Exercise 4:
COMSOL tutorial on bistable buckled beam simulation

Context: We want to study the buckling behaviors of an initially-straight beam with pinned-pinned boundary conditions (Fig. 1). The first task is to simulate the stable buckling deflections of the beam when axially preloaded by a controlled displacement Δl (Fig. 1b). Then, the second task is to evaluate the snap-through behaviors of the bistable buckled beam when an input rotation angle θ_{in} is applied at one of its pinned extremities (Fig. 1c). The final goal is to plot the different actuation characteristics (namely, the input moment M_{in}, the output pivot angle θ_{out}, and the strain energy of the beam) as a function of the input pivot angle θ_{in}. To that end, FEM computations are carried out on COMSOL using 2D static studies.

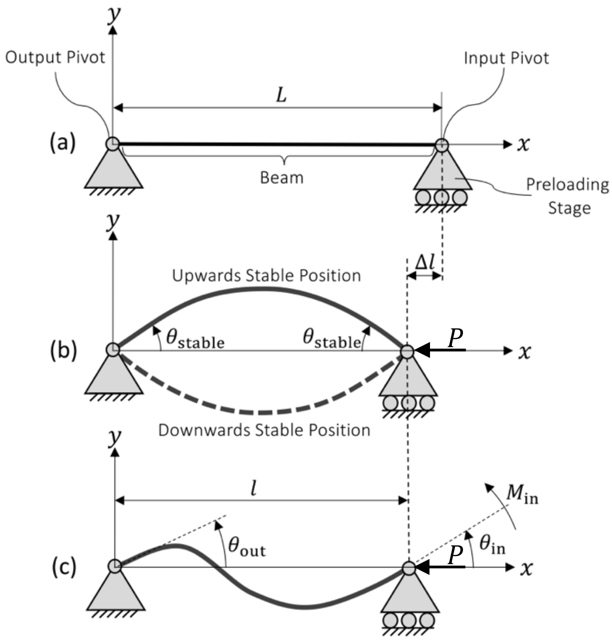


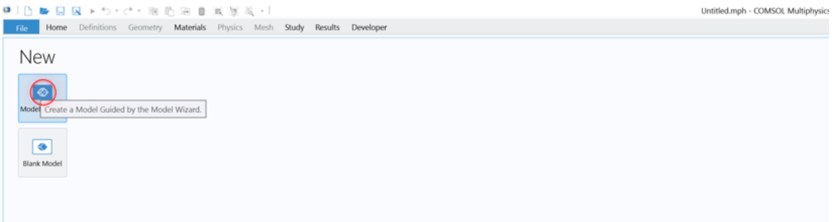
Figure 1: Pinned–pinned buckled beam: (a) undeflected, (b) buckled into one of its two stable positions using the preloading stage, and (c) where a moment is applied to the input pivot to switch between the two stable states.

You can find more information on the analytical modeling of this type of buckled beams in the following paper: https://doi.org/10.1016/j.mechmachtheory.2022.104874

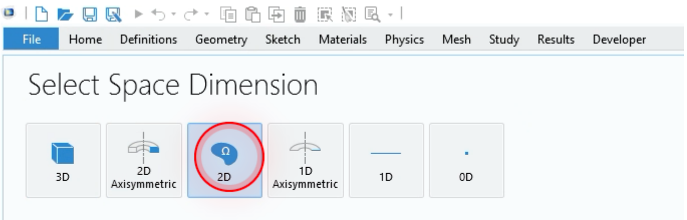
Task 1

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

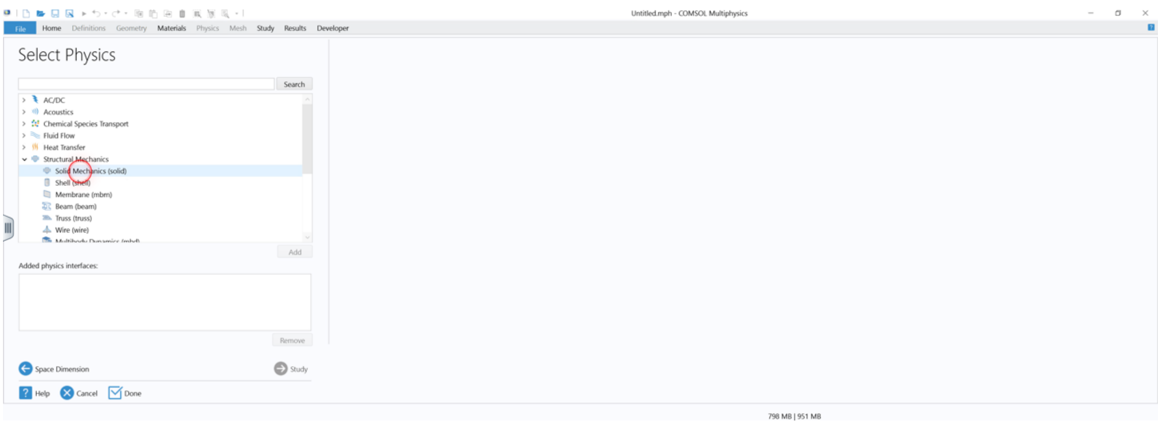
2. On the *New* model menu, select *Model Wizard.*



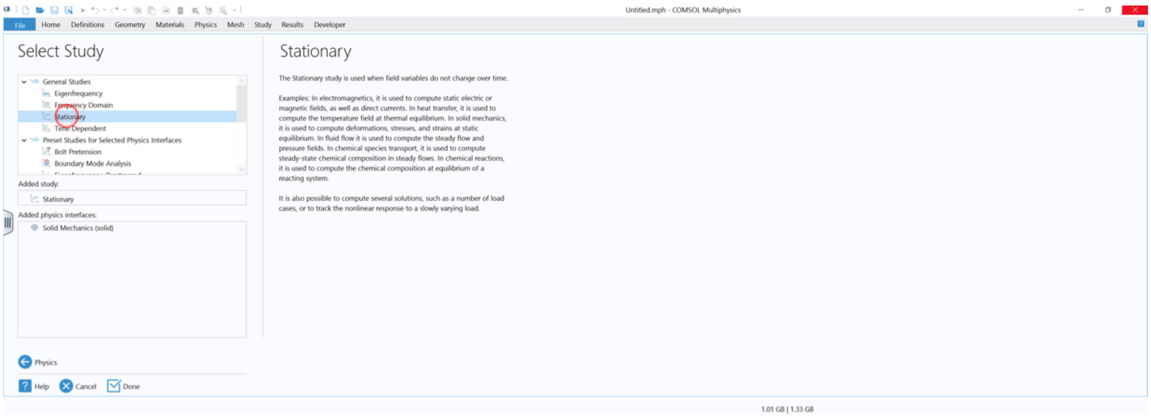
3. On the *Select Space Dimensions* menu, select *2D.*



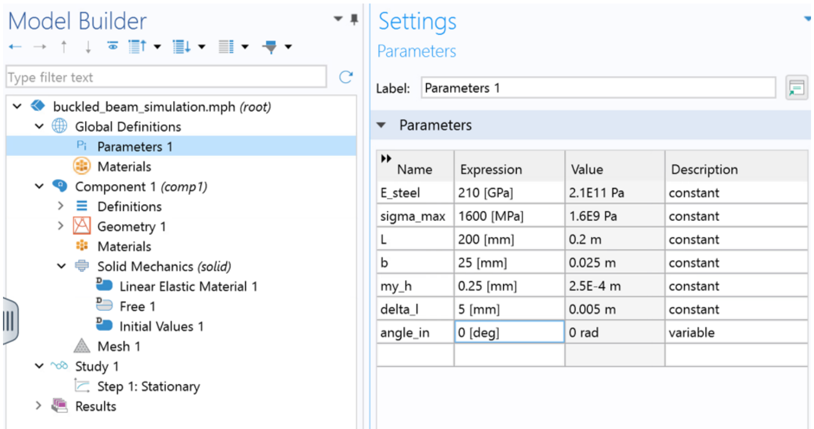
4. On the *Select Physics* → *Structural Mechanics* → *Solid Mechanics (solid).* Press *Add*, then *Study.*



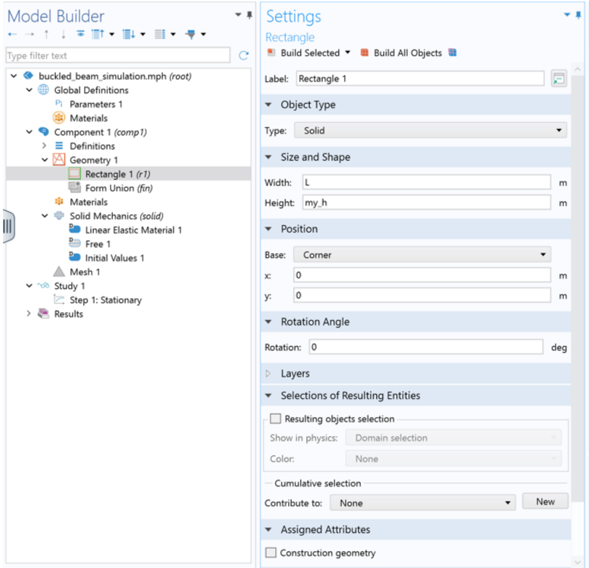
5. On the *Select Study* → *Stationary*, then *Done.*



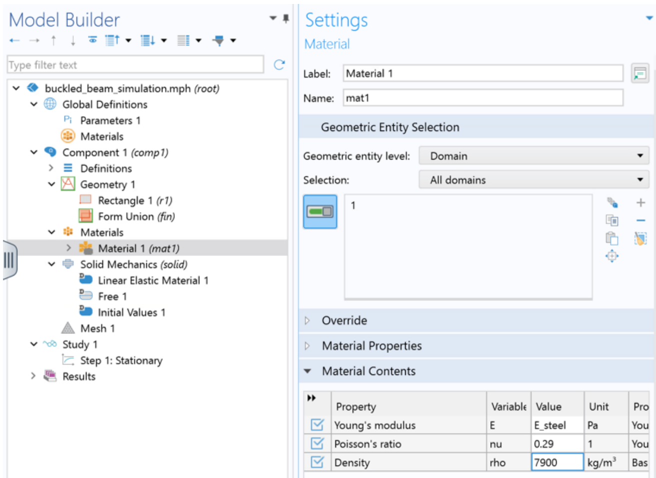
6. In *File* → *Save as…*, save the model with the name *buckled_beam_simulation.*
7. In the *Model Builder* → *Global Definitions* → *Parameters 1*, enter the parameters and the values as listed below. (N.B.: *h* is named *my_h* as "*h*" is reserved for the Planck constant in COMSOL).



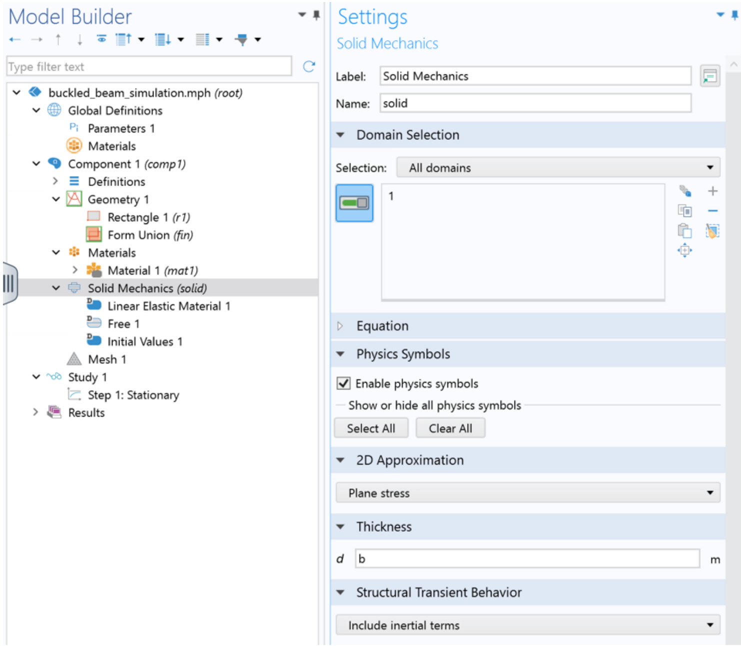
8. In *Component 1* → *Geometry*, right-click and add a *Rectangle*. In the settings of the *Rectangle*, enter the parameters as below. Click on *Build Selected*.



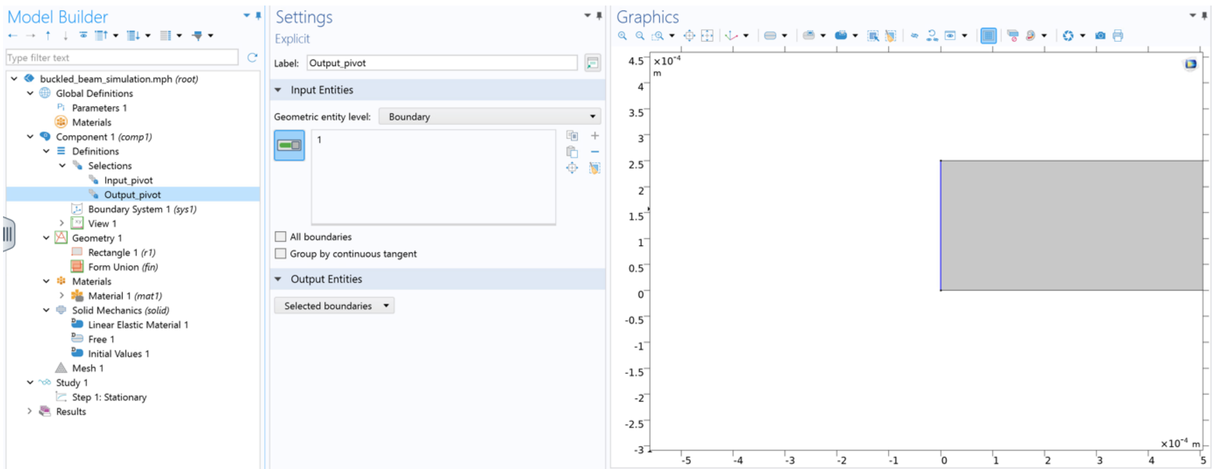
9. In *Component 1*, right click on *Materials*, click on *Blank Material*. In *Settings* of *Material 1*, add the following parameters. They correspond to the hardened Spring Steel (1.1274).



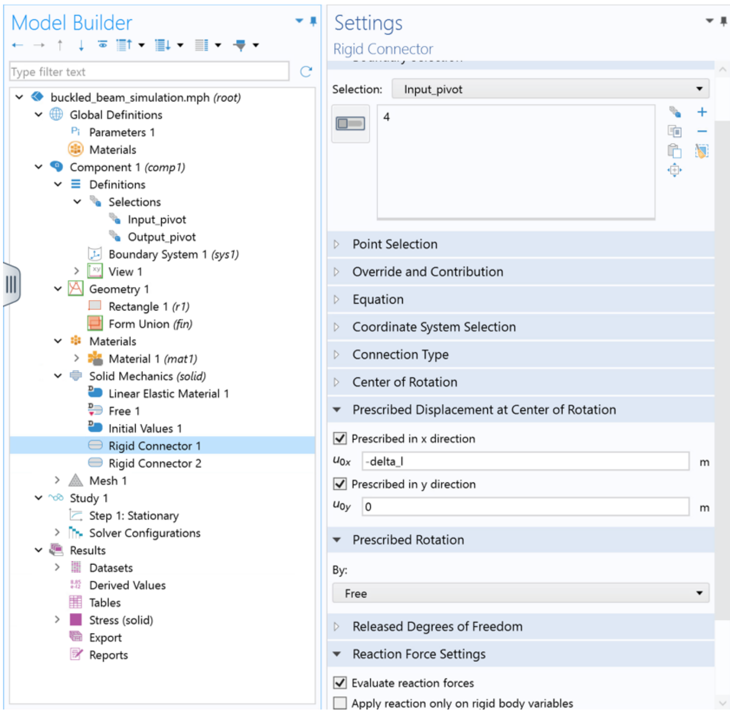
10. In the *Model Builder*, click on *Solid Mechanics*, then in the *Settings* pane, tick on the box *Enable physics symbols*, chose *2D Approximation* as *Plane stress* and set the out-of-plane *Thickness* of the model equal to *b*.



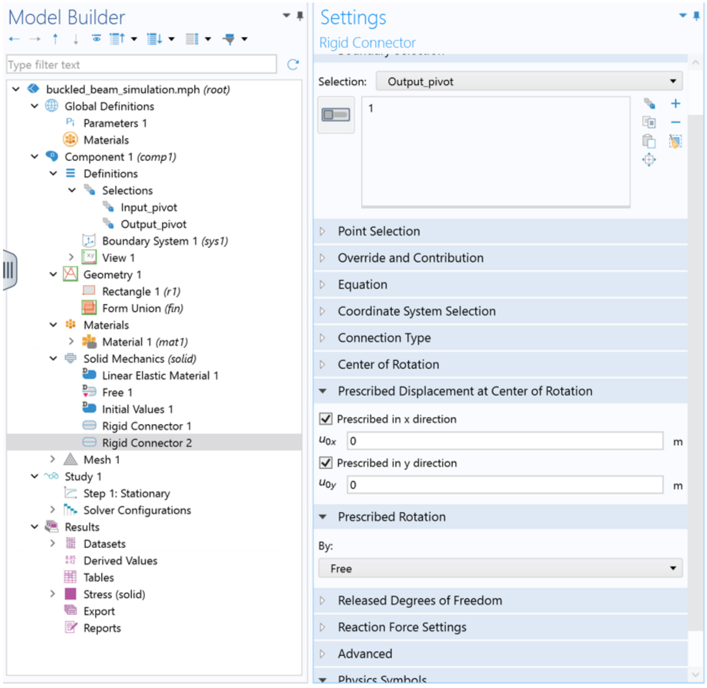
11. *Component 1* → *Definitions* → *Right click on Selections* → *Explicit*. Name the *Label Input_pivot*, set the *Geometry entity level* as *Boundary* and select the right edge of the beam. Use *Explicit* once again, and label the left beam extremity as *Output_pivot*.



12. Right click on *Solid Mechanics* → *Connections* → Add a *Rigid Connector*. In *Rigid Connector 1* → *Boundary Selection* → Select *Input_pivot*. Add a *Prescribed displacement* of *-delta_1* and 0 in x and y directions, respectively. Let the rotation free for the moment. Tick *Evaluate reaction forces* in *Reaction Force Settings* (this will be useful to evaluate the axial buckling load).



13. Add a second *Rigid Connector* corresponding to the *Output_pivot*. Add a *Prescribed displacement* of 0 in both x and y directions. Let the rotation free.



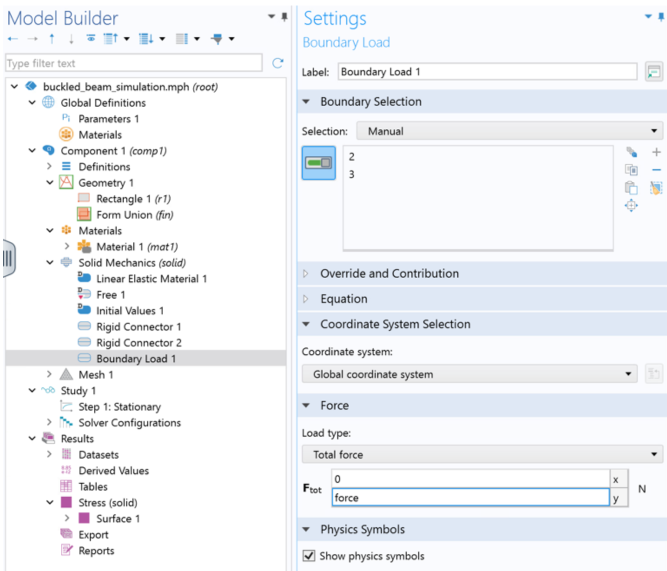
14. Right click on *Mesh 1* → Add *Mapped*. Then on right click on *Mapped 1* → Add *Distribution*. In *Distribution 1*, select the two short edges of the beam and insert 8 as the *number of elements*.

15. Once again, right click on *Mapped 1* → Add *Distribution*. In *Distribution 2*, select the two long edges of the beam and insert 100 as the *number of elements*. Click on *Build All* and verify the resulting mesh in *Graphics* window.
16. In *Study 1* → *Step 1: Stationary*, tick the *Include geometric nonlinearity* box and click on *Compute*.
17. Observe the deflected solution in *Stress (solid)*. You should see that the beam is straight and thus has not buckled. This compression state is unrealistic since any perturbation force applied on the side of the beam will lead to buckling. Even if this resulting deflection is strongly unstable, the FEM solver converges to this solution since it corresponds to a beam deflection equilibrium. In order to obtain a buckling solution, let's apply a lateral force to the beam to ensure that it buckles into one of its two stable states.
18. To this end, in *Global Definitions* → *Parameters 1*, enter a new variable named *force*.

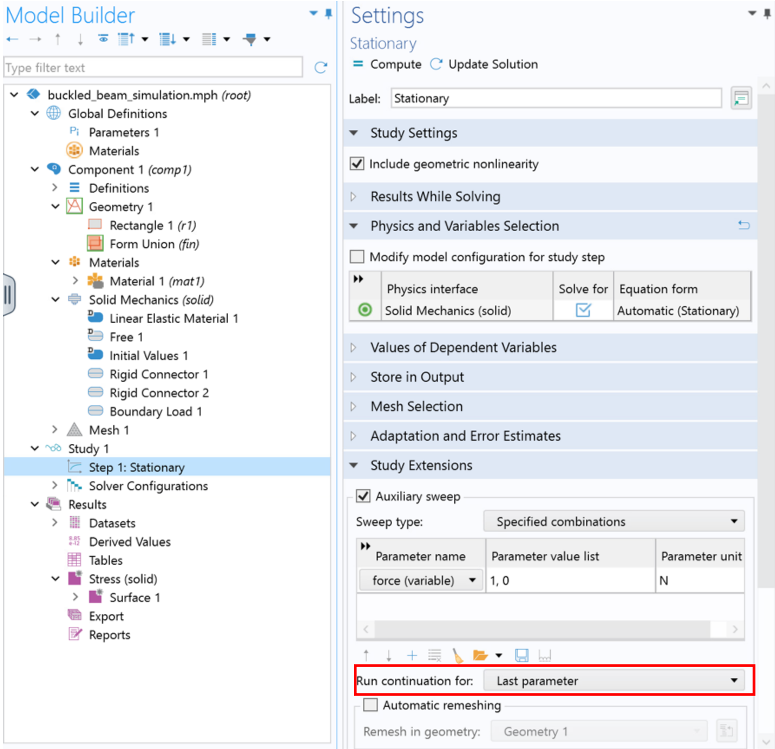
Parameters

Name	Expression	Value	Description
E_steel	210 [GPa]	2.1E11 Pa	constant
sigma_max	1600 [MPa]	1.6E9 Pa	constant
L	200 [mm]	0.2 m	constant
b	25 [mm]	0.025 m	constant
my_h	0.25 [mm]	2.5E-4 m	constant
delta_l	5 [mm]	0.005 m	constant
angle_in	0 [deg]	0 rad	variable
force	0 [N]	0 N	variable

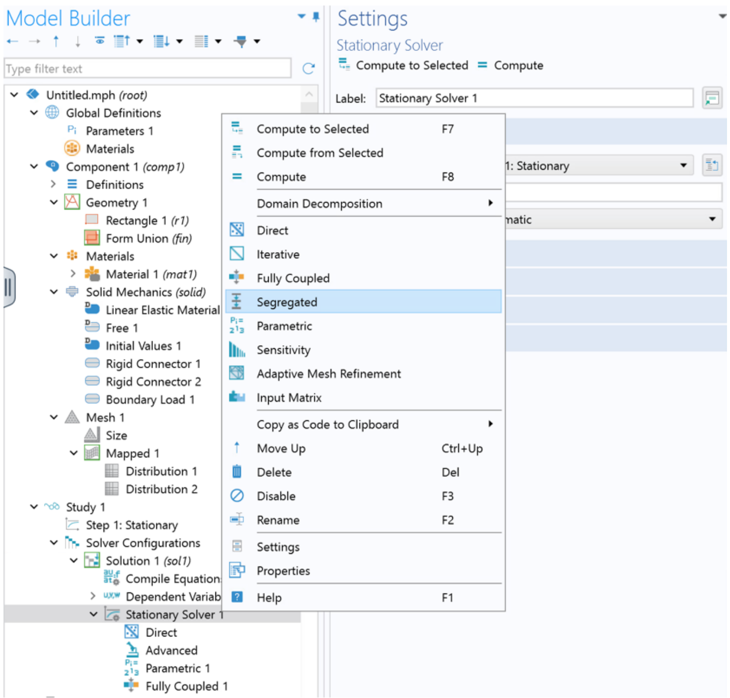
19. Right click on *Solid Mechanics* → Add a *Boundary Load*. In *Boundary Load 1* → *Boundary Selection* → select the two long edges of the beam. In *Load type* select *Total force*, and enter the variable *force* in the y direction.



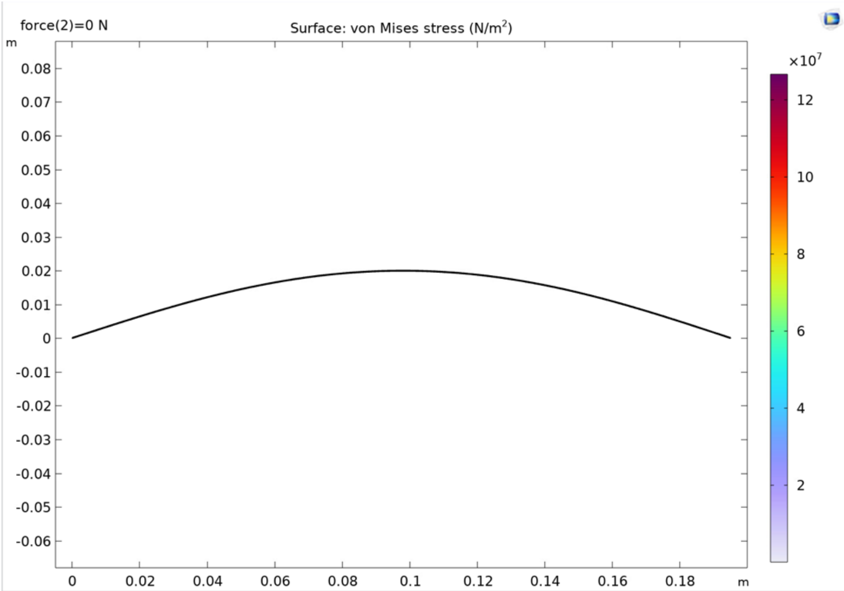
20. In *Study 1→ Step 1: Stationary → Study Extensions*, tick the box *Auxiliary sweep* and then click on the *Add* button and select *force* as the swept parameter. First, apply a lateral force of 1 N to force the beam to buckle upwards and then reduce the force to 0 N to obtain the stable deflection. Note that the solver automatically reuses the previous solution (i.e., *force*=1 N) to compute the next solution (i.e., *force*=0 N) if we use the default setting *Run continuation parameter for -> last parameter*.



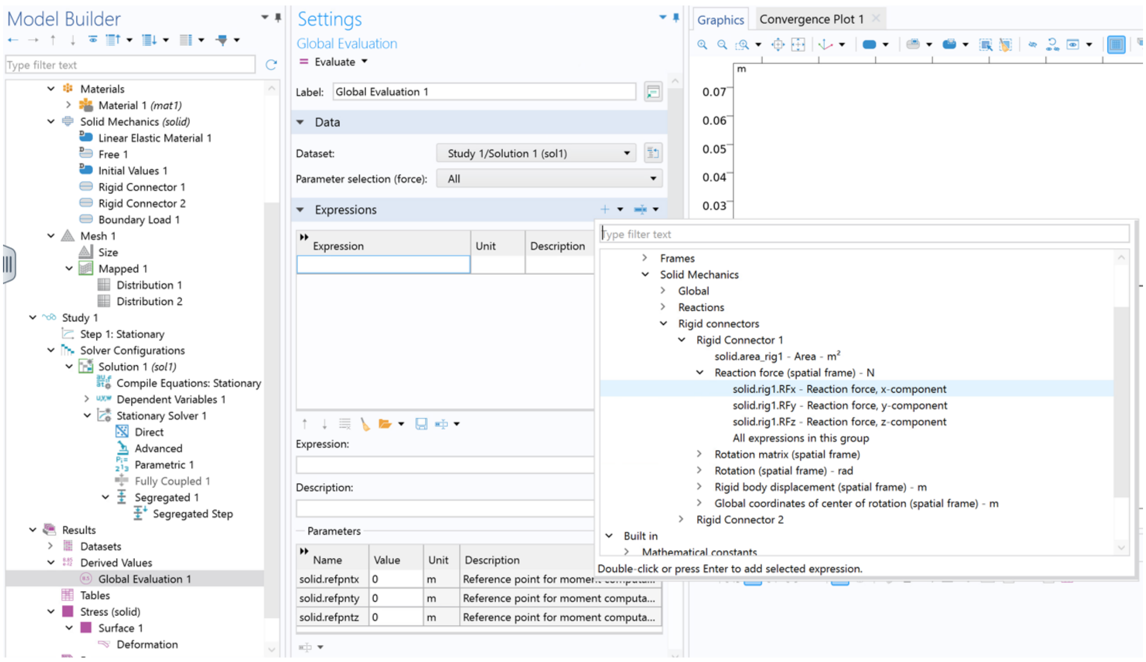
21. In *Study 1→ Solver Configurations → Solution 1 (sol1)*, right click on *Stationary solver 1* and click on *Segregated*. This will help the solver to converge. Press on *Compute*.



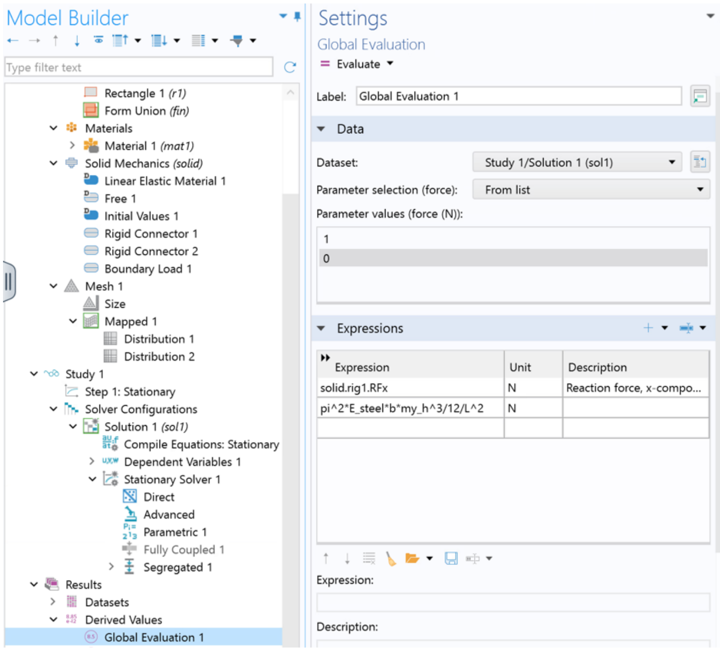
22. Observe the deflection solution in *Stress (solid)* when *force*=0, and visually verify that the beam deflection now corresponds to a **buckling stable equilibrium** rather than a **compression unstable equilibrium**.



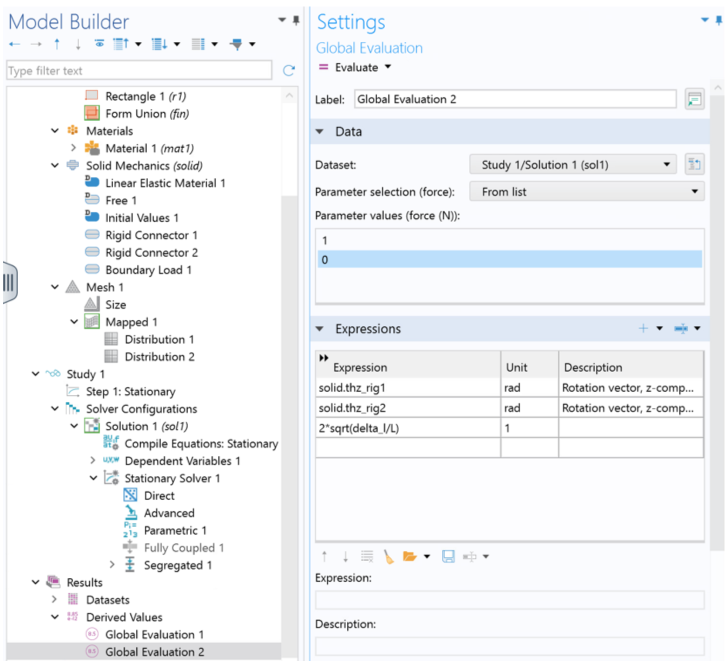
23. In *Results* → *Derived Values* → Add a *Global Evaluation.* In *Expressions*, pressing on the *Add* button, select *Solid Mechanics* → *Rigid connectors* → *Rigid connector 1* → *Reaction force* → *x-component.*



24. Still in *Expressions* of *Global Evaluation 1*, add the equation of the Euler's critical load (first mode of buckling) of a pinned-pinned beam. Compare the analytical and FEM values (when *force*=0) by pressing on *Evaluate*.



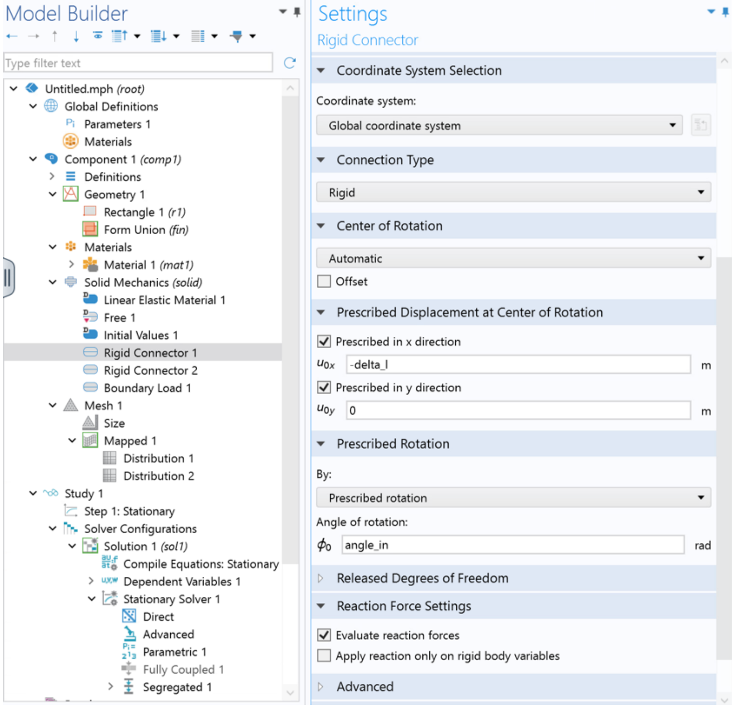
25. Add a second *Global Evaluation.* In *Expressions*, pressing on the *Add* button, select *Solid Mechanics → Rigid connectors → Rigid connector 1 → Rotation → z-component.* Proceed the same way to add the rotation of *Rigid connector 2* in *Expressions.* Add the equation of the stable angles of a pinned-pinned buckled beam (the formula is given in the paper https://doi.org/10.1016/j.mechmachtheory.2022.104874, Eq. (25)). Compare the analytical and FEM values (when *force*=0) by pressing on *Evaluate.*



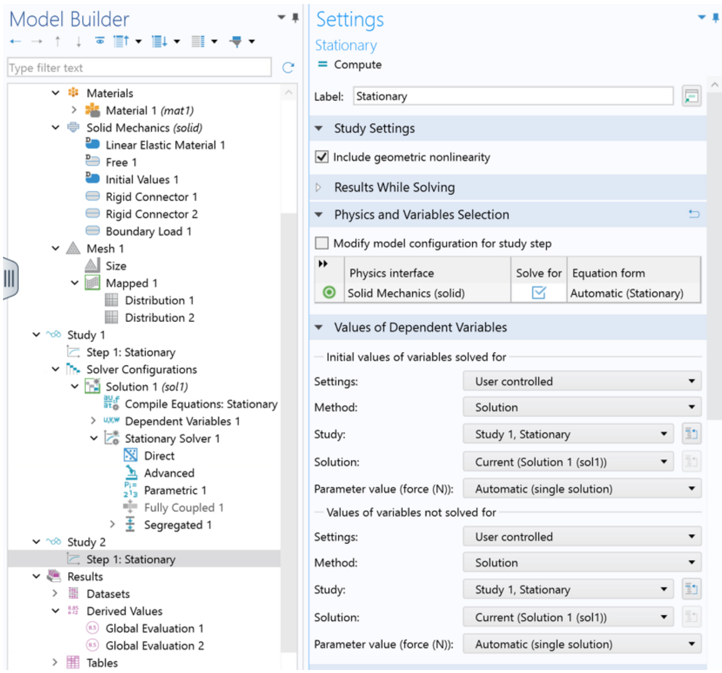
EPFL

Task 2

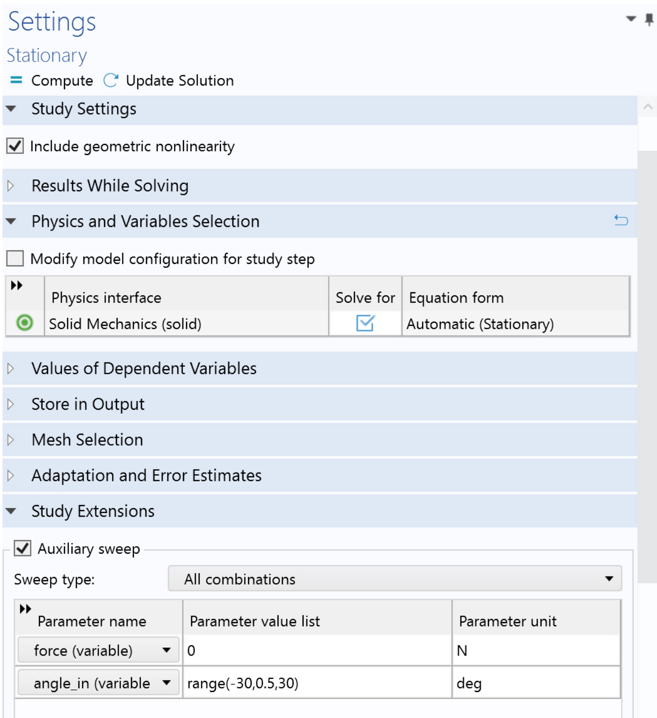
26. We will now actuate the beam at the input pivot. In *Model Builder → Solid Mechanics → Rigid Connector 1*, add a *Prescribed rotation* and enter the parameter *angle_in*.



27. Create a second study using the ribbon → *Study → Add Study → Stationary*. In *Study 2 → Step 1: Stationary*, tick the box *Include geometric nonlinearity*. In order to reuse the buckling solution of the first study (Study 1), in *Values of Dependent Variables*, select the following settings:

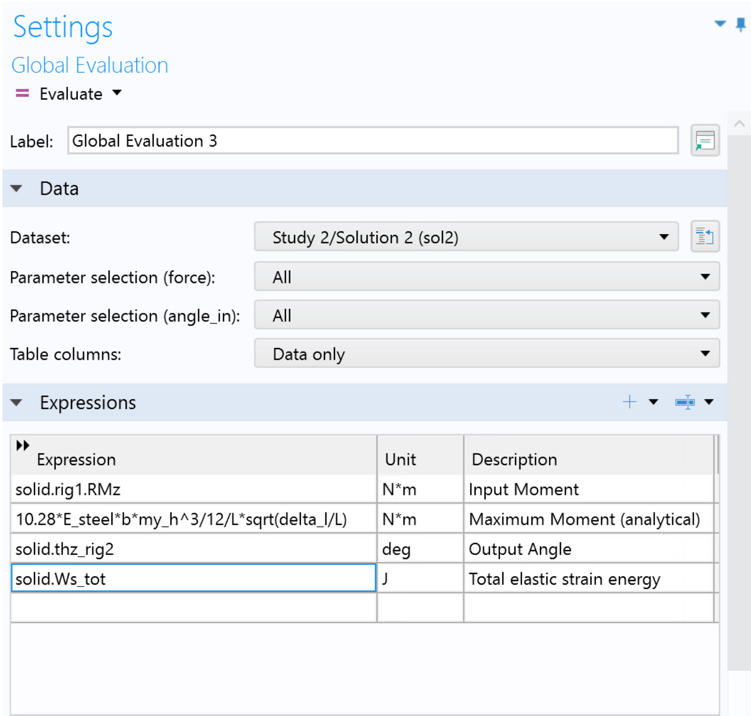


28. Right click on *Study 2* and click on *Show Default Solver*. Wait a for moment. Use a segregated approach as before: in *Study 2→ Solver Configurations → Solution 2 (sol2)*, right click on *Stationary solver 1* and click on *Segregated*. In *Step 1: Stationary* of *Study 2*, tick the box *Auxiliary sweep* and then click on the *Add* button to select *force* and *angle_in* as swept parameters. Fill the range and unit as specified. Click on *All combinations* in *Sweep type* because the two parameters have not the same number of values. Press on *Compute*.

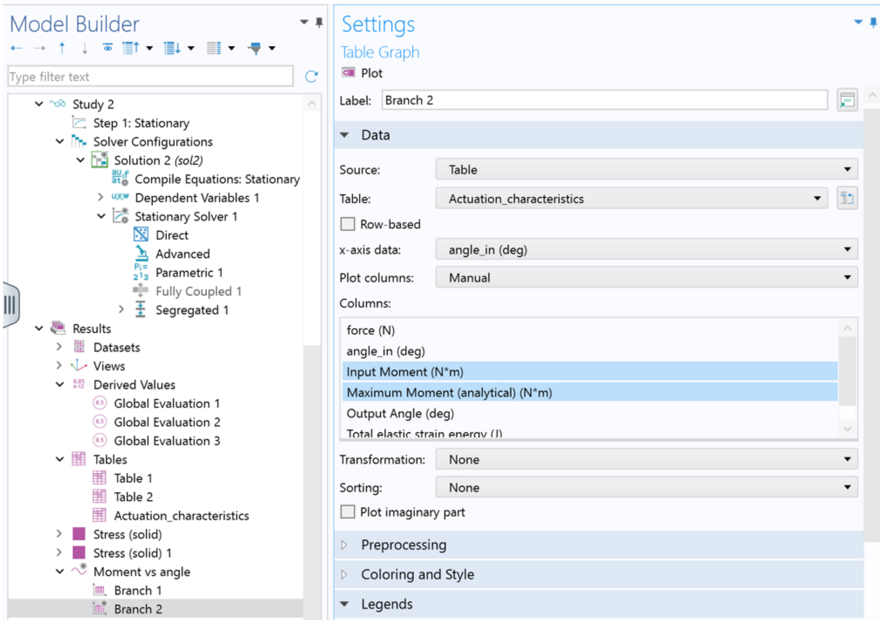


29. In the *Convergence Plot* window, try to locate the iteration number where the solver is computing the beam deflection at the snap-through instability. (Hint the solver requires more iterations to converge to find a single solution when the beam is starting to snap).

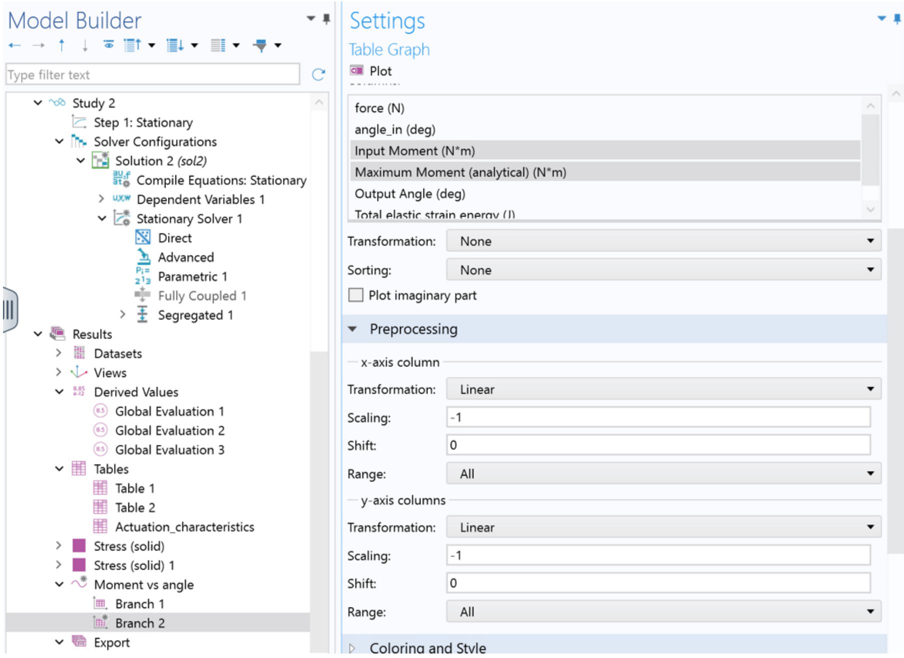
30. In *Results → Derived Values* → add a *Global Evaluation*. Select the dataset *Solution 2*. In *Expressions*, write the following parameters (with their respective description) then click on *Evaluate*.



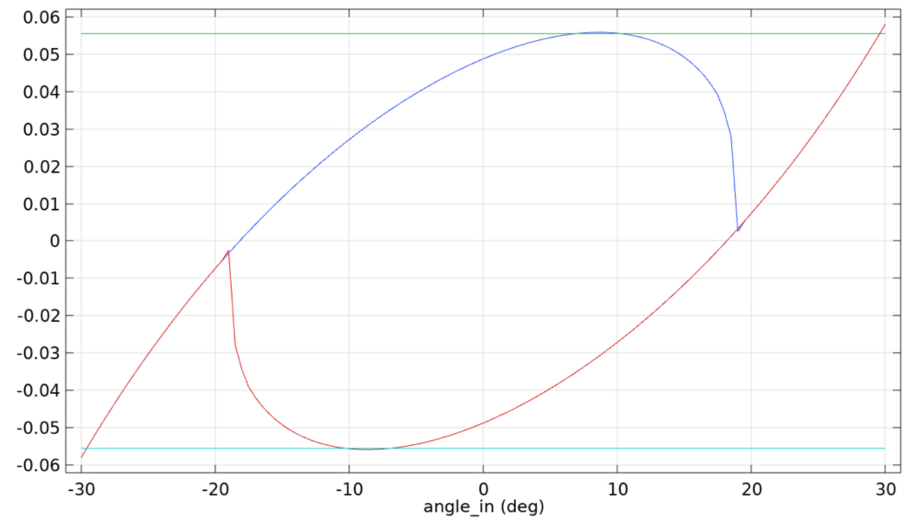
31. Rename the table that you have just created as *Actuation_characteristics.* Then, right click on *Results* and select *1D Plot Group.* Label this plot group as *Moment vs angle.* Right click on *Moment vs angle* and select *Table Graph,* two times. Rename the first table graph as *Branch 1* and the second one as *Branch 2.* Select the *Actuation_characteristics* as the table source, *angle_in* as the *x-axis data*, and *Input Moment* and *Maximum Moment (analytical)* as the *Plot columns* for both *Branch 1* and *Branch 2.*



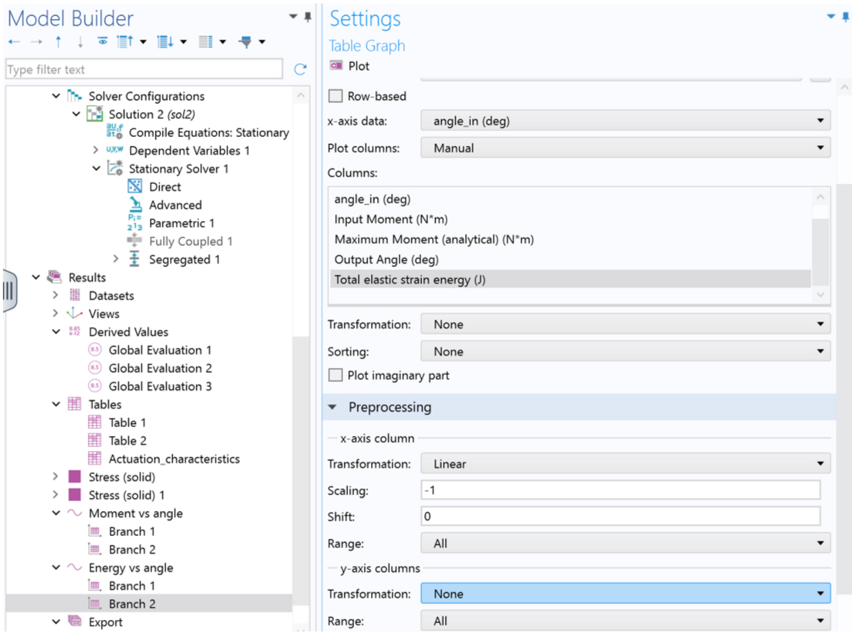
32. Now, only for Branch 2, in *Preprocessing* settings, apply a linear transformation with a scaling value of -1 for both the x and y axes.



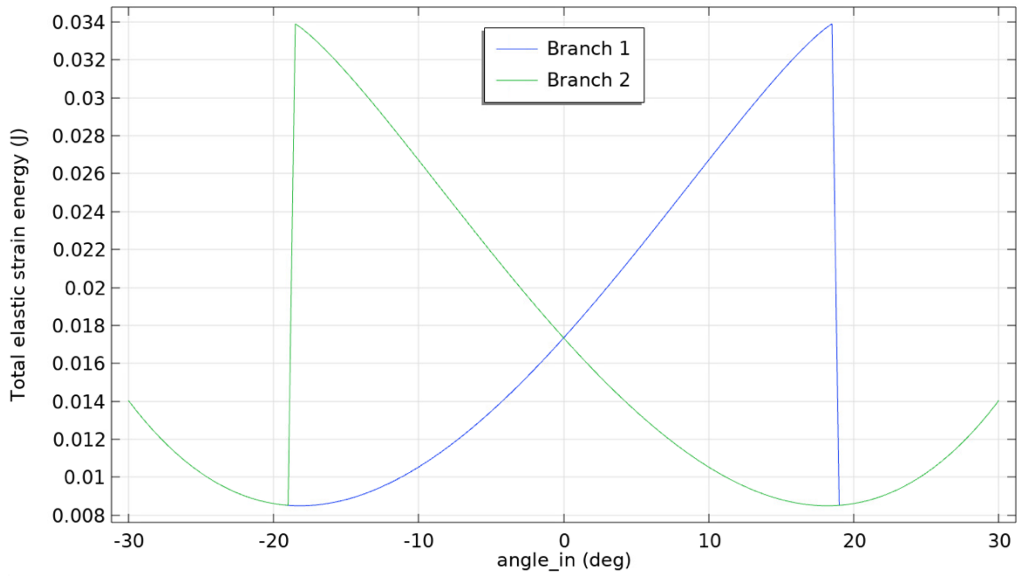
33. Click on *Moment vs angle* → *Plot.* You can now see the hysteretic curves of the input moment as a function of the input angle. The analytical limit moments are also plotted by upper and lower straight lines, in order to be compared with the FEM curves.



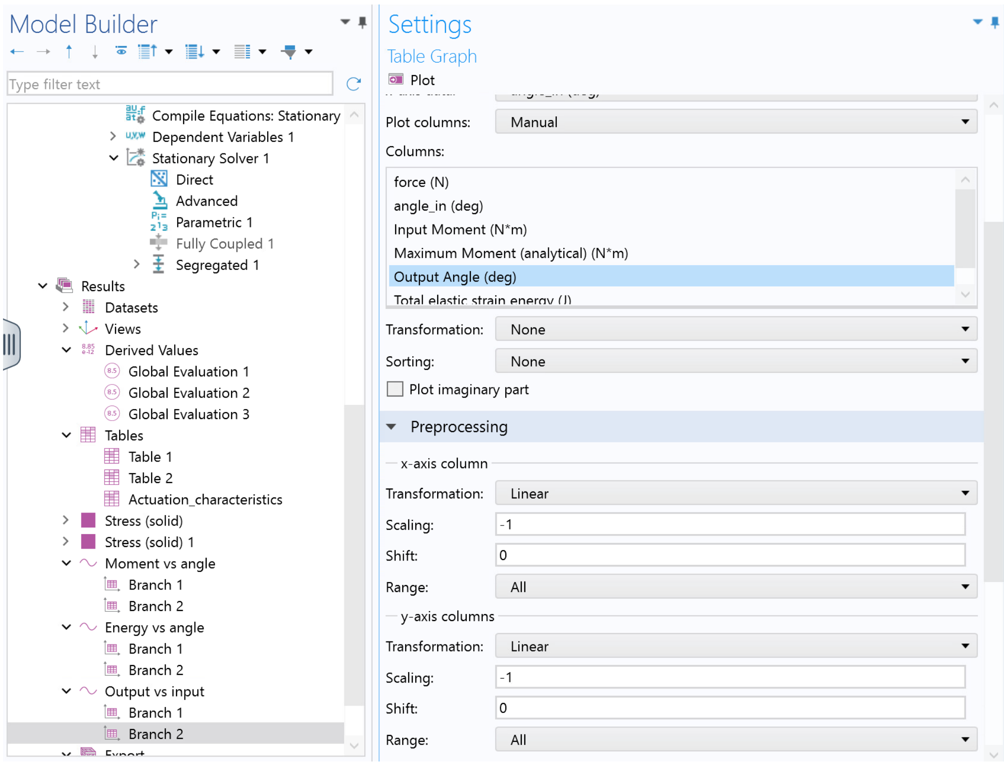
34. In this graph, it is possible to locate the stable and unstable states of the buckled beam. However, to analyze the stability of a structure, it is often easier to observe its potential energy. Let's thus compute the strain energy of the buckled beam. Right click on *Moment vs angle* and select *duplicate.* Rename the new 1D plot group as *Energy vs angle.* In *Energy vs angle,* in *Branch 1* and *Branch 2,* select *Total elastic strain energy* as the column. Remove the preprocessing transformation of the y axis for *Branch 2.* Click on *Plot.*



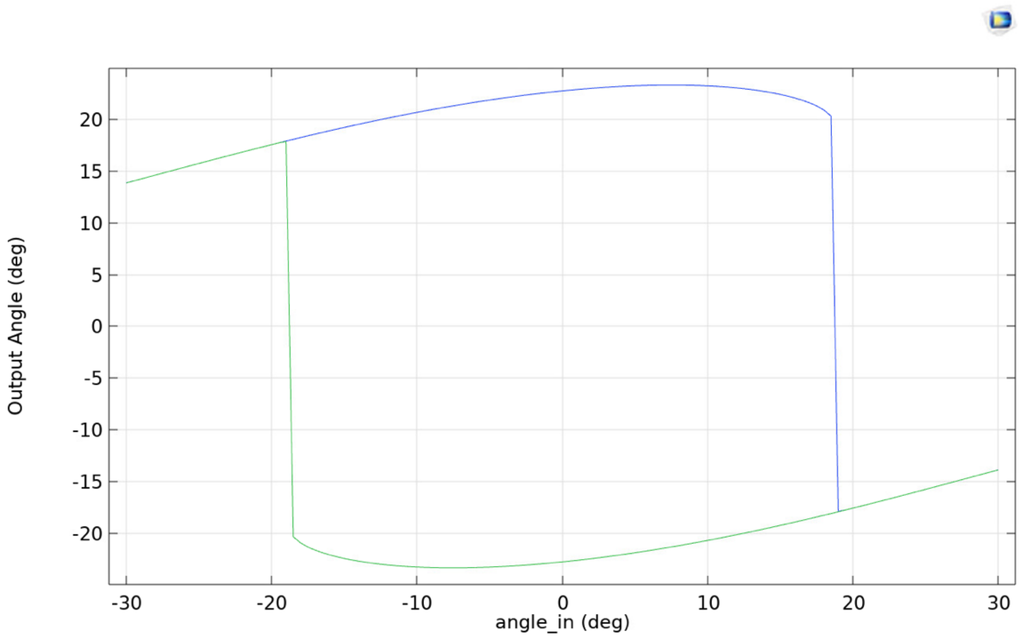
35. In the energy diagram, we can easily locate the stable equilibriums (local minima) and unstable states (local maxima or limit points).



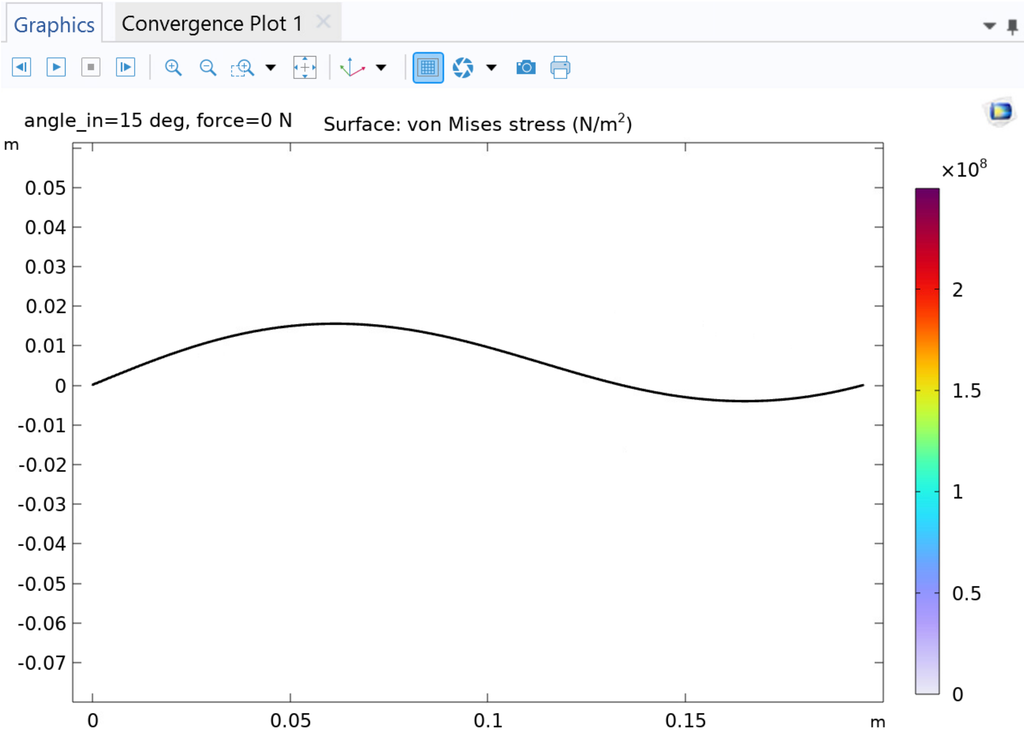
36. We can also plot the output angle as a function of the input angle. To do so, right click on *Moment vs angle* and select *duplicate.* Rename the new 1D plot group as *Output vs input.* In *Output vs input.,* in *Branch 1* and *Branch 2*, select *Output Angle* as the y-axis column. Click on *Plot.*



37. The graph of the output angle as a function of the input angle shows that the output pivot suddenly rotates to the opposite side when the beam snap-through is reached.



38. Let's observe this behavior in an animation. In *Results* → right click on *Export* → *Animation* → add *Player*. Select *Stress (solid) 1* as the subject. In the settings of *Animation* → *Playing* → *Repeat* → choose *Forever*. On the *Graphics* window, click on the *Play* button and enjoy watching the actuation of the buckled beam and its snap-through!



To summarize, we have seen in this tutorial:

- To apply and evaluate displacements, angles, forces and moments in a structure
- To plot load-deformation characteristics that are highly nonlinear (e.g., hysteresis and snap-through)
- To plot potential energy diagrams
- To use a segregated approach to subdivide the problem into two or more segregated steps to improve the solver convergence
- To observe and discuss the convergence plot
- And more particularly, how to simulate buckled beams:
    - Evaluation of the stable positions
    - Evaluation of the actuation characteristics