Exercise One: Creating and Starting a Fusion 360 Account

Create an Account:

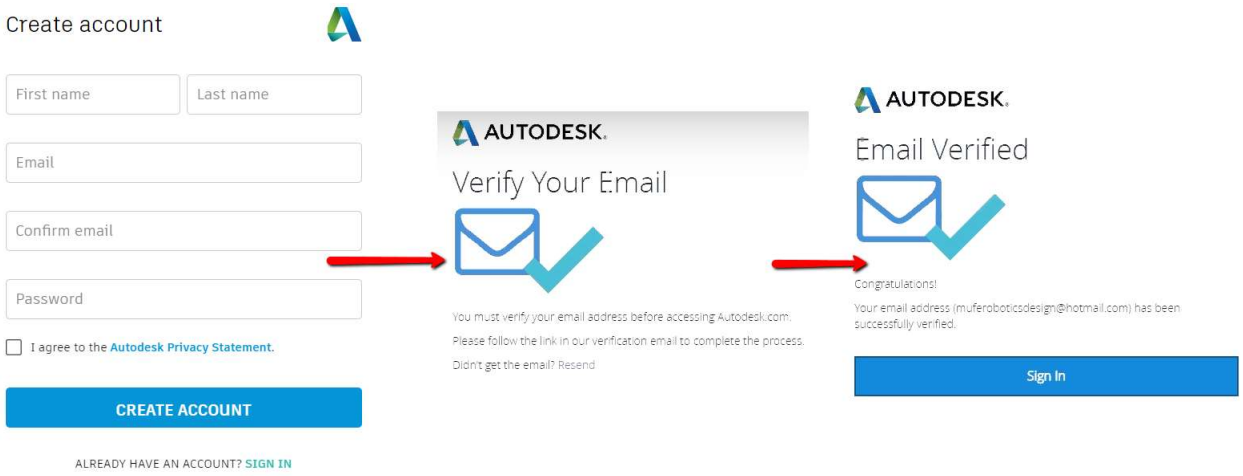
- Download: https://www.autodesk.com/products/fusion-360/overview
- For Registration: https://accounts.autodesk.com/register
- Education: https://www.autodesk.com/products/fusion-360/students-teachers-educators

The links above are for downloading Fusion 360 and to create your account.

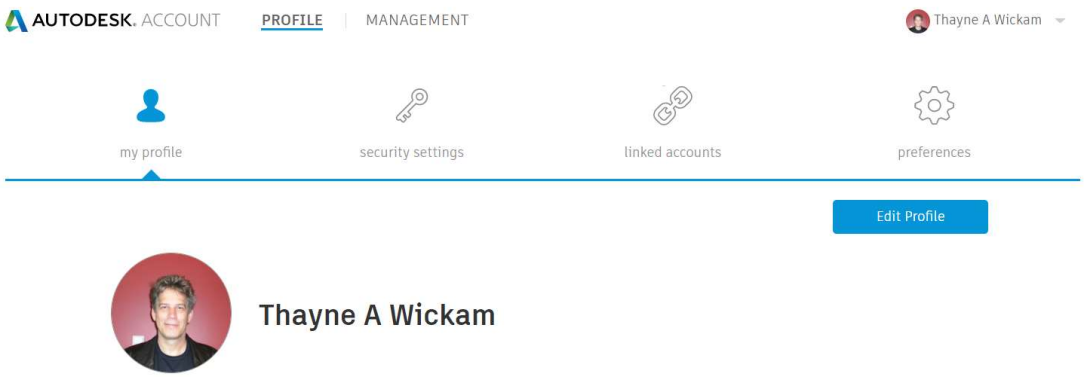
- Fusion 360 is cloud based, meaning the files you create are saved in a secured cloud.
- Files can be saved locally if needed.
- You are the only one that can control access to these files.

After the installation of Fusion 360:

1. Create an account to use it.
2. Verify your email address, by going to your email and replying to it.
3. You will receive a notification, that your email has been verified.



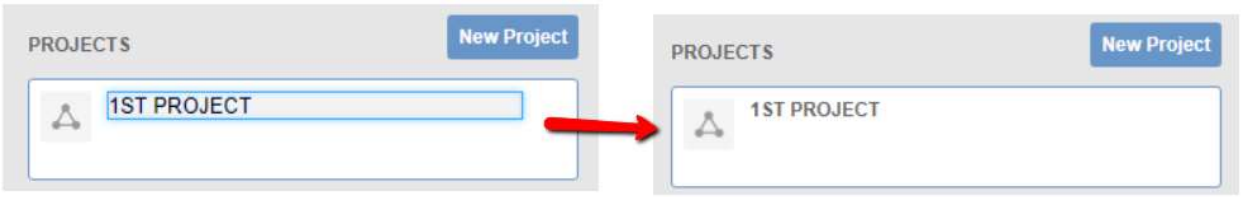
Once the account has been created, you will be able to use Fusion 360.



Exercise Two: Create a Project and Folder

Select the New Project button to Start a New Project. Provide it a Name and the Project is created when you hit Enter.

1. Create a Project called 1ST PROJECT.
2. Double click 1ST PROJECT to enter it.



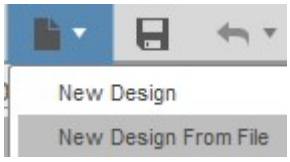
Select the New Folder button to Start a New Folder. Provide it a Name and the Folder is created when you hit Enter.

3. Create a Folder called 1ST PROJECT FOLDER.
4. Double click 1ST PROJECT FOLDER to enter it.

This will be the Project and Folder for these sessions.

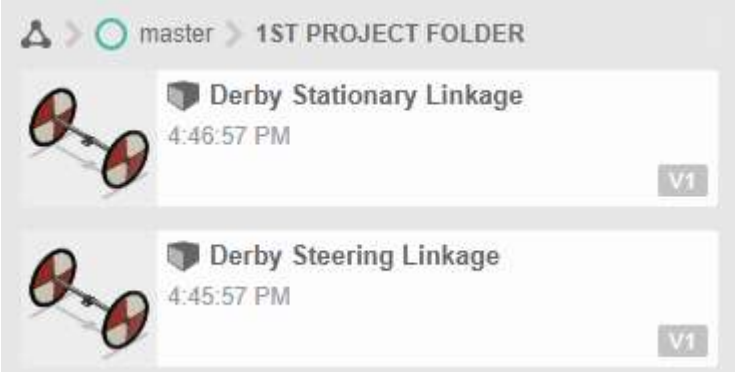
Exercise Three: Create Designs from External Files

1. Extract the files from Fusion 360 External Files.zip, to your C:/ drive.
2. In Fusion 360, go to File> New Design From File:



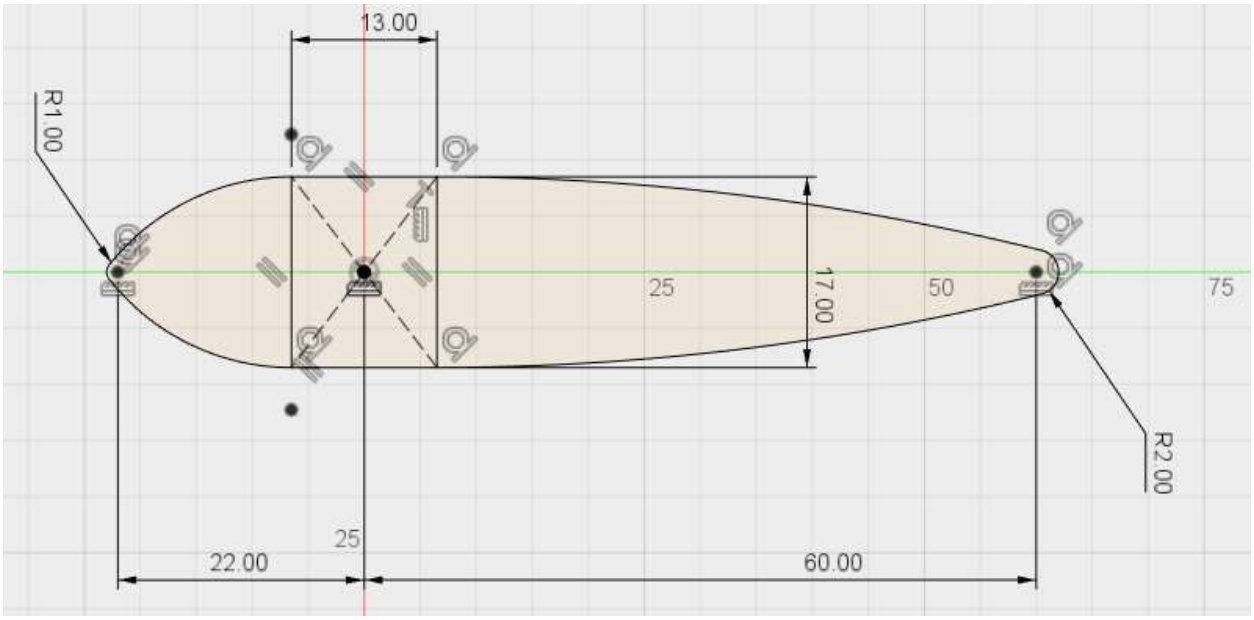
3. Go to C:/Fusion 360 External Files
4. Go to the Derby Steering Linkage folder and select “DERBY STEERING LINKAGE.sat”
5. Save the design as Derby Steering Linkage
6. The image should show up in the data panel
7. Go to the Derby Stationary Linkage folder and select “DERBY STATIOANRY LINKAGE.iges”
8. Save the design as Derby Stationary Linkage

The image should show up in the data panel:



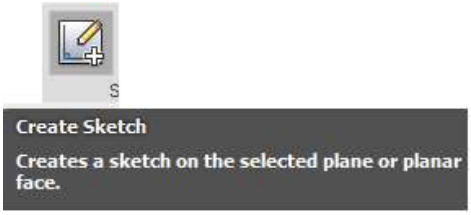
Exercise Four: Create a Sketch

Most base features start with a sketch. Sketches provide constraints and dimension driven details of the design. The dimensions drive the sketch for a more accurate design.

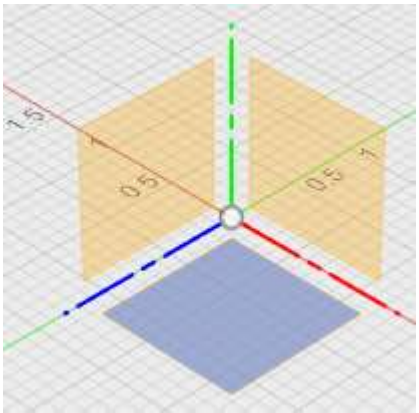


To develop the floorboard of the derby car we will start with a sketch on one of the origin planes.

1. Select the Create Sketch button.



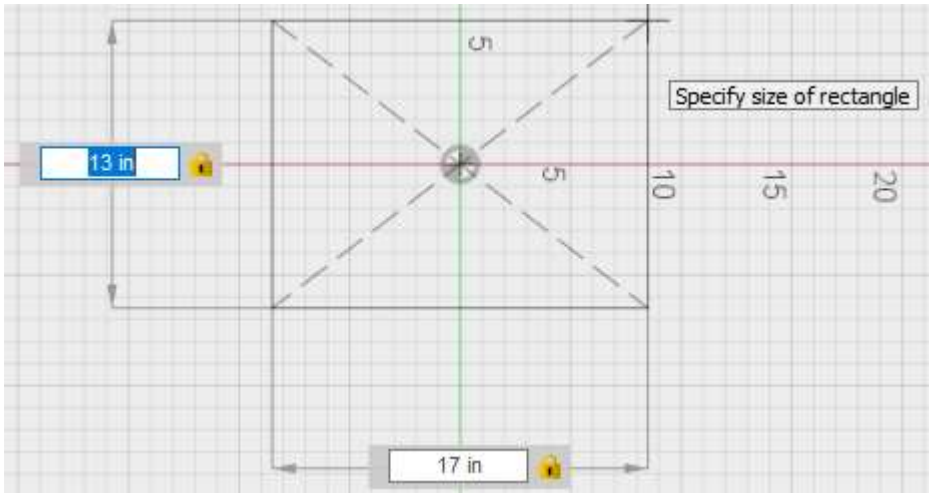
2. For the orientation needed, select the bottom plane, or the XZ plane.



The first part of the sketch will be anchoring the rectangle at the design's 0,0,0 location.

All design files have a zero origin for the X,Y and Z axis. Start the sketch where it is anchored to a center point. The example below shows a center rectangle conjoined at the center point. Length and width can be entered for the exact size of the rectangle.

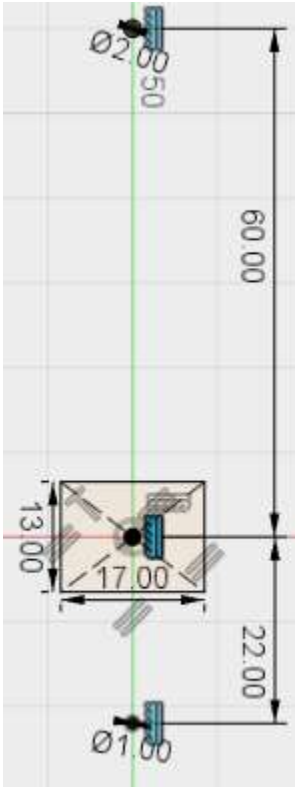
1. Create a center rectangle at the center point, 17 inches wide by 13 inches high.



Next will be the front and back arcs of the floorboard.

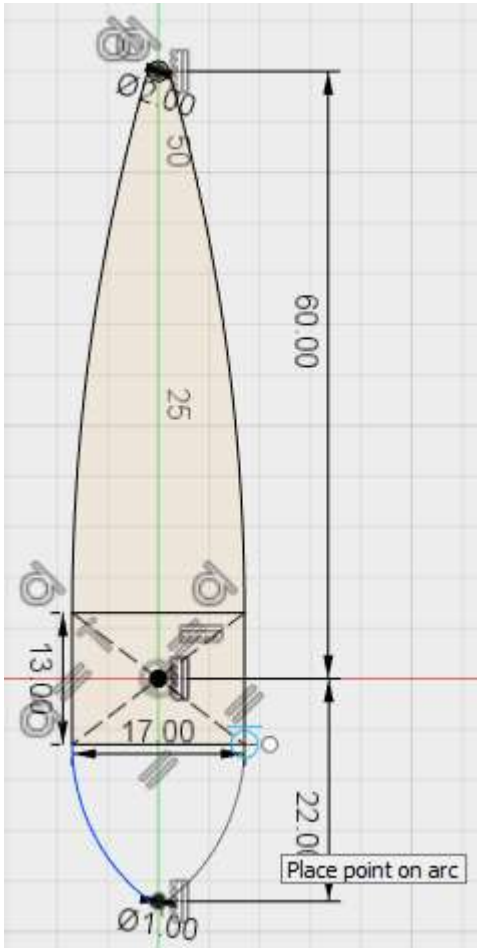
2. Create a Circle with a 2-inch diameter above the rectangle to where it is in line to the center point.
3. Select the center of the circle and the center point using a vertical constraint.

4. Create a 1-inch circle to the same requirements as the 2-inch circle.
5. Set the top circle 60-inches from the center point.
6. Set the 1-inch circle 22-inches from the center point.

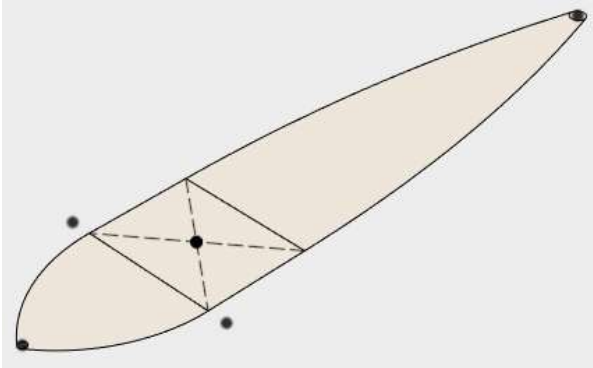


Now to connect the circles and rectangle with tangent arcs.

1. Create 3-point arcs between the rectangle and the two circles.
2. Add tangent constraints if not already applied.



3. Finish the sketch to get back to model mode, by hitting the Stop Sketch button.

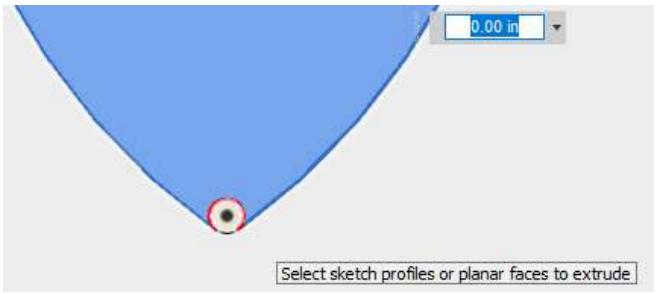


Exercise Five: Extrude a Sketch

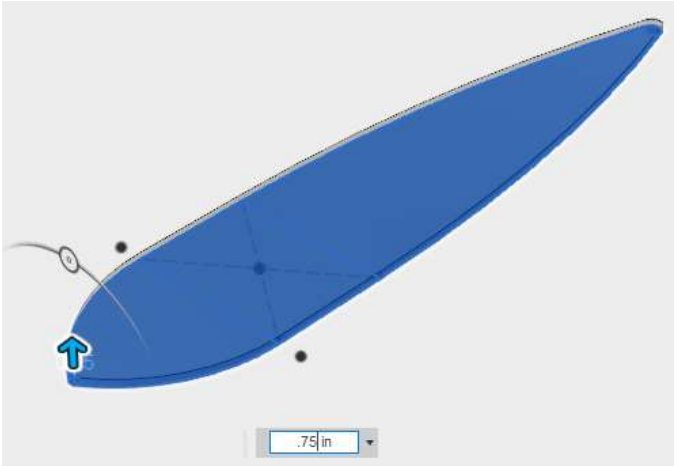
Now we can give the sketch thickness with an extrude.

1. Start the Extrude command.

2. Select all closed profiles in the sketch, including the circles.



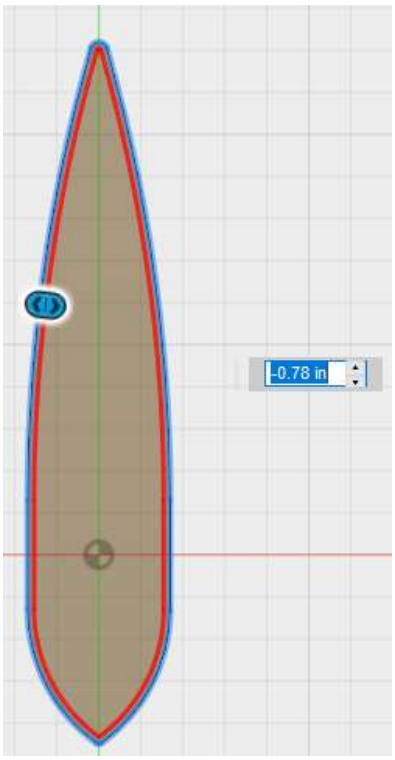
3. Extrude up to .75 inches.



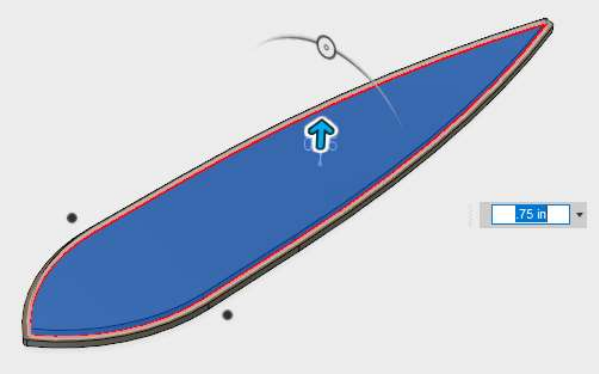
This will be the floorboard made from ¾ inch plywood.

Exercise Six: Create a Second Sketch

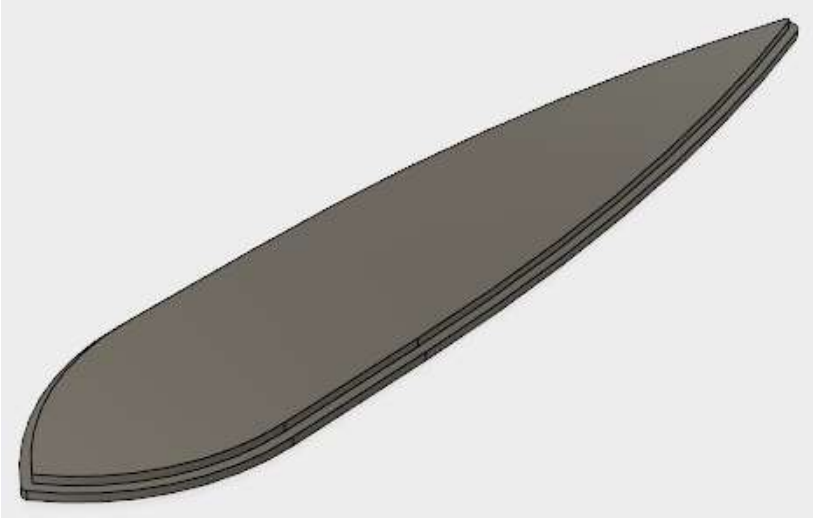
Now we can add a second ¾ inch plywood on top of the base feature, with a ¾ inch offset. This will be used to mount to top body and reinforce the floorboard.



1. Right Click the top of the floorboard.
2. Select Create Sketch.
3. Select the Offset command under sketch.
4. Pick the top edge of the floorboard
5. Type .75 and Enter to make a ¾-inch offset.
6. Stop the sketch
7. Extrude to .75-inches



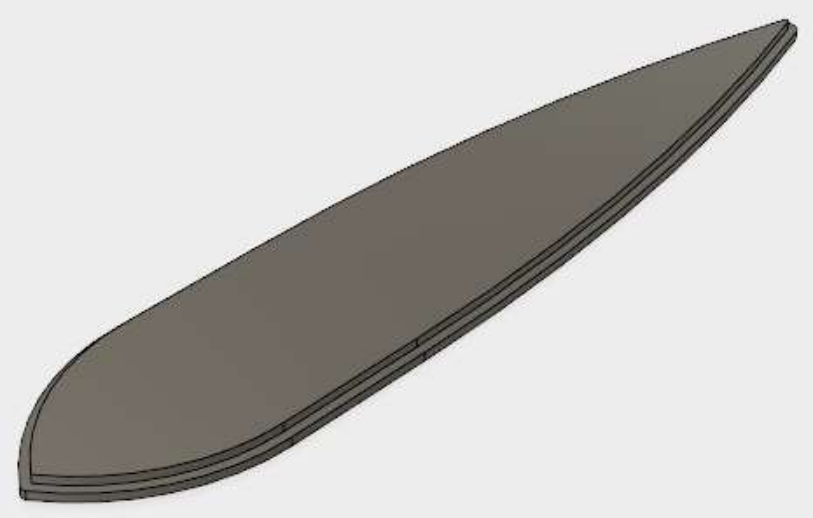
The final extrusion should provide a lip around the perimeter of the floorboard:



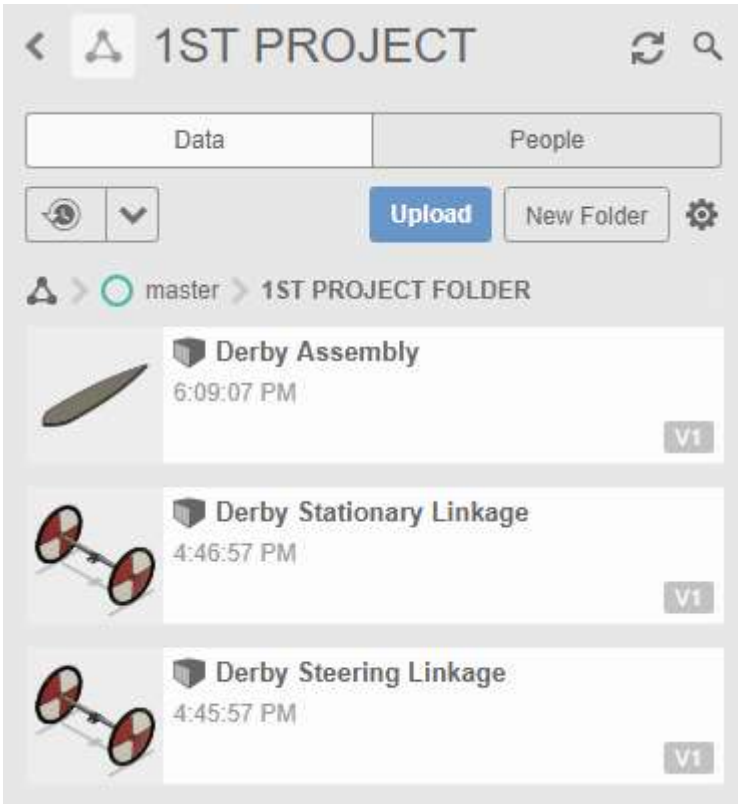
Exercise Seven: Save the File

Now we save the file to use it for the next session.

1. Pick the save button
2. Save the file as “Derby Assembly”



Once the file has been saved, it should show in the data panel under 1ST PROJECT FOLDER:


1ST PROJECT
Data
People
Upload
New Folder
master
1ST PROJECT FOLDER
Derby Assembly
6:09:07 PM
V1
Derby Stationary Linkage
4:46:57 PM
V1
Derby Steering Linkage
4:45:57 PM
V1

Fusion 360 One Hour Sessions – Session 1

For Session Two: From the first session's designs, learn how to add secondary features such as joins, cuts and intersects. Also learn how to use multiple bodies and how to create them into components. Understand how to align bodies and how to use joints with components.

1. Create Secondary Features
2. Creating Multiple Solids
3. Moving and Aligning Solids
4. Creating Components from Solids
5. Using Joints