Lab Assignment #9: Mechanism Design & Animation ME 170

In this lab you will learn to design a mechanism using Fusion's "Top-Down Design" functionality and you will create a mechanical mechanism called a "Slider-crank". Let me explain both these terms:

- **Top-Design Design:** Are you starting with a blank screen? Or do you have a library of components? This is the top-down versus the bottom-up design question. Most designs use both. You start by assembling one or more components previously designed (bottom-up design), then you design a new component from, or to match, these components directly in the CAD modeler's design space (top-down design).
- **Slider-crank:** A slider-crank mechanism is a four-link mechanism with three revolute joints and one prismatic, or sliding, joint. The rotation of the crank drives the linear movement the slider, or the expansion of gases against a sliding piston in a cylinder can drive the rotation of the crank (as in a car's internal combustion engine).

You will animate the mechanism in Fusion 360 and, optionally, study the kinematics using dynamic simulation with Inventor (another Autodesk product).

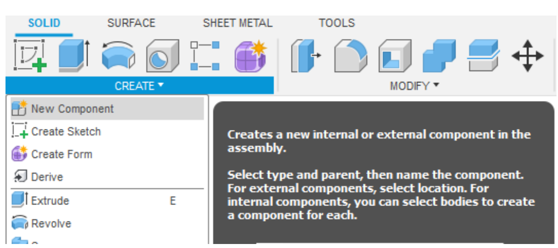
Part I. Create a Slider Mechanism in Fusion

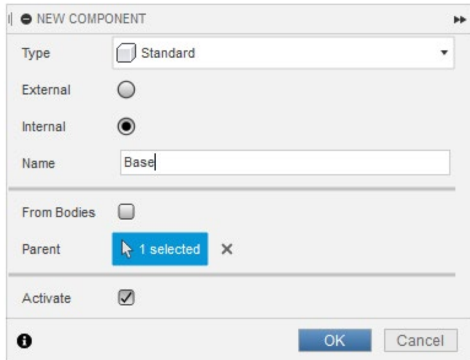
1) Create the Base Part

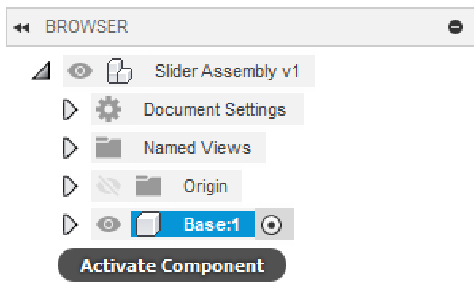
   a) Start Fusion, navigate to your ME170 Project and create a new folder named Lab 9

   b) Save a new file in this folder named "Slider Assembly"

   c) From the Create dropdown, select the New Component tool. This lets you create parts within the existing file (as opposed to creating them separately and then combining them). Enter "Base" as the Name of the part and press OK to create it.

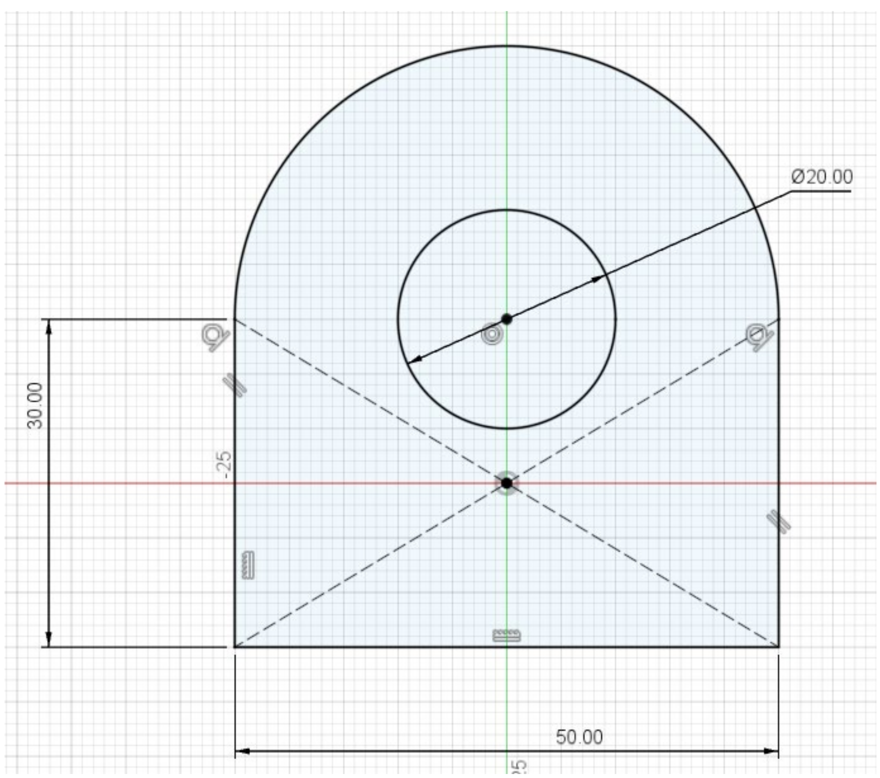






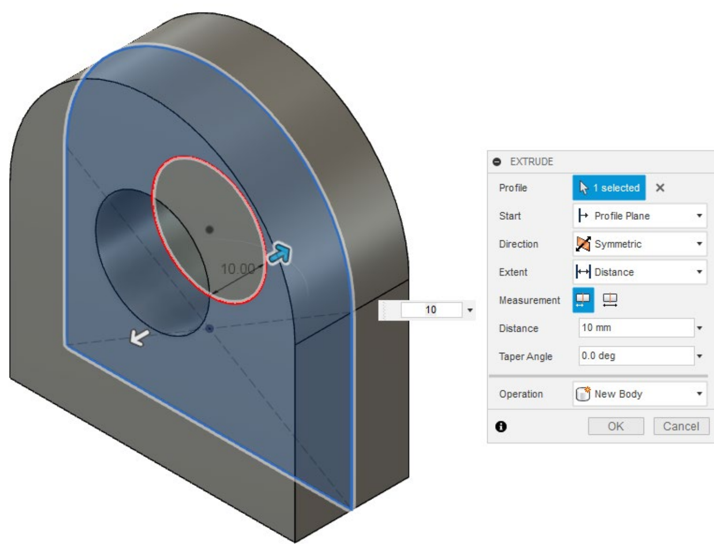
Note: The black dot in the Browser shows which component you are working on- this is important to keep track of.

d) Begin a Sketch on the XZ plane. Create the following profile:

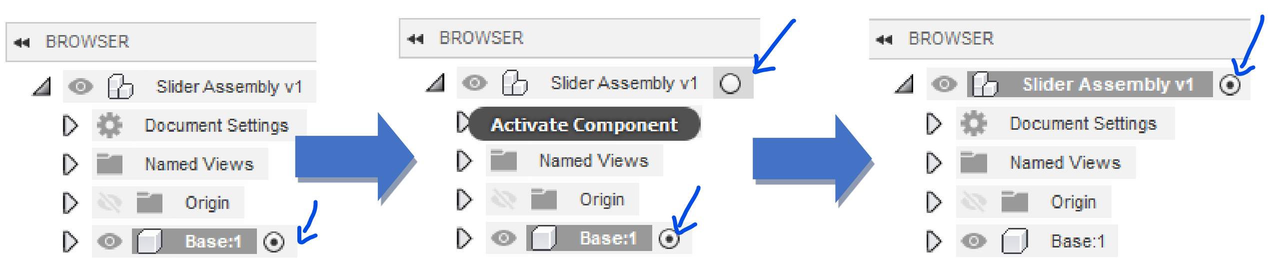


Recommended Order: Create a Center Rectangle at the origin with dimensions 30mm x 50 mm. Then create a Tangent Arc connecting the top corners of the rectangle. Then create a circle with diameter 20 mm with the same center point as the arc.

e) Finish the Sketch and Extrude the profile (excluding the center circle) symmetrically for a Distance of 10 mm (20 mm total).



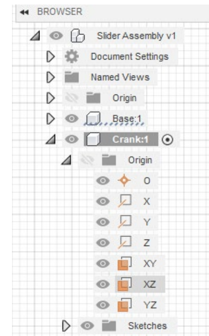
f) Activate the highest level by clicking the Activate Component option next to "Slider Assembly" in the browser. This means you are working within the overall assembly, not the specific component.



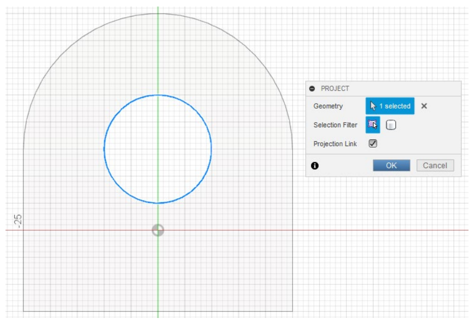
2) Create the Crank

a) While working in the highest layer, select New Component again, name the part "Crank", and press OK. The Base should be visible, but translucent. As you will see, we can reference its geometry, making the dimensioning and constraining of future parts significantly easier.

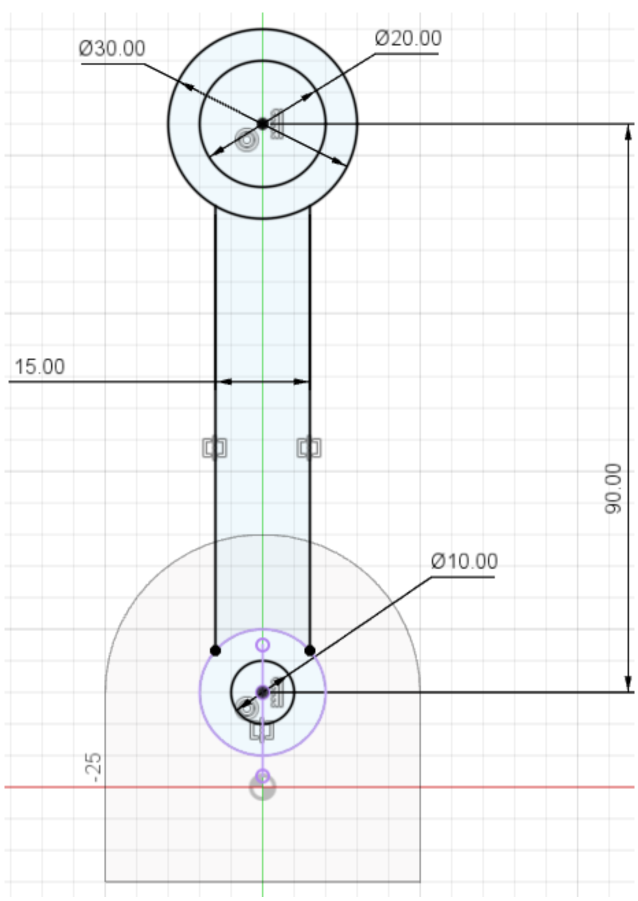
b) Create a Sketch on the XZ plane, taking care not to select one of the faces of the Base part. If needed, you can select from the Origin folder under Crank in the Browser.



c) Use the Project tool (hotkey p or in the Create dropdown) and select the inside circle of the Base part, then press OK.

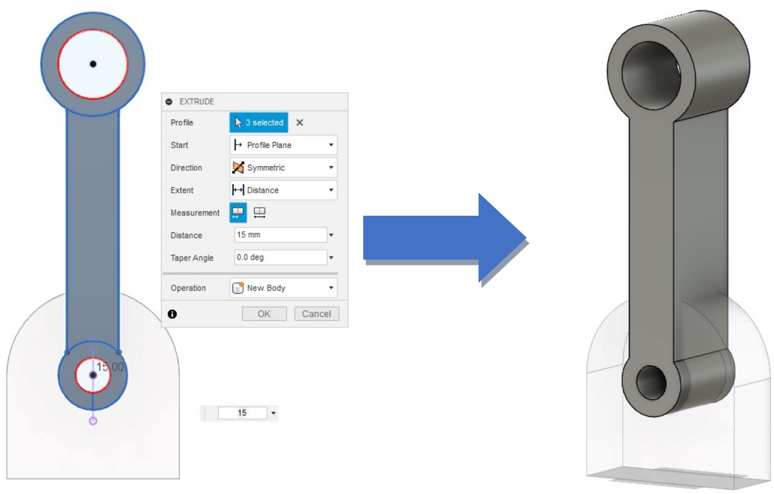


d) Create the following profile:

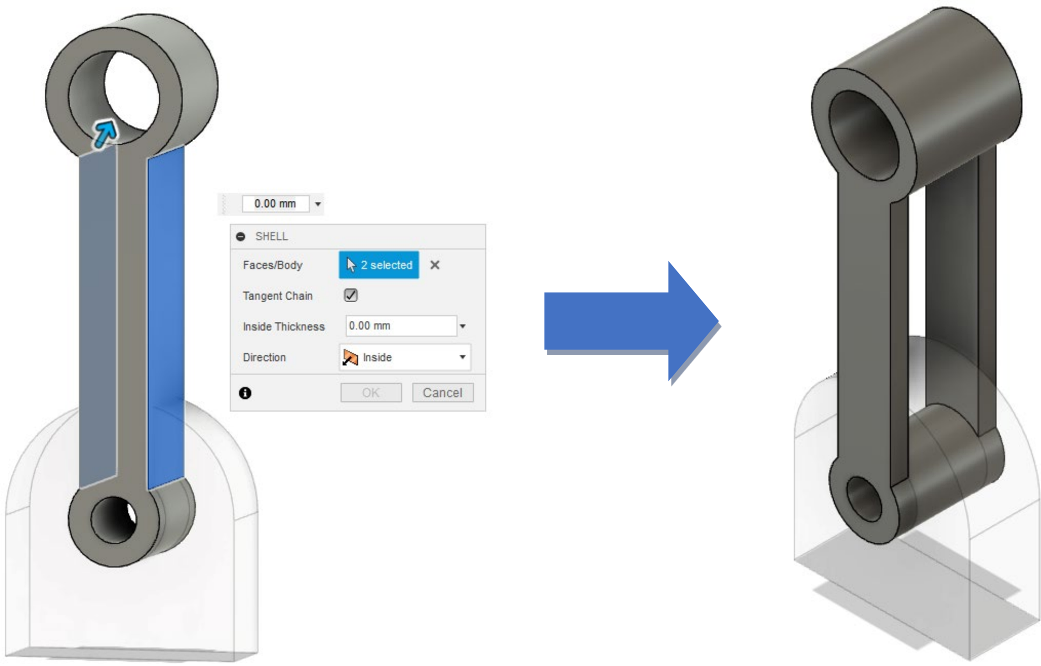


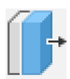
Recommended Order: Sketch two concentric circles (diameter 20 mm and 30 mm), then constrain them to be 90 mm directly above the projected circle. Sketch two vertical lines connecting the 30 mm circle and the projected circle. Make the lines symmetric about the vertical (Z) axis and Dimension the distance between them to 15 mm. Add a 10 mm diameter circle inside the projected circle.

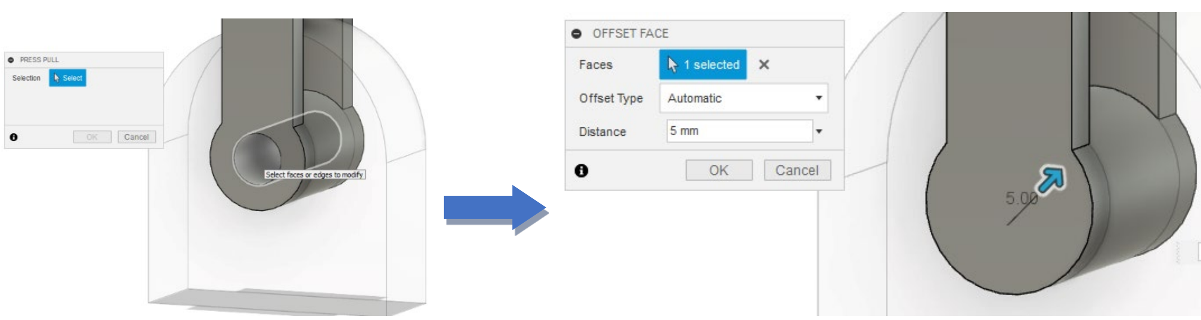
e) Press Finish Sketch and Extrude the profile (excluding the inner concentric circles) symmetrically for a Distance of 15 mm (30 mm total).



f) Use the Shell tool and select the two rectangular side faces of the part. Enter 5 mm as the Inside Thickness and press OK.



g) Select the Press Pull tool  (the first tool in the Modify toolbar) and select the inner face of the smaller hole. Enter 5 mm as the Distance to pull by, and press OK. This will close the empty space.

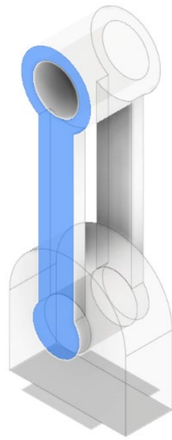


h) Activate the highest layer (Slider Assembly) and if you like, you can check interferences. Since the sketch was based on a projection of the first part, the two parts will fit perfectly with no interference.

3) Create the Connecting Rod

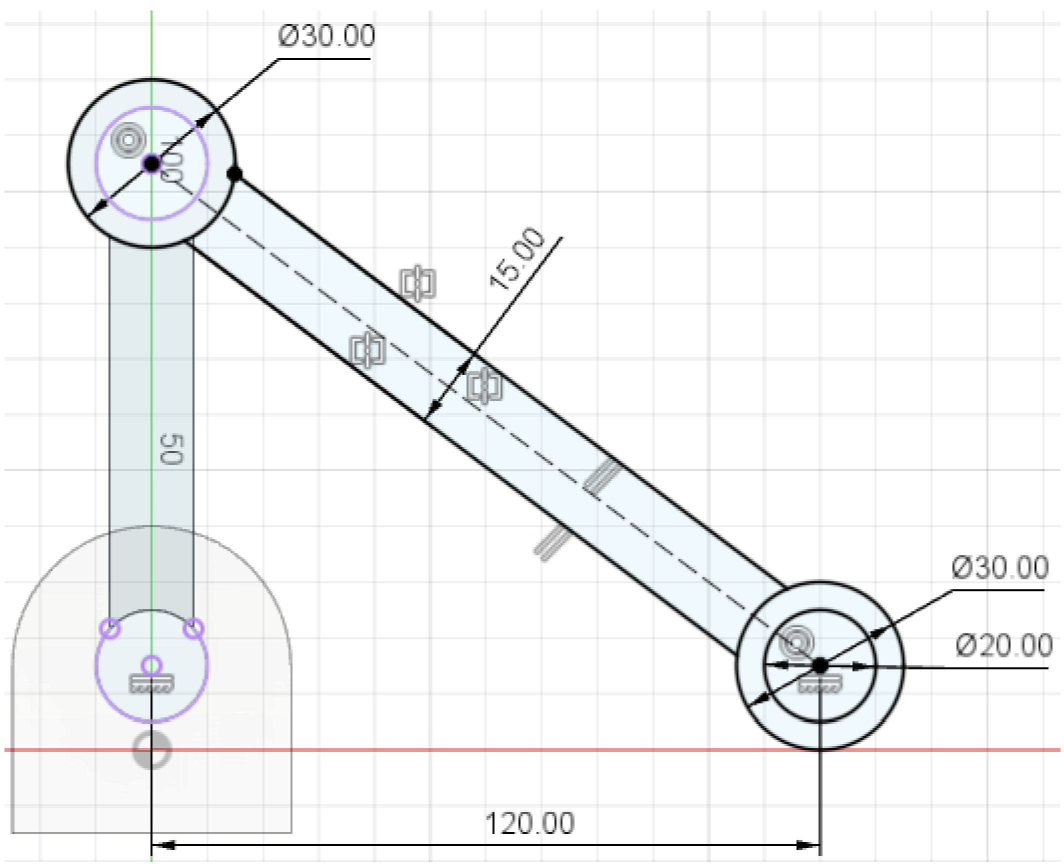
a) While working in the highest layer, select New Component again, name the part "Connecting Rod", and press OK. Both previous parts should be translucent, as the new part should be Active.

b) Create a Sketch on the Front face of the Crank:



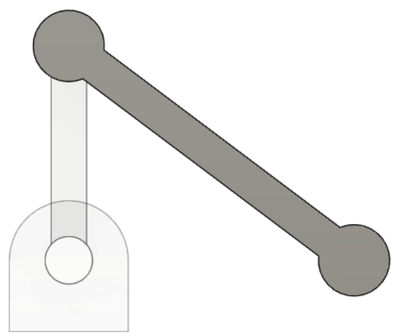
c) Project both the top and bottom circles of the Crank to define their center points.

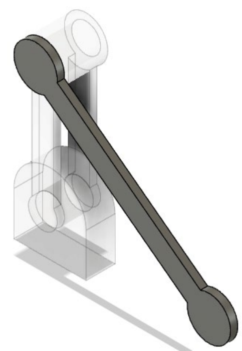
d) Create the following profile:



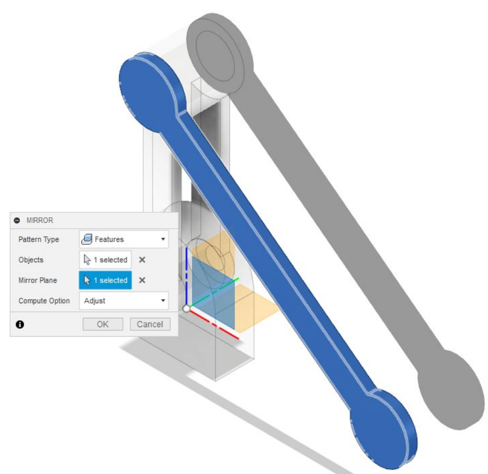
Recommended Order: Create a Circle with diameter 30 mm, then constrain it be horizontally alligned with the bottom circle of the Crank and 120 mm away. Using the same center point, create a Circle with diameter 20 mm. Create another 30 mm circle centered at the top circle of the Crank. Connect the two circles with two lines, and connect the two centerpoints with a Construction line (press X to toggle construction). Constrain the two lines to be symmetric about the Construction line, parallel to each other, and 15 mm apart.

e) Extrude the whole profile 5 mm (making sure the direction points away from the Crank).

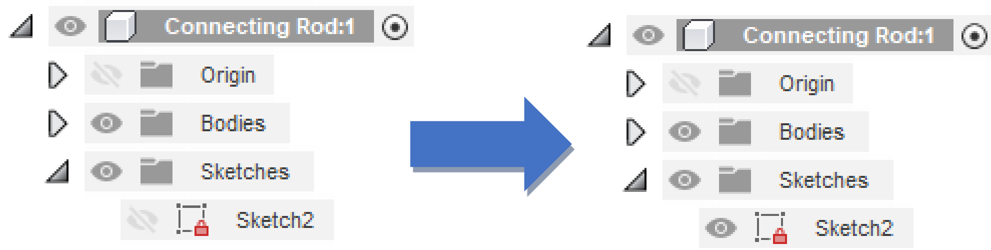




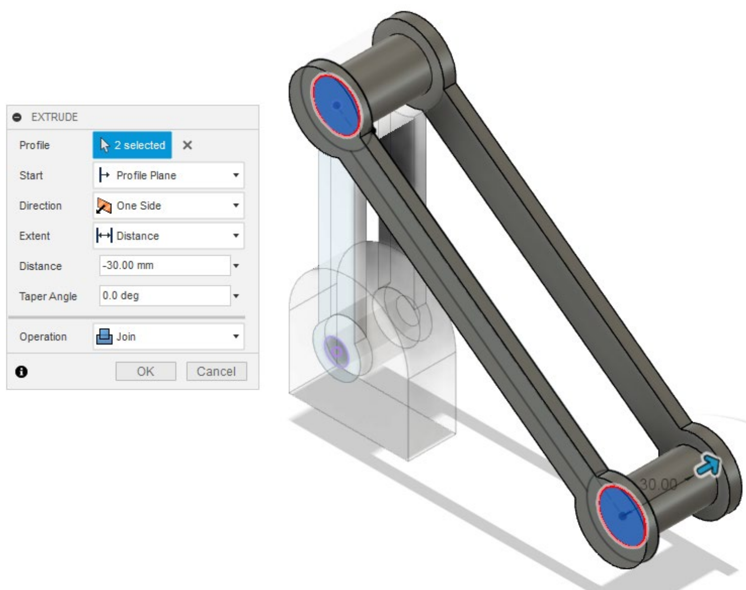
f) Mirror the Extrusion feature across the XZ plane. Make sure the Mirror Pattern Type is set to Features and you can select the extrusion from the timeline at the bottom-left.



g) In the Browser, open the Sketches folder under Connecting Rod and toggle visibility on the sketch by pressing the eye next to it. This allows you to create multiple features based on the same sketch.



h) Extrude the two inner circles sketched so that they connect the two existing Extrusions. The Distance is -30 mm, or you could change the Extent to "To Object" and select the opposite face.



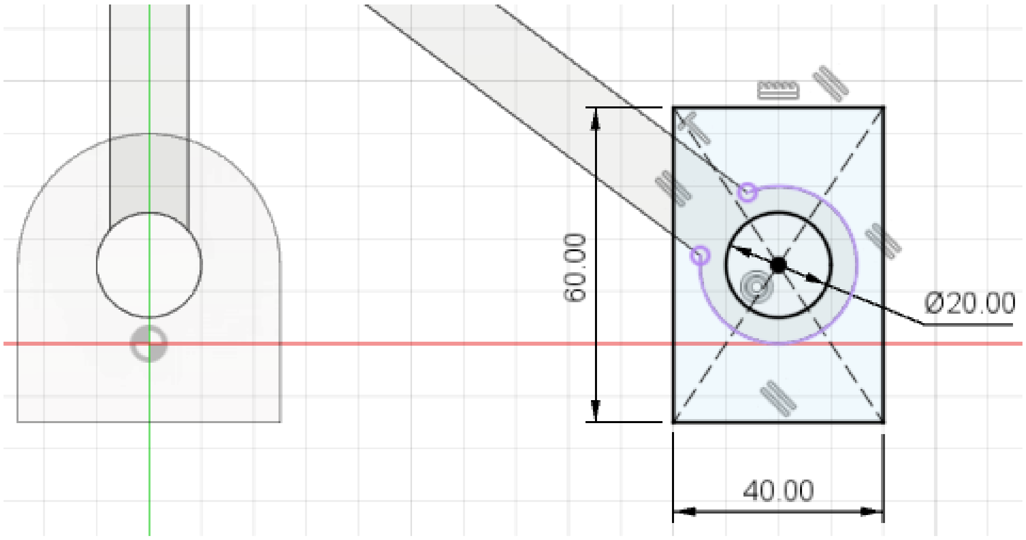
i) Untoggle visibility of the sketch and Activate the highest layer of the Assembly.

4) Create the Slider

a) While working in the highest layer, select New Component again, and name the part "Slider".

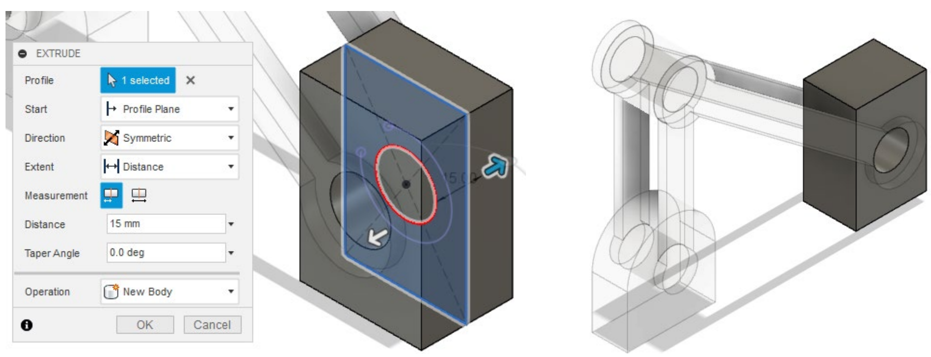
b) Create a Sketch on the XZ plane and Project the bottom-left circle (the one not connected to an existing part) in the Connecting Rod.

c) Create the following profile:



Reccomended Order: Create a Circle with diameter 20 mm at the centerpoint of the projection, then create a Center Rectangle at the same point with dimensions 60 mm x 40 mm.

d) Finish Sketch and Extrude the profile (excluding the inner 20 mm circle) symmetrically for a Distance of 15 mm (total 30 mm).

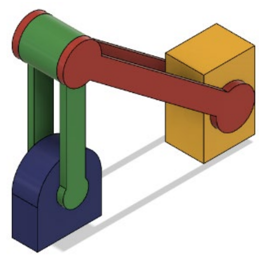


e) Activate the highest layer and make sure to Save.

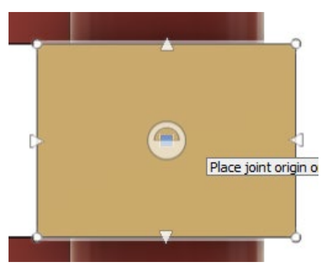
5) Define the Mechanism

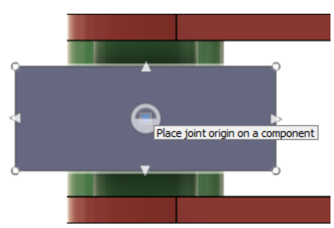
a) Right click on the Base in the Browser on the left, then choose Ground. A pin should appear next to the name.

b) Under the Modify dropdown, select the Appearance tool (hotkey A). In the Appearance popup, open the Paint folder and then the Glossy folder. Click and drag a unique color onto each of the 4 components to make them clearly distinguishable. When finished, press the Esc key to close the popup. The colors chosen are just an example:

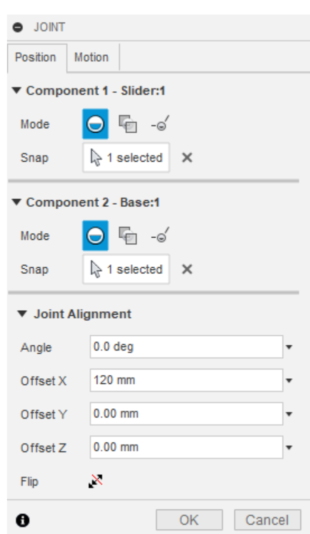


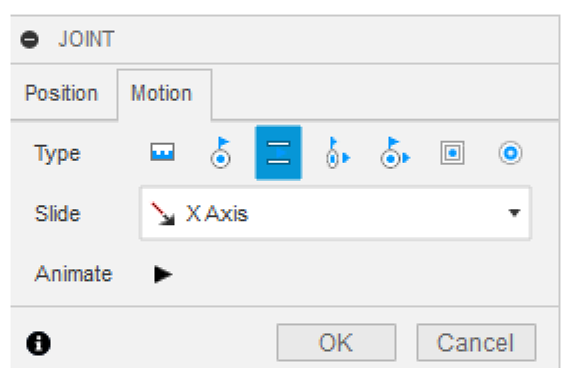
c) Select the Joint tool and then the centerpoints on the underside of the Base and Slider (do not press OK yet).



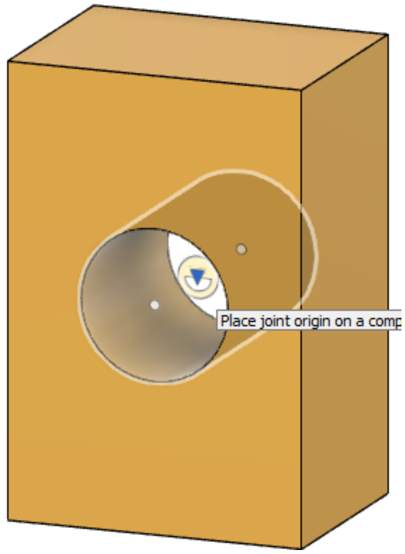


d) Make sure Flip is not selected (not highlighted blue) and set the X offset to 120 mm. Then, click Motion in the top right and change the Type to Slider, setting the Slide Direction to X Axis. Then press OK.



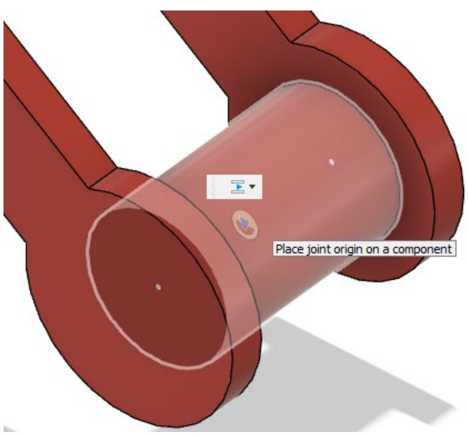


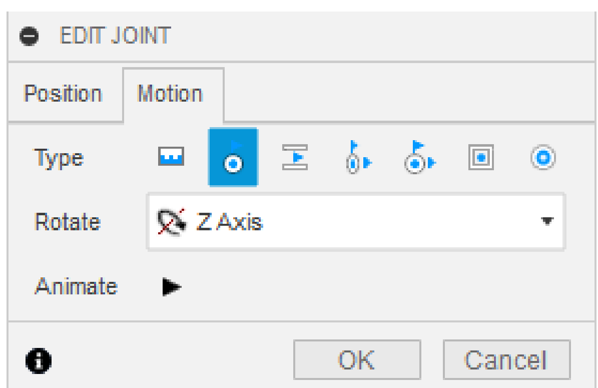
e) Toggle the visibility of the Connecting Rod. Create a new Joint, and for the first Component select the middle of the hollow cylinder in the Slider:



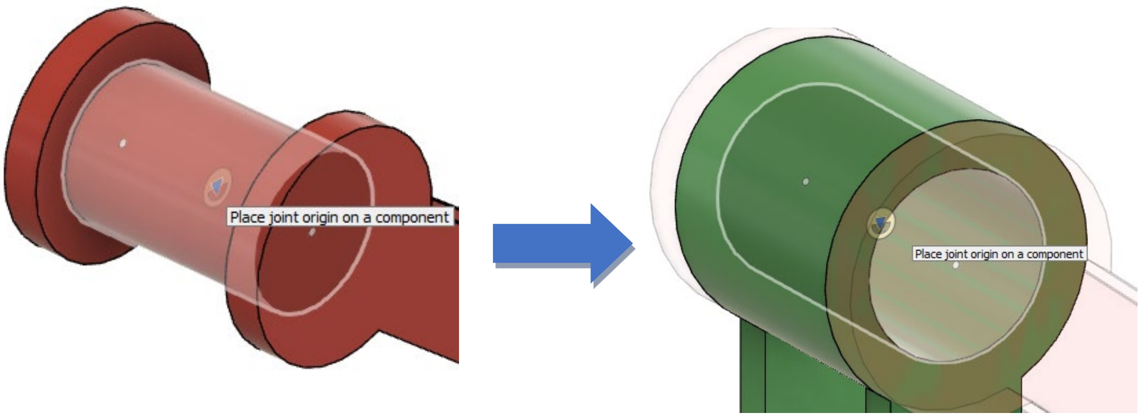
Note: If you have trouble selecting the center, on Windows hold the Ctrl key when the cylinder is highlighted, on Mac click once when the cylinder is highlighted but the cursor is not snapped on a point to select.

f) Toggle visibility of the Connecting Rod and the Slider. For the second component of the Joint, select the middle of the lower cylinder on the Connecting Rod. In the Motion tab, change the Type to Revolute and make sure the Z axis is selected. Then press OK and toggle visibility on the Slider.

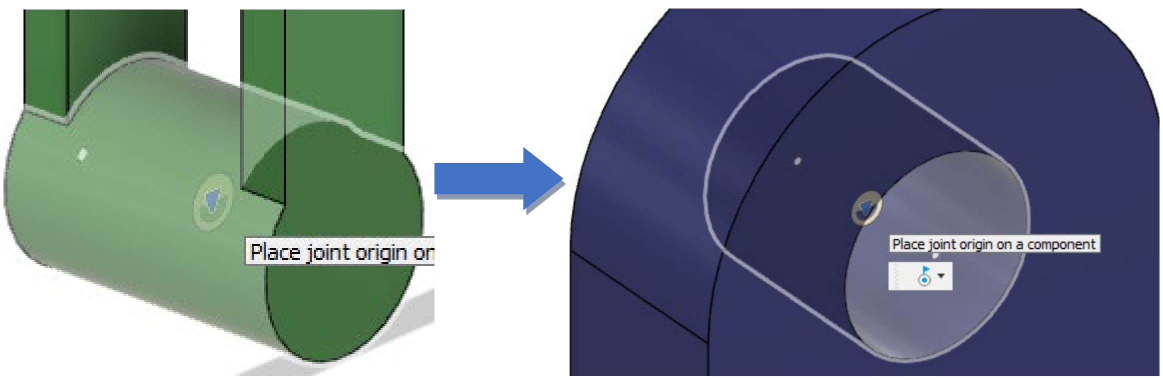




g) Repear the process for the Connecting Rod & Crank, as well as the Crank & Base, toggling visibility as needed. Both Joints should be of Type Revolute and Rotate around the Z Axis.



And



6) Animating the design model

a) Animate Joint Relationships

With 4 joints, the Slider Mechanism is set up and ready to animate. Open the Joints folder in the Browser, right-click on the Joint between the Base and the Crank, then click Animate Joint Relationships. This should demonstrate the motion of the mechanism (press Esc to end the animation). Notice how the velocity of the Slider changes, even with a constant angular velocity in the Crank.

b) Contact Sets

When you wish to animate a mechanism that is not connected by joints and links but by gears and/or cams you will need to create a Contact Set (or setup a Motion link.) It's as simple as clicking on ASSEMBLE > Enable contact sets, and clicking on the two components that contact each other (e.g. two gears). A short 2 min video explains how:

https://knowledge.autodesk.com/support/fusion-360/learn-explore/caas/screencast/Main/Details/d651acae-571b-415f-8318-e8dbb1371db9.html

c) Motion Link

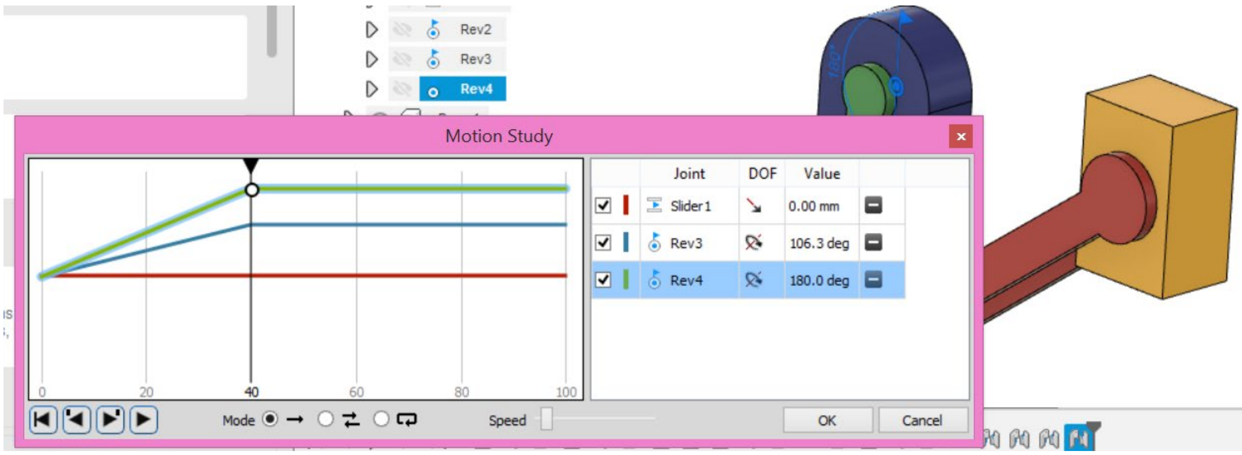
The 'Animate Joint Relationships' functionality works well for a closed kinematic link such as the slider-crank. However, ASSEMBLE > Motion Link allows you to include the movement of additional components of your product that are either not physically connected together, or where a belt or cable moves around pulleys. It allows you to 'link' the motion of one joint to another by giving the ratio of angles and/or distances. Many of your projects will need to use this to fully animate and present your Final Design Projects. A short 6 minute video explains how to do this for a gear drive, and linear screw drive: https://www.youtube.com/watch?v=jSaLy4RMnfY

SPRINGS: Fusion does not have a direct way of animating a spring, yet. However, you can fool the eye by creating two shorter springs overlapping each other and using a slider joint to animate them to look like one spring expanding and compressing (as long as you don't zoom in too close!) OR you can create joints between each spring section and do a visually clearer hack (but quite a bit of work!: https://www.youtube.com/watch?v=iwwy7mdfTHE

d) Motion Study

Another useful tool for some animation needs is through clicking on ASSEMBLE > Motion Study.

The following dialog box will open:



A short but excellent you tube video explains how to use "Motion Study":

https://www.youtube.com/watch?v=JWJI8yzPUdU

CHAIN or BELT DRIVES:

https://www.youtube.com/watch?v=F-mAVhdW8z8

e) Fusion has great easy to use visualization tools such as 'Animate Joint Relationships', 'Contact Sets', 'Motion link' and 'Motion study', but to get quantitative results, you will need to use dynamic simulation software. Fusion does not yet have such a feature but another Autodesk product, **Inventor**, does. Dynamic simulation requires greater computational power than some of your laptops may have, so this semester we have decided not to require you to do this part of the lab, however, the instructions are in Appendix A if you wish to learn how to do this. It may well come in handy for other projects and classes such as ME370.

Part II. Submission Requirements

a. Make sure your Slider Assembly" is in the Lab 9 folder and is shared with your TA.

b. Go to ME170 Blackboard website and the CAD LAB Assignments content area. Click directly on the "CAD LAB 9" assignment title and use the "Write Submission" button to type "Ready for Grading". As before, please do not go back and change your files in the Fusion Lab folder.

Appendix A

OPTIONAL

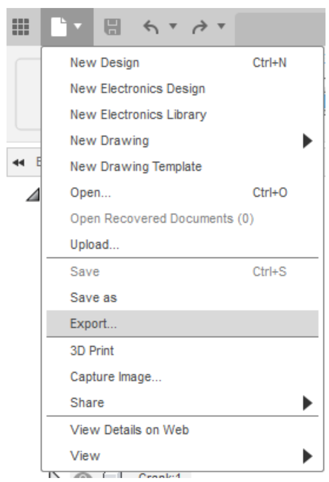
Dynamic Simulation using Autodesk Inventor

This will not be graded but useful to learn for later classes, such as ME370, or extra-curricular activities with design/build/compete projects.

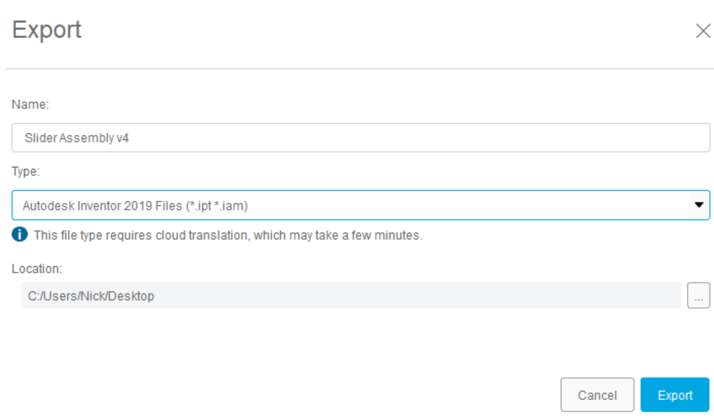
A1. Export the slider-crank assembly to Autodesk Inventor

1) Export the Assembly

   a) In the top-left corner of Fusion, select File, then select Export



   b) Change the Type to Autodesk Inventor Files, set the Location to a place you will remember, and press Export.



c) A progress screen will appear, it will likely take a few minutes to export. Once the status is Completed, select "Show in File Explorer."

d) Unzip the folder; there should be 5 files inside (4 for the individual parts and 1 for the assembly).

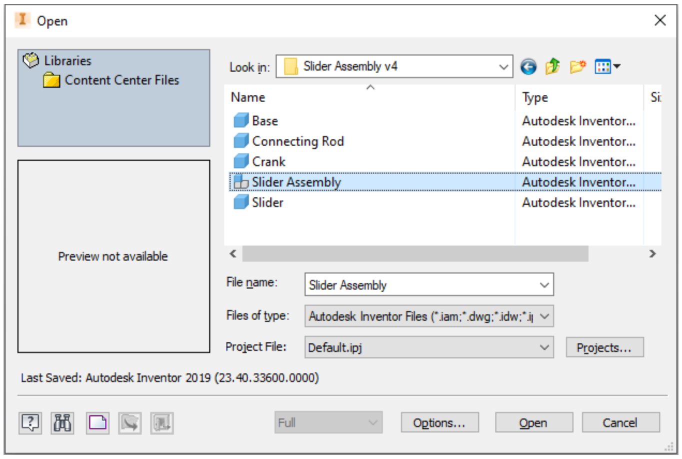
2) Download and Launch Inventor

a) If you do not already have access to Autodesk Inventor, navigate to www.autodesk.com/education/edu-software/overview and select "Get Started" under the Inventor Professional option.

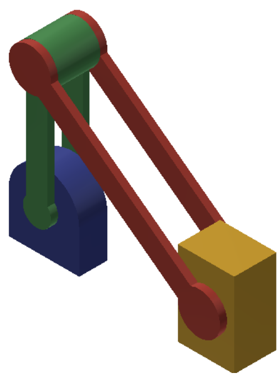
b) Enter your Autodesk account information (the same account you use for Fusion), then select "Get Product" under Inventor Professional.

c) Verify the version is appropriate for your computer and select Install. Follow the instillation prompts that appear (these will vary depending on your computer), and once finished, open the application.

d) At the home screen, press Ctrl + O to open a file. Navigate to where you saved the "Slider Assembly" folder and select the Slider Assembly file. This file should have a different logo than the other files in the folder, since in Inventor, Assembly files are different than Part files.



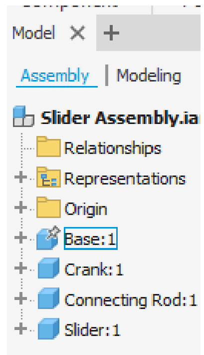
e) The Navigation tools in Inventor are the same as Fusion (e.g. middle-mouse button is Pan), and you should find that the assembly looks very much the same. Unfortunately, the Joints are not carried over, so they will have to be added back.



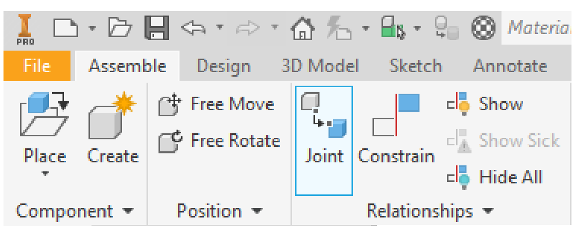
A2. Set Up and Conduct the Simulation

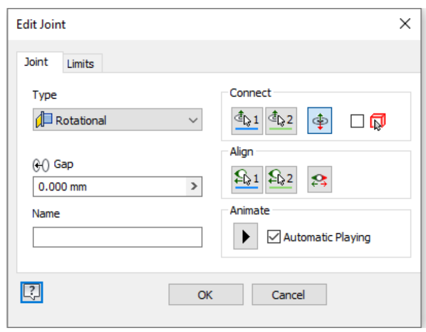
1) Redefine the Component Relationships

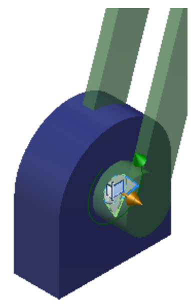
   a) Make sure your Base part is Grounded (this should have carried over), as shown by a pin in the symbol in the Browser. If not, right-click it and click "Grounded."



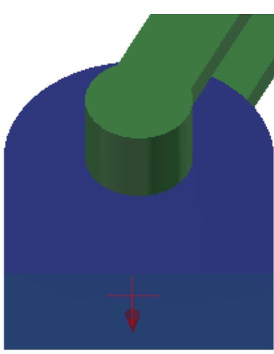
b) The Joint tool is located in the top toolbar. In Inventor, you can move parts without requiring a Capture Position, so move the components apart, until the parts that overlap are clearly visible. Use the Joint tool to add back in the rotational Joints between the Base & Crank, Crank & Connecting Rod, and Connecting Rod & Slider. Do not remake the Joint between the Base and the Slider.

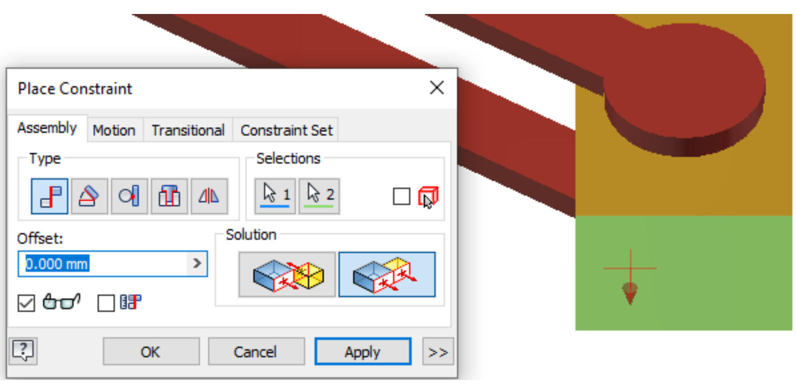






c) Select the Constrain tool (next to Joint), under Solution change Mate to Flush, and then select the bottom face of the Base and Slider. A click sound should play, and the two undersides should be lined up. Press OK to create the Constraint. You should be able to rotate the mechanism like in Fusion.

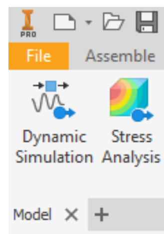




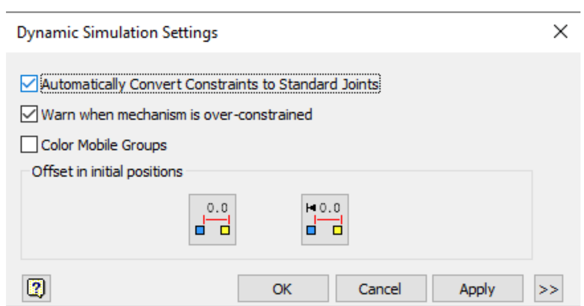
2) Set up the Simulation Environment

a) Along the very top toolbar, select the Environments tab.

b) In the Environments tab, open the Dynamic Simulation tool. You will receive an error message saying the mechanism is over-constrained, press OK.

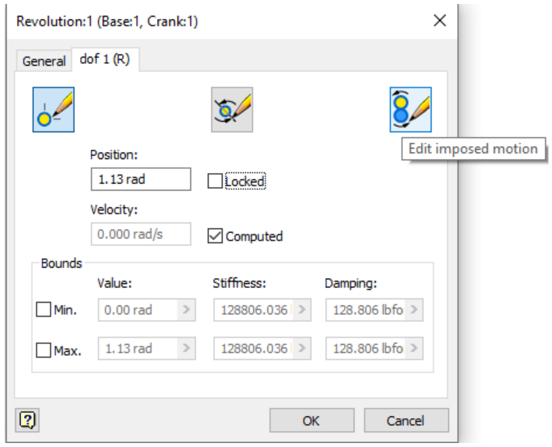


c) In the toolbar, select Simulation Settings, and then uncheck "Automatically Convert Constraints to Standard Joints." A pop-up will apear asking if you would like to maintains standard joints, press Yes.

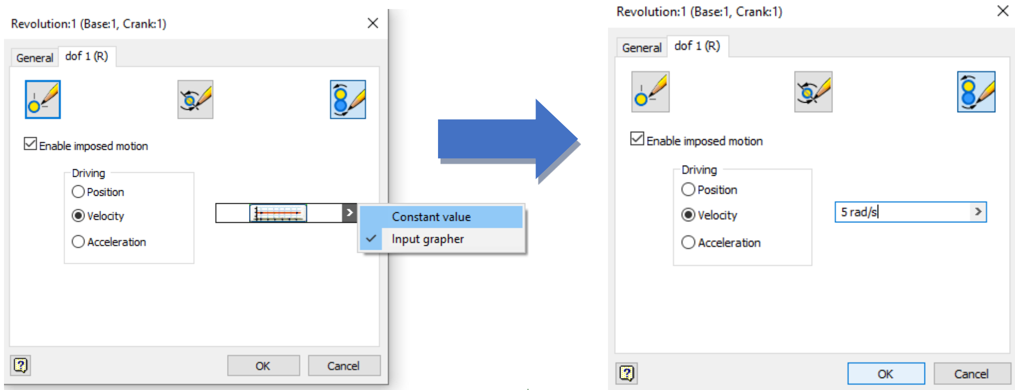


d) Locate the Standard Joints folder in the Browser on the left. Right-click on the Revolution between the Base & Crank and select Properties.

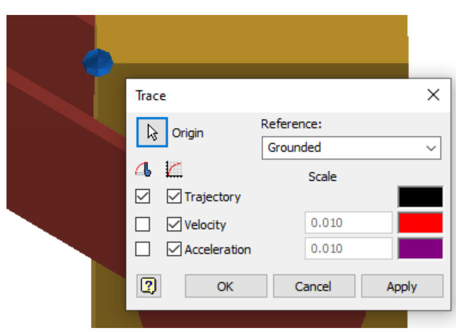
e) Select the dof 1 (R) tab, and then the Edit Imposed Motion button



f) Check "Enable imposed motion", then click the arrow next to the Input Grapher, then select Constant Value. Enter 5 rad/s and press OK.

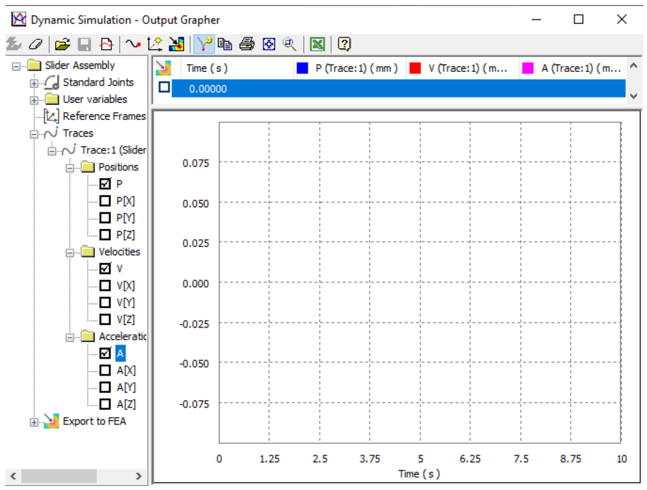


g) In the toolbar, select the Trace tool. Check off Trajectory, Velocity, and Acceleartion in the Output Trace Value column, then click on one of the corners of the Slider component to set it as the Origin. Press OK.

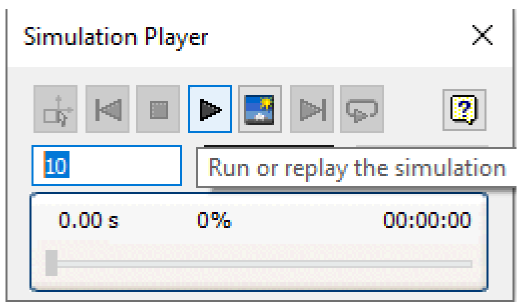


h) In the Simulation Player pop-up, change the time from 1s to 10s for a longer Simulation.

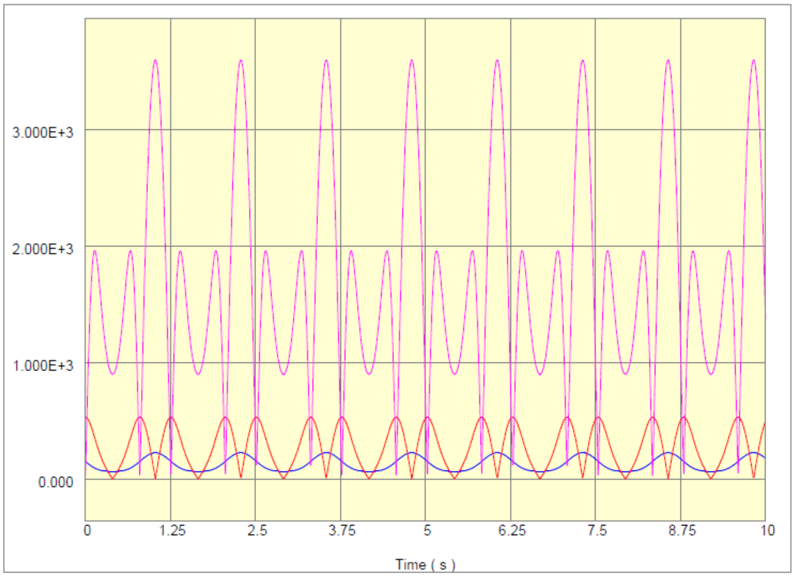
i) From the toolbar, select Output Grapher. Resize the window to make it larger, then expand the following folders in the list on the left: Slider Assembly > Traces > Trace 1, then open Position, Velocity, and Acceleration, and check off the boxes next to P, V, and A.



j) Press Play on the Simulation Player pop-up and watch.



k) As the Mechanism runs, the graph is created, and then finalized. In the Output Grapher window, select the Print tool. Choose “Microsoft Print to PDF” and then OK. Choose a location to save the .pdf to and then press Save. Submit this to Compass if you would like to impress us that you actually did this optional part if the lab ☺. Sorry, but there are no grade points per se. It should look something like this:



Position, Velocity, and Acceleration Plots

l) If you would like, you can experiment by changing the given values or testing other variables. To do so, press the Construction Mode button in the top-left of the Simulation Player pop-up and then make changes. Once finished, save your file in Inventor and close the program, make sure it is also saved in Fusion and close it as well.