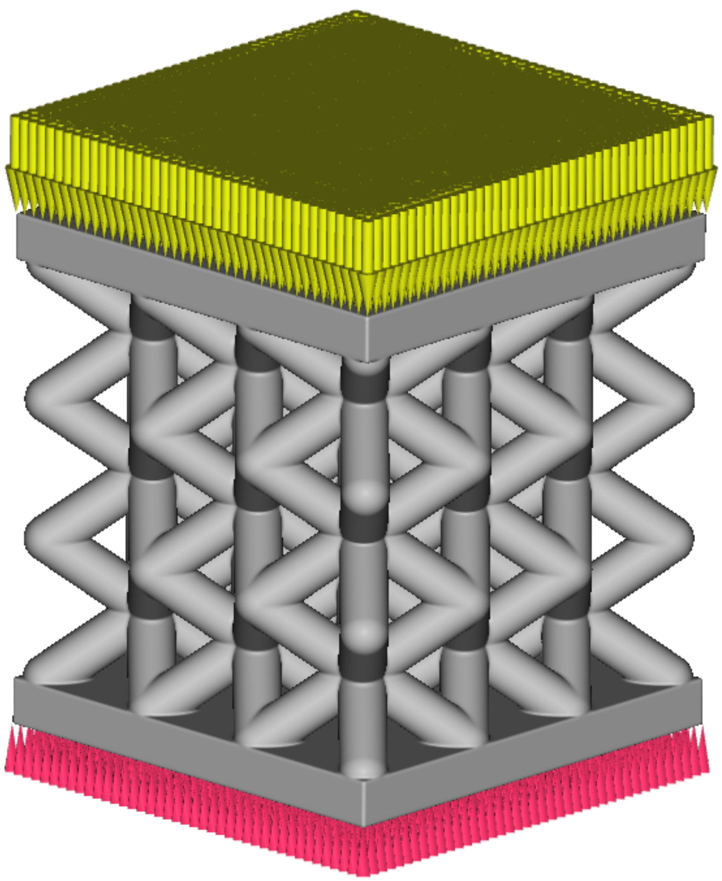
Follow Along: Homogenized Lattice Simulation

This video walks through an example of how to set up and run a simulation on a simple lattice structure using homogenization.

Please download the nTop File below to follow along with the tutorial. The completed file of this workflow will be available for download at the end of this course section.

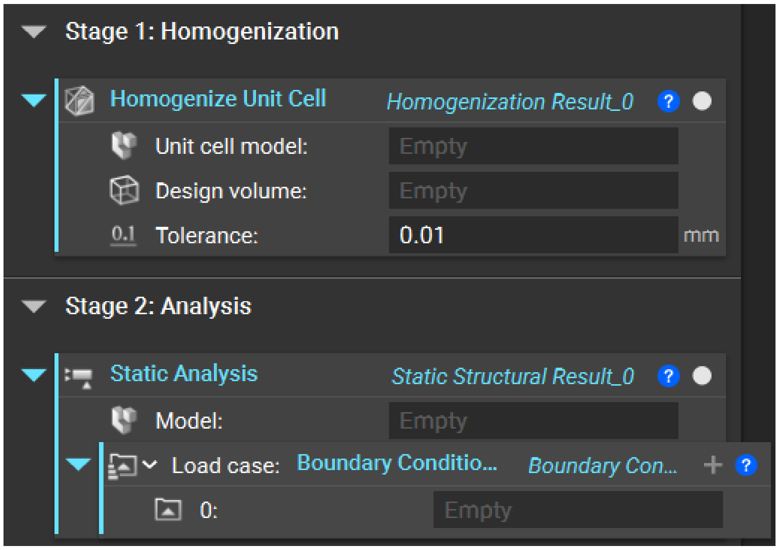
The starter file contains this lattice structure sandwiched between two plates as well as a pre-defined material. You will simulate this lattice with compressive load (1000 N) applied to the top plate and the bottom plate fixed in place.



Last updated on April 1, 2022 with version 3.21.4.

Step 1: Create a new section called 'Stage 1: Homogenization' and add the **Homogenize Unit Cell** block. Create another section called 'Stage 2: Analysis' then add the **Static Analysis** block.

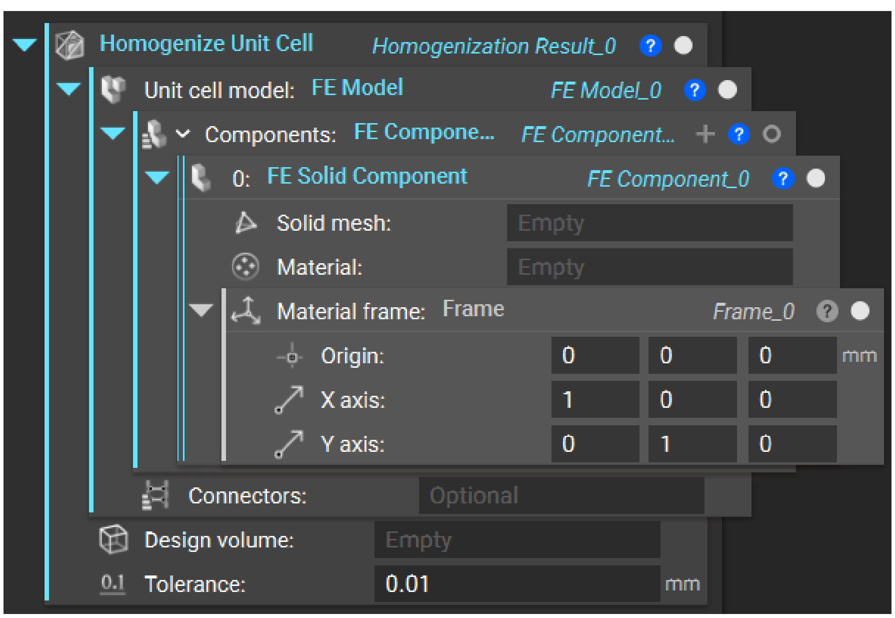
You will work backwards from each of these blocks to fill out all the necessary steps of each workflow.



Homogenization

Step 2: Starting with Homogenization, create the FE Model by double-clicking in the Model input and choosing **FE Model**

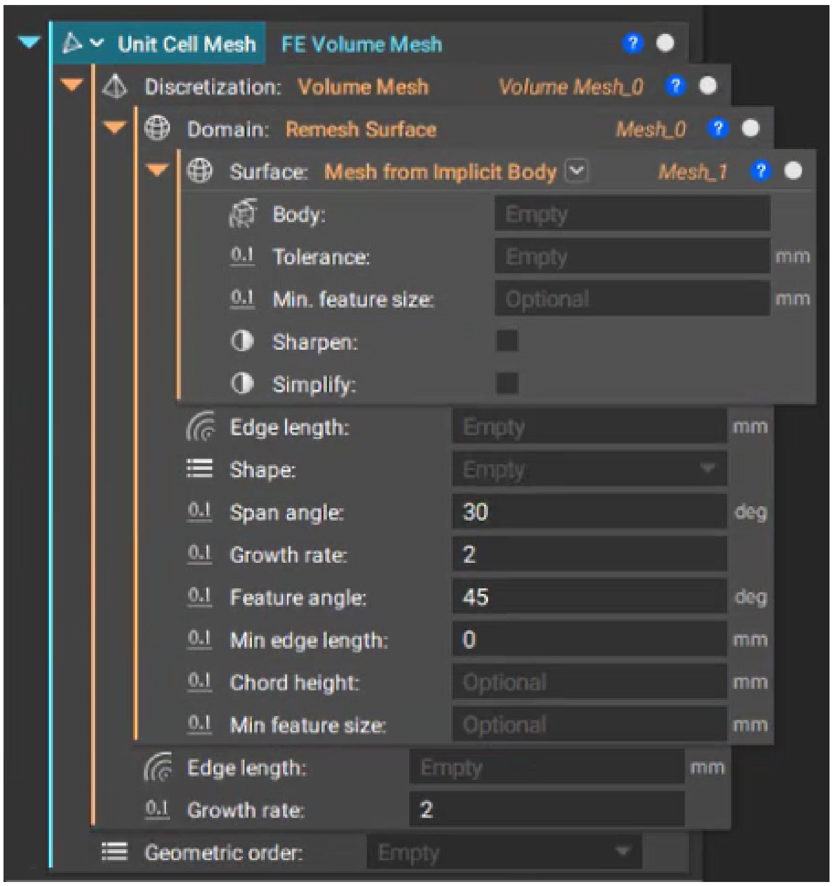
Then do the same in the components input of the **FE Model** and choose **FE Solid Component**.



Last updated on April 1, 2022 with version 3.21.4.

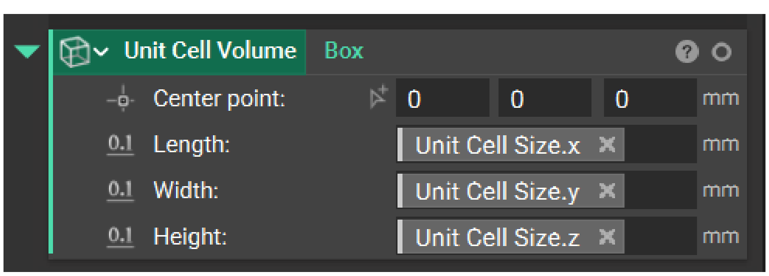
Step 3: For the Solid mesh input, generate a solid mesh of the lattice unit cell. Start by double-clicking in the Solid mesh input, choosing **FE Volume Mesh**, then making it into a variable called 'Unit Cell Mesh'.

Double-click into the first input of the **FE Volume Mesh** block and choose **Volume Mesh**. Do the same thing to fill out the next two blocks: **Remesh Surface** and **Mesh from Implicit Body**.



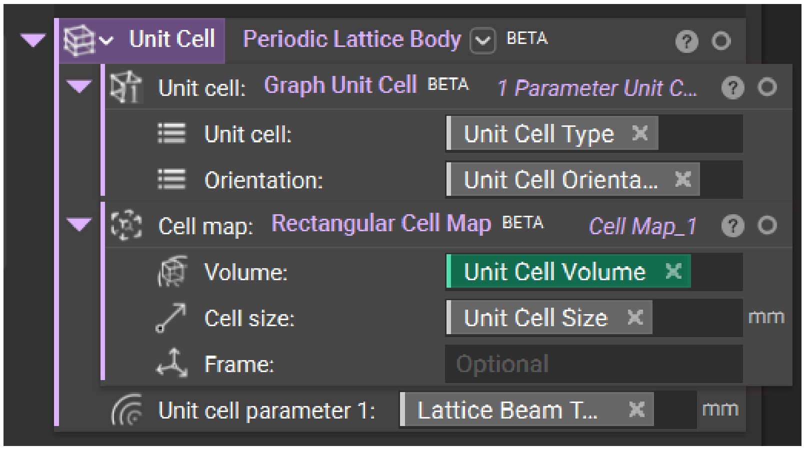
Step 4: Create a unit cell to use in the Body input of the **Mesh from Implicit Body** block.

Since the unit cell has to fit into a box, create a **Box** variable named 'Unit Cell Volume' with length, width, and height of the initial lattice's unit cell size in the x, y, and z directions respectively.

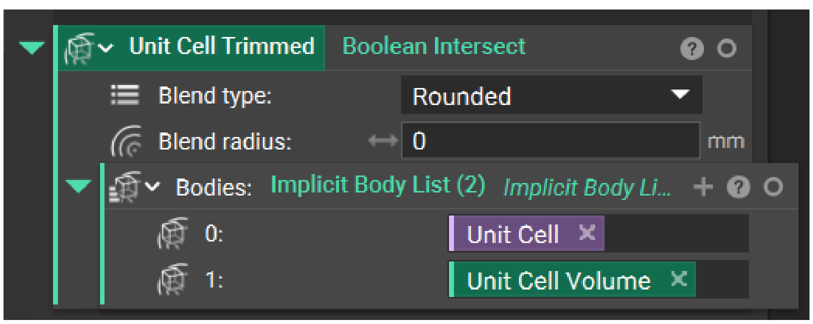


Last updated on April 1, 2022 with version 3.21.4.

Then create a lattice body using the **Periodic Lattice Body** block that is filled in with **Graph Unit Cell** and **Rectangular Cell Map**. Use the Unit Cell Volume box as this cell map's volume input and reference the unit cell type, orientation, size, and thickness from the original lattice. Make this lattice body a variable named 'Unit Cell'



Step 5: Trim the unit cell to the box using **Boolean Intersect**. Input this new implicit body into the meshing workflow previously created.



Step 6: Use the following inputs for the Unit Cell Mesh workflow:

Mesh from Implicit Body (2nd Overload)

- Tolerance 0.3 mm
- Sharpen iterations 1

Remesh Surface

- Edge length 0.25 mm - make into a variable called 'Mesh Size'
- Shape Triangle

Volume Mesh

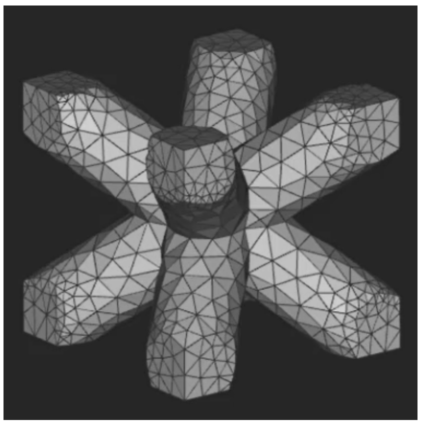
- Edge length 0.25 mm (use the variable from above)

FE Volume Mesh

- Geometric order Linear

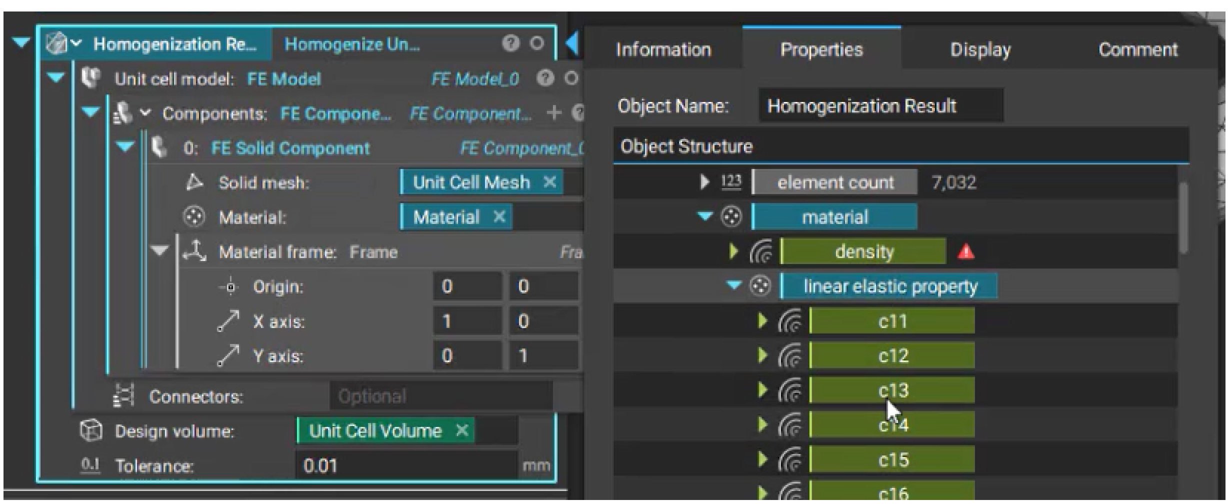
Check the properties of **Mesh from Implicit Body** and **Remesh Surface** blocks to ensure that the mesh is closed, edge manifold, and not self-intersecting.

Once the entire meshing workflow is complete, your mesh should look like the image below.



Step 7: Use the given Material variable as input for the **FE Solid Component** material to complete the **FE Model** for **Homogenize Unit Cell**.

Use the **Unit Cell Volume** box in the Design volume input.



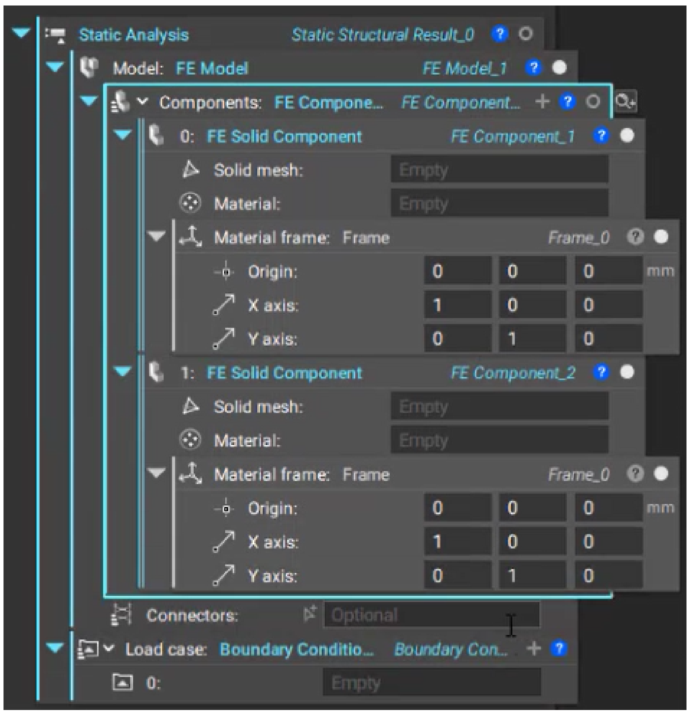
After the homogenization block finishes running, you will be able to read the compliance and stiffness matrix coefficients under *linear elastic property* in the block's properties and export them.

Last updated on April 1, 2022 with version 3.21.4.

Analysis

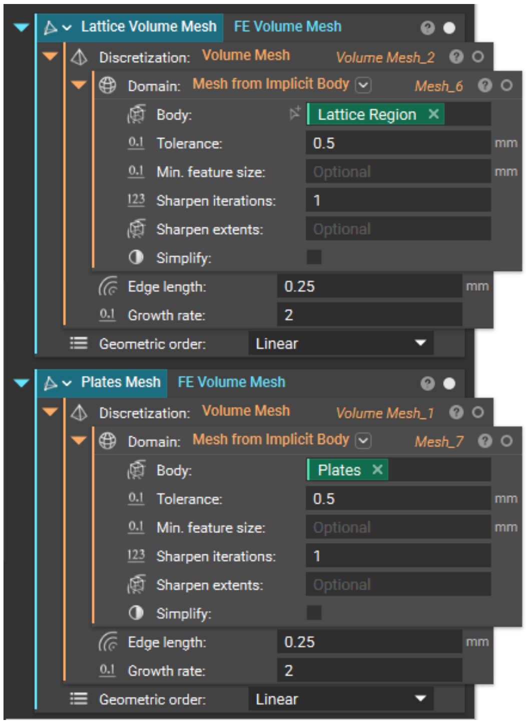
Step 8: Work backwards from the Static Analysis block to fill out all the necessary steps of this workflow. Create the FE Model by double-clicking in the Model input and choosing **FE Model**

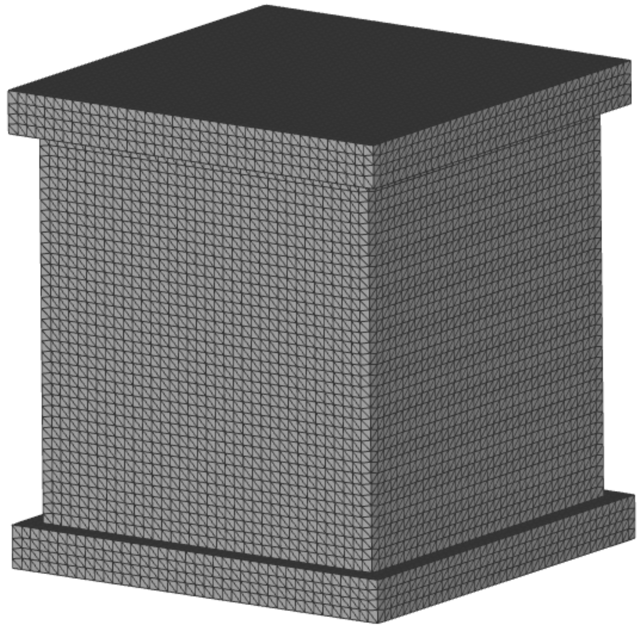
Then do the same in the components input of the **FE Model** and choose **FE Solid Component**. Add another component to the list and fill it with another **FE Solid Component**.



Step 9: Each component needs a mesh of the corresponding structure; solid mesh of the representative bulk lattice volume in the first **FE Solid Component** and solid mesh of the plates in the second **FE Solid Component**.

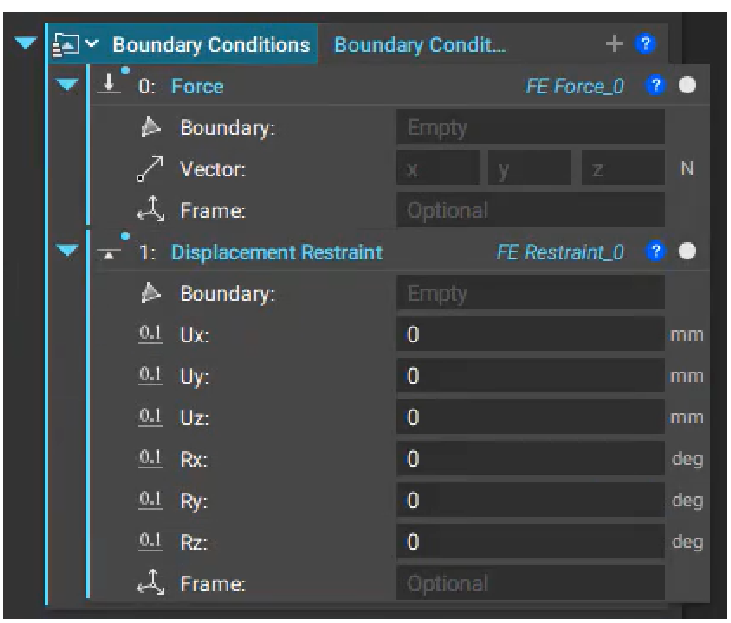
Create each of these using **FE Volume Mesh** block and the standard meshing workflow, leaving out the **Remesh Surface** because of the simple geometry. Make them into variables called "Lattice Volume Mesh" and "Plates Mesh" respectively.





Step 10: Place each completed mesh into their component and assign the material using the given Material variable.

Step 11: Make the Boundary Conditions List a variable and define the load case using the **Force** and **Displacement Restraint** blocks.

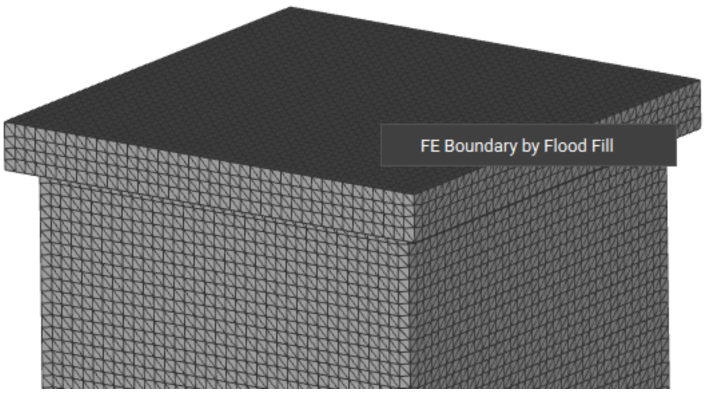


Last updated on April 1, 2022 with version 3.21.4.

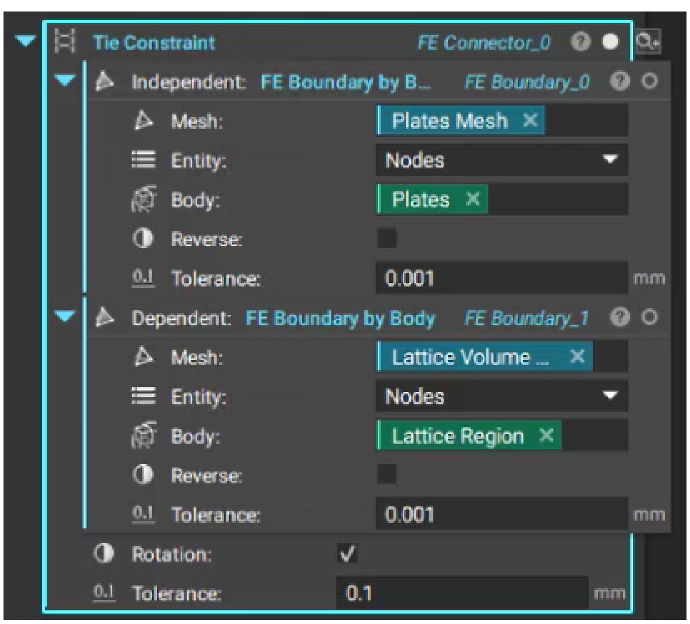
Step 12: Fill in the **Force** and **Displacement Restraint** boundary conditions.

Use force vector of 0, 0, -1000 N.

Select the boundary input by isolating the Solid Mesh and right-click on the top surface to select *FE Boundary by Flood Fill*. A block will be created, which you can drag into the input of the **Force** block. Repeat on the bottom plate surface to select the boundary for the **Displacement Restraint**.



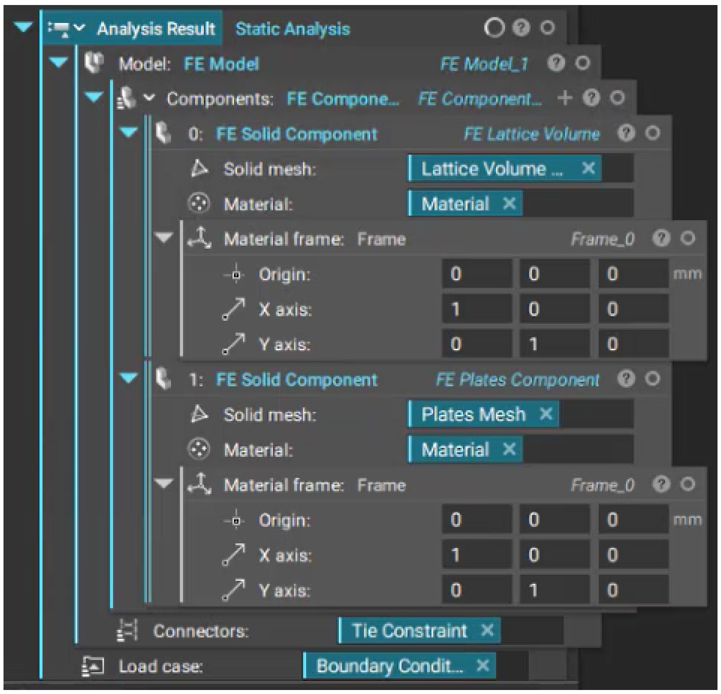
Step 13: With the FE Model and Boundary Conditions completed, you will get an error when Static Analysis block runs. This is because there are two components in the FE Model, but they are not connected. Add a **Tie Constraint** block to establish a connection between the two components.

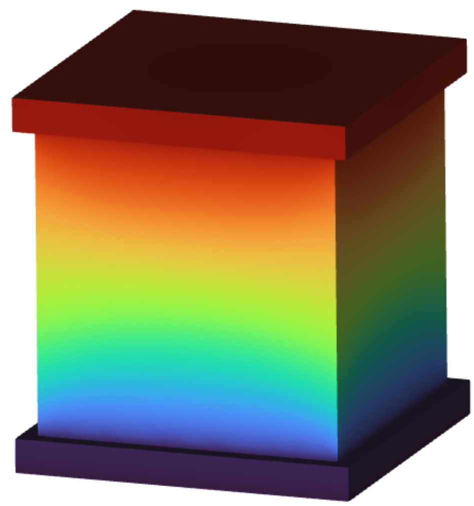


Last updated on April 1, 2022 with version 3.21.4.

Select the Independent and Dependent boundaries using two **FE Boundary by Body** blocks. Keep the tolerance to default as all needed nodes are captured with this value. Place this **Tie Constraint** in the Connectors input of the FE Model.

Step 14: With all the inputs in the previous steps filled in, the **Static Analysis** block will begin to run. Make it into a variable called ‘Analysis Result’. Isolate the block to see the results.





Last updated on April 1, 2022 with version 3.21.4.