Course in FEM – ANSYS Classic

Geometric modeling

Modeling

Programme for Lesson:

BUILD THE MODEL

- Modeling considerations
- Element Type
- Real Constants
- Material Properties
- Sections
- Geometry/Modeling
  - WorkPlane & Coordinate systems
  - Keypoints
  - Lines
  - Areas
  - Volumes
- Meshing

Review

Interpolation: *A computation of an approximate value to a function on the basis of another function given by values in two outer points to an interval.*

In the element method the values of the displacements $u_1, u_2, \ldots$ in the element nodal points are such points. These known values are degrees of freedoms in the element model and are computed/approximated by the solution of the system of algebraic system of equations.

Interpolation is the essence of the FE method. Using sufficiently fine element distribution with linear shape functions it is possible to model even relatively complex problems (geometrically as well as due stress state) applying simple interpolation functions.

Review

- Equilibrium for nodal forces and -moments is satisfied.
- Compatibility is satisfied in FE nodes.
- Equilibrium is not satisfied across the element boundaries.
- Compatibility is not necessarily satisfied across element boundaries. For the triangular and the rectangular element compatibility is satisfied as the element sides remain straight under deformation.
- Equilibrium is not satisfied for the individual element (due to the weak formulation – integral form).
- Compatibility is satisfied for the individual element, i.e. the displacement field must be continuous. This is automatically achieved by a proper formulation of the element shape functions, i.e. polynomial formulation.

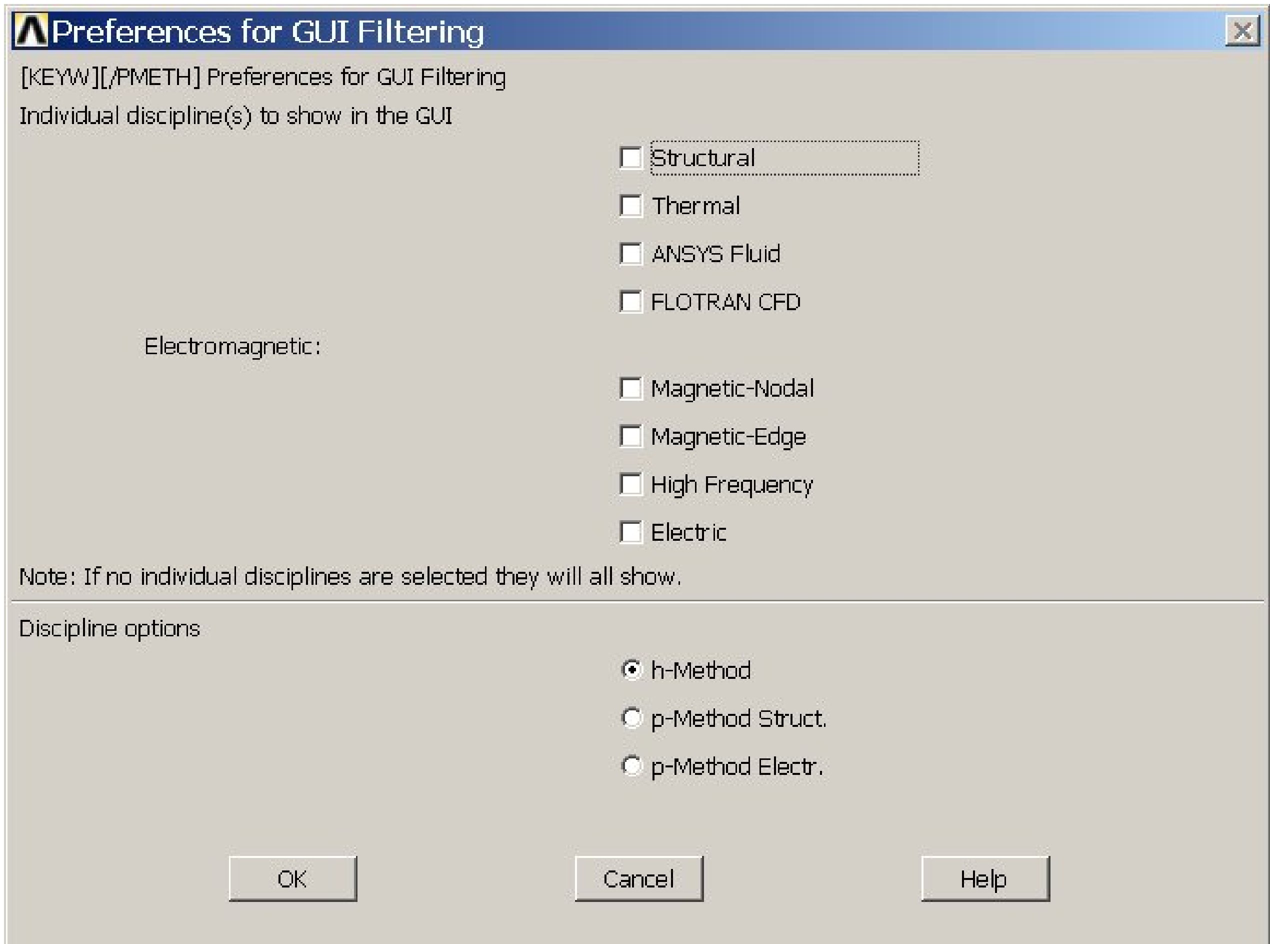
Modeling considerations

- As you begin your model generation, you will (consciously or unconsciously) make a number of decisions that determine how you will mathematically simulate the physical system:
  - What are the objectives of your analysis?
  - Will you need to vary/modify model data?
  - Will you need to change the geometric topology of the model, e.g. add holes to the model?
  - Will you model all, or just a portion, of the physical system?
  - How much detail will you include in your model?
  - What kinds of elements will you use? How dense should your finite element mesh be?
- In general, you will attempt to balance computational expense (CPU time, etc.) against precision of results as you answer these questions.
- The decisions you make in the planning stage of your analysis will largely govern the success or failure of your analysis efforts.

Modeling considerations

- Linear or Higher Order Elements
- Take Advantage of Symmetry
  - The axis of symmetry *must* coincide with the global Cartesian Y-axis.
  - Negative nodal X-coordinates are not permitted.
  - The global Cartesian Y-direction represents the axial direction, the global Cartesian X-direction represents the radial direction, and the global Cartesian Z-direction corresponds to the circumferential direction.
  - Your model should be assembled using appropriate element types:
    - For axisymmetric models, use applicable 2-D solids with KEYOPT(3) = 1, and/or axisymmetric shells. In addition, various link, contact, combination, and surface elements can be included in a model that also contains axisymmetric solids or shells. (The program will not realize that these "other" elements are axisymmetric unless axisymmetric solids or shells are present.)
- How Much Detail to Include
- Appropriate Mesh Density

Modeling considerations

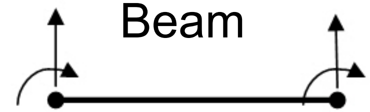


Modeling considerations

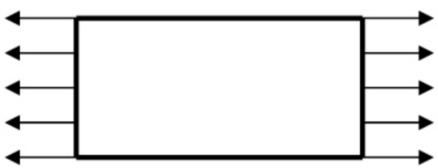
- Characterization of problem

Rod

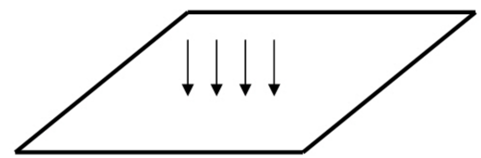
Beam



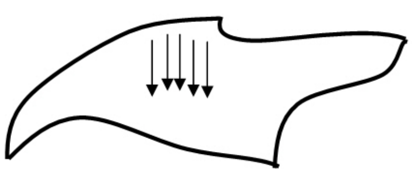
Disk



Plate



Shell



Solid

Modeling considerations

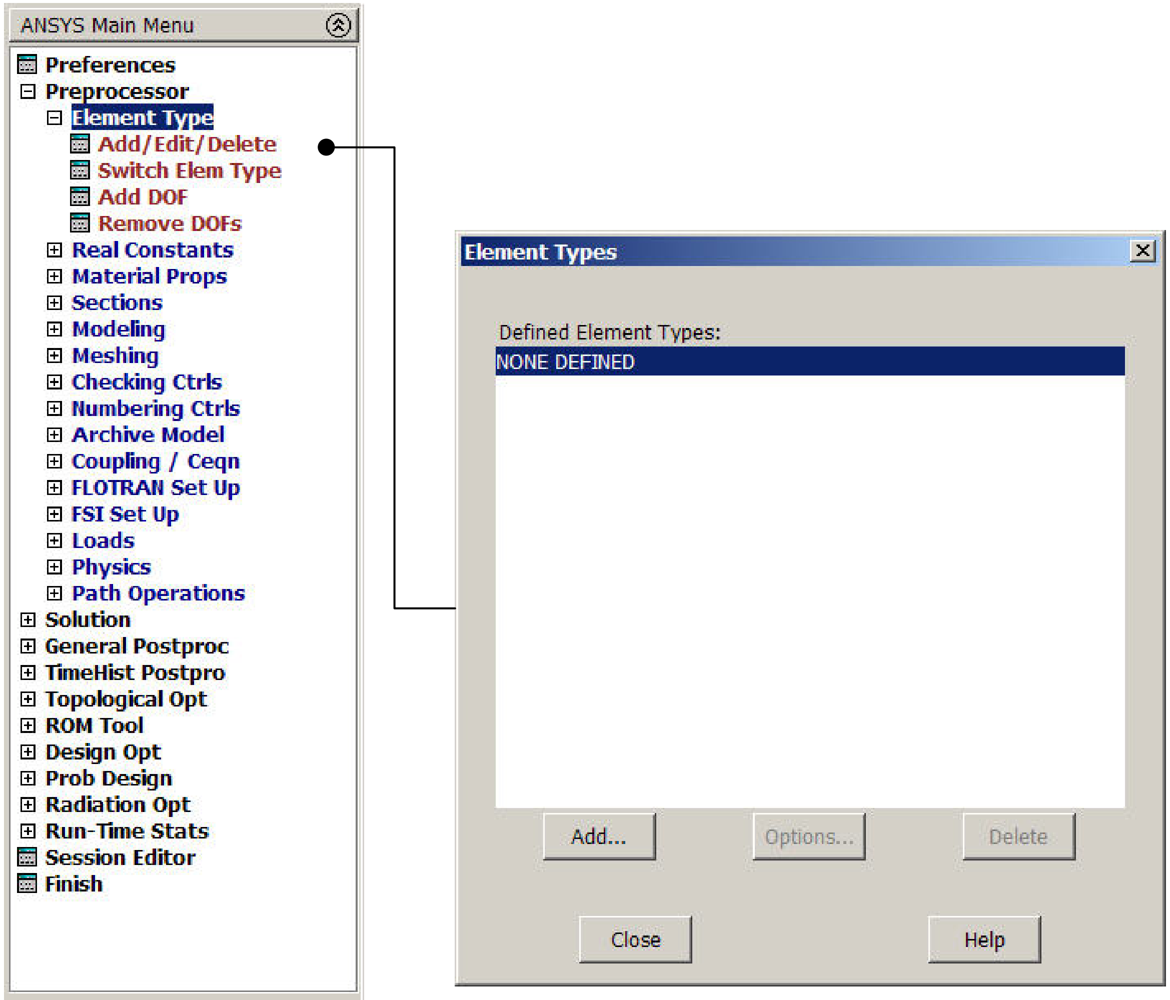
- The ANSYS program does not assume a system of units for your analysis.
- Units must however be consistent for all input data.

Element Type

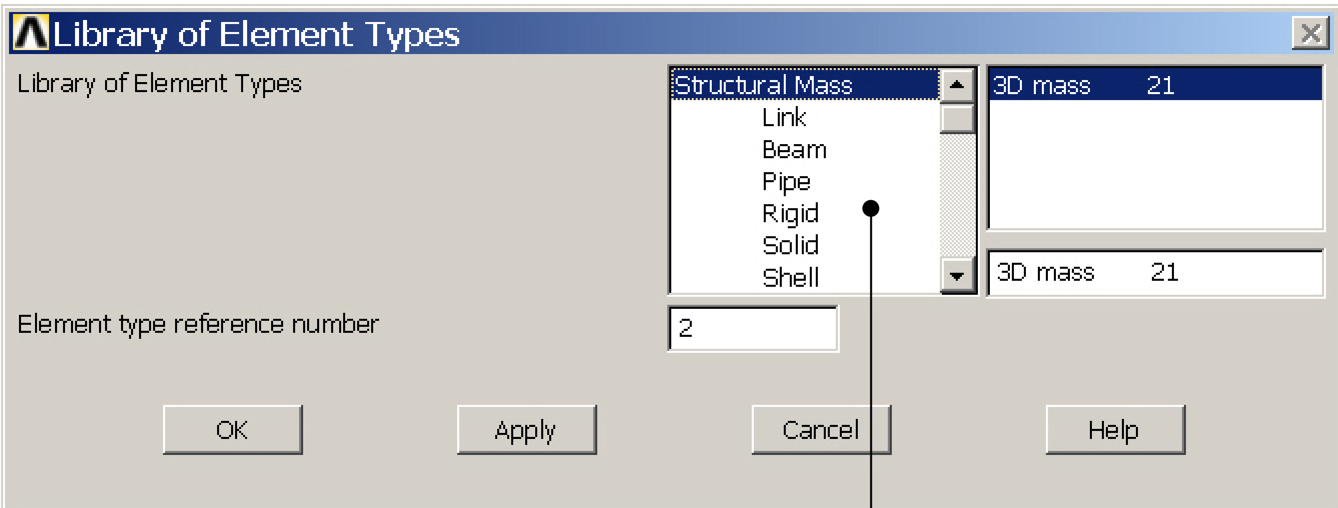
- BEAM
- CIRCUit
- COMBINation
- CONTACt
- FLUID
- HF (High Frequency)
- HYPERelastic
- INFINite
- INTERface
- LINK
- MASS
- MATRIX
- MESH
- Multi-Point Constraint
- PIPE
- PLANE
- PRETS (Pretension)
- SHELL
- SOLID
- SOURCe
- SURFace
- TARGEt
- TRANSducer
- USER
- VISCOelastic (or viscoplastic)

Element Type

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



Element Type



The ANSYS element library contains more than 150 different element types

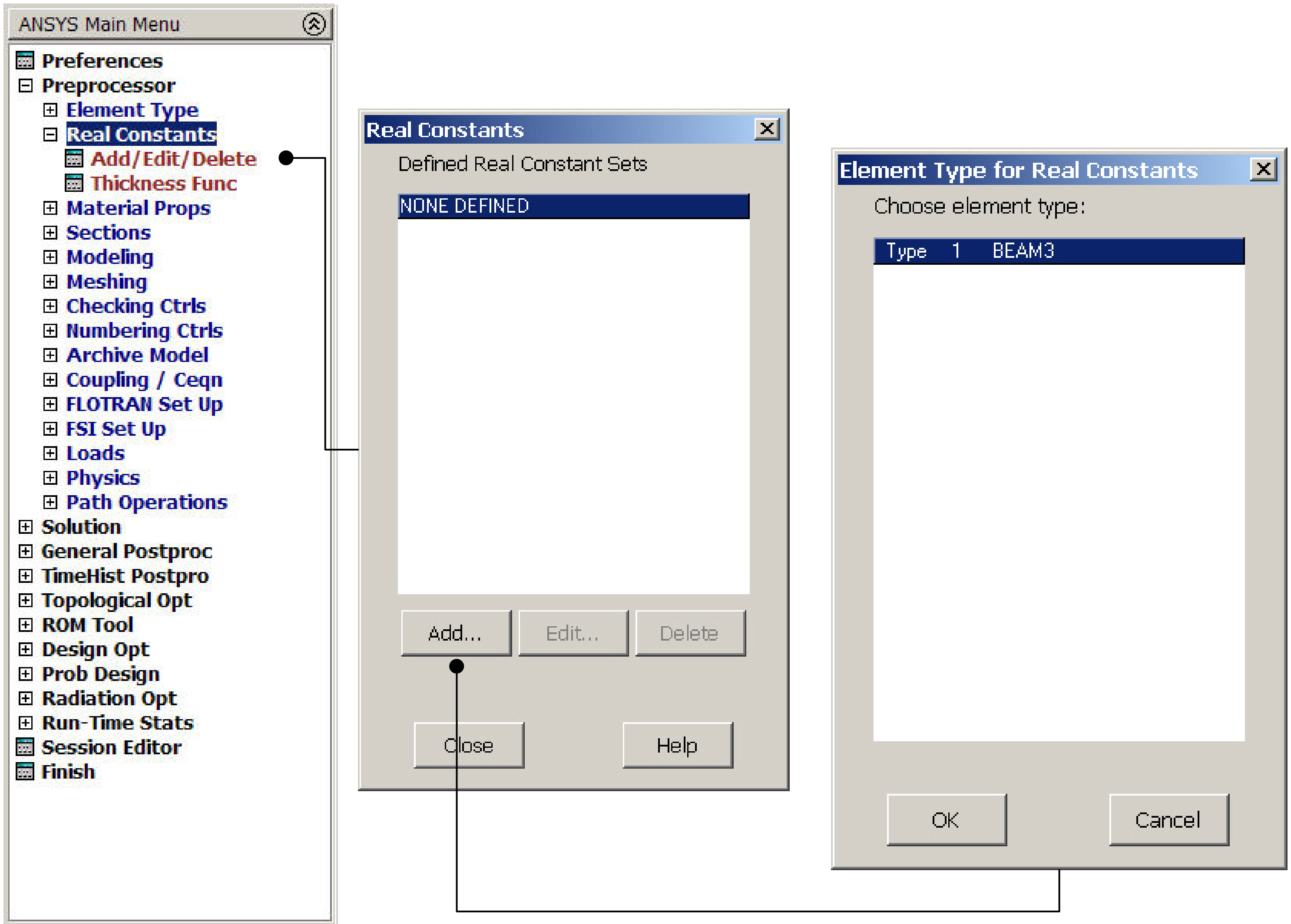
Each element type has a unique number and a prefix that identifies the element category

ET,1,BEAM4
ET,2,SHELL63

Element Type

- Many element types have additional options, known as KEYOPTs, and are referred to as KEYOPT(1), KEYOPT(2), etc. e.g.:
  – KEYOPT(9) for BEAM4 allows you to choose results to be calculated at intermediate locations on each element
  – KEYOPT(3) for SHELL63 allows you to suppress extra displacement shapes

Real Constants



Real Constants

- Element real constants are properties that depend on the element type, such as cross-sectional properties of a beam element
  - e.g. real constants for BEAM3, the 2-D beam element, are area (AREA), moment of inertia (IZZ), height (HEIGHT), shear deflection constant (SHEARZ), initial strain (ISTRN), and added mass per unit length (ADDMAS).
- Not all element types require real constants, and different elements of the same type may have different real constant values.

Real Constants

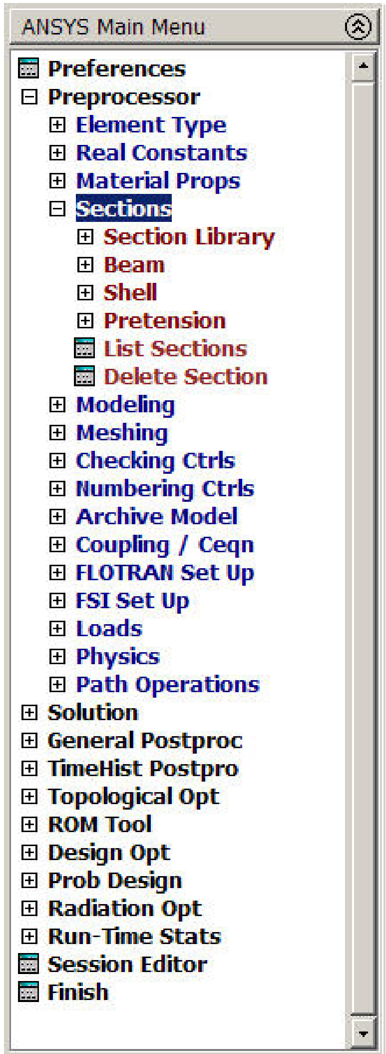
- For line and area elements that require geometry data (cross-sectional area, thickness, diameter, etc.) to be specified as real constants, you can verify the input graphically by using the following commands in the order shown:

  Utility Menu> PlotCtrls> Style> Size and Shape

  Utility Menu> Plot> Elements

- ANSYS displays the elements as solid elements, using a rectangular cross-section for link and shell elements and a circular cross-section for pipe elements. The cross-section proportions are determined from the real constant values.

Sections

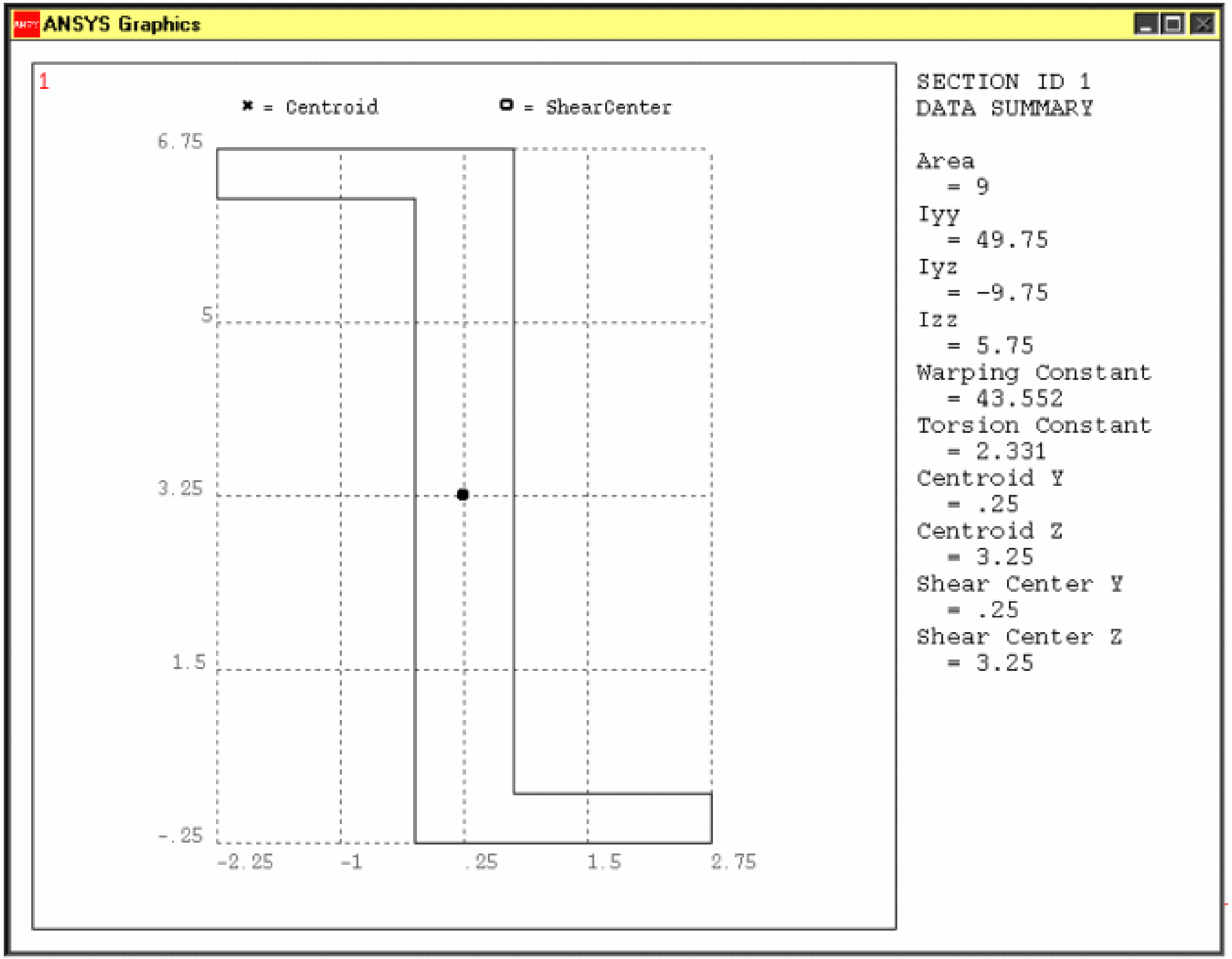


Building a model using BEAM44, BEAM188, or BEAM189, you can use the section commands (**SECTYPE**, **SECDATA**, etc.) or their GUI path equivalents to define and use cross sections in your models.

Sections

- A cross section defines the geometry of the beam in a plane perpendicular to the beam axial direction. ANSYS supplies a library of eleven commonly-used beam cross section shapes, and permits user-defined cross section shapes.
- When a cross section is defined, ANSYS builds a numeric model using a nine node cell for determining the properties (Iyy, Izz, etc.) of the section and for the solution to the Poisson's equation for torsional behaviour.

Sections



ANSYS Graphics
1
✖ = Centroid
○ = ShearCenter
6.75
5
3.25
1.5
-.25
-2.25
-1
.25
1.5
2.75
SECTION ID 1
DATA SUMMARY
Area
= 9
Iyy
= 49.75
Iyz
= -9.75
Izz
= 5.75
Warping Constant
= 43.552
Torsion Constant
= 2.331
Centroid Y
= .25
Centroid Z
= 3.25
Shear Center Y
= .25
Shear Center Z
= 3.25

Geometry/Modelling

- Creating a solid model within ANSYS.
- Using direct generation.
- Importing a model created in a computer-aided design (CAD) system.

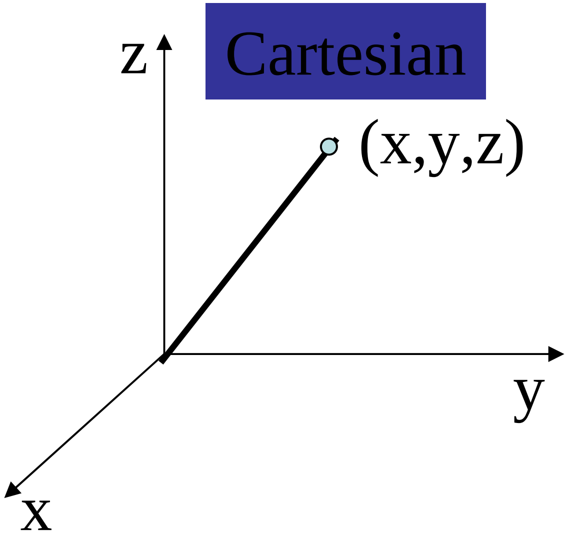
Coordinate systems

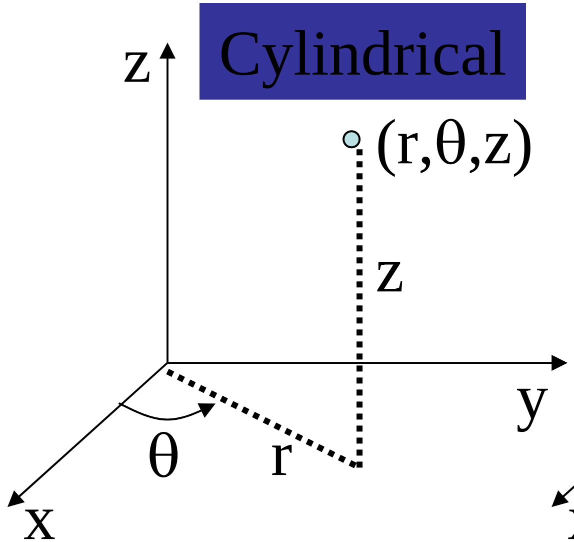
- *Global* and *local* coordinate systems are used to locate geometry items (nodes, keypoints, etc.) in space.
- The *display* coordinate system determines the system in which geometry items are listed or displayed.
- The *nodal* coordinate system defines the degree of freedom directions at each node and the orientation of nodal results data.
- The *element* coordinate system determines the orientation of material properties and element results data.
- The *results* coordinate system is used to transform nodal or element results data to a particular coordinate system for listings, displays, or general postprocessing operations (POST1).
- The working plane, which is separate from the coordinate systems discussed in this chapter, is used to locate geometric primitives during the modeling process.

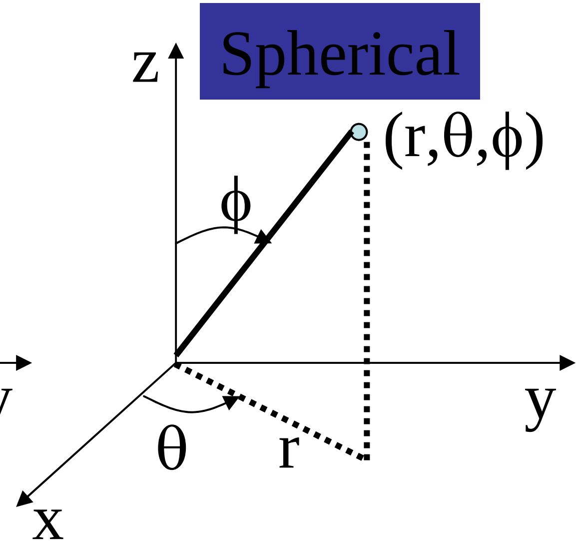
Coordinate systems

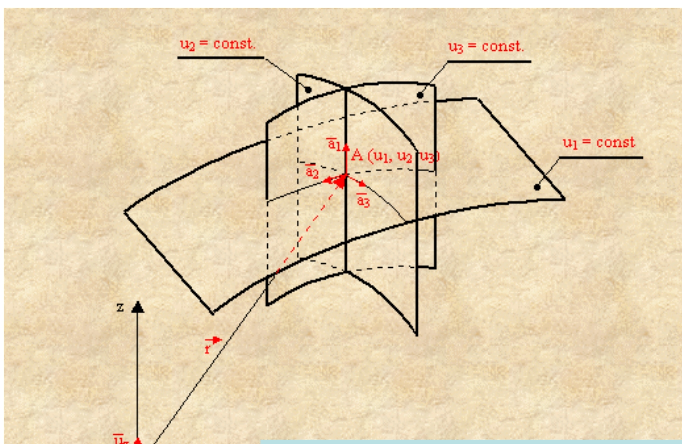
- *(a) Cartesian* (X, Y, Z components) coordinate system 0 (C.S.0)
- *(b) Cylindrical* (R, θ, Z components) coordinate system 1 (C.S.1)
- *(c) Spherical* (R, θ, φ components) coordinate system 2 (C.S.2)
- *(d) Cylindrical* (R, θ, Y components) coordinate system 5 (C.S.5)

Modeling (coordinates)

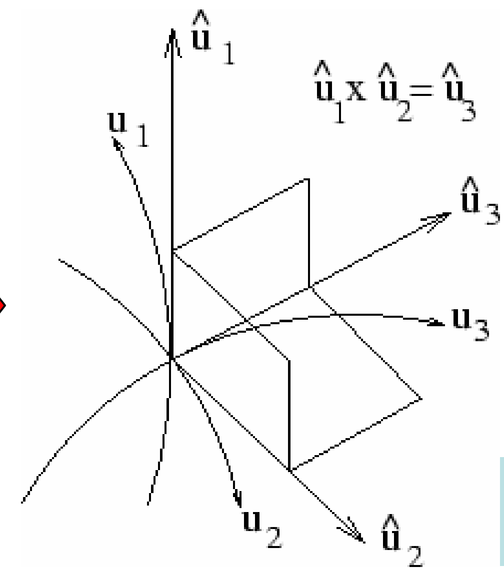






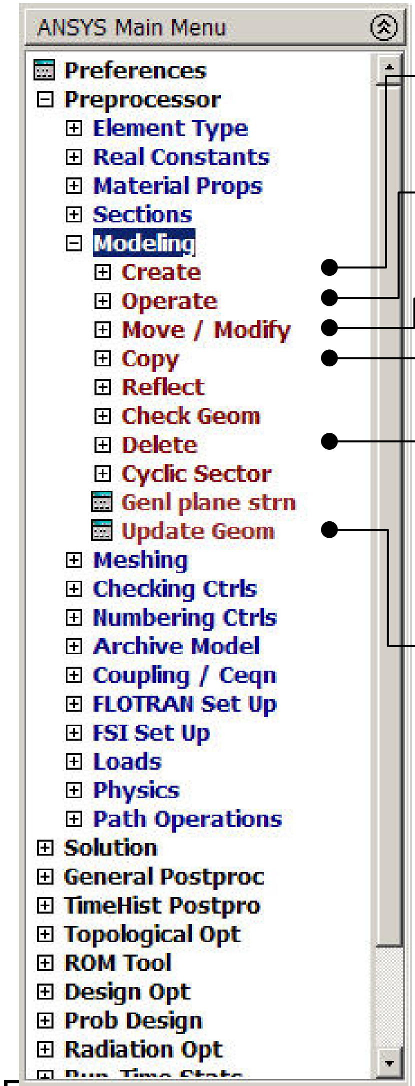


General Curvilinear Coordinates



General orthogonal Coordinates

Geometry/Modelling



Create – geometrical entities

Operate – perform Boolean operations

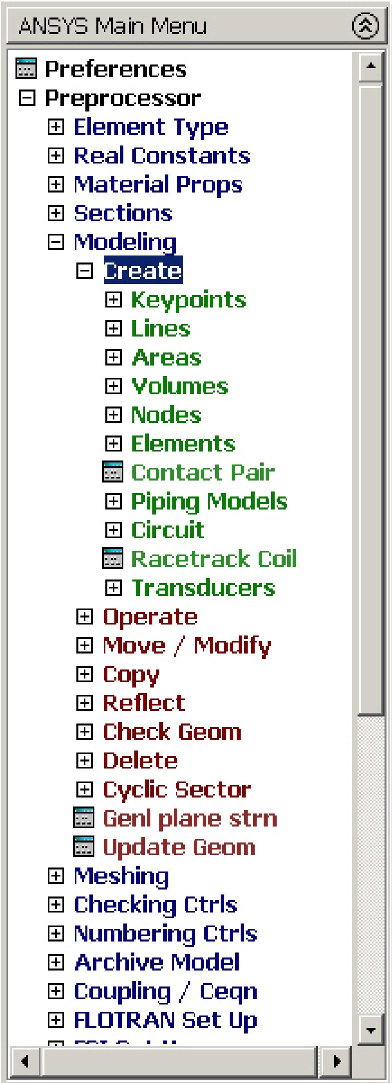
Move / Modify – move or modify geometrical entities

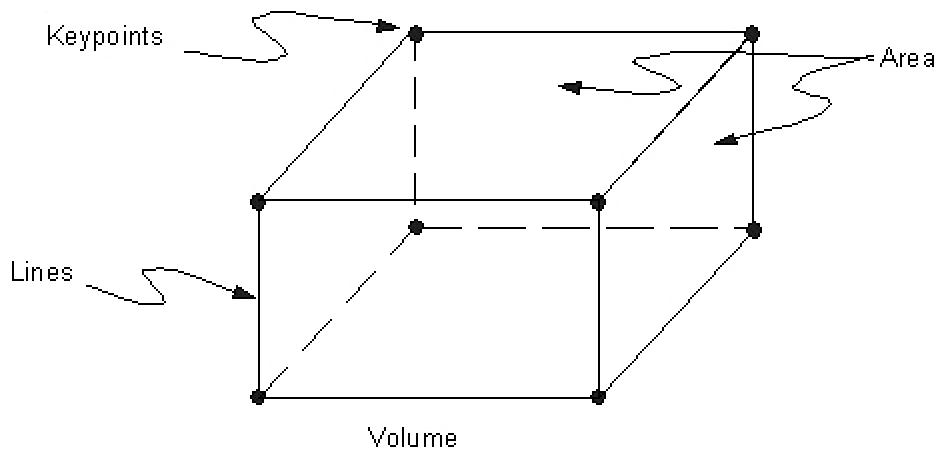
Copy – copy geometrical entities

Delete – geometrical entities

Update Geom – update the geometry in relation to for example buckling analysis

Modeling - Create



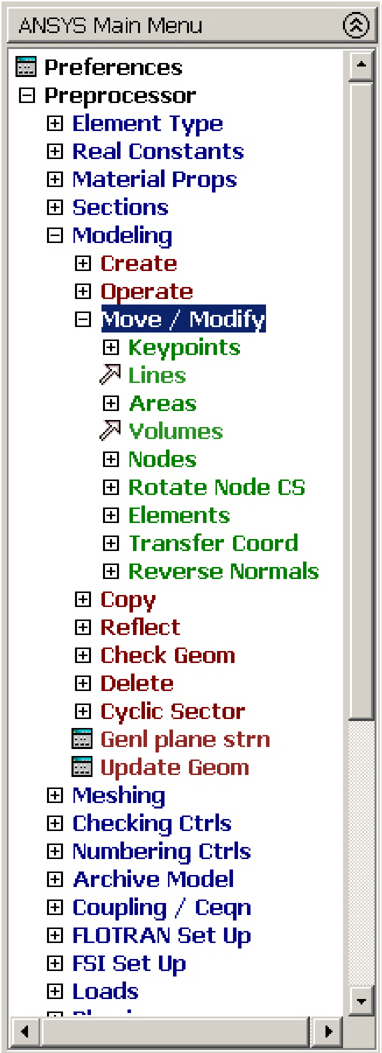


- The hierarchy of modeling entities is as listed below:
  - Elements (and Element Loads)
  - Nodes (and Nodal Loads)
  - Volumes (and Solid-Model Body Loads)
  - Areas (and Solid-Model Surface Loads)
  - Lines (and Solid-Model Line Loads)
  - Keypoints (and Solid-Model Point Loads)

Modeling - Operate

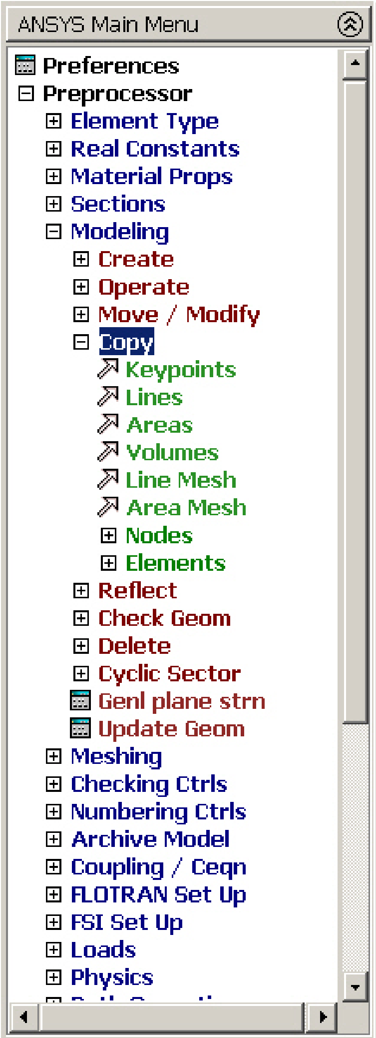
Perform geometrical operations in order to obtain new geometrical entities

Modeling - Move/Modify



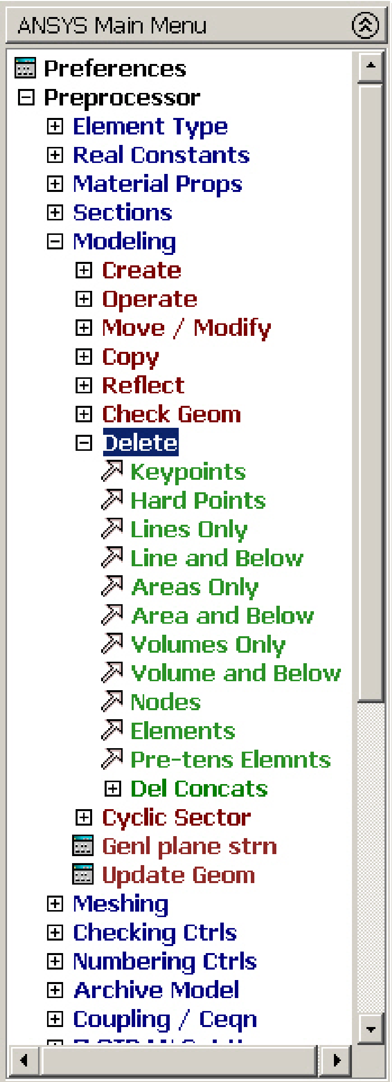
Move or modify locations or sizes of geometrical entities

Modeling - Copy



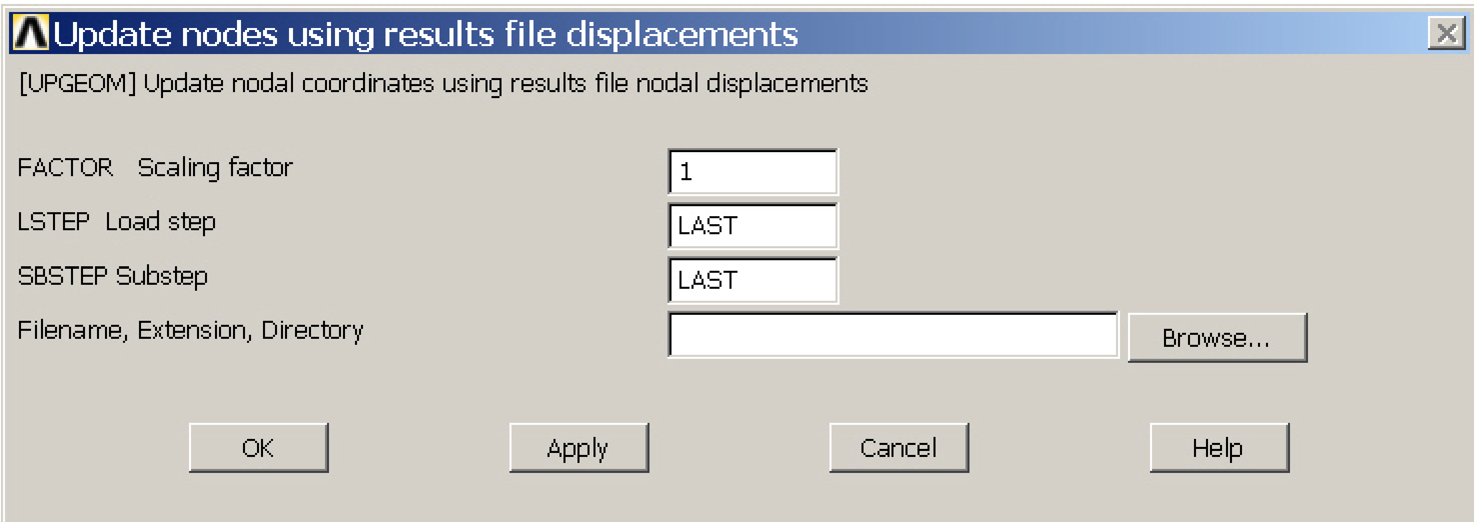
Copy geometrical entities to new geometrical entities with new locations

Modeling - Delete



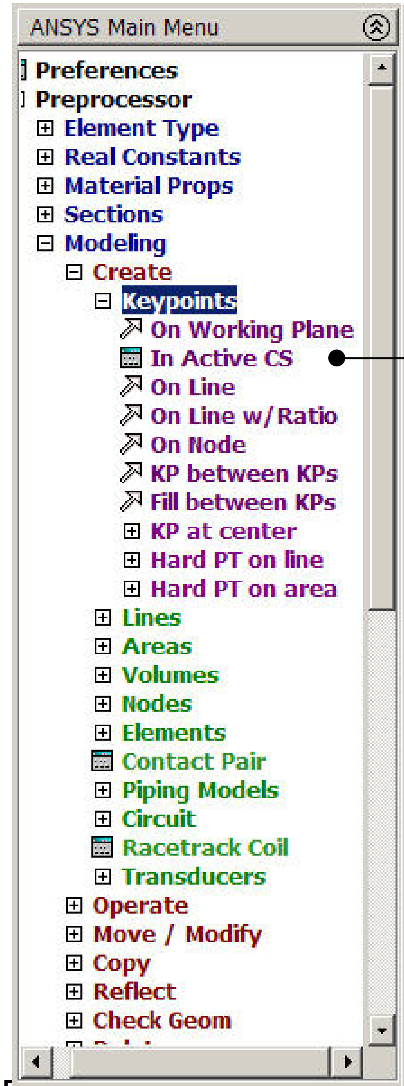
- The hierarchy of modeling entities is as listed below:
  - Elements (and Element Loads)
  - Nodes (and Nodal Loads)
  - Volumes (and Solid-Model Body Loads)
  - Areas (and Solid-Model Surface Loads)
  - Lines (and Solid-Model Line Loads)
  - Keypoints (and Solid-Model Point Loads)

Modeling - Update Geom

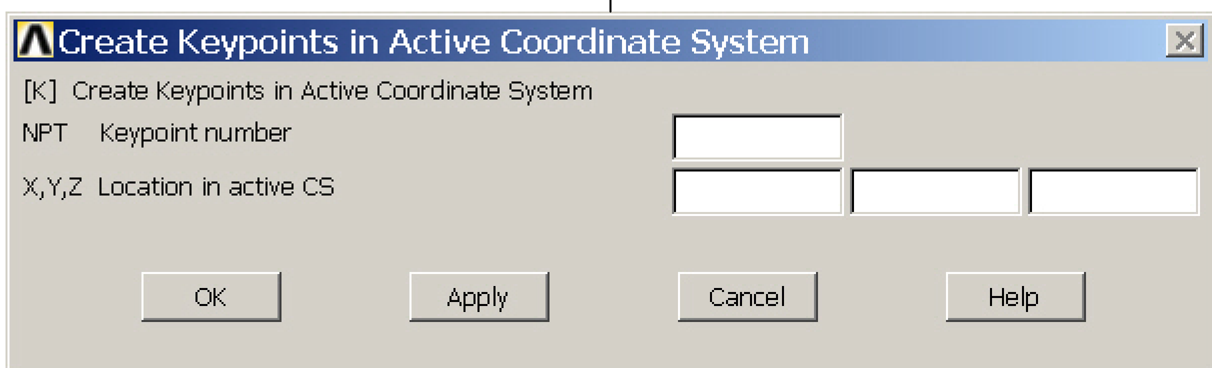


Adds displacements from a previous analysis and updates the geometry of the finite element model to the deformed configuration.

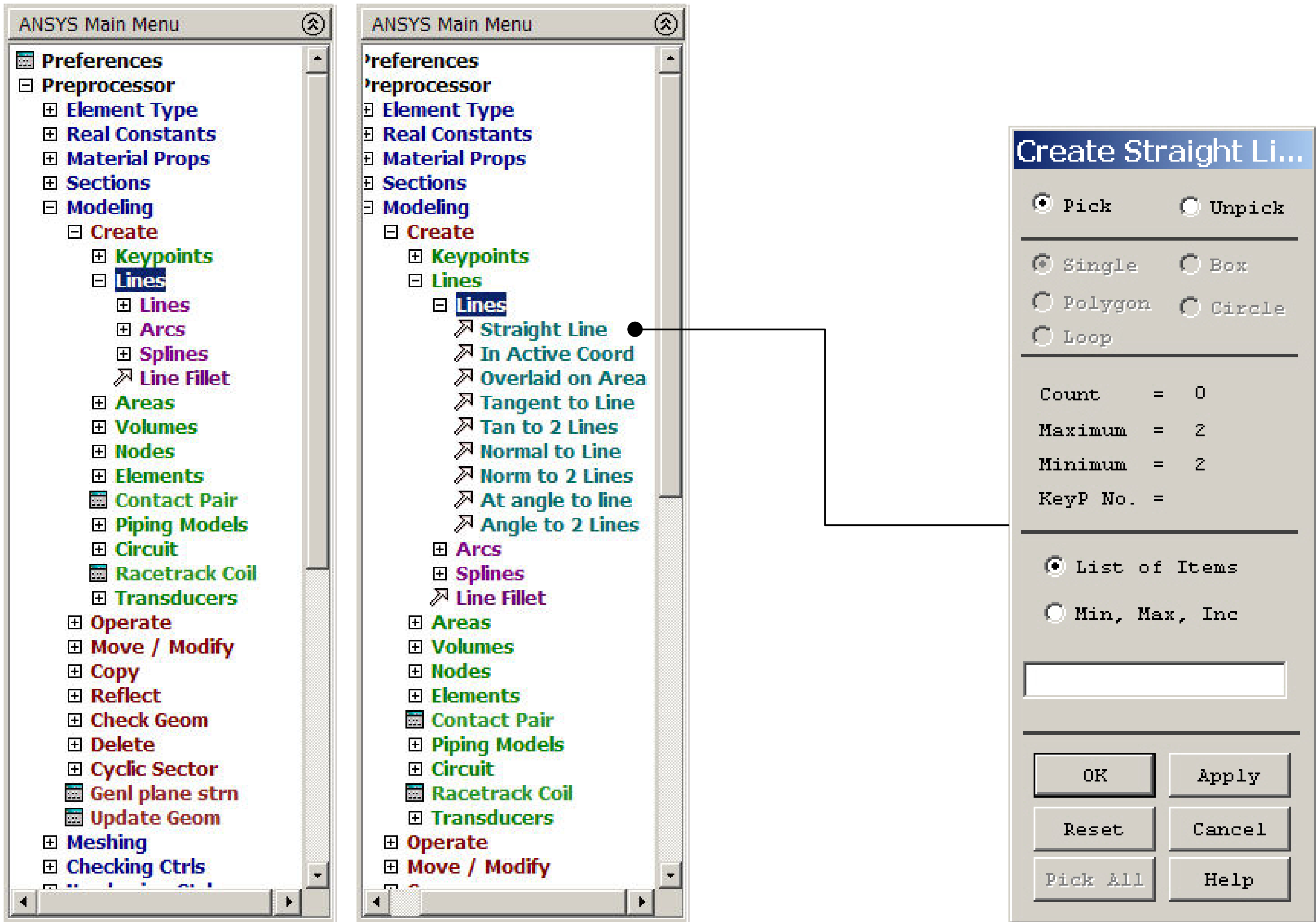
Create – Keypoints (In Active CS)



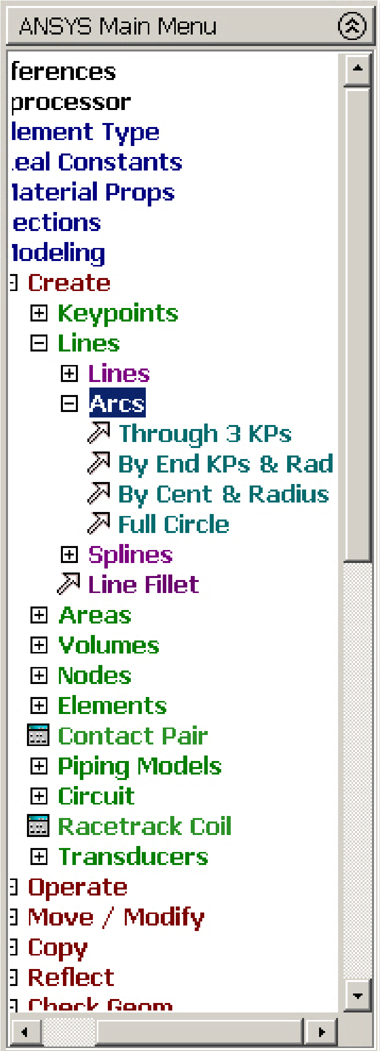
It is a good idea to use keypoints as reference points in the modeling phase

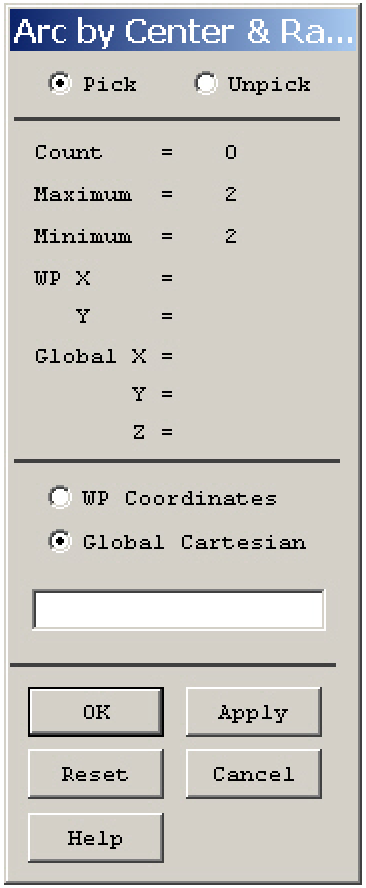


Create – Lines (Straight Line)

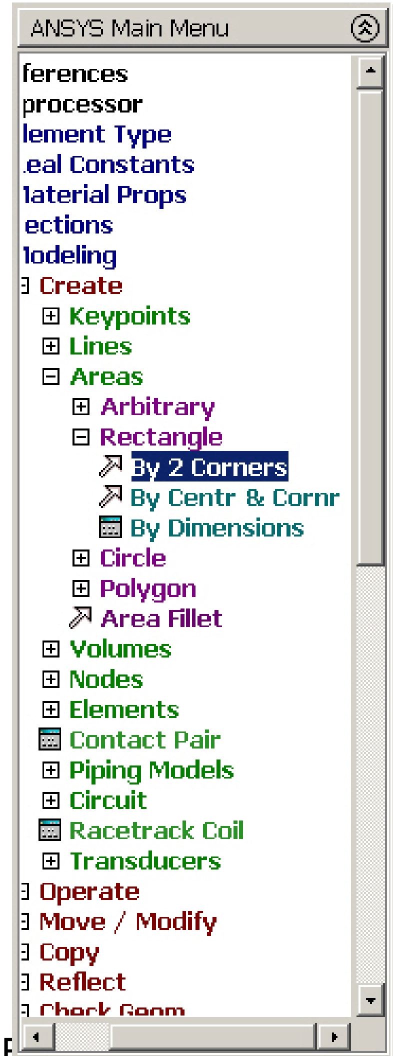


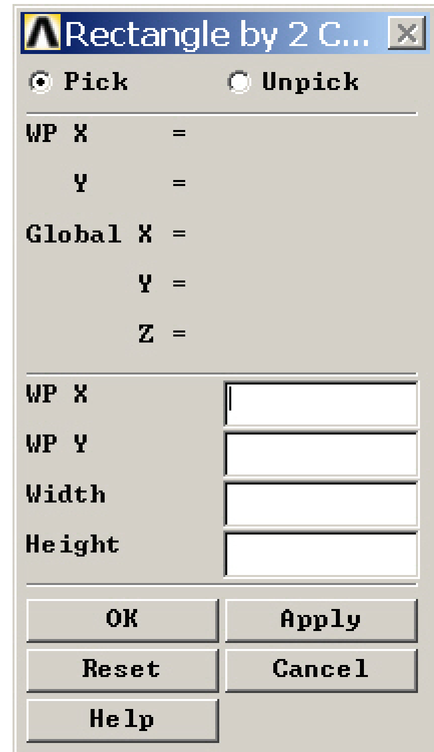
Create – Lines - Arcs



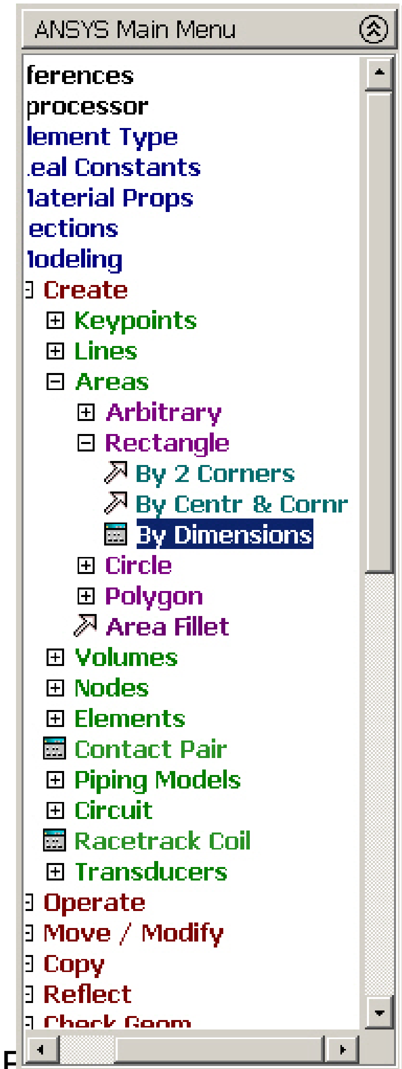


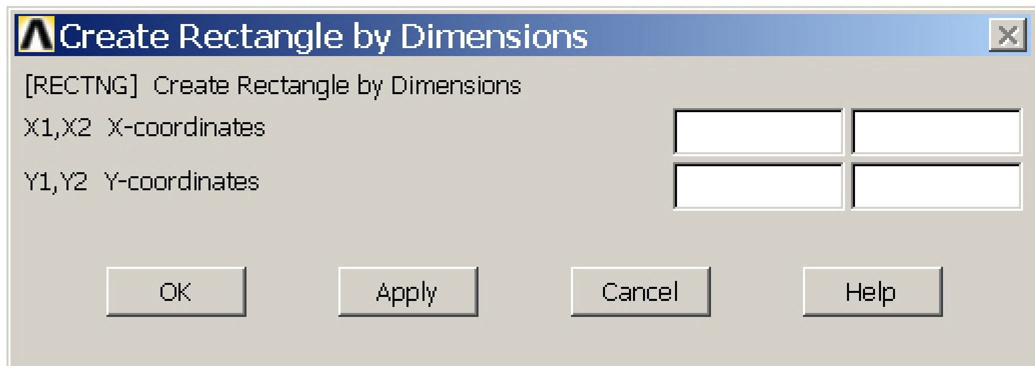
Create – Areas (By 2 Corners)



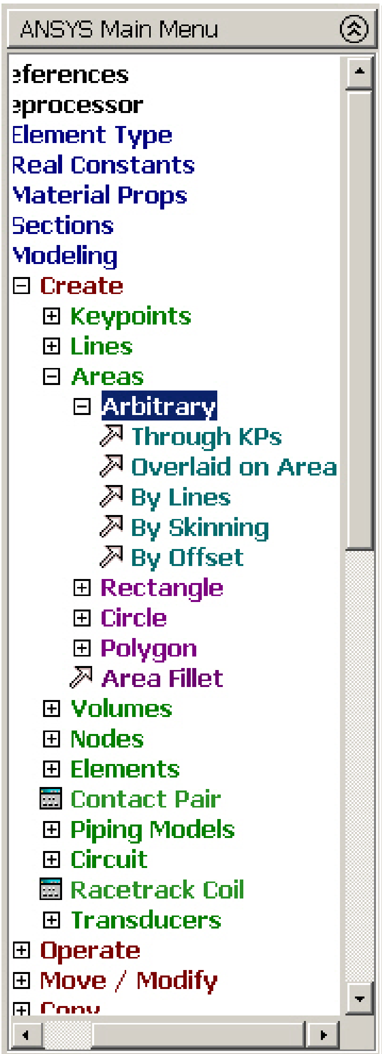


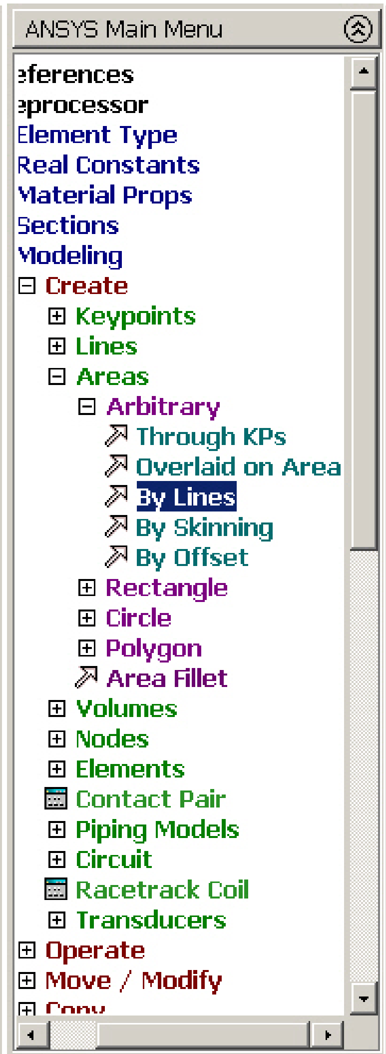
Create – Areas (By dimensions)

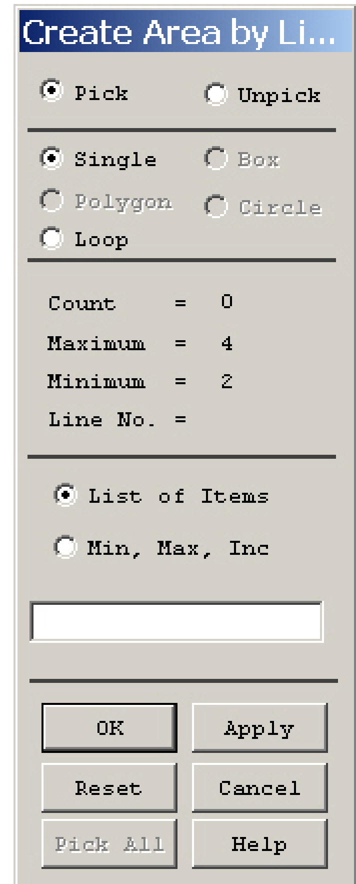




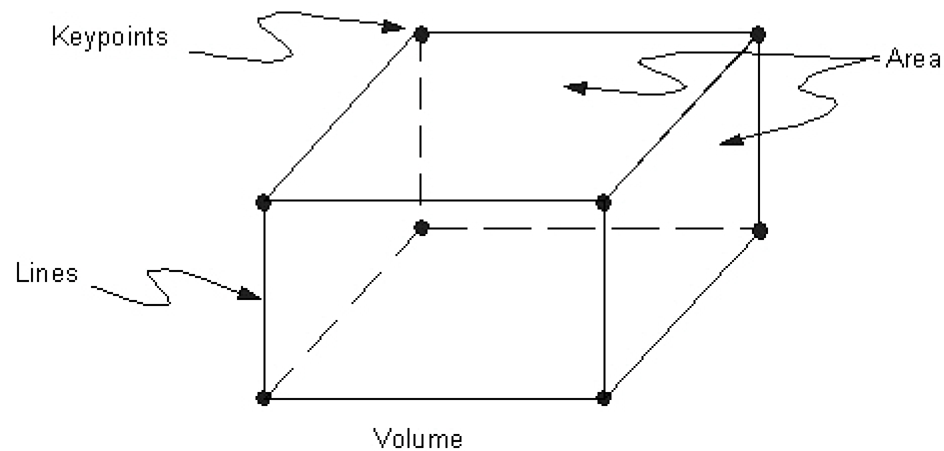
Create – Areas (By Lines)





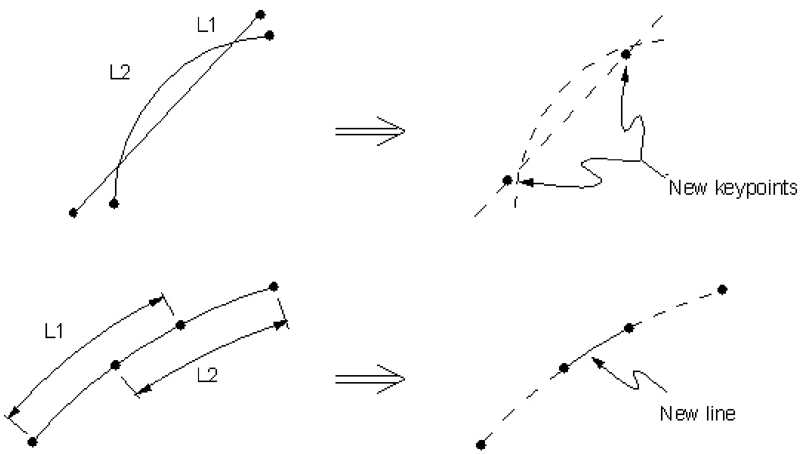


Create - Volumes

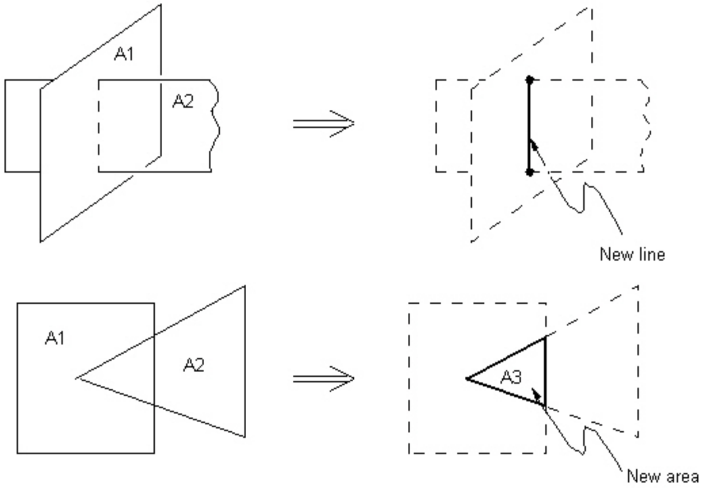


Booleans - Intersect

LINL (Line Intersect Line)

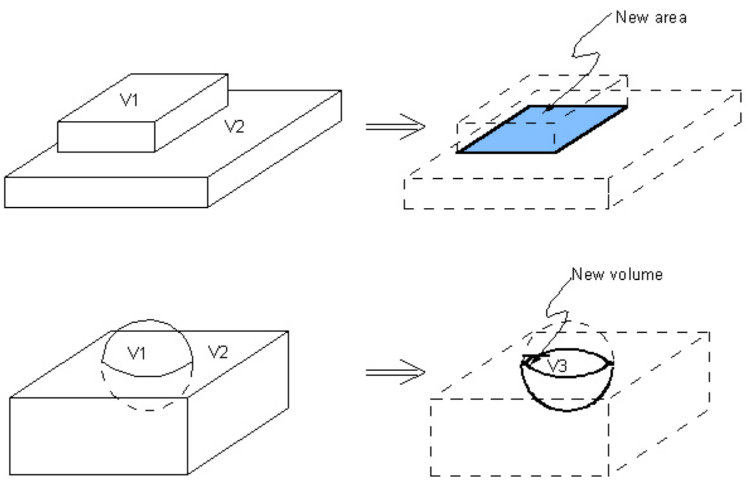


AINA (Area Intersect Area)

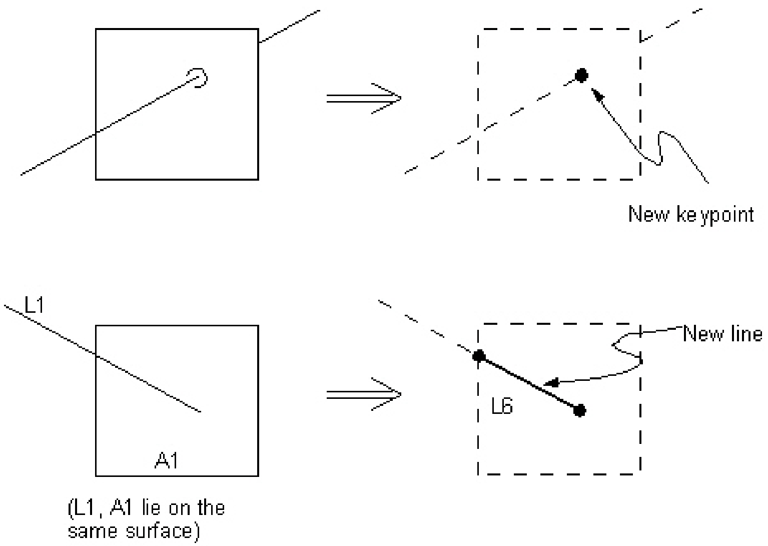


Booleans - Intersect

VINV (Volume Intersect Volume)

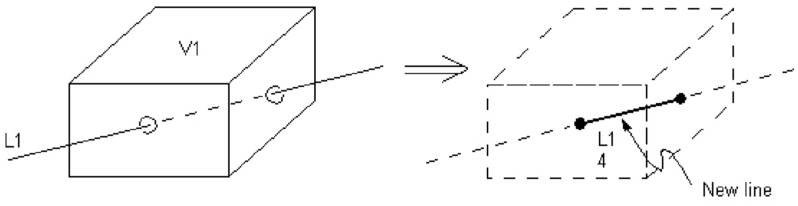


LINA (Line Intersect Area)

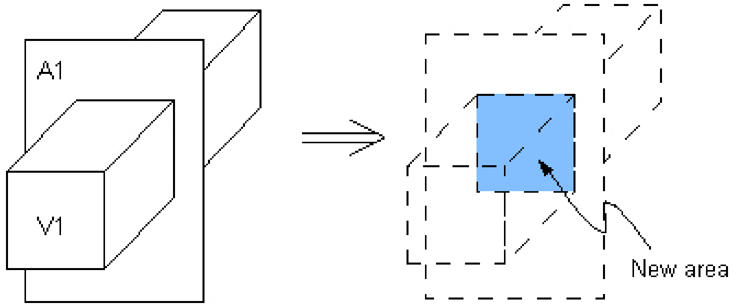


Booleans - Intersect

LINV (Line Intersect Volume)

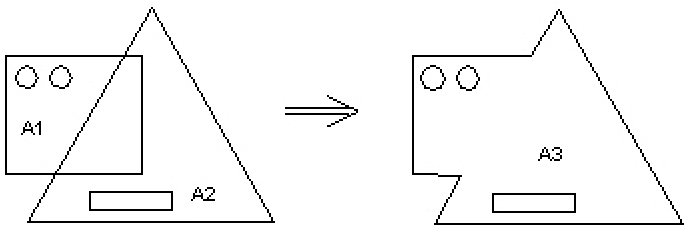


AINV (Area Intersect Volume)

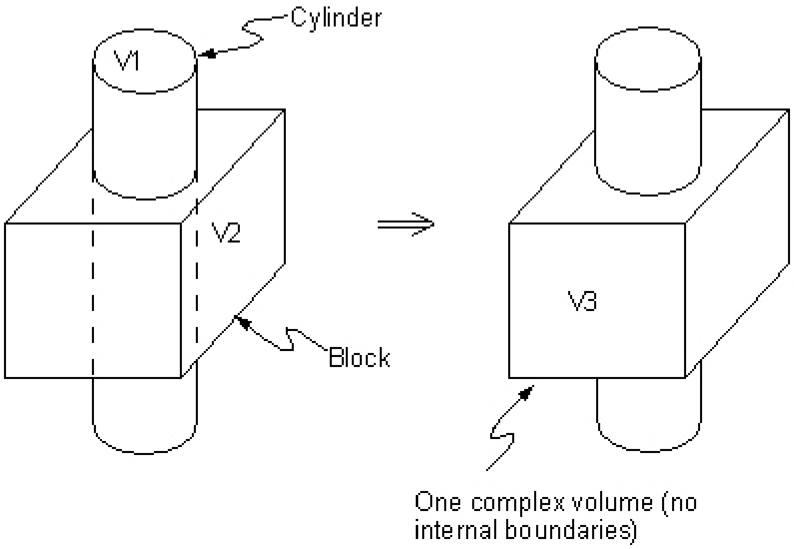


Booleans - Add

AADD (Add Areas)

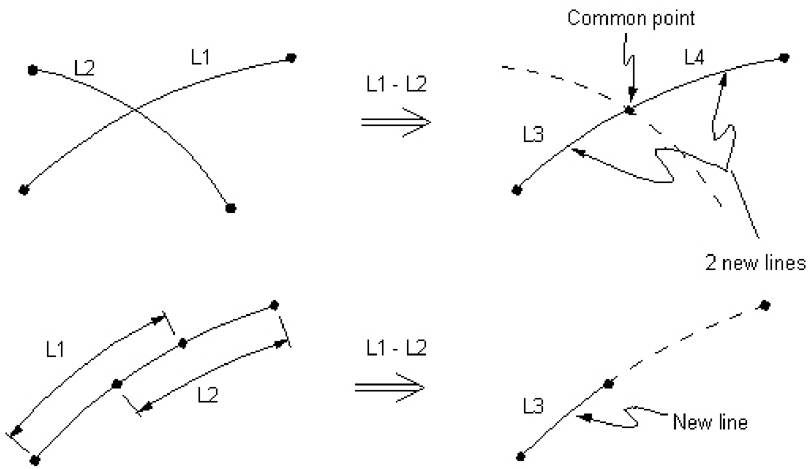


VADD (Add Volumes)

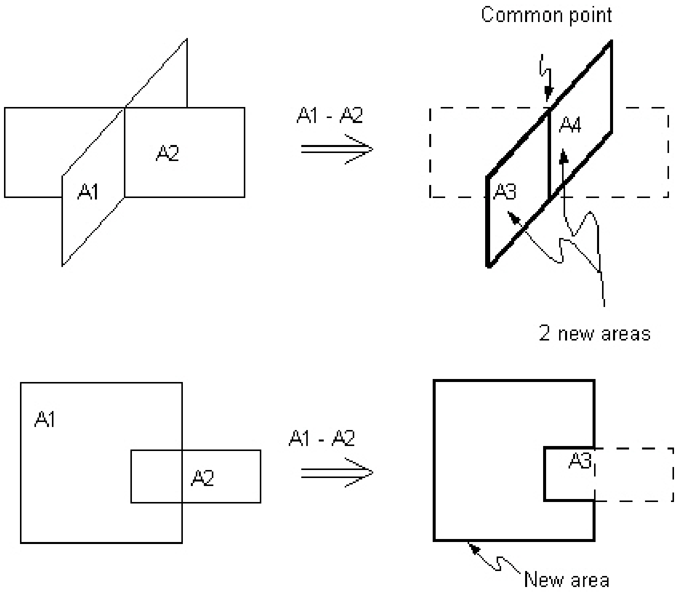


Booleans - Subtract

LSBL (Line Subtract Line)

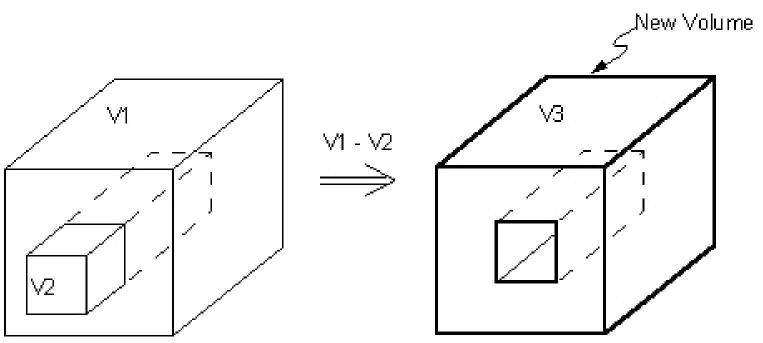


ASBA (Area Subtract Area)

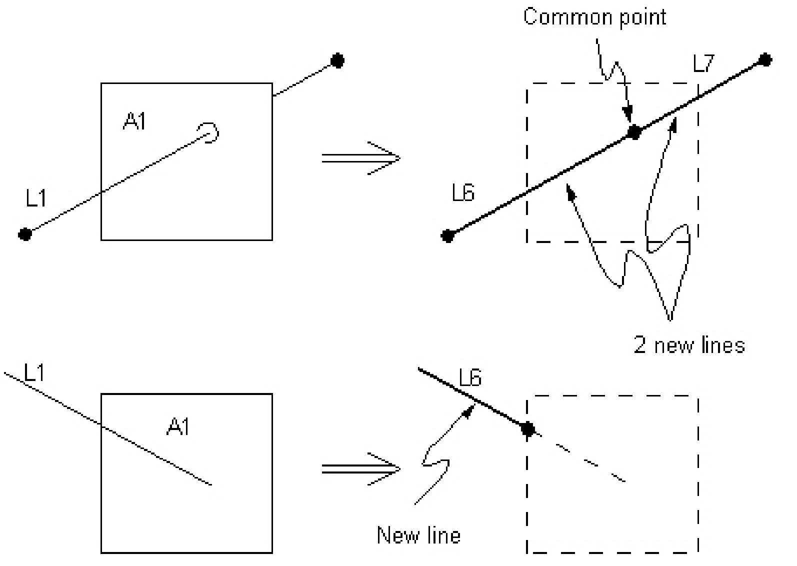


Booleans - Subtract

VSBV (Volume Subtract Volume)

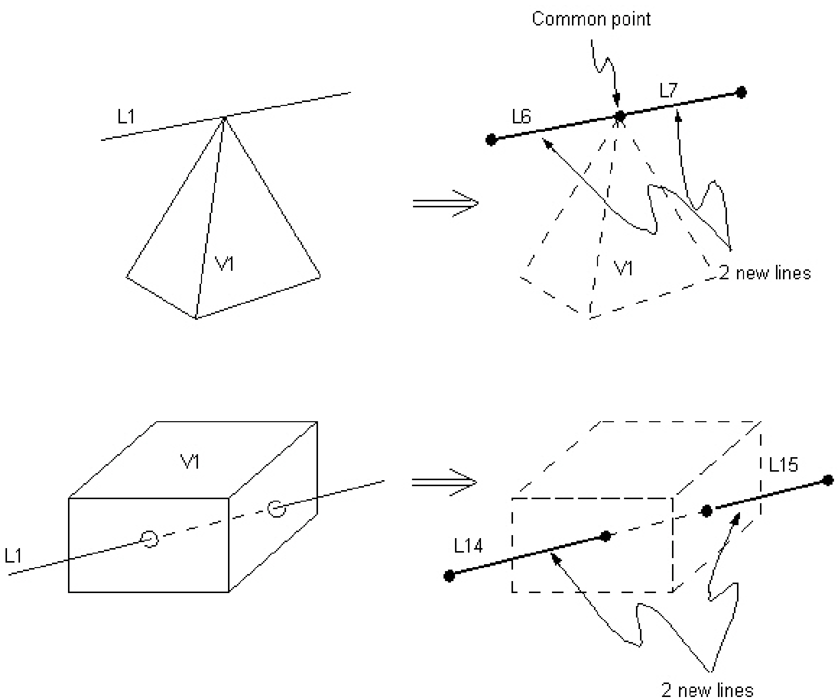


LSBA (Line Subtract Area)

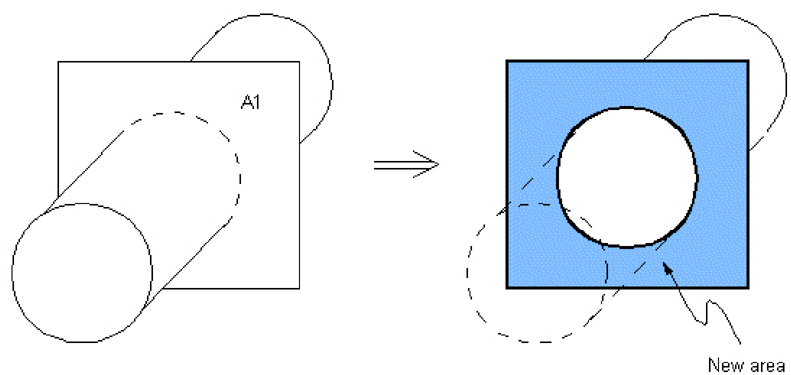


Booleans - Subtract

LSBV (Line Subtract Volume)

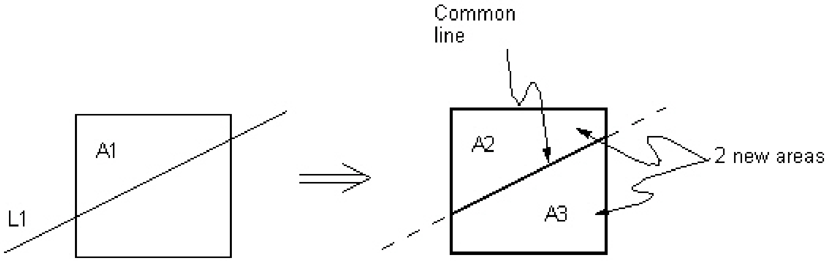


ASBV (Area Subtract Volume)

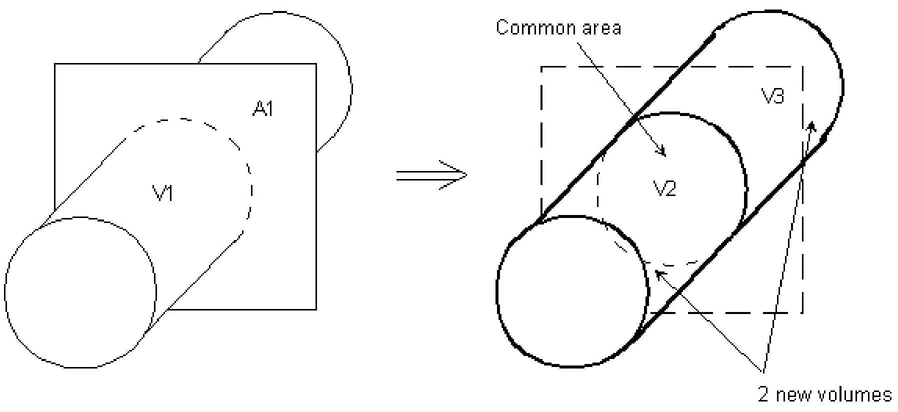


Booleans - Subtract

ASBL (Area Subtract Line)

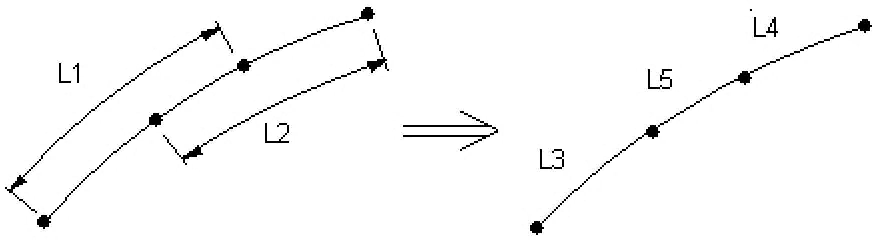


VSBA (Volume Subtract Area)

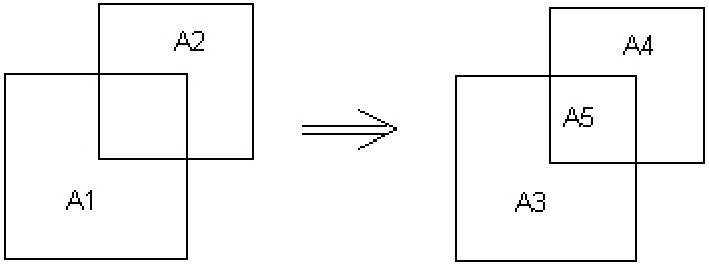


Booleans - Overlap

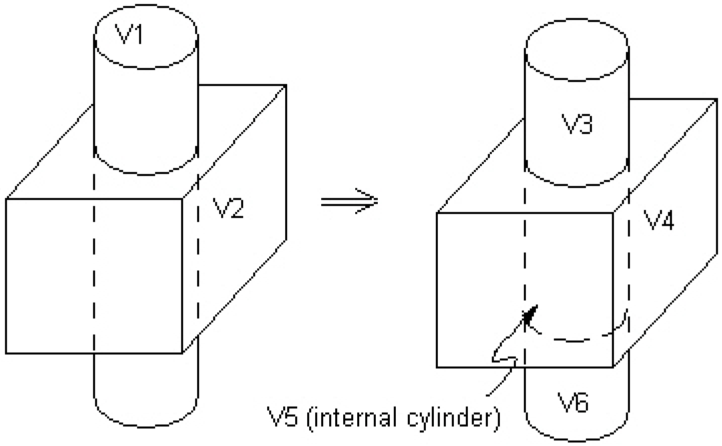
LOVLAP (Line Overlap Line)



AOVLAP (Area Overlap Area)

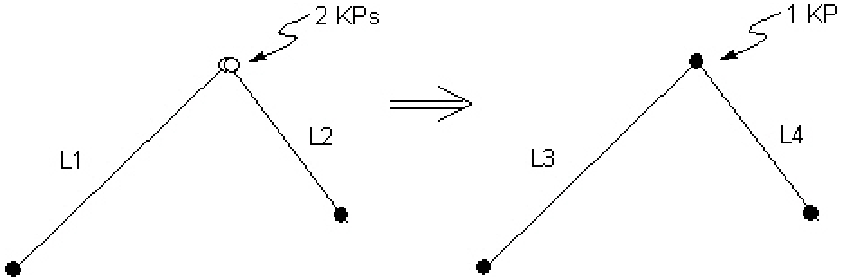


VOVLAP (Volume Overlap Volume)

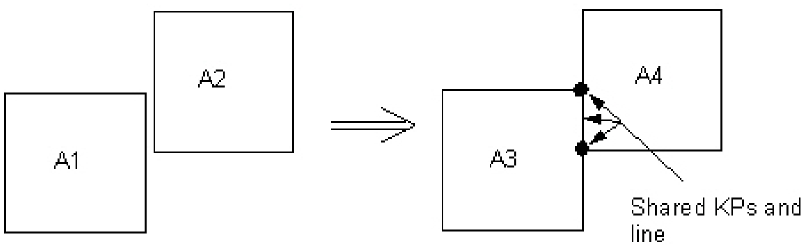


Booleans - Glue

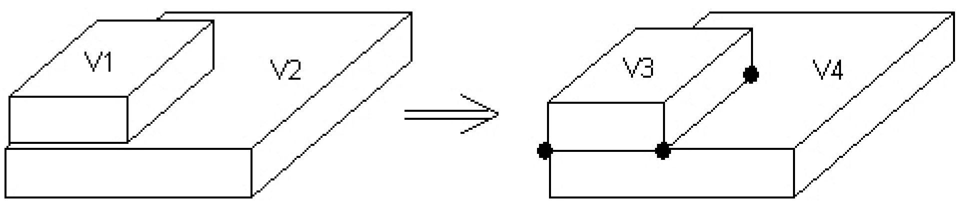
LGLUE (Line Glue Line)



AGLUE (Area Glue Area)



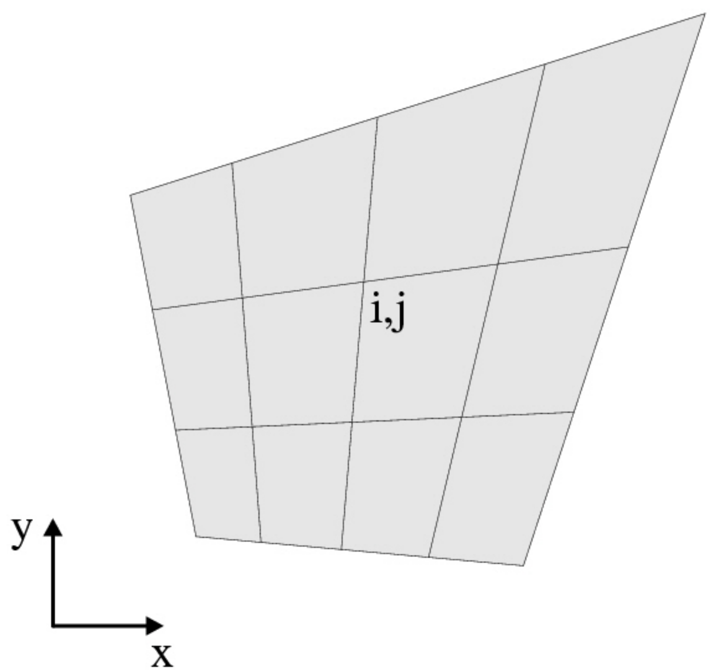
VGLUE (Volume Glue Volume)



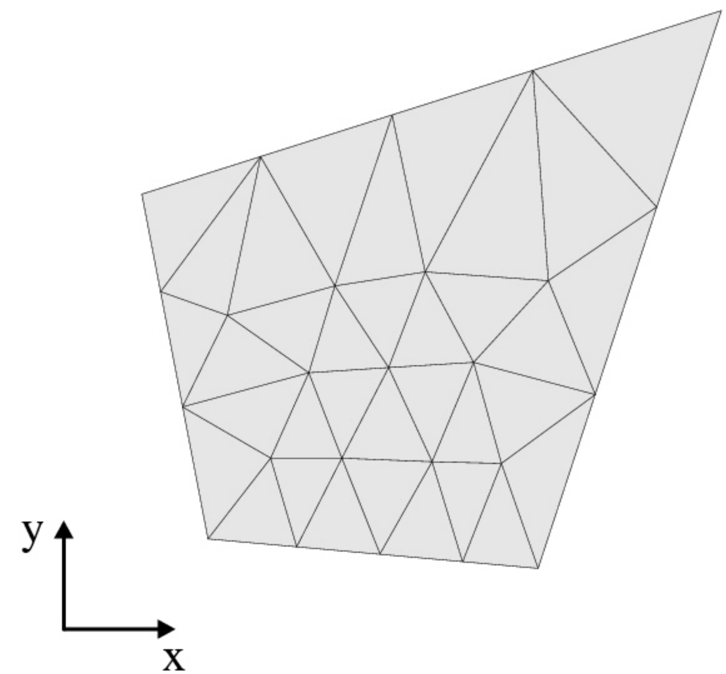
V3 and V4 share 4 keypoints,
4 lines, and an area

Mesh Generation Approaches

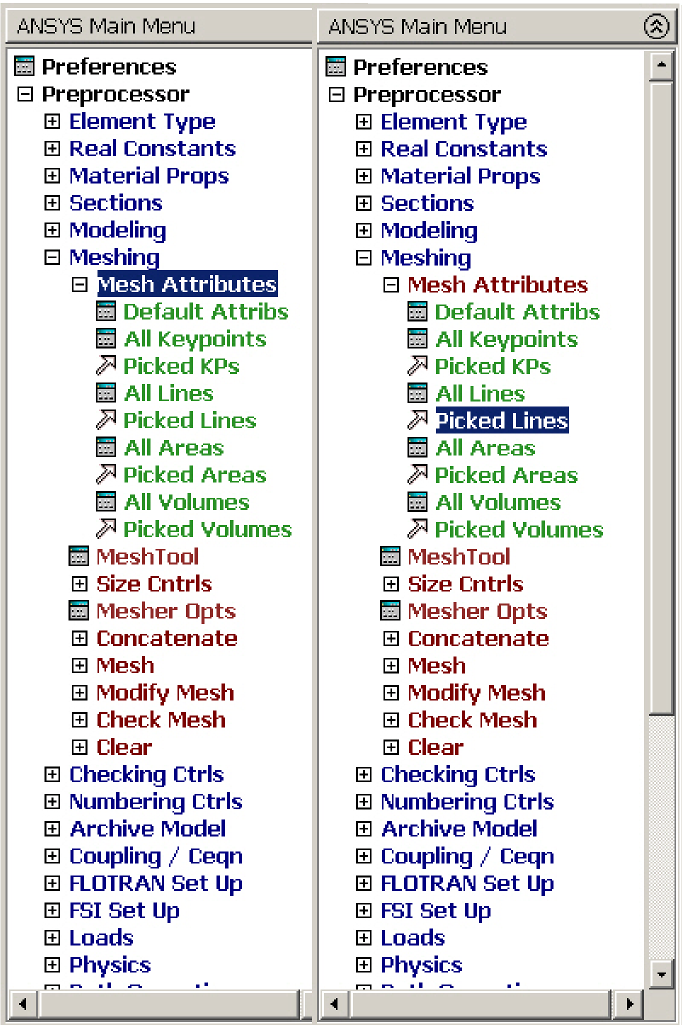
Structured discretization
Mapped meshing

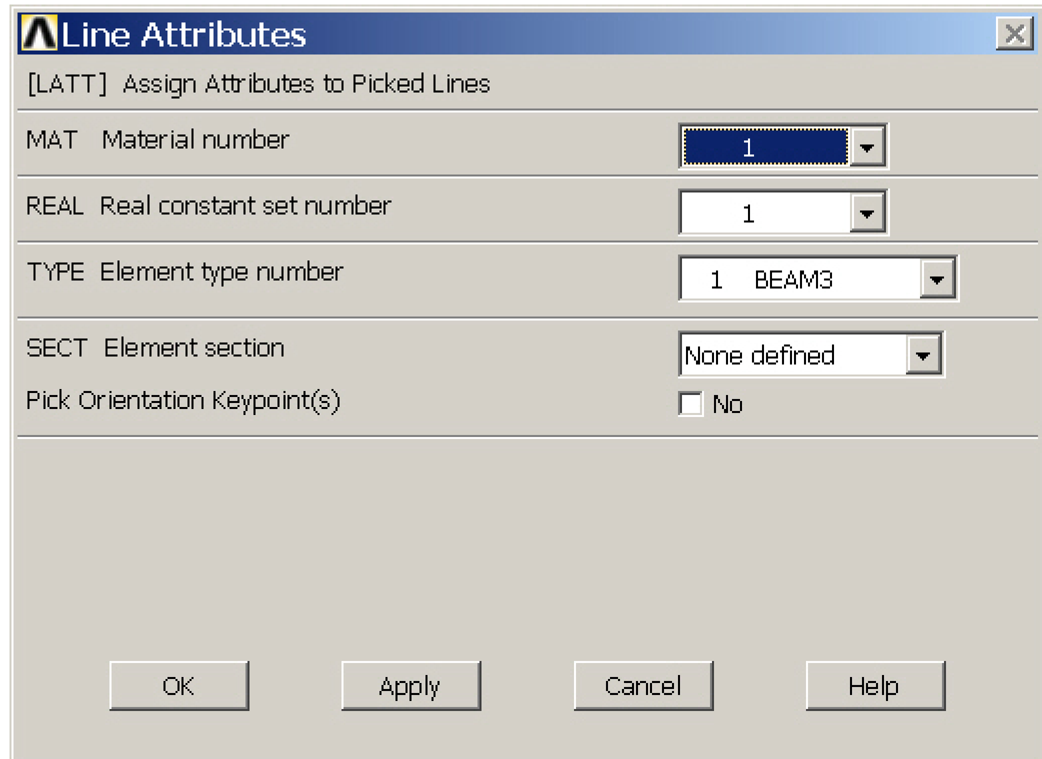


Unstructured discretization
Free meshing

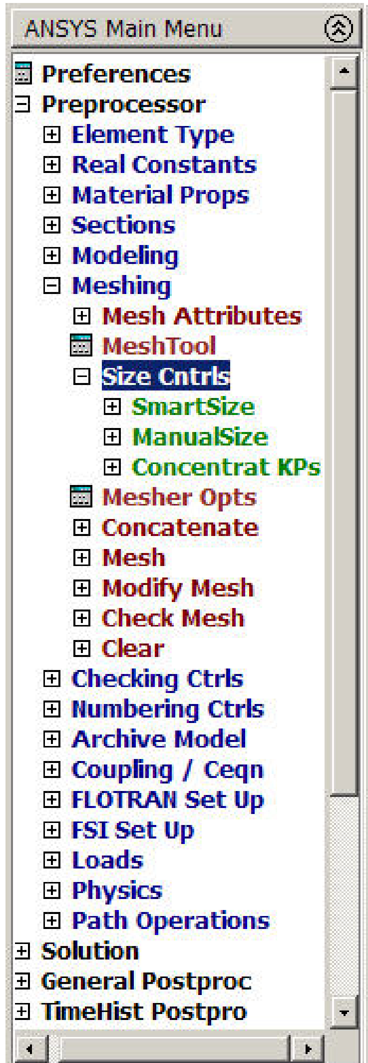


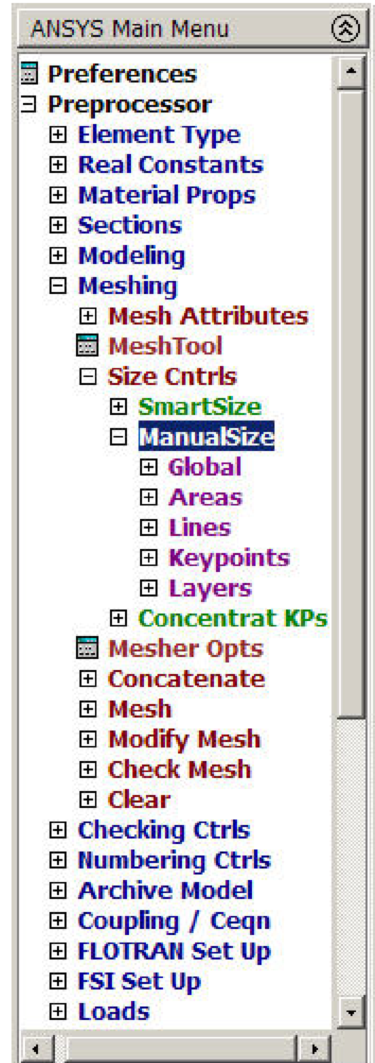
Mesh Attributes



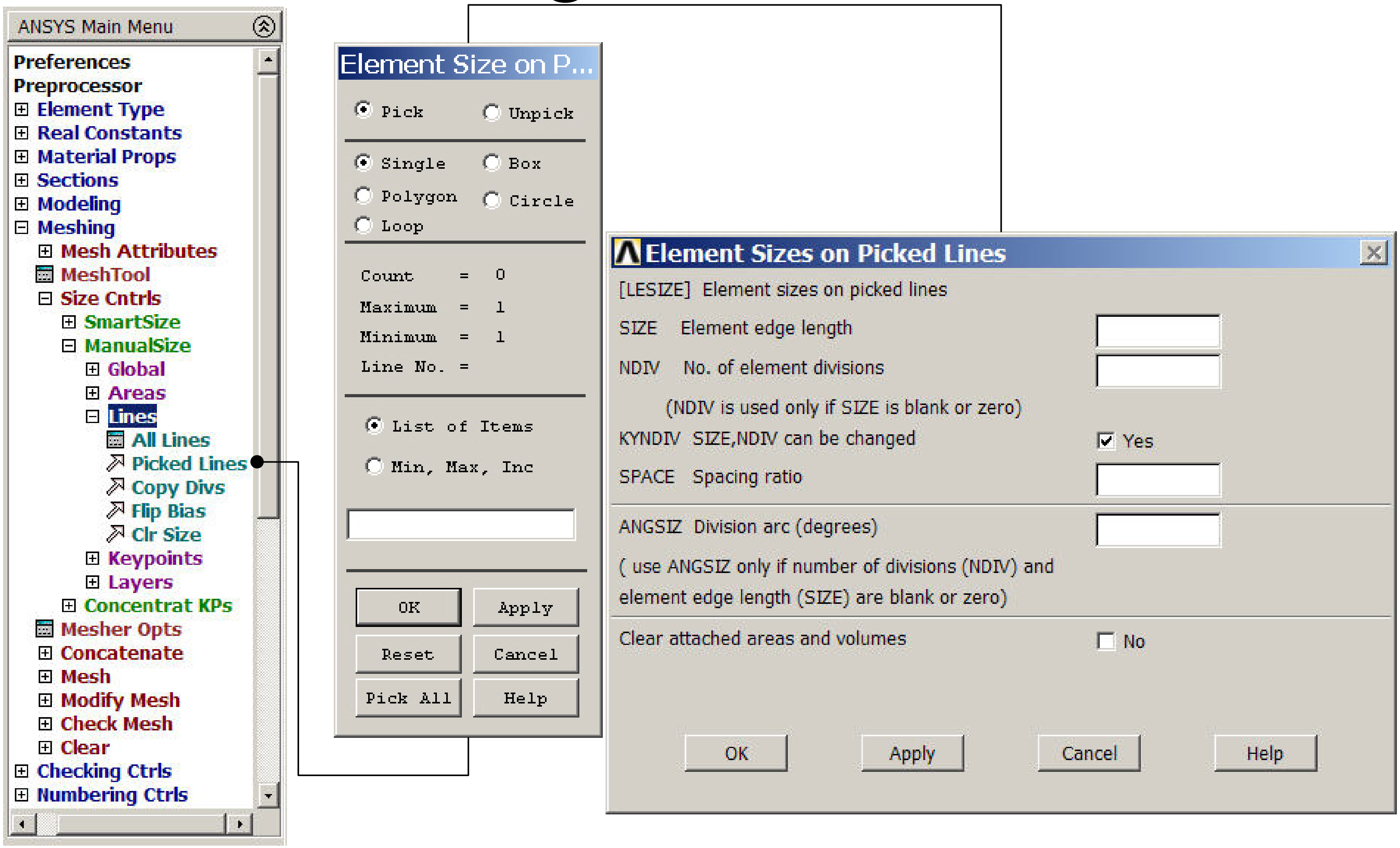


Meshing – Size Cntrls

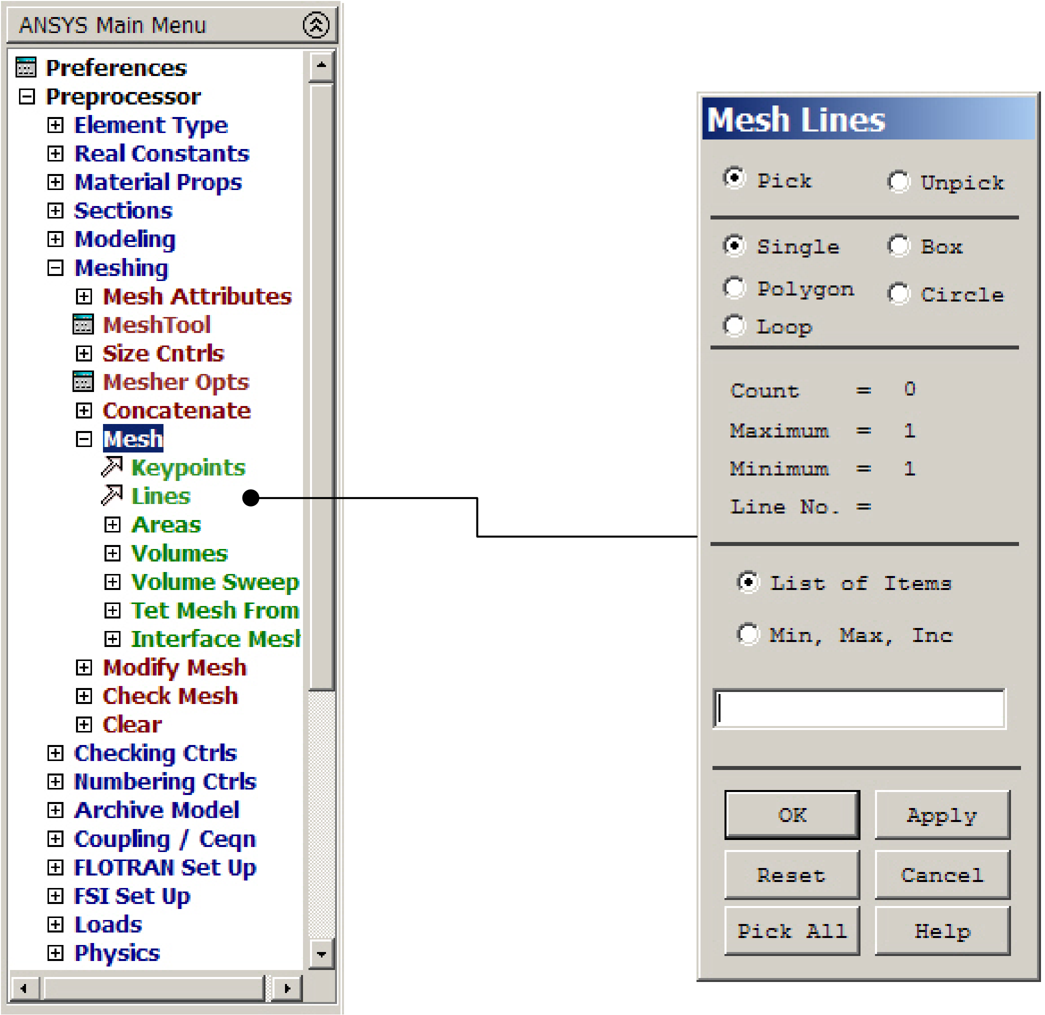




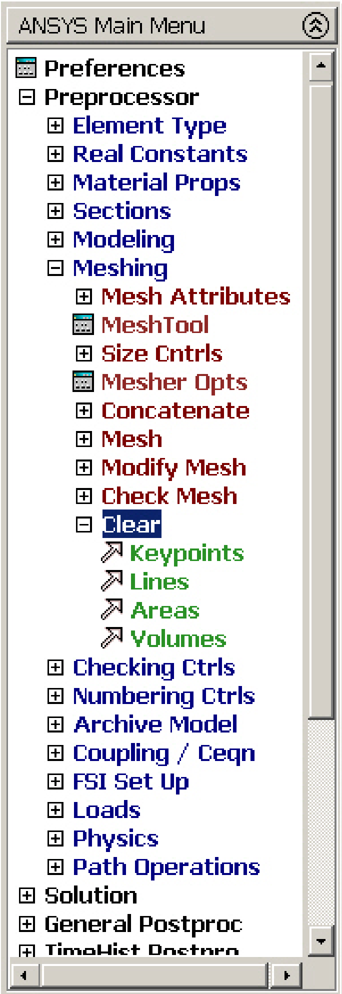
Meshing - ManualSize



Meshing - Lines



Meshing - Clear



Deletes nodes and area elements associated with selected lines, areas, or volumes.