Exercise 1:
COMSOL tutorial on flexure-based linear stage stiffness calculation

Context: We study a flexure-based linear stage composed of two leaf-springs (blades), we want to calculate the stiffness, the mechanical stress and the displacement of the structure for a given applied force. We will see in this simple example the basics of COMSOL software with the Structural Mechanics module. The studied structure is shown on the Figure 1.

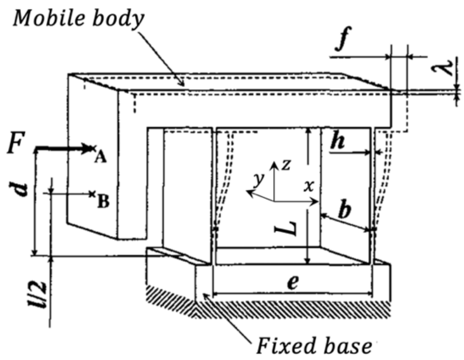
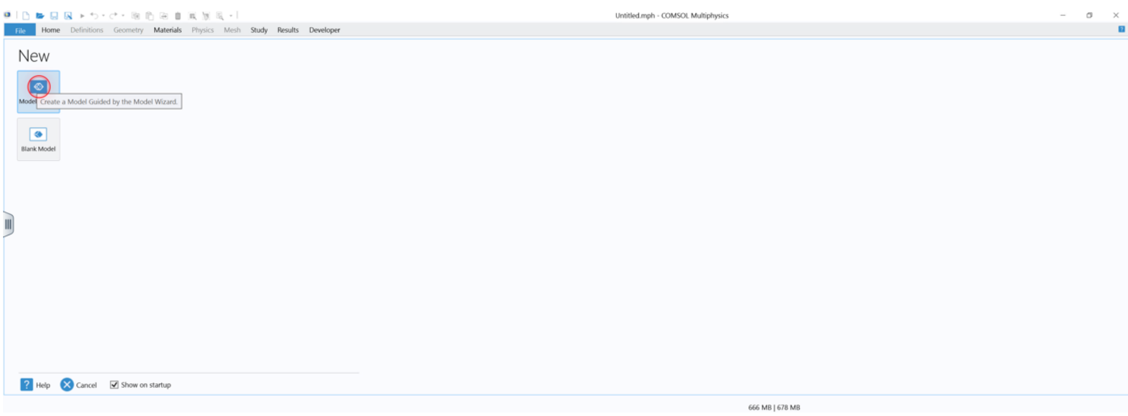


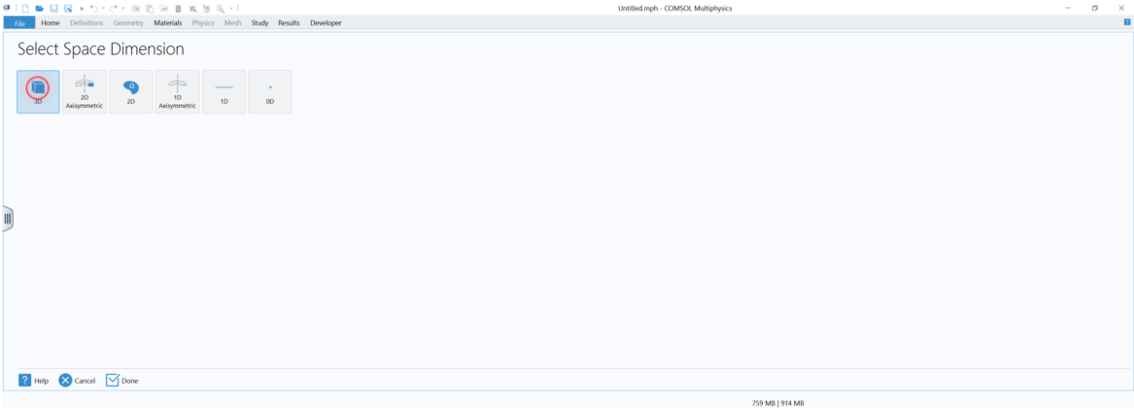
Figure 1: Flexible linear stage.

1. On the *VDI interface*, start COMSOL (Classkit License COMSOL Multiphysics 6.2).

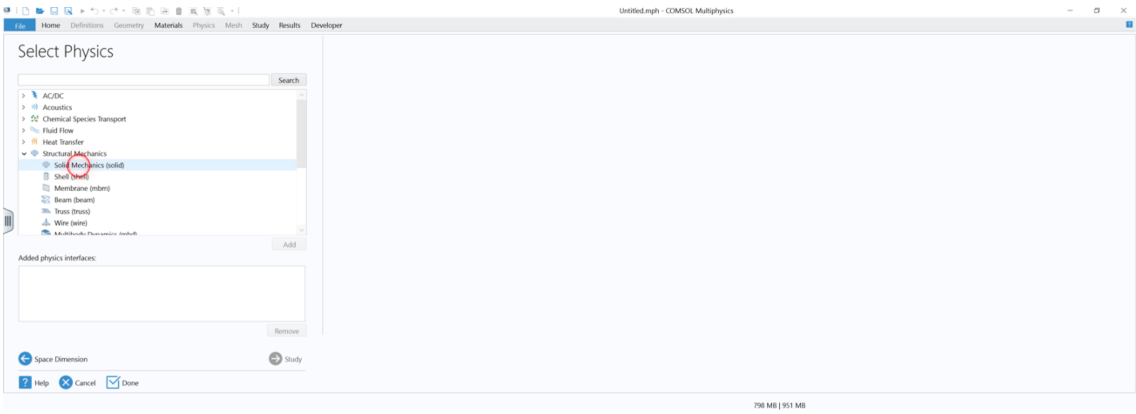
2. On the *New* model menu, select *Model Wizard.*



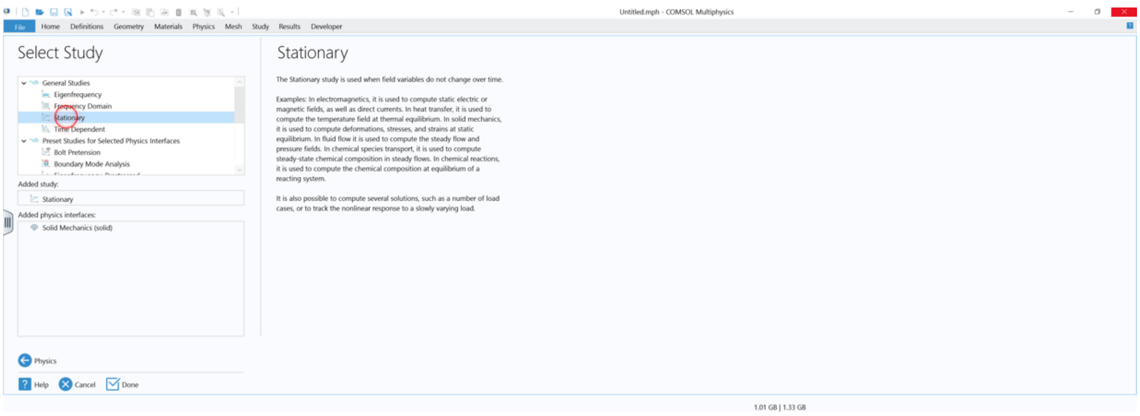
3. On the *Select Space Dimensions* menu, select *3D.*



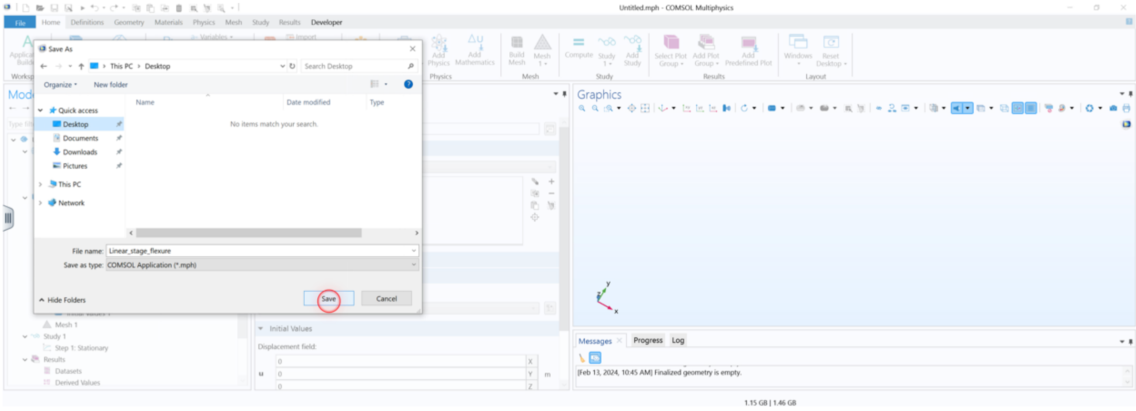
4. On the *Select Physics* → *Structural Mechanics* → *Solid Mechanics (solid).* Then press *Add*, then *Study.*



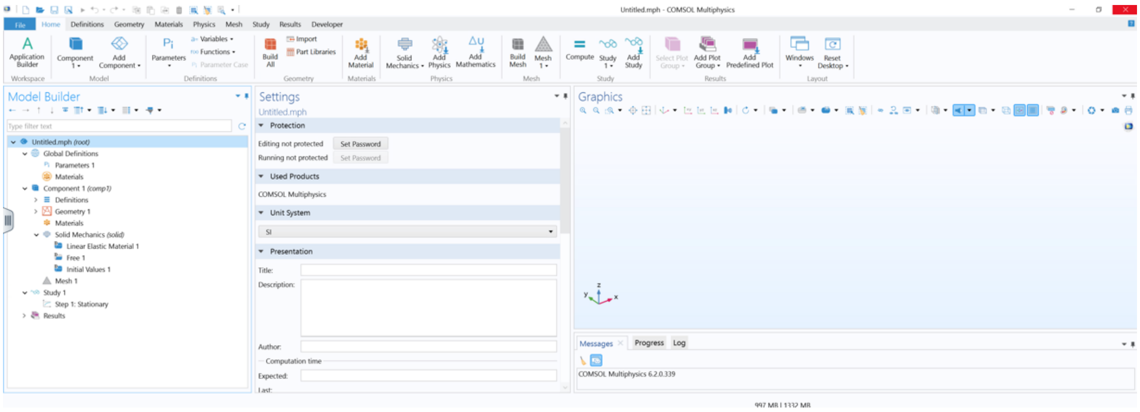
5. On the *Select Study* → *Stationary*, then *Done.*



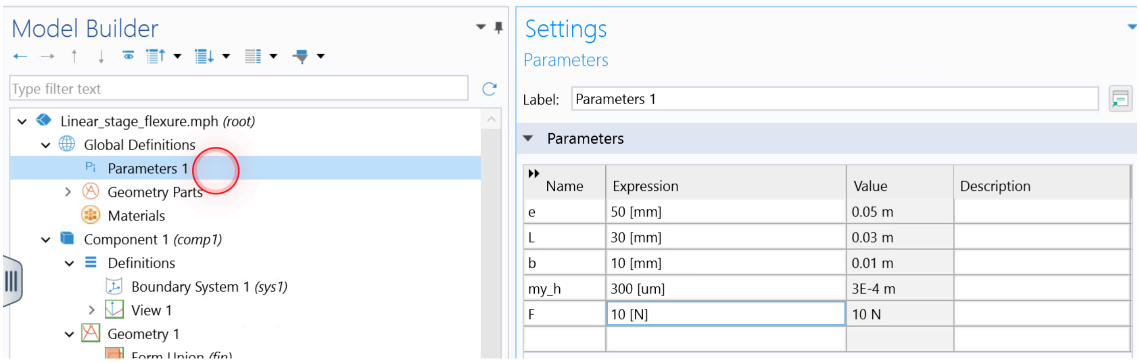
6. In *File* → *Save as..*, save the model with the name *linear_stage_flexure*.



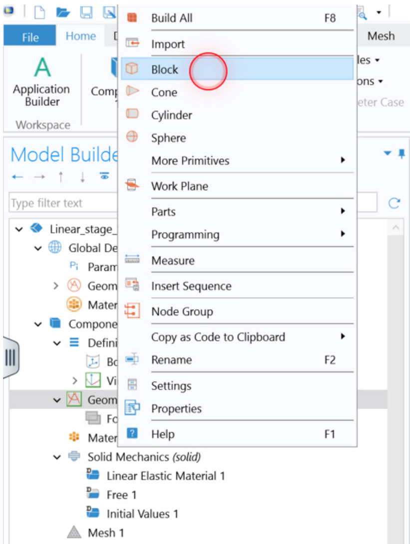
7. Explore briefly the COMSOL interface, the model builder and the ribbon bar.



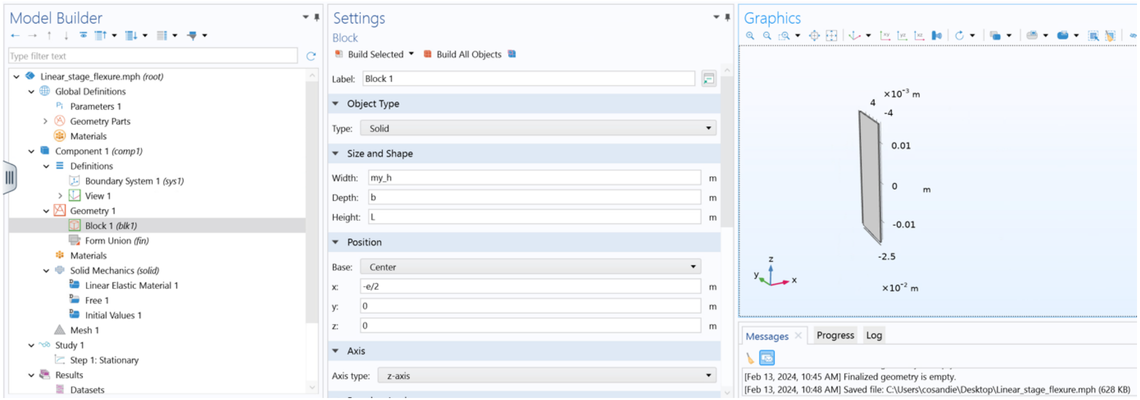
8. In the *Model Builder* → *Global Definitions* → *Parameters 1*, enter the parameters and the values as listed below. (N.B.: *h* is named *my_h* as "*h*" is reserved for the Planck constant in COMSOL).



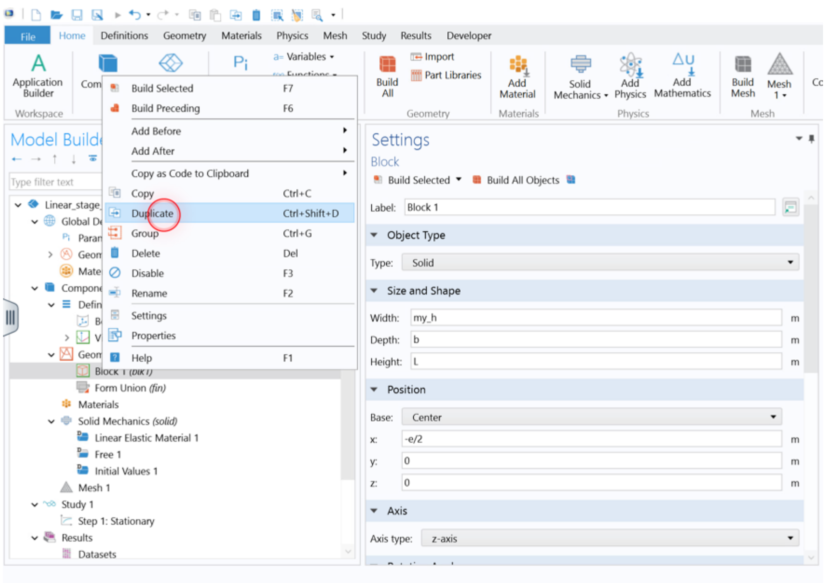
9. In *Component 1* → *Geometry*, right-click and add a *Block*.



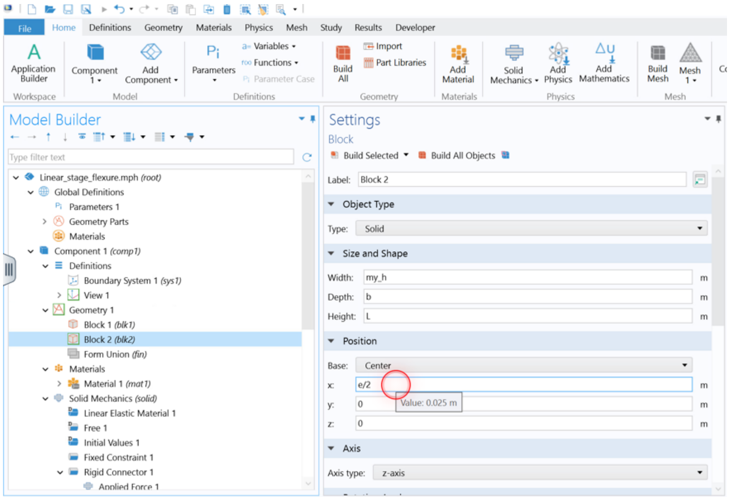
10. In the settings of the *Block*, enter the parameters as below. Click on *Build Selected*.



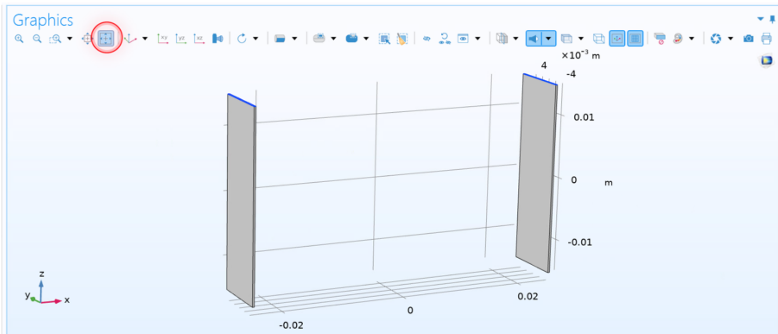
11. Right click on *Block 1* → *Duplicate*.



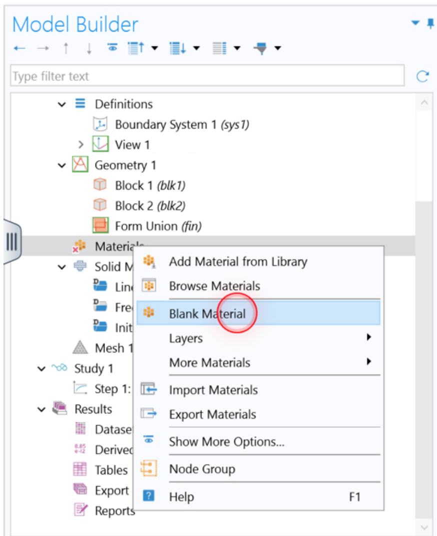
12. In *Block 2*, change the sign of the *x* base position from -e/2 to e/2. Click on *Build Selected.*



13. Become familiar with the graphical interface. Hold the left click for pivoting the geometry. Hold the middle click to zoom in and out and hold the right click to translate the geometry. To reset the view, click on the button shown below.



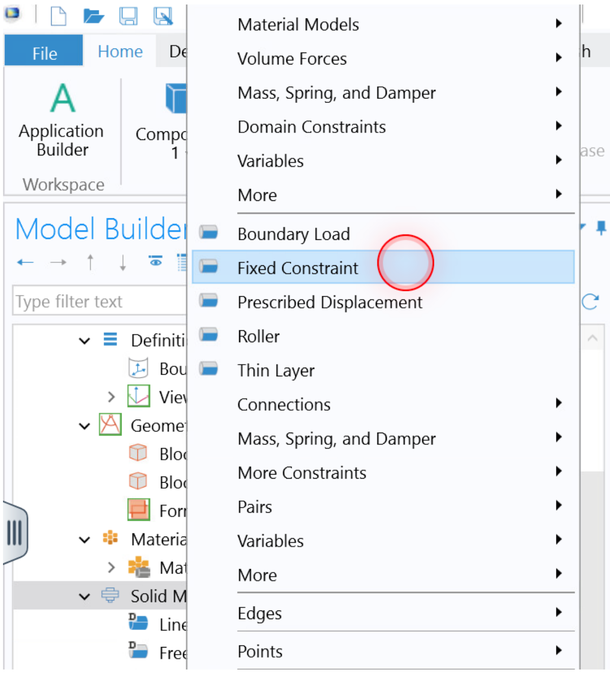
14. In the *Model Builder*, right click on *Material*, click on *Blank Material.*



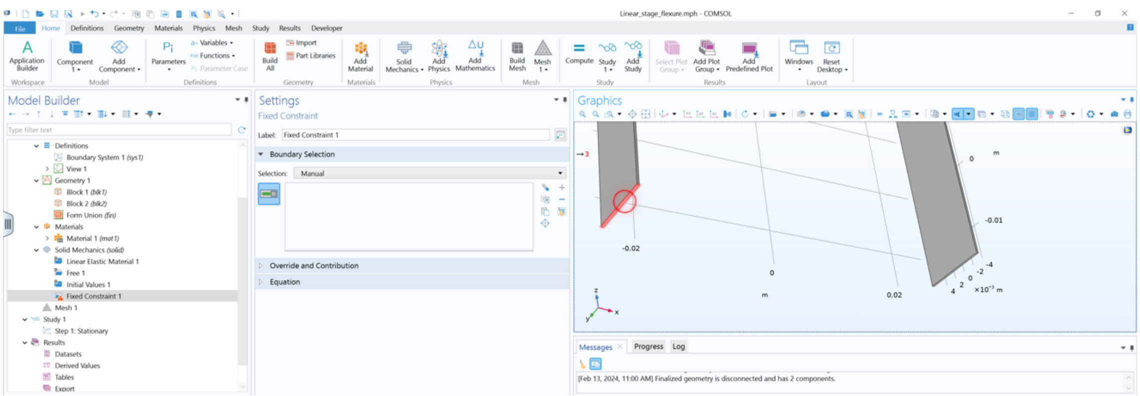
15. In *Settings*, add the following parameters. They correspond to stainless steel.

	Property	Variable	Value	Unit	Property group
☑	Young's modulus	E	200e9	Pa	Young's modulus and P...
☑	Poisson's ratio	nu	0.3	1	Young's modulus and P...
☑	Density	rho	7800	kg/m³	Basic

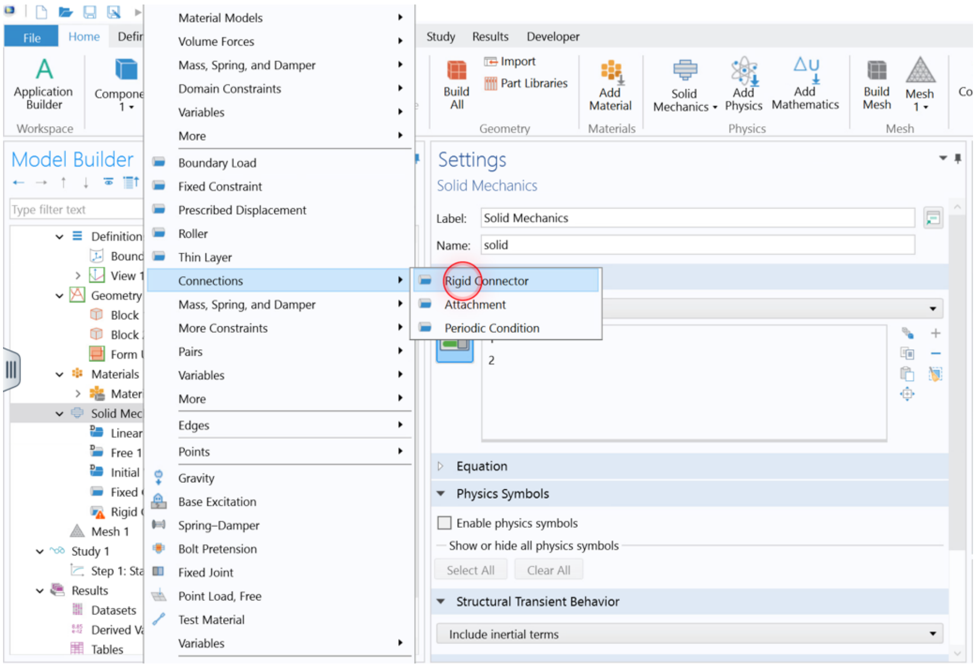
16. In the *Model Builder*, right click on *Solid Mechanics*, add *Fixed Constraint.*



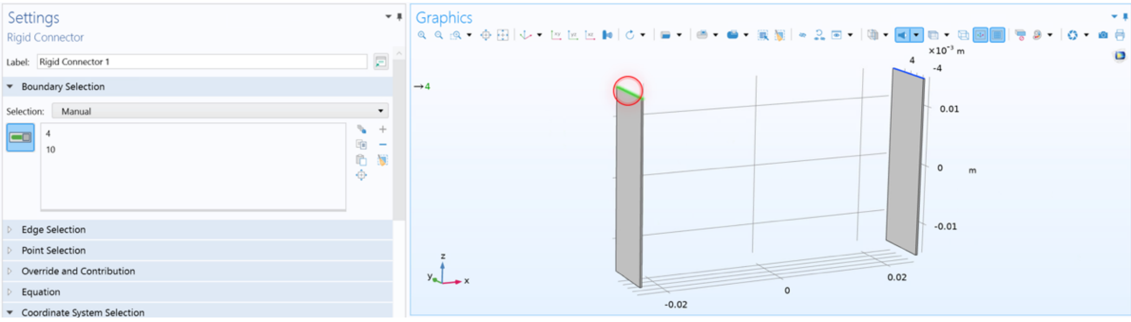
17. On the *Settings pane* → *Boundary Selection*, make sure the *Activate Selection* button is turned on and select the lower faces (blade's roots) towards the negative z-direction.



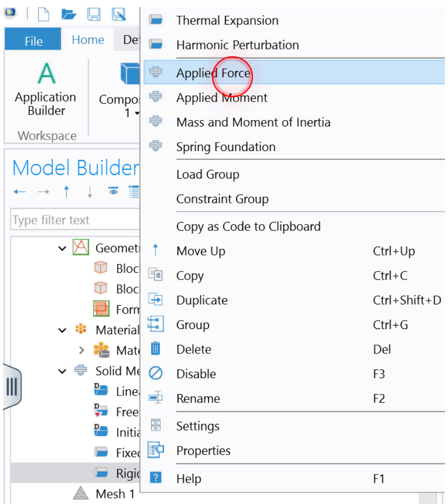
18. Right click in *Solid Mechanics* → *Connections* → *Rigid Connector.*



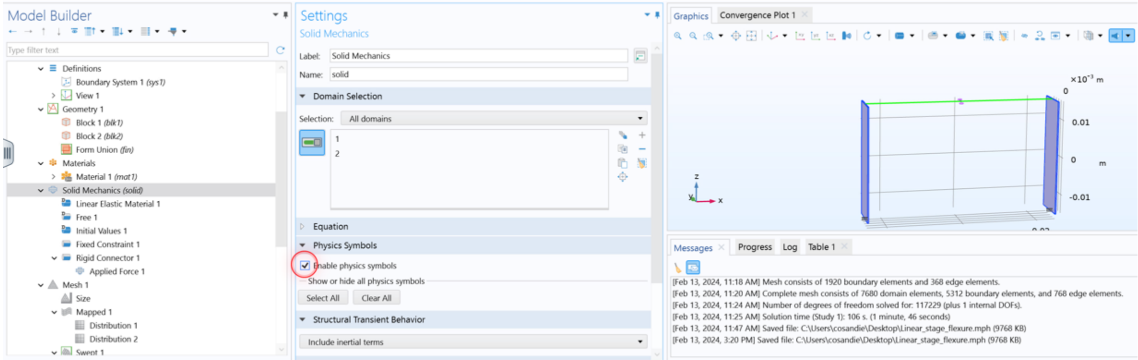
19. Select the upper faces (axis z +) of the blades.



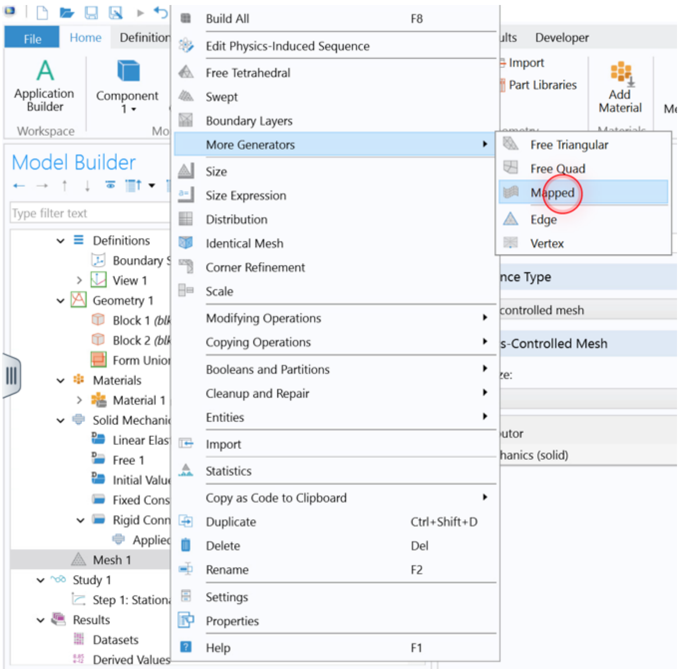
20. Right click on the *Rigid Connector* → *Applied Force.* In the *Settings* pane, put *F* as the x-component of the applied force.



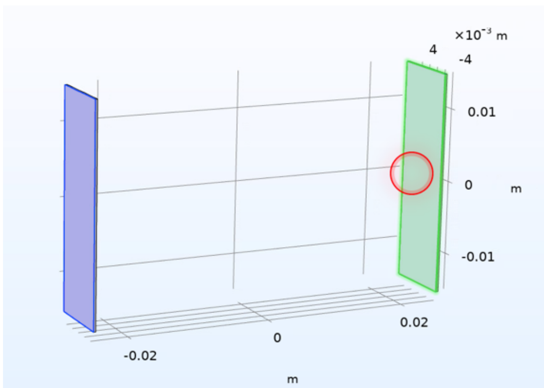
21. In the *Model Builder*, click on *Solid Mechanics*, then in the *Settings* pane, tick on the box *Enable physics symbols*. That makes appear the rigid connector linking the flexures and the fixed constraints.



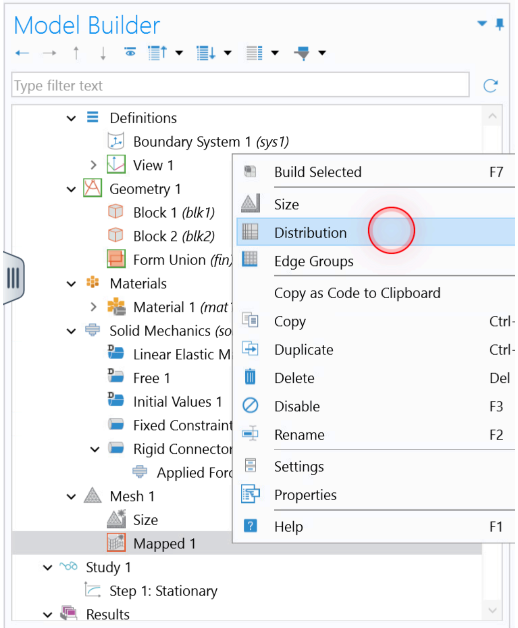
22. In the *Model Builder*, right click on *Mesh 1* → *More Generators* → *Mapped*.



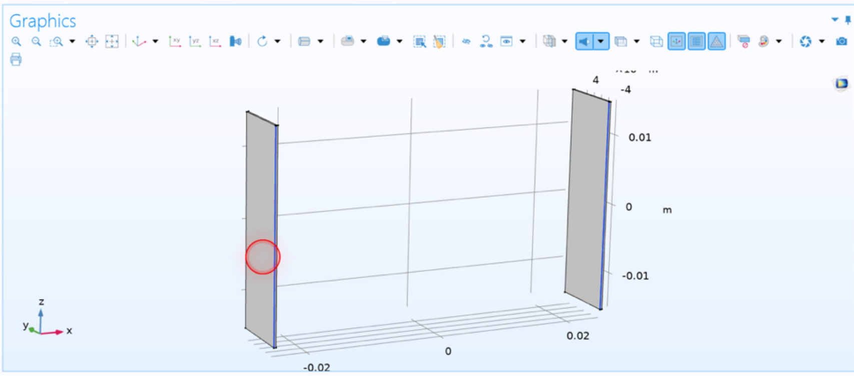
23. On the *Graphics* window, select one large face on each of the two blades.



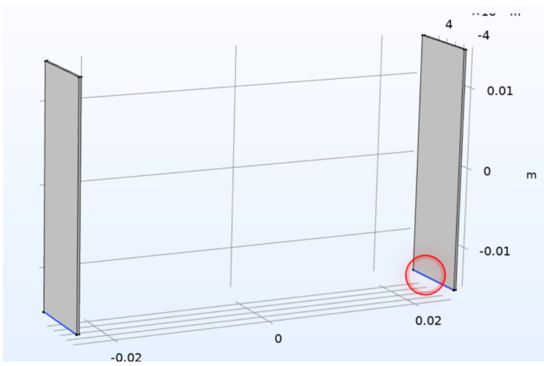
24. Right click on *Mapped 1* → *Distribution*.



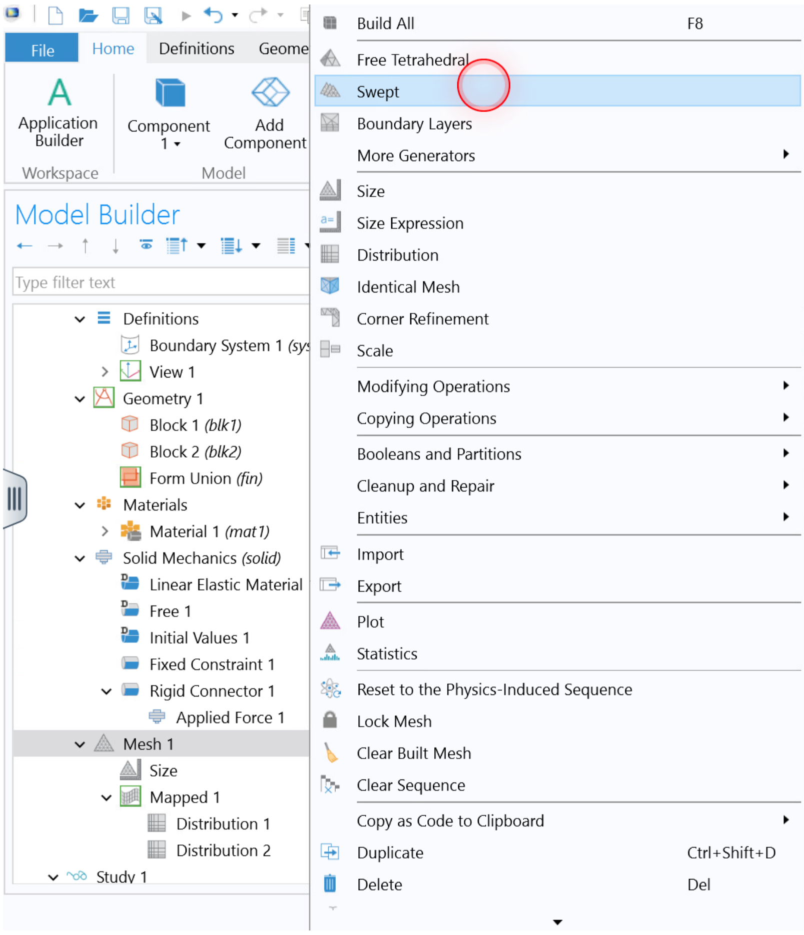
25. Select one long edge on each blade (one that belong to the faces that you selected on point 23). In *Number of elements*, in the *Settings* pane, put 80.



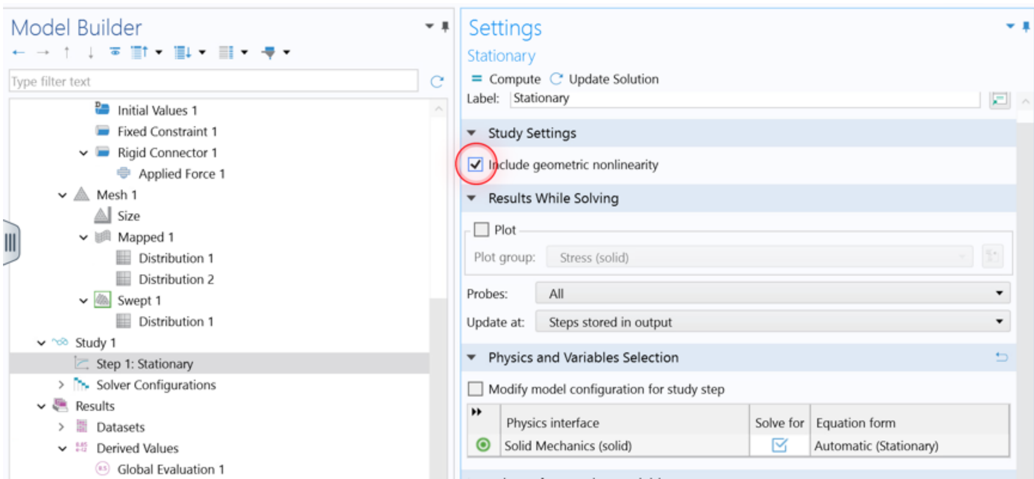
26. Right click on *Mapped 1* → *Distribution*. Select one middle edge on each blade (one that belong to the faces that you selected on point 23). In *Number of elements*, in the *Settings* pane, put 12. Click on *Build Selected*.



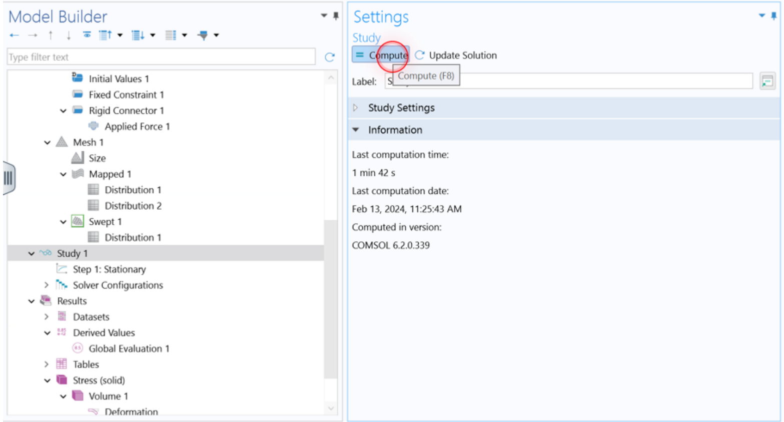
27. Right click on *Mesh 1* → *Swept*. Right click on *Swept 1* → *Distribution*. Make sure the two blades are selected. In *Number of elements*, in the *Settings* pane, put 4. Click on *Build Selected*. Verify visually that the volumes of both blades are fully meshed.



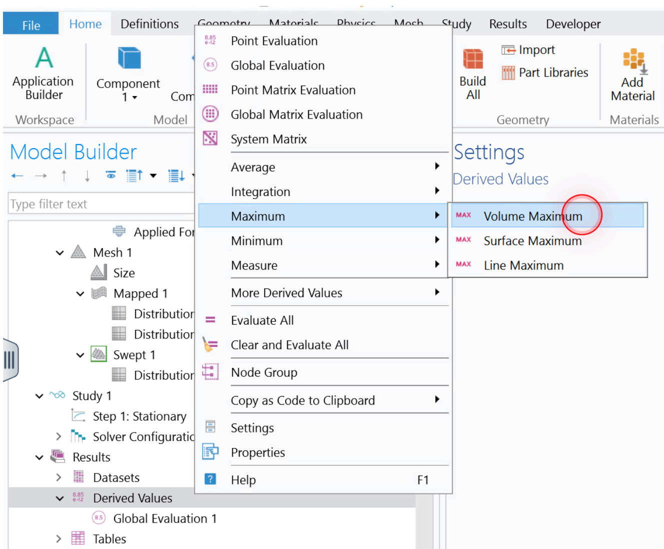
28. In *Study 1* → *Stationary*, tick the box *Include geometric nonlinearity*.



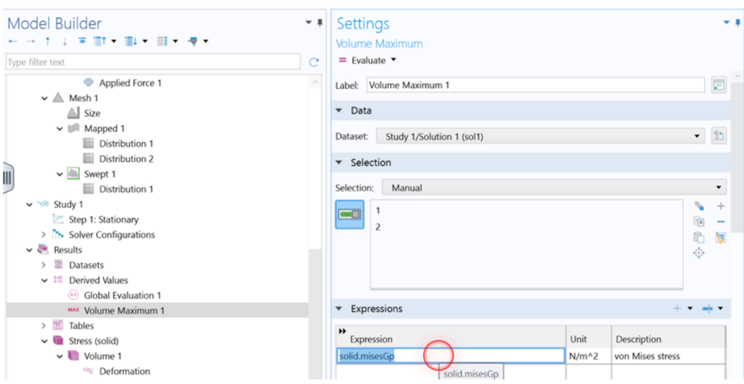
29. *Study 1* → *Settings pane* → *Compute*. Wait a couple of minutes!



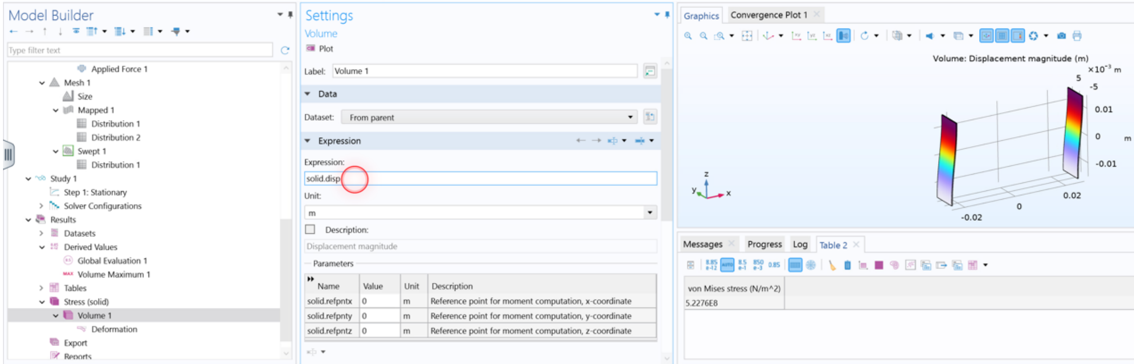
30. We will now evaluate the results! On the *Graphic* pane, evaluate the maximum von Mises stress value and location. Right click on *Derived Values* → *Maximum* → *Volume Maximum.*



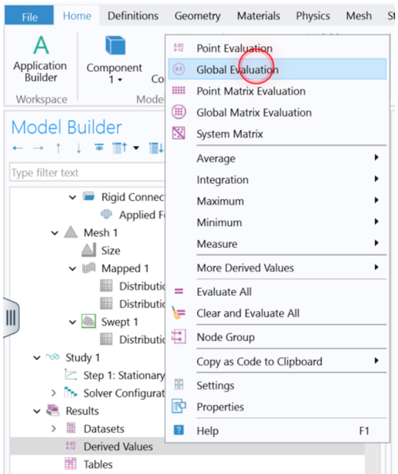
31. In the *Settings* pane, select the two blades bodies and in *Expression*, type *solid.misesGp* → *Evaluate*. Read the maximal stress in the *Table,* below the *Graphic* pane.



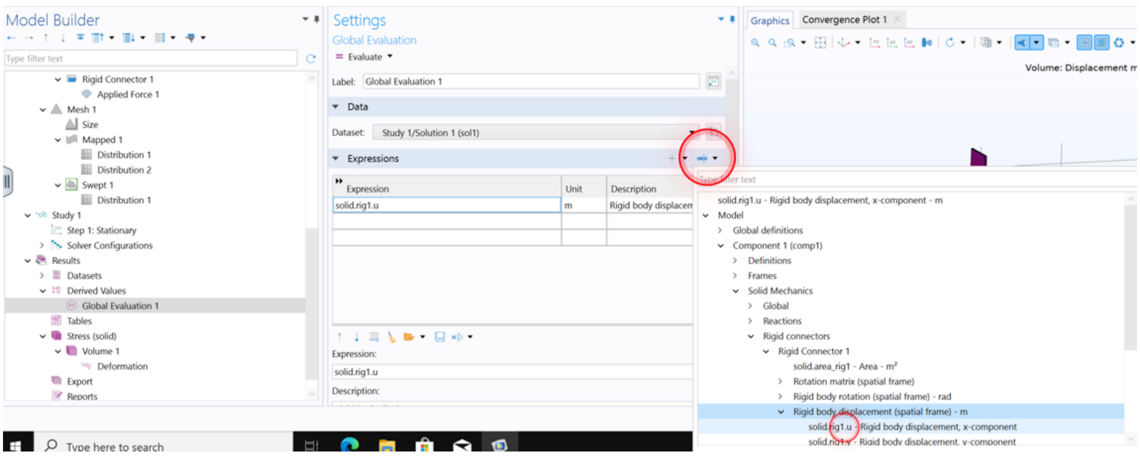
32. In *Results* → *Stress* → *Volume 1*, replace the expression by *solid.disp* → *Plot*. Asses visually the displacement.



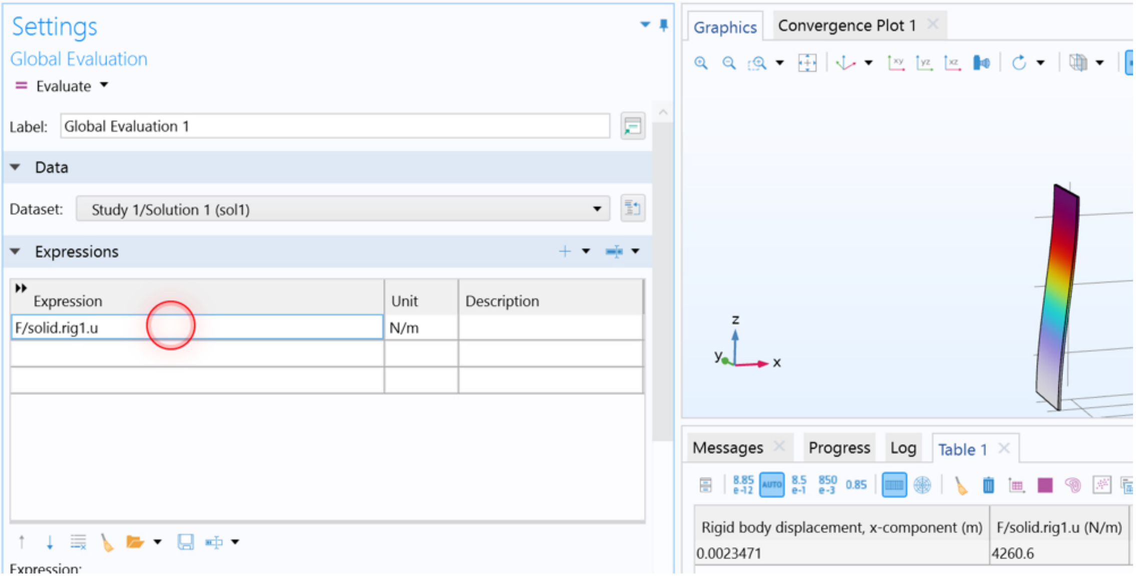
33. Right click on *Derived Values* → *Global Evaluation*.



34. In *Expression*, click on the little button on the right → *Component 1* → *Solid Mechanics* → *Rigid connector* → *Rigid Connector 1* → *Rigid body displacement* → *solid.rig1.u*. Evaluate and see the displacement value of the rigid connector 1 along the x-axis in the *Table*.



35. Finally, replace the expression *solid.rig1.u* by *F/solid.rig1.u* and evaluate. The stiffness of the linear stage is calculated along the *x*-axis.



Congratulations, you have now reached the end of this COMSOL tutorial and you are now able to model and simulate a basic mechanical structure!!