# INTRODUCTION TO THE SYSTEM

## 1. Introduction

The ANSYS computer program is a large-scale multi-purpose finite element program, which may be used for solving several classes of engineering analyses. The analysis capabilities of ANSYS include the ability to solve static and dynamic structural analyses, steady-state and transient heat transfer problems, mode-frequency and buckling eigenvalue problems, static or time-varying magnetic analyses, and various types of field and coupled-field applications. The program contains many special features which allow nonlinearities or secondary effects to be included in the solution, such as plasticity, large strain, hyperelasticity, creep, swelling, large deflections, contact, stress stiffening, temperature dependency, material anisotropy, and radiation. As ANSYS has been developed, other special capabilities, such as substructuring, submodeling, random vibration, kinetostatics, kinetodynamics, free convection fluid analysis, acoustics, magnetics, piezoelectrics, coupled-field analysis and design optimization have been added to the program. These capabilities contribute further to making ANSYS a multi-purpose analysis tool for varied engineering disciplines.

The ANSYS program has been in commercial use since 1970, and has been used extensively in the aerospace, automotive, construction, electronic, energy services, manufacturing, nuclear, plastics, oil, and steel industries. In addition, many consulting firms and hundreds of universities use ANSYS for analysis, research, and educational use. ANSYS is recognized worldwide as one of the most widely used and capable programs of its type.

## 2. What ANSYS Is

ANSYS finite element analysis software enables engineers to perform the following tasks:
- Build computer models or transfer CAD models of structures, products, components, or systems.
- Apply operating loads or other design performance conditions.
- Study physical responses, such as stress levels, temperature distributions, or electromagnetic fields.
- Optimize a design early in the development process to reduce production costs.
- Do prototype testing in environments where it otherwise would be undesirable or impossible (for example, biomedical applications).

The ANSYS program has a comprehensive graphical user interface (GUI) that gives users easy, interactive access to program functions, commands, documentation, and reference material. An intuitive menu system helps users navigate through the ANSYS program. Users can input data using a mouse, a keyboard, or a combination of both.

## 2.1. The ANSYS Environment

**Organization of the ANSYS Program**

The ANSYS program is organized into two basic levels:

- Begin level
- Processor (or Routine) level

The Begin level acts as a gateway into and out of the ANSYS program. It is also used for certain global program controls such as changing the jobname, clearing (zeroing out) the database, and copying binary files. When you first enter the program, you are at the Begin level.

At the Processor level, several processors are available. Each processor is a set of functions that perform a specific analysis task:

- For example, the general preprocessor (PREP7) is where you build the model,
- the solution processor (SOLUTION) is where you apply loads and obtain the solution,
- and the general postprocessor (POST1) is where you evaluate the results of a solution.
- An additional postprocessor, POST26 , enables you to evaluate solution results at specific points in the model as a function of time.

**Entering a Processor**

In general, you enter a processor by selecting it from the ANSYS Main Menu in the Graphical User Interface (GUI). For example, choosing *Main Menu>Preprocessor* takes you into PREP7 . Alternatively, you can use a command to enter a processor (the format is */name*, where name is the name of the processor).

**Table 1. Processors (Routines) Available in ANSYS**

| Processor | Function | GUI Path | Command |
|---|---|---|---|
| PREP7 | Build the model (geometry, materials, etc.) | Main Menu>Preprocessor | /PREP7 |
| SOLUTION | Apply loads and obtain the finite element solution | Main Menu>Solution | /SOLU |
| POST1 | Review results over the entire model at specific time points | Main Menu>General Postproc | /POST1 |
| POST26 | Review results at specific points in the model as a function of time | Main Menu>TimeHist Postpro | /POST26 |
| OPT | Improve an initial design | Main Menu>Design Opt | /OPT |
| AUX2 | Dump binary files in readable form | Utility Menu>File> List>Binary FilesUtility Menu>List>Files> Binary Files | /AUX2 |
| AUX12 | Calculate radiation view factors and generate a radiation matrix for a thermal analysis | Main Menu>Radiation Matrix | /AUX12 |
| AUX15 | Translate files from a CAD or FEA program | Utility Menu>File>Import | /AUX15 |
| RUNSTAT | Predict CPU time, wavefront requirements, etc. for an analysis | Main Menu>Run-Time Stats | /RUNST |

**Exiting from a Processor or ANSYS**

To return to the Begin level from a processor, pick *Main Menu>Finish* or issue the FINISH (or /QUIT) command. You can move from one processor to another without returning to the Begin level. Simply pick the processor you want to enter, or issue the appropriate command.

To leave the ANSYS program (and return to the system level), pick *Utility Menu>File>Exit*, to display the Exit from ANSYS dialog box, or use the /EXIT command. By default, the program saves the model and loads portions of the database automatically and writes them to the database file, Jobname.DB. If a backup of the current database file already exists, ANSYS writes it to Jobname.DBB. Options in the dialog box (and on the /EXIT command) allow you to save other portions of the database or to quit without saving.

**The ANSYS Database**

In one large database, the ANSYS program stores all input data (model dimensions, material properties, load data, etc.) and results data (displacements, stresses, temperatures, etc.) in an organized fashion. The main advantage of the database is that you can list, display, modify, or delete any specific data item quickly and easily.

No matter which processor you are in, you are working with the same database. This gives you basic access to the model and loads portions of the database from anywhere in the program. "Basic access" means the ability to select, list, or display an item.

**Saving the Database**

Because the database contains all your input data, you should frequently save copies of it to a file. To do this, pick *Utility Menu>File>Save* as Jobname.DB or issue the SAVE command. Either choice writes the database to the file Jobname.DB. If you use the SAVE command, you have the option to save:

- the model data only
- the model and solution data
- the model, solution and preprocessing data

To specify a different file name, pick *Utility Menu>File>Save as* or use the appropriate fields on the SAVE command. Any save operation first writes a backup of the current database file (if the database already exists) to Jobname.DBB. If a Jobname.DBB file already exists, the new backup file overwrites it. For a static or transient structural analysis, the file Jobname.RDB (a copy of the database) will be automatically saved at the first substep of the first load step.

**Restoring Database Contents**

To restore data from the database file, pick *Utility Menu>File>Resume* Jobname.DB or issue the RESUME command. This reads the file Jobname.DB. To specify a different file name, pick *Utility Menu>File>Resume from* or use the appropriate fields on the RESUME command.

**Clearing the Database**

While building a model, sometimes you may want to clear out the database contents and start over. To do so, choose *Utility Menu>File>Clear & Start New* or issue the /CLEAR command. Either method clears (zeros out) the database stored in memory. Clearing the database has the same effect as leaving and re-entering the ANSYS program, but does not require you to exit.

**ANSYS Program Files**

The ANSYS program writes and reads many files for data storage and retrieval. File names follow this pattern:   *Name.Ext*

Name defaults to the jobname, which you can specify while entering the ANSYS program or by choosing *Utility Menu>File>Change Jobname* (equivalent to issuing the /FILNAME command). The default jobname is FILE (or file).

Ext is a unique, two- to four-character ANSYS identifier that identifies the contents of the file. For example, Jobname.DB is the database file, Jobname.EMAT is the element matrix file, and Jobname.GRPH is the neutral graphics file. Some systems (such as PCs) truncate the extension to three characters. Also, the extension may be in lowercase, depending on the system.

**Table 2. ANSYS File types and Formats**

| File Type | File Name | File Format |
|---|---|---|
| Log file | Jobname.LOG | ASCII |
| Error file | Jobname.ERR | ASCII |
| Output file | Jobname.OUT | ASCII |
| Database file | Jobname.DB | Binary |
| Results file: | Jobname.xxx | |
| structural or coupled | Jobname.RST | |
| thermal | Jobname.RTH | Binary |
| magnetic | Jobname.RMG | |
| FLOTRAN | Jobname.RFL | |
| Load step file | Jobname.Sn | ASCII |
| Graphics file | Jobname.GRPH | ASCII (special format) |
| Element matrices | Jobname.EMAT | Binary |

**The Jobname.LOG File**

The Jobname.LOG file (also called the session log) is especially important, because it provides a complete log of your ANSYS session. The file opens immediately when you enter the ANSYS program, and it records all commands you execute, whether you execute those commands via GUI paths or type them in directly. You can read the Jobname.LOG file, view it while in ANSYS, edit it, and input it later.

The ANSYS program always appends log data to the log file instead of overwriting it. If you change the jobname while in an ANSYS session, the log file name does not change to the new jobname. For more information about Jobname.LOG, see Using the ANSYS Command Log.

## 2.2. Entering ANSYS on Windows Systems

Most ANSYS users who run the product on a Windows system access the product, related utilities, and the ANSYS help system via the
*Start>Programs>ANSYS x.x* menu on Windows (*ANSYSx.x means the actually installed version of the program e.g. ANSYS 12*).
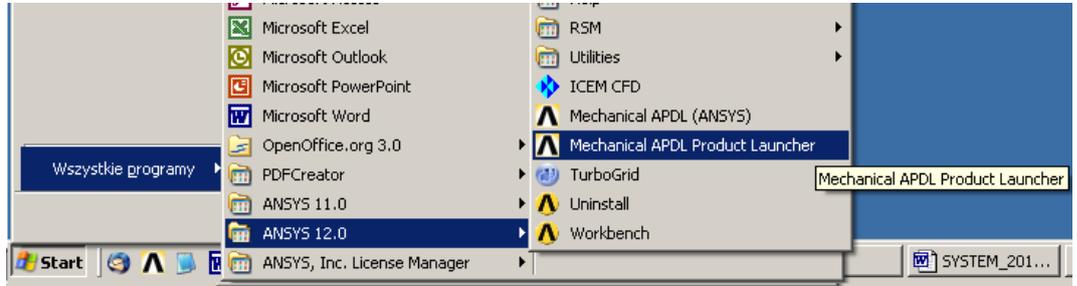
Here is a list briefly describing the function of each item in the ANSYS folder:

**Interactive -** displays a dialog box in which you define settings for an interactive session. When you select Interactive icon, the following dialog box appears:
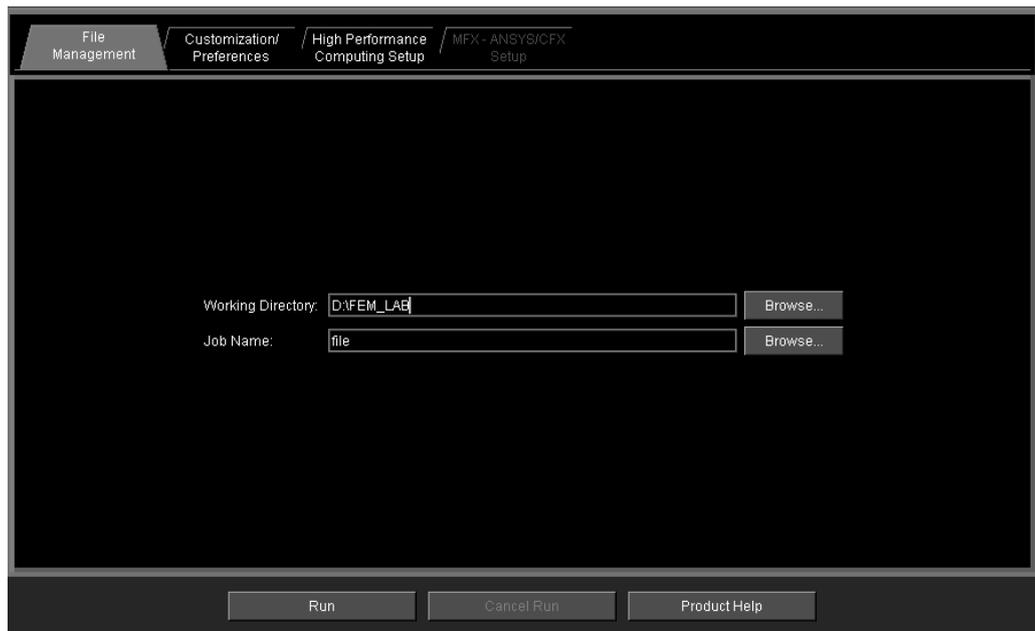
**ANS_ADMIN Utility - s**ets information for running ANSYS
**Batch**                              - launches the ANSYS program in batch mode
**CMAP Utility**              - **s**tarts the CMAP program
**DISPLAY Utility**          - **s**tarts the DISPLAY program.
**Help System**               - displays the ANSYS online help and manuals.
                                  Using the Online Help and Manuals discusses the online documentation
                                  and how to use it.
**Readme**                       - displays a file containing information about the current release of ANSYS.
**Run Interactive Now**   - launches an interactive ANSYS session immediately.
**Site Information**          - displays an information file maintained by your system administrator.

## 2.3. Using the ANSYS GUI

**Layout of the GUI**

For all ANSYS products except ANSYS/Professional, by default the ANSYS GUI has six main windows
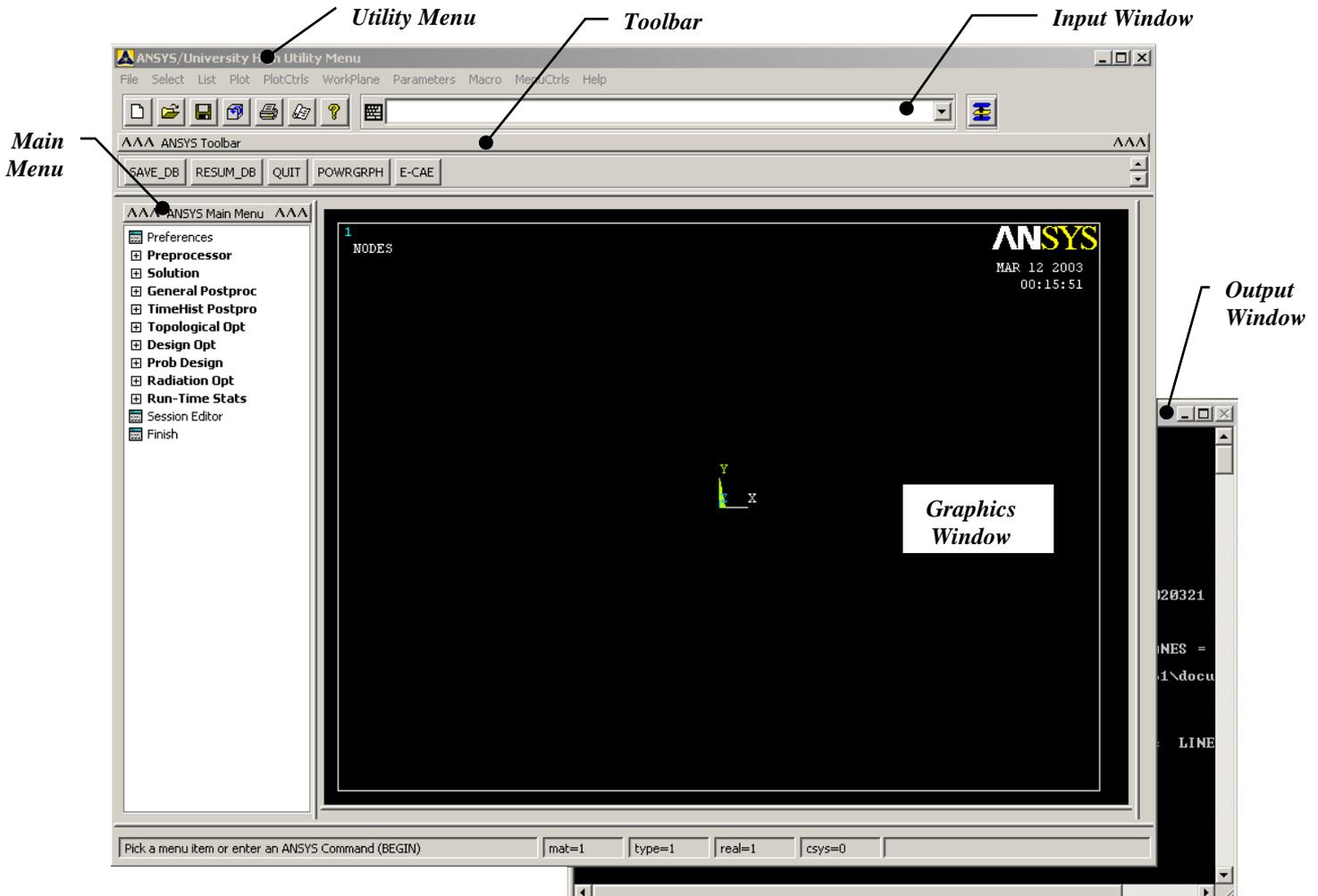


| | |
|---|---|
| *Utility Menu* | Contains utility functions that are available throughout the ANSYS session, such as file controls, selecting, graphics controls, and parameters. These functions are executed through smooth, cascading pull-down menus that lead directly to an action or dialog box. You also exit the ANSYS program through this menu |
| *Main Menu* | Contains the primary ANSYS functions, organized by processors (preprocessor, solution, general postprocessor, design optimizer, etc.). |
| *Toolbar* - | Contains push buttons that execute commonly used ANSYS commands and functions. You may add your own push buttons by defining abbreviations. |
| *Input Window* | Shows program prompt messages and allows you to type in commands directly. All previously typed in commands also appear for easy reference and access. |
| *Graphics Window* | A window where graphics displays are drawn. |
| *Output Window* | Receives text output from the program. It is usually positioned behind the other windows, but you can bring it to the front when necessary. |

## 3. Performing a Typical ANSYS Analysis

The ANSYS program has many finite element analysis capabilities, ranging from a simple, linear, static analysis to a complex, nonlinear, transient dynamic analysis. The analysis guide manuals in the ANSYS documentation set describe specific procedures for performing analyses for different engineering disciplines.

A typical ANSYS analysis has three distinct steps:

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

## 3.1. Building a Model

Building a finite element model requires more of your time than any other part of the analysis. First, you specify a jobname and analysis title. Then, you use the PREP7 preprocessor to define the element types, element real constants, material properties, and the model geometry.

### 3.1.1. Specifying a Jobname and Analysis Title

This task is not required for an analysis, but is recommended.

*Defining the Jobname*

The jobname is a name that identifies the ANSYS job. When you define a jobname for an analysis, the jobname becomes the first part of the name of all files the analysis creates.

If you do not specify a jobname, all files receive the name FILE or file, depending on the operating system. You can change the default jobname as follows:

- By using the initial jobname entry option when you enter the ANSYS program, either via the launcher or on the ANSYS execution command
- From within the ANSYS program, you can use either of the following:
  Command(s): /FILNAME
  GUI: *Utility Menu>File>Change Jobname*

*Defining an Analysis Title*

The /TITLE command (*Utility Menu>File>Change Title*), defines a title for the analysis. ANSYS includes the title on all graphics displays and on the solution output. You can issue the /STITLE command to add subtitles; these will appear in the output, but not in graphics displays.

### 3.1.2. Defining Element Types.

The ANSYS element library contains more than 150 different element types. Each element type has a unique number and a prefix that identifies the element category e.g.: BEAM4, PLANE77, SOLID96, etc.

The element type determines, among other things:

- The degree-of-freedom set (which in turn implies the discipline: structural, thermal, magnetic, electric, quadrilateral, brick, etc.)
- Whether the element lies in two-dimensional or three-dimensional space.

You must be in PREP7, the general preprocessor, to define element types. To do so, you use the ET family of commands (ET, ETCHG, etc.) or their GUI path equivalents:
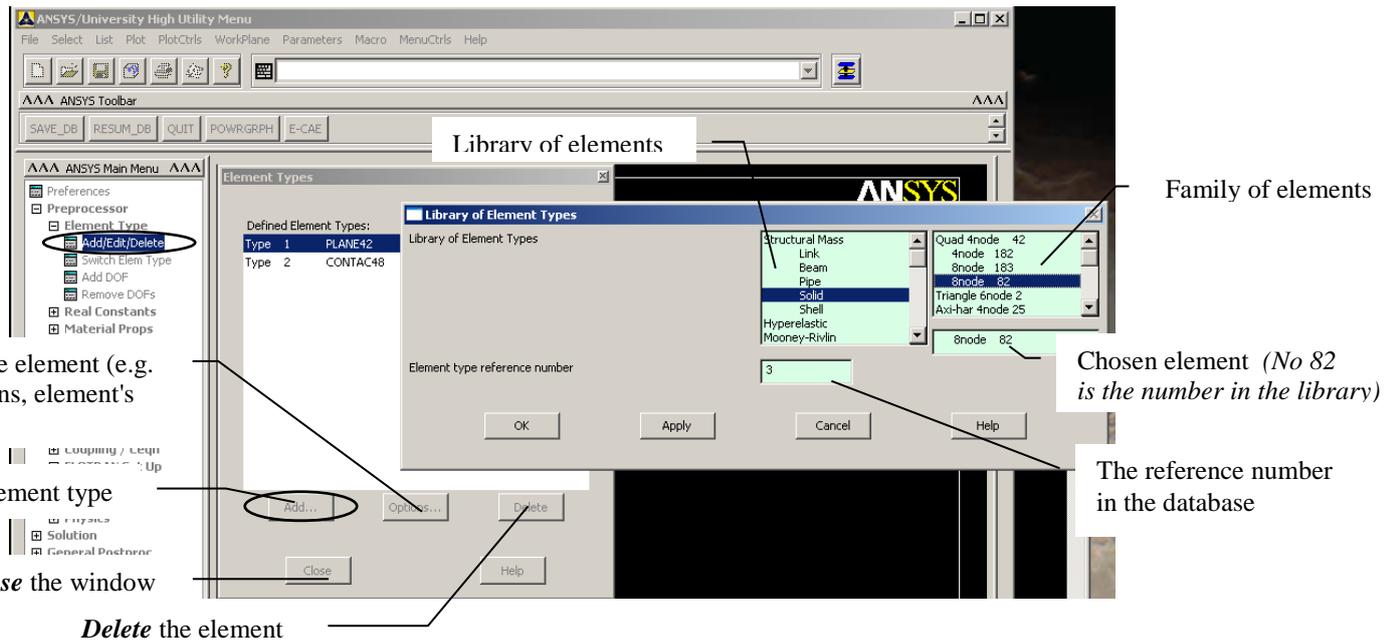


### 3.1.3 Defining Element Real Constants

Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element. Not all element types require real constants, and different elements of the same type may have different real constant values.

You can specify real constants using the R family of commands (R, RMODIF, etc.) or their equivalent menu paths:
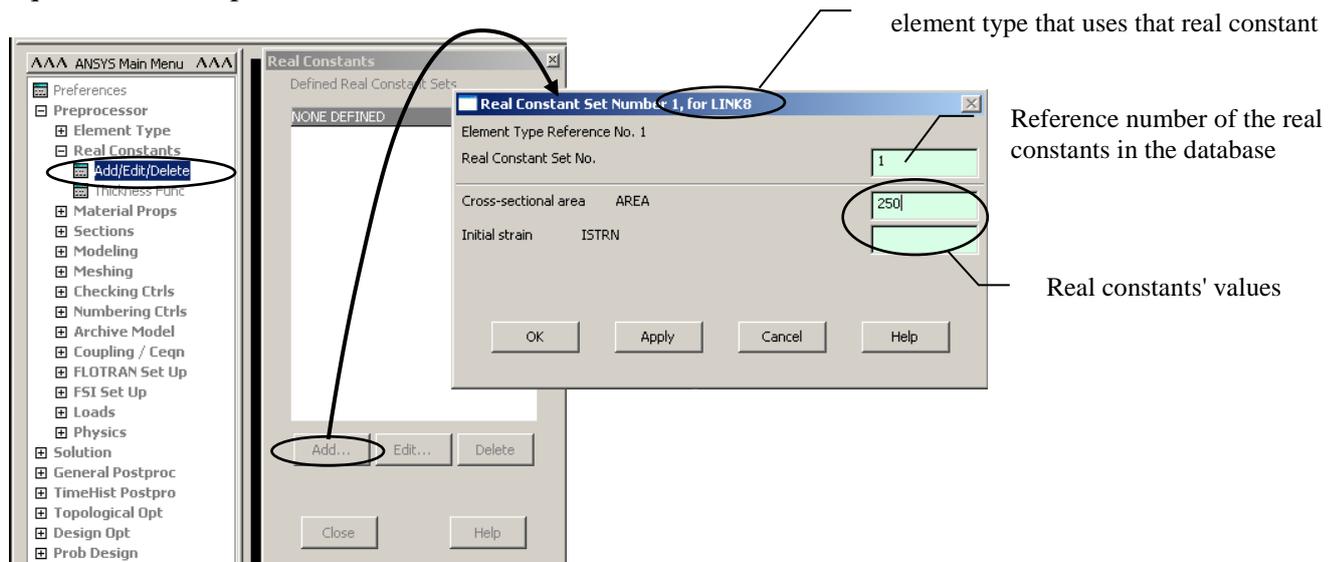
### 3.1.4. Defining Material Properties

Most element types require material properties. As with element types and real constants, each set of material properties has a material reference number. The table of material reference numbers versus material property sets is called the material table. Within one analysis, you may have multiple material property sets (to correspond with multiple materials used in the model). ANSYS identifies each set with a unique reference number.

Depending on the application, material properties can be:
- linear (Linear material properties can be constant or temperature-dependent, and isotropic or orthotropic)
- nonlinear (Nonlinear material properties are usually tabular data, such as plasticity data (stress-strain curves for different hardening laws), magnetic field data (B-H curves), creep data, swelling data, hyperelastic material data, etc.).
- Anisotropic Elastic Material Properties (Some element types accept anisotropic elastic material properties, which are usually input in the form of a matrix. (These properties are different from anisotropic plasticity, which requires different stress-strain curves in different directions.))

*Material Model Interface*
ANSYS includes an intuitive hierarchical tree structure interface for defining material models. A logical top-down arrangement of material categories guides you in defining the appropriate model for your analysis.

*Accessing the Interface*
You access the material model interface from ***Main Menu>Preprocessor>Material Props>Material Models***.

*Choosing Material Behavior*
The Material Models Available window on the right displays a list of material categories (for example, Structural, Thermal, Electromagnetics). If a category is preceded by a folder icon, there are subcategories available under the main category. When you double-click on the category, the subcategories appear indented, and below the category

*Entering Material Data*
Included in a data input dialog box is a table whose rows and columns you can alter depending on the requirements of the specific material property or model you have chosen.

*Temperature Dependent Data*
Initially, the table is set up for temperature independent data so the temperature field is grayed out. At this point, should you decide to enter data for various temperatures, you can quickly add columns of text fields for the data representing each temperature. You can add or delete the temperature dependent data at any time. You do not need to pre-determine if the data should be temperature dependent.
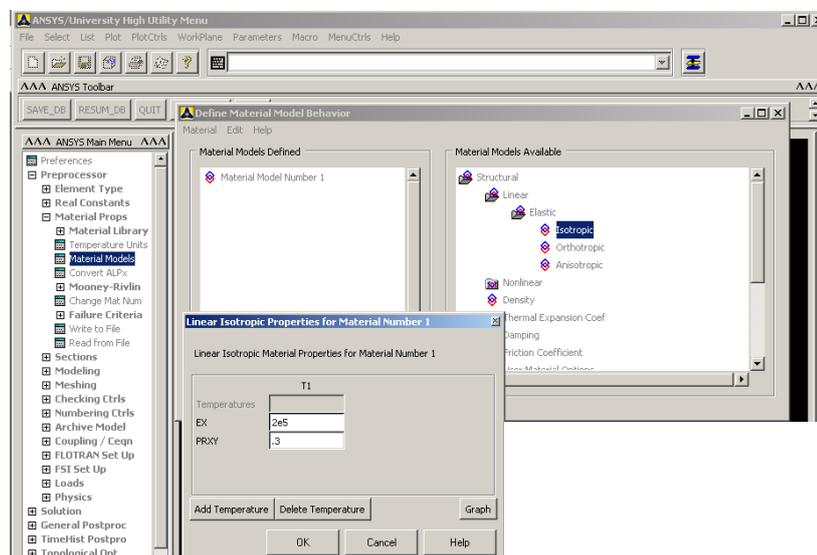
### 3.1.5. Creating the Model Geometry

Once you have defined material properties, the next step in an analysis is generating a finite element model, nodes and elements, which adequately describes the model geometry.
There are two methods to create the finite element model: *solid modeling* and *direct generation*. With *solid modeling*, you describe the geometric shape of your model, then instruct the ANSYS program to automatically mesh the geometry with nodes and elements. You can control the size and shape in the elements that the program creates. With *direct generation*, you "manually" define the location of each node and the connectivity of each element. Several convenience operations, such as copying patterns of existing nodes and elements, symmetry reflection, etc. are available.

*An Overview of Solid Modeling Operations*
The purpose of using a *solid model* is to relieve you of the time-consuming task of building a complicated finite element model by direct generation. Let's take a brief look at some of the solid modeling and meshing operations that you can use to speed up the creation of your final analysis model:

Building your model from the bottom up: Keypoints, the points that define the vertices of your model, are the "lowest-order" solid model entities. If, in building your solid model, you first create your keypoints, and then use those keypoints to define the "higher-order" solid model entities (that is, lines, areas, and volumes), you are said to be building your model "from the bottom up." Keep in mind that models built from the bottom up are defined within the currently active coordinate system.
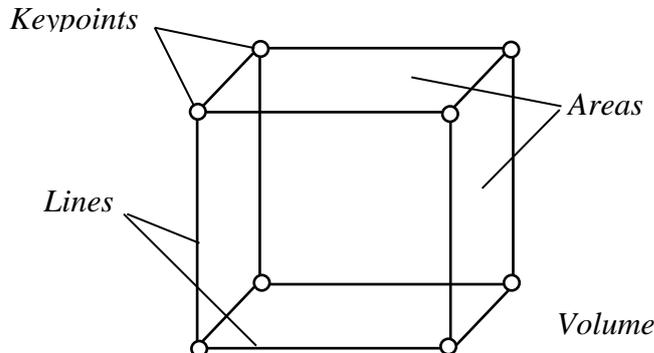


Building your model from the top down: The ANSYS program also gives you the ability to assemble your model using *geometric primitives*, which are fully-defined lines, areas, and volumes. As you create a *primitive*, the program automatically creates all the "lower" entities associated with it. If your modeling effort begins with the "higher" primitive entities, you are said to be building your model "from the top down." You can freely combine bottom up and top down modeling techniques, as appropriate, in any model. Remember that *geometric primitives* are built within the working plane while bottom up techniques are defined against the active coordinate system.

Keypoints

Lines , Arcs, Splines

Areas (Rectangles, Circles, Polygons, Area fillets )

Primitives

Volumes (*Rectangular prism, Cylinders, Prisms, Spheres, Cones, Torus* )
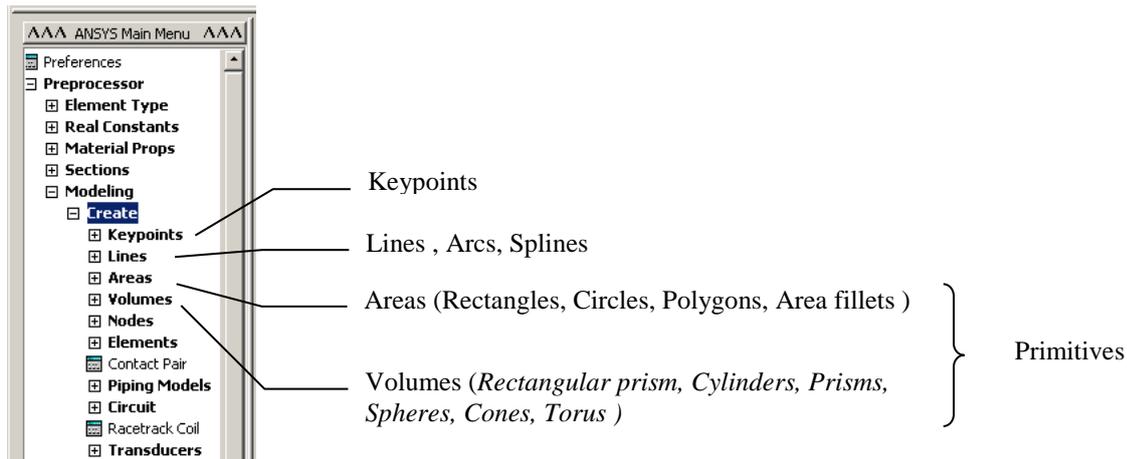
Using Boolean operators: You can "sculpt" your solid model using intersections, subtractions, and other Boolean operations. Booleans allow you to work directly with higher solid model entities to create complex shapes. (Both bottom up and top down creations can be used in Boolean operations.)

Extrude or Sweep

Extend Line

Intersect
Add
Substract
Divide
Glue
Overlap

Boolean operations

Scale
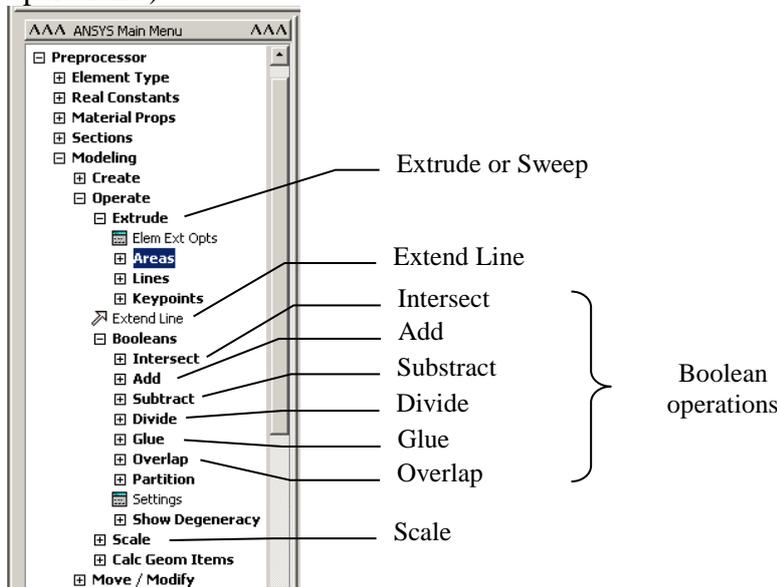
Dragging and rotating:
Boolean operators, although convenient, can be computationally expensive. Sometimes a model can be constructed more efficiently by dragging, or rotating.

Moving and copying solid model entities:
A complicated area or volume that appears repetitively in your model need only be constructed once; it can then be moved, rotated, and copied to a new location on your model. You might also find it more convenient to place geometric primitives in their proper location by moving them, rather than by changing the working plane.
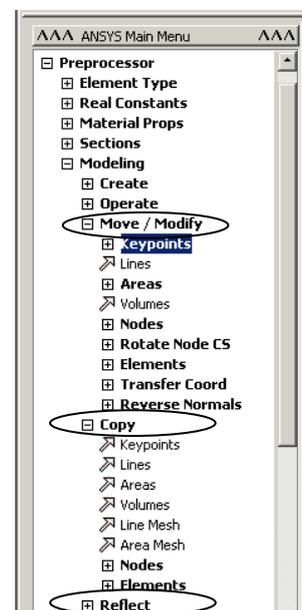
Meshing: Your ultimate objective in building a solid model is to mesh that model with nodes and elements. Once you have completed the solid model, set element attributes, and established meshing controls, you can then turn the ANSYS program to generate the finite element mesh. By taking care to meet certain requirements, you can request a "mapped" mesh containing all quadrilateral, all triangular, or all brick elements.

Define attributes

Size Controls

Clear mesh
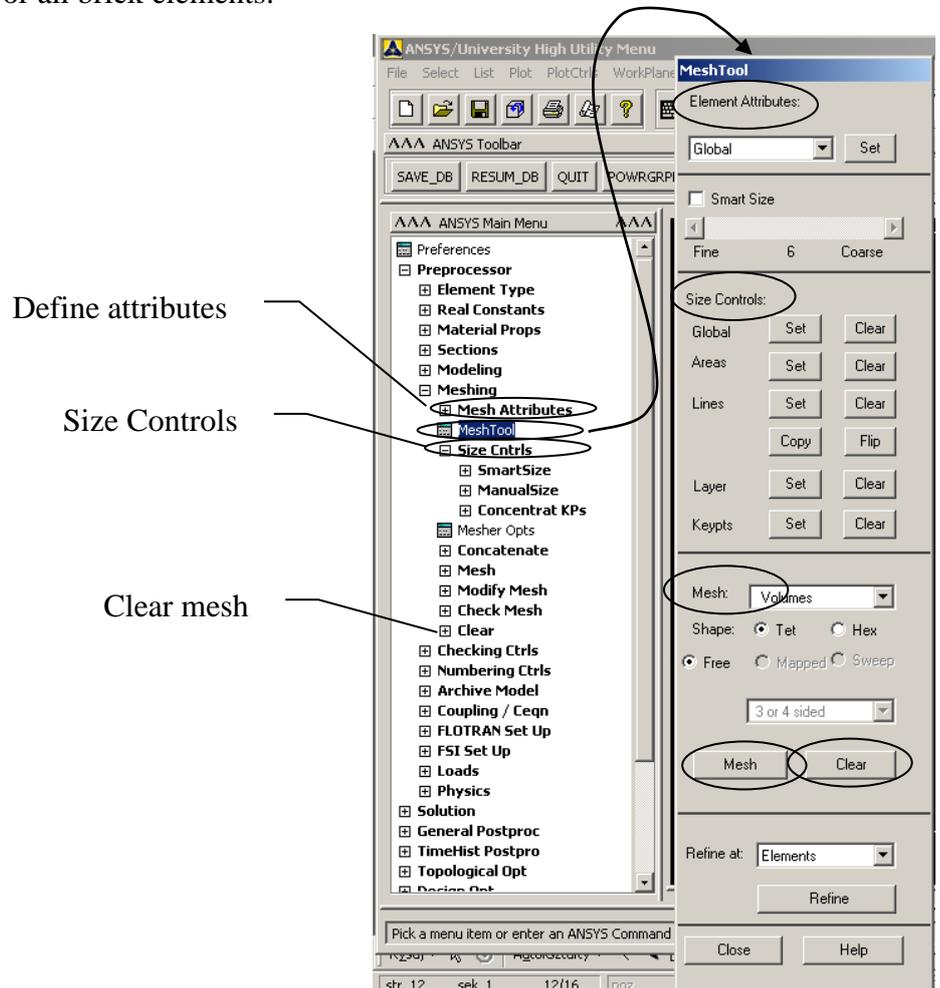
Revising your model (clearing and deleting): Before you can revise your model, you need to be aware of the hierarchy of solid model and finite element model entities. A lower order entity cannot be deleted if it is attached to a higher-order entity. Thus, a volume cannot be deleted if it has been meshed, a line cannot be deleted if it is attached to an area, and so forth. If an entity is attached to any loads, deleting or redefining that entity will delete the attached loads from the database. The hierarchy of modeling entities is as listed below:

| | |
|---|---|
| Highest | Elements (and Element Loads) |
| | Nodes (and Nodal Loads) |
| | Volumes (and Solid-Model Body Loads) |
| | Areas (and Solid-Model Surface Loads) |
| | Lines (and Solid-Model Line Loads) |
| Lowest | Keypoints (and Solid-Model Point Loads) |

If you need to revise a solid model after it has been meshed, you must usually first delete all the nodes and elements in the portion of the model to be revised, using the xCLEAR commands (*Main Menu>Preprocessor>Clear*).

## 3.2. Apply Loads and Obtain the Solution

In this step, you use the SOLUTION processor to define the analysis type and analysis options, apply loads, specify load step options, and initiate the finite element solution. You also can apply loads using the PREP7 preprocessor.

### *Defining the Analysis Type and Analysis Options*

You choose the analysis type based on the loading conditions and the response you wish to calculate. For example, if natural frequencies and mode shapes are to be calculated, you would choose a modal analysis. You can perform the following analysis types in the ANSYS program: static (or steady-state), transient, harmonic, modal, spectrum, buckling, and substructuring.
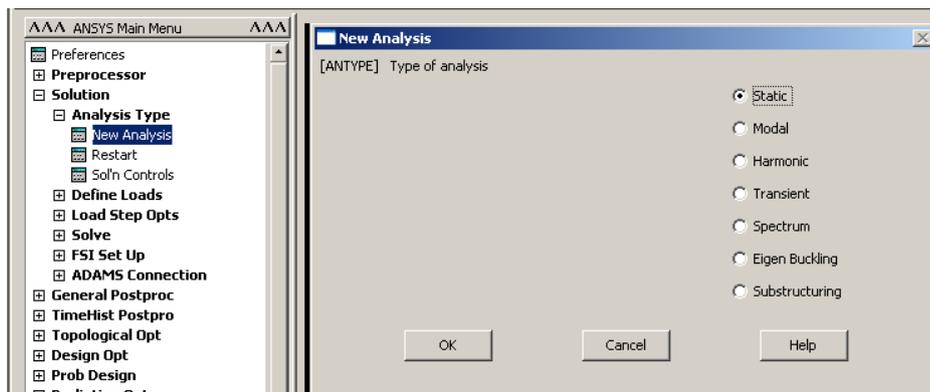


Not all analysis types are valid for all disciplines. Modal analysis, for example, is not valid for a thermal model. The analysis guide manuals in the ANSYS documentation set describe the analysis types available for each discipline and the procedures to do those analyses.

Analysis options allow you to customize the analysis type. Typical analysis options are the method of solution, stress stiffening on or off, and Newton-Raphson options.
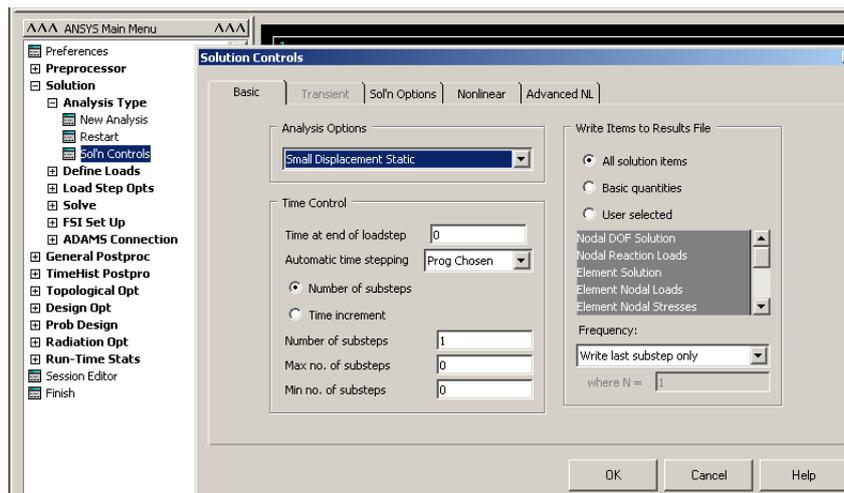


If you are performing a static or full transient analysis, you can take advantage of the Solution Controls dialog box to define many options for the analysis.

*Applying Loads*

The word loads as used in ANSYS documentation includes boundary conditions (constraints, supports, or boundary field specifications) as well as other externally and internally applied loads.
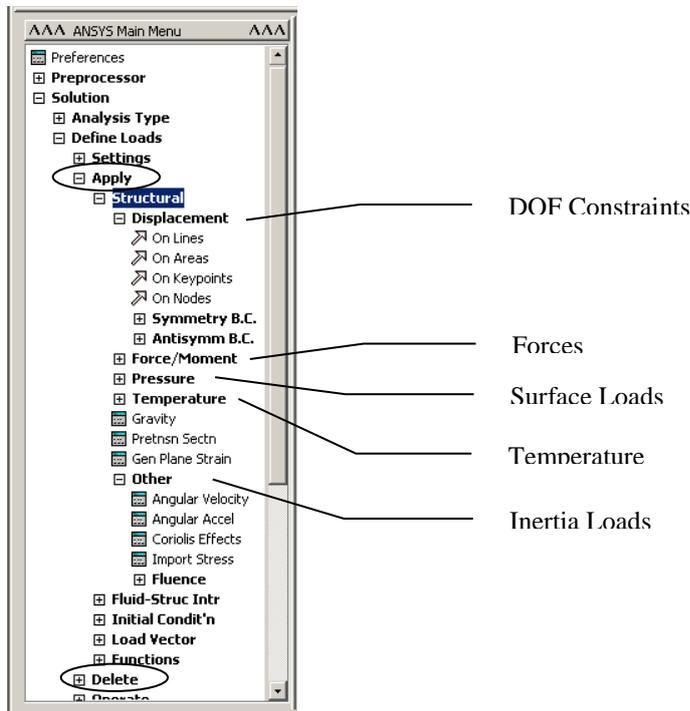


You can apply most of these loads either on the solid model (keypoints, lines, and areas) or the finite element model (nodes and elements).

Two important load-related terms you need to know are load step and substep. A load step is simply a configuration of loads for which you obtain a solution. In a structural analysis, for example, you may apply wind loads in one load step and gravity in a second load step. Load steps are also useful in dividing a transient load history curve into several segments.

Substeps are incremental steps taken within a load step. You use them mainly for accuracy and convergence purposes in transient and nonlinear analyses. Substeps are also known as time steps -- steps taken over a period of time.

Note: The ANSYS program uses the concept of time in transient analyses as well as static (or steady-state) analyses. In a transient analysis, time represents actual time, in seconds, minutes, or hours. In a static or steady-state analysis, time simply acts as a counter to identify load steps and substeps.

*Specifying Load Step Options*

Load step options are options that you can change from load step to load step, such as number of substeps, time at the end of a load step, and output controls. Depending on the type of analysis you are doing, load step options may or may not be required. The analysis procedures in the analysis guide manuals describe the appropriate load step options as necessary.

### *Initiating the Solution*

To initiate solution calculations, issue SOLVE command. The ANSYS program takes model and loading information from the database and calculates the results. Results are written to the results file (Jobname.RST, Jobname.RTH, Jobname.RMG, or Jobname.RFL) and also to the database. The only difference is that only one set of results can reside in the database at one time, while you can write all sets of results (for all substeps) to the results file.

You can solve multiple load steps in a convenient manner:
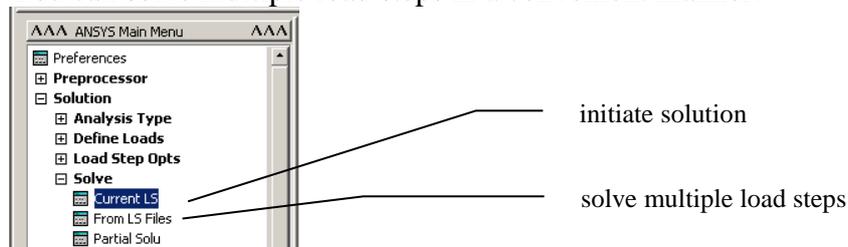
## 3.3. Review the Results

You use POST1, the general postprocessor, to review results at one substep (time step) over the entire model or selected portion of the model. The command to enter POST1 is /POST1 (*Main Menu>General Postproc*), valid only at the Begin level. You can obtain contour displays, deformed shapes, and tabular listings to review and interpret the results of the analysis. POST1 offers many other capabilities, including error estimation, load case combinations, calculations among results data, and path operations.



Presentation of nodal results

Selected entity

Component to present



Presentation of path results

Define path

Delete path

Map an item on the path

Plot path items

Path functions