Chapter 5
ANSYS Solution and Postprocessing

5.1 Overview

A typical ANSYS session, regardless of the discipline, involves the following steps:

1. Model Generation
   - Specify *jobname* (this step is optional but recommended).
   - Enter *Preprocessor.*
   - Define element types and options.
   - Define real constant for the element types (if the element type(s) require real constants).
   - Define material properties.
   - Create the model:
     - Build solid model (using either top-down or bottom-up approach).
     - Define meshing controls.
     - Create the mesh.
   - Exit the Preprocessor.
2. Boundary/Initial Conditions and Solution
   - Enter Solution Processor.
   - Define analysis type and analysis options.
   - Specify boundary/initial conditions:
     - Degree of freedom constraints.
     - Nodal force loads.
     - Surface loads.
     - Body loads.
     - Inertia loads.
     - Initial conditions (if the analysis type is transient).
   - Save database (this step is not required but is recommended).

The online version of this book (doi: 10.1007/978-1-4939-1007-6_5) contains supplementary material, which is available to authorized users

E. Madenci, I. Guven, *The Finite Element Method and Applications in Engineering Using ANSYS®*, DOI 10.1007/978-1-4899-7550-8_5

– Initiate solution.
– Exit the *Solution Processor.*

3. Review Results

– Enter the appropriate Postprocessor (*General Postprocessor* or *Time History Postprocessor*).
– Display results.
– List results.

The first step involves operations concerning the *ANSYS Preprocessor* and was covered in detail in Chap. 4. The operations pertaining to the solution and post-processing of the results are discussed in detail in this chapter. At the end, specific steps are demonstrated by considering a one-dimensional transient heat transfer problem.

5.2 Solution

After preprocessing, the model generation, including meshing, is complete. The user is ready to begin the solution phase of the ANSYS session. First, the analysis type is specified from among the three main types:

- Static.
- Transient (time-dependent).
- Submodeling and substructuring (discussed in Sects. 11.3 and 11.4).

If the problem under consideration falls into the *Structural Analysis* discipline, then there are additional analysis types, such as *modal*, *harmonic*, *spectrum*, and *eigenvalue buckling*. There are two main deciding factors in choosing the analysis type:

Loading conditions: If the boundary conditions change as a function of time ***or*** there are initial conditions, then the analysis type is *Transient*. However, if the analysis discipline is structural ***and*** if the loading is a sinusoidal function of time, then the analysis type is *Harmonic*. Similarly, if the loading is a seismic spectrum, the analysis type is *Spectrum*.

Results of interest: If the analysis discipline is structural ***and*** if the results of interest are the natural structural frequencies, then the analysis type is *Modal*. Similarly, if the interest is in determining the load at which the structure looses stability (buckles), then the analysis type is *eigenvalue buckling*.

The analysis type is specified by using the following menu path:

Main Menu > Solution > Analysis Type > New Analysis

This brings up the dialog box shown in Fig. 5.1. The user selects a particular analysis type by clicking on the corresponding radio-button and clicks ***OK***. The common

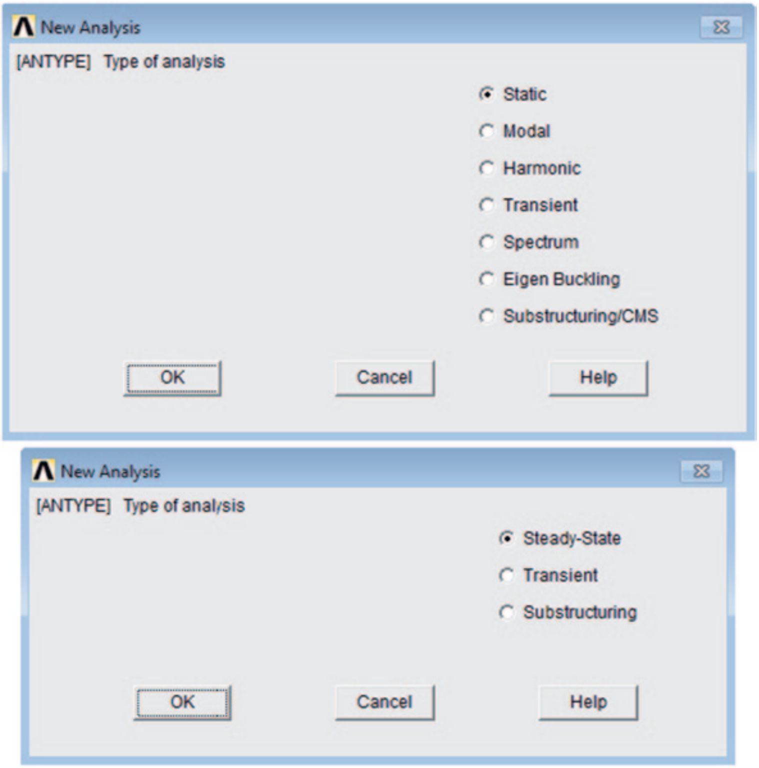


Fig. 5.1 Dialog boxes for selecting the type of analysis for structural (*top*) and thermal (*bottom*) disciplines

solution operations used in almost every ANSYS session are discussed in the following subsections.

5.2.1 Analysis Options/Solution Controls

ANSYS allows the user to select certain options during the solution phase. They are specified through either *Analysis Options* or *Solution Controls*. The *Analysis Options*, specific to the *Analysis Type*, permit the user to select the method of solution and related details; this step requires familiarity with the *Analysis Type*. *Analysis Options* can be specified by the following the menu path:

Main Menu > Solution > Analysis Type > Analysis Options

Because the *Analysis Options* are specific to the particular problems under consideration, related discussions are covered in various example problems throughout this book.

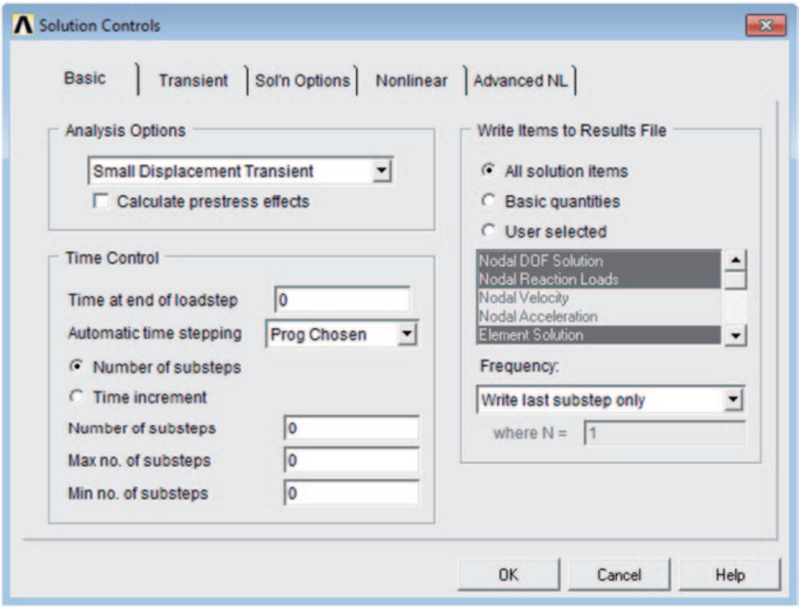


Fig. 5.2 *Solution controls* dialog box

In addition to the *Analysis Options*, the user has the option of specifying preferences through the *Solution Controls*. The main difference between the *Analysis Options* and the *Solution Controls* is that the *Solution Controls* are not specific to the *Analysis Type*. The same set of options in the *Solution Controls* can be used in a structural or thermal analysis. The *Solution Controls* dialog box can be activated by using the following menu path:

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

As shown in Fig. 5.2, the *Solution Controls* dialog box has five different tabs:

Basic: Involves selection of options specific to the analysis type, time-domain-related parameters, and results items to be written to the *Results File*.

Transient: This option provides control over the way the loading is applied (stepped or ramped over time), the damping coefficients, and the time integration parameters.

Sol'n Options: The equation solver is chosen under this option. Also, if the current analysis is a *Restart* from a previous analysis or is intended to be "restarted" later, this option controls the number of restart files to write and the frequency at which they are written.

Nonlinear: Involves nonlinear options, specification of the maximum number of equilibrium iterations, and the limits on physical values used to perform bisection when performing nonlinear structural analyses, such as plasticity deformation, creep, etc.

Advanced NL: This option is used to specify what the software should do when convergence is not achieved during a nonlinear analysis.

Within the *Solution Controls*, all of the options have default values that the user is **not** required to specify while performing the analysis. However, if the analysis fails to produce convergence, manual specification of these options may improve the chances of convergence. As part of the example problems solved throughout this book, the *Solution Controls* items are manually specified for several problems in Chaps. 8 and 9.

5.2.2 Boundary Conditions

In a well-posed mathematical problem, the conditions along the entire boundary must be known. These conditions are referred to as the *boundary conditions*, and they can be specified in three different ways:

Type I: Specification of the primary variable (degree of freedom).
Type II: Specification of variables related to the derivative of the primary variable.
Type III: Specification of a linear combination of the primary variable and its derivative.

In a *Structural* problem, the primary variables are the displacement components (see Sect. 2.2.1.3). When *Type I* boundary conditions are used, the displacement constraints are specified along a segment of the boundary. If tractions are specified along the boundary, the boundary conditions fall under *Type II* because the tractions are related to the derivatives of the displacement components. A special case of the traction boundary conditions is the point load (also called the *Force/Moment* load). When the structure is subjected to tractions over a rather small area of the boundary, it is reasonable to idealize this condition as a concentrated load applied at a point. While conducting an analysis with **`BEAM`** or **`SHELL`** element types, moment loads can also be applied. Displacements, pressures (normal tractions), forces, and moments can be specified using the following menu paths:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement
Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment
Main Menu > Solution > Define Loads > Apply > Structural > Pressure

In a *Thermal* problem, the temperature is the primary variable. Similar to *Structural* problems, *Type I* boundary conditions correspond to the specification of the primary variable, i.e., temperature over a portion of the boundary. Specified heat flux conditions fall under *Type II* boundary conditions. Finally, convective conditions correspond to *Type III* boundary conditions. Temperature, heat flux, and convective conditions along the boundaries can be specified using the following menu paths:

Main Menu > Solution > Define Loads > Apply > Thermal > Temperature
Main Menu > Solution > Define Loads > Apply > Thermal > Heat Flux
Main Menu > Solution > Define Loads > Apply > Thermal > Heat Flow
Main Menu > Solution > Define Loads > Apply > Thermal > Convection

Each of the boundary conditions discussed in this section can be applied on *Nodes* or on appropriate solid model entities such as *Keypoints*, *Lines,* or *Areas*. If they are applied on the solid model entities, ANSYS transfers them to the nodes when the solution is initiated. Although the paths for specification of boundary conditions are shown under the *Solution Processor*, it is possible to apply them under the *Preprocessor*.

5.2.3 Initial Conditions

A well-posed transient problem requires the specification of initial conditions. For *Structural* problems, initial conditions may involve components of displacement, rotation, velocity, or acceleration. In a *Thermal* problem, initial conditions are typically the temperature distribution within the domain. Initial conditions can be specified only when the *Analysis Type* is selected as *Transient*; if the *Analysis Type* is selected as *Static*, the specification of initial conditions does not appear as an option in the menus. Initial conditions can be specified using the following menu paths:

Main Menu > Solution > Define Loads > Apply > Initial Condit'n

5.2.4 Body Loads

Body loads can be generated internally or externally as the result of a physical field acting on the body. They act within the domain expressed volumetrically. Gravity, inertia loads, and temperature change represent body loads in a *Structural* problem. They can be specified using one of the following menu paths:

Main Menu > Solution > Define Loads > Apply > Structural > Temperature
Main Menu > Solution > Define Loads > Apply > Structural > Inertia > Angular Velocity
Main Menu > Solution > Define Loads > Apply > Structural > Inertia > Angular Accel
Main Menu > Solution > Define Loads > Apply > Structural > Inertia > Coriolis Effects
Main Menu > Solution > Define Loads > Apply > Structural > Inertia > Gravity

Heat generation within the domain is also represented as a body load for *Thermal* problems and can be specified using the following menu path:

Main Menu > Solution > Define Loads > Apply > Thermal > Heat Generat

5.2.5 Solution in Single and Multiple Load Steps

After completing the finite element mesh and specifying the loading conditions (boundary, initial, and body loads), the solution can be initiated using the following menu path:

Main Menu > Solution > Solve > Current LS

However, there are cases in which the loads are time-dependent, and the solution is achieved in multiple steps. Different load steps must be used if the loading on the structure changes abruptly. The use of load steps also becomes necessary if the response of the structure at specific points in time is desired. ANSYS accommodates the application of time-dependent loads through the use of multiple *Load Steps*. Time-dependent loading is commonly encountered in analyses involving the determination of dynamic response and viscoplastic- and creep-type material behaviors. Simulation of manufacturing processes also involves time-dependent thermal loading. Figure 5.3 illustrates different profiles of impact loading as a function of time. The solid lines designate the actual loading while the dashed lines denote the loading profiles as specified in ANSYS. The solid circles indicate the times at which a load step starts or ends. As observed in Fig. 5.3, ANSYS permits the user to specify either step or ramped loading. In all of the cases, the last load step is necessary in order to capture the response of the structure at times after the load is removed. The multiple load steps are also necessary in modeling a viscoplastic material subjected to thermal cycling, as shown in Fig. 5.4.

The following steps are used in order to use multiple load step solution method:

1. **Apply the initial conditions as explained in Sect. 5.2.3.**
2. **Apply the boundary conditions appropriate for the first load step.**
3. **Specify time-related parameters**: This is performed by using the following menu path:

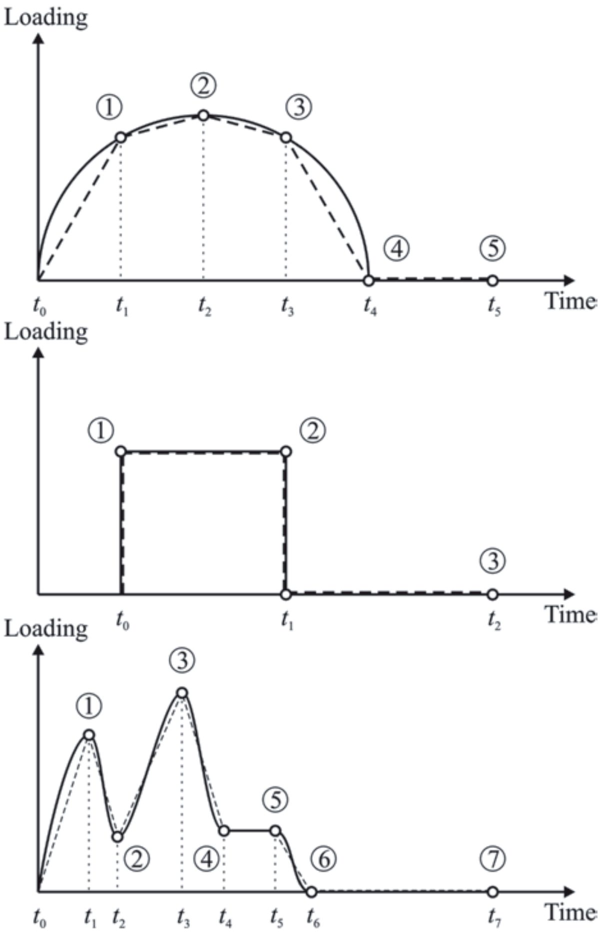
Main Menu > Solution > Load Step Opts > Time/Frequenc > Time—Time Step

This brings up the *Time and Time Step Options* dialog box (Fig. 5.5) Enter the *time at end of load step* (***TIME***) and the *time step size* (***DELTIM***), which is optional. Choose between whether the loads are applied in a stepped or ramped manner (***KBC***). If the *Automatic Time Stepping* is ***OFF***, then the user must specify the *time step size*. If the *time step size* is not specified and the *Automatic Time Stepping* is set as ***Prog Chosen*** (stands for program chosen), then ANSYS turns the *Automatic Time Stepping* ***ON***. If the *time step size* is specified and the *Automatic Time Stepping* is ***ON***, ANSYS starts the solution with the specified *time step size* and modifies it based on the convergence.

4. **Write Load Step file:** Load steps are written to *Load Step Files* by using the following menu path:

Main Menu >Solution > Load Step Opts > Write LS File

Fig. 5.3 Different profiles of impact loading as functions of time



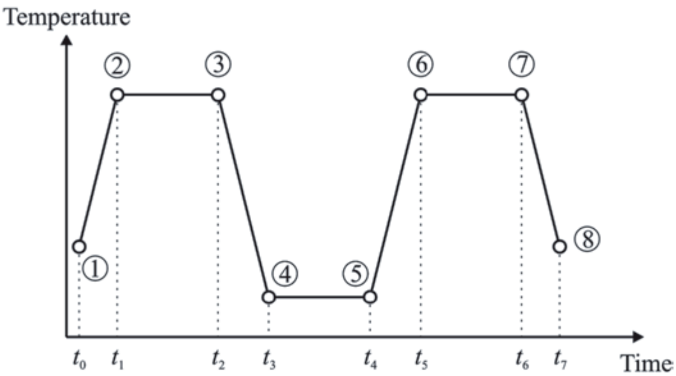


Fig. 5.4 Cyclic thermal loading

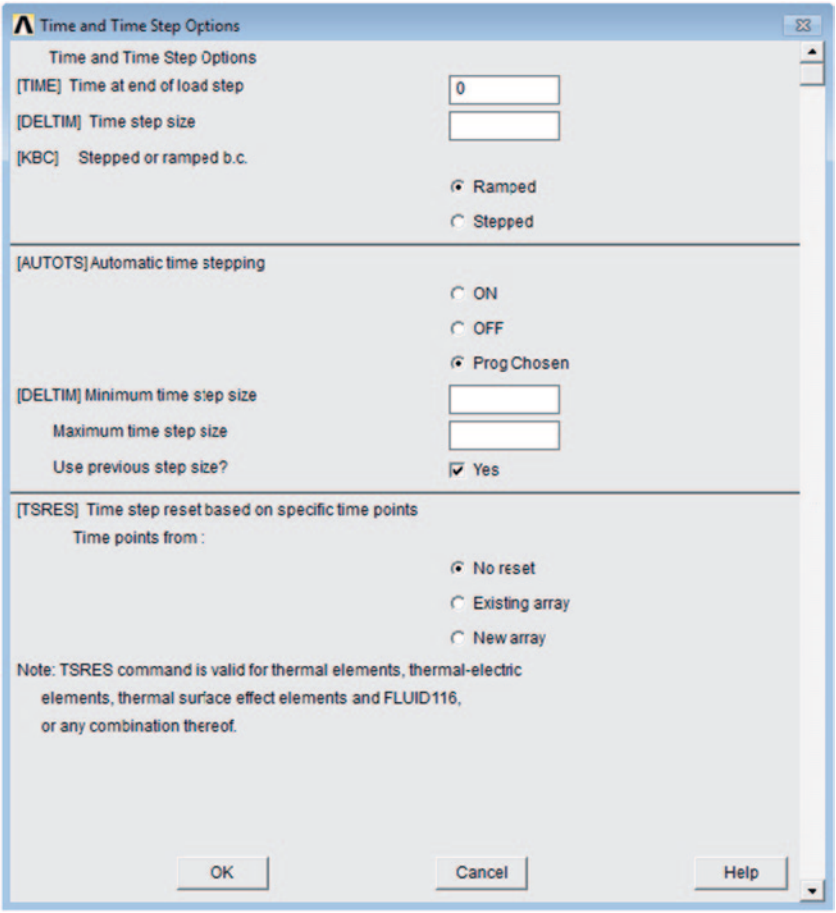


Fig. 5.5 *Time and time step options* dialog box

This brings up the *Write Load Step* File dialog box (Fig. 5.6). Enter the load step file number (***LSNUM***) and hit ***OK***. This file is stored in the *Working Directory* and contains all the solution options, the time, and time-related parameters, as well as the boundary conditions.

5. **Repeat steps 2–4 for the remainder of the load steps.**
6. **Initiate solution from *Load Step files*:** Once all of the *load step files* are written, the solution is initiated by using the following menu path:

Main Menu > **Solution** > **Solve** > **From LS Files**

which brings up the *Solve Load Step Files* dialog box. Enter the starting and ending load step file numbers (***LSMIN*** and ***LSMAX***) and hit ***OK***.

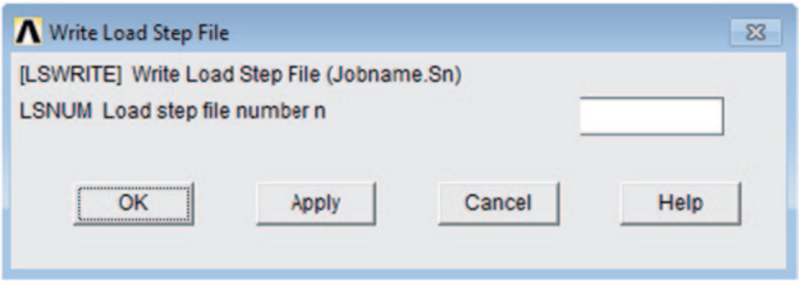


Fig. 5.6 *Write load step file* dialog box

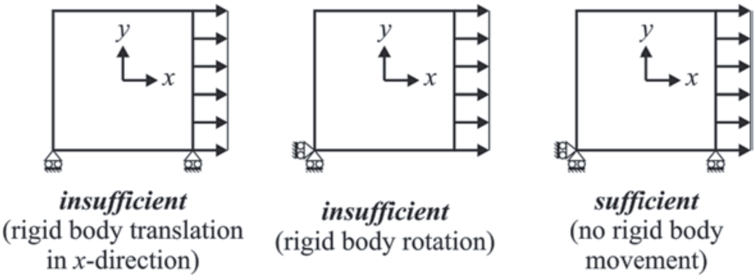


Fig. 5.7 Instability due to lack of constraints (*left* and *middle*); stable configuration

5.2.6 Failure to Obtain Solution

There are two common reasons why ANSYSfails to provide a solution:

Singular coefficient matrix: As shown in detail in Chap. 1, every finite element solution involves the solution of a system of equations with a known coefficient matrix (stiffness), an unknown degree of freedom vector, and a known right-hand-side (force) vector. If the coefficient matrix is singular, the solution fails. The most common reasons why the coefficient matrix becomes singular are as follows:

1. Instability in the structure due to lack of constraints in static structural analyses. This leads to rigid-body translations and rotations, which makes the stiffness matrix singular. As an example of this phenomenon, Fig. 5.7 shows three distinct constraint configurations applied on the same 2-D square structure subjected to a distributed tensile load in the *x*-direction. The first configuration involves two constraints, both suppressing displacements in the *y*-direction, along the bottom surface of the structure. Because there are no constraints suppressing displacements in the *x*-direction, the structure is free to move in the *x*-direction under the applied load, thus leading to a singular stiffness matrix. In the second configuration, displacements in both the *x*- and *y*-directions are suppressed at the

same point (bottom left corner). Although this configuration prevents rigid-body translations, it fails to prevent rigid-body rotation around the corner node where displacement constraints are applied, causing the stiffness matrix to become singular. Finally, in the third configuration, two corners are constrained in the y-direction, with one of them also constrained in the x-direction. This is a stable configuration, preventing all possible rigid-body movements, leading to a nonsingular stiffness matrix and thus a successful unique solution.

– Material properties that are physically impossible may make the coefficient matrix singular. Examples include zero or negative Young's modulus, thermal conductivity, density, or specific heat.
– There are structural elements within the ANSYS element library that carry loads only along their line of direction (**`SPAR`** elements simulating truss structures). Stability concerns of *Statics* apply to structures made up of these elements, and the user must make sure that the structure is stable.

2. ***Failed convergence***: In finite element analyses, problems involving nonlinearity are solved through iterations. As described in Chap. 2, these nonlinearities arise through the material behavior (plasticity, creep, viscoelasticity, viscoplasticity, etc.) or geometric configuration (large deformations) of the structure. The "correct" solution is approached in small *steps*, referred to as convergence iterations. If the problem is time-dependent, then the small *steps* are taken in the time domain. If the problem is not dependent on time (e.g., plasticity), these small steps are taken in the application of the loads. At the end of each iteration, ANSYS checks whether the solution satisfies a *convergence criterion* "built-in" for different analysis types. If the criterion is not satisfied, the last step is repeated with a smaller step size. This is repeated until the convergence criterion is satisfied. However, there are limits on the number of convergence iterations and, if a converged solution is not achieved within those limits, ANSYS terminates the solution process. Because each nonlinear analysis type is different, there is no straightforward answer as to what to do to improve the chances of a successful convergence. However, several nonlinear problems are considered in Chap. 10 that may give the reader some ideas on convergence considerations.

5.3 Postprocessing

After a solution is obtained in an ANSYS session, the user can review the results in either the *General Postprocessor* or the *Time History Postprocessor*. If the problem is static (or steady state), then the *General Postprocessor* is the only postprocessor where the results can be reviewed. However, if the problem is dependent on time (transient), both processors are useful for distinctly different tasks. The postprocessors and common postprocessing operations are discussed briefly in the following (5.3.1–5.3.6).

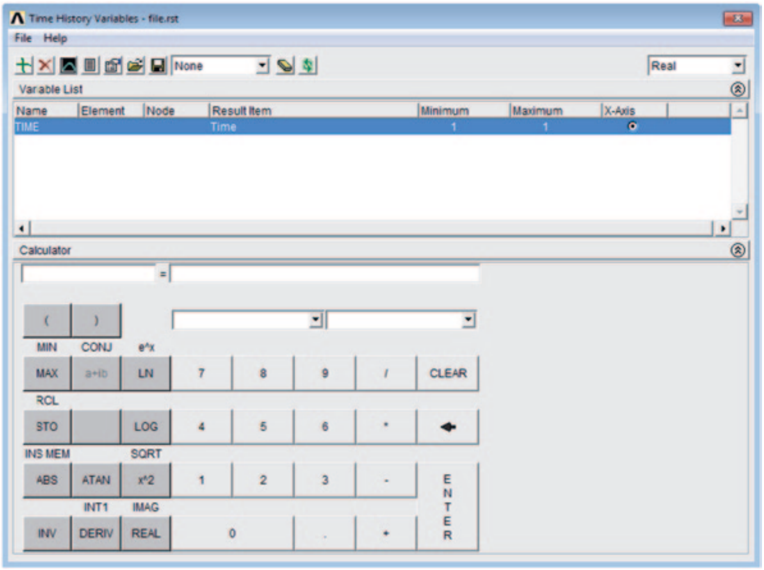


Fig. 5.8 *Time history variables* dialog box

5.3.1 General Postprocessor

In the *General Postprocessor*, the results of a solution at a specific time (if the problem is time dependent) are reviewed. Available options for review include graphical displays and a listing of results. It is also possible to perform sorting and mathematical operations on the results.

5.3.2 Time History Postprocessor

When the problem under consideration is time dependent, the time variation of the results at specific locations (nodes) are reviewed under the *Time History Postprocessor*. Upon entering this postprocessor, the *Time History Variables* dialog box appears (Fig. 5.8). This dialog box has three distinct areas: ***Toolbar***, ***Variables***, and ***Calculator***. The first four buttons (from the left) in the ***Toolbar*** are the most commonly used ones:

Add Data Button: This button is used to define new variables, such as displacements, temperatures, etc., at specific nodes.
Delete Data Button: Used for deleting defined variables.
Graph Data: Using this button, the user can plot the time variation of variables.
List Data: Similar to plotting, this button is used for listing the results as functions of time.

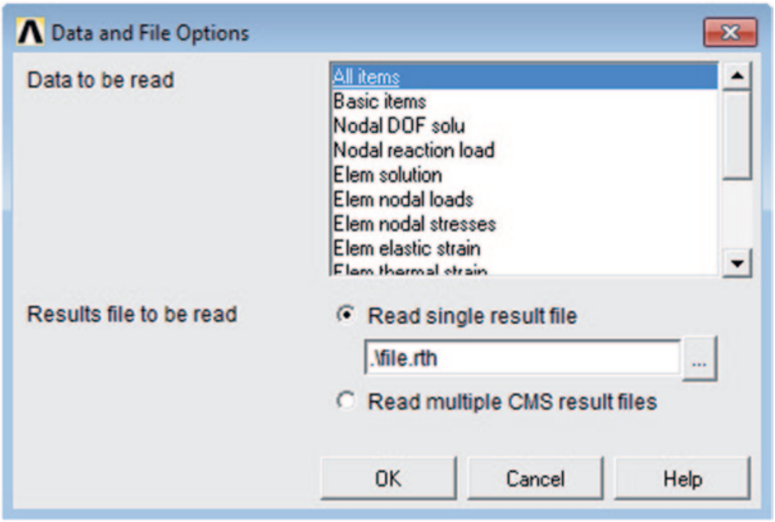


Fig. 5.9 *Data and file options* dialog box

As the new variables are defined, they appear in the ***Variables*** area. By default, ***TIME*** is the first variable and cannot be removed. In addition to the name of the variable, the ***Variables*** area includes useful information about the variable, such as its element or node number, what result item it corresponds to, and the range of its values.

The last item (located at the right-most side) is the ***X-Axis*** button, which enables the user to select which variable to display on the *x*-axis in the graphical representations.

An example problem (time-dependent heat transfer) demonstrating the use of the *Time History Postprocessor* is given in Sect. 5.4.

5.3.3 Read Results

The results, obtained through the *Solution Processor*, are saved in results files (***jobname.rst*** for structural, ***jobname.rth*** for thermal, and ***jobname.rfl*** for fluids problems), which are stored in the working directory. In order to review the results, the user needs to guide ANSYS so that the correct results file is selected. This is done by using the following menu path:

Main Menu > General Postproc > Data & File Opts

This brings up the *Data and File Options* dialog box (Fig. 5.9). The results file is selected by clicking on the browse button (button with three dots). After selecting the correct file, click on ***OK***.

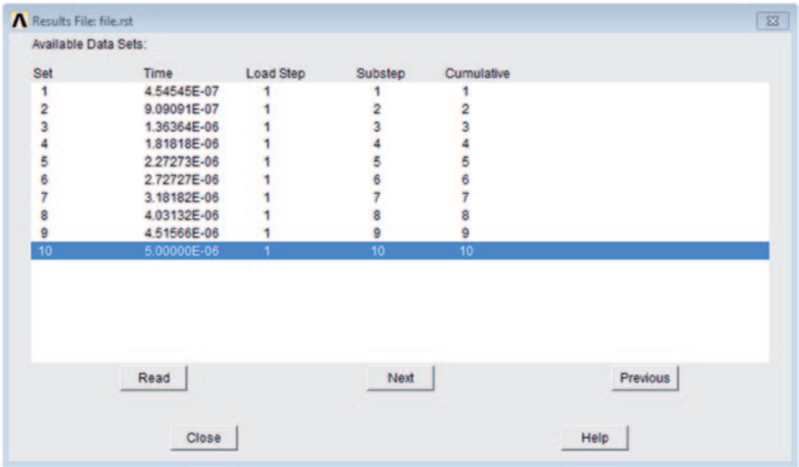


Fig. 5.10 *Results file* dialog box

If the solution does ***not*** involve multiple substeps and load steps, then there is only one results "set" the user can review. However, when the solution involves multiple substeps and load steps, there are many results sets and the user should select the correct (intended) one. The results sets can be selected using the following options:

First Set: Results related to the first available set are read into the database using the following menu path:

Main Menu > General Postproc > Read Results > First Set

Next Set: Results related to the set available immediately after the current set are read into the database using the following menu path:

Main Menu > General Postproc > Read Results > Next Set

Previous Set: Results related to the set available immediately before the current set are read into the database using the following menu path:

Main Menu > General Postproc > Read Results > Previous Set

Last Set: Results related to the last available set are read into the database using the following menu path:

Main Menu > General Postproc > Read Results > Last Set

Read By Picking: The following menu path is used for this option:

Main Menu > General Postproc > Read Results > By Pick

which brings up a dialog box (Fig. 5.10) listing the available results sets. The user selects the desired results set and clicks on ***Read*** and ***Close*** for the results to be read into the database.

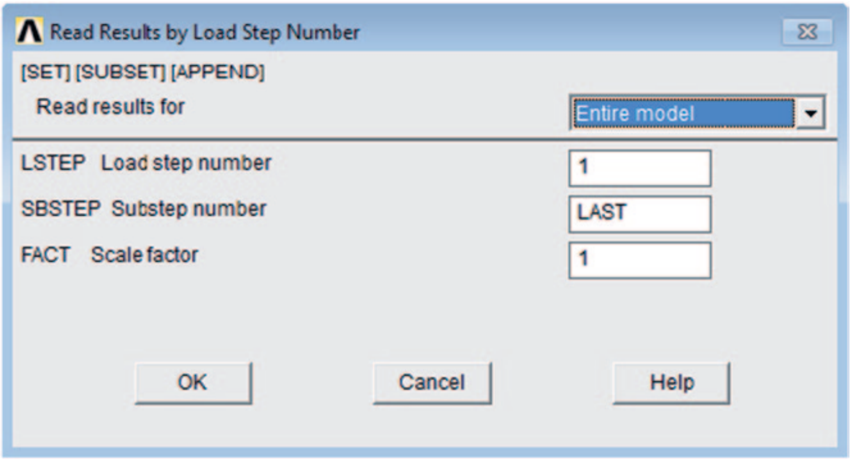


Fig. 5.11 *Read results by load step number* dialog box

Read By Load Step Number: Results related to a specific load step and substep are read into the database using the following menu path:

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

which brings up the *Read Results by Load Step Number* dialog box (Fig. 5.11) in which the user specifies the load step number (***LSTEP***) and substep (***SBSTEP***) number within that load step and clicks on ***OK*** for the results to be read into the database.

Read by Time: Results related to a specific time (or frequency) value are read into the database using the following menu path:

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

which brings up the *Read Results by Time or Frequency* dialog box (Fig. 5.12) in which the user specifies the value of time (or frequency) (***TIME***) and clicks on ***OK*** for the results to be read into the database.

5.3.4 Plot Results

After the desired results set is read into the database, the result quantities can be reviewed through graphics displays. The types of graphics displays include deformed shapes (structural analysis), contour plots, vector displays (thermal), and path plots.

In structural analyses, the deformed shape resulting from the applied loads and boundary conditions is displayed using the following menu path:

Main Menu > General Postproc > Plot Results > Deformed Shape

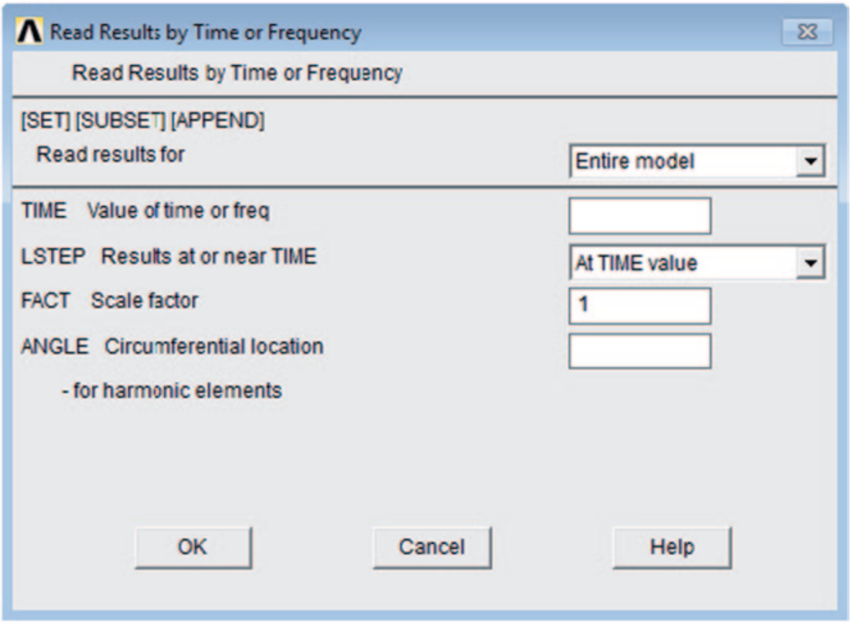


Fig. 5.12 *Read results by time or frequency* dialog box

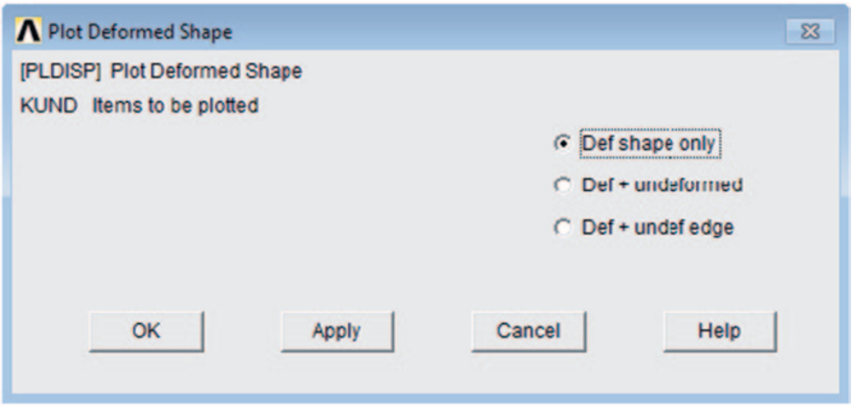


Fig. 5.13 *Plot deformed shape* dialog box

which brings up the *Plot Deformed Shape* dialog box (Fig. 5.13). The user is offered three distinct display modes:

- Display deformed shape only.
- Display deformed and undeformed shapes together.
- Display deformed shape with the outer boundary (edge) of the undeformed shape.

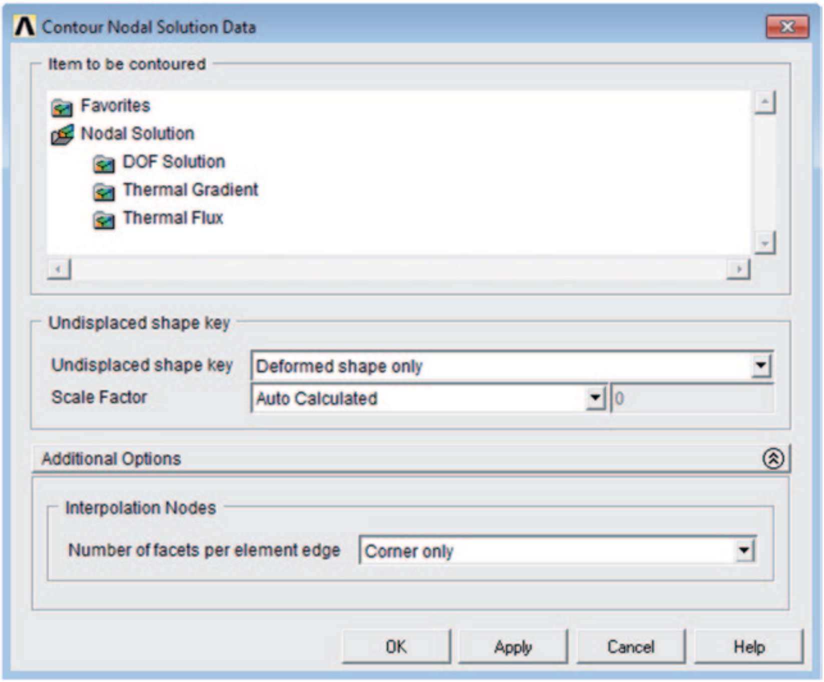


Fig. 5.14 *Contour nodal solution data* dialog box

After the user makes a choice and clicks on ***OK***, the deformed shape appears in the *Graphics Window*.

Contour plots are obtained using one of the following menu paths:

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

which brings up the *Contour Nodal* (*Element*) *Solution Data* dialog box (Fig. 5.14). In this dialog box, both the degree of freedom (DOF) solution (displacements, temperatures, etc.) and derived quantities (stresses, strains, fluxes, etc.) are available for plotting. Once the user makes the selection, upon clicking ***OK***, the contour plot appears in the *Graphics Window*.

Vector plots are obtained using the following menu path:

Main Menu > General Postproc > Plot Results > Vector Plot >Predefined

which brings up the *Vector Plot of Predefined Vectors* dialog box. Similar to the contour plots, this dialog box has two fields identifying the quantity to be plotted. Once the user makes a selection, upon clicking ***OK***, the vector plot appears in the *Graphics Window*.

In the ANSYS *General Postprocessor*, it is possible to obtain line plots along a path. Utilizing path plots involves:

Defining Paths: This can be performed by different methods, one of which uses the following menu path:

Main Menu > General Postproc > Path Operations > Define Path > By Nodes

which brings up a *Pick Menu* for the nodes defining the path to be picked. Upon clicking on ***OK*** in the *Pick Menu*, a dialog box appears asking for path specifications such as the user-defined name of the path and number of divisions between data points. Clicking ***OK*** in this dialog box finishes the path definition.

Mapping Quantities onto Paths: Once the paths are defined, quantities of interest are mapped onto paths by using the following menu path:

Main Menu > General Postproc > Path Operations > Map onto Path

which brings up the *Map Result Items onto Path* dialog box. The user specifies a unique label for the result item to be mapped and selects the result item; clicking ***OK*** completes the mapping operation.

Plotting Quantities on Graphs or on Geometry: The quantities mapped onto defined paths can be plotted using the following menu items:

Main Menu > General Postproc > Path Operations > Plot Path Item > On Graph
Main Menu > General Postproc > Path Operations > Plot Path Item > On Geometry

which brings up the *Plot of Path Items on Graph* (*Geometry*) dialog box. The user selects the path item to be plotted from the list of defined path items and clicks on ***OK***; the plot appears in the *Graphics Window*.

The operations related to path plots are demonstrated through an example problem in Sect. 5.4.

5.3.5 Element Tables

In ANSYS, each element type possesses numerous output quantities available upon completion of the solution. Although several of these quantities are offered by their names under the postprocessors, some are not directly accessible, and the user needs to take additional steps in order to access them. One important purpose of using *Element Tables* is to access these result items. Another important role of *Element Tables* is that they enable the user to perform arithmetic operations involving several result items. Element tables are defined by using the following menu path:

Main Menu > General Postproc > Element Table > Define Table

which brings up the *Element Table Data* dialog box (Fig. 5.15). In order to add new items to the element table, the user needs to click on the ***Add*** button, which brings up the *Define Additional Element Table Items* dialog box (Fig. 5.16). After selecting

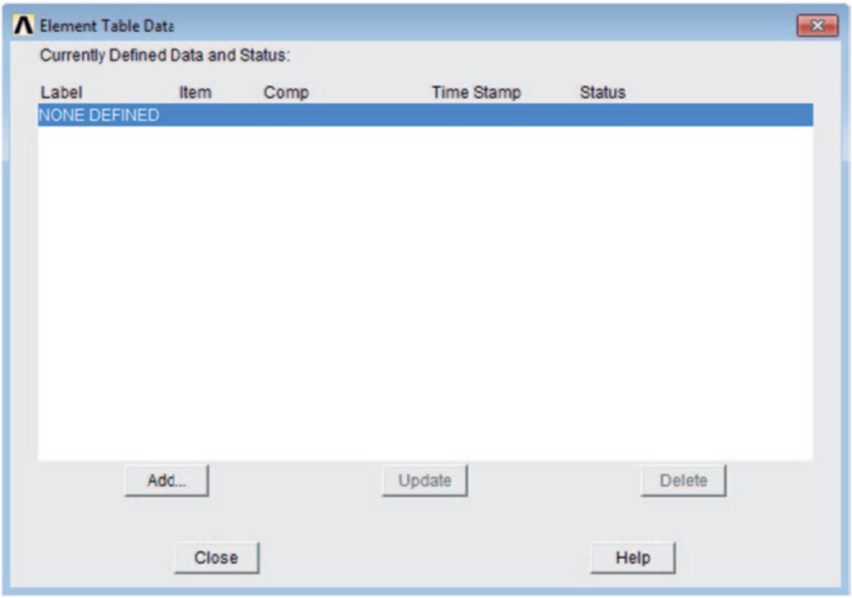


Fig. 5.15 *Element table data* dialog box

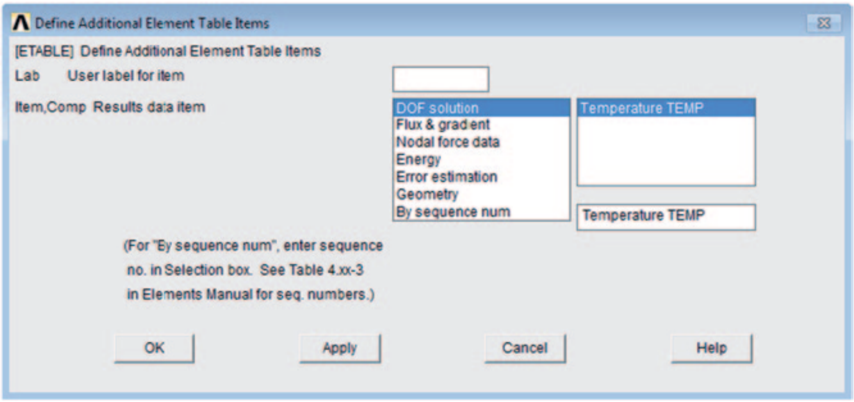


Fig. 5.16 *Define additional element table items* dialog box

the result quantity and specifying a user label for it, clicking on ***OK*** completes the element table item definition.

In order to explain the usage of element tables, an example based on the **PLANE55** element type is considered. In Table 5.1, the output quantities provided by the element type **PLANE55** are given. Table 5.2 lists the results quantities accessible through the element tables. For example, the heat flow rate per unit area across the element faces caused by input heat flux, denoted by ***HFLXAVG*** in Tables 5.1

Table 5.1 Output quantities provided by the **PLANE55** element type

Name	Definition
EL	Element number
NODES	Nodes: I, J, K, L
MAT	Material number
VOLU	Volume
XC, YC	Location where results are reported
HGEN	Heat generations HG(I), HG(J), HG(K), HG(L)
TG:X, Y, SUM	Thermal gradient components and vector sum at centroid
TF:X, Y, SUM	Thermal flux (heat flow rate/cross-sectional area) components and vector sum at centroid
FACE	Face label
AREA	Face area
NODES	Face nodes
HFILM	Film coefficient at each node of face
TBULK	Bulk temperature at each node of face
TAVG	Average face temperature
HEAT RATE	Heat flow rate across face by convection
HFAVG	Average film coefficient of the face
TBAVG	Average face bulk temperature
HFLXAVG	Heat flow rate per unit area across face caused by input heat flux
HEAT RATE/AREA	Heat flow rate per unit area across face by convection
HFLUX	Heat flux at each node of face

Table 5.2 Quantities obtained via the element table

Output quantity name	Element table input				
	Item	FC1	FC2	FC3	FC4
AREA	NMISC	1	7	13	19
HFAVG	NMISC	2	8	14	20
TAVG	NMISC	3	9	15	21
TBAVG	NMISC	4	10	16	22
HEAT RATE	NMISC	5	11	17	23
HFLXAVG	NMISC	6	12	18	24

and 5.2, is available for definition in the element tables. In Table 5.2, the matching item for this quantity is given as ***NMISC***, along with numbers 6, 12, 18, and 24 corresponding to different faces of the element. To store ***HFLXAVG*** at the 4th face of each element, in the *Define Additional Element Table Items* dialog box, ***By sequence num*** from the left list and ***NMISC*** from the right list must be selected. Entering 24 in the text field underneath the right list ensures that the ***HFLXAVG*** at the 4th face of each element will be stored in the element table.

Element tables are plotted and listed using the following menu paths:

Main Menu > **General Postproc** > **Element Table** > **Plot Elem Table**
Main Menu > **General Postproc** > **Element Table** > **List Elem Table**

It is also possible to perform arithmetic operations within each column or between the columns of the element table. Examples of such operations include: finding absolute values, finding the sum of each element table item, adding and multiplying element table items, etc.

5.3.6 List Results

Results of an ANSYS solution can be reviewed through lists. Although there are numerous different options for listing the results under postprocessors, only two of them are discussed in this section: nodal and element solutions. In order to list results computed at the nodes, the following menu path is used:

Main Menu > **General Postproc** > **List Results** > **Nodal Solu**

which brings up the *List Nodal Solution* dialog box. Once the user makes a selection as to what result quantities are to be reviewed and clicks on ***OK***, the list appearsw in a separate window. Similar to nodal solution listings, the element results are listed by using the following menu path:

Main Menu > **General Postproc** > **List Results** > **Element Solu**

The usage of this option is similar to the nodal solution lists.

5.4 Example: One-dimensional Transient Heat Transfer

Consider the one-dimensional transient heat transfer problem shown in Fig. 5.17. The problem is time dependent; therefore, in addition to thermal conductivity, the specific heat and the density of the material are taken into account. The governing equation for this problem is written as

$$\rho c \frac{\partial T}{\partial t} = \kappa \frac{\partial^2 T}{\partial x^2} \qquad 0 \le x \le l \tag{5.1}$$

with the boundary and initial conditions

$$\begin{aligned} T(x=0,t) &= T_a = 100 \\ T(x=l,t) &= T_b = 0 \\ T(x,t=0) &= f(x) = 0 \end{aligned} \tag{5.2}$$

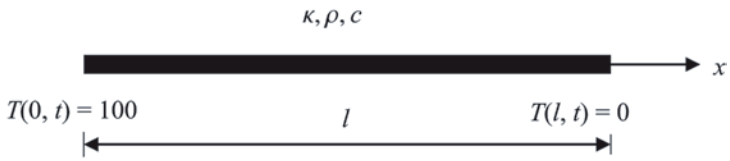


Fig. 5.17 One-dimensional transient heat transfer problem

The analytical solution for this problem is given by (Carslaw and Jaeger 1959, pp. 99–100):

$$T(x,t) = T_a + (T_b - T_a)\frac{x}{l} + \frac{2}{\pi}\sum_{n=1}^{\infty}\frac{T_b \cos n\pi - T_a}{n}\sin\frac{n\pi x}{l}e^{-\alpha n^2\pi^2 t/l^2}$$
$$+\frac{2}{l}\sum_{n=1}^{\infty}\sin\frac{n\pi x}{l}e^{-\alpha n^2\pi^2 t/l^2}\int_0^l f(x')\sin\frac{n\pi x'}{l}dx' \tag{5.3}$$

where $\alpha = \kappa/(\rho c)$. Substituting $T_a = 100$, $T_b = 0$, $f(x) = 0$, $l = 2$, $\kappa = 1$, $\rho = 10$, and $c = 3$, Eq. (5.3) yields

$$T(x,t) = 100 - 100\frac{x}{2} + \frac{2}{\pi}\sum_{n=1}^{\infty}\left(-\frac{100}{n}\right)\sin\frac{n\pi x}{2}e^{-n^2\pi^2 t/120} \tag{5.4}$$

When computing the exact solution using this equation, the number of terms, n, is truncated at 40 for satisfactory convergence.

Subjected to the boundary conditions indicated in Fig. 5.18 in the time range $0 \le t \le 5$, this problem is solved by using two-dimensional **`PLANE55`** elements in ANSYS.

The model is generated by using 4 element divisions along the vertical boundaries and 20 element divisions along the horizontal boundaries. The temperature variations along the midline at times $t = 0.1$, 0.5, and 5 are obtained by ANSYS and the exact solution is plotted.

Model Generation

- Specify the ***jobname*** as ***1d_dif*** using the following menu path:

Utility Menu > **File** > **Change Jobname**

  – In the dialog box, type ***1d_dif*** in the ***[/FILNAM] Enter new jobname*** text field; click on the check box for ***New log and error files*** to show ***Yes***; click on ***OK***.

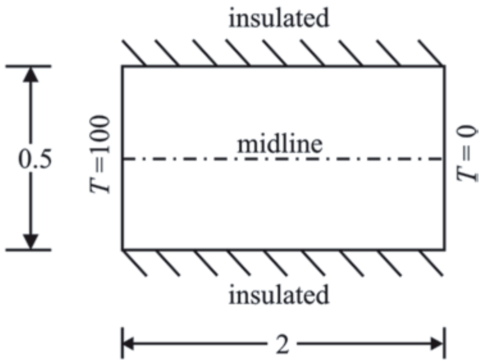


Fig. 5.18 Two-dimensional representation of the 1-D transient heat transfer problem and corresponding boundary conditions

- Define the element type (`ET` command) using the following menu path:

Main Menu > **Preprocessor** > **Element Type** > **Add/Edit/Delete**

  - Click on ***Add***.
  - Select ***Solid*** immediately below ***Thermal Mass*** on the left list and ***Quad 4node 55*** on the right list; click on ***OK***.
  - Click on ***Close***.

- Specify material properties (`MP` command) using the following menu path:

Main Menu > **Preprocessor** > **Material Props** > **Material Models**

  - In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on ***Thermal***, ***Conductivity***, and, finally, ***Isotropic***, which brings up another dialog box.
  - ***Enter 1 for KXX; click on OK.***
  - In the *Define Material Model Behavior* dialog box, in the right window, left-click on ***Specific Heat***, which brings up another dialog box.
  - Enter ***3*** for ***C***; click on ***OK***.
  - In the *Define Material Model Behavior* dialog box, in the right window, left-click on ***Density***, which brings up another dialog box.
  - Enter ***10*** for ***DENS***; click on ***OK***.
  - Close the *Define Material Model Behavior* dialog box by using the following menu path:

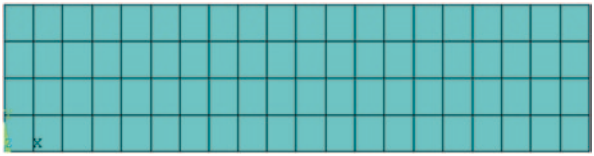
Material > Exit

- Create the solid model:

  - Create a rectangular area using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

Fig. 5.19 Mesh used in the analysis



- In the *Create Rectangle by Dimensions* dialog box, type ***0*** for ***X1***, ***2*** for ***X2***, ***0*** for ***Y1***, and ***0.5*** for ***Y2***; click on ***OK***.

• Create the mesh:

- Specify the number of elements along the vertical boundaries using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

- Pick the two vertical lines; click on OK.
- *Element Sizes on Lines* dialog box appears; type ***4*** in the text field corresponding to ***NDIV*** (the second text field), and uncheck the first check box; click on ***OK***.
- Specify the number of elements along the horizontal boundaries using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

- Pick the two horizontal lines; click on OK.
- *Element Sizes on Lines dialog box reappears; type 20 in the text field corresponding to NDIV (the second text field), and uncheck the first check box; click on OK.*
- Create the mesh using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- In the *Pick Menu*, click on ***Pick All***.
- Figure 5.19 shows the mesh.

• Save the model using the following menu path:

Utility Menu > File > Save as Jobname.db

The model is saved under the name ***1d_dif.db*** in the ***working directory***.

Solution

• Specify the analysis type as transient using the following menu path:

Main Menu > Solution > Analysis Type > New Analysis

- Click on ***Transient***; click on ***OK***.
- A new dialog box appears; click on ***OK***.

- Specify temperature boundary conditions along the vertical boundaries using the following menu path:

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

- Pick Menu appears; click on the Box radio-button and draw a rectangle around the nodes along the left vertical boundary; click on OK.
- *Apply TEMP on Nodes* dialog box appears; highlight ***TEMP***, enter ***100*** for ***VALUE Load TEMP value;*** click on ***Apply***.
- *Pick Menu* reappears; click on the ***Box*** radio-button and draw a rectangle around the nodes along the right vertical boundary; click on ***OK***.
- *Apply TEMP on Nodes* dialog box reappears; highlight ***TEMP***, enter ***0*** for ***VALUE Load TEMP value;*** click on ***OK***.

- Specify initial conditions within the domain using the following menu path:

Main Menu > Solution > Define Loads > Apply > Initial Condit'n > Define

- *Pick Menu* appears; click on ***Pick All***.
- *Define Initial Conditions* dialog box appears; select ***TEMP*** on the ***Lab*** pull-down menu; enter ***0*** in the ***VALUE*** text field; click on ***OK***.

- Specify time parameters using the following menu path:

Main Menu > Solution > Load Step Opts > Time/Frequenc > Time—Time Step

- Time and Time Step Options dialog box appears.
- As shown in Fig. 5.20, enter **5** in the **[TIME] Time at end of load step** text field and **5/100** in the **[DELTIM] Time step size** text field, and click on the **Stepped** radio-button for **[KBC]**; click on **OK**.

- Specify output controls using the following menu path:

Main Menu > Solution > Load Step Opts > Output Ctrls > DB/Results File

- Controls for Database and Results File Writing dialog box appears.
- As shown in Fig. 5.21, click on the **Every substep** radio-button for **FREQ File write frequency**; click on **OK**.

- Obtain solution using the following menu path:

Main Menu > **Solution** > **Solve** > **Current LS**

- Confirmation Window appears along with Status Report Window.
- Review status; if **OK**, close the Status Report Window and click on **OK** in the Confirmation Window.
- Wait until ANSYS responds with ***Solution is done!***

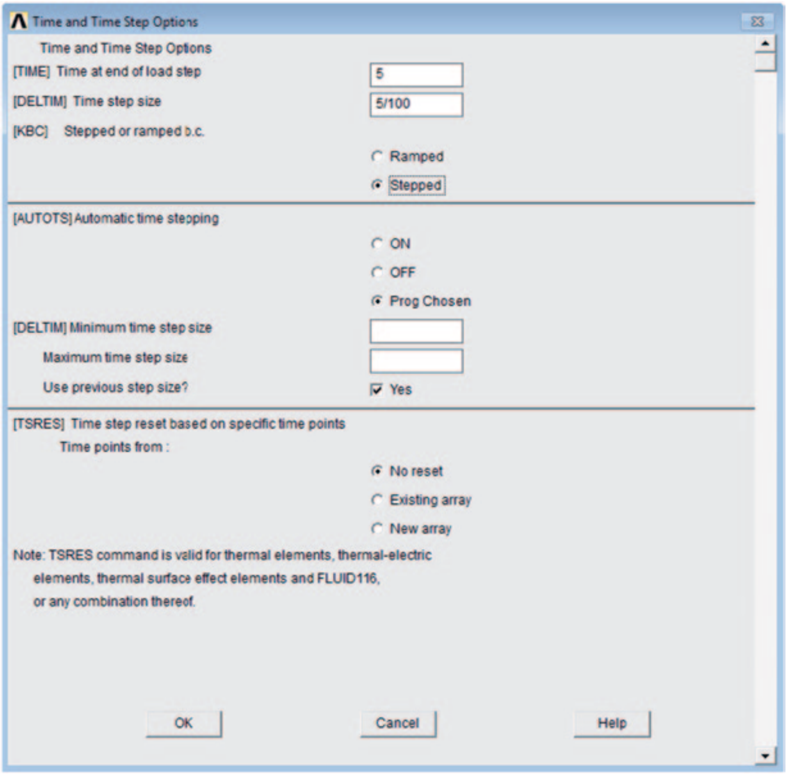


Fig. 5.20 *Time and time step options* dialog box used in the analysis

General Postprocessing

- Review results at the end of first substep using the following menu paths:

Main Menu > General Postproc > Read Results > First Set Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- Contour Nodal Solution Data dialog box appears; click on **DOF Solution** and **Nodal Temperature**; click on **OK**.
- Fig. 5.22 shows the contour plot of the temperature distribution at $t = 0.05$ as it appears in the Graphics Window.

- Review results at the "next" substep using the following menu paths:

Main Menu > General Postproc > Read Results > Next SetMain Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- Contour Nodal Solution Data dialog box appears; click on **DOF Solution** and **Nodal Temperature**; click on **OK**.
- Figure 5.23 shows the contour plot of temperature distribution at $t = 0.1$ as it appears in the Graphics Window.

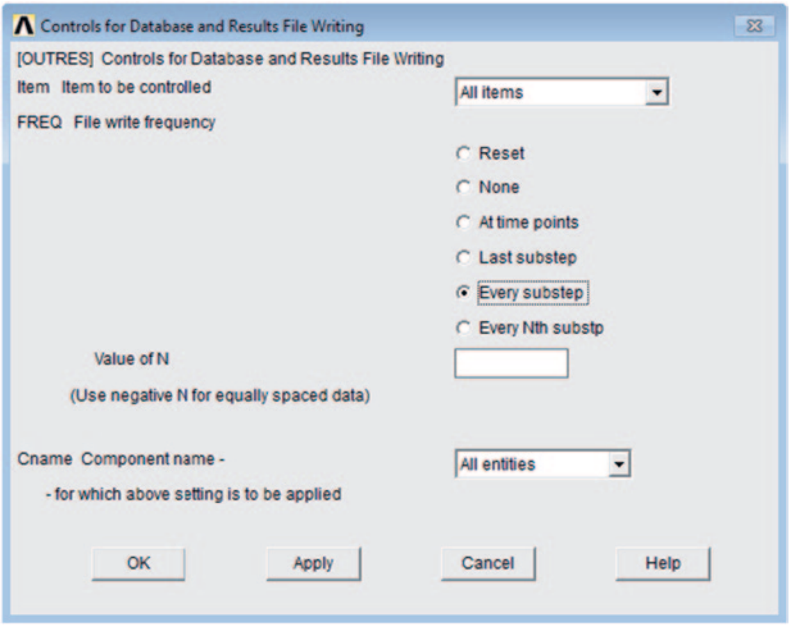


Fig. 5.21 *Output controls* dialog box used in the analysis

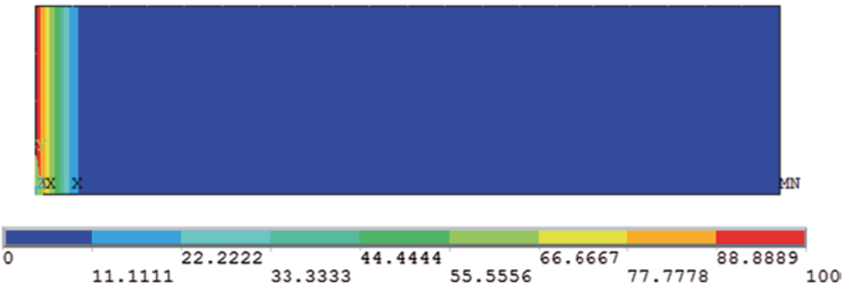


Fig. 5.22 Temperature distribution contour plot at $t = 0.05$

- Review results at $t = 0.5$ using the following menu path:

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

  – As shown in Fig. 5.24, Read Results by Time or Frequency dialog box appears; enter 0.5 for TIME Value of time or freq; click on OK.
  – View the temperature contours, as shown in Fig. 5.25.

- Review results at the "last" substep using the following menu paths:

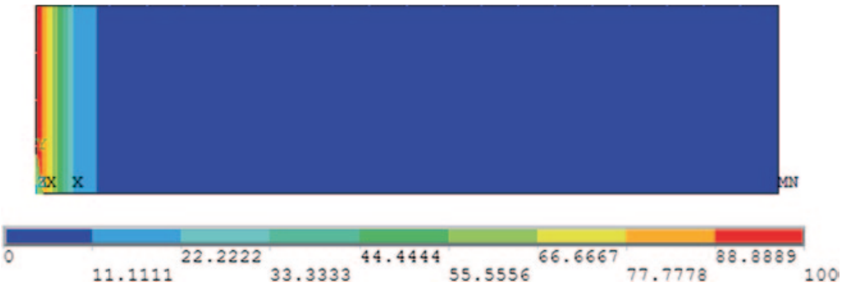


Fig. 5.23 Temperature distribution contour plot at $t=0.1$

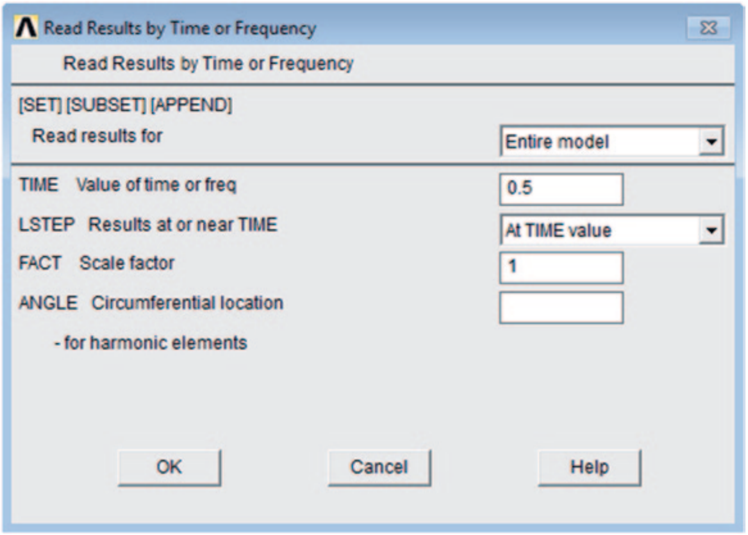


Fig. 5.24 *Read results by time or frequency* dialog box used in the analysis

Main Menu > General Postproc > Read Results > Last Set Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- Contour Nodal Solution Data dialog box appears; click on ***DOF Solution*** and ***Nodal Temperature***; click on **OK**.
- Figure 5.26 shows the contour plot of the temperature distribution at $t = 5$ as it appears in the Graphics Window.

- Review thermal flux vector plot at $t = 5$ using the following menu path:

Main Menu > General Postproc > Plot Results > Vector Plot > Predefined

- *Vector Plot of Predefined Vectors* dialog box appears; select ***Thermal flux TF*** on the right list and click on ***OK***.

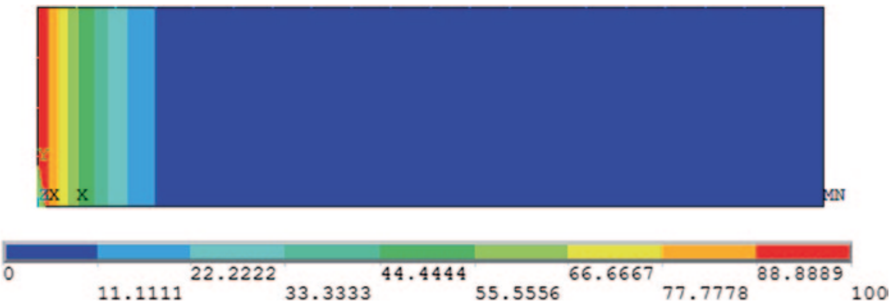


Fig. 5.25 Temperature distribution contour plot at $t=0.5$

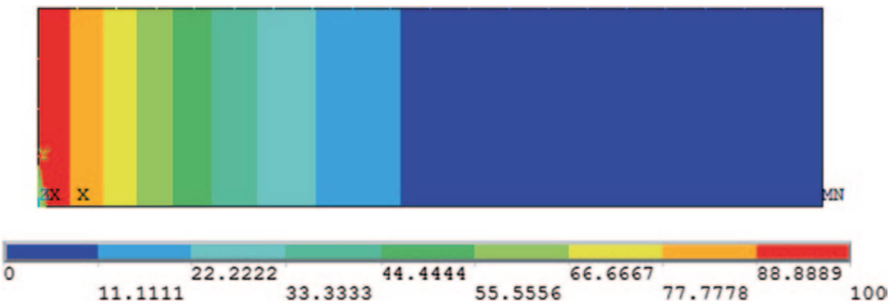


Fig. 5.26 Temperature distribution contour plot at $t=5$

  - Figure 5.27 shows the vector plot of the thermal flux as it appears in the *Graphics Window*.

- Review results by path plots:
  - Plot elements using the following menu path:

Utility Menu > Plot > Elements

  - Turn node numbering on using the following menu path:

Utility Menu > PlotCtrls > Numbering

  - *Plot Numbering Controls* dialog box appears; click on the check box for ***NODE Node numbers*** to show ***On***; click on ***OK***.
  - Define the path using the following menu path:

Main Menu > General Postproc > Path Operations > Define Path > By Nodes

  - ***Pick Menu appears; pick nodes 47 (x=0, y=0.25) and 24 (x=2, y=0.25); click on OK.***
  - *Define Path By Nodes* dialog box appears, as shown in Fig. 5.28; enter a unique name (e.g., ***y025***) identifying the path; click on ***OK***.

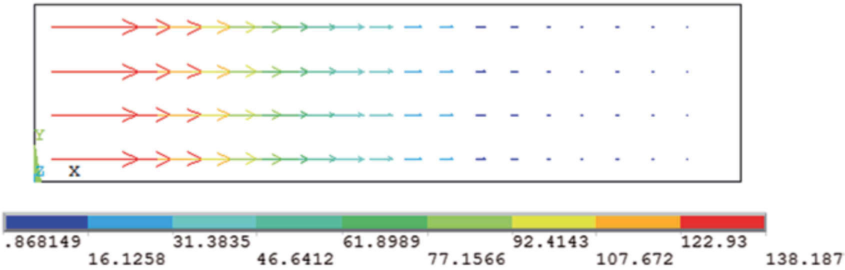


Fig. 5.27 Vector plot of thermal flux at $t = 5$

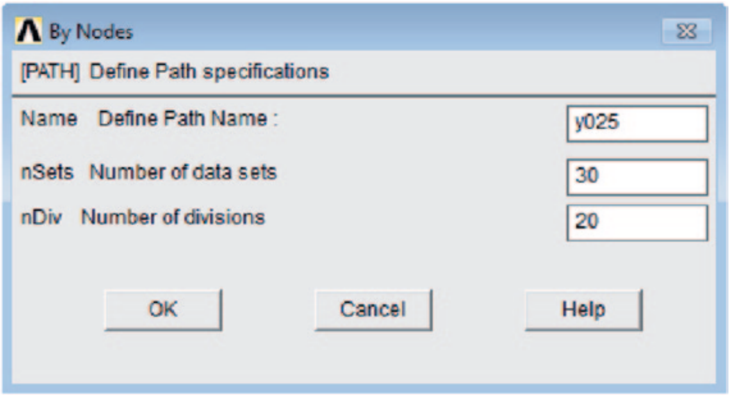


Fig. 5.28 Defining path specifications

- Close the *Path Status Information Window.*
- Turn node numbering off using the following menu path:

Utility Menu > PlotCtrls > Numbering

- In the *Plot Numbering Controls* dialog box, click on the check box for ***NODE Node numbers*** to show ***Off***; click on ***OK***.
- Plot the path on geometry using the following menu path:

Main Menu > General Postproc > Path Operations > Plot Paths

- Figure 5.29 shows the result of this action.

• Map the temperature results onto the defined path using the following menu path:

Main Menu > General Postproc > Path Operations > Map onto Path

- *Map Result Items onto Path* dialog box appears, as shown in Fig. 5.30; enter a unique name for the result item (e.g., ***t025***, note that this is different than the

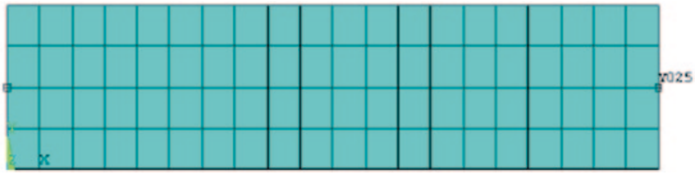


Fig. 5.29 Geometry plot of path `Y025`

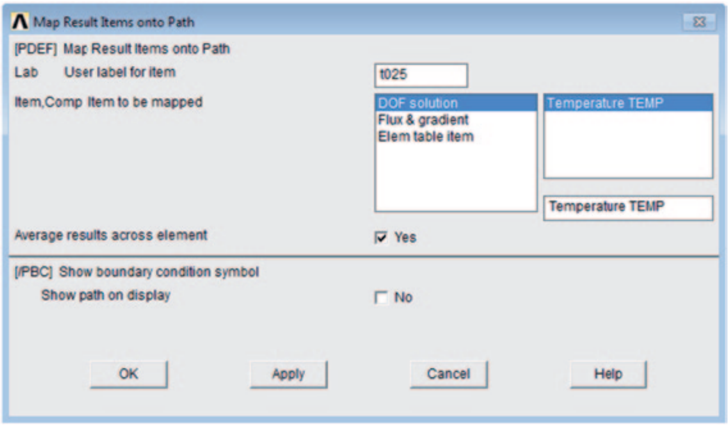


Fig. 5.30 *Map result items onto path* dialog box used for line plot of temperature along the path `Y025`

name given for the path); select ***DOF solution*** from the left list and ***Temperature TEMP*** from the right list for the item; click on ***OK***.

– Path plot the temperature results using the following menu path:

Main Menu > General Postproc > Path Operations > Plot Path Item > On Graph

– *Plot of Path Items on Graph* dialog box appears; select ***T025*** from the list; click on ***OK***.
– Observe the temperature variation along the path ***y025*** as it appears in the *Graphics Window*, as shown in Fig. 5.31.
– Map the flux results onto the defined path using the following menu path:

Main Menu > General Postproc > Path Operations > Map onto Path

– *Map Result Items onto Path* dialog box appears, as shown in Fig. 5.32; enter a unique name for the result item (e.g., ***q025***); select ***Flux & gradient*** from the left list and ***Thermal flux TFX*** from the right list for the item; click on ***OK***.
– Path plot the flux results using the following menu path:

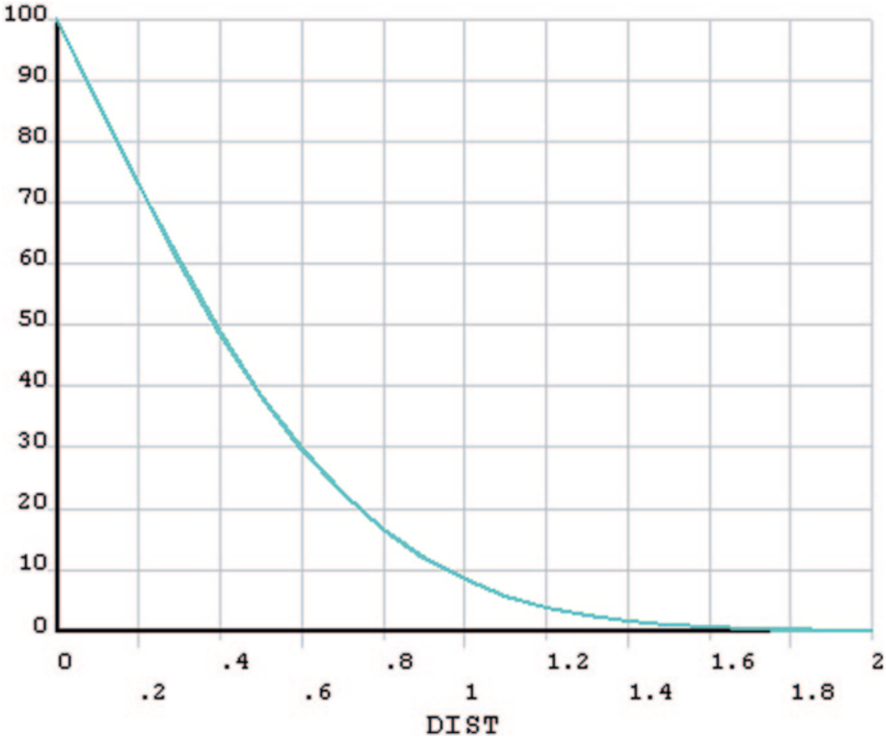


Fig. 5.31 Temperature variation line plot along the path **Y025** at $t=5$

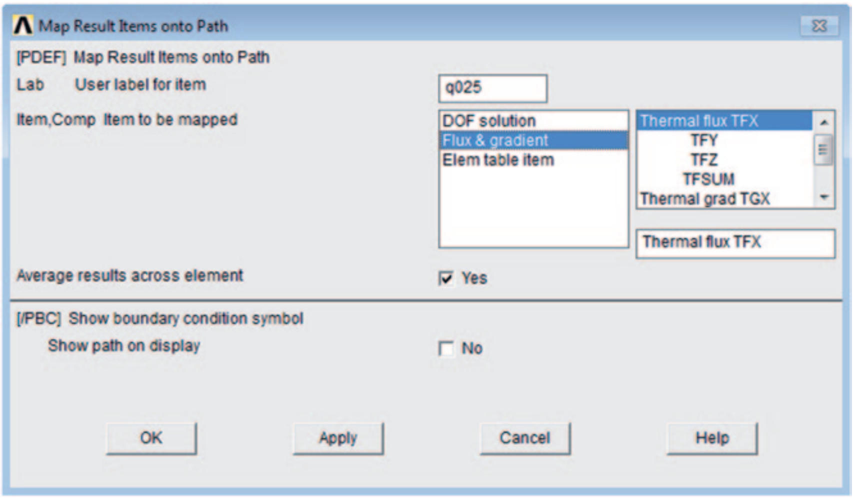


Fig. 5.32 *Map result items onto path* dialog box used for line plot of flux along the path **Y025**

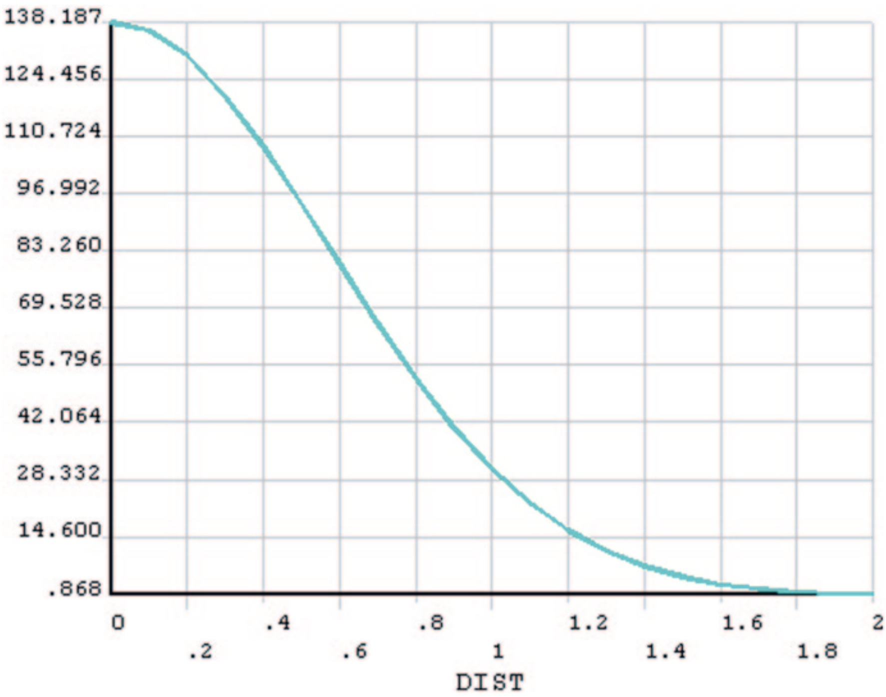


Fig. 5.33 Flux variation line plot along the path **`Y025`** at $t=5$

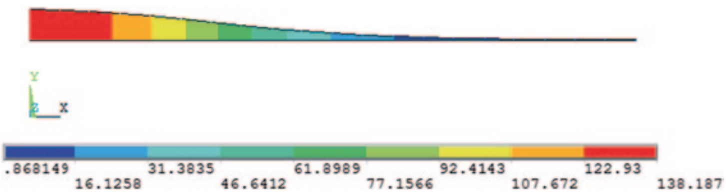


Fig. 5.34 Path plot on geometry for the variation of flux

Main Menu > General Postproc > Path Operations > Plot Path Item > On Graph

- *Plot of Path Items on Graph* dialog box appears; unselect ***T025*** and select ***Q025*** from the list; click on ***OK***.
- Observe the flux variation along the path ***y025*** as it appears in the *Graphics Window*, as shown in Fig. 5.33.
- Finally, plot the flux on actual geometry using the following menu path:

Main Menu > General Postproc > Path Operations > Plot Path Item > On Geometry

- *Plot of Path Items on Geometry* dialog box appears; select ***Q025*** from the list; click on ***OK***.
- Fig. 5.34 shows the path plot of thermal flux as it appears in the *Graphics Window*.

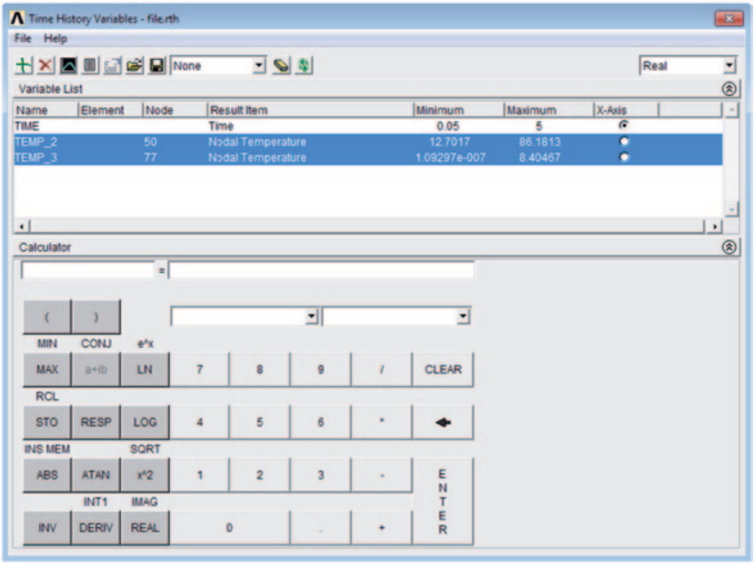


Fig. 5.35 *Time history variables* dialog box

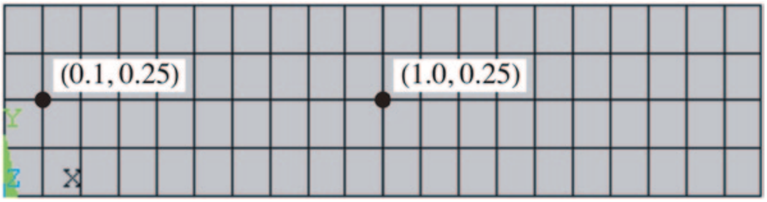


Fig. 5.36 Nodes to be picked to review the time-dependent behavior of temperature and flux

Time History Postprocessing

- Review time-dependent behavior of temperature at nodes located at $(0.1, 0.25)$ and $(1.0, 0.25)$ using the following menu path:

Main Menu > TimeHist Postpro

- *Time History Variables* dialog box appears (Fig. 5.35).
- Click on the button with the green plus sign at the top-left to define a variable.
- Add Time History Variable dialog box appears.
- Successively click on the items Nodal Solution, DOF Solution, and Nodal Temperature; click on ***OK***.
- Pick Menu appears; pick the node located at $x = 0.1, y = 0.25$ (as indicated in Fig. 5.36); click on ***OK***.
- Note the new variable ***TEMP_2*** in the Time History Variables dialog box.

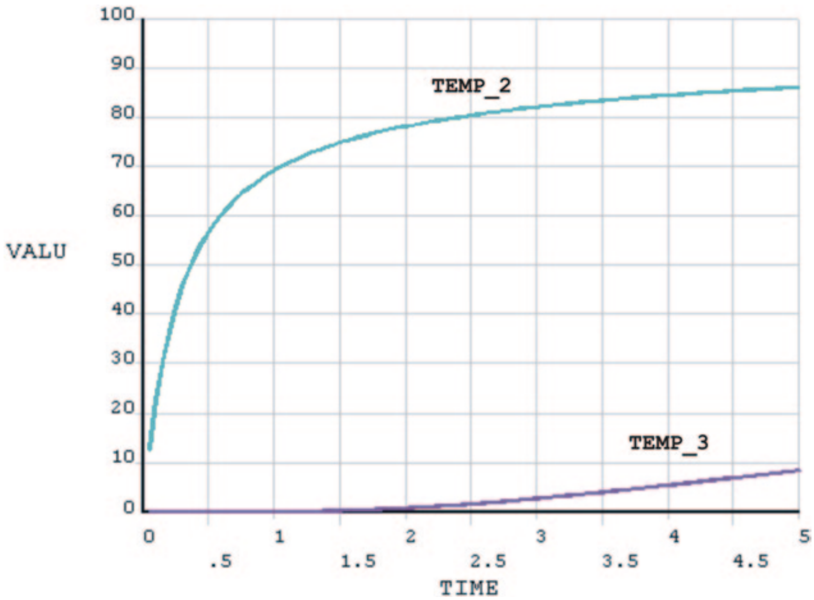


Fig. 5.37 Temperature variation over time at two nodes

- Add a new variable for temperature at the center node by clicking on the button with the green plus sign and successively clicking on the items ***Nodal Solution***, ***DOF Solution***, and ***Nodal Temperature***; click on ***OK***.
- Pick the node located at $x = 1, y = 0.25$ (as indicated in Fig. 5.36); click on ***OK***.
- Note the new variable ***TEMP_3*** in the *Time History Variables* dialog box.
- Highlight the rows ***TEMP_2*** and ***TEMP_3*** from the list (by pressing ***Ctrl*** on the keyboard and clicking on the rows with the left mouse button); click on the third from the left button to plot the time variation of these temperatures.
- The plot appears in the *Graphics Window*, as shown in Fig. 5.37.

- Review time-dependent behavior of thermal flux at nodes located at $(0.1, 0.25)$ and $(1.0, 0.25)$.
  - In the *Time History Variables* dialog box, highlight the rows ***TEMP_2*** and ***TEMP_3*** from the list and click on the second from the left button (button with a red cross) to delete the temperature variables.
  - In order to add thermal flux variables, click on the button with the green plus sign at the top-left to define a variable.
  - *Add Time History Variable* dialog box appears.
  - Successively click on the items ***Nodal Solution***, ***Thermal Flux***, and ***X-Component of thermal flux***; click on ***OK***.
  - The *Pick Menu* appears; pick the node located at ($x = 0.1, y = 0.25$) (as indicated in Fig. 5.36); and click on ***OK***.
  - Note the new variable ***TFX_2*** in the *Time History Variables* dialog box.

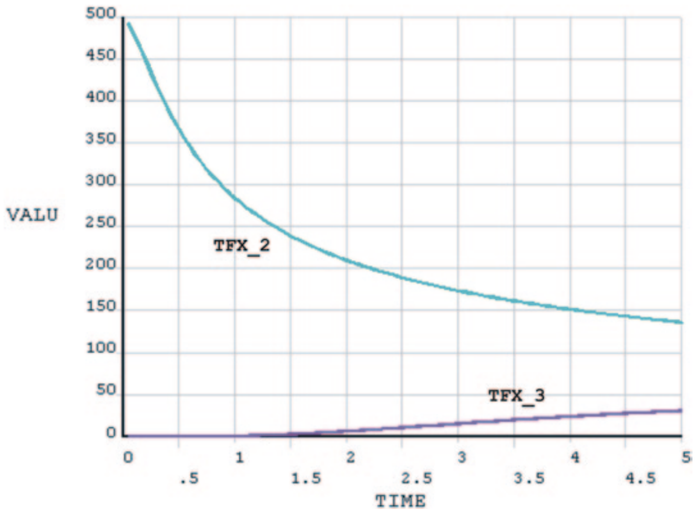


Fig. 5.38 Flux variation over time at two nodes

- Add a new variable for thermal flux at the center node by clicking on the button with the green plus sign and successively clicking on the items ***Nodal Solution***, ***Thermal Flux***, and ***X-Component of thermal flux***; click on ***OK***.
- Pick the node located at ($x = 1, y = 0.25$) (as indicated in Fig. 5.36); click on ***OK***.
- Note the new variable ***TFX_3*** in the *Time History Variables* dialog box.
- Highlight the rows ***TFX_2*** and ***TFX_3*** from the list (by pressing ***Ctrl*** on the keyboard and clicking on the rows with the left mouse button); click on the third from the left button to plot the time variation of these thermal fluxes.
- The plot appears in the *Graphics Window*, as shown in Fig. 5.38.
- Close *Time History Variables* dialog box.

Table 5.3 lists the temperature values along the midline ($y = 0.25$) obtained by ANSYS (columns 2–4) and the analytical solution given by Eq. (5.4) (columns 5–7) at times $t = 0.1$, 0.5, and 5. The analytical solution is obtained by using $n = 40$ in the series. Three separate ANSYS solutions are obtained with the final times $t = 0.1$, 0.5, and 5, each utilizing 100 equal time steps. Figure 5.39 shows a graphical comparison of the analytical and ANSYS solutions.

Table 5.3 Temperature values along the midline (y=0.25) obtained by ANSYS and Eq. (5.4) (t=0.1, 0.5, 5.0)

x	ANSYS			EXACT (n=40)		
	t=0.1	t=0.5	t=5.0	t=0.1	t=0.5	t=5.0
0.00	100.0000	100.0000	100.0000	100.0000	100.0000	100.0000
0.10	24.2230	56.7890	88.1810	22.0671	58.3882	86.2490
0.20	3.7816	26.6910	72.7800	1.4306	27.3322	72.9034
0.30	0.4497	10.7250	60.1760	0.0239	10.0348	60.3332
0.40	0.0443	3.7954	48.6770	9.84E–05	2.8460	48.8422
0.50	0.0038	1.2112	38.5010	2.09E–06	0.6170	38.6475
0.60	0.0003	0.3550	29.7610	1.88E–06	0.1015	29.8698
0.70	2.07E–05	0.0969	22.4760	1.73E–06	0.0126	22.5346
0.80	1.36E–06	0.0249	16.5800	1.56E–06	0.0012	16.5857
0.90	8.47E–08	0.0061	11.9450	1.40E–05	8.24E–05	11.9033
1.00	5.00E–09	0.0014	8.4047	1.24E–06	4.32E–06	8.3264
1.10	2.83E–10	0.0003	5.7753	1.09E–06	1.69E–07	5.6746
1.20	1.54E–11	6.94E–05	3.8761	9.43E–07	4.94E–09	3.7666
1.30	8.14E–13	1.46E–05	2.5411	8.08E–07	1.08E–10	2.4340
1.40	4.17E–14	3.00E–06	1.6271	6.80E–07	1.74E–12	1.5307
1.50	2.08E–15	6.00E–07	1.0169	5.58E–07	0.00E+00	0.9360
1.60	1.03E–16	1.17E–07	0.6187	4.41E–07	0.00E+00	0.5551
1.70	4.85E–18	2.25E–08	0.3628	3.27E–07	0.00E+00	0.3167
1.80	2.27E–19	4.24E–09	0.1981	2.17E–07	0.00E+00	0.1684
1.90	1.04E–20	7.61E–10	0.0868	1.08E–07	−1.69E–14	0.0723
2.00	0	0	0	−8.39E–15	−4.03E–15	−3.85E–15

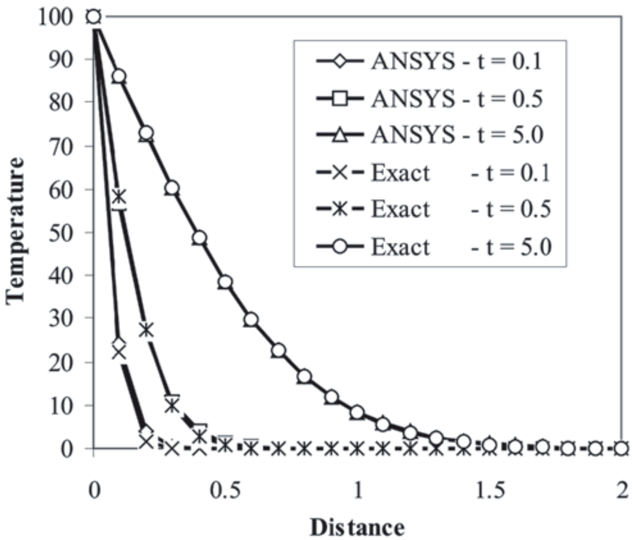


Fig. 5.39 Graphical comparison of ANSYS and analytical solutions