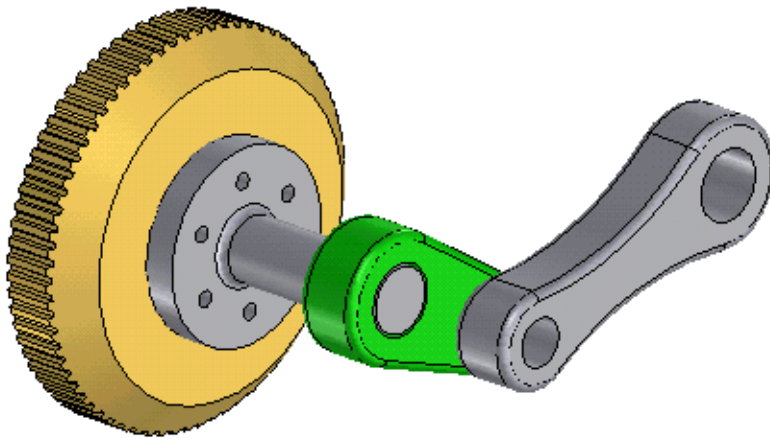


Creating and Editing Shapes Using 2D Cross-Sections

This tutorial focuses on IronCAD's advanced capabilities for creating and editing IntelliShapes using 2D geometry. It assumes you are already familiar with the basics of IronCAD, as introduced in the Getting Started section of the IronCAD documentation.

You will create the simple assembly of parts shown below. However, rather than creating the parts in the least number of steps possible, you will explore numerous alternatives for construction and modification.




Topics covered in this tutorial include :

- Setting Drawing Options
- Positioning Lines using Endpoint Dimensions, Curve Dimensions, and Curve Handles
- Dynamic Constraints Using the “Maintain End Conditions” Option
- Relocating the Axes of Endpoint Dimensions
- Drag-and-Drop IntelliShape modeling Vs. 2D Cross-section Modeling
- Creating a Hole Pattern
- Referencing other 3D Entities While in 2D Drawing Mode Using Project Edges
- Editing Curve Properties
- Creating and Maintaining Tangency Conditions
- Creating an Assembly
- Reusing 2D Cross-Sections via Cut and Paste and via Catalogs
- Rotating Lines and Curves
- Moving the Datum Lines
- Construction Lines
- Using the Right Mouse Button for More Power
- Linked Instances of a Cross-Section
- 2D Fillets Vs. 3D Blends

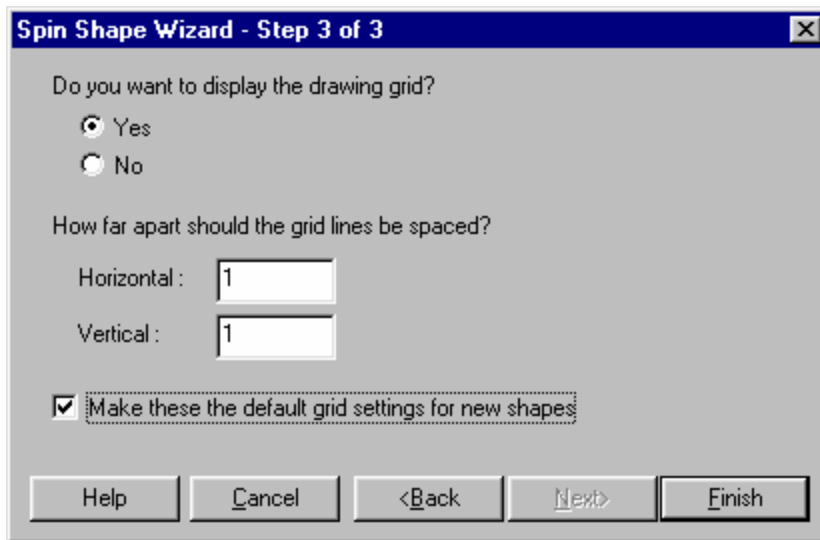
Creating the Flywheel with a Spin IntelliShape


A Spin IntelliShape is a 3D shape created by revolving a 2D profile (cross-section) about an axis. The profile must consist of one or more closed loops of connected 2D lines with no overlaps and no lines that cross over the spin axis (the Width axis).

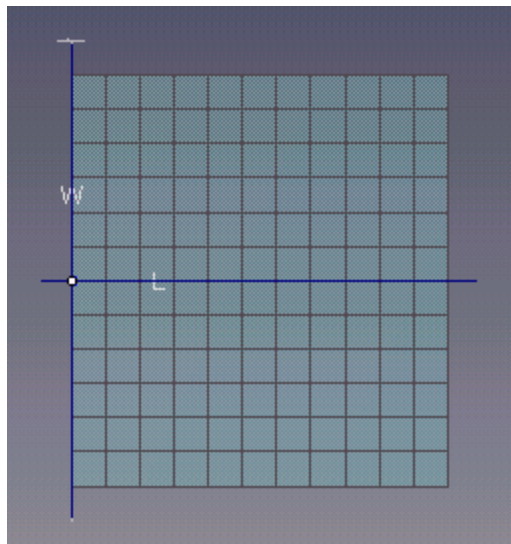
1. From the File menu, select New, Scene and click OK. From the Workspace (English) tab, select the template Gray.ics and click OK.
2. To create a spin IntelliShape, click the Spin tool  from the IntelliShape Creation toolbar, or click Create, IntelliShape, Spin from the menu bar, then click anywhere near the center of the screen to specify the location of the shape.

The Spin Shape Wizard appears.

3. Click Next then Next again to accept default values for the first two steps, then specify the settings for the third step as shown and click Finish.



4. To position your view normal to the drawing surface, click the LOOK At tool  and then click on the drawing surface. Use the other camera positioning tools as needed to orient your view so it looks approximately like this:

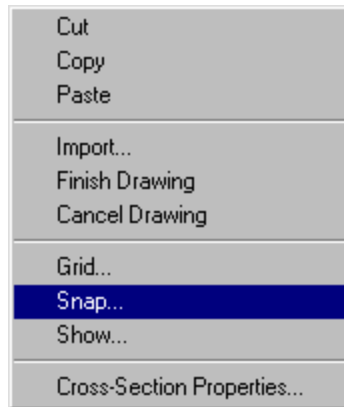


Tip

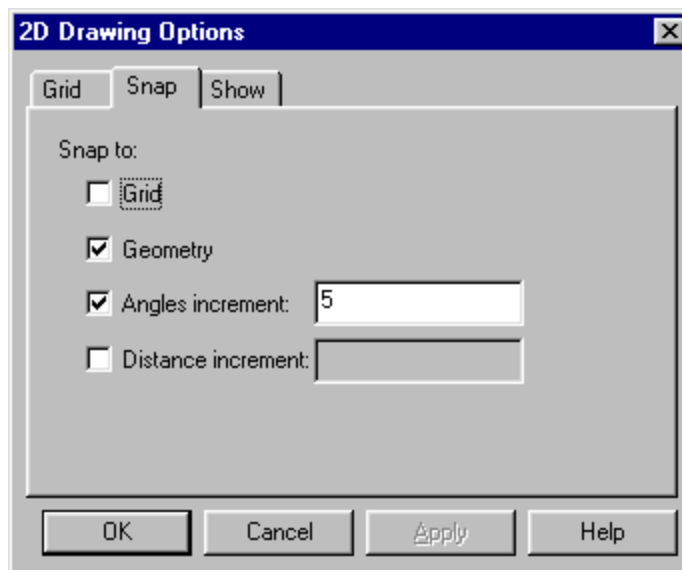
To control which toolbars are displayed, click **View, Toolbars** from the main menu bar. Alternatively, right-click on any blank space next to a toolbar and select from the list displayed.

Setting the Drawing Options

1. Right-click (click the right mouse button) anywhere in the view and select Snap... from the menu as shown. Alternatively, click Format, Grid... from the menu bar and select the Snap tap.




2. Set the Snap options exactly as shown and click OK.



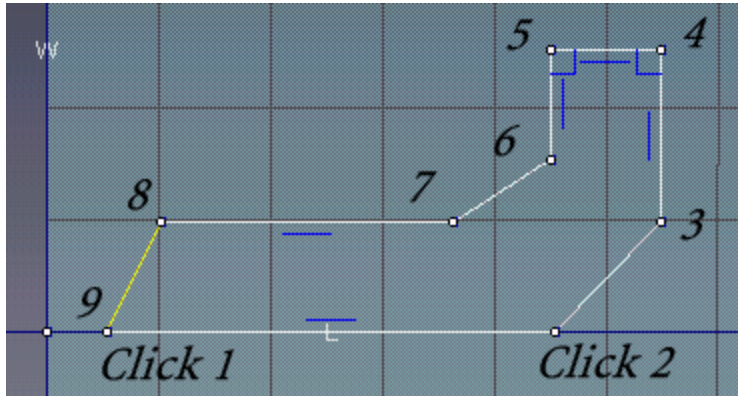
3. Click the Show Endpoint Positions tool from the 2D Editing toolbar so that it's depressed "in" as shown. Notice that this tool does not invoke a command. It just toggles on (in) and off (out) each time you click it.



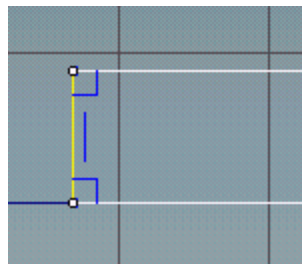
Drawing the Spin profile

1. Click the Polyline tool  from the 2D Drawing toolbar and draw the profile as shown. As you move the cursor between clicks, notice the snapping behavior and blue feedback lines that indicate geometric relationships such as horizontal, vertical, perpendicular, or tangent line conditions. Take advantage of this behavior to create the

horizontal and vertical lines as shown. Using the grid lines as a reference, draw the lines with approximately the dimensions shown below.



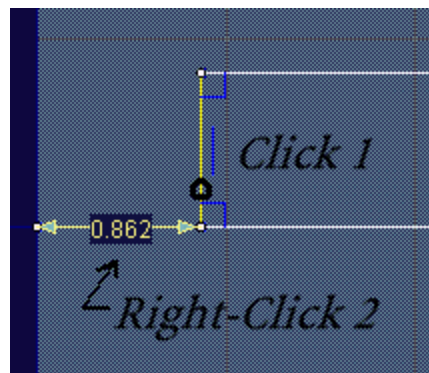
2. To end the Polyline command, press the Esc key or click the Polyline tool again.
3. To make the line segment between points 8 and 9 snap to vertical, click on point 8 and drag it to the left until it snaps into position. The blue feedback lines should indicate that the line is horizontal and perpendicular to its adjacent lines as shown.



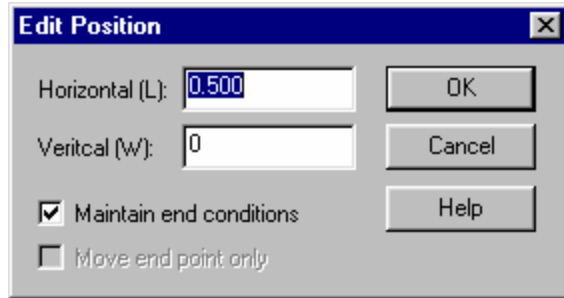
4. If you made a mistake and any line is not aligned as shown, just drag its endpoint until it snaps into the desired alignment.

Accurately Positioning Lines Using Endpoint Position Dimensions

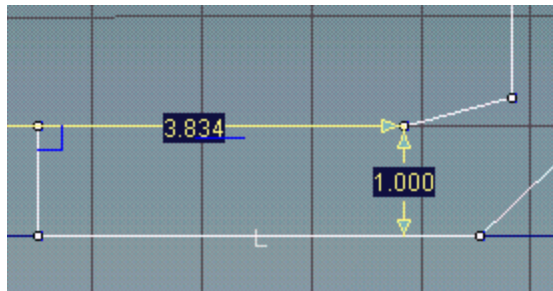
1. Select the leftmost vertical line below its midpoint as shown. Now move your mouse cursor over the horizontal endpoint dimension text until it changes to a "hand" icon, then right-click and select Edit Value....



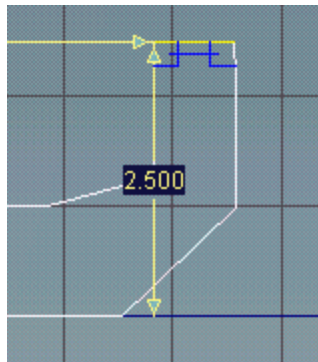
2. Change the Horizontal distance to 0.5 as shown then click OK.



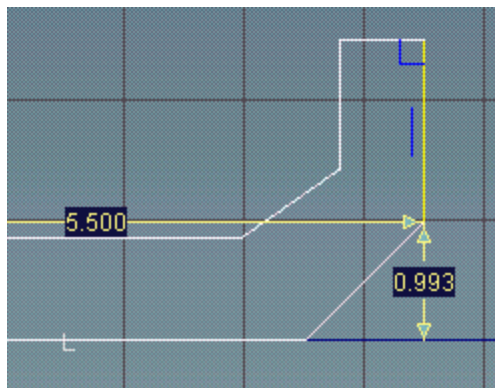
3. Repeat this same technique to position the adjacent horizontal line to 1.0 inches above the horizontal axis: select the line, right-click on its vertical endpoint dimension, select Edit Value ..., change the Vertical distance to 1.0 inches and click OK.



4. Select the next horizontal line as shown and change its vertical position to 2.5 inches.



5. Select the vertical line shown and change its horizontal distance to 5.5.

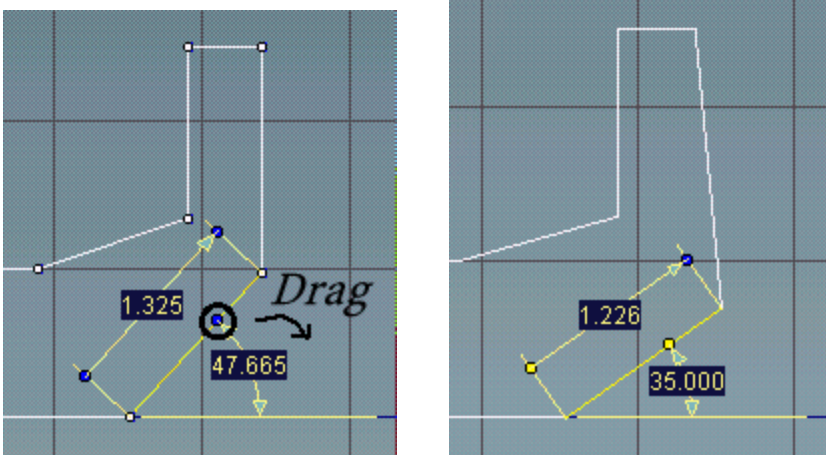



Accurately Positioning Lines Using Curve Dimensions

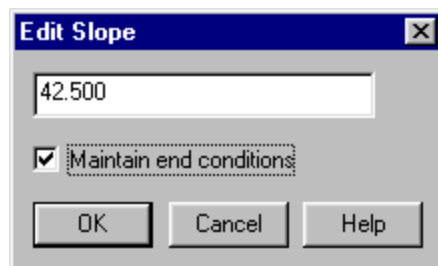
1. Click the Show Endpoint Positions tool to turn it OFF, and click the Show Curve Dimensions tool to turn it ON.



2. Select the angled line shown at a point above its midpoint (notice the difference if you select it below its midpoint). Place your cursor over the blue dot connected to the angular dimension and notice how your cursor changes to a hand. These blue dots are called **curve handles**. Drag the angular curve handle clockwise by a few degrees and notice what happens. Not only does the angled line move, but its adjacent vertical line also moves such that it is no longer vertical.



3. Click Undo  to return the line to its previous position.
4. To maintain the vertical orientation of the adjacent line, drag the angular curve handle again, but this time **hold down the Shift key as you drag the handle**. Snap the angle to exactly **30-degrees** (the angle should snap to 5-degree increments if you adjusted the Snap settings as described earlier). Notice that this time the adjacent line maintains its angle instead of moving its endpoint. This behavior is referred to as **maintain end conditions**.
5. Now suppose you've changed your mind and you want a different angle. While the angular line is still selected, right-click the angular dimension text and select **Edit Value...** Change the value to 42.5-degrees and select Maintain end conditions as shown, then click OK. Notice that checking Maintain end conditions has the same affect as holding down the shift key while dragging the curve handle.

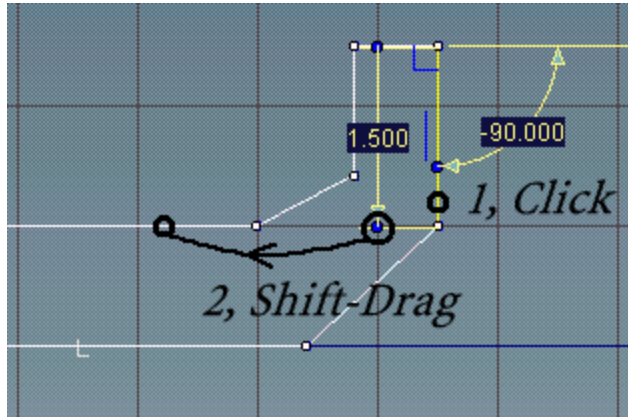


6. Now select the adjacent vertical line as shown. While holding the Shift key, grab the linear curve handle (not the angular curve handle) and drag it over the horizontal line as shown. Notice that because you are holding the shift key the lines are constrained to

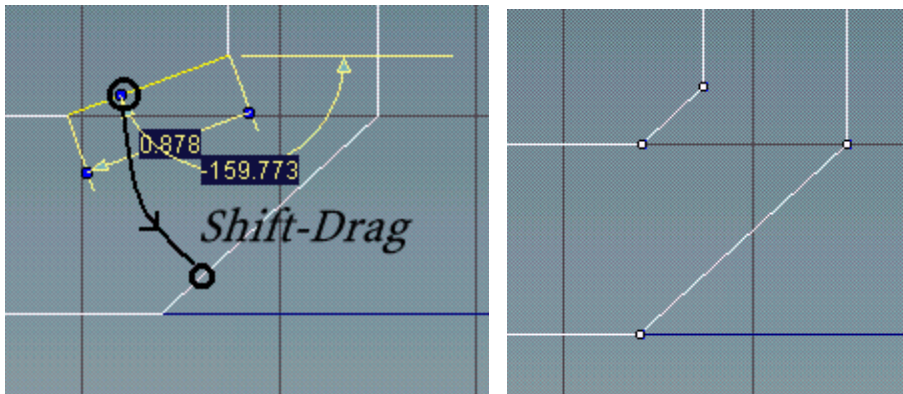
Note

The "Maintain End Conditions" behavior is sometimes referred to as "Dynamic Constraints". It allows you to apply temporary constraints when and where you need them. This is in sharp contrast to traditional parametric techniques, which are rigid and can become overly complex, causing your model to be difficult to understand and difficult to modify when unanticipated changes are required.

maintain their angles, yet you can snap to other geometry, including curve midpoints, circle centerlines, and more.



- To adjust the other angular line to also lie at 42.5 degrees, select it below its midpoint, press and hold the shift key, and drag the angular curve handle clockwise until your cursor is over the other angular line (the one that has already been set to 42.5 degrees). Be careful not to snap to the center point or an endpoint of this line. Notice that as you wave the cursor over the other line, the angle of the line being modified snaps to the same angle as the line the cursor is over!



Tip

If an item you need to select is obstructed by an Endpoint Dimension or Curve Dimension, just zoom in a little bit. The geometry of course will appear larger as you zoom in, but items such as curve handles and dimension text will remain the same size.

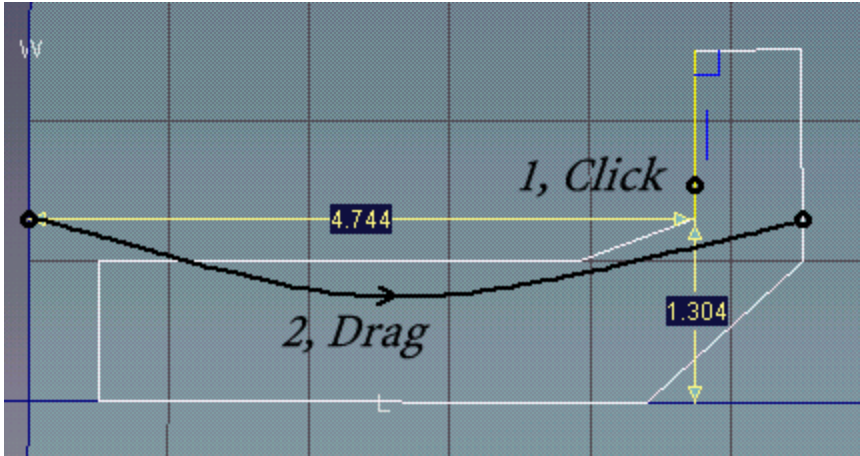
Relocating the Coordinate System Axes of Endpoint Dimensions

- Click the Show Curve Dimensions tool to turn it OFF, and click Show Endpoint Positions to turn it ON.



- Select the vertical line as shown below, then place your cursor over the left arrowhead of the horizontal dimension so that it changes to a "hand" icon. Click the arrowhead and drag it over to the vertical line to the right of the line selected until it snaps onto that line, then release the mouse button.

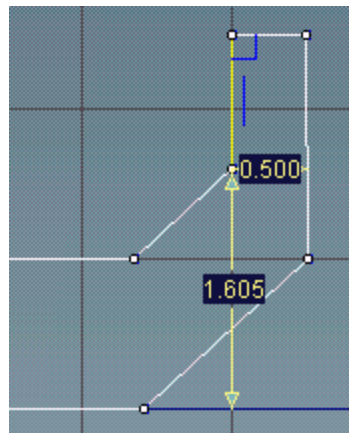
The axis for horizontal endpoint dimensions moves from the vertical axis over to the vertical line selected.



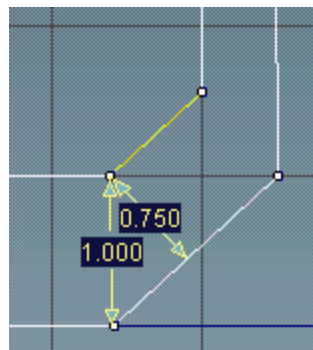
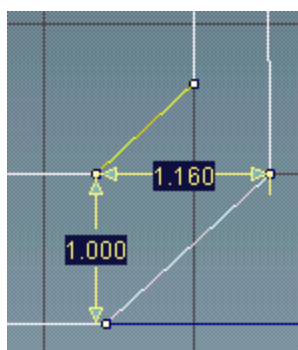
Tip

In addition to moving the attachment point of a dimension, you can also select and move the datum line (the vertical axis) that the dimension is attached to.

3. Right-click on the newly located vertical endpoint dimension, select Edit Value..., change the Horizontal value to 0.5 as shown, select Maintain end conditions and click OK.




4. Now select the uppermost angular line as shown. Notice that the horizontal endpoint dimension is still attached to the vertical line you moved it to in the previous step. To reposition it again, drag its rightmost arrowhead over to the other (parallel) angular line. Change its Horizontal value to 0.75 using the Maintain end conditions option.

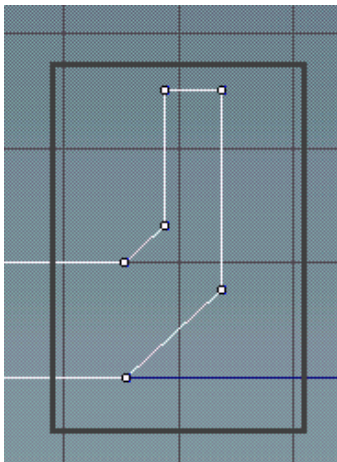


5. Drag the horizontal endpoint dimension back to the vertical axis.

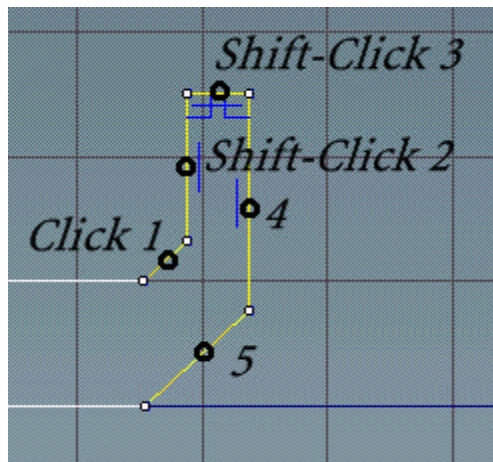
Positioning Multiple Lines at Once

Endpoint dimensions and curve handles may also be used to position multiple lines at once.

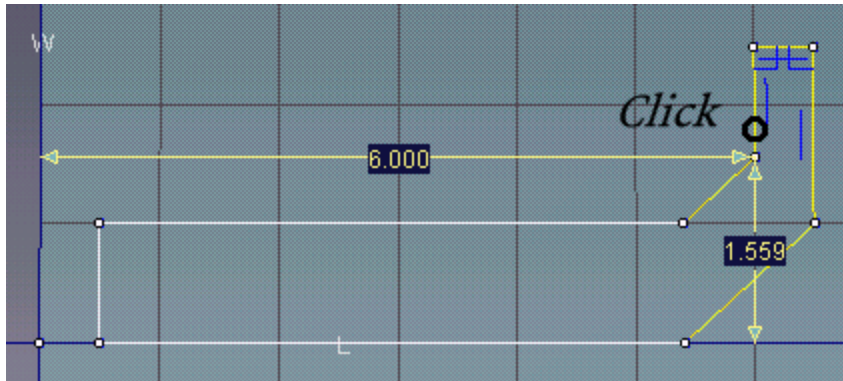
- Click the Box Select tool  on the Selection toolbar and draw a box around the lines as shown. Alternatively, you can select the lines by clicking one of them then Shift-clicking the rest (hold the Shift key while clicking).



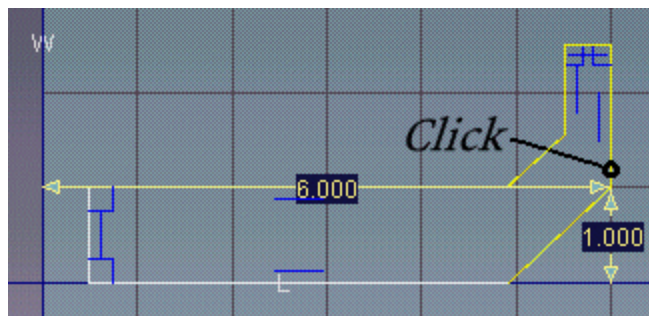
OR



- While all 5 lines are still selected, click the line shown and change its Horizontal distance to 6.0. Note that using the Maintain end conditions option is entirely optional in this case since the direction of movement is the same as the direction of the adjacent lines.



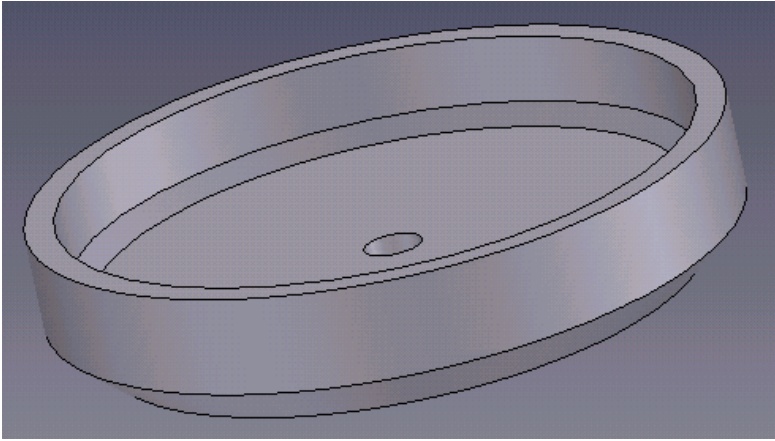
- Now suppose you've changed your mind and want to position this group of lines relative to the "outer" line instead of the "inner" line. To do this, first make sure all 5 lines are still selected (or box select them again if they are not), then click on the outer line as shown and change its Horizontal distance to 6.0.



- To spin the cross-section into a solid 3D shape, click Finish Shape from the Edit Cross-section menu. Alternatively, right-click anywhere on the drawing surface and

select Finish Drawing.

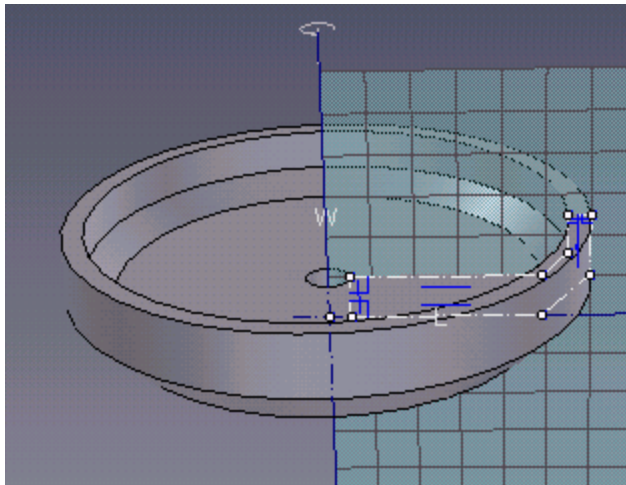
The cross-section is revolved (spun) 360-degrees about its vertical axis and a new 3D part is created.



Editing the Cross-section of an Existing IntelliShape

1. Select the newly created part at the IntelliShape level (Click on it until it highlights in yellow with red handles).
2. Right-click on the IntelliShape and select Edit Cross-Section.

The 2D drawing surface appears again and you are now back in 2D editing mode.




Enhancing the Visibility of 2D Lines Inside the Solid

Notice that the 2D lines are displayed in the context of the 3D part and may sometimes be difficult to view clearly, depending on your various rendering and color settings. To improve the visibility of the 2D lines, experiment with using one or more of the following techniques:

- Try using a different color for 3D part edges than for 2D line edges. For example, try black for part edges and white for 2D lines. To change these colors, select **TOOLS**, **Options...**, **Color** and edit the entries named **New 2D lines and curves** and **Part Edges**.
- Turn part edges off. From the main menu bar, select **Format**, **Rendering...** and deselect the **Draw part edges** option.

Tip

Whenever you see a red dot in your cross-section, it indicates a vertex that only belongs to a single line, as opposed to connecting two lines. You must eliminate all red dots before you can finish a shape. IronCAD can only create a shape if the cross section contains closed "loops" of lines without any overlaps or discontinuities. If you reposition a disconnected endpoint onto another endpoint and the red dot still does not go away, right-click on the dot and select **Connect**. If that still doesn't work, then the endpoints are slightly off, even though they may appear to be aligned.

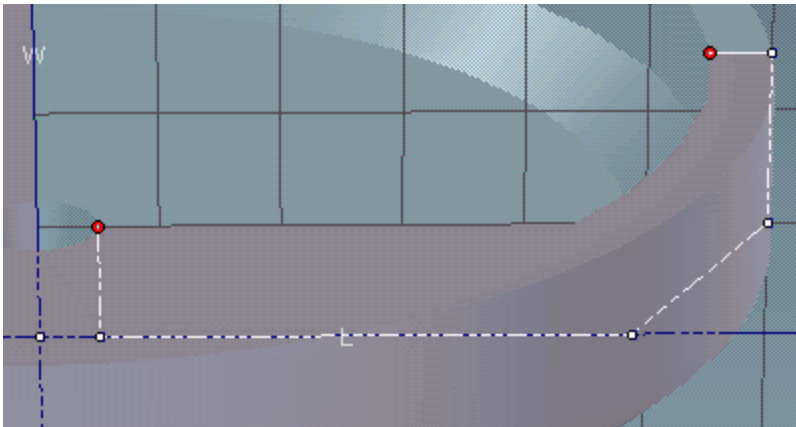
- Turn perspective off using the Perspective Camera tool  on the Camera toolbar. This is useful if your view is exactly normal to the drawing surface and you wish to see the exact relationship between 3D edges and 2D lines.

Using the Offset Command

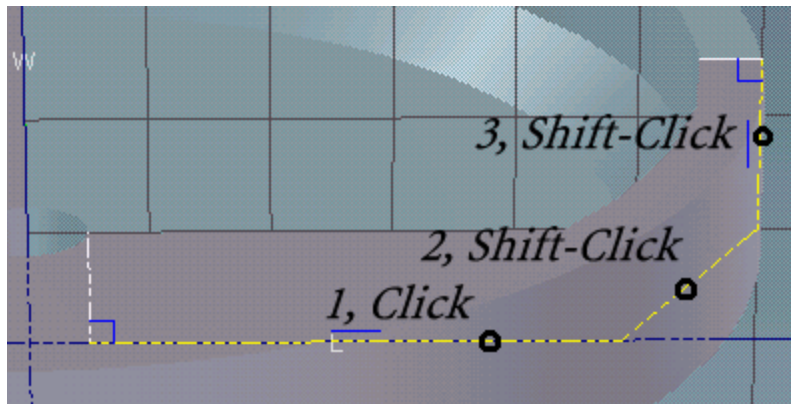
Now suppose your design requirements have changed unexpectedly and you want to use a constant wall thickness for the flywheel.


1. Delete the 3 inner lines as shown by right-clicking each line and choosing **Delete**, or by selecting each line (or all the lines at once using Shift-click) and pressing the **Delete** key.

The red dots indicate endpoints that are not connected to another line.



2. Select the 3 lines shown (click the first line, then Shift-click the remaining two lines).



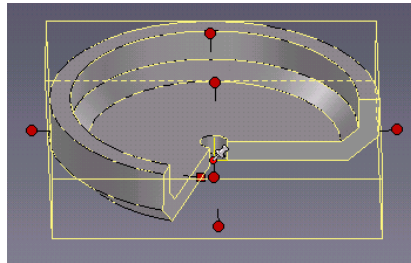
3. Click the Offset 2D Curves tool  from the 2D Editing toolbar. The Offset dialog appears.
4. Set the Distance to 0.75, leave the Number of copies at 1, and click Apply. If the lines are offset to the wrong side (below instead of above), click Flip Direction. Click OK.
5. To reconnect the lines, drag the two red dots to their closest respective endpoints.
6. Click Finish Shape.

Drag-and-Drop IntelliShape Modeling vs. 2D Cross-section

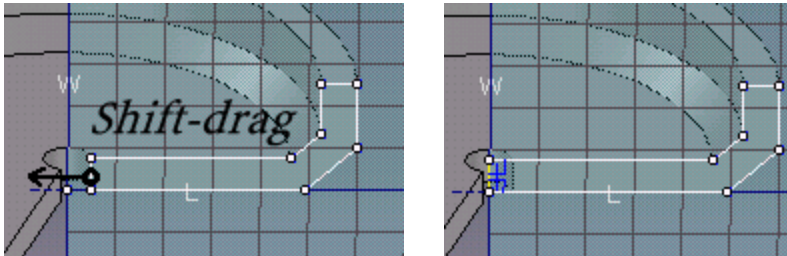
Modeling

In this section, you will first modify the 2D cross-section to effectively delete the hole in the center of the flywheel. Then, you'll recreate the hole as a separate IntelliShape feature using drag-and-drop from the standard shapes catalog.

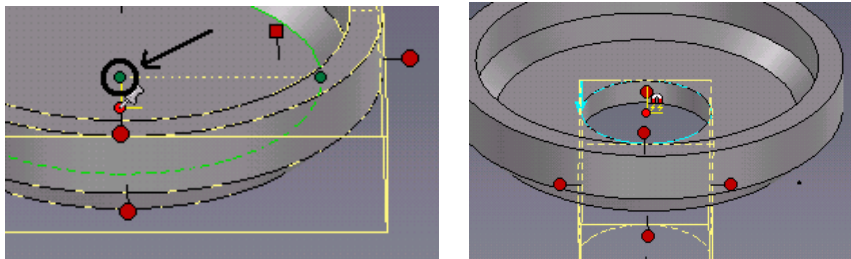
1. Select the flywheel shape at the IntelliShape level.
2. Grab the Spin Angle handle as shown and drag it approximately 100-degrees. Alternatively, you can right-click on the handle, select *Edit Value...*, type an exact angle, and click OK.



3. Right-click on the IntelliShape and select *Edit Cross-Section*.
4. To effectively delete the hole in the center of the flywheel, select the innermost vertical line as shown and shift-drag it until it snaps to the vertical axis. It's not absolutely required that you hold the Shift key while dragging, but if you don't you'll have to be careful to use the snapping behavior to maintain the perpendicularity with the two adjacent lines.



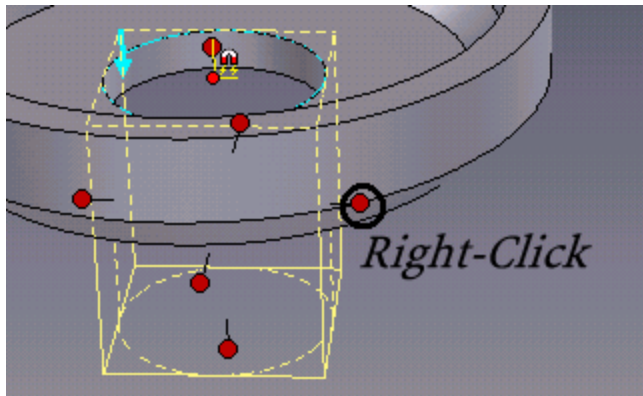
5. Click *Finish Shape* from the *Edit Cross-section* dialog.
6. Grab the Spin Angle handle again and drag it back to where it was (360-degrees). Alternatively, right-click on the handle and type 360 in the dialog.
7. Open the Shapes catalog on the right side of your screen and drag the H Cylinder shape to the center of the Flywheel as shown. As you're dragging the H Shape, notice that when your cursor touches the top surface the center point is highlighted with a dark green dot, which turns to bright green when your cursor touches it. This behavior is called **SmartSnap**.



8. To adjust the diameter of the hole to 2 inches, right-click the handle shown, select *Edit Sizebox...*, change the Length or Width value to 2.0 and click OK. Note that for this particular IntelliShape, the Length and Width values are constrained to be equal.


Tip

Tip



Setting Part and IntelliShape Names and Color

As your design starts to get more complex, you may want to start naming individual parts and their various IntelliShapes. It's also useful to add some color.

1. Open the Scene Browser using the Scene Browser tool  on the Standard menu.
2. Click the "+" symbol next to the flywheel part to display its component IntelliShapes.



3. To change the name of the part, click the part name once, wait a moment, then click it again and the name will become editable. Change the name to **Flywheel** and press Enter.

Alternatively, you can right click on the part name, select Part Properties..., select the General tab and change the User name.

4. Repeat this same procedure to change the name of Shape 1 to **Spin Profile**.



5. To change the color of the flywheel part, open the Surfaces catalog on the right side of the screen and drag a new SmartPaint item anywhere onto the part. For example, try using the Gold surface. Alternatively, you can select the part, then right-click on it and select Smart Paint... and edit the SmartPaint Properties.

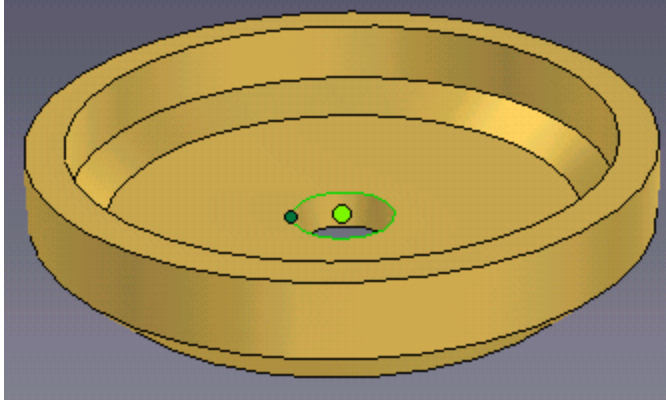
Creating a Bolt Hole Pattern

1. Open the Advshapes catalog, drag out the H Bolt Circle shape, and SmartSnap it to the center of the Flywheel as shown. If you have difficulty snapping to the center, make sure your view is oriented exactly as shown. This is important because SmartSnap only works when your cursor is over a face -- it cannot snap to "air". In other words, the flywheel must be oriented such that your cursor can touch both the center point and

Tip

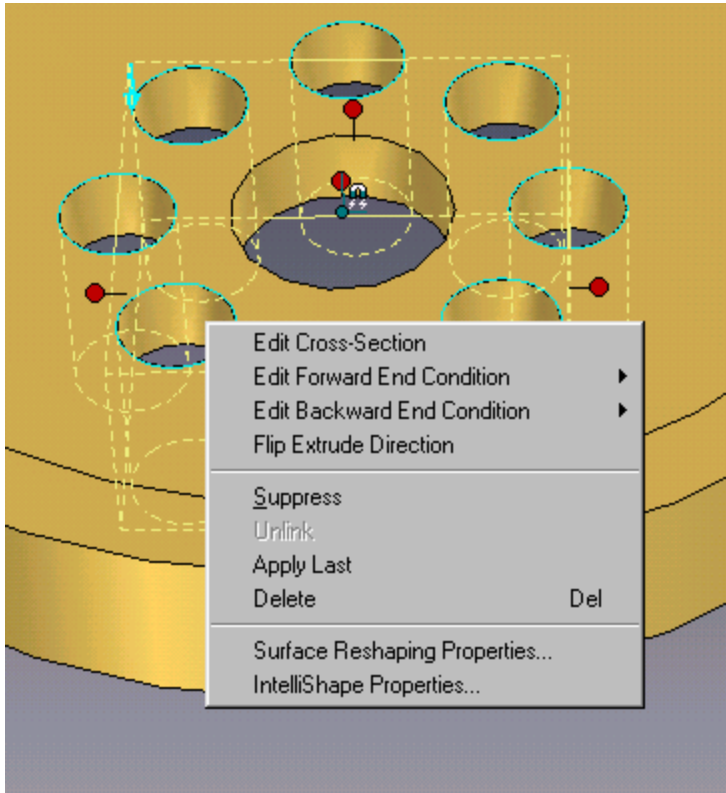
*If you don't see a tab for the Surfaces catalog, select the **Catalogs, Open...** menu and search for the catalog under the IronCAD installation directory (for example, C:\Program Files\Vds\IronCAD\Catalogs).*

the cylinder face at the same time.



SmartSnap only works when your cursor is over a face – it cannot snap to “air”. For example, to snap to the center of a cylinder edge, you must orient the camera such that your cursor can touch both the center point and the cylinder face at the same time

2. While the H Bolt Circle IntelliShape is selected, right-click on one of its Width or Length handles, select Edit Sizebox... and change the value to 4.0. This gives the bolt pattern a radius of 2.0 inches.
3. Now zoom in on one of the bolt holes so that you can easily select it, then right-click on the face of the hole as shown and select IntelliShape Properties...



4. Select the Variables tab and change the Number of Circles value to 6 and the Radius of Circles value to $0.25 * 2.54$ (the 2.54 multiplier is necessary because the variable units are centimeters). Click OK. A message will appear asking you if you want to replace the formula for the Radius of Circles parameter. Click Yes.

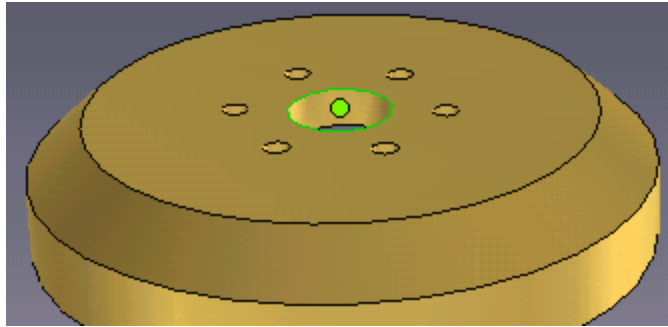
Any time you need to type a numeric value in IronCAD, you can type a fixed number or you can type a mathematical expression and let IronCAD calculate the value for you.

Creating the Crank Shaft


In this section, you will create a crankshaft part that mates with the flywheel.

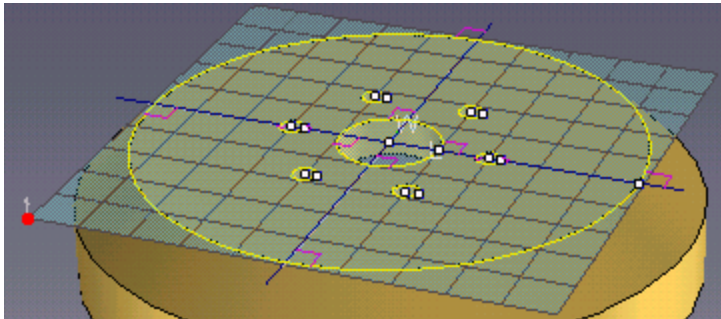
1. Position your view of the flywheel as shown, click the Extrude Shape tool from the IntelliShapes toolbar, then SmartSnap the origin of the extrude shape to the center of the flywheel as shown.

The Extrude Shape Wizard appears.




2. Select Stand alone so this shape will create a new part.
3. Click Next 3 times to accept the remaining default values, then click Finish.
A 2D drawing surface appears.

4. Click the Project Edges to Drawing Grid tool  from the 2D Editing toolbar, then click the top face of the Flywheel as shown.

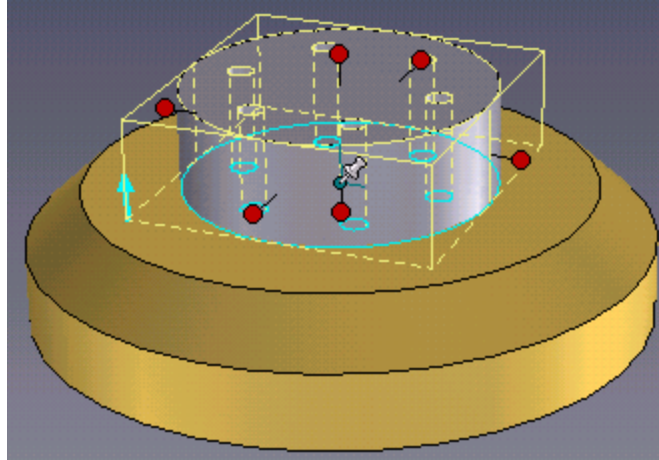


5. Press the Escape Key or click the Project Edges to Drawing Grid tool again to turn it off.
6. Click any blank spot on the drawing surface to deselect the lines.
7. Right-click on the outer (largest) circle, select Curve Properties... and change the Major Radius to 3.0.

Alternatively, you can turn on Show Curve Dimensions , select the circle, right-click on the radius dimension text, select Edit Value..., change the value to 3.0 and click OK.

8. Delete the innermost circle.
9. Click Finish Shape.

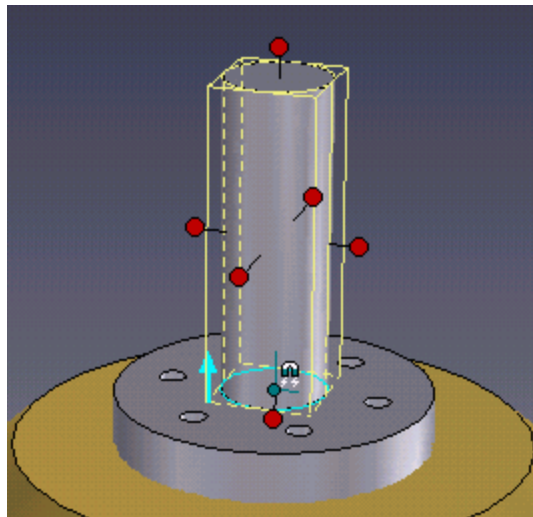
The 2D profile is extruded and a new part is created.



10. To adjust the height of the new shape, right-click the top (Height) IntelliShape handle, select Edit Sizebox..., change the Height value to 1.0 and click OK. Note that this value also could have been specified at the time of initial creation in the Extrude Shape Wizard dialog.
11. Pull on either the length or width handle and notice that the sizing behavior is not exactly what you might expect! In general, you should be careful about using the sizebox handles of custom IntelliShapes unless you fully understand the sizing behaviors. You can fully control these sizing behaviors, but that topic will not be covered in this section.
12. Click Undo to restore the shape to its previous size.

Adding More Shapes to the Crank Shaft

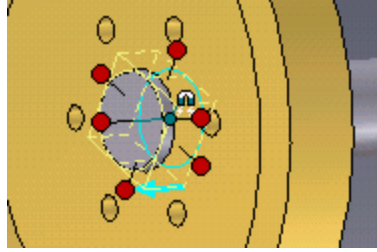
1. Drag a Cylinder IntelliShape from the Shapes catalog and snap it to the center of the crankshaft as shown. Adjust the Length and Width to 2.0 and the Height to 6.0.





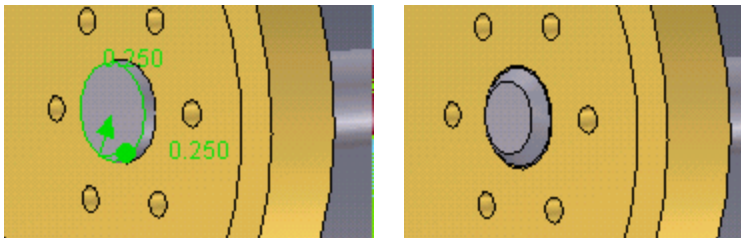
2. Now drag and drop a cylinder onto the other side of the crankshaft as shown. Adjust the Length and Width to 1.9 and the Height to 1.0.


Note

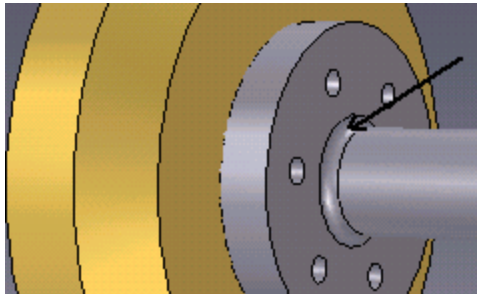
Notice that unlike many CAD systems, IronCAD does not require you to create each part in a separate environment then switch to an “assembly” mode to put them together. It’s very simple to create the parts in the context of their mating parts, enabling what VDS refers to as “fit by design”. Even so, the individual parts or subassemblies of an assembly may optionally be split out into separate linked files to enable different team members to work on different components of an overall assembly of parts.



3. To bevel the front edge of the crankshaft, click the Chamfer Edges tool  from the Face/Edge Edit toolbar, then click the front/top edge. Type in a distance of 0.25 for both sides of the chamfer, then click the Apply and Exit Command button .

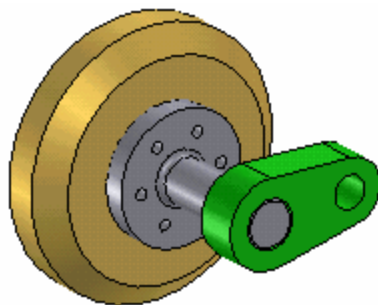


4. Repeat this same process using the Blend Edges tool  to blend the crankshaft edge to a radius of 0.25 as shown. Blends are also sometimes referred to as “rounds” or “fillets”.

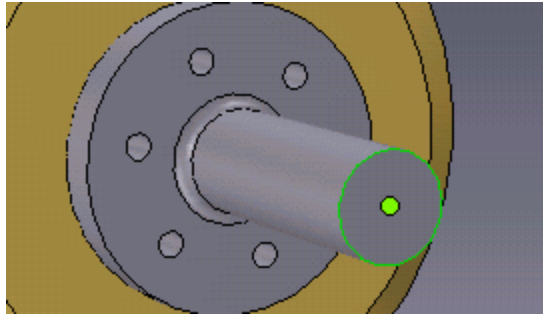



Creating the Crank Arm

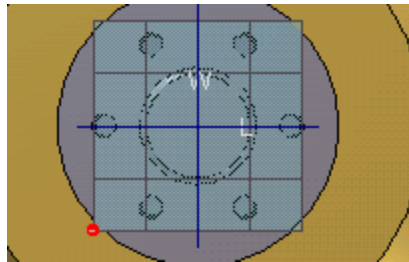
In this section, you will begin creating a crank arm that mounts on the end of the crankshaft as shown.



1. Click Extrude Shape  on the IntelliShapes toolbar, then SmartSnap the origin of the extrude shape to the center of the crankshaft end as shown.




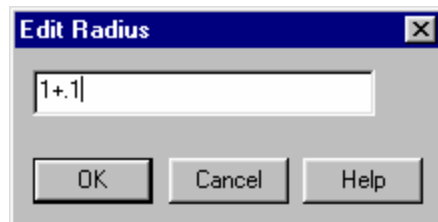
2. Click Stand alone in the Extrude Shape Wizard, then click Next and Next again.
3. Set the Distance to 2.0 then click Finish.
4. Use the Look At tool  and other camera tools as needed to position your view of the drawing grid approximately as shown.




5. Turn Show Curve Dimensions ON and Show Endpoint Positions OFF.

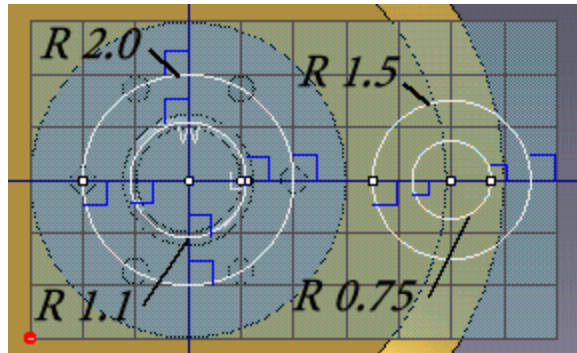


6. Use the Project Edges to Drawing Grid tool  to project the circular edge from the end face of the crankshaft, then turn the tool off by pressing the **Esc** key or clicking the tool button again.
7. Right-click on the circle's radius dimension and select Edit Value..., then add 0.1 to the value as shown and click OK.



8. Now use the Circle: Center Radius tool  to create 3 more circles as shown, for a total of two pairs of concentric circles. The two circles on the left should be centered at the drawing grid origin, and the two circles on the right should share a center point anywhere on the horizontal datum line. After creating the circles, turn off the Circle:

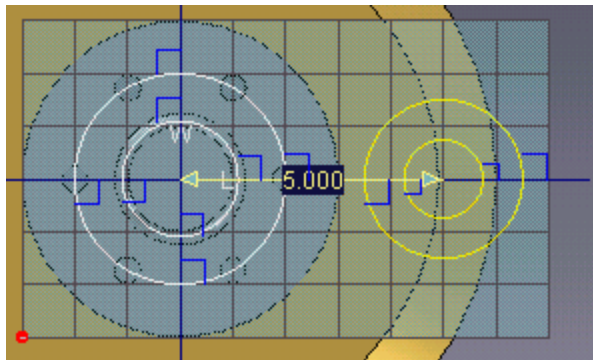
Center Radius tool and change the radii to 2.0, 0.75, and 1.5 as shown.



9. Turn Show Curve Dimensions OFF and Show Endpoint Positions ON.




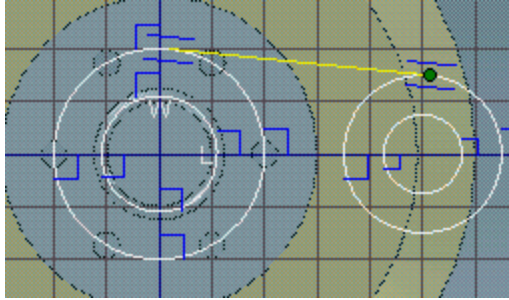
- 10.
11. Now select both of the rightmost circles at the same time (click one circle then shift-click the second, or use Box Select). Right-click the horizontal endpoint dimension, select Edit Value..., change the value to 5.0, and click OK.



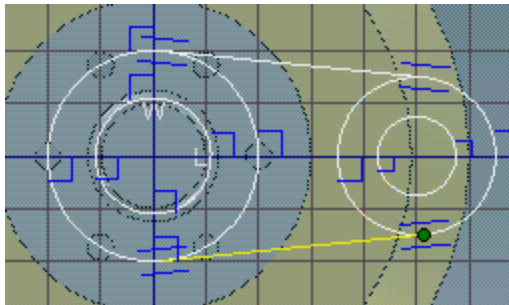
Creating Tangent Lines


Now you will connect the two outer circles with tangent lines and then trim away portions of the circles to form a single closed profile with two holes.

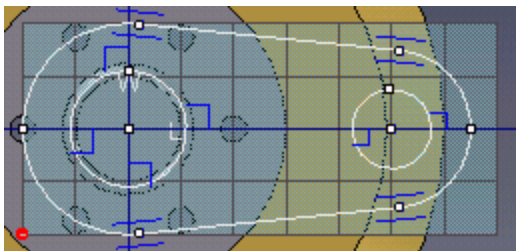
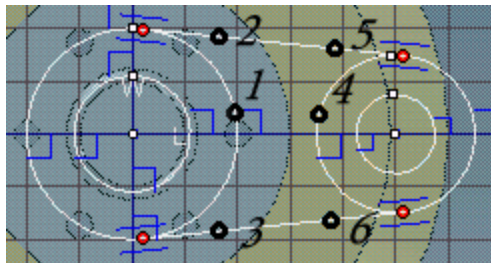
1. Click the Tangent Line tool  and select the left outer circle anywhere along the top half of the circle. Now wave your cursor around and notice that the line stays tangent to the circle no matter where you move your cursor. Notice also the prompt in the bottom left corner of the IronCAD window. The system is prompting you to either pick another point, or to right-click to specify a distance and angle.
2. Move your cursor along the top edge of the right outer circle until the line snaps to the tangent point of the circle as shown, then click that point.



3. While the Tangent Line tool is still active, repeat to create a tangent line along the bottom sides of the outer circles as shown, then press Esc to end the command.



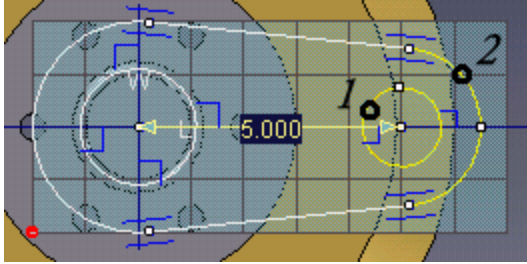
4. Click Trim Curve Between Curves  and trim the interior portion of the two outer circles by clicking the lines in the order show. For guidance, read the command prompt in the lower left corner of the screen. Turn the tool off when you're finished.



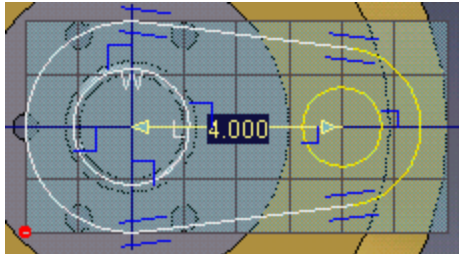
Maintaining Tangency Conditions

Now suppose you've changed your mind and you'd like the hole centers to be spaced at 4 inches instead of 5.

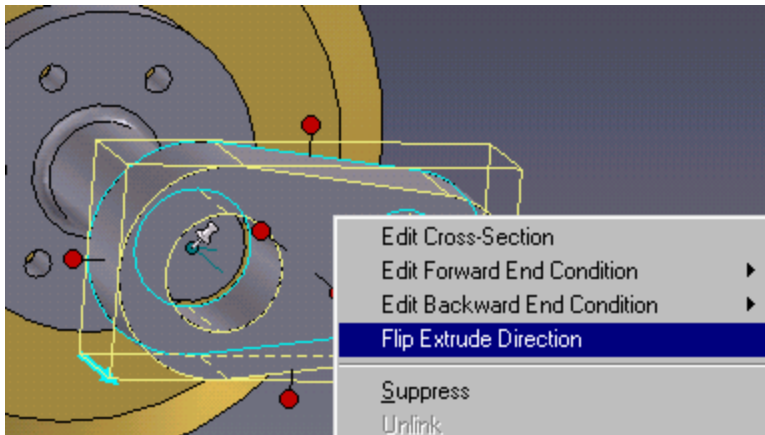
1. Select the two curves shown in the order shown by clicking the first curve then shift-clicking the second. It's **very important** that you select the curves in this order, as will be explained later.



2. Right-click the horizontal endpoint dimension and change its value to 4.0. Make sure the Maintain End Conditions option is selected and click OK. Notice that the lines remain tangent. If you had selected the two curves in the opposite order, the Maintain end Conditions options would not have been presented (since the inner circle is not connected to any other lines) and therefore the tangencies would have been lost!



3. Click Finish Shape to extrude the crank arm.
4. Right-click on the shape and select Flip Extrude Direction as shown.

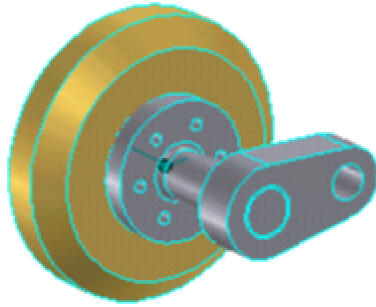



Creating an Assembly

In this section you'll organize the parts you've designed by combining them into a new assembly and adjusting their names and colors.

1. Make sure nothing is currently selected by clicking on the scene background or by choosing Deselect All from the Edit menu.
2. Select all three parts simultaneously by clicking one part then shift-clicking the other two. All three parts should be highlighted with cyan colored edges as shown.

Alternatively, you can select the parts from the Scene Browser .



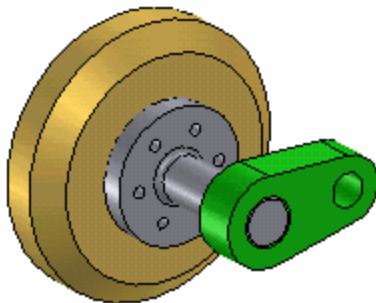
3. Select Create, Assembly from the main menu bar.
Open the Scene Browser  to view the newly created assembly.





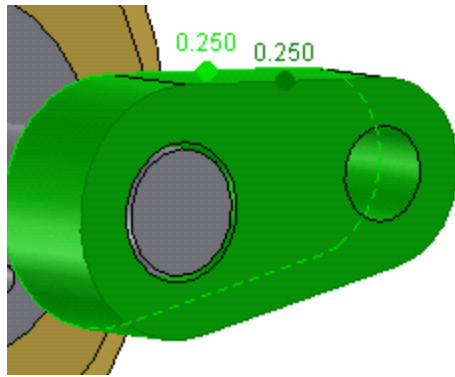
4. Rename the assembly and its component parts as shown.



5. Open the Surfaces catalog and drag the Shiny Green color onto the crank arm. If you have the assembly selected, or nothing selected, you will be prompted to choose whether to apply the new SmartPaint object to the entire assembly or only to the part. Choose Replace SmartPaint for this part only. If the part is selected when you drop the SmartPaint onto it, you will not be prompted and only the selected part will be affected.



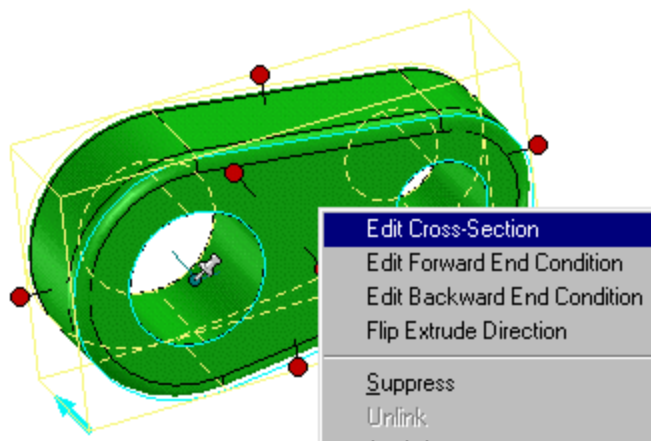
6. To blend the outer edges, click Blend Edges , set the radius to 0.25, select Extend to smoothly connected edges , then pick the two outer edges as shown and apply the command.




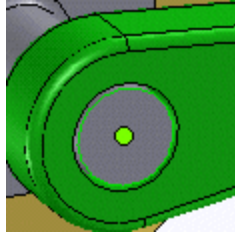
Reusing 2D Cross-Sections via Cut and Paste

Sometimes you will find it useful to copy some or all of the geometry from one 2D cross-section to another. One way to accomplish this is to simply cut and paste the geometry to and from the Windows® clipboard.

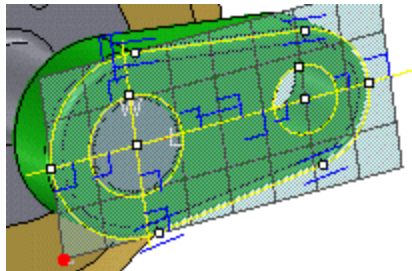
1. Select the Crank Arm at the IntelliShape level, then right click on the extruded shape and select Edit Cross-Section.



2. From the Edit menu, choose Select All Curves to select all curves on the 2D drawing grid *except* construction lines.
3. From the Edit menu, choose Copy. The selected curves are copied into the Windows® clipboard.
4. Click the Cancel button on the Edit Cross-section dialog (you don't need to edit the shape, you just need to copy its 2D geometry).
5. Click Extrude Shape  and SmartSnap the origin of the extrude shape to the center of the Crank Shaft as shown. When the Extrude Shape Wizard appears, select Stand alone then click Finish.




6. Select Edit, Paste and the previously copied 2D cross-section should now appear on the drawing grid!



7. Click Cancel to cancel the extrude command so you can try a different technique in the next section.


Saving 2D Cross-Section Geometry into a Catalog

IronCAD's catalog system provides yet another way to copy 2D geometry from one cross-section to another. Once you've saved a cross-section as a catalog item, you can use it at any time by simply dragging and dropping it onto a 2D drawing grid!

1. To create a new catalog, select Catalogs, New from the main menu bar. A new catalog tab appears at the bottom right corner of your screen with a default name such as "Catalog1".
2. Open the newly created catalog, then right-click anywhere in the catalog and select Paste. The 2D geometry copied to the Windows® clipboard in the previous section is now pasted into the new catalog.
3. To change the default name of "UnNamed" to "CrankArm", click twice slowly on the name to make it editable, then type a new name of "CrankArm" and press **Enter**. If you accidentally double-click the name (instead of pausing between the first and second click), this will actually open the catalog item in a new window. If this happens, just close  the new window and try again.

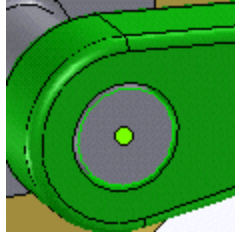


CrankArm

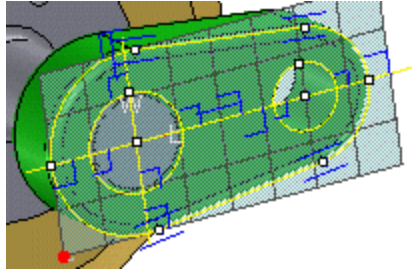
4. If you wish to make this catalog permanent and to give it a more meaningful name, select Catalogs, Save from the main menu bar and save the new catalog to a file.
5. Click Extrude Shape  and SmartSnap the origin of the extrude shape to the center of the Crank Shaft as shown. When the Extrude Shape Wizard appears, select Stand alone then click Finish.

Note



The 2D geometry should still be in the clipboard from the previous section, unless you have put something else there in the mean time. If it is no longer there, you'll need to repeat the steps from the previous section to copy it back onto the clipboard.

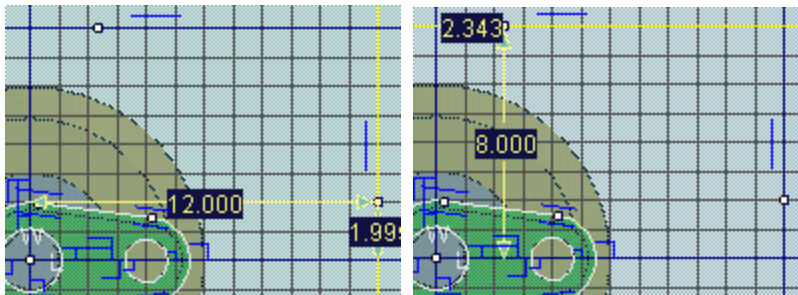



- Open the newly created catalog, then drag and drop the “CrankArm” item to anywhere on the drawing grid. The 2D cross-section appears on the drawing grid. Notice that the geometry always appears in its original location relative to the origin regardless of where you drop it on the grid.

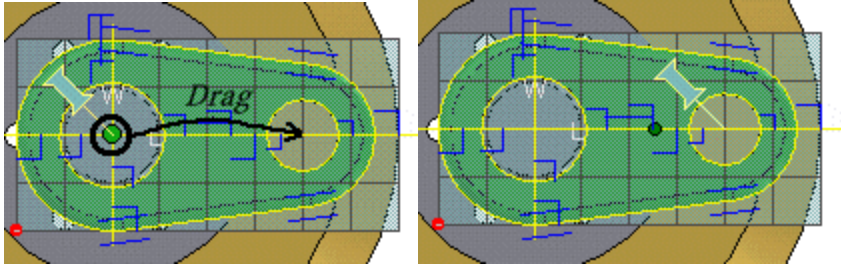


Rotating Lines and Curves

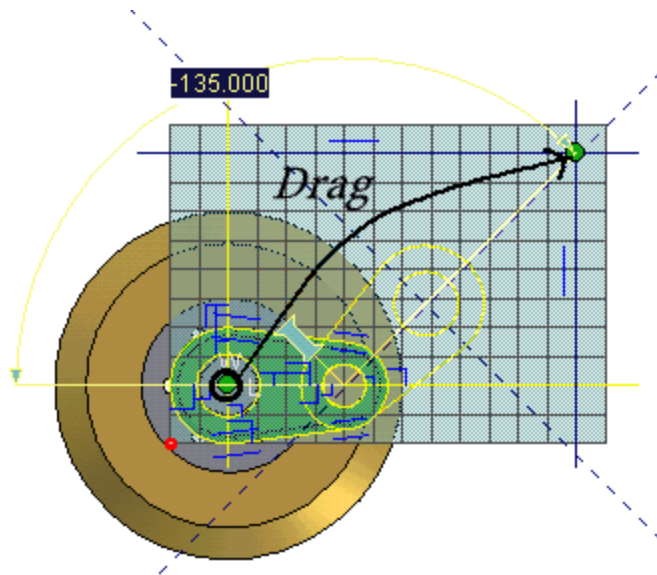
- Use the Vertical construction line tool  and the Horizontal construction line tool  to create the two construction lines approximately as shown (these tools are located on the 2D Construction toolbar). Use endpoint position dimensions to position the vertical line at a Horizontal distance of 12 and the horizontal line at a Vertical distance of 8.



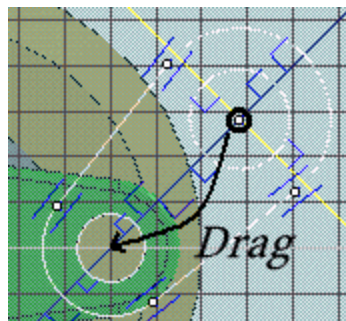
- Select all curves on the profile using either Edit, Select All or by typing Ctrl-A.
- Now **shift-click** the two construction lines you just created to deselect them. All lines on the drawing grid should be selected **except** these two construction lines, including the horizontal and vertical datum lines.
- Click the Rotate 2D Curves tool , then drag the pin icon from the center of the large circle to the center of the small circle as shown.



5. Drag the center of the large hole and snap it to the intersection of the two construction lines as shown.



6. Deselect all curves, then select the vertical datum line (which also has been rotated) and drag it down to the center of the small hole as shown.



7. Select the two circles **in the order shown** and use the endpoint dimension to change the Horizontal distance to 8. Be careful to select the circles in the correct order and to use the Maintain End Conditions option.

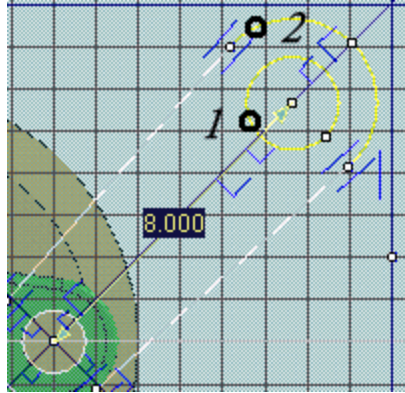
Tip

If you drag with the right mouse button you are presented with a

Tip

right drag technique will be used later in this tutorial.

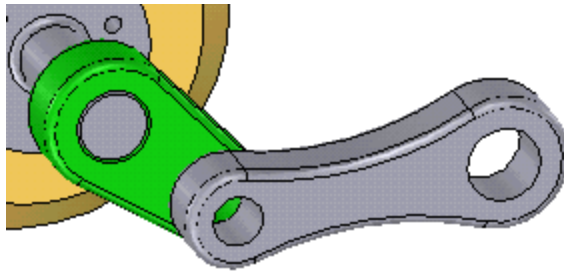
*The **Move 2D Curves** and **Scale 2D Curves** tools work very similarly to the Rotate tool, including the option to right-drag for Move or Copy.*




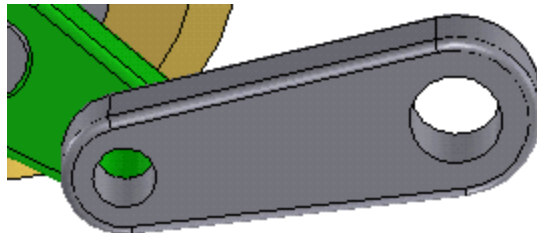
8. Click Finish Shape.


Specifying More Sophisticated Tangency Conditions

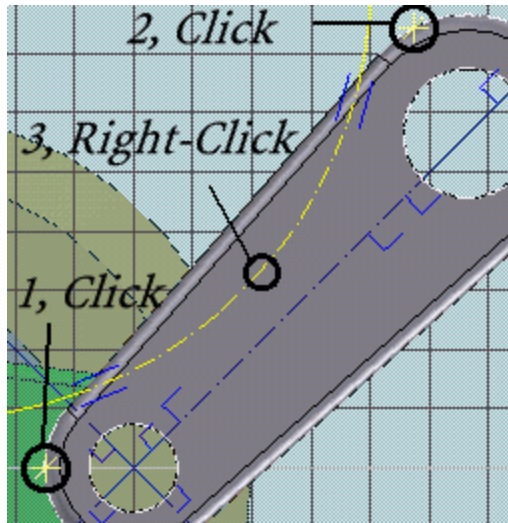
In this section, you will modify the connecting rod by replacing the straight line segments with tangent arcs to create the shape shown. You'll also learn how to handle Blend regeneration errors that sometimes occur when you make radical changes to cross-section geometry.




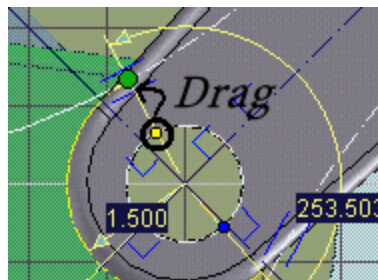
1. Use the Blend Edges  tool to blend the outer edges of the newly created part with a radius of 0.25 as shown. **It's important that you do not skip this step!**




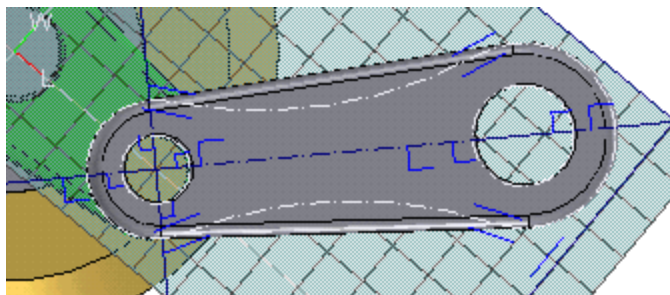
2. Edit the cross-section of the new part and delete the two straight lines from the profile.
3. Click the Circle: 2 Tangents 1 Point tool  and click the points in the order shown. For the third click, use the right mouse button to indicate that you want to enter a radius instead of pick a specific point.



4. Enter a radius of 10. Note that multiple solutions are possible and the system may not always choose the one you expect on your first try. If this happens, delete the circle and try again, attempting to right-click at a location as close to the desired radius as possible.
5. Repeat steps 3 and 4 to create another circle along the bottom side of the two arcs (symmetric about the datum line connecting the two hole centers).
6. Turn on Show Curve Dimensions , select the arc shown, and use one of its blue angle handles to snap its endpoint to the tangent circle as shown below. As you drag the handle, trace your cursor along the edge of the circle until the arc snaps to the exact tangent point. Repeat for the other side of this arc and then for both sides of the other arc.



7. Use the Trim Curve Between Curves tool  to trim away the outer side of the two large circles. Your cross-section should now look like this:



8. Click Finish Shape. The cross-section extrudes correctly, but a blending error occurs. The blends created in Step 1 fail to regenerate because the edges they were

Tip

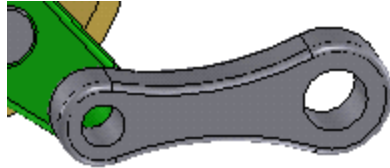
Numerous other commands also provide a right-click option to give you more power, including the following:

- **Circle: 1 Tangent 2 Points**
- **Circle: 2 Tangents 1 Point**
- **Circle: 3 Points**
- **Perpendicular line**
- **Tangent Line**

originally attached to have been deleted and replaced.

9. Click the Edit button in the error dialog and re-specify the blend just as you did in Step 1.

Alternatively, you may choose to Delete the failed blend (and recreate it later if you wish), or to Close the error message and ignore the failed blend for now (it will be displayed in the Scene Browser with an icon that has an “X” through it indicating it has failed).



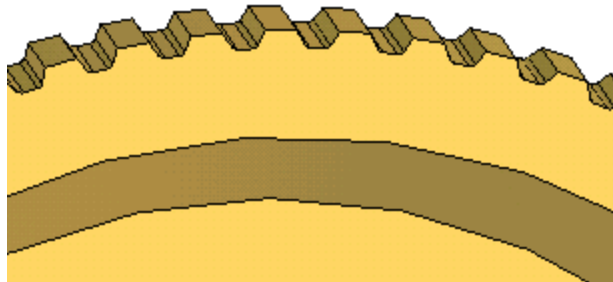
10. Rename this new part to “Rod” as shown.



Tip

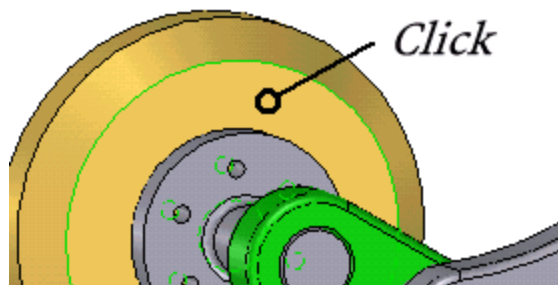
Projecting Edges as Construction Geometry

In this section, you'll go back to the Flywheel shape and add some gear teeth as shown.



Tip


1. Click Extrude Shape then click anywhere on the back face of the flywheel as shown.

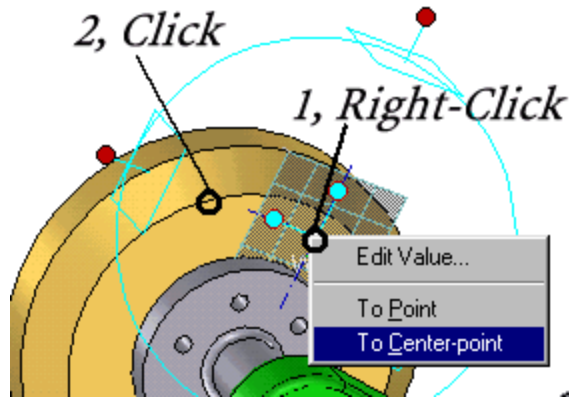


2. Select Remove Material in the Extrude Shape Wizard then click Finish.

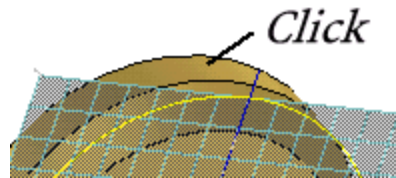
Now use the tribal to position the origin of the drawing surface at the center of the back face of

the flywheel.

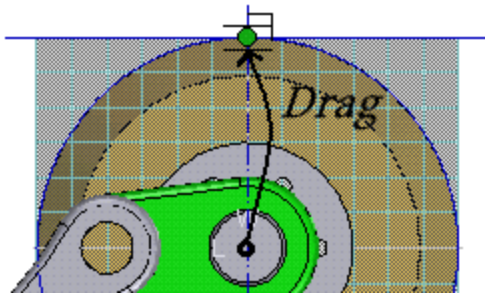
- Click the Triball tool , then **right-click** on the center of the triball as shown and select To Center-point. Now click the circular edge of the back face of the Flywheel as shown. The origin of the drawing surface should now be located at the center of the back face of the Flywheel.




- Turn off the Triball tool.
- Click the Construction Drawing tool , then click Project Edges  and click the outermost cylindrical face of the Flywheel as shown.



- Turn **off** the Construction Drawing and Project Edges tools then click any blank spot on the drawing surface to deselect the projected circle. Notice that the circle has been projected as construction geometry instead of regular geometry.
- Select the horizontal datum line and drag it up to the intersection of the vertical datum and the projected circle as shown.

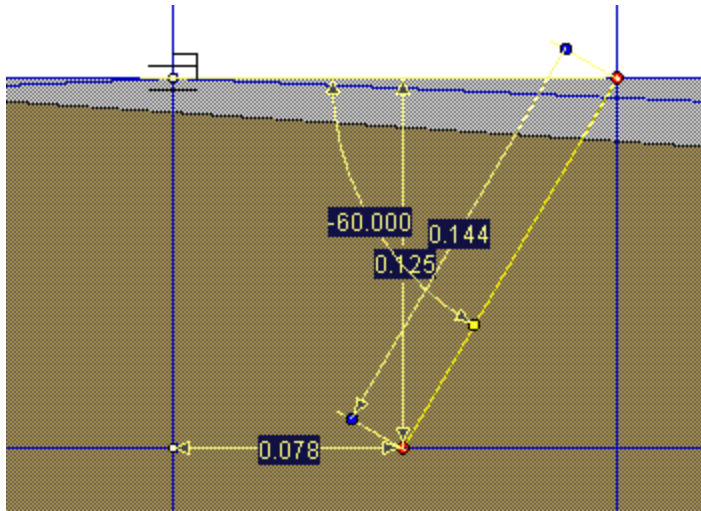


- Create a vertical construction line at a horizontal distance of 0.15 inches to the right of the vertical datum line, and a horizontal construction line at a vertical distance of 0.125 below the horizontal datum line.
- Click Two Point Line  and create a line segment of arbitrary length originating at the intersection of the two construction lines just created. Then use the angular curve handle to adjust the angle to -60 degrees as shown. Finally, use the linear curve handle

*When the **Construction Drawing** tool is active, all geometry created using any of the 2D drawing tools will be created as construction geometry.*



*To switch a line from construction geometry to regular geometry (or vice versa) after it has been created, right-click the line and select **Use Outline for Construction Only**.*

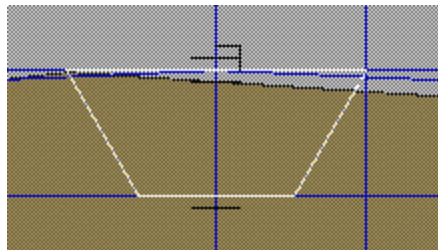
to snap the line to the horizontal construction line.



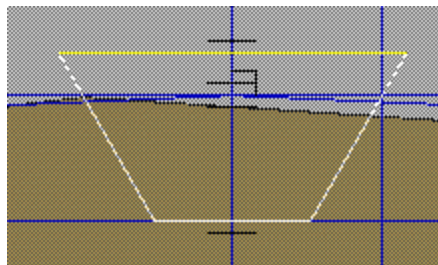
Note

Sometimes when you zoom in close on a circular edge, it may appear that the edge is jagged and inaccurate. Don't worry. The underlying geometry is completely accurate, but the graphics are displayed using a series of flat planes to approximate a curved surface.


10. While the line just created is selected, click Mirror Curves  then click the vertical datum line. A mirror copy of the line segment is created.
11. Use the Horizontal Line tool  to create two horizontal lines connecting the angled line segments as shown.

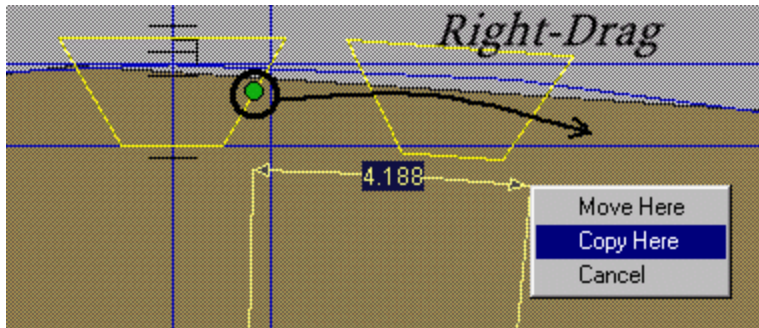


12. Shift-drag the topmost horizontal line upward by "just a little bit" as shown. It's important to hold down the shift key as you drag to ensure that the angles of the lines are maintained, but the exact distance of translation is unimportant.



You've just created a profile for cutting out material between two teeth, but now you need to make radial copies of this pattern all the way around the perimeter of the Flywheel.

13. Right-click any one of the 4 line segments comprising the trapezoid and choose Select Outline. All four lines should now be selected.
14. Click the Rotate 2D Curves tool  and use the **right mouse button** to drag the outline a few degrees clockwise as shown. When you release the mouse button, select the Copy Here option as shown.

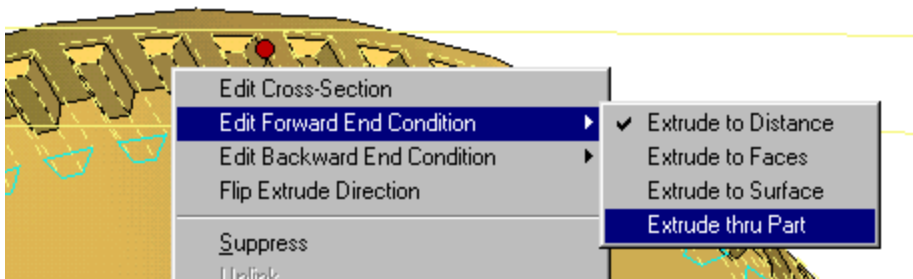


15. Enter 5 for Rotation Angle and $360/5 - 1$ for Number of copies.

16. Click Finish Shape.

Now you need to specify the depth of the cut to ensure that it goes all the way through the edge of the flywheel.

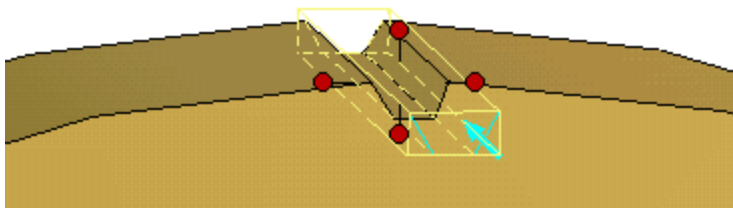
17. With the newly created shape selected in IntelliShape mode, right-click on one of its faces and select Edit Forward End Condition, then Extrude thru Part. This ensures that the extrusion always go through the entire part, regardless of any dimensional changes made to other features of the part.




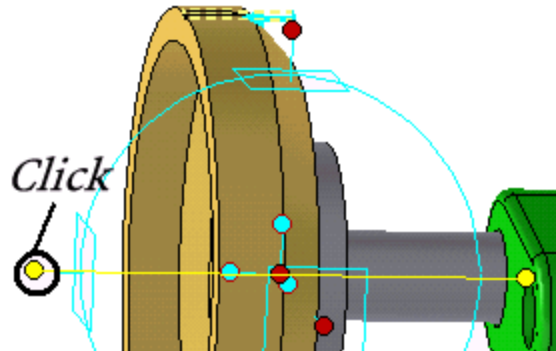
Creating Linked Instances of a Cross-Section

The technique you just used to cut the teeth in the Flywheel may not always be the best approach. In this section, you will explore an alternative technique that provides superior power and flexibility for future modifications.

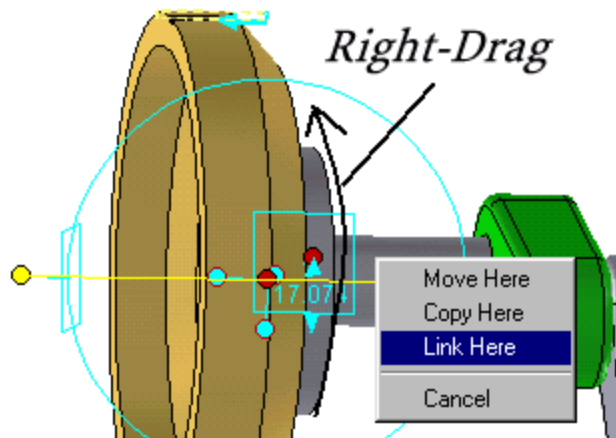
1. Edit the cross-section of the negative extrude shape created in the previous section.
2. Delete all of the trapezoidal shapes **except** for the original one created prior to copying the others. Using the Box Select tool will help to speed up the selection and deletion of the lines.
3. Click Finish Shape. You should now have only a single cutout as shown.




4. While the shape is selected, click the Triball , then click one of the handles on the axis of the Triball that runs parallel to the axis of the Crank Shaft as shown. The axis is highlighted yellow to indicate that it is “locked”. It is temporarily constrained to only allow motion about or along this axis



5. Now use the **right mouse button** to rotate the Triball about the locked axis using an upward dragging motion as shown. Upon releasing the mouse, select the Link Here option. Enter $360/5 - 1$ for Number and 5 for Angle then click OK.



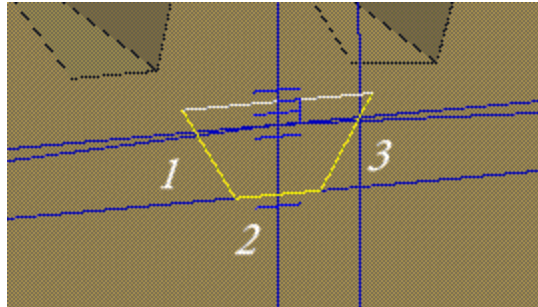
6. Turn OFF the Triball then open the Scene Browser  and notice the shapes listed under the Flywheel shape. Now instead of just one Extrude IntelliShape, you have 72 linked instances of a single Extrude shape! If you make a geometric change to any one of these shapes it will apply to all of them, as you shall see in the following section.



2D Fillets Vs. 3D Blends

You have already seen how Blend IntelliShapes can be used to round or fillet the edges of your parts. In this section you will use 2D fillets instead of 3D blends to round the edges of the teeth on the flywheel.



1. Select any one of the linked “tooth” IntelliShapes and edit its cross-section.
2. Select the three bottom lines of the trapezoid as shown.



3. Click Smooth 2D Corners , enter a Radius of 0.05 and click OK. The bottom corners are rounded with the specified radius.
4. Delete one of the rounded corners (right click the arc line segment and select Delete, or select it then press the Delete key). Notice that the sharp corner reappears.
5. To restore the rounded corner using another technique, click Fillet  and select the vertex of the corner to round, then click again to specify an approximate center for the arc. Press Esc to end the Fillet command.
6. Right-click on the rounded corner, select Curve Properties, and change the Radius to 0.05.
7. Click Finish Shape. Notice that this change is reflected in all linked instances of the shape.

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