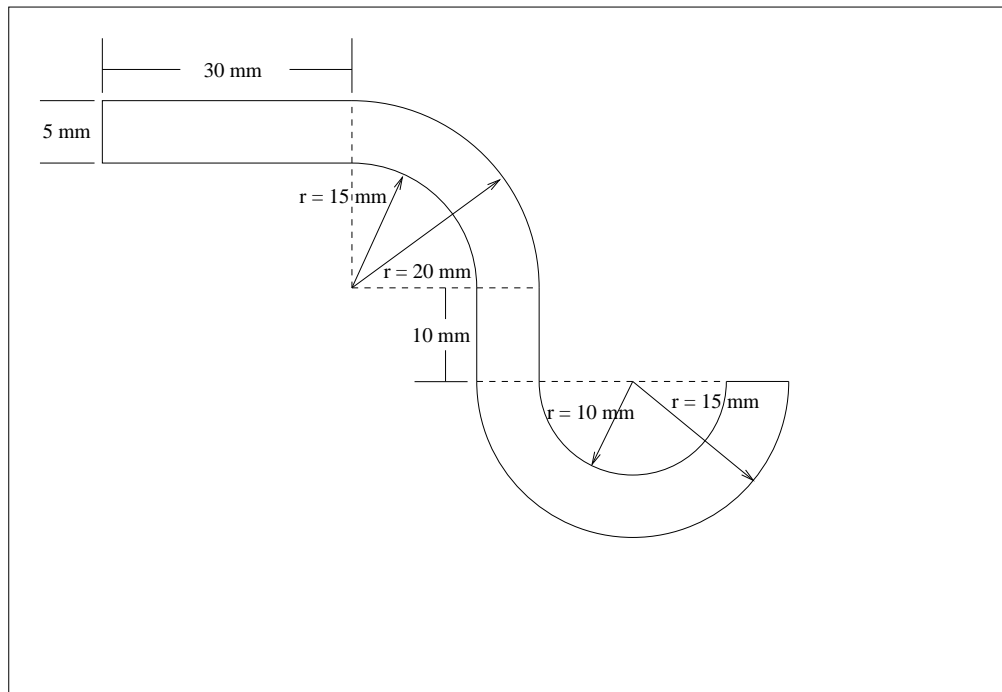


HOMEWORK 9: VEHICLE BRACKET ANALYSIS IN COMSOL

Please read the entire assignment before you start generating the model, this will help you plan each step of the process. Consider a hanger bracket used to support an exhaust pipe that is directly after a catalytic converter. A dimensioned schematic of the bracket is shown below. The bracket is 40mm wide, and extends out the page. The drawing plane is the x-y plane, and the extrusion is in the z-direction. (Hint: this is how you should generate the geometry and mesh)

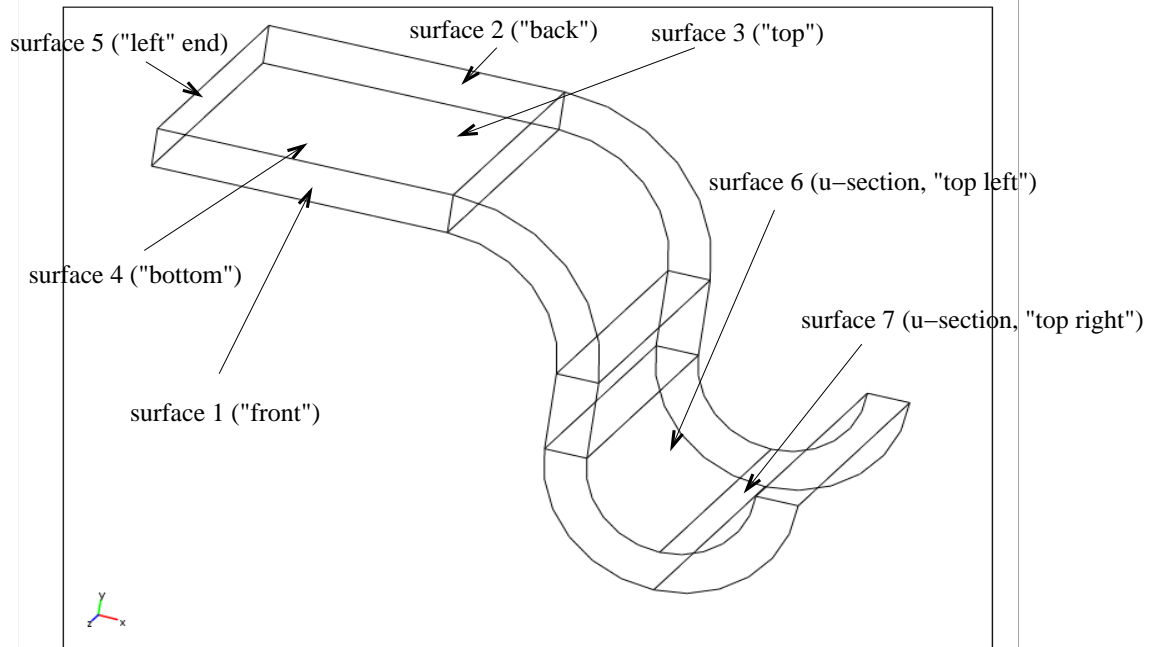


The material properties for AISI-302 stainless steel, which is used to make the bracket, are given below:

- Young's modulus, $E = 195 \times 10^9 Pa$
- Poisson ratio, $\nu = 0.30$
- Coefficient of thermal expansion, $\alpha = 17 \times 10^{-6} 1/K$

- Thermal conductivity, $k = 17W/(mK)$

The bracket is mounted to the frame of the vehicle, near the engine block, and supports a pipe in the curved u-section. Heat is transmitted to the bracket through the frame and contact with the pipe. The pipe will oscillate during the operation of the vehicle, but we will only focus on the maximum load generated during the cycle (which makes the problem static). The surfaces that are affected by the thermal and structural loads are labeled in the wire frame schematic below.

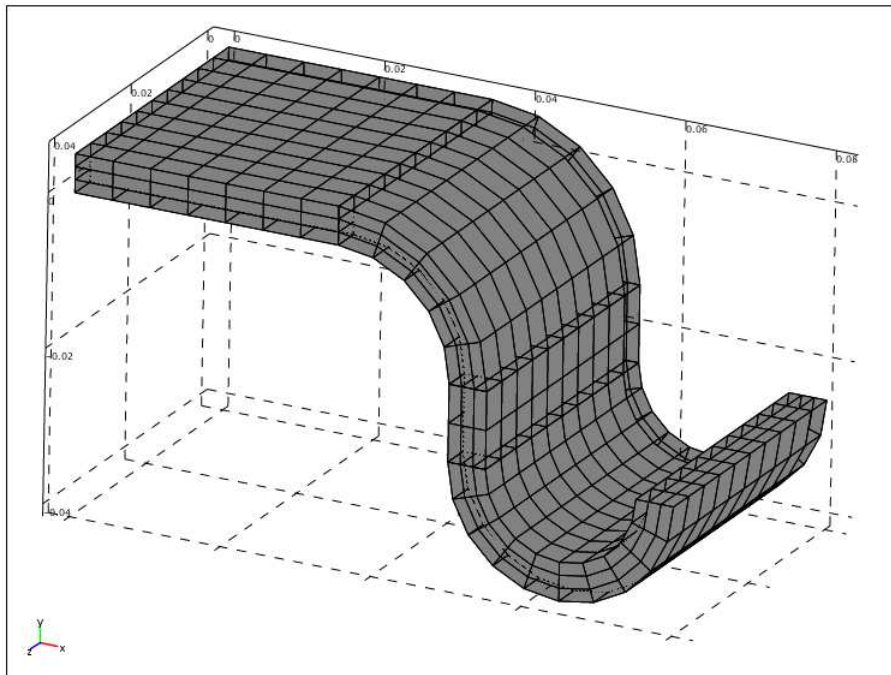


The boundary conditions that correspond to the physical system, as described above, are given below. The reference temperature of the bracket is 298K (which is set in the “Load” tab of the structural subdomain settings).

- Thermal
 - Surfaces 1, 2, 3, 4, 5 apply a temperature of 320K.
 - Surfaces 6 and 7 apply a temperature of 400K.

- All other surfaces leave as thermally insulated.
- Structural
 - Surfaces 1 and 2, fix z-direction.
 - Surfaces 3 and 4, fix y-direction.
 - Surface 5, fix x-direction.
 - Surfaces 6 and 7, apply a face load of -10000 N/m^2 in the y-direction.

Generate a mesh using trilinear elements (this is done using the “mapped mesh” in COMSOL). A sample coarse mesh is shown below, which contains 720 elements. You should generate a mesh that contains no less than 1000 elements but no more than 3000. You have control over how many elements are along each edge. You can choose to have more elements in certain regions if you deem it necessary.



- Solve the problem described above, using the appropriate mode in COMSOL, for a coupled thermal - structure problem.
- Generate a plot of your mesh and state how many total elements there are. Also, discuss if you put more elements on a particular region.
- Generate subdomain plots of the von Mises stress, the temperature, and z-direction displacement. Include these in your report.
- Where is the maximum stress and what is the value? Please include this in your report.
- What do you think causes the stress to occur where it does (there are several factors to consider)? What can be done to reduce the stress (again, there are several things)? Try to come up with a reasonable explanation.
- One thing to consider are the structural boundary conditions. Think of how they might be adjusted to reduce the stress. Try to rerun the problem with an adjustment and indicate what happens to the maximum stress, in terms of the value and location. Comment on this in your report.