

EMEM

Groupe 4

How to simulate our experiment using ANSYS

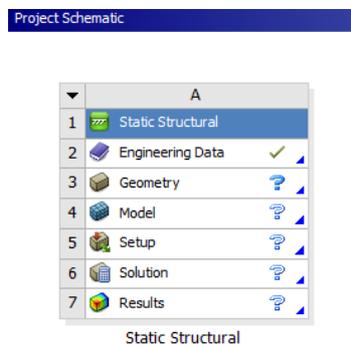
The tension experiment carried out gives us some valuables measured and extrapolated data. It is however interesting to carry out a simulation based on theoretical model to compare with our measurement and assess the accuracy of our experimental results . In this note, we will carry the Finite Element Method simulation using ANSYS solver .

1. SETTING UP

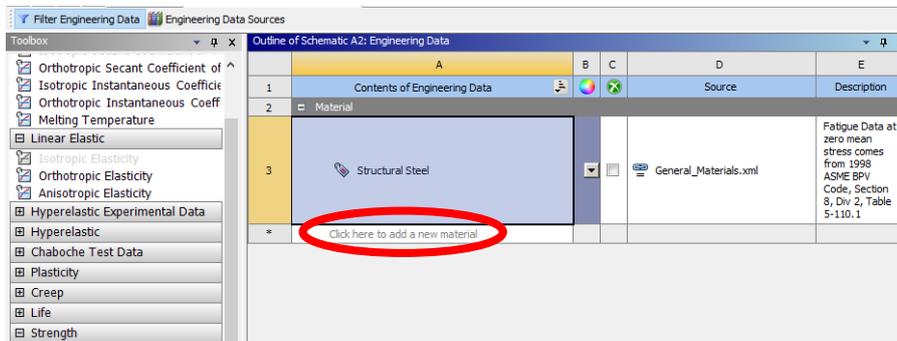
If you don't have ANSYS, is to download the student version of ANSYS , you can find it here :

<https://www.ansys.com/academic/students/ansys-student>

The First step is to create a project using the Static Structural template, which should gives you this schematic.



The second step is then to create our sample material which correspond to the Polyvinyl Siloxane. To do that we will create a custom material under Engineering Data (Double Click on it).



Then add a new material, rename it (in this example Double Elite 32) and add the following properties :

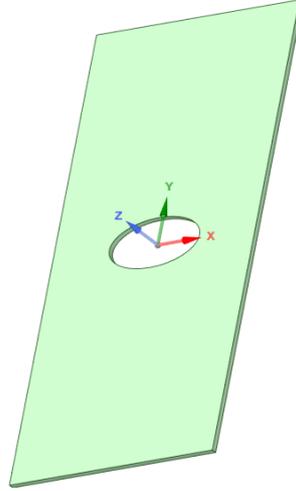
Properties of Outline Row 4: Double Elite 32				
	A	B	C	D E
1	Property	Value	Unit	
2	Material Field Variables	Table		
3	Density	1,2	g cm ⁻³	
4	Isotropic Elasticity			
5	Derive from	Young's Modulu...		
6	Young's Modulus	1,1E+06	Pa	
7	Poisson's Ratio	0,499		
8	Bulk Modulus	1,8333E+08	Pa	
9	Shear Modulus	3,6691E+05	Pa	

It's now time to head back to our project schematic workbench

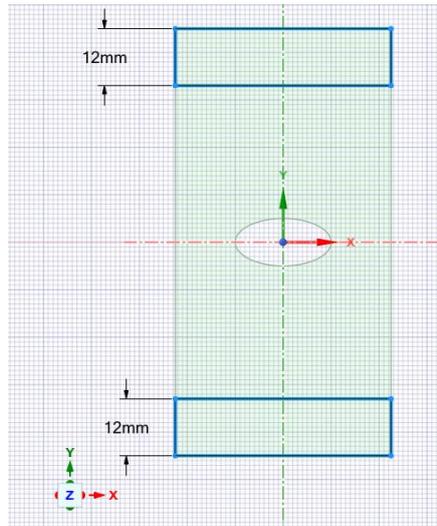
2. GEOMETRY

The second part of this tutorial consist in creating a proper geometry. Double click on Geometry, to open the ANSYS CAD software : SpaceClaim.

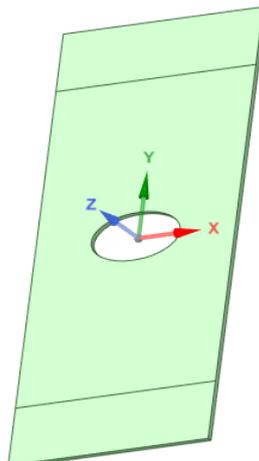
Start by creating your sample geometry by creating a centered esquisse in the XY plane. And use the pull tool to create a 3D shape with the right thickness. Here is an example for our geometry :



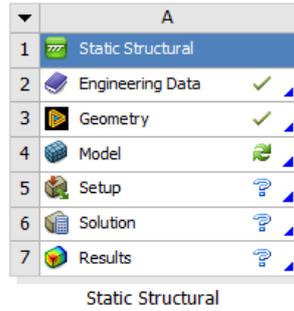
We then want to set up the 4 clamping surfaces on our sample to later applied the displacement of the Universal Testing Machine. To do that we will create the following esquisse on each face of our sample, consisting in two rectangle of 12 mm of width.



By ending the sketch editing, you should have a now modified 3D shape of your sample such as :

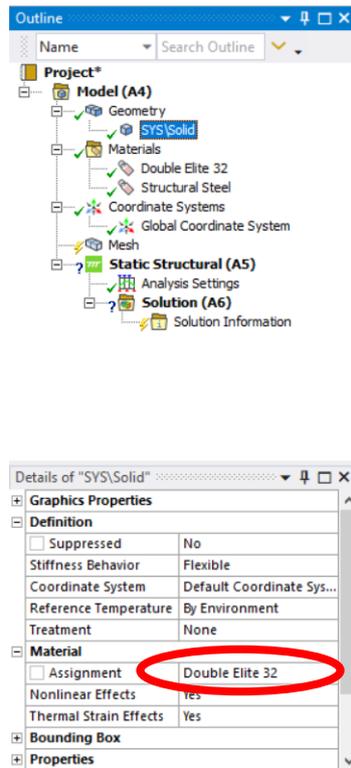


We can now save our work before quitting SpaceClaim. The project schematic should now look like that :

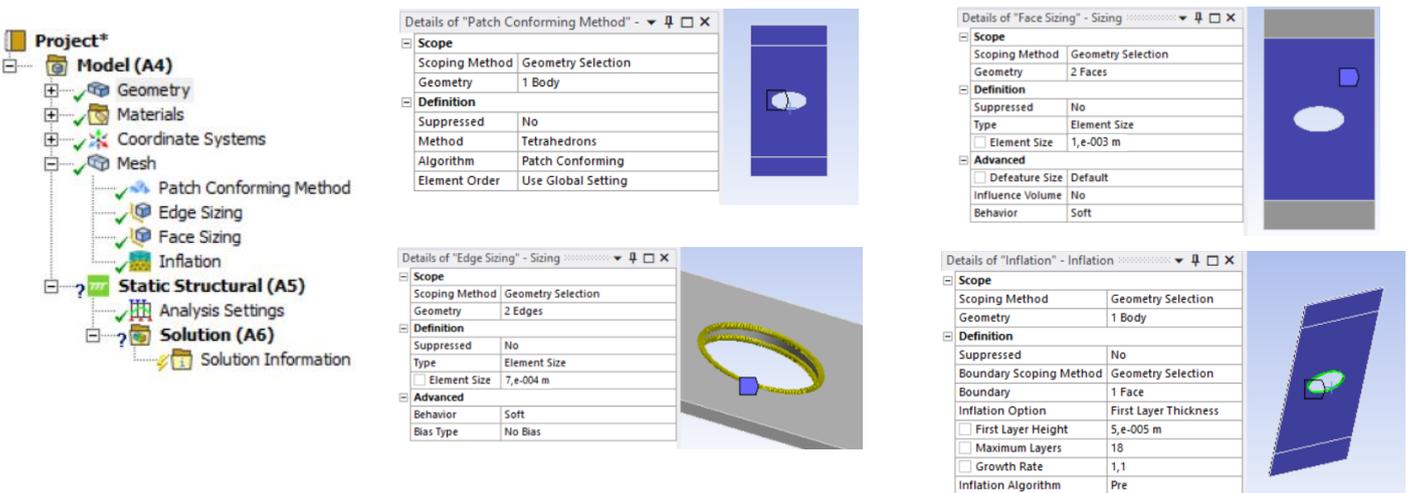


3. MESH

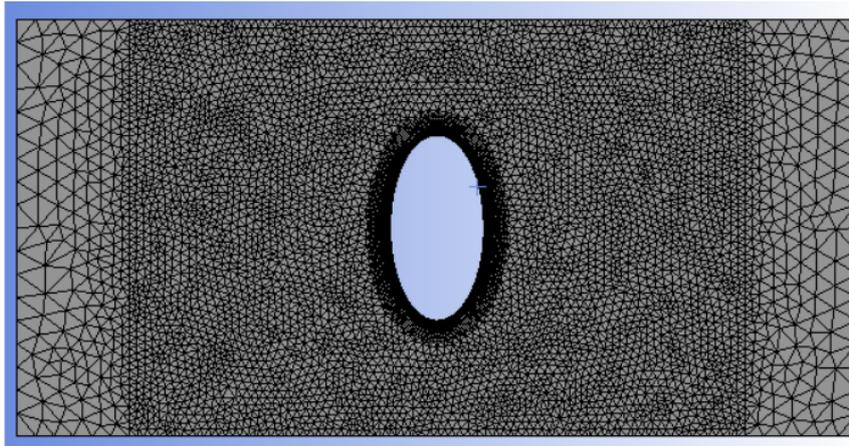
Start ANSYS Mechanical, our mesher and solver by clicking on Model. Then, the first step is to assign the previously defined material to our geometry.



The next step, is to create the mesh. For that, we will implement some simples features such as a edge and a surface sizing, a triangular method and an inflation. The goal here is to play with some parameters (mainly the element size) to have the most efficient mesh possible, not too precise (too long computation time) and not too coarse (imprecise). Here are the parameter used :



Then click on generate to create the mesh and visualize it by enabling “Show Mesh” on the Display tab. This the resultant mesh in our case :

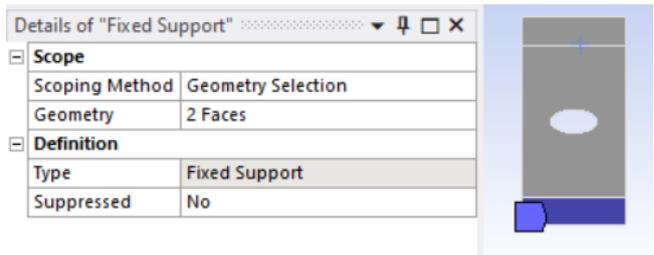


Finally, a convergence analysis can be done to assess the accuracy of the mesh by varying the different mesh parameters and investigate the changes induced on the simulation results.

4. CONDITIONS

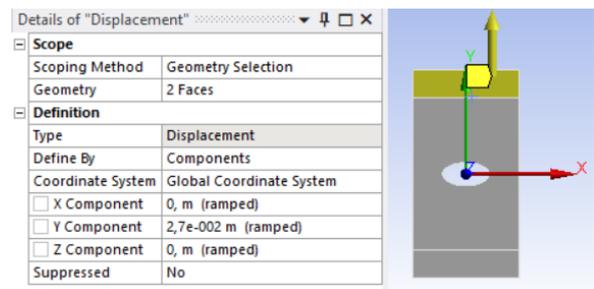
Then we need to setup our case by implementing the loading created during the test. To do that we will fixed the displacement of the lower clamp and force the vertical displacement in the upper clamp.

Insert a fixed support on the two lower clamping surface :

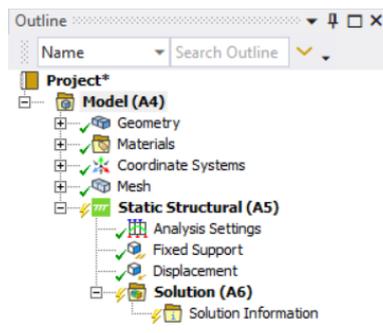


And insert a displacement on the two upper clamping surface :

Note that you have to force a constant value for each component of the displacement.



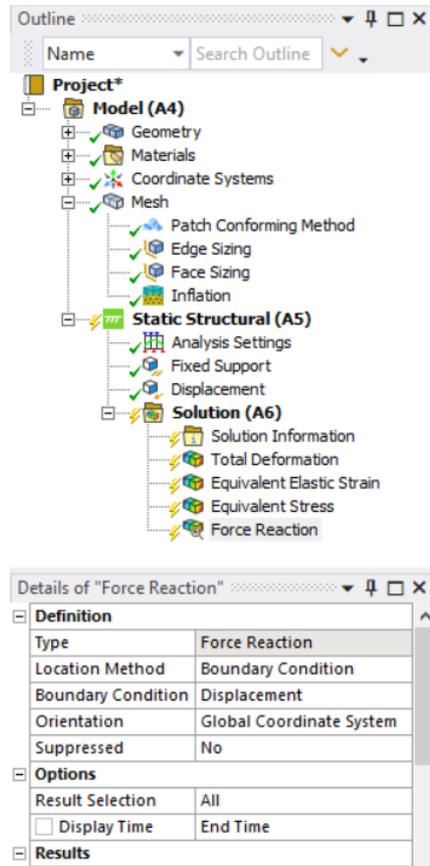
The outline window should now looks like that :



We are now ready to implement the solution results we are interested in.

5. SOLUTION

In our set up, we are interested in the deformation field, the Equivalent (von-Misses) strain field and the Equivalent (von-Misses) stress field. Moreover, by setting a Force Reaction probe on the upper surface to asses the accuracy of our simulation. The outline windows should now look like :



We are now ready to solve our case by pressing F5.