# 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

1.	Aim:	4
2.	Introduction:	4
١.	CFD analysis (Computational fluid dynamics)	4
II.	FEA analysis (Finite Element analysis)	4
3.	Method	5
a.	Computational fluid dynamics (CFD) Method	5
b	FEA analysis (Finite Element analysis) Method1	1
4.	Design specification:	5
a.	Sample model1	5
b	Modified model1	5
5.	Preliminary Design Drawings:1	5
6.	Results1	6
6	1 FEA Analysis1	6
	6.1.1 Sample model without modification1	6
	6.1.2 Modified model1	9
6	2 CFD Modelling (Computational fluid dynamics)2	1
	6.2.1 Sample model CFD Result	1
	6.2.2 Sample model modified CFD Result	4
	6.2.3Goals plot of sample bench2	7
	6.2.4 Goals plot of modified sample Bench2	8
7.Gl	ossary and list of Abbreviations2	8
8.Di	scussion:2	9
8.	1 CFD Analysis:	9
8.	2 FEA Analysis:	9
9.Re	eference:	0

# List of Figures

Figure 1: Project name	5
Figure 2: unit system	6
Figure 3: Analysis type	6
Figure 4	7
Figure 5: default fluid	7
Figure 6: Assigning the values for air at a velocity in velocity parameters	8
Figure 7: Computational domain	8
Figure 8: Adding equation goals	9
Figure 9: Running the Model	10
Figure 10: Export results to simulation	10
Figure 11: connections	11
Figure 12: Contact set	12
Figure 13: adding fixture geometry	12
Figure 14: Applying the force	13
Figure 15 : Mesh Details	14
Figure 16:Modified model	15
Figure 17: Von mises stress	16
Figure 18:Resultant displacement	17
Figure 19:Equivalent strain	18
Figure 20:Von Mises Stress	19
Figure 21:Resultant displacement	20
Figure 22:Equivalent strain	20
Figure 23: cut plot (Velocity)	21
Figure 24: cut plot 2(velocity)	21
Figure 25: surface plot 1(pressure)	22
Figure 26: surface plot 2(velocity)	22
Figure 27: Flow Trajectory	23
Figure 28: pressure (Modified Model)	24
Figure 29: Pressure (Modified model)	24
Figure 30: surface plot (pressure)	25
Figure 31: surface plot 2(velocity)	25
Figure 32: Flow trajectory (modified model)	26
Figure 33: 2D model of sample Bench and Modified Bench	31

#### List of Tables

Table 1: Goals plot of sample Bench	27
Table 2: Goals plot of modified sample Bench	28
Table 3: Glossary and list of Abbreviations	28

### 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 figure 1

Computational Domain			
Fluid Subdomains     Fluid Subdomains     Fains     Heat Sources     Porous Media     Gala	Configuration to add th Configuration: Configuration name:	e project Use Current Default	×
Goals     Goals     Coal Initial Meshes     Coals     Coals			>

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.

	Unit system:				
K m/s	System         CGS (cm-g-s)           FPS (th-b-s)         IPS (th-b-s)           IPS (th-b-s)         INMM (tmm-g-s)           SI (tm-kg-s)         USA	Path Pre-Defined Pre-Defined Pre-Defined Pre-Defined Pre-Defined	Cammer CGS (cm FPS (ftH) IPS (imH) NMM (m SI (im-kg USA	nt ⊷g•s) ⊷s) −s) m-g•s) s)	
e mi	Create new Parameter	Name:	SI (m-kg-s) (modified) Decimals in results display	1 SI unit equals to	^
gai	Main     Pressure & stress     Velocity     Mass	Pa m/s kg	.12 .123 .123	1 1 1	
kg	Cime Length Temperature Physical time Percentage	m K s %	.123 .12 .123 .12	1 1 1 1	~

#### Figure 2: unit system

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

Wizard - Analysis Type	Analysis type O Internel © External Physical Features Heat conduction in Radiation Time-dependent Gravity Pention	Consider closed cavities   Exclude cavities without flow conditions  Exclude internal space  Value  Colids	? ×
	Time-dependent		
	Rotation	- H	
	Free surface		
		Depen	dency 🛞

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.

	Fluids	Path	New »
	🗄 Gases		
	Liquids		
	🗄 Non-Newtonian Liquids		
	Compressible Liquids		
	🗄 Real Gases		
	🗄 Steam		
			Add
	Project Fluids	Default Fluid	Remove
14 10			
A Second			
	Flow Characteristic	Value	
States and a state of the state	Flow type	Laminar and Turbulent	
A state of the sta			
A STATE OF THE STA			
			())
and the second se			

Wizard - Default Fluid				?	$\times$
	Fluids Gases	Path	<u>^</u> N	ew	) () 
	Acetone	Pre-Defined			
	Ammonia	Pre-Defined			
	Butane	Pre-Defined			
	Chlorine	Pre-Defined Pre-Defined			
	Ethanol	Pre-Defined	~	٨dd	
	Project Fluids	Default Fluid	Re	move	
-1978					
	Elow Charactorictic	Value	_		
	Flow type	Laminar and Turbulent	~		
					»
	< Back	Next > Cancel	Н	əlp	

#### Figure 4

Figure 5: default fluid

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

60       10         30       10         11       10         12       10         11       10         12       10		Parameter	Value	
S0       10         30       10         11       10         12       10         12       10         12       10         12       10         12       10         12       10         12       10         12       10         12       10         12       10         12       10 <td>60 -</td> <td>Parameter Definition</td> <td>User Defined</td> <td><math>\sim</math></td>	60 -	Parameter Definition	User Defined	$\sim$
0       1       2       3       4       5       6       7       9       0	50-10	Thermodynamic Parameters		Louis I
30       0         20       10         11       10         11       10         11       10         11       10         11       10         11       10         12       10         12       10		Parameters	Pressure, temperature	$\sim$
10       10 <td< td=""><td></td><td>Pressure</td><td>101325 Pa</td><td>Laurant</td></td<>		Pressure	101325 Pa	Laurant
0       1       2       3       4       5       6       7       8       0       Time.s         0       1       2       3       4       5       6       7       8       0       Time.s         0       1       2       3       4       5       6       7       8       0       Time.s	0 000000000	Temperature	293.2 K	
Parameter Velocity     Defined by 3D Vector     Defined by 3D Vector     Velocity in X direction 0 m/s     Velocity in X direction -70 m/s     Velocity in Z direction -70 m/s     Turbulence Parameters     Coordinate System Dependency Dependency Dependency	3	Velocity Parameters		
0       1       20         0       1       20         0       1       20         0       1       3       4       5       6       7       9       0       Turbulence Parameters       Dependency         0       1       2       3       4       5       6       7       9       Dependency	10-	Parameter	Velocity	$\sim$
0       1       2       3       4       5       6       7       8       0       Time.s	F 0	Defined by	3D Vector	~
0       1       2       3       4       5       6       7       8       0       Turbulence Parameters         0       1       2       3       4       5       6       7       8       0       Turbulence Parameters       Dependency		Velocity in X direction	0 m/s	
Velocity in 2 direction       -70 m/s         Implementation       -70 m/s		Velocity in Y direction	0 m/s	
Image: State of the state		Velocity in Z direction	-70 m/s	
0 I 2 3 4 5 6 7 8 9 10 Time,s Coordinate System Dependency				
	No.			

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

- 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
- 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

 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.





 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

 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.



ons	Adaptive	Flow/T	hermal Effects	Notification	Remark	
Therr	nal options					
	nput tempe	erature				
0			4			
0	remperatur	es from	thermai study			
	Thermal st	udy:			Time step:	1
	For ea	ich nonli	inear time step	, use temper	ature from	
$\cap$	corre	espondir	ng time of tran	isient therma	analysis.	
0	Temperatur	e from S	SOLIDWORKS	Flow Simulati	on	_
SO	LIDWORKS	model	name :			
Co	nfiguration	name	:			
Ter	mperature f	from tim	ie step :			
Ter Refe	mperature f	from tim perature	ie step : at zero strain:	29	Kelvin (k	0 ×
Ter Refe	mperature f rence temp	from tim erature	ne step : at zero strain:	298	Kelvin (K	<) ×
Ter Refe Fluid	mperature f rence temp pressure o	from tim erature ption	ie step : at zero strain:	298	Kelvin (K	<) ×
Ter Refe Fluid	mperature f rence temp pressure o Include flui	from tim perature ption d pressu	ne step : at zero strain: ure effects fror	298 m SOLIDWOR	KS Flow Simulatio	<) ~
Ter Refe Fluid	mperature f rence temp pressure o Include flui	from tim perature ption d pressu yp\Desk	at zero strain: ure effects fror top\Bench\1\1	298 m SOLIDWOR	KS Flow Simulatio	() ~
Ter Refe Fluid	mperature 1 rence temp pressure o Include flui :\Users\ton	from tim perature ption d pressu yp\Desk	at zero strain: ure effects fror top\Bench\1\1	298 m SOLIDWOR .fld Assem1 SI D	KS Flow Simulatio	() ~ n 
Ter Refe Fluid C: SO	mperature 1 rence temp pressure o Include flui :\Users\ton ULDWORKS	from tim perature ption d pressu yp\Desk model	at zero strain: ure effects fror top\Bench\1\1 name :	298 m SOLIDWOR .fld Assem1.SLD Default	KS Flow Simulatio	() ~ n 
Ter Refe Fluid C: SO Co	mperature f rence temp pressure o Include flui CUSers\ton ULDWORKS nfiguration	from tim perature ption d pressu yp\Desk model name	at zero strain: ure effects fror top\Bench\1\1 name :	298 m SOLIDWOR .fld Assem1.SLD Default 128	KS Flow Simulatio	n 
Ter Refe Fluid C: SO Co Flo	mperature f rence temp pressure o Include flui (Users\ton DUDWORKS nfiguration w iteration	from tim perature ption d pressu yp\Desk model name no.	at zero strain: ure effects fror top\Bench\1\1 name : : : : : : : :	298 m SOLIDWOR fild Assem1.SLD Default 128 n fild file	KS Flow Simulatio	n 
Ter Refe Fluid C: SO Co Flo	mperature f rence temp pressure o Include flui CUSers\ton DLIDWORKS Infiguration w iteration USe refere	from tim perature ption d pressu yp\Desk model i name no. nce pres	at zero strain: ure effects fror top\Bench\1\1 name : : : ssure (offset) in uressure (offset)	294 m SOLIDWOR .fld Assem1.SLD Default 128 n .fld file P	KS Flow Simulatio	0
Ter Refe Fluid C: SO Co Flo	mperature 1 rence temp pressure o Include flui (\Users\ton ILIDWORKS nfiguration w iteration Use refere Define refere	from tim perature d pressu yp\Desk model i name no. nce pres	at zero strain: ure effects fror top\Bench\1\1 name : : : ssure (offset) in ressure (offset)	294 m SOLIDWOR .fld Assem1.SLD Default 128 n .fld file t)	KS Flow Simulatio	0)
Ter Refe Fluid C: SO Co Flo	mperature ( rence temp pressure o Include flui (\Users\ton HIDWORKS Infiguration w iteration I Use refere I Define refe Run as leg	from tim perature ption d pressu yp\Desk model name no. nce pres erence p acy stud	at zero strain: ure effects fror top\Bench\1\1 name : : ssure (offset) in ressure (offset) iy (exclude she	298 n SOLIDWOR .fld Assem1.SLD Default 128 n .fld file t) war stress)	KS Flow Simulatio	0 ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~ ~

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	-= 🔀	Mesh Details	-1=
Study name	Static 1 (-Default-)	Study name	Static 1 (-Default-)
Mesh type	Solid Mesh	Mesh type	Solid Mesh
Mesher Used	Curvature-based mesh	Mesher Used	Curvature-based mesh
Jacobian points	4 points	Jacobian points	4 points
Mesh Control	Defined	Mesh Control	Defined
Max Element Size	2.74923 in	Max Element Size	2.83493 in
Min Element Size	0.549847 in	Min Element Size	0.566987 in
Mesh quality	High	Mesh quality	High
Total nodes	8558	Total nodes	56788
Total elements	3407	Total elements	28171
Maximum Aspect Ratio	16.027	Maximum Aspect Ratio	34.222
Percentage of elements with Aspect Ratio < 3	49.9	Percentage of elements with Aspect Ratio < 3	93.5
Percentage of elements with Aspect Ratio > 10	0.998	Percentage of elements with Aspect Ratio > 10	0.0923
% of distorted elements (Jacobian)	0	% of distorted elements (Jacobian)	0
Remesh failed parts with incompatible mesh	Off	Remesh failed parts with incompatible mesh	Off
Time to complete mesh(hh:mm:ss)	00:00:02	Time to complete mesh(hh:mm:ss)	00:00:05
Computer name		Computer name	

a. Sample model mesh details

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)





#### 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

Goal						Progr	Use In		
Nam			Average	Minimu	Maximu	ess	Converg		
е	Unit	Value	d Value	m Value	m Value	[%]	ence	Delta	Criteria
GG									
Av									
Static									
Press		101314.	101314.	101314.	101314.			0.50248	0.52255
ure 1	[Pa]	3711	3299	1801	6826	100	Yes	5384	2422
GG									
Av									
Total									
Press		104257.	104257.	104257.	104257.			0.27515	1.60924
ure 1	[Pa]	8239	7257	6108	8859	100	Yes	7154	8161
GG									
Av									
Veloc		69.1369	69.1342	69.1300	69.1375			0.00752	0.03113
ity 1	[m/s]	6985	2391	3666	6459	100	Yes	7932	5774
GG									
Force		1640.05	1633.70	1624.75	1640.24			15.4943	145.359
1	[N]	1764	1736	5348	9723	100	Yes	7527	9459
GG									
Av									
Densi									
ty									
(Fluid	[kg/m	1.20346	1.20346	1.20346	1.20346			4.50641	9.34554
) 1	^3]	6962	6312	4829	9335	100	Yes	E-06	E-06
Equa									
tion									
Goal	No	0.01000	0.00996	0.00991	0.01000			9.27048	0.00085
1	unit	3635	5699	2302	5006	100	Yes	E-05	7626

#### Table 1: Goals plot of sample Bench

# 6.2.4 Goals plot of modified sample Bench

Goal						Progr	Use in		
Nam			Average	Minimu	Maximu	ess	Converg		
е	Unit	Value	d Value	m Value	m Value	[%]	ence	Delta	Criteria
GG									
Av									
Static									
Press		101317.	101317.	101317.	101317.			0.44026	0.47604
ure 1	[Pa]	3292	369	0009	5378	100	Yes	6326	9849
GG									
Av									
Total									
Press		104260.	104260.	104260.	104260.			0.47232	1.47229
ure 1	[Pa]	7178	5407	2988	7712	100	Yes	6432	1191
GG									
Av									
Densi									
ty									
(Fluid	[kg/m	1.20350	1.20350	1.20349	1.20350			5.49472	8.56316
)1	^3]	1455	1215	7311	2806	100	Yes	E-06	E-06
GG									
Av									
Veloc		69.1908	69.1863	69.1806	69.1925			0.01188	0.02875
ity 1	[m/s]	4613	4448	2945	1299	100	Yes	3534	9534
GG									
Force		1376.03	1369.88	1361.28	1376.03			14.7480	120.295
1	[N]	6875	8208	8781	6875	100	Yes	9338	282
Equa									
tion									
Goal	No	0.14742	0.14678	0.14587	0.14742			0.00154	0.01250
1	unit	5006	5342	8078	5006	100	Yes	6928	0264

Table 2: Goals plot of modified sample Bench

# 7. Glossary and list of Abbreviations

Table 3: Glossary and list of Abbreviations

S/N	Abbreviations	Exp
01	CFD	Com
02	FEA	Finit

Explanation Computational fluid dynamics 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: <a href="https://grabcad.com/library/bench-189">https://grabcad.com/library/bench-189</a>

# 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.