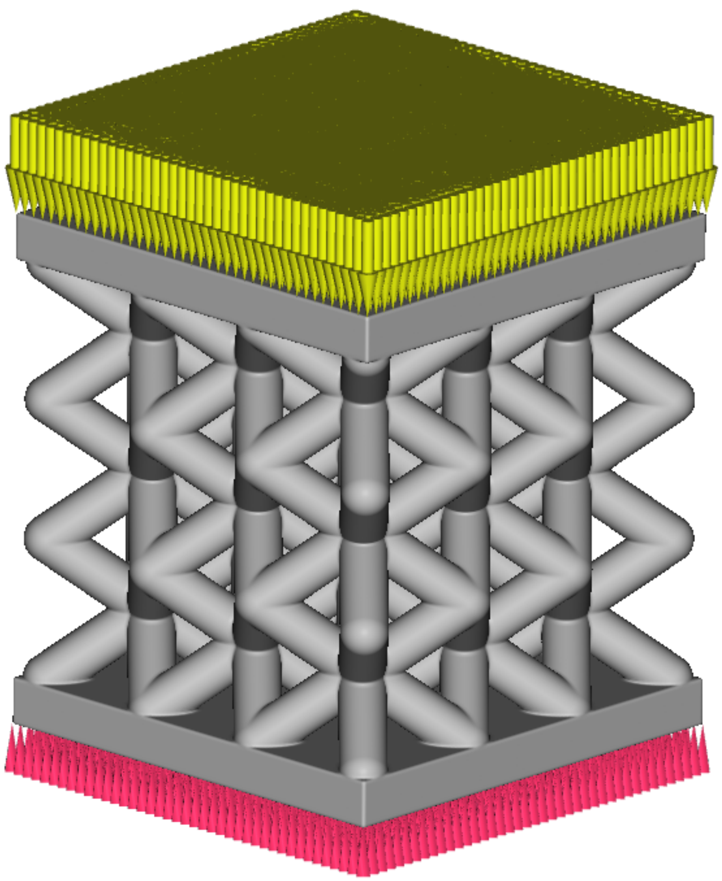
Follow Along: Lattice Simulation Using Solid Elements

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

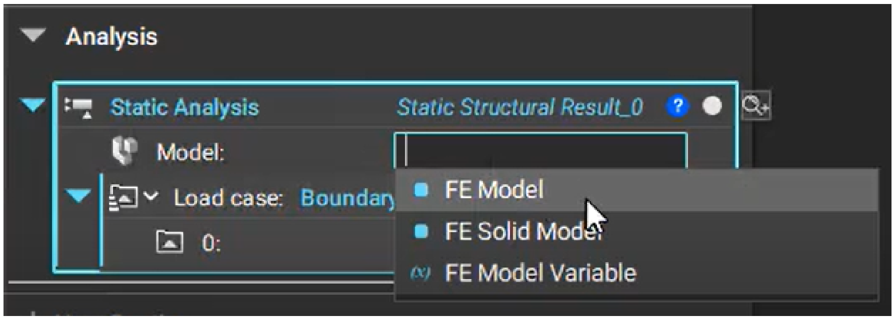
Please download the nTop File below to follow along with the tutorial. The completed file of this workflow will be available for download at the end of this course section.

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.

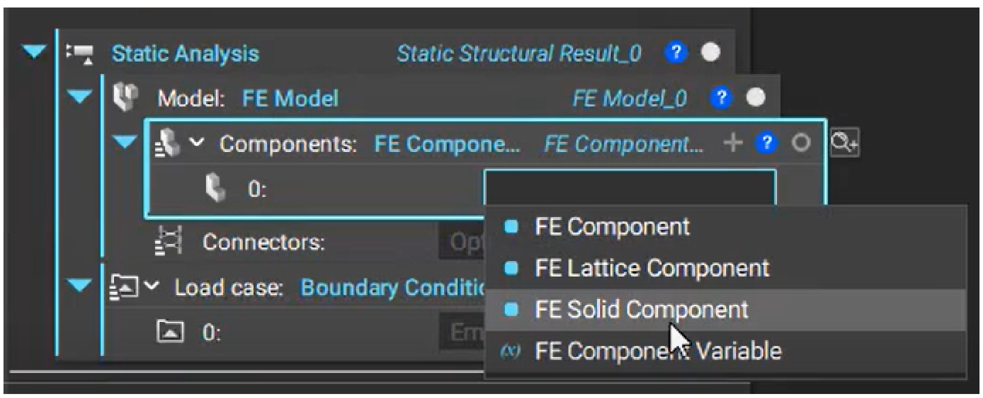


Last updated on April 1, 2022 with version 3.21.4.

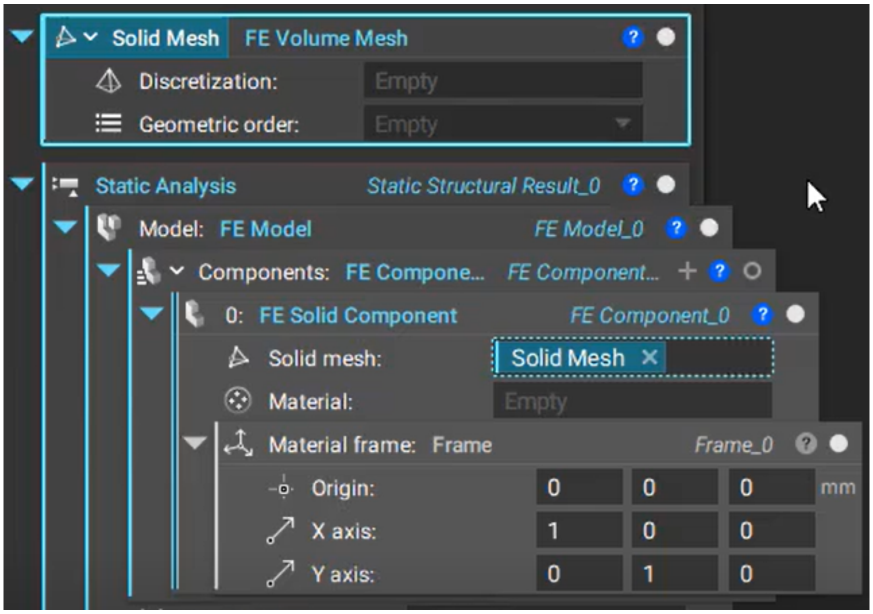
Step 1: Create a new section called 'Analysis' and add the **Static Analysis** block to it. Work backwards from this block to fill out all the necessary steps of this workflow. Create the FE Model by double-clicking in the Model input and choosing **FE Model**



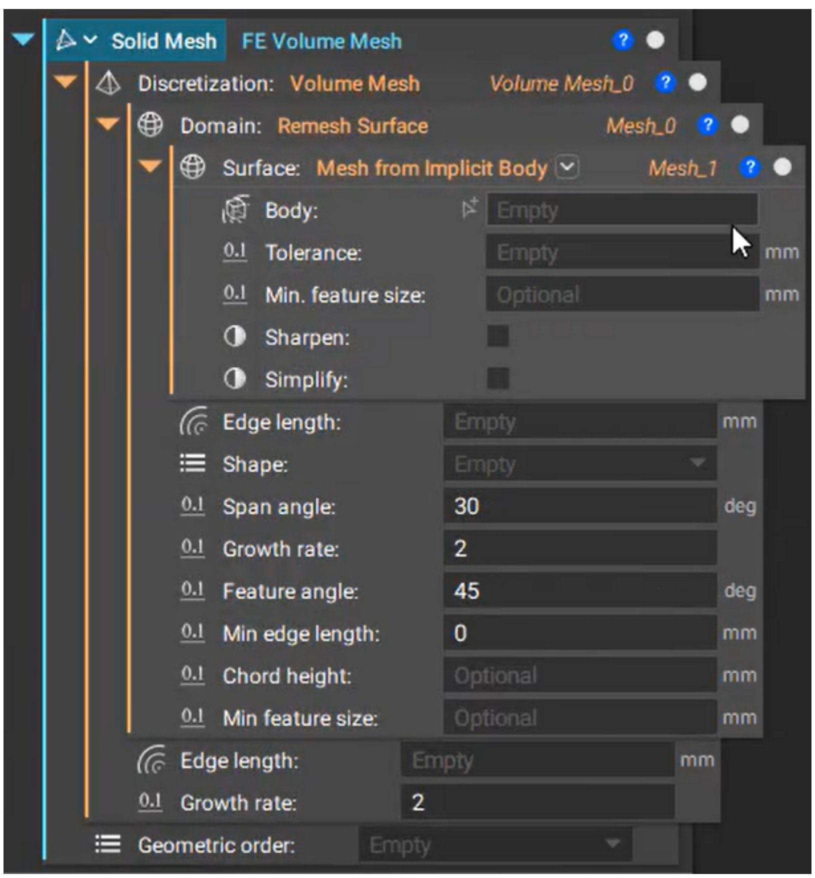
Then do the same in the components input of the **FE Model** and choose **FE Solid Component**



Step 2: For the Solid mesh input, generate a solid mesh of the lattice structure. Start by double-clicking in the Solid mesh input, choosing **FE Volume Mesh**, then making it into a variable called 'Solid Mesh'.



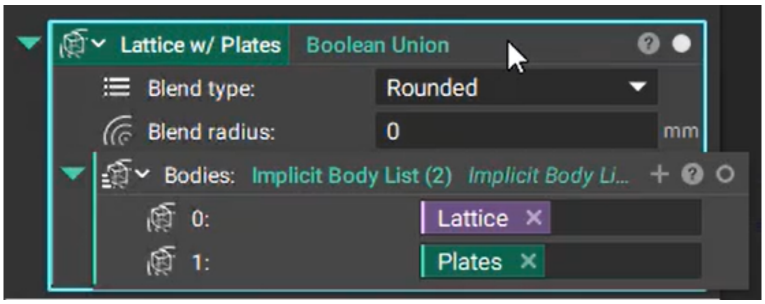
Step 3: Double-click into the first input of the **FE Volume Mesh** block and choose **Volume Mesh**. Do the same thing to fill out the next two blocks: **Remesh Surface** and **Mesh from Implicit Body**.



Last updated on April 1, 2022 with version 3.21.4.

Right-click on **Remesh Surface** and select Manual Run Mode. A pause button will appear next to it. Do the same for **Volume Mesh**. This is to make sure the entire workflow doesn't run until you let it, so that you can change the values and check the mesh quality before moving on to the next step.

Step 3: For the Body input of **Mesh from Implicit Body**, create a Boolean Union variable of the Lattice body and the Plates implicit body.



Step 4: Use the following inputs for the Solid Mesh workflow:

Mesh from Implicit Body (2nd Overload)

- Tolerance 0.5 mm
- Sharpen iterations 1

Remesh Surface

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

Volume Mesh

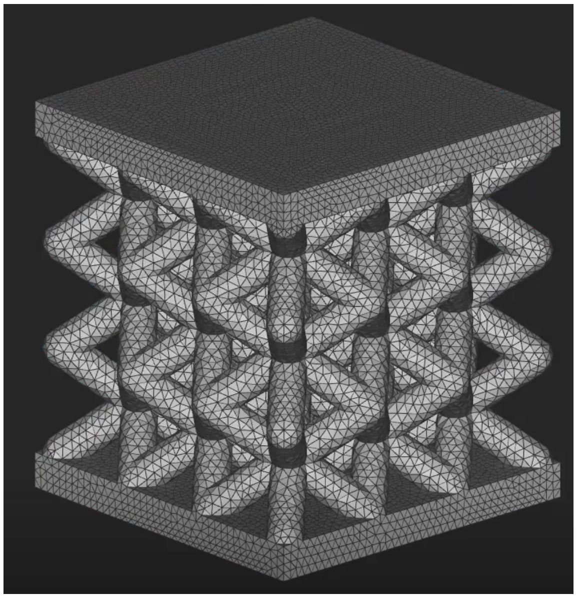
- Edge length 0.25 mm (use the variable from above)

FE Volume Mesh

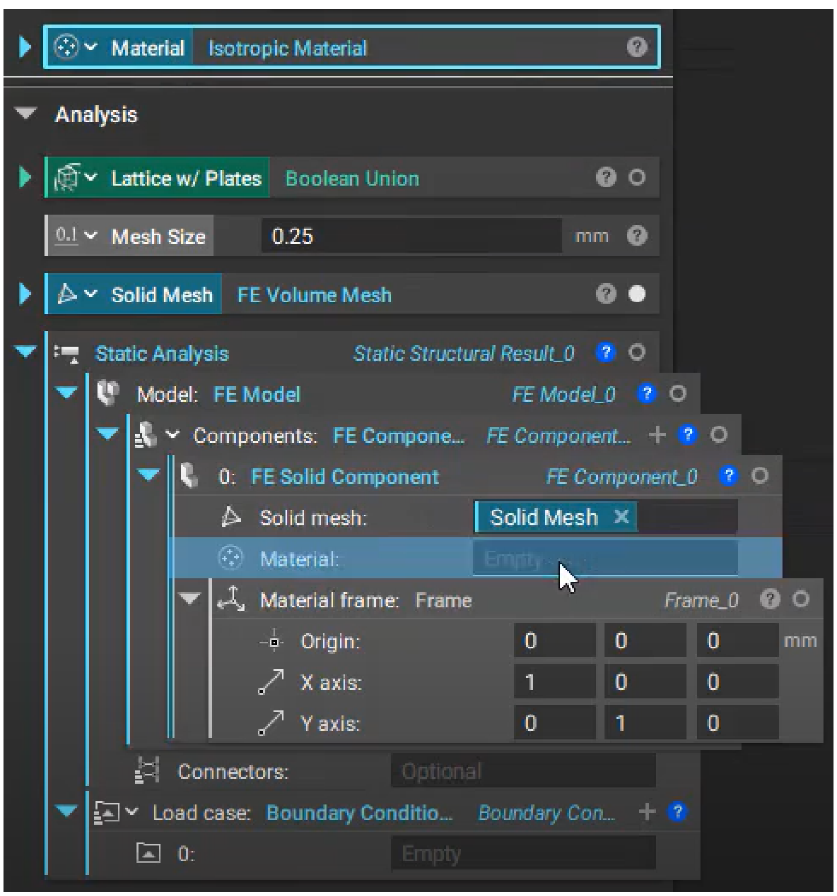
- Geometric order Linear

After filling out and running each block, check the properties 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.

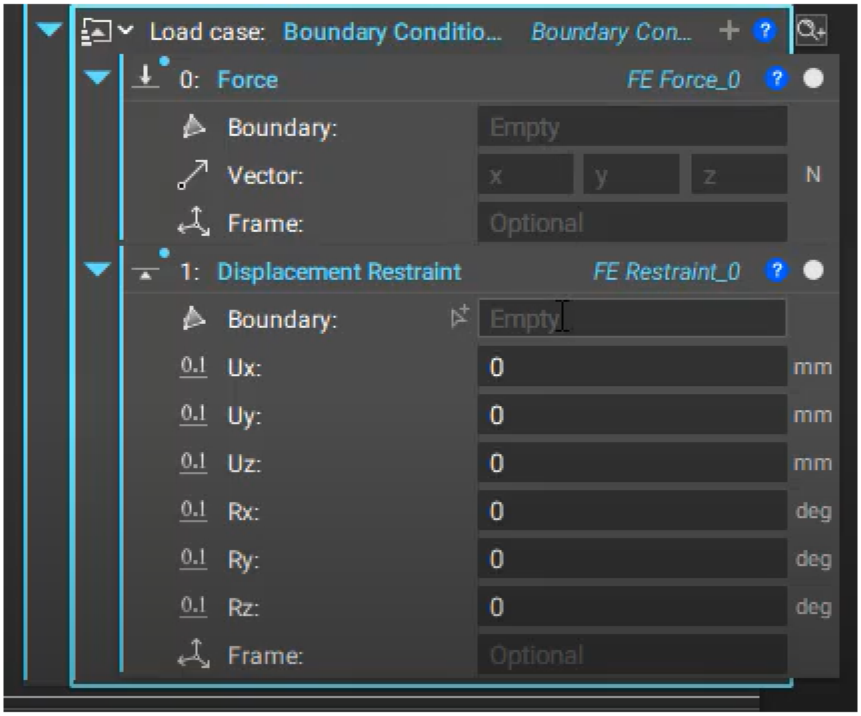


Step 5: Use the given Material variable as input for the **FE Solid Component** material to complete the **FE Model**.



Last updated on April 1, 2022 with version 3.21.4.

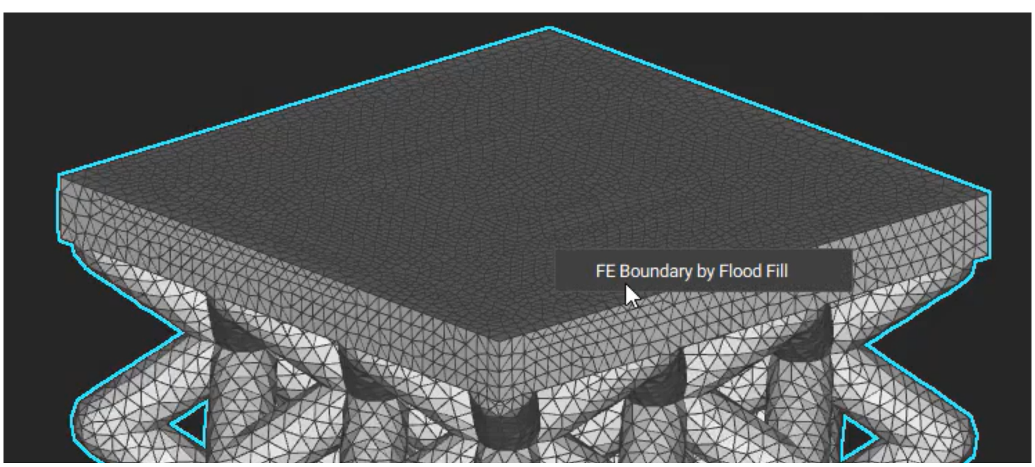
Step 6: Start filling out the load case in the boundary conditions list. You will need a **Force** block and a **Displacement Restraint** block.



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**.



Last updated on April 1, 2022 with version 3.21.4.

Step 8: 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.