The assignment required that a program be written to solve the projects finite element problem, however several simplifications had to be done to the original goal of modeling a gasket between a flange. Ultimately it was decided to model a two-dimensional cross section of a flange, gasket and bolt, and replace the contact stresses with equivalent pressures. In addition to being able to more directly leverage techniques taught in the course, it will provide an interesting comparison to the more thorough and complex model of Abaqus.

MATLAB was chosen as the platform to create the FEM program with. The two-dimensional cross section of the flange, gasket and bolt were created in GMSH and a mesh was generated with 3-node triangular elements. Combined with the material properties, the local stiffness matrices were derived. Instead of modeling the contact stresses, equivalent pressures were calculated with equations and values from the manufacture of the flange:

<TODO actual pretty equations>

Equivalent Pressure = (4\*BoltForce/EffectiveGasketDiameter \*\*2+ 16 \* Bending Moment on Flange/EffectiveGasketDiameter\*\*3) / 6894.7

With the pressures in place, the local load vectors were generated. The global stiffness matrix was assembled with MATLAB’s sparse matrix, and the global load vector was assembled. The boundary conditions were hard-coded into the program, eliminating rows and columns for node elements that were fixed or where the axis of symmetry cut the cross section. The displacements of the remaining nodes was found by inverting the global stiffness matrix and multiplying it with the remaining load vector. The reaction forces and strains where then computed and plots of the stress and displacements were generated. It was also reported out if the gasket had failed (exceeded its critical stress?).

Once the routine was completed, it was used to model the loading and unloading of the flange. To accomplish this, the FEM routine evaluated a quarter of the overall load. The displacements were found and applied to all of the nodes in the original mesh, which was then used to define a new mesh. Then another quarter of the pressure load was added and the displacements were found again, and this was repeated until we reached the maximum load. Then the pressure was stepped down in a similar manor. These results showed that the gasket <will|won’t> hold, and they <were reasonable close to|were completely off from> the Abaqus result.

We then attempted to model the effects of temperature, but…