Lecture Solidworks Tutorial: Basic Sketching and Assembly

Open SolidWorks

Select new document -> Part -> Ok
Select front plane

Select Sketch -> Circle
Click on origin and move cursor to draw circle

Select Sketch -> Smart dimension
Select checkmark
Select Features -> Extrude -> Enter Length 5 in -> Select checkmark
Save part

Select new document -> Part -> Ok

Select front plane

Click on origin and move cursor to draw circle

Select Sketch -> Smart dimension -> Enter 0.5 in
Click on origin and move cursor to draw circle

Select Sketch -> Smart dimension -> Enter 1.75 in

Select checkmark
Select Feature -> Extrude -> Enter Length 0.625 -> Select checkmark
Select Orientation -> Front

Select Sketch -> Circle -> Click on origin and draw circle
Select Sketch -> Smart Dimension -> 1.875

Select Features -> Extrude -> 0.0625 in -> Select checkmark
Save Part
Select new document -> Part -> Ok -> Select front plane

Select Sketch -> Line

Draw base of block

Select Sketch -> Tangent Arc

Connect the two free ends of the block with the arc.
Select Sketch -> Circle

Draw circle from center of tangent arc
Smart Dimension

Extrude -> 0.625 -> Checkmark
Select Feature -> Fillet
Select edges to fillet

Enter radius -> 0.5 in
Select Orientation -> Front View

Select Sketch -> Circle

Draw circle from center of arc

Smart Dimension -> 1.875 in
Select Features -> Extrude Cut -> Enter length 0.0625 in
Select Features -> Hole wizard -> Position
Select Orientation -> Top View -> Click on location of holes -> ok
Save Part

Select new document -> Assembly -> Ok -> Select Block (The part is anchored)
Select Assembly -> Insert Components

Repeat for all the components
Select Assembly -> Move Components

Move components into the positions needed
Select Mate

Pick the two places that will be joined

Outer bearing to inner block
Bearing face to block face

Shaft to bearing
Shaft face to block face
Block face to block face
Save Assembly