

Exercise 1: Axle Structural Static Analysis

The purpose of this exercise is to cover the basic functionality of the Mechanical Toolbar (MTB) in the context of performing an actual analysis. Details of each command, along with an explanation of why you are using the command, are provided prior to each step.

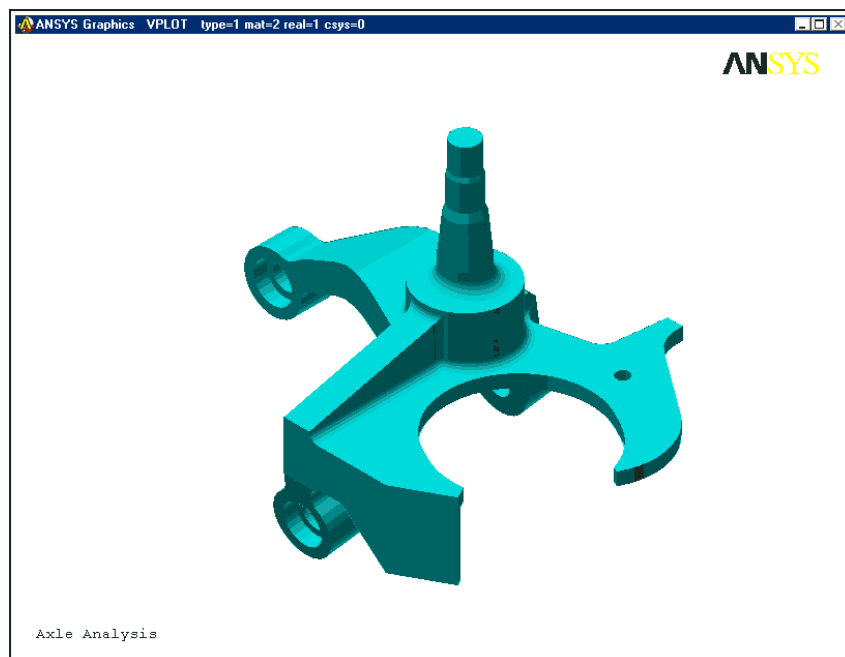
This will be a simple problem that will be performed entirely within the MTB to give the student a quick overview of the MTB functionality. This exercise will use the very minimum of input required to progress through the steps and obtain results. The model will employ the no defeature/repair option during import of a solid 3D part, MTB materials, Smartsize meshing, simple area constraints, and area force loading.

The analysis will be performed on a 3-D solid model of an automobile front wheel axle. The model has already been sufficiently defeatured to facilitate building the finite element model. You will use the MTB to determine maximum stress and deflection and if the axle can withstand the given loads without yielding.

Important: Please Read the Following

This exercise contains the step by step instructions to complete the problem described above. All instructions to the exercise are denoted by **Blue** colored text.

This exercise also contains detailed information about the functions on Mechanical Toolbar. The information that is provided covers the required functions necessary to complete this exercise. Other functions will be covered in later exercises. The information should be read before proceeding with the instructions. The detailed information is denoted by the **Black** colored text.



Outline:

1. Launch ANSYS/Professional With The MTB:

- 1.1.Launch ANSYS.
- 1.2.Activate Mechanical Toolbar

2. Setup:

- 2.1.Engineering Discipline.
- 2.2.Analysis Type
- 2.3.Unit System
- 2.4.Graphic Title
- 2.5.Toolbar Properties

3. Model:

- 3.1.Model Preparation
- 3.2.Importing Models
- 3.3.Viewing The model
- 3.4.Material Properties
- 3.5.Meshing the Model with SmartSizing
- 3.6.The Mesh Tool

4. Loads And Boundary Conditions:

- 4.1.Environment
- 4.2.Adding or Deleting Loads and B.C.'s

5. Solve:

- 5.1.Solve Now
- 5.2.Solve Later

6. Linear Elastic Solution Results

- 6.1.Results Item
- 6.2.Results Display
- 6.3.Reports

7. Conclusion

8. Additional Functions

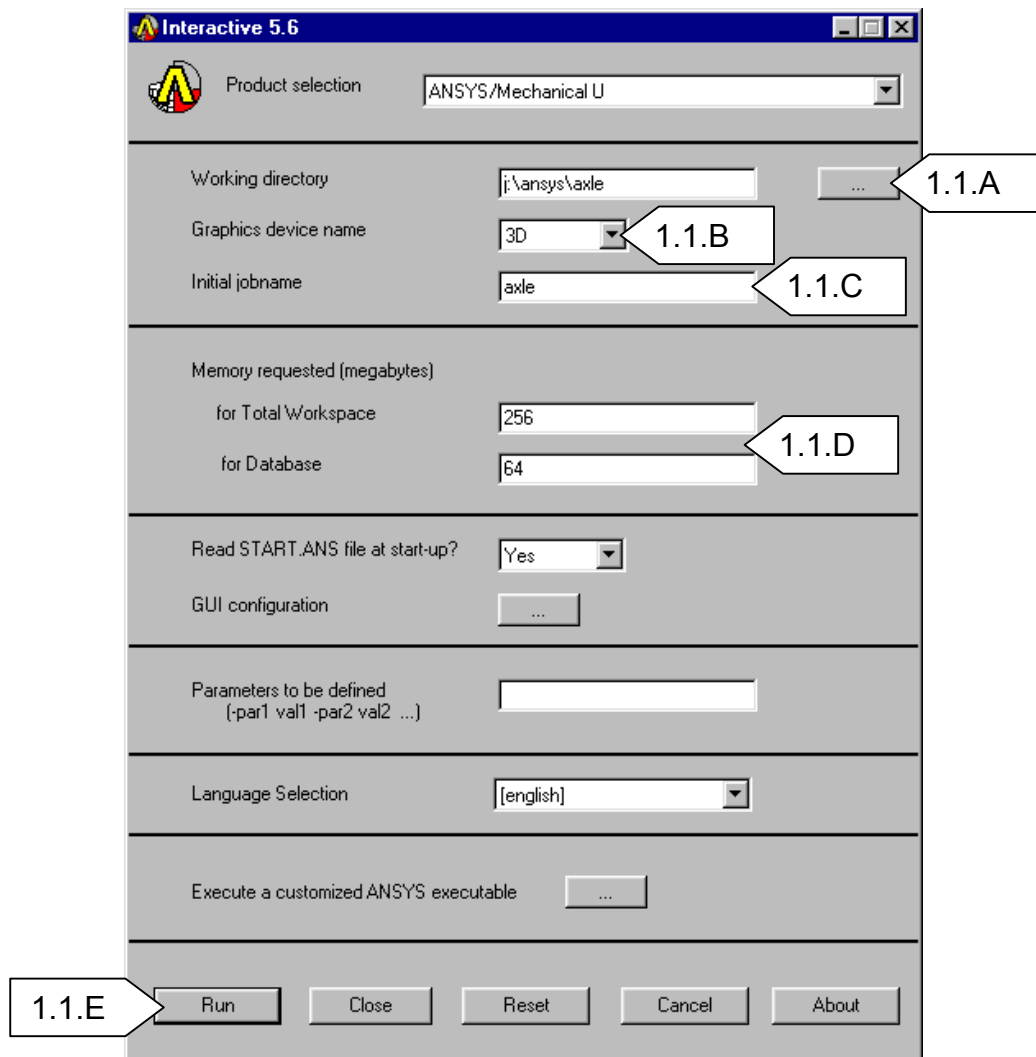
Step-by-step Instructions:

Before beginning this problem, create a separate folder on your computer for this job and copy the **axle** Parasolid part file (**axle.xmt_txt**) to this folder.

1. Launch ANSYS/Professional and Activate the Mechanical Toolbar (MTB)

1.1. Launch ANSYS using your start menu.

- A. Browse to select the working directory you just created for this job.
- B. Change the Graphics device name to **3D**
- C. Enter a job name (**axle**). All ANSYS files created for this problem will have a filename of axle followed by a unique extension.
- D. Change the *Memory Requested for Total Workspace* and for *Database* sizes for this job to be **256** and **64** respectively.
- E. Click **RUN** to start the ANSYS GUI.

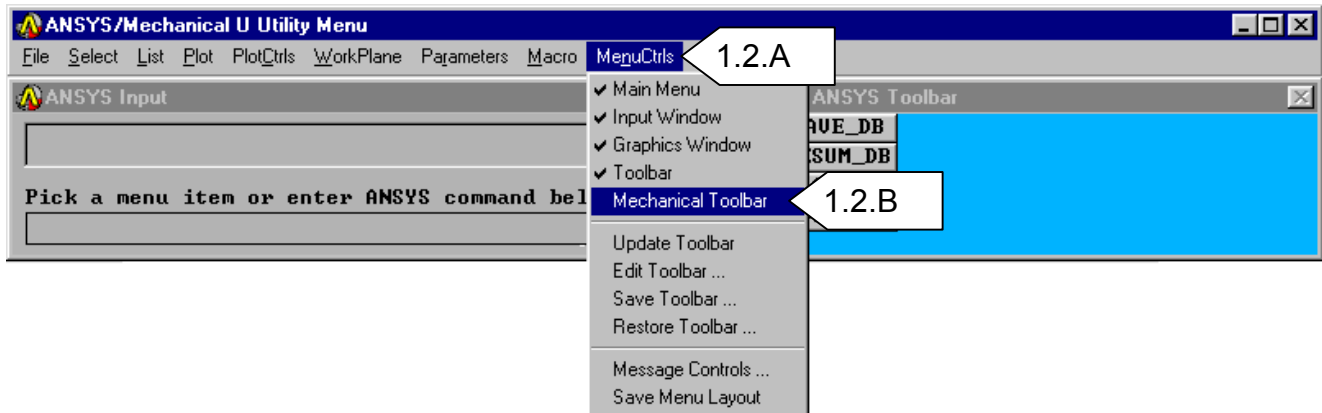


The following step is required only if you are **not** running ANSYS Professional.

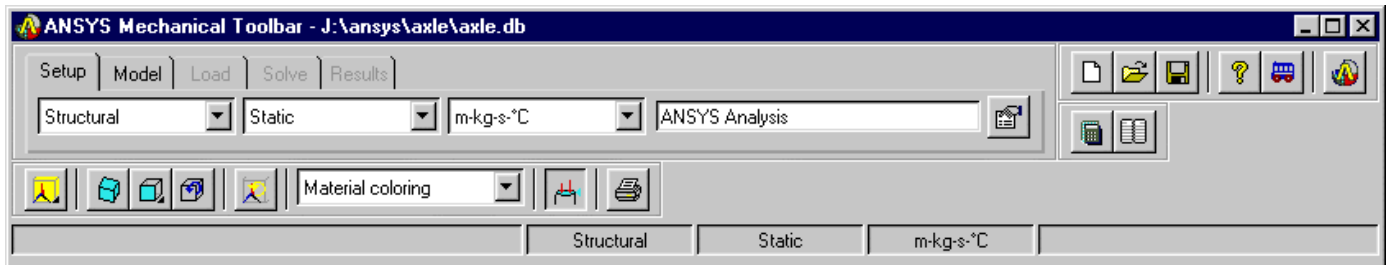
1.2. Activate the Mechanical Toolbar (MTB).

A. Click on **MenuCtrls**

B. Click on **Mechanical Toolbar**. The Mechanical Toolbar (MTB) will now appear and replace the Main Menu, Input Window, and the Toolbar.



By default the MTB will first display the Setup tab. This is where you will specify the analysis settings. The MTB options will vary depending on which Tab is currently active.

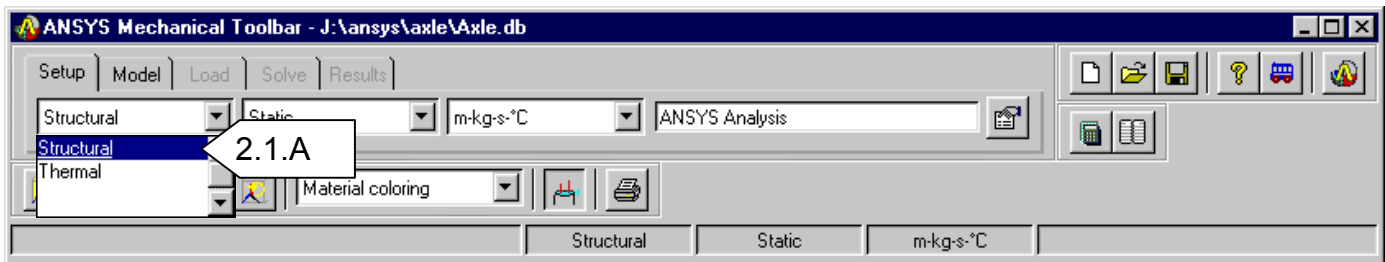


2. Setup

On the **Setup** tab, you are required to specify the engineering discipline, analysis type, unit system, and the graphics title for the analysis. You also have options for setting your MTB preferences and inputting user information.

2.1. Set the Engineering Discipline

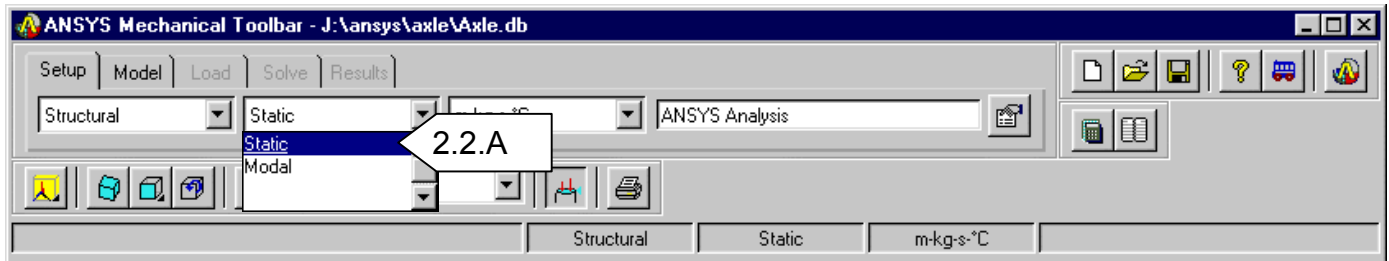
To set the engineering discipline, click on the Engineering Discipline drop down list and select either Structural or Thermal. The selected option will directly affect which options are presented to you in the *Analysis Type* drop down list.



A. Set the *Engineering Discipline* to Structural

2.2. Analysis Type

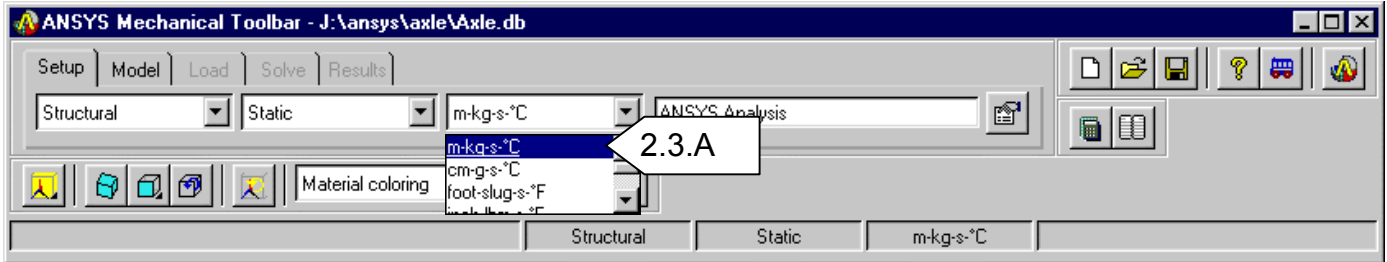
To set the analysis type, click on the Analysis Type drop down list and select the type. If the *Engineering Discipline* is set to *Structural*, you have the options of either Static or Modal. If the *Engineering Discipline* is set to *Thermal*, then you only have the option of Steady-State.



A. Set the *Analysis Type* to Static

2.3. Unit System

To set the unit system, click on the Unit System drop down list and select the unit system that you want to work in. You also have the option to define a new unit system. The unit system that you choose has no effect on existing data or completed analyses. It is only used to define labels on the plots for convenience. If you change units during the modeling process, ANSYS does not perform a unit conversion for you. That is your responsibility. Be consistent with your units.



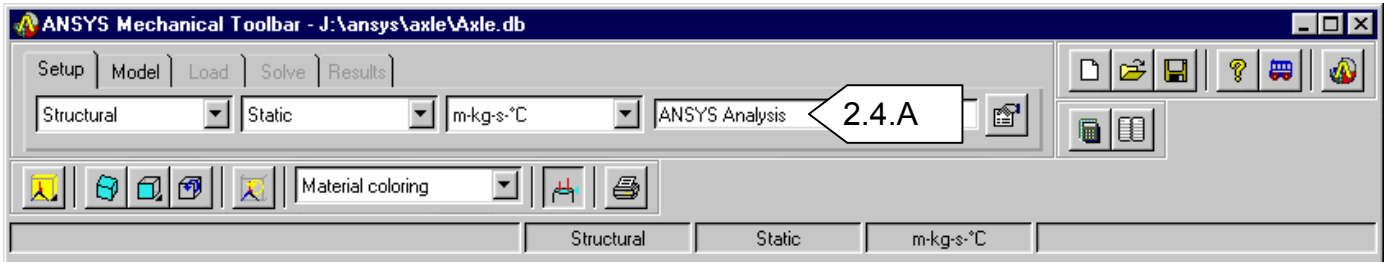
The unit system options include:

- m-kg-sec-°C
- cm-g-sec-°C
- foot-slug-sec-°F
- inch-lbm-sec-°F
- mm-kg-sec-°C
- Add Unit System

A. Set the *Unit System* to **m-kg-s-°C**

2.4. Graphic Title

To name the analysis, click in the Graphic Title input window, delete the default name (ANSYS Analysis), and type in the new name. You are allowed up to 72 alphanumeric characters.

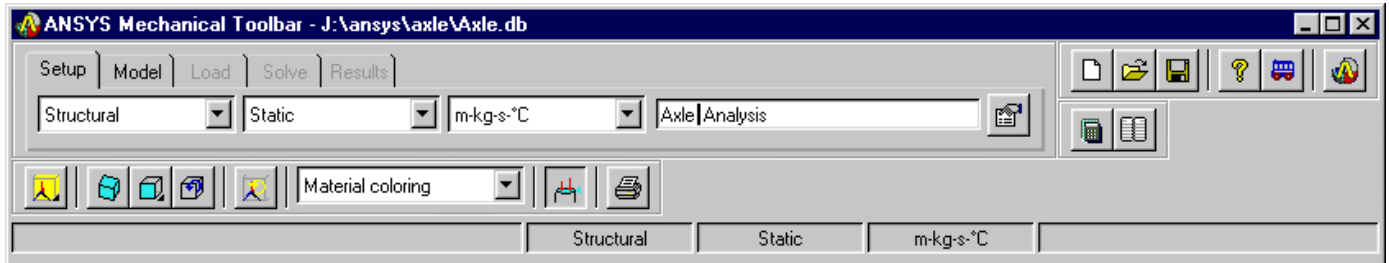


A. Enter a *Graphic Title* for the analysis: **Axle Analysis.**



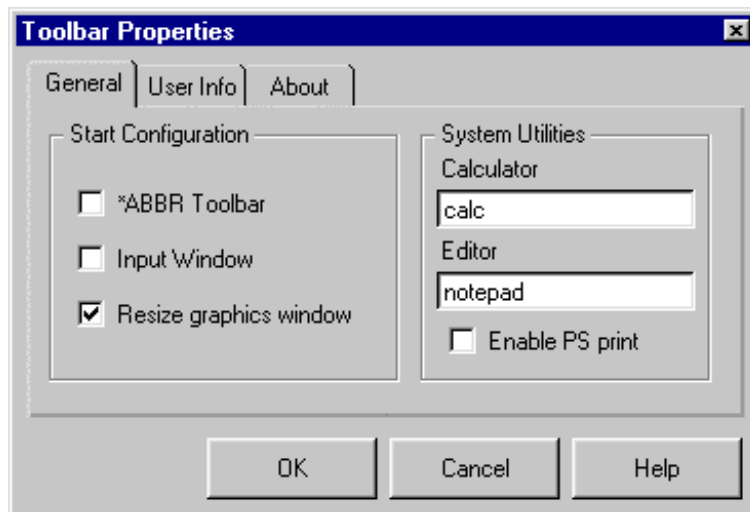
2.5. Toolbar Properties

Toolbar Properties button contains options to allow you to affect the behavior of the MTB. To modify these options, click on the Toolbar Properties button. Three tabs appear on the dialog box: General, User Info, and About.

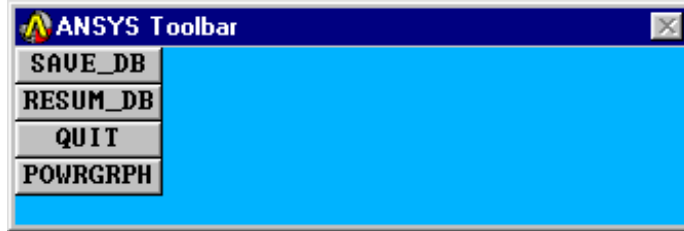


General Tab

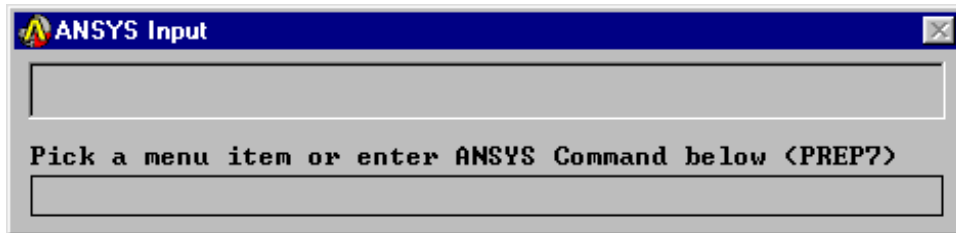
- Click the **General** tab to change the start-up configuration of the MTB, to define a system calculator or system editor, or to enable PostScript printing. (for Windows NT systems only)
- If the **ABBR Toolbar* or the *Input Window* has been turned off via the *MenuCtrls* in the full ANSYS menu system, you must turn off the Mechanical Toolbar and then turn it back on after choosing either of these configuration options from the Toolbar Properties dialog box. Otherwise, they will not appear in the interface.




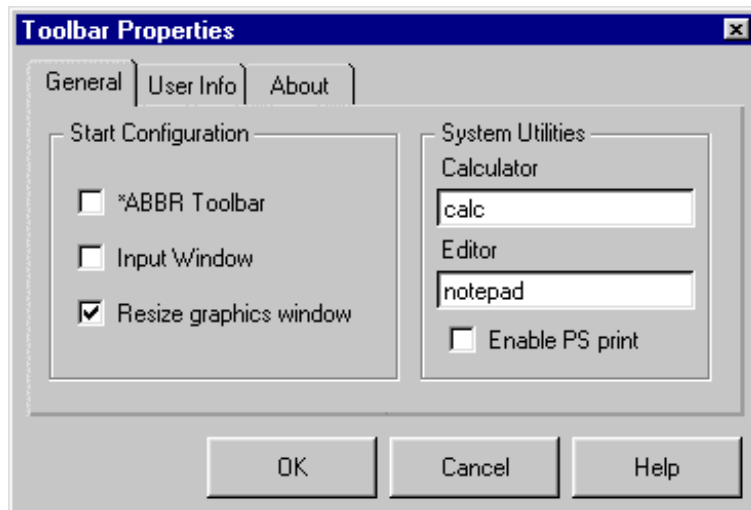
- Toggling ON the ***ABBR Toolbar** allows you to access the ANSYS GUI component known as the ***ABBR Toolbar**. This toolbar contains a set of push buttons that execute commonly used ANSYS functions. You can customize this toolbar by adding or removing functions.




- Toggling on the **Input Window** allows you to access the ANSYS GUI component known as the Input Window. Using this window, allows you to input commands directly to ANSYS.



- The **System Utilities-Calculator** allows you to input the name of your preferred system calculator in the text entry box. The MTB will start the defined calculator when you click the System Calculator button on the MTB. 

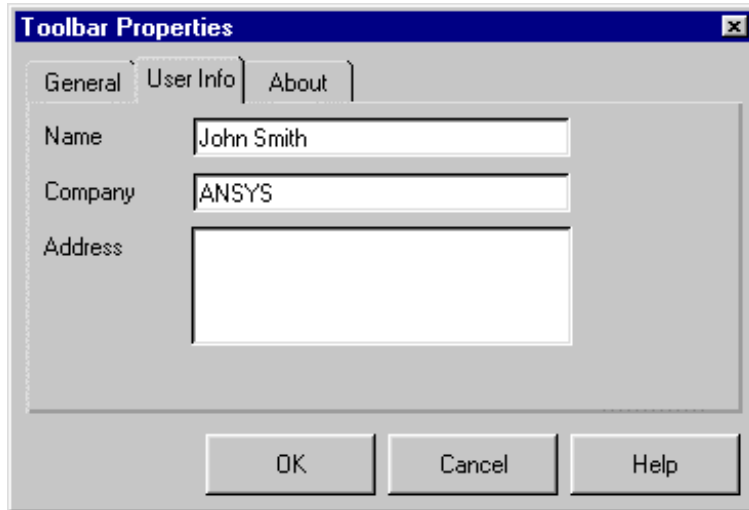


- The **System Utilities-Editor** allows you to input the name of your preferred system text editor. The Mechanical Toolbar will invoke the defined editor when you click the System Editor button on the MTB. 

- The **System Utilities-Enable PS print** (Windows NT systems only). Check this box if you are using a PostScript-enabled printer for better print quality.

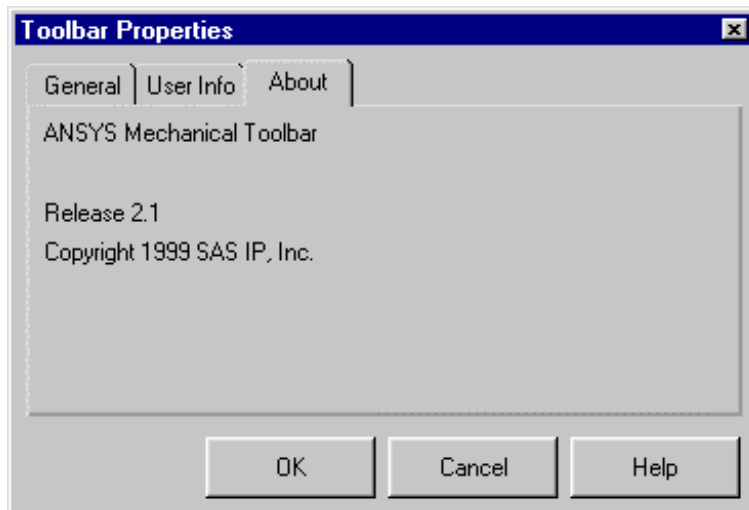
User Info Tab

- Click the **User Info** tab to provide user information for Analysis. You can record your name, company name, and company address in the corresponding text entry boxes. The Mechanical Toolbar uses these entries when creating the report.



About Tab

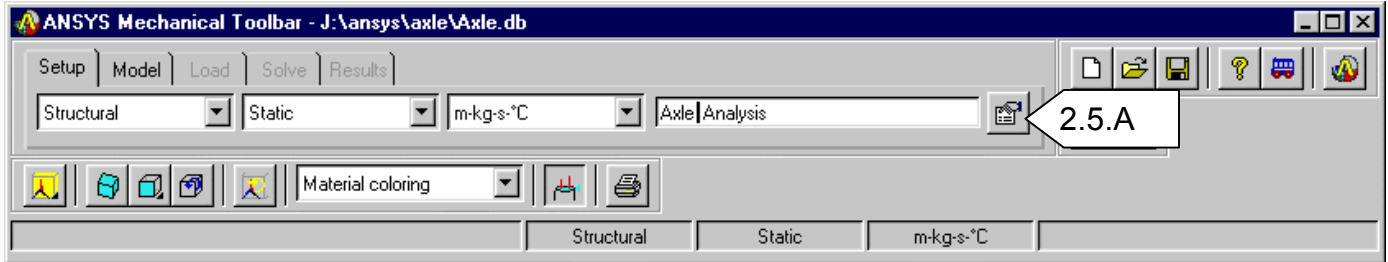
- Click the **About** tab to verify which version of the Mechanical Toolbar you are using. Just in case you forgot.



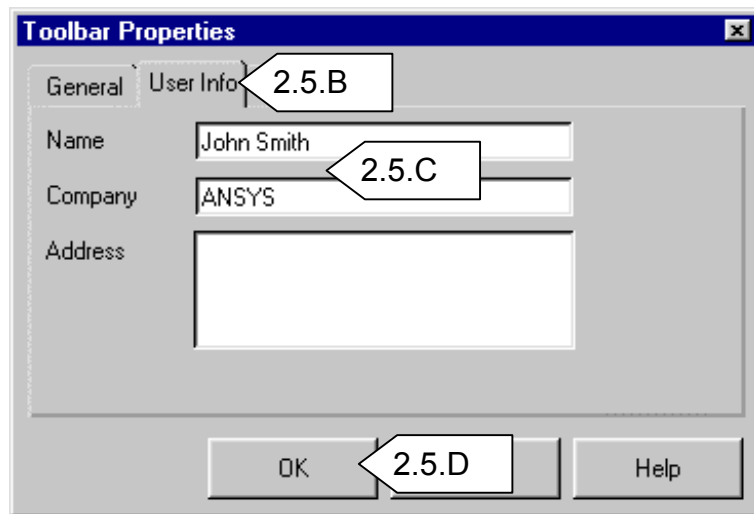
- To put your changes into effect and exit the Toolbar Properties dialog box, click **OK**. To put any Start Configuration changes into effect, you must also restart the Mechanical Toolbar.



A. Click on the **Toolbar Properties** button.



B. The Toolbar Properties dialog will appear. Click on the **User Info** tab.



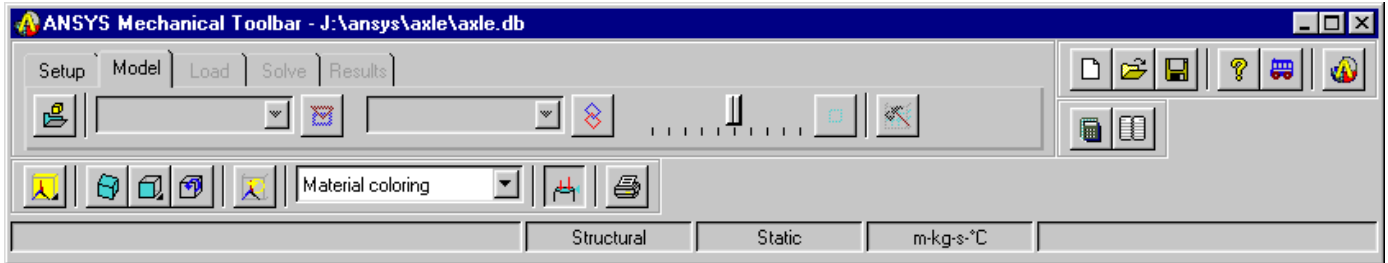
C. Enter your name, and if you want, company name and address.

D. Click **OK** when finished.

3. Model

Under the Model tab is where you import the geometry, assign shell element thickness, material properties and mesh the part.

With the MTB you do not have the capability to create the volumes or areas, therefore the geometry must be imported.



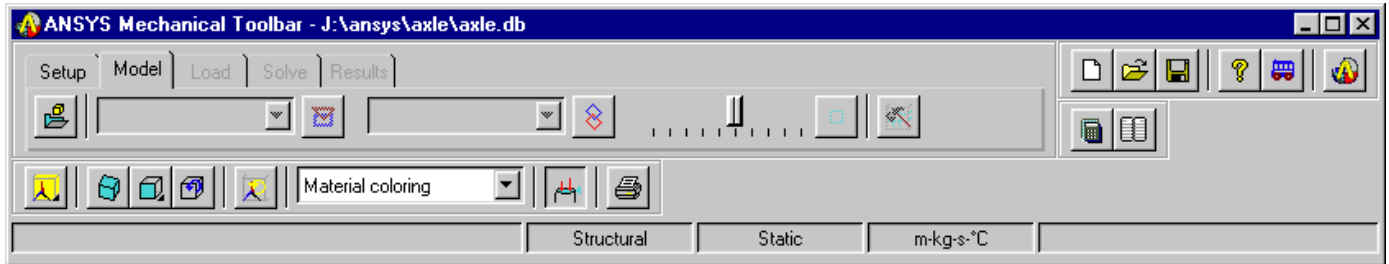
3.1. Model Preparation

The following are a few tips on how to prepare the geometry before importing it into ANSYS.

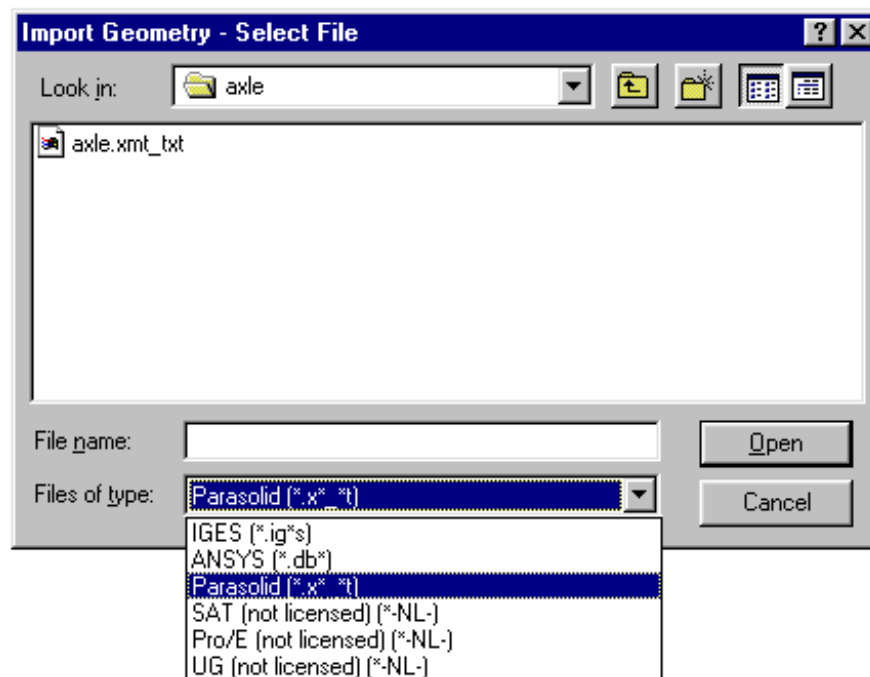
- Create a copy of the model in your CAD system. Modify the copy by simplifying the geometry to better facilitate the building of the finite element model.
- Suppress unnecessary features in the model such as external fillets, small holes, and any feature that is not essential to the analysis.
- Cut the model on all symmetry planes. Only keep the smallest unique piece that can be mirrored or copied (note that the loads and boundary conditions must also be symmetric).
- Remove or suppress everything from the model that isn't part of the model's geometry, such as dimensions, construction lines, etc.
- Either move the model near the geometric center of the coordinate system or create an output coordinate system near the geometric center of the model.
- Analyze the model and look for any small slivers or poor geometry. Remove these areas or correct the geometry.

3.2. Importing Models

- To load a model into the Mechanical Toolbar, click the **Import Geometry** button. The *Import Geometry – Select File* dialog will appear.

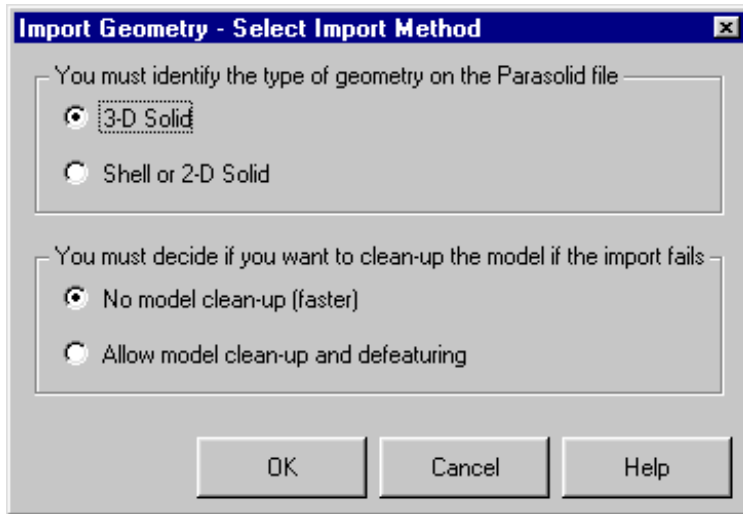


- Click on the **Files of type** drop down list and select one of the import types.



- Change the directory to the directory containing the file to be imported.
- Highlight the desired file name and click **Open**. The *Import Geometry - Select Import Method* dialog will appear.

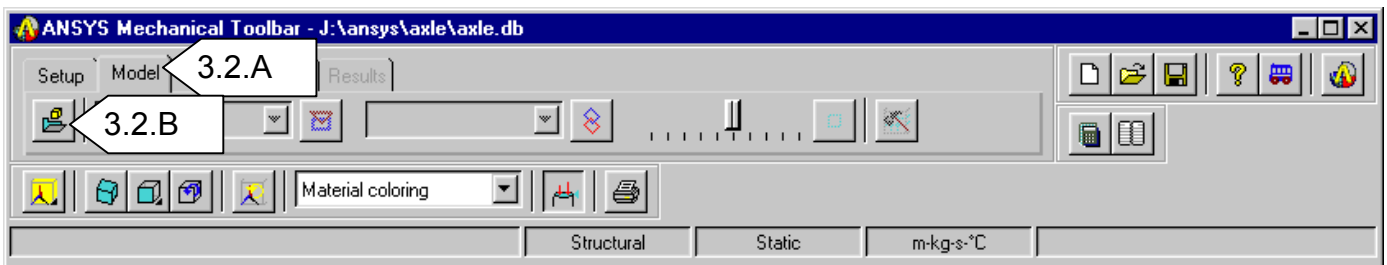
- Use the *Import Geometry - Select Import Method* dialog box to decide which method to use when importing a model. Specify whether the model is a 3-D Solid or a Shell or 2-D Solid.



- You have two additional choices to make concerning your model
 - **No model clean-up** - This is the preferred method; it's faster and more reliable.
 - **Allow model clean-up and defeaturing** - Using this method activates the defeaturing tools. A better method is to remove all unnecessary features from your model in the CAD system.
- Click **OK** to proceed with the geometry import.

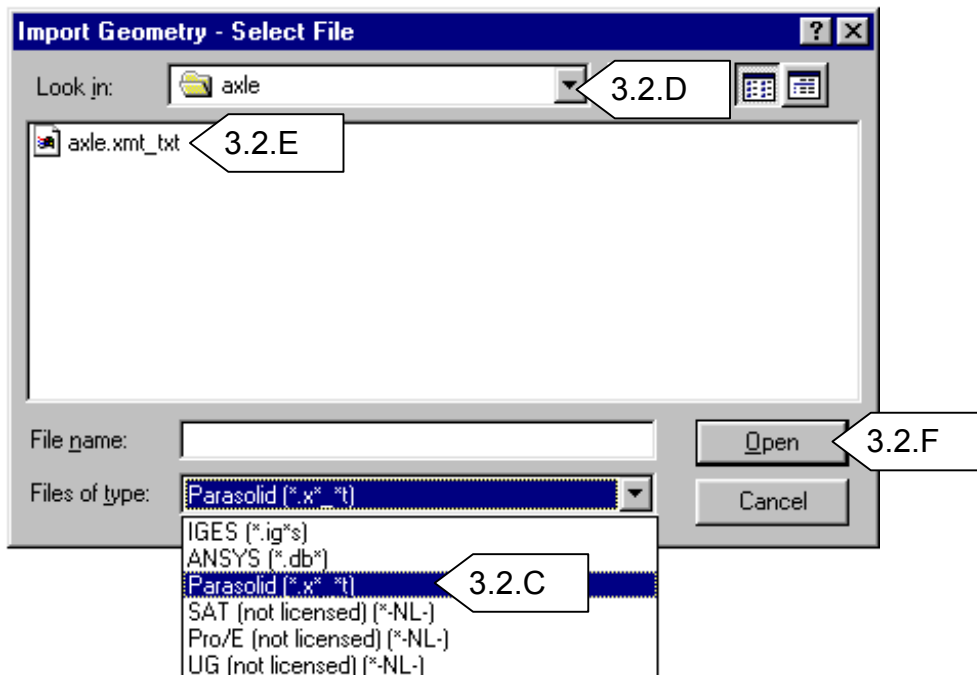


A. Import the geometry for this exercise. Click on the **Model** tab on the MTB.



B. Click on the **Import geometry** button to bring up the *Import Geometry - Select File* dialog box.

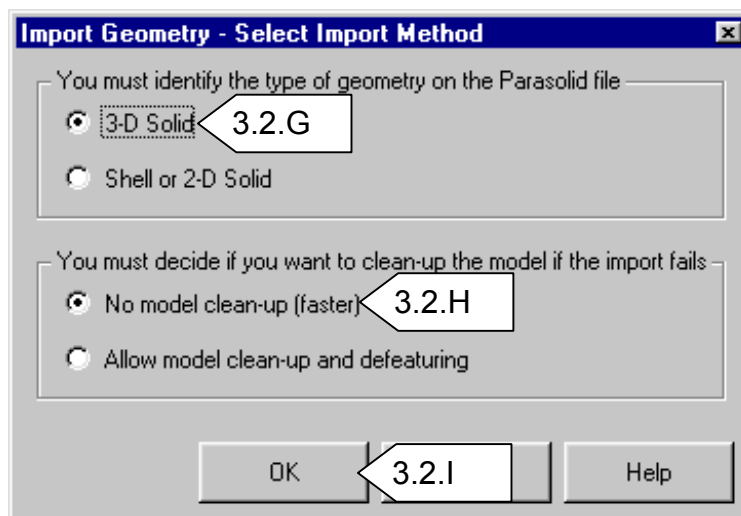
C. Change the **Files of type:** option to **Parasolid**.



D. Change the directory to the directory containing the axle parasolid.

E. In the file list, highlight the file named **axle.xmt_txt**.

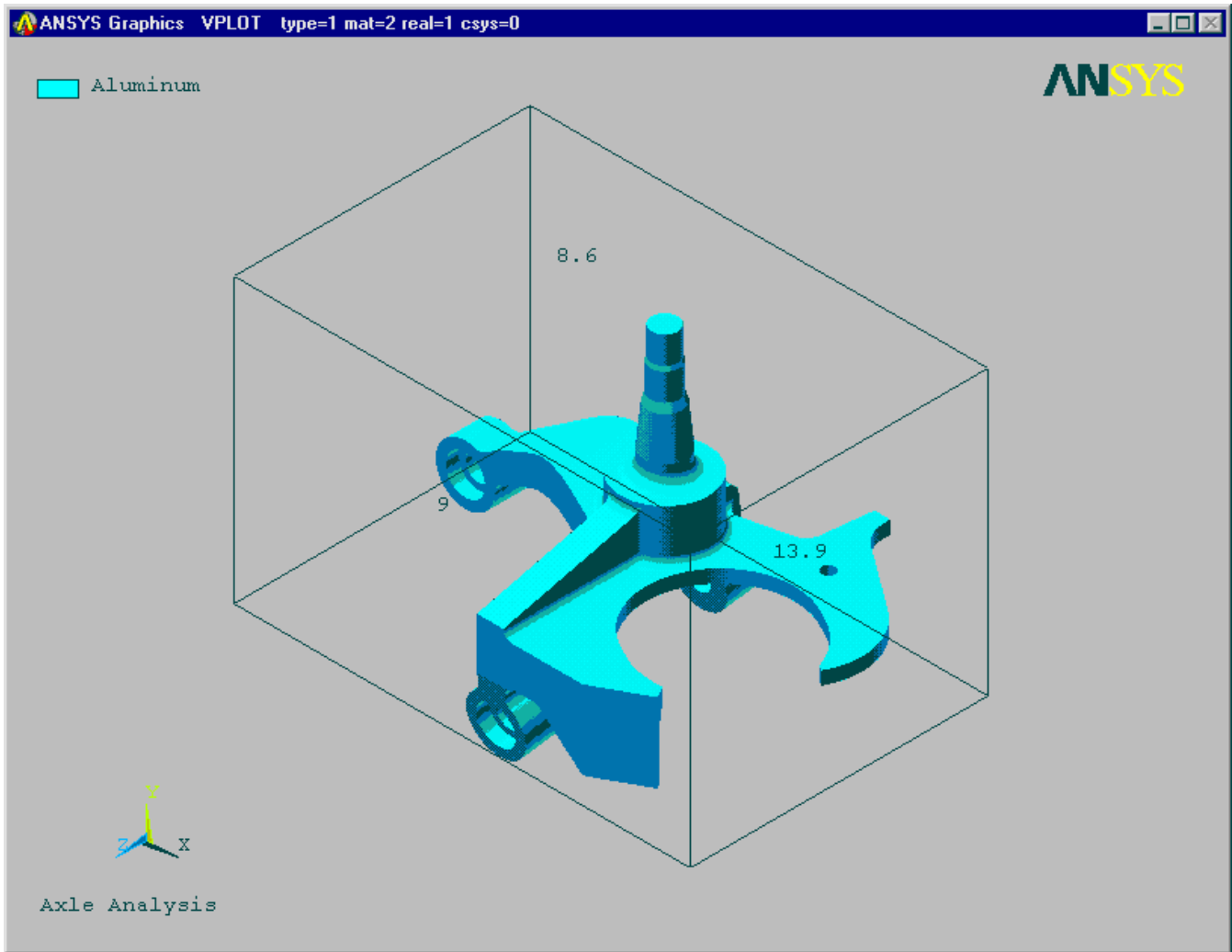
F. Click **Open**. The Import Geometry - Select Import Method dialog will appear.



G. Select **3-D Solid**.

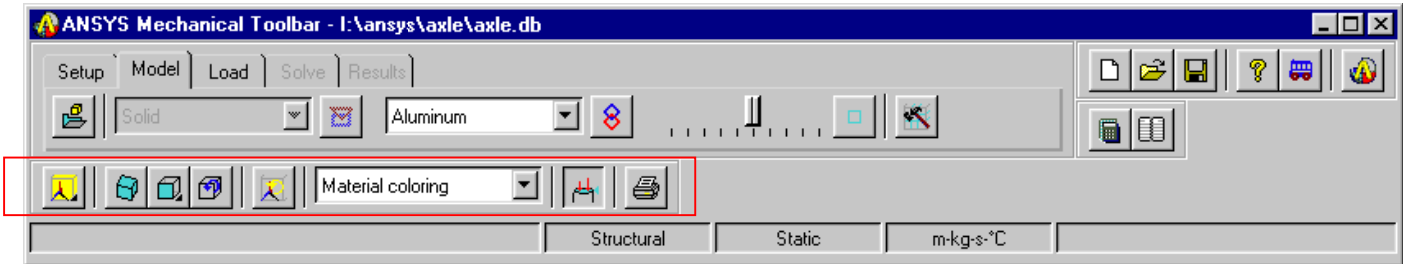
H. Leave the **No model clean-up (faster)** option set.

I. Click **OK**. ANSYS will import the Parasolid file and draw the part in the graphics window.



3.3. Viewing the model

There are several buttons on the Mechanical Toolbar that allow you to manipulate graphic plots. These buttons and their functions include:



- **Plot:** To access a fly-out toolbar of plotting options, do the following:
 - Position the cursor on the **Plot** button.
 - Press and hold the left mouse button until the fly-out toolbar appears.



- The fly-out toolbar contains the following buttons: Keypoint Plot, Line Plot, Area Plot, Volume Plot, Node Plot, and Element Plot.
- Release the left mouse button.
- Position the cursor on the desired fly-out button and click.

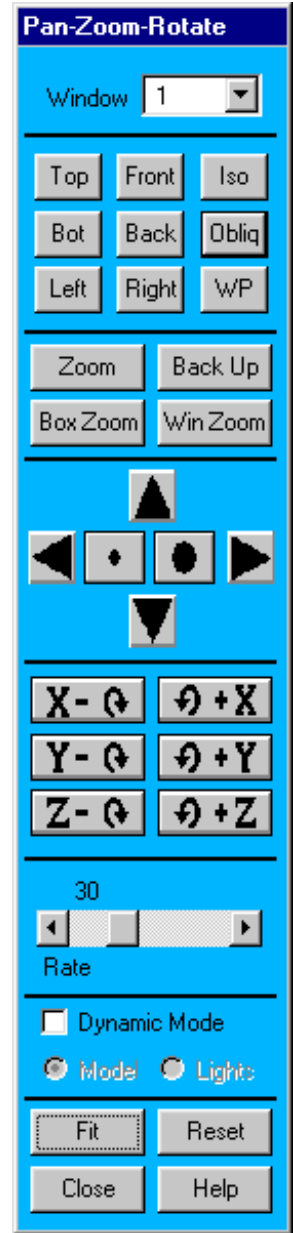
- **Oblique View:** Sets the view of the graphic window to oblique.
- **View:** Sets the view of the graphic window. To access a fly-out toolbar:
 - Position the cursor on the **View** button.
 - Press and hold the left mouse button until the fly-out toolbar appears.



- The fly-out toolbar contains the following View option buttons: Top View, Bottom View, Front View, Back View, Left View, Right View, Oblique View, Working Plane View, and Isometric View.



- **Pan-Zoom-Rotate:** This option activates the Pan-Zoom-Rotate control, which allows you to manipulate the model view by panning, zooming, or rotating the model. **Note:** This is a good option to leave on while working on the model.
 - To change the view, click on the desired view button (Top, Front, Iso, etc)
 - You can choose a Zoom method. Your options include: Zoom, Box Zoom, Win Zoom, and Back Up (UnZoom).
 - Pan/Zoom buttons: Pan up/down left/right using the arrows. Small dot zooms out and the large dot zooms in. The **Rate** slider located below the rotate buttons controls the amount of change in model size from zooming in or out.
 - The **Rotate** buttons allow you to define the axis of rotation. The rate slider that is below the rotate buttons controls the amount of rotation.
 - **Dynamic Mode:** This option allows you to use your mouse buttons to pan/zoom/rotate. When this is on, the left mouse button pans, the middle mouse button zooms, and the right mouse button rotates.
 - **Lighting:** This option allows you to control the light source location, intensity, and the reflectance of your model. When this option is ON the mouse buttons function as follows: Left Button: Move in X-direction to increase or decrease the reflectance of your model. Move in the Y-direction to change the intensity of the directional lights (there are two directional lights, one in front and one behind your model). Middle Button: Move in the X-direction to rotate the directional lights about the screen Z-axis. Move in the Y-direction to change the intensity of the ambient light. Right Button: Move in the X-direction to rotate the directional lights about the screen Y-axis. Move in the Y-direction to rotate the lights about the screen X-axis.
 - **Fit:** Automatically adjusts that amount of zoom in the active window so that your entire model can be seen within the window.
 - **Reset:** Removes any Pan-Zoom-Rotate changes in the active window. Your model will be displayed in its default orientation and sized to fit within the active widow.
 - **Close:** Dismisses the Pan-Zoom-Rotate dialog box.



- Alternatives to using the Pan-Zoom-Rotate control :
 - **Pan** – Hold down the **CTRL** key and press the left mouse button. Move the mouse left, right, up, or down to move the model in the desired direction.
 - **Zoom** – Hold down the **CTRL** key and press the middle mouse button. Move the mouse up to zoom in. Move the mouse down to zoom out. If you are using a two-button mouse, you may be able to mimic the action of a middle mouse button by holding down the **SHIFT** key and pressing the right mouse button.
 - **Rotate about Z axis** – Hold down the **CTRL** key and press the middle mouse button. Move the mouse left or right to rotate the model about the Z axis. If you are using a two-button mouse, you may be able to mimic the action of a middle mouse button by holding down the **CTRL-SHIFT** keys and pressing the right mouse button.
 - **Rotate about X or Y axis**–Hold down the **CTRL** key and press the right mouse button. Move the mouse to rotate the model about the X (up/down) or Y (left/right) axis.



- **Redraw Model:** Redraws (refreshes) the contents of the graphic window.

- **Render Model:** This option allows you to set the render options for the model. The render options include material coloring, material texturing, thickness, translucency, wireframe, gray scale, and show edges.

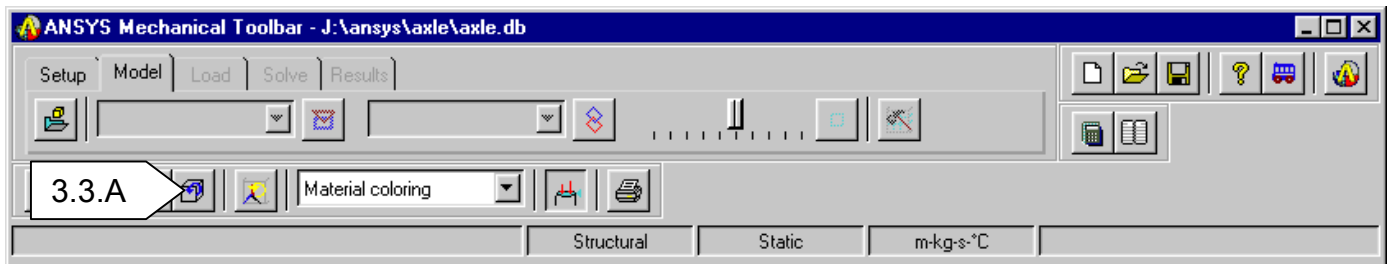


- **Boundary Conditions:** Toggles boundary condition symbols on and off.

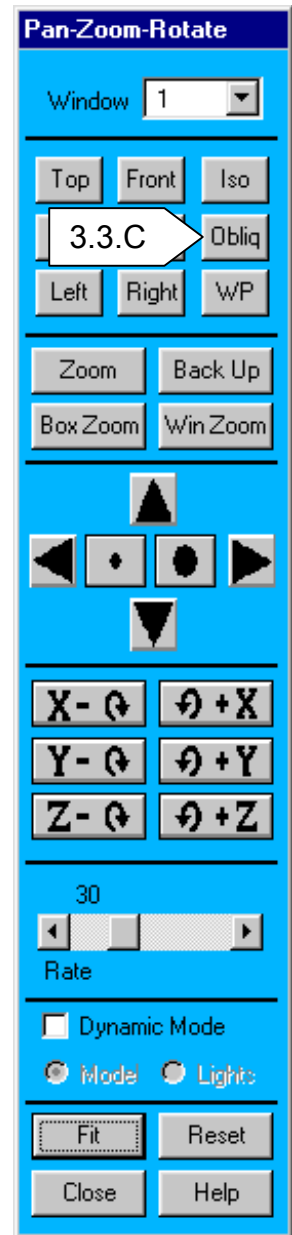
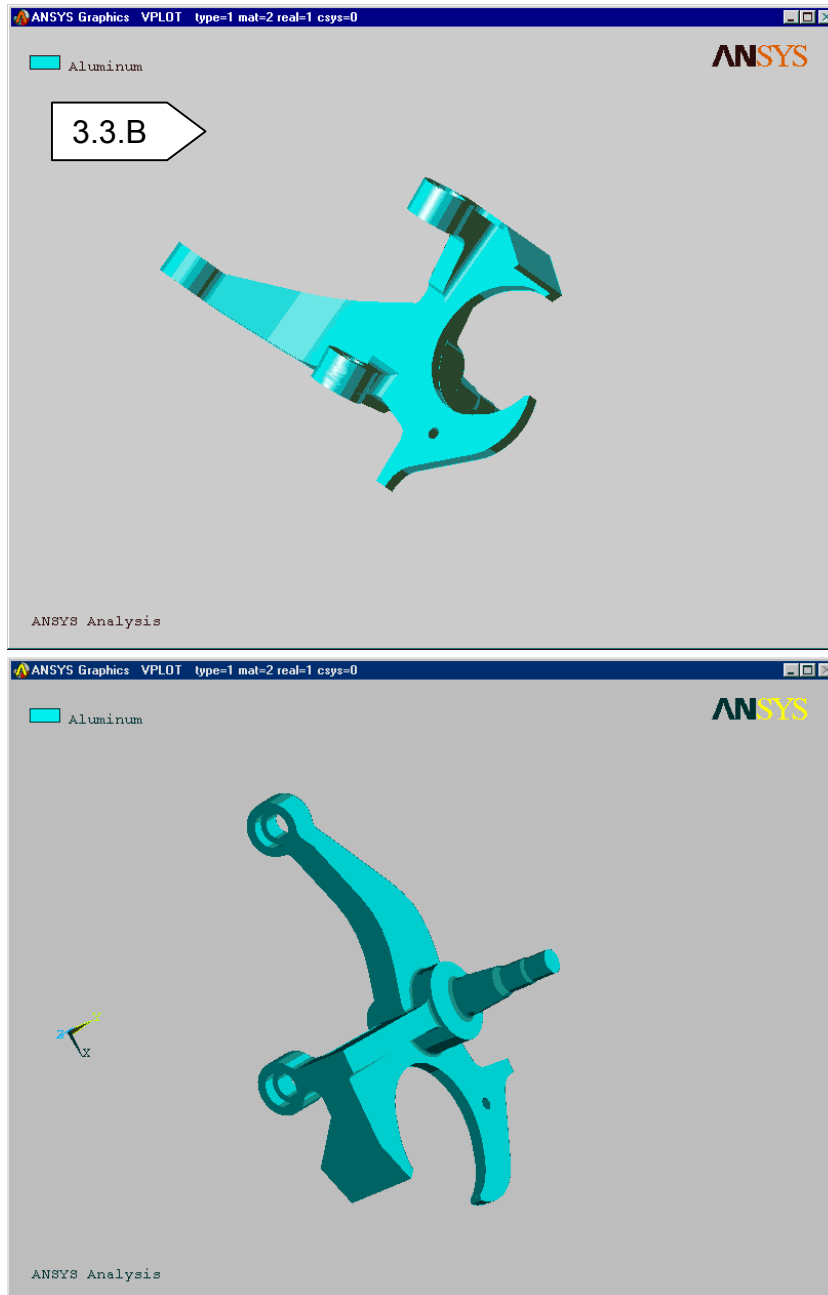


- **Print Hardcopy:** Prints a hardcopy of the graphic window contents.

A. Select the **Pan-Zoom-Rotate** button. The Pan-Zoom-Rotate dialog will appear .



- B. Thoroughly inspect the model by Panning, zooming, and rotating the model. Play with the light source option.
- C. Return the view to **Oblique** when you are finished. Leave the Pan-Zoom-Rotate control open for the rest of the exercise.



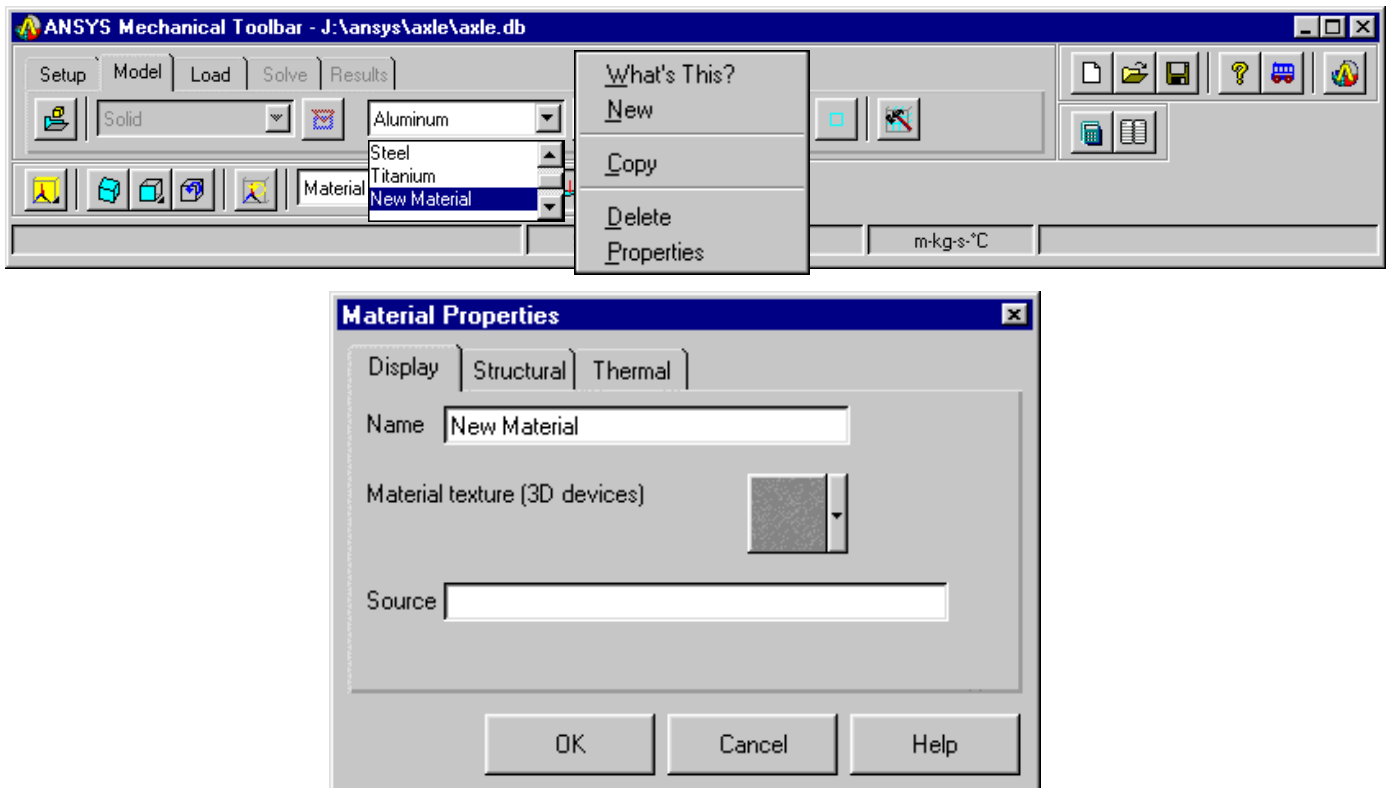
3.4. Material Properties

By default, there are three materials that are available for use with the Mechanical Toolbar: these are ANSYS-supplied aluminum, steel, and titanium. In addition to these ANSYS-supplied materials, you can add other materials to create your own personal material library.

Creating a Personal Material Library

To create a new material that will be added to your personal material library, you need to be on the Model tab. Follow these steps:

- Click on the **Default Material** drop down list and select **New Material**, or place the cursor over the *Default Material* drop down list, click the right mouse button and select **New** from the list of options. The Material Properties dialog box appears. Three tabs appear on the dialog box:



Display, Structural, and Thermal.

- On the **Display** tab, type a unique name for the new material into the Name window. Material names are limited to 32 characters.
- Click on the arrow to the right of the **Material Texture** button. When the texture options appear, click the texture of your choice to select it. The Mechanical Toolbar will use the texture that you choose to depict the material in graphic plots.

- Use the **Source** input window to record information about the source that you referred to for the definition of the material (up to 64 characters, including spaces).
- Click on the **Structural** or **Thermal** tab (as applicable) and type in the

The image shows the 'Material Properties' dialog box with the 'Structural' tab selected. The 'Physical Constants' section contains four input fields: 'Elastic modulus' (unit: psi), 'Specific Weight' (unit: lb/in³), 'Thermal expansion coeff' (unit: 1/°F), and 'Poissons ratio'. The 'Display' tab is also visible but not selected. At the bottom are 'OK', 'Cancel', and 'Help' buttons.

The image shows the 'Material Properties' dialog box with the 'Thermal' tab selected. The 'Physical Constants' section contains two input fields: 'Conductivity' (unit: Btu/sec/in °F) and 'Specific heat' (unit: Btu/lb °F). The 'Display' and 'Structural' tabs are also visible but not selected. At the bottom are 'OK', 'Cancel', and 'Help' buttons.

physical constants that are necessary to define the new material.

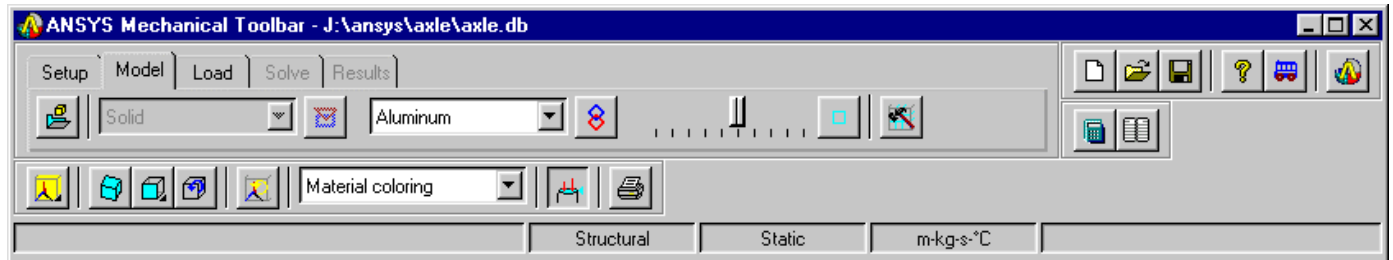
- Click **OK** to put your changes into effect and exit the Material Properties dialog box. The new material will be added to your personal material library and will always appear in the Default Material drop down list box on your system.


Copy, Delete, and Modifying Material Properties

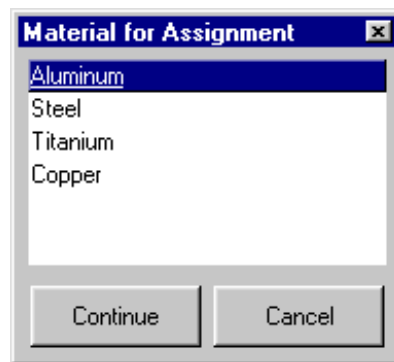
- Place the cursor over the *Default Material* drop down list, click the right mouse button and select **Copy, Delete, or Properties**.

Assigning Material Properties

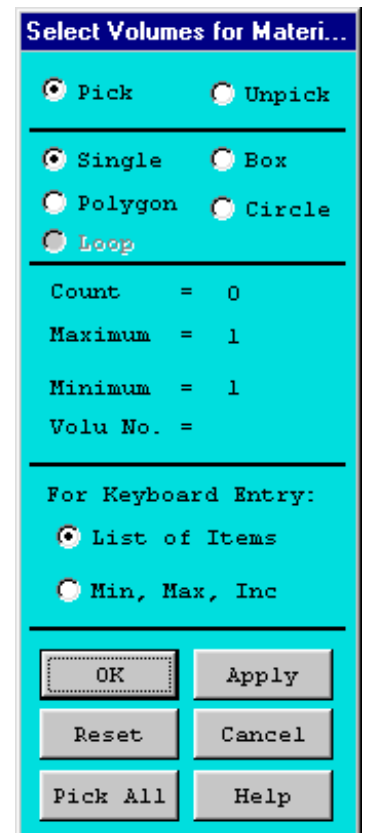
Any unassigned model entities will take on the material that is currently displayed in the Default Material drop down list. To assign a material to entities in a model, you need to be on the Model tab. Follow these steps:



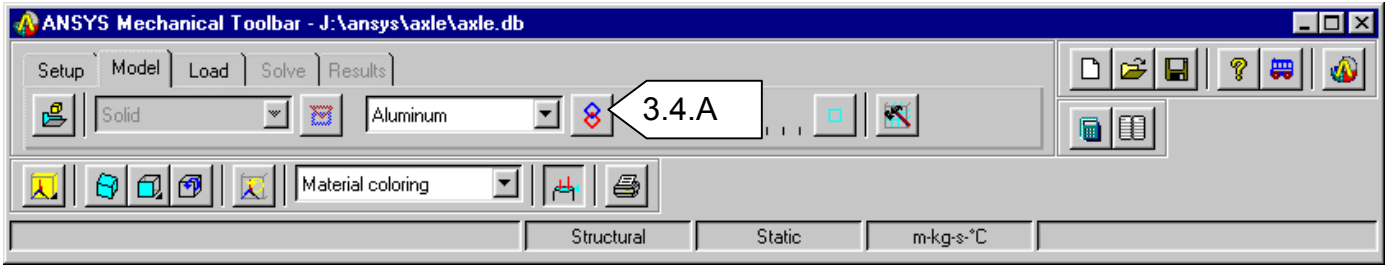
- 
 • Click on the **Assign Material** button. The Material for Assignment dialog will appear.



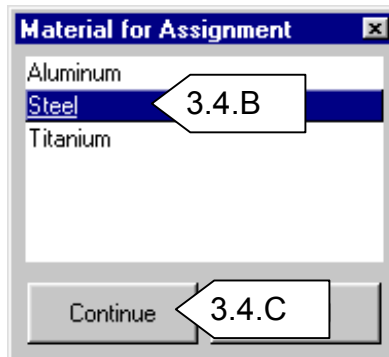
- In the *Material for Assignment* dialog box, select the desired material and then click **Continue**. The ANSYS Picker will appear.
- In the picker, make the desired picker selections (for example, Single pick, Box pick, and so on).
- In the graphic window, select those parts of the model to which you want to assign the material.
- Click **Apply** in the picker.
- Repeat steps 1 through 5 until you have assigned a material to all appropriate entities in the model.
- Click **OK** in the picker.



- A. Next we want to change the material properties from the default of Aluminum to Steel. Click on the **Assign Material** button in the MTB.

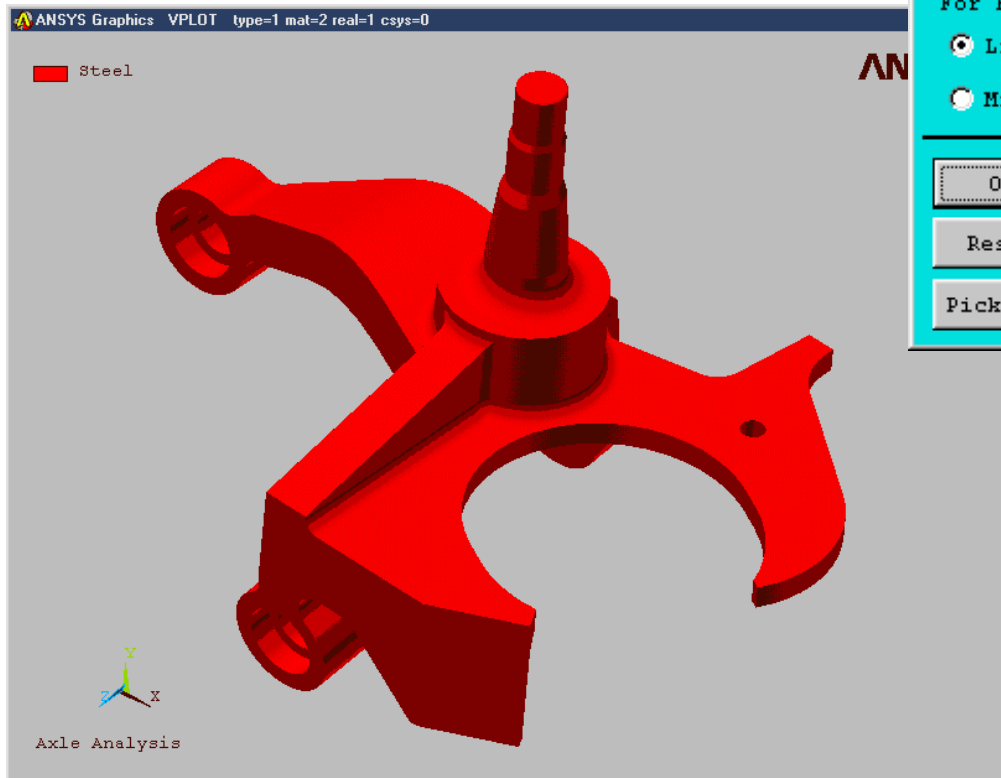


- B. ANSYS has built in properties for Aluminum, Steel and Titanium. Highlight **Steel** in the list.



- C. Click **Continue**.

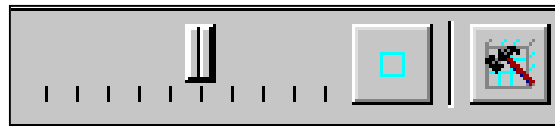
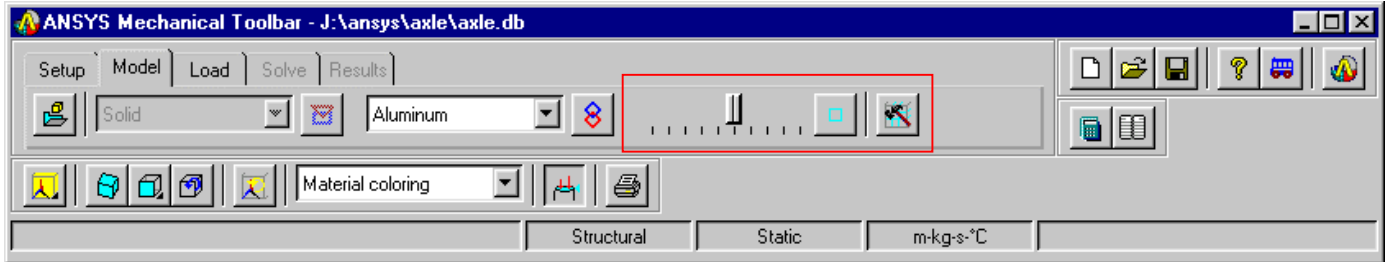
- D. The picker dialog will appear for you to select volumes for material assignment. Click the button that says **Pick all**. The plot will change to reflect the change in material.




3.5. Meshing the model with Smart Sizing

Meshing the model is performed on the Model tab.

- Click on the Mesh SmartSize slider and move it from left (fine mesh) to right (coarse mesh) to define the overall element size for the mesh. As you move the Mesh SmartSize slider, notice that the Mesh Model icon that appears to the right of the slider changes in size as you move the slider.



- You can use Smart Sizing to set different mesh sizes for different parts of a model. To do so, set a Smart Sizing level and mesh only those entities in the model that should have that level. Then continue setting levels and meshing entities (one or more entity at a time), until all entities in the model are meshed.
-  • Set the level of the Smart Size by moving the slider either to the left or the right, then select the Mesh Model button.



3.6. The MeshTool provides access to more advanced meshing controls and meshing operations.

Element Attribute Controls

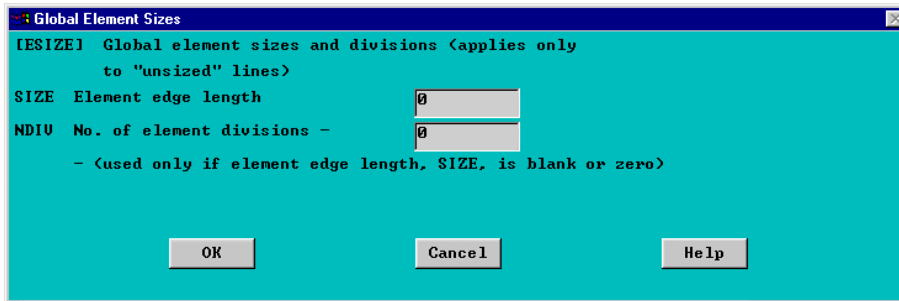
- Element Attribute Controls on the MeshTool are ignored when the Mechanical Toolbar is active.

SmartSizing Controls

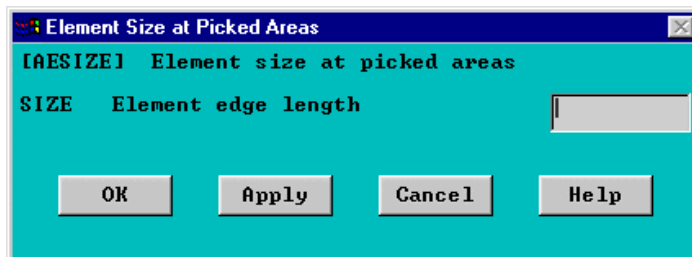
- Clicking the Smart Size check box toggles SmartSizing on and off. This option is identical to SmartSizing on the Mechanical Toolbar's Model tab.

Size Controls Provide more control over element size specifications.

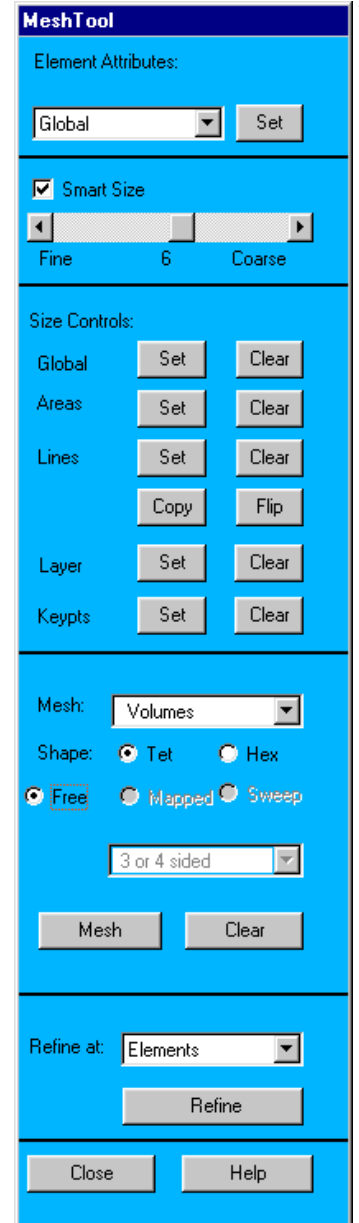
- **Global:** Controls the setting and clearing of global element edge lengths.
 - Clicking **Set** opens a dialog box for setting the global edge length



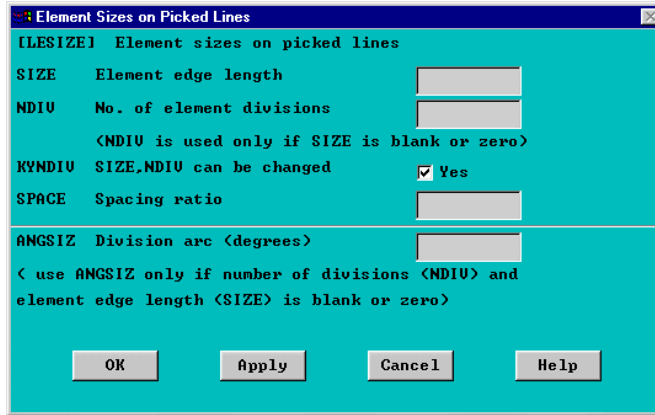
- Clicking **Clear** clears this specification.
- **Areas:** Controls the setting and clearing of element edge lengths on selected areas.
 - Clicking **Set** opens a picking dialog for selecting areas, followed by a dialog box for setting element edge lengths on the selected areas



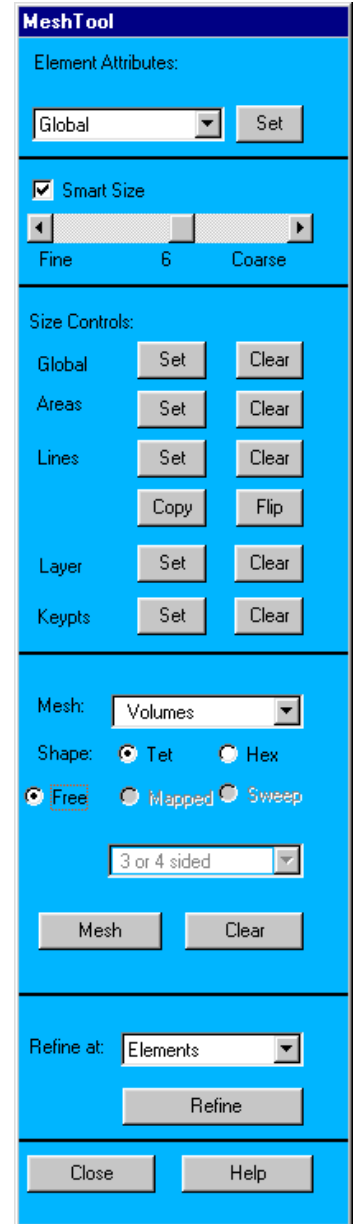
- Clicking **Clear** allows you to select areas and clear this specification.



- **Lines:** Controls the setting and clearing of the divisions and spacing ratios on selected unmeshed lines.
 - Clicking **Set** opens a picking dialog for selecting lines, followed by a dialog box for setting divisions.



- Clicking **Clear** opens a picking dialog for selecting lines to be cleared of divisions.
- Clicking **Copy** opens a picking dialog to let you copy line divisions (including spacing ratios) from one line onto other unmeshed lines. First, pick the line from which divisions will be copied, then click OK in the picking dialog. Next, pick the lines to which the divisions should be copied, then click OK in the picking dialog. If previously set line divisions exist, the copied divisions overwrite them.
- Clicking **Flip** opens a picking dialog to let you flip the spacing ratio of line divisions (from one end to the other) on an unmeshed line. First pick the line to be flipped, then click OK in the picking dialog.
- **Layer:** Layer meshing is used in the modeling of fluid flow. It is not of use to Mechanical Toolbar users.



- **Keypoints:** Controls the setting and clearing of the edge lengths of the elements near a selected keypoint or keypoints. Clicking Set opens a picking dialog for selecting keypoints, followed by a dialog box for setting edge lengths. Clicking Clear opens a picking dialog for selecting the keypoints for which you wish to clear the keypoint sizing specifications.

Meshing Operation Controls

- **Mesh:** Controls which type of entity is being meshed. For Mechanical Toolbar users, only Volumes (for solids) and Areas (for shells and planes) are valid choices.

- **Shape:** Controls the shape of the elements used to create a mesh (quadrilateral, triangle, hexahedral, or tetrahedral). Mechanical Toolbar users should leave this control set to Tet when meshing volumes, and Quad when meshing areas.
- **Mesh:** Starts the meshing operation. Clicking Mesh opens a picking dialog that lets you select the entity to be meshed. You can also start a meshing operation by clicking the Mesh Model button on the Model tab of the Mechanical Toolbar.
- **Clear:** Clears the selected volumes, areas, lines, and keypoints (i.e., vertices) of their meshes. Clicking Clear opens a picking dialog that lets you select the entity to be cleared.

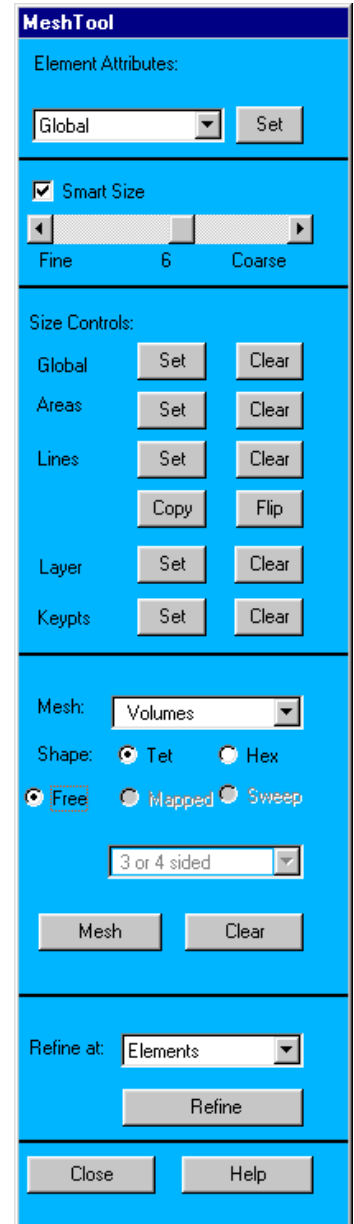


Refinement Controls

- **Refine at:** Controls the general location at which mesh refinement occurs. Clicking the drop down list box causes a list of available choices—Nodes, Elements, KeyPoints, Lines, Areas, and All Elems—to appear.
- **Refine:** Starts the refinement operation. Clicking Refine opens a picking dialog box that lets you select the specific area(s), line(s), etc. at which you want refinement to occur.

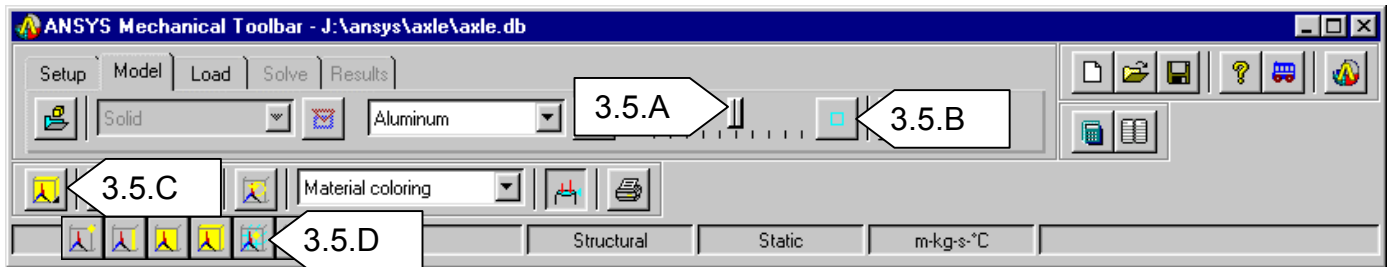
Close Closes the MeshTool.

Help Displays the full ANSYS product's help for the MeshTool.

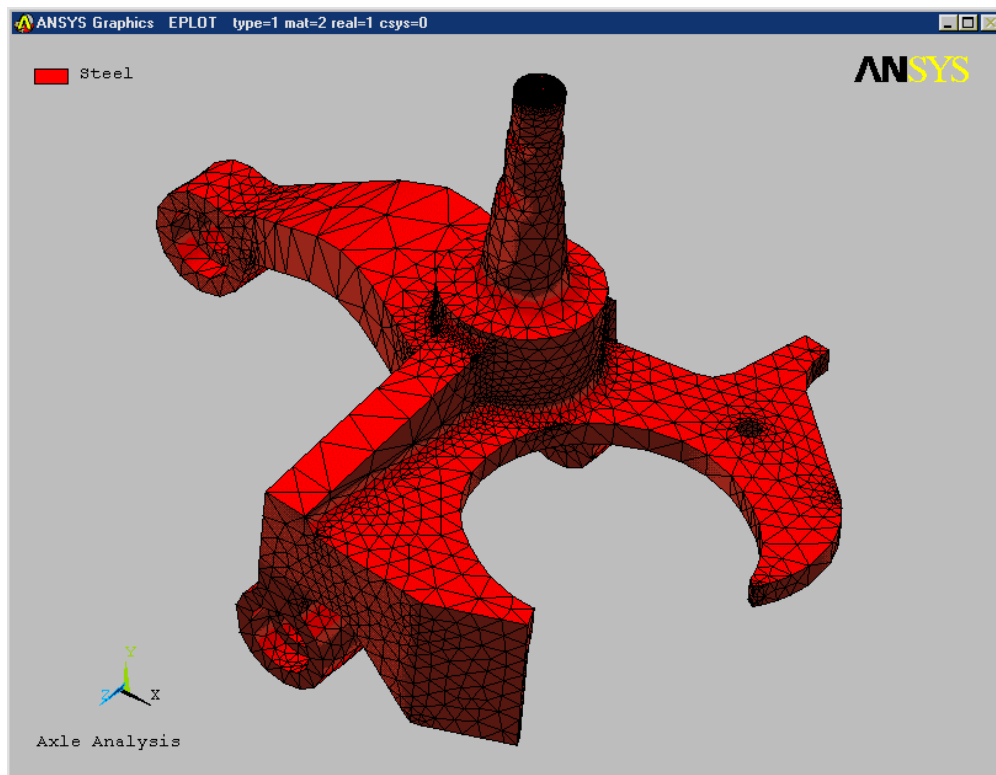


Meshing: We will use the default SmartSize meshing to create a mesh on the part. The resultant mesh will be good enough to run the preliminary analysis.

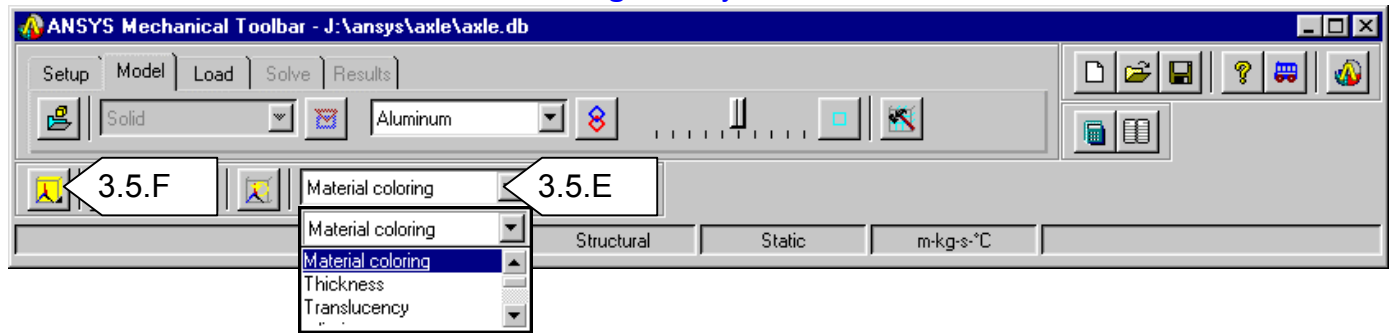
- A. The slider bar in the MTB controls the SmartSize mesh density in various levels from very fine (left most setting) to very course (right most setting). We will use the default (center setting).
- B. Since there is only 1 volume in the model, click on the **Mesh Model** button to create the mesh. The ANSYS meshing process may take a few minutes. When meshing is complete, the mesh will appear in the graphics window.
- C. Take a look at the nodes. In the MTB, click and hold on the **Plot** button momentarily until the fly-out options appear.



- D. Click on the **Plot Nodes Button**. Notice that the Render Model option has changed to **Show Edges**.



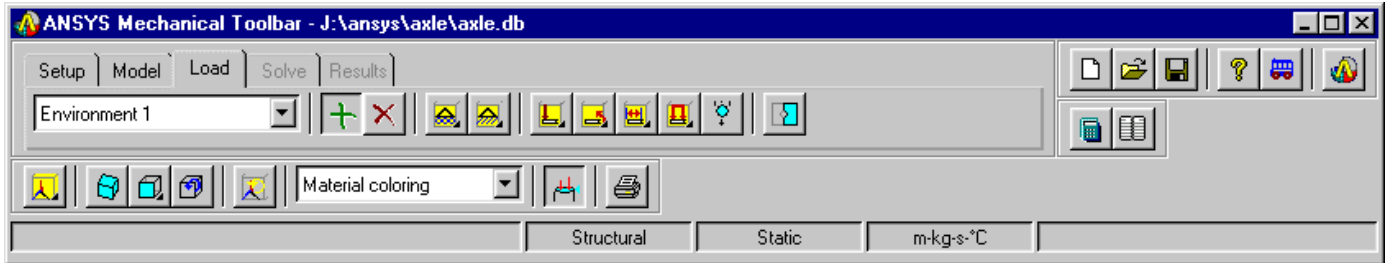
E. Click on **Render Model**. Try some of the other Render Model options.
Return to **Material Coloring** when you are finished



F. Click and hold on the Plot Button and pick **Plot Elements**.

4. Loads and Boundary Conditions

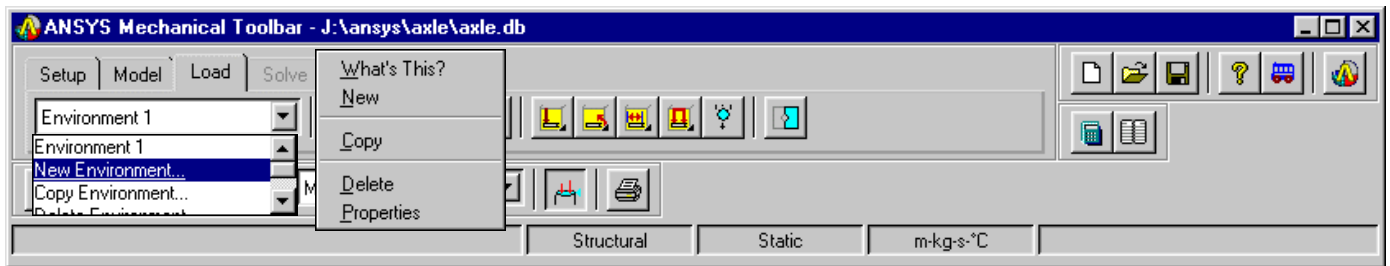
Now that we have completed the model definition phase, it's time to apply the loads and boundary conditions.



4.1. Environment

An environment is a set of boundary conditions (loadcases) applied to the model. You are allowed to define multiple environments. This is useful when you want to examine and compare the behavior of the model under different loads or boundary conditions.

The MTB allows you to add, copy, rename, or delete an environment.



New Environment

- To create a new environment and assign a name, perform the following:
 - Select the **New Environment...** option from the Load Environment drop down list box, or Right-click the mouse while it is positioned over the Load Environment drop down list and click on **New**. The *New Environment* dialog box appears.
 - Type the name of the new environment. Names are limited to 32



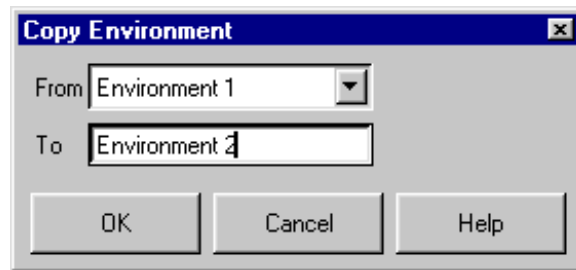
characters.

- Click **OK**. The new environment name now appears in the list of existing environment names.

Copy Environment

To copy an existing environment to a new environment, perform the following:

- Select the **Copy Environment...** option from the Load Environment drop down list box. The *Copy Environment* dialog appears.
- Select the environment to be copied in the From window, and type in the new environment name in the To window.

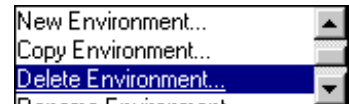


- Click **OK**

Delete Environment

To delete an environment perform the following:

- Select the **Delete Environment...** option from the Load Environment drop down list box. The *Delete Environment* dialog appears.
- Select the environment to be deleted and click **OK**.



Rename Environment

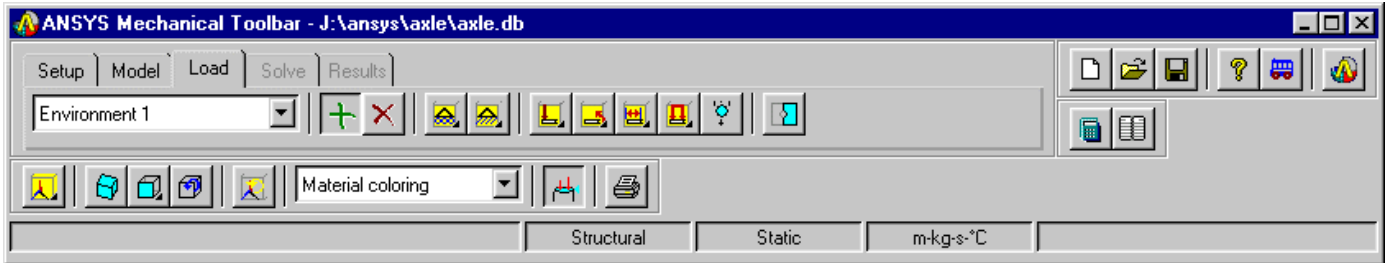
To Rename an environment perform the following:

- Select the **Rename Environment...** option from the Load Environment drop down list box. The *Rename Environment* dialog appears.
- Select the environment to be renamed.
- Type in a new name and click **OK**.

4.2. Adding or Deleting Loads and Boundary Conditions



In the Mechanical Toolbar, you specify whether you want to add or delete a load before selecting the specific type of load. Click the **Add B.C.** or the **Delete B.C.** button, and then click the button representing the load type you want to add or delete.



Make sure the proper Environment is set prior to creating a load or boundary condition.

After you click on a load type button, a picker appears that allows you to select the entity to which you are applying the load, or from which you are deleting the load.

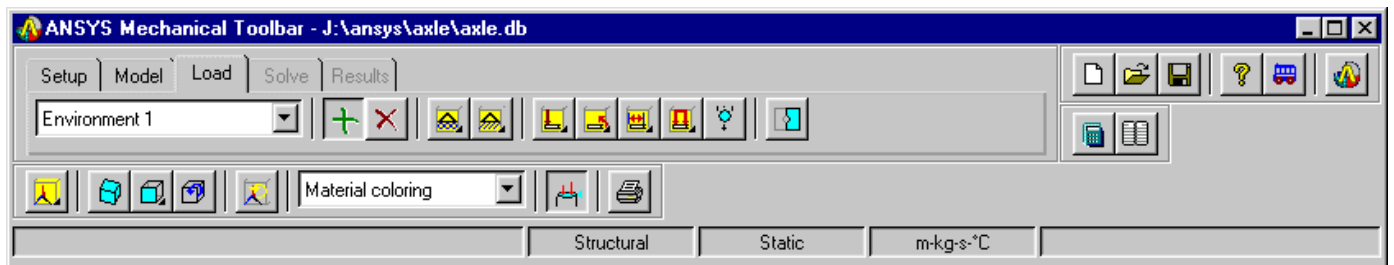
Boundary Conditions

To create boundary conditions

- Choose the environment for which this B.C. applies.
- Click the **Add B.C.** button.



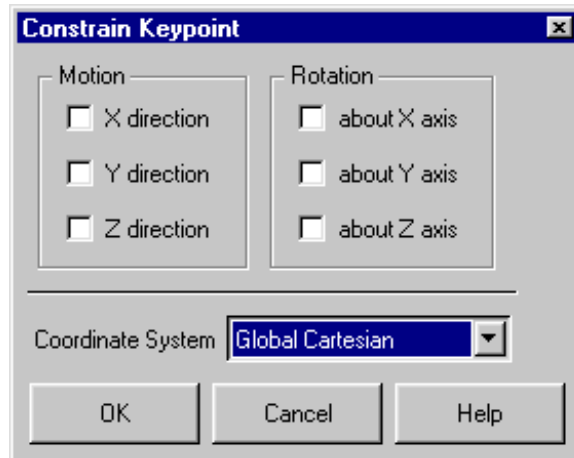
Constraint



- Select **Constraint** to remove one or more degrees of freedom from the selected object type.
- With the cursor positioned on the Constraint button, hold down the left mouse button. A fly-out toolbar appears. You can constrain a keypoint, a line, or an area.



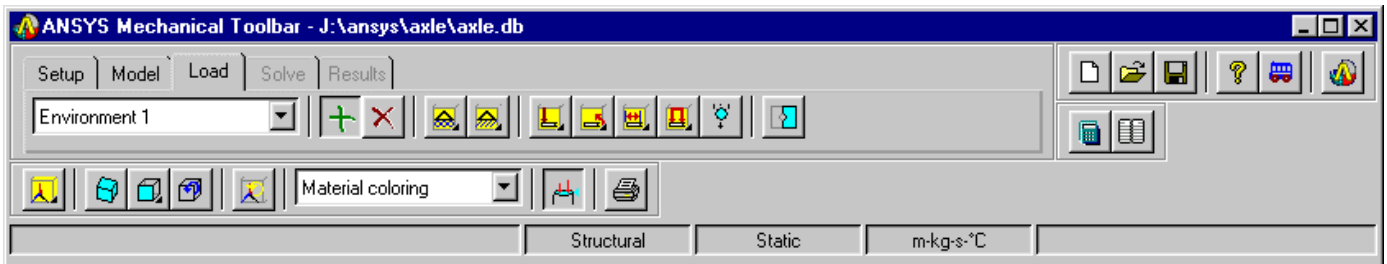
- Click on the button representing the entity constraint you want.
- A picker appears. Select the keypoints, lines, or areas to be constrained. Click **OK**. A *Constrain* dialog box appears. Select the direction(s) in which you want to constrain the model.



- Select the coordinate system you want to use to specify the direction if other than the default Global Cartesian system.
- Click **OK**.



Fixed



- Select **Fixed** to remove all degrees of freedom from the selected object type.
 - With the cursor positioned on the **Fixed** button, hold down the left mouse button. A fly-out toolbar appears. You can fix a keypoint, a line, or an area.



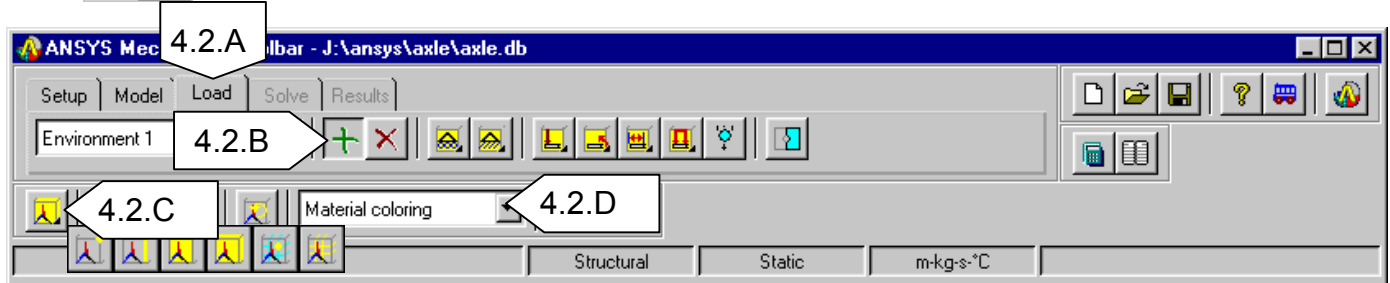
- Click on the button representing the entity you want to fix.
- A picker appears. Select the keypoints, lines, or areas to be fixed.
- Click **OK**.

Apply the boundary constraints to the model

A. Click on the **Load** tab in the MTB. Notice that the current environment is **Environment 1**.



B. Make sure the **Add** button is depressed.

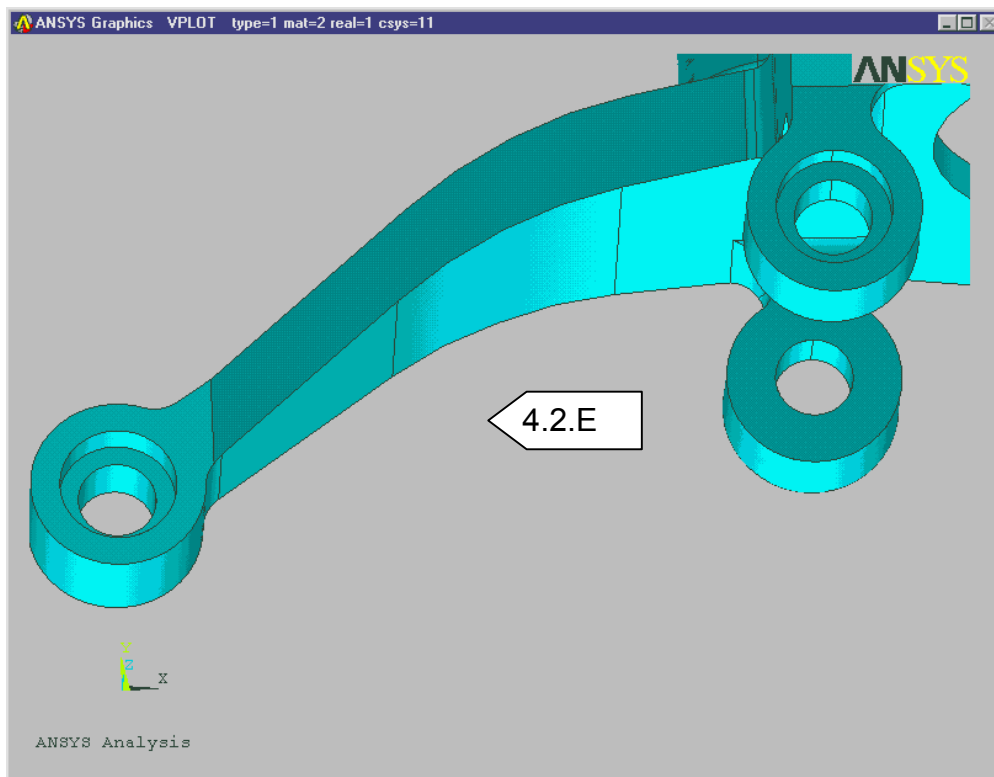


C. Change the graphics to a Volume plot to hide the elements. Click and hold the cursor on **Plot** to get the fly-out toolbar. Click on **Volume Plot**.

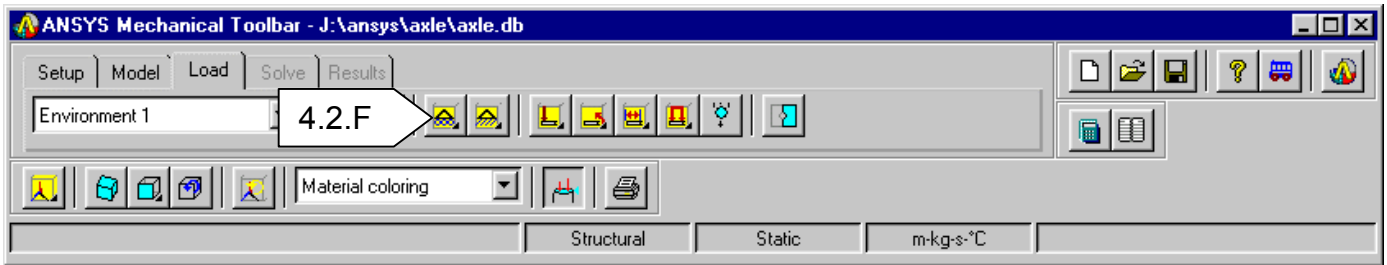
D. Change the Render Model option from **Material Coloring** to **Show Edges**. This will make it easier to view the areas



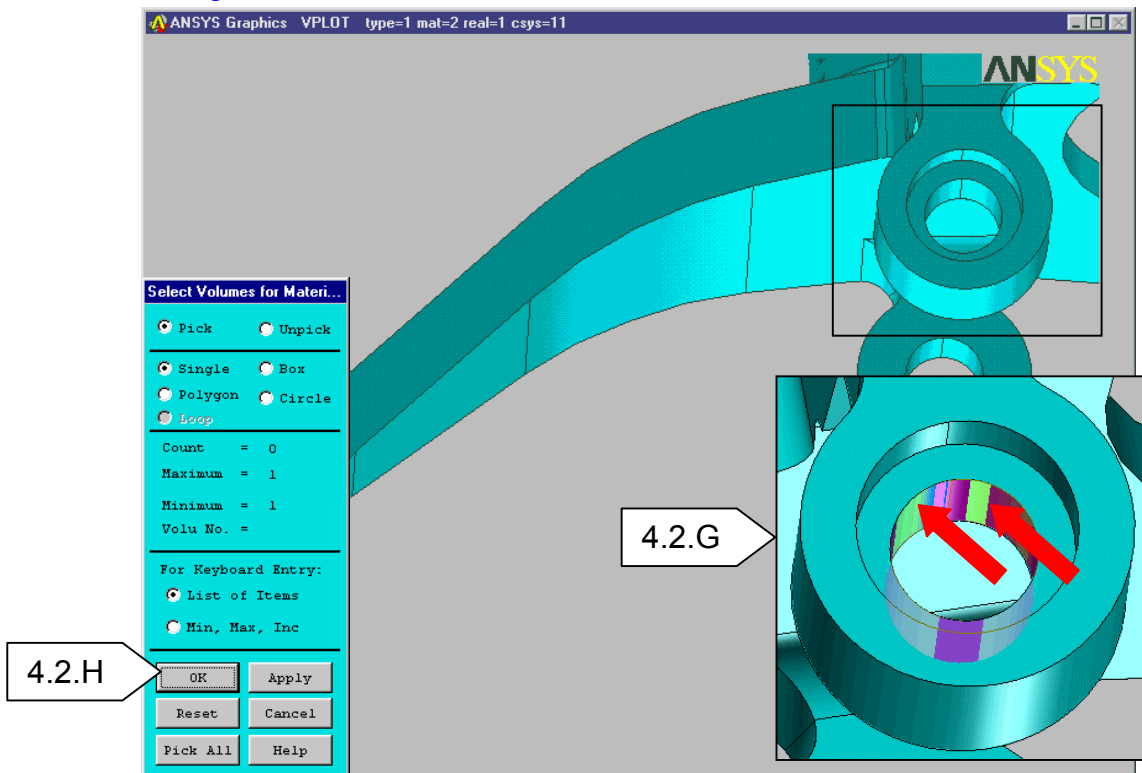
E. Click on the **Pan-Zoom-Rotate** button (if not already activated) and orient the plot approximately as shown



F. Click and hold the cursor on the **Constraint** button to get the fly-out toolbar. Click on **Constrain Area**. The Picker dialog will appear.

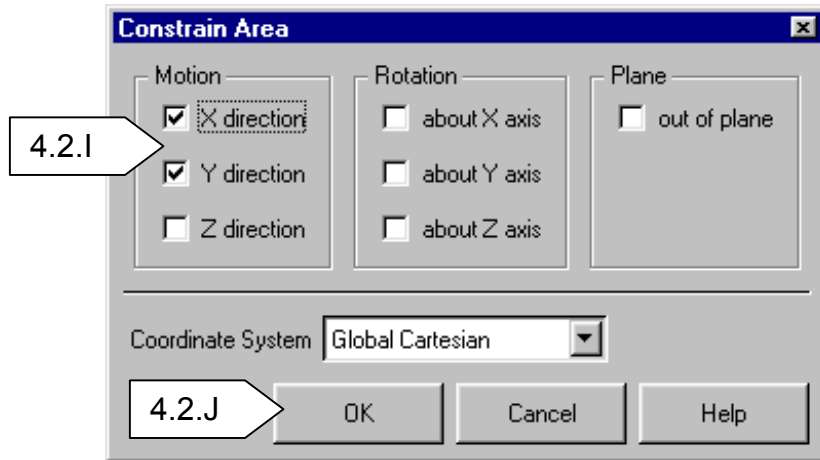


G. Pick the 2 inside face of the 3 holes as shown (total of 6 faces). You may want to zoom in on the individual holes to pick the faces and then **Fit** the view to move on to the next hole. **Tip:** Click and hold the left mouse button and move the cursor around the hole faces until the correct face highlights, then let go of the mouse button. (The selection does not occur until you release the mouse button). Do the same for all faces. If you accidentally select an incorrect area, click the right mouse button. The cursor will toggle from an up arrow (select) to a down arrow (deselect). You can then deselect the incorrect area. Click the right mouse button again to toggle back to select, and try again.



H. When all 6 faces have been selected, click **OK** on the Picker dialog. The **Constrain Area** dialog will appear.

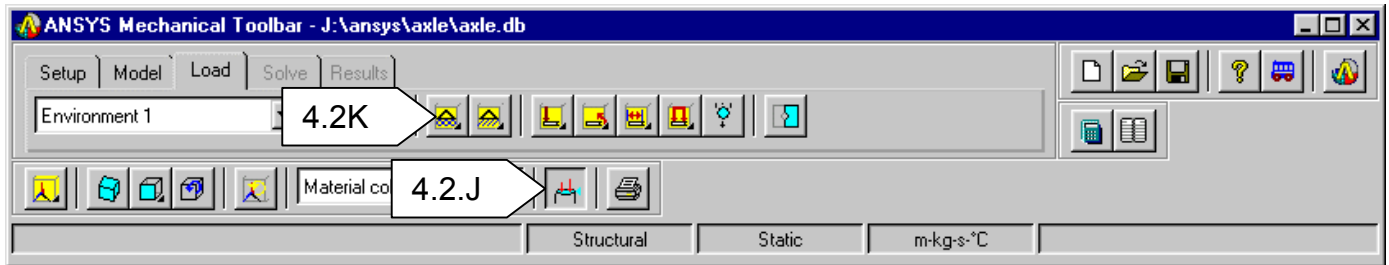
- I. We are going to constrain the inside faces of the holes from translating in the X and Y direction. Click on the check boxes next to **X direction** and **Y direction**.



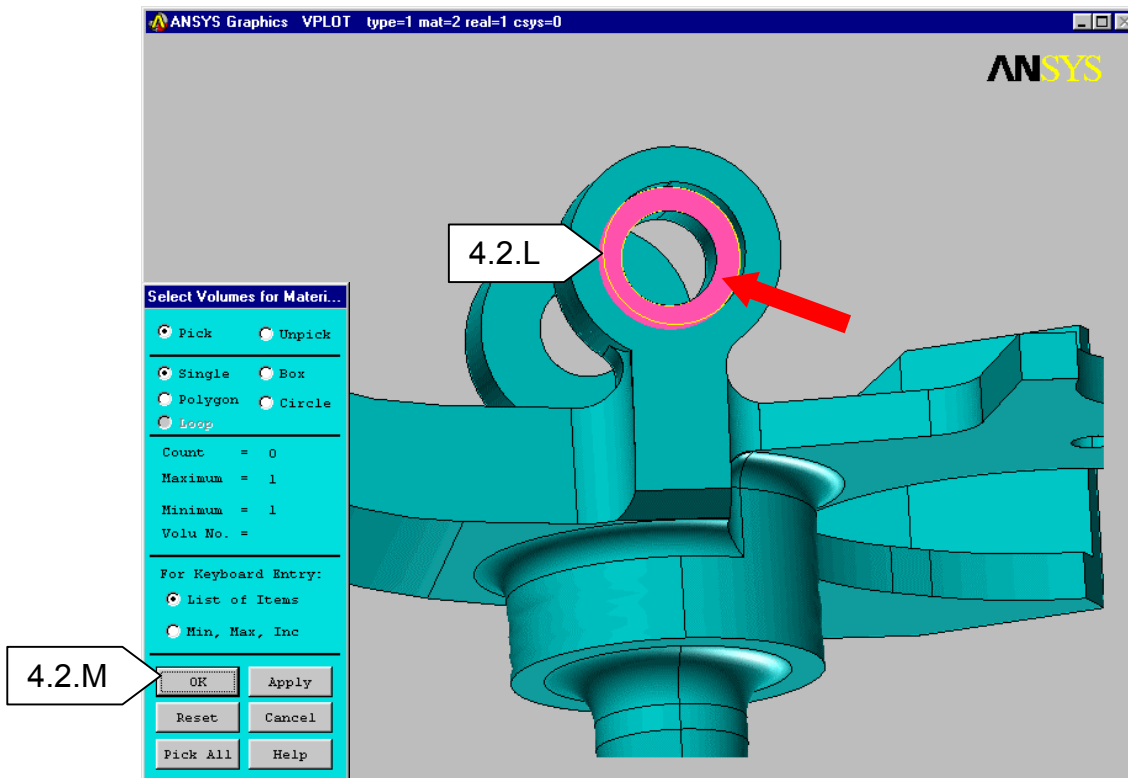
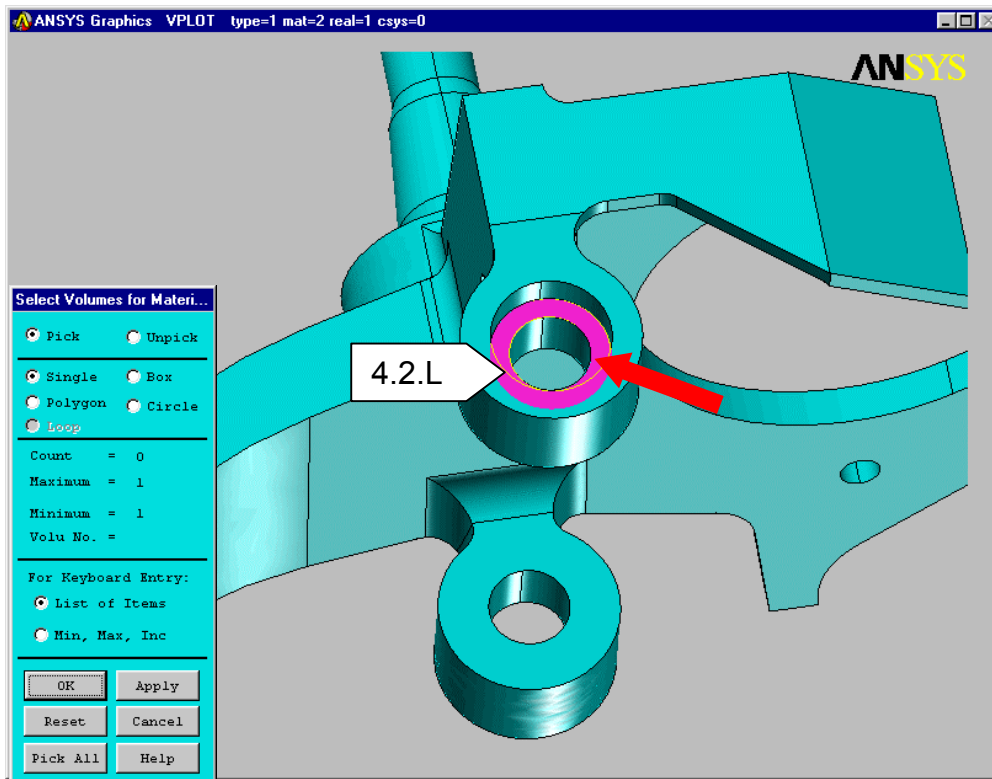
- J. Click **OK**. The symbols for the constraints will now appear on the inside hole faces of the model. If the symbols disappear, you may need to click on the **Boundary Conditions** button to display them.



- K. Click on the **Constrain Area** button again. The Picker dialog will appear.

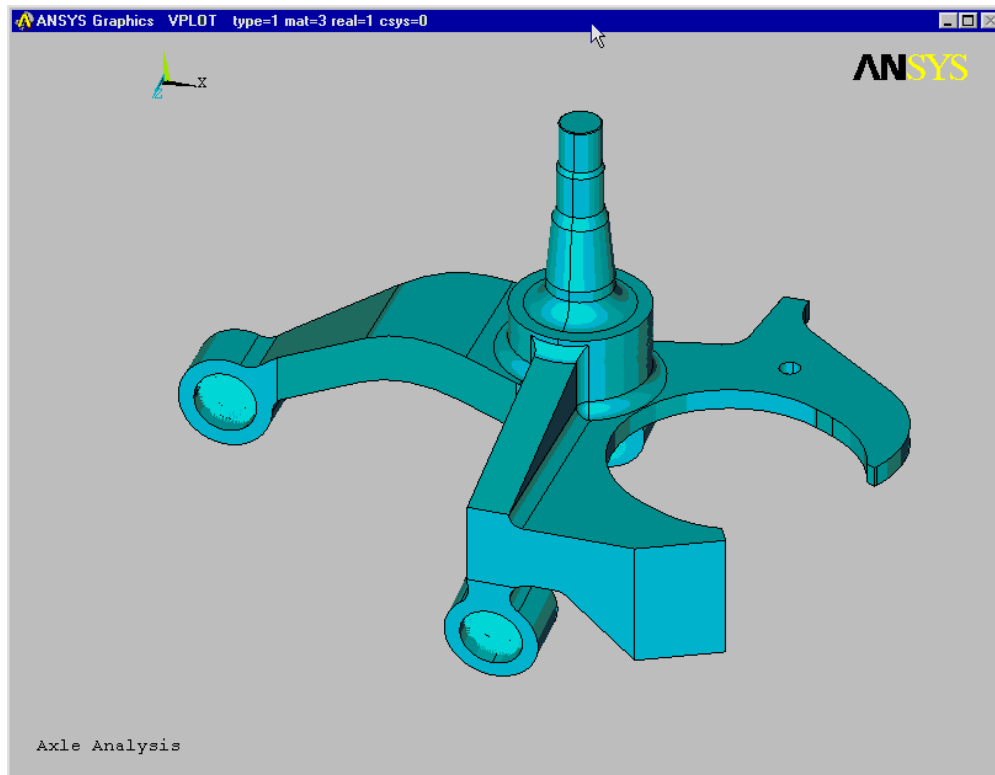
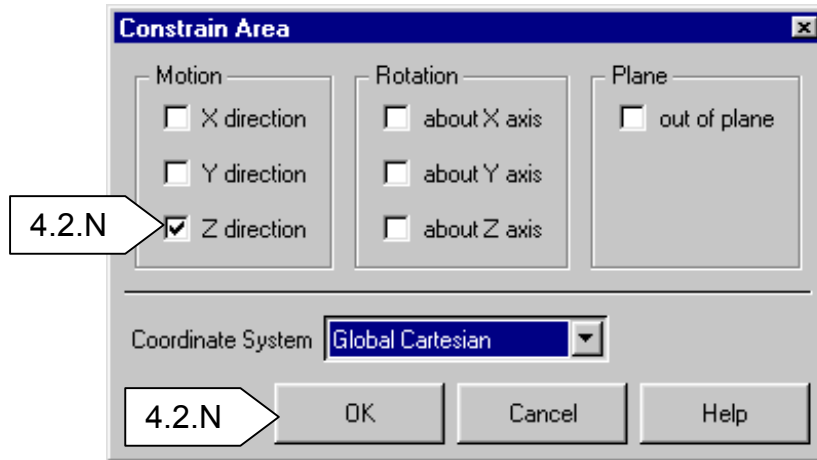


L. Pick the bottom face of the counterbored hole on both the upper and lower attachment locations as shown. (Just the 2 holes shown)



M. Click **OK** on the Picker dialog. The *Constrain Area* dialog will appear.

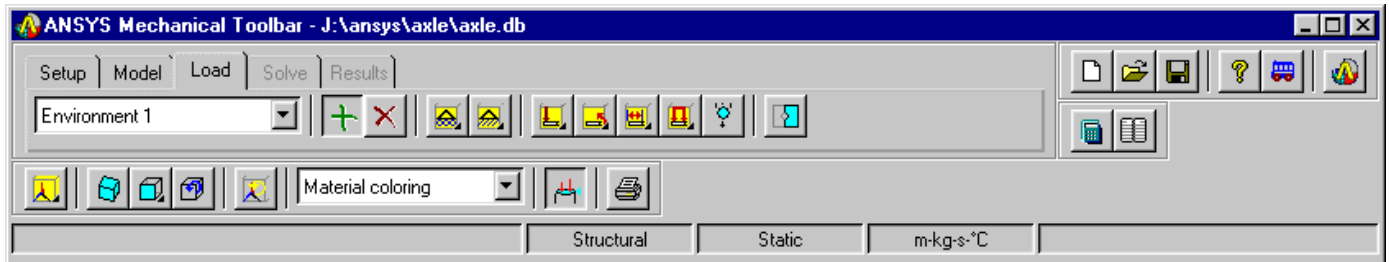
N. Constrain these areas in the Z direction only. Check on only the **Z direction** check box. Click **OK**. The constraint symbols will appear on these 2 faces.



4.3. Loads

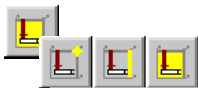
To create loads

- Choose the environment for which this Load applies.
- Click the **Add B.C.** button.

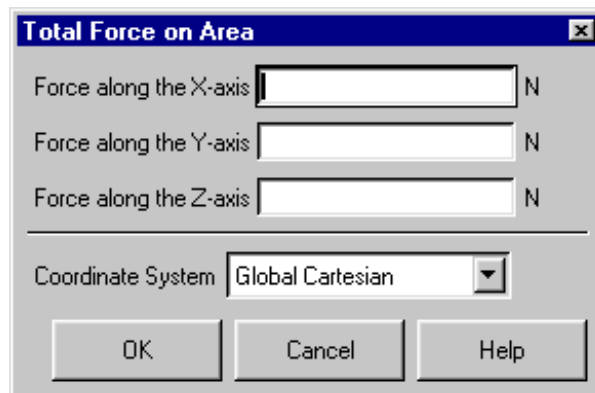


Force

- Select **Force** to apply a force to the selected object(s).
 - With the cursor positioned on the Force button, hold down the left mouse button. A fly-out toolbar appears. You can apply a force on a keypoint, a line, or an area.

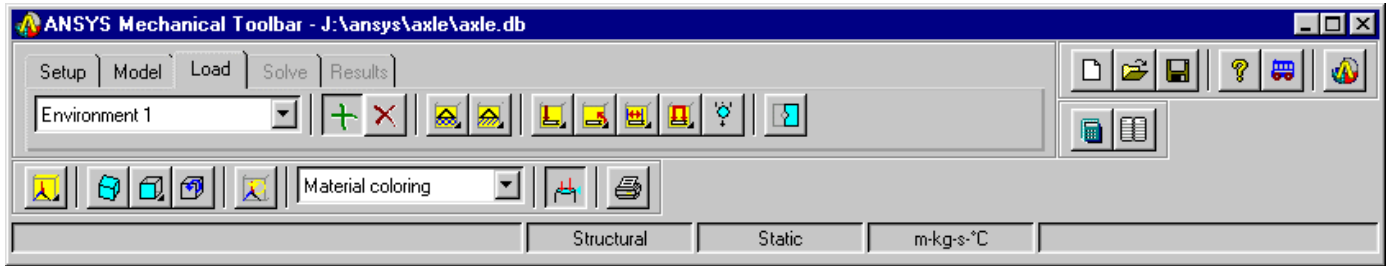


- Click on the button representing the type force you want.



- A picker appears. Select the keypoints, lines, or areas for applying the force. Click **OK**.
- A Total Force on dialog box appears. Specify the value of the force in the required direction(s). This is the total force applied to all selected objects.
- Select the coordinate system you want to apply if other than the Global Cartesian system. If you choose New Coordinate System, special conditions apply.
- Click **OK**.

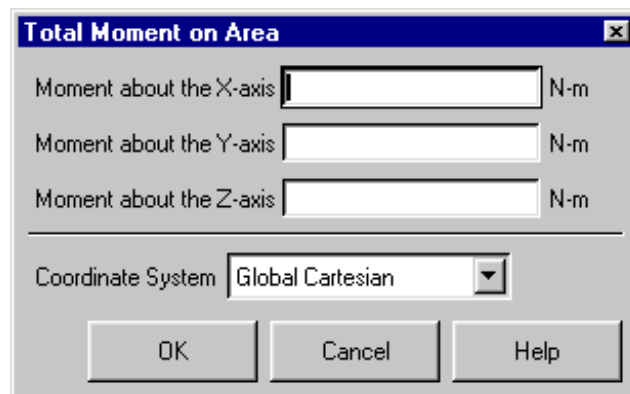
Moment



- Select **Moment** to apply a moment to the selected Object(s)
- With the cursor positioned on the Moment button, hold down the left mouse button. A fly-out toolbar appears. You can apply a moment to a keypoint, a line, or an area.

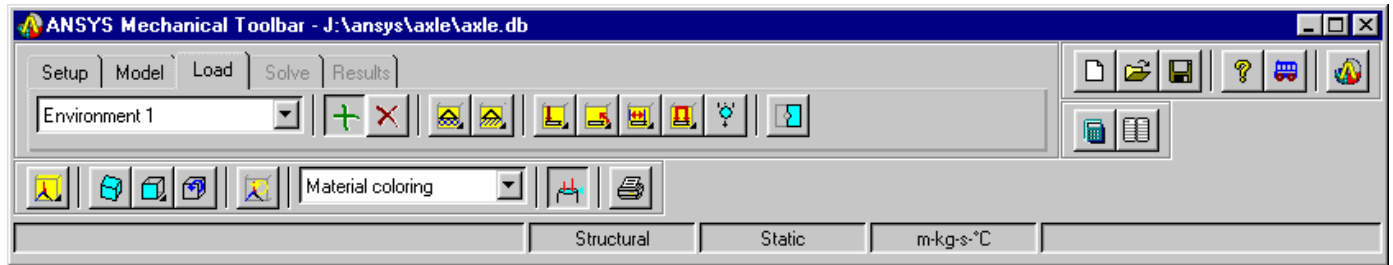


- Click on the button representing type of entity you want to apply moment to.
- A picker appears. Select the keypoints, lines, or areas to apply the moment to. Click **OK**.
- A *Total Moment on* dialog box appears. Specify the value of the moment in the required direction(s).



- Select the coordinate system you want to apply if other than the Global Cartesian system.
- Click **OK**.

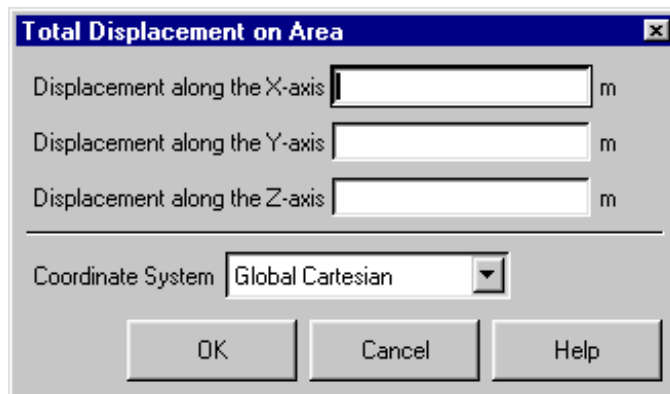
Displacement



- Select **Displacement** to set an initial displacement on the selected object.
 - With the cursor positioned on the Displacement button, hold down the left mouse button. A fly-out toolbar appears. You can displace a keypoint, a line, or an area.



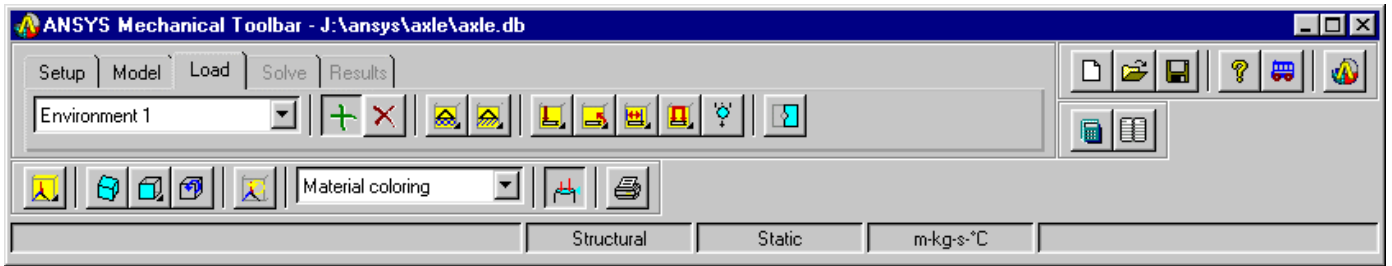
- Click on the button representing type of entity you want to apply displacement to.
- A picker appears. Select the keypoints, lines, or areas to be displaced. Click **OK**. A *Total Displacement on* dialog box appears.



Specify the displacement along the appropriate axes.

- Select the coordinate system you want to apply if other than the Global Cartesian system.
- Click **OK**.

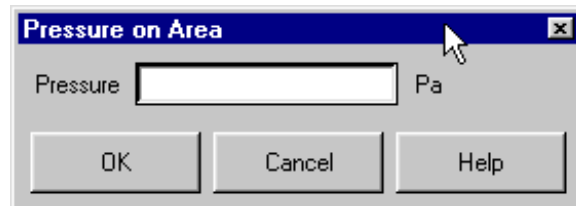
Pressure



- Select **Pressure** to apply a pressure to either a line or area
 - With the cursor positioned on the **Pressure** button, hold down the left mouse button. A fly-out toolbar appears. You can apply pressure to a line or an area.

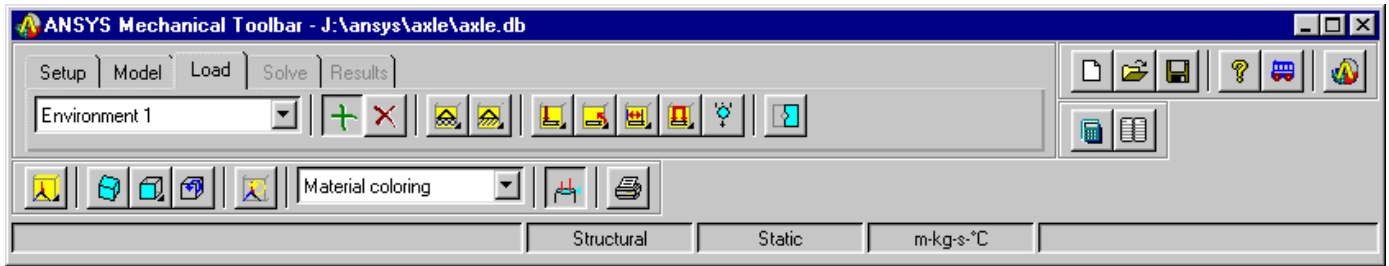


- Click on the button representing type of entity you want to apply pressure to.
- A picker appears. Select the lines or areas you are applying a pressure to and click **OK**.
- A Pressure on dialog box appears. Enter the pressure value you want to apply.

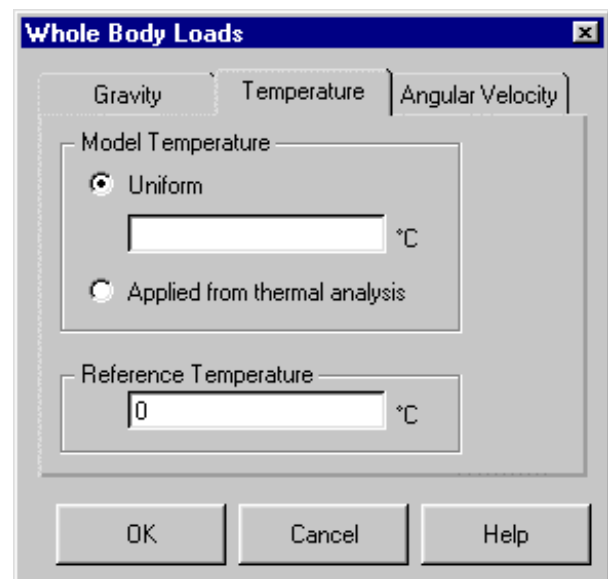
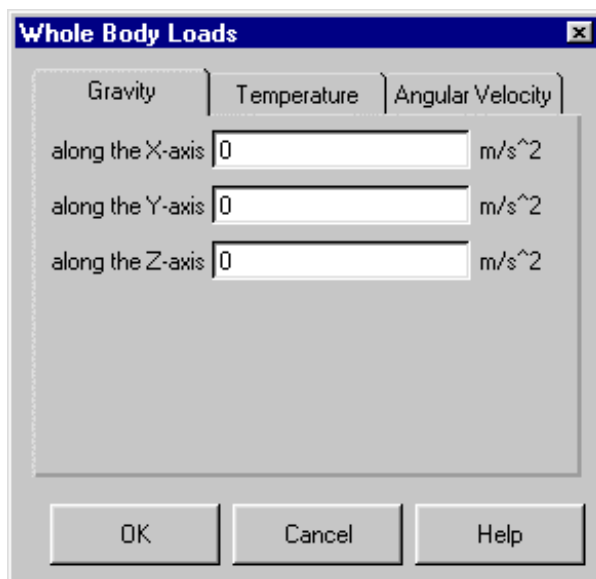


- Click **OK**.

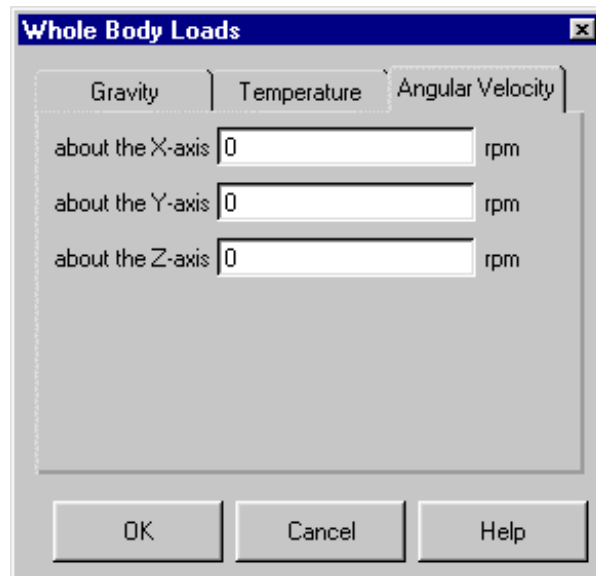
Body Load



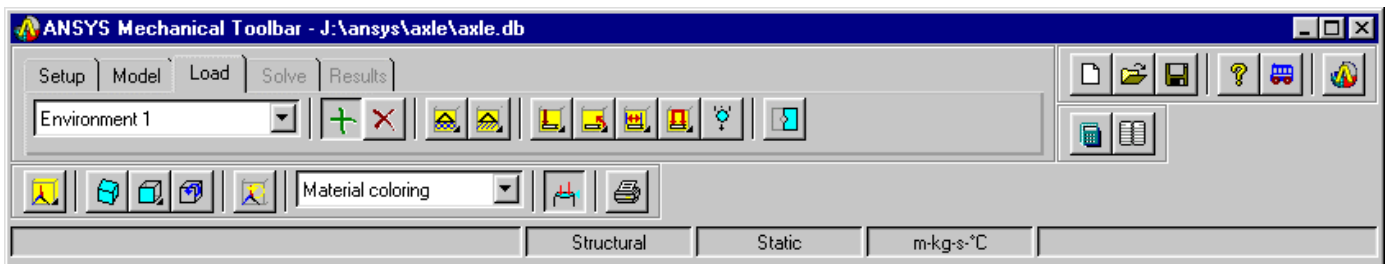
- Select **Body Load** to apply a volumetric or field load. In the MTB, you can apply gravity, temperature, and angular velocity as body loads.
- Click on the Body Load button.
- A Whole Body Loads dialog box appears.
- To apply gravity body loads, click the **Gravity** tab, enter the gravitational acceleration load values for the X, Y, and/or Z directions, then click **OK**.
- To apply temperature body loads, click the **Temperature** tab and make choices depending on whether the temperature load is uniform or is a result of a thermal analysis.
- For a uniform temperature load, click the **Uniform** radio button and enter the load temperature and the reference temperature, then click **OK**.



- To apply angular velocity body loads, click the Angular Velocity tab, enter the rotational speeds about the X, Y, and/or Z axes in revolutions per minute (RPM), then click **OK**.





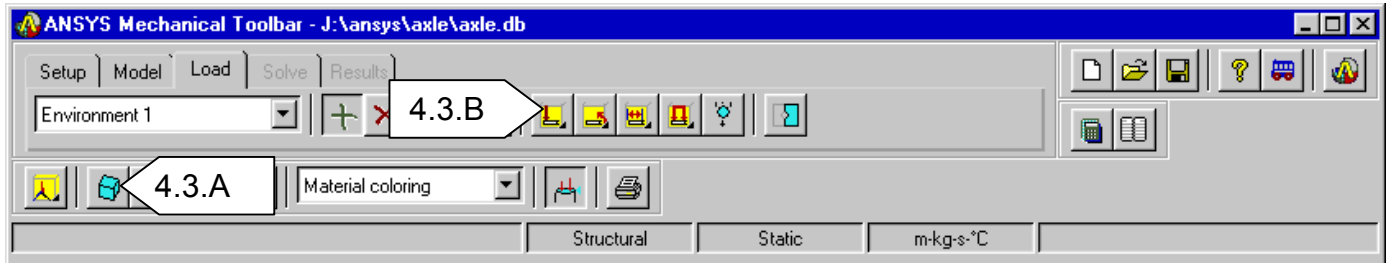
Model Symmetry



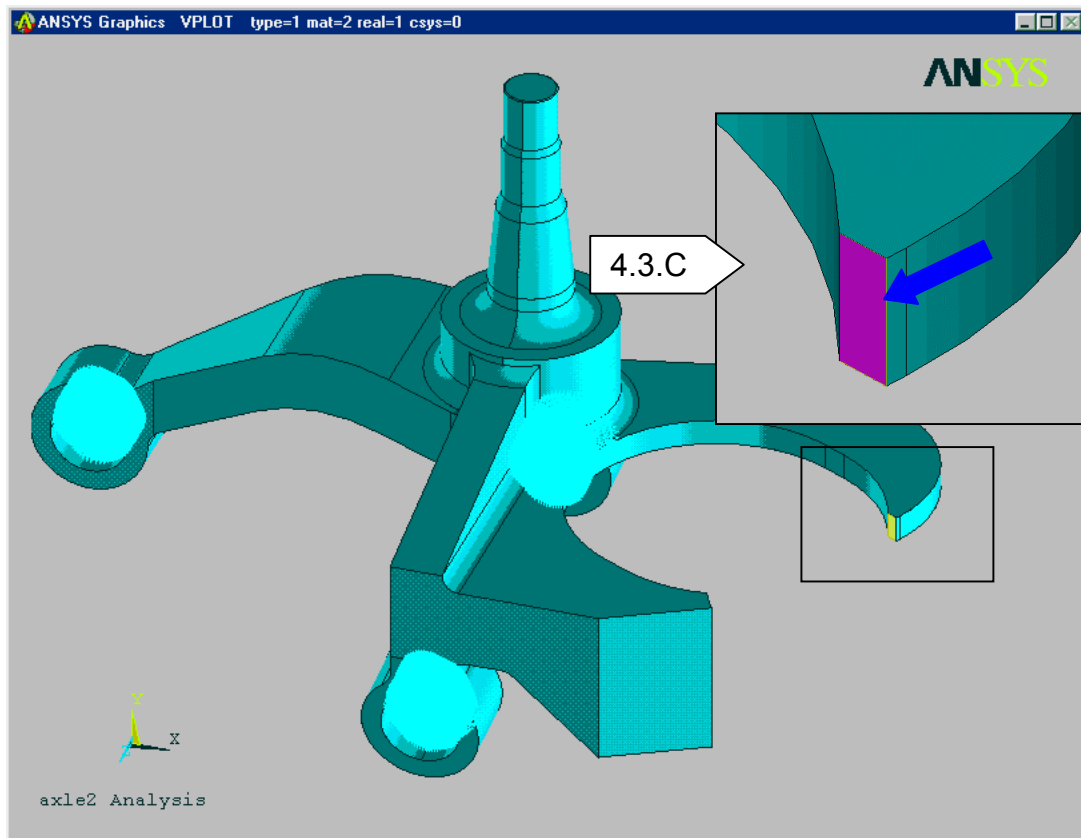
- Select **Model Symmetry**, if you are a constructing a symmetric model, to define symmetry boundary conditions. These can be represented as an area on a 3-D model or a line on a 2-D (or 3-D shell) model.
- Click the **Model Symmetry** button. A picker appears.
- Select the line (for a 2-D model or 3-D shell model) or area (for a 3-D model) on which to define the symmetry boundary conditions.
- Click **OK**.

Apply the loads to the model

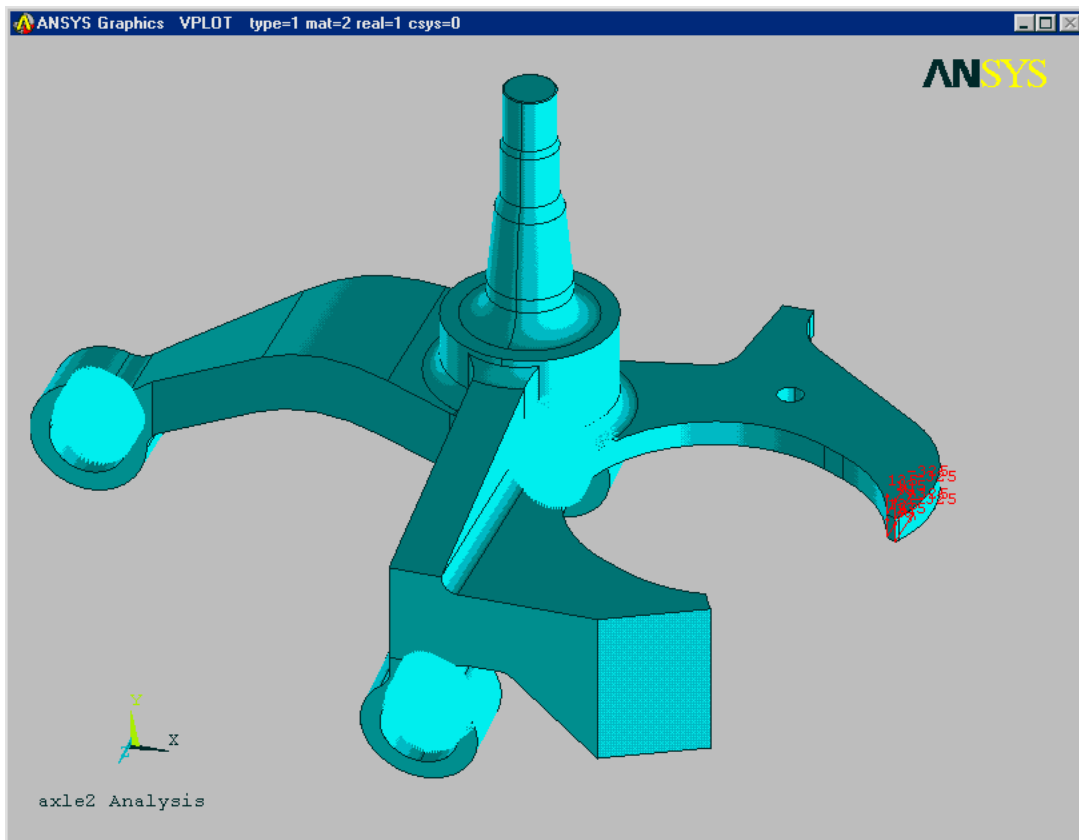
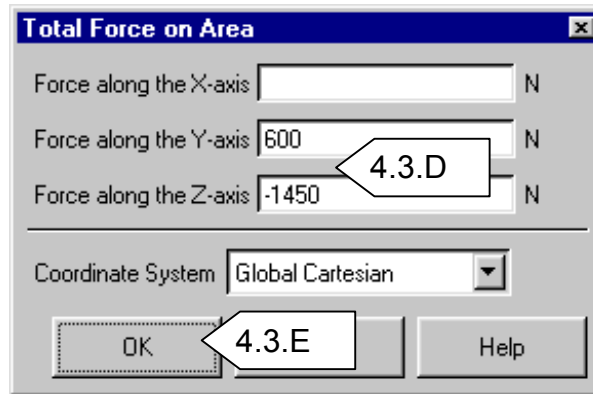
-  A. Change the view to better facilitate the loads application. Click on the **Oblique View** button.
-  B. Click and hold the left mouse button on **Force** to get the fly out toolbar. Click on **Area force**. The Picker dialog will appear



- C. Select the small area as shown (you may want to zoom up on the area) and click **OK**. The **Total Force on Area** dialog will appear.

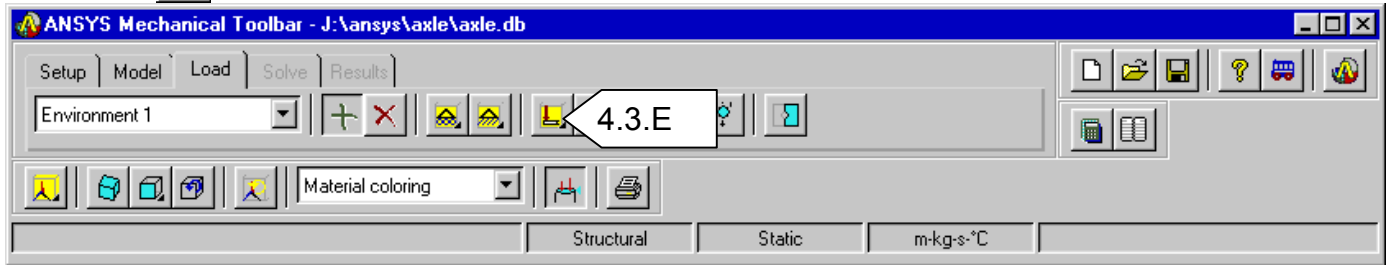


D. Input **600** newtons for the **force along the Y-axis** and **-1450** newtons for the **Force along the Z-axis**. Click on **OK**. The symbols for the force will appear on the small area.

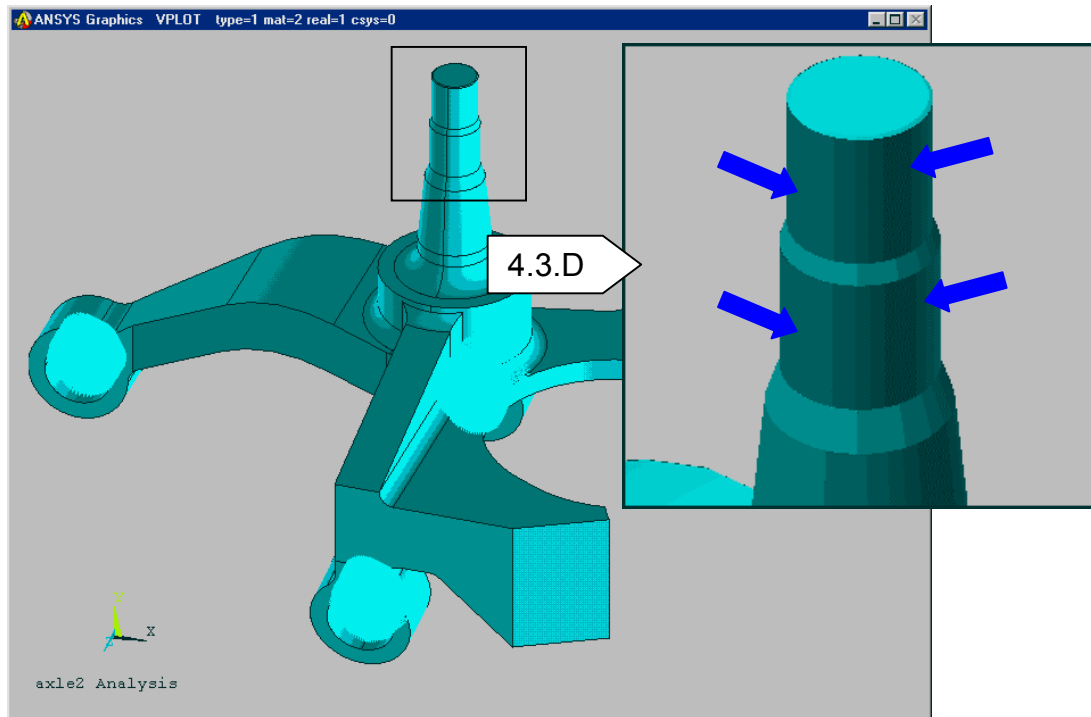




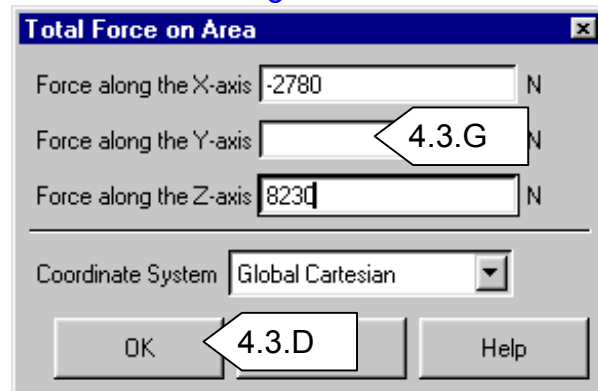
E. Pick the **Area Force** button again. The Picker will appear



F. Pick the 4 areas on the axle as shown and select **OK**



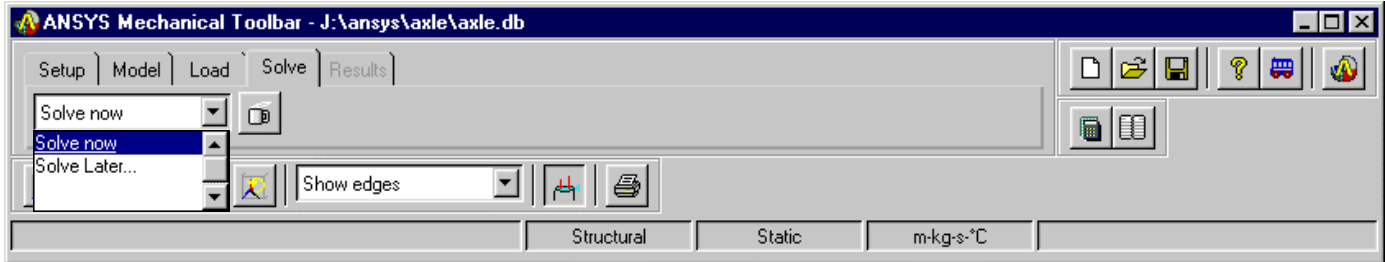
G. Input **-2780** newtons for the Force along the X-axis and **8230** newtons for the Force along the Z-axis.



H. Click on **OK**. The symbols for the forces will appear on the axle.

5. Model Solution

Use the **Solve** tab to solve the analysis. You can choose to solve now or solve later.

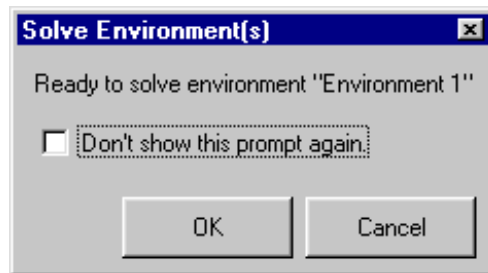


5.1. Solve Now

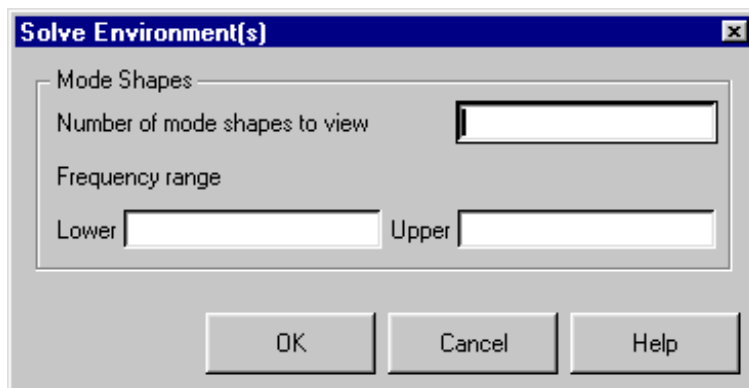


To solve now, select **Solve Now** from the *Solve Time* drop down list box and click the Solve Problem button.

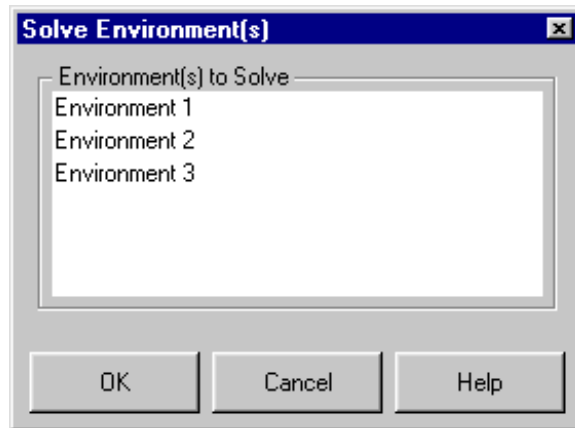
- If you defined a single environment for a structural static or thermal steady-state analysis, a Solve Environment(s) dialog box appears stating that the environment is ready to be solved. Click the **OK** button to initiate the solution.



- If you defined a single environment for a modal analysis, a Solve Environment(s) dialog box appears. Enter the number of mode shapes you want to view and the frequency range (if desired), then click OK to initiate the solution.



- If you defined multiple environments for a structural static or thermal steady-state analysis, a *Solve Environment(s)* dialog box appears. Choose the environment(s) you want to solve and click **OK** to initiate the solution(s).



- When the solution is finished, the Mechanical Toolbar brings up the Results tab and automatically displays an appropriate plot (based on the discipline) and a text window that lists the environment(s), and summary information based on the discipline.

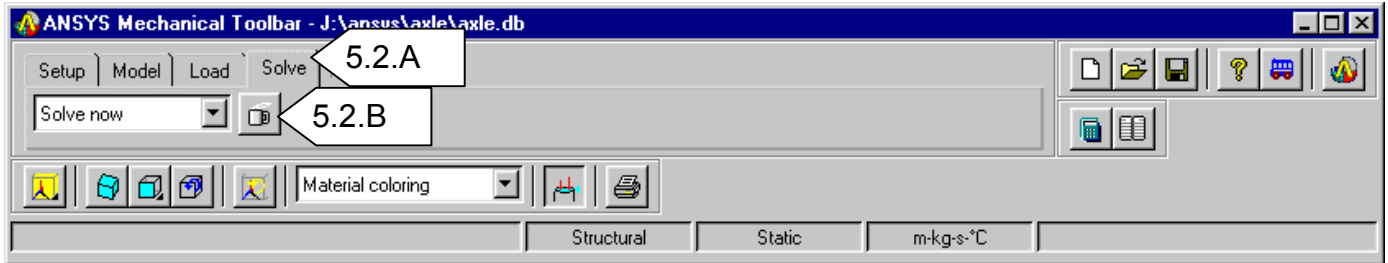
5.2. Solve Later

To solve later, follow these steps:

- Select **Solve Later...** from the *Solve Time* drop down list box. A Solve Time dialog box appears.
- Enter the date and time you want the solution to begin and click **OK**.
- Click the **Solve Problem** button. A *Solve Environment(s)* dialog box appears whose content varies depending on the number of environments you want to solve and whether the problem involves a modal analysis .
- Enter any required information in the Solve Environment(s) dialog box and click **OK**. A Solve Later Information dialog box appears which displays the solution start time that you entered, the working directory name and location where files will be stored, and a statement specifying that system specific processes must be running before using the solve later option.
- If these systems are running, click **OK** to accept the name and location of the working directory, or specify another name and location for the working directory by clicking **Browse...**, choosing the name and directory location in the *Solve Later Browse Working Directory* dialog box, then clicking **OK**.

Solve the exercise

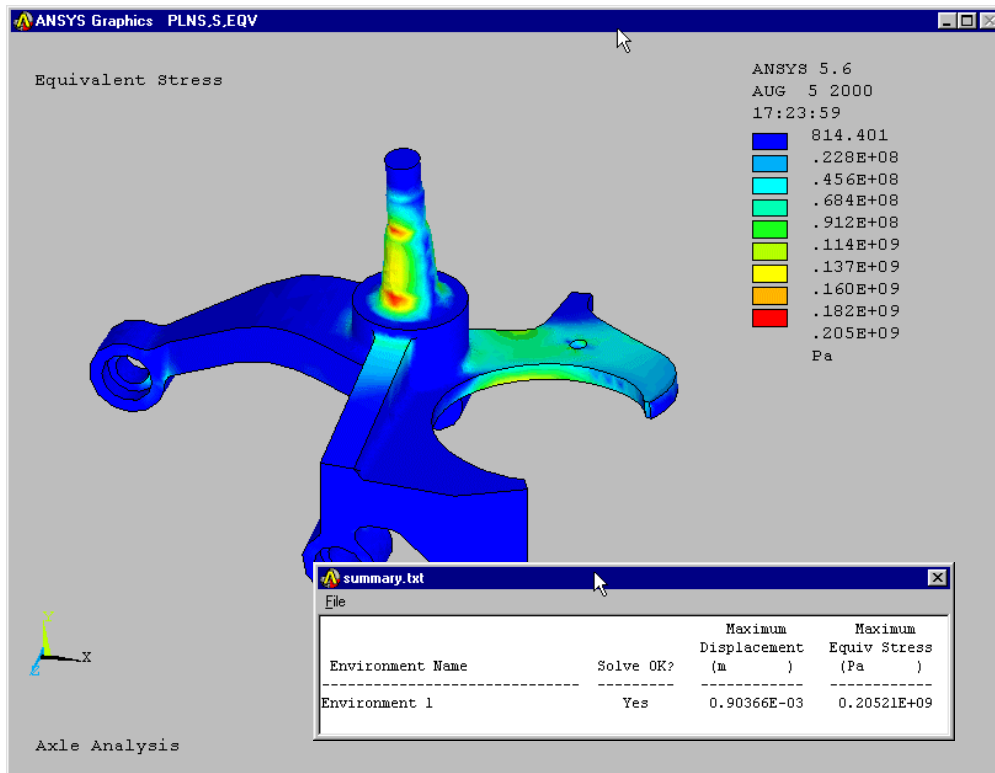
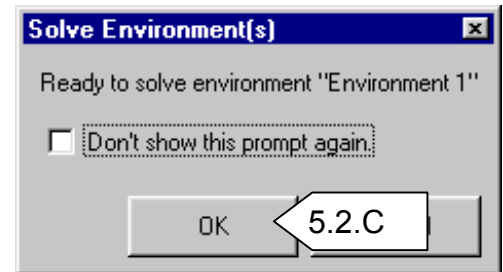
- A. Click on the **Solve** tab on the MTB.



- B. Click on the **Solve Problem** button. The *Solve Environment(s)* dialog will appear since there is only one environment.

- C. Click **OK** to proceed with the solution. This may take a few minutes

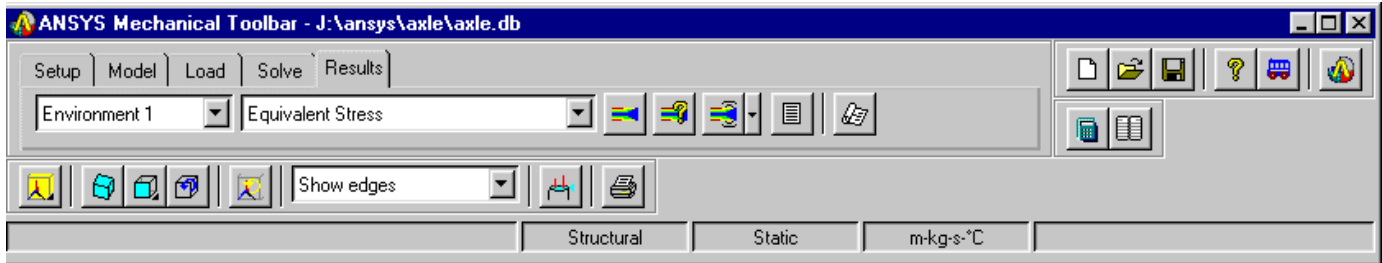
- D. When finished, a text window will appear showing that the solution successfully completed and will list the maximum displacement and stress. The graphics display will show the Von Mises Equivalent Stress plot. Note that the maximum stress value is well within the yield limit stress of steel.



6. Results

After completing a successful solution, it's now time to post process the model and take a look at the results. By default the MTB will select the Results tab after a successful solution.

To look at the results, click on the **Results** tab



If you solved multiple load environments, you can view the results for each of them by:

- Clicking on the Load Environment drop down list box.
- Clicking on the environment for which you want to see results.

6.1. Results Item

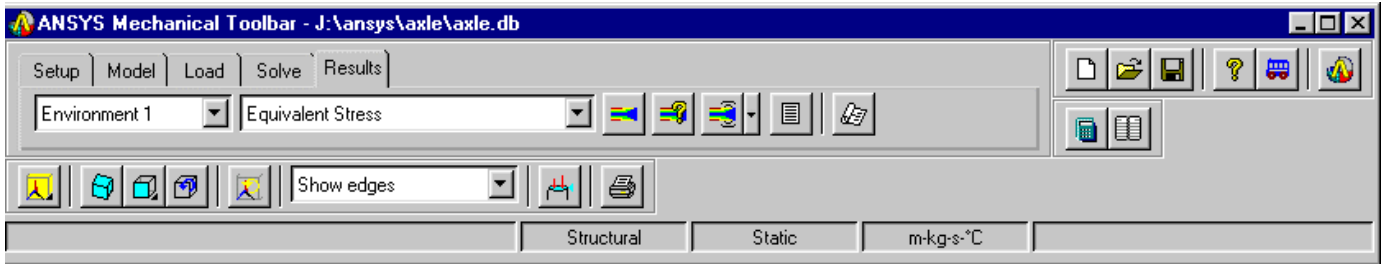
- You can view any of the following types of result information for a structural static analysis by clicking on the **Results Item** drop down list:
 - **Equivalent stress** - (Von Mises stress) a representation of any arbitrary stress state as a single positive stress value.
 - **Displaced shape** - physical displacement of the model.
 - **Stress intensity** - the difference between the maximum (1st) and minimum (3rd) principal stresses.
 - **1st and 3rd principal stresses** - The maximum and minimum principal stresses.
 - **Stress in the global X, Y, or Z direction** - individual stress components for each direction in the global Cartesian system.

-  After selecting the *Results Item* click on **Plot Results**

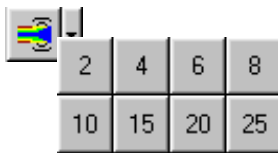


6.2. Results Display

You can plot, query, animate, or list the result item shown in the Result Item drop down list box.

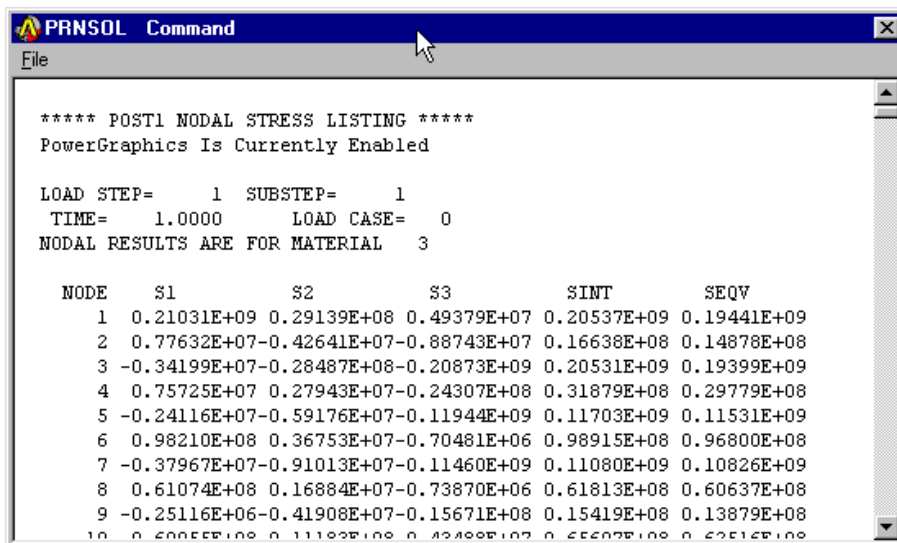


- Click on the **Plot Result** button to plot the selected *Results Item*
- To query the results, click the **Query Result** button. A picker appears. Hold the left mouse button down and drag it over the area of interest. Result values appear both on the plot as well as in the picker dialog.
- To animate the results, click the **Animate Result** button. Click the black arrow to the right of the icon to specify the number of frames to animate (default = 10 frames).



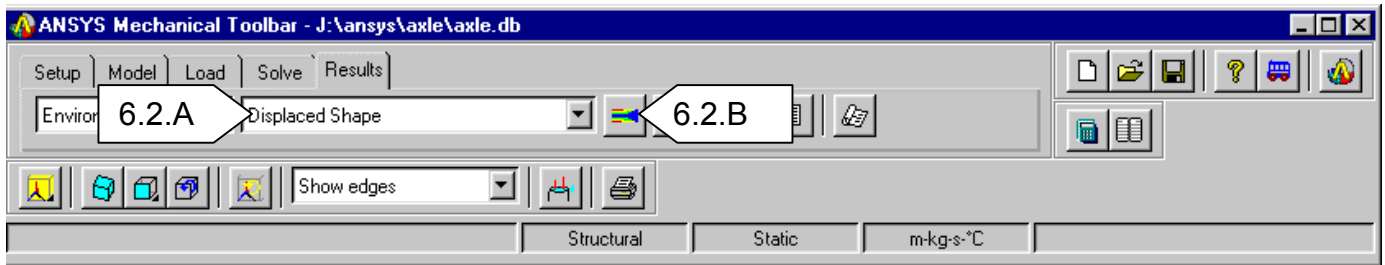
- An Animation Controller appears that allows you to start or stop the animation, play continuously or forward only, or apply delay.


- Click on the **List Result** button to see a tabular listing of the results.

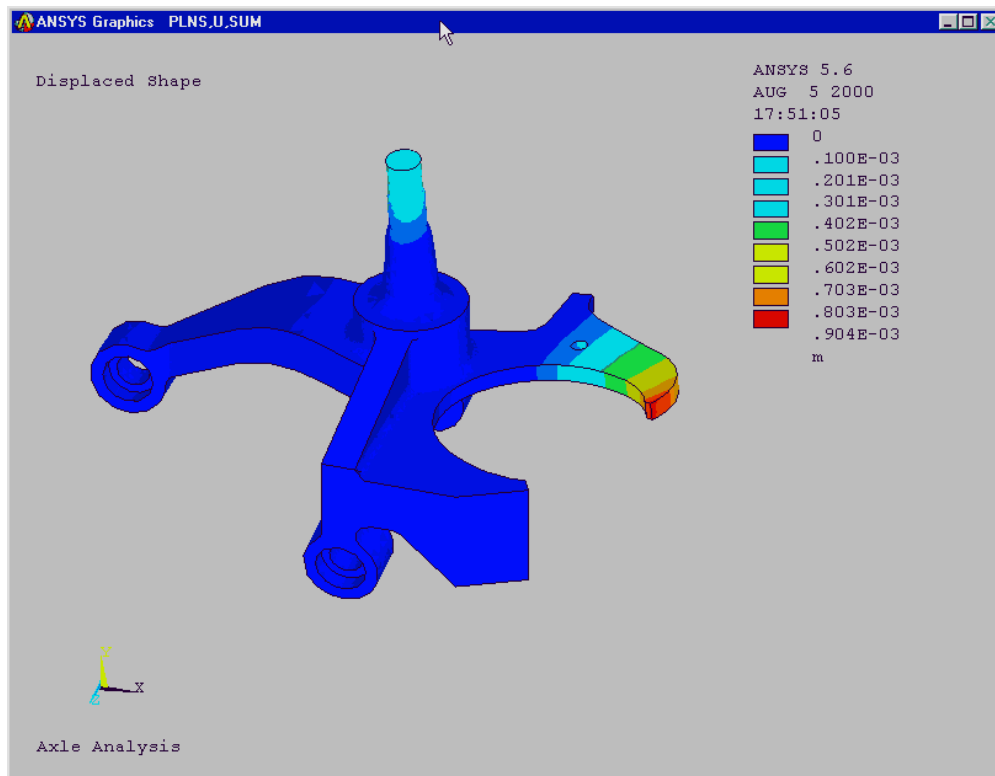


Let's view the Result Items for this exercise

A. Change the *Results Item* to **Displaced Shape**



-  B. Click on the **Plot Results** button. The Displacement Shape plot will appear. The displacement are small in comparison to the thickness of the model indicating good compliance with small displacement theory

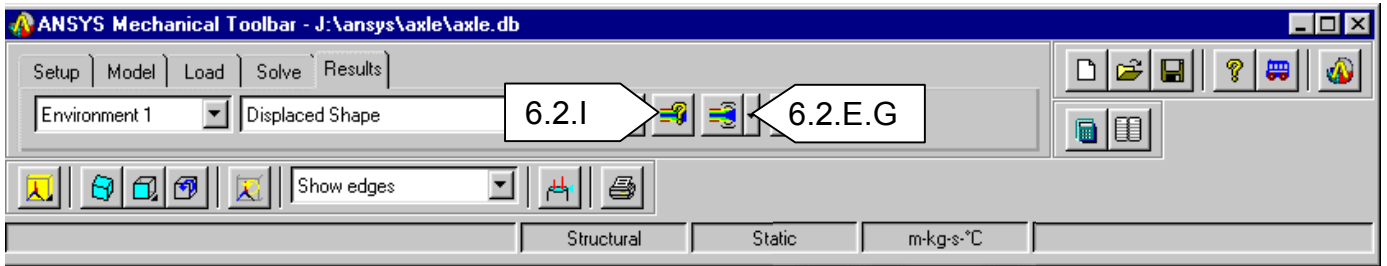


C. Plot the other result items for kicks.

D. Return the plot to **Equivalent Stress** when you are done having fun.



E. Animate the results. Click on the arrow next to the **Animate** button



F. Change the number of frames to 8

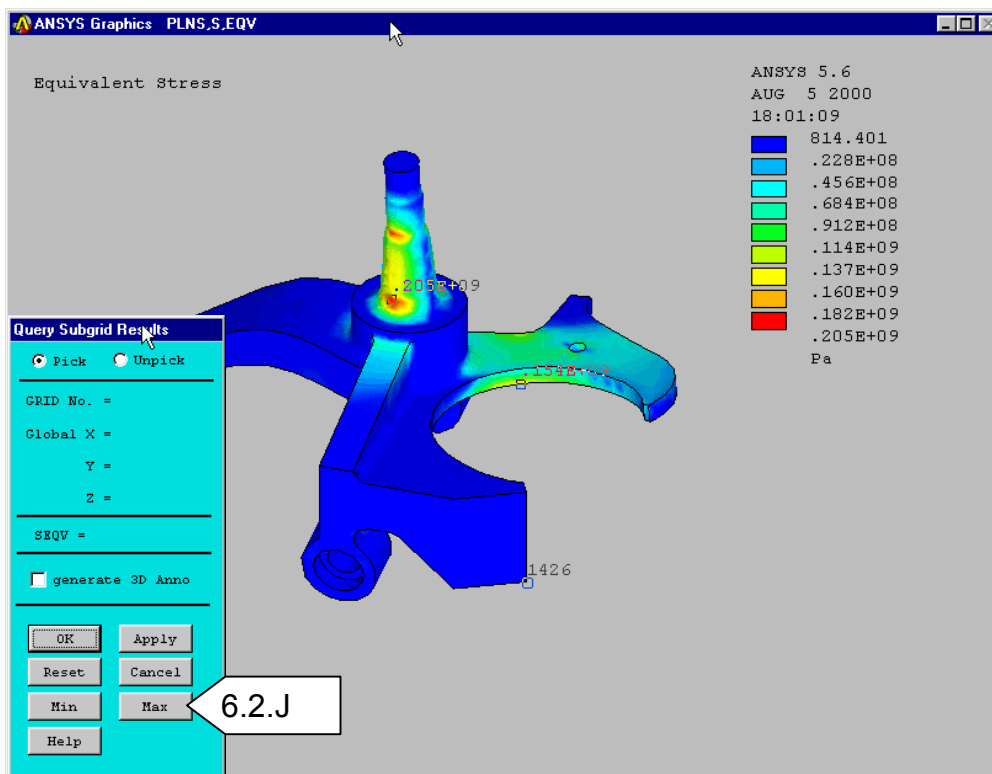
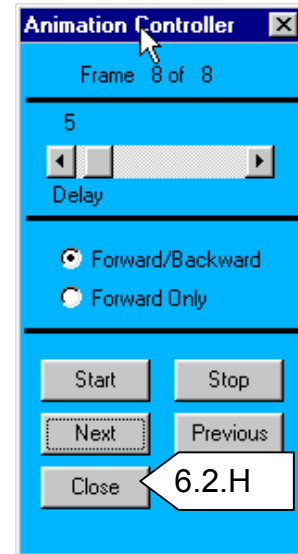
G. Click on the **Animate** button to start the animation

H. Click on **Close** when you are satisfied.

I. Query the results. Click on the **Query Results** button. The Picker will appear.

J. Click on the **Min** and **Max** buttons on the picker to display a label indicating max and min stress.

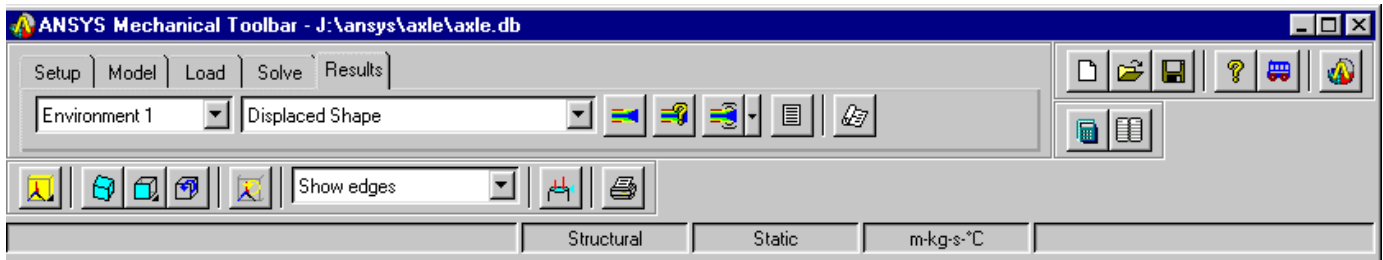
K. With your cursor click on some other areas to display the stress. If you click and hold the left mouse button down and move it over the model the system will temporarily display the stress values.





6.3. Reports

You can generate a report using a predefined format (template) or you can generate a report using a template that you created. You can also view an existing report. All reports are generated in standard HTML. You can forward them electronically, post them on a web site, or print them.



General Report Format

The General Report format includes the following items:

- **Title page** - name of analysis (taken from the Graphic Title entry in the Setup tab), name of analyst/designer (taken from the User Info tab [Toolbar Properties button within the main Setup tab]), date of report, links to Summary, Model Information, Analysis Information, and Results Information sections.
- **Summary** - ANSYS plot of the original model, text summary including information on analysis type, environments, and results data.
- **Model Information** - source of model file, ANSYS mesh plot, tabular details of the finite element model and material properties.
- **Analysis Information** - ANSYS plot and tabular listings of loads and boundary conditions.
- **Results Information** - ANSYS plots and tabular listings of results data.

To generate an HTML report using the predefined format, do the following:

- Click on the **Show Report** button. The *Report Options* dialog appears.
- Click on the **Create a new report** radio button.
- Click on the **General Report** option.
- Click **OK**. The report is generated in the default HTML browser, and a directory named **reportn** is created (where n increments with each report generated) that includes the report's HTML file and all graphics files associated with the report.

Viewing an Existing Report

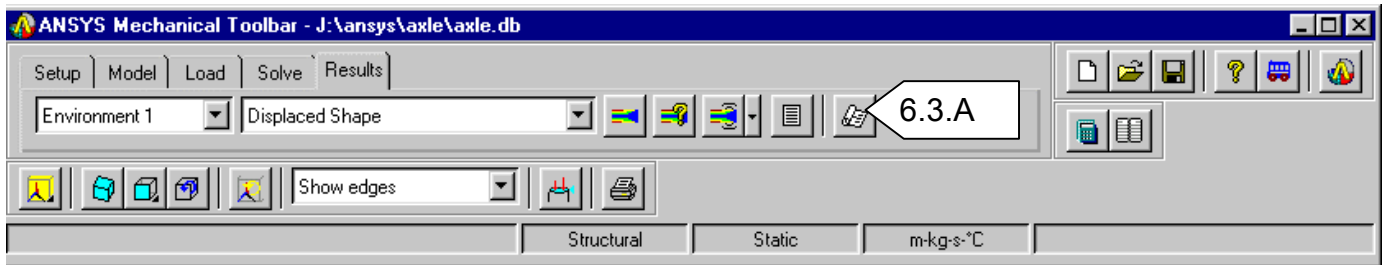
To view an existing report, do the following:



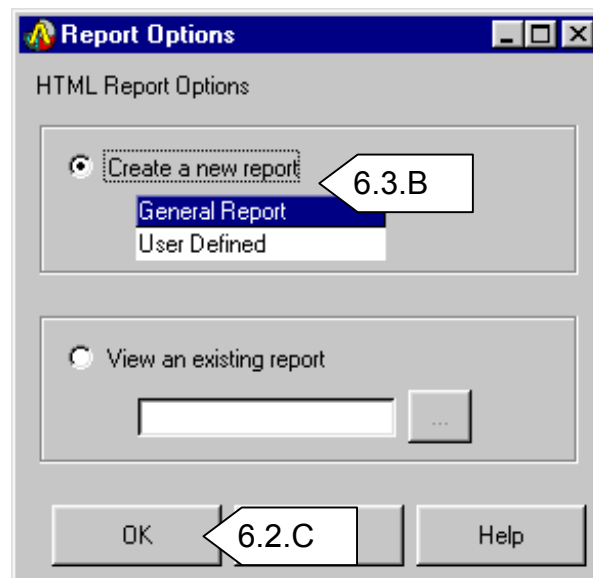
- Click the **Show Report** button. The Report Options dialog box appears.
- Click the **View an existing report** radio button.
- Type the directory path and file name of the HTML report file you want to view and click **OK** or use the **Browse** button to look through the directories and locate the report.

Create a Report

 A. Click on the **Show Report** button



B. Toggle ON the **Create a new report** options and click on **General Report**



C. Click **OK**. The report generation may take a few minutes. ANSYS will generate a professional looking report summarizing the model definition including element type, number of nodes and elements, applied loads, and constraints. All stresses, displacements and reaction force components will be plotted and summarized as well.

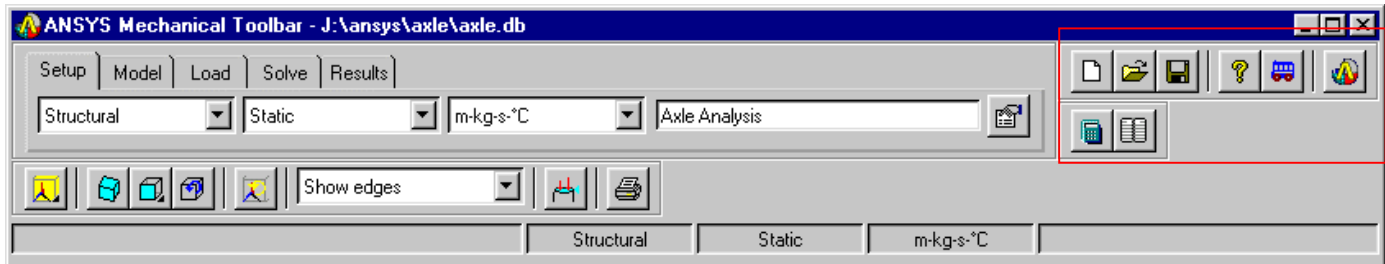
D. When complete, ANSYS will launch the report in your Internet browser. Take a few moments to review each section. This report can be customized and included in other documentation by cut and paste, or through hyperlinks.

E. For a sample HTML report, click [here](#).

7. Conclusion:

The true intent of this exercise was to guide you through the Mechanical Toolbar and explain the various options that are available to you. The analysis of the axle showed that the part displayed no signs of yield under the given loading condition. Mesh refinements can be made in localized areas to determine more accurate stress values, but we will explore that in later exercises.

8. Additional Functions:



8.1. New Model:

This option allows you to clear the database stored in memory and start with a new one.



8.2. Resume Model:

Restores the database from the database file as it was at the last time that it was saved. This button is valid only for resuming database files that were generated using the Mechanical Toolbar. If you want to bring an ANSYS model into the Mechanical Toolbar, you need to use the Import Geometry button instead.



8.3. Save Model:

Saves the current model to a database file.

[A. Let's save our work. Click on the **Save** button in the MTB.](#)



8.4. Context Help:

Invokes your system's default Web browser and displays the table of contents for the Mechanical Toolbar's HTML-based help.



8.5. Tour:

Displays the Mechanical Toolbar's tour help. The tour provides an overview of the Mechanical Toolbar, its controls, and how to use it to perform an analysis.



8.6. Fully Functional ANSYS:

Switches to full ANSYS functionality. Note that only one environment may be brought over.



8.7. System Calculator:

Invokes the system calculator Use the Toolbar Properties dialog box to define the system calculator.



8.8. System Editor:

Invokes the system editor (if one is available). Use the Toolbar Properties dialog box to define the system editor.