ME 305 Homework 5

For the purpose of analysis, all units used in this case are mm and N.

Model Components

To simulate strain gauge stress measurement on solid surface, the model is composed of 2 dragged solid components, 1 mapped solid components and a thin wall skill of 0.005 mm component with property of PSHELL and identical material as that of the solid, which is steel. The shell component is wrapping around the entre three solid components with all free edges and faces equivalence assigned to be used as strain gauge on surface for stress recovery.

Constrain Requirement

To avoid artificial stress concentration due to the Poisson's effect caused by in appropriate assignment of the boundary conditions, the DOF 1 and DOF 3 constrains are applied to the back side on the skin shell element nodes along a center line estimated from the geometry as shown in Figure 1. All DOF 2 on the flat surface of the back side are constrained.

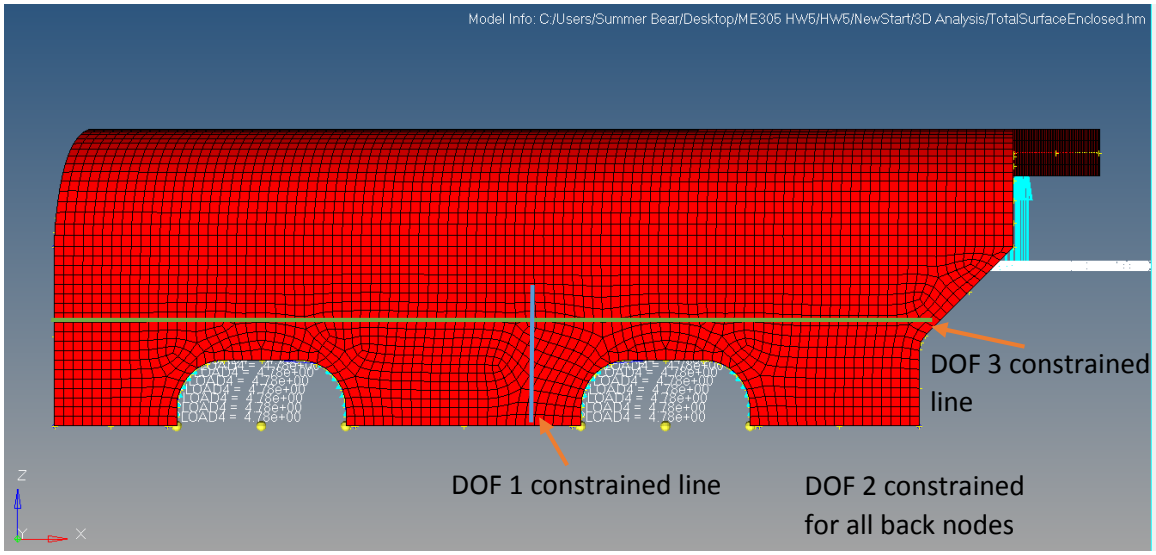

Figure 1: Boundary Condition Assignment Indication (Constrains not shown for clarity)

Pressure Load

The pressure load assigned on the elements are calculated based on the force divided by the area of the meshed elements measured within Hypermesh. With the given force applied at different location, the load applied is summarized in Table *1*.

Location	Force (N)	Area (mm^4)	Pressure Value (MPa)
Outside	800	87.954	9.096
Back	200	41.831	4.781

Table 1: Stress Prediction at Edge for Glued Model with 2.5mm Element Size

Element Size 1 mm

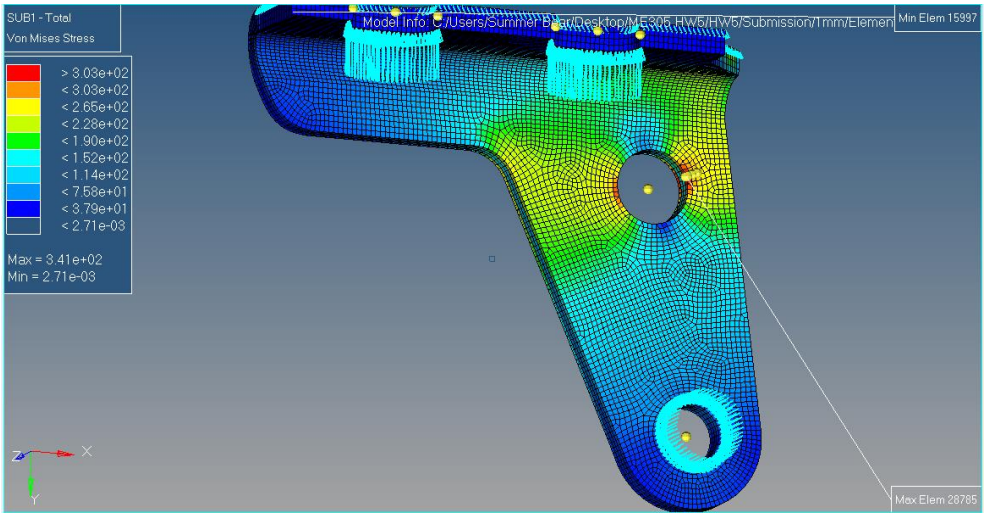

Figure 2: 1 mm Element Size Solid Elements Von Mises Stress Overview (unit: MPa)

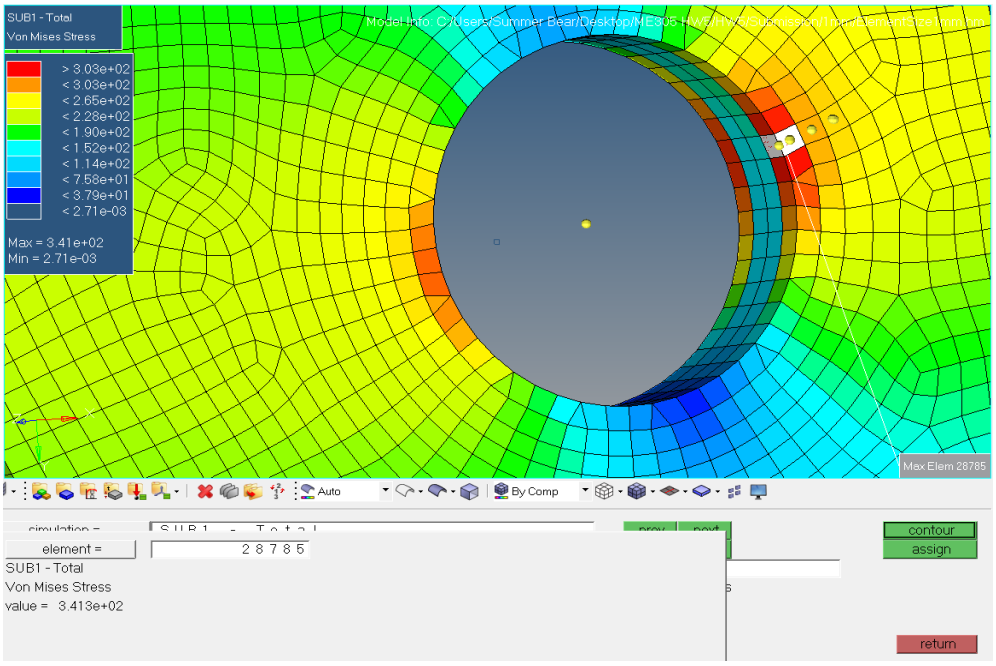

Figure 3: 1 mm Element Size Solid Elements Von Mises Stress Detailed View (unit: MPa)

The picture shows the maximum stress location in the solid elements. To obtain a more accurate measure from extrapolation, the stress value on the thin wall shell element is used as strain gauge on skin. The extrapolated value is obtained from the center point values and shown in Table *2*.

Point	Distance from maximum stress edge (mm)	Maximum Von Mises Stress(MPa)
1	0.492	445.5
2	1.493	383.3
3	2.485	364.2
Predicted	0	491.9

Table 2: Stress Prediction at Edge for Glued Model with 1 mm Element Size

Element Size 0.5 mm

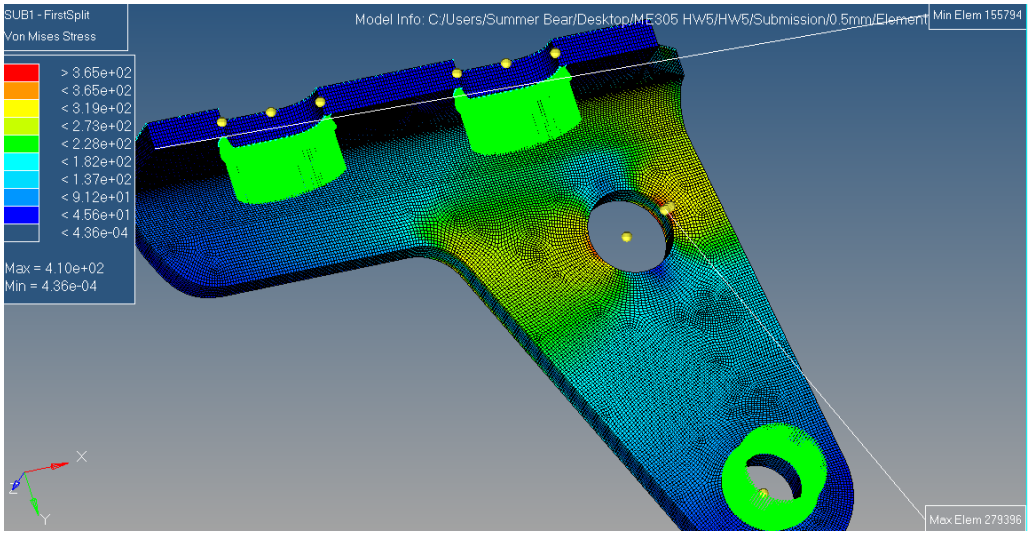

Figure 4: 0.5 mm Element Size Solid Elements Von Mises Stress Overview (unit: MPa)

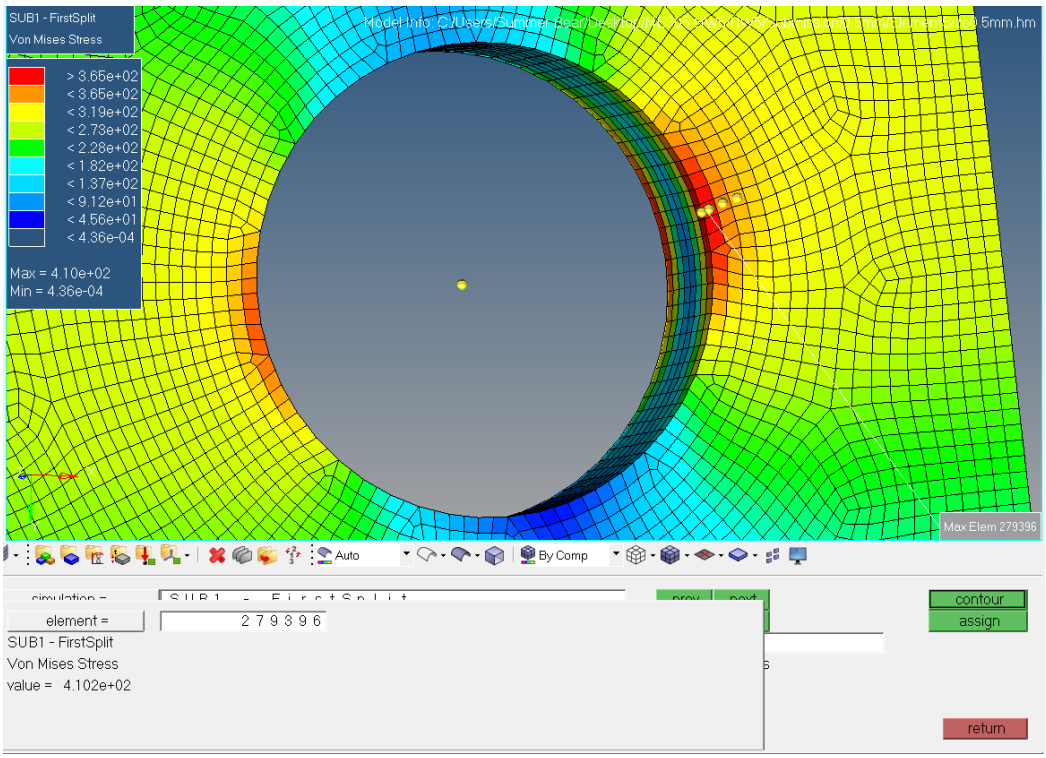

Figure 5: 0.5 mm Element Size *Solid Elements* Von Mises Stress *Detailed View* (unit: MPa)

The picture shows the maximum stress location in the solid elements. To obtain a more accurate measure from extrapolation, the stress value on the thin wall shell element is used as strain gauge on skin. The extrapolated value is obtained from the center point values and shown in Table *3*。

Point	Distance from maximum stress edge (mm)	Maximum Von Mises Stress(MPa)

1	0.2464	460.3
2	0.7392	414.9
3	1.239	389.9
Predicted	0	490.7

Table 3: Stress Prediction at Edge for Glued Model with 0.5 mm Element Size

Part d Stress Convergence

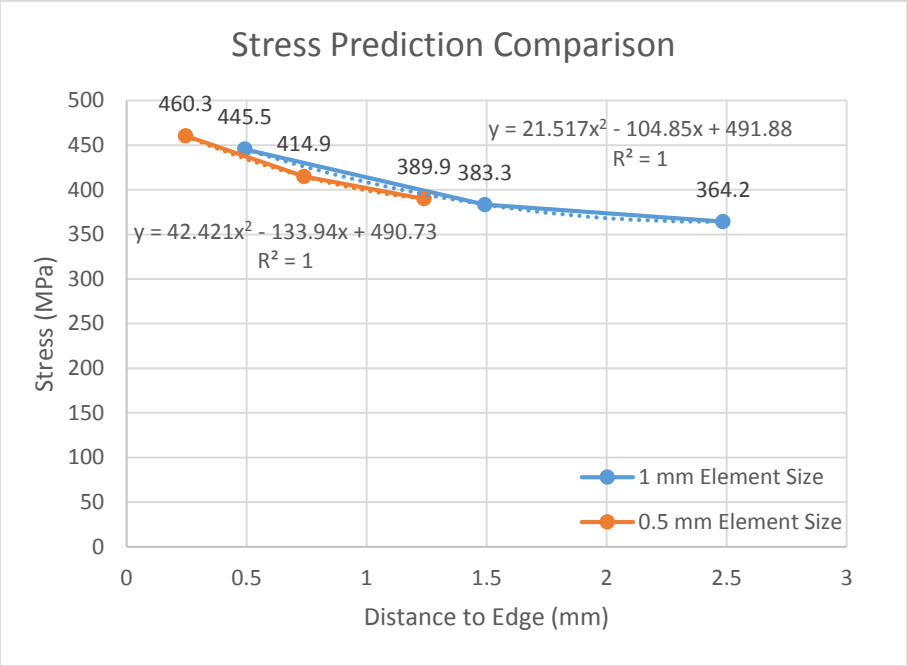

Figure 6: Stress Prediction Extrapolation Comparison

From the plot, we see that the result from the predicted models of 1mm and 0.5 mm element size have similar maximum stress location and similar Von Mises stress value which differs only by 0.26%. With the small difference between the splitting of elements, there is no need to perform another round of splitting and the result should be reasonably close to the maximum stress within the structure under the specified loading condition. As 0.26% is a lot smaller than 15% which is the commonly employed stress extrapolation accuracy requirement range, the result is justified to be within the accuracy requirement.