

Chapter 8

Linear Structural Analysis

A linear analysis is conducted if a structure is expected to exhibit linear behavior. The deformation and load-carrying capability can be determined by employing one of the analysis types available in ANSYS, static or dynamic, depending on the nature of the applied loading. If the applied loading is determined as part of the solution for structural stability, a buckling analysis is conducted. If the structure is subjected to thermal loading, the analysis is referred to as thermomechanical.

8.1 Static Analysis

The behavior of structures under static loading can be analyzed by employing different types of elements within ANSYS. The nature of the structure dictates the type of elements utilized in the analysis. Discrete or framed structures are suitable for modeling with rod- and beam-type elements. However, the modeling of continuous structures usually requires a three-dimensional model with solid elements.

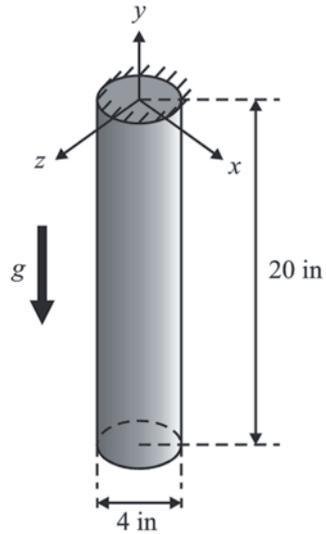
Under certain types of loading and geometric conditions, the three-dimensional type of analysis can be idealized as a two-dimensional analysis. If the component is subjected to in-plane loading only and its thickness is small with respect to the other length dimensions, it is idealized as a plane stress condition. If the component with a uniform cross section is long in the depth direction and is subjected to a uniform loading along the depth direction, it is idealized as a plane strain condition. If the component has a circular cross section and is subjected to uniform and concentric loading, it possesses axisymmetry. If thin structural components are subjected to lateral loading, the plate and shell elements are suitable for analysis.

8.1.1 Trusses

A truss is a structure that is made of straight structural members capable of carrying loads only in their own direction, i.e., no shear forces, no moments. Thus, each

The online version of this book (doi: 10.1007/978-1-4939-1007-6_8) contains supplementary material, which is available to authorized users

Fig. 8.1 Schematic of a bar deformed due to its own weight



member is under either axial tension or axial compression. These members are connected to each other by means of joints. It is assumed that loads can only be applied at the joints. **LINK180** is the element for modeling truss structures. The degrees of freedom at each node for truss elements are the displacement components u_x , u_y , and u_z . However, the vector sum of the deformations (elongation or contraction, *not* the displacements) is aligned with the direction of the element. Two example problems are given to demonstrate the usage of truss elements within ANSYS.

8.1.1.1 Elongation of a Bar Under its Own Weight Using Truss Elements

Consider a steel bar of uniform cross section whose upper end is supported such that it is fixed from translational motion. The mass density, elastic modulus, and Poisson's ratio of steel are $\rho = 0.284\text{lb/in}^3$, $E = 30 \times 10^6\text{psi}$, and $\nu = 0.3$, respectively. The radius and length of the bar are assumed to be $r = 2\text{in}$ and $l = 20\text{in}$, respectively, and the gravitational acceleration is $g = 386.2205\text{in/sec}^2$. The goal is to find the elongation of the bar at the lower end due to its own weight. The positive y -direction is the opposite direction of the gravitational acceleration, as shown in Fig. 8.1.

This problem can be solved using two-dimensional truss, two-dimensional axisymmetric plane, or three-dimensional elements. Since, we are interested in the elongation only, two-dimensional truss elements (**LINK180**) are used to obtain the solution.

Model Generation

- Specify the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
 - Select *Link* immediately below *Structural Mass* from the left list and *3D finite 180* from the right list; click on *OK*.
 - Click on *Close*.
- Specify real constants (**R** command) using the following menu path:

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete

- Click on *Add*.
 - Highlight *Type 1 Link 180*; click on *OK*.
 - Enter *12.5664* (calculated based on radius, $r = 2$ in) for *AREA*; click on *OK*.
 - Click on *Close*.
- Specify material properties for the bar (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on *Structural* and *Density*, which will bring up another dialog box.
- Enter *0.284* for *DENS*; click on *OK*.
- In order to specify the elastic modulus and Poisson's ratio, in the *Define Material Model Behavior* dialog box, in the right window, successively left-click on *Structural*, *Linear*, *Elastic*, and, finally, *Isotropic*, which will bring up another dialog box.
- Enter *30e6* for *EX* and *0.3* for *PRXY*; click on *OK*.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create keypoints (**K** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

- A total of 2 keypoints will be created.
 - Enter (x, y) coordinates of keypoint 1 as $(0, 0)$; click on *Apply*.
This action will keep the *Create Keypoints in Active Coordinate System* dialog box open. If the *NPT Keypoint number* field is left blank, then ANSYS assigns the lowest available keypoint number to the keypoint that is being created.
 - Repeat the same procedure for keypoint 2 using $(0, -20)$ for the (x, y) coordinates.
 - Click on *OK* (instead of *Apply*).
- Create a line (**L** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line

- *Pick Menu* appears; first pick keypoint 1, then keypoint 2; click on **OK**.
- Specify the number of divisions on the line (**LESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrl > ManualSize > Lines > Picked Lines

- *Pick Menu* appears; pick the line; click on **OK**.
- *Element Sizes on Picked Lines* dialog box appears; enter 20 for **NDIV**; click on **OK**.
- Create the mesh (**LMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Lines

- *Pick Menu* appears; pick the line; click on **OK**.
- Review elements.
 - Turn on element numbering using the following menu path:

Utility Menu > PlotCtrls > Numbering

- Select *Element numbers* from the first pull-down menu.
- Plot elements (**EPLOT** command) using the following menu path:

Utility Menu > Plot > Elements

- Turn off element numbering and turn on node numbering using the following menu path:

Utility Menu > PlotCtrls > Numbering

- Place a *checkmark* by clicking on the empty box next to **NODE Node numbers**.
- Select *No numbering* from the first pull-down menu.
- Click on **OK**.
- Plot nodes (**NPLOT** command) using the following menu path:

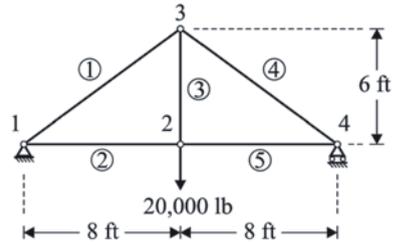
Utility Menu > Plot > Nodes**Solution**

- Apply displacement constraints (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick node 1 (upper end); click on **OK** in the *Pick Menu*.
- Highlight *All DOF*; click on **OK**.

Fig. 8.2 Schematic of the truss structure with symmetry



- Apply gravitational acceleration (**ACEL** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Inertia > Gravity > Global

- *Apply (Gravitational) Acceleration* dialog box appears.
- Enter **386.2205** for *ACELY*; click on **OK**.

- Obtain the solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review displacement values (**PRNSOL** command) using the following menu path:

Main Menu > General Postproc > List Results > Nodal Solution

- Click on *Nodal Solution*, *DOF Solution*, and *Y-component of displacement*; click on OK.
- The list appears. Note that the value for the *y*-displacement at node 2 (lower end) is listed as **-0.73124E-03** (in inches).

8.1.1.2 Analysis of a Truss Structure with Symmetry

Consider the steel truss structure shown in Fig. 8.2, which possesses symmetry with respect to the ordinate. Node and element numbers are also shown in this figure. Element 3 has a cross-sectional area of $A = 20 \text{ in}^2$, while the other elements have $A = 10 \text{ in}^2$. The elastic modulus for all of the elements is $E = 30 \times 10^6 \text{ psi}$. The goal is to find the displacements at the nodes and the stresses in the elements. Due to the

symmetry condition, only half the geometry is modeled with appropriate boundary conditions, i.e., the x -displacement at nodes 2 and 3 is zero and the applied force at node 2 is halved. Also, for the element located along the symmetry line, one half of the cross-sectional area is used.

The solution obtained using ANSYS is as follows:

Model Generation

- Specify the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Link* immediately below *Structural Mass* from the left list and *3D finite 180* from the right list; click on *OK*.
- Click on *Close*.

- Specify real constants (**R** command) using the following menu path:

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete

- Click on *Add*.
- Highlight *Type 1 Link 180*; click on *OK*.
- Enter *10* for *AREA*; click on *OK*.
- Click on *Close*.

- Specify material properties for the bar (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on *Structural*, *Linear*, *Elastic*, and, finally, *Isotropic*, which will bring up another dialog box.
- Enter *30e6* for *EX* and *0* for *PRXY*; click on *OK*. Click on *OK* in the subsequent warning message.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create nodes (**N** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS

- A total of 3 nodes will be created.
- Enter the (x, y) coordinates of node 1 as $(0, 0)$; click on *Apply*.
- Repeat the same procedure for nodes 2 and 3 using $(96, 0)$ and $(96, 72)$, respectively, for the (x, y) coordinates.
- After entering the coordinates for node 3, click on *OK* (instead of *Apply*).

- A total of 3 elements will be created. Element 1 is defined by nodes 1 and 3 [1–3]. Similarly, elements 2 and 3 are defined by nodes [1–2] and [2–3], respectively. Create elements (**E** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru Nodes

- *Pick Menu* appears; create elements by picking two nodes at a time and clicking on **Apply** in between.
- Observe the elements created after clicking on **Apply** in the *Pick Menu*.
- Repeat until element 3 is created; click on **OK**.

Solution

- Apply displacement constraints (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick node 1; click on **OK** in the *Pick Menu*.
- Highlight **UY**; click on **Apply**.
- *Pick Menu* reappears; pick nodes 2 and 3; click on **OK** in the *Pick Menu*.
- Click on **UY** to remove the highlight then click on **UX** to highlight.
- Click on **OK**.

- Apply force boundary conditions (**F** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

- *Pick Menu* appears; pick node 2; click on **OK** in the *Pick Menu*.
- Select **FY** from pull-down menu and enter **-10000** for *Force/moment value*; click on **OK**.

- Obtain the solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status, if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review displacement values (**PRNSOL** command) using the following menu path:

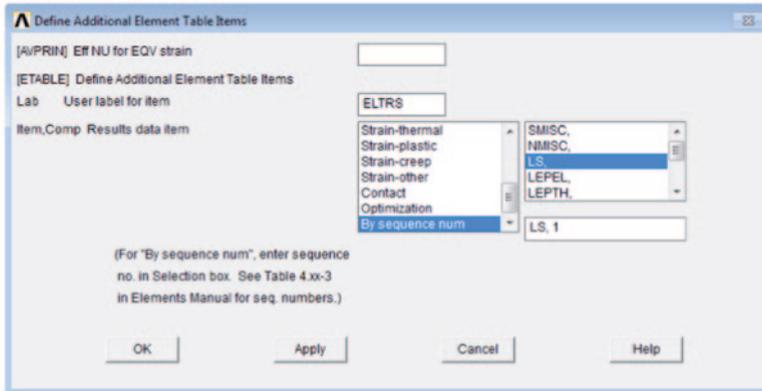


Fig. 8.3 Dialog box for retrieving element results based on sequence numbers

Main Menu > General Postproc > List Results > Nodal Solution

- *List Nodal Solution* dialog box appears. Click on **DOF Solution** and **Y-component of displacement**; click on **OK**.
- The list appears with the values for the y -displacement at nodes 2 and 3 as $-0.19200E-01$ and $-0.16800E-01$, respectively.
- Review element stress values (**ETABLE** command) using the following menu path:

Main Menu > General Postproc > Element Table > Define Table

- *Element Table Data* dialog box appears. Click on **Add**, which brings up the *Define Additional Element Table Items* dialog box. Enter a label (**Lab**) for element stresses, say **ELSTRS**. Scroll down in the left list; click on **By Sequence num**; click on **LS** in the right list. Finally, enter **LS,1** in the last text field, as shown in Fig. 8.3; click on **OK**.
- Note that the element table **ELSTRS** is now listed in the *Element Table Data* dialog box; click on **Close**.
- List the element table (**PRETAB** command) using the following menu path:

Main Menu > General Postproc > Element Table > List Elem Table

- In the *List Element Table Data* dialog box, highlight **ELSTRS**; click on **OK**.
- The list appears with stresses in elements 1, 2, and 3 as -1666.7 , 1333.3 , and 1000 , respectively.

8.1.2 Beams

A beam is a structural member capable of carrying axial, shear, and moment loads. Unlike truss members, loads can be applied anywhere along the beam geometry.

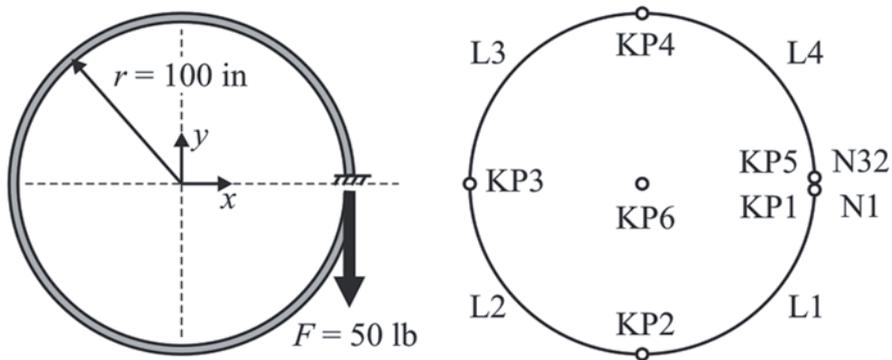


Fig. 8.4 Schematic of a circular steel ring and the corresponding solid model (*left*) and *Beam Tool* dialog box (*right*)

ANSYS provides several element types for modeling beams. The most commonly used one is **BEAM188** for two- and three-dimensional analyses, respectively. At each node, both displacements and rotations are the degrees of freedoms for structural beam elements ($u_x, u_y,$ and θ_z for 2-D; $u_x, u_y, u_z, \theta_x, \theta_y,$ and θ_z for 3-D). Two example problems are considered in this section for the demonstration of the usage of beam elements within ANSYS.

8.1.2.1 Analysis of a Slit Ring

A circular steel ring with a slit, as shown in Fig. 8.4, is subjected to a 50-lb vertical force acting in the negative y -direction at the termination point while translations and rotations are constrained in every direction. The ring has a solid circular cross section with radius 1 in. The structure is modeled using beam elements with cross-sectional area $A = \pi$, elastic modulus $E = 30 \times 10^6$, Poisson's ratio $\nu = 0.3$, and moment of inertia $I_{zz} = \pi/4$. The goal is to find the displacements at the nodes and the moment diagram. The solid model used in the ANSYS solution is also shown in Fig. 8.4 (left), with the keypoint and line numbers indicated.

Model Generation

- Specify the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Beam* immediately below *Structural Mass* from the left list and *2node 188* from the right list; click on *OK*.
- Click on *Close*.

- Specify geometry for the beam (**SECTYPE** command) using the following menu path:

Main Menu > Preprocessor > Sections > Beam > Common Sections

- *Beam Tool* dialog box appears; select the solid circular section from the **Sub-Type** pull-down menu.
- Enter **I** for **R**.
- Exit from the *Beam Tool* dialog box (Fig. 8.4 (right)) by clicking on **OK**.
- Specify material properties for the beam (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**, which will bring up another dialog box.
- Enter **30e6** for **EX** and **0.3** for **PRXY**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create keypoints (**K** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

- A total of 6 keypoints will be created.
- Enter the (x, y) coordinates of keypoint 1 as **(100, 0)**; click on **Apply**.
This action will keep the *Create Keypoints in Active Coordinate System* dialog box open. If the **NPT Keypoint number** field is left blank, then ANSYS assigns the lowest available keypoint number to the keypoint that is being created.
- Referring to Fig. 8.4, repeat the same procedure for keypoints 2, 3, 4, 5, and 6 using **(0, -100)**, **(-100, 0)**, **(0, 100)**, **(100, 0)**, and **(0, 0)**, respectively, for the (x, y) coordinates.
- After generating keypoint 6, click on **OK** (instead of **Apply**).
- Note that keypoints 1 and 5 are coincident. This is intentional, so the slit can be modeled properly.

- Create arcs (**LARC** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Lines > Arcs > By End KPs & Rad

- A total of 4 lines (arcs) will be created.
- *Pick Menu* appears; pick keypoints 1 and 2 (end points of the arc); click on **OK** in the *Pick Menu*.
- Pick keypoint 6 (center of the arc); click on **OK** in the *Pick Menu*.

- *Arc by End KPs & Radius* dialog box appears; enter **100** for **RAD Radius of the arc**.
- Click on **Apply**; line 1 is created.
- Repeat this procedure for lines 2, 3, and 4 using keypoint pairs (2, 3), (3, 4), and (4, 5), respectively. All lines use keypoint 6 as the center and 100 as the radius.
- Specify the number of divisions on all lines (**LESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrl > ManualSize > Lines > All Lines

- *Element Sizes on All Selected Lines* dialog box appears; enter **10** for **NDIV**; click on **OK**.
- Create the mesh (**LMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Lines

- *Pick Menu* appears; click on **Pick All**.

Solution

- Apply displacement constraints (**D** command) using the following menu path:

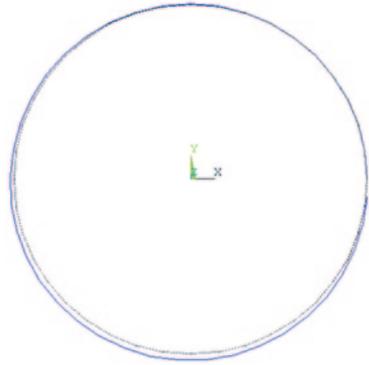
Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick one of the nodes at $x=100$ and $y=0$. There are two nodes at this location: nodes 1 and 32. When picking, ANSYS asks the user which one of the nodes is to be picked. Click on the **Next** button in this *Warning Window* so that it shows **Node 32**; click on **OK** in the *Pick Menu*.
- *Apply U, Rot on Nodes* dialog box appears; highlight **All DOF**; click on **OK**.
- Apply force boundary conditions (**F** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

- *Pick Menu* appears; this time pick node 1 (instead of node 32); click on **OK** in the *Pick Menu*.
- Select **FY** from the pull-down menu and enter **-50** for **Force/moment value**; click on **OK**.
- Obtain the solution (**SOLVE** command) using the following menu path:

Fig. 8.5 Deformed shape of the steel ring under applied boundary conditions



Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Review the deformed shape (**PLDISP** command) using the following menu path:

Main Menu > General PostProc > Plot Results > Deformed Shape

- Select **Def + undeformed**; click on **OK**.
- The deformed shape is shown in Fig. 8.5 as it appears in the *Graphics Window*.

- Store bending moment values in the element table (**ETABLE** command) using the following menu path:

Main Menu > General Postproc > Element Table > Define Table

- *Element Table Data* dialog box appears; click on **Add**.
- *Define Additional Element Table Items* dialog box appears. Enter a label name, say MZI, in the User label for item text field. In the left list, scroll down to select **By sequence number** and select **SMISC** in the right list. Finally, type **SMISC,3** in the last text field; click on **Apply** (Fig. 8.6).
- Repeat this procedure for **MZJ** using **SMISC, 16**. When done, click on **OK** (instead of **Apply**).
- Note that **SMIS3** now appears in the list in the *Element Table Data* dialog box. Exit from the *Element Table Data* dialog box by clicking on **Close**.
- Plot the moment diagram (**PLLS** command) using the following menu path:

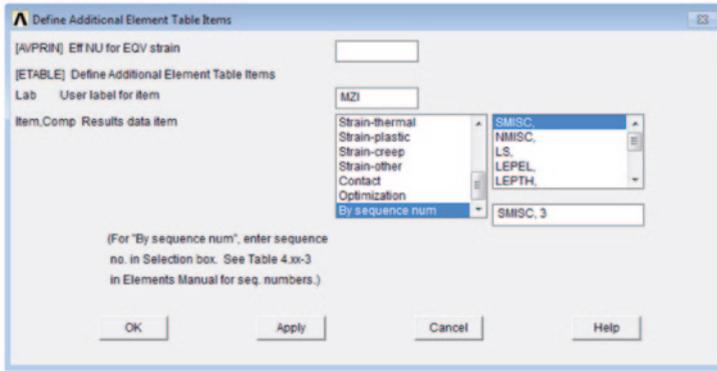


Fig. 8.6 Define Additional Element Table Items dialog box for extracting nodal moment values

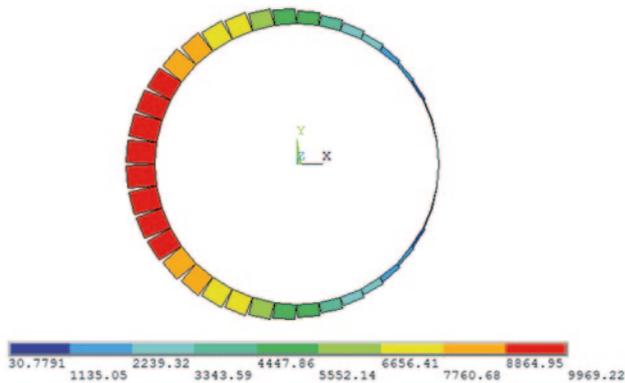


Fig. 8.7 Moment diagram of the steel ring

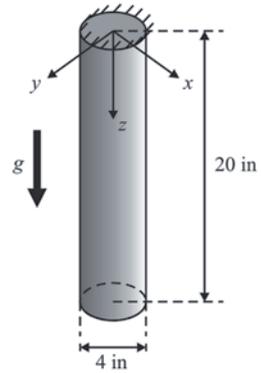
Main Menu > General Postproc > Plot Results > Contour Plot > Line Elem Res

- Plot Line-Element Results dialog box appears; click on **OK**.
- Figure 8.7 shows the resulting moment diagram as displayed in the Graphics Window.

8.1.3 Three-Dimensional Problems

Almost all engineering problems are three-dimensional (3-D) by nature. However, depending on the specific geometry, loading conditions, and quantities of interest, it is common to approach the problem with the idealization of a lower dimensionality. If a representative idealization cannot be utilized, then a three-dimensional model must be created. The most commonly used three-dimensional structural element is, **SOLID185** which is an 8-noded brick element. The degrees of freedom at each

Fig. 8.8 Schematic of a bar deformed due to its own weight



node for 3-D problems are u_x , u_y , and u_z . Determining the deformation of a bar under its own weight using three-dimensional elements in ANSYS is demonstrated in the following.

The elongation of a bar due to its own weight was modeled in Sect. 8.1.1.1 using two-dimensional link elements. That solution provided the displacement of the bar in the longitudinal direction, and the same cross-sectional area is assumed. Three-dimensional elements provide the change in the cross-sectional area, as well as the displacement components. The reference frame shown in Fig. 8.8 is used in the 3-D solution.

Model Generation

- Specify the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Solid* immediately below *Structural Mass* from the left list and *Brick 8node 185* from the right list; click on *OK*.
- Click on *Close*.

- Specify material properties for the bar (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on *Structural* and *Density*, which will bring up another dialog box.
- Enter **0.2839605** for *DENS*; click on *OK*.
- In order to specify the elastic modulus and Poisson's ratio, in the *Define Material Model Behavior* dialog box, in the right window, successively left-click on *Structural*, *Linear*, *Elastic*, and, finally, *Isotropic*, which will bring up another dialog box.

- Enter **30e6** for **EX** and **0.3** for **PRXY**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create a volume (**CYLIND** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > By Dimensions

- *Create Cylinder by Dimensions* dialog box appears. Enter **2** for **RAD2**, **20** for **Z2**, and **90** for **THETA2**; click on **OK**.
- Create additional volumes by reflection (**VSymm** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Reflect > Volumes

- *Pick Menu* appears; click on **Pick All** button, which brings up the *Reflect Volumes* dialog box.
- Click on the **Y-Z Plane X** radio-button; click on **Apply**.
- *Pick Menu* reappears; click on **Pick All** button and in the *Reflect Volumes* dialog box click on the **X-Z Plane Y** radio-button; click on **OK**.
- Glue the volumes (**VGLUE** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Volumes

- *Pick Menu* appears; click on **Pick All** button.
- Specify the global element size (**ESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size

- *Global Element Sizes* dialog box appears; enter **1** for **SIZE**; click on **OK**.
- Create the mesh (**VMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Volumes > Mapped > 4 to 6 sided

- *Pick Menu* appears; click on **Pick All**.

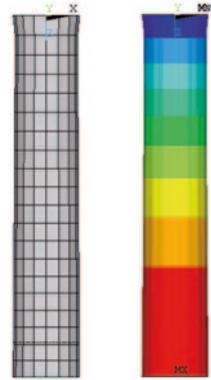
Solution

- Apply displacement constraints (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick all the nodes at $z=0$ (use different viewpoints if necessary); click on **OK** in the *Pick Menu*.
- Highlight **All DOF**; click on **OK**.

Fig. 8.9 Deformed shape (*left*) and contour plot of the z-displacement (*right*) of the bar due to its own weight



- Apply gravitational acceleration (**ACEL** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Inertia > Gravity > Global

- Apply (*Gravitational*) *Acceleration* dialog box appears.
- Enter **-386.2205** for **ACELZ**; click on **OK**.

- Obtain the solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- Confirmation Window appears along with Status Report Window.
- Review status, if OK, close the Status Report Window; click on **OK** in the Confirmation Window.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review the deformed shape (**PLDISP** command) using the following menu path:

Main Menu > General PostProc > Plot Results > Deformed Shape

- Select **Def + undef edge**; click on **OK**.
- The deformed shape is shown in Fig. 8.9 as it appears in the *Graphics Window*.

- Review z-displacement contours (**PLNSOL** command) using the following menu path:

Main Menu > General PostProc > Plot Results > Contour Plot > Nodal Solu

- Contour Nodal Solution Data dialog box appears. Click on **Nodal Solution**, **DOF Solution**, and then **Z-component of displacement**; click on **OK**.
- The contour plot is shown in Fig. 8.9 as it appears in the *Graphics Window*.

- Review displacement values (**PRNSOL** command) using the following menu path:

Main Menu > General Postproc > List Results > Nodal Solution

- Click on *Nodal Solution*, *DOF Solution*, and then *Z-component of displacement*; click on *OK*.
- The list appears in a separate window. It is a long list of *z*-displacements.
- At the bottom of the window maximum displacement value is printed as **0.72386E-03**.

8.1.4 Two-Dimensional Idealizations

As mentioned in Sect. 6.2.2, the reduction of the dimensionality of a problem from three to two through an idealization may reduce the computational cost significantly. There are three distinct two-dimensional idealizations: plane stress, plane strain, and axisymmetry.

Plane stress and strain idealizations are discussed in Sects. 6.2.2.1 and 6.2.2.2, respectively. Therefore, the descriptions given in the following subsections are brief.

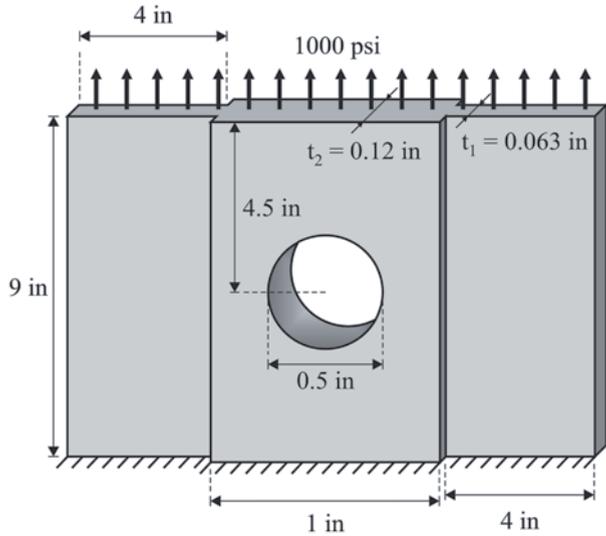
8.1.4.1 Plane Stress

In a structural problem, if one of the dimensions is much smaller than the in-plane dimensions, and if the structure is subjected to only in-plane loads along the boundary, then the plane stress idealization is valid. It reduces the computational cost significantly without a loss of accuracy in the quantities of interest. Plane stress idealization is demonstrated by considering a plate with a circular hole and a composite plate under axial tension.

Analysis of a Plate with a Circular Hole

A square plate (9×9 in²) with a circular hole (radius $r=0.25$ in) is subjected to uniformly distributed tensile loading (1000 psi) in the vertical direction along its top surface while being fixed along the bottom surface (Fig. 8.10). The plate is stiffened by means of increased thickness, from 0.063 to 0.12 in. Plane stress idealization is used in the ANSYS solution, as the plate is thin and there are no lateral loads. The material properties are given as elastic modulus $E=10 \times 10^6$ psi and Poisson's ratio $\nu=0.25$. The goal is to obtain the displacement and stress fields resulting from the applied boundary conditions.

Fig. 8.10 Geometry and loading of the plate with a circular hole



Model Generation

- Define the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Solid* immediately below *Structural Mass* in the left list and *Quad 4 Node 182* in the right list; click on *OK*.
- Click on *Options*.
- *PLANE182 element type options* dialog box appears; select the *Plane strs w/ thk* item from the pull-down menu corresponding to *Element behavior K3*.
- Click on *OK*; click on *Close*.
- Specify the thickness information using real constants (**R** command) using the following menu path:

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete

- *Real Constants* dialog box appears; click on *Add*. Click on *OK*; *Real Constants Set Number 1 for PLANE182* dialog box appears.
- Type **0.063** in the *Thickness THK* text field; click on *Apply*.
- Change the *Real Constant Set No.* from **1** to **2** and modify the *Thickness THK* text field to be **0.12**; click on *OK*.
- Exit from the *Real Constants* dialog box by clicking on *Close*.
- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- *Define Material Model Behavior* dialog box appears. In the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**, which brings up another dialog box.
- Enter **10e6** for **EX** and **0.25** for **PRXY**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create a square area (**RECTNG** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

- In the *Create Rectangle by Dimensions* dialog box, enter **0** and **0.5** for **X1** and **X2** and **0** and **0.5** for **Y1** and **Y2**; click on **OK**.
- Create a circular area for the hole geometry (**PCIRC** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions

- In the *Create Circle by Dimensions* dialog box, type **0.25** for **Outer radius**; click on **OK**.
- Subtract the circle from the rectangle (**ASBA** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

- *Pick Menu* appears; pick the rectangle; click on **OK**; pick the circle; click on **OK**.
- Create additional rectangular areas (**RECTNG** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

- In the *Create Rectangle by Dimensions* dialog box, enter **0.5** and **4.5** for **X1** and **X2** and **0** and **0.5** for **Y1** and **Y2**; click on **Apply**.
- Now, enter **0.5** and **4.5** for **X1** and **X2** and **0.5** and **4.5** for **Y1** and **Y2**; click on **Apply**.
- Finally, enter **0** and **0.5** for **X1** and **X2** and **0.5** and **4.5** for **Y1** and **Y2**; click on **OK**.
- Glue the areas (**AGLUE** command) using the following menu path:

Fig. 8.11 Areas after gluing operation

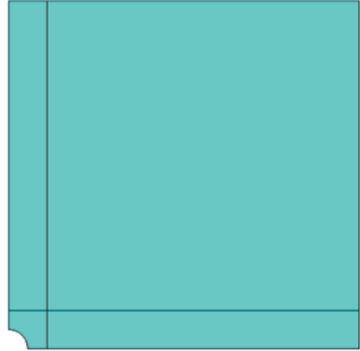
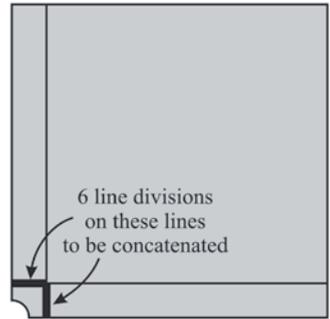


Fig. 8.12 Number of divisions on identified lines



Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Areas

- *Pick Menu* appears; click on **Pick All** button.
- The areas appear in the *Graphics Window*, as shown in Fig. 8.11.

- Specify the global element size (**ESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size

- *Global Element Sizes* dialog box appears; enter **0.1** for **SIZE**; click on **OK**.

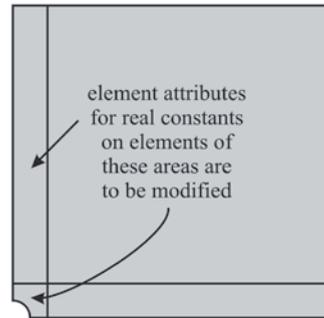
- Specify the number of divisions on selected lines (**LESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

- *Pick Menu* appears; pick the two lines identified in Fig. 8.12; click on **OK**.
- *Element Sizes on Picked Lines* dialog box appears; enter 6 for **NDIV** and remove the checkmark next to **KYNDIV SIZE, NDIV can be changed** so that it shows **No**; click on **OK**.

- Concatenate lines (**LCCAT** command) using the following menu path:

Fig. 8.13 Element attributes on elements attached to identified areas



Main Menu > Preprocessor > Meshing > Concatenate > Lines

- *Pick Menu* appears; pick the two lines identified in Fig. 8.12; click on **OK**.
- Create the mesh (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- *Pick Menu* appears; click on **Pick All**.
- Modify the real constant set attribute of the elements corresponding to the thicker portion of the plate (**EMODIF** command) using the following menu path:

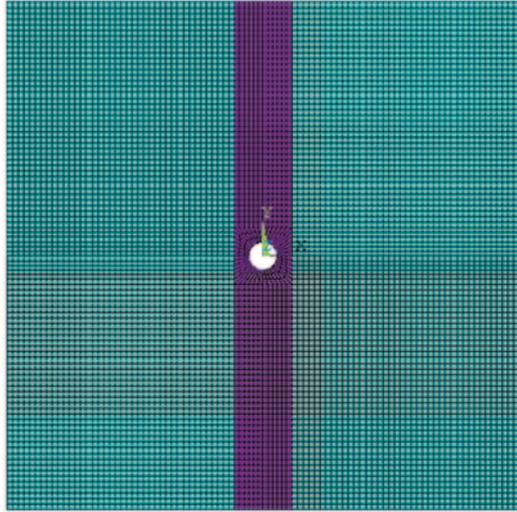
Main Menu > Preprocessor > Modeling > Move/Modify > Elements > Modify Attrib

- *Pick Menu* appears; pick the elements corresponding to the areas indicated in Fig. 8.13 (click on the **Box** radio-button in the *Pick Menu* and draw a rectangle in the *Graphics Window* to pick the elements). Clicking on **OK** brings up the *Modify Elem Attributes* dialog box.
- Select **Real const REAL** from the pull-down menu and enter 2 in the **II New attribute number** field; click on **OK**.
- Create two successive reflective symmetric meshes (**ARSYM** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Reflect > Areas

- *Pick Menu* appears; click on **Pick All**.
- *Reflect Areas* dialog box appears; click on the **Y-Z plane X** radio-button; click on **Apply**.
- *A Warning Window* appears; click on **OK**.
- *Pick Menu* reappears; click on **Pick All**.
- *Reflect Areas* dialog box reappears; click on the **X-Z plane Y** radio-button; click on **OK**.
- Plot elements (**EPLOT** command) using the following menu path:

Fig. 8.14 Elements of the plate, as they appear in the *Graphics Window*



Utility Menu > Plot > Elements

- Although it is not apparent through visual inspection, there are duplicate entities (keypoints, lines, and nodes) along the symmetry lines, thus there is no continuity. Therefore, merge duplicate entities using the following menu path:

Main Menu > Preprocessor > NumberingCtrls > Merge Items

- In the dialog box, select *All* from the first pull-down menu; click on *OK*.
- Plot elements with different colors based on their real constant numbers using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears. Select *Real const num* from the first pull-down menu (corresponding to *Elem/Attrib numbering*) and select *Colors only* from the second pull-down menu (corresponding to *[/NUM] Numbering shown with*); click on *OK*.
- Plot elements (**E**PL**O**T command) using the following menu path:

Utility Menu > Plot > Elements

- Figure 8.14 shows the corresponding element plot with different colors¹ based on material numbers.

¹ Colors have not been used in the printed version of the figures. See the accompanying CD-ROM for color versions of the figures.

Solution

- Apply displacement constraints (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the nodes along the bottom surface of the plate (click on the **Box** radio-button in the *Pick Menu* and draw a rectangle in the *Graphics Window* to pick the nodes); click on **OK** in the *Pick Menu*.
- Highlight both **UX** and **UY**; click on **OK**.

- Apply surface force (pressure) boundary conditions (**SF** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Nodes

- *Pick Menu* appears; pick the nodes along the top surface of the plate; click on **OK** in the *Pick Menu*.
- Type **-1000** (negative 1000) for **VALUE Load PRES value**; click on **OK**.
- Pressure, by definition, acts normal toward the body along the surface. The direction of action in reference to the global coordinate system does not affect whether it is positive or negative. The only factor that dictates the sign is whether it acts toward or away from the body. Therefore, in order to apply the tensile loading, it is necessary to apply negative pressure.

- Obtain the solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review the normal stress contour plots in the *x*- and *y*-directions (**PLNSOL** command) using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- Click on **Nodal Solution, Stress, and X-component of stress**; click on **OK**.
- The contour plot of σ_{xx} appears in the *Graphics Window*, as shown in Fig. 8.15.
- The Contour plot of σ_{yy} is obtained similarly by selecting the **Y-component of stress** from the list and clicking on **OK** (shown in Fig. 8.16).

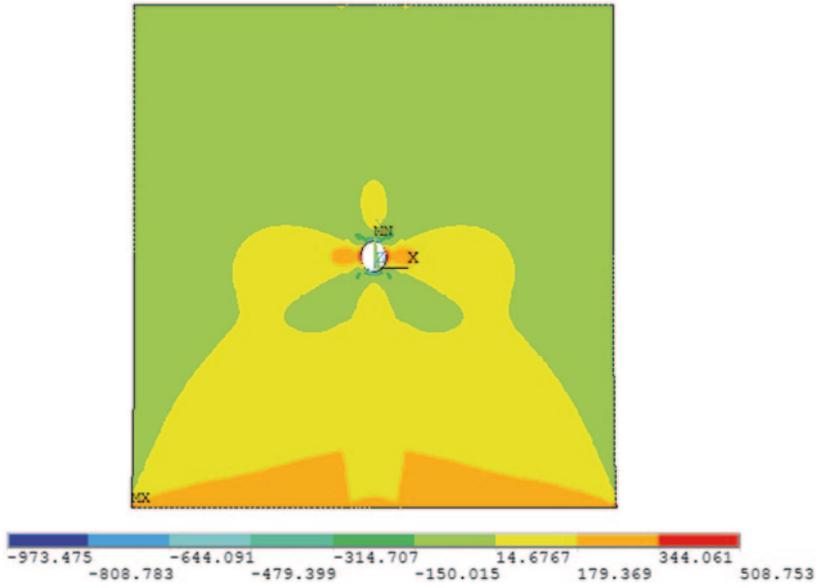


Fig. 8.15 Contour plot of σ_{xx}

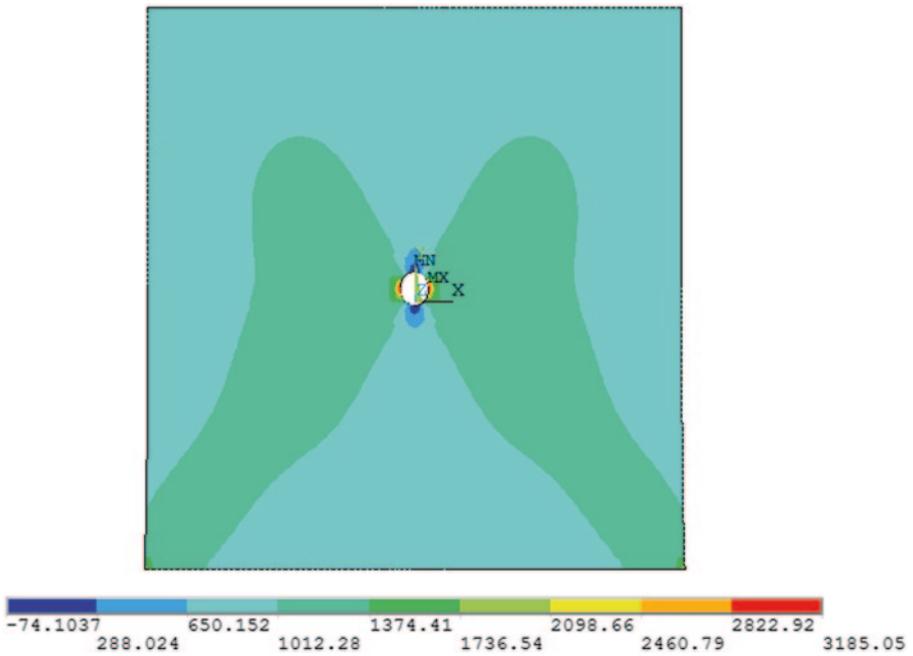


Fig. 8.16 Contour plot of σ_{yy}

- Review the variation of stresses along a path by means of a line plot. This operation requires the path to be defined first, followed by mapping the solution items of interest onto the path and, finally, obtaining the plot. The path that is defined in this case lies along the positive x -axis, starting from the left boundary of the hole and ending at the left boundary of the plate. Define the path (**PPATH** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Define Path > By Nodes

- *Pick Menu* appears; pick the nodes with (x, y) coordinates (0.25, 0) and (4.5, 0); click on **OK**.
- *By Nodes* dialog box appears; enter a name describing the path, say *hrz*, in the **Define Path Name** text field; click on **OK**.
- Close the *PATH Command Status Window*.
- Map results onto path (**PDEF** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Map onto Path

- *Map Result Items onto Path* dialog box appears; select **Stress** from the left list and **Y-direction SY** from the right list; click on **OK**.
- Obtain line plot of σ_{yy} along the path (**PLPATH** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Plot Path Item > On Graph

- *Plot of Path Items on Graph* dialog box appears; select **SY**; click on **OK**.
- Figure 8.17 shows the line plots of σ_{xx} and σ_{yy} along the defined path.

Composite Plate Under Axial Tension

A fiber-reinforced square plate, shown in Fig. 8.18, is subjected to a uniform stress field of 20 ksi along the top and bottom boundaries. The sides of the plate are 10 in long, and the fibers are oriented at a 45° angle to the global Cartesian coordinate system. Material properties are specified as $E_1 = 10 \times 10^3$ ksi, $E_2 = 30 \times 10^3$ ksi, $G_{12} = 15 \times 10^3$ ksi, and $\nu_{12} = 0.1$. The goal is to find the displaced shape.

Model Generation

- Define the element type (**ET** command) using the following menu path:

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

- Click on **Add**.
- Select **Solid** immediately below **Structural Mass** in the left list and **Quad 4 Node 182** in the right list; click on **OK**.
- Click on **Close**.

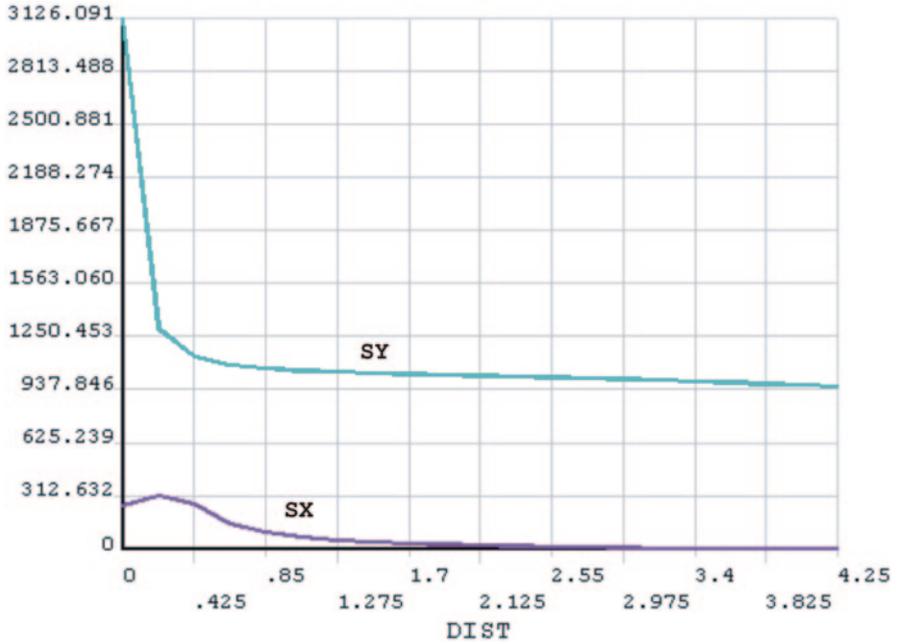
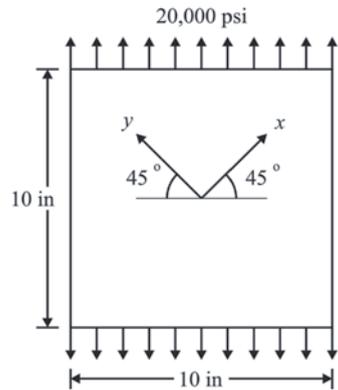


Fig. 8.17 Line plots of σ_{xx} and σ_{yy} along the defined path

Fig. 8.18 Schematic of the composite plate, fiber orientation, and loading



- Specify material properties by typing the following four commands in the *Input Field* (at the end of each command, hit the Enter key to execute):

```

MP, EX,1,10E6
MP, EY,1,30E6
MP, PRXY,1,0.1
MP, GXY,1,15E6
    
```

- Create keypoints (**K** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

- A total of 4 keypoints will be created.
- Enter (x, y) coordinates of keypoint 1 as $(-5, -5)$; click on *Apply*.
This action will keep the *Create Keypoints in Active Coordinate System* dialog box open. If the *NPT Keypoint number* field is left blank, then ANSYS assigns the lowest available keypoint number to the keypoint that is being created.
- Repeat the same procedure for the keypoints 2, 3, and 4 using $(5, -5)$, $(5, 5)$, and $(-5, 5)$, respectively, for the (x, y) coordinates.
- Once keypoint 4 is created, click on *OK* (instead of *Apply*).

- Create the area through keypoints (**A** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > Through KPs

- *Pick Menu* appears; pick keypoints 1 through 4 (in sequence); click on *OK*.
- Material properties refer to the fiber directions. However, the global Cartesian coordinates and the fiber directions are at an angle of 45° . Therefore, the *element coordinate system* needs to be aligned with the fiber orientation. For this purpose, create a local coordinate system (**CLOCAL** command) using the following menu path:

Utility Menu > WorkPlane > Local Coordinate Systems > Create Local CS > At Specified Loc

- *Pick Menu* appears; type **0, 0, 0** in the text field in the *Pick Menu*; click on *OK*,
- A dialog box appears; type **45** in the *THXY Rotation about local Z* text field; click on *OK*; local coordinate system 11 is created.
- Align the element coordinate system with local coordinate system 11 (**ESYS** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh Attributes > Default Attrs

- *Meshing Attributes* dialog box appears. Select **11** from the *ESYS Element coordinate sys* pull-down menu; click on *OK*.
- Switch the active coordinate system to global Cartesian using the following menu path:

Utility Menu > Work Plane > Change Active CS to > Global Cartesian

- Specify the number of divisions on all lines (**LESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All Lines

- *Element Sizes on All Selected Lines* dialog box appears; enter **20** for **NDIV**; click on *OK*.

- Mesh the square (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- *Pick Menu* appears; click on **Pick All**.

Solution

- Apply displacement constraints (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the center node, i.e., $x = 0$ and $y = 0$; click on **OK** in the *Pick Menu*.
- Highlight both **UX** and **UY**; click on **Apply**.
- *Pick Menu* reappears; pick the right-side center node, i.e., $x = 5$ and $y = 0$; click on **OK** in the *Pick Menu*.
- Remove the highlight on **UX**, leaving **UY** highlighted; click on **OK**.
- Apply surface force (pressure) boundary conditions (**SF** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Nodes

- *Pick Menu* appears; pick the nodes along the top and bottom surfaces of the plate; click on **OK** in the *Pick Menu*.
- Type **-20000** (negative 20000) for **VALUE Load PRES value**; click on **OK**.
- Pressure, by definition, acts normal toward the body along the surface. The direction of action in reference to the global coordinate system does not affect whether it is positive or negative. The only factor that dictates the sign is whether it acts toward or away from the body. Therefore, in order to apply the tensile loading, it is necessary to apply negative pressure.
- Obtain the solution (**SOLVE** command) using the following menu path:

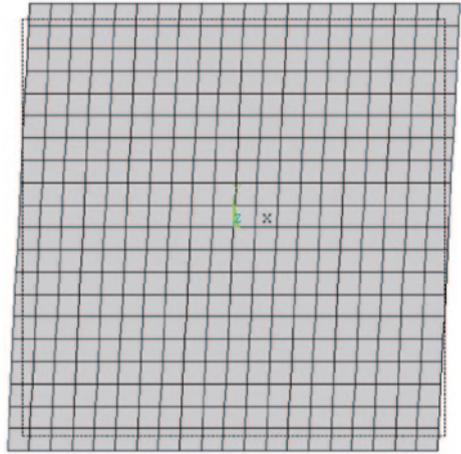
Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status, if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review the deformed shape (**PLDISP** command) using the following menu path:

Fig. 8.19 Deformed shape of the composite plate



Main Menu > General PostProc > Plot Results > Deformed Shape

- Select *Def + undef edge*; click on *OK*.
- The deformed shape is shown in Fig. 8.19 as it appears in the *Graphics Window*.
- Review the *x*-displacement at the top-right and the *y*-displacement at the top-left nodes (**PRNSOL** command) using the following menu path:

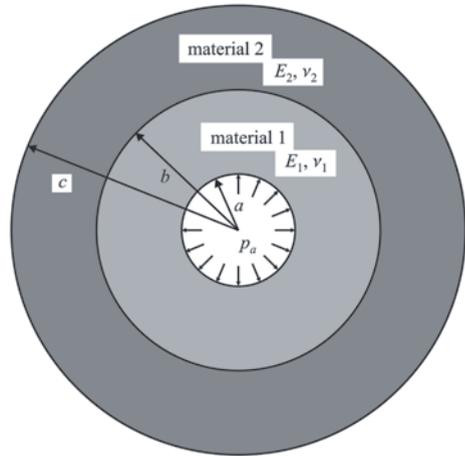
Main Menu > General Postproc > List Results > Nodal Solution

- Click on *Nodal Solution, DOF Solution*, and then *Displacement vector sum*; click on *OK*.
- The list appears. The *x*-displacement at the top-right node (node 22) is given as $0.45E-2$, and the *y*-displacement at the top-left node (node 42) is given as $0.45E-2$.
- In ANSYS, results can also be listed (or displayed) in different coordinate systems. By default, the *Results Coordinate System* is aligned with the Global Cartesian. Align the *Results Coordinate System* with local coordinate system 11 defined earlier using the following menu path:

Main Menu > General Postproc > Options for Outp

- *Options for Output* dialog box appears. Select *Local system* from the first pull-down menu and enter *11* for *Local system reference no*; click on *OK*.
- Now, review the nodal displacements one more time using the following menu path:

Fig. 8.20 Plane strain representation of a bi-material cylindrical pressure vessel under internal pressure



Main Menu > General Postproc > List Results > Nodal Solution

- When the results are transformed to the local coordinate system, the x -displacement at the top-right node becomes $0.6364E-2$, and the y -displacement at the top-left node becomes $0.16499E-2$. Corresponding analytical solution values are $0.6364E-2$ and $0.16495E-2$, producing negligible error values.

8.1.4.2 Plane Strain

In a structural problem, if one of the dimensions is significantly longer than the other dimensions defining a uniform cross-sectional area, and if the structure is subjected to only uniform lateral loads, then plane strain idealization is valid. Similar to plane stress idealization, because the number of nodes and elements in the model is reduced drastically, utilization of plane strain idealization leads to significant savings in computational cost without loss of accuracy in the quantities of interest. Stresses in a bi-material cylindrical pressure vessel are used to demonstrate plane strain idealization.

A bi-material cylinder is subjected to internal pressure, p_a , as shown in Fig. 8.20. The radius of the hollow portion is a , and the thicknesses of the inner and outer cylinders are $(b-a)$ and $(c-a)$, respectively. Perfect contact with no slipping is assumed along the interface, implying displacement continuity. Elastic properties of the inner and outer cylinders are (E_1, ν_1) and (E_2, ν_2) , respectively. The goal is to compute the stress field. The problem is solved with ANSYS using $E_2/E_1 = 0.5$, $\nu_a = \nu_b = 0.33$, $b/a = 2$, and $c/a = 4$, with $a = 1$, $p_a = 1$, and $E_1 = 2$.

Model Generation

- Define the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Solid* immediately below *Structural Mass* in the left list and *Quad 4 Node 182* in the right list; click on *OK*.
- Click on *Options*.
- *PLANE182 element type options* dialog box appears; select *Plane strain* item from the pull-down menu corresponding to *Element behavior K3*.
- Click on *OK*; click on *Close*.

- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- The inner and outer cylinder will have material reference number 1 and 2, respectively. *Define Material Model Behavior* dialog box appears. In the right window, successively left-click on *Structural*, *Linear*, *Elastic*, and, finally, *Isotropic*, which brings up another dialog box.
- Enter *2* for *EX* and *0.33* for *PRXY*; click on *OK*.
- Add new material model using the following menu path:

Material > New Model

- Click on *OK* in the new dialog box.
- In the right window, successively left-click on *Structural*, *Linear*, *Elastic*, and, finally, *Isotropic*; Enter *1* for *EX* and *0.33* for *PRXY*; click on *OK*.
- When finished, close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create partial hollow circular areas (**PCIRC** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions

- In the *Create Circle by Dimensions* dialog box, type *2* for *Outer radius*, *1* for *Inner radius*, *0* for *Theta1*, and *90* for *Theta2*; click on *Apply*.
- Now, type *4* for *Outer radius*, *2* for *Inner radius*, *0* for *Theta1*, and *90* for *Theta2*; click on *OK*.

- Glue the areas (**AGLUE** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Areas

- *Pick Menu* appears; click on *Pick All* button.

- Create the mesh. Since the problem involves two dissimilar materials, the inner circle (material 1) will be meshed first. Then the *default material attribute* will be changed to material 2 for the outer circle. Specify global element size (**ESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size

- *Global Element Sizes* dialog box appears; enter **0.1** for **SIZE**; click on **OK**.
- Mesh the inner circle (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- *Pick Menu* appears; pick the inner circle; click on **OK**.
- Change the *default material attribute* to 2 (**MAT** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh Attributes > Default Attrs

- *Meshing Attributes* dialog box appears. Select **2** from the second pull-down menu; click on **OK**.
- Mesh the outer circle (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- *Pick Menu* appears; pick the outer circle; click on **OK**.

Solution

- Apply displacement constraints (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the nodes along $x=0$ (coincident with y -axis); click on **OK** in the *Pick Menu*.
- Highlight **UX**; click on **Apply**.
- *Pick Menu* reappears; pick the nodes along $y=0$ (coincident with x -axis); click on **OK** in the *Pick Menu*.
- Highlight **UY** and remove the highlight on **UX**; click on **OK**.
- Apply surface force (pressure) boundary conditions along the inner circular boundary. Since the boundary is circular, it is convenient to first switch to *Cylindrical Coordinates* and then select the nodes.
 - Switch to *Cylindrical Coordinates* (**CSYS** command) using the following menu path:

Utility Menu > WorkPlane > Change Active CS to > Global Cylindrical

- Select nodes along the circular boundary (**NSEL** command) by using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose **By Location** in the second pull-down menu and type **1** in the **Min, Max** text field; click on **OK**. Because the active coordinate system is cylindrical, any reference to the *x*-coordinate is treated as a reference to the *r*-coordinate by ANSYS.
- Now, apply pressure boundary conditions (**SF** command) by using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Nodes

- *Pick Menu* appears; click on **Pick All**.
- Type **1** for **VALUE Load PRES value**; click on **OK**.
- Select everything (**ALLSEL** command) using the following menu path:

Utility Menu > Select > Everything

- Switch back to *Cartesian Coordinates* (**CSYS** command) using the following menu path:

Utility Menu > WorkPlane > Change Active CS to > Global Cartesian

- Obtain the solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with **Solution is done!**

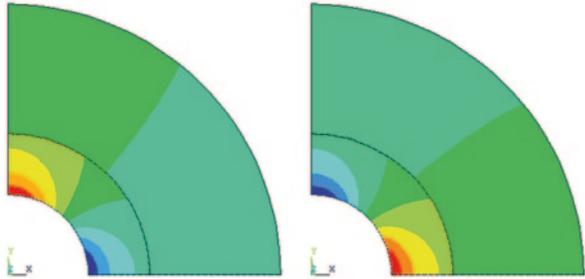
Postprocessing

- Obtain contour plots for σ_{xx} and σ_{yy} (**PLNSOL** command) using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- Click on **Stress** and **X-component of stress**; click on **OK**.
 - The contour plot appears in the *Graphics Window*, as shown in Fig. 8.21 (left).
 - Repeat the same procedure for σ_{yy} , which produces the contour plot given in Fig. 8.21 (right)
- Since the problem possesses a circular geometry, it is often more useful to examine the stresses in cylindrical coordinates. For this purpose, change the results coordinate system to the global cylindrical system (**RSYS** command) using the following menu path:

Fig. 8.21 Normal stresses in Cartesian coordinates: in x -direction, σ_{xx} (left), and in y -direction, σ_{yy} (right)



Main Menu > General Postproc > Options for Outp

- *Options for Output* dialog box appears. Select **Global cylindric** from the first pull-down; click on **OK**.
- Now, obtain contour plots for σ_{rr} and $\sigma_{\theta\theta}$ (**PLNSOL** command) using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- Select **X-component of stress** from the list; click on **OK**.
 - The contour plot appears in the *Graphics Window*, as shown in Fig. 8.22 (left).
 - Repeat the same procedure for $\sigma_{\theta\theta}$, which produces the contour plot given in Fig. 8.22 (right).
- Review the variation of stresses along a path by means of a line plot. Define the path (**PPATH** command) using the following menu path:

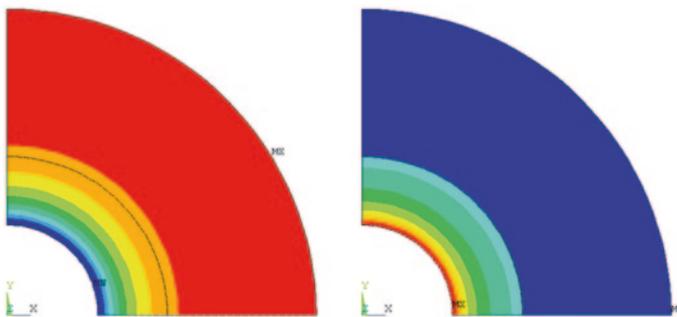


Fig. 8.22 Normal stresses in cylindrical coordinates: in r -direction, σ_{rr} (left), and in θ -direction, $\sigma_{\theta\theta}$ (right)

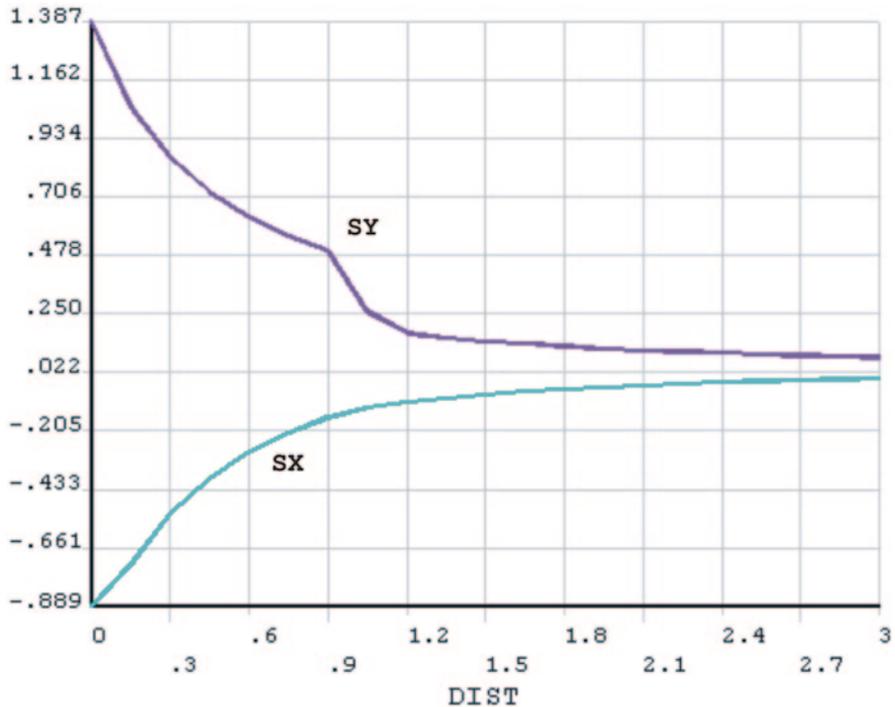


Fig. 8.23 Radial and hoop stresses along $y=0$

Main Menu > General Postproc > Path Operations > Define Path > By Nodes

- *Pick Menu* appears; pick the nodes with (x,y) coordinates $(1, 0)$ and $(4, 0)$; click on **OK**. The path lies along the positive x -axis, starting from the boundary of the hole and ending at the left boundary of the structure.
- *By Nodes* dialog box appears; enter a name describing the path, say *hrz*, in the **Define Path Name** text field; click on **OK**.
- Close the *PATH Command Status Window*.
- Map results onto path (**PDEF** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Map onto Path

- *Map Result Items onto Path* dialog box appears; select **Stress** from the left list and **Y-direction SY** from the right list; click on **OK**.
- Obtain line plot of $\sigma_{\theta\theta}$ along the path (**PLPATH** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Plot Path Item > On Graph

- *Plot of Path Items on Graph* dialog box appears; select **SY**; click on **OK**.
- Figure 8.23 shows the line plots of σ_{rr} and $\sigma_{\theta\theta}$ along the defined path.

8.1.4.3 Axisymmetric

In a solid of revolution, location of a point in the body can conveniently be identified by cylindrical coordinates, r , θ and z , with z being the axis of rotation. When a solid of revolution is subjected to loading that can also be obtained by revolution about the z -axis, then the results become independent of θ . This is called an *Axisymmetric Condition*. Two problems are considered here.

Deformation of a Bar Due to its Own Weight Using 2-D Axisymmetric Elements

Deformation of a bar under its own weight was modeled in Sect. 8.1.1.1 using two-dimensional link elements. The problem was solved using three-dimensional brick elements in Sect. 8.1.3. The solution to this problem also can be obtained using axisymmetric elements as the geometry and the loading (gravity) exhibit conditions for axisymmetry. The following axisymmetric solution utilizes the reference coordinate frame shown in Fig. 8.1.

Model Generation

- Define the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Solid* immediately below *Structural Mass* in the left list and *Quad 4 Node 182* in the right list; click on *OK*.
- Click on *Options*.
- *PLANE182 element type options* dialog box appears; select the *Axisymmetric* item from the pull-down menu corresponding to *Element behavior K3*.
- Click on *OK*; click on *Close*.

- Specify material properties for the bar (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on *Structural* and *Density*, which will bring up another dialog box.
- Enter *0.2839605* for *DENS*; click on *OK*.
- In order to specify the elastic modulus and Poisson's ratio, in the *Define Material Model Behavior* dialog box, in the right window, successively left-click on *Structural*, *Linear*, *Elastic*, and, finally, *Isotropic*, which will bring up another dialog box.
- Enter *30e6* for *EX* and *0.3* for *PRXY*; click on *OK*.

- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create the rectangle defining the axisymmetric cross section (**RECTNG** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

- In the *Create Rectangle by Dimensions* dialog box, enter **0** and **2** for *X1* and *X2* and **0** and **-20** for *Y1* and *Y2*; click on **OK**.
- Specify the global element size (**ESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size

- *Global Element Sizes* dialog box appears; enter **0.2** for *SIZE*; click on **OK**.
- Create the mesh (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Free

- *Pick Menu* appears; click on **Pick All**.

Solution

- Apply displacement constraints (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the nodes at $y=0$; click on **OK** in the *Pick Menu*.
- Highlight **All DOF**; click on **OK**.
- Apply gravitational acceleration (**ACEL** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Inertia > Gravity > Global

- *Apply (Gravitational) Acceleration* dialog box appears.
- Enter **386.2205** for *ACELY*; click on **OK**.
- Obtain the solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review y -displacement contours (**PLNSOL** command) using the following menu path:

Main Menu > General PostProc > Plot Results > Contour Plot > Nodal Solu

- Click on *DOF Solution* and *Y-component of displacement*; click on **OK**.
- The contour plot is shown in Fig. 8.24 as it appears in the *Graphics Window*.

- Review displacement values (**PRNSOL** command) using the following menu path:

Main Menu > General Postproc > List Results > Nodal Solution

- Click on *DOF Solution* and *Y-component of displacement*; click on **OK**.
- The list appears in a separate window. It is a long list of z -displacements. At the bottom of the window, the maximum displacement value is printed as $-0.72469E-03$.

Analysis of a Circular Plate Pushed Down by a Piston Head

An aluminum circular plate with a diameter of 40 in is pushed down by a steel piston head, as shown in Fig. 8.25. The piston head has two sections with diameters 20 and 2 in. The elastic modulus and Poisson's ratio for the aluminum plate are given as $E_{al} = 10 \times 10^6$ psi and $\nu_{al} = 0.35$, respectively, whereas the corresponding properties for steel are $E_{st} = 30 \times 10^6$ psi and $\nu_{st} = 0.3$. The aluminum plate is clamped along the boundary (all degrees of freedom constrained). The goal is to obtain the displacement and stress fields when the piston is pushed down (at the top) by an amount of 0.1 in. This problem possesses the conditions necessary for axisymmetry to be employed. Following is the solution utilizing axisymmetric elements in ANSYS.

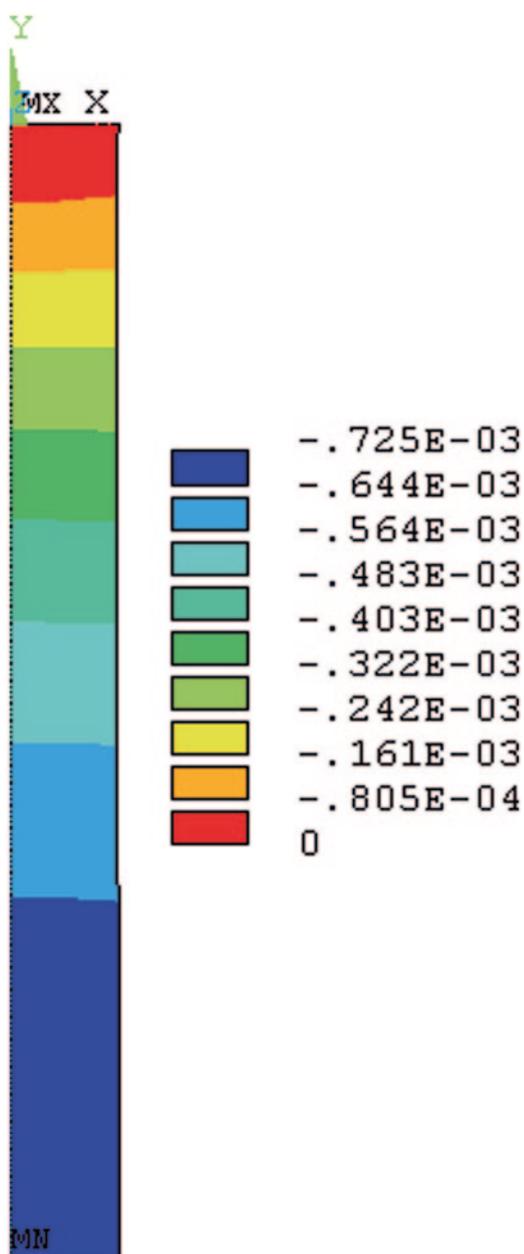
Model Generation

- Define the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Solid* immediately below *Structural Mass* in the left list and *Quad 4 Node 182* in the right list; click on **OK**.
- Click on *Options*.
- *PLANE182 element type options* dialog box appears; select *Axisymmetric* item from the pull-down menu corresponding to *Element behavior K3*.

Fig. 8.24 Contour plot of z-displacement of a bar elongated due to its own weight



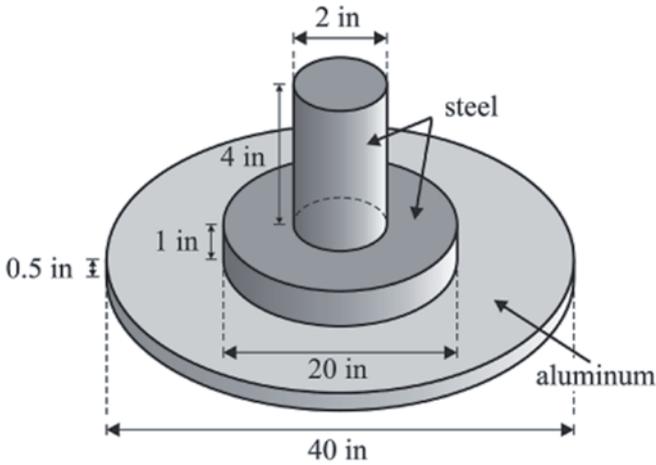


Fig. 8.25 Schematic of a circular plate pushed down by a piston head

- Click on **OK**; click on **Close**.

- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**, which will bring up another dialog box.
- Enter **10e6** for **EX** and **0.35** for **PRXY**; click on **OK**.
- Add new material model using the following menu path:

Material > New Model

- Click on **OK** in the new dialog box.
- In the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**; Enter **30e6** for **EX** and **0.3** for **PRXY**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Three rectangles defining the geometry will be created and overlapped. Create the rectangles defining the axisymmetric cross section (**RECTNG** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

- In the *Create Rectangle by Dimensions* dialog box, enter **0** and **20** for **X1** and **X2** and **0** and **0.5** for **Y1** and **Y2**; click on **Apply**.

- Now, enter **0** and **10** for **X1** and **X2** and **0** and **1.5** for **Y1** and **Y2**; click on **Apply**.
- Finally, enter **0** and **1** for **X1** and **X2** and **0** and **5.5** for **Y1** and **Y2**; click on **OK**.

- Overlap the rectangles (**AOVLAP** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas

- *Pick Menu* appears, click on **Pick All**.
- The overlapping operation produces six areas (started with three), sharing lines along the interfaces.

- Specify the global element size (**ESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size

- *Global Element Sizes* dialog box appears; enter **0.2** for **SIZE**; click on **OK**.

- Create the mesh for the aluminum plate (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- *Pick Menu* appears; pick the bottom row of rectangles (corresponding to the aluminum plate); click on **OK** in the *Pick Menu*.
- Plot the areas (**APLOT** command) using the following menu path:

Utility Menu > Plot > Areas

- Change default element attribute for material number from 1 to 2 (**MAT** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh Attributes > Default Attribs

- *Meshing Attributes* dialog box appears; select **2** from the **[MAT] Material number** pull-down menu; click on **OK**.
- Create mesh for the steel piston (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- *Pick Menu* appears; pick the rectangles corresponding to the steel piston; click on **OK** in the *Pick Menu*.
- Plot elements with different colors based on their material numbers using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears. Select **Material numbers** from the first pull-down menu (corresponding to **Elem/Attrib numbering**) and select **Colors only** from the second pull-down menu (corresponding to

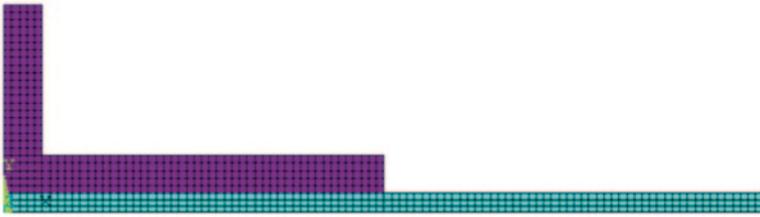


Fig. 8.26 Element plot with different colors based on material numbers

[/NUM] Numbering shown with); click on **OK**. Figure 8.26 shows the corresponding element plot with different colors based on material numbers.

Solution

- Apply displacement constraints along the periphery of the aluminum plate (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the nodes along the right boundary ($x=20$); click on **OK** in the *Pick Menu*.
- Highlight **All DOF**; click on **OK**.

- Apply displacement constraints along the top surface of the steel piston (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the nodes along the top boundary ($y=5.5$); click on **OK** in the *Pick Menu*.
- Remove the highlight on **All DOF** and highlight **UY**.
- Enter **-0.1** in the text box for **VALUE Displacement value**; click on **OK**.

- Obtain the solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status, if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review the deformed shape (**PLDISP** command) using the following menu path:

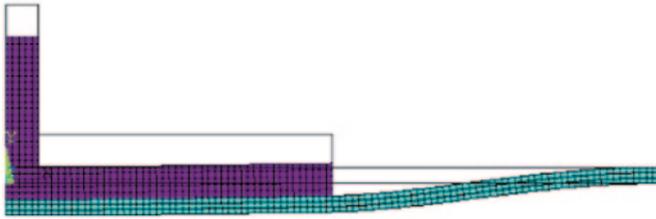


Fig. 8.27 Deformed shape with undeformed edge

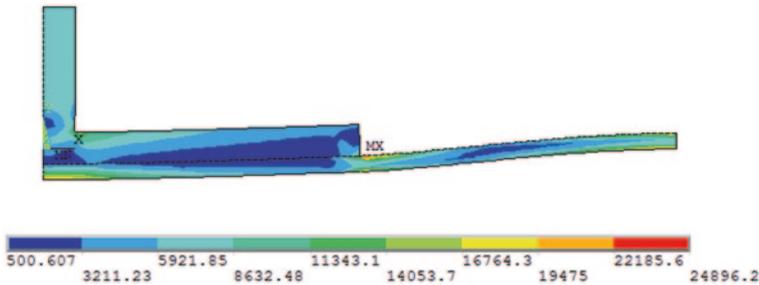


Fig. 8.28 Equivalent stress contours

Main Menu > General PostProc > Plot Results > Deformed Shape

- *Plot Deformed Shape* dialog box appears; select the **Def + undef edge** radio button; click on **OK**.
- Corresponding deformed shape is shown in Fig. 8.27.
- Review the equivalent stress (von Mises) contour plot (**PLNSOL** command) using the following menu path:

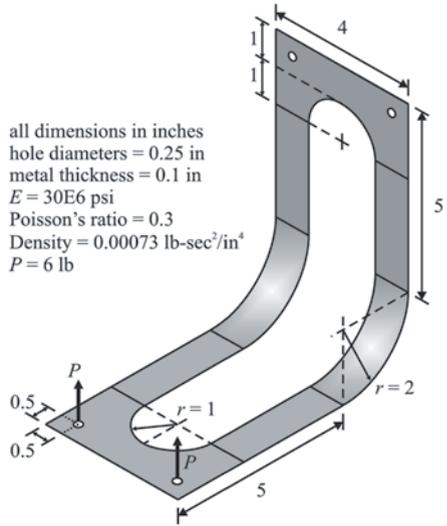
Main Menu > General PostProc > Plot Results > Contour Plot > Nodal Solu

- *Contour Nodal Solution Data* dialog box appears. Click on **Stress** and scroll down to select **von Mises stress**. Click on **OK**.
- Figure 8.28 shows the corresponding contour plot

8.1.5 Plates and Shells

Many engineering structures involve plates and shells where one dimension is much smaller than the other two. When these thin members are flat and only in-plane loads are applied, the problem can be solved using *Plane Stress* idealization. However, if they are curved and/or subjected to both in-plane and out-of-plane loads, it is necessary to solve the problem in 3-D using shell elements. At each node of the shell elements, both displacements and rotations are the degrees of freedom. Three problems are solved utilizing shell elements.

Fig. 8.29 Geometry, material properties, and loading on the bracket



8.1.5.1 Static Analysis of a Bracket

The bracket shown in Fig. 8.29 is clamped at the two top holes and is subjected to static vertical loading at the bottom two holes. Due to the symmetry in geometry, only one quarter of the structure is modeled at first. Once the top-left quarter is modeled and meshed, two symmetric reflection operations are utilized to create the rest of the bracket. The goal is to create the finite element model and obtain the static solution.

Model Generation

- Specify the *jobname* as *bracket* using the following menu path:

Utility Menu > File > Change Jobname

- In the dialog box, type *bracket* in the *[/FILNAM] Enter new jobname* text field; click on the checkbox for *New log and error files* to show *Yes*; click on *OK*.

- Define the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Shell* immediately below *Structural Mass* in the left list and *Elastic 4node 181* in the right list; click on *OK*.
- Click on *Close*.

- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**, which will bring up another dialog box.
- Enter **30e6** for **EX** and **0.3** for **PRXY**; click on **OK**.
- In the *Define Material Model Behavior* dialog box, in the right window, left-click on **Density**, which will bring up another dialog box.
- Enter **0.00073** for **DENS**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Specify the thickness for the shell (**SECTYPE** command) using the following menu path:

Main Menu > Preprocessor > Sections > Shell > Lay-up > Add/Edit

- *Create and Modify Shell Sections* dialog box appears; enter **0.1** for **Thickness**.
- Exit from the *Create and Modify Shell Sections* dialog box by clicking on **OK**.
- Create the solid model.
 - Move *Working Plane origin* using the following menu path:

Utility Menu > WorkPlane > Offset WP by Increments

- *Offset WP* dialog box appears; type **0**, **3**, **-2** in the **X**, **Y**, **Z Offsets** text field; click on **OK**.
- Create a rectangular area using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

- In the *Create Rectangle by Dimensions* dialog box, type **-2** for **X1**, **0** for **X2**, **0** for **Y1**, and **2** for **Y2**; click on **OK**.
- Create a circular area using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions

- In the *Create Circle by Dimensions* dialog box, type **1** for **Outer radius**, **90** for **Theta1**, and **180** for **Theta2**; click on **OK**.
- Subtract the circle from the rectangle using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

- *Pick Menu* appears; pick the rectangle; click on **OK**; pick the circle; click on **OK**.

- Move the *Working Plane* origin to the top-left hole center using the following menu path:

Utility Menu > WorkPlane > Offset WP by Increments

- *Offset WP* dialog box appears; type -1.5 , 1.5 in the *X, Y, Z Offsets* text field (because only *x*- and *y*-increments are entered, no move will be applied in *z*-direction); click on **OK**.
- Create a circular area for the top-left hole using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions

- In the *Create Circle by Dimensions* dialog box, type $0.25/2$ for *Outer radius*, 0 for *Theta1*, and 360 for *Theta2*; click on **OK**.
- Subtract the circle from the rest of the area using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Subtract > Areas

- *Pick Menu* appears; pick the large area and click **OK**; pick the circle; click on **OK**.
- Move the *Working Plane* in order to create the additional rectangular area using the following menu path:

Utility Menu > WorkPlane > Offset WP by Increments

- *Offset WP* dialog box appears; type -0.5 , 0.5 in the *X, Y, Z Offsets* text field; click on **OK**.
- Create additional rectangular area using the following menu path:

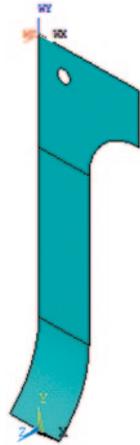
Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

- In the *Create Rectangle by Dimensions* dialog box, type 0 for *X1*, 1 for *X2*, -2 for *Y1*, and -5 for *Y2*; click on **OK**.
- In order to create a curved area, create keypoints that define the axis of rotation using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

- *Create Keypoints in Active Coordinate System* dialog box appears; type 51 for *NPT Keypoint number* and 0 in the *X, Y, Z Location in active CS* text fields; click on **Apply**.
- In the same dialog box, type 52 for *NPT Keypoint number* and -0.5 for *x* and 0 for *y* and *z* in the *X, Y, Z Location in active CS* text fields; click on **OK**.
- Plot areas using the following menu path:

Fig. 8.30 Solid model of a quarter of the bracket



Utility Menu > Plot > Areas

- Create the curved area by sweeping the line at the bottom around an axis defined by the last two keypoints created using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Extrude > Lines > About Axis

- *Pick Menu* appears; the user is first asked to pick the line to be swept, and then to pick the keypoints defining the axis that the line to be swept about.
- Pick the horizontal line at the bottom; click on **OK**; type **51** in the text field in the *Pick Menu* and hit **Enter** on the keyboard; type **52** followed by hitting **Enter** on the keyboard; click on **OK**.
- *Sweep Lines about Axis* dialog box appears; type **45** for *ARC Arc length in degrees*; click on **OK**.
- Click on the *Isometric View* button.
- Figure 8.30 shows the result of this action.
- Although the areas created appear to be connected, ANSYS treats them as independent of each other (not connected). Therefore, the areas must be glued to each other. This is achieved by using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Areas

- *Pick Menu* appears; click on **Pick All**.
- Create the mesh.
 - Specify the number of elements around the hole using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

- Pick the four circular segments defining the hole; click on **OK**.
- *Element Sizes on Lines* dialog box appears; type **2** in the text field corresponding to **NDIV** (the second text field); uncheck the first checkbox; click on **OK**.
- Specify mesh density in the vicinity of the top-left corner using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Keypoints > Picked KPs

- *Pick Menu* appears; pick the top-left keypoint; click on **OK**.
- *Element Size at Picked Keypoints* dialog box appears; type **0.3** for **SIZE Element edge length** text field; click on **OK**.
- Specify global mesh density using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size

- *Global Element Sizes* dialog box appears; type **0.5** for **SIZE Element edge length** text field; click on **OK**.
- Create the mesh using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Free

- In the *Pick Menu*, click on **Pick All**.
- *A Warning Window* appears; click on **OK**.

- Save the model using the following menu path:

Utility Menu > File > Save as Jobname.db

The model will be saved in the *Working Directory* under the name **bracket.db**.

- Create a reflective symmetric mesh using the following menu path:

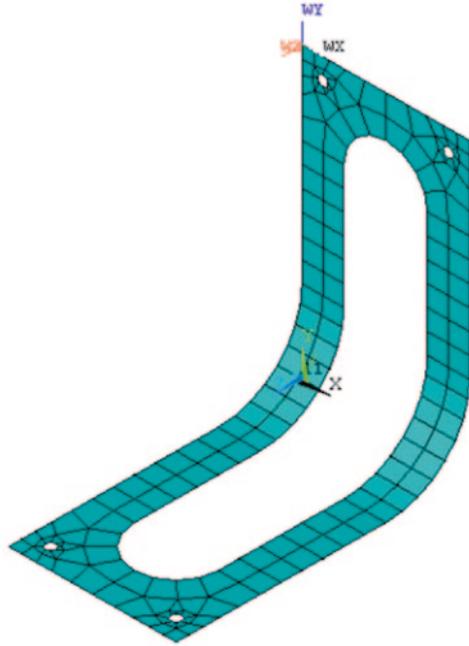
Main Menu > Preprocessor > Modeling > Reflect > Areas

- *Pick Menu* appears; click on **Pick All**.
- *Reflect Areas* dialog box appears; click on the **Y-Z plane X** radio-button; click on **OK**.
- Although it is not apparent through visual inspection, there are duplicate entities (keypoints, lines and nodes) along the symmetry line, thus there is no continuity. Therefore, merge duplicate entities using the following menu path:

Main Menu > Preprocessor > Numbering Cntrls > Merge Items

- In the dialog box, select **All** from the first pull-down menu; click on **OK**.
- Create a second reflective symmetric mesh.
 - For this purpose, create a local coordinate system using the following menu path:

Fig. 8.31 Bracket after meshing and two reflection operations



Utility Menu > WorkPlane > Local Coordinate Systems > Create Local CS > At Specified Loc

- *Pick Menu* appears; type **0, 0, 0** in the text field in the *Pick Menu*; click on **OK**.
- A dialog box appears; type **-45** in the **THYZ Rotation about local X** text field; click on **OK**.
- Create a reflective symmetric mesh using the following menu path:

Main Menu > Preprocessor > Modeling > Reflect > Areas

- *Pick Menu* appears; click on **Pick All**.
- *Reflect Areas* dialog box appears; click on the **X-Z plane Y** radio-button; click on **OK**.
- Plot elements using the following menu path:

Utility Menu > Plot > Elements

- Figure 8.31 shows the **isometric** view of the mesh after the reflection.
- Merge duplicate entities using the following menu path:

Main Menu > Preprocessor > NumberingCtrls > Merge Items

- In the dialog box, select **All** from the first pull-down menu; click on **OK**.
- Define *components* for future use.

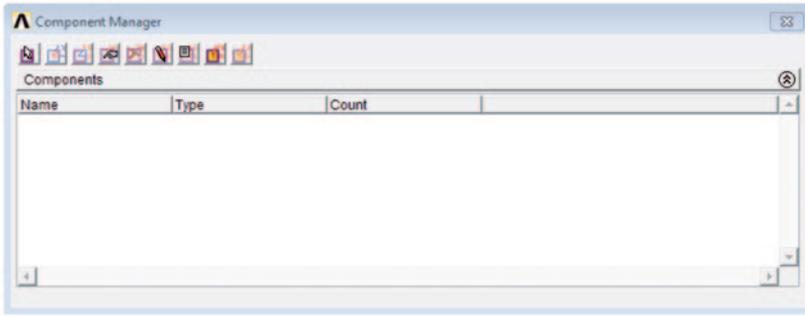


Fig. 8.32 *Component Manager* dialog box (*left-most button* is used for creating components)

- For this purpose, create a local coordinate system at the center of the top-left hole using the following menu path:

Utility Menu > WorkPlane > Local Coordinate Systems > Create Local CS > At Specified Loc

- *Pick Menu* appears; type $-1.5, 4.5, -2$ in the text field in the *Pick Menu*; click on **OK**.
- A dialog box appears; select *Cylindrical 1* in the *KCS Type of coordinate system* pull-down menu.
- **Delete -45** in the *THYZ Rotation about local X* text field; click on **OK**.
- Select nodes along the top-left hole by using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose **By Location** in the second pull-down menu and type $0.25/2$ in the *Min, Max* text field; click on **OK**. Because the active coordinate system is cylindrical, any reference to the *x*-coordinate will be treated as a reference to the *r*-coordinate by ANSYS.
- Create the component by using the following menu path:

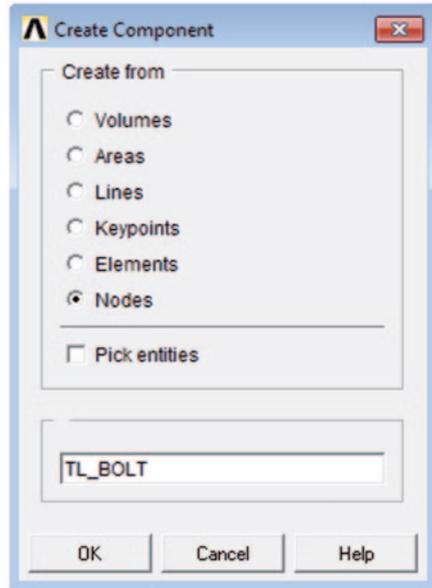
Utility Menu > Select > Component Manager

- *Component Manager* dialog box appears (Fig. 8.32); click on the first button on the left (**Create Component** button).
- *Create Component* dialog box appears; click on the *Nodes* radio-button and name the component by typing *TL_BOLT* (stands for top-left bolt) in the text field (Fig. 8.33); click on **OK**.
- Close the *Component Manager*.
- Create components for top-right, bottom-left, and bottom-right bolts in the same manner. The origin of the local *cylindrical* coordinates for each of these are given as

TR_BOLT: $1.5, 4.5, -2$

BL_BOLT: $-1.5, -2, 4.5$ and use -90 for the *THYZ Rotation about local X*

Fig. 8.33 Dialog box for creating components



BR_BOLT: 1.5, -2, 4.5 and use -90 for the *THYZ Rotation about local X*

- Save the model using the following menu path:

Utility Menu > File > Save as Jobname.db

Solution

- Constrain displacement and rotation degrees of freedom along the top-left and -right holes. For this purpose, first select the components created earlier for these holes (*TL_BOLT* and *TR_BOLT*) using the following menu path:

Utility Menu > Select > Comp/Assembly > Select Comp/Assembly

- A dialog box appears; click on the *by component name* radio-button; click on **OK**.
- A new dialog box with the components listed appears; highlight *TL_BOLT*; click on **OK**. This action selects the nodes along the top-left hole.
- Specify the displacement boundary conditions using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; click on **Pick All**.
- In the new dialog box, highlight *All DOF*; click on **OK**.

- Repeat the same procedure for the top-right hole (*TR_BOLT*).
- Apply force boundary conditions.
 - For this purpose, create a local coordinate system at the center of the bottom-left hole using the following menu path:

Utility Menu > WorkPlane > Local Coordinate Systems > Create Local CS > At Specified Loc

- *Pick Menu* appears; type *-1.5, -2, 4.5* in the text field in the *Pick Menu*; click on *OK*.
- A dialog box appears; select *Cylindrical 1* in the *KCS Type of coordinate system* pull-down menu and type *-90* in the *THYZ Rotation about local X* text field; click on *OK*.
- Select the keypoints along the bottom-left hole using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose *Keypoints* in first pull-down menu and *By Location* in the second pull-down menu; type *0.25/2* in the *Min, Max* text field; click on *OK*.
- Apply forces on the keypoints using the menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

- *Pick Menu* appears; click on *Pick All*.
- In the new dialog box, select *FY* from pull-down menu and enter *6/4* for *Force/moment value*; click on *OK*.
- A *Warning Window* appears, informing the user that boundary conditions applied to solid modeling entities overwrite those that may have already been applied to finite element entities (nodes and elements) directly.
- Close the *Warning Window*.
- Repeat the same procedure for the bottom-right hole (use *1.5, -2, 4.5* for the local coordinate system origin and *-90* for the *THYZ Rotation about local X*).
- Select everything using the following menu path:

Utility Menu > Select > Everything

- Save the model using the following menu path:

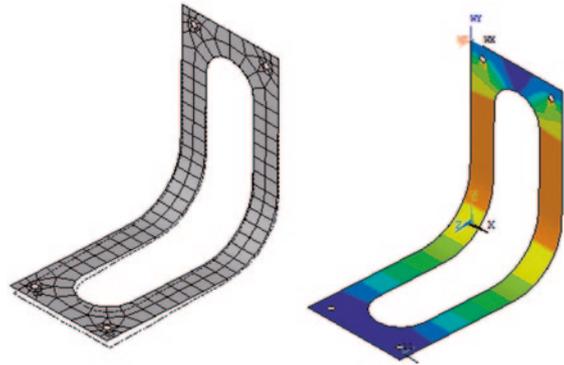
Utility Menu > File > Save as Jobname.db

- Obtain the solution using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.

Fig. 8.34 Deformed shape (*left*), and contour plot of equivalent (von Mises) stresses (*right*)



- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- If a dialog box with a message reminding the user about previous warnings issued appears, click on **Yes**.
- Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Review the deformed shape using the following menu path:

Main Menu > General PostProc > Plot Results > Deformed Shape

- Select **Def + undef edge**; click on **OK**.
- The deformed shape is shown in Fig. 8.34 (left) as it appears in the *Graphics Window*.

- Review the stress contours using the following menu path:

Main Menu > General PostProc > Plot Results > Contour Plot > Nodal Solu

- Click on **Stress**. Scroll down and click on **von Mises stress**; click on **OK**.

The equivalent stress contour plot is shown in Fig. 8.34 (right) as it appears in the *Graphics Window*.

8.1.5.2 Analysis of a Circular Plate Pushed Down by a Piston Head

The circular aluminum plate pushed down by a steel piston was analyzed in Sect. 8.1.4.3.2 by employing only axisymmetric elements. The geometry of the problem is shown in Fig. 8.25. The same problem is solved in this section by using a combination of shell and 3-D solid elements.

Model Generation

- Define the element types (**ET** command) using the following menu path:

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

- *Element Types* dialog box appears; click on **Add**.
- Select **Shell** immediately below **Structural Mass** in the left list and scroll down to select **8node 281** in the right list; click on **Apply**.
- Select **Solid** immediately below **Structural Mass** in the left list and scroll down to select **20node 186** in the right list; click on **OK**.
- Define *keyoptions* on the shell element by highlighting **Type 1 SHELL281** in the *Element Types* dialog box and clicking on **Options**.
- *SHELL281 element type options* dialog box appears; select **All layers** from the **Storage of layer data K8** pull-down menu.
- Click on **Close** to exit from the *Element Types* dialog box.

- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**, which brings up another dialog box.
- Enter **10e6** for **EX** and **0.35** for **PRXY**; click on **OK**.
- Add new material model using the following menu path:

Material > New Model

- Click on **OK**.
- In the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**; Enter **30e6** for **EX** and **0.3** for **PRXY**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Specify the thickness for the shell (**SECTYPE** command) using the following menu path:

Main Menu > Preprocessor > Sections > Shell > Lay-up > Add/Edit

- *Create and Modify Shell Sections* dialog box appears; enter **0.5** for **Thickness**.
- Select **User-Input-Location** from the **Section Offset** pull-down menu. Enter **0.25** in the text box for **User Defined Value**.
- Exit from the *Create and Modify Shell Sections* dialog box by clicking on **OK**.

- Create quarter circular areas (**PCIRC** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Circle > By Dimensions

- In the *Create Circle by Dimensions* dialog box, type **1** for *Outer radius*, **0** for *Theta1*, and **90** for *Theta2*; click on *Apply*.
- Modify *Outer radius* to be **10**; click on *Apply*.
- Finally, modify *Outer radius* (one more time) to be **20**; click on *OK*.
- Overlap the areas (**AOVLAP** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Areas

- *Pick Menu* appears; click on *Pick All*.
- Create quarter cylindrical volumes (**CYLIND** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Volumes > Cylinder > By Dimensions

- In the *Create Cylinder by Dimensions* dialog box, type **10** for *Outer radius*, **0** for *Z1*, **1** for *Z2*, **0** for *Theta1*, and **90** for *Theta2*; click on *Apply*.
- Modify *Outer radius* to be **1** and *Z2* to be **5**; click on *OK*.
- Overlap the volumes (**VOVLAP** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Overlap > Volumes

- *Pick Menu* appears; click on *Pick All*.
- Specify size controls for meshing. First, the number of divisions on specific lines will be specified, followed by specification of the global element size.
 - Select lines at $0.25 \leq x \leq 0.75$ and $0.25 \leq y \leq 0.75$ (**LSEL** command) using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose *Lines* from the first pull-down menu and *By Location* in the second pull-down menu. Type **0.25,0.75** in the *Min, Max* text field; click on *Apply*.
- Now, select *Y-coordinates* and the *Also Select* radio-buttons without changing the text (**0.25, 0.75**) in the *Min, Max* text field; click on *OK*.
- A total of 12 lines are selected. Specify the number of element divisions along the selected lines (**LESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

- *Pick Menu* appears; click on *Pick All*.

- *Element Sizes on Picked Lines* dialog box appears; enter **4** for **NDIV**. Remove the checkmark next to **KYNDIV SIZE, NDIV can be changed** so that it shows **No**; click on **OK**.
- Specify global element size (**ESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Global > Size

- *Global Element Sizes* dialog box appears; enter **0.5** for **SIZE**; click on **OK**.
- Select everything (**ALLSEL** command) using the following menu path:

Utility Menu > Select > Everything

- Generate the mesh in two stages: (i) the mesh associated with the aluminum plate is created using **SHELL281** elements followed by (ii) the generation of the mesh for the steel piston head using **SOLID186** elements.
- Select areas attached to the volumes (**ASLV** command) using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose **Areas** from the first pull-down menu and **Attached to** in the second pull-down menu. Select **Volumes** and **From Full** radio-buttons; click on **Apply**.
- Now, click on **Invert** button. This inverts the selection, i.e., selected areas are unselected and vice versa. At this point, areas associated with the aluminum plate are selected.
- Create the mesh for the aluminum plate (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- *Pick Menu* appears; click on **Pick All**.
- Select everything (**ALLSEL** command) using the following menu path:

Utility Menu > Select > Everything

- Change the *default element type attribute* to **2** (**TYPE** command) and *default material attribute* to **2** (**MAT** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh Attributes > Default Attribs

- *Meshing Attributes* dialog box appears. Select **2 SOLID186** from the first pull-down menu and select **2** from the second pull-down menu; click on **OK**.
- Create the mesh for the steel piston head (**VMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Volumes > Mapped > 4 to 6 sided

- *Pick Menu* appears; click on **Pick All**.
- Obtain an isometric view of the mesh using the following menu path:

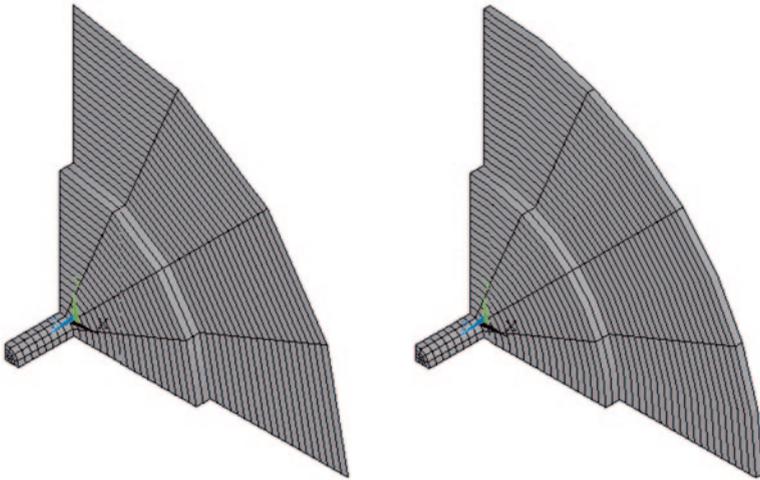


Fig. 8.35 Elements plotted without (*left*) and with (*right*) thickness information from real constants and curved surfaces

Utility Menu > PlotCtrls > Pan Zoom Rotate

- *Pan Zoom Rotate* window appears; click on **ISO** button.
- Plot elements with different colors based on their material numbers using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears. Select **Material numbers** from the first pull-down menu (corresponding to **Elem/Attrib numbering**) and select **Colors only** from the second pull-down menu (corresponding to **[/NUM] Numbering shown with**); click on **OK**. Figure 8.35 (left) shows the corresponding element plot with different colors based on material numbers.
- Note in the element plot that the aluminum plate elements do not have a thickness. This is because the thickness information is stored in *real constants* and the **SHELL281** elements are plane elements. However, for visualization purposes, it is possible to plot plane elements with their thickness (**/ESHAPE** command) using the following menu path:

Utility Menu > PlotCtrls > Style > Size and Shape

- *Size and Shape* dialog box appears. Place a checkmark next to **[/ESHAPE] Display of element shapes based on real constant descriptions** so that it shows **On**. In the same dialog box, select **2 facets/edge for [/EFACET] Facets/element edge** so that the elements with curved edges and surfaces are shown correctly (**/EFACET** command). Click on **OK**.
- Figure 8.35 (right) shows the corresponding element plot with elements having curved edges/surfaces and thickness.

- Finally, merge duplicate entities (**NUMMRG** command) using the following menu path:

Main Menu > Preprocessor > Numbering Ctrl's > Merge Items

- In the dialog box, select *All* from the first pull-down menu; click on **OK**.

Solution

- Apply degree of freedom (DOF) constraints along the outer boundary of the aluminum plate. Since the boundary is circular, it is convenient to first switch to *Cylindrical Coordinates* and then select the nodes.
 - Switch to *Cylindrical Coordinates* (**CSYS** command) using the following menu path:

Utility Menu > WorkPlane > Change Active CS to > Global Cylindrical

- Select nodes along the circular boundary (**NSEL** command) by using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose *Nodes* in the first pull-down menu and *By Location* in the second pull-down menu. Click on *X coordinate* and *From Full* radio-buttons; type *20* in the *Min, Max* text field; click on **OK**. Because the active coordinate system is cylindrical, any reference to the *x*-coordinate is treated as a reference to the *r*-coordinate by ANSYS.
- Now, apply DOF constraints (**D** command) by using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; click on **Pick All**.
- In the new dialog box, highlight *All DOF*; click on **OK**.
- Select everything (**ALLSEL** command) using the following menu path:

Utility Menu > Select > Everything

- Switch back to *Cartesian Coordinates* (**CSYS** command) using the following menu path:

Utility Menu > WorkPlane > Change Active CS to > Global Cartesian

- Apply degree of freedom (DOF) constraints along the top surface of the steel piston.
 - Select nodes (**NSEL** command) by using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose **By Location** in the second pull-down menu; click on the **Z coordinate** and **From Full** radio-buttons; type **5** in the **Min, Max** text field; click on **OK**.
- Apply DOF constraints (**D** command) by using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; click on **Pick All**.
- In the new dialog box, remove the highlight on **All DOF** and highlight **UZ**. Enter **-0.1** in the text box for **VALUE Displacement value**; click on **OK**.
- Apply symmetry conditions along the $x=0$ and $y=0$ planes for the entire structure.
 - Select nodes (**NSEL** command) by using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose **Nodes** in the first pull-down menu and **By Location** in the second pull-down menu. Click on the **X coordinate** and **From Full** radio-buttons; type **0** in the **Min, Max** text field; click on **OK**.
- Apply DOF constraints (**D** command) by using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; click on **Pick All**.
- In the new dialog box, remove the highlight on **UZ** and highlight **UX**. Enter **0** in the text box for **VALUE Displacement value**; click on **OK**.
- Select nodes (**NSEL** command) by using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose **Nodes** in the first pull-down menu and **By Location** in the second pull-down menu. Click on the **Y coordinate** and **From Full** radio-buttons; type **0** in the **Min, Max** text field; click on **OK**.
- Apply DOF constraints (**D** command) by using the following menu path:

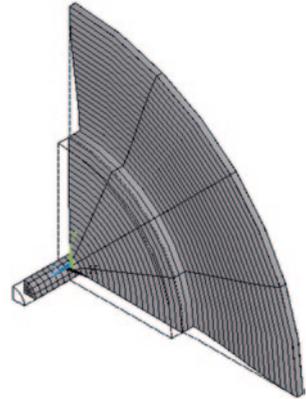
Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; click on **Pick All**.
- In the new dialog box, remove the highlight on **UX** and highlight **UY**; click on **OK**.
- Select everything (**ALLSEL** command) using the following menu path:

Utility Menu > Select > Everything

- Obtain the solution using the following menu path:

Fig. 8.36 Isometric view of the deformed shape



Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Review the deformed shape using the following menu path:

Main Menu > General PostProc > Plot Results > Deformed Shape

- Select ***Def + undef edge***; click on **OK**.
- The isometric view of the deformed shape is shown in Fig. 8.36 as it appears in the *Graphics Window*.
- Review the equivalent stress (von Mises) contour plot (**PLNSOL** command) using the following menu path:

Main Menu > General PostProc > Plot Results > Contour Plot > Nodal Solu

- *Contour Nodal Solution Data* dialog box appears. Click on ***Stress***. Scroll down and select ***von Mises stress***; click on **OK**.
- Figure 8.37 shows the corresponding contour plot.

8.1.5.3 Analysis of an Axisymmetric Shell with Internal Pressure

Consider the pressure vessel shown in Fig. 8.38 with elastic properties $E = 10 \times 10^6$ psi and $\nu = 0.3$. Its radius changes, as shown in Fig. 8.38, while the

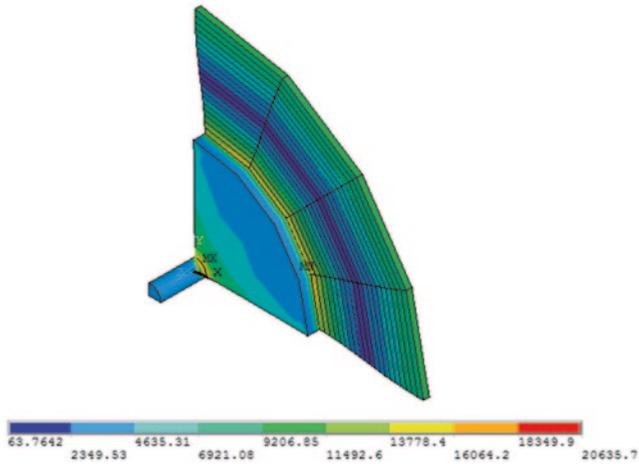
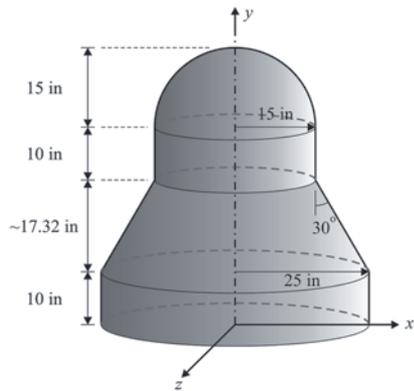


Fig. 8.37 Equivalent stress contours

Fig. 8.38 Schematic of the axisymmetric shell with internal pressure



thickness remains constant, $t = 0.25$ in. The internal pressure is 300 psi. The goal is to find the meridional and hoop stresses in the shell. Examination of the geometry and loading reveals that the problem is axisymmetric. Therefore, axisymmetric shell elements within ANSYS are utilized in this section.

Model Generation

- Define the element type (**ET** command) using the following menu path:

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

- Click on *Add*.

Table 8.1 Keypoint numbers and coordinates for the axisymmetric shell

Keypoint number	x	y
1	25	0
2	25	10
3	15	27.32
4	15	37.32
5	0	52.32
6	0	37.32

- Select *Shell* immediately below *Structural Mass* in the left list and *Axisym 2node 208* in the right list; click on *OK*.
- Click on *Close*.

- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on *Structural*, *Linear*, *Elastic*, and, finally, *Isotropic*, which will bring up another dialog box.
- Enter *10e6* for *EX* and *0.3* for *PRXY*; click on *OK*.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Specify the thickness for the shell (**SECTYPE** command) using the following menu path:

Main Menu > Preprocessor > Sections > Shell > Lay-up > Add/Edit

- *Create and Modify Shell Sections* dialog box appears; enter *0.25* for *Thickness*.
- Exit from the *Create and Modify Shell Sections* dialog box by clicking on *OK*.

- Create keypoints (**K** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

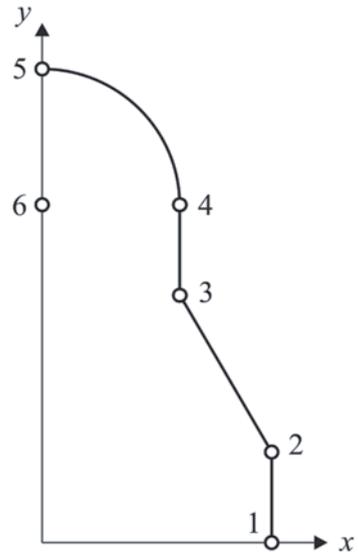
- In the *Create Keypoints in Active Coordinate System* dialog box, type, *25* for *X* and *0* for *Y*; click on *Apply* (keypoint 1 is created).
- Referring to the schematic of keypoints shown in Fig. 8.39 and tabulated in Table 8.1, repeat this procedure for keypoints 2 through 6.

- Create straight lines (**L** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line

- *Pick Menu* appears; pick keypoints 1 and 2; line 1 is created.

Fig. 8.39 Schematic of the keypoints used in the solid model



- Repeat this for lines 2 and 3 using keypoint pairs 2–3 and 3–4, respectively; click on **OK** in the *Pick Menu*.

- Create an arc (**LARC** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Lines > Arcs > By End KPs & Rad

- *Pick Menu* appears; pick keypoints 4 and 5 (end points of the arc); click on **OK** in the *Pick Menu*.
- Pick keypoint 6 (center of the arc); click on **OK** in the *Pick Menu*.
- *Arc by End KPs & Radius* dialog box appears; enter **15** for **RAD Radius of the arc**.
- Click on **OK**; line 4 is created.

- Specify the number of divisions on the lines (**LESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > All Lines

- *Element Sizes on Picked Lines* dialog box appears; enter **20** for **NDIV**; click on **OK**.

- Create the mesh (**LMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Lines

- *Pick Menu* appears; click on **Pick All**.

Solution

- Apply degree of freedom (DOF) constraints at end points (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the bottom node, i.e., $(x, y) = (25, 0)$; click on **OK**.
- In the dialog box, select **UY**; click on **Apply**.
- *Pick Menu* reappears; pick the top node, i.e., $(x, y) = (0, 52.32)$; click on **OK**.
- In the dialog box, remove the highlight on **UY** and highlight **UX**; click on **OK**.

- Apply pressure (**SFE** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Elements

- *Pick Menu* appears; click on **Pick All**.
- In the new dialog box, enter **300** for **Value Load PRES value**; click on **OK**.

- Obtain the solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Review the deformed shape (**PLDISP** command) using the following menu path:

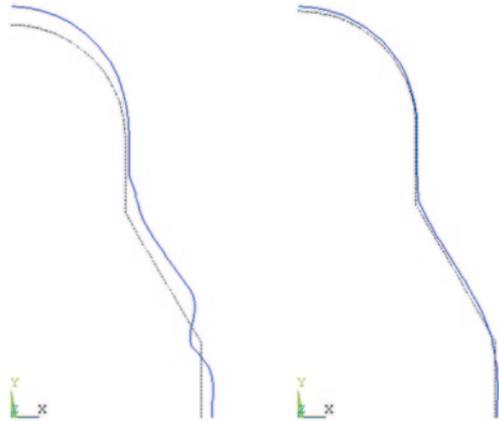
Main Menu > General PostProc > Plot Results > Deformed Shape

- Select **Def + undef edge**; click on **OK**.
- The deformed shape is shown in Fig. 8.40 (left) as it appears in the *Graphics Window*. It is clear from the figure that the bottom end of the conical section exhibits unexpected displacements/rotations. Problems with real applications (using realistic material properties, geometry, and loads) seldom produce displacements that can be visually detected. Therefore, ANSYS scales the displacements when displaying the deformed shape.
- Change the displacement scaling (**/DSCALE** command) using the following menu path:

Utility Menu > PlotCtrls > Style > Displacement Scaling

- This brings up the *Displacement Display Scaling* dialog box. Note the number **25.253596413609** in the **User specified factor** field. This means that the

Fig. 8.40 Deformed shape with automatic scaling (*left*; amplified ~ 23 times) and manual scaling (*right*; amplified 5 times)



displacements are amplified by a factor of approximately 22, so they can be clearly viewed. In order to change this setting, click on the *User specified* radio-button; replace the existing scaling factor with the desired value. Figure 8.40 (right) shows the deformed shape amplified by a factor of 5.

- Store element stresses in the element table (**ETABLE** command) using the following menu path:

Main Menu > General PostProc > Element Table > Define Table

- *Element Table Data* dialog box appears; click on *Add*.
- *Define Additional Element Table Items* dialog box appears; assign a user-defined label for the plate mid-plane meridional stresses, say **STMR**, in the *Lab User label for item* field.
- In the left list, scroll down and select *By sequence num*; in the right list, select **SMISC**.
- Finally, in the last field, type **SMISC,18**; click on *Apply*. There are several quantities that are stored in sequences, i.e., **EPEL**, **NL**, or **SMISC**. The information as to which quantity is stored under which sequence is given in *element help pages*. In this particular example, the help page for **SHELL208** contains tables explaining which quantities are stored under which sequence.
- Similarly, store plate mid-plane hoop stresses by assigning a user label, say **STHP**, and entering **SMISC,19**; click on *OK*.
- Click on *Close* in the *Element Table Data* dialog box.

- View element table quantities (**PRETAB** command) using the following menu path:

Main Menu > General PostProc > Element Table > List Elem Table

- *List Element Table* dialog box appears; select **STMR** and **STHP**; click on *OK*.
- Element numbers and requested element table items are displayed in columns in a separate window. Figures 8.41 and 8.42 show the meridional and hoop stresses, respectively (plotted outside ANSYS).

Fig. 8.41 Meridional stresses in the axisymmetric shell

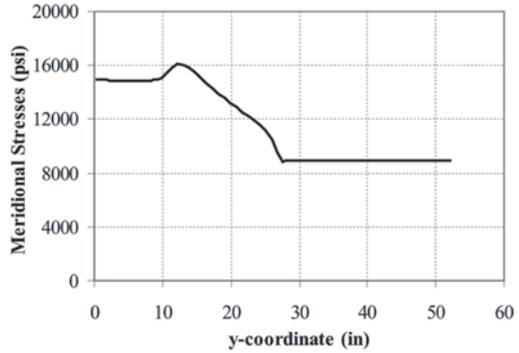
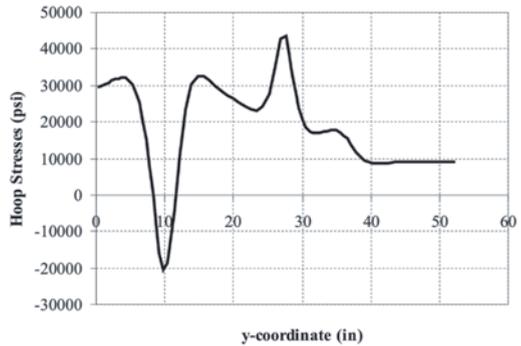


Fig. 8.42 Hoop stresses in the axisymmetric shell



8.1.5.4 Analysis of a Layered Composite Plate

A 10 in \times 10 in square composite plate with a stacking sequence of $[45^\circ/0^\circ/-45^\circ/90^\circ]$ is subjected to tensile loading of 100 MPa in the y -direction, as shown in Fig. 8.43. Unidirectional ply properties are $E_L = 161$ GPa, $E_T = 9$ GPa, $\nu_{LT} = 0.26$, and $G_{LT} = 6.1$ GPa. The subscripts L and T designate longitudinal (fiber direction) and transverse (perpendicular to fiber direction), respectively. Each ply has a thickness of 0.16 mm. The goal is to find the displacement and stress fields in the plate.

Model Generation

- Define the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Shell* immediately below *Structural Mass* in the left list and *Elastic 4node 181* in the right list; click on *OK*.

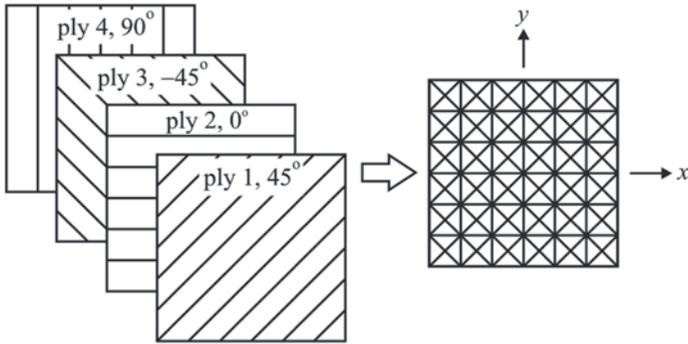


Fig. 8.43 Schematic of the layered composite plate

- Click on *Close*.

- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Orthotropic**, which will bring up another dialog box.
- Enter **161e9** for **EX**, **9e9** for **EY** and **EZ**, **0.26** for **PRXY** and **PRXZ**, **0.01** for **PRYZ**, **6.1e9** for **GXY** and **GXZ**, and **1e9** for **GYZ**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Specify layer information using sections (**SECTYPE** command) using the following menu path:

Main Menu > Preprocessor > Sections > Shell > Lay-up > Add/Edit

- Click on **Add layer** button until there are four rows to accommodate four layers.
- Dialog box for layer information appears; material number (**Material ID**), orientation angle (**Orientation**), and thickness (**Thickness**) information for each layer (ply) are entered in this dialog box. Enter the related quantities, as shown in Fig. 8.44.
- Exit from the *Create and Modify Shell Sections* dialog box by clicking on **OK**.

- Create keypoints (**K** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

- In the *Create Keypoints in Active Coordinate System* dialog box, type, **-5** for **X** and **-5** for **Y**; click on **Apply** (keypoint 1 is created).
- Repeat this procedure for keypoints 2 through 4 using the following data:

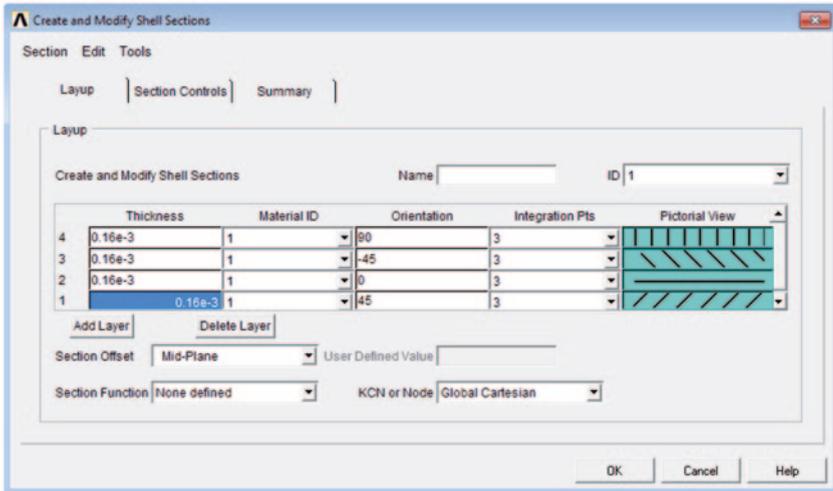


Fig. 8.44 Dialog box for entering layer information

$$\begin{aligned} x_2 &= 5 & y_2 &= -5 \\ x_3 &= 5 & y_3 &= 5 \\ x_4 &= -5 & y_4 &= 5 \end{aligned}$$

– After creating keypoint 4, click on **OK** instead of *Apply*.

- Create the area (**A** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > Through KPs

– *Pick Menu* appears; pick keypoints 1, 2, 3, and 4 in this order; click on **OK** in the *Pick Menu*.

- Specify the number of divisions on all lines (**LESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrl > ManualSize > Lines > All Lines

– *Element Sizes on All Selected Lines* dialog box appears; enter **40** for **NDIV**; click on **OK**.

- Create the mesh (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

– *Pick Menu* appears; click on **Pick All**.

Solution

- Apply degree of freedom (DOF) constraints at the center node and right mid-node (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the center node, i.e., $(x, y) = (0, 0)$; click on **OK**.
 - In the dialog box, select **UX** and **UY**; click on **Apply**.
 - *Pick Menu* reappears; pick the right mid-node, i.e., $(x, y) = (5, 0)$; click on **OK**.
 - In the dialog box, remove the highlight on **UX** (leaving only **UY** highlighted); click on **OK**.
- Constrain z degrees of freedom (DOF) in all nodes (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; click on **Pick All**.
 - In the dialog box, remove the highlight on **UY** and highlight **UZ**; click on **OK**.
- Plot nodes for clarity (**NPLOT** command) using the following menu path:

Utility Menu > Plot > Nodes

- Apply pressure (**SFE** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Nodes

- *Pick Menu* appears; using the **Box** radio-button in the *Pick Menu*, pick the nodes along $y = 5$ and $y = -5$; click on **OK**.
 - In the new dialog box, enter $-100E6 * 4 * .16E-3$ for **Value Load PRES value**; click on **OK**.
- Obtain the solution (**SOLVE** command) using the following menu path:

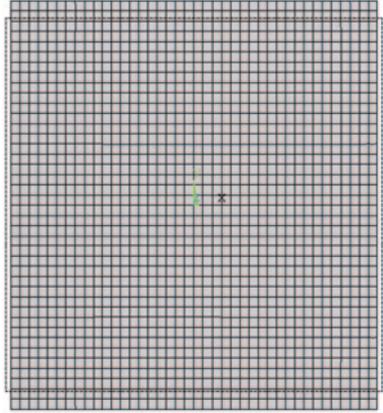
Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review the deformed shape (**PLDISP** command) using the following menu path:

Fig. 8.45 Deformed shape of the composite plate under uniaxial tension in the y -direction



Main Menu > General PostProc > Plot Results > Deformed Shape

- Select *Def + undef edge*; click on *OK*.
- The deformed shape is shown in Fig. 8.45 as it appears in the *Graphics Window*.
- Obtain a contour plot of the y -displacement (u_y) (**PLNSOL** command) using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- Select *DOF Solution* and *Y-component of displacement*; click on *OK*.
- The contour plot appears in the *Graphics Window*, as shown in Fig. 8.46.

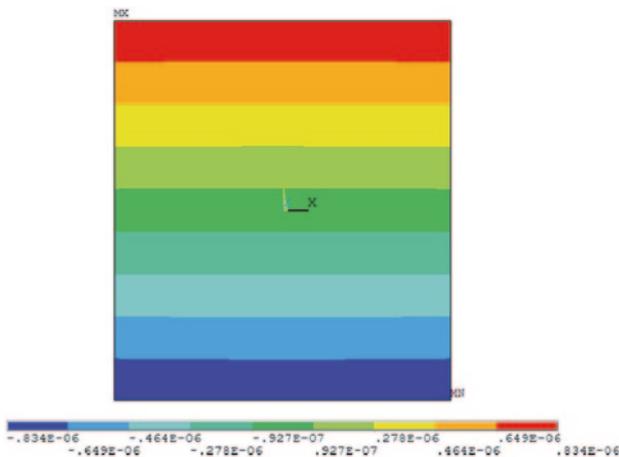


Fig. 8.46 Contour plot of the y -displacement of the composite plate

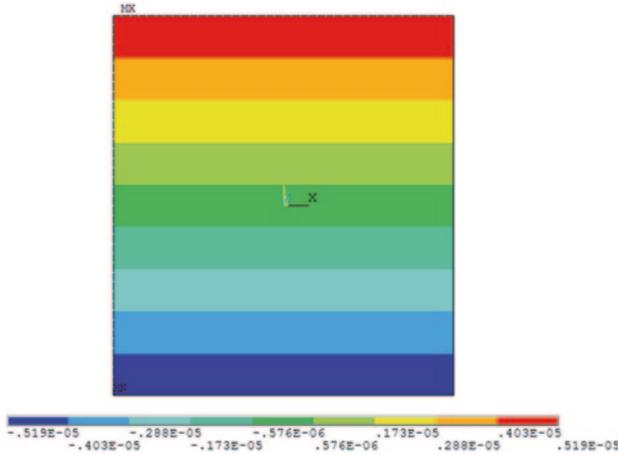


Fig. 8.47 Contour plot of the y -displacement of the composite plate utilizing two-dimensional plane stress idealization with orthotropic properties

The same problem can be solved following the procedure given in Sect. 8.1.4.1.2 using the following equivalent orthotropic material properties for the laminate:

$$\begin{aligned}
 E_x = E_y &= 61.7 \text{ GPa} \\
 \nu_{xy} &= 0.3 \\
 G_{xy} &= 23.8 \text{ GPa}
 \end{aligned}$$

The contour plot for the y -displacement (u_y) obtained by using this two-dimensional approximation is shown in Fig. 8.47.

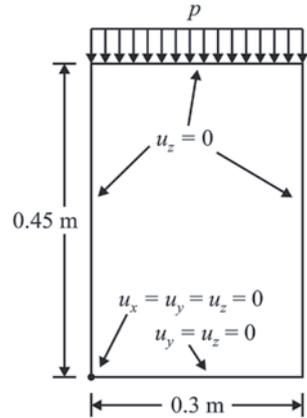
8.2 Linear Buckling Analysis

If the component is expected to exhibit structural instability, the search for the load that causes structural bifurcation is referred to as a “buckling load” analysis. Because the buckling load is not known a priori, the finite element equilibrium equations for this type of analysis involve the solution of homogeneous algebraic equations whose lowest eigenvalue corresponds to the buckling load, and the eigenvector represents the primary buckling mode.

There are two approaches in the ANSYS program for buckling analysis: (i) eigenvalue buckling (linear), and (ii) non-linear buckling. The first is considered here.

Eigenvalue buckling is used for calculating the theoretical buckling load of a linear elastic structure. Since it assumes the structure exhibits linearly elastic behavior, the predicted buckling loads are overestimated (unconservative). Steps involved in a typical *Eigenvalue Buckling* analysis are:

Fig. 8.48 Schematic of the rectangular plate



- Build the model.
- Obtain the static solution.
- Obtain the *eigenvalue buckling* solution.
- Expand the solution.
- Review the results.

A static solution is needed to establish the stiffening of the structure under the applied load (*stress stiffening*). There are several buckling modes (theoretically, infinitely many!) in a structure. The first buckling mode is the one requiring the smallest load. The user specifies the number of buckling modes to be extracted. An *eigenvalue buckling* solution simply calculates the buckling loads for each of these modes. The solution is then *expanded* to include the deformation patterns in the structure (mode shapes) corresponding to the buckling loads. Results are reviewed in the *General Postprocessor*.

As an example, a rectangular plate is subjected to uniform compressive loading along its top edge while the bottom edge is constrained to move in the direction of loading, as shown in Fig. 8.48. The plate is 0.45 m long, 0.3 m wide, and 0.003 m thick. It is made of steel with elastic modulus $E=200$ GPa and Poisson's ratio $\nu=0.32$. The goal is to find the first four buckling modes and their corresponding buckling loads under given constraints and loading configuration.

Model Generation

- Define the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Shell* immediately below *Structural Mass* in the left list and *Elastic 4node 181* in the right list; click on *OK*.
- Click on *Close*.
- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**, which will bring up another dialog box.
- Enter **200e9** for **EX** and **0.32** for **PRXY**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Specify the thickness for the shell (**SECTYPE** command) using the following menu path:

Main Menu > Preprocessor > Sections > Shell > Lay-up > Add/Edit

- *Create and Modify Shell Sections* dialog box appears; enter **0.003** for **Thickness**.
- Exit from the *Create and Modify Shell Sections* dialog box by clicking on **OK**.
- Create the solid model, a rectangular area in this case, using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

- In the *Create Rectangle by Dimensions* dialog box, type **0** for **X1**, **0.3** for **X2**, **0** for **Y1**, and **0.45** for **Y2**; click on **OK**.
- Turn line numbering on using the following menu path:

Utility Menu > PlotCtrls > Numbering

- Place a checkmark in the square box next to **LINE Line numbers**; click on **OK**.
- Plot lines using the menu path:

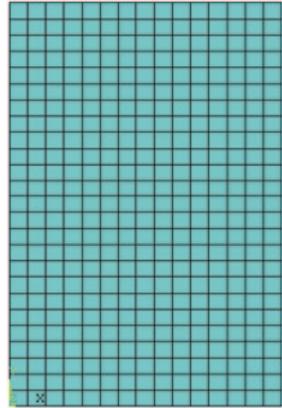
Utility Menu > Plot > Lines

- Specify the number of elements on selected lines for mapped meshing. On lines 1 and 3, use **15** divisions; on lines 2 and 4, use **25** divisions. Use the following menu path for this action:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

- Pick lines 1 and 3; click on **OK**.
- *Element Sizes on Picked Lines* dialog box appears; type **15** in the text field corresponding to **NDIV** (the second text field), and uncheck the first checkbox; click on **Apply**.
- Repeat this procedure for the next set of lines (2 and 4) with their corresponding number divisions as **25**. After specifying the number of divisions for lines 2 and 4, click on **OK** in the *Element Sizes on Picked Lines* dialog box.

Fig. 8.49 Mesh of the rectangular



- Create the mesh using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- Pick the area; click on **OK**.
- The mesh should appear in the *Graphics Window*, as shown in Fig. 8.49.

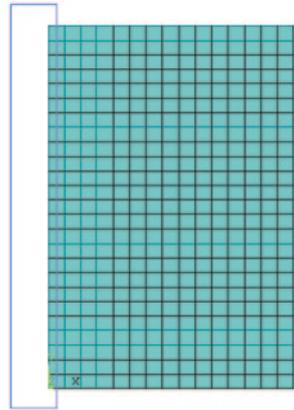
Solution

- Constrain the out-of-plane displacements (z -displacements) of the nodes along the entire boundary using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- In the *Pick Menu* click on the radio-button next to **Box**. This enables the user to pick several nodes at a time by drawing an area in the *Graphics Window*. Move the mouse pointer to a location slightly left and above the top-left corner of the meshed area.
- Click on the left mouse button (*without* releasing it) and draw a rectangle that encloses only the nodes along the $x=0$ boundary; release the left button (Fig. 8.50). Observe that each selected node is identified by a small square.
- Similarly, select nodes along all boundaries ($x=0.3$, $y=0$, and $y=0.45$); click on **OK** in the *Pick Menu*.
- Highlight **UZ**; click on **OK**.
- Apply remaining displacement constraints in the same manner as in the previous step.
 - Constrain displacements in the y -direction along the $y=0$ boundary.
 - Constrain the displacement in the x -direction at the boundary point $(x, y)=(0,0)$.

Fig. 8.50 Selecting nodes using plate, the *Box* option



- Apply the uniform load along the $y=0.45$ boundary using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Nodes

- Pick the nodes along the $y=0.45$ boundary; click on **OK**.
- Type **1**; click on **OK**.
- The eigenvalue buckling analysis calculates a scaling factor for the existing loads; therefore, if a unit load is applied, the scaling factor yields the buckling load.
- Turn on pre-stress effects using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- A dialog box appears; place a checkmark in the box next to **Calculate pre-stress effects**; click on **OK** (Fig. 8.51).
- Obtain the static solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with **Solution is done!**
- Exit *Solution Processor* using the following menu path:

Main Menu > Finish

- Re-enter the *Solution Processor* and change the analysis type to eigenvalue buckling using the following menu path:

Main Menu > Solution > Analysis Type > New Analysis

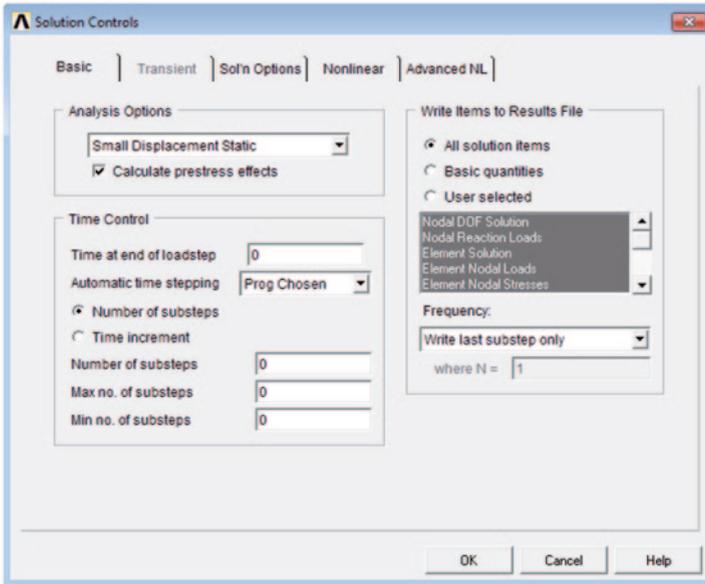


Fig. 8.51 *Solution Controls* dialog box (*Basic* tab shown)

- Click on *Eigen Buckling*; click on *OK*.

- Set analysis options using the following menu path:

Main Menu > Solution > Analysis Type > Analysis Options

- Type *4* in the text field next to *NMODE No. of modes to extract*; click on *OK*.
- In the new dialog box, click on *OK*, leaving the settings at their default values.

- Instruct ANSYS to expand modes using the following menu path:

Main Menu > Solution > Load Step Opts > ExpansionPass > Single Expand > Expand Modes

- Type *4* in the text field next to *NMODE No. of modes to extract*; click on *OK*.

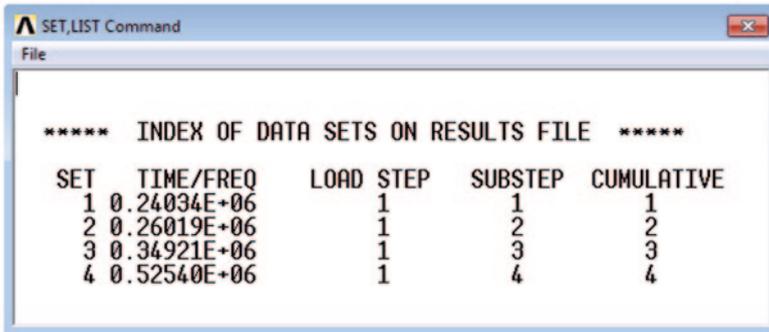
- Obtain the eigenvalue buckling solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on *OK* in the *Confirmation Window*.
- Wait until ANSYS responds with *Solution is done!*

Postprocessing

- Review the buckling loads using the following menu path:



The screenshot shows a window titled "SET,LIST Command" with a "File" menu. The main content is a table titled "***** INDEX OF DATA SETS ON RESULTS FILE *****". The table has five columns: SET, TIME/FREQ, LOAD STEP, SUBSTEP, and CUMULATIVE. The data is as follows:

SET	TIME/FREQ	LOAD STEP	SUBSTEP	CUMULATIVE
1	0.24034E+06	1	1	1
2	0.26019E+06	1	2	2
3	0.34921E+06	1	3	3
4	0.52540E+06	1	4	4

Fig. 8.52 List of buckling loads for different buckling modes

Main Menu > General PostProc > Results Summary

- The list will appear in a new window, as shown in Fig. 8.52. The critical load for the first mode is given as 0.24034E+06 Pa (0.24034 MPa) in this list. This means that when the applied load p is increased to this value, the plate will buckle in the first mode.
- Review the buckling modes.
 - Read the results for the first buckling load using the following menu path:

Main Menu > General PostProc > Read Results > First Set

- Obtain contour plot of the z -displacement using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- Click on *DOF Solution* and *Z-component of displacement*; click on *OK*.
- The contour plot will appear in the *Graphics Window*, as shown in Fig. 8.53.
- Read the results for the second buckling load using the following menu path:

Main Menu > General PostProc > Read Results > Next Set

- Obtain contour plot of the z -displacement using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- Click on *DOF Solution* and *Z-component of displacement*; click on *OK*.
- The contour plot will appear in the *Graphics Window*, as shown in Fig. 8.54.
- Repeat for modes 3 and 4 to obtain plots similar to those given in Fig. 8.55 and 8.56.
- Review the buckling mode shapes.
 - Read the results for the desired mode (as shown in previous step) and plot the deformed shape using the following menu path:

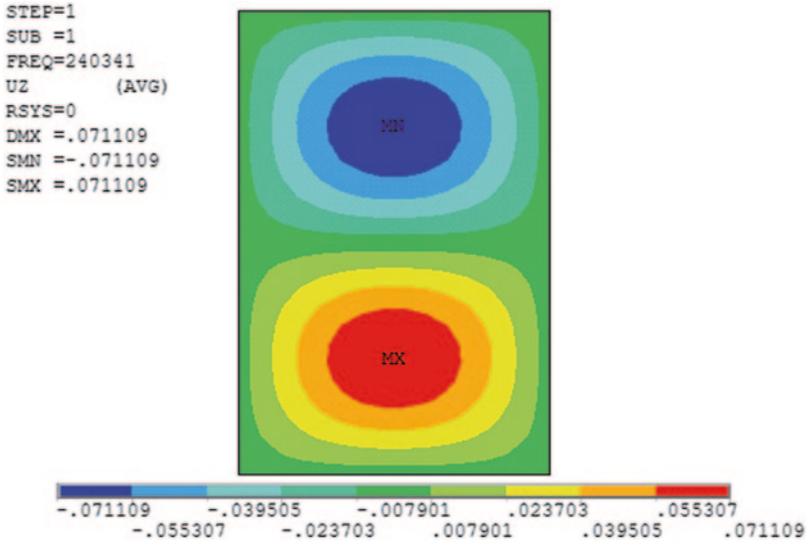


Fig. 8.53 Contour plot of u_z field (z-displacement) under first buckling mode

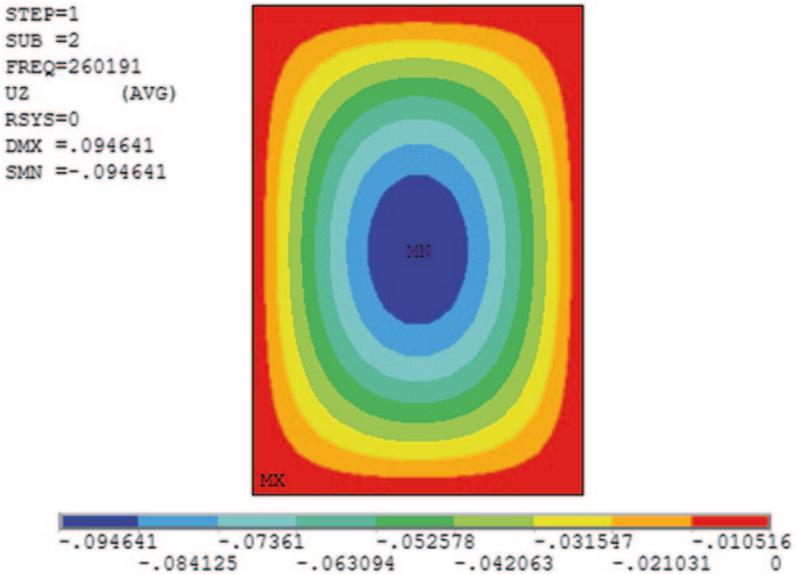


Fig. 8.54 Contour plot of u_z field (z-displacement) under second buckling mode

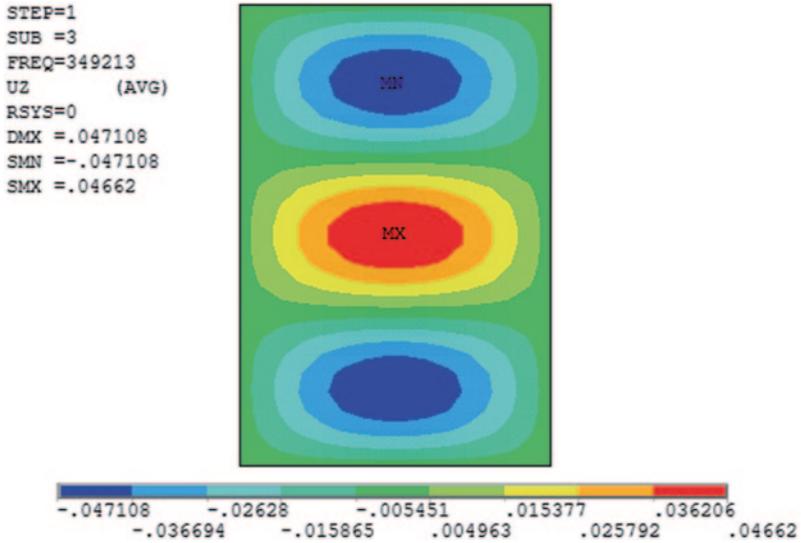


Fig. 8.55 Contour plot of u_z field (z-displacement) under third buckling mode

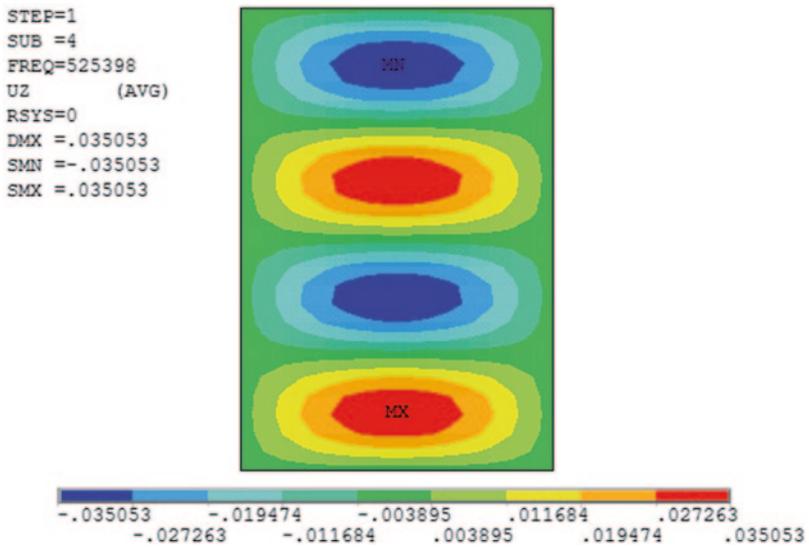


Fig. 8.56 Contour plot of u_z field (z-displacement) under fourth buckling mode

Main Menu > General Postproc > Plot Results > Deformed Shape

- Click on the *Def shape only* radio-button; click on **OK**.
- Change the viewpoint to isometric using the menu path:

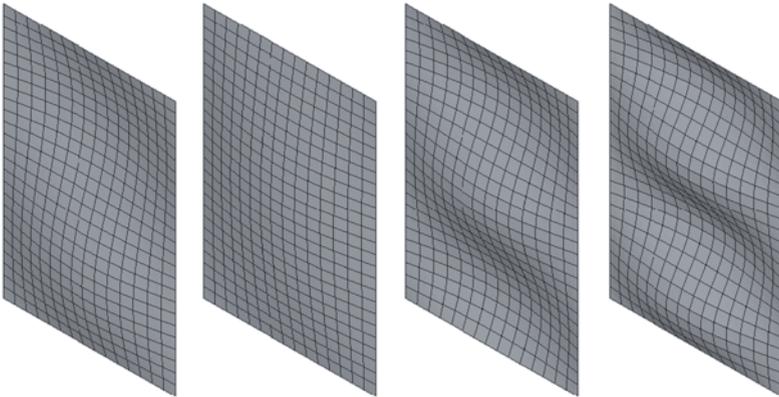


Fig. 8.57 Buckling modes 1 through 4 (from *left to right*)

Utility Menu > PlotCtrls > Pan Zoom Rotate

- In the *Pan Zoom Rotate* window, click on the *Iso* button.
- Figure 8.57 shows the first four mode shapes of the plate.

8.3 Thermomechanical Analysis

Thermal strains and stresses constitute an important part of the design considerations for many practical engineering problems. They become especially critical when materials with different coefficients of thermal expansion form interfaces.

As an example of a thermomechanical analysis with ANSYS, consider an electronic device containing a silicon die (chip), epoxy die-attach substrate, and a molding compound, as shown in Fig. 8.58. A common cause of failure in electronic devices is the thermal stresses at elevated temperatures caused by a coefficient of thermal expansion mismatch. In the ANSYS solution, plane strain idealization is utilized. The device is subjected to a uniform temperature increase of 30°C. Material properties of the constituent materials are given in Table 8.2. The goal is to obtain displacement and stress fields.

Model Generation

- Define the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Solid* immediately below *Structural Mass* in the left list and *Quad 4 Node 182* in the right list; click on *OK*.
- Click on *Options*.

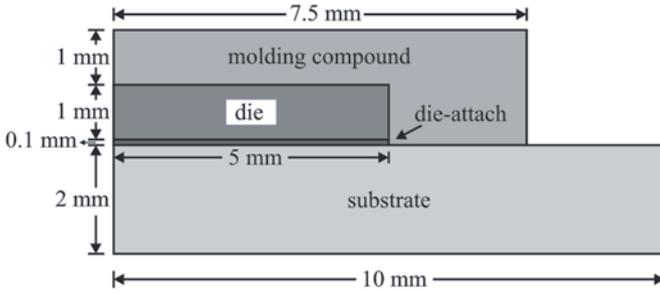


Fig. 8.58 Geometry of the electronic package

Table 8.2 Properties of the constituent materials in the electronic package

	E (GPa)	ν	α ($10^{-6}/^{\circ}\text{C}$)	Material reference number
Substrate	22	0.39	18	1
Die-attach	7.4	0.4	52	2
Silicon	163	0.278	2.6	3
Molding compound	15	0.25	16	4

- *PLANE182 element type options* dialog box appears; select **Plane strain** item from the pull-down menu corresponding to **Element behavior K3**.
- Click on **OK**; click on **Close**.
- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- *Define Material Model Behavior* dialog box appears. In the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**, which brings up another dialog box.
- Referring to Table 8.2, enter **22E9** for **EX** and **0.39** for **PRXY**; click on **OK**.
- In the right list, successively left-click on **Structural**, **Thermal Expansion**, **Secant Coefficient**, and, finally, **Isotropic**, which brings up another dialog box.
- Enter **18E-6** for **ALPX**; click on **OK**.
- Add new material model using the following menu path:

Material > New Model

- Repeat the procedure for the remaining materials (2 through 4) referring to Table 8.2.
- When finished, close the *Define Material Model Behavior* dialog box by using the following menu path:

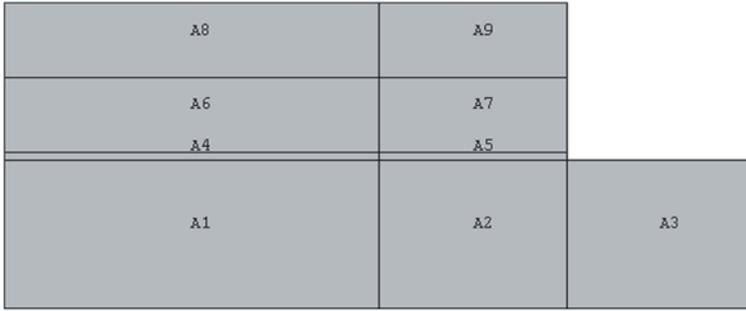


Fig. 8.59 Solid model of the electronic package

Material > Exit

- Create rectangles as identified in Fig. 8.59 (**RECTNG** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Rectangle > By Dimensions

- *Create Rectangle by Dimensions* dialog box appears. Referring to Table 8.3, enter **0** for *X1*, **5E-3** for *X2*, **0** for *Y1*, and **2E-3** for *Y2*; click on **Apply**.
- Repeat the procedure for the remaining areas (2 through 9). When creating Area 9, click on **OK** after entering the coordinates.

- Glue the areas (**AGLUE** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Operate > Booleans > Glue > Areas

- *Pick Menu* appears; click on **Pick All** button.

- Mesh the areas (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided

- *Pick Menu* appears; click on **Pick All**.
- At this point, all the elements have *Material Reference Number* 1. Attributes can be changed after the elements are created. For this purpose, areas are selected first. Then the elements that are attached to the selected areas are selected. Finally, elements are modified so they have the correct attributes. The correspondence between the areas and material numbers are given in Table 8.3. Select areas (**ASEL** command) using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; select **Areas** from the first pull-down menu and **By Num/Pick** from the second pull-down menu; click on **OK**.
- *Pick Menu* appears; pick areas 5, 7, 8, and 9; click on **OK**.
- Now, select the elements that are attached to the selected areas (**ESLA** command) using the following menu path:

Table 8.3 Coordinates defining the areas and the corresponding material reference numbers

Area number	<i>X1</i>	<i>X2</i>	<i>Y1</i>	<i>Y2</i>	Material reference number
	(mm)				
1	0	5	0	2	1
2	5	7.5	0	2	1
3	7.5	10	0	2	1
4	0	5	2	2.1	2
5	5	7.5	2	2.1	4
6	0	5	2.1	3.1	3
7	5	7.5	2.1	3.1	4
8	0	5	3.1	4.1	4
9	5	7.5	3.1	4.1	4

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; select **Elements** from the first pull-down menu; select **Attached to** from the second pull-down menu. Click on the **Areas** radio-button; click on **OK**.
- Modify the attributes of the selected set of elements (**EMODIF** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Move/Modify > Elements > Modify Attrib

- *Pick Menu* appears; click on **Pick All**, which brings up the *Modify Elem Attributes* dialog box.
- Select **Material MAT** from the pull-down menu and enter **4** in the **II New attribute number** field; click on **OK**.
- Repeat this procedure for area 4 (material reference number 2) and area 6 (material reference number 3).
- When finished, select everything (**ALLSEL** command) using the following menu path:

Utility Menu > Select > Everything**Solution**

- Apply displacement constraints (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the nodes along $x=0$ (y -axis); click on **OK** in the *Pick Menu*.
- Highlight **UX**; click on **Apply**.
- *Pick Menu* reappears; pick the bottom-left corner node ($x=0, y=0$); click on **OK** in the *Pick Menu*.

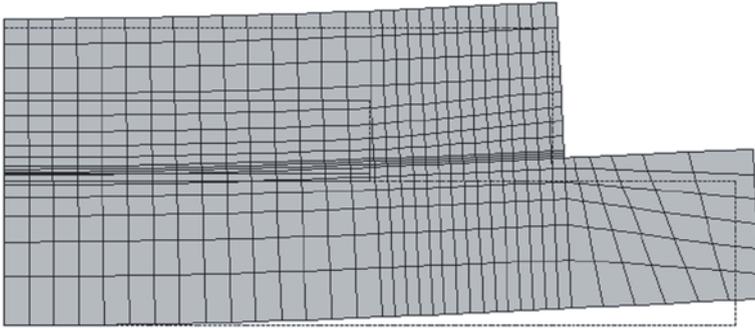


Fig. 8.60 Deformed shape of the electronic package under thermal load

- Remove highlight *UY* (leave the *UX* highlighted); click on **OK**.
- Apply the thermal load (**TUNIF** command) using the following menu path:
Main Menu > Solution > Define Loads > Apply > Structural > Temperature > Uniform Temp
 - *Uniform Temperature* dialog box appears; enter **30** for **TUNIF**; click on **OK**.
- Obtain the solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window*; click on **OK** in the *Confirmation Window*.
- Wait until ANSYS responds with ***Solution is done!***

Postprocessing

- Review the deformed shape (**PLDISP** command) using the following menu path:

Main Menu > General PostProc > Plot Results > Deformed Shape

- Select **Def + undef edge**; click on **OK**.
- The deformed shape is shown in Fig. 8.60 as it appears in the *Graphics Window*.
- Obtain the normal stress in the *y*-direction and shearing stress contour plots (**PLNSOL** command) using the following menu path:

Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu

- In order to obtain the view of the normal stresses in the *y*-direction, select **Stress** and **Y-component of stress**; click on **OK**.
- The resulting contour plot, along with a zoomed-in view of the critical junction, is shown in Fig. 8.61.

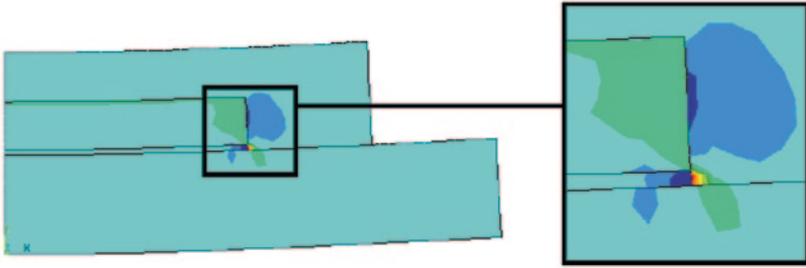


Fig. 8.61 Contour plot of the normal stress (σ_{yy}) in the y -direction: in the entire package (*left*) and in the vicinity of the die/die-attach interface (*right*)

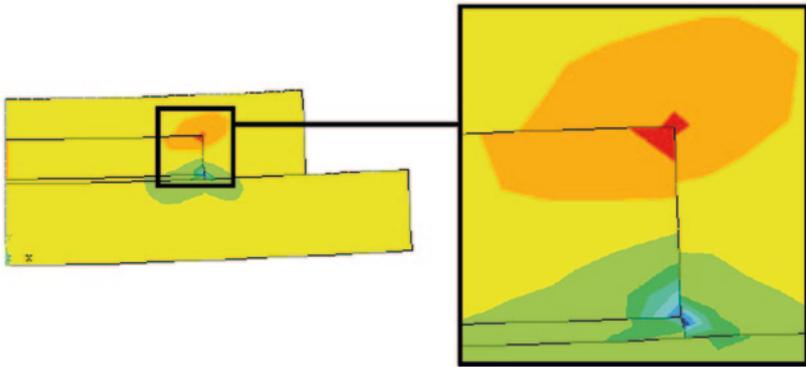


Fig. 8.62 Contour plot of the shear stress (σ_{xy}): in the entire package (*left*) and in the vicinity of the die/die-attach interface (*right*)

- Similarly, in order to view the shear stresses, select **Stress** and **XY Shear stress**; click on **OK**.
 - The resulting contour plot, along with a zoomed-in view of the critical junction, is shown in Fig. 8.62.
- Plot elements (**EPLOT** command) using the following menu path:

Utility Menu > Plot > Elements

- Review variation of stresses along paths by means of line plots. Two paths are defined, both of which are vertical. The first path passes through the vertical cross section where the die and the die-attach terminate and form an interface with the molding compound ($x = 5mm.$). The second path is located approximately in the middle of the die and die-attach. Both paths are plotted in Fig. 8.63 and 8.64 (element edges are removed in Fig. 8.64 for clarity). Define the path (**PPATH** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Define Path > By Nodes

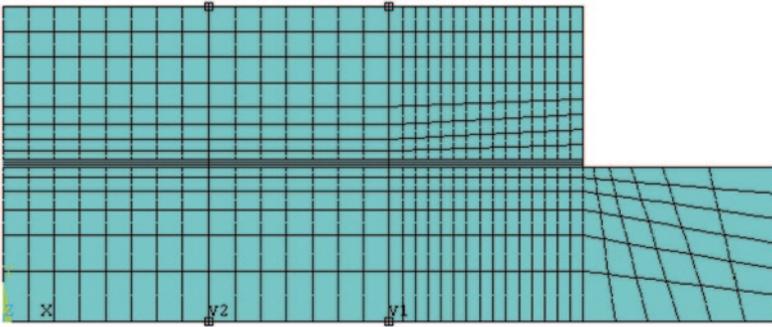


Fig. 8.63 Element plot with paths V1 and V2 identified

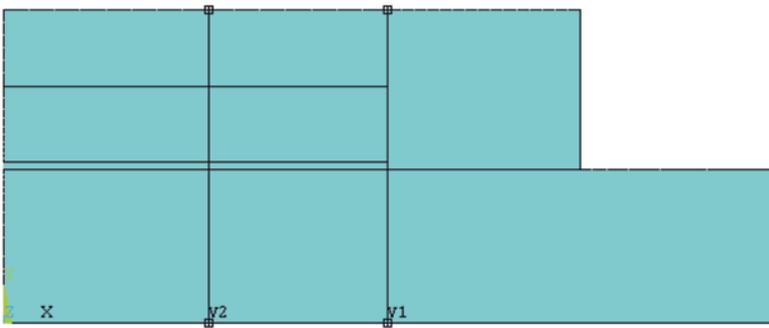


Fig. 8.64 Element plot (element edges removed) with paths V1 and V2 identified

- *Pick Menu* appears; pick the two nodes indicated with small squares, as shown in Fig. 8.63 (corresponding to the path V1); click on **OK**.
- *By Nodes* dialog box appears; enter a name describing the path, say **V1**, in the **Define Path Name** text field; click on **OK**.
- Close the *PATH Command Status Window*.
- Define a second path as indicated in Fig. 8.63 and 8.64 (corresponding to the path V2); enter the name as **V2**.
- When multiple paths are defined, only one path is active at a given time, and mapping of results is performed on the active path. Activate the path V1 (**PATH** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Recall Path

- *Recall Path* dialog box appears. Select **V1**; click on **OK**.
- Map results onto path (**PDEF** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Map onto Path

- *Map Result Items onto Path* dialog box appears; select **Stress** from the left list and **Y-direction SY** from the right list; click on **Apply**.

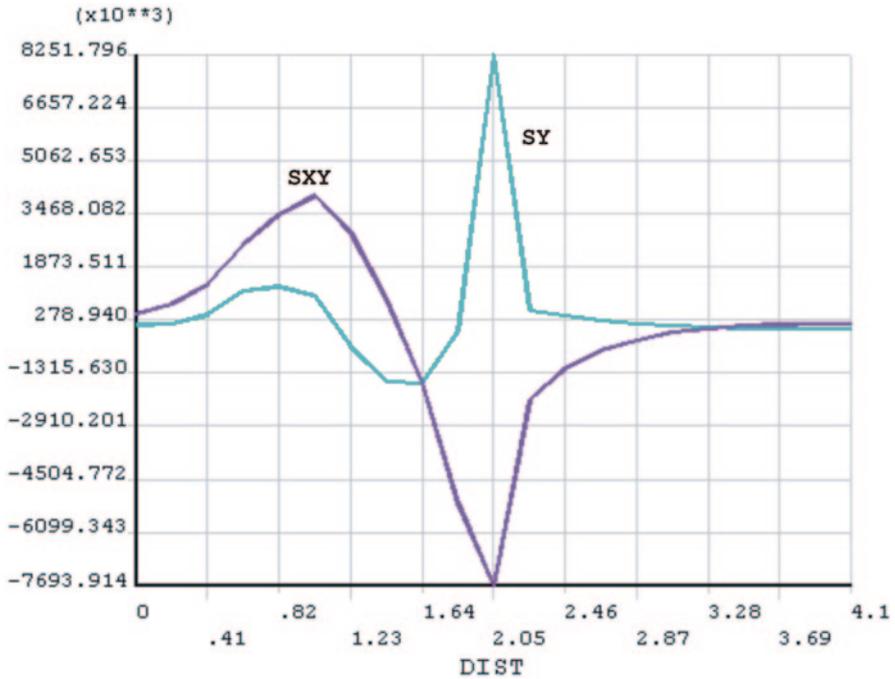


Fig. 8.65 Line plots of σ_{yy} and σ_{xy} along path V1

- *Map Result Items onto Path* dialog box remains active; select **Stress** from the left list and **XY-shear SXY** from the right list; click on **Apply**.
- Now, select **Stress** from the left list and scroll down in the right list to select **von Mises SEQV**; click on **OK**.
- At this point, normal stress in the y -direction (σ_{yy}), xy shear stress (σ_{xy}), and equivalent stress (σ_{eqv}) values are mapped onto path V1. Obtain line plot of σ_{yy} and σ_{xy} along the path V1 (**PLPATH** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Plot Path Item > On Graph

- *Plot of Path Items on Graph* dialog box appears; select **SY** and **SXY**; click on **OK**.
- Figure 8.65 shows the line plots of σ_{yy} and σ_{xy} along the defined path.
- Now, obtain line plot of σ_{eqv} in the same graph with σ_{yy} and σ_{xy} along path V1 (**PLPATH** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Plot Path Item > On Graph

- *Plot of Path Items on Graph* dialog box appears; add **SEQV** to the existing selection (**SY** and **SXY**); click on **OK**.

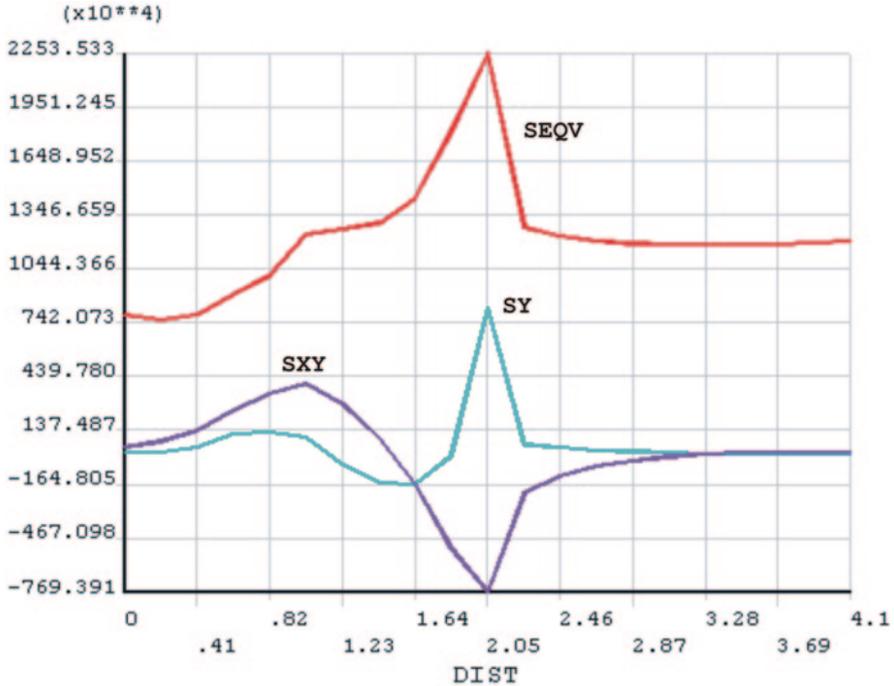


Fig. 8.66 Line plots of σ_{yy} , σ_{xy} , and σ_{eqv} along path V1

- Figure 8.66 shows the resulting line plot.
- Similar plots can be obtained for stresses along path V2. For this purpose, the user needs to activate path V2, followed by the mapping of quantities. Figure 8.67 shows the variation of σ_{yy} , σ_{xy} , and σ_{eqv} along the path.

8.4 Fracture Mechanics Analysis

Computation of fracture parameters, such as the stress intensity factors or energy release rate, using finite element analysis requires either a refined mesh around the crack tip or the use of “special elements” with embedded stress singularity near the crack tip. Although conceptually the stress intensity factors are obtained in a straightforward manner, finite element analyses with conventional elements near the crack tip always underestimate the sharply rising stress-displacement gradients.

Instead of trying to capture the well-known $1/\sqrt{r}$ singular behavior with smaller and smaller elements, Henshell and Shaw (1975) and Barsoum (1976, 1977) introduced a direct method by shifting the mid-side node of an 8-noded isoparametric quadrilateral element to the one-quarter point from the crack tip node. Relocating

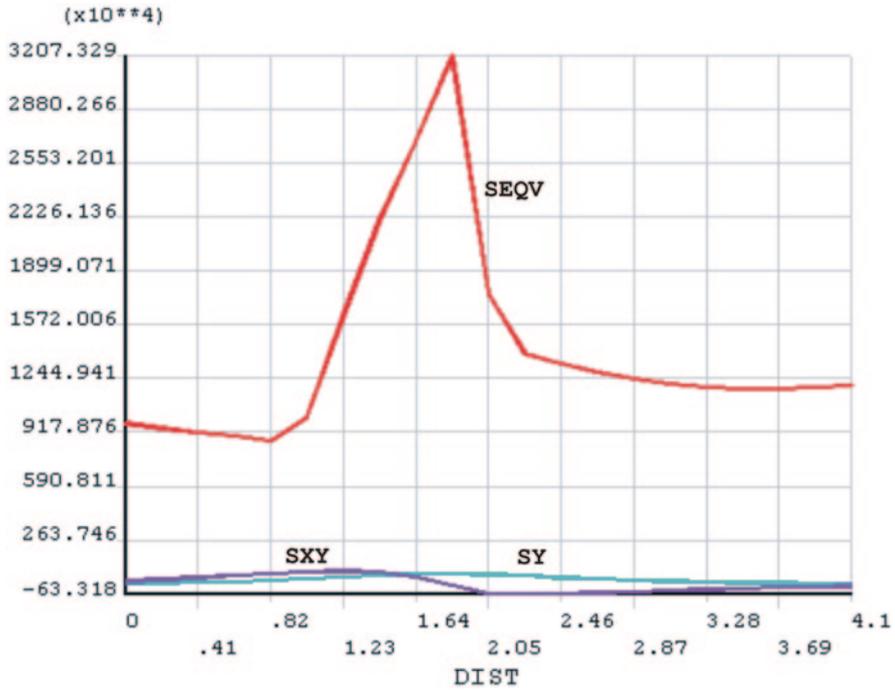


Fig. 8.67 Line plots of σ_{yy} , σ_{xy} , and σ_{eqv} along path V2

the mid-side nodes to the one-quarter point achieves the desired $1/\sqrt{r}$ singular behavior. In the case of linear elastic deformation, the elements **PLANE183** (2-D, 8-noded quadrilateral), and **SOLID186** (3-D, 20-noded brick) in ANSYS are used to obtain the well-established singular stress field by shifting the mid-side nodes one-quarter away from the crack tip.

Once an accurate stress field is obtained, fracture parameters (i.e., stress intensity factors, J -integral, and energy release rate) can be calculated within the ANSYS postprocessor.

As an extension of the node collapsing approach, Pu et al. (1978) showed that the stress intensity factors, K_I and K_{II} for opening and sliding modes, respectively, can be computed directly from the nodal displacements on opposite sides of the crack plane as

$$K_I = G \sqrt{\frac{2\pi}{r_0}} \frac{u_y(r_0, \theta = \pi) - u_y(r_0, \theta = -\pi)}{(\kappa + 1)}$$

and

$$K_{II} = G \sqrt{\frac{2\pi}{r_0}} \frac{u_x(r_0, \theta = \pi) - u_x(r_0, \theta = -\pi)}{(\kappa + 1)}$$

Fig. 8.68 Displacements at nodal points located behind the crack tip

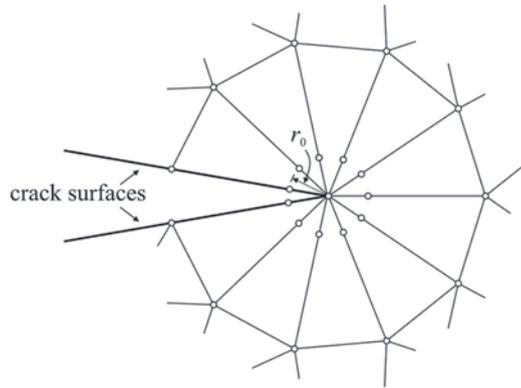
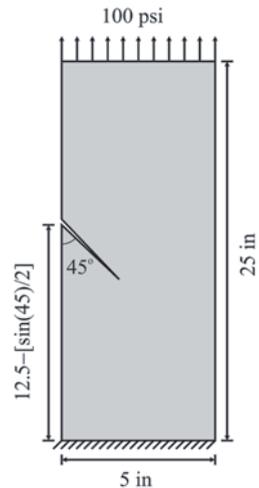


Fig. 8.69 Geometry of the strip with inclined edge crack



in which r_0 , usually restricted to one or two percent of the crack length, is the distance from the crack tip to the first side-node behind the crack tip, as shown in Fig. 8.68. The shear modulus is G , and the parameters are $\kappa = 3 - \nu/1 + \nu$ and $\kappa = 3 - 4\nu$ for plane stress and strain idealizations, respectively.

Under plane strain assumptions, the computation of the stress intensity factors within ANSYS is demonstrated by considering a strip with an inclined edge crack, as shown in Fig. 8.69. The crack is 1 in long and has an inclination angle of 45°. The width and length of the strip are 5 and 25 in, respectively. The bottom surface of the strip is constrained in both directions while the top surface is subjected to a tensile load of 100 psi. Elastic modulus and Poisson's ratio of the strip are 30×10^6 psi and 0.3, respectively.

In the ANSYS solution, special meshing around the crack tip is utilized. It relocates the mid-side nodes to one-quarter away from the crack tip. Coordinates of the keypoints are listed in Table 8.4. Note that keypoint 2 is located at the crack tip, and

Table 8.4 Coordinates of the keypoints

Keypoint no.	x	y
1	0	$12.5 + \frac{\sin(45)}{2}$
2	$\cos(45)$	$12.5 - \frac{\sin(45)}{2}$
3	0	$12.5 + \frac{\sin(45)}{2}$
4	0	0
5	5	0
6	5	$12.5 - \left\{ [5 - \cos(45)] \times \tan(45) + \frac{\sin(45)}{2} \right\}$
7	5	25
8	0	25

Table 8.5 Line-keypoint correspondence in the solid model

Line no.	Keypoint 1	Keypoint 2
1	4	5
2	5	6
3	6	2
4	2	1
5	1	4
6	6	7
7	7	8
8	8	3
9	3	2

keypoints 1 and 3 are coincident, each belonging to the opposite crack faces. Line numbers with their corresponding keypoints are listed in Table 8.5. The goal is to obtain stress intensity factors, as well as the displacement and stress fields.

Model Generation

- Define element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Solid* immediately below *Structural Mass* from the left list and *Quad 8node 183* on the right list; click on *OK*.
- Click on *Options*.

- *PLANE183 element type options* dialog box appears; select **Triangle** from the pull-down menu corresponding to **Element shape K1** and **Plane strain** from the pull-down menu corresponding to **Element behavior K3**.
- Click on **OK**; click on **Close**.

- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- *Define Material Model Behavior* dialog box appears; in the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**, which brings up another dialog box.
- In the new dialog box, enter **30E6** for **EX** and **0.3** for **PRXY**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Change the default angular unit to degrees (***AFUN** command) using the following menu path:

Utility Menu > Parameters > Angular Units

- *Angular Units for Parametric Functions* dialog box appears; select **Degrees DEG** from the pull-down menu; click on **OK**.

- Create keypoints (**K** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

- *Create Keypoints in Active Coordinate System* dialog box appears. Referring to Table 8.4, enter **0** and **12.5+SIN(45)/2** for **X** and **Y**, leaving the text fields for **NPT** and **Z** blank. Click on **Apply**.
- Repeat the procedure for the remaining keypoints (2 through 8). When creating keypoint 8, click on **OK** after entering the coordinates.

- Create lines (**L** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line

- *Pick Menu* appears, prompting the user to pick two keypoints forming the line. Referring to Table 8.5, pick the correct keypoints. When picking keypoint 1 or 3, ANSYS displays a warning message informing the user that there are two coincident keypoints at the particular location. By clicking on the **Next** button in this message, pick the correct keypoint.

- Turn line and keypoint numbering on (**/PNUM** command) using the following menu path:

Utility Menu > PlotCtrls > Numbering

- *Plot Numbering Controls* dialog box appears. Click on the boxes next to **KP Keypoint numbers** and **LINE Line numbers** (this places checkmarks), and select **Numbers only** from the *[/NUM] Numbering shown* with the pull-down menu. Click on **OK**.
- Plot lines (**L PLOT** command) using the following menu path:

Utility Menu > Plot > Lines

- Figure 8.70 shows the line plot with both keypoint and line numbers printed. Observe that keypoints 1 and 3 and lines 4 and 9 are coincident.
- Two areas are created. The first area utilizes lines 1–5 while the second area is formed by lines 6, 7, 8, 9, and 3. Create areas using lines (**AL** command) using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > By Lines

- *Pick Menu* appears, prompting the user to pick lines forming the area. Pick lines 1 through 5 (in this order) and click on **OK** in the *Pick Menu*. When picking line 4, ANSYS informs the user that there are two coincident lines at the picked location. Make sure to pick line 4.
- Repeat the same procedure for the second area by picking lines 6, 7, 8, 9, and 3. Similar to the previous case, make sure to pick line 9 (instead of line 4).
- For the stress intensity factor calculations, a local coordinate system aligned with the crack faces is needed. Create a local coordinate system using 3 keypoints (**CSKP** command) using the following menu path:

Utility Menu > WorkPlane > Local Coordinate Systems > Create Local CS > By 3 Keypoints

- *Pick Menu* appears; pick keypoints 2, 6, and 7 (in this order) and click on **OK** in the *Pick Menu*.
- *Create CS By 3 KPs* dialog box appears; click on **OK**.
- The local coordinate system is now active. Activate the global Cartesian coordinate system (**CSYS** command) using the following menu path:

Utility Menu > WorkPlane > Change Active CS to > Global Cartesian

- Specify keypoint 2 to be the crack tip so that the elements around it have the singular stress capability (**KSCON** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > Concentrat KPs > Create

- *Pick Menu* appears; pick keypoint 2; click on **OK** in the *Pick Menu*.
- *Concentration Keypoint* dialog box appears; enter *1/20* for **DEL R Radius of 1st row of elems** and *6* for **NTHET No of elems around circumf**. Select **Skewed 1/4pt** from the **KCTIP midside node position** pull-down menu and click on **OK**.
- Specify mesh density around keypoints (**KE SIZE** command) using the following menu path:

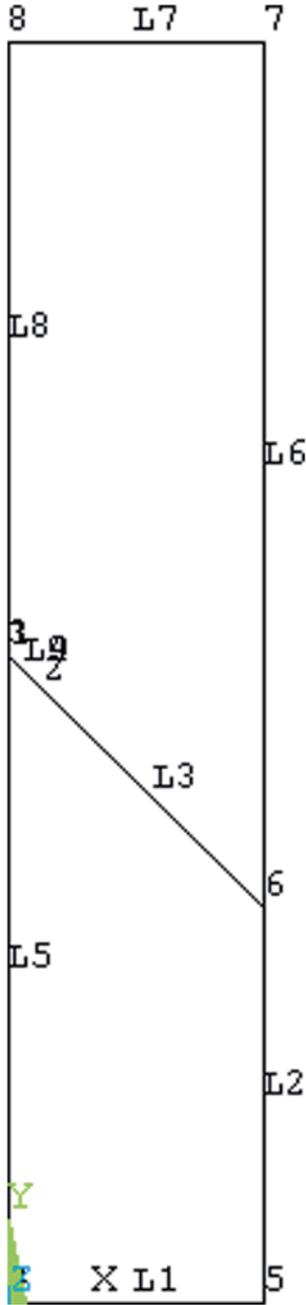


Fig. 8.70 Line plot with both keypoint and line numbers printed

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Keypoints > All KPs

- *Element Size at All Keypoints* dialog box appears; enter $5/3$ for **SIZE Element edge length**; click on **OK**.
- Specify mesh density around specific keypoints (**KESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Keypoints > Picked KPs

- *Pick Menu* appears; pick keypoints 1 and 3 (since these keypoints are coincident, click on the location twice to pick both of them); click on **OK** in the *Pick Menu*.
- *Element Size at Picked Keypoints* dialog box appears; enter $1/3$ for **SIZE Element edge length**; click on **OK**.
- Specify mesh density around crack tip (**KESIZE** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Keypoints > Picked KPs

- *Pick Menu* appears; pick keypoint 2; click on **OK** in the *Pick Menu*.
- *Element Size at Picked Keypoints* dialog box appears; enter $1/30$ for **SIZE Element edge length**; click on **OK**.
- Mesh the areas (**AMESH** command) using the following menu path:

Main Menu > Preprocessor > Meshing > Mesh > Areas > Free

- *Pick Menu* appears; click on **Pick All**.
- Close the *Warning Window*.
- Zoom in around the crack tip and observe the mesh pattern around it (Fig. 8.71).

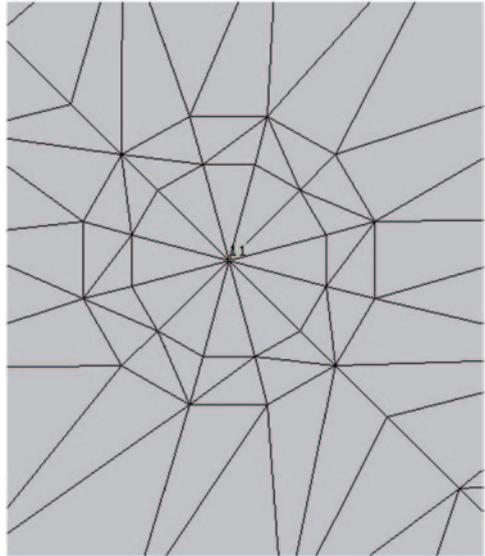
Solution

- Apply displacement constraints (**D** command) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; pick the nodes along $y=0$ (x -axis); click on **OK** in *Pick Menu*.
- *Apply U, ROT on Nodes* dialog box appears; highlight **UY**; click on **Apply**.
- *Pick Menu* reappears; pick the bottom-left corner node ($x=0, y=0$); click on **OK** in *Pick Menu*.
- *Apply U, ROT on Nodes* dialog box reappears; highlight **UX** (leave the **UY** highlighted) and click on **OK**.
- Apply surface load (**SF** command) using the following menu path:

Fig. 8.71 Mesh pattern around the crack tip



Main Menu > Solution > Define Loads > Apply > Structural > Pressure > On Nodes

- *Pick Menu* appears; pick the nodes along $y=25$; click on **OK** in *Pick Menu*.
 - *Apply PRES on nodes* dialog box appears; enter **-100** for **VALUE Load PRES value**; click on **OK**.
- Obtain solution (**SOLVE** command) using the following menu path:

Main Menu > Solution > Solve > Current LS

- *Confirmation Window* appears along with *Status Report Window*.
- Review status; if OK, close the *Status Report Window* and click on **OK** in the *Confirmation Window*.
- If a dialog box with a message reminding the user about previous warnings issued appears, click on **Yes**.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review deformed shape (**PLDISP** command) using the following menu path:

Main Menu > General PostProc > Plot Results > Deformed Shape

- Select **Def shape only**; click on **OK**.
 - The deformed shape near the crack is shown in Fig. 8.72 as it appears in the *Graphics Window*.
- Activate local coordinate system 11 (**CSYS** command) using the following menu path:



Fig. 8.72 Deformed shape of the crack (*left*) and the deformed shape in the vicinity of the crack tip (*right*)

Utility Menu > WorkPlane > Change Active CS to>Specified Coord Sys

- Change Active CS to Specified CS dialog box appears; enter **11** for **KCN Coordinate system number**; click on **OK**.
- Enforce the use of the same coordinate system (11) for the results calculations and display (**RSYS** command) using the following menu path:

Main Menu > General Postproc > Options for Outp

- Options for Output dialog box appears; select **Local system** from the [**RSYS**] **Results coord system** pull-down menu and enter **11** for **Local system reference no.**; click on **OK**.
- In order to calculate stress intensity factors, a path along the crack faces in the vicinity of the crack tip is defined (**PPATH** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Define Path > By Nodes

Pick Menu appears; a total of 5 nodes are needed for this operation. Crack tip node needs to be picked first, followed by the two nodes closest to the crack tip along the top crack face. Finally, the two nodes closest to the crack tip along the bottom crack face are picked. The crack tip node number is 18 in this particular problem. Node numbers for the two nodes closest to the crack tip along the top and bottom faces are 47 and 48, and 521 and 522, respectively. Before picking the nodes, it is recommended that the user zoom in around the crack tip and plot elements as shown in Fig. 8.73. Pick nodes 18, 47, 48, 522, and 521 (in this order) and click on **OK**. The nodal locations to be picked are also shown in Fig. 8.73 (denoted by small squares).

- *By Nodes* dialog box appears; enter a path name (say *crack*); click on **OK**.
- Close the new information window.
- Calculate stress intensity factors (**KCALC** command) using the following menu path:

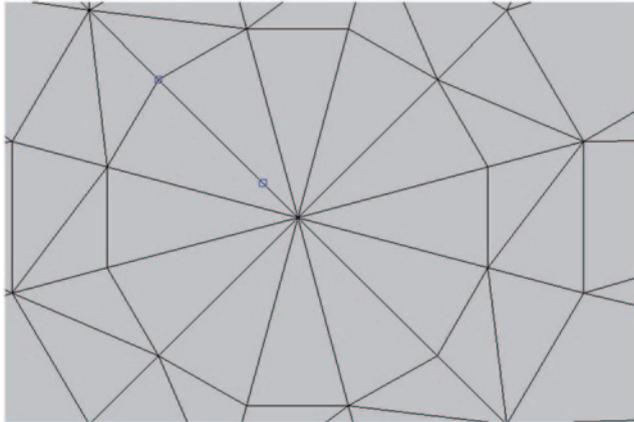


Fig. 8.73 Elements around the crack tip is zoomed in for picking operation

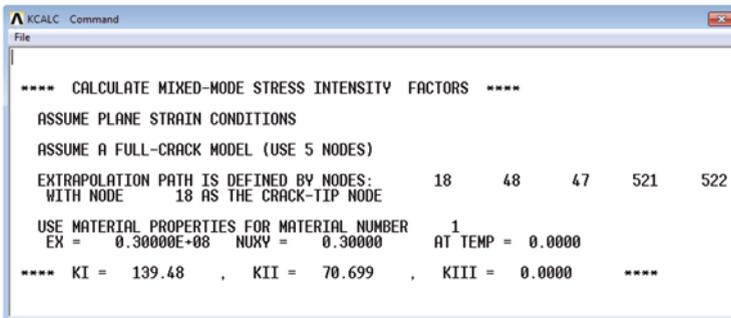


Fig. 8.74 Stress intensity factor results

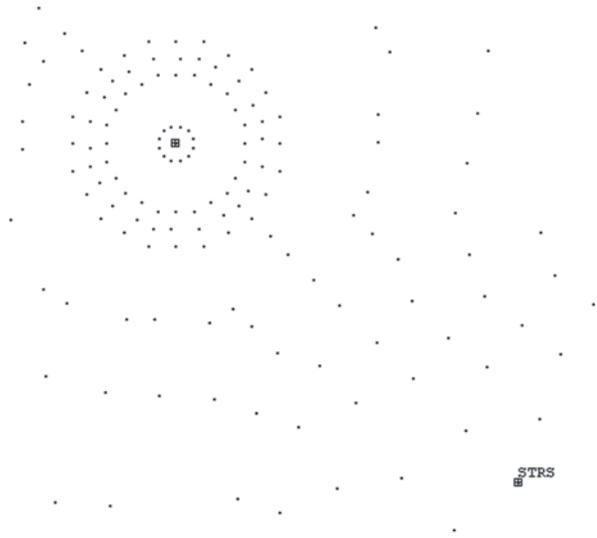
Main Menu > General PostProc > Nodal Calcs > Stress Int Factr

- *Stress Intensity Factor* dialog box appears; select **Plane strain** from the **KPLAN Disp extrapolat based on** pull-down menu and select **Full-crack model** from the **KCSYM Model type** pull-down menu. Click on **OK**.
- Stress intensity factors are reported in a separate window (KI=139.48, KII=70.699), as shown in Fig. 8.74.
- Review normal and shear stresses ahead of the crack tip, in the direction of the crack. For this purpose, define a new path (**PPATH** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Define Path > By Nodes

- *Pick Menu* appears; pick nodes 18 (crack tip) and 34, as shown in Fig. 8.75; click on **OK** in the *Pick Menu*.
- *By Nodes* dialog box appears; enter a path name (say **strs**); click on **OK**.

Fig. 8.75 Nodal plot showing the starting (crack tip, node 18) and ending (node 34) nodes for the path definition



- Close the new information window.
- Map stresses onto the path *strs* (**PDEF** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Map onto Path

- *Map Result Items onto Path* dialog box appears; select **Stress** from the left list and **Y-direction SY** from the right list; click on **Apply**.
- *Map Result Items onto Path* dialog box reappears; select **Stress** from the left list and **XY-shear SXY** from the right list and click on **OK**.
- Plot stresses along the path on the graph (**PLPATH** command) using the following menu path:

Main Menu > General Postproc > Path Operations > Plot Path Item > On Graph

- *Plot of Path Items on Graph* dialog box appears; select **SY** from the list; click on **OK**. The graph appears in the *Graphics Window*, as shown in Fig. 8.76. Similarly, shear stresses are plotted by selecting item **SXY** in the *Plot of Path Items on Graph* dialog box (shown in Fig. 8.77).

8.5 Dynamic Analysis

There are three commonly used dynamic analysis types in ANSYS: (i) modal analysis, (ii) harmonic analysis, and (iii) transient analysis.

The results related to these types of analyses can be reviewed in both postprocessors (*General Postprocessor* and *Time History Postprocessor*). The *General Postprocessor* is used to review results over the entire model at specific

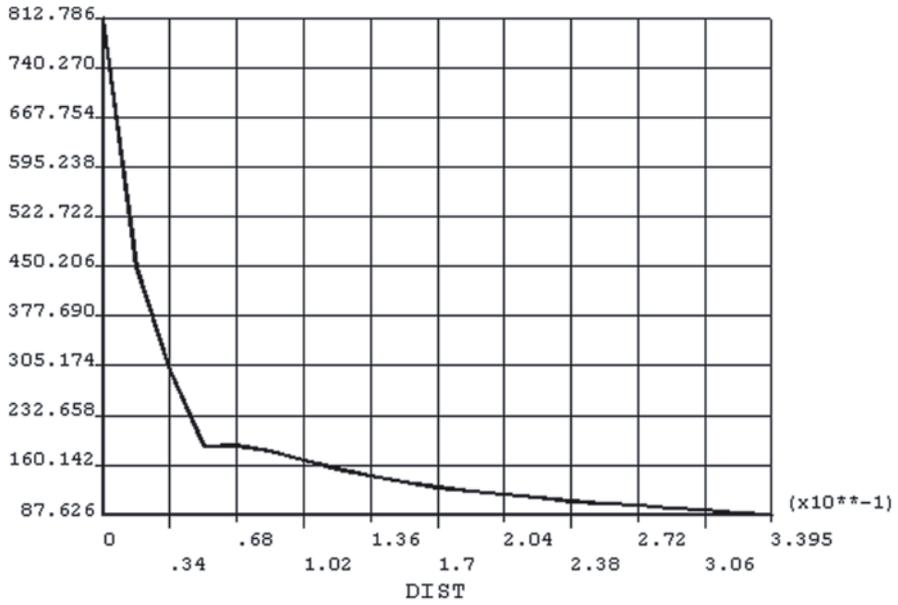


Fig. 8.76 Normal stress σ_y (in coordinate system 11) ahead of the crack tip

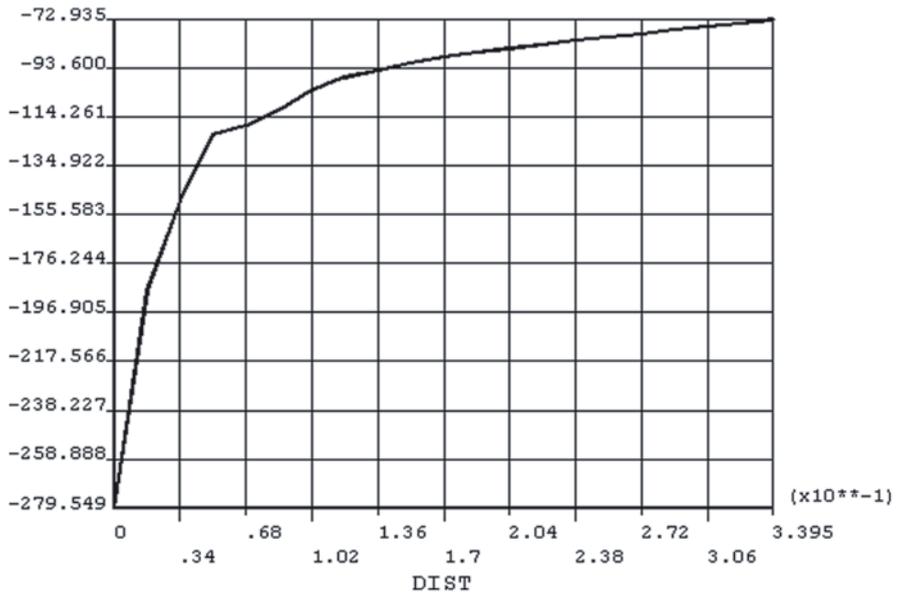


Fig. 8.77 Shear stress σ_{xy} (in coordinate system 11) ahead of the crack tip

times or frequencies while the *Time History Postprocessor* allows the user to review results at specific nodes in the model over the entire time or frequency range.

8.5.1 Modal Analysis

If the structural vibration is of concern in the absence of time-dependent external loads, a modal analysis is performed. Because the structural frequencies are not known a priori, the finite element equilibrium equations for this type of analysis involve the solution of homogeneous algebraic equations whose eigenvalues correspond to the frequencies, and the eigenvectors represent the vibration modes. The following steps are used in a typical modal analysis in ANSYS:

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

In certain cases, especially if the model has of a large number of degrees of freedom, it is advantageous to define *Master Degrees of Freedom* (MDOF). This procedure condenses the full matrices describing the structure into a smaller size, thus reducing the computational cost significantly. The only boundary conditions that are permissible in *modal analysis* are zero displacements. Any constraints/loads that are non-zero are ignored in the analysis. Once the modal analysis is complete, the solution is *expanded* to find results related to the complete structure—not just the MDOF. The results include natural frequencies, mode shapes, and corresponding parametric (relative) stress fields.

8.5.1.1 Modal Analysis of a Bracket

The bracket shown in Fig. 8.29 is clamped at the two top holes. The reduced method of modal analysis is used here, which requires master degrees of freedom. Master degrees of freedom are automatically selected by ANSYS. The reduced modal analysis is then expanded for the number of modes desired (in this case, four). The goal is to obtain the modal frequencies and corresponding mode shapes.

Model Generation

The finite element model of this bracket is created in a separate problem (Sect. 8.1.5.1). Therefore, the model will not be regenerated here; it will be generated interactively using an input file. It is worth noting that the components consisting of the nodes along the holes are defined in the input file.

- Create the model by reading the input file “*bracket.inp*” from the CD-ROM location *Input_Files\ch08\bracket.inp* using the following menu path:

Utility Menu >File > Read Input from

- Browse for the *bracket.inp* file in the folder given above; click on **OK**.
- Wait until the model is generated.

Solution

- Specify the *Analysis Type* as *Modal*, using the following menu path (**ANTYPE** command), which will bring up the *New Analysis* dialog box:

Main Menu > Solution > Analysis Type > New Analysis

- Click on the *Modal* radio-button; click on **OK**.
- Print 4 reduced mode shapes using the following menu path:

Main Menu > Solution > Analysis Type > Analysis Options

- Click on the *Reduced* radio-button; click on **OK**, which will bring up another dialog box.
- Enter **4** for *PRMODE*; click on **OK**.
- Specify the master degrees of freedom using the following menu path:

Main Menu > Solution > Master DOFs > Program Selected

- In the dialog box, type **20** for *NTOT Total no. of master DOF*; click on the checkbox to show **Yes** for *NRMDF Exclude rotational DOF*; click on **OK**.
- Expand the first four modes using the following menu path:

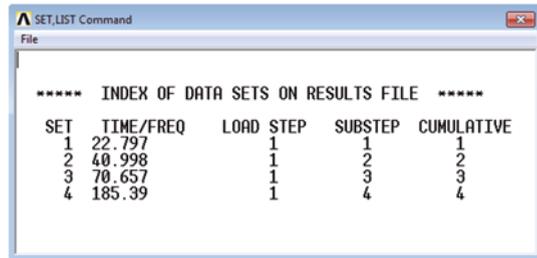
Main Menu > Solution > Load Step Opts > ExpansionPass > Single Expand > Expand Modes

- Enter **4** for *NMODE*; click on **OK**.
- Constrain displacement and rotation degrees of freedom along the top-left and -right holes. For this purpose, first select the components created earlier for these holes (*TL_BOLT* and *TR_BOLT*) using the following menu path:

Utility Menu > Select > Comp/Assembly > Select Comp/Assembly

- A dialog box appears; click on the *by component name* radio-button; click on **OK**.
- A new dialog box with the components listed appears; highlight *TL_BOLT*; click on **OK**. This action selects the nodes along the top-left hole.
- Specify the displacement boundary conditions using the following menu path:

Fig. 8.78 Listing of the mode frequencies



The screenshot shows a window titled "SET,LIST Command" with a menu bar containing "File". The main content area displays the following text:

```

***** INDEX OF DATA SETS ON RESULTS FILE *****
SET   TIME/FREQ   LOAD STEP   SUBSTEP   CUMULATIVE
  1   22.797       1     1         1
  2   40.998       1     2         2
  3   70.657       1     3         3
  4  185.39        1     4         4

```

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; click on **Pick All**.
- In the new dialog box, highlight **All DOF**; click on **OK**.
- Repeat the same procedure for the top-right hole (**TR_BOLT**).
- Select everything (**ALLSEL** command) using the following menu path:

Utility Menu > Select > Everything

- Obtain the solution using the following menu path:

Main Menu > Solution > Solve > Current LS

- Close the */STATUS* information window.
- Click on **OK** to start the solution.
- If a dialog box with a message reminding the user about previous warnings issued appears, click on **Yes**.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review the results using the following menu path:

Main Menu > General Postproc > Results Summary

- The mode frequencies appear in a separate window, as shown in Fig. 8.78.
- Plot the mode shapes.
 - Set the solution to **First Set** using the following menu path:

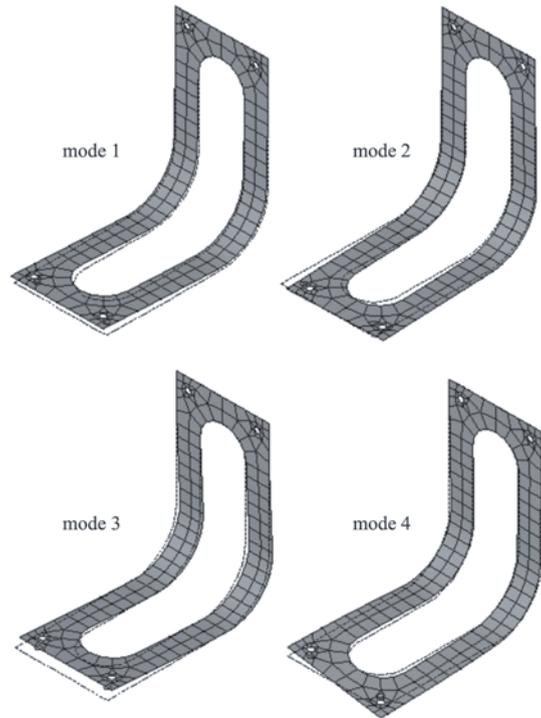
Main Menu > General Postproc > Read Results > First Set

- Plot the mode shape using the following menu path:

Main Menu > General Postproc > Plot Results > Deformed Shape

- Click on **Def + undef edge** radio-button; click on **OK**.
- The mode shape is shown in Fig. 8.79 as it appears in the *Graphics Window*.

Fig. 8.79 Mode shapes 1 through 4



- Set the solution to **Second Set** using the following menu path:

Main Menu > General Postproc > Read Results > Next Set

- Plot the mode shape using the following menu path:

Main Menu > General Postproc > Plot Results > Deformed Shape

- Click on **Def + undef edge** radio-button; click on **OK**.
- The mode shape is shown in Fig. 8.79 as it appears in the *Graphics Window*.
- Repeat for the third and fourth modes.

8.5.1.2 Vibration of an Automobile Suspension

An automobile suspension system is simplified to consider only two major motions of the system: (i) up-and-down linear motion of the body, and (ii) pitching angular motion of the body.

The body is idealized as a lumped mass with weight, W , and radius of gyration, r , as shown in Fig. 8.80. The equivalent finite element model is depicted in Fig. 8.81. The numerical values of the geometric parameters used in Fig. 8.80 are as follows: $l_1=4.5$ ft, $l_2=5.5$ ft, and $r=4$ ft; the gravitational acceleration is $g=32.2$ ft/s².

Fig. 8.80 Schematic of the automobile suspension system

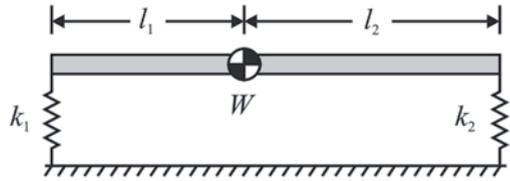


Fig. 8.81 Finite element method model of the automobile suspension system

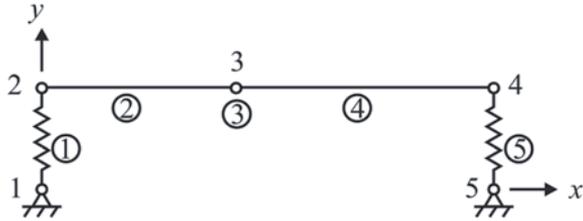


Table 8.6 Node numbers and coordinates of the automobile suspension system model

Node number	x	y
1	0.0	0.0
2	0.0	1.0
3	4.5	1.0
4	10.0	1.0
5	10.0	0.0

The corresponding coupled frequencies, f_1 and f_2 , are to be determined. The elastic modulus of the beam is $E=4 \times 10^9$ psf, the lumped weight is $W=3220$ lb, the spring constants are $k_1=2400$ lb/ft and $k_2=2600$ lb/ft. The nodal coordinates and element properties and connectivity are given in Tables 8.6 and 8.7, respectively.

Model Generation

- Define the element types (**ET** command) using the following menu path, which brings up the *Element Types* dialog box:

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

- Click on *Add*.
- Specify element type reference number 1 as **BEAM188** (*Beam* in the left list immediately below *Structural Mass*; *2 node 188* in the right list); click on **OK**.
- Click on *Add* for element type reference number 2.
- Specify element type reference number 2 as **COMBIN14** (*Combination* in the left list, *Spring-damper 14* in the right list); click on **OK**.

Table 8.7 Finite element method model of the automobile suspension system

Element number	Attributes			Nodes	
	Element type reference number	Real const. & sections number	Material reference number	1	2
1	2	Real-1	1	1	2
2	1	Sect-1	1	2	3
3	3	Real-2	1	3	
4	1	Sect-2	1	3	4
5	2	Real-3	1	4	5

- Highlight **COMBIN14** entry by clicking on it; click on *Options*.
 - Select *2-D longitudinal* for **K3**; click on *OK*.
 - Click on *Add* for element type reference number 3.
 - Specify element type reference number 3 as **MASS21** (*Structural Mass* in the left list, *3D mass 21* in the right list).
 - Highlight **MASS21** entry by clicking on it; click on *Options*.
 - Select *2-D w rot inert* for **K3**; click on *OK*.
 - Click on *Close*.
- Specify the real constants (**R** command) using the following menu path:

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete

- Click on *Add*.
 - Specify the left spring constant (k_1) as real constant set number 1:
 - Highlight *Type 2 COMBIN14* entry by clicking on it; click on *OK*.
 - Enter **2400** for **K**; click on *OK*.
 - Specify the mass properties as real constant set number 2:
 - Click on *Add*.
 - Highlight *Type 3 MASS21* entry by clicking on it; click on *OK*.
 - Enter **100** for **MASS** and **1600** for **IZZ**; click on *OK*.
 - Specify the right spring constant (k_2) as real constant set number 3:
 - Highlight *Type 2 COMBIN14* entry by clicking on it; click on *OK*.
 - Enter **2600** for **K**; click on *OK*.
 - Click on *Close*.
- Specify geometry for the beam (**SECTYPE** command) using the following menu path:

Main Menu > Preprocessor > Sections > Beam > Common Sections

- *Beam Tool* dialog box appears; enter **1** for **B** and **1** for **H**; click on *Apply*.
 - Repeat the same procedure for **ID 2**. When done, click on *OK* (instead of *Apply*).
- Specify material properties for the beam (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**, which brings up another dialog box.
- Enter **4e9** for **EX**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create the nodes (**N** command) using the following menu path, which brings up the *Create Nodes in Active Coordinate System* dialog box:

Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS

- A total of 5 nodes will be created.
- Referring to Table 8.6, enter the **x**- and **y**-coordinates of node 1; click on **Apply**. This action will keep the *Create Nodes in Active Coordinate System* dialog box open. If the **Node number** field is left blank, then ANSYS will assign the lowest available node number to the node that is being created.
- Repeat the same procedure for the nodes 2, 3 and 4.
- After creating node 5, click on **OK** (instead of **Apply**).
- Create the elements (the element attributes and connectivity information are given in Table 8.7).
 - Referring to Table 8.7, element 1 is a spring element (*type 2*) with spring stiffness 2400 lb/ft (*real constant set number 1*), composed of nodes 1 and 2. By default, the *element attributes* are set to 1. Before element 1 is created, the element type attribute needs to be specified as 2. Perform this operation by using the following menu path, which will bring up the *Element Attributes* dialog box:

Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes

- Select **2 COMBIN14** in the **[TYPE] Element type number** pull-down menu; click on **OK**. Any element that is created after this point will have these attributes (i.e., element type number 2, material number 1, and real constant set number 1).
- Create element 1 (**E** command) using the following menu path, which brings up a *Pick Menu*:

Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru Nodes

- Pick (by clicking in the *Graphics Window*) nodes 1 and 2; click on **OK** in the *Pick Menu*.

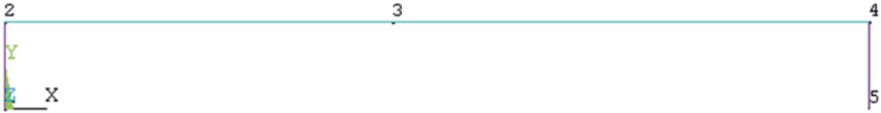


Fig. 8.82 Elements of the automobile suspension system model

LIST ALL SELECTED ELEMENTS. (LIST NODES)									
ELEM	MAT	TYP	REL	ESV	SEC	NODES			
1	1	2	1	0	1	1	2		
2	1	1	1	0	1	2	3	0	
3	1	3	2	0	1	3			
4	1	1	2	0	2	3	4	0	
5	1	2	3	0	1	4	5		

Fig. 8.83 Listing of the elements

- Create elements 2 through 5 in the same manner, each time setting the correct attributes as explained above.
- The elements should appear in the *Graphics Window*, as shown in Fig. 8.82.
- The commands equivalent to changing attributes are: **TYPE**, **MAT**, **REAL**, and **SECNUM**
- Obtain and review the list of elements by using the following menu path:

Utility Menu > List > Elements > Nodes + Attributes

Observe attributes (element type, real constants, material properties, and section numbers) and node numbers for each element (Fig. 8.83).

Solution

- Specify the *Analysis Type* as *Modal*, using the following menu path (**ANTYPE** command), which will bring up the *New Analysis* dialog box:

Main Menu > Solution > Analysis Type > New Analysis

- Click on the *Modal* radio-button; click on **OK**.

- Expand the first mode (**MXPAND** command) using the following menu path:

Main Menu > Solution > Load Step Opts > ExpansionPass > Single Expand > Expand Modes

- Enter *1* for *NMODE*; click on **OK**.

- Print two reduced mode shapes (**MODOPT** command) using the following menu path:

Main Menu > Solution > Analysis Type > Analysis Options

- Click on the *Reduced* radio-button; click on **OK**, which brings up another dialog box.
- Enter **2** for **PRMODE**; click on **OK**.

- Specify the master degrees of freedom (**M** command) using the following menu path:

Main Menu > Solution > Master DOFs > User Selected>Define

- After picking node 3, click on **OK** in the *Pick Menu*, which brings up the *Define Master DOFs* dialog box.
- Select **UY** for **Lab1** and **ROTZ** for **Lab2–6**; click on **OK**.

- Specify boundary conditions (**D** command):

- Constrain *x*- and *y*-displacements at nodes 1 and 5 using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- Pick nodes 1 and 5; click on **OK** in the *Pick Menu*, which brings up the *Apply U, ROT on Nodes* dialog box.
- Highlight **UX** and **UY**; click on **OK**.
- Constrain *x*-displacement at node 3 in the same manner.

- Instruct ANSYS to print the solution at every *substep* using the following menu path (**OUTPR** command):

Main Menu > Solution > Load Step Opts > Output Ctrl's > Solu Printout

- Select **Nodal DOF solu** from the first pull-down menu.
- Click on the *Every substep* radio-button; click on **OK**.

- Obtain the solution using the following menu path (**SOLVE** command):

Main Menu > Solution > Solve > Current LS

- Close the */STATUS information window*.
- Click on **OK** to start the solution.
- If a dialog box with a message reminding the user about previous warnings issued appears, click on **Yes**.
- Wait until ANSYS responds with **Solution is done!**

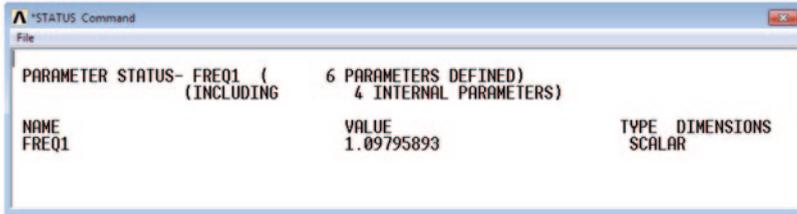


Fig. 8.84 First mode frequency for the automobile suspension system model

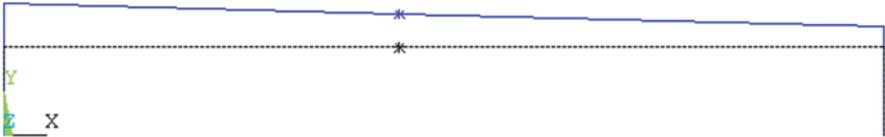


Fig. 8.85 First mode shape for the automobile suspension system model

Postprocessing

- Retrieve the coupled frequencies and store them in user-defined parameters using the following menu path:

Utility Menu > Parameters > Get Scalar Data

- Highlight *Results data* in the left list, *Modal results* in the right list; click on **OK**, which brings up another dialog box.
- Enter *freq1* for *Name of parameter to be defined* field, and enter *1* for *Mode number N*; click on **OK**.
- Repeat the procedure above to define parameter *freq2* corresponding to *Mode 2*.

- List the parameters defined above using the following menu path:

Utility Menu > List > Other > Named Parameter

- Highlight *freq1* or *freq2* from the list of parameters; click on **OK**.
- Figure 8.84 shows the outcome of this action for *freq1*.

- Plot the first mode shape.

- Set the solution to *First Set* using the following menu path:

Main Menu > General Postproc > Read Results > First Set

- Plot the mode shape using the following menu path:

Main Menu > General Postproc > Plot Results > Deformed Shape

- Click on *Def+ undeformed* radio-button; click on **OK**.
- The mode shape is shown in Fig. 8.85 as it appears in the *Graphics Window*.

8.5.2 Harmonic Analysis

When a structure is subjected to cyclic loading, the resulting response is expected to be cyclic as well. ANSYS provides the user with the capability to solve this class of problems through the *Harmonic* analysis option. The restrictions on a harmonic analysis are:

- All loads must be sinusoidal functions of time.
- All loads must have the same frequency.
- The structure must exhibit linearly elastic behavior (no geometric and material nonlinearities).

Sinusoidal loads are specified through the parameters *amplitude*, *phase angle*, and *forcing frequency range*. *Amplitude* is the peak value of the load, and *phase angle* is the time lag between multiple loads that are out of phase with each other. On the complex plane, it is the angle measured from the real axis. Finally, *forcing frequency range* is the frequency range of the harmonic load (in cycles/time).

8.5.2.1 Harmonic Analysis of a Bracket

A bracket, shown in Fig. 8.29, experiences a harmonic (cyclic) loading at two points of application. The forcing frequency varies from 0 to 400 Hz; however, the two loads are 120° out of phase, i.e., $F_1 = 1.5 \sin \omega t$ and $F_2 = 1.5 \sin(\omega - 120)t$. The amplitude of each load is 1.5 lb. The bracket is clamped at its two upper bolt holes. The structural damping ratio is 0.03. The mode superposition method is used to calculate the harmonic response. This method requires a modal analysis to be performed first. A reduced modal analysis is used, so the harmonic analysis results in a reduced solution, which is then expanded using *Expansion Pass*. Note that the phase angle input is crucial for the expansion pass. *Time History Postprocessing* of the reduced analysis reveals the appropriate phase for expansion.

Model Generation

The finite element model of this bracket was created in Sect. 8.1.5.1 (modeling of a bracket). Therefore, the model will not be regenerated here; it will be generated using the input file *bracket.inp*. It is worth noting that the components consisting of the nodes along the holes are defined in the input file.

- Specify the *jobname* as *br_harm* using the following menu path:

Utility Menu > File > Change Jobname

- In the dialog box, type *br_harm* in the *[/FILNAM] Enter new jobname* text field; click on the checkbox for *New log and error files* to show *Yes*; click on **OK**.

- Create the model by reading the input file **bracket.inp** from the CD-ROM location/*Input_Files\ch08\bracket.inp* using the following menu path:

Utility Menu > File > Read Input from

- Browse for the **bracket.inp** from the folder given above; click on **OK**.
- Wait until the model is generated.

Solution

- Specify the *Analysis Type* as *Modal*, using the following menu path (**ANTYPE** command), which brings up the *New Analysis* dialog box:

Main Menu > Solution > Analysis Type > New Analysis

- Click on **Modal** radio-button; click on **OK**.

- Specify the *reduced modal analysis* method using the following menu path:

Main Menu > Solution > Analysis Type > Analysis Options

- Click on the **Reduced** radio-button; click on **OK**.
- A new dialog box appears; click on **OK**.

- Constrain displacement and rotation degrees of freedom along the top-left and -right holes. For this purpose, first select the components created earlier for these holes (**TL_BOLT** and **TR_BOLT**) using the following menu path:

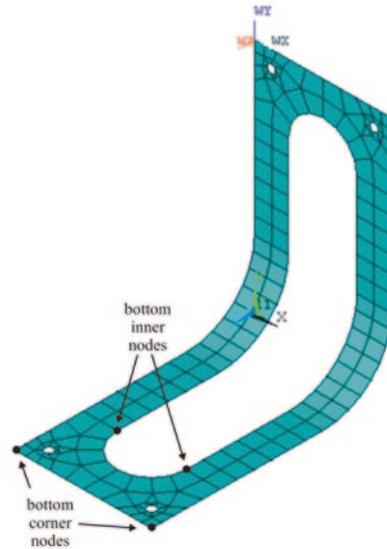
Utility Menu > Select > Comp/Assembly > Select Comp/Assembly

- A dialog box appears; click on the **by component name** radio-button; click on **OK**.
- A new dialog box with the components listed appears; highlight **TL_BOLT**; click on **OK**. This action selects the nodes along the top-left hole.
- Specify the displacement boundary conditions using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; click on **Pick All**.
- In the new dialog box, highlight **All DOF**; click on **OK**.
- Repeat the same procedure for the top-right hole (**TR_BOLT**).
- Specify the master degrees of freedom at the four nodes of interest (as shown in Fig. 8.86) and specify that an additional 40° of freedom are to be selected automatically by ANSYS.
 - Select the bottom corner nodes using the following menu path:

Fig. 8.86 Nodes at which master degrees of freedom are specified



Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose **Nodes** from the first pull-down menu and choose **By Location** in the second pull-down menu; click on the radio-button for **Z coordinates** and type **5** in the **Min, Max** text field; click on **Apply**.
- In the same dialog box (*Select Entities* dialog box), click on the radio-button for **X coordinates** and type **-1.99, 1.99** in the **Min, Max** text field; click on the radio-button for **Unselect**; click on **OK**.
- Specify the master degrees of freedom using the following menu path:

Main Menu > Solution > Master DOFs > User Selected>Define

- Click on **Pick All** in the *Pick Menu*, which brings up the *Define Master DOFs* dialog box.
- Select **All DOF** for **Lab1**; click on **OK**.
- Select bottom inner nodes using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose **Nodes** from the first pull-down menu and choose **By Location** in the second pull-down menu; click on the radio-button for **X coordinates** and type **-1.01, 1.01** in the **Min, Max** text field; click on the radio-button for **From Full**; click on **Apply**.
- In the same dialog box (*Select Entities* dialog box) click on the radio-button for **Z coordinates** and type **3** in the **Min, Max** text field; click on the radio-button for **Reselect**; click on **OK** (the left node among the two is referred to as *bottom inner left node* in the postprocessing).
- Specify the master degrees of freedom using the following menu path:

Main Menu > Solution > Master DOFs > User Selected>Define

- Click on *Pick All* in the *Pick Menu*, which brings up the *Define Master DOFs* dialog box.
- Select *All DOF* for *Lab1*; click on *OK*.
- Select everything using the following menu path:

Utility Menu > Select > Everything

- Specify additional 40 master degrees of freedom using the following menu path:

Main Menu > Solution > Master DOFs > Program Selected

- In the dialog box, type *40* for *NTOT Total no. of master DOF*; click on the checkbox to show *Yes* for *NR MDF Exclude rotational DOF*; click on *OK*.
- Save the model using the following menu path:

• **Utility Menu > File > Save as Jobname.db**

- Obtain the modal solution using the following menu path:

Main Menu > Solution > Solve > Current LS

- Close the */STATUS information window*.
- Click on *OK* to start the solution.
- If a dialog box with a message reminding the user about previous warnings issued appears, click on *Yes*.
- Wait until ANSYS responds with *Solution is done!*

- Exit the *Solution Processor* using the following menu path:

Main Menu > Finish

- Specify the *Analysis Type* as *Harmonic*, using the following menu path, which bring up the *New Analysis* dialog box:

Main Menu > Solution > Analysis Type > New Analysis

- Click on *Harmonic* radio-button; click on *OK*.
- A *Warning Window* appears; click on *OK*.

- Specify the *Mode Superposition* method using the following menu path:

Main Menu > Solution > Analysis Type > Analysis Options

- Select *Mode superpos'n* from the *Solution method* pull-down menu; click on *OK*.
- *Mode Sup Harmonic Analysis* dialog box appears; click on radio-button for *Cluster at modes*; click on *OK*.

- Apply force at bottom-left and -right corner nodes.

- Select bottom-left corner node using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose **Nodes** from the first pull-down menu and choose **By Location** in the second pull-down menu; click on the radio-button for **Z coordinates** and type **5** in the **Min, Max** text field; click on the **From Full** radio-button; click on **Apply**.
- In the same dialog box (*Select Entities* dialog box) click on the radio-button for **X coordinates** and type **-2** in the **Min, Max** text field; click on the **Reselect** radio-button; click on **OK**.
- Define scalar parameter using the following menu path:

Utility Menu > Parameters > Scalar Parameters

- In the text field within the dialog box, type **AMPL=1.5**; click on **Accept**. Close the *Scalar Parameters* dialog box by clicking on **Close**.
- Apply the force on previously selected node (bottom-left corner) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

- Click on **Pick All** in the *Pick Menu*.
- In the new dialog box, select **FY** from the first pull-down menu and type **AMPL** in the **VALUE Real part of force/mom** text field; click on **OK**.
- Select bottom-right corner node using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose **Nodes** from the first pull-down menu and choose **By Location** in the second pull-down menu; click on the radio-button for **Z coordinates** and type **5** in the **Min, Max** text field; click on the **From Full** radio-button; click on **Apply**.
- In the same dialog box (*Select Entities* dialog box) click on the radio-button for **X coordinates** and type **2** in the **Min, Max** text field; click on the **Reselect** radio-button; click on **OK**.
- Select *degrees* as the unit for angles using the following menu path:

Utility Menu > Parameters > Angular Units

- In the dialog box, select **Degrees** for **[*AFUN] Units for angular**; click on **OK**.
- Define scalar parameter using the following menu path:

Utility Menu > Parameters > Scalar Parameters

- In the text field within the dialog box:
- Type **PHASE=120**; click on **Accept**.
- Type **FR=AMPL * COS(PHASE)**; click on **Accept**.
- Type **FI=AMPL * SIN (PHASE)**; click on **Accept**.
- Click on **Close**.
- Apply the force on previously selected node (bottom-right corner) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

- Click on *Pick All* in the *Pick Menu*.
- In the new dialog box, select *FY* from the pull-down menu and type *FR* in the *VALUE Real part of force/mom* text field and type *FI* in the *VALUE2 Imag part of force/mom* text field; click on *OK*.
- Select everything using the following menu path:

Utility Menu > Select > Everything

- Specify the *harmonic frequencies* using the following menu path:

Main Menu > Solution > Load Step Opts > Time/Frequenc > Freq and Sub-steps

- Type *0* in the first text field and *400* in the second text field.
- Type *40* for *NSUBST* text field; select the *Stepped* radio-button; click on *OK*.
- Specify the *damping* using the following menu path:

Main Menu > Solution > Load Step Opts > Time/Frequenc>Damping

- Type *0.03* in the *DMPRAT* text field (third text field); click on *OK*.
- Save the model using the following menu path:

Utility Menu > File > Save as Jobname.db

- Obtain the harmonic solution using the following menu path:

Main Menu > Solution > Solve > Current LS

- Close the *STATUS information window*.
- Click on *OK* to start the solution.
- If a dialog box with a message reminding the user about previous warnings issued appears, click on *Yes*.
- Wait until ANSYS responds with *Solution is done!*

Postprocessing

- Enter the *Time History Postprocessor* using the following menu path:

Main Menu > TimeHist Postpro

- *Time History Variables* dialog box appears. Click on the button with the green plus sign at the top-left to define a variable.
- *Add Time History Variable* dialog box appears.
- Successively click on *Nodal Solution*, *DOF Solution*, and *Y-component of displacement*; in the text field, type *UYRCNR*; click on *OK*.
- *Pick Menu* appears; pick the bottom-right corner node; click on *OK*.
- Note the new variable *UYRCNR* in *Time History Variables* dialog box.

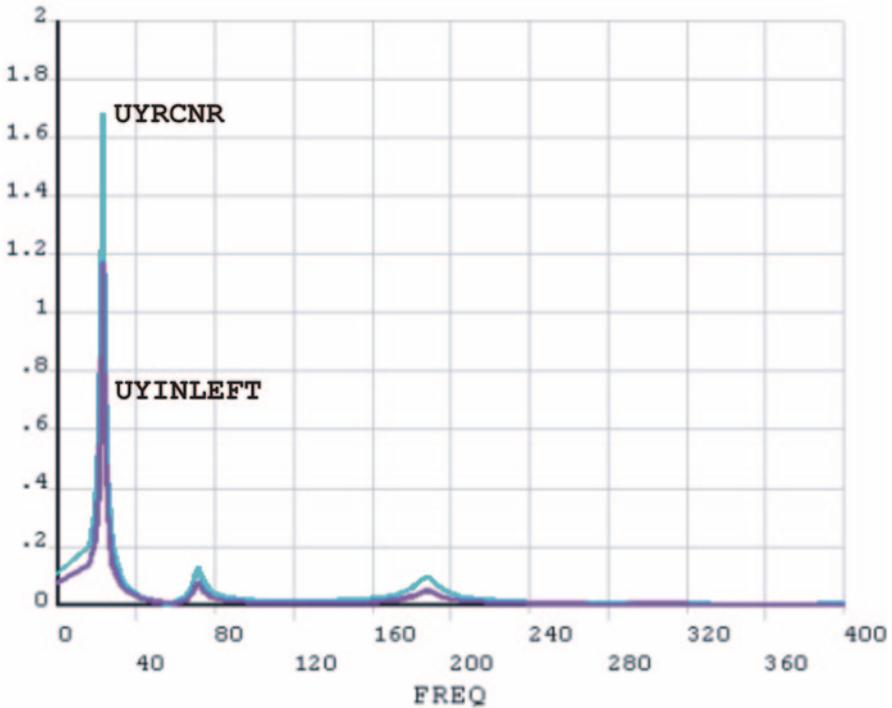


Fig. 8.87 Amplitudes of the displacements at two locations as functions of frequency under harmonic loading

- Add new variable for displacement in y -direction at the bottom inner-left node by clicking on the button with the green plus sign and successively clicking on *Nodal Solution*, *DOF Solution*, and *Y-component of displacement*; in the text field, type **UYINLEFT**; click on **OK**.
- Pick the bottom inner-left node; click on **OK**.
- Note the new variable **UYINLEFT** in *Time History Variables* dialog box.
- Highlight the rows **UYRCNR** and **UYINLEFT** from the list (by pressing **Ctrl** on the keyboard and clicking on the rows with left mouse button); click on the third from left button to plot the variation of these displacements.
- The plot appears in the *Graphics Window*, as shown in Fig. 8.87.
- Change the frequency axis (x -axis) limits in order to take a closer look at the first peak using the following menu path:

Utility Menu > PlotCtrls > Style > Graphs > Modify Axes

- In the dialog box, under *[XRANGE]*, select the radio button for *Specified range* and type **16** and **37** in the *XMIN*, *XMAX Specified X range* text fields; click on **OK**.
- Obtain the plot with the new range using the following menu path:

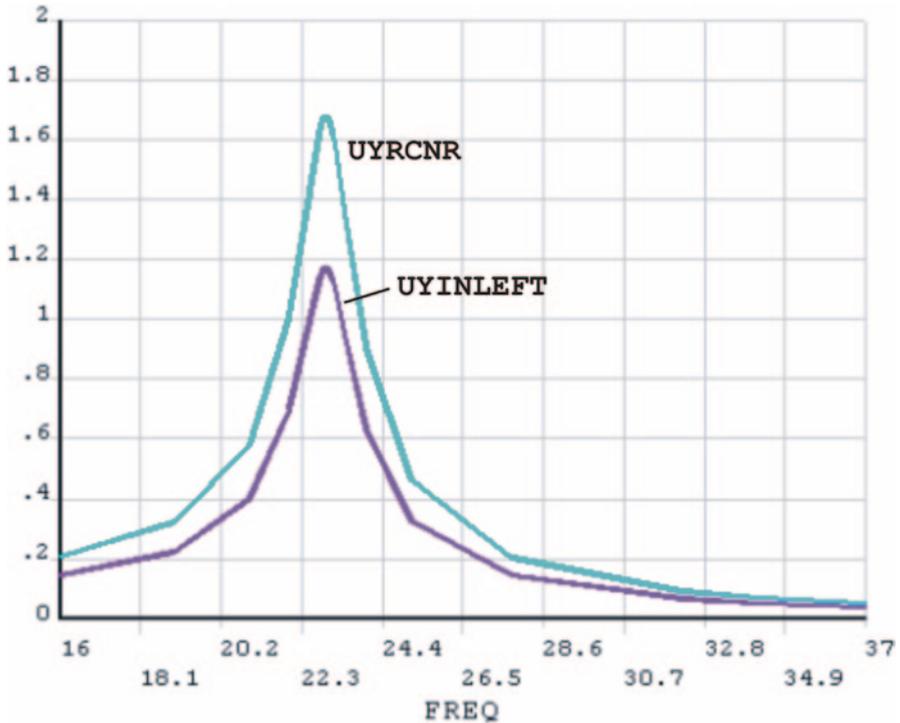


Fig. 8.88 Close-up view of amplitudes of the displacements at two locations as functions of frequency under harmonic loading

Utility Menu > Plot > Replot

- The plot appears in the *Graphics Window*, as shown in Fig. 8.88.
- Reset the frequency axis (x -axis) limits using the following menu path:

Utility Menu > PlotCtrls > Style > Graphs > Modify Axes

- In the dialog box, under *[XRANGE]*, select the radio button for *Auto calculated*; click on *OK*.
- In order to plot the phase angles, change the settings of the time history graphing using the following menu path:

Main Menu > TimeHist Postpro > Settings > Graph

- Select *Phase Angle* from the pull-down menu for *[PLCPLX] Complex variable*; click on *OK*.
- Click on the third from left button in *Time History Variables* dialog box to plot the variation of these phase angles.
- The plot appears in the *Graphics Window*, as shown in Fig. 8.89.

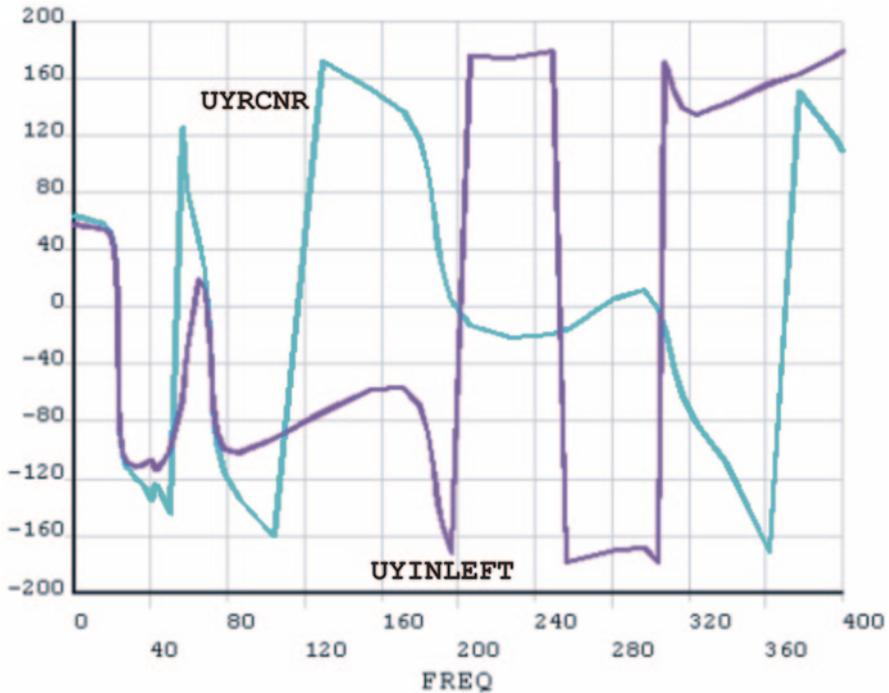


Fig. 8.89 Phase angles of the displacements at two locations as functions of frequency under harmonic loading

8.5.2.2 Harmonic Analysis of a Guitar String

A stainless-steel guitar string of length $l=710$ mm and diameter $d=0.254$ mm is stretched between two rigid supports by a tensioning force $F_1=84$ N applied at the right end (refer to Fig. 8.90). The string is struck at a location $c=165$ mm from the left end with a force $F_2=1$ N. The elastic modulus and density of the string are $E=190$ GPa and $\rho=7920$ kg/m³, respectively. The fundamental frequency f_1 is determined using truss elements (**LINK180**), and a *Mode Superposition Harmonic Response Analysis* is performed. In order to perform the harmonic analysis, static and modal analyses are first performed.

Model Generation

- Specify the *jobname* as **guitar** using the following menu path:

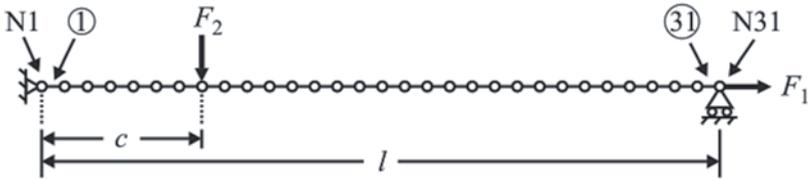


Fig. 8.90 Finite element model of a guitar string

Utility Menu > File > Change Jobname

- In the dialog box, type *guitar* in the *[/FILNAM] Enter new jobname* text field; click on the checkbox for *New log and error files* to show *Yes*; click on **OK**.
- Define element types using the following menu path, which brings up the *Element Types* dialog box:

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

- Click on *Add*.
- Specify *element type reference # 1* as **LINK180** (Select *Link* immediately below *Structural Mass* from the left list and *3D finit str 180* from the right list); click on **OK**.
- Click on *Close*.
- Specify real constants using the following menu path:

Main Menu > Preprocessor > Real Constants > Add/Edit/Delete

- Click on *Add*.
- Specify the cross-sectional area as real constant set number 1:
- Highlight *Type 1 LINK180* entry by clicking on it; click on **OK**.
- Enter **50671E-12** for *Cross-sectional area AREA*; click on **OK**.
- Click on *Close*.
- Specify material properties for the string using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on *Structural*, *Linear*, *Elastic*, and, finally, *Isotropic*, which will bring up another dialog box.
- Enter **190e9** for *EX*; click on **OK**.
- In the *Define Material Model Behavior* dialog box, in the right window, left-click on *Density*, which will bring up another dialog box.
- Enter **7920** for *DENS*; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

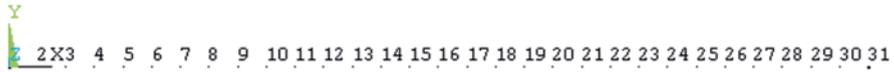


Fig. 8.91 Nodes of the guitar string finite element model



Fig. 8.92 Elements of the guitar string finite element model

Material > Exit

- Create nodes using the following menu path, which brings up the *Create Nodes in Active Coordinate System* dialog box:

Main Menu > Preprocessor > Modeling > Create > Nodes > In Active CS

- Type *1* for *NODE Node number*; type *0* for *x-* and *y-*coordinates; click on *Apply*.
- In the same dialog box, type *31* for *NODE Node number*; type *0.71* for *x-*coordinate and *0* for *y-*coordinate; click on *OK*.
- Fill between nodes using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Nodes > Fill Between Nds

- *Pick Menu* appears; pick the two nodes; click on *OK*.
- A new dialog box appears; click on *OK*.
- The nodes should appear in the *Graphics Window*, as shown in Fig. 8.91.

- Create elements using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Thru Nodes

- *Pick Menu* appears; pick the first two nodes (nodes 1 and 2); click on *OK*.
- Create the rest of the elements by copying element 1 using the following menu path:

Main Menu > Preprocessor > Modeling > Copy > Elements > Auto Numbered

- *Pick Menu* appears; pick the element; click on *OK*.
- A new dialog box appears; type *30* for *ITIME Total number of copies*; click on *OK*.
- The elements should appear in the *Graphics Window*, as shown in Fig. 8.92.

Static Solution

- Specify the *Analysis Type* as *Static*, using the following menu path, which will bring up the *New Analysis* dialog box:

Main Menu > Solution > Analysis Type > New Analysis

- Click on **Static** radio-button; click on **OK**.
- Specify boundary conditions.
 - Constrain *x*- and *y*-displacements at node 1, and *y*-displacement at the remaining nodes using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- Pick node 1; click on **OK**.
- Highlight **All DOF**; click on **Apply**.
- Click on the radio-button for **Box** in the *Pick Menu*; select all the nodes except node 1; click on **OK**.
- Highlight **UY**, **remove** the highlight on **All DOF** by clicking on it; click on **OK**.
- Apply the axial force using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

- Pick the last node (node 31); click on **OK**.
- In the new dialog box, select **FX** from the pull-down menu and type **84** for **VALUE Force/moment value**; click on **OK**.
- Specify this analysis to be a *prestressed* analysis using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- A dialog box with tabs appears. By default, the **Basic** tab is active.
- On the left side, under **Analysis Options**, put a checkmark for **Calculate prestress effects**.
- On the right side, under **Write Items to Results File**, click on the radio-button for **All solution items** and select **Write every substep** from the **Frequency** pull-down menu.
- Click on **OK**.
- Save the model using the following menu path:

Utility Menu > File > Save as Jobname.db

The model will be saved in the *Working Directory* under the name *guitar.db*.

- Obtain the solution using the following menu path:

Main Menu > Solution > Solve > Current LS

- Close the */STATUS* information window.
- Click on **OK** to start the solution.
- Wait until ANSYS responds with ***Solution is done!***
- Exit the *Solution Processor* using the following menu path:

Main Menu > Finish**Modal Solution**

- Specify the *Analysis Type* as *Modal*, using the following menu path, which will bring up the *New Analysis* dialog box:

Main Menu > Solution > Analysis Type > New Analysis

- Click on ***Modal*** radio-button; click on **OK**.
- A *Warning Window* appears; click on **OK**.
- Specify *analysis options* using the following menu path:

Main Menu > Solution > Analysis Type > Analysis Options

- Click on the ***Block Lanczos*** radio-button; type **6** for *No. of modes to extract* (first text field).
- Place a checkmark for ***[PSTRES] Incl prestress effects***; click on **OK**, which brings up another dialog box.
- Click on **OK**.
- Modify boundary conditions.
 - Delete constraints in the *y*-direction at nodes 2 through 30 (leaving the displacement constraints at nodes 1 and 31 intact) using the following menu path:

Main Menu > Solution > Define Loads > Delete > Structural > Displacement > On Nodes

- Click on the radio-button for ***Box*** in the *Pick Menu*; select all the nodes except nodes 1 and 31; click on **OK**.
- Select ***UY*** from the pull-down menu; click on **OK**.
- Save the model using the following menu path:

Utility Menu > File > Save as Jobname.db

The model will be saved in the *Working Directory* under the name ***guitar.db***.

- Obtain the solution using the following menu path:

Main Menu > Solution > Solve > Current LS

- Close the */STATUS* information window.
- Click on **OK** to start the solution.
- Wait until ANSYS responds with ***Solution is done!***
- Exit the *Solution Processor* using the following menu path:

Main Menu > Finish**Harmonic Solution**

- Specify the *Analysis Type* as *Harmonic*, using the following menu path, which will bring up the *New Analysis* dialog box:

Main Menu > Solution > Analysis Type > New Analysis

- Click on ***Harmonic*** radio-button; click on **OK**.
- A *Warning Window* appears; click on **OK**.
- Specify the *mode superposition* method using the following menu path:

Main Menu > Solution > Analysis Type > Analysis Options

- Select ***Mode superpos'n*** from the **[HROPT] Solution method** pull-down menu.
- Select ***Amplitud + phase*** from the **[HROUT] DOF printout format** pull-down menu; click on **OK**.
- Click on **OK** in the new dialog box.
- Modify the boundary conditions.
 - Delete the axial force using the following menu path:

Main Menu > Solution > Define Loads > Delete > Structural > Force/Moment > On Nodes

- Pick the last node (node 31); click on **OK**.
- In the new dialog box, select ***FX*** from the pull-down menu; click on **OK**.
- Apply the vertical force using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

- Pick node 8; click on **OK**.
- In the new dialog box, select ***FY*** from the pull-down menu and type ***-I*** for ***VALUE Real part of force/mom***; click on **OK**.
- Specify the *harmonic frequencies* and enforce step loading using the following menu path:

Main Menu > Solution > Load Step Opts > Time/Frequenc > Freq and Sub-steps

- Type **0** in the first text field and **2000** in the second text field for **HARFRQ**.
- Type **250** for **NSUBST** text field; select the **Stepped** radio-button; click on **OK**.

- Save the model using the following menu path:

Utility Menu>File>Save as Jobname.db

The model will be saved in the *Working Directory* under the name **guitar.db**.

- Obtain the solution using the following menu path:

Main Menu > Solution > Solve > Current LS

- Close the */STATUS information window*.
- Click on **OK** to start the solution.
- Wait until ANSYS responds with **Solution is done!**

- Exit the *Solution Processor* using the following menu path:

Main Menu > Finish

Postprocessing

- Enter the *Time History Postprocessor* using the following menu path:

Main Menu > TimeHist Postpro

- *Time History Variables* dialog box appears. Click on the button with the green plus sign at the top-left to define a variable.
- *Add Time History Variable* dialog box appears.
- Successively click on the items **Nodal Solution**, **DOF Solution**, and **Y-component of displacement**; click on **OK**.
- *Pick Menu* appears; pick the midpoint node (node 16); click on **OK**.
- Note the new variable **UY_2** in *Time History Variables* dialog box.
- Highlight the row **UY_2** from the list; click on the third from left button to plot the variation of these displacements.
- The plot appears in the *Graphics Window*, as shown in Fig. 8.93.

8.5.3 Transient Analysis

Practical engineering problems having a non-cyclic transient loading can be solved using the *Transient* analysis in ANSYS. Typically, the following steps are used for a *Transient* analysis:

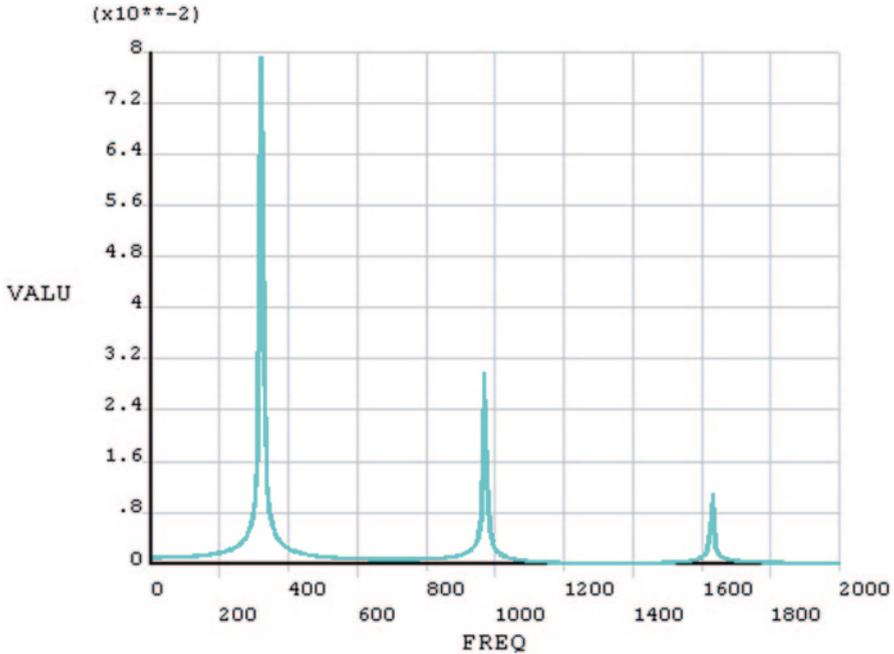


Fig. 8.93 Amplitudes of the displacement at the midpoint node as a function of frequency under harmonic loading

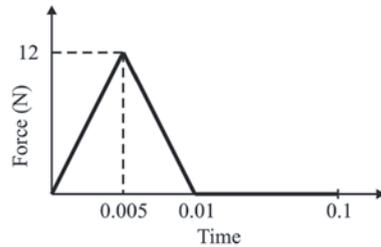
Build the model.
 Specify initial conditions.
 Specify *Solution Controls*.
 Apply loads.
 Write *Load Step File*.
 Apply/change loads and *Solution Controls* for the next *Load Step(s)* and write *Load Step File(s)*.
 Obtain solution from *Load Step Files*.
 Review the Results.

Solution Controls are specified using the *Solution Controls* dialog box, which is accessed through the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

There are five tabs in the *Solution Controls* dialog box. The first two tabs, **Basic** and **Transient**, are sufficient for most transient analyses. The **Basic** tab contains options to specify nonlinear geometry (large deformation), time at the end of the load step, load step and substep numbers and sizes, automatic time stepping, and the amount and frequency of results data to be saved. The **Transient** tab involves options related to time integration and damping specifications.

Fig. 8.94 Time-dependent loading on the bracket



8.5.3.1 Dynamic Analysis of a Bracket

The bracket shown in Fig. 8.29 is clamped at the two top holes and subjected to an impact load at the bottom-left hole. The load is 12 lb, in the positive z -direction, and is applied for 0.01 s. Three load steps with ramped loading are used; as shown in Fig. 8.94, the times at the end of the first, second, and third load steps are 0.005, 0.01, and 0.1 s, respectively, with a time step size of $1\text{E}-4$ s. The objective is to obtain the time-dependent response of the bracket.

Model Generation

The finite element model of this bracket was created in Sect. 8.1.5.1 (modeling of a bracket). Therefore, the model will not be regenerated here; it will be generated using an input file. It is worth noting that the components consisting of the nodes along the holes are defined in the input file.

- Create the model by reading the input file ***bracket.inp*** from the CD-ROM location/*Input_Files\ch08\bracket.inp* using the following menu path:

Utility Menu > File > Read Input from

- Browse for the ***bracket.inp*** from the folder given above; click on ***OK***.
- Wait until the model is generated.

Solution

There are three distinct loading regiments: (1) from zero loading to maximum loading, (2) from maximum loading to complete removal of the load, and (3) time period after load removal during which the transient response of the bracket is sought. Each of these distinct periods will be defined as a *Load Step*. Each load step will be written to a load step file prior to solution. Once all the load step files are written, the solution will be obtained using these files. Displacement constraints do not change throughout the analysis; they will be specified only once at the beginning.

- Specify the *Analysis Type* as *transient* using the following menu path:

Main Menu > Solution > Analysis Type > New Analysis

- Click on *Transient*; click on *OK*.
- A new dialog box appears; click on *OK*.
- Constrain displacement and rotation degrees of freedom along the top-left and -right holes. For this purpose, first select the components created earlier for these holes (*TL_BOLT* and *TR_BOLT*) using the following menu path:

Utility Menu > Select > Comp/Assembly > Select Comp/Assembly

- A dialog box appears; click on the *by component name* radio-button; click on *OK*.
- A new dialog box with the components listed appears; highlight *TL_BOLT*; click on *OK*. This action selects the nodes along the top-left hole.
- Specify the displacement boundary conditions using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- *Pick Menu* appears; click on *Pick All*.
- In the new dialog box, highlight *All DOF*; click on *OK*.
- Repeat the same procedure for the top-right hole (*TR_BOLT*).
- Apply the load at the bottom-left hole.
 - For this purpose, create a local coordinate system at the center of the bottom-left hole using the following menu path:

Utility Menu > WorkPlane > Local Coordinate Systems > Create Local CS > At Specified Loc

- *Pick Menu* appears; type *-1.5, -2, 4.5* in the text field in the *Pick Menu*; click on *OK*.
- A dialog box appears; select *Cylindrical 1* in the *KCS Type of coordinate system* pull-down menu and type *-90* in the *THYZ Rotation about local X* text field; click on *OK*.
- Select the keypoints along the bottom-left hole using the following menu path:

Utility Menu > Select > Entities

- *Select Entities* dialog box appears; choose *Keypoints* in first pull-down menu; choose *By Location* in the second pull-down menu; type *0.25/2* in the *Min, Max* text field; click on *OK*.
- Apply forces on the keypoints using the menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Keypoints

- *Pick Menu* appears; click on *Pick All*.

- In the new dialog box, select **FZ** from pull-down menu and enter **12/4** for **Force/moment value**; click on **OK**.
- Select everything using the following menu path:

Utility Menu > Select > Everything

- Set solution options for the first load step using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- *Solution Controls* dialog box appears. By default, the **Basic** tab is active.
- On the left side, under **Time Control**, type **0.005** for **Time at end of loadstep**; select **On** from **Automatic time stepping** pull-down menu.
- Click on the radio-button for **Time increment** and type **0.0001** in **Time step size** text field.
- On the right side, under **Write Items to Result File**, click on the radio-button for **All solution items** and select **Write every substep** from the **Frequency** pull-down menu.
- Activate the **Transient** tab (by clicking on it).
- Under **Full Transient Options**, click on the radio-button for **Ramped loading**; click on **OK** to close the *Solution Controls* dialog box.

- Write first *load step file* using the following menu path:

Main Menu > Solution > Load Step Opts > Write LS File

- Type **1**; click on **OK**.
- The first load step is written to a file.

- Remove the load at the midpoint node using the following menu path:

Main Menu > Solution > Define Loads > Delete > Structural > Force/Moment > On Keypoints

- Click on **Pick All**.
- Select **FZ** from the pull-down menu; click on **OK**.

- Set *solution options* for the second load step using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- Activate the **Basic** tab. On the left side, under **Time Control**, type **0.01** for **Time at end of loadstep**, leave the other options unchanged.
- Click on **OK** to close the *Solution Controls* dialog box.

- Write the second *load step file* using the following menu path:

Main Menu > Solution > Load Step Opts > Write LS File

- Type **2**; click on **OK**.
- The second load step is written to a file.

- Set solution options for the third load step using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- On the left side, under *Time Control*, type *0.1* for *Time at end of loadstep*; leave the other options unchanged.
- Click on **OK** to close the *Solution Controls* dialog box.

- Write the third *load step file* using the following menu path:

Main Menu > Solution > Load Step Opts > Write LS File

- Type *3*; click on **OK**.
- The third load step is written to a file.

- Obtain the solution using the following menu path:

Main Menu > Solution > Solve > From LS Files

- *Solve Load Step Files* dialog box appears.
- Type *1* for *LSMIN Starting LS file number* and *3* for *LSMAX Ending LS file number*; click on **OK**.
- Wait until ANSYS responds with *Solution is done!*

Postprocessing

- Review time-dependent behaviors of *y*-displacement at the bottom-left and -right corner nodes using the following menu path:

Main Menu > TimeHist Postpro

- *Time History Variables* dialog box appears.
- Click on the button with the green plus sign at the top-left to define a variable.
- *Add Time History Variable* dialog box appears.
- Successively click on the items *Nodal Solution*, *DOF Solution*, and *Y-component of displacement*; click on **OK**.
- Pick Menu appears; pick the bottom-left corner node; click on **OK**.
- Note the new variable *UY_2* in Time History Variables dialog box.
- Add new variable for displacement in *y*-direction at the bottom-right corner node by clicking on the button with the green plus sign and successively clicking on the items *Nodal Solution*, *DOF Solution*, and *Y-component of displacement*; click on **OK**.
- Pick the bottom-right corner node; click on **OK**.
- Note the new variable *UY_3* in Time History Variables dialog box.
- Highlight the rows *UY_2* and *UY_3* from the list (by pressing **Ctrl** on the keyboard and clicking on the rows with left mouse button); click on the third from left button to plot the time variation of these displacements.
- The plot appears in the *Graphics Window*, as shown in Fig. 8.95.
- Close *Time History Variables* dialog box.

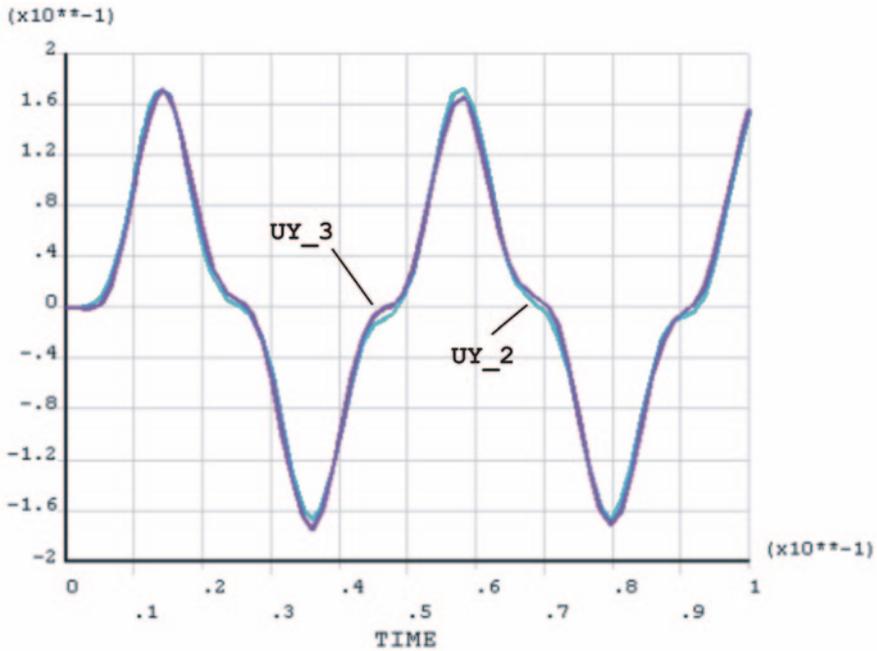


Fig. 8.95 Time variation of displacements in the y -direction at two locations

8.5.3.2 Impact Loading on a Beam

A 3-ft-long steel beam is clamped at both ends and subjected to an impact loading of 100 lbf that lasts 0.0002 s at the midpoint (Fig. 8.96). It is assumed that the loading increases linearly from 0 to 100 lbf in 0.0001 s, and decreases in the same manner, as demonstrated in Fig. 8.97. The beam has a square cross-sectional area (1×1 in) with material properties $E=29 \times 10^6$ psi and $\nu=0.32$. The objective is to solve for the time-dependent response of the beam for 0.5 s.

The problem is solved twice: (i) without structural damping, and (ii) with structural damping.

Model Generation

- Define the element type (**ET** command) using the following menu path:

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

- Click on *Add*.
- Select *Beam* immediately below *Structural Mass* from the left list and *2node 188* from the right list; click on *OK*.
- Click on *Close*.

Fig. 8.96 Schematic of the beam under impact loading

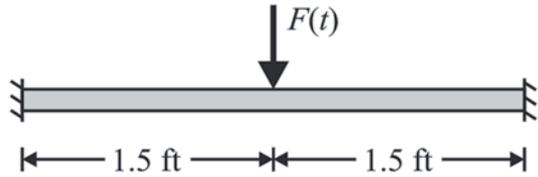
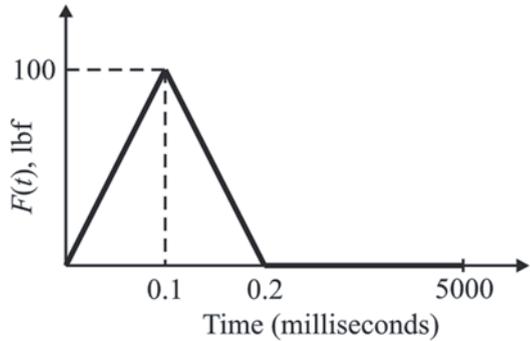


Fig. 8.97 Time-dependent loading



- Specify geometry for the beam (**SECTYPE** command) using the following menu path:

Main Menu > Preprocessor > Sections > Bea > Common Sections

- *Beam Tool* dialog box appears; enter **1** for **B** and **1** for **H**; click on **OK**.
- Exit from the *Beam Tool* dialog box by clicking on **OK**.

- Specify material properties (**MP** command) using the following menu path:

Main Menu > Preprocessor > Material Props > Material Models

- In the *Define Material Model Behavior* dialog box, in the right window, successively left-click on **Structural**, **Linear**, **Elastic**, and, finally, **Isotropic**, which will bring up another dialog box.
- Enter **29e6** for **EX** and **0.32** for **PRXY**; click on **OK**.
- In the *Define Material Model Behavior* dialog box, in the right window, left-click on **Density**, which will bring up another dialog box.
- Enter **0.2836** for **DENS**; click on **OK**.
- Close the *Define Material Model Behavior* dialog box by using the following menu path:

Material > Exit

- Create the solid model. In this problem, the solid model is a line, which is composed of two keypoints. Therefore, first create the keypoints, then the line.
 - Create keypoints using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS

- In the *Create Keypoints in Active Coordinate System* dialog box, type – **18** for **X** and **0** for **Y**; click on **Apply** (keypoint 1 is created).
- Type **18** for **X** and **0** for **Y**; click on **OK** (keypoint 2 is created)
- Create the line using the following menu path:

Main Menu > Preprocessor > Modeling > Create > Lines > Lines > Straight Line

- Pick keypoints 1 and 2; line is created; click on **OK** in the *Pick Menu*.
- Specify the number of elements on the line as **30** divisions using the following menu path:

Main Menu > Preprocessor > Meshing > Size Cntrls > ManualSize > Lines > Picked Lines

- Pick the line; click on **OK**.
- *Element Sizes on Lines* dialog box appears; type **30** in the text field corresponding to **NDIV** (the second text field), and uncheck the first checkbox; click on **Apply**.
- Create the mesh using the following menu path:

Main Menu > Preprocessor > Meshin > Mesh > Lines

- Pick the line; click on **OK**.

Solution Without Structural Damping

There are three distinct loading regiments: (1) from zero loading to maximum loading, (2) from maximum loading to complete removal of the load, and (3) time period after load removal during which the transient response of the beam is sought. Each of these distinct periods will be defined as a *Load Step*. Each load step will be written to a load step file prior to solution. Once all the load step files are written, the solution will be obtained using these files. Displacement constraints do not change throughout the analysis; they will be specified only once at the beginning.

- Specify the *Analysis Type* as *transient* using the following menu path:

Main Menu > Solution > Analysis Type > New Analysis

- Click on **Transient**; click on **OK**.
- A new dialog box appears; click on **OK**.
- Specify displacement constraints using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Displacement > On Nodes

- Pick the nodes on either end; click on **OK** in the *Pick Menu*.

- Highlight *All DOF*; click on **OK**.
- Apply the load at the midpoint node (located at the origin for picking conveniently) using the following menu path:

Main Menu > Solution > Define Loads > Apply > Structural > Force/Moment > On Nodes

- Pick the midpoint node; click on **OK**.
- Select *FY* from the pull-down menu and type -100 in the *VALUE Force/moment value* text field; click on **OK**.
- Set *solution options* for the first *load step* using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- *Solution Controls* dialog box appears. By default, the *Basic* tab is active.
- On the left side, under *Time Control*, type $1e-4$ for *Time at end of loadstep*, type *20* for *Number of substeps*, and type *20* for *Min no. of substeps*.
- On the right side, under *Write Items to Results File*, click on the radio-button for *All solution items* and select *Write every substep* from the *Frequency* pull-down menu.
- Activate the *Transient* tab (by clicking on it).
- Under *Full Transient Options*, click on the radio-button for *Ramped loading*; click on **OK** to close the *Solution Controls* dialog box.
- Write the first *load step file* using the following menu path:

Main Menu > Solution > Load Step Opts > Write LS File

- Type *I*; click on **OK**.
- The first load step is written to a file.
- Remove the load at the midpoint node using the following menu path:

Main Menu > Solution > Define Loads > Delete > Structural > Force/Moment > On Nodes

- Pick the midpoint node; click on **OK**.
- Select *FY* from the pull-down menu; click on **OK**.
- Set *solution options* for the second *load step* using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- Activate the *Basic* tab. On the left side, under *Time Control*, type $2e-4$ for *Time at end of loadstep*; leave *20* unchanged for both *Number of substeps* and *Min no. of substeps*.
- On the right side, under *Write Items to Results File*, click on the radio-button for *All solution items* and select *Write every substep* from the *Frequency* pull-down menu.
- Click on **OK** to close the *Solution Controls* dialog box.

- Write the second *load step file* using the following menu path:

Main Menu > Solution > Load Step Opts > Write LS File

- Type **2**; click on **OK**.
- The second *load step* is written to a file.

- Set *solution options* for the third *load step* using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- On the left side, under **Time Control**, type **0.5** for **Time at end of loadstep** and type **200** for both **Number of substeps** and **Min no. of substeps**.
- On the right side, under **Write Items to Results File**, click on the radio-button **for All solution items** and select **Write every substep** from the **Frequency** pull-down menu.
- Click on **OK** to close the Solution Controls dialog box.

- Write the third *load step file* using the following menu path:

Main Menu > Solution > Load Step Opts > Write LS File

- Type **3**; click on **OK**.
- The third *load step* is written to a file.

- Obtain the solution using the following menu path:

Main Menu > Solution > Solve > From LS Files

- Solve *Load Step Files* dialog box appears.
- Type **1** for **LSMIN Starting LS file number** and **3** for **LSMAX Ending LS file number**; click on **OK**.
- Wait until ANSYS responds with **Solution is done!**

Postprocessing

- Review time-dependent behaviors of *y*-displacement at the midpoint, reaction force in *y*-direction at the left support, and reaction moment about *z*-axis at the left support using the following menu path:

Main Menu > TimeHist Postpro

- *Time History Variables* dialog box appears.
- Click on the button with the green plus sign at the top-left to define a variable.
- *Add Time History Variable* dialog box appears.
- Successively click on the items **Nodal Solution**, **DOF Solution**, and **Y-component of displacement**; click on **OK**.
- *Pick Menu* appears; pick the midpoint node; click on **OK**.
- Note the new variable **UY_2** in *Time History Variables* dialog box.

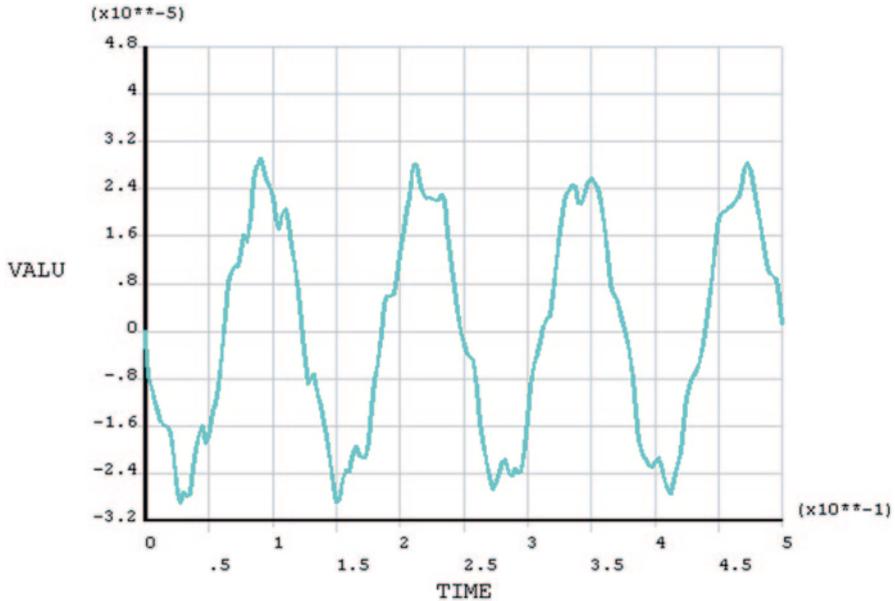


Fig. 8.98 Time variation of the y -displacement at the midpoint

- Highlight the row of **UY_2** from the list; click on the third from left button to plot the time variation of this displacement.
- The plot appears in the *Graphics Window*, as shown in Fig. 8.98.
- Note that in this plot the displacement has a periodical behavior with no sign of decay. This is because no damping was specified in the solution. A solution with damping will be obtained in the next section.
- Click on the fourth from left button to list the time variation of this displacement in a separate window.

Solution with Structural Damping

- Repeat the previous solution in its entirety with one difference: when specifying solution options, specify *structural damping* as 0.005. This is accomplished using the following menu path:

Main Menu > Solution > Analysis Type > Sol'n Controls

- Activate the **Transient** tab in the *Solution Controls* dialog box.
- Under **Damping Coefficients**, type 0.005 for **Stiffness matrix multiplier (BETA)**; click on **OK** to close the *Solution Controls* dialog box.
- Note that this needs to be done before each *load step* is written and that all the time and *substep* information needs to be re-entered.

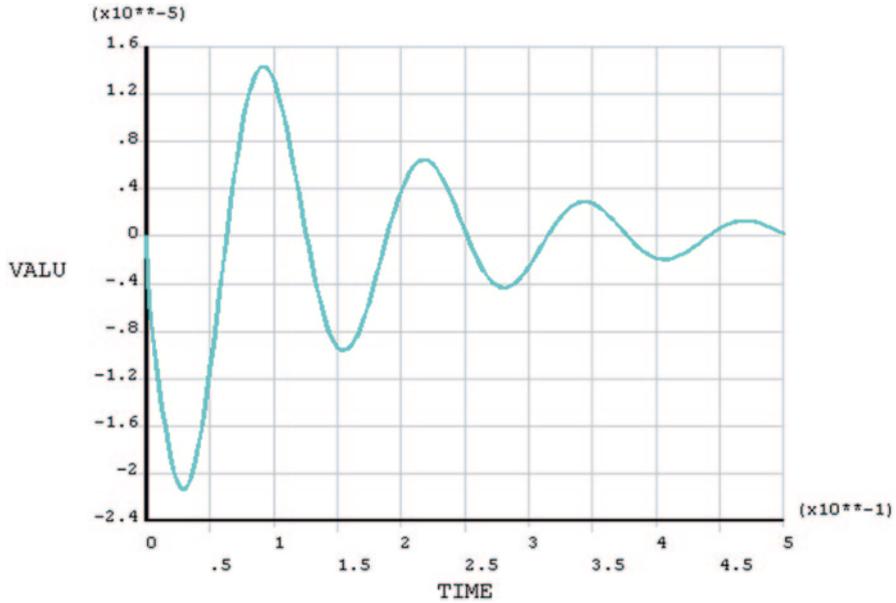


Fig. 8.99 Time variation of the y -displacement at the midpoint when damping is included in the analysis

Postprocessing

- Review the time-dependent behavior of the y -displacement at the midpoint in the same manner as was done for the case without damping. The corresponding response is given in Fig. 8.99 as it appears in the *Graphics Window*.

8.5.3.3 Dynamic Analysis of a 4-bar Linkage

Consider the 4-bar linkage shown in Fig. 8.100. The far-left node is rotated about the z -axis in a clockwise direction. The modeling part of the analysis utilizes direct nodal and elemental definition. Since this model is a flexible kinematics analysis, numerous beam elements are defined for each bar. To reduce the initial vibrations caused by loading, loads are applied in two time steps. A four-revolution displacement is imposed over a time period of 0.6 s. This is equivalent to a 400 rev/min (rpm) displacement.

The following input listing is used for model generation and solution:


```

MP,PRXY,2,0.3          ! POISSON'S RATIO FOR MATERIAL 2
MP,DENS,2,2.587799e-4 ! DENSITY FOR MATERIAL 2
MP,DAMP,2,0.000278    ! DAMPING MULTIPLIER FOR MATERIAL 2
N,1,0,0                ! CREATE NODES
N,2,2.5,0
N,3,2.5,0
CLOCAL,11,0,2.5,0,0,66.024 ! DEFINE LOCAL CS 11
CSYS,11                ! SWITCH TO LOCAL CS
N,4,2.5,0              ! CONTINUE CREATING NODES
N,5,5.5,0
N,6,8.5,0
N,7,11,0
N,8,11,0
CSYS,0                 ! SWITCH TO GLOBAL CARTESIAN CS
*AFUN,DEG              ! USE DEGREES FOR TRIG. FUNCTIONS
CLOCAL,12,0,2.5+11*COS(66.024),11*SIN(66.024),0,-73.21
                        ! DEFINE LOCAL
                        ! COORDINATE SYSTEM (CS) 12
CSYS,12                ! SWITCH TO LOCAL CS 12
N,9,2.5,0              ! CONTINUE CREATING NODES
N,10,5.25,0
N,11,8,0
N,12,10.5,0
E,1,2                  ! CREATE BEAM ELEMENTS FOR THE
E,3,4                  ! FIRST TWO BARS
E,4,5
E,5,6
E,6,7
MAT,2                  ! SWITCH TO MATERIAL 2
E,8,9                  ! CREATE BEAM ELEMENTS FOR THE
E,9,10                 ! LAST BAR
E,10,11
E,11,12
MAT,3                  ! SWITCH TO MATERIAL 3
TB,JOIN,3,1,1,JNS4    ! ACTIVATE JOINT MATERIAL MODEL
                        ! WITH NONLINEAR ELASTIC STIFFNESS
TBPT,,1E7              ! DEFINE NONLINEAR STIFFNESS
                        ! BEHAVIOR IN LOCAL ROTX
TB,JOIN,3,1,1,JNS6    ! ACTIVATE JOINT MATERIAL MODEL
                        ! WITH NONLINEAR ELASTIC STIFFNESS
TBPT,,1E7              ! DEFINE NONLINEAR STIFFNESS
                        ! BEHAVIOR IN LOCAL ROTZ
TYPE,2                 ! SWITCH TO ELEMENT TYPE 2
SECTYPE,2,JOINT,REVO  ! DEFINE REVOLUTE JOINT AT
SECJOINT,,11,11       ! NODES 2 AND 3
SECTYPE,3,JOINT,REVO  ! DEFINE REVOLUTE JOINT AT
SECJOINT,,11,11       ! NODES 7 AND 8
SECNUM,2               ! SWITCH TO SECTION NUMBER 2
E,2,3                  ! CREATE REVOLUTE JOINT ELEMENTS AT
                        ! NODES 2 AND 3
SECNUM,3               ! SWITCH TO SECTION NUMBER 3
E,7,8                  ! CREATE REVOLUTE JOINT ELEMENTS AT
                        ! NODES 7 AND 8
TYPE,3                 ! SWITCH TO ELEMENT TYPE 3
E,2                    ! DEFINE THE MASS AT NODE 2

```

```

E, 3                ! DEFINE THE MASS AT NODE 3
E, 7                ! DEFINE THE MASS AT NODE 7
E, 8                ! DEFINE THE MASS AT NODE 8
FINISH             ! FINISH
/SOL               ! ENTER SOLUTION PROCESSOR
*AFUN, RAD         ! USE RADIANS FOR TRIG. FUNCTIONS
ANTYPE, TRAN, NEW ! DEFINE NEW ANALYSIS AS TRANSIENT
NLGEOM, 1         ! INCLUDE LARGE DEFLECTION EFFECTS
D, 1, UX, , , 12, 11, UY ! SUPPRESS X AND Y DISPLACEMENTS
                  ! AT NODES 1 AND 12
D, ALL, UZ        ! SUPPRESS Z DISPL. AT ALL NODES
D, 1, ROTZ, -0.041888 ! SPECIFY A SMALL ROTATION OF
                  ! -0.041888 RAD (2.4 DEG) AT NODE 1
ACEL, 209.4, 324.7 ! SPECIFY GRAVITATIONAL
                  ! ACCELERATION IN X AND Y
                  ! DIRECTIONS
TIME, 0.001       ! SPECIFY THE TIME AT THE END OF
                  ! THIS LOAD STEP AS 0.001 SECONDS
NSUBST, 1         ! USE ONLY ONE SUBSTEP
TIMINT, OFF       ! TURN OFF TRANSIENT EFFECTS
NEQIT, 1         ! USE 1 EQUILIBRIUM ITERATION
                  ! FOR THE SUBSTEP
CNVTOL, F, 1, 0.001 ! SPECIFY CONVERGENCE TOLERANCE
CNVTOL, M, 1, 0.001 ! FOR FORCE AND MOMENT
OUTRES, ALL, ALL  ! WRITE RESULTS FOR ALL SUBSTEPS
                  ! TO THE DATABASE FILE
SOLVE             ! OBTAIN SOL'N FOR THIS LOAD STEP
D, 1, ROTZ, -25.13274 ! SPECIFY A ROTATION OF -25.13274
                  ! RAD (1440 DEG) AT NODE 1
TIME, 0.6        ! SPECIFY THE TIME AT THE END OF
                  ! THIS LOAD STEP AS 0.6 SECONDS
NSUBST, 299      ! USE w99 SUBSTEPS
TIMINT, ON       ! INCLUDE TRANSIENT EFFECTS
NEQIT, 100      ! USE A MAXIMUM OF 100 EQUILIBRIUM
                  ! ITERATIONS FOR EACH SUBSTEP
KBC, 0           ! APPLY THE LOAD (ROTATION) IN A
                  ! RAMPED FASHION
SOLVE           ! OBTAIN SOL'N FOR THIS LOAD STEP

```

Postprocessing

- Review time-dependent behavior of the torque at node 1 using the following menu path:

Main Menu > TimeHist Postpro

- *Time History Variables* dialog box appears.
- Click on the button with the green plus sign at the top-left to define a variable.
- *Add Time History Variable* dialog box appears.

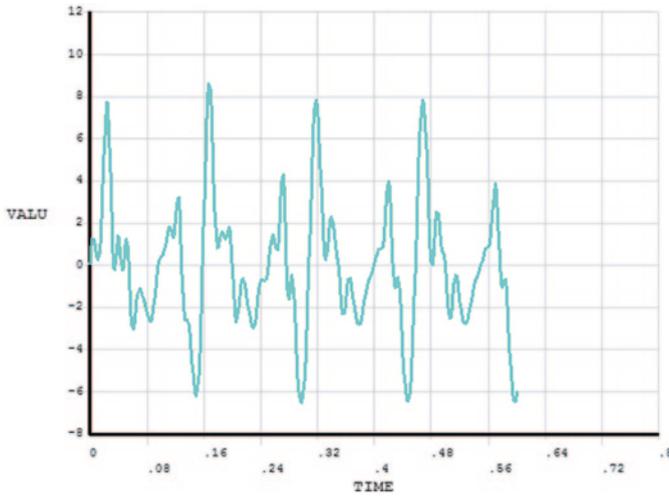


Fig. 8.101 Time variation of the moment at the *left* joint (node 1)

- Successively click on **Reaction Forces**, **Structural Moments**, and **Z-component of moment**; click on **OK**.
- *Pick Menu* appears; pick node 1; click on **OK**.
- Note the new variable **MZ_2** in the *Time History Variables* dialog box.
- Highlight the row **MZ_2** from the list; click on the third from left button to plot the time variation of this displacement.
- The plot appears in the *Graphics Window*, as shown in Fig. 8.101.
- Plot the nodes using the following menu path:

Utility Menu > Plot > Nodes

- Add new variable for displacement in the *y*-direction at node 2 by clicking on the button with the green plus sign and successively clicking on **Nodal Solution**, **DOF Solution**, and **Y-component of displacement**; click on **OK**.
 - *Pick Menu* appears; pick node 2. Since nodes 2 and 3 are coincident, ANSYS asks the user which one to pick. Select node 2, click on **OK**. Click on **OK** in the *Pick Menu*.
 - Note the new variable **UY_3** in the *Time History Variables* dialog box.
 - Highlight the row **UY_3** from the list; click on the third from left button to plot the time variation of this displacement.
 - The plot appears in the *Graphics Window*, as shown in Fig. 8.102.
 - Close the *Time History Variables* dialog box.
- Obtain an animation of the motion of the structure.
 - Enter *General Postprocessor* and read the results using the following menu path:

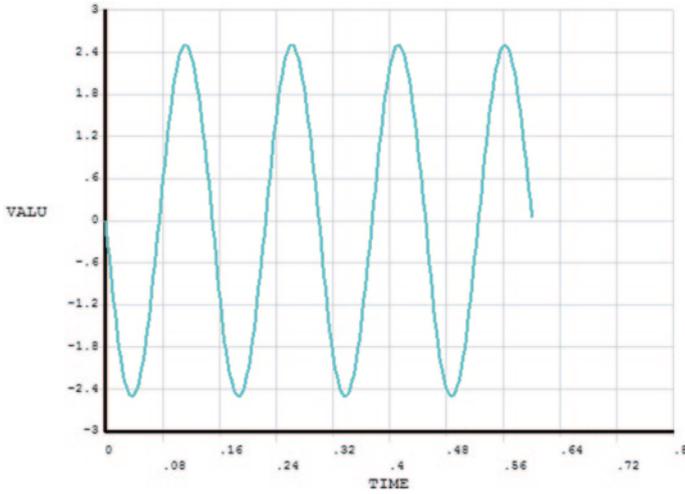


Fig. 8.102 Time variation of the y -displacement at node 2

Main Menu > General Postproc > Read Results > Last Set

- Plot deformed shape using the following menu path:

Main Menu > General Postproc > Plot Results > Deformed Shape

- Select *Def+ undeformed*; click on *OK*.
- Obtain animation using the following menu path:

Utility Menu > PlotCtrls > Animate > Over Time

- *Animate Over Time* dialog box appears. Enter **100** for *Number of animation frames*. Enter **0.05** for *Animation time delay (sec)*; click on *OK*.