Alibre Atom3D Exercise Manual
Introduction to creating parts and assemblies
Creating the link

New concepts in this lesson include:

• Setting Sketch Grid options
• Creating simple sketch figures by snapping to the sketch grid
• Extruding sketches into solid bodies
• Interacting with the view using Pan, Zoom, and Rotate
• Changing part color
1. Open a new **Part** workspace from the Alibre Atom3D **Home** window.
1. Click on the Alibre Gem
2. Click on System Options
1. Under Parts/Assemblies>Sketching, enable the Display Grid and Snap to Grid options.
2. Change the values to .500 inch for X, Y, and Z.
3. Press the OK button.
1. If necessary, click on the Part Modeling tab in the ribbon.
2. Click on the XY Plane in the Design Explorer.
3. Click on the Activate 2D Sketch icon.
1. Position the mouse cursor near the **Origin** point. Roll the mouse wheel forward to zoom in. The view of the work area should be similar to the screen capture (approximately 5 grid squares visible, vertically).

2. Click on the **Circle** tool on the **Sketch Figures** tab in the ribbon.
1. Position the mouse cursor over the **Origin** point. The point's color change confirms that the mouse cursor is positioned properly. *Click and release* to create the center point of the circle.

2. Move the mouse cursor a distance of one grid square to the right, along the **X Axis**. *Click and release* to complete the circle; this circle should be 1.000”. Confirm that the results are similar to the screen capture.
1. Click on the **Part Modeling** tab in the ribbon
2. Click on the **Extrude Boss** icon in the upper-right corner of the screen.
1. In the Extrude Boss dialog box, confirm that the Sketch entry box contains Sketch<1>.
2. Change the Type drop-down menu to the Mid Plane option.
3. Confirm that the Depth entry box has a value of 5 inches. Enter that value if it does not.
4. Click OK.
1. Confirm similar results for the Extrude Boss operation.
2. Select the ZX Plane in the Design Explorer.
3. Click Activate 2D Sketch icon in the Part Modeling tab.
1. The view of the work area should be similar to the screen capture.
2. Click on the **Circle tool**.
1. Position the mouse cursor over the **Origin** point.  *Click and release* to create the center of the circle.

2. Move the mouse cursor to the position indicated; this circle should also be 1.000”. *Click and release* to complete the circle.
1. Press the spacebar and a context sensitive menu will appear.
2. Click on Extrude Boss icon.
1. In the **Extrude Boss** dialog box, confirm that the **Sketch** entry box contains **Sketch<2>**.
2. Change the **Type** drop-down menu to the **Mid Plane** option.
3. Confirm that the **Depth** entry box has a value of **5 inches**. Enter that value if it does not.
4. Click OK.
1. Confirm similar results for the **Extrude Boss** operation.
1. You can pan the model by holding down the middle mouse button (scroll button) and moving the mouse. This will keep the model in the same orientation in the workspace but move it left, right, up, and down.

2. Press and hold the middle mouse button and move the part in the workspace to the left.
1. By scrolling the middle mouse button you can zoom into or away from the current model. Moving your finger towards you will zoom out and moving your finger away from you will zoom in. Note, the zoom action will be centered around the location of the mouse.

2. With the mouse cursor in the center of the workspace, zoom away from the model.
1. Holding both mouse buttons while moving the mouse cursor will rotate the model (about the center of volume) on the screen allowing you to view the model from a different angle.

2. After rotating the model, press the Isometric icon in the ribbon.
1. Right click anywhere on the part.
2. From the pop-up menu select **Color Properties**.
3. Click on the **Color** button in the **Color Properties** dialog box.
1. Select a bright green color in the **Color** dialog box by clicking on one of the colored squares in the **Basic colors** area.
2. Click **OK** in the **Color** dialog box.
3. Click **OK** in the **Color Properties** dialog box.
1. Confirm similar results for the color change.
2. Save this Part to a convenient location on your hard drive with the name: **Link.** (not shown).
3. Close the Part workspace (not shown).
Creating the crank

New concepts in this lesson include:

- Using the Offset Sketch tool
- Using the Trim Sketch tool
- Creating and removing 3D material with extrusions
- Creating rounded corners with Fillets
- Sketching on model faces
1. Open a new Part workspace from the Alibre Atom3D Home window.
1. If necessary, click on the **Part Modeling** tab in the ribbon.
2. Click on the **XY Plane** in the **Design Explorer**.
3. Click on the **Activate 2D Sketch** icon.
1. Position the mouse cursor near the **Origin** point. *Click and hold the middle mouse button,* then drag the view downwards and to the left. Release the mouse button.
1. On the Part Modeling tab, click **Activate 2D Sketch** and select the **XY Plane** in the Design Explorer (not shown).

2. Click on the **Rectangle** tool in the **Sketch Figures** group in the **2D Sketching** tab.
1. Position the mouse cursor over the **Origin** point. Click and release to create the first corner of the rectangle.

2. Move the mouse cursor upwards and to the right.

3. The measurement indicator at the bottom of the screen should be **Width: 3.500** and **Height: 5.000**. Click and release to complete the rectangle.
1. Make sure you are in the **2D Sketching** tab.
2. Select the **Offset** command.
3. Click to select the top, left and bottom segments of the rectangle. The segments will change color upon selection. The segments will change.
4. Note that as you select them, the segments will appear in the **Figures to Offset** box (not shown).
1. Confirm the preview of the **Offset** results.
2. Confirm the **Distance** value in the **Offset** dialog box is **0.5 inches**.
3. Click **OK**.
1. Confirm the result of the **Offset** operation. 3 of the 4 figures of the rectangle should be offset by .5".
1. Make sure you have the 2D Sketching tab active.
2. Select the Trim command.
3. Click on the line segment indicated. The segment will get deleted.
1. Confirm the results of the Trim operation.
2. Click on the Deactivate Sketch icon in the ribbon.
3. Click the Extrude Boss icon.

When you click on the Deactivate Sketch icon in the 2D Sketching tab you will automatically be brought to the Part Modeling tab.
1. Confirm that Sketch<1> is the Sketch to be extruded.
2. Select Mid Plane as the Type of extrusion.
3. Enter a value of 1.5 inches for the Depth.
4. Click OK.
1. Confirm a similar result for the **Extrude Boss** operation.
1. Select the Fillet icon.
1. Click on the four edges indicated. A preview of the Fillet operation is displayed.
2. Enter a value of 0.75 inch for the Radius in the Fillet Edges dialog box.
3. Click Apply then Close.
1. Confirm similar results for the **Fillet** operation.
2. Click on the face indicated.
3. Click **Activate 2D Sketch**.
1. Confirm that the view of the work area is similar to the screen capture.
2. Click on the **Circle** tool.
1. Move the mouse cursor to the position shown (2.50" to the right of the origin). Click and release to create the center point of the circle.

2. Move the mouse cursor to the right a distance of one grid square; this will create a Ø1.000”. Click and release to complete the circle.
1. Select the **Concentric Constraint** option.
2. Click on the perimeter of the circle.
3. Click on the circular edge indicated.
1. Confirm similar results for the **Concentric** constraint. Take note of the Concentric constraint symbol that appears near the circle’s center. The circle and the circular edge should be concentric, and the circle’s center point should move to the right.

2. Click on **Extrude Cut** tool in the **Cut (Remove Material)** group in the **Part Modeling** tab (or click on the **Deactivate Sketch** icon).
1. In the Extrude Cut dialog box, confirm that the Sketch entry box displays Sketch<2>.
2. Select Through All as the Type from the drop down menu.
3. Click OK.
1. Confirm similar results for the **Extrude Cut** operation.
1. Hold down the left and right mouse buttons together over the model to **Rotate**.
2. Position the mouse cursor as shown. Click and hold the mouse buttons while moving the mouse to the right (approximately as shown). Release the mouse buttons.
1. Confirm similar results for the view rotation.
2. Click on the **Part Modeling** tab in the ribbon and click on the **Activate 2D Sketch** icon.
3. Click on the face indicated.
1. The view of the work area should be similar to the screen capture.
1. Click on the **Circle** tool.
2. Position the mouse cursor 2.50” above the origin. Click and release to create the center of the circle.
3. Move the mouse cursor one grid square to the right; this will create a Ø1.000” circle. Click and release to complete the circle.
1. Click on the Part Modeling tab in the ribbon.
2. Click on the Extrude Boss icon.
1. Confirm that Sketch<3> is the Sketch to be extruded.
2. Confirm that To Depth is the Type of extrusion.
3. Enter a value of 6.0 inches in the Depth entry box.
4. Click OK.
1. Confirm similar results for the **Extrude Boss** operation.
2. Click on the indicated face.
3. Click **Activate 2D Sketch**.
1. The view of the work area should be similar to the above image.
2. Position the mouse near the center of the screen, hold down the Middle Mouse button, and drag the mouse to the left.
1. Click on the **Rectangle** tool.

2. Position the mouse cursor as shown. Click and release to create the first corner of the rectangle.

3. Move the mouse cursor to the right and upwards until the measurement indicator attached to the cursor displays **Width: 10.000** and **Height: 1.000**. Click and release to complete the rectangle.
1. Click on the Part Modeling tab and click on the Extrude Boss icon.
2. Confirm that Sketch<4> is the Sketch to be extruded.
3. Confirm that To Depth is the Type of extrusion.
4. Enter a value of 0.5 inches for the Depth.
5. Click OK.
1. Click on the indicated face.
2. Click Activate 2D Sketch.
1. Click on the **Circle** tool.

2. Position the mouse cursor as shown. Click and release to create the center of the circle.

3. Move the mouse cursor to the right a distance of one grid square. Click and release to complete the circle.
1. Click on the Part Modeling tab and click on the Extrude Boss icon.
2. Confirm that Sketch<5> is the Sketch to be extruded.
3. Confirm that To Depth is the Type of extrusion.
4. Enter a value of 5 inches for the Depth.
5. Click OK.
1. Confirm similar results for the **Extrude Boss** operation.
1. Select the **Fillet** icon.
2. Click on the face indicated.
1. Confirm the preview of the Fillet operation.
2. Enter a value of 0.5 inches for the Radius in the Fillet Edges dialog box.
3. Click Apply then Close.
1. Select the **Part Color** icon.
2. Click on the **Color** button in the **Color Properties** dialog box.
3. Select a medium blue color in the **Color** dialog box.
4. Click **OK** in the **Color** dialog box.
5. Click **OK** in the **Color Properties** dialog box.
1. Confirm similar results for the color change.
2. Save this Part to the same location as the Link with the name: Crank (not shown).
3. Close the Part workspace (not shown).
Creating the flywheel

New concepts in this lesson include:

• Using patterns to decrease your work
• Creating 3D material with the Revolve Boss tool
1. Open a new Part workspace from the Alibre Atom3D Home window.
1. Click on the Part Modeling tab in the ribbon.
2. Click on the YZ Plane in the Design Explorer.
3. Click on the Activate 2D Sketch icon.
1. Click on the **Rectangle** tool.

2. Position the mouse cursor over the **Origin point**. *Click and release* to create the first corner of the rectangle.

3. Move the mouse cursor upwards and to the right until the measurement indicator attached to the cursor displays: **Width: 5.000** and **Height: 3.500**. *Click and release* to complete the rectangle.
1. Select the **Offset** command.
2. Click to select the **right**, **left**, and **bottom** segments of the rectangle. The segments will change color upon selection.
1. Confirm the preview of the Offset results.
2. Confirm the Distance: value in the Offset dialog box is 0.5 inch.
3. Click OK.
1. Confirm the result of the **Offset** operation.
1. Select the **Trim** command.
2. Click on the line segment indicated.
1. Confirm the result of the **Trim** operation.
2. Click on the **Part Modeling** tab in the ribbon (or click on the **Deactivate Sketch** icon).
3. Click on the **Extrude Boss** icon in the upper-right corner of the screen.
1. Confirm that Sketch<1> is the Sketch to be extruded.
2. Select Mid Plane as the Type of extrusion.
3. Enter a value of 1.5 inches for the Depth.
4. Click OK.
1. Confirm similar results for the **Extrude Boss** operation.
1. Click on the **Fillet** tool.
2. Move the **Fillet Edges** dialog box such that the entire part is visible.
3. Click on the four edges indicated.
1. A preview of the Fillet operation is displayed.
2. In the Fillet Edges dialog box, change the **Radius**: value to **0.75 inch**.
3. Click **Apply** then **Close**.
1. Click on the face indicated.
2. Click on **Activate 2D Sketch**.
1. Click on the **Circle** tool.
2. Move the mouse cursor to the position shown. **Click and release** to create the center point of the circle.
3. Move the mouse cursor to the right a distance of one grid square. **Click and release** to complete the circle.
1. Select the **Concentric** option.
2. Click on the perimeter of the circle.
3. Click on the circular edge indicated.
1. Confirm similar results for the **Concentric** constraint. Take note of the **Concentric** constraint symbol that appears near the circle's center.

2. Click on the **Part Modeling** tab.

3. Click on **Extrude Cut**.
1. In the **Extrude Cut** dialog box, confirm that the **Sketch** entry box displays **Sketch<2>**.
2. Select **Through All** from the **Type** drop down menu.
3. Click **OK**.
1. Confirm similar results for the **Extrude Cut** operation.
1. Rotate the model by holding down both of the mouse buttons and dragging.

2. *Click and hold* both mouse buttons while moving the mouse cursor upward (approximately as shown). Release the mouse buttons.
1. Click on the face indicated.
2. Click on **Activate 2D Sketch**.
1. The view of the work area should be similar to the screen capture.
1. Click on the **Circle** tool.

2. Position the mouse cursor as shown. Count the grid squares to confirm the location as the center of the rectangular face in the background. **Click and release** to create the center of the circle.

3. Move the mouse cursor one grid square to the right. **Click and release** to complete the circle.
1. Click on the Part Modeling tab in the ribbon.
2. Click on the Extrude Boss icon.
3. Confirm that Sketch<3> is the Sketch to be extruded.
4. Confirm that To Depth is the Type of extrusion.
5. Enter a value of 6.0 inches for the Depth.
6. Click OK.
1. Confirm similar results for the **Extrude Boss** operation.
2. Click on the face indicated.
3. Click on **Activate 2D Sketch**.
1. The view of the work area should be similar to the screen capture.
2. Position the mouse cursor over the visible, circular face. Roll the mouse wheel backwards to zoom out until the visible geometry is roughly half of its original size.
1. The view of the work area should be similar to the screen capture.
1. Click on the Rectangle tool.
2. Position the mouse cursor as shown. *Click and release* to create the first corner of the rectangle.
3. Move the mouse cursor downwards and to the right until the measurement indicator displays **Width: 1.000** and **Height: 4.000**. *Click and release* to complete the rectangle.
1. Click on the Part Modeling tab in the ribbon.
2. Click on Extrude Boss.
3. Confirm that Sketch<4> is the Sketch to be extruded.
4. Confirm that To Depth is the Type of extrusion.
5. Enter a value of 0.5 inch for the Depth.
6. Click OK.
1. Confirm similar results for the **Extrude Boss.**
1. Select the Reference Axis icon.
2. Click on the face indicated in the screen capture: the cylindrical face of the shaft.
3. A preview of the Axis is displayed.
4. Click Apply then Close.
1. Click on the **Circular Pattern** icon.
2. Click on the face indicated.
1. Confirm that Extrusion<5> appears in the Features to Pattern: entry box.
2. In the Copies: entry box, enter 6.
3. Select the Complete Revolution option.
4. Click on Axis 6 in the Design Explorer.
5. Click OK.
1. Click on the YZ Plane in the Design Explorer.
2. Click on Activate 2D Sketch.
1. Click on the **Rectangle** tool.

2. Position the mouse cursor as shown.  *Click and release* to create the first corner.

3. Move the mouse to the position indicated.  *Click and release* to complete the rectangle.
1. Click on the Part Modeling tab in the ribbon.
2. Click on the Revolve Boss tool.
3. Confirm that Sketch<5> appears in the Sketch to revolve: entry box.
4. Click in the Axis entry box and select Axis 6.
5. Click OK.
1. Confirm similar results for the **Revolve Boss** operation.
1. From the Quick Start menu select Isometric (+X, +Y, +Z) View.
1. Confirm similar results for the view re-orientation.
1. Select the **Part Color** icon.
2. Click on the **Color** button in the **Color Properties** dialog box.
3. Select a light blue color in the **Color** dialog box.
4. Click **OK** in the **Color** dialog box.
5. Click **OK** in the **Color Properties** dialog box.
1. Save this Part to the same location as the Link and Crank with the name: Pulley (not shown).
2. Close the Part workspace (not shown).
Creating the base

New concepts in this lesson include:

• Turning off the Sketch Grid
• Using Dimensions and Sketch Constraints to place and size sketch figures
1. Open a new **Part** workspace from the Alibre Atom3D **Home** window.
1. If necessary, click on the Part Modeling tab in the ribbon.
2. Click on the XY Plane in the Design Explorer.
3. Click on the Activate 2D Sketch icon.
This lesson will utilize another common way to create geometry in Alibre Design (relative to the previous lessons in this tutorial). Extensive use of constraints, especially dimensions (also called parameters) characterize this method of creating and controlling geometry.

1. Go to System Options>Parts/Assemblies>Sketching.
2. Turn off the Display Grid and Snap to Grid options and click OK.
1. Click on the Rectangle tool.
2. Position the mouse cursor over the Origin point. Click and release to create the first corner.
3. Move the cursor upwards and to the right an arbitrary distance (approximately as shown). Click and release to complete the rectangle.
1. Click on the **Dimension** tool.
2. Click on the bottom edge of the rectangle
1. Establish the position of the dimension by moving the cursor downward, then click and release.
1. The **Dimension** tool displays the precise length of the line segment as it currently exists.
1. Enter 12 in the entry box.
2. Click on the green checkmark or press the Enter key on the keyboard.
1. Confirm the result of the **Dimension**. Take note that dimensions in Alibre Design control the elements to which they're applied, rather than merely documenting them.

2. **Click Reset View** to center the sketch in the workspace.
1. Click on the left segment of the rectangle.
1. Establish the position of the dimension by moving the cursor to the left, then click and release.
1. The **Dimension** tool displays the precise length of the line segment as it currently exists.
1. Enter 10 in the entry box.
2. Click on the green checkmark or press the *Enter* key on the keyboard.
3. Click *Reset View* to center the sketch in the workspace.
1. Click on the Part Modeling tab in the ribbon.
2. Click on Extrude Boss.
1. Confirm that Sketch<1> is the Sketch to be extruded.
2. Confirm To Depth is the Type of extrusion.
3. Enter a value of 1 inch for the Depth.
4. Click OK.
1. Confirm similar results for the **Extrude Boss** operation.
1. Click on the face indicated.
2. Click on **Activate 2D Sketch**.
1. Click on the **Rectangle** tool.

2. Position the mouse cursor as shown (approximately). First click and release to create the first corner.

3. Move the mouse cursor to the position indicated (approximately). Click and release to complete the rectangle.
1. Click on the Line sketch tool.
1. Use the **Line** tool to draw an angled rectangle as shown (approximately).
1. Click on the **Dimension** tool.
2. Click on the indicated segment to dimension.
3. Position the dimension as shown (approximately). Click and release to place the dimension.
1. Enter 2 for the dimension value. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).
1. The **Dimension** tool should still be enabled from the previous step. If it is not still enabled then click on the **Dimension** tool.

2. Click on the indicated segment to dimension.

3. Position the dimension as shown (approximately). Click and release to place the dimension.
1. Enter 3 for the dimension value. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).
1. The **Dimension** tool should still be enabled from the previous steps. If it is not still enabled then click on the **Dimension** tool.

2. Click on the indicated segment to dimension.

3. Position the dimension as shown (approximately). Click and release to place the dimension.
1. Enter 2 for the dimension value. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).
1. The **Dimension** tool should still be enabled from the previous steps. If it is not still enabled then click on the **Dimension** tool.

2. Click on the indicated segment to dimension.

3. Position the dimension as shown (approximately). Click and release to place the dimension.
1. Enter 3 for the dimension value. Click on the green checkmark or press the `Enter` key on the keyboard (not shown).
1. Click on the **Dimension** tool.
2. Click on the indicated segment.
3. Click on the indicated segment. The dimension now applies to the angle between the two selected segments.
4. Position the dimension as shown (approximately). Click and release to place the dimension.
1. Enter 135 for the angular dimension value. Click on the green checkmark or press the Enter key on the keyboard (not shown).
1. Confirm the results of the angular dimension. The angled line will rotate relative to the horizontal line.
1. From the **Constraints** group, select the **Collinear** option.
1. Click on the indicated line segment.
2. Click on the indicated Edge to complete the Collinear constraint.
1. Confirm the results of the **Collinear** constraint.
1. From the **Constraints** group, select the **Coincident** option.
1. Click on the indicated **Node**. Nodes are found at the endpoints of all sketch entities and change color upon selection.

2. Click on the indicated **Edge** to complete the **Coincident** constraint.
1. Confirm the result of the previous **Coincident** constraint.
2. Click on the indicated **Node**.
3. Click on the indicated **Edge** to complete the **Coincident** constraint.
1. Click on the **Dimension** tool.
2. Click on the segment indicated.
3. Click on the **Edge** indicated.
4. Position the dimension as shown. Click and release to place it.
1. Enter 4.5 as the dimension value. Click on the green checkmark or press the Enter key on the keyboard (not shown).
1. Click on the **Part Modeling** tab in the ribbon and click on the **Extrude Boss** icon.

2. Confirm that **Sketch<2>** is the **Sketch** to be extruded.

3. Confirm **To Depth** is **Type** of extrusion.

4. Enter a value of **5 inches** in the **Depth** entry box.

5. Click **OK**.
1. Confirm similar results for the **Extrude Boss** operation.
1. Click on the Fillet icon in the Geometry Transform group.
2. Position the Fillet Edges dialog box such that the entire part is visible.
3. Click on the edges indicated.
1. A preview of the Fillet operation is displayed.
2. In the Fillet Edges dialog box, change the Radius value to 1 inch.
3. Click Apply then Close.
1. Right click on the face indicated.
2. From the popup menu select **Activate 2D Sketch**.
1. Click on the **Circle** tool.
2. Position the mouse cursor as shown (approximately). Click and release to create the center.
3. Move the mouse to the position indicated (approximately). Click and release to complete the circle.
1. Select the **Concentric** sketch constraint.
2. Click on the perimeter of the circle.
3. Click on the circular edge shown in the screen capture.
1. Confirm that the result of the **Concentric** constraint is similar to the screen capture. Take note of the **Concentric** constraint symbol.
2. Right click and select **Dimension** from the popup menu.
1. Click on the perimeter of the circle.
2. Position the dimension approximately as shown. Click and release to place the dimension. Enter a value of 1 for the dimension.
3. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).
1. Click on the **Part Modeling** tab in the ribbon and click on the **Extrude Cut** icon.

2. Confirm that the **Sketch<3>** is the **Sketch** to be extruded.

3. Select **Through All** as the **Type** of extrusion.

4. Click **OK**.
1. From the Quick Access toolbar
2. Select the first **Isometric** option.
1. Click on the face indicated and enter into sketch mode.

You can do this either by clicking on the Part Modeling tab in the ribbon and then click on the Activate 2D Sketch icon, or you can right click in the workspace and select Activate 2D Sketch from the popup menu.
1. Position the mouse cursor over the face indicated; and roll the mouse wheel forward until the visible geometry is roughly twice as large.
1. Click on the Circle tool.
2. Position the mouse cursor as shown (approximately). Click and release to create the center.
3. Move the mouse cursor to the position indicated (approximately). Click and release to complete the circle.
1. Select the **Concentric** sketch constraint.
2. Click on the perimeter of the circle.
3. Click on the circular edge shown in the screen capture.
1. Confirm similar results for the **Concentric** constraint. Take note of the Concentric constraint symbol.

2. Enter into dimension mode. (Not Shown)
   
   You can do this either by clicking on the **Standard Dimension** icon, or you can right click in the workspace and select **Dimension** from the popup menu.
1. Click on the perimeter of the circle.
2. Position the dimension approximately as shown. Click and release to place the dimension.
3. Enter a value of 1 for the dimension. Click on the green checkmark or press **Enter** on the keyboard (not shown).
1. Click on the **Part Modeling** tab in the ribbon and click on the **Extrude Cut** icon.
2. Confirm that the Sketch<4> is the Sketch to be extruded.
3. Select **To Geometry** as the **Type** of extrusion.
1. With the **Extrude Cut** dialog box still open, rotate the model (press and hold both mouse buttons while moving the mouse cursor) until it is in the approximate orientation.

2. Click in the **End Target** textbox (it will highlight light blue).

3. Click on the face indicated above.

4. Click on the **OK** button.
1. Confirm similar results for the **Extrude Cut** operation.
1. From the Quick Access toolbar
2. Select the second **Isometric** option.
1. Confirm similar results for the view re-orientation.
1. Select the Part Color icon.
2. Click on the Color button in the Color Properties dialog box.
3. Select a light red color in the Color dialog box.
4. Click OK in the Color dialog box.
5. Click OK in the Color Properties dialog box.
1. Confirm similar results for the color change.
2. Save the Part to the same location as the Link, Crank and Pulley, with the name Base (not shown).
3. Close the Part workspace (not shown).
Putting the parts together

New concepts in this lesson include:

- Inserting Parts into the Assembly workspace
- Using the Quick Constraint tool to fit the parts together into a moving assembly
1. Open a new **Assembly** workspace from the Alibre Atom3D **Home** window.
1. The **Insert Part/Subassembly** dialog will open automatically.
2. Select the **Pulley** in the **Insert Part/Subassembly** dialog box.
3. Click **OK**.
1. Do not click any mouse buttons yet, but move the mouse around. Take note of the transparent copy of the Pulley attached to the cursor.
1. Position the mouse cursor as shown. Every mouse click in the workspace will result in an additional part being inserted into the assembly. Click and release once to create a single instance of the Pulley at the location indicated.

2. Click the **Finish** button in the **Inserting** dialog box.

3. Click the **Front View** button from the Quick Access toolbar.
1. Confirm that the view of the workspace is similar to the screen capture.
1. From the Insert group, select the Insert Design icon.
2. Select the Link in the Insert Part/Subassembly dialog box.
3. Click Open.
1. Position the mouse cursor as shown. Click and release once to create a single instance of the Link at the location indicated.

2. Click the Finish button in the Inserting dialog box.

3. Click the Front View button from the Quick Access toolbar.
1. The view of the work area should be similar to the screen capture.
1. From the **Insert** group, select the **Insert Design** icon.
2. Select the **Crank** in the **Insert Part/Subassembly** dialog box.
3. Click **Open**.
1. Position the mouse cursor as shown. Click and release once to create a single instance of the Crank at the location indicated.

2. Click the Finish button in the Inserting dialog box.

3. Click the Front View button from the Quick Access toolbar.
1. The view of the work area should be similar to the screen capture.
1. Position the mouse cursor as shown.
2. Click and hold the left and right mouse buttons while moving the mouse up (approximately as shown). Release the mouse buttons.
1. Confirm (or continue) the view-rotation such that the indicated faces are visible: the inner faces of both cylindrical holes, and the flat face of the Link (as shown).
1. Position the mouse cursor over the center of the **Link** part, and roll the mouse wheel forward to zoom in until the visible geometry roughly doubles in size.

2. The view of the work area should be similar to the screen capture.
1. From the **Constrain** group, select the **Quick** command.
1. Click on the indicated face
2. Click on the indicated face.
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.
1. At this point the Quick Constraint dialog box should still be open.
2. Click on the indicated face
3. Click on the indicated face.
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.
1. Click on the indicated face.
2. Click on the indicated face.
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.
1. Click on the indicated face.
2. Click on the indicated face.
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.
3. Close the Assembly Constraints dialog box.
4. Place your cursor over the top face of the Link part and use the scroll wheel to zoom out.
1. The view of the work area should be similar to the screen capture.
1. From the **Insert** group, select the **Insert Design** icon.
2. Select the **Base** in the **Insert Part/Subassembly** dialog box.
3. Click **Open**.
1. Position the mouse cursor (with the transparent preview of the base) over the Origin point. Proper positioning is confirmed by the color change of the Origin point. Click and release once to create a single instance of the part.

2. Click on the Finish button on the Inserting dialog box.
1. Right click on the Base item that is listed in the Design Explorer. (Not Shown)
2. Select the Anchor Part command. (Not Shown)
3. Confirm anchored part in the Design Explorer.
1. Select the **Quick Constraint** command.
1. Click on the indicated face
2. Click on the indicated face.
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.

Note: Since the Base is anchored it cannot move. In this case, the Crank will always move in relation to the Base; the order of selection is irrelevant.
Rotate and Zoom the workspace so you can easily see the following items.

1. The inside surface on the Base.
2. The cylindrical shaft on the Crank.
1. Click on the indicated face
2. Click on the indicated face.
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.
3. Close the Assembly Constraints dialog box.
1. Position the mouse cursor over the Link part, and roll the mouse wheel backward to zoom out slightly, until the view is similar to the screen capture.
1. Position the mouse cursor over the rounded end-face of the Crank part. Click and hold the mouse button; then move the mouse around to simulate turning the crank by hand. Notice that the mechanism responds as it would in the real world.

2. Save this Assembly to the same location as the part files with the name: Universal Joint. (not shown)

3. Close the Assembly workspace (not shown).

This concludes the Tutorial