Analysis and Design of Bracket Using Abaqus CAE Software

Gusman, Lucas

EML 4507   11/26/2019

***Abstract*— Real world systems and processes can be modeled as Finite Element Models on paper or with the help of computers to facilitate their analysis. However, this type of approach although resourceful, requires meticulous steps to ensure that your model properly translates to the real system. This project conducted studies on the convergence of results depending on the number of nodes used in analysis, the limitations and properties of different element types, and proper design changes to achieve mass reduction without increasing stress levels in the wrong areas.**

***Index Terms*—Element Types, FEA, Mesh, Mass Reduction**

I. Introduction

ONE physical system can be expressed in many different ways through finite element (FE) models. However, these models need to properly model the system's reaction to its physical environment. The way boundary conditions, loads, and element types/sizes are defined is critical to get a FE model to yield useful results corresponding to reality. To do that, engineers need to have a solid understanding of the mechanics and material properties ruling over the system of interest, as well as of how FE analysis gets to its results. Knowing these principles, one can model a system without making incorrect assumptions or interpreting results incorrectly. This project's main objective was to show the impact element definitions have on the model results. To do that, a bracket design was put through the same loading environment several times but with the analysis being conducted with different element types and sizes. Then, once the best analysis set up was found, a study was conducted to minimize the mass of the bracket while keeping its functionality intact.

The loads and boundary conditions required to model this environment were provided in the instructions, along with geometry dimensions and constrains, material specifications, and guidelines for each of the element studies to be conducted. The bracket was designed as a triangle with rounded points hollowed out to allow fasteners and with a smaller triangular cut out at its center (Fig. 1). To define how the part was constrained, the project defines the two bottom holes as fixed. To conclude the environment definition, a load of 10,000 N was applied at the top hole, in the horizontal direction.

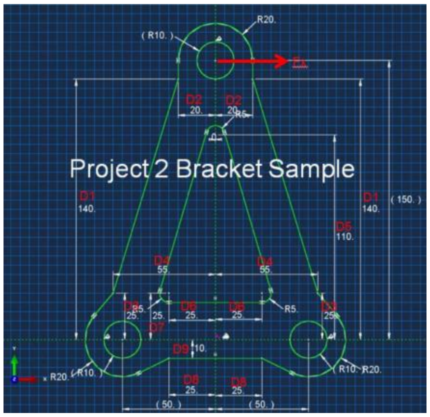


Fig. 1: Diagram of bracket design along with dimensions, boundary conditions, loads applied, and geometric constrains. Given in the project instructions.

This report will include results and discussions dealing with: a convergence study on the accuracy of FE results as the number of nodes is increased, an element comparison study, and a mass minimization study.

II. Approach

Modeling the Part Geometry

To start, a model of the bracket was made on Abaqus. The bracket was modeled as a deformable 2D planar *shell*. The initial model was dimensioned and constrained as shown in the project proposal. After achieving a fully constrained geometry (Fig. 2), the part was then assigned a *solid*, *homogenous* section with a plane stress/strain thickness of 3mm.

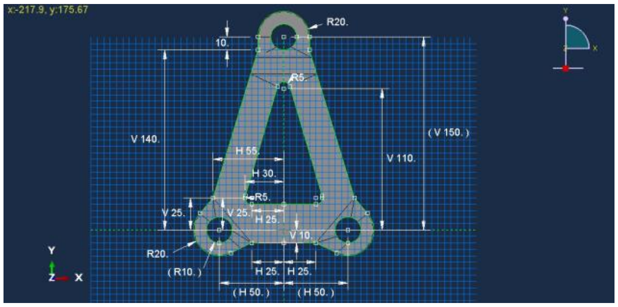


Fig. 2: Fully constrained geometry as requested in *Task 1*.

The material properties were then specified with a density of 7850 kg/m^3, a thickness t = 3mm, a Young's modulus E = 206.8 GPa, Poisson's ratio of 0.29, and a maximum allowable stress of 800 MPa. For easy reference, Table I presents all given material properties.

TABLE I
MATERIAL PROPERTIES

Material Property	***Value***
Density (ρ)	7850 kg/m^3
Thickness (t)	3 mm
Young's Modulus (E)	206.8 GPa
Poisson's Ratio (ν)	0.29
Maximum Stress (σ)	800 MPa

An instance of the part was then created so boundary conditions could be defined. A *linear perturbation* step was also made to allow for an applied load to be defined.

Modeling the Loading Environment

To correctly model the required environment, boundary conditions and loads had to be applied at the center of the bracket holes. For this, reference points were made at the center of each hole. MPC, beam type, constraints were set at each of those reference points with their respective hole circumferences. To accomplish this, the center reference point was selected as the control point (master node) while the circumference was selected as the slave nodes.

With the completed center point set up, boundary conditions were then applied. The centers of the bottom two holes were set to be fixed to the ground, which meant constraining their displacements in the x and y directions to be zero ($u = v = 0$). A concentrated load of 10,000 N was then applied in the x direction to the center of the upper hole (Fig. 3).

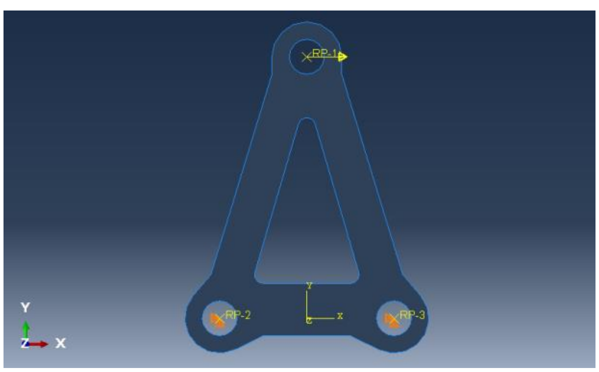


Fig. 3: Loading environment as specified in project requirements. In orange are pinned boundary conditions and the yellow arrow the applied load.

Convergence Study

A convergence study was conducted to verify that the finite element solution converges to a value as more elements are added. For this study, the x-displacement at the load application point was to be analyzed as a function of the number of nodes used in the mesh.

To start, an untreated mesh was created. This default mesh, being asymmetrical and using multiple element types, was later used for comparison as new meshes were created (Fig. 4).

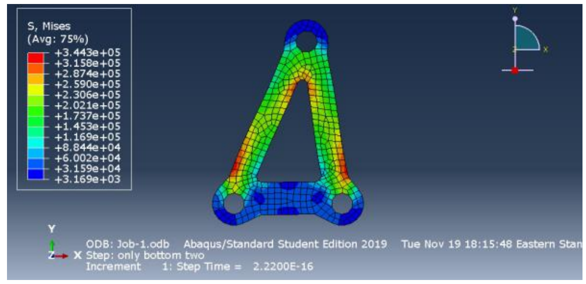


Fig. 4: Stress study ran using untreated mesh.

Using mesh control, this untreated mesh was set to only use 4-node, quadrilateral elements. The next step was to refine the meshing process. For that, the part was partitioned into several different sections, allowing for better control of the meshing process in problematic areas. With the part face partitioned, a sweep technique was used to create a more organized and symmetric mesh.

The project suggested that the convergence study would be conducted for results using 400, 600, 800, and 1,000 nodes. Abaqus allows the user to set the global element size which dictates the number of elements and, therefore, the number of nodes used. Since the complexity of the part did not allow for a smooth relation between global element size and number of nodes, trial and error would take a significant amount of time given the fact that multiple meshes would need to be created for each part of the study. To resolve this, a MATLAB code was developed to quickly calculate the number of nodes used given a global element size. The code and explanation of it can be found in the Appendix. A mesh using each of the suggested number of nodes was created and the load environment was the studied for each of them.

Element Type Study

The second study conducted was on the type of element used in the mesh. Following project guidelines, meshes using 4-node quadrilateral elements, 8-node quadrilateral elements, 3-node triangular elements, and 6-node triangular elements were to be analyzed. To achieve each of these element types, mesh control was once again used. In mesh control, *quad* elements were used for the quadrilateral studies and *tri* elements were selected for the triangular ones. To control the number of nodes in these elements, *element type* was used to switch between linear and quadratic geometric orders. It is also important to note that reduced integration was unselected for all of these studies.

The factors of interest during this study were the displacement of the load application point and the maximum von Misses stress recorded on the part.

Mass Reduction Process

For this final study, the project requests that the mass of the bracket is decreased by changing 9 design variables. This could of course be done randomly, however, following basic concepts learned in mechanics of materials, a few design changes standout as having a big impact on the mass and a small impact on the maximum stress experienced by the part.

III. Results

During the convergence study four different mesh models, each using a different number of nodes, were studied. For the first model, a global element size of 6.25 was used, which yielded 456 nodes. With this set up, the displacement of the load node of interest was of was of $4.141\ X\ 10^{-7} mm$. A displacement plot was exported to show its distribution through out the part, however since it is basically the same for all iterations and the only node of interest is the load application point, it will only be shown once (Fig. 5).

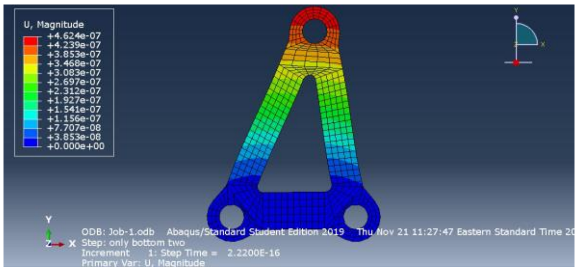


Fig. 5: Plot of displacements for the ~400 node set up.

For the second one, a global size of 5 was used, which yielded 631 nodes. With this set up, the displacement at the node of interest was found to be $4.152\ X\ 10^{-7} mm$.

For the third one, a global size of 4.41 was used, which yielded 807 nodes. With this set up, the displacement at the node of interest was found to be $4.160\ X\ 10^{-7} mm$.

Lastly, a global size of 4 was used to yield 949 nodes. With this set up, the displacement of the pint of interest was found to be.

Using the values found above, a convergence study was conducted. A plot of the displacements found and the number of nodes used was created to help visualize a convergence pattern, if there was any (Fig. 6).

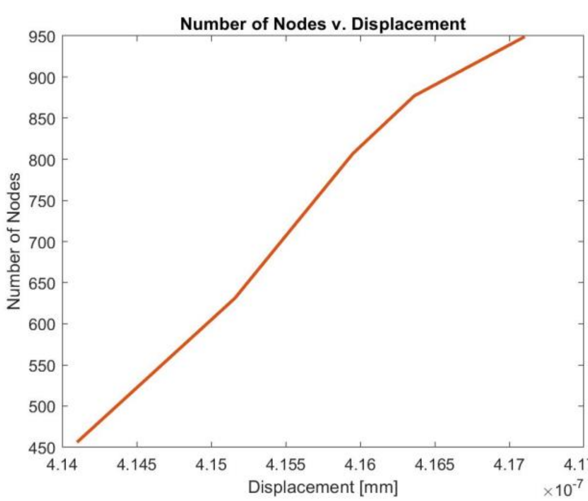


Fig. 6: Plot for convergence study, displacement v. number of nodes.

Then, for the element type study, four different element types were used and their results analyzed. First, for a mesh using a 4-node quadrilateral, the maximum von Misses stress recorded was of $5.25\ X\ 10^{8} Pa$ (Fig, 7). A displacement of $4.171\ X\ 10^{-7} mm$ was recorded at the node of interest.

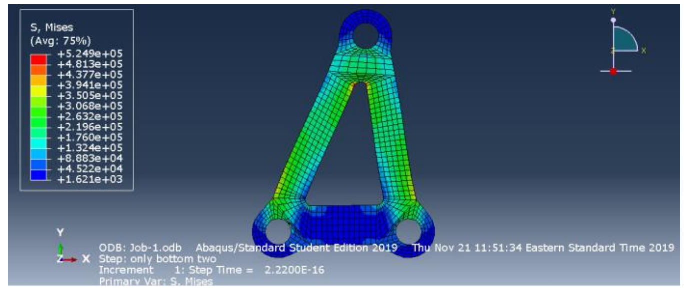


Fig. 7: Stress distribution for 4-node quadrilateral element analysis.

Then, for a mesh using an 8-node quadrilateral, the maximum von Misses stress recorded was of $5.38\ X\ 10^{8} Pa$ (Fig, 8). A displacement of $4.186\ X\ 10^{-7} mm$ was recorded at the node of interest.

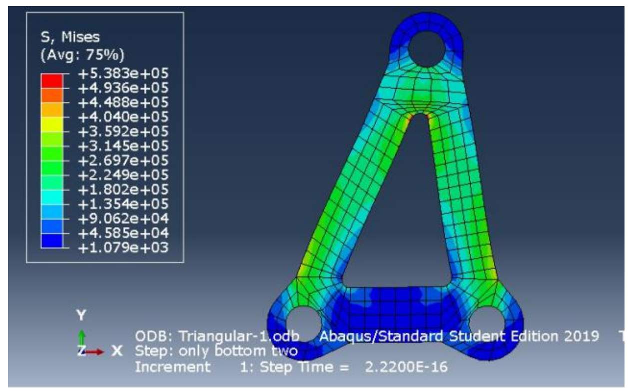


Fig. 8: Stress distribution for 8-node quadrilateral element analysis.

Then, for a mesh using 3-node triangular elements, the maximum von Misses stress recorded was of $3.65\ X\ 10^{8} Pa$ (Fig, 9). A displacement of $4.147\ X\ 10^{-7} mm$ was recorded at the node of interest.

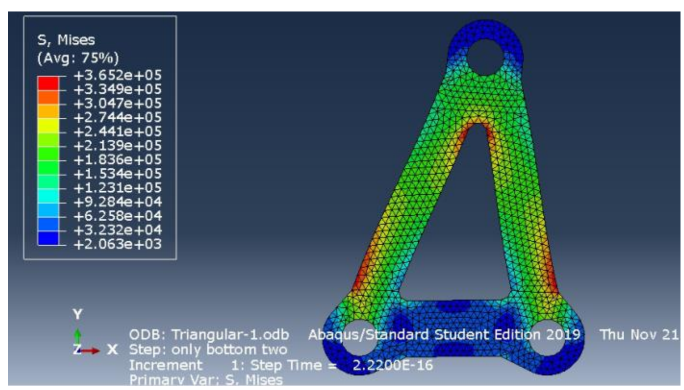


Fig. 9: Stress distribution for 3-node triangular element analysis.

Finally, for a mesh using 6-node triangular elements, the maximum von Misses stress recorded was of (Fig, 10). A displacement of $4.188\ X\ 10^{-7} mm$ was recorded at the node of interest.

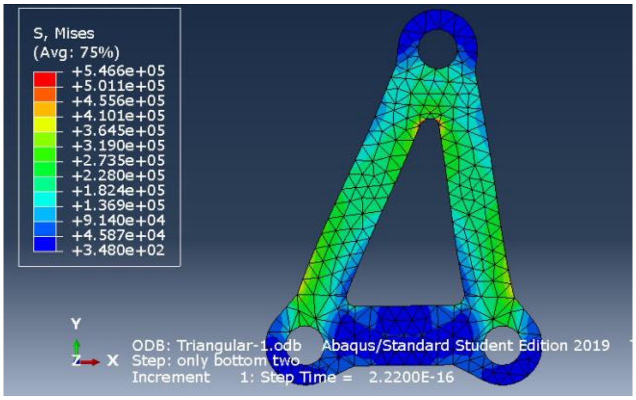


Fig. 10: Stress distribution for 6-node triangular element analysis.

Lastly, a mass reduction study was conducted on the bracket design. To reduce the mass of the bracket, the bracket was shortened over all, the inside slot was made bigger, and its bottom plate was made smaller. The stress analysis on the final design is shown in Fig. 11. The reduced mass design had a weight of 196g while the original design weighed 248g, yielding a little over a 20% mass reduction.

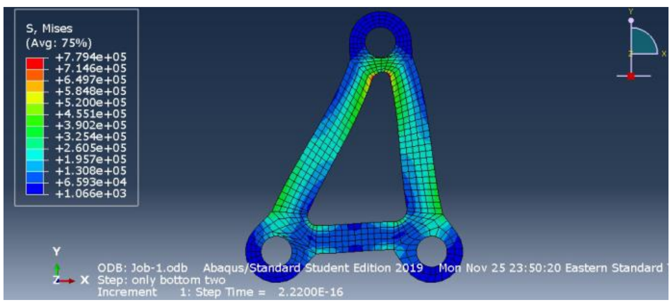


Fig. 11: Stress distribution for the reduced mass design.

IV. Discussion

After iterating through the different sizes of mesh and analyzing the convergence study plot, it could be said that the displacement value for the node of interest tends to converge to a single value as more nodes are added. Looking at the plot in Fig. 6, there is no massive evidence of an asymptote. However, a decreasing slope trend can be seen between each increase in number of nodes. That along with the fact that the real displacement value must be a finite number, it is safe to assume that the model should converge to that real value if more nodes kept being added on. This assumption, however, could not be further explore in the models due to the Abaqus restriction of 1,000 nodes for analysis on the Student License. If this constrain was not present, efforts would be made to conduct more studies with larger numbers of nodes to expand the convergence plot found up to where the final value converges. If this is true, then it proves that a larger number of nodes is not always the best way to make a FE model more accurate. For this project, adding more nodes would still give a more accurate result. However, that does not imply that putting 1 million nodes in will give me a better answer than maybe 10,000 nodes since the trend is for the displacement value to converge after a certain number of nodes.

Then, after iterating through different element types, the 8-node quadrilateral element was found to be the best element type to be used for the purposes of this project. This conclusion came from the fact that it yielded both the highest maximum stress and highest displacement readings from all of the trials. It is important to note that a higher maximum stress reading does not always mean a better model. Sometimes, a high aspect ratio in a mesh or other bad mesh scenarios might report a erroneous stress concentration, and that stress might end up being higher than the prior maximum stress. Care was taken in this study to make sure this would not happen, so the maximum stress readings were always checked to see if they came from the expected area of stress concentration (the top slot fillet). Some of the other element types gave satisfactory readings also, like the 4-node quadrilateral and the 6-node triangular elements. However, the 3-node triangular element reported stress levels that were only about 65% the stress levels reported by the other elements. That happened because of the limitations of these Constant Strain Triangles (CST). These elements only report constant values through each element. When compared to a quadrilateral element that has the ability to have a linear difference (or quadratic for a 8-node one) between each node, its accuracy drops significantly. The 6-node triangle however performed better than its 3-node counterpart because of these three extra nodes. These newly added nodes allowed the element to have variation within it, making it more accurate.

For the mass study, it was found that the mass of the original bracket design could be decreased by a little more than 20% until it would reach yielding stress levels. This was achieved by changing nine given design dimensions. The first step taken was to shorten the bottom plate connecting the bottom two holes (D9 on the project diagram). That was the first step taken because it was in a low stress concentration area, meaning changes there wouldn't have a big impact on the high stress areas. Then the radii of the inner slots were increased. This was done because those radii had a significant impact on the overall weight of the part. However, the top radius also did have a big impact on the maximum stress, as it was the stress concentrator where that ax stress was at. Thus, increasing that radius meant increasing the maximum stress experienced by the part. To counter this increase in stress, the over all height of the part (D1 and D5) was decreased. This decrease in height also meant a decrease in the moment arm the applied load had, which in turn decreased the stress caused by it. After applying all these changes, the bracket weighed 196g and had a maximum stress reading of $7.794\ X\ 10^8 Pa$, just short of the yield strength of the given material at $8.00\ X\ 10^8 Pa$.

V. Conclusion

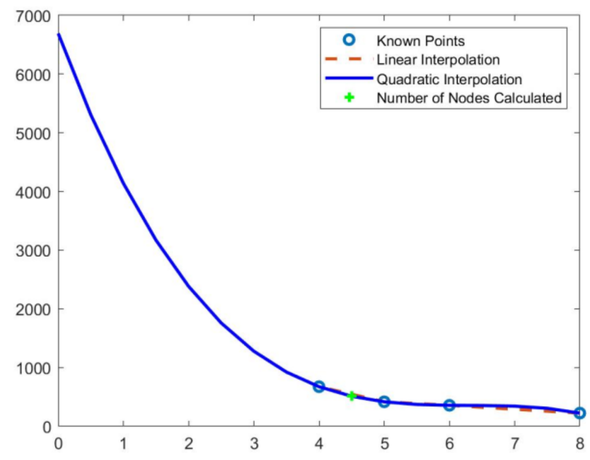
The main objectives of this project were to understand the impact a mesh has on the results of a FE model and to understand how to properly make design changes to an existing design. It was found that with the limitations of the student version of Abaqus, not enough nodes could be used to find the real displacement and stress values experienced by the given part. However, the conducted convergence study proved that just adding more nodes randomly would not be the best idea since the results tend to converge to a value, therefore meaning there is a number of nodes past which adding more won't make the results more accurate. It was also found that, over all, quadrilateral elements are more reliable than triangular elements since they have the ability to better represent stress and displacements in between element nodes when compared to triangular elements. Just as important as the element type and number of nodes, is the meshing technique and face partitioning. Partitioning the different faces of a part allows for better control over the mesh. Custom mesh structures can be created for specific areas, for example a high stress area can be assigned a mesh with more elements since the distribution of stress and readings are important at that specific point.

When it comes to mass reduction in existing designs, engineers need to keep in mind concepts of mechanics of materials and other structural classes. Mass reduction needs to be done first in low stress areas since those won't affect the critical areas of the given part. Mass reduction near a high stress concentration should be avoided as much as possible, and when it is not then measures need to be taken to lower the stress value around that area.

APPENDIX

MATLAB code to help find approximate number of nodes from given global element size. This code was written using known element and node values corresponding to their respective global element sizes given by Abaqus. After getting these known values, a simple linear interpolation was initially done to find the values in between these points. However, it was found that linear interpolation was not very accurate for this relationship. So, the *polyfit* function was used instead to try fitting polynomials of different degrees between these points. A second degree polynomial was found to be accurate enough for the purposes of this task. A plot of the interpolated function, given points, and calculated number of nodes for a global element size of 4.5 for 4-node quadrilateral elements is shown as an example.

```
% Quick FEA Proect 2 Help
clc; clear;
globsize  = 8;
elements  = 224;
nodes     = 291;

globsize2 = 4;
elements2 = 674;
nodes2    = 806;

globsize3 = 6;
elements3 = 357;
nodes3    = 445;

globsize4 = 5;
elements4 = 417;
nodes4    = 521;

gg        =[globsize,globsize3,globsize4,globsize2];
ee        =[elements,elements3,elements4,elements2];
nn        =[nodes,nodes3,nodes4,nodes2];

p           = polyfit(gg,ee,3);
mnodes      = (nodes3-nodes2)/(elements3-elements2);
gsize       = input(' ENTER GLOBAL SIZE: ');
gg          = [gg,gsize];
f1          = polyval(p,gg);
x1          = 8:-.5:0;
f2          = polyval(p,x1);

figure(3)
plot(gg(1:end-1),ee,'o',gg,f1,'--',x1,f2,'b')
hold on
spot        = (max(gg)-gsize)*2+1;
newelements = f2(spot);
newnodes    = newelements*(nodes/elements);
fprintf('The number of nodes you will get is~
%g\n',newnodes)
plot(gsize,f2(spot),'+k')
```



MATLAB code used to plot convergence curve. The file “ConvergenceStudy” is a .csv file that kept track of the number of nodes and global element sizes for each one of the studies ran. The code then read that file and plotted the different values against eachother.

```
%convergence Study
clc; clear;

T = readtable('ConvergenceStudy.csv');
T = T.Variables;
nodes = T(:,1);
disp = T(:,2);

figure(1)
plot(disp,nodes,disp,nodes,'LineWidth',2)
title('Number of Nodes v. Displacement')
xlabel('Displacement [mm]')
ylabel('Number of Nodes')
```

REFERENCES

[1] N. H. Kim,” Lecture Notes”, WEIL 270, 2019.
[2] N. H. Kim,” Project 2 Assignment”, Canvas, 2019.