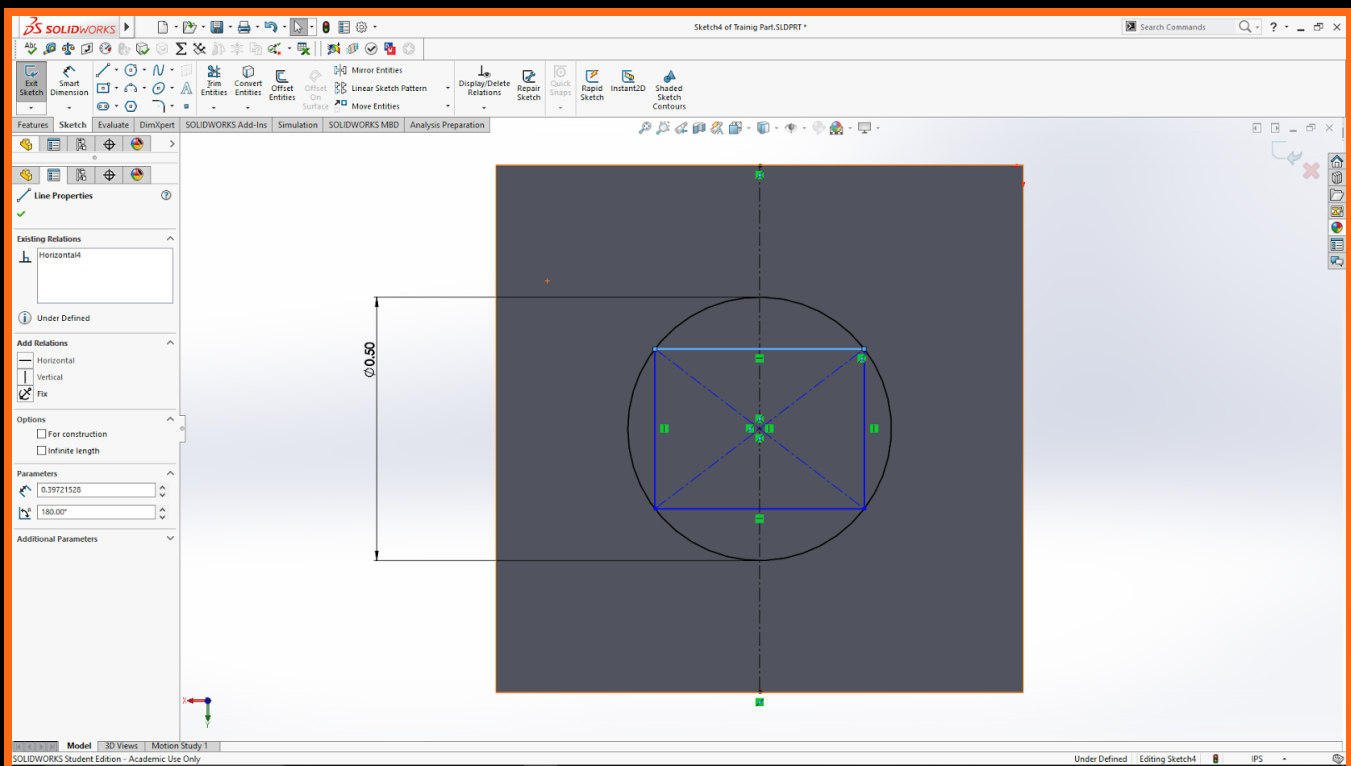


2017 Advanced SolidWorks Information

Topics Reviewed-

Construction Sketches-

- Any part of a sketch can be turned into a construction line or shape. When a feature of a sketch is turned into a construction sketch it will no longer interfere or be a part of an extrude or an extruded cut. Construction sketches are used for reference and relations.
- Right click on a line or feature and a menu where the feature tree is located will appear. Select the lines or features you want to use for construction. Now, select the box “For construction” under options. Select the lines or features you want to use for construction. Now, select the box “For construction” under options.



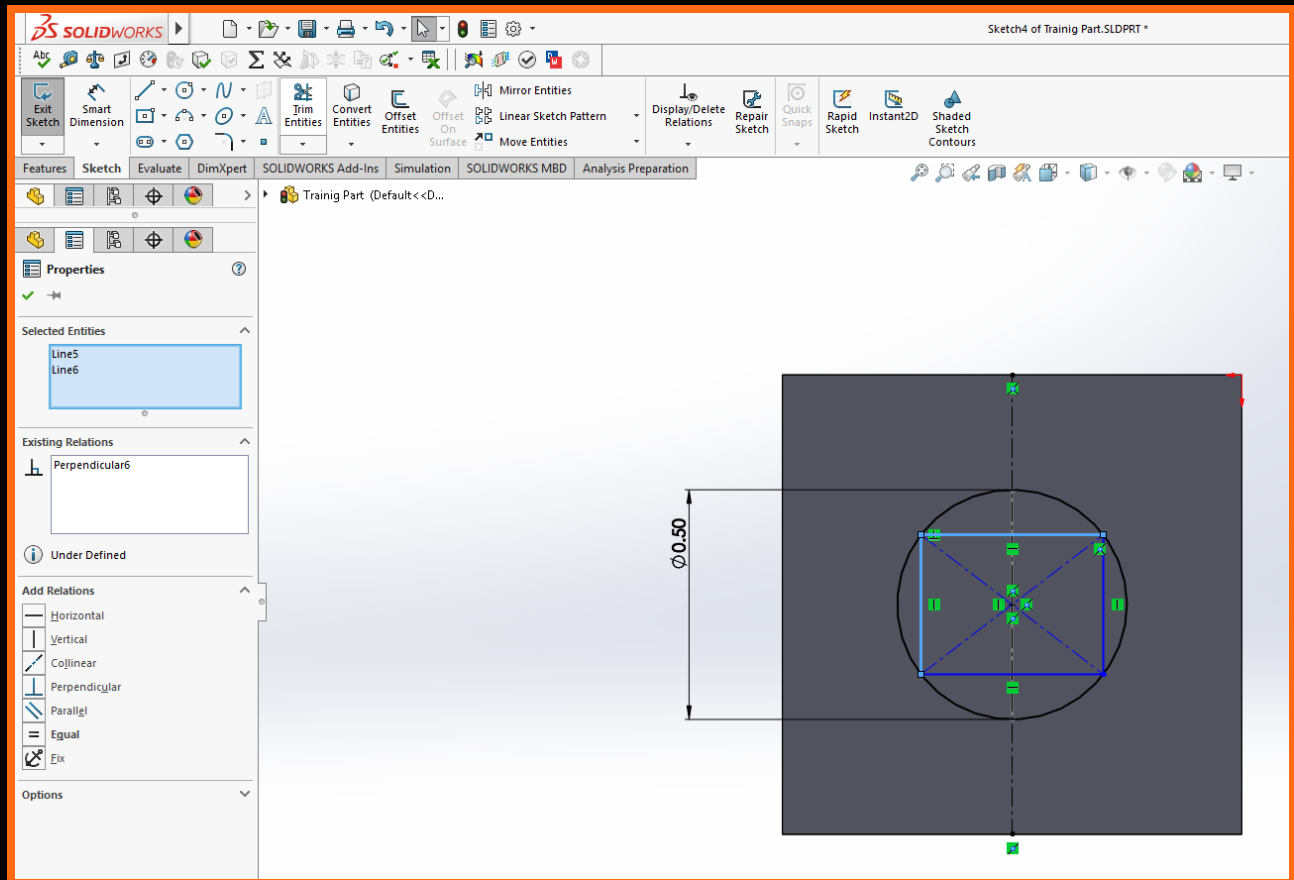
Example-

- In this example the top horizontal line that is highlighted is the selected feature that is going to be turned into a construction line.
- The “Relations and Options” toolbar is present in the left hand side of the photo.
- To make a feature for construction, select the feature and select “For construction” under options.



Sketch Relations-

- Sketch relations are the mates of a sketch and they are shown by the small green boxes next to lines and splines. The easiest way to work with them is to right click on the line or spline you want to add to or remove from sketch relations and the vertical toolbar will appear. To select more lines or splines, hold control and click to select the lines and/or splines. Then add or remove relations.



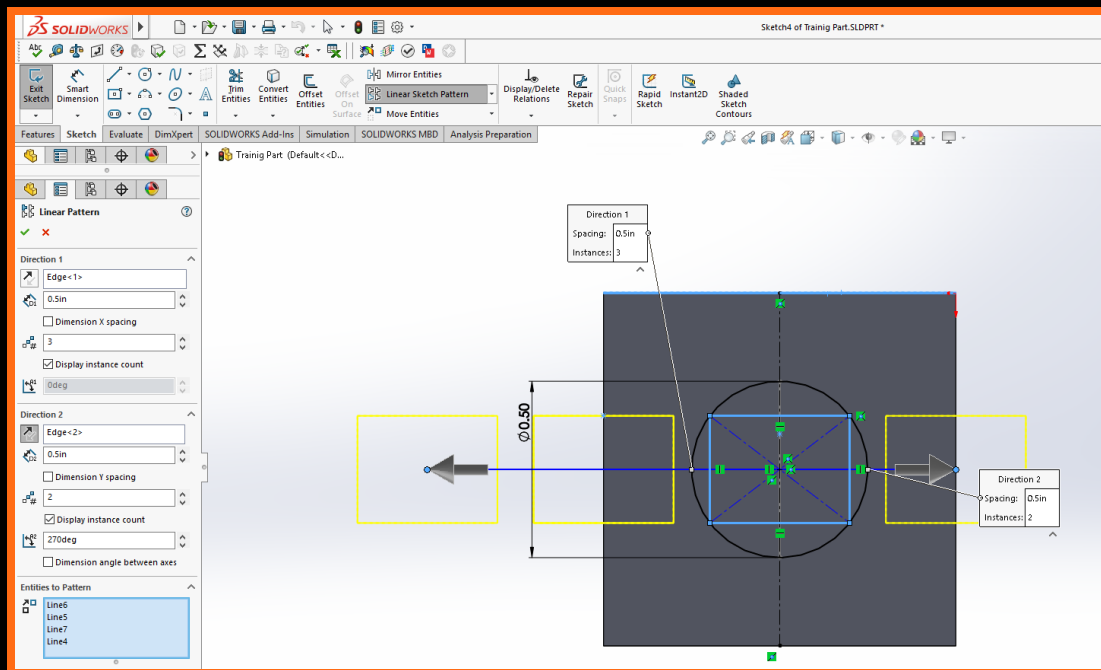
Example-

- Here you can see that the “Relations and Options” toolbar is present on the left hand side of the screen.
- The relations are the small green boxes present on the sketch, the symbol in each of the boxes represent the different relations.
- Some shapes and the placement of them generate relations upon creation (ie. squares like above).
- Here the top highlighted line and the right vertical line are the features of the sketch that the relation is going to be applied to.(Select by right click and holding control to select another feature).



Linear and Circular Sketch Patterns-

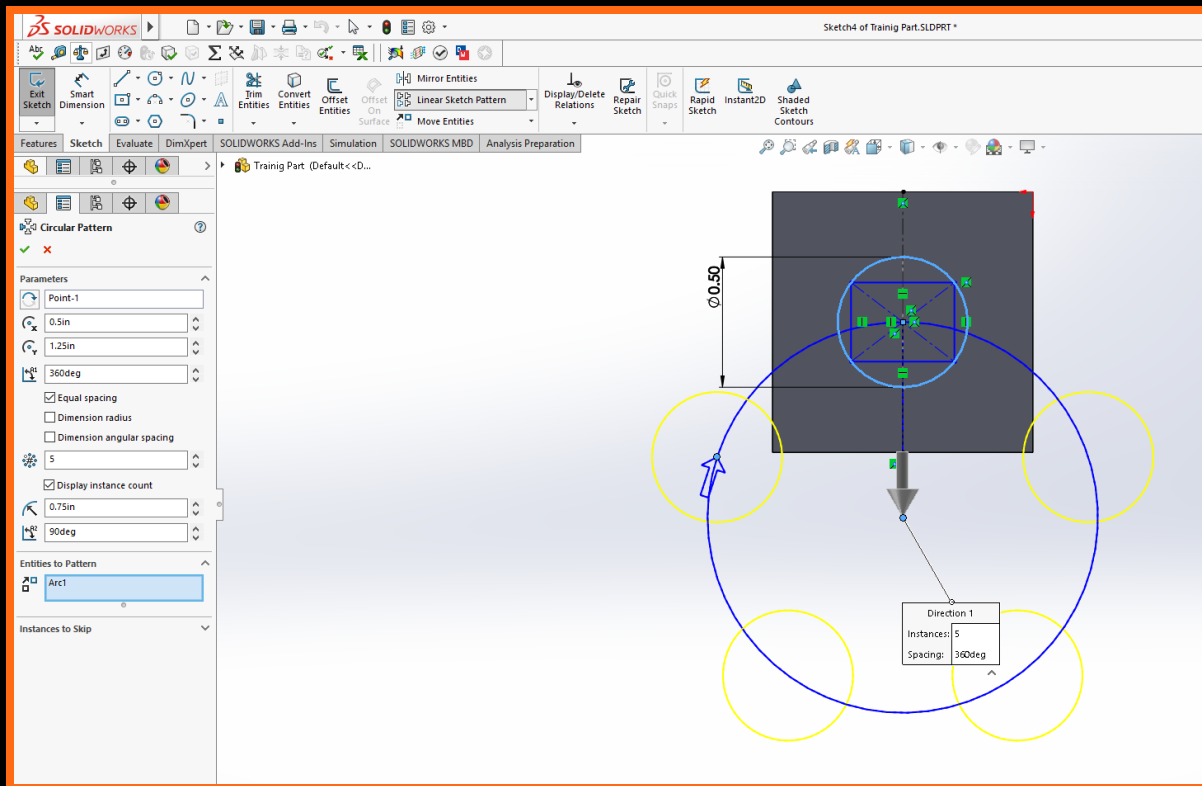
- A single feature or multiple parts of a sketch can be replicated in any direction or at any angle within the same sketch using sketch patterns.
- After the initial sketch that you want to be replicated is finished and fully defined, select either “Linear Sketch Patterns” or “Circular Sketch Patterns” under the drop down within the sketch tab of the horizontal tool bar.
- For linear sketch patterns you will then need to select an edge or axis to replicate your pattern on, the distance at which you want to offset the next replication of the pattern from the center of the other, the number of replications, and what you are going to be replicating. There is also an option of selecting an angle at which to replicate your pattern.
- For circular sketch patterning you will need to select a point that you want all of your patterns to revolve around, the offset on the x and y direction from the origin, the degrees you want to continue the pattern, the number of replications, the radius of the patterns arc, degrees offset from the x axis, and what you want to be patterned.



Example-

- This is a linear sketch pattern example.
- The sketch being replicated is the square in the center of the circle.
- After the sketch of the square was created and fully defined “Linear Sketch Pattern” was selected in the sketch toolbar.





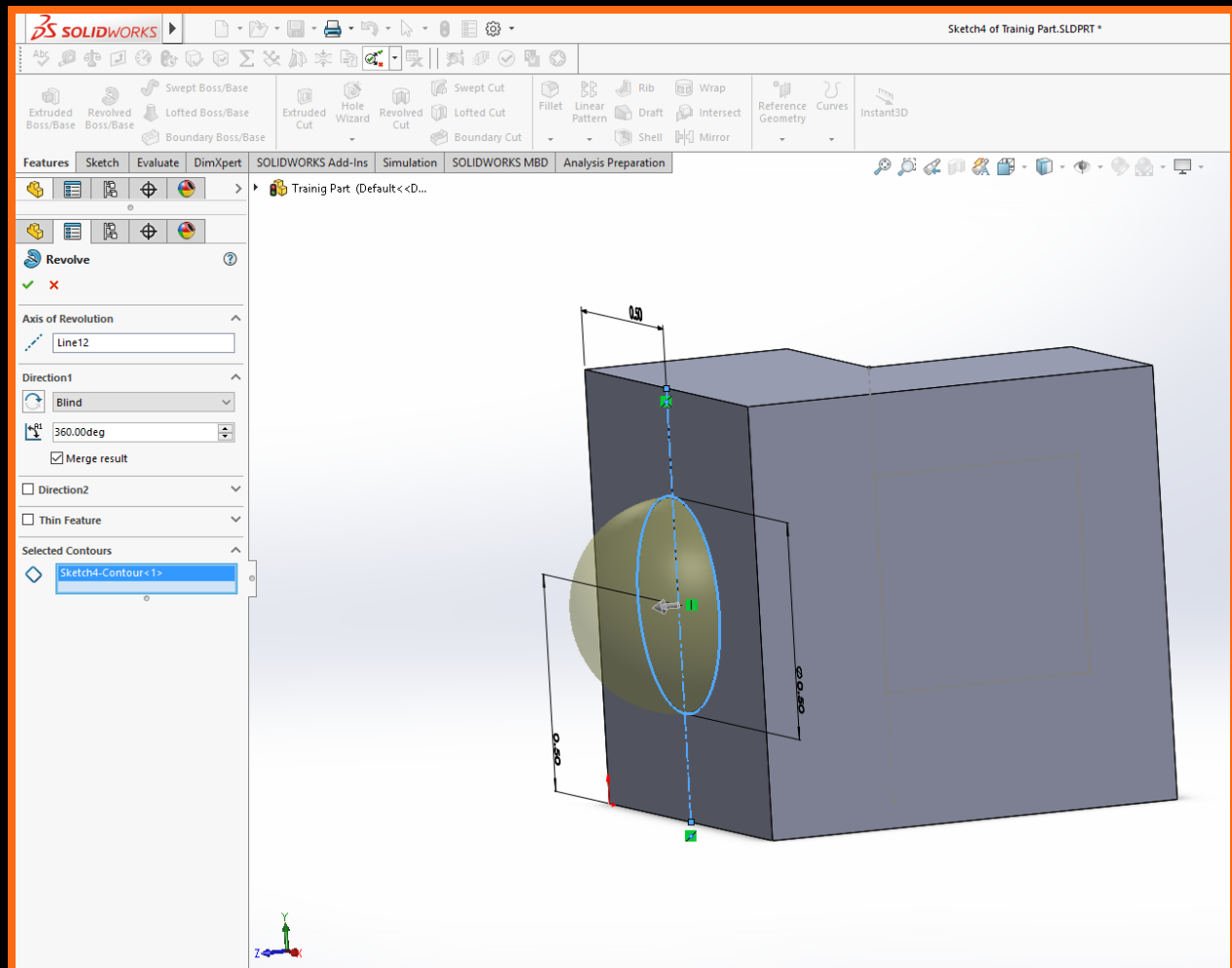
Example-

- This is a circular sketch pattern example.
- The sketch being replicated is the circle around the square.
- After the sketch of the circle was created and fully defined “Circular Sketch Pattern” was selected in the sketch toolbar under the drop down with “Linear Sketch Patterns”.
- The second and third options down are the offset in the x and y direction from the origin.



Revolve Tool-

- The revolve tool is used to create an extrude or cut extrude with a constantly curved edge that is revolved around an axis.
- Create the shape that you would like to be revolved and a construction line as the point of central revolution. Select the revolve tool, the features to be revolved, and axis of revolution. Then input the degrees it needs to be revolved around the axis or any of the other revolve distance options in the drop down bar.



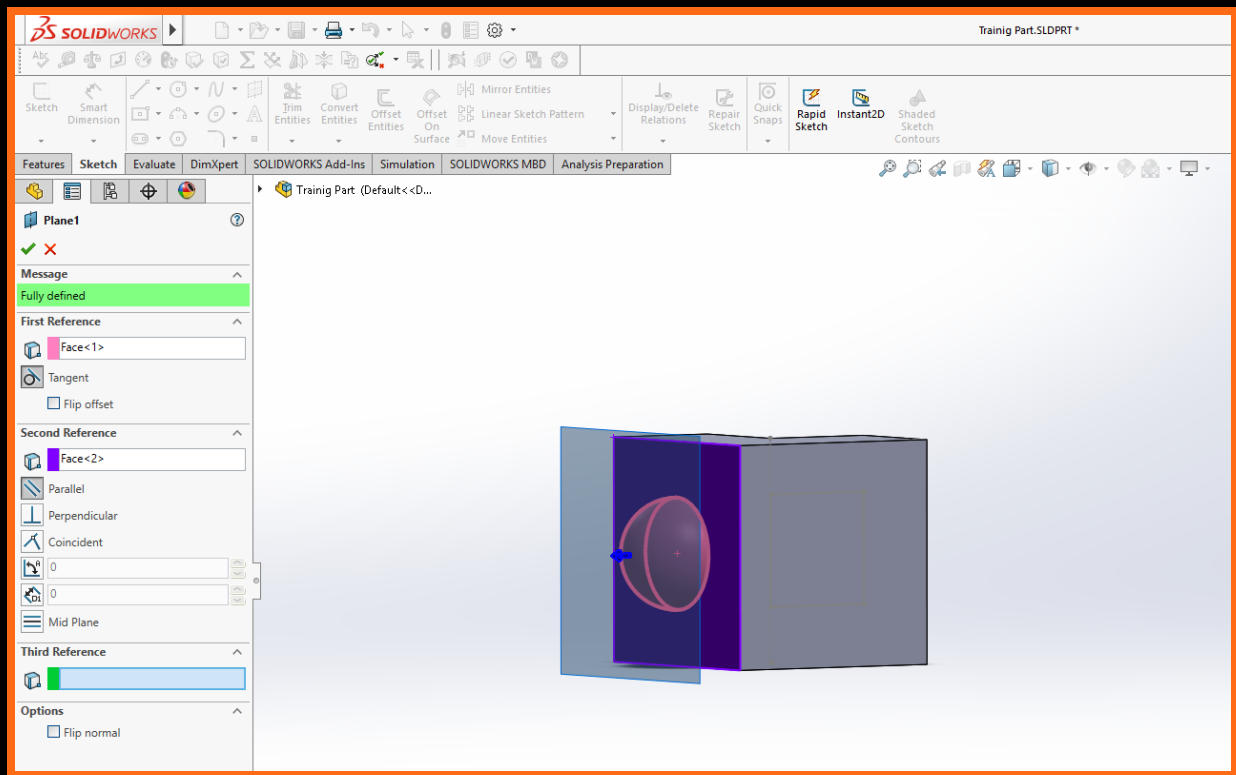
Example-

- The axis of revolution in this example is the highlighted and dotted construction line and the shape being revolved is the circle placed on the center of the axis of revolution.
- The axis of revolution does not need to be in the center of the shape being revolved.
- There are multiple options for extruding your revolve, such as "Up To Surface".



Reference Geometry-

- Reference geometry can be used within sketches, parts, and assemblies.
- The two most common types of reference geometry are planes and points. Reference geometry is not like a sketch in the sense that you can only use the plane you are sketching on with just that sketch. Reference geometry can be referenced for any feature of a part or assembly.
- Sketching on a reference geometry plane or having a reference geometry point is the same as on any other plain, except that it is created for a specific purpose. This goes back to the idea that SolidWorks files are just a compilation of vectors which is shown as planes and faces.



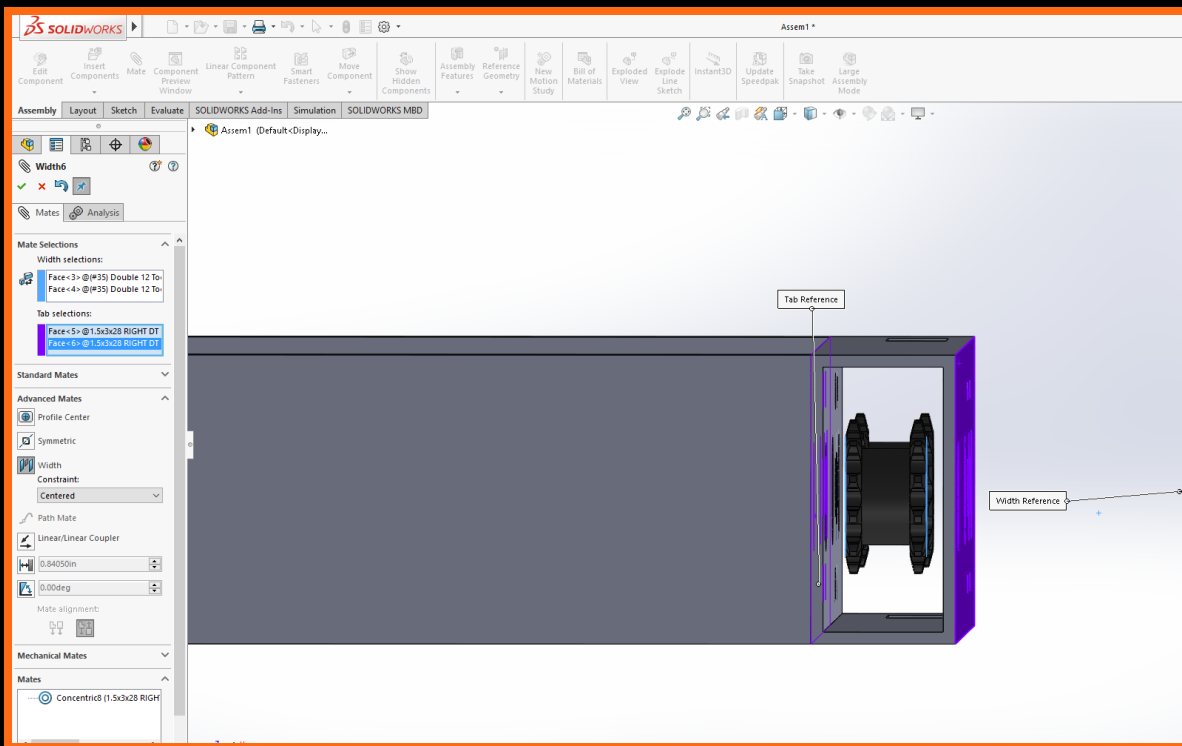
Example-

- In this example, the light shaded face furthest to the left is the plane that is being created.
- The rounded revolved surface is the first reference, the plane is being created tangent to its face.
- The second reference is the flat surface that the revolve is coincident to. The plane is going to be created parallel to this face.
- In the top left in the reference geometry toolbar there is a message that states that the plane is now “fully defined”. Reference geometry, just like anything else, is a part that should always be fully defined.



Advanced Mates-

- Advanced mates are used in assemblies for convenience and simplicity. The most common advanced mate that you are likely to use in FRC is the width mate.
- The width mate allows you to center an object by in between two faces, points, or segments within as assembly.
- This can prevent a lot of reworking that could need to be done if the distance of something was to change. (ex. You have two rails distance mated together and two rails running between them like in a drivetrain. You decide you want to lengthen the rails running in between to widen the drive base. You would then have to change your distance mate and probably others. If you had used a width mate that adjustment would be made for you.)
- Find advanced mates in the “Mates” toolbar. There is a drop down that you can select under the types of basic mates.



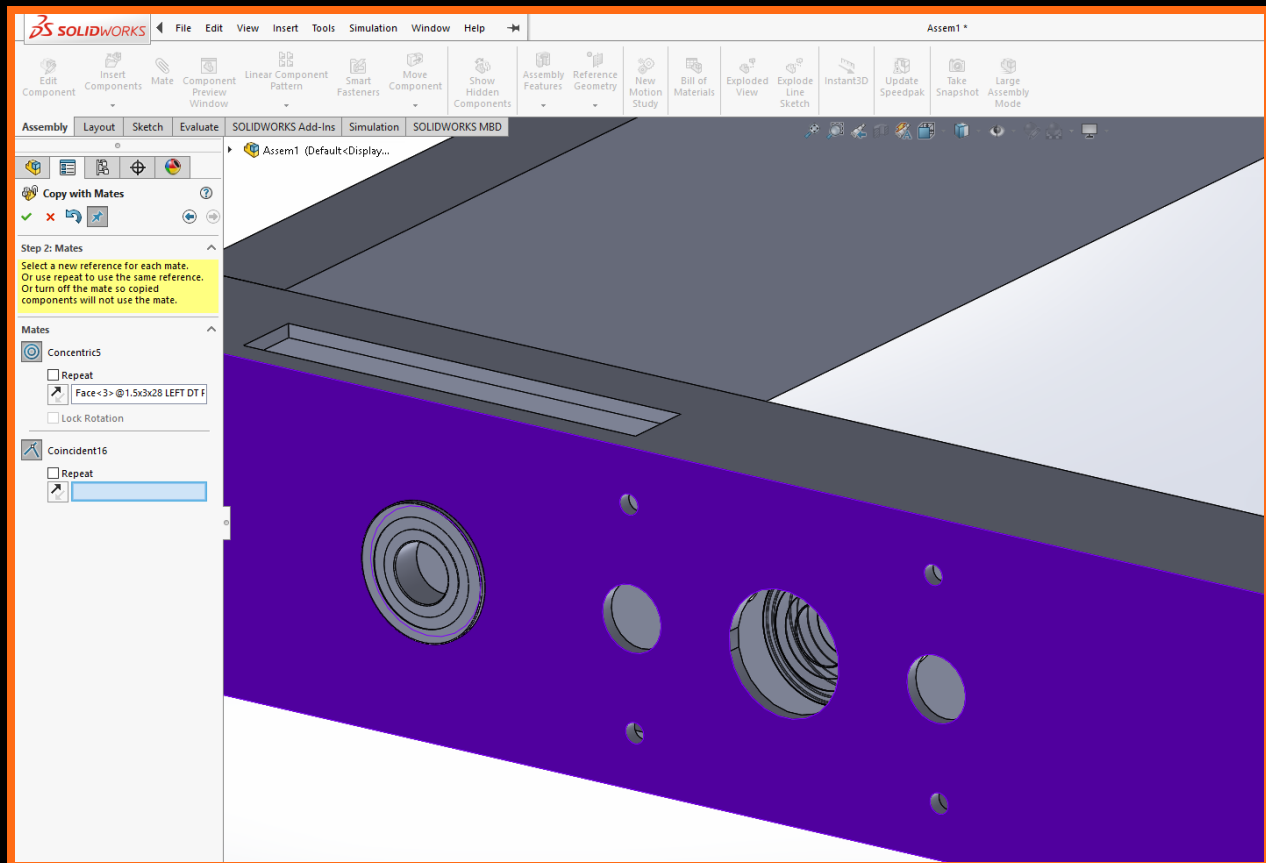
Example-

- In this example a sprocket is being width mated inside a drive rail.
- The width selections would be the two outside faces of the sprocket and the two tab selections would be either the two inside or two outside faces of the rail itself.
- To use distance mate to mate you would have to find the width of the drive rail and the width of the sprocket then subtract the two, then divide them by two. That is a considerably longer process and more prone to error than just using a width mate.



Copy With Mates-

- Copy with mates is a great tool when you need to place multiples of the same part that has the same mates in an assembly(ex. bearings or flywheels)
- Once you have all the mates you want to be copied with the specific part; right click and select “Copy with Mates”
- Once you are in copy with mates press the white arrow to go forward and then select the new faces that you want the corresponding mates to mate to.



Example-

- In this example a bearing is being copied.
- The highlighted box on the left, below a type of mate, is the mate it wants you to select a face for.
- In the image, and on your screen, the face that the original mate was mated to is highlighted for reference as to indicate what mate you are currently selecting for. This is needed when there are multiples of the same type of mate. In this example it is the highlighted front facing side of the rail.



Other Tips-

Ctrl Gesture -

- Quick way to copy a part within an assembly
- Hold control, select part, drag away and drop part to duplicate.

Mouse Gestures-

- You can set up mouse gestures to make your CADing even more efficient
- Set up mouse gestures by going into the “Tools” drop down by the SolidWorks logo, selecting “Customize”, and then going to the “Mouse Gestures” tab.

Large Assemblies Mode-

- Large assemblies mode is very helpful when you get deep into designing a robot.
- Large assemblies mode will reduce your lag and improve the load being put on your laptop
- You can set an assembly to large assemblies mode by going to the “Tools” dropdown in the menu bar that appears when you select the arrow next to the SolidWorks logo in the top left corner.

Split View of an Assembly-

- When designing a robot there are times you are going to want to see how certain components work with others; for instance, the compression for an intake. This can be hard to see when you have a large file like a robot where something can be hidden on the interior. In SolidWorks we can manipulate assemblies to be able to view any part of our assembly clearly.
- Using reference geometry, add a plane where you want to view the assembly. Often this is just splitting the robot in half.
- After you have the plane where you want, right click the plane and select “Section View” in the pop up toolbar, then select the plane you want to view it from.

Changing Documents-

- For different parts or assemblies you might want different document units or additional decimals.
- To change these select the ISP pop up in the bottom right of your screen.



Additional Resources-

Basic Understanding-

- [This video](#) will give someone a better understanding of what is actually happening when you use a feature in SolidWorks.
- Within SolidWorks there are very basic tutorials that will help someone who is just learning how to use some of the tools. Access these by selecting the “New Document” icon when you first open SolidWorks. Then select “Tutorials” there are many tutorial options that you can use specific to what you need to learn.
- [Team 1114 the Simbots](#) have made a great series of SolidWorks tutorials for all skill levels.

Advanced Resources-

- [Team 1114 the Simbots](#) have a made a great series of SolidWorks tutorials for all skill levels.
- There is a YouTube page specifically devoted to SolidWorks tutorials. [This is their page link.](#)
- [This](#) a an advanced SolidWorks handbook that is dense, but has a lot of material.

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