Go Kart Optimization

ME 4041: Computer Graphics & Computer Aided-Design Project

Vladislav Pankadzh

Tyler McDaniel

Zach Flegle

Zach Greenfield

Spring 2018

04/25/2018

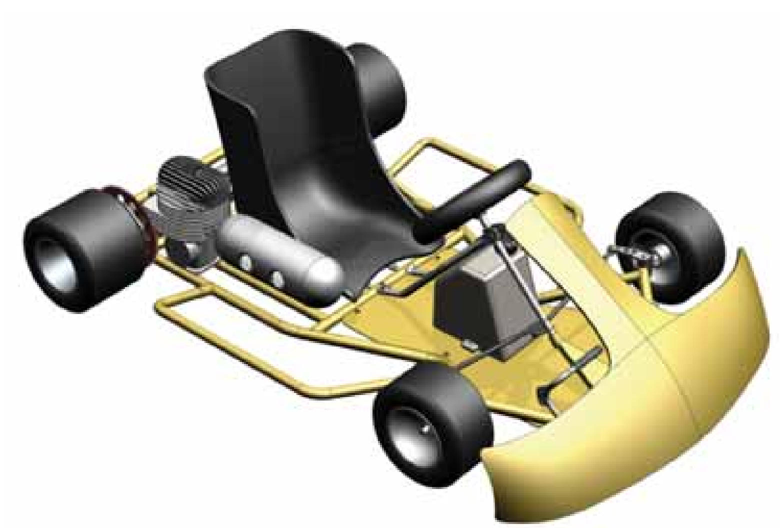


Table of Content:

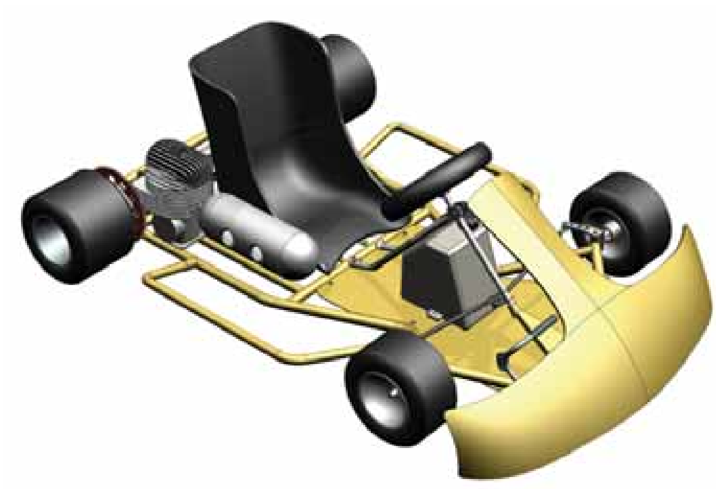
Introduction:

Go Kart vehicles are used for a reactional and competitive racing. Currently to compete in a professional Go Kart racing event, consumers must pay premium price which often is due to the brand name and not reflected in performance. The primary objective of this design project is to redesign an existing, cheaper Go Kart vehicle to improve the performance and functionality and to ensure it can withstand the higher mechanical, thermal and aerodynamics forces required for the professional racing. Target market is an Automotive enthusiasts and automotive clubs in schools and colleges interested in a high-performance vehicle for recreational and educational use.

Objectives:

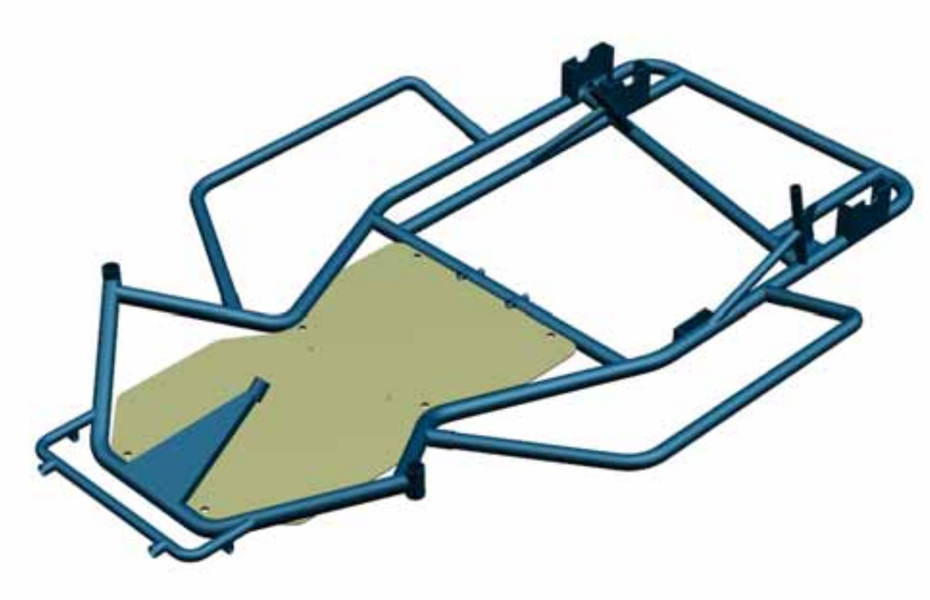
The goal of redesign of the Go Kart focuses on improving the performance and functionality of existing designs. The goal will be to reduce weight and improve strength by using stronger lighter materials while special attention will be taken to ensure that all parts can withstand the additional forces associated with the increased power and reduced weight. Additionally, a focus of this project will be to improve the design to provide quick and simple service and maintenance.

Final Assembly/Model:



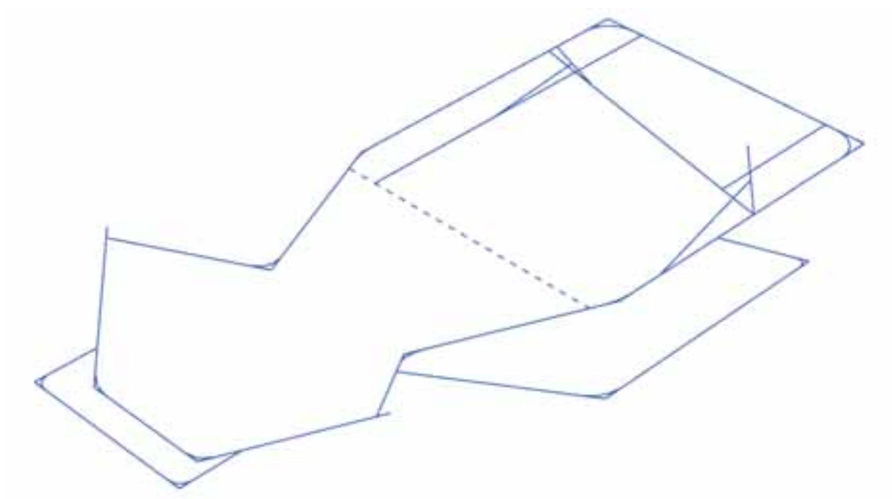
Modeling:

Chassis Subassembly:

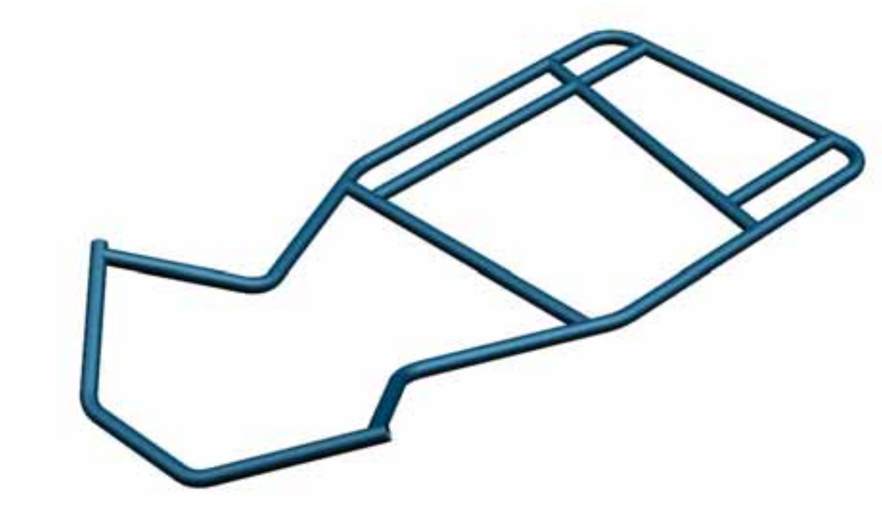


Chassis

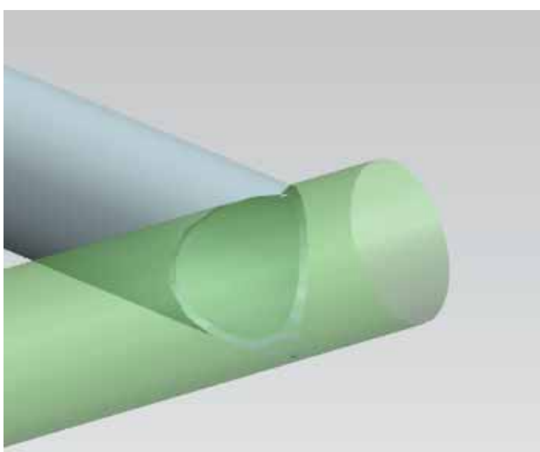
The chassis interfaces with all the subassemblies for the vehicle. The chassis was first started by creating a wire frame of the chassis. The wire frame is created by setting the end points of the chassis tubing, the points are connected by lines. Finally, arcs are places at the intersection of 2 lines where a bend would occur. the wire frame model is shown below.



Once the wireframe model is completed, the chassis tubes are created by using the tube command. The main chassis has a tubing size of 1.125” OD and 1” ID. In order to create the tubing, the wire frame lines that have acres on at least one end are trimmed using the trim curve command to create a line segment the starts/ends at the end of the acres. Once the line segments have been trimmed, the tube command can be used to create the main chassis, shown below.

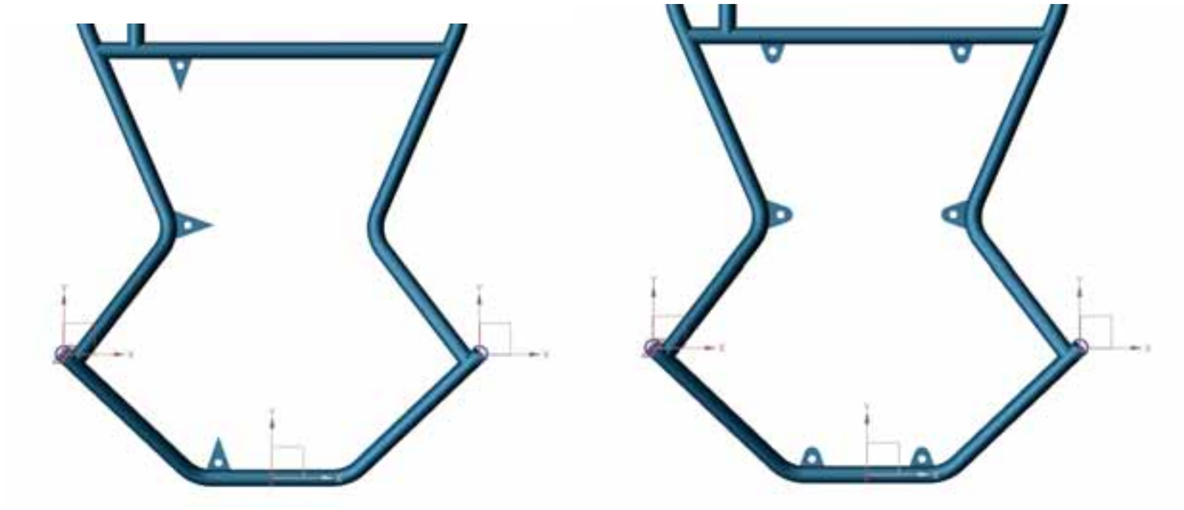


When using the tubing command, the tubes will intersect at the common end points. To resolve this issue, the trim body command is used to trim the end of the tube so that the ends with conform to the face of the other tube. Once the tubes are trimmed, the unite command is used to unite the separate bodies into one. An example of the trimmed end of the tube is shown below.

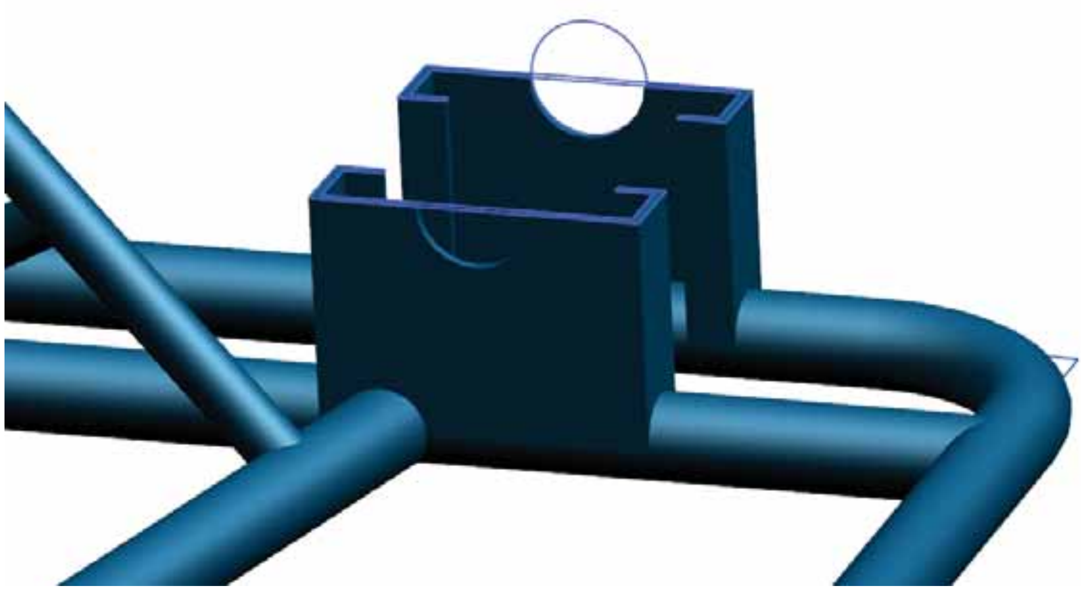


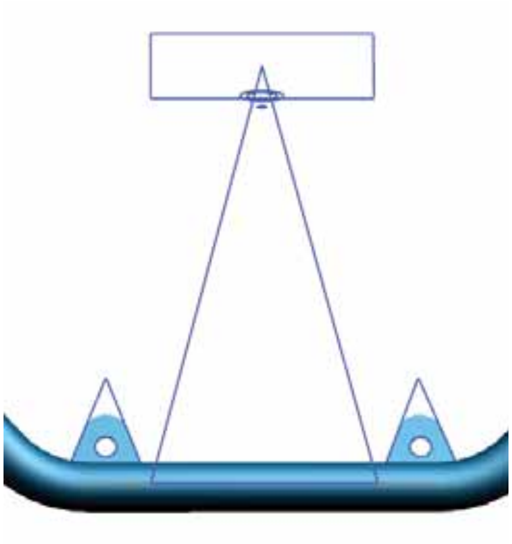
The next step is to create the front wheel bracket mounts. This is done by creating a datum coordinate system at the end of the tube. A sketch is preformed on the XY plane of the datum coordinate system. The sketch is extruded to create the front wheel bracket mount. This process is repeated for the other front bracket mount.

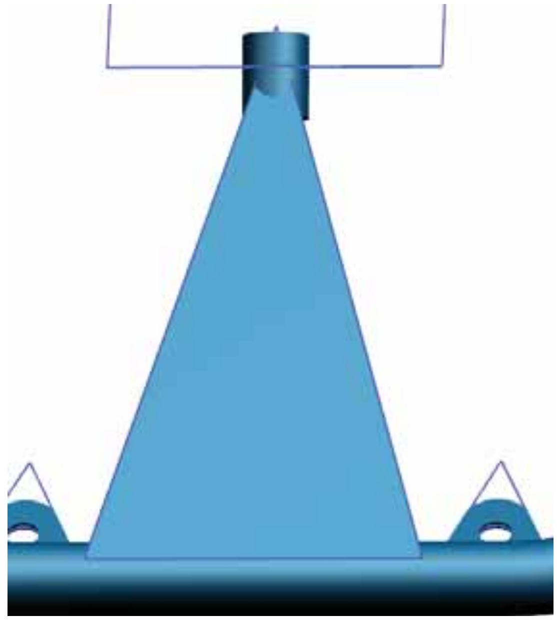
The floor panel mounts are created by sketching out the tabs that stick out of the chassis on the world coordinate system XY plane. The left side is created first and is then mirrored. To finish the tabs, the corners are rounded and the trim body and unite command are used to fix them to the main frame.

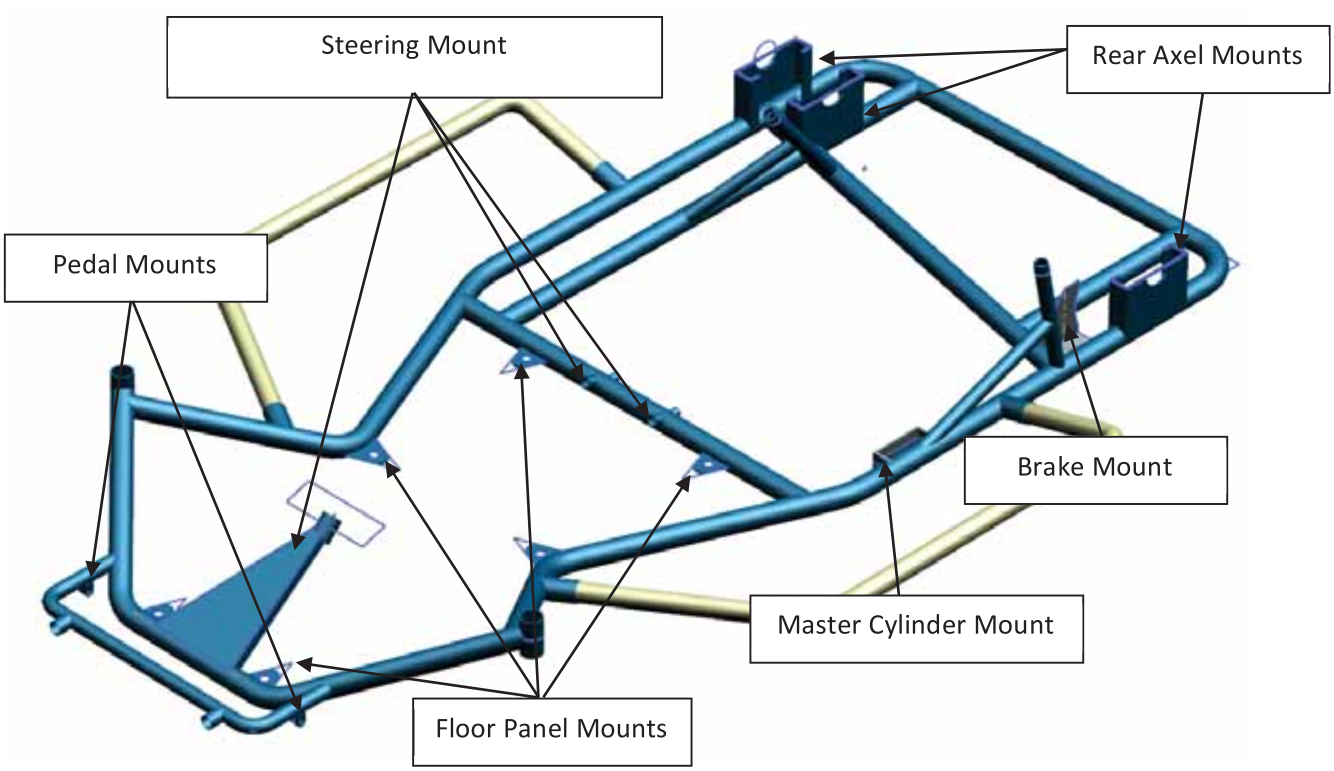


The rear axle mounts are created by first making a datum plane, then sketching the mount profile and extruding the sketch. The extrusion is then trimmed and united with the chassis frame. This process is repeated for all the additional mounts, pedal, brake, steering. The rear axle and steering mounts are shown below.





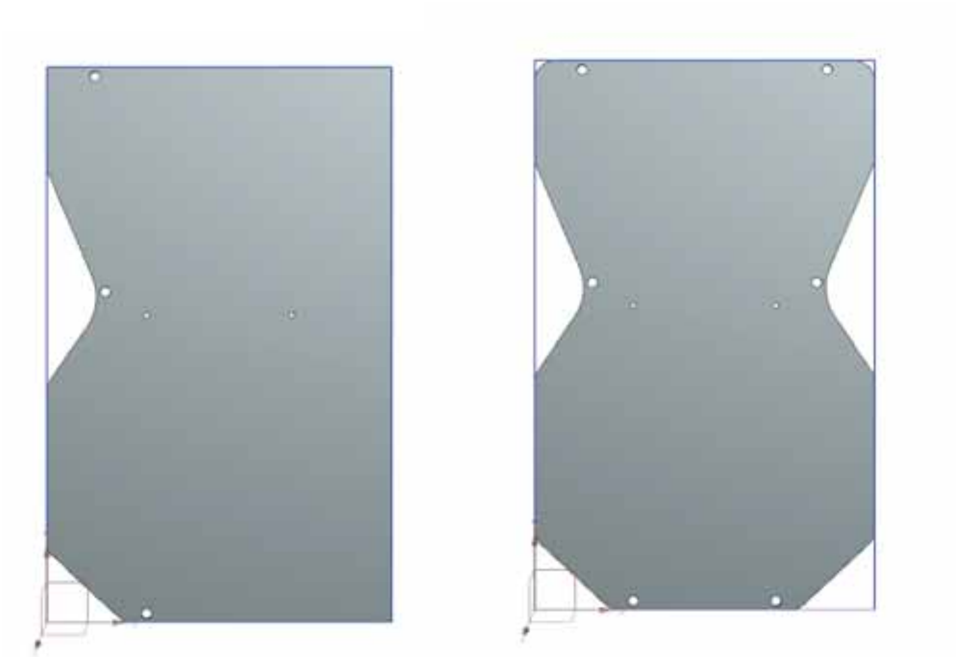




This process is repeated until all the various mounting tabs are created on the chassis. Once all the mounting tabs are created, the Floor panel is inserted into the model and constrained to the mounting tabs on the chassis.

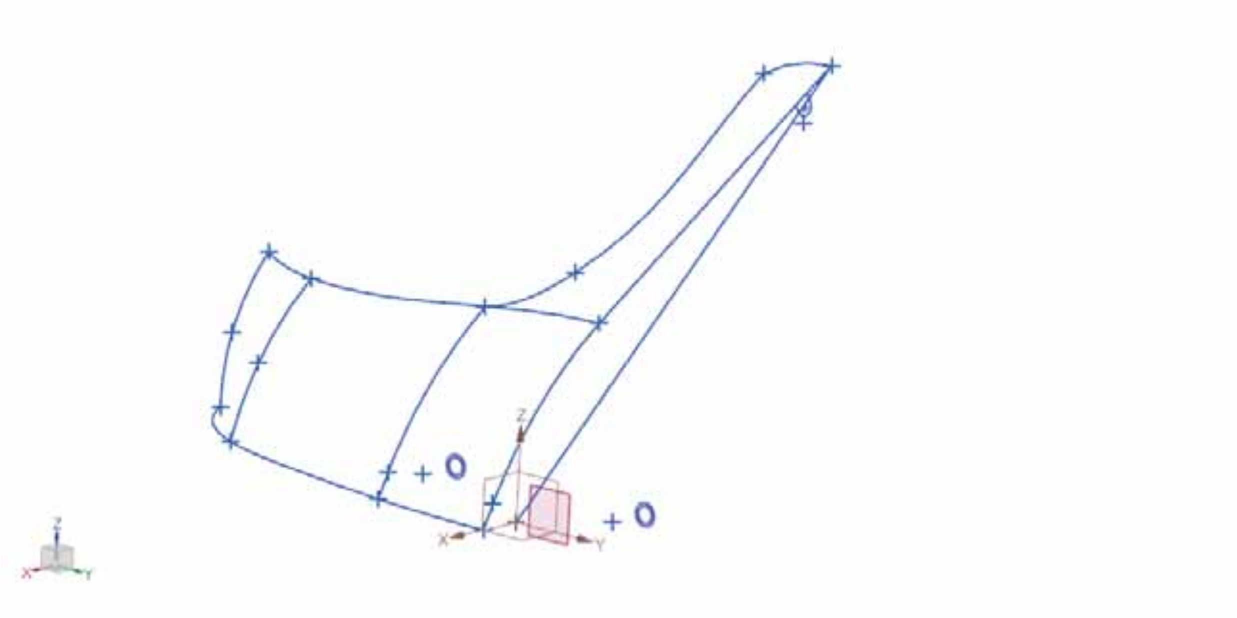
Floor panel

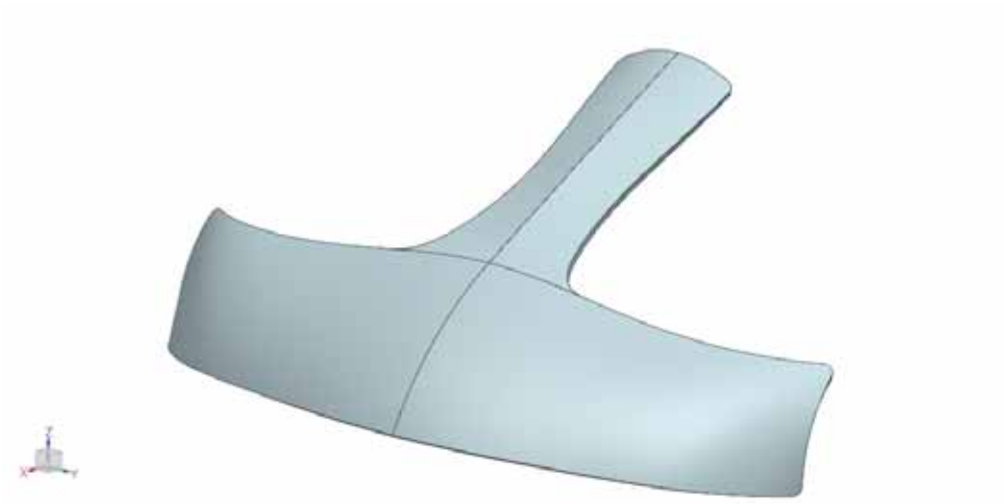
The floor panel is created by a simple sketch and extrusion. Once the main extrusion is completed, the shape of the cutouts are sketched and subtracted from the body including the bolt holes. This is done for one side and is mirror on the other side.



Body panel

The body panel is used to reduce the drag and protect the driver. It is created using multiple curves to define the outline and surface of the body panel. The surface was created by using through curve mesh and mirroring the left surface to create both sides of the body panel. Thicken is used to create a solid body and additional mounts are extruded.

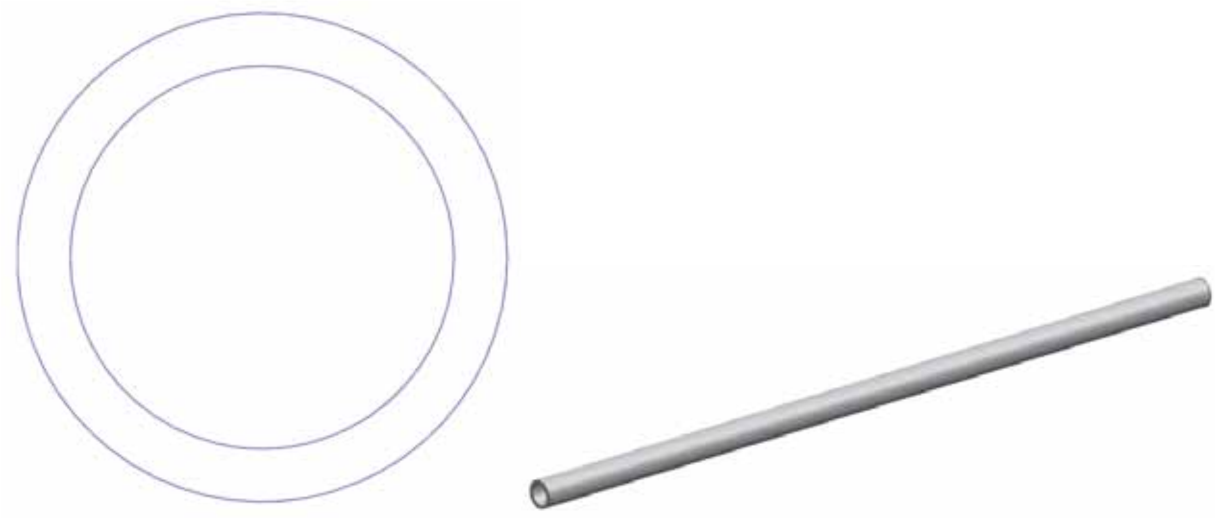




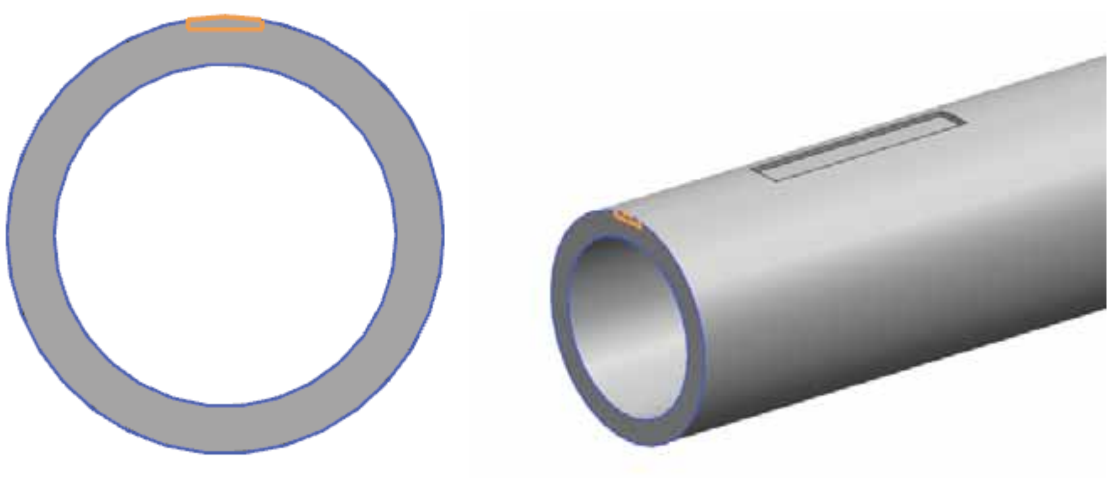
Rear Axle Subassembly:

Rear Axle

The rear axle was a relatively simple part. First a sketch was created with one circle of 35 mm diameter. A second circle was made by offsetting the first one 3.8 mm inward to meet the minimum international specifications. The sketch was then extruded 41 inches.

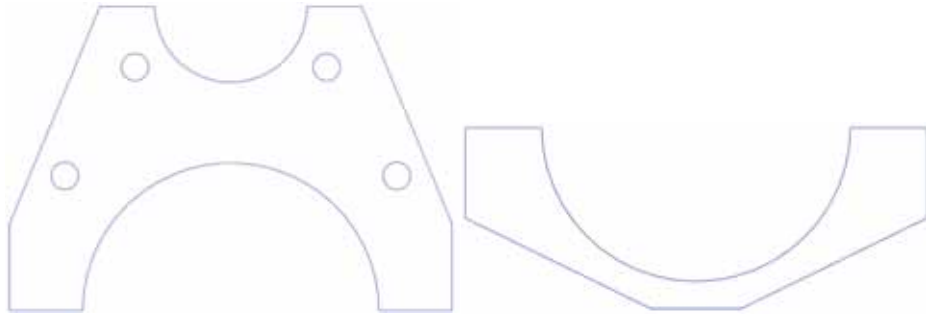


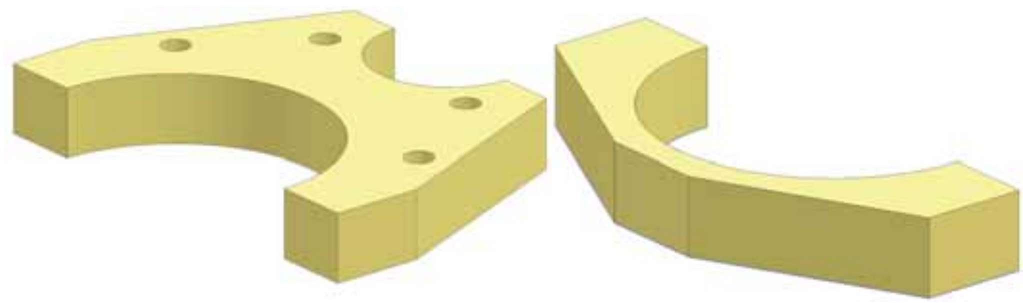
To make the cutouts for the keys to attach the wheel mounts, a new sketch was made on one end of a slot 1 mm deep and was extruded at the proper distance from the end of the axle then mirrored to the other side.



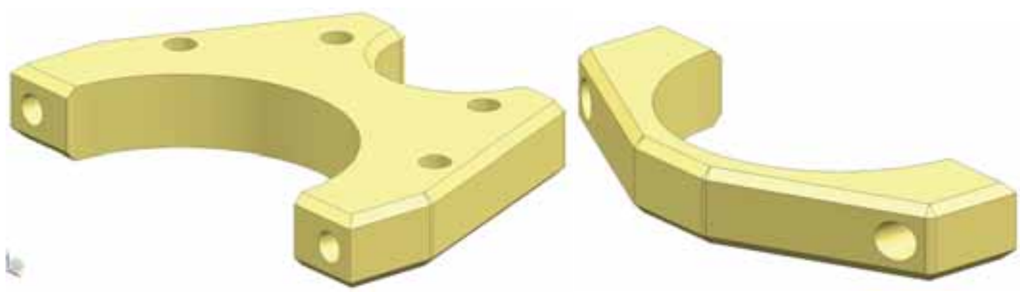
Axle Mounts

The axle mounts are made of three main parts. The first two parts, Axle Mount A and Axle Mount B, were made by sketching the proper geometry then extruding by 15 mm.

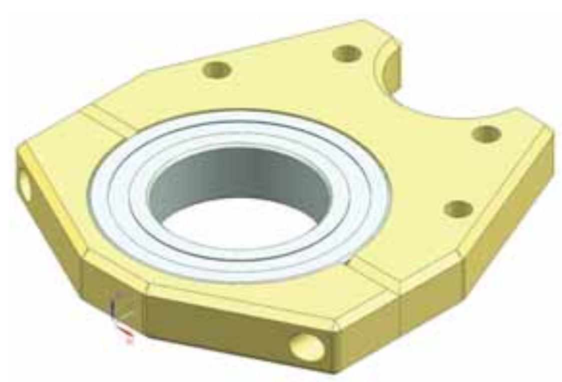




The outside edges were then chamfered and the bolt holes to connect the two pieces were added.



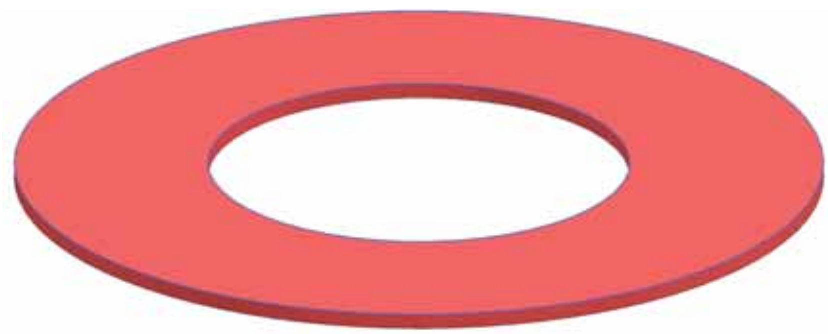
Finally, the pieces were assembled by aligning the bolt holes and the bearing from McMaster-Carr was inserted.



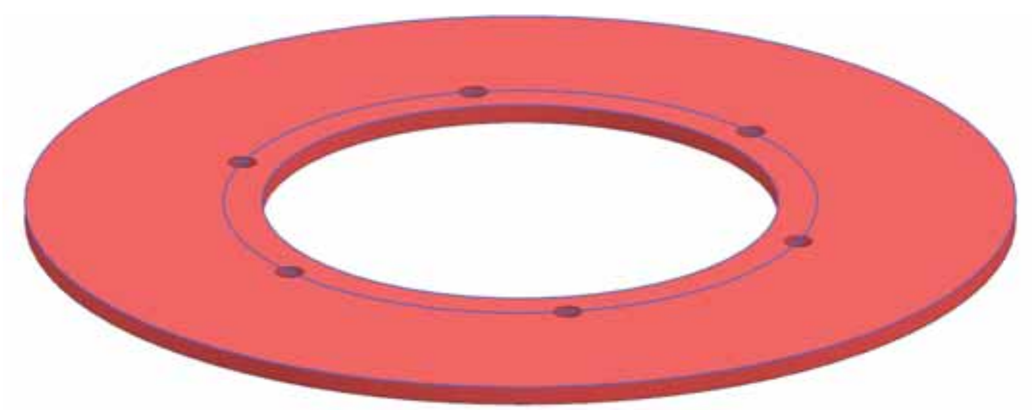
The axle mounts attach to the axle with a small piece of pipe that has an inner diameter equal to the outer diameter of the axle and an outer diameter equal to the inner diameter of the bearing. It was a simple extrusion like the axle with the addition of a pinhole.

Sprocket Assembly

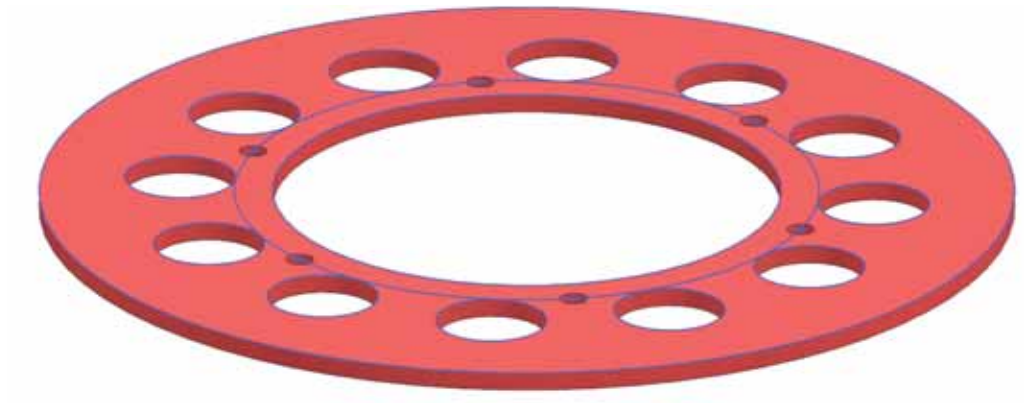
The sprocket assembly consists of two sprocket halves and a mount. The sprocket halves are identical and were made by first sketching and extruding the overall shape.



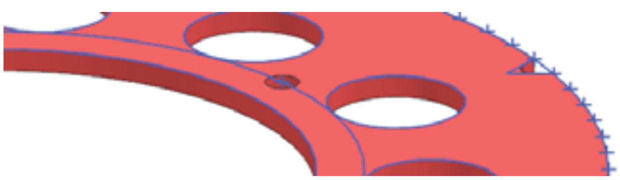
Then the bolt holes were added by sketching a circle with six equally spaced points along it that were extruded using the hole function.



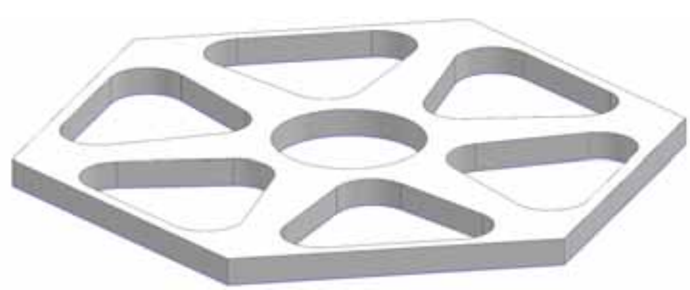
More holes were added to reduce weight.



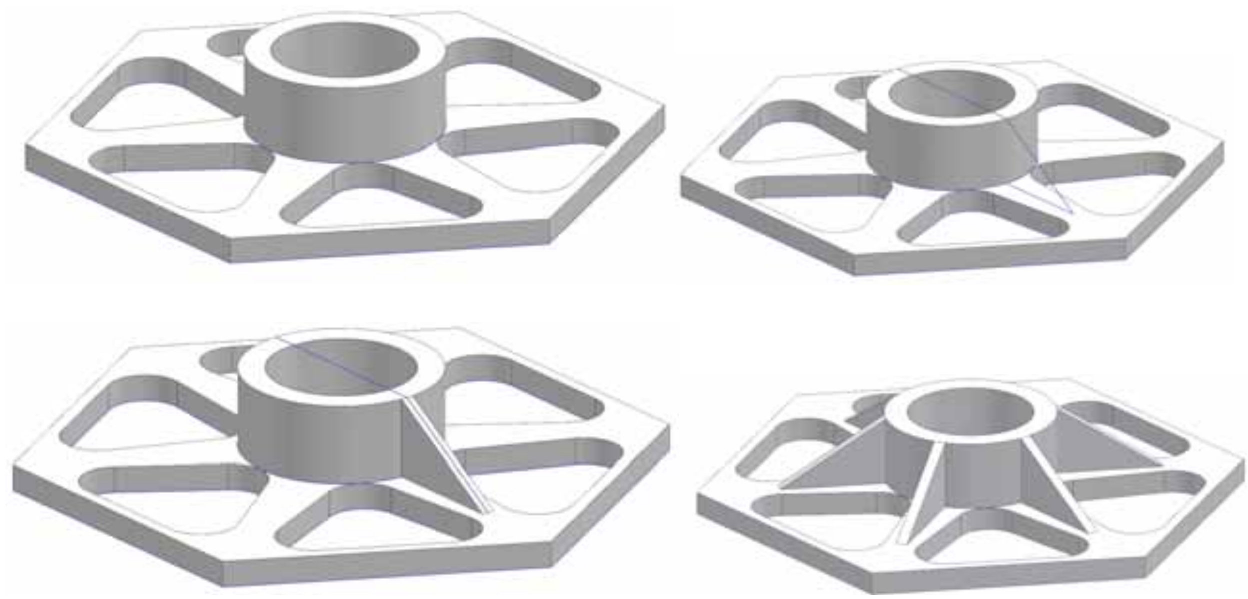
The gear notches were created by removing a triangle and blending the inside edge before patterning the feature 88 times around the outside before finally trimming half of the body.



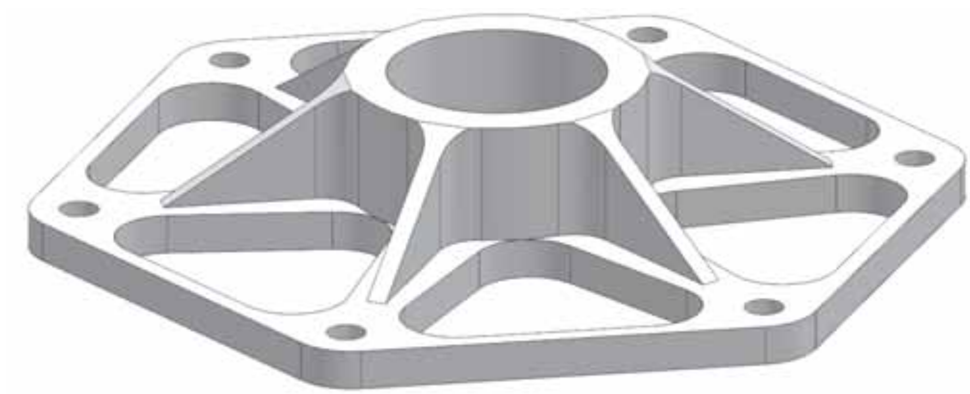
The sprocket mount was made by first sketching and extruding the overall geometry.



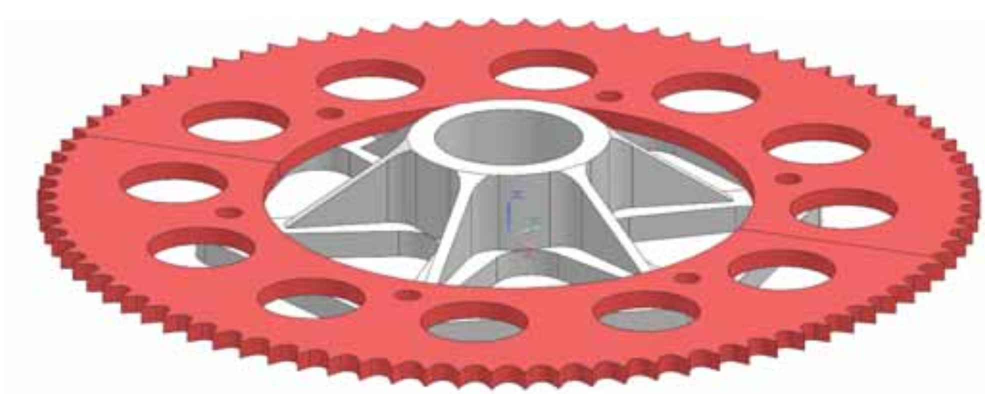
The center tube was sketched to be tangent to the cutouts then extruded before a support fins were added by sketching and extruding one and patterning it to each support arm.



Lastly, all major edges were blended and the bolt holes were added.

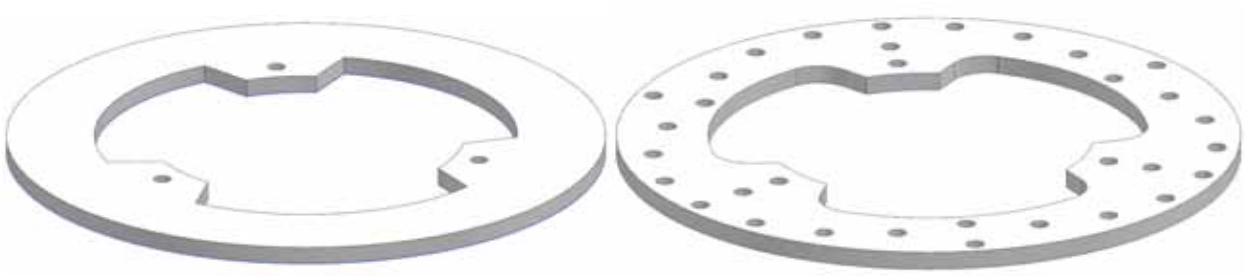


The entire sprocket assembly was made by applying concentric constraints on the bolt holes on the mount and two of the sprocket halves.



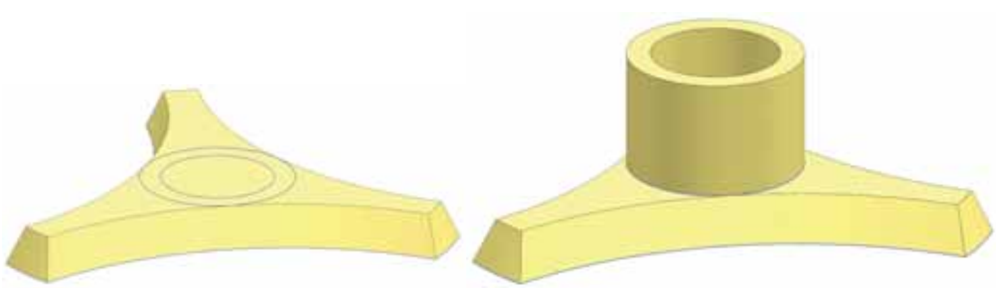
Brake Rotor Assembly

The brake rotor assembly is comprised of the brake rotor and brake rotor mount. The rotor itself was fairly simple. The overall shape was sketched and extruded then the holes were added and the major edges were blended.



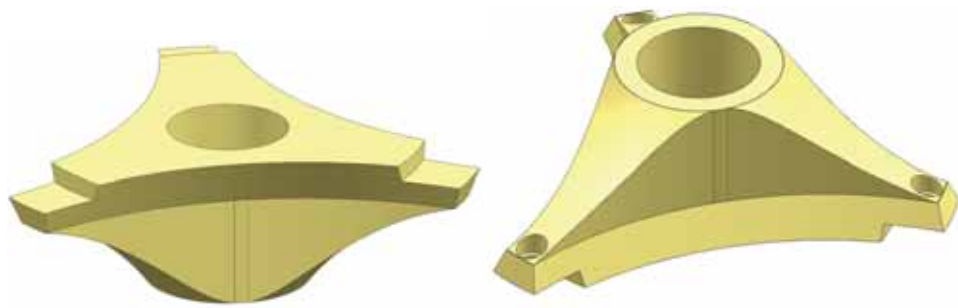
The mount was created by first sketching the overall base geometry, then extruding it with a draft angle.

The central tube was made by sketching and extruding a circle in the middle of the upper face.

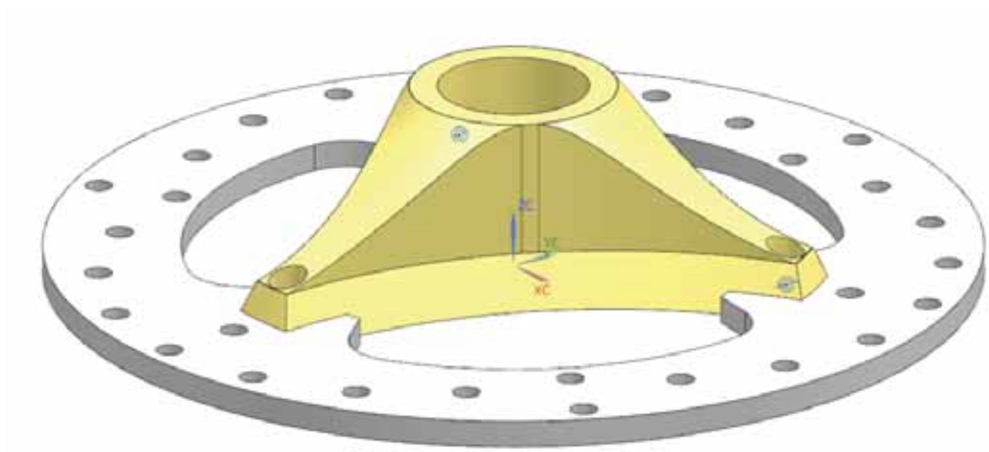


The slopes from the base to the top of the tube were made by revolving a sketch then subtracting the areas between each arm of the base.

The ends of each arm were cut to allow the rotor to be set into the mount, then the bolt holes were made with a counterbore on each arm.

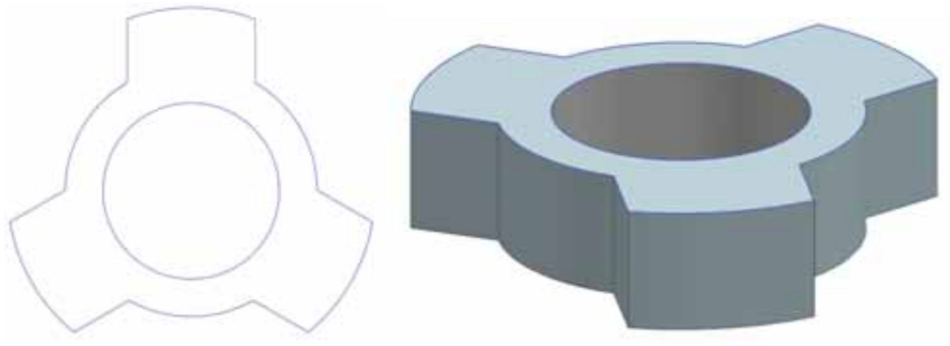


The two parts were assembled by applying concentric constraints to the bolt holes on each part.

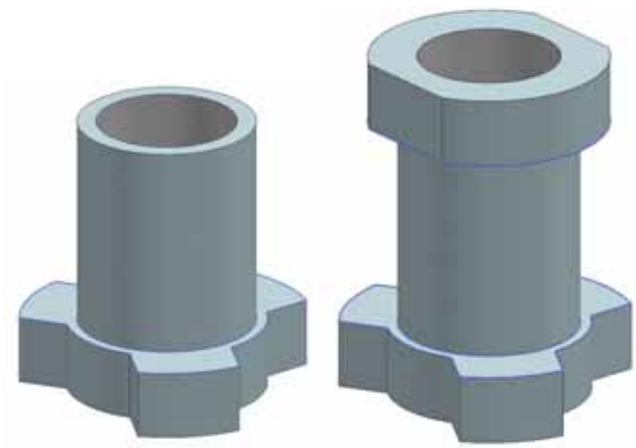


<u>Wheel Mounts</u>

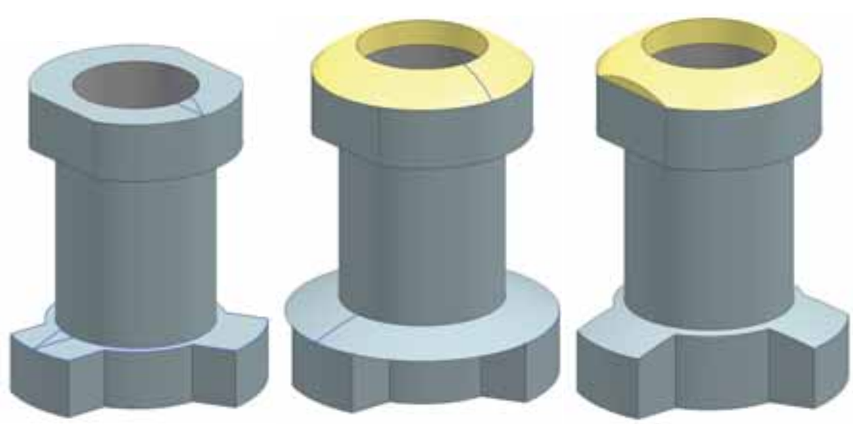
The wheel mounts connect the wheels to the axle. First, the part that connects to the wheel is sketched and extruded.



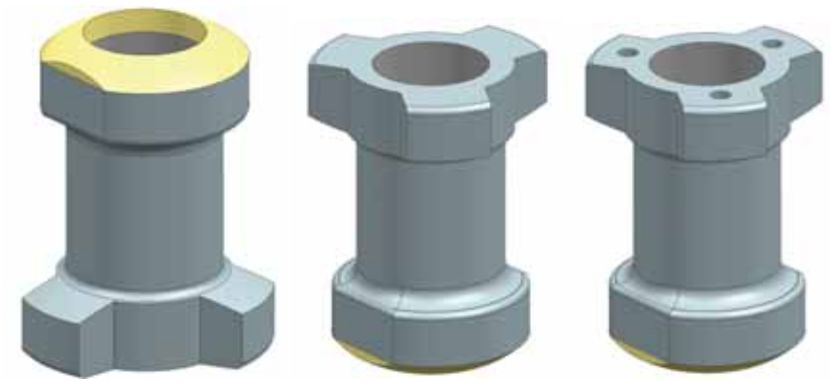
Then the middle and top portions are added on in the same manner.



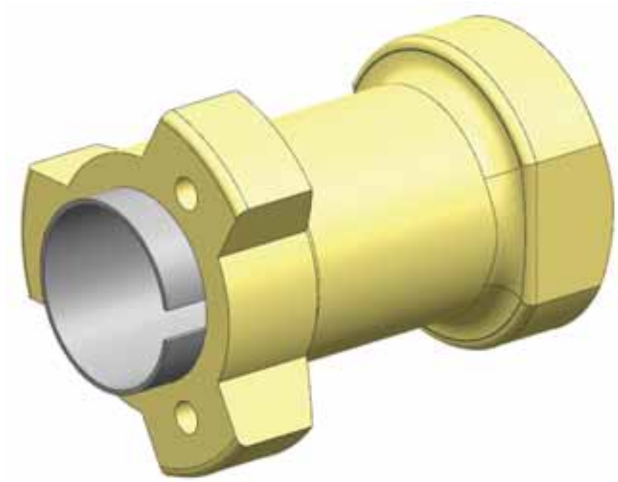
The curved portions were both created by revolving sketches then cutting away portions of the revolved feature.



A series of edge blends were then applied to blend the three portions of the mount together and holes were added to attach the wheel to the mount.

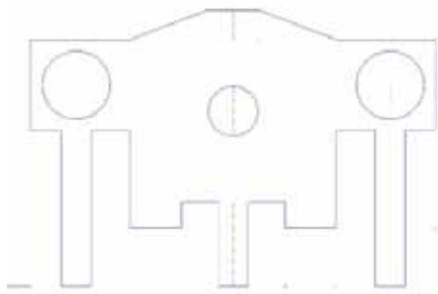
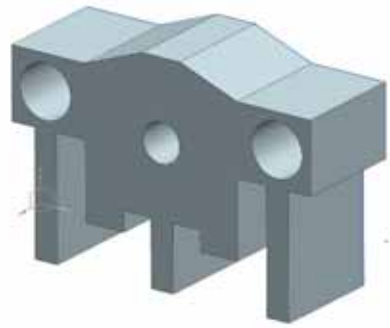


A thin tube portion was added to the side the wheels mount to for a snug fit and a slot was added for the key to connect to the axle and all of the extrusions were united into one solid body.

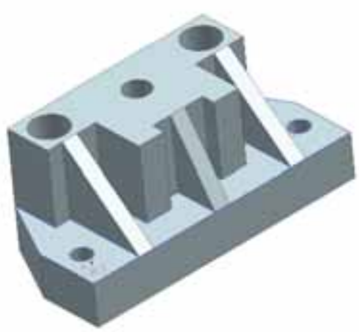
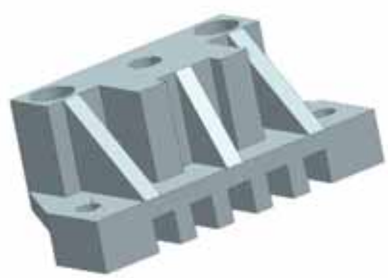
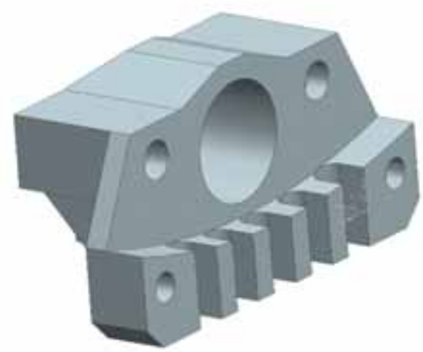


Brakes

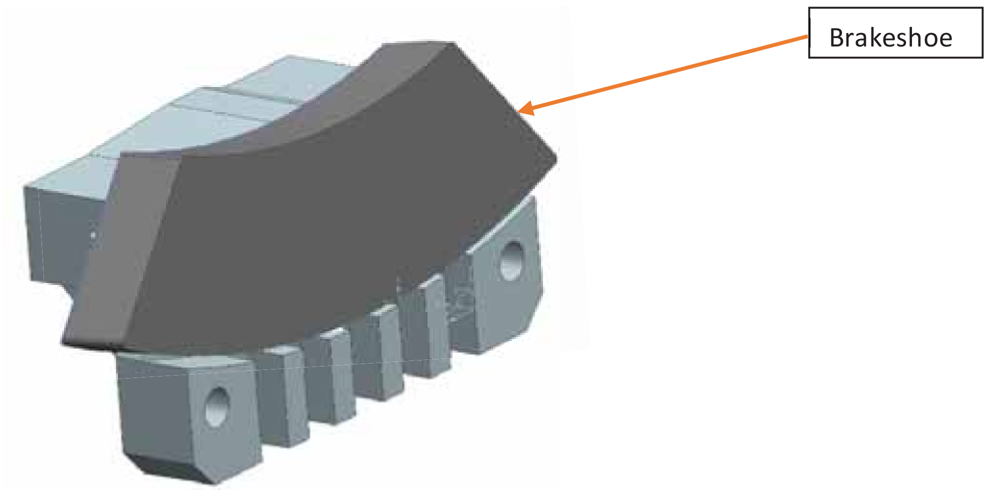
Rear Axle assembly also includes the brakes. Brakes are relatively complicated geometrical part. The initial step for modeling breaks is to create a half of the sketch that can be mirrored onto another half. Sketched is then extruded to a certain height and chamfer command is used to get rid of weight to optimize the brakes.

The following step is to create an extrusion for the flat part of the brake, that holds the brake shoes part. These ridges have a rectangle outer ridges and smooth inner corners.

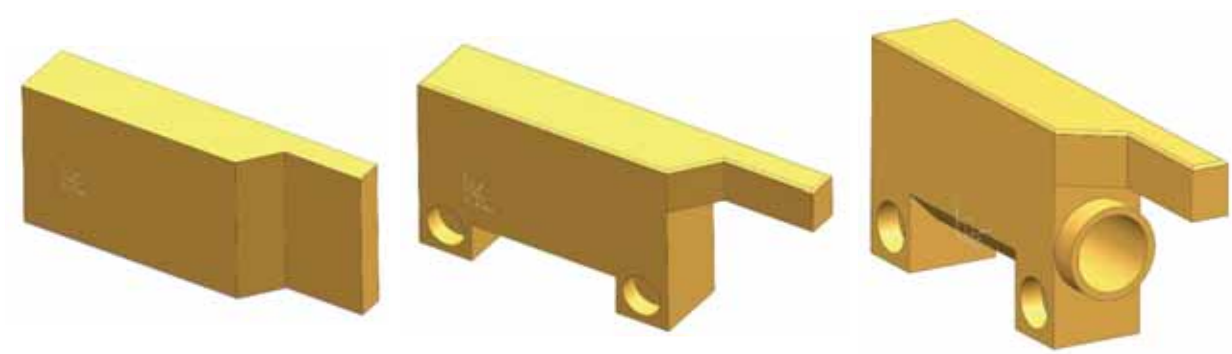
  

The final step is to create a round opening for the brake shoe and the cylinder that pushes brake shoe against the sprockets to create friction. In addition to a cylindrical extrusion on the other side, where brake line is connected to. There are two types of brakes with different cylindrical extrusions on different ridges.

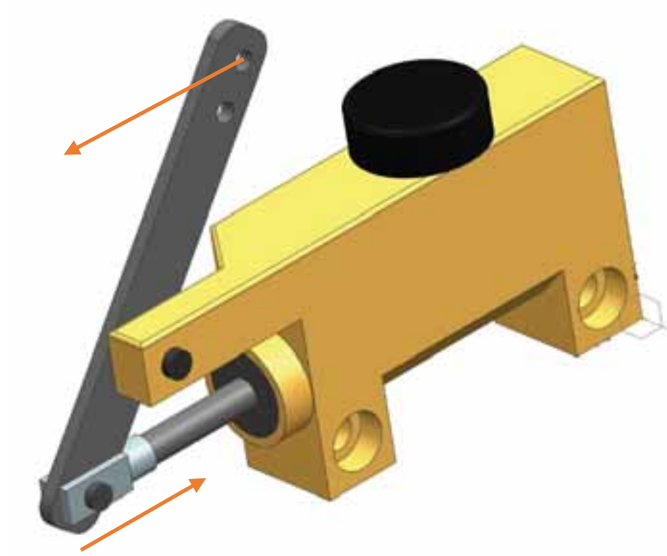


Brake cylinder

For this part an extrusion is created, and then additional features are added in. Cylindrical cut outs are added on the sides so the brake cylinder can be fixed to the frame of the go kart.

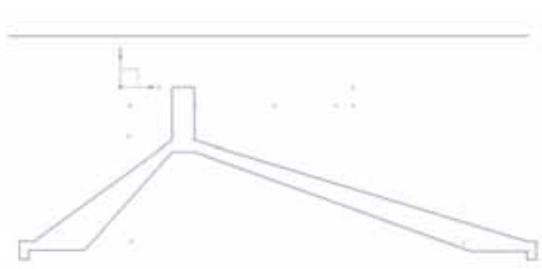


This part is hollow and contains a cylinder inside. Once the driver pushes on the brake pedal, the wire pulls the lever on the side of this part and increases the pressure of the pressurized break fluid that then pushes break shoes against the sprocket to slow car down. This part has a cap on the top to control the level of the brake fluid.



Rear Wheels

Rear wheels are modeled after a real wheel tire used for go karts. Wheels are not symmetric and are modeled with furthest half from the center to be positioned closer to the rear axle.



Rear Tires

Rear tire is modeled after the rear wheel to make sure it would precisely fit. A sketched is create and then extruded via a rotation to create a tire.

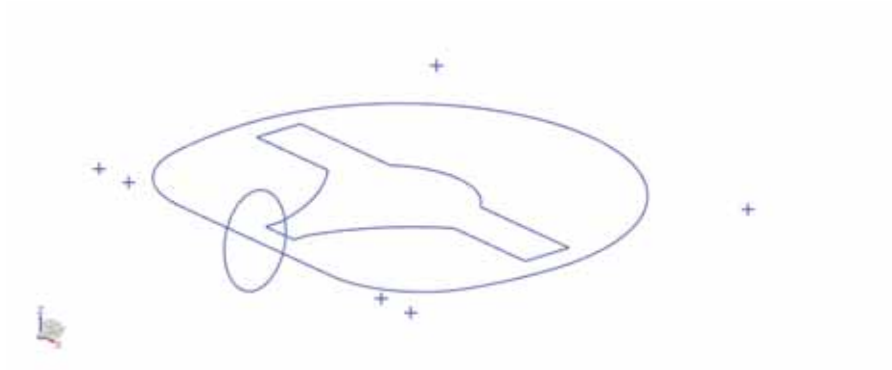
Front Wheels and Steering Assembly:

Front Wheels and Steering Assembly provides stabilization and steering functions. In this assembly it was important to design all parts to be compatible with each since they all interact dynamically. The full assembly can be seen below in figure below. The steering wheel is controlled by the driver and rotates about the

The steering wheel is controlled by the driver and rotates about the steering column. The rotation of the steering column forces the push rods on either side to translate and push the front wheel brackets. This motion rotates the tires and turns the go kart.

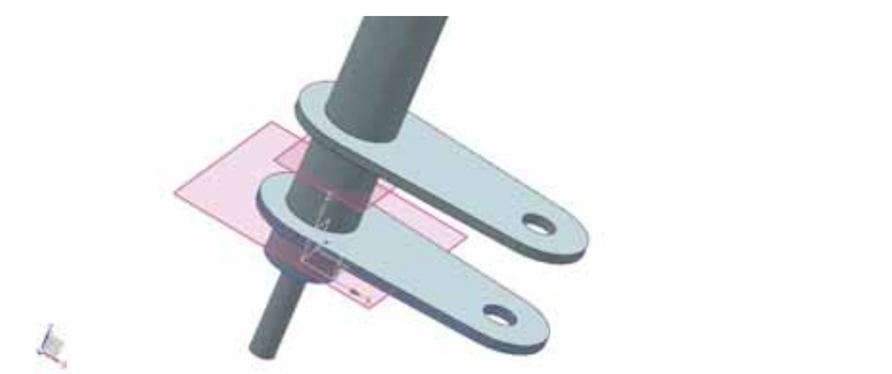
Steering Wheel

The steering wheel is modeled by first creating the outline of the shape. An elliptical profile is then swept through that profile to create the wheel. The Mounting points of the steering wheel are made by sketching a profile and extruding it to three offset datum planes. Finally, the holes needed to secure the steering wheel to the steering column were extruded from the solid body.



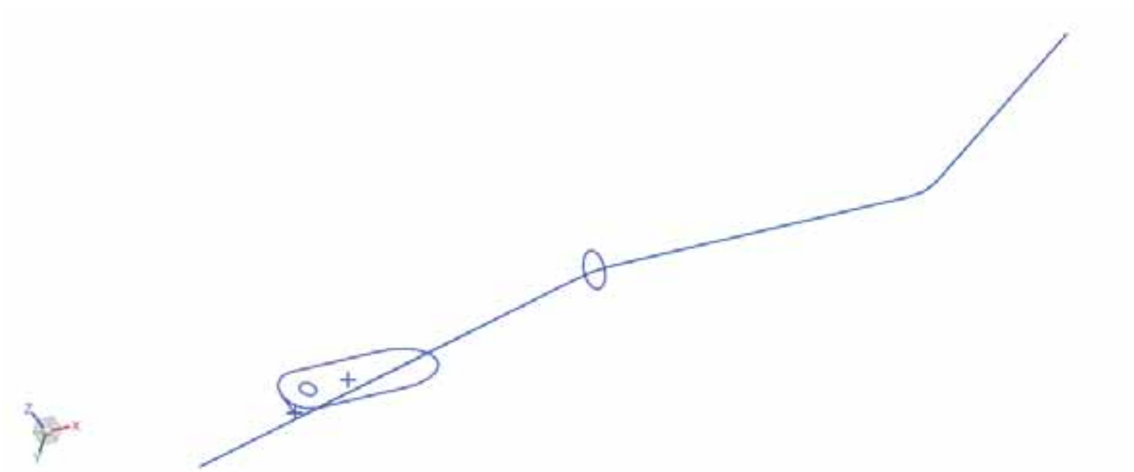
Steering Column

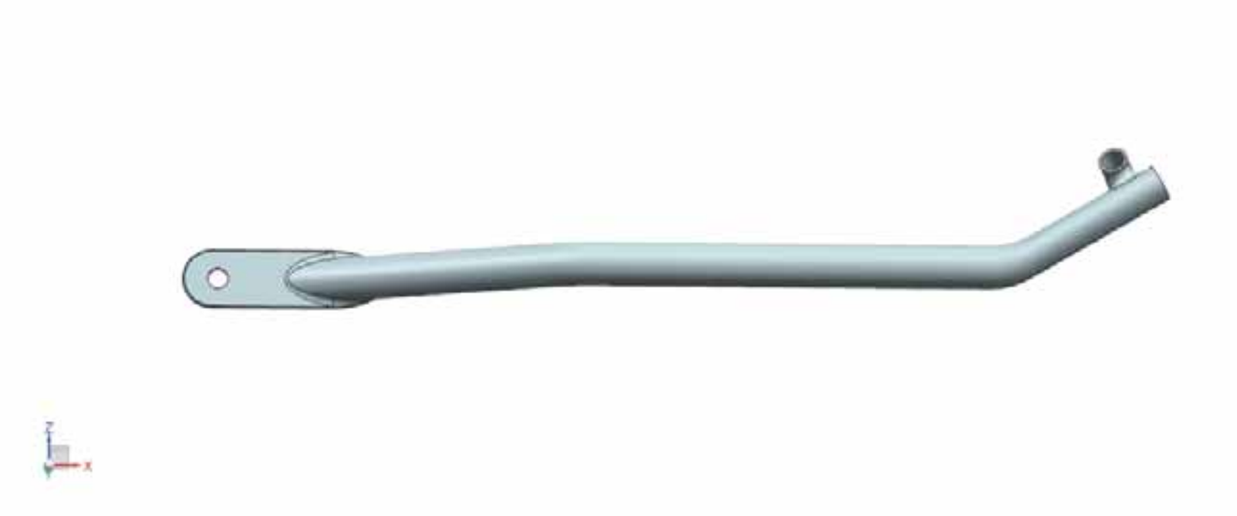
The steering column is created by extruding a circular profile to create a cylinder. The two mounting points for the push rods are made by extruding each offset profile. This is where the push rods connect and translate the rotational motion of the steering wheel into the linear motion of the push rods.



<u>Steering Column Mounts</u>

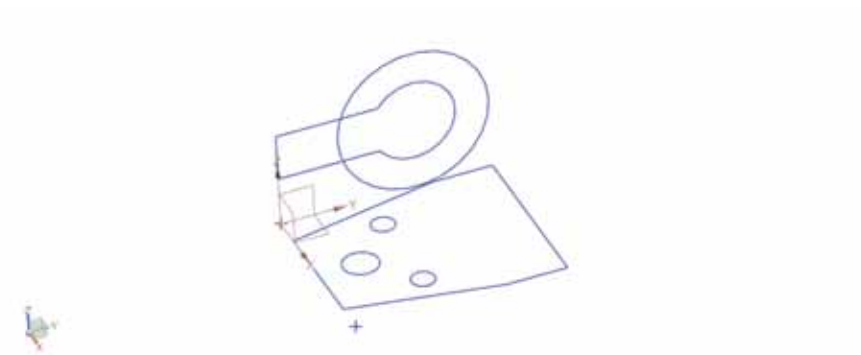
The steering column mounts are modelled by creating a three dimensional curve using control points. A circular profile is then swept through that curve to create the unique shape of the mount. The two mounting points are made by extruding profiles with circular mounting points on datum planes at offset angles. Because there is a left and a right mount, the left mount was mirrored across a plane in order to create the right mount.





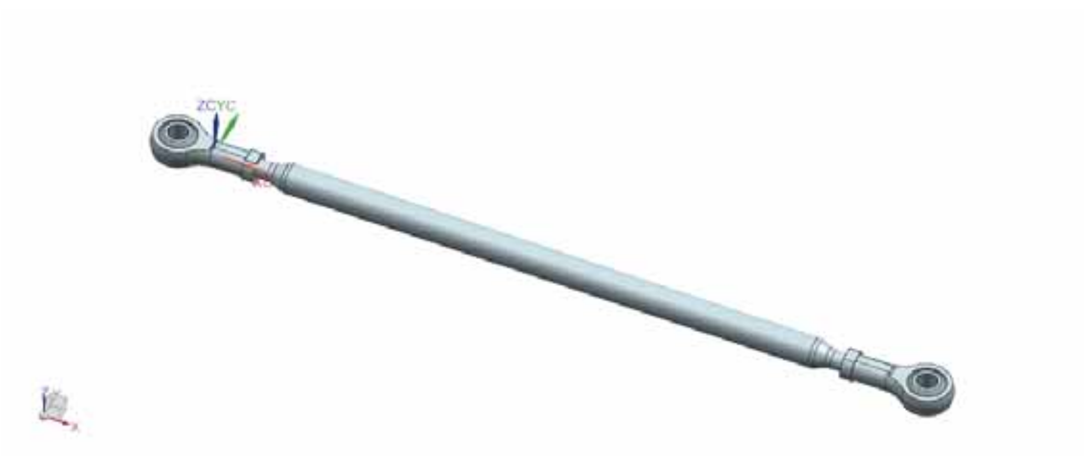
Steering Column Bushing

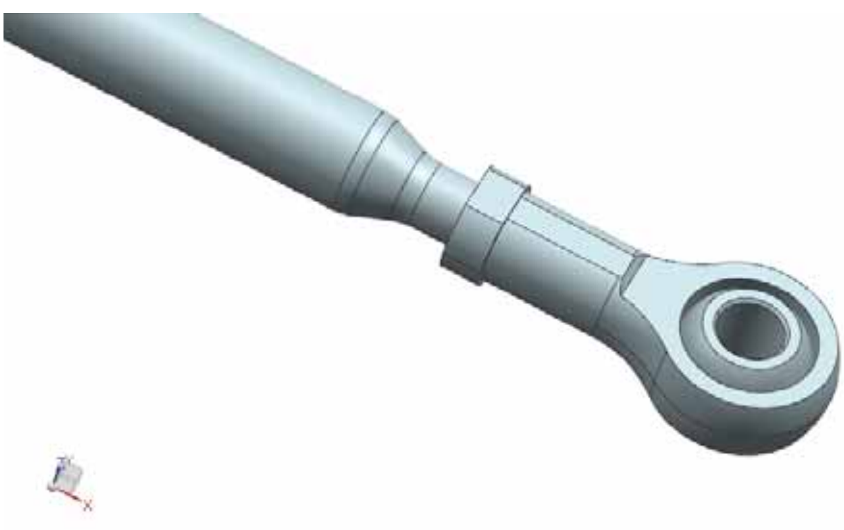
The steering column bushing allows the mounts to connect to the steering column without interfering with the rotational motion. This is modeled by extruding the unique profile to create a solid body. From here the various mounting holes are extruded from the solid body.



Push Rods

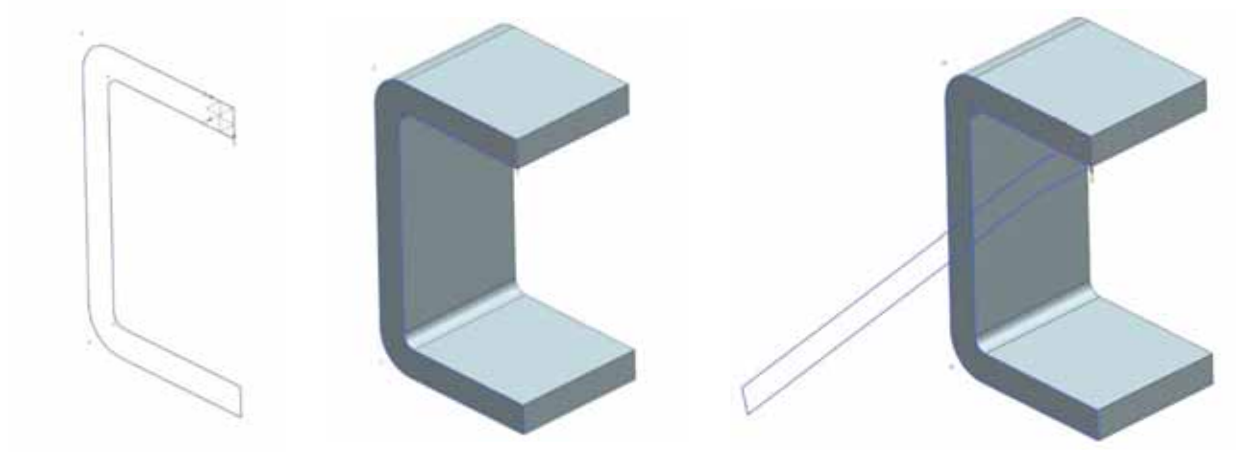
The push rod assembly is made from three unique parts. The central connecting rod was made by extruding a circular profile. The socket joint model was downloaded from McMaster-Carr because it was a purchased part. The Ball that fits into the socket joint was made by revolving a circular profile and then extruding the necessary cuts to add a hole in the center and two flat sides. The overall assembly contains one connecting rod, two sockets, and two balls. Thus allows the push rods to connect the Front wheel bracket to the steering column.



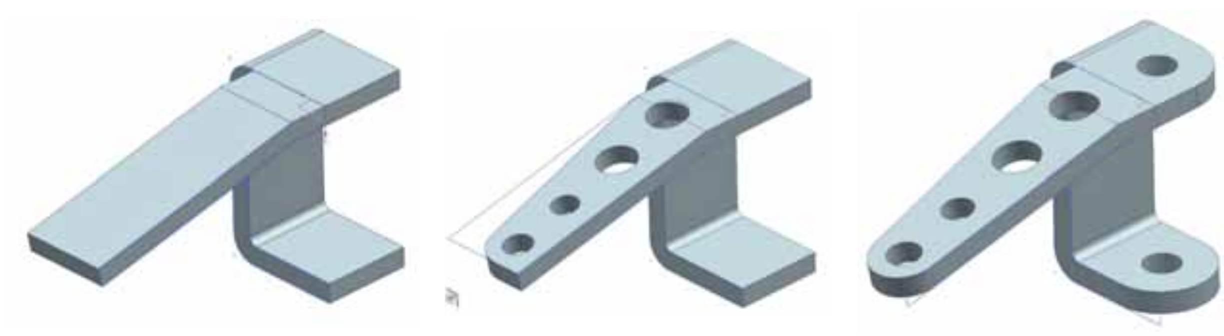


Front Wheel Bracket

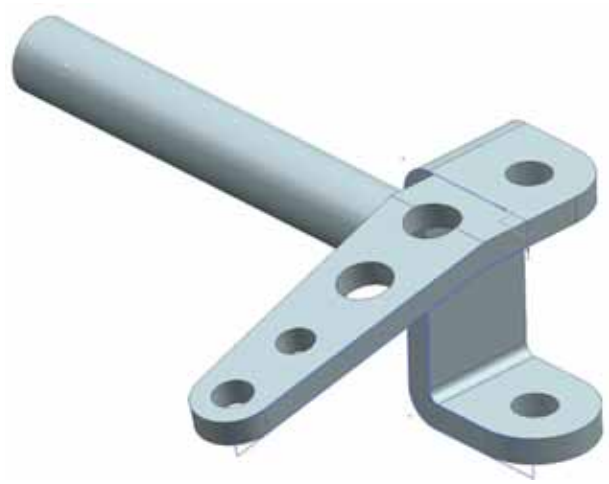
An important part of the front wheel assembly is the wheel bracket that connects the wheel with chassis and the steering columns. As seen below the first step is to sketch one of the cross sections and then extrude it.

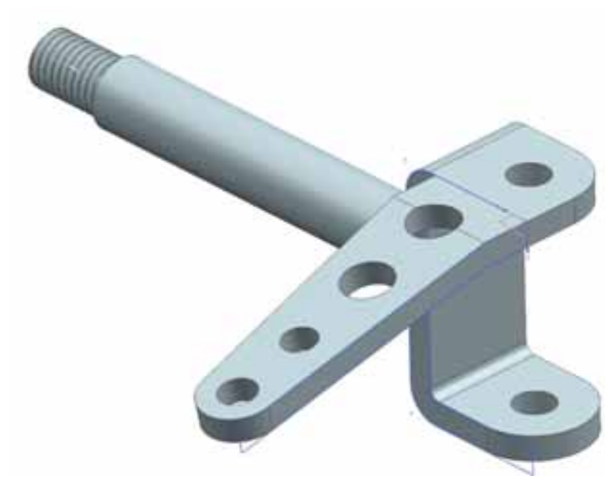


Once the part that attaches to chassis is extruded, the part that connects to the steering columns is sketched and extruded at a slight angle to accommodate for chassis geometry.



The part that connects to the chassis has two openings on each side. The part that connects to the steering column has an opening at the end and has a decreasing cross-sectional area. An addition of other three openings is done due to the design optimization and helps to reduce the weight.

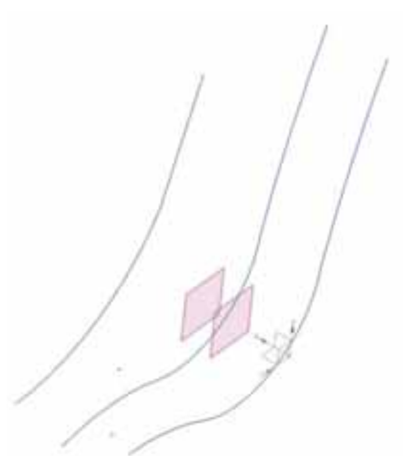


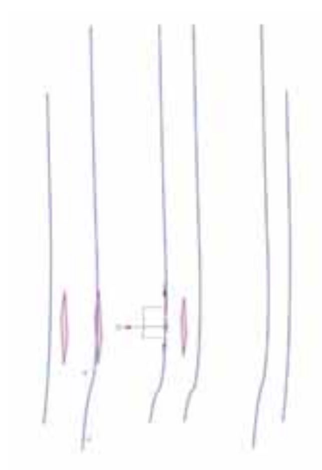


The last part of the wheel bracket is the part that connects to the front wheel. It is extruded from the side of the wheel brackets with an addition of thread part at the end to bold the wheel bearing down. Additional face blends are added to reduce the stress of the forces from the wheel.

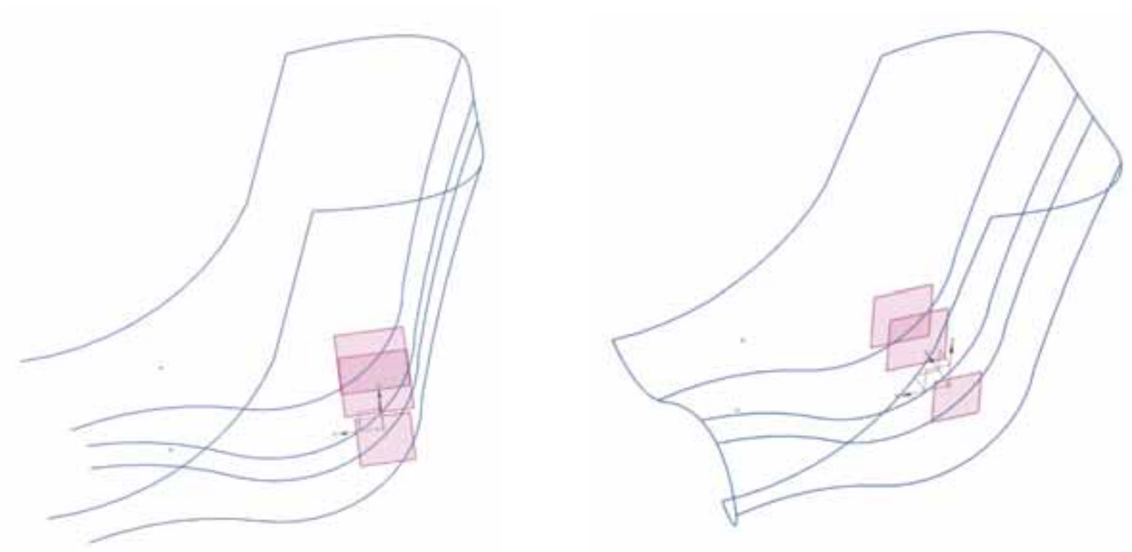
Seat

One of the challenging parts of the go kart design is a driver's seat, as it has multiple curves and must be designed to be as comfortable as possible. Since the sides of the seat are symmetric, the first step is to sketch multiple curve lines at specific datum planes distances and then mirror them to the other side.

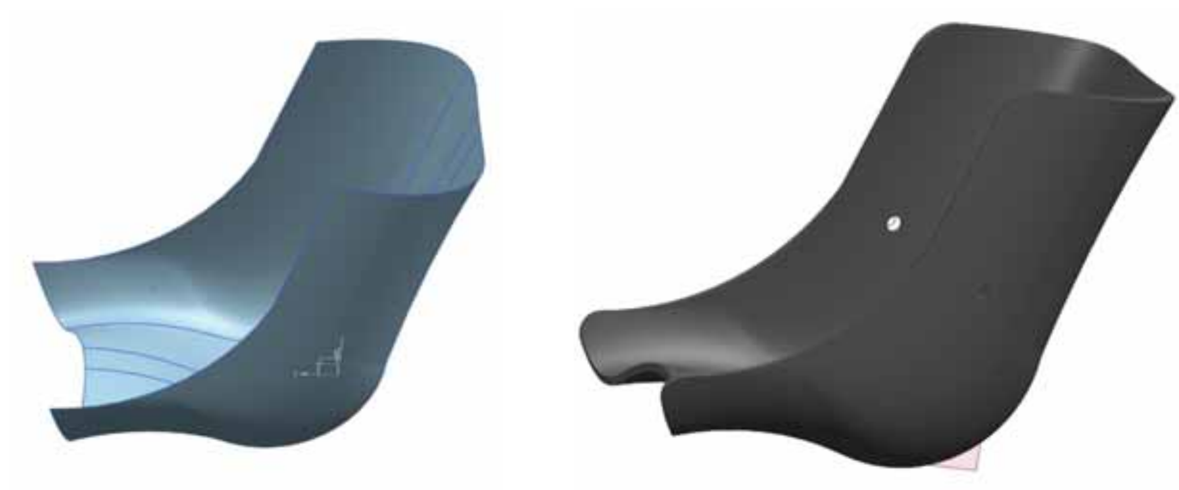




When the main curves are completed, next step is to connect them using polynomial curves to the highest order to get a smoothest curve. In this case best fitting polynomial to the order of 6 is used.



Once all curves are connected, the surface of the seat can be created. In this case command Through Curve Mesh is used. Once surface is created, volume is added using command Thicken. The seat is adjusted by creating openings that are used to fix it to the base of chassis.



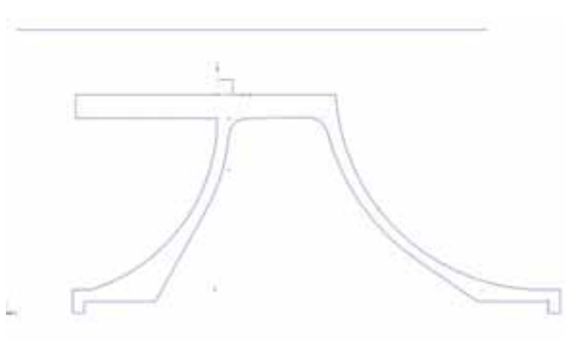
Pedals

Pedals are relatively simple part that is created using 3d curve sketch with few fillet commands. Once the curve is created a command Tube is used and a specific tubing inner and outer diameter is set.

Additional details are added by using pattern and extrude commands like adding a patterned pedal part where feet touch the pedals to reduce slipping, and where springs and cable are attached.

Front Wheel

Front wheels are created by designing a cross sectional sketch of wheel and then using rotational extrusion command and creating a final part. This tire also uses separate bearings part.



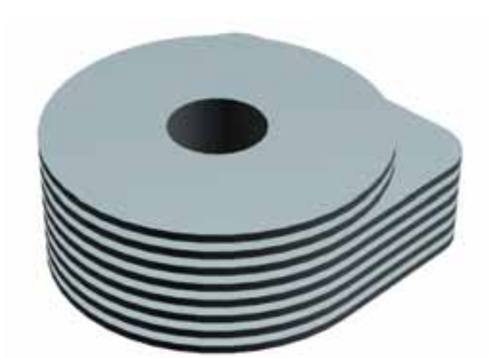
Front Tire

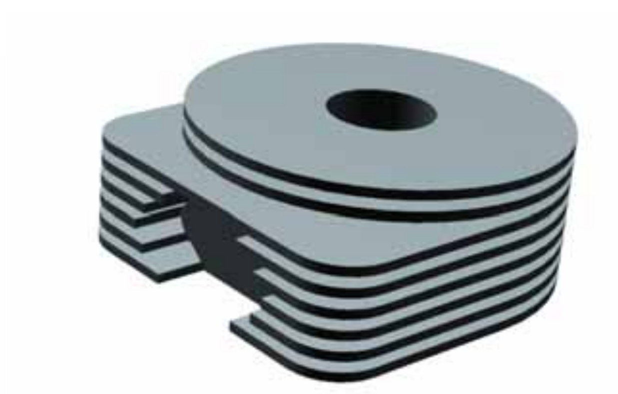
Front tires is a simple part that is designed by sketching the cross sectional that closely aligns with the front wheel. Front tire has smaller dimensions than the rear tire.

Powertrain:

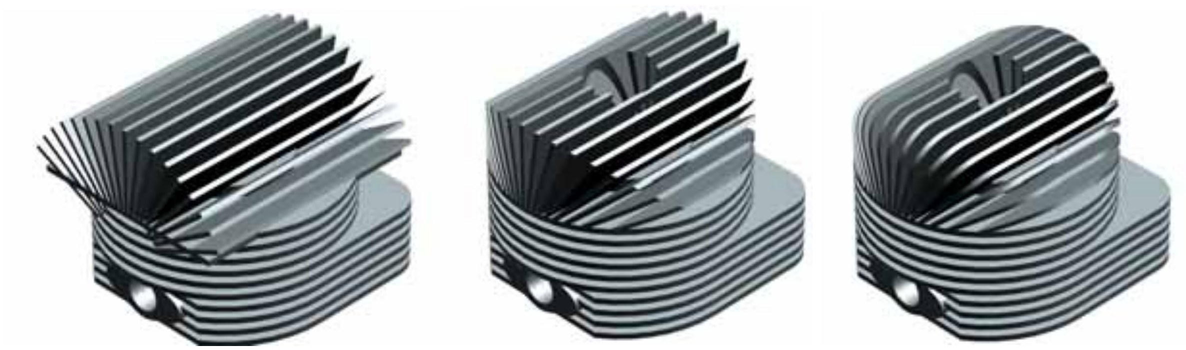
Engine

The powertrain was only designed to be a placeholder for where an engine would go. The engine is created by extrude a hollow cylinder and then extruding the fin and patterning the fins. The intake and exhaust parts are extruded on the engine.

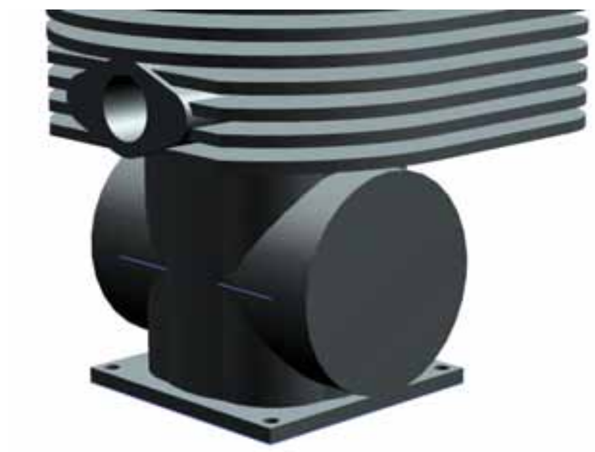




The head of the engine is created next by extruding a sketch of half of the fins and mirroring the fins. The fins are rounding using edge blend and the fin bodies are trimmed using trim body and are united with the engine body.

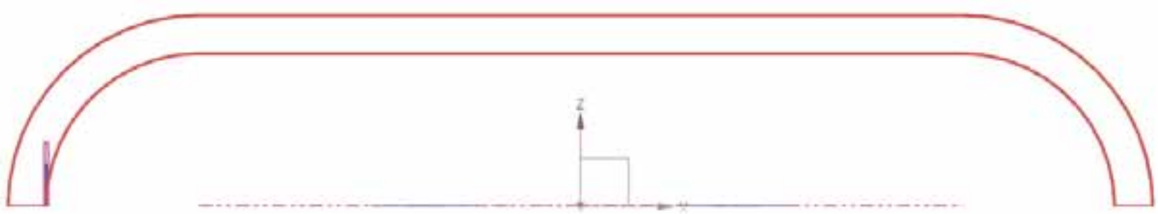


The last step is to extrude a crank body with mounting holes to function as a placeholder in the assembly.



Intake

The intake is created by revolving a sketch of the cross-sectional area of the intake. Then the engine mounting plate is extruded along with a hollow cylinder connecting the internal of the intake to the engine. finally, two holes are cut from the side of the intake to allow air to enter the intake and proceed to the engine.



Analysis:

Chassis FEA

The chassis torsional stiffness is calculated by applying a fixed constraint at the rear suspension mounting points, which is the rear axle brackets, and by applying an equal and opposite force of 100 lbf at the front suspension mounting points. The deflection in the vertical direction is used to determine the torsional stiffness of the chassis by using equations 3.1-3.3. The vehicle has a track width of Xx inches.

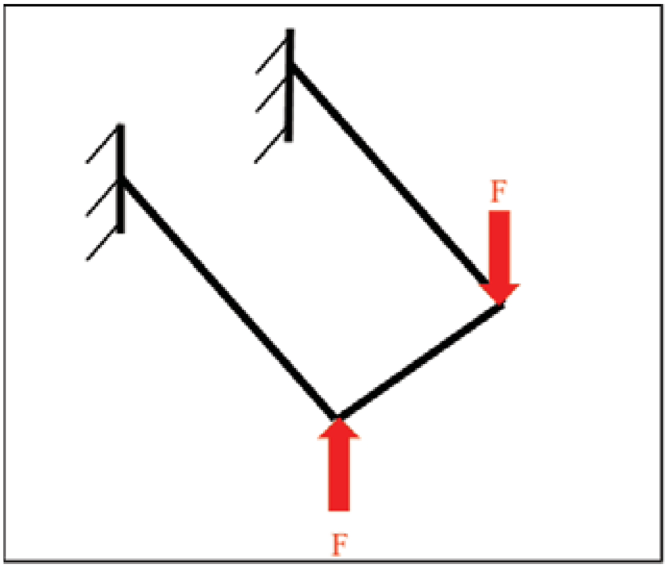
$$\theta = \tan^{-1}\frac{Z_{Front\ Left} - Z_{Front\ Right}}{Track\ Width}$$

$$M = \frac{100 * Track\ Width}{12}$$

$$Torsional\ Stiffness = \frac{M}{\theta}$$

Using The beam element equations, 3.4-3.6, for the simplified chassis, figure 3.1. the deflection for the front suspension nodes can be calculated. E = 28000ksi, I=0.0295in^4,

L_1=41.825in, L_2=27.9in, F_1=100lbf, F_2=-100lbf. The deflection in the vertical direction is determined to be 2.5 inches.

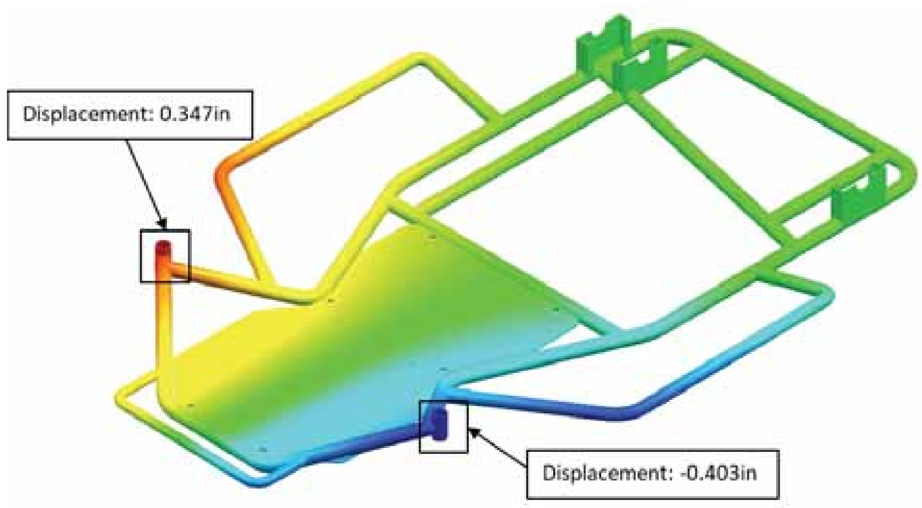


$$\begin{bmatrix} F_1 \\ M_1 \\ F_2 \\ M_2 \end{bmatrix} = \frac{EI}{L_1^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} d_1 \\ \varphi_1 \\ d_2 \\ \varphi_2 \end{bmatrix}$$

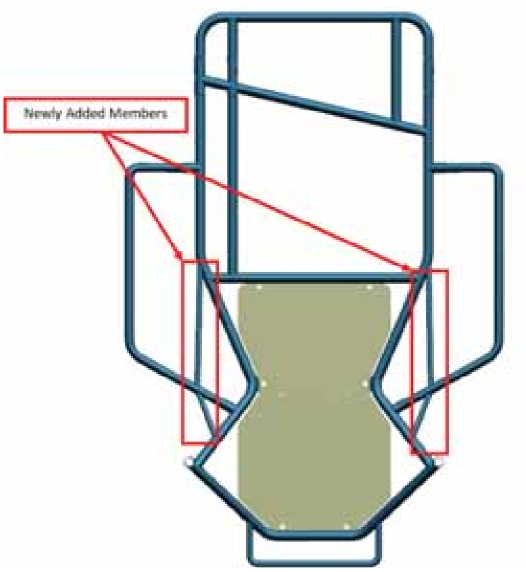
$$\begin{bmatrix} F_2 \\ M_2 \\ F_3 \\ M_3 \end{bmatrix} = \frac{EI}{L_2^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} d_2 \\ \varphi_2 \\ d_3 \\ \varphi_3 \end{bmatrix}$$

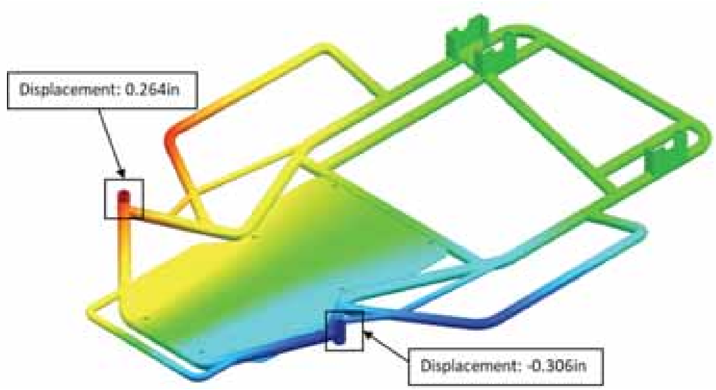
$$\begin{bmatrix} F_3 \\ M_3 \\ F_4 \\ M_4 \end{bmatrix} = \frac{EI}{L_1^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} d_3 \\ \varphi_3 \\ d_4 \\ \varphi_4 \end{bmatrix}$$

The simulation is set up with the same fixed constraints on the rear of the chassis and the equal but opposite forces on the front suspension points. The simulation, figures 3.2-3.3) produce a deflection in the z of approximately 0.347 inches for the right and -0.403 inches for the left, this results in a torsional stiffness of 360 ft-lbs/deg.



To increase the torsional stiffness additional members were added to the frame. The revised chassis is shown below along with the FEA results. The delefection of the front suspension points are reduced to 0.264 inches for the right and -0.306 inches for the left. This results in a torsional stiffness of 450 ft-lbs/deg which is a 24% increase in torsional stiffness.



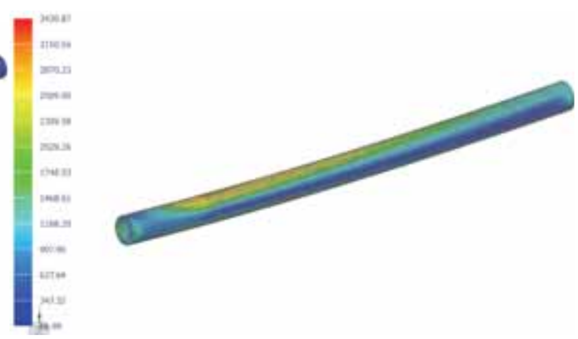


The different in the simulation results and the hand verification results can be attributed to the over simplification of the hand verification model. The simplified version of a chassis used in hand calculations does not account for the cross members in the chassis that would help prevent/resist twisting of the chassis. Given the missing cross members, it is expected that the hand verification results would yield higher deflection of the front nodes versus the simulation results of the full chassis model.

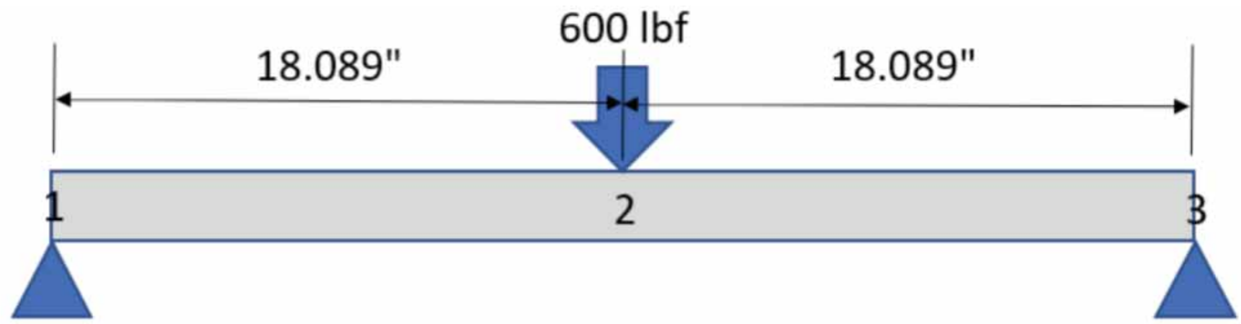
Rear Axle FEA

In order to reduce weight, the dimensions of the rear axle were minimized. The 35 mm axle outer diameter was kept due to availability of parts for it, meaning that the thickness could be no less than 3.8 mm per international standards. The axle is also required to be made out of steel. Assuming a max weight of kart and rider to be 500 pounds, the axle was tested to a safety factor of 2 by using 1000 pounds total weight. To properly distribute the force, it was assumed that 60% of the total weight (600 lbs) rests on the back axle. That was further divided between the three axle mounts, with the one near the back left wheel taking half (300 lbs) of it and the two near the back right evenly splitting the other half (150 lbs each).

This led to a maximum deformation in the center of the axle of 0.0144 inches and a maximum stress of 3430.87 psi, much less than the yield stress of 20000 psi.



To verify the FEA by hand, the axle was first simplified to a beam supported on each end. The entirety of the 600 pounds was applied to the middle of the beam rather than being spread out among each mount location, leading to the following diagram and hand calculations.



$$\begin{bmatrix} F_1 \\ M_1 \\ 600 \\ 0 \\ F_3 \\ M_3 \end{bmatrix} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 \\ 6L & 4L^2 & -6L & 2L^2 & 0 & 0 \\ -12 & -6L & 24 & 0 & -12 & 6L \\ 6L & 2L^2 & 0 & 8L^2 & -6L & 2L^2 \\ 0 & 0 & -12 & -6L & 12 & 6L \\ 0 & 0 & 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} * \begin{bmatrix} 0 \\ 0 \\ d_2 \\ \varphi_2 \\ 0 \\ 0 \end{bmatrix}$$

$$F_2 = 600\ lbf$$

$$M_2 = 0$$

$$d_1 = d_3 = 0"$$

$$\varphi_1 = \varphi_3 = 0$$

$$E = 30014109.4466\ psi$$

$$L = 18.089"$$

$$I = \frac{\pi}{4}\left(r_o^4 - r_i^4\right) = \frac{\pi}{4}(0.689^4 - 0.539^4) = 0.1107\ in^4$$

$$\frac{EI}{L^3} = \frac{30014109.4466 * 0.1107}{18.089^3} = 561.34\frac{lbf}{in}$$

$$600\ lbf = 561.34\frac{lbf}{in} * 24 * d_2 \rightarrow d_2 = 0.0445"$$

$$0 = 561.34\frac{lbf}{in} * 8 * 18.089^2 * \varphi_2 \rightarrow \varphi_2 = 0$$

$$F_1 = F_3 = -12 * 561.34\frac{lbf}{in} * 0.0445" = -299.76\ lbf$$

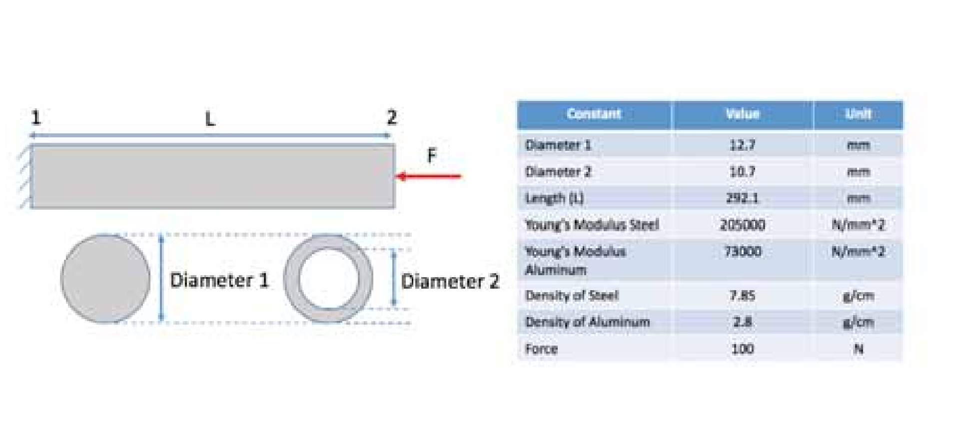
$$M_1 = -M_3 = -6 * 561.34\frac{lbf}{in} * 18.089 * 0.0445 = -2711.1 = -225.9\ ft - lbs$$

Based on the hand calculations, the maximum deflection should be 0.0445 inches in the middle of the axle. This is 3.09x greater than the FEA result, but the difference is expected due to the forces being simplified to one force in the middle where the max deflection occurs.

Push Rod FEA

The push rod assembly consists of two ball joints connected by a metal rod with a diameter 0.5 inches. This rod connects the steering column to the wheel bracket and translates the rotational input from the driver into rotational movement of the wheels. This translation of movement from the steering wheel assembly to the front wheel assembly is what allows the driver to change the direction of the go kart.

The rod connecting the two ball joints of the push rod assembly is currently manufactured of solid a steel. Because this rod receives relatively low compressive forces during operation, there is an opportunity to remove some weight. Three options are evaluated to determine the best method of optimizing the weight reduction of the push rod assembly. The first method consists of changing the material from steel to aluminum. The second option involves manufacturing the rod from a hollow rod of steel. The third option involves using a hollow aluminum tube. Each method is analyzed by loading the rod with a 140 N load and evaluating the associated deformation. This force is double the expected loading and therefore gives the analysis a factor of safety of two. This simulates the expected maximum forced from turning the steering column. The goal is to minimize deformation while reducing the mass of the connecting rod. This maintains the handling of the vehicle while reducing the overall weight of the assembly. The figure below shows the set up for the analysis including constants utilized in the calculations.



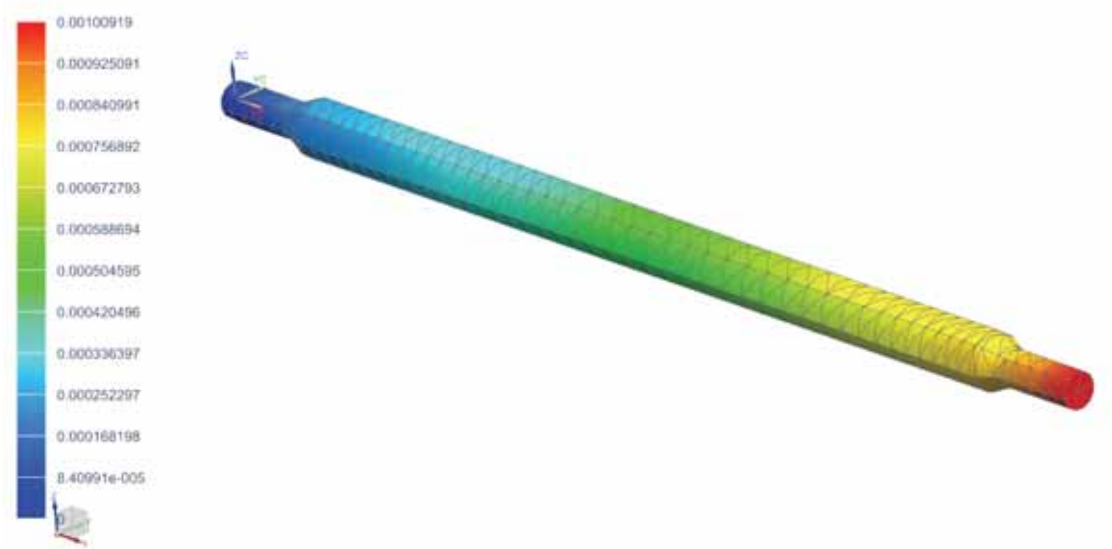
The expected deformation that each rod will withstand is calculated using equations shown below.

$$\begin{bmatrix} F_1 \\ F_2 \end{bmatrix} = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{bmatrix} dx_1 \\ dx_2 \end{bmatrix} \quad (1)$$

$$k = \frac{A * E}{L} \quad (2)$$

Hand calculations were performed using a cylinder to simplify the geometry and calculations. These results are used to validate the computer aided engineering solution. The three scenarios were also tested using Siemens NX to simulate the same compressive load. These simulations have more accurate geometry which produces a more reliable analysis tool. The deformation results for all three scenarios can be seen below.

Deformation Results						
Scenario	Material	Rod Composition	Max Deformation Hand Calculations (in)	Max Deformation FEA	mass (lbs)	Mass Change (%)
1	Steel	Solid	0.000262	0.000383	0.644	0.0
2	Aluminum	Solid	0.001090	0.001009	0.162	74.7
3	Steel	Hollow	0.000903	0.001095	0.156	71.0
4	Aluminum	Hollow	0.002535	0.0030535	0.066	89.7



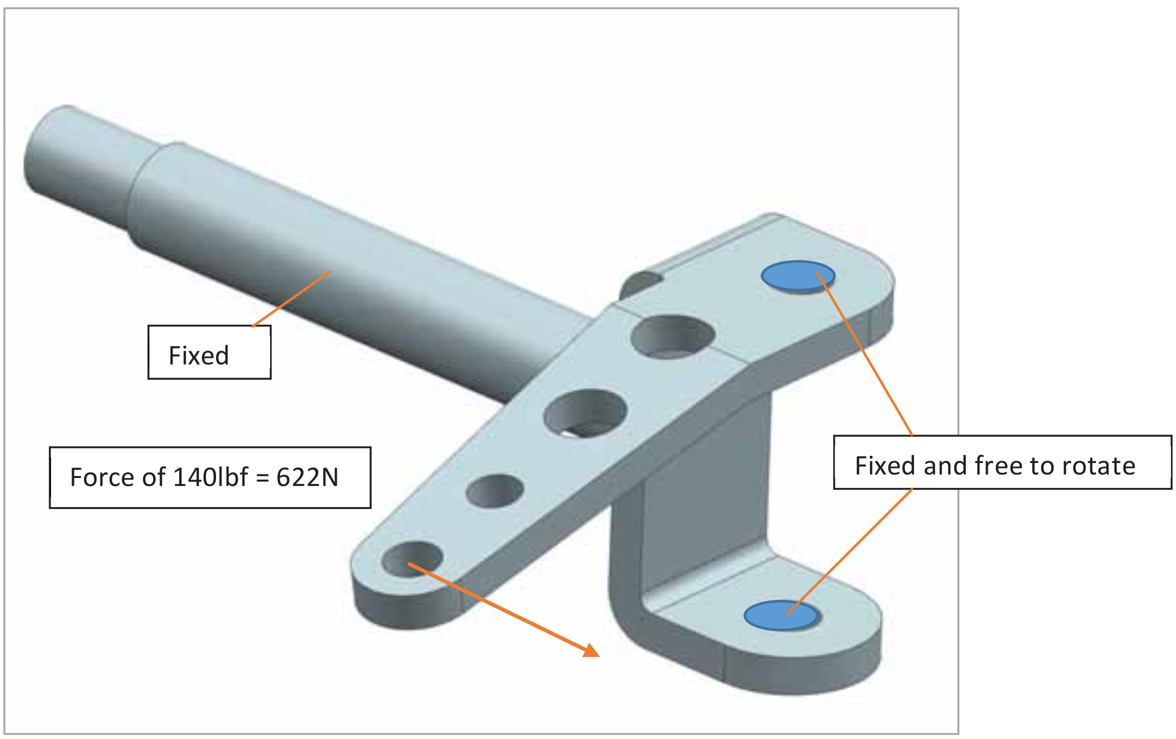
Figure

Based on these results, the hand calculations and FEA results have confirmed the same affects that each scenario would have on the deformation of the connecting rod. The FEA results differ slightly because the hand calculations were oversimplified to simplify calculations. Overall, this proved that changing materials or the geometry in scenarios 2 and 3 would have similar effects on the maximum deformation. Further analysis was performed to determine how each scenario would affect the overall mass of the assembly. This proved that scenario 2 was able to reduce the weight by 74.681 percent while only increasing the maximum deformation to 0.001090 mm. Similarly, scenario 3 was able to reduce the weight by 70.984 percent but increased the deformation to 0.001095 mm. Option 4 has a maximum deformation of 0.0030535 and would reduce the weight by 89.7 percent. This analysis proves that choosing a solid aluminum rod is the best solution to minimize weight while maintaining the stiffness of the rod.

<u>Front Wheel Bracket FEA</u>

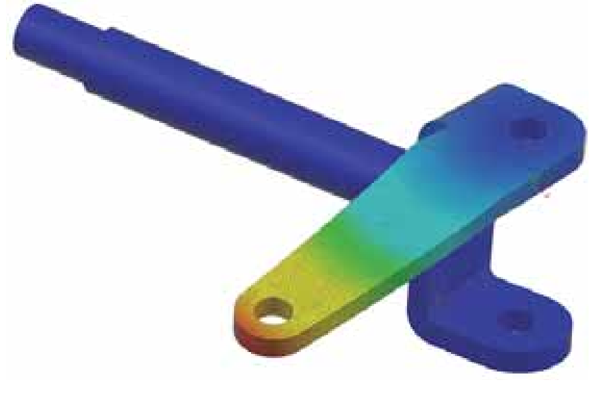
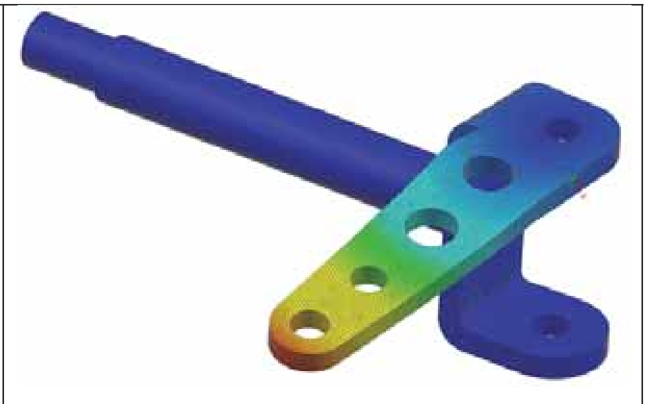
One of the design approaches that helps to maximize a specific characteristic of a part is a modification of its geometry. In the following example Front Bracket was modified to have less weight by having multiple openings and still keep its deflection change to a minimum.

Material	E	L	b	h	I	EI/L^3
Aluminum 6061	**6.89*10^9 Pa**	**0.0889 m**	**0.00635 m**	**0.02857 m**	**1.23467E-08m^4**	**121077**

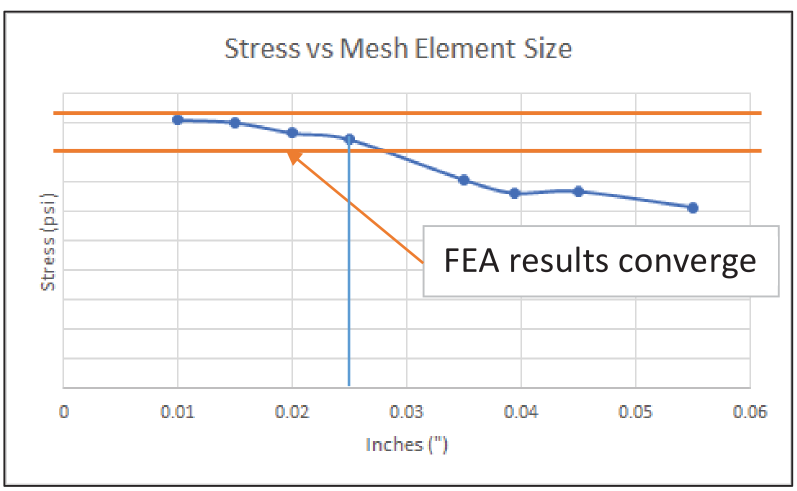


Front Wheel Bracket Part, forces and constrains.

FEA Analysis:

	
Original Design	**New Optimized Design**
Deflection: 0.0231”	**Deflection: 0.0252”**
	Deflection increase under 5% **Weight reduction of 4.5%**

To have a more precise FEA analysis results, mesh size was decreased, and optimal mesh size was found to be 0.025”. Optimal mesh size for FEA analysis was found by performing multiple FEAs and observing max stress convergences as in graph below. At 0.025” mesh size, time to perform the FEA is optimal. This helps to reduce the computing power therefore cost.



Time to perform FEA	**Mesh Size**
5min	0.035”
12min	0.030”
<u>20min</u>	**<u>0.025”</u>**
86min	0.020”

<u>FEA Verification:</u>

For the FEA verification, front wheel bracket was simplified and modeled as cantilever beam with a constant cross-sectional area.

$$\begin{matrix} 0 \\ M_1 \\ F_2 \\ 0 \end{matrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{bmatrix} 0 \\ 0 \\ d_2 \\ \varphi_2 \end{bmatrix} \rightarrow \begin{bmatrix} M \\ F \end{bmatrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & -6L \\ -6L & 4L^2 \end{bmatrix}\begin{bmatrix} d_2 \\ \varphi_2 \end{bmatrix}$$

$$\begin{bmatrix} M_1 \\ 622 \end{bmatrix} = \begin{bmatrix} 1452935.66 & -64582.98987 \\ -64582.98987 & 3827.618533 \end{bmatrix}\begin{bmatrix} d_2 \\ \varphi_2 \end{bmatrix}$$

FEA analysis for the Front Wheel Bracket has shown that the maximum displacement result under a 140lbf side load is about 0.0252” which is satisfactory for this design use. To back up this FEA analysis, hand-drawn calculations are performed. In this a case a simplification to a cantilever beam will be sufficient and by using above matrix formula, the displacement calculations for d_2 is performed. By solving a system of equations, the answer for d_2 is 0.01671”. The hand calculated answer is different, but it is within the same magnitude. The difference in deflections in FEA and hand calculations is due to the simplification of the beam model and that the part has multiple holes and different cross-section throughout. Calculating a precise moment of inertia would improve results.

Summary and Future Work:

Optimization of Go Kart design project has been successful in achieving its goal of optimizing existing Go Kart for performance increase to compete in professional racing events. The stiffness of the chassis was increased to improve handling. The choice of axel was validated through FEA analysis. Finally, the front wheel bracket and push rod were analyzed and redesigned to reduce weight while maintaining stiffness.

Future work would include a further design and optimization of chassis and addition of a more powerful engine and analysis to confirm that the parts can withstand the additional forces. Also, in the future more body panels would be designed, and their aerodynamic performance would be analyzed to validate their design.