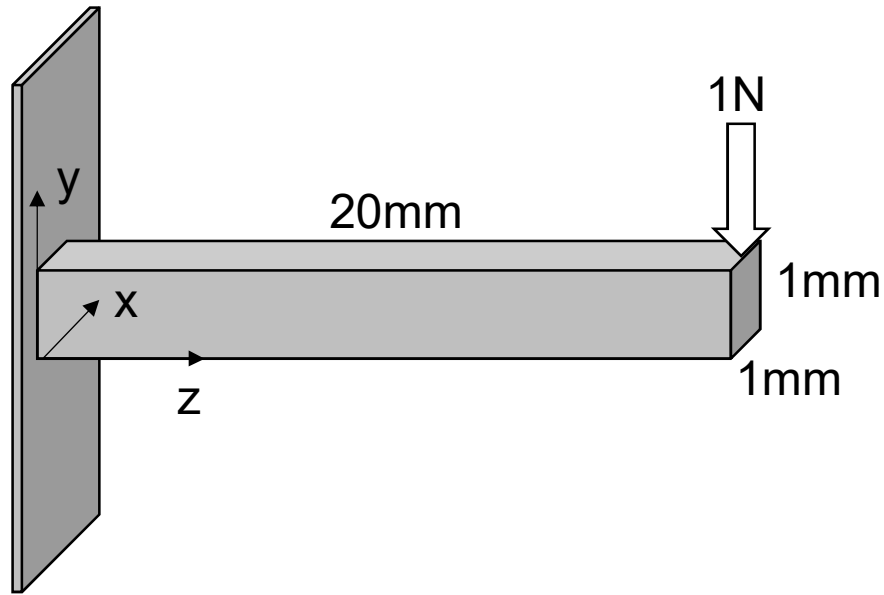


# Finite element analysis of beam bending

imaiy@cc.saga-u.ac.jp



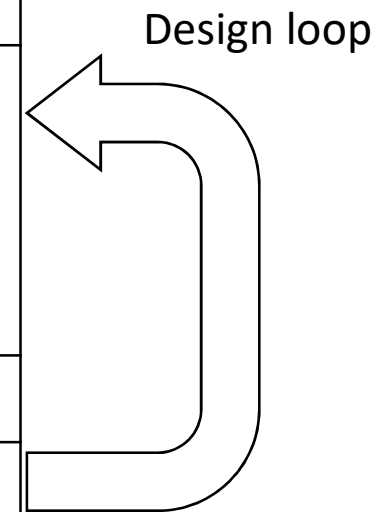
$$\text{Displacement} = \frac{P \cdot L^3}{3EI}$$

$$I = \frac{bh^3}{12}$$

Iron bar

P	N	1
L	m	20e-3
E	Pa	211e9
I	m <sup>4</sup>	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

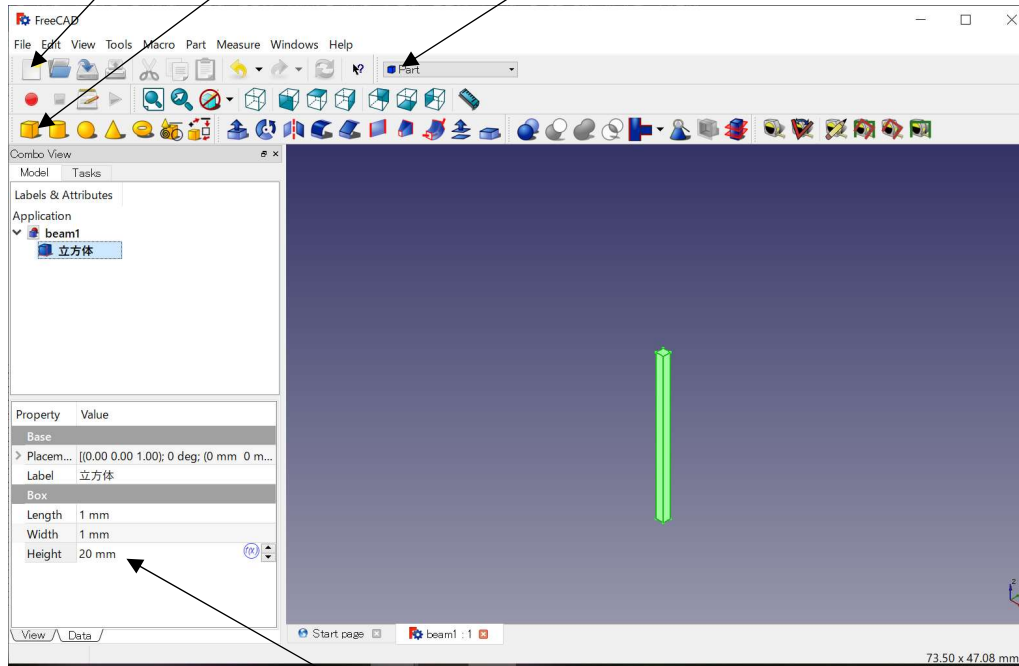


# Creating geometries on FreeCAD

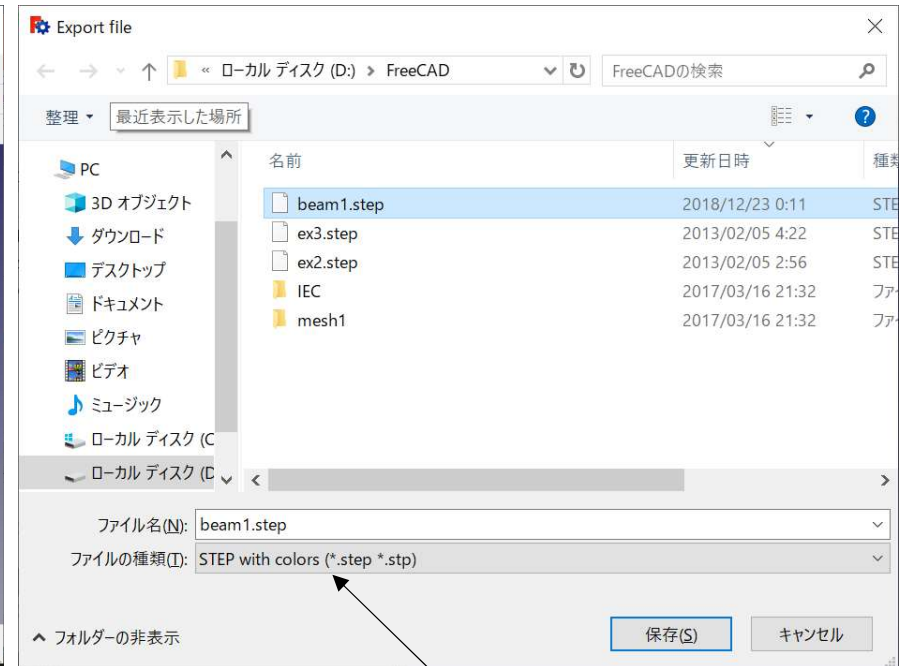
1:New

3:Cube solid

2:Part



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



5:File>Export  
Select STEP format

# Import geometries on SimScale

## New Project

Create New Project ×

beam1

Private Project  DISABLED [Upgrade my plan](#)

beam example

Choose a category:

- Validation
- Testing**
- Professional
- Learning & Teaching
- Other

Default unit system:

SI (Meter, Kilogram, Second, Kelvin)

Drag and drop your CAD files here  
or click here to add your files

Create project

## Geometries > Upload

beam1 - SimScale Workbench × +

← → ↻ 🏠 <https://www.simscale.com/workbench>

beam1 ⚙️

- Geometries (0) +
  - Upload**
  - Import from Onshape
- Meshes (0)
- Simulations (0)

Job status

## Select STEP file

Upload ×

Drag and drop your file(s) here  
or click here to add your file(s)

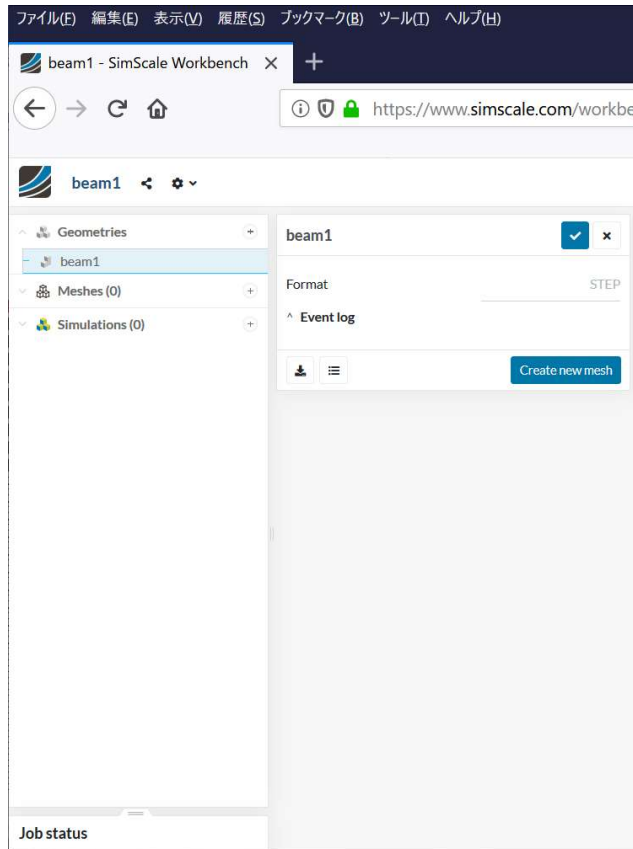
0% Pending

beam1  
STEP

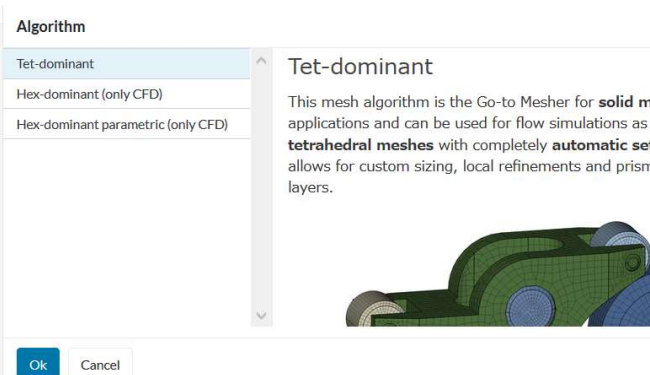
Upload

# Meshing on SimScale

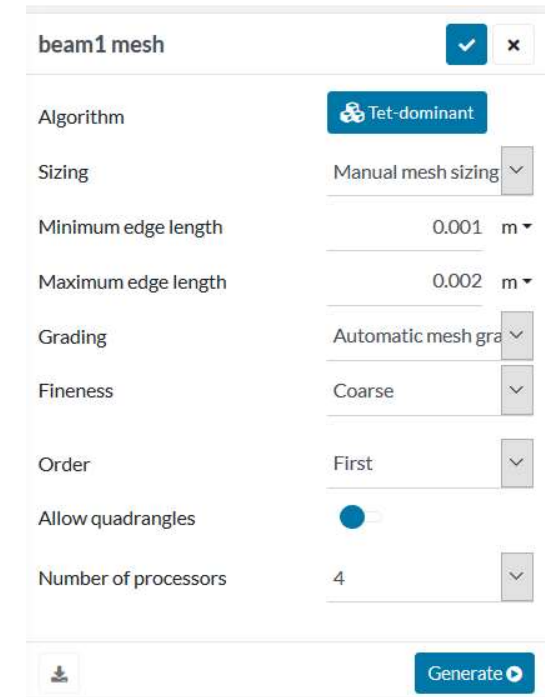
## Create new mesh



## Tet-dominant



## Edge length(Manual)



# Assign materials

## Static Analysis

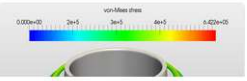
Analysis

- Static
- Dynamic
- Heat transfer
- Thermomechanical
- Incompressible
- Incompressible (LBM)
- Compressible
- Convective heat transfer
- Conjugate heat transfer

### Static Analysis

This analysis type is used to determine structures or components caused by the loads - inertia and damping effects are either linear or nonlinear. Types of non deformations, plasticity, hyperelasticity

Learn more...



0.00e+00 2e+5 3e+5 4e+5 5.00e+05

von Mises stress

Ok Cancel

## Materials (iron)

beam1 - SimScale Workbench

ファイル(E) 編集(E) 表示(V) 履歴(S) ブックマーク(B) ツール(T) ヘルプ(H)

← → ↺ 🏠 ⓘ 🔒 https://w 80% 🔍 検索 ⬇️ ⏪ ⏩ ☰

beam1 Dashboard Public Projects Forum Help yimai

### Iron

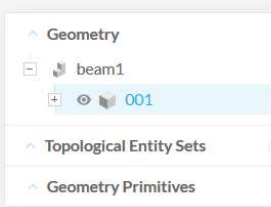
Material behavior: Linear elastic  
Directional dependency: Isotropic  
(E) Young's modulus: 211000000000 Pa  
(ν) Poisson's ratio: 0.29  
(ρ) Density: 7874 kg/m<sup>3</sup>

Assignment (1 Volume)

- 001

Job status

- beam1 mesh Mesh

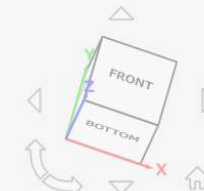


Geometry

- beam1
  - 001

Topological Entity Sets

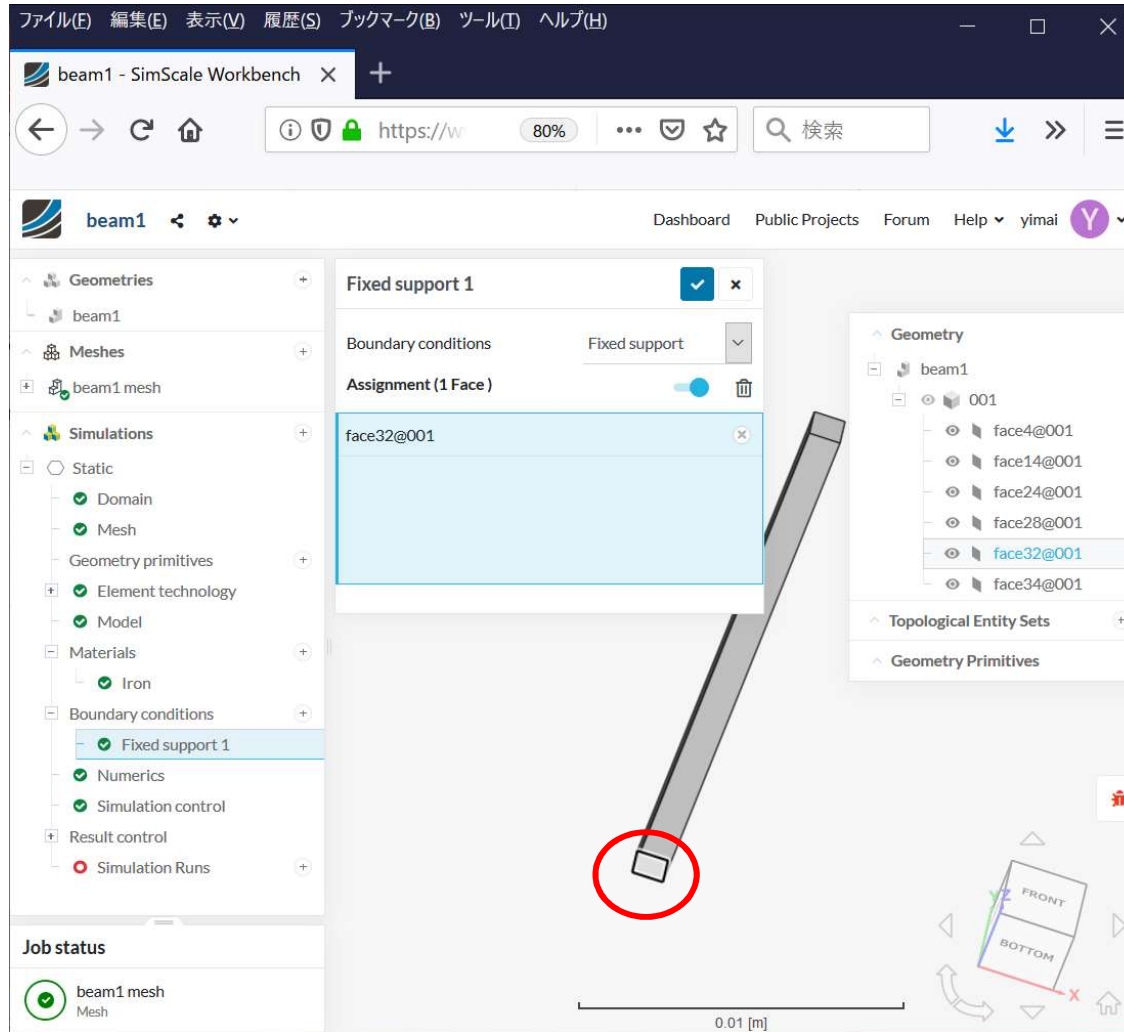
Geometry Primitives



0.01 [m]

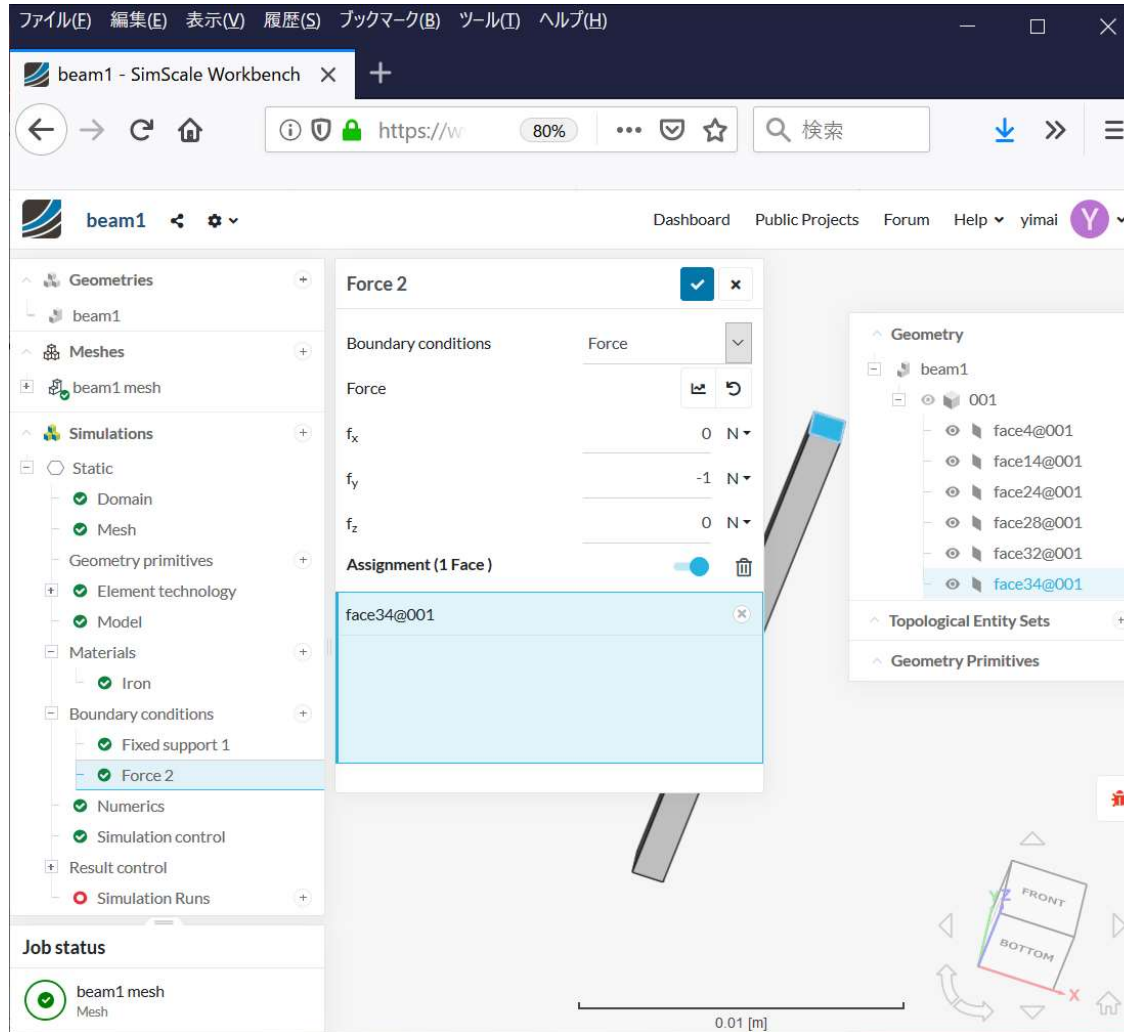
# Boundary Conditions 1

## Fixed Support (z=0 plane)



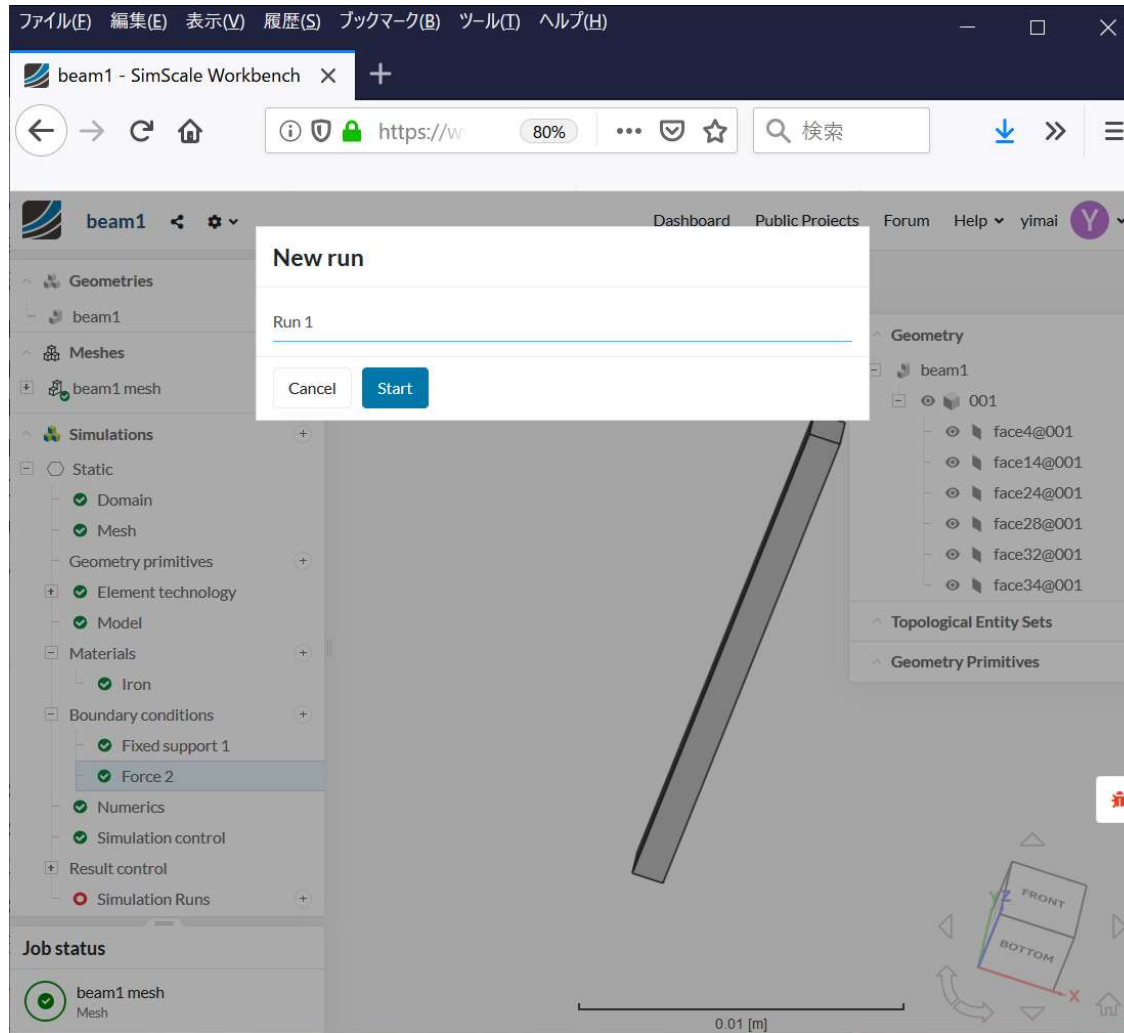
# Boundary Conditions 2

Force ( $F_y = -1\text{N}$ ,  $z = 0.002$  plane)



# Simulation

## Simulation Runs





# Simulation

## Simulation Runs

The screenshot displays the SimScale Workbench interface for a project named 'beam1'. The browser address bar shows the URL 'https://w' with a search icon and a refresh button. The SimScale logo and 'beam1' are visible in the top left, along with navigation links for 'Dashboard', 'Public Projects', 'Forum', and 'Help'. A user profile icon for 'yimai' is in the top right.

The left sidebar contains a tree view of the project structure:

- beam1
  - Meshes
    - beam1 mesh
  - Simulations
    - Static
      - Domain
      - Mesh
    - Geometry primitives
      - Element technology
      - Model
    - Materials
      - Iron
    - Boundary conditions
      - Fixed support 1
      - Force 2
    - Numerics
    - Simulation control
    - Result control

The main workspace shows a 3D model of a beam. A 'Run 1' window is open, displaying the following information:

Run 1	
Creation time	Dec 23, 2018 1:52 AM
End time	--
Progress	30% Queued 0 min - 0.0 core hours

The 'Geometry' panel on the right lists the following entities:

- beam1
  - 001
    - face4@001
    - face14@001
    - face24@001
    - face28@001
    - face32@001
    - face34@001

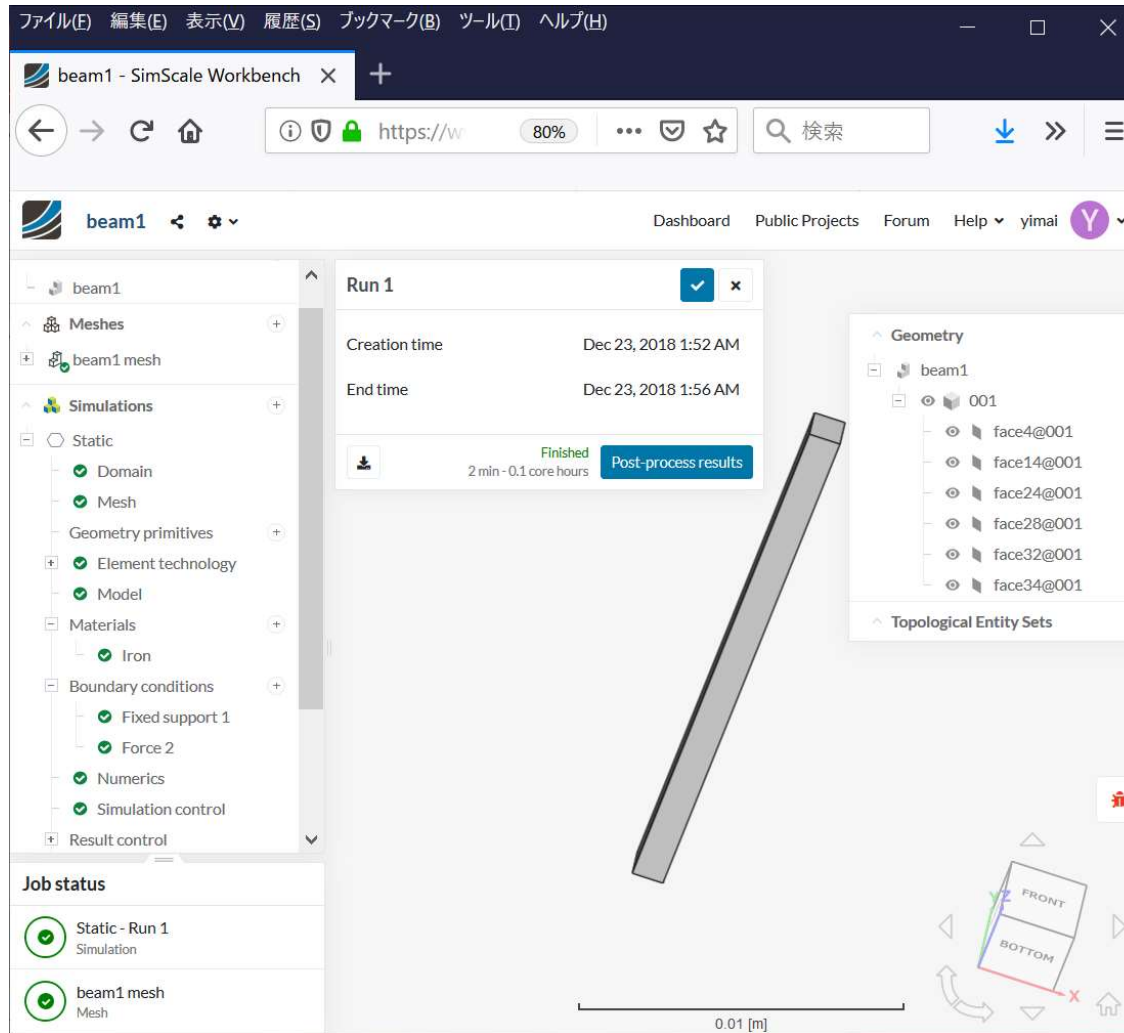
The 'Topological Entity Sets' panel is currently empty. The 'Job status' panel at the bottom left shows the following items:

- Static - Run 1 Simulation
- beam1 mesh Mesh

The 3D view includes a coordinate system with 'FRONT' and 'BOTTOM' labels, a scale bar indicating 0.01 [m], and navigation controls.

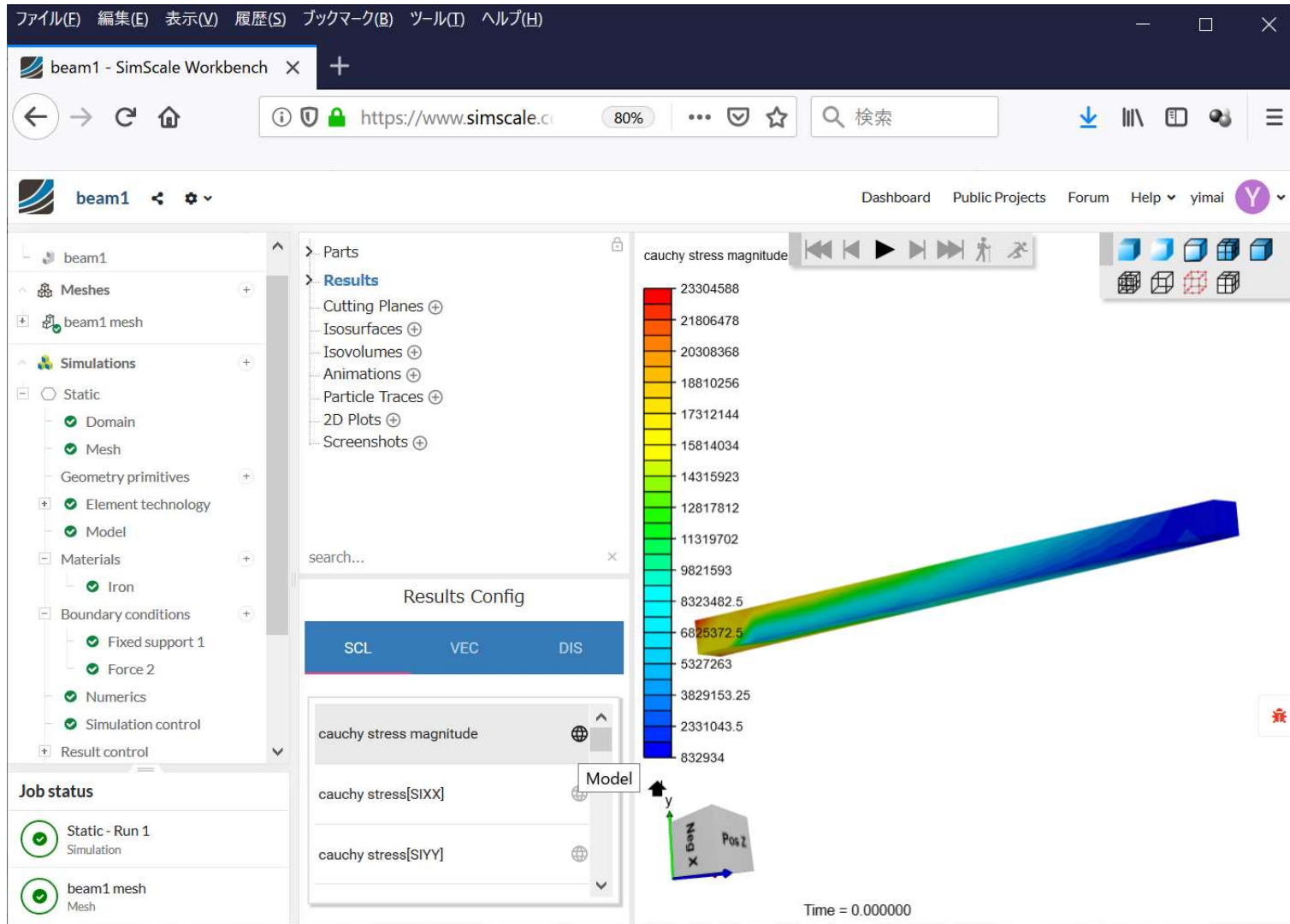
# Simulation

## Simulation Finish -> Post process



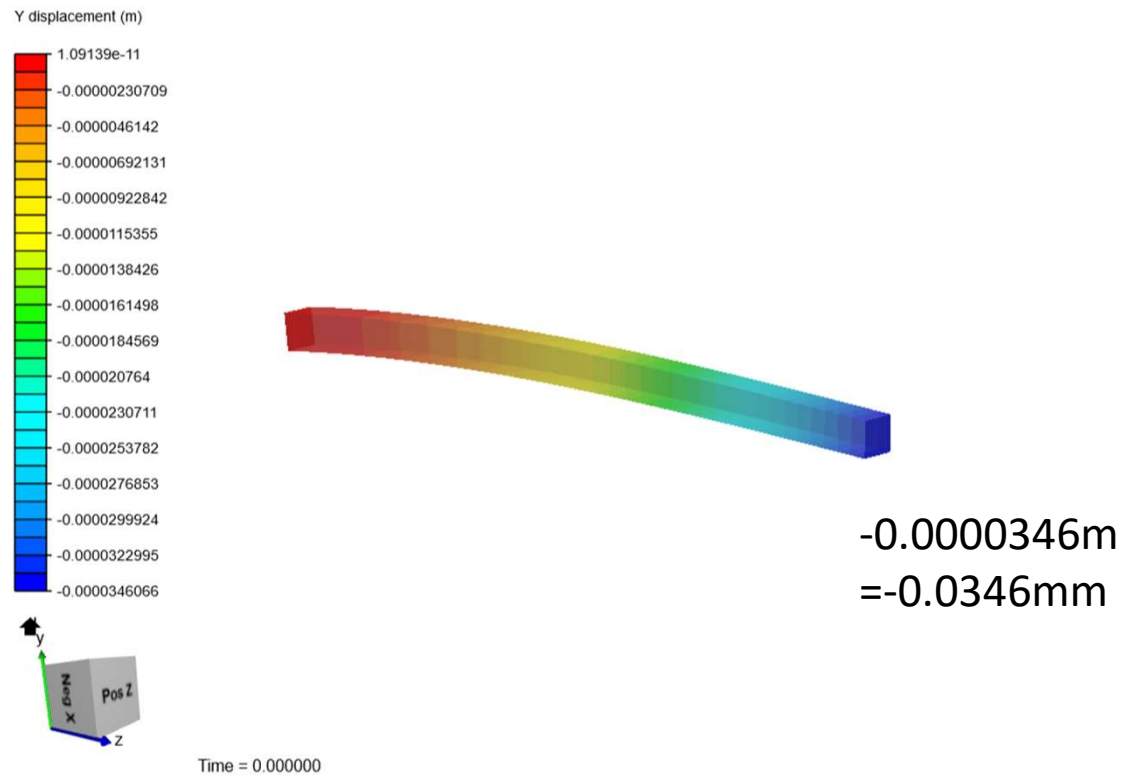
# Post-Process

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



# Post-Process

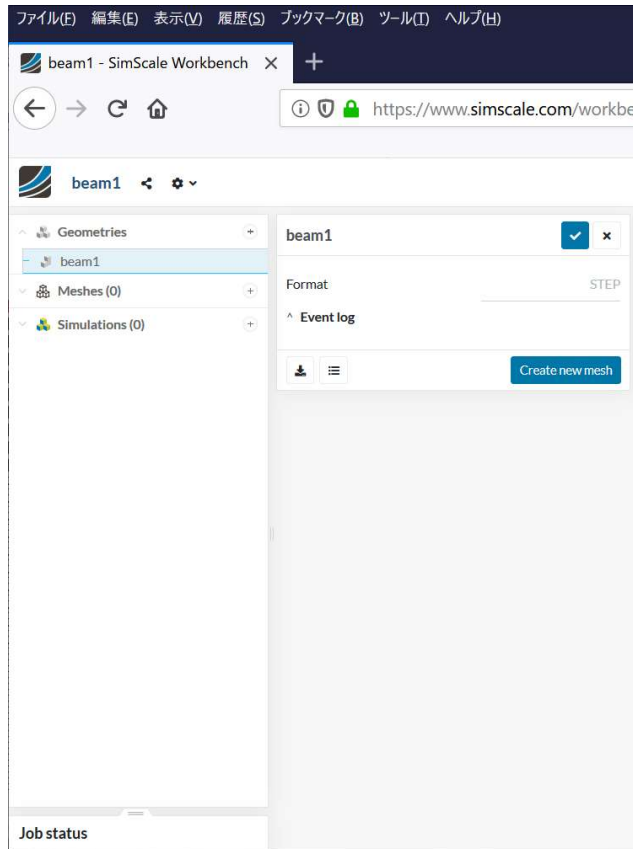
Y displacement at free end is larger than theory.



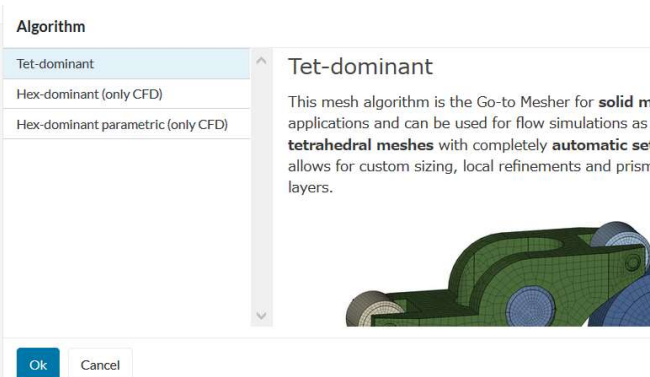
Re-calculate with smaller (fine) mesh.

# Re-Meshing on SimScale

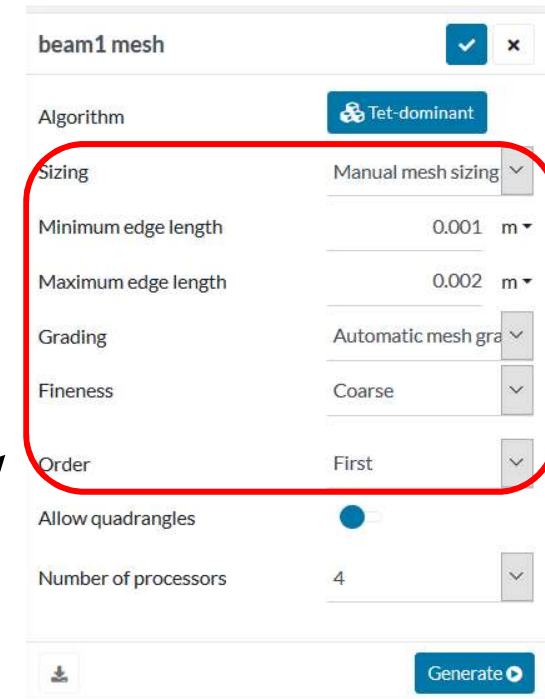
## Create new mesh



## 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.