Exercise 2:
COMSOL tutorial on linear VS nonlinear 2D FEM calculation

Context: We study a flexure-based linear stage composed of two leaf-springs (blades), we want to calculate the parasitic vertical displacement and the axial stiffness for several axial displacements. We will see in this example the difference between the linear and non-linear FEM calculation. This is illustrated in Figure 1 left and the studied structure is shown on the Figure 1 right. It is the same structure as in Tutorial 1, however this time we study it in 2D.

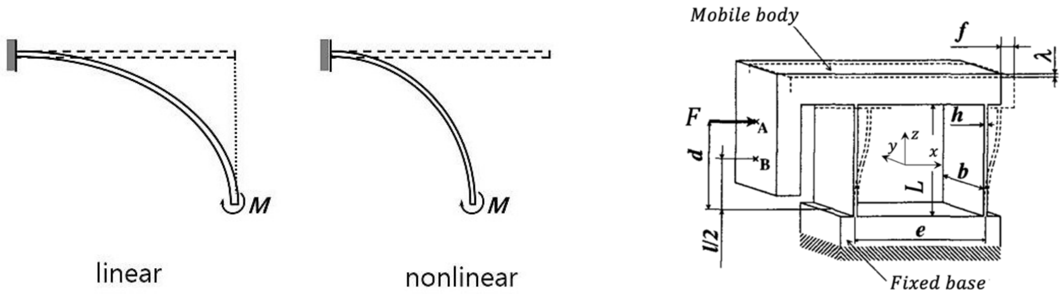
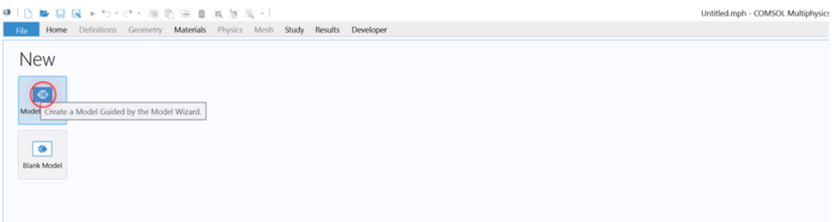


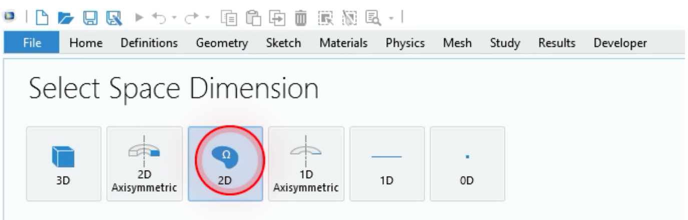
Figure 1: Linear versus Non-linear deformation – Left.
Flexible linear stage – Right.

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

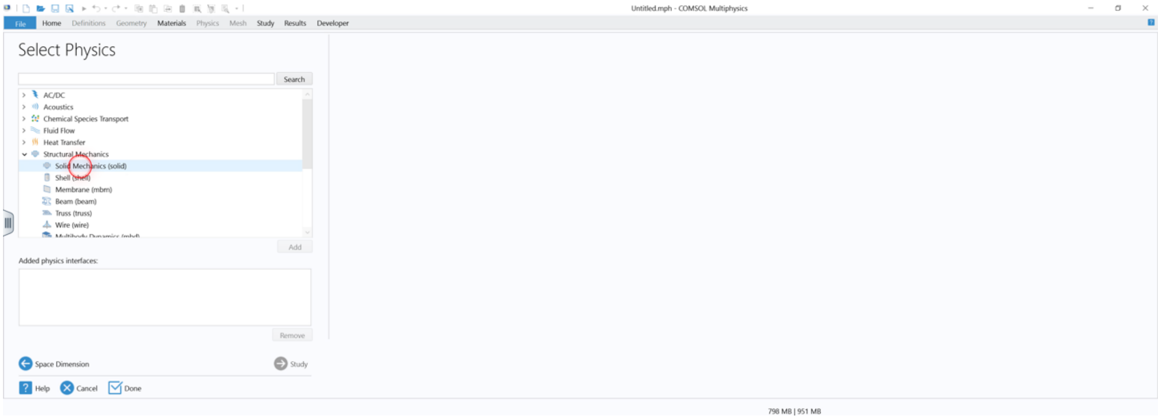
2. On the *New* model menu, select *Model Wizard.*



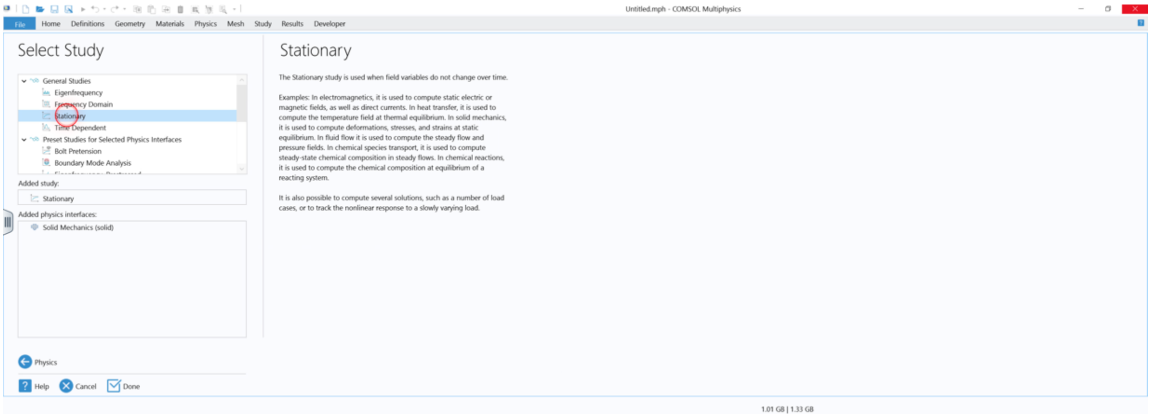
3. On the *Select Space Dimensions* menu, select *2D.*



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

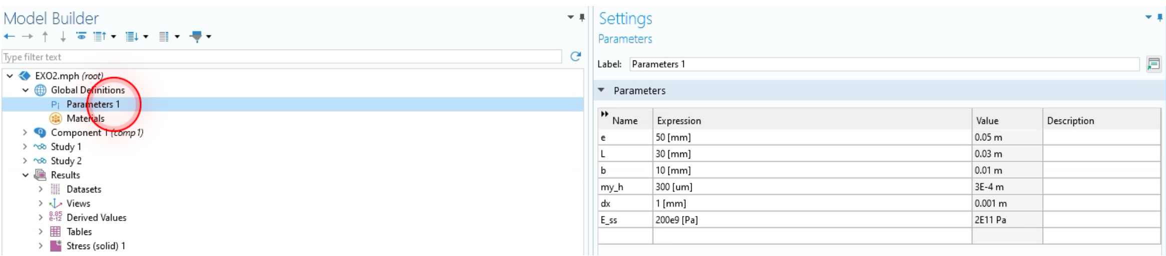


5. On the *Select Study → Stationary*, then *Done*.



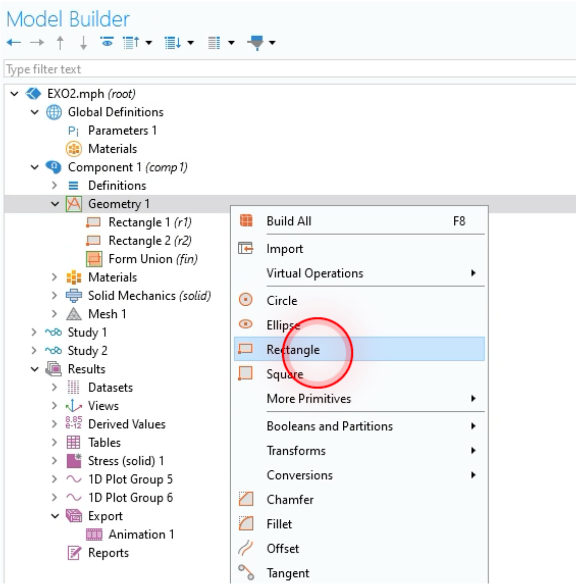
6. In *File → Save as..*, save the model with the name *lin_vs_nonlin*.

7. 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).

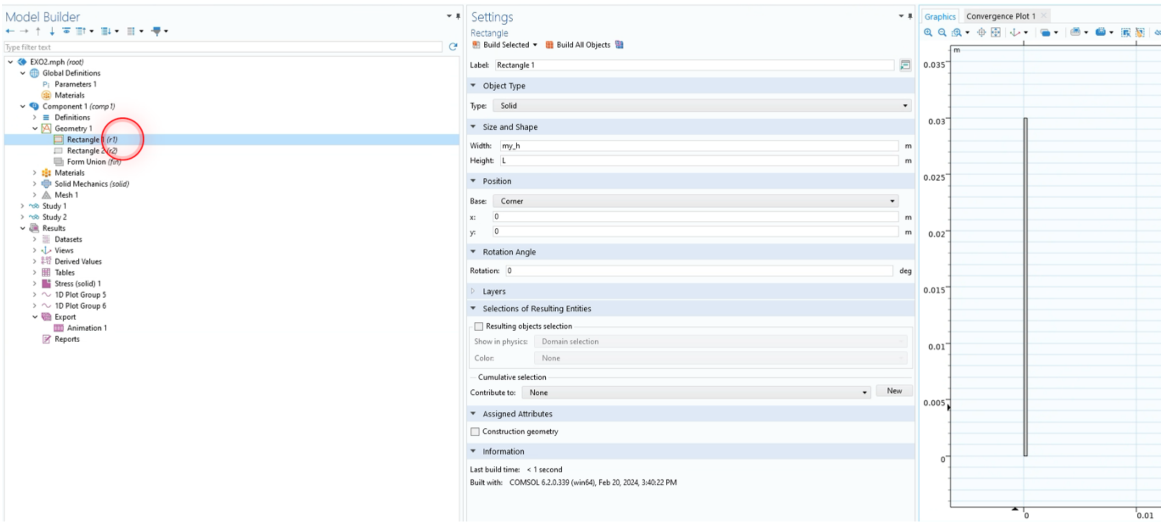


Name	Expression	Value	Description
e	50 [mm]	0.05 m	
L	30 [mm]	0.03 m	
b	10 [mm]	0.01 m	
my_h	300 [um]	3E-4 m	
dx	1 [mm]	0.001 m	
E_ss	200e9 [Pa]	2E11 Pa	

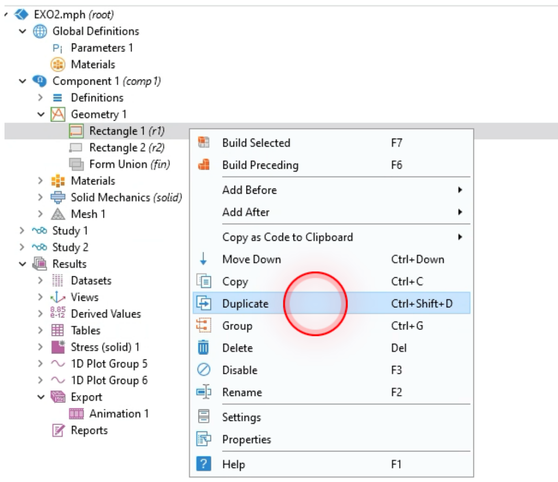
8. In *Component 1* → *Geometry*, right-click and add a *Rectangle*.



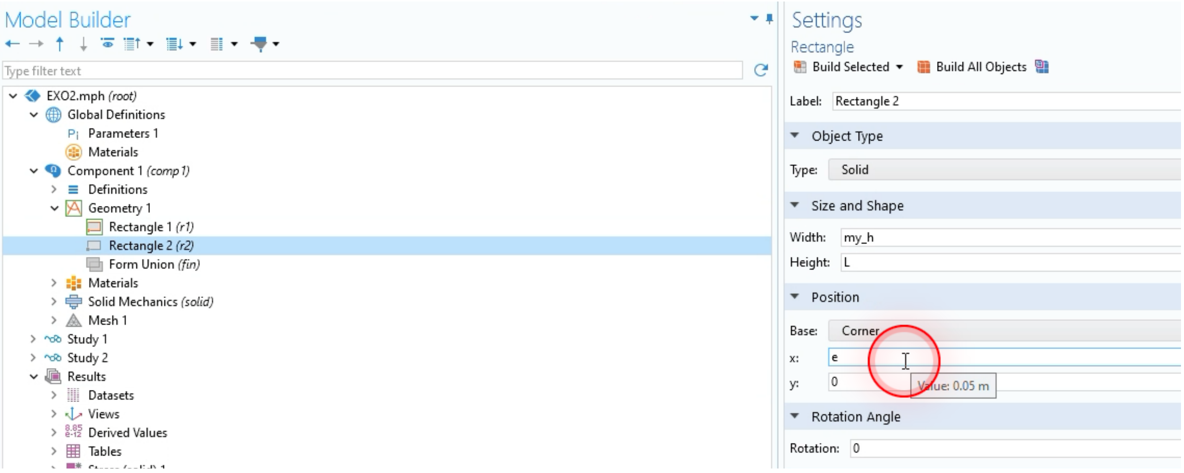
9. In the settings of the *Rectangle*, enter the parameters as below. Click on *Build Selected*.



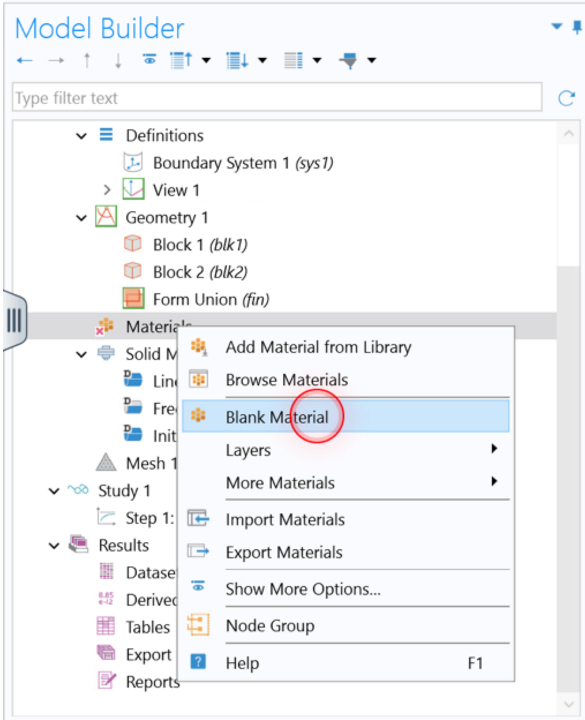
10. Right click on *Rectangle 1* → *Duplicate*.



11. In the settings of *Rectangle 2*, change the x-component of the *Corner* position from *0* to *e*. → *Build All Objects*.



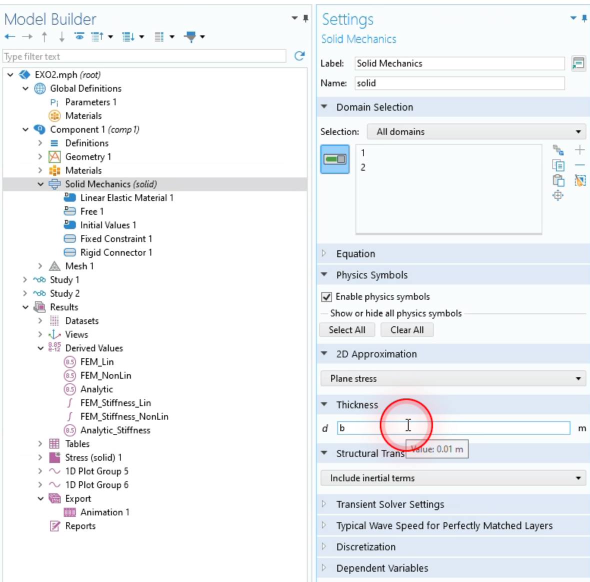
12. In the *Model Builder*, right click on *Material*, click on *Blank Material*.



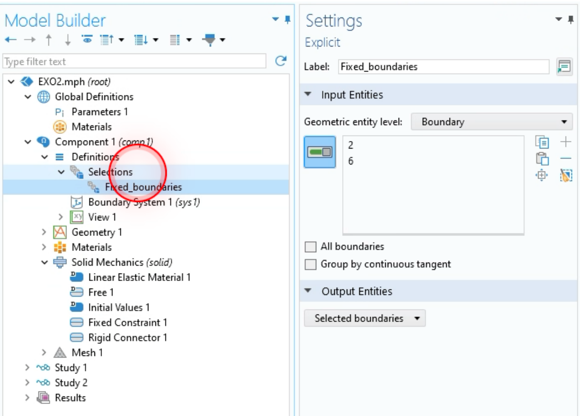
13. In *Settings* of *Material 1*, add the following parameters. They correspond to stainless steel.

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

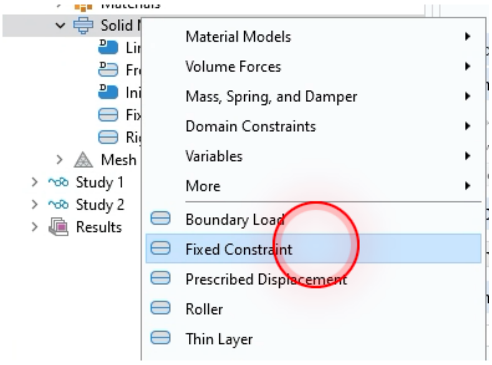
14. In the *Model Builder*, click on *Solid Mechanics*, then in the *Settings* pane, tick on the box *Enable physics symbols*, chose *2D Approximation* as *Plane stress* and set the out-of-plane *Thickness* of the model equal to *b*. Notice that in 2D simulations, the plane strain approximation can lead to increased stiffness and stress linked to Poisson ratio.



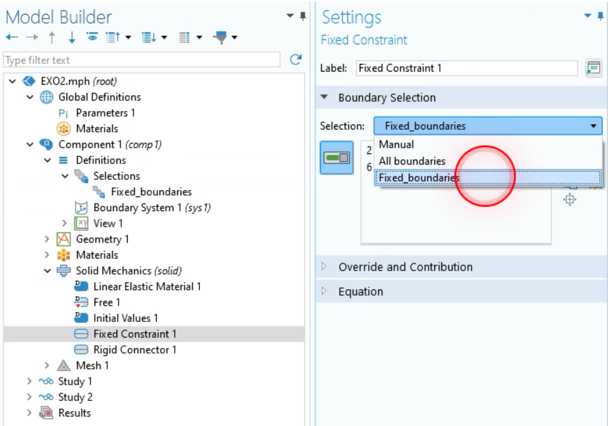
15. *Component 1* → *Definitions* → *Right click on Selections* → *Explicit*. Name the *Label Fixed_boundaries*, set the *Geometry entity level* as *Boundary* and select the two lower edges of the blades.



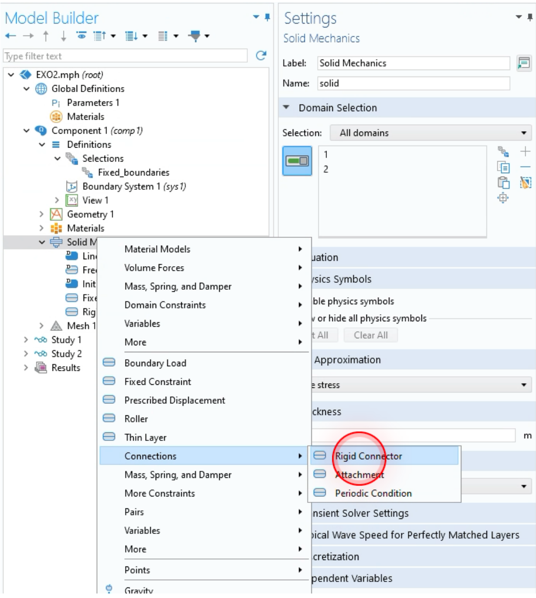
16. Right click on *Solid Mechanics* → Add *Fixed Constraint.*



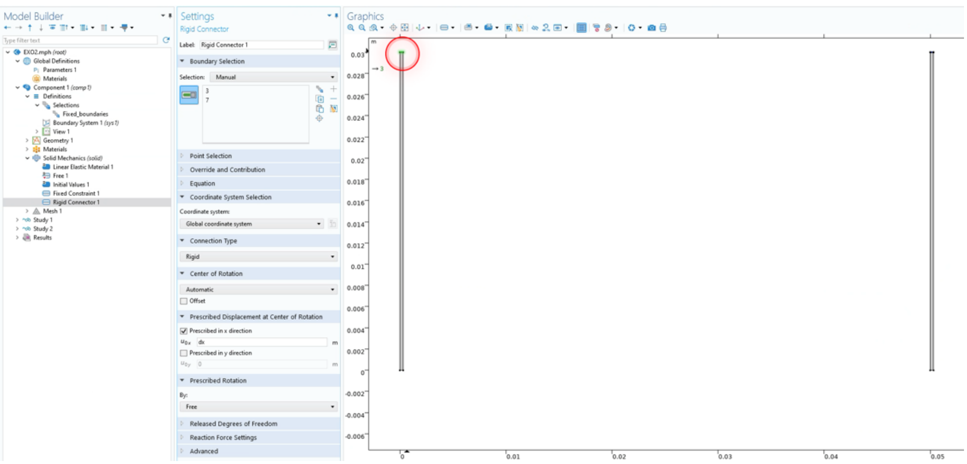
17. *Fixed Constraint 1* → *Boundary Selection* → Select the *Fixed_boundaries* previously defined.



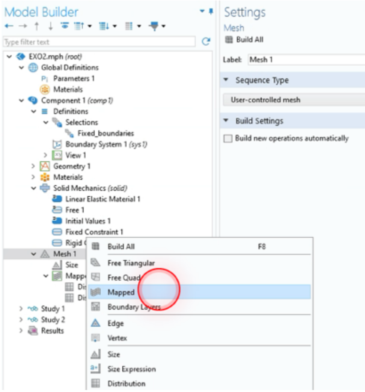
18. Right click on *Solid Mechanics* → *Connections* → Add a *Rigid Connector.*



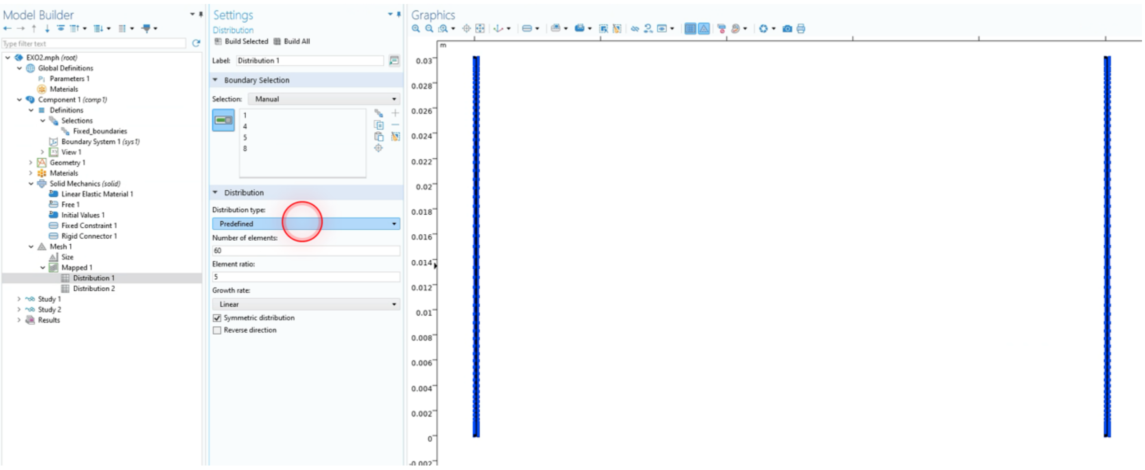
19. *Rigid Connector 1* → Select the two upper edges of the blades. Add a *Prescribed displacement* in x direction, enter the dx as value. Let the y displacement free.



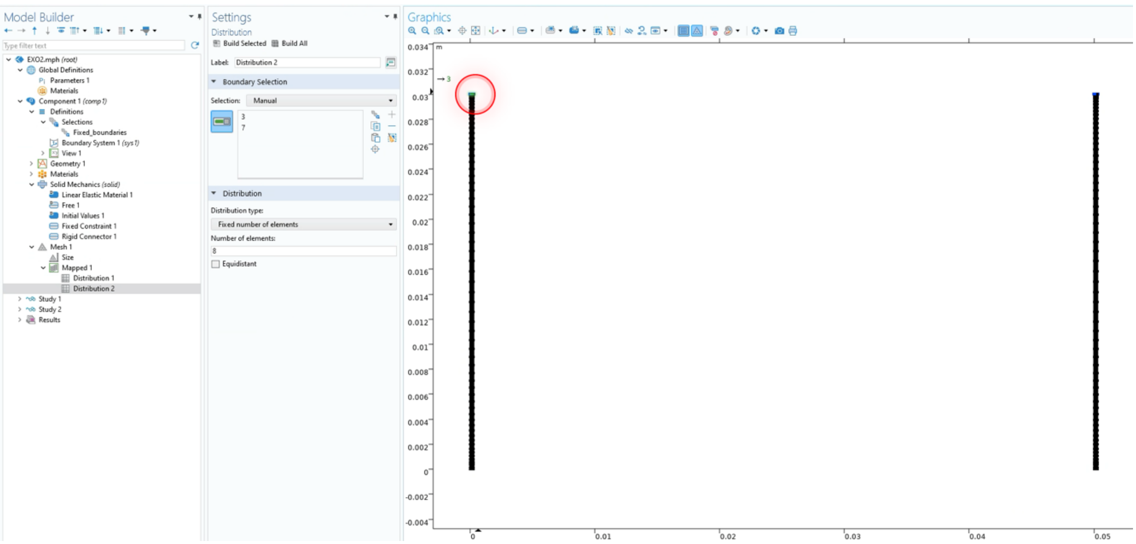
20. Right click on *Mesh 1* → Add *Mapped*. Then on right click on *Mapped 1* → Add *Distribution*.



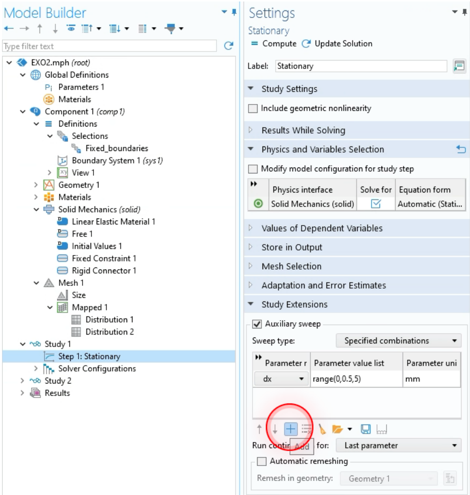
21. In *Distribution 1*, select the four long edges of the blades and enter the parameters as below. Applying non-evenly distributed elements along the mesh is good for calculating accurately the stress at the roots of the blades, where its value and gradients are maximal.



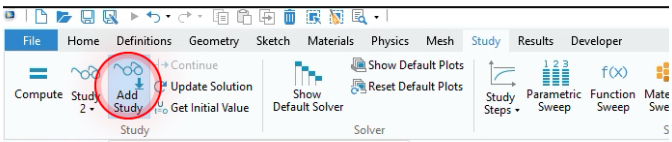
22. Right click on *Mapped 1* → Add *Distribution*. In *Distribution 2*, select the two upper surfaces of the blades and insert 8 as *number of elements*.

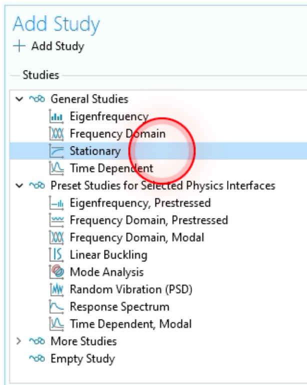


23. In *Study 1* → *Step 1: Stationary* → *Study Extensions*, tick the box *Auxiliary sweep* and then click on the *Add* button and select *dx* as the swept parameter. Fill the range and unit as specified. The deformation will be calculated for each position in the range.

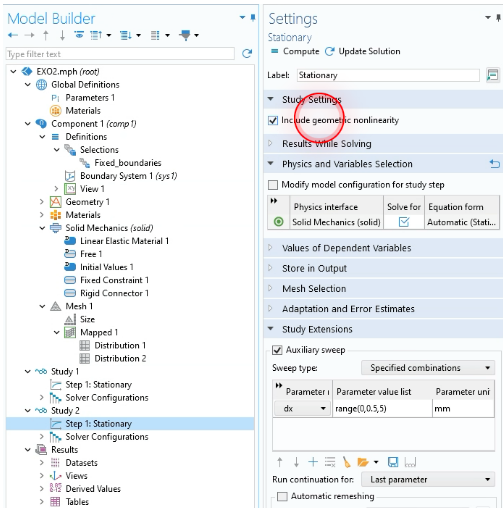


24. Right click on *Study 1* → *Compute*. Then go on the *ribbon* → *Study* → *Add Study* → add a *Stationary study*.

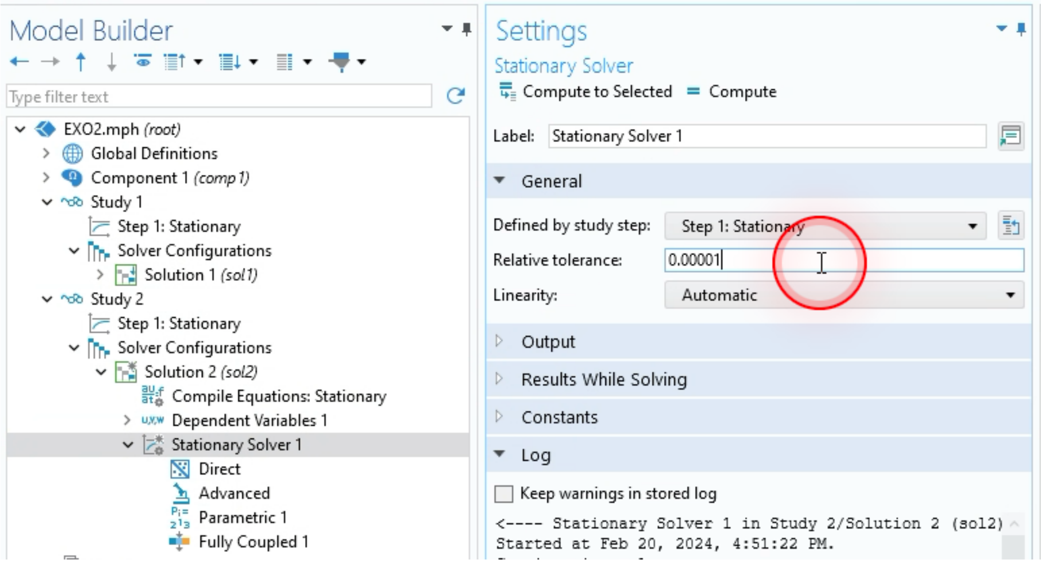




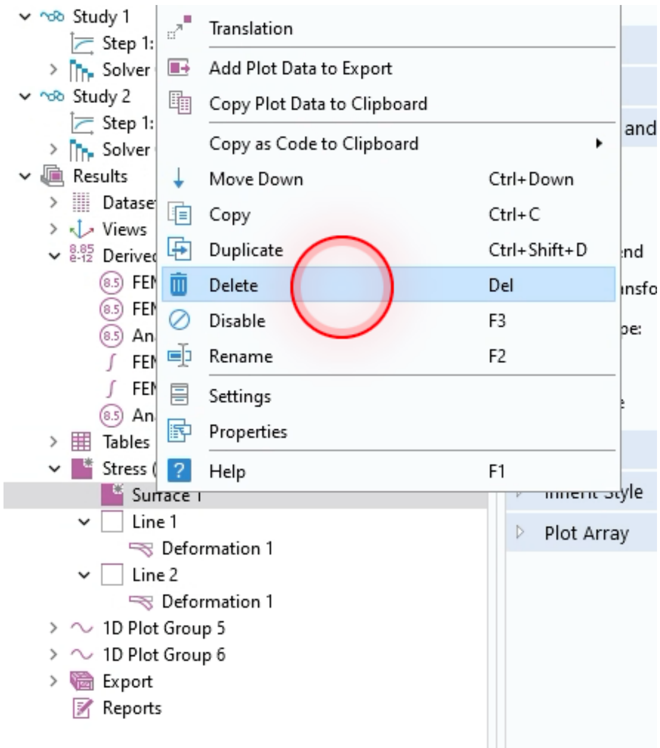
25. Set the same parameter as in *Study 1*, except this time you tick the *Include geometric nonlinearity* box. Right click on *Study 2* → *Compute*.



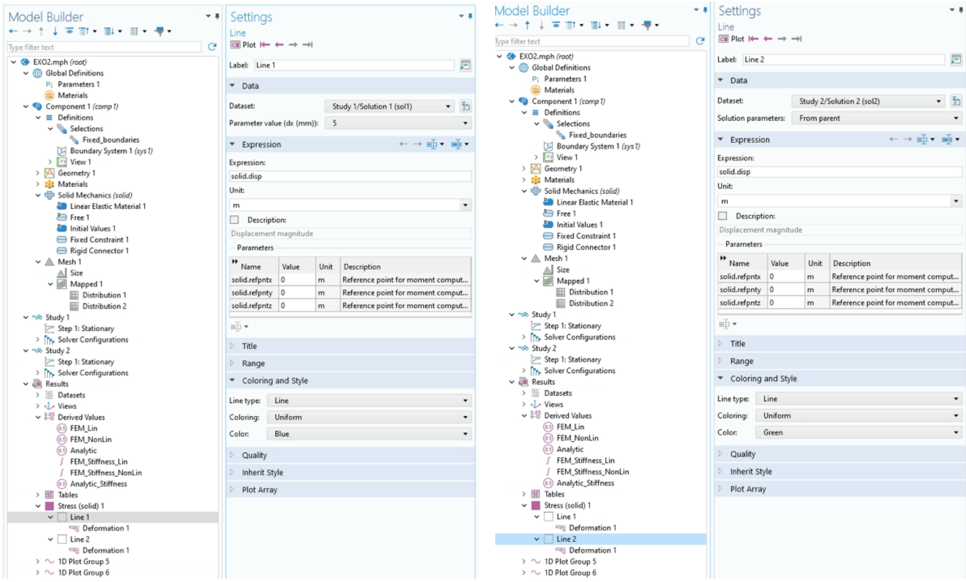
26. In *Study 2 → Solver Configurations → Stationary Solver 1 → General →* set the *relative tolerance* to 0.00001. Right click on *Study 2 → Compute.* In case we let the default value of the *relative tolerance*, the stiffness curve would be tainted with numerical noise. Let us notice that we run the *Study 2* calculation twice, as the *Solver Configuration* is created during the first run. Doing so allows us to configure the solver before the second run. Note that it is also possible to access the solver configuration by right clicking on the study and running the *Get Initial Value* command.



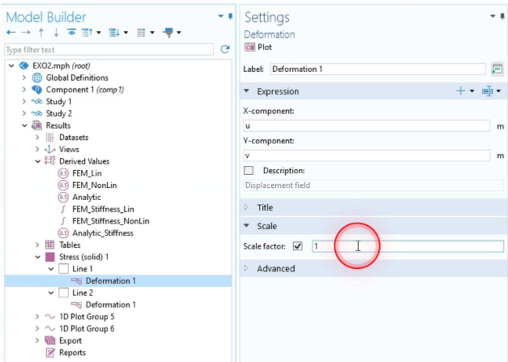
27. We will now **compare visually the deformation of the blades** in the case of linear and non-linear calculation. In *Results → Stress →* Right click on *Surface → Delete.*



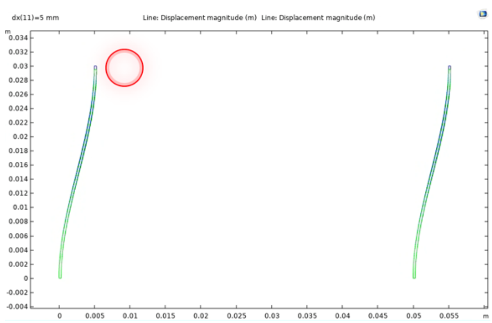
28. Right click on *Stress* → add *Line* two times and set the parameters as below. Be sure to set the maximal displacement in both cases (*Parameter value* and *Solution parameters*). Additionaly, right click on Line 1 and Line 2 and add Deformaiton.



29. In *Stress* → *Line 1* (and *Line 2*) → *Deformation 1*, set the *scale factor* to 1.

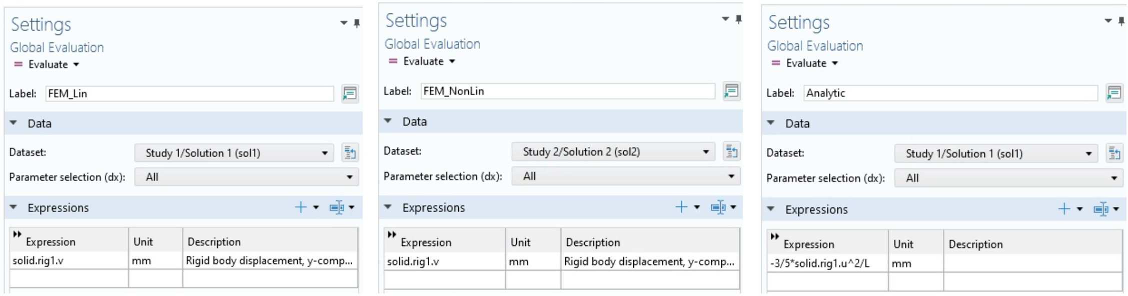


30. In the *Graphics* window, you should see the deformed blades in blue and in green, respectively as calculated from the linear and nonlinear analysis. Look carefully at the top of the blades. There is no “shortening” of the blades in the case of the linear analysis. The non-linear analysis provides more realistic result.

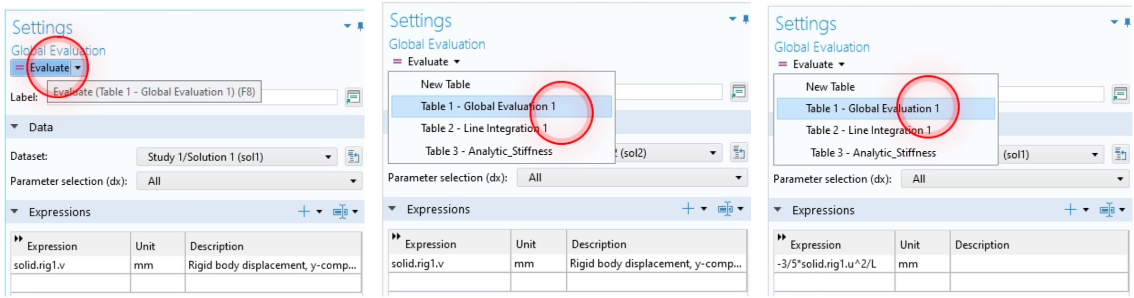


31. We will now **evaluate the vertical displacement of the translation stage** given by the linear calculation, by the non-linear calculation and by the analytical formula. In *Results*

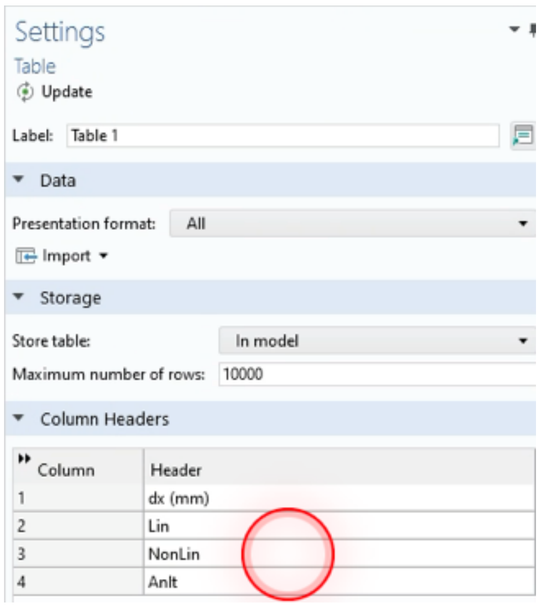
→ *Derived Values* → Add three times *Global evaluation*. In the three of them, fill respectively the following settings for the *Label*, the *Dataset* and the *Expression*.



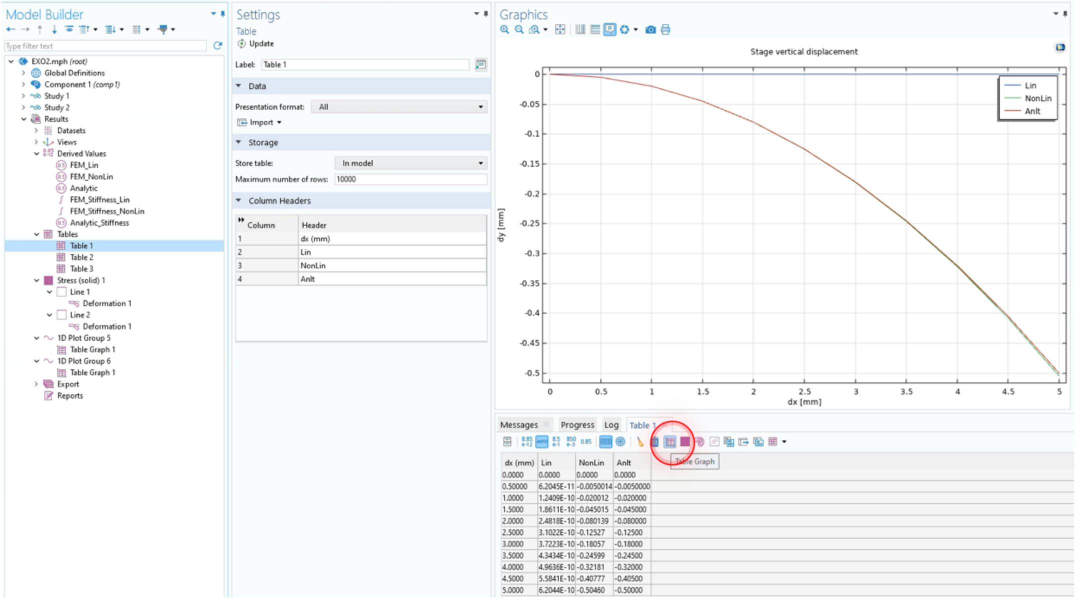
32. In the *FEM_Lin* evaluation, click on *Evaluate*. Then on the second and third evaluation, click on the little arrow and evaluate in the *Table 1*.



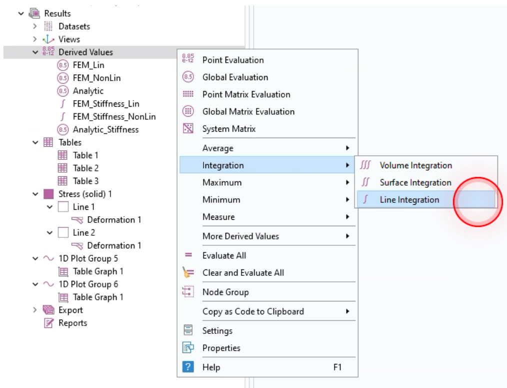
33. In *Results* → *Tables* → *Table 1*, fill these names for the *Headers* of the *Columns*.



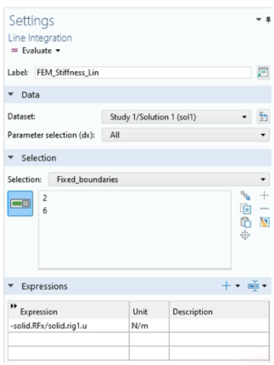
34. In *Table 1*, click on the *Table Graph* button. You can see that in the case of the linear calculation, the vertical displacement of the stage is null, while the non-linear analysis and the analytical formula match well, with a parabolic shape.

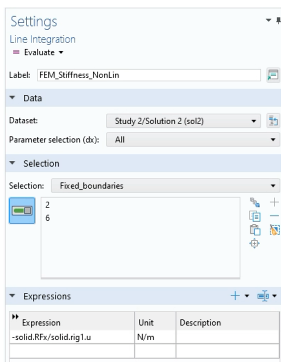


35. We will now **evaluate the axial stiffness of the translation stage** given by the linear calculation, by the non-linear calculation and by the analytical formula. In *Results* → right click on *Derived Value* → *Integration* → add *Line Integration* two times.

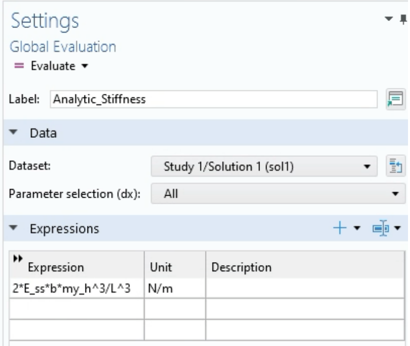


36. In the two of them, fill respectively the following settings for the *Label*, the *Dataset* and the *Expression*. For the *Selection*, select the previously defined *Fixed_boundaries* set of edges.

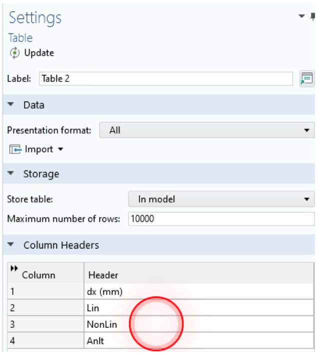




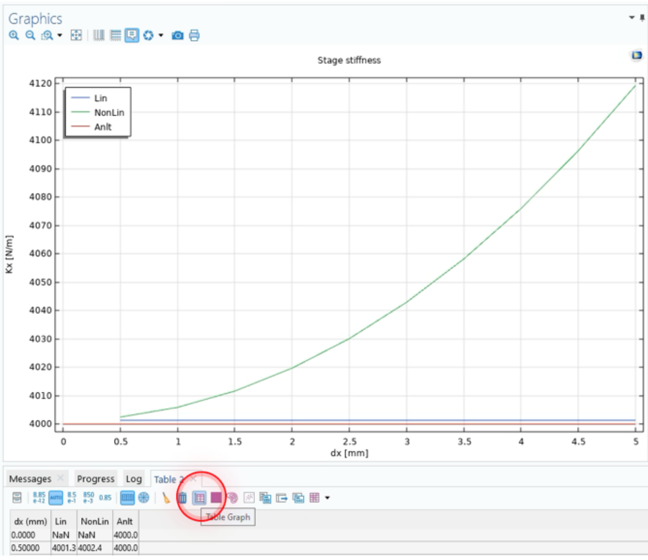
37. In *Results* → *Derived Value* → Add a *Global evaluation*. In settings, fill the following settings for the *Label*, the *Dataset* and the *Expression*. The later corresponds to the analytic formula of the stage's axial stiffness.



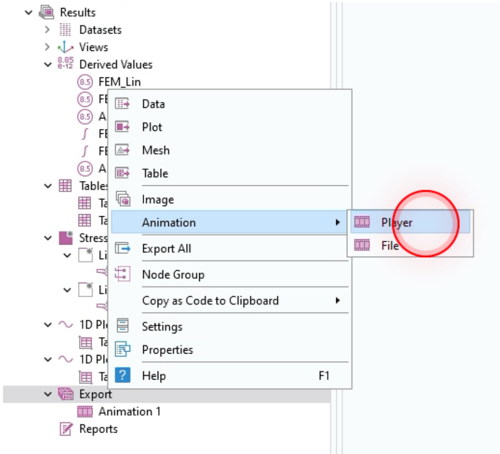
38. In the *FEM_Stiffness_Lin* evaluation, click on the *Evaluate* button. Then in the *FEM_Stiffness_NonLin* and *Analytic_Stiffness*, *Evaluate* in Table 2. In *Results* → *Tables* → *Table 2*, fill these names for the *Headers* of the *Columns*.



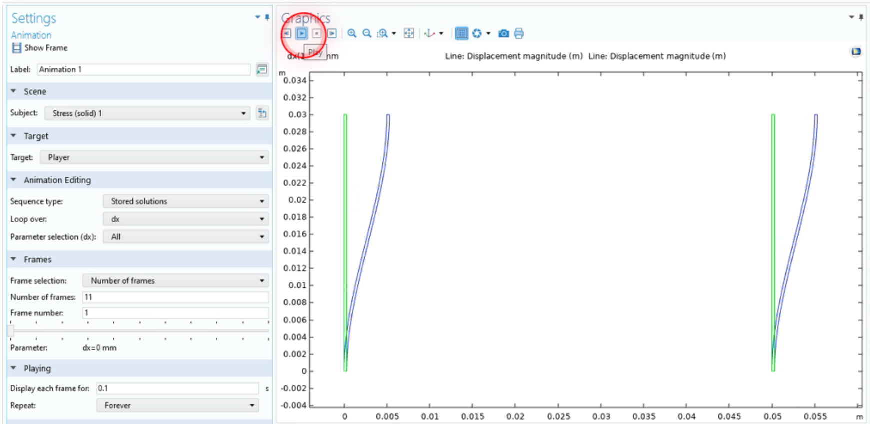
39. In *Table 2*, click on the *Table Graph* button. You can see that the linear calculation gives a constant value and is close to the value given by the formula. Additionally, the non-linear stiffness calculation gives the same value as the linear and analytic approaches, but increases as the axial displacement gets larger.



40. Finally, we will **animate the translation stage**. In *Results* → right click on *Export* → *Animation* → add *Player*.



41. In the settings of *Animation* → *Playing* → *Repeat* → choose *Forever*. On the *Graphics* window, click on the *Play* button and enjoy watching the motion of the stage!



To summarize, we have seen in this tutorial:

- 2D FE modeling.
- Predefine sets of boundaries to be used at a later stage.
- How to sweep over positions to calculate several deformations.
- How to mesh with custom distributions of elements to match the stress level and gradient.
- How to evaluate the trajectory of a rigid connector.
- How to evaluate the reactions forces while integrating (summing) along the edges where fixed constraints are set. Small individual reaction forces are summed on nodes when *Line Integration* is evaluated.
- How to animate the FE model.
- The difference between the linear and non-linear FEA.
- The agreement between the linear and non-linear FEA and the analytical formulae.