

ANSYS – Car Rear Window

Type of Problem: Structural

Type of Analysis: Static

Difficulty: Beginner

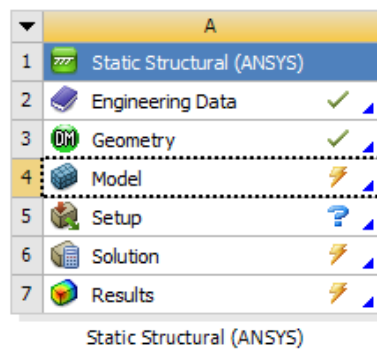
Software: ANSYS 12 Workbench

Description:

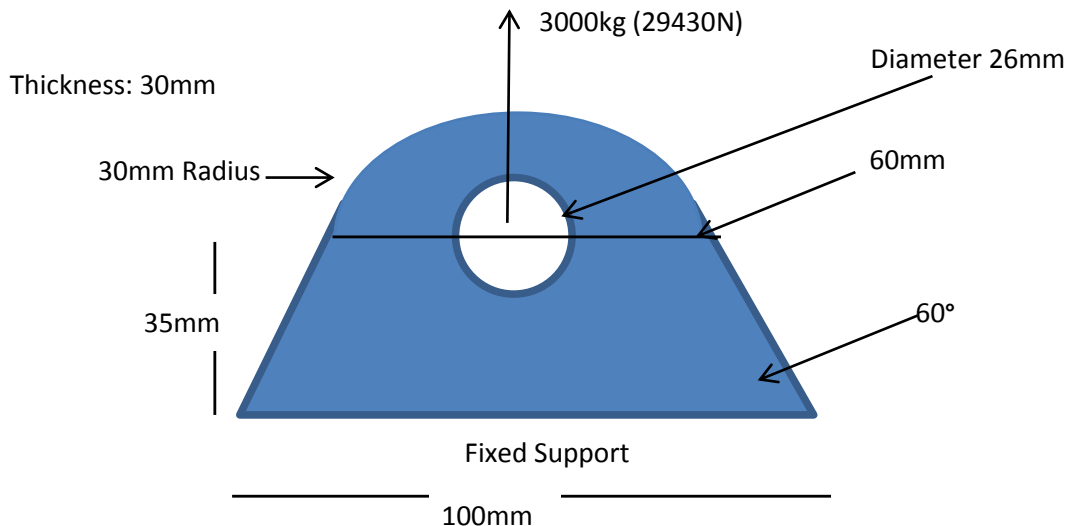
I will design and model a simple lug to show quick you can use ANSYS12 workbench to do quick design checks, and get a better understanding of how stress acts on lugs.

Open ANSYS 12 Workbench and drag a Static structural analysis into the project schematic. Then right click on Geometry and select New Geometry.

7



Sketch up our simple lug as below.



Once sketched, go back to the project schematic and click Model it is time to make refine our mesh. We know

ANSYS – Car Rear Window

Next we must insert our parameters. Our lug is going to have a 3000kg (29430N) force acting on it. We add the force by click static structural. The environment options should appear in the top left corner.



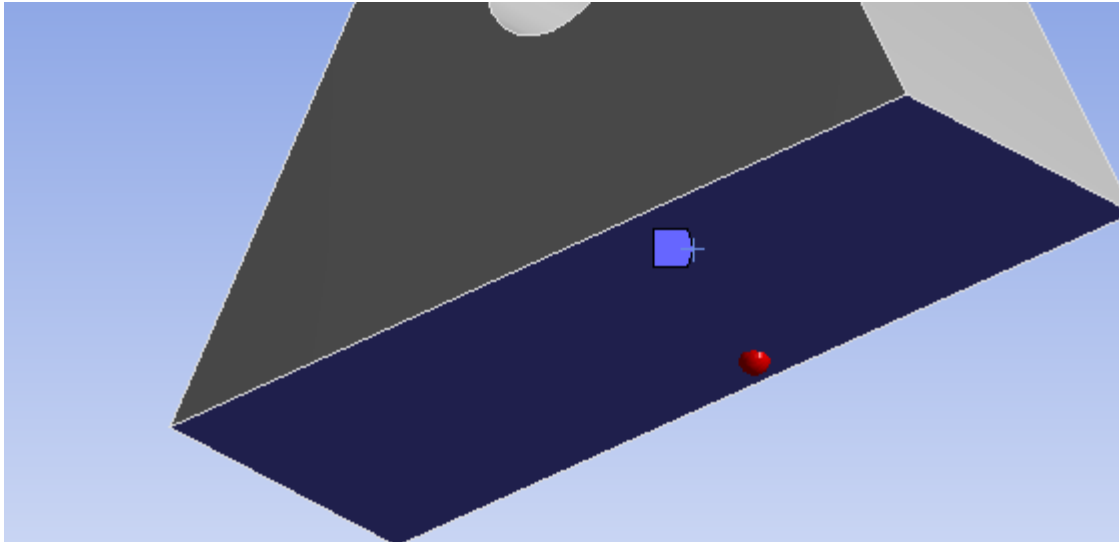
Click on Loads and press Remote Force Load. Click on the inside of our lug. At the bottom left click on Define By and select Components.

Set the Y component to 29430 N (3000kg x 9.81)

Details of "Remote Force"	
<input type="checkbox"/> Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Coordinate System	Global Coordinate System
<input type="checkbox"/> X Coordinate	0. m
<input type="checkbox"/> Y Coordinate	3.5e-002 m
<input type="checkbox"/> Z Coordinate	1.5e-002 m
Location	Click to Change
<input type="checkbox"/> Definition	
Type	Remote Force
Define By	Components
<input type="checkbox"/> X Component	0. N (ramped)
<input type="checkbox"/> Y Component	29430 N (ramped)
<input type="checkbox"/> Z Component	0. N (ramped)
Suppressed	No
Behavior	Deformable
<input type="checkbox"/> Advanced	

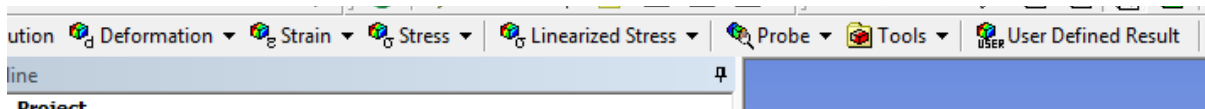
Now we have to fix the lug down to something. In this case we are going to assume it's completely fixed. To do this, go back to the top left corner and select Supports. Select Fixed Support and select the bottom of the lug.

ANSYS – Car Rear Window



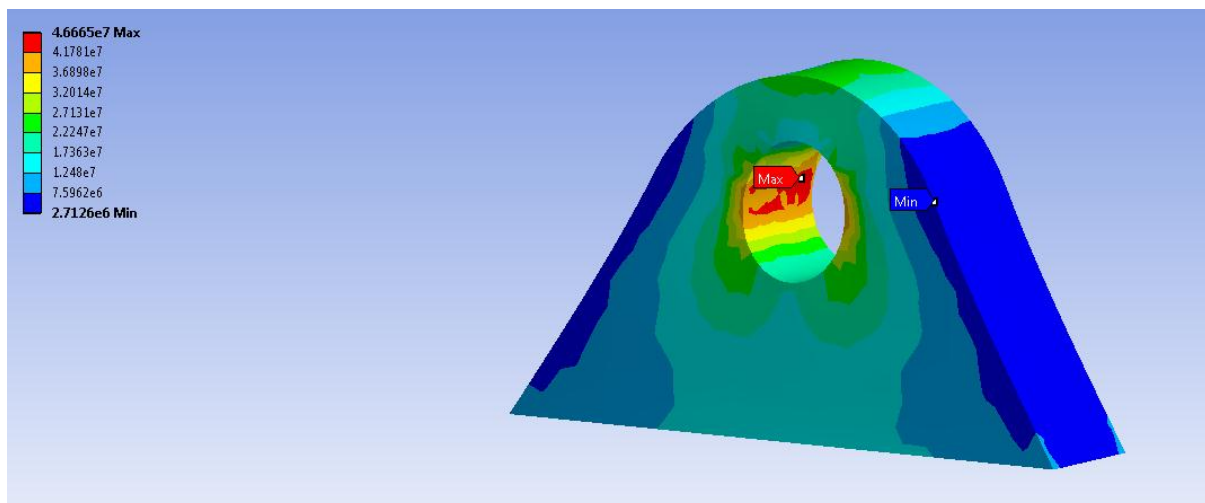
Now we need to select the type of solution we want. I want to know whether the lug can support the load, so I will do a Von Mises stress analysis and determine if the max Von Mises strength is less than the Yield Strength of the lug. (Assume for this plate 350MPa)

To do this click on Static Structural -> Solution. Then press Stress in the top left, select Von Mises.



Time to solve. Press the Solve button in the top left corner. (Little Lightning Bolt)

Once solved click on Equivalent Stress under your Solution



As we can see the maximum stress occurs where we thought it would and is about 46MPa which is less than our 350MPa. Hence the lug meets our design criteria.

ANSYS – Car Rear Window

Remember to always check models with hand calculations to ensure that your model is at least in the right ballpark. It's very easy to make a small mistake, that gives you an answer however it makes no sense whatsoever.

Fundamental rule of ANSYS:

Crap in = Crap out