EPFL

Exercise 3:
COMSOL tutorial on frequential analysis

Context: We study the vibrations modes of blade samples in two different materials: stainless steel (E_{ss} = 200 GPa) and titanium (E_{Ti} = 114 GPa). For each material, three sample blades with different thicknesses are tested: 300 um, 350 um and 400 um. Their length is 40 mm and their width is 10 mm. A mass M = 15 g is located at the tip of each blade. We will **calculate analytically** the **three first modes of vibration** along the directions X, Y and Z (Figure 1) and compare with the FEM results given by the **eigenfrequency** analysis for the **three stainless-steel sample blades**. Then we calculate the maximum **von Mises stress** for each of the **six sample blades** when a displacement of the tip of f = 10 mm is imposed (Figure 2). Note that the results obtained here will be used for fatigue tests on a later exercise.

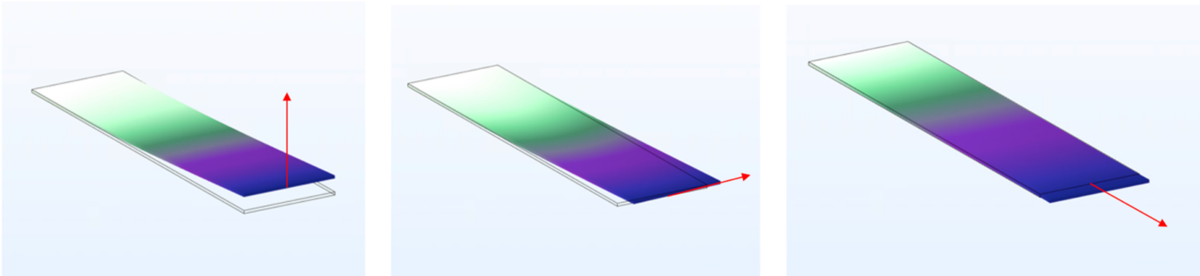


Figure 1: The three first vibrations modes of a blade with a point mass at its extremity.

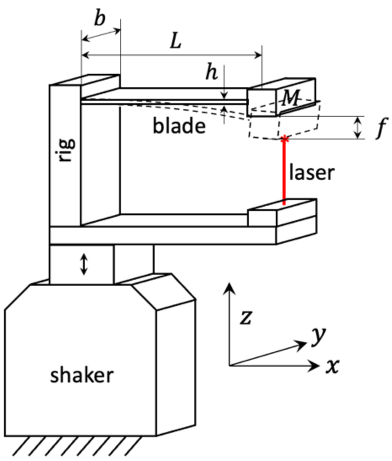
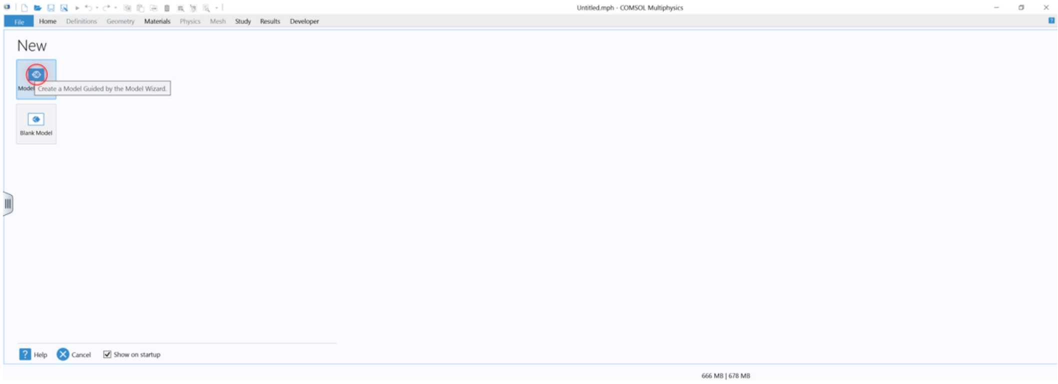


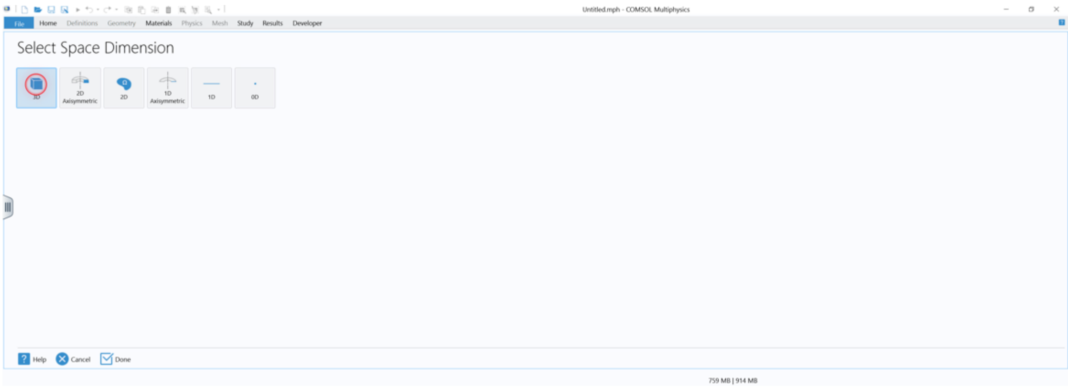
Figure 2: Setup for measuring the vibration of a blade.

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

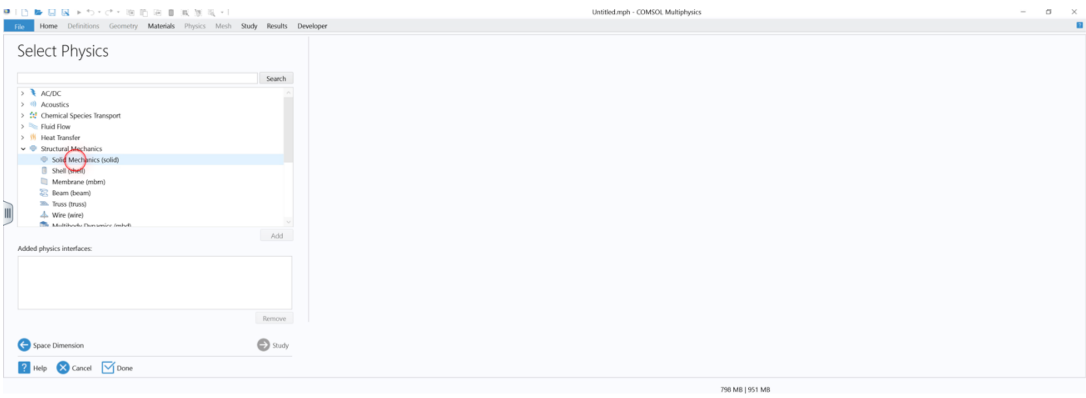
2. On the *New* model menu, select *Model Wizard.*



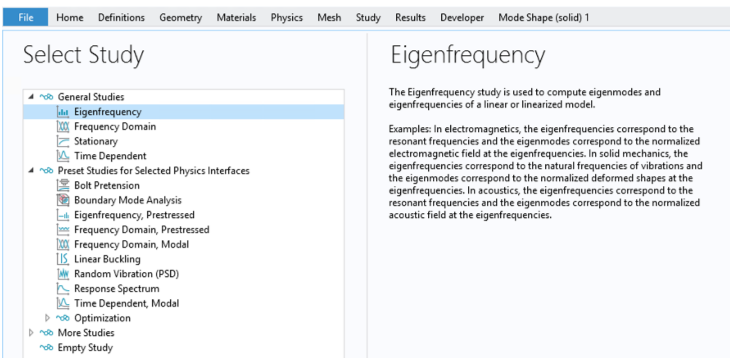
3. On the *Select Space Dimensions* menu, select *3D.*



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



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

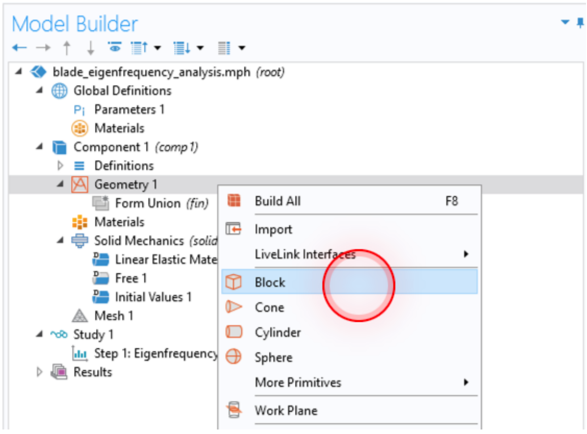


6. In *File* → *Save as..*, save the model with the name *blade_eigenfrequency_analysis*.

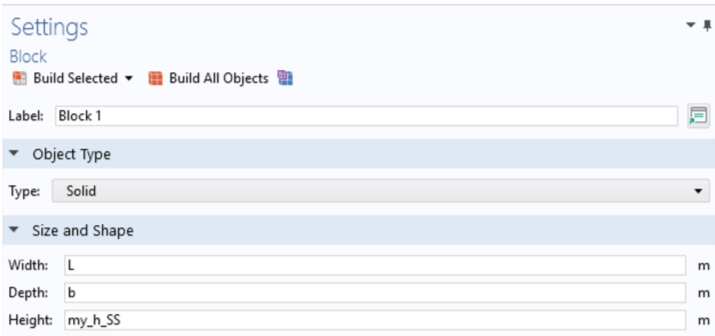
7. In the *Model Builder* → *Global Definitions* → *Parameters 1*, enter the parameters and the values as listed below.

Name	Expression	Value	Description
L	40 [mm]	0.04 m	
b	10 [mm]	0.01 m	
my_h_SS	300 [um]	3E-4 m	
my_h_Ti	200 [um]	2E-4 m	
dz	10 [mm]	0.01 m	

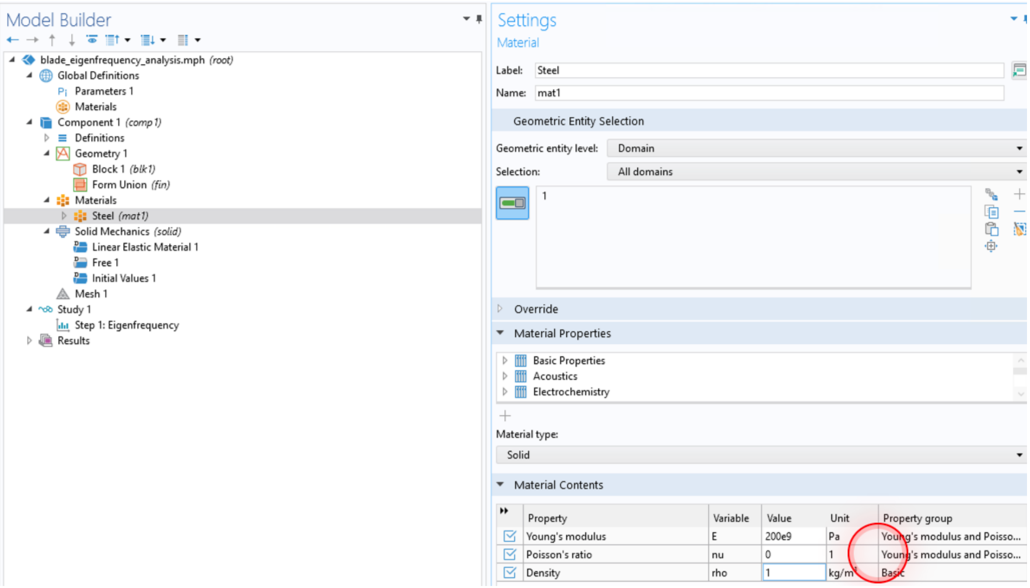
8. In *Component 1* → *Geometry 1*, right-click and add a *Block*.



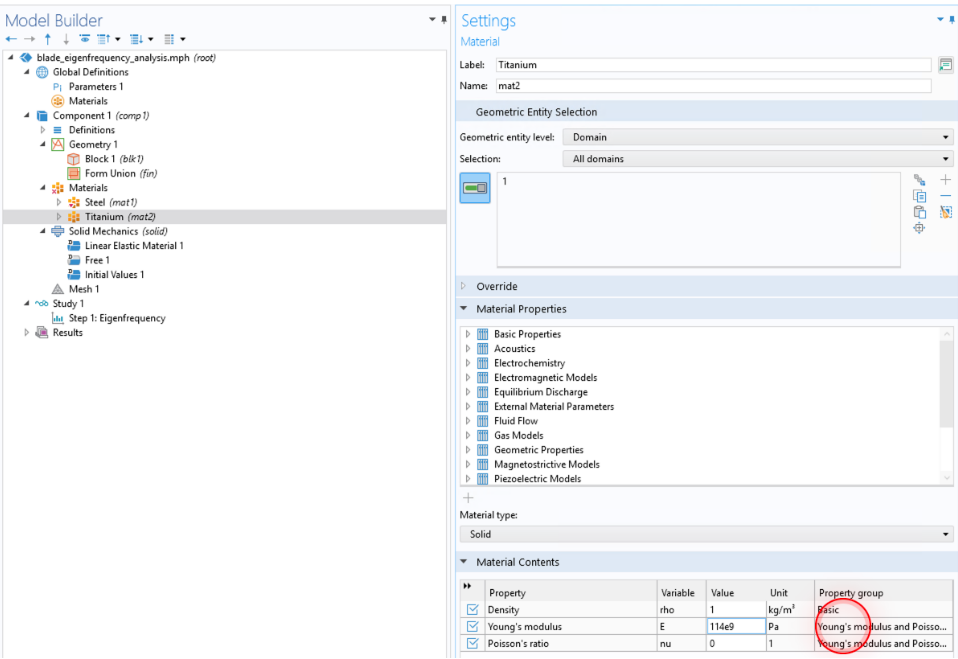
9. In the settings of the *Block*, enter the parameters as below. Click on *Build Selected*.



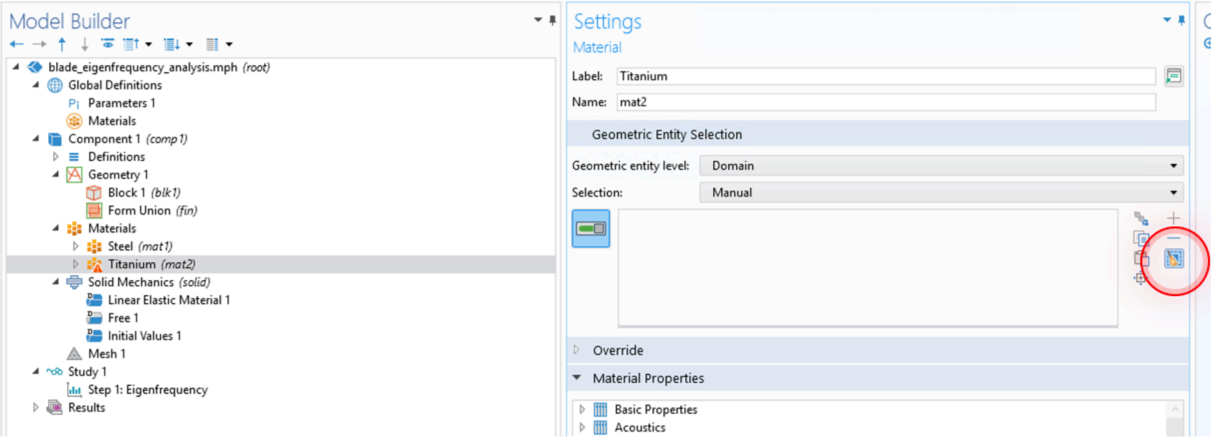
10. Right click on *Materials*, add a *blank material* and insert the parameters as below. Let us notice that the *Poisson's ratio* is set as zero because this allows us to better match the analytical calculation. Also, the density is set to 1 kg/m^3 as we neglect the mass of the blades and setting it to 0 would cause calculation issues. Also, label the material as *Steel*.



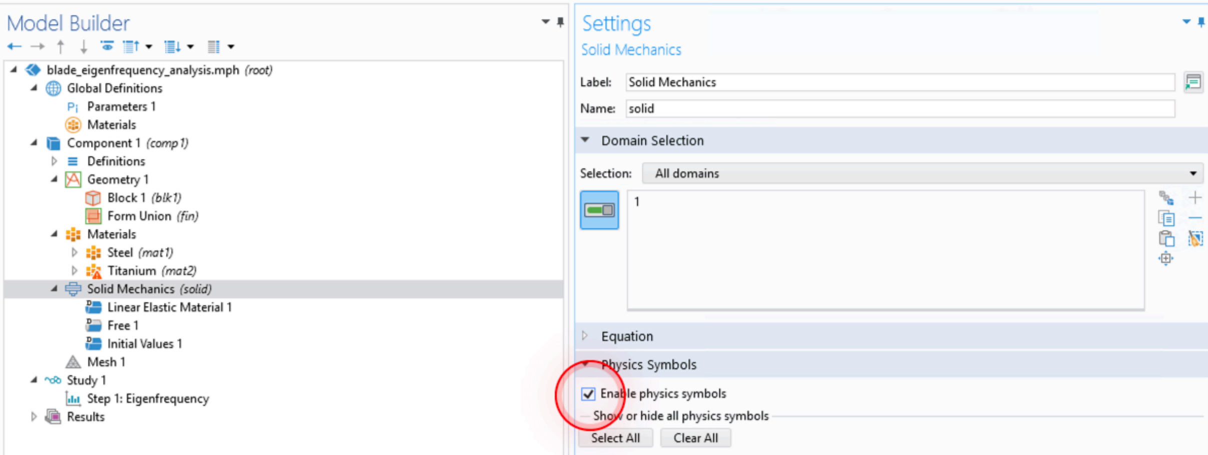
11. Duplicate the material, name its label "*Titanium*" and replace the value of the *Young modulus*.



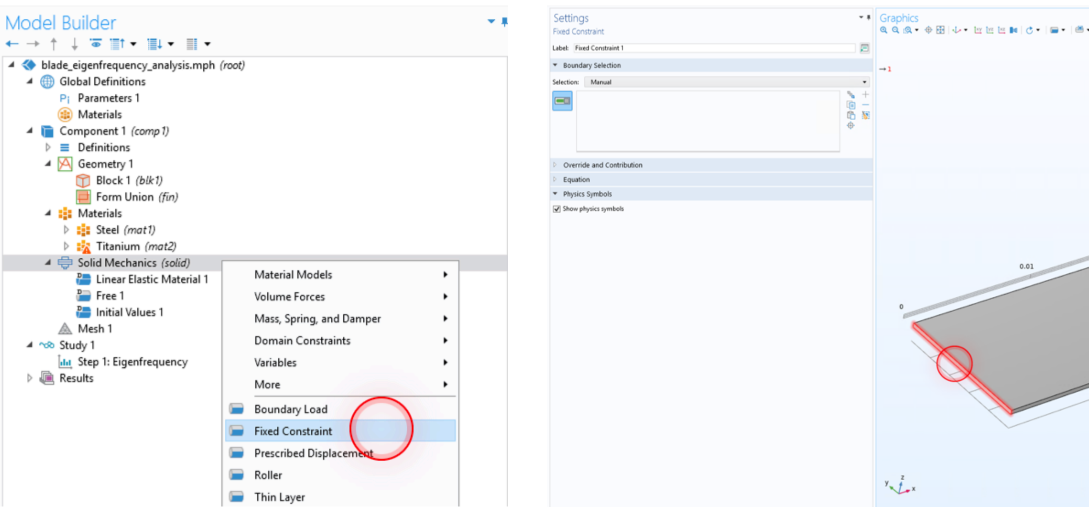
12. In *Materials* → *Titanium*, click on *Clear Selection*, so the material of the blade is set again to stainless steel.



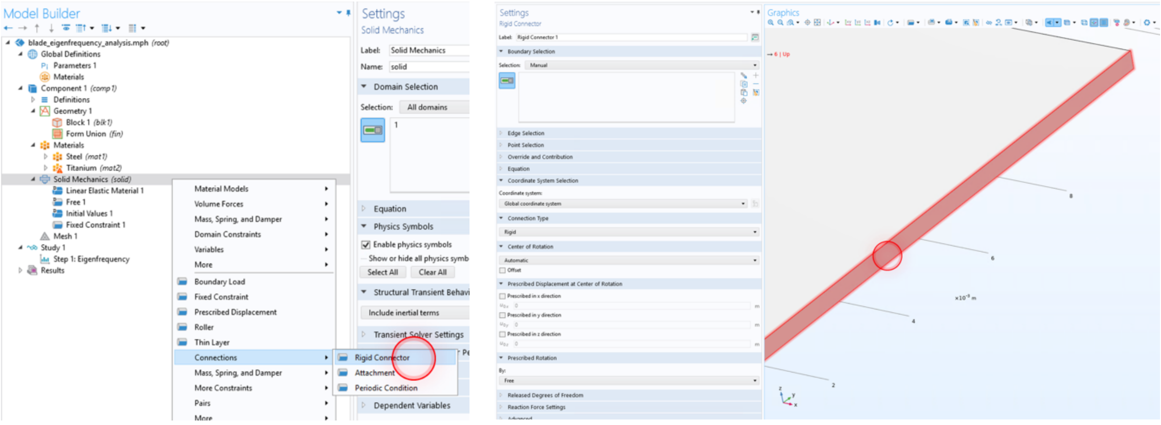
13. In *Solid Mechanics*, tick the box *Enable physics symbols*.



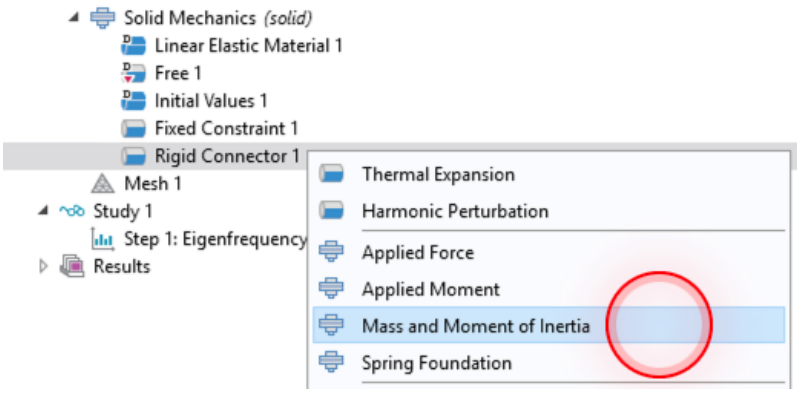
14. Right click on *Solid Mechanics*, add *Fixed constraints*. Then select the left face of the blade (boundary in -x direction).



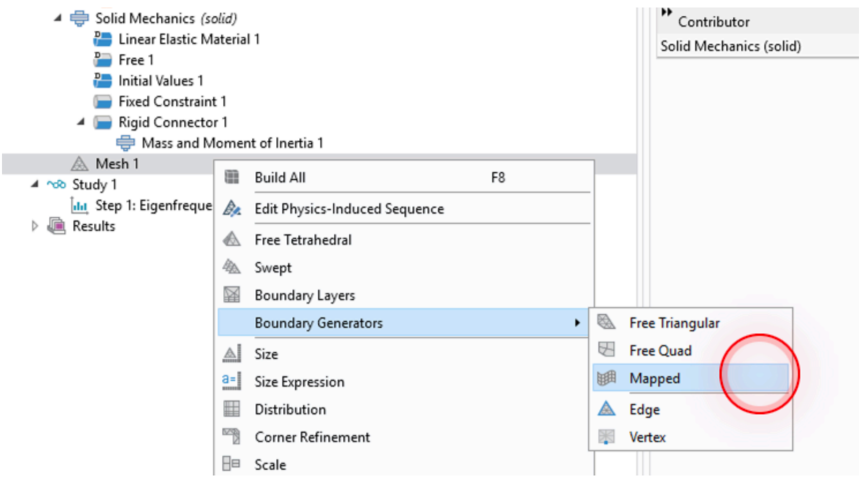
15. Right click on *Solid Mechanics* → *Connections* → add *Rigid Connector*. Then select the right face of the blade (boundary in +x direction).



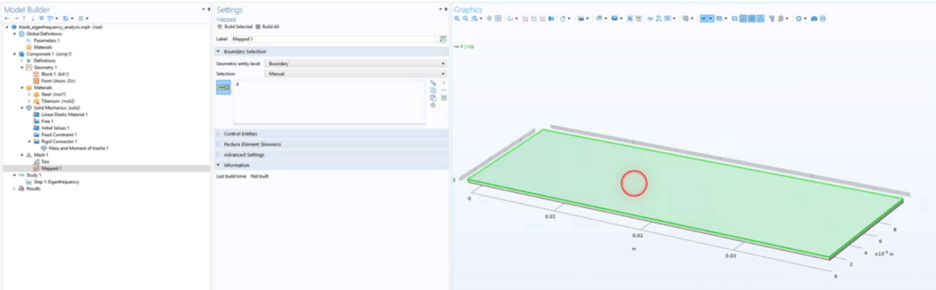
16. Right click on *Rigid Connector 1* → add *Mass and Moment of Inertia*. Set the mass value to 0.015 kg and no inertia.



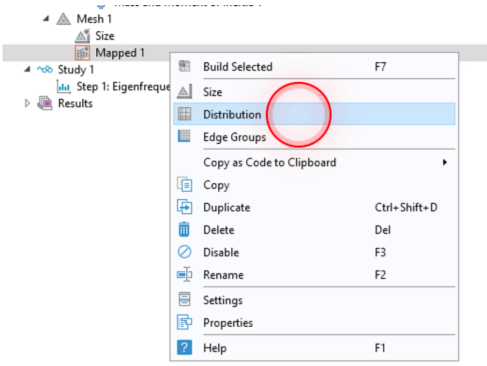
17. Right click on *Mesh 1*→ *More Generators* → add *Mapped.*



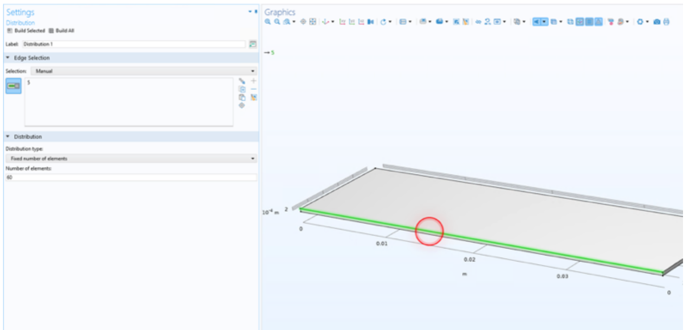
18. Select the top face of the blade (boundary in +z direction).



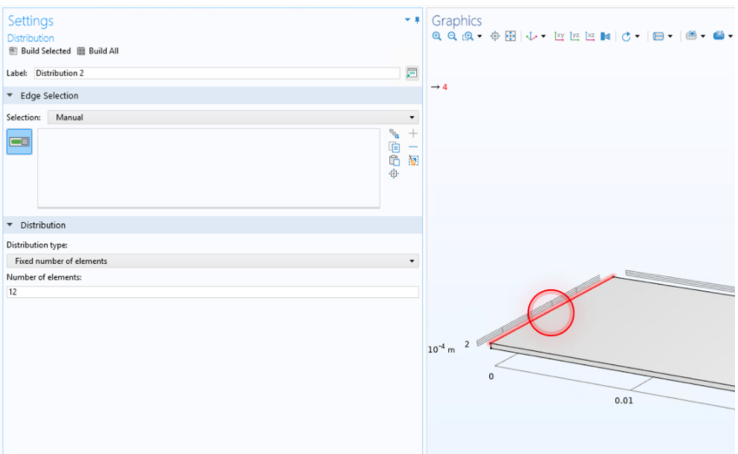
19. Right click on *Mapped 1* → add *Distribution*. Do that two times.



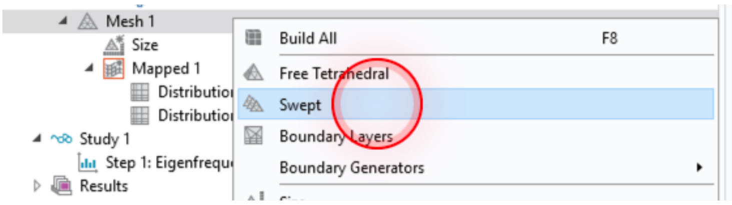
20. In the first *Distribution*, select one of the long edges of the top face of the blade. Insert 60 as *number of elements*.



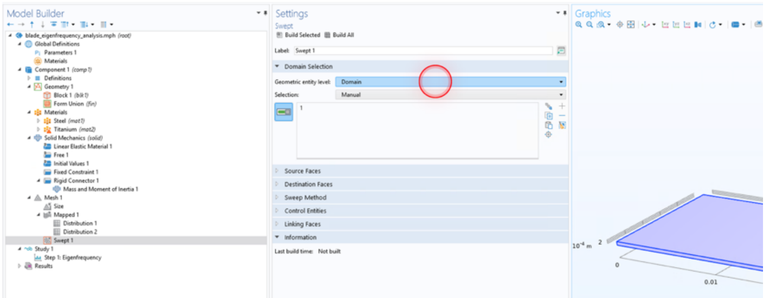
21. In the second *Distribution*, select one of the short edges of the top face of the blade. Insert 12 as *number of elements*.



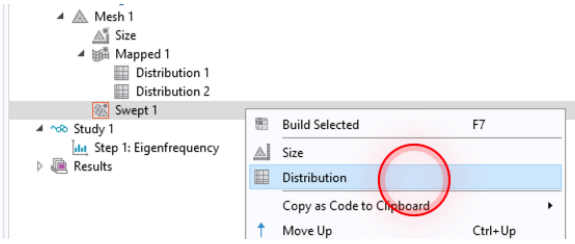
22. Right click on *Mesh 1*→ add *Swept.*



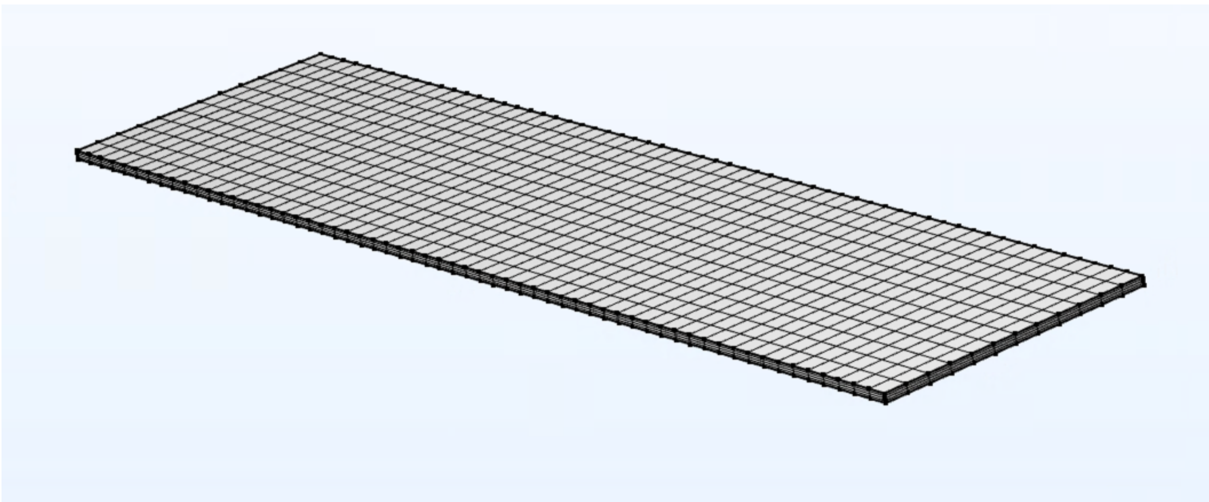
23. In *Swept 1* → *Domain Selection*, choose *Domain* and select the blade as domain.



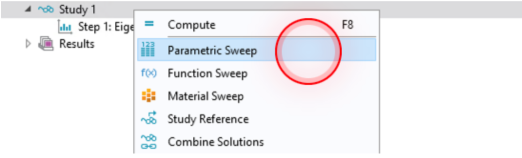
24. Right click on *Swept 1* → add *Distribution.* Set 4 as *number of elements* → *Build All.*

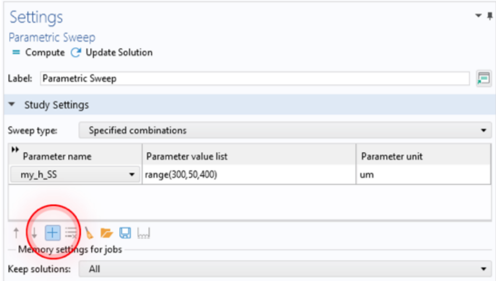


25. Your mesh should look like this.

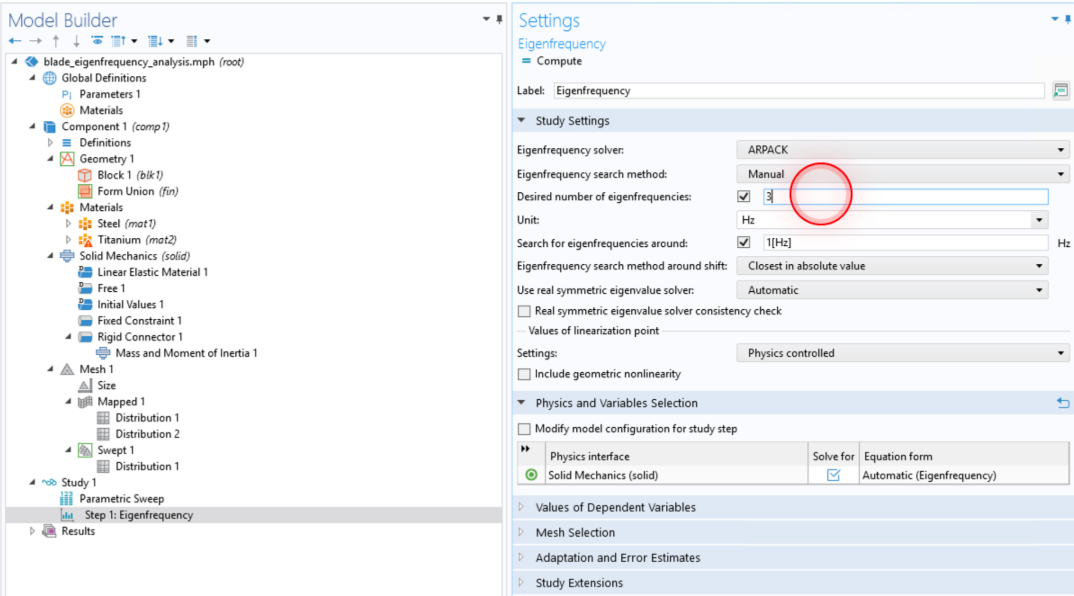


26. Right click on *Study 1* → add *Parametric Sweep*. Press the *plus button* and set the parameters as below.

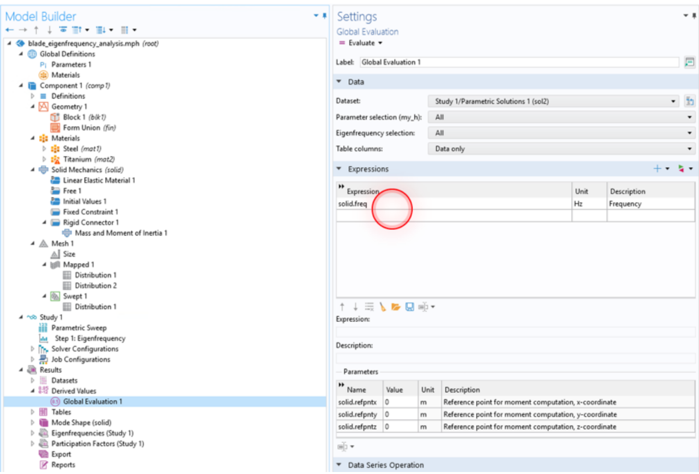




27. In *Study 1* → *Step 1*: *Eigenfrequency* → *Study Settings* → *Desired number of Eigenfrequencies*, enter 3. Then right click on *Study 1* → *Compute*.



28. You have now access to the nine eigenfrequencies values (three modes times three thicknesses). In *Results* → *Derived Values* → add *Global* Evaluation. In *Data* → select the *Dataset Study 1/Parametric Solutions 1*. In expression, add *solid.freq* → *Evaluate*. The eigenfrequencies values will be shown in *Table 1*.

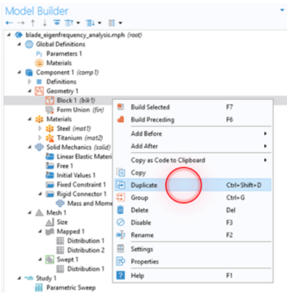


29. Use the form on Moodle (FLEX_FORMULAIRE.pdf) to calculate analytically the X, Y and Z translation stiffnesses in the Excel file EXO_3.xlsx, also on available on Moodle. Hint: the three modes are *bending*, *transversal bending* and *traction*. Finally, calculate the corresponding frequencies with the following formula:

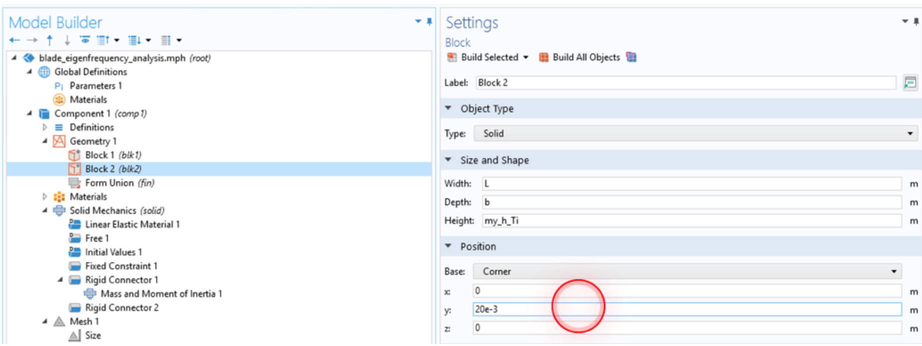
$$f_{X,Y,Z} = \frac{1}{2\pi}\sqrt{\frac{K_{X,Y,Z}}{M}}$$

30. Compare the values given by the FE model and by the analytical formulae and explain the potential mismatch between the two sets of values.

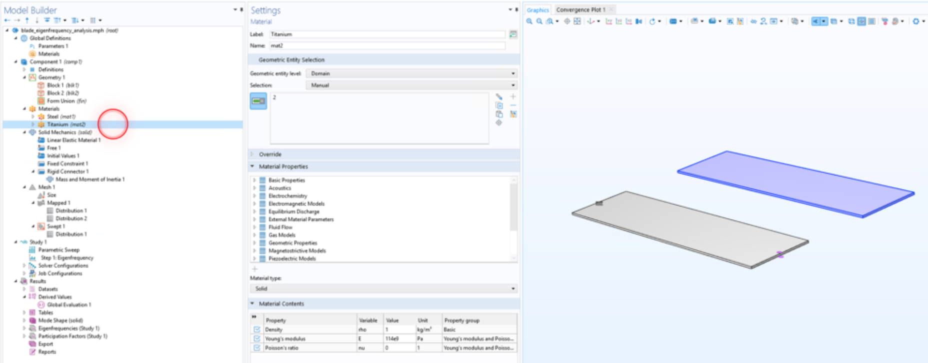
31. **We will now add the Titanium blade, assign the material, copy and paste the mesh and perform a static analysis to calculate the von Mises stress**. Right click on *Block 1* in *Geometry 1* → *Duplicate*.



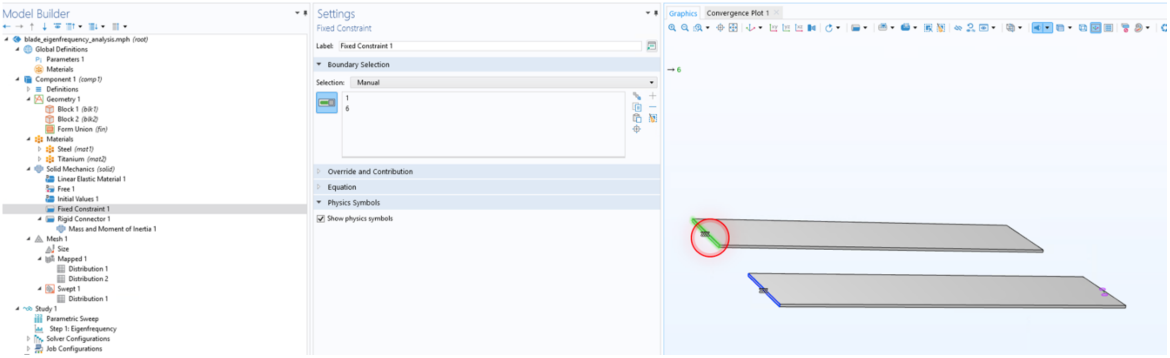
32. In *Settings* → *Size and Shape* → change the *Height* to *my_h_Ti*. In *Settings* → *Position* → change the *corner y-position* to +20 mm → *Build All Objects*.



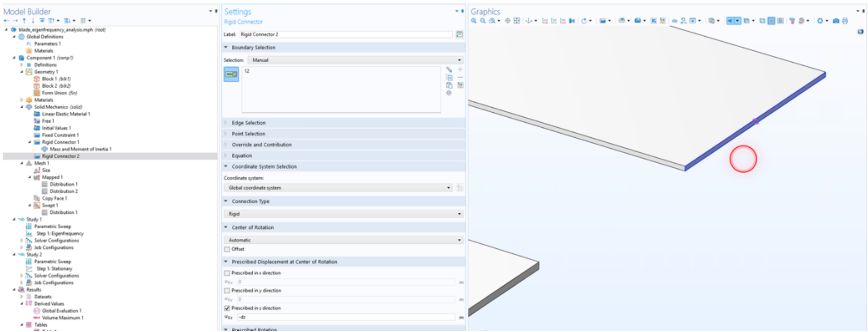
33. In *Materials* → click on *Titanium* → Select the previously created blade.



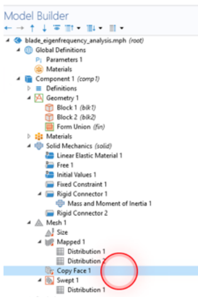
34. In *Solid Mechanics* → click on *Fixed Constraint 1* → select the fixed extremity of the second blade.



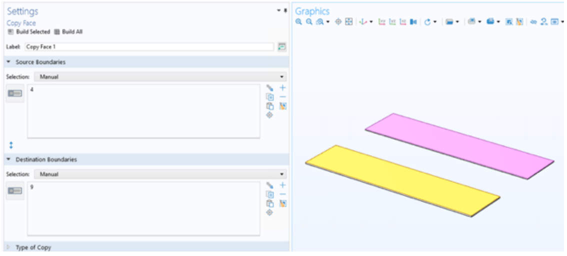
35. Duplicate the *Rigid Connector 1*. *Rigid Connector 2* → select the moving end of the second blade. Set the *Prescribed Displacement* in *z* direction to *-dz* (do it also for the *Rigid Connector 1*).



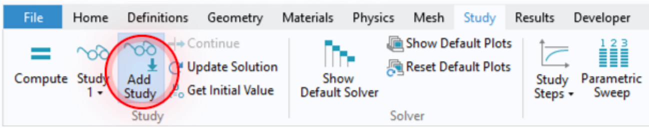
36. Right click on *Mesh 1* → *Copying Operations* → add *Copy Face*. Drag and drop *Copy Face* between *Distribution 2* and *Swept 1*.



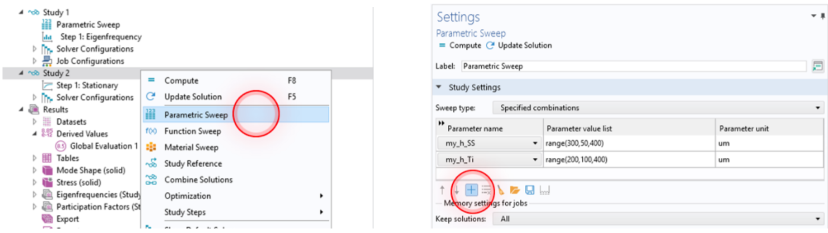
37. In *Settings* → Select the *Source* as the upper face of the steel blade and the *Destination* as the upper face of the titanium blade. In *Swept 1*, add the second blade to the selection. → *Build All.*



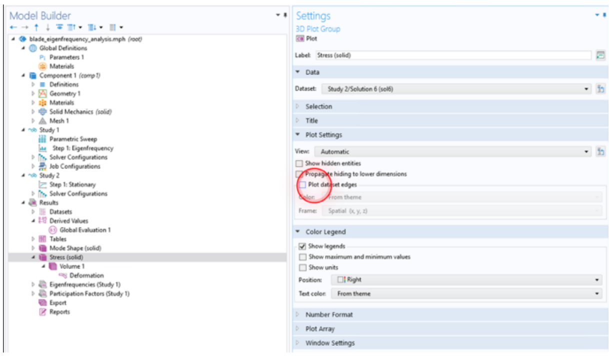
38. In the *Ribbon* → *Study* → *Add Study* → *Stationary* → + *Add Study*



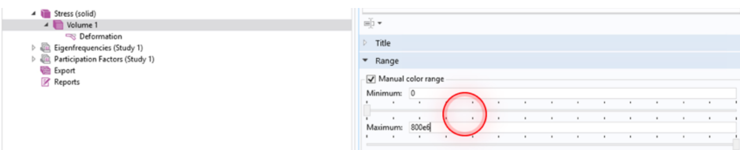
39. Right click on *Study 2* → add *Parametric Sweep* → set the same parametrization as in shown.



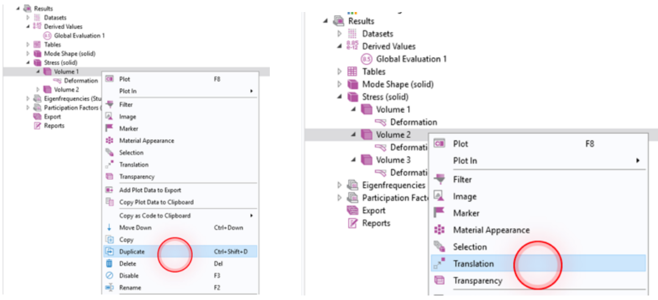
40. Right click on *Study 2* → *Compute* → Wait. In *Results* → *Stress* → untick the box *Plot dataset edges* → *Plot.*



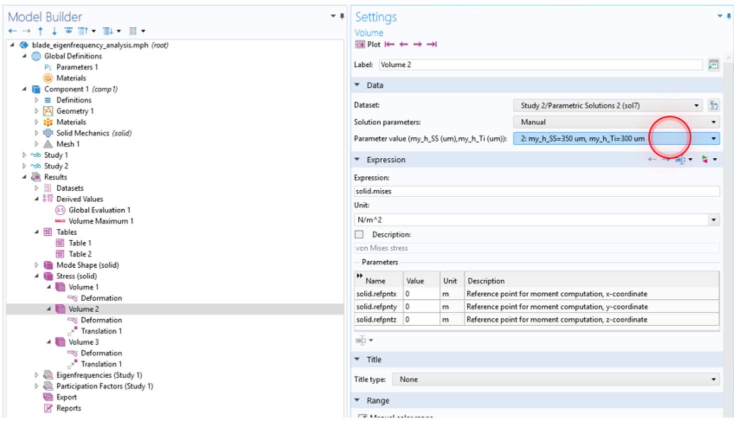
41. In *Stress* → *Volume 1* → *Settings* → *Data* → *Dataset* → select *Study 2/Parametric Solutions 2.* In *Range* → set the *Minimum* to 0 and the *Maximum* to 800e6. In *Stress* → *Volume 1* → *Deformation*, set the scale factor to 1.



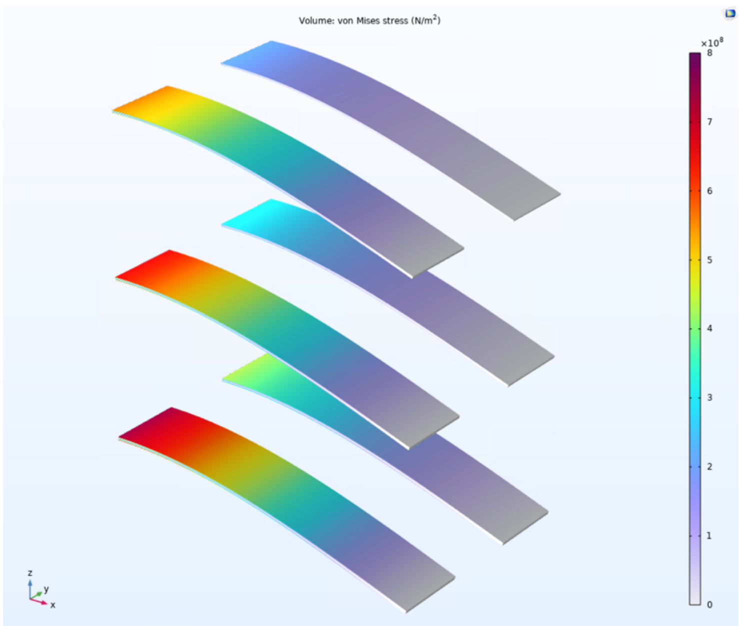
42. Right click on *Volume 1* → *Duplicate* (twice). Then, right click on *Volume 2* → add *Translation*. In *Translation* settings, set the *z* value to -20 mm. Do the same for *Volume 3* with a translation value of -40 mm.



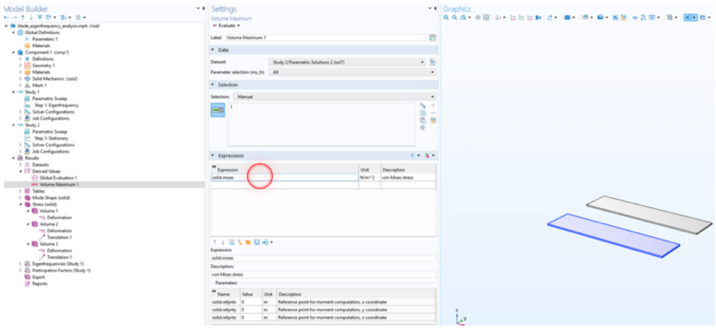
43. In *Volume 1* → *Data* → *Parameter value* → select the first set of parameters. In *Volume 2* and *3*, select the second and third set of parameters.



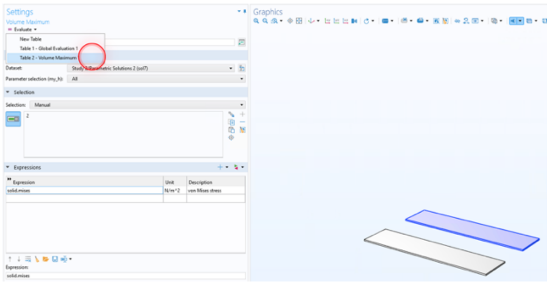
44. In *Volume 2* (and *Volume 3*) → *Title* → *None*. In *Volume 2* (and *Volume 3*) → *Coloring and Style* → untick the box *Color legend*. You should now see the six studied blades:



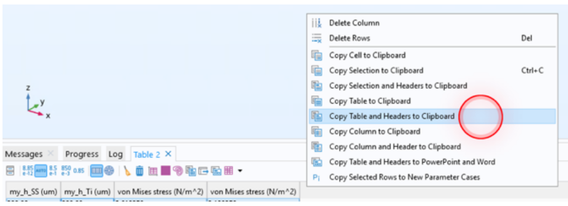
45. In *Results* → *Derived Values* → *Maximum* → add *Volume Maximum* → *Data* → *Dataset* → select the *Study 2/Parametric Solutions 2*. In *Selection* → select the stainless steel blade. In *Expression* → type *solid.mises* → *Evaluate*.



46. Unselect now the steel blade and select the other blade. *Evaluate* in the same table (*Table 2*).



47. Right click on the *Table 2* → *Copy Table and Headers to Clipboard*. Paste the values in your excel sheet and store them in a cold place for future use during EXO 6.



To summarize, we have seen in this tutorial:

- Eigenfrequency study.
- Analytical formulae derivation of eigenfrequencies.
- Comparison with FE values.
- Copy of a mesh from a domain to another domain.
- Multi-material simulation.
- Parametric sweep over a geometrical dimension.
- Display of several parameters' solutions in the same graphic viewport.

Answer to the point 30: shear contribution is not considered in formula, while it is considered in FEA. Shear is neglectable in the calculation of axial and bending stiffness and is not in the case of transversal stiffness.