ANSYS - 3D Curved Beam step 6

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

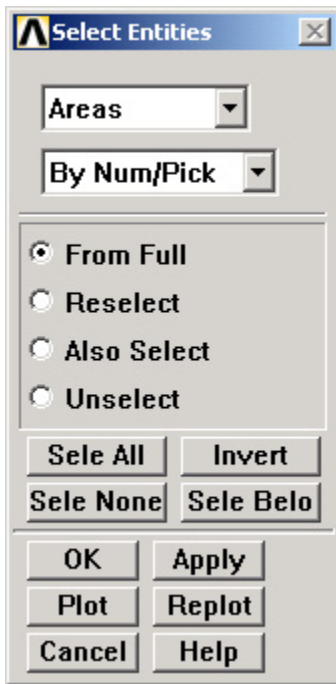**Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Nodes**

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.

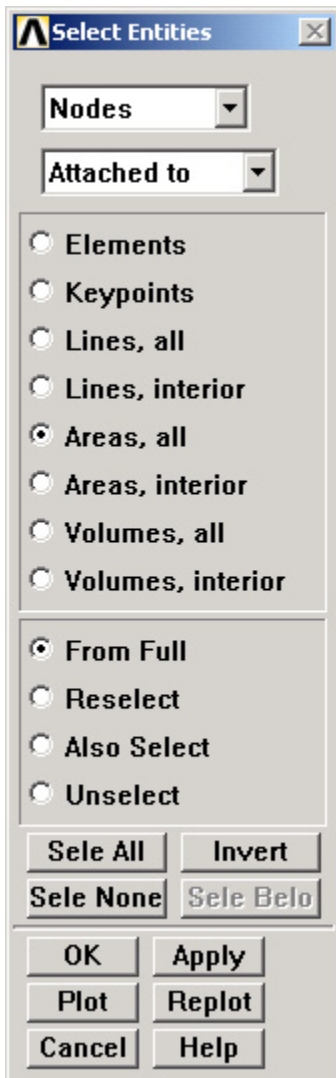**Utility Menu > Select > Entities**

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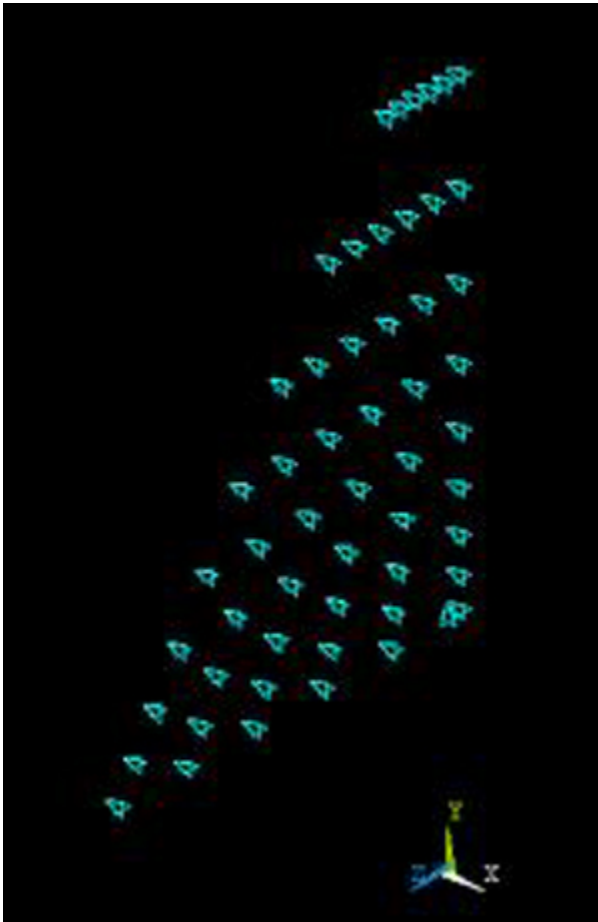
We can use *Pick All* since only the nodes on face 1 are currently selected. ANSYS commands apply only to the currently selected entities.

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:

Result = $1e-4*(72.2e-3 - sqrt(\{X\}^2+\{Y\}^2))$

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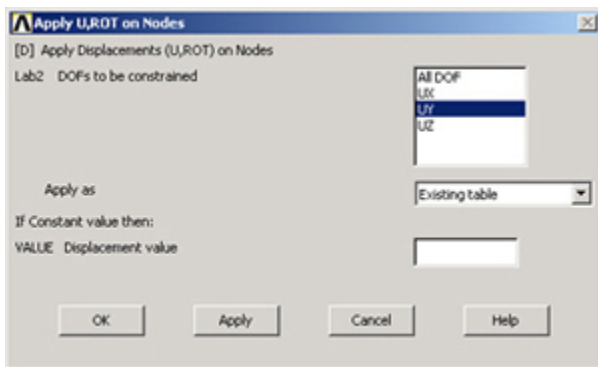
We have defined only one table (*VFACE2*) and that is automatically selected. Click *OK*.

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