3. Stress Analysis

Overview

- ***Stress analysis*** **is a general term used to describe analyses where the results quantities include stresses and strains. It is also known as** ***structural analysis*****.**
- **As described in Chapter 2, ANSYS allows several types of stress analyses:**

Static	**Transient Dynamic**
Modal	**Spectrum**
Harmonic	**Explicit Dynamics**

- **In this chapter, we will use a linear static analysis to describe the steps involved in an analysis. By mastering these steps, you can quickly learn how to do other analyses.**

...Overview

- **Topics covered:**

A. Analysis Steps

Every analysis involves three main steps:

- **Preprocessing**
  - **Create or import the model geometry**
  - **Mesh the geometry**
- **Solution**
  - **Apply loads**
  - **Solve**
- **Postprocessing**
  - **Review results**
  - **Check the validity of the solution**

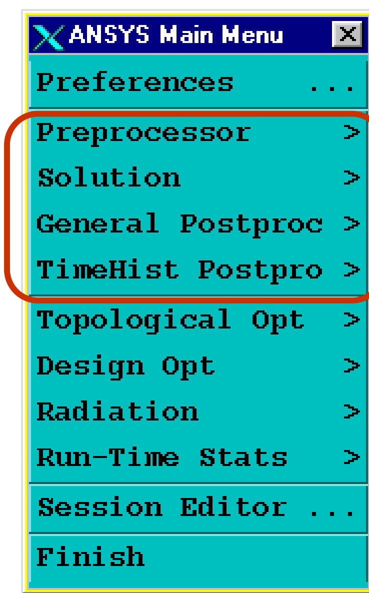
Preprocessing

Solution

Postprocessing

...Analysis Steps

- **Notice that the ANSYS Main Menu is also organized in terms of preprocessing, solution, and postprocessing.**



...Analysis Steps

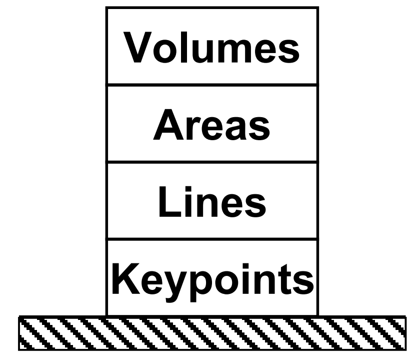
- **The preprocessor (called PREP7 in ANSYS) is where you provide the majority of the *input* to the program.**
- **Its main purpose is to generate the finite element model, which consists mainly of nodes, elements, and material property definitions. You can also use PREP7 to apply loads.**
- **Usually begins with definition of the model geometry.**
- **A *solid model* model is typically used to represent model geometry.**
  - **A CAD-type mathematical representation that defines the geometry of the structure.**
  - **May consist of solids or just surfaces, depending on what is being modeled.**

B. Geometry

- **A typical solid model is defined by volumes, areas, lines, and keypoints.**
  - ***Volumes* are bounded by areas. They represent solid objects.**
  - ***Areas* are bounded by lines. They represent faces of solid objects, or planar or shell objects.**
  - ***Lines* are bounded by keypoints. They represent edges of objects.**
  - ***Keypoints* are locations in 3-D space. They represent vertices of objects.**

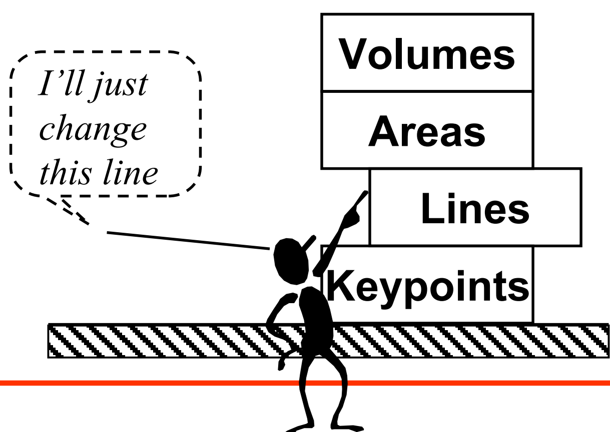
...Geometry

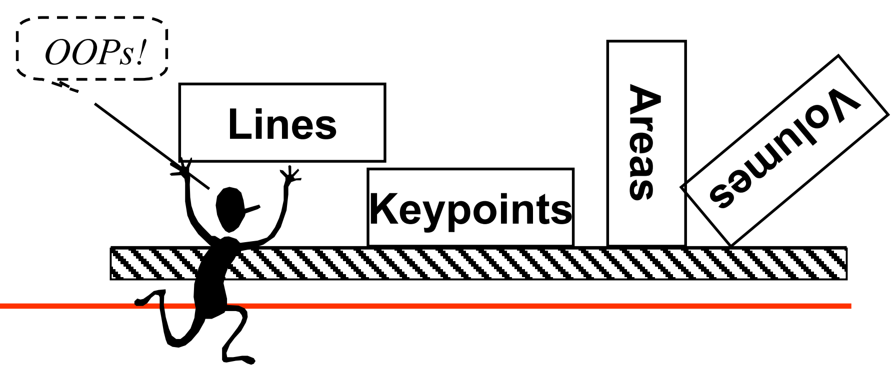
- There is a built-in *hierarchy* among solid model entities. Keypoints are the “foundation” entities. Lines are “built” from the keypoints, areas from lines, and volumes from areas.



- This hierarchy holds true regardless of how the solid model is created.

- ANSYS will not allow you to delete or modify a lower-order entity if it is attached to a higher-order entity. (Certain types of modifications are allowed… discussed later.)



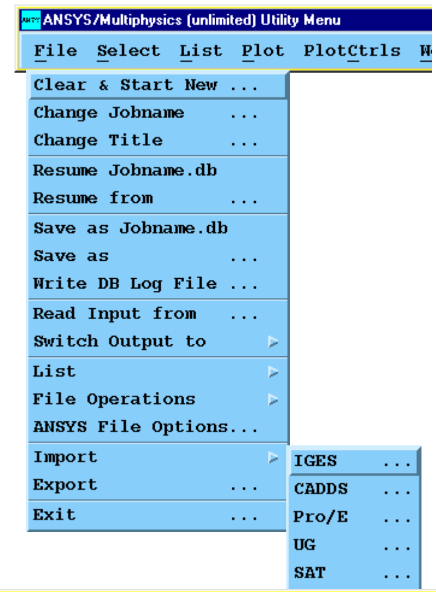


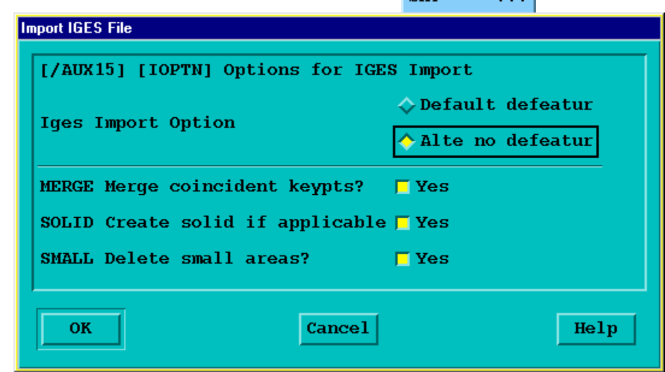
...Geometry

- You can either *create* a solid model in ANSYS or *import* it from another software package.
- Details of both methods will be presented later. For now, we will briefly discuss how to import an IGES file and scale the geometry if needed.
- IGES (Initial Graphics Exchange Specification) is a way to transfer solid model geometry from one software package to another.
  - An IGES file is ASCII, allowing it to be easily transported between computer systems.
  - Most packages, including ANSYS, allow you to *write* as well as *read* an IGES file.

...Geometry

- **To import an IGES file into ANSYS:**
  - Utility Menu > File > Import > IGES...
    - **In the resulting dialog box, choose the *alternate* method* (**Alte no defeatur**) and press OK (defaults for everything else).**
    - **In the second dialog box, choose the desired file and press OK.**
  - **OR use the** IGESIN **command:**
    - /aux15
    - ioptn,iges,alte
    - igesin,*filename,extension,directory*
    - finish



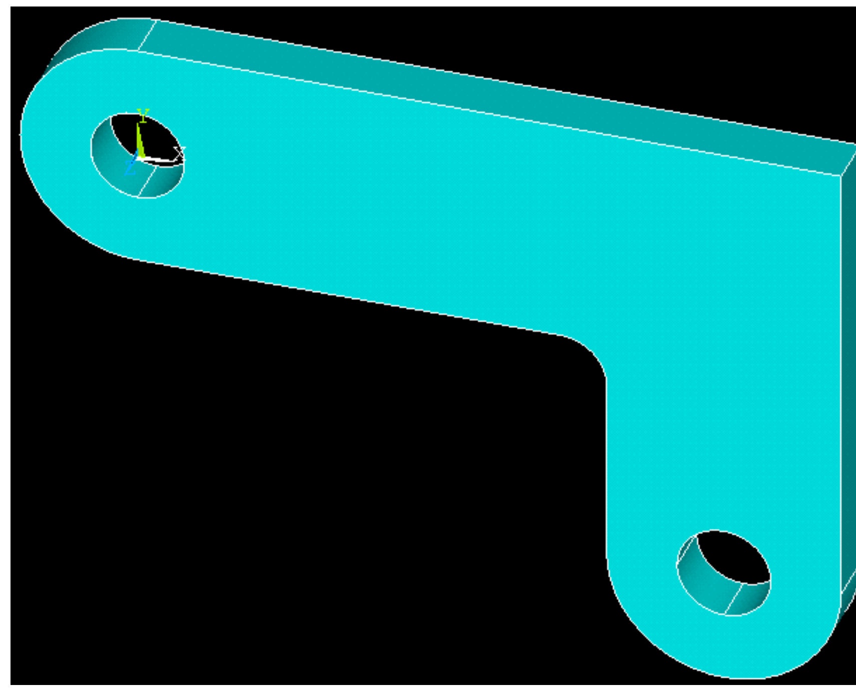


* **Details about the default vs. alternate method and the other options will be presented later.**

...Geometry

- **When the import is completed, ANSYS will automatically plot the geometry.**

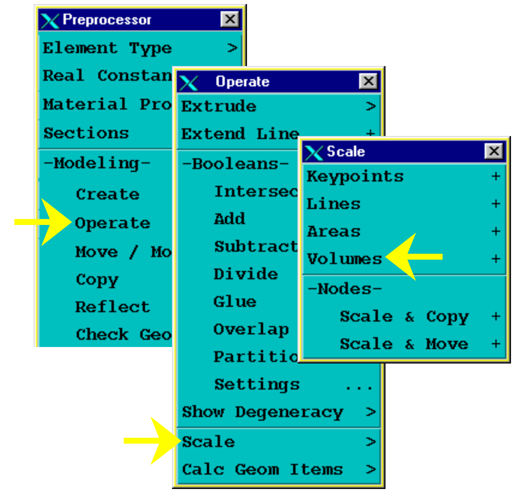
- **You may then modify the geometry as needed.**
  - **ANSYS allows many operations on the solid model, which we will describe later.**
  - **For now, we will discuss how to *scale* the model to a different set of units. (Note: Scaling is NOT available for a *default* IGES import.)**

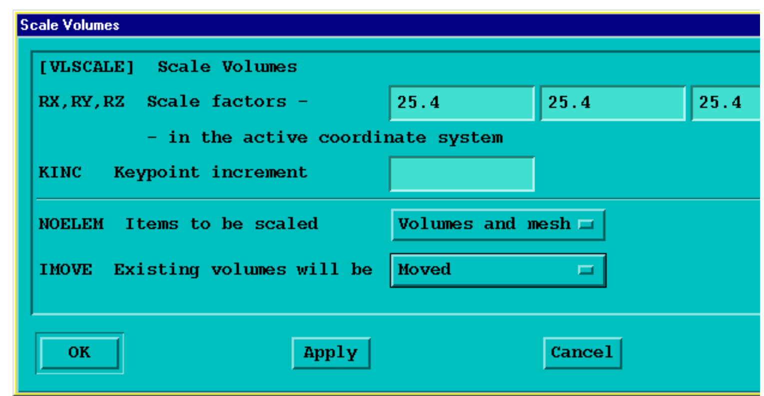


...Geometry

- **Scaling is typically needed when you want to convert the geometry to a different set of units, say from inches to millimeters.**

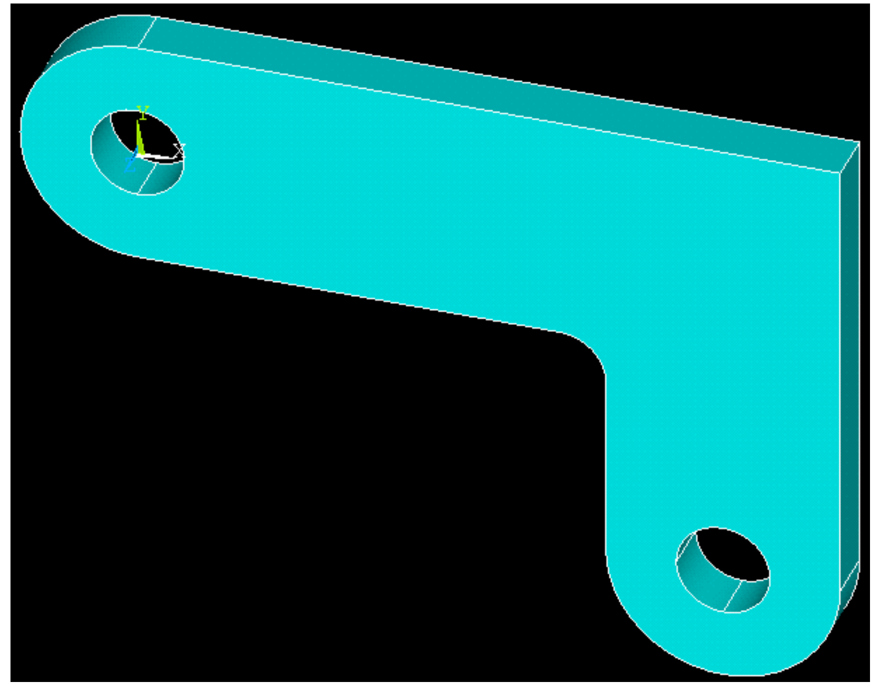
- **To scale a model in ANSYS:**
  - **First save the database --** Toolbar > SAVE_DB **or** SAVE **command.**
  - **Then** Main Menu > Preprocessor > Operate > Scale > Volumes **(choose the highest-level entity available in the model)**
    - [Pick All] **to pick all volumes**
    - **Then enter desired scale factors for RX, RY, RZ and set IMOVE to "Moved" instead of "Copied"**
  - **Or use the** VLSCALE **command:**
    - vlscale,all,,,25.4,25.4,25.4,,,1





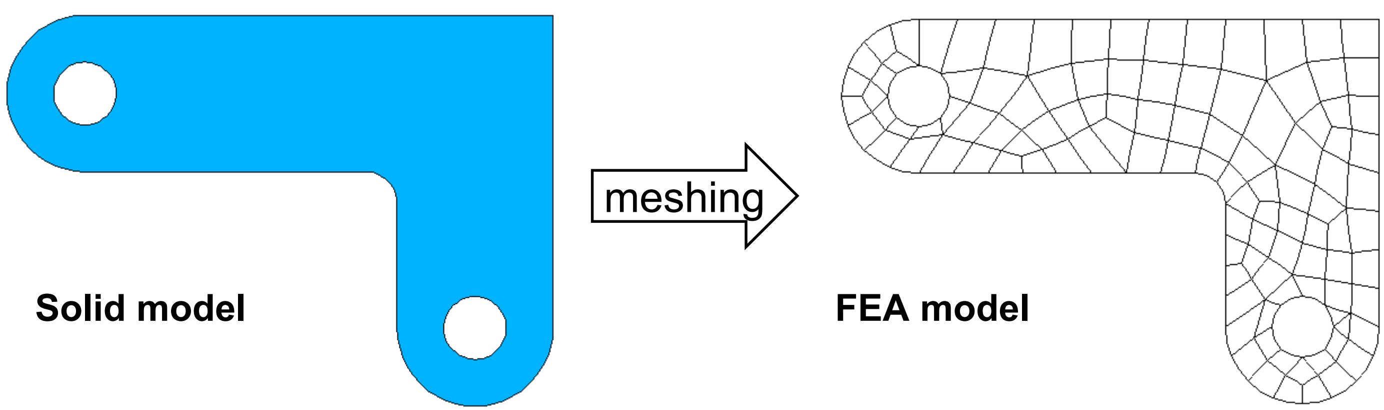
...Geometry

- **Preprocessing**
  - **Geometry**
  - **<u>Meshing</u>**
- **Solution**
  - **Loading**
  - **Solve**
- **Postprocessing**
  - **Review results**
  - **Check validity of solution**



C. Meshing

- *Meshing* **is the process used to "fill" the solid model with nodes and elements, i.e, to create the FEA model.**
  - **Remember, you need nodes and elements for the finite element solution, not just the solid model. The solid model does NOT participate in the finite element solution.**



Stress Analysis - Preprocessing

...Meshing

- There are three steps to meshing:
  - Define element attributes
  - Specify mesh controls
  - Generate the mesh

- ***Element attributes*** are characteristics of the finite element model that you must establish prior to meshing. They include:
  - Element types
  - Real constants
  - Material properties

...Meshing

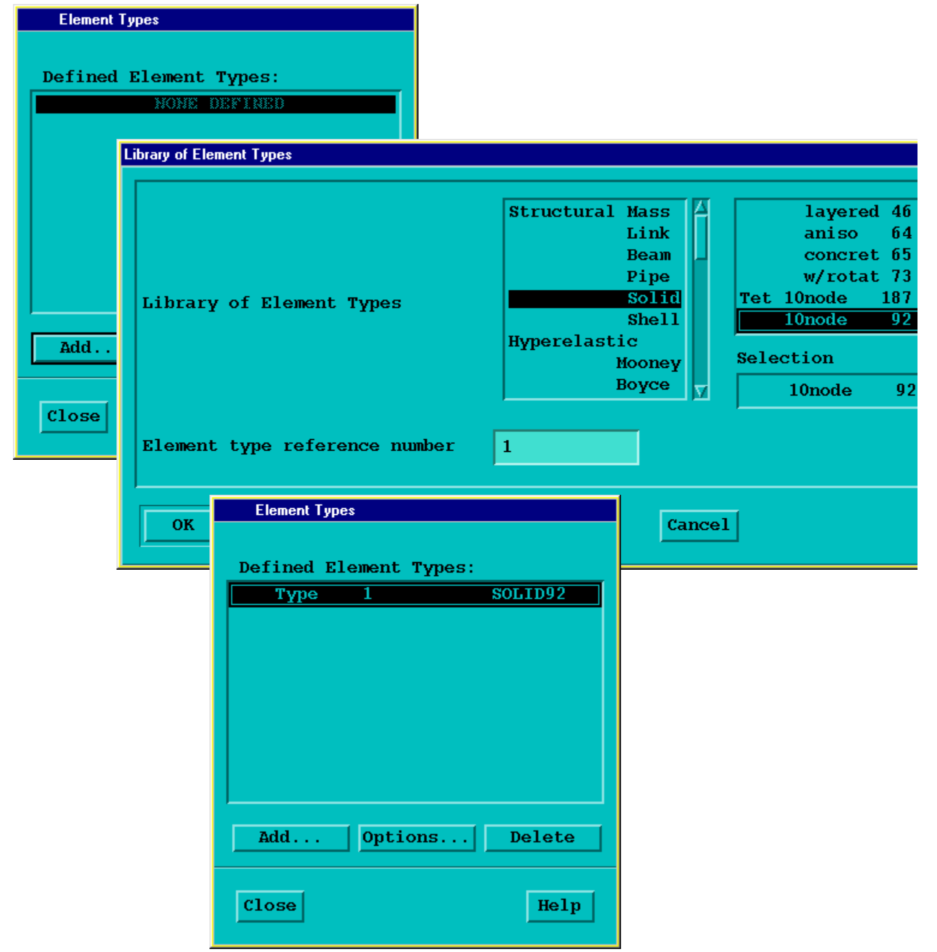
Element Type

- **The element type is an important choice that determines the following element characteristics:**
  - **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 (X-Y plane only), or 3-D.**
  - **Assumed displacement shape -- linear vs. quadratic.**
- **ANSYS has a "library" of over 150 element types from which you can choose. Details on how to choose the "correct" element type will be presented later. For now, let's see how to define an element type.**

Stress Analysis - Preprocessing

...Meshing

- **To define an element type:**
  - Preprocessor > Element Type > Add/Edit/Delete
    - [Add] **to add new element type**
    - **Choose the desired type (such as SOLID92) and press OK**
    - [Options] **to specify additional element options**
  - **Or use the** ET **command:**
    - et,1,solid92



...Meshing

- **Notes:**
  - **Setting preferences to the desired discipline (**Main Menu > Preferences) **will show only the element types valid for that discipline.**
  - **You should define the element type early in the preprocessing phase because many of the menu choices in the GUI are filtered out based on the current DOF set. For example, if you choose a structural element type, thermal load choices will be "grayed out" or not shown at all.**

...Meshing

Real Constants

- **Real constants are used for geometric properties that cannot be completely defined by the element's geometry. For example:**
  - **A beam element is defined by a line joining two nodes. This defines only the length of the beam. To specify the beam's cross-sectional properties, such as the area and moment of inertia, you need to use real constants.**
  - **A shell element is defined by a quadrilateral or triangular area. This defines only the surface area of the shell. To specify the shell thickness, you need to use real constants.**
  - **Most 3-D solid elements do not require a real constant since the element geometry is fully defined by its nodes.**

...Meshing

- **To define real constants:**
  - Preprocessor > Real Constants
    - [Add] **to add a new real constant set.**
    - **If multiple element types have been defined, choose the element type for which you are specifying real constants.**
    - **Then enter the real constant values.**
  - **Or use the** R **family of commands.**
- **Different element types require different real constants, and some don’t require any real constants. Check the *Elements Manual*, available on-line, for details.**

...Meshing

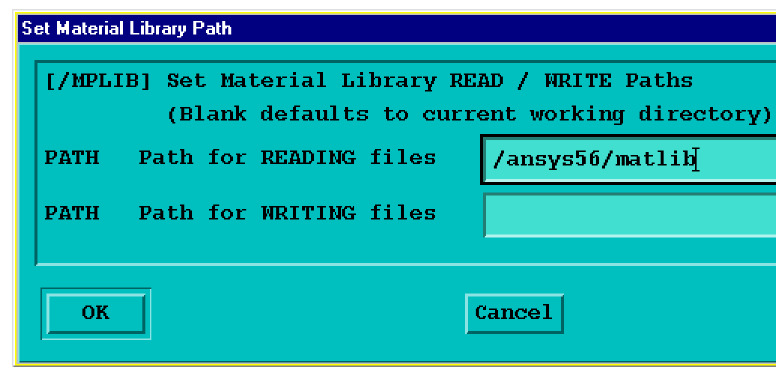
Material Properties

- **Every analysis requires *some* material property input: Young's modulus EX for structural elements, thermal conductivity KXX for thermal elements, etc.**
- **There are two ways to define material properties:**
  - **Material library**
  - **Individual properties**

...Meshing

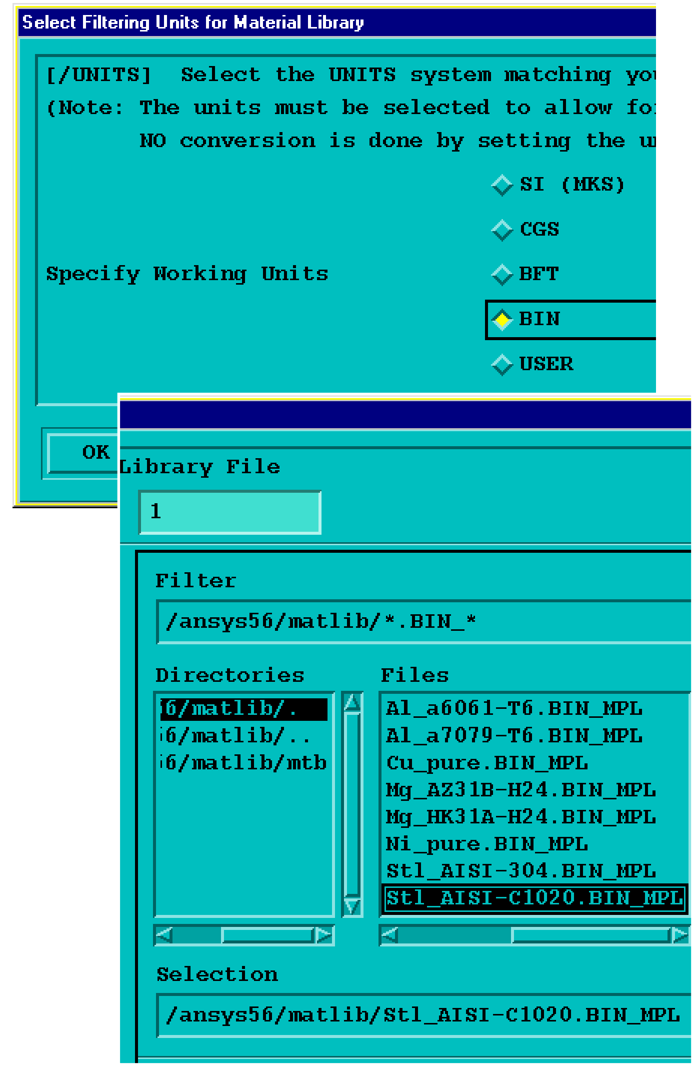
Using the Material Library

- **This method allows you to choose a predefined set of properties for a given material.**
- **ANSYS supplies typical structural and thermal properties (linear only) for some common materials, but we strongly recommend that you create your own material library.**
- **To choose a material from the library:**
  - **First define the library path.**
    - Preprocessor > Material Props > Material Library > Library Path
      - **Enter the location from which to READ material data, e.g,** /ansys56/matlib.
    - **Or use the** /MPLIB **command.**



...Meshing

- Then “import” a material from the library.
  - Preprocessor > Material Library > Import Library
    - Choose the units system. This is used only to filter the list of files shown in the subsequent dialog. ANSYS has no knowledge of units and does NOT do unit conversion.
    - Choose the desired material file, such as steel AISI C1020.
  - **Or use the** MPREAD **command with the LIB option.**

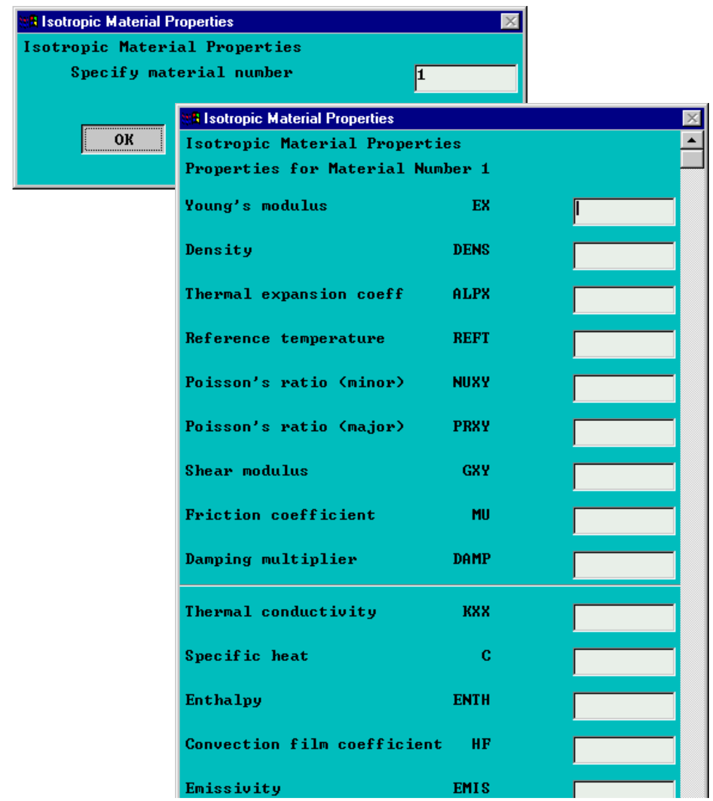


...Meshing

Specifying Individual Material Properties

- **Instead of choosing a material name, this method involves directly specifying the required properties.**

- **To specify individual properties:**
  - Preprocessor > Material Props > Isotropic
    - **Specify material number, usually 1.**
    - **Then enter the individual property values.**
  - **Or use the** MP **command.**



...Meshing

A Note on Units

- **You do not need to tell ANSYS the system of units you are using. Simply decide what units you will use, then make sure all of your input is consistent.**
  - **For example, if the model geometry is in inches, make sure that all other input data — material properties, real constants, loads, etc. — are in terms of inches.**
- **ANSYS does NOT do units conversion! It simply accepts all numbers you input without questioning their validity.**
- **The command** /UNITS **allows you to specify a units system, but it is simply a recording device to let other users of your model know what units you used.**

...Meshing

Specifying Mesh Controls **is the second step in meshing.**

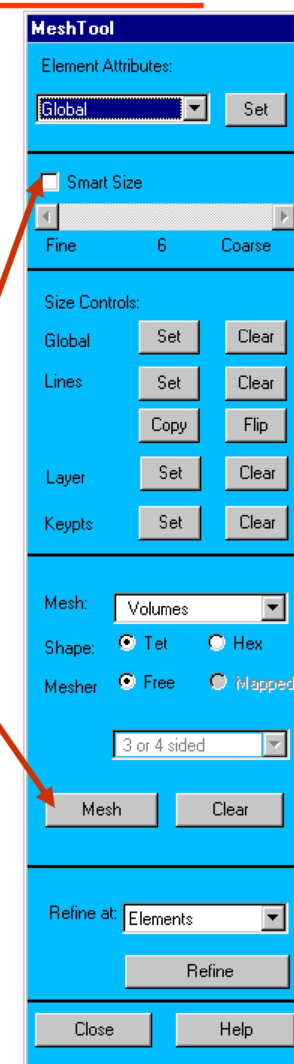
- **Many mesh controls are available in ANSYS. For now, we will present a simple method of specifying mesh density, called SmartSizing.**
- ***SmartSizing*** **is an algorithm that assigns element divisions to all lines in the model based on line length, curvature, and proximity to holes, etc.**
- **You simply specify a "size level" ranging from 1 (very fine mesh) to 10 (very coarse mesh), and ANSYS takes care of the rest.**

...Meshing

- **The MeshTool is the best way to specify mesh controls:**
  - Preprocessor > MeshTool**.**
  - **Activate SmartSizing. Size level defaults to 6.**

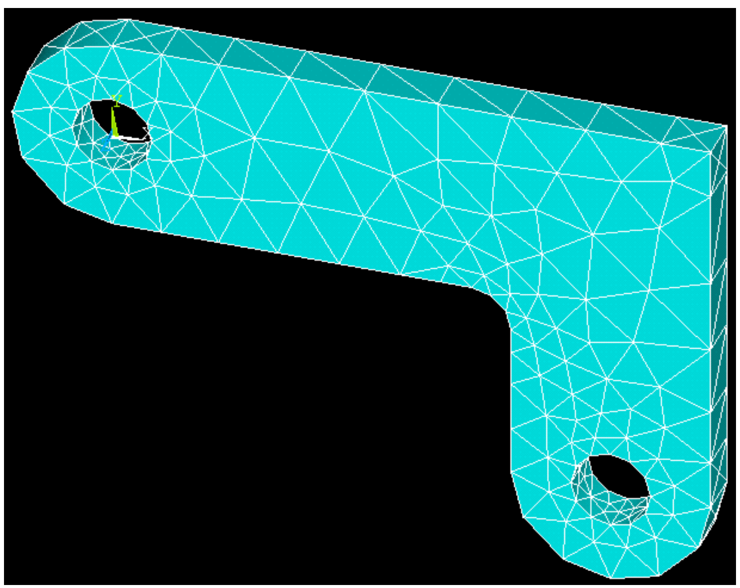
Generating the mesh **is the final step in meshing.**

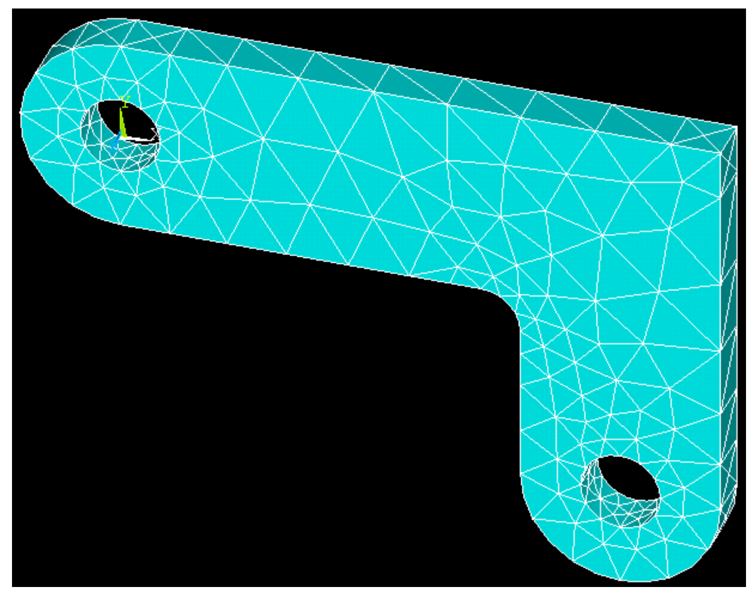
- **First save the database.**
- **Then press** [Mesh] **in the MeshTool.**
  - **This brings up a picker. Press** [Pick All] **in the picker to indicate all entities.**



...Meshing

- **When the meshing is complete, ANSYS will automatically plot the elements.**
  - **The default element plot shows all element edges as straight lines even for a quadratic element type.**
  - **To show curved element edges, issue** /EFACET,2 **(or** Utility Menu > PlotCtrls > Size and Shape…**).**

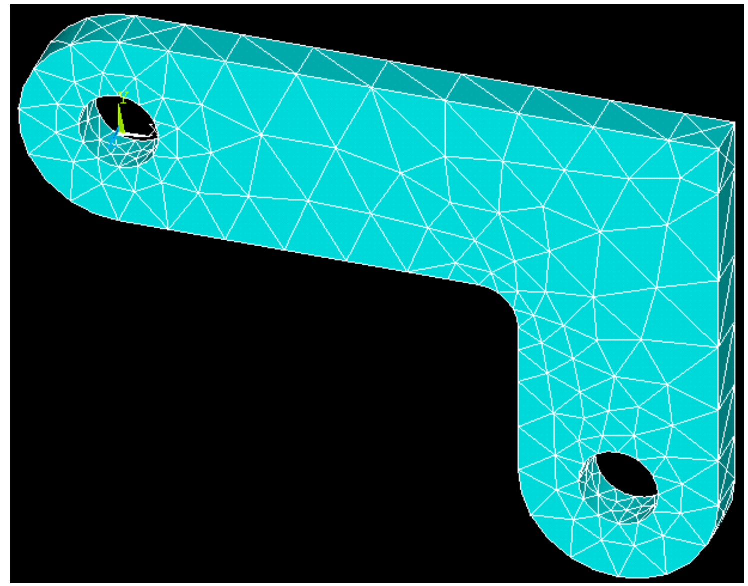




Stress Analysis - Preprocessing

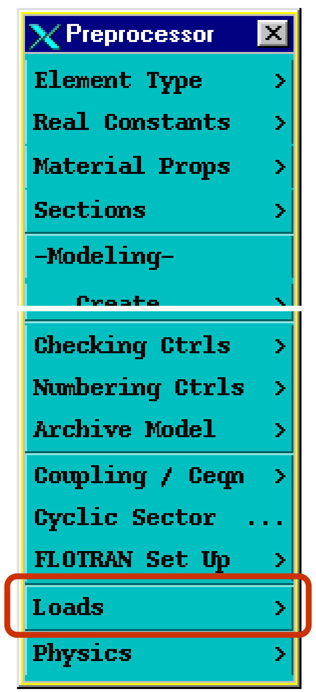
…Meshing

- **Preprocessing**
  - **Geometry**
  - **Meshing**
- **Solution**
  - **<u>Loading</u>**
  - **Solve**
- **Postprocessing**
  - **Review results**
  - **Check validity of solution**



D. Loading

- The solution step is where we apply loads on the object and let the solver calculate the finite element solution.
- Loads are available both in the Solution and Preprocessor menus.



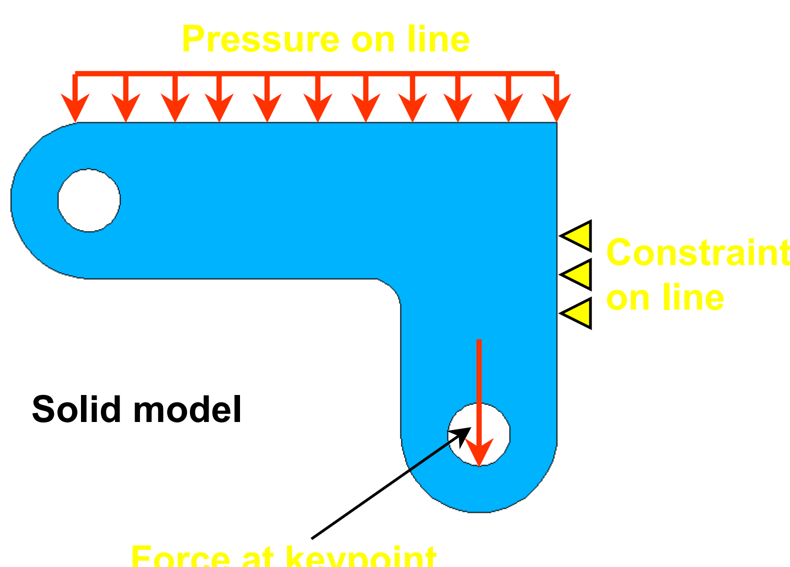
...Loading

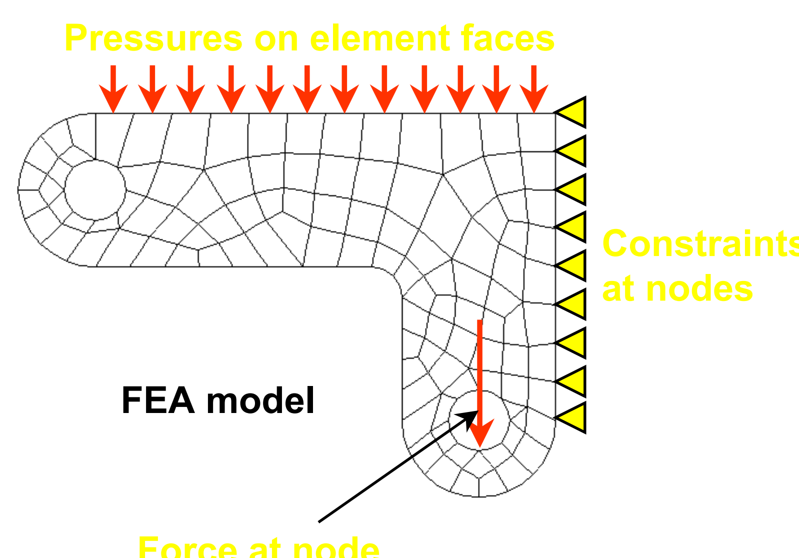
- **There are five categories of loads:**

DOF Constraints	**Specified DOF values, such as displacements in a stress analysis or temperatures in a thermal analysis.**
Concentrated Loads	**Point loads, such as forces or heat flow rates.**
Surface Loads	**Loads distributed over a surface, such as pressures or convections.**
Body Loads	**Volumetric or field loads, such as temperatures (causing thermal expansion) or internal heat generation.**
Inertia Loads	**Loads due to structural mass or inertia, such as gravity and rotational velocity.**

...Loading

- **You can apply loads either on the solid model or directly on the FEA model (nodes and elements).**
  - **Solid model loads are easier to apply because there are fewer entities to pick.**
  - **Moreover, solid model loads are independent of the mesh. You don't need to reapply the loads if you change the mesh.**





...Loading

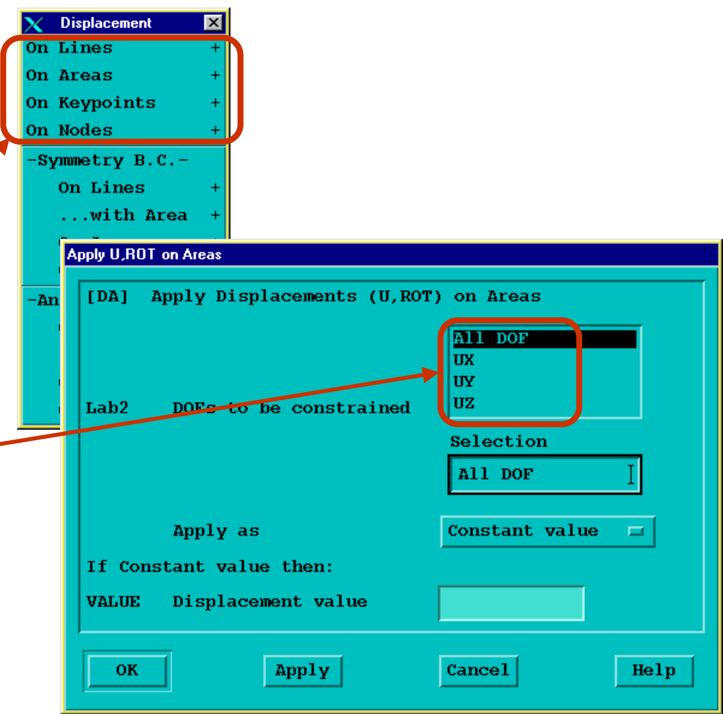
- **Regardless of how you apply the loads, the solver expects all loads to be in terms of the finite element model. Therefore, solid model loads are *automatically transferred* to the underlying nodes and elements during solution.**
- **We will now discuss how to apply the following types of structural loads:**
  - **Displacement constraints**
  - **Pressures**
  - **Gravity**

...Loading

Displacement Constraints

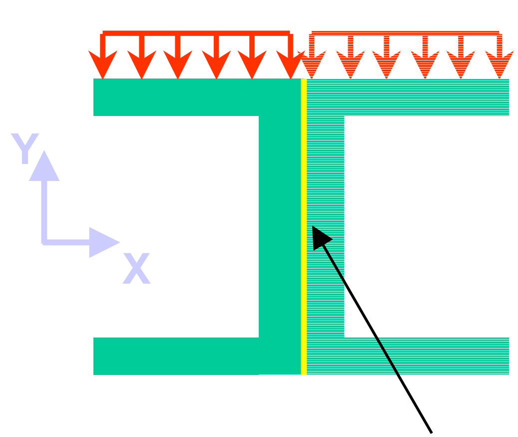
- **Used to specify where the model is fixed (zero displacement locations).**
- **Can also be non-zero, to simulate a known deflection.**
- **To apply displacement constraints :**
  – Solution > -Loads- Apply > Displacement
    - **Choose where you want to apply the constraint.**
    - **Pick the desired entities in the graphics window.**
    - **Then choose the constraint direction. Value defaults to zero.**

  – **Or use the** D **family of commands:** DK, DL, DA, D.

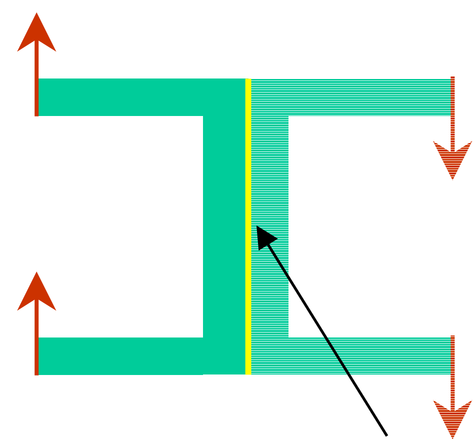


...Loading

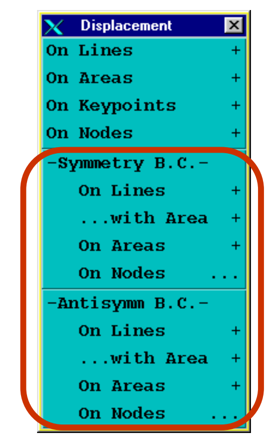
- **Displacement constraints are also used to enforce *symmetry* or *antisymmetry* boundary conditions.**
  - **Symmetry BC: Out-of-plane displacements and in-plane rotations are fixed.**
  - **Antisymmetry BC: In-plane displacements and out-of-plane rotations are fixed.**



Symmetry Boundary
UX=0
ROTY=ROTZ=0



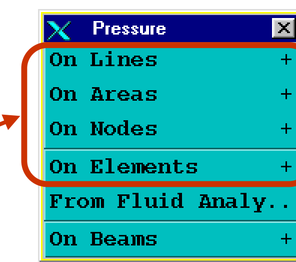
Antisymmetry Boundary
UY=UZ=0
ROTX=0

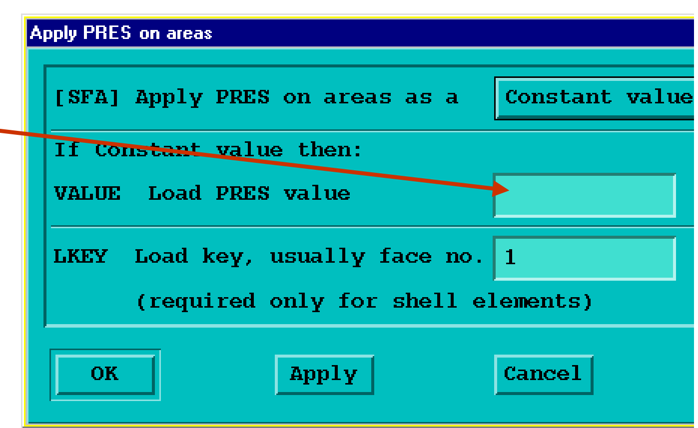


...Loading

Pressures

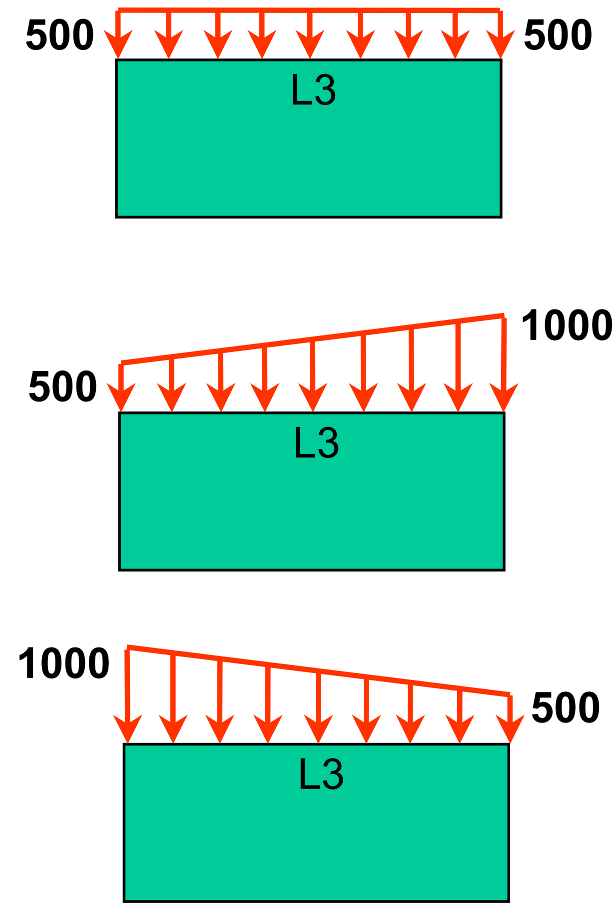
- **To apply a pressure:**
  - Solution > -Loads- Apply > Pressure
    - **Choose where you want to apply the pressure -- usually on lines for 2-D models, on areas for 3-D models.**
    - **Pick the desired entities in the graphics window.**
    - **Then enter the pressure value. A positive value indicates a compressive pressure (acting towards the centroid of the element).**
  - **Or use the** SF **family of commands:** SFL, SFA, SFE, SF.





...Loading

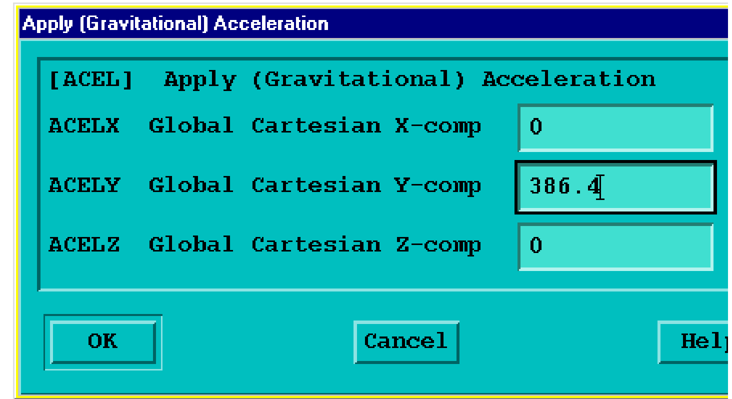
- **For a 2-D model, where pressures are usually applied on a line, you can specify a tapered pressure by entering a value for both the I and J ends of the line.**
- **I and J are determined by the line direction. If you see the taper going in the wrong direction, simply reapply the pressure with the values reversed.**



...Loading

Gravity

- **To apply gravitational acceleration:**
  - Solution > -Loads- Apply > Gravity
  - **Or use the** ACEL **command.**

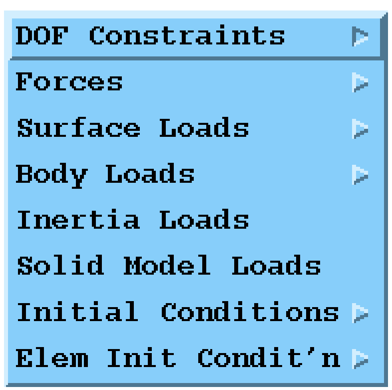


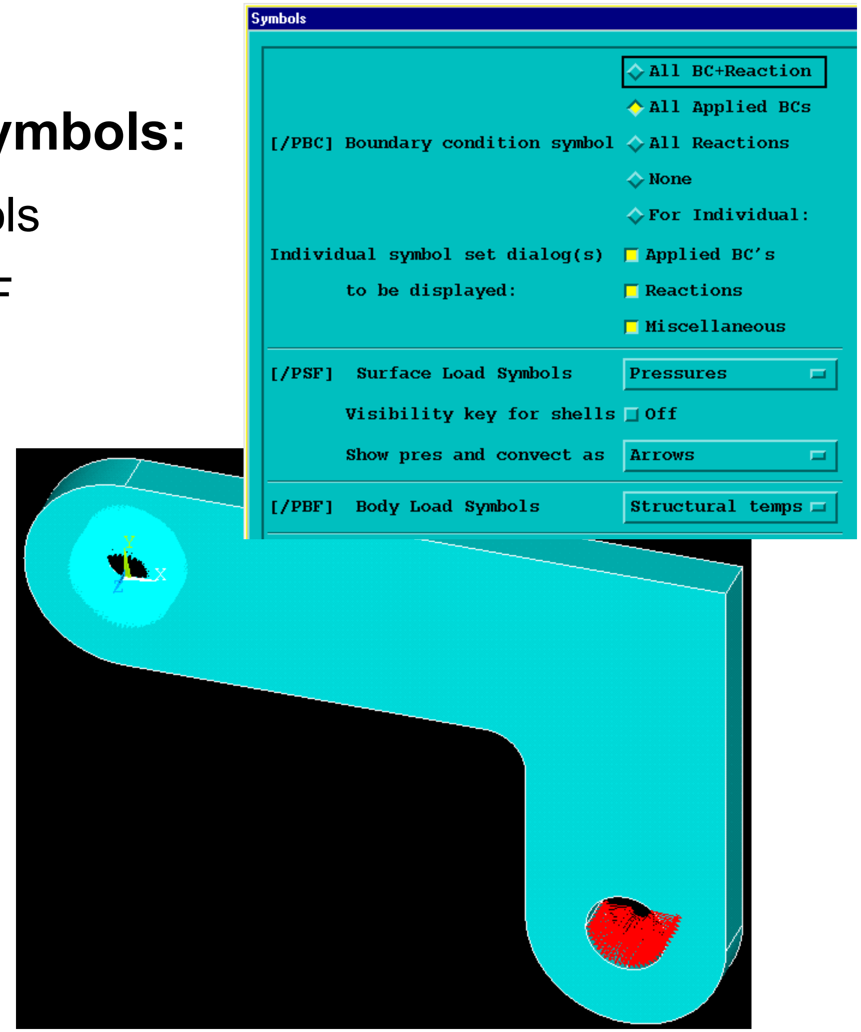
- **Notes:**
  - **A <u>positive</u> acceleration value causes deflection in the <u>negative</u> direction. If Y is pointing upwards, for example, a positive ACELY value will cause the structure to move downwards.**
  - **Density (or mass in some form) must be defined for gravity and other inertia loads.**

...Loading

Verifying applied loads

- **Plot them by activating load symbols:**
  - Utility Menu > PlotCtrls > Symbols
  - **Commands --** /PBC, /PSF, /PBF
- **Or list them:**
  - Utility Menu > List > Loads >





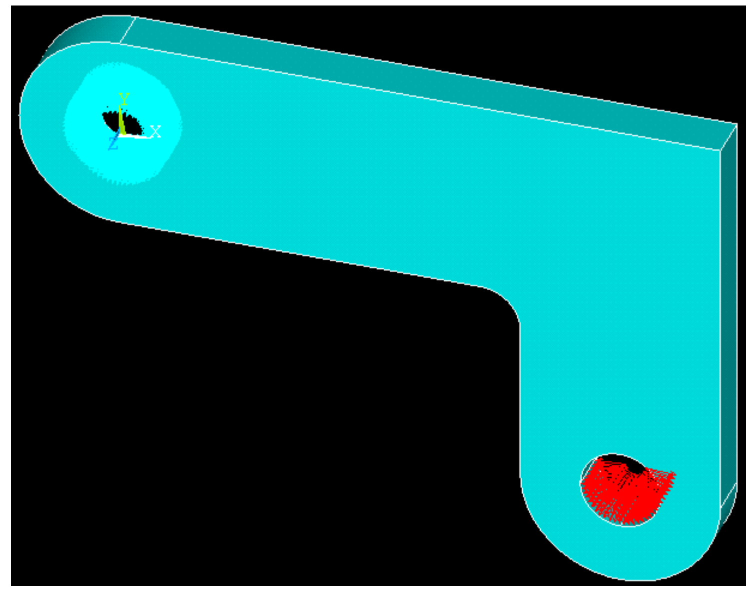
...Loading

Modifying and Deleting Loads

- **To modify a load value, simply reapply the load with the new value.**
- **To delete loads:**
  - Solution > -Loads- Delete >
  - **When you delete solid model loads, ANSYS also automatically deletes all corresponding finite element loads.**

...Loading

- **Preprocessing**
  - **Geometry**
  - **Meshing**
- **Solution**
  - **Loading**
  - **Solve**
- **Postprocessing**
  - **Review results**
  - **Check validity of solution**

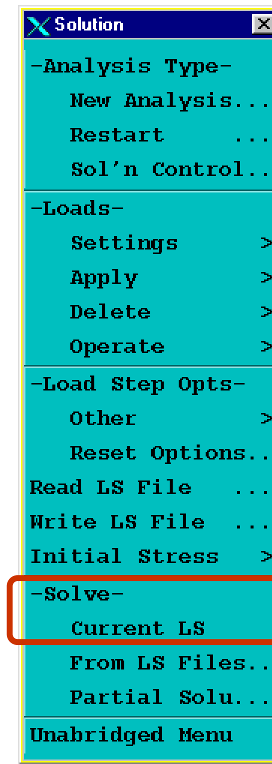


E. Solve

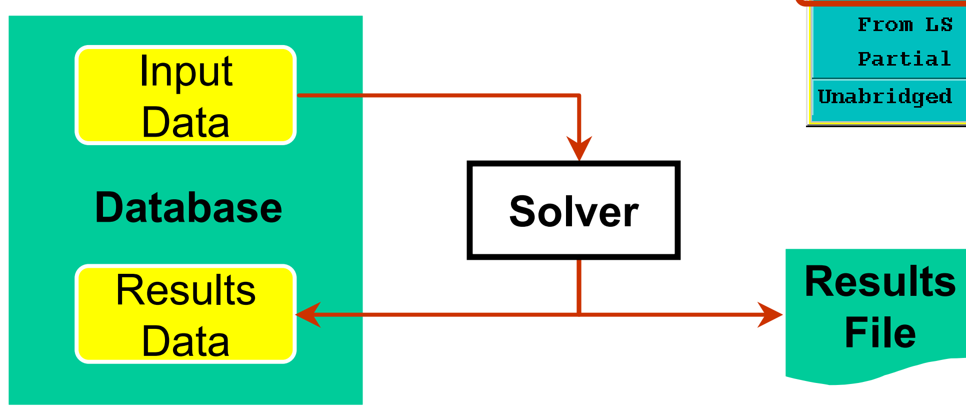
- The *solve* step is where you let the solver calculate the finite element solution.
- First, it is a good idea to review and check your analysis data, e.g:
  - Consistent units
  - Element types, options, and real constants
  - Material properties
    - Density if inertia loading
    - Coefficient of thermal expansion if thermal stress
  - Mesh density, especially in stress concentration regions
  - Load values and directions
  - Reference temperature for thermal expansions

...Solve

- **To initiate the solve:**
  - **First save the database!**
  - **Then:**
    - Solution > -Solve- Current LS
    - **Or issue the** SOLVE **command.**



- **The solver writes results data to the in-memory database <u>and</u> to the results file, *jobname*.rst (or .rth, .rmg, .rfl).**

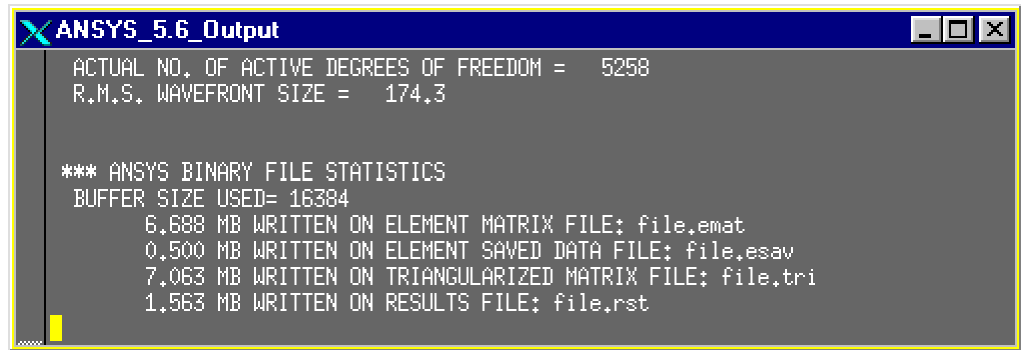


...Solve

- **During solution, ANSYS provides a lot of useful information in the Output Window, such as:**
  - **Mass properties of the model**
    - **The mass calculation is quite accurate; centroid and mass moment calculations are rough approximations**
  - **Range of element matrix coefficients**
    - **May indicate a problem with material properties or real constants if maximum/minimum ratio > 1.0E8**
  - **Model size and solver statistics**
  - **Summary of files written and their sizes:**
    - jobname.emat **- element matrix file**
    - jobname.esav **- element saved data file**
    - jobname.tri **- triangularized matrix file**
    - jobname.rst **- results file**

…Solve

- **Preprocessing**
  - **Geometry**
  - **Meshing**
- **Solution**
  - **Loading**
  - **Solve**
- **Postprocessing**
  - – **Review results**
  - – **Check validity of solution**



F. Reviewing Results

- **Postprocessing is without doubt the most important step in an analysis. You may be required to make design decisions based on the results, so it is a good idea not only to review the results carefully, but also to check the validity of the solution.**
- **ANSYS has two postprocessors:**
  - **POST1, the General Postprocessor, to review a single set of results over the entire model.**
  - **POST26, the Time-History Postprocessor, to review results at selected points in the model over time. Mainly used for transient and nonlinear analyses. (Not discussed in this course.)**

...Reviewing Results

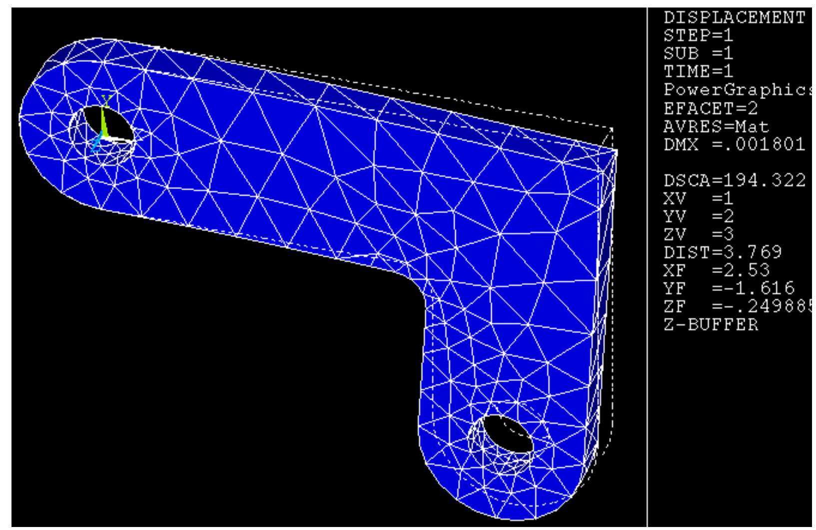
- **Reviewing results of a stress analysis generally involves:**
  - **Deformed shape**
  - **Stresses**
  - **Reaction forces**

<u>Deformed Shape</u>

- **Gives a quick indication of whether the loads were applied in the correct direction.**
- **Legend column shows the maximum displacement, DMX.**
- **You can also *animate* the deformation.**

...Reviewing Results

- **To plot the deformed shape:**
  - General Postproc > Plot Results > Deformed Shape
  - **Or use the** PLDISP **command.**
- **For animation:**
  - Utility Menu > PlotCtrls > Animate > Deformed Shape
  - **Or use the** ANDISP **command.**



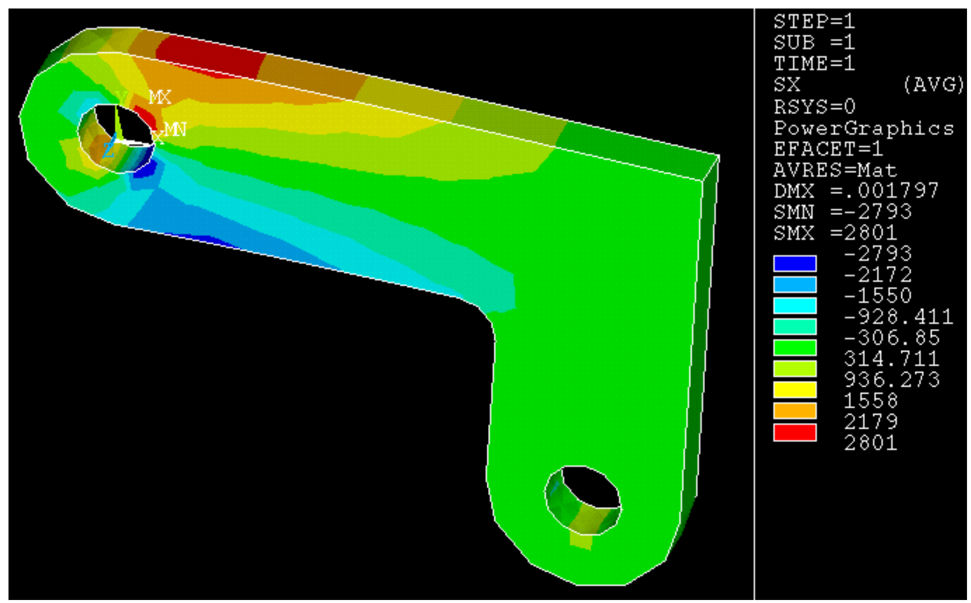
...Reviewing Results

Stresses

- **The following stresses are typically available for a 3-D solid model:**
  - **Component stresses —** SX, SY, SZ, SXY, SYZ, SXZ **(global Cartesian directions by default)**
  - **Principal stresses —** S1, S2, S3, SEQV **(von Mises),** SINT **(stress intensity)**
- **Best viewed as contour plots, which allow you to quickly locate "hot spots" or trouble regions.**
  - ***Nodal solution:*** **Stresses are averaged at the nodes, showing smooth, continuous contours.**
  - ***Element solution:*** **No averaging, resulting in discontinuous contours.**

...Reviewing Results

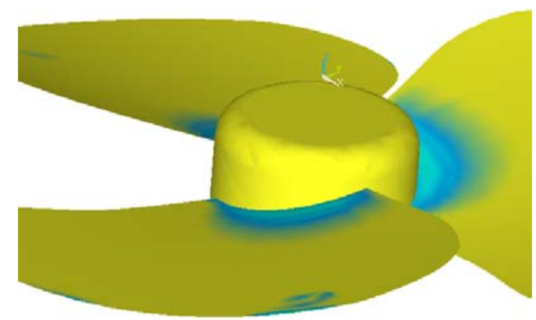
- **To plot stress contours:**
  - General Postproc > Plot Results > Nodal Solu… **or** PLNSOL **command**
  - General Postproc > Plot Results > Element Solu… **or** PLESOL **command**
- **You can also *animate* stress contours:**
  - Utility Menu > PlotCtrls > Animate > Deformed Results... **or** ANCNTR **command**



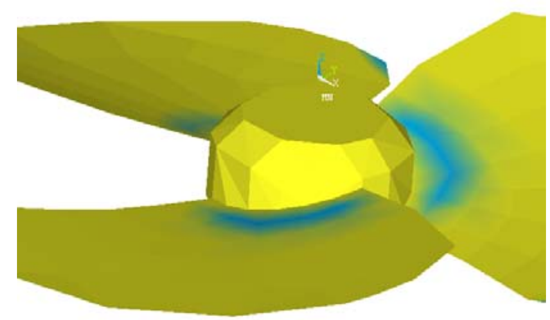
...Reviewing Results

A Note on PowerGraphics

- **It is the default graphics setting (**/GRAPH,POWER**).**
- **Plots only the visible surfaces and ignores everything "underneath."**



- **Advantages:**
  - **Faster replot, crisp graphics.**
  - **Smooth, almost photo-realistic displays.**
  - **Prevents stress averaging across material and real constant boundaries.**

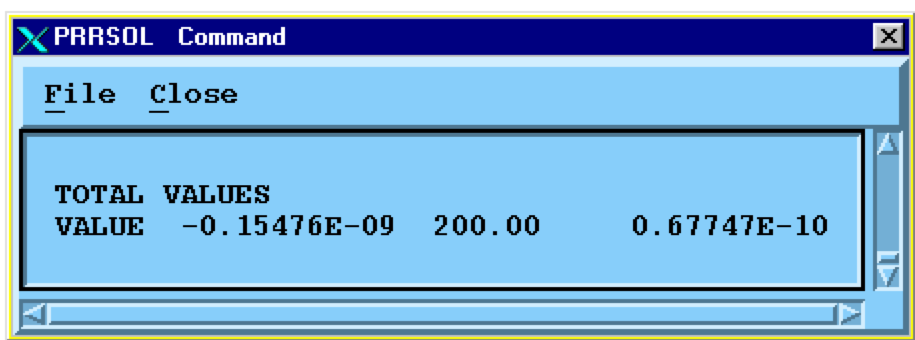


- **To deactivate PowerGraphics (or activate "full graphics"):**
  - Toolbar > POWERGRPH
  - **Or issue** /GRAPH,FULL**.**

...Reviewing Results

Reaction Forces

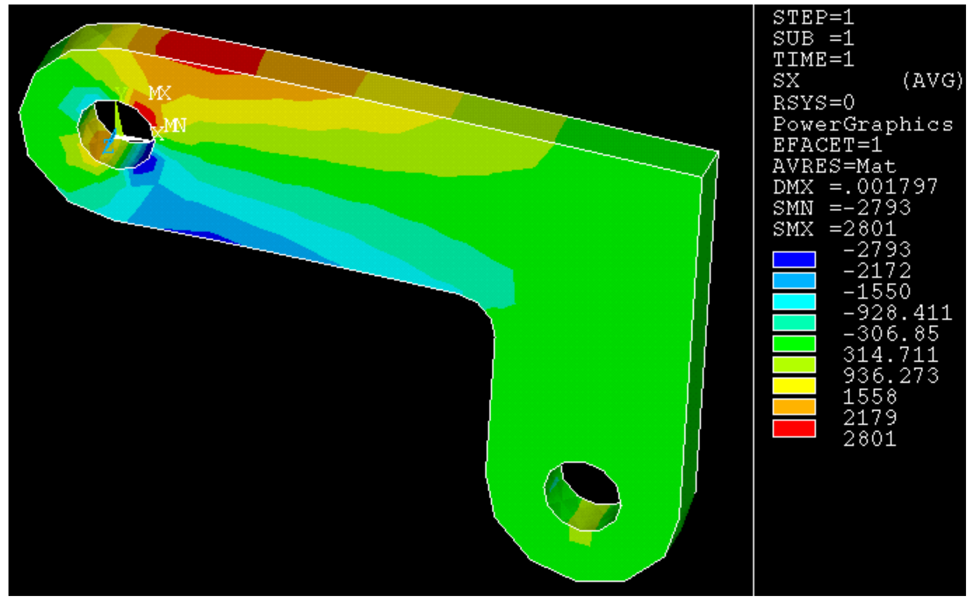
- **The sum of the reaction forces in each direction must equal the sum of applied loads in that direction.**
- **Best viewed as a listing:**
  - General Postprocessor > List Results > Reaction Solution… **or** PRRSOL **command**



Stress Analysis - Postprocessing

...Reviewing Results

- **Preprocessing**
  - **Geometry**
  - **Meshing**
- **Solution**
  - **Loading**
  - **Solve**
- **Postprocessing**
  - **Review results**
  - **<u>Check validity of solution</u>**



G. Checking Validity of Solution

- **It is always a good idea to do a "sanity check" and make sure that the solution is acceptable.**
- **What you need to check depends on the type of problem you are solving, but here are some typical questions to ask:**

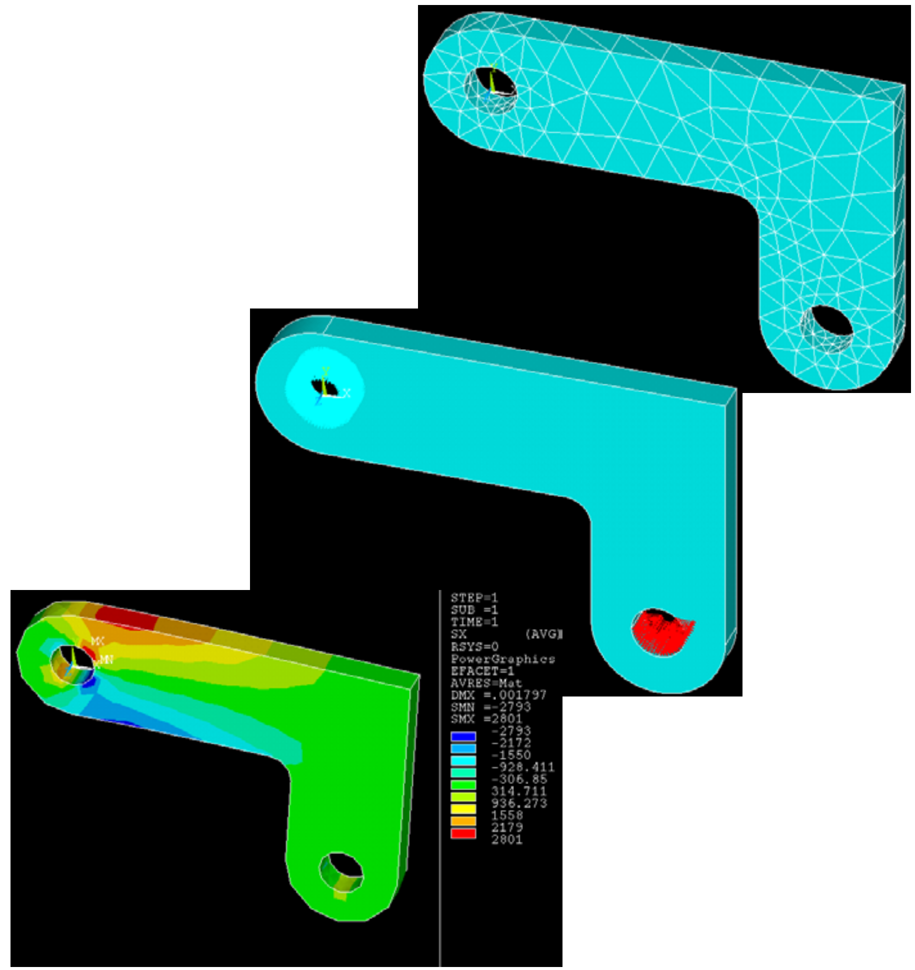
- ***Do the reaction forces balance the applied loads?***
- ***Where is the maximum stress located?***
  - **If it is at a singularity, such as a point load or a re-entrant corner, the value is generally meaningless. (We will discuss more about this in Chapter 5.)**
- ***Are the stress values beyond the elastic limit?***
  - **If so, the load magnitudes may be wrong, or you may need to do a nonlinear analysis.**

...Checking Validity of Solution

- ***Is the mesh adequate?***
  - This is always debatable, but you can gain confidence in the mesh by using error estimation data (discussed in Chapter 14).
  - Other ways to check mesh adequacy:
    - Plot the element solution (unaveraged stresses) and look for elements with high stress gradients. These regions are candidates for mesh refinement.
    - If there is a significant difference between the nodal (averaged) and element (unaveraged) stress contours, the mesh may be too coarse.
    - Similarly, if there is a significant difference between PowerGraphics and full graphics stresses, the mesh may be too coarse.
    - Re-mesh with twice as many elements, re-solve, and compare the results. (But this may not always be practical.)

Stress Analysis

- **Preprocessing**
  - **Geometry**
  - **Meshing**
- **Solution**
  - **Loading**
  - **Solve**
- **Postprocessing**
  - **Review results**
  - **Check validity of solution**



H. Workshop

- **This workshop consists of two problems:**

  2A. Lathe Cutter

  2B. 2-D Corner Bracket Tutorial

  Refer to your *Workshop Supplement* for instructions.