Example 2/5

Courtesy of Justin Jones

An example of writing that reflects the understanding of finite element theory.
Also, an example of writing figure captions.

of these will be further explained as well as the specific parameters used for this report in the following passages.

Geometry

One of the earliest decisions an engineer faces when modeling is how to choose the appropriate geometry. This may be an obvious decision based on structural requirements that need to be modeled as three dimensional solid bodies, but this may not always be necessary. Some times it may be advantageous for the engineer to model the part as a two dimensional planar object.

Two dimensional planar models are limited to the ones that have lengths long enough or a thickness small enough, such as sheet metal, that allow them to be analyzed using the plane theory of elasticity. There are two general planar methods, plane strain or plane stress. Plane stress is a state of stress in which the normal stress and shear stress in planes normal to the principal plane are zero [5]. Plane strain is the state of strain where the normal strain and the shear strain normal to the principal plans are zero [5]. This type of simplification may greatly reduce the computational tax necessary to model the part and should be used when appropriate.

Another simplification that can be utilized when modeling is symmetry or anti-symmetry. Often when modeling objects will be very complex or large that the mesh will result in a model that will be computationally heavy. In order to reduce this computational tax there are several different types of symmetry, such as mirror, axial, cyclic, and repetitive that can be used. By using symmetry the number of elements necessary to model the object will be significantly reduced as well as the degrees of freedom, and as such care has to be taken with the boundary conditions and applied loads [5]. For example if there is mirror symmetry about the y axis, translation in the x direction where the part interfaces with the axis of symmetry must be fixed. It would also be necessary to fix rotation about the y axis.

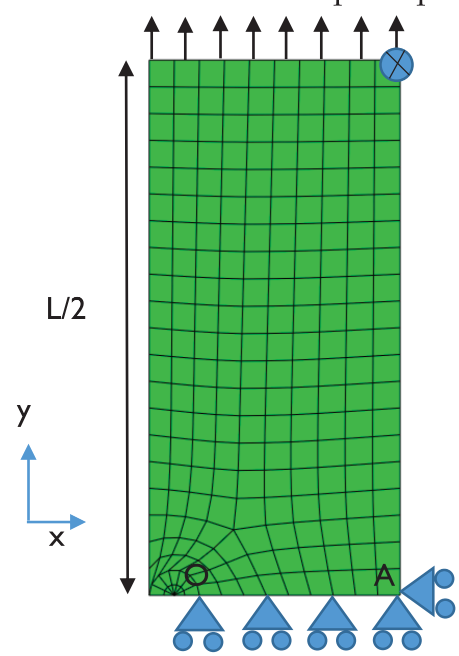

Figure 1. Basic Geometry, BC and Loading of part. The axis of symmetry is fixed in the y-direction, and free along the crack length. There is a 450MPa Pressure load across the top. To restrict translation the x-direction is fixed on the node across from the crack and the z-direction is fixed in the top right corner to restrict rotation about the x axis.

Briefly, a specific example that worked for this report, a two dimensional plane strain geometry with mirror symmetry about the x axis was used. This is due to the thickness being a factor of three less than the length. Also, an isotropic elastic material with a modulus of elasticity of 68.9 GPa and poisons ratio of 0.33 was used for the material properties making the bar model the behavior of aluminum 6061 T6. The overall dimension were a height of 200 mm, width of 95 mm, and a crack length of 9.5mm.

Mesh

After defining the parts geometry, interfaces, and material properties the next step is to select an appropriate mesh. Some considerations include which type of element to be used. For a two dimensional object the options include quadrilateral elements and triangular elements. These describe the shape of the elements themselves and should be chosen based on geometry to result in good meshes. Then there is the option whether or not to make the shape function of the elements linear or quadratic. A linear element is one that's shape function changes linearly between nodes. Where a quadratic element would have a quadratic shape function. The quadratic shape function would result in a higher order approximation of the solution and should result in a smaller error, as long as all the parameters are sufficiently smooth [5], but again would require more computational power to solve.

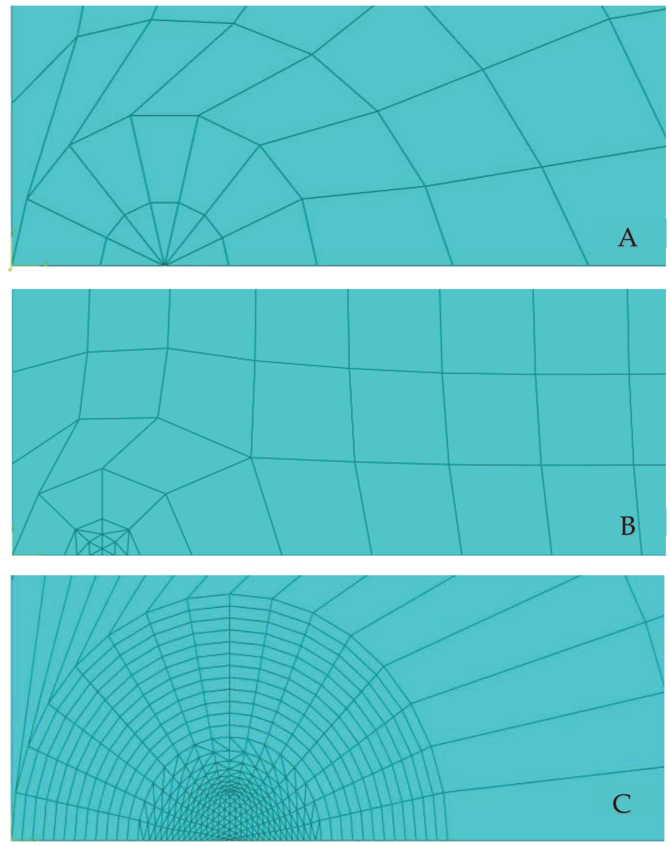

Figure 2. Process of refining the mesh at the crack tip. A. is a course mesh, B. is medium mesh, and C is a fine mesh.

When the type of elements have been selected it is typical to run a course mesh first. This is beneficial to gain quick results especially for complex models. Once initial results are generated the engineer can refine the mesh in regions of interest, such as areas with stress concentrations like fillets, holes, and cracks. A finer resolution of the mesh will result in a better approximation of the solution. However, again there is a trade of in computational time since there will be more equations to solve for. With ever finer meshes the approximate solution will converge on the analytical solution [5]. One method to ensure convergence is called a patch test, which was not performed for this project.

For the edge crack problem for this report two half circle partitions were made with the center at the crack tip and a radius of 2mm and 9.5 mm in order to have more control over refining the mesh. Within the smallest circle at the crack tip, linear