ANSYS 12 - LaminatePlate - Setup (Physics)

⚠ This Tutorial is Under Construction!

Author: John Singleton, Cornell University

4. Setup (Physics)

Supports

The laminated plate will require fixed displacement boundary conditions for each of its four sides. First orient the the view such that, you are looking down the z axis onto the xy plane. The y axis should be pointing upward and the x axis should be pointing to the right. The support on the left side of the plate will be dealt with first. In order to specify a fixed displacement, *(Right Click) Static Structural < Insert < Displacement*. Rename "Displacement" to "Left Side Displacement". Next, use the edge selection filter, select the left plate side and apply it to the geometry selection for the Displacement condition. Next, set the y and z components of the displacement to zero while leaving the x component free. Repeat this process for the right side of the plate. Rename "Displacement 2" to "Right Side Displacement". Note that for the right side the displacement conditions will be the same as the left side. Now, the top side will be deal with. Once again insert a Displacement condition. Rename "Displacement 3" to "Top Side Displacement". Use the edge selection filter to select the top edge and then apply it as the geometry selection. Next, specify the x and the z component of the displacement to zero while leaving the y component free. Repeat, this process for the bottom edge. Rename "Displacement 4" to "Bottom Side Displacement". The displacement components for the bottom side are the same as those for the top side.

Pressure

To create the uniform pressure *(Right Click) Static Structural > Insert > Pressure*. Then apply the face of the plate as the geometry selection and set the magnitude of the pressure to 1 Pa.

Composite Setup

ANSYS Workbench does not have a GUI for composites, thus other measures must be taken. In order to model the laminate plate two command snippets must be inserted within Workbench and a new coordinate system must be introduced

Command Snippet 1

In order to create the first command snippet *(Expand) Geometry > (Right Click) Surface Body > Insert > Commands*. Many commands are needed to specify the composite material properties of the laminate plate. These commands are listed below in between the two lines of asterisks. Copy and paste the commands below into the Commands snippet, that you just created. Notice, that the purpose of each command is displayed to the right.

```
***************************************************************************

/prep7              !Call the Preprocessor

et,matid,181         !Specify the element type, shell 181
keyopt,matid,3,2     !Use full integration
mpdel,all,matid       !Delete all previously assigned material properties
tbdel,all,matid       !Delete all previouly assigned tabular data

MP,EX,1,25E6         !Define Young's Modulus: X Direction
MP,EY,1,1E6          !Define Young's Moduls: Y Direction
MP,EZ,1,1E6          !Define Young's Modulus: Z Direction
MP,GXY,1,5E5          !Define Shear Modulus XY
MP,GYZ,1,2E5          !Define Shear Modulus YZ
MP,GXZ,1,5E5         !Define Shear Modulus XZ

MP,PRXY,1,0.25        !Define Major Poisson's ratio XY
MP,PRYZ,1,0.01        !Define Major Poisson's ratio YZ
MP,PRXZ,1,0.25        !Define Majoy Poisson's ration XZ
```

```
sect,1,shell            !Specify the section type
secdata,.025,matid,0    !Define thickness & orientation for layer 1
secdata,.025,matid,90   !Define thickness & orientation for layer 2
secdata,.025,matid,90   !Define thickness & orientation for layer 3
secdata,.025,matid,0    !Define thickness & orientation for layer 4
secoffset,MID           !Define thickness location
```

****

Command Snippet 2

In order to create the second command snippet *(Right Click) Static Structural > Insert > Commands*. Next, copy and paste the commands listed below in between the lines of asterisks into the second Command Snippet.

****

```
fini        !Exit current Module
/prep7      !Call the preprocessor
etcon,off   !Disable ANSYS from changin any KEYOPTS
fini        !Exit the preprocessor
/solu       !Call the Solve Module
```

****

Go to Step 5: Solution

See and rate the complete Learning Module

Go to all ANSYS Learning Modules