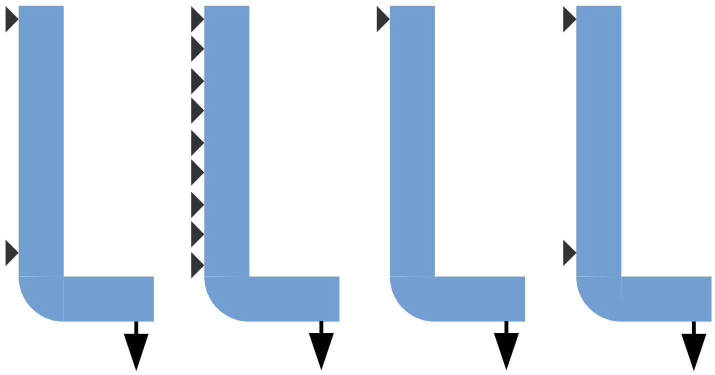
Effects of Constraint on Stress

Consider the following angle brackets. Each shares the same geometry but is constrained in a slightly different way.



In all cases except the fourth, the triangle indicates an `ENCASTRE` boundary condition. In the fourth, `ENCASTRE` the upper anchor but allow rotation around the lower anchor. The load will be 650 N.

Module *Part*

Create a `2D`, `deformable`, `planar shell` part. Name this part `Bracket`. Sketch a part with width 5 cm, height 30 cm, and projecting width 15 cm. (Select and use a consistent unit system throughout.)

Module *Property*

We will utilize our typical steel:

Material Property & Value	Name
$E = 2\times10^{11}$ Pa	Young's modulus
$\nu = 0.3$	Poisson's ratio
$\rho = 7800$ kg·m^{-3}	density

1. Create a section named `BracketSection` of category `Solid` and type `Homogenous`. Assign material `Steel` and the currect profile `BracketProfile`.
2. Assign section `BracketSection` to the bracket.

Module *Assembly*

1. Create a dependent instance of the part named `Bracket`.
2. Create geometry sets as necessary to apply the boundary conditions above.

Module *Step*

One step (each) is required:

Step	Details
`General`	Apply a 650 N tensile force.

1. Create a step of the type `General; Static, General`.
    1. Name this step `Load` with the following description: `Apply load of 650 N`.
    2. In the *Edit Step* dialog box, include the effects of geometric nonlinearity by toggling on `Nlgeom`. Specify *Time period* of `1` and *Incrementation→Initial increment size* of `0.1`.
2. The next default step is `Output requests`. In this case, the default output database output requests created by Abaqus/CAE for each step will suffice and we do not need to create any additional output database output requests. If you do need to request output to the restart file:
    1. In the *Main Menu*, select *Output→Restart Requests* from the *Main Menubar* of the *Step* module.
    2. Write data to the restart file `Every 1 increment`.

Module *Load*

1. In the first step, create a `Mechanical Concentrated` force named `Force` that applies a `-250` N tensile force to the proper set for the anchor.
2. Clamp or restrain the bracket as appropriate for the anchor points.

Module *Mesh*

1. Seed and mesh the bracket section with appropriately sized `Quadratic plane stress elements` (`CPS8`).
2. Create jobs as appropriate.
3. Save your model in a model database file, and submit the job for analysis.

Module *Job*

You can open the *Job Monitor* in the *Job Manager* dialog box. Here you can see the *Log*, *Errors*, *Warnings* and other output of the `Bracket-`*`n`* job.

Module *Visualization*

1. Enter the *Visualization* module, and open the output database, `Bracket-`*`n`*`.odb`, created by this job.
2. Plot deformation and von Mises stress as appropriate.

For your submission, include screen shots and report the value and location of maximum von Mises stress in each.