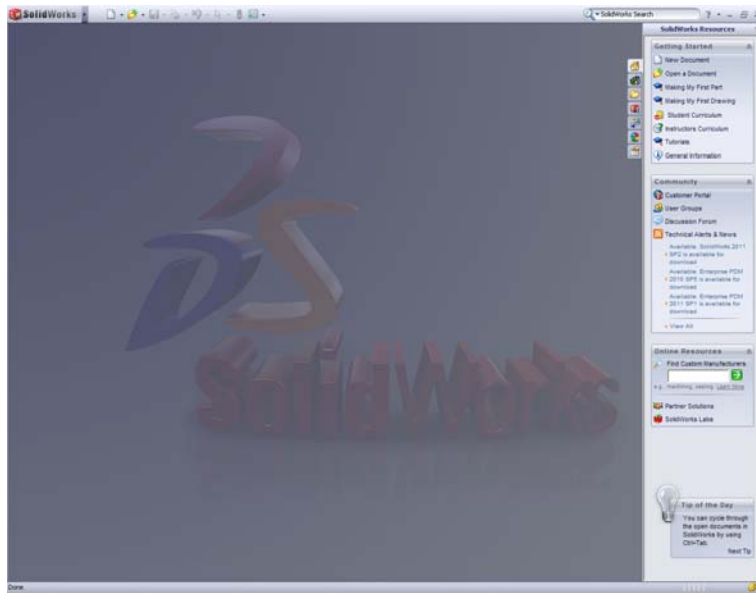
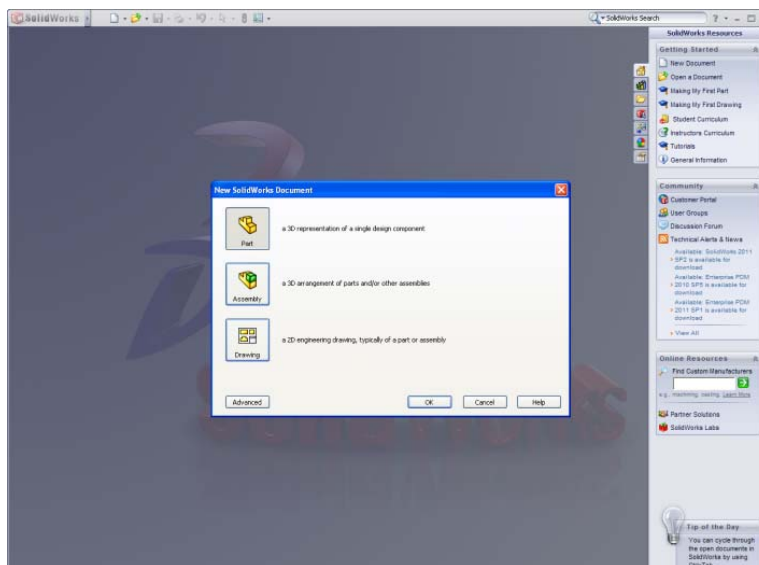


Lecture Solidworks Tutorial: Basic Sketching and Assembly

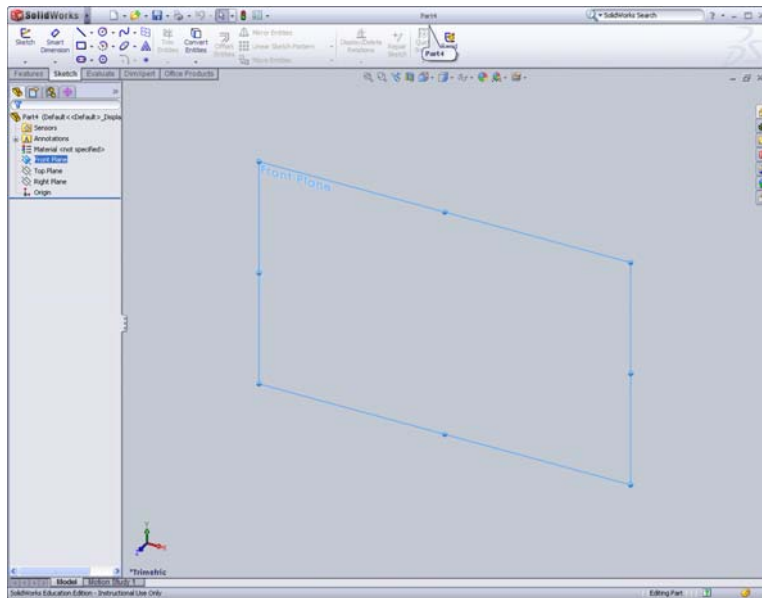
Open SolidWorks



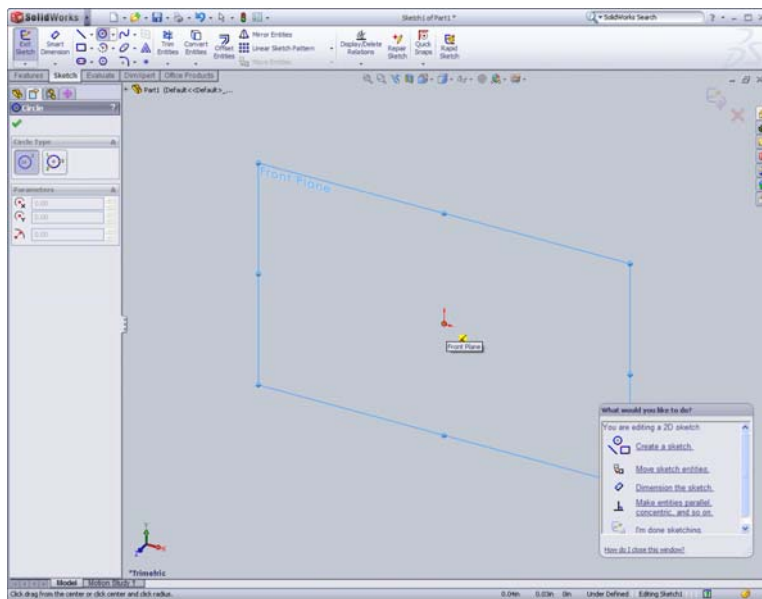
Select new document -> Part -> Ok



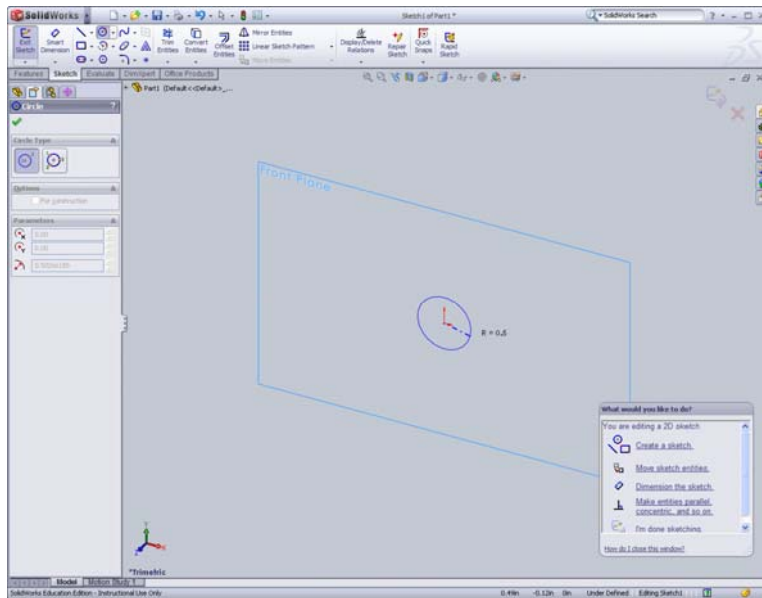
Select front plane



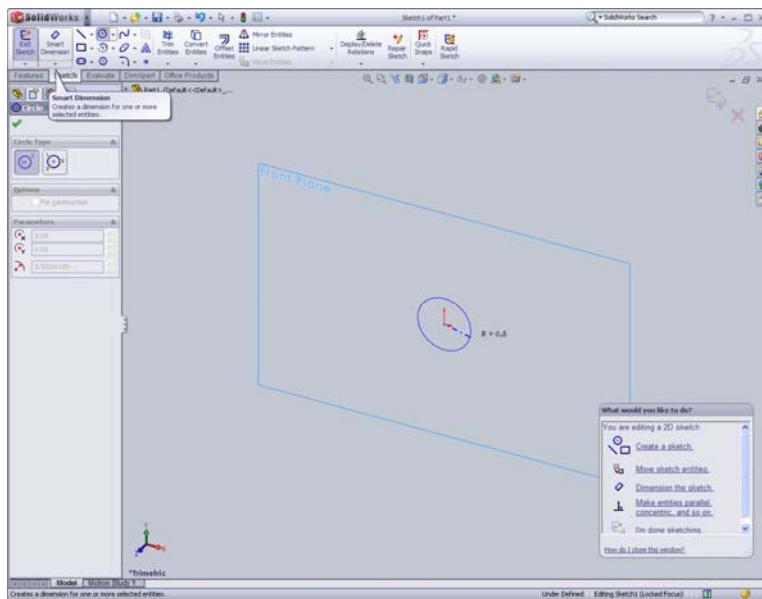
Select Sketch -> Circle

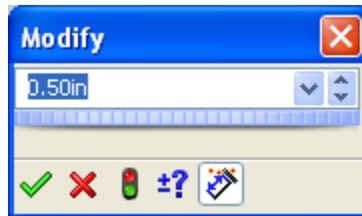
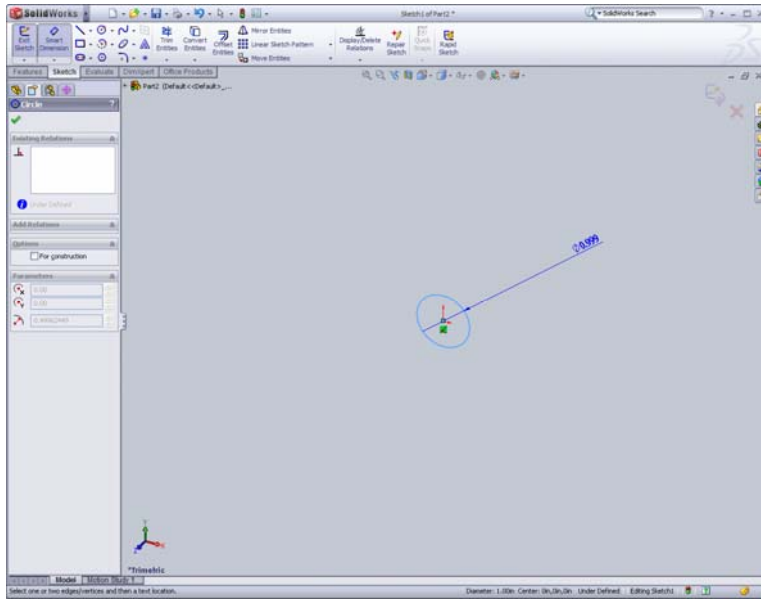


Click on origin and move cursor to draw circle

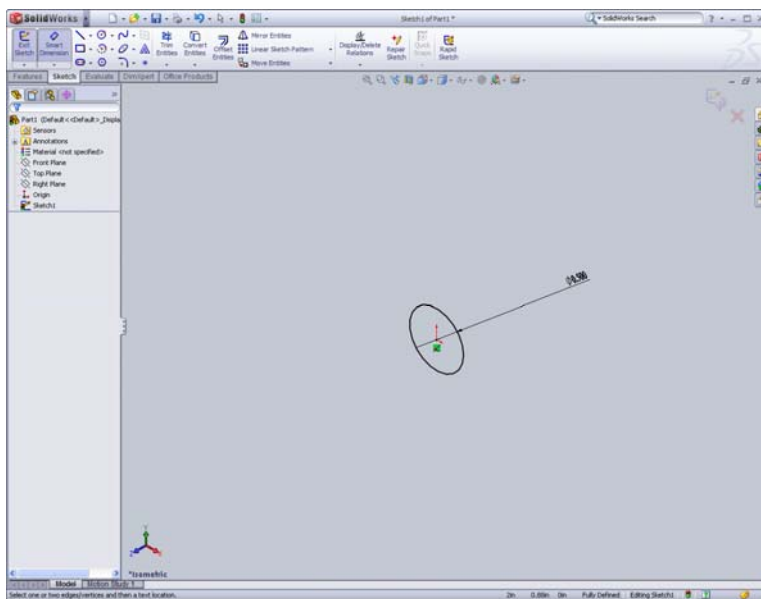


Select Sketch -> Smart dimension

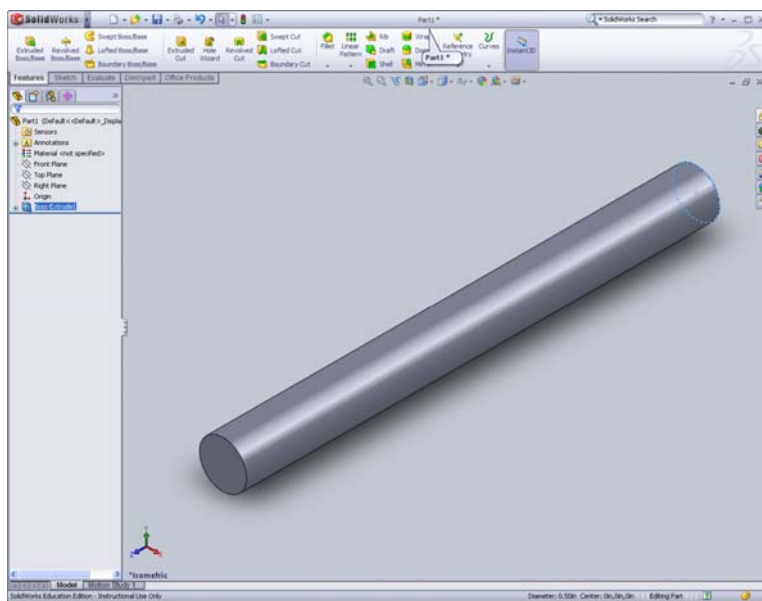
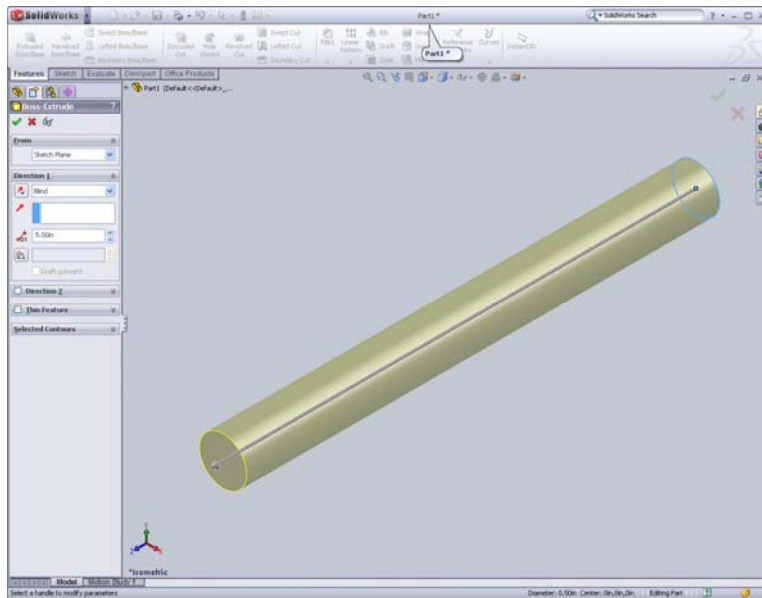




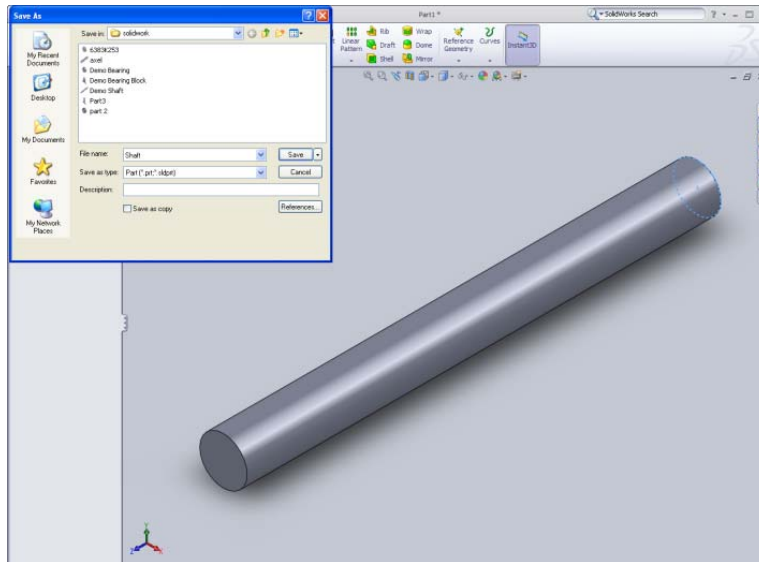
Select checkmark



Select Features -> Extrude -> Enter Length 5 in -> Select checkmark



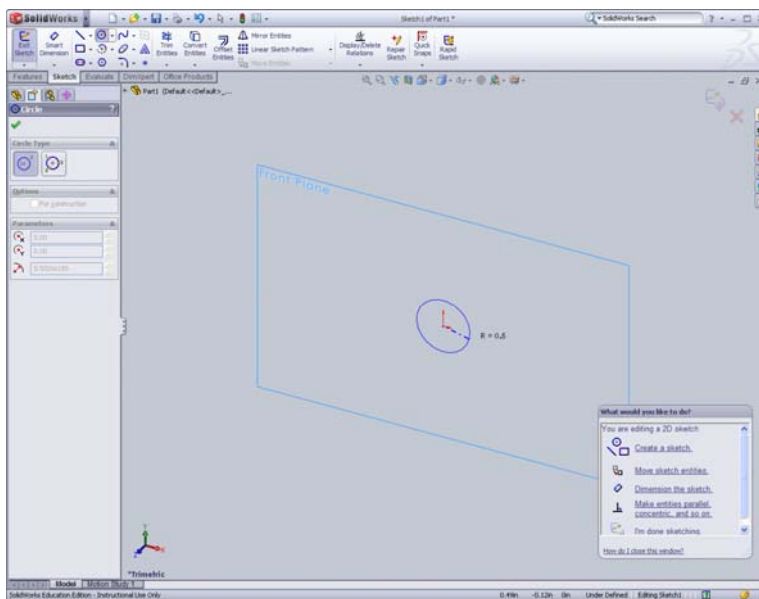
Save part



Select new document -> Part -> Ok

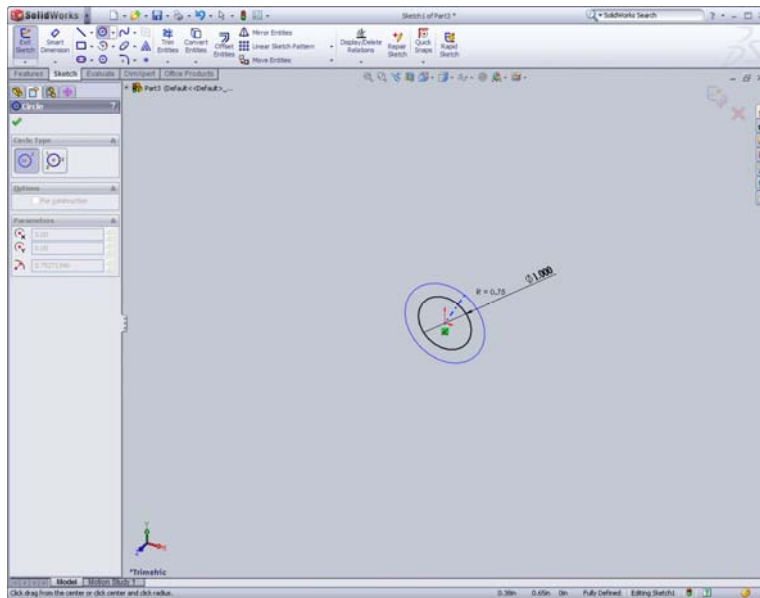
Select front plane

Click on origin and move cursor to draw circle

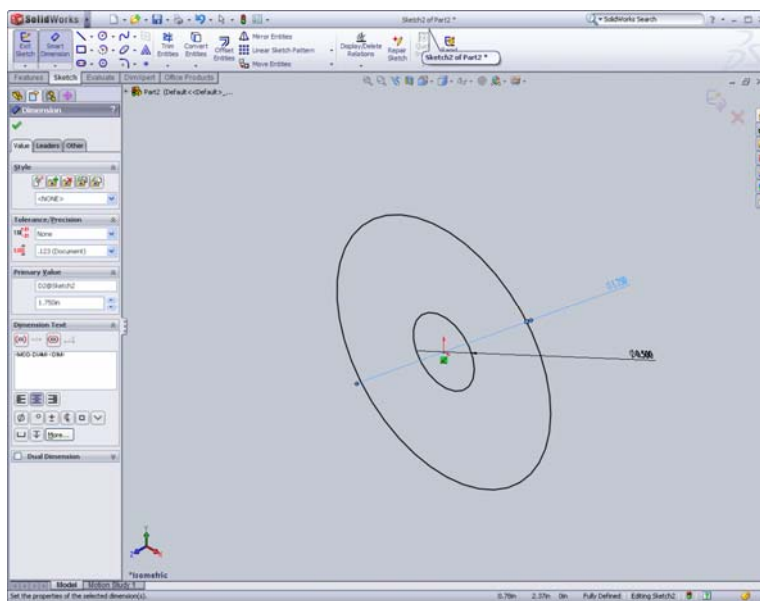


Select Sketch -> Smart dimension -> Enter 0.5 in

Click on origin and move cursor to draw circle

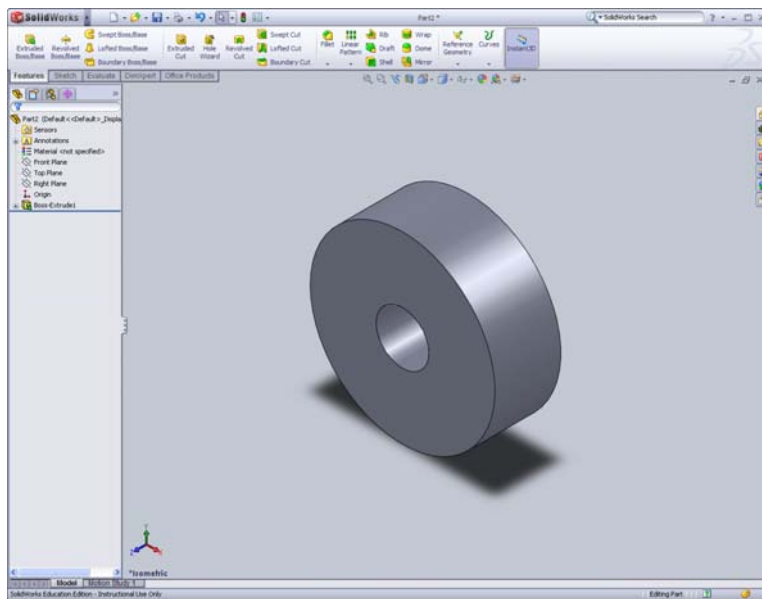
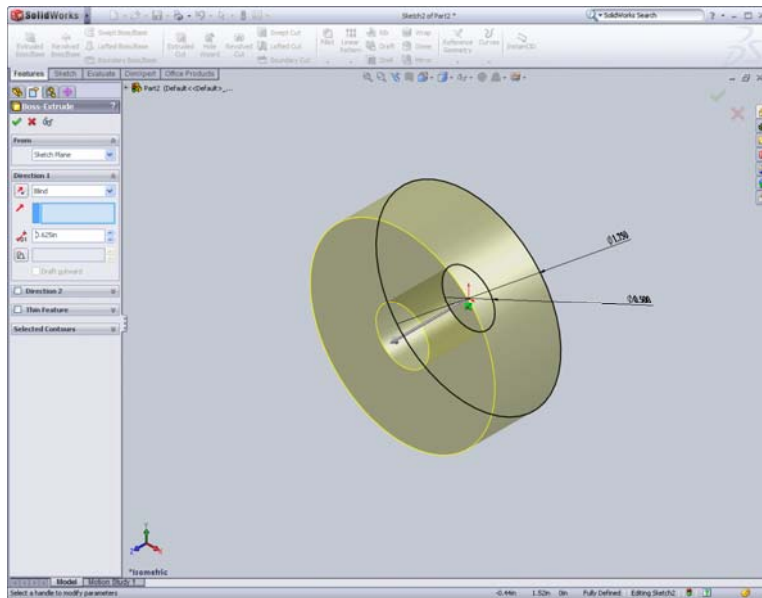


Select Sketch -> Smart dimension -> Enter 1.75 in

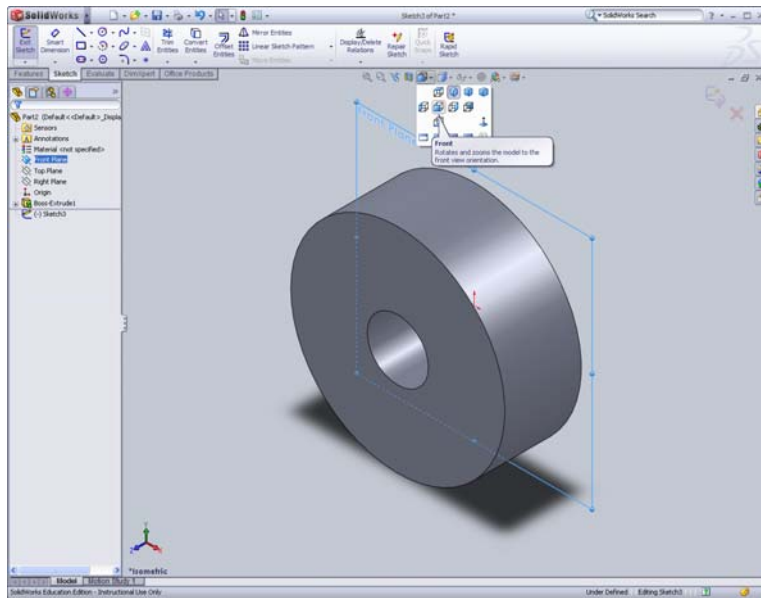


Select checkmark

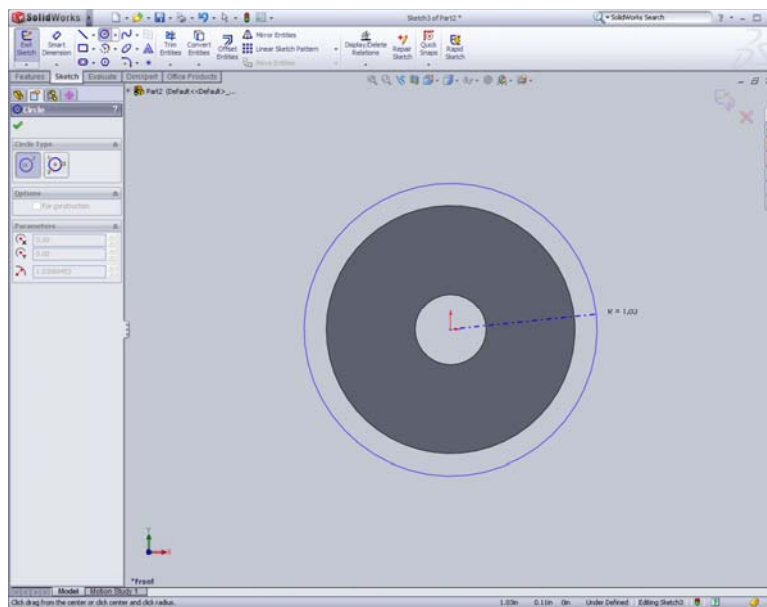
Select Feature -> Extrude -> Enter Length 0.625 -> Select checkmark



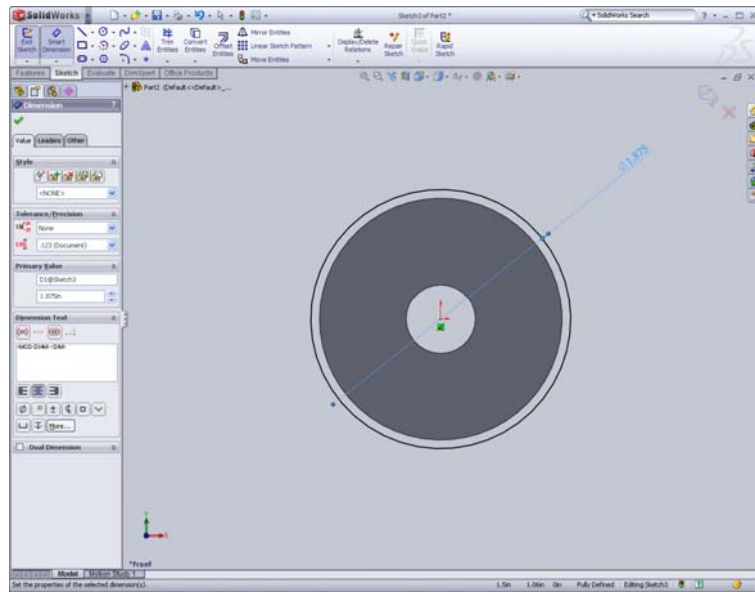
Select Orientation -> Front



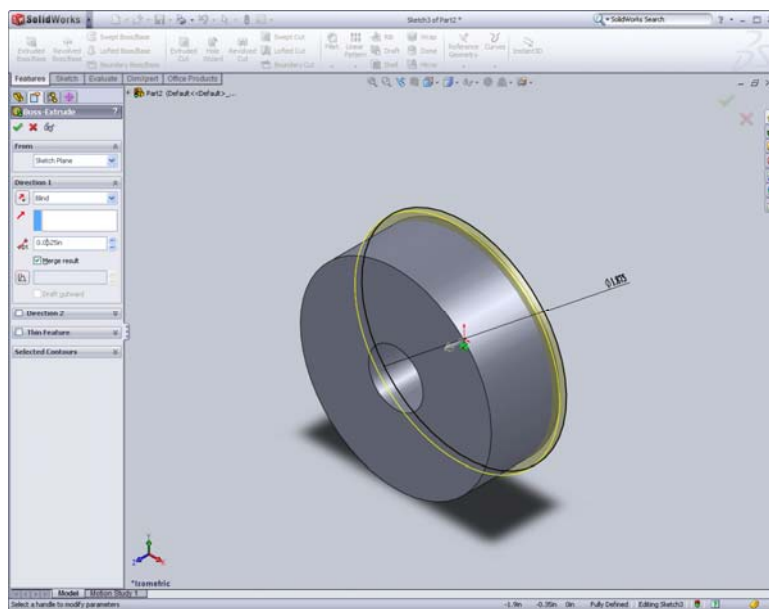
Select Sketch -> Circle -> Click on origin and draw circle

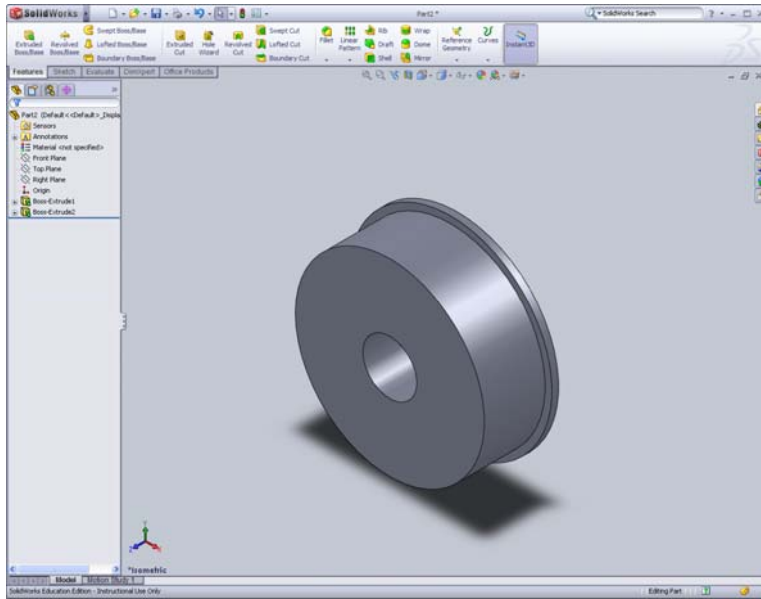


Select Sketch -> Smart Dimension -> 1.875

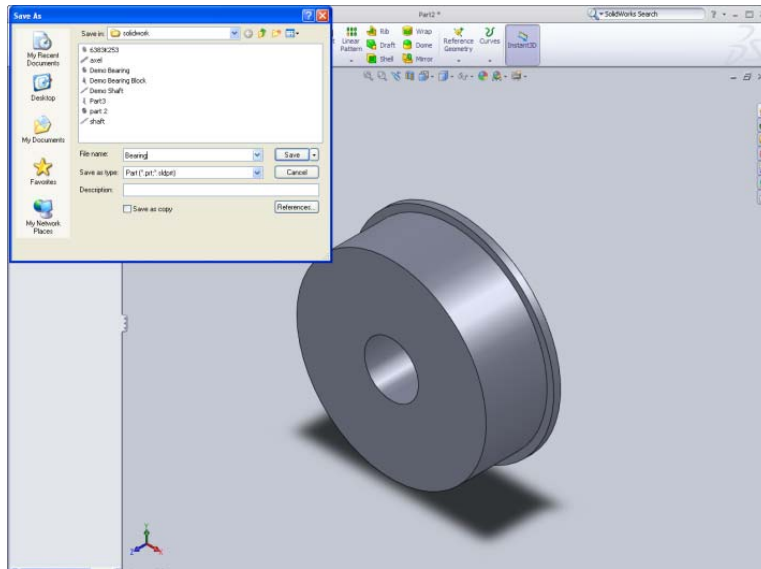


Select Features -> Extrude -> 0.0625 in -> Select checkmark





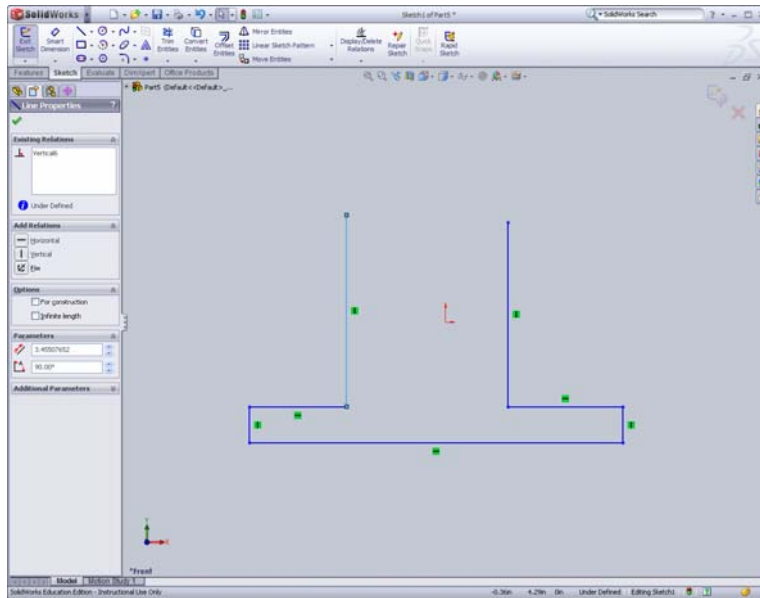
Save Part



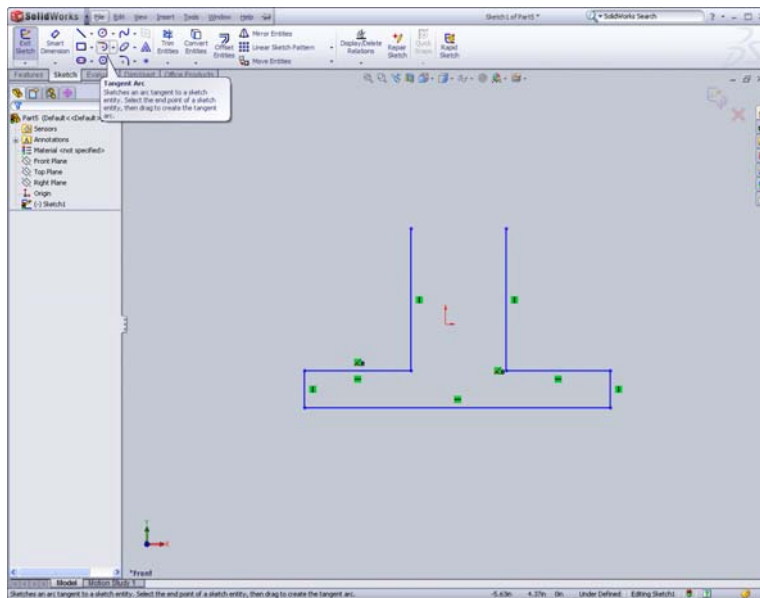
Select new document -> Part -> Ok -> Select front plane

Select Sketch -> Line

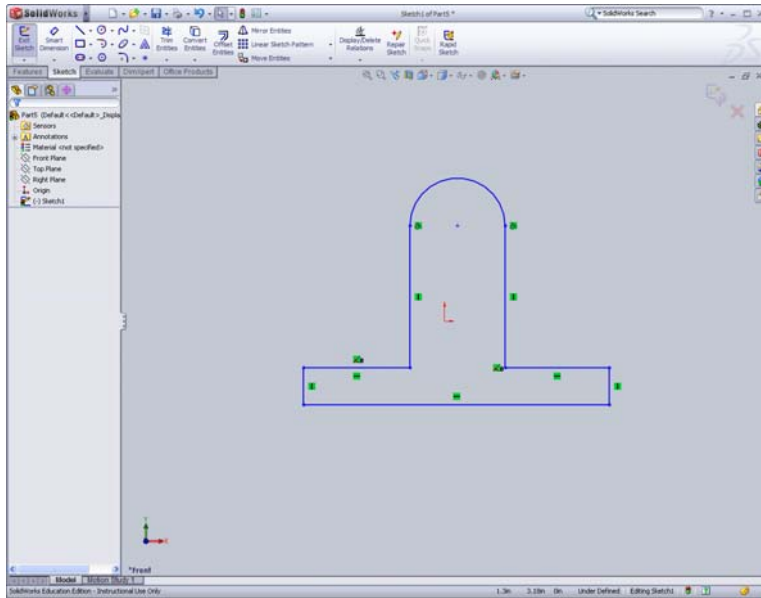
Draw base of block



Select Sketch -> Tangent Arc

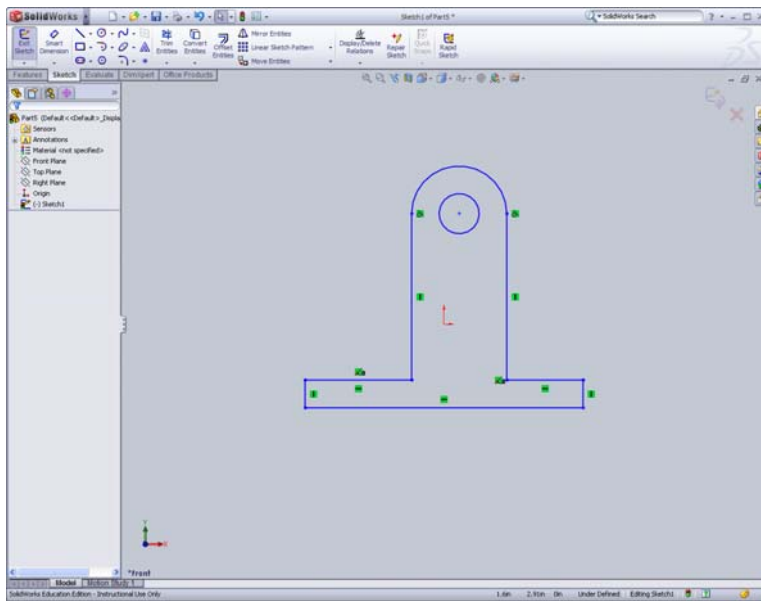


Connect the two free ends of the block with the arc.

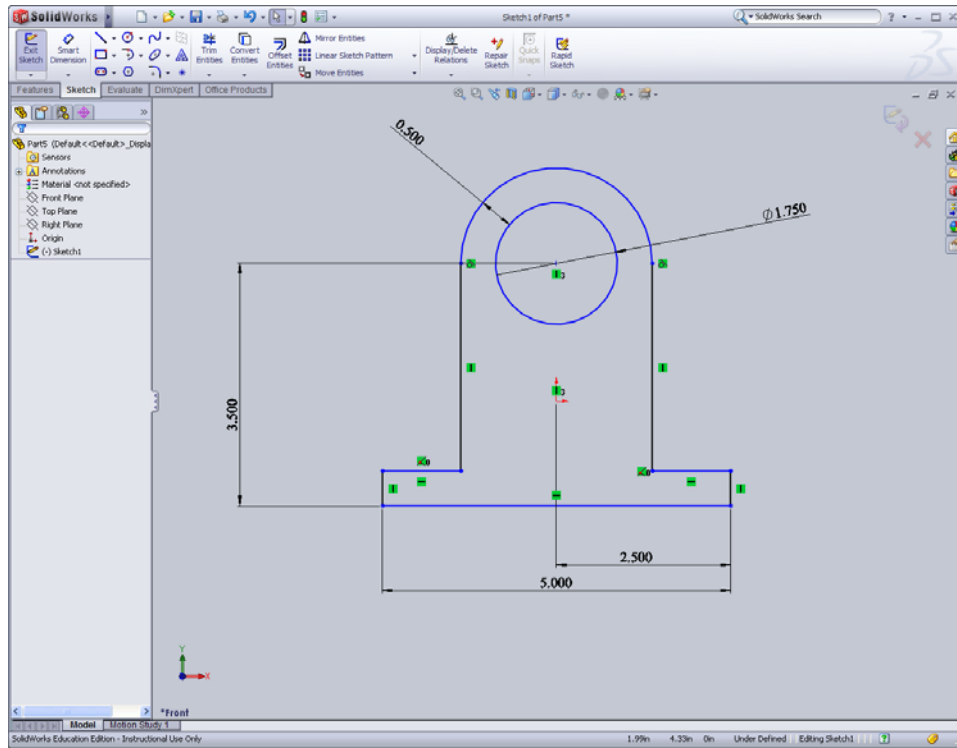


Select Sketch -> Circle

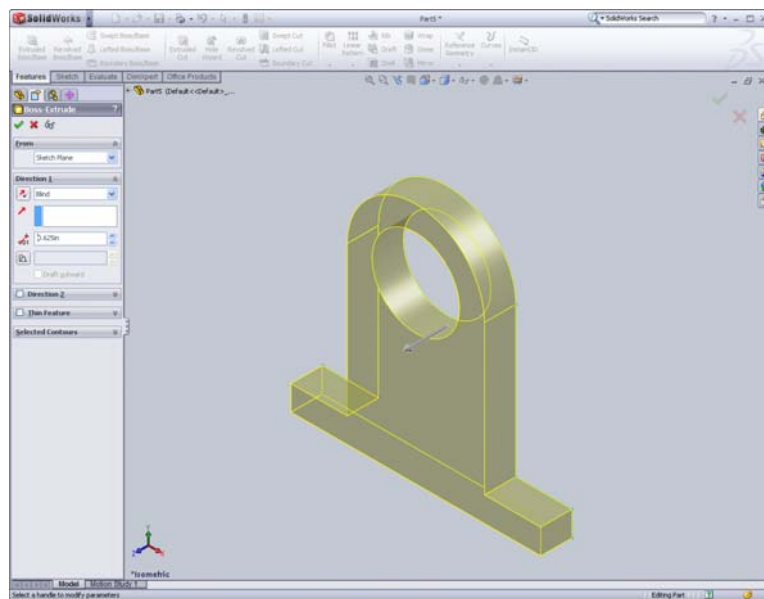
Draw circle from center of tangent arc

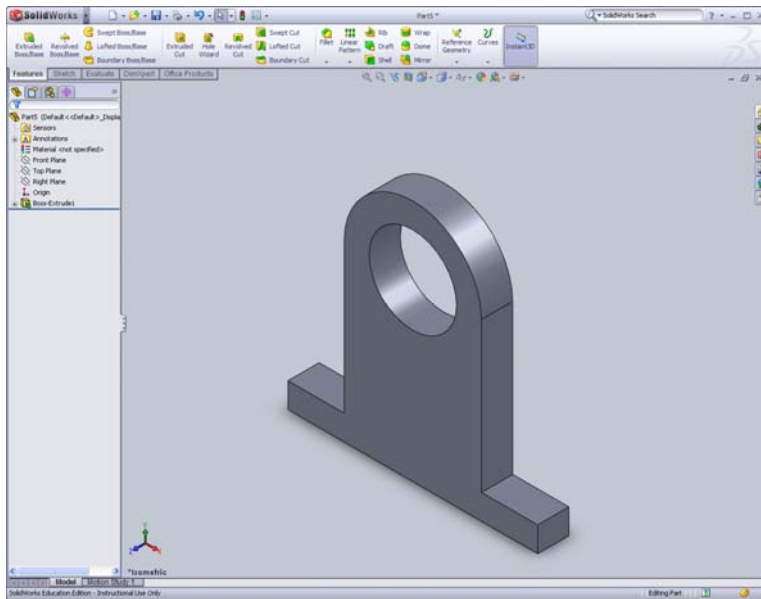


Smart Dimension

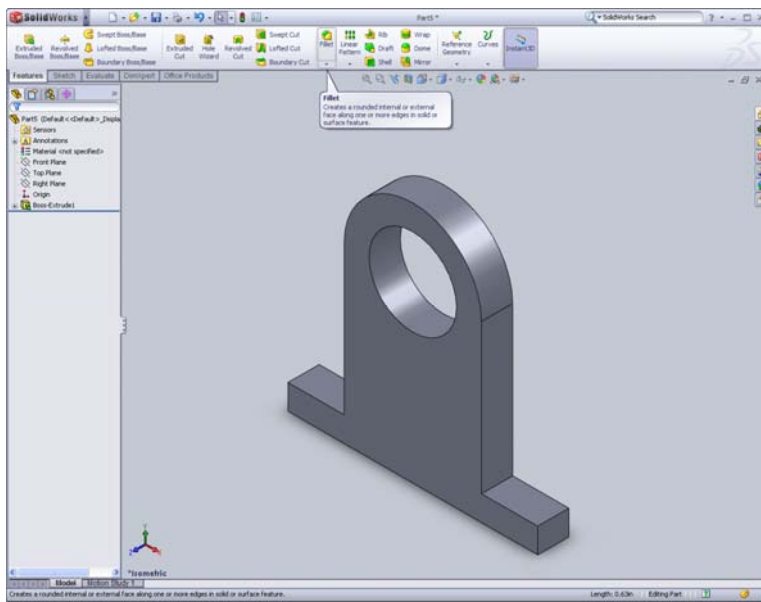


Extrude -> 0.625 -> Checkmark

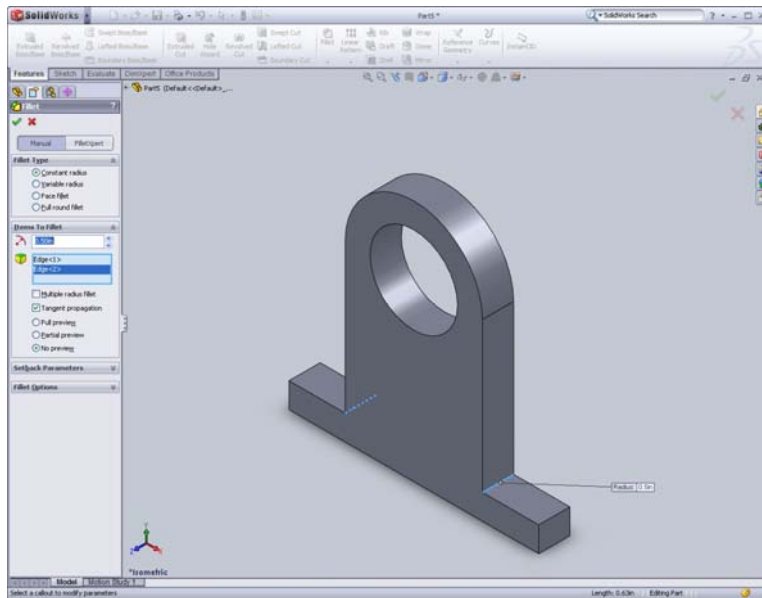




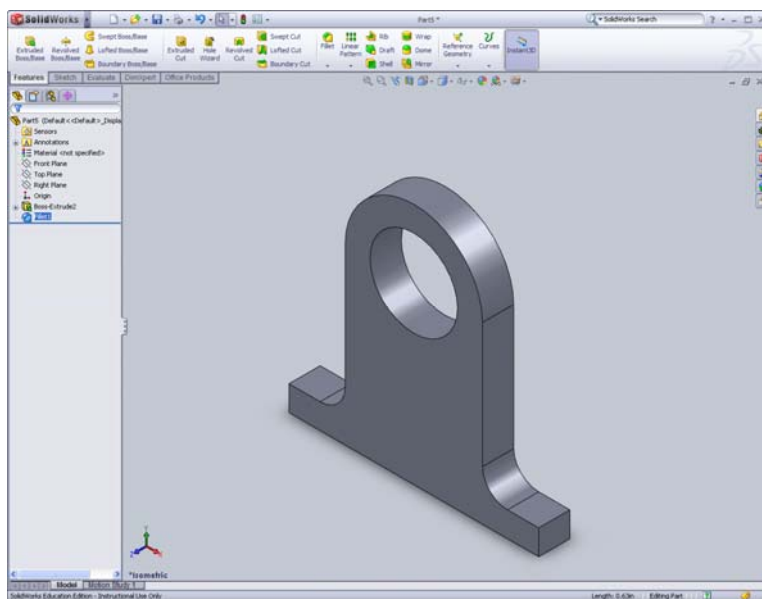
Select Feature -> Fillet



Select edges to fillet



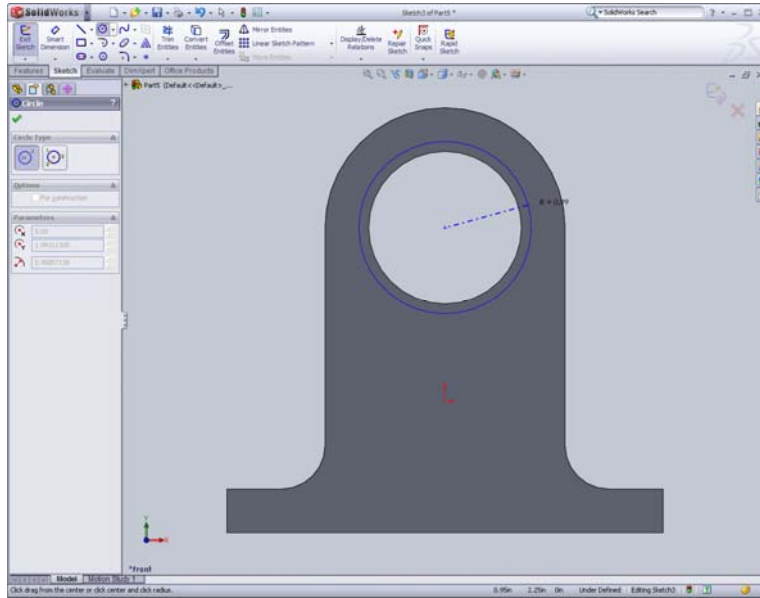
Enter radius -> 0.5 in



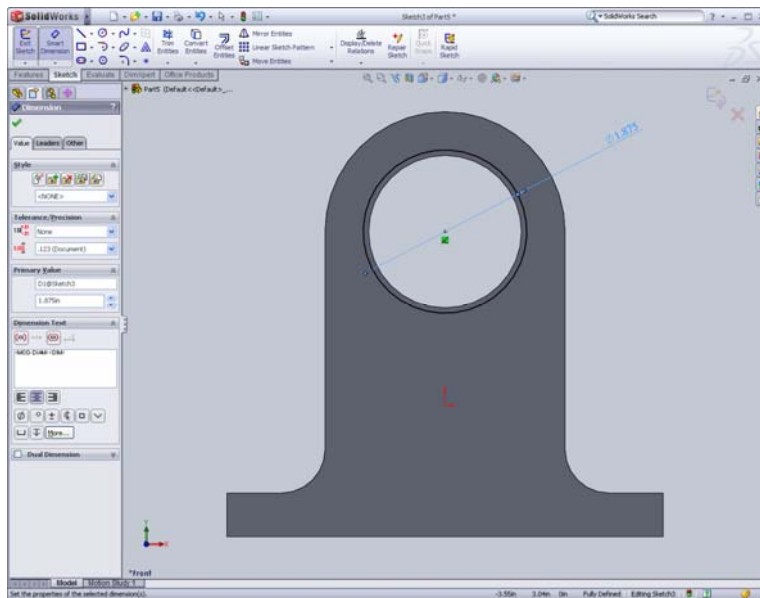
Select Orientation -> Front View

Select Sketch -> Circle

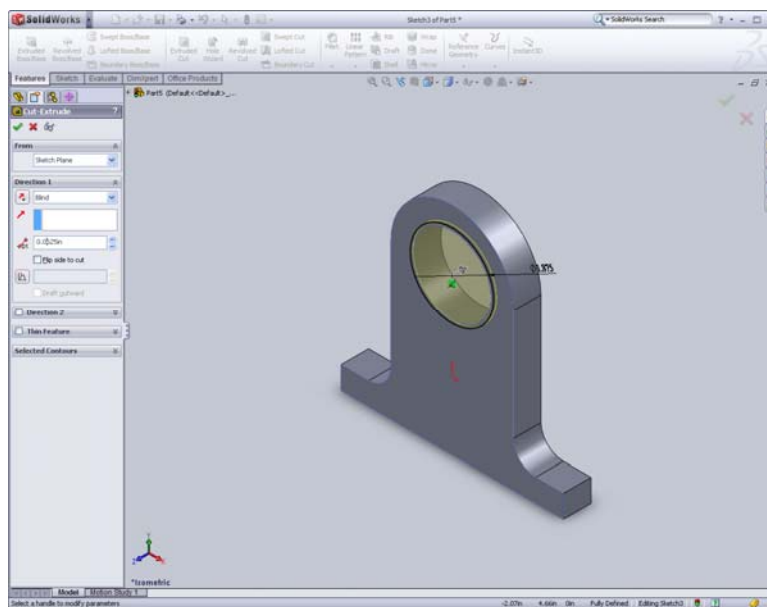
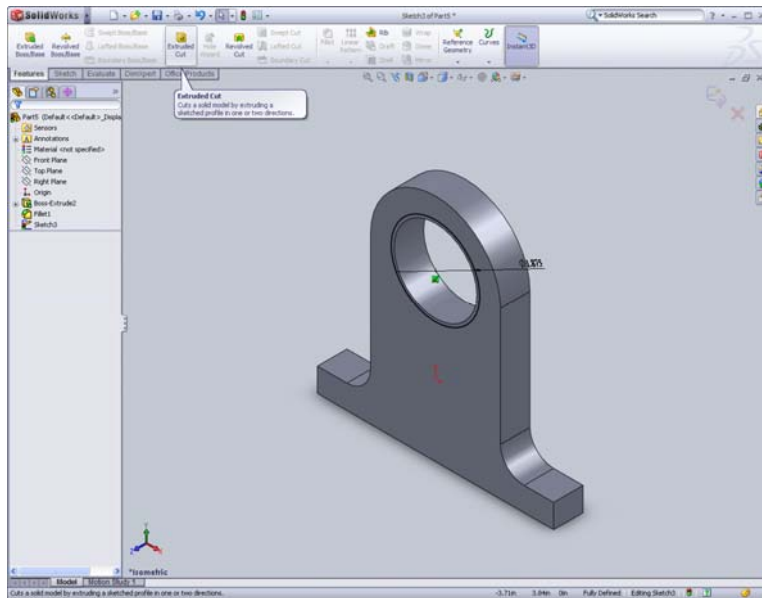
Draw circle from center of arc

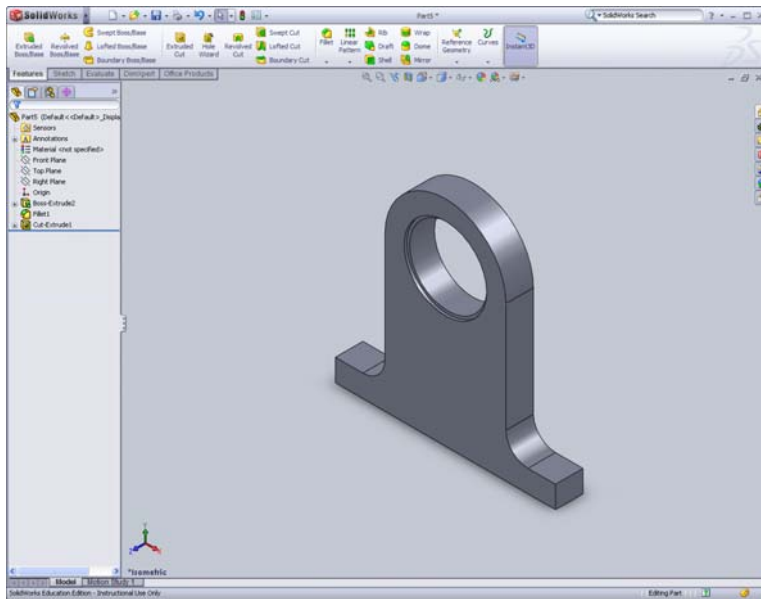


Smart Dimension -> 1.875 in

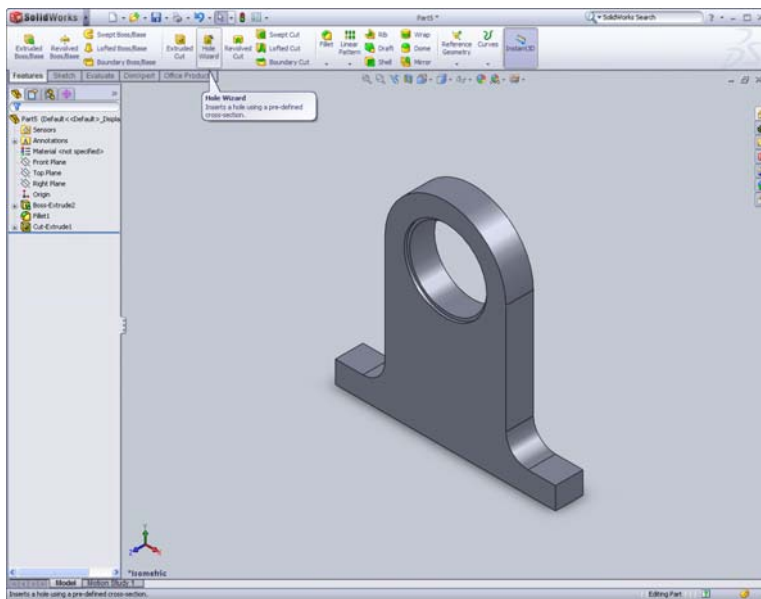


Select Features -> Extrude Cut -> Enter length 0.0625 in

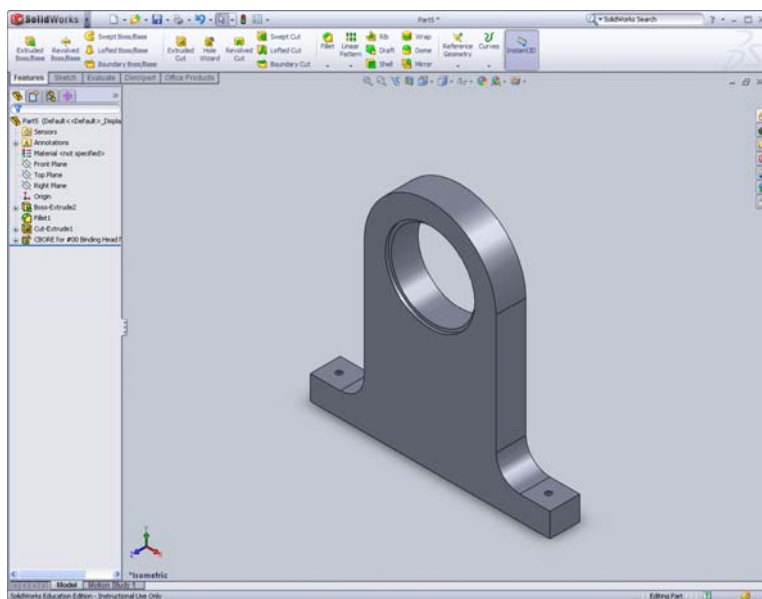
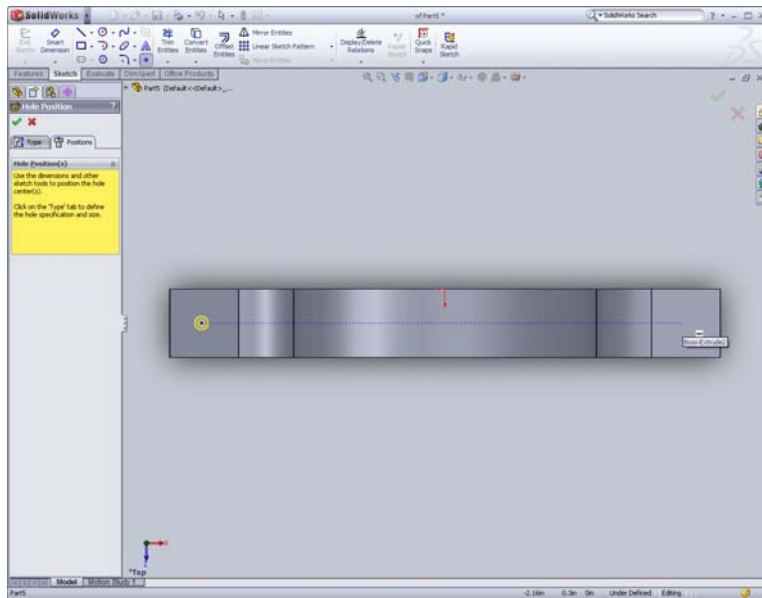




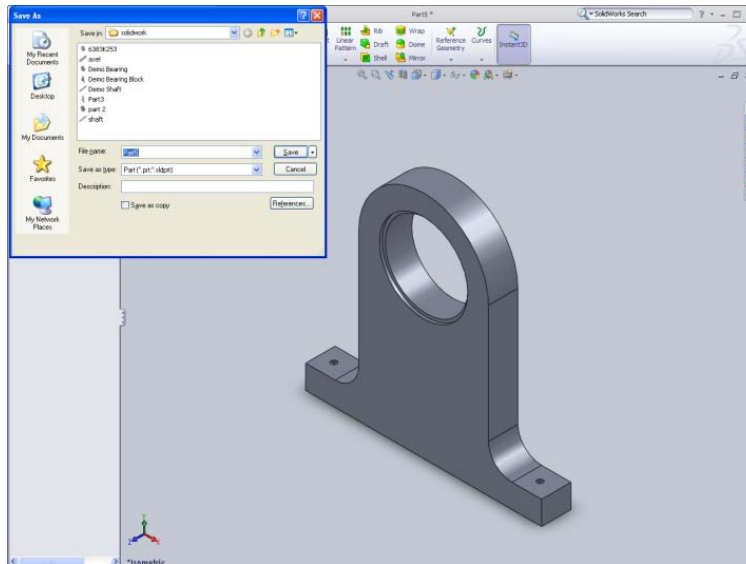
Select Features -> Hole wizard -> Position



Select Orientation -> Top View -> Click on location of holes -> ok

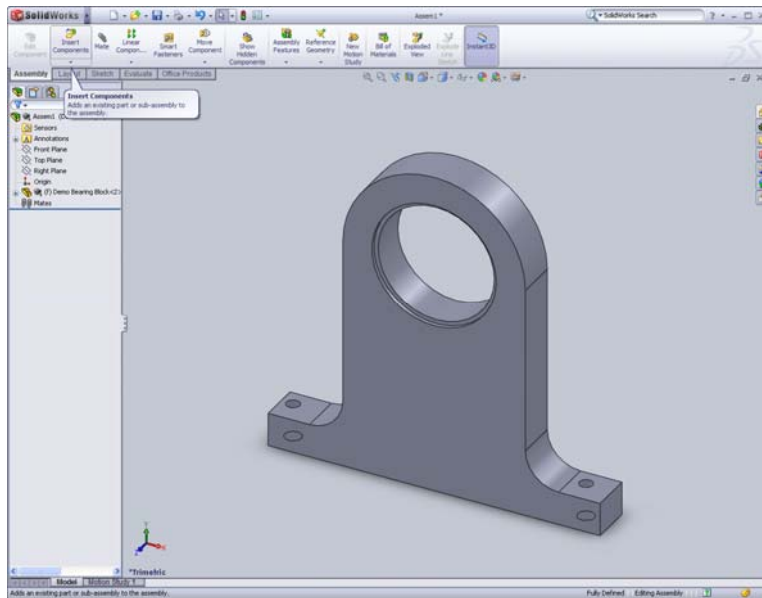


Save Part

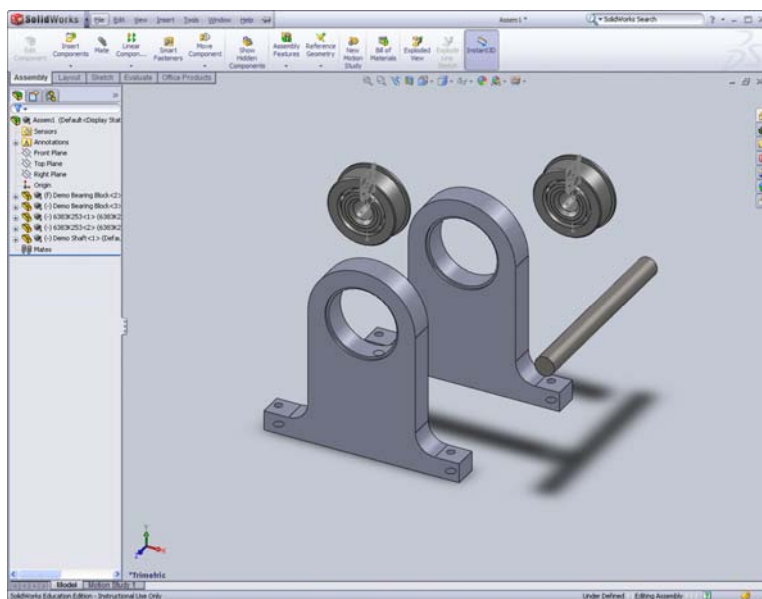


Select new document -> Assembly -> Ok -> Select Block (The part is anchored)

Select Assembly -> Insert Components

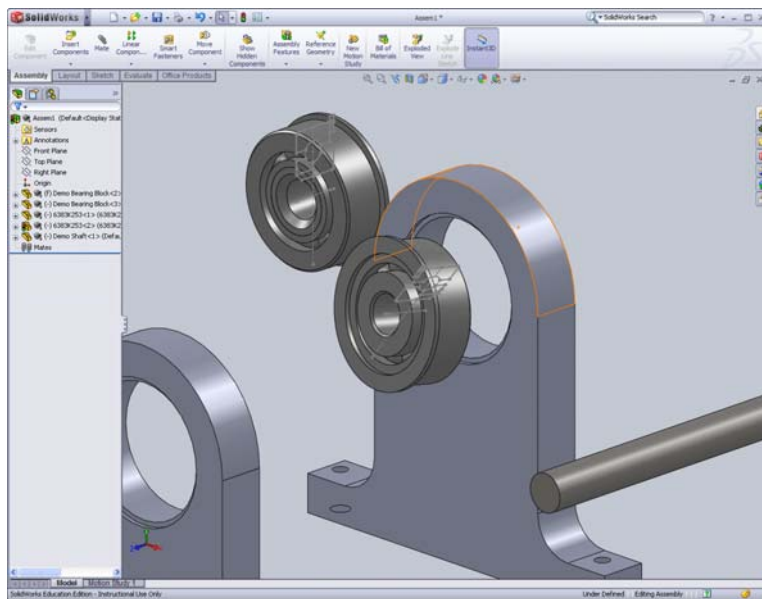
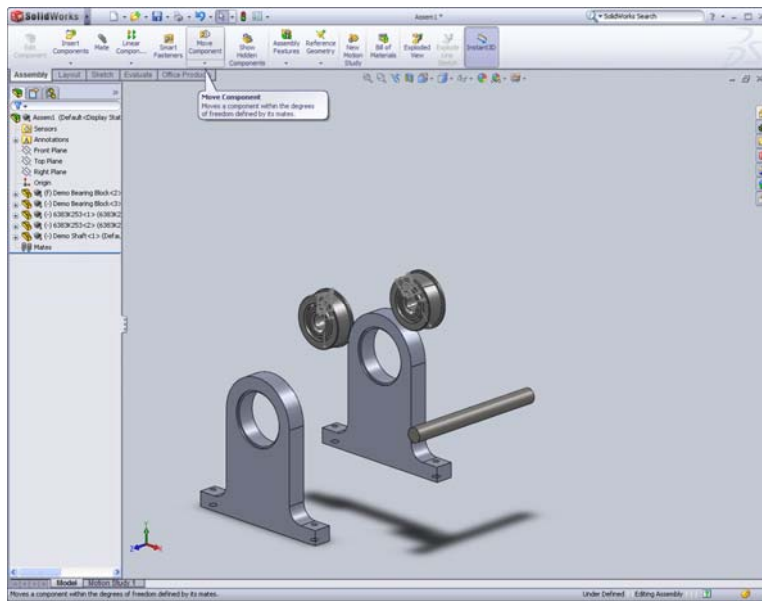


Repeat for all the components



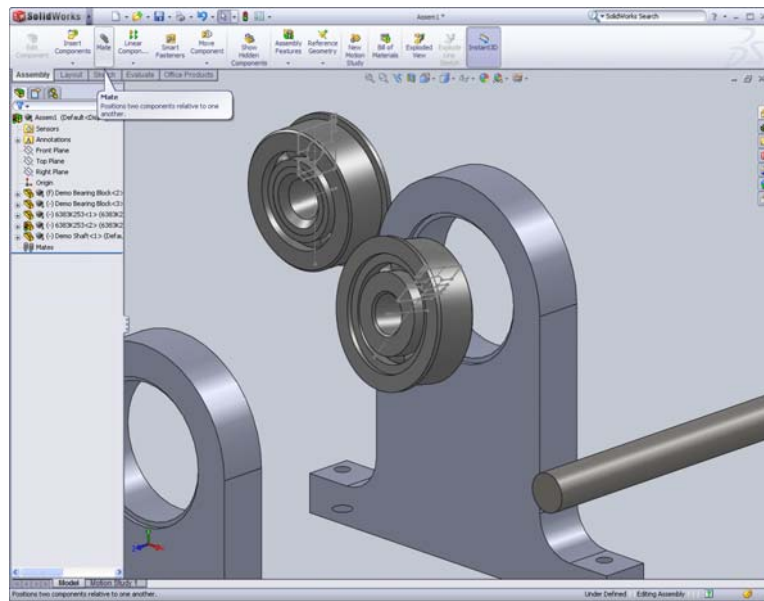
Select Assembly -> Move Components

Move components into the positions needed

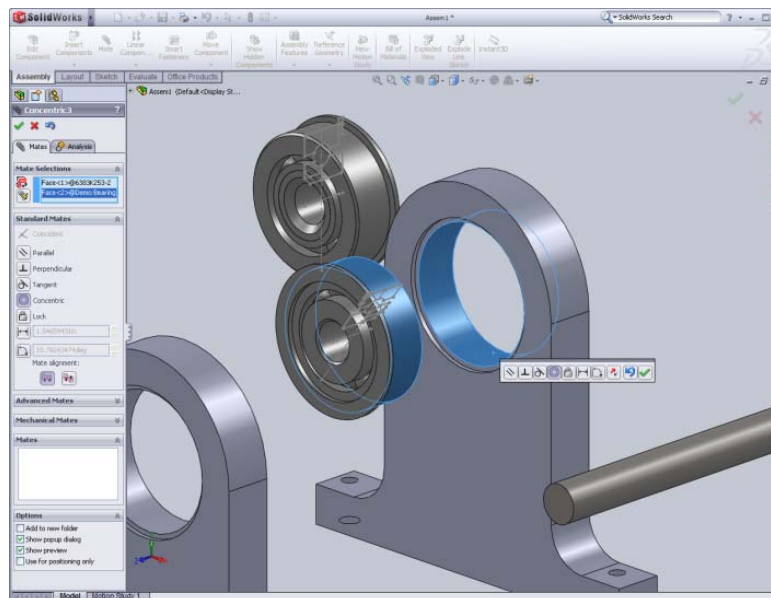


Select Mate

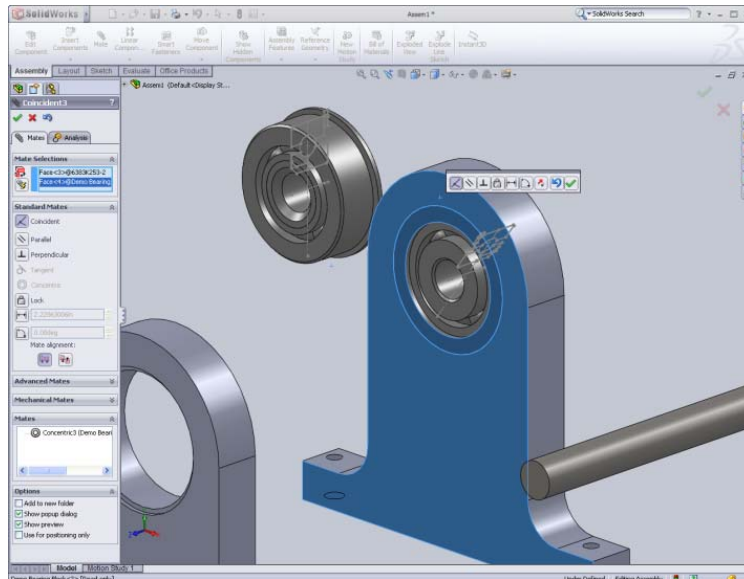
Pick the two places that will be joined



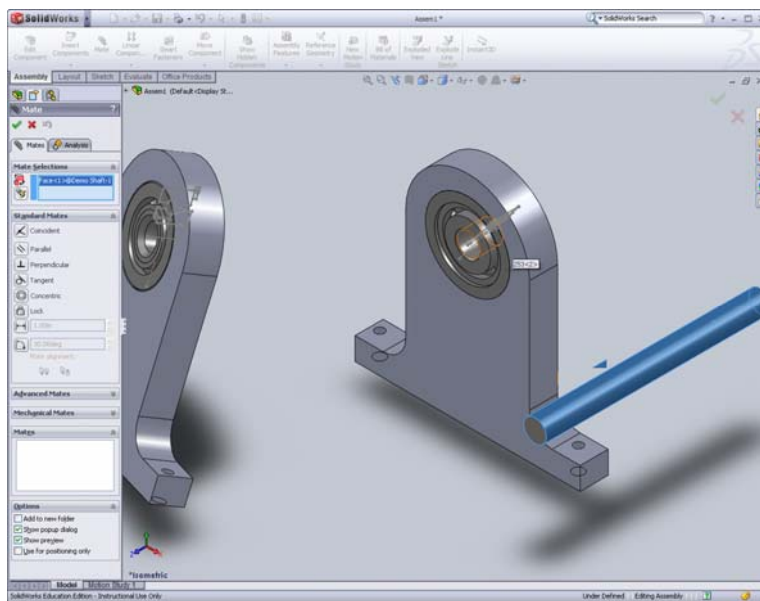
Outer bearing to inner block

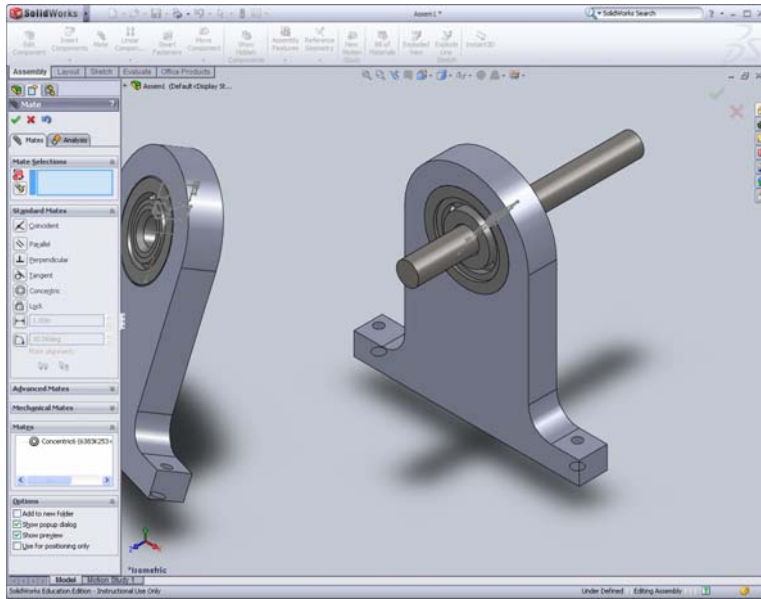


Bearing face to block face

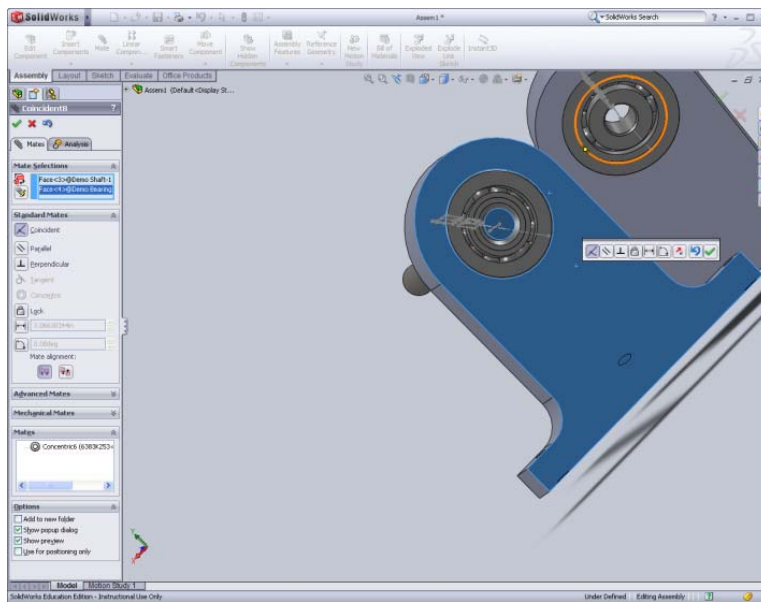


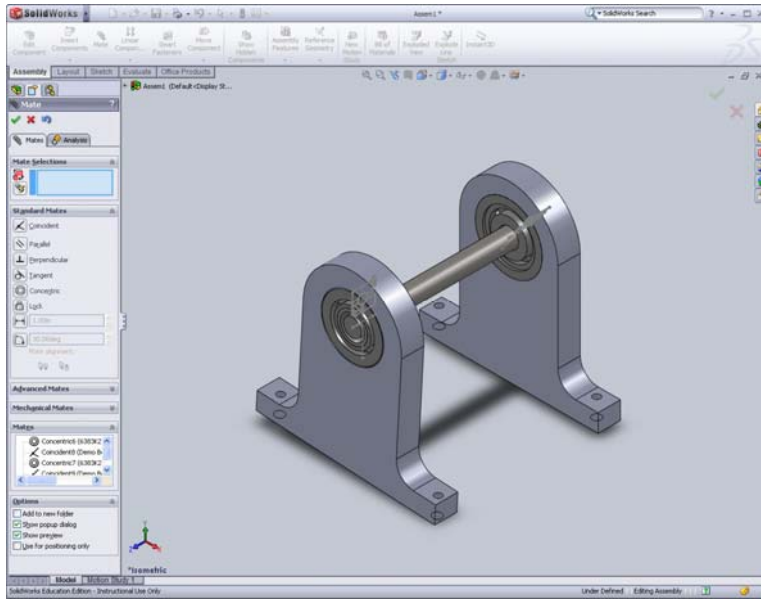
Shaft to bearing



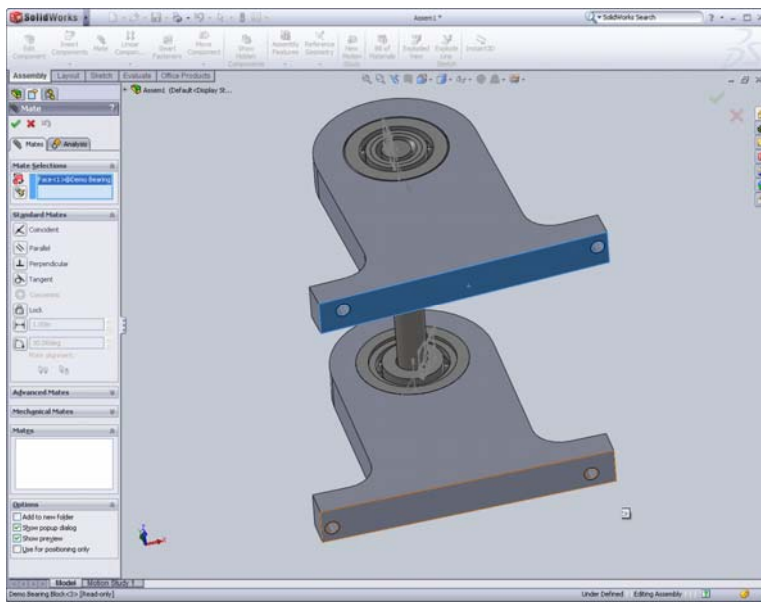


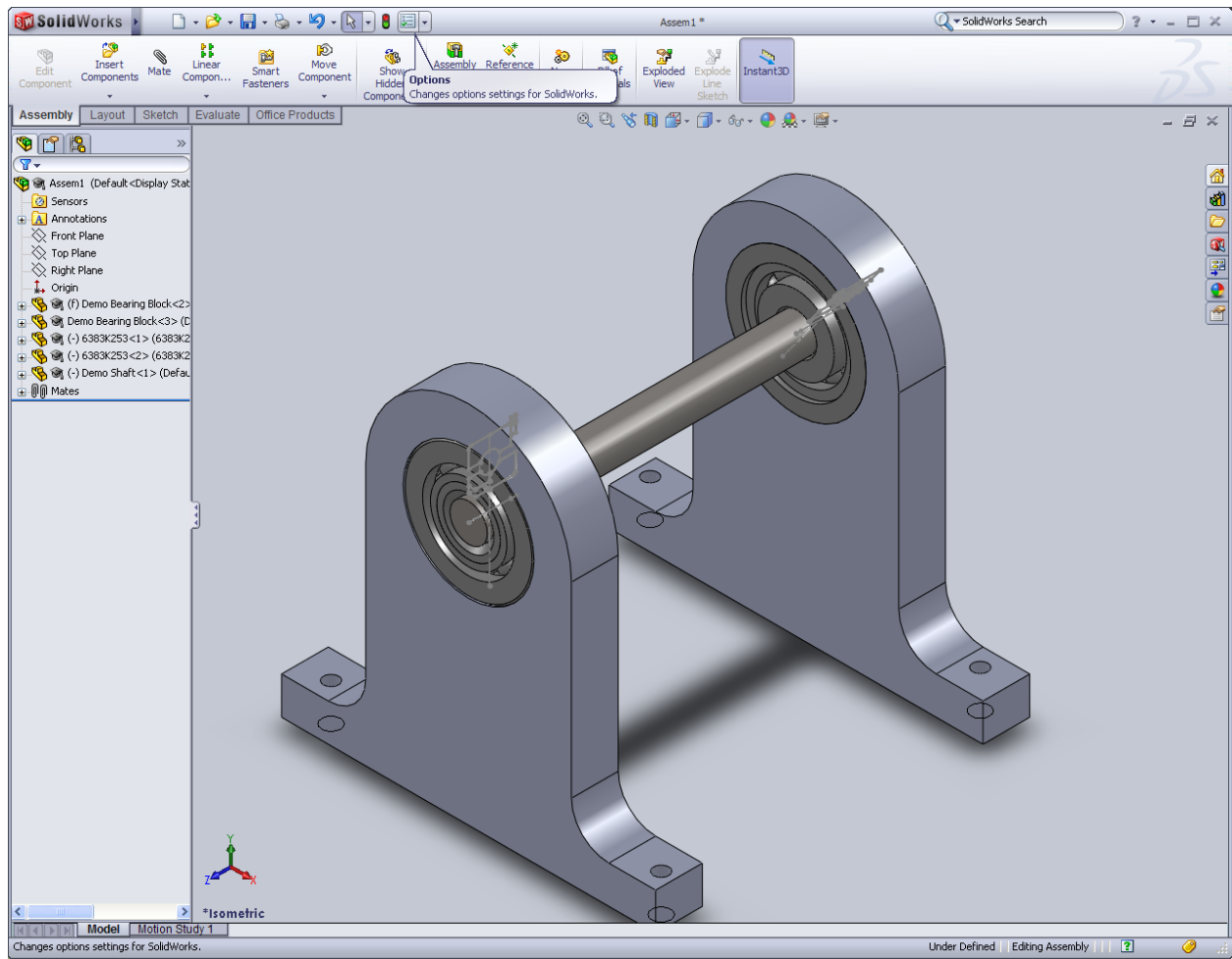
Shaft face to block face





Block face to block face





Save Assembly