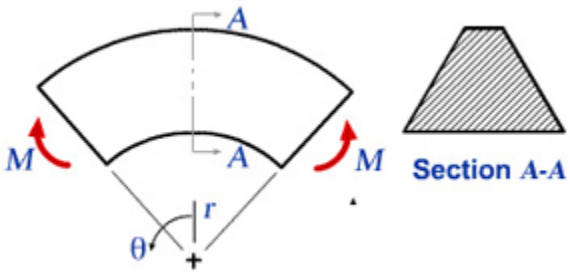
ANSYS - 3D Curved Beam

Author: Rajesh Bhaskaran, Cornell University

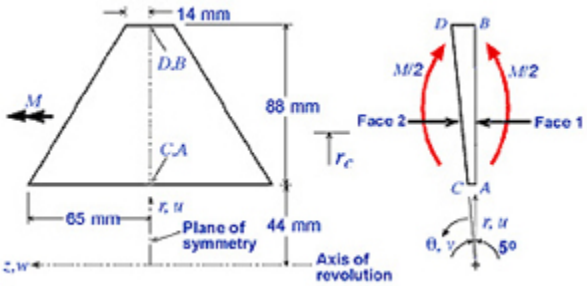
Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

Problem Specification

The problem considered here is the curved beam of uniform trapezoidal cross-section in example 6.15 of Cook et al. The beam is bent in its own plane by moments M. The problem is not axisymmetric because displacements have circumferential as well as radial and axial components. So we use 3D solid elements rather than axisymmetric elements. The geometry can nevertheless be described in cylindrical coordinates.



We would like to obtain the stresses for the trapezoidal cross-section AA shown above. Stresses in the curved beam do not vary with , so we can reduce the model and analyze only a typical slice between two closely spaced radial planes as shown below. The angle between AB and CD is taken to be 5 deg. as suggested by Cook et al.



The bending moment *M* must be applied indirectly in the reduced model since we don't know *a priori* the circumferential stress distribution it produces on the cross-section. Instead, we'll prescribe *displacements* such that radial plane sections remain plane and a pure moment load acts on the model i.e. no net force acts on it. The moment *M* can be computed from the stress distribution on the cross-section obtained from FEA. Stresses scale linearly with the applied moment. So the stresses associated with a prescribed moment M_p can be obtained by multiplying the computed stresses by the ratio M_p/M.

The z-constant plane containing A,B,C and D is a symmetry plane. So only half the cross-section needs to be modeled.

Boundary Conditions

The nodal d.o.f. in the radial (u), circumferential (v), and axial (w) directions are constrained as follows:

Face 1	Face 2
u=0 at node A	.
v=0 at all nodes	*v*=0.0001(r_c-r)at all nodes
w=0 along AB	*w*=0 along CD

All remaining d.o.f. are unrestrained. Setting u=0 at A prevents rigid body motion in the r-direction. Setting v=0 on face 1 nodes prevents circumferential motion of face 1. Setting w=0 on ABCD imposes symmetry about the middle r- plane. The above BC on face 2 nodes causes face 2 to remain plane as it rotates about a z-parallel axis at r=r_c. The factor 0.0001 is arbitrarily chosen. At the outset, the appropriate value of r_c is not known. The right value of r_c will give a pure bending load so that the radial reaction R_A at node A is zero. Two preliminary FE analysis with guess values of r_c=60mm and r_c=70mm were done. The respective R_A values turn out to be 2001N and 357N. By linear extrapolation, R_A=0 when r_c=72.2mm. So we'll use r_c=72.2mm in our analysis. (Since this is a pedagogical exercise, I've decided to be nice and give you the r_c value to use. In the real world, you'd of course have to figure it out yourself).

Go to Step 1: Start-up and preliminary set-up

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

Step 1: Start-up and preliminary set-up

Create a folder

Create a folder called *cbeam* at a convenient location. We'll use this folder to store files created during the session.

Start ANSYS

Start > All Programs > ANSYS 11.0 > ANSYS Product Launcher

Enter the location of the folder *cbeam* that you just created as your **Working Directory** by browsing to it.

Enter `cbeam` as your ***Job Name***. So all files generated during this ANSYS session will have *cbeam* as the prefix. Click on ***Run***.

Resize the ANSYS windows as shown in this snapshot so that you can read instructions in your browser window and implement them in ANSYS.

Set Preferences

Main Menu > Preferences

In the *Preferences for GUI Filtering* dialog box, click on the box next to **Structural** so that a tick mark appears in the box.

Recall that this is an optional step that customizes the graphical user interface so that only the menu options valid for structural problems are made available.

Go to Step 2: Specify element type and constants

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

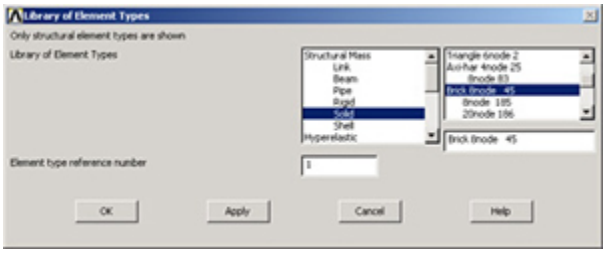
Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

Step 2: Specify element type and constants

Specify Element Type

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

Under ***Structural Mass***, pick ***Solid*** in the left field and ***Brick 8-node 45*** in the right field. Click **OK**.



Close the *Element Types* dialog box and also the *Element Type* menu.

Specify Element Constants

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

This brings up the *Element Type for Real Constants* menu with a list of the element types defined in the previous step. We have only one element type and it is automatically selected. Click **OK**.

You should get a note saying "Please check and change keyopt setting for element SOLID45 before proceeding." This means that there are no real constants to be specified for this element, as you might recall from the plate tutorial.

Close the *Real Constants* menu.

Save Your Work

Toolbar > SAVE_DB

Go to Step 3: Specify material properties

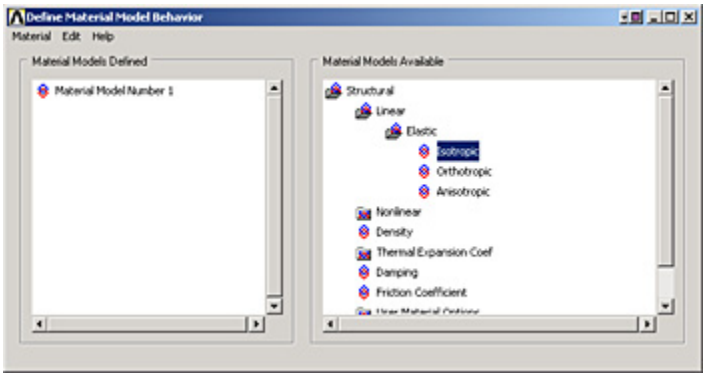
See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 3: Specify material properties

Main Menu > Preprocessor > Material Props > Material Models

In the *Define Material Model Behavior* menu, double-click on ***Structural***, ***Linear***, ***Elastic***, and ***Isotropic***.



Enter `200e9` for Young's modulus **EX**, `0.3` for Poisson's Ratio **PRXY**.

Click **OK**. Close the *Define Material Model Behavior* menu.

Save Your Work

Toolbar > SAVE_DB

Go to Step 4: Specify geometry

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 4: Specify geometry

We'll first create keypoints corresponding to the eight vertices of the model and then generate a volume from the keypoints. The keypoints will be created in the cylindrical coordinate system. Four of the keypoints are the vertices A,B,C and D shown in the figure of the geometry. The other four keypoints have the same *r* and as A,B,C and D but are displaced in the *z*-direction with respect to them.

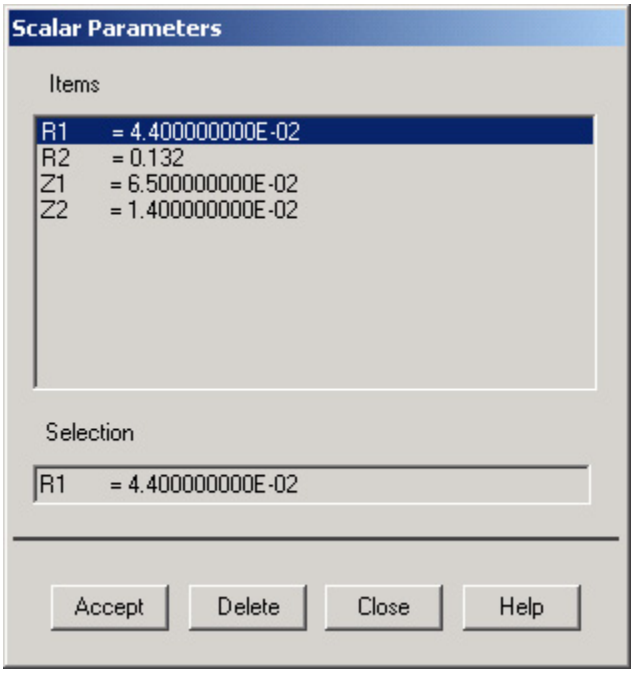
Create Scalar Parameters

For convenience, we'll create scalar parameters for the geometric dimensions in SI units.

Utility Menu > Parameters > Scalar Parameters

Enter the following parameters, clicking **Accept** after each. Check the figure of the geometry to see what dimension each parameter corresponds to.

```
R1=44e-3
R2=R1+88e-3
Z1=65e-3
Z2=14e-3
```



Click **Close**.

Switch to Cylindrical Coordinate System

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

Check that ANSYS reports the active coordinate system in the *Output* window :

```
ACTIVE COORDINATE SYSTEM SET TO     1  (CYLINDRICAL) Z- CYL. AXIS
```

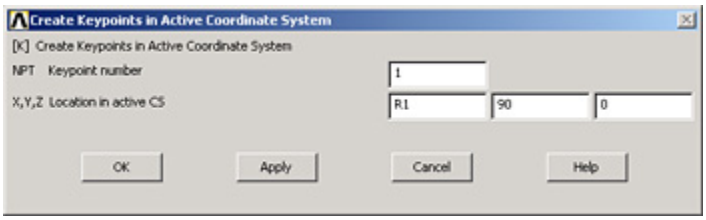
The reference number that ANSYS uses for the cylindrical coordinate system is 1 (the Cartesian system is 0).

Save your work: **Toolbar > SAVE_DB**

Create Keypoints

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

When the active coordinate system is set to cylindrical, **X**, **Y**, and **Z** in the menus refer to the cylindrical coordinates *r*, (in degrees) and *z,* respectively. Remember to make this mental substitution as you enter the keypoint coordinates. Also, you can use the tab key to move the cursor to the next entry field. Don't forget to change the keypoint number as you enter the coordinates of the keypoints.



Enter the keypoint locations (think about where each one lies as you enter its coordinates):

Keypoint 1: **X**=R1, **Y**=90, **Z**=0, Click **Apply**.

Keypoint 2: **X**=R1, **Y**=95, **Z**=0, Click **Apply**.

Keypoint 3: **X**=R1, **Y**=95, **Z**=Z1, Click **Apply**.

Keypoint 4: **X**=R1, **Y**=90, **Z**=Z1, Click **Apply**.

Keypoint 5: **X**=R2, **Y**=90, **Z**=0, Click **Apply**.

Keypoint 6: **X**=R2, **Y**=95, **Z**=0, Click **Apply**.

Keypoint 7: **X**=R2, **Y**=95, **Z**=Z2, Click **Apply**.

Keypoint 8: **X**=R2, **Y**=90, **Z**=Z2, Click **OK**.

Save your work: **Toolbar > SAVE_DB**

Then click on the isometric view button  on the right toolbar.

Higher Resolution Image

Create Volume

We'll next generate a volume from the 8 keypoints. The order of the keypoints should be around the bottom first and then the top.

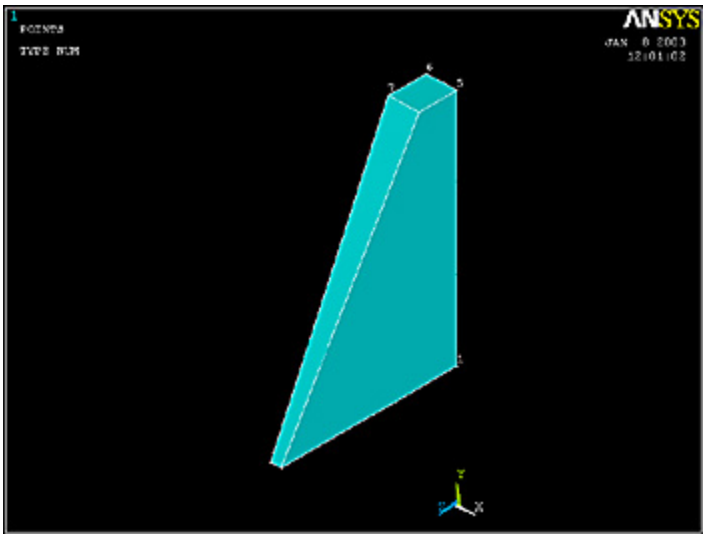
Switch to the Cartesian coordinate system for generating the volume:

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

The lines (i.e. edges) connecting the keypoints that ANSYS generates during the volume creation are "straight" in the active coordinate system. Since we want these edges to be straight, the active coordinate system needs to be Cartesian rather than a curvilinear system like the Cylindrical.

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

Pick the 8 keypoints in the order in which they are numbered. Click **OK** in the pick menu.



Plot Lines

Let's take a look at the lines that ANSYS generated in the volume creation process:

Utility Menu > Plot > Lines

Save Your Work

Toolbar > SAVE_DB

Go to Step 5: Mesh geometry

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 5: Mesh geometry

Bring up the *MeshTool*:

Main Menu > Preprocessor > MeshTool

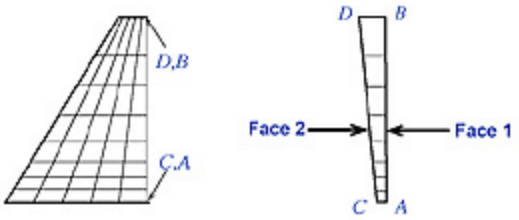
The *MeshTool* is used to control and generate the mesh.

Set Meshing Parameters

The element type and material property set to be used in meshing are automatically selected since we have only one of each. To check this using the *Mesh Tool*, make sure **Global** is selected under **Element Attributes** and click on **Set**. You will see that the correct element type and material number are already selected in the *Meshing Attributes* menu. There is no real constant set required for the SOLID45 element with default options (which we haven't changed). Click **Cancel**.

Set Mesh Size

Two views of the FE mesh we want to use are shown in the figure below.

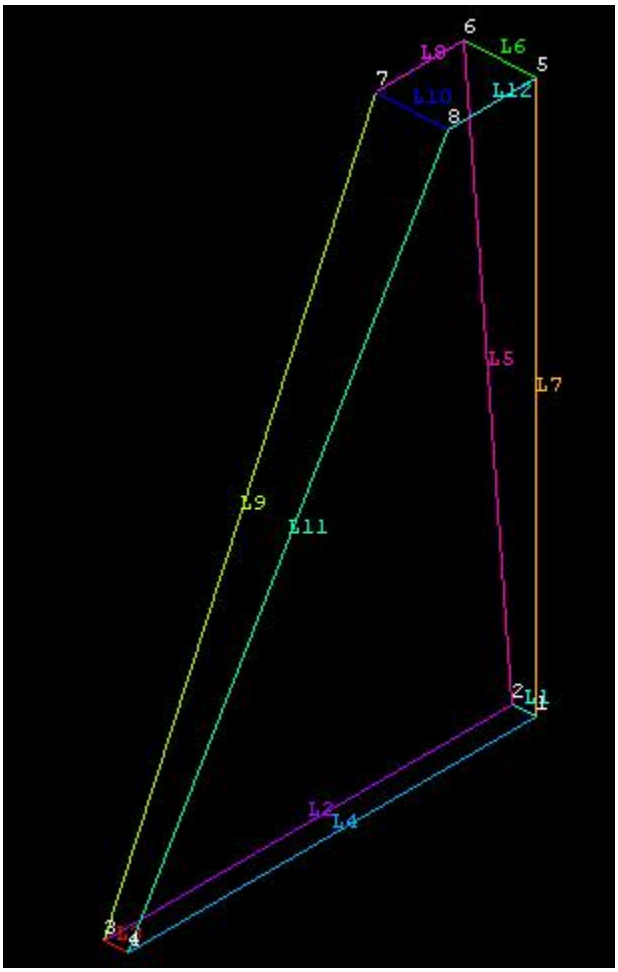


Curved beam theory predicts that the stress gradients will be highest on the edge nearest the center of curvature. So we want to use a graded mesh such that the radial length of elements increases as we go from edge AC towards edge BD. The *spacing ratio* parameter is used to control mesh grading in ANSYS. If positive, the spacing ratio is the ratio of last division size to first division size. Of course, the direction of line AB, for example, can be from A to B or B to A. We need to know the direction in order to determine which is the first division and which is the last. The way to determine the line direction is shown below. The default spacing ratio is one i.e. no mesh grading. Along lines AB and CD, we will use a spacing ratio of 0.3 (with the mesh spacing increasing from A to B and C to D).

Turn on line and keypoint numbers:

Utility Menu > PlotCtrls > Numbering

Turn on **Keypoint numbers** and **Line numbers** and click **OK**.



Utility Menu > List > Lines > OK

Looking at the table and figure, we see that the line 7 (or L7) goes from keypoint 5 (vertex *B*) to keypoint 1 (vertex *A*). So the first division is that next to *B* and the last division is that next to *A*. Since we want the radial mesh spacing to decrease from *B* to *A*, the spacing ratio is 0.3 (rather than 1/0.3). Similarly, determine the direction and spacing ratio for L5, L9 and L11.

Let's make a table summarizing the number of divisions (*NDIV*) and spacing ratio (*SPACE*) for each line.

Line no.	*NDIV*	*SPACE*
L1,L3,L6,L10	1	1
L2,L4,L8,L12	5	1
L7,L9,L11	8	0.3
L5	8	1/0.3

Your keypoint and line numbers may be different from the above and you should make your own table to account for that. Refer to the table while following the instructions given below for specifying no. of divisions and spacing ratio for each line.

> If you have trouble selecting the correct line below, **hold down** the left mouse button until the line is selected and then release the left button. If you want to **deselect** a line, right-click to go into deselect mode, left-click on the line to be deselected and right-click again to go back into select mode.

In *MeshTool*, under **Size Controls** and **Lines**, click **Set**. This brings up a pick menu.

Pick lines L1,L3,L6, and L10 and click **OK** in the pick menu. Enter `1` for ***No. of element divisions*** and click **Apply**. (The **Spacing Ratio** field can be left blank since the default value is one.)

Pick lines L2,L4,L8, and L12 and click **OK** in the pick menu. Enter `5` for ***No. of element divisions*** and click **Apply**.

Pick lines L7,L9, and L11 in the *Graphics* window and click **OK** in the pick menu. Enter `8` for ***No. of element divisions***, `0.3` for ***Spacing Ratio*** and click **Apply**.

Pick line L5 in the *Graphics* window and click **OK** in the pick menu. Enter `8` for ***No. of element divisions***, `1/0.3` for ***Spacing Ratio*** and click **OK**.

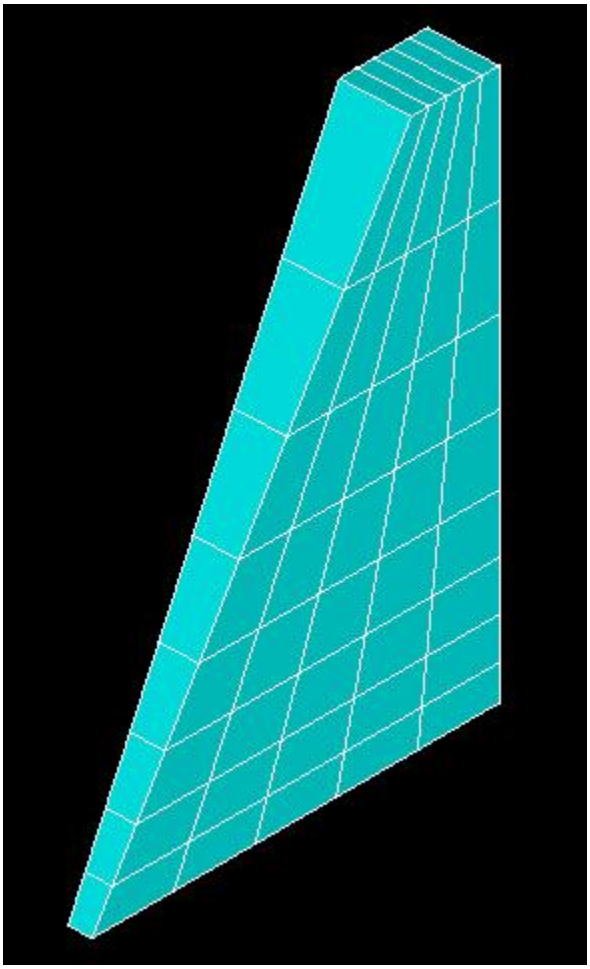
Mesh Volume

We'll mesh the volume using hexahedral elements (rather than tetrahedral elements). So choose **Hex** under **Shape** in the *MeshTool*.

Make sure **Volumes** is selected in the drop-down list next to **Mesh** in the *MeshTool*. This means the geometry component to be meshed is a volume. Click on 

.

Click on **Pick All** in the pick menu. The volume is meshed.



Utility Menu > PlotCtrls > Pan, Zoom, Rotate

Look at various views of the mesh and satisfy yourself that it looks right.

Close the *MeshTool*.

Save your work: **Toolbar > SAVE_DB**

Go to Step 6: Specify boundary conditions

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Step 6: Specify boundary conditions

Recall that the BCs for face 1 are:
u=0 at node A (keypoint 1)
v=0 at all face 1 nodes
w=0 along AB (line L7)

These BCs are in the cylindrical coordinate system. Switch to this coordinate system:

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

We'll work with areas while specifying the BCs. So plot areas: **Utility Menu > Plot > Areas**

Rotate Nodal Coordinate System

In ANSYS, the boundary constraints are applied in the nodal coordinate system which by default is parallel to the global Cartesian system. Since we want to apply the constraints in the global Cylindrical coordinate system, we need to rotate the nodal coordinate system into the active coordinate system (i.e. Cylindrical) using the nrotat command.

Type `nrotat,all` in the *Input* window.

To see the help page for *nrotat*, type help,nrotat in the *Input* window.

Apply u=0 at Node A

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

Select node at A (keypoint 1) in the lower-right corner and click ***OK*** in the pick menu. Select ***UX*** for ***DOFs to be constrained***. You can leave the ***Displacement value*** blank since the default is zero. Click ***OK***. You'll see an arrow symbol in the *Graphics* window indicating that the node A is constrained in the radial direction.

Select Nodes on Face 1

ANSYS provides extensive capabilities, referred to as "select logic", for selecting a subset of the full model using various criteria. We'll use select logic to select the nodes on face 1. We'll first select the area corresponding to face 1 and then select the nodes attached to this area.

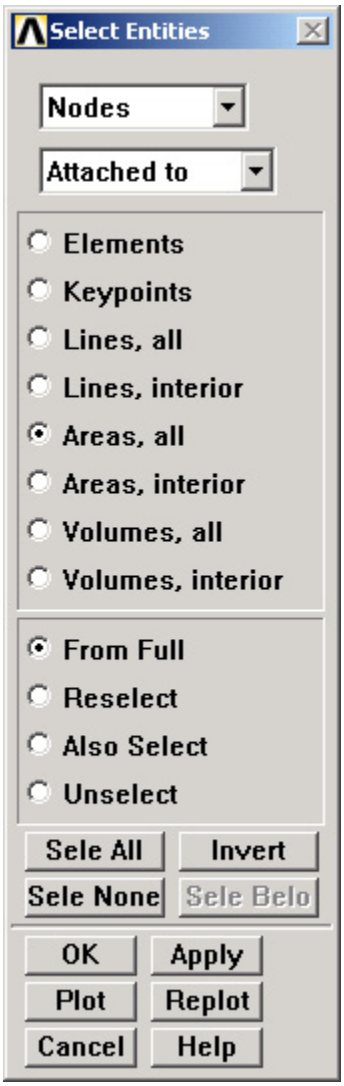
Utility Menu > Select > Entities

Select ***Areas*** from the pull-down menu at the top. Make sure ***By Num/Pick*** is selected below that. Click ***Apply***.

Hold down the left mouse button until face 1 is picked. Click ***OK*** in the pick menu.

Only the area corresponding to face 1 is selected currently. Verify this: **Utility Menu > Plot > Areas.**

Next we'll select the nodes attached to the selected area. In the *Select Entities* menu, select ***Nodes*** from the pull-down menu at the top and ***Attached*** to below that. Select ***Areas, All*** below that. Click ***Apply***.

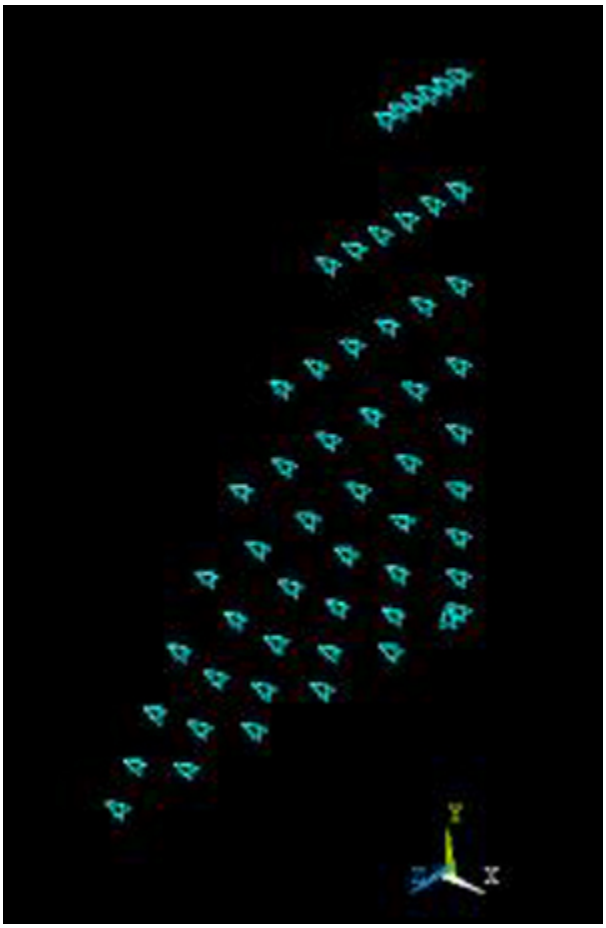


Check that only nodes attached to face 1 are currently selected: **Utility Menu > Plot > Nodes**

Apply v=0 on Face 1

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

Pick All nodes in the pick menu. Select ***UY*** for ***DOFs to be constrained*** and click ***OK***. You'll see arrow symbols in the *Graphics* window indicating that the nodes on face 1 are constrained in the circumferential direction.



We can use ***Pick All*** since only the nodes on face 1 are currently selected. ANSYS commands apply only to the currently selected entities.

Select Nodes Along AB

Plot lines: Utility Menu > Plot > Lines

In the *Select Entities* menu, select ***Lines*** from the pull-down menu at the top and ***By Num/PickApply*** below that. Click ***OK***.

Click on line AB (L7) and ***OK*** in the pick menu.

Next we'll select the nodes attached to the selected line. In the *Select Entities* menu, select ***Nodes*** from the pull-down menu at the top and ***Attached to*** below that. Select ***Lines, All*** below that. Click ***Apply.***

Check that only nodes attached to line AB are currently selected: **Utility Menu > Plot > Nodes**

Apply w=0 Along AB

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

Pick All nodes in the pick menu. Select ***UZ*** for ***DOFs to be constrained*** and click ***OK***.

Define Function

Recall that the BCs for face 2 are:
v=0.0001(r_c-r) at all face 2 nodes
w=0 along CD (line L5)

Since the BC on v is a function of the spatial coordinates, we need to define a function to apply this BC. Bring up the function editor:

Utility Menu > Parameters > Functions > Define/Edit...

You can enter the function using the calculator buttons or type it in. The variables such as ***TIME***, ***X***, ***Y*** etc. that are available for defining functions are in the pull-down list below the ***Result*** field. For entering the spatial coordinates *X* and *Y*, use the pull-down menu. Enter the function:

Note that variables are enclosed in squiggly brackets.

Save the function: **Function Editor > File > Save**

Use vface2.func for the filename.

Close the function editor.

Define Table from Function

ANSYS doesn't allow the user to use functions directly while applying loads to a model. Instead, one has to go through the additional step of using a "Function Loader" that retrieves the function and loads it as a *Table* array. The *Table* array can then be applied to the model. The process is not exactly elegant but then we are engineers.

Utility Menu > Parameters > Functions > Read From File

Select *vface2.func* and click ***Open***.

Enter vface2 for ***Table parameter name***.

Observe that ANSYS displays the equation that will be used in creating the *Table* array. Click ***OK***.

Select Nodes on Face 2

Start by selecting the whole model to undo previous selects.

Utility Menu > Select > Everything

Utility Menu > Plot > Areas

To select the nodes on face 2, we'll follow the same procedure as for face 1.

Utility Menu > Select > Entities

Select ***Areas*** from the pull-down menu at the top. Select ***By Num/Pick*** below that. Click ***Apply***.

Hold down the left mouse button until face 2 is picked. Click ***OK*** in the pick menu.

Only the area corresponding to face 2 is selected currently. Verify this: **Utility Menu > Plot > Areas.**

Next we'll select the nodes attached to the selected area. In the *Select Entities* menu, select ***Nodes*** from the pull-down menu at the top and ***Attached to*** below that. Select ***Areas, All*** below that. Click ***Apply***.

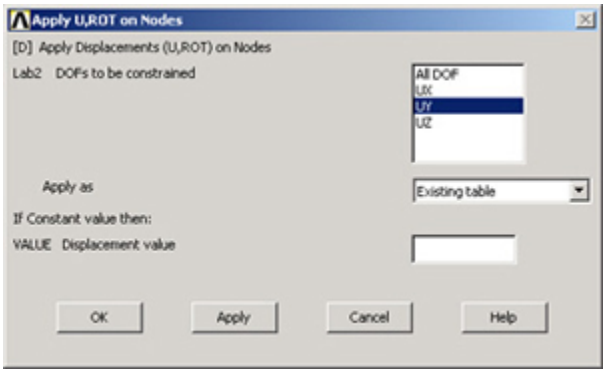
Check that only nodes attached to face 2 are currently selected: **Utility Menu > Plot > Nodes**

Apply BC for v on Face 2

We'll use the *vface2* table that we created to apply this BC.

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

Pick All nodes in the pick menu. Select ***UY*** for ***DOFs to be constrained***. Select ***Existing table*** under ***Apply as*** and click ***OK***.



We have defined only one table (***VFACE2***) and that is automatically selected. Click ***OK***.

You'll see arrow symbols in the *Graphics* window indicating that the nodes on face 2 are constrained in the circumferential direction.

Select Nodes Along CD

Plot lines: **Utility Menu > Plot > Lines**

In the *Select Entities* menu, select ***Lines*** from the pull-down menu at the top and ***By Num/Pick*** below that. Click ***Apply***.

Click on line CD (L5) and ***OK*** in the pick menu.

Next we'll select the nodes attached to the selected line. In the *Select Entities* menu, select ***Nodes*** from the pull-down menu at the top and ***Attached to*** below that. Select ***Lines, All*** below that. Click ***Apply***.

Check that only nodes attached to line CD are currently selected: **Utility Menu > Plot > Nodes**

Apply w=0 Along CD

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

Pick All nodes in the pick menu. Select ***UZ*** for ***DOFs to be constrained***. Select ***Constant value*** under ***Apply as*** and click ***OK***.

Utility Menu > Select > Everything

Utility Menu > Plot > Volumes

Save your work:**Toolbar > SAVE_DB**

Go to Step 7: Solve!

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

Step 7: Solve!

Enter Solution Module:

Main Menu > Solution

Enter check in the *Input* window. If the problem has been set up correctly, there will be no errors or warnings reported. If you look in the *Output* window, you should see the message: The analysis data was checked and no warnings or errors were found.

Main Menu > Solution > Solve > Current LS

Review the information in the */STATUS Command* window. Close this window.

Click ***OK*** in *Solve Current Load Step* menu.

ANSYS performs the solution and a window should pop up saying "Solution is done!". Close the window.

Verify that ANSYS has created a file called *cbeam.rst* in your working directory. This file contains the results of the (previous) *solve*.

Go to Step 8: Postprocess the results

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

Step 8: Postprocess the results

Enter the postprocessing module to analyze the solution.

Main Menu > General Postproc

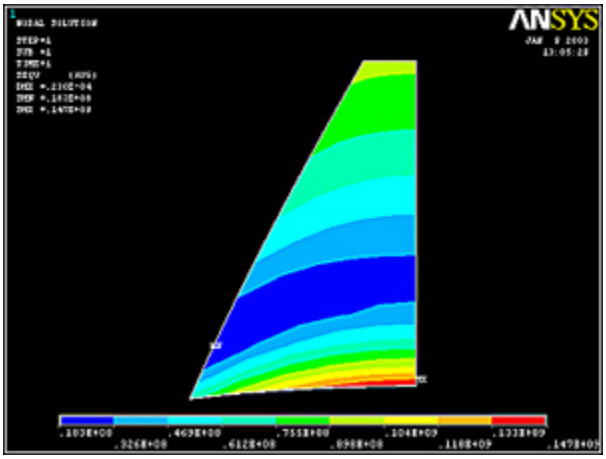
Plot von Mises Stress

To display the von Mises stress distribution as *continuous* contours, select

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

Select ***Stress*** from the left list, ***von Mises SEQV*** from the right list and click OK.

Utility Menu > PlotCtrls > Pan,Zoom,Rotate > Right



(Click Picture for Larger Image)

The maximum von Mises stress is 147 MPa and occurs at the bottom on the symmetry line.

Plot Circumferential Stress

is the *SY* stress component in cylindrical coordinates in ANSYS. Activate the cylindrical coordinate system for results display (you need to do this even if you were working in the cylindrical system in the preprocessor):

Main Menu > General Postproc > Options for Outp

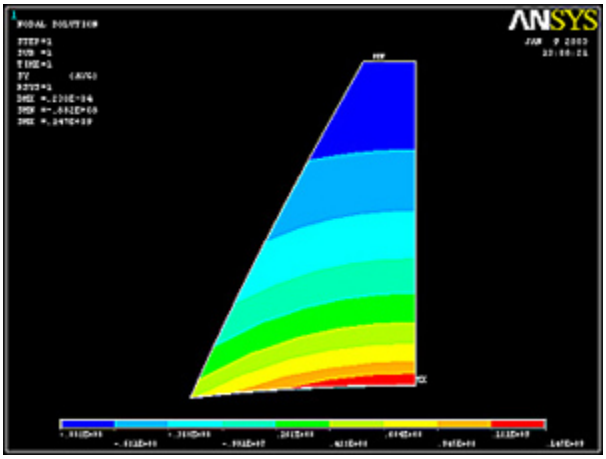
Select **Global Cylindric** for **Results Coord System**.

To display thestress distribution over face 1 as *continuous* contours, select

Utility Menu > PlotCtrls > Pan,Zoom,Rotate > Right

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

Select ***Stress*** from the left list, ***Y-direction SY*** from the right list and click **OK**.



(Click Picture for Larger Image)

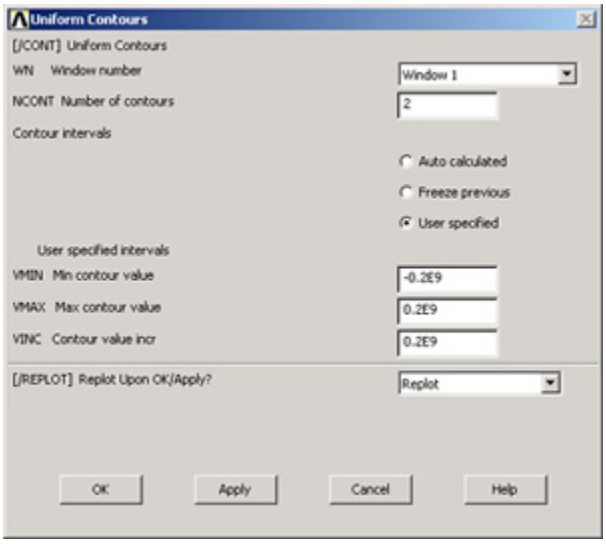
Check where the maximum (*MX*) and minimum (*MN*) values occur in the plot. The circumferential stress is tensile (positive) and compressive (negative) on the inner and outer portions of the cross-section, respectively. Is this what you'd have expected? Thecontours are more closely spaced at smaller *r* values. This agrees with the prediction of curved beam theory that the stress gradients will be highest on the edge nearest the center of curvature.

Plot Neutral Axis

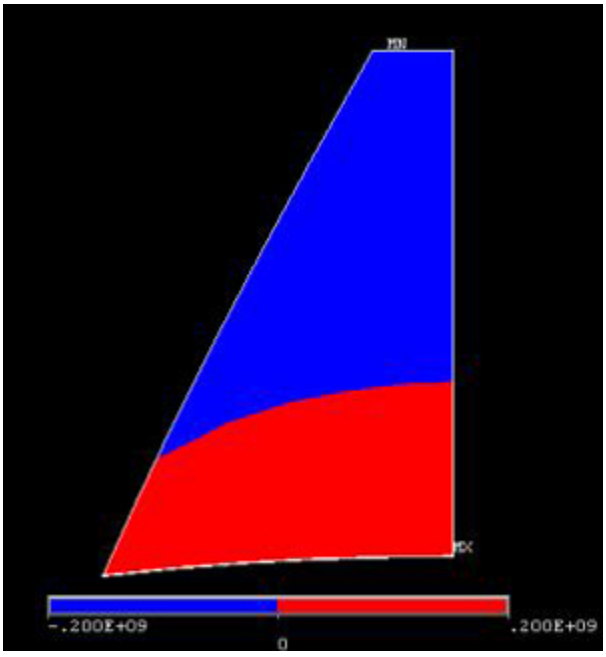
The neutral axis is the locus of points where is zero. To visualize the neutral axis, we'll change the contour levels that are plotted.

Utility Menu > PlotCtrls > Style > Contours > Uniform Contours

Enter 2 for ***Number of contours***, and choose ***User specified*** for ***Contour Intervals***. Enter ***Min contour value***=-0.2E9, ***Max contour value***=0.2E9, and ***C ontour value incr***=0.2E9. Click ***OK***.



This plots the regions with positive and negative values in different colors. In the red region, 0<<200MPa and in the blue region, -200MPa<<0. So the boundary between the two colors is the neutral axis.



The FEA results indicate that the neutral axis is curved, contrary to the assumption in mechanics of materials theory.

Plot Radial Stress

In cylindrical coordinates, the radial stress is the *SX* stress component.

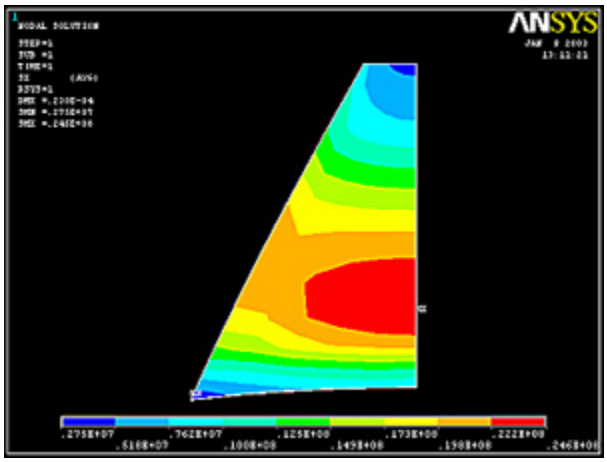
Main Menu > General Postproc > Plot results > Nodal Solu...

Select ***Stress*** from the left list, ***X-direction SX*** from the right list and click ***OK***.

Change contour plot options back to original:

Utility Menu > PlotCtrls > Style > Contours > Uniform Contours

Enter 9 for ***Number of Contours***, and choose Auto calculated for ***Contour Intervals***. Click ***OK***.



(Click Picture for Larger Image)

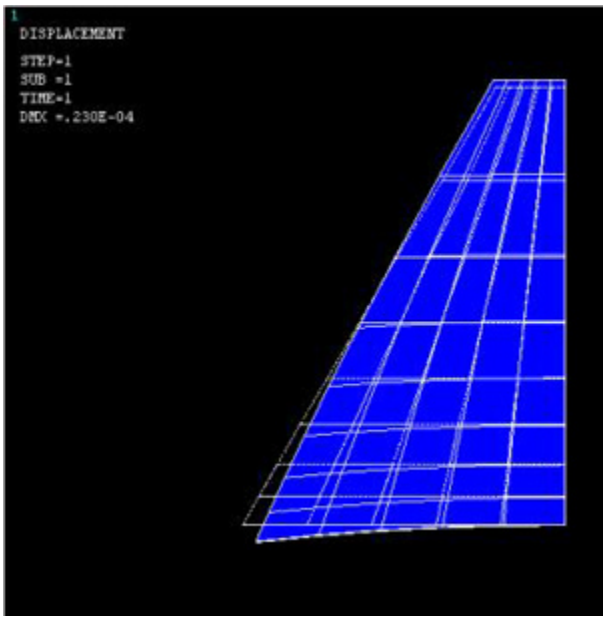
The radial stress is tensile over the entire cross-section.

Plot Deformed Shape

Main Menu > General Postproc > Plot Results > Deformed Shape

Select ***Def + undeformed*** and click ***OK***.

This plots the deformed and undeformed shapes in the *Graphics* window. The maximum displacement *DMX*=0.230e-4 m.



Animate the deformation:

Utility Menu > PlotCtrls > Animate > Deformed Shape

Select ***Def + undeformed*** and click ***OK***. ***Select Forward Only*** in the *Animation Controller*.

From the animation of the deformation of the cross-section, check that the following BCs are satisfied: u=0 at node A, w=0 along AB and CD. (Note that the z-direction is from right to left in the above view).

The radial stress is tensile, so the radial dimension becomes larger as expected. We saw that the circumferential stress is, respectively, tensile and compressive on the inner and outer portions of the cross-section. You can check that the stress *SZ* parallel to the axis of revolution is small. Therefore, the Poisson effect should cause the inner and outer portions, respectively, to contract and expand in the z-direction as is observed. Circumferential tension on the inner portion pulls material toward the center of curvature. Outer corners are more flexible than the central portion. So it makes sense that the outer, bottom corner moves inward with respect to central point A (Cook et al). Thus, the deformation we gets looks reasonable.

Animate the front view:

Utility Menu > PlotCtrls > Pan,Zoom,Rotate > Front

Utility Menu > PlotCtrls > Animate > Deformed Shape

Select ***Def + undeformed*** and click ***OK***. ***Select Forward Only*** in the *Animation Controller*.

From this animation, check that the BCs for v on both faces are satisfied.

Save your work: **Toolbar > SAVE_DB**

Go to Step 9: Validate the results

See and rate the complete Learning Module

Go to all ANSYS Learning Modules

Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve!
8. Postprocess the results
9. Validate the results

Step 9: Validate the results

Simple Checks

Does the deformed shape look reasonable and agree with the applied BCs? We checked this in step 8.

Do the reactions at the supports balance the applied forces for static equilibrium? To check this, select

Main Menu > General Postproc > List Results > Reaction Solu

Select ***All struc forc F*** for ***Item to be listed*** and click ***OK***.

There are no applied forces in this problem, so the total reaction force should be zero for equilibrium. The total reaction force in the radial direction (*FX*) is 4.6 N which is close to zero. We can lower it even further by refining our estimate of r_c. The total reaction forces FY in the cirumferential direction and FZ in the axial direction are small but not zero. This is possible because FX is small but not zero. So the structure is in equilibrium to a reasonable degree of approximation.

Refine Mesh

Let's repeat the calculations on a mesh with twice the no. of mesh divisions in the radial and axial directions while retaining a single division on AC and BD. We need to reset NDIV and SPACE on the following lines:

Line no.	*NDIV*	*SPACE*
L2,L4,L8, L12	10	1
L7,L9,L11	16	0.3
L5	16	1/0.3

Let's use a different jobname for the refined mesh case. Change jobname: **Utility Menu > File > Change Jobname**

Enter cbeam2 as the ***New jobname*** and click ***OK***.

Main Menu > Preprocessor > Meshing > MeshTool

Delete the current mesh: Select ***clear*** under ***Mesh***: and ***Pick All*** in the *pick* menu. The mesh is deleted.

Utility Menu > Plot > Lines

Under ***Size Controls*** and ***Lines***, click ***Set***. This brings up a pick menu.

Pick lines L2,L4,L8, and L12 and click ***OK*** in the pick menu. Enter 10 for ***No. of element divisions,*** leave ***Spacing Ratio*** blank and click ***Apply***.

Pick lines L7,L9, and L11 in the *Graphics* window and click ***OK*** in the pick menu. Enter 16 for ***No. of element divisions***, 0.3 for ***Spacing Ratio*** and click ***Apply***.

Pick line L5 in the *Graphics* window and click ***OK*** in the pick menu. Enter 16 for ***No. of element divisions***, 1/0.3 for ***Spacing Ratio*** and click ***OK***.

Select Volumes for ***Mesh***: and Hex for ***Shapes***:, then click ***Mesh***.

Since we applied the BCs to the finite-element model rather than the solid geometry model, the BCs were deleted along with the mesh. So we have to reapply the BCs again. Repeat step6 to reapply the BCs. It might feel like a chore but consider it as good practice. Since the *vface2* table for applying the BC on face 2 already exists, you need not recreate the function or the table.

Save your work: **Toolbar > SAVE_DB**

This will create the file *cbeam2.db* in your working directory.

After reapplying the BCs, solve the problem as in step7.

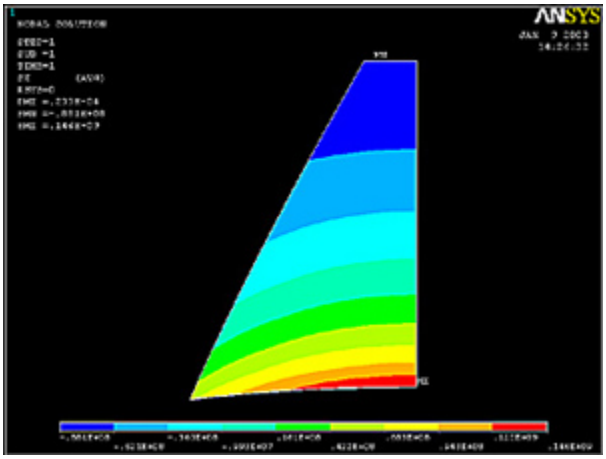
Plot Circumferential Stress

Display thestress distribution over face 1:

Utility Menu > PlotCtrls > Pan,Zoom,Rotate > Right

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

Select ***Stress*** from the left list, ***Y-direction SY*** from the right list and click ***OK***.



(Click Picture for Larger Image)

Compare this result with the plot obtained on the coarser mesh. The results on the two meshes compare well indicating that the coarse mesh provides good resolution. Similarly, compare the von Mises stress results on the two meshes.

Exit ANSYS

Utility Menu > File > Exit

Select ***Save Everything*** and click ***OK***.

Reference

Cook, R.D., Malkus, D.S., Plesha, M.E., and Witt, R.J., *Concepts and Applications of Finite Element Analysis*, Fourth Edition, John Wiley and Sons, Inc., 2002.

Back to Problem Specification

See and rate the complete Learning Module

Go to all ANSYS Learning Modules