BINANI TECHNOLOGIES

MECHANICAL DESIGN INTERNSHIP REPORT

By Arunkumar V

Date of Presentation:
16/09/2022

TABLE OF CONTENTS:

- INTRODUCTION
- TASK 1 (Design and Analysis of Subframe)
  - Analysis
  - Iteration
  - Conclusion
- TASK 2 (Design and Analysis of Control Arm - FBR)
  - Analysis
  - Iteration
  - Conclusion
- TASK 3 (Design of Chassis Rail)
  - Design in Solidworks
  - Conclusion
- ASSEMBLY OF COMPONENTS
  - Design in Solidworks
  - Mates
  - Conclusion

Introduction

Solidworks is used to design, assemble, and create the model. Solidworks is a 3D modelling programme that allows you to create complex 3D forms and structures. ANSYS is a finite-element modelling tool that can be used to solve a wide range of mechanical problems numerically.It's often used to evaluate a structure's or component's strength, toughness, elasticity, temperature distribution, electromagnetism, fluid flow, and other properties.We will examine the stress concentration location with the help of ansys and take the necessary measures to reduce it.In order to make the component safer to use in any condition, the factor of safety should be greater than or equal to 2.

TASK 1

(DESIGN AND ANALYSIS OF REAR SUBFRAME)

Input Data :

Total GVW of vehicle - 3.5 T.Weight distribution for the rear part - 52 %. All the mounting points were given and subframe needs to be developed from the hard points Consider UDL due to mass on the side top beams.

Objective:

A design of rear subframe needs to be done.The subframe needs to be optimized such that it can handle heavy load and it should posses a Factor of Safety more than 2.The weight of the subframe should be strictly less than 30 kg

Units:

Stress - MPa, Total Deformation - mm, Force - N, Strain - mm/mm (No units)

Concept :- Application of UDL on both the side of rectangular tubes and analysing the result obtained.

REAR SUBFRAME :

- The purpose of using a subframe in an automobile is to distribute high local loads over a wider area of body structure (most relevant in thin-walled monocoque body designs) and to isolate vibration and harshness from the rest of the body.The subframes are generally bolted or welded to the vehicle chassis rails
- This task examines the design, analysis, optimization, and validation of the rear subframe of an automobile suitable for usage in a vehicle. In order to significantly reduce the bulk of the sub-frame while maintaining performance and structural integrity, a new design was developed without altering the mounting locations from the base design. Improving the vehicle's overall efficiency is the primary reason behind the decrease in mass. One of the key elements that affects how efficiently a vehicle operates is its mass.
- Hydroformed tubes are generally used while manufacturing

ANALYSIS

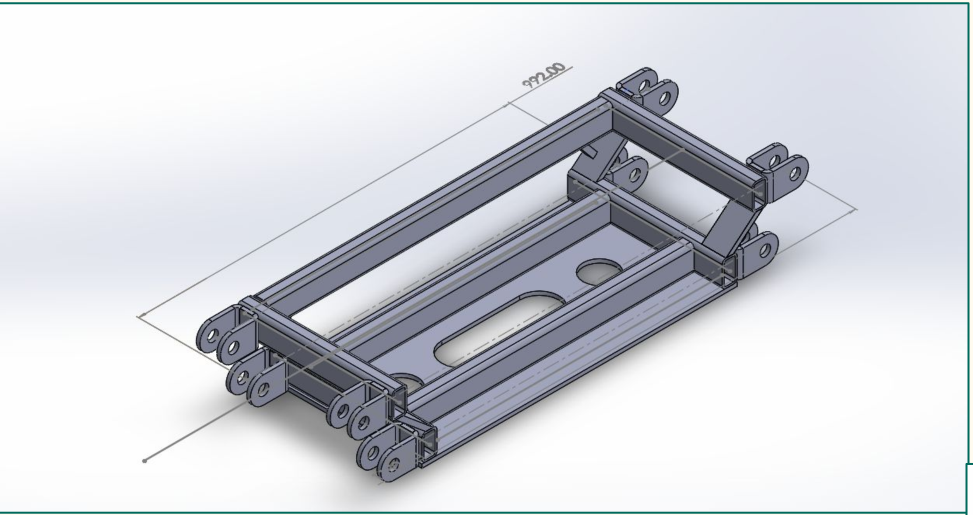
PROCEDURE FOR ANALYSIS

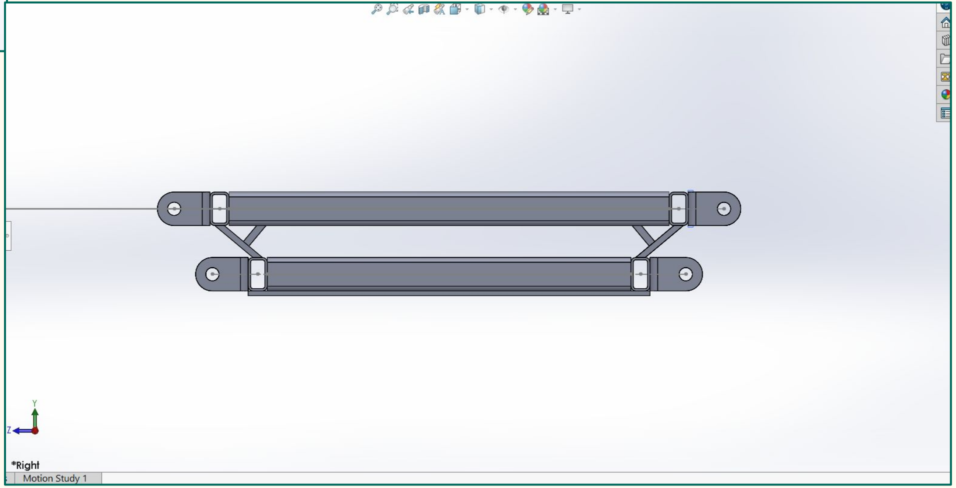
- Solidworks - Design
- Ansys Workbench - Analysis

Design in SOLIDWORKS

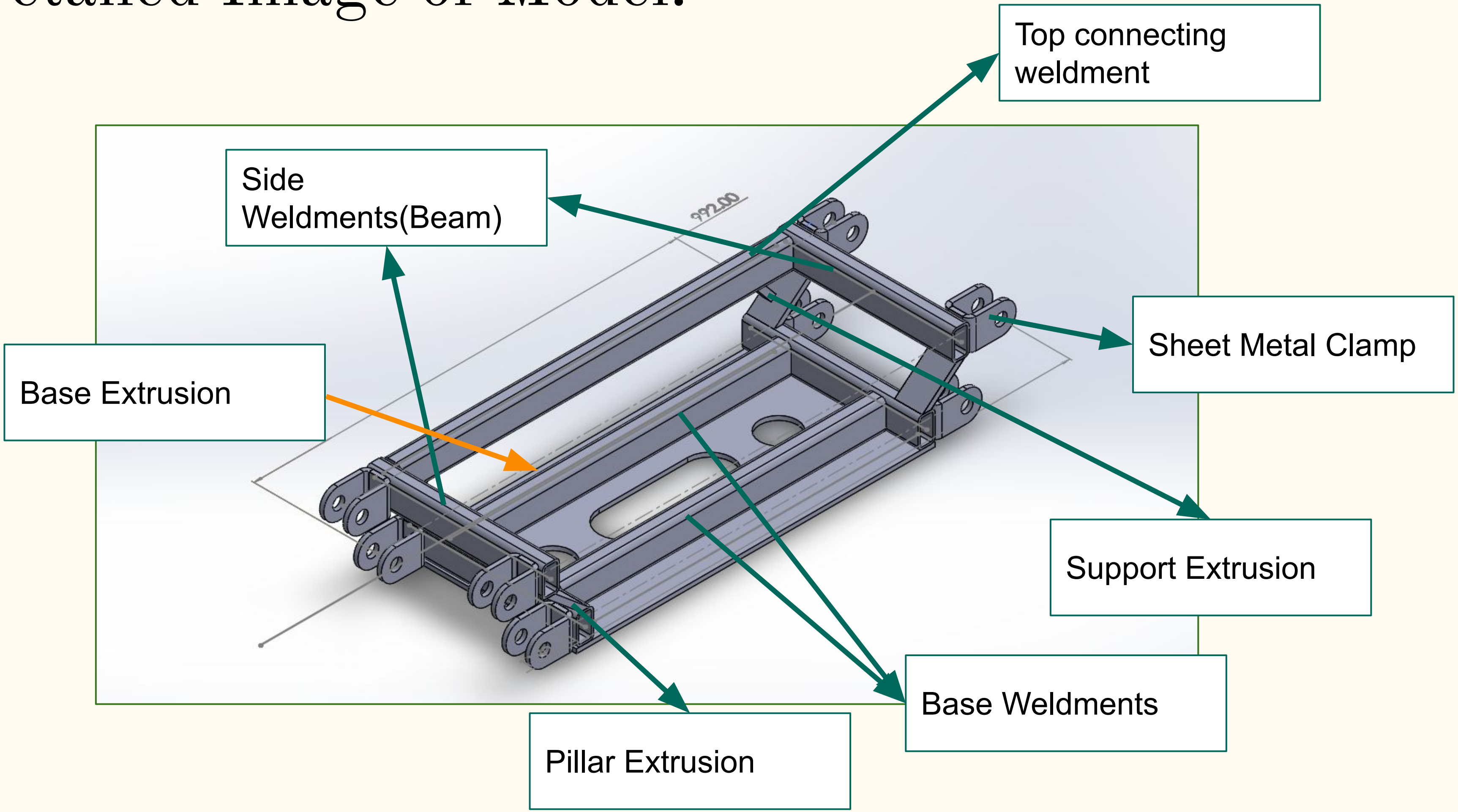
- Solidworks was used to design the body. The body is made of weldments, sheet metal, and extrusions. The positions of hard points are fixed.
- At the top, we'll construct an upper beam that connects the other perpendicular beams.For stability and to lessen stress concentration, I inserted a few supporting extrusions.
- Sheet metal is used to make the clamps.Extrusions are used to join the weldments in order to increase the load bearing capability.The beam's initial thickness was used as 5 mm.Initial flange thickness is around 10 mm.
- Sheet metal is strong and durable.Parts made from sheet metal can withstand great pressure.So the clamps at the end for suspension are made with sheet metal.
- Weldments are used at the top end because stress from the chassis can be transferred without much deformation. Weldments in general have more strength than the parent assembled bodies.So it is very suitable. It also has less weight in comparison to extrusion. I used weldments of 70mm x 40mm x 5mm dimensions.
- After saving the file in step format, we need to import it to Ansys for analysis.

Initial Model





Detailed Image of Model:



ANSYS PART :

- Material applied for our structure will be ST52
- It has a yield strength of 355 MPa.
- Factor of Safety (ductile material) =Yield strength/Allowable strength
- In order to ensure that material does not suffer unexpected stress or strain in any driving circumstance, it is always preferable to have a high factor of safety. This also assures that the car's subframe will not be readily damaged in the event of an accident.
- After assigning the material properties,we will import the step file of our model into ansys
- Here we can check or redefine dimensions if we need.
- We will do all of these in Ansys SpaceClaim

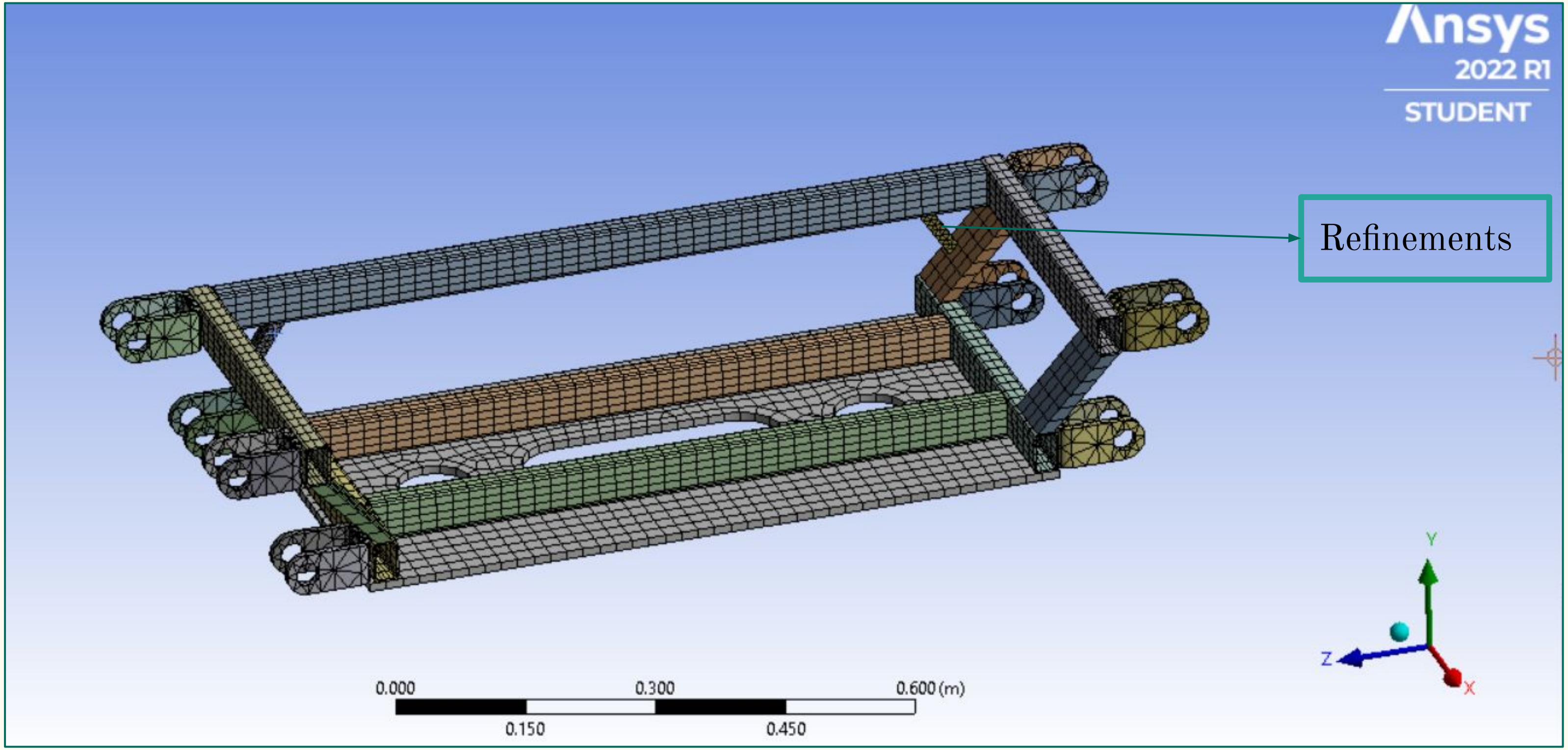
Material Properties(only ST52 -3):

	St52-3	SAE 1040
Young's Modulus (GPa)	210	200
Poisson's Ratio	0.3	0.29
Tensile Yield Strength (MPa)	355	415
Tensile Ultimate Strength (MPa)	520	600

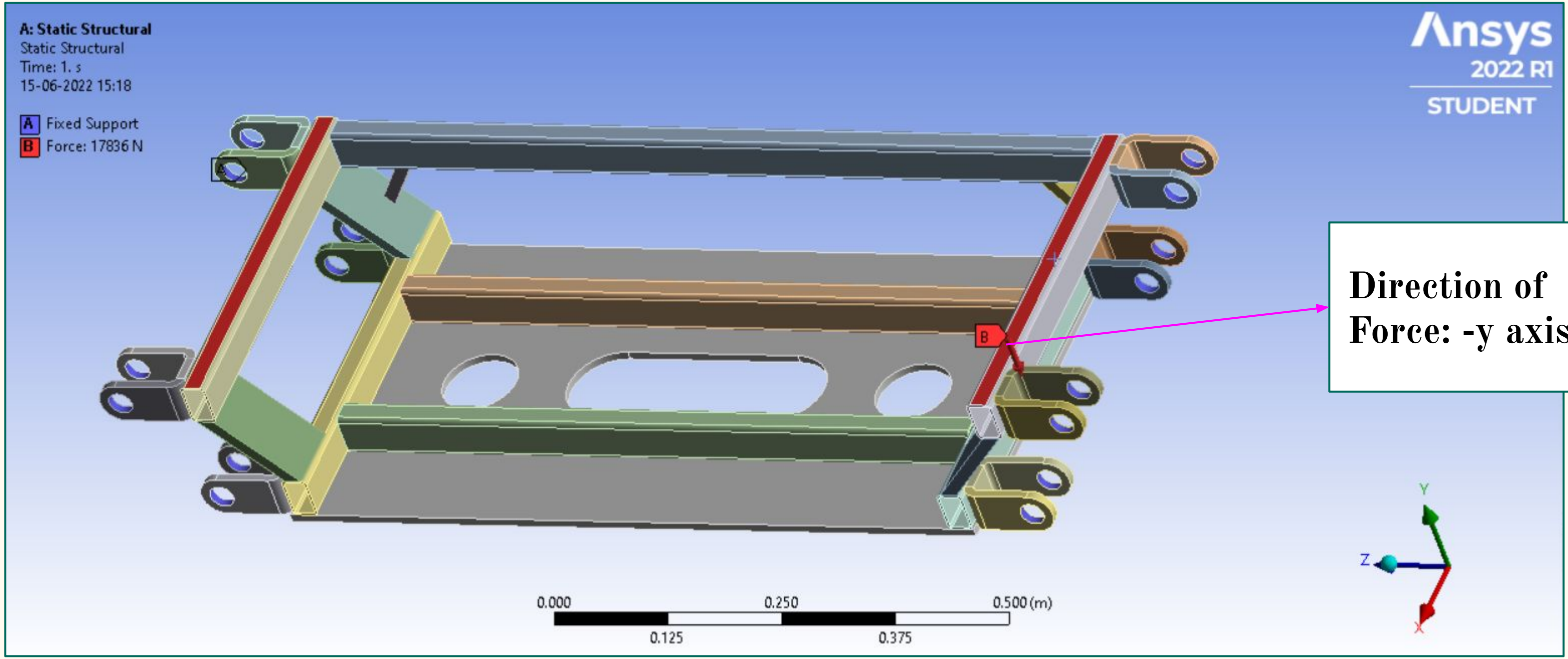
Density of the material : 7850 kg/m3

Ansys Mechanical

- As our next step, we will open Ansys mechanical. After that, we will check whether all the contacts are proper. Connections should be of the **bonded** type. As of now, we will weld to join the pieces.
- Generally, ansys will create a contact group on its own. If we want, we can create new contacts with a specific contact type.
- After that, we will give a mesh command to our structure. We'll construct a global mesh with elements that are 10 mm in size.
- We also created refinement for our small extrusions that we added for beam support. This refinement feature will ensure that those parts will be considered as different nodal shapes (i.e Tetragonal) rather than one dimensional node.
- We will select the span angle centre option to be medium instead of coarse.
- Now we will generate the mesh.



MESH CREATION - Element size 10 mm , Span Angle Centre - Medium

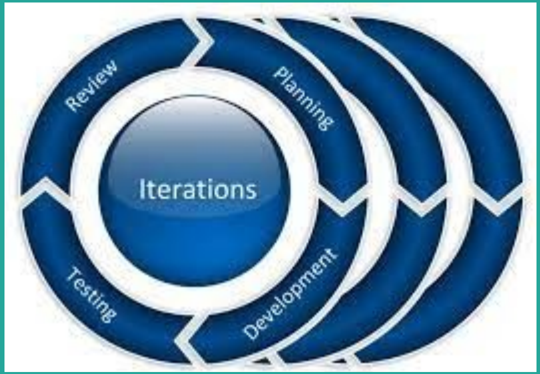


Direction of Force: -y axis

- After the completion of mesh generation we will apply fixed support for our model
- Fixed support is given at the face of each clamp .So a total of 16 faces will be selected.After that we will apply load of 17836 N as UDL along the top surface of the weldment.This weight will be delivered from chassis due to other components present in rear end.The constraints are fixed support and UDL on top surface

- After that we will run the solver.The Ansys solver will solve a large number of equations, as well as the boundary condition, and provide solutions for each individual element. As a result, we can calculate the maximum von Mises stress and total deformation generated in the material as a result of the applied load.
- The maximum stress that we get from ansys should be less than or equal to the allowable stress of the material.
- Along with that the excess material present in flanges are removed.So final thickness become 5 mm.

ITERATIONS

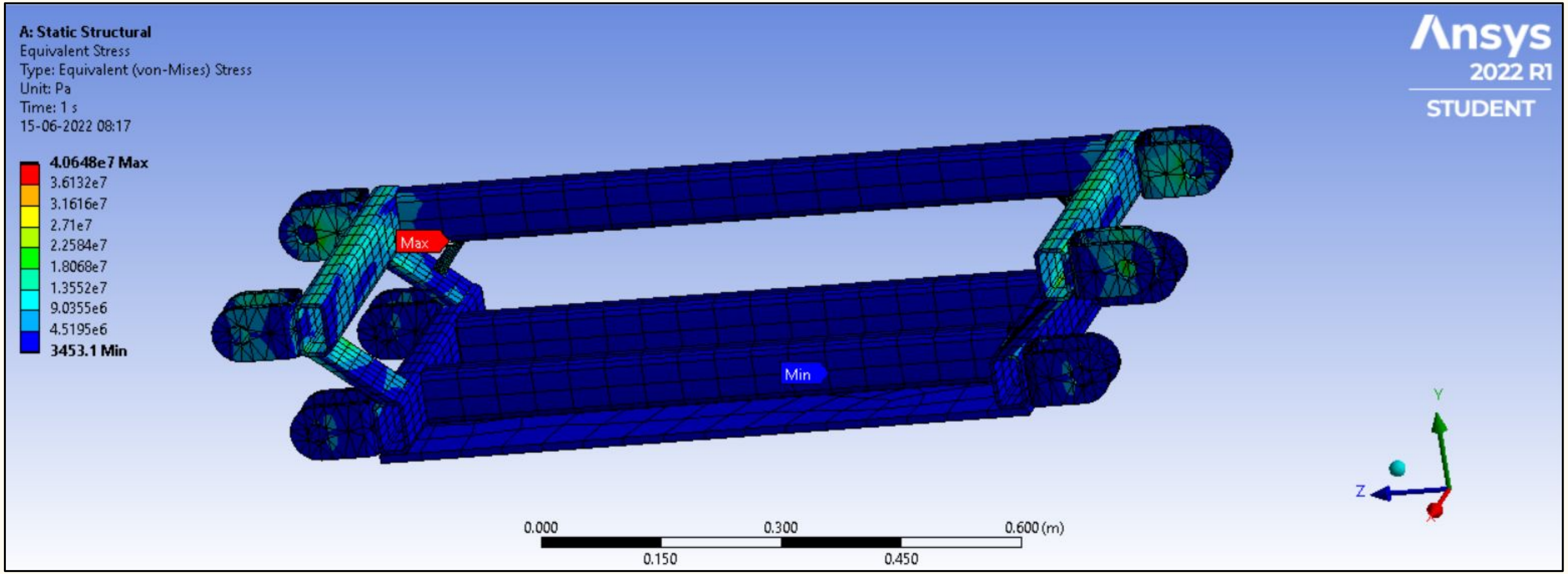


INITIAL CASE (BEFORE ITERATION)

- The initial value of maximum stress from my model is around 40.3 MPa.The maximum stress occurs at the inside base of rectangular tube
- Here we can see that we got FOS more than what we want.This indirectly signifies that more material is used for making the structure.Inorder to be cost effective we will reduce the dimensions and thickness of the weldments and increase the support extrusion length.
- Weight of the model is around 58 kg initially

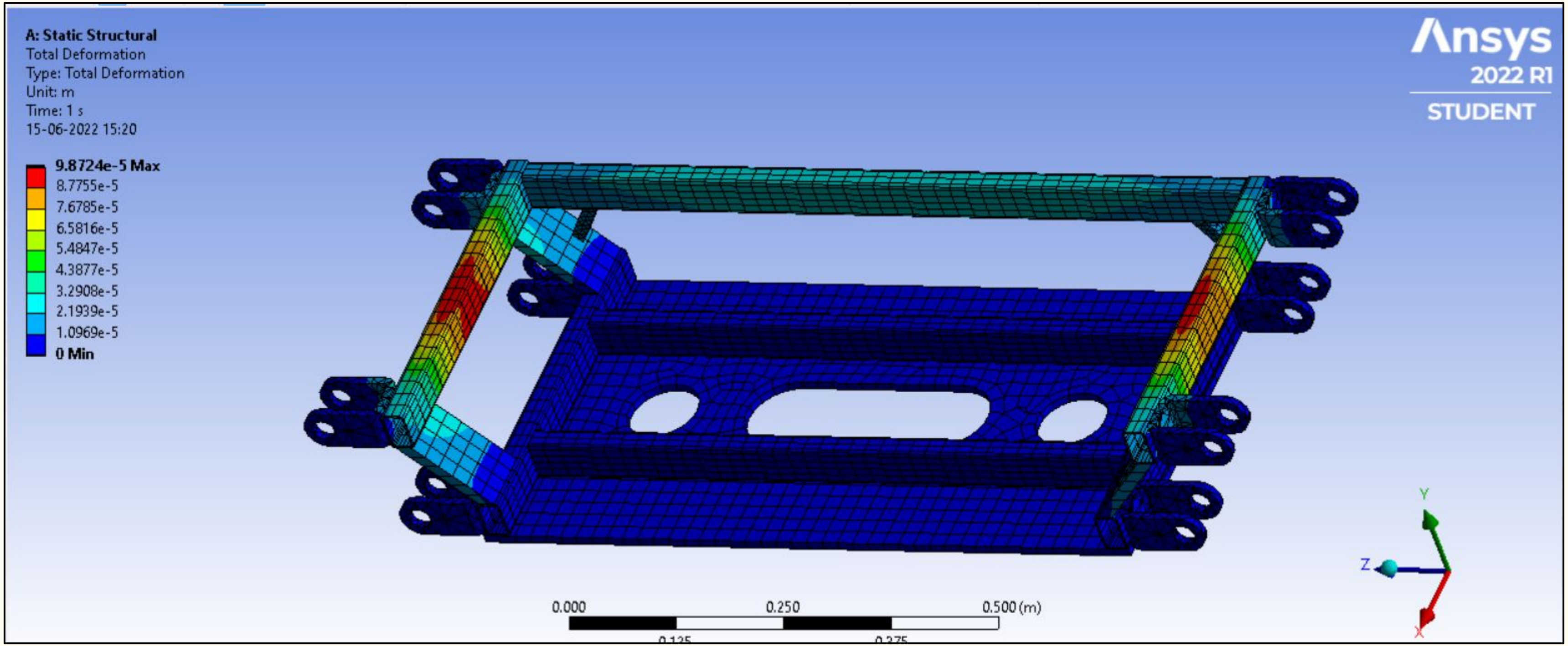
Initial CASE

Maximum Von Mises Stress:



Initial CASE

Maximum Total deformation:



ITERATION 1

ITERATION 1:

DRAWBACK:

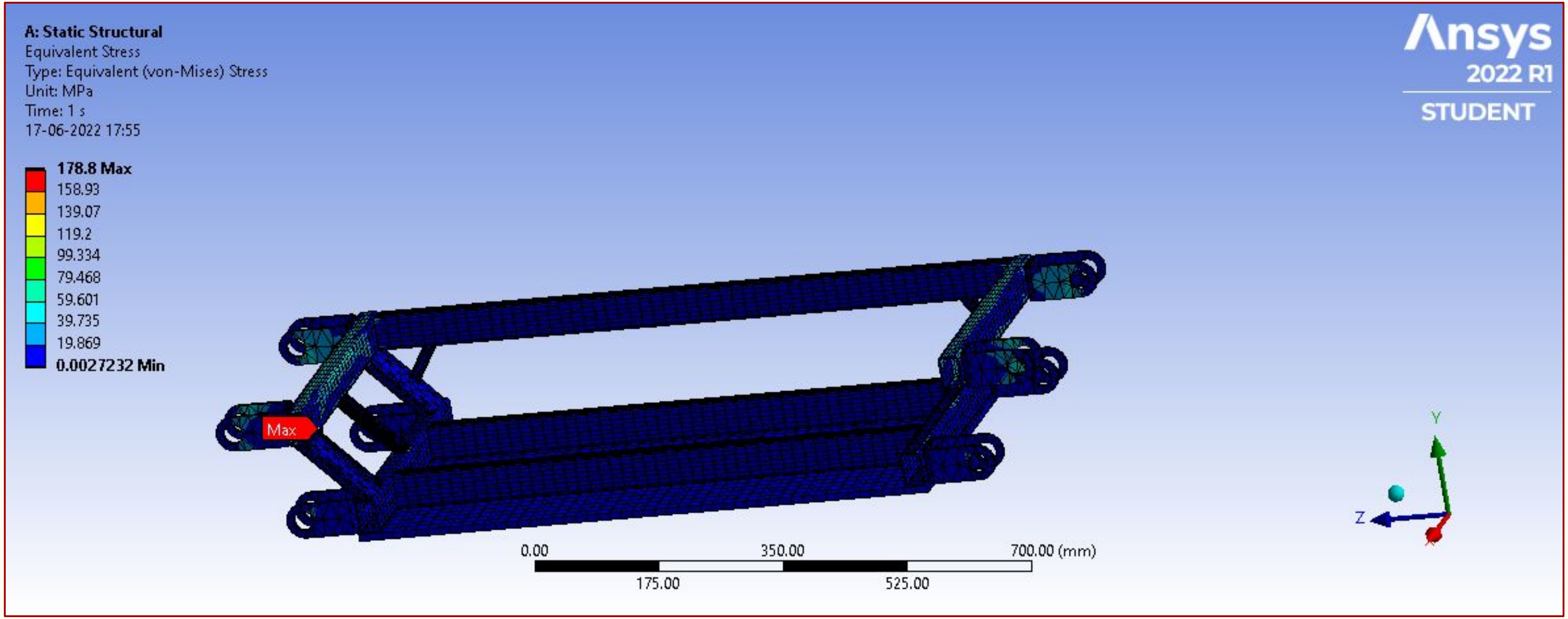
- The reason for this iteration is to provide a reduced FOS along with decrease in material wastage.Weight of this model is comparatively more
- Material consumed in this iteration is structural steel.

SOLUTION :

- Reduction in size of weldments.Final weldment dimension is 50mm x 30mm x 2.6mm. Decreased the thickness of flanges by half.
- As a result, the model obtained weighed around 47.4 kg.
- Size of support extrusion for top beam has increased
- In order to improve stability, an extrusion was added to the middle of the subframe side. My hypothesis is that the engine will be attached to the axle through the empty space.

ITERATION 1:

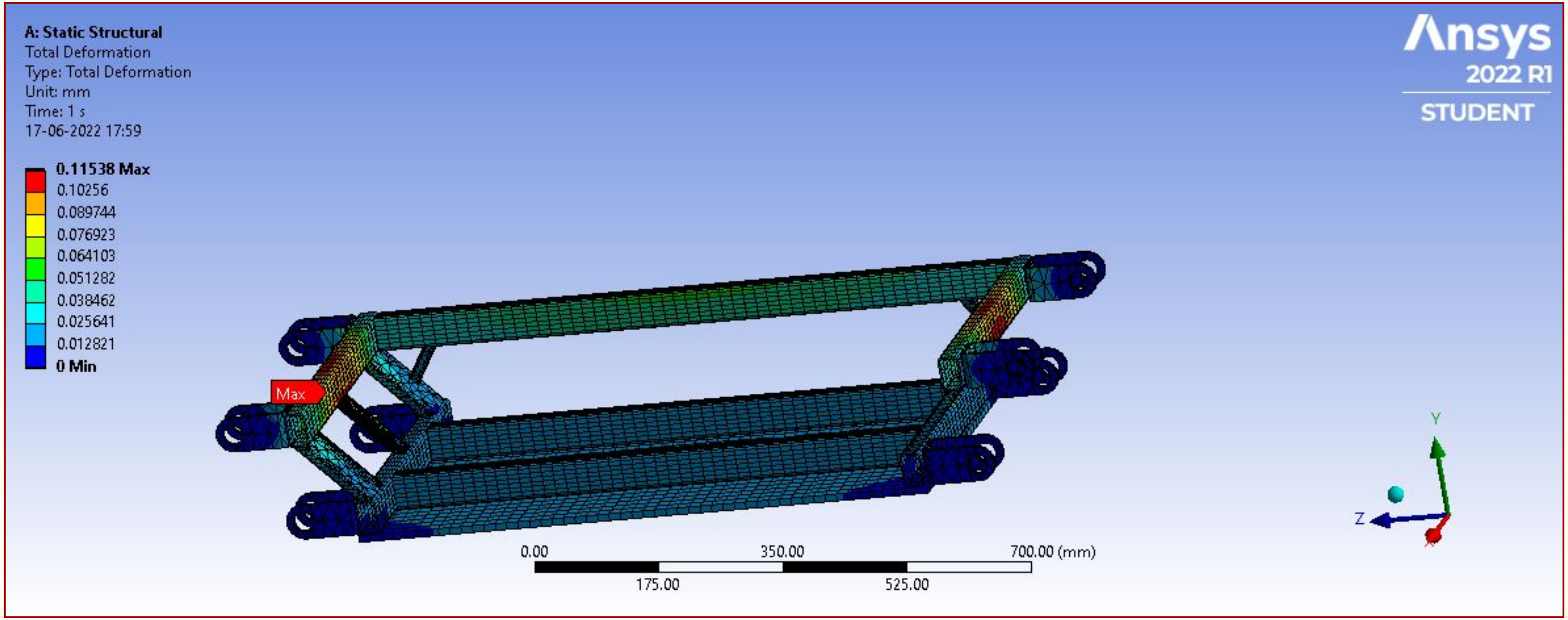
Maximum Von Mises Stress:



Maximum Stress Value: 178.8 MPa
Maximum Permissible value: 180 MPa

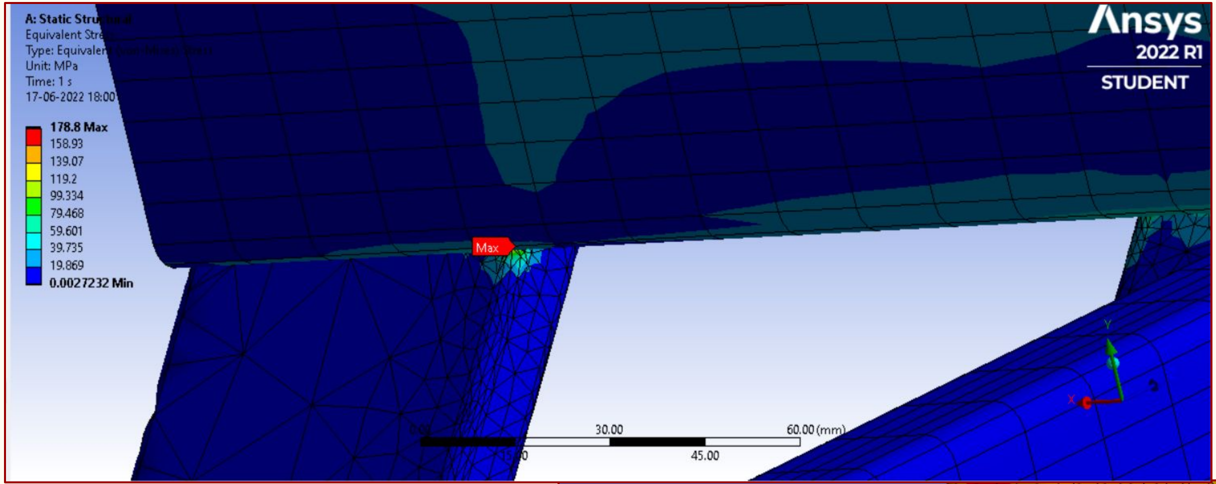
ITERATION 1:

Maximum Total deformation:

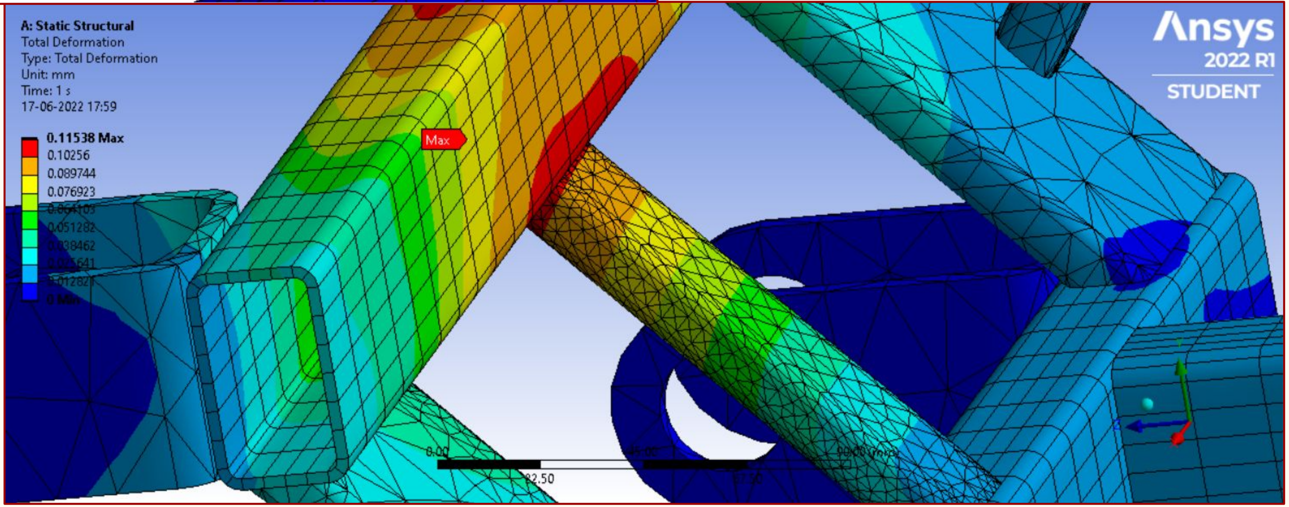


Maximum Total Deformation Value: 0.115 mm
Maximum Permissible value: 0.4 mm

Region of MAX STRESS AND DEFORMATION



Maximum at Rectangular tube outer base



Maximum at middle of the weldment

Result of Iteration 1:

Maximum Stress obtained : 178.8 MPa

Maximum Total Deformation obtained : 0.11 mm

Weight of Model : 47.4 Kg

Material Used : Structural Steel

Factor Of Safety : 1.9845

Weldments thickness got reduced to 2.6 mm

ITERATION 2

ITERATION 2

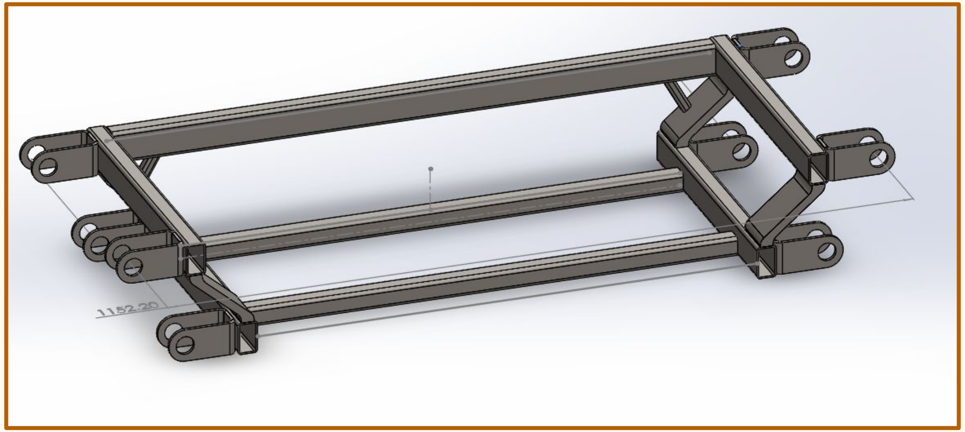
Drawbacks:

- Weight of the model should be less than 30 kg.
- Stress limit should be maintained in the process
- Connection between axle and engine should be provided.So space should be allocated
- ST52 material should be used in place of structural steel

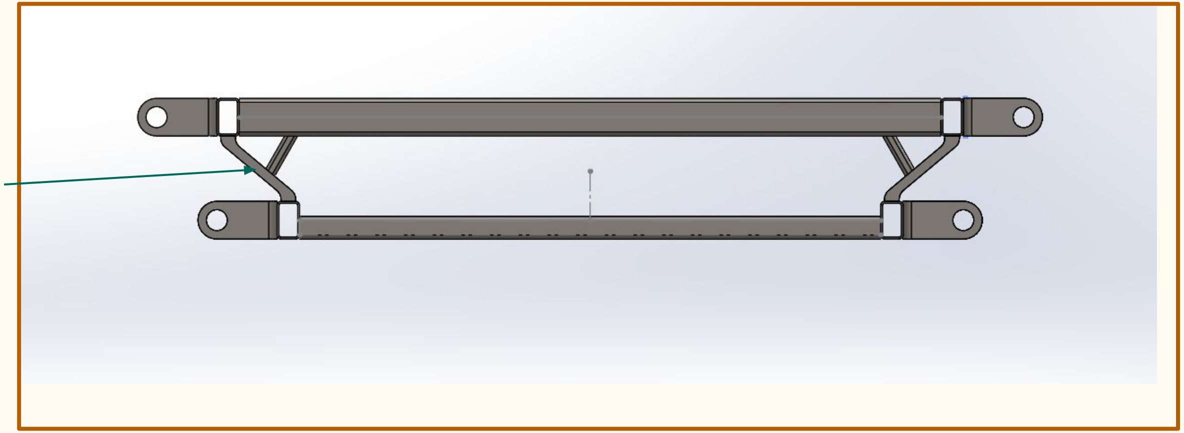
Solution :

- I removed the excess material present in pillars (Extrusions) by reducing its thickness
- I changed the profile of pillar extrusions in such a way that the reaction force exerted will be normal to it.This reduces the stress concentration region and leads to a better solution
- Side extrusion for support has been removed to facilitate the engine - axle connection
- The base extrusion was removed to reduce model weight
- As a result I generated a 10mm mesh for better analysis.
- Bottom rectangular tubes were replaced with square tubes
- Fillets were added to reduce stress concentrations

Model

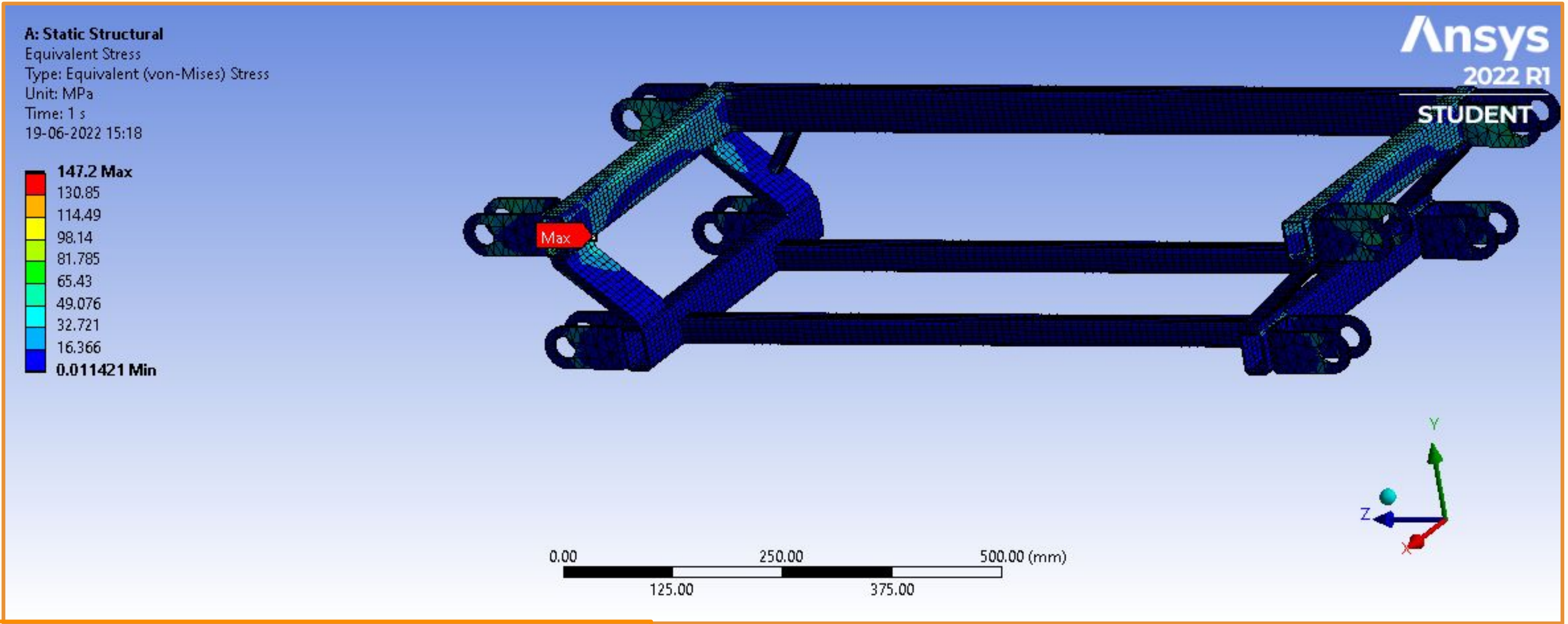


New optimized
Extrusion profile



ITERATION 2:

Maximum Von Mises Stress:

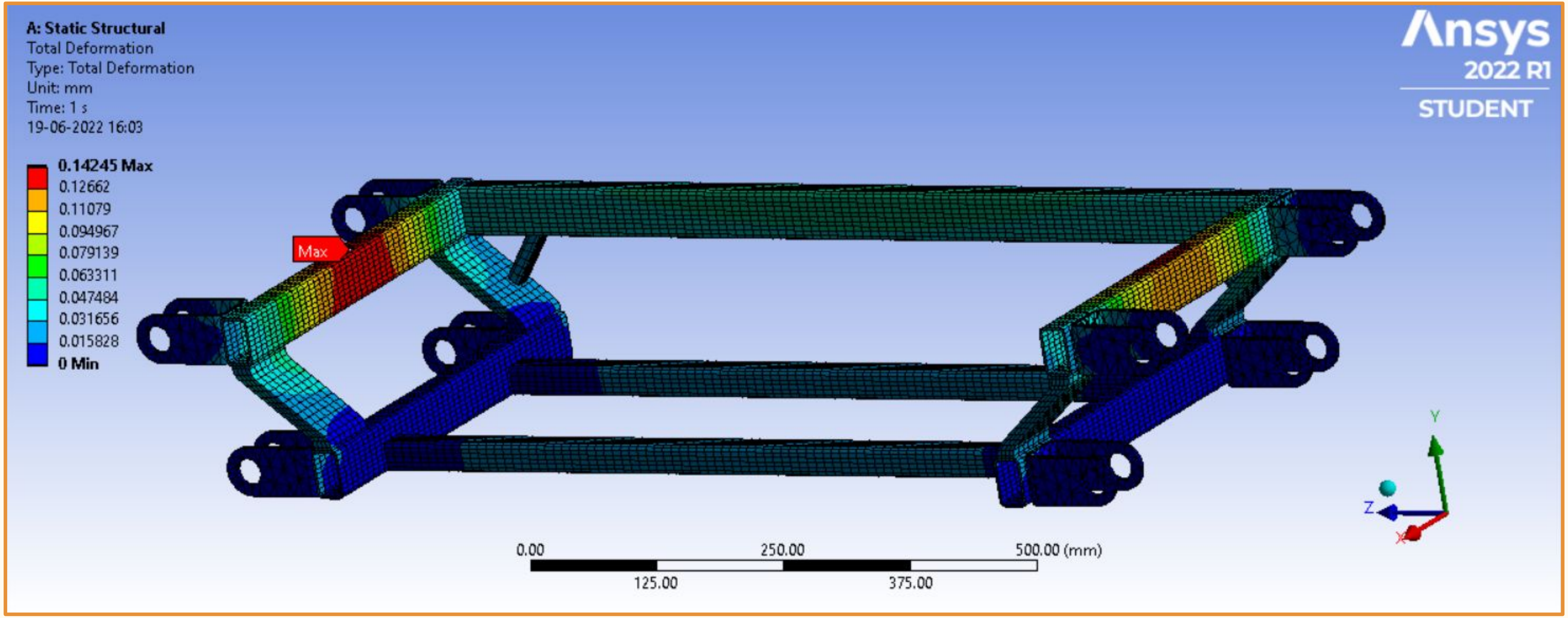


Maximum Stress Value: 147.2 MPa
Maximum Permissible value: 177.5 MPa
F.O.S = 2.411

Mesh element size of 5 mm was given with medium span angle centre

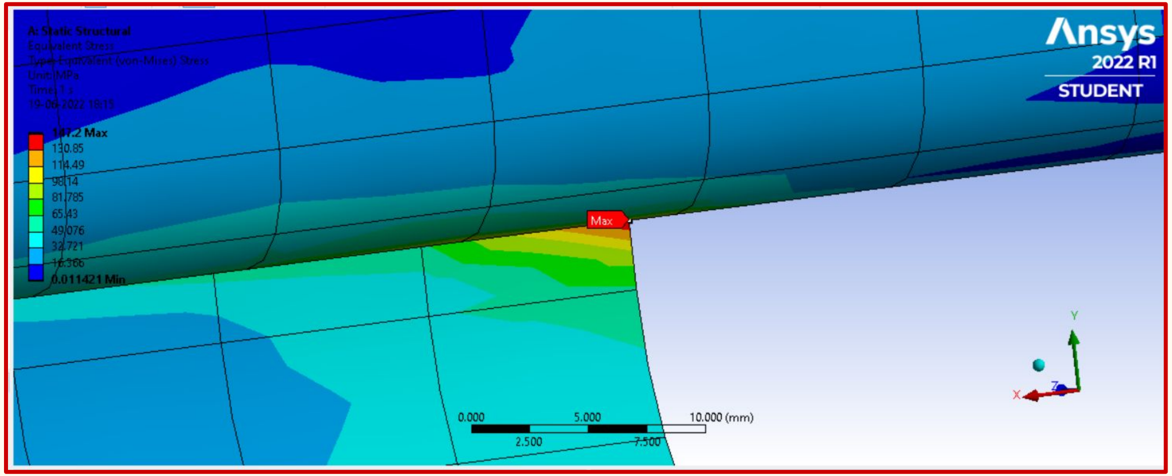
ITERATION 2:

Maximum Total deformation:



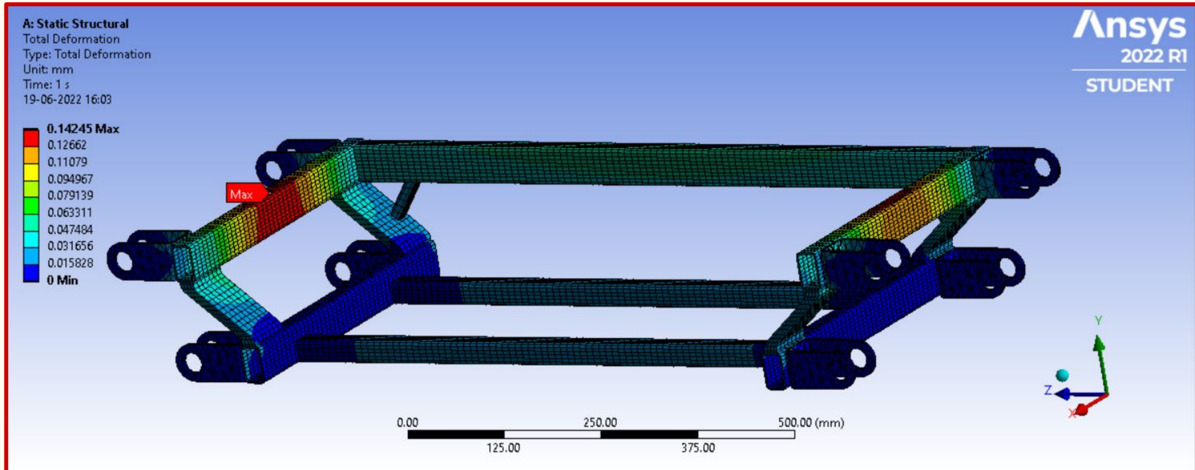
Maximum Total Deformation Value: 0.142 mm
Maximum Permissible value: 0.4 mm

Region of Max stress and deformation:



Maximum stress is located at the edge of pillar extrusion

Maximum deformation is found at the middle



Result of Iteration 2:

Maximum Stress obtained : 147.2 MPa

Maximum Total Deformation obtained : 0.14 mm

Weight of Model : 19.688 Kg

Material Used : ST52

Factor Of Safety : 2.411

Flange thickness was reduced to 5 mm.

Pillar extrusion was reduced by 5mm on all sides.

Support pillar extrusion was removed

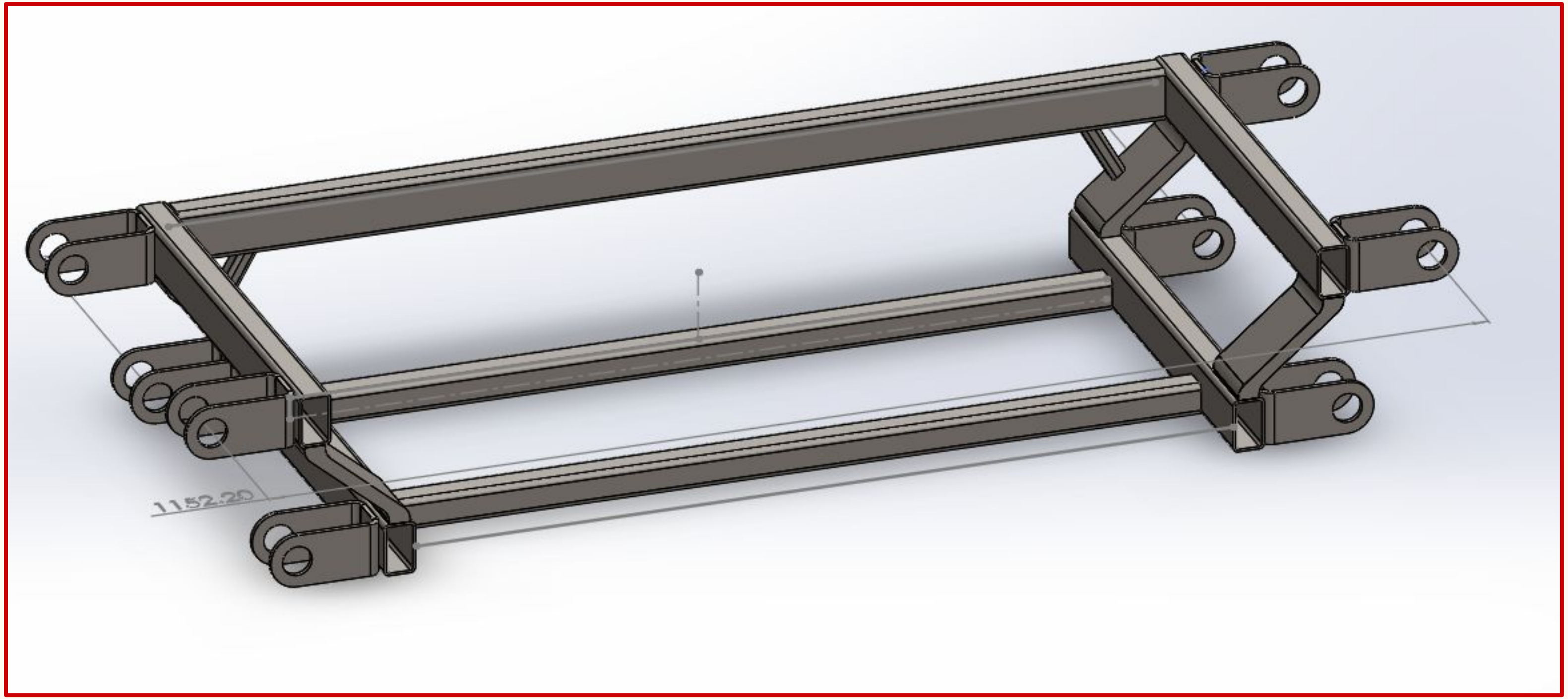
DATA COMPARISON

	Maximum Stress (MPa)	Maximum Total deformation(mm)	Weight of rear subframe(kg)
Initial CASE	40.64	0.098	58
Post Iteration 1	178.8	0.11538	47.4
Post Iteration 2	147.2	0.14245	19.69

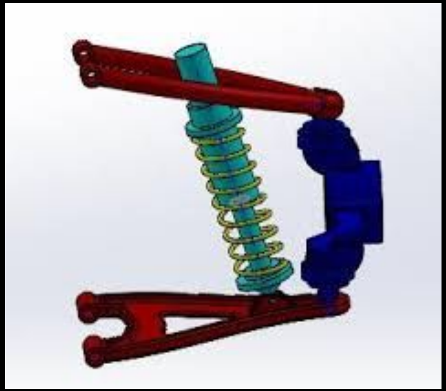
Note: Material was changed from structural steel to ST52 from Iteration 1


Conclusion

Finalized Model :

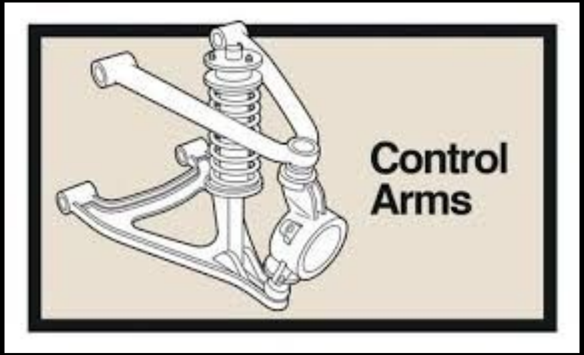


As a result of second iteration, the model we obtained weighed 19.688 kg. Excess parts are removed and stress distribution was even due to the profile shape.Maximum stress obtained is around 147.2 MPa.Hence we get a F.O.S around 2.411.The total deformation is around 0.14 mm which is less than the maximum limit(0.4 mm).Flange thickness were reduced by half.Most of the stress concentration occurs at the corner of pillar extrusion.Most of the deformation occurs at the middle.So from two successful iterations we got our finalized model. All the mounting points are maintained in its respective position in all the iterations.



TASK 2

(DESIGN AND ANALYSIS OF CONTROL ARM-FBR)



OBJECTIVE:

- To design the Front Bottom Right control arm using Solidworks
- To analyse the effect of load in the control arm
- To modify the design and to iterate again
- To get a model with less weight and more FOS

UNITS:

Stress - MPa , Total deformation - mm , Fatigue life - no of cycles,
Safety factor - No unit

CONCEPT:

Standard load of 9000N will be applied to the model in Z direction.According to the results obtained iteration of the model will be done

CONTROL ARM - FBR(Front Bottom Right) :

- A vehicle's suspension is the system of shocks, springs, and linkages underneath the car body that connects the chassis to the wheels. Its purpose is to support the vehicle's weight and control its ride quality, handling abilities, and overall dynamics while in motion.
- The control arms are one of the core components of a suspension system and serve as the direct connection points between the front wheel assemblies and the vehicle's frame. The control arms allow a driver to steer a car while also guiding the wheels up and down with the road surface.
- Many vehicles have an upper and a lower control arm for each front wheel, connecting to the highest and lowest steering knuckle points. This architecture makes for a more substantial assembly, ensuring balanced wheel control and stability.
- A control arm purpose is straightforward. It connects the steering knuckle to the frame and stabilizes the vehicle by allowing the chassis and the wheels to move in unison while the vehicle is in motion. Ultimately, control arms help achieve coordination between the suspension and steering systems, dampening the ride and giving the driver the ability to maneuver the vehicle.

ANALYSIS

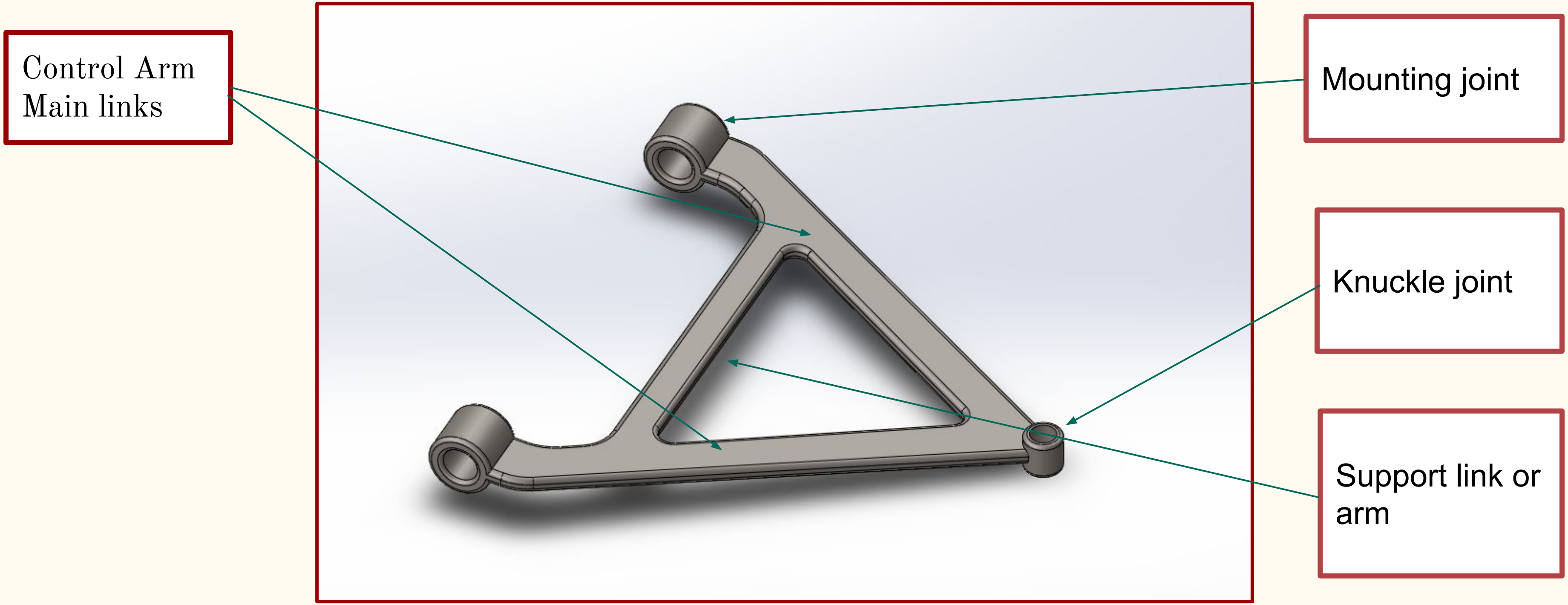
PROCEDURE FOR ANALYSIS

- Solidworks - Design
- Ansys Workbench - Analysis

DESIGN IN SOLIDWORKS

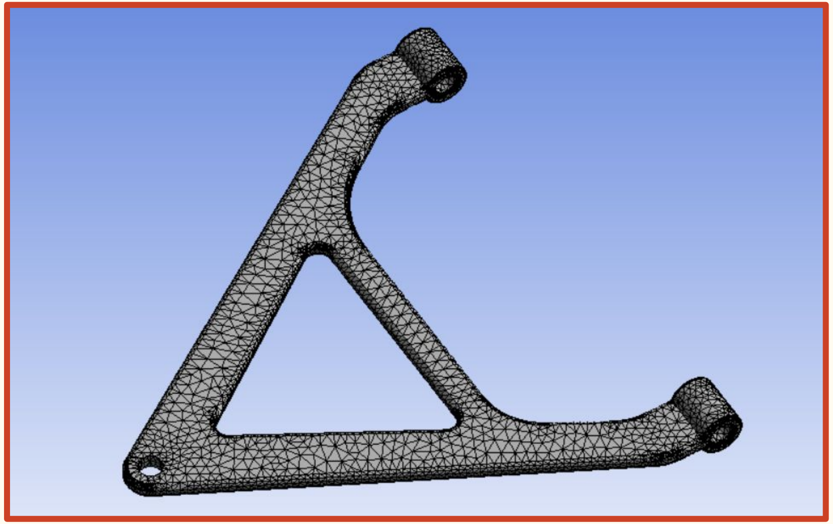
- The model was designed in solidworks .The lengths of the control arm were taken from provided hardpoints
- The design was made over the hardpoints considering it at the middle of the clamps.
- This design composed of only extruded parts.As the final product should will be done using casting,I used only extrusions
- Fillets were given inorder to reduce accumulation of stress
- The size of mounting joint were given as per my previous subframe clamp distance
- From the results obtained in analysis, changes were made

INITIAL DESIGN



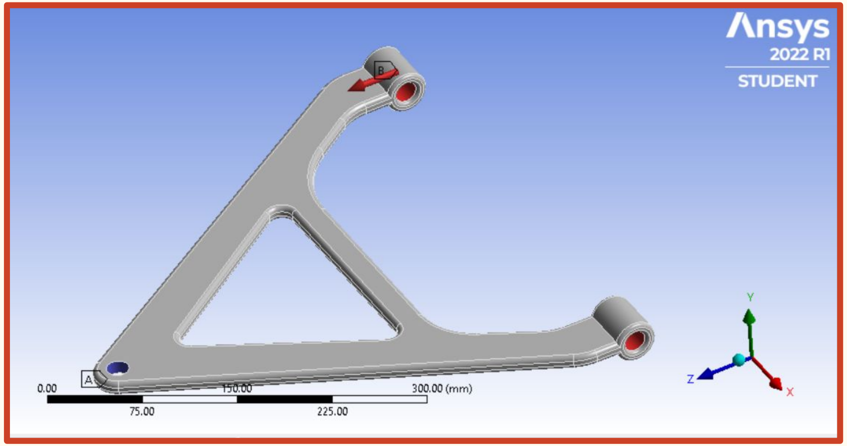
ANALYSIS IN ANSYS

- I used ansys workbench for analysis of my model under loading.
- The design was imported through .STEP file format from Solidworks
- Then a mesh size of 5mm was given.I setted the span angle centre to ‘fine’.As a result of this we can get better results at the corner
- Respective forces and fixed supports were given to the model
- Force of 9000N will act from either +z direction or from -z direction
- We will obtain Safety factor, Equivalent stress, Total deformation and Life of the model from ANSYS
- Material used is Structural Steel
- These results obtained will be used to analyse the design



MESH CREATED
ELEMENT SIZE : 5mm
SPAN ANGLE CENTRE : Fine

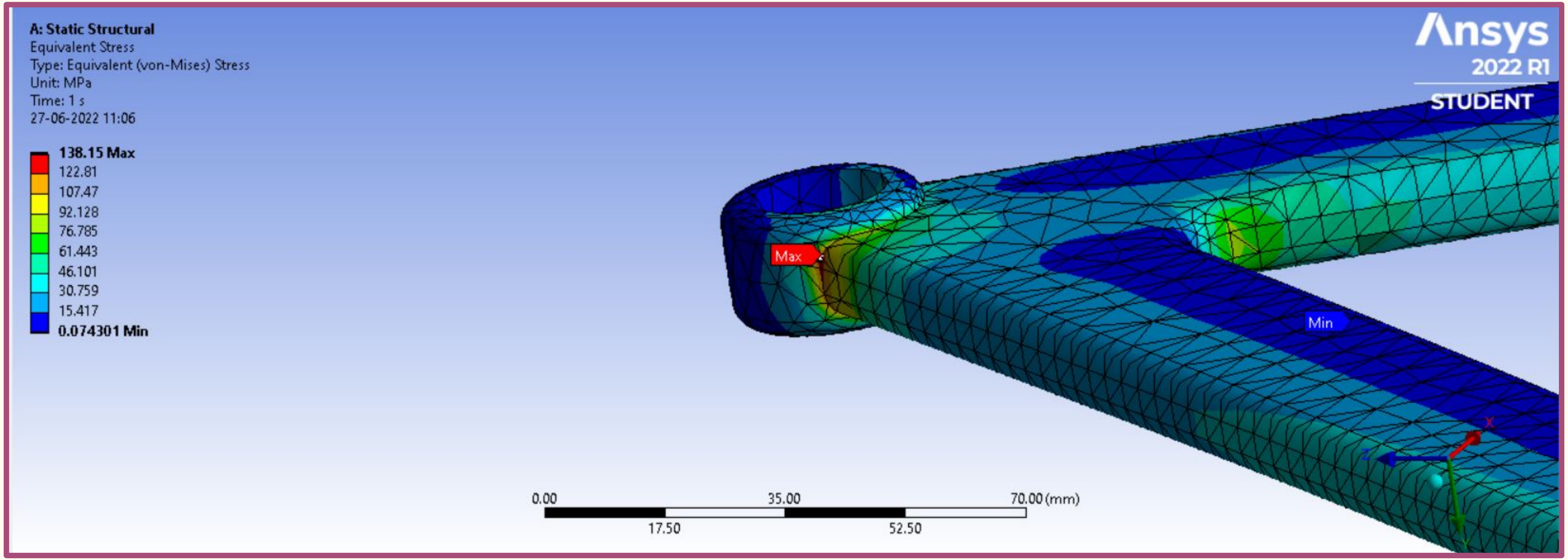
FORCE + FIXED SUPPORT
FORCE APPLIED: 9000 N



ITERATION

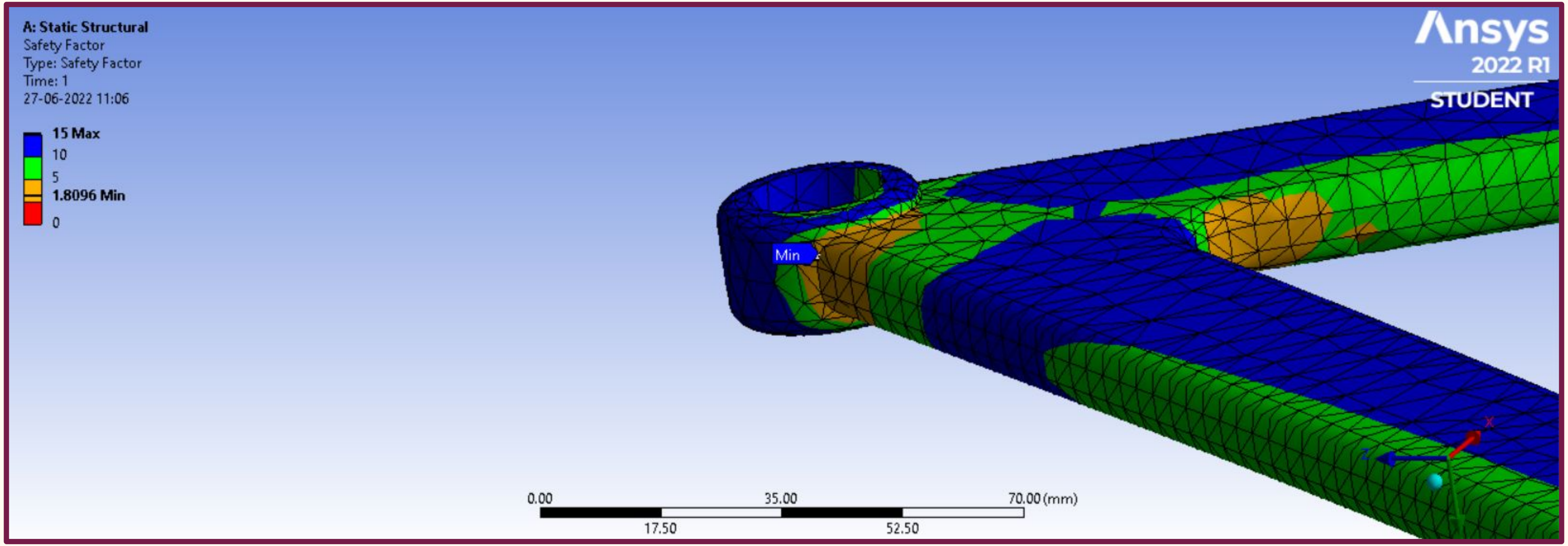
BEFORE ITERATION

Maximum Stress:



BEFORE ITERATION

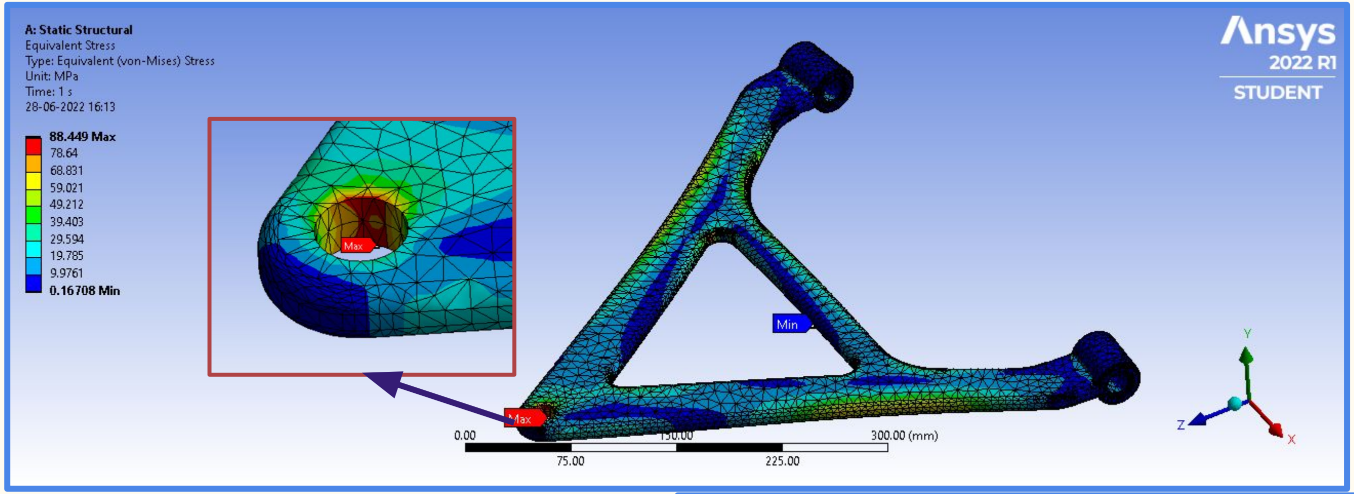
SAFETY FACTOR LIMIT:



Changes Made to the design

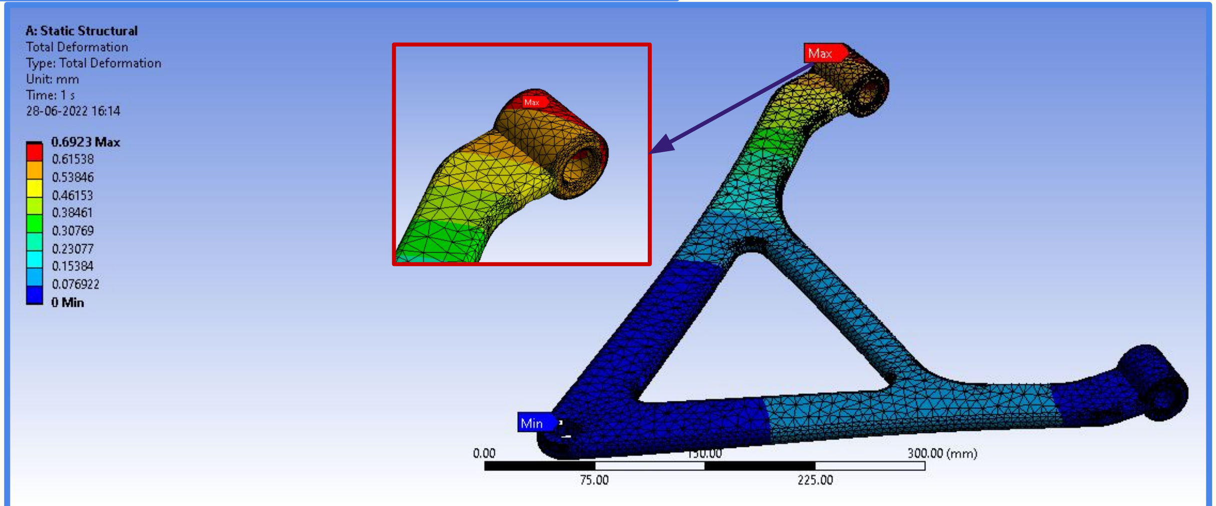
- Removed the excess extrusion present at the knuckle joint
- Reduces the extra thickness of control arm
- Decreased the thickness of mounting joints
- Additional fillets were given to reduce the stress accumulation at the corners
- More fillet size was given to the corner with more length
- Increased the support arm length by 5 mm

POST ITERATION

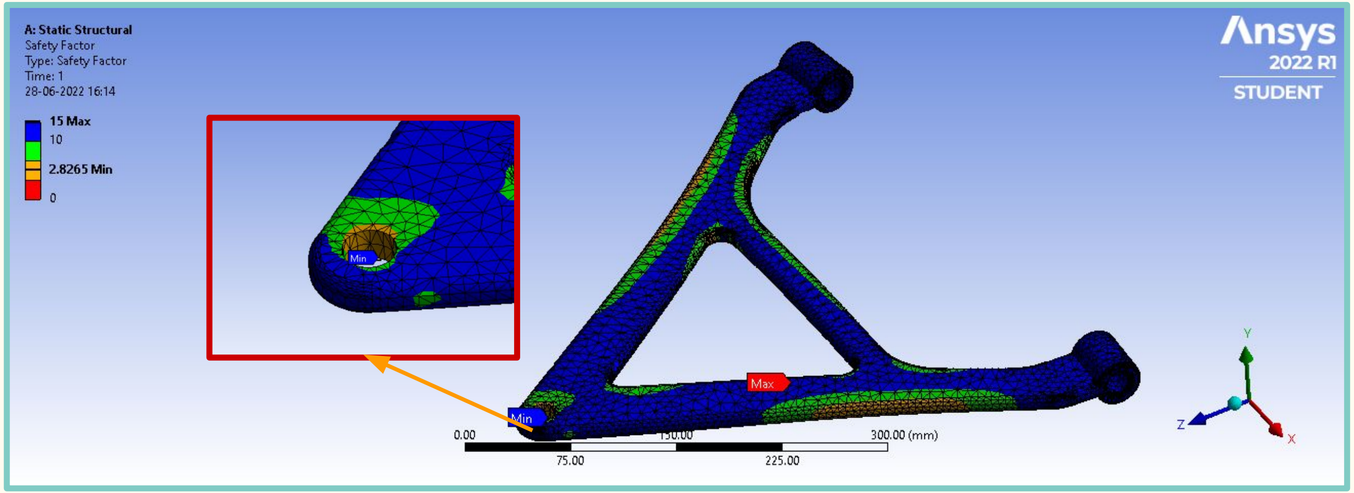


MAXIMUM EQUIVALENT VON MISES STRESS

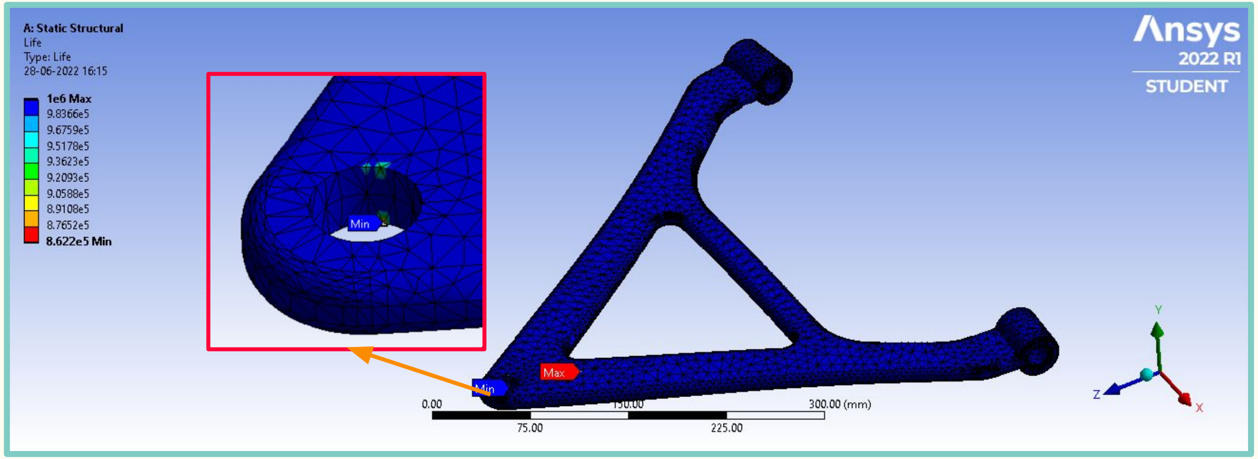
MAXIMUM TOTAL DEFORMATION



POST ITERATION

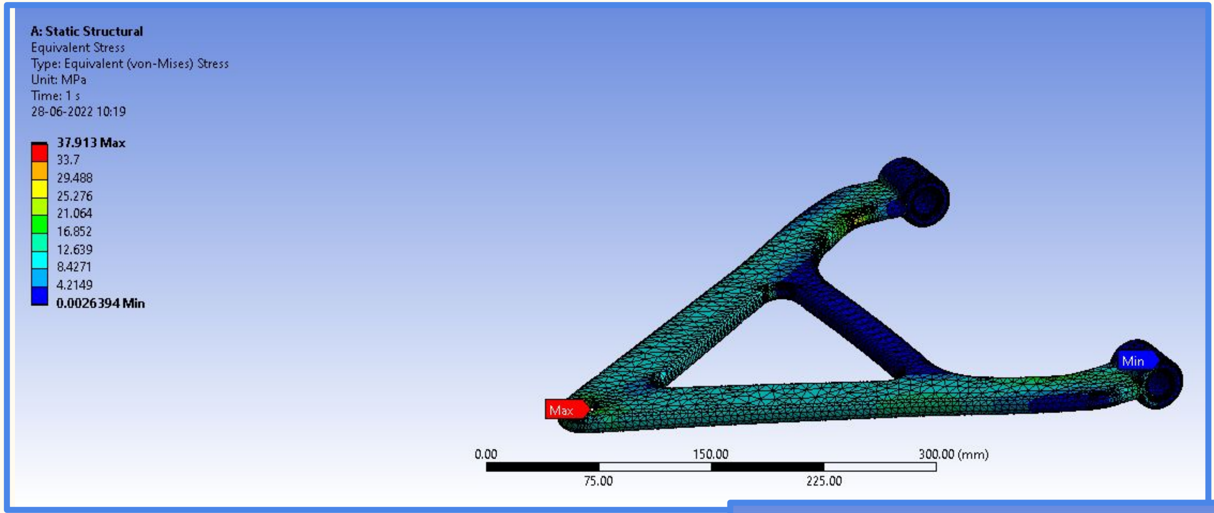


MINIMUM SAFETY FACTOR LIMIT



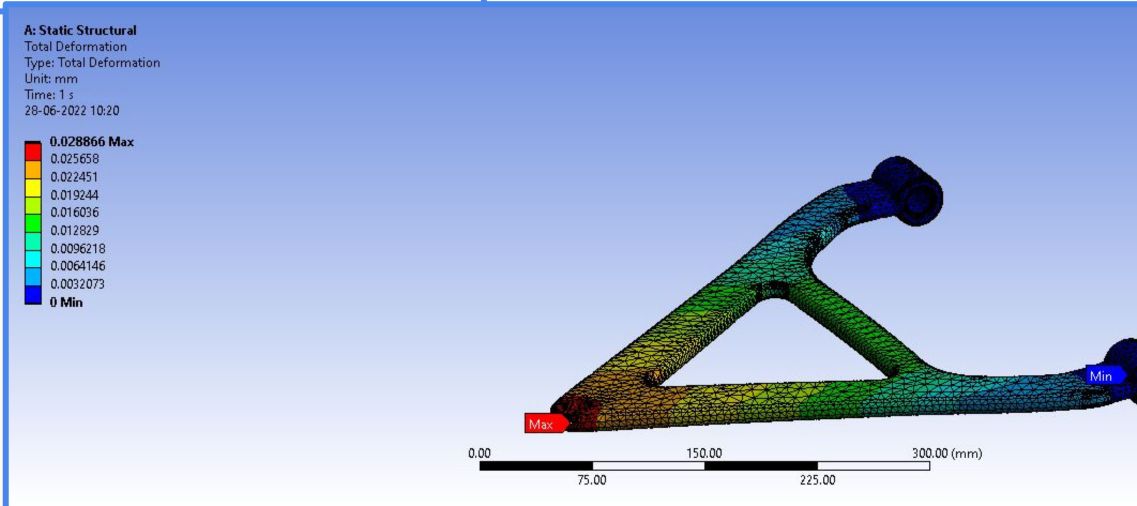
MINIMUM FATIGUE LIFE

POST ITERATION (Force from Knuckle joint);

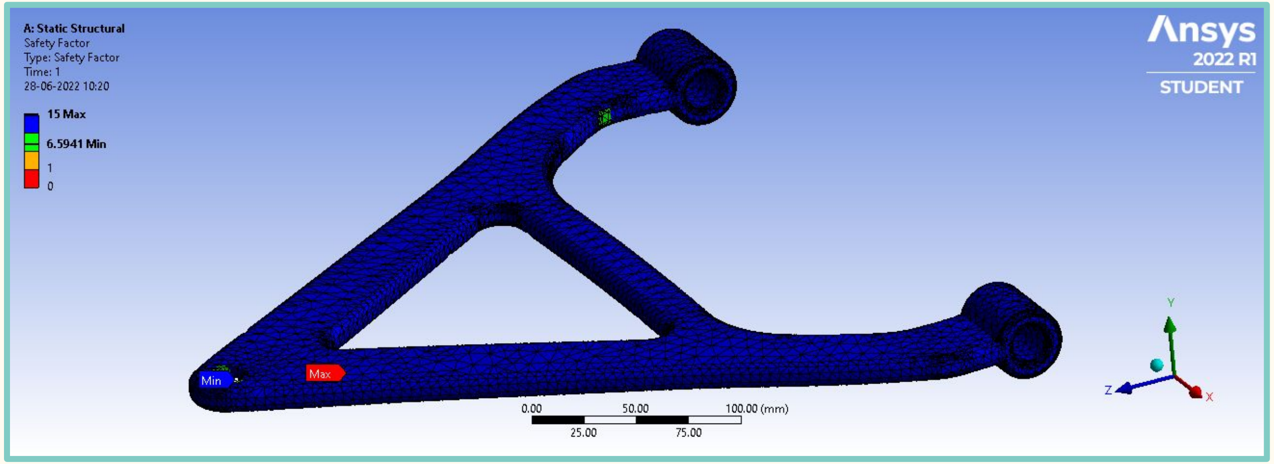


MAXIMUM EQUIVALENT VON MISES STRESS

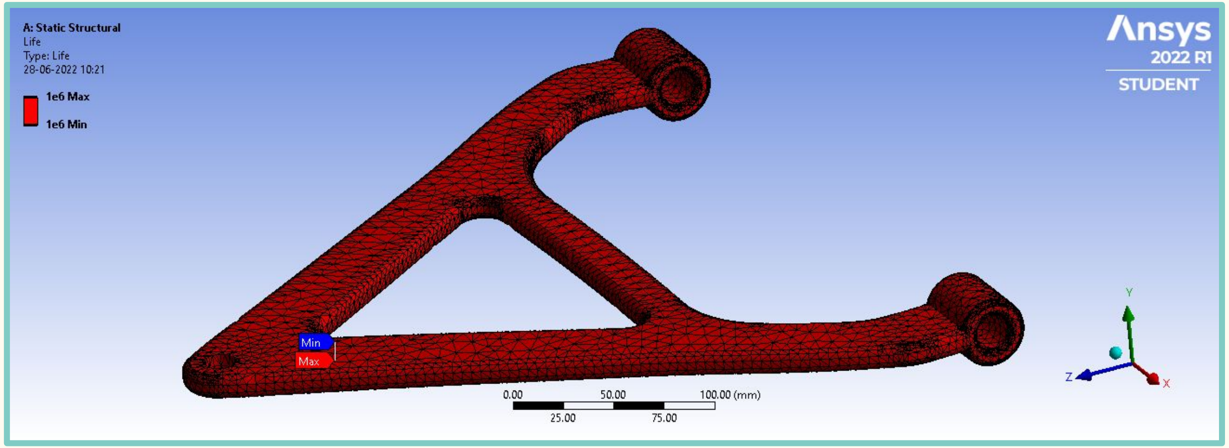
MAXIMUM TOTAL DEFORMATION



POST ITERATION(Force from Knuckle):



MINIMUM SAFETY FACTOR LIMIT



MINIMUM FATIGUE LIFE

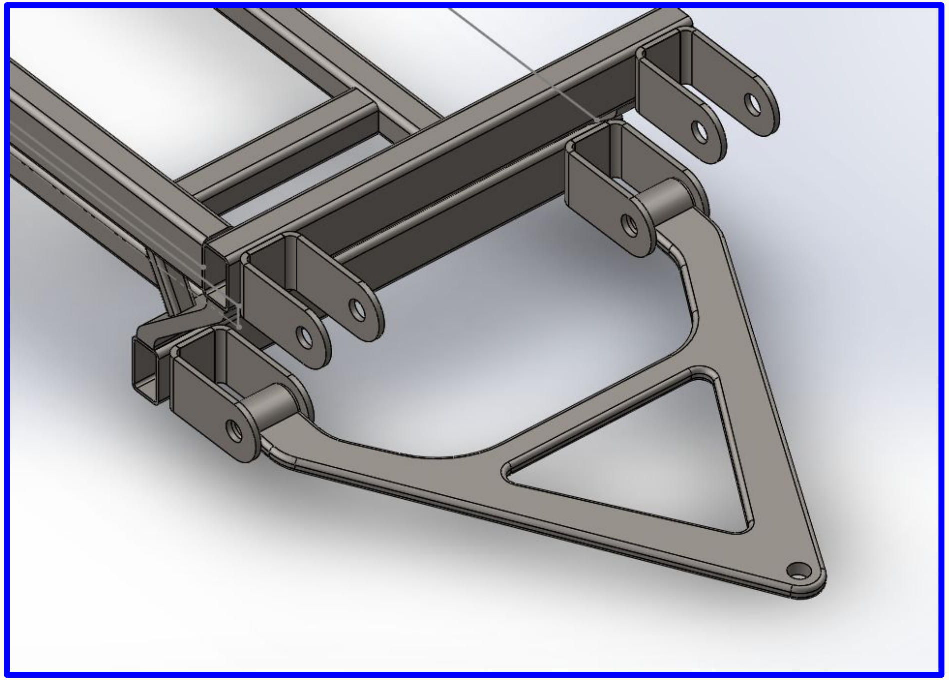
DATA COMPARISON

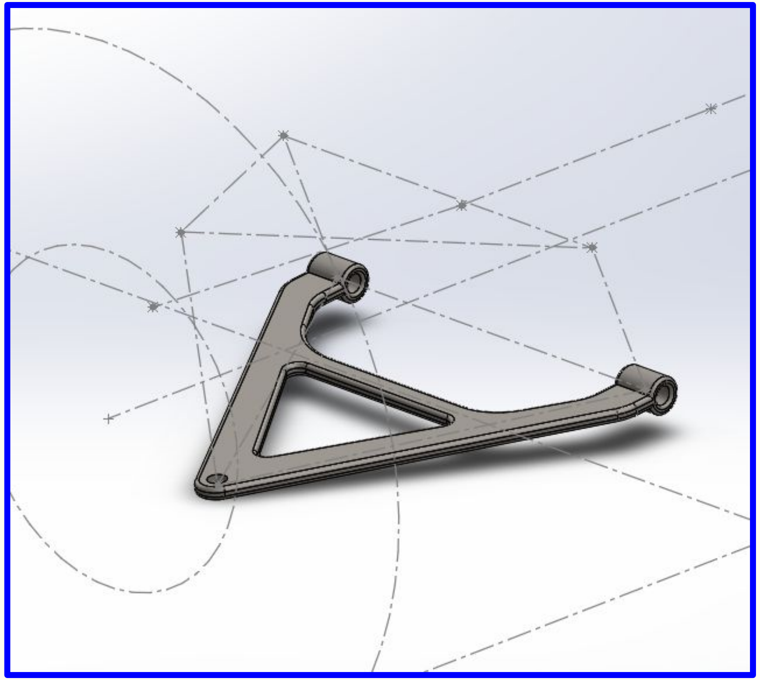
	Maximum Stress(MPa)	Maximum Total deformation(mm)	Safety factor limit	Minimum Life cycle(x 10^5)	Weight of the control arm(kg)
Before Iteration	138.15	0.732	1.8096	5.021	4.55
Post Iteration	88.449	0.6923	2.8265	8.622	3.94

LIMITING CASE: Minimum Safety factor - 2 ,Max Weight of control arm - 5 kg

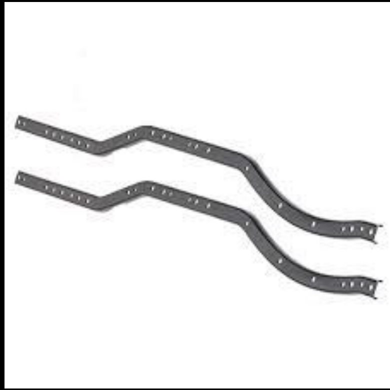

Conclusion

FINALIZED DESIGN





- The weight of the model is under limit and it holds well under both the direction of load applied
- Fillets given in these model increases its endurance (i.e life)
- The model is also perfectly attaching in the subframe design
- The safety obtained is more than 2.5.
- The region of maximum stress is around knuckle joint corner and its value is considerably fine
- Thus as a result of Iteration 1, we have obtained a more sustainable model



TASK 3

(DESIGN OF CHASSIS RAILS)

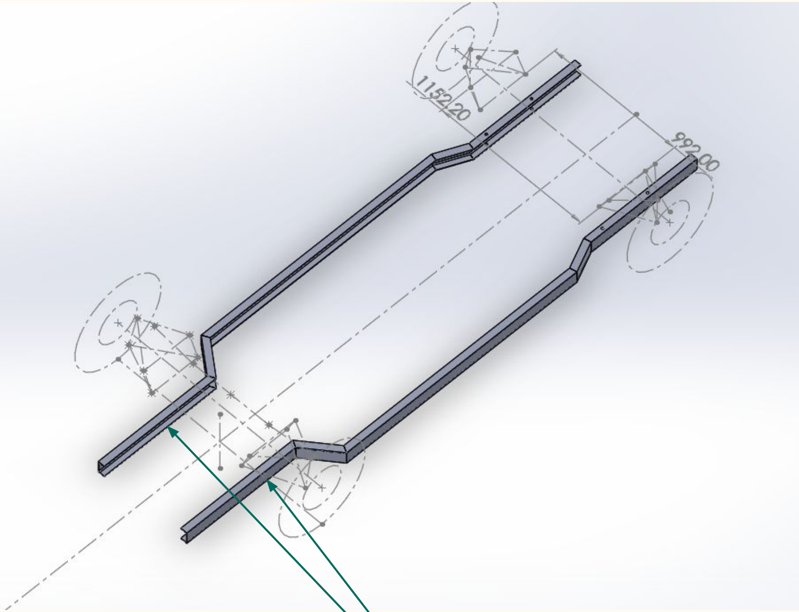
CHASSIS RAIL :

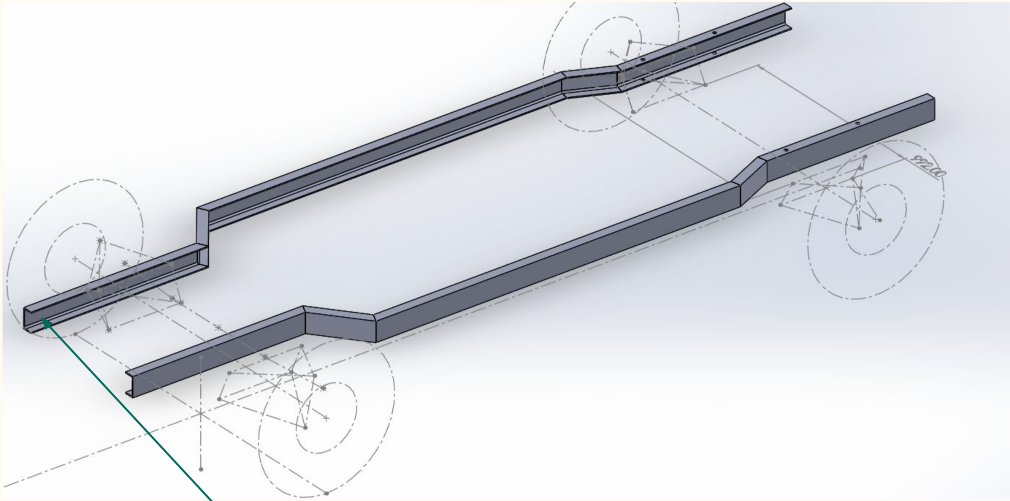
- Chassis rails transfer the load to the subframe.It also acts as a holder for the battery box.
- Chassis rails are bolted to the top portion of the subframe.
- Typically, the material used to construct vehicle chassis and frames is carbon steel or aluminium alloys to achieve a more lightweight construction.
- In the case of a separate chassis, the frame is made up of structural elements called rails or beams. These are ordinarily made of steel channel sections, which are made by folding, rolling, or pressing steel plate.
- The battery box is contained in this case by a C-shaped profile.

DESIGN IN SOLIDWORKS

- C channel beams are used in the design of the chassis rail.
- They are weldments with size of 100 x 10 mm.
- Using the hard point references, a sketch of the weldment profile was created.
- After constructing one side, the mirror property was applied to create the opposite side.
- In addition, I considered the battery box dimensions while determining the centre width and total length.
- Thus the design of chassis rails were made

Chassis rail design





Chassis rails

C Shaped profile


Conclusion

Conclusion:

- The design of the chassis rail was made by weldments.
- The battery box was placed in between the maximum width regions.
- Provided dimensions were used to design the chassis rails.
- The width and height of its position were taken from hard points.
- Thus, the final chassis rail design was made.

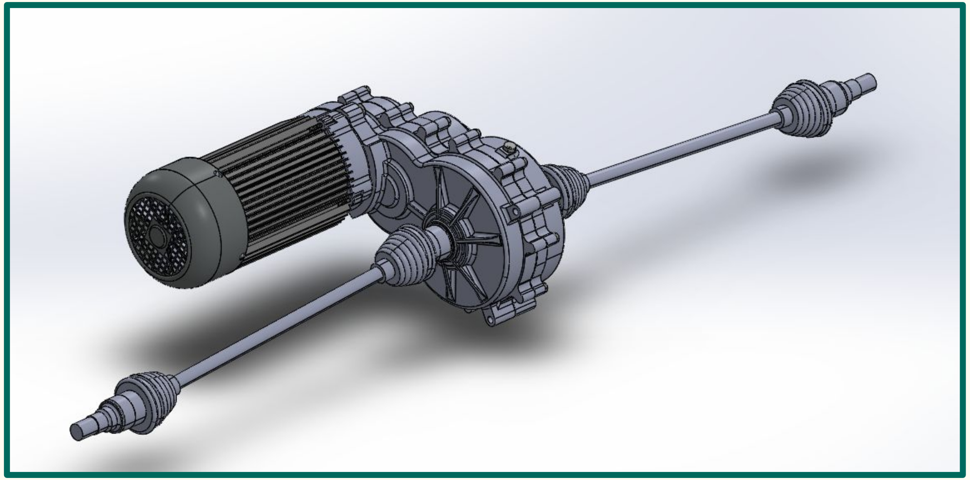
ASSEMBLY

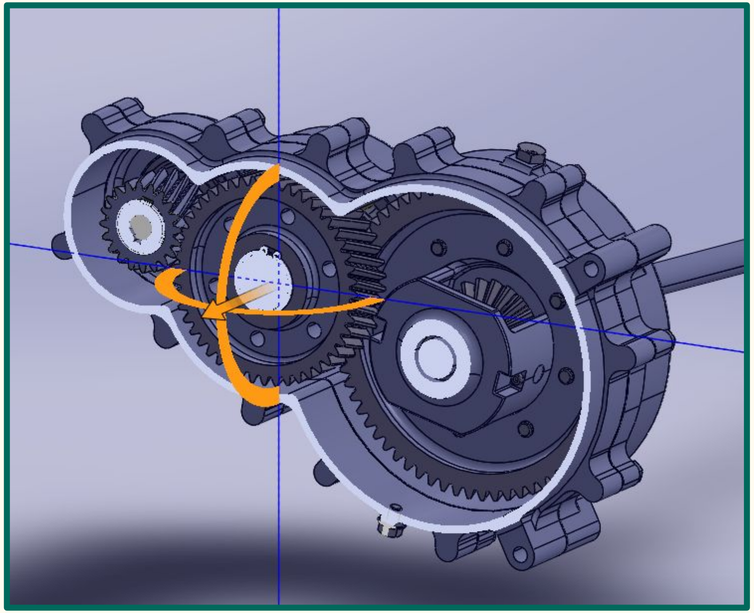
(Assembly of provided and designed components)

OBJECTIVE:

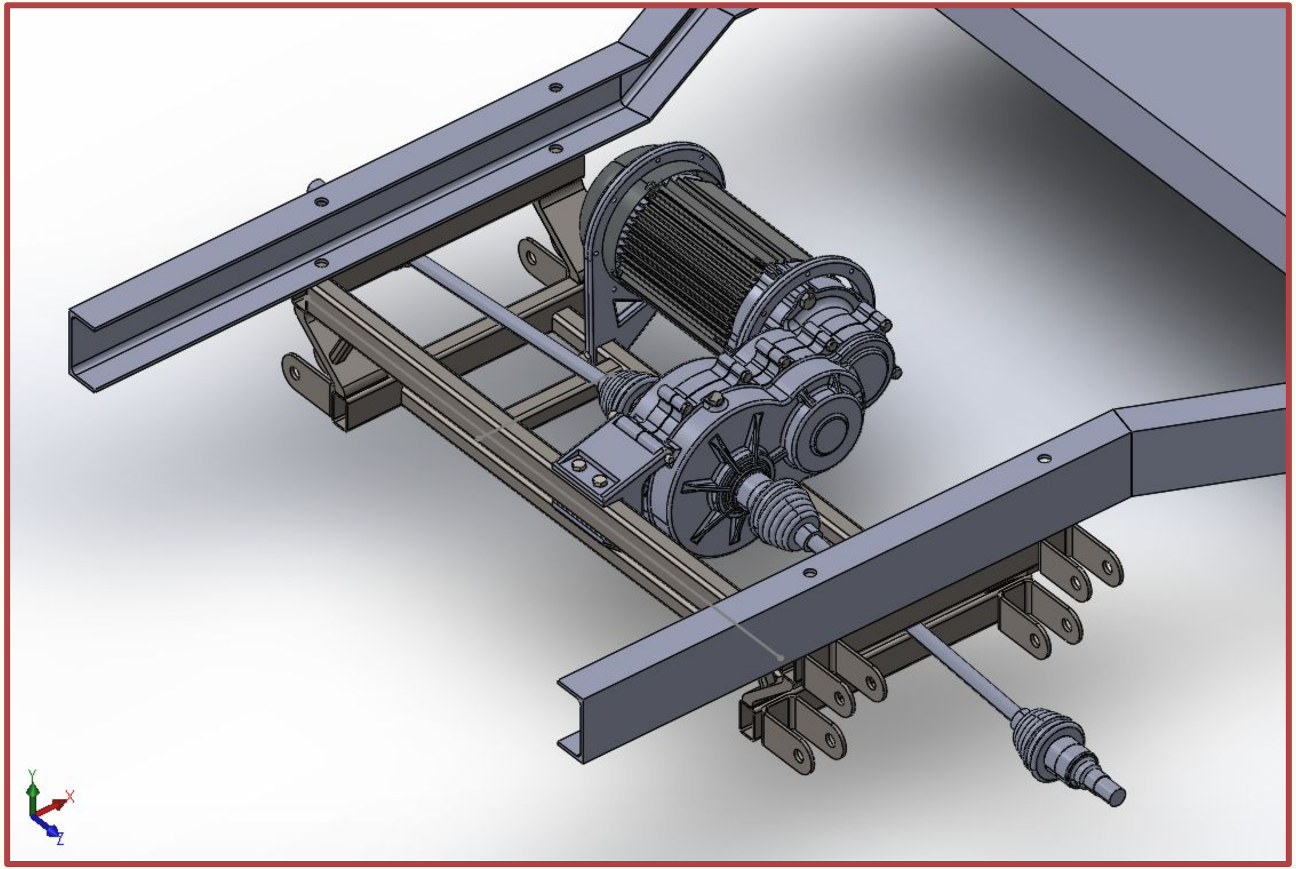
- To assemble all the early made components to make a basic infrastructure of modular frame
- To position battery box in side the Chassis rails without intersection
- To design support mountings for motor and gearbox and assemble it
- Motor and Gearbox part files were designed and proper support were given in addition to it.
- To insert bolt and to provide proper spacing for rear axle to move independently

Design of Differential Gearbox and Motor



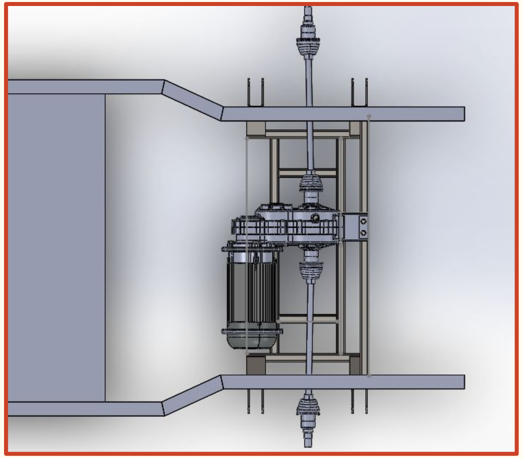


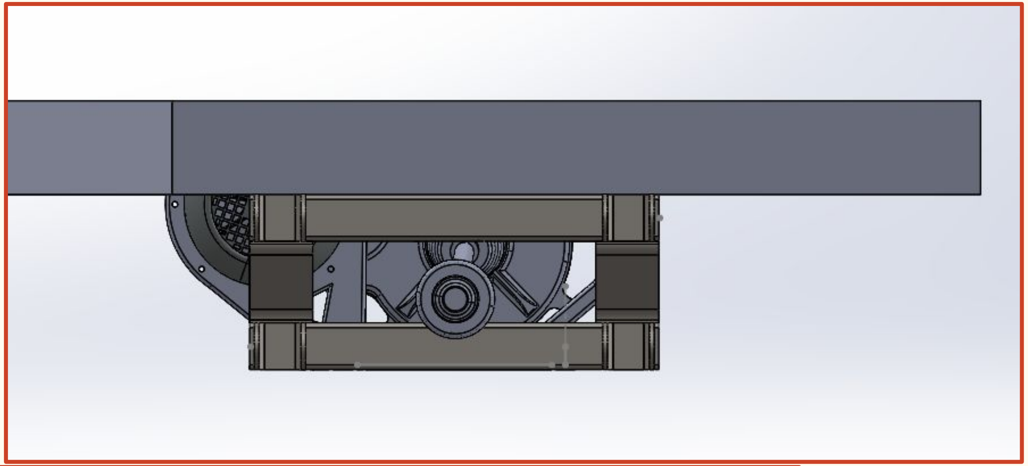
ISOMETRIC VIEW OF REAR END

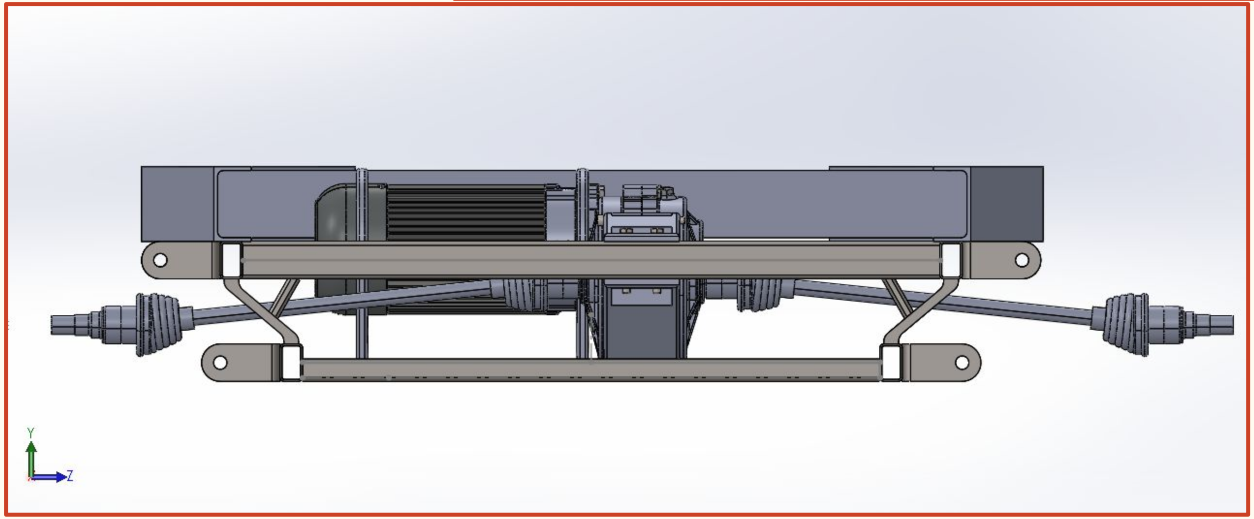


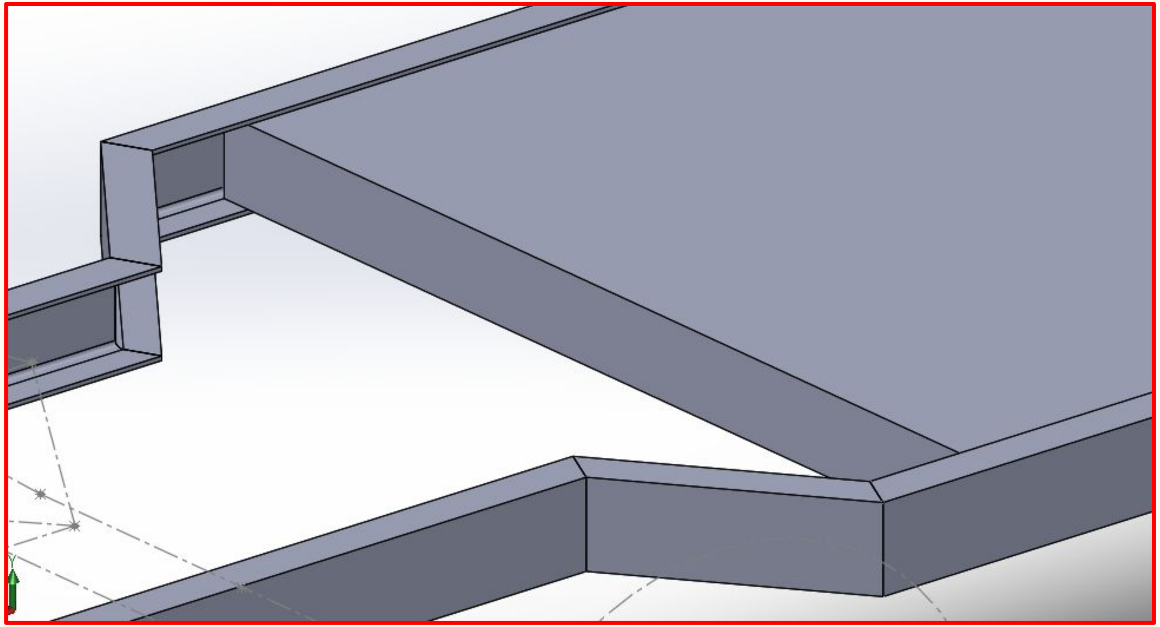
MATES :

- On both rails, a coincident mate was assigned to the battery box.
- Coincident mate was also given to the chassis rail such that the total weight is only focused on weldments and not on flanges.
- As a result, we have the most space for the rear axle to move.
- Concentric mate was given to the support structure at the weldment and coincident mate was also applied at the gearbox connection.
- Extra support was such that the motor would hold steadily in its place. Lack of this support might cause unnecessary torsional force on the gearbox and other components.
- Here, concentric and coincident mates were used.
- Then holes were created for the components, and fasteners (bolts) were used to fill the holes.
- The Smart fastener feature was used for this process.

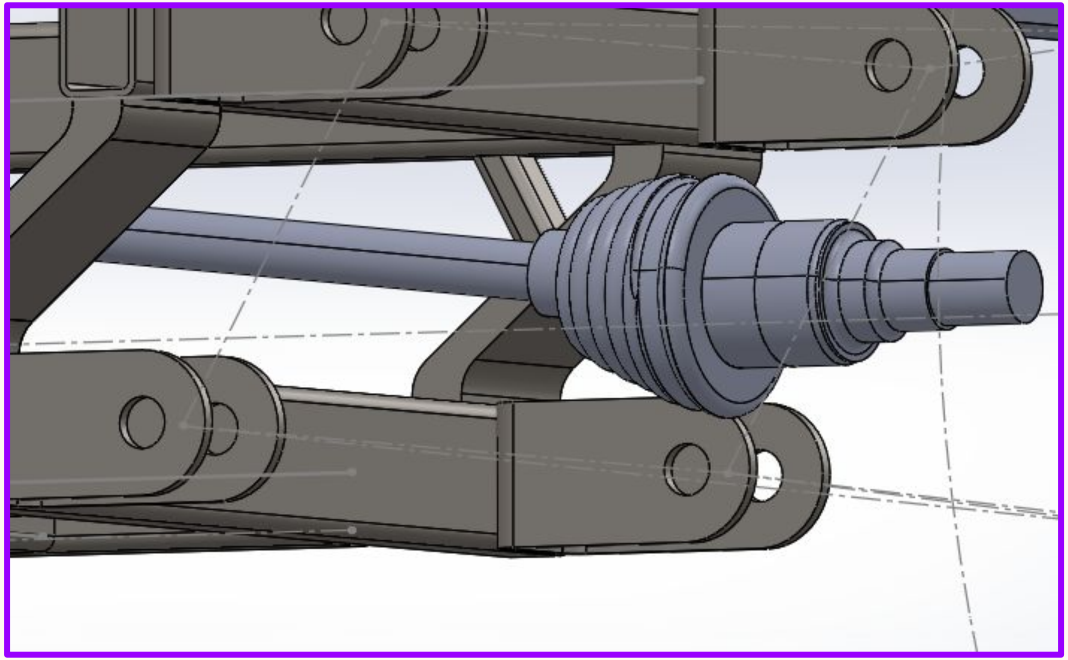





Y
Z

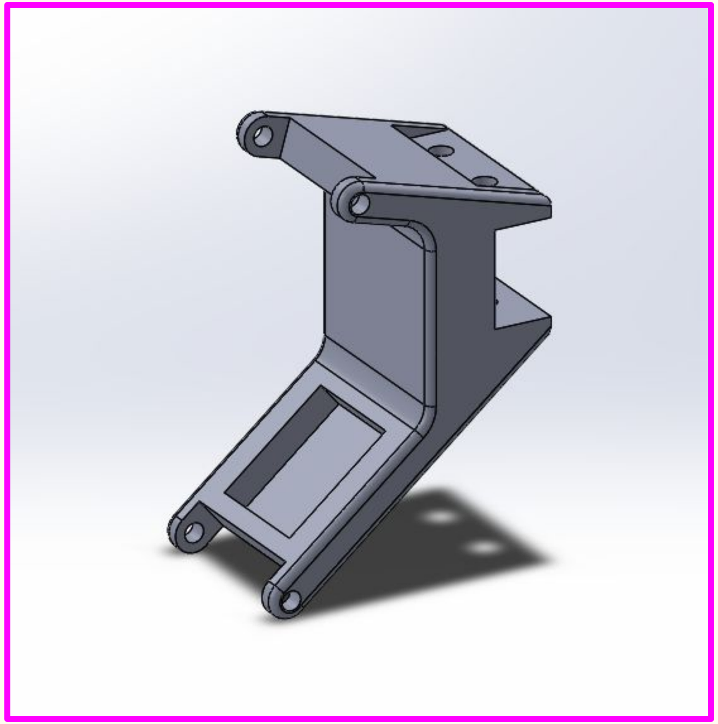


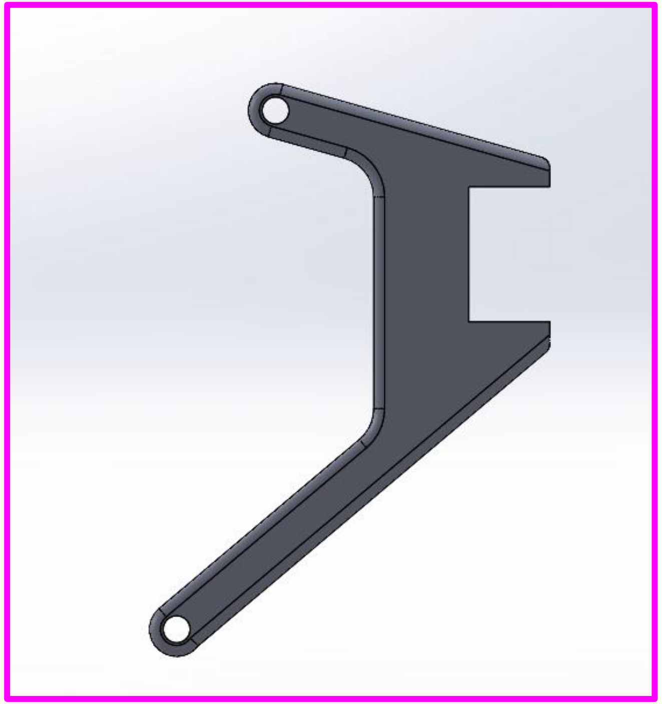
The battery box was inserted in between the chassis rails without intersection.This is done with the help of coincident mate



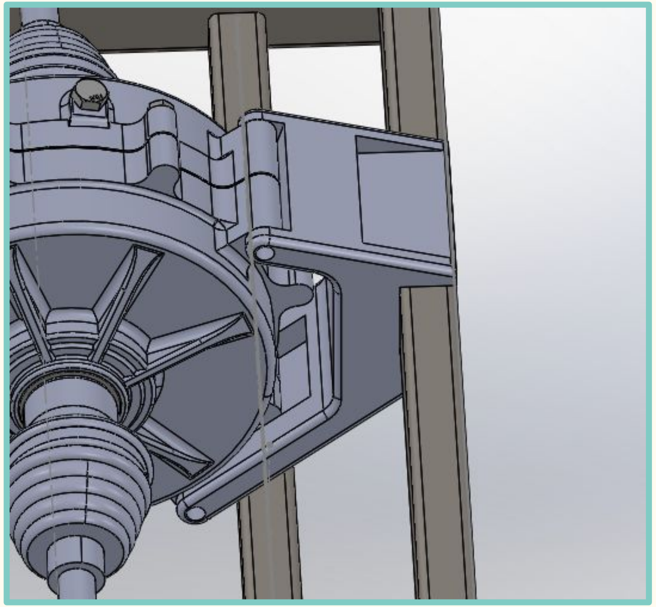
The assembly of gearbox and motor were made.The gap was made by making concentric mate with the midpoint of knuckle joint line.

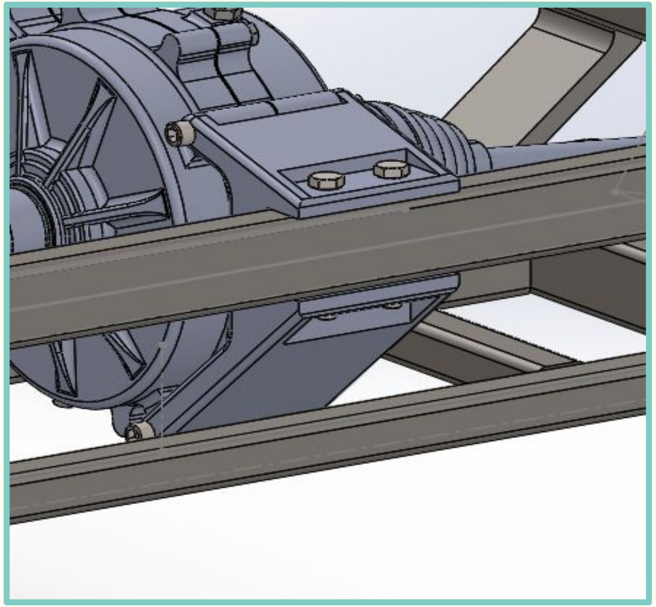
SUPPORT MOUNT FOR GEAR BOX





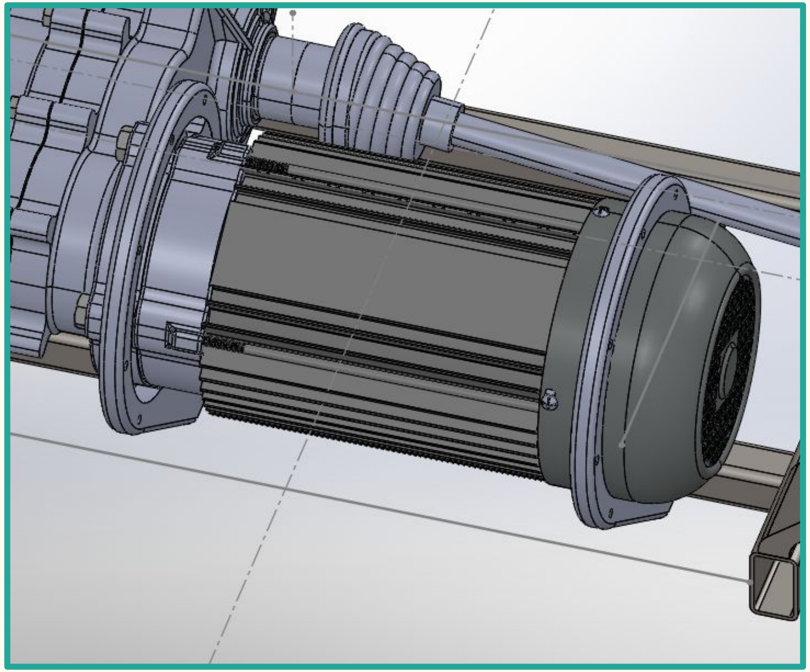
POSITION IN ASSEMBLY

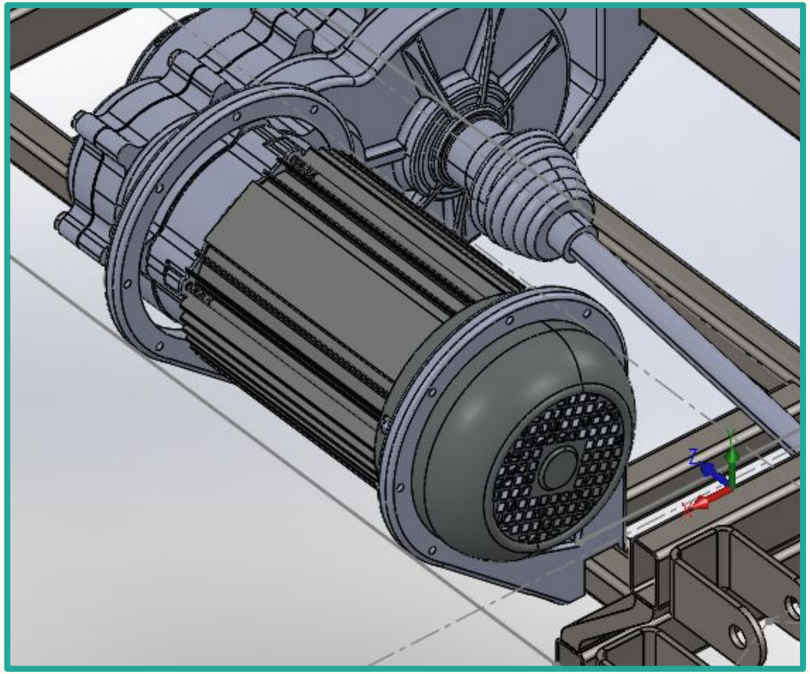


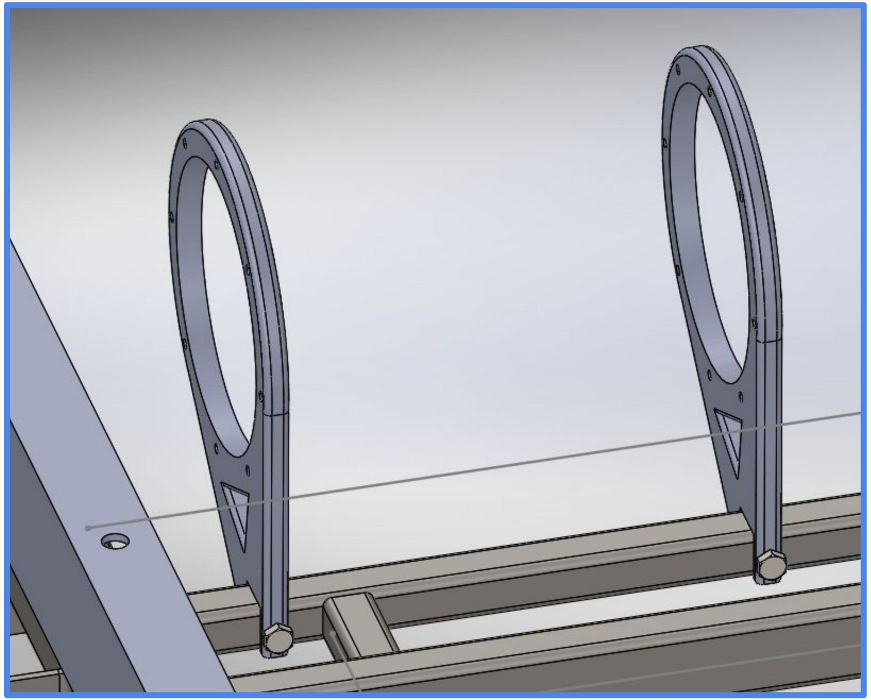


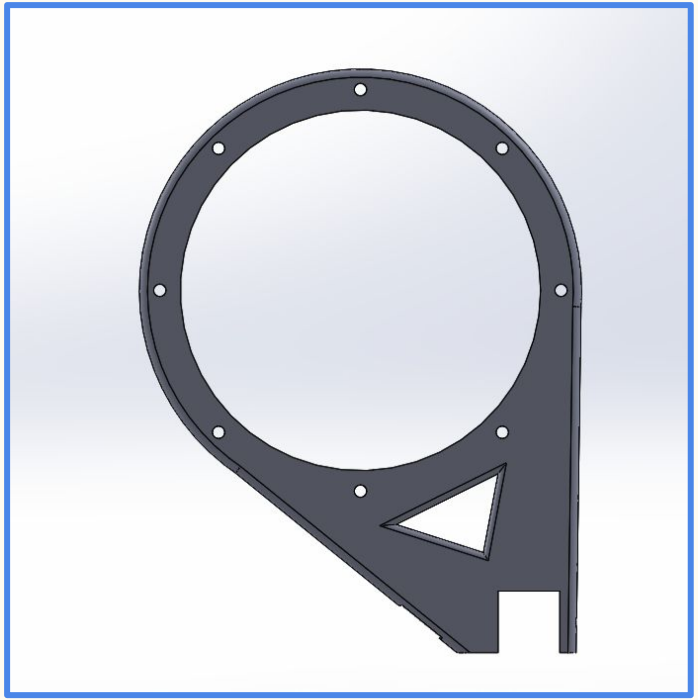
FINAL IMAGE WITH M10 HEX BOLT and HX-SHCS

EXTRA SUPPORT FOR MOTOR WITH M10 BOLT FOR MORE STABILITY

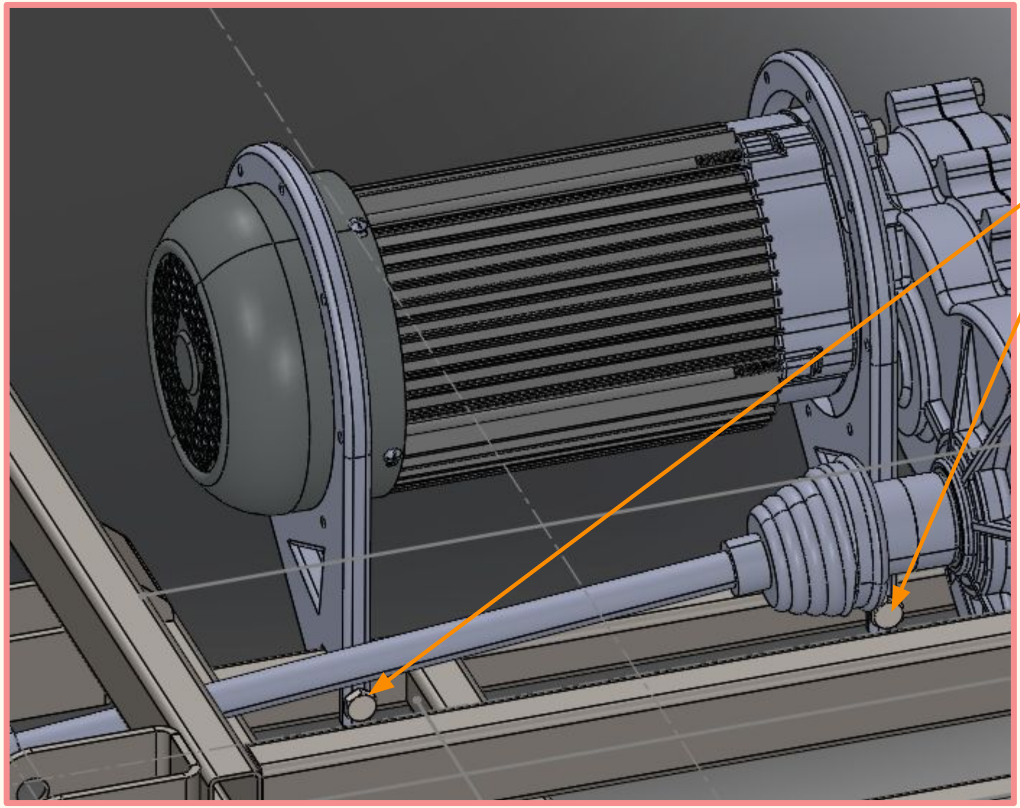






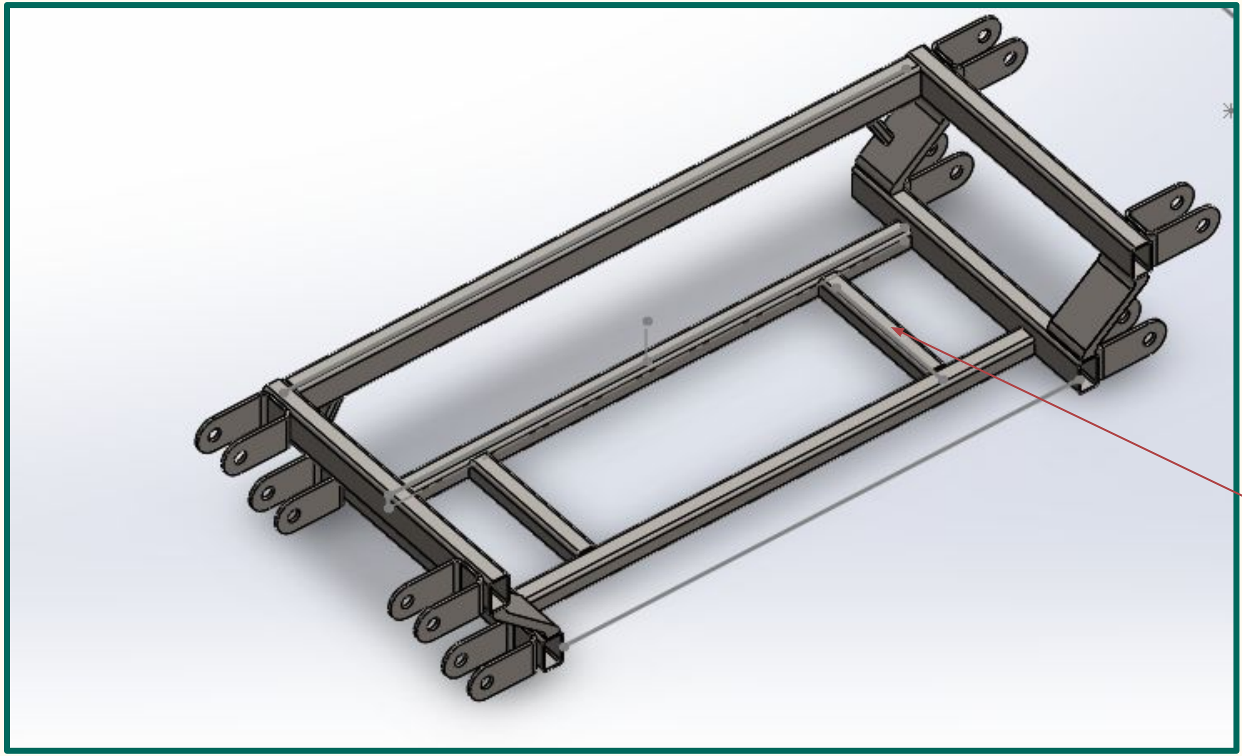


This design is made such that the total weight of motor will be spread through these two supports to Square tube.Additional support were given to square tube in order to maintain rigidity and to prevent twist.Coincident mate was given



M10 HEX BOLTS

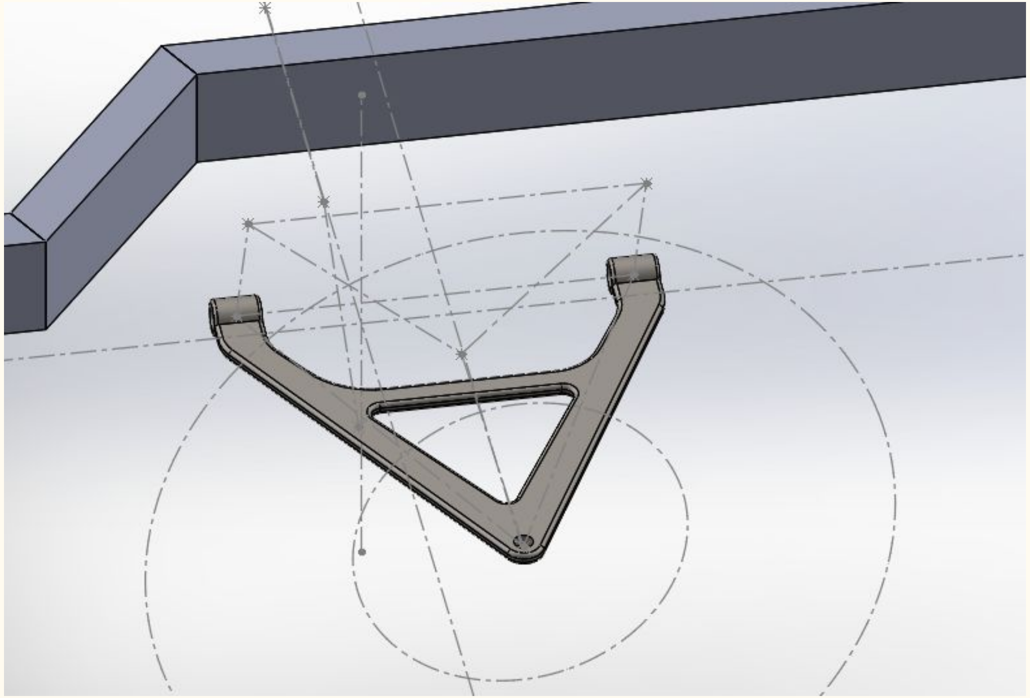
FINAL MODIFIED REAR SUBFRAME



Extra support Tube

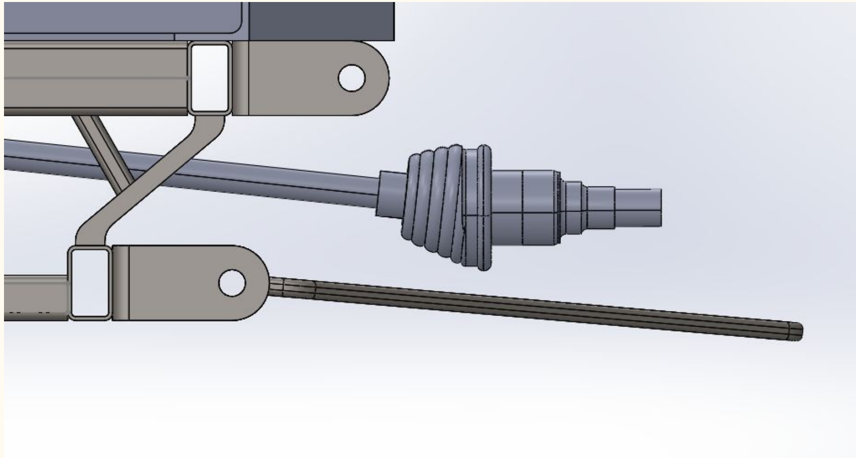
Note : The extra tubes were added as support.It can be removed if design sustains loading without it

CONTROL ARM - ASSEMBLY

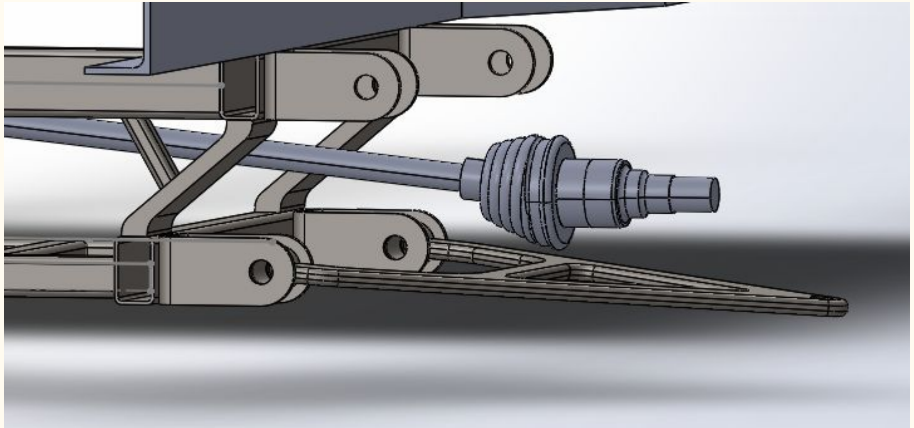


I am assigned to do the front bottom right control arm.Here we can notice that it coincides perfectly around the hardpoints.

SIDE VIEW



ISOMETRIC VIEW



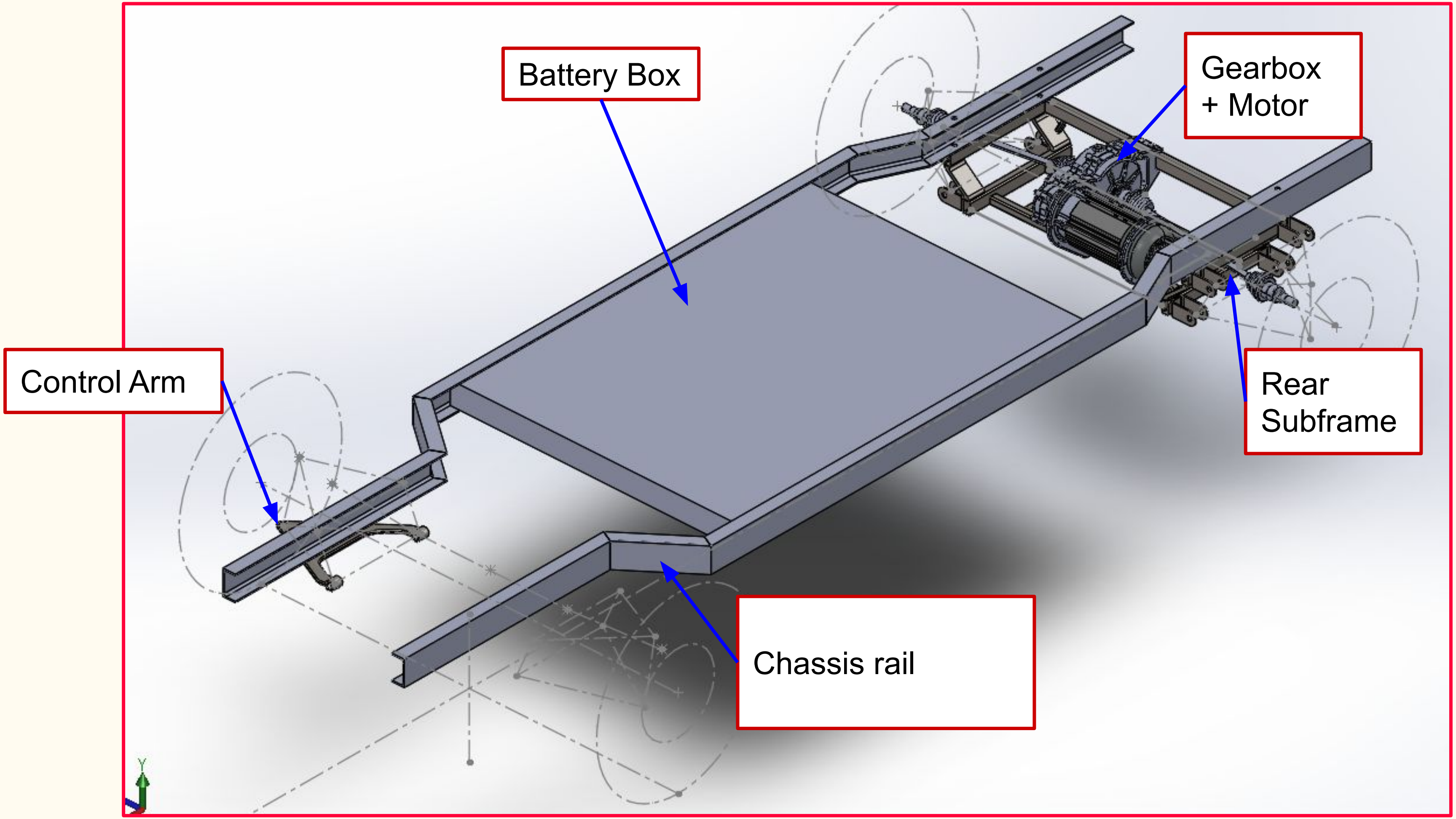
If control arm was attached to the rear subframe, this is how it would look.We can see that the control arm matches perfectly with the rear subframe mounting clamp ends.


Conclusion

CONCLUSION:

- From the early made components, proper matings were given provided that there is no intersection between components surface
- Extra square tubes were inserted as our model has significantly less weight (nearly 20 kg) than the max limit, so it is feasible to include these tubes for extra support
- Bolts were attached to complete the design.Sizing of the bolts used were 10 mm.
- Extra supports were given to the engine and gearbox
- The rear axle got enough space to move up and down freely
- Thus the final assembly of all the components was done

FINAL FULL VIEW OF THE ASSEMBLY



MONTH 2

Group task assigned with another group member and mentor

Task Flow:

- Calculation of forces under new loading scenarios
- Material selection
- Design work of front subframe
- Design work of Control Arm
- Ansys analysis of control arm
- Ansys analysis of subframe
- Inclusion of damper in the design
- Finalized Stress and Total Deformation values

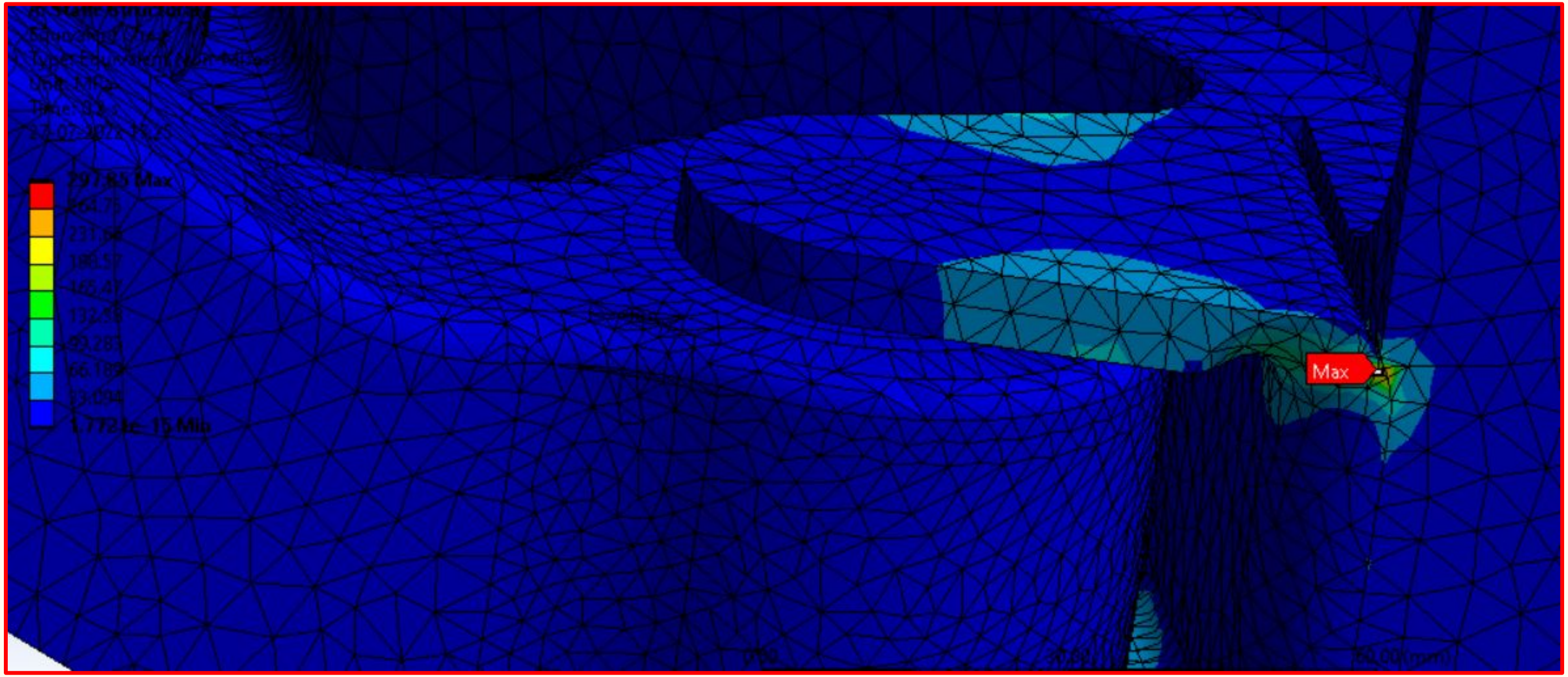
MATERIAL ALLOTTED: ST52 FOR SUBFRAME
Cost:Rs 80 per kg
Cheap and durable

	St52-3
Young's Modulus (GPa)	210
Poisson's Ratio	0.3
Tensile Yield Strength (MPa)	355
Tensile Ultimate Strength (MPa)	520

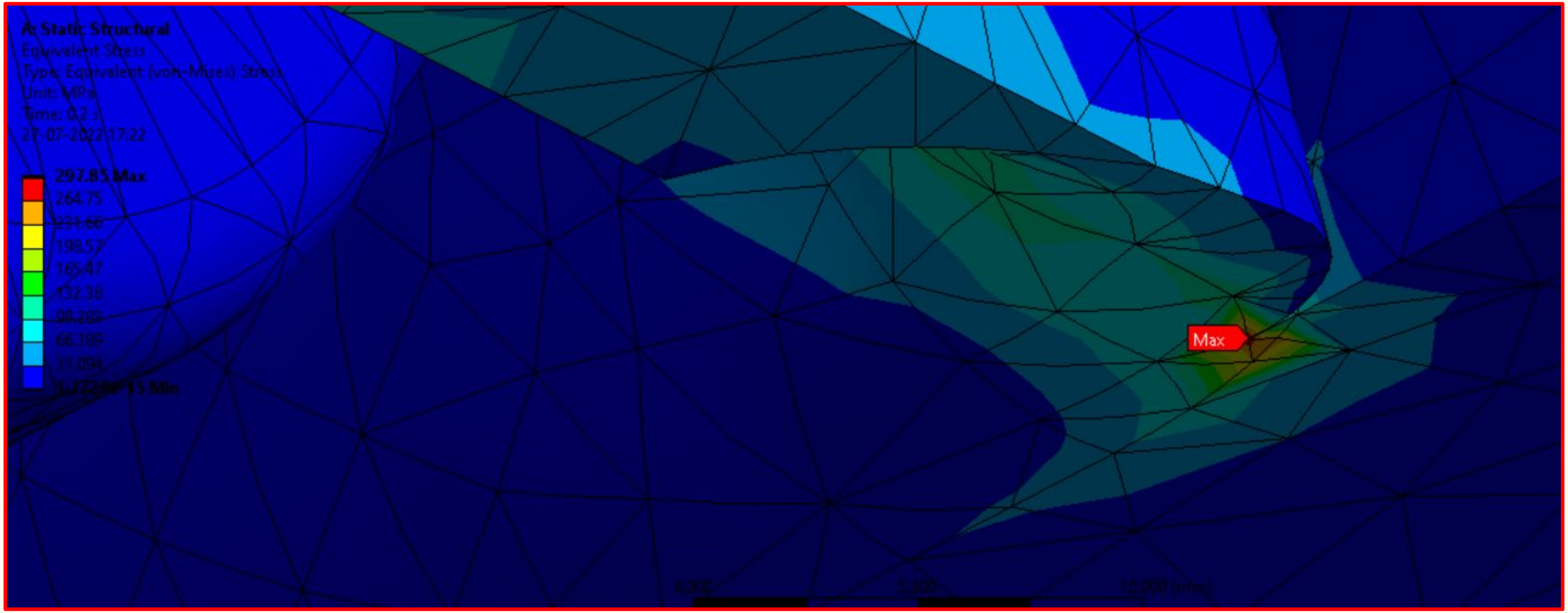
MATERIAL ALLOTTED: Aluminium 6061 - T6 for Control Arm
Cost: Rs 260 per kg

Material	Density (Kg/m^3)	Young's Modulus (N/mm^2)	Poisson Ratio	Yield Strength (N/mm^2)
Aluminium 6061-T6	2731	69000	0.3	294

Stress Value at the connection zone tends to be maximum

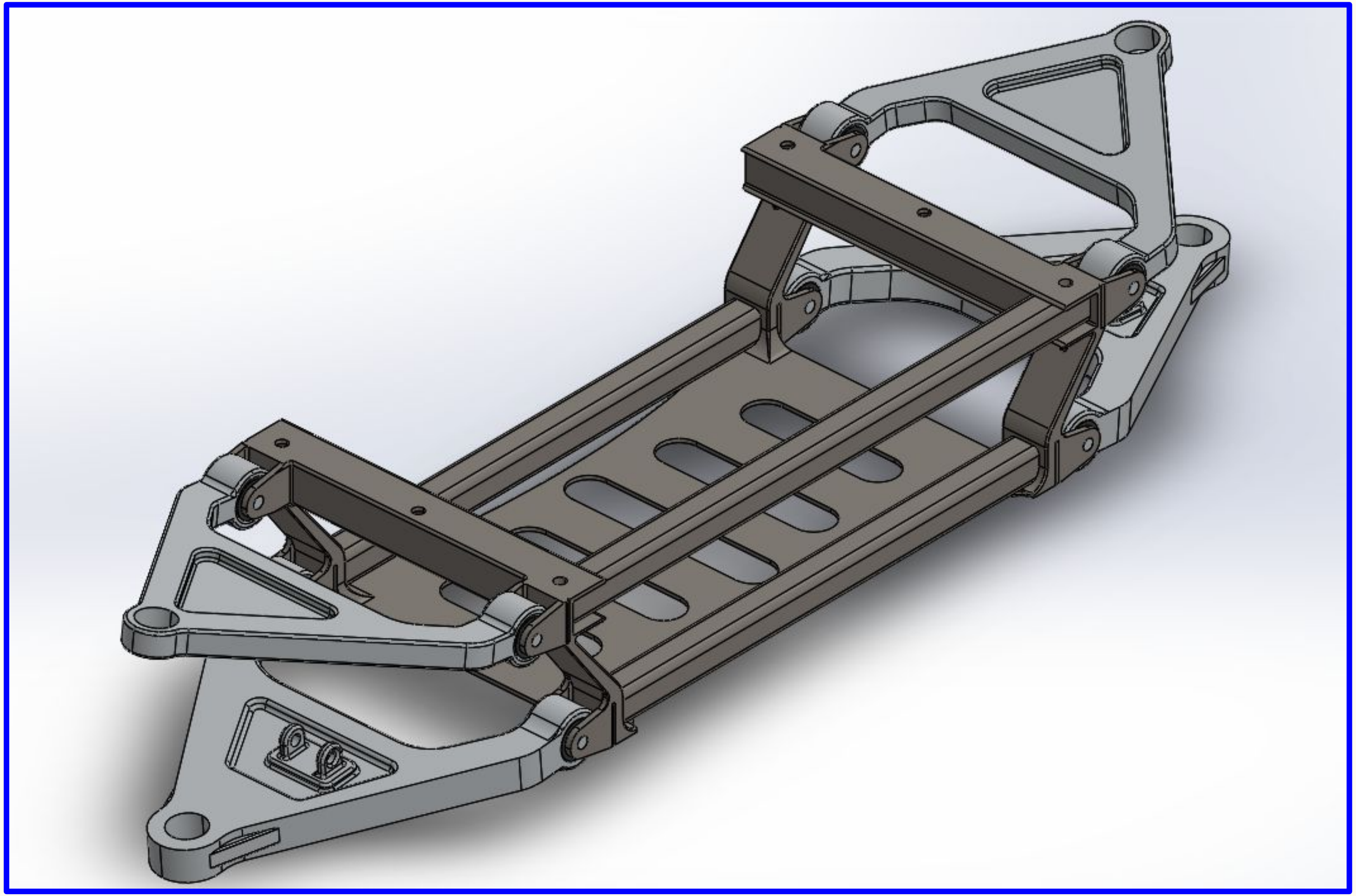


Stress Value at the connection zone tends to be maximum

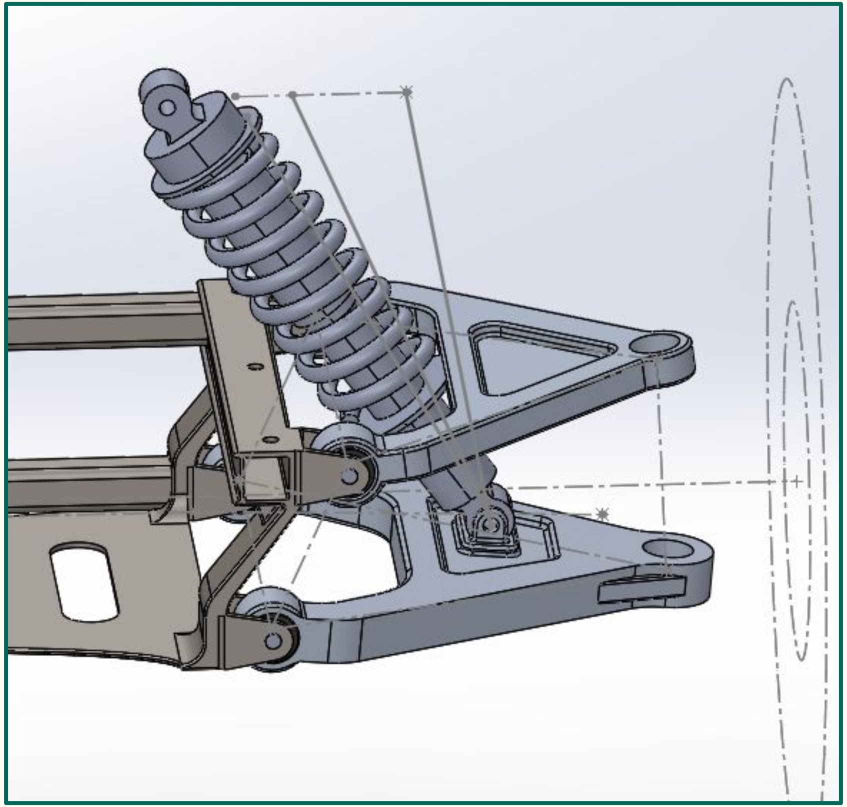


Maximum Total Deformation occurred: at the point of application of knuckle joint

Finalized Design Model



DAMPER INTEGRATED WITH CONTROL ARM SUSPENSION




THANK YOU