A

Practical activity Report submitted for

Computer Aided Design & Analysis (UME411)

Online Test 02: Analysis of Air Vessel

THAPAR INSTITUTE OF ENGINEERING AND TECHNOLOGY,

(A DEEMED TO BE UNIVERSITY), PATIALA, PUNJAB

INDIA

Feb-June 2021

ACKNOWLEDGEMENT

This project would not have been possible without the guidance and support of my course coordinate and Lecturer Dr A S JAWANDA. With his guidance we learnt manier projects which would be very useful as industrial knowledge in future. With is guidance I am able to bring this project .

INDEX

Introduction

This project features the one of the most important components of Feed Pump i.e. **Air Vessel** which is used in reciprocating feed pump in a Boiler.

An Air vessel is bolted to the top of the Valve chest and its main function is to maintain uniform flow of water and to save the power required to drive the pump.

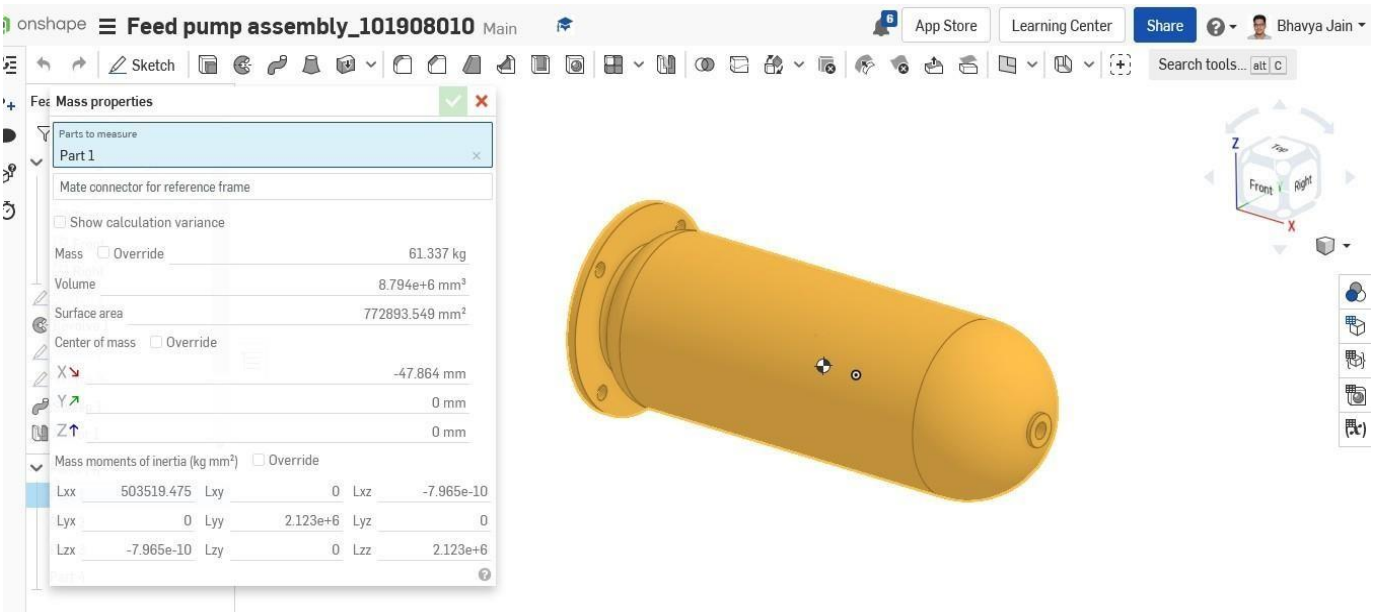
The pump has to develop a pressure of 12 bar to the boiler having volume of 500L.

Basically, we'll be doing analysis on the basis of parameters assigned to us.

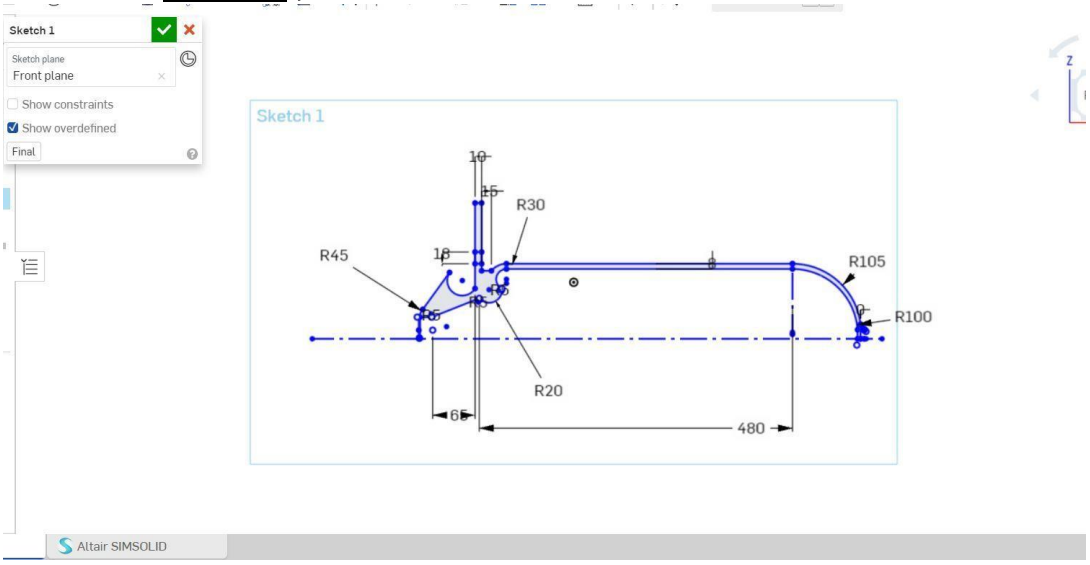
Image Reference website :
https://www.google.com/url?sa=i&url=http%3A%2F%2Fbulkmaterialhandlingequipments.com%2Fproduct%2Fair-vessel%2F&psig=AOvVaw0lPB_dkgJRJrLXEYa-4B32&ust=1622989290454000&source=images&cd=vfe&ved=0CAIQjRxqFwoTCPj3xMLYgPECFQAAAAAdAAAAABAE

ANALYSIS

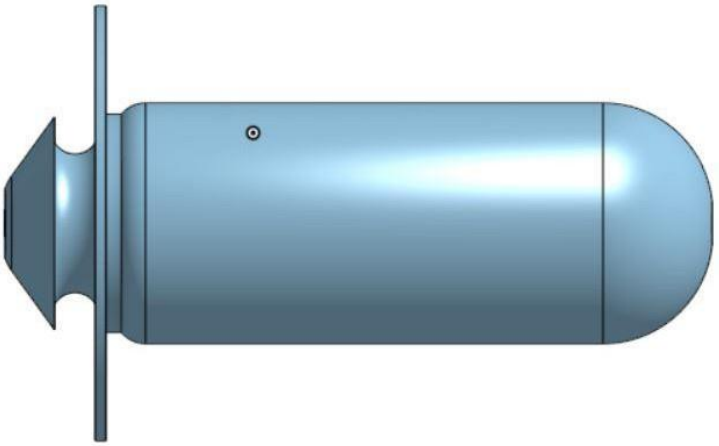
I first checked the mass of initial model, which comes out to be **50.33 kg**and the model looks like this.



And the **Sketch** ,

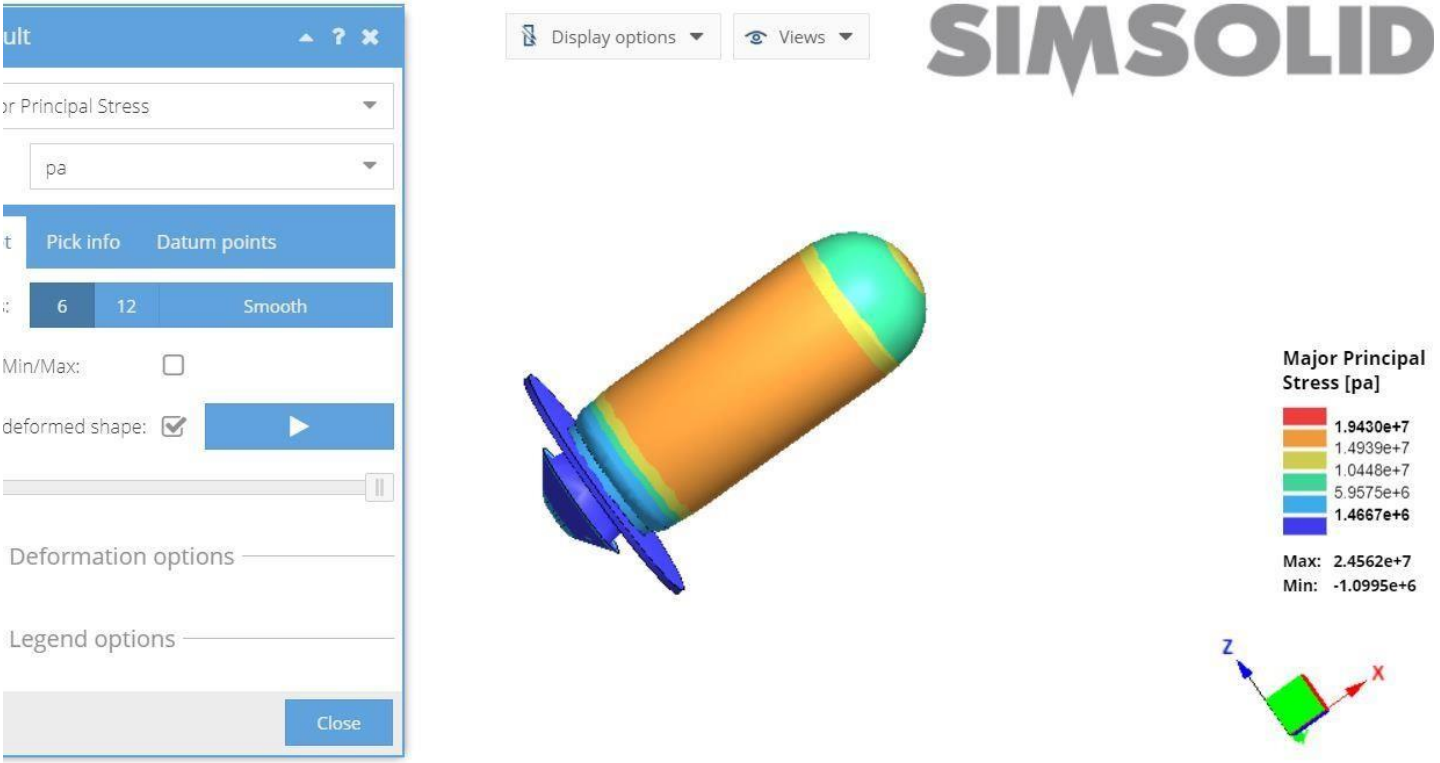


Now ,firstly we closed holes of air vessel from both sides ,



I tried different variation of thickness (6mm,5mm…) but then found my optimum stress distribution on thickness of 4mm.

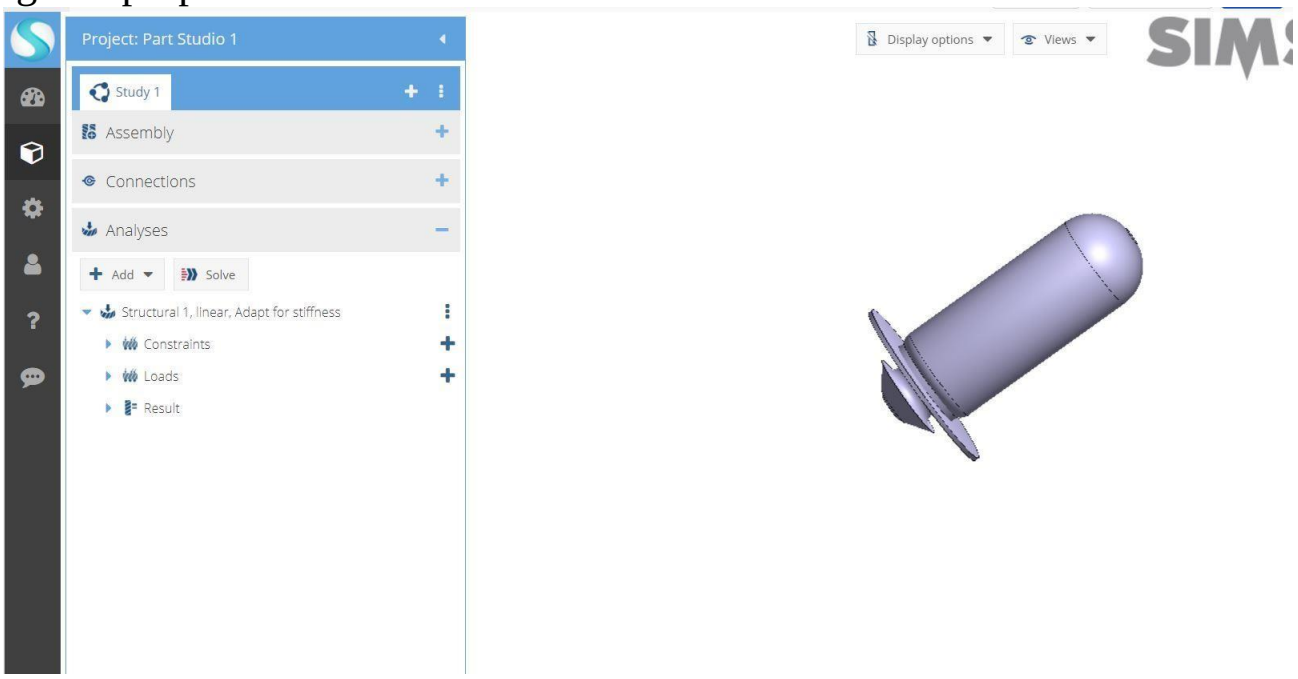
I have done few of my iterations of Analysis on Altair solid here they are:



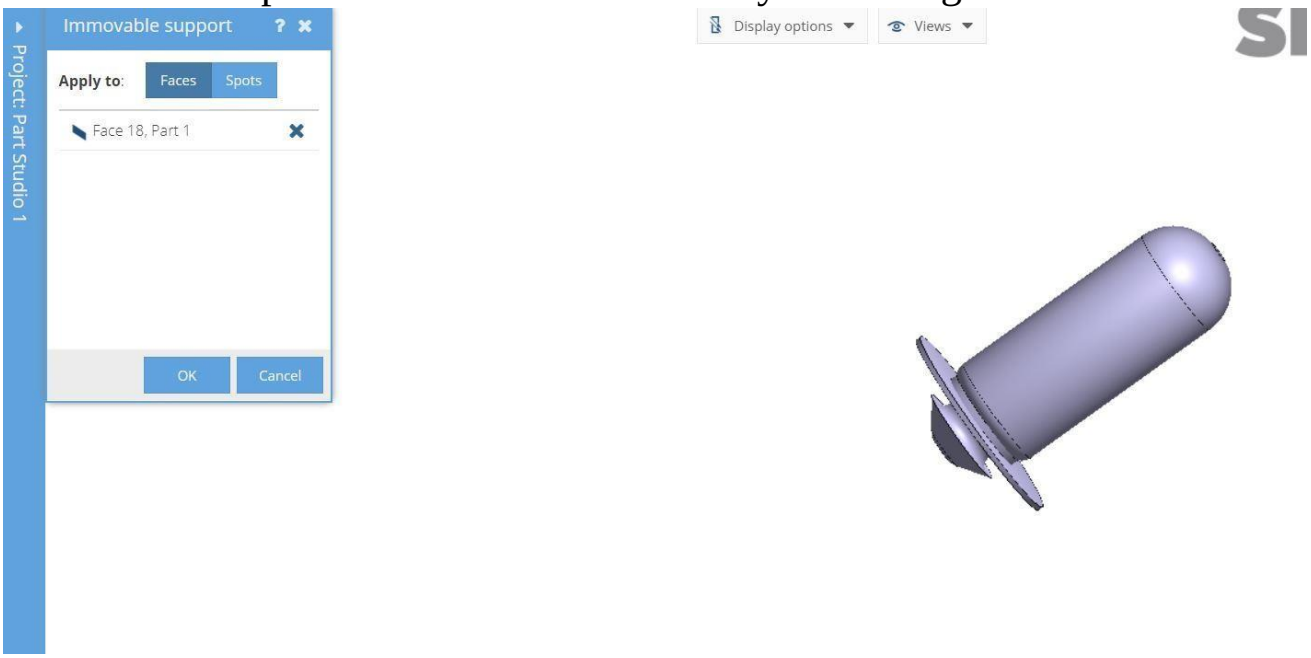
Keeping thickness around 6mm gave a max stress of 194.30Mpa. The mass was reduced to 19 Kg, But the stress distribution is not completely uniform.

So I reduced the thickness further to **4mm,** and Now we moved to **Altair Simsolid** to perform iterations.

Here we add our Air Vessel for analysis and define a material Cast iron with given properties.



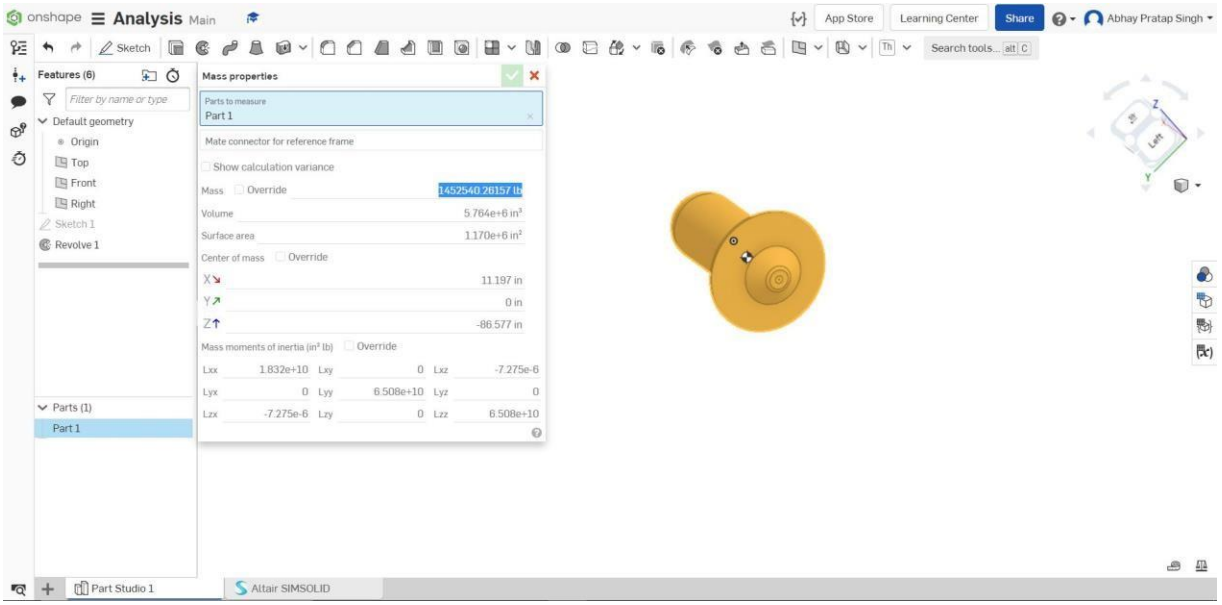
Now in Analysis section, we go to Constraints and select **Immovable support** and fixed that part of Air vessel as shown by dots in fig. Below:

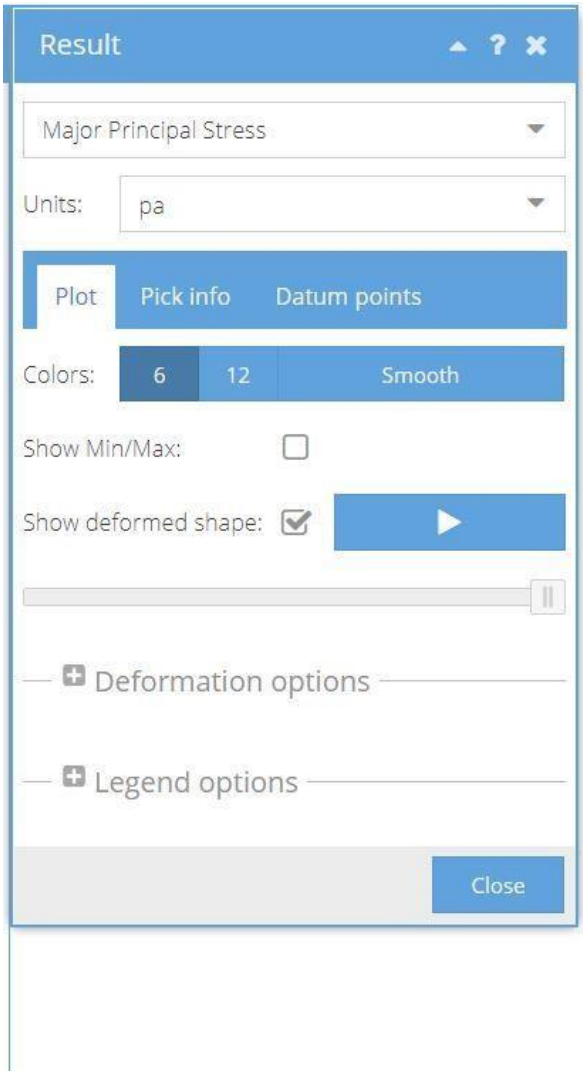


Now we go to **Loads** and select **Pressure** and by selecting inner and outer parts of Air vessel, we give **12 Bar** and **1 Atm** pressure respectively.

d

Now we move to **Result** Section and we select **Major Principal Stress** and result is as shown in fig:

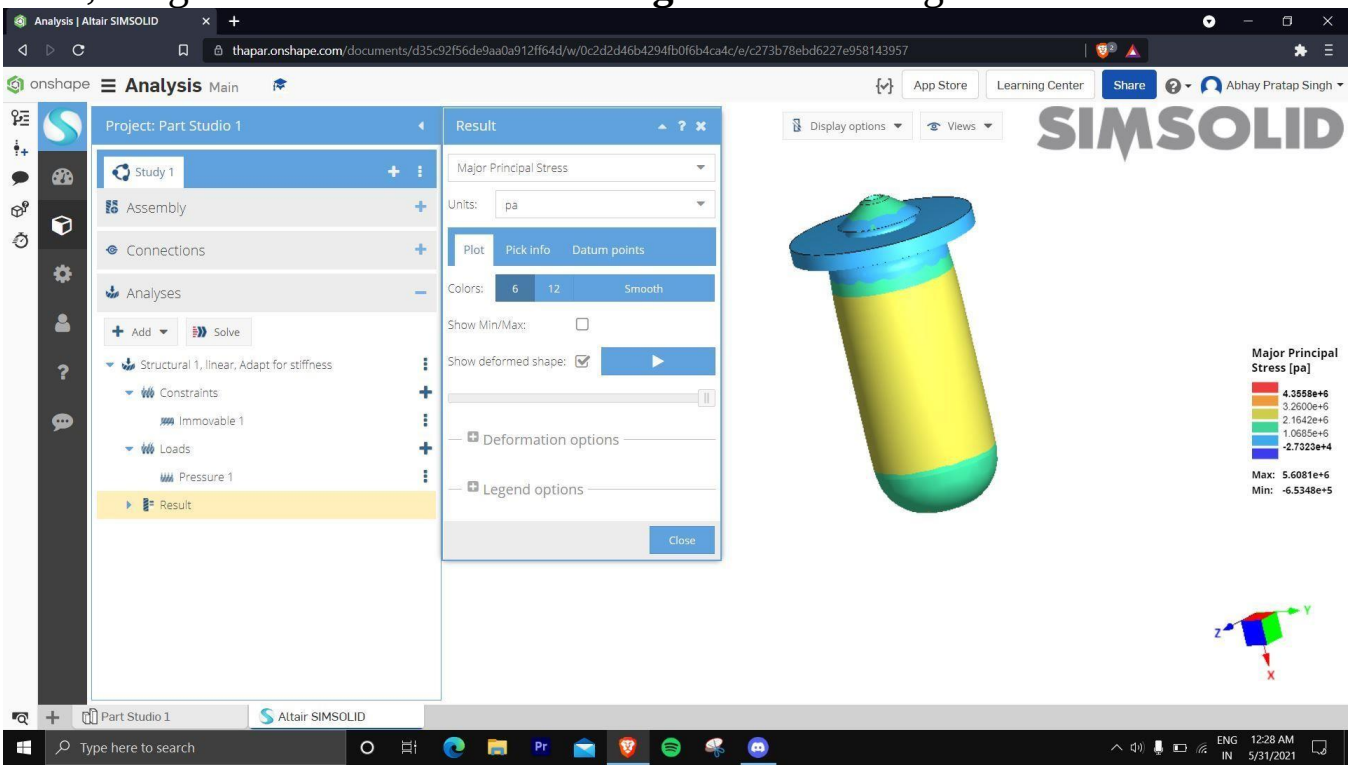


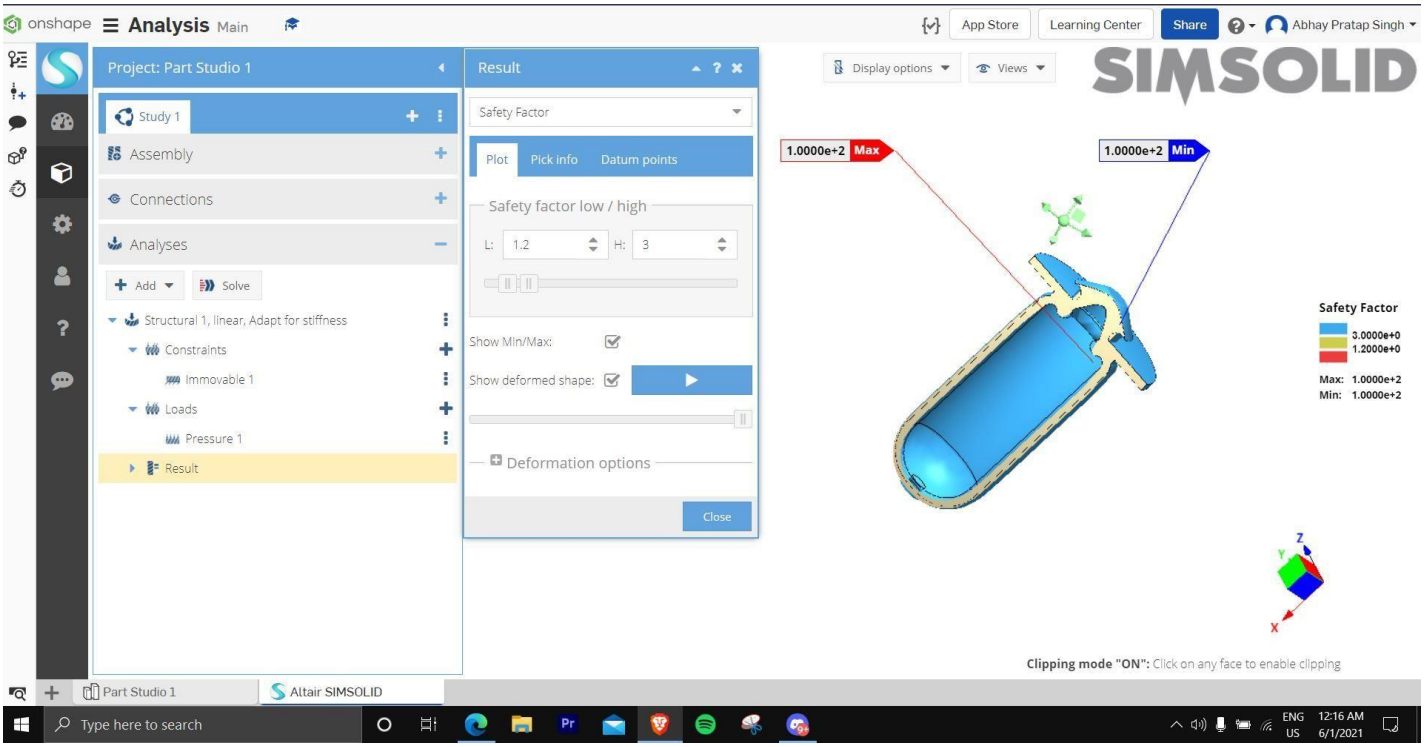

Result
Major Principal Stress
Units:
pa
Plot
Pick info
Datum points
Colors:
6
12
Smooth
Show Min/Max:
Show deformed shape:
Deformation options
Legend options
Close


Display options
Views
SIM

This gives a more **uniform stress distribution** here the max stress comes out to be **24.572MPa** which gives a FOS of **1.0000 e+2**.

Now, we get reduced Mass as **14.77Kg** as shown in fig below:





REFLECTIONS

I. Learnings

This project helped us expand our knowledge of FEA analysis on different platforms like SIMSOLID. SIMSOLIDS is an online platform hence more reliable as at autosaves on network itself which is further ost more efficient I am more confident now to perform FEA of any model not only on CREO but on different platforms as well. This project also tested our engineering IQ. Study through the analysis that how the model actually behaves upon defining different constraints and to observe that which part of the model is experiencing max or min stresses. A lot of knowledge is gathered in this project

II. The effect of other components in the assembly on the design of the air vessel?

- ✓ The Valve chest needs to have the same radius as that of the air vessel opening so that it can fix on top of the valve. Thus the diameter of the top hole of Valve chest effects the air vessel design.
- ✓ Plunger is used to decrease inside pressure of air vessel.

III. What other loading conditions are relevant for the Air vessel design which are not considered in this simulation?

One atm pressure should be applied.

CONCLUSION

I concluded from this project is that before concerning about the FOS we should check for uniform thickness of geometry. We should get a reliable and correct FOS (Factor of safety) as a final result

The Factor of Safety is directly proportional to the stress acting on the model which depends on the thickness of the model.