
Solidworks Lesson 5 - Assembly Configurations and Motion

UCF Engineering



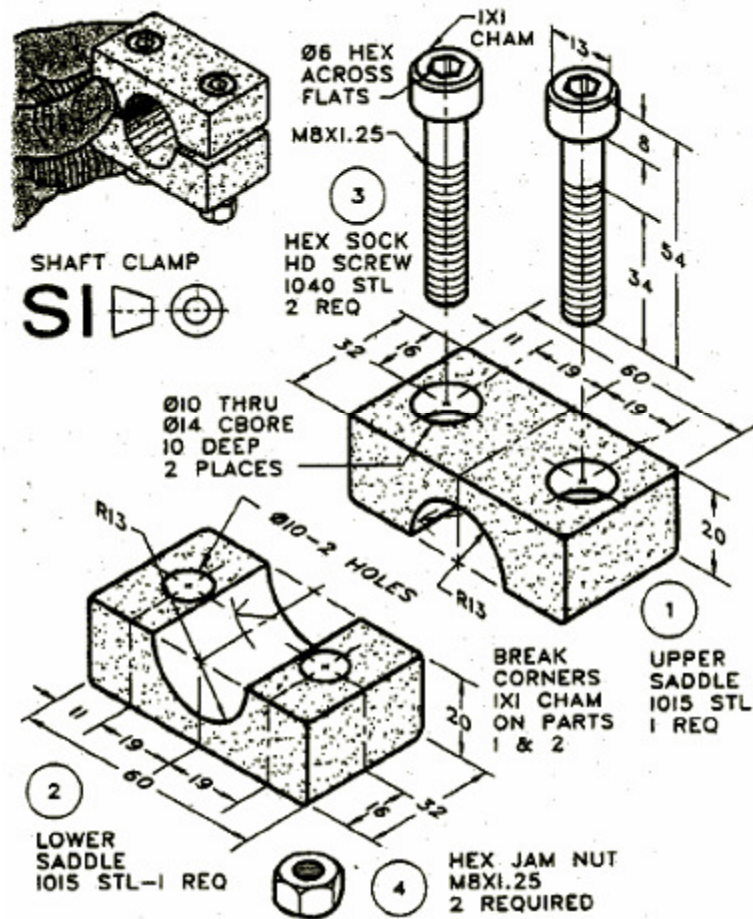
University of Central Florida
College of Engineering and Computer Science

Solidworks

- During the last lesson we learned how to construct assemblies using the basic mating constraints that are present in SolidWorks.
 - The first thing that we are going to do this lesson is look at how we can create alternate configurations for the assembly, in order to represent exploded or section views.
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Solidworks

Begin by opening the assembly that we created last lesson.



Configuration Manager

Oftentimes, when we are creating assemblies we want to show an exploded view which illustrates to the reader how to assemble/disassemble the part.

SolidWorks provides us an excellent tool in the Configuration Manager to allow us to construct exploded views of our part.

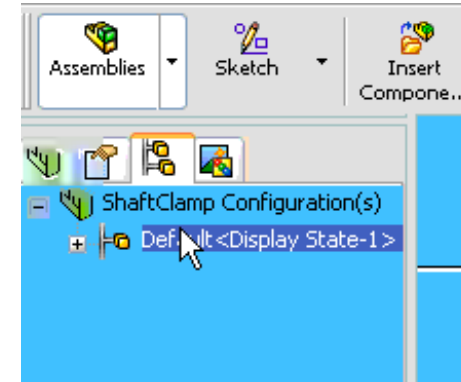
Configuration Manager

Using the configuration manager we can create different “configurations” of our part which we can turn on and off on the fly.

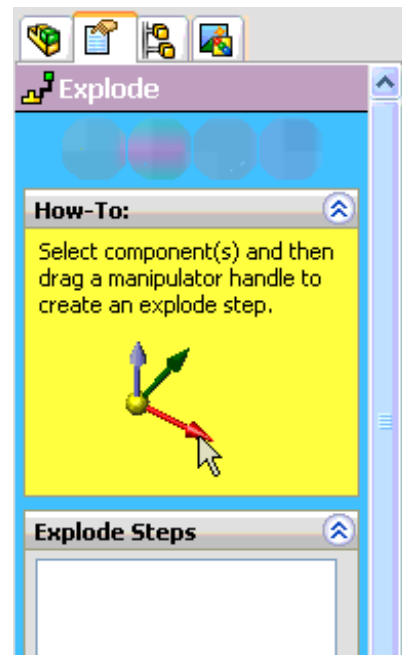
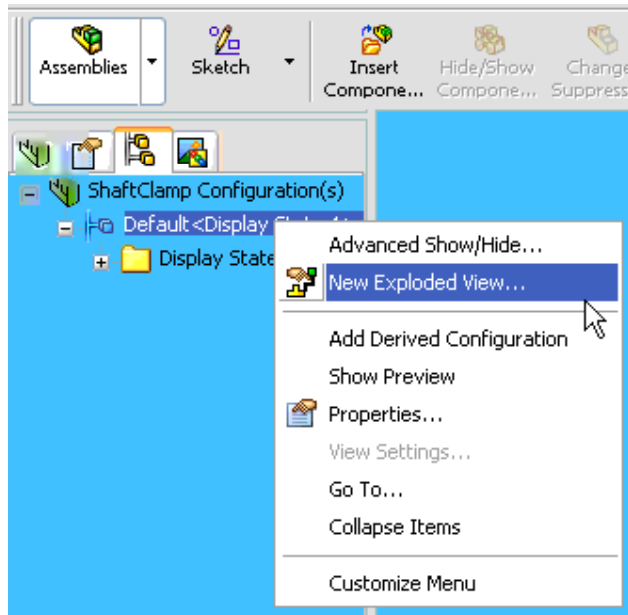
This way, we only have to create a view one time, then we can display it at any point with the click of a button.

Configuration Manager

Click the THIRD TAB that is with the Feature and Property Manager.

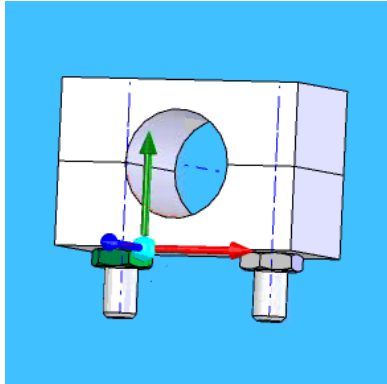


To create an EXPLODED view. RIGHT CLICK on DEFAULT. Choose NEW EXPLODED VIEW.



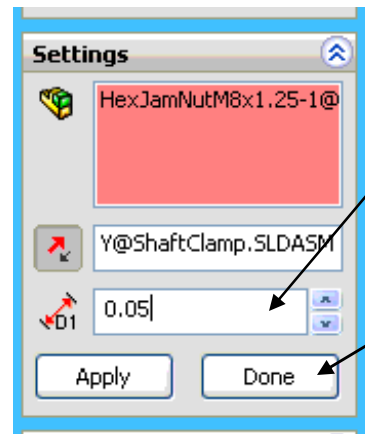
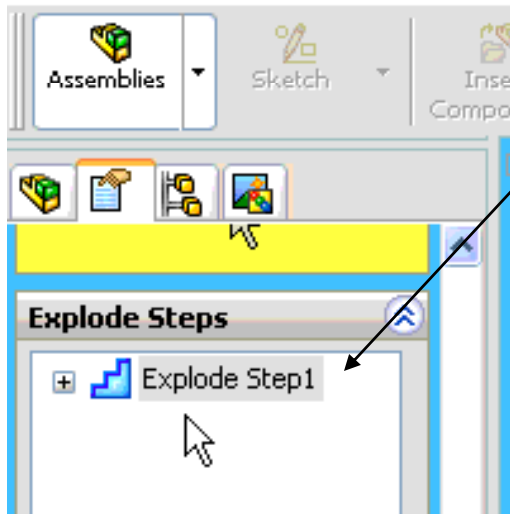
Property Manager will then open.

Creating an Exploded View

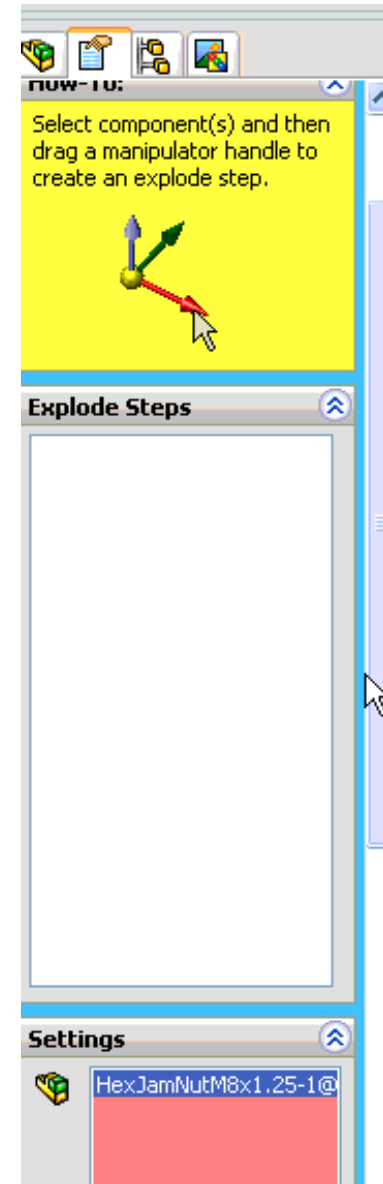


Click on one of the HEX Nuts. You will see it appear under settings. You will also see an axis. Click and Hold on any chosen direction to DRAG the part away from the main assembly. Click on the vertical arrow and drag the hex nut below.

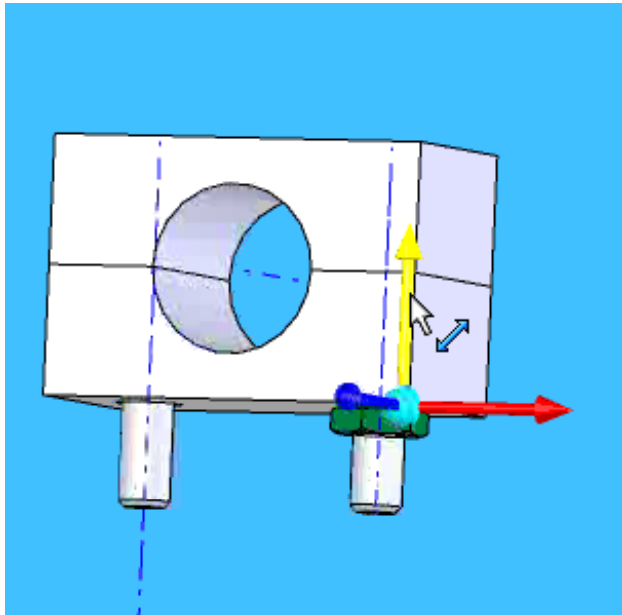
You will then see an “explode step”. Double click on this and change the distance to 0.05m.



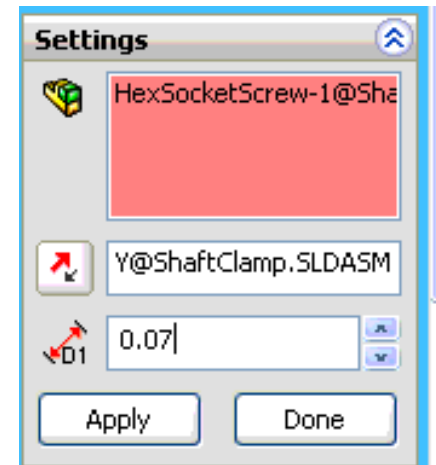
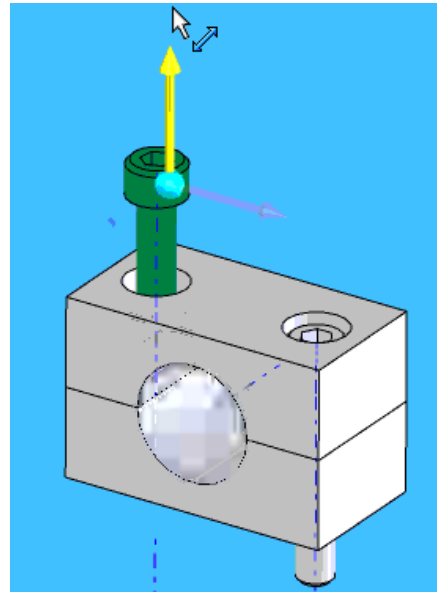
Click APPLY then DONE!



Creating an Exploded View

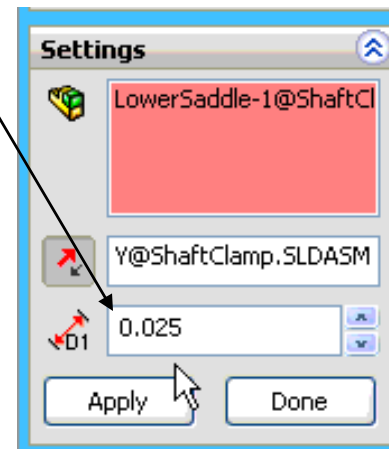
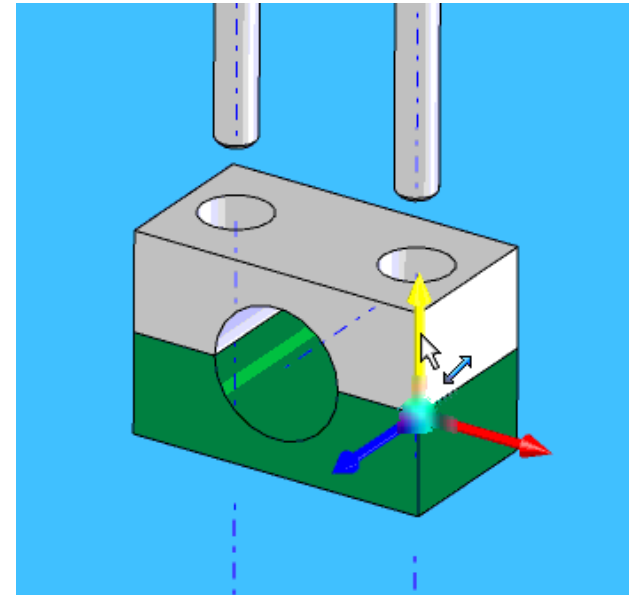
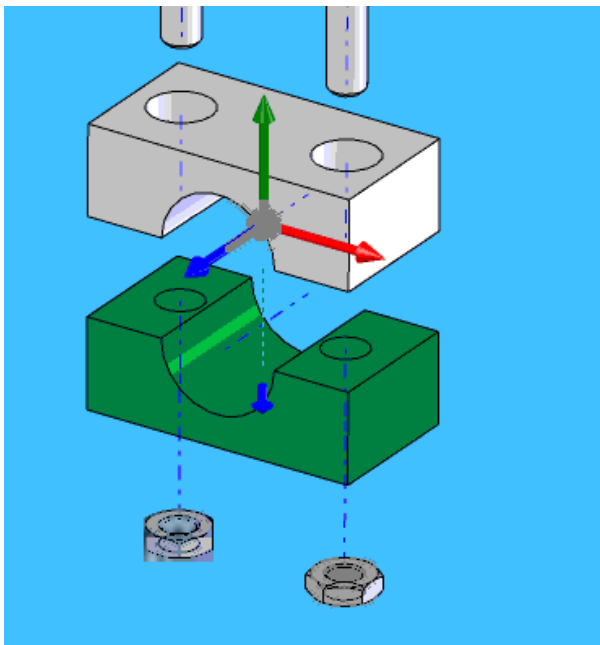


Repeat procedure for second hex nut then repeat for each hex screw. The screw should be pulled UPWARD and the distance set to 0.07m. Repeat for other screw.



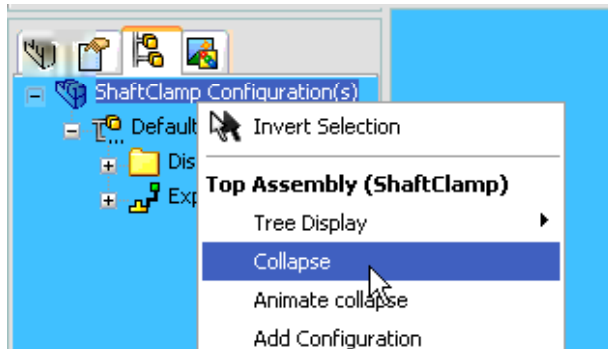
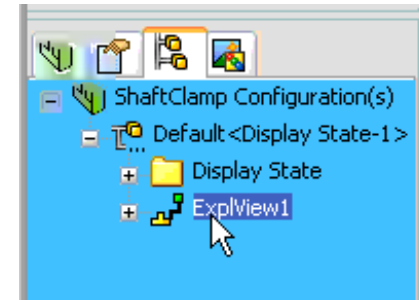
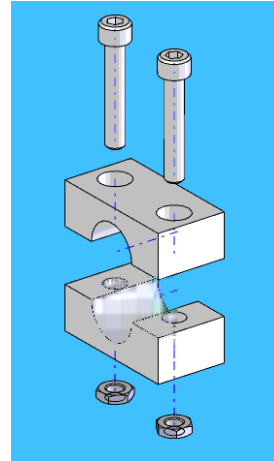
Creating an Exploded View

Now click on the lower saddle and move it downward 0.025 m.

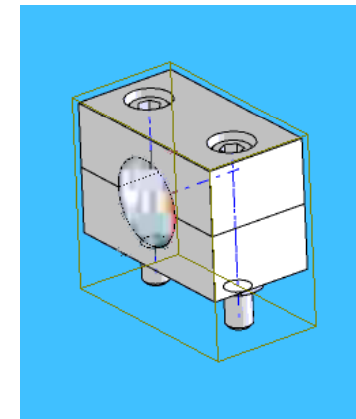


Creating an Exploded View

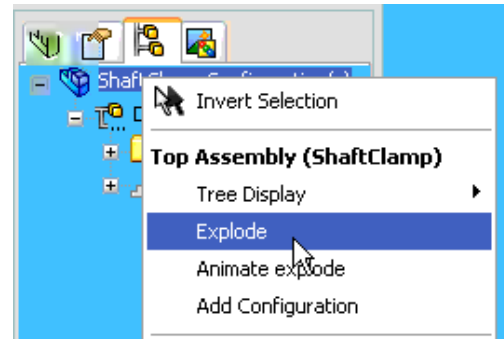
Click OK in configuration manager or hit OK to exit sketch. You should see after exiting a NEW “Explview1”



If you RIGHT CLICK on the original assembly, you can collapse the assembly.

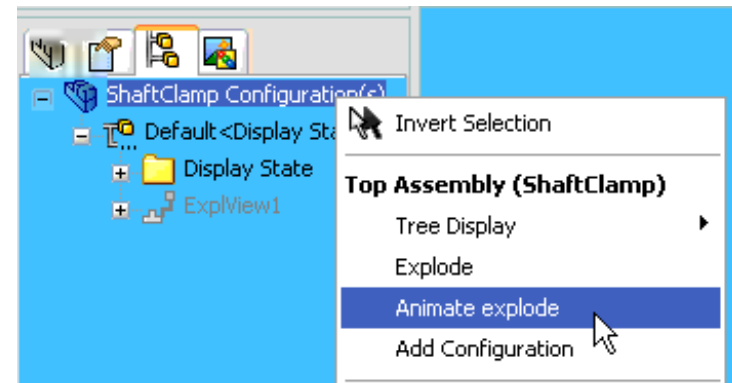


Likewise, if you RIGHT CLICK on the assembly again in configuration manager, you can EXPLODE the assembly.

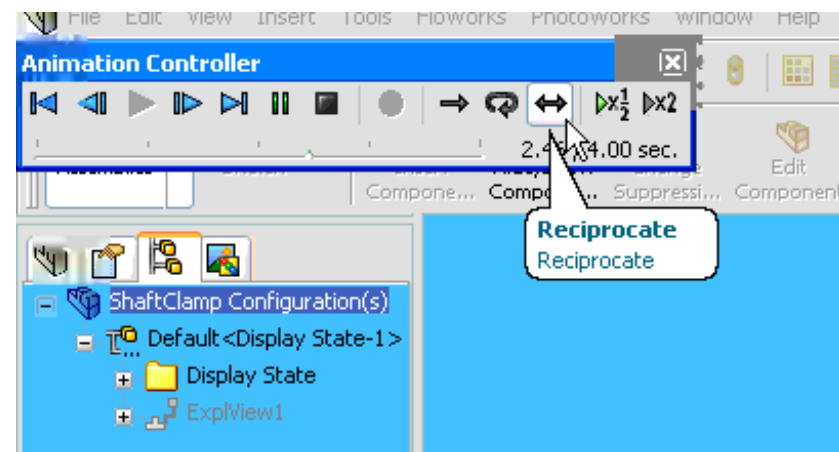


Animating Configurations

To animate the collapse/explode, right click on the configuration and choose **ANIMATE EXPLODE** or **COLLAPSE**.



You can have it continue to explode and collapse if the **RECIPROCATE** button is pressed on the Animation controller.



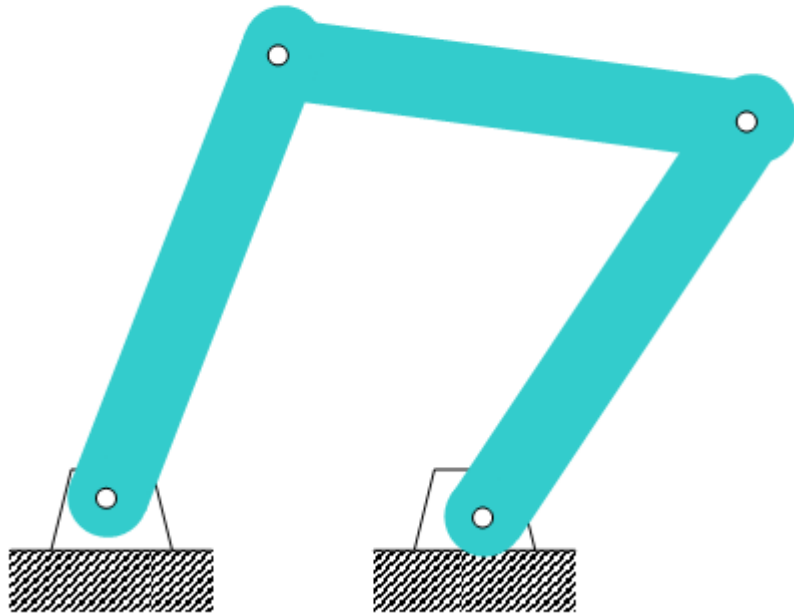
Kinematics in Solidworks

So far we have been making sure our assemblies are fully defined, however, we can also use SolidWorks to analyze motion of parts if we only partially constrain our assemblies.

This is very useful for design of kinematic systems where the motion is often complex

Kinematics in Solidworks

Lets looks at a simple 4-Bar assembly:



The 4-Bar linkage is an amazingly simple mechanism but the analysis is fairly complex (kinematics is a junior/senior level course).

However, we can look at the motion very easily by building a simple SolidWorks model.

Kinematics in Solidworks

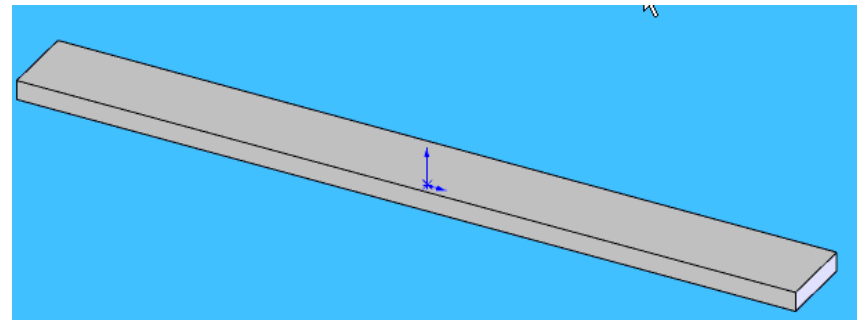
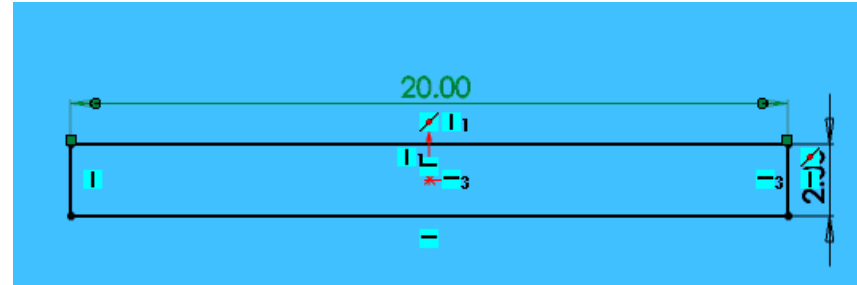
Let's begin by making a new part.
Change the units to the IPS system.

Choose EXTRUDED/BOSS Base and build from the TOP PLANE.

Make a rectangle around the origin and use the MIDPOINT of 1 vertical edge and 1 horizontal edge to add HORIZONTAL and VERTICAL relations with the origin.

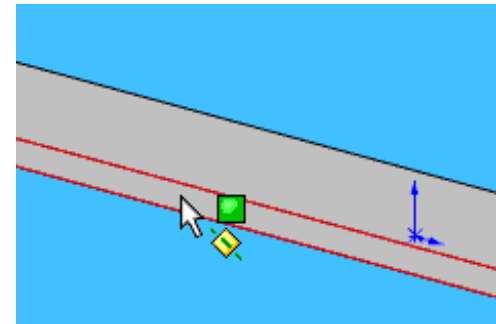
Dimension the horizontal edge as 20 in and the vertical edge as 2 in.

Exit the sketch and EXTRUDE the part to 0.50 in.

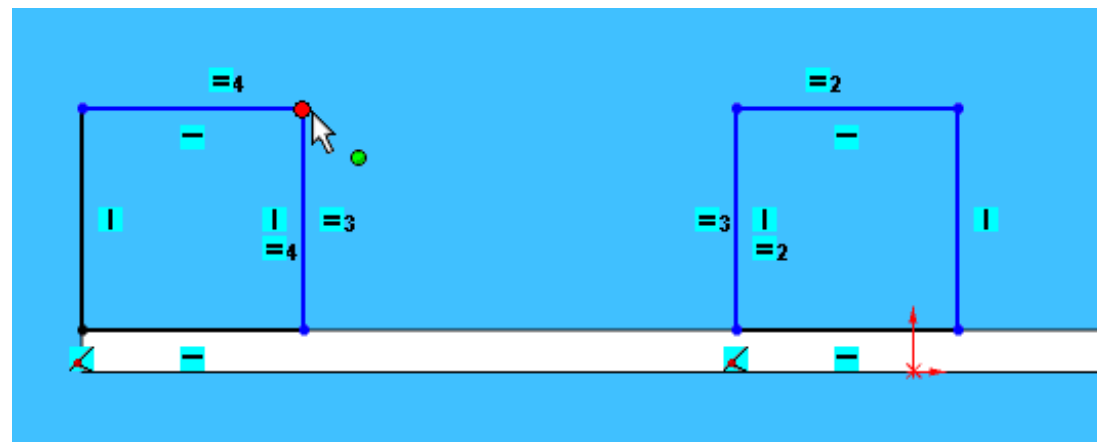


Kinematics in Solidworks

Now lets make another extrude on the base we just made. Choose EXTRUDE BOSS/BASE. Choose the front edge and then change your view to “normal to”.

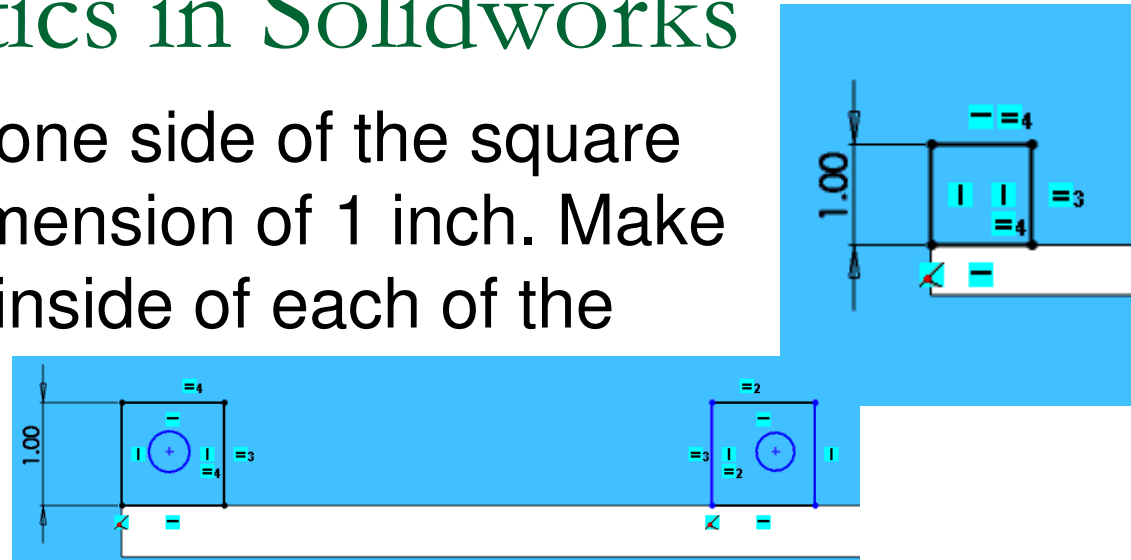


Make 2 rectangle with one coincident with the base's top left edge. Make sure the second rectangle's bottom is also coincident with the base's edge. Click 1 vertical and 1 horizontal edge of EACH rectangle using CTRL and add an EQUAL relation.

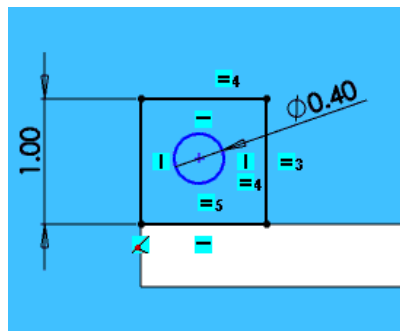


Kinematics in Solidworks

Dimension one side of the square with a dimension of 1 inch. Make 2 circles inside of each of the squares.



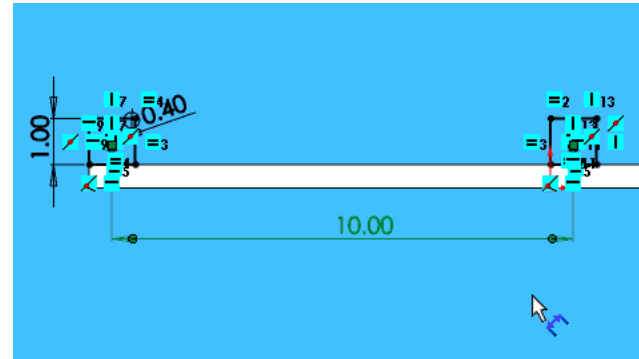
Click on each circle using CTRL and add an EQUAL relation. Then dimension one of the circles to have a diameter of 0.4 in.



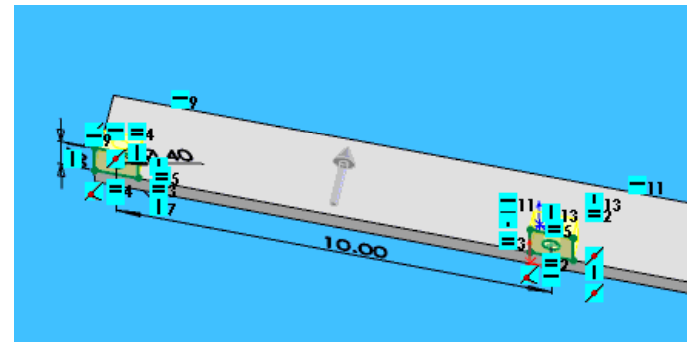
Center EACH circle inside the square by right clicking on a vertical and horizontal side of the square. Choose MIDPOINT. Using CTRL, click the origin of the circle and add the appropriate horizontal or vertical relation.

Kinematics in Solidworks

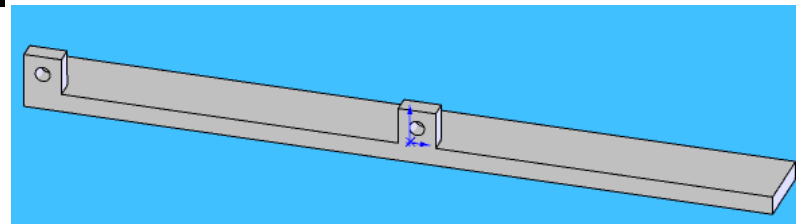
Now dimension the distance between the center of the circle to be 10 inches.



Click OK to exit the sketch, then EXTRUDE to 0.5 inches. You may need to FLIP the direction so that you are extruding inward.

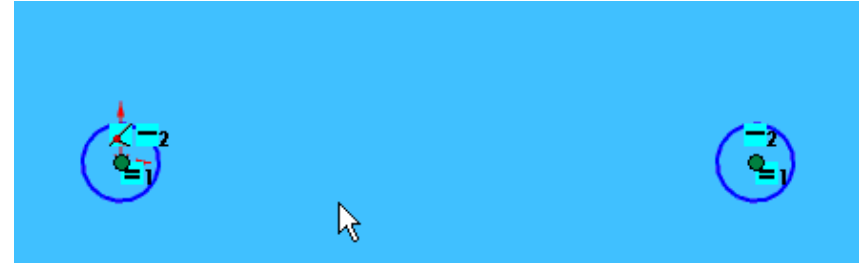


Save part as 4bar_base

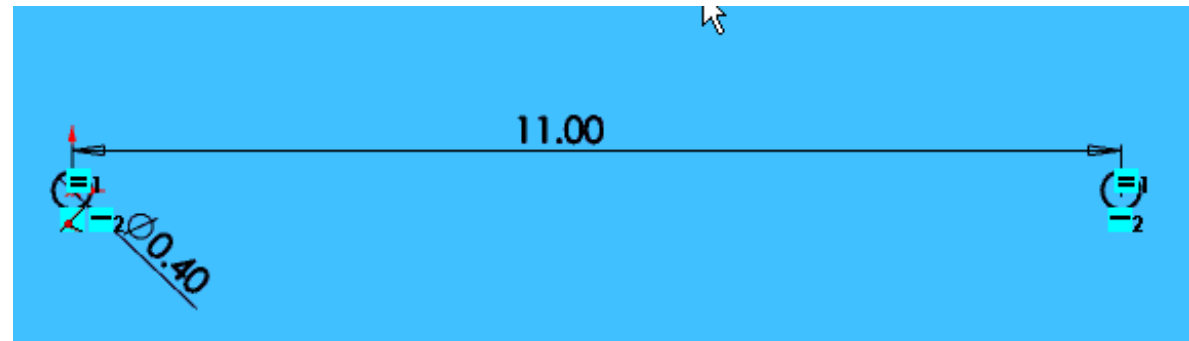


Kinematics in Solidworks

Choose NEW part. Change units once again to IPS. Then Extrude Boss/Base. Choose FRONT PLANE. Make 2 circles. Add EQUAL and HORIZONTAL relations to each.

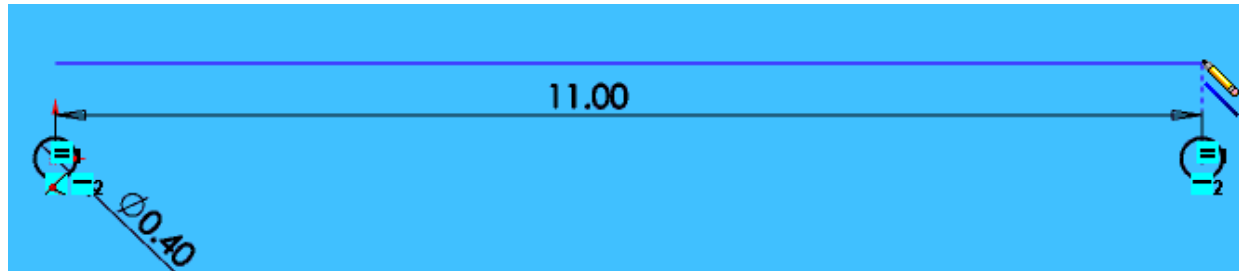


Dimension one of the circles with a diameter of 0.40 inches. Dimension the distance from center to center to the circles to be 11 inches.

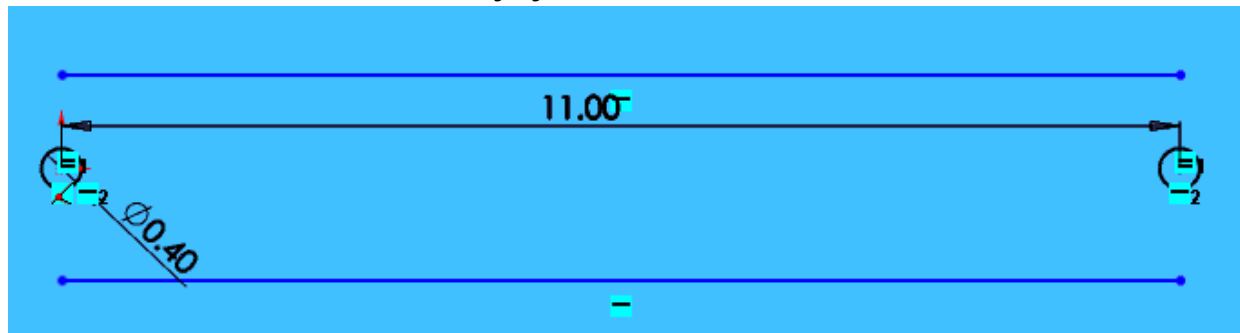


Kinematics in Solidworks

To make the bar, choose LINE. Start by placing the cursor ABOVE the left circle yet over the center of the circle. You will see a reference vertical line show up. Begin there and go horizontally till you are above the right circle. Look for the vertical reference line again to know when to stop.

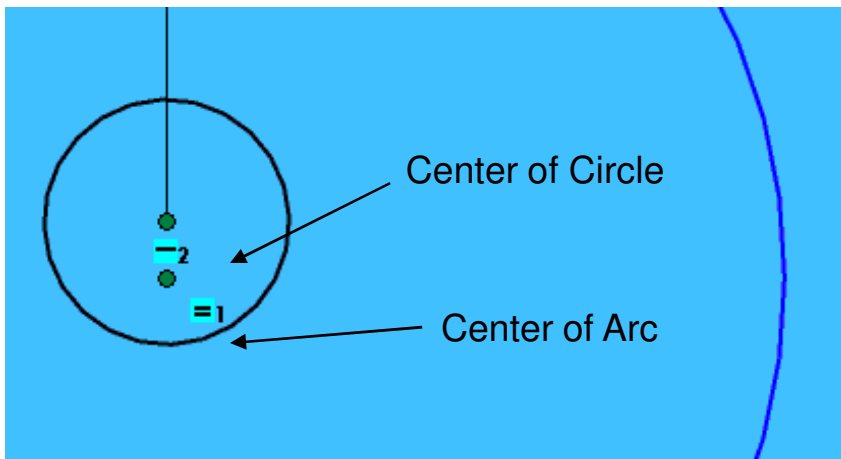
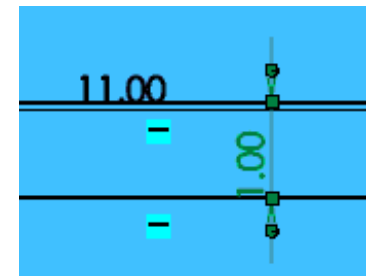
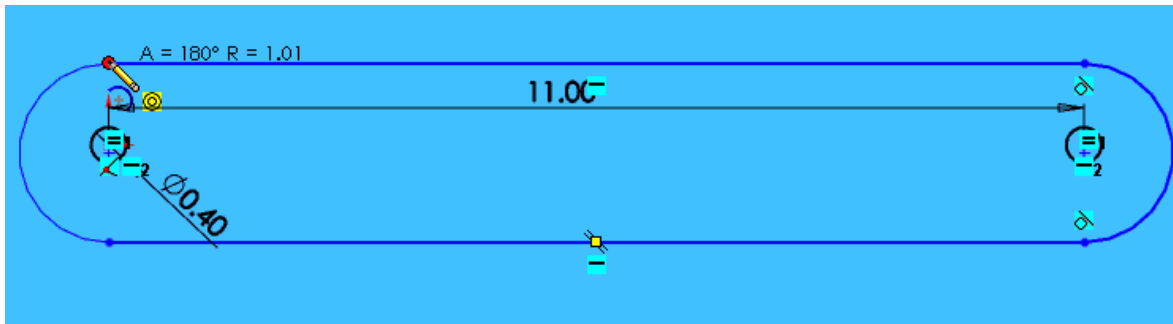


Draw another line the same way you did before YET draw it under the circles.



Kinematics in Solidworks

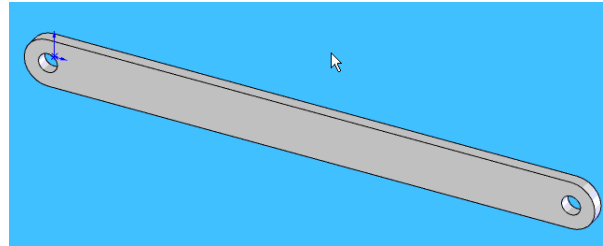
Select TANGENT ARC and draw an arc from the ends of the top line to the ends of the bottom line.



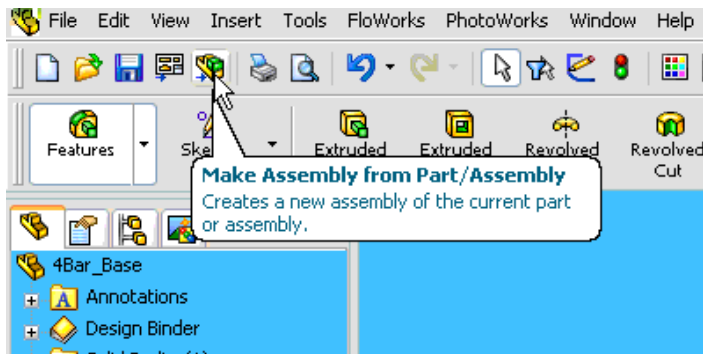
Now add a MERGE relation between the center of each arc and the center of each circle. Lastly, dimension the width of the bar to be 1 inch. Then exit the sketch.

Kinematics in Solidworks

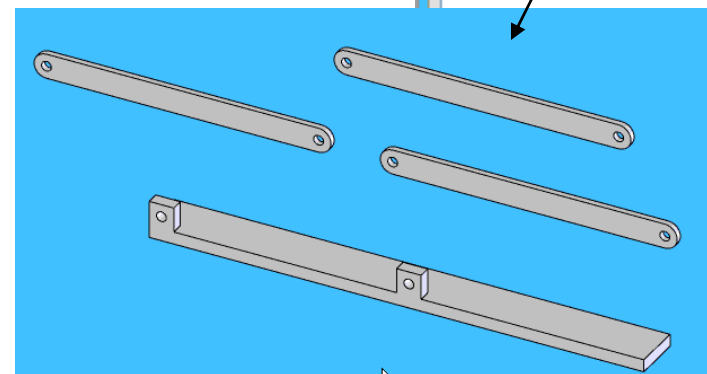
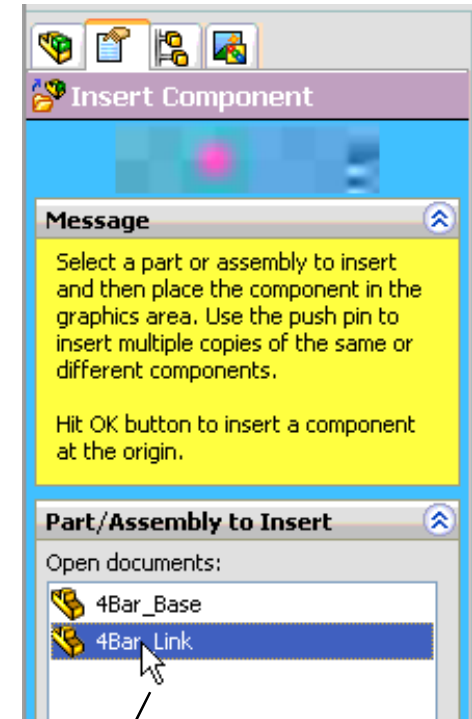
Extrude the part 0.25 inches.
Save as:
4bar_link



View the BASE window or the FIRST part you made and choose MAKE ASSEMBLY FROM PART. Make sure units are in IPS.



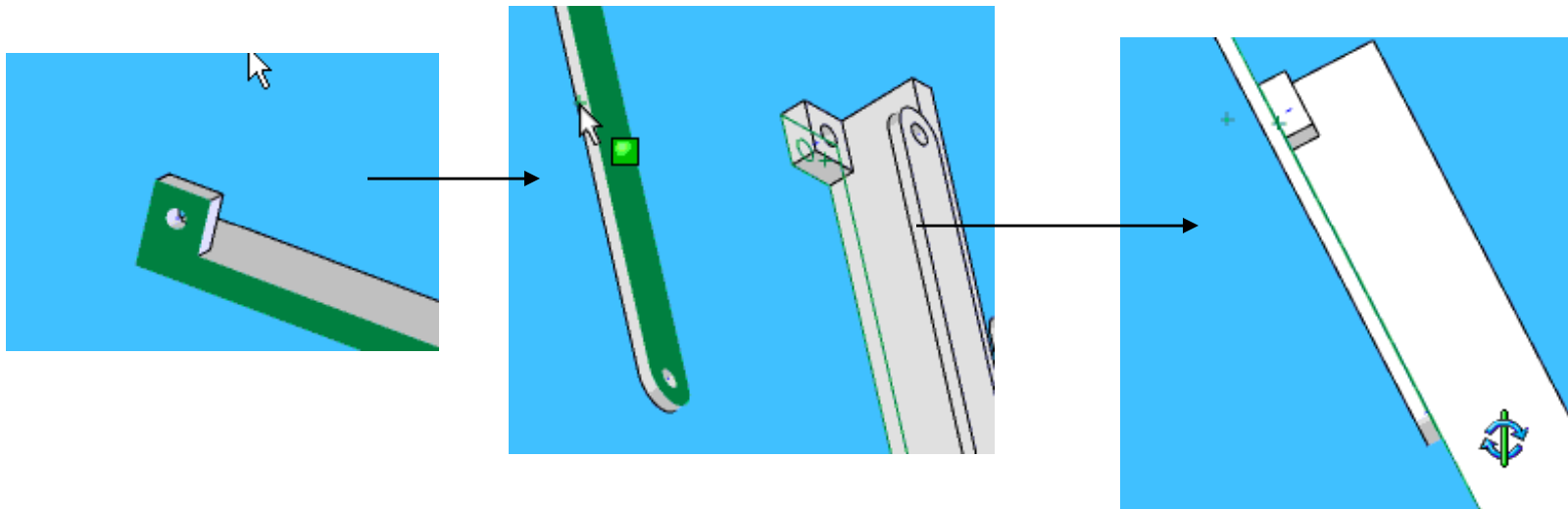
Insert your BASE first, then insert 3 LINKS into the viewing plane.



Kinematics in Solidworks

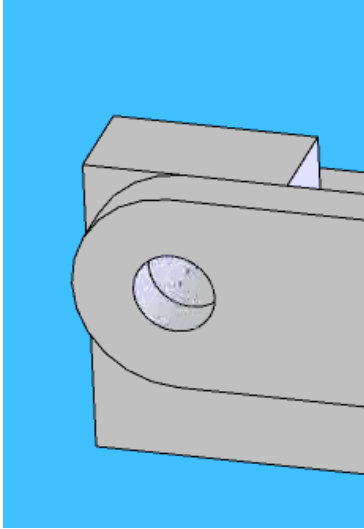
Before we MATE then parts together. Turn on the TEMPORARY AXIS, by clicking the small glasses in the viewing area.

Choose MATE. Click on the front surface of the base and the back surface of one of our links.

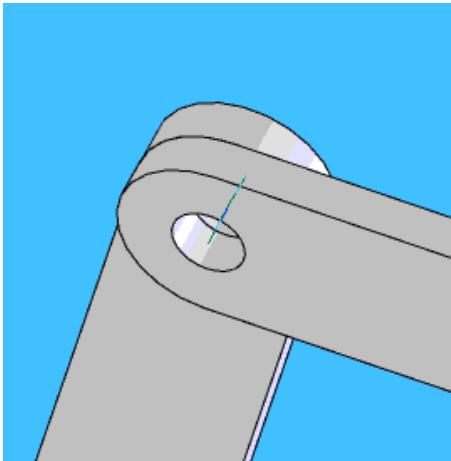
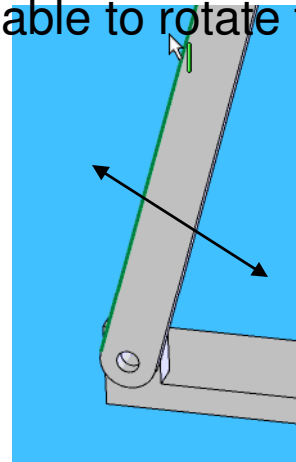


This will give us alignment as if we were looking down from above.

Kinematics in Solidworks



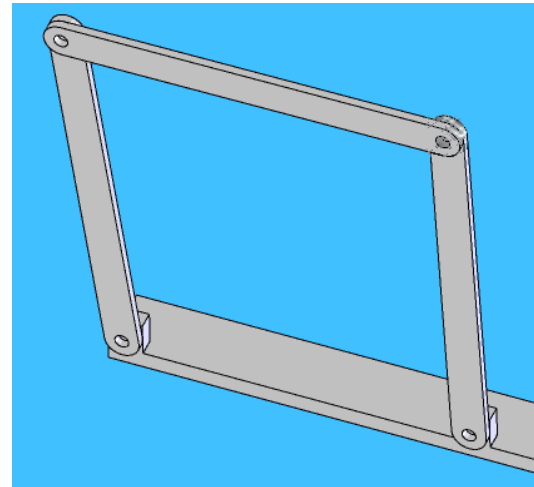
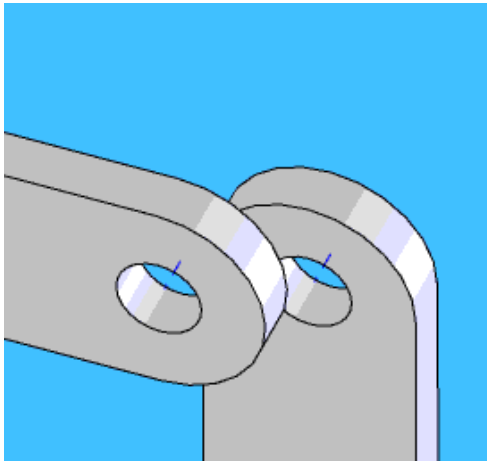
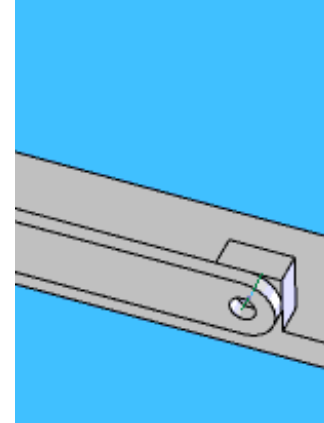
Click MATE. Choose the axis inside the left circle on one bar and the axis inside the left circle on our base. Click OK. If you click on the right side of the bar and hold the mouse button, you should be able to rotate the bar around.



Click MATE, choose the BACK side of the second link and the FRONT side of the first link to align them. Then mate their axes together as shown.

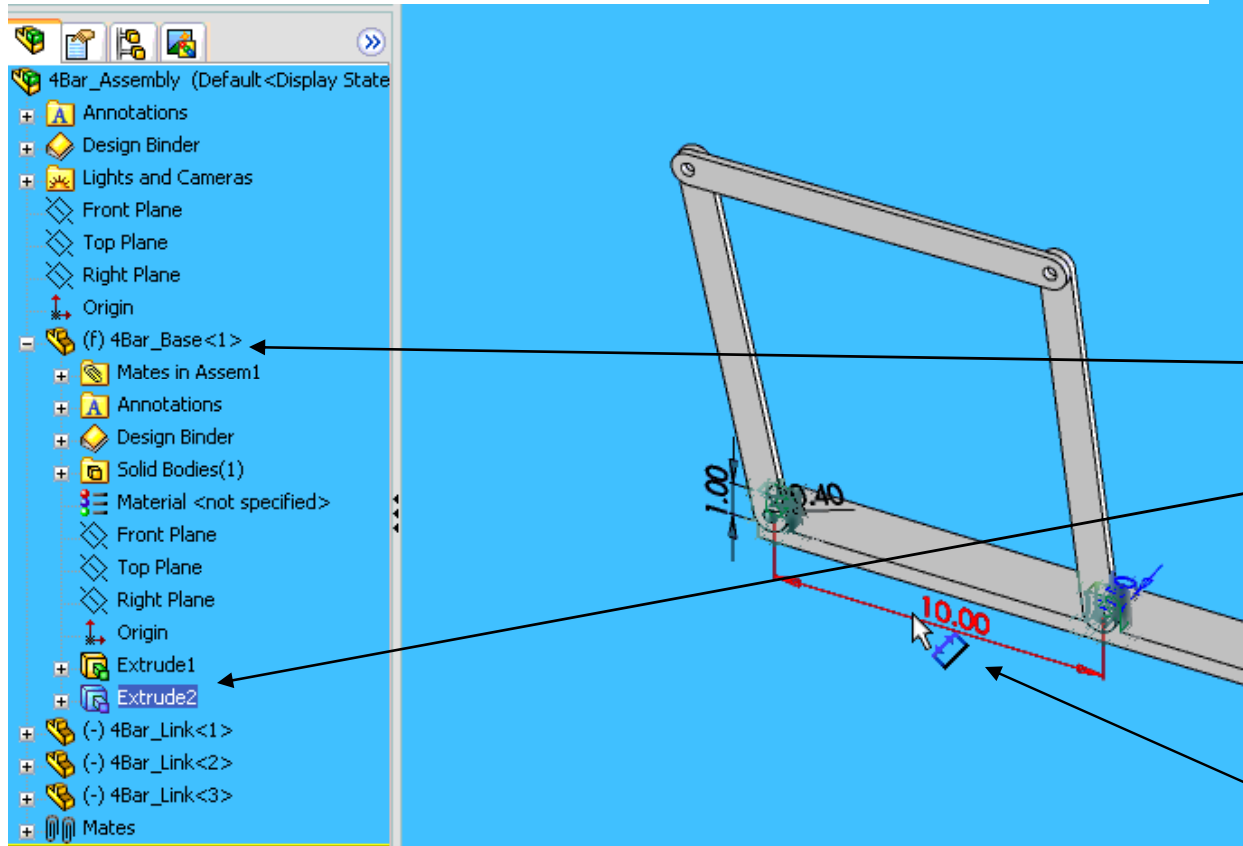
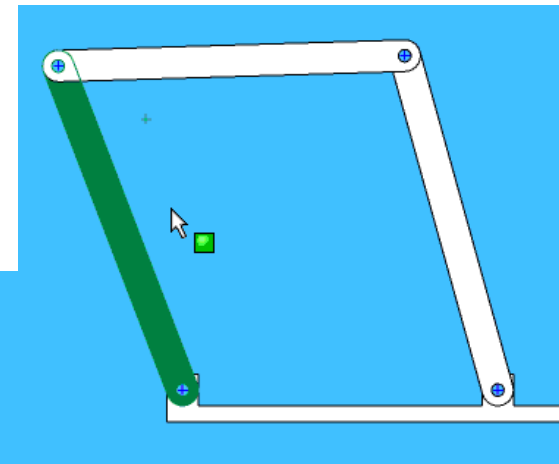
Kinematics in Solidworks

Make sure you click OK between each MATE. Click MATE. Now mate the BACK side of the third link to the FRONT side of the base. Mate the axis of each circle as well. Move the 2nd and 3rd link CLOSE to each other. Then MATE the axis of each circle together. It is always a good idea to bring mates close to each other BEFORE mating them together.



Kinematics in Solidworks

Click on the left side to rotate the entire system. Save file as: **4bar_assembly**

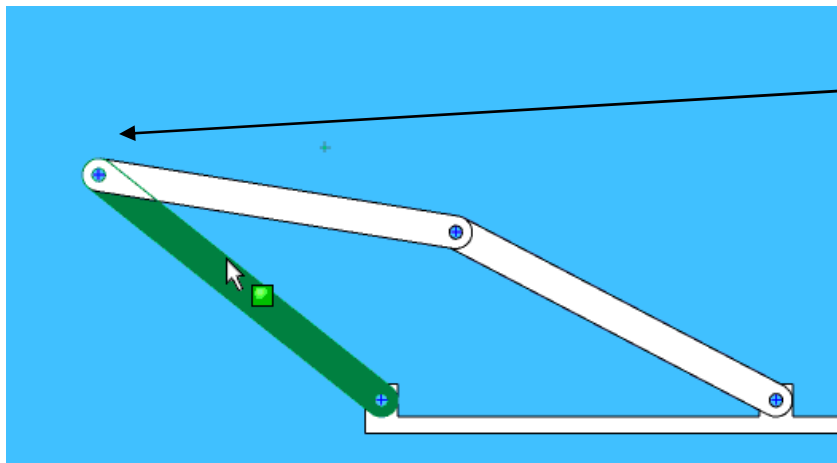
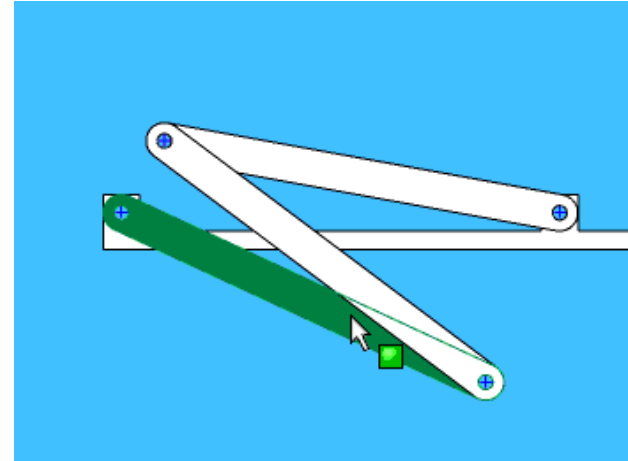


Double click on the **BASE** in feature manager then click on the second extrude to edit the feature. Change the distance between the circles from 10 inches to 12 inches.

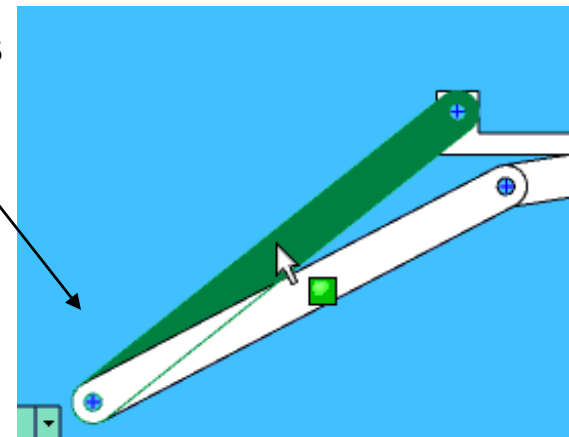
You may need to click the stop light button to RE-BUILD.

Kinematics in Solidworks

Change your view to FRONT, then try to move the system now. You should discover that all the mates still exist, yet the motion of the assembly is different than before.



Stall Points



Kinematics in Solidworks

Try now changing the length of the LINK from 11 to 8. Be sure to click RE-BUILD. Notice the entire assembly changes.

Can you think of anything that use the 4 bar linkage idea? **Hint:** It is on your CAR!
