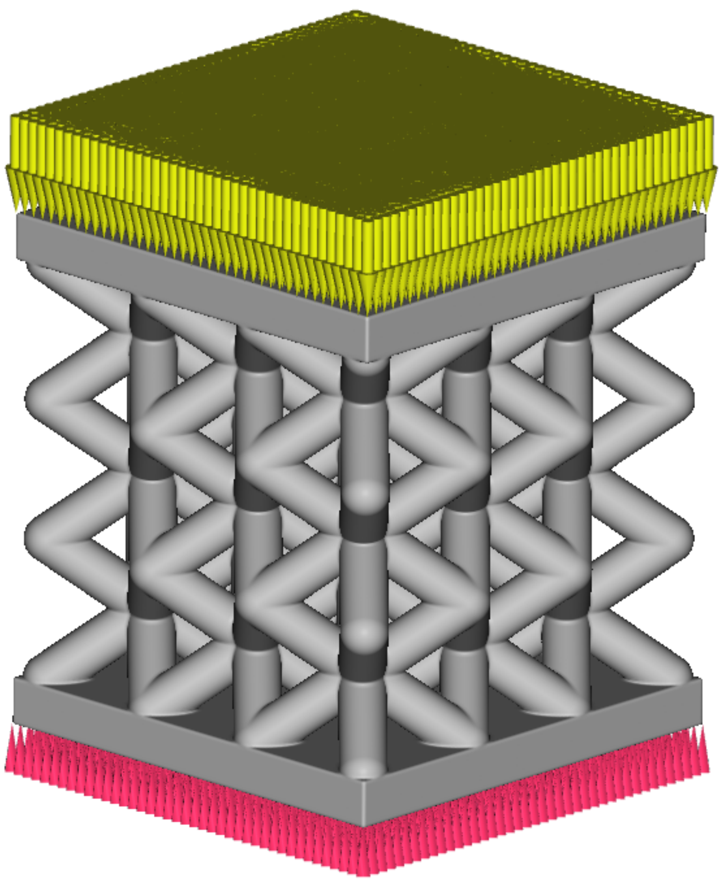
Follow Along: Lattice Simulation Using Solid Elements

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

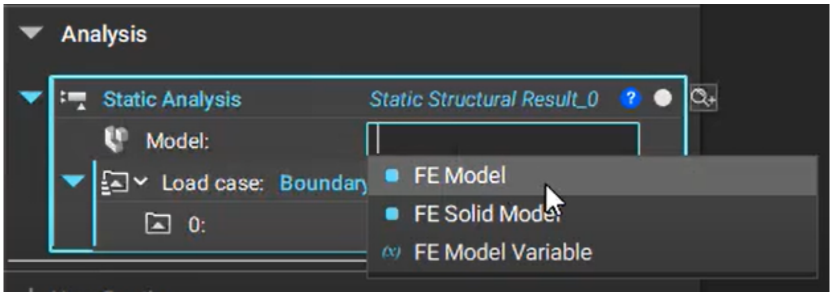
Please download the nTop File below to follow along with the tutorial. The completed file of this workflow will be available for download at the end of this course section.

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.

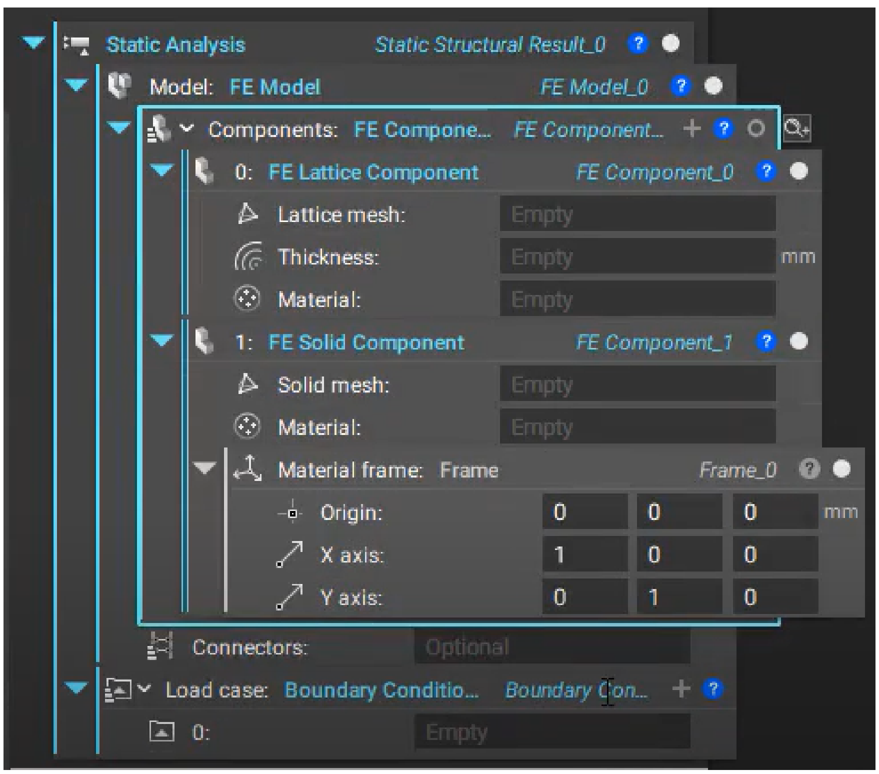


Last updated on April 1, 2022 with version 3.21.4.

Step 1: Create a new section called 'Analysis' and add the **Static Analysis** block to it. Work backwards from this 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**



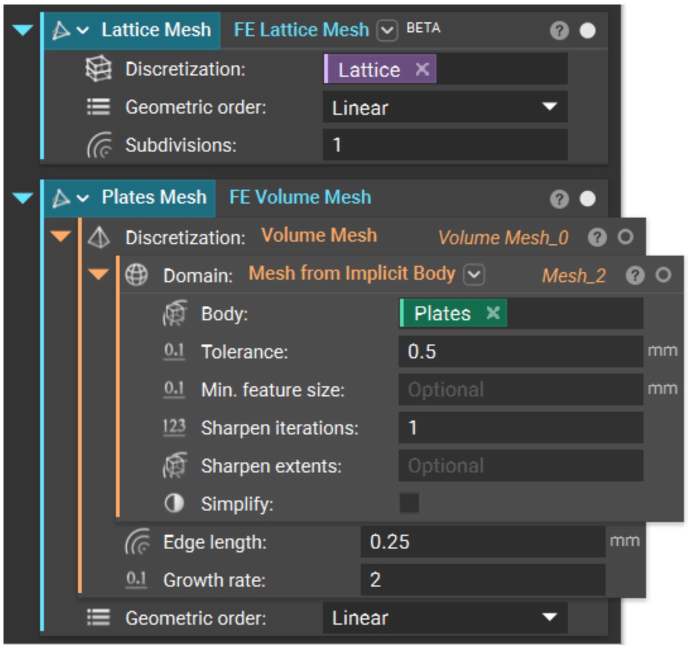
Then do the same in the components input of the **FE Model** and choose **FE Lattice Component**. Add another component to the list and fill it with an **FE Solid Component**.

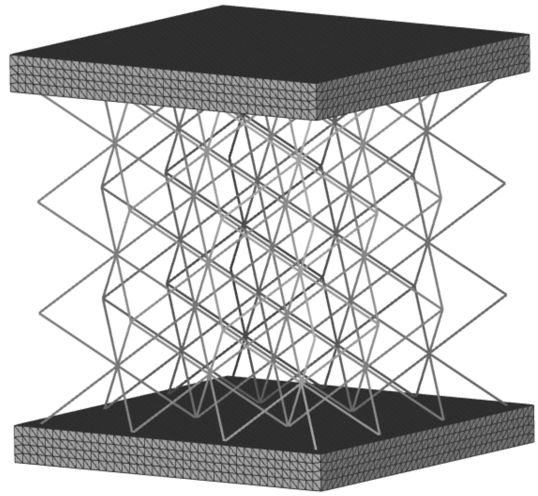


Step 2: Each component needs a mesh of the corresponding structure; beam mesh of the lattice in the **FE Lattice Component** and solid mesh of the plates in the **FE Solid Component**.

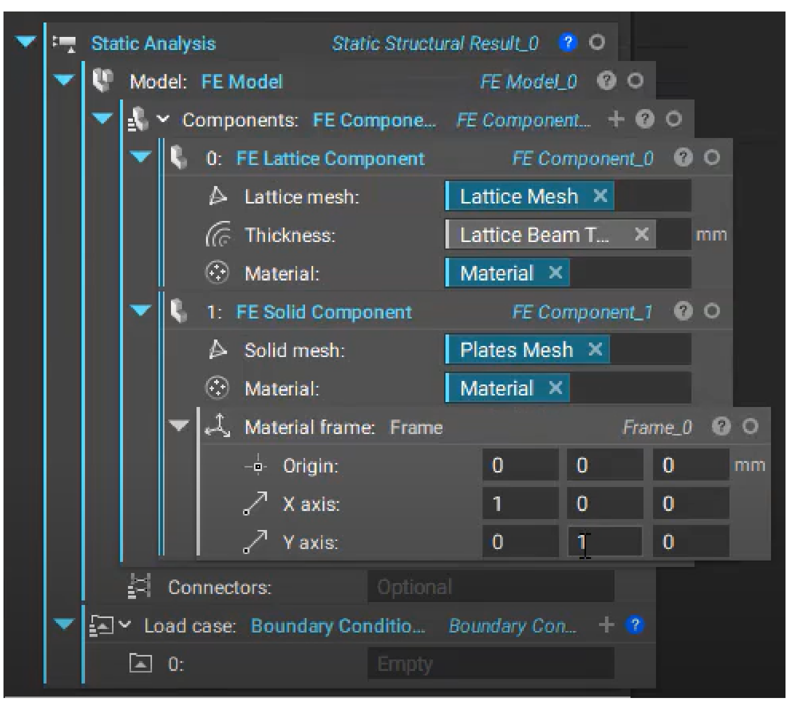
Create the former using the **FE Lattice Mesh** block and the latter using **FE Volume Mesh** block and the standard meshing workflow, leaving out the **Remesh Surface** because of the simple plates geometry.

Use the input values shown in the image below.



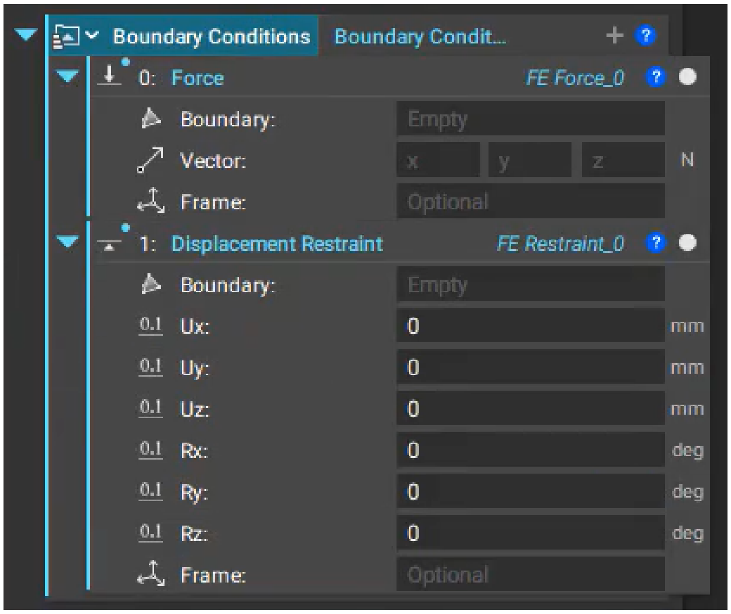


Step 3: Place each completed mesh into their component and assign the material using the given material variable. Drag the Lattice Beam Thickness variable into the thickness input of the FE Lattice Component.



Last updated on April 1, 2022 with version 3.21.4.

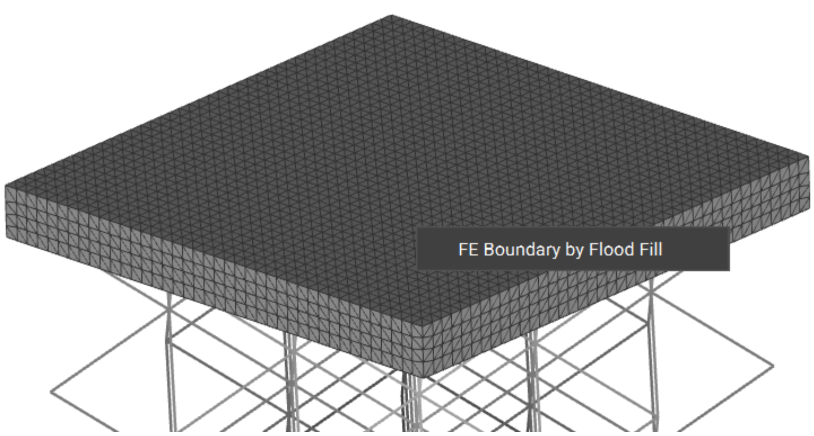
Step 4: Make the Boundary Conditions List a variable and define the load case using the **Force** and **Displacement Restraint** blocks.



Step 7: Fill in the **Force** and **Displacement Restraint** boundary conditions.

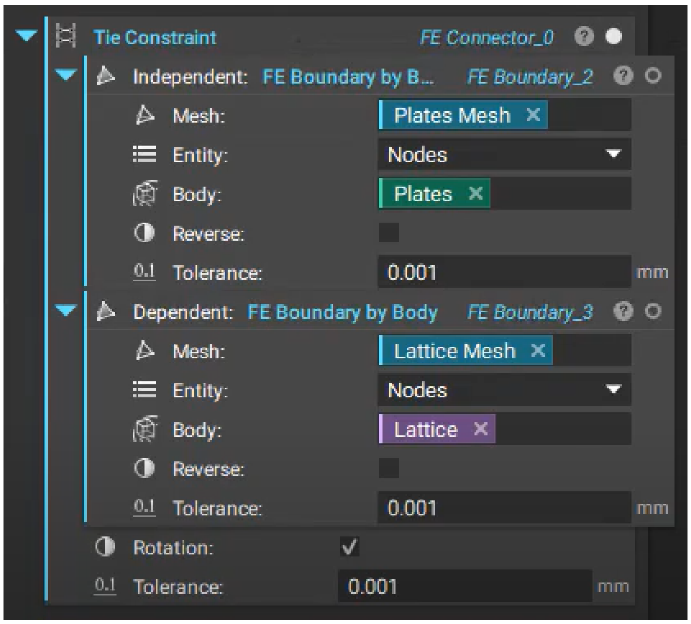
Use force vector of 0, 0, -1000 N.

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**.

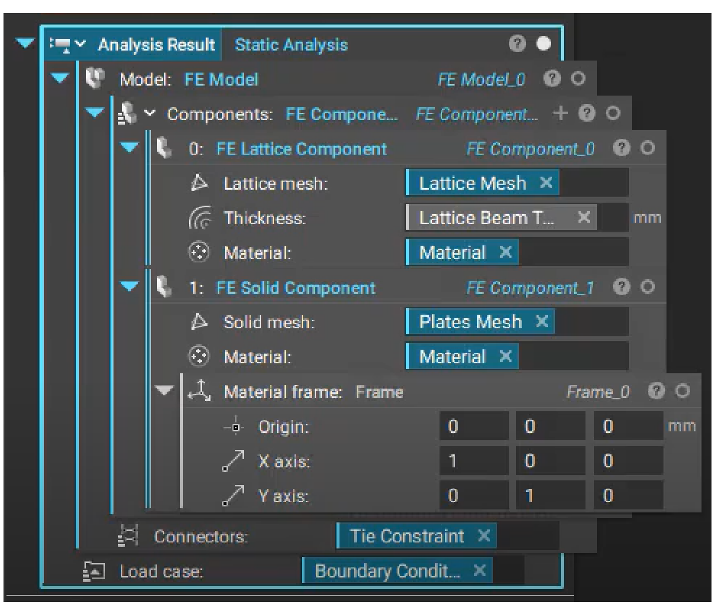


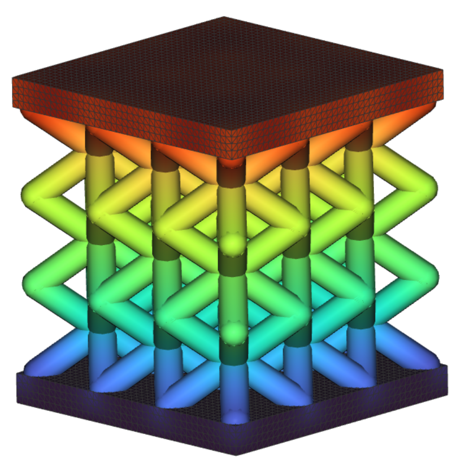
Step 8: With the FE Model and Boundary Conditions completed, you will get an error when Static Analysis block runs. This is because there are two components in the FE Model, but they are not connected. Add a **Tie Constraint** block to establish a connection between the two components.

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 9: 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.