MEMORANDUM

4/8/2026

TO: Eric Ocegueda, Assistant Professor, Mechanical Engineering
FROM: Apolonio Ordaz, ME 328 – 02
SUBJECT: Lab 1: Intro to FEA

Overview

The goal of this lab was to get familiar with utilizing and running Finite Element Analysis in SolidWorks by analyzing a simply supported aluminum beam under a uniform pressure. We ran simulations using three different elements, solid, shell and beam, as well as testing different boundary conditions and how they affected the calculated values. Hand calculations were performed before running simulations in order to gauge if each respective simulation was producing accurate results.

Hand Calculations

Below is the problem that we were analyzing for the duration of this lab along with the hand calculations performed in Appendix A:

Using beam tables for a simply supported beam with a uniform distributed load, the maximum bending stress and maximum deflection were calculated using the following beam properties: h = 1 in, b = 10 in, L = 10 ft, and P = 5 psi.

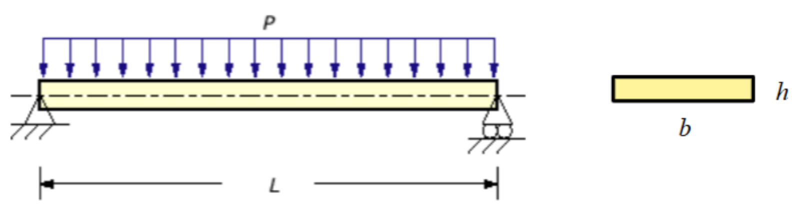


Figure 1. Simply supported Beam for ME 328 Lab 1.

The calculated values were as follows:

Max vonMises Stress: 54 kpsi

Max Deflection: 15.3 inches

A summary of the analysis done in this lab is discussed below, with the recorded data shown in Table 1. Note for beam analysis, the degrees of freedom do not increase as expected, this may have been to a software error, multiple analysis were performed to try and correct the error to no avail.

Table 1. Summary of Max vonMises Stress and Max Deflection with change of Fixtures, Element, and Element size.

Element Type	Element Size (in)	Degrees of Freedom	Max vonMises Stress (kpsi)	Max Deflection (in)
Solid-Fixed Face	8	1365	25.87	2.70
Solid-Sliding Face	8	1374	39.17	5.94
Solid-Edge	8	1404	52.55	15.05
Solid-Edge	4	3859	53.86	15.28
Solid-Edge	2	12356	54.00	15.32
Solid-Edge	1	71841	54.02	15.32
Shell	8	569	53.81	15.28
Shell	4	2527	54.01	15.31
Shell	2	7931	54.03	15.31
Shell	1	30261	54.03	15.32
Beam	8	90	54.00	15.31
Beam	4	1356	54.00	15.31
Beam	2	1356	54.00	15.31
Beam	1	720	54.00	15.31

Boundary Conditions and Fixtures.

We evaluated three different ways to constrain the simply supported beam and each one yielded different result.

The first boundary condition we used we did a fully fixed geometry on the faces of the beams. This meant we locked the beam in all three axes' including the rotation. This was a good starting point but because of the over constrained nature of the fixture it made the beam stiffer than reality and was not accurate to the simple supported beam. Because of this the vonMises and deflection came out lower than the hand calculations predicted at 25.87 kpsi and 2.701 inches, respectively.

The second boundary condition we used restricted the left side of the beam completely on three axis and the right face of the beam on two axes, letting it slide allowing it to move. This better represents a pinned and rolled support from the assignment problem but there were still issues as the locked faces would not allow the beam to rotate as a simply supported beam would. This was notable in the vonMises stresses and deflection calculated which were better but not quite the calculated values at 39.17 kpsi and 5.941 inches, respectively.

The third boundary condition was much better as we only restricted the edges instead of the whole face. This finally let the ends rotate freely when the beam deflected which was much more representative of how a pinned/roller support would behave. The resulting vonMises and deflection numbers calculated were more precise and aligned with the hand calculated values at 52.55 kpsi and 15.05 inches, respectively.

Element Types

We used three different types of elements in this lab, solid, shell, and beam. The difference in the type of element used was a tradeoff between accuracy and time it took to compute the model.

Solid elements model the full 3D shape and represented by tetrahedrons as seen in Figure 2. Because of this inherent shape it does poorly when calculating bending. This can be improved by using smaller elements, however there is a tradeoff between the time to compute and accuracy. As seen in Appendix B, as we decrease the size of the elements we converged to a more accurate calculation. The 8-inch element gave 52.55 kpsi and 15.05 inches respectively, while the 1-inch elements gave 54.02 kpsi and 15.32 inches, respectively.

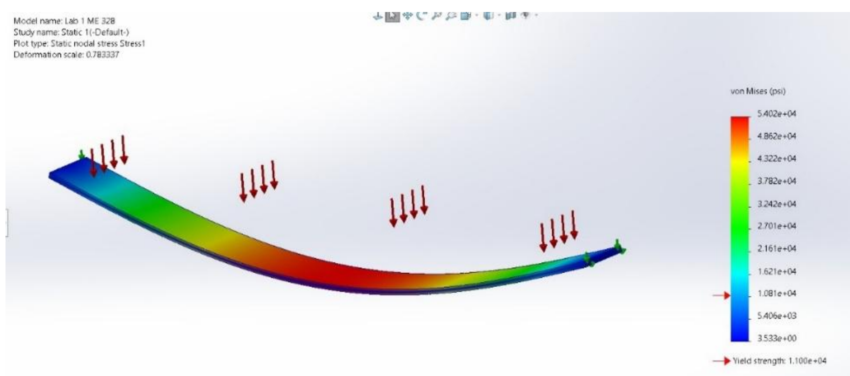


Figure 2. Solid Element vonMises Stress Contour Plot

Shell elements use planar triangles and are used to represent thin parts as seen in Figure 3. This allows the calculations to be quicker and more accurate as it is able to account for the bending that the solid elements could not account for. Since the beam we are analyzing is relatively flat, the shell elements converged faster as seen in Appendix B. For a shell element size of 8 inches, we got 53.81 kpsi and 15.28 inches respectively, and for the 1-inch shell elements we got 54.04 kpsi and 15.32 inches, respectively.

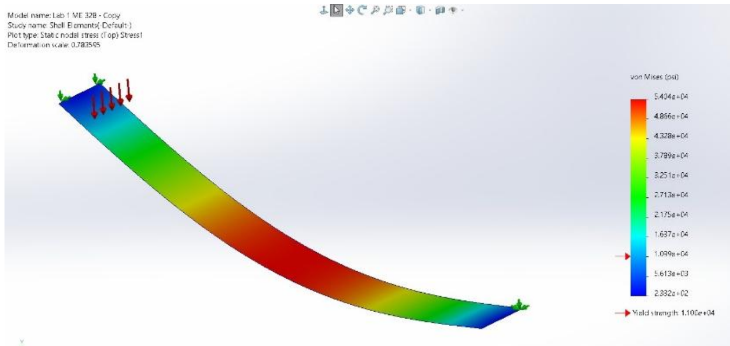


Figure 3. Shell Element VonMises Stress Contour

Beam elements are the simplest and can give calculated numbers very quickly. This is due to the fact that there are predetermined equations that already describe the bending behavior of beams. This beam analysis is showcased in Figure 4. As the beam elements decreased in size the calculated values stayed pretty much constant at 54.00 kpsi and 15.31 inches, respectively. This means that even at the larger element size of 8 inches, the beams were able to accurately calculate the vonMises stresses as well as the overall deflection.

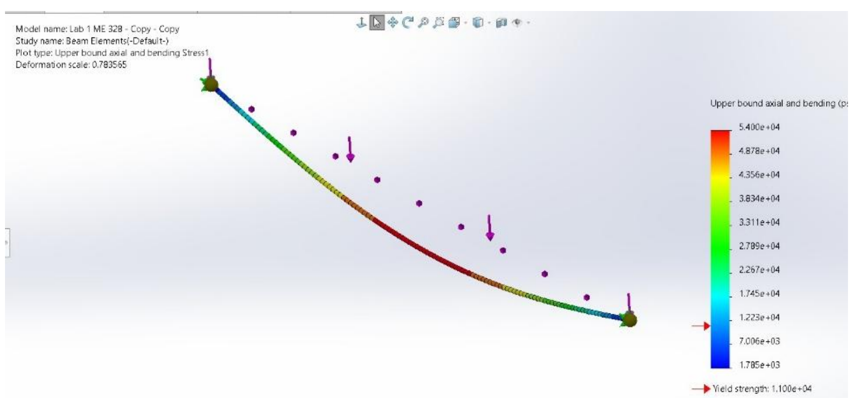


Figure 4. Beam Element VonMises Stress Contour Plot

Element Size and Mesh Convergence

Element size is extremely critical to obtaining correct results. When looking at the convergence study it is noticeable that as we decrease the element size, which increases the number of elements present, the stress and deflections calculated crept closer to the numbers that were calculated by hand. Cutting the element sizes in half shows that as the element gets smaller and smaller, it gives more accurate results because of more points where the FEA is calculating the vonMises Stress criterion and deflections that were present. This causes a convergence of values that can be seen for both vonMises

Stresses and Deflections can be seen for all three Solid, Shell, and Beam elements in figures 5, 6, and 7, respectively.

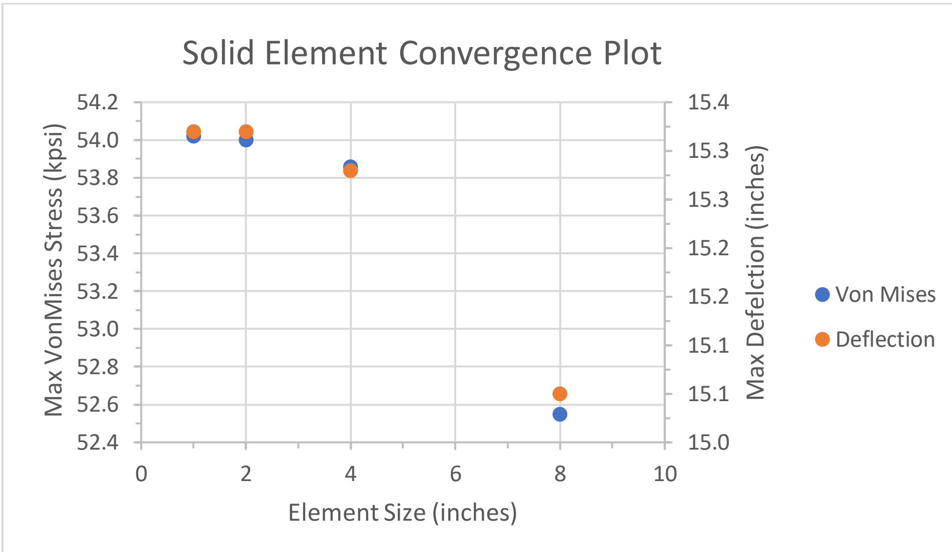


Figure 5. Solid Element vonMises Stress and Deflection Convergence Plot

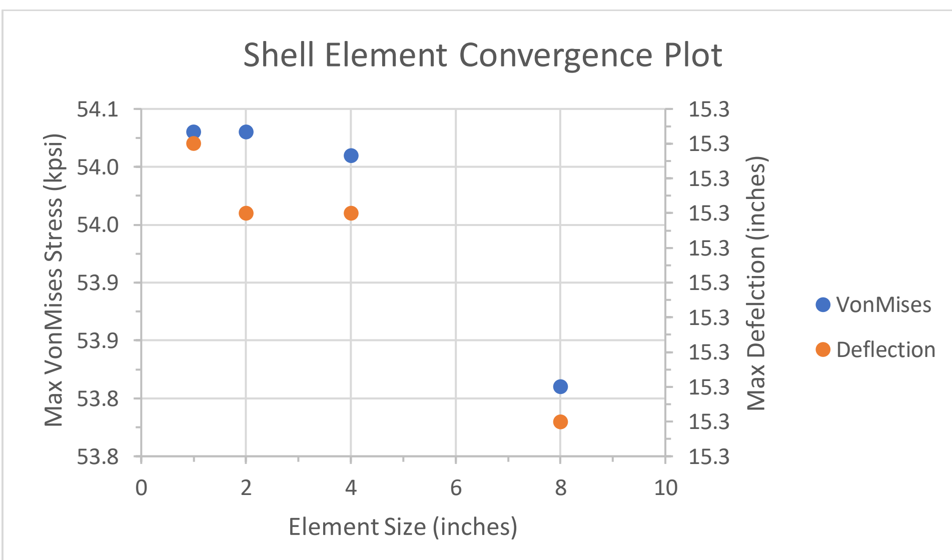


Figure 6. Shell Element vonMises Stress and Deflection Convergence Plot

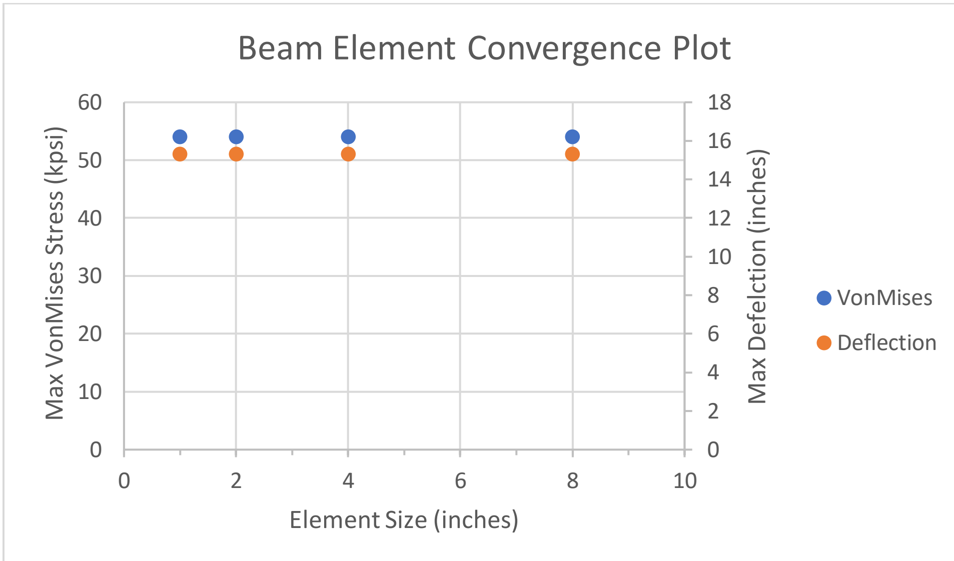


Figure 7. Beam Element vonMises Stress and Deflection Convergence Plot

The issue we run into when only running one simulation with one element mesh size is there is no way to know where we lie on this convergence curve. Both Figures also showcase that the type of element used also dictates the rate of which convergence occurs. Shell and Beam elements converge much faster than the solid elements for our simply supported beam. This lets us know that both element type and mesh size are important in obtaining accurate results from our FEA analysis.

Importance of Hand Calculations

This lab clearly demonstrates that it is important to have an idea of what numbers you expect to see when running FEA. The first two boundary conditions looked correct and produced a relatively good analysis of the beam, however because we solved the problem by hand we knew we had simulated values that were not correct. Without the hand calculations there would be no other way to know that our FEA analysis was wrong. This allowed us to pivot and double check our simulation to better represent the simply supported beam. An issue arises when a number cannot be calculated by hand. In this case, which is the whole reason to utilize FEA, it is still useful to get a ballpark estimate of what numbers we expect to see. This is much better than operating blind as we have a basis to stand on. In this case we can utilize changing elements and mesh sizes to better analyze our problems.

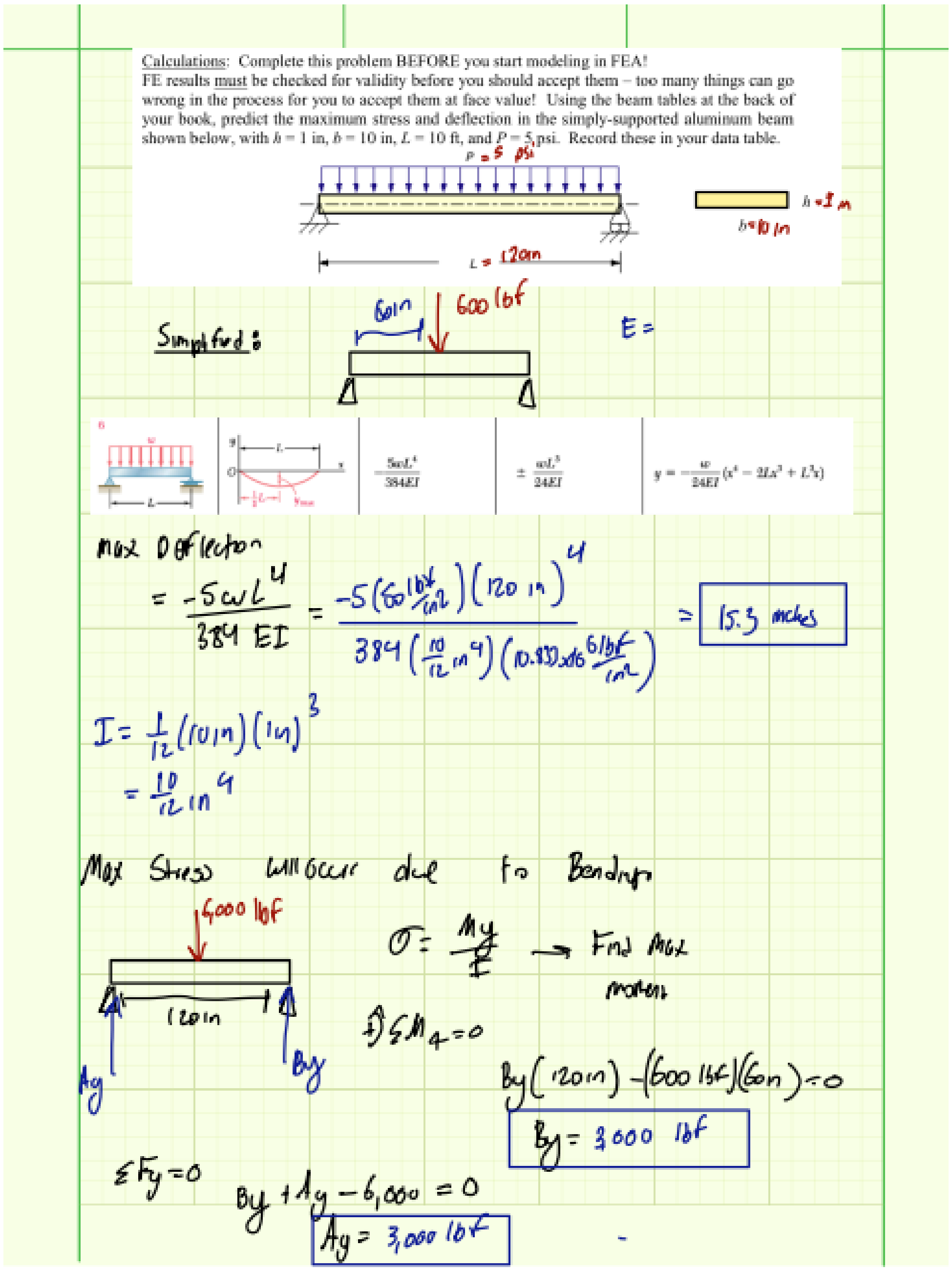
Reliability of FEA Results

Based on this lab it is apparent that FEA does not automatically or always give the correct answer. Our simply supported beam, analyzed with the same loading condition, and just changing the boundary conditions, produced vastly different values for the vonMises Stress and deflections. The FEA study runs and gives numbers based off of how we define the parameters. FEA is just a tool, and it is up to the engineering utilizing this tool to make sure they are inputting the correct parameters. The bigger emphasis is on the engineer to understand the problem, performing convergence studies, and validating results with simple hand calculations. Thus, the FEA simulation will give answers that are only as reliable as the Engineering being able to understand and utilize the software correctly and accurately.

CAL POLY

Appendix A

Hand Calculations



Calculations: Complete this problem BEFORE you start modeling in FEA! FE results must be checked for validity before you should accept them – too many things can go wrong in the process for you to accept them at face value! Using the beam tables at the back of your book, predict the maximum stress and deflection in the simply-supported aluminum beam shown below, with $h = 1$ in, $b = 10$ in, $L = 10$ ft, and $P = 5$ psi. Record these in your data table.

$P = 5$ psi

$h = 1$ in

$b = 10$ in

$L = 120$ in

Simplified:

60 in

600 lbf

E =

6		$\frac{5wL^4}{384EI}$	$\pm \frac{wL^3}{24EI}$	$y = -\frac{w}{24EI}(x^4 - 2Lx^3 + L^3x)$

Max Deflection

$$= \frac{-5wL^4}{384\,EI} = \frac{-5\left(5\,\frac{lbf}{in^2}\right)(120\text{ in})^4}{384\left(\frac{10}{12}\text{ in}^4\right)\left(10.33\times10^6\,\frac{lbf}{in^2}\right)} = 15.3 \text{ inches}$$

$$I = \frac{1}{12}(10\text{ in})(1\text{ in})^3 = \frac{10}{12}\text{ in}^4$$

Max Stress will occur due to Bending

6000 lbf

$\sigma = \frac{My}{I} \rightarrow$ Find Max moment

120 in

Ay

By

$\circlearrowleft \Sigma M_A = 0$

$By(120\text{ in}) - (6000\text{ lbf})(60\text{ in}) = 0$

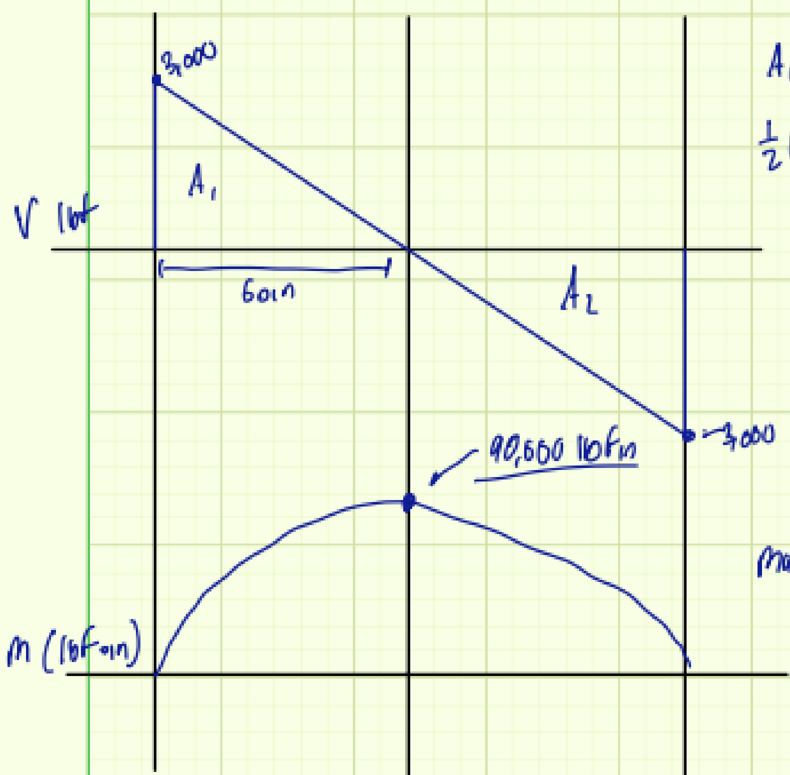
$By = 3000$ lbf

$\Sigma F_y = 0$

$By + Ay - 6{,}000 = 0$

$Ay = 3{,}000$ lbf

Shear Diagram



$A_1 = A_2 =$

$\frac{1}{2}(3{,}000)(60\,in) = \boxed{90{,}000\ lbf\text{-}in}$

$\max M = 90{,}000\ lbf\text{-}in$

$$\sigma = \frac{My}{I} = \frac{(90{,}000\ lbf\text{-}in)(0.5\ in)}{\frac{1}{12}(10\,in)(1\,in^2)}$$

$$\boxed{\sigma_{max} = 54\ kpsi}$$

CAL POLY

Appendix B
Lab Data Table

ME 328 Lab 1 – Finite Element Analysis

Data Table

Hand Calculations:

Max vonMises Stress: 54 kpsi

Max Deflection: 15.3 in

Element Type	Element Size (in)	Degrees of Freedom	Max vonMises Stress (kpsi)	Max Deflection (in)
Solid – Fixed Face	8	1365	25.87	2.701
Solid – Sliding Face	8	1374	39.17	5.941
Solid – Edge	8	1404	52.55	15.05
Solid – Edge	4	3859	53.86	15.28
Solid – Edge	2	12356	54.00	15.32
Solid – Edge	1	71841	54.02	15.32
Shell	8	569	53.81	15.28
Shell	4	2527	54.01	15.31
Shell	2	7931	54.03	15.31
Shell	1	30261	54.04	15.32
Beam	8	90	54.06	15.31
Beam	4	1856	54.00	15.31
Beam	2	1356	54.00	15.31
Beam	1	720	54.00	15.31