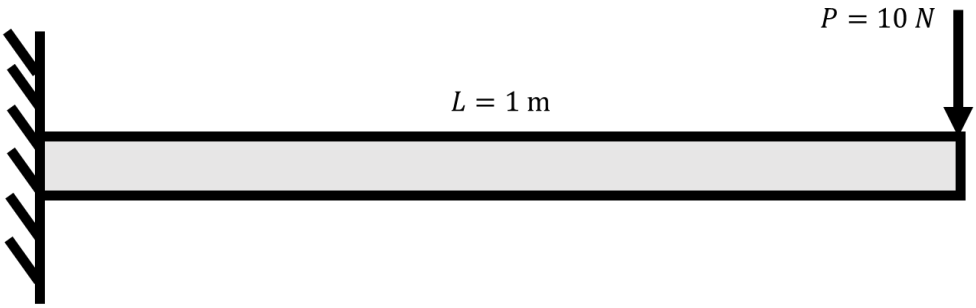
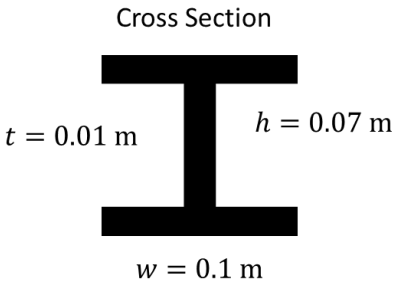




These step-by-step examples are broken up by module. Make sure you have the correct module selected for each part. Any text that is bold, indicates the prompt in the Prompt Area.

Part Module

1) Click Icon, Create Part (*A dialog box will pop up*)
   - **a) Fill out the Create Part dialog**
     - i) Modeling Space: 2D Planar
     - ii) Base Feature: Wire
     - iii) Approximate Size: 5
     - iv) Click Continue…
   - **b) Sketch the section for the wire**.
     - i) Click Icon, Create Lines: Connected
       - (1) **Pick a starting point for the line – or enter X,Y:** 0, 0
       - (2) **Pick an end point for the line – or enter X,Y:** 1, 0
       - (3) Click the Red X to cancel the procedure
     - ii) Click Done in the prompt area since we are done with our sketch

Property Module

1) Click Icon, Create Material (*An Edit Material dialog box will pop up*)
   - a) Name: Aluminum
   - b) Click Mechanical > Elasticity > Elastic
   - c) Young's Modulus: 70e9
   - d) Poisson's Ratio: 0.3
   - e) Click OK
2) Click Icon, Create Section
   - a) Create Section dialog
     - i) Category: Beam
     - ii) Type: Beam

iii) Click Continue…

b) Edit Beam Section dialog

i) Click Icon, Create Beam Profile 

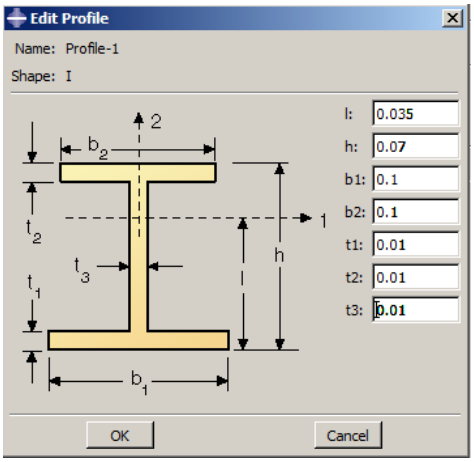
(1) Shape: I

(2) Click Continue...

(3) Fill in Numbers

(4) Click OK

ii) Click OK



3) Click Icon, Assign Section 

a) Select the regions to be assigned a section

i) Click on the part

ii) Click Done

b) Fill out the Edit Section Assignment dialog

i) Click OK

4) Click Icon, Assign Beam Orientation

a) Select the regions to be assign a beam section orientation

i) Click on the part

b) Enter an approximate n1 direction (tangent vectors are shown)

i) Leave as default: 0, 0, -1

ii) *This defines the orientation of the cross section. The 1 direction is defined in the Edit Profile dialog shown above.*

c) Click OK to confirm input

i) Click OK

d) Click Red X

Assembly Module

1) Click Icon, Create Instance (a dialog box will appear) 

a) Click OK

Step Module: The step defines the type of analysis you will run

1) Click Icon, Create Step 

a) A "Create Step" dialog box will appear

i) Ensure "Static, General" is selected

ii) Click Continue…

b) A *"Edit Step" dialog box will appear*

i) Click OK

Skip the Interaction Module (we only have one part so it does not interact with other parts)

Load Module

1) Click Icon: Create Load 

a) Fill out the Create Load dialog

i) Types for Selected Step: Concentrated force

ii) Click Continue...

b) Select points for the load

i) Click on the far right point of the part

ii) Click Done in prompt area (or click middle/scroll button)

   c) Fill out the Edit Load dialog
      i) CF2: -10 (CF2 stands for concentrated force 2-direction_
      ii) Click OK
2) Click Icon: Create Boundary Condition 
   a) Fill out the Create Boundary Condition dialog
      i) Types for Selected Step: Displacement/ Rotation
      ii) Click Continue…
   b) Select regions for the boundary condition
      i) Click the left point
      ii) Click Done in prompt area (or click middle/scroll button)
   c) Fill out the Edit Boundary Condition dialog
      i) Check U1, U2, UR3 boxes (corresponding to a clamped BC)
      ii) Click OK

Mesh Module
1) To the right of the Module, next to Object, select Part (instead of Assembly)
2) Click Icon, Seed Part 
   a) Set the data using the Global Seeds dialog
      i) Approximate global size: 0.05
      ii) Click OK
   b) Seeding definition complete
      i) Click Done in prompt area (or click middle/scroll button)
3) Click Icon, Mesh Part 
   a) OK to mesh the part? Click Yes

Skip the Optimization Module

Job Module

1) Click Icon, Create Job 
   a) Click Continue…
   b) Click OK
2) Click Icon, Job Manager  (Top one)
   a) Click Submit
3) Wait for job to finish running, in the Message area you will see "Job Job-1 completed successfully."
4) Click Results to transition to Visualization Module

Visualization Module

1) Click Icon, Plot Contours on Deformed Shape 
2) Change size of Legend Text
   a) Viewport > Viewport Annotation Options
   b) Legend tab > Set Font… > Size 14
3) Visualize the Cross Section
   a) View > ODB Display Options…
   b) Check Rend beam profiles
   c) Click OK
   d) *Notice that the stresses are constant for each cross-section. If you are interested in how the stresses vary throughout the cross section, you must use a 3D model.*