Abaqus – exercise 3
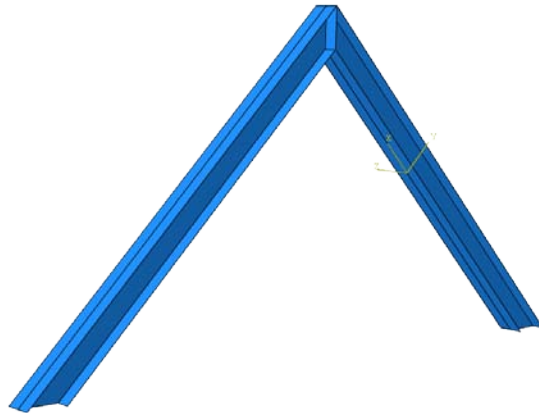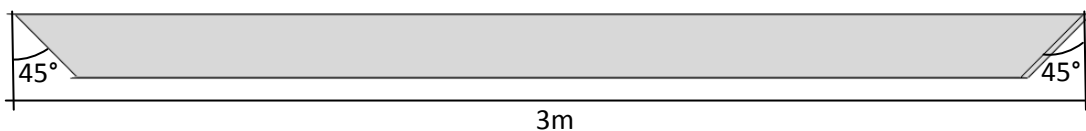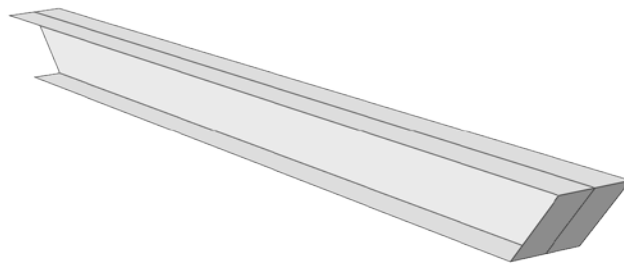
The structure below is going to be analyzed.

First make a model of the I-shaped beam below, (h=200mm, w=200mm, t=9mm, L=3m). There is a 9mm thick plate at one end of the beam.

45°   45°

3m

Create Part -> 3D, Shell, Planar -> Draw the base shape (The web of the beam), (Livet på balken)

To create the flanges:

First: create a datum plane that lies where the flange is going to be placed.

Then: *Create Shell: Planar* and draw a flange of the beam on the created plane.
Do this procedure for both flanges.

Property:
Create a steel material E=201 GPa, v=0.3
Create a shell section, t=9mm
Assign section

Assembly:

Instance the part two times. At the second time, click in the box "Auto-offset from other instances".

Use position constraints to position the two parts so that the ends with a plate lie to each other. See the figure for position constraints.
There is only need for the "Edge to Edge" constraint.
Assume one of the parts as being fixed and the other movable.

F=30 kN
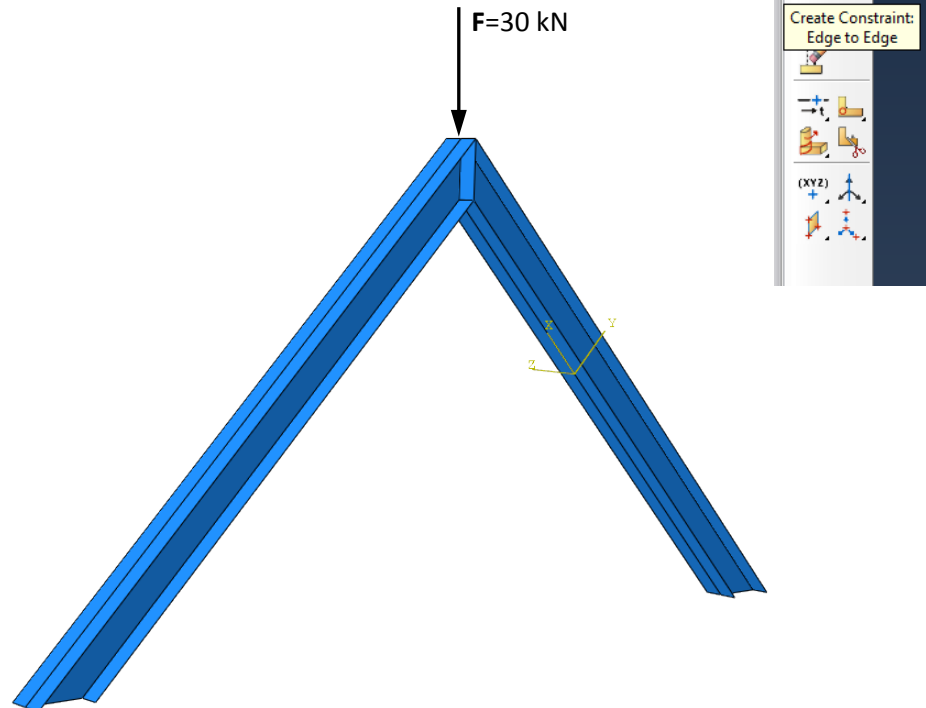
Step:
Create Static, General step

Interaction
At the assembly step the two parts were positioned but they are still unconnected. How they will interact is given in the "Interaction module".
First note that the visibility of the parts can be turned on and off in: "View -> Assembly Display Options -> Instance" dialogue box.

To create a full tied interaction:
1. "Create Constraint -> Tie -> Continue.."
2. Turn the visibility of one of the parts off.
3. Select surface for the master type.
4. Select the two areas on the visible part where the parts meet to be the master surface.
5. Choose brown or purple for the side of the surface pointing towards the other part.
6. Turn the visibility of this part to off and the other part to on.
7. Select surface for the slave type.
8. Select the two areas on the other part where the parts meet to be the slave surface.
9. Click OK in the Edit Constraint dialogue box.
10. Turn the visibility of both parts to on.

Load:

Create the point load and add fixed displacements at the two bottom ends.

Mesh:

1. Make mesh on Part
2. Assign Mesh Controls -> Element shape = Quad; Technique = Structured
3. Assign Element Type: Geometric Order=Linear; Family: Shell elements
4. Seed Part
5. Mesh Part

Create job -> Submit job -> Results -> Visualize