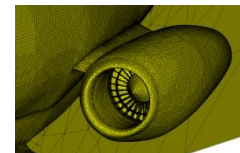# Using ANSYS finite element package

Dr. Anna A. Nasedkina

nasedkina@math.sfedu.ru

Institute of Mathematics,

Mechanics and Computer Science

Southern Federal University
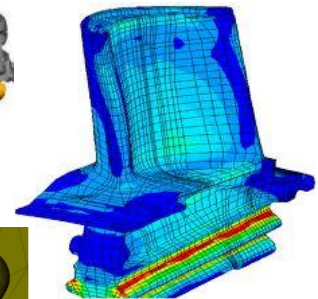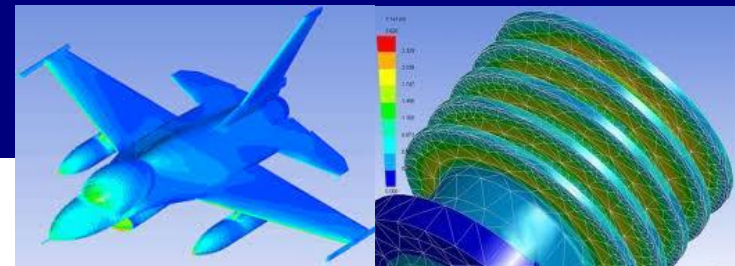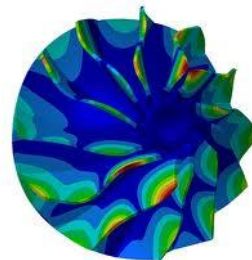
Outline

- Product Launcher
- Graphical User Interface
- ANSYS file system and database
- Commands and input files
- Solid modeling: top-down, bottom-up, boolean operations
- Defining element types, element key options
- Meshing methods: free and mapped, mesh density control
- Defining material properties, use of units
- Applying different types of boundary conditions: symmetry, force, pressure
- Viewing and listing results
- Post-processing for transient analysis

How to launch ANSYS Mechanical APDL

- **Use Product Launcher**
- **Specify Simulation Environment, License, Working Directory and Job Name**
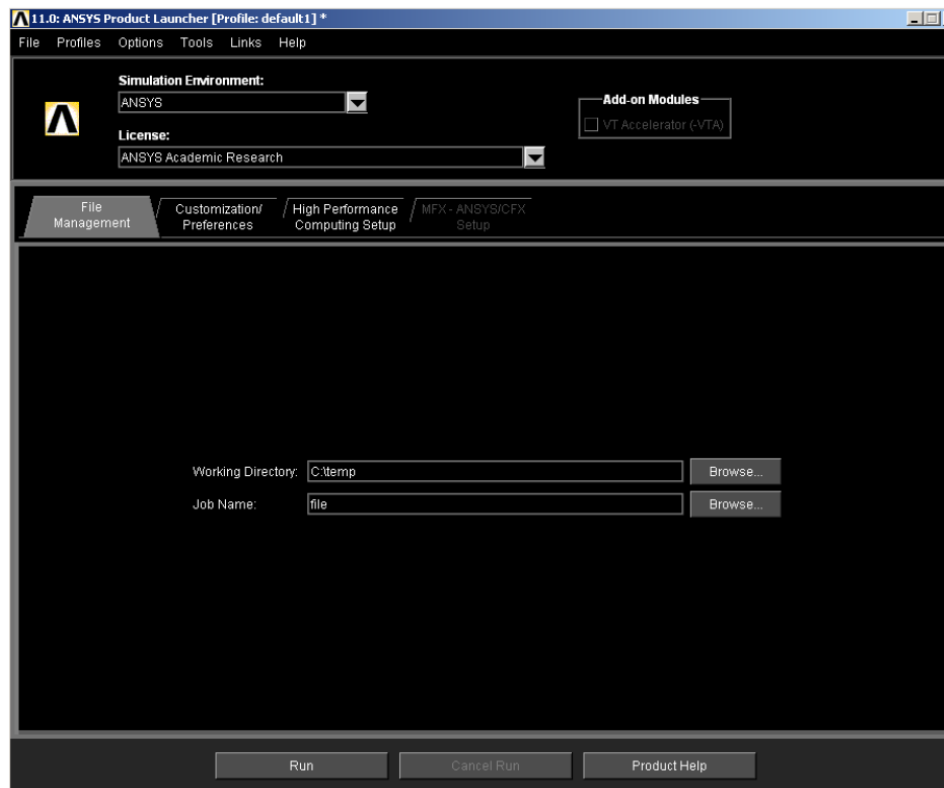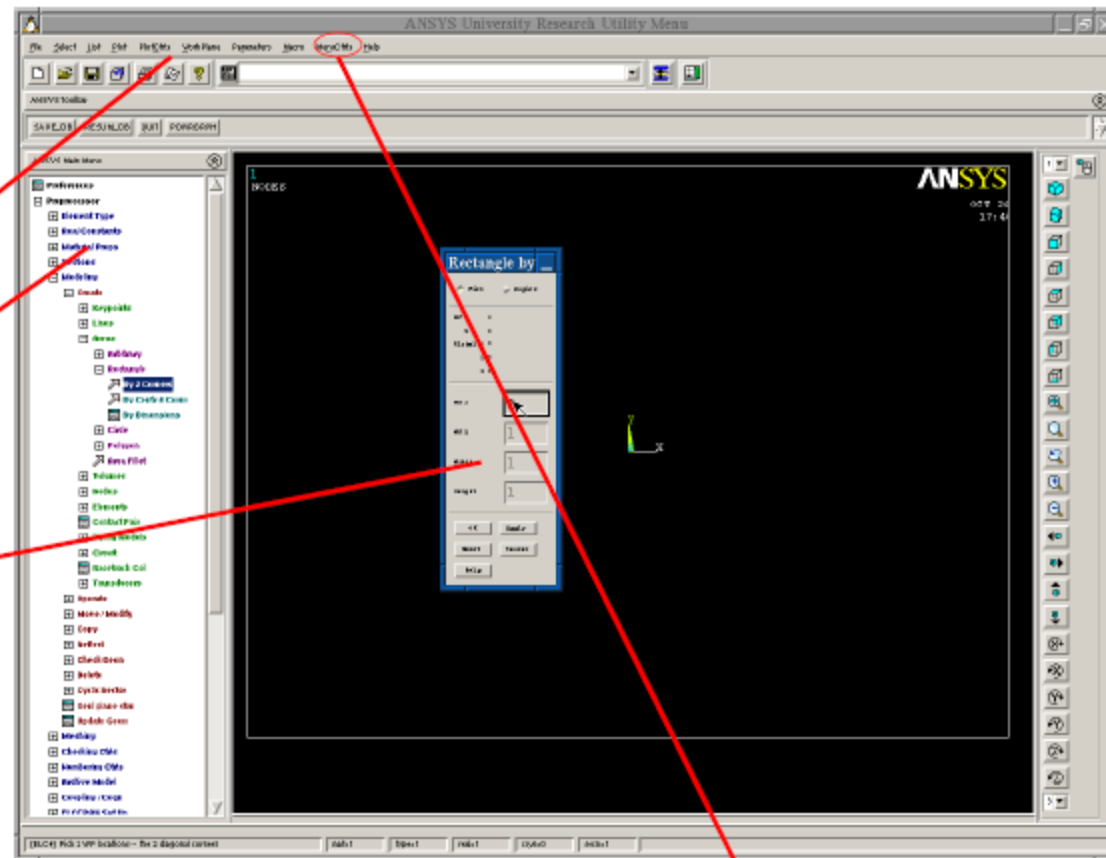
ANSYS graphical user interface
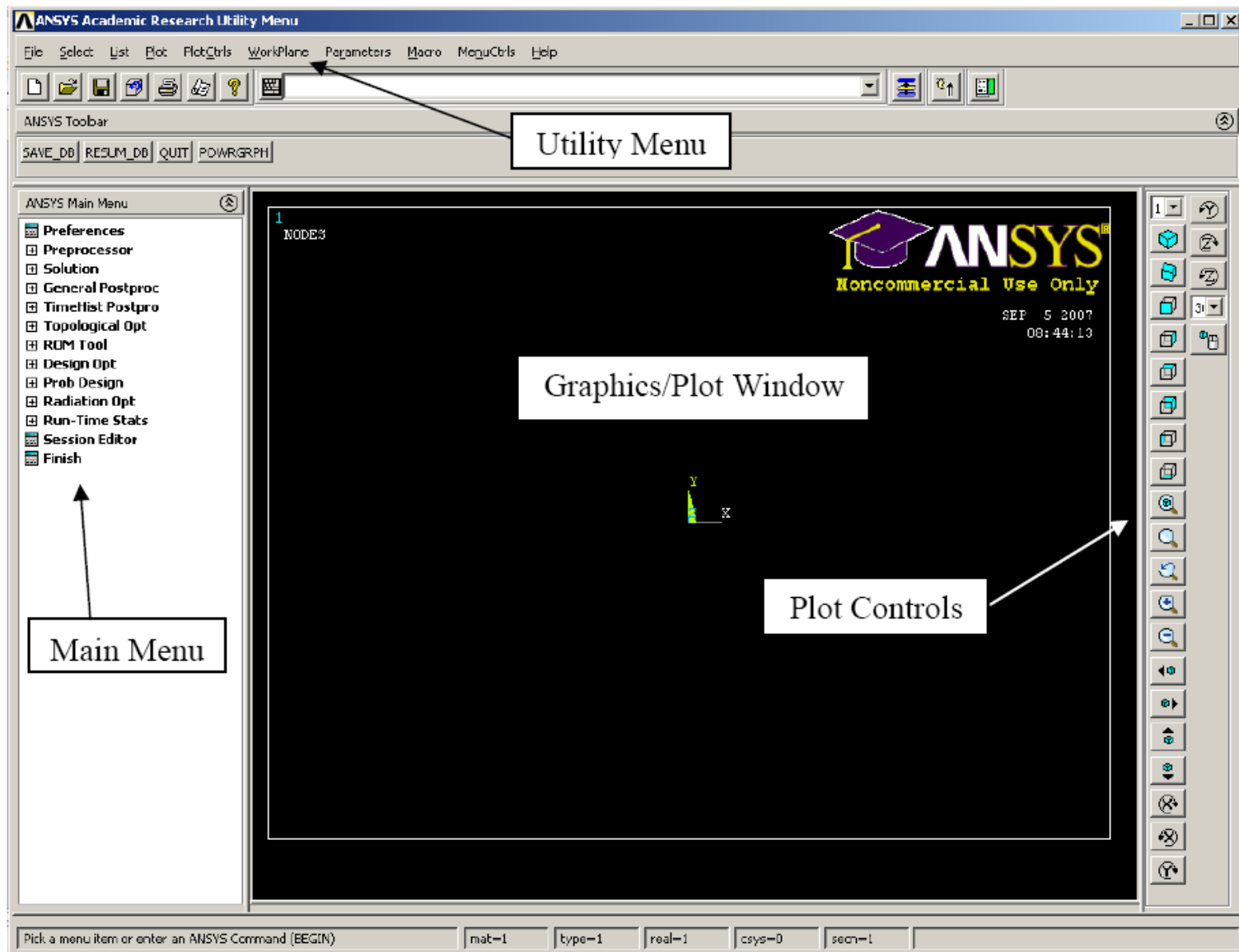
ANSYS graphical user interface

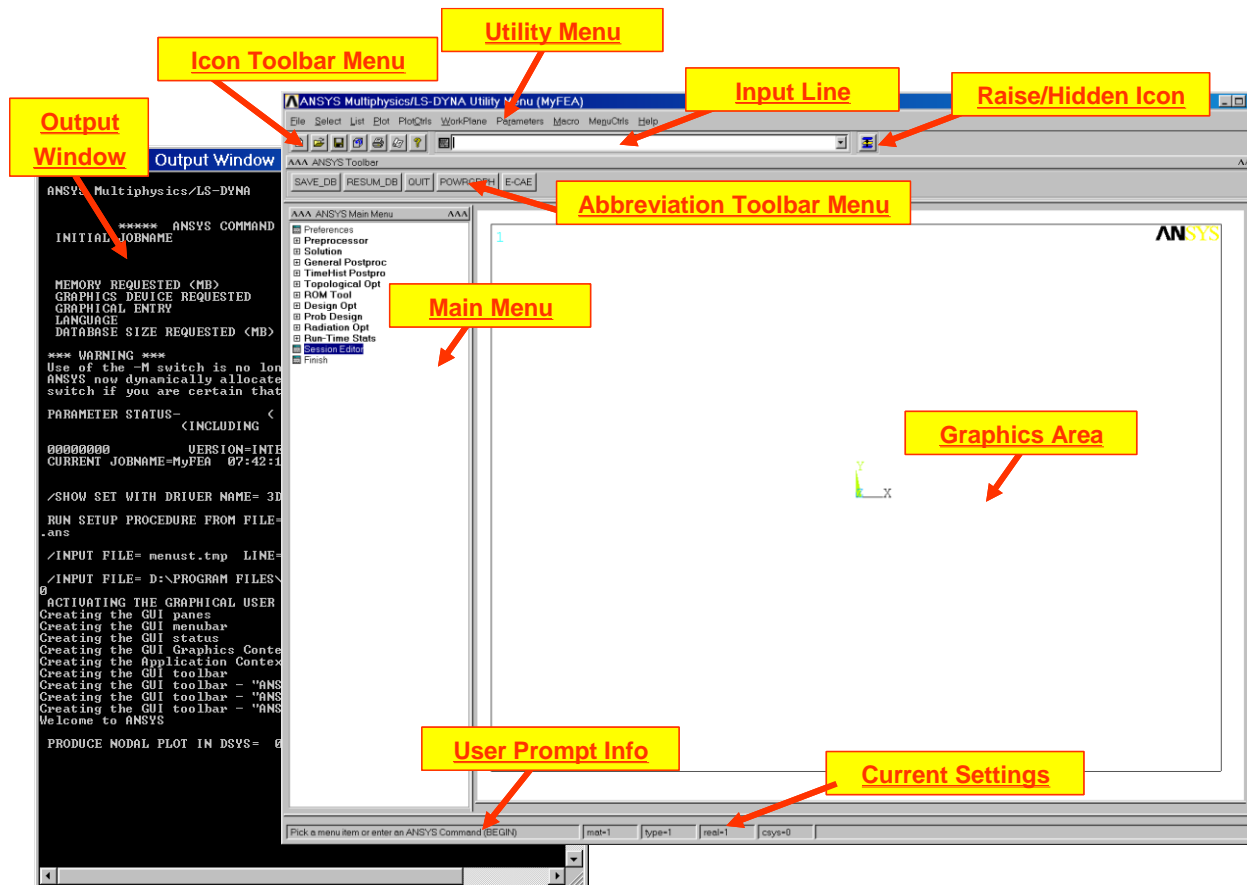You can do things graphically using the menus

at the top

and the side

of the screen

This will often bring up a dialog box

or ask you to pick things with the cursor

NOTE: You can change font size using the *MenuCtrls > Font Selection* menu

ANSYS graphical user interface and output window

Initial parameters

Set initial parameters:

1) **Utility Menu → File → Change Jobname…** or **Change Directory…**(In case you forgot when you started ANSYS.)

2) **Utility Menu → File → Change Title…** (This step is not necessary, but it is a good idea. The Title will appear on plots and in other output files.)

3) **ANSYS Main Menu → Preferences →** Set the **Preferences for GUI Filtering** (This step is not necessary, but it is a good idea. Commands not related to a particular analysis type will be filtered out of the GUI. In this class, we will perform Structural and Thermal analyses. Also, we will generally use the h-method Discipline option.)
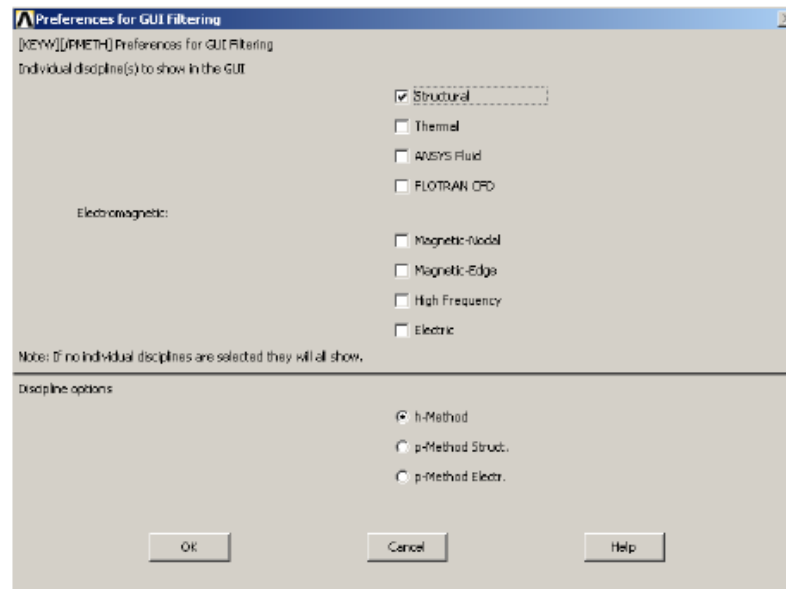
ANSYS file system

❖ **Database and Files:** The term ANSYS database refers to the data ANSYS maintains in memory as you build, solve, and postprocess your model. The database stores both your input data and ANSYS results data:
– **Input data** -- info you must enter, such as dimensions, material properties, and load data.
– **Results data** -- quantities that ANSYS calculates, such as displacements and stresses.

❖ **Typical files in Ansys**
jobname.db, .dbb: Database file, binary. Compatible across all supported platforms.
jobname.log: Log file, ASCII. Contains a log of every command issued during the session. If you start a second session with the same jobname in the same working directory, ANSYS will *append* to the previous log file (with a time stamp).
jobname.err: Error file, ASCII. Contains all errors and warnings encountered during the session. ANSYS will also *append* to an existing error file.
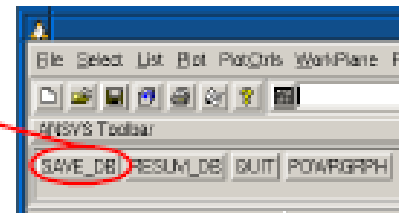*jobname*.rst, .rth, .rmg, .rfl: Results files, binary. Contains results data calculated by ANSYS during solution. Compatible across all supported platforms.

ANSYS database

- You can save ANSYS models as a database file

This can be done here

or with the SAVE command

- Once you have done this if you list the directory contents it now contains a file: *file.db*

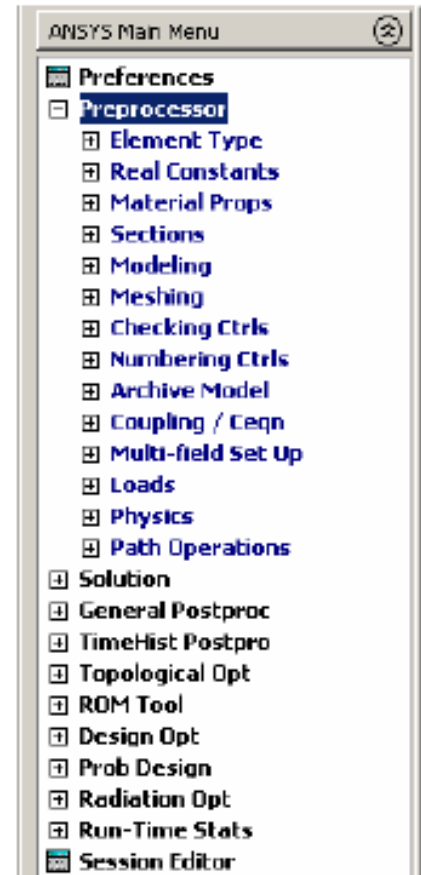- You can resume from your database each time you start ANSYS

File > Resume from OR RESUME

Overview of simulation stages in ANSYS

- **<u>Preprocessor</u>**
- **Solution**
- **Postprocessor**
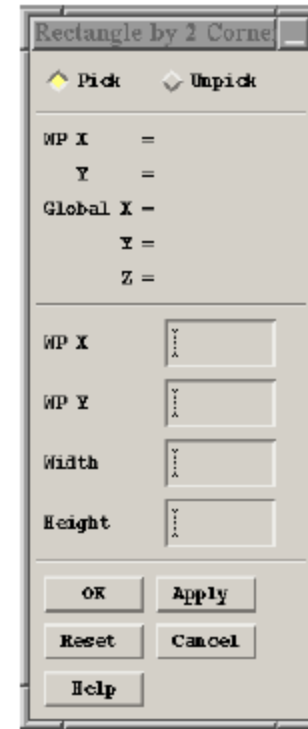
ANSYS Main Menu → Preprocessor

1) Define **Element Type**
2) Define **Real Constants** (if necessary)
3) Define **Material Properties**
4) Create the geometric **Model** (define in ANSYS or import geometry from Pro/E or SolidWorks)
5) Create a **Mesh** for the model

ANSYS Main Menu

- Preferences
- Preprocessor
  - Element Type
  - Real Constants
  - Material Props
  - Sections
  - Modeling
  - Meshing
  - Checking Ctrls
  - Numbering Ctrls
  - Archive Model
  - Coupling / Ceqn
  - Multi-field Set Up
  - Loads
  - Physics
  - Path Operations
- Solution
- General Postproc
- TimeHist Postpro
- Topological Opt
- ROM Tool
- Design Opt
- Prob Design
- Radiation Opt
- Run-Time Stats
- Session Editor
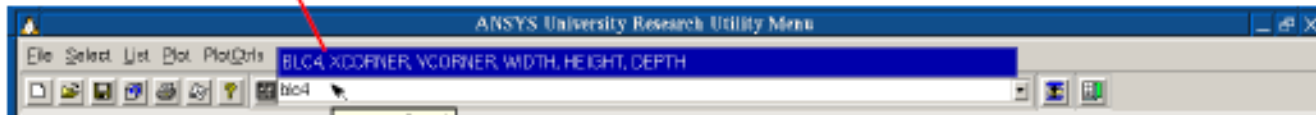
Graphical input

- Try it! From the side menu select:

Preprocessor > Modeling > Create > Areas > Rectangle > By 2 corners

- This should bring up this dialog box

- WP X and WP Y are the coords of the bottom left corner of the rectangle

- Type a value for these and the width and height and click OK

- You should get a cyan rectangle

Rectangle by 2 Corne

◇ Pick   ◇ Unpick

WP X   =
   Y   =
Global X =
     Y =
     Z =

WP X

WP Y

Width

Height

OK   Apply

Reset   Cancel

Help

Using commands in ANSYS

- **Command line mode can be used instead of interactive mode**

- **Commands are written in APDL – ANSYS Parametric Design Language**

- If you type a command in the command box, ANSYS will tell you what it expects as input data

- Typing the command in has the same effect as running it graphically and the command will also be stored in the log file as before

- To get help, simply type *help* in the command box, or use the help menu to select *Help Topics*

ANSYS input files

■ **Input file is a text file .inp или .txt**

- You can programme in ANSYS using input files
- These are just text files containing a list of ANSYS commands:

```
!plate: ANSYS input file for plate with hole
/FILNAM,plate ! Name the file
/PREP7 ! Enter the preprocessor
ET,1,PLANE42 ! Define the element type - 4 node solid element
UIMP,1,EX, , ,200000, ! Define material properties
K,1,2.5,0,, ! Define keypoints
LARC,1,2,11,2.5, ! Define arcs
LSTR, 1, 4 ! Define straight lines
AL,3,6,4,1 ! Define areas
lesize,3,,,15 ! Define mesh size
```

- Purple text is comment (designated by '!') for info only

Input files versus graphical input

- They are small, large models can be built with only a few commands
- They allow you to review what you've done easily
- They allow you to modify models very quickly

I have built and meshed a square in ANSYS, now I want to change the length of the model

Graphically

Either: Modify keypoints and then
remesh the areas of the model

Or: Start again from scratch

File > Clear and start new

With an input file

Change 2 values in input file and run again:

BLC4, *val1* , *val2* , *val3* , *val4*

File > Read input from

Parameterization in input files

- Parameterisation can be very useful

```
/PREP7
ET,1,42

!Define the parameters of my problem
xlength = 10
ylength = 20

xmesh = 5
ymesh = 15

!Use the parameters to define geometry and mesh the model
BLC4, -xlength/2 , -ylength/2 , xlength , ylength
LSEL,s,loc,x,0
LESIZE,all,,,xmesh
LSEL,s,loc,y,0
LESIZE,all,,,ymesh

AMESH,all
```
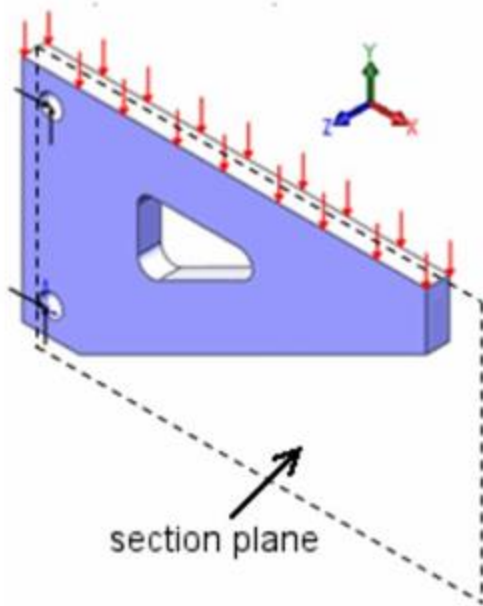
We can define the mesh and geometry using just these 4 values

3D and 2D simulation

Cases of 2D simulation:

- **Plane stress**
- **Plane strain**
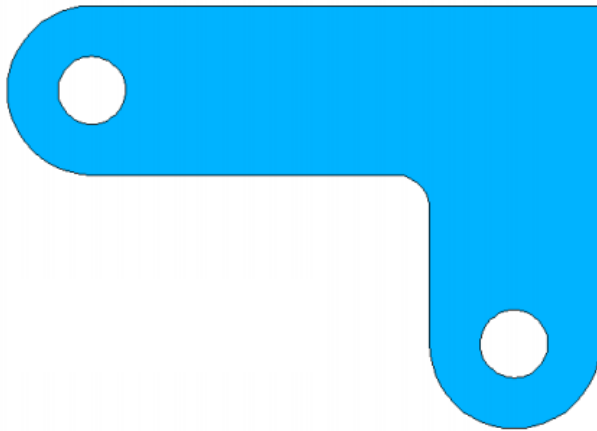- **Axial symmetry**
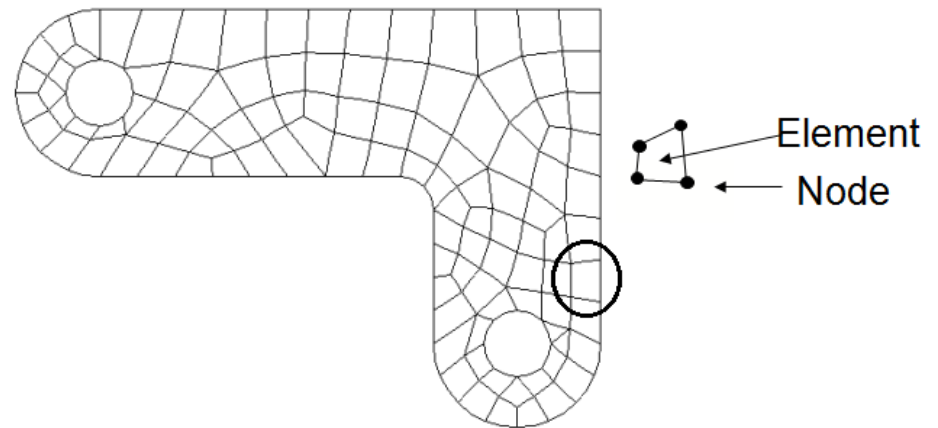
Plane Stress
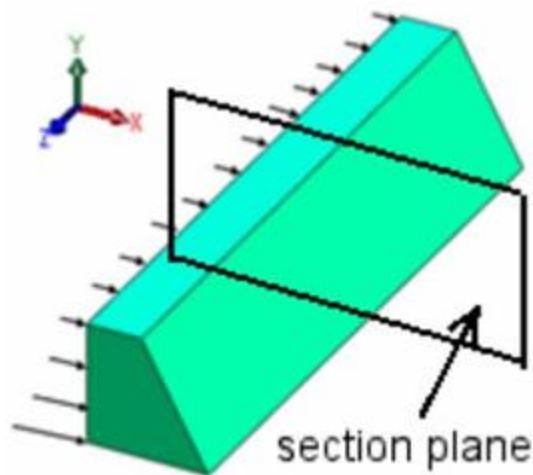
Plane Strain
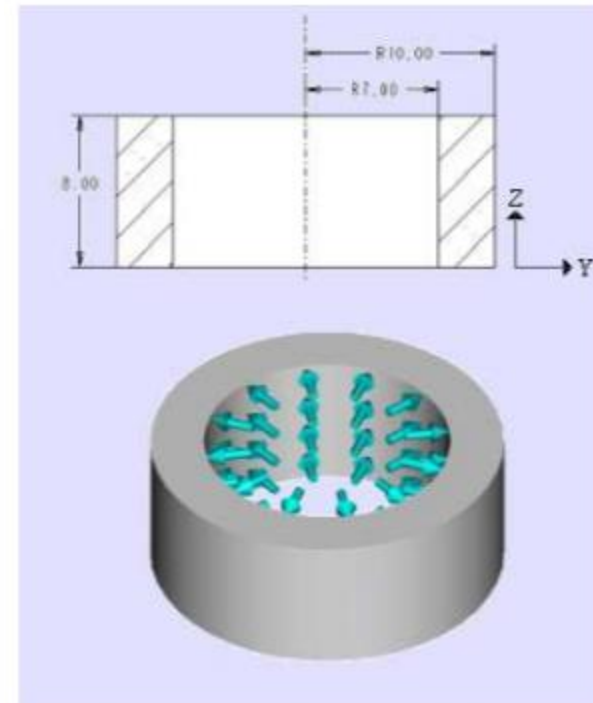
Axisymmetric

Solid and finite element simulation

- **Solid model**
- **Finite element model**

- Solid model of a bracket
- Finite element model of a bracket

Element
Node

Solid modeling in ANSYS

❖ **<u>Solid Modeling</u>**

Be defined as the process of creating solid models in CAD system.
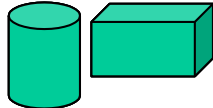
❖ **<u>Definitions</u>**

–A *solid model* is defined by volumes, areas, lines, and keypoints.

–Volumes are bounded by areas, areas by lines, and lines by keypoints.

–Hierarchy of entities from low to high as

   keypoints < lines < areas < volumes

–You cannot delete an entity if a higher-order entity is attached to it. Also, a model with just areas and below, such as a shell or 2-D plane model, is still considered a solid model in ANSYS terminology.
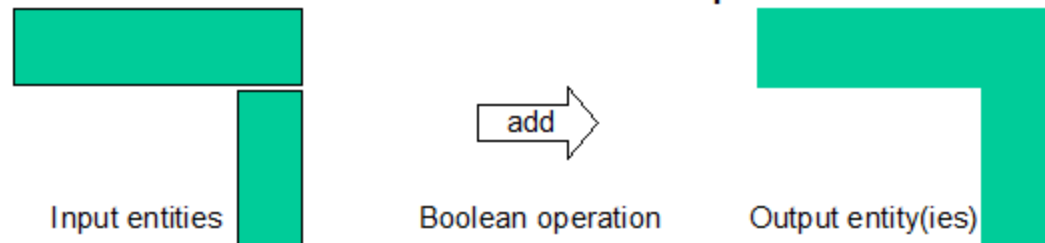
Volumes	
Areas	
Lines	
Keypoints	

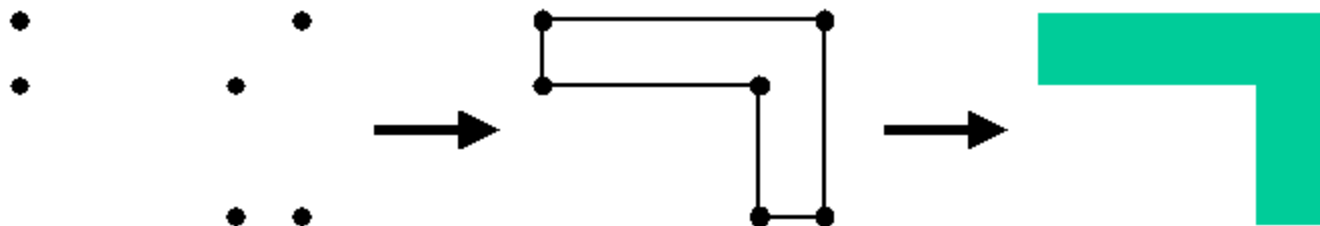# Methods of solid modeling in ANSYS

- **Top-down**
- **Bottom-up**

❖ **Top-down modeling**
starts with a definition of volumes (or areas), which are then combined in some fashion to create the final shape.

Input entities | Boolean operation | Output entity(ies)

add

❖ **Bottom-up modeling**
starts with keypoints, from which you "build up" lines, areas, etc.

ANSYS primitives

The volumes or areas that you initially define are called *primitives*, which are basic entities for the top-down method. ANSYS contains the following 2D and 3D *primitives:*

Rectangle   Circle   Polygon

Block   Cylinder   Prism   Sphere   Cone   Torus
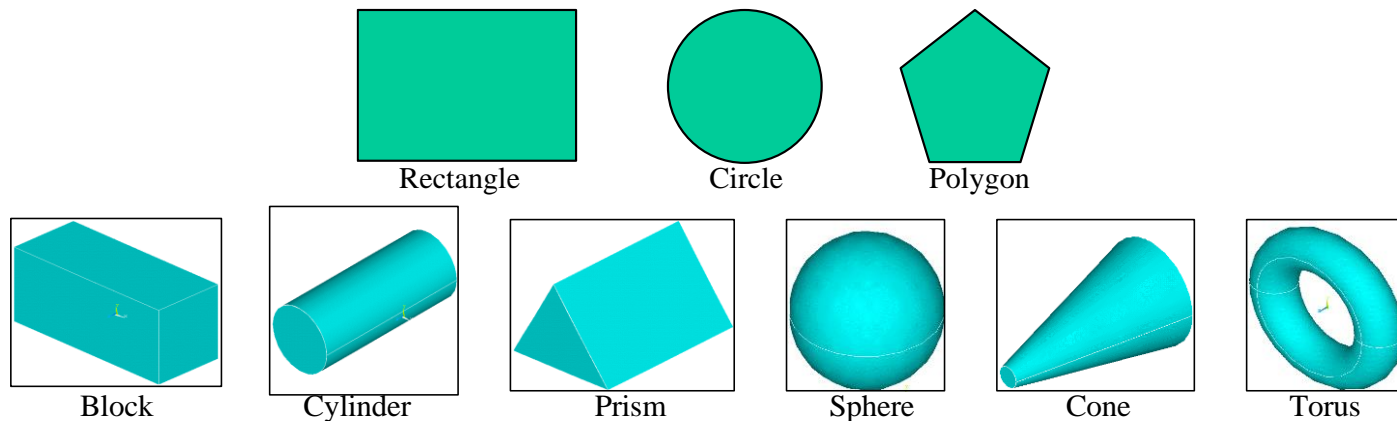
Boolean operation in top-down method

The final shape of an object is usually not as simple as primitives. However, it is likely doable to combine a number of primitives through a series of proper *Boolean operations*. The "input" to Boolean operations can be any geometric entity, ranging from simple primitives to complicated volumes generated in previous steps.
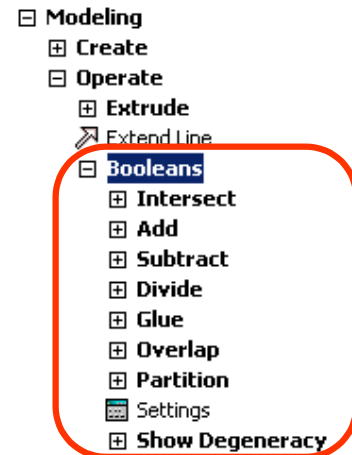
❖ **Boolean operations** are computations involving combinations of geometric entities. ANSYS Boolean operations include add, subtract, intersect, divide, glue, and overlap. •All Boolean operations are available in the GUI:

`Main Menu > Preprocessor > Modeling > Operate > Booleans`

Examples of some boolean operations

❖ **Add:** Combines two or more entities into one.

❖ **Glue:** Attaches two or more entities by creating a common boundary between them, which is uesful when you want to maintain the distinction between entities (such as for different materials).

❖ **Overlap:** Same as glue, except that the input entities overlap each other.

❖ **Subtract:** Removes the overlapping portion of one or more entities from a set of "base" entities, which can be useful for creating holes or trimming off portions of an entity.
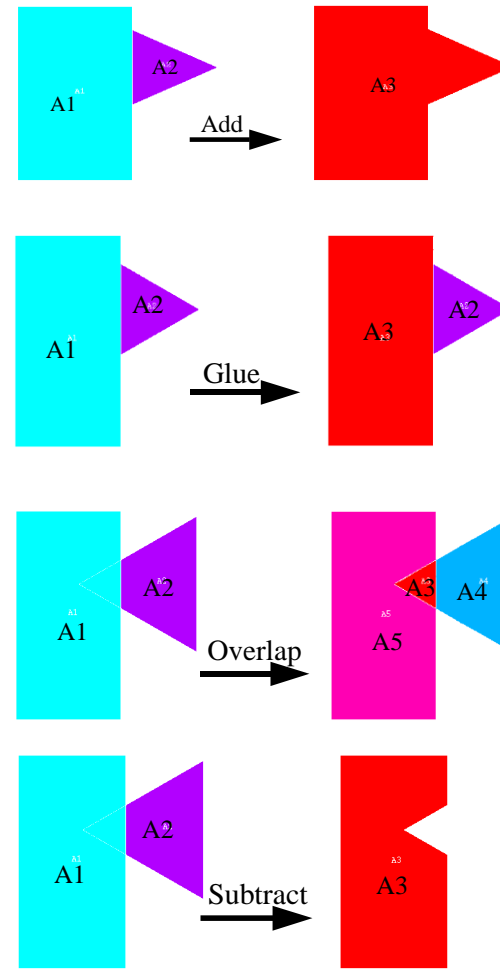
Defining element types in ANSYS

<u>Element Type</u>

The element type is an important choice that determines the following element characteristics:

- Degree of Freedom (DOF) set. A thermal element type, for example, has one dof: TEMP, whereas a structural element type may have up to six dof: UX, UY, UZ, ROTX, ROTY, ROTZ.
- Element shape -- brick, tetrahedron, quadrilateral, triangle, etc.
- Dimensionality -- 2-D solid (X-Y plane only), or 3-D solid.
- Assumed displacement shape -- linear vs. quadratic.

To define an element:

```
Main Menu>Preprocessor>Element Type> Add/Edit/Delete>Add
```
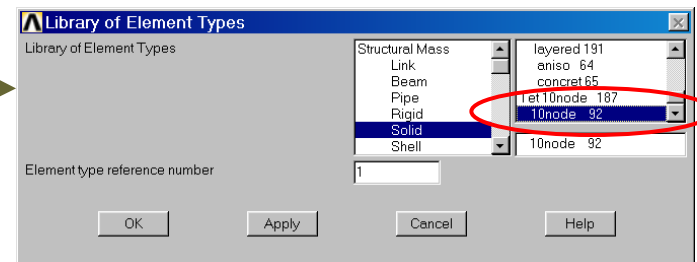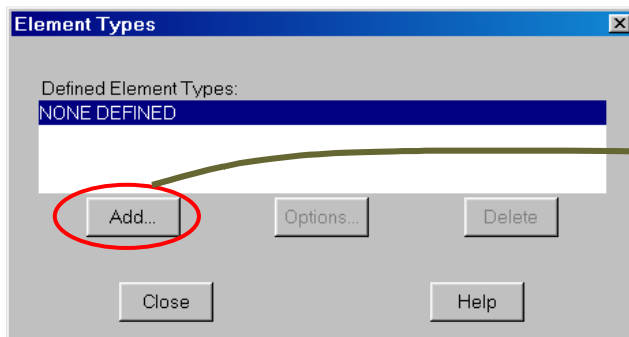
Meshing methods: free and mapped

There are two main meshing methods: Free and Mapped.

Free Mesh – Has no element shape restrictions.

• The mesh does not follow any pattern.

• Suitable for complex shaped areas and volumes.

• Suitable for complex shaped areas and volumes.

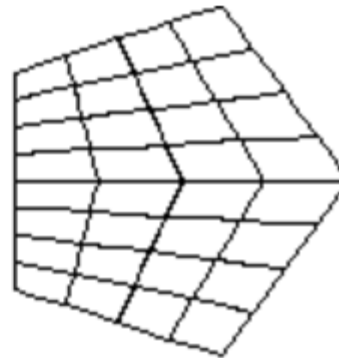• Volume meshes consist of high order tetrahedral (10 nodes), large dof.

Mapped Mesh – Restricts element shapes to quadrilaterals (areas) and hexahedra (volume)

•Typically has a regular pattern with obvious rows of elements.

•Suitable only for "regular" shapes such as rectangles and bricks.
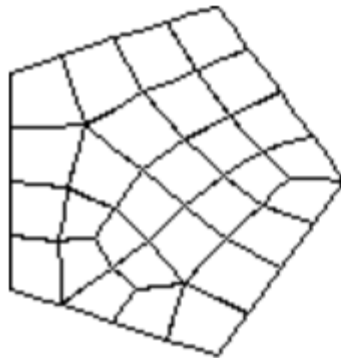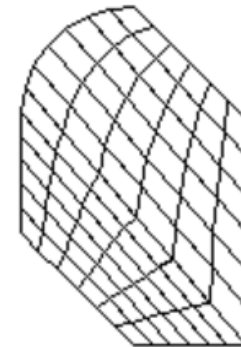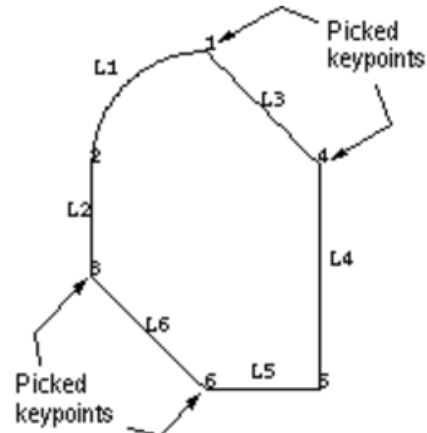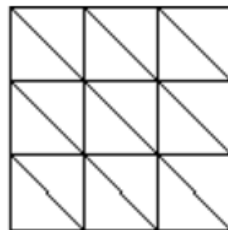
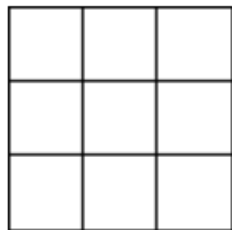Free mesh   Mapped mesh
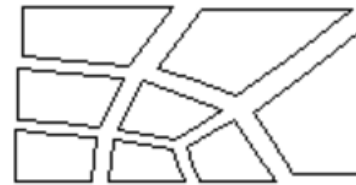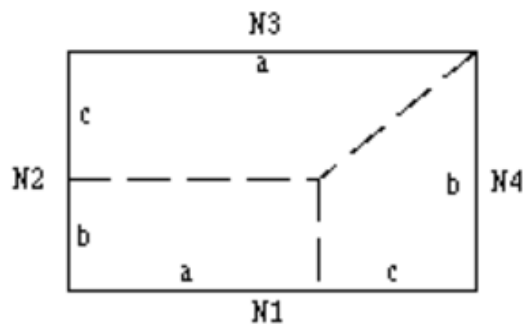
Area mapped mesh

Simplified mapped meshing (AMAP)

Area mapped meshes

Examples of transition mapped quadrilateral meshes

Area mapped mesh: transition mapped mesh
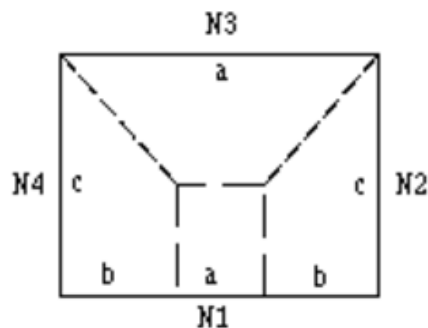

Side 1: has N1 divisions where N1 = a + c
Side 2: has N2 divisions where N2 = b + c
Side 3: has N3 divisions where N3 = a
Side 4: has N4 divisions where N4 = b

(that is, N2−N4 = N1−N3)

For example:

a = 3
b = 2
c = 3

N1 = 6
N2 = 5
N3 = 3
N4 = 2

N2−N4 = N1−N3

Side 1: has N1 divisions where N1 = a + 2*b
Side 2: has N2 divisions where N2 = c
Side 3: has N3 divisions where N3 = a
Side 4: has N4 divisions where N4 = c

(that is, N2=N4 and the difference between N1 and N3 is even)

For example:

a = 3
b = 4
c = 2

N1 = 11
N2 = 2
N3 = 3
N4 = 2

N2=N4 and the difference between N1 and N3 is even

Volume mapped mesh

Opposite edges must have equal number of divisions

(a) Brick Shaped Volume

Edges on sides must have equal number of divisions

Edges on face must have even and equal number of divisions

(b) Prism Shaped Volume

Number of divisions must be equal and even on all edges
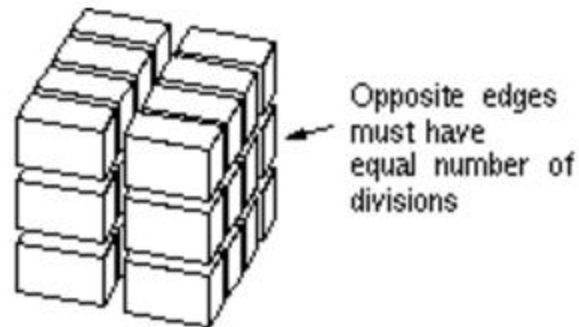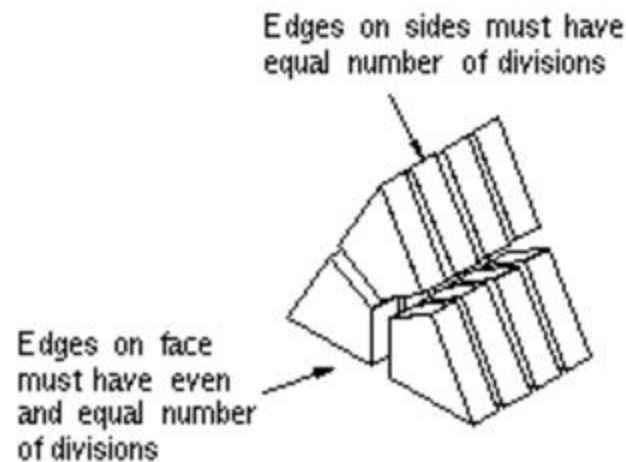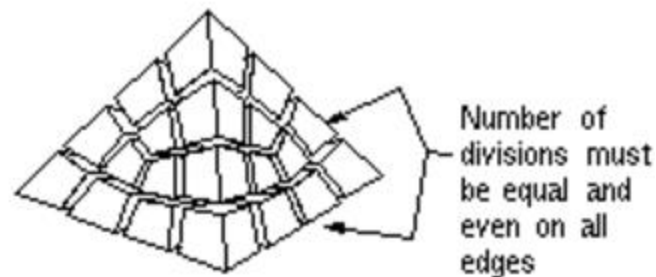
(c) Tetrahedral Shaped Volume

Transition mapped mesh
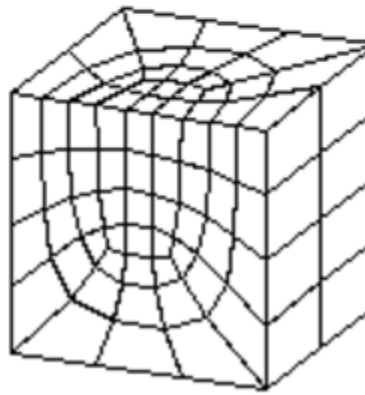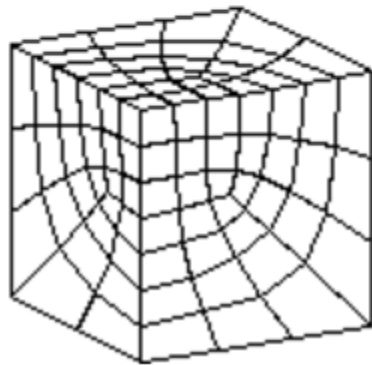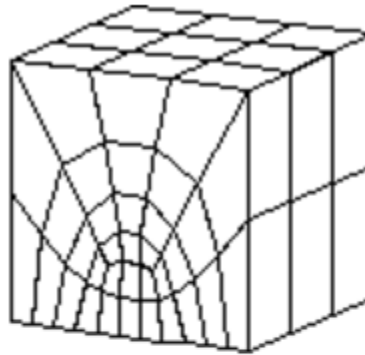
Examples of area free mesh
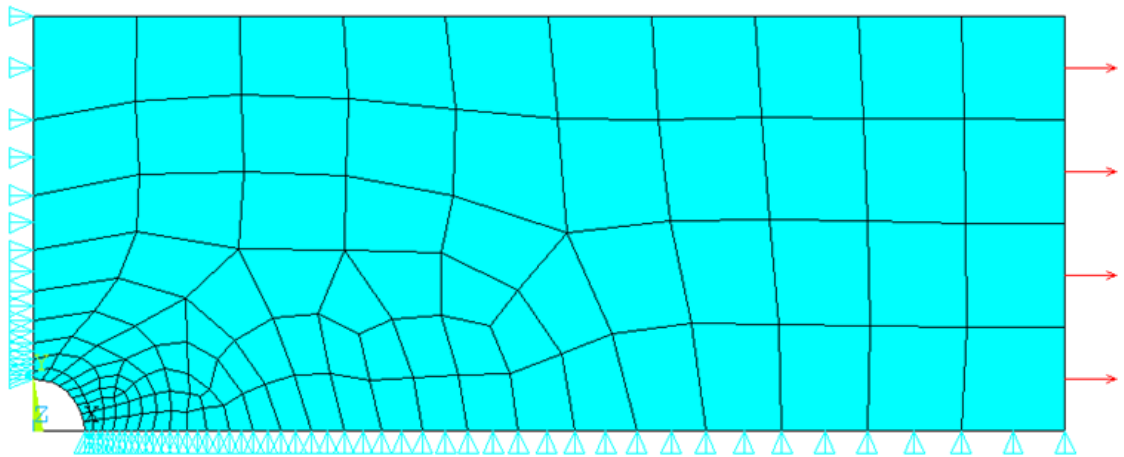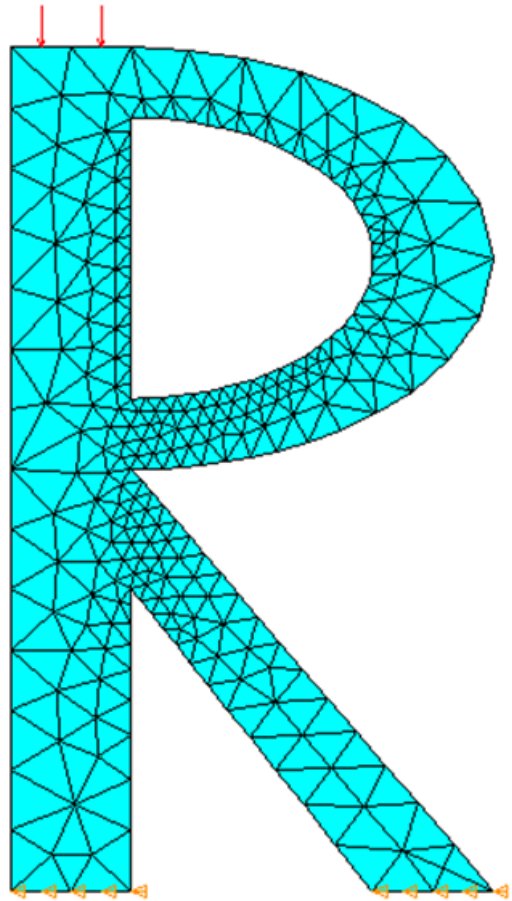
Examples of volume free mesh
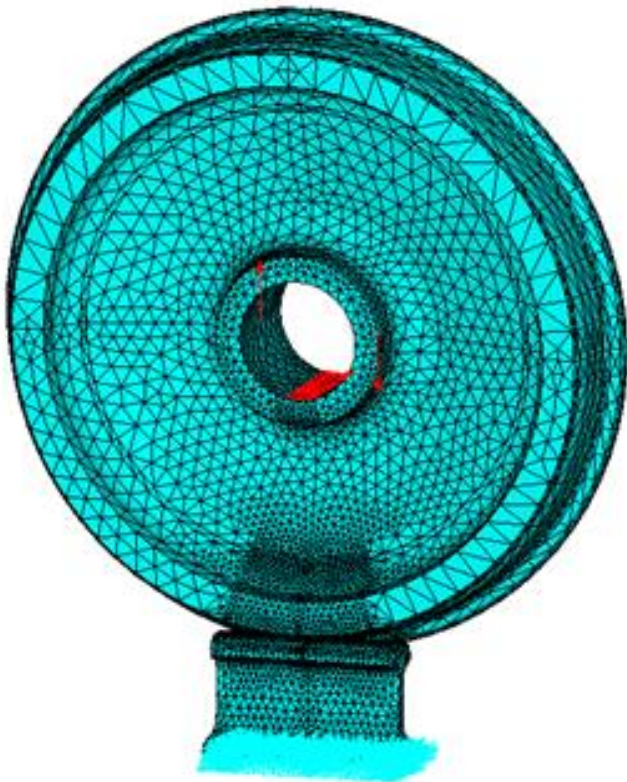
Mesh density control

❖ **Mesh Density Control**

ANSYS provides many tools to control mesh density, on a global and local level:

–Global controls: SmartSizing; Global element sizing; Default sizing

–Local controls: Keypoint sizing; Line sizing; Area sizing

To bring up the MeshTool:

```
Main Menu > Preprocessor > Meshing > MeshTool
```

❖ **SmartSizing:** by turning on SmartSizing, and set the desired size level. Size level ranges from 1 (fine) to 10 (coarse), defaults to 6. Then mesh all volumes (or all areas) at once, rather than one-by-one.Advanced SmartSize controls, such as mesh expansion and transition factors, are available by

```
Main Menu>Preprocessor>Meshing>Size Cntrls>SmartSize>Adv Opts
```
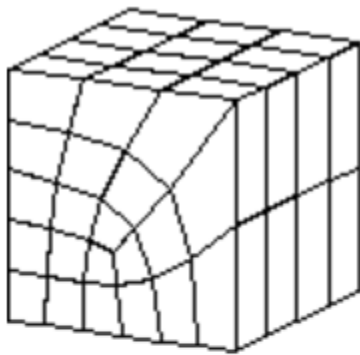
❖ **Global Element Sizing:** Allows you to specify a maximum element edge length for the entire model (or number of divisions per line):

Go to "Size Controls", "Global" ,and click [Set] or

```
Main Menu>Preprocessor>Meshing>Size Cntrls > ManualSize
>Global >Size
```
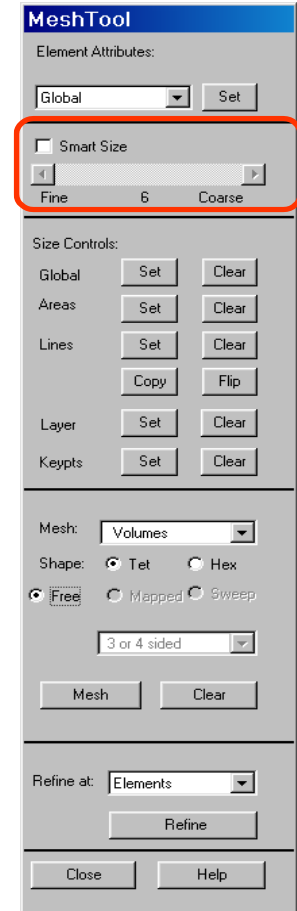
Material properties and units in ANSYS

<u>Material Properties</u>

Every analysis requires *some* material property input: Young's modulus (EX), Poisson's ratio (PRXY) for structural elements, thermal conductivity (KXX) for thermal elements, etc.

To define the material properties:

     `Main Menu>Preprocessor>Material Props>Material Models`
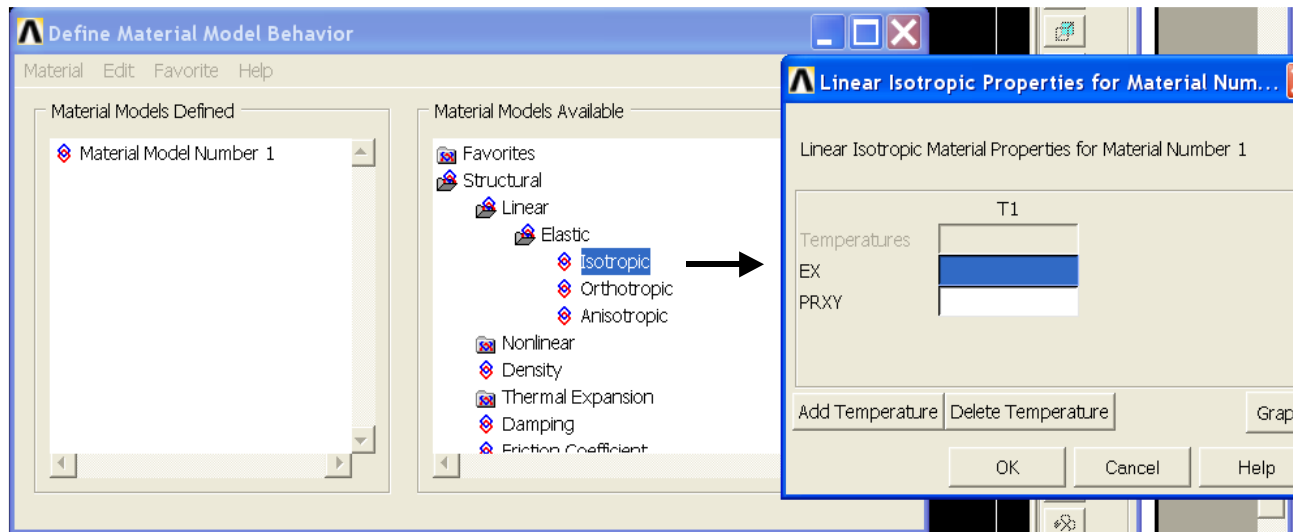
More than one set of material properties can be defined when needed.

❖ Unit – The ANSYS program does not assume a system of units for your analysis. You can use any system of units

Example: structural analysis problem

- **Tension of the plate with a hole**
- 2D (plane stress)
- Use of symmetry: consider ¼ of the plate
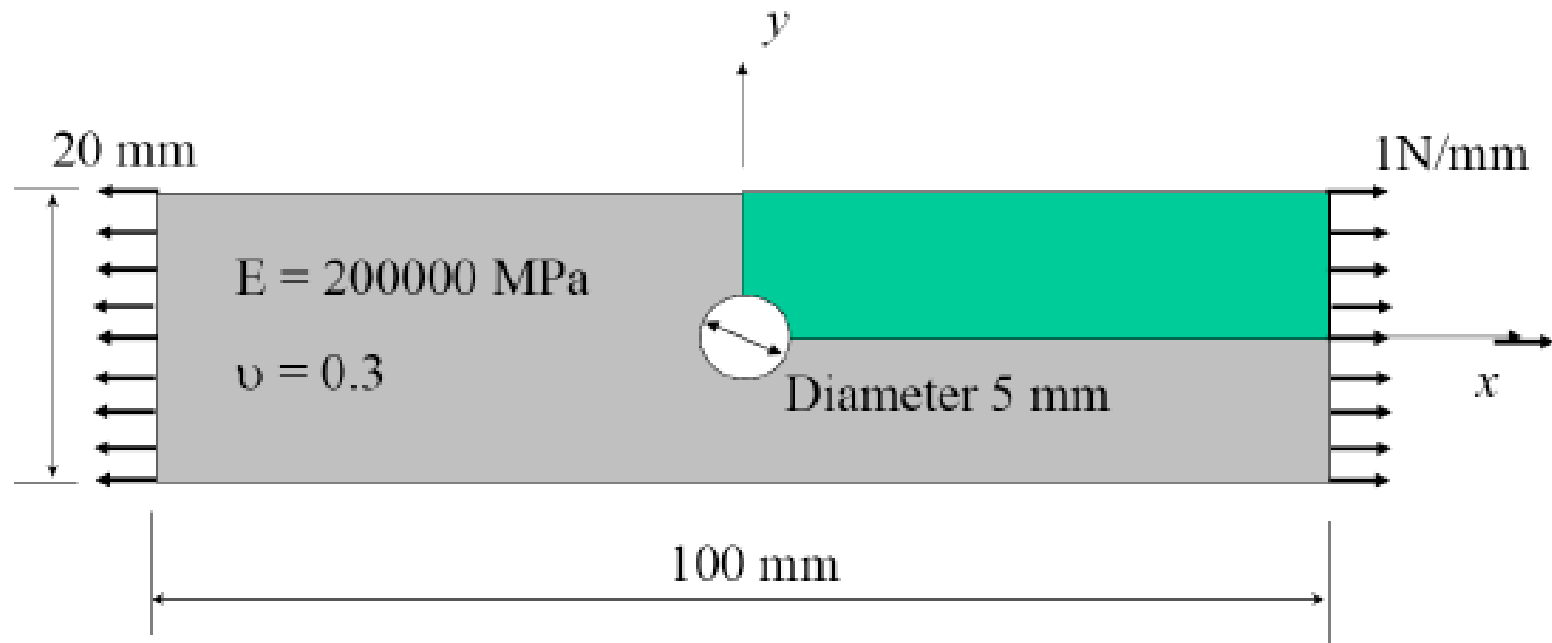
Stress concentration at hole in plate ?

Use of units in ANSYS

- ANSYS knows nothing about the units you are using
- It is important to be consistent
- For this example we will choose mm as our unit of length and MPa for stress
- The consistent units of mass, time and force are then grams, ms and N respectively

Suggested input file: keypoints generation

```
!plate: ANSYS input file for plate with hole
/FILNAM,plate              ! Sets the filename – all output files will be named plate from now on
/PREP7                     ! Enter the preprocessor – all geometry creation is done here
ET,1,PLANE42               ! Define the element type - 4 node solid element, type help 42 for more info
UIMP,1,EX, , ,200000,      ! Define material properties, this is the Young's modulus in the x direction
UIMP,1,NUXY, , ,0.3,       ! Define material properties, this is the value of Poisson's ratio for the material
K,1,2.5,0,,                ! Define keypoints, format is: k,knum,kx,ky knum is the number of the point
K,2,1.768,1.768,,          ! knum is the number of the point, kx is the x coordinate, ky is the y coordinate
K,3,0,2.5,,
K,4,10,0,,
K,5,10,10,,
K,6,0,10,,
K,7,50,0,,
K,8,50,10,,
K,11,0,0,,
```


KEYPOINT LOCATIONS
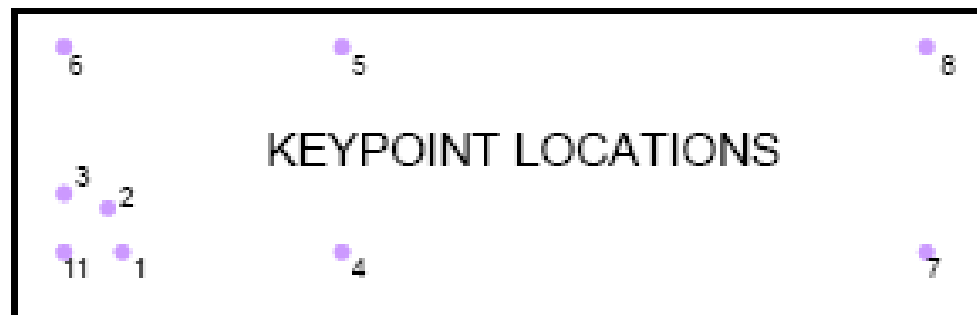
Suggested input file: lines, areas and mesh generation

```
LARC,1,2,11,2.5,        ! Define arcs
LARC,2,3,11,2.5,        ! format is: larc, pt1,pt2,cntr,radius
LSTR, 1, 4              ! Define straight lines between kypts
LSTR, 2, 5
LSTR, 3, 6
LSTR, 4, 5
LSTR, 7, 8
LSTR, 5, 6
LSTR, 4, 7
LSTR, 5, 8
AL,3,6,4,1              ! Define areas using line numbers
AL,4,8,5,2
AL,9,7,10,6
```

```
lesize,3,,,15   ! Define mesh size
lesize,4,,,15   ! The number of elements
lesize,5,,,15   ! is specified per line
lesize,1,,,8
lesize,2,,,8
lesize,6,,,8
lesize,7,,,8
lesize,8,,,8
lesize,9,,,12
lesize,10,,,12
amesh,1,3,1   !Mesh Areas
! Then define boundary conditions, loads
! Solve the model and post-process
```

LINE NUMBERS AND (MESH SIZING)

For 'mapped meshing' we need consistent sizing in the direction of the arrows – this can reduce the number of elements and generally creates a better mesh

AREA NUMBERS and FINAL MESH GEOMETRY
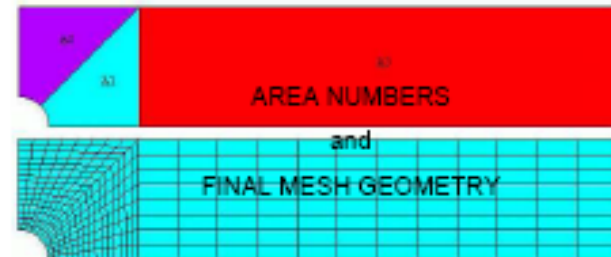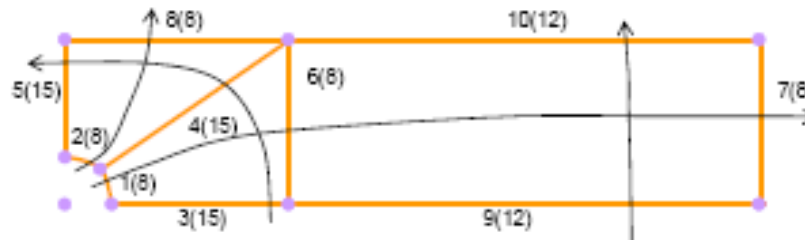
Merging common boundaries

- If you define areas and lines using BLC4 and other commands like it, they will not share boundaries
- You can use NUMMRG,all,*tol* to merge common boundaries (tol is the tolerance, usually use very small value)

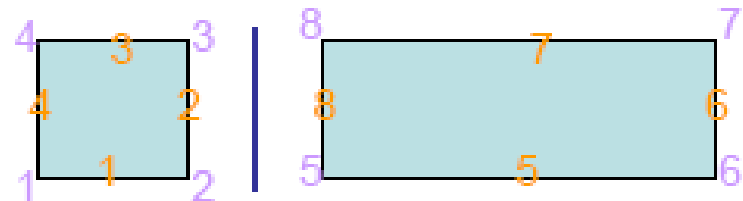Definition of model geometry

Lines and kypts before NUMMRG

Lines and kypts after NUMMRG
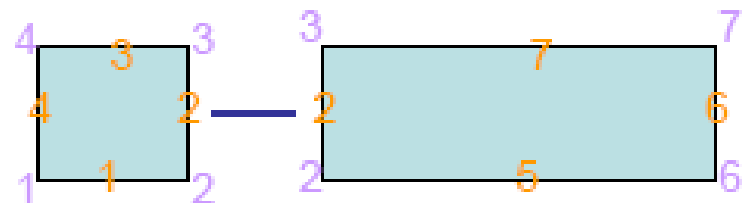
Applying boundary conditions: symmetry

GRAPHICAL METHOD

Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > *Choose*

and then pick the nodes (or lines) you want using the cursor

COMMAND METHOD

Select nodes by location using NSEL,s,loc,x,min,max

where x,y or z is coordinate direction, min is the minimum value in this direction, max is the maximum value (if only one value is given, just the nodes at this exact location are selected).

Apply DOF constraints to selected nodes using D,nnum,DOF,val

where nnum is the number of the node to apply the constraint to (use all for all selected nodes), DOF is the label of the DOF (in this case either UX or UY) and val is the value of the DOF.

e.g.   NSEL,s,loc,x,0

       D,all,ux,0

To display the DOF constraints on your model use the menu:

Plotctrls > Symbols

and select the 'ALL APPLIED BCs' option

Applying boundary conditions: force

GRAPHICAL METHOD

Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > *Choose*
and then pick the nodes (or lines) you want using the cursor

COMMAND METHOD

Select nodes by location using NSEL,s,loc,*x,min,max*

where x,y or z is coordinate direction, min is the minimum value in this direction, max is the maximum value (if only one value is given, just the nodes at this exact location are selected).

Apply forces to selected nodes using F,*nnum,LAB,val*

where nnum is the number of the node to apply the force to (use all for all selected nodes), LAB is the label of the force (in this case either FX or FY) and val is the value of the force.

e.g.   NSEL,s,loc,x,50

F,all,fx,??

Applying boundary conditions: pressure

❑ Pressure is distributed force (Surface Load)

GRAPHICAL METHOD

Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > *Choose*
and then pick the nodes (or lines) you want using the cursor

COMMAND METHOD

Select nodes by location using NSEL,s,loc,*x,min,max*

where x,y or z is coordinate direction, min is the minimum value in this direction, max is the maximum value (if only one value is given, just the nodes at this exact location are selected).

Apply pressures to selected nodes using SF,*nnum*,PRES,*val*

where nnum is the number of the node to apply the pressure to (use all for all selected nodes), PRES tells ANSYS to apply a pressure and val is the value of the pressure.

e.g.   NSEL,s,loc,x,50
        SF,all,PRES,1*zthick

Elements and their key options

- 2d structural element, PLANE 42

The nodal forces, if any, should be input per unit of depth for a plane analysis (except for KEYOPT(3) = 3)

KEYOPT(3) defines element behavior:

0 = Plane stress (default) (stress in z direction = 0)

1 = Axisymmetric

2 = Plane strain (strain in z direction = 0)

3 = Plane stress with thickness input

We want to calculate the plane stress solution, forces are per unit of depth in z direction so the effective zthick = 1. So we should apply a pressure of 1*1 = 1.

We can set a different thickness value by setting KEYOPT,*ITYPE*,3,3 (ITYPE is the element type number = 1) and R,*ISET*,*THK* (ISET is the real constant set number = 1 and *THK* is the thickness value).

Positive pressures act into the element and we want to apply the load acting out, therefore we apply a pressure of -1.

Finite element model with applied boundary conditions; running the solver

- What should the loaded model look like?
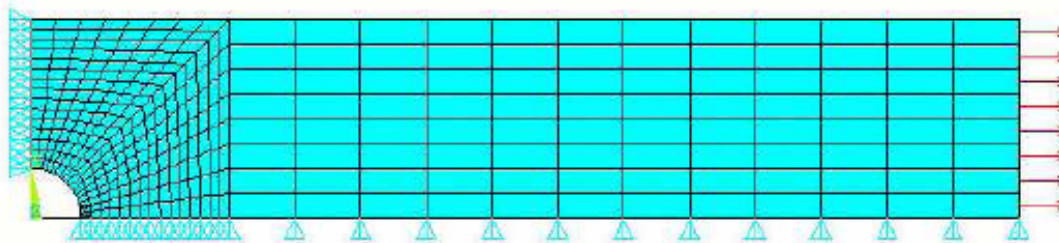
- So now we can solve the problem

GRAPHICAL METHOD

Select > Everything
Solution > Solve > Current LS

COMMAND METHOD

allsel
/solu
solve

# Viewing and listing results (General Postprocessor)

- In the steady state we will only use the General Postproc (/POST1)
- We can view results using commands, but it is often easier to do this graphically
    General Postproc > Read results > Last Set
- A few examples:
    General Postproc > Plot results > Contour plot > Nodal Solution

DOF solution > X-component of displacement

Stress > X-component of stress
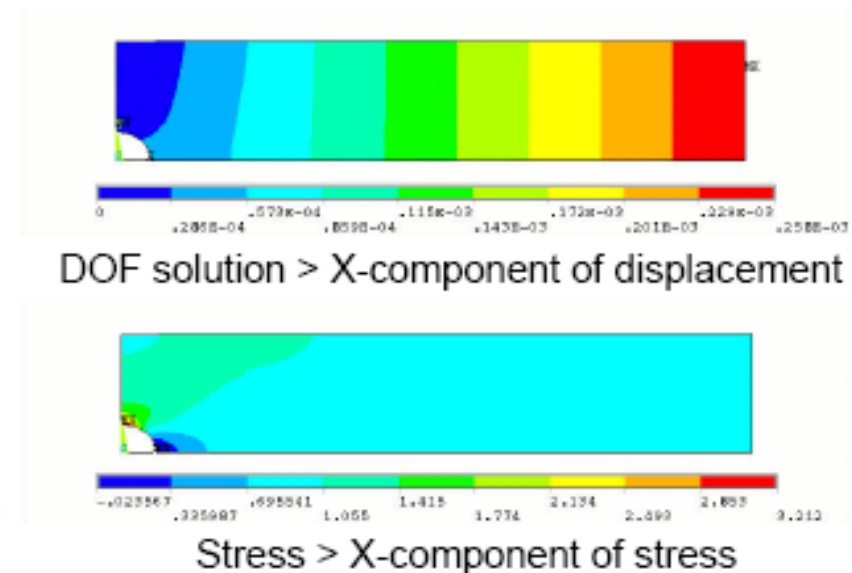
- We can do this through:
    List > Results > Nodal solution
    General Postproc > List results > Nodal solution
- This will list all selected nodes, so we might want to select fewer nodes graphically or with NSEL

Post-processing for transient analyses

- TimeHist Postproc has a graphical interface which lets us add variables and plot them
- If we want to look at the variation of displacement of a node with the applied load we can add it as a variable
- Type eplo to plot the elements of your model, make sure you can see the region you're interested in clearly
- First click on + to add a new variable
- Then select the results type to add and click OK
- Then select a node from the model and click OK
- You should see a new variable has appeared
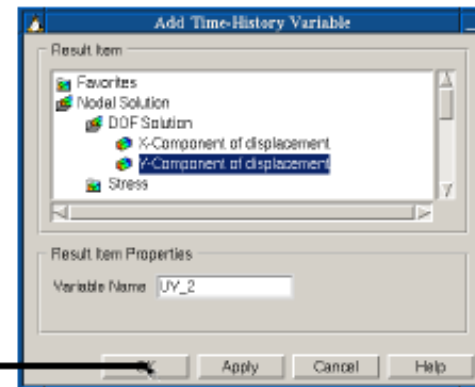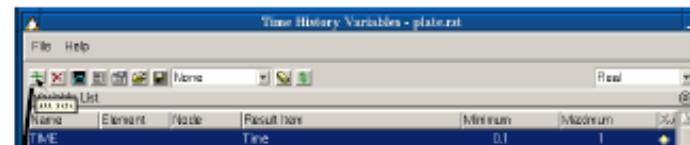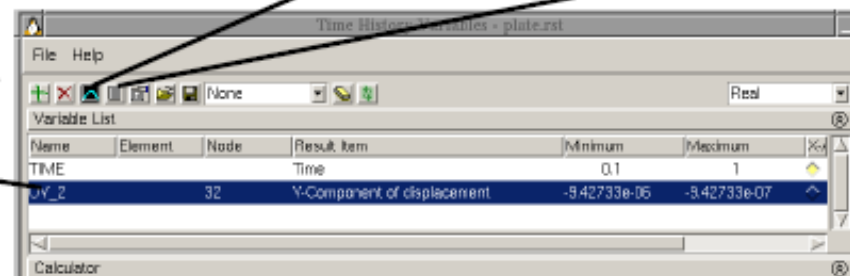
Advanced solution options

- Here we solved the response of the model using a linear solution, applying all the load at once

- If we wish to apply the load in stages we can solve the model over a number of substeps using the command NSUBST,*NSBSTP,NSBMX,NSBMN* (you should set these equal e.g. NSUBST,10,10,10)

- If we want to write results at every step we can use the command OUTRES,ALL,ALL

- When we view the results in /POST1 we may want to look at not just the last results

General Postproc > Read results > By Pick

- If our structure undergoes large deformations we may need to include non-linear geometric effects. For more information see the help for NLGEOM and Chapter 8 of the ANSYS help *Structural Analysis Guide > Nonlinear structural analysis*

Graphical representation of solution process for transient analysis