**EXERCISE 3 - SKETCH FEATURES**

**Goal**

The goal of this exercise is to become familiar with using the Sketcher.

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**Task 1:** Create a sketch feature in a new part.

- Start a new session of Creo Parametric
- Be sure you have added and applied the configuration options listed on page 4
- Click File, New or click the icon □
- Enter < 4455-001 > for the name of the part (don't include the <> symbols)
- Click OK in the New dialog box
- Select the RIGHT datum plane for the sketch plane
- Click the Sketch icon □ in the Model tab of the ribbon
- Notice the system has selected the TOP datum plane for the Top orientation reference (the positive surface normal of the TOP datum plane points to the top of the monitor)
- Notice the two dashed lines indicating the references selected for you by the system (these are not centerlines, these are references)
- Click the Rectangle icon then sketch as shown below, align two sides of the rectangle to the dashed reference lines, click the middle mouse button, ignore the dimensions for now

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**Tip**

If you spin the model while in the sketcher, click the Sketch View icon in the Graphics toolbar to return to the sketch view.
- Click the Select icon

- Double click each dimension and enter the values as shown in the previous figure

- Click the checkmark to complete the sketch

- Orient the model using the Standard Orientation

- The result is shown below

- Save the part

- Read the message in the message area

- Click File, Close or click the icon

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**Task 2: Create a sketch feature in a new part.**

- Click File, New or click the icon

- Enter < 4455-200 > for the name of the part then click OK in the dialog box

- Select the FRONT datum plane for the sketch plane

- Click the Sketch icon

- Notice the system has selected the RIGHT datum plane for the Right orientation reference (the positive surface normal of the RIGHT datum plane points to the right side of the monitor)

- Notice the two dashed lines indicating the references selected for you by the system (these are not centerlines, these are references)

- Click the Line icon

- Sketch a horizontal line aligned with the horizontal reference; click the left mouse button to start the line and another left click to end the line

- Click the middle mouse button to end the line chain
• Sketch another horizontal line above the first line and equal length as shown below
• Click the 3 Point / Tangent End icon then sketch two tangent arcs as shown below

![Diagram of a horizontal line and two tangent arcs]

• Click the Normal icon then create two dimension and enter the values as shown above
• Click the checkmark to complete the sketch
• Orient the model using the Standard Orientation
• The result is shown below

![Diagram showing the result]

• Save the part
• Read the message in the message area
• Click File, Close or click the icon
Task 3: Create a sketch feature in a new part.

- Click File, New or click the icon 
- Enter <4455-201> for the name of the part then click OK in the dialog box
- Select the RIGHT datum plane for the sketch plane
- Click the Sketch icon 
- Notice the system has selected the TOP datum plane for the Top orientation reference (the positive surface normal of the TOP datum plane points to the top of the monitor)
- Sketch a vertical construction centerline aligned with FRONT
- Sketch a horizontal line for the base of the triangle using the Line icon start on the left of the centerline and sketch across to the right side of the centerline, the system will 'snap' when the sketch is symmetrical
- Complete the triangle with two additional lines
- Click the Circular Fillet icon then select two locations, one near each corner
- Click the Equal icon then select the three arcs
- Create the dimensions and enter the values as shown below
- Click the checkmark to complete the sketch
- Orient the model using the Standard Orientation
- Save the part
- Read the message in the message area
- Click File, Close or click the icon

**Task 4:** Create a sketch feature in a new part.

- Click File, New or click the icon
- Enter < 4455-202 > for the name of the part then click OK in the dialog box
- Select the RIGHT datum plane for the sketch plane
- Click the Sketch icon
- Sketch the geometry as shown below

- Create the dimensions and enter the values as shown above
- Flex the sketch by modifying the 800 dimension to 1200
- Change several other dimensions to see how the sketch reacts
- Click the checkmark to complete the sketch
Task 5: Create a sketch feature in a new part.

- Click File, New or click the icon 📚.
- Enter < 4455-203 > for the name of the part then click OK in the dialog box.
- Select the FRONT datum plane for the sketch plane.
- Click the Sketch icon 🖋️.
- Sketch a vertical geometry centerline aligned with RIGHT ⬇️.
- Sketch the right half of the sketch, use the Circular Trim icon 🔋 to create the arcs, ignore the dimensions for now.
- Click the Select icon 👀 then select all the entities (press and hold the left mouse button and drag a box around the entire sketch).
- Click the Mirror icon 💪 then select the centerline as shown below.

- Orient the model using the Standard Orientation.
- Save the part.
- Read the message in the message area.
- Click File, Close or click the icon 📚.
• Create the dimensions as shown in the previous figure, ignore the default values for the dimensions as they are created

• Click the Select icon \( \text{Select} \) then drag a box around all the dimensions using the left mouse button

• Click the Modify icon \( \text{Modify} \)

• Remove the checkmark next to Regenerate in the Modify Dimensions dialog box

• Enter the appropriate value for each dimension as shown in the previous figure

• Click the checkmark in the Modify Dimensions dialog box

• Notice the sketch updates with the new dimension values

• Click the checkmark to complete the sketch

• Orient the model using the Standard Orientation

• The result is shown below

![Sketch Features Diagram]

• Save the part

• Click File, Close or click the icon \( \text{File Close} \)

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**Task 6:** Create a sketch feature in a new part.

• Click File, New or click the icon \( \text{File New} \)

• Enter \(< 4455-004 >\) for the name of the part then click OK in the dialog box

• Select the TOP datum plane for the sketch plane

• Click the Sketch icon \( \text{Sketch} \)

• Notice the two dashed lines indicating the references selected for you by the system (these are not centerlines, these are references)

• Sketch a horizontal and a vertical construction centerline aligned with the appropriate references
• Sketch an arc using **Center and Ends** to locate the center of the arc at the intersection of the two centerlines as shown below.

• Sketch three tangent arcs to complete the shape as shown below.

• Add the constraints and dimensions as shown below; as dimensions are created ignore the default values.
- Click the Select icon
- Drag a box around all the dimensions using the left mouse button
- Click the Modify icon
- *Remove* the checkmark next to *Regenerate* in the *Modify Dimensions* dialog box
- Enter the appropriate value for each dimension as shown in the previous figure
- Click the checkmark in the *Modify Dimensions* dialog box
- Click the Shade Closed Loops icon to toggle off the shading
- Notice the closed loop is no longer shaded
- Click the Shade Closed Loops icon again to toggle on the shading
- Click the Feature Requirements icon
- Read the information then *Close* the Feature Requirements dialog box
- Click the checkmark to complete the sketch
- Orient the model using the Standard Orientation
- The result is shown below

- Save the part
- Click File, Close or click the icon
Task 7: Create three sketch features in a new part.

- Click File, New or click the icon 
- Enter <4455-205> for the name of the part then click OK in the dialog box
- Select the FRONT datum plane for the sketch plane
- Click the Sketch icon
- Sketch and dimension the geometry as shown below

![Sketch 1]

- Modify the dimension values as shown above
- Click the checkmark to complete the sketch
- Click the Sketch icon
- Click Use Previous in the Sketch dialog box
- Sketch the geometry including a vertical construction centerline as shown below

![Sketch 2]
- Create the dimensions and enter the values as shown in the previous figure
- Click the checkmark to complete the sketch
- Click **Sketch** icon
- Click **Use Previous** in the **Sketch** dialog box
- Sketch the geometry including a horizontal construction centerline as shown below

![Sketch Example]

- Create the dimensions and enter the values as shown above
- Click the checkmark to complete the sketch
- Orient the model using the Standard Orientation
- The result is shown below

![3D Sketch Result]

- **Save the part**
- Click **File, Close** or click the icon
Task 8: Create a sketch feature in a new part.

- Click File, New or click the icon.
- Enter <4455-206> for the name of the part then click OK in the dialog box.
- Select the FRONT datum plane for the sketch plane.
- Click the Sketch icon.
- Sketch and dimension the geometry including a vertical geometry centerline as shown below (the geometry centerline is used because we want to revolve this sketch in the next exercise).

To create these dimensions, click the Normal icon, select the entity, select the centerline, select the entity again, then click middle to place the dimension.

- Click the checkmark to complete the sketch.
- Orient the model using the Standard Orientation.
- Save the part.
- Click File, Close or click the icon.

Use Circular Fillet to create this arc then dimension to the sharp corner here.
Task 9: Create a sketch feature in a new part.

- Click File, New or click the icon.
- Enter <4455-207> for the name of the part then click OK in the dialog box.
- Select the TOP datum plane for the sketch plane.
- Click the Sketch icon.
- Sketch and dimension the geometry including a vertical construction centerline as shown below.

- Click the checkmark to complete the sketch.
- Orient the model using the Standard Orientation.
- Save the part.
- Click File, Close or click the icon.