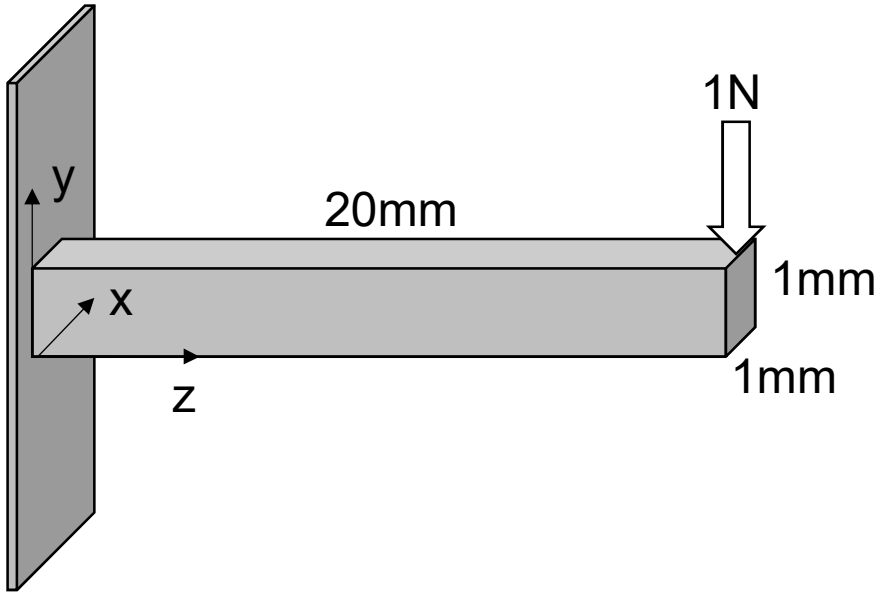
Finite element analysis of beam bending

imaiy@cc.saga-u.ac.jp



Displacement=P*L^3/(3EI)
I=bh^3/12
Iron bar

P	N	1
L	m	20e-3
E	Pa	211e9
I	m4	8.33e-14
Disp.	m	1.52e-4

	workflow	software
pre-process1	Creating geometries Export geometries in STEP format	FreeCAD
pre-process2	Import geometries Creating mesh (discretizing) Assign Materials Set fix points Set loading points	SimScale
processing	Run simulations	SimScale
post-process	Check displacement and stress	SimScale

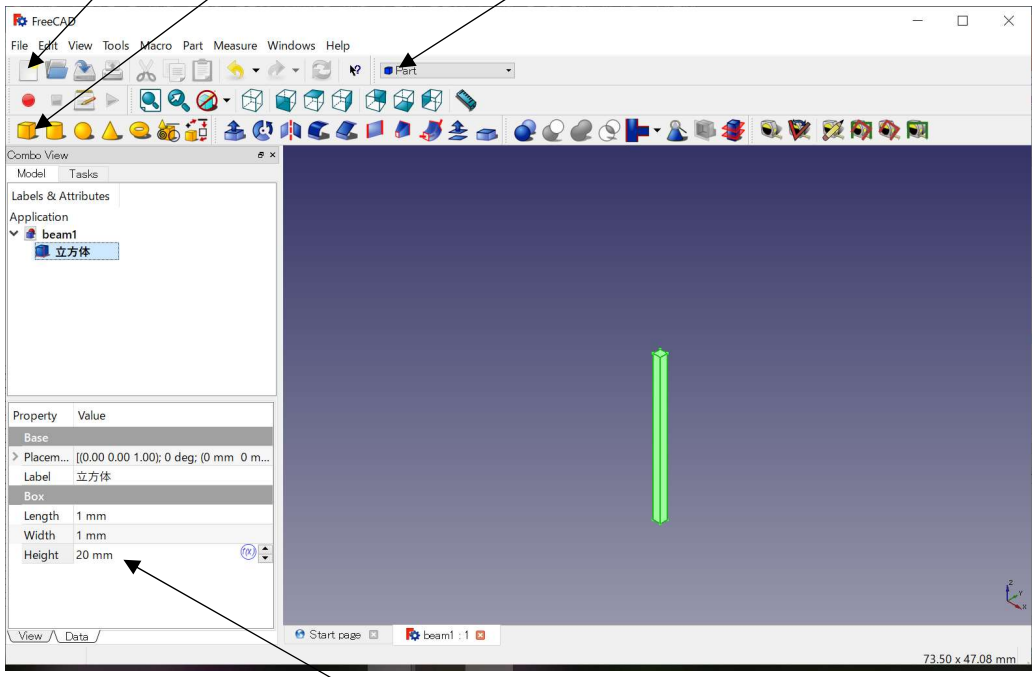
Design loop

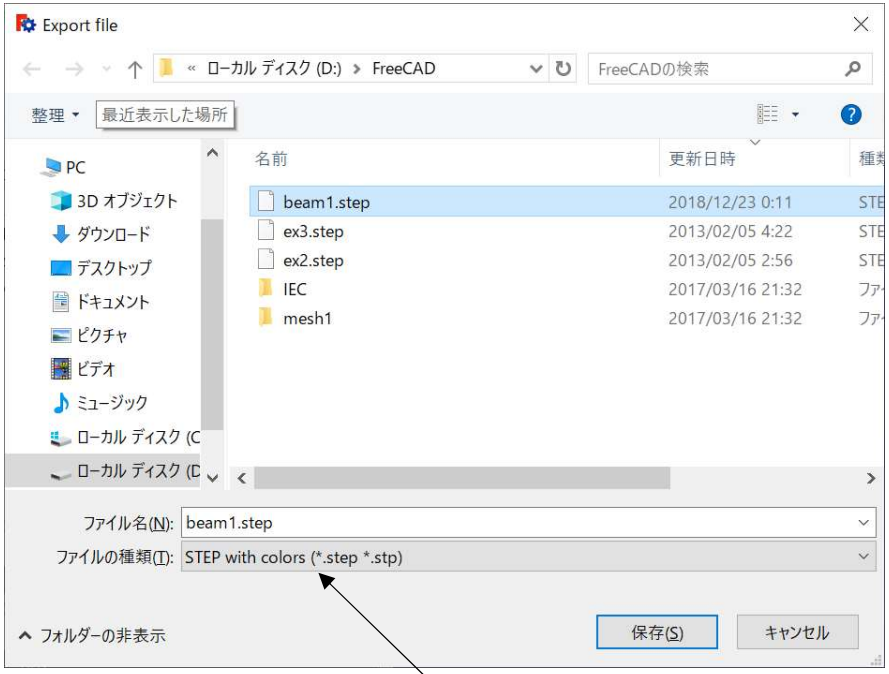
Creating geometries on FreeCAD

1:New

3:Cube solid

2:Part



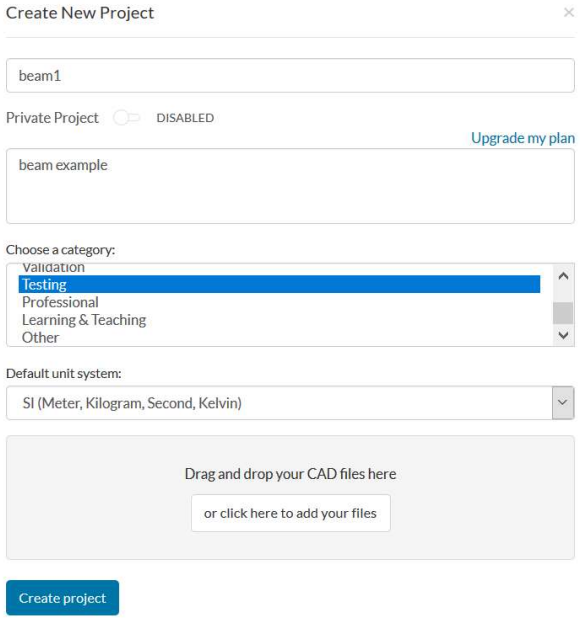


4:Length=1mm, Width=1mm, Height=20mm

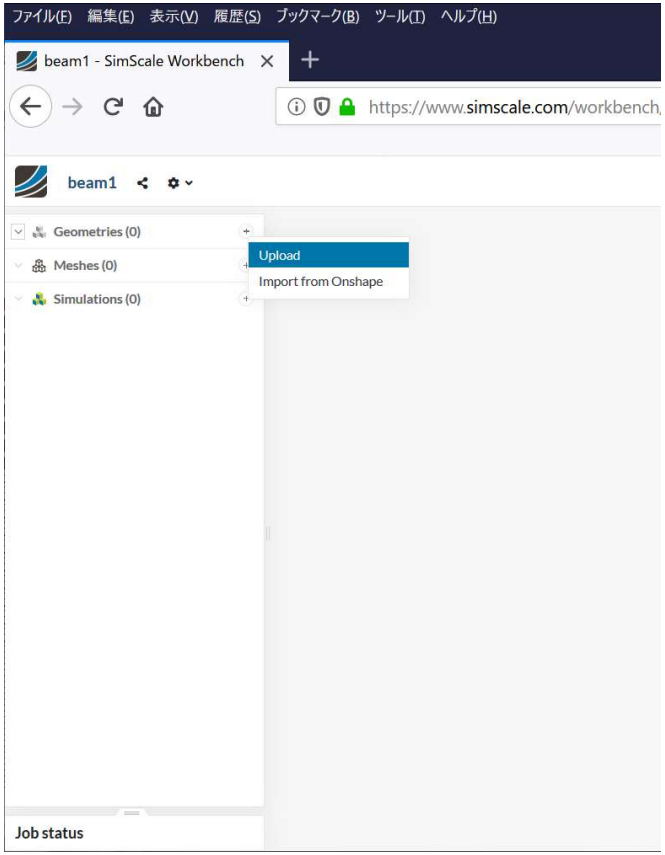
5:File>Export
Select STEP format

Import geometries on SimScale

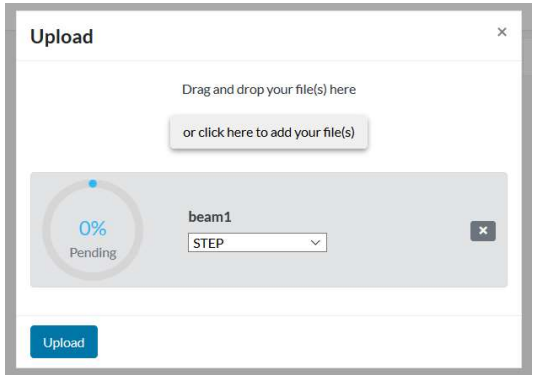
New Project



Geometries > Upload

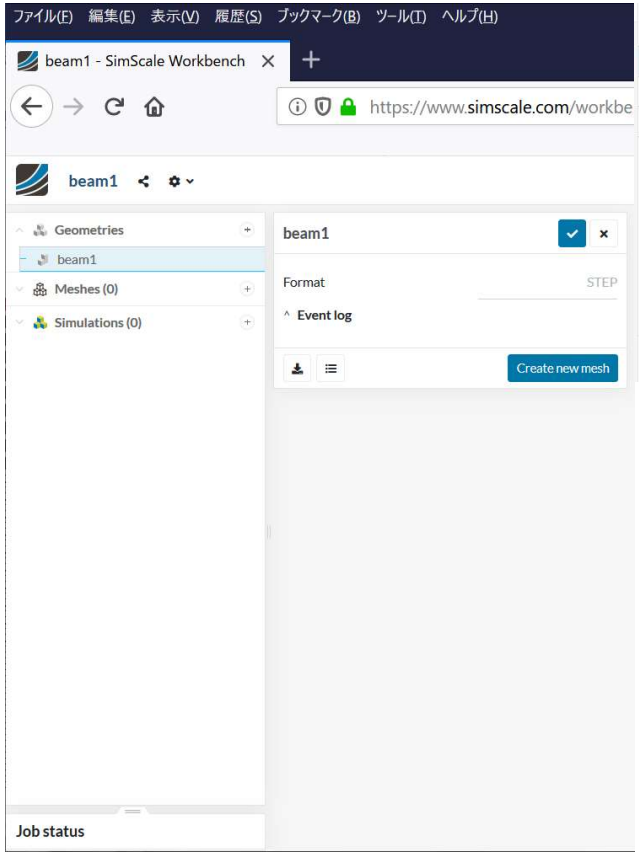


Select STEP file

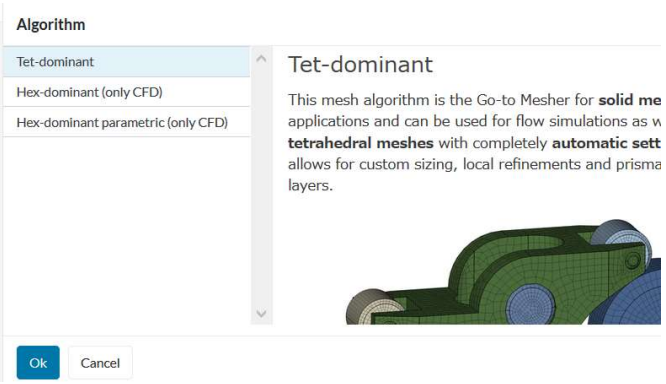


Meshing on SimScale

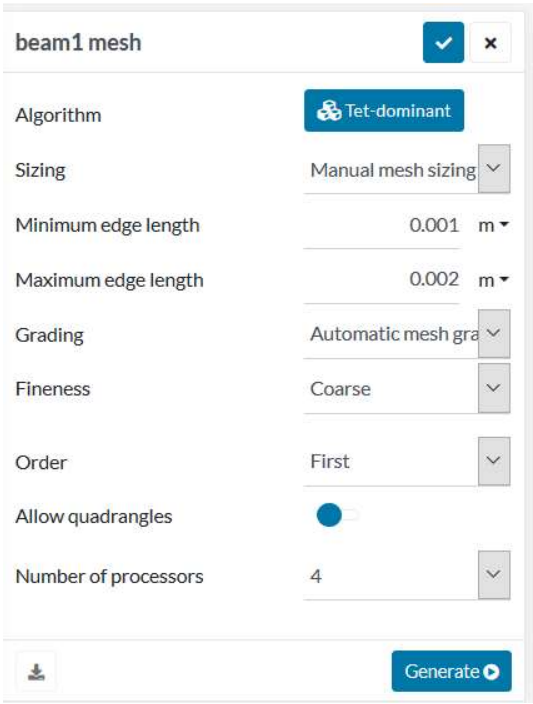
Create new mesh



Tet-dominant

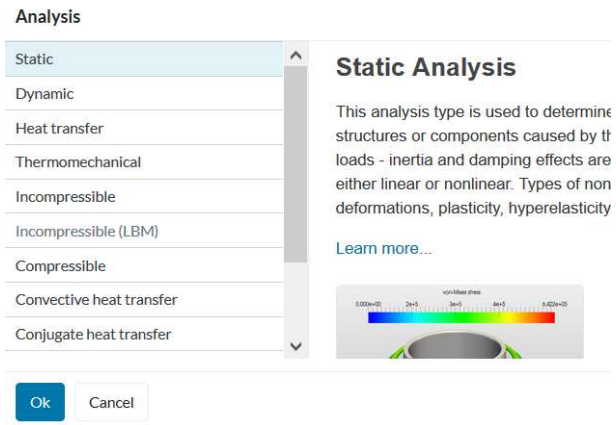


Edge length(Manual)

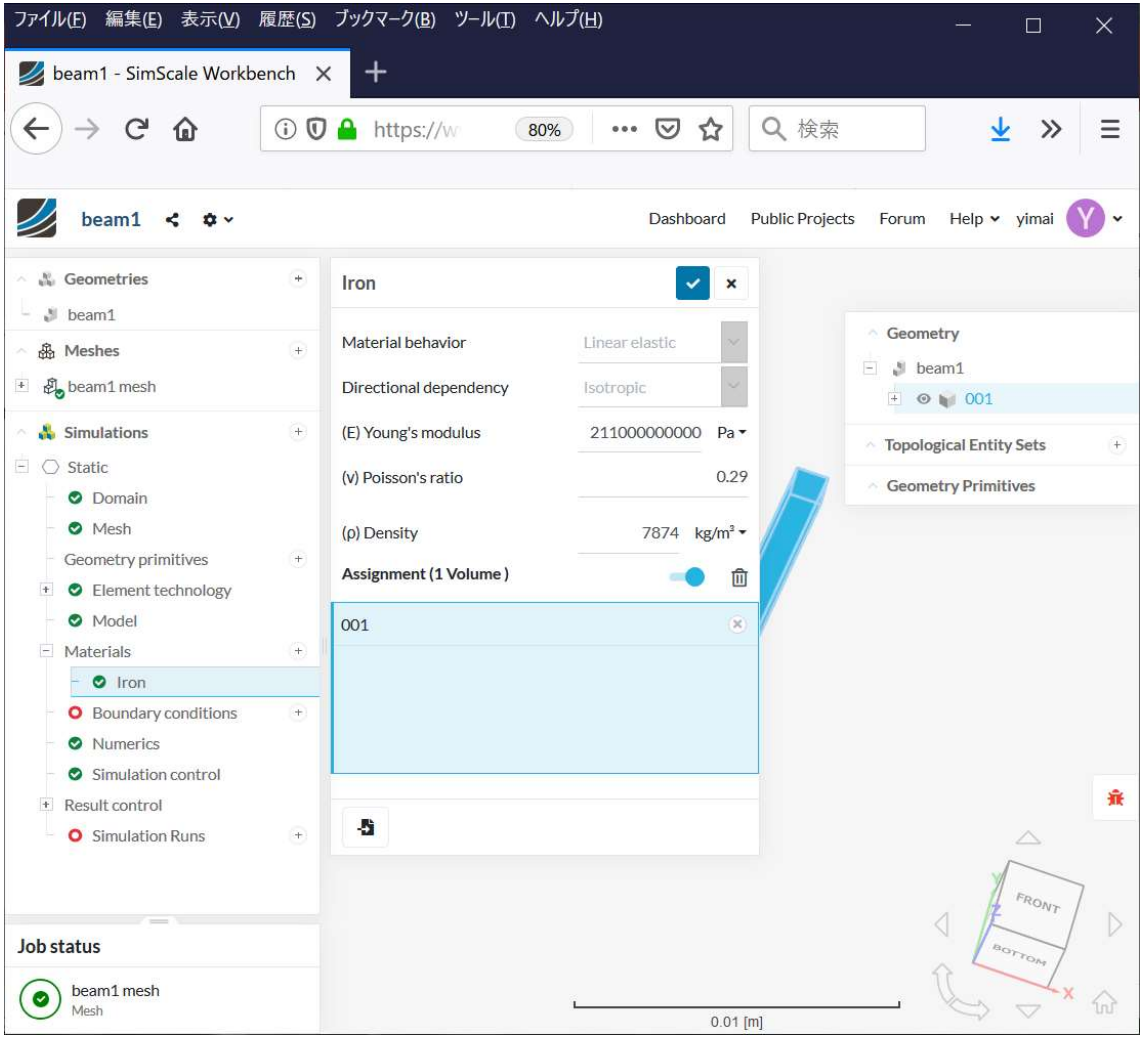


Assign materials

Static Analysis

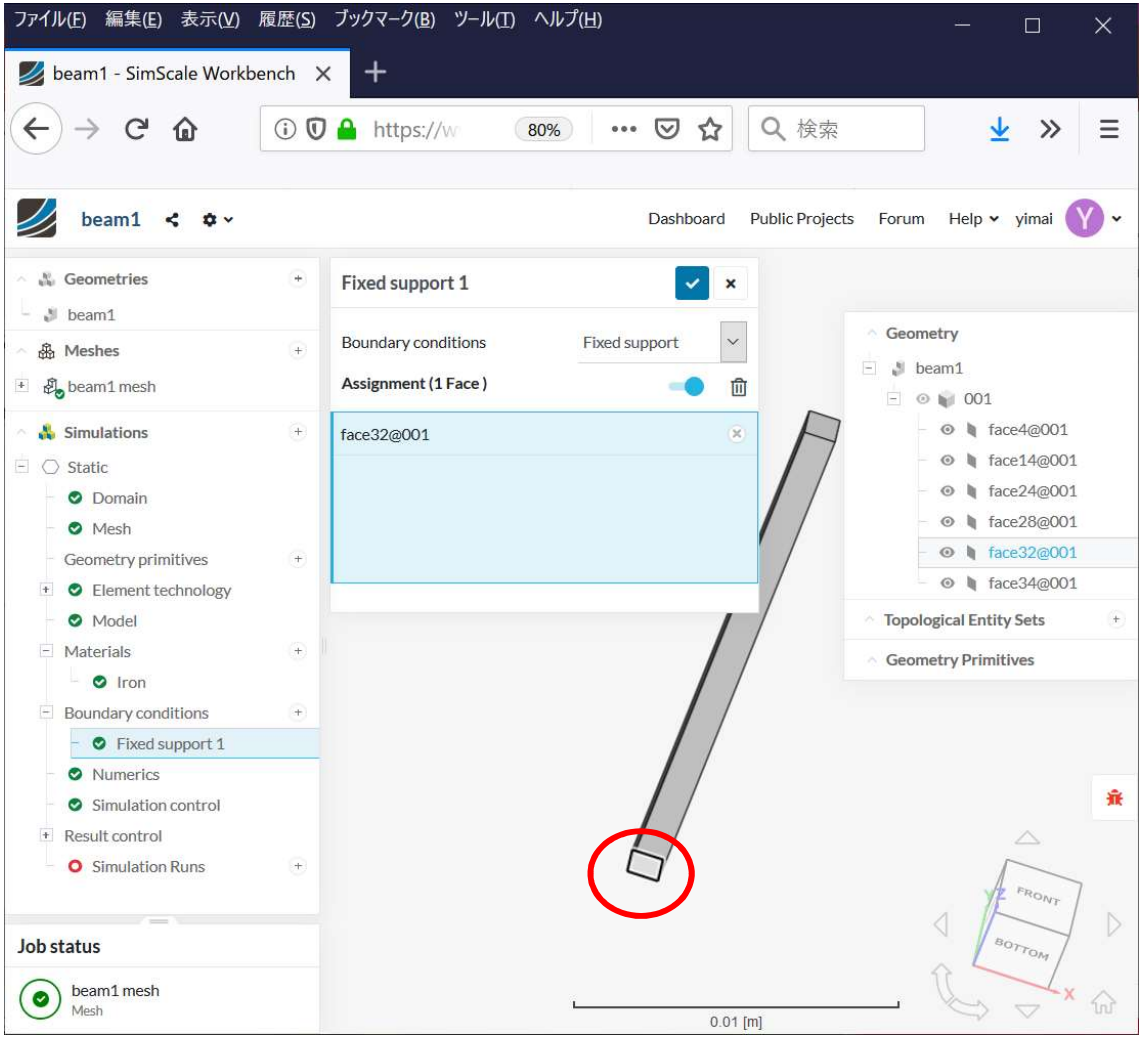


Materials (iron)



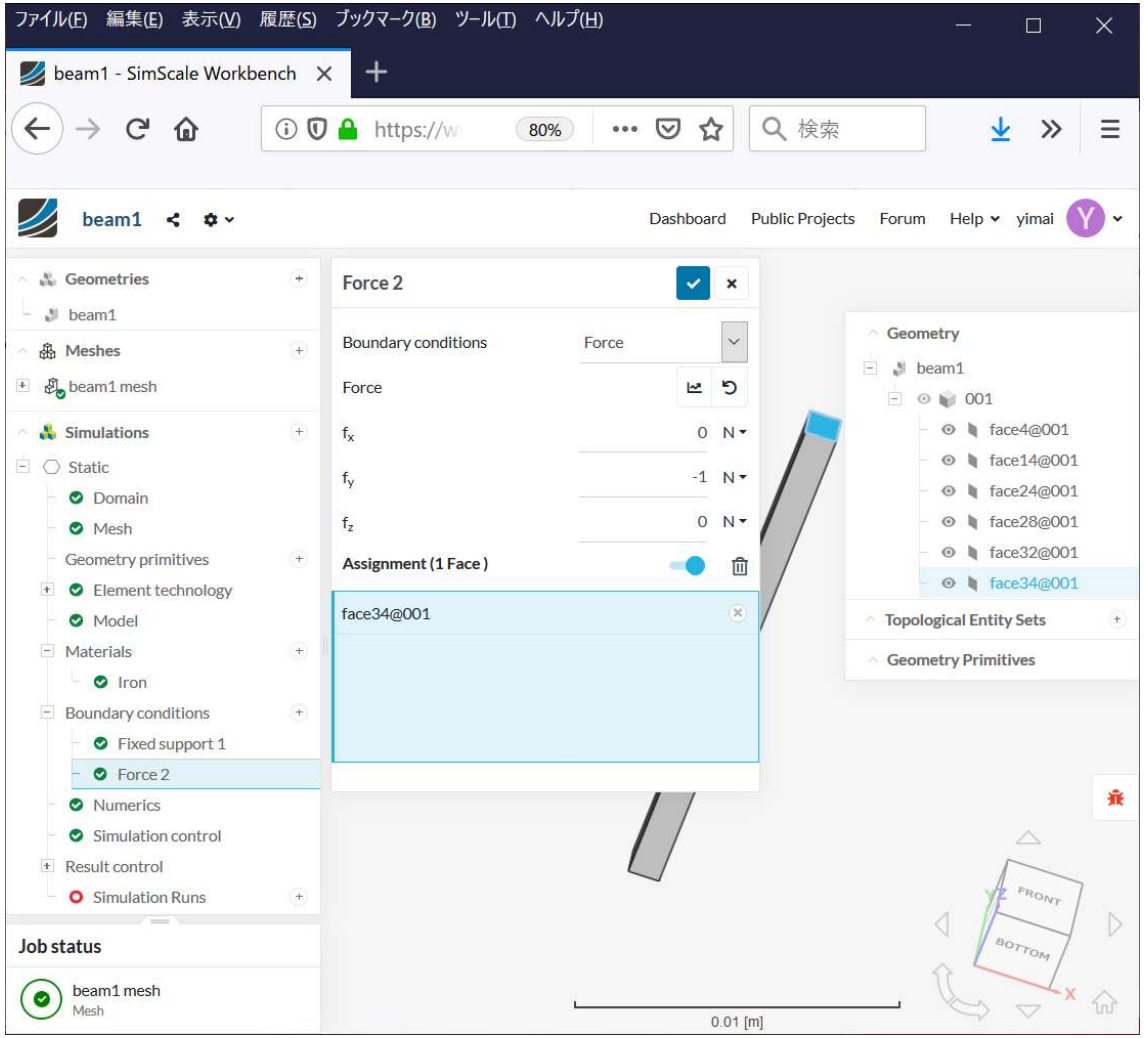
Boundary Conditions 1

Fixed Support (z=0 plane)



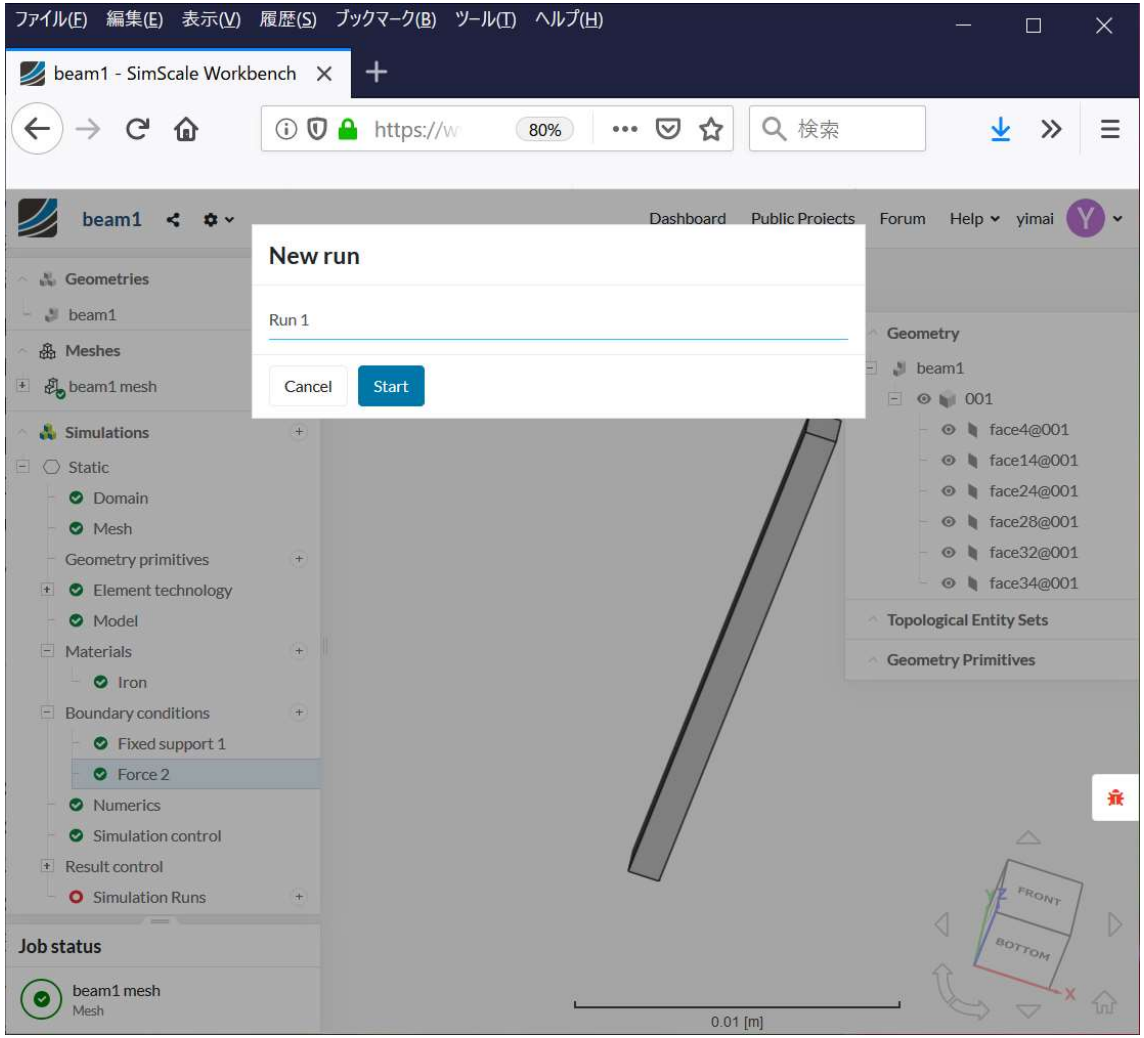
Boundary Conditions 2

Force (Fy=-1N, z=0.002 plane)



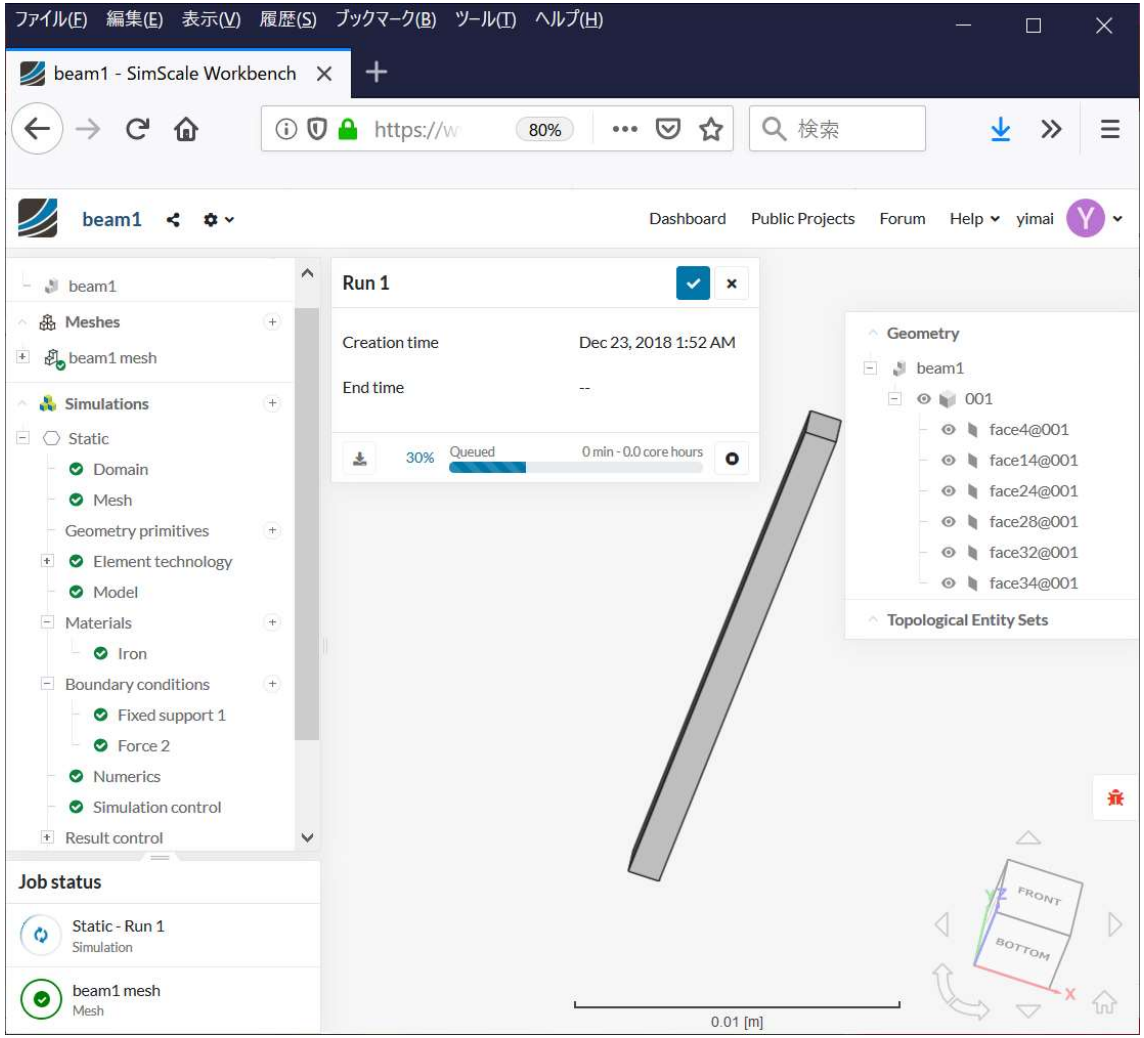
Simulation

Simulation Runs



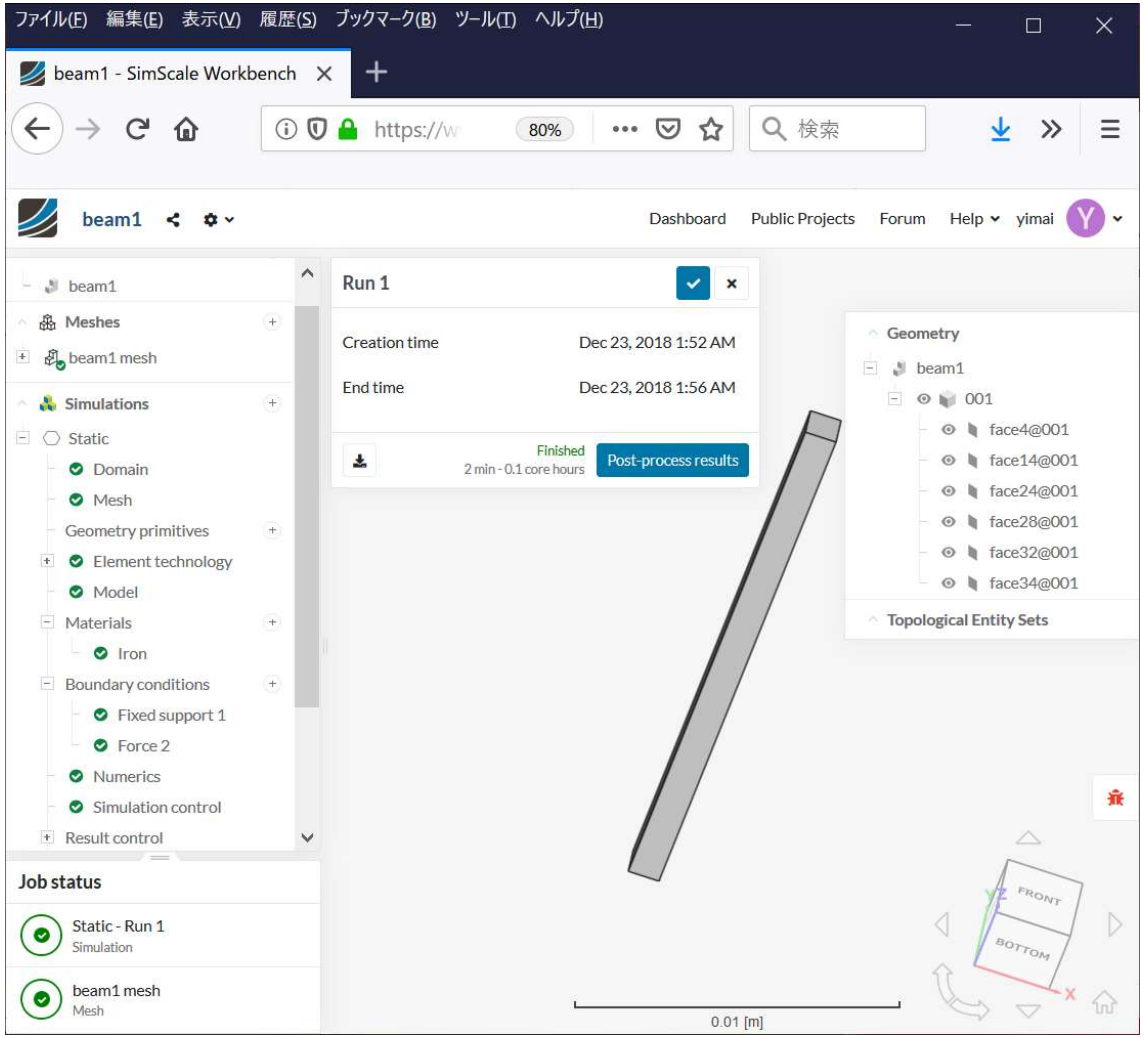
Simulation

Simulation Runs



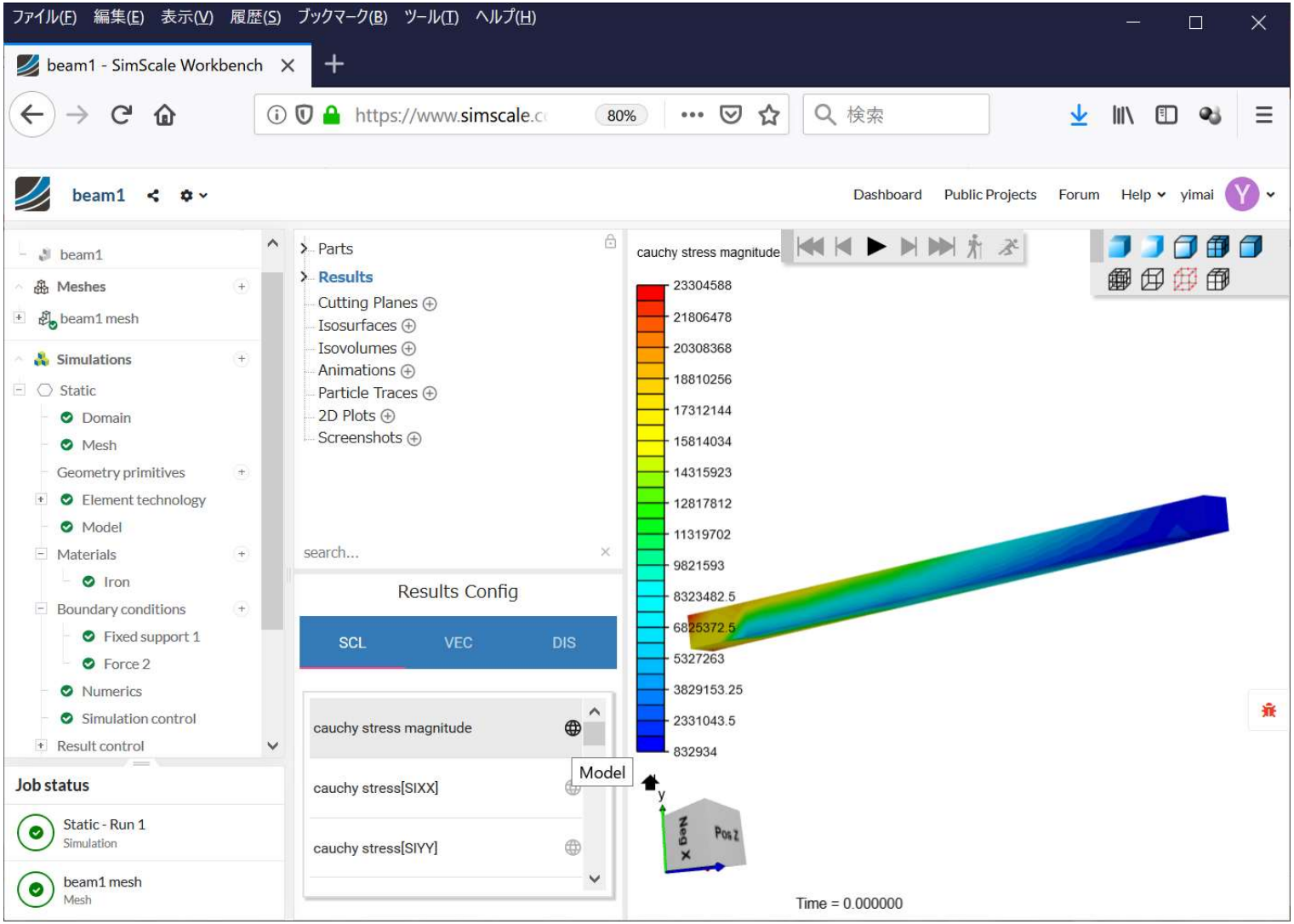
Simulation

Simulation Finish -> Post process



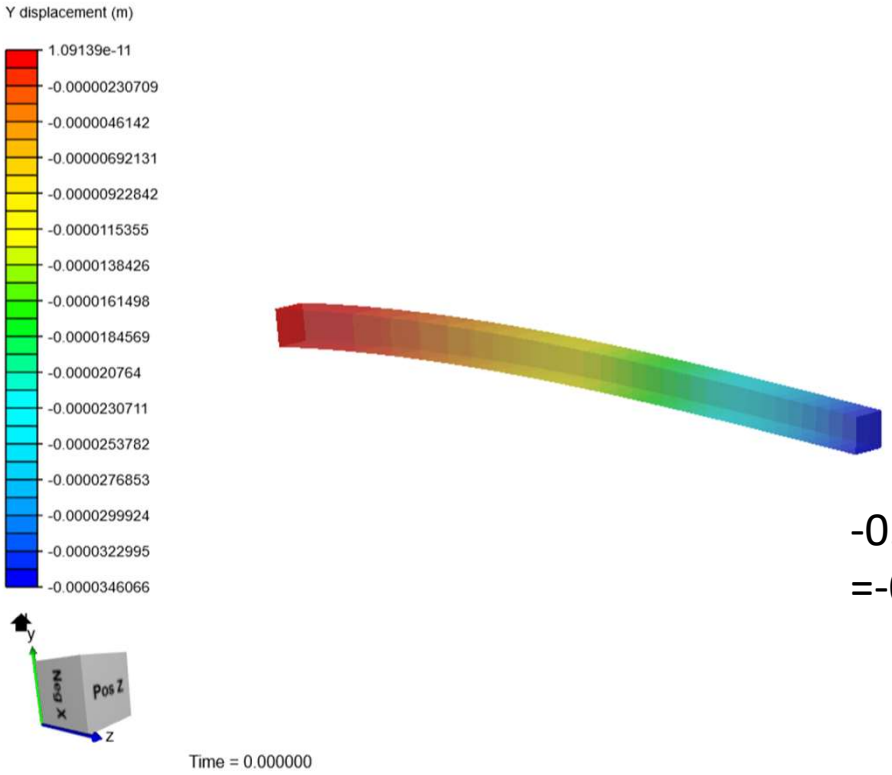
Post-Process

Stress at z = 0 is large. Stress at z=0.002 is small.



Post-Process

Y displacement at free end is larger than theory.

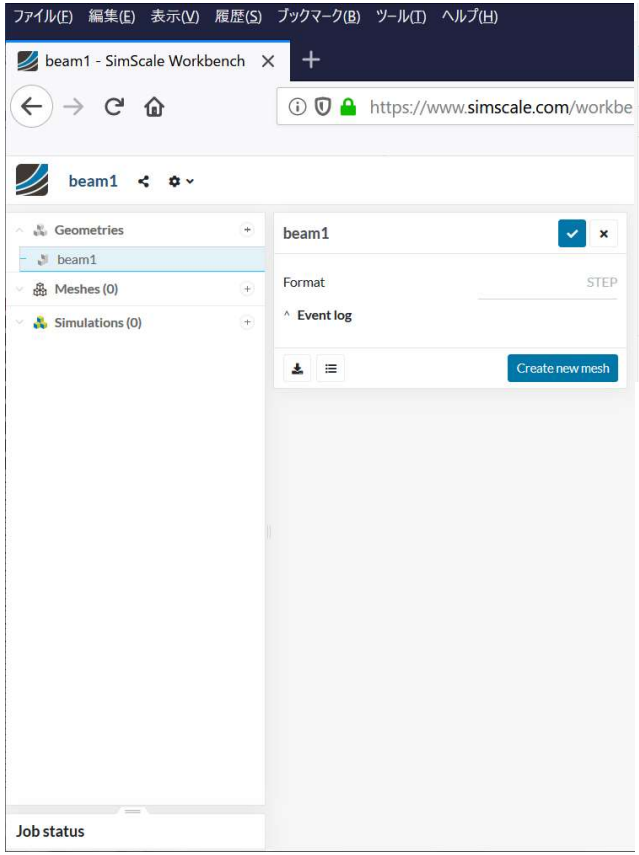


-0.0000346m
=-0.0346mm

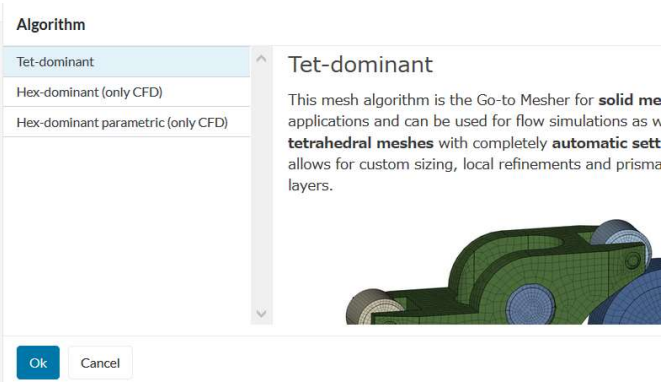
Re-calculate with smaller (fine) mesh.

Re-Meshing on SimScale

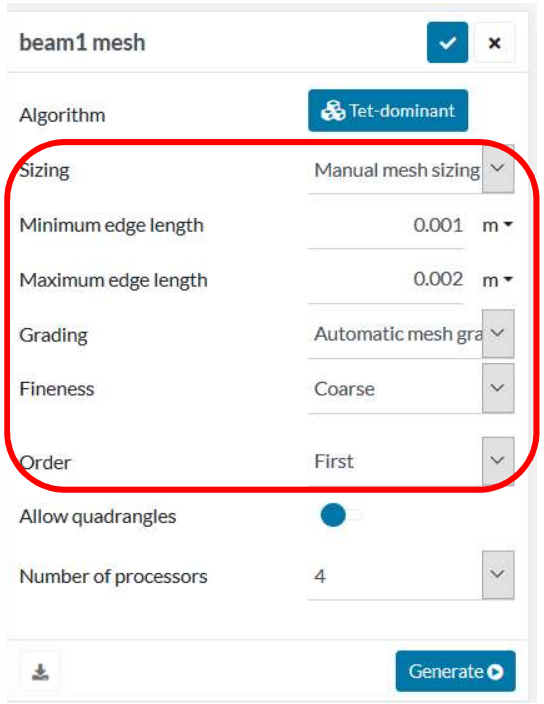
Create new mesh



Tet-dominant



Edge length



Change here

Mesh size and FEA

Mesh size	Small(fine)	Medium	Large(coarse)
Accuracy			
Time consuming			

It is important to identify the relationship between accuracy and mesh size at an early stage.