# Problem 1 – Direct Stiffness Method using MATFESolver

## Part 1) The MATFE Solver

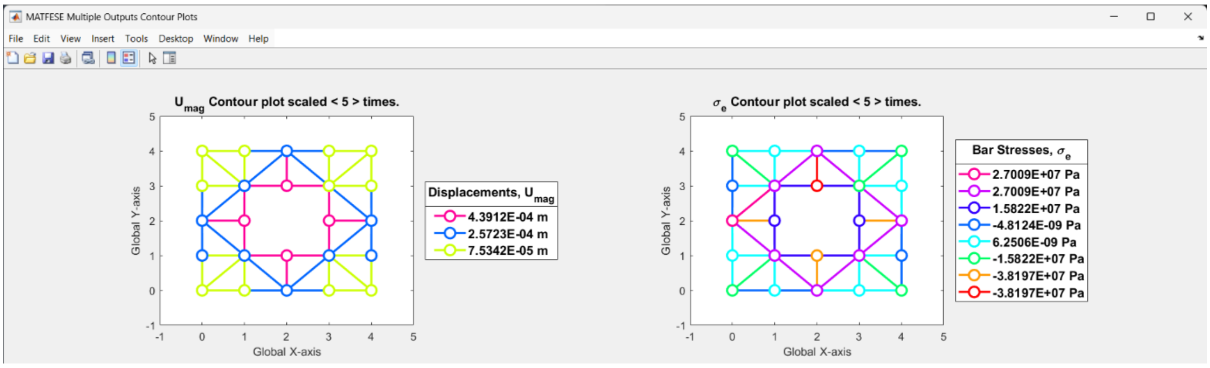
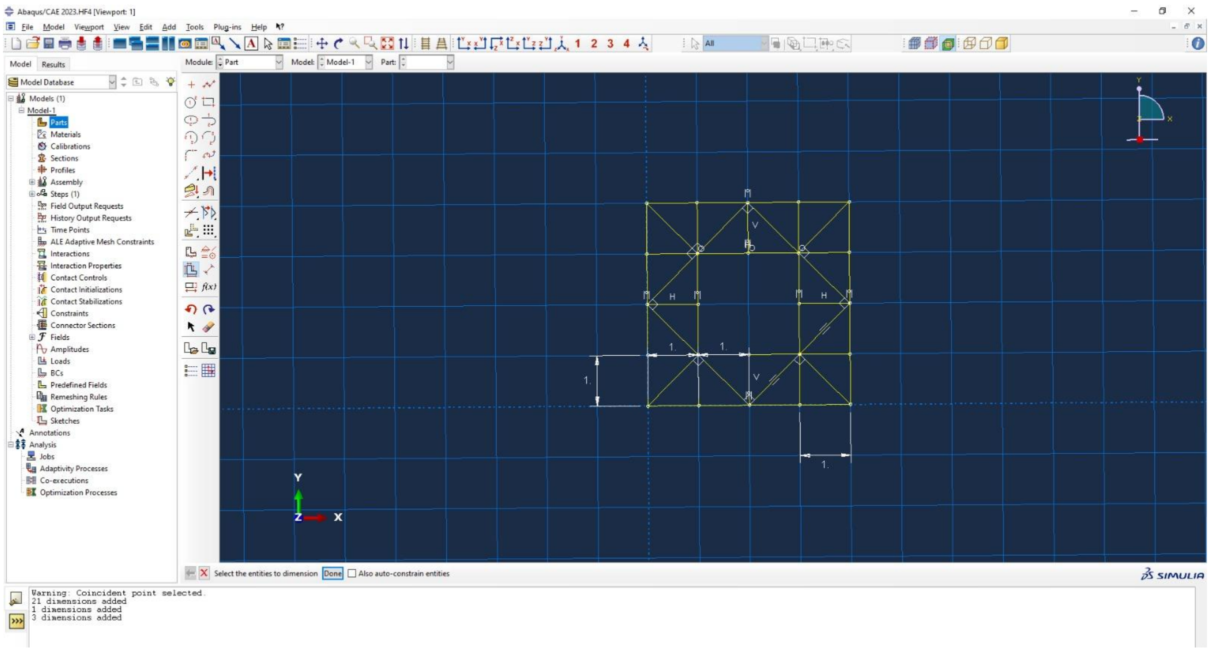



*Figure 1.01: Contour Plots from MATLAB*

The figure above shows 4 contour plots, the main two being the displacement and stress plots. The plots show the stresses experienced by the steel truss-based containment structure with dimensions of $2x2x1m^3$. Figure a shows the displacement scaled by a factor of 5 (for visualisation reasons) and figure b shows the displacements (also scaled by the same factor) for the same structure due to the pressurised container.

## Part 2) Abaqus



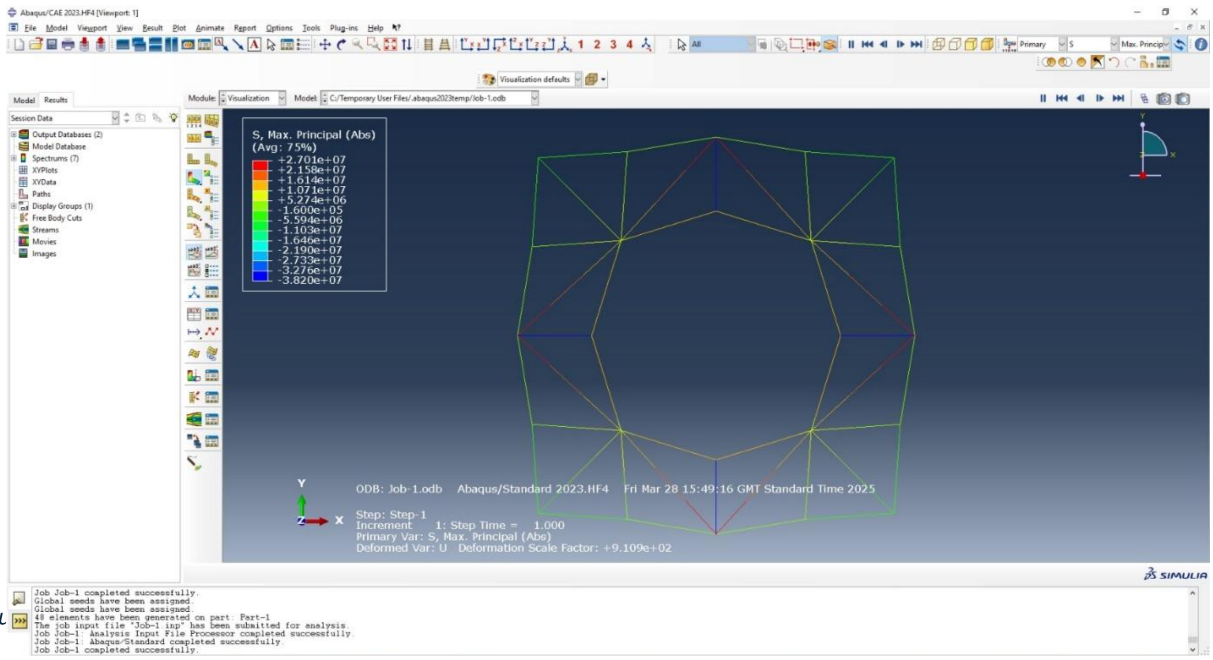




*Figure 1.03: Stress Plot of geometry (S. Max Principle)*

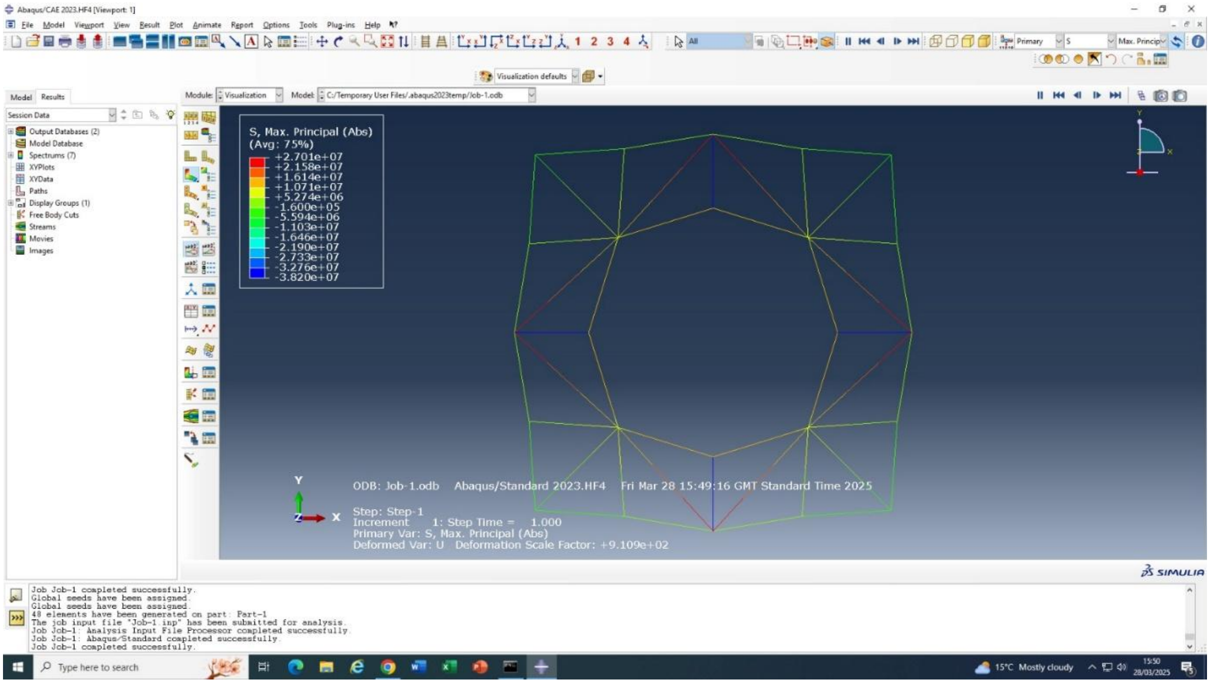


*Figure 1.04: Displacement plot*

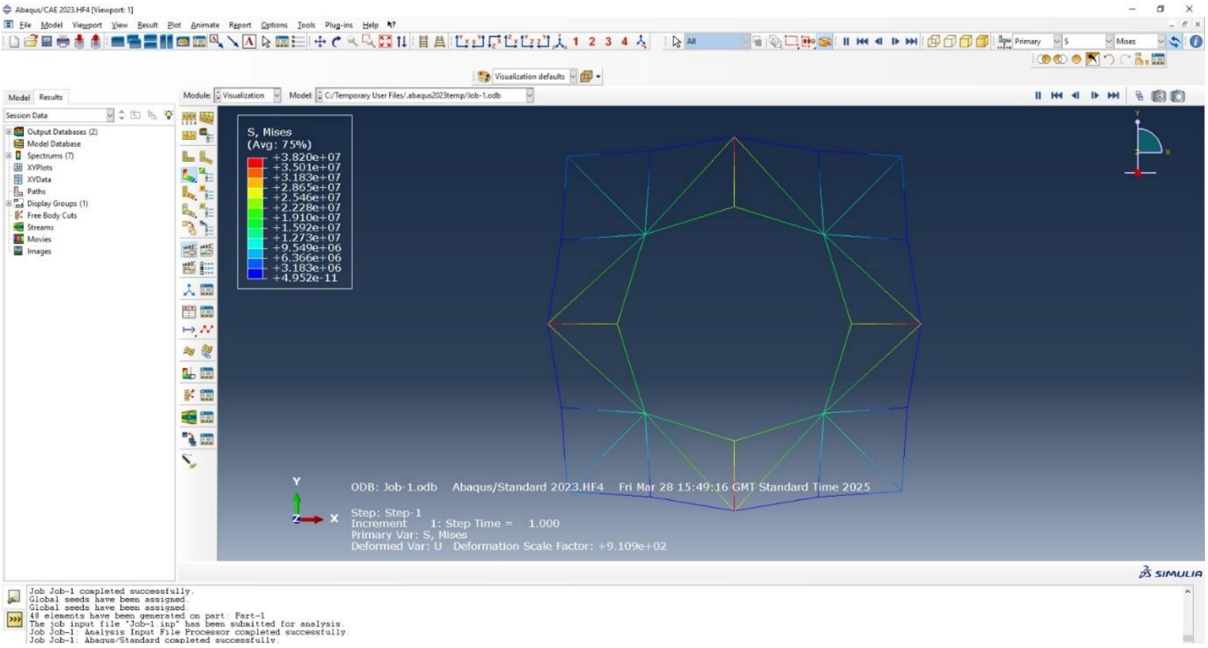


*Figure 1.05: Stress plot (Mises)*

## Part 2) Continued - Discussion

Figures 1.03 and 1.04 show the stress and displacement from the Abaqus and 1.01 shows the stress and displacement for the MATLAB version. In terms of displacement, the numerical values for both are very similar with both showing deformations of $4.39 \times 10^{-4}$ . This shows that the MATLAB version is successful in calculating the displacements just as well as an Finite Element software. However, the only difference that could be noted is the visual deformations is much more exaggerated in Abaqus compared to MATLAB which showed the original plot but with the displacement values (and colours).

In terms of stress, the plot in figure 1.01 shows the bar stress around $2.7 \times 10^{7}$. However, the MATLAB version doesn't display the von mises (average) result and as such there is a discrepancy, however, when browsing through the different types of stress plots the MATLAB version produces a similar result to the S.Max Principal plot which suggests that the MATLAB script is giving the max values instead of the average.

Overall, the use of MATLAB as a FE solver proved to be quite successful and shows how it could potentially be used for different element types (e.g. beams) in the future as well because of the similarity to Abacus in terms of results.

## Part 3) Removal of redundant elements

On Abacus

Using the displacement contour (figure 1.04) there is a variation in the displacement among the trusses. Furthermore, there is also a symmetrical nature that can be seen as well. To fix the over design problem, use the stress contour plot and the dark blue trusses experience the least (almost minimal) displacement (shown in white circles). However, upon revisiting the problem, the white circles are boundary condition nodes (or are connected to the boundary condition nodes) and therefore cannot be removed. As such, whilst following the same ideology described above, the next set of elements that could be removed are the ones displayed in yellow in figure1.06 below.

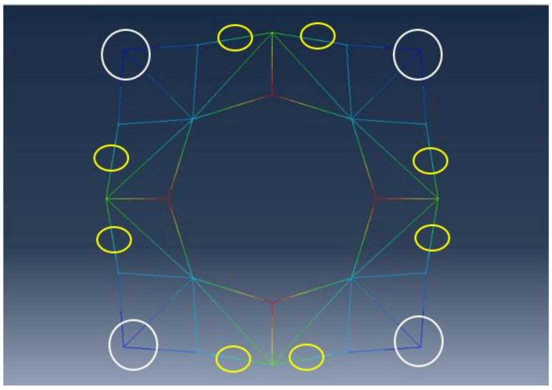


*Figure 1.06: Over designed analysis*

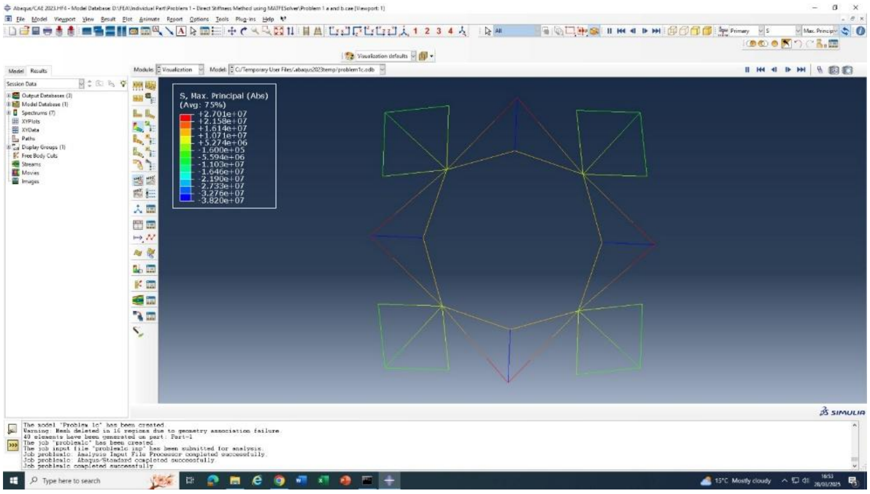


*Figure 1.07: Post Removal S. Max*

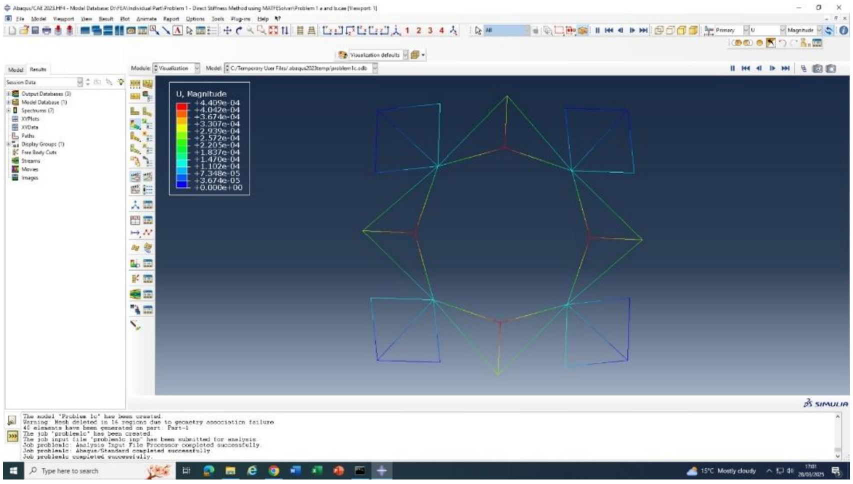


*Figure 1.08: Post Removal Displacement*

By comparing the max numerical values in figures 1.07 &1.08 with 1.03 and 1.04 respectively, there is no significant change in values and as result those trusses can be removed, which would cut material cost, without impacting the structural integrity of the containment structure (assuming the pressure is the same).

<u>On MATLAB</u>

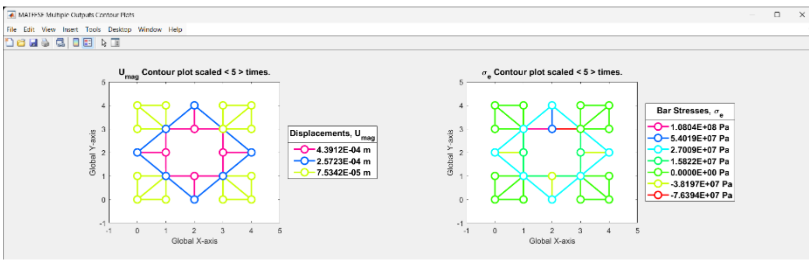



*Figure 1.09: Post Removal (MATLAB)*

Figure 1.09 shows the MATLAB version of the same thing, the displacement values in 1.09 are the same as the displacement values shown in figure 1.01. This further demonstrates the redundancy of the elements that were removed and highlights MATLABs capabilities for 2D analysis.

## Part 4)

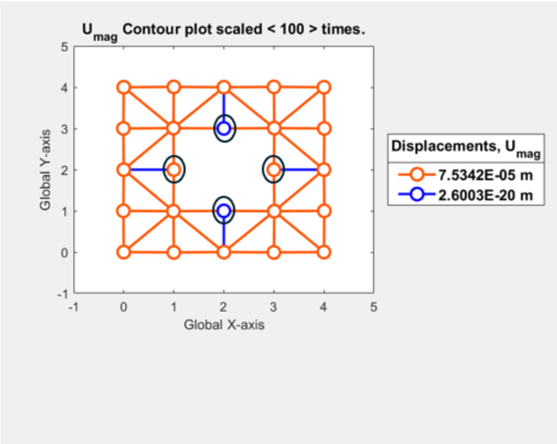



*Figure 1.10: MATLAB with roller supports in 4 central nodes (circled)*

Figure 1.10 shows the new deformation for the original structure (in part A), this shows that there are lower displacement, now the displacement is approximately 7.05 x $10^{-5}$ m and therefore higher stresses can be handled by the structure as it now supported.

## Part 5) Determining Maximum Pressure

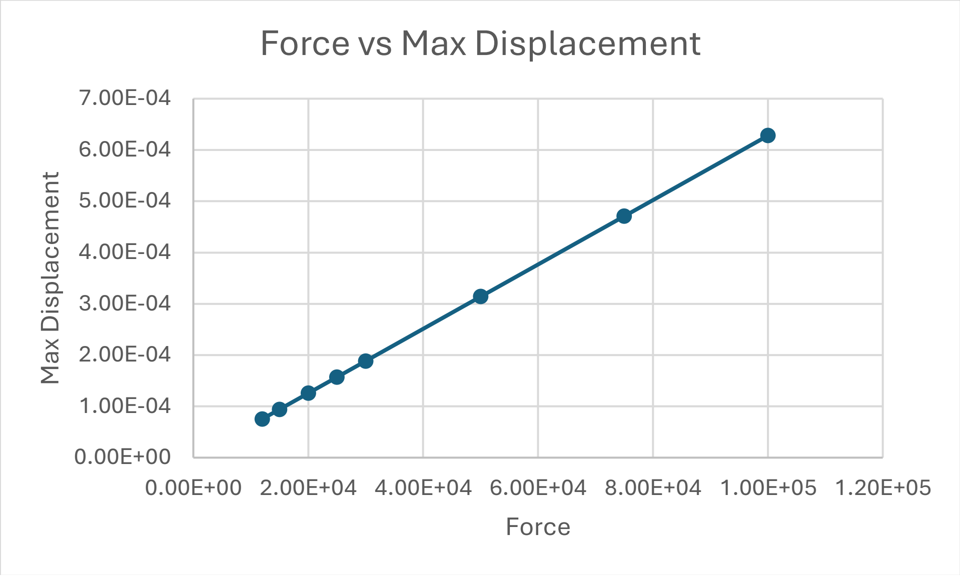
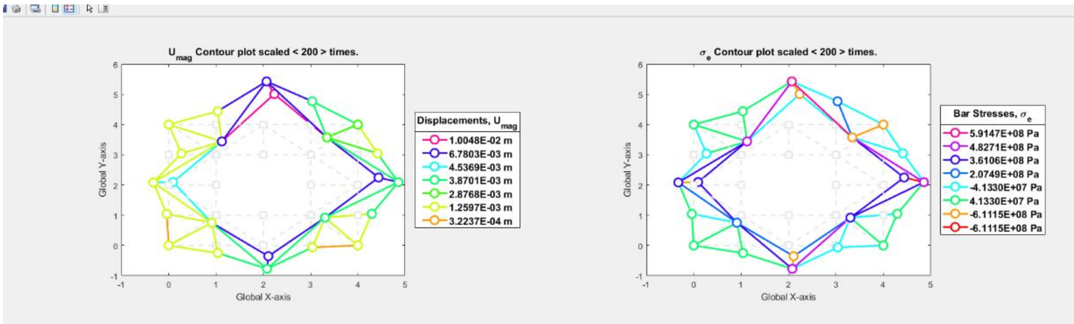



*Figure:1.11 Excel graph of various forces and displacements*

The graph above shows the force applied and the maximum displacements experienced by the structure, (after the roller supports) the maximum pressure is determined by increasing the force applied and then plotting the displacement to see the linear relationship and as such the new maximum force that the structure can tolerate is nearly 70kN which is equivalent to approximately 4.3 x $10^{-4}$ m (which is what the original was) and as such the maximum pressure would be 35kN/$m^2$ which is a significant increase from the initial 6kPa.

## Part 6) Verification Testing with variable loads

- I




This figure above demonstrates the displacement and stresses the structure experiences under the variable loads. The displacement contour plot shows the new deformed profile of the structure.

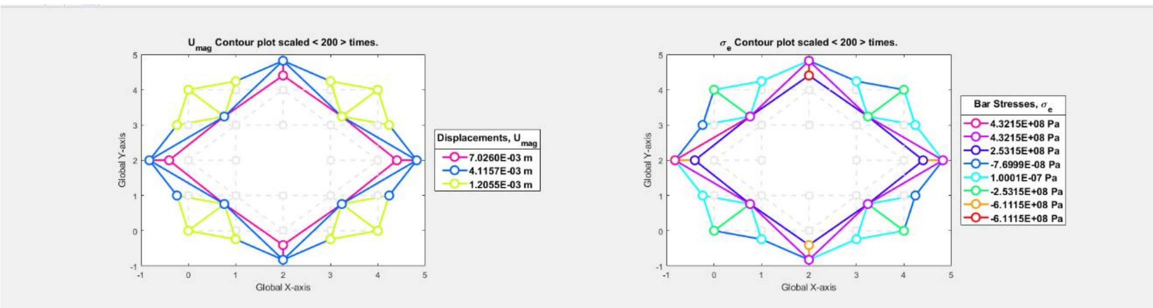
- II

The maximum displacement is very high under these new loads (relative to the other ones seen in 1.1 etc) the max displacement reached a value of approximately 1 x $10^{-2}$ m.

## Part 7) Radical Re-Design

- A) Hollow Truss

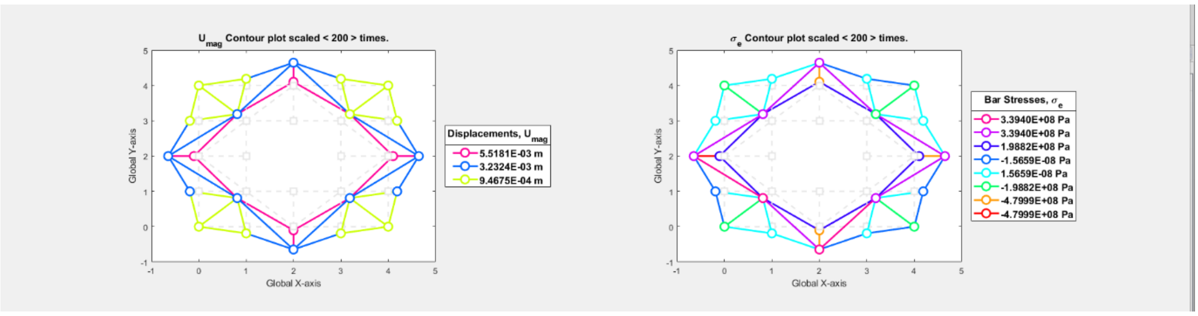
To redesign the structure by replacing the cylindrical bars with hollow trusses, the effective diameter of the hollow trusses was worked out to be 5mm and as such its effective area was calculated to be approximately $2 \times 10^{-5}$ m$^2$. With the new area, the original model was modified in MATLAB and due to the smaller cross-sectional area both the stress and strains increased significantly within the structure which is demonstrated in the figures below.




The re-design proved to be a failure, as the displacement far exceeded the original displacement in 1.1 (part A) with the displacement reaching approximately 7mm. Addittionaly the stresses also increased sgnificantly from 2.7 x$10^7$ to over 4.3 x$10^8$ N/m$^2$.

- B) Square Cross-Section

If the cylindrical bars were replaced by 5x5mm square trusses, the effective cross-sectional area would be 25x$10^{-6}$ m$^2$. However, the MATFE solver requires an input value for the diameter as well as such the equivalent diameter was calculated to be approximately $5.642 \times 10^{-3}$. In putting the data into the solver yields the contour plots shown below.




The displacement plot shows that the applied pressure of 6kPa is so significant in this situation relative to the one in 1.1 (part A) that all the nodes experienced significant displacment as much as 5.5 mm, whereas the original in qs1.1 (part A) only had displacements about 4.4 x$10^{-4}$ m. This increase of a factor of 10 (roughly) demonstarates that the redeisgn may not be a successful idea if using the same boundary conditions as the original but with a square cross-sectional truss and a reduced cross-sectional area.

Concluding remarks to part 7

The figures in part 7- A and 7-B the demonstrate that the redesign may not be a good decision as the stresses and displacements increased significantly for both options. Although the contour plots are scaled and hence the movement exxagerated, it still displays that the nodes were physicallly shifted from their original position which would compromise the strucutral integrity.