APPLIED COMPUTATIONAL MODELLING PROJECT REPORT

Topic: CFD And FEA Analysis of Initial and Final design sample Bench

Tutor: DR.Mohammad AL-Rawi

Student: Tony pauly

Student ID:19490954

Contents

List of Figures

List of Tables

1. Aim:

Aim of this project is to do FEA and CFD simulation on Bench on solid works and find out the draw backs. Then redesign the selected solid works model and compare the results with the old model.

2. Introduction:

Solid work is a software for designing the mechanical parts and runs the designed samples and comparing the results with the real-world conditions. In the solid works software, we are using the two different type of analysis

I. CFD analysis (Computational fluid dynamics)

Computational fluid dynamics (CFD) is a branch of fluid mechanics which analyses and solve problems involving fluid flow using numerical analysis and data structure. Computers are used to perform the calculations needed to simulate the fluid's free-stream flow, and the fluid (liquid and gases) interaction with boundary conditions specified surface. Better solutions can be found with high-speed supercomputers, which are often necessary to solve the biggest and most complicated problems.

II. FEA analysis (Finite Element analysis)

FEA is the process by which simulation technology is used to check how a product design responds to physical effects such as bending, heat, vibration, fluid flow and other consequences. With FEA simulation tool, designs can be evaluated early in the design cycle, determining what will cause premature failures, exploring design change quickly to reduce cast and weight and determining the safety factor of the product.

Here in this project, outside bench is the selected model which is downloaded from the grab CAD software. So, then the next step is to open the bench model in solid works and runs the model in the CFD analysis under a specified boundary condition. After that, the model is analysed by using FEA analysis. After that, the results are evaluated and finds out the causes of the failure. Then the model is redesigned for resolving the factors effects the failure. The results of the redesigned bench are compared with the bench which is downloaded from the grab cad.

3. Method

a. Computational fluid dynamics (CFD) Method

1. Open the Bench file by using the solid works software.
2. Then click on the **flow simulation**, then click on the **Wizard** which is in the top left corner.
3. After clicking the wizard button, a box will appear which is shown in the figure1

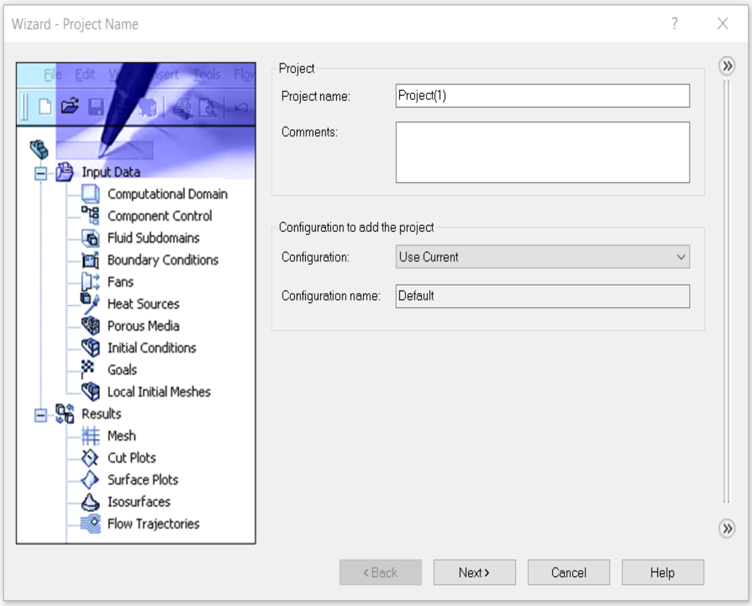


Figure 1: Project name

4. Then click on next, then click on SI units from the unit system in the dialog box and then click next.

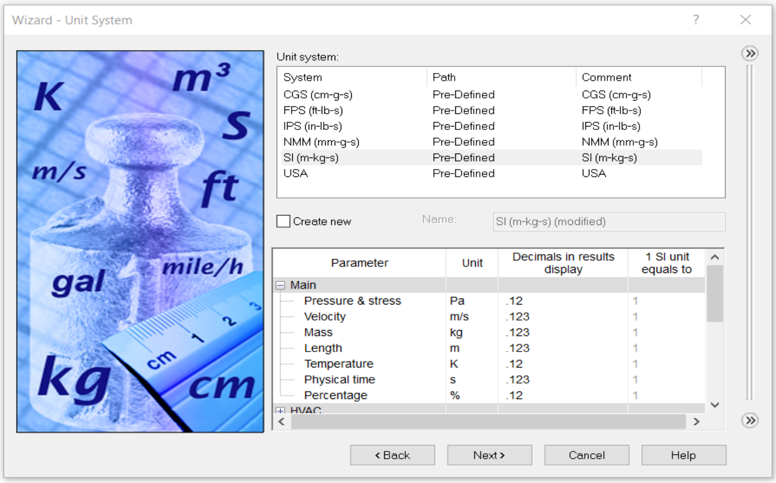


Figure 2: unit system

5. In this dialog box click on External
6. Click on Exclude cavities without flow conditions and Exclude internal pipes-> click on next

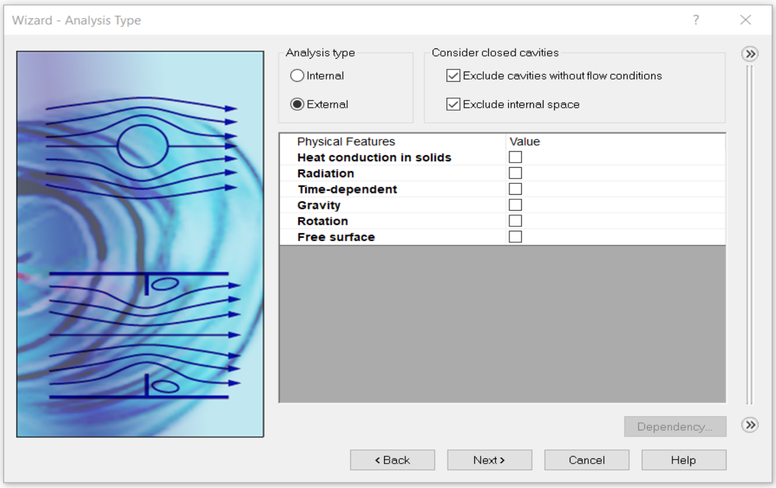


Figure 3: Analysis type

7. Here in this model we are selecting the air because these models are normally used in the plain areas like parks, near the lake side etc. so, in New Zealand the chances of high velocity wind are high.

So, click on the **gases** on the dialog box and select **air**-> then press **add** button -> click **next.** Which is shown in the figure 4 and 5.

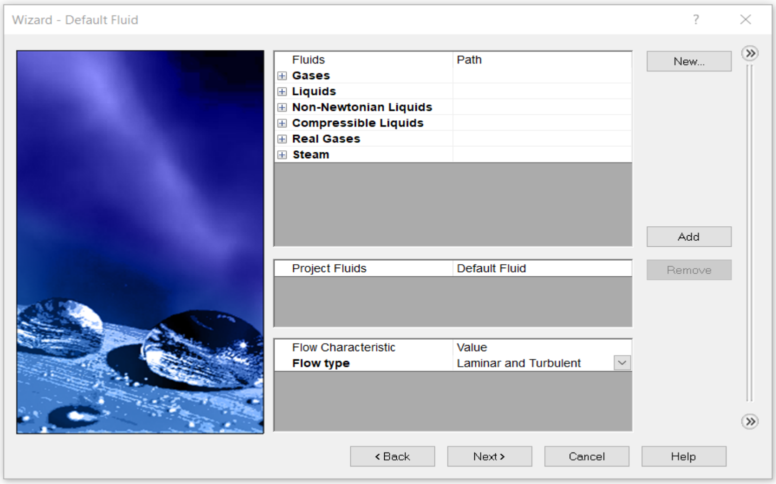


Figure 4

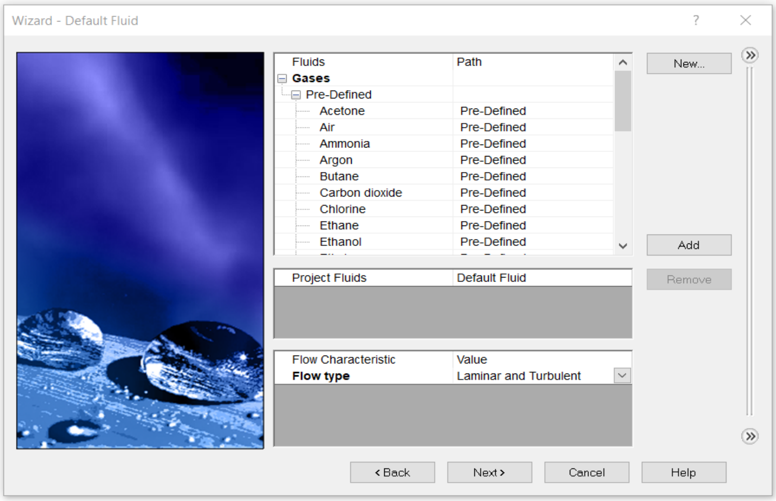


Figure 5: default fluid

8. In the parameters, there is a velocity parameter and the air (wind) can flow in z direction at 70m/s. Then the mark it as -70 m/s in velocity in z directions. Which is shown in the figure 6

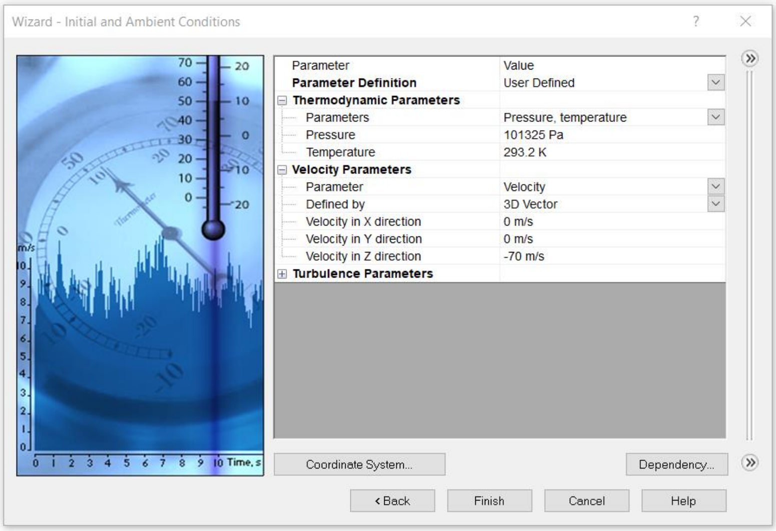


Figure 6: Assigning the values for air at a velocity in velocity parameters.

9. After completion of the wizard dialog box, a computational domain box will appear which is shown in the figure 7. If the domain is overlapping, then right click on the computational domain-> click edit definition. Then adjust the domain by using the arrows, which is shown in the figure 7

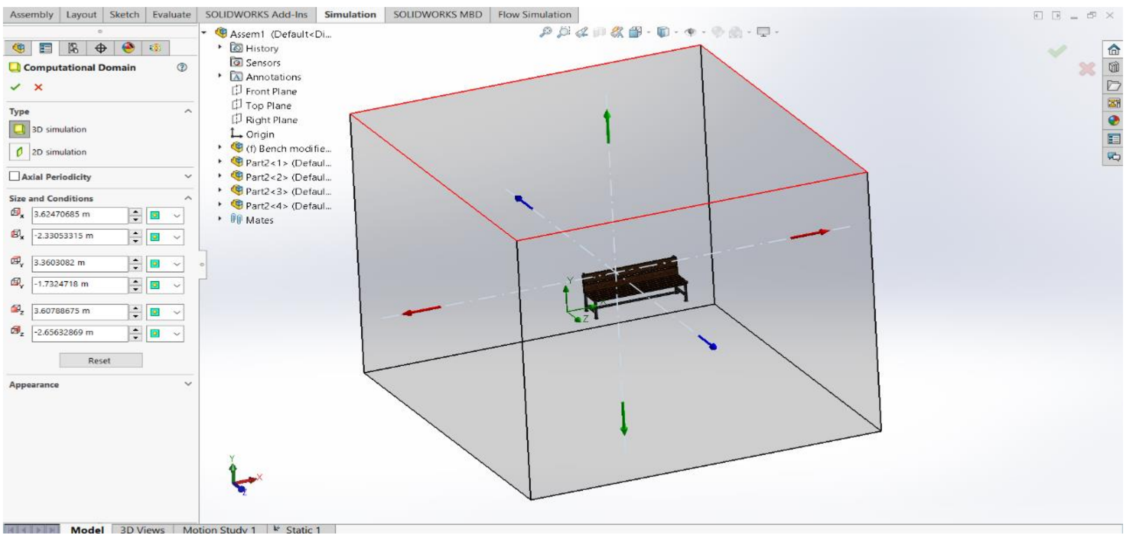


Figure 7: Computational domain

10. The next step is setting the goals by right clicking the goals, then select insert global goals. Then select static pressure, total pressure, density, average velocity, and force. Then click ok. Which is shown in the figure 8.

    Also, we need to add the equation goals for finding the drag force by right clicking the goals and select Equation goals. And add equation as (2*{drag force})/ (({Density of air} *{Velocity of air} ^2) *Area of the bench.

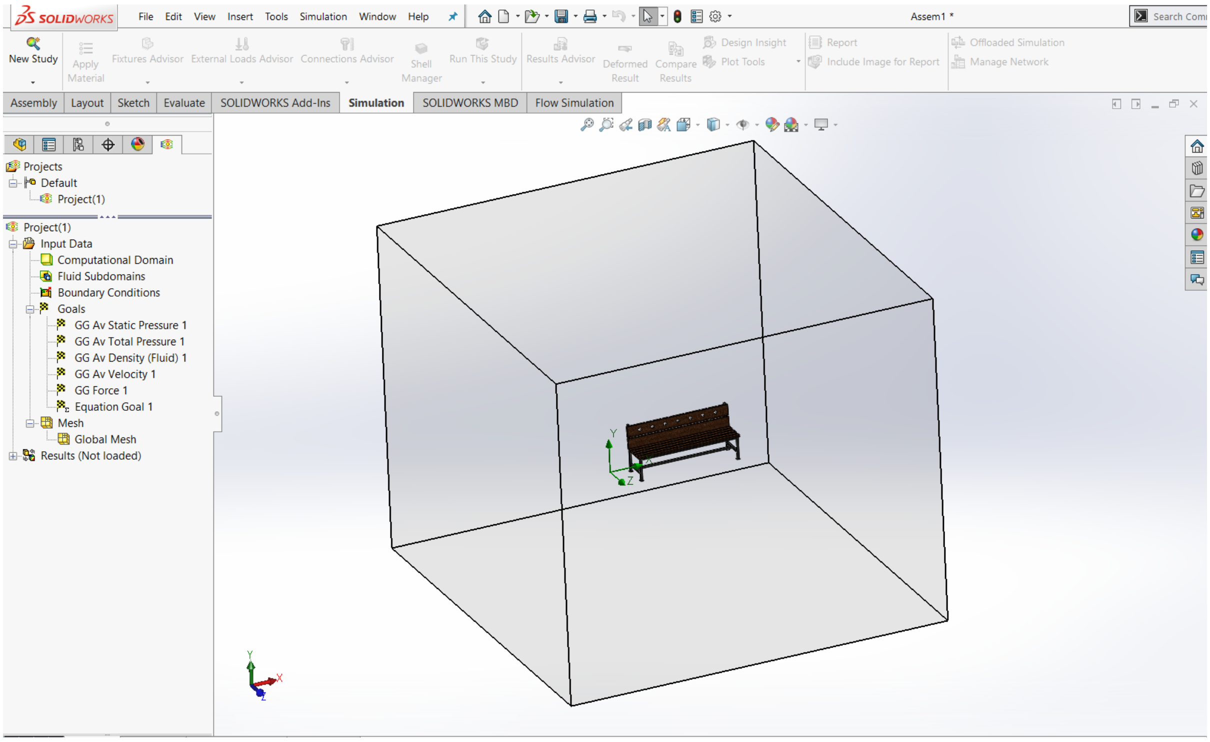


Figure 8: Adding equation goals

11. The next step is to run the simulation, by right clicking the Tools in the top of task bar, then select flow simulation-> select solve-> select Run (shown in the figure 9). After running the results, note down the reading in the results.

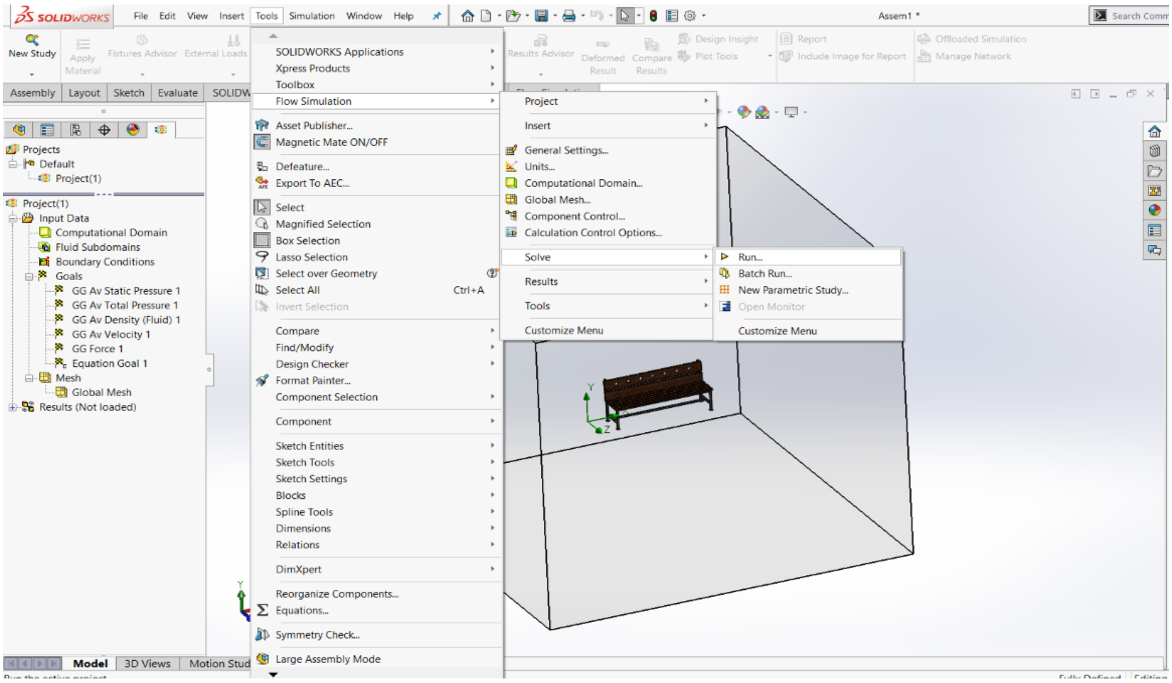


Figure 9: Running the Model

12. After running the results, then we need to export the flow simulation. So, select tools-> click on flow simulation-> click on Tools -> then click on Export results to simulation.

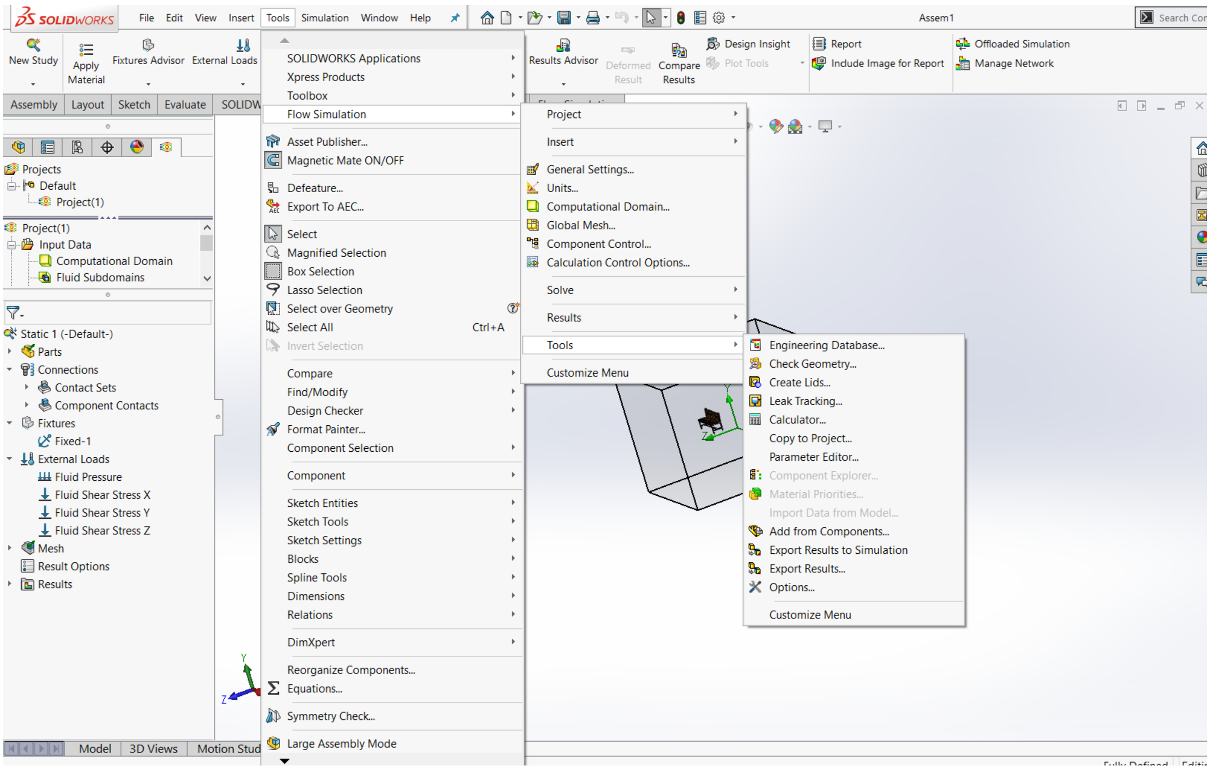


Figure 10: Export results to simulation

b. FEA analysis (Finite Element analysis) Method

1. To run the FEA analysis, first click on New study-> click on static -> click ok. Then the new study for FEA analysis will appear in the left side
2. Then the next step is to assign the material to the model (Bench). So, right clicking the parts, click the apply/edit material. Then select Delrin from the plastics then click apply, then click ok.
3. The main reason for selecting Delrin from plastics, because of no corrosive and low cost. Also, it last for long period of time.
4. Next step is the connections, right click on connections-> click on contact set, then the contact set dialogue box will appear, which is shown in the figure 11.

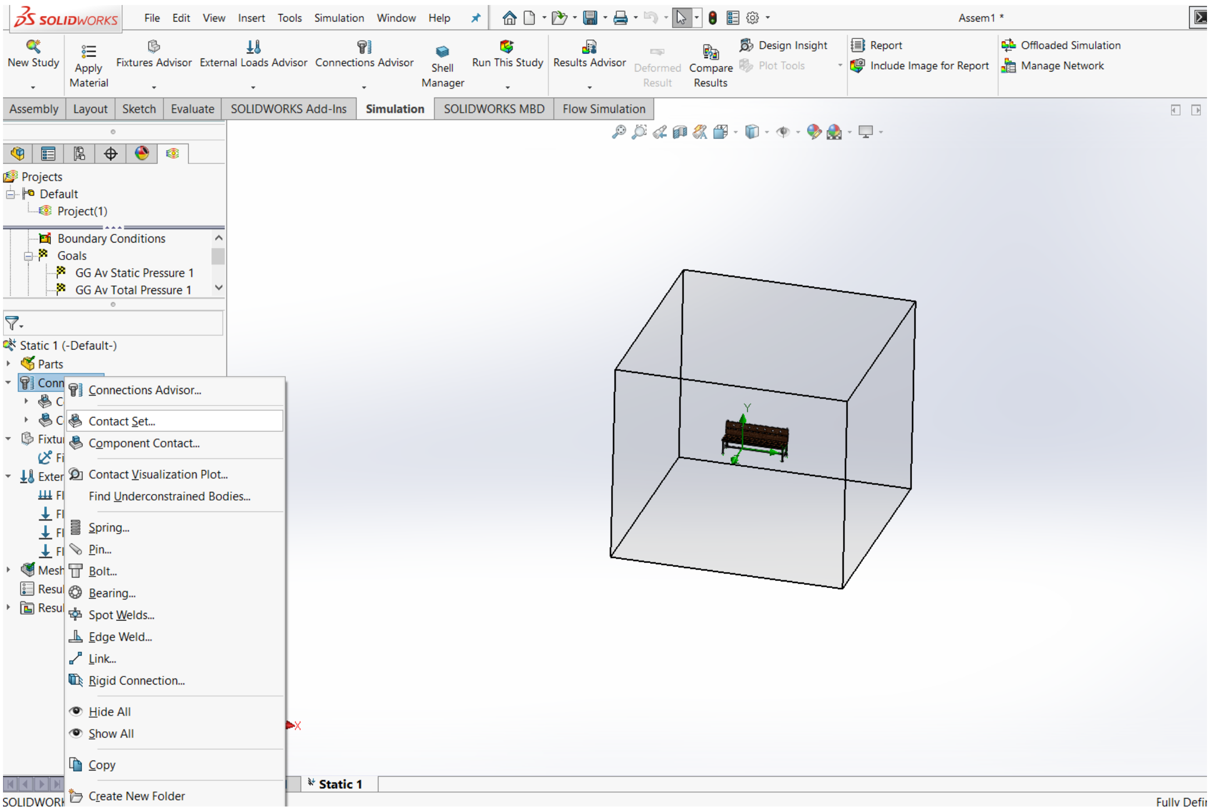


Figure 11: connections

5. Then select the **Automatically find contact** set -> in the components box click on **find contact set**. In the results box, convert No penetration into bonded then

select the four different components in the result box. Click the plus sign -> click ok shown in figure 12.

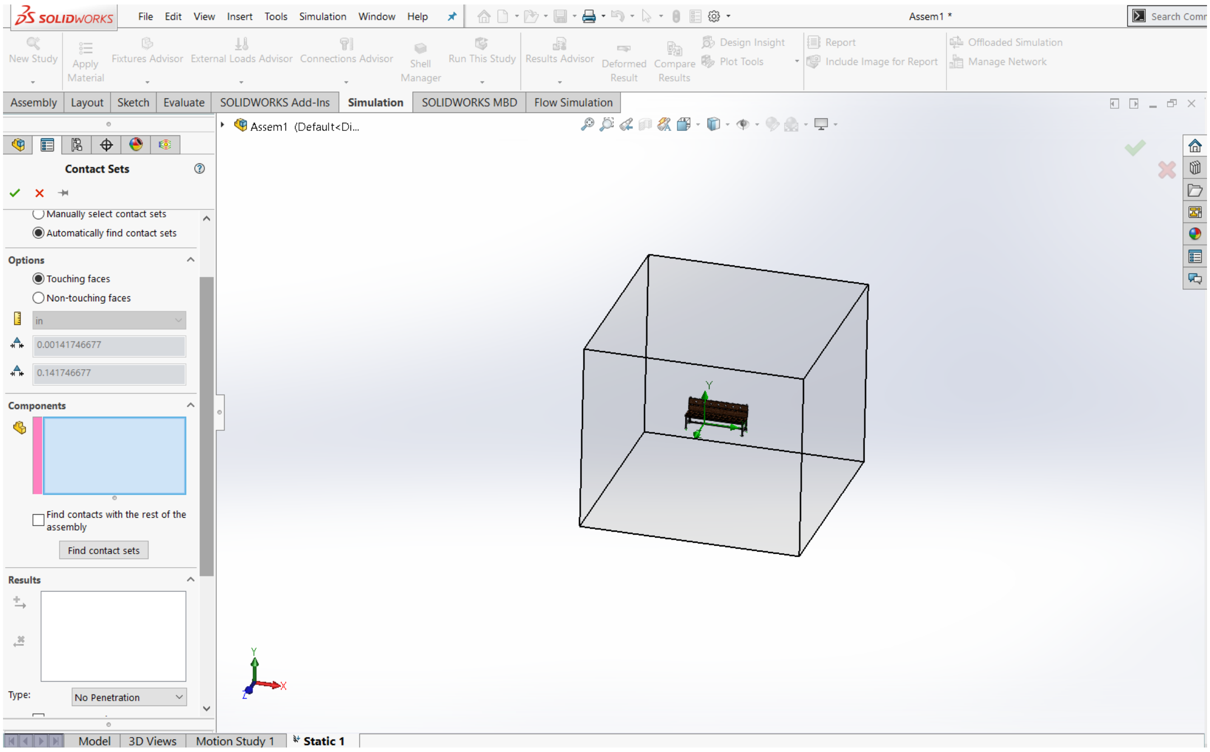


Figure 12: Contact set

6. Adding the fixtures, by right clicking the fixtures and select fixed geometry. Then the next step is to select the places to make fixture which is shown in the figure 13.

Figure 13: adding fixture geometry

7. After applying the fixtures, the next step is to add the external load by right clicking the External load -> select flow effects. Which is show in the figure 14.

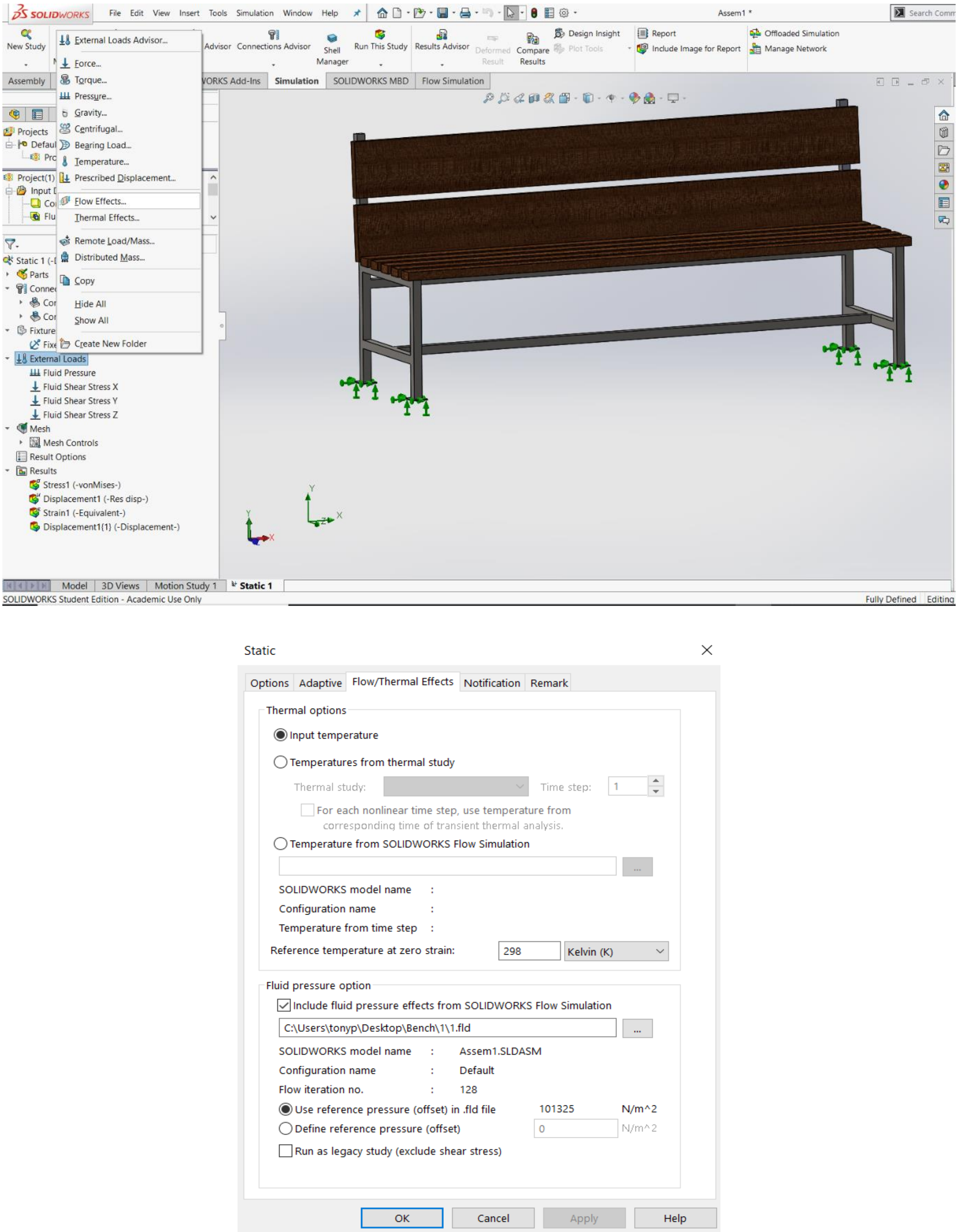


Figure 14: Applying the force

A dialog box will appear in flow/Thermal effects, in the fluid pressure options, make a tick mark in the include fluid pressure effects from SOLIDWORKS flow simulation. Then click doted button near the Fluid pressure option-> then select the 1.fld file -> click ok. Which is shown in the above figure.

8. The next step is to create the mesh to the selected model
   Right click on the mesh, then click create mesh. after that, a dialogue box will appear in add the mesh according to our needs. Then click ok.

Mesh Details

Study name	Static 1 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Curvature-based mesh
Jacobian points	4 points
Mesh Control	Defined
Max Element Size	2.74923 in
Min Element Size	0.549847 in
Mesh quality	High
Total nodes	8558
Total elements	3407
Maximum Aspect Ratio	16.027
Percentage of elements with Aspect Ratio < 3	49.9
Percentage of elements with Aspect Ratio > 10	0.998
% of distorted elements (Jacobian)	0
Remesh failed parts with incompatible mesh	Off
Time to complete mesh(hh:mm:ss)	00:00:02
Computer name	

a. Sample model mesh details

Mesh Details

Study name	Static 1 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Curvature-based mesh
Jacobian points	4 points
Mesh Control	Defined
Max Element Size	2.83493 in
Min Element Size	0.566987 in
Mesh quality	High
Total nodes	56788
Total elements	28171
Maximum Aspect Ratio	34.222
Percentage of elements with Aspect Ratio < 3	93.5
Percentage of elements with Aspect Ratio > 10	0.0923
% of distorted elements (Jacobian)	0
Remesh failed parts with incompatible mesh	Off
Time to complete mesh(hh:mm:ss)	00:00:05
Computer name	

b. Modified model mesh details

Figure 15 : Mesh Details

9. Final step is to run the model at given conditions, by clicking **Run this study.** Then note down the readings from the results.

4. Design specification:

a. Sample model

Here in this project, outside bench is the selected model which is downloaded from the grab cad. So, here in this model both the CFD and FEA is analysed. In the CFD the air the moving gas. We can see the different type of benches are installed in parks, near the lakes etc. so, sometimes a high velocity wind can blow through the plain surfaces like parks, near the lakes or rivers. Therefore, it is important to analyse the flow by using the CFD and FEA analysis and the model is drawn in the 2D drawings, which is shown in the figure 32. Also, the material which selected is the plastics which is not corrosive and long lasting.

b. Modified model

After getting the results form the sample bench model. The next set is to redesign the model with improved efficiency. So, here I have done some modifications which is shown in the second image of figure 32.

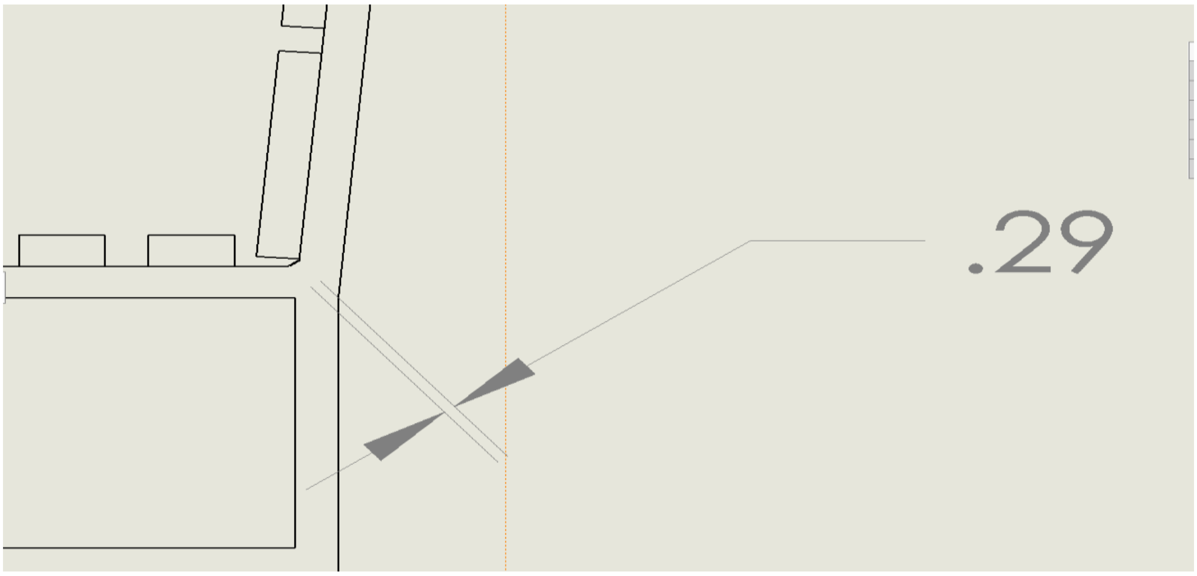


Figure 16:Modified model

5. Preliminary Design Drawings:

It is shown in 2D, which is shown in the last page

6. Results

6.1 FEA Analysis

6.1.1 Sample model without modification

a. Von Mises Stress

Maximum Von Mises stress- 7.588e+01 Mpa

Minimum Von Mises stress- 3.19e-03 Mpa

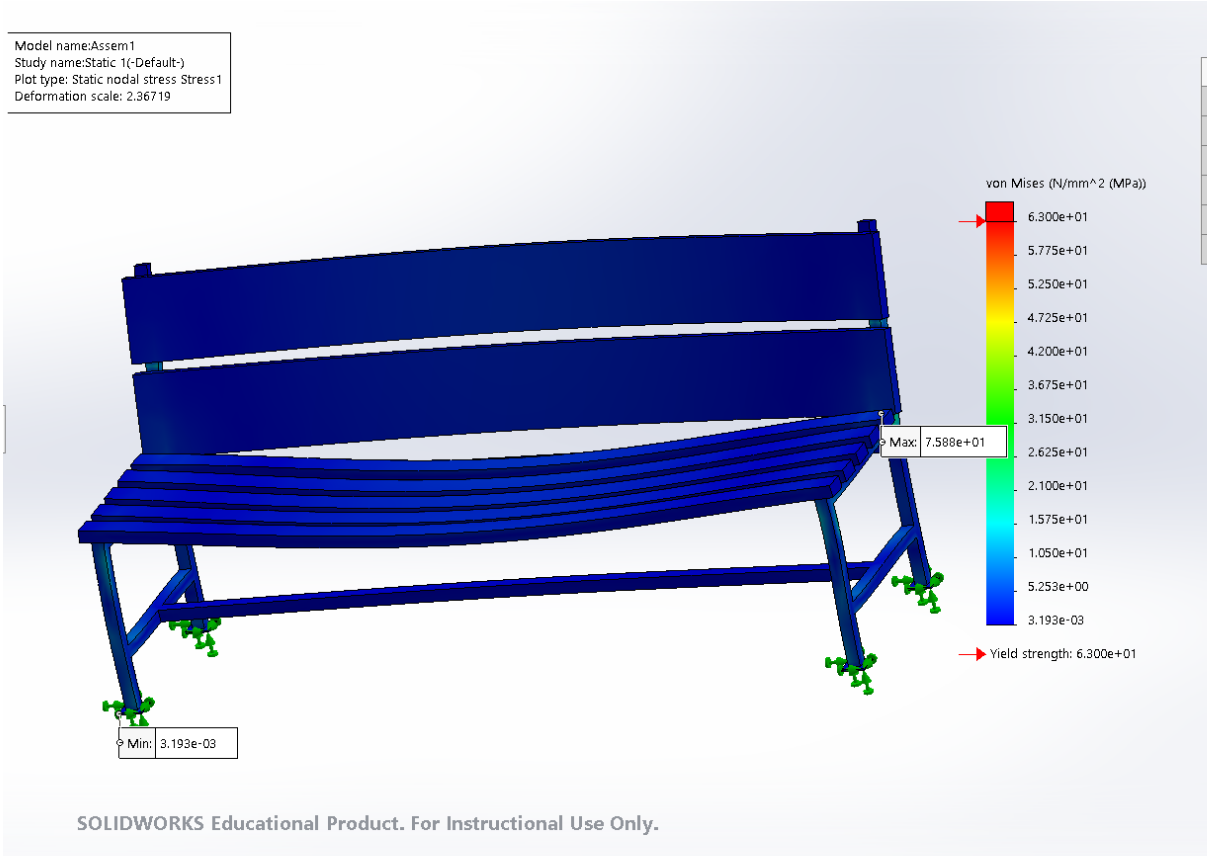


Figure 17: Von mises stress

b. Resultant displacement

Maximum Resultant displacement – 6.740e+07 mm

Minimum Resultant displacement – 1.00e-30 mm

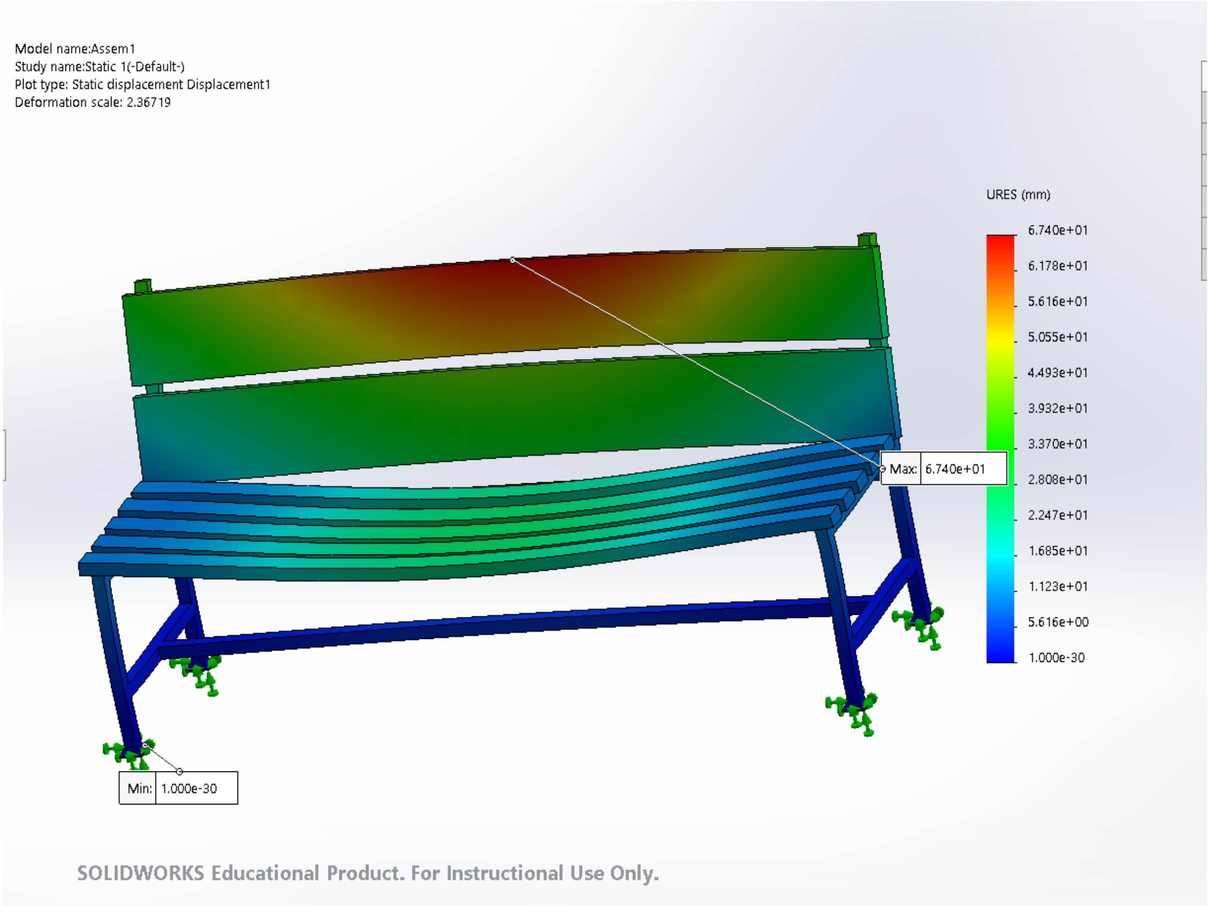


Figure 18:Resultant displacement

c. Equivalent strain

Maximum Equivalent strain – 7.820-03

Minimum Equivalent strain – 1.443-06

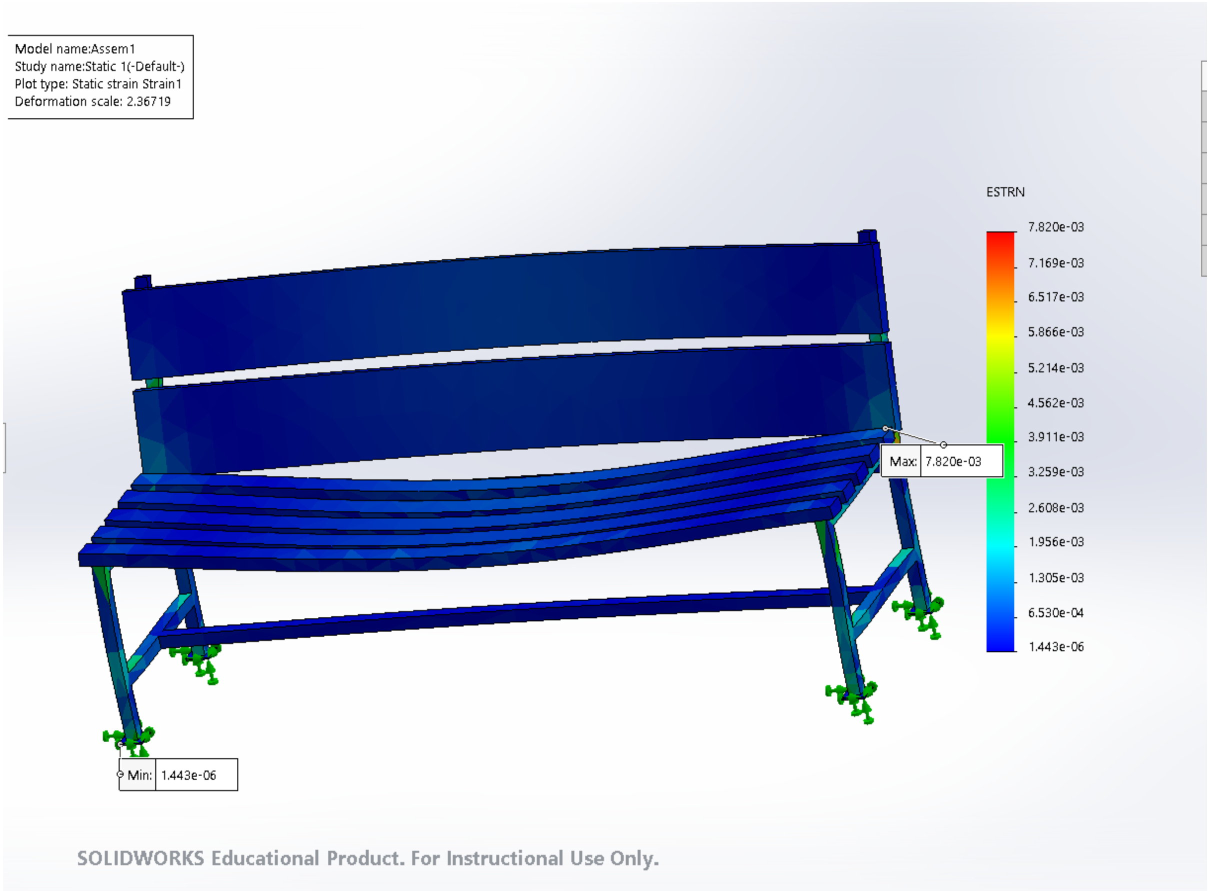


Figure 19:Equivalent strain

6.1.2 Modified model

a. Von Mises Stress

Maximum Von Mises stress- 3.149e+01 Mpa

Minimum Von Mises stress- 7.320e-04 Mpa

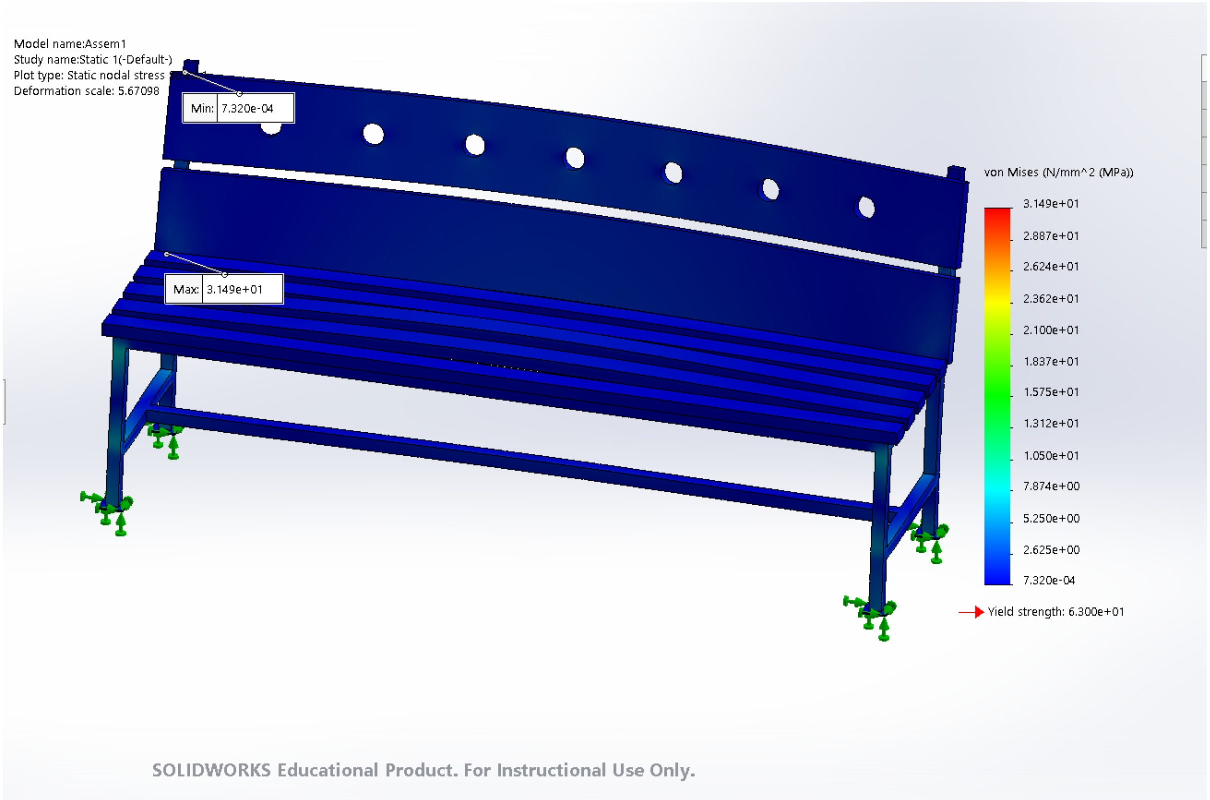


Figure 20:Von Mises Stress

b. Resultant displacement

Maximum Resultant displacement – 2.805+01 mm

Minimum Resultant displacement – 1.00e-30 mm

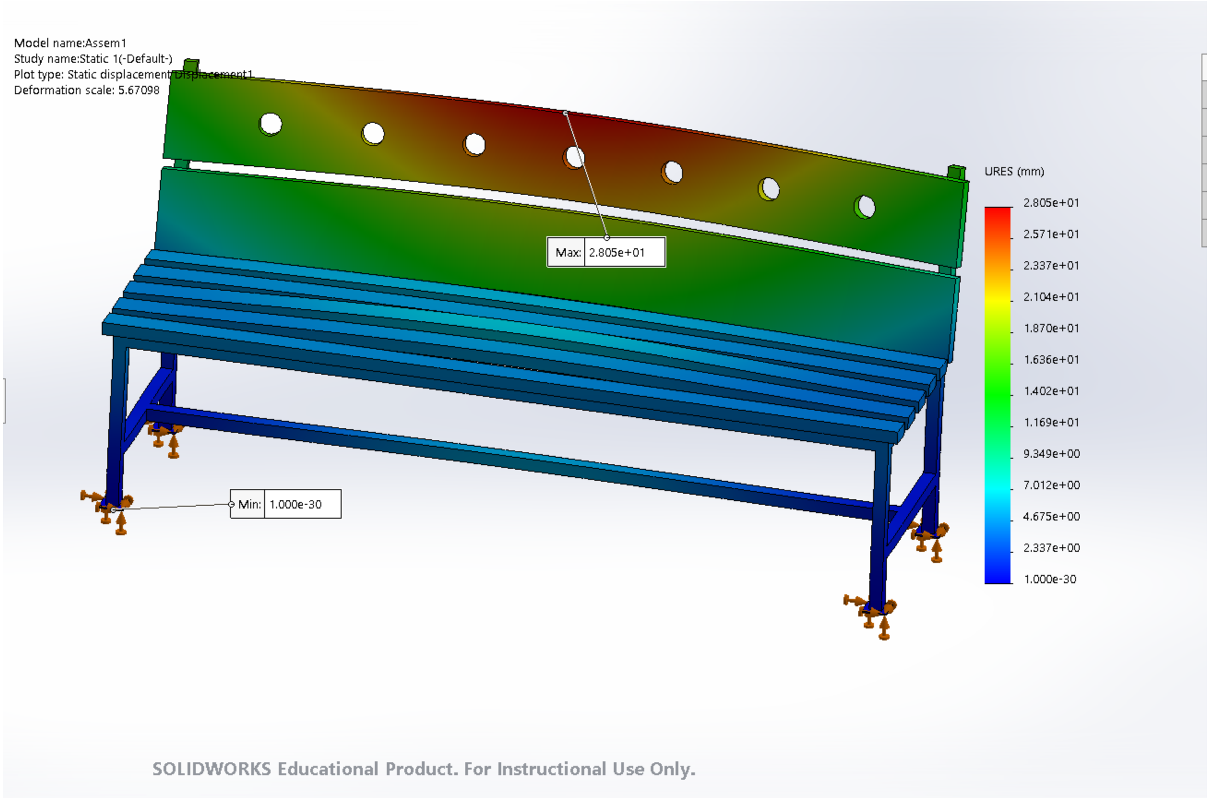


Figure 21:Resultant displacement

c. Equivalent strain

Maximum Equivalent strain – 7.820-03

Minimum Equivalent strain – 1.443-0

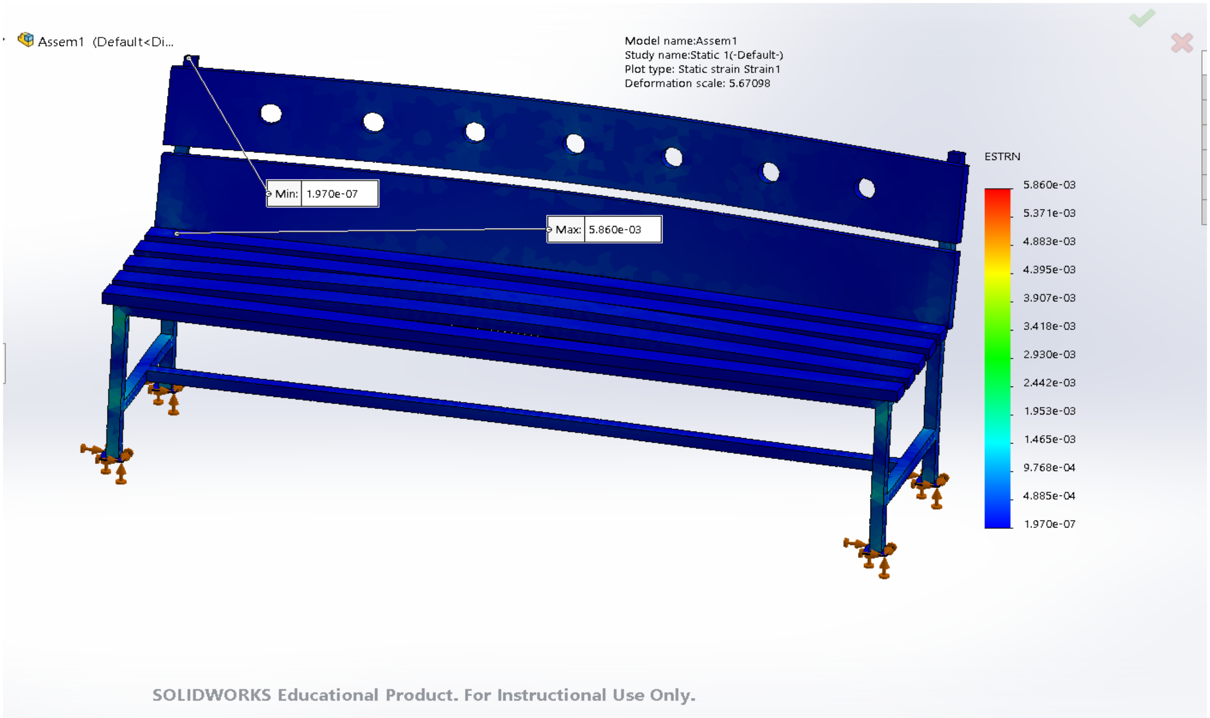


Figure 22:Equivalent strain

6.2 CFD Modelling (Computational fluid dynamics)

6.2.1 Sample model CFD Result

1. Cut plot 1(velocity)

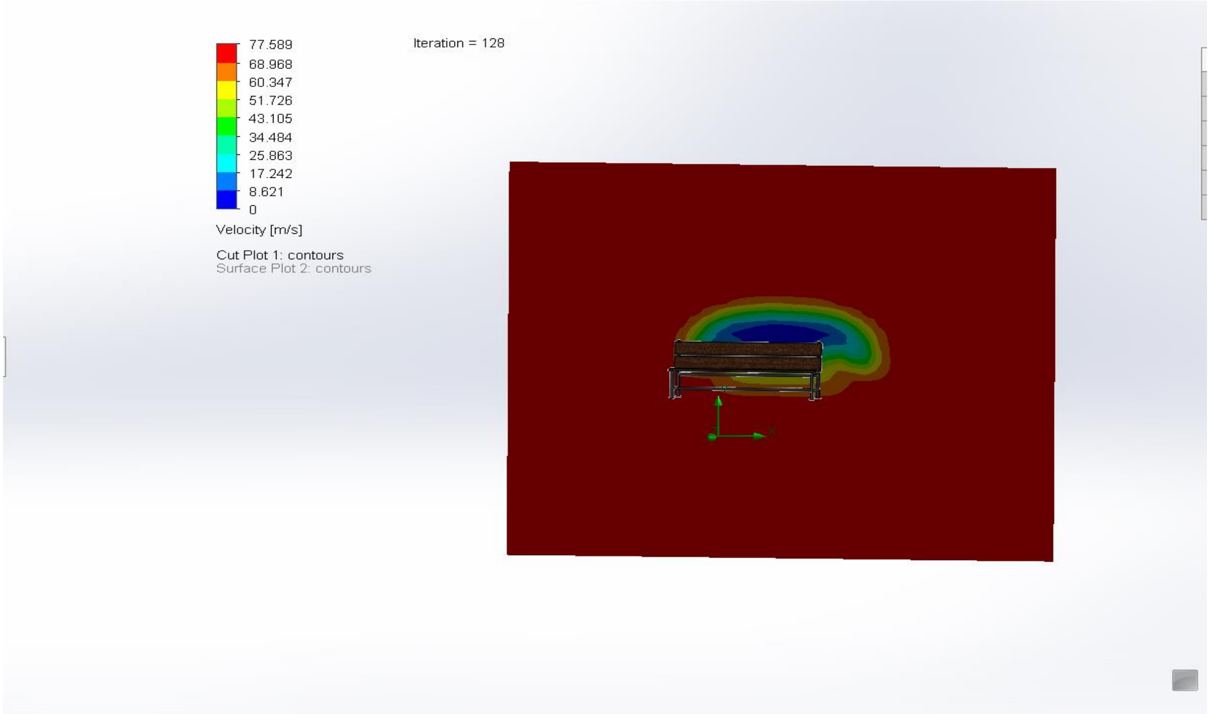


Figure 23: cut plot (Velocity)

2. Cut plot 2 (pressure)

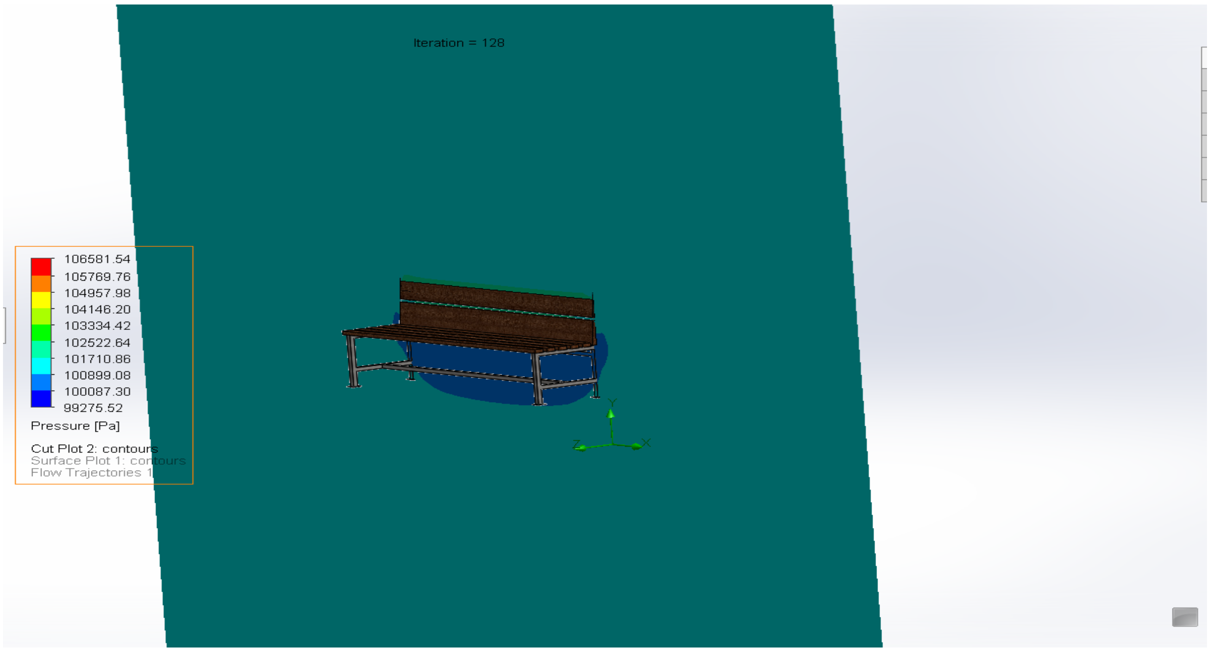


Figure 24: cut plot 2(velocity)

3. Surface plot 1(pressure)

Figure 25: surface plot 1(pressure)

4. Surface plot 2(velocity)

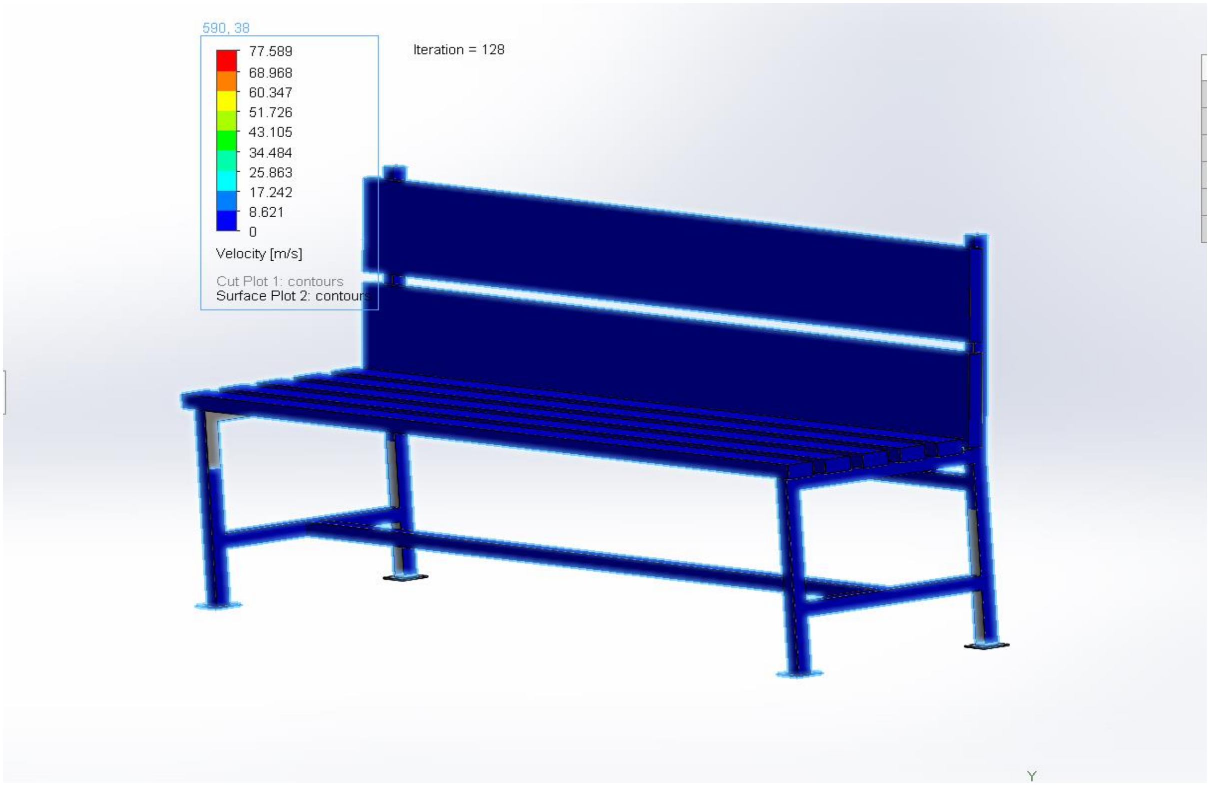


Figure 26: surface plot 2(velocity)

5. Flow Trajectory

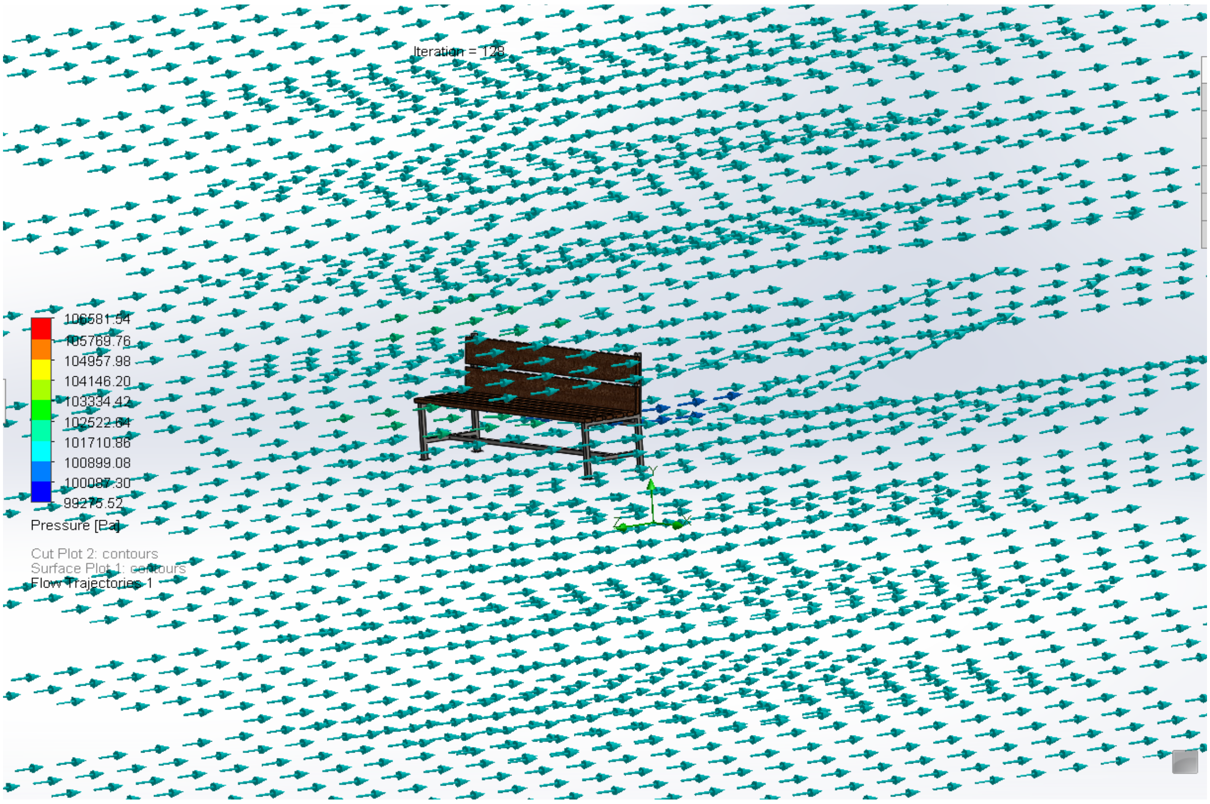


Figure 27: Flow Trajectory

6.2.2 Sample model modified CFD Result

1. Cut plot 1(pressure)

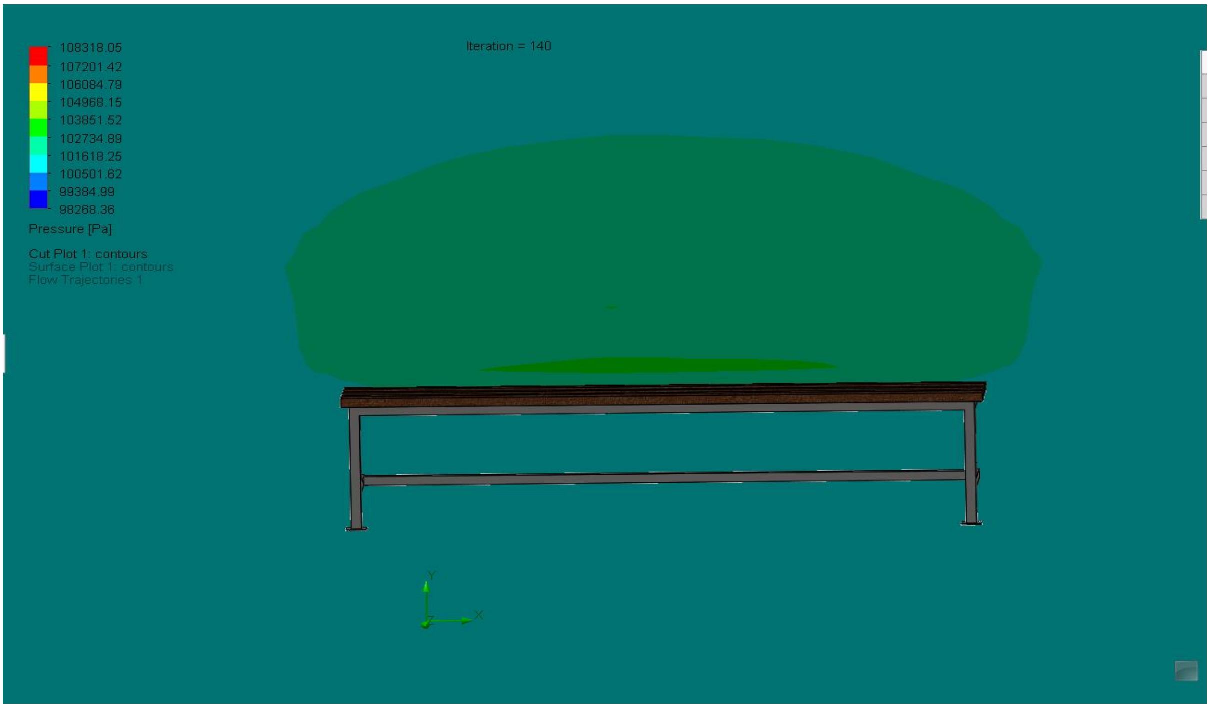


Figure 28: pressure (Modified Model)

2. Cut Plot 2(Velocity)

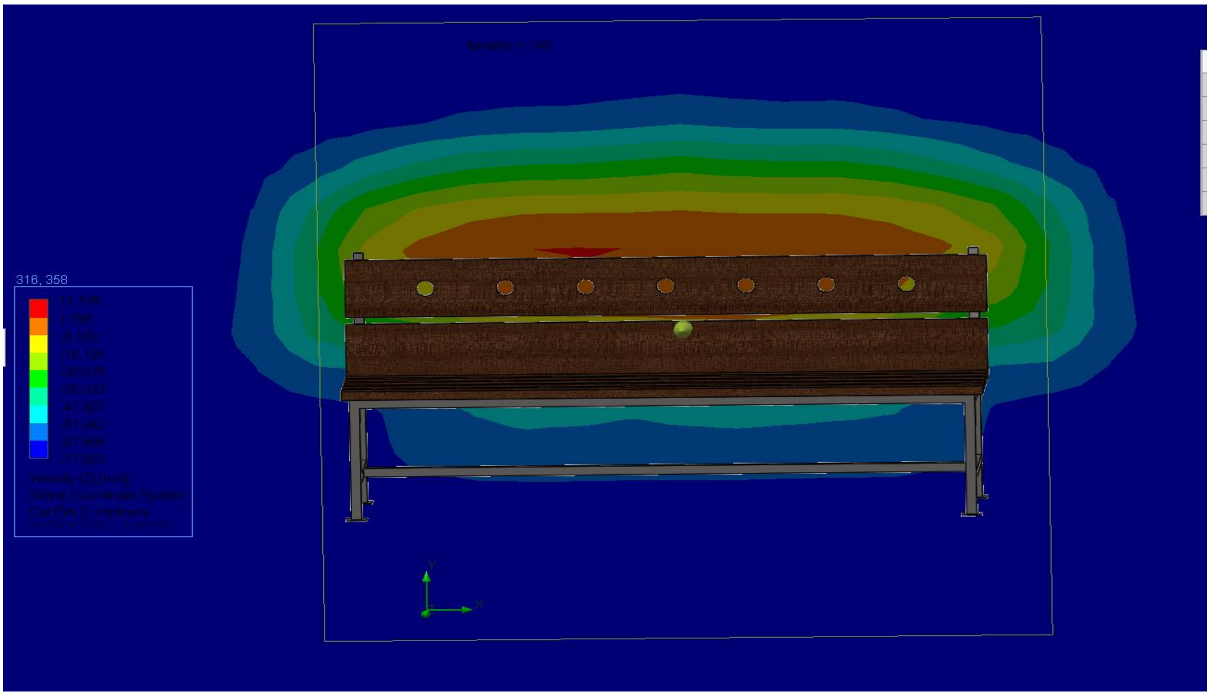


Figure 29: Pressure (Modified model)

3. Surface plot 1

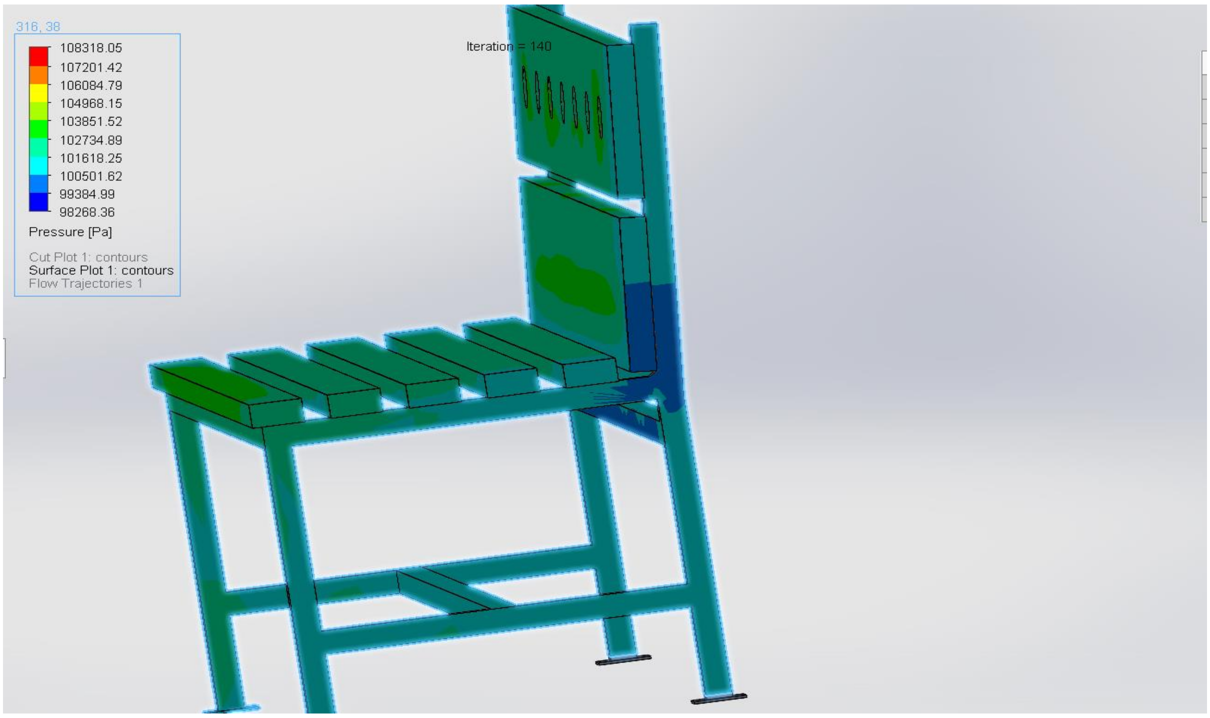


Figure 30: surface plot (pressure)

4. Surface plot 2 (Pressure)

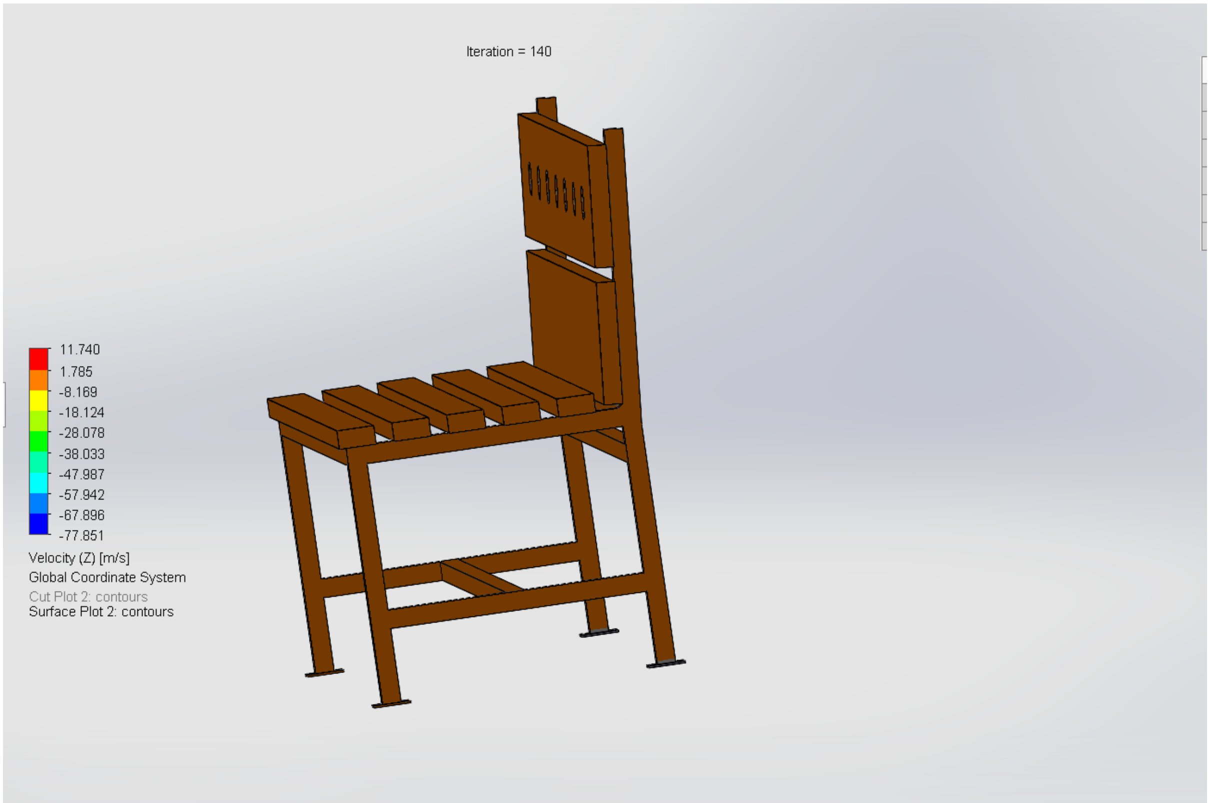


Figure 31: surface plot 2(velocity)

5. Flow trajectory

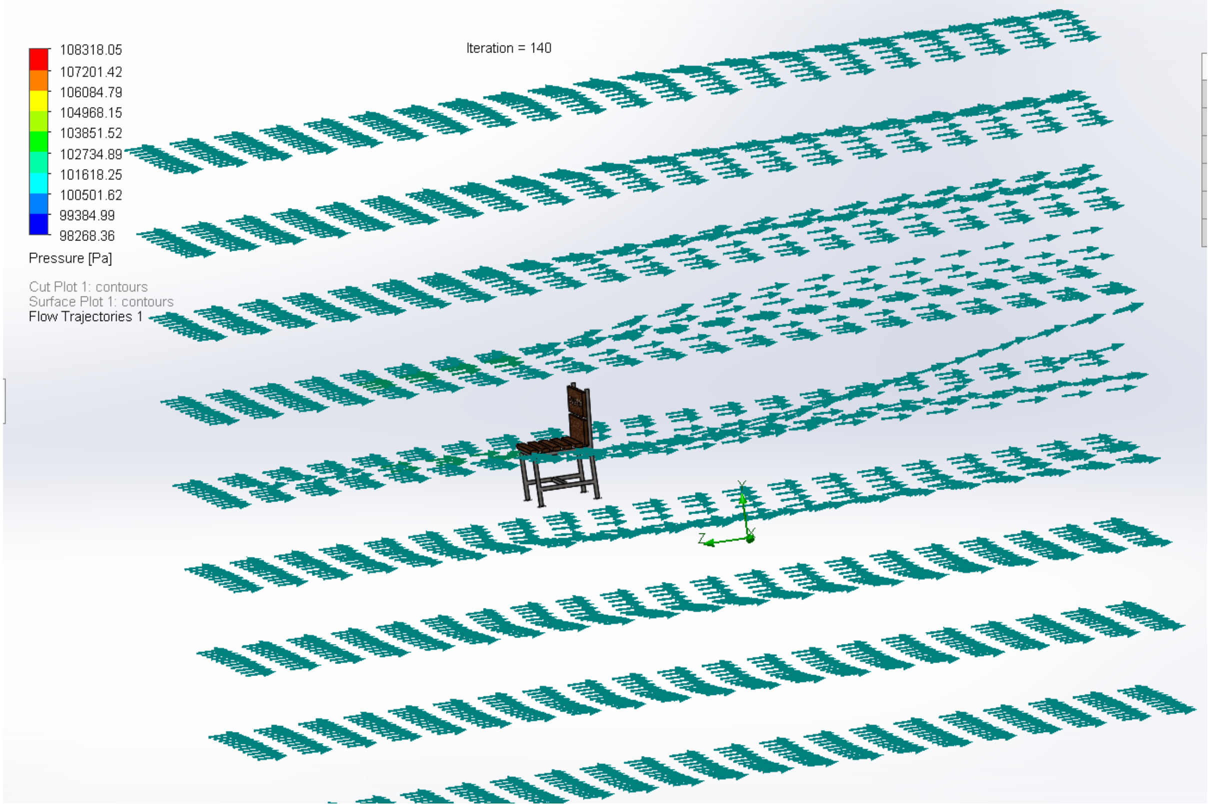


Figure 32: Flow trajectory (modified model)

6.2.3Goals plot of sample bench

Table 1: Goals plot of sample Bench

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use In Convergence	Delta	Criteria
GG Av Static Pressure 1	[Pa]	101314.3711	101314.3299	101314.1801	101314.6826	100	Yes	0.502485384	0.522552422
GG Av Total Pressure 1	[Pa]	104257.8239	104257.7257	104257.6108	104257.8859	100	Yes	0.275157154	1.609248161
GG Av Velocity 1	[m/s]	69.13696985	69.13422391	69.13003666	69.13756459	100	Yes	0.007527932	0.031135774
GG Force 1	[N]	1640.051764	1633.701736	1624.755348	1640.249723	100	Yes	15.49437527	145.3599459
GG Av Density (Fluid) 1	[kg/m^3]	1.203466962	1.203466312	1.203464829	1.203469335	100	Yes	4.50641E-06	9.34554E-06
Equation Goal 1	No unit	0.010003635	0.009965699	0.009912302	0.010005006	100	Yes	9.27048E-05	0.000857626

6.2.4 Goals plot of modified sample Bench

Table 2: Goals plot of modified sample Bench

Goal Name	Unit	Value	Averaged Value	Minimum Value	Maximum Value	Progress [%]	Use in Convergence	Delta	Criteria
GG Av Static Pressure 1	[Pa]	101317.3292	101317.369	101317.0009	101317.5378	100	Yes	0.440266326	0.476049849
GG Av Total Pressure 1	[Pa]	104260.7178	104260.5407	104260.2988	104260.7712	100	Yes	0.472326432	1.472291191
GG Av Density (Fluid) 1	[kg/m^3]	1.203501455	1.203501215	1.203497311	1.203502806	100	Yes	5.49472E-06	8.56316E-06
GG Av Velocity 1	[m/s]	69.19084613	69.18634448	69.18062945	69.19251299	100	Yes	0.011883534	0.028759534
GG Force 1	[N]	1376.036875	1369.888208	1361.288781	1376.036875	100	Yes	14.74809338	120.295282
Equation Goal 1	No unit	0.147425006	0.146785342	0.145878078	0.147425006	100	Yes	0.001546928	0.012500264

7.Glossary and list of Abbreviations

Table 3: Glossary and list of Abbreviations

S/N	Abbreviations	Explanation
01	CFD	Computational fluid dynamics
02	FEA	Finite Element Analysis

8.Discussion:

8.1 CFD Analysis:

Initial design:

- In the initial design of the bench, the force is 1640.249723 N. Therefore, the chances of failure are comparatively high. The main reason for this much force is that the air flow is directly act in the vertical surface of the bench and there is only a small gap. Hence the air flow should need to deviate, hence the force will be created.

Final Design:

- In the final design the force reduced to 1376.036875 N by adding number of holes in the vertical surface of the bench. So, the air will flow through the holes then the force will consequently reduce.
- From the goal plots which is shown in table 1 and 2, the pressure remains constant in both the initial and final design. The main reason for the constant pressure is that the CFD analysis the bench is fixed or stationary. There is not any other moving part, Hence the pressure remains constant.

8.2 FEA Analysis:

Initial Design:

- In the sample bench model, the assembled part which is run through the FEA analysis and then the results are note. So, here the maximum Von Mises stress for this is 7.588e+01 Mpa. However, the yield stress of the Delrin the 6.300e+01. If the von mises stress exceeds the yield stress the failure occurs. Here, when the wind blows at 70 m/s the sample bench model of 7.588e+01 von Mises stress, the material fails in the FEA analysis. Therefore, if the material fails at a given conditions then the sample bench model should need to be modified with more efficient than the last model.
- The main reason for the failure is that the area of cross section is the one among the factors for causing failure. If the there is less cross the chances of failure are relatively high.

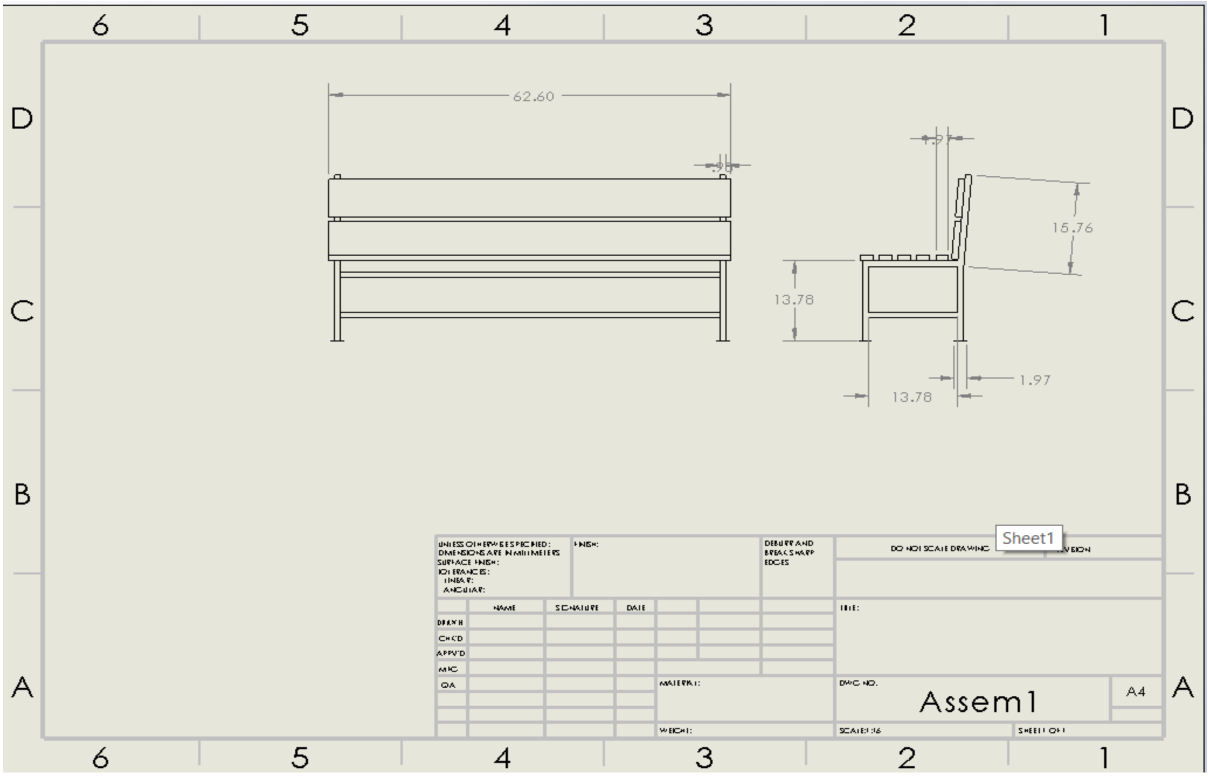
Final design:

- Here main areas of failure are near the edges of the bench which is shown in the figure 17. So, to avoid this failure one of the common methods is the adding chamfer in the edges of the bench frame, which is shown in the figure 16. After adding the chamfer on either side of the bench and runs through the FEA analysis. The main function of the chamfer will reduce the stress concentration which reduces the damage in the edges of the assembled material. Then the Von Mises stress is 3.149e+01 M pa and the yield strength of the Delrin material is 6.300e+01. Hence the von mises is less than yield stress. There for, the material will be in the safe condition when the air blows at a speed of 70 km/hr towards the bench.

9.Reference:

https://grabcad.com/library/bench-189

2D sketch of initial and final design



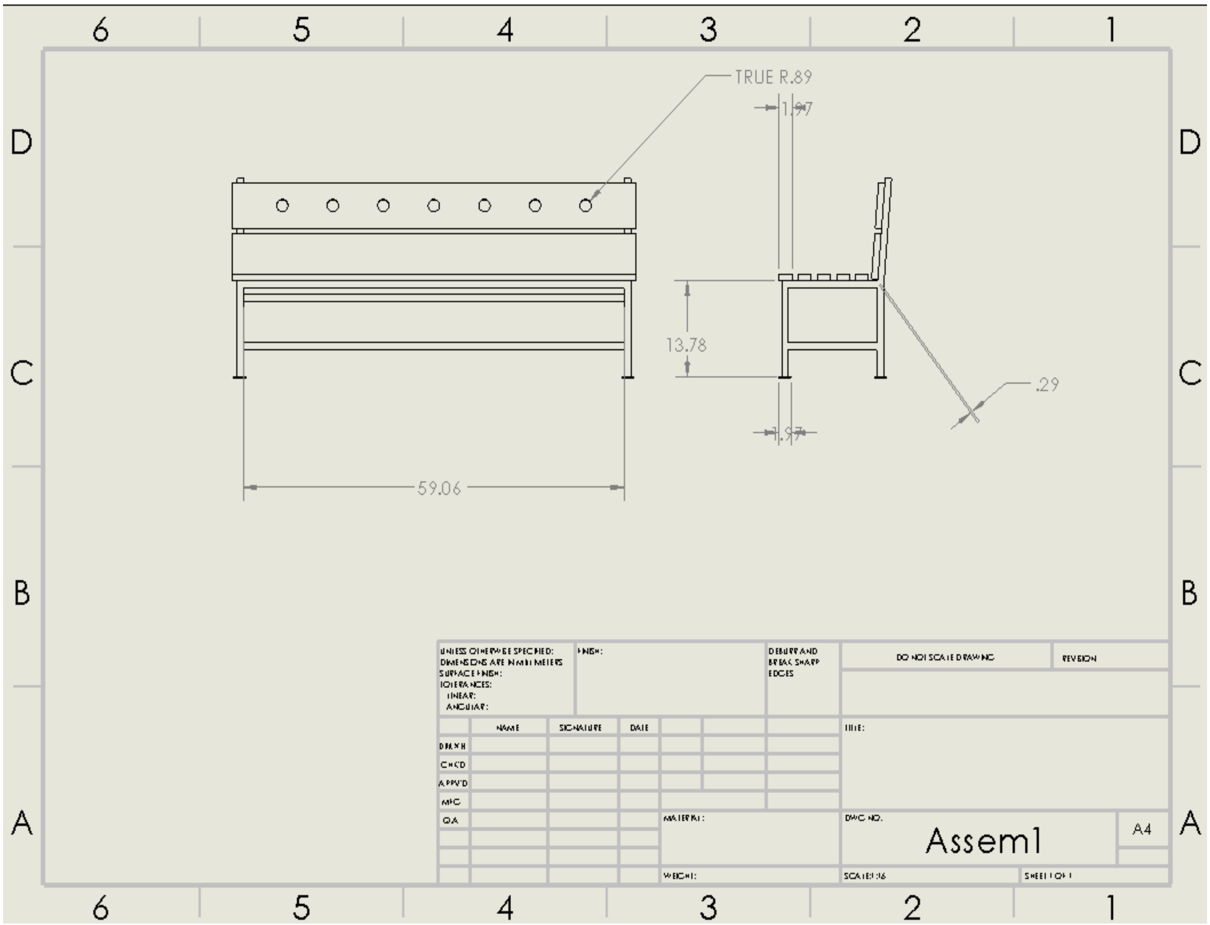
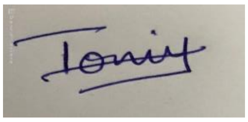


Figure 33: 2D model of sample Bench and Modified Bench

Student Declaration

I have not copied any part of this report from any other person's work, except as correctly referenced. No other person has written any part of this report for me.

1. **Student Name:** Tony Pauly



Student declaration of the above ___________________________________ signed.