A step-by-step guide to simulating an ant-like piezoelectric robot

Nasim Rezaei[1], Rebecca Saive[1]

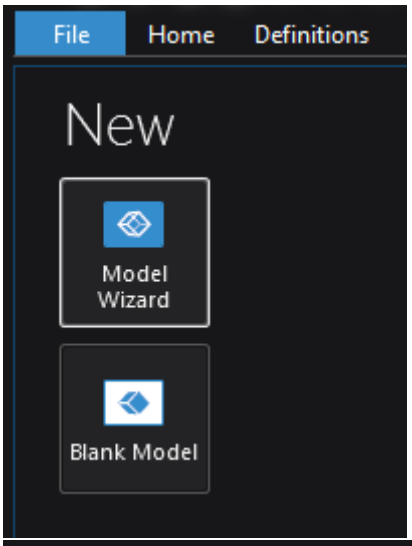
[1] University of Twente, Enschede, 7522NB, the Netherlands

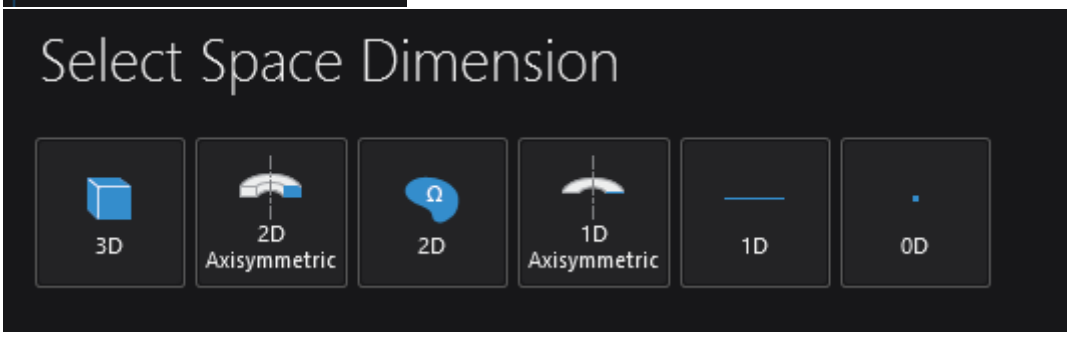
This tutorial is about building up a COMSOL model for a robot ant. In this model, we have a robot with a size of a few centimeters and which is based on piezoelectric ceramics. This robot has already been built by the University of Michigan and some COMSOL models only for the legs' operation are already existing and are published [1], [2].

From what we know, the full ant robot had not been computationally simulated before. Here, we provide a tutorial on how to successfully achieve this task. Furthermore, we uploaded a video on YouTube to visualize the results: https://youtu.be/ePyD8TAQt3I

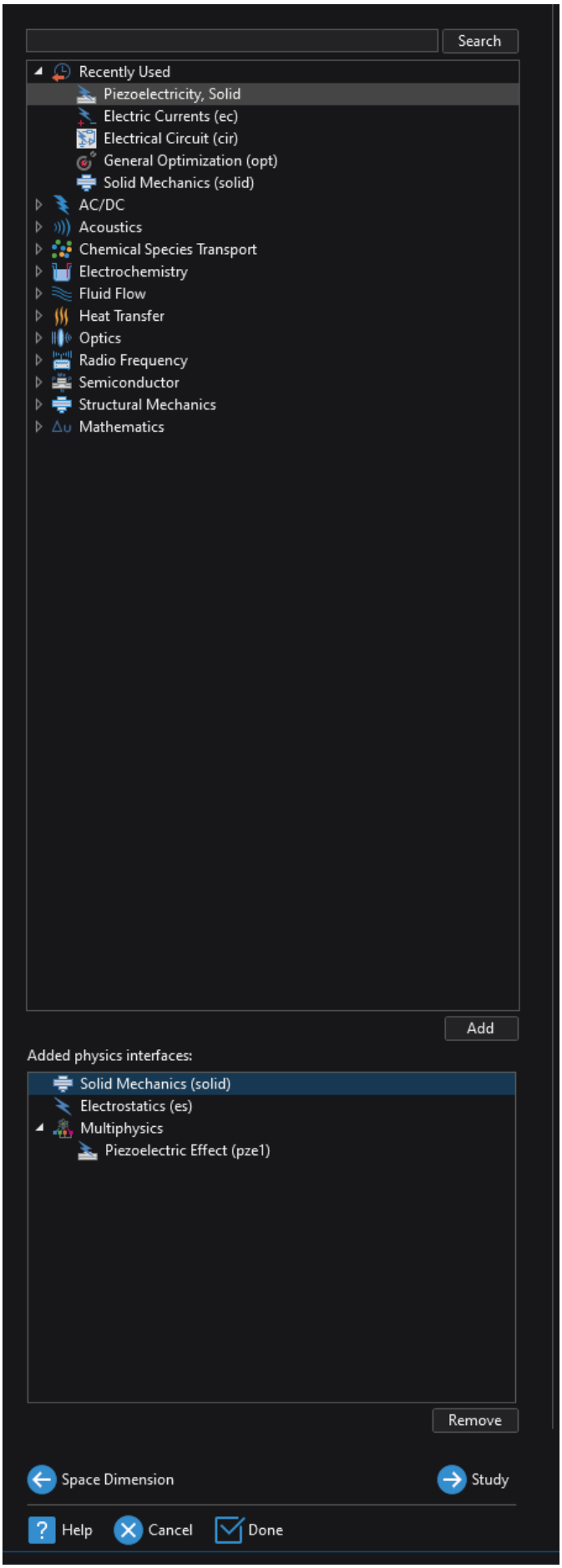
So the trick, in order to have a moving robot on a surface, is to activate friction and contact features in COMSOL. This is quite a challenging feature and the simulations might encounter convergence issues.

We start from a blank model in COMSOL. Go to model wizard and press 3D.

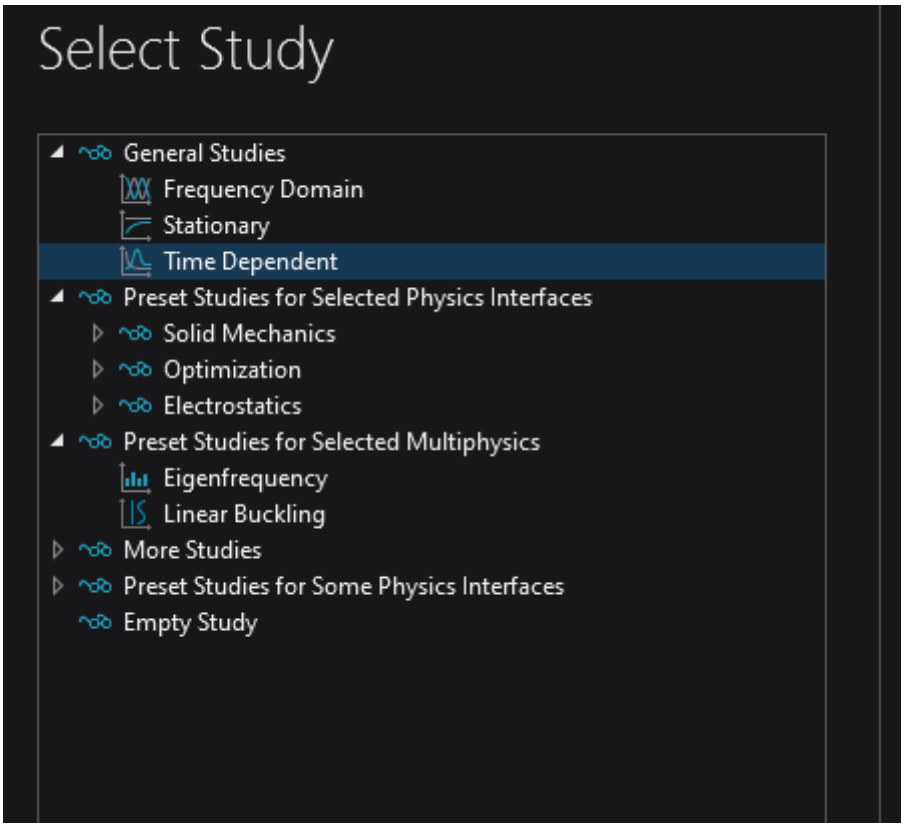




In this model we need to use piezoelectricity, which consists of two main modules, "solid mechanics" and "electrostatics".

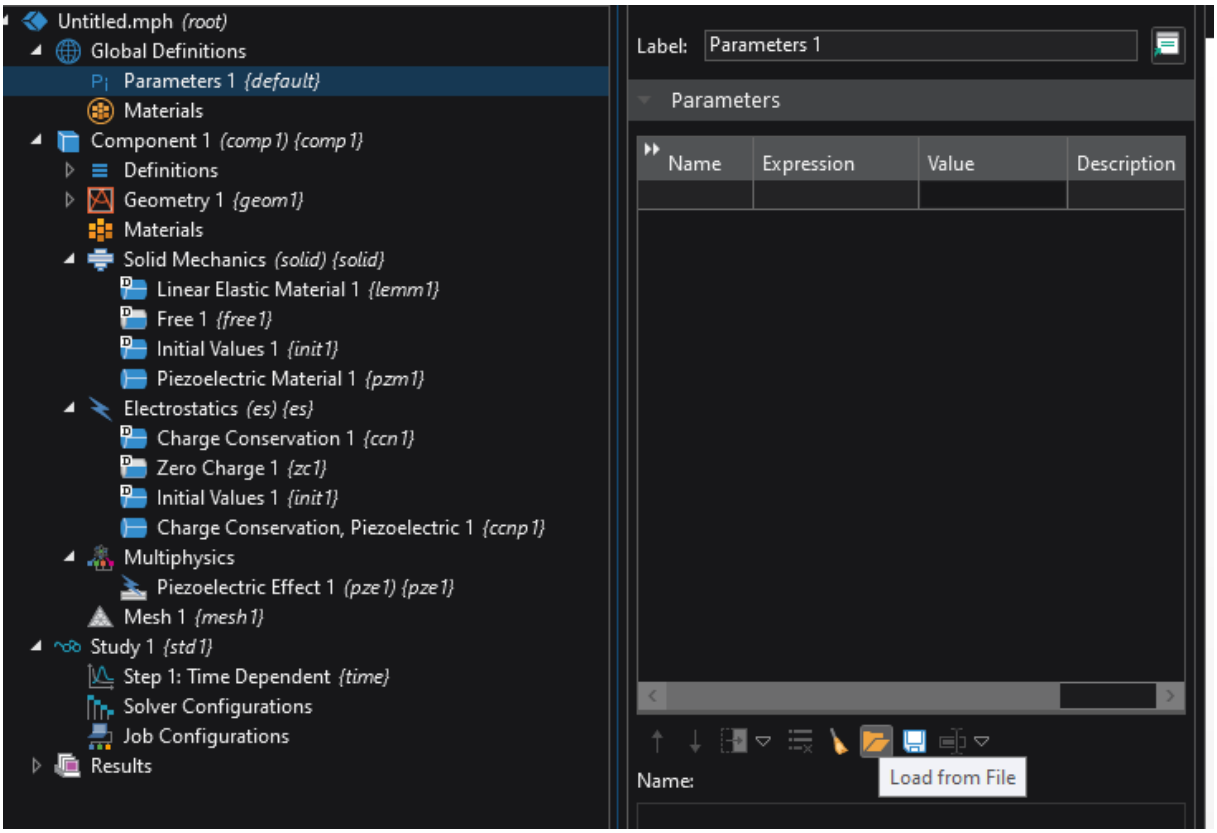

Search
Recently Used
Piezoelectricity, Solid
Electric Currents (ec)
Electrical Circuit (cir)
General Optimization (opt)
Solid Mechanics (solid)
AC/DC
Acoustics
Chemical Species Transport
Electrochemistry
Fluid Flow
Heat Transfer
Optics
Radio Frequency
Semiconductor
Structural Mechanics
Mathematics
Add
Added physics interfaces:
Solid Mechanics (solid)
Electrostatics (es)
Multiphysics
Piezoelectric Effect (pze1)
Remove
Space Dimension
Study
Help
Cancel
Done

Press study. In this simulation we need to work in time domain. Therefore, we use time dependent study.

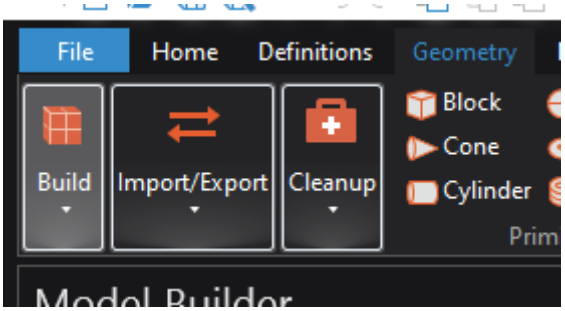


So now, we have a blank model. In order to speed up this tutorial I will import the already built geometry model and also the parameter set. In any case, building a model from scratch can be done with different approaches depending on the user. The final geometry should be the same.

Under “global definitions”, press “parameters”, and then press “load from file”.

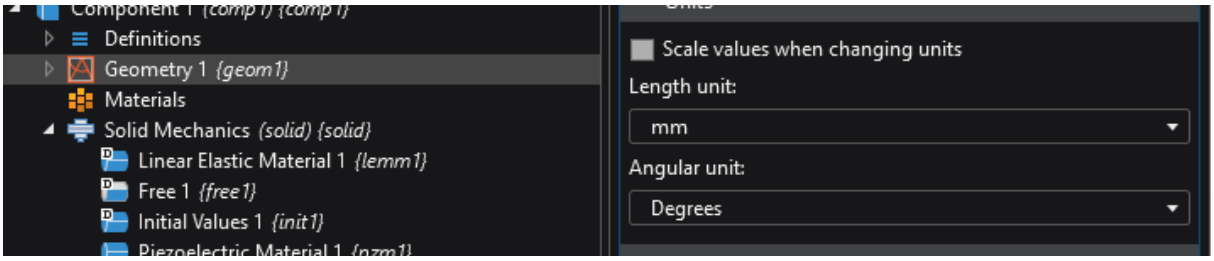


Then select your parameter set in the form of text files and open it. After importing the parameters, it's time to import the geometry. Under the "geometry" tab press "import/ export", press "import" and then browse, select your geometry file with the extension " mphbin". After the selection press import.

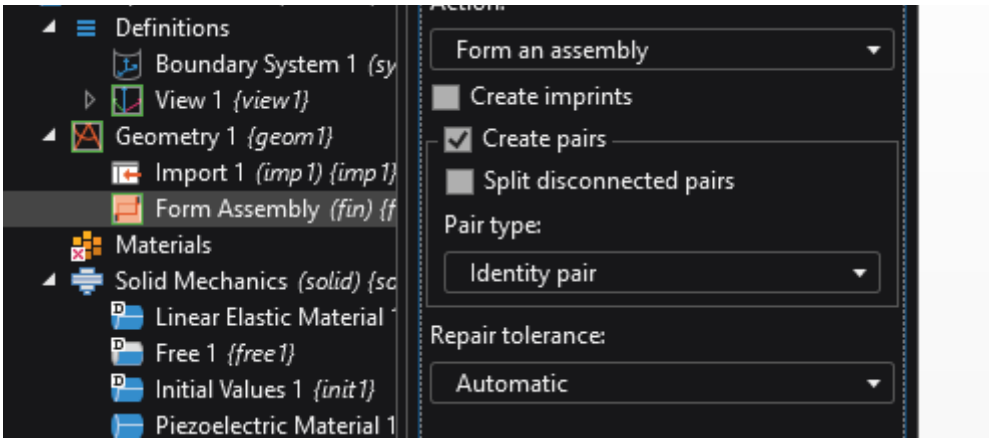


It is important to note that, importing a finalized geometry means that, its parameters are not changeable. Therefore the parameter set that is imported cannot be used for altering the size of the geometry. However, what I call "soft variables" can be changed in the model, e.g. "frequency, voltage, etc".

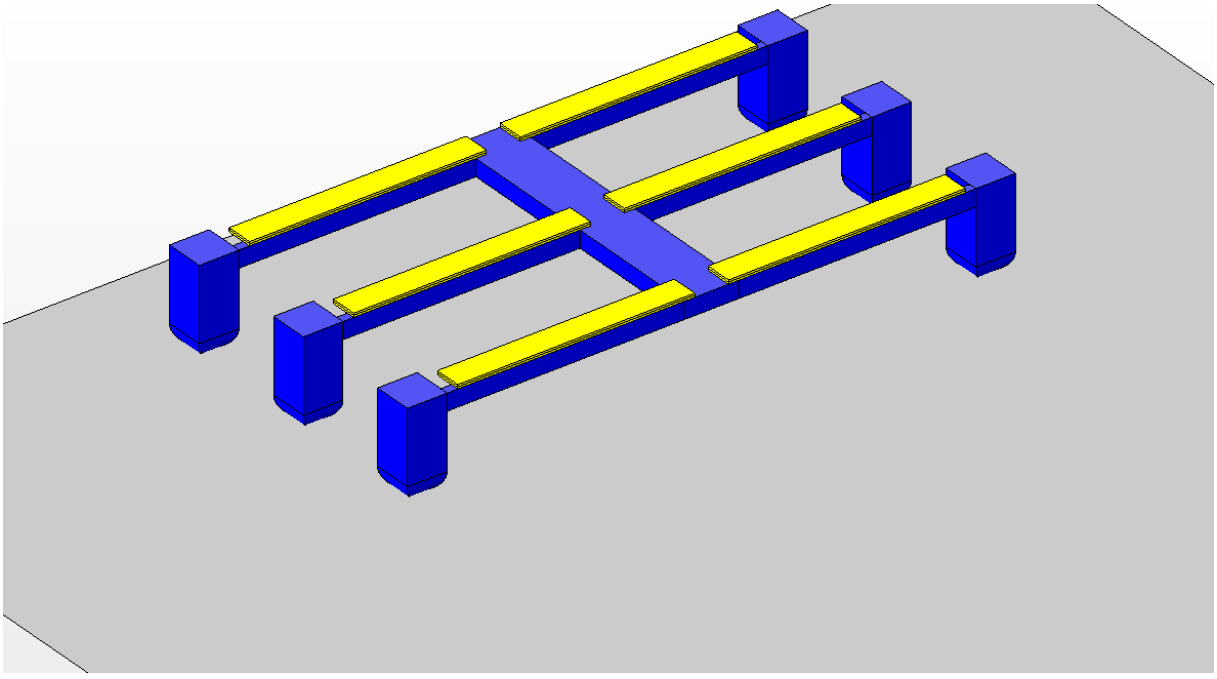
You should remember to select the correct length units, otherwise the geometry that is imported will be scaled according to the default unit. If that happens you can press on geometry tab and then change the units according to the desired geometry units. In our case the desired unit is millimeters. You should remember to unselect "scale values when changing units" option.



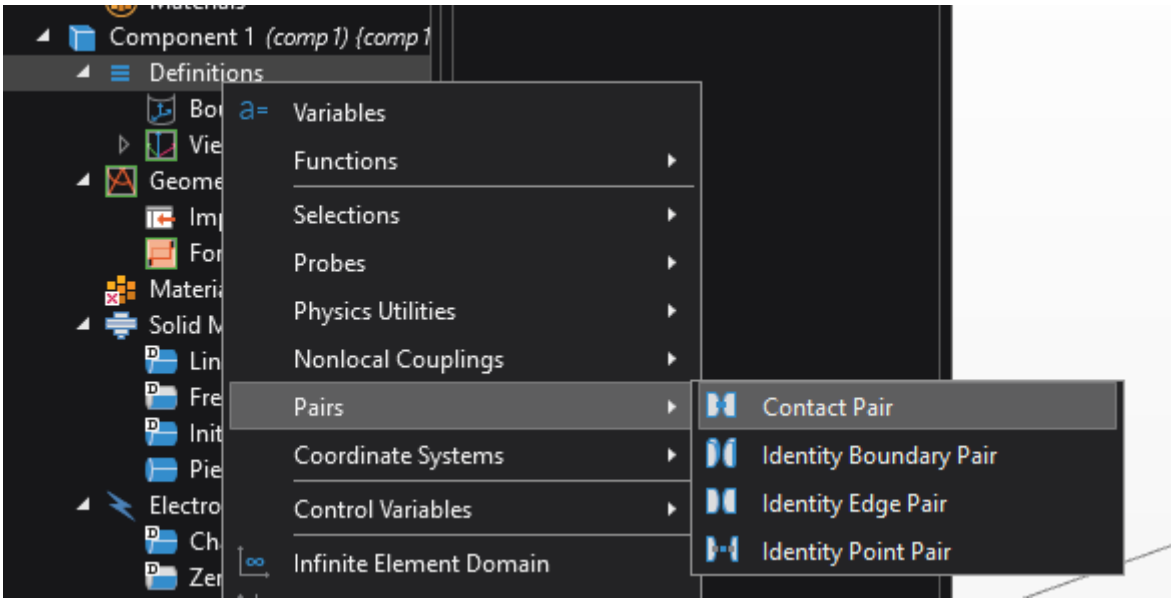
Also, instead of “form union” you need to press “form an assembly”. Select “create pairs” and for the pair type you need to select “identity pair”. This step is necessary in order to prevent the robot body from fusing into the ground level.



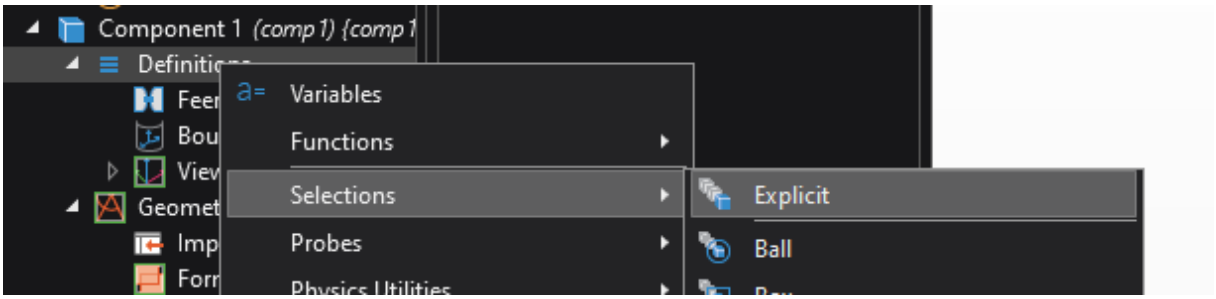
Wider explanations about this geometry: as you can see this geometry consists of three main parts, firstly a ground floor (in grey), then the main robot body (in blue), which consists of the 3D-printed polymer mainframe, including legs, and then the piezoelectric arms (in yellow). The piezoelectric arms are biomorphic piezoelectrics made of PZT ceramic.



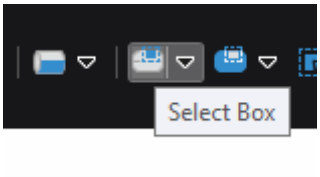
Now, we need to establish a contact between the feet and the ground. For that, under the “definitions” tab, right click on “definitions”, press “pairs” and then select “contact pair”.

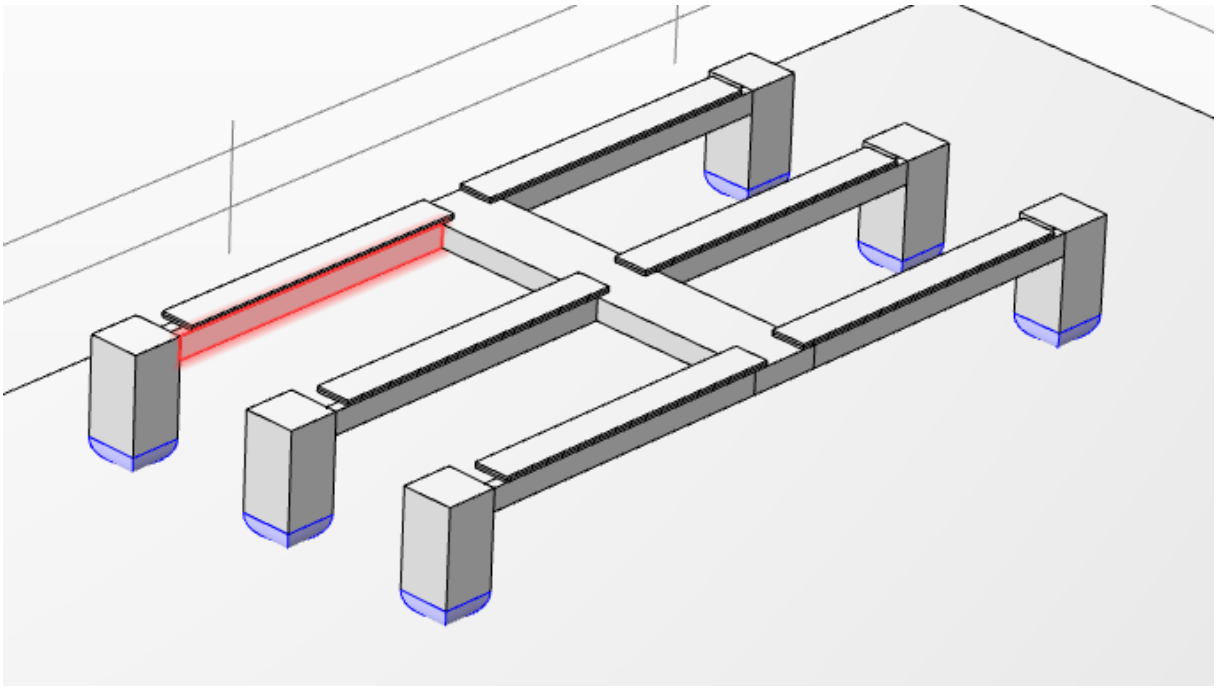


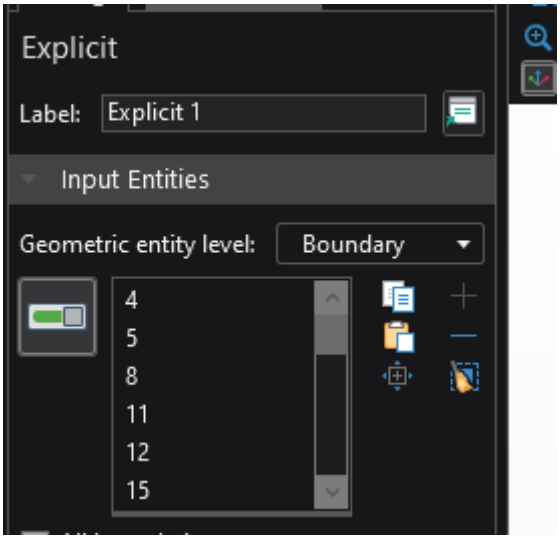
For the “source”, we select the ground boundary. Generally, when selecting contact pairs, the source boundaries are the ones that are stiffer or harder materials and the “destination” boundaries are generally the softer ones or sometimes the moving parts. Now for the destination boundaries, we can do two things: either we can select the bottom surface of the feet one by one, or we can form an “explicit selection” of those surfaces and then do a onetime selection, which makes everything easier. This is how it is done: right-click on “definitions”, press “selections” and then “explicit”:



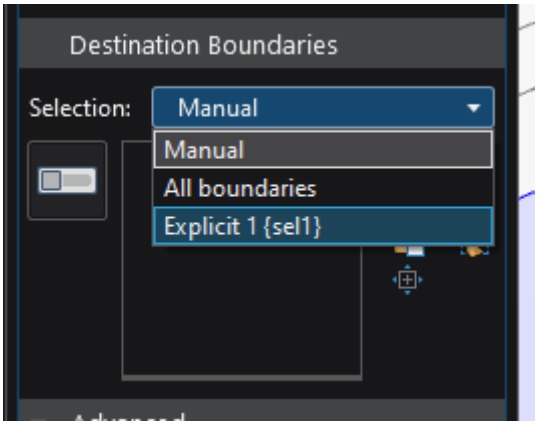
For the “geometric entry level”, press boundary, and then select the boundaries that you would you like to use as the destination boundaries. For a faster selection, you can use the option “select box”, which means that whichever boundary you include in your selection, by dragging the mouse cursor, will be selected in the explicit selections.





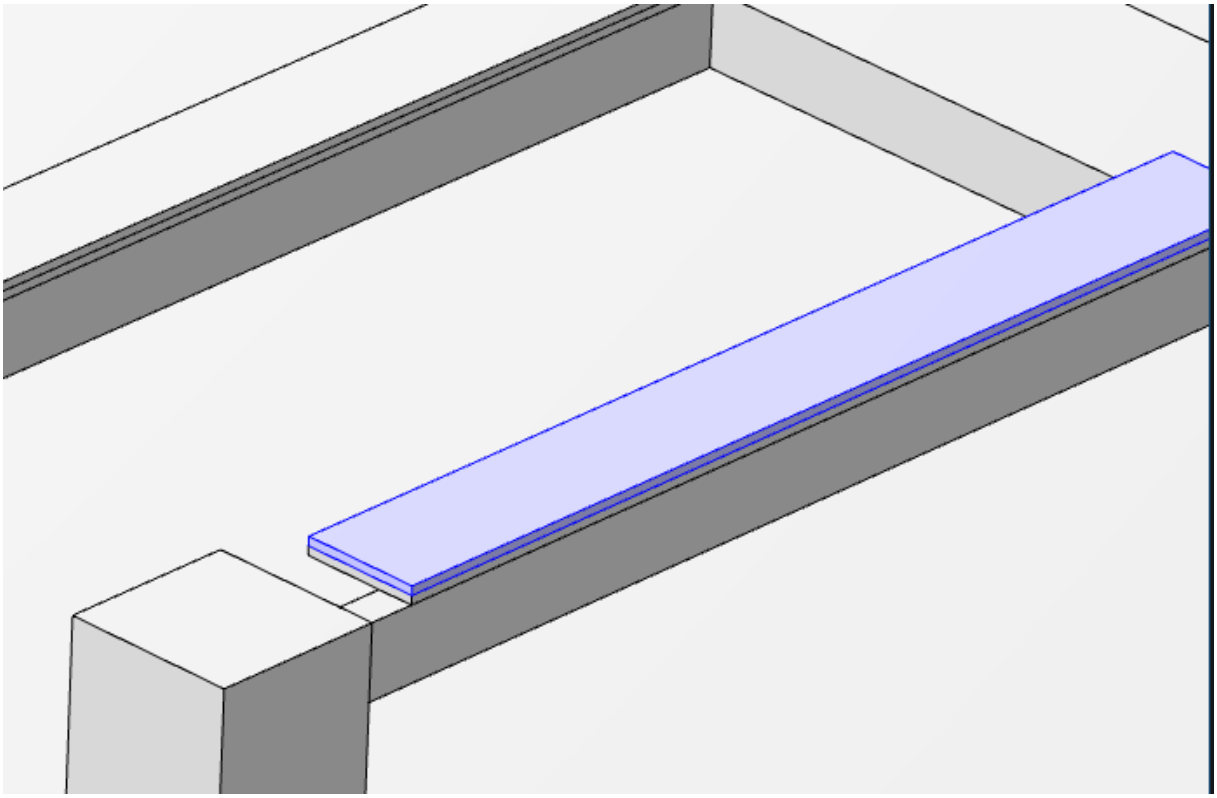


Now back to the feet contact selections for the destination boundaries, simply select “Explicit 1” and all the feet’s lower boundaries will be selected.



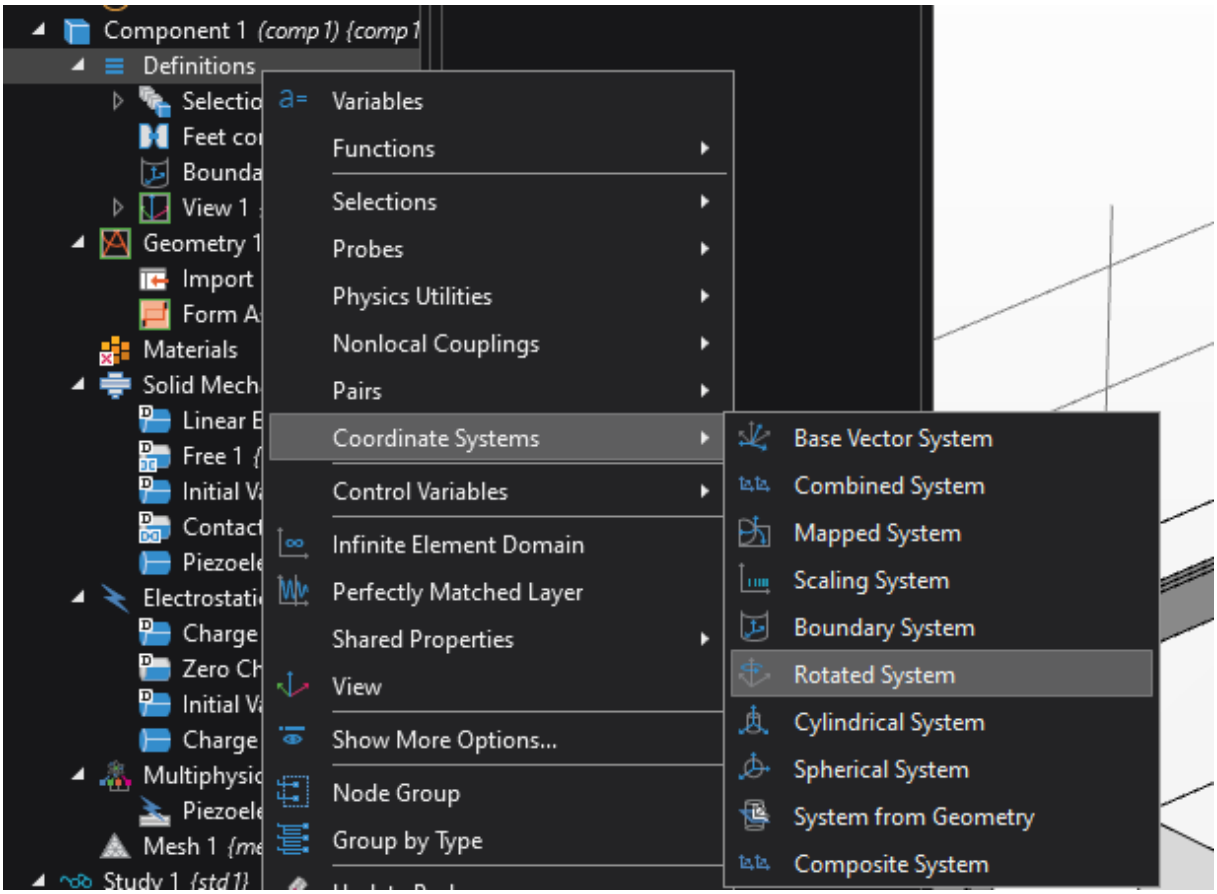
As a search method you should choose “direct”. This makes the simulations a bit slower, but it’s more accurate for the case when the object is moving along the surface and therefore, new contact points need to be searched every time there is a movement.

We can also rename the “Explicit 1” selection into “feet BND” for future reference. We need to make another two sets of selections for the different polarizations of PZT ceramics. This is because for bimorph PZT, we need to have different sets of coordinate systems. Again, right click on “definitions”, then “selections”, then “explicit”. For “label” write down “PZT +Z”. This is for the PZT ceramics with +Z polarization. As the geometric entry level, select Domain. Then select the top PZT ceramics that are on each leg; therefore six top PZT to ceramics should be selected:



Repeat the same process once again, for “PZT – Z” and this time select the six bottom PZT ceramics.

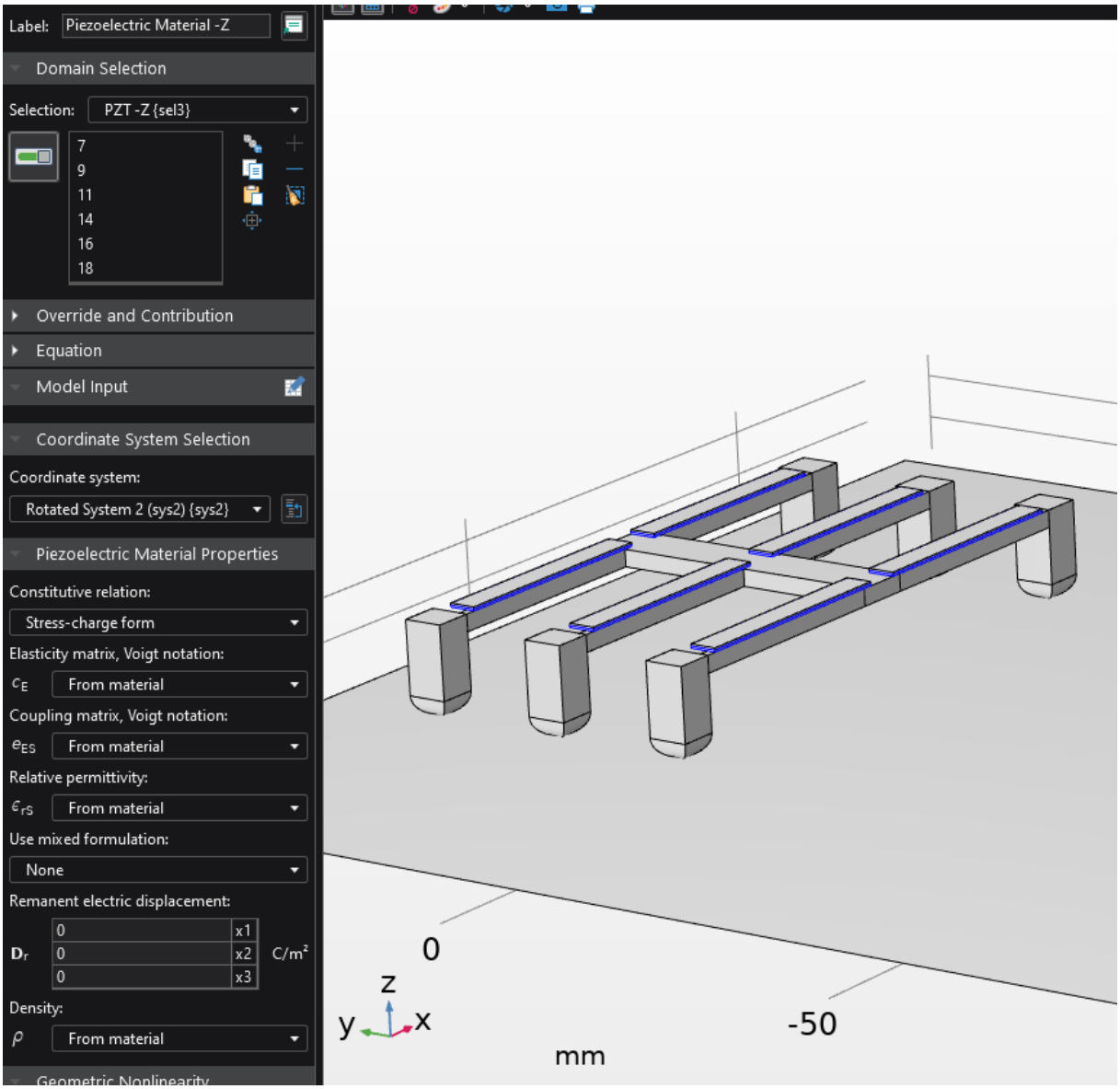
Now you need to define a rotated coordinate system, such that the polarity of the bottom PZT ceramics can be in reference to *this* coordinate system. From “definitions” and then “coordinate system”, select “rotated system”.



Change the β value from 0 to pi, and keep the rest the same.

Now it is time to define the materials. We have two main materials, the polymer body and PZT. Right click on “materials” and select “browse library”. Search for “PLA” and find “polylactic acid” under “polymers”. Choose this material. Also, search for “PZT- 5H” and select that one too. The main body should be assigned to PLA, and the ceramics should be assigned to PZT.

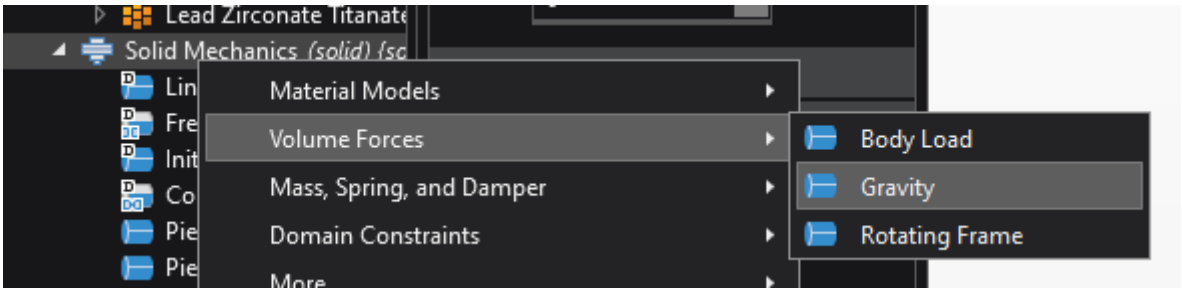
It Is time to set the settings for “solid mechanics”. By default, there should be a “piezoelectric material” under “solid mechanics”. But we need two piezoelectric materials. Rename the first one as “piezoelectric material – Z”, for which the coordinate system should be “rotated system 2”. Also the selection should be “PZT – Z”, i.e. all the bottom PZT layers:



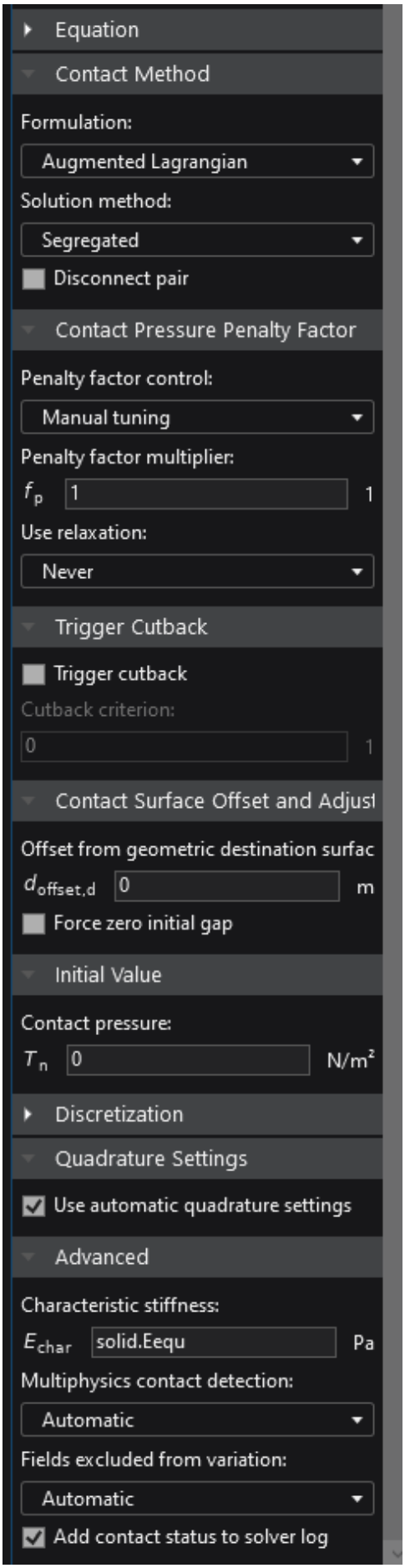
For the rest of PZT ceramics define “piezoelectric material +Z” by right-clicking on “Solid mechanics” → “Material models” → “Piezoelectric material” and keep the coordinate system to default. The main body domains should be included in “linear elastic material”. You should remember that the default features of PLA do not include Young’s modulus, density and Poisson’s ratio. Insert the following settings to complete the material properties:

Property	Variable	Expression
Young's modulus	E	1.36[GPa]
Density	rho	1.25[g/cm^3]
Poisson's ratio	nu	0.3

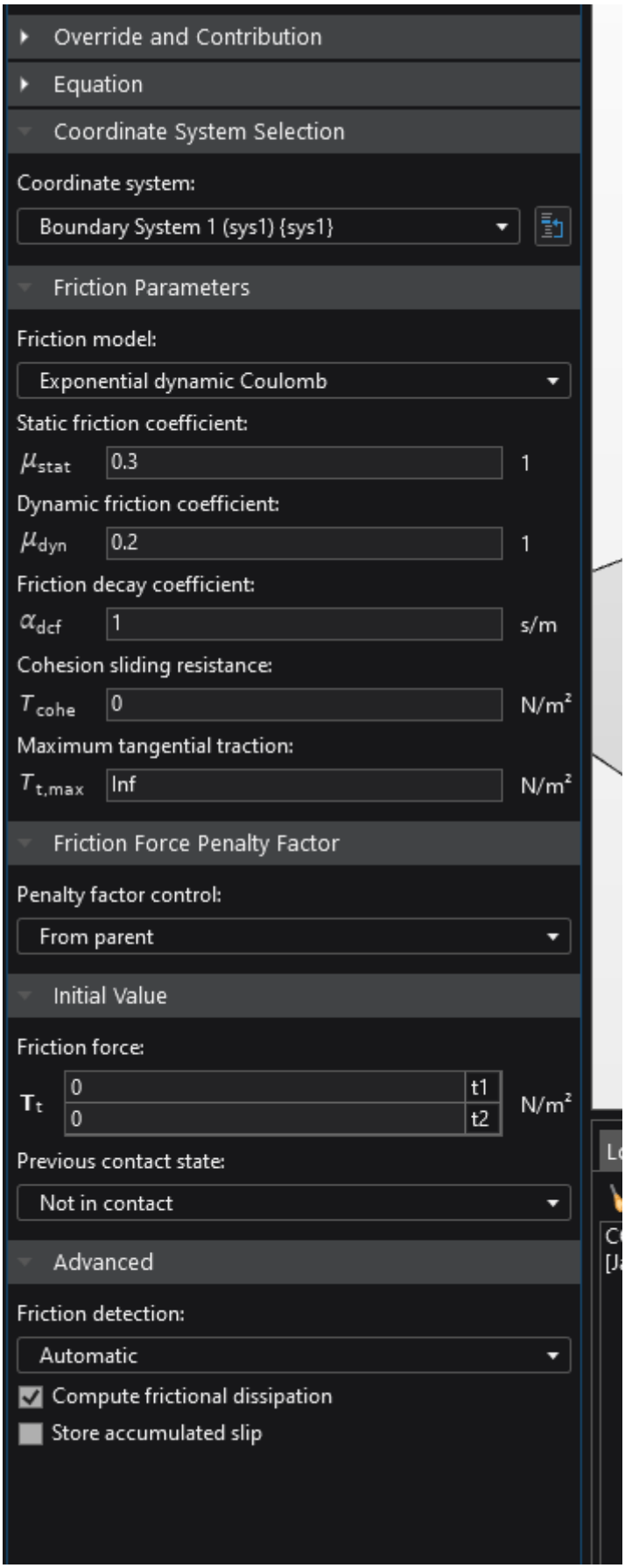
Next, gravity should be activated. Choose all domains for this:



Then, contact settings should be done. For “pairs”, “feet contact” selection group must be selected. As formulation “Augmented Lagrangian” should be chosen. The rest should be done according to the picture below:

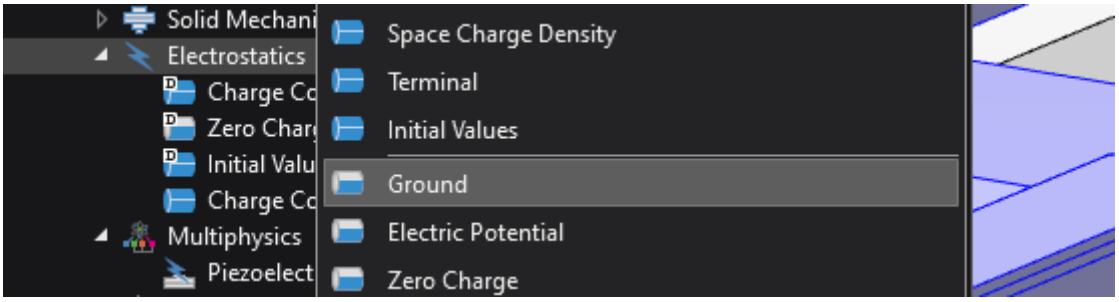

Equation
Contact Method
Formulation:
Augmented Lagrangian
Solution method:
Segregated
Disconnect pair
Contact Pressure Penalty Factor
Penalty factor control:
Manual tuning
Penalty factor multiplier:
f_p 1 1
Use relaxation:
Never
Trigger Cutback
Trigger cutback
Cutback criterion:
0 1
Contact Surface Offset and Adjust
Offset from geometric destination surfac
$d_{offset,d}$ 0 m
Force zero initial gap
Initial Value
Contact pressure:
T_n 0 N/m²
Discretization
Quadrature Settings
Use automatic quadrature settings
Advanced
Characteristic stiffness:
E_{char} solid.Eequ Pa
Multiphysics contact detection:
Automatic
Fields excluded from variation:
Automatic
Add contact status to solver log

Then right click on contact 1 and choose “friction”:



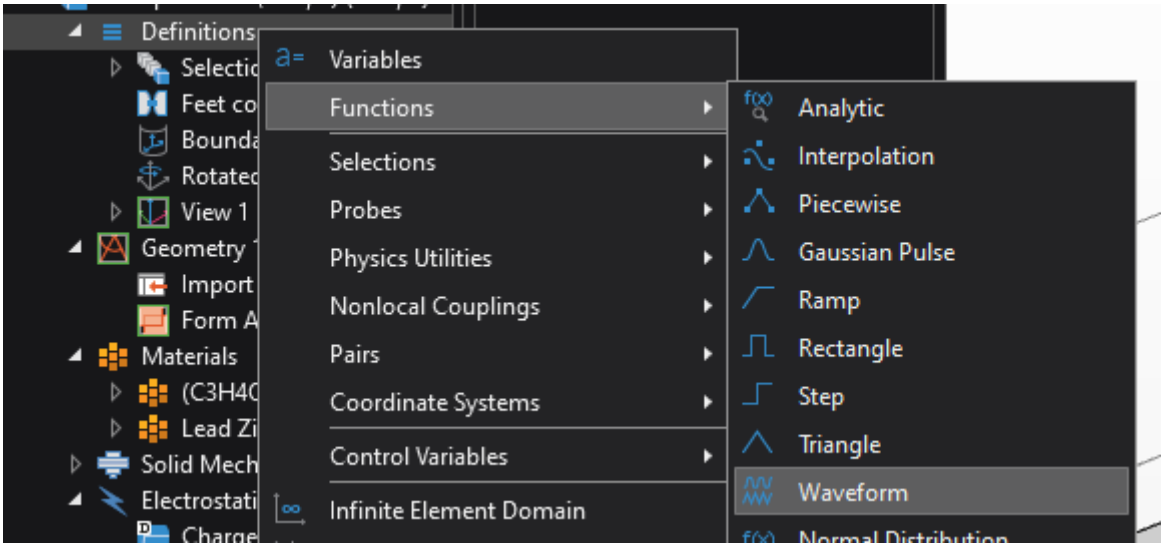
at this point“ solid mechanics” settings are done. Now, it is time to adjust the settings for “electrostatics”. Under “charge conservation piezoelectric 1” choose all the PZT ceramics.

Next right click on “electrostatics” and choose “ground”.

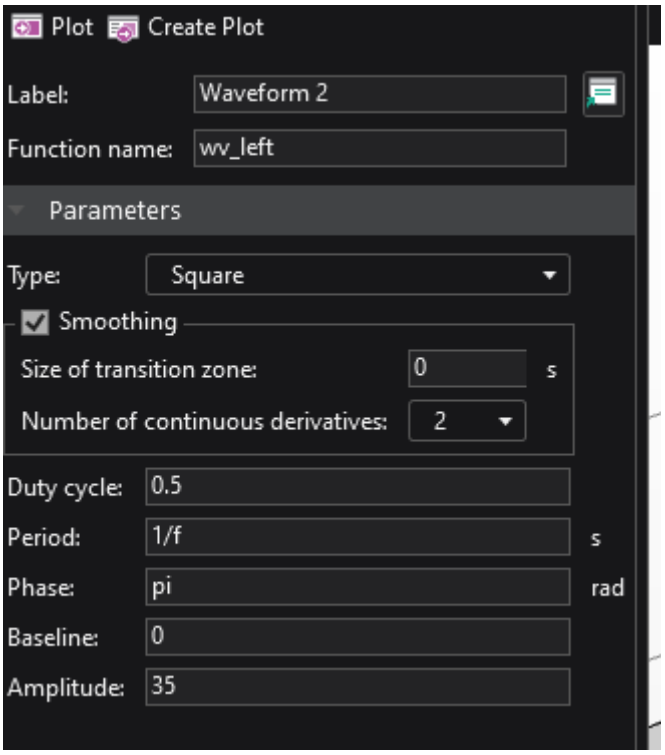


All inner PZT boundaries must be selected for ground potential interfaces. In a smarter and faster way, all these boundaries can be included in one explicit selection.

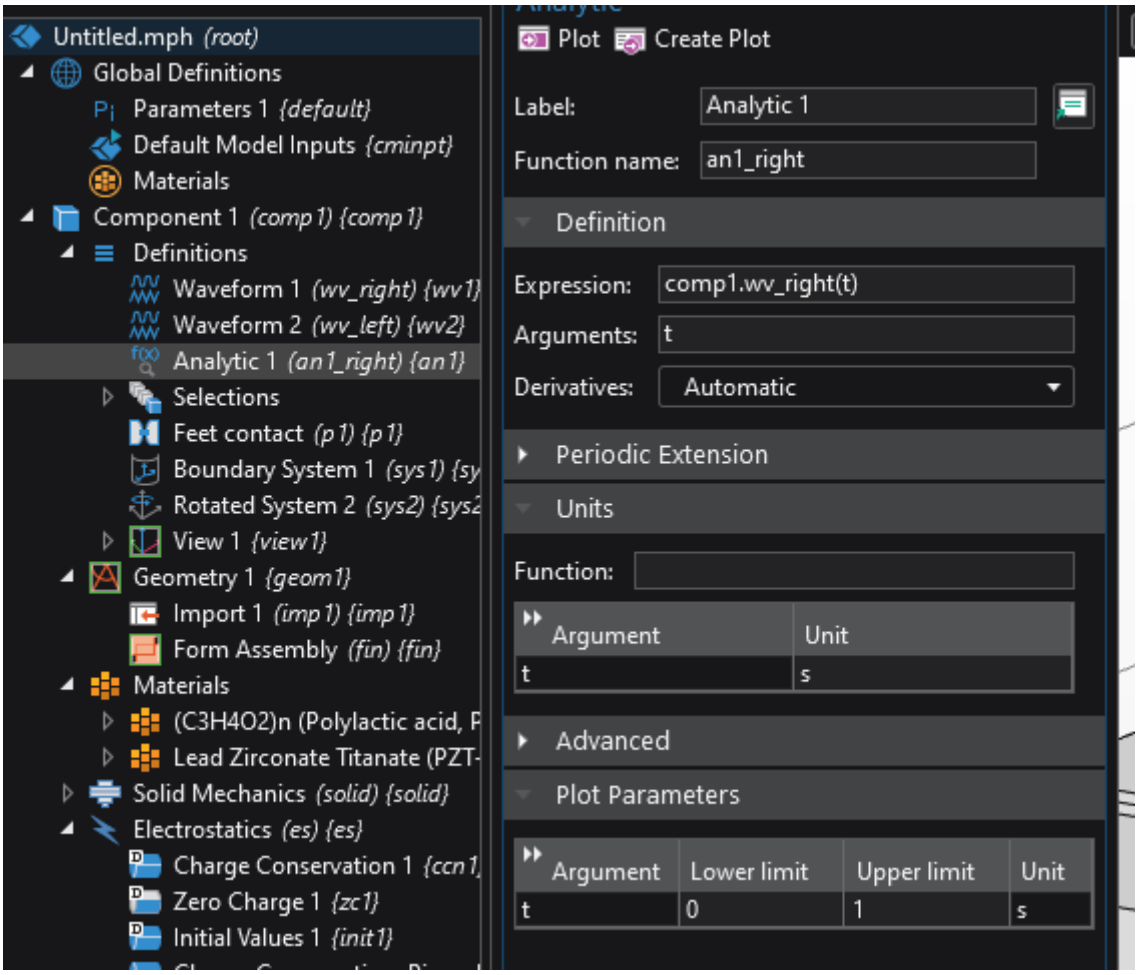
At this point, we need to do the settings for electric potentials. These are the voltage potentials that will be applied to the terminals and will cause the actuation of the PZT ceramics. It should be noted that the legs of this robot are alternatively excited with square wave forms that are 180 degrees out of phase with each other. We need to define these waveforms under the “definitions” tab.



For the first waveform, the function name is “ wv_right”. “Type” is “square”, size of transition zone is 0, period is 1/f and amplitude is 35. This is because we want to excite the legs with voltages that are 35 volts in amplitude. Afterwards, build a second waveform, but this time, with “phase” equal to Pi radians. You can do this by “duplicating” the first waveform.

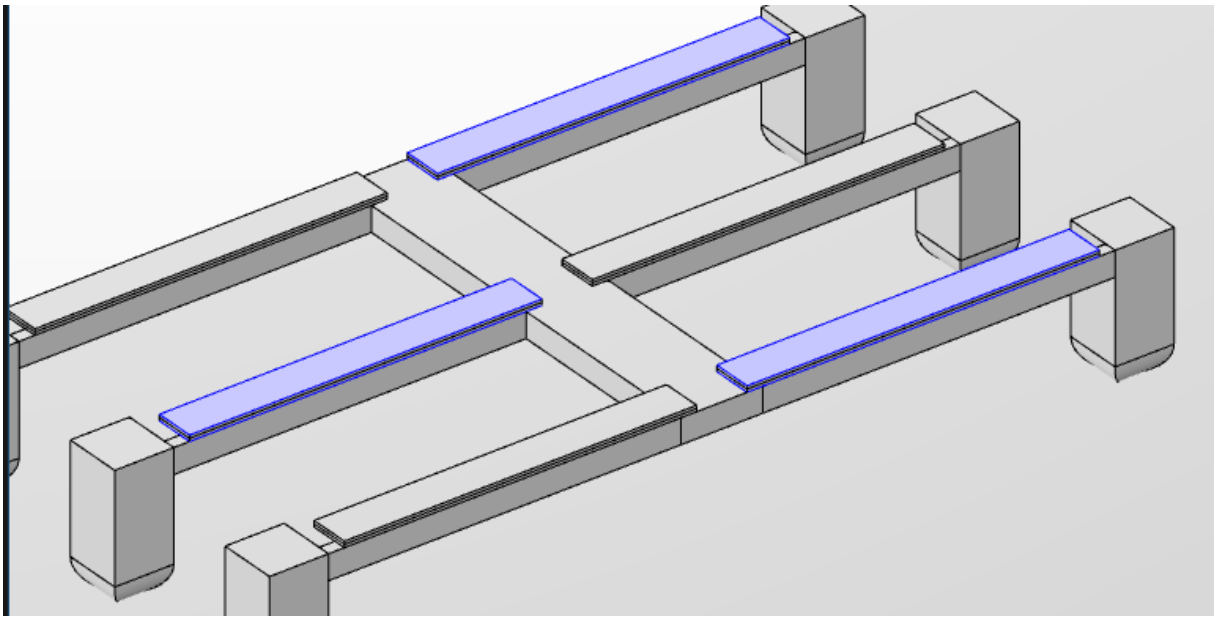


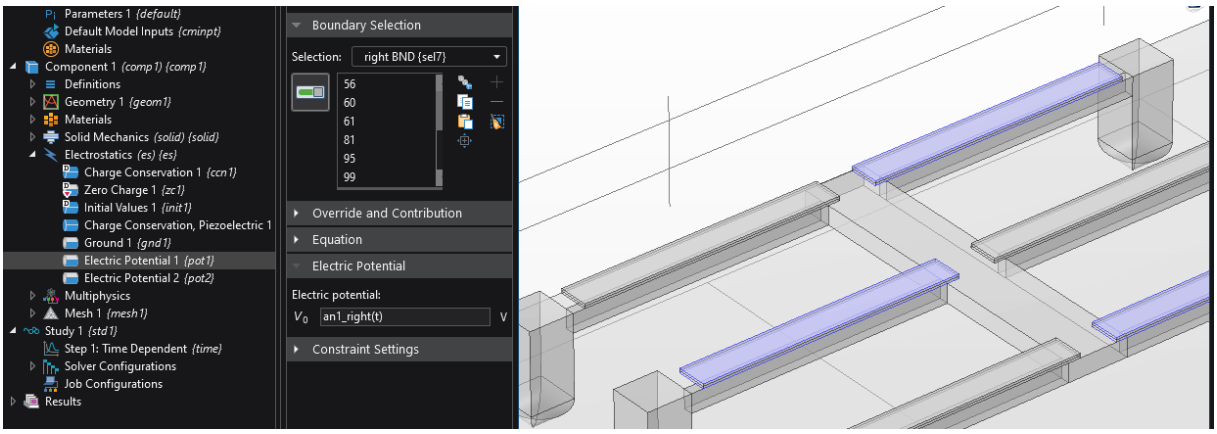
Now we are going to build two analytical functions, in which the abovementioned two waveforms are used. Again, right-click on “definitions”, and under “functions”, select “analytic”. The following settings must be entered. The function name is “an1_right”. As for the “expression”, we are calling “wv_right” with the argument “t” in brackets, as shown below. Remember to change the units into seconds.



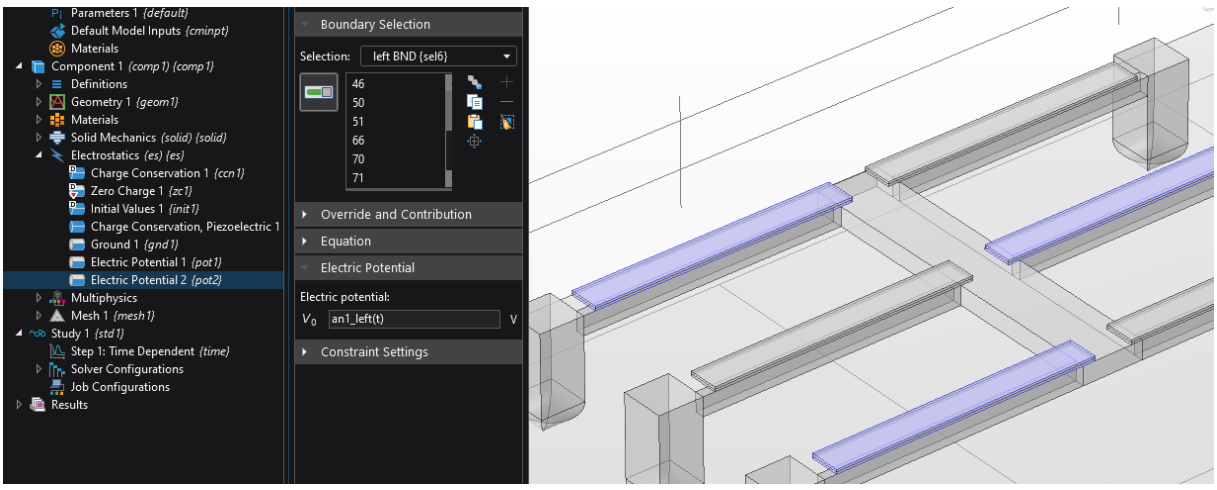
Do the same for the second analytical function. This way we generate 2 square wave forms that are 180 degrees out of phase with respect to each other and have the frequency of 100 Hz. We can now apply these voltages to the alternative legs as shown in picture below. Remember that only the topmost and the bottommost boundaries of each bimorph PZT ceramic should be selected.

Also remember that the bottom most boundaries of the ceramics consist of three or four separate boundary pieces and all of them must be selected. You can make the selection easier by creating an explicit selection of those boundaries once, and later using them quickly. Next, right-click on “electrostatics” and select “electrostatic potential”. As electric potential V0, you should choose the first analytic function.

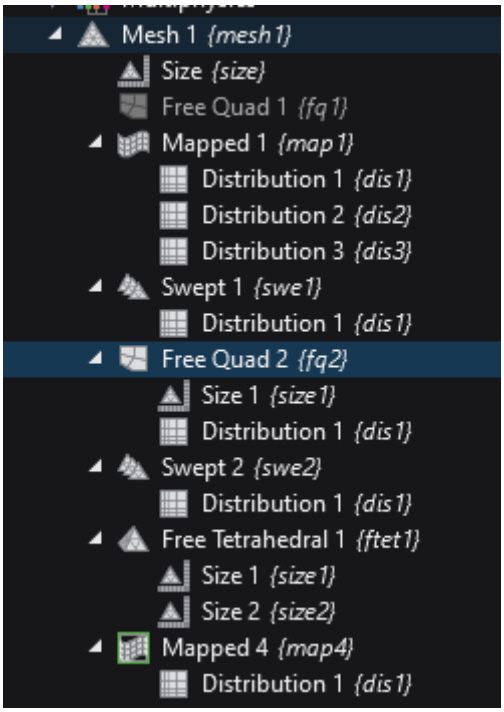




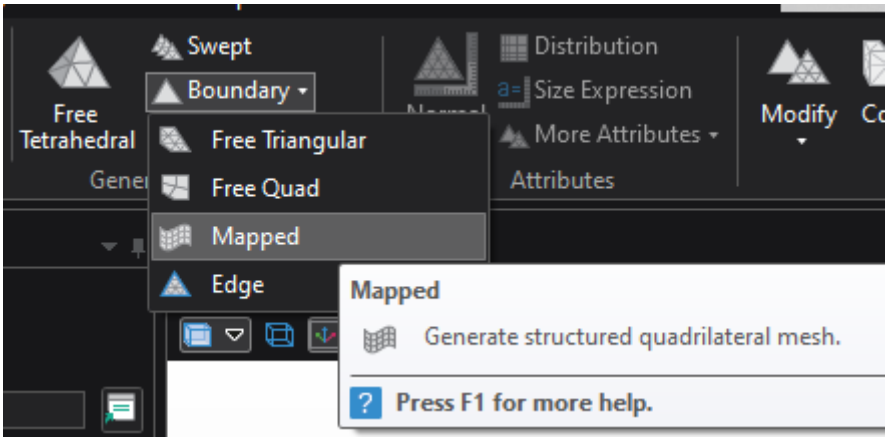
Repeat this step for the other set of electric potentials for the opposite legs.



Next we move to the meshing part. Right click on mesh 1 and select “edit Physics-induced sequence”. You’re going to have the following settings:

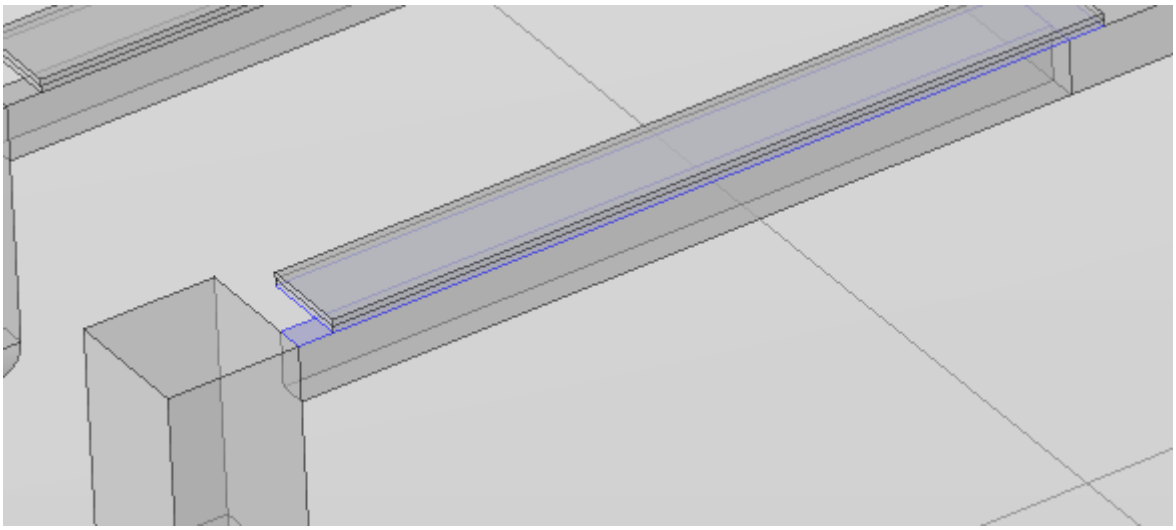


First create a mapped mesh:

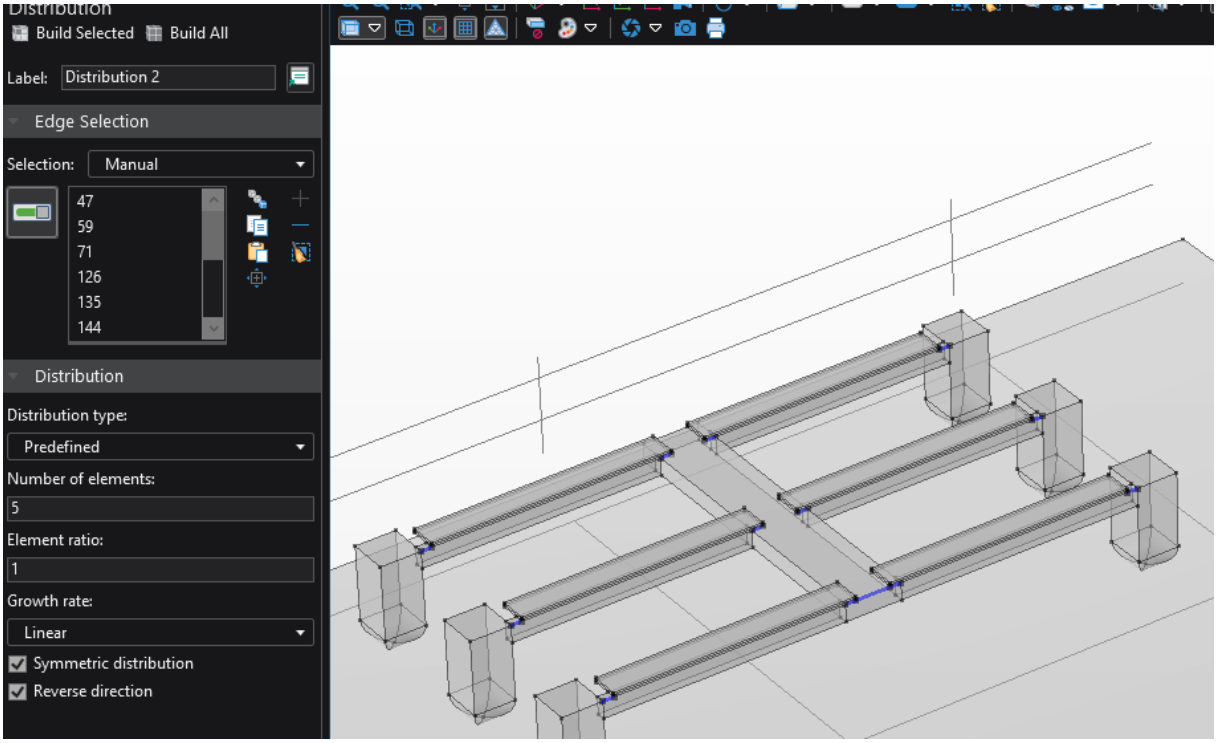


This involves the bottom side of the PZT ceramics and the upper side of the legs. Choose the bottom side of the PZT ceramics for “Boundary selection”.

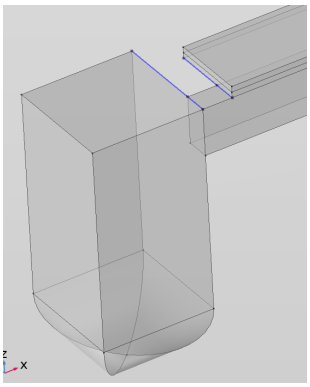
Also, you need to add 3 “distributions” for different edges by right-clicking on “Mapped 1”.



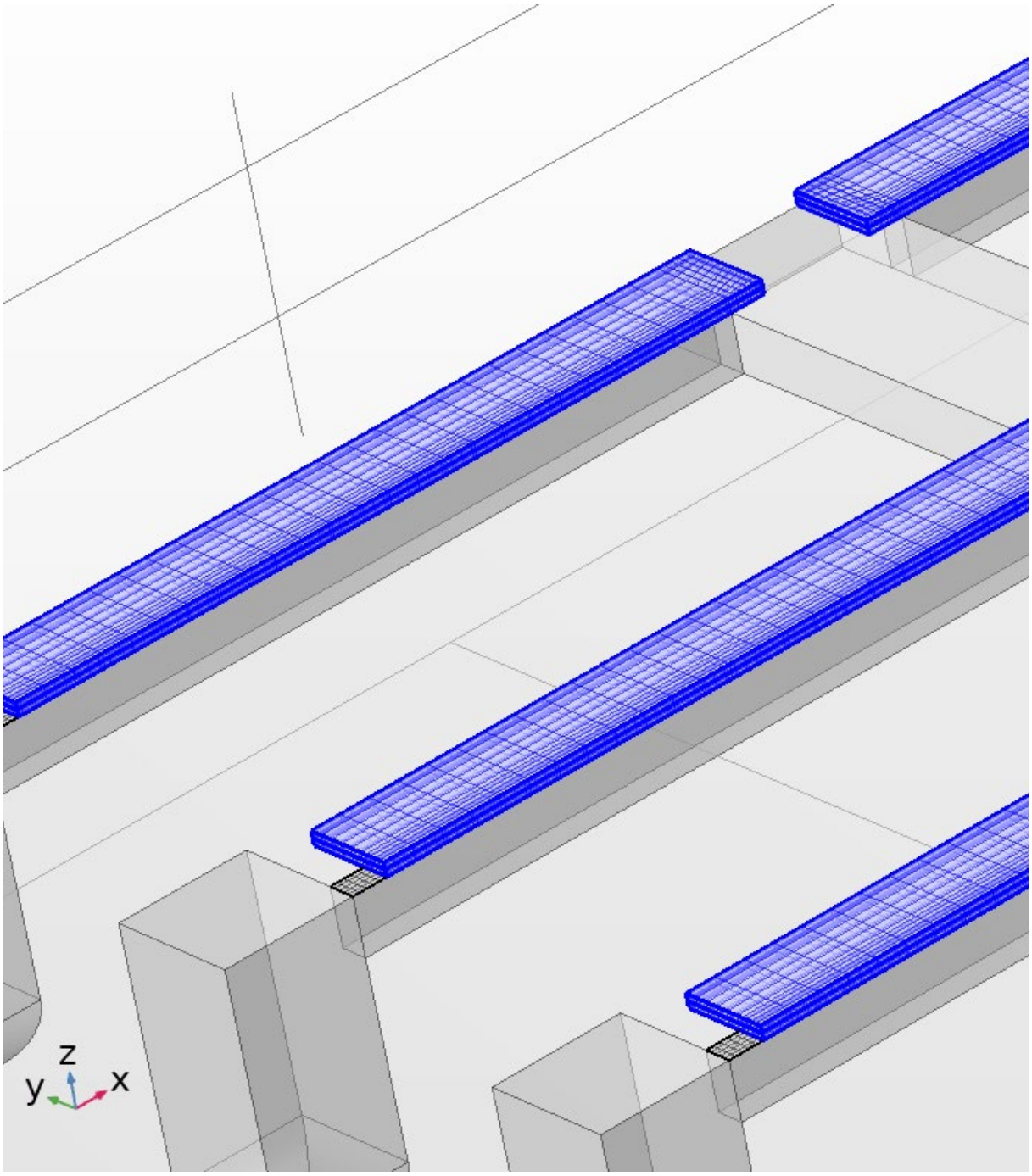
The second distribution is for the following edges:



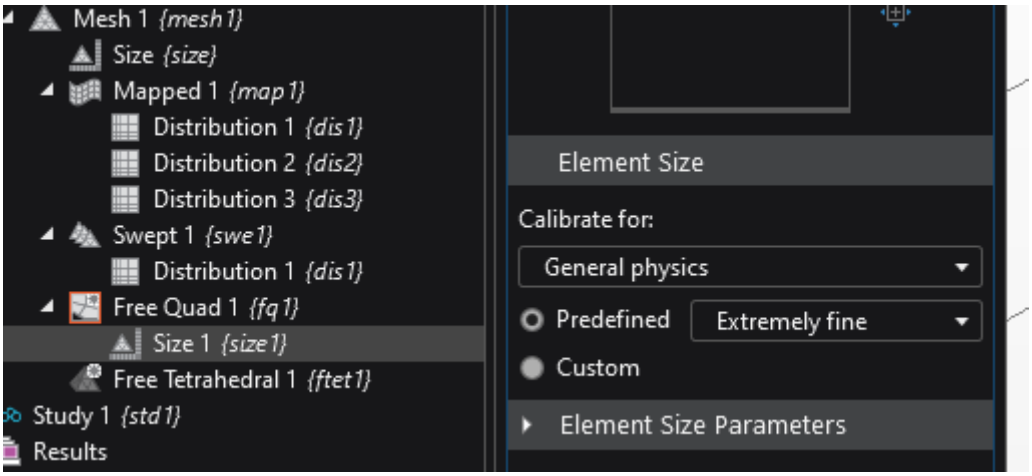
And finally the third distribution in this mapped mesh:

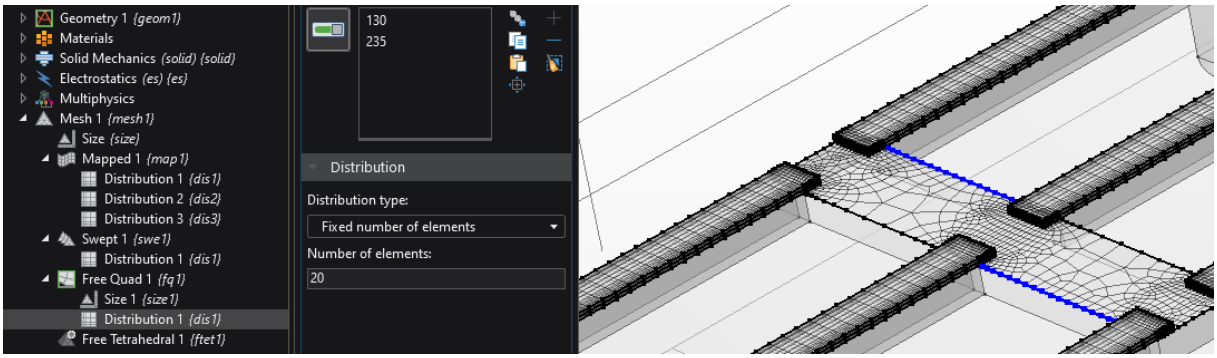


Next using the “swept mesh” the ceramics will be meshed. So far, the mesh should look like the following:

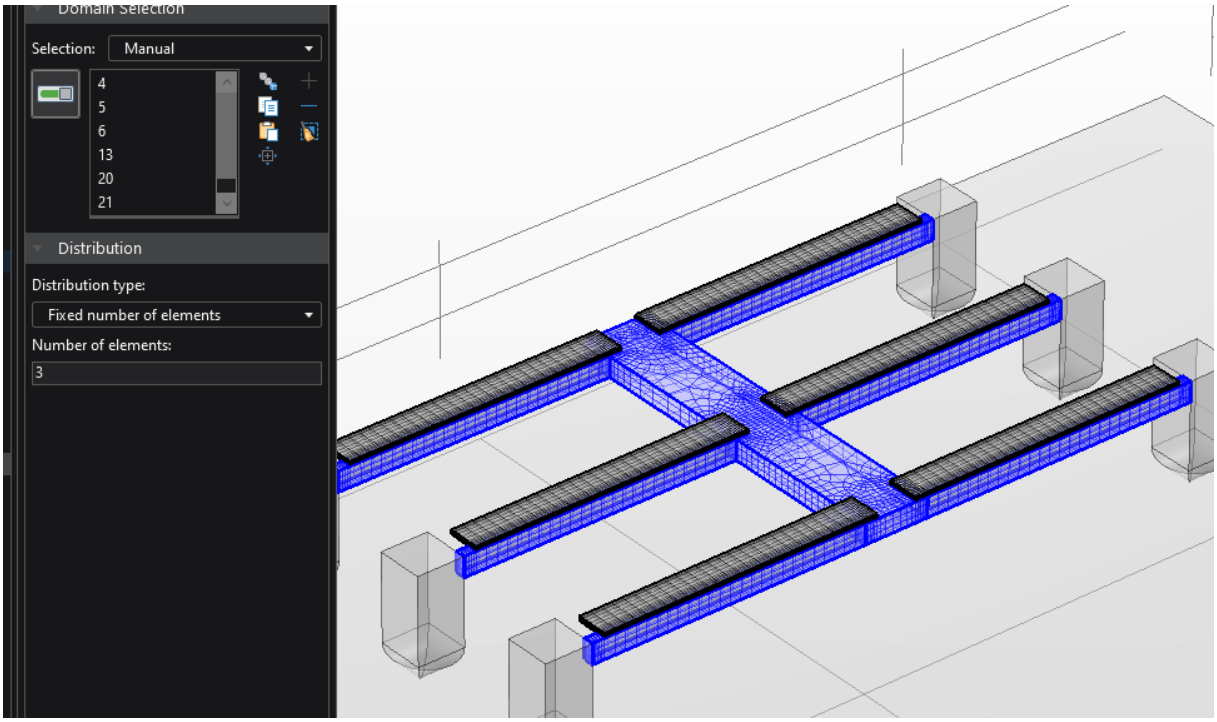


Next we mesh the top surface of the robot’s belly using “free quadrilateral” mesh. Remember to add a size and a distribution sub node, so that the mesh size is fine enough.

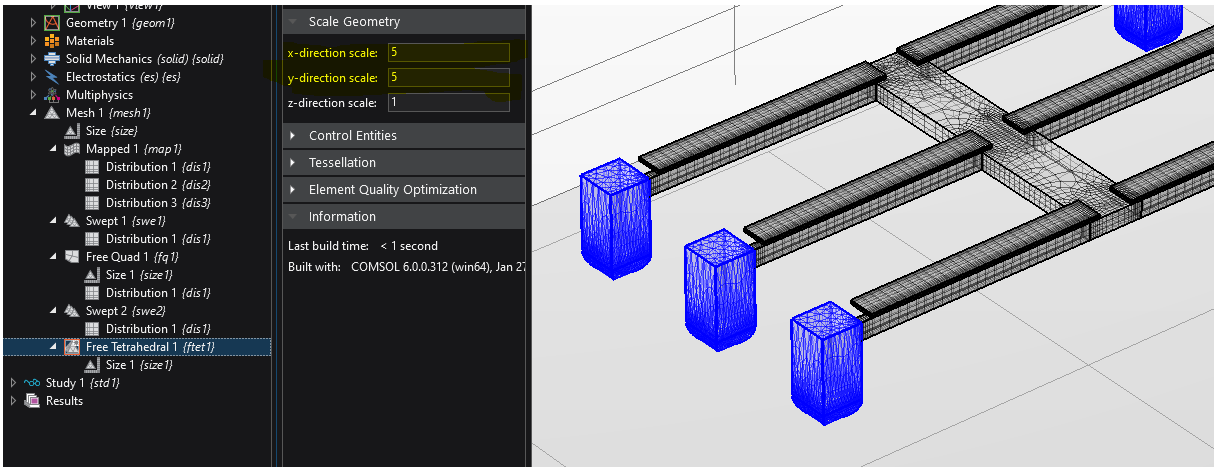


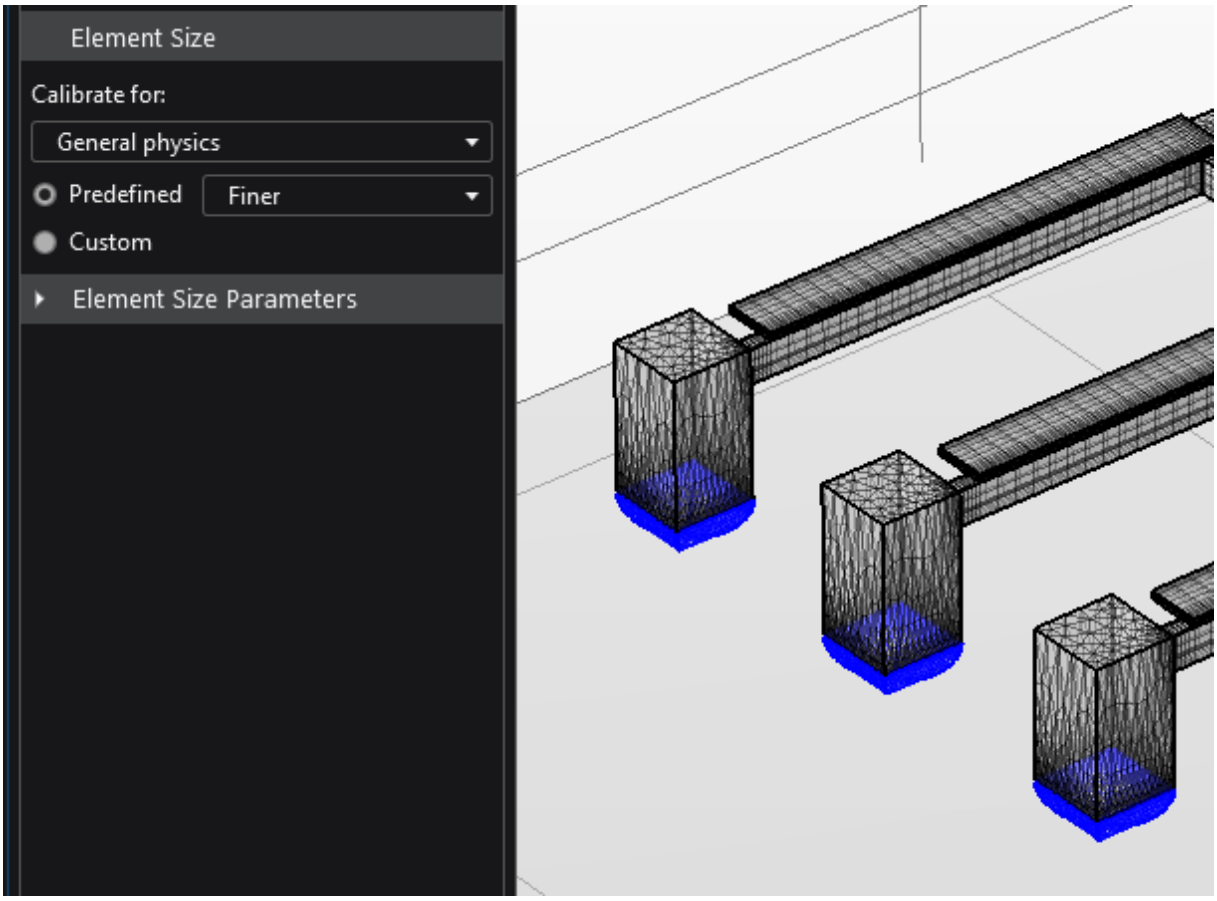


We can now extend the mesh to the entire horizontal part of the robot body, i.e. every part of the polymer except the vertical feet. This can be done by adding a swept node:

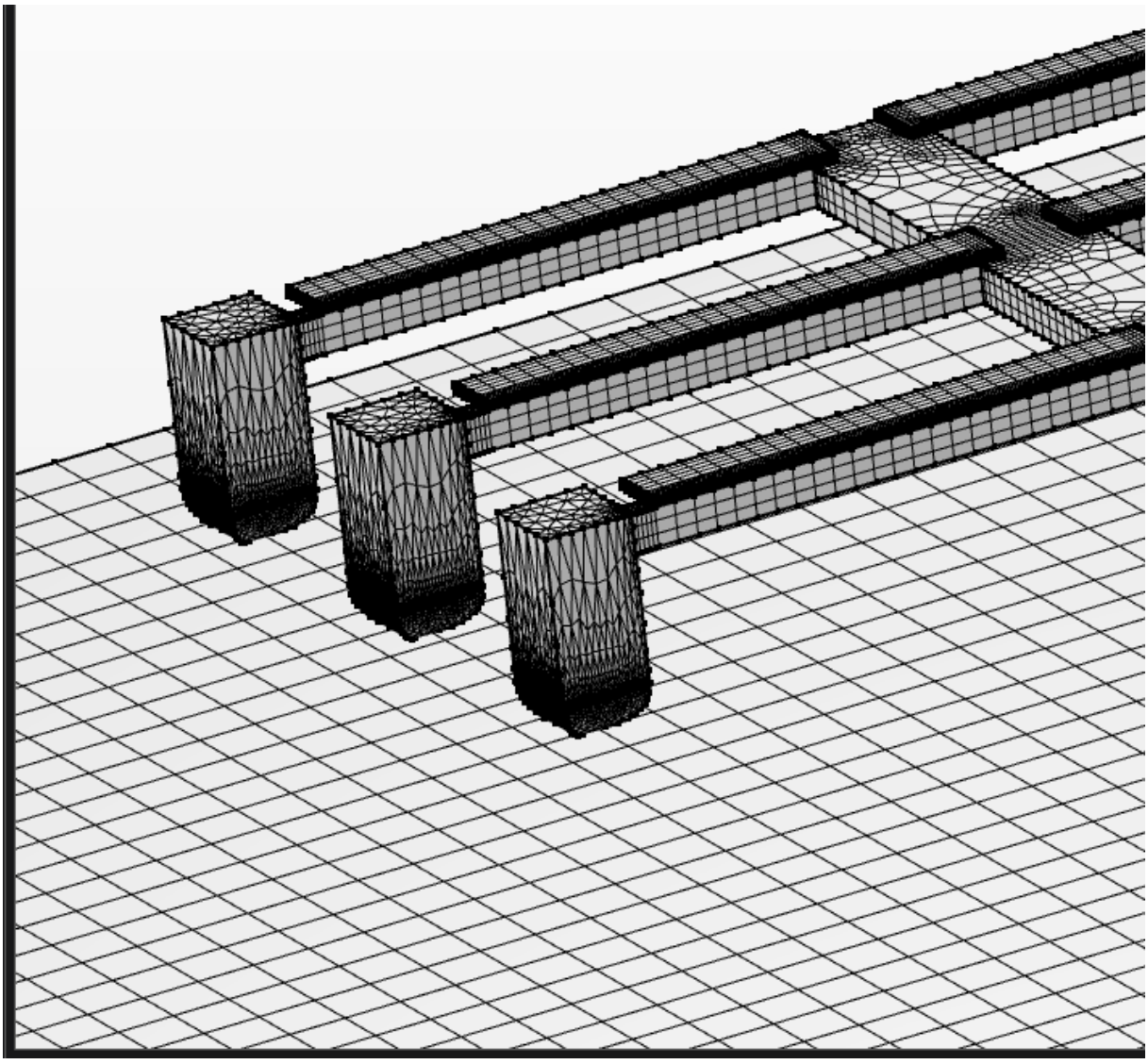


A “Free tetrahedral” with a “size” sub node can be used to mesh the feet. The size sub node is applied to the bottom surface of the feet:



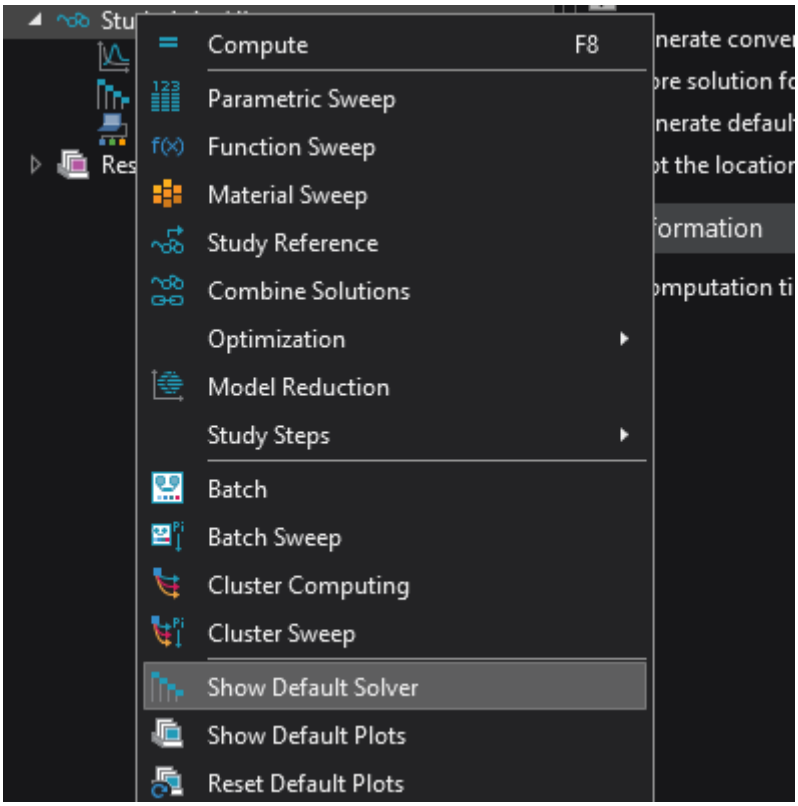


And finally a mapped mesh with a distribution of 50 elements per edge is added for the ground surface.

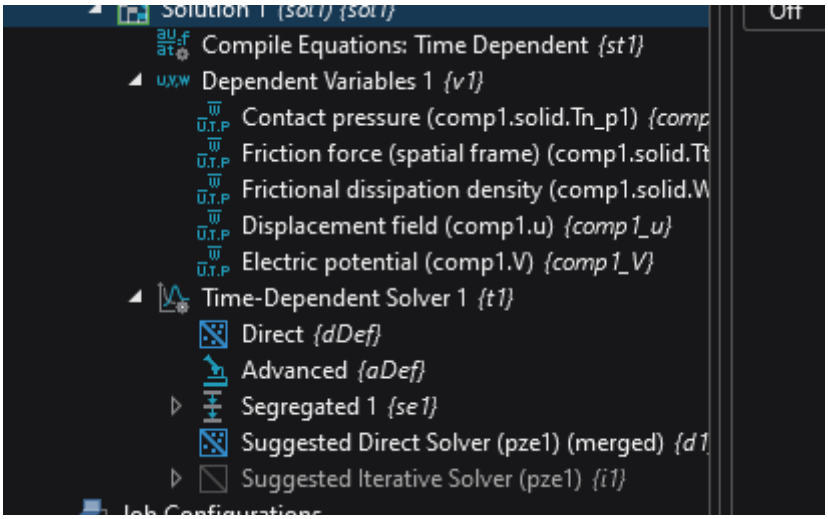


Now it’s time to take care of the study settings for the time dependent study. When simulating contacts with friction, the study settings are very critical. They basically determine whether or not the simulation will converge and whether or not the results are trustworthy. In the first part insert “range(0,0.001,0.2)” for output times.

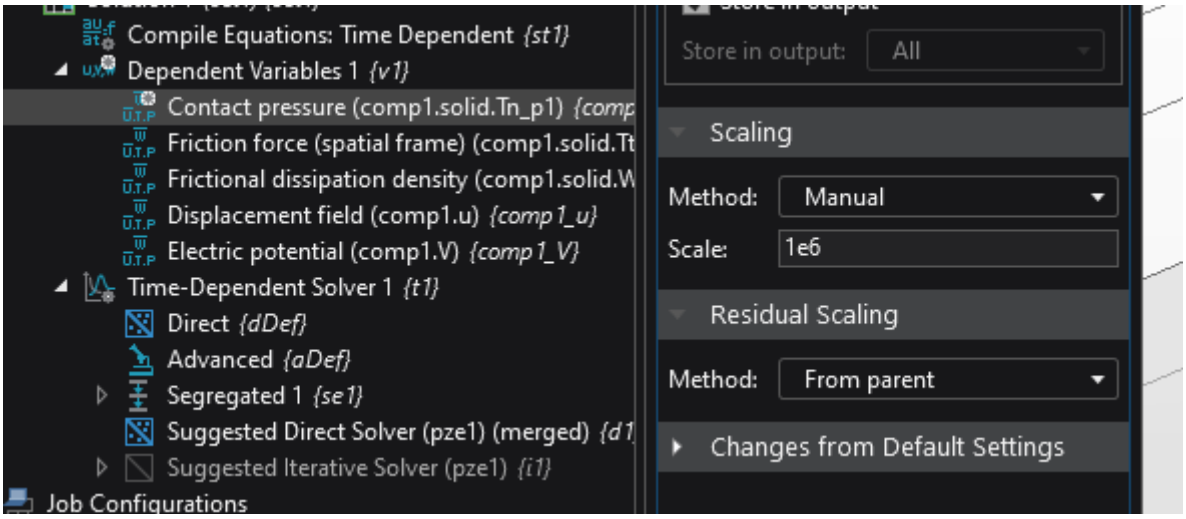
Then right click on “study 1” and select “show default solver”.

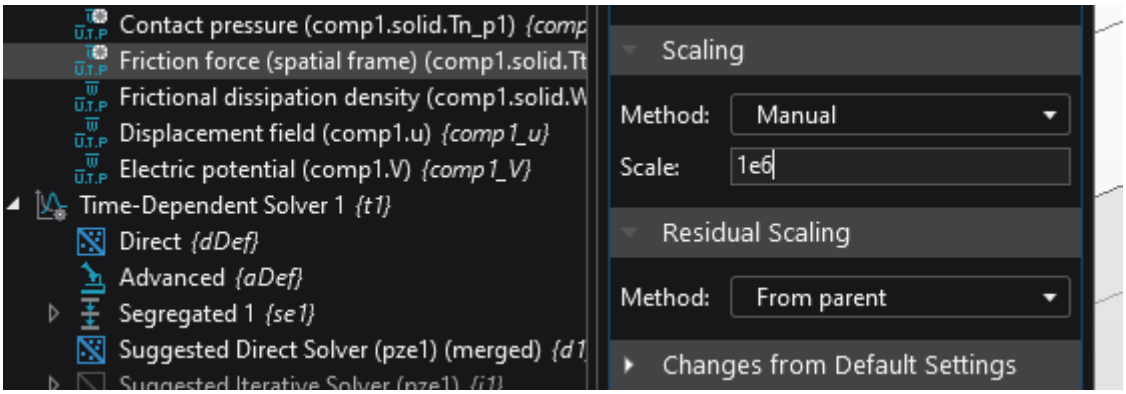


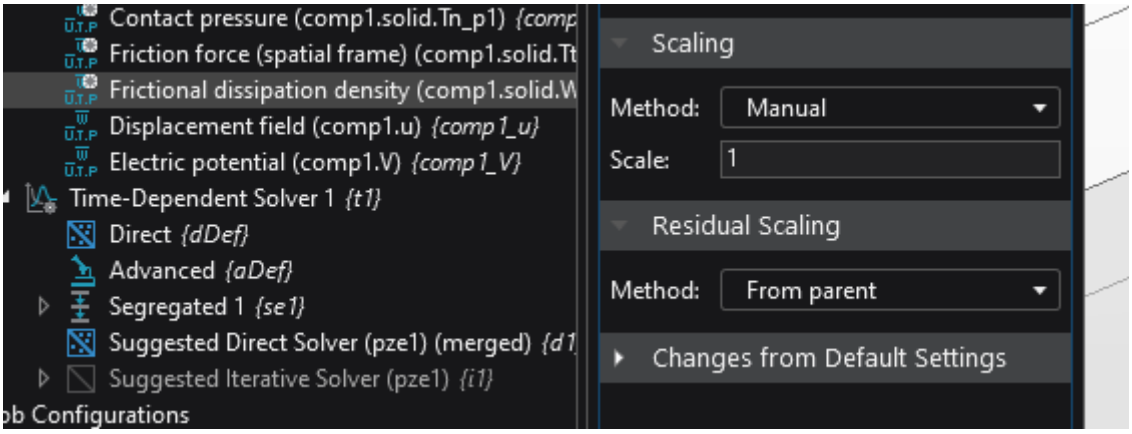
Under “solution 1” there are two main sections that need to be manipulated until convergence is achieved. These sections are the “dependent variables” and “time- dependent solver”.

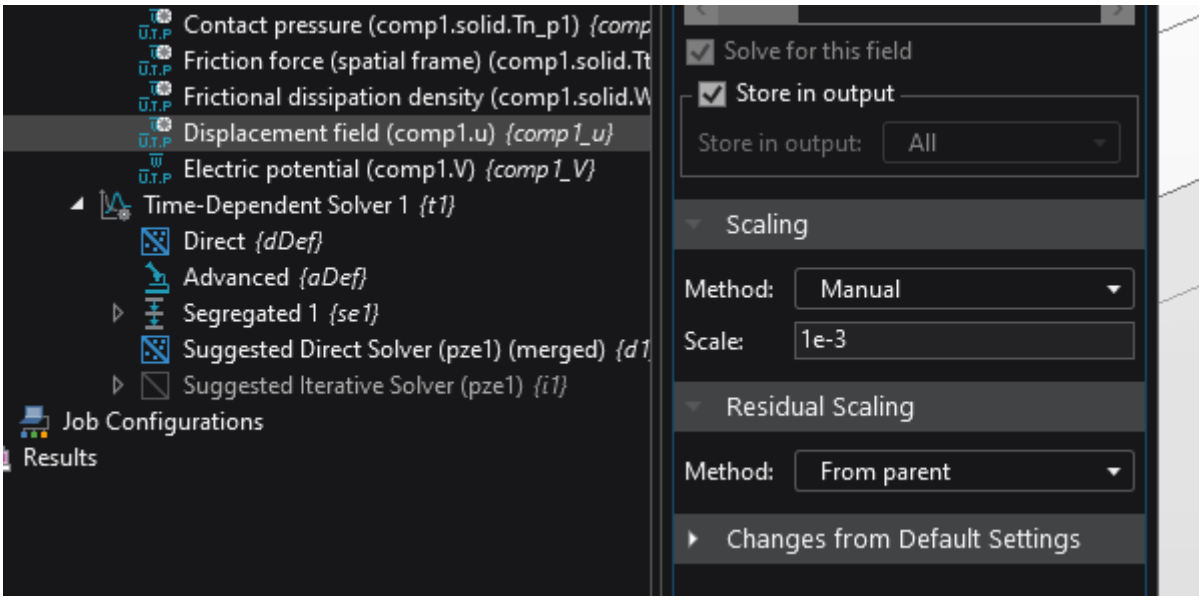


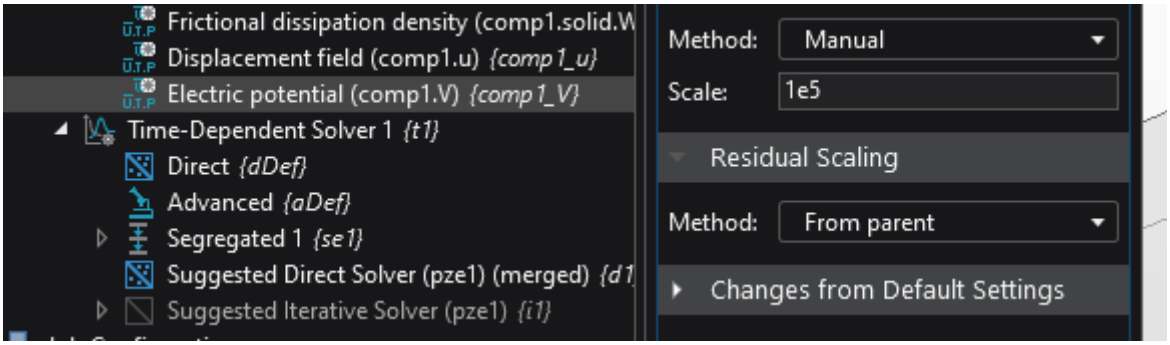
When using the “Augmented Lagrangian” formulation, it is necessary to manually scale the variables in a contact problem. If you cannot estimate the contact pressure before the solution, you may need to do the analysis in two passes, where you first compute an estimate of the contact pressure using the Penalty formulation. The scaling of the contact pressure is used when checking the convergence, so if a too high value is used, there is a risk that the results are not correct. https://www.comsol.com/support/knowledgebase/1102


Compile Equations: Time Dependent {st1}
Dependent Variables 1 {v1}
Contact pressure (comp1.solid.Tn_p1) {comp
Friction force (spatial frame) (comp1.solid.Tt
Frictional dissipation density (comp1.solid.W
Displacement field (comp1.u) {comp1_u}
Electric potential (comp1.V) {comp1_V}
Time-Dependent Solver 1 {t1}
Direct {dDef}
Advanced {aDef}
Segregated 1 {se1}
Suggested Direct Solver (pze1) (merged) {d1
Suggested Iterative Solver (pze1) {i1}
Job Configurations
Store in output: All
Scaling
Method: Manual
Scale: 1e6
Residual Scaling
Method: From parent
Changes from Default Settings

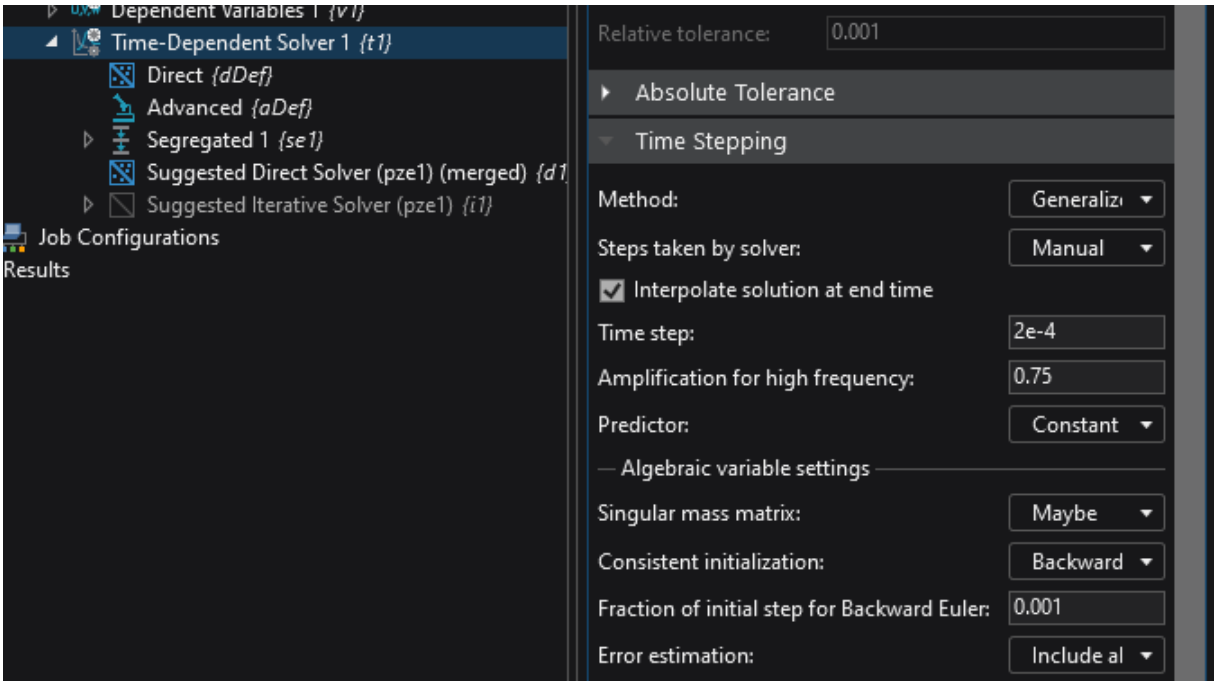

Contact pressure (comp1.solid.Tn_p1) {comp
Friction force (spatial frame) (comp1.solid.Tt
Frictional dissipation density (comp1.solid.W
Displacement field (comp1.u) {comp1_u}
Electric potential (comp1.V) {comp1_V}
Time-Dependent Solver 1 {t1}
Direct {dDef}
Advanced {aDef}
Segregated 1 {se1}
Suggested Direct Solver (pze1) (merged) {d1
Scaling
Method: Manual
Scale: 1e6
Residual Scaling
Method: From parent
Changes from Default Settings


Contact pressure (comp1.solid.Tn_p1) {comp
Friction force (spatial frame) (comp1.solid.Tt
Frictional dissipation density (comp1.solid.W
Displacement field (comp1.u) {comp1_u}
Electric potential (comp1.V) {comp1_V}
Time-Dependent Solver 1 {t1}
Direct {dDef}
Advanced {aDef}
Segregated 1 {se1}
Suggested Direct Solver (pze1) (merged) {d1
Suggested Iterative Solver (pze1) {i1}
ob Configurations
Scaling
Method: Manual
Scale: 1
Residual Scaling
Method: From parent
Changes from Default Settings

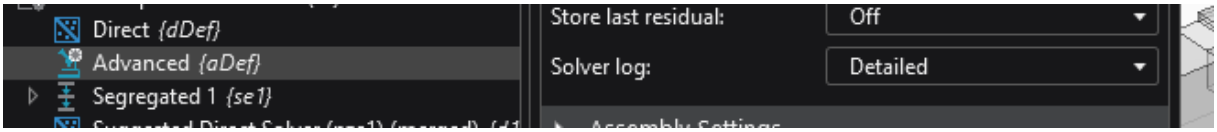




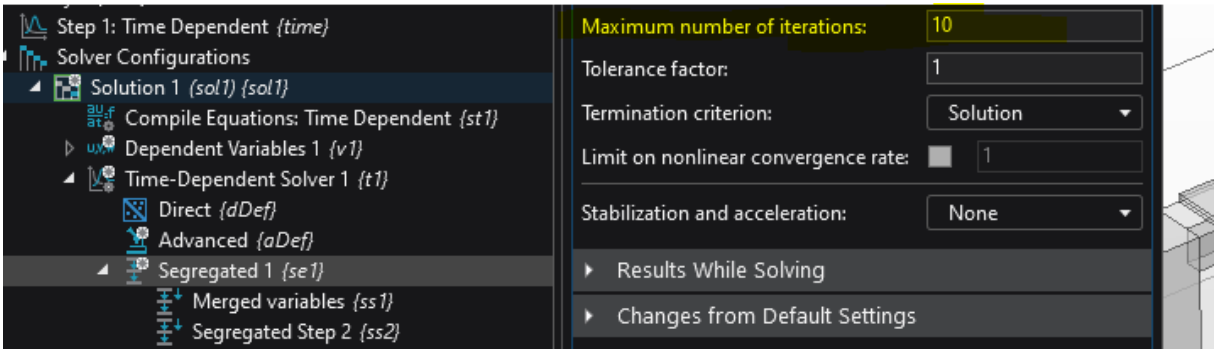
If you keep time stepping to the default value, the simulation may never converge, as it will probably choose a very small time step. Therefore, you need to change the “steps taken by the solver” into “manual” and then select a different time step. This time step depends on the signal and also the time resolution degree of the physical phenomenon that is expected by the user.



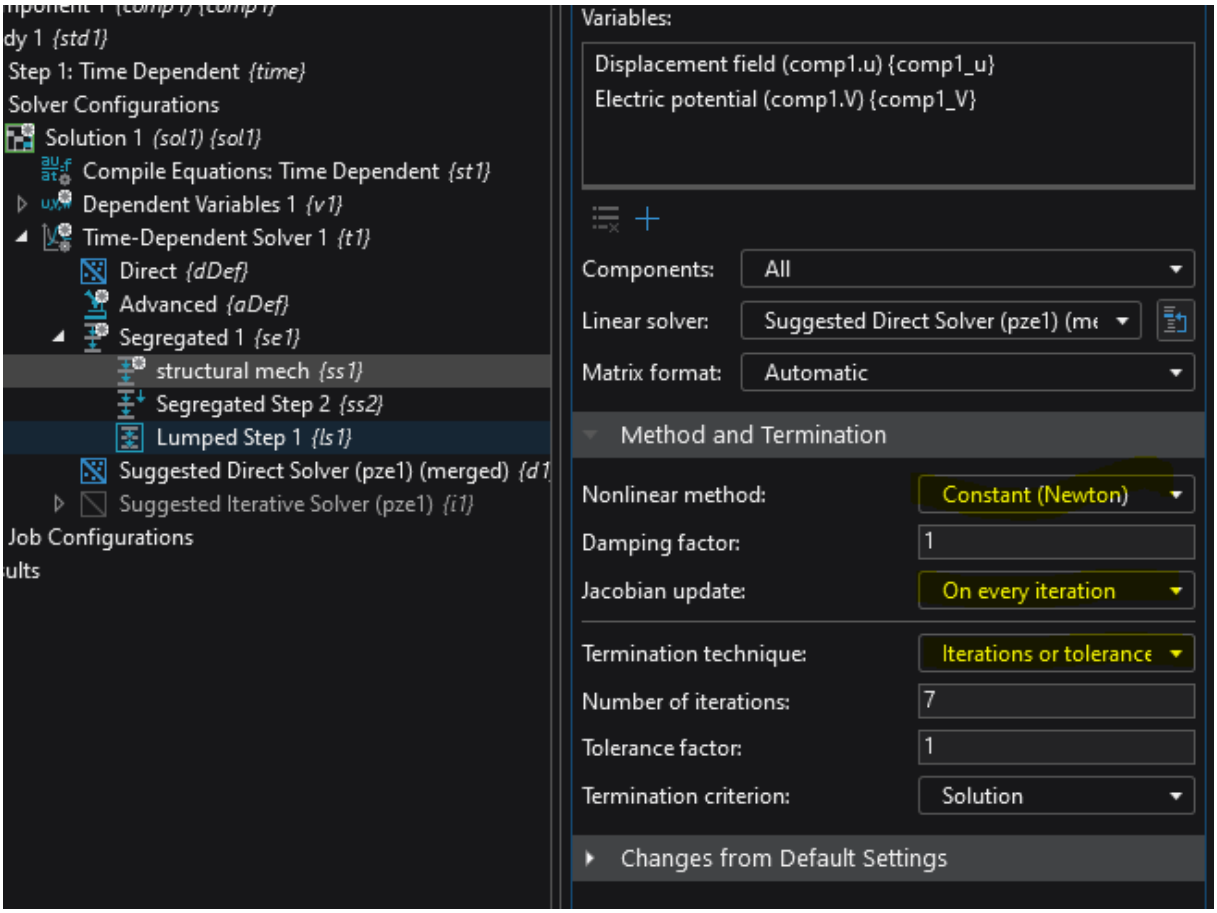
One more tip: if you change “solver log” to “detailed”, while solving the simulation, more details about the status of contacts will be shown in messages.



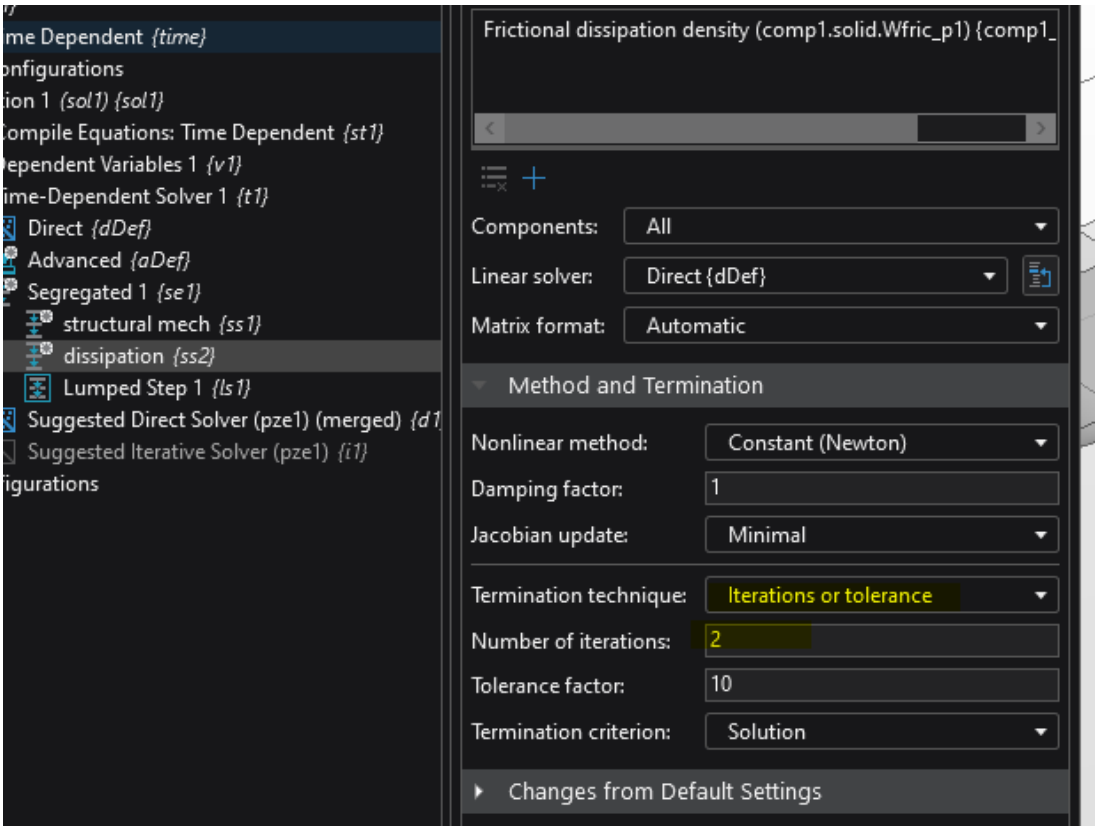
Now it is time for some details about the segregated variables. These settings as follows, are only for this simulation and are attained based on trial and error.



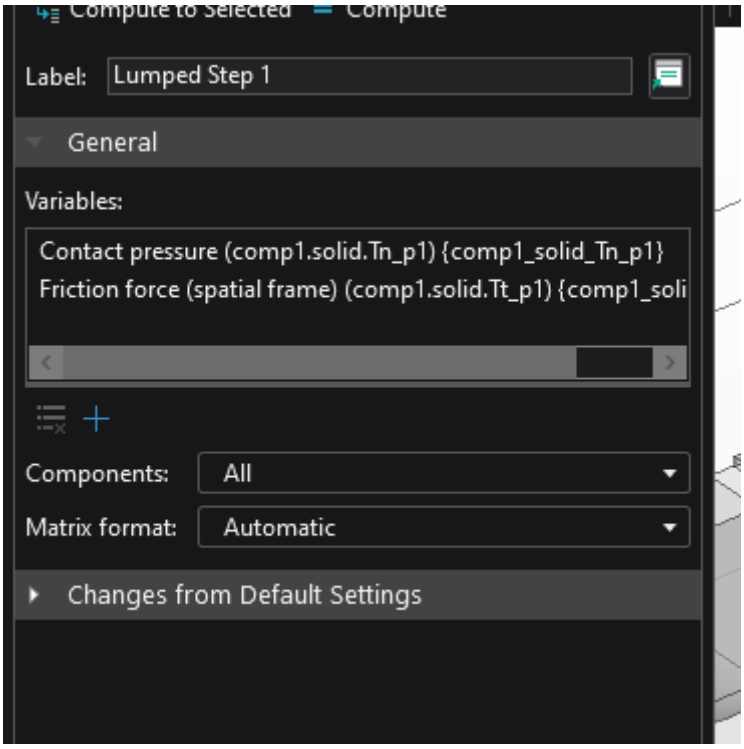
By default “displacement field and electric potential” are placed in one group as “merged variables”. We rename them as “structural mech”.



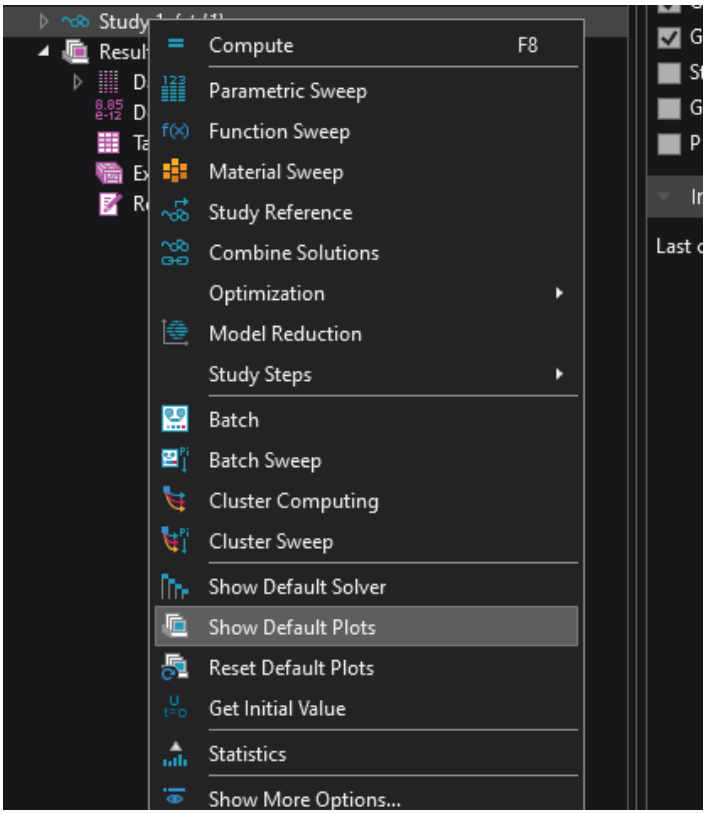
“frictional dissipation density” is in a separate segregated step and we rename it to “dissipation”.



And finally “contact pressure and friction force” are placed into a “lump step”. We do not change anything in the settings of this lump step.

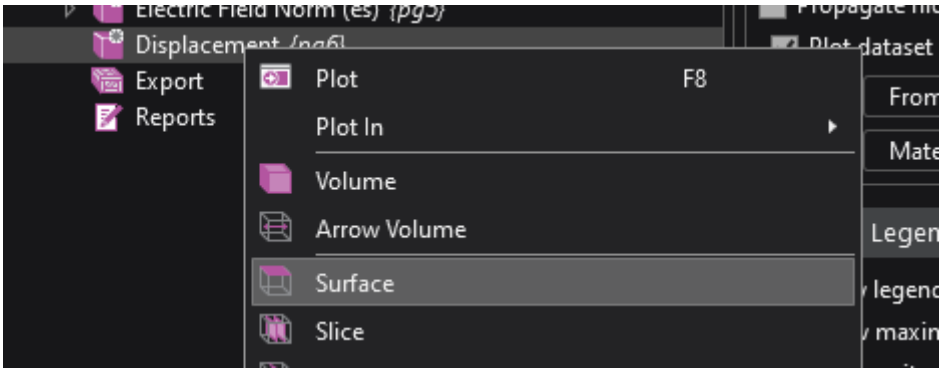


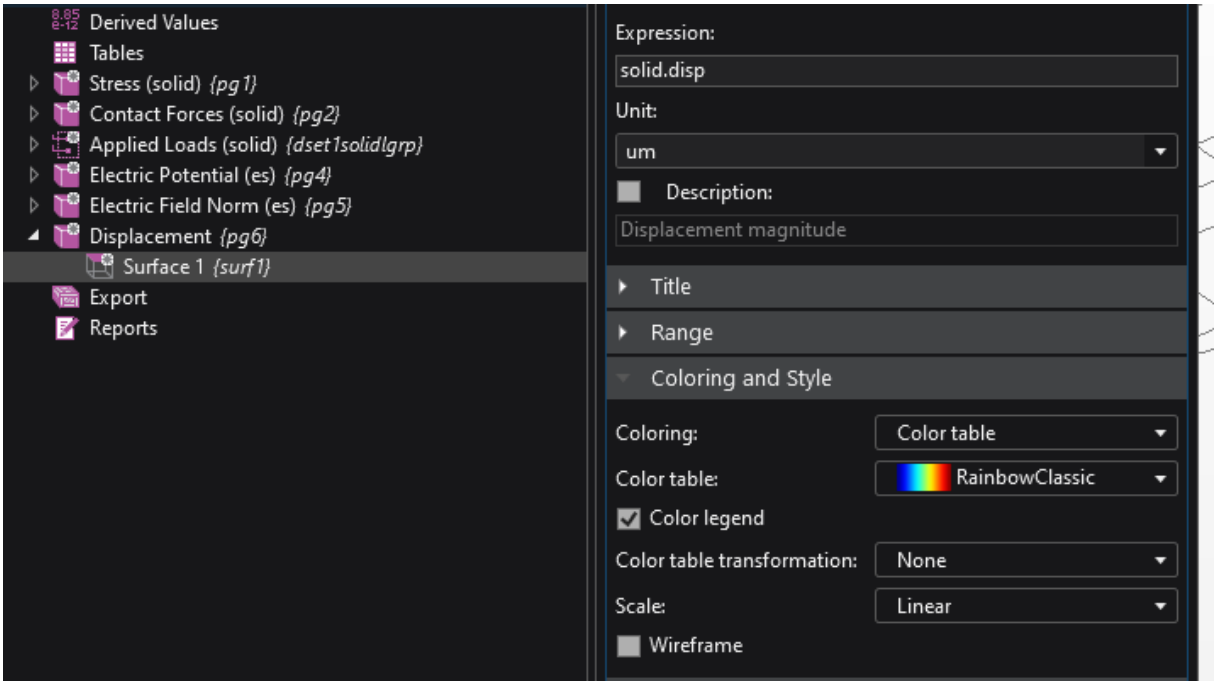
You can also choose to watch the results while solving. These results can be, for example, the displacement of the robot during the simulation. To do this, first you need to show some default plots and then make some adjustments to them.



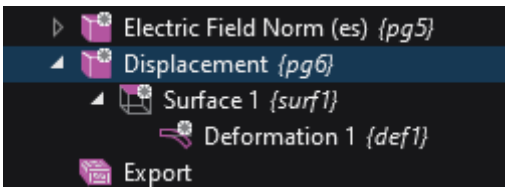
One of the default plots is the “stress” in the structure. But I myself, am more interested in observing the displacement of the structure as time passes. This can be done by generating a new result:

Right-click on results and make a 3D plot group. Rename it as “displacement”. Then right-click on this result and select “surface”.

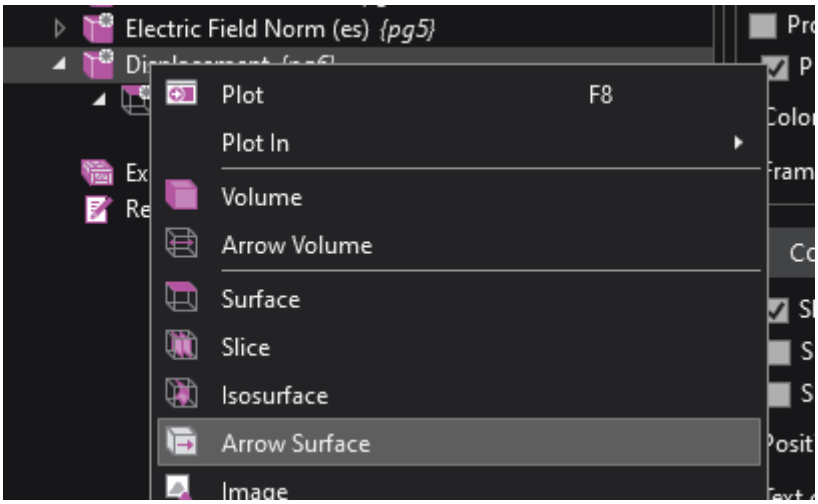




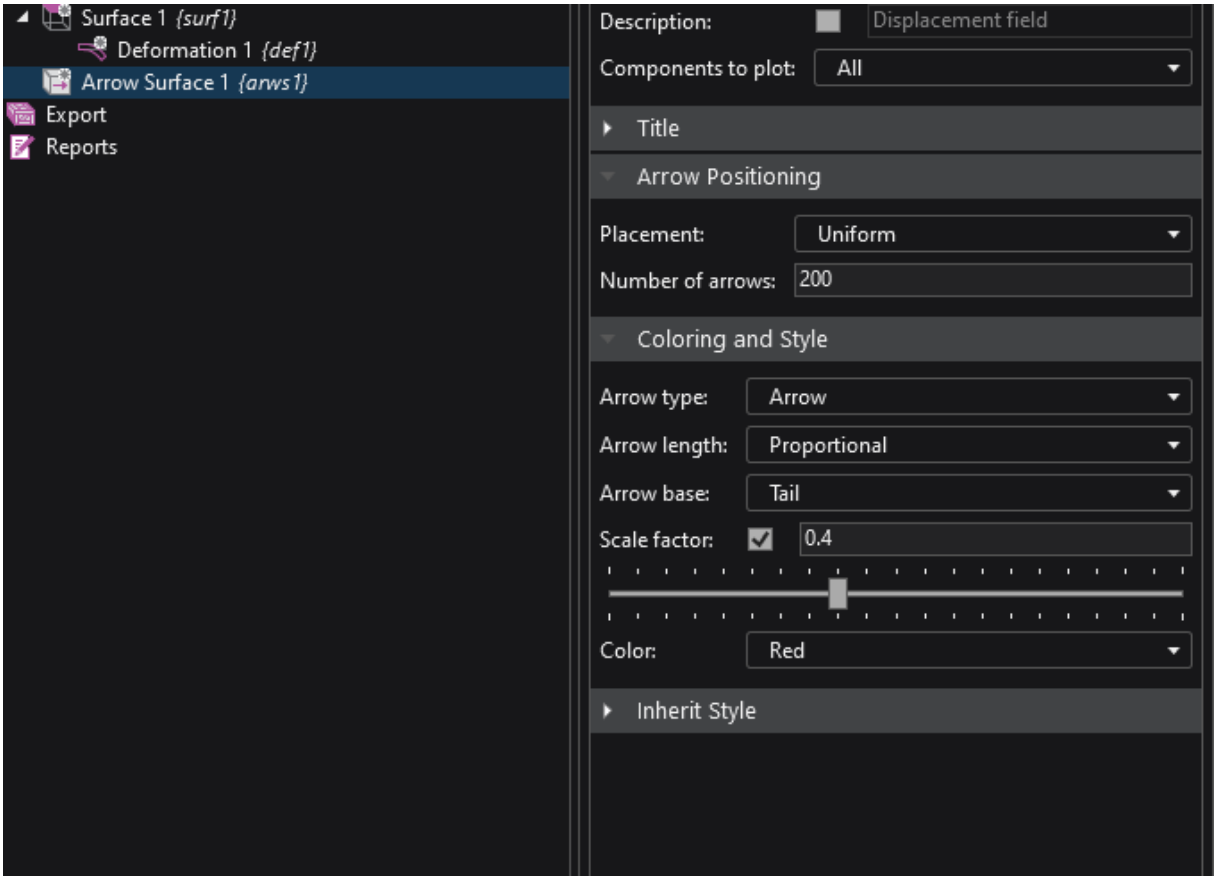
If you are also interested in seeing how the structure is “deforming”, then you need to activate deformation. For that, right-click on surface 1 and choose deformation.



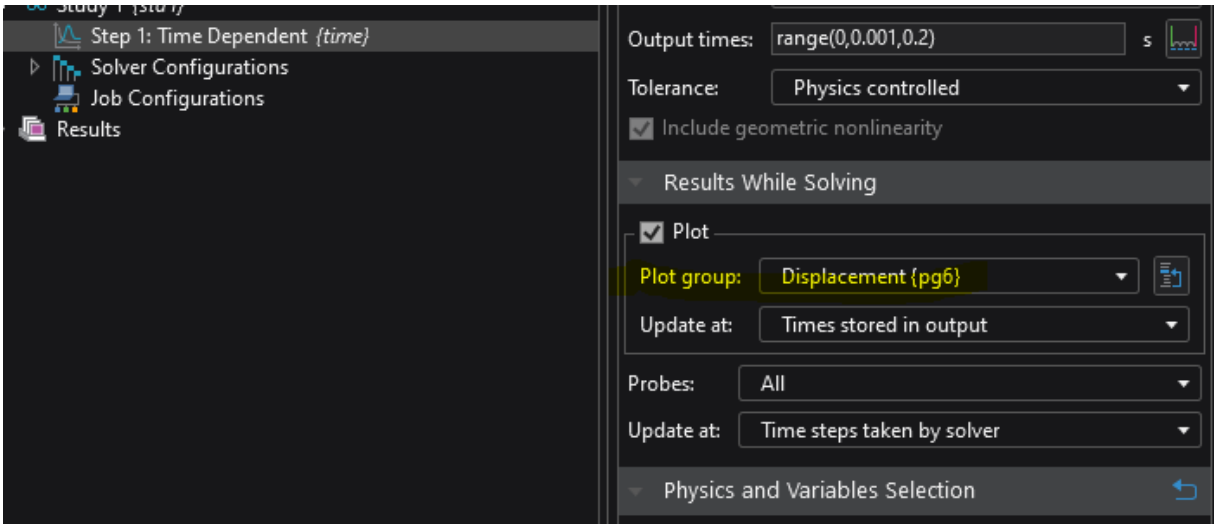
I am also interested in seeing the direction of the movements everywhere in the structure. Therefore, I would like to have an “arrow surface” added to the same plot.



The only settings that needs to be changed here, is “scale factor”. Reset that to 0.4.



Now go back to “study 1” → “step 1”, and change “results while solving”. Choose the desired plot group to be plotted, while the simulation is running.



This model is now ready to be simulated.

Bibliography

[1] K. Patel, J. Qu, and K. R. Oldham, “Tilted Leg Design for a Rapid-Prototyped Low-Voltage Piezoelectric Running Robot; Tilted Leg Design for a Rapid-Prototyped Low-Voltage Piezoelectric Running Robot,” 2018.

[2] J. Qu, C. B. Teeple, and K. R. Oldham, “Modeling Legged Microrobot Locomotion Based on Contact Dynamics and Vibration in Multiple Modes and Axes,” *Journal of Vibration and Acoustics, Transactions of the ASME*, vol. 139, no. 3, Jun. 2017, doi: 10.1115/1.4035959.