accuracy, for a coarse mesh, increasing *NZONE* to even its maximum limit might not yield an accurate solution for the hidden method.
- For axisymmetric models, about 20 axisymmetric sectors provide reasonably accurate form factors. Elements should have reasonable aspect ratios when they are expanded to a 3-D model.
- LINK32 elements, which are used as superimposed radiation surface elements in 2-D planar or axisymmetric models, do not directly support the axisymmetric option. In axisymmetric models, therefore, be sure to delete (or unselect) them before doing the thermal analysis.

Theoretically, the summation of view factor from any radiating surface to all other radiating surfaces should be 1.0 for a closed system. This is printed as ***** FORM FACTORS ***** TOTAL = *Value* for each radiating surface if the view factors for radiating surfaces are printed using the **MPRINT,1** command. For open systems, the summation should always be less than 1.0. One way of checking whether the view factor calculations are correct or not is to use the **MPRINT,1** command, and check if the summation of view factors for any radiating surface exceeds 1.0. This can happen if the non-hidden method is inadvertently used for calculating view factors between radiating surfaces with blocking effects. For more information, see Section 6.4: Radiation Matrix Method in the *ANSYS, Inc. Theory Reference*.

4.6. Using the Radiosity Solver Method

Offered in the ANSYS Multiphysics, ANSYS Mechanical, and ANSYS Professional programs only, this method also works for generalized radiation problems involving two or more surfaces receiving and emitting radiation. The method is supported by all 3-D/2-D elements having a temperature degree of freedom.

Elements supported for the radiosity method include:

FLOTRAN

FLUID141
FLUID142

ANSYS 2-D

PLANE13
PLANE35
PLANE55
PLANE67
PLANE77

The following radiosity topics are available:

- 4.6.1. Procedure
- 4.6.2. Further Options for Static Analysis

4.6.1. Procedure

The Radiosity Solver method consists of five steps:

1. Define the radiating surfaces.
2. Define Solution options.
3. Define View Factor options.
4. Calculate and query view factors.
5. Define load options.

4.6.1.1. Defining the Radiating Surfaces

You define the radiating surfaces by performing the following tasks:

1. Build the thermal model in the preprocessor (PREP7). Radiating surfaces support symmetry conditions in some cases; see Section 4.7: Advanced Radiosity Options for information on modeling symmetry for radiating surfaces. Symmetry conditions are not supported for FLOTRAN analyses using the radiosity method. For the Radiosity Solution Method radiating surfaces are faces of a 3-D model or sides of a 2-D model. In the Radiosity Solver Method, you can have up to ten enclosures, with surfaces radiating to each other.
2. Flag the radiation surfaces for a given emissivity and enclosure number using the **SF**, **SFA**, **SFE**, or **SFL** command. For all surface or line facets radiating to each other, issue the same enclosure number.

To specify temperature dependent emissivity, issue the **SF**, **SFA**, **SFE**, or **SFL** command with *VALUE* = -N. Emissivity values are from the EMIS property table for material N [MP]. Negative value of enclosure number is required for FLUID141 and FLUID142 elements to model radiation occurring between surfaces

inside the fluid domain. Positive value of enclosure number corresponds to radiation between surfaces in the solid domain.

Since radiation can pass through a fluid region and impact on a solid, you can apply the surface-to-surface radiation load on a fluid/solid interface, as well as on external model boundaries. In this case, you should apply the RDSF load to either the fluid or solid element faces, or the solid entity defining the interface. If you apply the load to more than one face, FLOTRAN applies the boundary conditions on only one face and issues a message that it skipped duplicate boundary conditions.

3. Verify the flagged radiation surfaces for properly specified emissivity, enclosure number and direction of radiation:

Command(s): /PSF

GUI: Utility Menu> PlotCtrls> Symbols

To apply radiation surface loads on the SHELL57 or SHELL157 elements, you must specify the face number with the exterior or interior orientation to properly flag it. You can use the **SF**, **SFA**, or **SFE** commands to apply these loads. The **SF** and **SFA** commands apply the radiation surface loads only on face 1 of the shell element. To apply radiation surface loads on face 2 or on both faces of the shell elements, use the **SFE** command. See SHELL57 and SHELL157 in the *ANSYS Elements Reference* for information on face orientation and numbering.

4.6.1.2. Defining Solution Options

For radiation problems, you must also define the Stefan-Boltzmann constant in the appropriate units:

Command(s): STEF

GUI: Main Menu> Preprocessor> Radiation Opts> Solution Opt

Main Menu> Radiation Opt> Radiosity Meth> Solution Opt

Main Menu> Solution> Radiation Opts> Solution Opt

If you define your model data in terms of degrees Fahrenheit or degrees Celsius, you must specify a temperature offset:

Command(s): TOFFST

GUI: Main Menu> Preprocessor> Radiation Opts> Solution Opt

Main Menu> Radiation Opt> Radiosity Meth> Solution Opt

Main Menu> Solution> Radiation Opts> Solution Opt

Next, select the Radiosity Solver and choose a direct solver or an iterative solver (default). You can also specify a relaxation factor and convergence tolerance for the heat flux:

Command(s): RADOPT

GUI: Main Menu> Preprocessor> Radiation Opts> Solution Opt

Main Menu> Radiation Opt> Radiosity Meth> Solution Opt

Main Menu> Solution> Radiation Opts> Solution Opt

If you are analyzing an open enclosure problem, you must specify the ambient temperature or the ambient node for each enclosure.

Specify the space temperature for the ambient radiation:

Command(s): SPCTEMP

GUI: Main Menu> Preprocessor> Radiation Opts> Solution Opt

Main Menu> Radiation Opt> Radiosity Meth> Solution Opt

Main Menu> Solution> Radiation Opts> Solution Opt

The **SPCTEMP** command specifies a space temperature for each enclosure. You can also list or delete all specified space temperatures using this command.

To specify a space node for each enclosure, use one of the following:

Command(s): SPCNOD

GUI: Main Menu > Preprocessor > Radiation Opts > Solution Opt

Main Menu > Radiation Opt > Radiosity Meth > Solution Opt

Main Menu > Solution > Radiation Opts > Solution Opt

If the ambient is another body in the model, you must specify the space node for the ambient radiation using the **SPCNOD** command for each enclosure. The **SPCNOD** command specifies a space node for each enclosure. The Radiosity Solver retrieves the nodal temperature for the specified node as the ambient temperature. You can also list or delete all specified space nodes using this command.

Note — In FLOTRAN in an axisymmetric radiosity analysis, you need to specify a space node even if the enclosure is closed.

To specify debug level for radiosity convergence output for FLOTRAN, use one of the following.

Command(s): FLDATA5, OUTP, DRAD

GUI: Main Menu > Preprocessor > FLOTRAN Set Up > Additional Out > Print Controls

The debug level defaults to 0 (none). A level of 1 (standard) provides final convergence information. A level of 2 (full) provides complete information for each global iteration.

4.6.1.3. Defining View Factor Options

To calculate new view factors for either 3-D or 2-D geometry, you can specify various options:

Command(s): HEMIOPT

GUI: Main Menu > Preprocessor > Radiation Opts > View Factor

Main Menu > Radiation Opt > Radiosity Meth > View Factor

Main Menu > Solution > Radiation Opts > View Factor

HEMIOPT allows you to set the resolution for 3-D view factor calculation using the Hemicube method. The default resolution is 10. Increasing the value increases the accuracy of the view factor calculation.

Command(s): V2DOPT

GUI: Main Menu > Preprocessor > Radiation Opts > View Factor

Main Menu > Radiation Opt > Radiosity Meth > View Factor

Main Menu > Solution > Radiation Opts > View Factor

V2DOPT allows you to select options for 2-D view factor calculation. The geometry type can be set to either 2-D plane or axisymmetric (defaults to plane). You can also define the number of divisions (defaults to 20) for an axisymmetric geometry.

ANSYS uses different algorithms to calculate the form factors for 2-D and 3-D models respectively. It assumes a 3-D model by default. The 2-D models may be either planar (*NDIV* value = 0), or axisymmetric (*NDIV* value > 0), with planar as the default. Axisymmetric models are expanded internally to a 3-D model, with *NDIV* representing the number of axisymmetric sections. For example, setting *NDIV* to 10 indicates ten sections, each spanning 36 degrees. This expansion is done only for view factor calculation, and not for the thermal solution.

The **V2DOPT** command also allows you to select hidden or non-hidden viewing option (defaults to hidden).

- The non-hidden method calculates the form factors from every element to every other element regardless of any blocking elements.
- The hidden method (default) first uses a hidden-line algorithm to determine which elements are “visible” to every other element. (A “target” element is visible to a “viewing” element if their normals point toward each other and there are no blocking elements.) Then, form factors are calculated as follows:

- Each radiating or “viewing” element is enclosed with a unit hemisphere (or a semicircle in 2-D).
- All target or “receiving” elements are projected onto the hemisphere or semicircle.
- To calculate the form factor, a predetermined number of rays are projected from the viewing element to the hemisphere or semicircle. Thus, the form factor is the ratio of the number of rays incident on the projected surface to the number of rays emitted by the viewing element. In general, accuracy of the form factors increases with the number of rays. You can increase the number of rays via the **NZONE** field on the **V2DOPT** command.

For more information, see the discussion on hidden and non-hidden options and axisymmetric geometry in Section 4.5: Using the AUX12 Radiation Matrix Method earlier in this chapter and Section 6.4: Radiation Matrix Method in the *ANSYS, Inc. Theory Reference*

You can specify whether new view factors should be computed or if existing values should be used:

Command(s): VFOPT

GUI: Main Menu > Preprocessor > Radiation Opts > View Factor

Main Menu > Radiation Opt > Radiosity Meth > View Factor

Main Menu > Solution > Radiation Opts > View Factor

VFOPT, *Opt* allows you to compute view factors and write them to a file (*Opt* = NEW). If view factors already exist in the database, this command also allows you to deactivate the view factor computation (*Opt* = OFF). OFF is the default upon encountering the second and subsequent **SOLVE** commands in **/SOLU**. After the first SOLVE command, ANSYS uses view factors existing in the database, unless they are overwritten by the **VFOPT** command.

VFOPT allows you to output view factors in ASCII or binary file format. Binary is the default.

4.6.1.4. Calculating and Querying View Factors

Next, you calculate the view factors. You can also query the view factor database and calculate an average view factor.

Compute and store the view factors:

Command(s): VFCALC

GUI: Main Menu > Radiation Opt > Radiosity Meth > Compute

List the calculated view factors for the selected source and target elements by querying the view factor database and calculate the average view factor:

Command(s): VFQUERY

GUI: Main Menu > Radiation Opt > Radiosity Meth > Query

You can retrieve the calculated average view factor using ***GET,Par,RAD,,VFAVG**. For FLOTRAN, you can retrieve the net heat rate lost by an enclosure using ***GET,Par,RAD,n,NETHF**.

4.6.1.5. Defining Load Options

Next, you specify an initial temperature if your model starts at a uniform temperature. You then specify the number or size of the time steps and specify a ramped boundary condition.

To assign a uniform temperature to all nodes, use one of the following:

Command(s): TUNIF

GUI: Main Menu > Solution > Define Loads > Settings > Uniform Temp

Set the number or size of time steps, using one of the following:

Command(s): NSUBST or DELTIM

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Time/Frequenc > Freq and Substps or Time and Substps

Main Menu > Preprocessor > Loads > Load Step Opts > Time/Frequenc > Time-Time Step

Due to the highly nonlinear nature of radiation, you should specify ramped boundary conditions:

Command(s): KBC

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Time/Frequenc > Time-Time Step

4.6.2. Further Options for Static Analysis

You can also solve a static problem using a false transient approach.

The analysis would include the following three steps:

1. Issue a constant density and specific heat for the model using the **MP** command. You should use a typical value of unit density and specific heat for the approach. The exact value for density and specific heat are not important as the problem finally approaches a steady-state solution.

2. Specify a transient analysis using one of the following:

Command(s): ANTYPE

GUI: Main Menu > Solution > Analysis Type > New Analysis

3. Run the quasi static radiation analysis to steady-state, using one of the following:

Command(s): QSOPT

GUI: Main Menu > Preprocessor > Loads > Load Step Opts > Time/Frequenc > Quasi-Static

The **QSOPT** command is available only when **SOLCONTROL** is ON. You can set the tolerance for the steady-state temperature using the **OPNCONTROL** command.

Depending on the material properties of the model (that is, density, specific heat, and thermal conductivity), temperature changes may be small at the beginning of a transient. With **QSOPT** on and the final time set to the default value (**TIME** = 1), you may obtain a solution before the true steady-state is reached. To obtain the true steady-state solution, use one of the following strategies:

- Tighten the steady-state temperature tolerance on the **OPNCONTROL** command. Be aware, though, it may take a long time to reach the true steady-state solution.
- Increase the final time (**TIME**) and the time step size (**DELTIM**) so that large temperature changes are captured at later time.

4.7. Advanced Radiosity Options

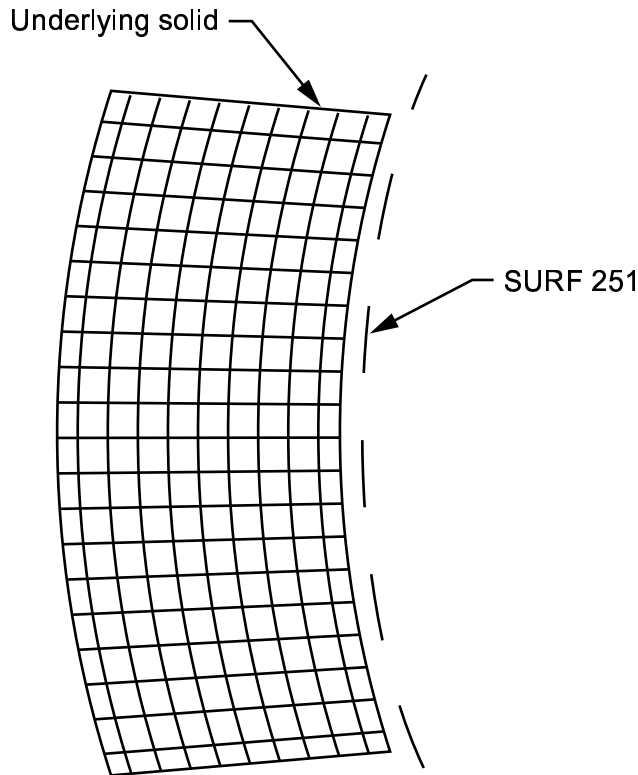
Use the advanced radiosity options to reduce the number of surface elements and then use symmetry to reduce the problem size. You must understand ANSYS' basic radiosity capabilities before using the advanced options.

The advanced radiosity options work with the same elements as the basic radiosity capability, except for FLUID141 and FLUID142, which are not supported for these advanced options.

1. Build the model in the preprocessor.
2. Select the appropriate set of solid elements to be flagged.
3. Apply any appropriate radiosity settings.

- Specify decimation parameters for the selected solid elements. Decimation allows you to use fewer radiation surface elements than there are underlying solid or shell element faces. Figure 4.4: “Decimation” illustrates this concept.

Figure 4.4 Decimation



Command(s): RDEC

GUI: Main Menu > Solution > Radiation Opt > Advanced Solution Option > Decimation Options > Define Specifications

Where different parts of the thermal model differ in size significantly, you should decimate these parts separately. Otherwise, smaller parts of the thermal model can be overdecimated.

You should estimate the number of radiosity surface elements on a decimated mesh before specifying the degree of decimation. The number should be enough to represent the original surface. For example, you would not want to represent a sphere using only five surface elements.

The goal of decimation is to reduce the time required for view factors calculation, as well as the heat flux calculation. For a small model with a small degree of decimation, the time saved for the view factors calculation could be offset by the amount of time required for the decimation calculations. Therefore, we recommend using decimation only for sufficiently large models.

- Specify symmetry options for the selected solid elements.

Command(s): RSYMM

GUI: Main Menu > Solution > Radiation Opts > Advanced Solution Option > Radiation Symmetry Options > Clear Symmetry

Use this command to specify either the plane of symmetry (POS) for planar reflection or the center of rotation (COR) for cyclic repetition. Note that POS reflection is NOT the same as COR repetition. Figure 4.5: “Planar Reflection” illustrates how the original sector is duplicated about a plane. Figure 4.6: “Cyclic

Repetition (Two Repetitions Shown)” illustrates how the original sector is duplicated about a center point.

The figures below show the results of planar and cyclic repetition. Issue **RSYMM**,,X for duplication around the X axis (Figure 4.5: “Planar Reflection”). Issue **RSYMM**,,n for a cyclic repetition (Figure 4.6: “Cyclic Repetition (Two Repetitions Shown)” uses **RSYMM**,,11; only 2 repetitions are shown in the figure).

Figure 4.5 Planar Reflection

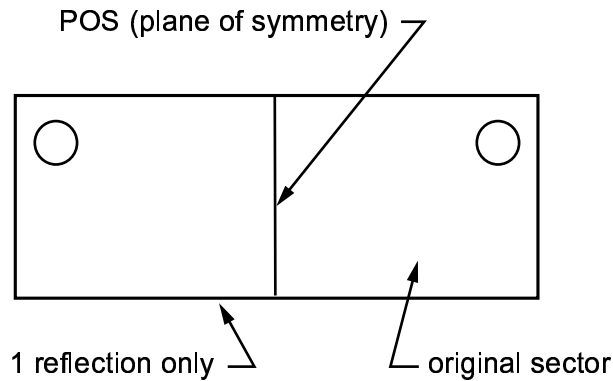
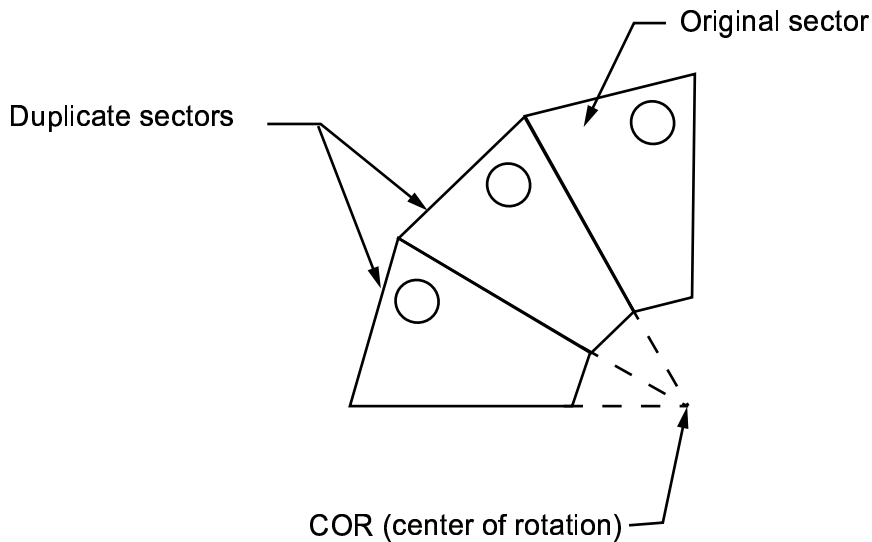
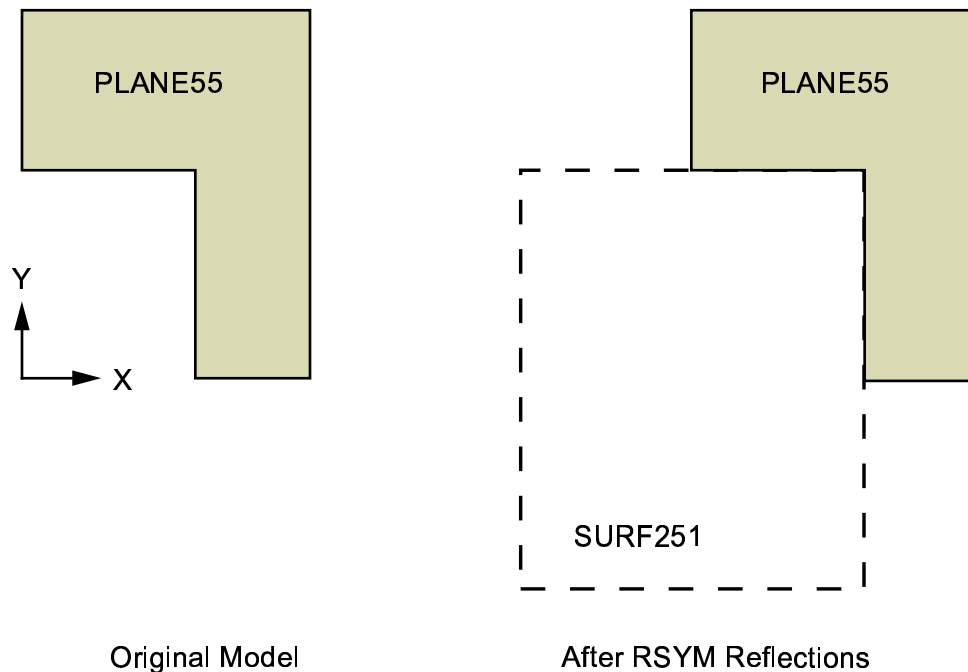


Figure 4.6 Cyclic Repetition (Two Repetitions Shown)



If you issue **RSYMM** more than once, each command will be processed in the order issued. For example, you could issue the following to turn condensation on, conduct a planar reflection about the global X axis, and then conduct a planar reflection about the global Y axis:

```
rsym,cond,,,,ON
rsym,,,x
rsym,,,y
```

Figure 4.7 Multiple RSYMM Commands

6. Generate the radiosity surface elements, SURF251/SURF252. Select the solid elements that you have flagged (using **SF**,,RDSF) and issue the following:

Command(s): RSURF

GUI: Main Menu > Radiation Opt >

If you need to regenerate the surface mesh (for example, unsatisfactory degree of decimation, improper symmetry reflection, etc.), delete the unsatisfactory results (**RSURF**,clear,last), adjust your decimation or symmetry parameters, and reissue the **RSURF** command.

The **RSURF** command applies symmetry reflections only to radiosity surface elements created by the current **RSURF** command, even if other elements are selected. You must use **RSURF** to create the surface elements; you cannot create SURF251/SURF252 elements manually using the **E**, **ESURF**, or **AMESH** commands.

7. Solve the model, and postprocess as usual. You can postprocess radiation heat flux using the NMISC records in SURF251 and SURF252.

If you save your database or model information (either through a **SAVE** or **CDWRITE** operation), the mapping information is automatically saved to a **.rsm** file if SURF251 and SURF252 elements are present in the model. A **.rsm** is useful for restarting your analysis. Without the **.rsm** file, you need to issue **RSURF**,DELE and then reissue **RSURF**,CREATE to recreate the mapped SURF251 and SURF252 elements. Doing so can be time-consuming for very large models.

To resume an analysis after you've issued a **SAVE** or **CDWRITE** and exited the ANSYS session:

1. Resume your database or **.cdb** file using **RESUME** or **CDREAD**. The mapping information is automatically saved to an **.rsm** file if SURF251 and SURF252 elements are present in the model. The **.rsm** file will be located in the directory specified by the **SAVE** or **CDWRITE** command.
2. Solve the model, and postprocess as usual.

You can also create the mapping (**.rsm**) file manually without issuing **SAVE** or **CDWRITE**. Issue the following command:

```
RSOPT,SAVE,file,ext,dir
```

where *file*, *ext*, and *dir* are the name, extension, and location of the file.

You can also read the **.rsm** manually (for example, if the **.rsm** file is located in a different directory than your database or **.cdb** file). Issue the following command:

```
RSOPT,LOAD,file,ext,dir
```

where *file*, *ext*, and *dir* are the name, extension, and location of the file.

There is no GUI equivalent for the **RSOPT** command.

Multi-field Restriction: When doing a multi-field analysis, we recommend that you first create all of the physics meshes in the database (either by using **MFIM** commands or by using meshing commands) and then create the SURF251/SURF252 elements using the appropriate combination of **RSYMM**, **RDEC**, and **RSURF** commands. At this point, you can save the file (via **SAVE** or **CDWRITE**), which will create the **.rsm** file for a later restart.

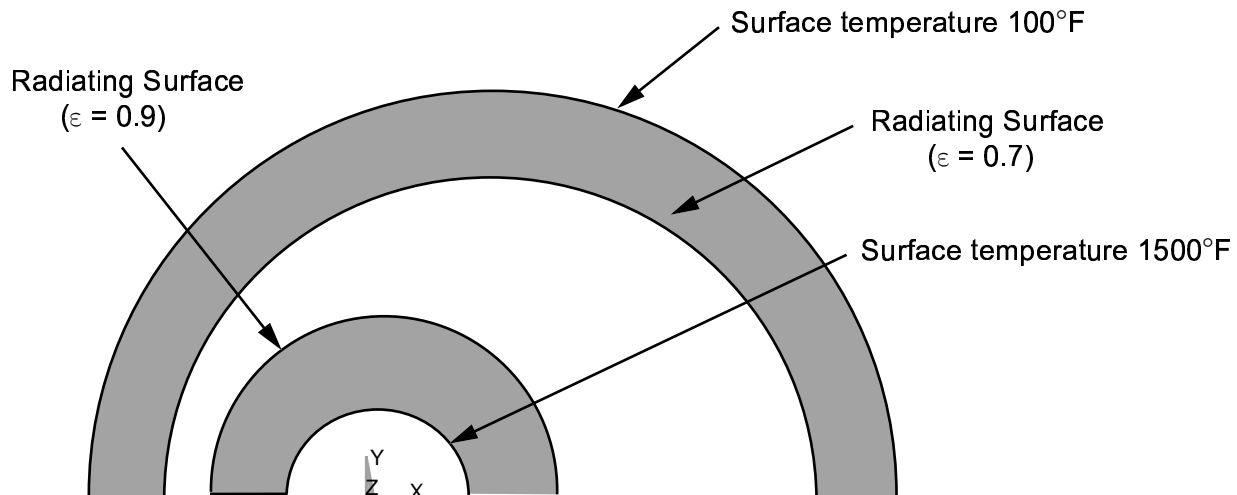
4.8. Example of a 2-D Radiation Analysis Using the Radiosity Method (Command Method)

This section describes how to do a steady-state thermal radiation analysis of a of a conical fin using the Radiosity Solver method by issuing a sequence of ANSYS commands, either while running ANSYS in batch mode or by issuing the commands manually during an interactive ANSYS session.

4.8.1. The Example Described

In this example, two circular annulus radiating to each other are considered. The outer surface of the inner annulus has an emissivity of 0.9. Its inner surface is maintained at a temperature of 1500°F. The inner surface of the outer annulus has an emissivity of 0.7, and its outer surface is maintained at a temperature of 100°F. The space temperature is maintained at 70°F.

Figure 4.8 Annulus



4.8.2. Commands for Building and Solving the Model

The following sequence of ANSYS commands builds and solves the finite element model. Text preceded by an exclamation mark (!) is comment text.

```

/TITLE,RADIATION BETWEEN CIRCULAR ANNULUS
! Example for 2-D radiation analysis using the radiosity method
/PREP7
CYL4,0,0,.5,0,.25,180    ! Circular annulus 1
CYL4,0.2,0,1,0,.75,180  ! Circular annulus 2
ET,1,PLANE55            ! 2-D thermal element
LSEL,S,LINE,,1
SFL,ALL,RDSF,.9, ,1,    ! Radiation boundary condition on inner annulus
LSEL,S,LINE,,7
SFL,ALL,RDSF,.7, ,1,    ! Radiation boundary condition on outer annulus
LSEL,S,LINE,,3
DL,ALL, ,TEMP,1500,1    ! Temperature on inner annulus
LSEL,S,LINE,,5
DL,ALL, ,TEMP,100,1    ! Temperature on outer annulus
ALLSEL
STEF,0.119E-10          ! Stefan-Boltzmann constant
TOFFST,460              ! Temperature offset
RADOPT,0.5,0.01,0,     ! Radiosity solver options
SPCTEMP,1,70           ! Space temperature for enclosure 1
V2DOPT,0.0,0,0,       ! 2-D view factor options
ESIZE,0.05,
AMESH,ALL
MP,KXX,1,.1            ! Thermal Conductivity
FINISH
/SOLU
TIME,1
DELTIM,.5,.1,1
NEQIT,1000
SOLVE
FINISH
/POST1
ASEL,S,AREA,,1
NSLA,S,1
PRNSOL,TEMP
FINISH

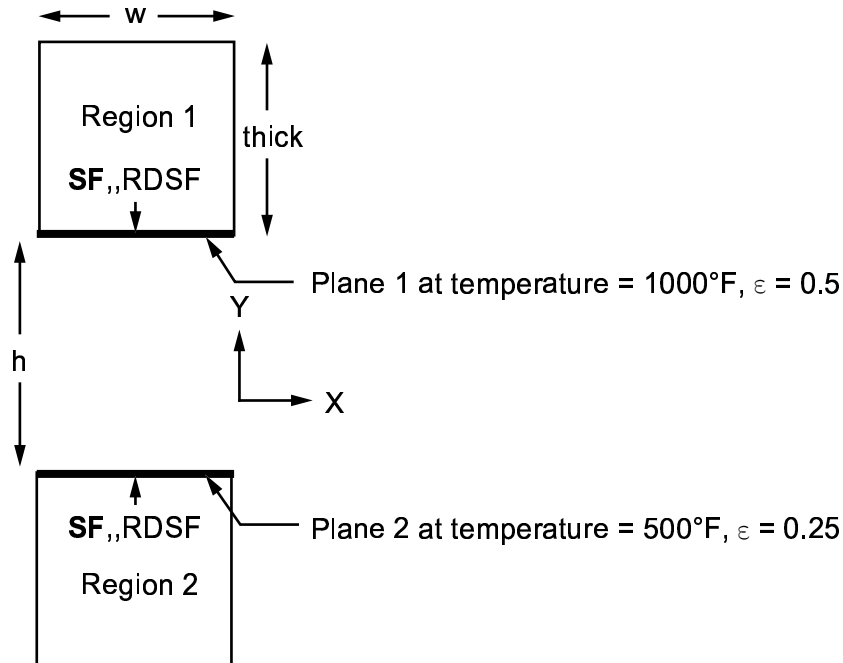
```

4.9. Example of a 2-D Radiation Analysis Using the Radiosity Method with Decimation and Symmetry (Command Method)

This section describes how to do a steady-state thermal radiation analysis of two parallel planes using decimation and symmetry by issuing a sequence of ANSYS commands, either while running ANSYS in batch mode or by issuing the commands manually during an interactive ANSYS session.

4.9.1. The Example Described

In this example, two parallel planes are considered for radiation. The underlying regions are meshed using PLANE55 elements. The first plane has a temperature of 1000°F and an emissivity of .5, and the second plane has a temperature of 500°F and an emissivity of .25.

Figure 4.9 Problem Geometry

4.9.2. Commands for Building and Solving the Model

The following sequence of ANSYS commands builds and solves the finite element model. Text preceded by an exclamation mark (!) is comment text.

```

/title,Radiation Problem Using Radiosity Surface Elements
/prep7
w      = 1
thick  = 1
h      = 0.06
!
tempoff = 270           ! Conversion to absolute temp
sbc    = 5.67e-8       ! Stefan-Boltzman constant
T1     = 1000
T2     = 500
emiss1 = 0.5
emiss2 = 0.25
!
rectng,-0.5*w,,0.5*h,0.5*h+thick
rectng,-0.5*w,,-0.5*h,-0.5*h-thick
!
et,1,55
mp,kxx,1,1
mshape,0,2D
mshkey,1
esize,0.125
lesize,all,,,,-1.5
amesh,all
!
! Specify temp/emissivity/rdsf on plane 1
!
nselect,s,loc,y,0.5*h
sf,all,rdsf,emiss1,1
d,all,temp,T1
!
! Specify temp/emissivity/rdsf on plane 2
!
nselect,s,loc,y,-0.5*h
sf,all,rdsf,emiss2,1
d,all,temp,T2

```

```

nset,all
allsel          ! Select PLANE55 nodes/elements for RSURF command

!
! Specify radiation options
!
tofst,tempoff
stef,cbc
radopt,1.0,1.0e-5,0,10000,,0.9
v2dopt,0,0,0,400
spctemp,1,0
vfopt,new,,,,asci
fini
/solu

rdec,,0.5      ! Set decimation to 50 percent reduction
rsymm,,0,x     ! Specify reflection about the x-axis
rsurf          ! Generate SURF251 elements and store in database

nlist          ! Includes SURF251 nodes
elist          ! Includes SURF251 elements
save
time,1
deltim,1

solve
fini
/post1
set,last
nset,s,loc,y,0.5*h   ! Select nodes of plane 1 and get nodal reaction
prsol
nset,s,loc,y,-0.5*h ! Select nodes of plane 2 and get nodal reaction
prsol
nset,all
*get,radnh,RAD,1,nethf ! Get the net outgoing radiant heat flux
! This should equal reaction 1 + reaction 2
*stat

!using nmisc element records to get net heat rate/emissivity/temp/
!enclosure/area/etc.

esel,s,type,,2 !select surf251
etable,elmarea,nmisc,4 ! Get element areas
etable,elmradnf,nmisc,7 ! Get element net outgoing radiant heat flux
smult,elmradnh,elmarea,elmradnf ! Multiply area*flux, store as heats
etable,elmradnf,erase
ssum          ! Get net area net heats.
! Net heat should = reaction 1 + reaction 2

!report element centroid & enclosure

etable,elmcenx,nmisc,1 ! Get element centroid x-coord
etable,elmceny,nmisc,2 ! Get element centroid y-coord
etable,elmcenz,nmisc,3 ! Get element centroid z-coord
etable,elmenc1,nmisc,18 ! Get element enclosure number
pretab,elmenc1,elmcenx,elmceny,elmcenz

!report element avg temp, emiss

etable,elmtmp,nmisc,5 ! Get element average temp
etable,elmemiss,nmisc,6 ! Get element average emissivity
pretab,elmtmp,elmemiss

!report netheatflux = emit+refl-inc

etable,elmradnf,nmisc,7 ! Get element net outgoing radiant heat flux
etable,elmradem,nmisc,8 ! Get element emitted heat flux
etable,elmradre,nmisc,9 ! Get element reflected heat flux
etable,elmradin,nmisc,10 ! Get element radiant heat flux
pretab,elmradnf,elmradem,elmradre,elmradin
fini

```

Index

A

ANSYS

- and thermal analysis, 1–1
- methods for radiation analysis, 4–1

ANSYS Professional, 2–1, 3–1

automatic time stepping, 2–10, 3–1, 3–8

AUX12 analysis method, 4–3

- guidelines for, 4–8

C

change of phase, 1–2

conduction, 1–1

contour displays, 2–14

convection, 1–1, 2–1, 2–6

convection film coefficient, 1–1

convergence tolerances, 3–8

coupled-field analysis, 1–2

E

element types, specifying, 2–4

elements

- for steady-state thermal analysis, 2–1

- for transient thermal analysis, 3–2

LINK31, 4–2

- superimposing on radiating surfaces, 4–3

- surface effect elements, 4–2

emissivity, 4–1, 4–5

enclosure, 4–1

F

form factors, calculating, 4–5, 4–7

function boundary conditions

- defining loads with, 2–7

G

geometry, choosing 2-D or 3-D, 4–5

H

heat flow rates, 2–1, 2–6

heat fluxes, 2–1, 2–6

heat generation rates, 2–1, 2–6

heat transfer, 1–1

heat transfer coefficients

- defining with functions, 2–7

I

interface

- material model, 1–2

internal heat generation, 1–2

K

keytime array, 3–4

keytimes, 3–4

L

load step options

automatic time stepping, 2–10, 3–1, 3–8

convergence tolerances, 2–10, 3–8

database and results file output, 2–11, 3–10

extrapolating results, 2–11, 3–10

for steady-state thermal analysis, 2–8

line searching, 2–10, 3–8, 3–13

number of equilibrium iterations, 2–10, 3–8

number of substeps, 2–9, 3–6

predictor option, 2–10, 3–8

printed output, 2–11, 3–10

solution control, 3–6

stepped or ramped loads, 2–9, 3–1, 3–6

terminating an unconverged solution, 2–10, 3–8

time integration effects, 3–8

time option, 2–9, 3–6

time step size, 2–9, 3–6

transient integration parameters, 3–8

load stepping

- applying in a transient thermal analysis, 3–4, 3–4

load vs. time curve, 3–1

loads

applying in a transient thermal analysis, 3–4

applying in steady-state thermal analysis, 2–5, 2–7

applying using table and function boundary conditions, 2–7

applying using TABLE array parameters, 2–5

stepped or ramped, 3–1

time-dependent, 3–1

M

magnetic-thermal analysis, 1–2

material model interface, 1–2

material properties

defining constant properties, 2–4

defining temperature-dependent properties, 2–4

defining values for, 2–4

monitor diagnostics results

monitor results in real time, 2–9, 3–6

N

Newton-Raphson option, 2–12

O

offset temperature, 2–12

P

parameters
 table type, 2–7
phase change, 3–13
POST1, 3–12
POST26, 3–12

R

radiating surfaces, 4–1
radiating surfaces, defining, 4–3
radiation, 1–1, 4–1
 definition of, 4–1
 methods of radiation analysis, 4–1
radiation link elements, 4–2
radiation matrix, 1–1, 4–1, 4–3, 4–5, 4–6
radiosity solver, 4–1, 4–9
resuming an analysis, 2–13

S

solution control options, 3–6
solvers
 Algebraic Multigrid (AMG) solver, 2–12
 frontal solver, 2–12
 Incomplete Cholesky Conjugate Gradient (ICCG)
 solver, 2–12
 Jacobi Conjugate Gradient (JCG) solver, 2–12
 JCG out-of-memory solver, 2–12
 PCG out-of-memory solver, 2–12
 Pre-Conditioned Conjugate Gradient (PCG) solver,
 2–12
 selecting, 2–12
space node, 4–1, 4–3, 4–3, 4–5, 4–7
space temperature, 4–1
steady-state thermal analysis, 2–1
 applying loads in, 2–5, 2–7
 building a model, 2–4
 definition of, 1–2, 2–1
 elements used in, 2–1
 examples of, 2–15
 linear analyses, 2–1
 load step options for, 2–8
 nonlinear analyses, 2–1
 reviewing results from, 2–13
Stefan-Boltzmann constant, 4–1, 4–5
stepped or ramped loads, 2–9, 3–6
surface effect elements, 4–2

T

table array parameters, 2–7
 defining loads with, 2–7
table listings of results data, 2–14
temperature, 2–1, 2–5

 applying a temperature DOF constraint, 3–3
 applying non-uniform starting temperature, 3–3
 deleting, 3–3
 specifying reference temperature, 3–3
 specifying uniform temperature, 3–3
temperature offset, 4–1
temperature-dependent film coefficient, 2–4
thermal analysis
 applications of, 1–1
 performed by ANSYS, 1–1
 purpose of, 1–1
 types of, 1–2
thermal gradients, 2–1
thermal stresses, 1–1
thermal-structural analysis, 1–2
time integration effects, 3–8
time step optimization, 3–8
time step size, 3–6
time stepping
 defining strategy in a transient thermal analysis, 3–4
 defining via tables, 3–4
time-dependent loads, 3–1
transient integration parameters, 3–8
transient thermal analysis, 3–1
 building a model, 3–2
 defining load steps, 3–4, 3–4
 defining time-stepping strategy, 3–4
 definition of, 1–2, 3–1
 elements used in, 3–2
 examples of, 3–14
 reviewing results as graphics or tables, 3–13
 reviewing results from, 3–11
 reviewing results in POST1, 3–12
 reviewing results in POST26, 3–12
 setting initial conditions for, 3–2
 specifying loads and load step options, 3–4
 time stepping via table, 3–4
 using the radiation matrix in, 4–6

V

variables, 3–12
vector displays, 2–14
view factors, 4–1