

# Simulating a traction test on COMSOL 5.6

Wanting to compare the results of the experiment with that of a simulation, we used COMSOL 5.6 to simulate a traction test. In order to obtain results, a few considerations had to be made, which will be discussed in this document.

## Sample geometry

Mould dimensions were used for the external dimensions of the sample giving a 45x95x2mm rectangle. An ellipse was cut inside with semi-axes  $a = 10[mm]$  and  $b = 5[mm]$ . The final geometry is displayed on Figure 1 just below.

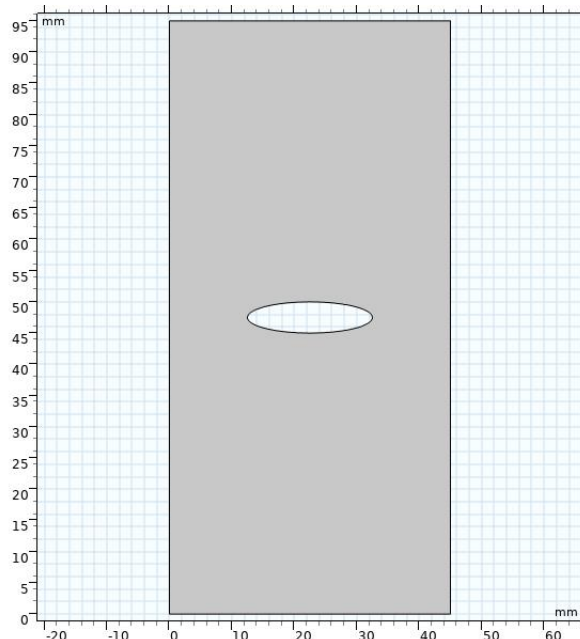


Figure 1: Geometry of the sample

As you can see, the geometry is kept in 2D, but we will set the thickness of the plate in the **Solid Mechanics** node relative to the physics of the problem.

## Material characterisation

The material properties of Elite Double 32 Zhermack were looked up, and we selected the following:

Material Property	Value
Density $\rho$	1050 [ $kg/m^3$ ]
Poisson ratio $\nu$	0.499
Young's Modulus $E$	1.3 [ $MPa$ ]
Lamé parameter 1 (shear modulus) $\mu$	$4.44 * 10^5$ [ $Pa$ ]
Bulk modulus $K$	1.75 [ $GPa$ ]
Lamé parameter 2: $\lambda = K - 2/3\mu$	1.749 [ $GPa$ ]

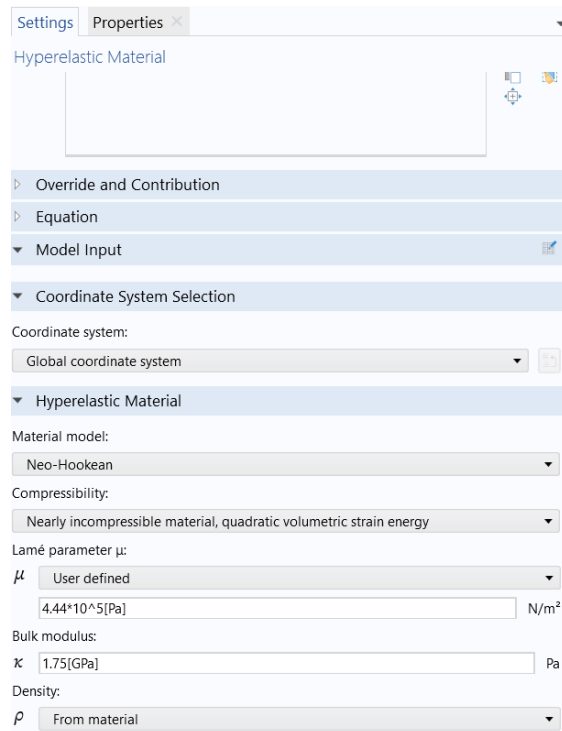
### References:

[1] <https://www.azom.com/properties.aspx?ArticleID=920>

[2] Pai Wang et al., Harnessing Buckling to Design Tunable Locally Resonant Acoustic Metamaterials, PHYSICAL REVIEW LETTERS, July 2014, <https://journals.aps.org/prl/abstract/10.1103/PhysRevLett.113.014301>

# Physics

- Solid Mechanics
- Initial Value: zero for all displacement fields and structural velocity fields
- Hyperelastic material
  - Neo-hookean model
  - Nearly incompressible, quadratic volumetric strain energy (because Poisson ratio is very close to 0.5)
  - Use  $\mu, K, \rho$  to define the material



- Prescribed displacement on the bottom boundary of the rectangle: 0 in the vertical direction
- Prescribed displacement on the top boundary of the rectangle: impose a parameter that will vary. Set the vertical displacement to  $d$  for the displacement imposed by the traction machine, we will do an auxiliary sweep to do the traction test. For this, define a parameter in the **Global definitions**. We will define the sweep in the **Study** part later on.

Parameters

Label: Parameters 1

Name	Expression	Value	Description
a	10[mm]	0.01 m	semi-major axis
b	2.5[mm]	0.0025 m	semi-minor axis
L	95[mm]	0.095 m	Length
H	45[mm]	0.045 m	Width
u_traction	1[mm/s]	0.001 m/s	traction machine velocity
density	1050[kg/m^3]	1050 kg/m <sup>3</sup>	density
Lame_mu	4.44*10^5[Pa]	4.44E5 Pa	shear modulus
displacem...	0.1[mm]	1E-4 m	displacement parameter

Now it is important to set the stress/strain situation on the 2D plate. This is done in the Solid Mechanics general node as follows:

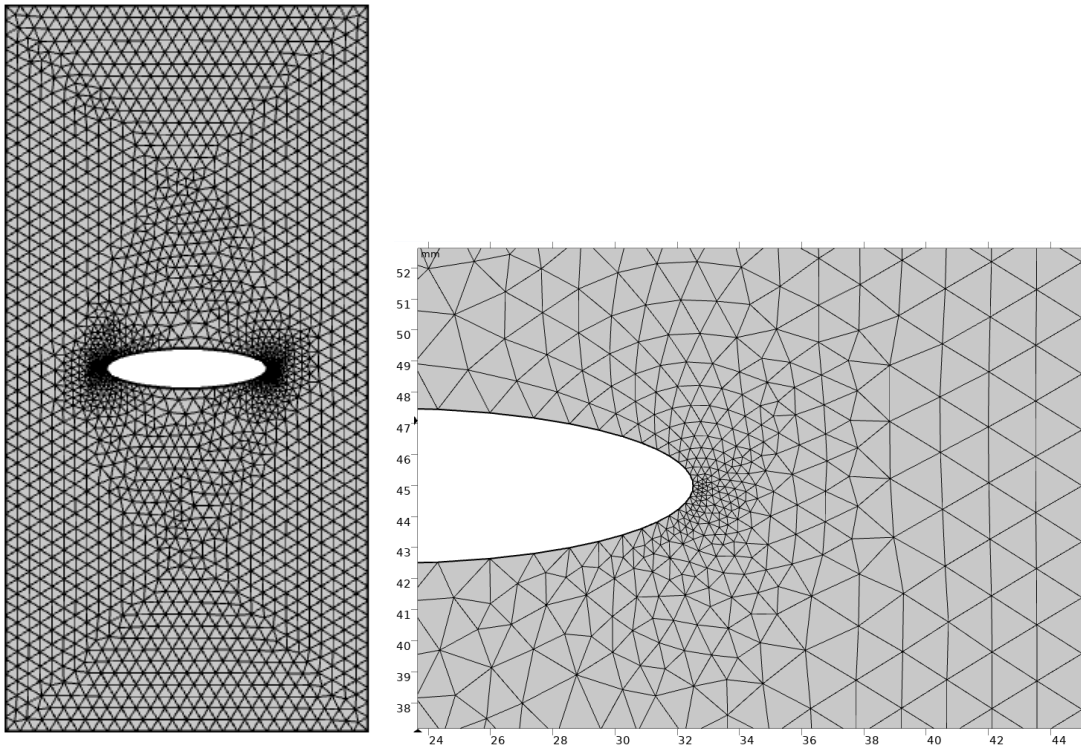
- Select Plane Stress
- Define the thickness of the sample here (see on the following image)



## Mesh

The most important is to refine the mesh around the elliptic hole in order to capture the stress in an accurate way in this region of interest.

- Physics controlled mesh
- Extra fine



## Study definition

- Select Stationary study
- Introduce a parametric sweep on the displacement imposed on the top boundary of the sample
  - Select  $d$  as the sweeping parameter
  - Define unit: [mm]
  - Define parameter value list: start: 0mm, step: 0.1mm, end: 8mm  
 = Range between 1 and 8 [mm], which corresponds to the elongation of our sample during the experiment, with an imposed velocity of 1 [mm/s] this corresponds to approximately 8s of experiment.  
 = Now, the camera takes 10 frames per second, so we need to take steps of 0.1 mm

See on the image below:

Settings Properties

Parametric Sweep

Compute Update Solution

Label: Parametric Sweep

Study Settings

Sweep type: Specified combinations

Parameter name	Parameter value list	Parameter unit
displacement (d)	range(0[mm],0.1[mm],8[mm])	mm

Memory settings for jobs

Keep solutions: All

Output While Solving

Plot

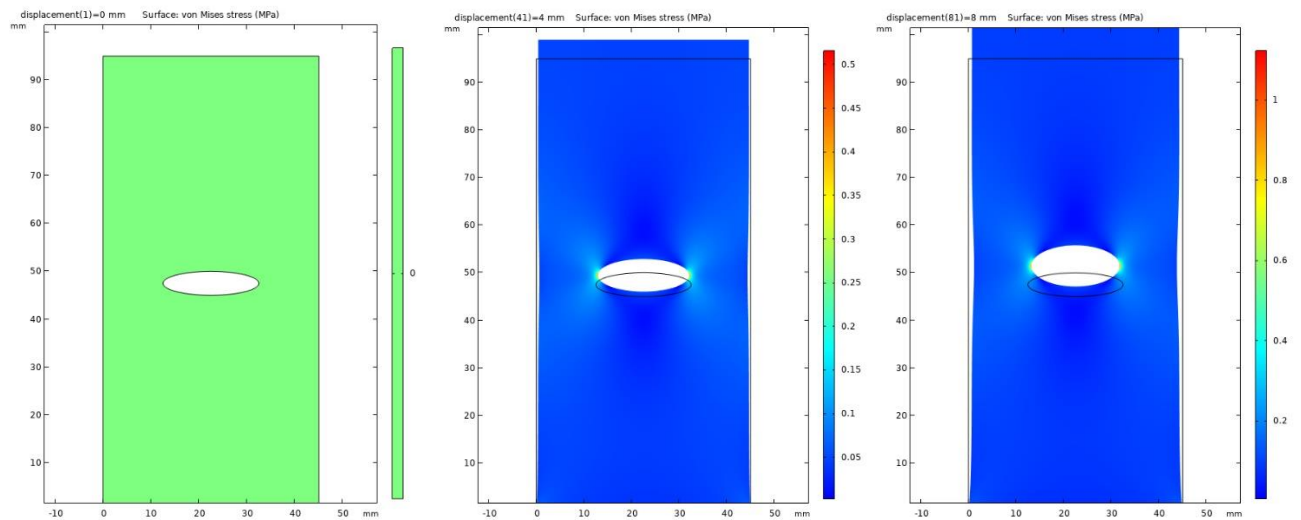
Plot group: Stress (solid)

Probes: All

Plot every step so that you can see the evolution of the stress in the sample during the traction test

## Results

Von mises Stress in the sample for three of the displacement values (0mm, 4mm and 8mm)



Comparison of the maximum stress (at the tip of the elliptic crack) and the stress applied at infinity (on the top boundary of the sample):

