IRONCAD

THE FASTEST AND EASIEST WAY TO 3D

Getting Started Guide

Copyright © 1995 - 2011 IronCAD, LLC All rights reserved.

Partial Copyright © 2011 CAXA Technology, Co., Ltd.

IRONCAD, INOVATE, IRONCAD DRAFT, TeamVault, InnovationSuite, SmartSnap, SmartRender, SmartDimension, IntelliShape, TriBall, SmartMotion, IronTool, Design Flow, Direct Face Modeling, Hyper-operability, SmartPaint, SmartUpdate, SmartBehavior, IntelliFace, IntelliSurface, e-Engineering, and IronCAD the Next Industrial Revolution, are registered trademarks of IronCAD or its Licensor.

CAXA DRAFT is copyrighted by CAXA Technology Co., LTD

CAXA, CAXA DRAFT, and CAXA DRAFT MECHANICAL are registered trademarks of CAXA Technology Co., LTD or its Licensor.

ACIS, SAT, and InterOp are registered trademarks of Spatial Technology, Inc.

Parasolid © 2011 Siemens Industry Software Limited. All Rights Reserved.

D-CubedTM 2D DCM, 3D DCM, CDM © 2011 Siemens Industry Software Limited. All Rights Reserved.

Granite is a registered trademark of PTC, Inc.

OpenHSF and HOOPS are trademarks of Tech Soft 3D.

SentinelLM is a trademark of SafeNet Inc.

Adobe Illustrator, Adobe Acrobat, Adobe 3D PDF, Adobe 2D PDF, and Postscript are trademarks of Adobe Systems Inc.

Microsoft®, MS-DOS, Windows, Windows NT, Visual Basic®, Windows 2000, Windows XP, Windows 98, Vista, Window 7, Visual Basic for Applications and Excel are registered trademarks or trademarks of Microsoft Corporation in the United States and/or other countries.

TIFF is a trademark of Aldus Corporation, an Adobe Company.

AutoCAD, DXF, DWG, and 3D Studio are registered trademarks of Autodesk, Inc.

"TEIGHA", the TEIGHA logo, "Open Design Alliance", and the Open Design Alliance logo are trademarks of the Open Design Alliance in the United States and/or other countries. DWG is the native and proprietary file format for AutoCAD® and a trademark of Autodesk, Inc. The Open Design Alliance is not associated with Autodesk.

TARGA is a registered trademark of Truevision, Inc.

Copyright © 1995 - 2011 IronCAD

Apple and Macintosh are registered trademarks of Apple Computer Corporation.

The Graphic Interchange Format is the copyrighted property of CompuServe Incorporated. GIF is a service mark of CompuServe Incorporated.

The Kodak Photo Access CD is the copyrighted property of, and Kodak is a registered trademark of Eastman Kodak Company.

Libtiff is the copyrighted property of Sam Leffler and SGI: ©1988-1995 Sam Leffler; ©1991-1995 SGI.

Netpbm is the copyrighted property of its individual authors.

This software is based, in part, on the work of the Independent JPEG Group.

IronCAD acknowledges the School of Architecture Property and Planning, University of Auckland, as the source of many of the image scans contained in the IRONCAD™ image library. IronCAD also acknowledges the following sources for software included with IRONCAD:

- qvlib VRML reader - SGI.
- NETPBM Library - Jef Poskanzer, et al.
- Zlib - the general purpose compression library is courtesy of Jean-loup Gailly and Mark Adler.
- libpng - the PNG image file format library is courtesy of its contributing authors and Group 42, Inc.

Yafray is copyrighted ©2001 by Alejandro Conty Estévez and Alfredo de Greef and is licensed under the LGPL license.

- IRONCAD Advanced Rendering Add-in is free software: you can redistribute it and/or modify it under the terms of the GNU General Public License as published by the Free Software Foundation, either version 3 of the License, or (at your option) any later version.
- This Add-in is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU General Public License for more details.
- A copy of the GNU General Public License and the Lesser GPL can be found in the ReadMe Folder. If not, see <http://www.gnu.org/licenses/>.
- To obtain the source code for the Advanced Rendering Add-in which is a derivative work of Yafray, please contact IronCAD support for details.

A portion of IRONCAD uses OpenEXR which is Copyrighted (c) 2002 by Industrial Light & Magic, a division of Lucas Digital Ltd. LLC All rights reserved. Additional license information can be found in the ReadMe folder under OpenEXRLicense.txt

A portion of IRONCAD use Radiance software (http://radsite.lbl.gov/) developed by the Lawrence Berkeley National Laboratory (http://www.lbl.gov/)." Additional license information can be found in the ReadMe folder under RadianceLicense.txt.

Copyright © 1995 - 2011 IronCAD

Additional designations used by manufacturers and sellers to distinguish their products and all other products or name brands are registered trademarks or trademarks of their respective holders.

Use, duplication or disclosure by the U.S. Government Departments and Agencies are as set forth in DFARS252.227-7013.

The IRONCAD Getting Started Guide

IRONCAD is a computer aided design (CAD) system using 3D solid modeling techniques to build a representation of an object, a design, a building or whatever you can conceive. This model can be as crude or as precise as you need. IRONCAD can be used as a very quick conceptual modeler to establish the early impact of a design or as a precise detail modeling system. Traditional 2D drawings are created by taking snapshots of the 3D models, in this way IRONCAD keeps both model and drawing in step.

IRONCAD was the one of the first systems to bring full 3D design visualization, drawing creation and animation into the mainstream of desktop computing. IRONCAD uses drag-and-drop solid modeling to provide the easiest part design/drawing environment available, whilst offering the same level of performance and compatibility found in higher-cost 3D applications.

TABLE OF CONTENTS

1. Design Project Phases ____ *8*

1.1. Building the part ____ **8**

1.2. Assembling multiple parts ____ **8**

1.3. Creating a 2D drawing of the part ____ **8**

1.4. Rendering the part ____ **8**

1.5. Communicating the part ____ **8**

2. Distinct Environments ____ *9*

2.1. 3D Scenes ____ **9**

2.2. 2D Drawings ____ **9**

2.3. History Based not History Bound ____ **9**

3. IRONCAD – Quick Start Guide ____ *10*

3.1. Starting IRONCAD ____ **10**

3.2. Creating Your First 3D Model in IRONCAD ____ **11**

3.2.1. Creating a 3D Scene Document ____ 11

3.2.2. Using Drag & Drop in the Scene ____ 14

3.2.3. Resize Using Precise Values ____ 17

3.2.4. SmartSnap ____ 23

3.2.5. Using the TriBall ____ 29

3.3. Communicating the Part ____ **32**

3.3.1. Creating a 2D drawing ____ 33

3.3.2. Creating a Photo-Realistic Rendering ____ 45

3.3.3. Exporting the 3D model ____ 54

3.4. Working with Assemblies to Complete the Design ____ **56**

3.4.1. Designing an Assembly ____ 57

3.4.2. Working with Boolean Operations ____ 69

3.4.3. Using Intelligent Features ____ 72

3.4.4. Using an Existing Intelligent Hole for the Basis of a New Hole ____ 73

3.4.5. Creating Sketch Based Features ____ 79

3.4.6. Working with Blends/Chamfers ____ 85

3.4.7. Positioning with Constraints ____ 88

3.4.8. Adding Draft ____ 95

3.4.9. Handling Errors in Feature Creation ____ 98

3.4.10. Creating Configurations of Different States ____ 99

3.4.11. Creating Exploded Assembly Drawings ____ 103

4. Using 2D Sketches in Part Design ____ *112*

4.1. Shape Handles vs. Sizebox Handles 113
4.1.1. Positioning of Features and Faces using Shape Handles 113
4.2. Using the Basic 2D Editing tools 117
4.2.1. Creating the Flywheel using a Spin IntelliShape 118
4.3. Creating Precise Sketches 127
4.3.1. Positioning Lines Using Endpoint Position Dimensions 127
4.3.2. Accurately Positioning Lines Using Curve Dimensions 129
4.3.3. Curve Handles 130
4.3.4. Changing the Orientation of Endpoint Dimensions 133
4.3.5. Positioning Multiple Lines Simultaneously 134
4.3.6. Finishing the Spin IntelliShape 136
4.4. Editing the Cross-section of an Existing IntelliShape 139
4.4.1. Setup the Part for Editing the Sketch: 139
4.4.2. Using the Offset Command 141
4.5. Using 2D Constraints 146
4.5.1. Adding Dimensional constraints 146
4.5.2. Adding Geometry Constraints 150
4.5.3. Making changes predictable 152
4.5.4. How Constraints Affect the Behavior of Shape Handles 157
4.5.5. Precise Model Changes Using Shape Handles 157
4.5.6. Overlapping Geometry Errors 158
4.5.7. Adding Full Design Intent to the Flywheel 159
4.6. Drag-and-Drop IntelliShape Modeling vs. 2D Cross-section Modeling 165
4.6.1. Remove of the Sketch Center Bore 165
4.7. Setting Part and IntelliShape Names and Color 168
4.7.1. Changing the color of the part 170
4.8. Using the Advance Shapes Catalog 172
4.9. Saving the part 174
4.10. Working with Multiple Parts in a Single File 175
4.10.1. Creating the Crankshaft 175
4.10.2. Adding the Shaft 179
4.10.3. Creating the Crank Arm 185
4.10.4. Creating Tangent Lines 191
4.10.5. Creating Tangency and Concentric Constraints 192
4.11. Parametric Constraints with Expressions 197
4.11.1. Parametrically Control the Crank Arm 197
4.12. Creating an Assembly 202
4.12.1. Making an Assembly from Existing Components 202
4.13. TriBall and Linked Components 204

4.13.1. Using the TriBall to Create Linked Shapes ____ 204

4.14. Adding Intelligence Using Parameters ____ 209

4.14.1. Parameter Table Format ____ 210

4.14.2. Adding Control to the Tooth Form ____ 211

4.14.3. Adding User Defined Parameters ____ 213

4.14.4. Establishing the Angle between the Teeth ____ 215

4.14.5. Relating the Diameter to the Number of Teeth ____ 216

4.15. Design Variations ____ 218

4.15.1. Adding Top Level Parameters ____ 219

4.15.2. Applying Design Variations Intelligence ____ 221

4.15.3. Testing the Design Variations ____ 223

5. The IRONCAD TriBall ____ 229

5.1. Anatomy of the TriBall ____ 229

5.1.1. TriBall Zone 1 ____ 231

5.1.2. TriBall Zone 2 ____ 231

5.1.3. TriBall Zone 3 ____ 233

5.1.4. Repositioning the TriBall ____ 234

5.2. Practical Examples using the TriBall ____ 235

5.3. Using the TriBall's Orientation Handles to position parts ____ 236

5.4. Using the TriBall's center point to position parts ____ 236

5.5. Temporarily constraining (declaring) an axis of the TriBall ____ 237

5.6. Parallel to Edge command ____ 238

5.7. Perpendicular to Face command ____ 238

5.8. Drag-and-Drop method of repositioning the TriBall ____ 239

5.9. To Point command ____ 239

5.10. Aligning the Dovetail component ____ 240

5.11. The TriBall Reverse command ____ 242

5.12. Point to Point command ____ 242

5.13. To Center Point command ____ 245

5.14. Using the TriBall's 'Copy in Plane' option ____ 246

5.15. Using the TriBall to create linked copies ____ 249

5.16. Orientating using existing faces to 'snap to' ____ 251

5.17. Using increment snapping with the TriBall ____ 252

5.18. Creating a Radial Array Pattern with the TriBall ____ 253

6. IRONCAD Drawing Creation ____ *256*

6.1. Basic Drawing Creation ____ **258**

6.1.1. Creating the Standard Views ____ 258
6.1.2. Changing the View Scale and Rendering options ____ 260
6.1.3. Moving Views ____ 262
6.1.4. Creating a Datum Reference Frame ____ 262
6.1.5. Adding SmartDimension's ____ 264
6.1.6. Modifying Individual Dimensions ____ 270
6.1.7. Adding a Feature Control Frame (Geometric Tolerance) ____ 272
6.1.8. Creating center lines ____ 274
6.1.9. Creating Reference Curves ____ 275
6.1.10. Multi-Sheet Drawings and Supplementary Views ____ 278
6.1.11. Adding a Detail View ____ 282
6.1.12. Creating an Auxiliary View ____ 283
6.1.13. Orientating center marks ____ 285
6.1.14. Creating a Hole Table ____ 286
6.1.15. Using the Sheet Drawing tools ____ 288
6.1.16. Styles and Layers ____ 290
6.1.17. Adding a Layer ____ 291

6.2. Part/Drawing associativity ____ **293**

6.3. Exporting the Drawing ____ **294**

7. IRONCAD sheet metal application ____ *297*

7.1. Sheet metal stock ____ **298**

7.2. Adding a bend with stock ____ **299**

7.2.1. Changing a bends angle ____ 300
7.2.2. Re-orientating length handles ____ 302

7.3. Adding bends with combined stock ____ **304**

7.3.1. Bend ____ 304
7.3.2. Out Bend ____ 304
7.3.3. In Bend ____ 304

7.4. Adding a Bend Relief ____ **306**

7.5. Adding standard punch features ____ **310**

7.6. Adding round corners – Vertex Fillets ____ **313**

7.7. Re-using existing features via the catalog ____ **313**

7.8. Adding a cluster of holes ____ **314**

7.9. Manipulating features with the TriBall ____ **314**

7.10. Using the Miter tool ____ **315**

7.11. Using standard catalog items in the sheet metal design process ____ **316**

7.12. Unfolding the part 318

7.13. Creating a 2D drawing of the sheet metal part 319

7.14. Defining custom punch shapes 321

8. IRONCAD Surface Modeling 325

8.1. Creating 3D Curves 325

8.2. Create Surfaces from Solids 332

8.3. Creating Surfaces 332

8.3.1. The Sweep Surface 332

8.3.2. Mesh Surface 334

8.3.3. Patch Surface 334

8.3.4. Ruled Surface 335

9. Photorealistic Rendering 338

9.1. Adding realism to the screen display 338

9.1.1. The Scene Background 338

9.2. Basic Rendering 341

9.2.1. Scene Exposure 341

9.3. Applying a Surface from a catalog 342

9.4. Modifying a surface texture 342

9.5. Saving a Surface Texture to a catalog 343

9.6. Setting a material property with a surface texture 343

9.7. Changing the finish of individual surfaces 344

9.8. Adding more realism with HDRI background lighting 345

9.9. Adding discrete lights 345

9.10. Finishing the scene 346

9.11. Adding a decal 347

9.12. Completing the rendering 348

9.13. Poster sized images 349

9.13.1. Ensure what you see is what you get 349

10. Animating with IRONCAD 352

10.1. Fundamental control of animation segments 352

10.1.1. Length, Height and Width 352

10.1.2. Key 352

10.1.3. TriBall 352

10.2. Animating the I-beam tool arrangement _____ 353
10.2.1. Opening the Top Plate _____ 353
10.2.2. Opening the Bottom Plate _____ 355
10.2.3. Animating the Guide Pins _____ 356
10.2.4. Adding a new animation segment to an existing animation path _____ 357
10.3. The SmartMotion Editor _____ 359
10.4. Changing the start point of an animation segment _____ 359
10.5. Creating a dynamic cut-away section. _____ 360
10.6. Exporting the animation _____ 362

1. DESIGN PROJECT PHASES

Within IRONCAD, there are five potential phases to any design project that you may encounter:

1.1. Building the part

First, create the part from IntelliShapes (IRONCAD's term for Features with Intelligence). Choose from appropriate shapes in the catalogs or create 2D profiles to be extended into custom 3D shapes.

1.2. Assembling multiple parts

To reflect the product structure or when it is desirable to manipulate multiple parts as one, join them as an assembly. While this option facilitates manipulating several objects simultaneously, each component of the assembly retains its individuality. In addition, Assembly Features which are cut operations that affect parts within assembly can be applied.

1.3. Creating a 2D drawing of the part

To communicate the design to many different departments of the company, it is essential to be able to create a 2D drawing of the 3D part/assembly. Simply choose the appropriate drawing size and drop the required views of the 3D model onto it. Alternative views, dimensions, annotation as well as other 2D drawing elements can then associate to the 3D model so that any changes to the model will be automatically reflected in the 2D drawing.

1.4. Rendering the part

As an additional communication tool, add colors and textures, lighting effects, bumps, reflections, and transparency to provide photo-realistic part renderings for inclusion in brochures, web sites, or simply to help communicate the idea graphically to other people.

1.5. Communicating the part

Finally, the part can be communicated to other companies with ease. IRONCAD provides extensive translation features for exporting your part to other software packages as well as providing many communication tools such as views and viewable packages that can be sent to the design chain for feedback.

Of course, not all projects will require all six of the above phases. Your job may be complete after the building and rendering phases for example.

2. DISTINCT ENVIRONMENTS

IRONCAD provides similar, but functionally different, user interfaces for completing a design project: the 3D scene and the 2D drawing. In addition, IRONCAD offers unique modeling environments to fit the design task at hand.

2.1. 3D Scenes

3D part design takes place in a scene. Scenes are composed of single or multiple parts/assemblies, created using the ACIS or Parasolid modeling kernel (and even a mixture that we can discuss in an advance lesson). They are saved as documents with an '.ics' extension.

2.2. 2D Drawings

Creation of 2D drawing views of 3D parts/assemblies takes place in the drawing environment. Drawings contain associative views of existing 3D parts/assemblies, and any additional annotations required to fully describe them. They are saved as documents with an .icd or .exb extension (depending on the drawing environment used).

2.3. History Based not History Bound

In addition to the two environments, IRONCAD uses a technique known as history based modeling in the 3D Scene. Each shape (or feature) of the design is recorded as an individual item and can be modified at will. However, unlike the vast majority of 3D design packages, IRONCAD does not suffer from limitations imposed by the order in which these features are created - in other words IRONCAD is not 'history bound'. For example, you might want to stretch a feature created at the early stages of the design to match the size of a feature added towards the end of the design – this is a very simple operation in IRONCAD, however in most other systems a feature can only 'see' features presented before it in the history tree. This rigid structure can also provide a unique design process and is also offer as an optional design model in IRONCAD called Structured Parts. Simply toggle the design type to create parts in either design mode that fits your specific design task at hand.

3. IRONCAD – QUICK START GUIDE

The concepts of IRONCAD are very quickly adopted. This section will introduce the basic functions of IRONCAD so that you can be productive in a matter of a few hours. The topics covered will be:

- Creating a concept 3D model
- Re-sizing the features to reflect desired dimensions
- Adding a few detail features
- Creating a photorealistic rendering
- Adding chamfers and blends
- Creating a detail drawing
- Building an assembly and detailing it.

This quick start guide will introduce the concepts of IRONCAD's unique drag and drop modeling. After the initial part has been modeled and a range of communication tools explored, it will be used to design a tooling assembly creating a range of parts through to a fully exploded and detailed assembly.

3.1. Starting IRONCAD

From the Windows Program group, navigate to the IRONCAD group and start IRONCAD.

If you have installed the system as part of a trial, IRONCAD will present a splash screen telling you how many days you have left of your trial and will open a web page to point you to a number of resources available from the IRONCAD web site. You will also see a check box that hides the trial license splash screen until the end of the trial. Clicking this will speed up the starting procedure, however it will also hide the number of days left of the trial (which can be seen in the Application Title bar for future reference). After starting IRONCAD, the opening screen of IRONCAD displays four options:

- ***Create a new Scene Document***

  A Scene Document is the IRONCAD 3D design environment. This is where designs from a nut to a complete battleship are produced.

- ***Create a new Drawing Document***

Communicating designs by 2D drawings will remain a requirement for a while to come. In IRONCAD, 2D drawings are created either by creating a 'free' 2D geometry or by taking snapshots of an existing 3D model. In the latter case, IRONCAD maintains the links between the model and the drawing ensuring that the drawing and the 3D model keep in step with one another.

- ***Create a new CAXA Draft Document***

  For those who are familiar with AutoCAD, drawings can be produced in a familiar environment. As with the standard IRONCAD drafting tool, views are created from the 3D model and links are maintained so that the drawing always remains in step with the 3D design.

- ***Open an Existing Document***

  A bit self explanatory really!

As with all sections of the guide the amount of detail provided by the text will reduce as the sections progress. Hopefully this will provide a clear introduction that does not repeat itself unnecessarily and hence speed through the key areas of IRONCAD.

3.2. Creating Your First 3D Model in IRONCAD

In this section, we will create a model that represents the IRONCAD I-beam icon. The process will introduce:

- Creating a 3D Scene document
- Drag and Drop conceptual modeling
- Resizing to precise dimensions
- SmartSnap
- The TriBall

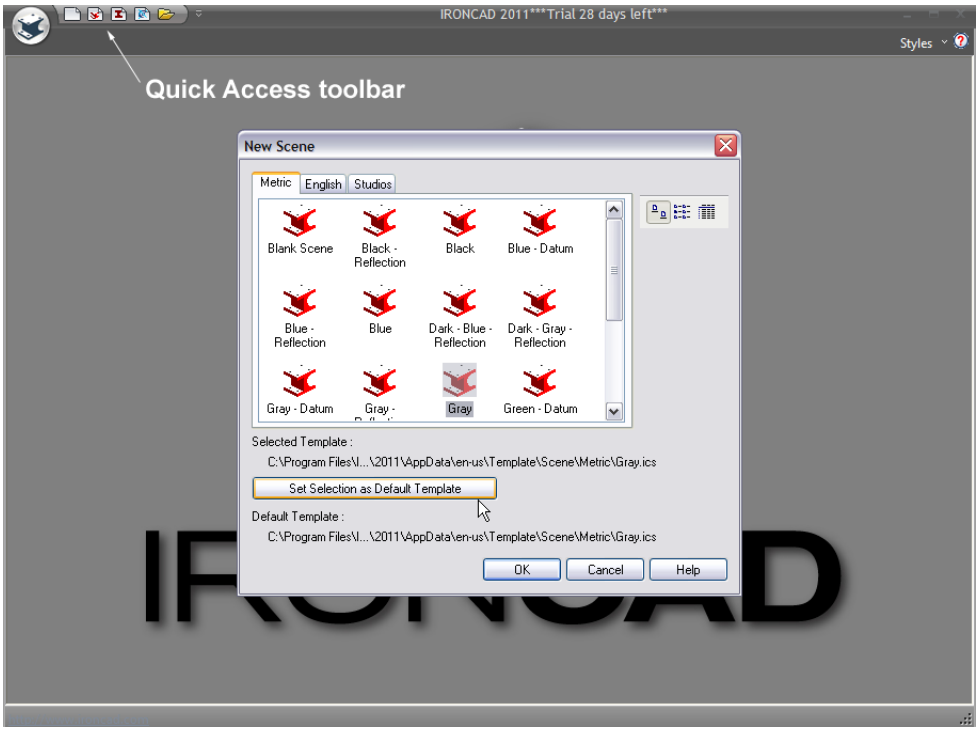
The whole process will be covered in 23 steps. The model can be produced in less than 2 minutes by an IRONCAD user familiar with these techniques. This tutorial should take 20 minutes including reading the text and becoming familiar with the concepts.

3.2.1. Creating a 3D Scene Document

1. **Select 'Create a New Scene Document'**

   IRONCAD will display a panel of pre-defined scene templates. Select to the Metric tab.

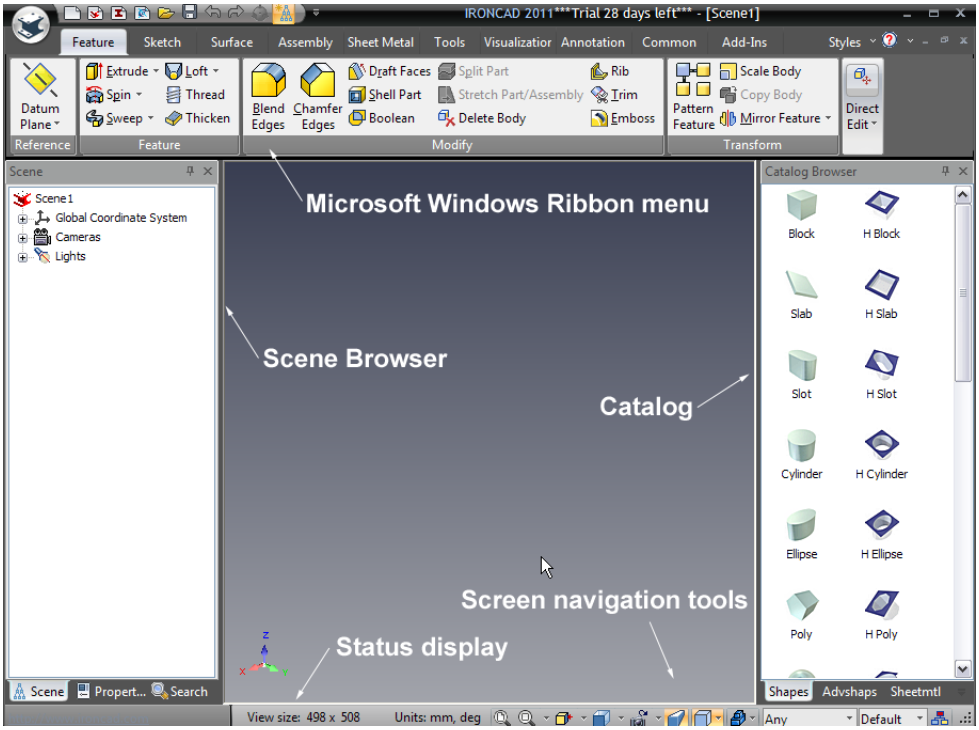
You will see a tab at the bottom of the panel allowing the current selection to be set as the default scene. IRONCAD has a shortcut icon on the windows Quick Access Toolbar - clicking on this will start a new scene using the default created in the previous step.



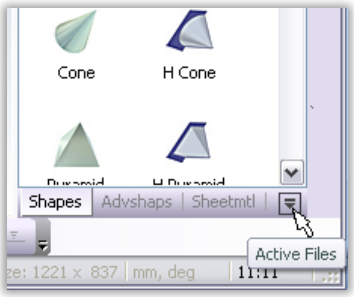
On the Metric tab, select “Gray” and click on “Set Selection to Default Template”. This can be reset by simply creating a new scene (but not using the Quick Access icon for New Scene).

The IRONCAD display combines standard Microsoft Windows functionality with IRONCAD specific functionality.

The top of the screen uses a standard Microsoft Windows Ribbon bar. Down the left hand side of the screen is the Scene Browser and on the right hand side is the catalog browser. On the bottom is the status display providing information about the scene, a useful single line prompt bar, and screen navigation tools to the right hand side.



The default catalog opened at startup is the Shapes catalog – this can be identified by its name found on the bottom of the catalog browser.



If you do not see the Shapes catalog, either:

- Click on the down arrow by the catalog names and select Shapes from the list,

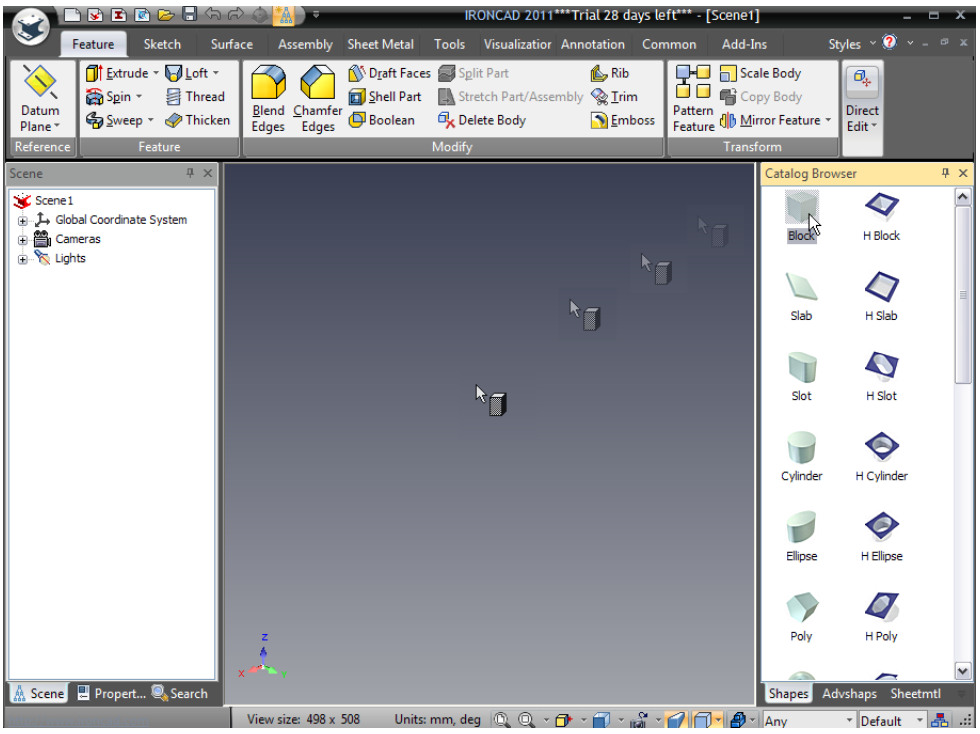
or if it's not listed:

- Select the Common tab on the Windows Ribbon bar,
- From the Catalogs panel, select Open and then browse to the Scene folder (IRONCAD should go straight to this folder, however if not it is found under Catalogs of your IRONCAD installation folder),
- Open the Metric folder and select Shapes.icc (the abbreviation for IRONCAD catalog).

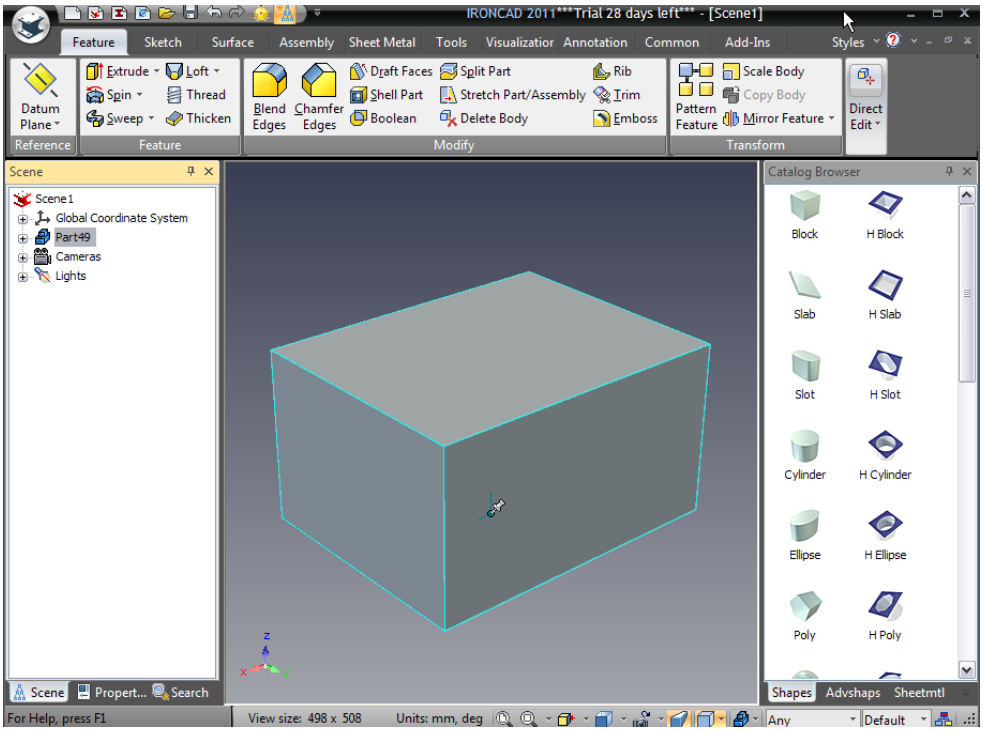
3.2.2. Using Drag & Drop in the Scene

2. Drag & Drop the Block Shape Into the Scene

Using the left mouse button, click on and hold the mouse button pressed on the Block catalog shape icon. With the left mouse button pressed, drag the mouse back into the scene and let then let go of the mouse button.



A block will be shown in the scene as displayed in the next image.



3. Rotate the Viewpoint of the Model

If you have a 3 button mouse click, the middle button and move the mouse around the screen. This moves the viewpoint not the model. This can also be achieved by pressing the F3 key and then the left mouse button. Deselecting this viewing mode is accomplished by pressing the Escape key or F3 again or simply releasing the middle mouse button.

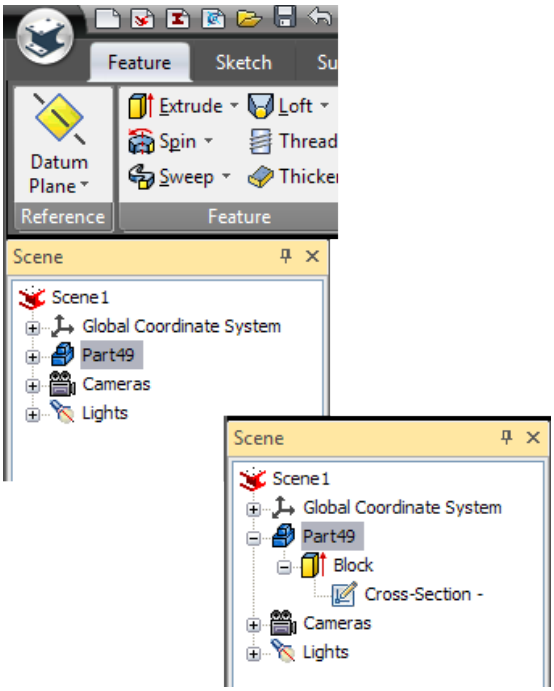
4. Part Level Selection

The Block is likely to be a gray color with black edges (if not, click in the background to deselect the current selection). Click once with the left mouse button on the Block. At this point, the edges turn a cyan color, in the scene browser IRONCAD displays a blue icon representing the part, and the part name is highlighted in the scene browser. This is referred to as 'part mode'.

5. Expand the Part in the Scene Browser

In the Scene Browser, you will notice a '+' by the side of the part icon. Click on the '+' and IRONCAD will expand the display to show the contents of the part in a very similar way to a standard windows explorer action. After

expanding, you will see another icon this time called “Block” in a yellow color with an arrow indicating this is an extruded part. You will notice this can be further expanded to reveal the originating 2D profile of the part – more of that later.



6. Select to the IntelliShape Level

Clicking on the part a second time will change the highlight from cyan to yellow (in the scene browser the highlight will move from Part to Block). At this point, IRONCAD will display 6 handles on each face of the block (representing a box around the geometry). The handles will be a short thin line at 90 degrees to each surface terminating in a red sphere. When the boundary is shown in yellow the item is in “IntelliShape” mode.

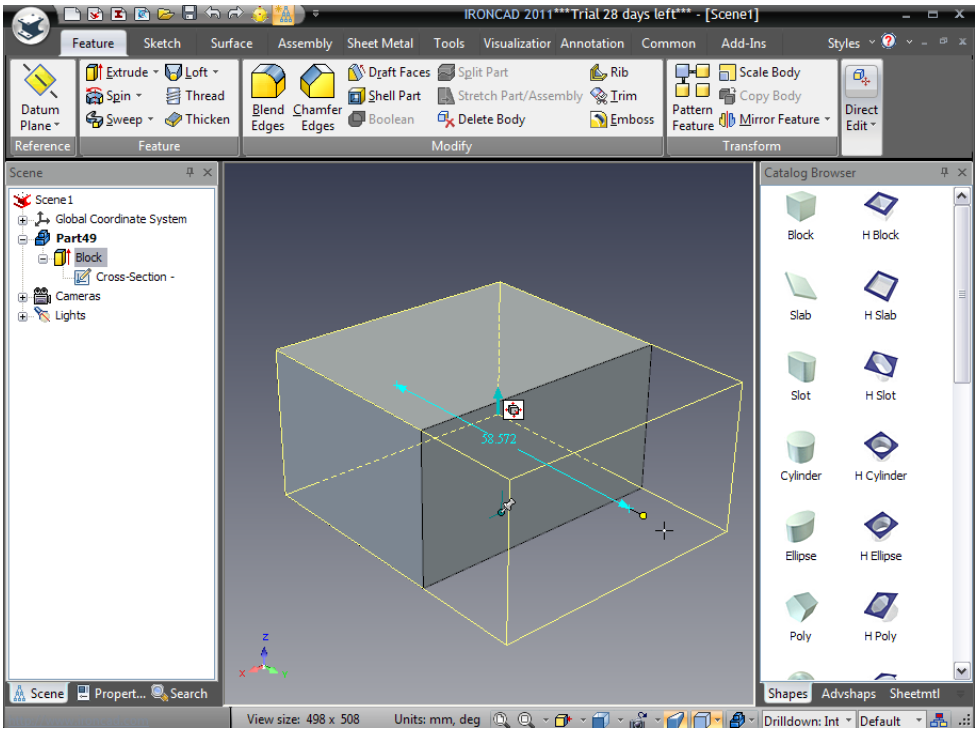
7. Select an IntelliShape Handle

Move the cursor over one of the red spheres and click and hold the left mouse button. IRONCAD will change the red sphere to a yellow sphere.

8. Resize the IntelliShape by Push & Pull of Handles

With the mouse button pressed, move the handle by moving your mouse. The yellow bounding box will move with the handle. In addition, IRONCAD

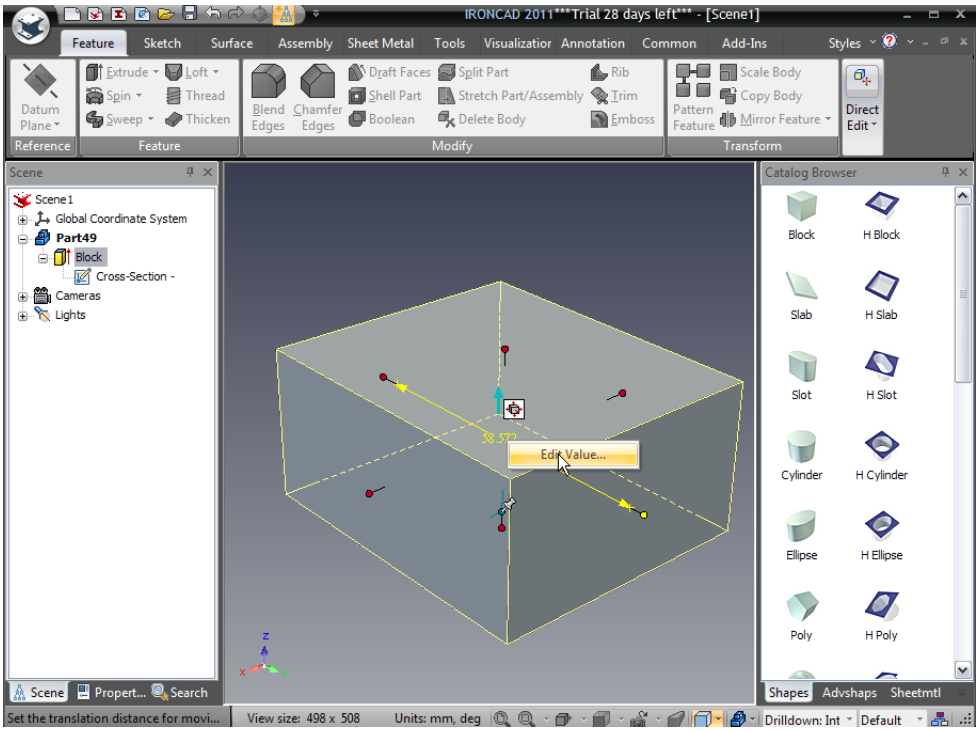
displays a dimension in line with the movement. When you release the mouse button, the model will reflect the new position of the handle.



3.2.3. Resize Using Precise Values

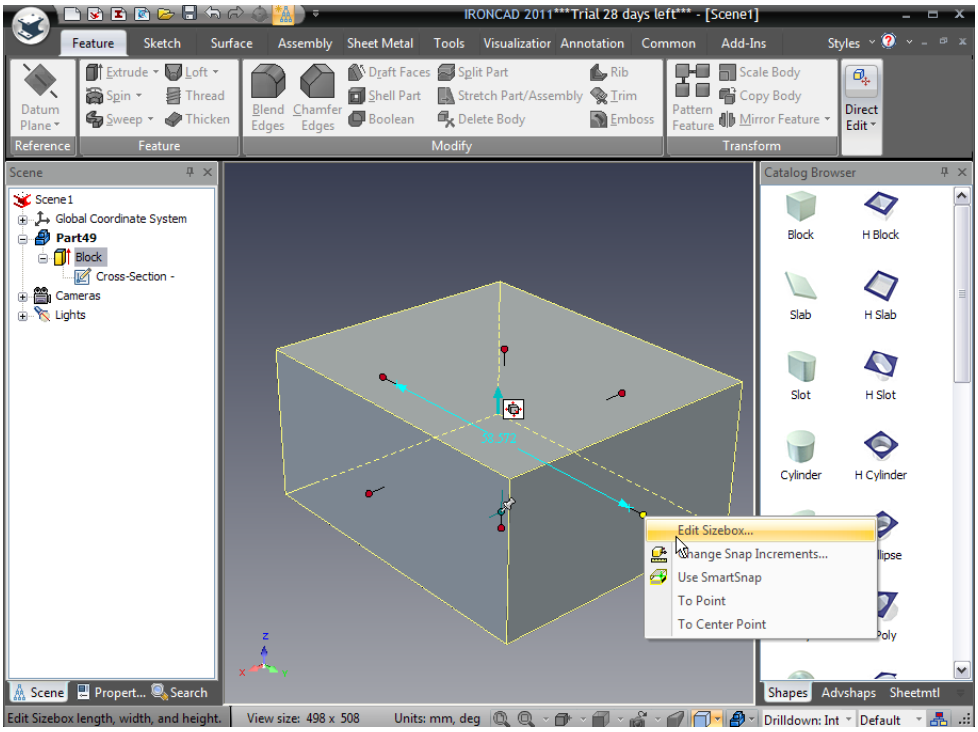
9. Set a Specific Dimension Size on the IntelliShape

Right-click on the displayed dimension that is shown after releasing the mouse button to set a size for the block. IRONCAD will provide a dialog box in which you can set the specific value. Also, you can double-click on the dimension to access the edit control as well.



10. Editing Sizebox Handle Values

If the dimension is not shown, right-click on the handle and it will change to yellow and IRONCAD will display a dialog box. Selecting the first option, Edit Sizebox, results in another dialog box showing the Length, Width, and Height of the shapes Sizebox. The highlighted value in the dialog is the value of the current handle selected. When changing this value, the IntelliShape Sizebox will change with respect to the handle directly opposite the one selected (the dimension is the total distance between the handles). If you edit the non-selected value, the Sizebox will change symmetrically. For this example, type in any value for the shape to re-size.



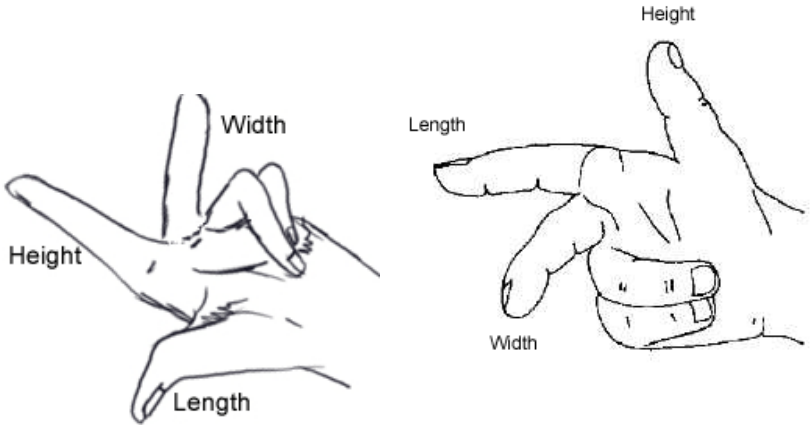
You have now created your first model, stretched it, and applied a specific size to it. This is one of the fundamental concepts of the freedom of IRONCAD which is essentially sketching in 3D using the left mouse button and adding precision with the right mouse button.

11. Sizebox and Length, Width and Height

Each item used in IRONCAD uses the concept of a Sizebox, which is essentially a bounding box that encompasses the extents of the item – be it a block, a part, or an assembly. As each item potentially sits at different orientations to others in the part of assembly, IRONCAD uses the concept of Length, Width, and Height not X, Y, and Z. While this may seem strange to some, it is actually one of the concepts that frees the constraints imposed by other systems. IRONCAD does use X, Y, and Z but only where it makes more sense.

Each Sizebox has an anchor which is in essence the datum point for the Sizebox. IRONCAD uses the anchor to orientate items with respect to other items for drag and drop and when changing values in the Sizebox dialog boxes. To reduce the amount of clutter on the screen the anchor is represented by two thin lines, one longer than the other. The longer one is the Height, the shorter one is aligned with the Length direction. The Width

direction follows the left hand rule or right-hand (depending on your region):

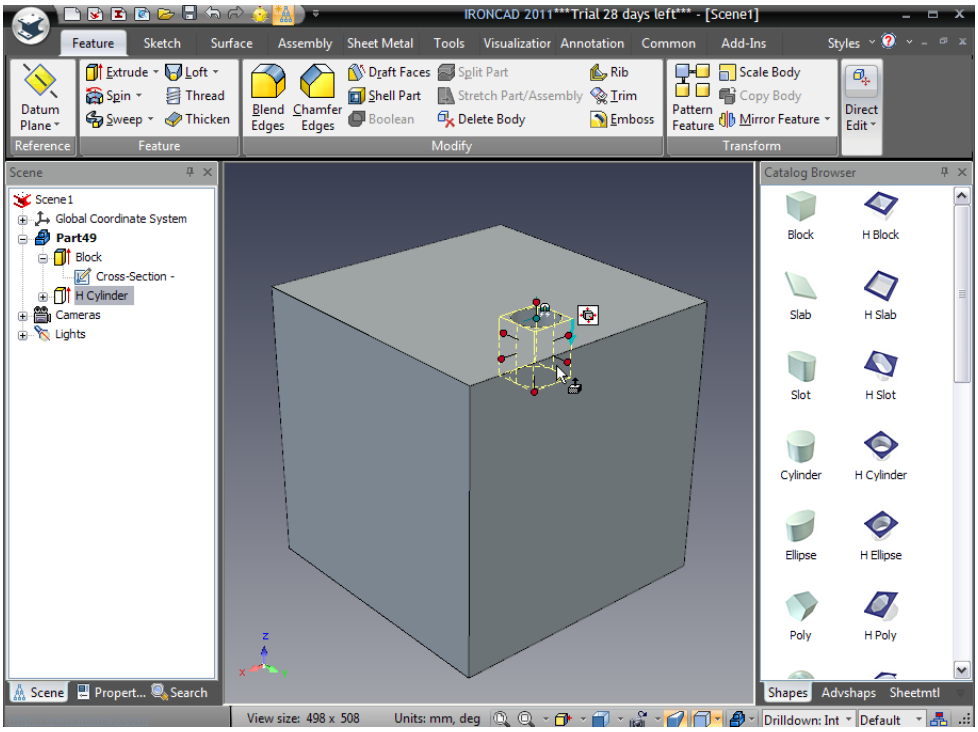


Back to the model, re-select the handle used previously with the right mouse button and select “Edit Sizebox”. Set each value to 50. IRONCAD will change the block into a 50mm cube. You will notice:

- The selected handle moves with respect to its opposing handle,
- The other two handles will change the Sizebox symmetrically about the center to adjust the size.

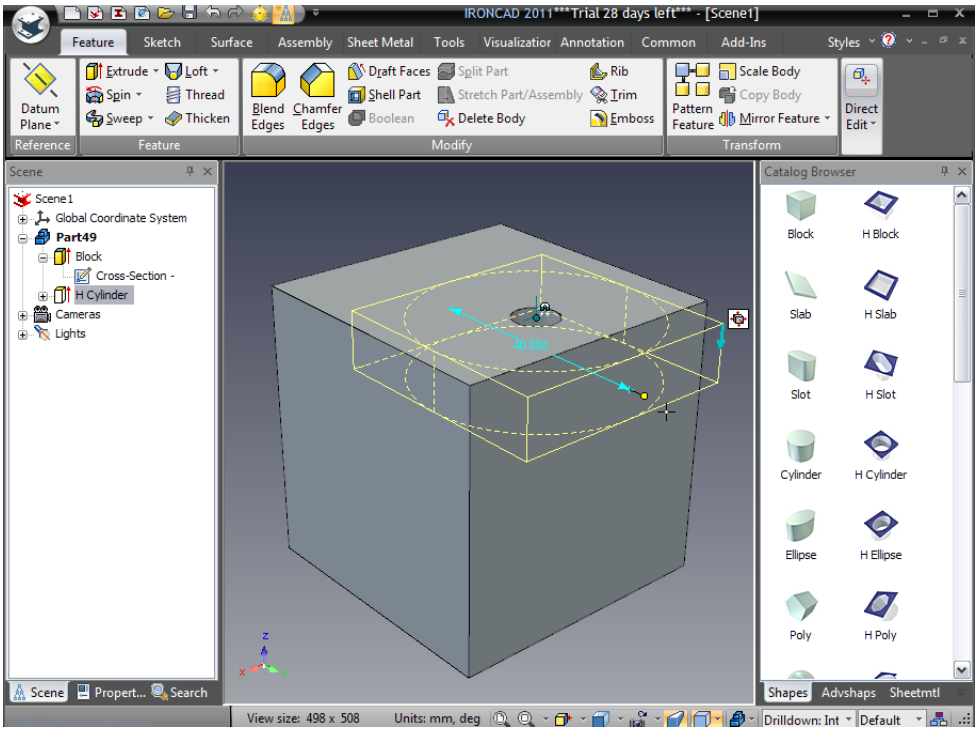
12. Adding Additional Features to the Block

From the catalog, using the left mouse button, drag an H Cylinder onto the existing block. IRONCAD will cut a hole into the block where you drop the item and show the Sizebox handles. The cylinder is added to the structure in the scene browser – its icon being a white extrusion suggesting a hole. The name H Cylinder is the name of the catalog entry and this name can be changed to help identify the feature later in the design if needed by clicking twice on the name (not a double click).



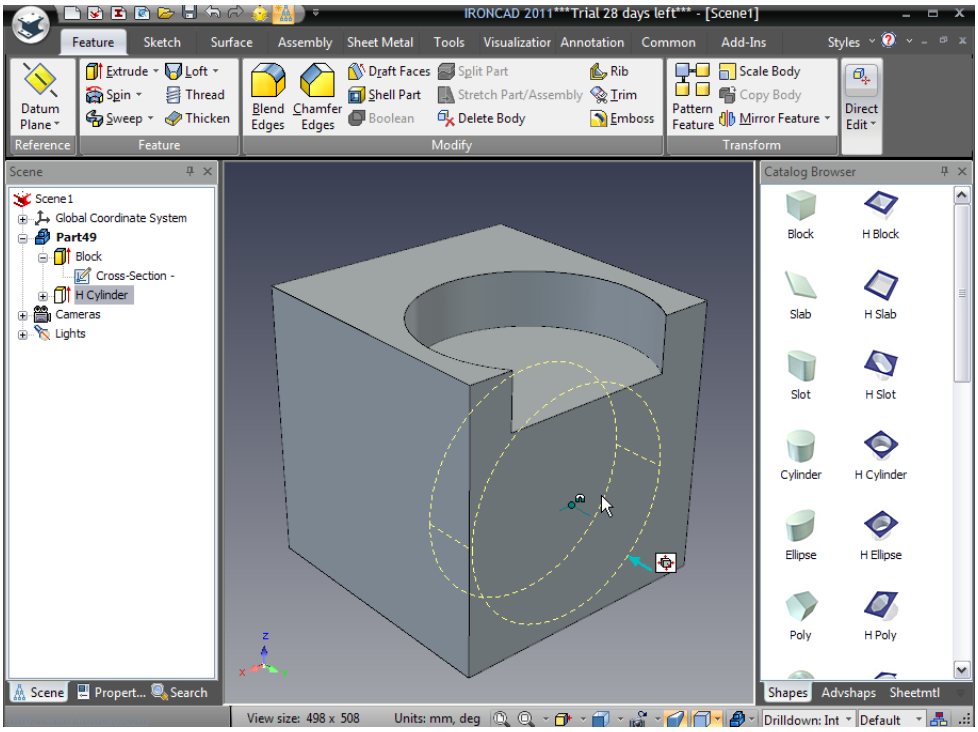
13. Changing the Size of an IntelliShape

Drag one of the handles defining the diameter and its corresponding handle will change too. Certain catalog items such as cylinders operate such that the length and width handles are linked – changing one will cause the other to change by the same amount. This can be set in the properties which we can discuss later in this tutorial.



14. Cruising Shapes by Drag and Drop

With the hole highlighted at the IntelliShape level, click and hold the left mouse button when the mouse pointer is over any of the surfaces of the hole and drag the hole over other faces of the block – this is referred to by some as cruising. Letting go, the mouse button will move the H Cylinder to that face.



3.2.4. SmartSnap

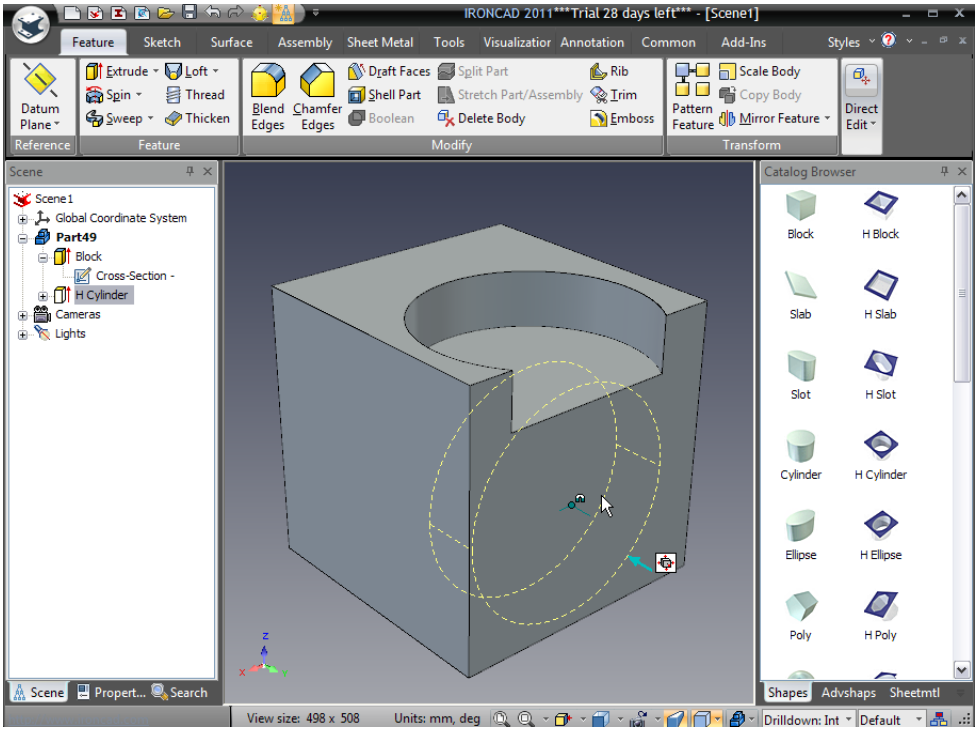
"Cruising" a shape allows complete freedom to explore the design. However, to position the shape accurately you can add dimensions, use the TriBall (introduced later), or use SmartSnap.

Every face and edge of an IRONCAD model has intrinsic intelligence relating to specific points such as end points and midpoints of edges, centroids of a surface, or the surface itself. SmartSnap is accessed by pressing the shift key at the same time as dragging.

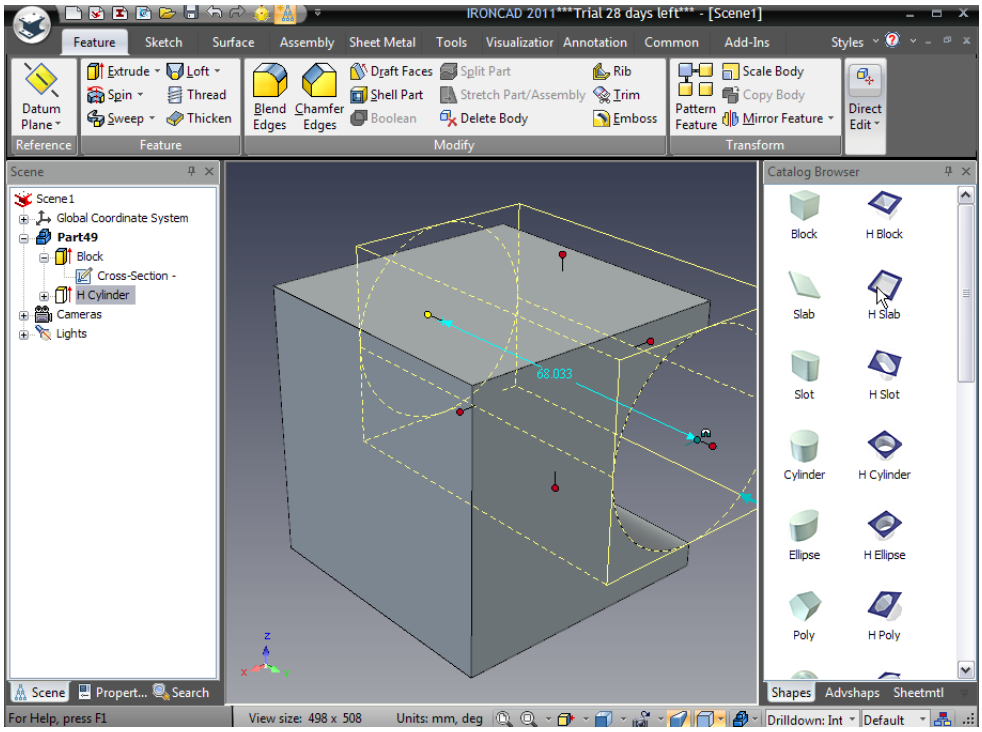
15. Precisely Position Using SmartSnap

Select the hole at IntelliShape mode and position the mouse pointer over the anchor of the hole. Click and hold the left mouse button. Now click and hold the shift key while dragging the hole around. IRONCAD will snap the hole to face centers, end points of edges and edge mid points as you cruise the hole about the Block.

Take note of the graphics feedback and how the hole snaps in position. Use the functionality to snap the hole to one of the edges of the block. Release the mouse first and then the Shift key to maintain the snap.



Drag the hole so that it cuts all the way across the face and by one of the methods used above set the diameter to 30mm so the model looks like the image below.

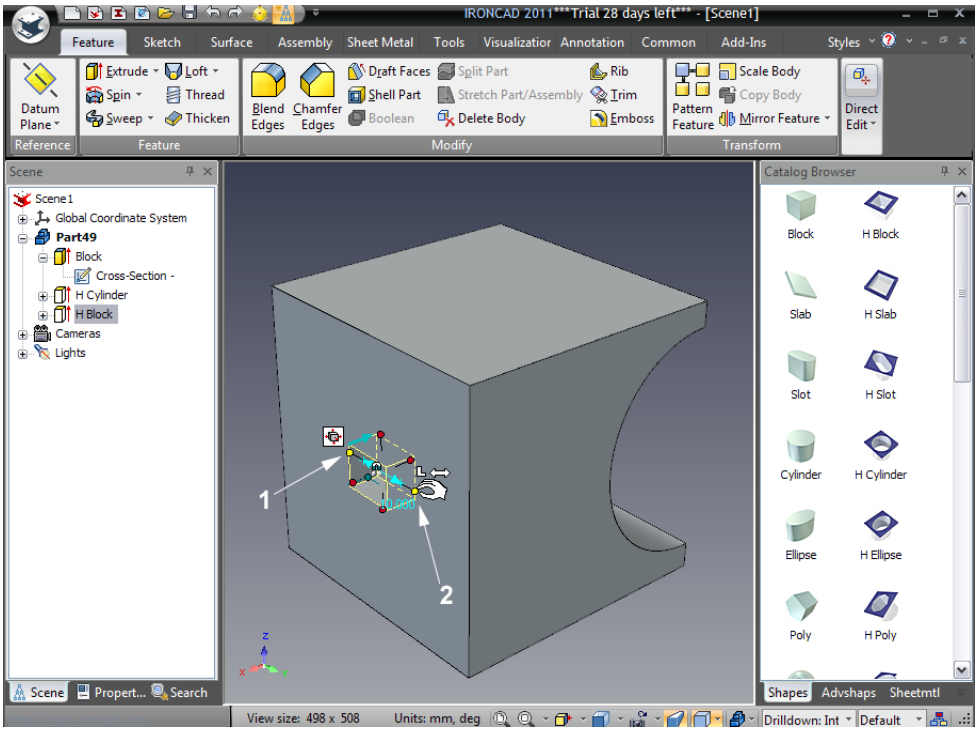


16. Add an Additional H Block IntelliShape

Now drag a H Block from the catalog onto the face of the Block opposite to the one with the hole cutting across it (in other words the back side of the block).

17. Resize the H Block Symmetrically

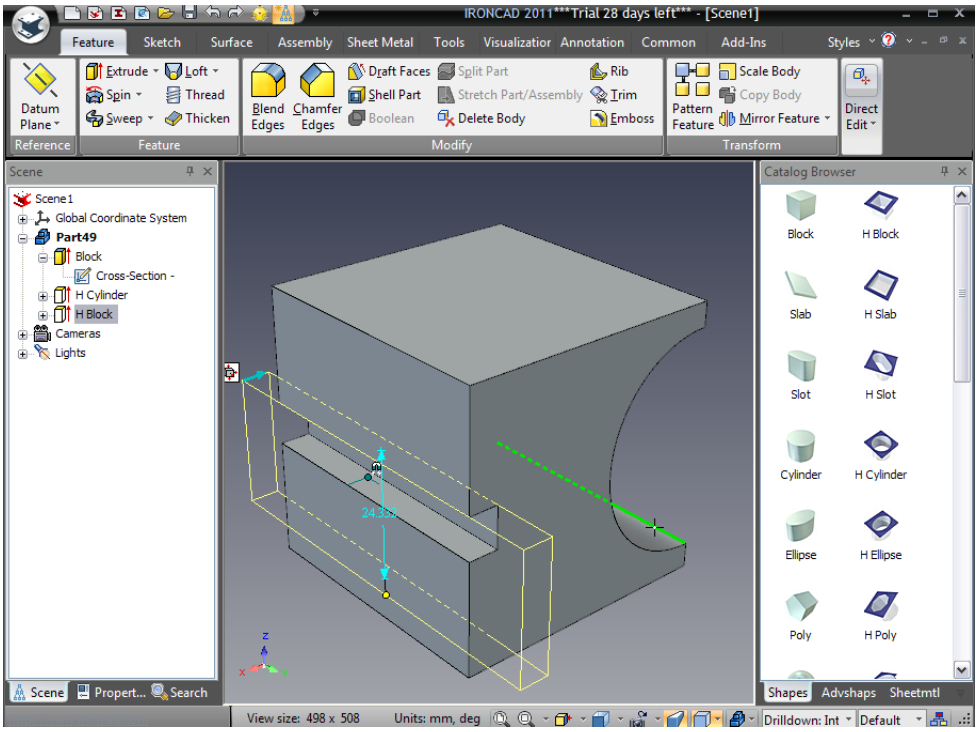
To stretch the block so that it cuts the sides of the Block in the same direction as the hole, select one of the handles (1) in the image below and then select its opposing handle (2) while pressing the CTRL key. IRONCAD will change both handles to yellow and dragging one will cause the opposite handle to move the same amount (or symmetrically). Drag the handle so that the H Block cuts right across the entire face.



Note: Try CTRL clicking multiple handles and notice the change can happen in multiple directions at the same time.

18. Using SmartSnap to Precisely Size to Geometry

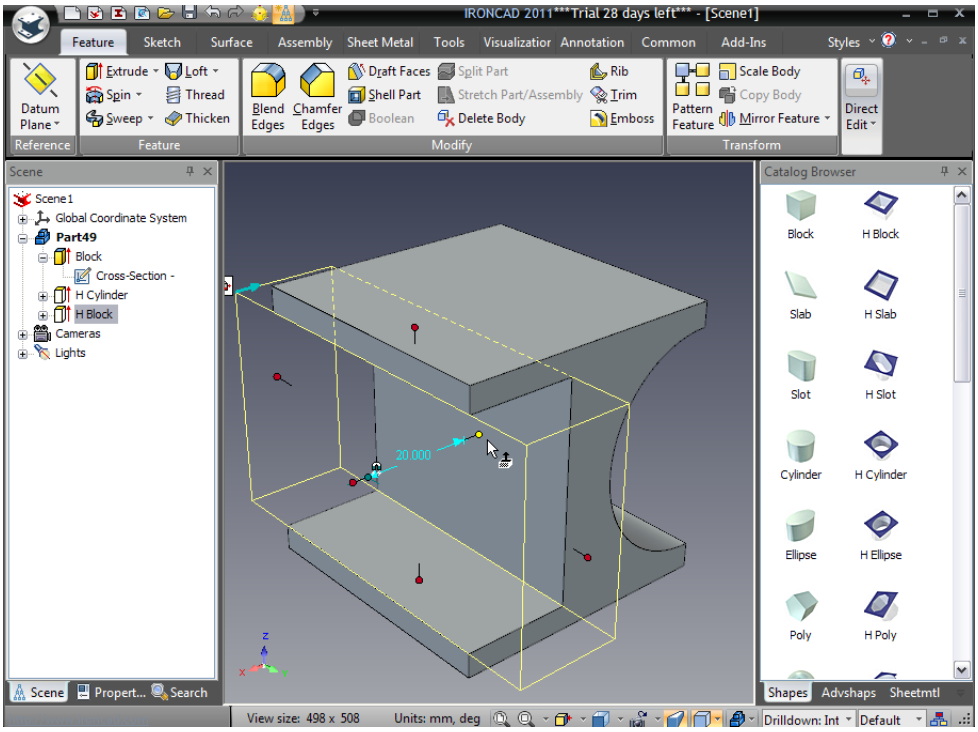
SmartSnap can also be used with IntelliShape handles to precisely size. Select one of the handles defining the height of the H Block and while pressing the shift key, drag the handle. However this time while the mouse button and shift key are pressed, move the mouse around the model. SmartSnap will snap to geometry elements around the model with the green highlights.



Using this technique make the H Block the same height as the diameter of the H Cylinder by matching both edges of the hole where it intersects the opposite face.

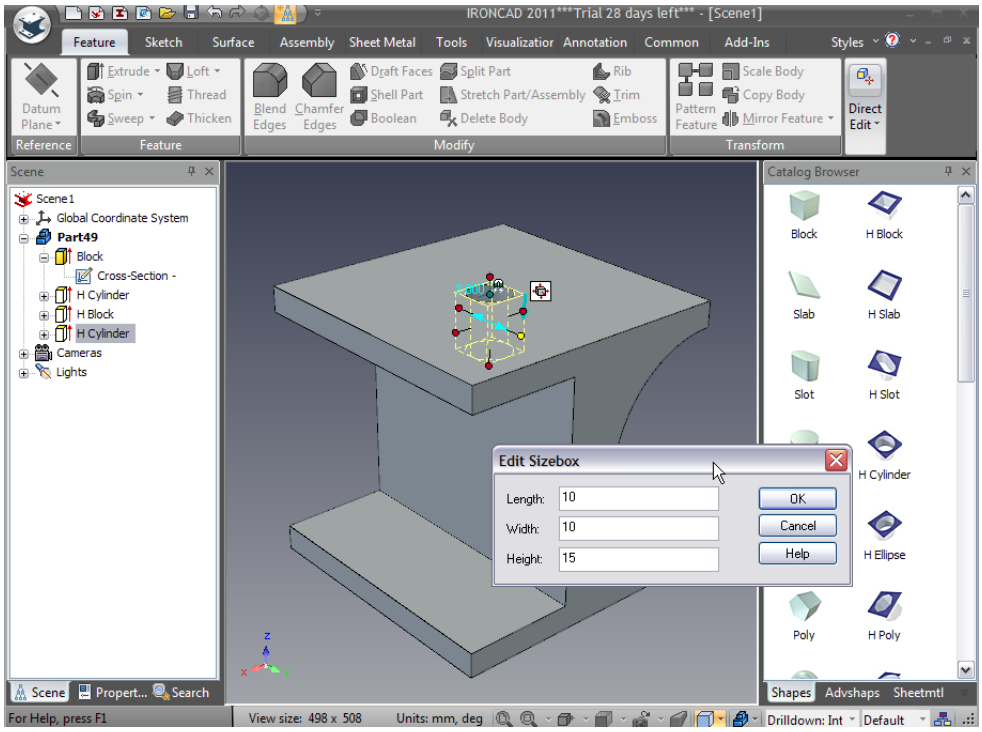
19. Set the Depth of the H Block

Now set the depth of cut for the H Block to be 20mm as show in the next image.



20. Add a hole to the Center of the Top Face

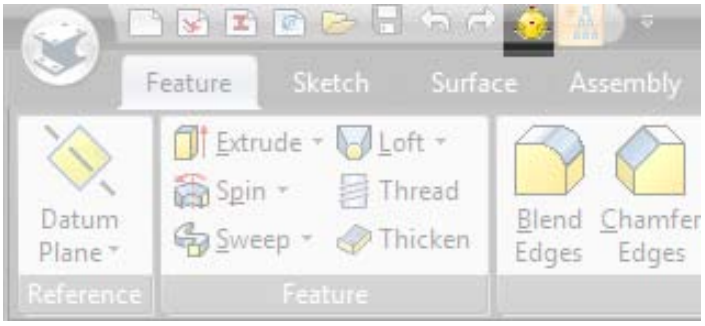
On the top face, shown in the next image, add a hole to the center of the face and set its diameter to 10mm and depth 15mm.



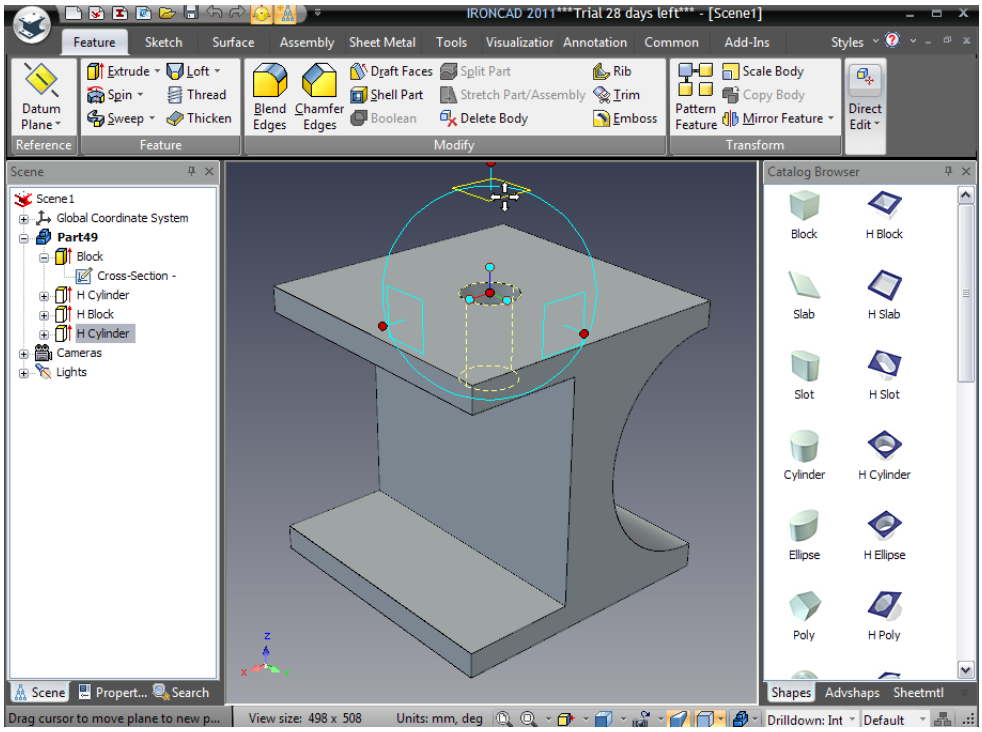
3.2.5. Using the TriBall

21. Reposition a Hole with the TriBall

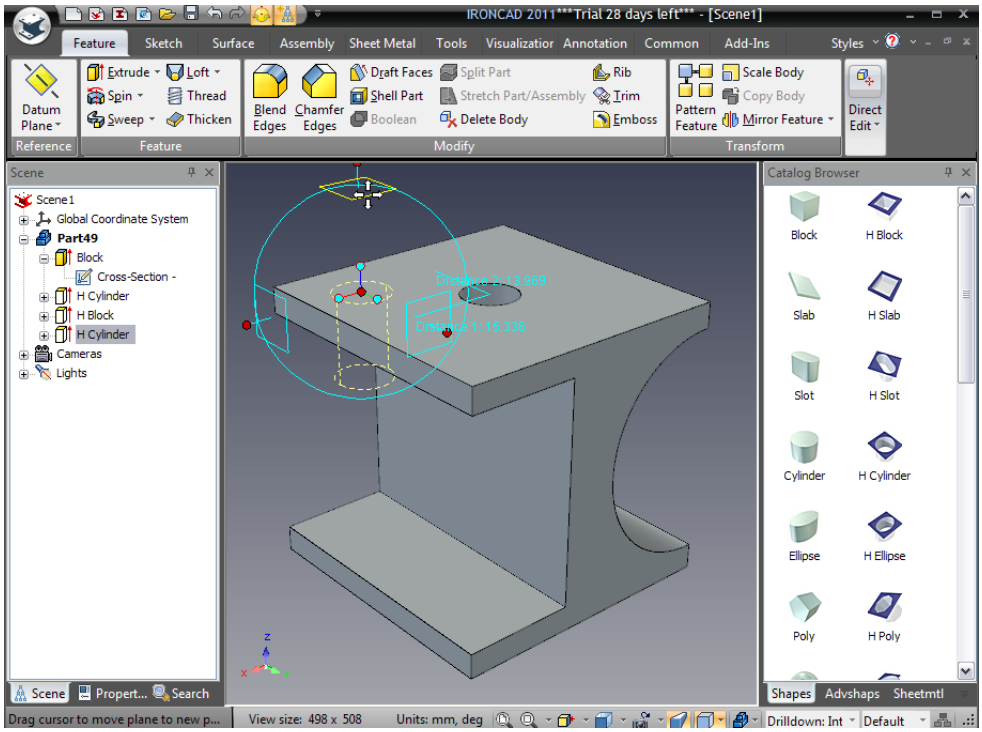
With the latest hole selected at the IntelliShape level activate the TriBall by clicking on its icon on the Quick Access toolbar:



IRONCAD will display the TriBall on the newly created hole. Move the mouse pointer over the Plane control (square handles). The display of the TriBall will change so that the plane icon changes to yellow and the mouse pointer changes to 4 arrows at 90 degrees to each other.



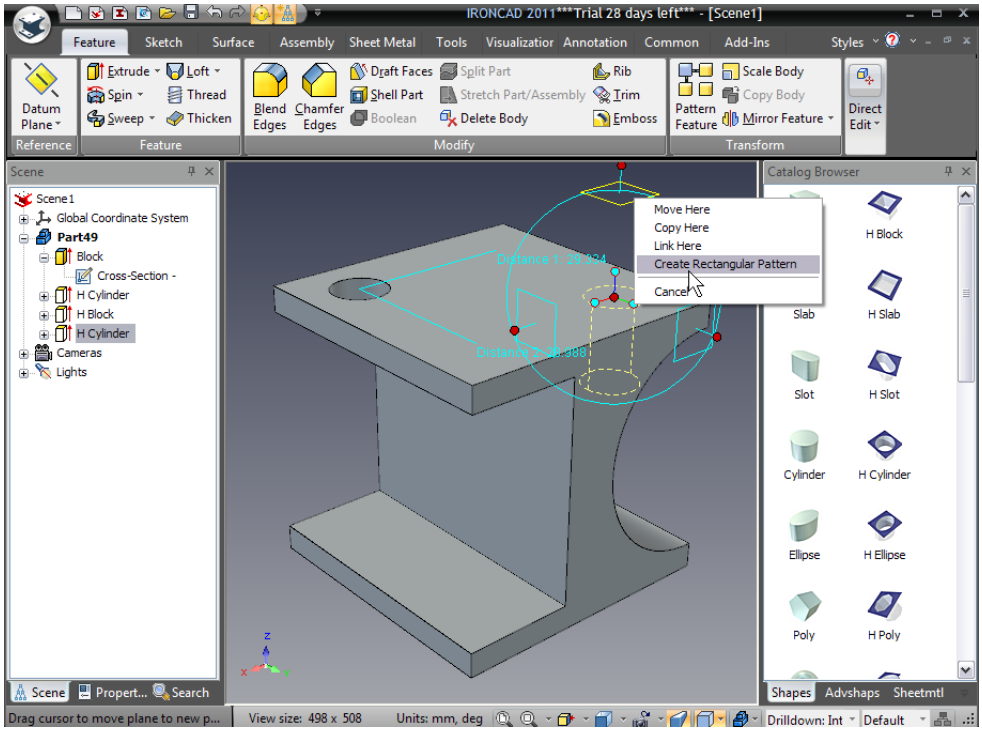
With the right hand mouse button pressed, move the hole as shown:



Releasing the mouse button will cause IRONCAD to display a control dialog box with options. Select Move Here and enter 15mm for both values.

22. Creating a Rectangular Pattern with the TriBall

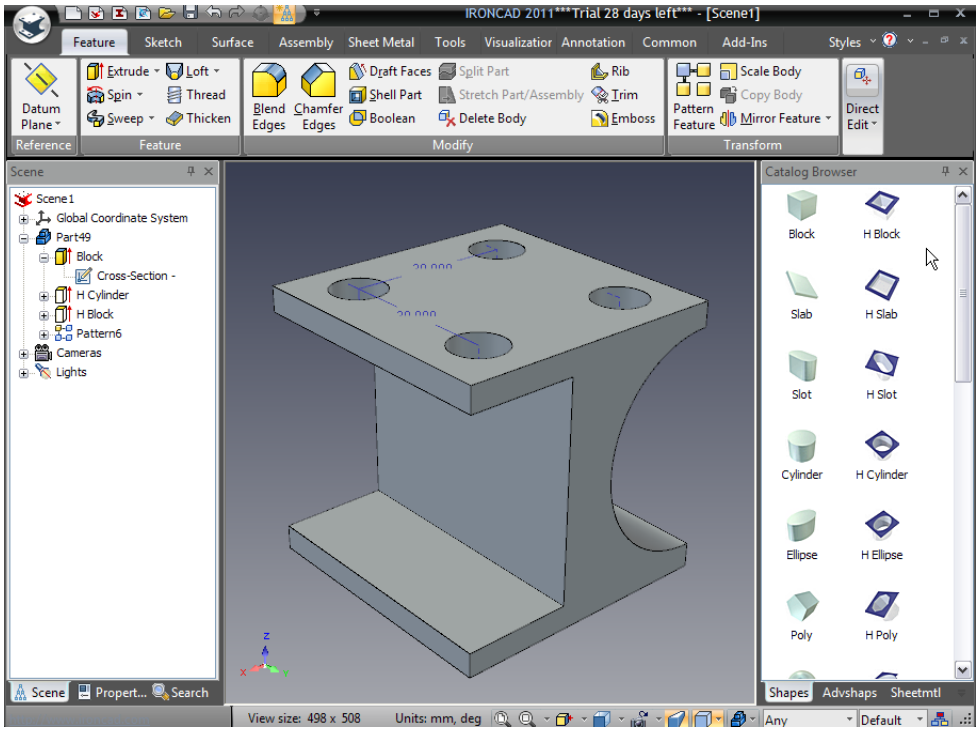
While the TriBall is selected, repeat the steps above and move the hole to the opposite corner of the face. However, instead of using the option Move Here, select Create Rectangular Pattern.



23. Finishing the Hole Pattern

Enter 2 for the both number entries and 30mm for both distance values to set the correct spacing on the top face.

You will now have a model of the IRONCAD I-Beam icon. More importantly you will have covered the core functions of IRONCAD giving a good foundation for modeling a wide range of parts using IRONCAD.



The key topics covered in this section were:

- Creating a 3D Scene document
- Drag and Drop conceptual modeling
- Resizing to precise dimensions
- SmartSnap
- TriBall

3.3. Communicating the Part

IRONCAD 3D parts can be communicated to the outside world in many ways. Some of the common ways in IRONCAD include:

- 2D detail drawings
- Photorealistic rendering
- Exporting using neutral 3D format models including:
  - ACIS, Parasolid, IGES, STEP and STL

This section will take the model created in the previous section and introduce the three primary options.

3.3.1. Creating a 2D drawing

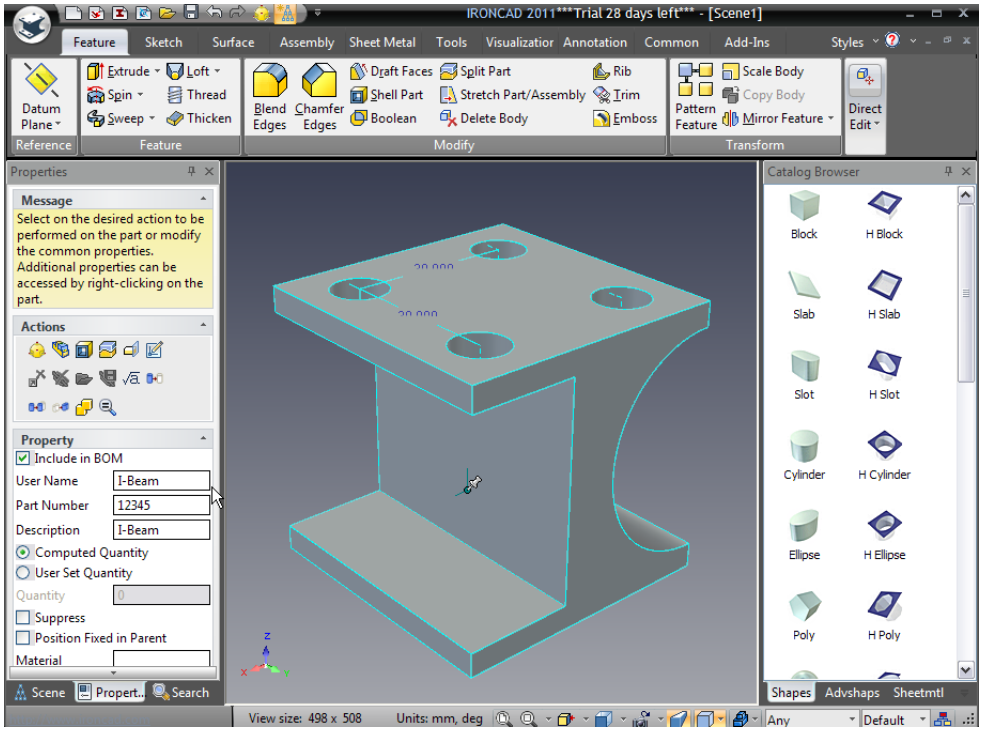
This section introduces the IRONCAD embedded drafting system used to create detail drawings from the 3D models created in the scene environment.

Users create Views which are standard views, sections and detail views that are essentially snapshots of the model. IRONCAD maintains links between the 3D model and the 2D drawing. Any changes to the model are automatically reflected in the detail drawing. In addition, IRONCAD possess the functionality to update the model from the drawing environment that will be covered by a later section of this guide.

1. Save the part

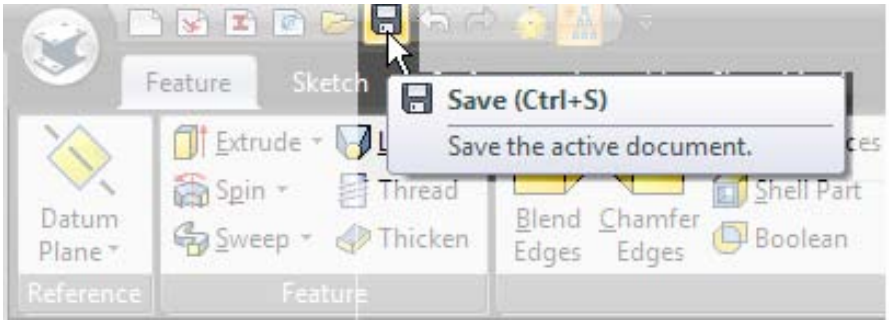
IRONCAD needs the name of the part in order to create the links between drawing and model. It's also a good idea to identify the part for future reference such as with its description and part number that can be used in annotation processes like the Bill of Material.

In the scene browser, right-click on the blue part icon and select the Properties tab on the bottom of the scene browser.



Enter I-Beam into the Description box and type 12345 in the Part Number box. You can optionally change the User Name to I-Beam as well.

Click the Save icon on the Quick Access Ribbon Toolbar and give the part a recognizable name (i.e. I-Beam).



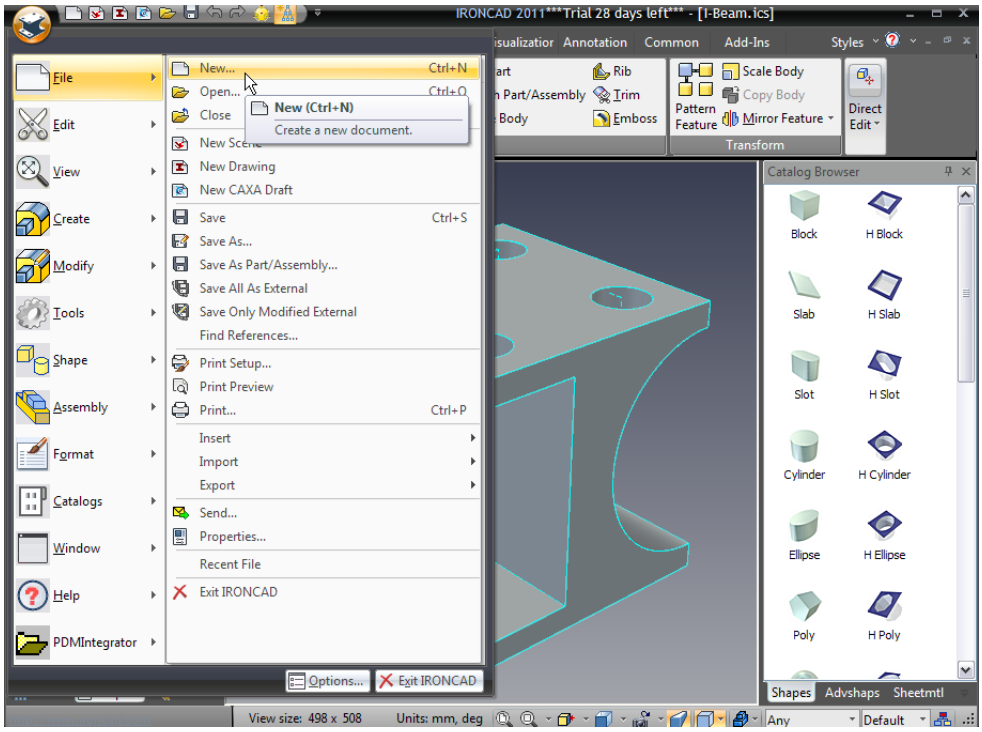
To help identify the part in the working window session, IRONCAD will add the part name to the main window title bar.

2. Default Save Location

IRONCAD will, by default, save the part to the IRONCAD folder created in My Documents (this can be changed in the Tools/Options/Directories).

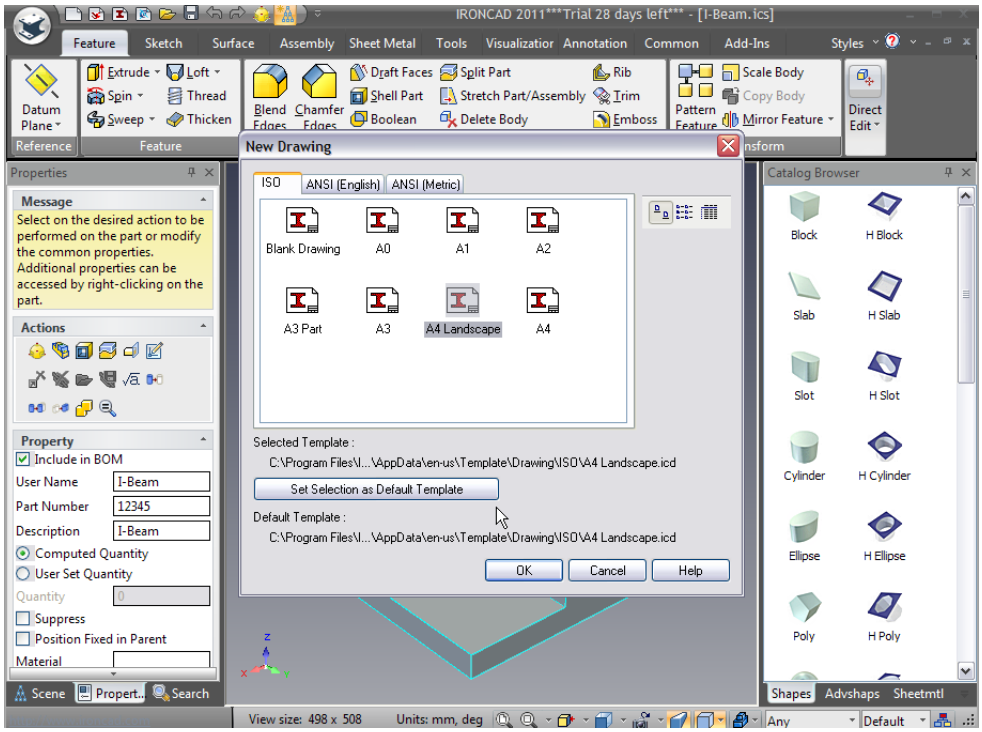
3. Create a New A4 Drawing

A. To create a new drawing, from the Windows Home button select: File>New.



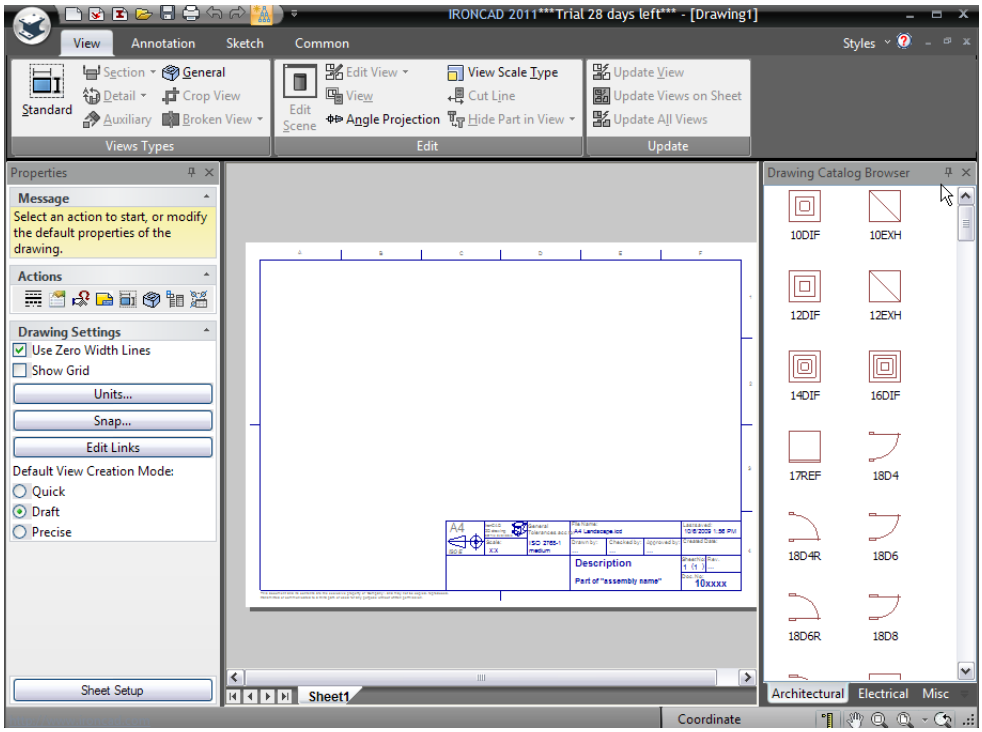
B. From the resulting dialog box, select drawing and then click OK.

C. From the ISO tab, select A4 Landscape.



As with the scene selection dialog box, the tab can be clicked to set this template as the default drawing template.

IRONCAD will now display a drawing sheet with the Properties browser on the left hand side and the catalog area on the right hand side.

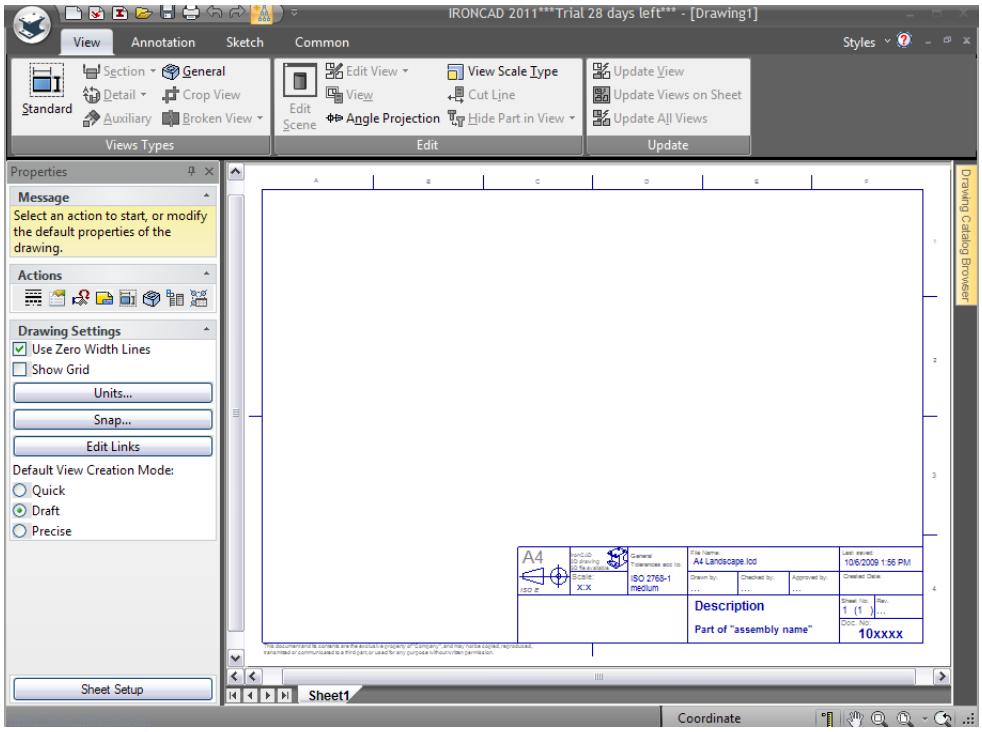


4. Hiding the Catalogs for Later Use

The 2D catalogs will not be required during this session and therefore switching them off will provide a greater screen area to work with. Click on the ‘push pin’ on the top right hand side of the catalog title bar. If you later need items from the catalog, ‘hover’ the mouse pointer over the orange tab on the right hand side of the screen, where the catalog once was to automatically display the catalog once more.

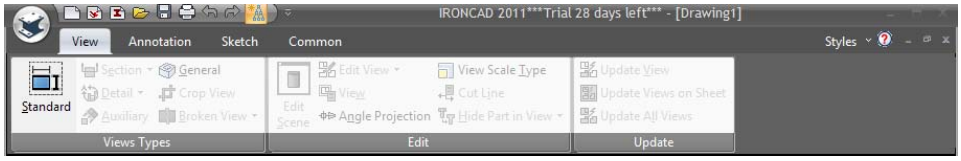
5. Fit the Contents to the Window

Press F8 to fit the contents of the drawing to the screen area (this hotkey is the same in the scene environment too).

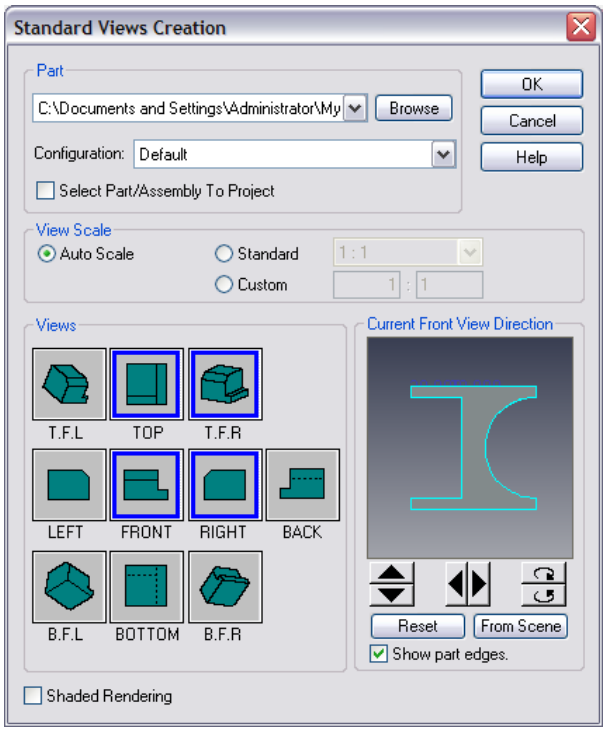


6. Accessing the Standard View Creation Dialog

From the Windows Ribbon bar, select the View tab if it not selected. Click on the Standard view icon.



IRONCAD will display a view creation dialog box. Use the arrows under the graphic image of the part to alter the position of the view so that the round cut is on the right hand side as shown in the next image.



7. Select the Desired Views

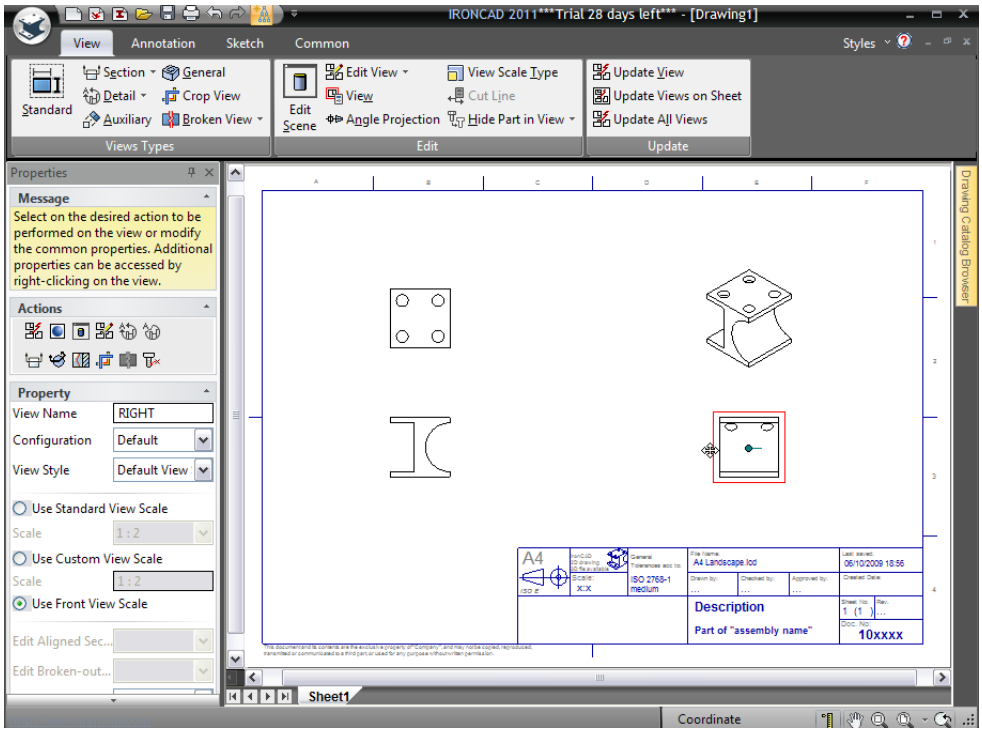
Click the views required for the drawing. In the previous operation, the image defines the front view which can be considered as the 'master' layout view. For this example, select the front, right, top and TFR icons (TFR generates an isometric view) and click OK.

Note: if you struggle to get an orthographic front view, you can use the 3D to achieve the orientation. Currently, the original model has been created at a non-orthogonal orientation in the parent scene. Return to the scene by minimizing the current window. In the scene view, select the 'Look At' icon or press F7 and point on one of the faces of the part. IRONCAD will orientate the view orthographically. Return to the IRONCAD drawing and in the view creation dialog select the From Scene button to access this orientation.

8. Place the Standard Views

After clicking ok to the previous dialog, IRONCAD will clear the view creation dialog box and display the views as a group of red bounding

boxes. Position the group by moving the mouse and click the left mouse button to place the views. To reposition a view, click on the view to activate a red bounding box for that view. Move the cursor over one of the border lines and the cursor will change to 4 arrows. Click on the border to drag and reposition the view.

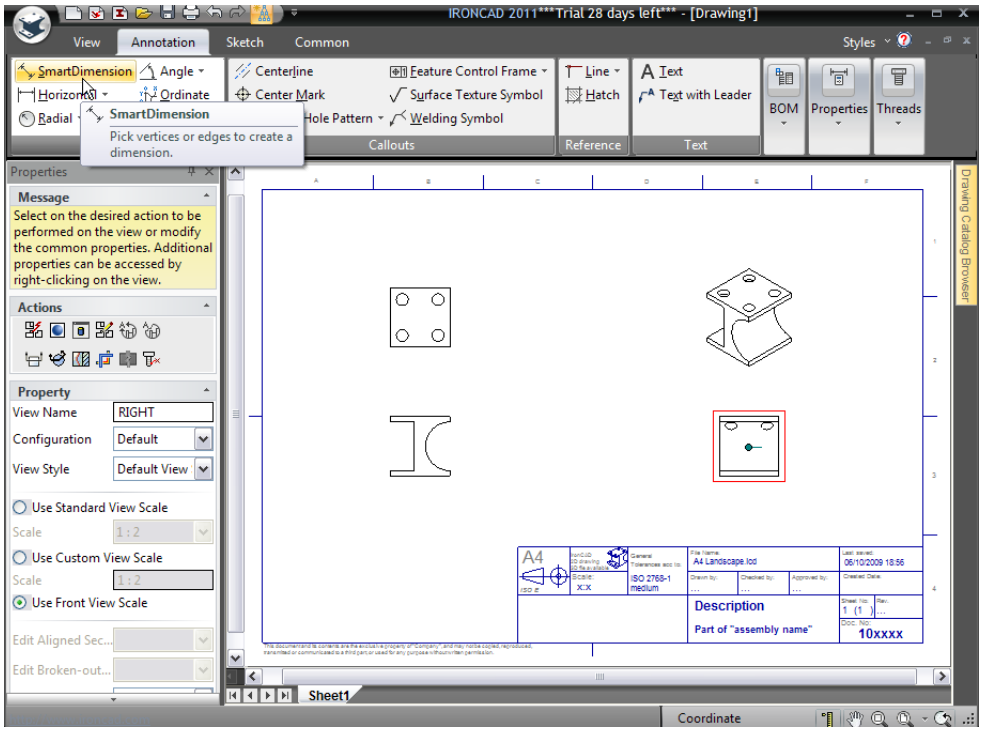


The repositioned view will maintain its alignment with the master view (Front View). The alignment can be broken if required by right-clicking on the view and selecting Break Alignment under the View Alignment options.

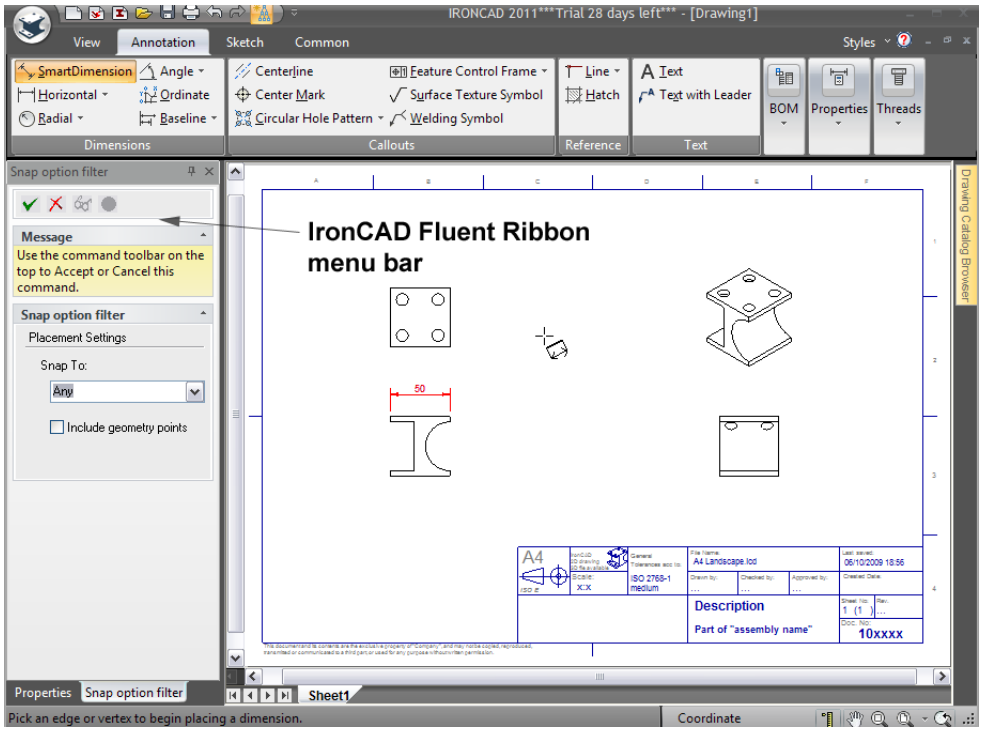
Selecting the Front view (the ‘master’) and moving it will also move the other views related to it.

9. Adding Annotations

Adding dimensions is easy. Select the Annotation tab and click on the SmartDimension icon.



In the Top View, click on the top most line of the view. IRONCAD will display a dimension. The dimension will move along with the cursor. To position the dimension, move it to the appropriate position and click the left mouse button. IRONCAD will stay in the selected dimension mode until the Apply and exit button is selected (the tick icon at the top of the IRONCAD Property Panel menu bar on the left hand side of the screen) or you select ESC.



10. To Dimension Hole Positions

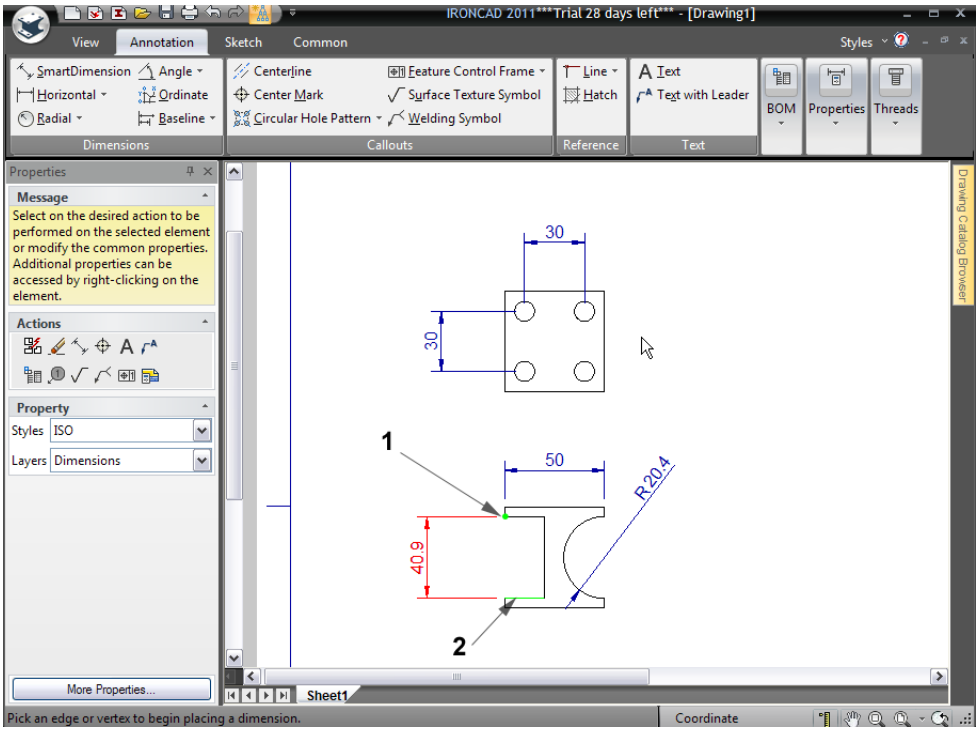
If SmartDimension isn't selected, click on the menu option. Position the cursor over one of the circles and with the Shift key pressed, click the left mouse button. IRONCAD will use the center of the circle as the first dimension location. Now select the vertical side of the block. IRONCAD will create the dimension and orientate it normal to the selected edge.

11. To Dimension the Radius

Using the SmartDimension, simply select the arc of the cut in the Front View. IRONCAD will automatically create a radial dimension.

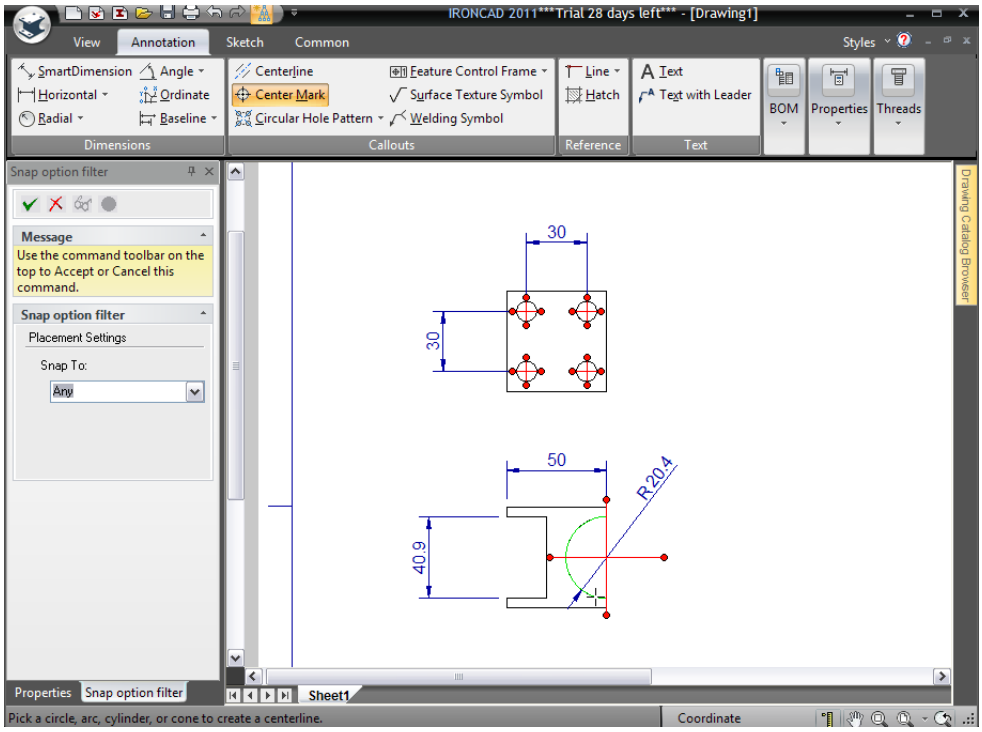
12. To Dimension the Square Cut

First select one of the outer corners of the cut (1) followed by the opposite side line (2) noting that the line will determine the direction of the dimension.



13. Adding Centerlines

To finish the drawing, add center lines using the Center Mark tool. Select the tool from the Annotation tab of the Windows Ribbon bar and then click on each arc/circle.



Complete the command by clicking on the apply 'tick' icon on the IRONCAD Property Panel.

14. Changing the Appearance of Dimensions

To change the appearance of a dimension, right-click on the dimension itself and select the appropriate modifier. For Example: On the Radial Dimension change:

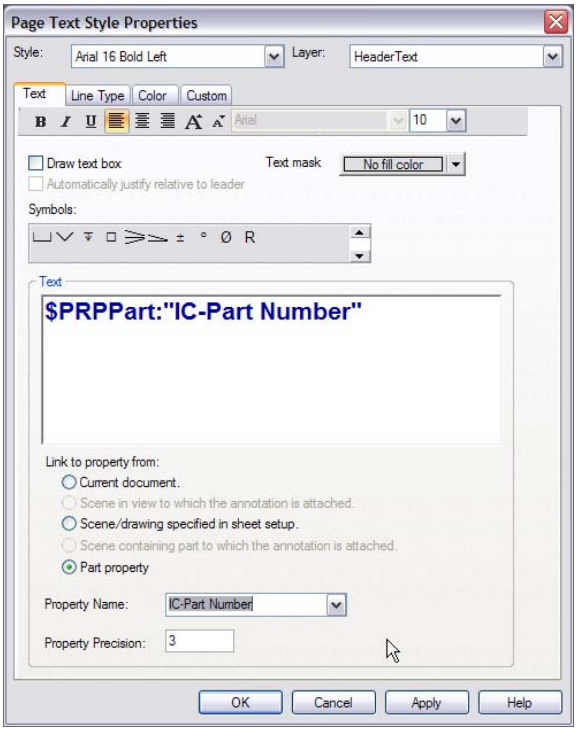
- It's format to On/Horizontal
- Flip it's arrow direction by clicking on Flip Inside/Outside Arrow
- And it's precision to 3 decimal places

15. Editing Title Block Text

Change the name in the title block by right-clicking on the existing text and selecting Edit Text. Note: By default, some templates place the title block text on a locked layer. To unlock the text, select on the Common tab of the Windows Ribbon bar and select Styles and Layers under the Styles group. In the bottom right side of the dialog, uncheck the locked on the template (or template text) layer.

16. Tying Text to Part Properties

To change the part number to read from the data specified in the scene, right-click on the part number text in the title block and select Properties. In the resulting properties panel, click the Part property radio button and in the Property Name drop list select IC_Part Number.



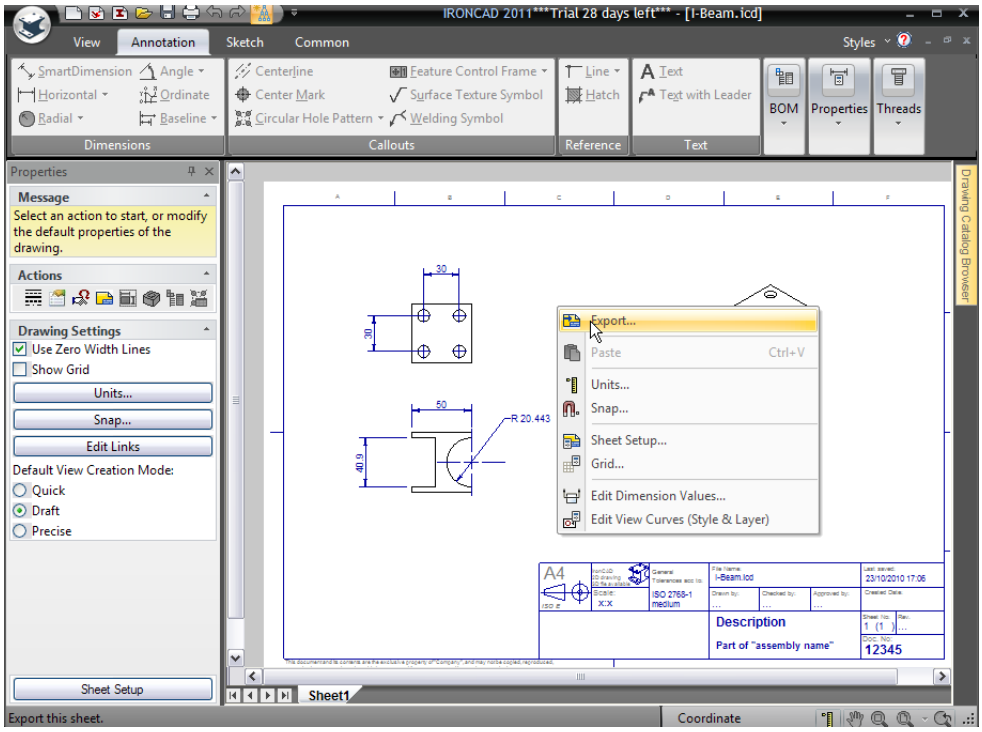
IRONCAD will replace the number provided on the original sheet and replace it by getting information from the model contained in the drawing.

17. Saving the Drawing

Save the drawing in the same way the scene was saved.

18. Exporting the Drawing

To export the drawing, right-click in any free part of the drawing and select Export.



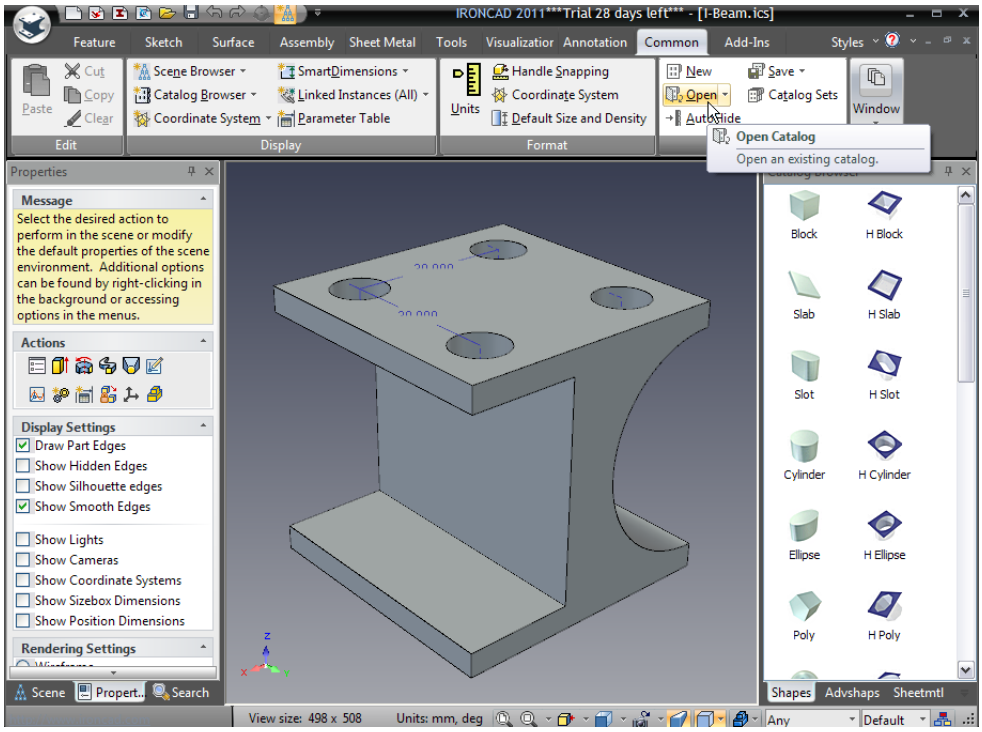
From the resulting menu, select a format such as PDF. Setting the options such as quality and resolution as might be required.

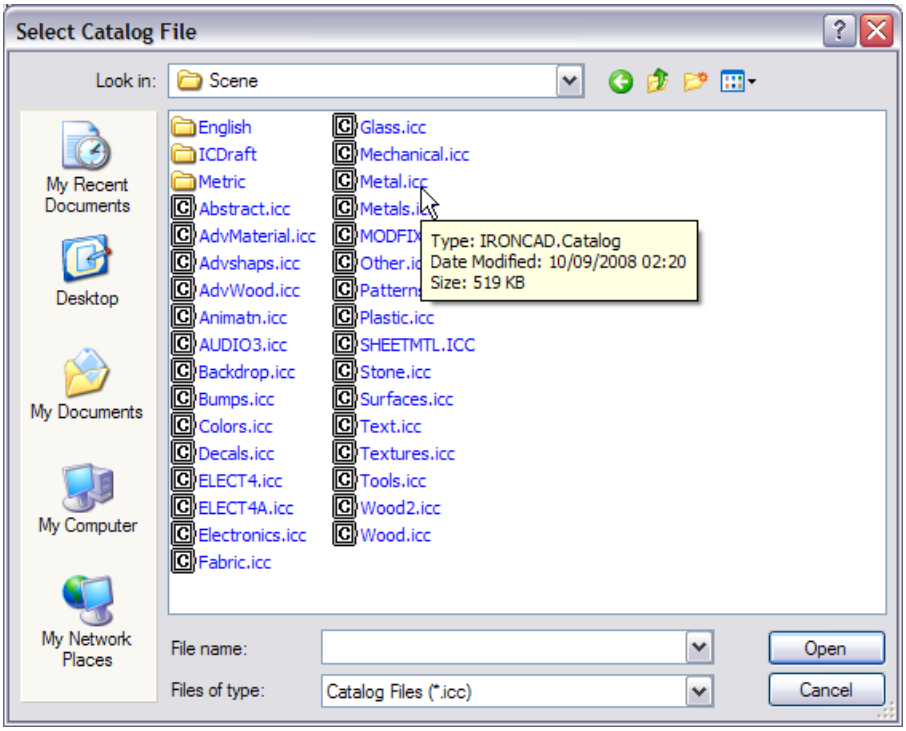
3.3.2. Creating a Photo-Realistic Rendering

IRONCAD has the tools available to generate superb rendered images. Textures and finishes can be applied to the part using the same drag and drop functionality used to build the part.

1. Open Rendering Catalogs

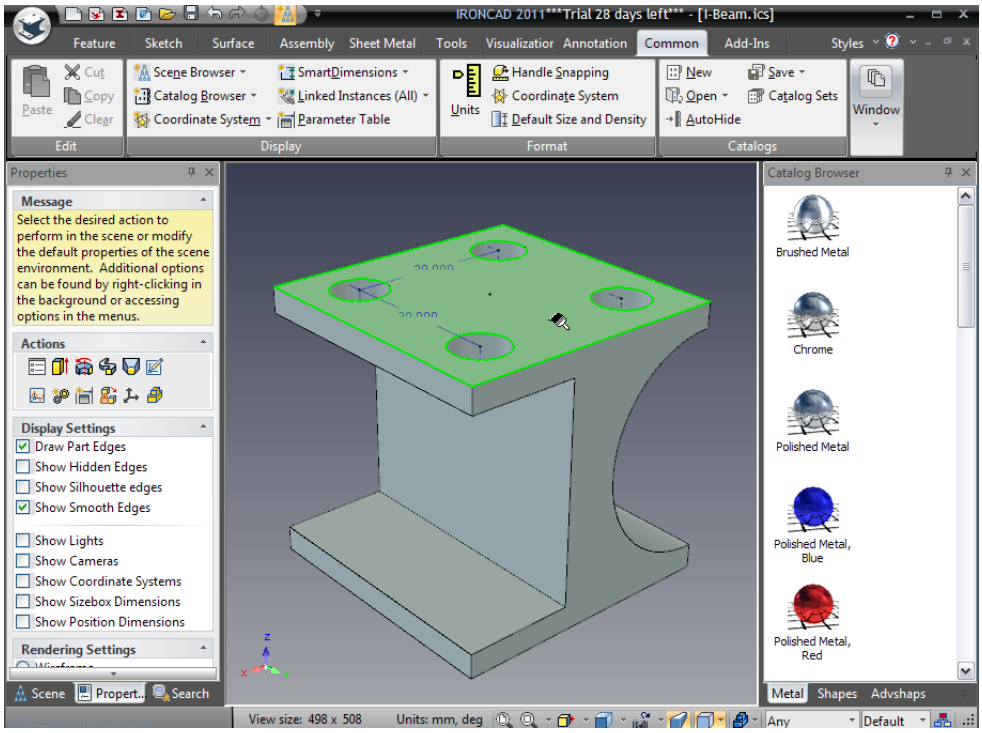
Return to the IRONCAD 3D scene by closing the drawing and open the Metal catalog by selecting to the Common tab and selecting Open. Select the Scene folder and select Metal.ics (not Metals.icc).


IRONCAD 2011***Trial 28 days left*** - [I-Beam.ics]
Feature
Sketch
Surface
Assembly
Sheet Metal
Tools
Visualizator
Annotation
Common
Add-Ins
Styles
Cut
Copy
Clear
Paste
Edit
Scene Browser
Catalog Browser
Coordinate System
SmartDimensions
Linked Instances (All)
Parameter Table
Display
Units
Handle Snapping
Coordinate System
Default Size and Density
Format
New
Save
Open
Catalog Sets
Autoslide
Window
Open Catalog
Open an existing catalog.
Properties
Message
Select the desired action to perform in the scene or modify the default properties of the scene environment. Additional options can be found by right-clicking in the background or accessing options in the menus.
Actions
Display Settings
Draw Part Edges
Show Hidden Edges
Show Silhouette edges
Show Smooth Edges
Show Lights
Show Cameras
Show Coordinate Systems
Show Sizebox Dimensions
Show Position Dimensions
Rendering Settings
Scene
Propert..
Search
Block
H Block
Slab
H Slab
Slot
H Slot
Cylinder
H Cylinder
Ellipse
H Ellipse
Poly
H Poly
Shapes
Advshaps
Sheetmtl
View size: 498 x 508
Units: mm, deg
Any
Default


Select Catalog File
Look in:
Scene
My Recent Documents
Desktop
My Documents
My Computer
My Network Places
English
ICDraft
Metric
Abstract.icc
AdvMaterial.icc
Advshaps.icc
AdvWood.icc
Animatn.icc
AUDIO3.icc
Backdrop.icc
Bumps.icc
Colors.icc
Decals.icc
ELECT4.icc
ELECT4A.icc
Electronics.icc
Fabric.icc
Glass.icc
Mechanical.icc
Metal.icc
Metals.icc
MODFIX
Other.icc
Pattern
Plastic.icc
SHEETMTL.ICC
Stone.icc
Surfaces.icc
Text.icc
Textures.icc
Tools.icc
Wood2.icc
Wood.icc
Type: IRONCAD.Catalog
Date Modified: 10/09/2008 02:20
Size: 519 KB
File name:
Files of type:
Catalog Files (*.icc)
Open
Cancel

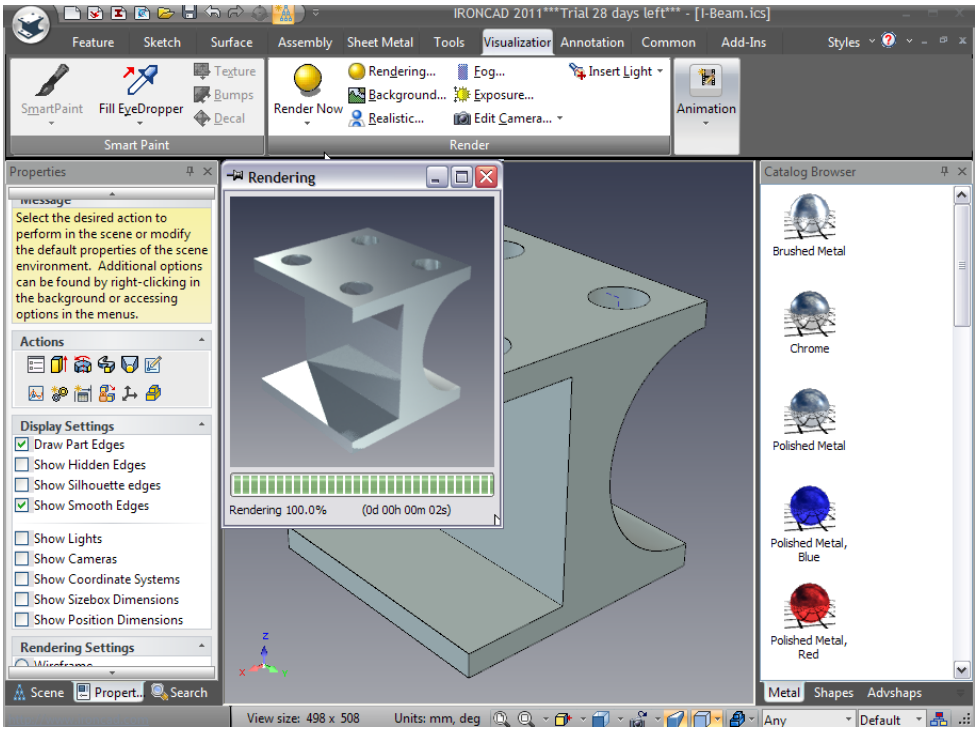
2. Applying a Material to a Part

Drag Brushed Metal from the catalog onto the part. Rotate the view so that its orientation looks like that of the Windows home button.



3. Create a Rendering Preview

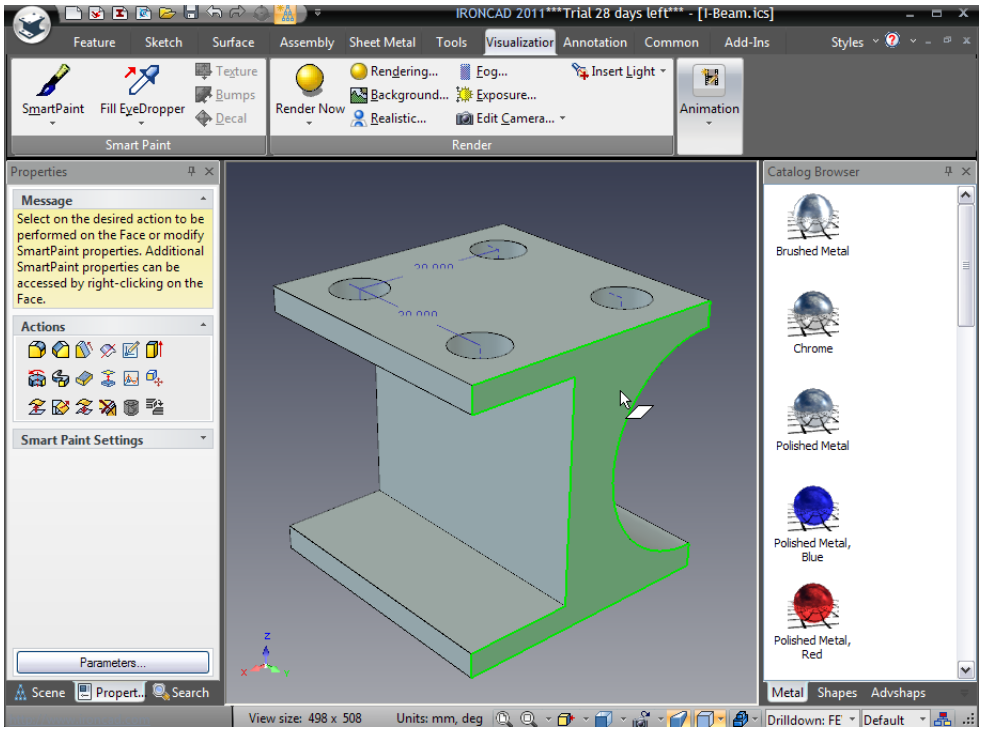
Open the Visualization tab on the Windows Ribbon bar and select Render Now. IRONCAD will produce a small preview image of the I-Beam with a brushed metal texture.



Close this window when IRONCAD has finished rendering.

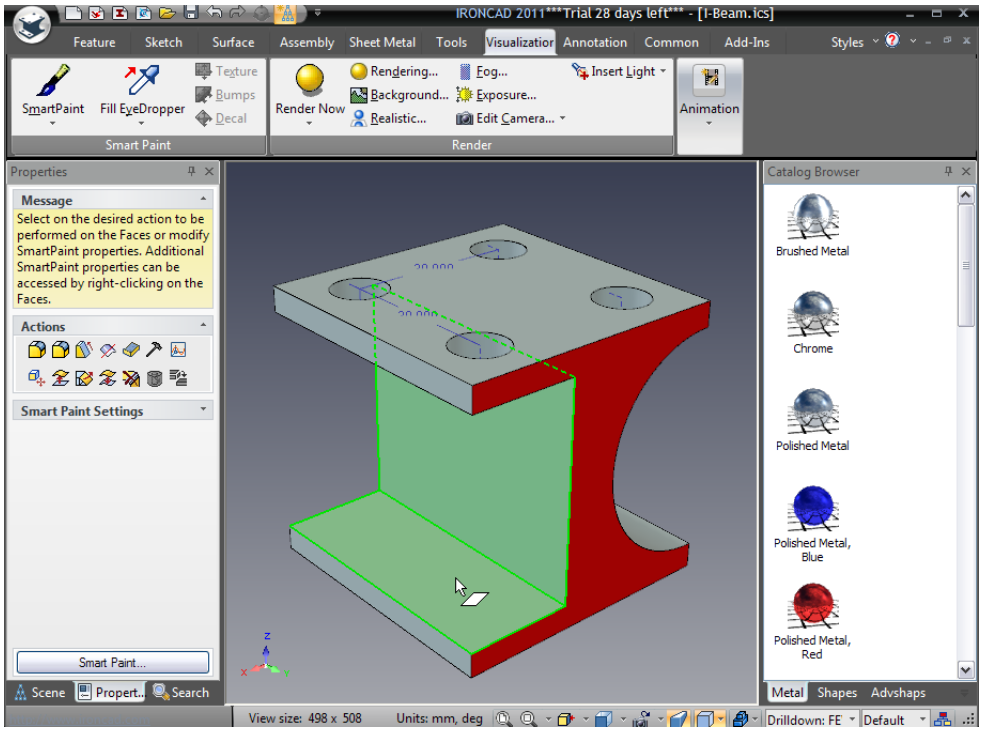
4. Changing the Material on Select Faces/Surface

To add a little more impact, certain faces need a different color and/or finish. To change the finish on an individual surface, click on the required part at the surface/face 3 times (Part, IntelliShape, Face/Edge/Surface Level).



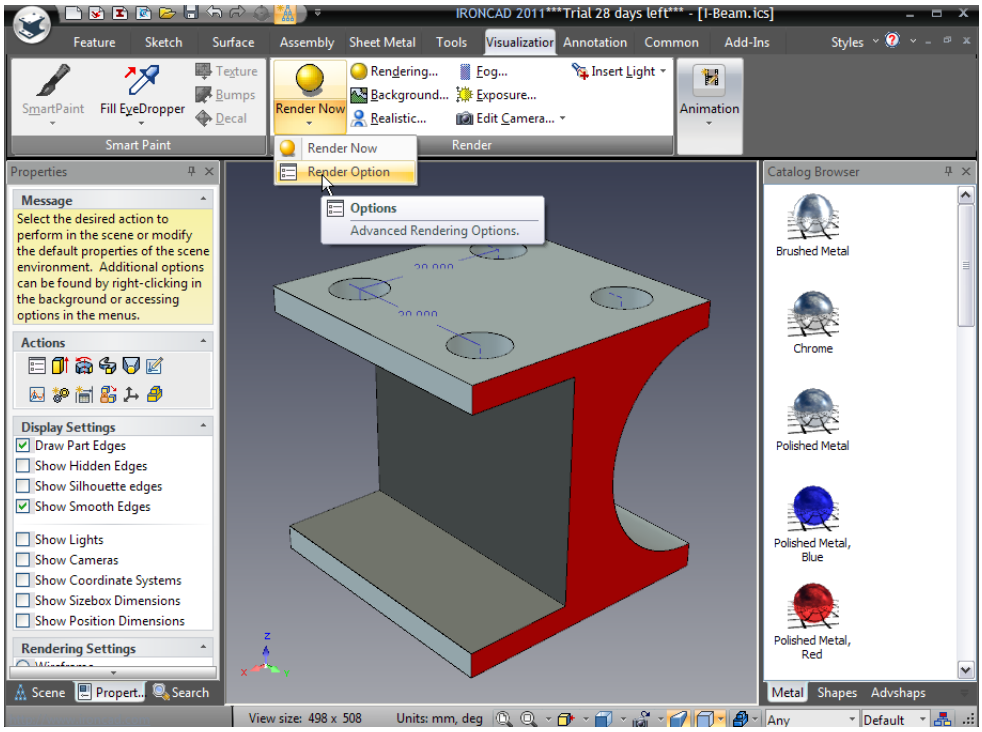
The surface will be highlighted with a bright green boundary and a green color to the surface. Drag the Polished Metal, Red onto this surface.

Next, drag the Chrome material to the visible vertical surface of the square cut and the horizontal surface of the square cut.

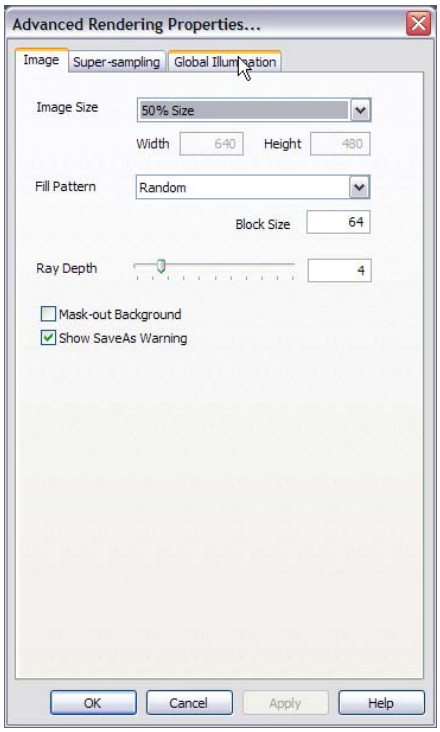


5. Access the Advance Rendering Properties

Click on the drop down arrow on the Render Now icon to open up the Advanced Rendering Properties dialog box.



IRONCAD will display the Advanced Rendering properties panel.



6. **Change the Advance Rendering Settings**

On the Image tab:

- Change the Image Size to full Window using the drop down, and
- Set the Fill Pattern to Spiral using the drop down options

On the Super-sampling tab:

- Change the Quality setting to High (this provides high definition to the edges)

On the Global Illumination tab:

- Set the Type to Background Illumination and Quality to High

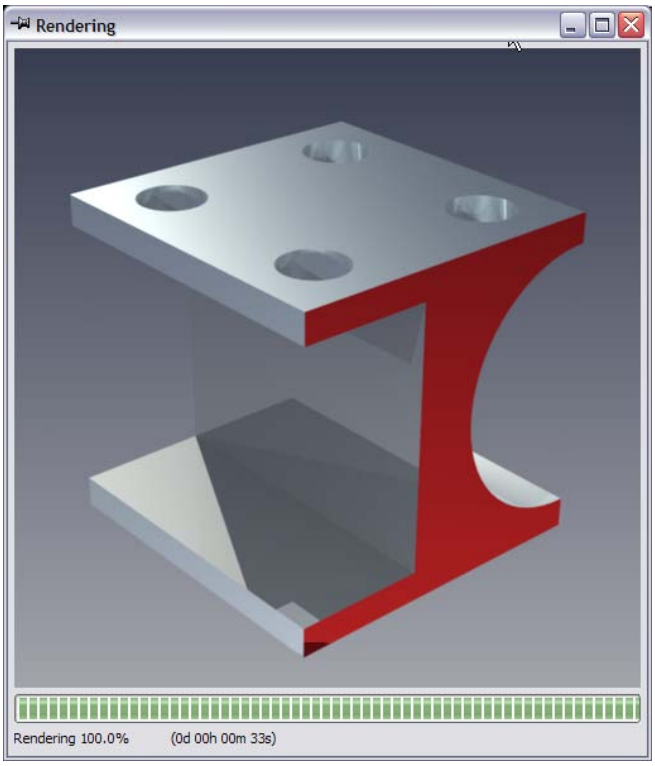
Click OK

7. **Set the Scene to Use Perspective**

Make sure the scene is displayed with perspective applied on the bottom right tool bar. Realistic rendering perform better using this setting since this is more realistic. Orthographic renderings may take a longer to render depending on the setting.

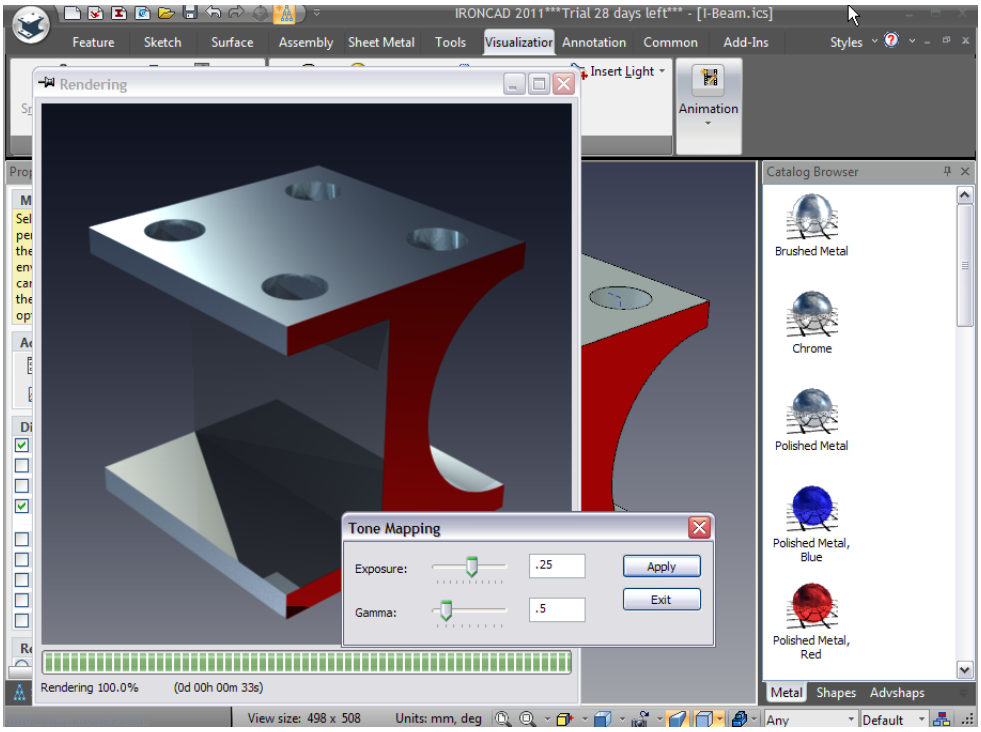
8. **Render with the New Advance Setting**

On the Render Options icon drop down select Render Now. IRONCAD will proceed to produce a photo realistic image of the I-Beam.



9. Adjusting the Final Rendering and Saving

When the rendering has finished, right-click in the render view and select Adjust Tone Mapping to adjust the settings to alter the rendered image appearance. For example: set Exposure to 0.25000 and Gamma to 0.5000 and hit the Apply button.



IRONCAD will immediately change the rendered view. When you are satisfied with the settings, click Exit. Then right-click the view and select Save As. Give the image your own name or just hit Save. The 'type' drop down allows you to change the file type of the image if you do not like the default bmp setting. Also, select the option button to adjust the quality for various exported types.

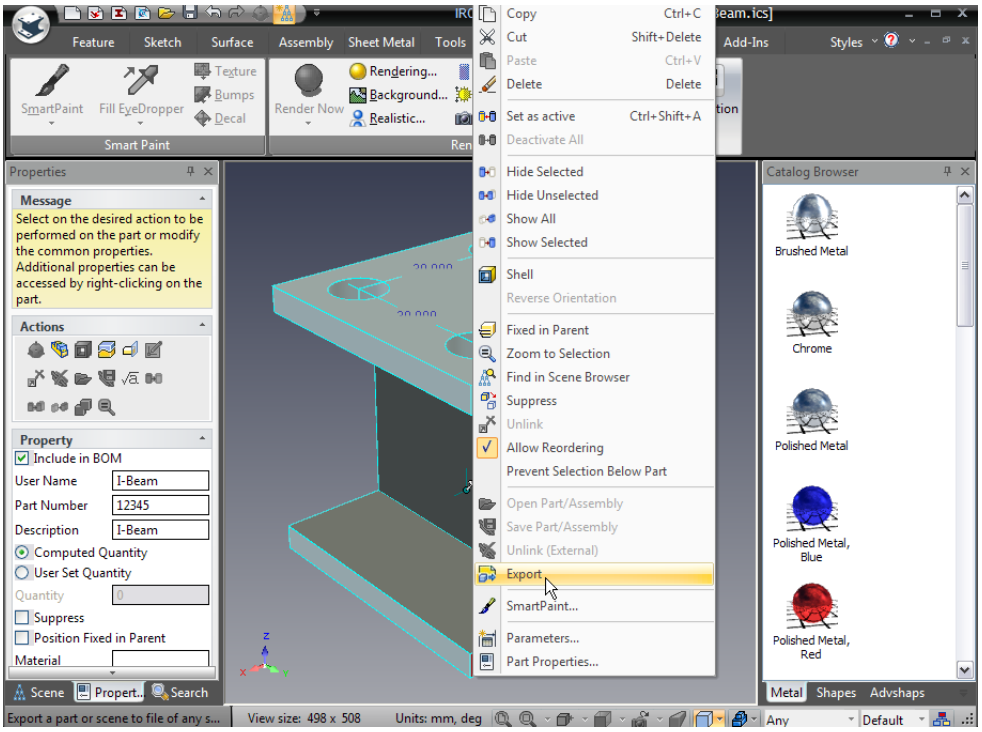
3.3.3. Exporting the 3D model

One of the real benefits of creating a 3D model is to be able to let others share your idea and to capitalize on a wide range of manufacturing techniques such as rapid prototyping. To export the model in any of the popular 3D formats, such as stl:

1. **Select the part (edges highlighted in cyan)**

2. **Select Export**

   Right-click in the scene and select Export from the drop down menu.

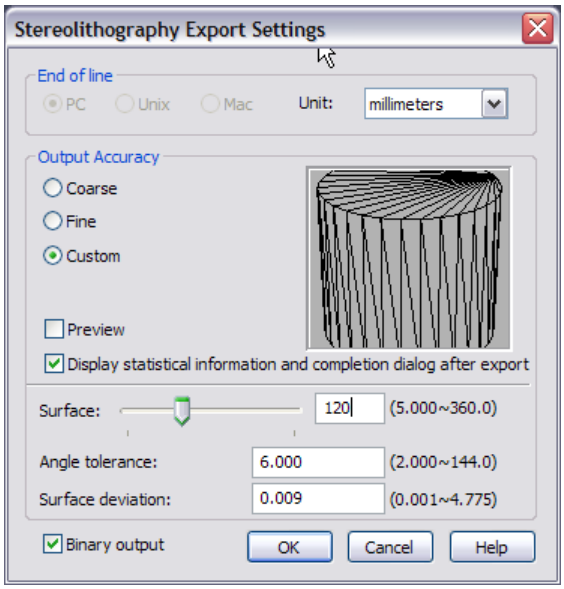


3. Select the Desired Type and Location

From the drop down, select Stereolithography (*.stl) and enter a name for the file to be exported. Set the desired location for the exported part.

4. Select the Desired Options

Select the Custom radio button and set the Binary output check box, as the vast majority of systems accept binary data and it results in smaller file sizes. Change the number of surface facets to a value recommended by the stereolithography process to be employed say 120 (3 degree facets on a cylinder). Hit OK.



This completes the basics of communicating your parts. The key topics covered in this section were:

- Creating 2D Drawing Layouts
- Rendering Parts
- Exporting Parts to Standard Format.

Later in this learning guide, we will go deeper into each of these sections to explore additional aspects of each capability.

3.4. Working with Assemblies to Complete the Design

IRONCAD's unique 3D Scene environment makes it seamless to work between part and assembly modes. This section will cover the following topics that will lead to the final design and drawing of our 3D model:

- Designing an Assembly
- Working With Boolean Operations
- Using Intelligent Feature
- Using Intelligent Features as a Basis for New Features
- Creating Sketch Based Features
- Working with Blend/Chamfers
- Positioning with Constraints
- Adding Draft
- Handling Errors In Feature Creation
- Creating Configurations of Different States

- Creating Exploded Assembly Drawings

This section will take the model created in the previous section introduce these primary sections to result in a completed design of our model.

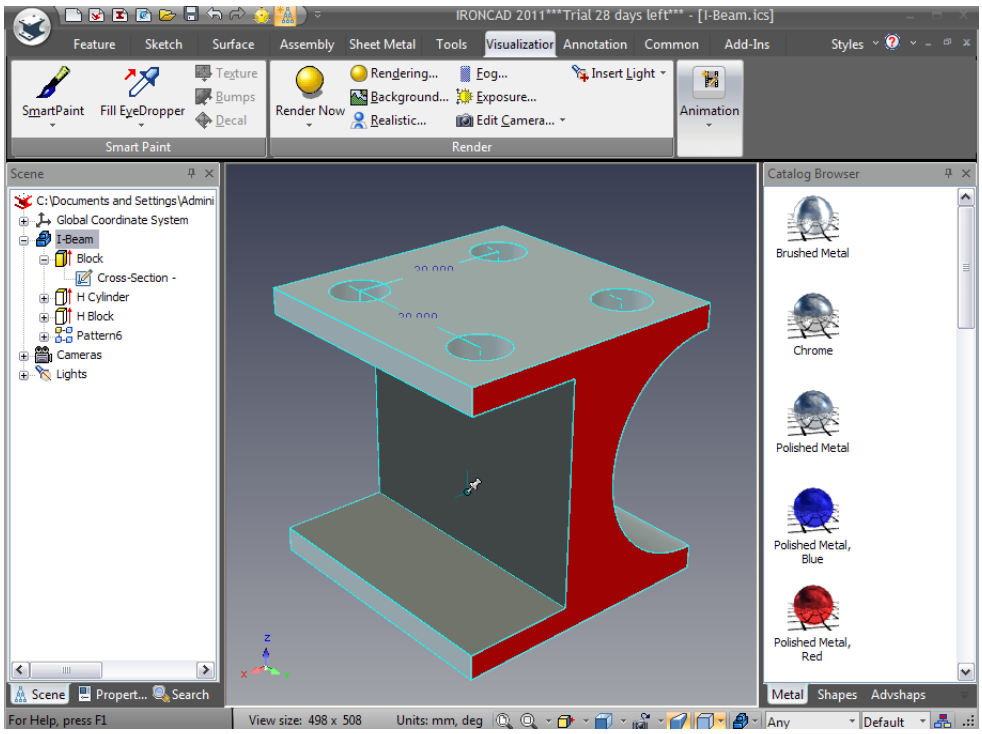
3.4.1. Designing an Assembly

IRONCAD provides true Top Down assembly modeling which is arguably the most natural way to design anything. Top Down modeling is when all the components making up a design are designed in conjunction with one another. In this section, the I-Beam model will be used to design a simple injection tool (note: This is not an example of injection tool design).

The first part of this exercise is to define the tooling plates.

1. Start with the Previous Model and Default Scene

In the IRONCAD Property Panel ensure the Scene Browser tab is active so the screen looks something like the image below.



2. **Copy the I-Beam**

   Right-click on the I-Beam model and select Copy from the drop down menu

3. **Paste the I-Beam and Repeat**

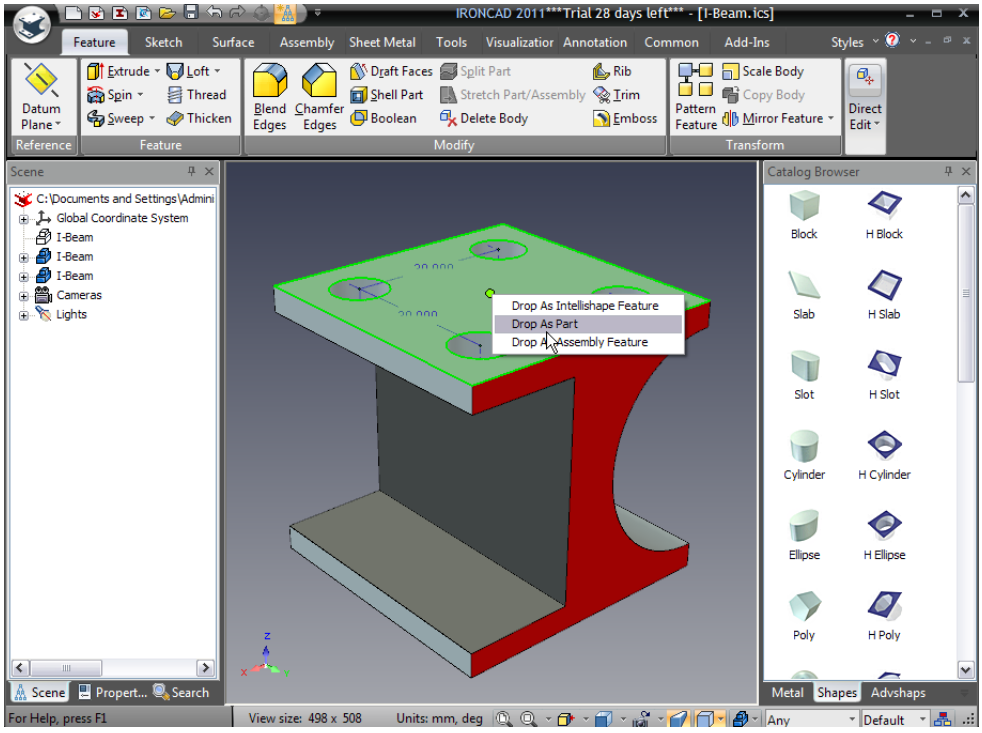
   Right-click again and select Paste. A second I-Beam part will be shown in the Scene Browser. Repeat the last operation so that the Scene Browser shows 3 I-Beam parts.

4. **Using Suppress to Hide**

   In the Scene Browser, right-click on the first I-Beam part and select Suppress from the drop down menu. The blue icon will be replaced by a white wire frame icon.

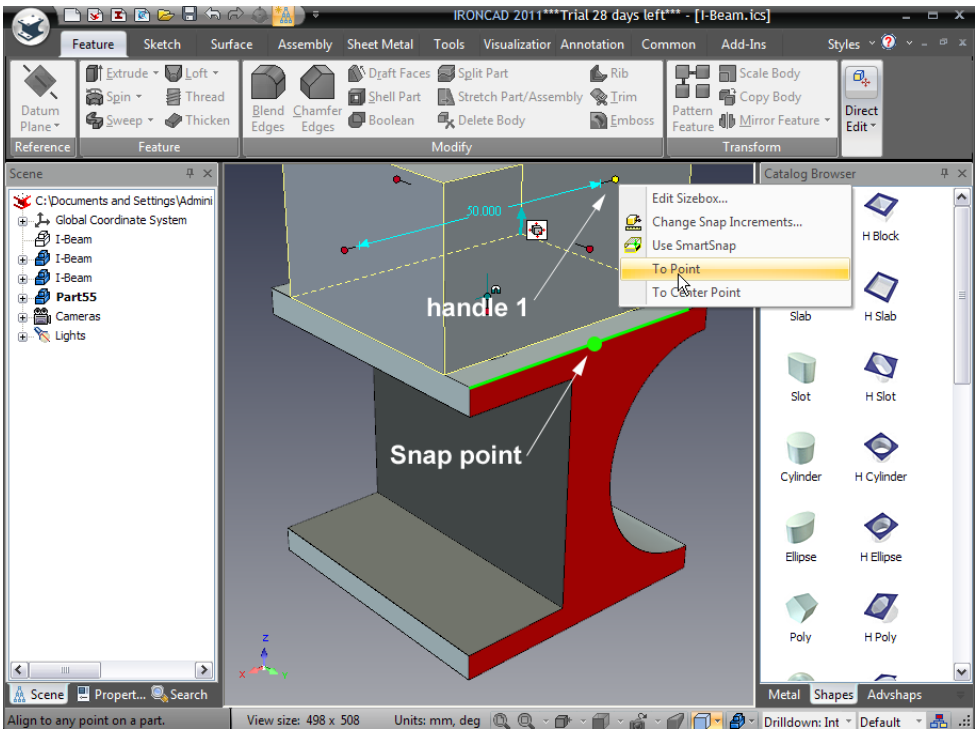
5. **Drag & Drop as a Part from the Catalog**

   From the catalog tabs, select the Shapes catalog. Using the right mouse button, click on the Block icon and drop it onto the center of the top face of the displayed model (remembering to use the SmartSnap feedback to ensure the center point of the face is used to drop the new part on). IRONCAD will display a drop down menu allowing the option to drop as a Part or IntelliShape. Select the Part option.



6. Using Precise Handle Snapping Options

Click on the new part to activate IntelliShape mode. Referring to the image below, right-click on the Sizebox handle (1) and select 'To Point' from the drop down menu.



Sweep the cursor over the edge shown. IRONCAD will highlight the edge in green and when the center point of the line is reached it will be highlighted with a green dot, as shown in the image above.

7. Resize To Midpoint Using Handle Placement

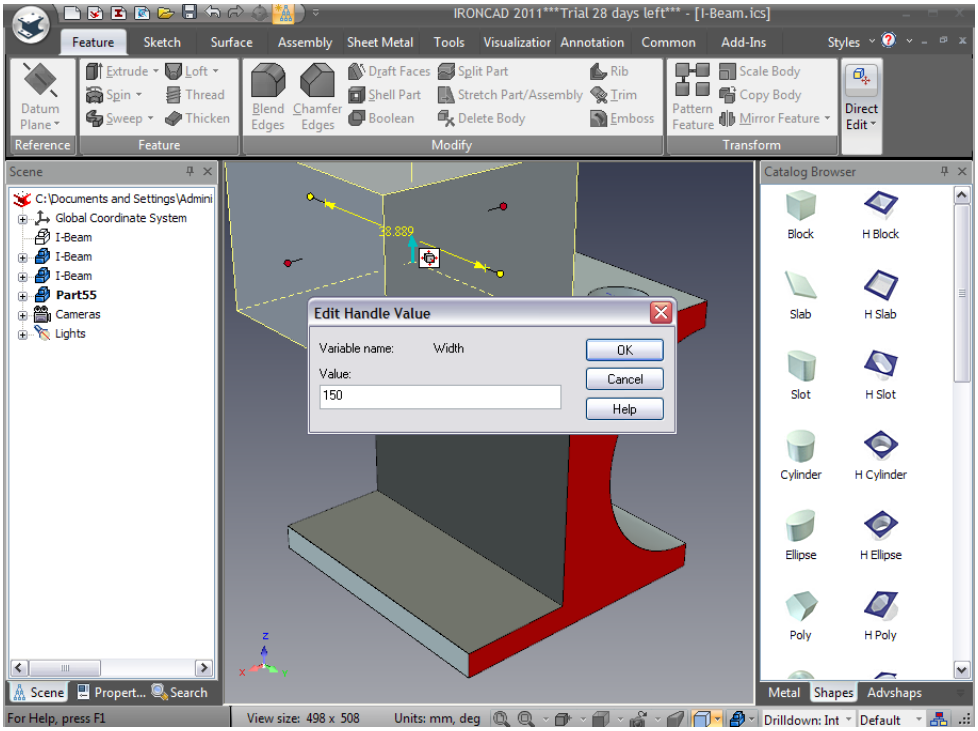
Select the midpoint of the top face edge of the I-Beam model to precisely snap to this location using this handle placement option.

8. Set Thickness

Now click on the opposite handle of the new block and change the value to 35mm to offset the thickness from the midpoint previous location outward.

9. Symmetrically Size the New Part

With the new part highlighted at IntelliShape level, select the 2 handles that go across the part in the same direction as the circular and square cuts on the I-Beam model using the CTRL click technique covered earlier. Change the value to 150mm.



IntelliShapes have 2 modes - Sizebox and Shape. Up until now we have used Sizebox mode.

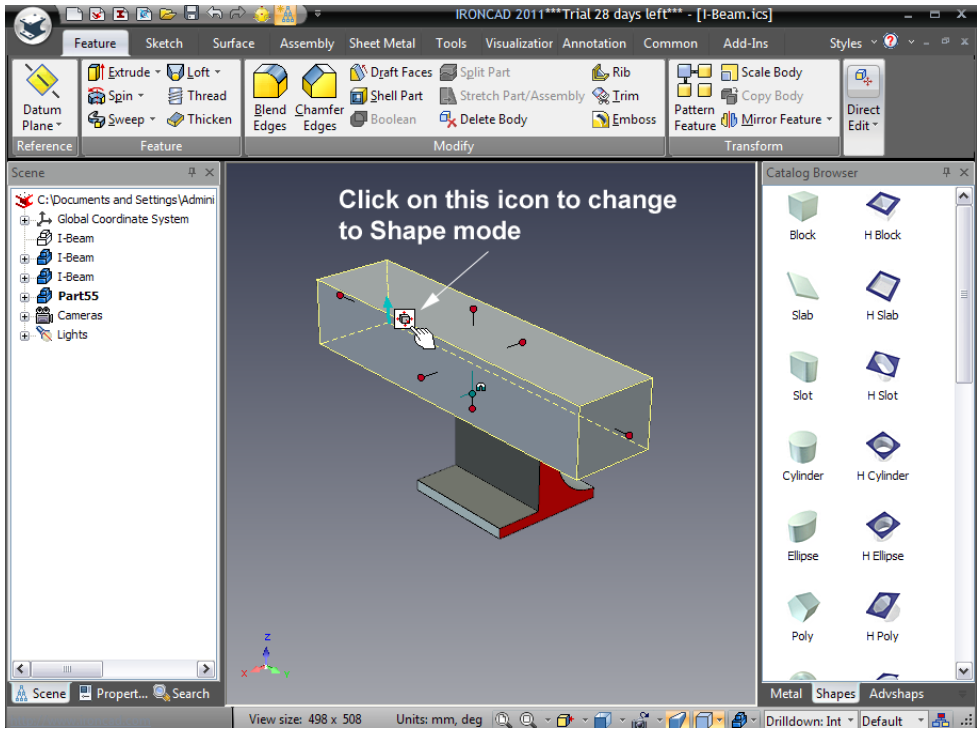
Models can also be stretched using the Shape mode. Shape handles give you direct access to the underlying 2D geometry (or Sketch) defining the IntelliShape. To change to this mode, click the white toggle icon located near the blue arrow on the extrude Intellishape. The Sizebox handles will disappear being replaced by cyan geometry and 2 triangular handles at the top and bottom of the extrusion (representing the extrusion height). As the mouse moves over each piece of geometry defining the shape, a triangular handle will be displayed on that geometry span allowing it to be dragged or positioned relative to some other feature in the scene.

10. Fit the Contents to the Scene

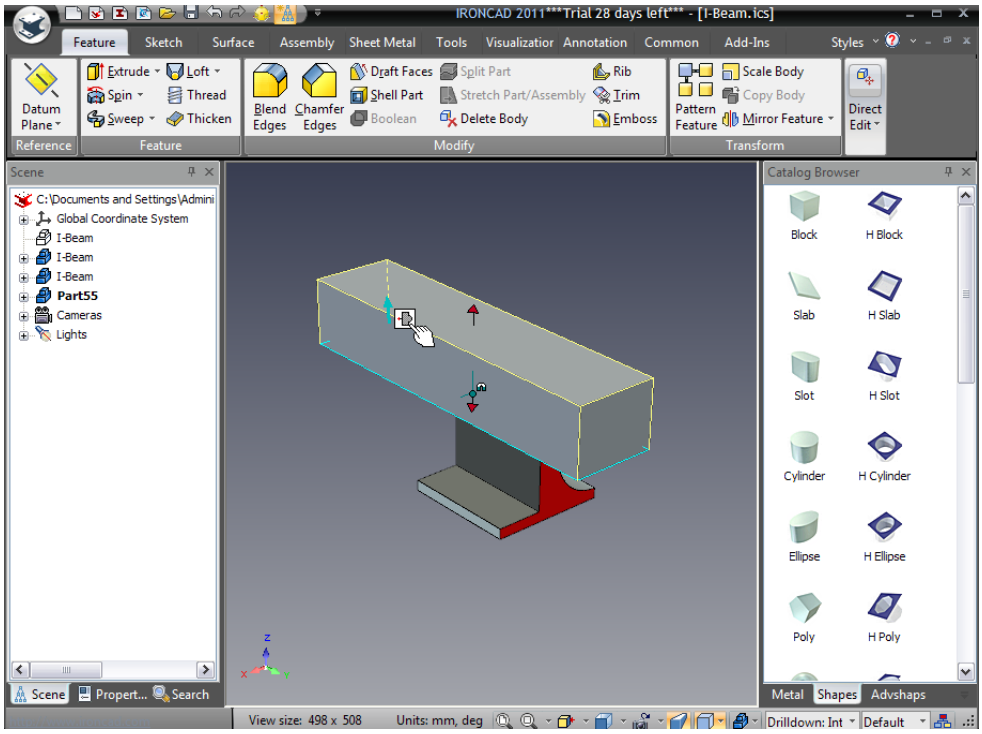
If necessary, press F8 to fit the scene window to the models.

11. Change to the Shape Handle Mode

Referring to the next image, click once on the Sizebox toggle icon to change the mode from Sizebox (yellow bounding box with red spherical handles) to Shape (cyan profile with red triangular handles).

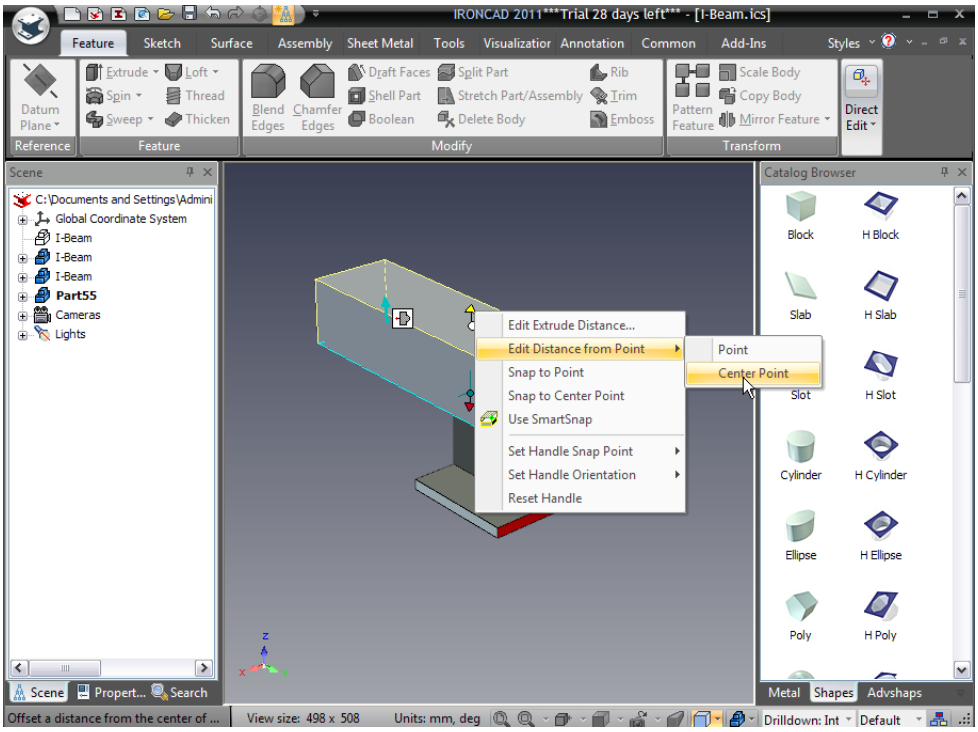


The display should look like this after changing modes:



12. Edit the Extrusion Height from a Point Location

Right-click on the top handle (it will turn yellow) and from the resulting drop down menu, select Edit Distance from Point. Choose the point type as Center Point.

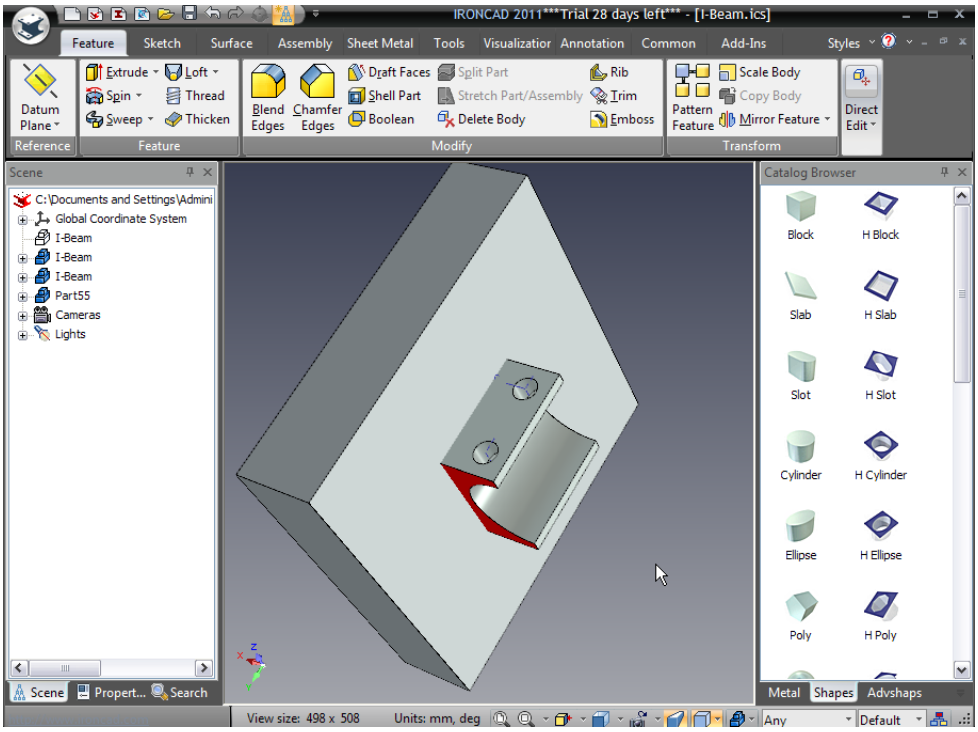


Select the edge of the round cut feature on the I-Beam, and enter 75mm in the resulting pop-up dialog. This will set the height location for our block part from the center of the I-Beam.

13. Set the Opposite Width of the Block

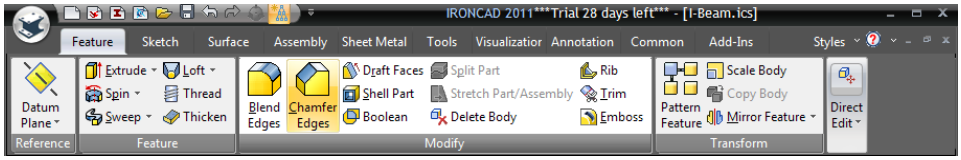
Next, click on the IntelliShape mode icon and change the mode back to Sizebox. Now select the lower handle (opposite of the previous handle selected) and change the value to 150mm.

Rotate the view and you will now have a block cutting vertically through the I-Beam model much like the next image.



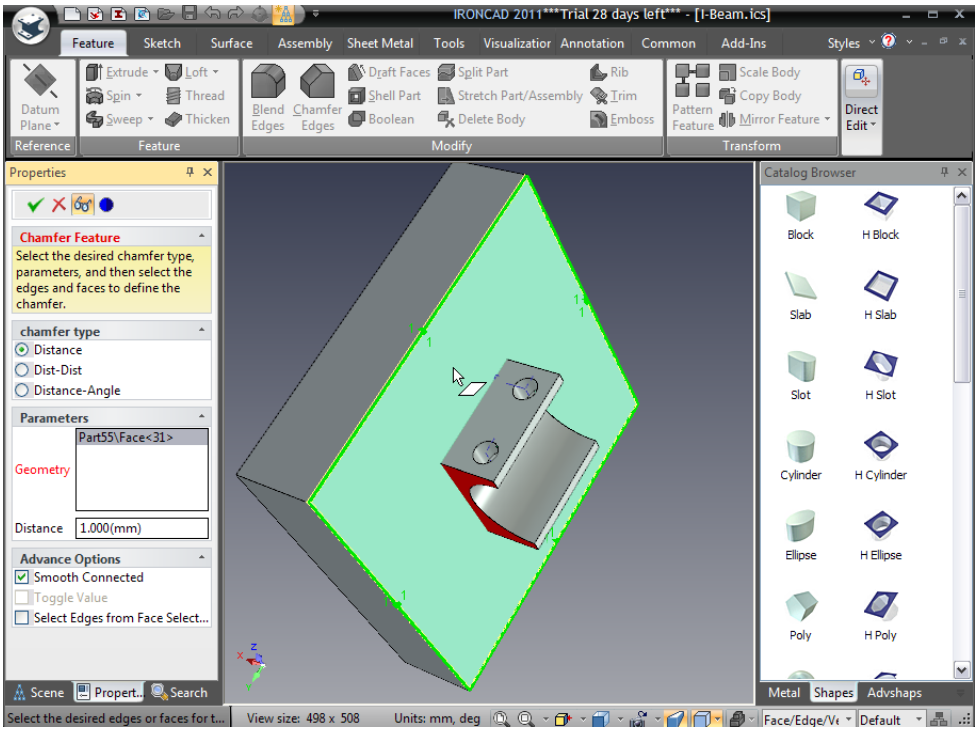
14. Using Chamfers

From the Feature tab of the Windows Ribbon bar, select Chamfer Edges.



15. Adding Chamfers to the Bottom Face of the Block

IRONCAD will display a set of feature controls in the property browser for the chamfer command. Set the distance to 1mm. Select the face that cuts the I-Beam part (bottom face) and then click on OK (the green checkmark icon).

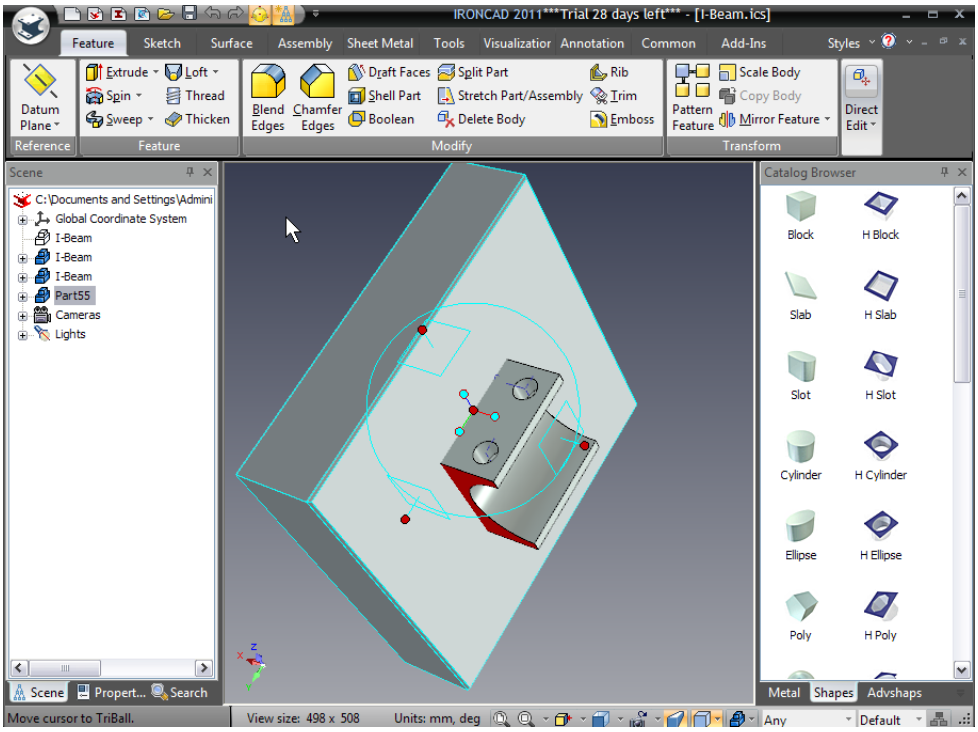


IRONCAD will produce chamfers on the edges highlighted during the selection process. Feel free to add chamfers to other edges of the block. You could try selecting individual edges by clicking on an edge till it is highlighted (referred to as drilling down). When an edge is highlighted in green, either click on the Chamfer Edges icon used previously or right-click on the edge and select Chamfer from the drop down menu.

To create the opposing tool plate:

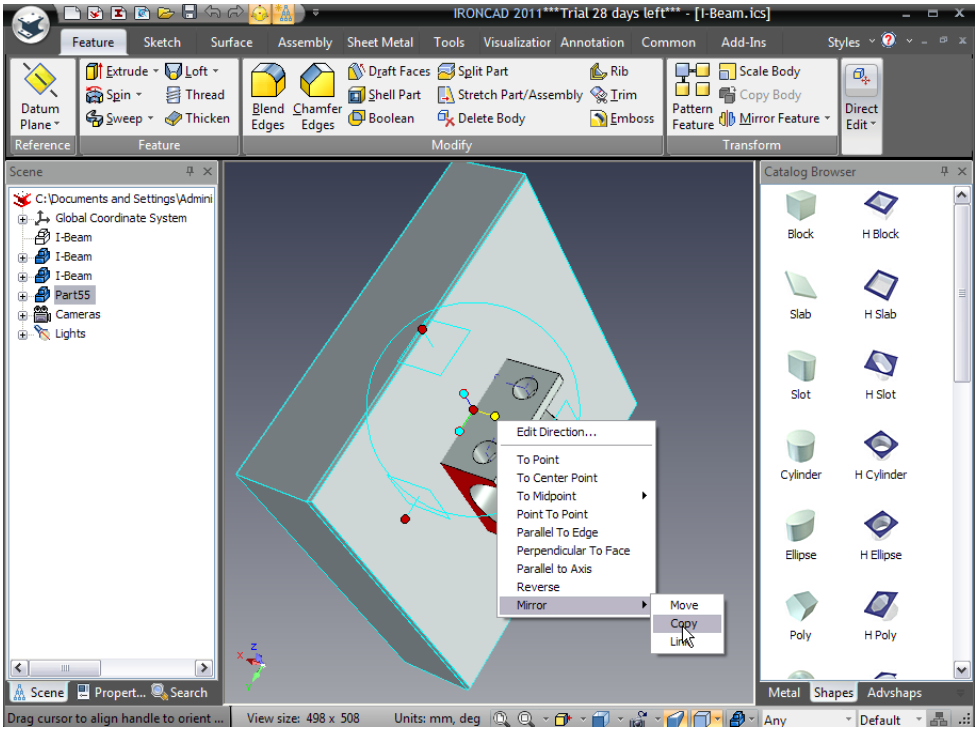
16. Activate the TriBall on the Created Plate

Select the tool plate created above and activate the TriBall.



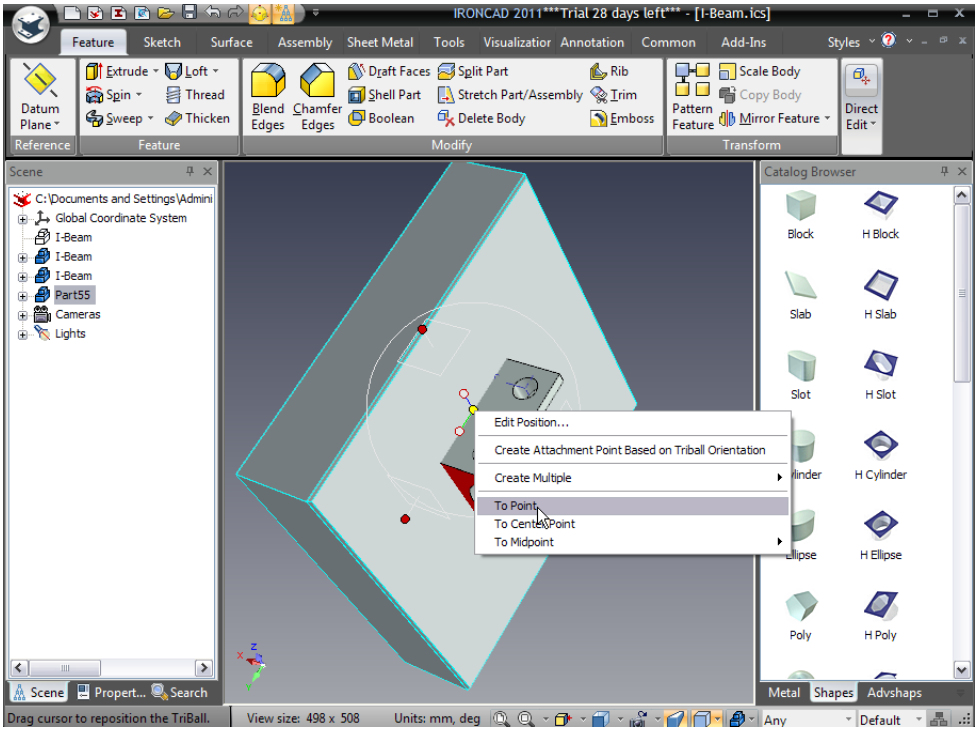
17. Create a Mirror Copy of the Plate

Right-click on the handle shown and select Mirror and then Copy.



IRONCAD will create the second tool plate mirrored about the TriBall location. Press F10 to deactivate the TriBall.

Note: If the TriBall does not appear to be on the chamfered face, press the space bar while the TriBall is active. Next, right-click on the center red sphere and select To Point.



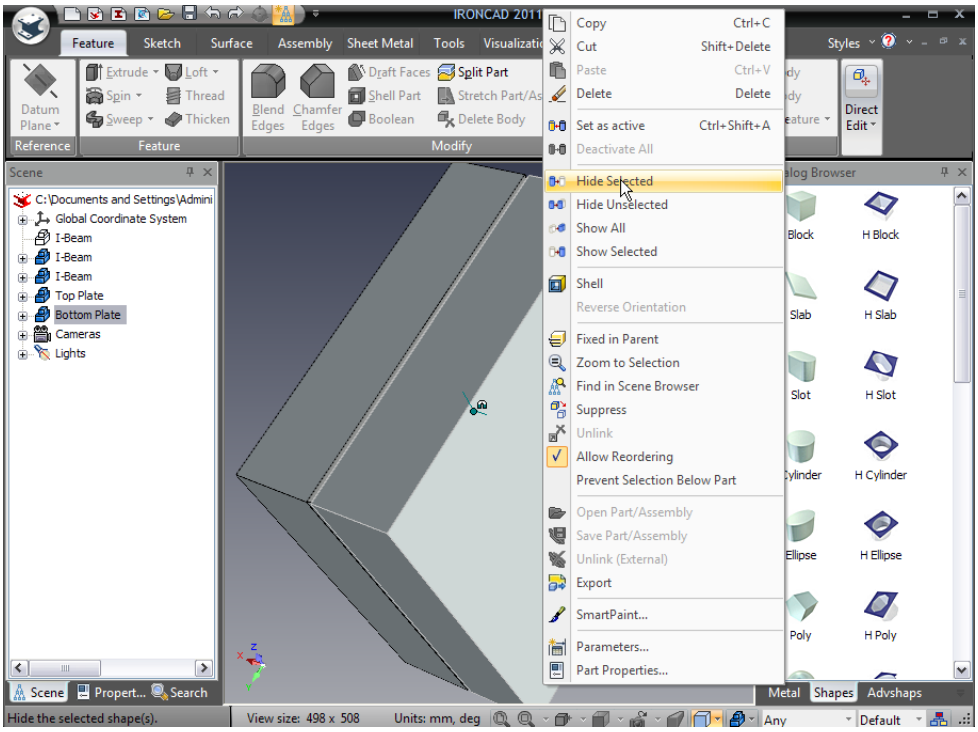
Select the face that has the chamfers and press the space bar again. The TriBall can be positioned independently of the part when it is white (Position TriBall Only Mode).

18. Rename the Plates in the Scene Browser

Click twice (not double click) on the part names of the 2 new parts in the Scene Browser and change their names to Top Plate and Bottom Plate.

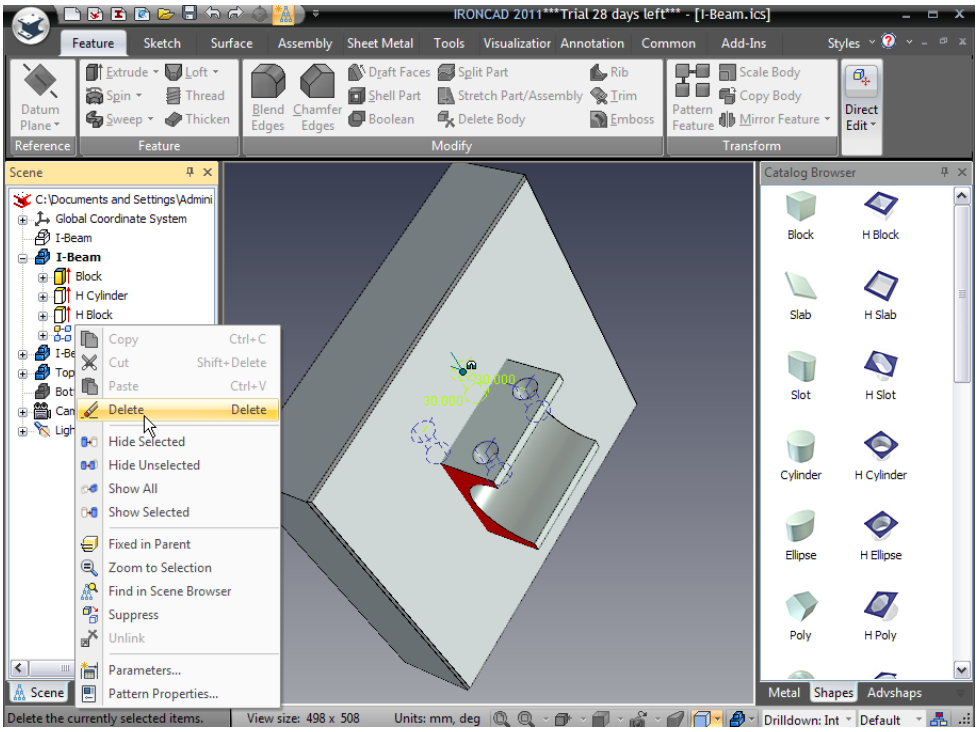
19. Hiding Parts from the Display

To simplify the display a little, right-click on the Bottom Plate and select Hide Selected from the down menu.



20. Remove the I-Beam Hole Pattern

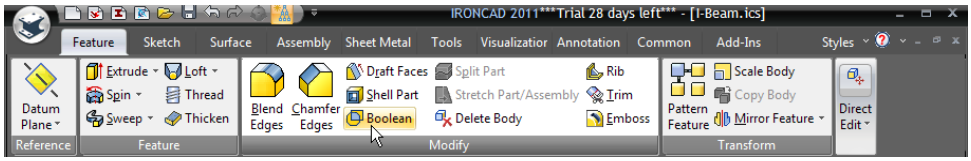
In the Scene Browser, expand the two I-Beam models trees. Right-click on the Pattern entry and select Delete to remove the top face holes as these will be produced as a secondary operation later.



3.4.2. Working with Boolean Operations

21. Use the Boolean Operation to Create Cavities in the Tool

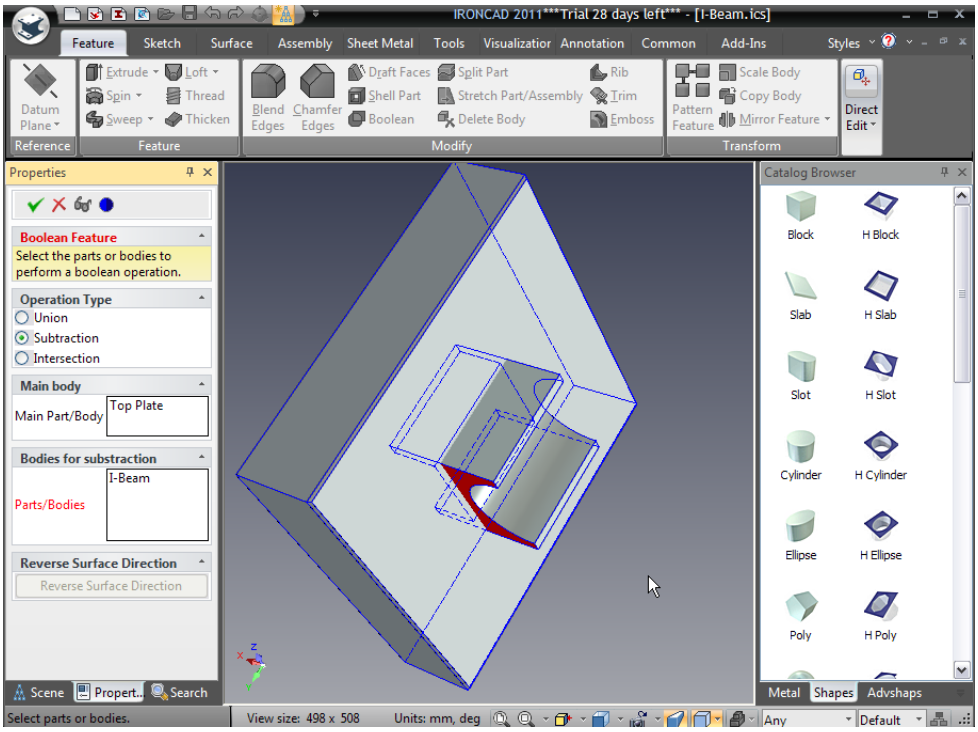
To create the cavities for the tool, select Boolean from the Feature tab of the Windows Ribbon bar.



22. Set the Type to Subtraction

In the property browser for the Boolean Operation, set the Operation Type to Subtraction:

- For the Main body, click on the Top Plate (the prompt Main Part/Body should be highlighted in red. If not, click in the box by the prompt),
- For the Bodies for subtraction, click in the Parts/Bodies box and then select the I-Beam.



Select OK to complete the operation.

23. Hide the Extra I-Beam to See the Boolean Result

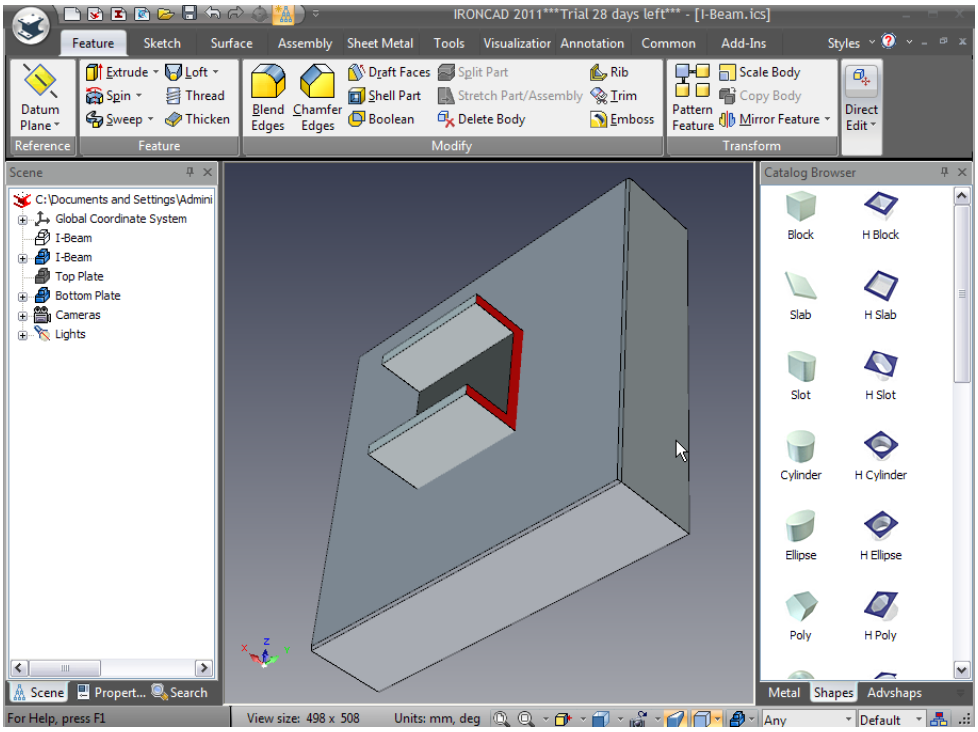
Right-click on the remaining I-Beam and select Hide Selected. IRONCAD will remove the remaining I-Beam (the copy we made earlier) from the display and show the Top Plate complete with a cavity made from the second copy I-Beam.

24. Un-Hide the Hidden Parts

In the scene background, right-click and select Show all to display all the parts that are not suppressed. Suppressing removes an item from the display until it is actually un-suppressed hence the original I-Beam will remain 'switched off'.

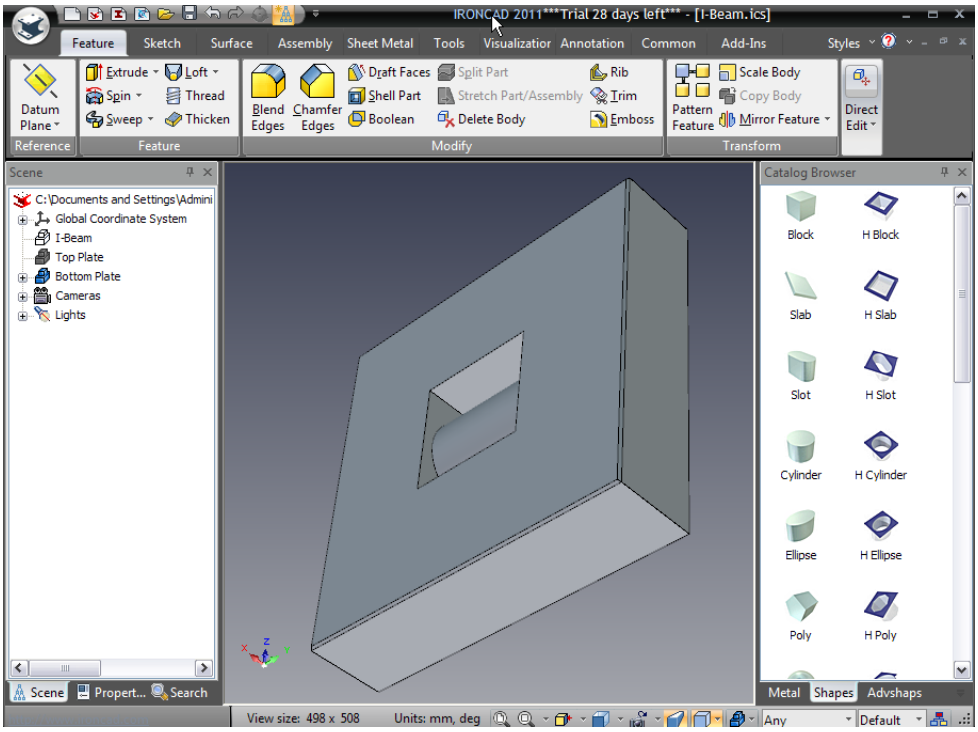
25. Hide the Top to Work on the Bottom

In the scene browser, right-click on the Top Plate and select Hide Selected from the resulting menu. Rotate the scene so that the viewing position is onto the remaining I-Beam part.



26. Repeat the Boolean on the Bottom Plate

Repeat the previous process to produce the cavity in the Bottom Plate by removing the remaining I-Beam from the Bottom Plate using the Boolean command.



The scene will show the Bottom Plate with the part cavity sunk into it.

3.4.3. Using Intelligent Features

To add a hole for the injector (sprue hole – in our case we will refer to it as the center counterbored hole), rotate the view so that you are looking on the back of the tooling plate.

27. Open the Tools Catalog and Create a Custom Hole

Open the Tools catalog and drag a custom hole onto the center of the plate.

28. Select Counterbored Hole

IRONCAD will display the Custom Hole dialog box. Ensure the units are Metric using the Unit drop down menu. Select the plane counterbored hole option (second icon from the left on the top row).

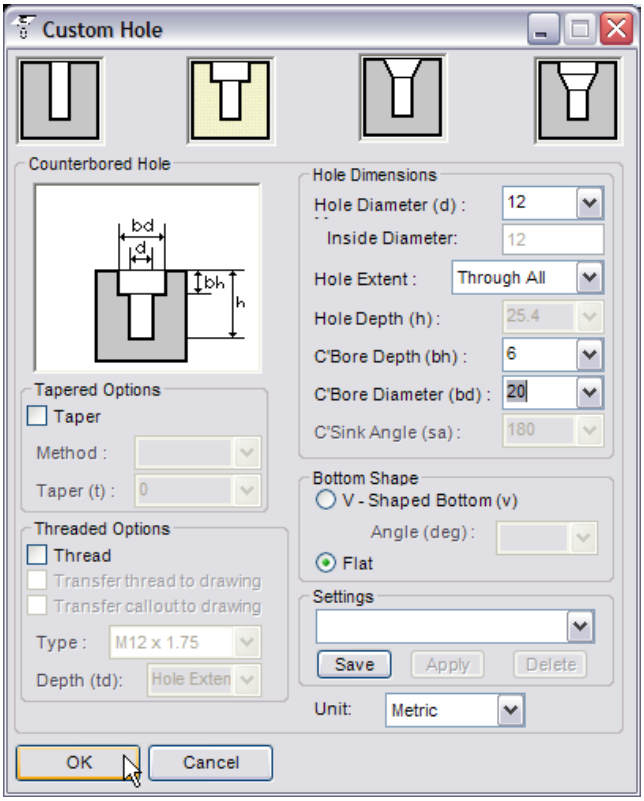
29. Specify the Counterbored Hole Size

Set the following values:

- Hole diameter to 12

- Hole Extent to Through All
- C'Bore Depth to 6mm
- C'Bore Diameter to 20mm

Click OK to create the hole



Values do not have to be selected using the drop downs. They can be entered manually. Using this tool to create features, IRONCAD will enable special properties to allow editing later. Once the features is created, select the feature and right-click to display an option called Ad-On Properties. Selecting this will redisplay the Custom Hole properties panel and enable direct changes to the feature.

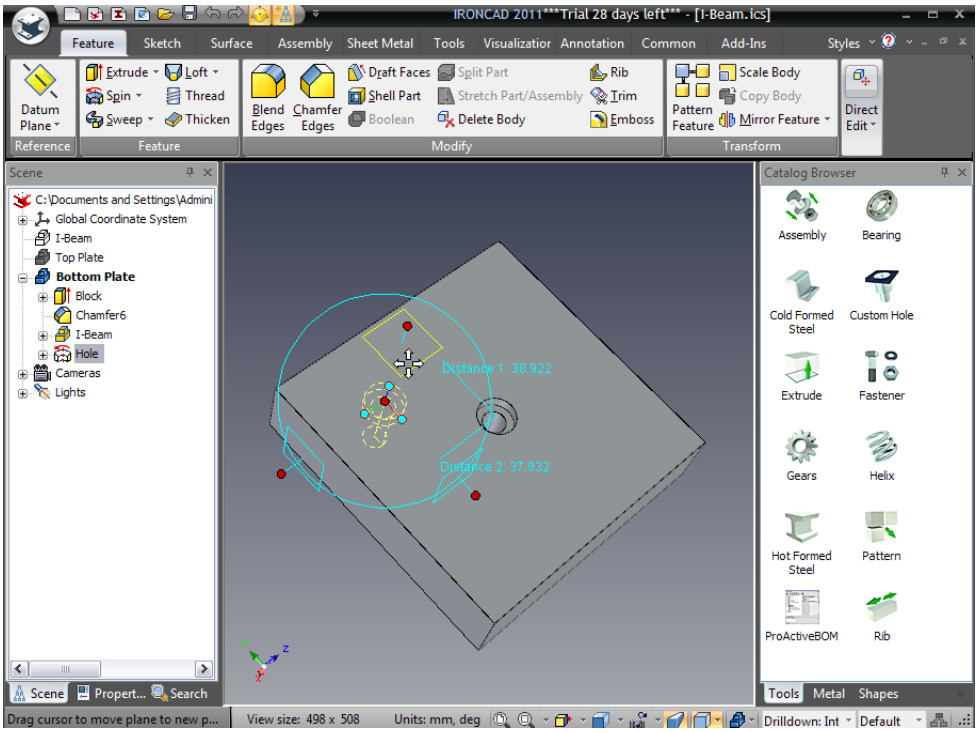
3.4.4. Using an Existing Intelligent Hole for the Basis of a New Hole

To create the outer Guide Pin Holes on the tooling plate:

30. Move the Custom Hole to the Corner

Select the custom hole at IntelliShape level and activate the TriBall. Using the Plane icon of the TriBall, right-click and drag the counterbored hole

towards a corner of the plate and use the copy function with a value of 55 for both distances.

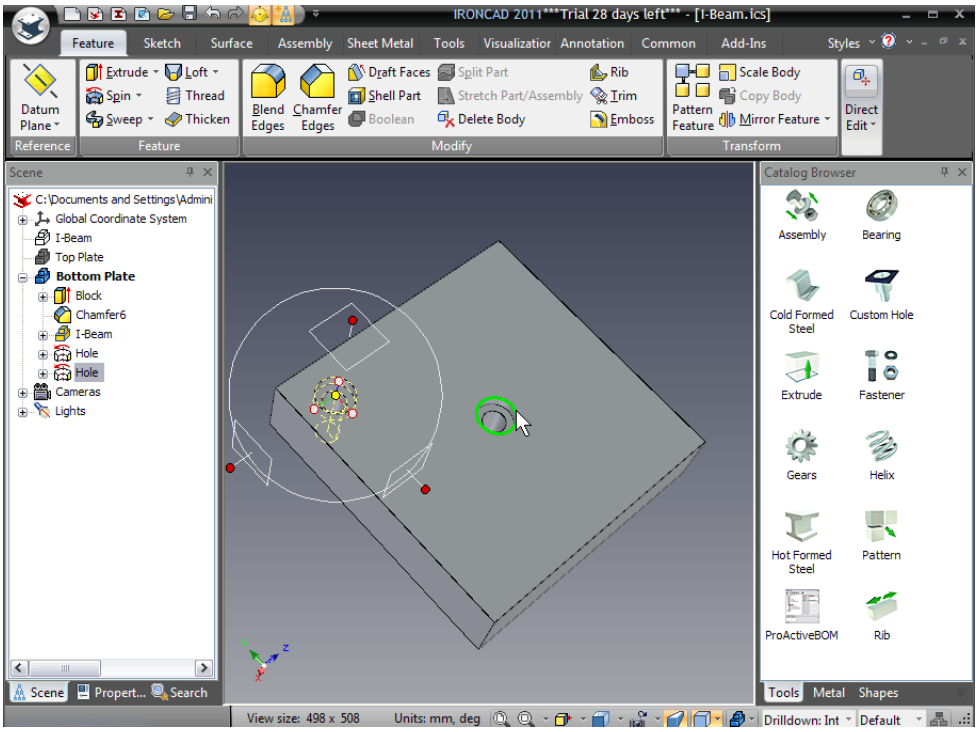


31. Select the Newly Create Hole for Copy

Deactivate the TriBall. Select the newly create hole and activate the TriBall again.

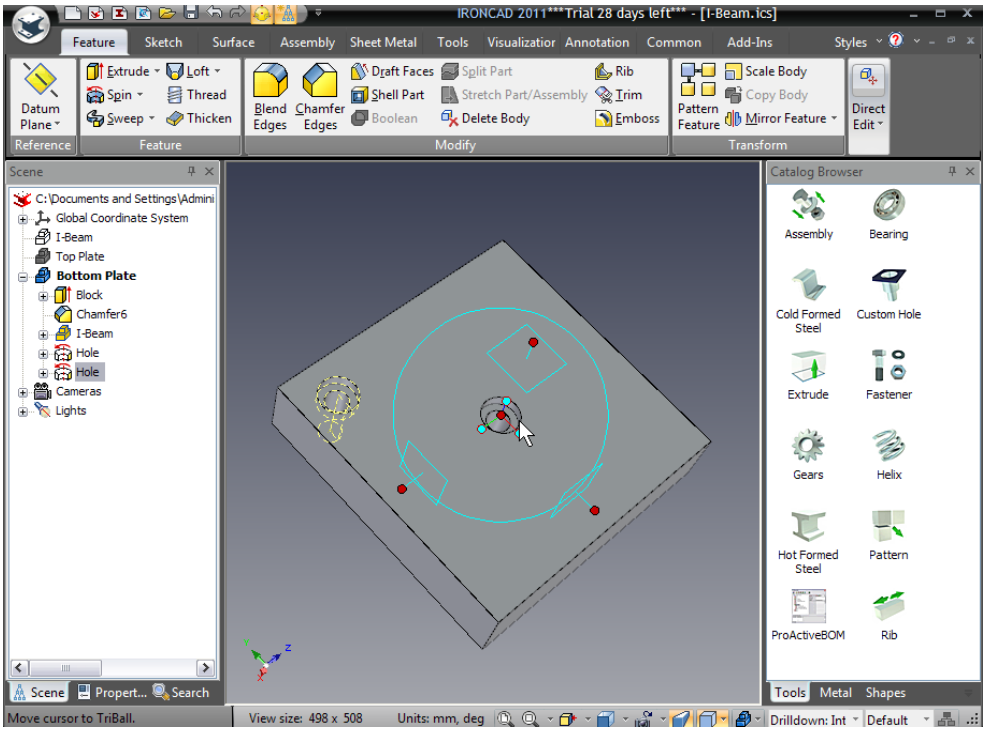
32. Deactivate the TriBall to Position it Only

With the TriBall active on the newly created hole, press the spacebar so that the TriBall can be repositioned. The TriBall will turn from cyan to white.



33. Reposition to the Center of the Counterbored Hole

Right-click on the center red sphere and select To Center Point. Select one of the edges of the center counterbored hole.

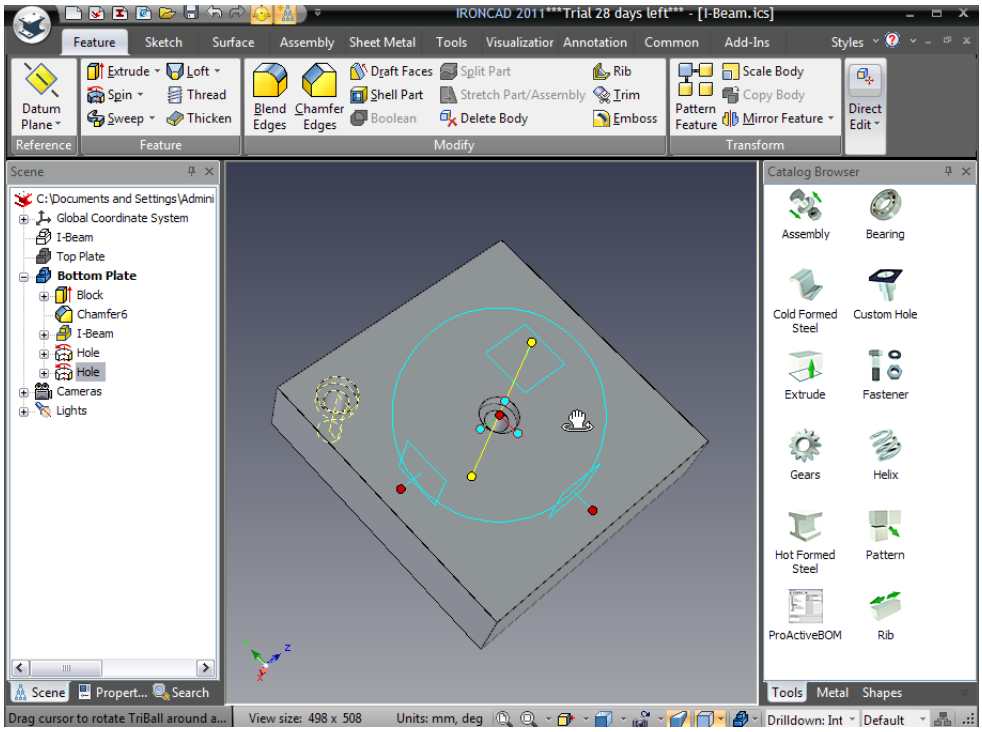


34. Enable the TriBall and Part Positioning

The TriBall will move to the center of the counterbored hole. Hit the spacebar again to lock the TriBall position.

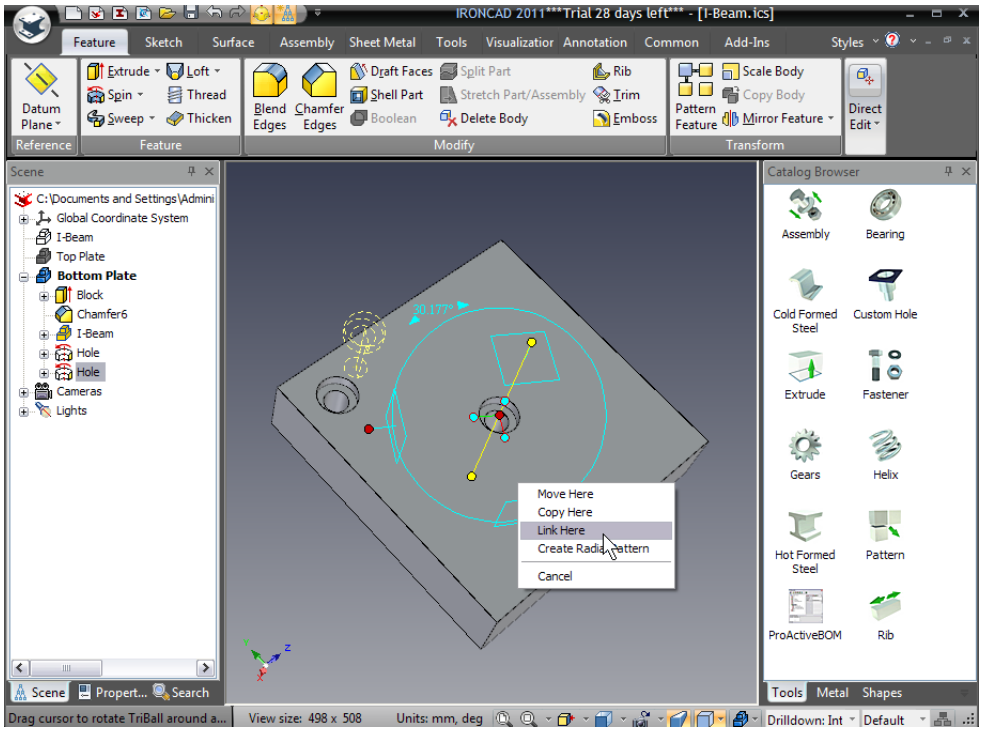
35. Select an Axis of the TriBall

The outer 3 handles of the TriBall perform actions along (translation) or around (rotation) the defined axes. Click on the red sphere on the outer handle emanating from the axis of the central counterbored hole. The axis will highlight in yellow.



36. Rotate About the Selected Axis

Move the mouse pointer inside the TriBall circle. The mouse pointer changes to a clenched fist with an arrow around the wrist as shown in the previous image. Right-click and drag the mouse around the axis a short amount and release the mouse button. IRONCAD will display a small drop down menu.



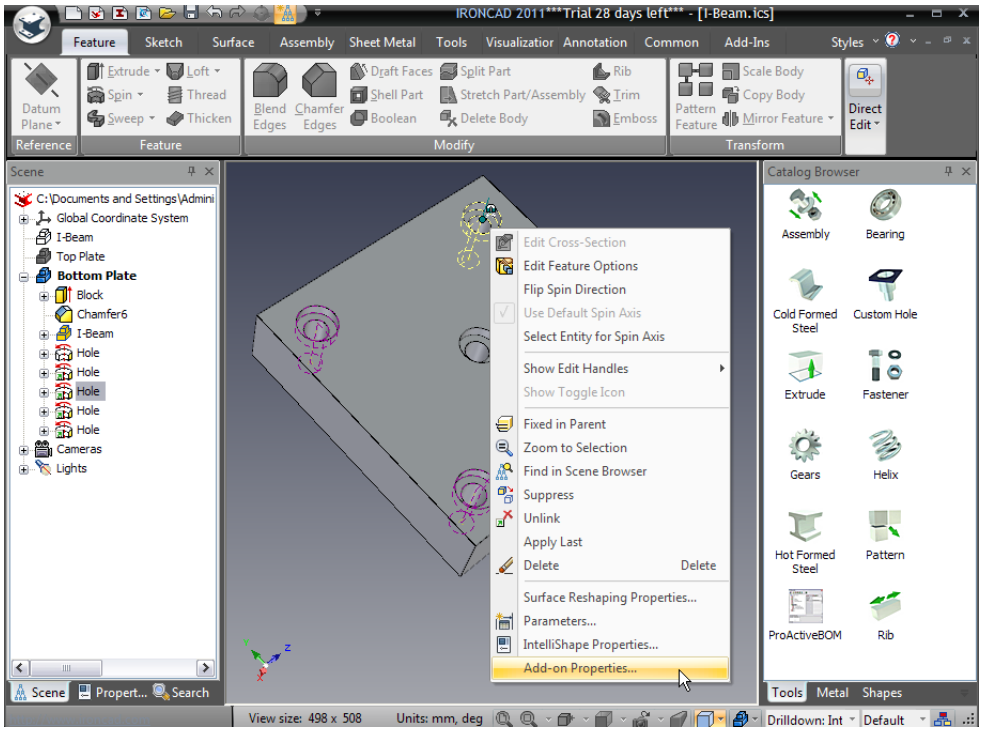
37. Make Linked Copies of the Hole

Select Link Here in the pop-up dialog. In the resulting menu, set the number to 3 and the Angle to 90 degrees. Hit OK and deactivate the TriBall.

To set the size of the Guide Pin holes:

38. Edit the Add-on Properties of the Guide Pin Hole

Click on any one of the guide pin hole so it highlights in yellow. Right-click and select Add-on Properties.



39. Set a New Value and All Links Update

Change the Hole Diameter to 18mm and the counter bore (C'Bore) to 24. Press the return key to accept the command.

Since all the Guide Pin holes are linked, all the hole diameters change.

The Bottom Plate is now completed. Next, the geometry of the Bottom Plate will be used to generate the holes in the Top Plate.

3.4.5. Creating Sketch Based Features

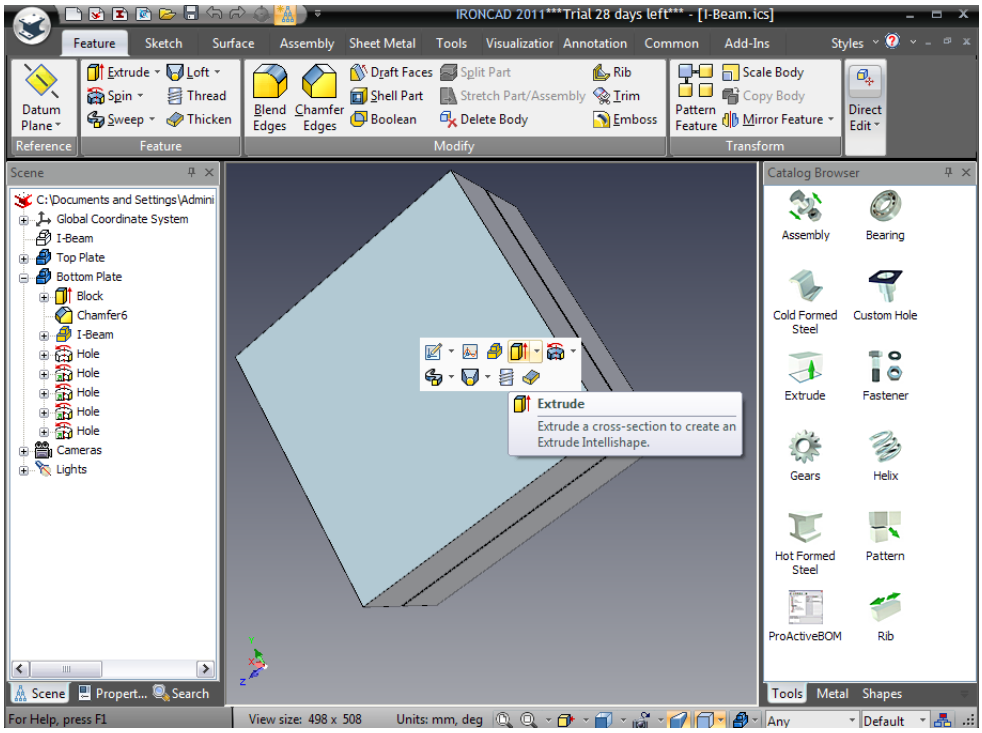
IRONCAD can create new features based on new 2D geometry and from geometry created from existing features as the basis of new features.

40. Show All Hidden Components

Move the mouse over the scene background. Right-click and select Show All. IRONCAD will display the Top Plate. Rotate the viewpoint to look on the bottom of the Top Plate.

41. Access the Quick Access Toolbar Commands

With nothing selected, press the 'S' key. This is a hotkey giving direct access to the most common functions used at the current selection level.

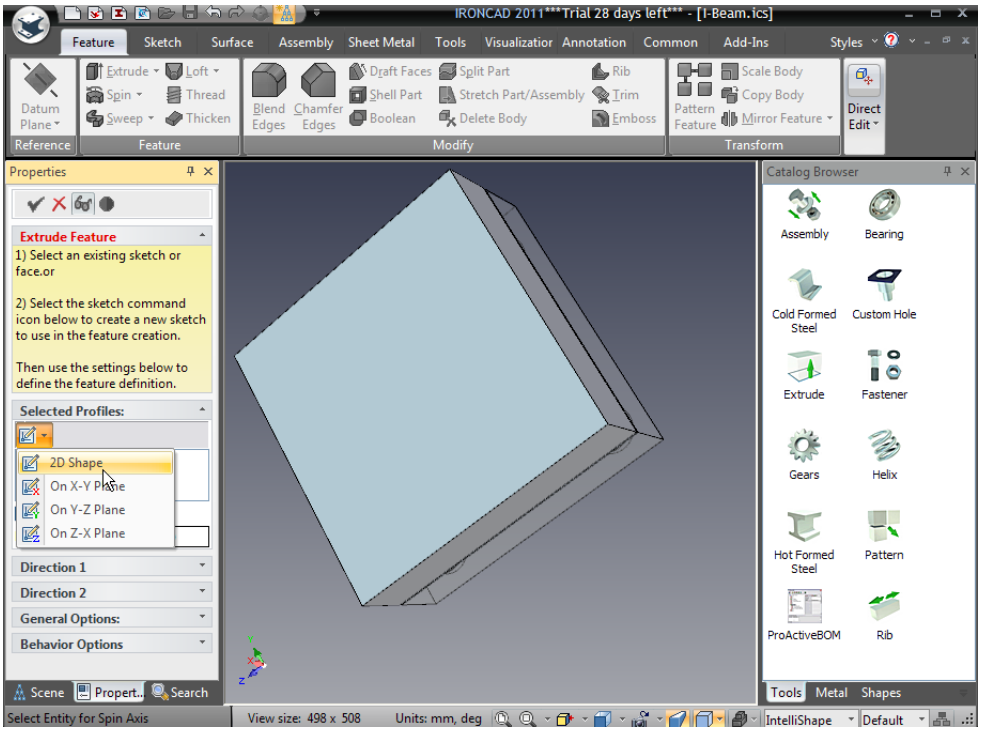


42. Select Extrude from the Quick Access Toolbar

IRONCAD changes the property browser to begin the extrude command. The 'Select a Part from Scene' radio button should be active (this is to add the newly feature to a selected part). Click on the Top Plate to continue.

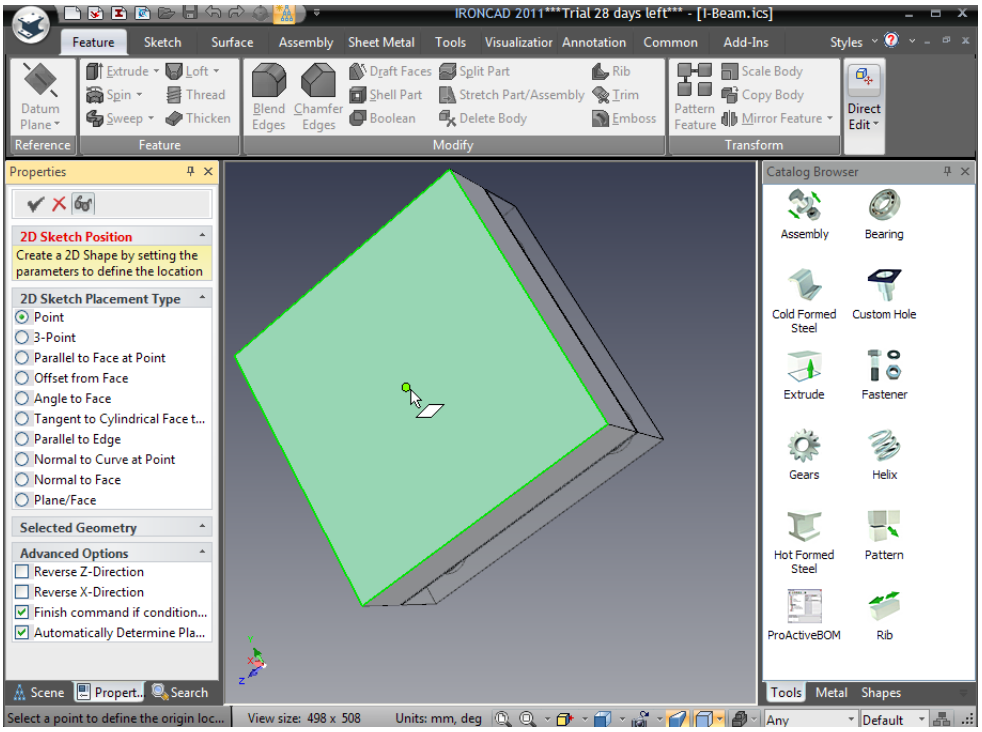
43. Start a 2D Shape (Sketch)

The property browser will now reflect the extrude command options. In the Selected Profiles panel, click the sketch icon and set the profile sketch plane to '2D Shape'.



44. Select a Point for the Sketch Placement Location

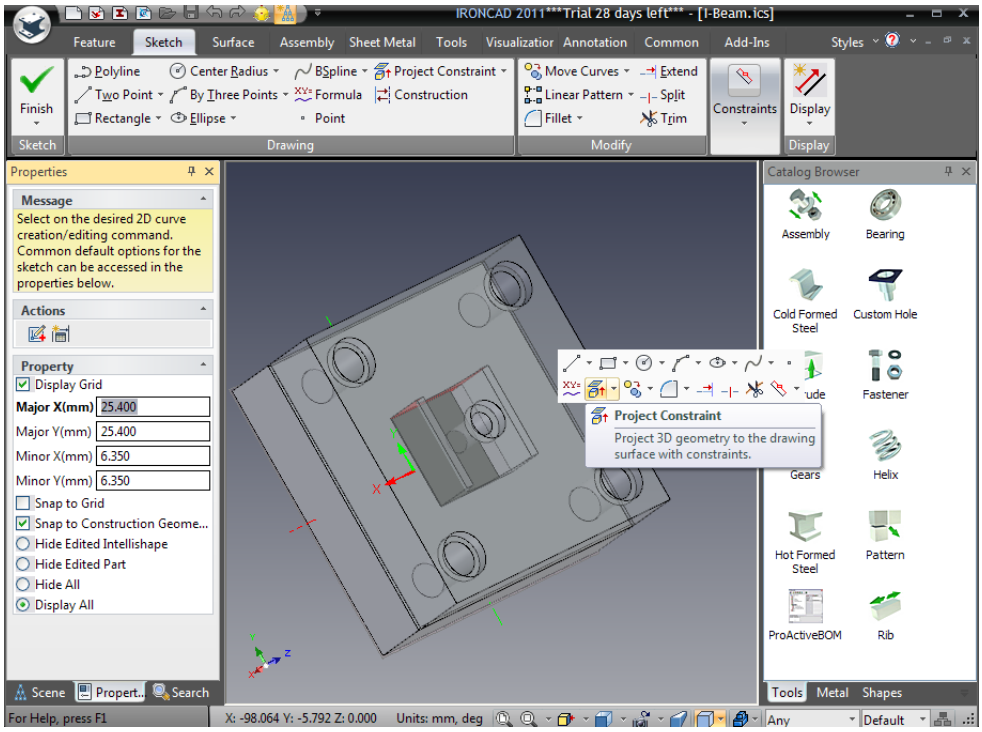
Next in the 2D Sketch Placement Type, click on Point and indicate the center of the Top Plate face.



IRONCAD will rotate the viewing point to look orthogonally onto the bottom face of the Top Plate.

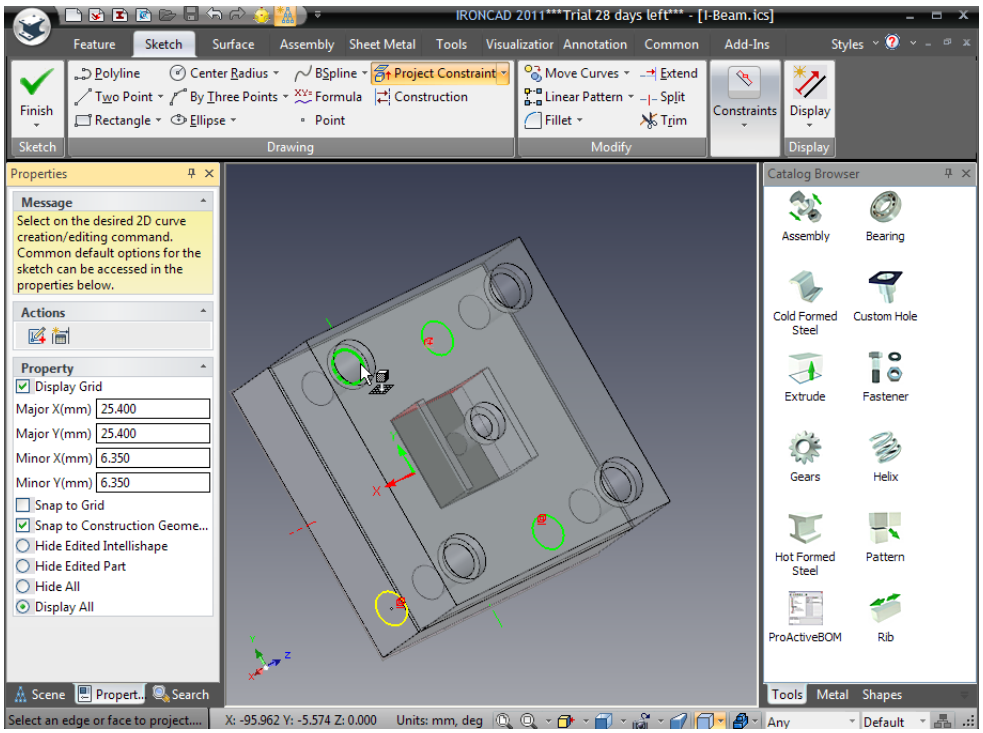
45. Using the Project Constraint Tool

Rotate the viewing point so that the model is seen from the Bottom Plate. IRONCAD renders the Bottom Plate in a semi-transparent way helping to keep the emphasis on the part being affected (i.e. the Top Plate). Click the 'S' hotkey and select the Project Constraint tool.



46. Select the Holes from the Bottom Plate

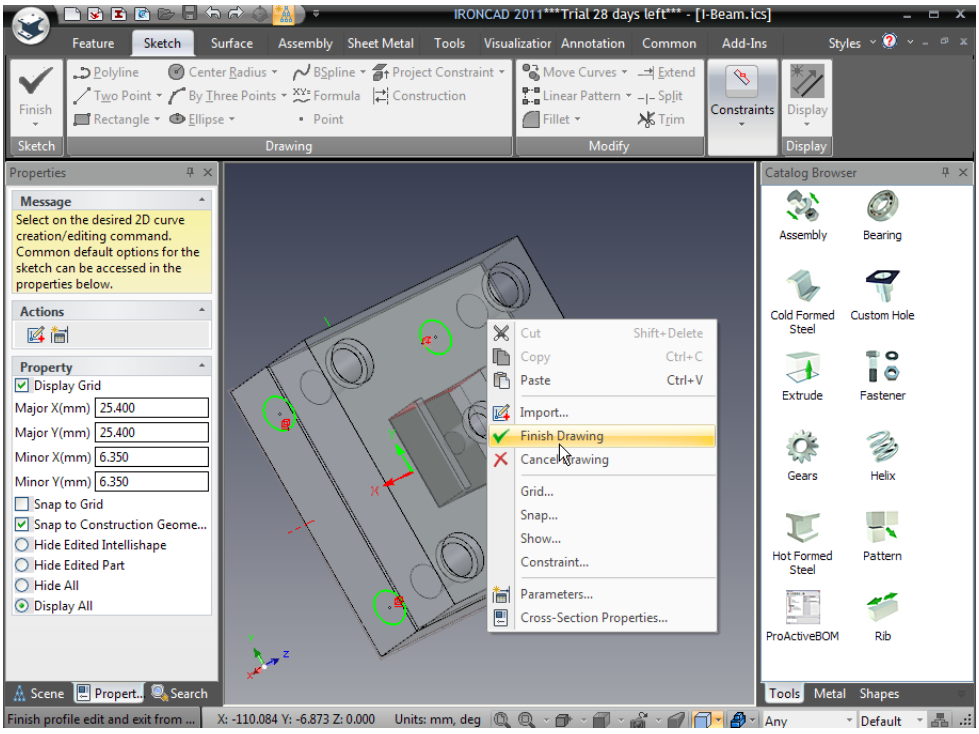
Select the inner bore of the four Guide Pin holes as shown.



IRONCAD will project these edges onto the sketch plane for the Top Plate and apply a relationship, or constraint, so that any changes in the Bottom Plate will be reflected in the features created in the Top Plate.

47. Complete the Sketch

Press the Esc key to terminate the action and then click the right mouse button and select Finish Drawing to complete the sketch.



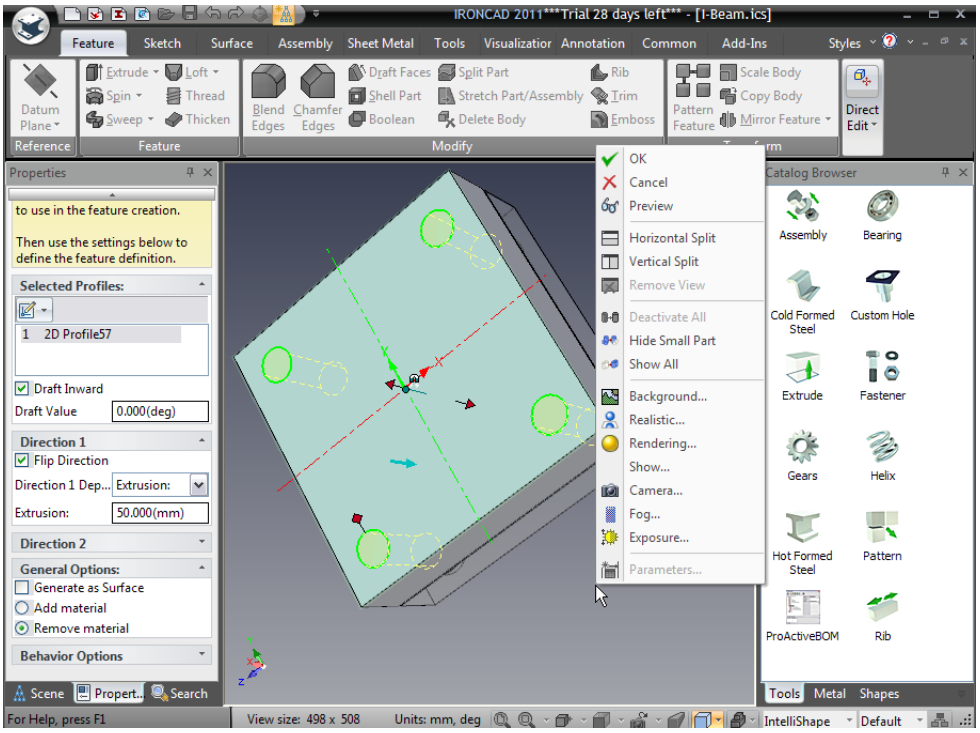
48. Enter the Options for the Extrude

IRONCAD will rotate the view back to the pervious viewing direction. The property browser will be updated to reflect the actions appropriate to the operation being completed (Extrude). Set the following options:

- In the Direction 1 panel, check the Flip Direction box
- In the General Options panel, select the Remove material radio button.

49. Complete the Extrusion of the Guide Pin Holes

Right-click in the scene and select OK. IRONCAD will now cut the four Guide Pin holes in the Top Plate.



3.4.6. Working with Blends/Chamfers

Next, we will create the Guide Pins for the plates.

50. Rotate the Part to the Top of the Bottom Plate

Rotate the view to look on the top face of the Bottom Plate.

51. Drop a New Part for the Start on the Pin

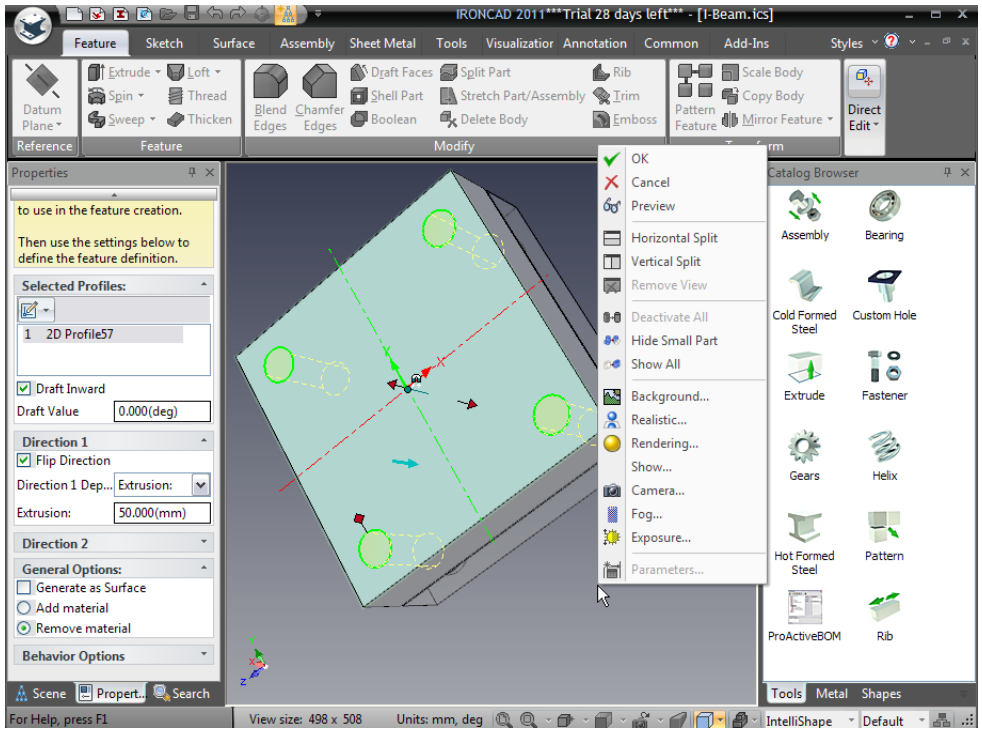
Drag a Cylinder from the Shapes catalog onto the top face of the Bottom Plate using the right mouse button. Select ‘Drop As Part’. Resize it to be 23.5mm diameter and 6mm deep by clicking on the new part until the handles are shown then right-clicking on a handle and setting the Sizebox values.

52. Add a Second Cylinder Feature

Add a second cylinder to the center of the new cylinder (left click and drag). Resize it to be 17.9mm in diameter and 60mm high.

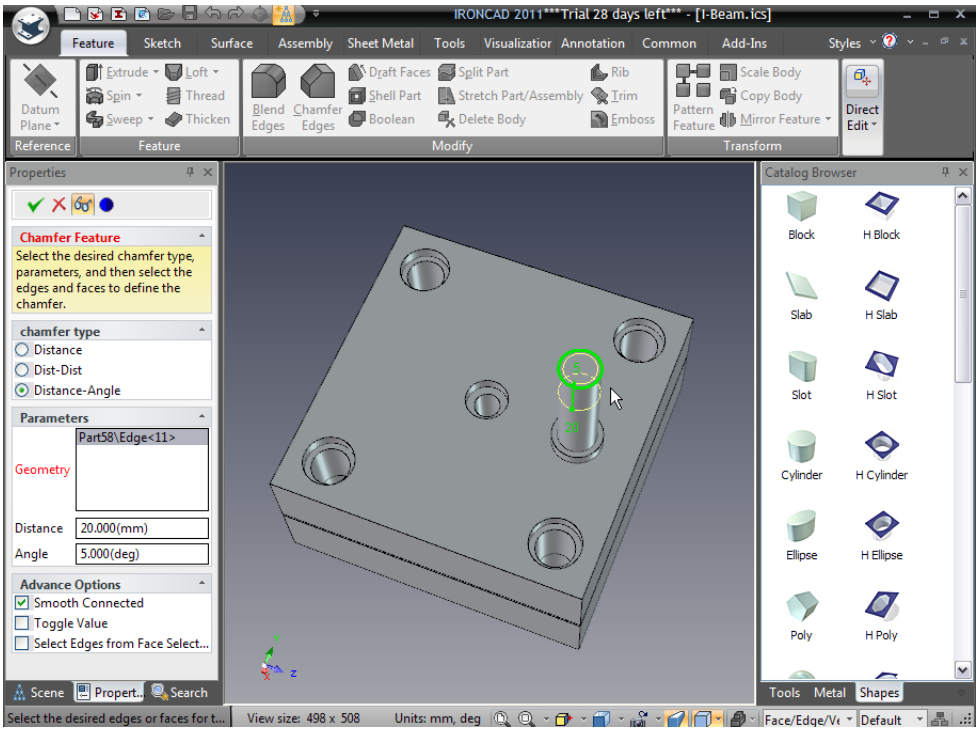
53. Enable Chamfer

With the Guide Pin highlighted in cyan, press the ‘S’ hotkey and select Chamfer Edges.



54. Set the Chamfer Values

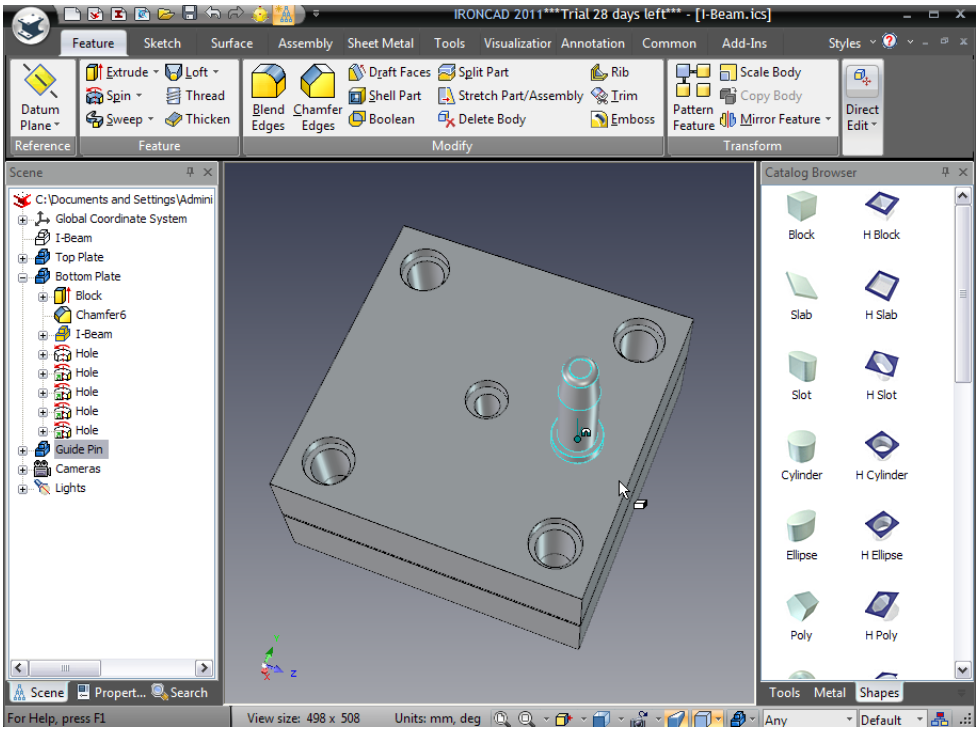
In the Chamfer property browser panel, set the Distance–Angle radio button. Set the distance to be 20mm and the angle to be 5 degrees and click on the top edge of the Guide Pin. IRONCAD will preview the chamfer.



Click Toggle Radius Values if the chamfer doesn't run down the cylinder as shown.

55. Add a Blend to the Tip of the Pin

In a similar manner add a 3mm blend to the edge at the tip of the Guide Pin. Rename the part in the Scene Browser to Guide Pin.

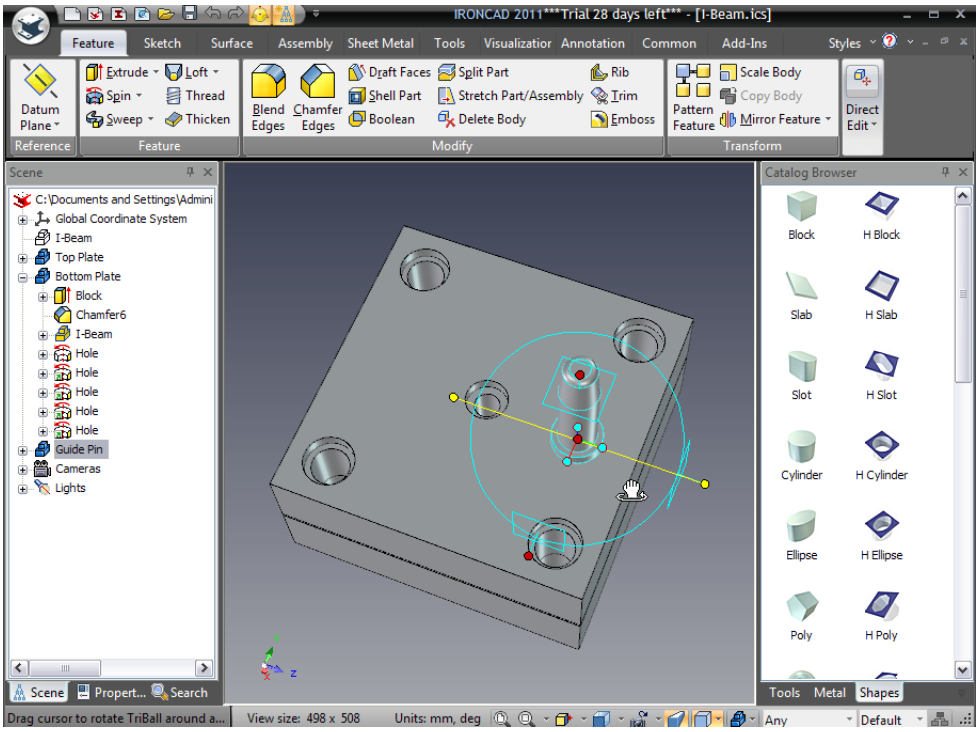


3.4.7. Positioning with Constraints

The Guide Pins should mate with the holes in both the Top Plate and Bottom Plates. The TriBall is a valid tool for this positioning purpose. However if the holes change position, the Guide Pins would not update using the TriBall method since there are no relationships between these parts and features. In this case, the positioning and control constraints functions provided by IRONCAD would be more suitable.

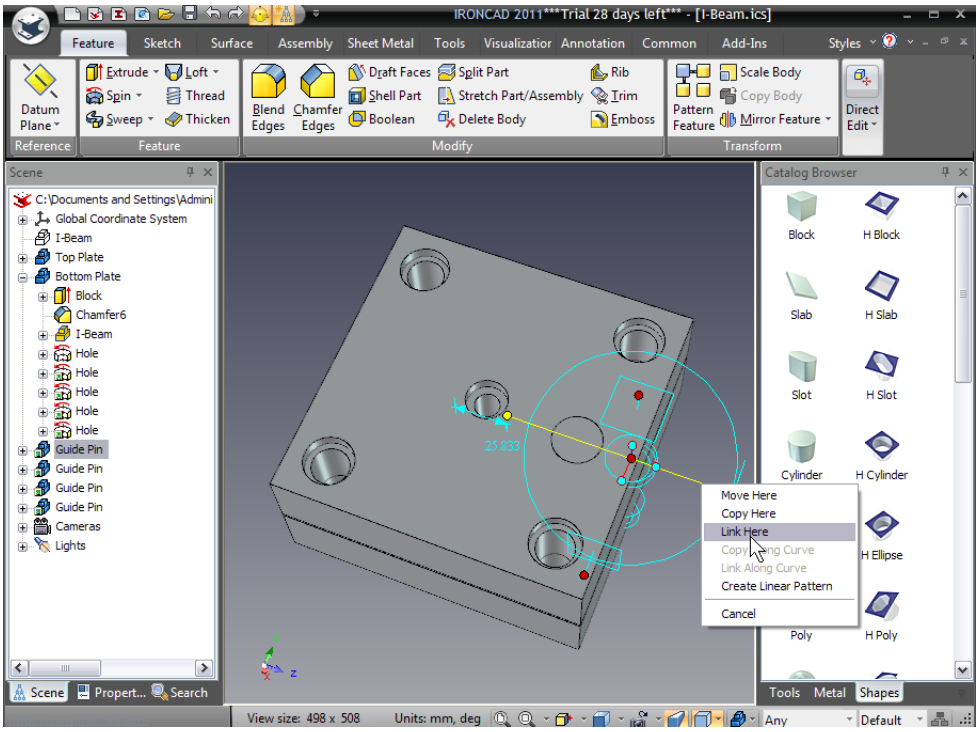
56. Rotate the Pin with the TriBall

With the Guide Pin selected, activate the TriBall and select one of the outer axis handles that are parallel to the Bottom Plate face as shown in the next image. Move the mouse pointer inside the TriBall circle and using the right mouse button rotate the Guide Pin 180 degrees to point through the Bottom Plate.



57. Create 3 Link Copies of the Pin

While the TriBall axis is active, right-click on the active handle outside of the TriBall and drag the Guide Pin a short distance and release the mouse button.



From the resulting drop down menu create 3 linked copies of the Guide Pin and then deactivate the TriBall.

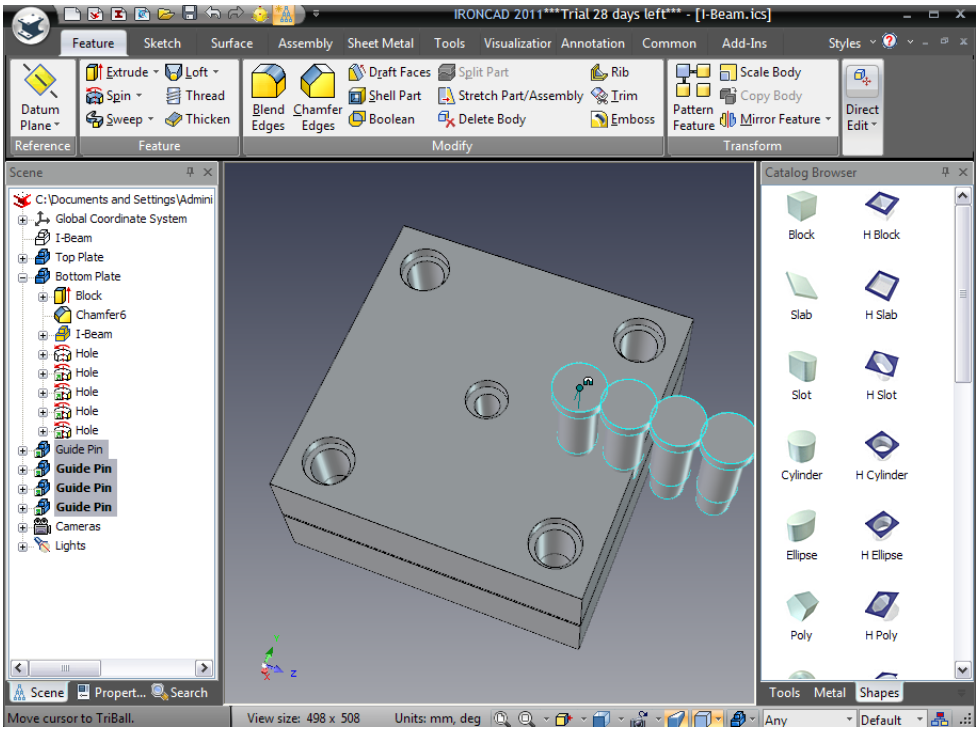
The TriBall could be used to position each Guide Pin with respect to their locating holes at this point. However in this case, it would be more prudent to constrain the Guide Pins to each locating hole.

58. Select All the Guide Pine

In the Scene Browser, select the four Guide Pins by CTRL clicking each of the Guide Pins.

59. Move the Pins into a Visible Area of the Scene

Activate the TriBall and using the right mouse button click and drag the four Guide Pins away from the Bottom Plate by 50mm as shown in the image. Deactivate the TriBall.

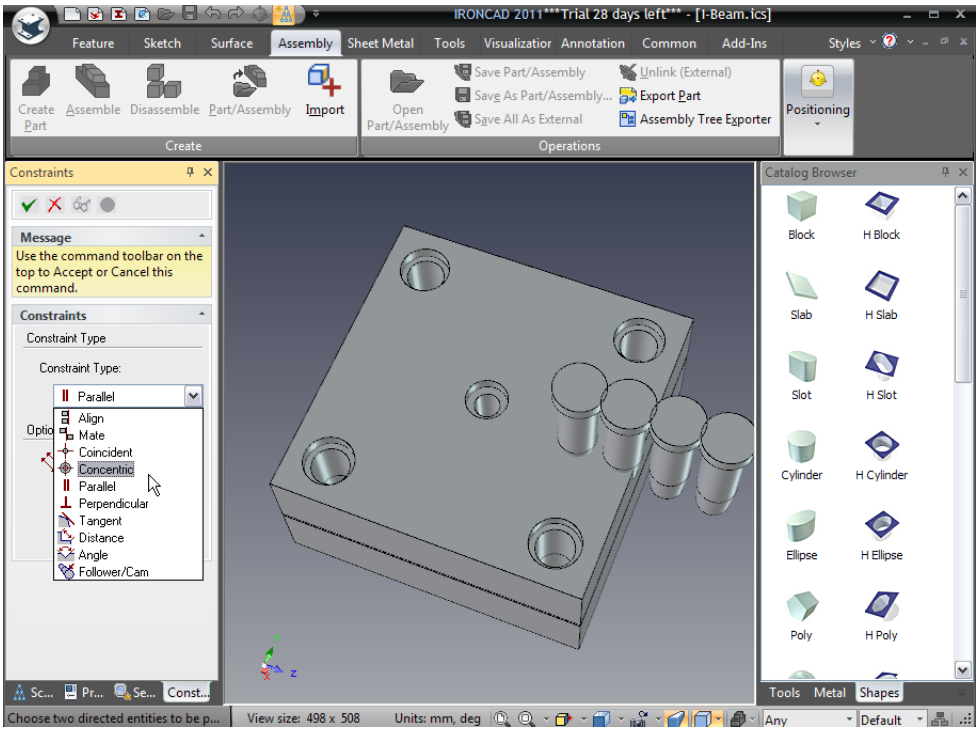


60. Activate the Positioning Constraint Tool

Select the Assembly tab and on the extreme right of the Windows Ribbon bar, select the Positioning Constraints tool.

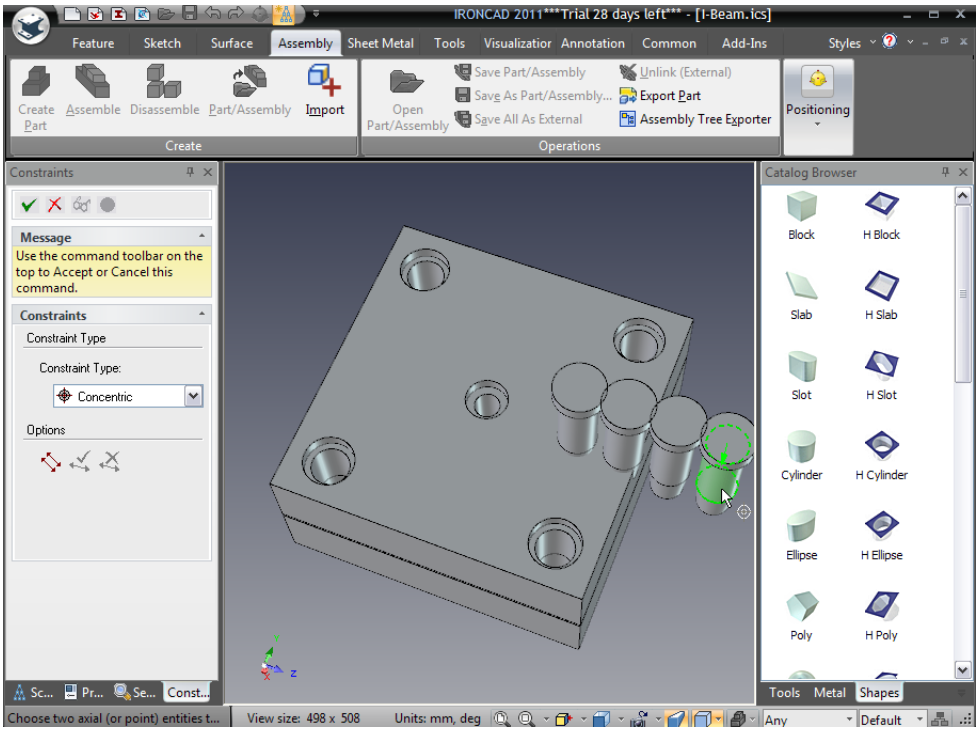
61. Select Concentric

In the IRONCAD Property Browser, select Concentric.

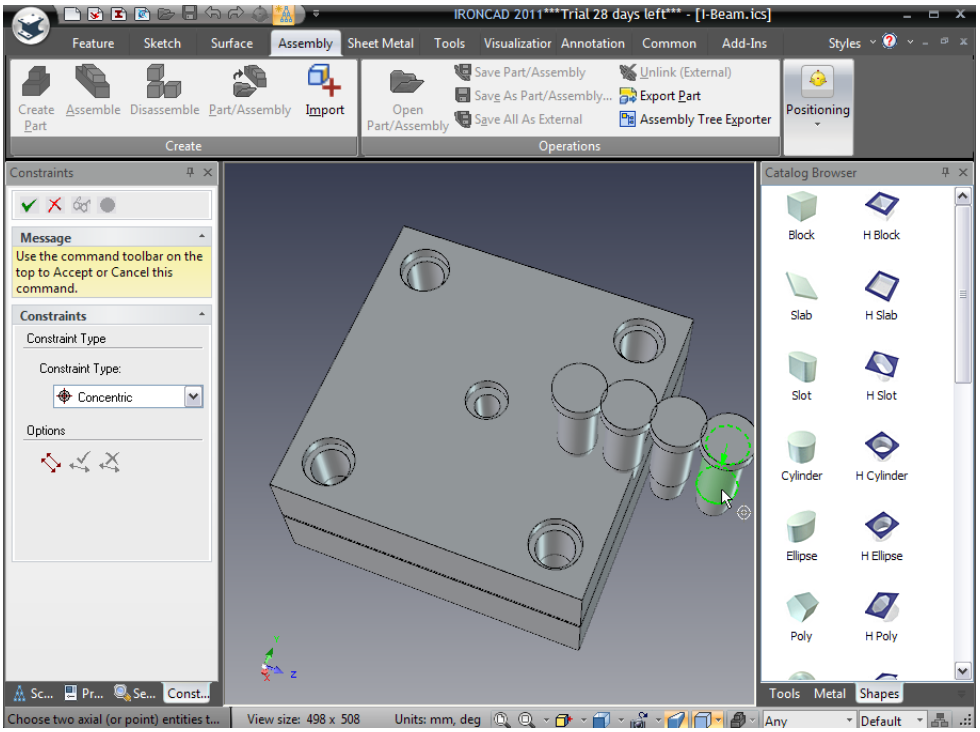


62. Position the Guide Pin to a Hole

Click on the 15.9mm portion of one of the Guide Pins. Then click on the bore of one of the Guide Pin holes. IRONCAD will snap the selected Guide Pin to its mating hole concentrically.



IRONCAD will 'ghost out' the linked pins making it easier to see the receiving geometry.



63. Accept and Repeat to All Pins

Click the left mouse button to accept the constraint positioning (Right-click would cancel the current positioning). Repeat the process for the remaining Guide Pins. When all Guide Pins are constrained, click the Apply and Exit icon (the green tick at the top of the Property Browser).

In the real world, dowels would be used to ensure the tool is assembled in the right orientation. This design will use an offset Guide Pin.

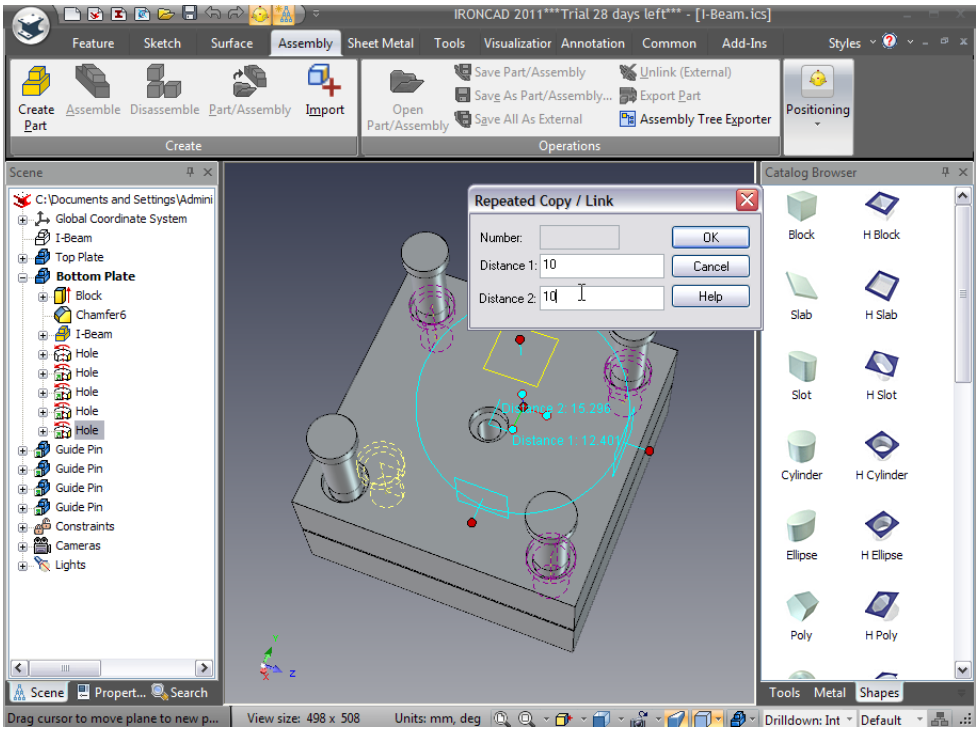
64. Select a Guide Pin Hole

In the scene browser, expand the feature tree of the Bottom Plate and select one of the Guide Pin holes that have a green arrow on its icon (the green arrow indicates the hole is linked).

IRONCAD will highlight the selected hole in yellow and any liked copies of the hole will be highlighted in purple.

65. Reposition a Hole Using the TriBall

Activate the TriBall and move the hole towards the center of the plate using the Plane function of the TriBall. It doesn't matter that the TriBall shows over the Counterbored (Sprue) hole. Move the hole 10mm in both directions.



The corresponding Guide Pin will move and the hole in the Top Plate will move in sync with the guide Pin Hole as the geometry was created by not only projecting the geometry of the hole, but also adding a constraint to the projected geometry keeping the resultant geometry in sync with the hole it was created from.

3.4.8. Adding Draft

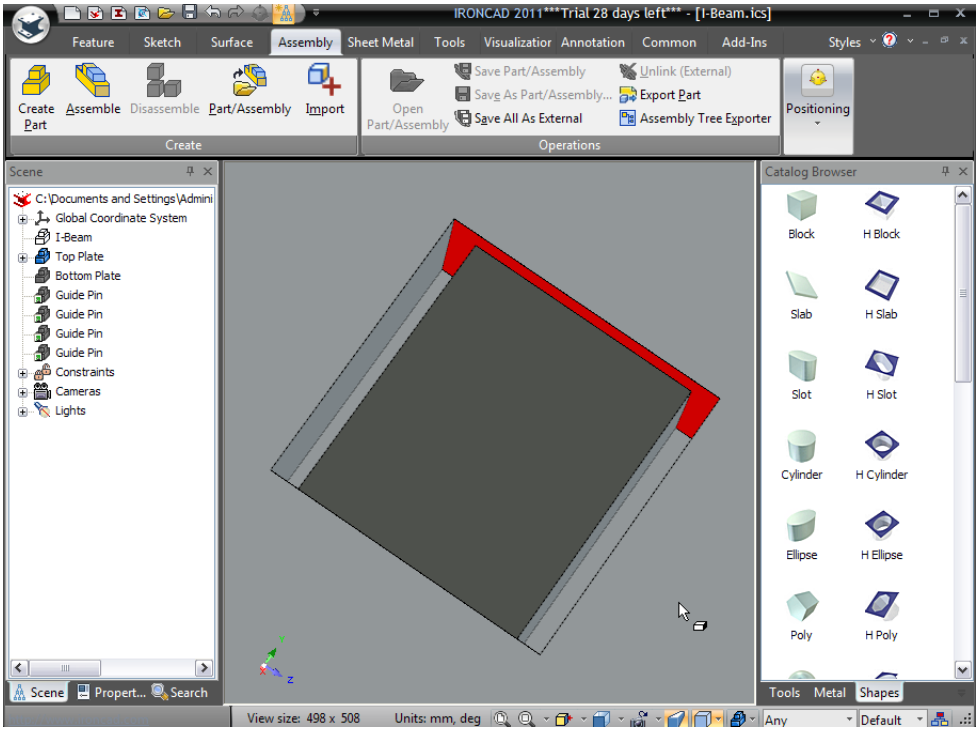
In the real world, the cavities produced by the Boolean operations would need a draft, or release, angle to be added to the side walls defining the molded component. IRONCAD can apply draft at many stages of the design. This example assumes the tooling proposal has been agreed and therefore draft gets added to the cavity as a final operation.

66. Hide UnSelected Parts

Select the Top Plate, and right-click on the mouse. Select Hide Unselected, IRONCAD will remove all the other components from the screen display.

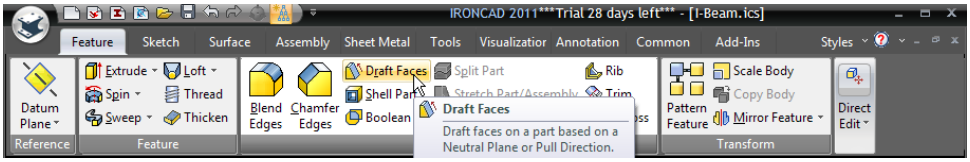
67. Zoom in to the Cavity

Zoom into the cavity using the camera tools. Notice that some of the surfaces will have inherited the color of the I-Beam when it was removed from the plate using the Boolean operation earlier.



68. Select Draft Faces

Switch to the Feature tab on the Windows Ribbon bar and select the option, Draft Faces.



69. Set the Draft Options and Neutral Plane

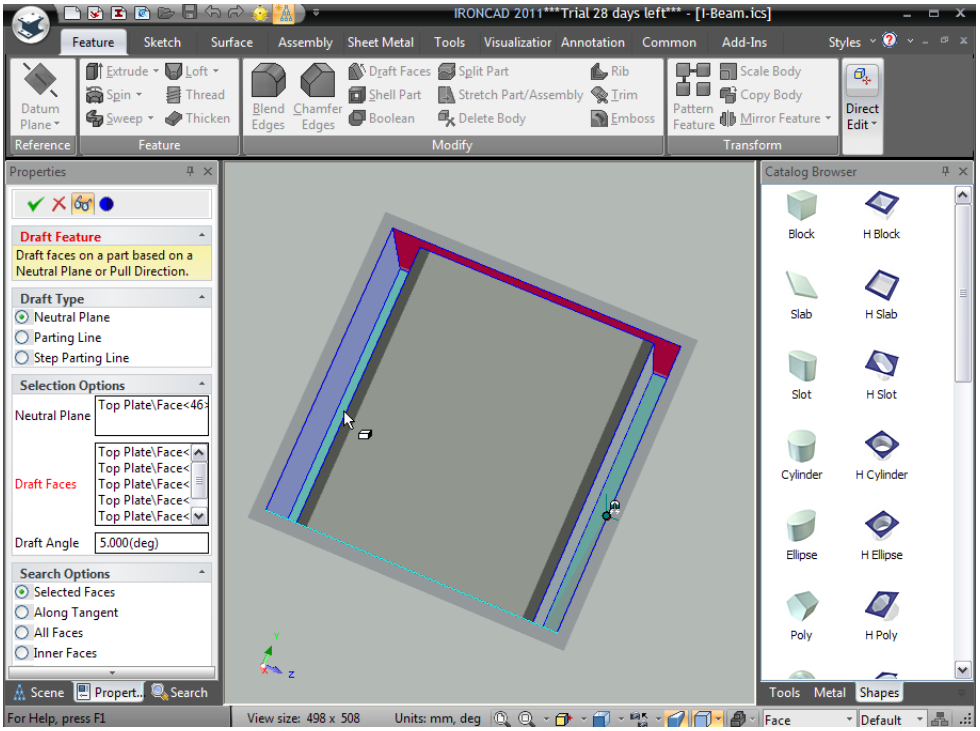
IRONCAD will change the Property Browser to the Draft Feature panel. Set Draft Type to Neutral Plane and select the face at the bottom of the pocket as shown in the next image.

70. Verify the Direction of Pull

Zoom out and check that the cyan arrow is pointing to the open end of the pocket. This arrow determines the direction of the draft angle so the final shape can withdraw from the tool.

71. Select the Faces to Draft

Zoom back in and select all the faces in the direction of the arrow. IRONCAD will show the extent of the draft as a preview.



72. Specify the Draft Angle

Set the Draft Angle to 2 degrees and click the OK checkmark. To change the draft angle once the operation is completed, click onto one of the drafted faces until it highlights in a pale yellow (all other faces drafted as part of that operation will also highlight). Right-click the mouse and select Edit Option.

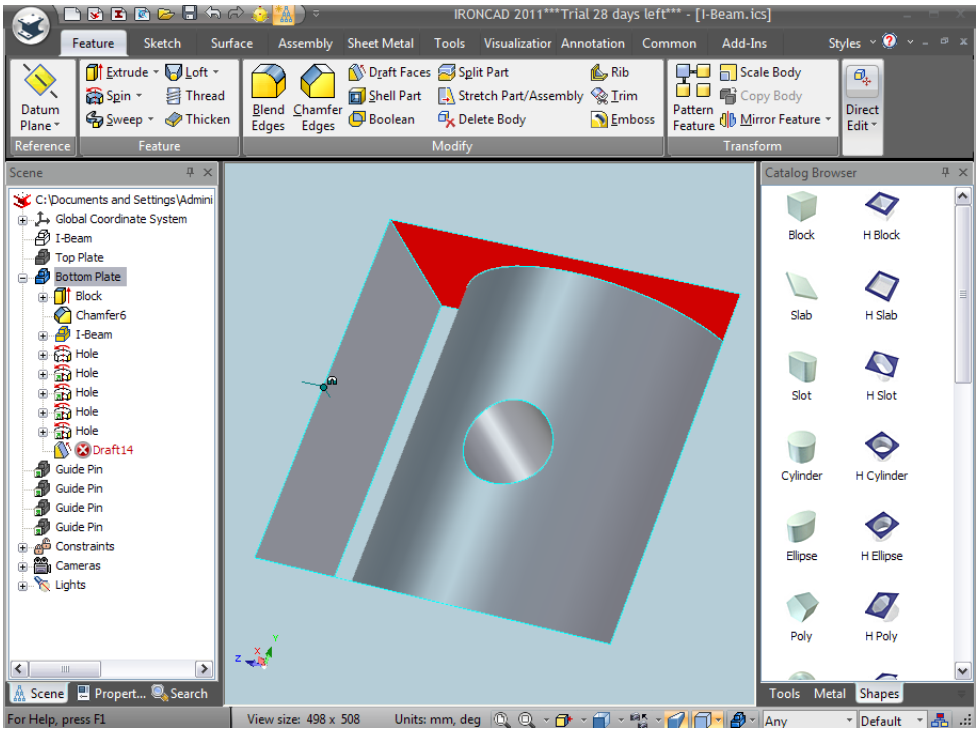
3.4.9. Handling Errors in Feature Creation

It is inevitable that at some point in a design, a desired feature cannot be created for some reason such as trying to put a 20mm blend around a 1mm wide feature or adding draft to a cylindrical face.

73. Repeat the Draft on Bottom Plate

Repeat the process on the Bottom Plate and include the curved surface. IRONCAD will give you an error as drafting a curved surface is not possible in this direction. Accept the error message and expand the feature tree in the scene browser.

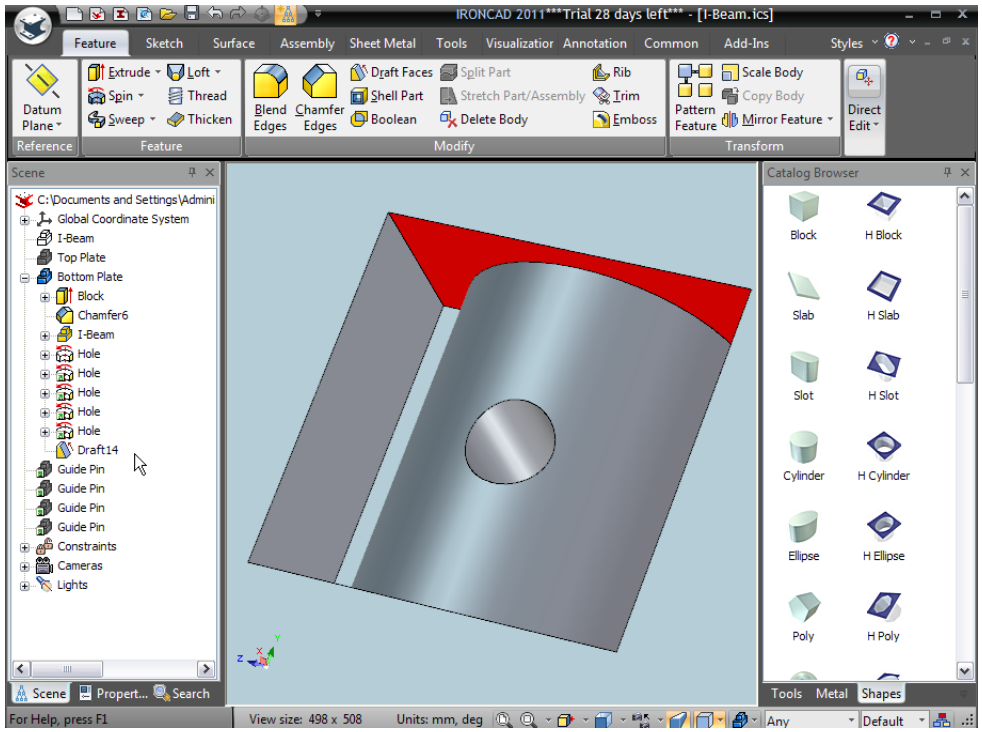
IRONCAD will clearly identify the feature with the error by changing its name to red and drawing a cross by the side of the name.



74. Correcting the Error

To clear the error, right-click on the entry in the scene browser and select Edit Option from the resulting menu. In the Draft Faces panel, select the curved surface, right-click and select Delete. Finish the option by selecting OK.

The error will have been resolved.



75. Display All the Parts

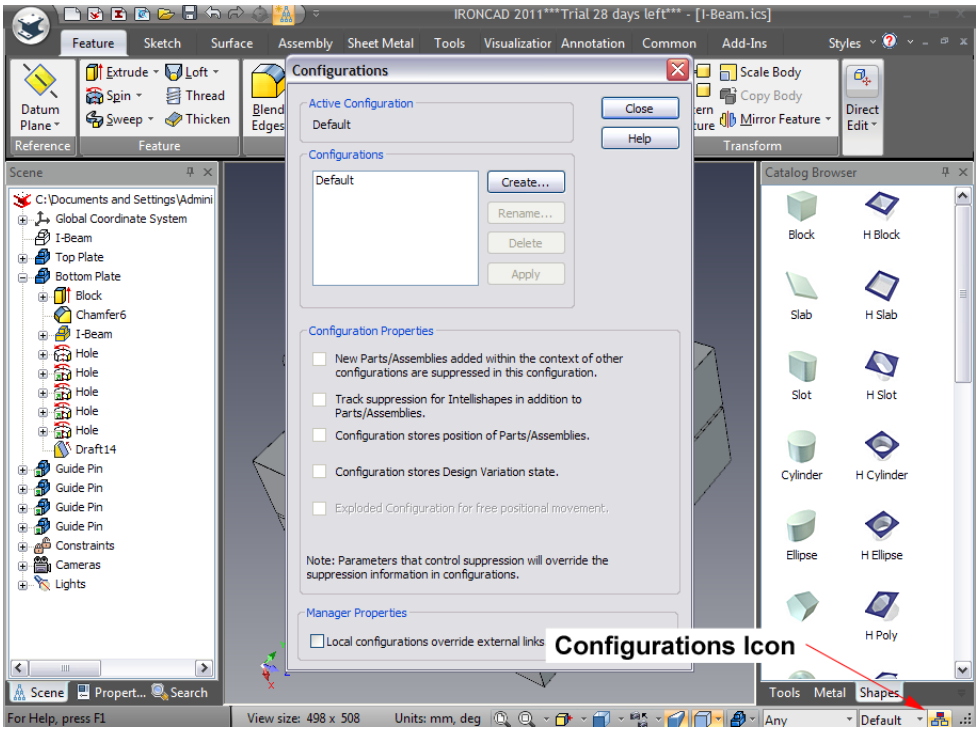
Right-click somewhere over the scene background and select Show All.

3.4.10. Creating Configurations of Different States

A configuration is a state in the design where positions and suppression of items can be remembered by IRONCAD for say an exploded assembly. Configurations can remember positions, display states, and even positions of IntelliShapes allowing the communication and evaluation of ‘what if’ scenarios. In this case the objective of this exercise is to create an exploded assembly.

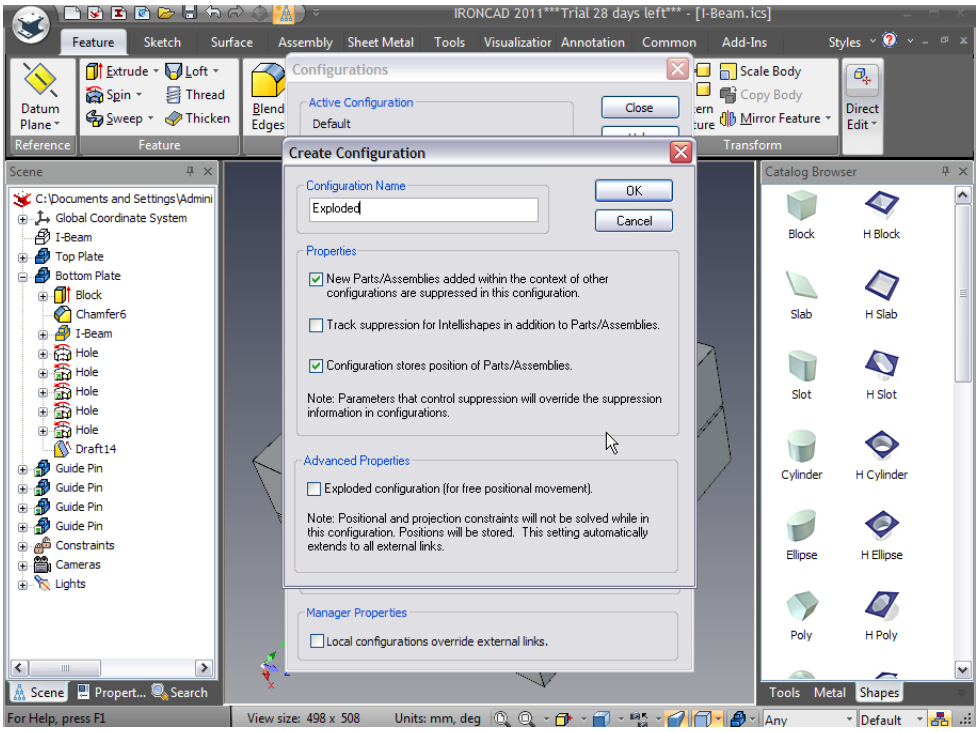
76. Enable the Configuration Dialog

On the bottom right of the IRONCAD window, click on the Configuration icon. IRONCAD will display the Configurations panel.



77. Create a New Configuration

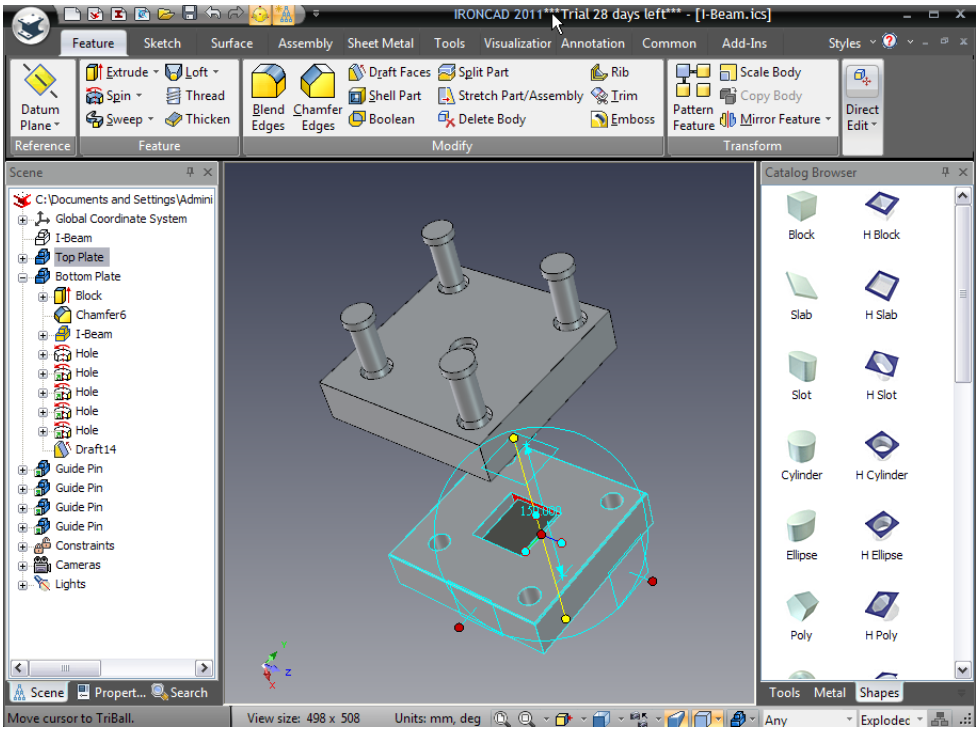
Click on the Create button. In the subsequent panel, enter Exploded for the Configuration Name and check ‘New Parts/Assemblies added. . . .’ and select Configuration stores position of Parts/Assemblies. Click OK to complete the definition of the configuration.



IRONCAD will update the Configurations panel with the new configuration setting. It will also change the name of the active configuration on the status bar by the configurations icon.

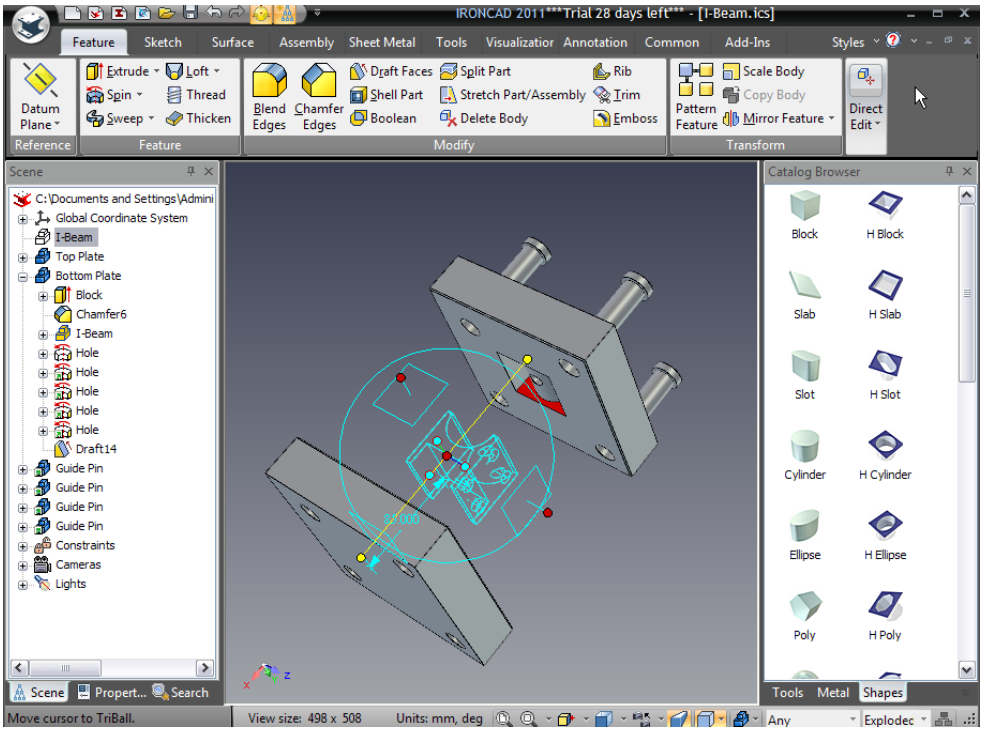
78. Reposition the Top Plate

Select the Top Plate at part mode (highlighted with cyan edges) and activate the TriBall. Move the Top Plate away from the Bottom Plate by 150mm as shown.



79. Move the Suppressed I-Beam

Deactivate the TriBall and select the suppressed I-Beam part on the scene browser. Re-activate the TriBall and move the suppressed I-Beam away from the Bottom Plate by 85mm.



80. Save the New Exploded File

Deselect the TriBall and re-orientate the view as shown above and save the scene.

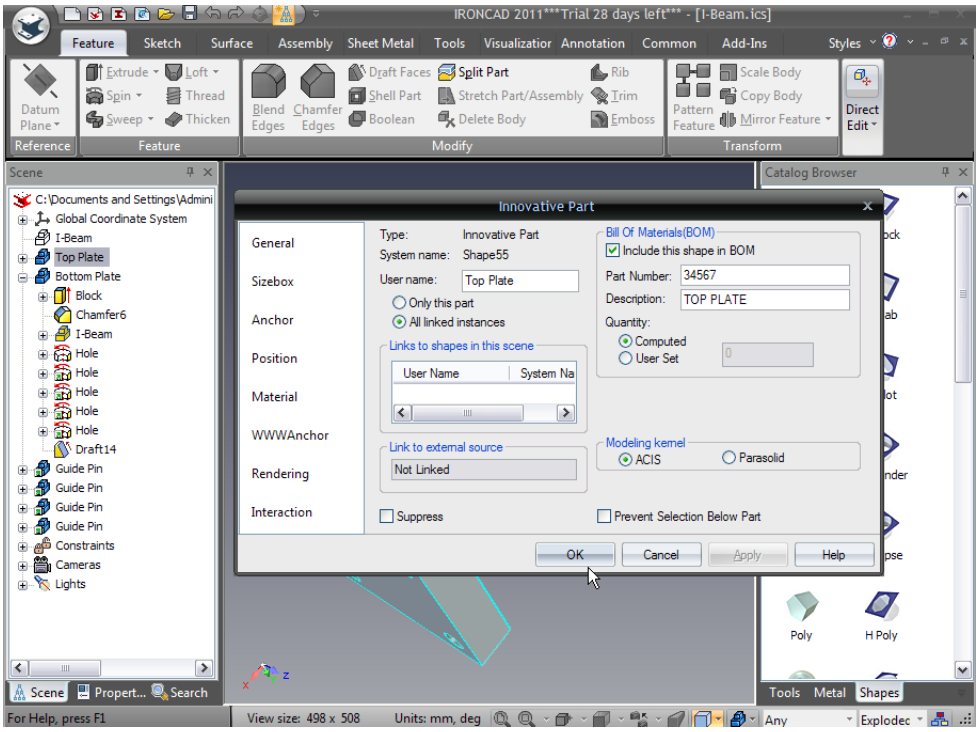
Now the scene will contain a configuration (default) as the assembled and an exploded configuration that can be toggled.

3.4.11. Creating Exploded Assembly Drawings

A common task in the design environment is for the provision of Bill of Materials data and exploded assembly drawings. IRONCAD provides links directly between the drawing and the 3D part for each component in the assembly.

81. Fill in Properties of Top Plate

Right-click on the Top Plate in the scene browser and select properties. Fill in the Part Number and Description boxes e.g. 34567 and TOP PLATE, and click OK. Note: You can also access this in the Property Browser



82. Repeat for Bottom Plate

Repeat this for the Bottom Plate using a different number.

83. Apply Properties to a Guide Pin

Again follow the same process on the Guide Pin using a different number. Notice as the four Guide Pins are linked they all inherit the data entered automatically.

84. Create a New Drawing

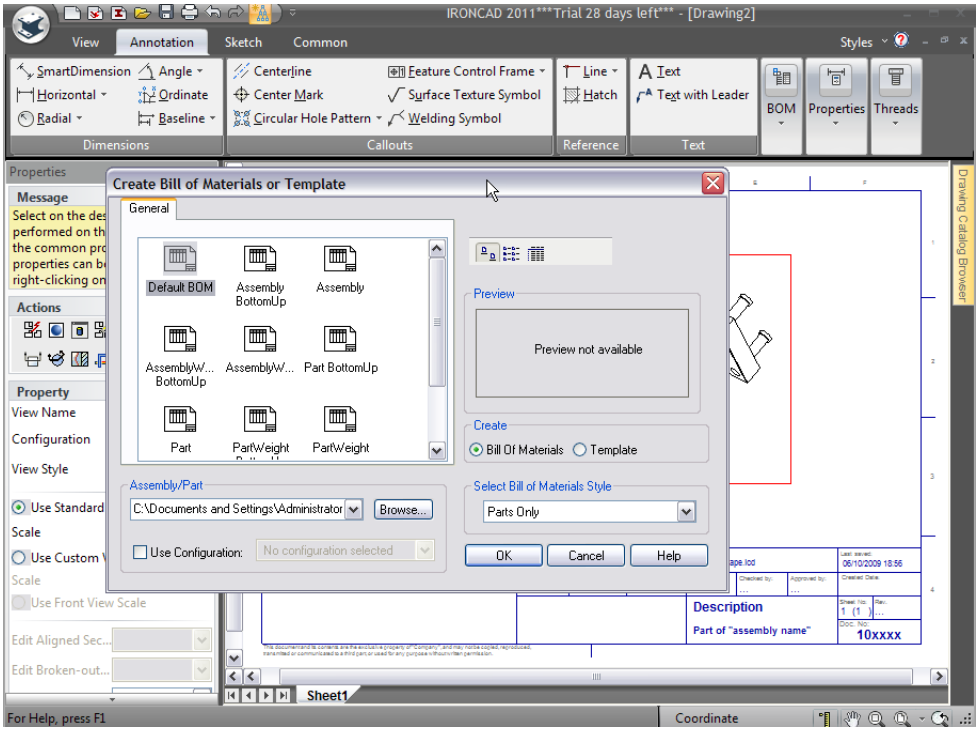
Click on the IRONCAD home button and create an new drawing (File > New > Drawing) using the A4 Landscape template used previously.

85. Create a General View

Select the View tab followed by the General View option. On the General View Creation panel, ensure the Configuration says Exploded and then click on the From Scene option and click OK. Place the drawing view on the drawing sheet.

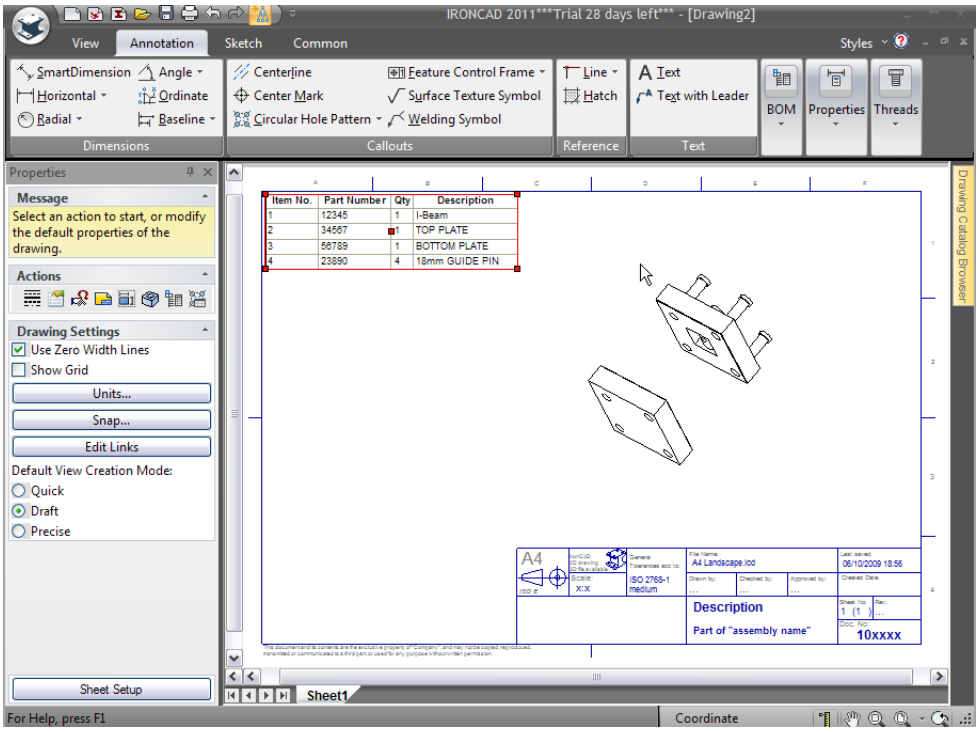
86. Create a Bill of Material

Open the Annotation tab and click the BOM (Bill of Materials) icon. IRONCAD will display the Create Bill of Materials panel. Select Default BOM and click OK.



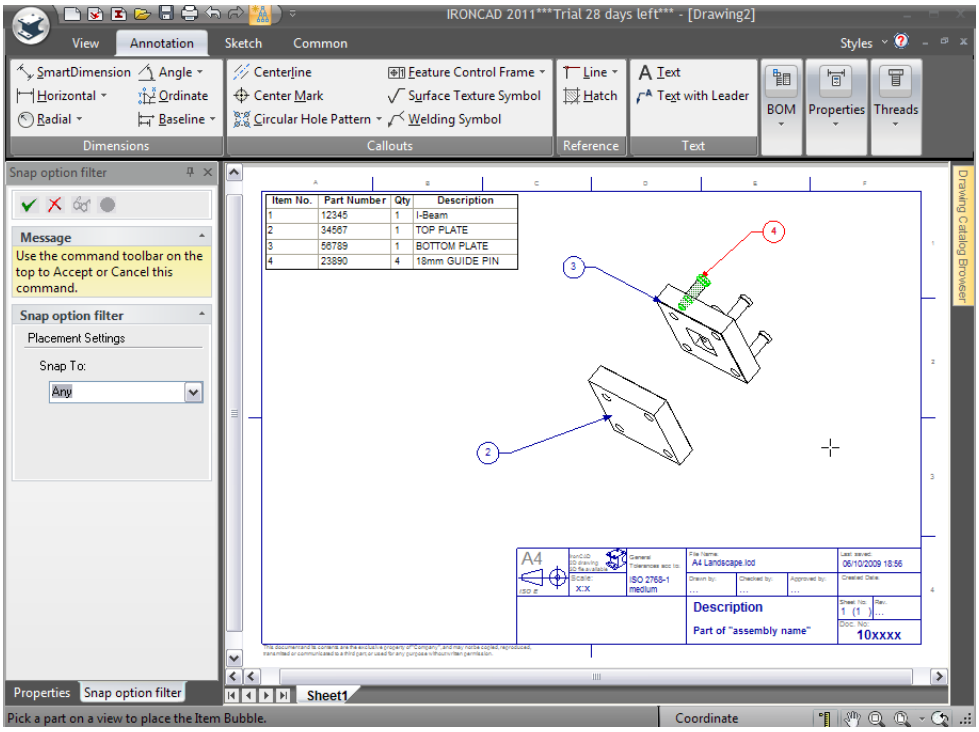
87. Place the BOM Table

IRONCAD will create the Bill of Materials and display a red bounding box for the table. Drag the BOM to the desired position and click to place.



88. Create Item Bubble Callouts

Once the BOM has been created, the drawing can be annotated with Item Bubbles. Select Item Bubble from the BOM panel. Click on the components of the drawing and position the Item Bubble. IRONCAD will give graphics feedback of the item selected in the case several items are very close to each other. The numbers of the Item Bubbles match the numbers in the BOM.

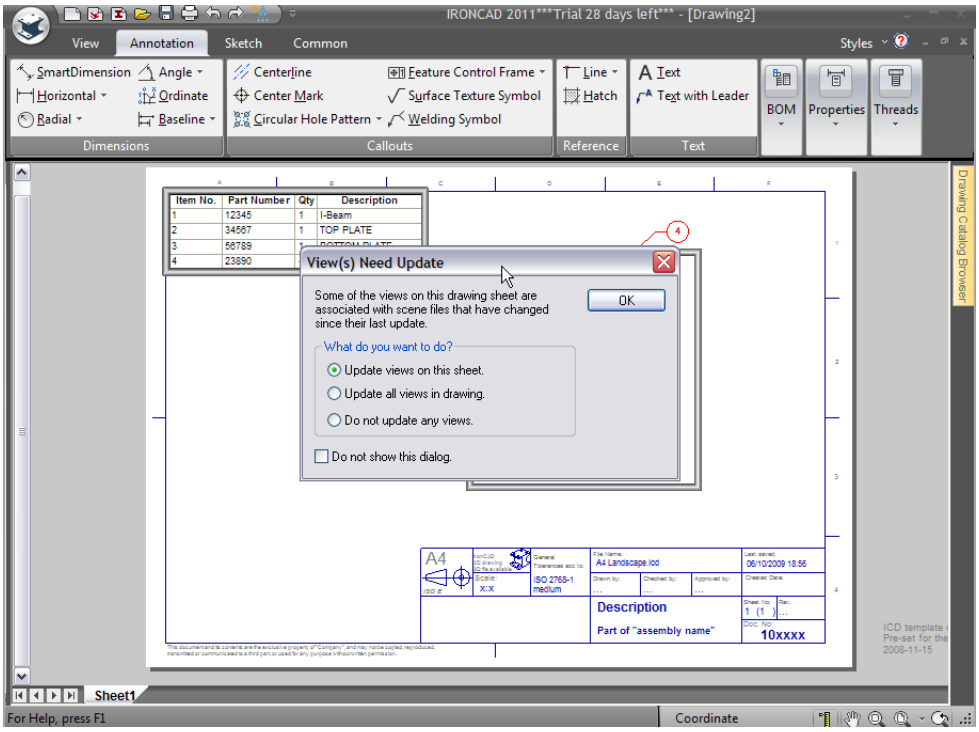


89. Un-Suppress the I-Beam

Minimize the drawing and in the scene, right-click on the I-Beam and un-suppress it by clicking on the Suppress option. At the same time change the part description of the I-Beam to MOULDED I-BEAM.

90. Return to the Drawing

Restore the drawing window (from the Common Tab or Quick Access Toolbar). IRONCAD will recognize a change has happened to the defining scene, draw notification boxes around any views and the BOM, and display the View(s) Need Update panel. Click OK.



91. Automatic Updates

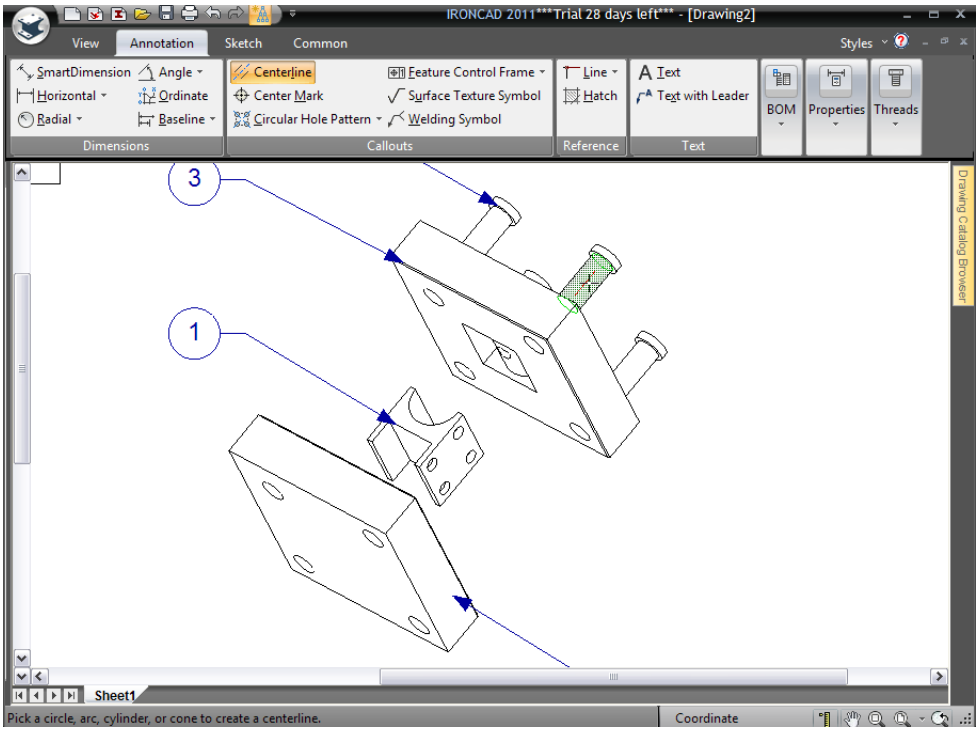
The BOM will be updated with the new description for the I-Beam and the drawing updated to show the I-Beam.

92. Add an Item Bubble for the I-Beam

Add the Item Bubble to the I-Beam and if needed, change the position of the Item Bubbles by selecting on the bubble itself and dragging it to a more pleasing position. To reposition the other end of the Item Bubble, select it at the arrow location and drag it to a new location.

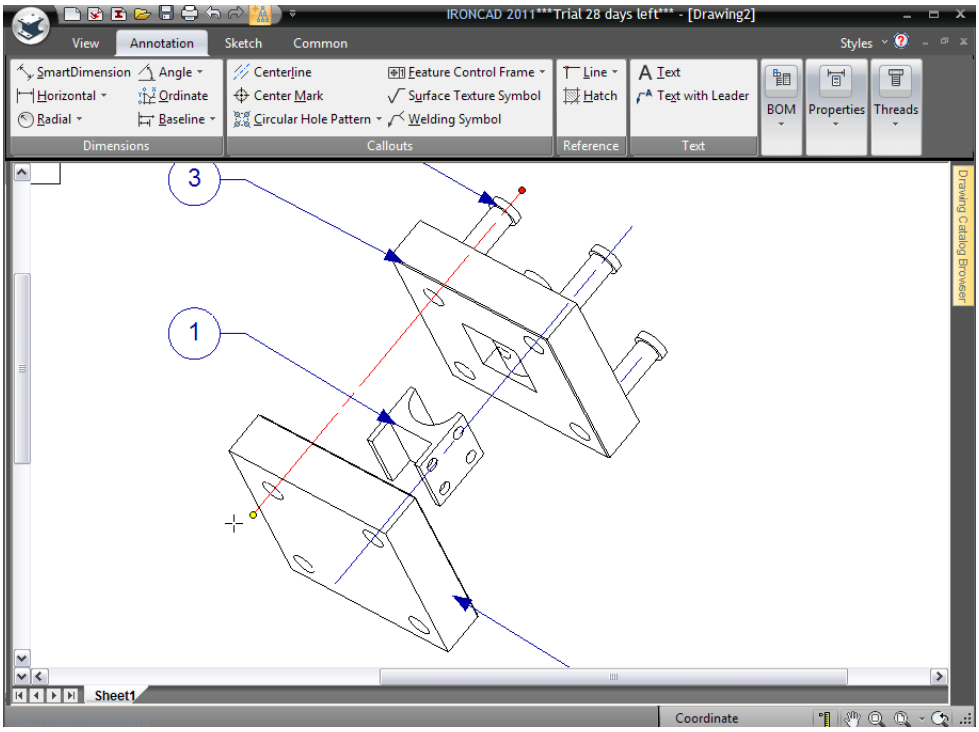
93. Add Center Lines for Guide Alignment

Add center lines to show the alignment of the Guide Pins through the assembly by selecting the Center line option and indicating the cylindrical portion of the Guide Pin (zooming in might help this operation).



94. Extend the Center Lines

IRONCAD will draw a center line on the selected portion of the component. To extend it through the assembly, click on the drawn center line and drag the red end sphere to the desired position.



95. Update the Title Block

Right-click on the title and the drawing number and replace the default text with an appropriate title and number using the Edit Text option (Note: Remember the Layer may be locked).

96. Show True Width Lines

The A4 Landscape template is set up to use very thin lines to help with dimension positioning and selection. Turn this option off by right-clicking anywhere on free drawing area (not within a projected view area) and select Sheet Setup. Un-check the option 'Use zero-width lines'.

97. Save the Drawing

Save the drawing giving it a new name such as I-Beam tool arrangement (by default IRONCAD assigns the drawing the name of its controlling 3D scene. At the same time IRONCAD will save the defining scene (without changing its name).

This completes the section of the Quick Start Guide. The key topics covered in this section include:

- Designing an Assembly
- Working With Boolean Operations
- Using Intelligent Feature
- Using Intelligent Features as a Basis for New Features
- Creating Sketch Based Features
- Working with Blends/Chamfers
- Positioning with Constraints
- Adding Draft
- Handling Errors In Feature Creation
- Creating Configurations of Different States
- Creating Exploded Assembly Drawings

This concludes our first quick pass introduction to IRONCAD. Hopefully this Quick Start Guide has provided you with an overall first introduction to the design process in IRONCAD as well as some of the fundamental features of the application. This Chapter focused on:

- Creating a concept 3D model
- Re-sizing the features to reflect desired dimensions
- Adding a few detail features
- Creating a photorealistic rendering
- Adding chamfers and blends
- Creating a detail drawing
- Building an assembly and detailing it.

Feel free to expand on your tooling part to gain a better understanding of the IRONCAD design process. In the next chapters, we will expand on some of the concepts covered in this section to give you a greater depth of the capabilities.

4. USING 2D SKETCHES IN PART DESIGN

IRONCAD models are not restricted to being built from the range of standard IntelliShape building blocks provided by the IRONCAD Catalog system. Sometimes it may be more convenient to create a shape from a 2D cross section, or sketch as it is commonly known. In fact all of the 3D shapes provided in the catalogs are created from 2D sketches and can be used as a basis for other features or even complete parts.

Custom shapes created from 2D sketches operate in a very similar way to standard IntelliShapes in that they can be modified by stretching handles. However by default, these handles apply to the individual elements, or spans, of the defining geometry.

This chapter 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 part design. This chapter will guide you through the process to create a simple assembly of parts. 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 chapter include:

- Shape Handles vs. Sizebox Handles
- Using the Basic 2D Editing tools
- Creating precise sketches using Endpoint Dimensions, Curve Dimensions, and Curve Handles
- Editing Cross-Sections of Existing IntelliShapes
- 2D Constraints
- Drag-and-Drop IntelliShape Modeling vs. 2D Cross-section Modeling
- Setting Part and IntelliShape Names and Colors
- Using the Advance Shapes Catalog
- Saving the Part
- Creating a Crank Shaft
- Adding the Shaft
- Creating the Crank Arm
- Parametric Constraints with Expression
- Creating an Assembly
- TriBall and Linked Components
- Adding Intelligence Using Parameters
- Design Variations

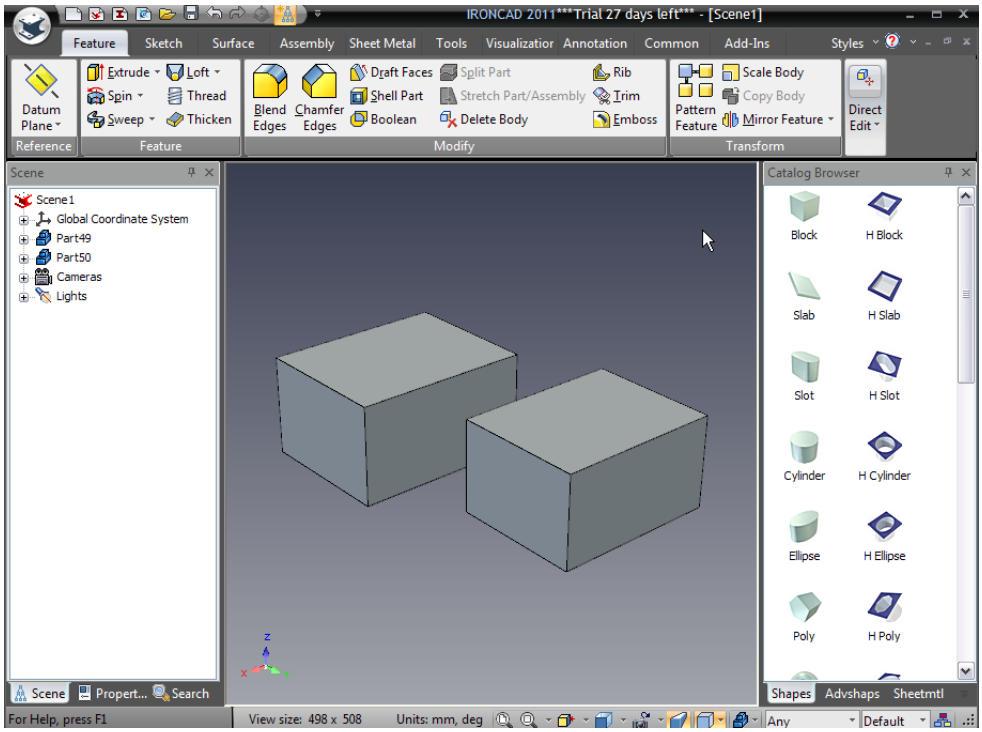
4.1. Shape Handles vs. Sizebox Handles

Previously, IntelliShapes were used to create a simple model. Any changes to the geometry were accomplished as a result of stretching the handles controlling the Sizebox of the IntelliShape or by the use of SmartDimensions.

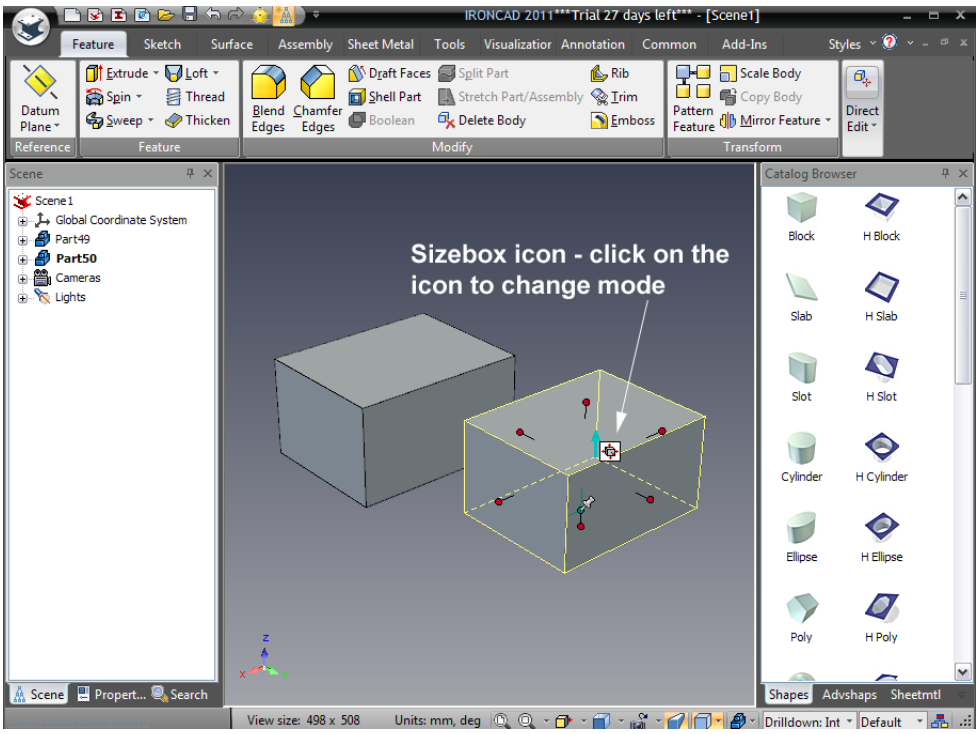
IRONCAD does not limit its dynamic modeling to Sizebox stretching. The defining sketch of the IntelliShape can also be pushed, pulled, and accurately positioned relative to other elements of the scene removing the need to use controlling sketch dimensions. Handles displayed on the sketch spans are known as Shape Handles. They differ in shape (i.e. red triangles) and also add greater positioning control giving complete control over the design without adding superfluous clutter around the design. They are also a fundamental part of designing within the context of the assembly.

4.1.1. Positioning of Features and Faces using Shape Handles

Faces of models can be positioned and sized using handles that IRONCAD attaches to the individual geometry spans defining the feature. To explore this powerful set of functionality, drag 2 blocks from the catalog as shown in the image below:

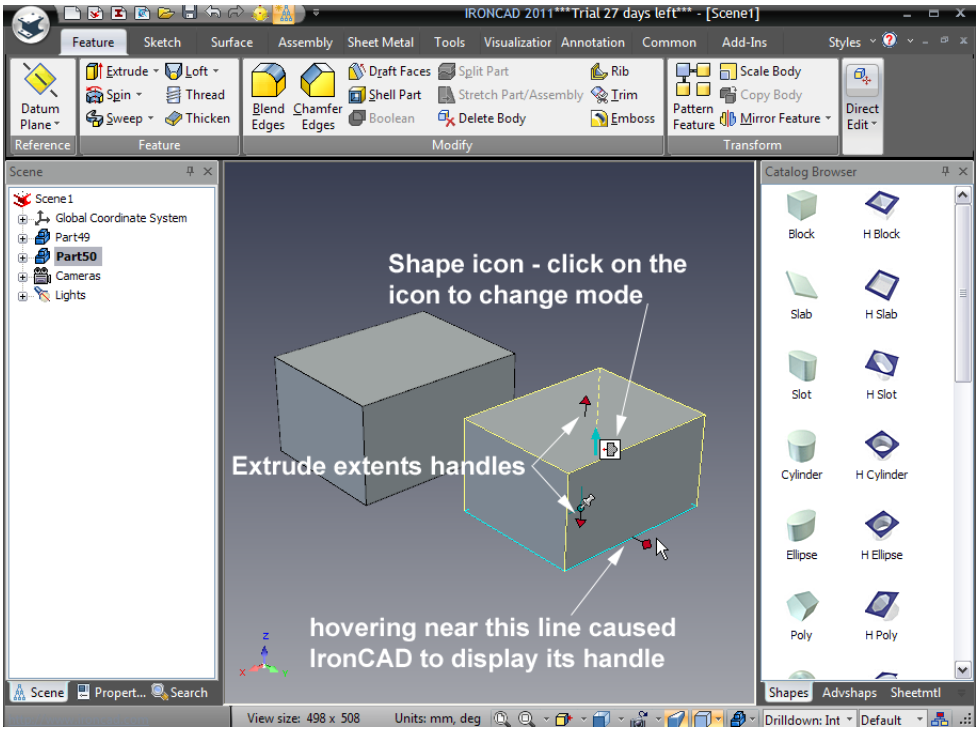


For this exercise it doesn't matter if these blocks form a single part or 2 separate parts. Drill down to display the IntelliShape Sizebox handles on one of the parts. You will notice an icon by the side of the Sizebox (located near the extrusion arrow direction). This icon indicates the current 'mode' the IntelliShape is currently active in. By clicking this icon, the 'mode' is changed from 'Sizebox mode' to 'Shape mode'.

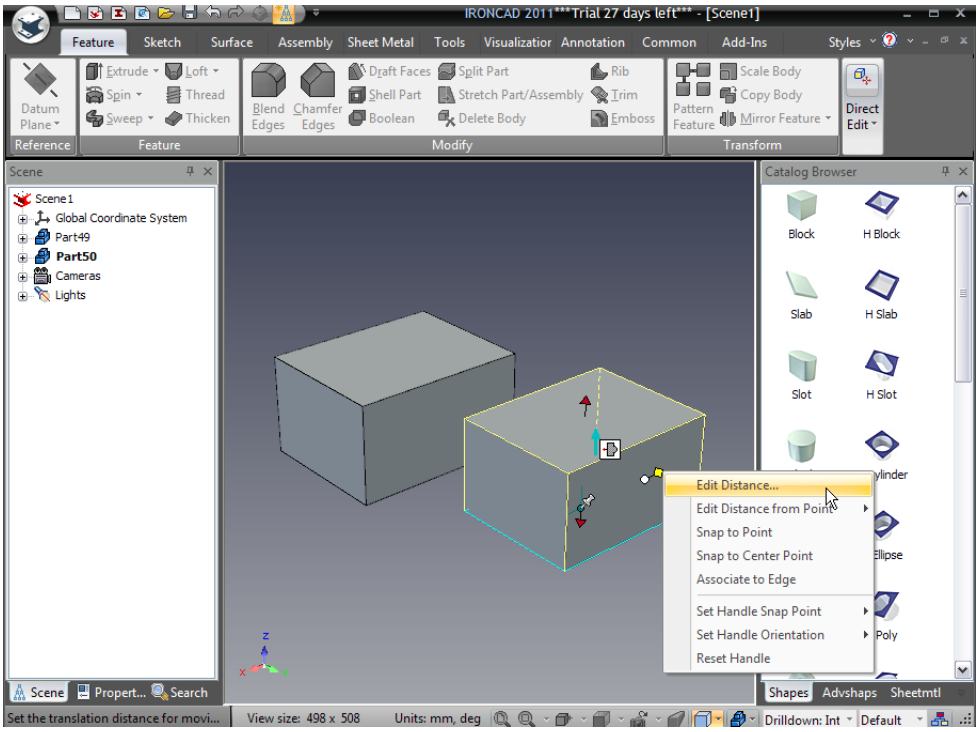


With the part highlighted in Shape mode, the familiar Sizebox handles will disappear and be replaced by two triangular handles defining the extrude direction and extents. In addition, the defining sketch will be displayed in cyan.

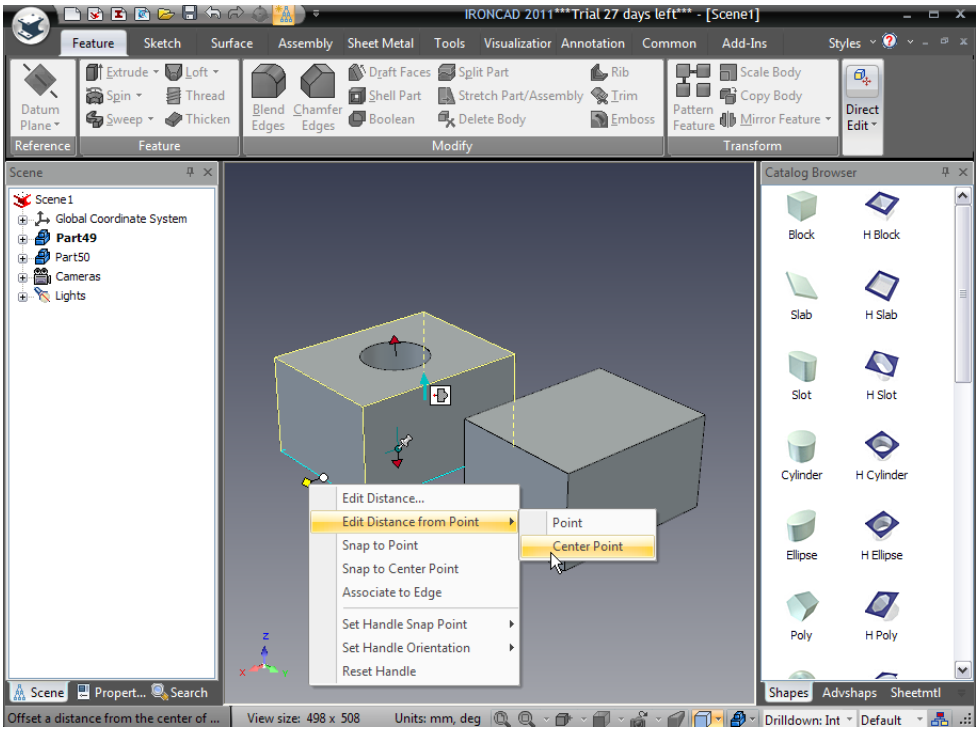
Moving the mouse pointer over one of the cyan spans results in IRONCAD displaying diamond shaped handle that you can click on a drag. This results in shape changes similar to those when dragging the Sizebox handles. However, right-clicking on a diamond shaped handle results in a pop-up menu giving you fine positional control of that geometry span (and hence the face).



For example: Select the first option ‘Edit Distance’. IRONCAD displays a dialog box for inputting a value. Typing a positive value will result in the face moving in the direction of the handle. A negative value results in the face moving in the opposite direction.



Now drop an H Cylinder shape from the catalog onto one of the existing IntelliShapes. Select one of the original block IntelliShapes and click on the mode icon to activate shape mode. Now, as before, move the mouse pointer over a geometry span and right-click the diamond shaped handle. This time select ‘Edit Distance from Point’. IRONCAD will display a 2 option drop down menu. Select Center Point and select the center of the hole just added (or the edge). Enter a value to position the face relative to the hole.



Repeat this operation for the other part in the scene. This shows 2 key design and productivity features of IRONCAD:

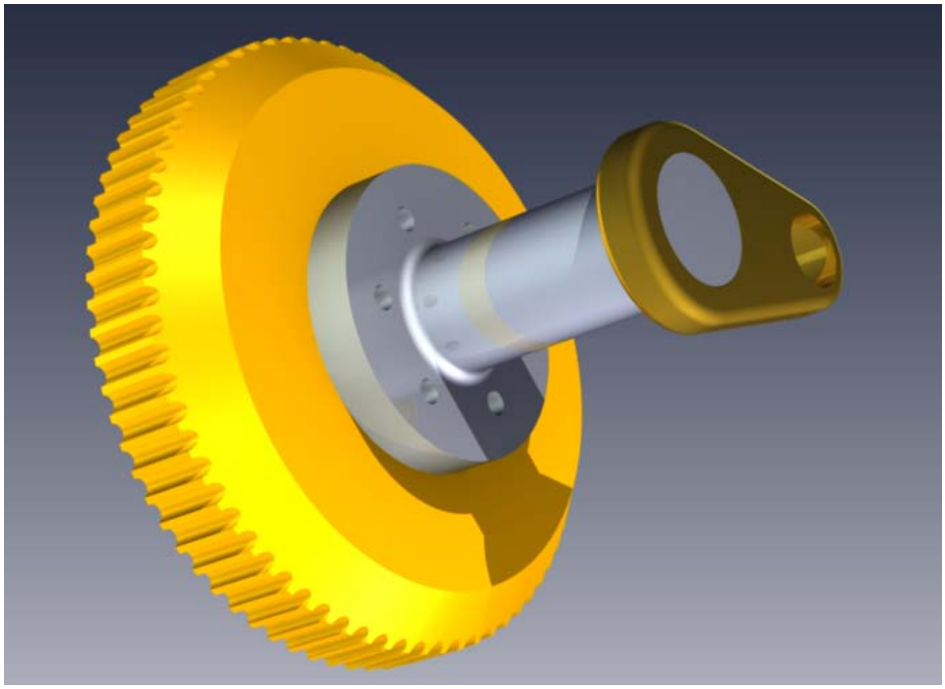
A. IRONCAD is not bound by the order in which the geometry is created. In the above example, the hole was created after the block yet its geometry was available to help position a face on the holes parent block.
B. Geometry can be sized directly and dynamically without introducing more data and functions to the design process.

It is worth experimenting more with this mode changing for sizing and positioning. One quick tip, if you use shape mode for the H Cylinder you can set its radius by Edit Distance from Point and then choose Center Point which will indicate the center of the H Cylinder as the reference.

4.2. Using the Basic 2D Editing tools

This section will introduce a more traditional approach to part design (i.e. by sketching geometry that reflects the final form of an item). The sketch will then be 'sized' so that its meets a set of dimensional criteria.

Sketch geometry can also be controlled so that any changes are predictable in nature. This approach is referred to as constraining the geometry. We will also introduce the concept of constraints and then show that IRONCAD freely mixes sketch based features with drag and drop based features. The component will then be used as a reference to create additional components ending up with the assembly shown below.



4.2.1. Creating the Flywheel using a Spin IntelliShape

A spin IntelliShape is a 3D shape created by revolving (spinning) a 2D sketch profile (cross-section) about an axis. The sketch profile must consist of one or more closed loops of connected geometry elements with no overlaps. Geometry that crosses the spin axis will cause an error.

The following section will introduce both the generation of a 2D sketch and the generation of a Spin IntelliShape. To help ensure results are consistent with the steps described within this section and for the values to make sense, begin a new scene and ensure the units match the examples.

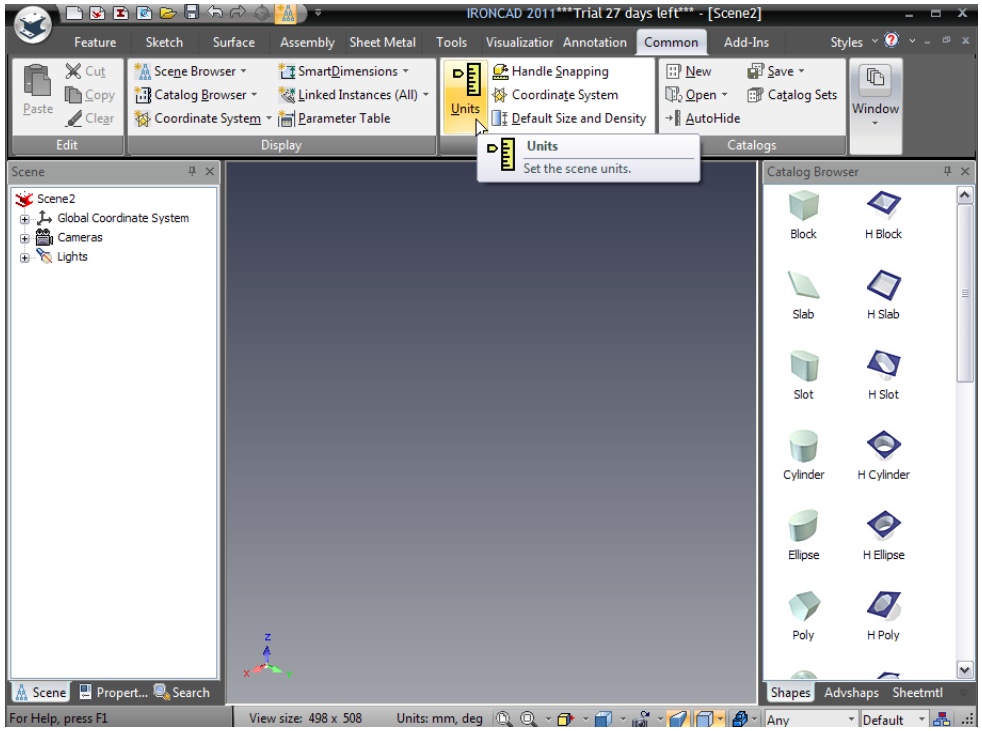
As in previous sections, this step by step guide will gradually reduce the amount of dialog presented as the exercise develops. If you just want to step

through the exercise simply follow the numbered steps. However, you might find information in the dialog to be useful.

1. Create a New Scene

From the IRONCAD 'home' button, select File followed by New. From the resulting dialog box, select Scene and click OK. From the Metric tab, select the template Gray and click OK.

IRONCAD will start a new scene with a fading gray background. Note: by checking the Status Line at the bottom right hand side of the IRONCAD window, you will quickly see the units are set to mm (millimeters).

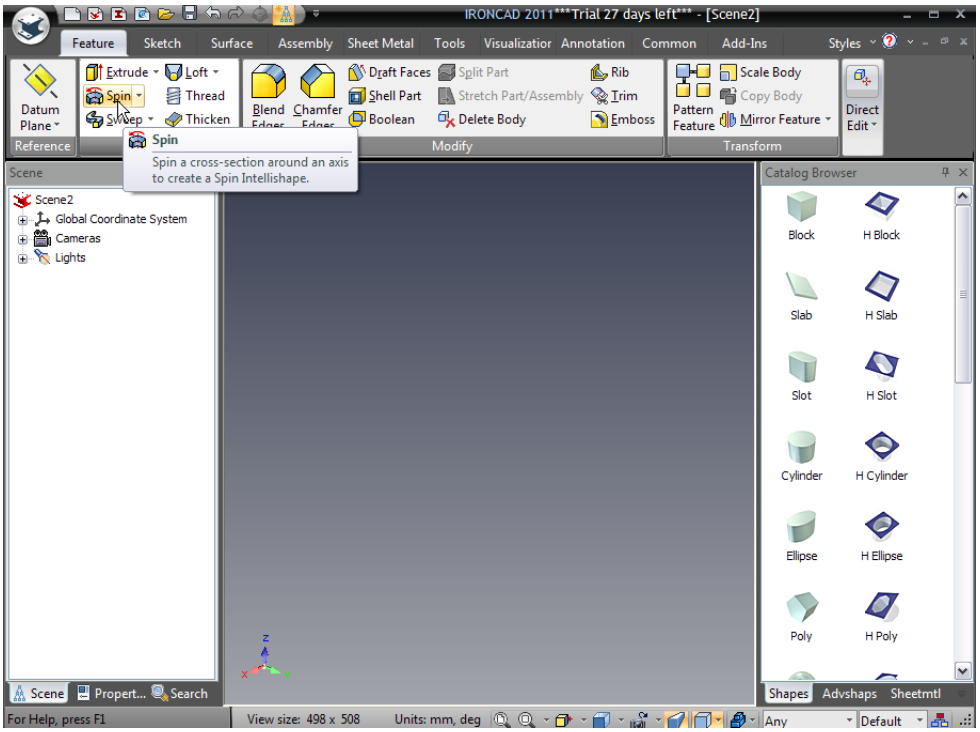


If the current scene units are not millimeters, go to the Common tab and click on the Units icon. Set the Length value displayed in the dialog box to Millimeters via the drop down list.

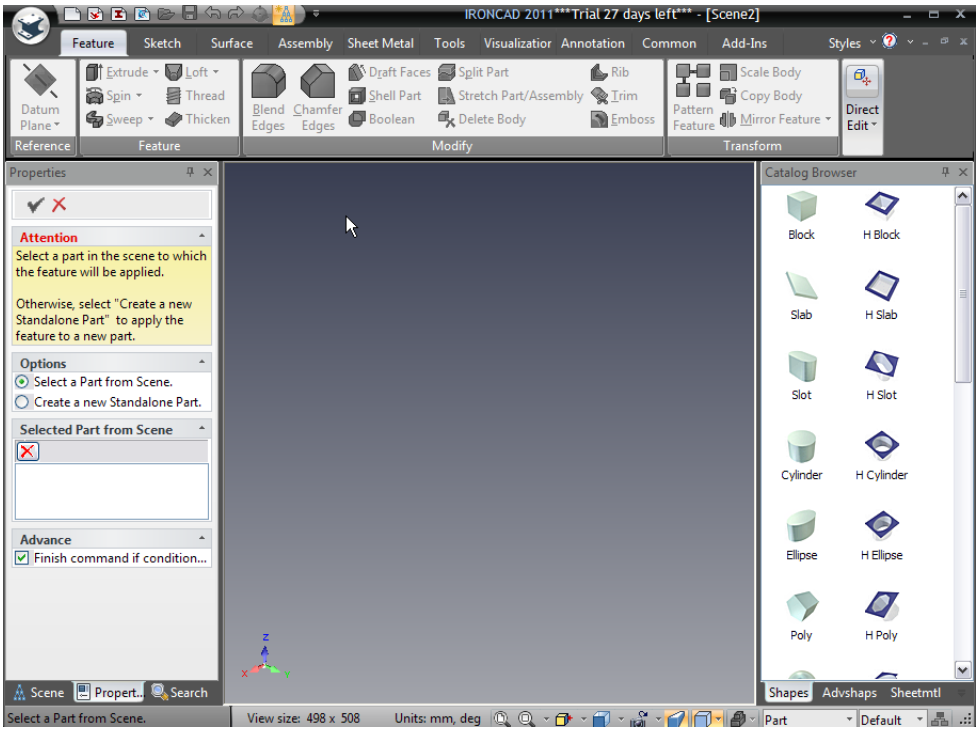
IRONCAD presents two options for creating a new Spin IntelliShape: The classic Spin Shape Wizard and for IRONCAD XG and later a more interactive Spin Feature. This section will introduce the Spin Feature method.

2. Select the Spin Feature Command

From the Feature Tab, select the “Spin” Feature tool.



IRONCAD will display the Command Property Browser on the left hand side of the screen.



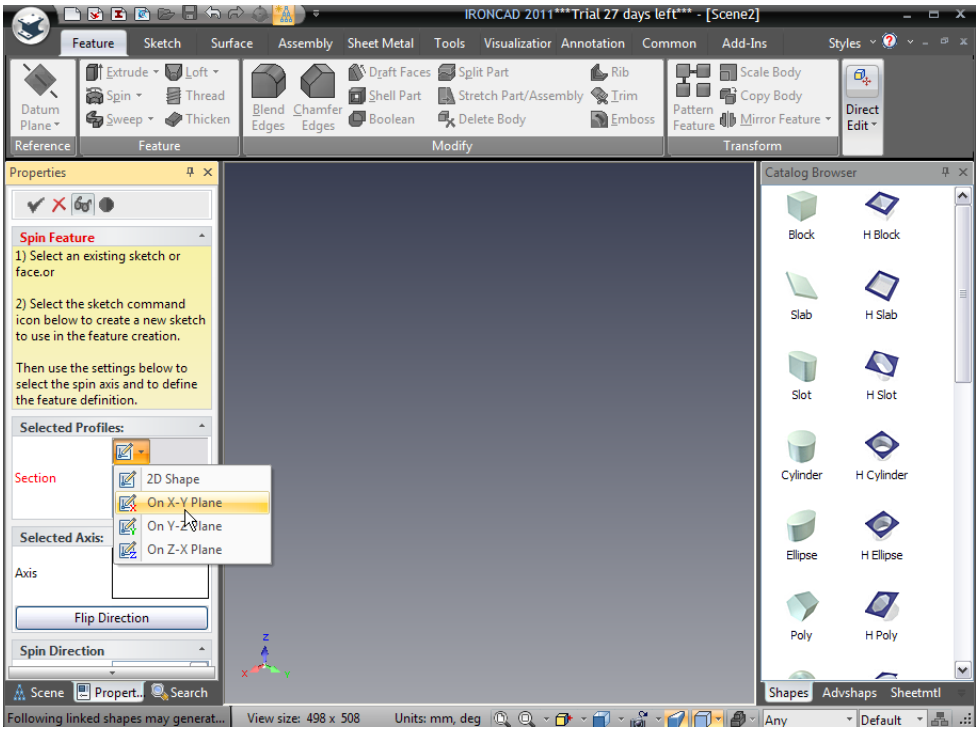
The Command Property Browser is designed to provide a logical step of operations for the creation of a shape. It is consistent in its approach for creation of all IntelliShapes. Furthermore, it presents the same tools to modify the IntelliShape as virtually all aspects of the IntelliShape are 'interchangeable'.

3. Create a New Standalone Part

In the Options panel, select the Create a new Standalone Part.
IRONCAD will then refresh the Command Property Browser to give options to use an existing sketch, face of a part to define the sketch geometry, or to create a new sketch as the basis of the IntelliShape.

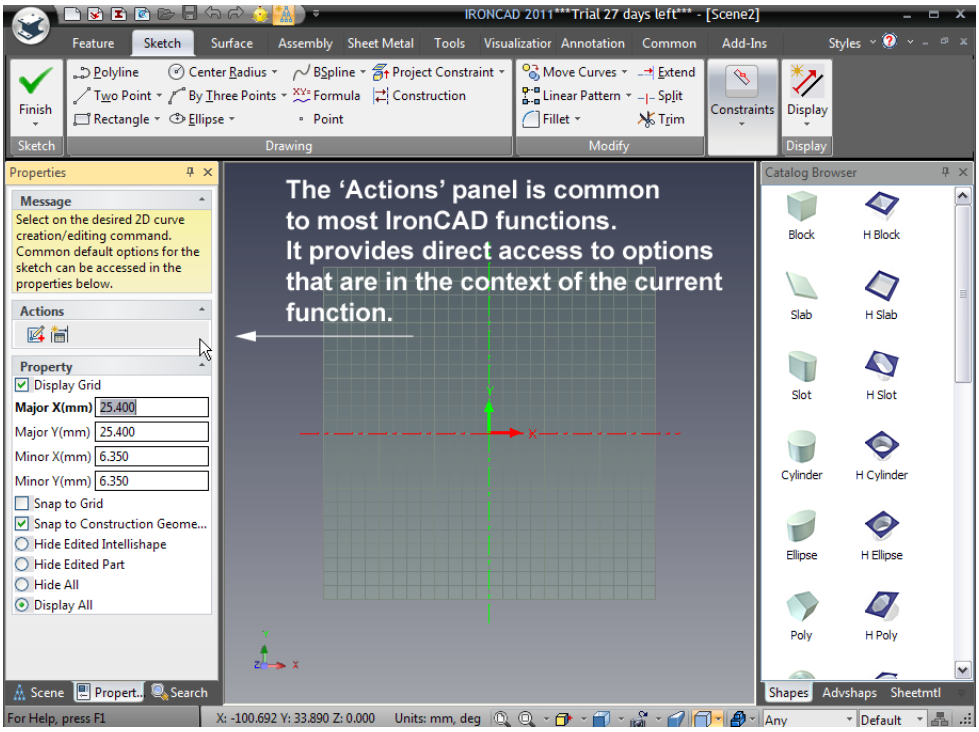
4. Create a New Sketch

In the Selected Profiles panel (this will have the word Section highlighted in red indicating that this box is expecting an input), use the drop down icon as shown and select on X-Y Plane.



IRONCAD will display a sketch grid in the scene window while orientating the view to look orthogonally onto the sketch grid. Additionally, IRONCAD changes the Property Browser to reflect this stage of the sketch process.

Here you can set the values for the grid spacing as well as options such as ‘snap to grid’ and a range of display options in the event you are working on a part that makes it tricky to see the geometry being created.

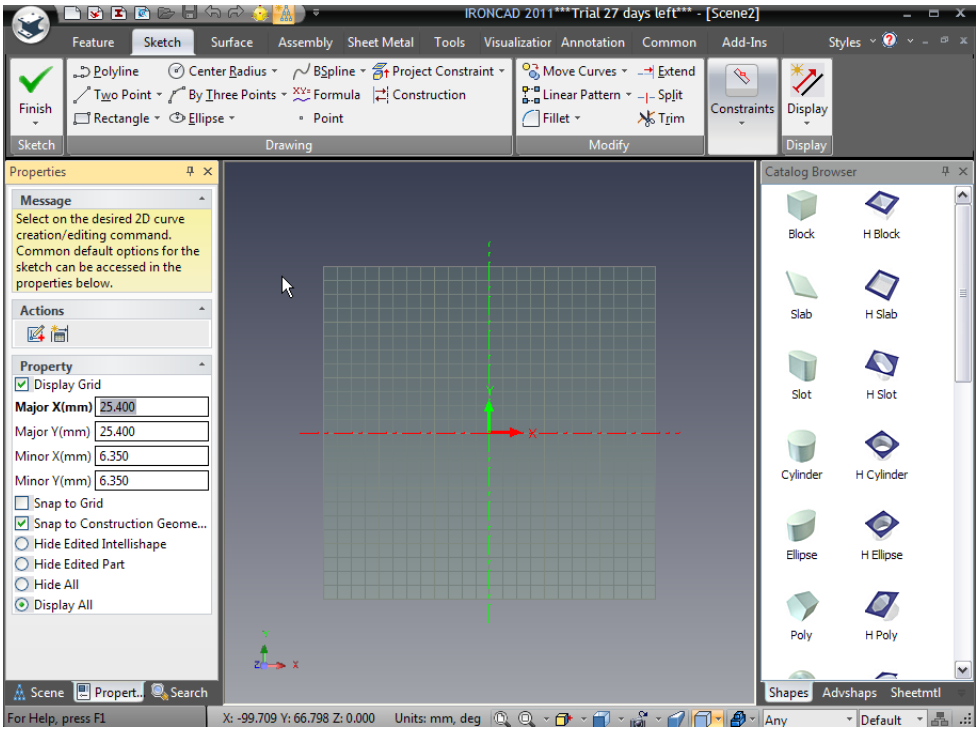


IRONCAD will also display two icons within the Actions section. These icons give direct access to context sensitive operations. In this case, they provide quick access to the import of an existing 2D piece of geometry in say DXF format and to the parameter table.

Context based icons increase productivity by reducing the number of interactions required to get the same results.

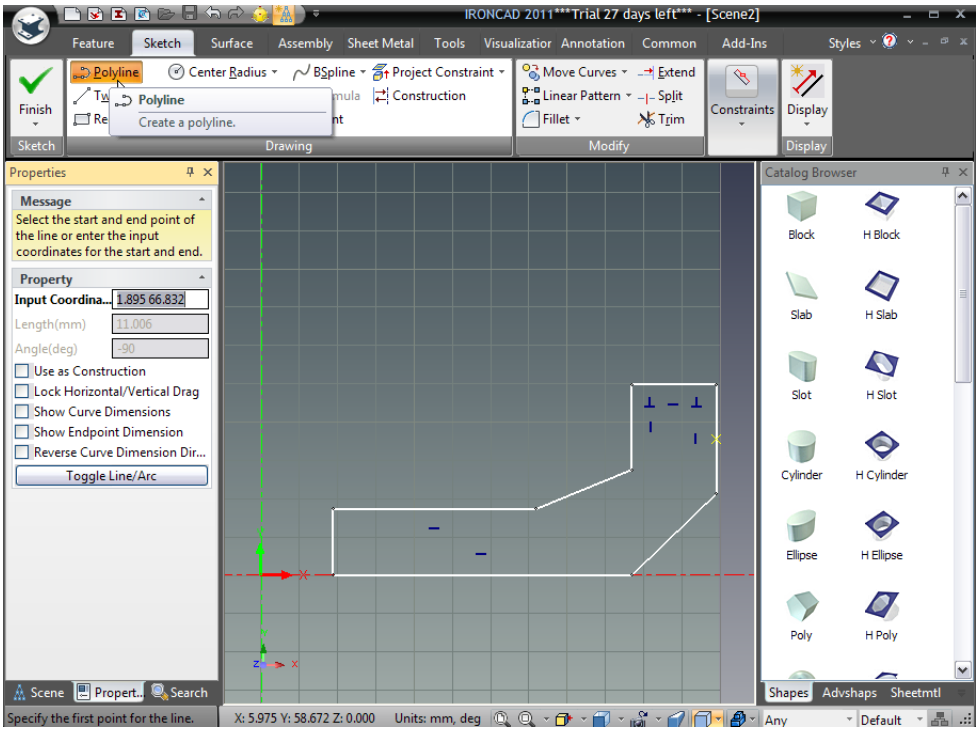
5. Drawing the Spin Profile

IRONCAD will now activate the Sketch tab of the Windows Ribbon bar. IRONCAD will use the Y-Axis as the spin axis. Accordingly, all the sketch geometry should be drawn on the X positive side of the sketch plane. Geometry can touch or co-exist with the Y-Axis but not cross it (creating overlap failures).



6. Create the Shape using a Polyline

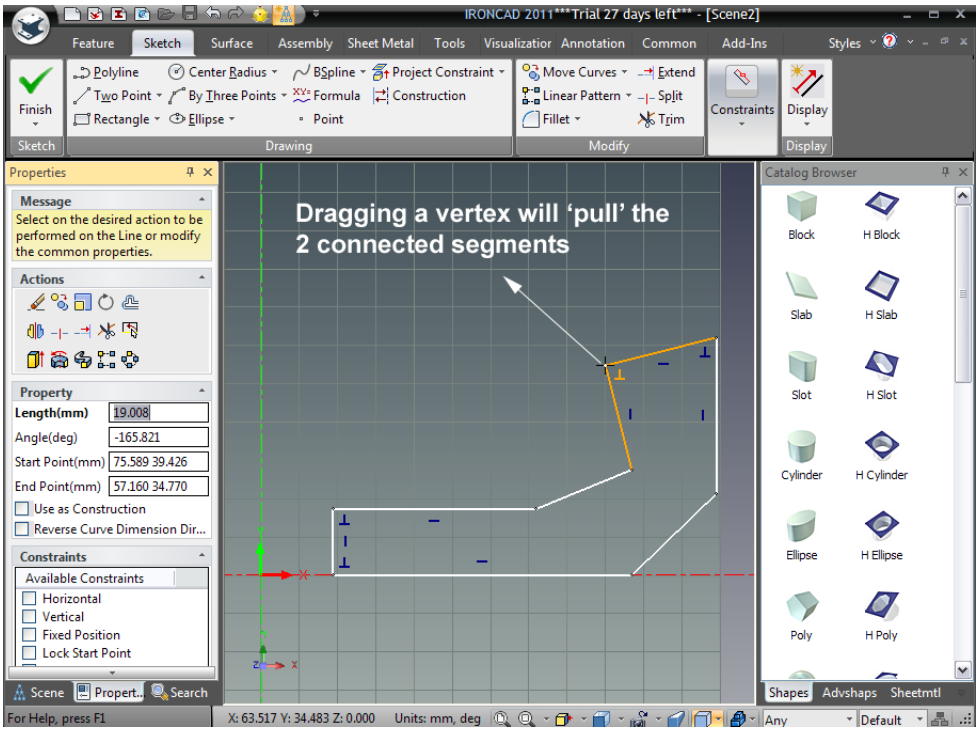
Click the Polyline tool on the Sketch toolbar. Using the grid lines as a reference, draw the profile approximately, as shown.



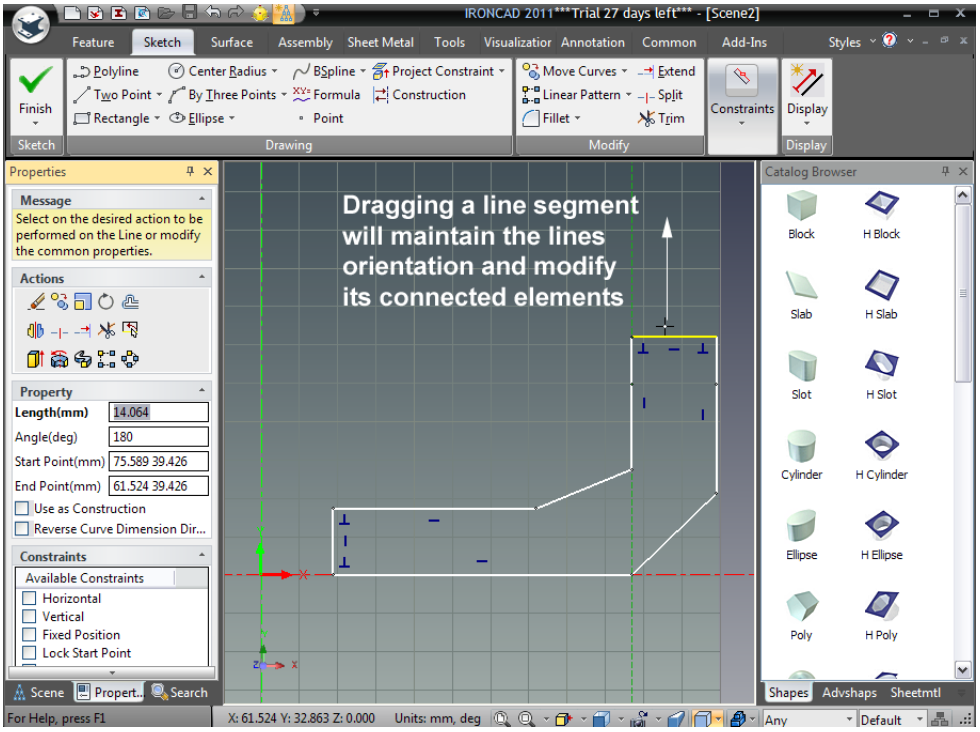
As you move the cursor between clicks, notice the snapping behavior and feedback lines that indicate geometric relationships such as horizontal, vertical, perpendicular, parallel, or tangent line conditions. Take advantage of this behavior to create the horizontal and vertical lines, as shown. To end the Polyline command, press the Esc key or click the Polyline tool again.

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

It is worth experimenting with modifying the profile by dragging lines and vertices (dragging requires you to click on the item (i.e. vertex or line) using the left mouse button and holding the mouse button down while you move the mouse around the screen). You will notice that a vertex moves freely when you drag it:



However, a line only moves perpendicular to its length if you click on a line and drag it:



4.3. Creating Precise Sketches

Unlike many systems, IRONCAD does not require you to place dimensions on the geometry. It has knowledge of the curve position, length, and angle. Therefore, it can present this information to you when you select a curve, if you so wish!

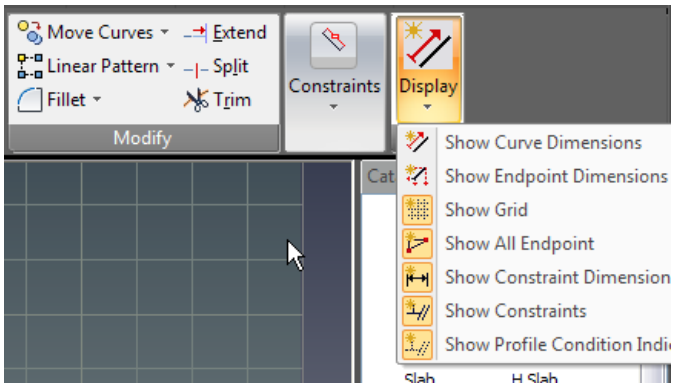
4.3.1. Positioning Lines Using Endpoint Position Dimensions

Displaying endpoint dimensions allows the quick and accurate positioning of lines and other key positioning points (like arc or circle center positions) relative to the sketch X and Y datum lines (or other geometry if redefined).

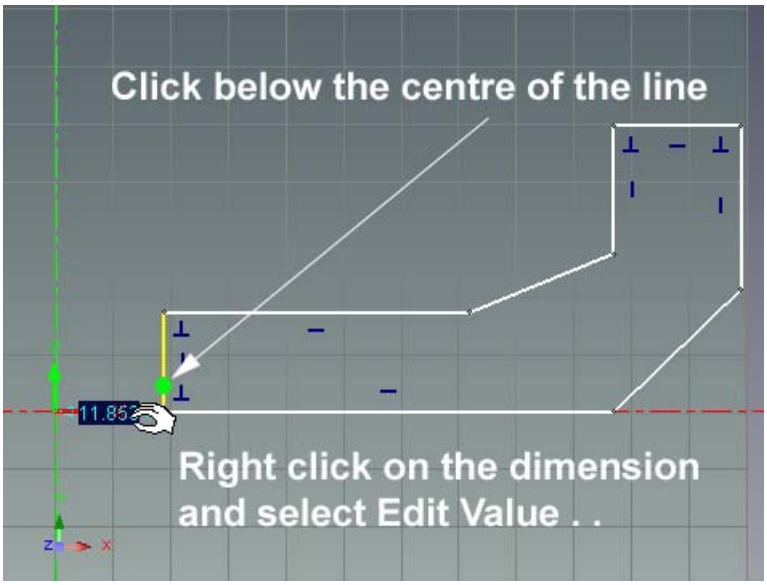
These dimensions are not “locked down” or “constrained” so you can move them freely at any time.

1. Turn on the Endpoint Dimensions

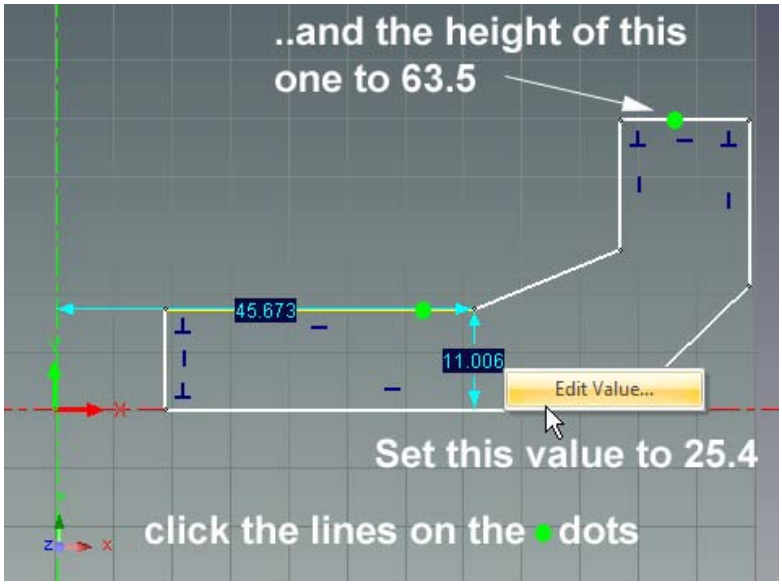
Click on the Display icon found on the right hand side of the Sketch tab of the Windows Ribbon bar. These options allow you to toggle the display setting of a number of sketch related options. Click the ‘Show Endpoint Dimensions’ option so that its icon is highlighted.



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 and then right-click select Edit Value (or double-click on this dimension).

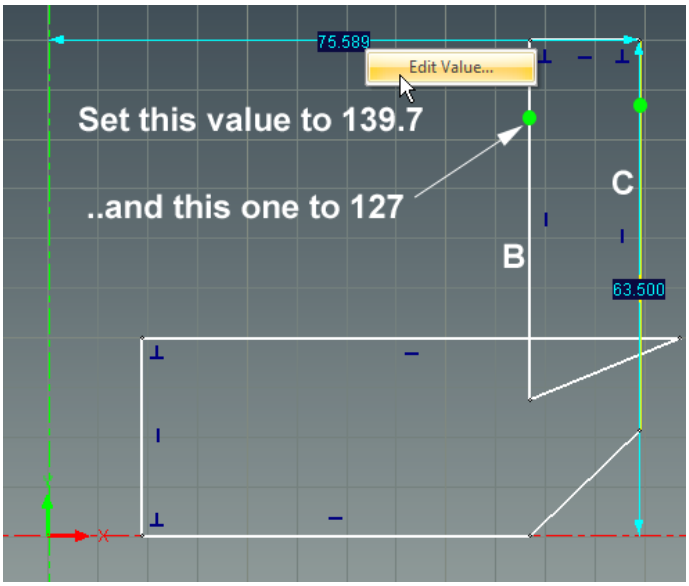


Change the Horizontal distance to 12.7, as shown, and then click OK. Repeat this same technique to position the upper, adjacent horizontal line to 25.4 mm. Select the line at the top (not the one on the vertical X-axis) and right-click on its vertical endpoint dimension. Select Edit Value and change the vertical distance to 25.4 mm and click OK. Repeat this for the upper most line and set it to 64.5mm.

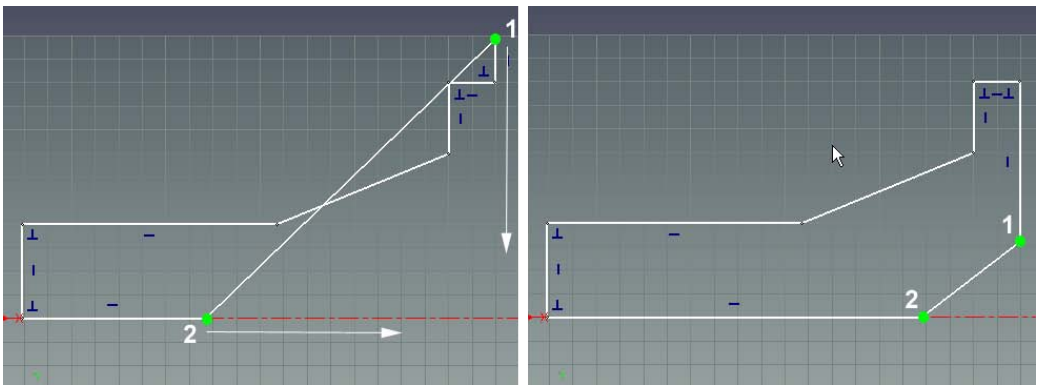


Next, select the vertical lines shown as B and C in the next image and set their horizontal distances to 127 for line B and 139.7 for line C. Even if the geometry

was the wrong scale at the beginning, adjusting the dimensions will pull it into the right shape and size even if it looks like the one below.

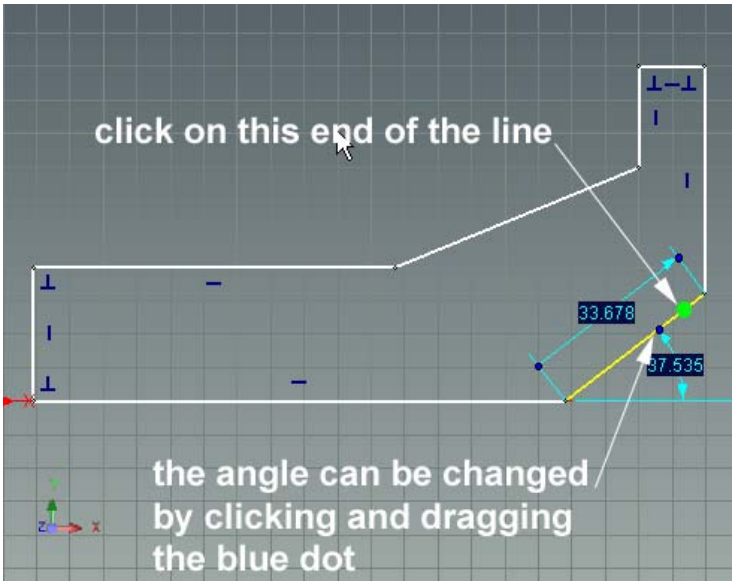


If the sketch has turned itself inside out as shown below, simply grab the offending vertex or vertices and drag them to the right to get the correct position.



4.3.2. Accurately Positioning Lines Using Curve Dimensions

Using endpoint dimensions is useful for positioning geometry with respect to the sketch X and Y datum. However, more control is provided by IRONCAD using Curve Dimensions. Using the Display panel introduced in the previous section, turn off 'Show Endpoint Dimensions' by clicking on the option and turn on 'Show Curve Dimensions'.



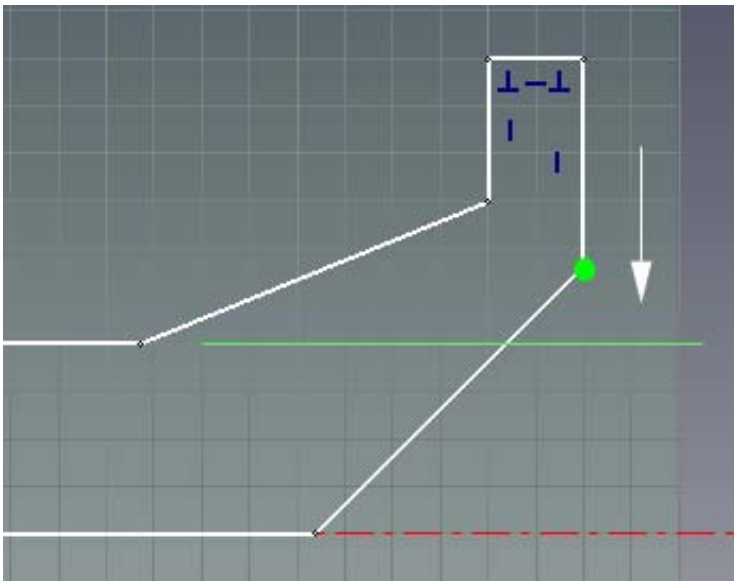
Select the angled line above its midpoint (notice the difference if you select it below its midpoint).

Placing the cursor over the blue dot connected to the angular dimension, notice how the cursor changes to a hand. These blue dots are called curve handles. Drag the angular curve handle clockwise by a few degrees to visually change the angle of the line.

Now precisely set the angle by changing the value of the angular dimension. Right-click on the value of the angular dimension text and select 'Edit Value' (or again, double-click). Change the value to 45 and then click OK.

4.3.3. Curve Handles

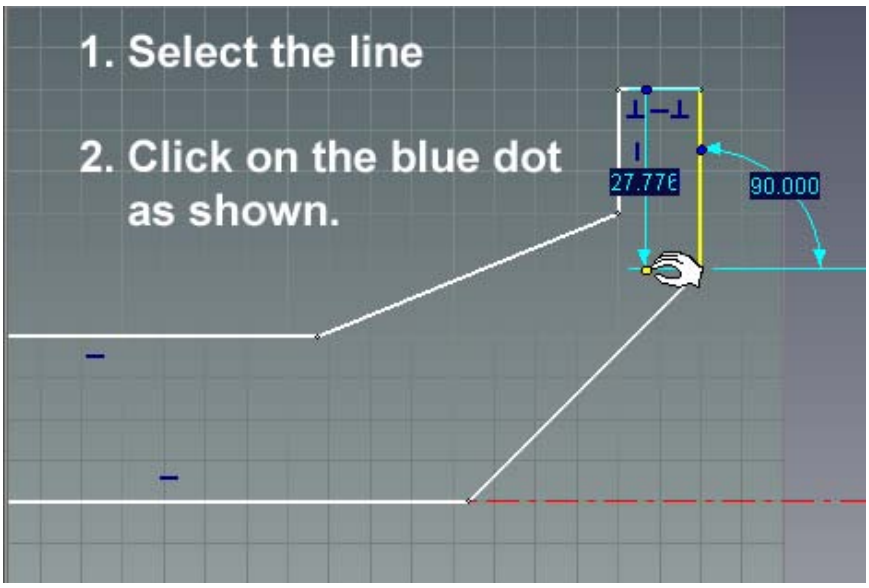
Curve handles are provided on end points of curve dimension too. These enable the length of a curve to be changed in relation to other sketch based curves without changing orientations or creating un-necessary reference geometry.



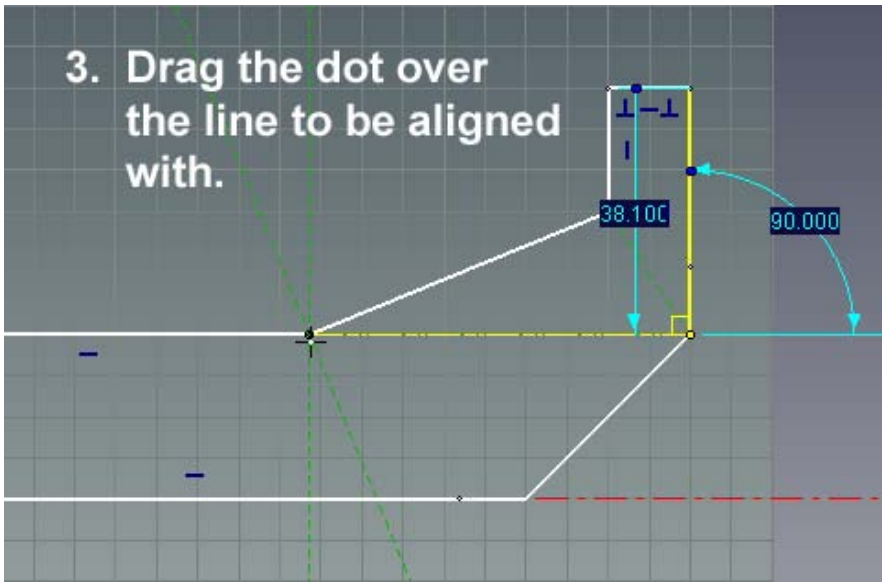
For example: in the image above, we want to move the lower vertex of the right hand vertical line to be in aligned with the horizontal line earlier set to 25.4 without changing the position of the vertical line or the angle of the 45 degree line we just set.

1. **Use the Curve Handle to Maintain Curve End Conditions**

   Select the line to adjust. IRONCAD will add the curve dimensions with blue dots at either end of the dimension and a blue dot on the curve. The blue dots at the ends of the dimension are known as linear curve handles.

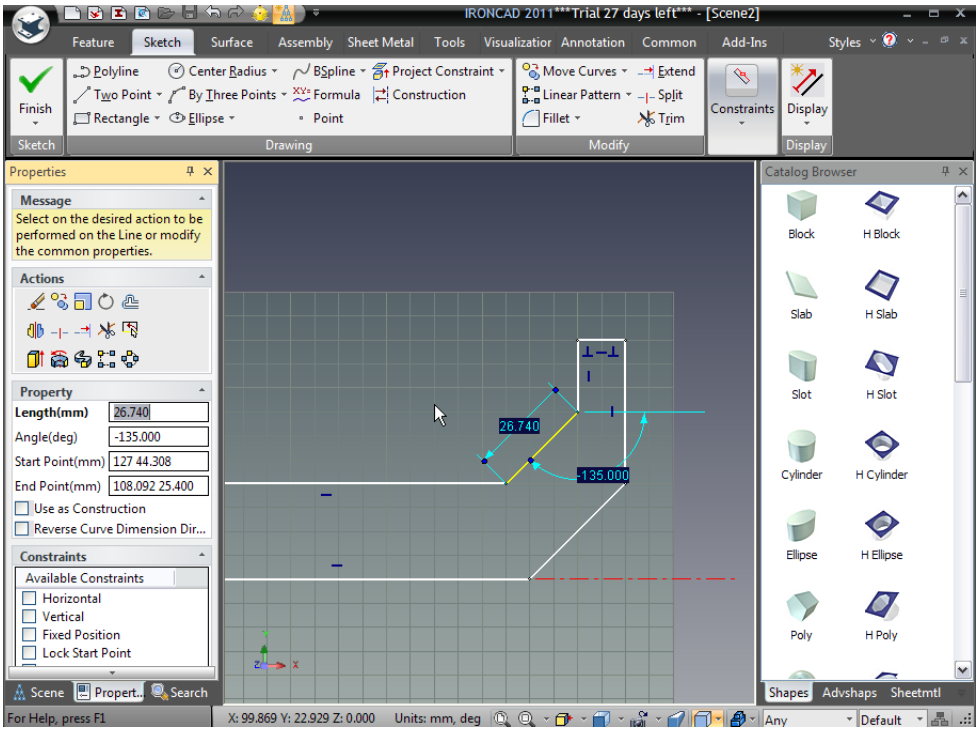


Click on the lower linear curve handle and drag it over to the line we set at 25.4 or over the vertex as shown, and release.



The line will extend until its lower point is in line with the 25.4 line used as a reference without altering the angular relationship of the vertical line and the 45 degree line.

Now adjust the uppermost angular line to also lie at 45 degrees, as shown next without worrying about its length.



4.3.4. Changing the Orientation of Endpoint Dimensions

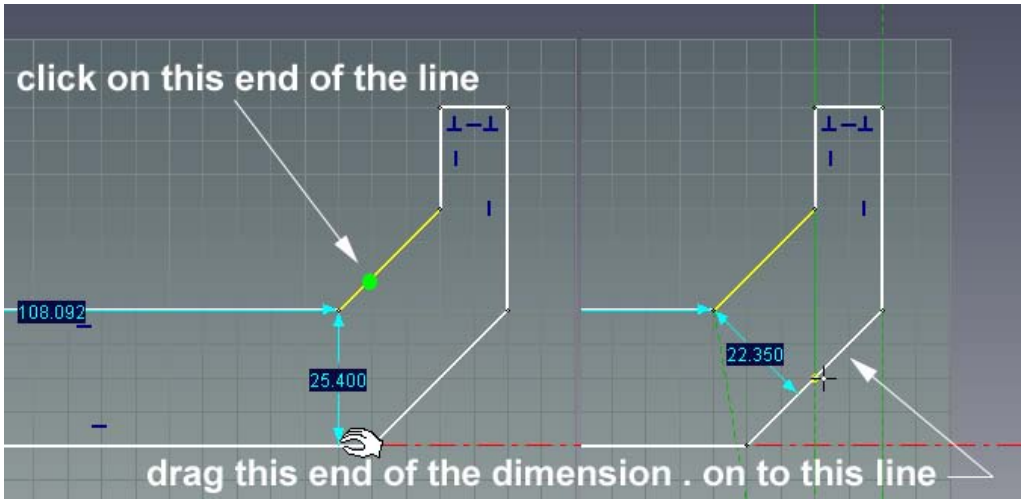
Previously, elements of the sketch were positioned with respect to the sketch X and Y datum using end point dimensions. By default, endpoint dimension are aligned and attached to the X and Y datum lines. However this is not always practical. IRONCAD provides a simple facility to allow other lines to be referenced for endpoint dimension irrespective of the lines position and orientation.

Consider the last line changed to 45 degrees. Traditionally to position this line to be a specific distance from the other 45 degree line, either new construction geometry would be required or it would be controlled by adding a dimension. In IRONCAD however, the same can be achieved by simply dragging the endpoint dimension to the 'new' reference line and then changing its value, as shown next. This technique is especially useful when creating geometry that is not aligned to the X and Y datum.

Switch "Show Endpoint Dimensions" "ON" by clicking on the Display icon. You may leave 'Show Curve Dimensions' switched on. IRONCAD will happily display both Endpoint Dimensions and Curve Dimensions which can be very

useful but it can also be a little crowded on small geometry spans. For the illustration, only the Show Endpoint Dimensions function was active.

Select the uppermost angular line, as shown. Drag its lower arrowhead over to the other angular line until it snaps onto the line. IRONCAD gives visual feedback by tracing a pale yellow dot over the new reference geometry. The endpoint dimension should now look something like the image below.



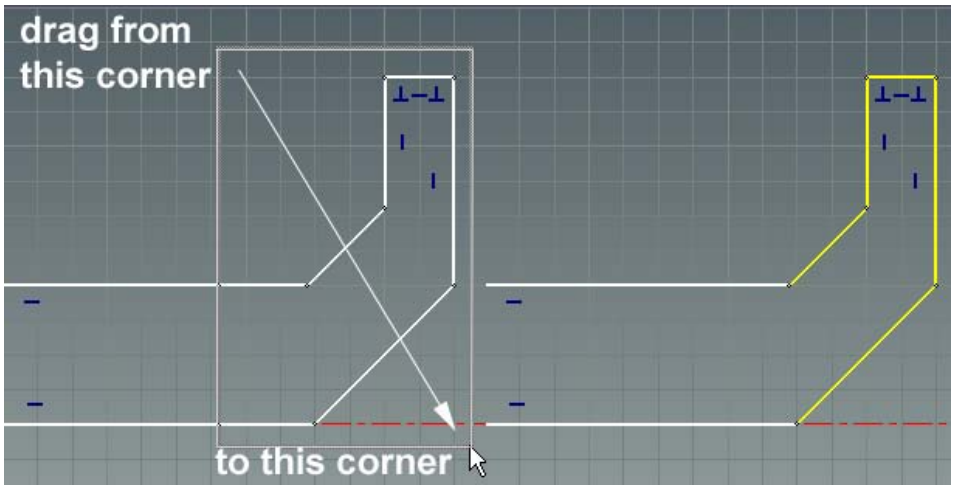
Change the value of the endpoint dimension to 19.

When you no longer require this orientation for endpoint dimensioning, simply drag the dimension arrowhead back to the horizontal axis. This can be important as unexpected changes may occur if the orientation is not reset.

4.3.5. Positioning Multiple Lines Simultaneously

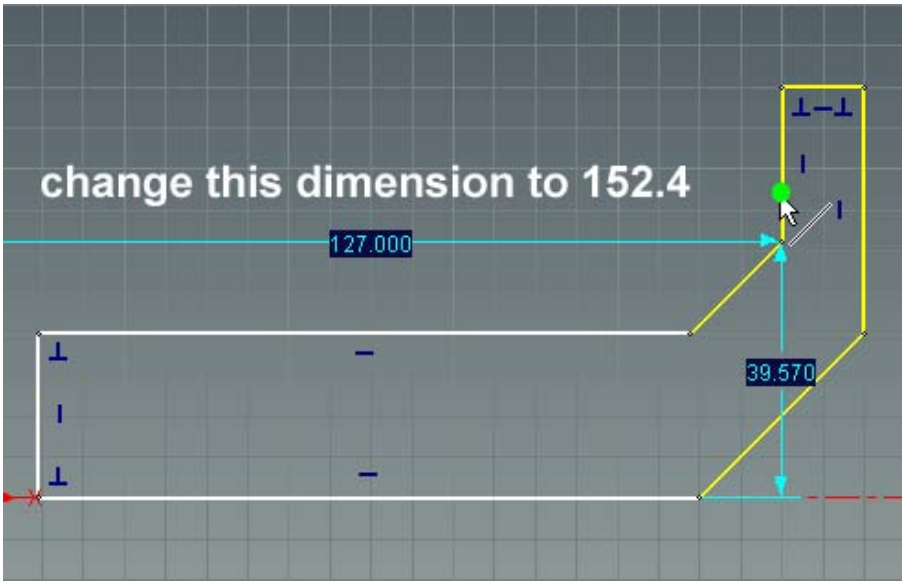
Endpoint dimensions and curve handles can be used to position multiple lines simultaneously.

Without selecting any icon or piece of geometry, draw a box around the geometry by simply clicking and dragging the mouse to create a box around the lines being selected as shown next.



IRONCAD will highlight the selected curves using a pale yellow (by default). Note: using a left to right motion will select only the curves fully within the box boundary. However, clicking and dragging from right to left will include all lines crossed by the boundary in addition to those wholly within the boundary.

While all five lines are still selected, refer to the next image and click the line shown and change the horizontal dimension to 152.4 (this is an endpoint dimension – enable this if you need to see it).



As an alternative to box selection, multiple items can be selected by clicking one of them and then Shift-clicking the rest (hold the Shift key while clicking). Furthermore, if you box select and want to remove an item from the selection,

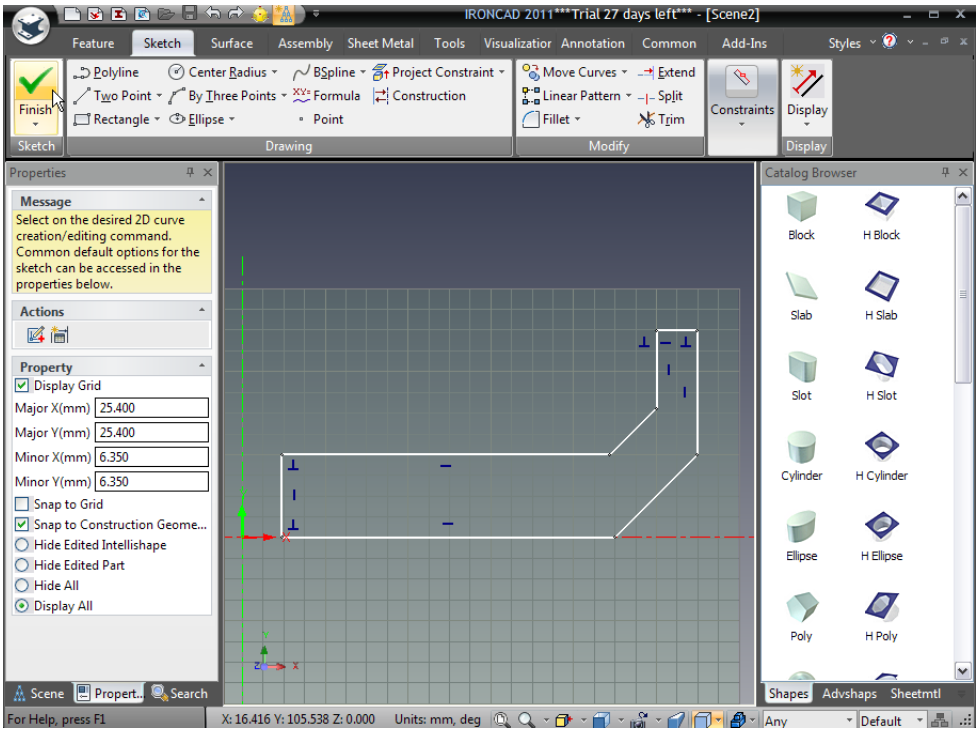
you can remove a curve from the selection by Shift-clicking the items to remove from the selection.

4.3.6. Finishing the Spin IntelliShape

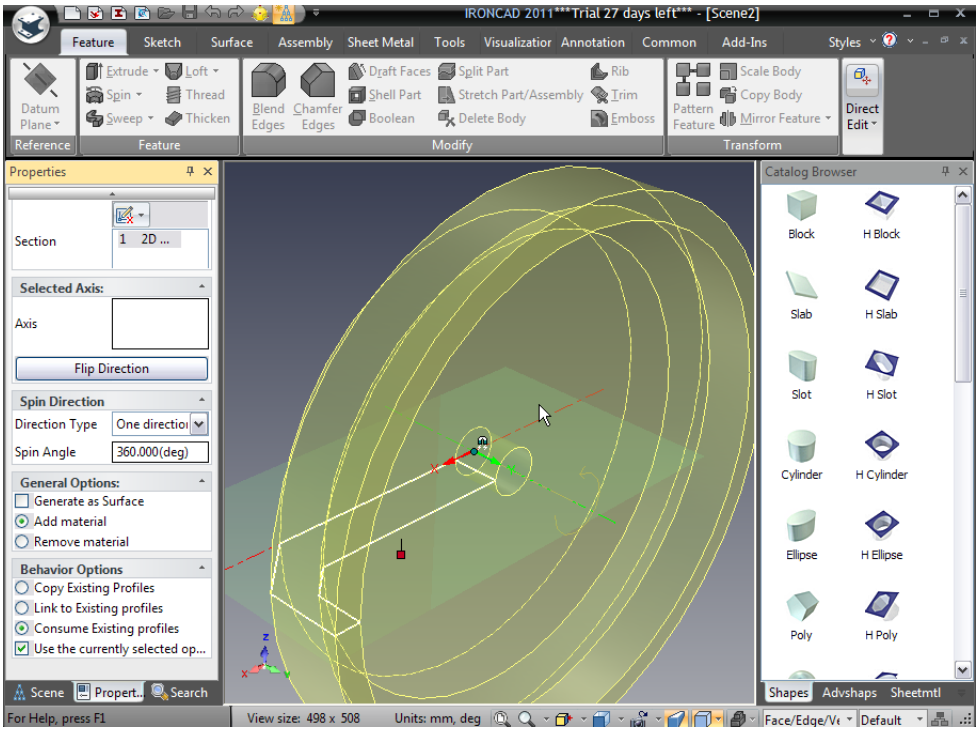
The sketch is now ready for turning into a 3D part spinning the cross-section into a solid 3D shape,

1. Click Finish

Click the green Finish check mark icon on the top left of the Sketch tab of the Windows Ribbon bar (or right-click in the background of the sketch and select Finish).



IRONCAD will then return the Command Property Browser and preview the final 3D shape. Take a minute to review the options presented by the lower three sections of the Command Property Browser.



In the Spin Direction panel:

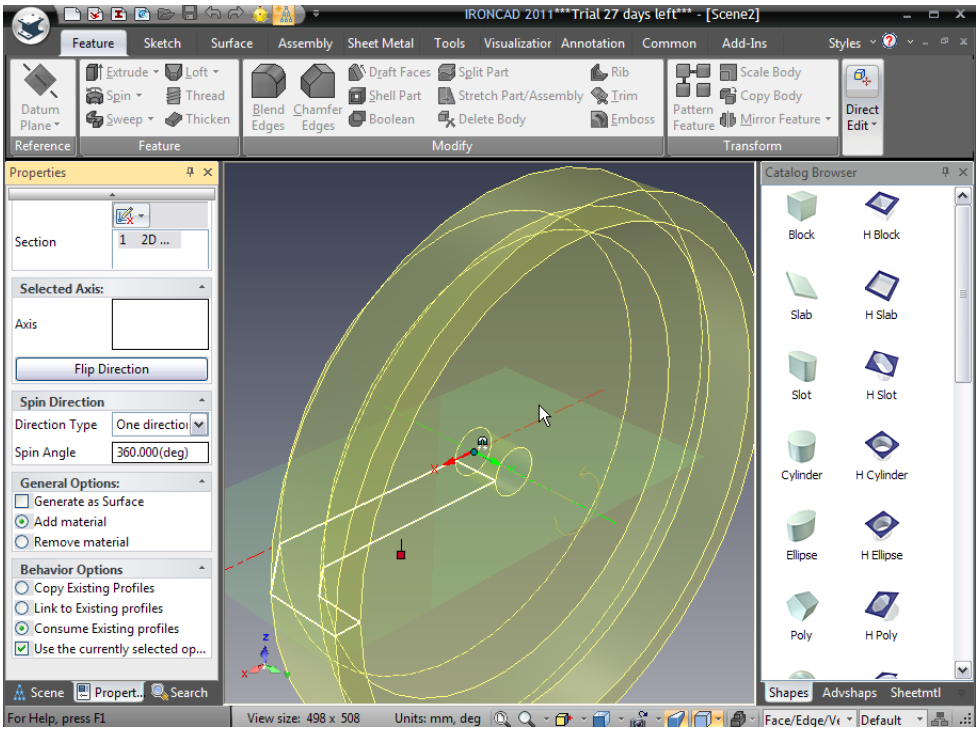
- Direction Type: Enables the item to be rotated to meet an existing item or just spun through an angle
- Spin Angle: This is self explanatory. The angle the spin is swept through to create the shape. For this exercise accept the default of 360 degrees.

In the General Options panel:

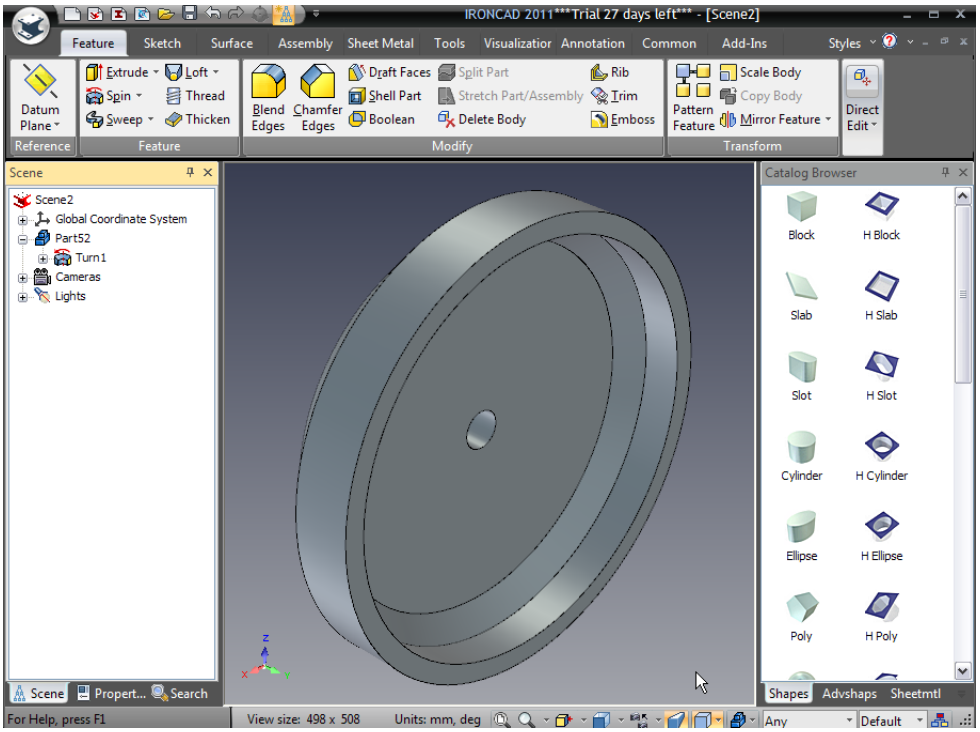
- Generate as Surface: With this option checked, IRONCAD generates a surface model not a solid model. The sketch need does not need to be a closed profile for a surface model.
- Add material and Remove material: The decision point for making a solid part or a hole. Use Add material in this case as we are making a solid shape.

In the Behavior Options panel:

The options presented within this panel control how the sketch you have just created is treated. In this case we want to use the third option, 'Consume Existing profiles', as we do not need the profile for other shapes or functions in this design.



To complete the part, click on the green check mark icon at the top of the Command Property Browser or right-click over the model and select OK.



Don’t forget to save the model!

4.4. Editing the Cross-section of an Existing IntelliShape

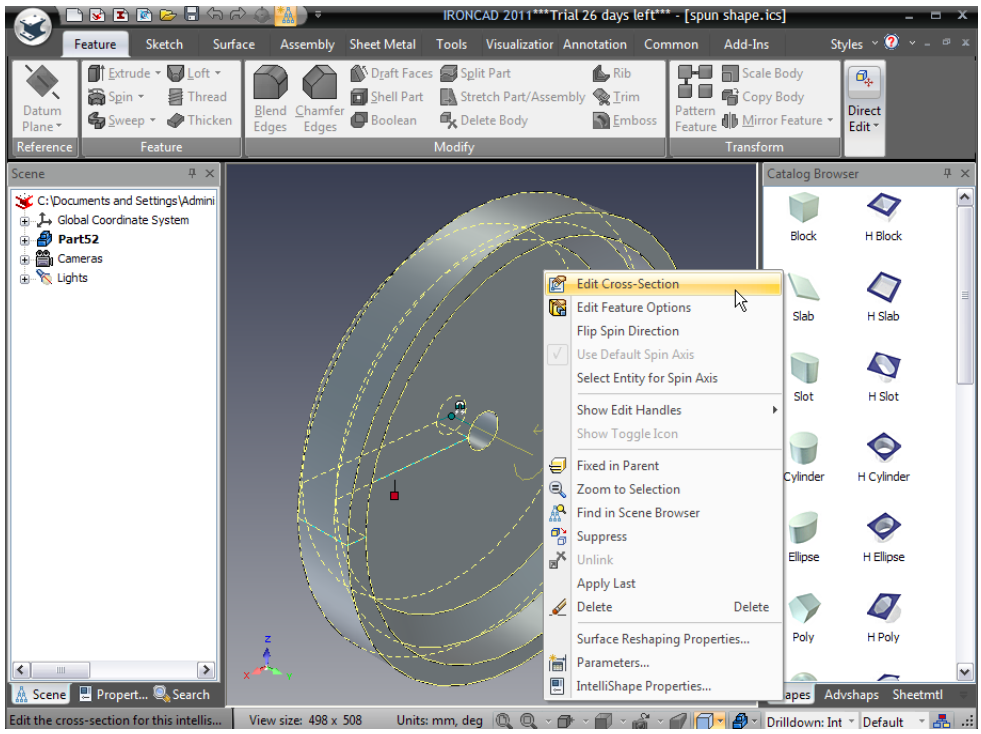
In many circumstances, the shape you have defined will undergo changes as the design develops and evolves. These changes can be accomplished by dragging shape handles and altering geometry. However it may be quicker and easier to directly edit the sketch geometry itself.

The following section introduces additional 2D sketch functions such as using existing geometry to create new geometry and adding a degree of intelligence to the sketch to ensure changes generate predictable results.

4.4.1. Setup the Part for Editing the Sketch:

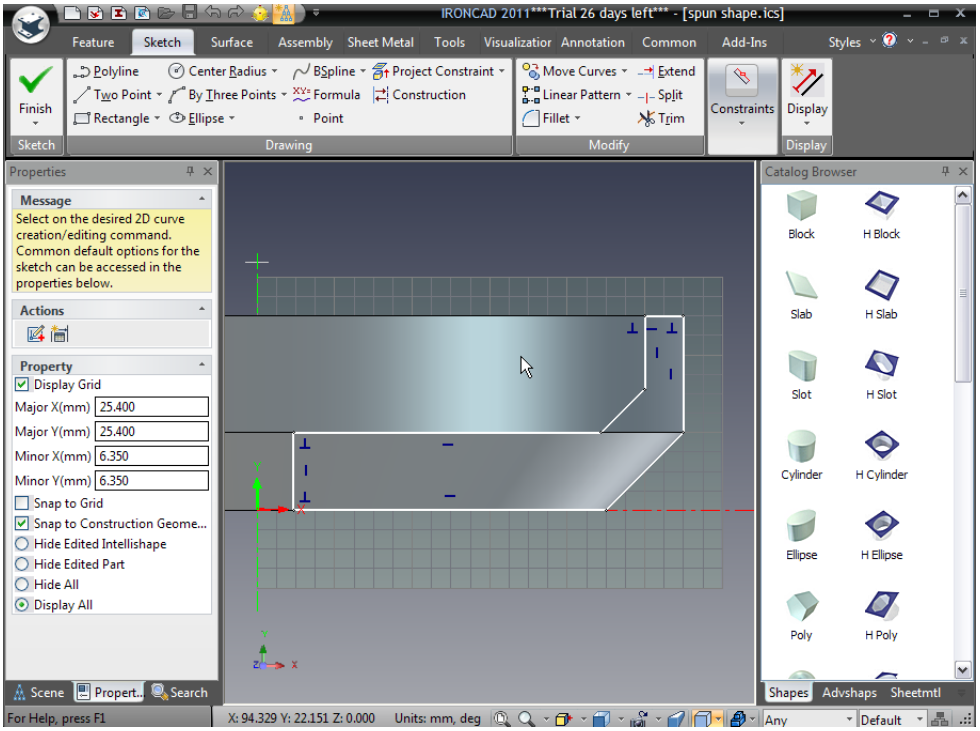
1. Select the Part at the Feature Level

Select the flywheel 3D model at the IntelliShape level (i.e. the edges of the 3D part are highlighted in a pale yellow and the sketch geometry will be visible as shown next).



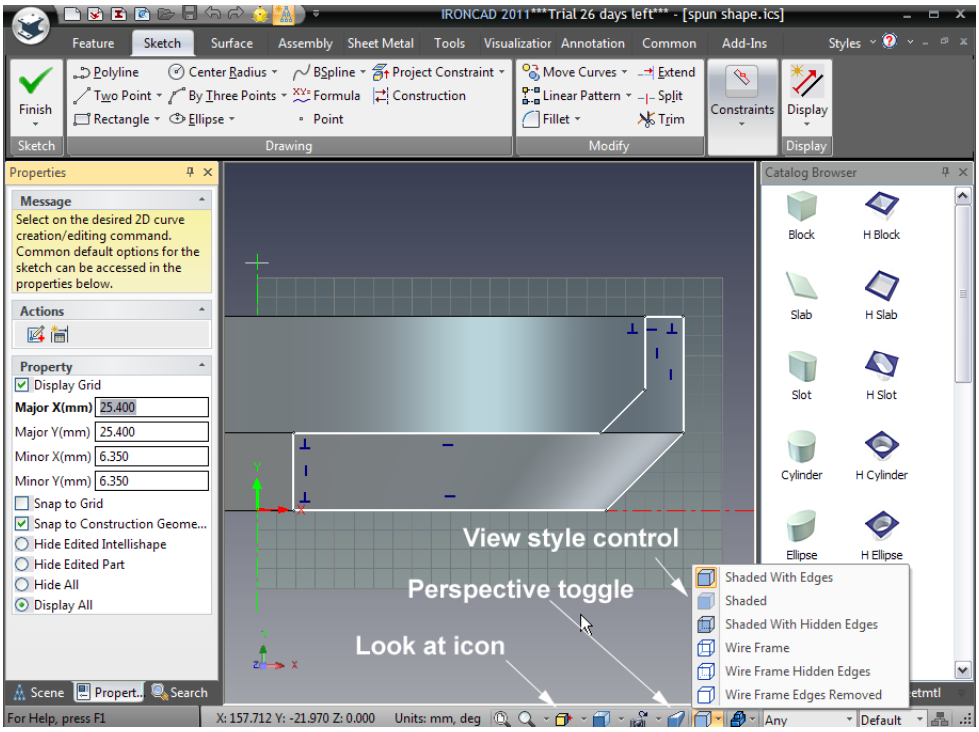
2. Right-Click Edit Cross-Section

Right Click on the IntelliShape (away from any handles that might be visible) and from the drop down menu select the first option, Edit Cross-Section. If you right-click over a handle, the drop down menu will not display the menu option Edit Cross-Section.



The 2D sketch will be displayed against the background of the 3D solid model. The sketch should be quite clearly visible. Visibility can be affected by display settings and your preferred color scheme as well as the complexity of the scene.

To improve the visibility of the 2D sketch experiment with:



A. **Turning perspective on or off** by clicking the Perspective icon toggles (perspective on and off). With perspective on, you might find that geometry does not appear to line up with edges of the part. However with perspective off, the geometry will align correctly.

B. **Turn part edges off** using the view style control and selecting Shaded. It is worth experimenting with the other settings to become familiar with the various display options.

During your experimentation, if you have altered your viewpoint and want to quickly return to looking square on to the sketch plane, press F7 (or click on the icon on the lower right toolbar). The mouse pointer will change to a pointing finger. Simply click on the sketch plane and IRONCAD will re-orientate the view to look directly onto the sketch.

4.4.2. Using the Offset Command

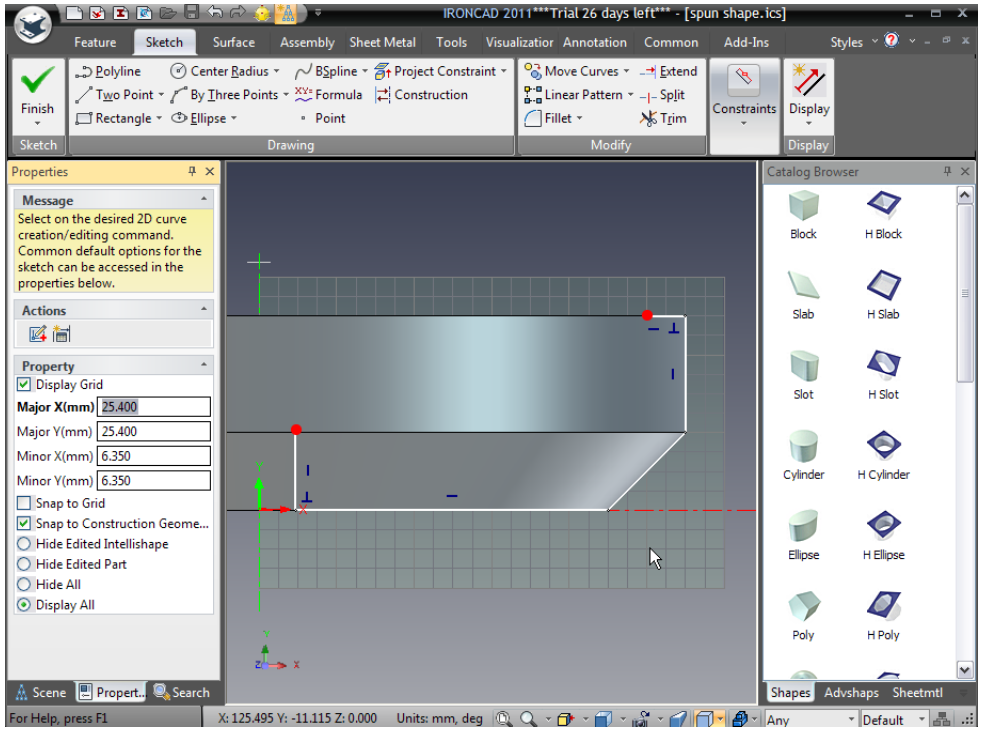
Suppose the design has changed so the flywheel requires a constant wall thickness. You can quickly and easily edit the sketch defining the flywheel.

1. Delete the Three (3) Inner Lines:

A. Either by right-clicking each line and choosing Delete individually,

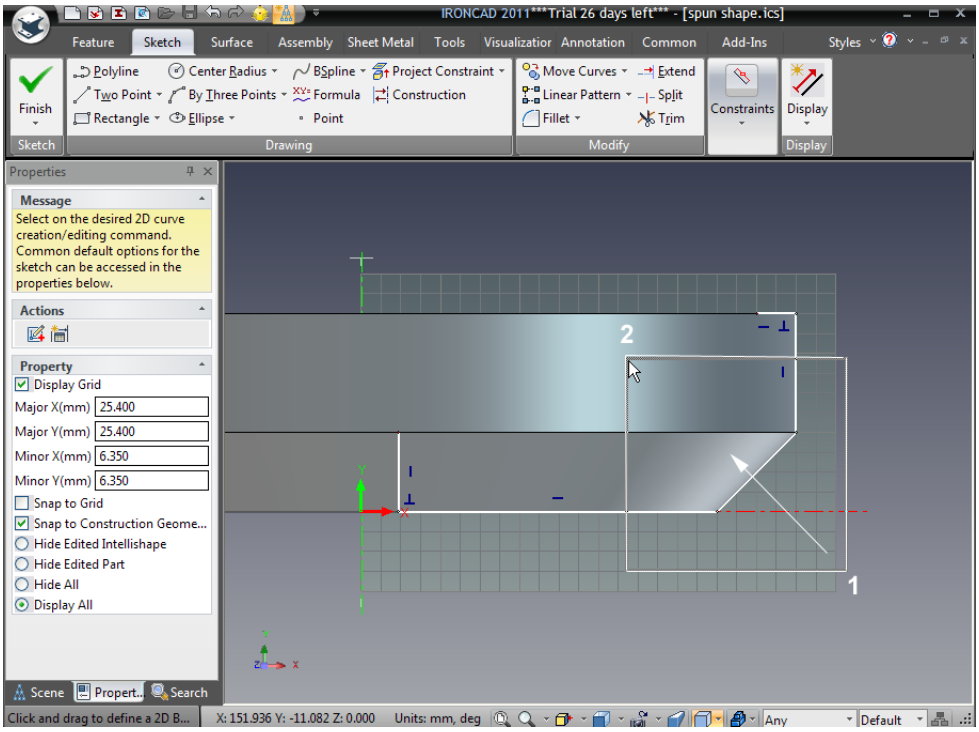
B. Selecting each line (or all the lines at once using Shift-click) and pressing the Delete key,
C. Or by using box selecting and pressing the Delete key.

The result should look like the following image. The red dots indicate endpoints of curves that are not connected to any other curve.



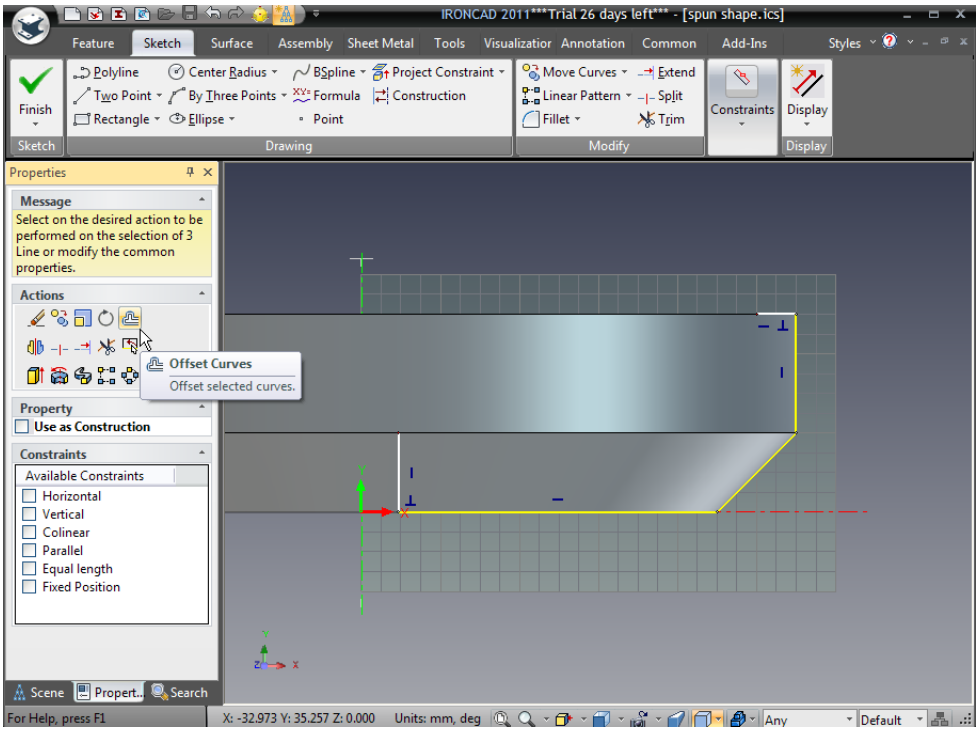
2. Select the Three (3) lines shown:

A. Either by clicking the first line, and then Shift-click the remaining two lines,
B. Or by box selecting from bottom right (1) to top left (2) as shown next:



3. Start the Offset Command to Offset the Selected Curves

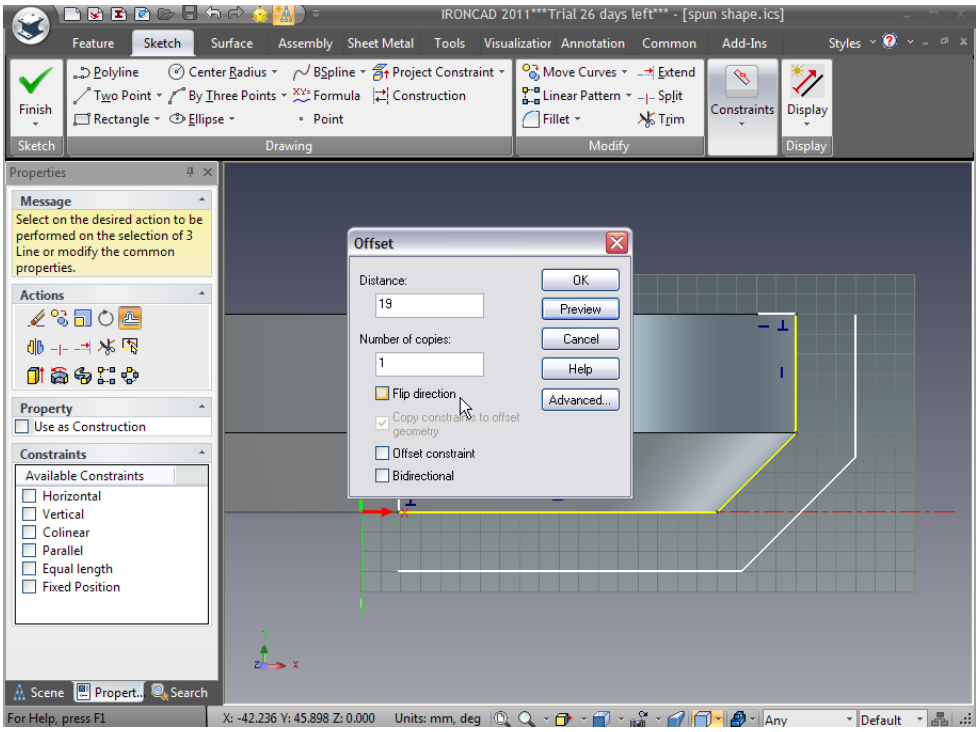
From the Actions panel on the Command Property Browser, select Offset Curves.



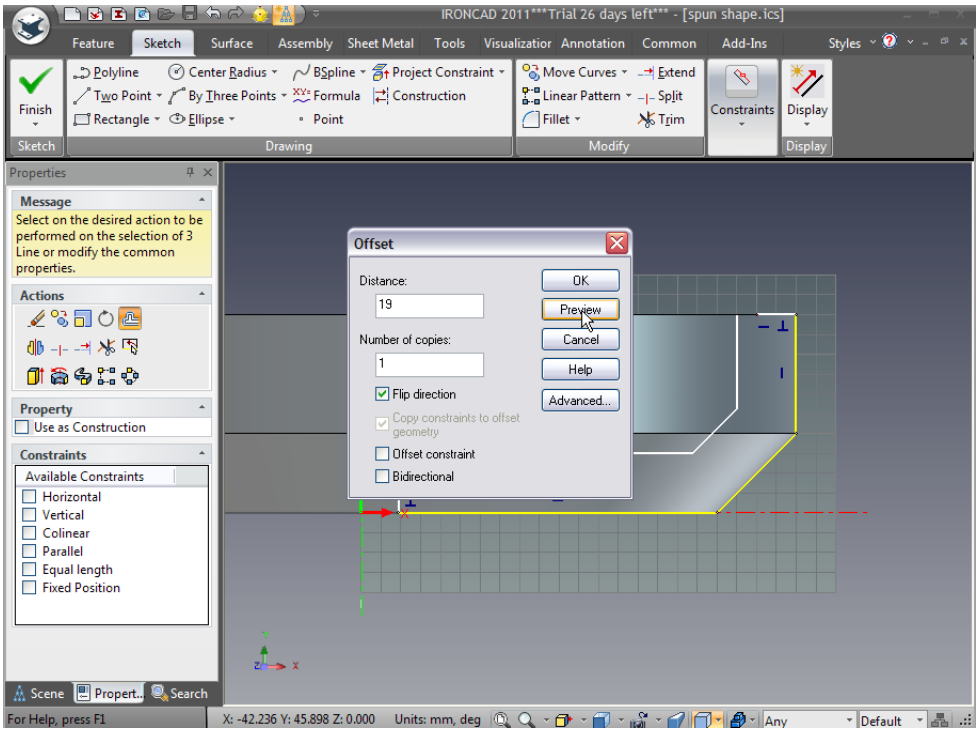
IRONCAD will display the Offset dialog box.

4. Specify 19mm in the Distance

Enter 19 in the Distance box leaving the Number of Copies at 1 and click Preview.



If the lines are offset to the wrong side (below instead of above), click Flip Direction and then click OK. The result will look like the following image.



5. **Connect the Disconnected Curves**

   Finally reconnect the line endpoints by dragging the red dots of the top right horizontal line and the bottom left vertical line to their endpoints of the offset geometry.

6. **Finish the Sketch**

   Right-click and select the green check mark to finish the drawing.

4.5. Using 2D Constraints

Up to this point, we have used IRONCAD's freedom to easily create and modify shapes without too much concern about the outcome. This freedom is especially useful when you're in the conceptual design phase. However, there are times when you want to tightly control the design and enforce precise geometrical relationships and dimensions were this is sometimes referred to as 'design intent'.

IRONCAD provides the tools to operate in this manner using specific curve dimensions and geometric constraints. This includes adding relationships between elements of your design often referred to as parametric design. You can enforce as much or as little "design intent" control, as desired.

This section will introduce the use of constraints to control design intent.

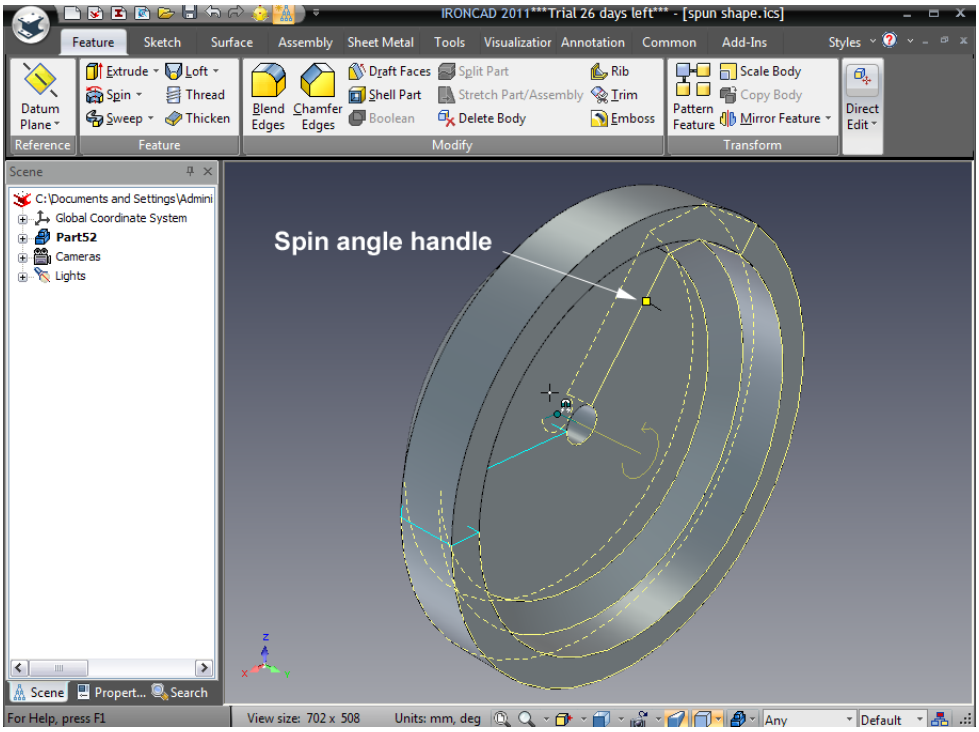
4.5.1. Adding Dimensional constraints

With Spin IntelliShapes, it can be advantageous to 'open' up the shape to get a clear view of the sketch profile. This is useful for editing the sketch and for dragging shape handles in the context of an assembly.

1. **Select the Flywheel Shape at the IntelliShape Level**

2. **Pull the Spin Handle**

   Grab the Spin Angle handle, as shown, and drag it approximately 100 degrees (alternately you can right-click on the handle, select Edit Value and enter an exact value for the extent of the spin angle).

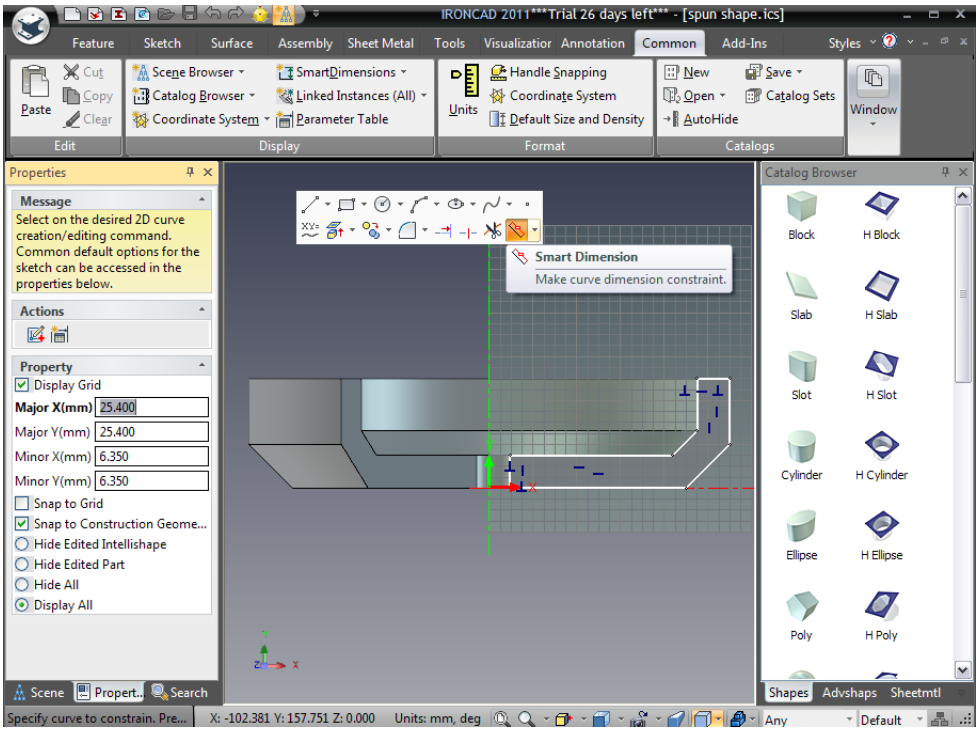


3. Edit the Cross-Section

Right-click on the IntelliShape and select ‘Edit Cross-Section’.

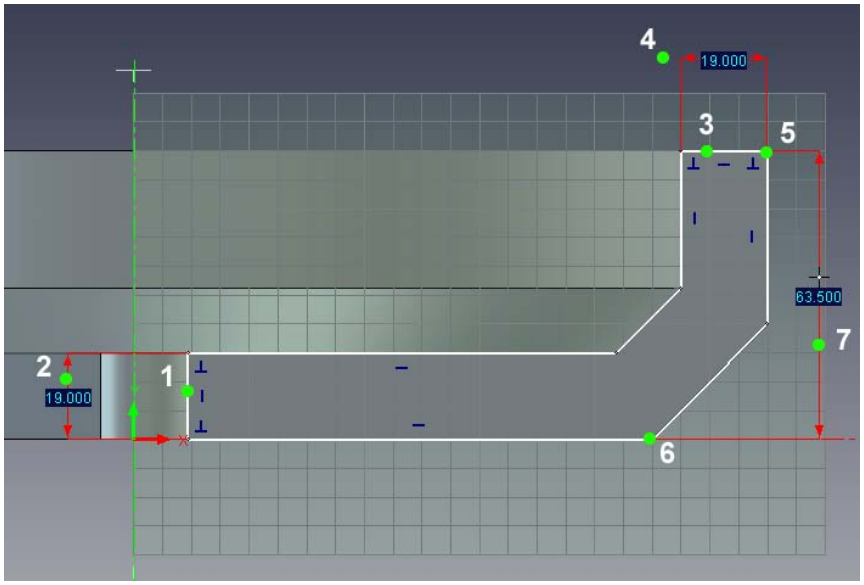
4. Start the SmartDimension Command

Next, click the ‘S’ key and select Smart Dimension as shown.

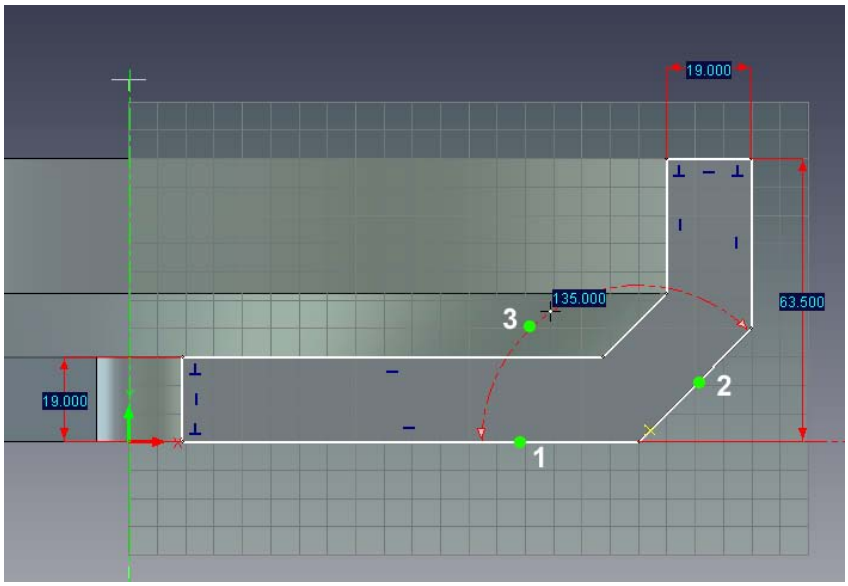


5. Add Dimensions

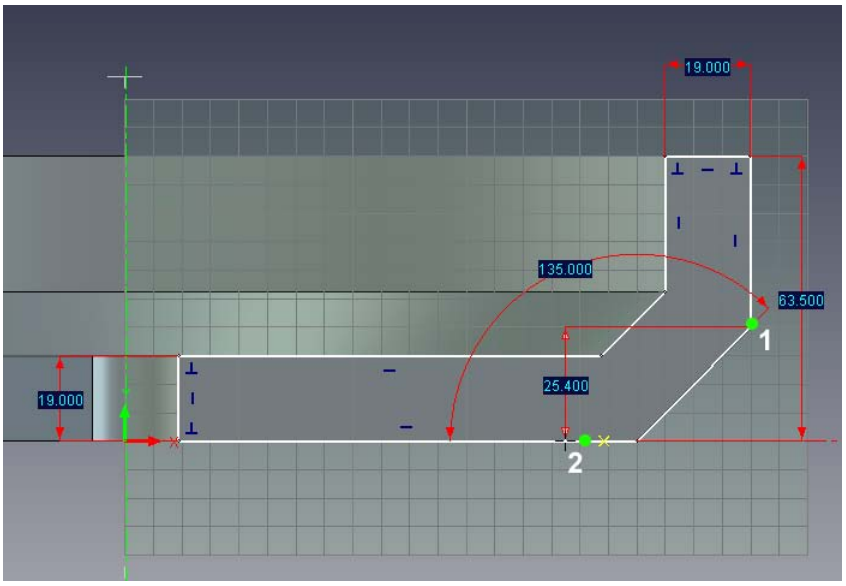
Add the dimensions for the left hand vertical line by clicking on the line (1) and then positioning the dimension (2). Add a dimension for the top most, short horizontal line by clicking on the line at (3) and positioning the dimension (4). Being able to click away from the dimension position can be useful when keeping the sketch tidy and readable.



For the last dimension shown, click on curve endpoints (5) and (6). The dimension will be drawn directly between the points. Orient the dimension either as horizontal, vertical, or as “shortest distance between two points” simply by moving the cursor until it snaps into the desired orientation (7). Apply an Angular Smart Dimension by clicking the two lines that define the angle (1) and (2) in the next image. IRONCAD senses the two lines are at an angle to each other and creates the appropriate type of dimension. Before clicking to indicate where the dimension should be placed, move the cursor in a circular motion around the intersection point of the two angled lines. IRONCAD will display the angle appropriate to the current cursor position.



Add vertical dimensions for the angled line by indicating the end of the line (1) and the horizontal line (2) and position the dimension appropriately.



For clarity you can reposition a dimension at any time after placing it:

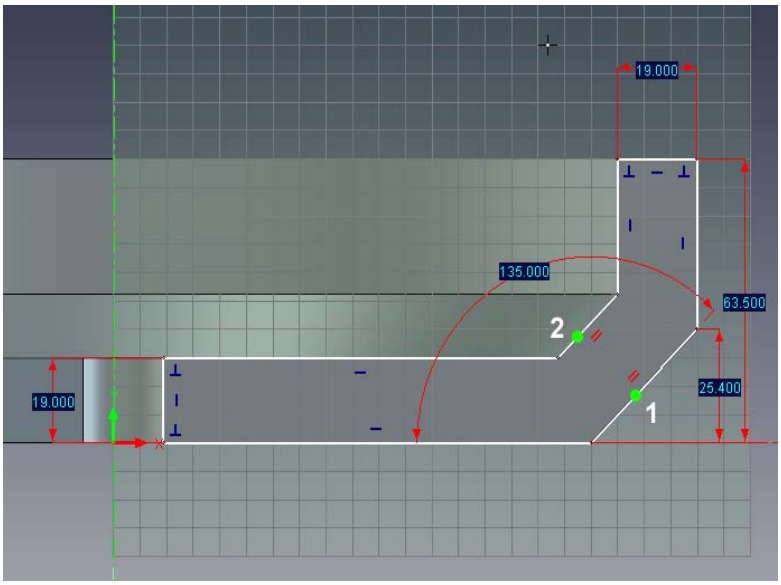
A. End any currently active command by pressing the Esc key or clicking on the active command icon.
B. Position the cursor over the dimension (IRONCAD will change the cursor to a hand).
C. Using the left mouse button, click and drag the dimension text to a new location.

4.5.2. Adding Geometry Constraints

Geometry constraints work in conjunction with dimensional constraints to convey the design intent. Geometry constraints do not need to have dimensions on the geometric elements to convey the attitude of an element or set of elements. The easiest geometry constraint to visualize is the perpendicular constraint. By applying a perpendicular constraint to two elements, IRONCAD will maintain the angle between the two elements to be 90 degrees. Similarly, a collinear constraint will ensure that two elements are maintained in a collinear fashion.

1. Apply a Parallel Constraint

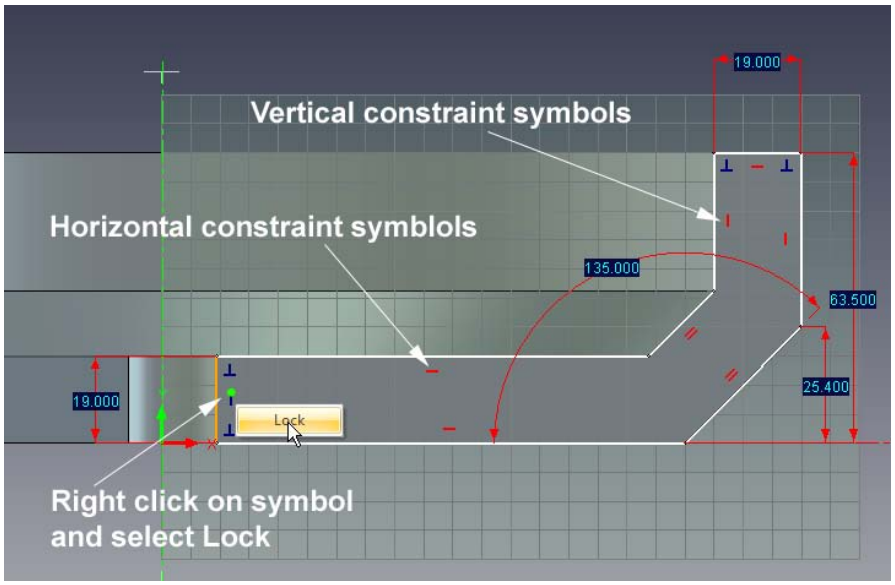
To ensure the two angled lines remain parallel, from the Constraints panel, select Parallel and select the two angled lines (1) and (2) shown in the next image:



IRONCAD will respond by adding a symbol of two red parallel lines by each line controlled. As the first line has a dimensional control, this control is applied to the second line thereby eliminating the need for a second angular dimension.

2. Create Constraints from Existing Profile Indicators

Referring to the next image, right-click on each blue icon (or profile indicators) that indicate horizontal or vertical lines and select Lock. Profile indicators automatically appear to guide you during the creation. If you need the physical constraint, simply lock it.

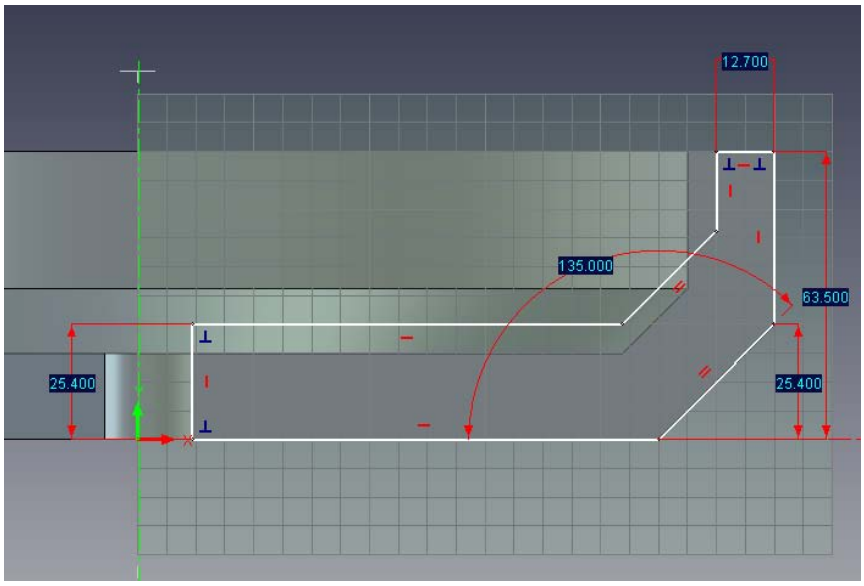


4.5.3. Making changes predictable

At this point, the sketch will still be shown as a collection of white curves and should resemble the previous image. Modifying dimensions at this point will change the curve. However, the changes may result in some unforeseen results as the profile is not fully constrained. For example, changing the horizontal 19mm dimension, which side should IRONCAD move?

1. **Modify Dimensions to See the Affect**

   To modify the value of an existing dimension, simply right-click on the dimension text and select Edit (or double-click). Use this technique to change the dimensions to the values shown.



2. Experiment with dragging lines and vertices, as shown next:

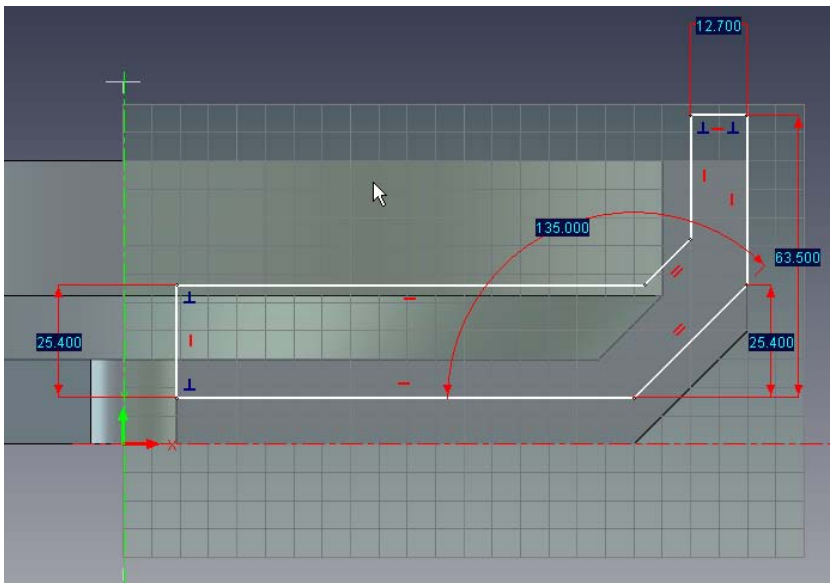
Each time, return the profile to the previous state by using Undo. The profile can move and stretch to some extent, but only in a manner that does not violate any of the dimensions and constraints. This is possible because the profile is not yet fully constrained. If the edits to the profile were accepted at this point (right-click Finish Drawing), the same behavior would be applied to the 3D model.

It should have been apparent that the sketch moves away from the X and Y axes in the 2D sketch (and in 3D the position changes with respect to the part anchor). This might not be desirable, nor provide the predictable design intent required.

To add predictability to the design intent, the profile must be dimension or constrained relative to a fixed point. One way to do this is to right-click on a vertex and Select Lock Position. Another way is to dimension a vertex relative to some other fixed point, such as the X and Y origin. For this profile, it makes more sense to relate the sketch to the origin rather than locking down any particular vertex.

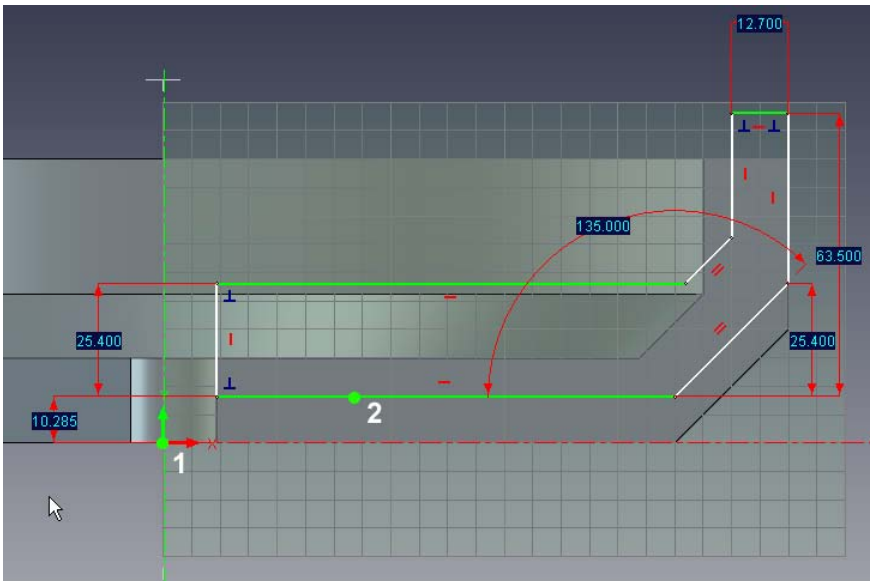
3. Move the Profile by Drag

With IRONCAD in the position of editing the profile, drag the profile away from the original position as shown in the previous image so that the sketch looks like the next image.



4. Add a Dimension from the Geometry to the Origin

Add a Smart Dimension by indicating the origin of the sketch first (2), followed by the lower horizontal line (1) as shown.

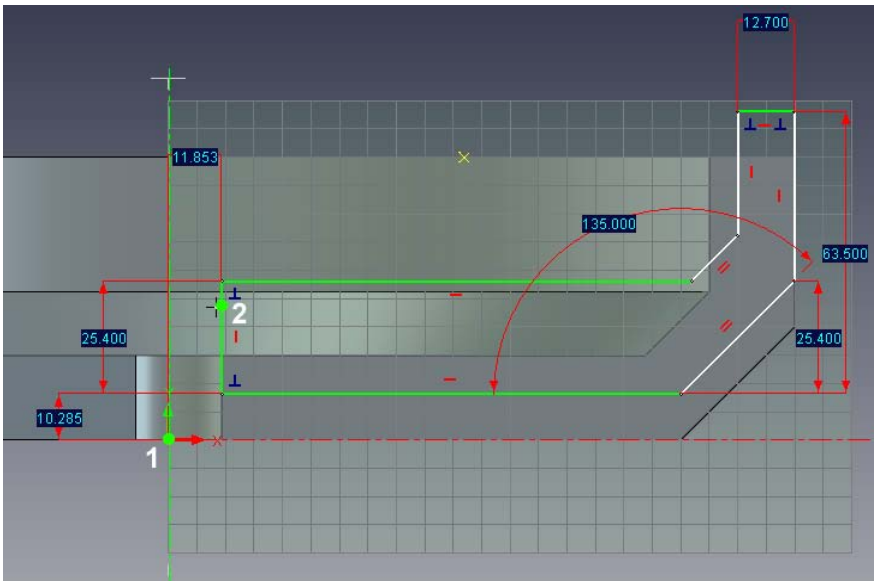


As the profile becomes predictably controlled, the geometry elements will turn green. For example in the previous image, the 3 horizontal lines become green since each one has its orientation constrained (the horizontal constraints) and the vertical positions relative to the origin have controlling dimensions.

5. Repeat for the Central Hole

Use the same technique to position the curve defining the central hole of the flywheel.

You should end up with an image something like the one below.



Again IRONCAD will change any element fully controlled to a green color.

6. Change the X Y Position

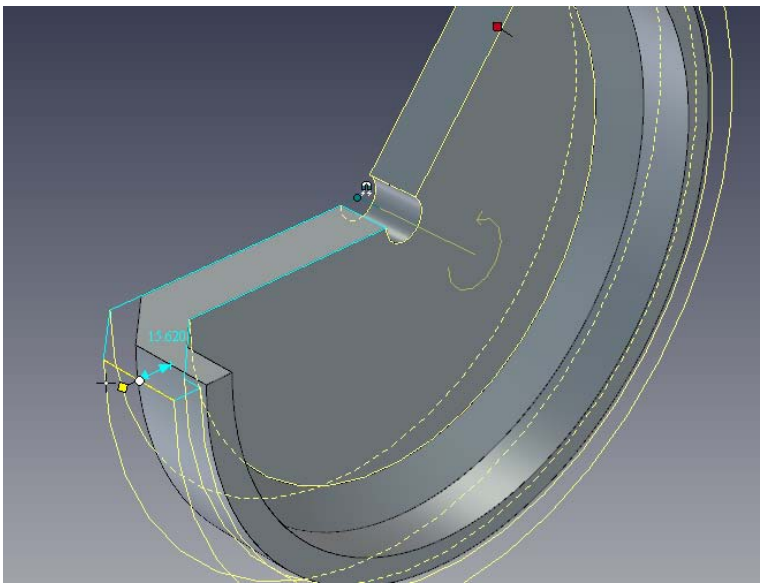
Edit the dimension value positioning the sketch to the X axis to 0 (zero) and the bore radius of the flywheel to 10 as shown below.

7. Finish the Sketch

Right-click on the sketch and select Finish Drawing for IRONCAD to complete this phase and return to the updated 3D model.

4.5.4. How Constraints Affect the Behavior of Shape Handles

The flywheel is now becoming more predictable in the way it behaves under modification. With the view arranged as shown below select the flywheel at the IntelliShape level and try pulling the shape handle corresponding to each line segment. The handles corresponding to the white curve segments will move but in a predictable way. However, the handles on the curve segments shown as green in the sketch will not drag.



As the handles are pulled and pushed, IRONCAD displays the amount of this change. This is great for visualizing the changes and rough positioning.

4.5.5. Precise Model Changes Using Shape Handles

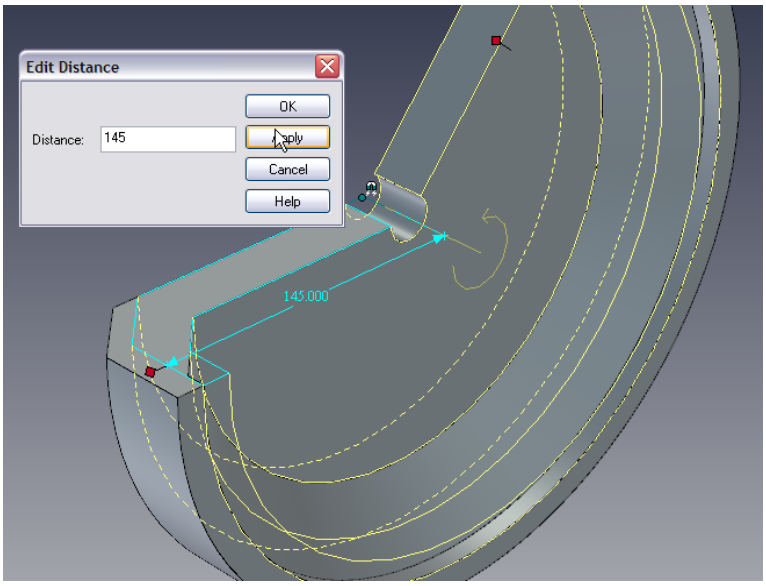
Using the right hand mouse button, select and drag a handle will result in the familiar pop-up menu giving access to precise control. For example, to change the outside diameter of the flywheel to say, 290:

1. **Select the Intellishape**

   Select the part at the IntelliShape level (the part edges will be pale yellow and the sketch shown as cyan lines)

2. **Edit the Distance of a Curve Handle**

   Position the cursor over the outer line and right-click on the handle. From the pop-up menu, select Edit Distance from Point followed by Center Point.

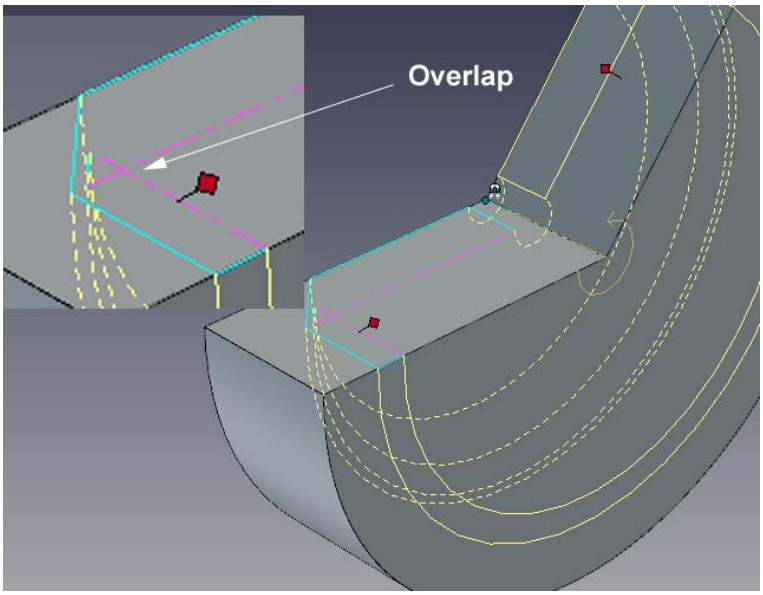


The cursor will display a dot by the pointer, and as you track over the 3D model, cylindrical surfaces and edges will be highlighted in green. Click on one of these cylindrical features and enter 145.

IRONCAD will update the part so that the radius of the outer rim is 145mm. This function is extremely useful for rapidly and accurately sizing and positioning features from other locations on the geometry.

4.5.6. Overlapping Geometry Errors

At this point, the model is not fully constrained leaving it open to unpredictable changes or changes that result in geometrical errors such as overlapping and crossing geometry elements as shown next.



In this example, the inner rim wall has been pulled inward causing the inner angled face to invert. The first clue is the most obvious one, IRONCAD displays an error message detailing the part identity and the feature that has failed. The second clue given that a problem exists, lies in the model. It doesn't vanish from the screen, just resorts to the bounding solid geometry with no features. The third clue lies in the geometry where the offending items will have been turned to magenta to help in rapidly identifying the cause of the failure.

Even if the geometry has failed, IRONCAD recognizes the model. The error can be reversed using the Undo command or any of the offending elements repositioned.

4.5.7. Adding Full Design Intent to the Flywheel

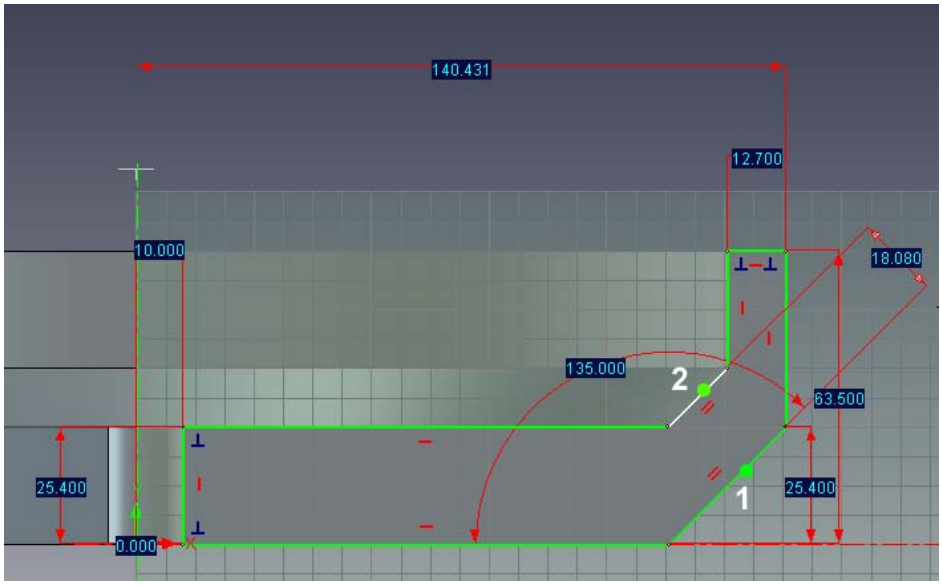
To remove the possibility of geometrical issues and make the shape completely predictable, the model needs to have the sketch geometry fully constrained.

1. **Edit the Cross-section**

   Select the part at the IntelliShape level, right-click and select Edit Cross-Section. IRONCAD will re-orientate the view to look directly onto the sketch.

2. Add SmartDimensions

Press the 'S' key and select the Smart Dimensions tool. Add a Smart Dimension from the outer vertical line and the Y axis. Next add a dimension between the two angled lines as shown next.



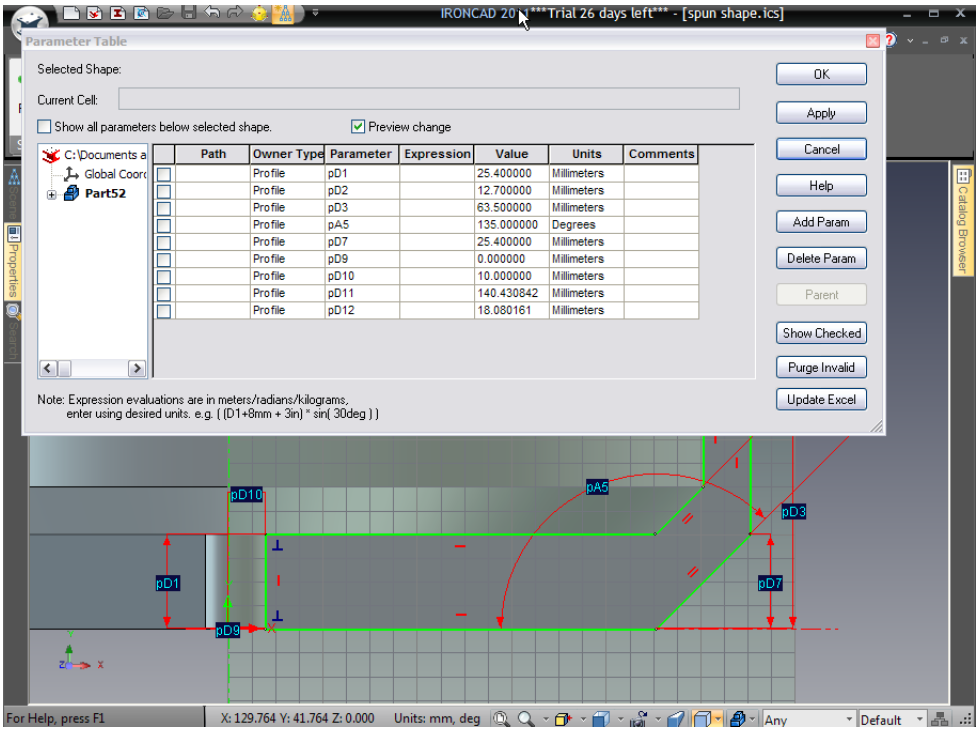
When the first line (1) is selected IRONCAD will draw a dimension for that line, selecting the second line (2) causes IRONCAD to create the dimension between the two lines.

The whole sketch profile will change to green indicating the sketch is fully constrained.

3. Display Parameter Table

Press the Esc key to cancel the command. Right-click the mouse button when the cursor is over the sketch plane (but not over a dimension!). From the drop down menu, select Parameters.

IRONCAD will display a spreadsheet like table over the scene. Hover the mouse pointer over one of the dimension values and IRONCAD will change the display of the dimensions from discrete values to a series of codes matching the Parameter column entries in the table.

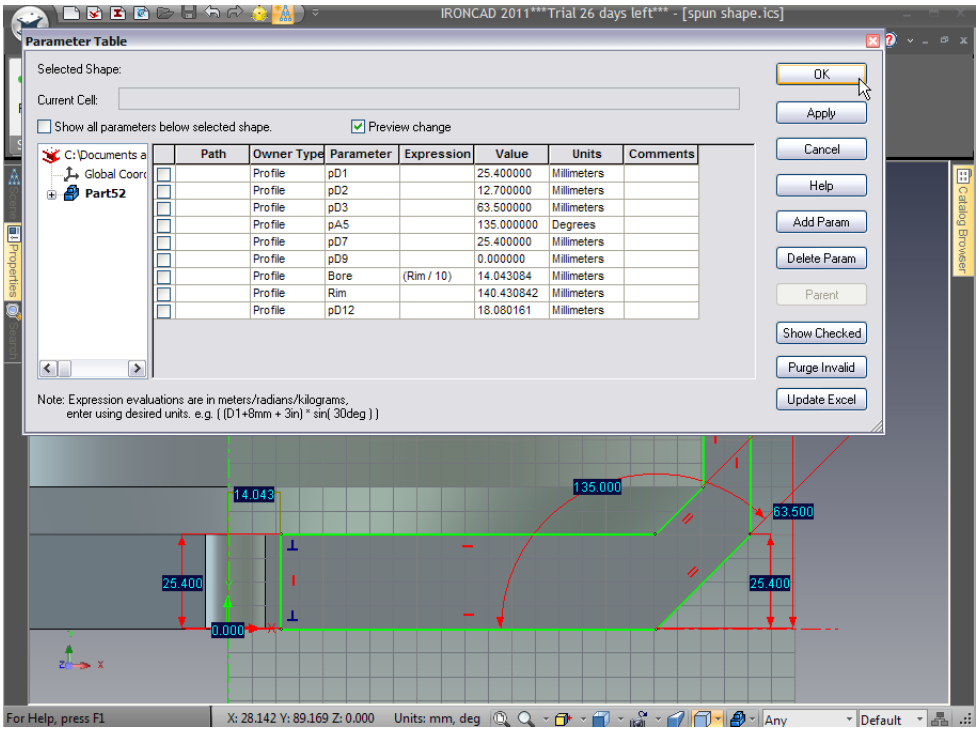


4. Change Parameter Information

Select the parameter in the table that corresponds to the central bore (in the screen image this would be pD10) and change its name to Bore.

Now change the name of the last dimension added for the rim of the flywheel to Rim.

To the right of the parameter list is a column titled Expression. In the Expression box by the parameter Bore, enter Rim/10 and click the Apply button.



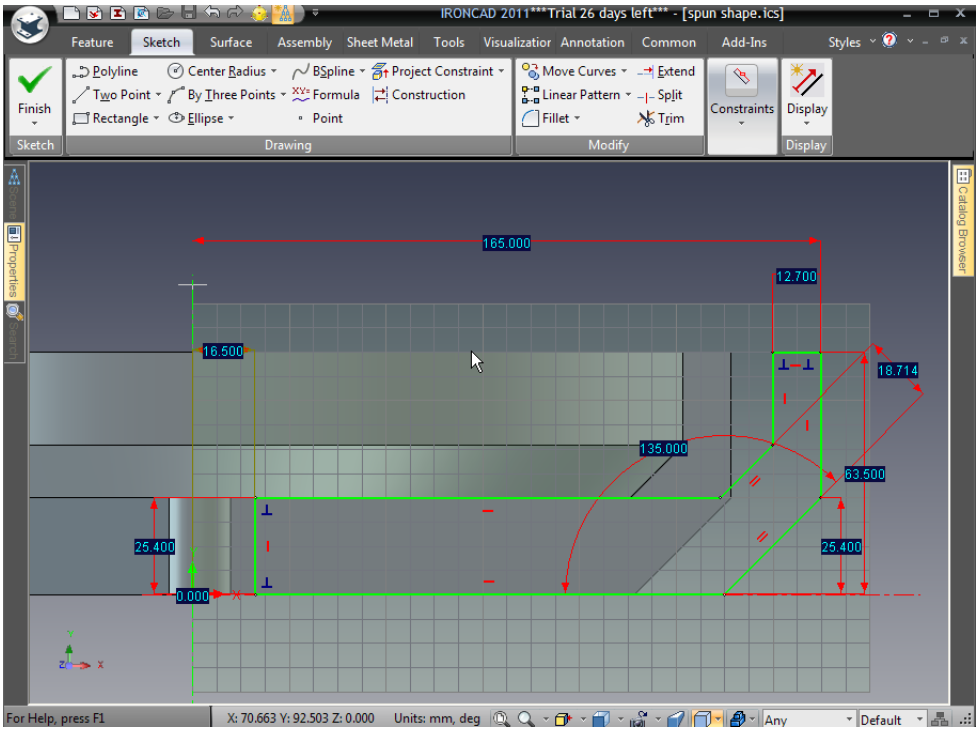
A number of things happen:

A. The value of Bore has changed in line with the current outside diameter, as IRONCAD dynamically evaluates the expression,
B. The dimension leader lines change color from red to a brownish green color indicating that that dimension is driven by an expression,
C. The system generated names for the dimensions are changed to the names given in the parameter table.

Click OK on the Parameter Table, IRONCAD will change the dimensions back to dimension values rather than parameter codes.

5. Change a Dimension Value to See the Expression Update

Right-click on the dimension defining the rim and change it to 165. IRONCAD will change the bore to 16.5. Next, try dragging curves around. Nothing changes confirming the sketch profile is fully constrained.



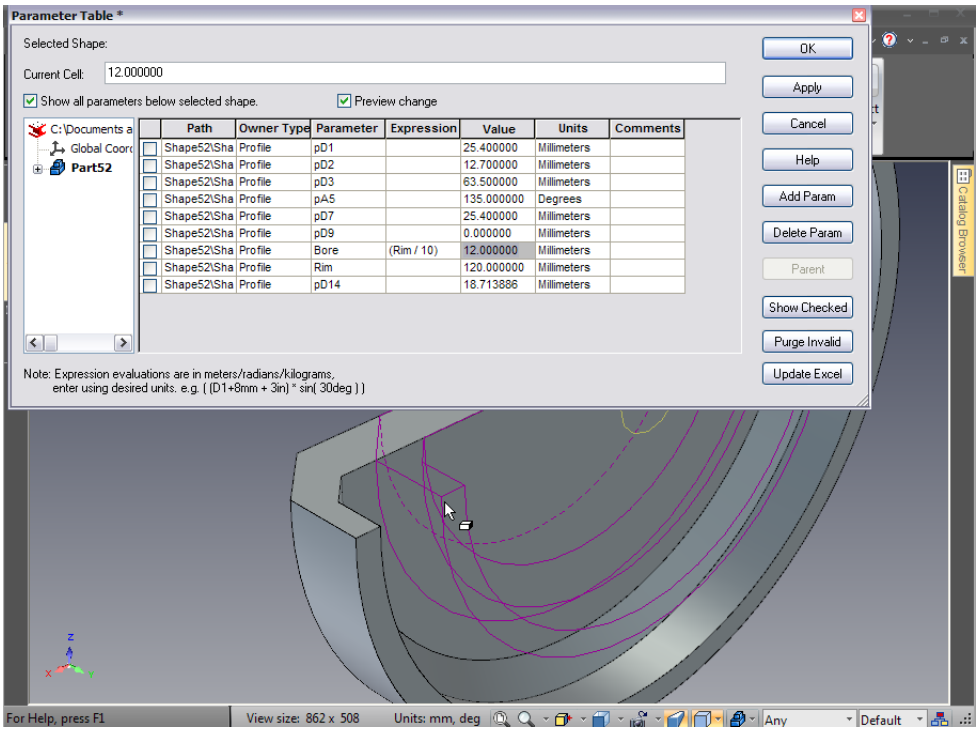
6. Finish the Sketch

Right-click and select Finish Drawing.

The flywheel geometry can be edited now by directly accessing the Parameter Table.

7. Access the Parameter Table to Edit

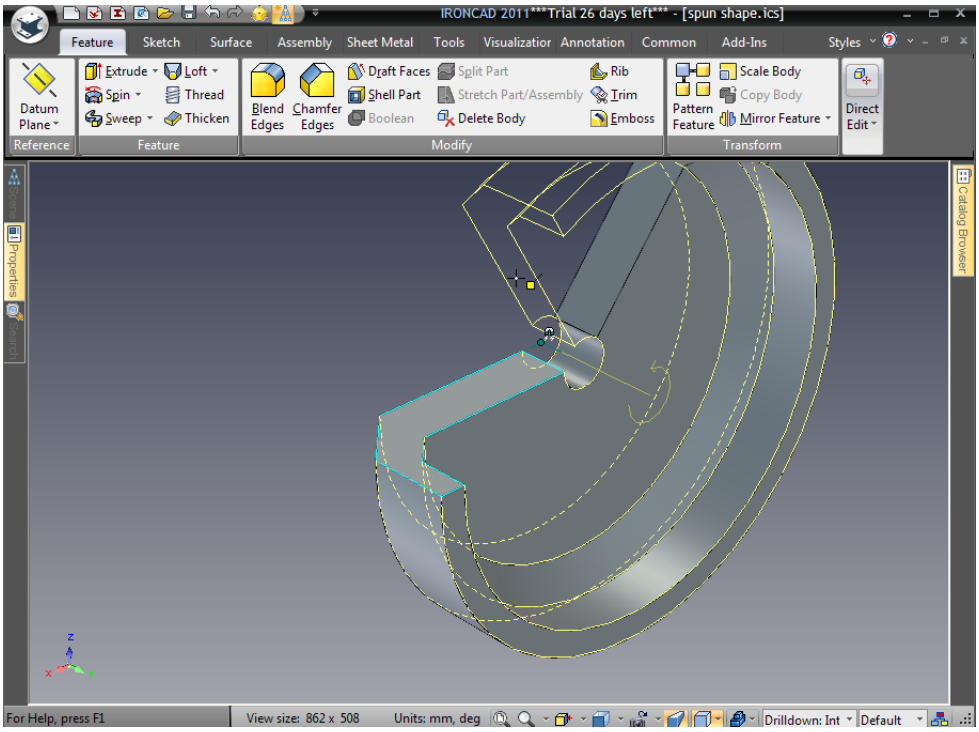
Right-click anywhere in the scene and select Parameters. Click in the check box at the top left hand side of the Parameter Table for Show all parameters and ensure Preview change is checked. Now change the value of the Rim parameter to 120.



IRONCAD will update the value of 'Bore' and give a preview of the resultant change on top of the current model. Clicking Apply will update the 3D model fully.

8. Close the Spin

To finish the model, select the part at the IntelliShape level and 'swing' the spin handle back to close the shape off again (or right-click the handle and set the spin angle to 360).



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

As a design evolves, a mix of styles and techniques will be used to accomplish the end result. This is a matter of personal preference or simply down to how the design evolves.

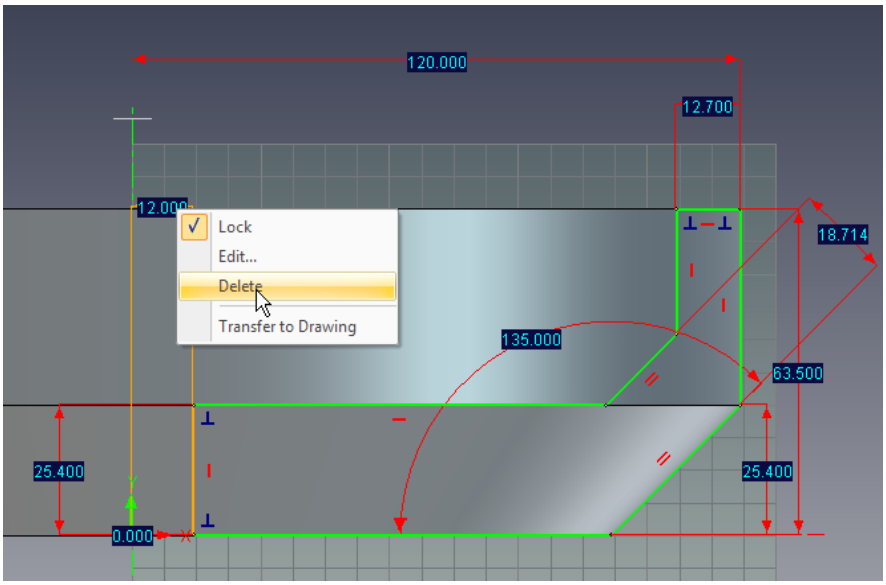
In this section, the controlled center bore will be replaced with a separate IntelliShape feature using the drag-and-drop method from the standard Shapes catalog and then a bolt hole pattern will be added using another catalog based feature.

4.6.1. Remove of the Sketch Center Bore

1. **Edit the Cross-section of the Flywheel.**

2. **Deleting the Bore Hole**

   Right-click on the horizontal dimension used to define the bore and select Delete from the pop-up menu.



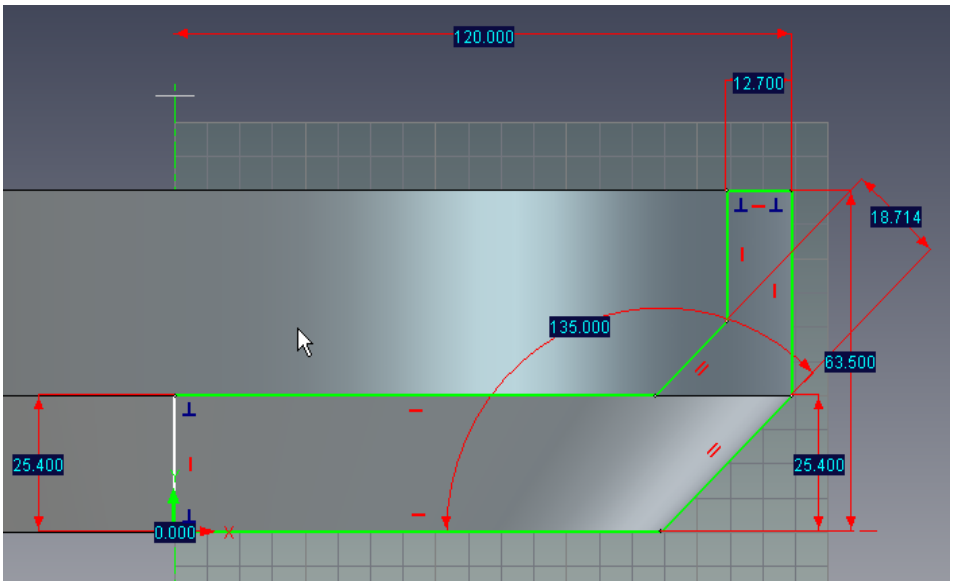
IRONCAD will display a warning saying the expression associated to that dimension will be removed. Click OK.

The curve defining the bore will return to white showing that it is under constrained. It is however still constrained by the vertical geometrical constraint and its end points controlled by the upper and lower horizontal curves as shown.

Click on the white curve and drag it until it snaps to the Y axis.

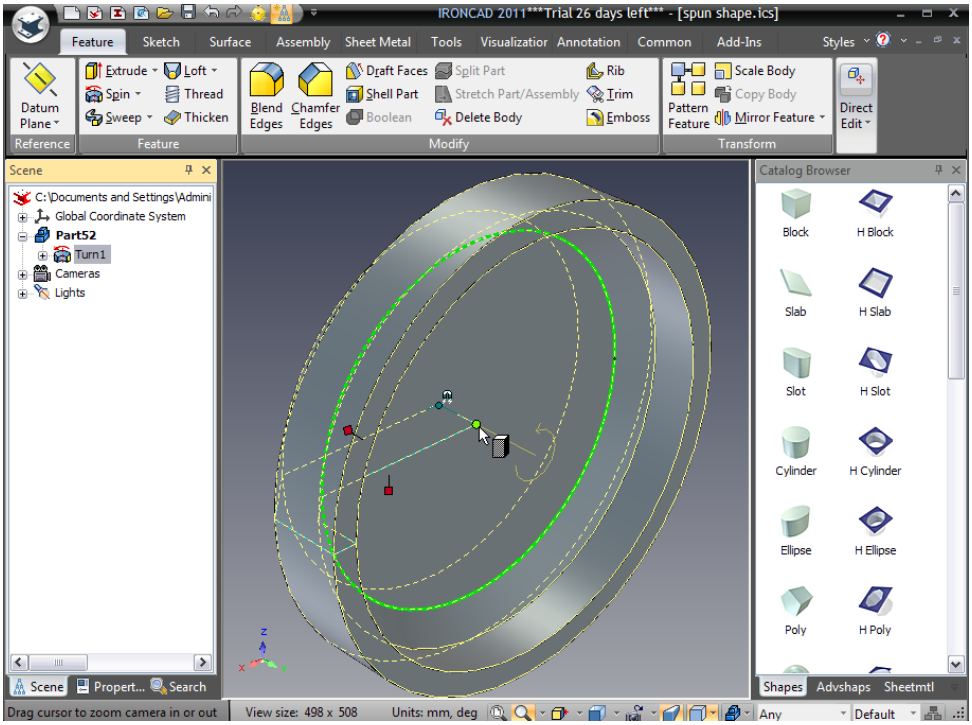
3. Finish the Sketch

Right-click and select Finish Drawing. The center bore will have been removed.



4. Add a Hole IntelliShape

Open the Shapes catalog on the right side of your screen and drag a H Cylinder shape to the center of the flywheel, as shown.



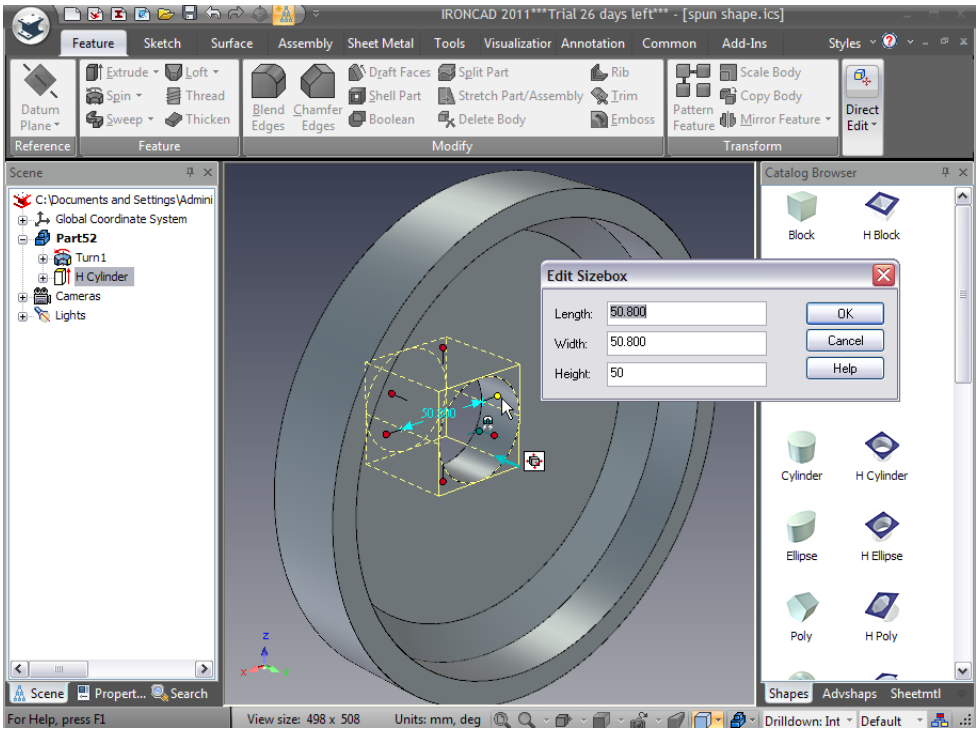
As you're dragging the H Cylinder, notice that when your cursor touches a surface it is highlighted in green with the edges in a brighter green. When

the center disk of the flywheel is touched the center point is highlighted with a dark green dot which turns to bright green when your cursor touches it. This behavior is an example of SmartSnap feedback. Use it to ensure that you snap to the exact center of the flywheel (i.e. when the center dot turns bright green drop the H Cylinder by letting go the mouse button).

The H Cylinder will be highlighted as an IntelliShape when it is dropped.

5. **Set the Hole Diameter**

Adjust the diameter of the hole to 50.8mm by right-clicking on the handle shown.

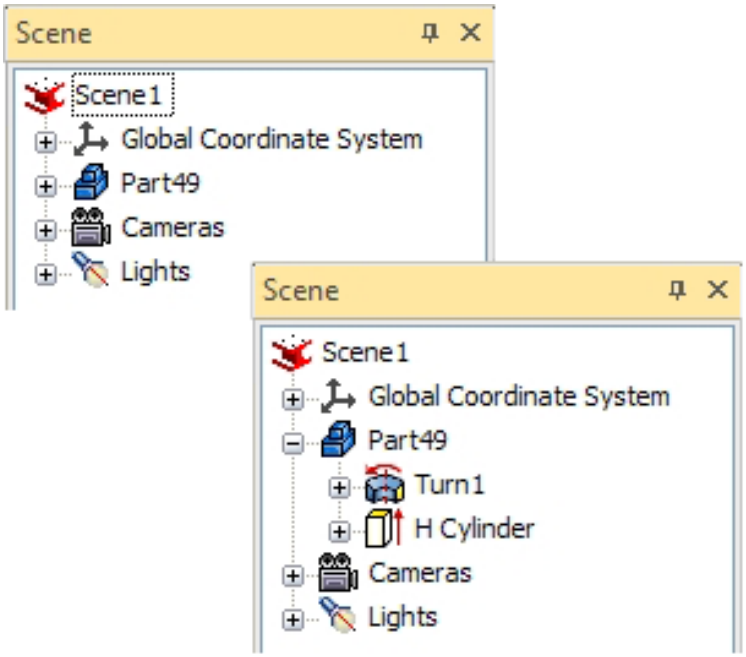


4.7. Setting Part and IntelliShape Names and Color

As the design starts to become more complex, it will become useful to start naming individual parts and their various IntelliShapes to help identify one feature from another. It can also be useful to add some color to a part to help differentiate parts.

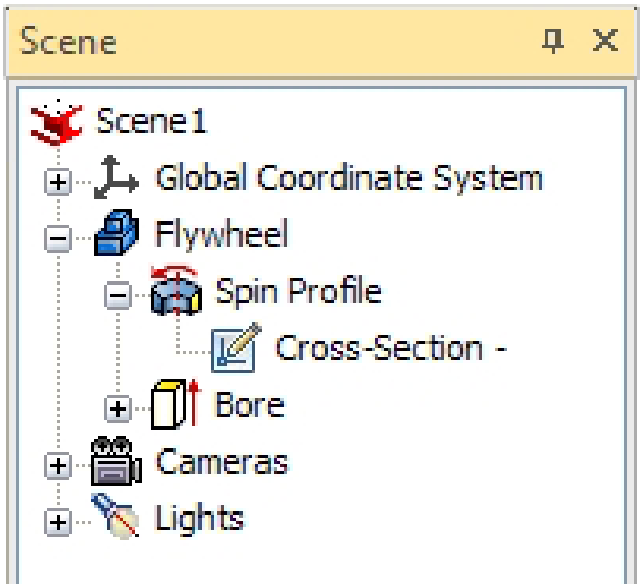
If the Scene Browser is not open, select the Scene Browser icon on the Quick Access Toolbar

The flywheel part is shown as Partxx (in our case Part49). In the image below, click the "+" symbol next to the part name to display its component IntelliShapes.



To change the name of the part, click the part name (Part49) once, wait a moment, and then click it again and the name will become editable. Change the name to Flywheel and press Enter.

Repeat this same procedure to change the name Turn1 to Spin Profile and H Cylinder to Bore



In the above image, you can see that by expanding the "+" by Spin Profile you get direct access to the cross section without having to drill down through a complex set of IntelliShapes.

4.7.1. Changing the color of the part

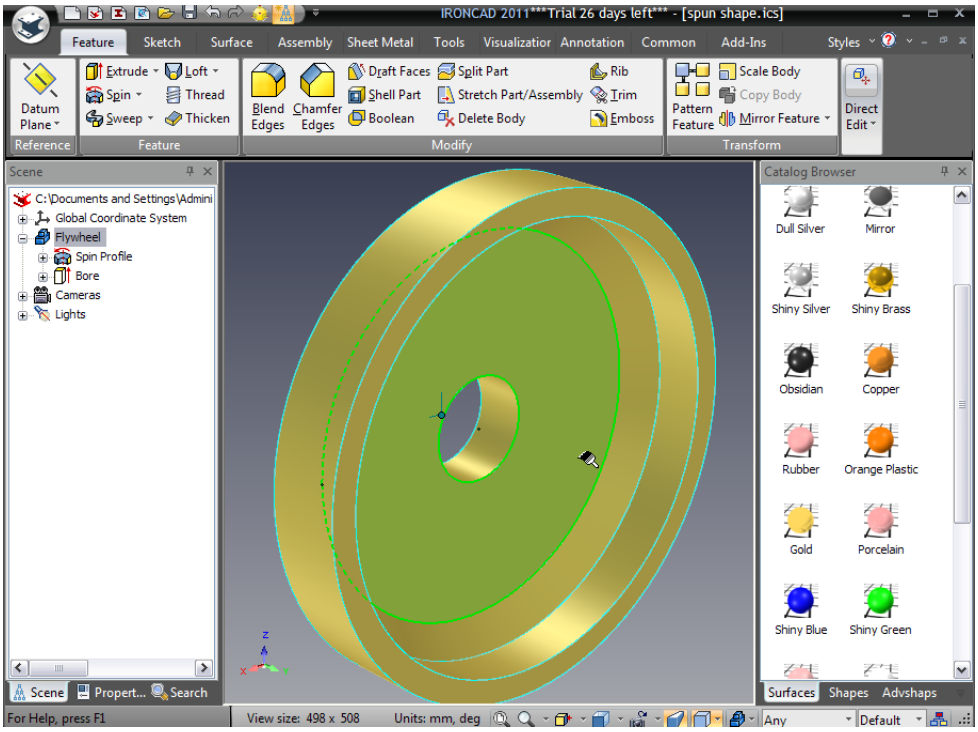
There are 3 ways to change the color of the part:

A. You can simply set a color to differentiate it from another part using SmartPaint.
B. You can drop a catalog surface finish onto the part if you want to produce a good quality color image
C. You can tune every last aspect of the surface finish to make a truly photorealistic image.

The latter will be covered in detail in its own section.

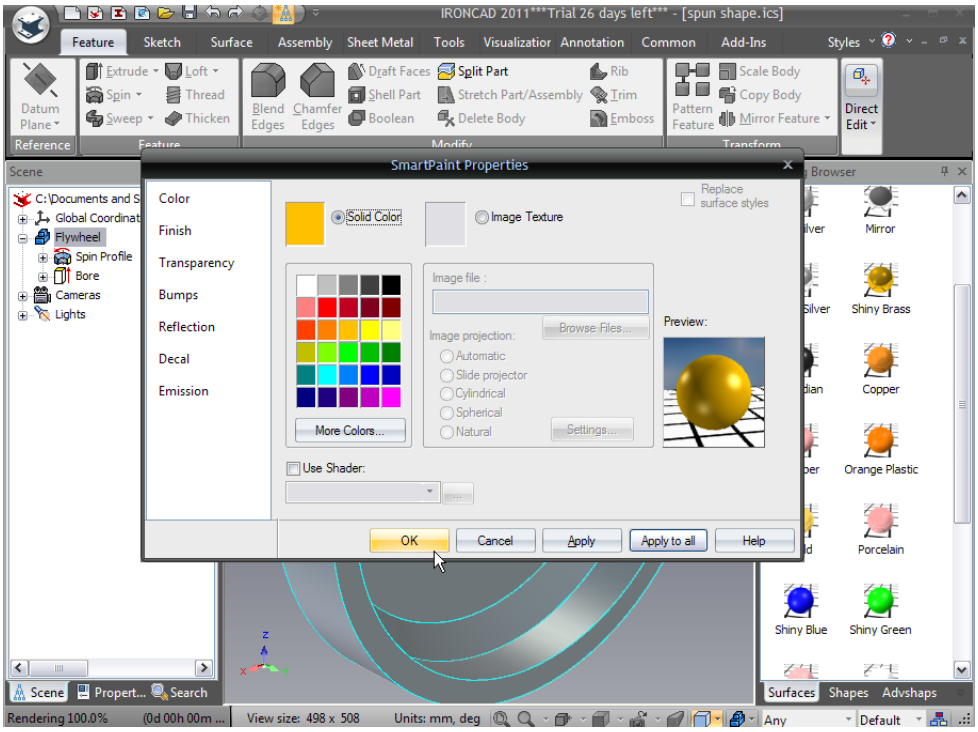
1. To use a predefined surface finish:

A. Open the Surfaces Catalog. If it's not available, select the Common tab in the Windows Ribbon bar. In the Catalogs group, select Open. IRONCAD will display the file browser and navigate to the Scene folder beneath Catalogs and select Surfaces.icc.
B. Drag, for example, Gold onto the Flywheel



2. To use SmartPaint:

A. Right-click on the flywheel
B. Select SmartPaint from the drop down menu
C. Select a color from the color swatch
D. Click OK



4.8. Using the Advance Shapes Catalog

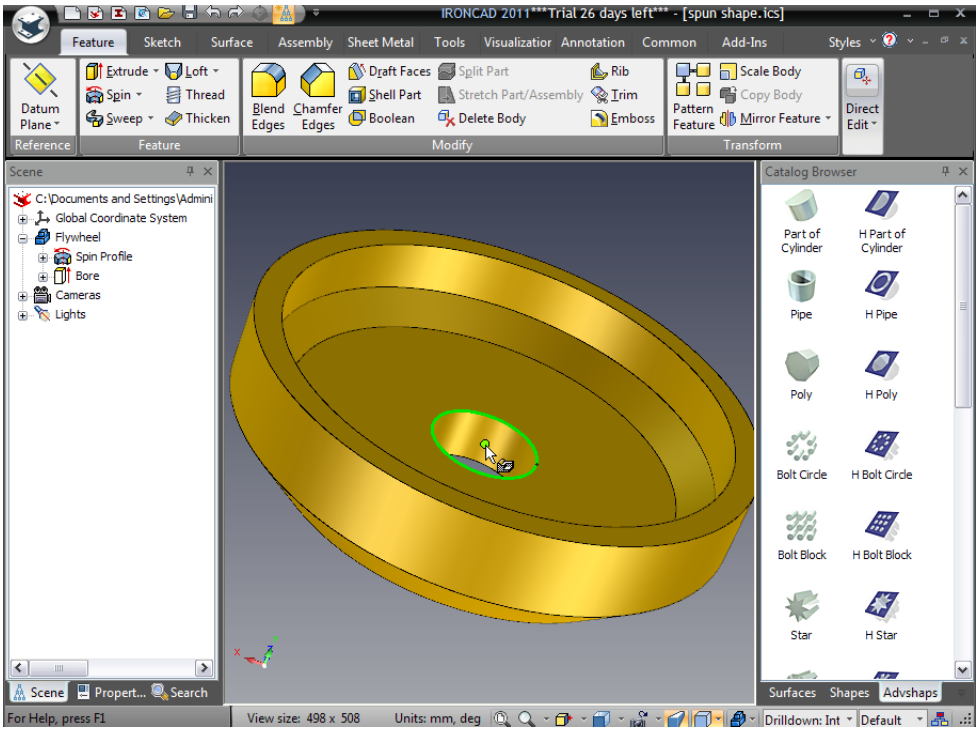
The Advance Shapes Catalog (Advshapes.icc), extends the capabilities of regular IntelliShapes. Each shape in this catalog contain extra variable information for controlling the shape. Essentially, the catalog can be viewed as Parametric IntelliShapes. This part of the exercise will show one of the shapes and how it can be used.

1. **Open the Advshapes Catalog.**

   If the catalog tab is not visible, click the Active Files down arrow to the right of the catalog tabs and select Advshapes from the list.

2. **Add the H Bolt Circle**

   Drag the H Bolt Circle from the catalog using SmartSnap feedback as used to position the flywheel bore hole to center the feature onto the flywheel. However, this time highlight the bore as it is touched and not the face of the flywheel as shown next.



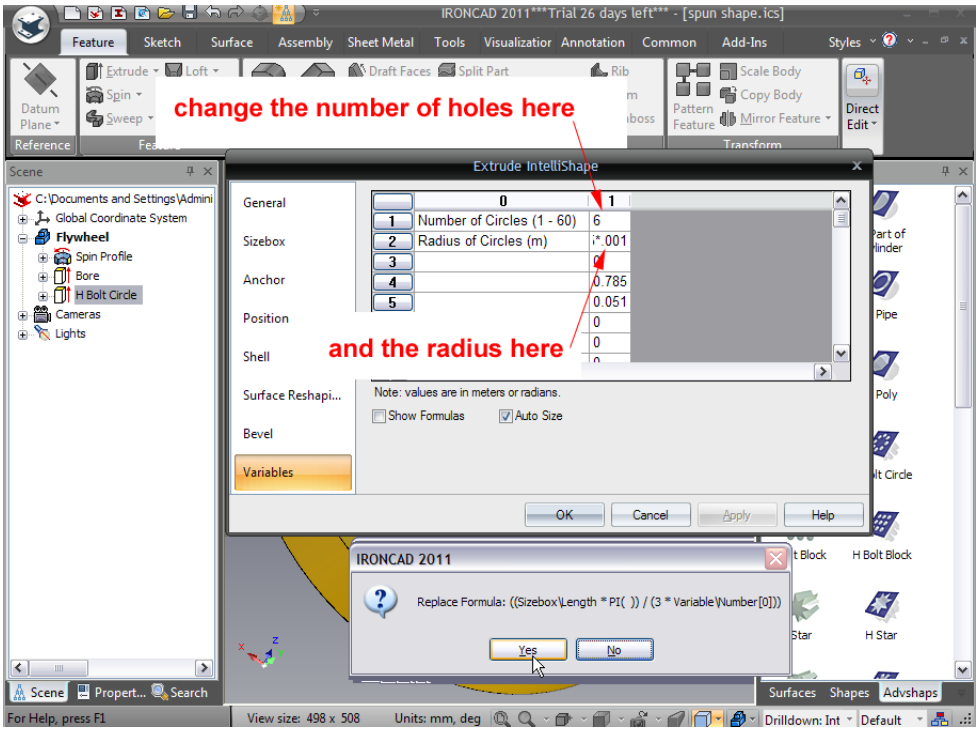
To ease the snapping procedure, make sure the view is oriented as shown. The center point of the bore must be over the bore surface otherwise it will not be selectable. This is because SmartSnap feedback only works when the cursor is over a face. IRONCAD cannot snap to "air." In other words, the flywheel must be oriented such that your cursor can touch both the center point and the cylinder face at the same time.

3. Set the Size of the Bolt Circle

While the H Bolt Circle IntelliShape is selected, right-click on one of its Width or Length handles. Select Edit Sizebox and set the value to 101.6. This gives the bolt pattern a radius of 50.8 mm.

4. Set the Variables in the Pattern

The scene browser will show an entry H Bolt Circle. Right-click on the entry and select IntelliShape Properties from the drop down menu.



Select the Variables tab and change the Number of Circles value to 6 and the Radius of Circles value to 6.35 * .001 (the .001 multiplier is necessary because the variable units are meters). As shown previously, a message will appear confirming the replacement of the formula for the Radius of Circles. Click 'Yes'.

On the Extrude IntelliShape properties box click OK to accept the changes.

4.9. Saving the part

At this point in the design, it would be good to save the part. As with all software tools it is wise to save work regularly. IRONCAD does not force each part used in the design to be saved to an individual file (it is optional). This decision really depends on the work flow and personal style.

In this case, the model is being designed on the fly in the context of other parts within the assembly. This is referred to as 'In Context' design or 'Top Down' design. To those ends, there may have a final structure in mind. However, this structure may change as the design evolves. Again, IRONCAD's freedom enables the design methodology to evolve therefore all that is needed is to save the design!

From the IRONCAD home button, select File followed by Save. Navigate to the appropriate work folder, if different to the one displayed by IRONCAD, enter the name of the part ‘Flywheel Assembly’ and click Save.

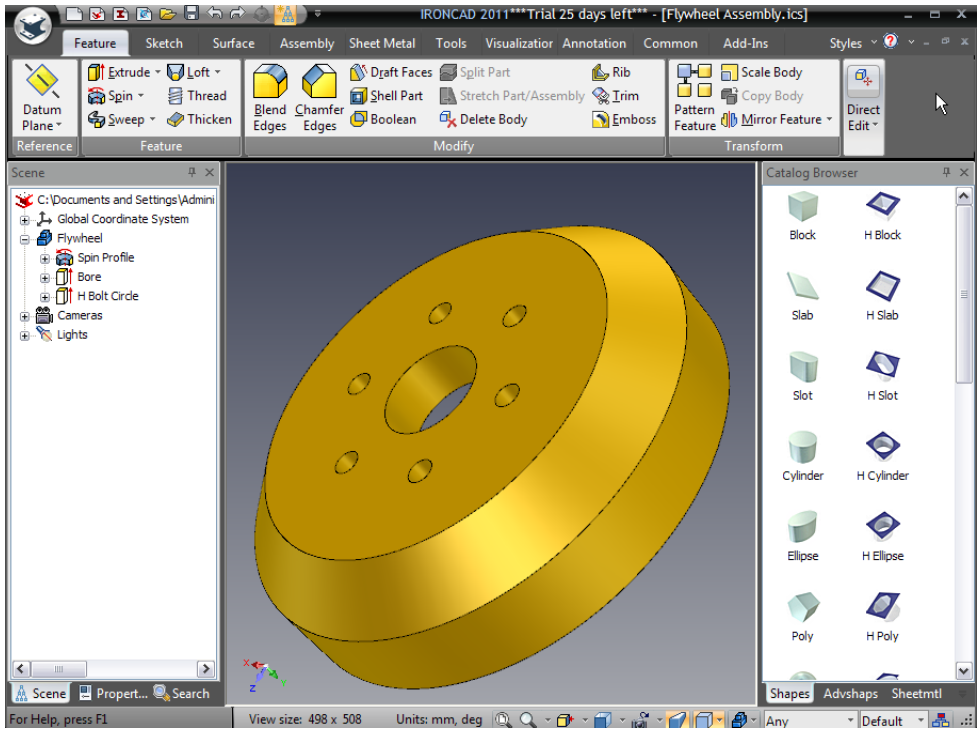
4.10. Working with Multiple Parts in a Single File

As mentioned above, IRONCAD can work “in Context”. This applies to adding multiple parts to a single file. In the following exercise, we will add additional parts in context to the flywheel we already created.

4.10.1. Creating the Crankshaft

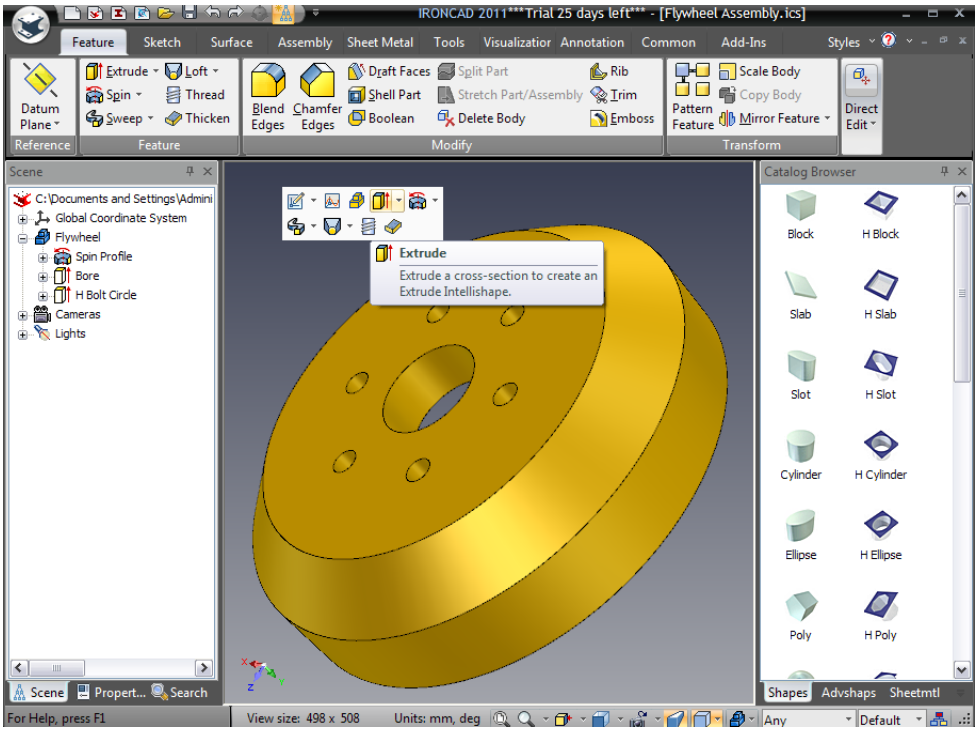
Following the theme of Top Down design, this section extends the design to create a simple crankshaft to mate with the flywheel.

1. Position the View of the Flywheel as Shown.

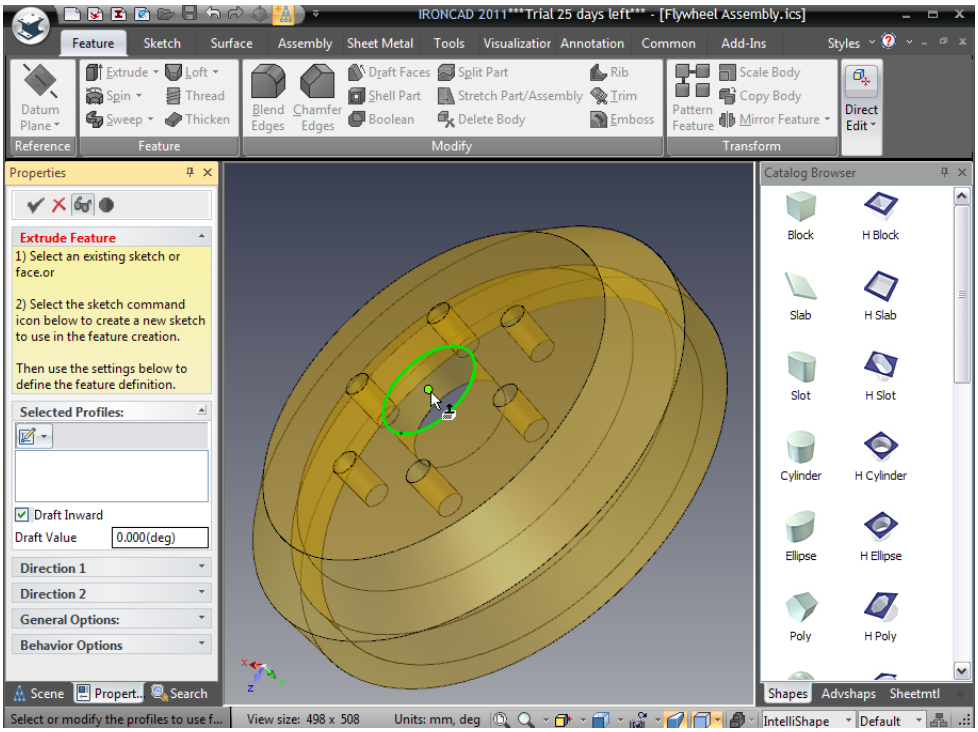


2. Start an Extrude as a New Part

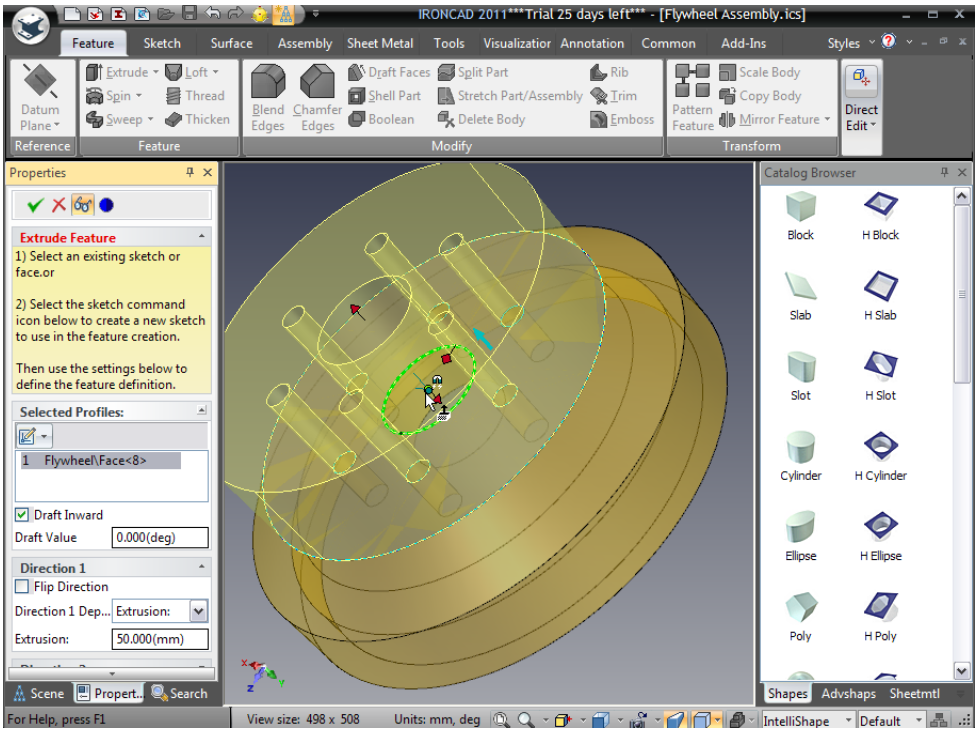
With nothing selected, press the ‘S’ key and select the Extrude option.



IRONCAD will display the familiar Command Property Browser. Select Create a New Standalone Part. The Flywheel part will become translucent at this point. Navigate to the center of the Flywheel bore and when the center dot turns bright green, click the left mouse button.



IRONCAD has entered a reference to the Flywheel face in the Selected Profiles dialog box and immediately produces a preview of a solid part created from all the edges present on the rear face, as shown next.



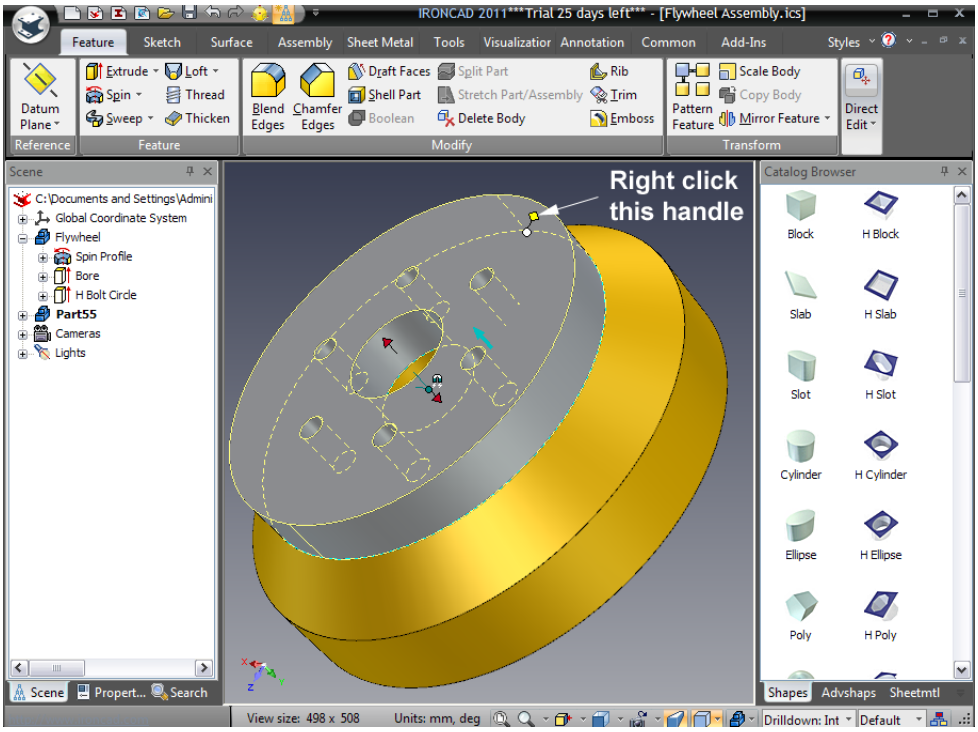
In the Direction 1 panel, set the Extrusion distance to 25.4 and ensure Add material is checked in the General Options panel. Right-click in the scene and click OK, or click on the green check mark at the top of the Command Property Browser.

With 4 simple interactions IRONCAD has created a new part to mate with the Flywheel.

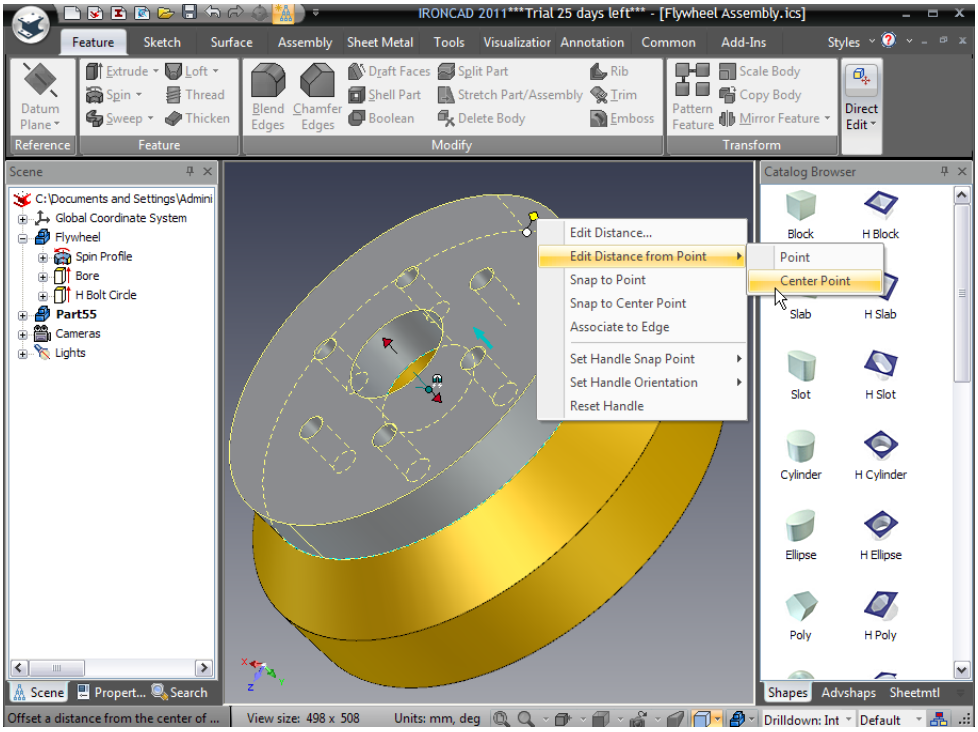
Using the 2D editing tools encountered previously the Crankshaft Flange will be sized appropriately.

3. Size the New Crankshaft

Select the new part at the IntelliShape level and move the mouse pointer somewhere near the cyan curve defining the outer diameter of the part. Right-click on the handle displayed by IRONCAD such as shown in the image below:



From the drop down menu, select Edit Distance from Point, followed by Center Point.

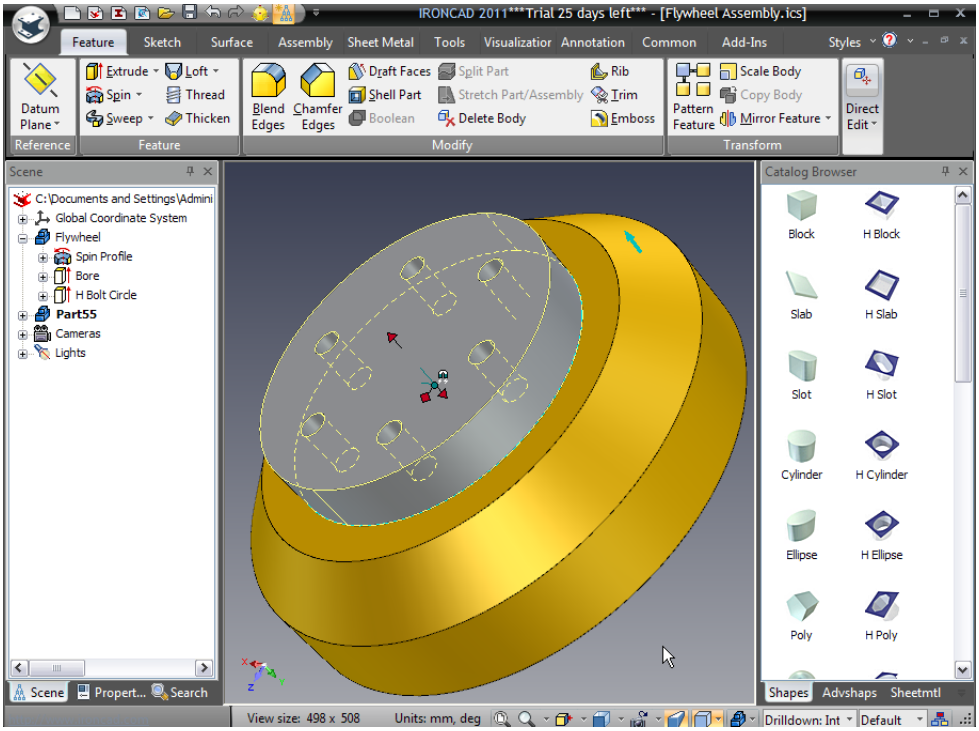


Select any outer cylindrical surface or circular edge and enter 76.2.

4. **Remove the Center Bore**

To remove the bore from the part, repeat the operation by right-clicking on the handle attached to the curve defining the bore. But this time, select 'Snap to Center Point' from the drop down menu, and select the outer rim of the new part.

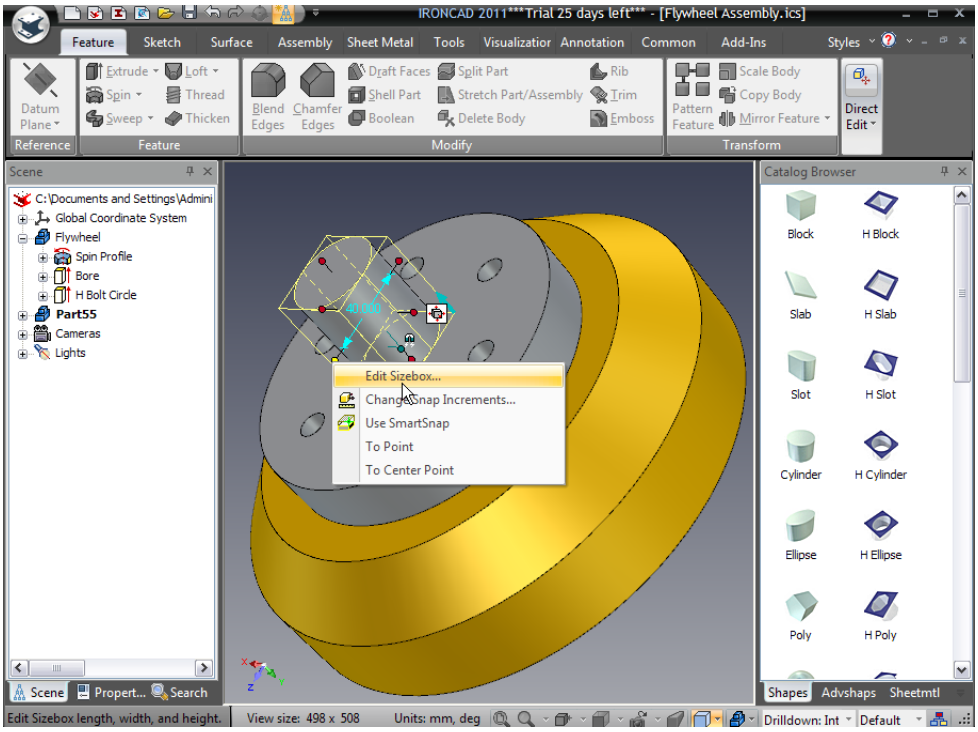
IRONCAD will remove the center bore hole. The assembly should look like the following.



4.10.2. Adding the Shaft

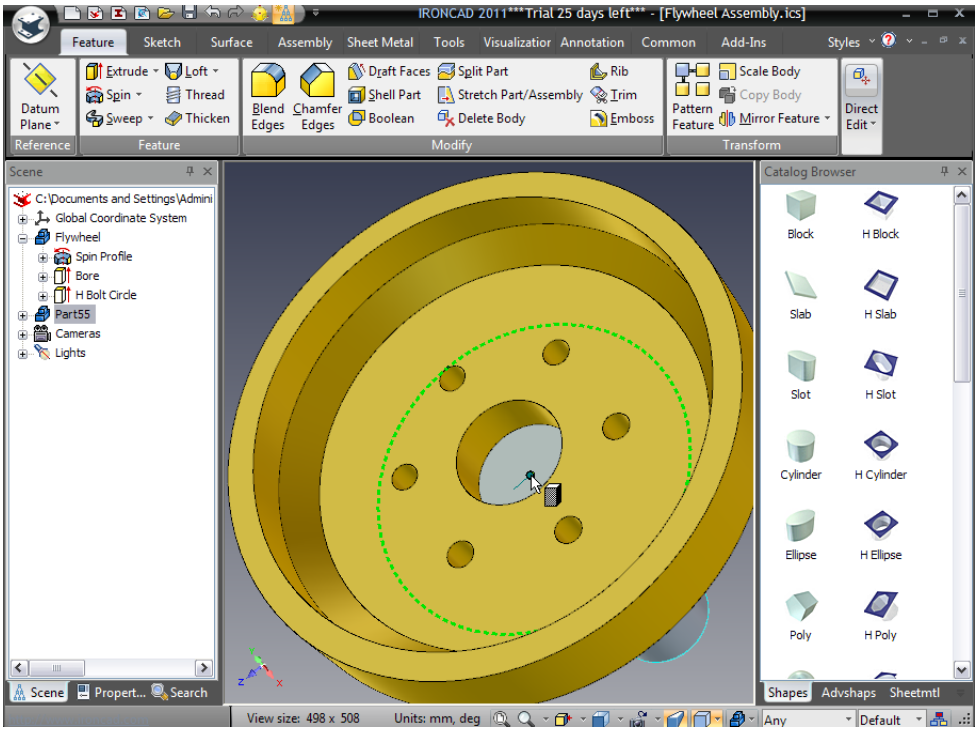
1. **Add a Cylinder to the Outer Side**

Drag a Cylinder IntelliShape from the Shapes catalog and snap it to the center of the crankshaft, as shown. Adjust the diameter to 50.8 and the Height to 152.4.



2. Add a Cylinder to the Inner Side

Rotate the model so you can see the inside of the flywheel and drag and drop a new cylinder onto the other side of the crankshaft, as shown.

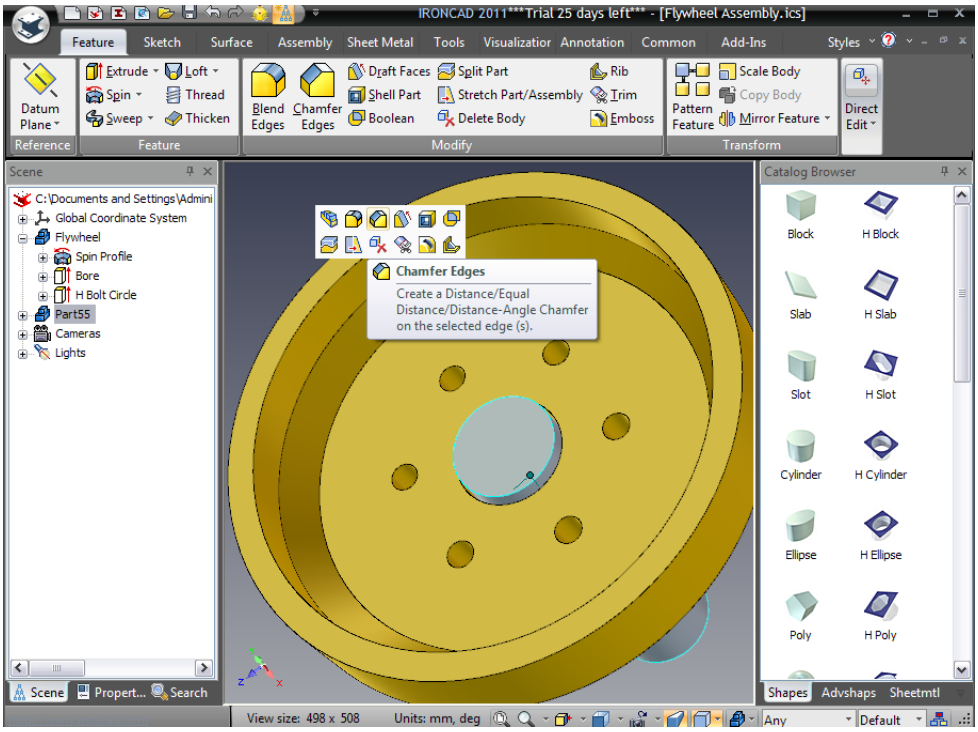


IRONCAD will display SmartSnap feedback showing the edge of the Crankshaft flange and its center point. Dropping the new cylinder onto the bright green centre dot will ensure it is central to the Crankshaft even though there is not much of the Crankshaft flange visible.

Adjust the diameter to 48.3 (i.e. Sizebox Length or Width) and the Height to 31.8.

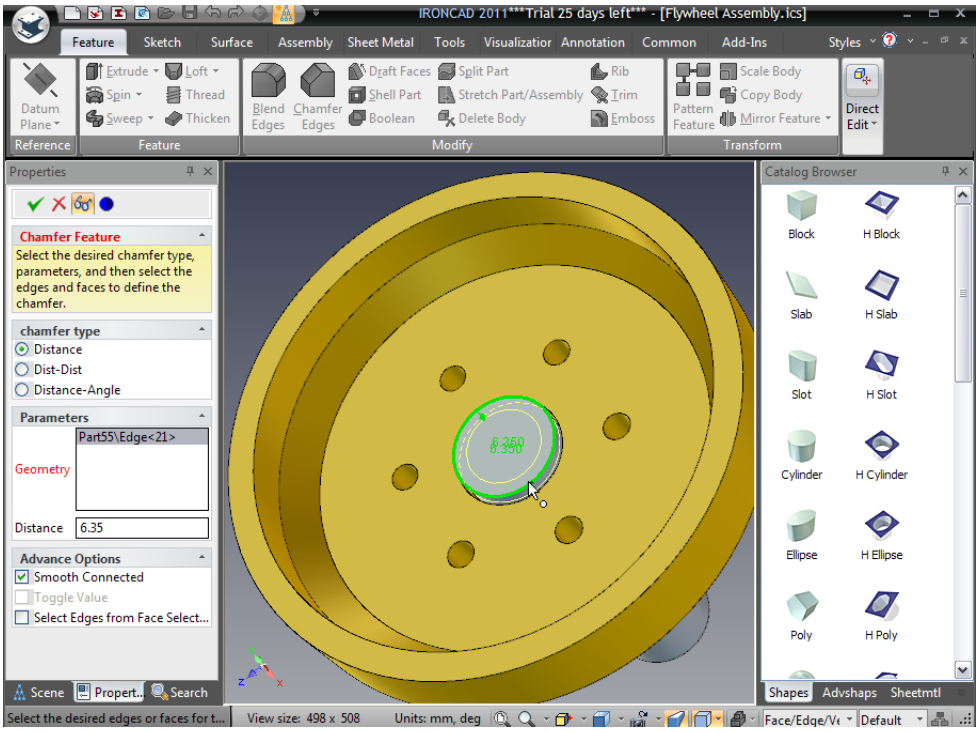
3. Add a Chamfer to the Cylinder

To add a chamfer to this new cylinder, click on the Crankshafts part name in the scene browser to highlight the part in cyan (i.e. part mode). Press the S key and select the Chamfer Edges tool. Alternatively click on the Chamfer Edges icon in the Modify panel found under the Feature tab of the main Windows Ribbon bar.



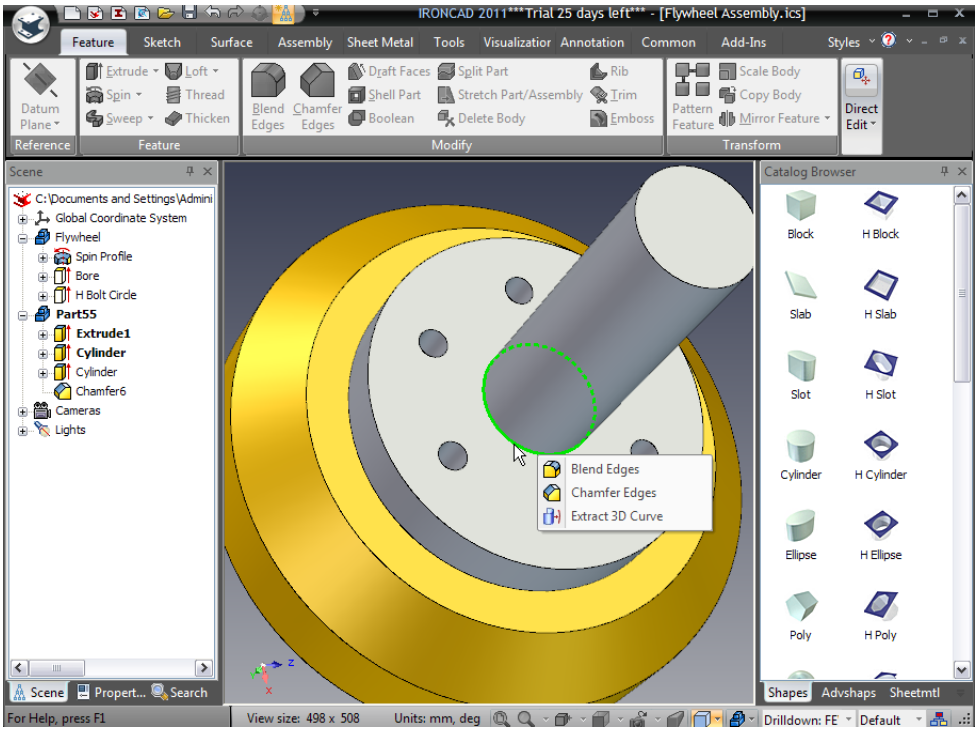
IRONCAD will display the Command Property Browser. Ensure the chamfer type is set to Distance and set the distance to 6.35.

Select the end face of the last cylinder created. IRONCAD will display a preview and the chamfer. Accept the chamfer by either pressing the Return key or by clicking the green check mark at the top left of the Command Property Browser.

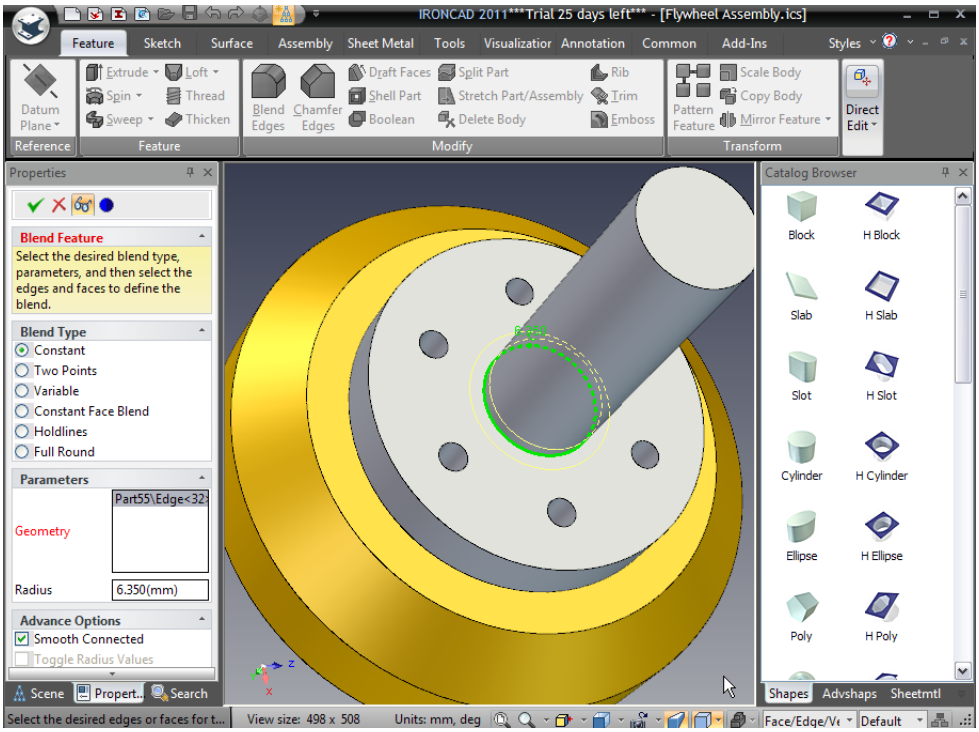


4. Add a Blend to Between the Cylinder and Base

Rotate the view to add a blend to the interface between the flange and the shaft. Introducing an alternative method to create blends and chamfers, position the cursor over the edge created at the intersection of the shaft and the flange. Use the left mouse button, click once to highlight the shape at the IntelliShape level and a second time to highlight the edge in green. When the edge is highlighted in green, press the right hand mouse button and select Blend Edges from the pop-up menu.



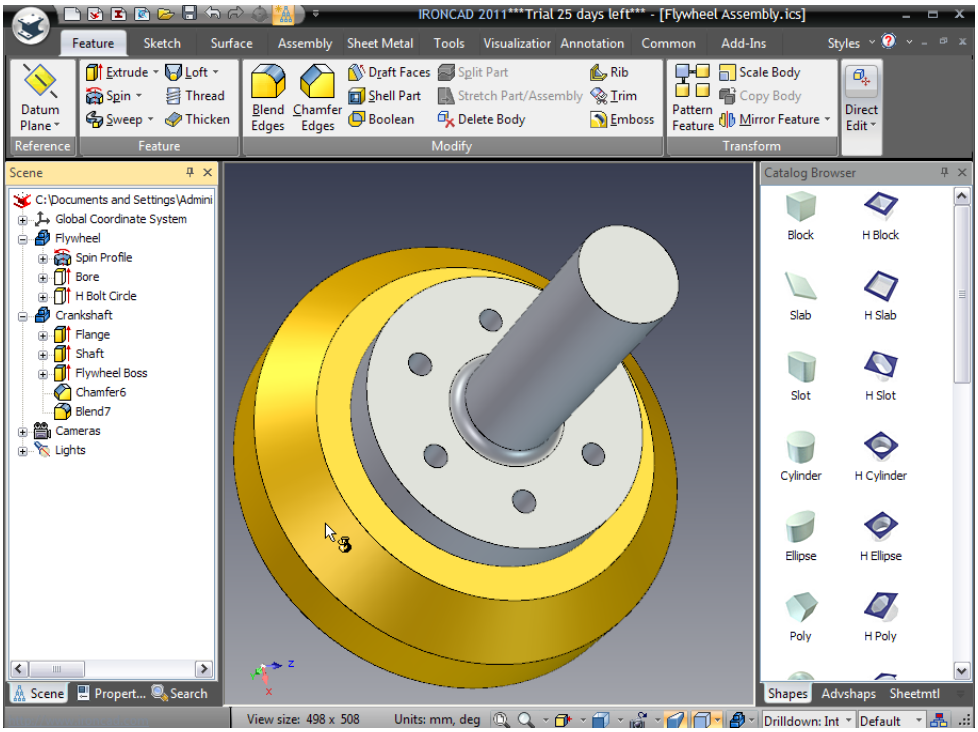
Ensure the Blend Type is set to Constant, set the radius to 6.35 (IRONCAD remembers the previously used value for the radius so no action should be needed). IRONCAD displays a preview of the blend, hit the Return key or select the green tick.



Note: A blend is sometimes referred to as a ‘fillet’ or a ‘round’. In addition, clicking on a face resulting in all the edges being highlighted in green and then selecting the blend or chamfer commands will result in IRONCAD attempting to put a blend on every edge highlighted.

5. Rename the Crankshaft and Save

Finally rename the part to Crankshaft, the extrusion to Flange and the two cylinders to Shaft and Flywheel Boss and save the scene.



4.10.3. Creating the Crank Arm

A Crank Arm is required to complete the design to mount on the end of the crankshaft.

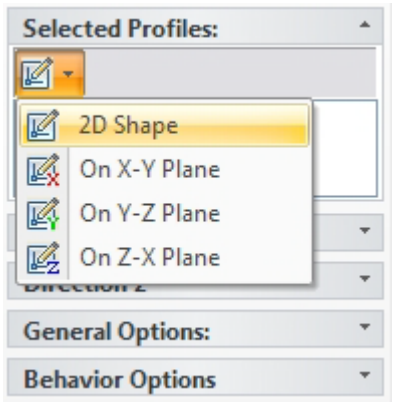
1. Create an Extrude Part

With nothing selected, either press the S key and select Extrude or click on the Extrude icon on the Feature tab of the Windows Ribbon bar.

IRONCAD will display the Command Property Browser. Click ‘Create a new Standalone Part’.

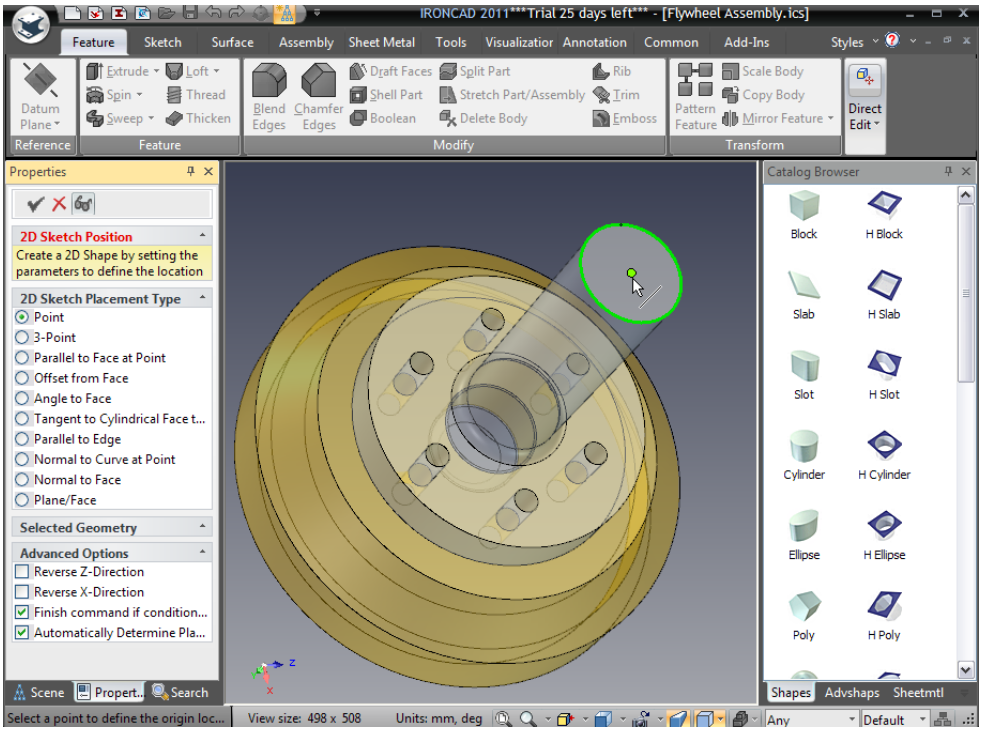
2. Select New 2D Shape

Before selecting anything in the scene expand the icon in the Selected Profiles panel (below), and select 2D Shape.

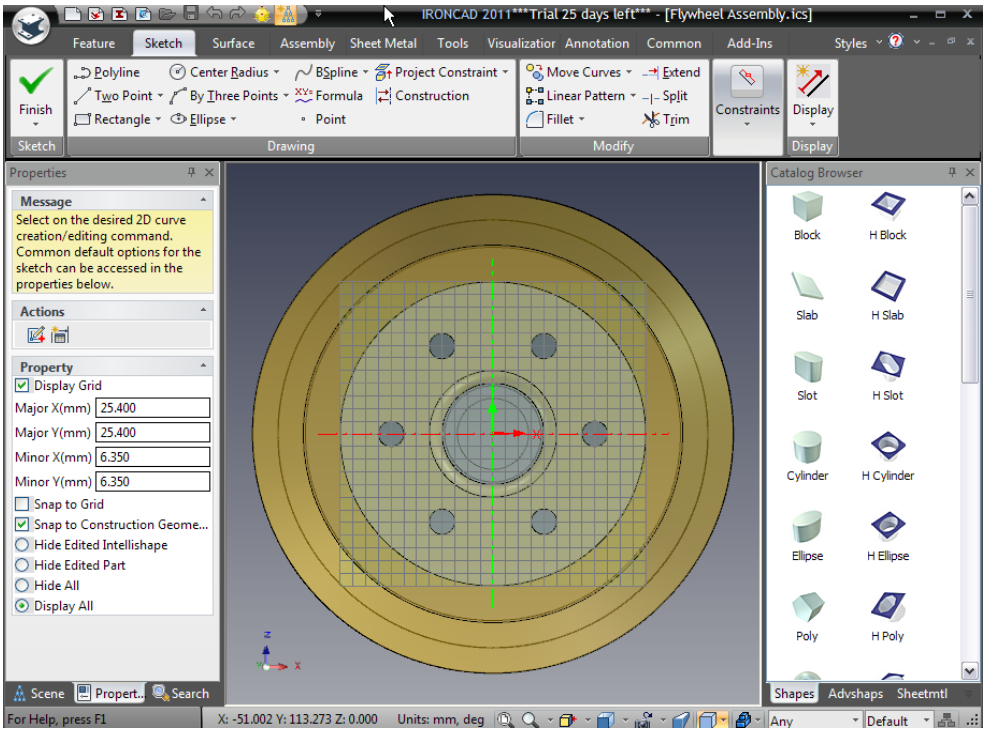


3. Place the Sketch at a Point

The Command Property Browser will now present a number of options to place the sketch. Select ‘Point’ and indicate the centre point of the end of the shaft.



IRONCAD will rotate the view and present the now familiar sketch plane. It should look something like the next image.



4. Project Constrain Existing Geometry

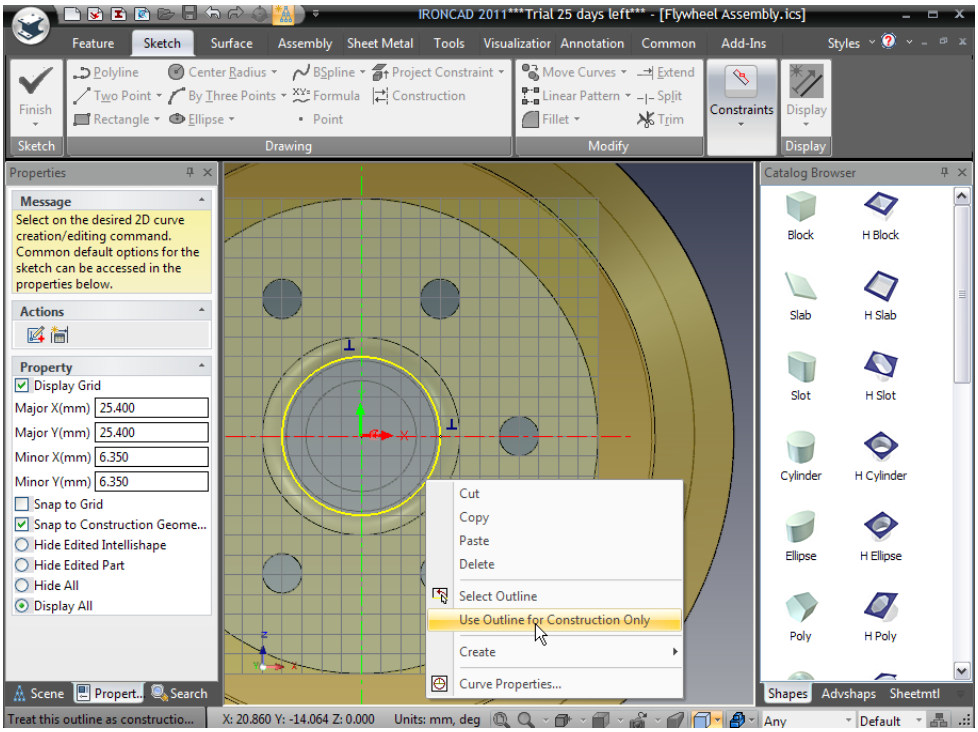
Either press the 'S' key and select Project Constraint or click on the Project Constraint tool on the Drawing panel of the Sketch tab. Select the end face of the shaft. IRONCAD will project a circle onto the sketch plane and constrain the projected circle back to the diameter of the shaft. Any changes to the shaft will result in the diameter of the circle changing. The circle will be drawn in green indicating that it is constrained.

5. Exit the Project Constraint Tool

After projecting the edge, turn the projection tool OFF by clicking it or press the Esc key once.

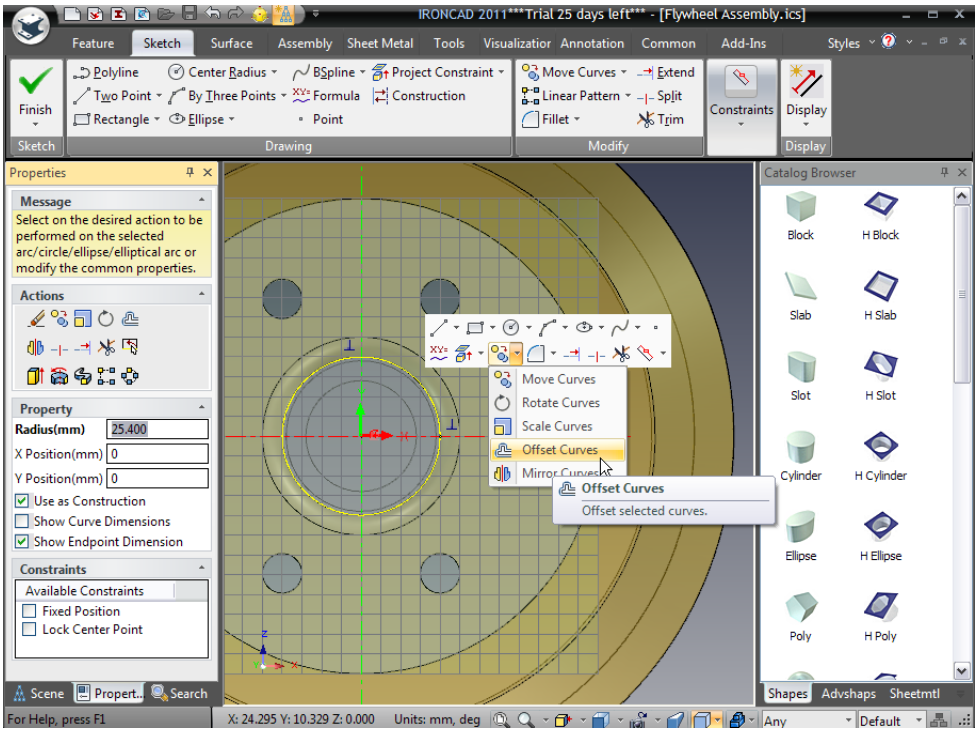
6. Turn the Projection into a Construction Element

Zoom in so you maximize the view of the circle on your screen. Right-click on the circle and select 'Use Outline for Construction Only'. IRONCAD will change the circle to a dark blue thin line and will not include this element in the finished profile.

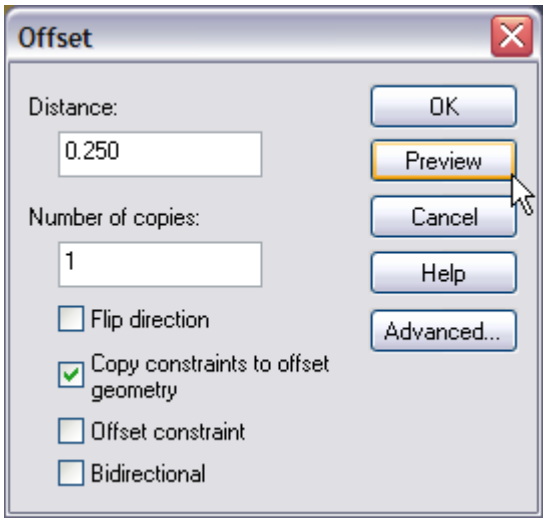


7. Offset Curves

With the circle highlighted, press the S key and expand the Move Curves icon and select Offset Curves.



IRONCAD will display the Offset Curves dialog box. Set the Distance to 0.25.

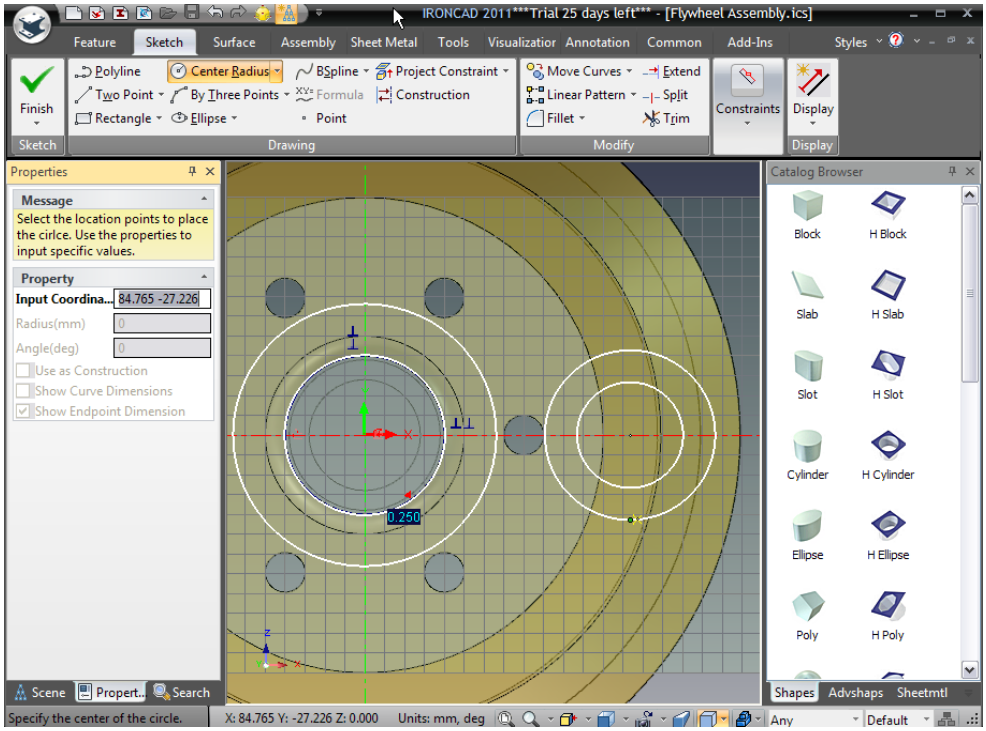


Click Preview. If IRONCAD draws the new geometry inside the selected circle, click on Flip direction. As the distance being offset is very small enter a value such as 5, check the offset side and then set the value back to 0.25. Ensure Copy constraints to offset geometry, and Offset constraint are both checked and complete the offset by clicking OK.

IRONCAD will produce a circle that is 0.25mm greater in radius than the shaft. By applying the Copy constraints and the Offset constraint, any changes in the shaft diameter will be reflected by this feature. In short we have defined the fit between the shaft and the bore of the Crank. Also, notice that the offset geometry is set to real geometry and not construction. New curves are set to be real curves even if generated from construction elements. To create construction curves by default, it can be enabled in the IRONCAD Properties Panel under an item labeled “Use as Construction”.

8. Create Additional Circles

Now use the Circle Center Radius tool to create three more circles, as shown, for a total of two pairs of concentric circles.



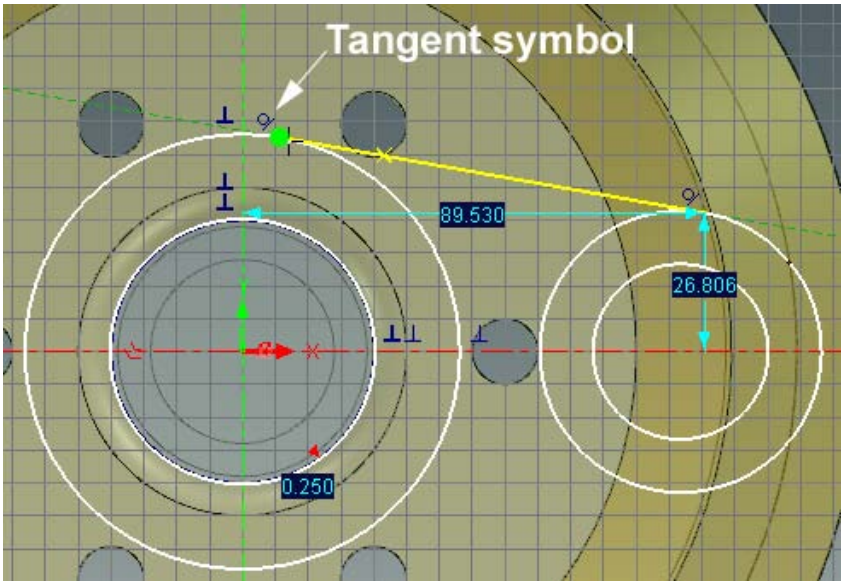
4.10.4. Creating Tangent Lines

To complete the Crank, the two outer circles need connecting with tangent lines and then trim away portions of the circles to form a single closed profile with two holes.

1. Create a Tangent Line

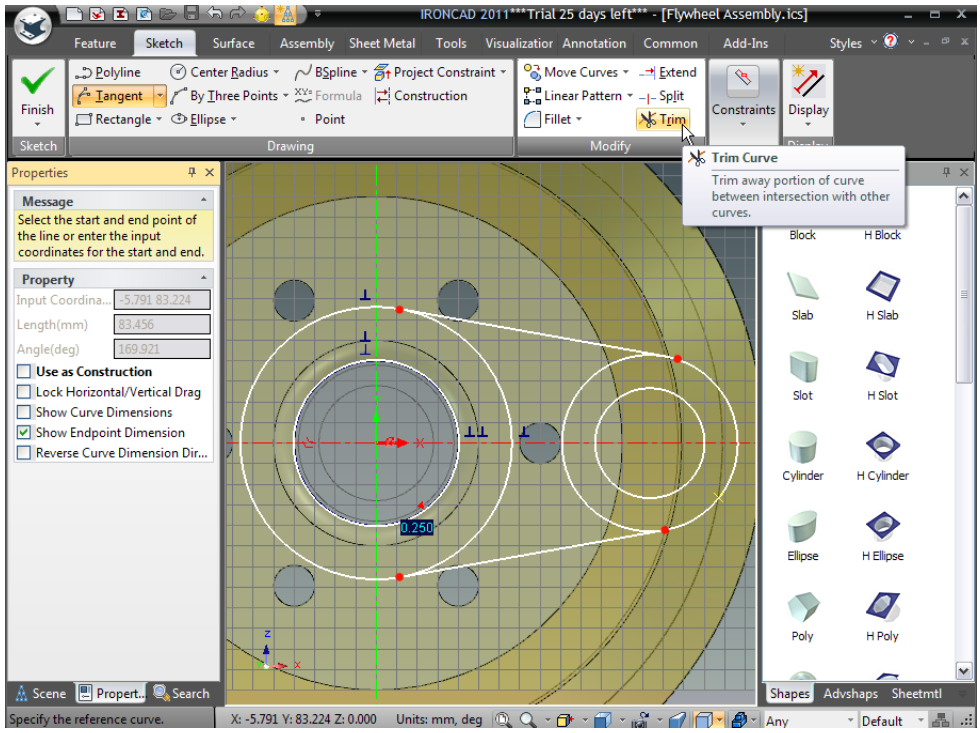
Press the 'S' key and click the pull down by the Two Point line tool and select Tangent.

Move the cursor along the top edge of the right outer circle. IRONCAD will place a yellow X on the circle where the circle is touched and draw a green dot on the circle as the cursor traces around the circle. Click the left mouse button and drag the cursor over and towards the left hand outer circle. IRONCAD will give feedback of the line angle and position. Touch the left hand circle with the cursor and the line will drop onto the circle and trace around the circle as the cursor moves along the curve. As the cursor approaches the tangent point, IRONCAD will snap the line to the tangent point, changing the tracing green dot to a large green dot and draw a small blue symbol indicating you have a tangent.



Repeat the process for the lower tangent line. Press the Esc key to cancel the operation.

IRONCAD will now have 4 white circles, two lines and 4 red dots (the red dots showing the end of the lines are not connected to any other geometry).



2. Trim the Excess Geometry

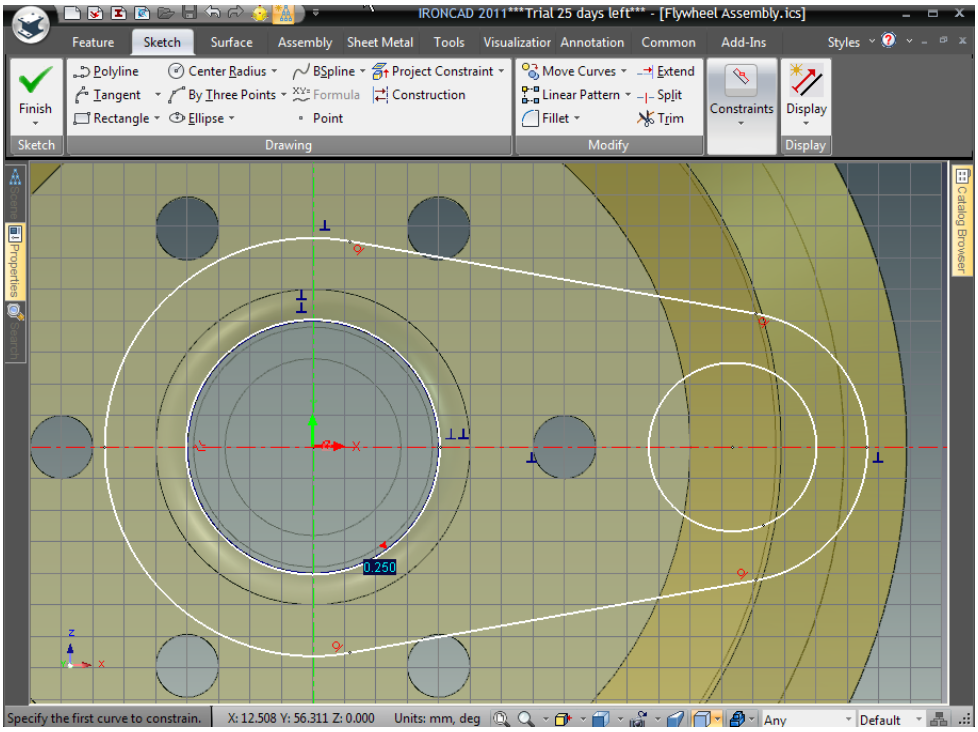
Select the Trim tool from the Modify panel and select the portion of the circle between the 2 tangent lines. The red dots should vanish indicating the geometry is a cleanly connected set of curves.

4.10.5. Creating Tangency and Concentric Constraints

As with the Flywheel, it would be good to complete the design intent by fully constraining the 2D sketch.

1. Lock Existing Profile Indicators or Add New Constraints

If you can see the blue tangency icons, right-click on each of them and select ‘Lock’. If you cannot see the tangency icons, select the ‘Tangent’ tool from the constraint panel. Add the tangent constraint by clicking on the arc and then on the line. This needs to be repeated for each line/arc connection. IRONCAD will then show a red tangency symbol at each junction.



2. Create Concentric Constraints

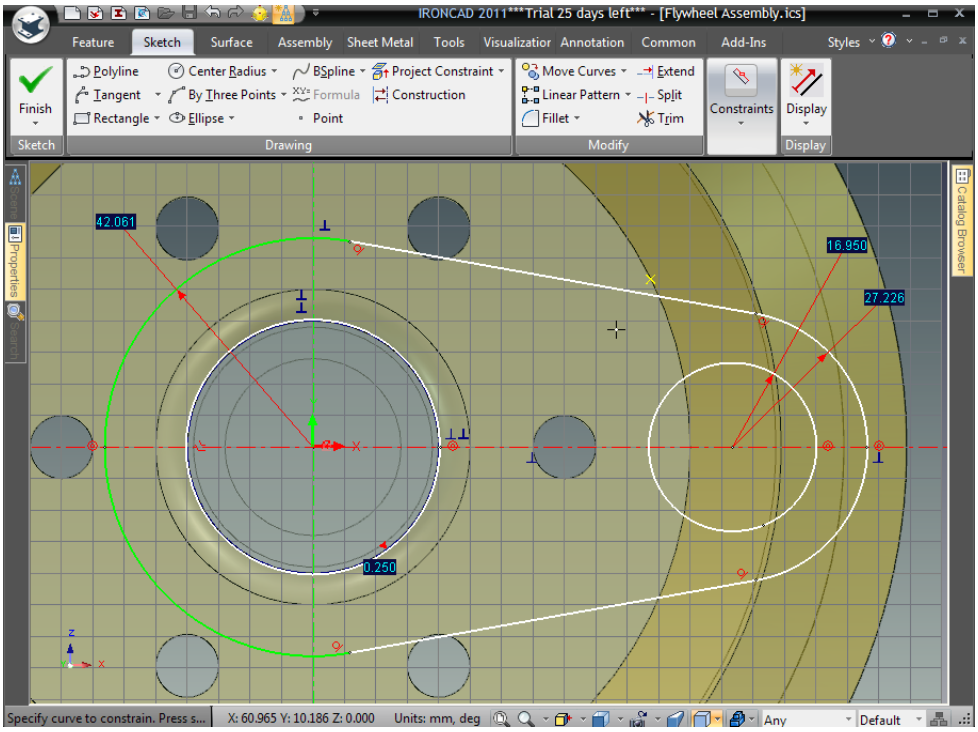
The left inner circle is constrained to the shaft and does not need to be locked.

Select the Concentric constraints tool and select the left hand inner circle and its associated outer arc. Next repeat this action for the right hand pair.

3. Add Dimensions

Using the Smart Dimension tool, add a dimension for the outer left arc and one each for the right hand outer arc and inner circle (click once to select the circle or arc and once again to specify the dimension location).

IRONCAD will change the left hand outer arc to green to signify that it is fully constrained.



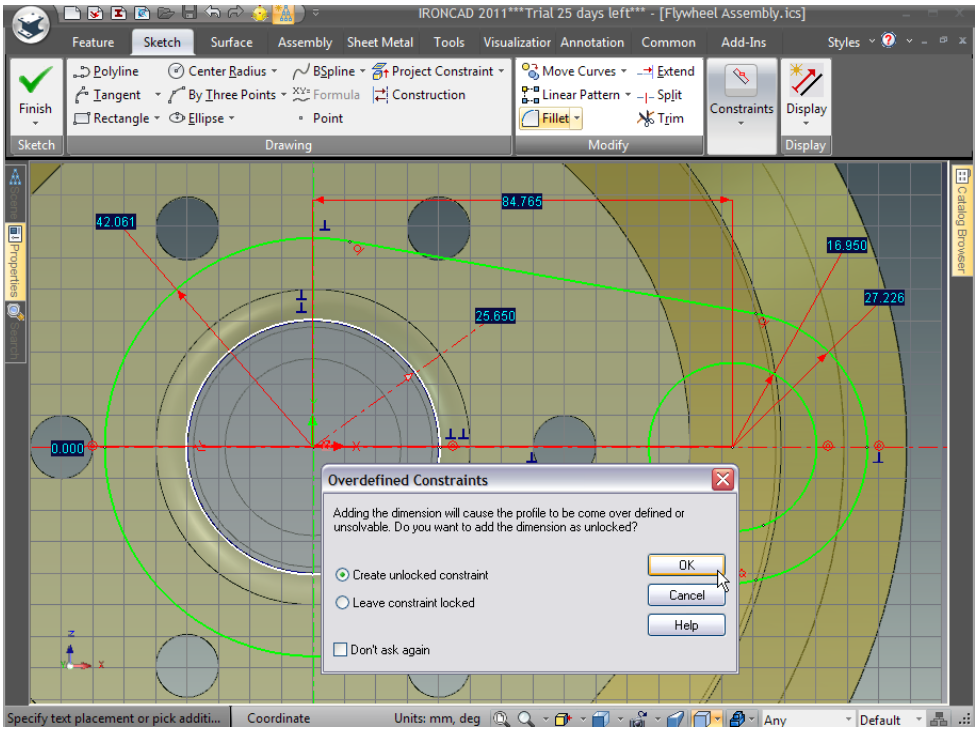
.

With the Smart Dimension tool, dimension the horizontal distance between the two inner circles. Click one circle, then click the other circle and then click to locate the dimension.

Add another dimension between the circle centers, but this time align it vertically and make sure it's set to 0.0. If the two circles are on the same horizontal axis, the third point for the dimension would be on that axis and IRONCAD will create the dimension even though its value is zero.

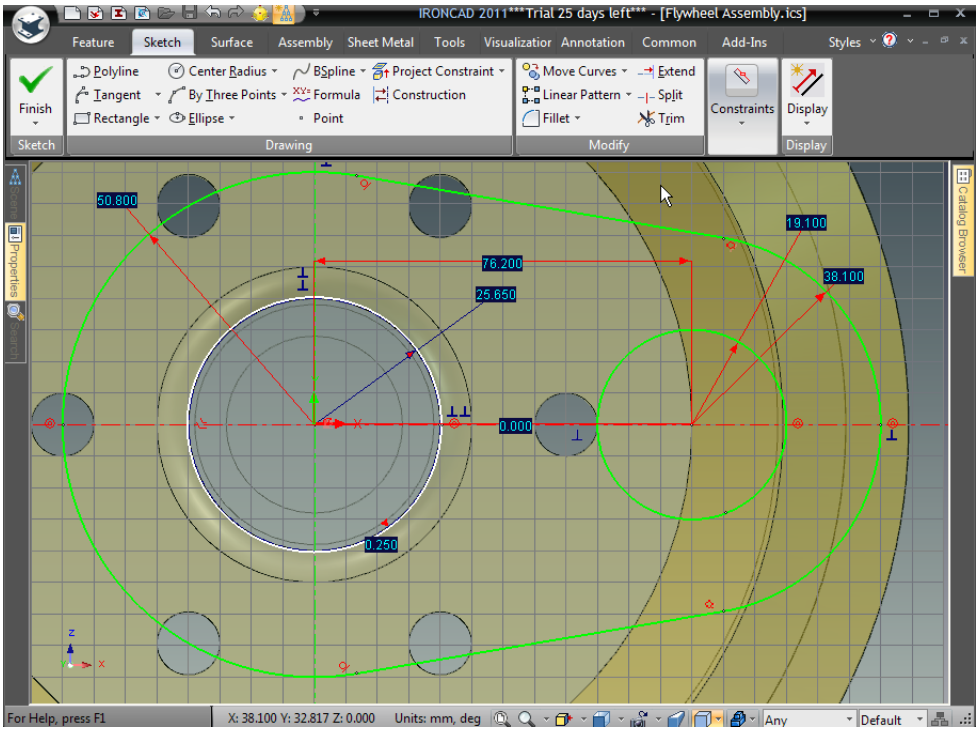
At this point, all the geometry except for the circle tied to the shaft diameter, should turn green showing the sketch is fully constrained.

Add one final dimension to the white circle. IRONCAD will report an issue of a potential error due to over constraining the profile. Select the option that allows the dimension to be placed as a reference dimension. This will be colored blue to differentiate it from the defining dimensions which are red.



Now edit each dimension to achieve the values shown.

A. Left Large Arc Radius = 50.8
B. Distance from Center to Center = 76.2
C. Right Arc Radius = 38.1
D. Right Circe Radius = 19.1

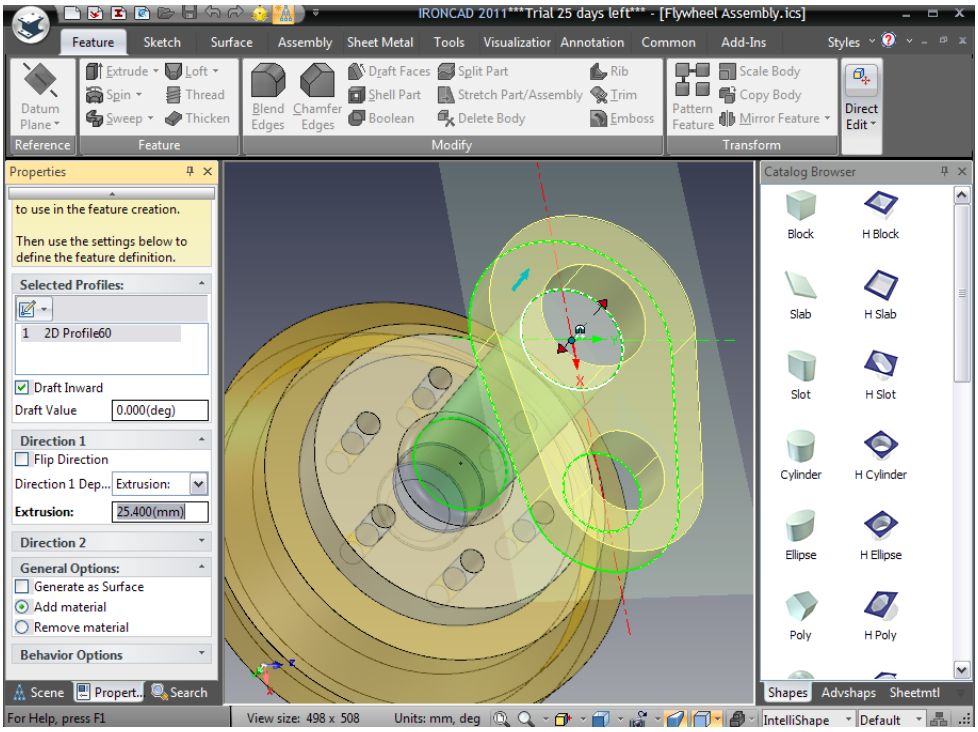


4. Finish the Sketch

Right-click and select Finish Drawing to complete the sketch.

5. Finish the Extrude

IRONCAD will display a preview of the Crank in place. In the Direction 1 panel, click on the Flip Direction check box and set the extrusion depth to 25.4.



Hit the Return key once to accept the changes to the panel contents and a second time to complete the operation.

6. **Rename the Part and Save**

   Change the name of the part to Crank Arm and save the scene!

4.11. Parametric Constraints with Expressions

Earlier in this section, the use of parametric relationships was introduced. This section will expand on that knowledge.

4.11.1. Parametrically Control the Crank Arm

Next, we will add parameters with expressions to control the crank arm to size parametrically when we edit the values.

1. **Access the Parameter Table Outside of an Edit Mode**

   In the scene browser, expand the Crank Arm by clicking on the "+" by the part name. Under this part will be a feature 'Extrude1'. Expand this feature too to reveal the cross section used to define the part. Right-click on the Cross Section and select Parameters.

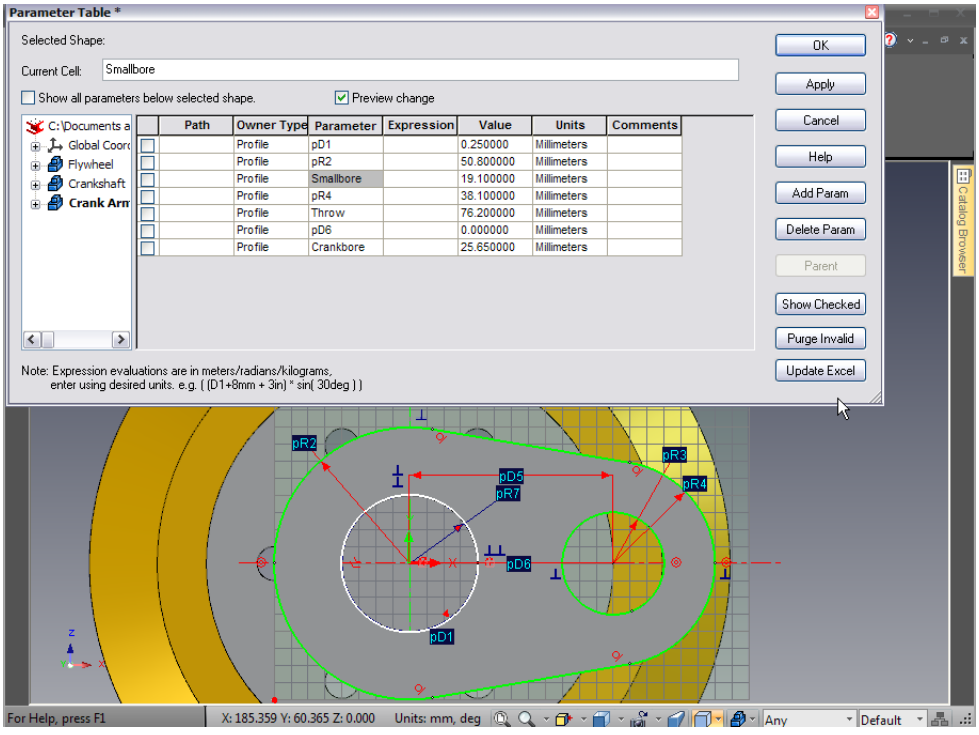
IRONCAD will display the parameter table for the Cross Section. The parameters will probably be recognizable for this part, however in a larger part the default parameter names will not mean much. Click OK to close the parameter table.

2. Access the Parameter Table within the Sketch

Right-click on the Cross Section in the scene browser and this time select Edit. IRONCAD will display the defining sketch.

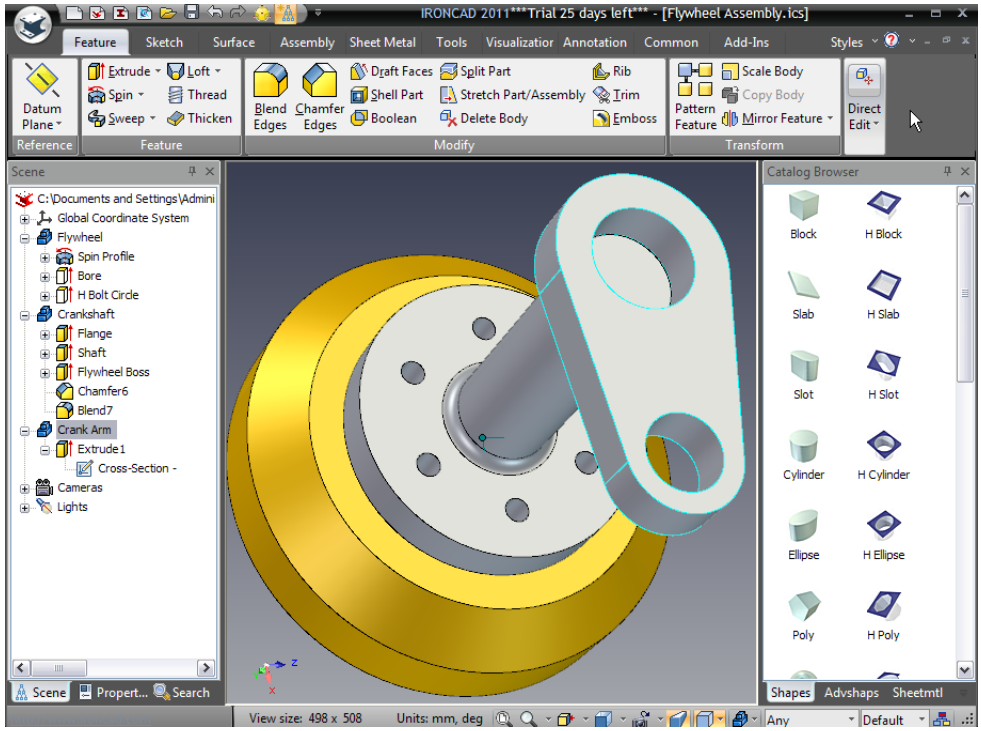
Right-click on the sketch plane and select Parameters from the drop down menu. IRONCAD will change the dimension values to match the parameter codes listed in the table (if not position the cursor over one of the dimensions – IRONCAD will update the view).

Edit a few of the parameter names to become meaningful i.e. 'Throw' for the distance between the bores, 'Crankbore' for the main bore (the reference dimension) and 'Smallbore' for the small right hand bore (IRONCAD does not allow spaces, or non alpha numeric characters in the name). Notice the clearance set at the beginning of the design phase of the Crank Arm is in the parameter list.



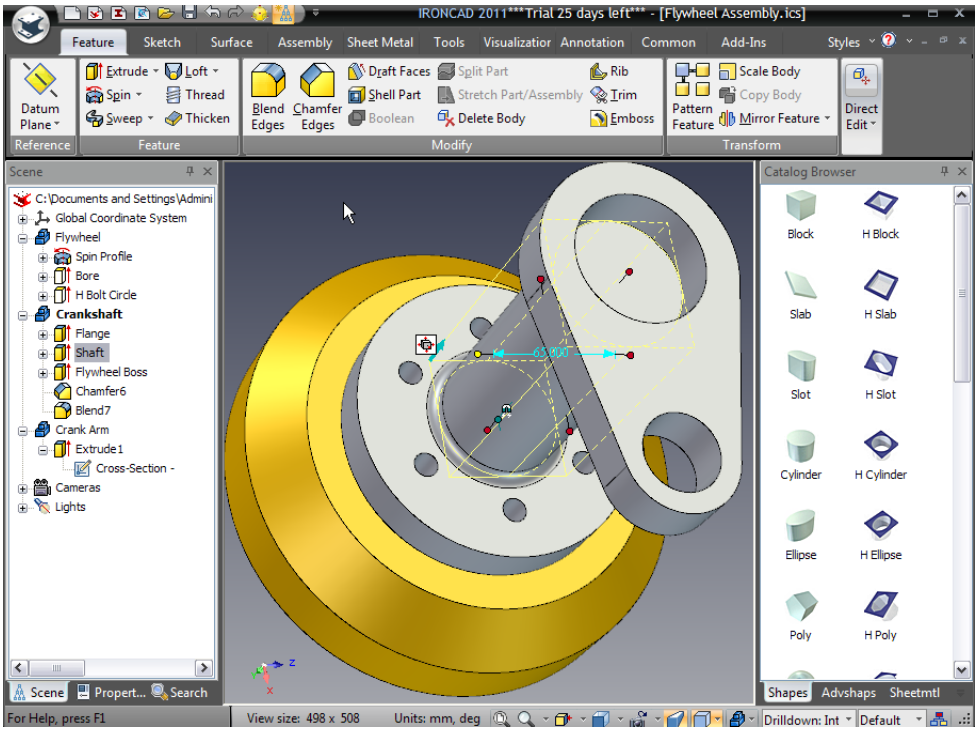
Enter ‘Smallbore+10mm’ in the expression box for the radius of the right hand outer arc. It is essential to tell IRONCAD what the units are with the value. In the expression box for ‘Throw’, enter Smallbore*5 and in the expression box for the large end radius, enter Crankbore+17.5mm.

The dependent dimension values are now calculated correctly. They are also highlighted in a different color than the red (independent) dimensions. Click OK to apply the changes and then click the Finish check mark.



3. Change a Dimension to See the Parameter Update

Select the cylinder that forms the main shaft of the Crank and change its diameter to 65.



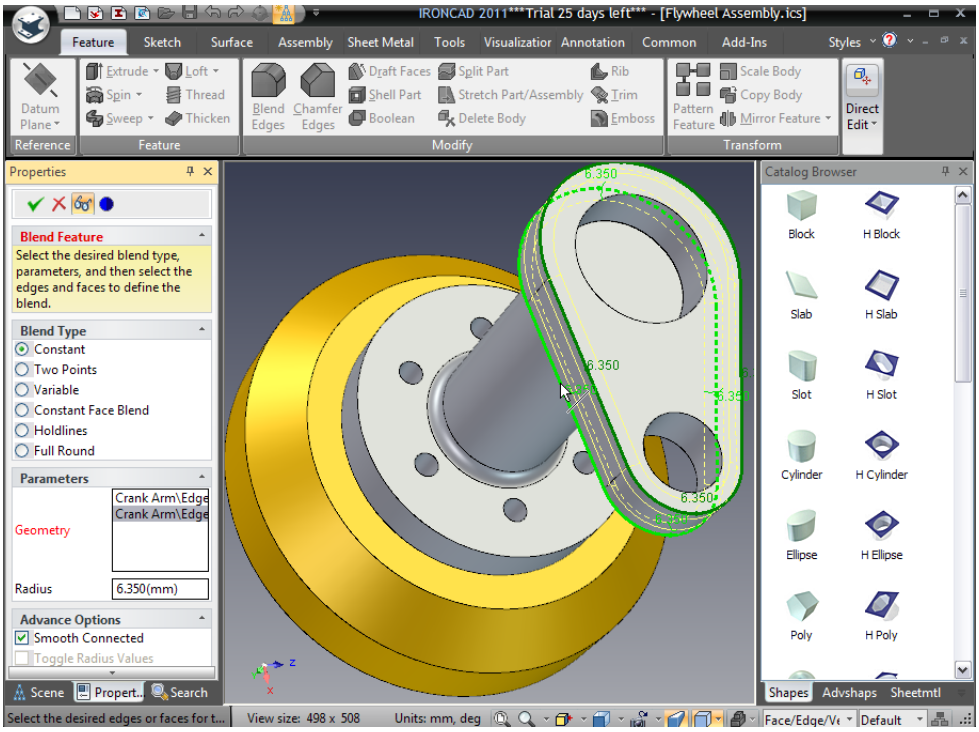
The Crank Arm changes in a predictable way based on the diameter of the shaft. We could go further and enter a whole raft of expressions to ensure the Crank is always the right size for the diameter of the shaft.

4. Blend the Crank Arm

To finish off the crank, we will add a blend to the outer edges on the front face and on the back face in a single operation.

From the Feature tab, select the Blend Edges tool. Ensure the Smooth Connected is checked in the Advanced Options panel of the IRONCAD Property Panel and set the Radius to 6.35mm.

Select one of the edges on the front face of the crank.



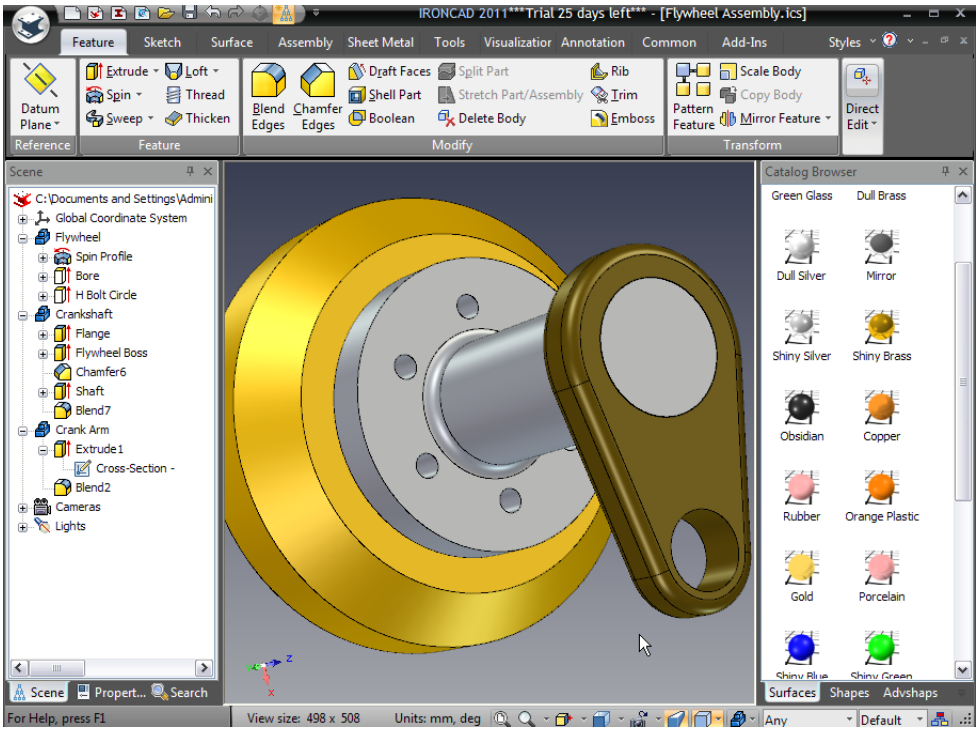
And then one of the visible edges on the back face of the crank. Then click the finish icon on the top of the Command Property Browser.

5. Add A Brass Material

Open the Surfaces catalog and drop Brass onto the Crank.

6. Flip the Extrude

Before finishing the part, align the Crank Arm with the end face of the Crankshaft. Select the Crank Arm at the IntelliShape level. Right-click and select Flip Extrude Direction.



7. Save the Scene

4.12. Creating an Assembly

The scene in progress is in effect an assembly of a few components. If this was part of a larger design, it may be required to structure the design as a series of components and assemblies to reflect the actual build structure. In this section the parts designed so far will be combined into a new assembly.

4.12.1. Making an Assembly from Existing Components

Making an assembly from existing components is a simple action of selecting and grouping in IRONCAD.

1. Deselect Everything

Make sure everything in the scene is deselected by clicking on the scene background.

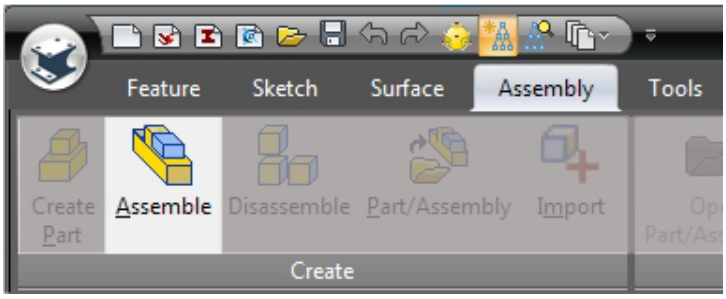
2. Select All Three Parts

Select all three parts simultaneously by clicking one part then shift-clicking the other two or by clicking in the background and pressing Ctrl A. All three

parts should be highlighted with cyan colored edges. Alternately, you can select the parts from the Scene Browser.

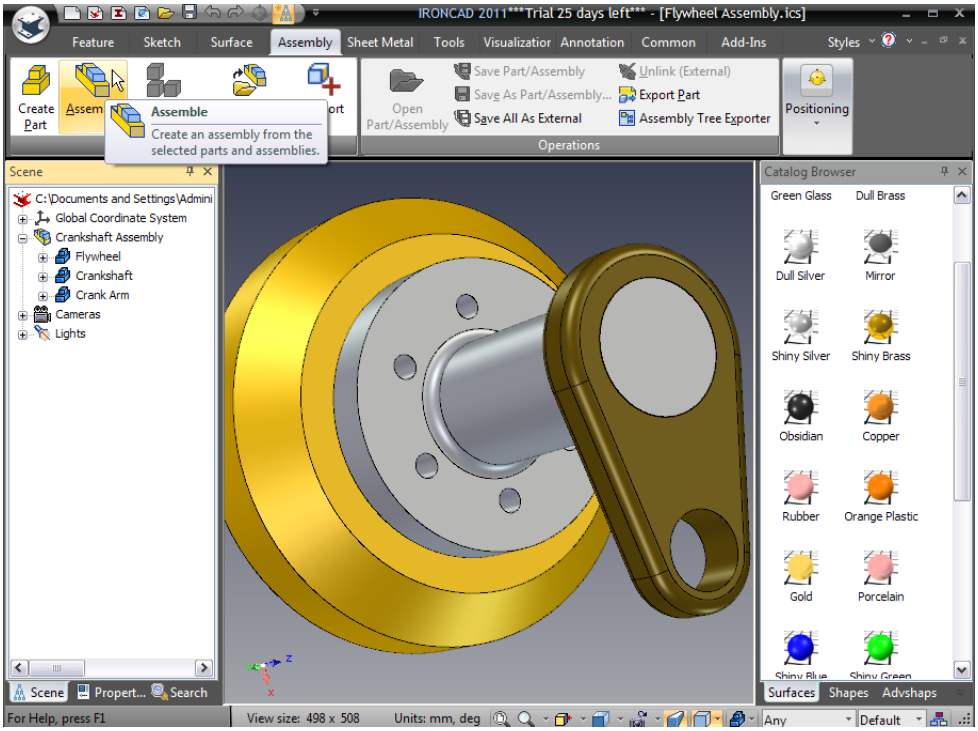
3. Click Assemble

Click on the Assembly tab and select the Assemble tool.



4. Rename the Assembly

If the Scene Browser is not open, open it to view the newly created assembly. Rename the assembly to Crankshaft Assembly.



4.13. TriBall and Linked Components

The design will be completed by adding simple gear teeth to the Flywheel component by introducing two powerful concepts – the TriBall and Linked shapes.

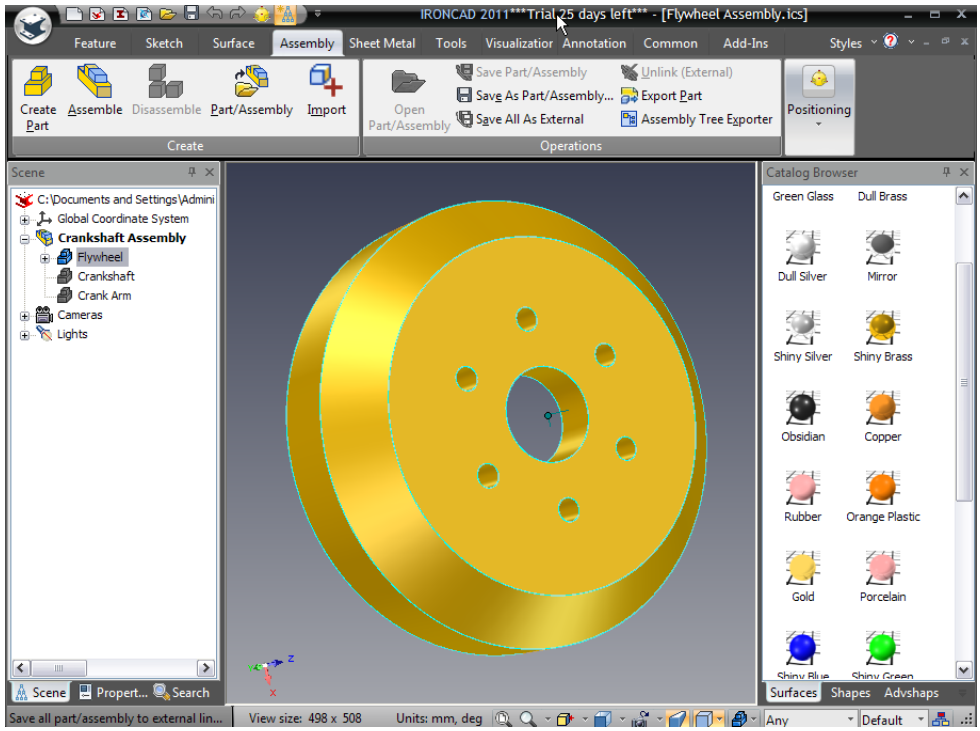
The TriBall is unique to IRONCAD. It provides positioning, rotation, and copying tools for all aspects of IRONCAD design.

4.13.1. Using the TriBall to Create Linked Shapes

This section will use the TriBall to create gear teeth from linked IntelliShapes.

1. Hide Unselected Components

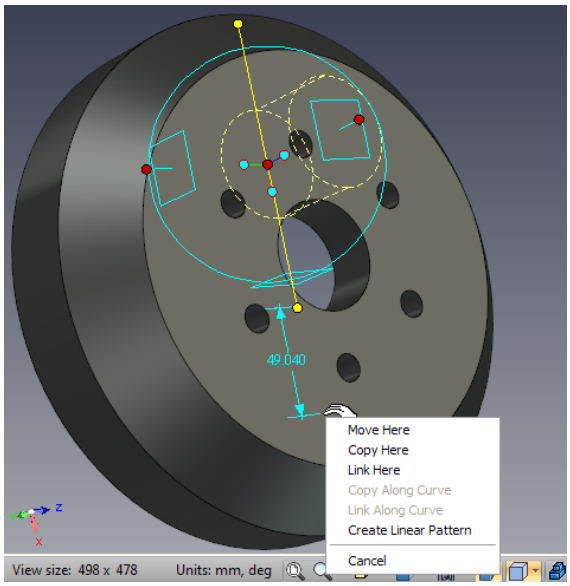
Select the Flywheel so its edges are highlighted cyan. Right-click and select Hide Unselected from the resulting pop-up menu. IRONCAD will remove the other components from the screen to reduce confusion.



2. Copy an Existing Shape Using the TriBall

Select the IntelliShape defining the bore of the Flywheel and activate the TriBall either by selecting the TriBall icon on the quick access toolbar or pressing F10.

Right-click on one of the red handles of the TriBall running in plane with the Flywheel (as shown in the next image. The Flywheel color has been changed in this image to clearly show the TriBall) and drag the bore towards the outer rim of the Flywheel and release the mouse button.



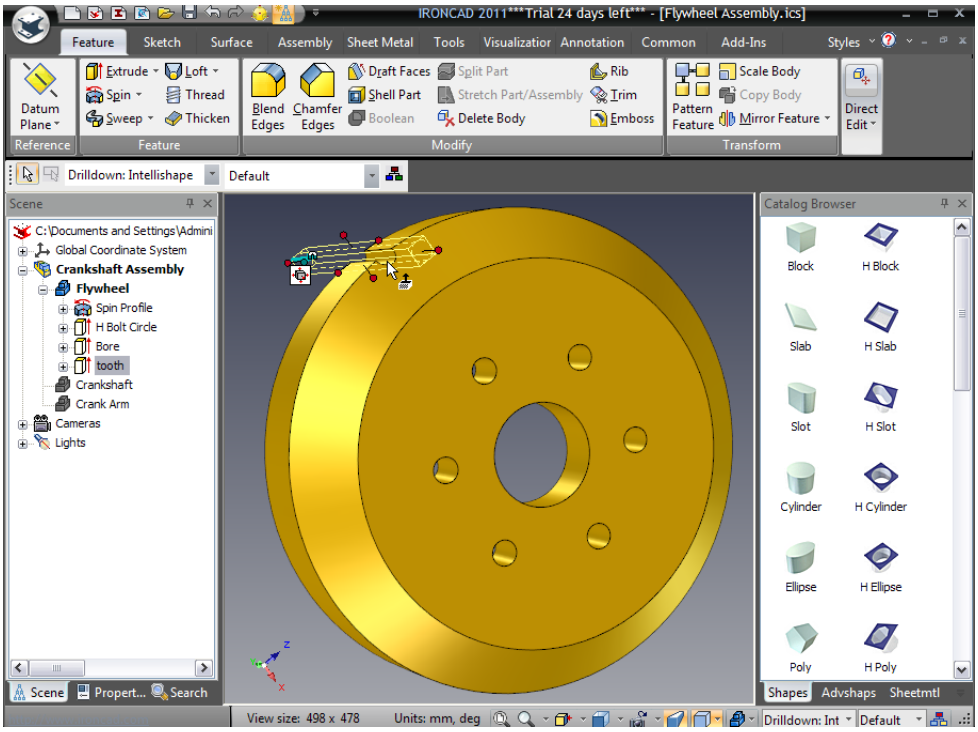
IRONCAD will display a pop-up menu. Select Copy Here and in the data entry box, enter 1 for the Number and 120 for the Distance and click OK.

3. Rename the New Shape

In the scene browser, change the name of the copied 'Bore' shape to 'Tooth'.

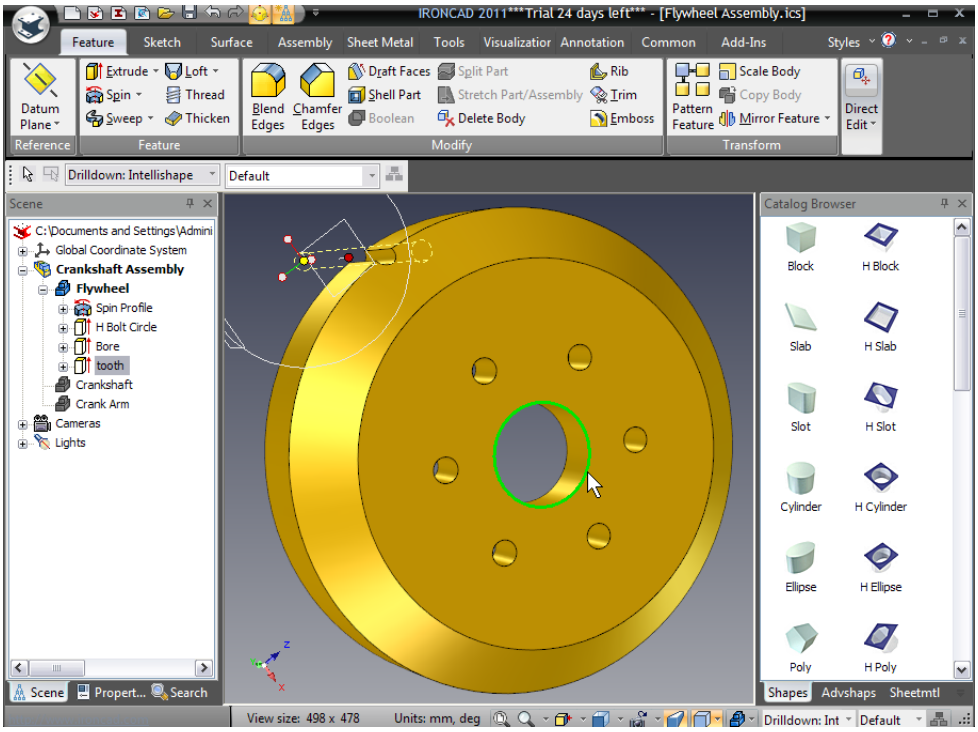
4. Change the Length and Depth

Change the diameter to 10mm and drag the length through the part.



5. Relocate the TriBall

With the 'Tooth' selected, activate the TriBall by pressing F10. IRONCAD will display the TriBall on the anchor of the feature. However the tooth needs to be rotated about the center of the Flywheel. With the TriBall active, press the Spacebar on the keyboard. The TriBall will change from cyan to white indicating the mode has changed from 'manipulate the item' (or Move Shape and Anchor) to 'manipulate the TriBall' (or Position the TriBall Only). Right-click on the center red point of the TriBall and select 'To Center point' from the resulting pop-up menu. Indicate the edge of the center bore.

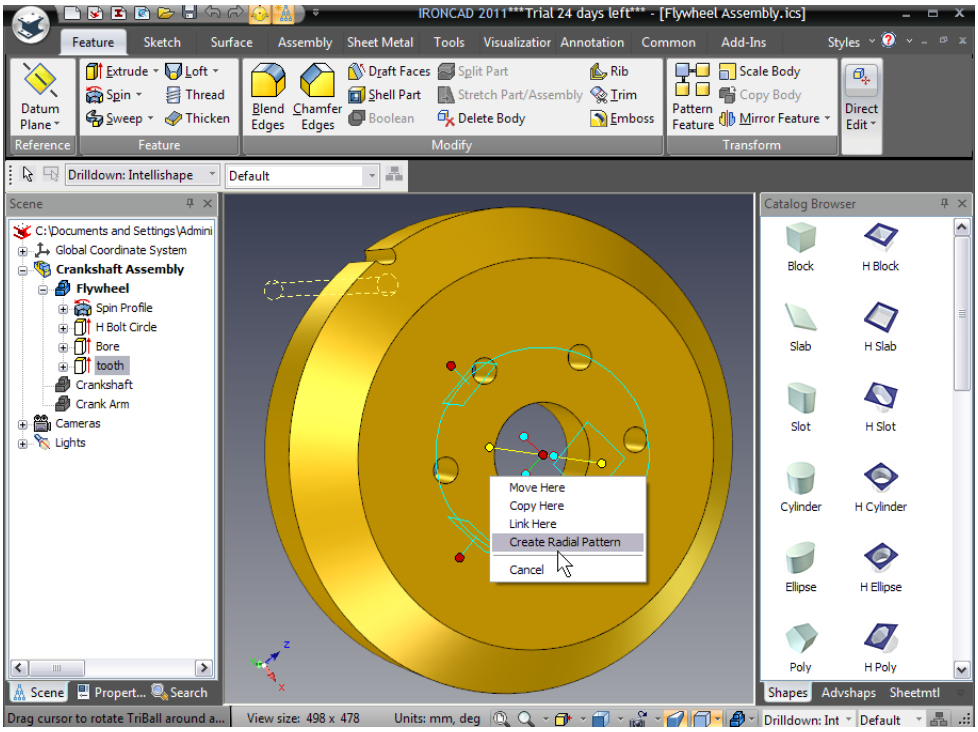


With the TriBall on the center of the Flywheel, press the Spacebar to change the TriBall mode back to 'manipulate an item'.

6. Create the Radial Pattern

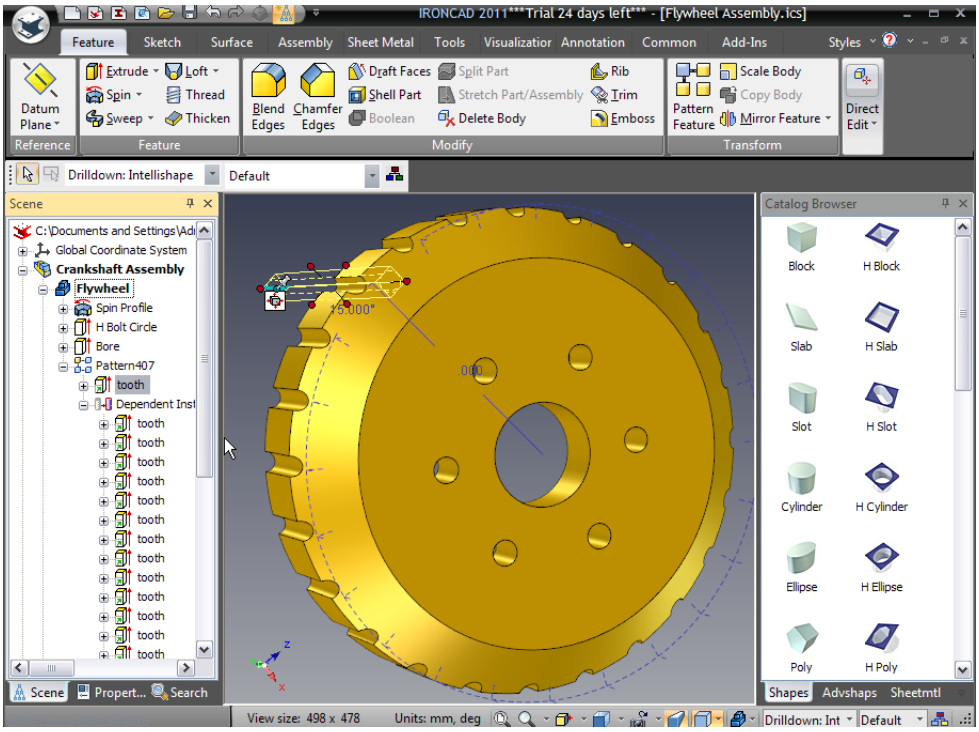
Click on the red handle emanating from the bore center (as shown). The axis will change to yellow and be drawn fully through the TriBall.

Positioning the cursor inside the TriBall outer circle will result in the cursor changing to a 'grabbing hand with an arrow rotating around its wrist'. Keeping the cursor within the TriBall boundary, click and hold the right mouse key and drag the mouse a small distance. IRONCAD will rotate the tooth around the Flywheel's central axis as the cursor moves.



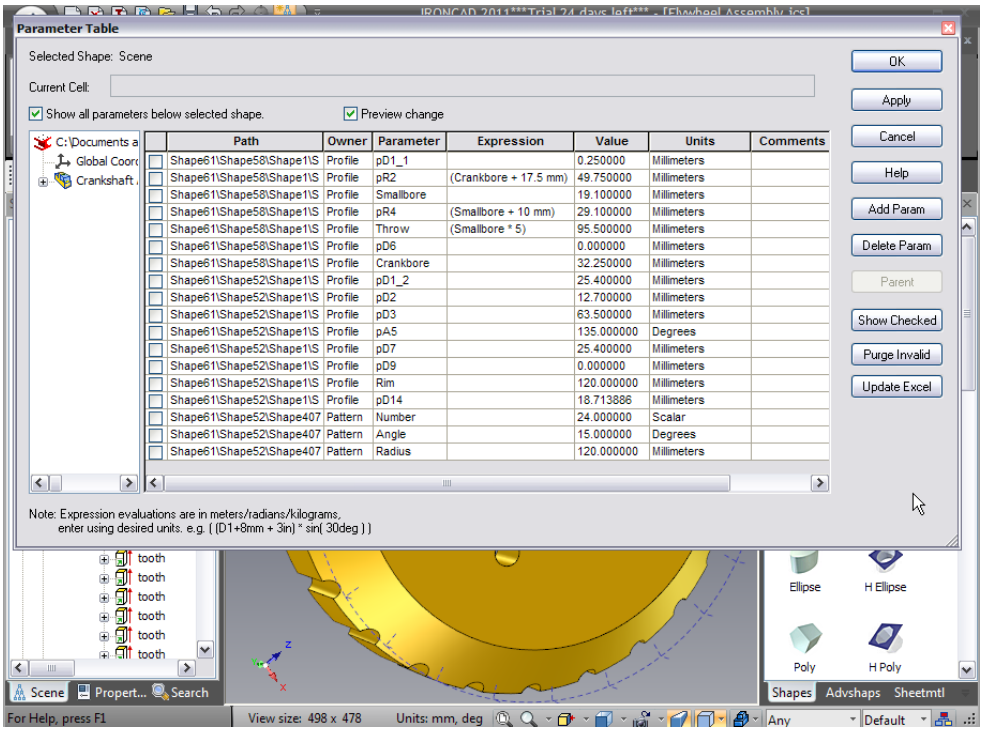
Releasing the mouse button will cause a pop-up menu to be displayed. Select ‘Create Radial Pattern’ and enter 24 for the Number and 15 for the Angle.

IRONCAD will create 23 linked copies of the original tooth. Changing any one would cause the other linked copies to change. Each tooth feature shown in the scene browser will have a green arrow on its icon showing the feature to be linked (the same notification is used for linked parts, assemblies, etc). As IRONCAD created this set of features as a pattern, the features are grouped under an entry called Pattern which can be renamed like any other feature. To change the values of the pattern, click on the pattern name. IRONCAD will highlight the primary values in bright green on the model. Right-clicking and selecting Edit will reveal the editable settings of the pattern.



4.14. Adding Intelligence Using Parameters

When creating the 2D profile earlier in this section, Smart Dimensions were used to rationalize the geometry and add relationships between different elements of the sketch. Smart Dimensions are also referred to as parameters. Using the Radial Pattern command to create the gear profile results in IRONCAD generating the input required for the pattern as a set of parameters too. The parameters are managed, or owned, according to the part of the model tree where they were created. For example: for the original 2D cross section, the parameters are listed in the parameter table as being owned by Profile and the ones for the pattern are owned by Pattern as shown next:



One of the main reasons for this is to simplify the process for editing and managing parameters both at the originating level and subsequent levels such as with Design Variations – IRONCAD's way to build part families.

Parameters can be added to help structure and follow the design intent of the part. For example: tying the outside diameter of the Flywheel to a certain type of drive belt. In the following example, the Flywheel will be designed to fit a specific toothed belt requirement (i.e. the size of the tooth and the linear distance between teeth). From these two values and the number of teeth required on the Flywheel, IRONCAD will rebuild the geometry of the Flywheel and any other component referencing the Flywheel.

4.14.1. Parameter Table Format

The parameter table is arranged in a similar way to a spreadsheet (i.e. a number of rows and columns). The rows reflecting the contents of each parameter and the columns containing specific data (such as parameter name, parameter value, etc).

One of the most significant points to remember is the parameter table evaluates all expressions in 'system' units (for example: meters, radians and kilograms). It is not the purpose of this guide to run through the methods and

complex examples of the mathematics that can be performed. To those ends, a much more simplified approach will be used, one that can be used as a fallback position to help resolve complex scenarios.

The basic building blocks of the formula to establish the radius of the rim defined previously are:

Rim = *rimcircumference*/2 x PI (defines the radius of the rim)
rimcircumference = Number x *pitchdistance* (defines the circumference of the rim)
pitchdistance = *land* + 2 x *toothrad* (defines the linear pitch of one complete tooth)

Pretty straightforward really: Rim is the radius of the circumference created by the number of teeth multiplied by the pitch of the teeth. The variables break down as follows:

Rim - the parameter set earlier in the 2D section
Number – this is a parameter set by the TriBall Pattern function
toothrad – the radius of the circle defining the tooth
land – the amount of material between each tooth

In addition to the primary user defined variables to simplify the expressions, the following variables should be created:

PI() – the value of *Π*
radnumber – the actual value of a radian

4.14.2. Adding Control to the Tooth Form

To set the size of the tooth so that it can be used in the expression, we will edit the cross-section.

1. Edit the Cross-Section

Rright-click on one of the tooth entries in the scene browser and select Edit Cross-Section.

2. **Lock the Axis**

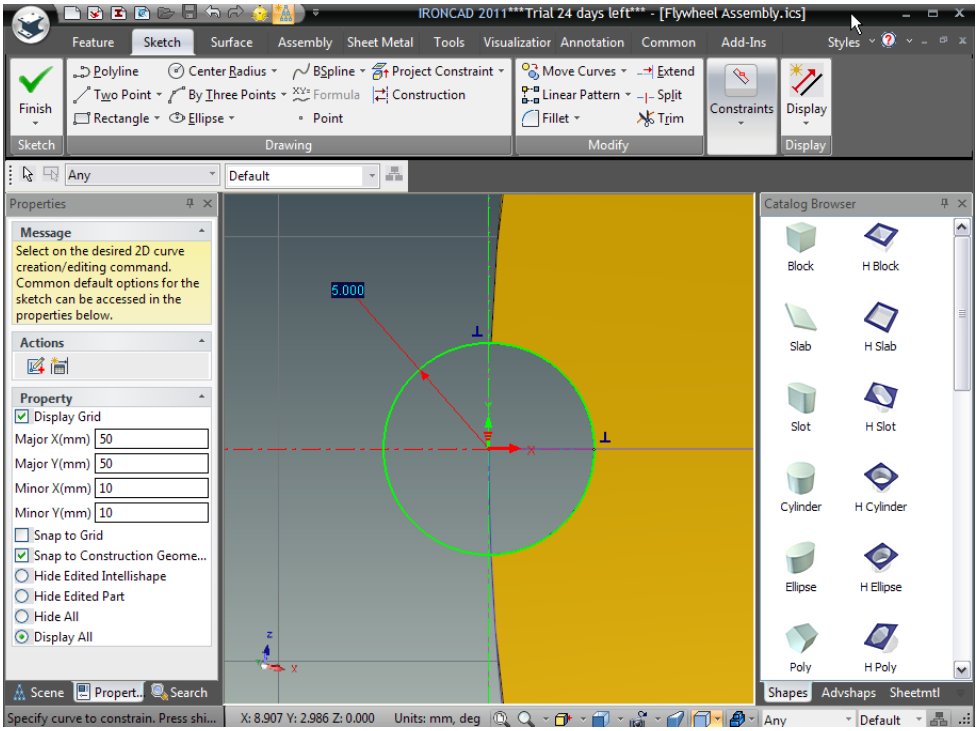
With the cross section displayed on the screen, right-click on the X-Y origin and select Lock.

3. **Add a Dimension**

Next add a Smart Dimension to the circle. At this point the circle should be colored green with a red dimension indicating a radius of 5mm as shown next.

4. **Accept the Changes**

Accept the changes by right-clicking and selecting Finish Drawing or click on the green check mark on the main menu.

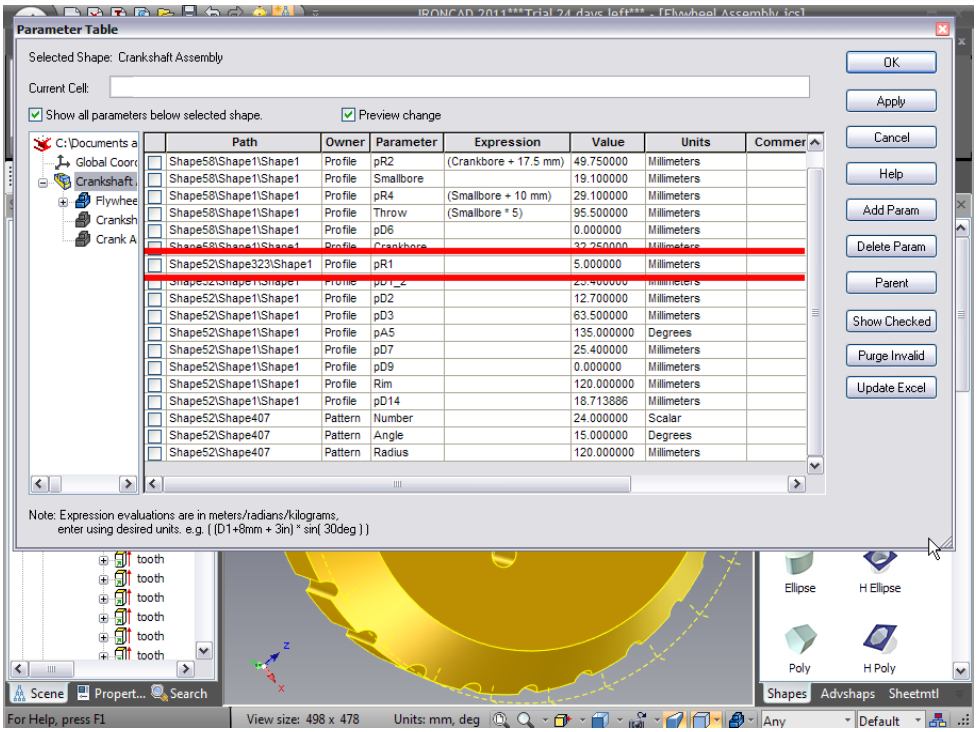


IRONCAD will create an entry in the parameter table reflecting the addition of the 5mm radial dimension.

5. **Access the Parameters**

Right-click anywhere in the scene and select Parameters .IRONCAD will display the parameter table. Ensure the Show all parameters below selected shape check mark is checked. In the scene browser shown by the

parameter table, ensure the Crankshaft Assembly is selected. In the next image, the new parameter can be seen as pR1. ‘pR1’ can be identified by its value (5.00000 millimeters) and has a unique identity in the Path column.

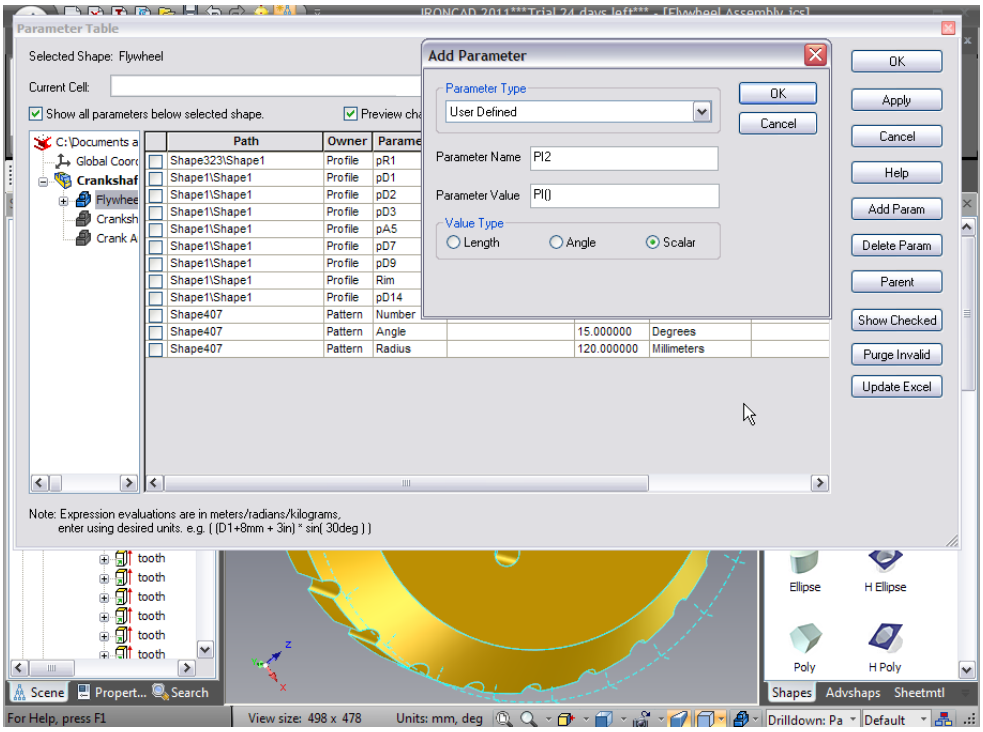


4.14.3. Adding User Defined Parameters

Now in the parameter table’s scene browse, select the Flywheel component. IRONCAD will reduce the list in the parameter table to those entries created when defining the profile. As mentioned previously, a number of variables need to be defined to help build the simple formulas to be used to create the design intent for the Flywheel.

1. Add a Parameter

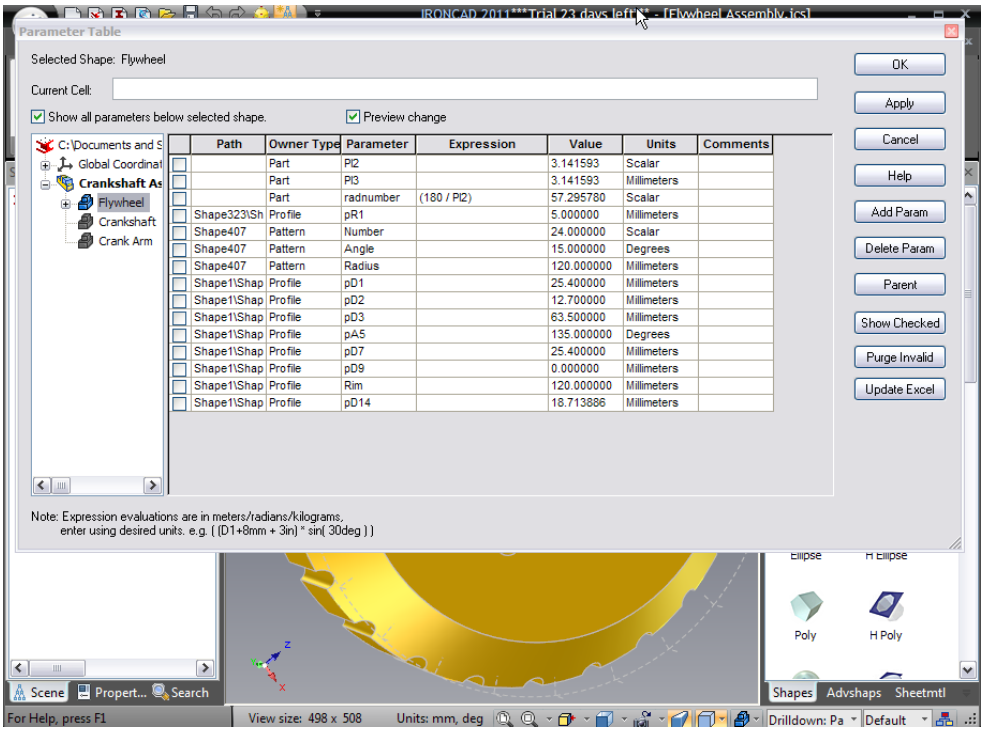
With the Flywheel component highlighted, select ‘Add Param’. In the resulting dialog box enter PI2 in the Parameter Name box, in the Parameter Value box enter PI() and set the Value Type to Scalar (i.e. a unit-less value).



Clicking OK will cause the data entered to be inserted into the parameter table.

2. Add Additional Parameters

Next add the following parameters in the same way:
Name – radnumber, Parameter Value =0 and Value Type to Scalar. This variable establishes an actual value for a radian.



3. Setting the Expression

To establish the value for the '*radnumber*' parameter, (once created) left click in the Expression column by the '*radnumber*' parameter and enter: 180/PI2 followed by pressing the Return key.

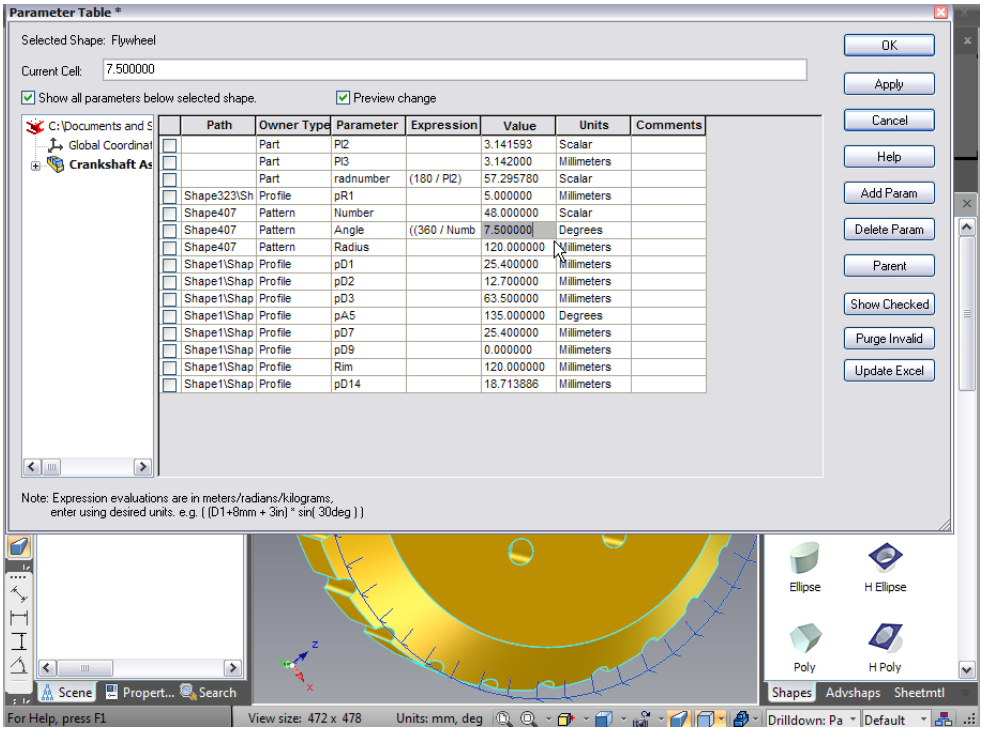
IRONCAD will update the value column to include the numeric value of 1 radian i.e. 57.295780.

4.14.4. Establishing the Angle between the Teeth

As mentioned previously, parameters are evaluated with respect to system based units and angles must therefore be in terms of radians.
The pattern function created three parameters automatically - Number, Radius and Angle. For the full flywheel, the Angle between each tooth will be the same, therefore in the expression box alongside Angle enter the following: (360/Number)**radnumber*

IRONCAD will evaluate the expression but not change the value of the Angle parameter. To test, click on the value associated to the Number parameter and enter, say, 48. To accept this value click another value box or press Return.

IRONCAD should update the Angle to 7.5. Additionally the graphical representation of the pattern will change by increasing the number of positions for the cylinder used in the pattern.



4.14.5. Relating the Diameter to the Number of Teeth

The outside diameter of the Flywheel rim will be a function of the number of teeth and the pitch of those teeth. For the simplicity of this example, the impact of 'wrapping' the pitch around the circumference will be ignored as in all practical senses it is so small. However in the practical world, this would have to be resolved.

The circumference of the Flywheel rim in this example is the number of teeth multiplied by the pitch of the teeth. The land between each tooth will be 4mm for this example. Therefore the pitch will be the width of the feature created by the patterned hole plus the land (i.e. using the value created earlier):

(pR1*2)+4

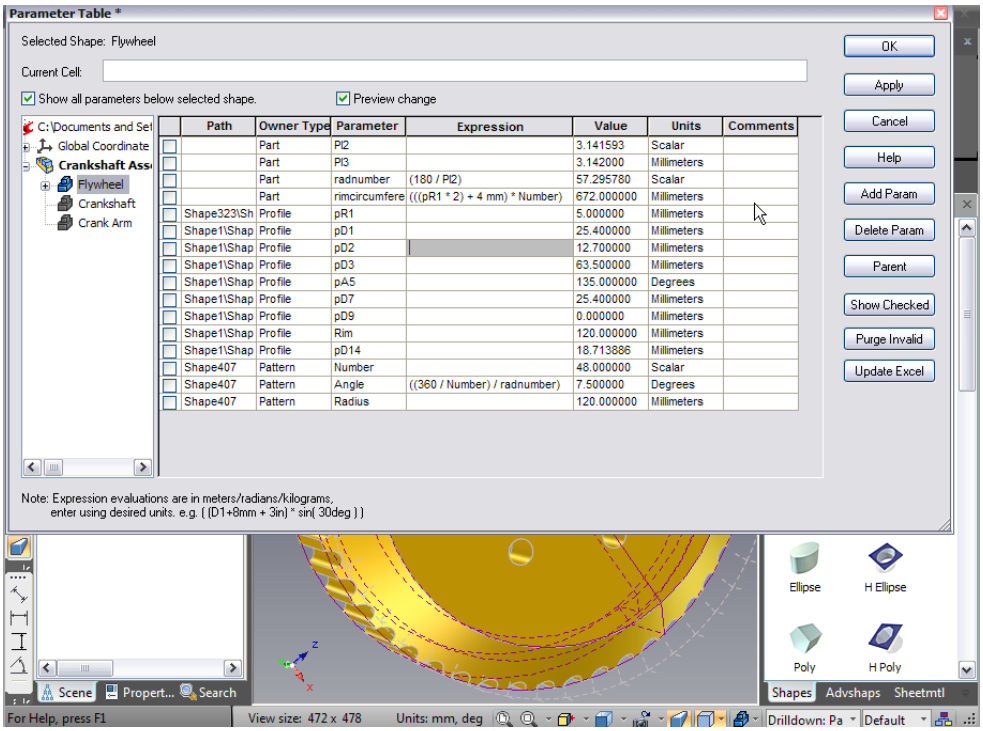
Add a parameter for the circumference of the rim by clicking on Add Param and then entering:

Name – rimcircumference, Parameter Value =0 and Value Type to Length and click OK.

Next, in the expression box alongside the newly created parameter enter:

((pR1*2)+4mm)**Number*

Remember to add the 'mm' after the 4 to tell IRONCAD this is an addition of 4 millimeters not 4 meters. The expression has also been grouped between parentheses to ensure the expression is evaluated correctly. In the next image, the *rimcircumference* parameter has been evaluated to 672mm.



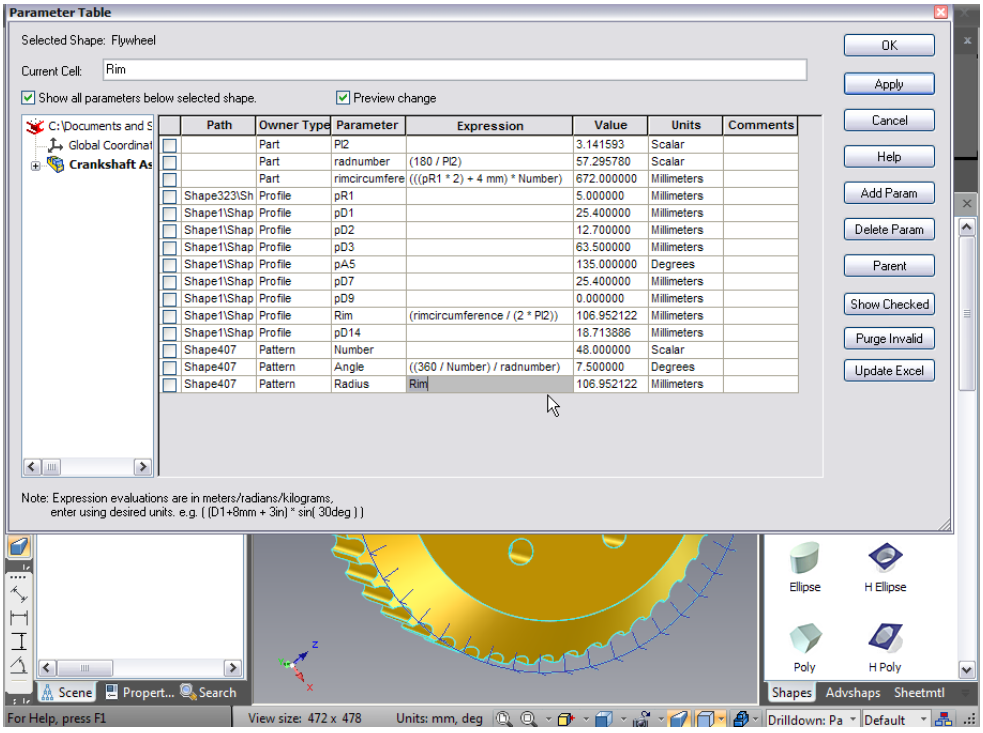
At this point, the circumference is not related to the diameter of the Flywheel. Currently the diameter of the Flywheel is defined by revolving its profile around an axis. The diameter is therefore twice the value associated with the parameter Rim (assigned when defining the profile). Therefore:

Rim = rimcircumference/2**Π*

In the expression box alongside the parameter Rim enter the following:

rimcircumference/(2*PI2)

IRONCAD will update the value relating defining the Rim and any other parameter relating to the parameter Rim. At this point however, the pattern still has the teeth being produced at the original diameter. Correct this by associating the Radius parameter to the Rim parameter by simply typing Rim into the expression panel alongside the Radius parameter.



IRONCAD will update the value of the pattern and provide a preview of the updates.

At this point the Flywheel Assembly is now controlled from the Number of Teeth entered into the value box for the parameter Number. Try changing the number of teeth to 60 or even 72.

Close the parameter table and save the file.

4.15. Design Variations

Put simply, the Design Variations function of IRONCAD allows the creation of part or assembly families based on one or more variables. For example, in the Flywheel Assembly created previously, to set the size of the assembly the parameter table has

to be accessed and the parameter changed. This assumes a certain level of knowledge and leaves the model open to other changes by design or by error.

Design Variations provides a higher level of automation by presenting predictable model variants yet still allowing a degree of flexibility. In the Flywheel Assembly, the whole assembly is built based on a set of parameters feeding from the parameter Number which was automatically created as a result of adding the pattern of teeth. In a typical supply situation, a company may offer standard variants directly from its catalog. Design Variations provides the functionality to pre-define these variants. It goes one stage further and allows the user to define custom scenarios too.

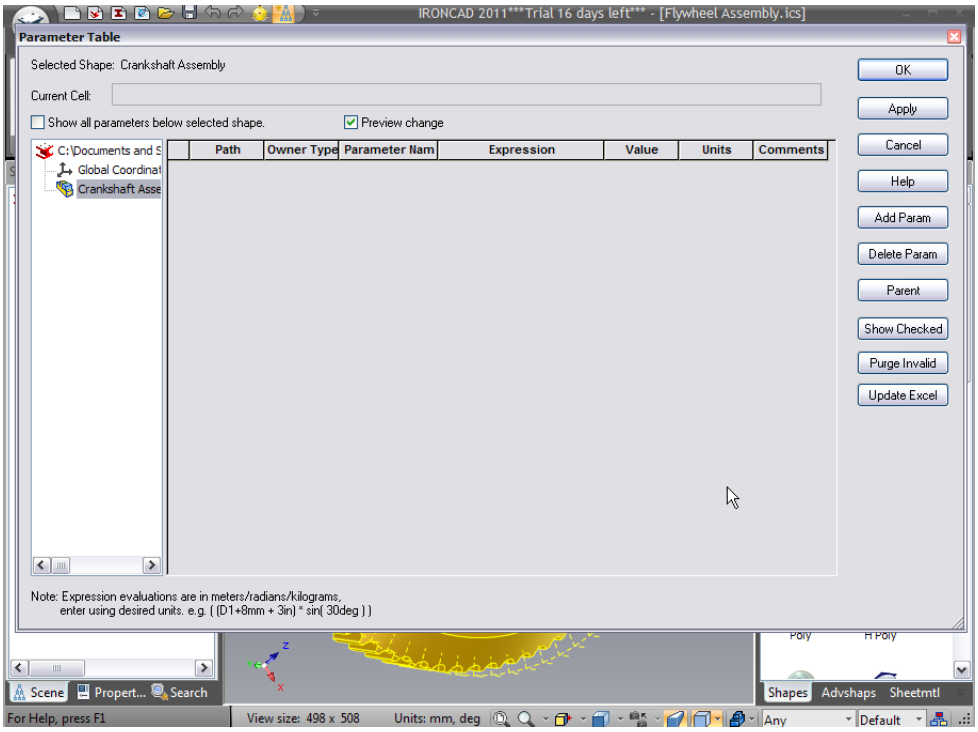
Working from the top level in the part structure, Design Variations hide all the controlling parameters and relationships from the novice or non-expert user of IRONCAD. As with most things, it's easier to explain with an example using the Flywheel Assembly saved at the previous session.

4.15.1. Adding Top Level Parameters

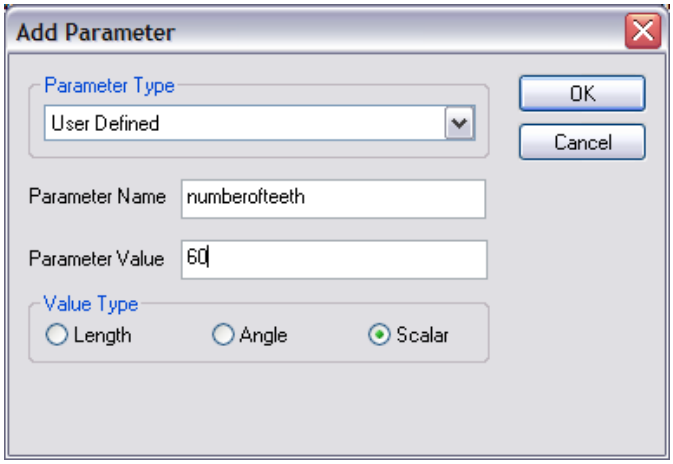
Top level parameter is necessary to control Design Variations and serves as an easy way to define the driving parameters for a design.

1. **Add Top Level Parameters**

   With the Crankshaft Assembly selected, right-click on the Crankshaft Assembly and select parameters from the menu. Ensure the 'Show all parameters below selected shape' check box is not ticked.



Next, add a parameter 'numberofteeth' and set its Parameter Value to 60 and its Value Type to Scalar as shown:



IRONCAD will create a parameter called *'numberofteeth'*.

2. Set the Expression for the Flywheel

Expand the parameter table's scene browser and select the Flywheel part and check the 'Show all parameters below selected shape' box. IRONCAD will display all the parameters for the Flywheel. In the Expression box

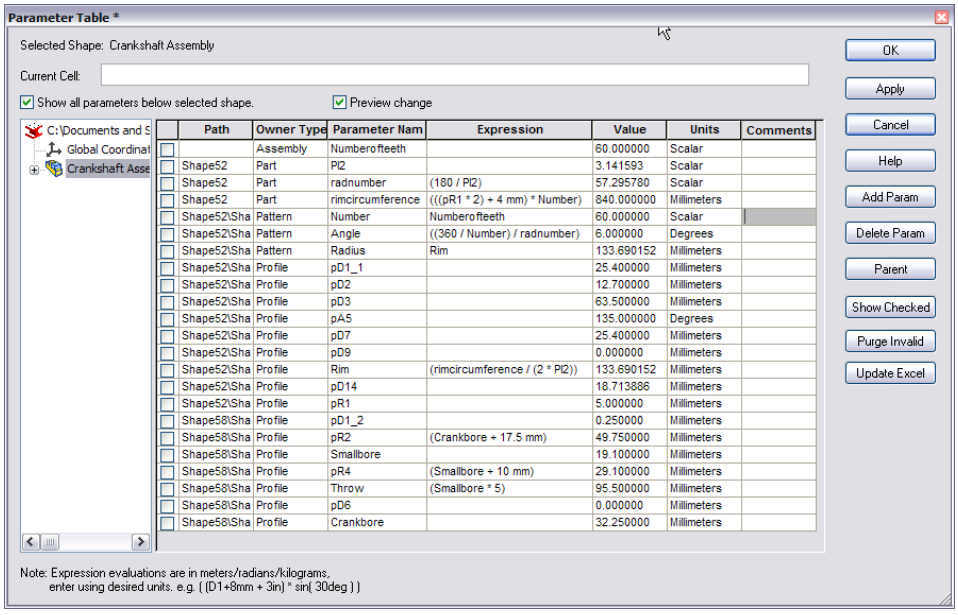
alongside the Number parameter, type ‘numberofteeth’. IRONCAD will relate the Flywheel parameter to the *numberofteeth* parameter.

3. Set the Length of the Crank Arm

It would seem appropriate to relate the length of the Crank Arm to the diameter of the Flywheel. Click on the Crank Assembly in the parameter table’s scene browser to display all the parameters defining the part. Change the expression for the parameter Throw to be: Rim*1.5

IRONCAD will change the length of the Crank Arm appropriately.

The result is that for a single change of *numberofteeth,* the whole assembly will be re-calculated as shown in the parameter table example.



Path	Owner Type	Parameter Nam	Expression	Value	Units	Comments
	Assembly	Numberofteeth		60.000000	Scalar	
Shape52	Part	PI2		3.141593	Scalar	
Shape52	Part	radnumber	(180 / PI2)	57.295780	Scalar	
Shape52	Part	rimcircumference	(((pR1 * 2) + 4 mm) * Number)	840.000000	Millimeters	
Shape52\Sha	Pattern	Number	Numberofteeth	60.000000	Scalar	
Shape52\Sha	Pattern	Angle	((360 / Number) / radnumber)	6.000000	Degrees	
Shape52\Sha	Pattern	Radius	Rim	133.690152	Millimeters	
Shape52\Sha	Profile	pD1_1		25.400000	Millimeters	
Shape52\Sha	Profile	pD2		12.700000	Millimeters	
Shape52\Sha	Profile	pD3		63.500000	Millimeters	
Shape52\Sha	Profile	pA5		135.000000	Degrees	
Shape52\Sha	Profile	pD7		25.400000	Millimeters	
Shape52\Sha	Profile	pD9		0.000000	Millimeters	
Shape52\Sha	Profile	Rim	(rimcircumference / (2 * PI2))	133.690152	Millimeters	
Shape52\Sha	Profile	pD14		18.713886	Millimeters	
Shape52\Sha	Profile	pR1		5.000000	Millimeters	
Shape58\Sha	Profile	pD1_2		0.250000	Millimeters	
Shape58\Sha	Profile	pR2	(Crankbore + 17.5 mm)	49.750000	Millimeters	
Shape58\Sha	Profile	Smallbore		19.100000	Millimeters	
Shape58\Sha	Profile	pR4	(Smallbore + 10 mm)	29.100000	Millimeters	
Shape58\Sha	Profile	Throw	(Smallbore * 5)	95.500000	Millimeters	
Shape58\Sha	Profile	pD6		0.000000	Millimeters	
Shape58\Sha	Profile	Crankbore		32.250000	Millimeters	

4. Exit the Parameter Table

Accept the changes to the parameter table by clicking OK.

4.15.2. Applying Design Variations Intelligence

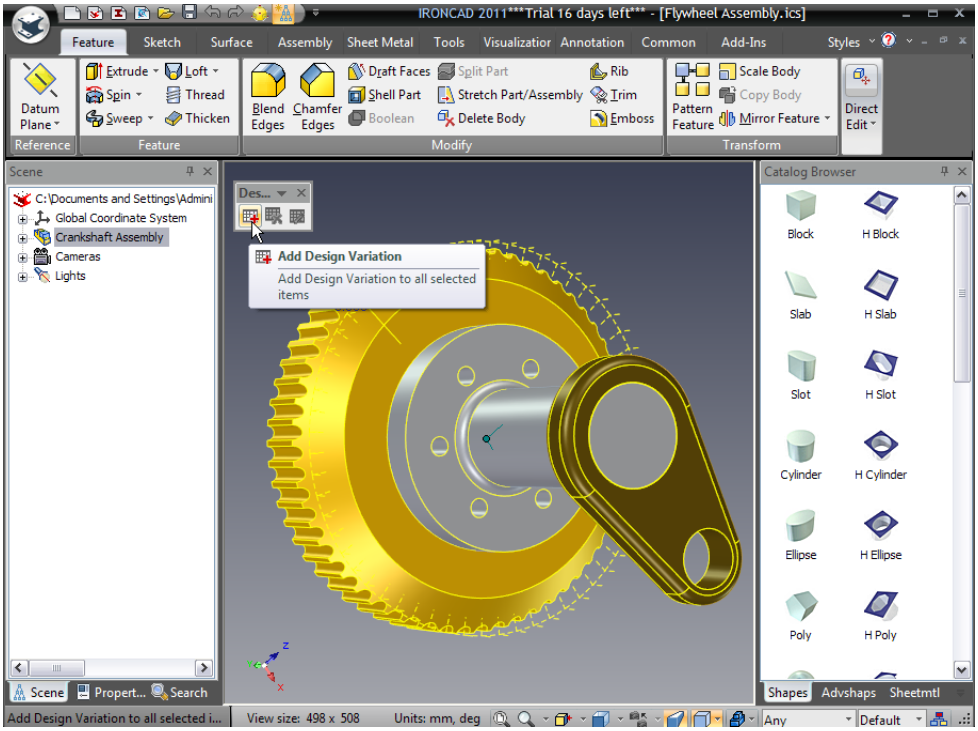
To add the intelligent design Variations, we will need to display the Design Variation tools.

1. **Open the Design Variations Toolbar**

   Click the IRONCAD home button and select View followed by Toolbars. IRONCAD will display the list of available toolbars. Select Design Variations. IRONCAD will display a small (3 icon) toolbar. (Note: This can also be found on the Add-ins tab of the Windows Ribbon bar).

2. **Add a Design Variation**

   Select the Crank Assembly so that all of the components are highlighted in yellow (by default) and select the Add Design Variations icon.



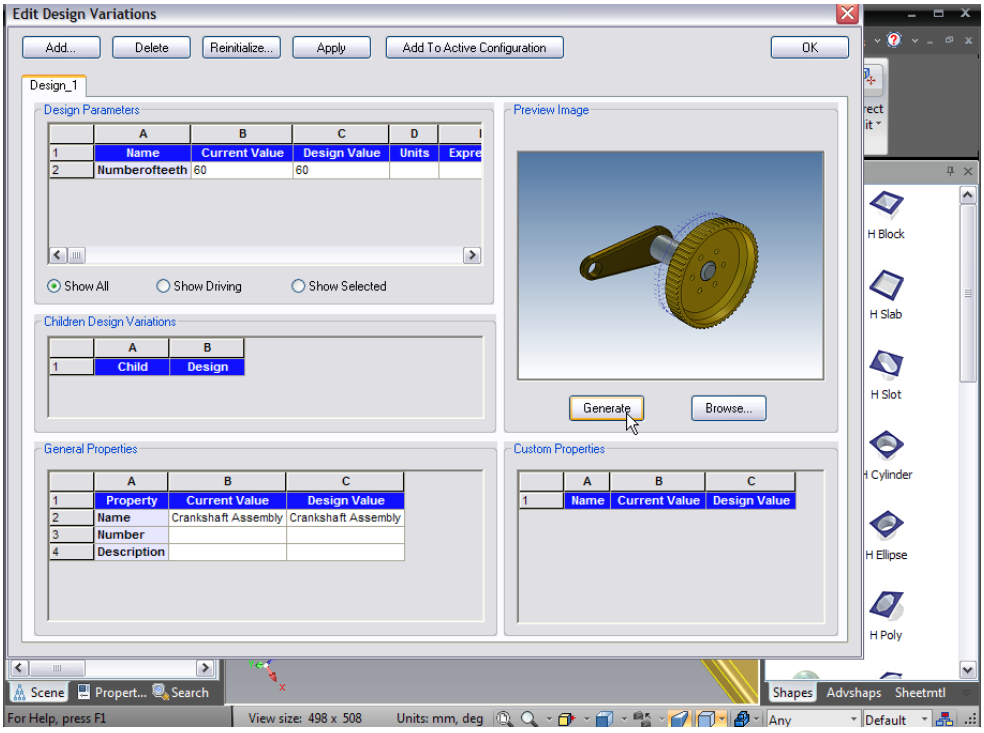
   This will cause the other two icons to become available and disable the Add Design Variations icon.

3. **Edit the Design Variation**

   Selecting the Edit Design Variations icon causes IRONCAD to display the Design Variations panel. In this panel, IRONCAD displays the design parameters that are available to Design Variations. As this section introduces Design Variations it will deal with the basic creation of additional designs based on the existing Flywheel Assembly. Click on the Add button at the top of the panel, IRONCAD will present a dialog asking

for a name to represent the model about to be created. Enter 72 Teeth as the Name. Ensure the Current option is selected and say OK.

IRONCAD will create a new design panel (tab). In the Design Parameters section, replace the Design Value with 72 and then click on Generate below the preview image. The new design with 72 teeth will be created. Repeat the process setting variants with 102 teeth and 50 teeth.



4. Accept the New Design Variations

Click OK to accept the changes.

4.15.3. Testing the Design Variations

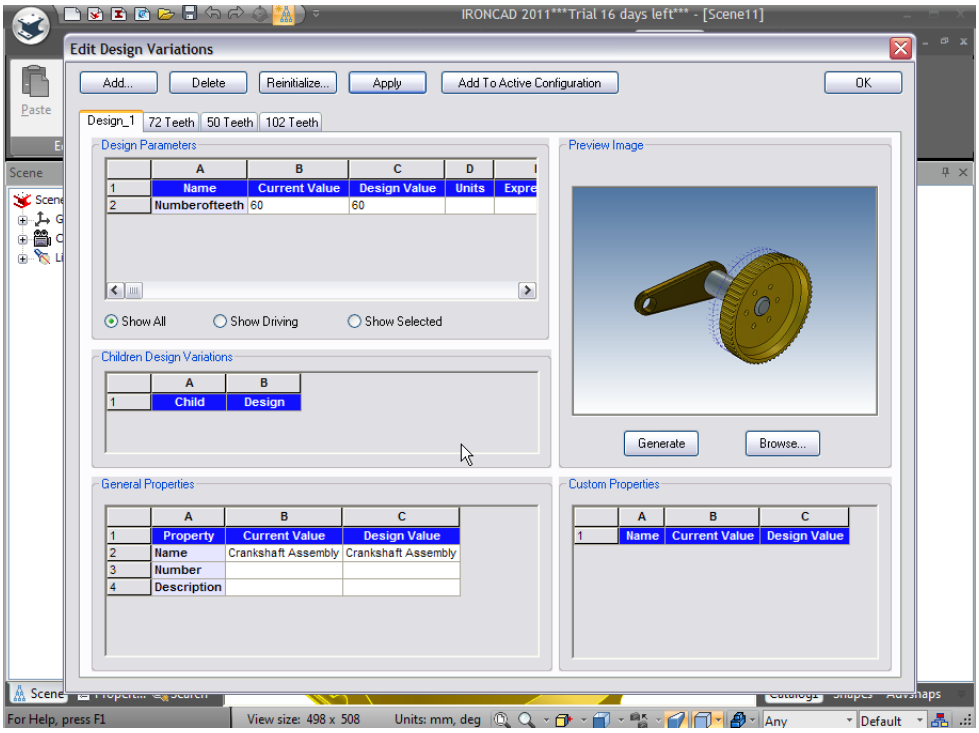
Now that the Design Variations have been created, it is good to cycle through the variations to ensure that they are returning the expected results.

1. **Place the Assembly into a New Catalog**

   Open a new catalog by going to the Common tab on the Windows Ribbon bar and selecting New. IRONCAD will display a new, empty catalog. Select the Crankshaft Assembly and drag it onto the catalog.

2. **Create a New Scene to Use the New Catalog Entry**

   Create a new scene and drag the catalog item into the scene.



   IRONCAD will immediately open the Design Variations panel.

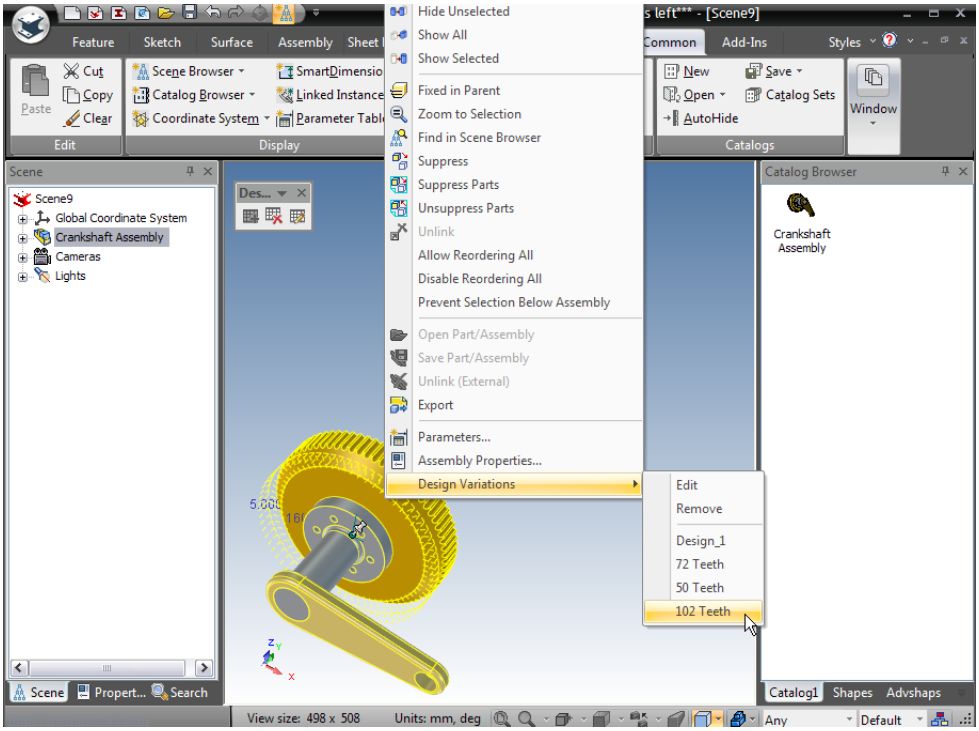
3. **Select a Design Variation**

   Select the 50 Teeth tab. click Generate and once the model has refreshed click OK.

   IRONCAD will create a new Crankshaft Assembly using the values entered into Design Variations.

4. Change the Design Variation

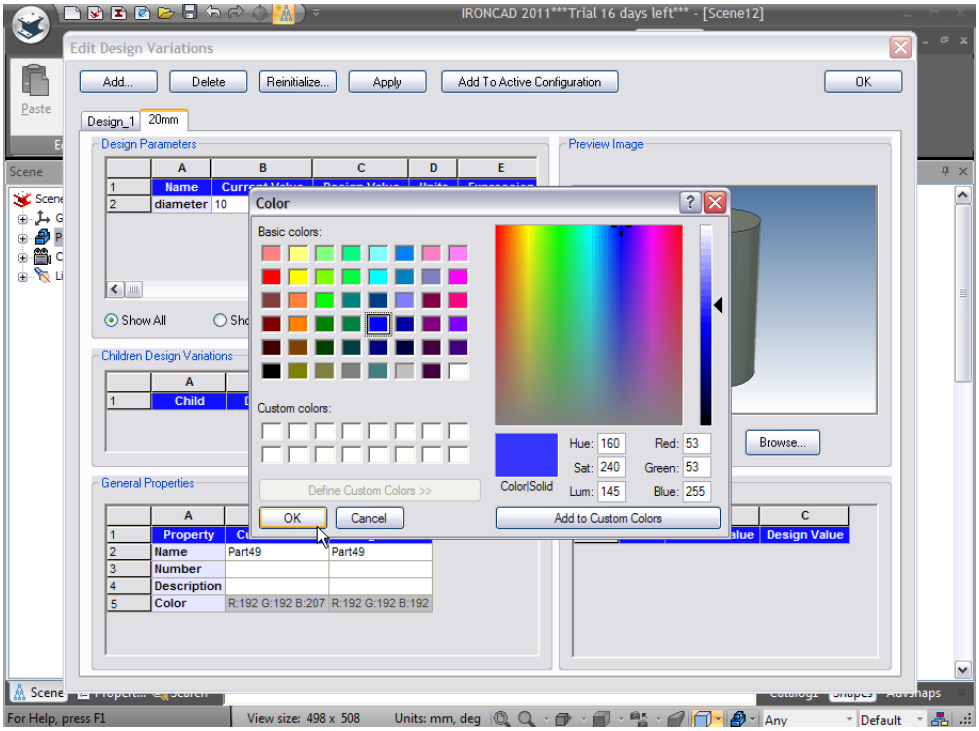
Right-click on the new Crankshaft Assembly to reveal a new entry in the menu for Design Variations. Select the 102 Teeth version and IRONCAD will immediately update the assembly.



From the same menu, custom variants can be created via the Edit option. Selecting Edit displays the familiar Design Variations control panel.

The above example uses a single value to update an assembly. The same techniques can be used at the part level at which time Design Variations can also apply a color to reflect different parts. For example, in a new scene, drag a Cylinder onto the scene and select it at the IntelliShape level. Now right-click on the IntelliShape and select 'Edit Cross-section'. Right-click on the center point and say Lock. Then add a Smart Dimension to the circle and accept the edit by right-clicking and saying Finish Drawing.
In the scene browser select the cylinder at part level, or select the cylinder in the scene so it is highlighted with cyan edges. Right-click and select parameters. Now add a parameter, Diameter, with a value of 20 and type length. Click the Show all Parameters option and in the Expression box against the dimension created (likely to be pR1) enter the word Diameter

to reflect the part level parameter created. The cylinder diameter will update appropriately. Close the parameter dialog box by clicking on OK. Select the cylinder at part level and click on the Add Design Variations icon followed by Edit Design Variations. The now familiar Design Variations panel will be displayed. However this time, in the General Properties pane at the lower left corner of the Design Variations panel there will be an entry for color.

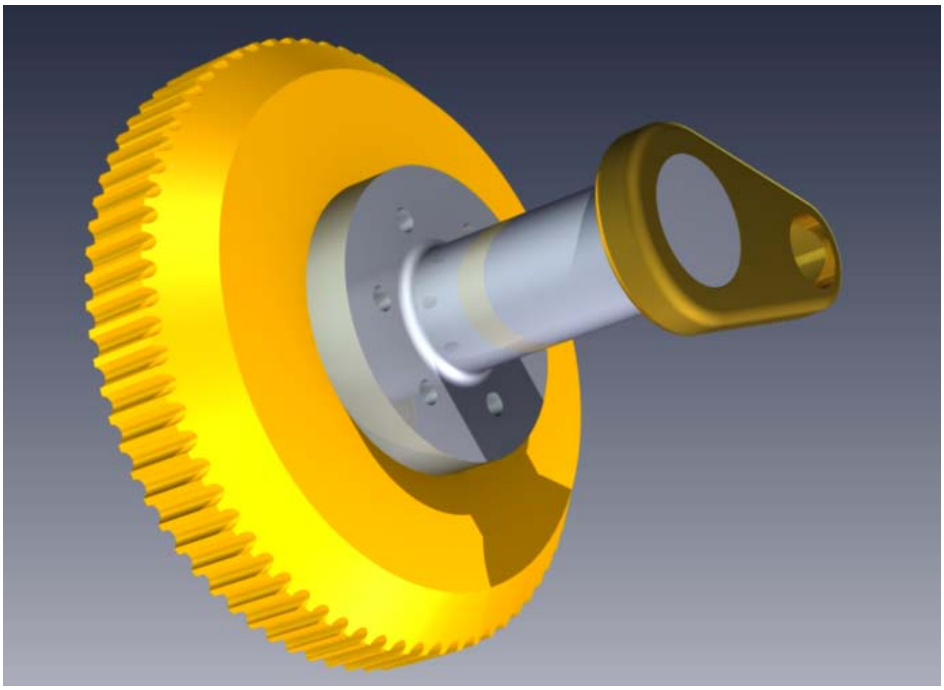


Add a new Design Variation, say 20mm. Change the Design Value for the diameter to 20 and click in the Design Value color box in the General Properties pane. IRONCAD will display the standard color picker to select a color from. Select a color and accept the selection by clicking OK followed by Generate to update the preview model.

Close the Design Variations panel by clicking OK. Right-click on the cylinder and select 20mm from the Design Variations list. The cylinder will change diameter and color to reflect the current variant.

The same steps can be used to set the Part Name, Description, and custom property added to the parts. Basically the selected Design Variation can define the new part properties.

This concludes the section on using 2D geometry to design and control IRONCAD parts. This chapter focused on IRONCAD's advanced capabilities for creating and editing IntelliShapes using 2D geometry. This chapter guided you through the process to create a simple assembly of parts. At this stage, your assembly should look similar to the next image:



Topics covered in this chapter included:

- Shape Handles vs. Sizebox Handles
- Using the Basic 2D Editing tools
- Creating precise sketches using Endpoint Dimensions, Curve Dimensions, and Curve Handles
- Editing Cross-Sections of Existing IntelliShapes
- 2D Constraints
- Drag-and-Drop IntelliShape Modeling vs. 2D Cross-section Modeling
- Setting Part and IntelliShape Names and Colors
- Using the Advance Shapes Catalog
- Saving the Part
- Creating a Crank Shaft
- Adding the Shaft
- Creating the Crank Arm
- Parametric Constraints with Expression

- Creating an Assembly
- TriBall and Linked Components
- Adding Intelligence Using Parameters
- Design Variations

In the next chapter, we will focus on the advance capabilities of the TriBall.

5. THE IRONCAD TRIBALL

Hailed by some as “the most useful tool in the history of CAD,” the TriBall is a powerful and flexible tool for performing 3D spatial transformations in IRONCAD. Most of the examples here show how to position individual parts. It is important to know that the TriBall can be used in many other applications within IRONCAD, including:

- Part and Assembly Positioning
- Feature and Profile Plane Positioning
- Direct Face Modeling, Loft, and Sweep Path Manipulation
- Key Frame Animation Path Manipulation
- Texture Mapping Placement
- Camera, Point and Spot Light Positioning
- Anchor and Attachment Point Positioning

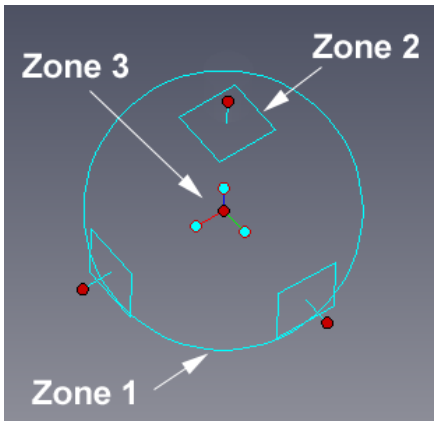
This section will break the TriBall down into 3 simple functional zones and then demonstrate, via examples, some of the more advanced capabilities of the TriBall such as:

- Orientating an item using the TriBall
- Build an assembly using the TriBall
- Reposition and orientate the TriBall with respect to its parent item
- Rotating, copying and linking an item using the TriBall.

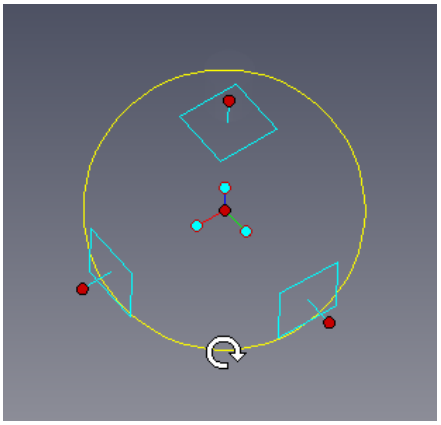
Although the TriBall was introduced in the Quick Start Guide, taking a little time to understand the 3 functional zones of the TriBall will crystallize the concepts and power of the TriBall.

5.1. Anatomy of the TriBall

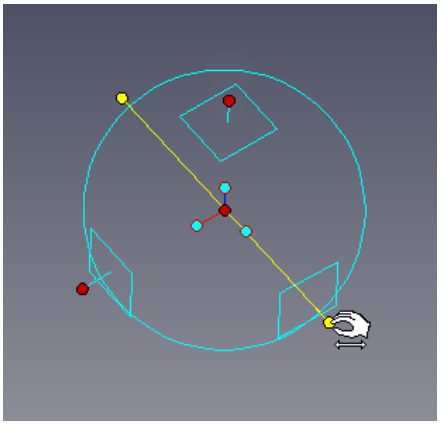
The TriBall provides tools to freely move an object about the plane of the screen (Zone 1), along defined axes or in planes (Zone 2) using the familiar left click for visual positioning or right-click for controlled precise movements. In addition the TriBall provides control of position and orientation of the item being manipulated (Zone 3). The Zone 3 tools are also used to alter and control the position of the TriBall.



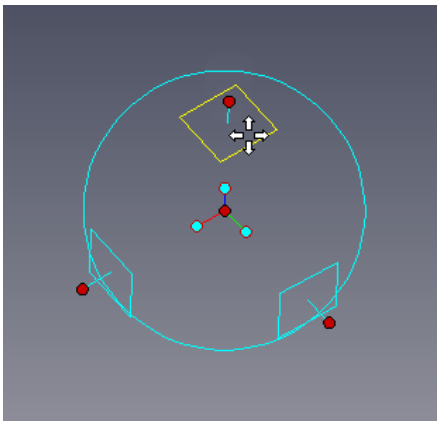
In addition, IRONCAD provides visual feedback via the symbol by the cursor:



Rotate/copy in the plane of the screen



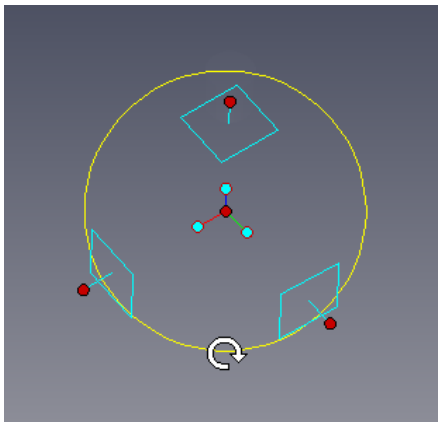
Rotate/copy about a defined axis



Move/copy in plane

5.1.1. TriBall Zone 1

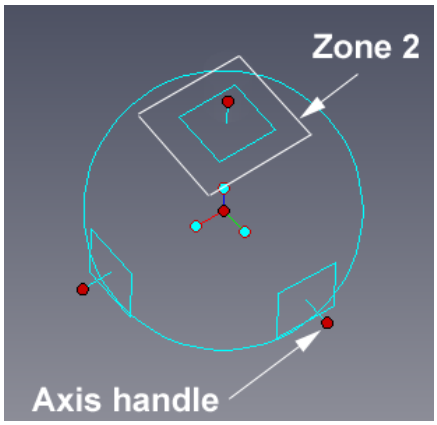
Zone 1 is the circle drawn on screen used to define the outer boundary of the TriBall. By placing the cursor over the boundary circle the mouse cursor changes to a clockwise rotating arrow and the boundary circle changes from cyan to yellow.



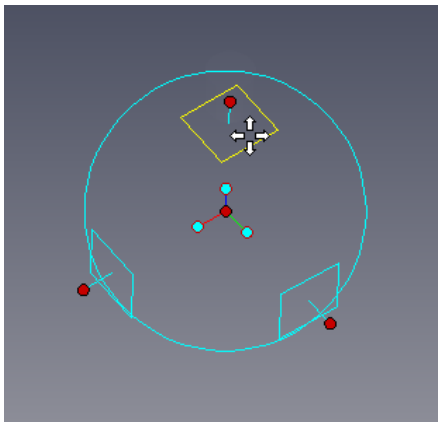
Clicking and dragging the left mouse button while over the boundary circle will rotate the item about the current TriBall center position in the plane of the screen.

5.1.2. TriBall Zone 2

Zone 2 controls movement along defined axes or planes. Zone 2 controls are probably the most used functions of the TriBall.



On the outside of the boundary circle are 3 squares that represent individual planes and as such are at 90 degrees to each other. Emanating from the center of each plane is a handle, the axis handle. The planes and axis handles are orientated with respect to the anchor of the item being manipulated. Positioning the cursor over one of the plane symbols results in the cursor graphic changing to 4 arrows perpendicular to each other.

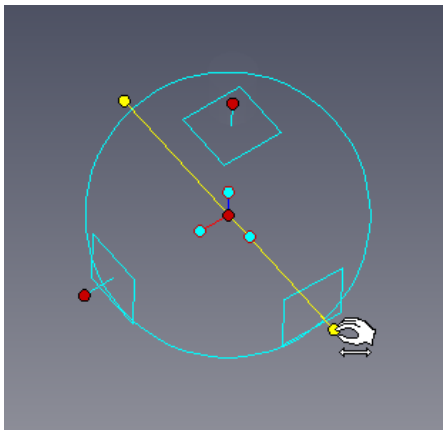


Left clicking and dragging with the plane highlighted in yellow will visually move the item in that plane, right-clicking will result in a dialog giving precise control. As an aid IRONCAD will display a pair of dimensions to give feedback of the movement.

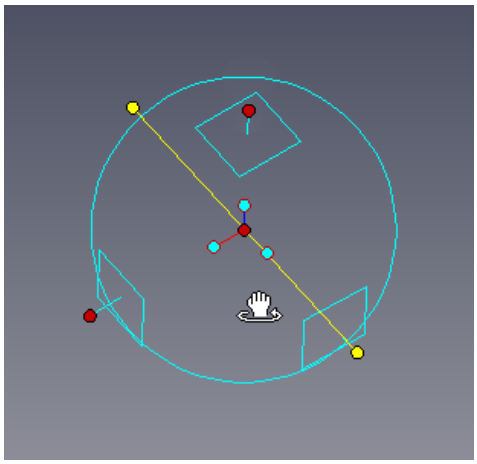
The axes handles are used for three primary actions:

- Moving in a defined direction,
- Rotating about an axis,
- Adding extra controlling the direction of movement when positioning using Zone 3 functions.

Clicking on one of the handles will highlight the handle passing through the TriBall this declares an axis for movement or control even when simply clicking on the handle and dragging in one movement.

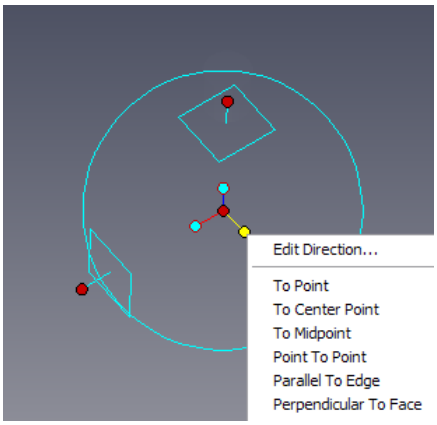


Again, IRONCAD will give feedback of any positional changes and change the cursor shape. With the axis 'declared' moving the mouse pointer inside of the outer boundary circle will cause IRONCAD to change the cursor shape one more time. By clicking and moving the cursor (keeping it inside the boundary circle) the item will be rotated about the declared axis.



5.1.3. TriBall Zone 3

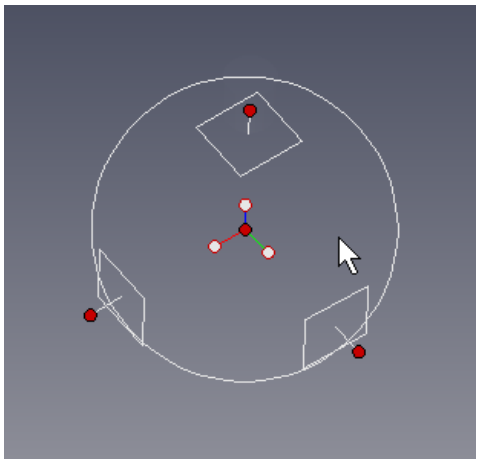
Zone 3 refers to the inner three orientation handles with cyan ends and the center red point. The controls provided by Zone 3 impact the orientation of the item being manipulated or the TriBall such as parallel to an edge etc. To view the full range of options, right-click on one of the cyan handles.



The center red point (the center handle) provides a quick way to freely drag the item to an existing point on any model in the scene, simply click on the center handle and, with the mouse button pressed, drag to the desired point on any model in the scene. To assist, IRONCAD will provide SmartSnap feedback to position the item.
Furthermore, if an axis handle is selected, IRONCAD will align the item being manipulated along the axis but aligned to the selected point.

5.1.4. Repositioning the TriBall

To TriBall would not be quite so powerful if its position was fixed within an item or space. By pressing the space bar when the TriBall is active results in the TriBall base color changing from cyan to white and any resulting manipulations impacting the TriBall itself and not the item.



5.2. Practical Examples using the TriBall

The following examples will introduce the fundamentals of Zones 2 and 3 in a practical way. At the time of installation IRONCAD installs two scene files under the Tutorials folder, by default (for 2011) this is:

Program Files\IRONCAD\2011\Tutorials\en-us.

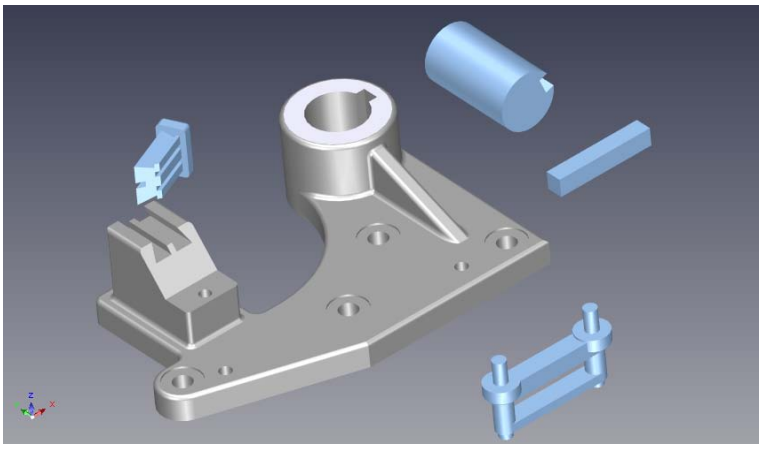
The two scene files are “triball1.ics” and “triball2.ics”. The examples will show the power provided by the TriBall by:

- Using the center handle of the TriBall
- Using the inner “orientation handles” of the TriBall
- Using the space bar to detach and reposition the TriBall
- Temporarily constraining (declaring) an axis of the TriBall
- Increment snapping with the TriBall
- Using the TriBall to copy radial / linear array patterns
- “To Point” command
- “Point to Point” command
- “Parallel to Edge” command
- “Perpendicular to Face” command
- “To Center Point” command
- “Parallel to Axis” command
- “Reverse” command
- Creating “Patterns”

IRONCAD has three keyboard commands for the TriBall:

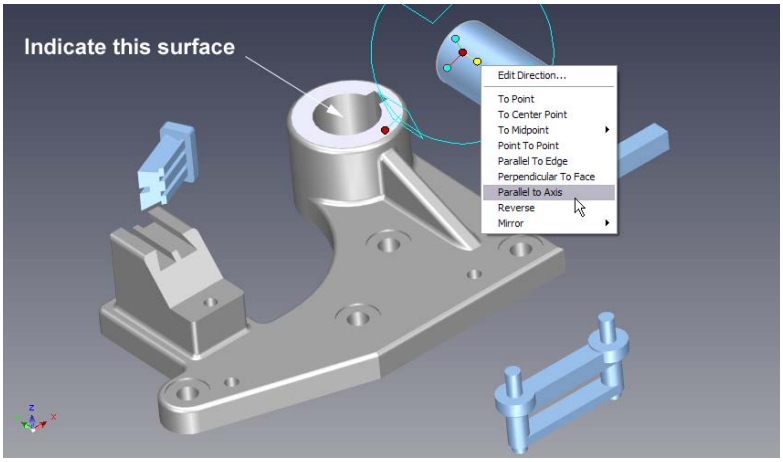
F10	Turn the TriBall on/off
SPACE	Detach / attach the TriBall with selected object
CTRL	Activate incremental snapping for translation / rotation

Open the file, **triball1.ics**, it should look something like the following image (the background has been changed to help show the scene contents better).



5.3. Using the TriBall's Orientation Handles to position parts

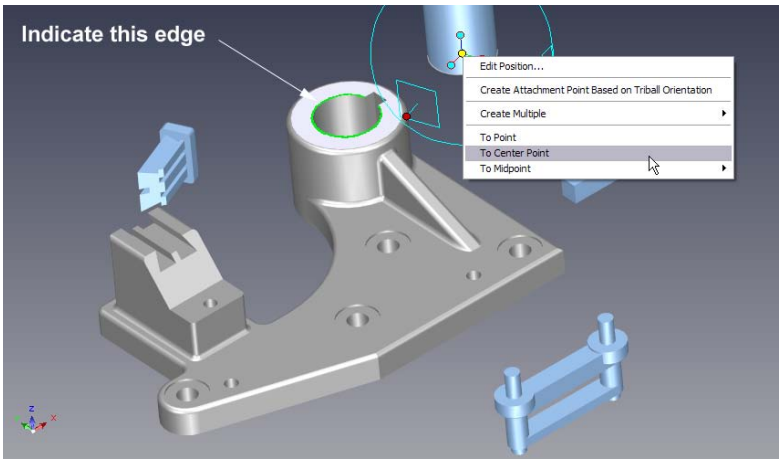
Select the shaft shown and turn on the TriBall.
Right-click on the orientation handle (Zone 3) running parallel to the axis of the shaft (as shown) and choose 'Parallel to Axis' from the resulting pop-up menu.



Then click on the cylindrical surface of the main, gray part shown in the image. This will cause the shaft's selected axis to be aligned with the axis of the hole. Note that, in this case, the outer surface of the hole could have been chosen instead of the inner surface and the result would have been identical.

5.4. Using the TriBall's center point to position parts

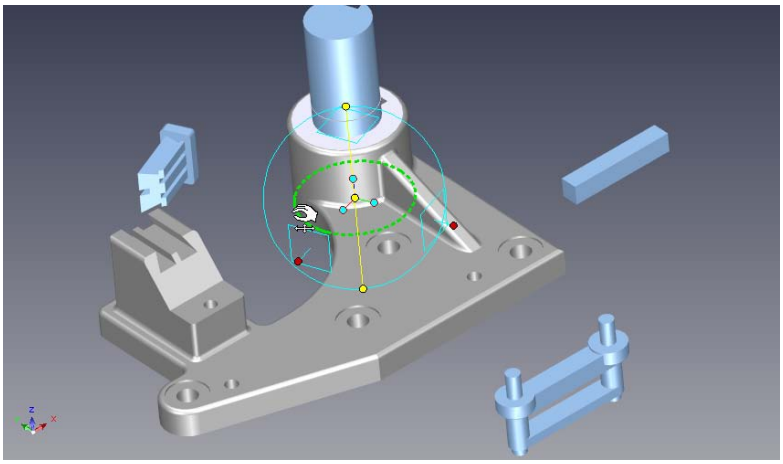
To move the shaft over to the hole's center, right-click the center of the TriBall and choose 'To Center-point' from the resulting pop-up menu.



Then click on the circular edge shown. This will move the TriBall center (and the shaft) to the center point of the target selection.

5.5. Temporarily constraining (declaring) an axis of the TriBall

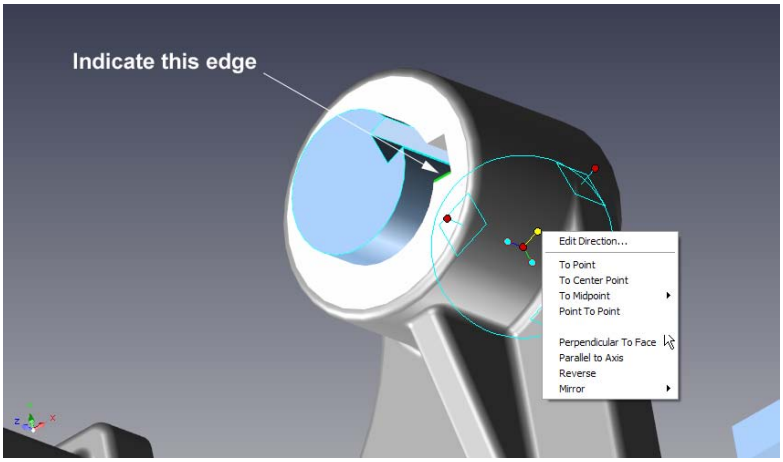
Now slide the shaft down to the base of the hole by first clicking on the top outer (Zone 2) TriBall handle shown. This action will cause the vertical axis of the TriBall to become highlighted in yellow, which means the TriBall is now temporarily constrained to move/ rotate only along/about that axis.



Now drag the center of the TriBall by left clicking and holding the mouse button pressed on the TriBall center red dot to the lower circular edge as shown. By declaring the axis, the cursor is effectively free to move outside of the TriBall to enable other features or parts to be selected. The shaft should “slide” down the constrained vertical axis and snap perfectly into alignment with the bottom of the hole.

5.6. Parallel to Edge command

Manipulate the viewing position to get a clearer view of the keyway in the shaft and housing. To align the keyway, right-click on the center orientation handle (Zone 3) shown and choose 'Parallel to Edge' from the resulting pop-up menu.

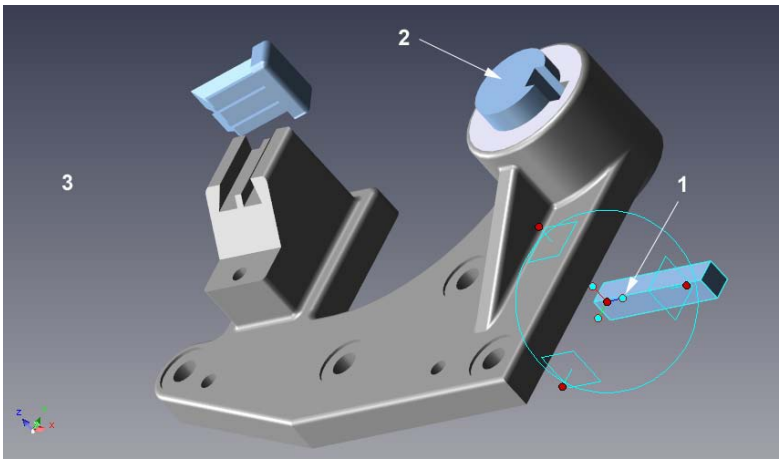


Next, click on the edge shown on the keyway. This will cause the selected axis of the TriBall to become aligned with the target edge by rotating about the TriBall center point.

Turn off the TriBall either by pressing F10 or by clicking on its icon.

5.7. Perpendicular to Face command

To insert the key into the keyway, select the key and turn on the TriBall.

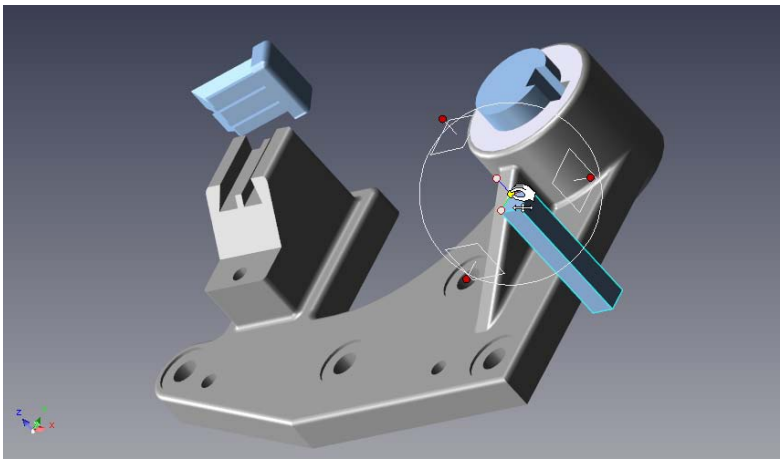


Align the key with the keyway by right-clicking on the center orientation handle shown (1) and choose 'Perpendicular to Face' from the resulting pop-up menu. Then click the

top surface of the shaft positioned previously (2). This will cause the selected axis of the TriBall to become aligned perpendicular with the target face. Click in a blank area (3) of the scene to deselect the selected axis.

5.8. Drag-and-Drop method of repositioning the TriBall

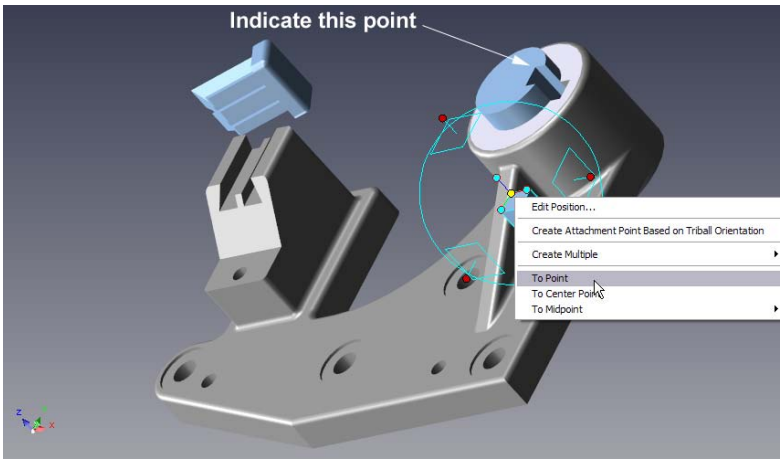
Relocate the TriBall on the part by pressing the Space bar. The TriBall color will now change to white, indicating that it is “detached” and can be moved independently of the part. Drag the center of the TriBall to the corner of the key, as shown (zoom in, if necessary).



Press the Space bar again to reattach the TriBall to the part (color returns to blue).

5.9. To Point command

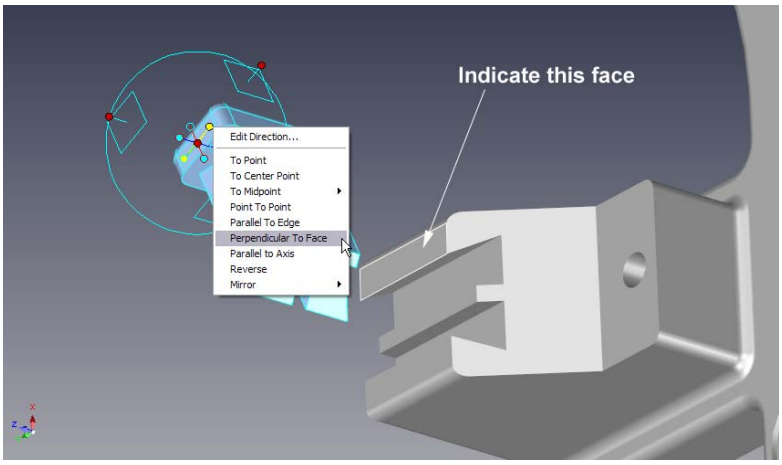
Position the key into the keyway by right-clicking the center of the TriBall and choose ‘To Point’ from the resulting pop-up menu and then select the corner point of the shaft. Alternately, drag the center of the TriBall to the corner point of the shaft. Both approaches have the same result.



The key should now be assembled into the keyway, turn off the TriBall either by pressing F10 or by clicking on its icon.

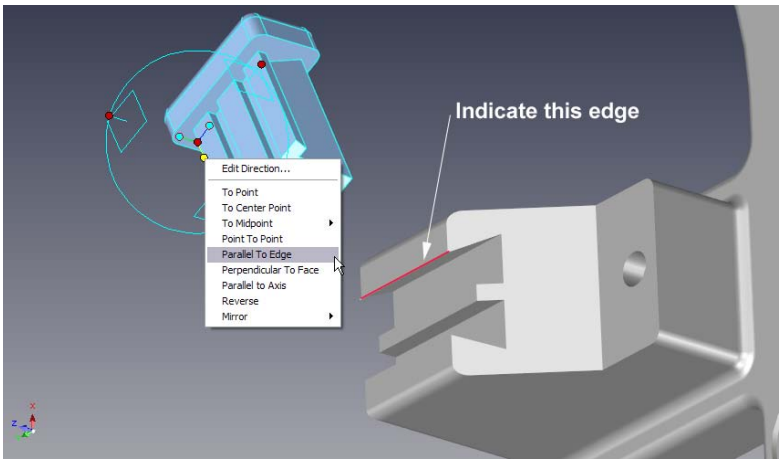
5.10. Aligning the Dovetail component

Select the dovetail part and turn on the TriBall. Referring to the image below, right-click on the orientation handle shown and choose 'Perpendicular to Face' from the resulting pop-up menu.

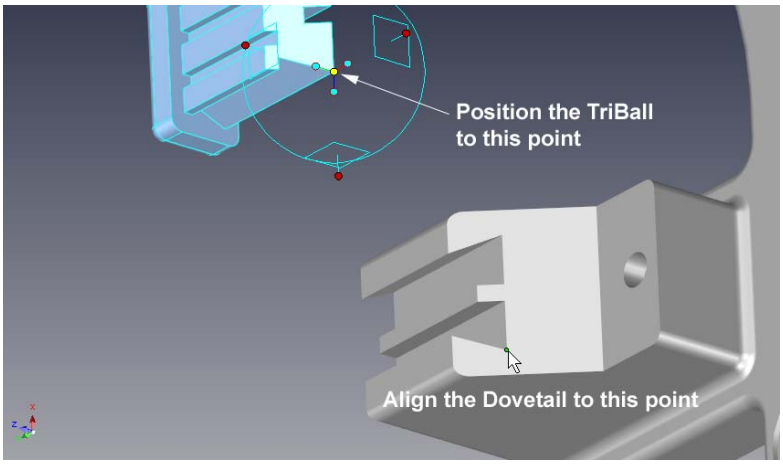


Click the face shown in the image and then deselect the handle by left clicking in the scene background.

To align the Dovetail component to the dovetail grooves right-click on the orientation handle parallel to the grooves and choose 'Parallel to Edge' from the resulting pop-up menu and then indicate the edge shown in the image below.



Click in a blank area of the scene to deselect the selected axis.
Relocate the TriBall on the part by pressing the space bar. The TriBall color will now change to white, indicating that it is “detached” and can be moved independently of the part.
Now, drag the center of the TriBall to the corner shown. Then press the space bar again to reattach the TriBall to the part (color returns to blue). Once the TriBall has been locked in place, left click on the center point of the TriBall and drag the Dovetail component into position by indicating the point shown in the following image. IRONCAD will help in this process by using the SmartSnap feedback.

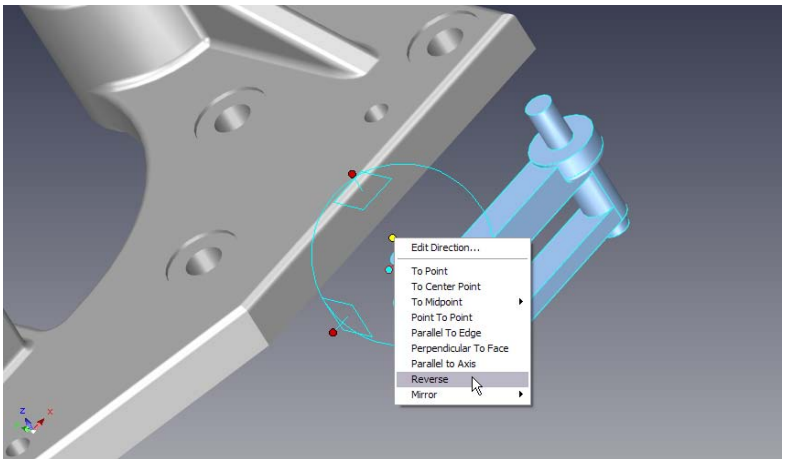


Alternately, right-click the center of the TriBall and choose ‘To Point’ from the resulting pop-up menu and then select the same point.

The part should now be assembled correctly.

5.11. The TriBall Reverse command

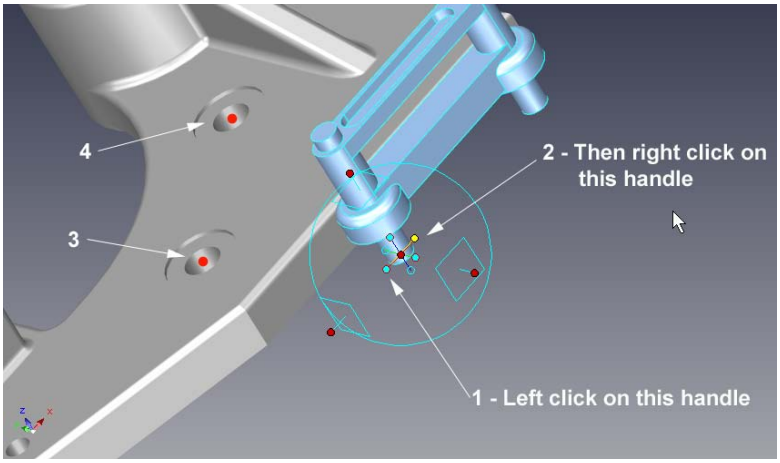
If the TriBall is still active on the Dovetail component, turn it off and select part shown and turn the TriBall back on.



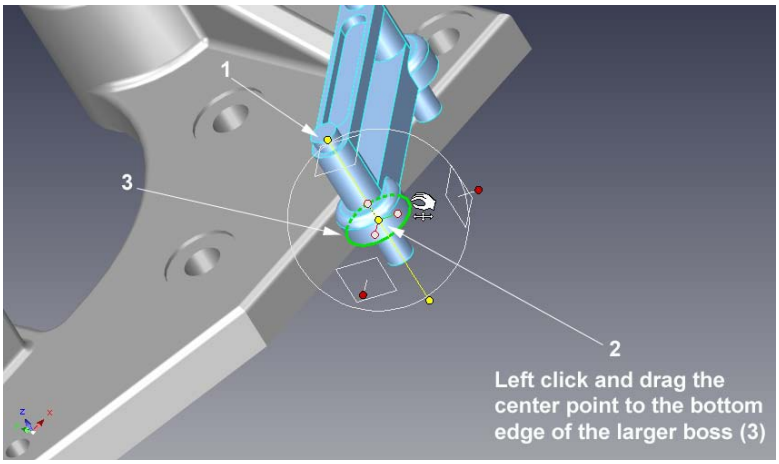
Right-click on the top orientation handle shown and choose ‘Reverse’ from the resulting pop-up menu. This will “flip” the part 180 degrees in the direction of the selected axis (note: reverse is different to mirroring the part).

5.12. Point to Point command

The TriBall is displayed with the minimum of handles to help maintain a clean working environment, however when a handle is selected IRONCAD displays the opposite handle. For example, to align the pegs with the holes, select the orientation handle in the next image (1) by left clicking. IRONCAD will display the opposite handle (2) first right-click on newly display orientation handle will indicate to IRONCAD the direction of manipulation. From the pop-up menu select ‘Point to Point’ and select the target hole center points (3) followed by (4). This will cause the selected axis of the TriBall to become aligned parallel to a virtual line between the two target points. Undo the command and this time indicate the hole center points in the opposite sense.

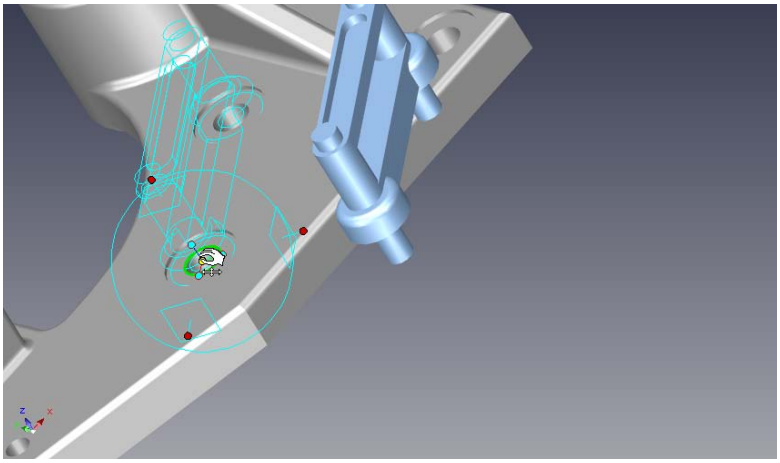


Relocate the TriBall on a feature of the part reflecting how it would be assembled by pressing the space bar. IRONCAD will change the TriBall color to white. Next referring to the following image, click on the top outer TriBall handle (1). This action will cause the vertical axis of the TriBall to become highlighted in yellow, which indicates that the TriBall is now temporarily constrained to move/rotate only on that axis.

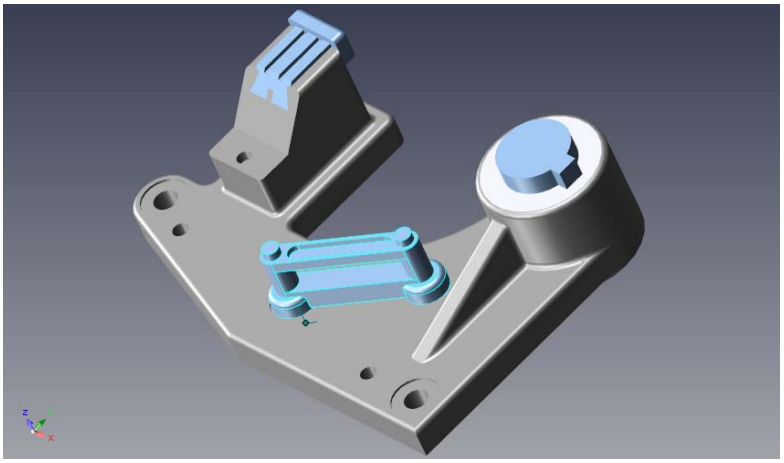


With the axis constrained and the TriBall highlighted in white, drag the center of the TriBall (2) to the lower circular edge (3). The TriBall should “slide” up the constrained vertical axis and snap perfectly into alignment with the base of the peg. Lock the TriBall to the new position by pressing the space bar and the TriBall will turn back to cyan.
Click in a blank area of the scene to deselect the selected axis.

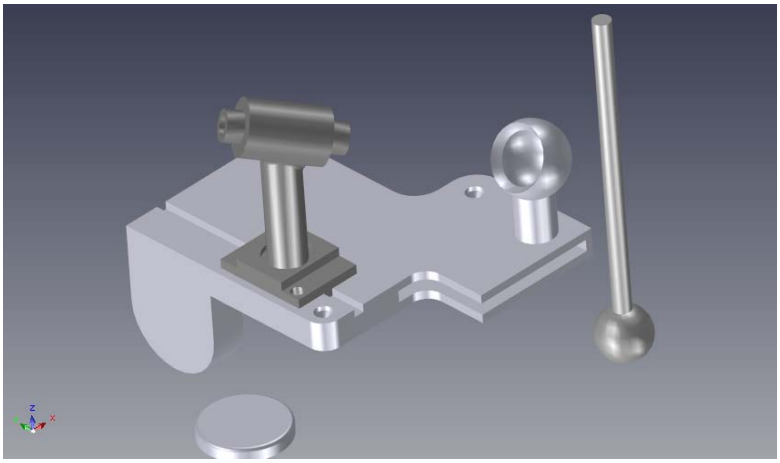
To place the pegs in the holes, simply drag the center of the TriBall to the center of the hole.



An alternative method to achieve the same result is to right-click on the center of the TriBall and select 'To Center Point' from the resulting pop-up menu and then click the circular edge of the hole.
The assembly is now complete and should look like the following image:



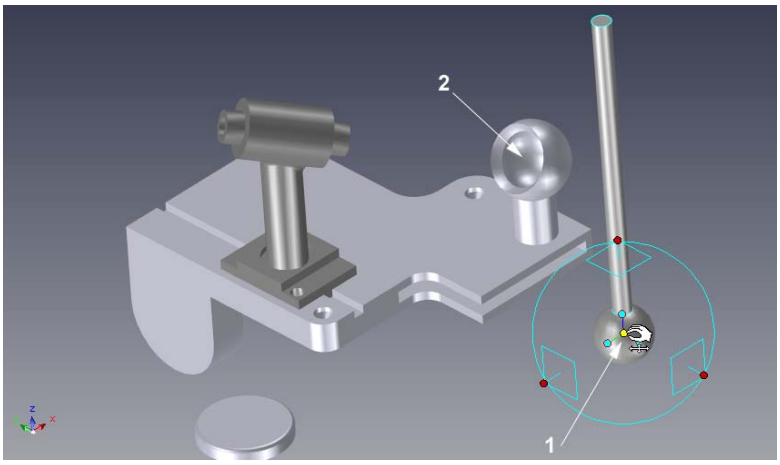
To explore some of the other positional functions open ***TriBall2.ics*** found in the tutorials folder.



TriBall2 should look like the above image (as with the previous example the background has been changed to help illustrate the TriBall).

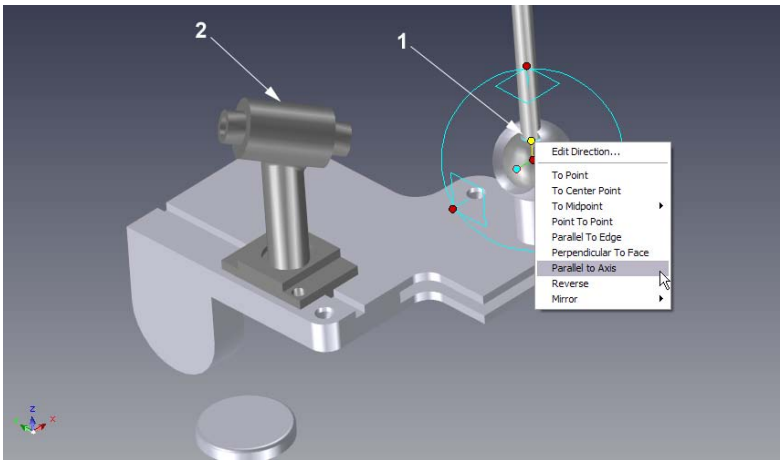
5.13. To Center Point command

To position the ball-ended shaft in the socket, first select it and turn on the TriBall. Move the shaft to the center of the socket by first right-clicking on the center of the TriBall (1) and selecting 'To Center Point' from the resulting pop-up menu. Then click the inner spherical surface, (2). In this case, the outer spherical surface could be chosen as the target, as well, which would have the same result since both of the spherical surfaces are concentric.



Referring to the following image, to align the shaft with its guide hole right-click on the top orientation handle (1) and choose 'Parallel to Axis' from the pop-up menu. Next, click on the cylindrical surface (2) which will highlight in green when the cursor touches it. This will cause the shaft's selected axis to be aligned with the axis of the hole. Note

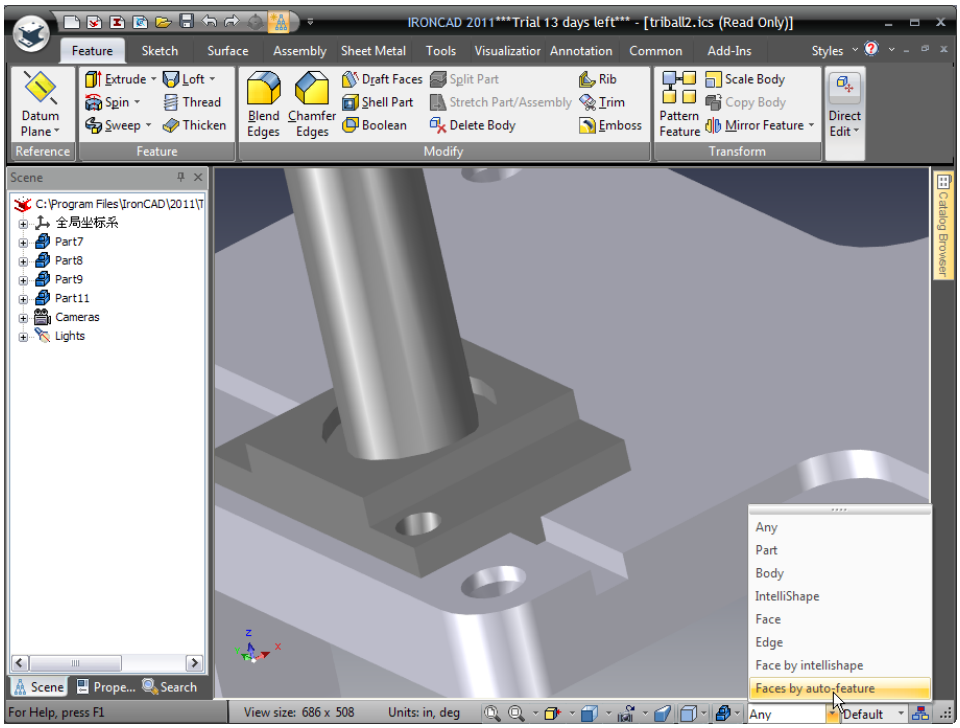
that there are several other co-axial cylindrical surfaces that could be used for the target selection which would have the same result.



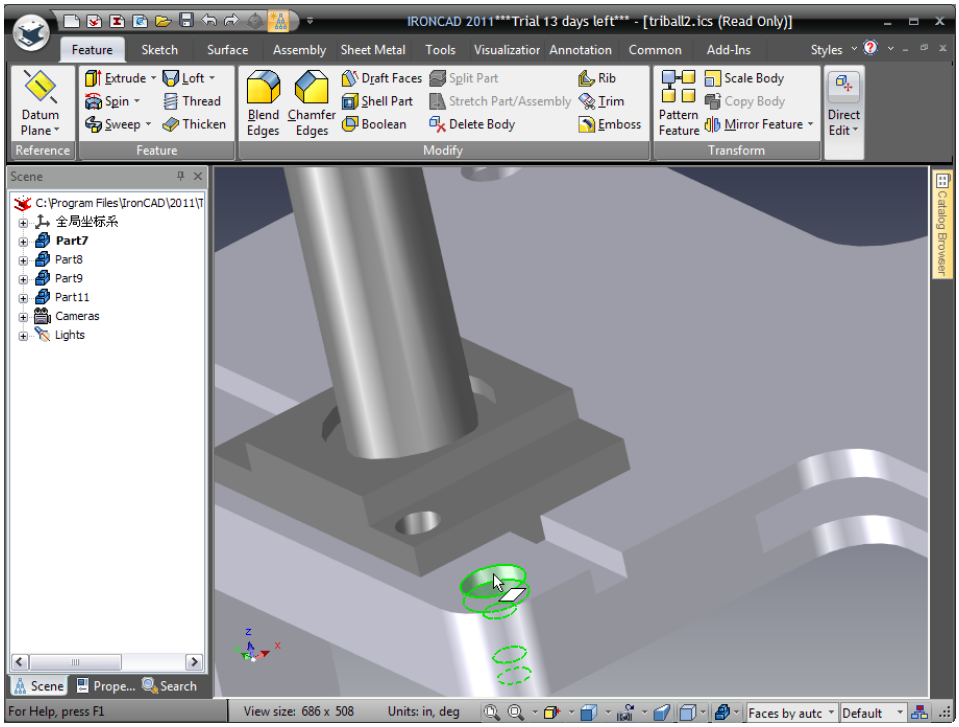
Deselect the TriBall.

5.14. Using the TriBall's 'Copy in Plane' option

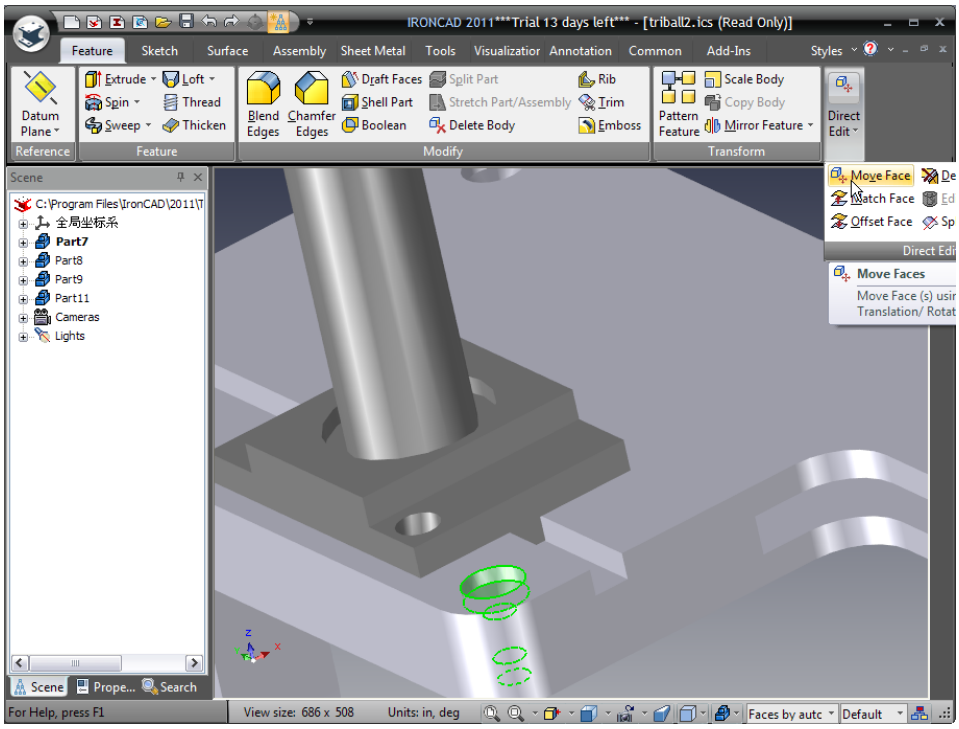
Using the TriBall, IRONCAD can modify features of imported geometry.
For example, the base part of TriBall2.ics is an imported part i.e. the part does not comprise any IntelliShapes or features. This can be seen by expanding the part in the scene browser, the structure shows Shape 15 only. Zoom into the part as shown in the following image and change the Selection Filter to 'Face by auto-feature'



Next select the counter-bored hole when its surface is highlighted as shown, noticing the cursor symbol changes to a white parallelogram when a surface is touched. IRONCAD will recognize the surfaces creating the counter-bored hole as a feature.

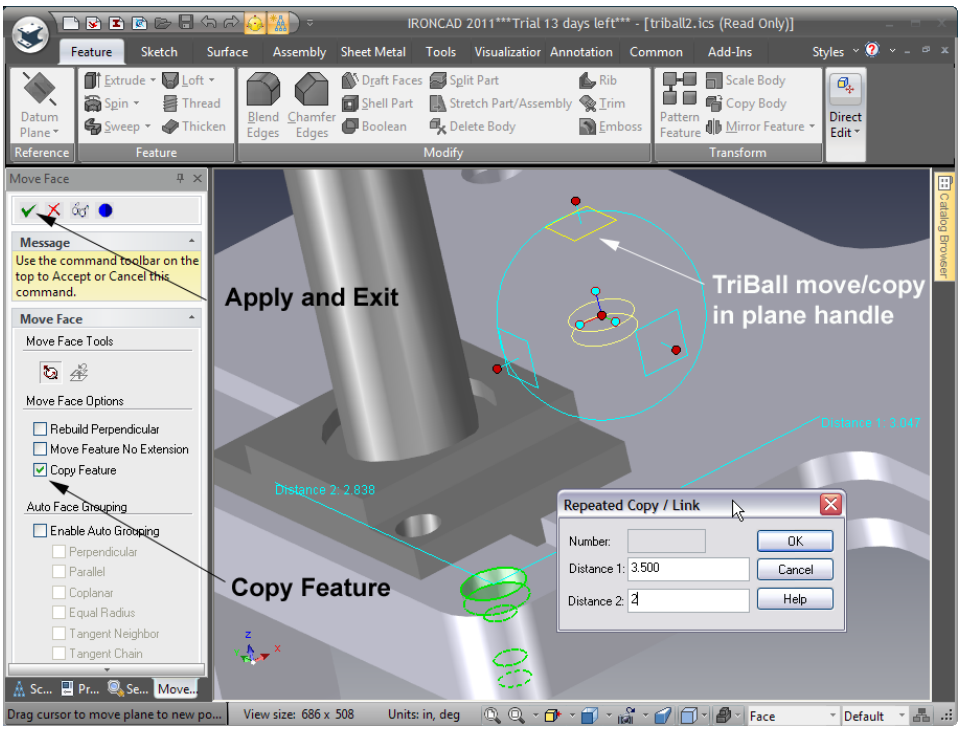


From the Windows Ribbon bar select the ‘Move Face’ tool from the Direct Edit panel. IRONCAD will activate the TriBall.



IRONCAD will also display the Property Panel for controlling the action of the TriBall appropriate to the current selection

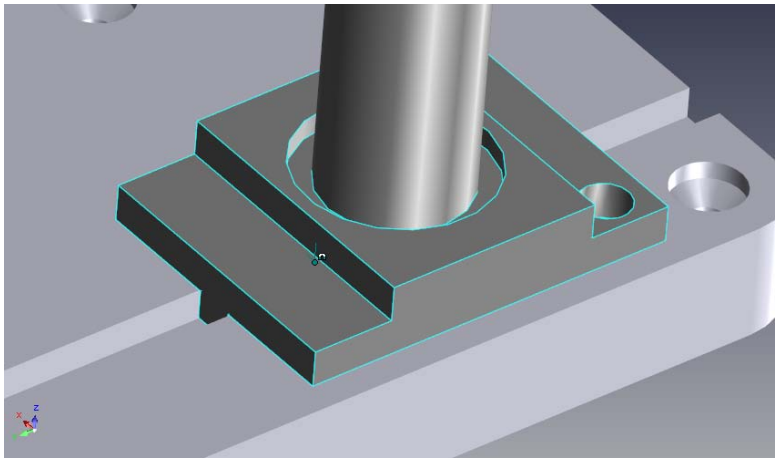
On the IRONCAD Property Panel menu, check ‘Copy Feature**’.** Next right-click and drag the top plane handle (the one parallel to the top face of the plate indicated in the image below) of the TriBall. In the data entry panel, enter 3.5 for “Distance 1” and 2 for “Distance 2” and click OK.



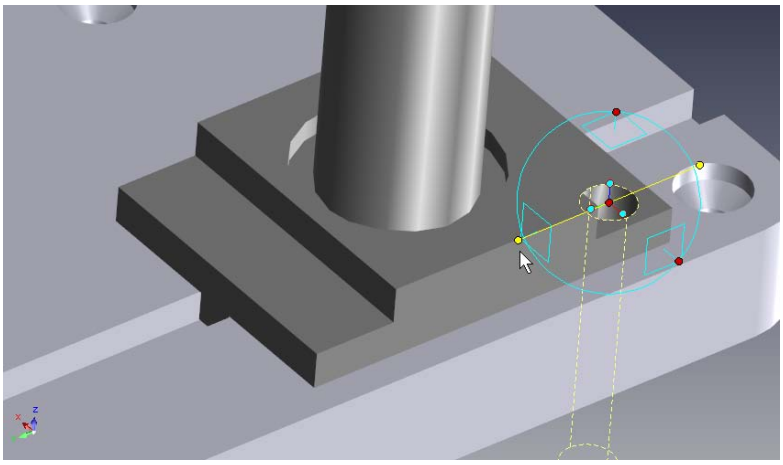
Finish the operation by clicking on the 'Apply and Exit' icon, in addition ensure the Selection Filter is reset to Any

5.15. Using the TriBall to create linked copies

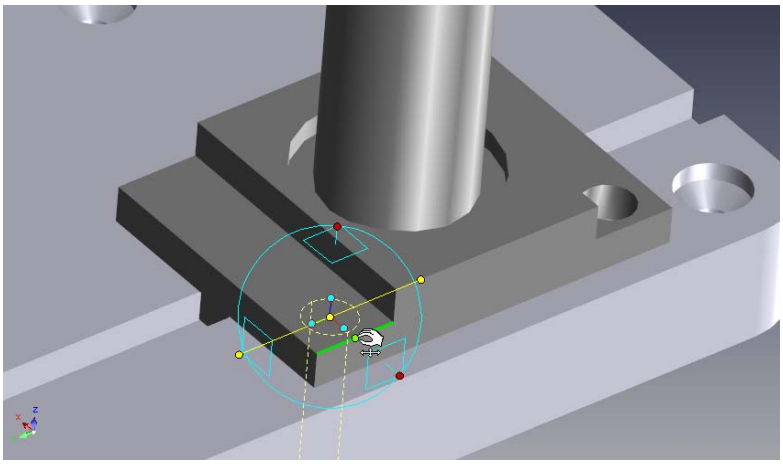
The SmartSnap feedback facility provided by IRONCAD reduces the need to enter dimensions and distances providing a high degree of efficiency to the design process. For example in the current example, the hole in the shaft support base needs to be copied onto the other flange face. Using the TriBall, the hole can be dragged into place and linked to the existing one so that any changes to either hole can be reflected automatically into its linked instance.



To use the TriBall to make a linked copy of the hole IntelliShape first, select the hole and turn on the TriBall.



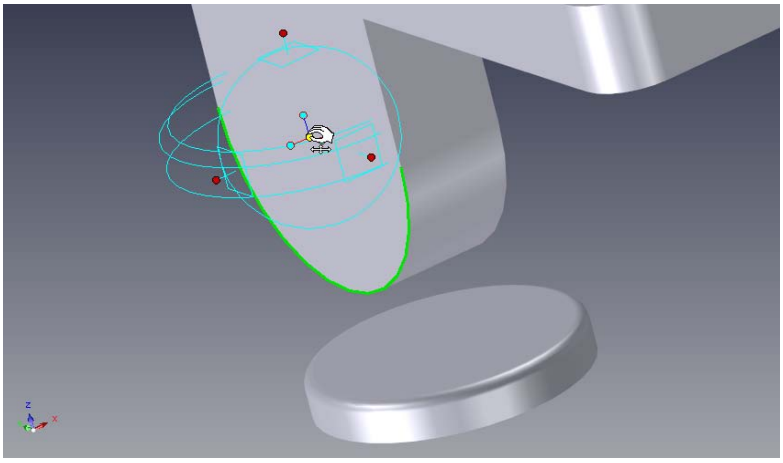
Click the outer TriBall handle, as shown. As in the previous examples, this action will cause that axis of the TriBall to become highlighted in yellow, which means that it is now temporarily constrained to move/rotate only on that axis.
Clicking and holding the right mouse button, drag the center of the TriBall to the midpoint of the edge shown using SmartSnap feedback. The TriBall should “slide” along the constrained axis and snap into alignment with the midpoint of the target edge.



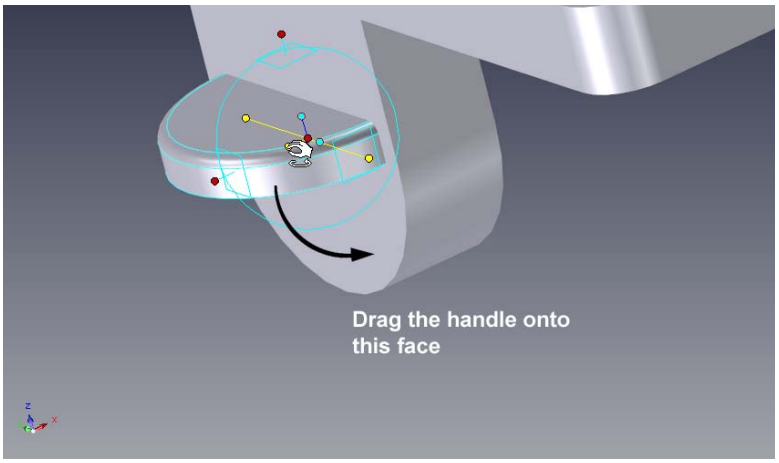
When you release the mouse button, choose ‘Link Here’ from the pop-up menu and then OK.
Deselect the TriBall tool.

5.16. Orientating using existing faces to ‘snap to’

Select the disk shown and turn on the TriBall. Left click on the TriBall’s center point and drag the center of the TriBall to the center point on the base, as shown. IRONCAD will highlight the center point of the cylindrical face and its edge using SmartSnap feedback.



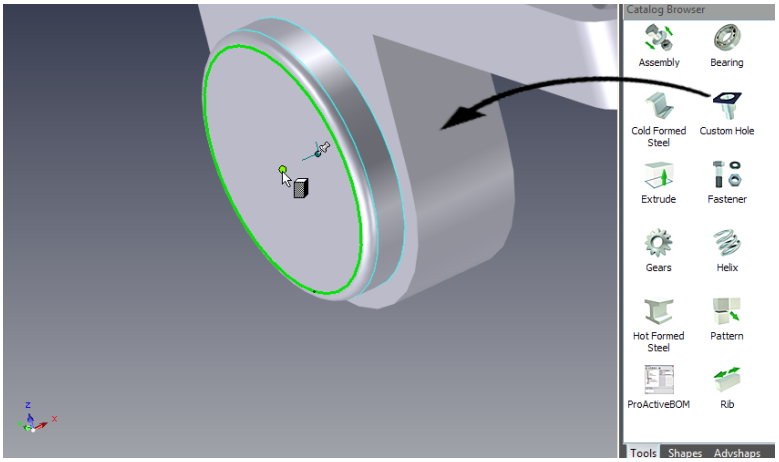
Click the outer TriBall handle, as shown, to constrain the axis. Now, drag the indicated orientation handle to the base surface shown.



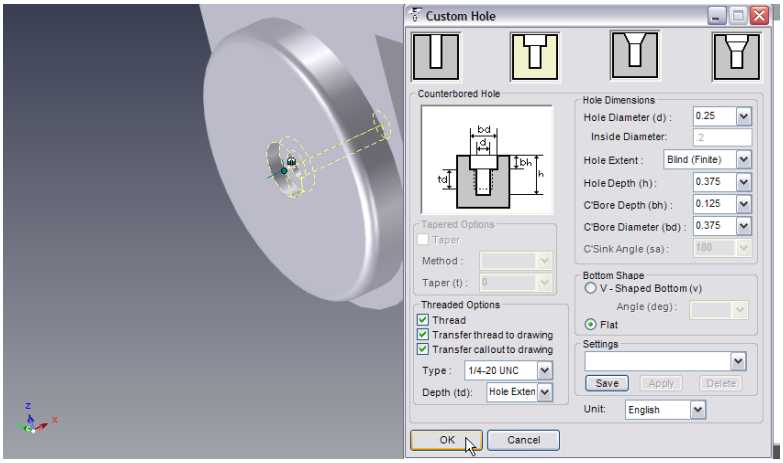
This is an example of using the drag-and-drop capabilities of the orientation handles as opposed to the “right-click/select” methods.
Turn off the TriBall.

5.17. Using increment snapping with the TriBall

The TriBall can be manipulated and positioned using snap increments defined on the fly. For example, add a threaded counter-bored hole by dragging a Custom Hole from the Tools catalog and dropping it onto the center of the disk, as shown.

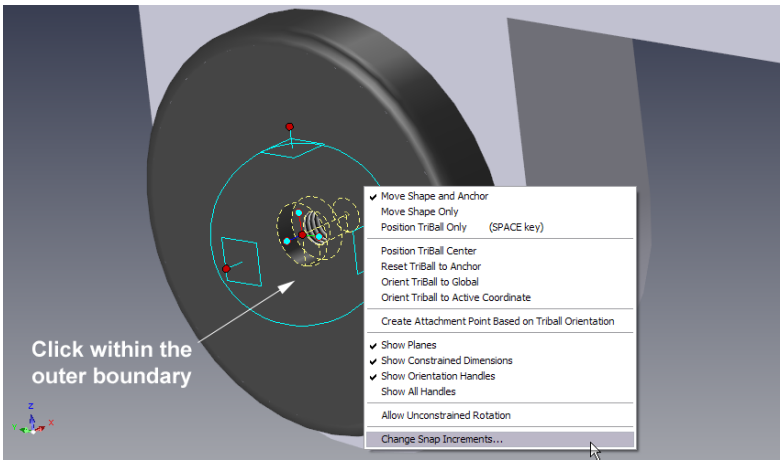


In the resulting Custom Hole dialog box, set the diameter of the counter-bore to 0.375 and the depth of the counter-bore to 0375 also check the thread option and edit the values, as shown and then click OK.

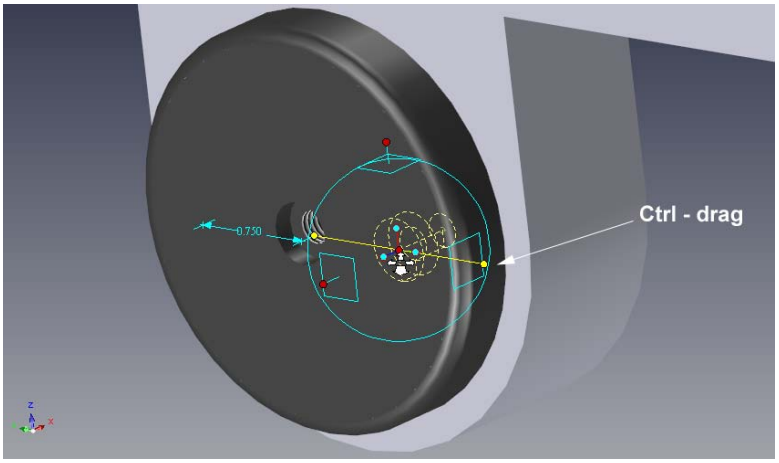


IRONCAD will construct a counter-bored and threaded hole in the center of the disk.

Select the custom hole just created and turn on the TriBall (the disc's color has been changed to highlight the TriBall more clearly).



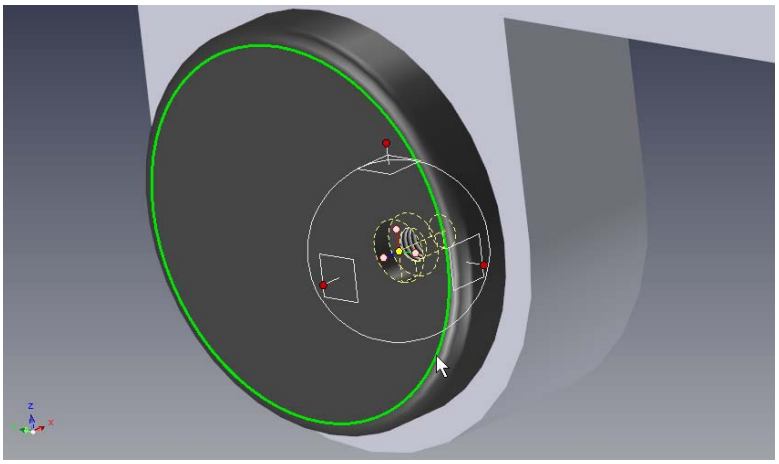
Then, right-click in an empty area inside the TriBall (i.e. not on a handle) and choose 'Change Snap Increments...' from the resulting pop-up menu.
Enter Distance value of 0.125 in the TriBall Snap Increments dialog and 45 in the Angle Increment box and then click OK.
To use the snap feature of the TriBall, hold down the Ctrl key and drag the outside TriBall handle to the right until the distance value displayed is 0.75**,** and release the mouse button.



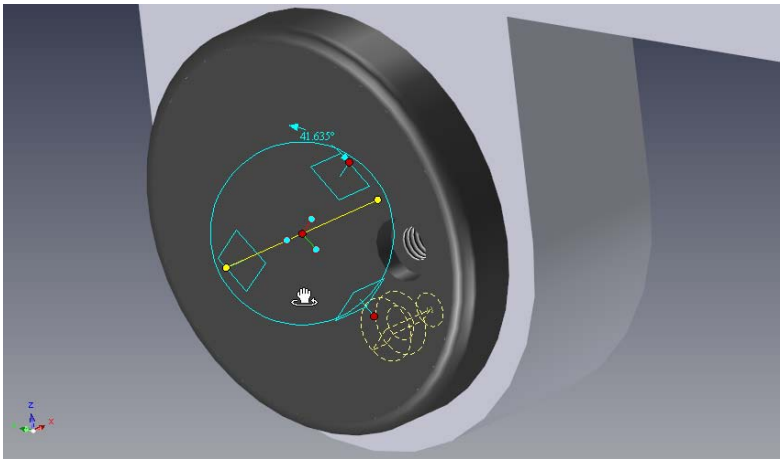
5.18. Creating a Radial Array Pattern with the TriBall

A common requirement in design is to produce a regular pattern of holes or features. The TriBall provides functions for making both radial and rectangular patterns of items. Right-click dragging and releasing on a plane handle provides the functionality for Rectangular Patterns whereas right-click dragging around an axis handle accesses the

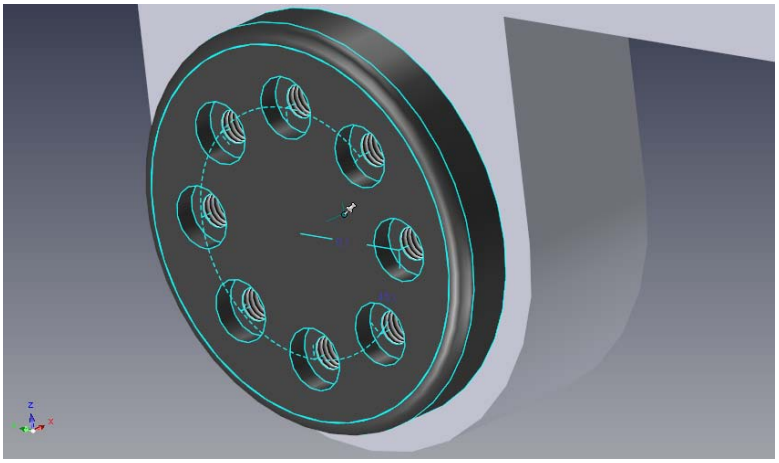
Radial Pattern function. This example explores the capability to make a radial array of holes around the center of the disk.
The TriBall must first be relocated to the center of the disk. To do this select the custom hole, activate the TriBall and press the Space bar. Right-click the center of the TriBall and choose 'To Center Point' and then click the circular edge of the disk.



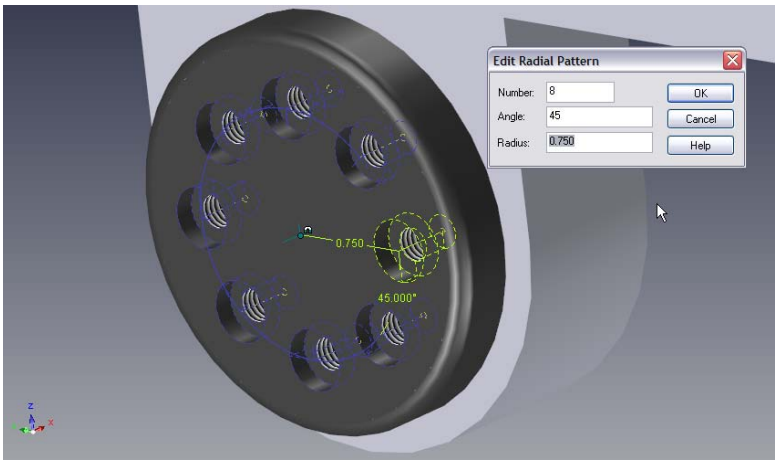
The TriBall will move to the center of the disk.
Press the space bar again to reattach the TriBall to the part (the color of the TriBall should change back to cyan). To create the radial array of holes about the center, click the outer TriBall handle shown to specify the axis of rotation.



Positioning the mouse inside the bounding circle will change the cursor to a gripping hand with an arrow around the wrist. Right-mouse-drag in an empty area within the TriBall ensuring the cursor is not on a handle. Release the right mouse button and choose Create Radial Pattern from the resulting pop-up menu.
Enter 8 in the Number box and 45 in the Angle box then click OK.



The part will resemble the image above. To edit the ‘Radial Pattern’ click on any one of the counter-bored holes. IRONCAD will highlight the radial pattern values in a bright green color. Right-click on one of the values to display a dialog box with the number of holes, the angle between each hole, and the radius of the pattern as shown below.



This completes the introduction to the TriBall. In this section the TriBall was broken down into 3 simple functional zones and the more advanced capabilities of the TriBall demonstrated, such as:

- Orientating an item using the TriBall
- Build an assembly using the TriBall
- Reposition and orientate the TriBall with respect to its parent item
- Rotating, copying and linking an item using the TriBall

The next chapter will introduce IRONCAD’s drawing creation capabilities.

6. IRONCAD DRAWING CREATION

IRONCAD provides a wealth of tools to create 2D drawings from existing 3D models or directly into the 2D drawing space for drawings such as schematics.

The fundamentals of an IRONCAD drawing are:

1. Create a 3D scene,
2. Save the part with a known name
3. Create a drawing using a pre-defined drawing boarder (template)
4. Add views and dimensions

IRONCAD maintains the relationships between the 3D scene and the drawing hence the essential step of giving the scene a name and saving it. Without this the drawing cannot hold a pointer to the drawing. From this simple standpoint all views, sections, detail views etc. are created.

Drawings can be of single parts or assemblies. They can be single or multi-sheet drawings. This chapter explores some common functions of drawing creation and introduces some of the more advanced features of IRONCAD's drawing creation capabilities later on. Although the chapter can be read in isolation from other chapters, it assumes a basic knowledge of the IRONCAD drawing creation concepts (introduced in the Quick Start section). The basis for the tutorial section is the scene file **drawingpart.ics** located in the tutorials folder of the IRONCAD installation folder usually:

Program Files\IRONCAD\201x\Tutorials\en-us (201x represents the version of IRONCAD)

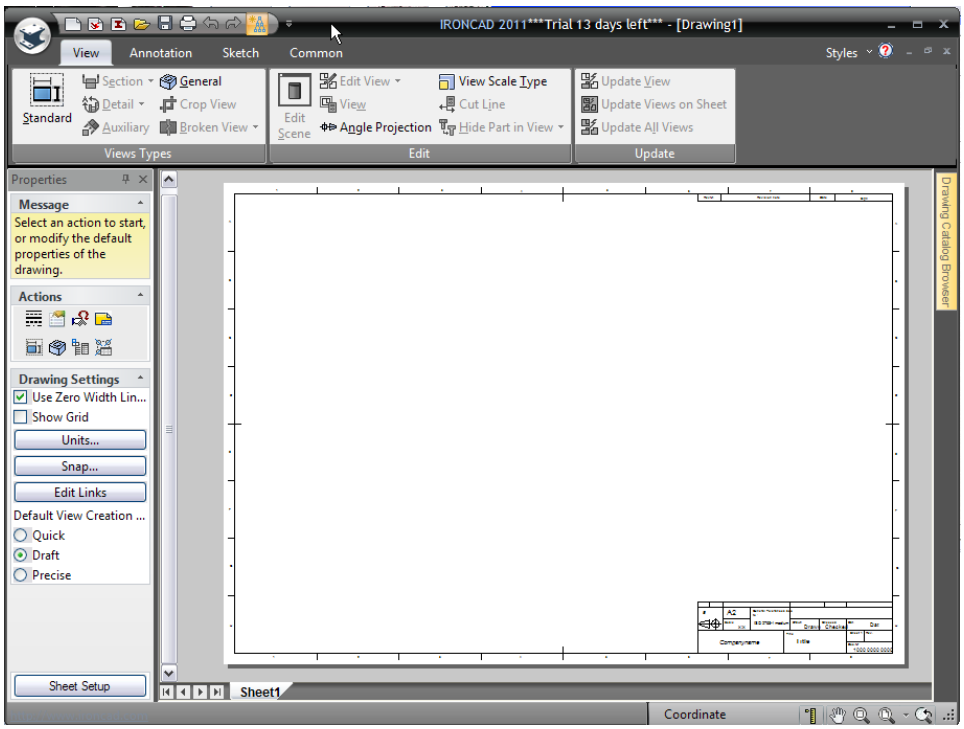
Topics covered in this chapter:

- Creating Standard Views
- Changing the View Scale and Rendering Options
- Moving Views
- Datums, Dimensioning & Tolerancing
- Centerlines and Reference Curves
- Section, Detail, Auxiliary, and General Views
- Multi-Sheet Drawings
- Hole Table
- Sheet Drawing Tools
- Styles and Layers
- Part/Drawing Associativity

6.1. Basic Drawing Creation

6.1.1. Creating the Standard Views

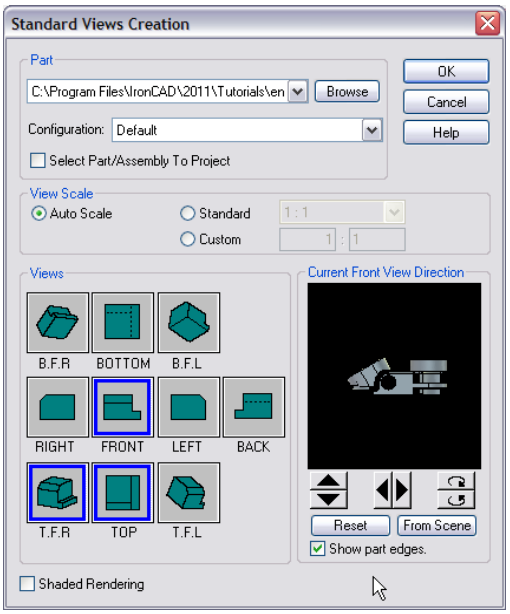
Open the scene file ***drawingpart.ics*** found in the tutorials folder created on installation, click on the IRONCAD Home Button and from the File menu, select New followed by Drawing, and then OK.
IRONCAD is delivered with three basic drawing standards. Each template within each standard can be customized to suit specific company standards. For clarity and consistency of results when following this section, from the ISO tab, select the template A2.icd and then choose OK.



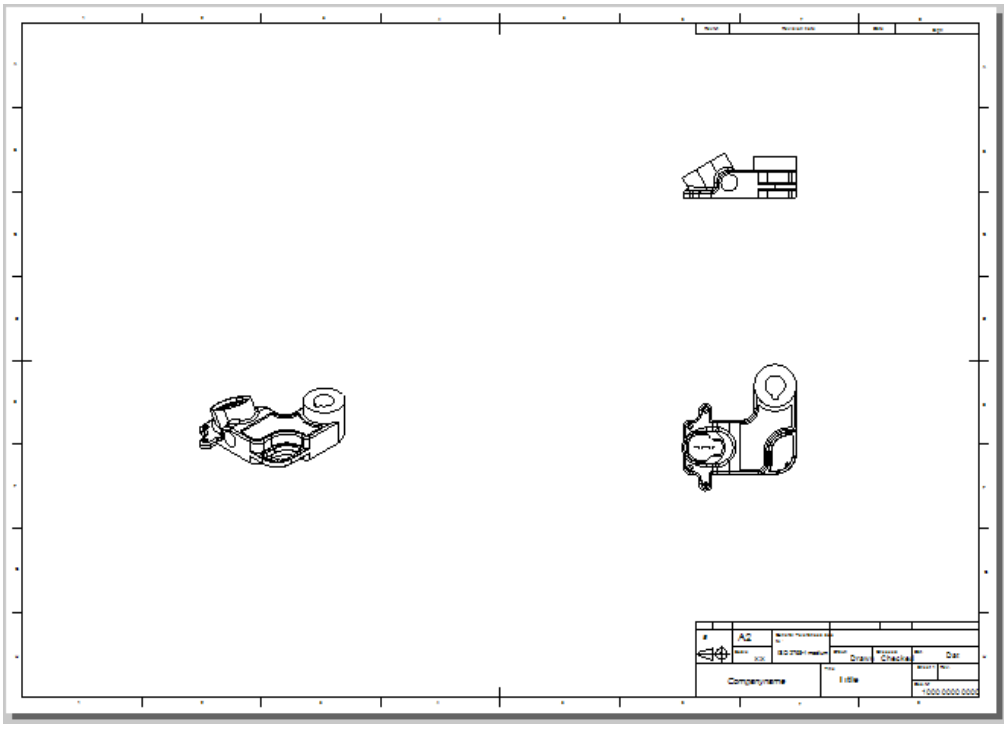
IRONCAD will display an empty drawing sheet and open the Property Panel down the left hand side of the screen. The Windows Ribbon bar will reveal 4 primary tabs:

View – setting up the drawing views, angle of projection etc.
Annotation – dimensions, notes etc.
Sketch – adding and editing geometry elements
Common – utilities such as fonts, copy & paste, styles, layers etc.

On opening a new drawing, IRONCAD will present the View tab, click on the 'Standard' icon.
The Standard Views Creation dialog box appears.

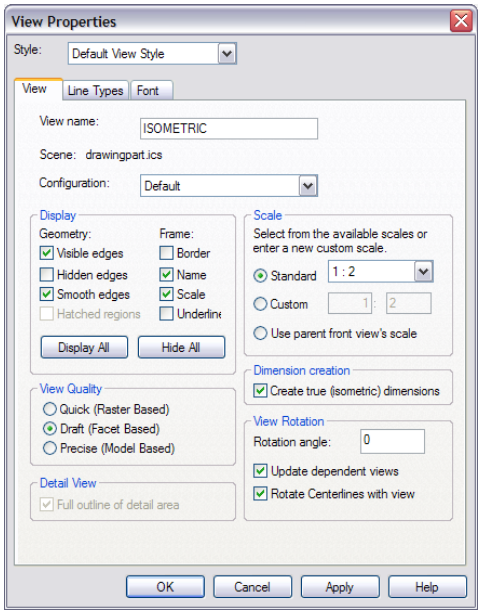


If the drawingpart.ics part isn't open choose the Browse button and browse and select the file ***drawingpart.ics***. Using the image above as a reference, click on the arrows to orientate the scene in the model window as shown. The model window presents the Front view of the model on the drawing, select the views Top, Front and TFR, as shown and then click OK. IRONCAD will display the selected views and left-click to place the views on the drawing sheet.

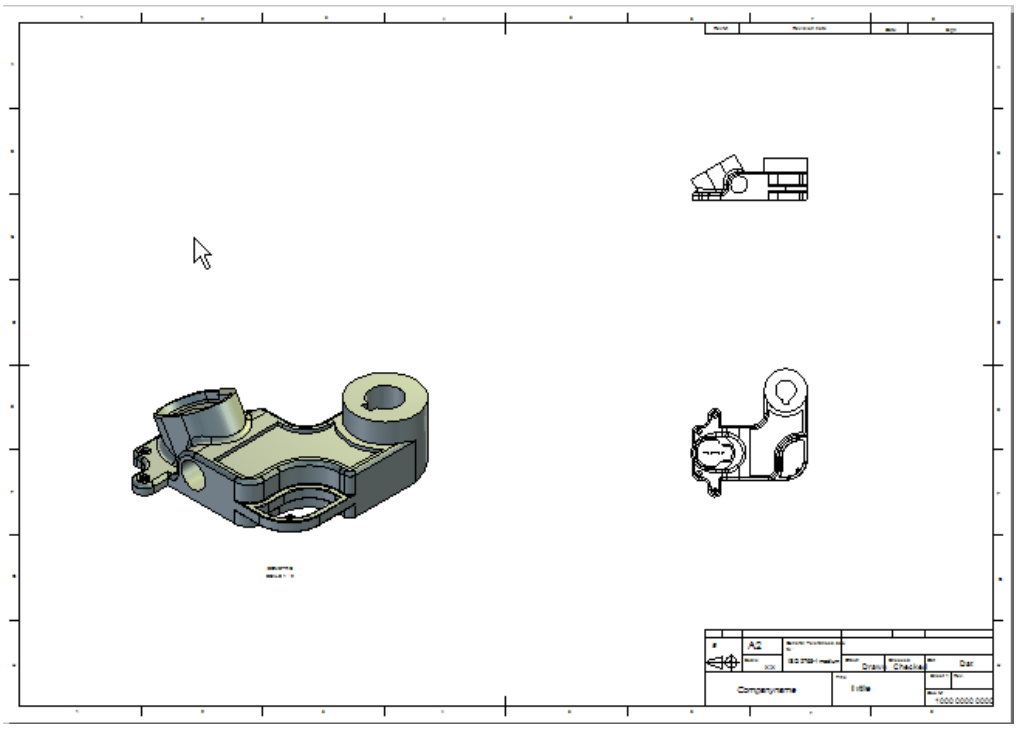


6.1.2. Changing the View Scale and Rendering options

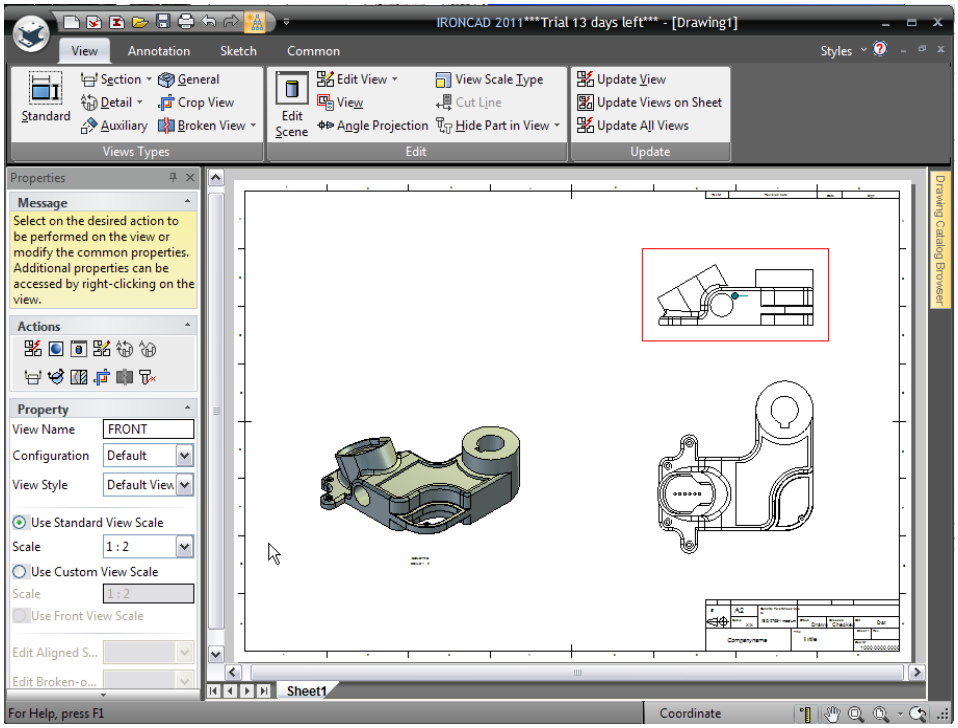
Right-click on the isometric view (the T.F.R. view) and choose Properties, and then change the scale to 1:2 by selecting the Standard option and use the drop down to select the scale. Also, change the view name to ISOMETRIC and activate the check boxes Name and Scale and select OK.



The name and scale should now appear below the view. Right-click on the isometric view and choose Shaded Rendering**.** If Shaded Rendering is not available for selection, check the mode used for creating the view by selecting 'Properties' and then in the View Quality section set the option for 'Quick (Raster Based)' or Draft (Facet Based).



Left click on the front view. IRONCAD will change the Property Panel to the properties of the view, select the Use Standard View Scale radio button and set the scale to 1:2. IRONCAD will respond by changing any other views created at the same time as the front view, not already altered as above, to the same scale. The screen and drawing should now look like the following image.



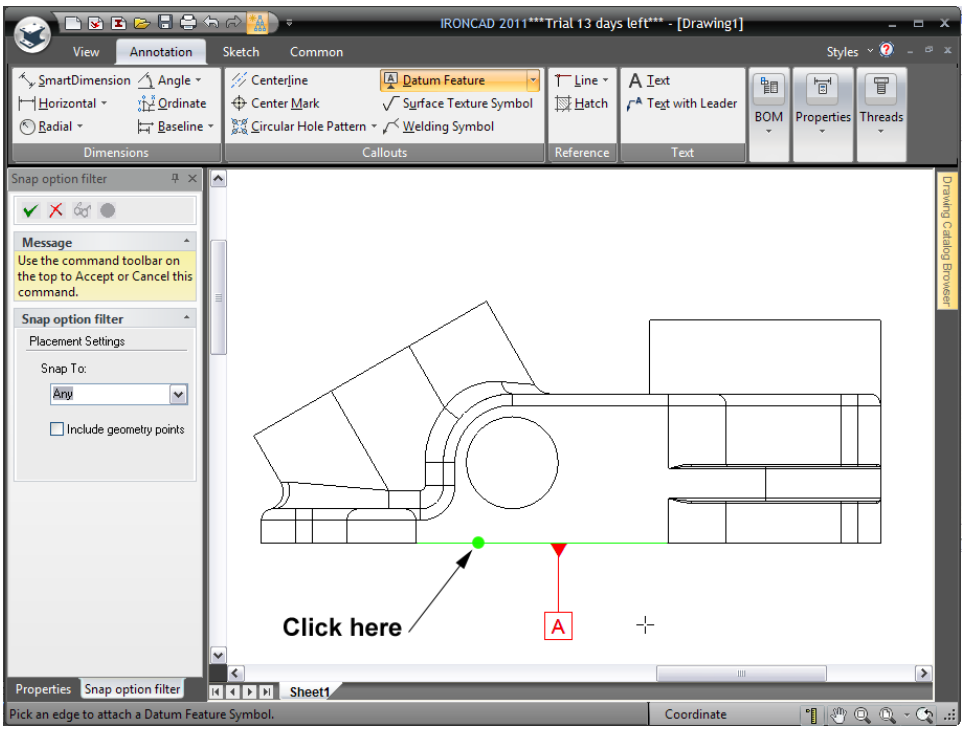
If the Top View did not change to match the Front View, select it and look at the properties in the Property Panel. The view scale option should be set to Use Front View Scale. If it isn't selected, select it.

6.1.3. Moving Views

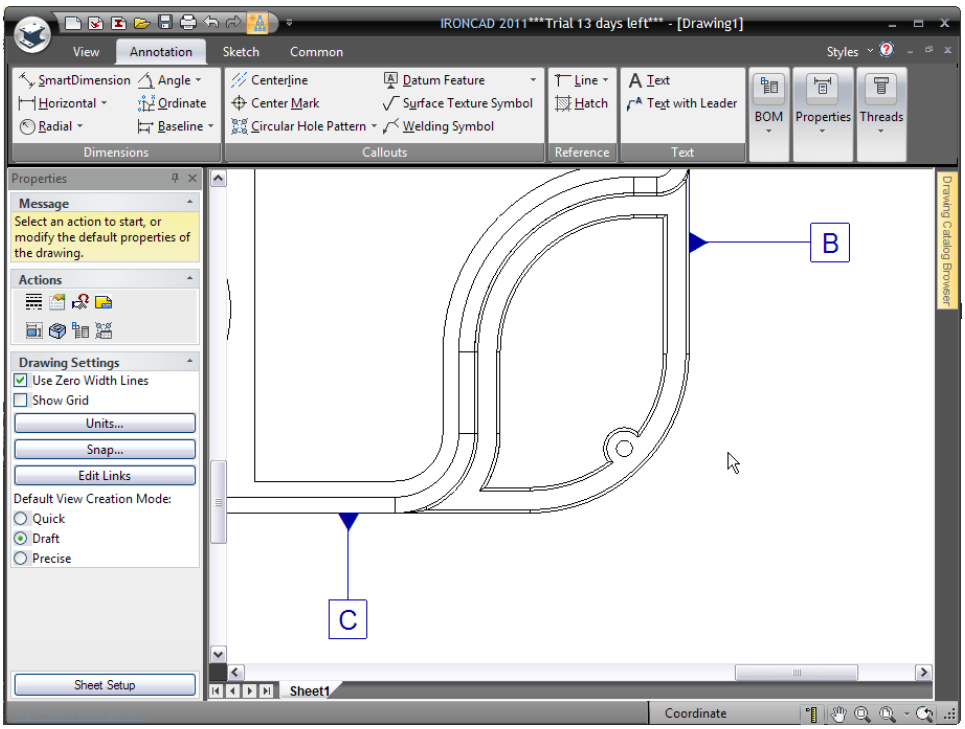
Select the Top view, the red Frame Border will be displayed.
Moving the cursor over to the Frame Border will change the cursor to a four way arrow. Click with the left mouse and drag the view, noting the automatic constrained alignment with the front (or "parent") view.

6.1.4. Creating a Datum Reference Frame

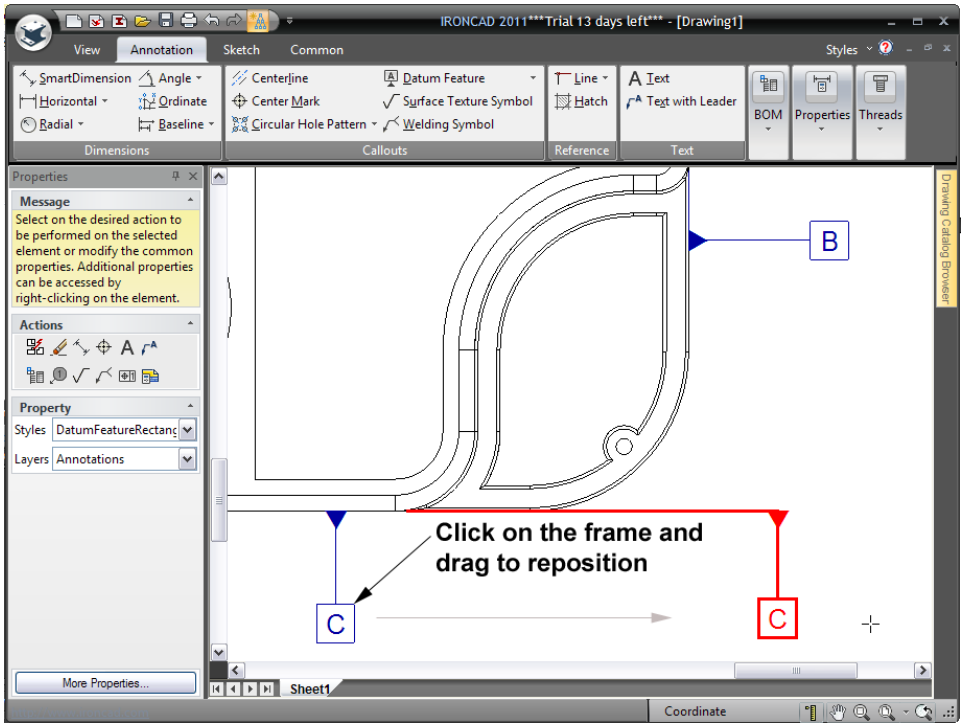
Zoom in on the Front view and select the Annotation tab on the Windows Ribbon bar. In the Callouts section, click on the down arrow by Feature Control Frame and select the 'Datum Feature' tool. Select the bottom edge of the part, and drag the label to locate it as shown.



After clicking to place the label, choose OK to accept the default "A" value (the primary datum). Zoom into the Top view and using the same technique, add the secondary (B) and tertiary (C) datum's, as shown.

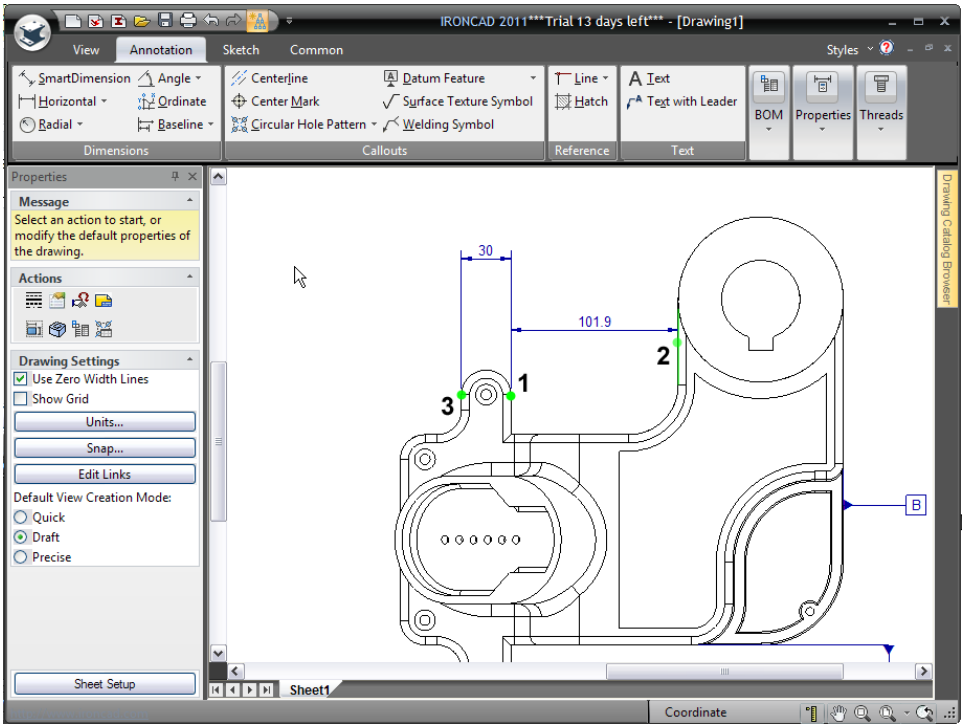


To reposition a datum frame, select it and drag it to a new position. Right-clicking on the datum frame opens a pop-up menu giving access to the properties providing the tools to change its form, its identity etc.



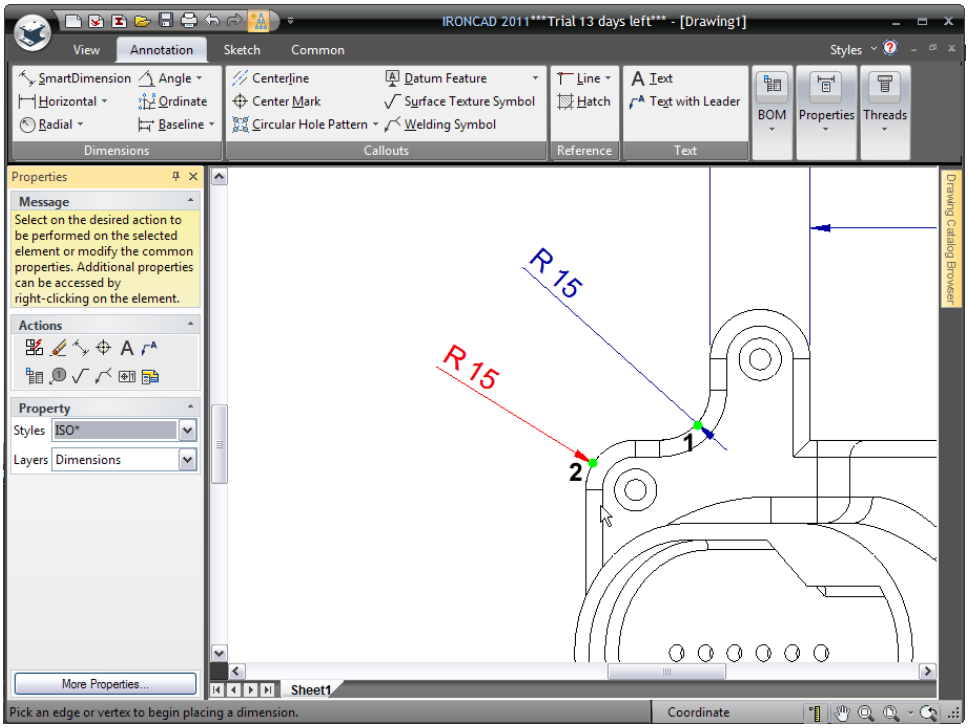
6.1.5. Adding SmartDimension's

From the Dimensions section of the same Annotation tab, select the SmartDimension tool and add the point-to-edge SmartDimension shown, by clicking point 1 indicated and the edge (2) in that order shown.

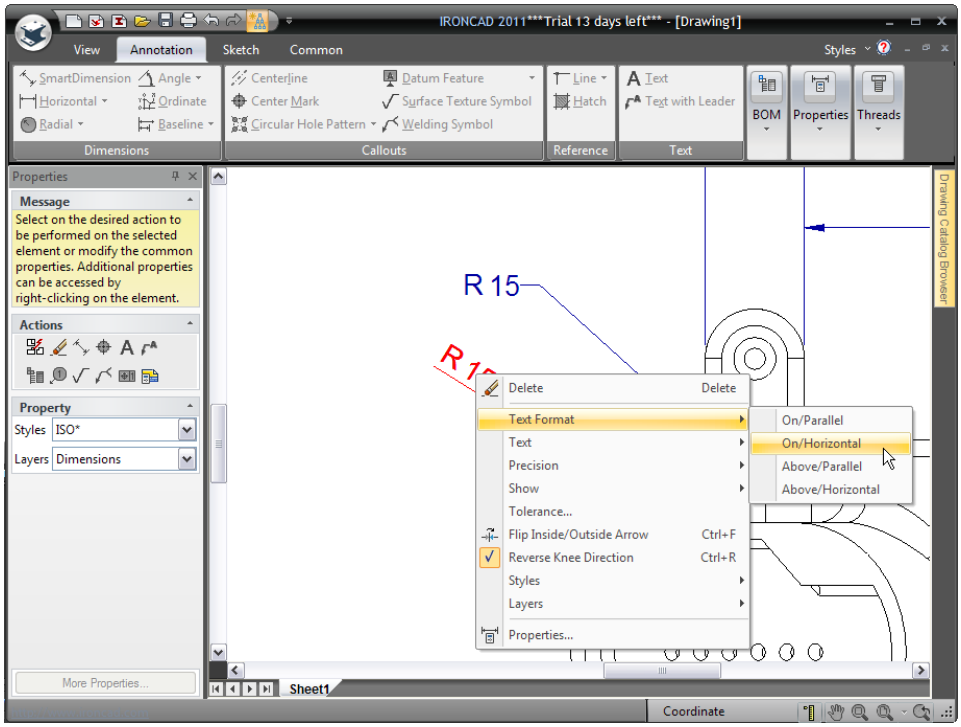


Now add the point-to-point SmartDimension shown by clicking point 1 again followed by point 3. Note that the dimension measures the perpendicular distance from the point to the line.

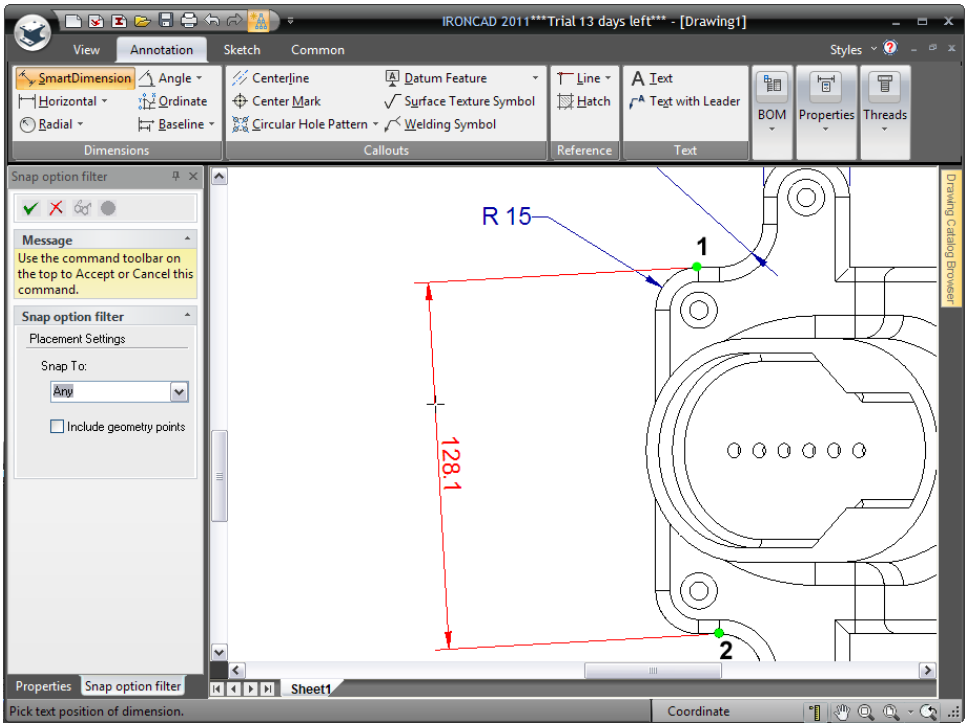
Use the SmartDimension tool to add the two radial dimensions shown in the next image. When clicking on any circular arc (except for its midpoint) with the SmartDimension tool, a radial dimension is automatically inferred.



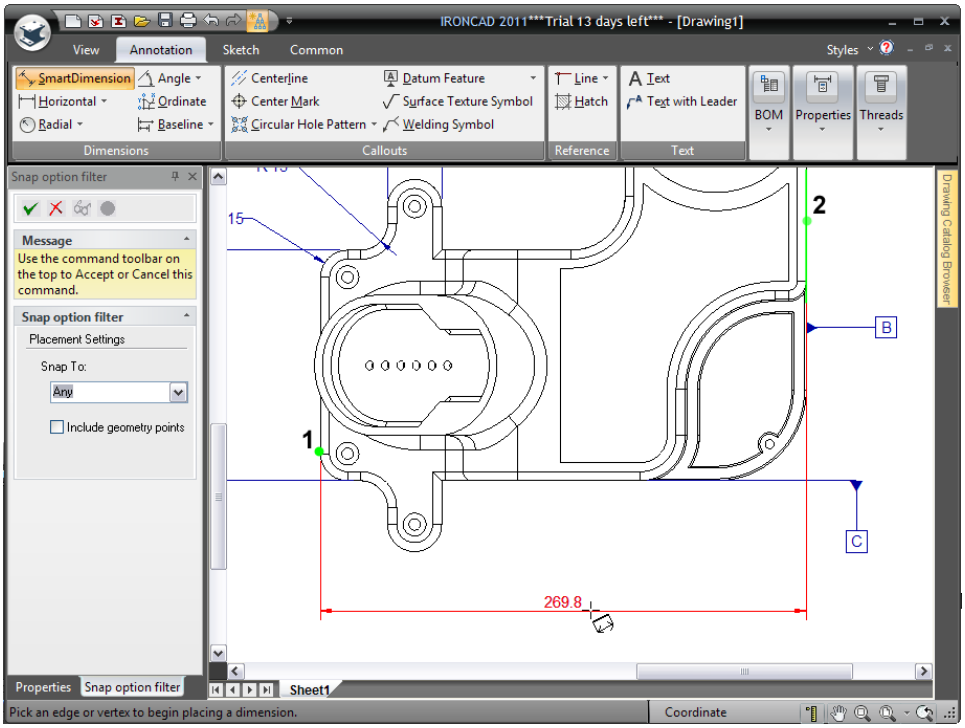
To change the orientation, the number of decimal places, the direction of the arrow etc., right-click on the dimension and set the appropriate option i.e. setting the dimension to be horizontal:



Select the SmartDimension tool and add the point-to-point SmartDimension shown by clicking the two points indicated, in the order shown. IRONCAD will draw a dimension perpendicular to the two points indicated. While the dimension is still active (or selected), pressing the TAB key toggles to a horizontal or vertical measurement (clicking the right hand mouse button also toggles the dimension orientation).

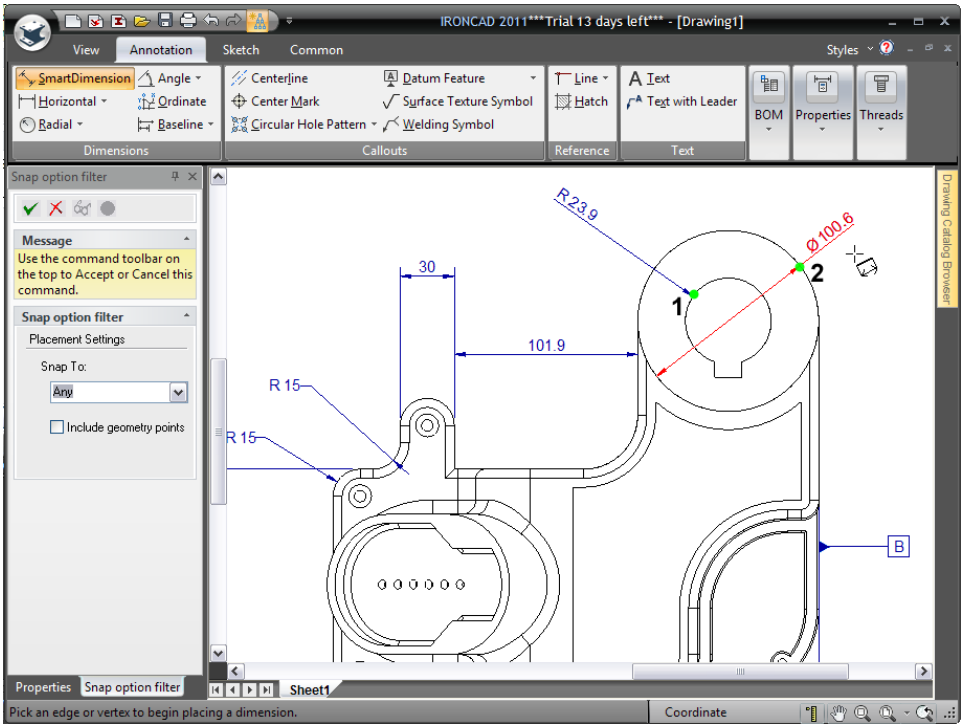


Next using the SmartDimension tool, add the point-to-line SmartDimension shown, by clicking the two locations indicated, in the order shown in the following image.

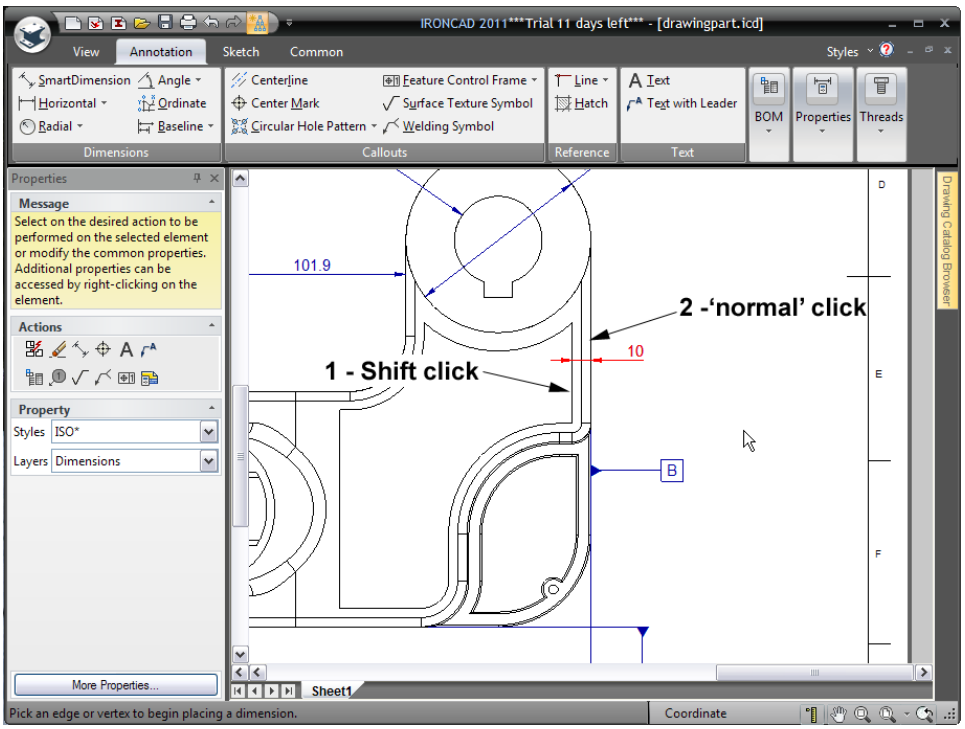


The SmartDimension tool, by default, works using single click modes i.e. touch an arc it creates an arc dimension, touch a line and a line dimension is created etc. In the image below use the SmartDimension tool to add the radial dimension (1). Again, note that when clicking on any circular arc (except for its midpoint) with the SmartDimension tool, a radius dimension is automatically inferred.

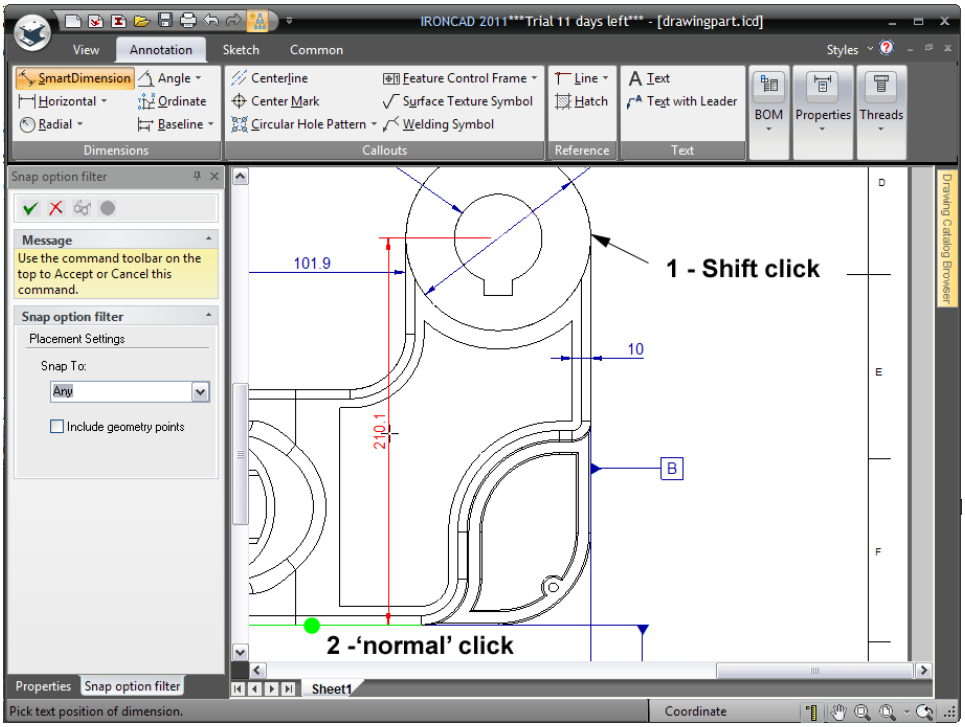
Use the SmartDimension tool to add the diameter dimension to the point (2). In a similar way to adding an arc dimension to a circular arc, clicking on a circle with the SmartDimension tool will generate a diameter dimension.



Pressing the shift key while selecting over-rides the default single click mode for example Use the SmartDimension tool to add the line-to-line dimension for the side rib shown below holding the Shift key for the first click.



In a similar way, to add a dimension from the boss center, hold the Shift key for the first click on the outside diameter and the for the second point click on the line (2). The Shift key will override the “single-click” behavior of the SmartDimension tool and use the center point of the circle as the point from which to measure, the second click (on the line) will orientate the dimension. After clicking the line, position the dimension to the right of Datum B.

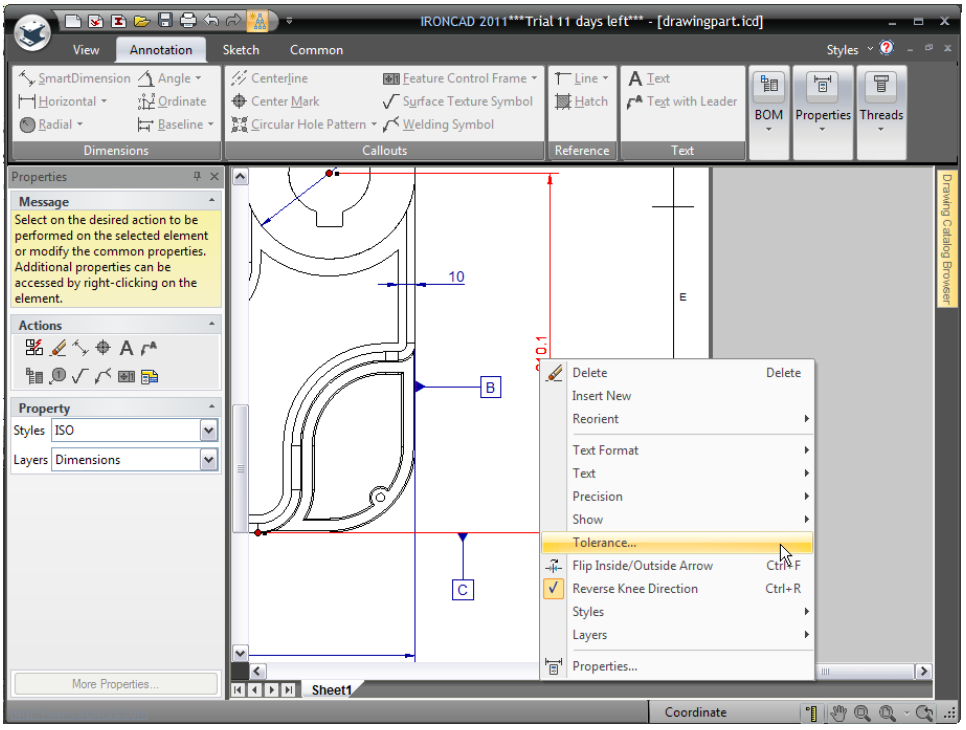


Click on the Apply and Exit check mark in the IRONCAD Property Panel to complete the dimensioning (or turn off the command).

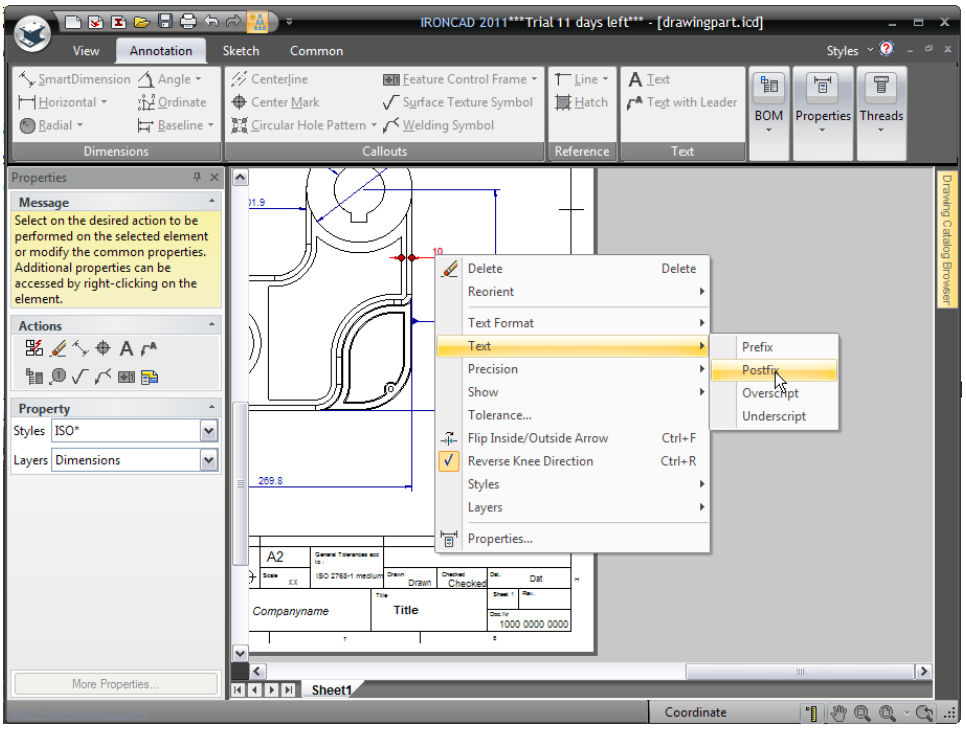
6.1.6. Modifying Individual Dimensions

Dimensions can be changed in many ways such as in the application of tolerances, adding a box to indicate the dimension is a ‘Basic’ dimension, changing the number of decimal places etc.

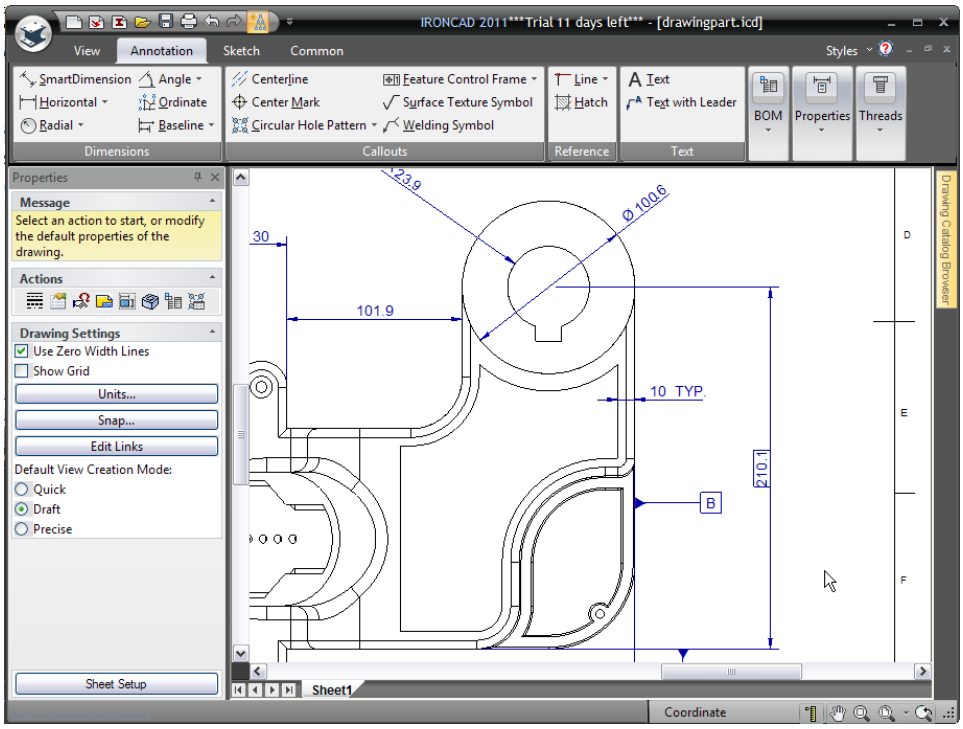
To change the last dimension created (210.1) to a Basic dimension, position the mouse over the dimension and right-click the mouse. From the pop-up menu select ‘Tolerance…’ and in the dimension properties box select ‘Basic dimension’ and then click OK.



Similarly, to add the postfix 'TYP.' to the rib line-to-line dimension (10) follow the same procedure, this time select Text followed by Postfix. IRONCAD will directly open the Text tab of the Dimension Properties dialog with the cursor in the Postfix entry. Type TYP and hit OK.

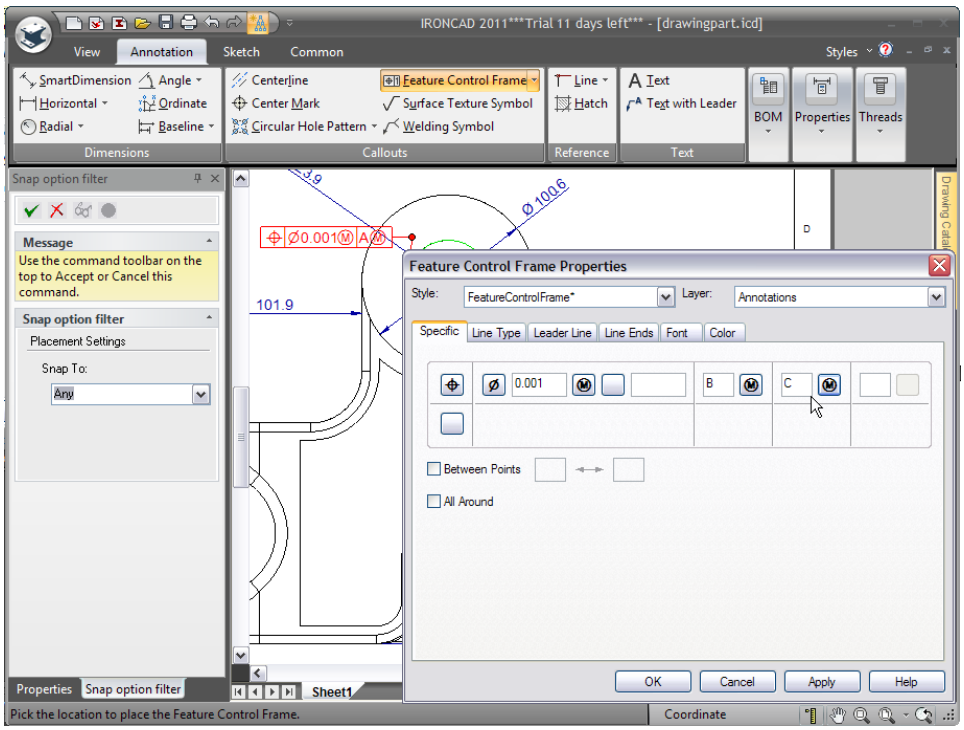


The drawing should resemble the following image after the last two dimension edits.

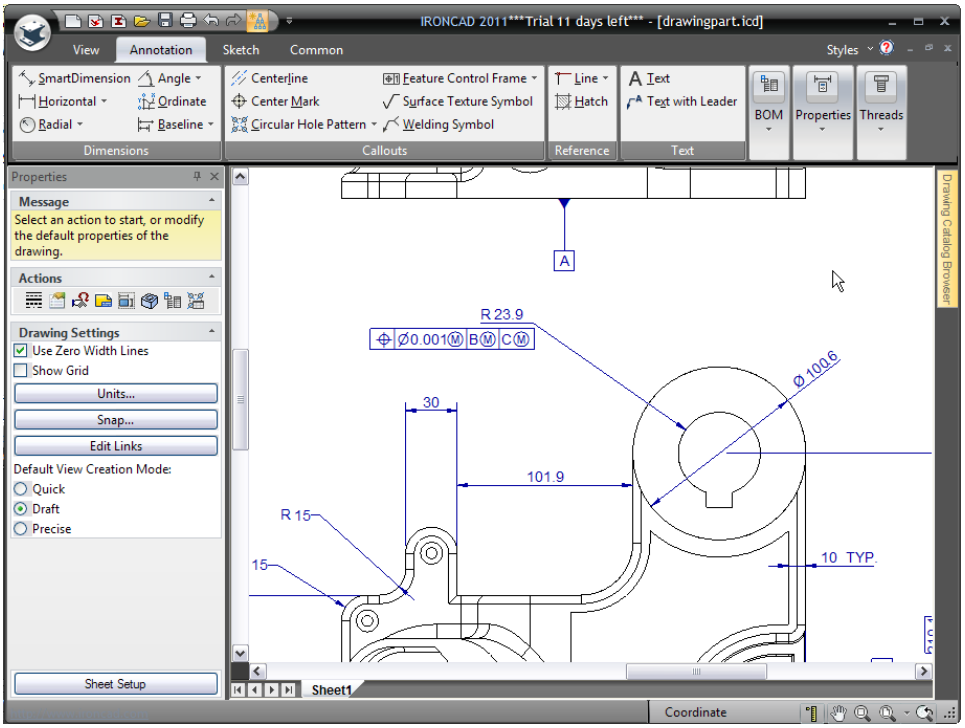


6.1.7. Adding a Feature Control Frame (Geometric Tolerance)

A Feature Control Frame (or Geometric Tolerance) is added in a similar way to the Datum Reference Frame created earlier.
From the Callouts section of the Windows Ribbon bar select the Feature Control Frame option (or click on the down arrow by the Datum Feature option if it is shown). Click on the inner circle, and then click to place the frame. IRONCAD will display the Feature Control Frame Properties dialog box where each attribute of the control can be entered. After entering the desired attributes, click OK followed by Apply and Exit on the IRONCAD Property Panel.



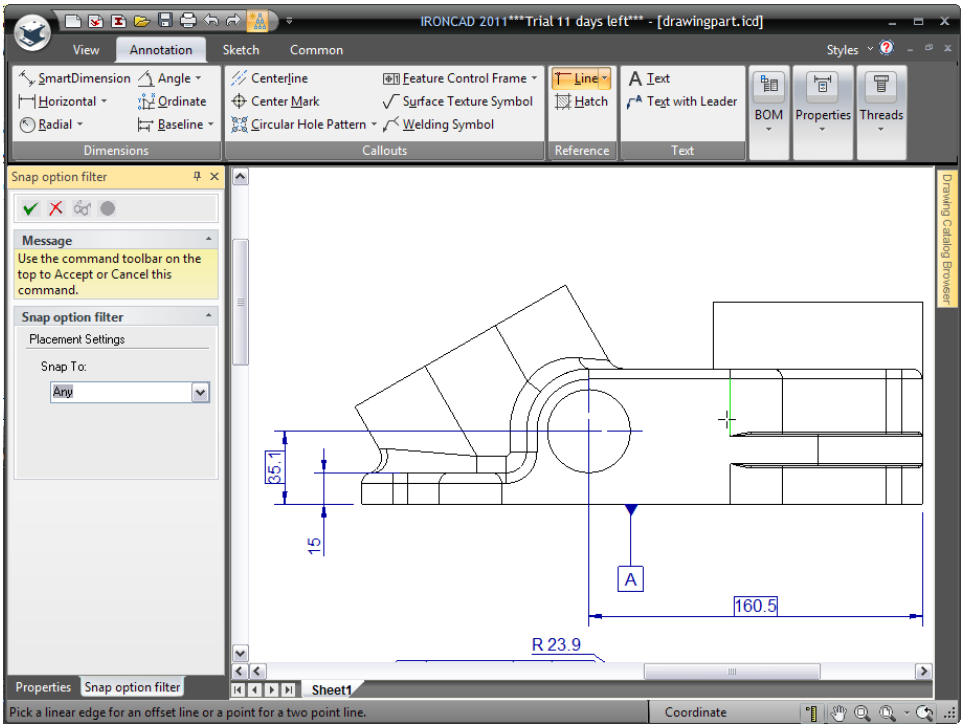
Reposition the Feature Control Frame as required. In the event the Feature Control Frame is to be appended to a dimension i.e. the leader line is not required, position the mouse over the Feature Control Frame and press the right mouse button. From the resulting pop-up menu, deselect Show Leader. After right-clicking on the radial dimension defining the hole and setting its 'Text Format' to horizontal and positioning the Feature Control Frame, the drawing should now resemble the image shown next.



6.1.8. Creating center lines

Zoom in on the Front View and add a centerline to the hole shown below by clicking the ‘Center Mark’ option from the Callouts section of the Windows Ribbon bar, then click on the circle, as shown. IRONCAD draws four extension lines which can be sized by selecting the centerline and dragging the red handles at the end of each line segment. Dimensions can be applied to these lines.

Size the center lines and add the dimensions, as shown.



For views displaying cylindrical features looked on from the side (and not the top) IRONCAD uses the Centerline function. The front view contains a number of cylindrical surfaces shown from the side. From the Callouts section, select the 'Centerline' function and trace the cursor over the front view. IRONCAD will highlight the cylindrical surfaces as the cursor passes across them. Clicking the left mouse button will cause a center line to be drawn. As with center marks, the length of center lines can be extended by dragging the red end handle.

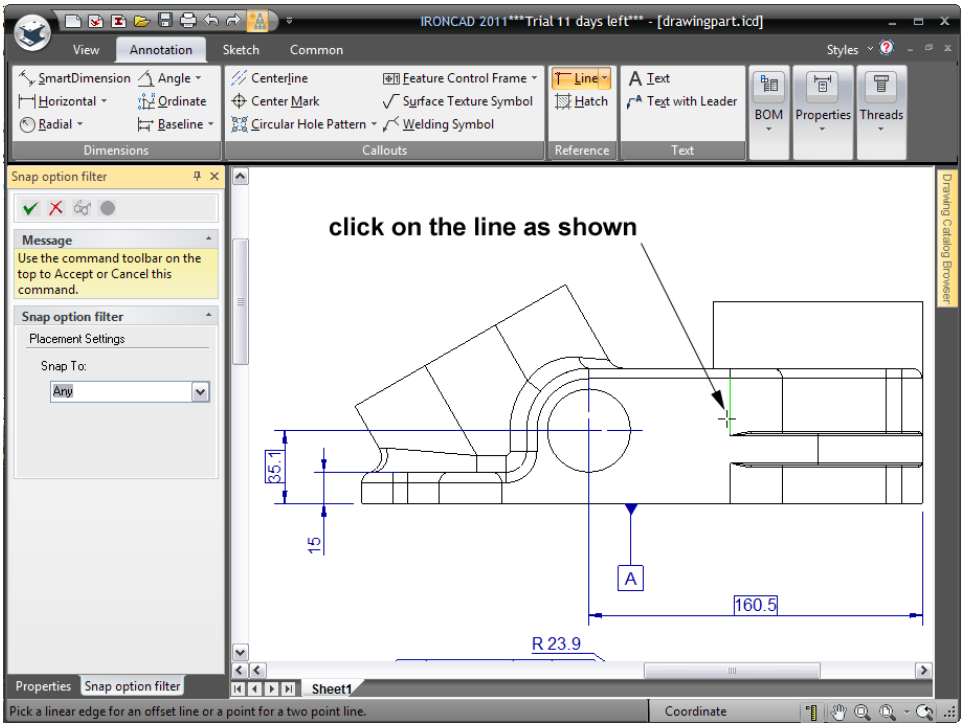
As with most functions IRONCAD remains in the selected mode (in this case Centerline) until the 'Apply and Exit' command on the IRONCAD Property Panel is selected or ESC is pressed.

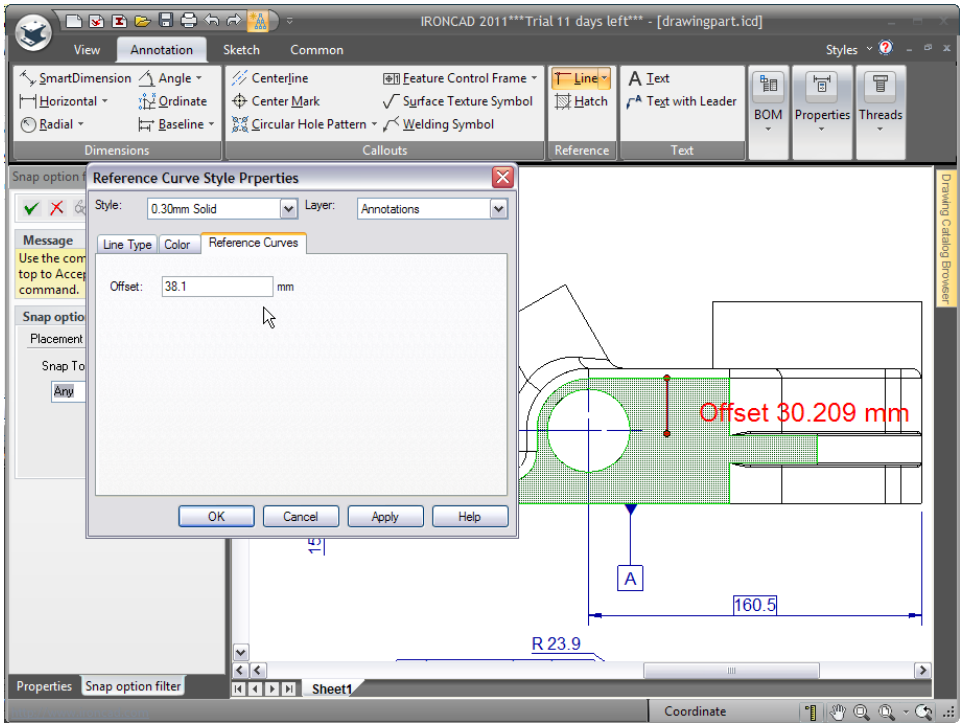
6.1.9. Creating Reference Curves

A reference curve is a line or circle which can be placed arbitrarily on the model and is useful for showing say a limit to a specific surface finish or treatment. Reference geometry is associative to the part and can also be used in dimensioning.

The reference curve function is located in the Reference section of the Annotation tab. By clicking on the down arrow by the option 'Line' will reveal the types of reference curve function available. To add a reference line to the model select the 'Line' button, clicking the line, as shown, dragging the line to

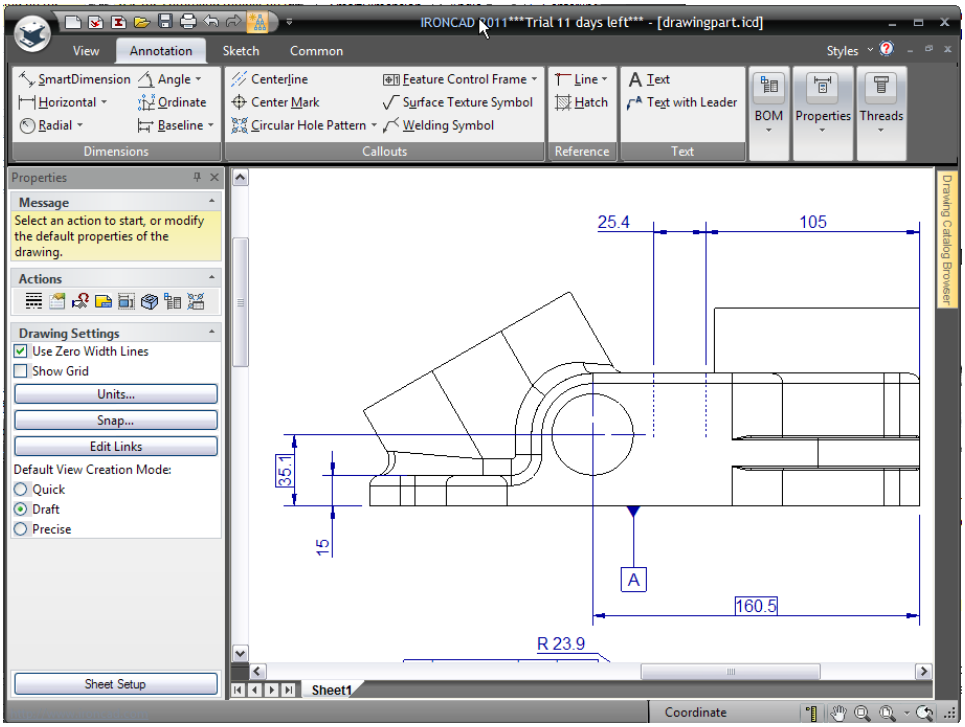
the left, and then left clicking again to place it visually, or right-clicking the position to place the reference line precisely – in this case we want the offset to be 38.1.





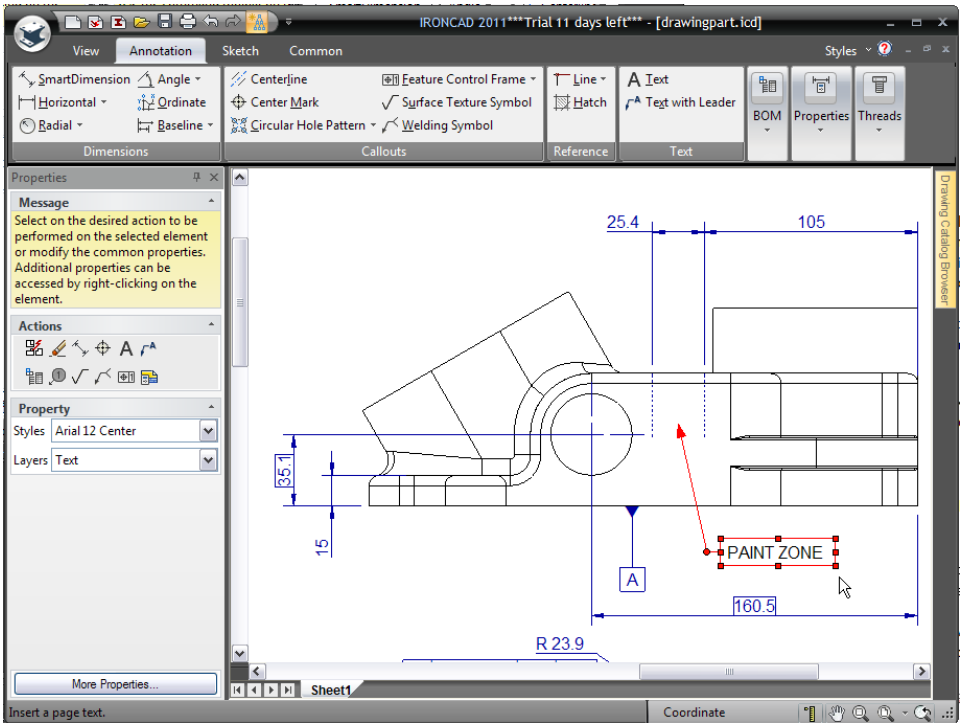
To change the format of the line i.e. from a solid line to a dotted line, either select a different line type prior to clicking OK when setting the offset prior to selecting Accept and Exit, select the Properties tab at the bottom of the IRONCAD Property Panel and in the Styles box change the line type to Dot or right-click on the reference line after completing the command and select Properties.
It is worth noting the layer can also be changed in this action.

Repeat the steps above to add another Reference Line, but with an offset of 12.7. Now add a SmartDimension's to the Reference Lines, as shown below:



Right-clicking on a dimension followed by selecting 'Flip Inside/Outside Arrow' will change the arrow positions. Furthermore when tidying up the drawing, when a dimension is selected and being repositioned, positioning the mouse pointer over a dimension arrow will change the pointer to a grabbing hand. With the 'grabbing hand' cursor dragging the highlighted dimension (the cursor changes to a small cross) on to another dimension leader arrow will snap the active dimension arrow to the indicated dimension arrow.

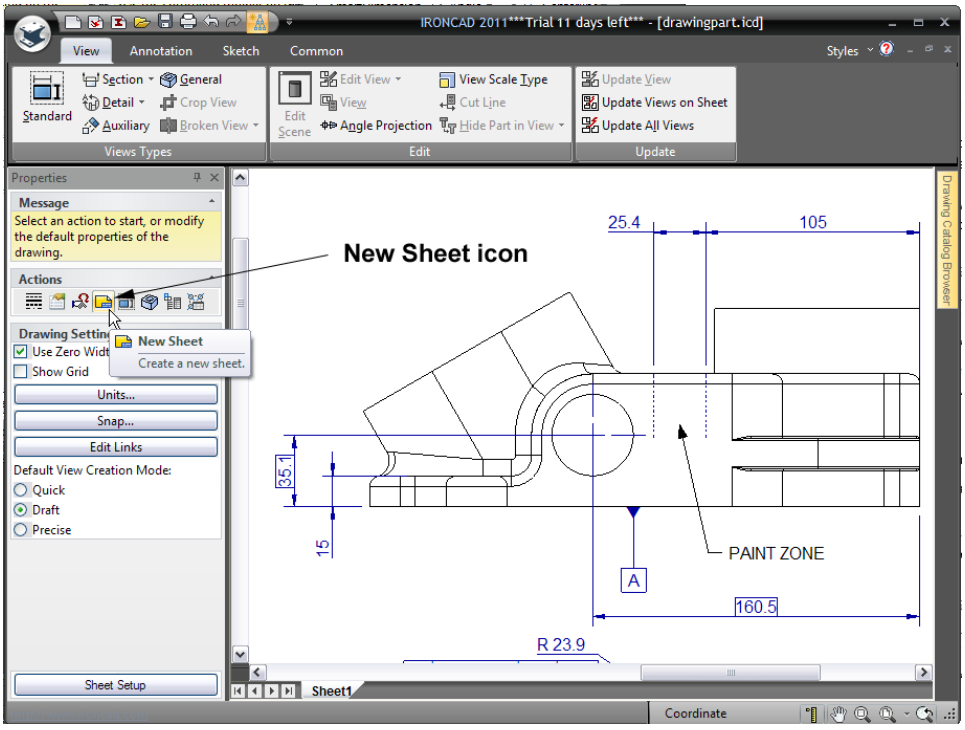
Next add a textbox, as shown, by clicking the' Text with Leader' tool, clicking on the part, dragging to the location shown, and then entering PAINT ZONE as the text.



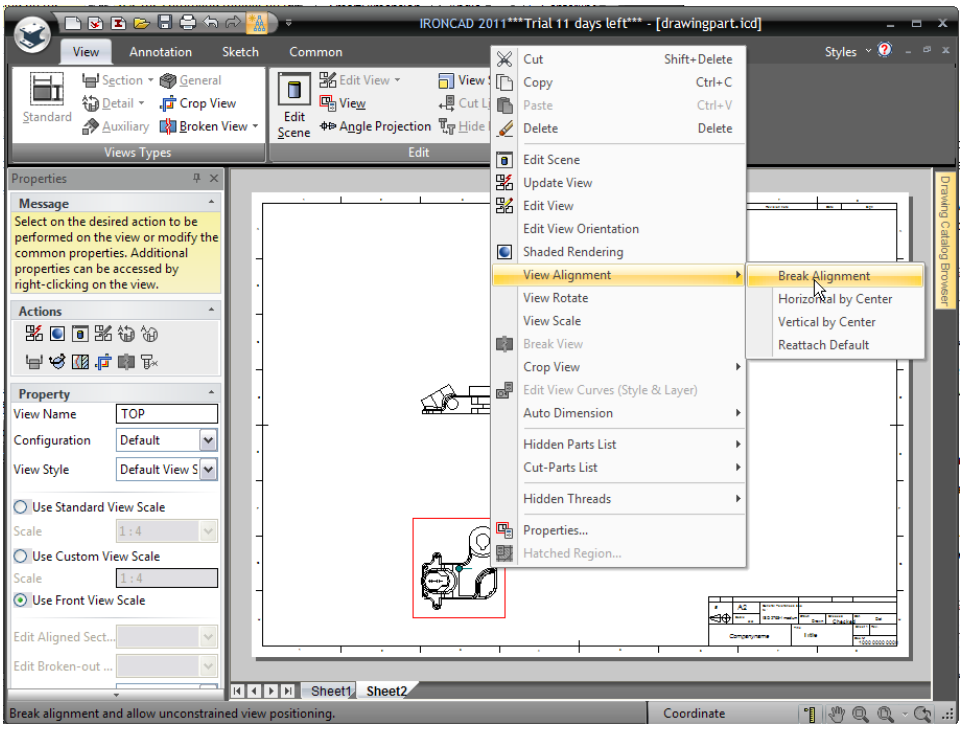
6.1.10. Multi-Sheet Drawings and Supplementary Views

As mentioned earlier drawings can comprise multiple sheets within IRONCAD all pointing back to the originating model. To create a new sheet, open the View tab. IRONCAD will not only change the Windows Ribbon bar it will refresh the Property Panel also changing the Actions panel to the most appropriate set of functions supplementing those found on the Windows Ribbon bar.

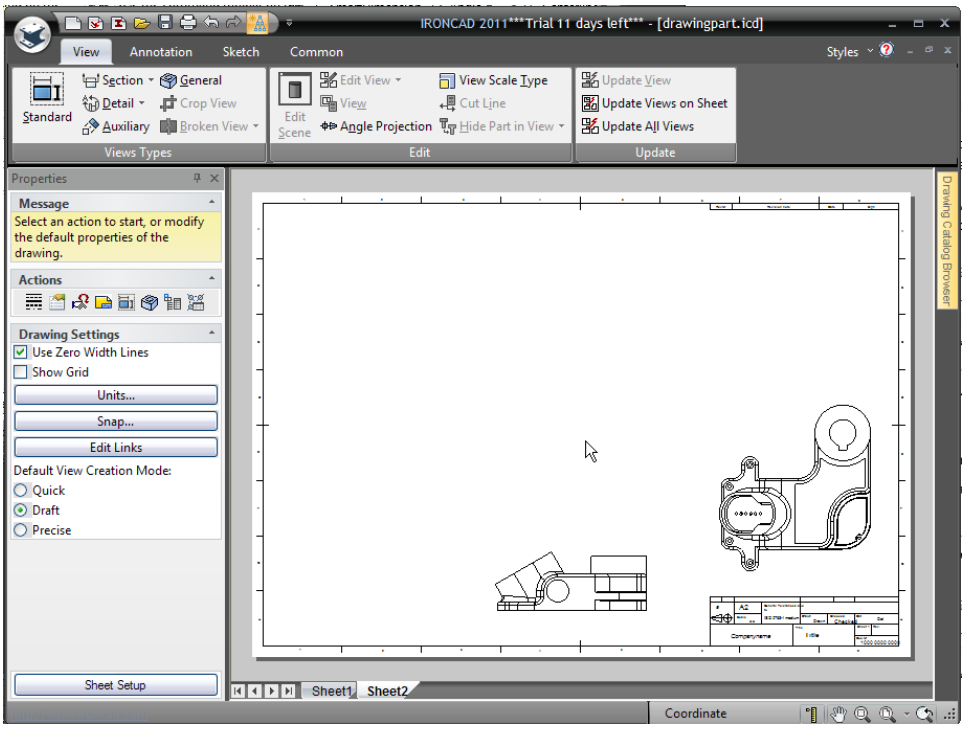
To create a new sheet, select the New Sheet icon highlighted next. IRONCAD will display the New Sheet dialog box, from the ISO tab, select the template A2, and then choose OK.



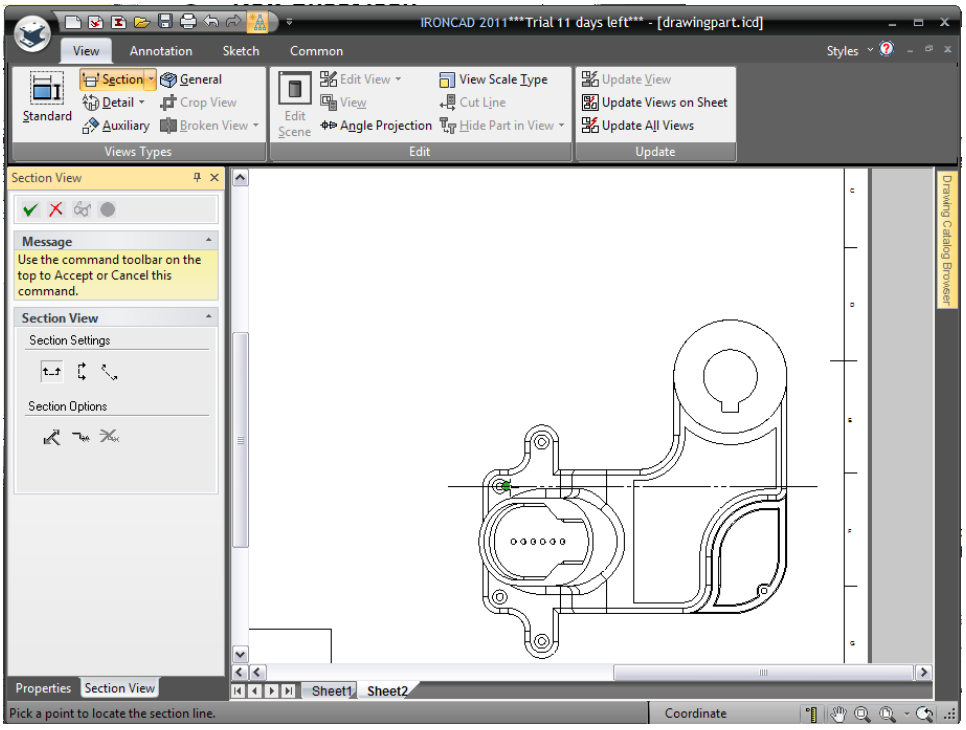
Now add two views to the new sheet by clicking the Standard View tool and choose the Top and Front view and then choose OK. The Top and Front views appear on the new sheet. Right-click on the top view and choose 'View Alignment>Break Alignment'.



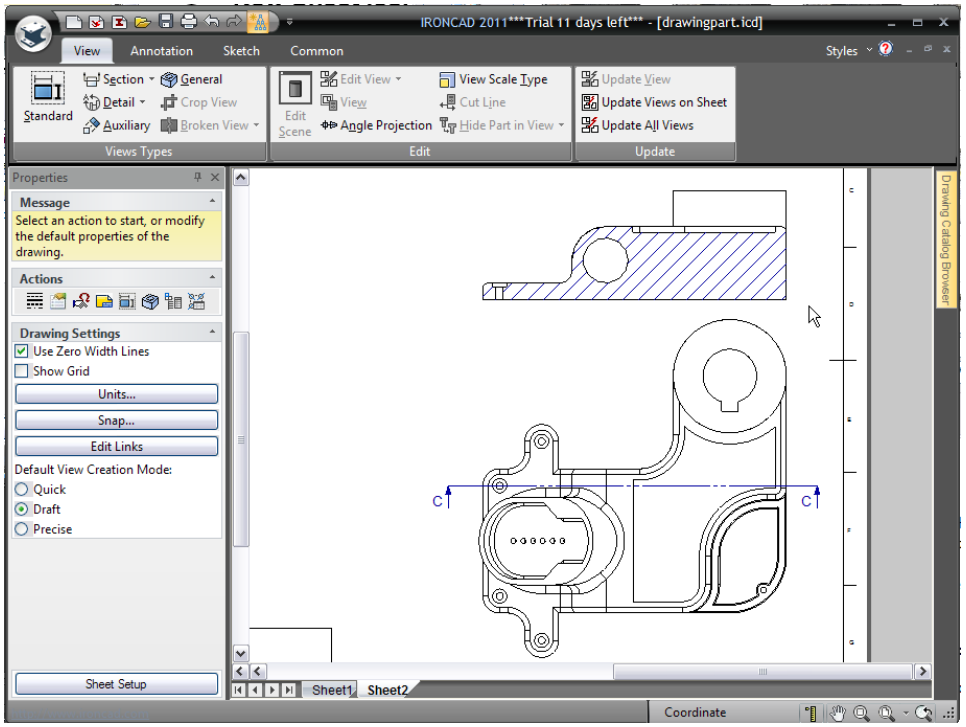
Manually rearrange the views, as shown. (To move a view, select it and then drag its border.) Set the scale of the views to 1:2.



Zoom in on the Top view. Select the Section tool (found in the View Types section of the Windows Ribbon bar) and then click the location shown to place the section line. Note the section line snaps to cylindrical feature centers.



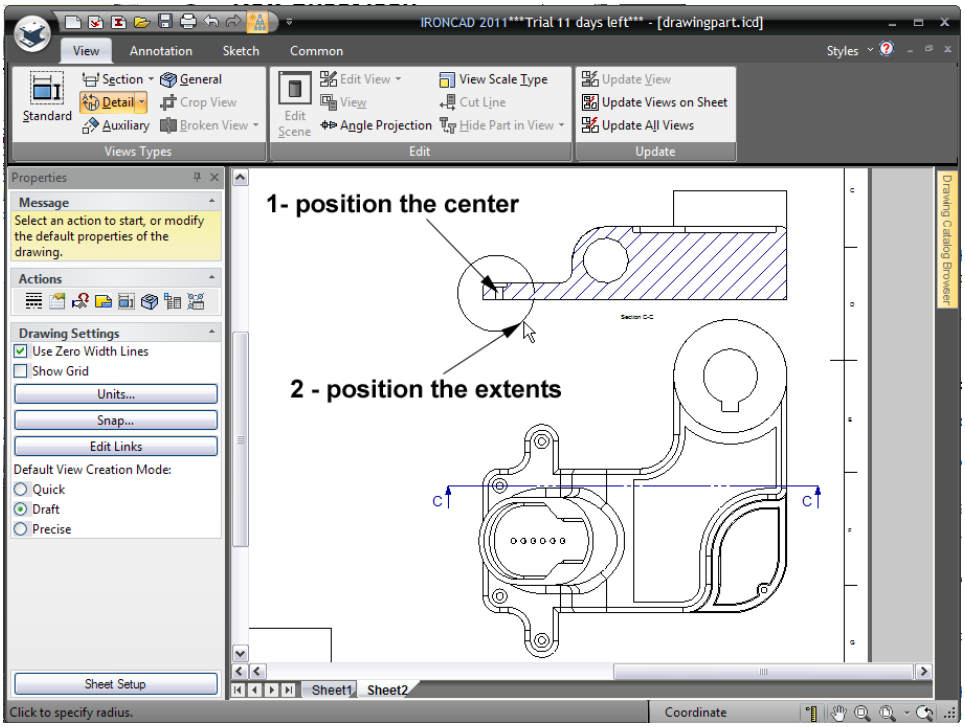
Click on the green check mark Accept and Exit button on the Property Panel, and then position the new view, as shown. Note that the view is automatically aligned with the section line direction.



Click on the section view, IRONCAD will change the Property Panel to the primary properties of the section view, check the Name option. At this point it is worth exploring the hatch options by clicking on the Hatch Properties button at the bottom of the Property Panel.

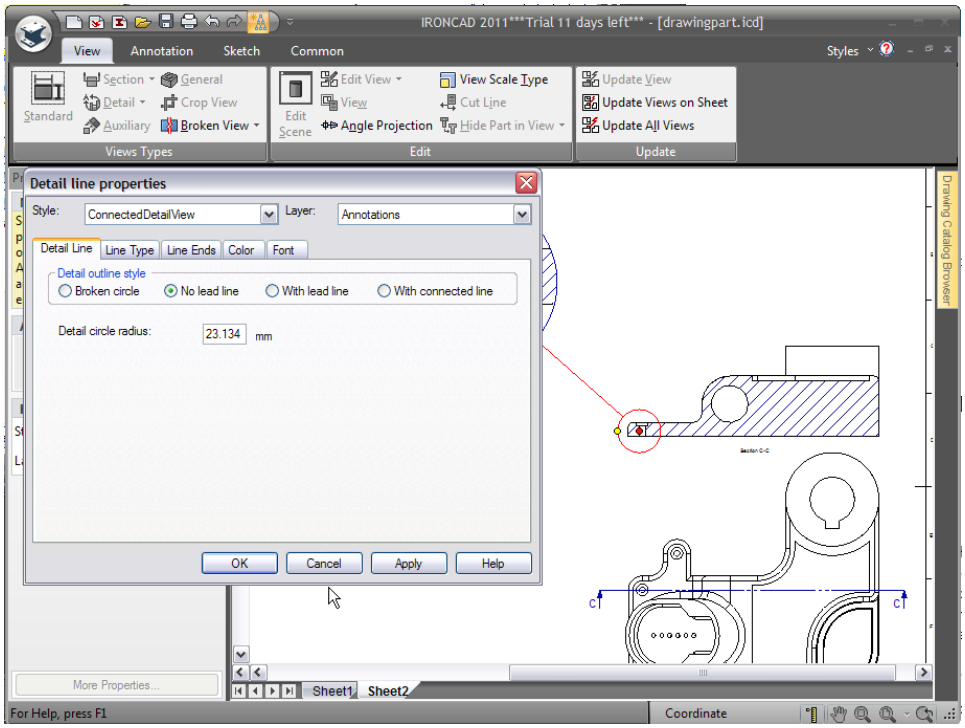
6.1.11. Adding a Detail View

To add a detail view, select Detail from the View Types section of the Windows Ribbon bar and then click the location shown to place the center point of the detail area. Drag the mouse to the approximate diameter shown and then click to complete the detail area.



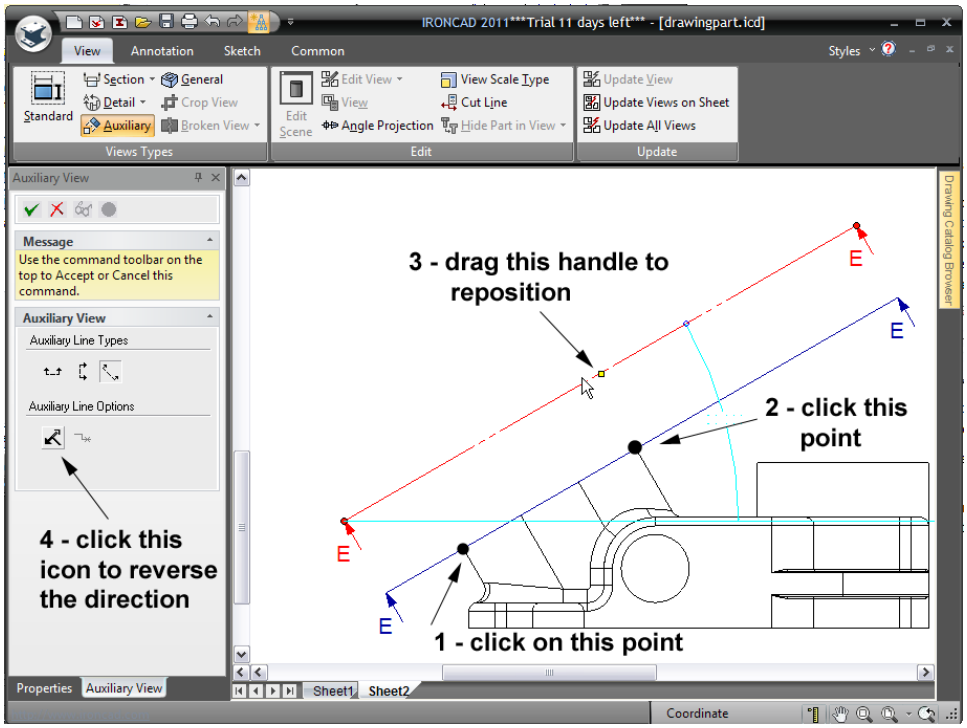
Click on the detail view and in the Property Panel turn ON the Name and Scale check boxes. It is worth reviewing the options presented by the Property Panel.

Right-clicking on the actual detail area boundary will reveal further options for the display of the detail view such as if the views are connected with a leader line.



6.1.12. Creating an Auxiliary View

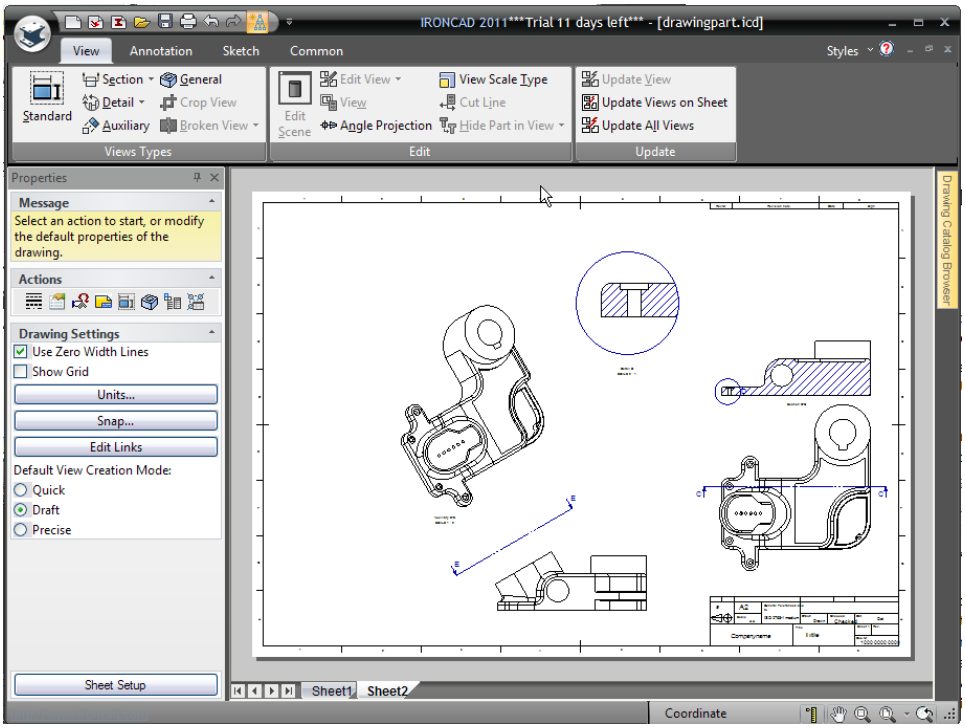
Zoom in on the Front view, then select the 'Auxiliary View' tool and referring to the next image click the two locations shown, in the order indicated, to place an auxiliary line. Next drag the view line to an appropriate position using its drag handle, and finally click the 'Flip Direction' tool on the Auxiliary Line Property Panel to direct the auxiliary line arrows towards the part (if required). Click on Accept and Exit to create the auxiliary view.



Zoom out and click to place the view, as shown. Note that the view is automatically aligned with the auxiliary view direction.

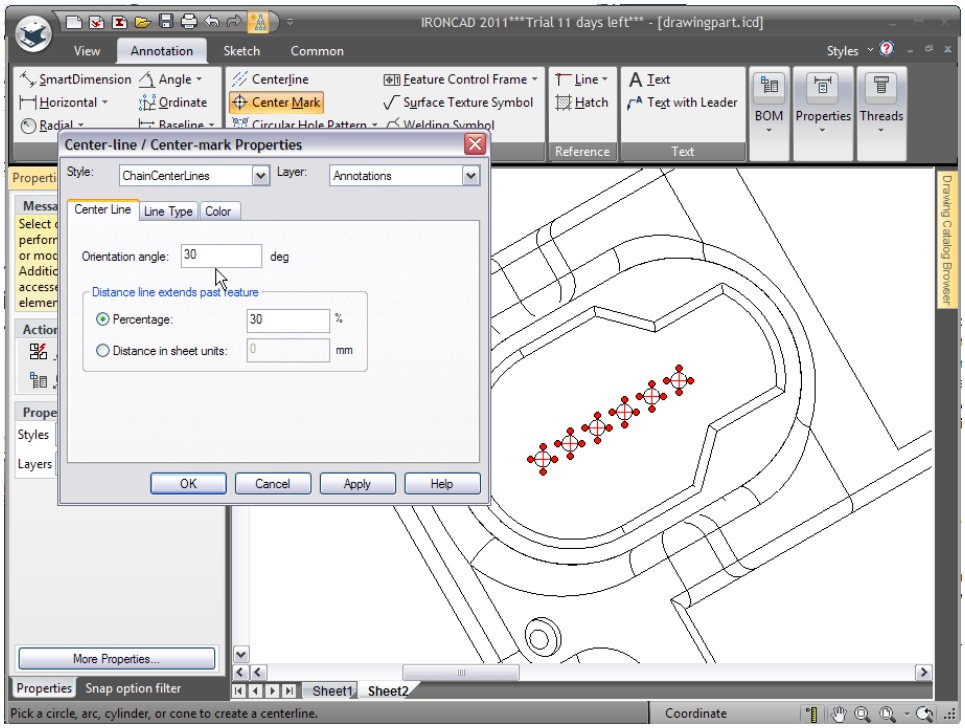
Select the auxiliary line and size it, as shown, with the red endpoint handles and click Update Views on Sheet – found in the update panel of the Windows Ribbon bar.

As with previous views, click on the Auxiliary view and turn on the name. The sheet should now resemble that shown in the following image.



6.1.13. Orientating center marks

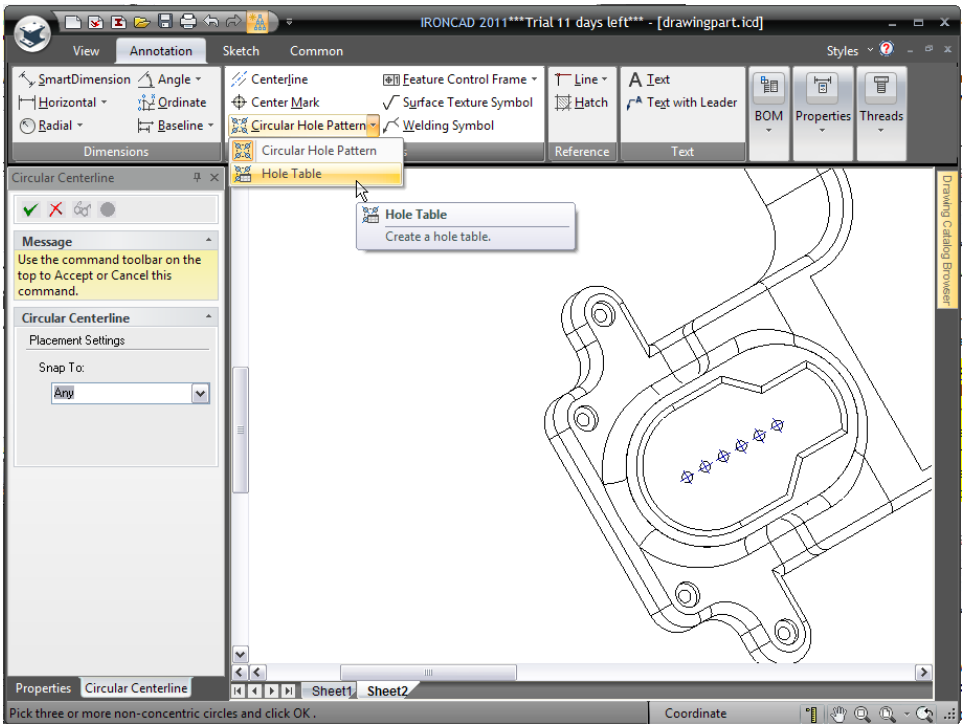
Zoom in on the Auxiliary view to create center marks on the small holes. Before hitting the Accept and Exit command, select the properties tab on the bottom of the Property Panel, followed by More Properties and rotate the center marks 30 degrees by setting the Orientation angle to 30 degrees.



An alternative method is to right-click on a center mark and access this function from the Properties dialog box.

6.1.14. Creating a Hole Table

It is often useful to position holes using a table rather than discrete dimensions. In this example a Hole Table will be used to position the small holes previously used as an example of how to re-orientate center marks. From the Annotation tab, either select Hole Table from the drop down by Circular Hole Pattern or from the icon on the Actions panel shown next.

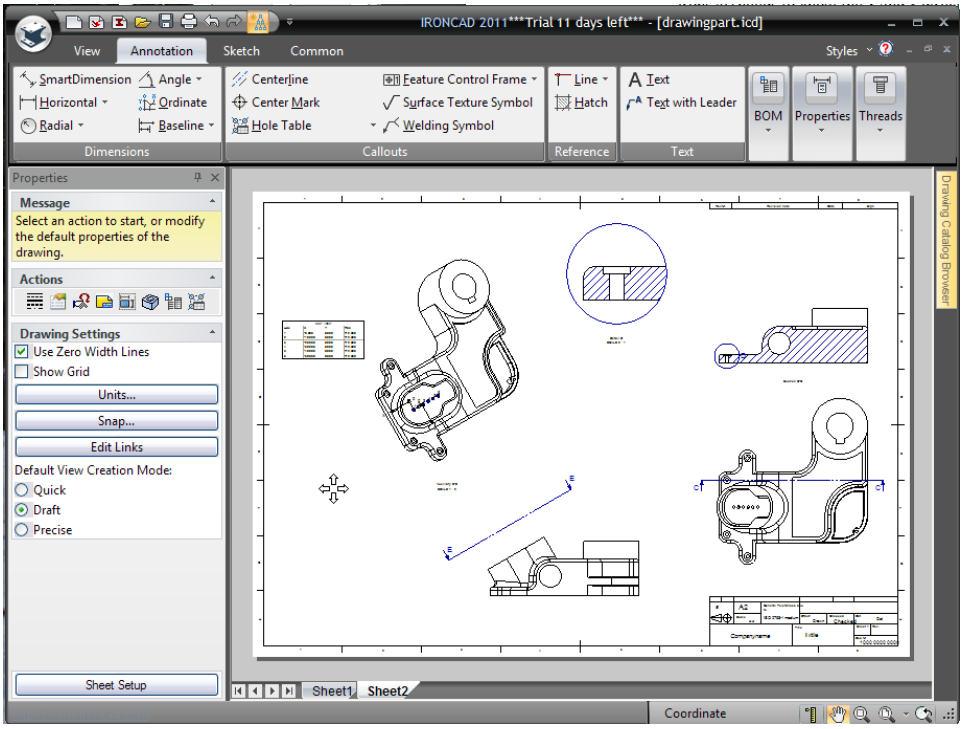


IRONCAD will produce an empty hole table, place the Hole Table anywhere on the sheet. The hole table can be repositioned later.

IRONCAD needs to know the X and Y direction for the hole table, click the centerlines of the first hole to define the X and Y axes for the Hole Table zooming in if necessary.

Now click each of the remaining holes to be included in the Hole Table, the cursor will highlight the hole before selecting it. After selecting the last hole hit the Esc key.

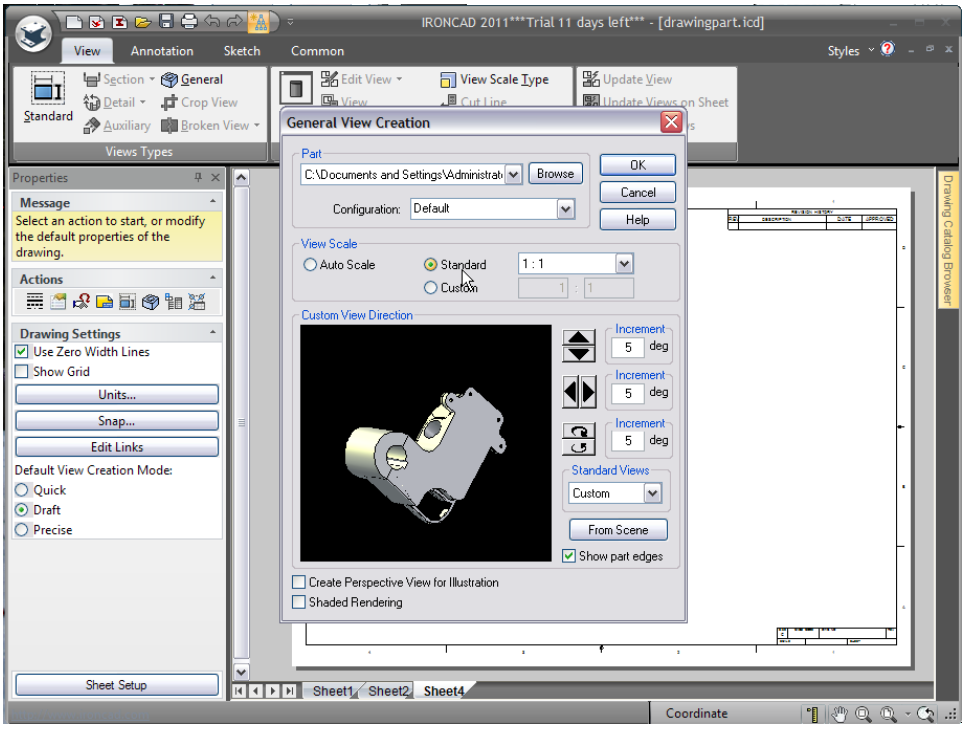
Zoom out and position the Hole Table in a more appropriate place. If the Hole Table axes point the wrong way, reverse them by right-clicking on the X-Y origin symbols and choosing the appropriate action. Additionally the hole labels can be repositioned for clarity by clicking on each label and dragging it to a clearer position Modify the display using the properties of the Hole Table so that the view resembles the next image.



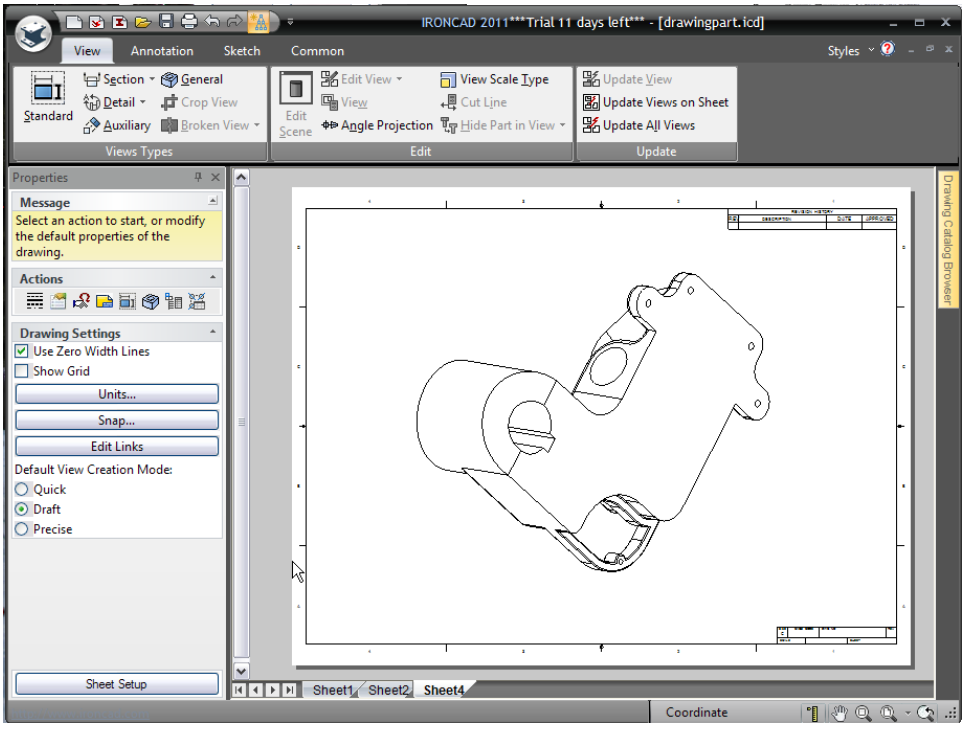
6.1.15. Using the Sheet Drawing tools

Add a new sheet to the drawing, however this time choose the template C Size Continuation from the ANSI (English) tab.

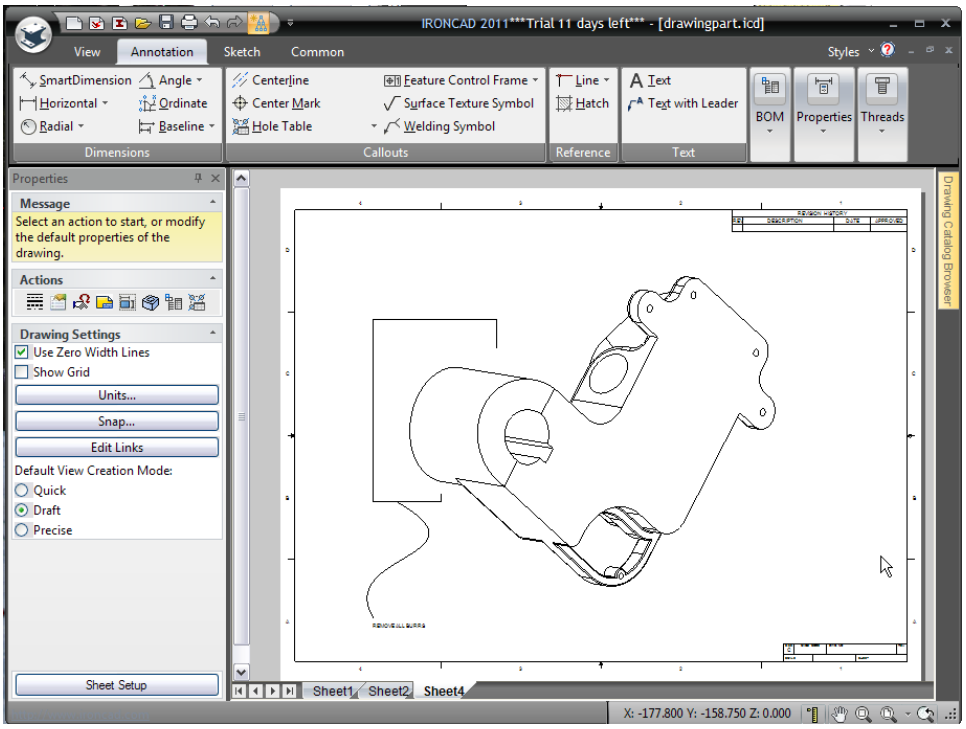
Click the General View tool in the View Types section of the View tab. Use the preview orientation controls to orient your view, as shown, and then click OK to create the view. It is worth noting the values used for the angle increments can be edited by clicking in the boxes and new values typing in.



Set the scale to 1:1 in the View Scale panel and then choose OK. The sheet should resemble the following image.



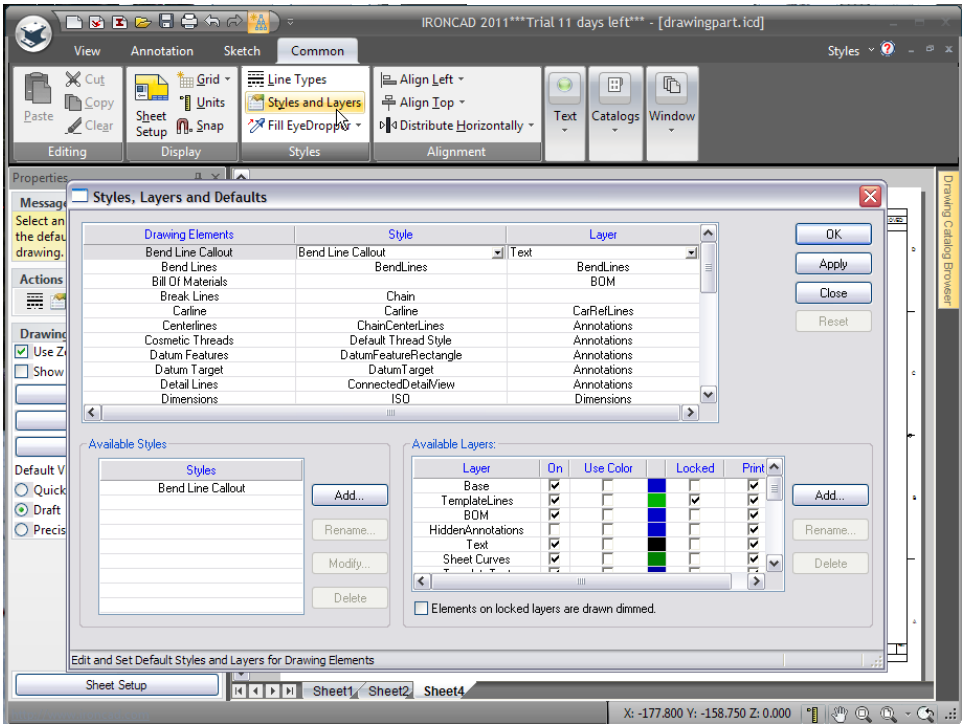
IRONCAD provides a number of 2D drawing tools that can be used to create 2D support geometry for annotations. The Sheet Drawing Tools are located on the Sketch tab of the Windows Ribbon bar. Use the Polyline, Bezier, and Text tool to create the annotation shown here. When adding text with the Text tool, click and drag to create a text box in the desired location and then insert the text.



6.1.16. Styles and Layers

There are a variety of options available for changing the appearance of a drawing, such as dimension styles and layers. For example, a dimension's style can be changed individually by right-clicking the dimension to change. The most common options such as the precision, tolerances, etc. are made immediately available on the resulting right-click menu, choosing Properties from this menu will give deeper access to the style, color layer etc. of the selected dimension. On the other hand, changing the styles for the entire drawing at once can be achieved using Styles and Layers.

To change the dimension style for the entire drawing, first open up the Common tab on the Windows Ribbon bar and select the Styles and Layers tool in the Styles section.



In the Drawing Elements list, choose Dimension. IRONCAD will display a list of available Styles, select ISO as the named style and click the Modify button. Select the Text tab and set:

- Font: Arial Narrow
- Style: Regular i.e. not bold or italic
- Size: 12
- Color: Blue

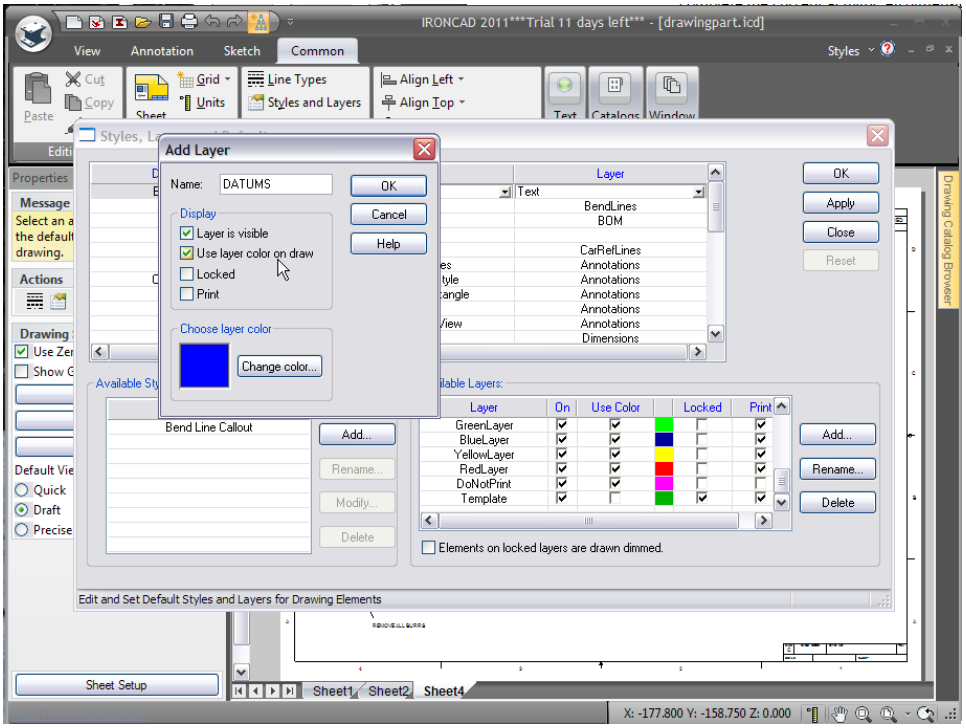
Then choose OK to close the Dimension Properties dialog box and OK to complete the current activity. All dimensions on the drawing, and on every sheet of the drawing, should now update to reflect the changes.

6.1.17. Adding a Layer

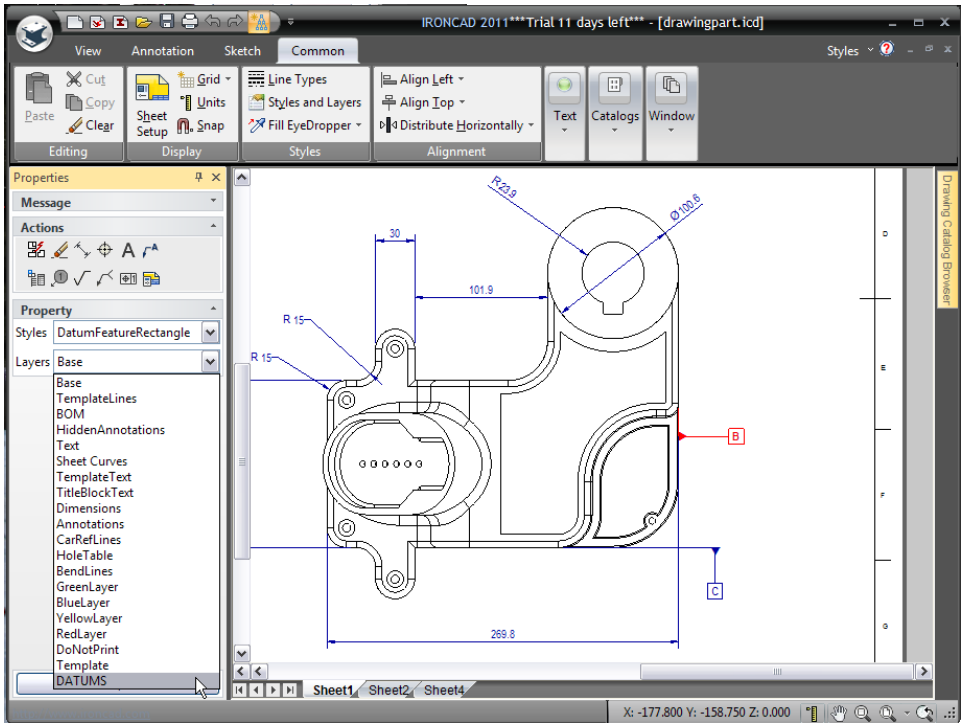
Layers are useful for organizing drawing elements. For example, a new layer can be added specifically for datum's, and it can be assigned a particular color so all datum's can be clearly distinguished on the drawing.

Open the Styles, Layers and Defaults properties box as the previous paragraphs relating to Styles and Layers. In the Available Layers section of the Styles, Layers and Defaults dialog box click 'Add...'

IRONCAD displays the Add Layer dialog box. Enter DATUMS for Name, change the Color to Blue, and select the checkbox 'Use layer color on draw', click OK twice.



To assign the B datum on Sheet 1 of the drawing to the DATUMS layer, select the datum B item and change its layer to DATUMS in the Layers box on the Property Panel.



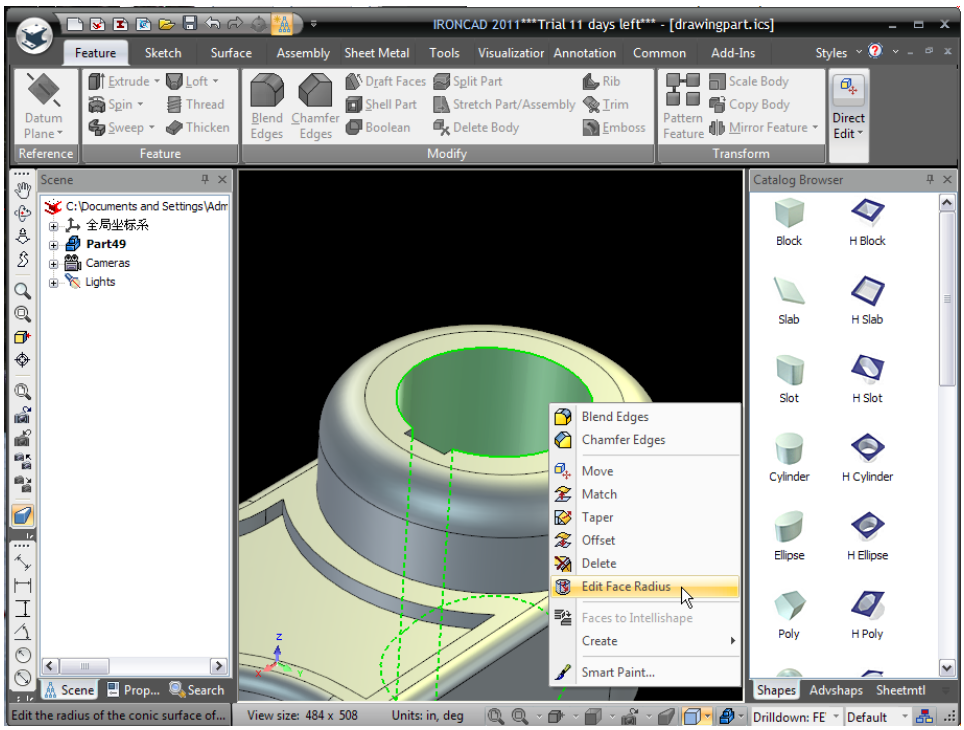
Repeat this process to move the A and C datum's to the DATUMS layer. Now the color of all the datum's can easily be modified by simply changing the color of the DATUMS layer.

6.2. Part/Drawing associativity

All the views contained within the drawing are snapshots of the originating 3D model. When a change is made to the geometry of a 3D part, the changes are updated in the associated drawing(s). In the majority of cases, the dimensions will also automatically update.

On Sheet 1, right-click on the isometric view and choose 'Edit Scene'. The 3D part file will open.

Select the outer circular edge of the feature shown, right-click, choose Blend Edges, enter a constant radius of 0.5, and click the OK button to apply the blend. Drill down to select the inner cylindrical face of the hole shown, right-click, choose Edit Face Radius and enter a value of 1.125



Minimize the IRONCAD scene and switch back to the drawing.

IRONCAD will present a dialog box informing you the drawing is out of sync with the model. You can update the drawing automatically or postpone the update if you need to zoom out and view all the possible ramifications of the change. Click OK to update all views. The drawing updates to reflect the changes in the part file.

Note that the radial dimension has automatically updated to reflect the model change. However, the diameter dimension may now be a magenta color, indicating that it is not valid and should be deleted then re-applied. This happened because the edge to which the dimension was originally applied was destroyed by the creation of the new blend.

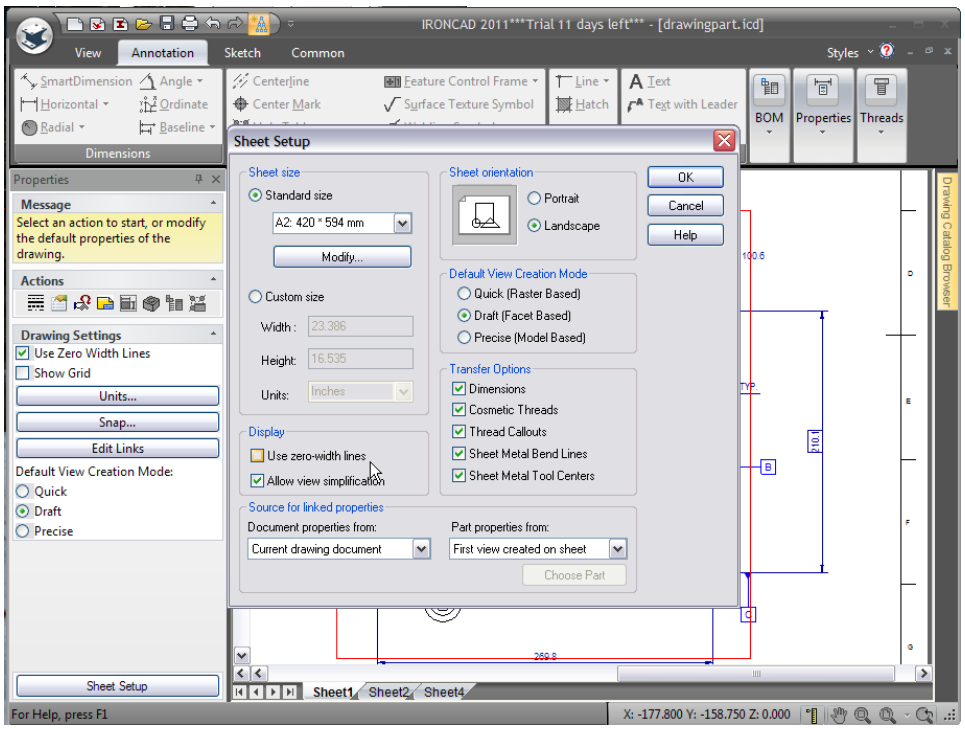
It is also worth noting that if the changes were a revision to the model say to Issue A, if the drawing and the model are in session i.e. both open saving the model as 'Drawingpart-A' would result in the internal pointer of the drawing being changed to the new 3D model identity.

6.3. Exporting the Drawing

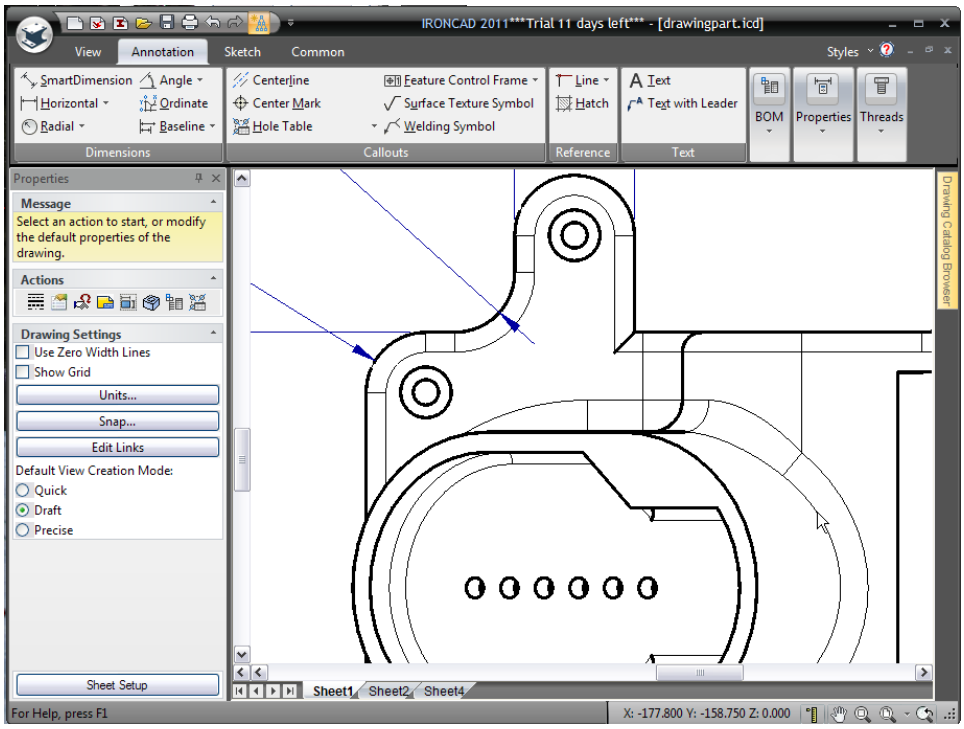
Up to this point all of the drawing elements of the drawing are shown in a single, thin line style. This is due to one of the default settings of the A2 drawing template, namely the option to 'Use zero-width lines' is checked in the Sheet Setup panel.

To access the Sheet Setup panel, right-click in an empty area of the drawing i.e. outside of any drawing view and select 'Sheet Setup' from the menu.

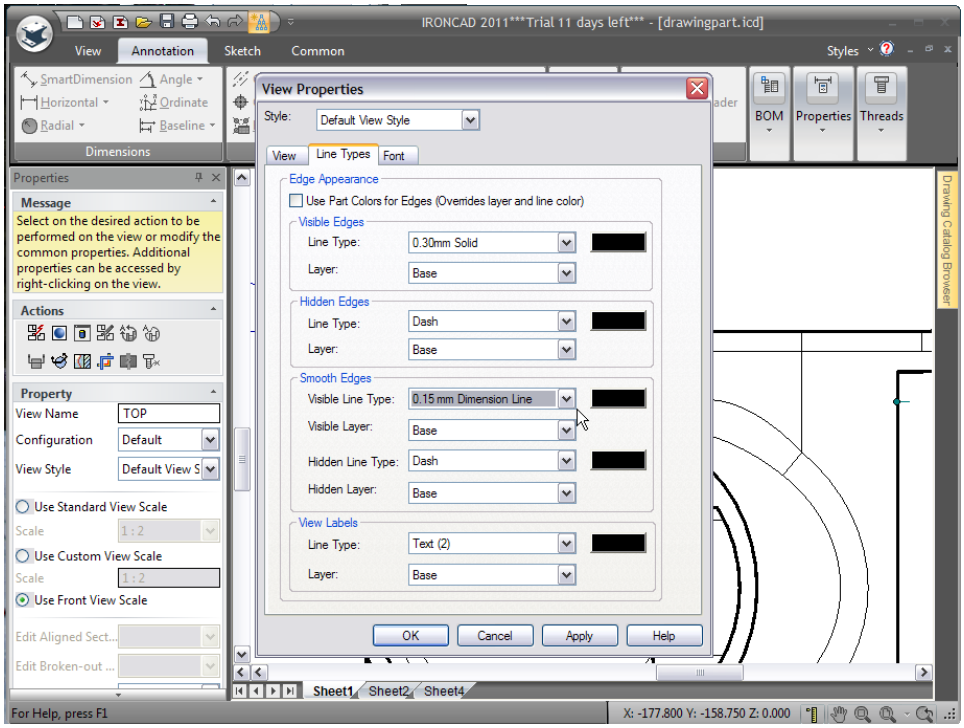
In the resulting options panel un-check 'Use zero-width lines' option.



Returning back to the drawing, zoom into the Top view as shown below, IRONCAD will now draw the feature lines in a a heavier type than blend edges.



In other drawing templates this behavior may be different. If it is right-click on the view properties and in the Line Types panel set the Smooth Edges to a desired line style and color.



With the drawing element styles set the drawing can be exported in a number of different formats. For Example, to export the drawing as a pdf file, right-click on an empty area of the drawing and select export, enter a name and the relevant settings. IRONCAD will export the drawing.

This ends the section introducing IRONCAD's drawing creation tools. The chapter covered:

- Creating Standard Views
- Changing the View Scale and Rendering Options
- Moving Views
- Datum's, Dimensioning & applying Tolerances
- Centerlines and Reference Curves
- Section, Detail, Auxiliary, and General Views
- Multi-Sheet Drawings
- Hole Table
- Sheet Drawing Tools
- Styles and Layers
- Part/Drawing Associativity

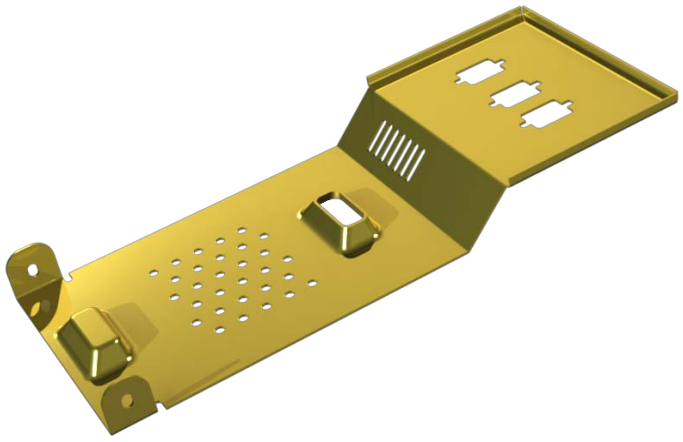
The next chapter will introduce the fundamentals of IRONCAD's sheet metal design capabilities.

7. IRONCAD SHEET METAL APPLICATION

IRONCAD is provided with a purpose designed capability to design parts for sheet metal fabrication. The system works in a way familiar to those who regularly use fabrication techniques using punching and folding machines. The models created using the sheet metal tools can be unfolded to provide 2D drawing layouts of the flat patterns.
The following guide provides an overview of some of the functionality provided by IRONCAD to design parts for fabrication using punching and folding machines. It is worth noting that the tools provided by IRONCAD classed as sheet metal can also be used for a range of other design requirements such as packaging cases and even folded, flexible circuit membranes.
The topics covered below include:

- Sheet metal stock
- Adding and editing bends
- Editing and adding Bend Reliefs
- Use of standard punch shapes
- Adding corner radii
- Use of catalogs for commonly used items
- The miter command
- Unfolding and drawing creation
- Creating a custom punch

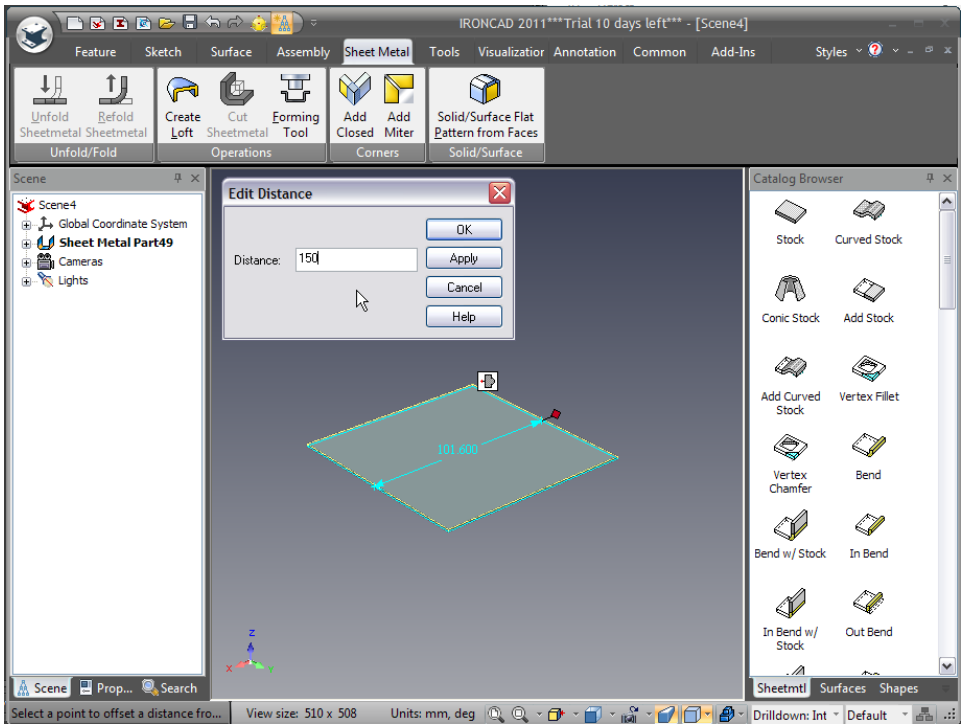
As with other models created in IRONCAD, sheet metal parts can be used in photo realistic rendering. At the end of this guide the part designed should resemble the image below which has had a gold finish applied.



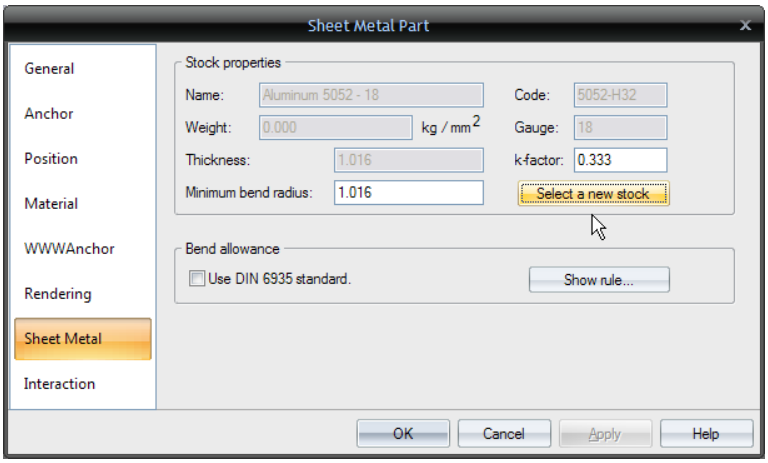
7.1. Sheet metal stock

All sheet metal fabrications are based on off the shelf sheet material thicknesses. IRONCAD is provided with the most popular sheet stocks materials. The list of stock and its contents can be edited to reflect any locally used materials. To do so, search for the file Tooltbl.txt taking care that any edits follow the format of the data (its best to copy and paste then edit individual entries rather than create fresh entries). The stock determines the standard bend radii applied in the design process. Bend radii however can be changed locally within the model.

Create a new metric scene using the Gray template and open the 'Sheetmetal' catalog. Drag a Stock shape from the catalog onto the scene and select it at the IntelliShape level. Immediately the difference becomes apparent as the IntelliShape is highlighted in 'Shape mode' rather than in 'Sizebox mode' as can clearly be seen in the image below.



Touch the right hand curve and right-click on the handle to set the length of the part to 150mm from the opposite end. Similarly set the width to 100mm.
Click into the scene background to deselect the part and then right-click on the part and select Part Properties from the menu.

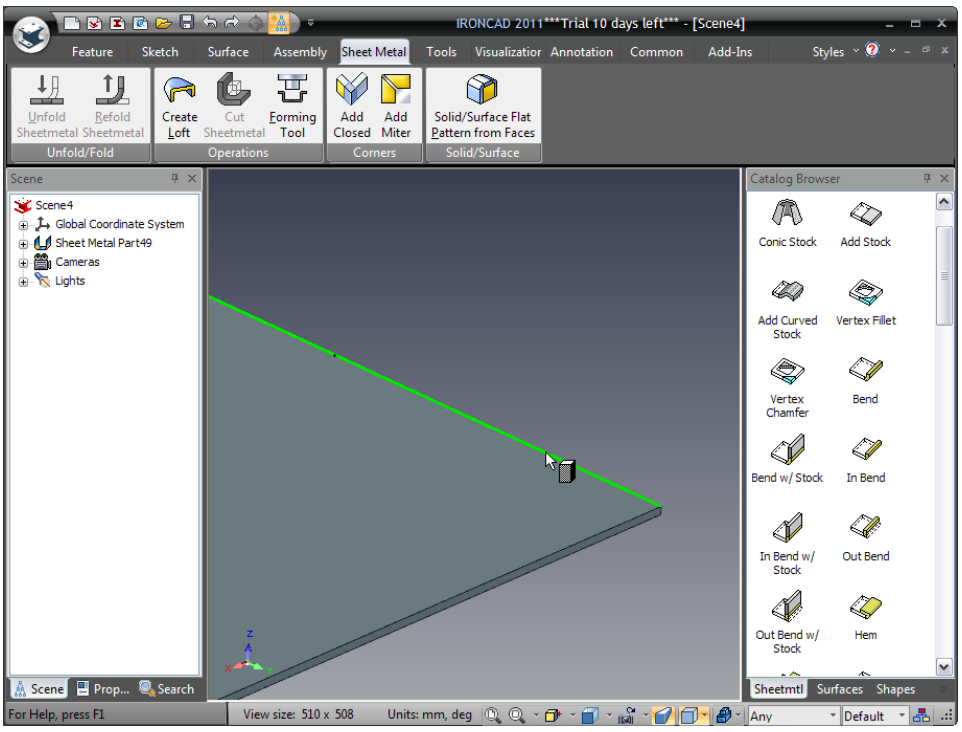


In the Sheet Metal Part properties box a dedicated tab is provided to control all aspects of the standard part functions such as bend radius, bend calculation parameter (k-factor) and a button for selecting new stock material. For this example the default Aluminum 5052 – 18 will be used.

7.2. Adding a bend with stock

IRONCAD provides a number of options for adding a bend, some of which will be used within this guide. Zoom into the model to get a clearer view of the right hand end of the stock part as shown below.

Drag an 'Out Bend w/Stock' shape onto the top edge of the stock as shown. Indicating the top edge of the stock will cause the bend to be upwards, the bottom edge would cause the bend to go downwards.

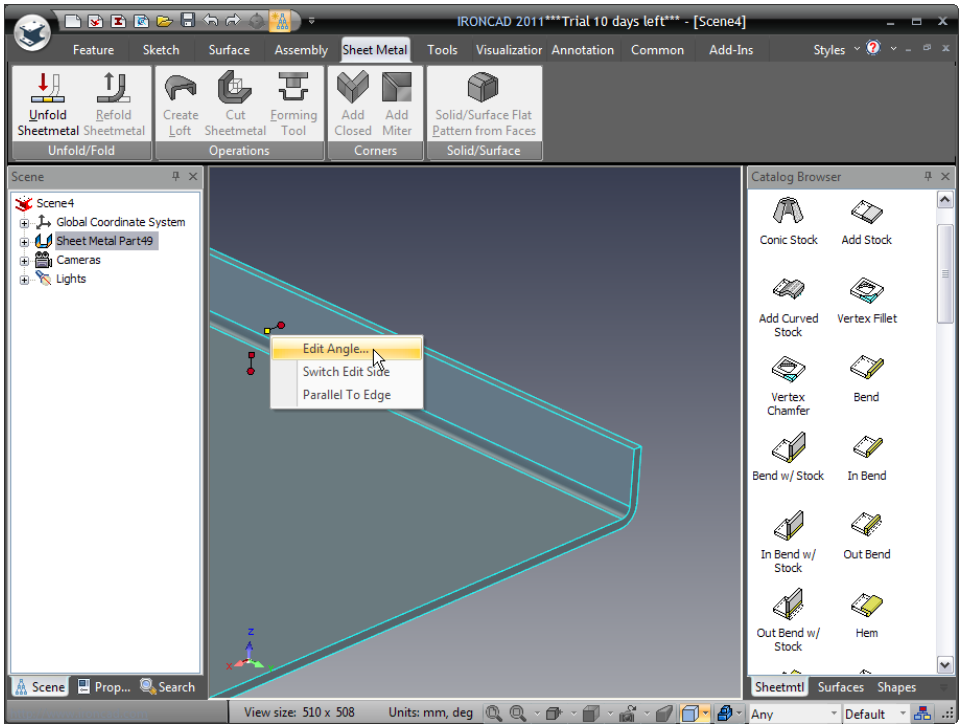


IRONCAD will create a 90 degree bend with a piece of stock on the bend. It is worth noting two things:

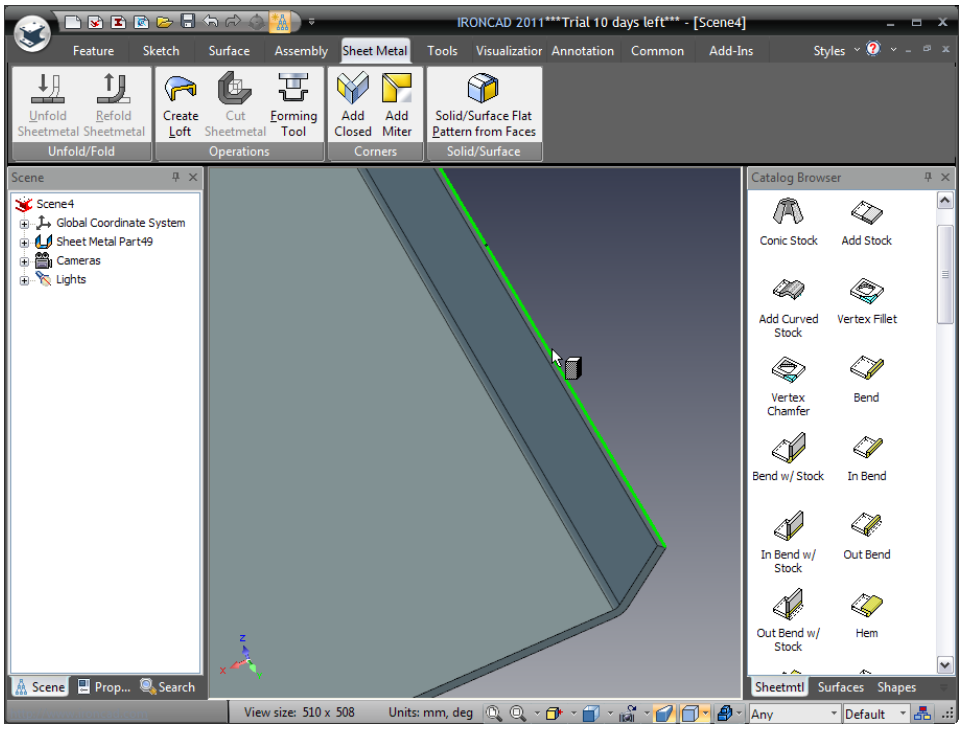
1. Bends with stock allow the stock material to be treated separately to the bend i.e. resized along the bend width not just in height (or even deleted),
2. Out bend means the outer surface of the bend will be in line with the original edge to which the bend was applied.

7.2.1. Changing a bends angle

'Hovering' the mouse over the central area of the bend when the part is shown in cyan results in two pairs of red control handles, one pair for length/position the other for angle. The angle handles are square, the length round. Referring to the image below set the angle of the newly added flange to 45 degrees.

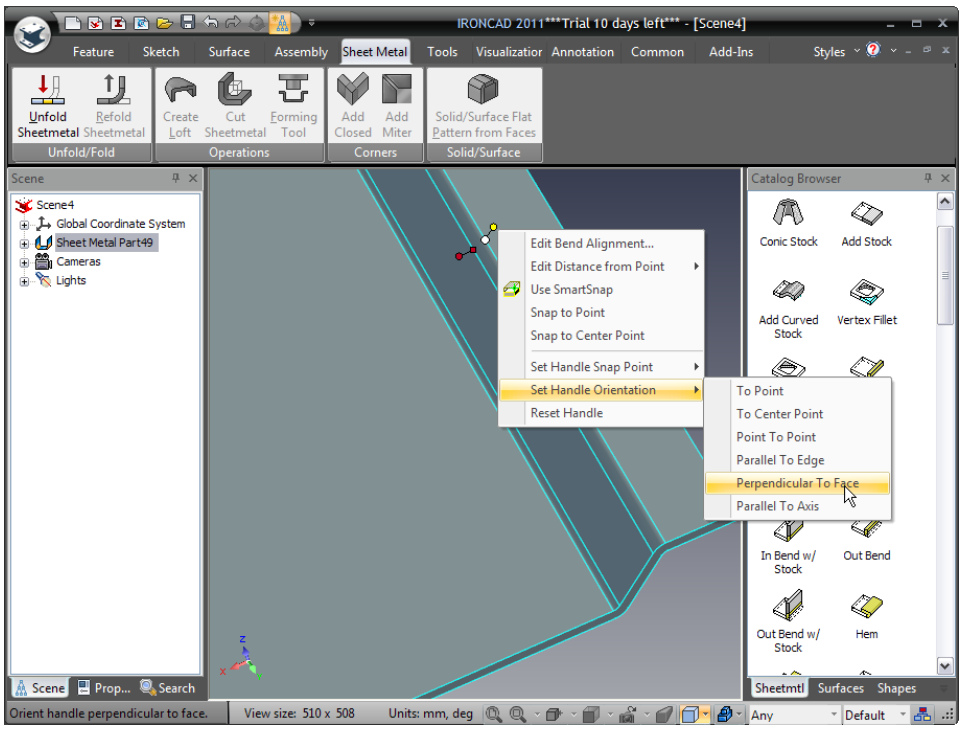


Rotate the view to look on the model so that both edges of the new flange are visible and add another 'Out Bend w/Stock' again to the outer edge as shown and as in the previous example set its angle to 45 degrees so that the newly created flange is parallel to the original stock piece.

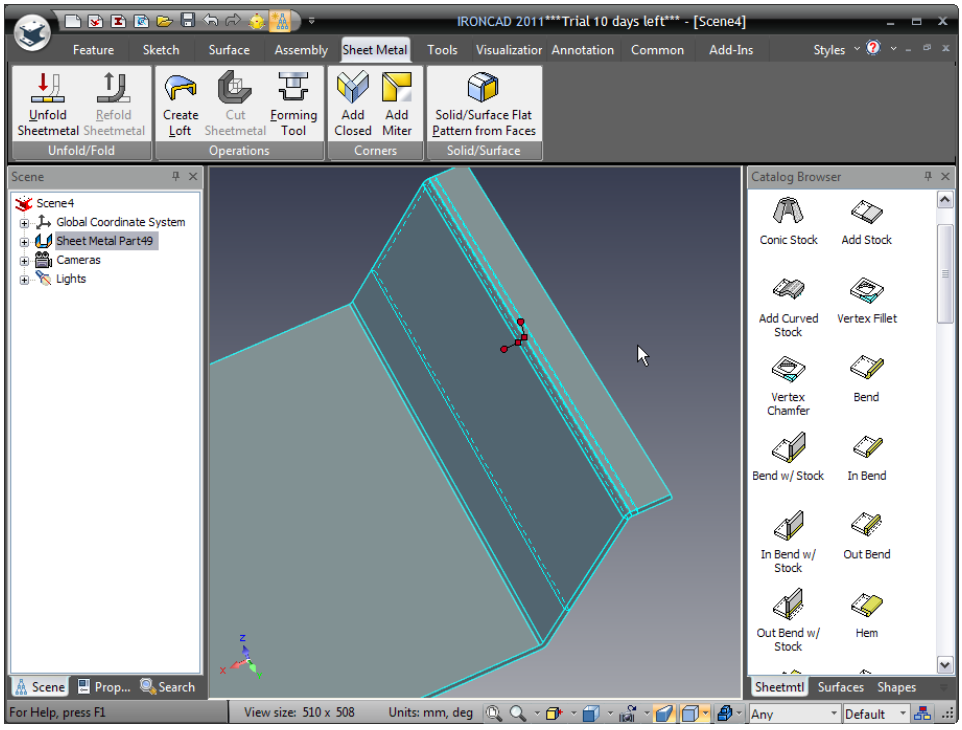


7.2.2. Re-orientating length handles

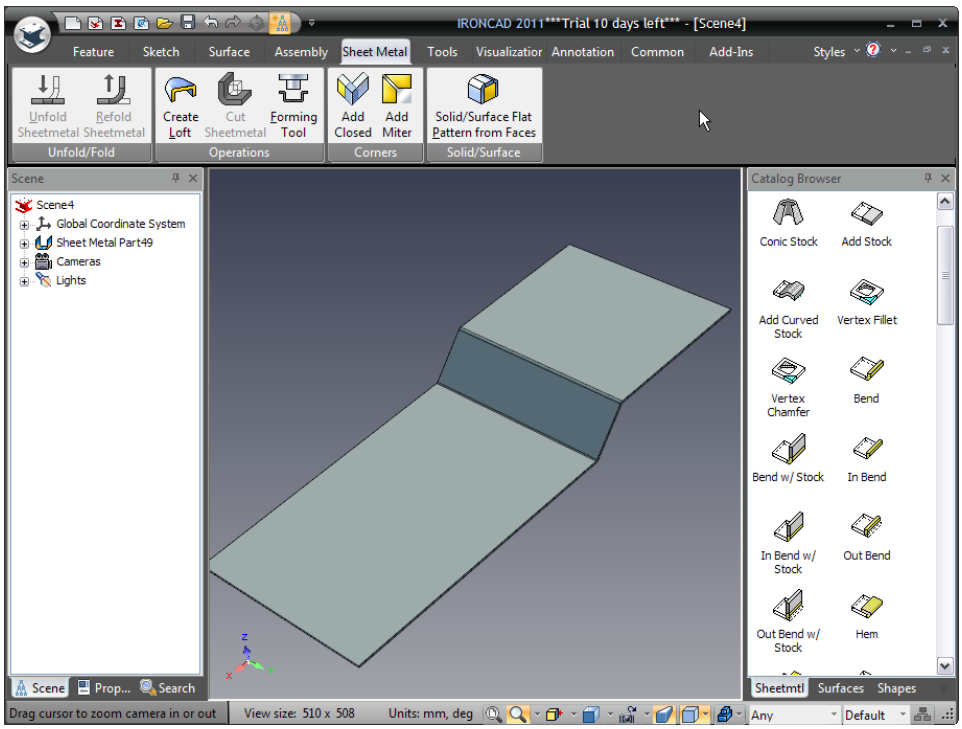
The top face of the new flange needs to be 20mm from the original stock top face however the preceding flange is at 45 degrees, and hence so is its length handle. To ensure the distance is measured perpendicular to the original stock, right-click on the length handle this time for the latest flange added and set the handle orientation to be 'Perpendicular to Face' and select the top face of the starting stock.



With the handle now perpendicular to the starting stock, right-click again on the handle and ensure it is on the face to be moved by choosing the 'Set Handle Snap Point', followed by 'To Point' and select the top face of the flange to move again. Next, right-click on the length handle and set the distance to 20mm. IRONCAD will add stock to the first flange to match the dimension entered.



Zoom out and set the end edge of the short flange so that the overall length of the part is 250mm as shown below.



7.3. Adding bends with combined stock

Previously the bends added had a separate stock shape added, these shapes add flexibility to the stock. However there are circumstances where the stock applied with the bend needs to be related to the bend. These shapes are identified in the catalog as simply In Bend, Out Bend and Bend.

7.3.1. Bend

A Bend is an item where the actual bend portion is added onto the parent stock and the attached stock simply added to the bend. The position of the flange is determined by the size of the bend radius.

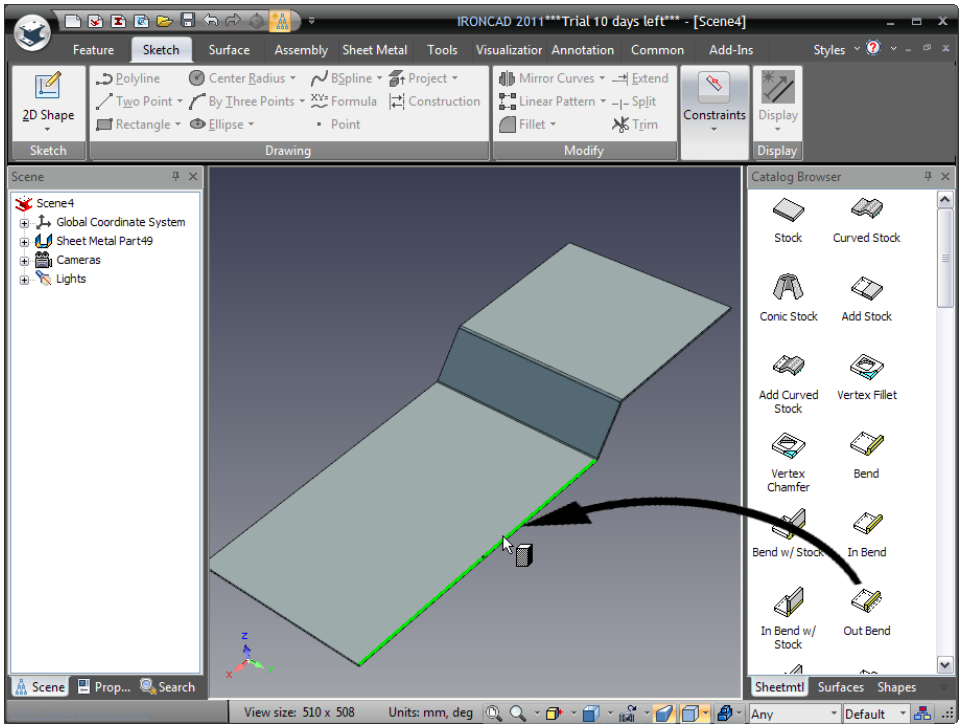
7.3.2. Out Bend

In this case the outer face of the added flange will be created in line with the edge to which the bend is added.

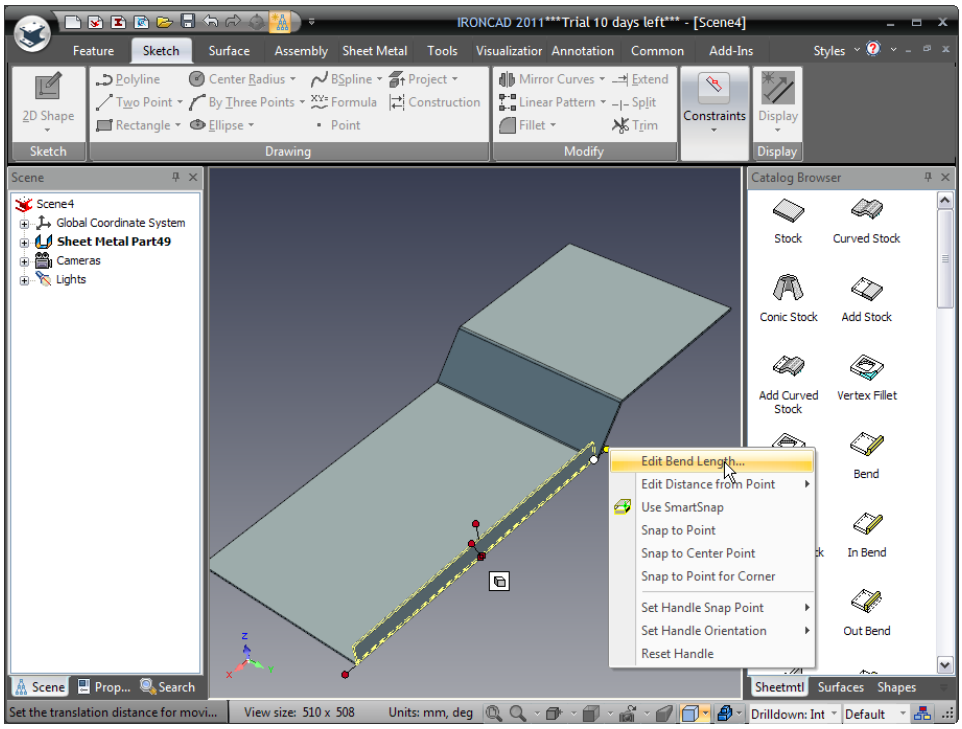
7.3.3. In Bend

An In Bend is one where the inner face of the added flange is aligned with the edge to which the bend is added.

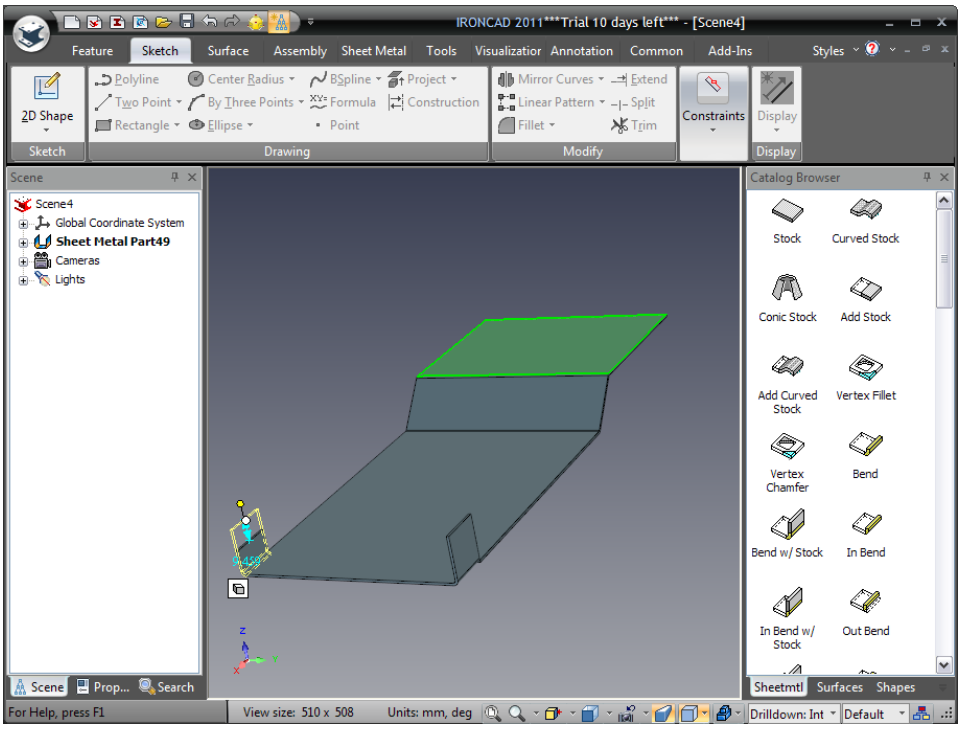
In the following example the controlling size is from the width of the original stock onto which the flanges are to be added i.e. the part will have to fit inside something, in which case the appropriate type of bend would be an ‘Out Bend’. Add an ‘Out Bend’ shape from the catalog to the edge shown in the following image.



IRONCAD will add the bend and flange in the same way as previous bends, however this time it will leave the addition highlighted with ‘Sizebox Handles’. Resize the bend to 20mm using the handle closest to the 45 degree face by right-clicking on the handle shown and selecting ‘Edit Bend Length’.

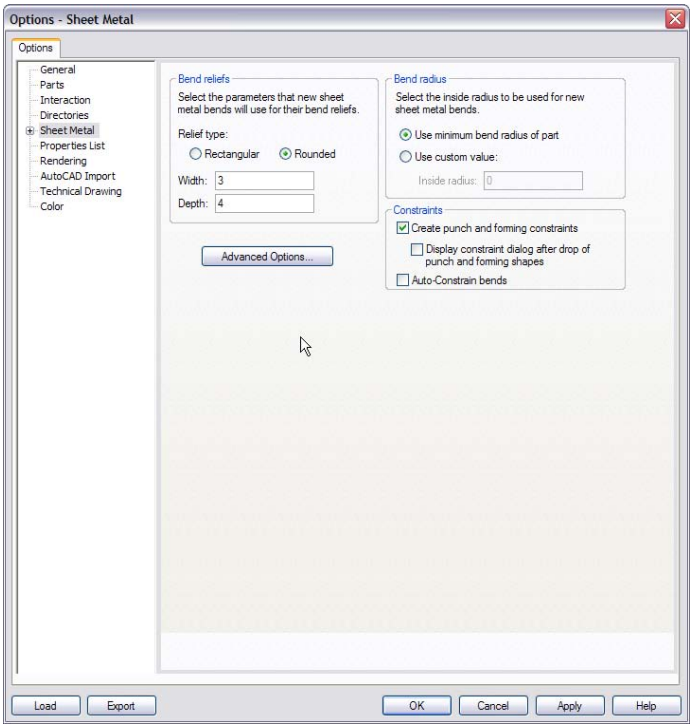


Add another Out Bend to the opposite edge and resize it by right-clicking and selecting ‘Snap to Point’. Next align the upper most edges of both additions to the top face of the upper platform using SmartSnap functionality i.e. selecting the handle with the left mouse button and dragging the handle while the shift key is selected to indicate the upper platform as shown below.

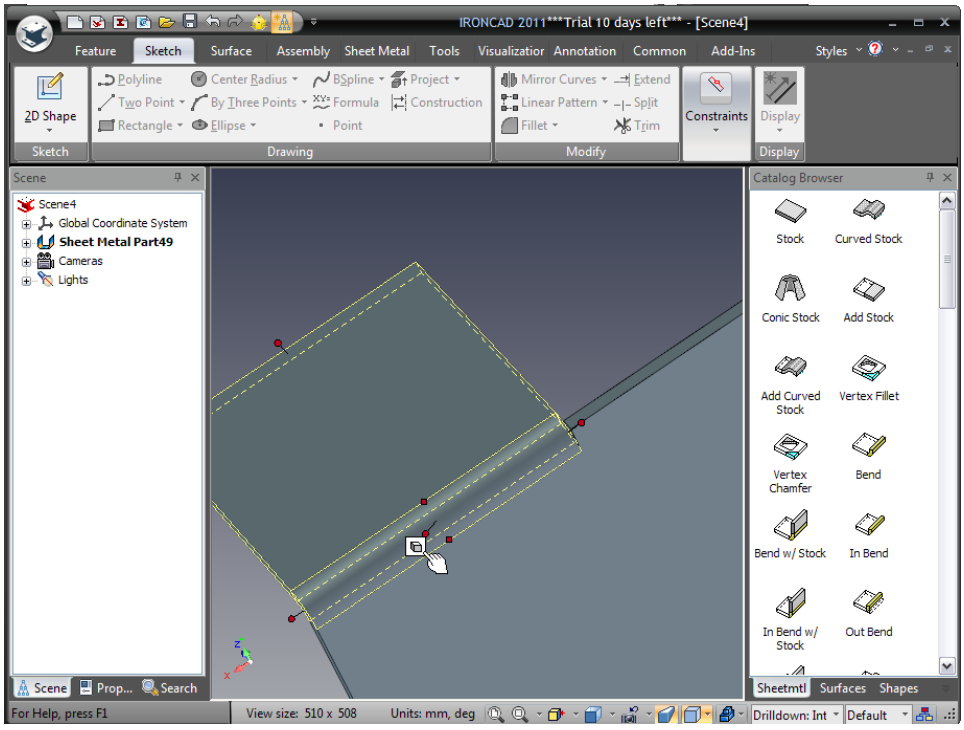


7.4. Adding a Bend Relief

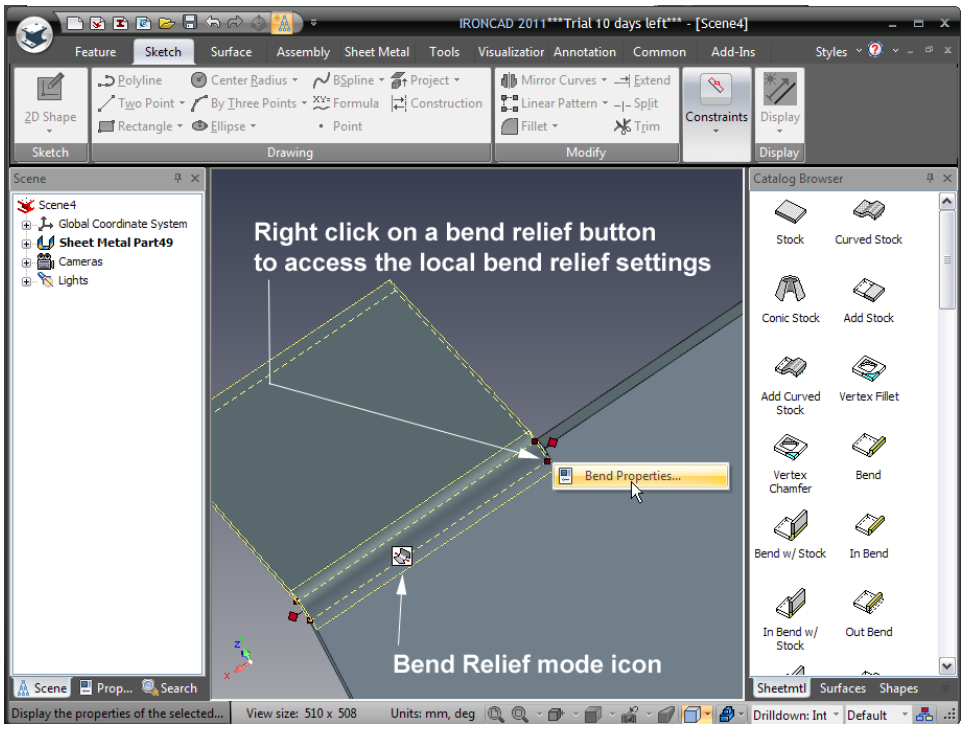
Bend reliefs are provided to allow bends to be produced in the real world by creating a notch for the bend to terminate at. Bend reliefs can be set per bend so that the design can be modified with respect to specific functional requirements or globally by accessing the Sheet Metal controls panel. To set the global options, select the IRONCAD home button, followed by Tools and then Options. Open the Sheet Metal tab and set the bend relief to a width of 3mm, a depth of 4mm and the Relief type to be rounded.



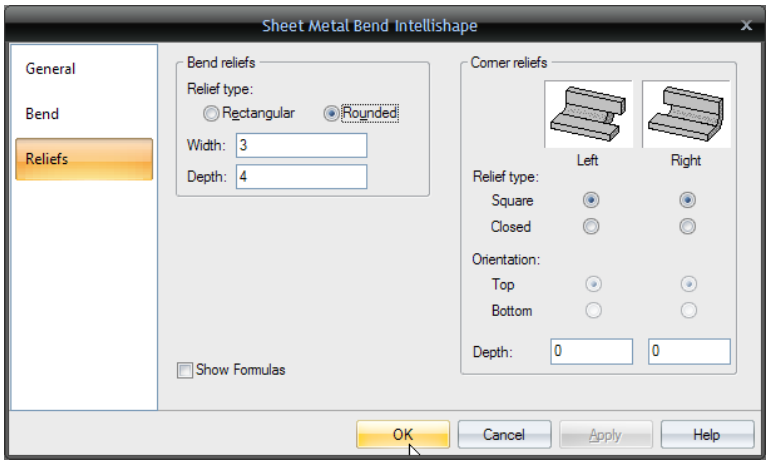
These settings will be applied to NEW sheet metal parts created from this point on. Zoom into one of the new tabs created and select it at the IntelliShape level. IRONCAD will display the tabs controlling handles for length angle height etc. and a mode button very similar to the 'Sizebox/Shape' mode button of standard IntelliShapes. Click the mode button to activate the bend relief handles/button.



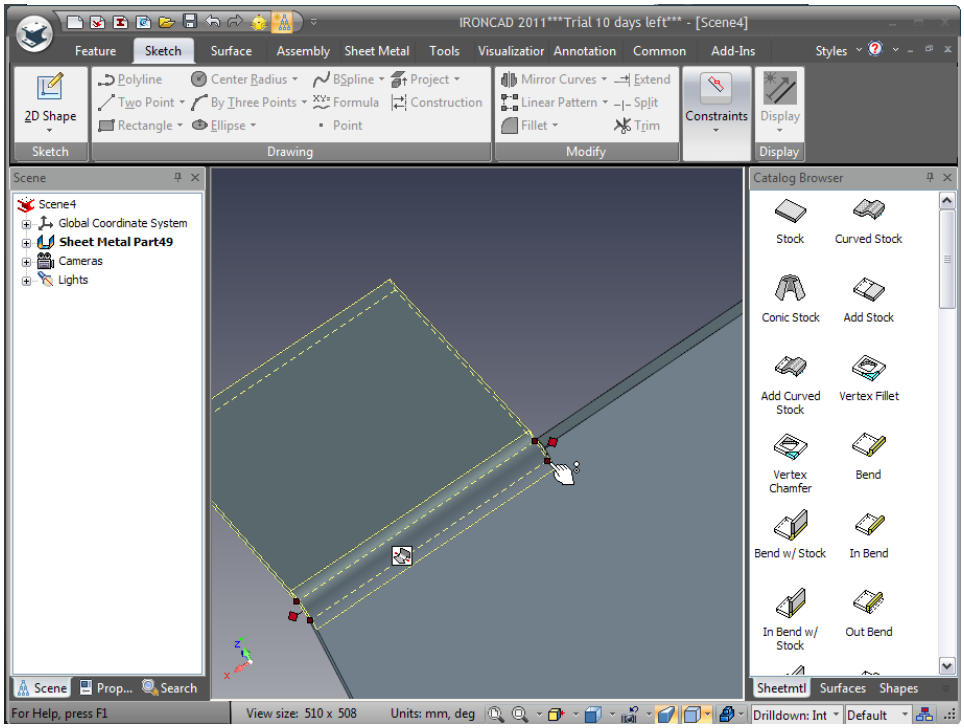
IRONCAD will change the buttons graphic to represent a bend with reliefs applied. As the global settings are only applied to a new part, with the mode set to Bend Relief, right-click on one of the bend relief buttons and access the Bend Relief properties to be applied to that specific bend.



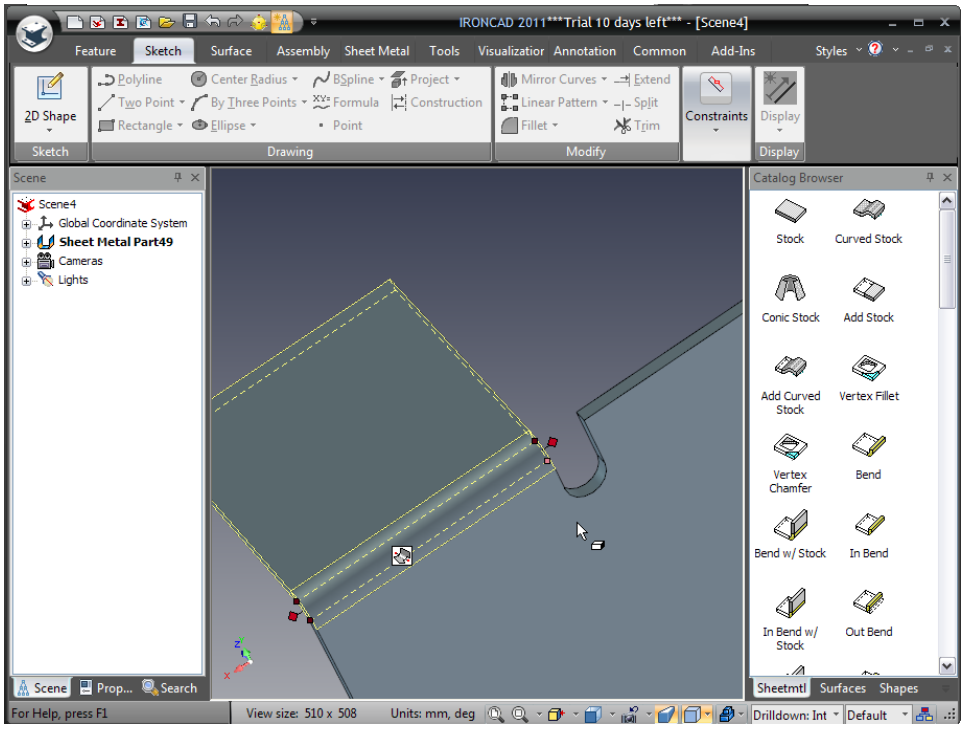
Set the width to 3, the depth to 4 and the Relief type to Rounded and click OK.



Now set the bend relief by clicking on the handle at the side of the bend where the relief is required.



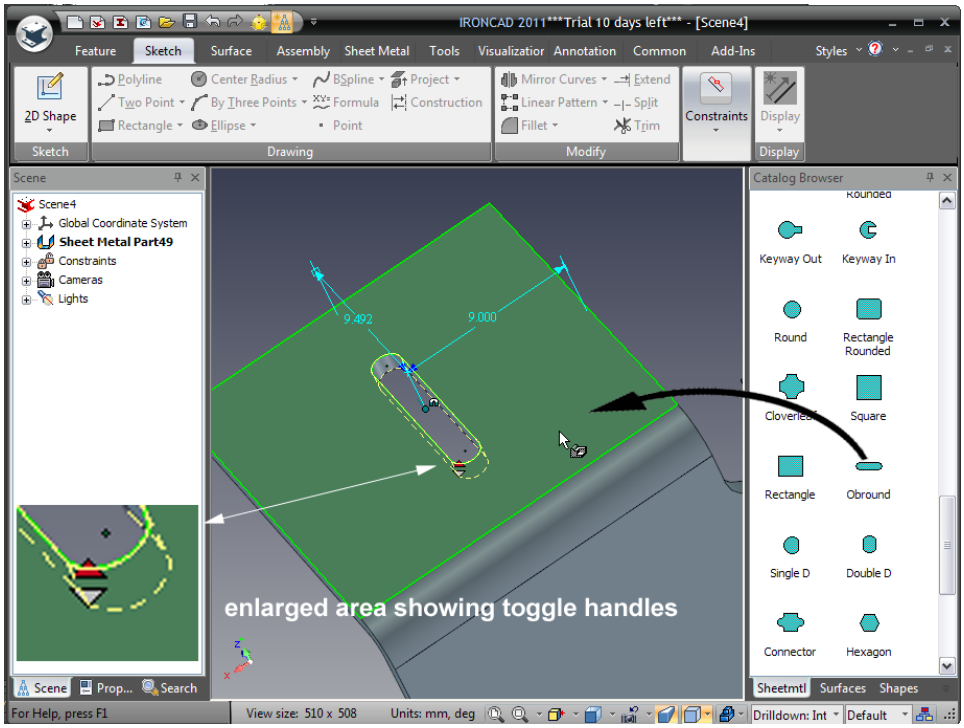
The resulting bend relief should look like the following image.



Add the same type of bend relief to the matching tab on the opposite side.

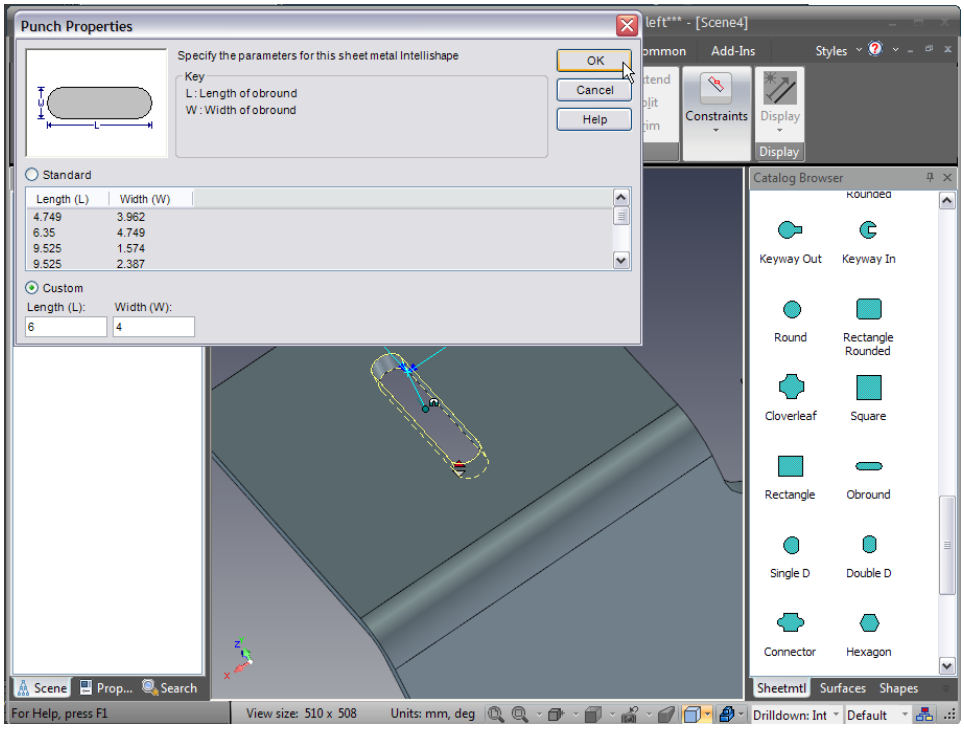
7.5. Adding standard punch features

The sheet metal catalog also presents a number of commonly found punch shapes with associated dimensions. For example to add and ‘Obround’ hole to newly created tabs, drag an ‘Obround’ cutout from the catalog onto the tab as shown.

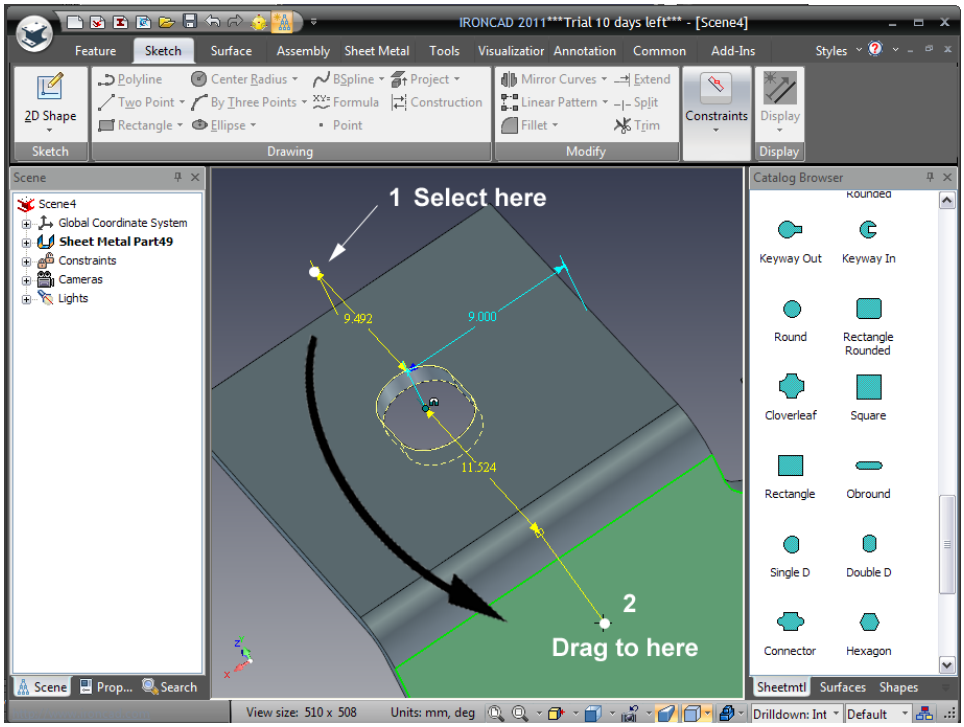


Clicking on the red toggle handle will cause IRONCAD to cycle through the standard tooling options for this feature. IRONCAD will display a small green dot by the handle to click when the appropriately sized shape is produced. Alternatively, right-click on the 'Obround' cutout and select 'Tooling Properties'.

In the Punch Properties control panel, select Custom and set the Length to 6 and width to 4 and click OK.



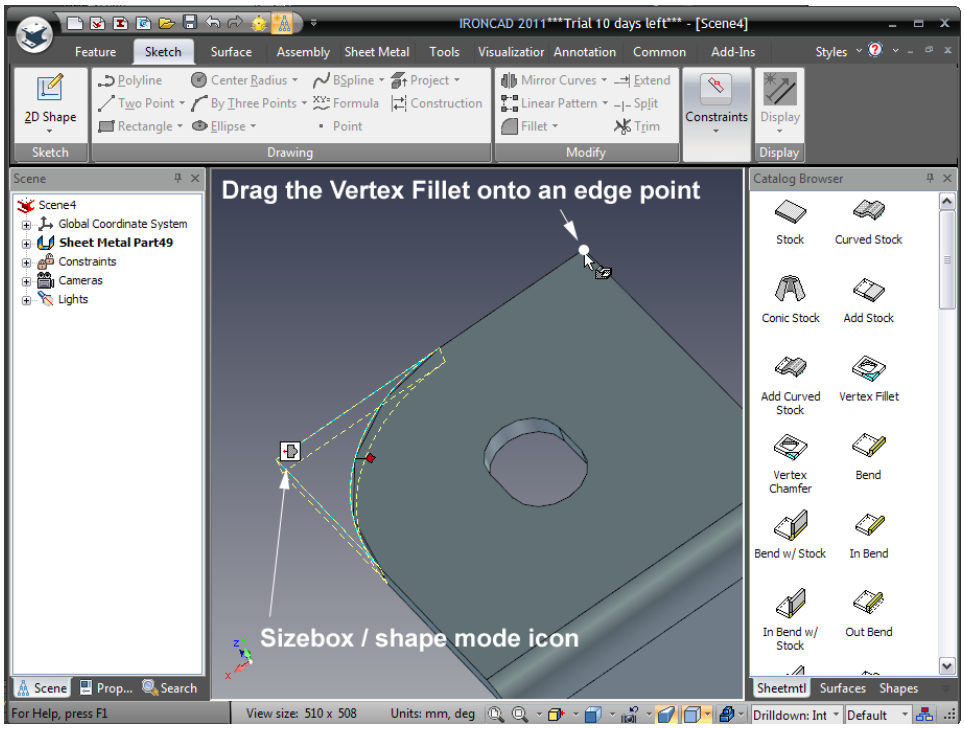
To position the 'Obround' relative to the rear face of the part, click on the SmartDimension (1) and drag the dimension onto the rear face. IRONCAD will highlight the rear face when it has been selected as shown below:



Right-click on the dimension value and set it to 10.

7.6. Adding round corners – Vertex Fillets

A Vertex Fillet is a corner radius specifically designed for sheet metal parts. Unlike its standard 3D counterpart, this item is dropped onto a point (or vertex). Drag a Vertex Fillet to the two corners shown.



To set the size of the Vertex Fillet to 8mm, click on the 'Sizebox/shape' mode icon and right-click on one of the handles and set the sizebox to 8 and click OK.

7.7. Re-using existing features via the catalog

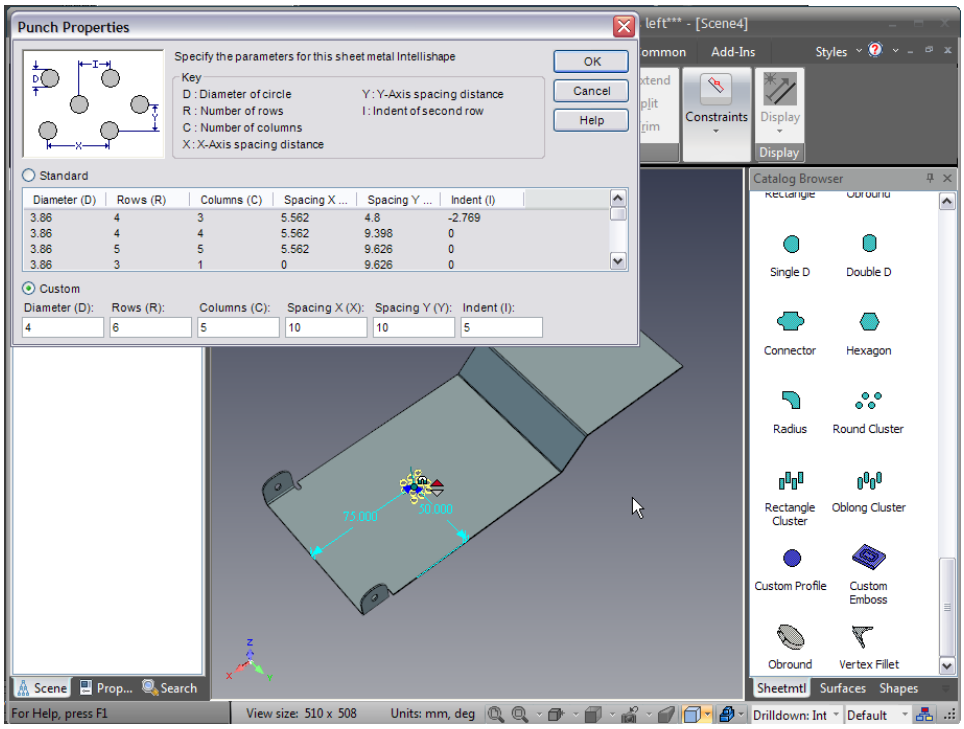
The current design has used standard catalog functions with some modifications. To simplify re-use of these features with the current settings simply drag the features back onto the catalog.

Highlight the feature at the IntelliShape level using a left mouse click and then drag the feature onto the 'Sheetmetal' catalog(or a new, personal catalog) and release the mouse button. IRONCAD will add the selected shape to the catalog for subsequent re-use. If the shapes are to be used beyond this design session, click twice on the name and give the feature a useful identity.

Using this feature, drag the ‘Obround’ and a Vertex Fillet onto the catalog and use them to complete the tab on the opposite side.

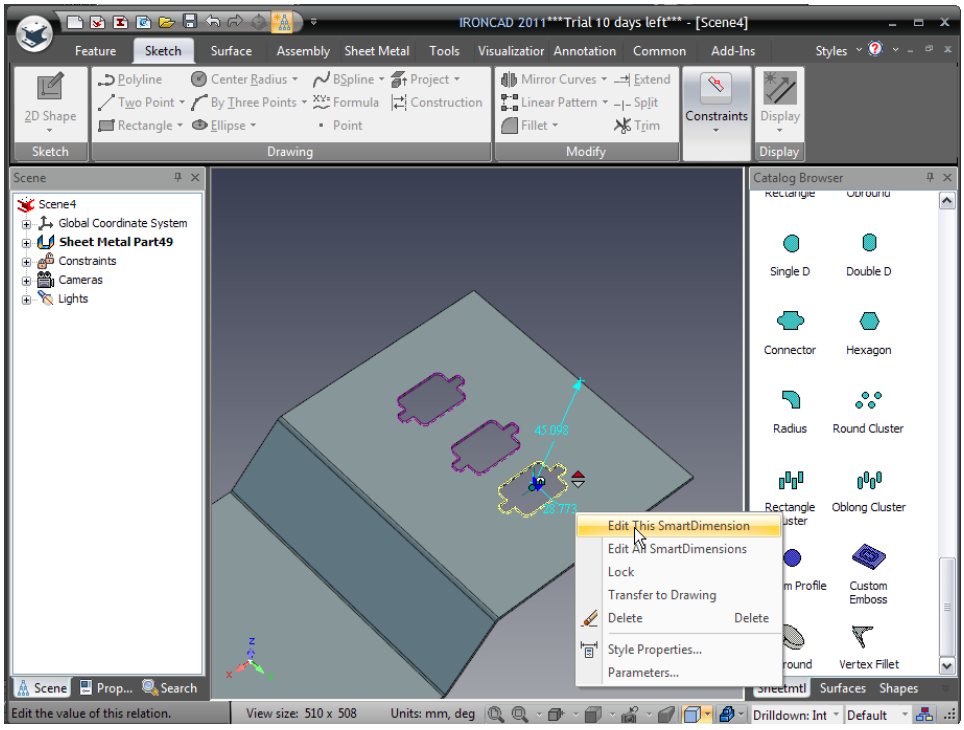
7.8. Adding a cluster of holes

Using a similar technique to the ‘Obround’ drag the ‘Round Cluster’ catalog item onto the part and set its position to 50mm from the side and 75mm from the bottom edge. Right-click on the cluster and set the Tooling Properties to Custom with the hole diameter being 4, the number of rows set at 6, the number of columns should be 5. The X and Y spacing should be 10 and the Indent 5.



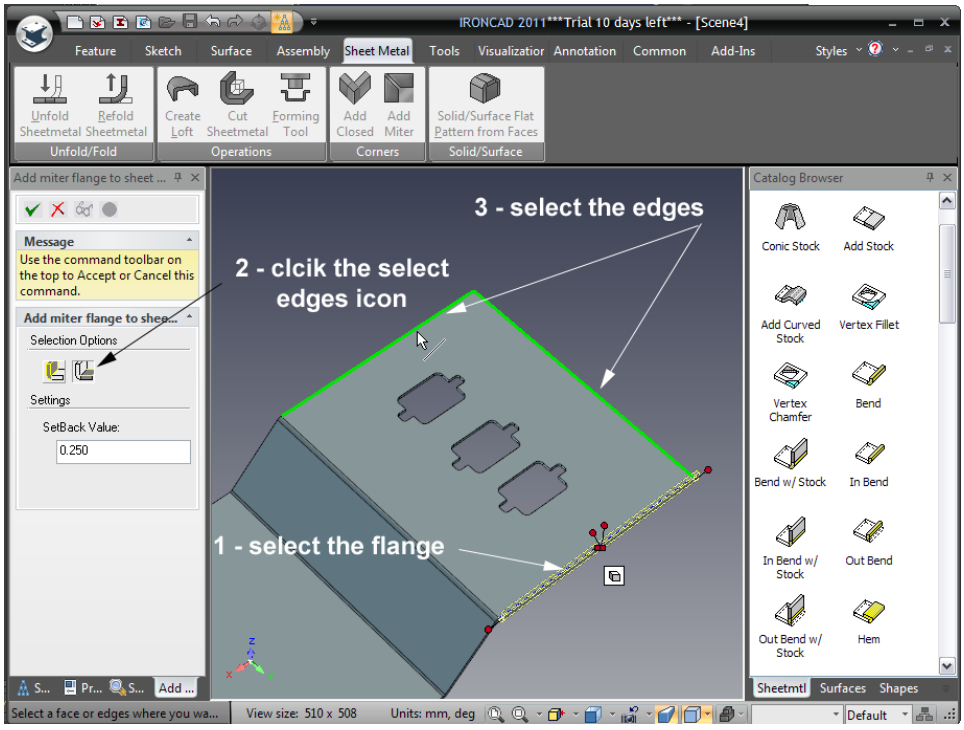
7.9. Manipulating features with the TriBall

As with any function of IRONCAD, punch features can be manipulated with the TriBall. Add a Connector catalog shape to center of the upper flange and activate the TriBall. Rotate the connector through 90 degrees and then create one linked copy each side of the first one dropped without worrying about the position. Referring to the following image set the center of the first connector hole to be 20mm from the edge.
Repeat the process to position the second one 50mm from the same edge and the third 80mm, again from the same edge.



7.10. Using the Miter tool

From the Windows Ribbon bar, select the Sheet Metal tab. Add a ‘Bend’ from the catalog onto the edge shown in the image. If not already open, display the Sheetmetal tab of the Windows Ribbon bar and select the Add Miter tool from the Corners panel menu. By default IRONCAD’s selection option will be in the mode to ‘Select bends’. Select the last bend added and then click the ‘Select Edges’ tool on the IRONCAD Property Panel and select the other two edges of face and click Apply and Exit.

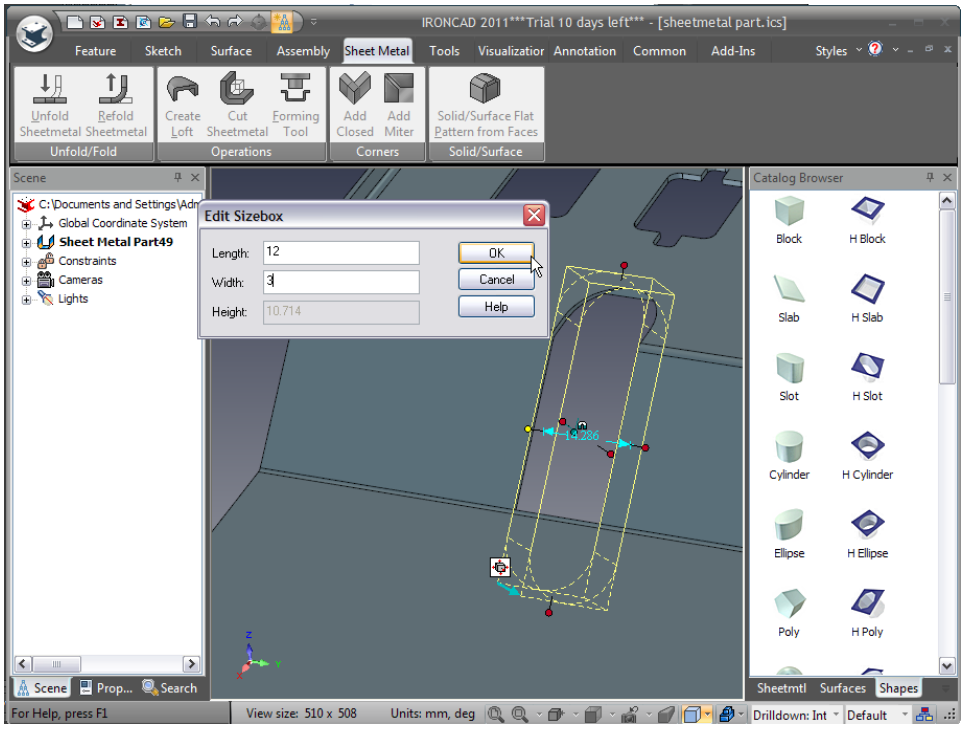


IRONCAD will extend the bends applied and apply the correct treatment to enable the part to be made correctly as shown, to see this treatment rotate the part to look at the corners produced.

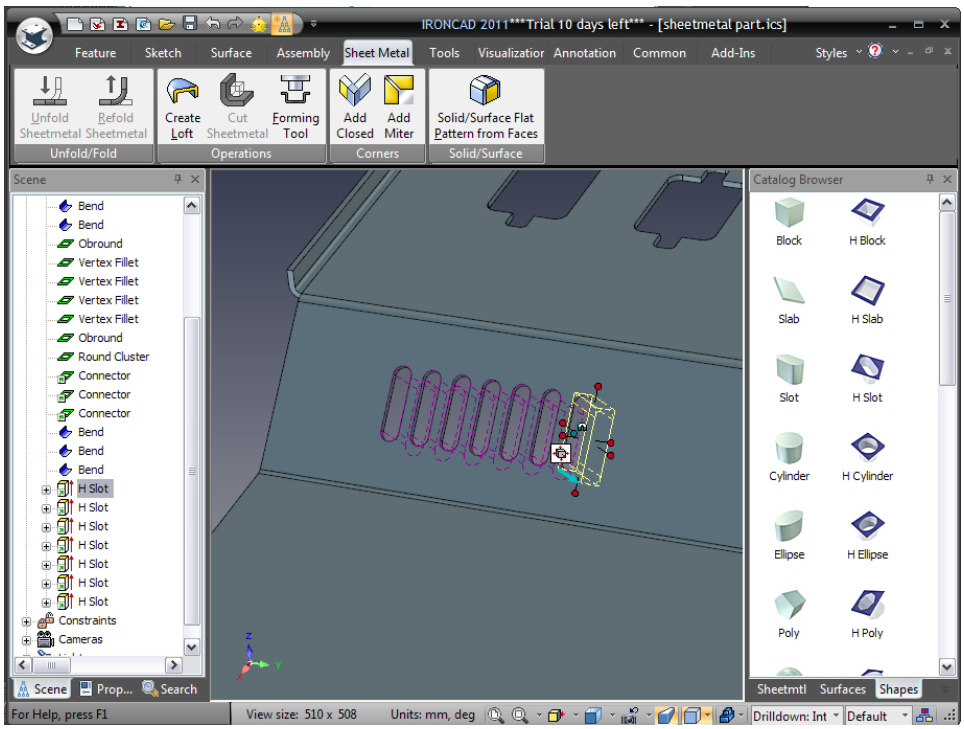
7.11. Using standard catalog items in the sheet metal design process

IRONCAD does not restrict the functions of sheet metal design to the items provided in the Sheet Metal catalog. Standard hole shapes can be used too.

Open the Shapes catalog and drag a H Slot onto the center of angled face using SmartSnap feedback and resize it to be 3mm wide by 12mm long. Notice the depth entry is grayed out.



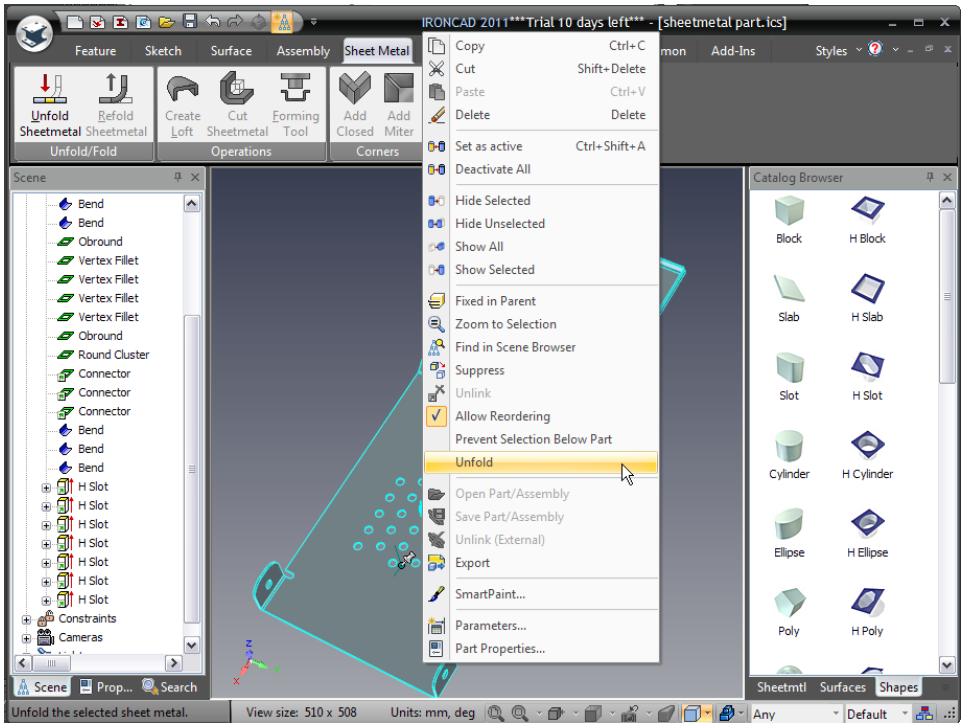
Using the TriBall create 6 linked copies of the H Slot going left across the part at a spacing of 5mm.



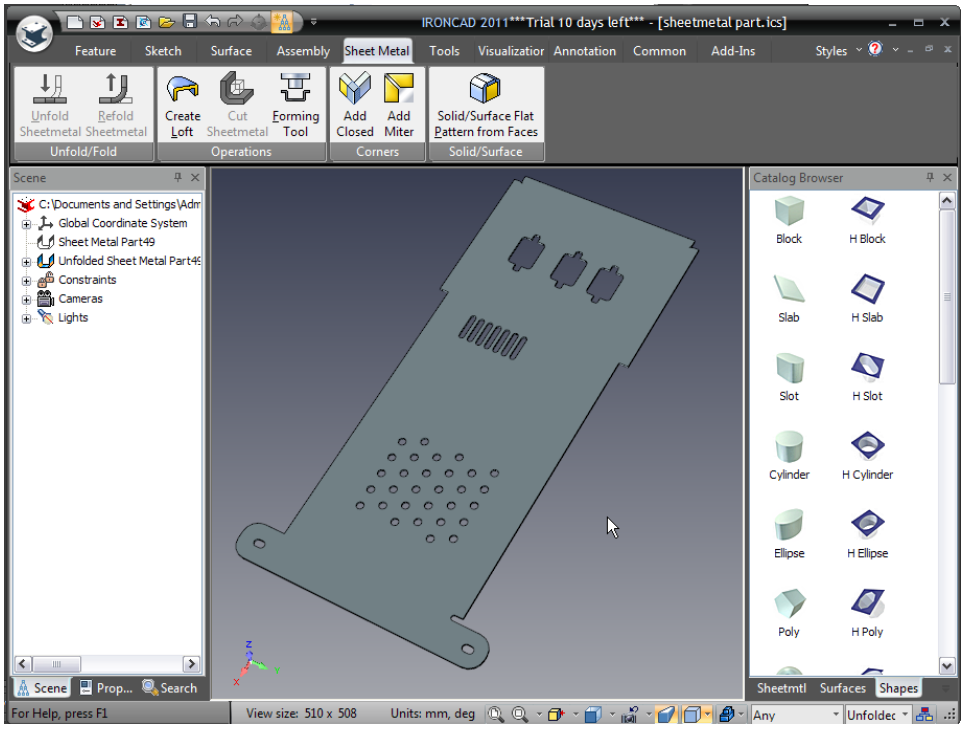
Open the part in the scene browser, IRONCAD will display sheet metal specific functions and ‘standard’ IRONCAD features that behave as standard IntelliShapes. Change the length of one of the H Slots to be 15mm and all will follow suit.

7.12. Unfolding the part

Zoom out to see the complete part and after right-clicking on the part select Unfold.



IRONCAD will create a new configuration and unfold all the faces with respect to the starting stock part.

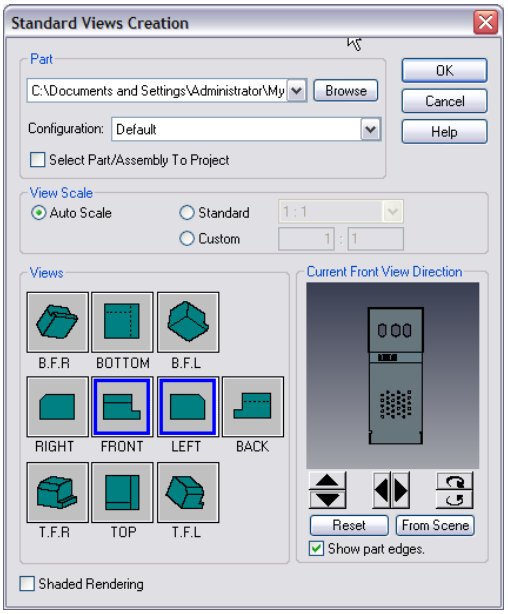


7.13. Creating a 2D drawing of the sheet metal part

Creating a 2D drawing follows the standard IRONCAD method for creating a drawing. In this section the drawing will comprise both the unfolded and the folded views of the part. Prior to creating the drawing, right-click on the unfolded part and click on Unfold again. IRONCAD will refold the part. Select the cluster of holes and right-click on the dimensions positioning the cluster and click on 'Transfer to Drawing'. IRONCAD will append the dimension with a '>' symbol. As with all IRONCAD models, the part needs to be saved before a drawing can be created.

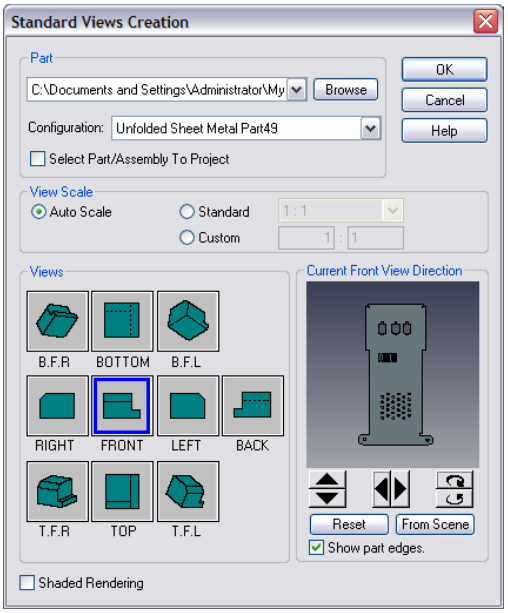
From the IRONCAD home button, select File followed by New. From the resulting menu click drawing and OK. IRONCAD will display the familiar set of 2D templates, select A2 from the ISO set of templates.

From the View tab, select the Standard view icon and orientate the part as shown and create a Front and Side view.



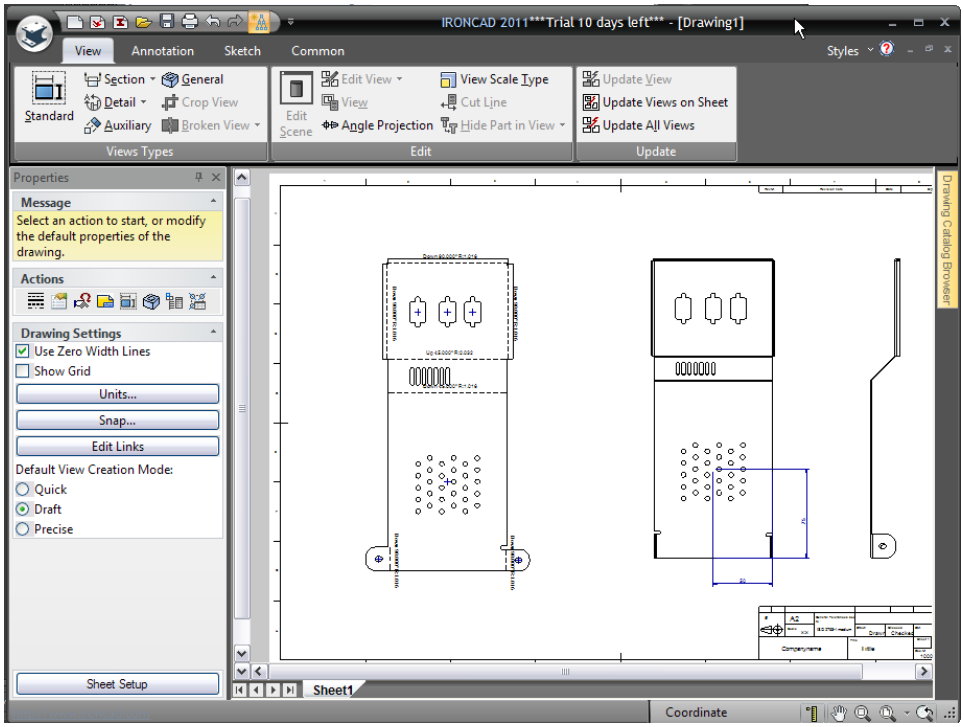
When the part was saved the model displayed was the 3D representation in the Default configuration, in the Standard View Creation dialog box IRONCAD selects this configuration for the view creation. Click Ok and position the views.

To add the unfolded view, within the IRONCAD drawing environment click on the 'Standard' view icon again. This time click on the Configuration pull down and select the 'Unfolded' configuration. Orientate the view to look onto the unfolded sheet and select Front only and click OK.



In the drawing space, IRONCAD will have drawn both the folded view and the unfolded view of the part. On the folded part, the dimensions positioning the cluster of holes will have been transferred from the model. These can be repositioned by dragging them.

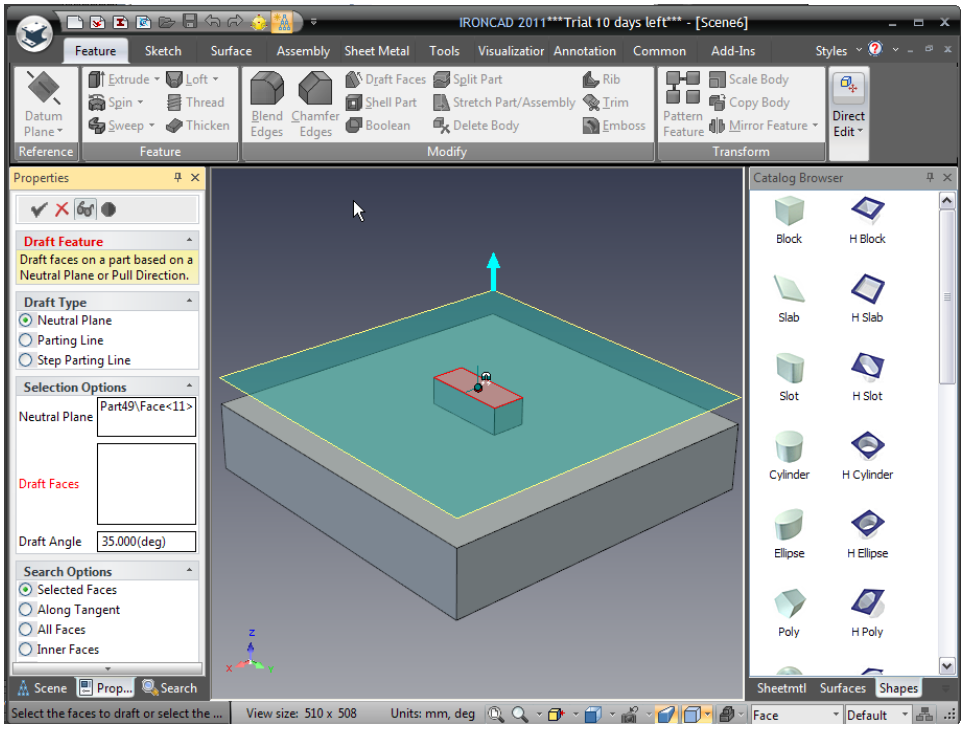
On the flat pattern the bend lines will have been drawn along with instruction on what to do at those bends along with center marks for the standard punch tool features. The drawing can be dimensioned and annotated using the standard tools provided.



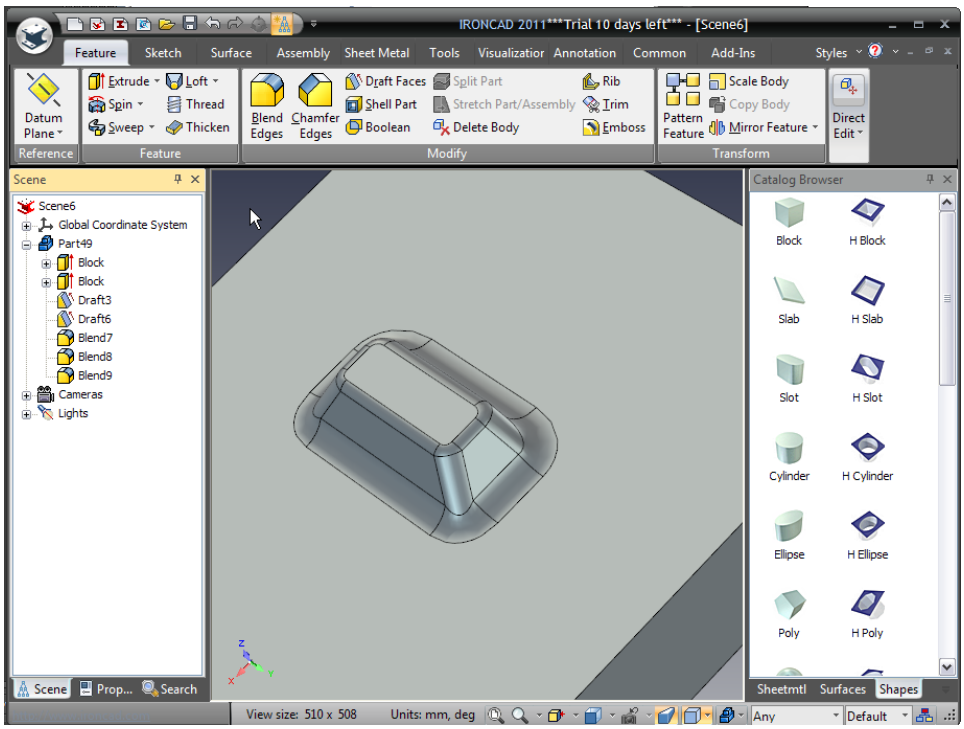
7.14. Defining custom punch shapes

Sheet metal projects do not always use the standard punch shapes found in the IRONCAD catalog so IRONCAD provides a function to produce custom punch forms. Open a new scene and drop a Block onto the scene. Resize the block to be 100mm in length and Width and 20mm in height. Add onto the first block an additional block 8mm high with length 25 and width 10mm centered on the first block as shown in the following image.

Add draft to the side walls by selecting the Draft Faces tool from the Features tab of the Windows Ribbon bar. Indicate the top face of the second block ensuring the arrow points away from the block. Set front and two side faces to 35 degrees and the back face to 10



Add 3.5mm blends to the side edges, 5mm to the interface between the two blocks and a 2mm blend on the top face. The part should look similar to the view below.



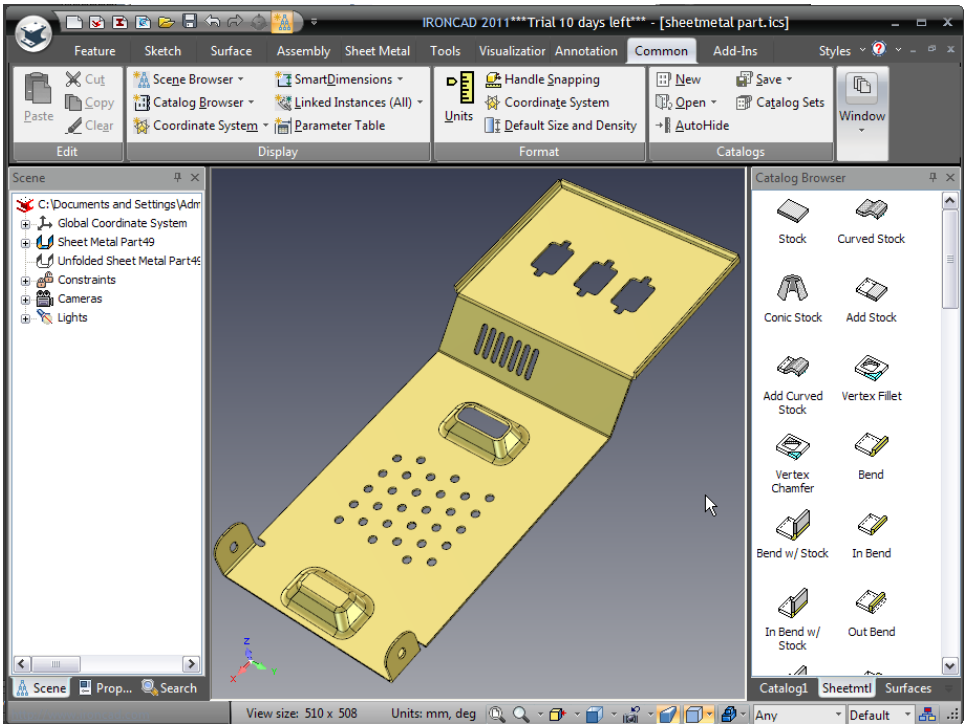
Open the Sheet Metal tab and select the Forming Tool option. For the Stopping Face indicate the top face of the first block. This in essence is the face that will stop against the part and click OK. IRONCAD will change the front face to cyan and add an entry to the part tree in the scene browser.

Open the Common tab on the Windows Ribbon bar and select 'New' from the Catalogs panel. IRONCAD will open a new catalog.

Click on the forming tool and drag it to the new catalog and click twice on the name in the catalog and change its name to Forming Punch.

In the scene browser, remove the 2mm blend from the top face of the punch. Next, right-click on the Forming Tool entry in the tree and select 'Edit Tool option'. If the word Face is not already highlighted in red, click in the 'Faces to Remove' panel and select the top face of the second block and click OK.
IRONCAD will highlight this face in yellow. Drag this tool onto the catalog and rename it 'Cutting Punch'.

Minimize or close the scene defining the punches and open the sheet metal part. Rotate the view to look on the back of the part. From the new catalog, drag the Forming Punch onto the part below the cluster of holes and the Cutting Punch above the cluster of the holes. IRONCAD will create a raised section with a closed top face at the bottom of the part and a similar form above the cluster, however this one will have a hole cut through. After changing the color of the part the scene should resemble the next image.



This completes the section on IRONCAD sheet metal design. In this section the topics covered include:

- Adding and editing bends and bend reliefs
- Use of standard punch shapes
- Adding corner radii
- Use of catalogs for commonly used items
- The miter command
- Unfolding and drawing creation
- Creating a custom punch

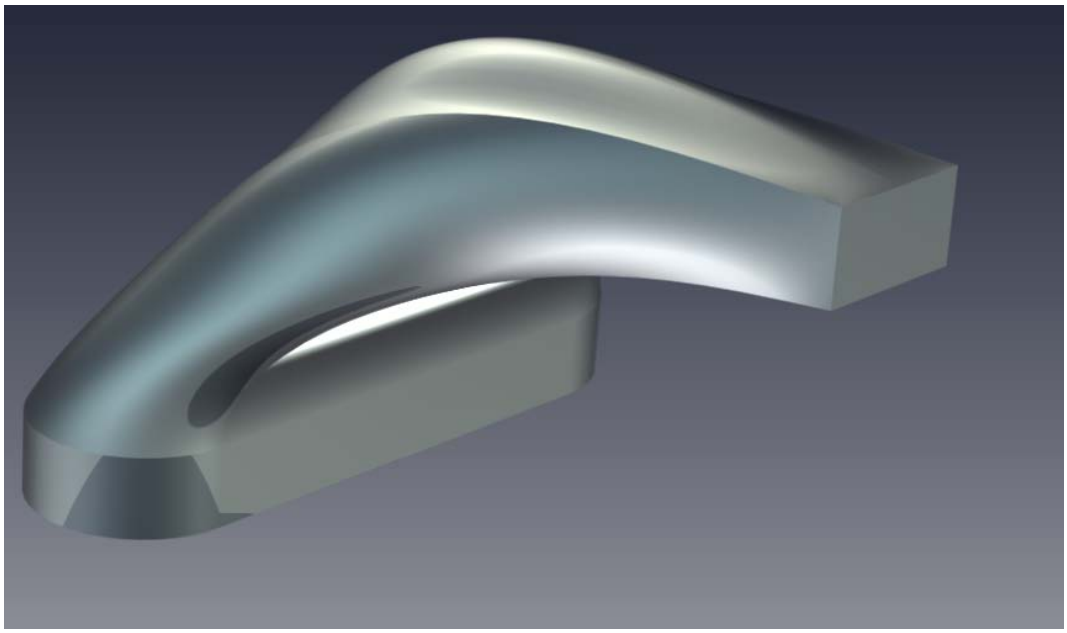
The next chapter will introduce the basics of IRONCAD's surface modeling capabilities.

8. IRONCAD SURFACE MODELING

This chapter introduces some of IRONCAD's Surface Modeling functionality and 3D curve creation capabilities. The chapter covers the use of surface modeling to create a Solid Model of a faucet, the topics covered are:

- Creation of 3D Curves from points,
- Creating Surfaces from Solids
- Create Sweep, Mesh, Edge and Ruled Surfaces

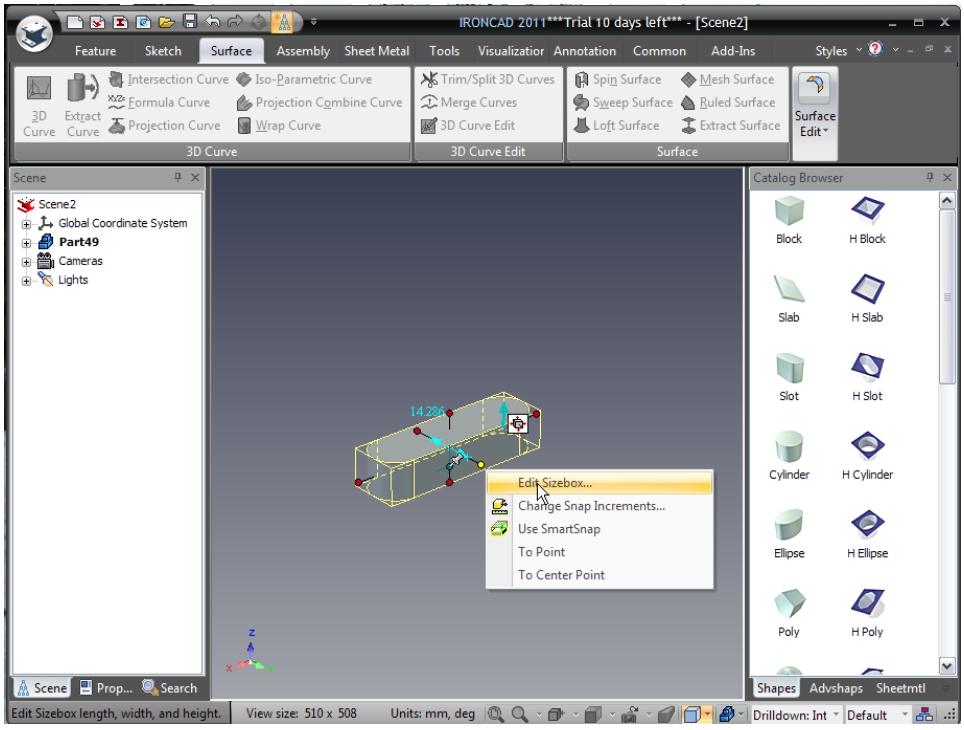
The Ruled Surface section closes the surface model into a fully bounded model that IRONCAD will recognize as a solid part.
At the end of the section the part should resemble the following:



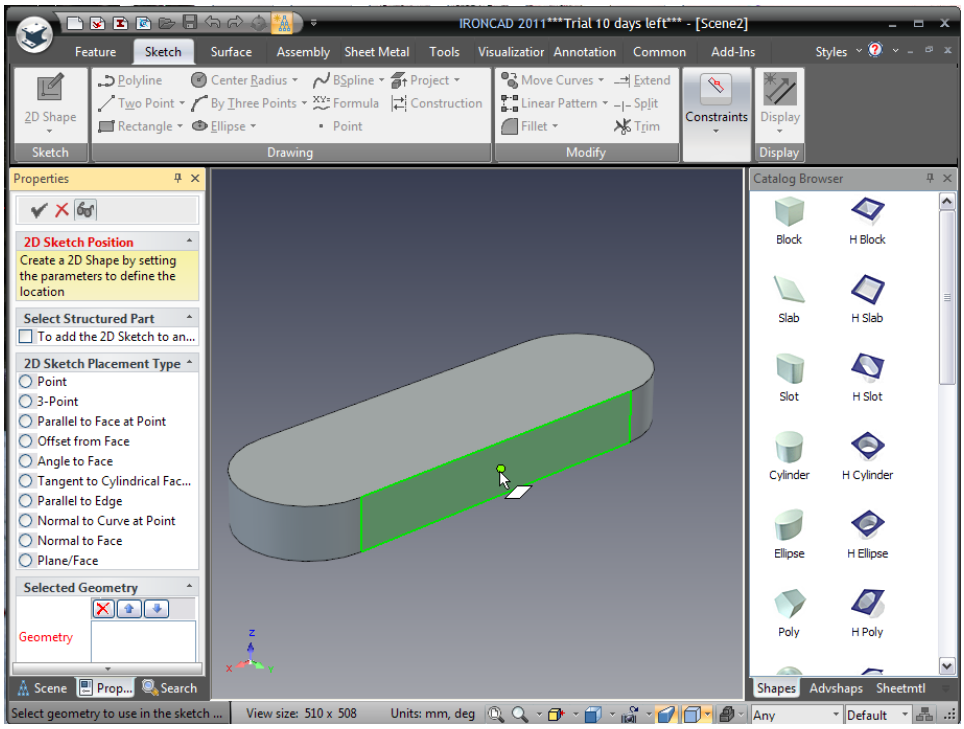
8.1. Creating 3D Curves

From the IRONCAD home button select File, New and select Scene. In the Metric tab, select the template labeled Gray and click OK.

From the Shapes catalog, drag a Slot onto the scene.

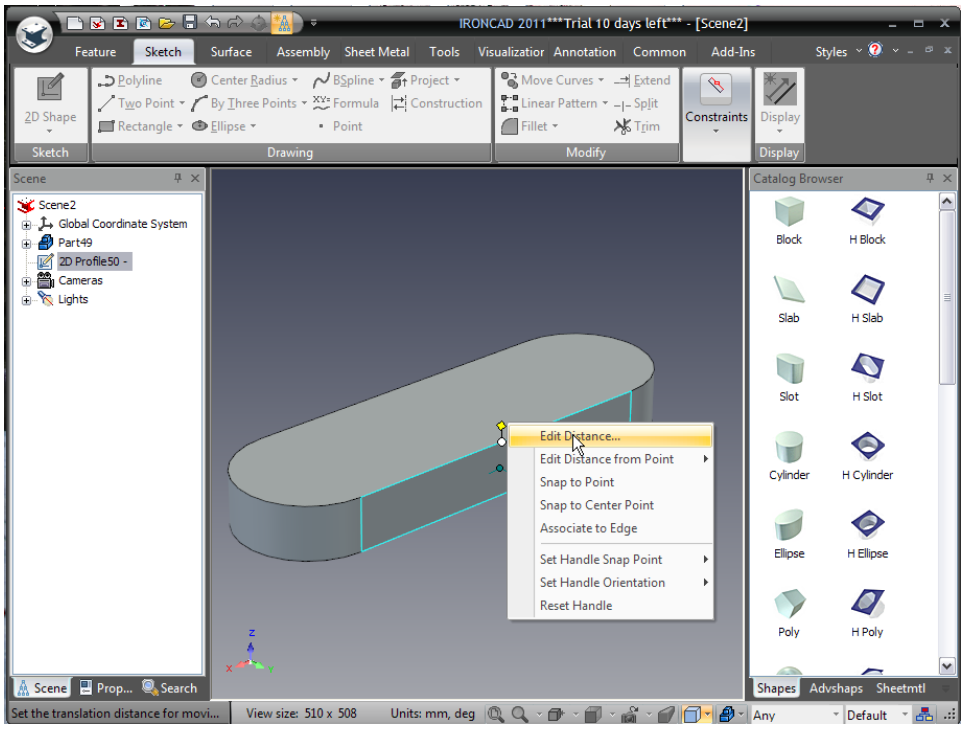


Select the slot at the IntelliShape editing level and resize setting the Length to 130, the Width to 40, the Height to 15, and then click OK. Open the Sketch tab of the Windows Ribbon bar and select the 2D Shape icon and using SmartSnap feedback indicate the center of the rectangular front face of the Slot part as shown.

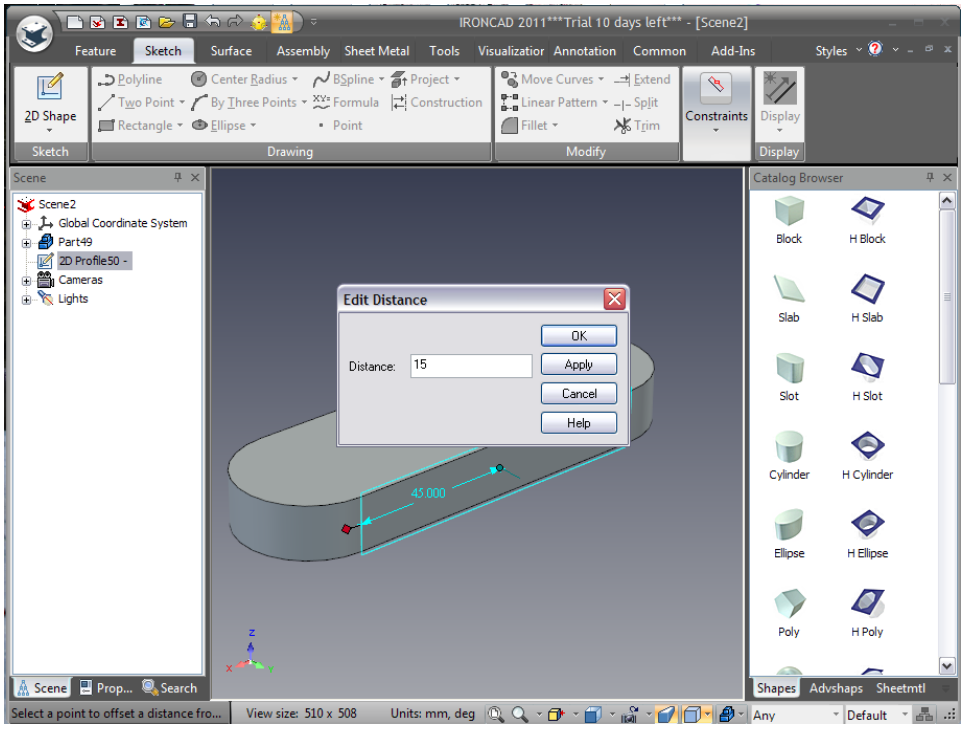


IRONCAD will re-orientate the view to look directly onto the face. In the Sketch tab of the Windows Ribbon bar, select Project from the pull down by Project Constraints and select the front rectangular face. Click the Finish icon to accept the shape and return to the scene.

Using the handles resize the rectangle of the 2D shape to 17mm x 30mm by touching the top curve and right-clicking on the handle, select Edit Distance and enter 1mm as shown below (use -1 if the handle points into the model).

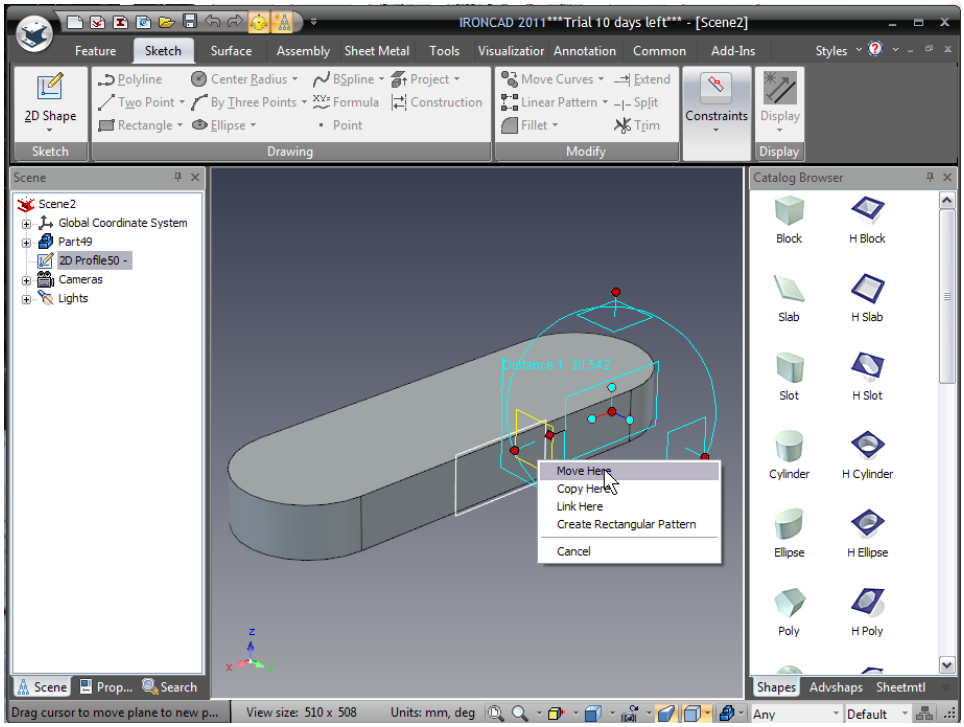


Repeat the process for the lower edge of the rectangle. To size the width, touch the left hand short side and right-click on the handle selecting Edit Distance from Point and indicate the face center as the reference point, set the distance to 15mm.

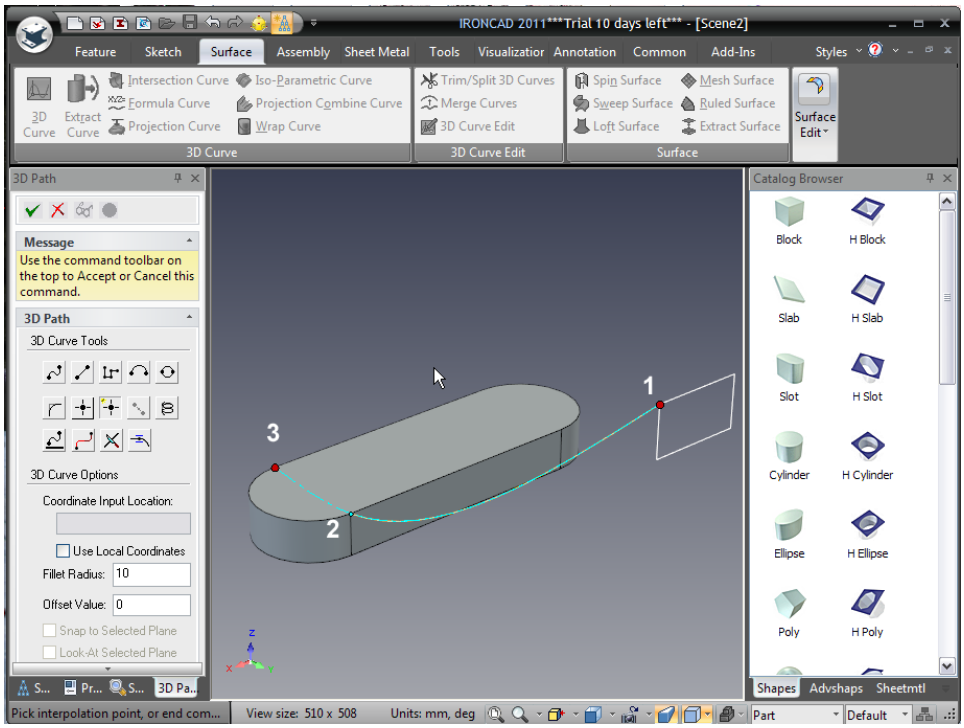


Repeat this for the other short side.

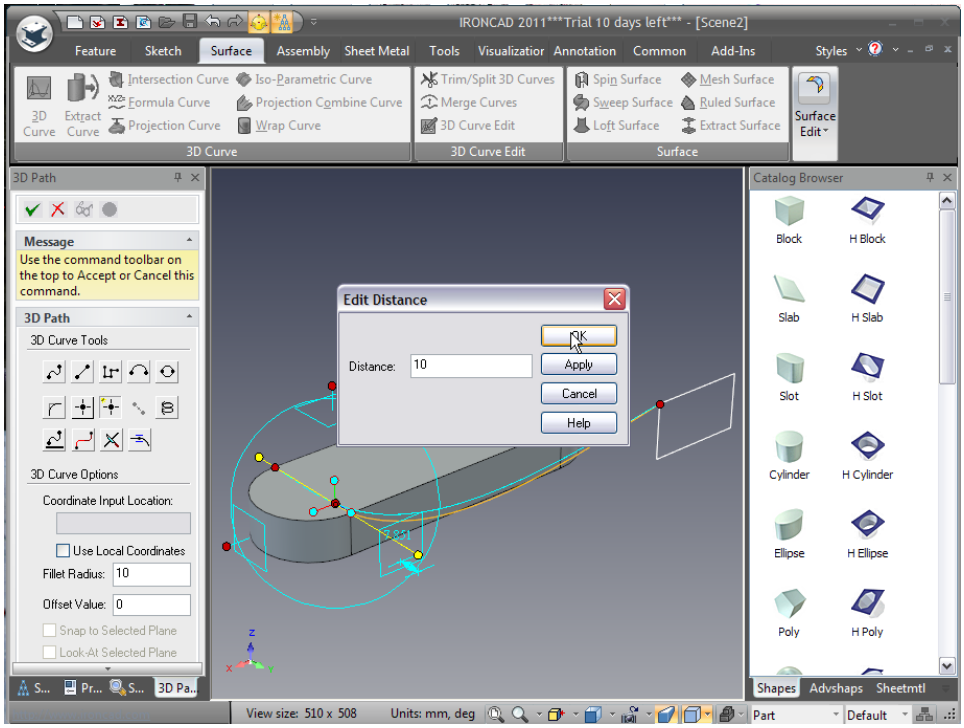
Activate the TriBall and move the 2D profile out from the face 100mm and up 65mm.



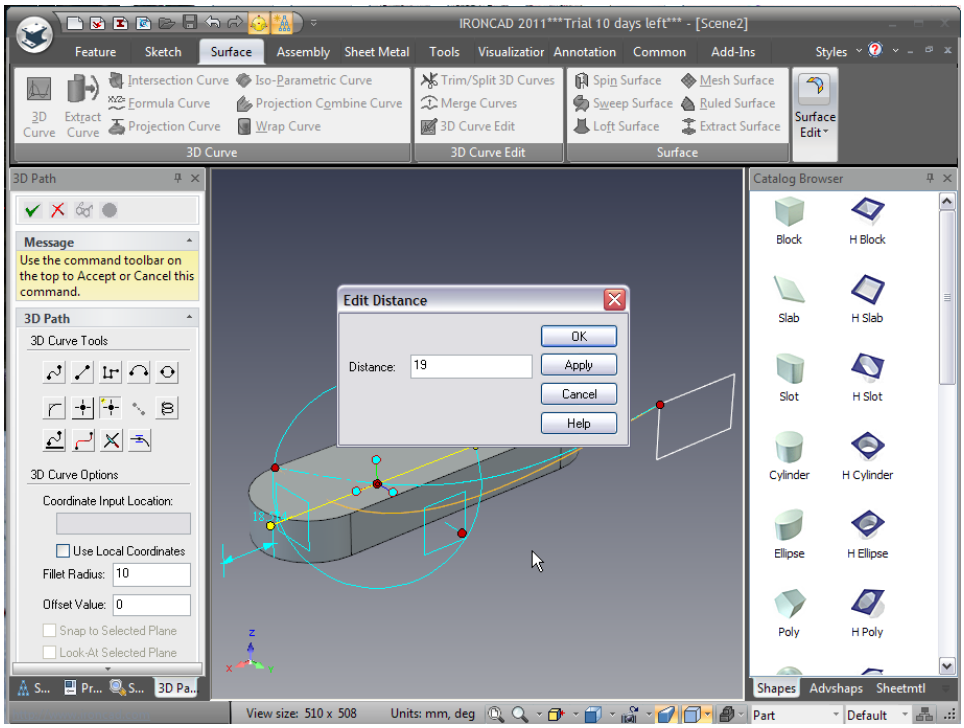
Select the Surface tab from the Windows Ribbon bar and click on the '3D Curve' icon tool and pick the three points shown below ensuring the three points are 'snapped' onto the end points of the model entities (the points will highlight green when selected).



Press Esc to stop creating 3D points for the curve. Next select the second point created and activate the TriBall. Move the point back 10mm towards the third point of the 3D curve, as shown.



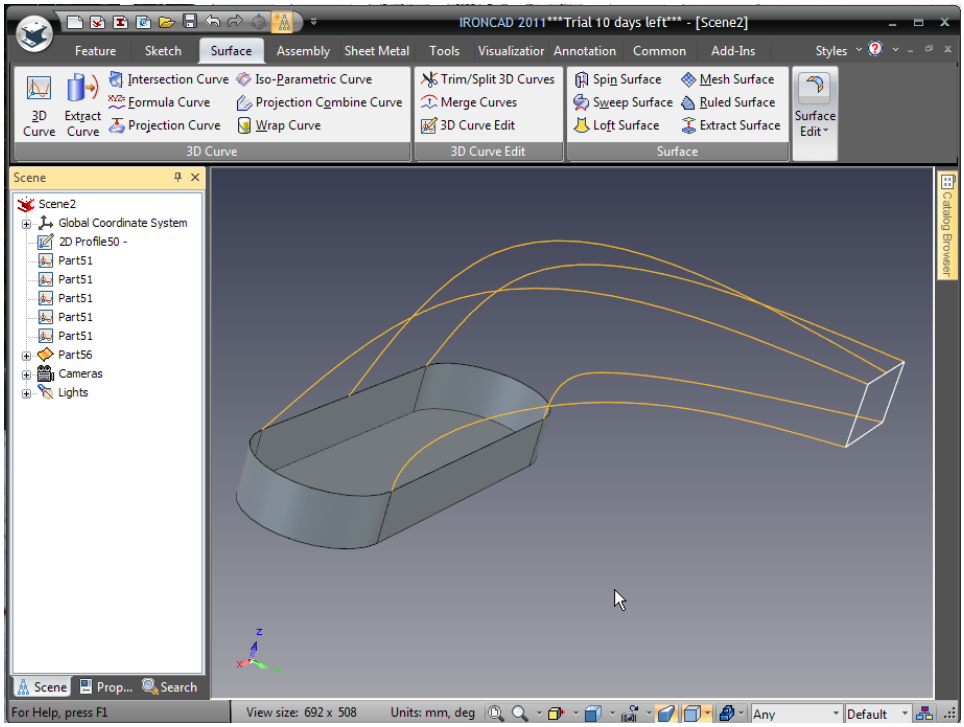
Then move it to the right (towards the middle of the Slot) 19mm.



Finally move the point up 44mm although the exact location isn't critical and can be adjusted to vary the shape of the faucet. Click Apply to finish the 3D Curve.

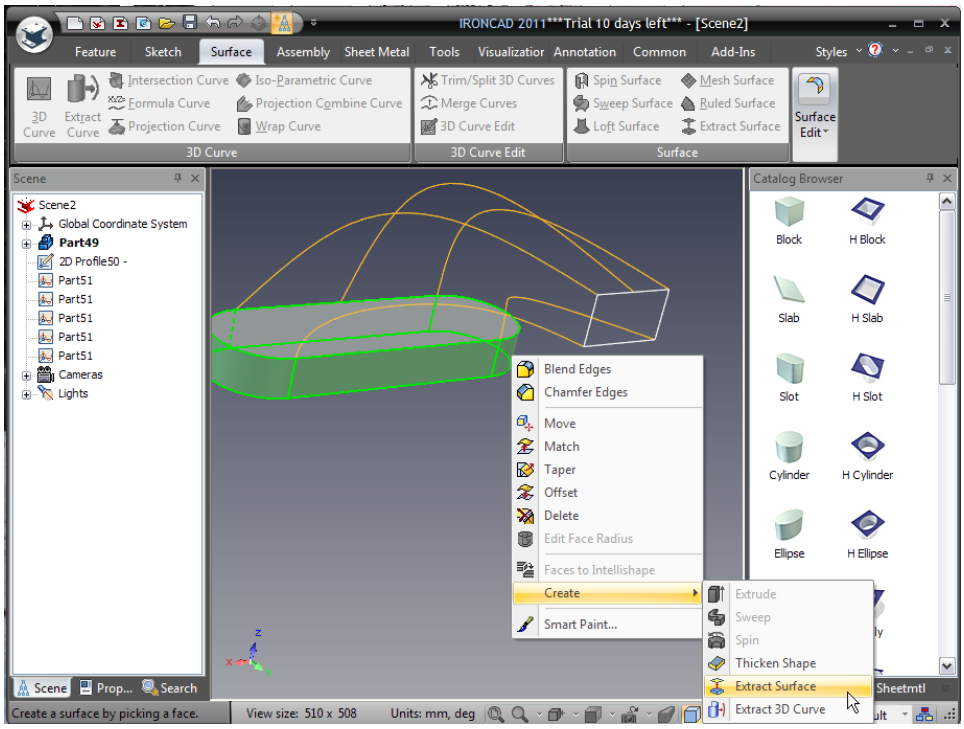
Repeat the 3D Curve to connect the other 3 points on the Slot to the other corners of the rectangle. It's a good idea to use the TriBall to mirror the curves from one side to the other to insure symmetry. Also, create a curve down the middle of the faucet, using the same TriBall technique, to give it a more raised surface in the middle of the top face.

The Curves should look similar to the following image when you are finished.



8.2. Create Surfaces from Solids

Using the selection filter set the selection type to ‘faces’. Next select all the faces except the top face on the original solid slot component using ‘Shift+Select’.



Right-click anywhere on the selected faces and choose the ‘Create Surface Shape’ this will provide a surface model “starting point” to complete the rest of the model. Delete the original slot solid model as it is no longer required.

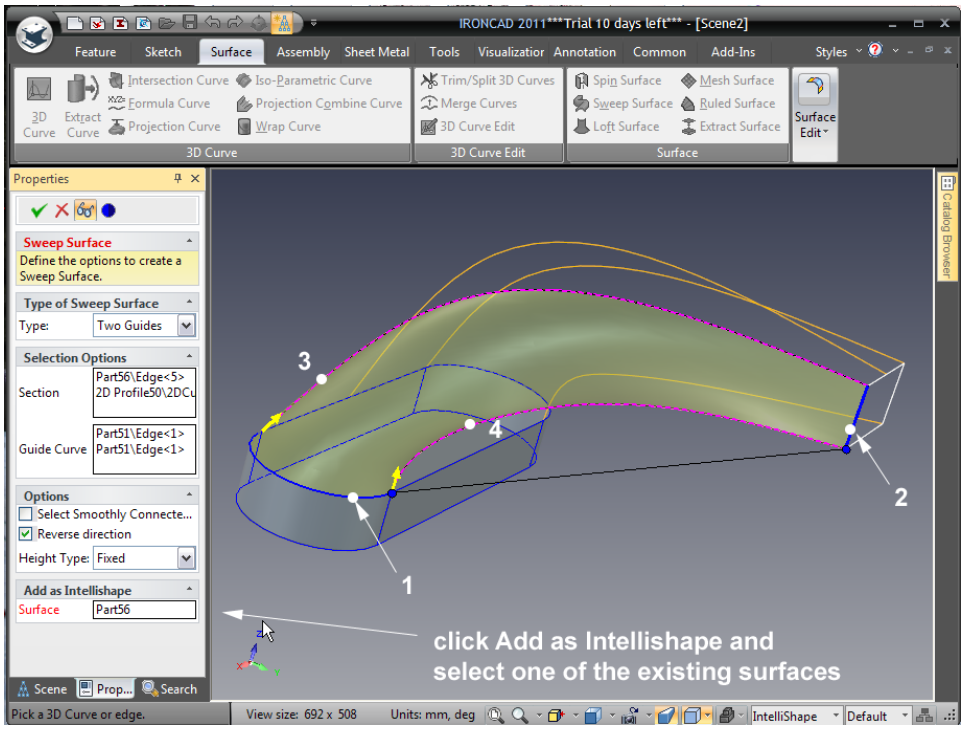
8.3. Creating Surfaces

8.3.1. The Sweep Surface

The Sweep Surface provides many options for driving (or sweeping) cross sections along “rails” to create unique shapes. Open the Surface tab of the Windows Ribbon bar and click the Sweep Surface icon in the Surface panel.
To create the side surfaces requires IRONCAD to go from a semi-circular form to a straight edge along two controlling edges. In the IRONCAD Property Panel set the Type of Sweeping Surface to Two Guides.
Select the upper semi-circular edge of slot shape towards one of the curves to be used as a guide curve (1) followed by the corresponding curve of the end rectangle, again towards the same guide curve (2). IRONCAD will highlight

these curves in blue and place a blue dot at the end of each curve. These dots should present themselves on the same guide curve. If they don't lie on the same guide curve right-click on the last entry in the section panel, delete it and reselect the curve closer to the guide curve.

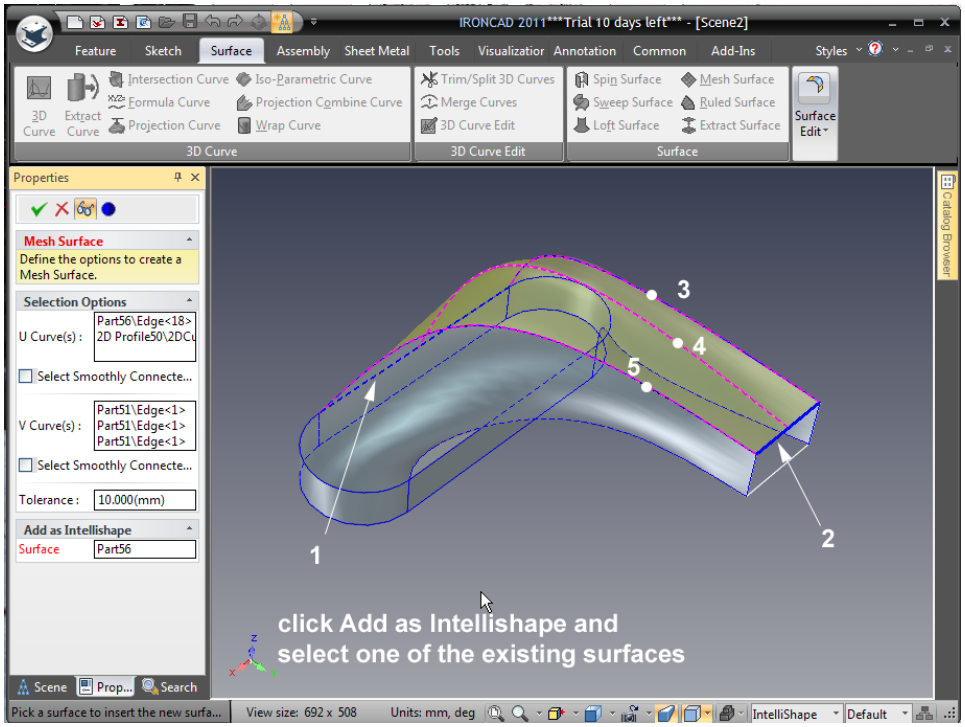
With the two section curves selected, IRONCAD will automatically enter the next two selections into the Guide Curve box. Select one of the 3D curves defining the guide and then the other. IRONCAD highlights the guide curves in magenta and places an arrow at the start point for the guide. Both guide curves should start at the first section curve selected (the semi-circular curve) and follow the same direction. When selecting a guide curve if the direction is not correct click in the Reverse direction check box and IRONCAD will reverse the direction of the curve.



Click in the 'Add as IntelliShape' box and select one of the surfaces extracted from the slot. This adds the new surface to the existing surface model extracted from the original slot component. This manages the scene contents as a single model not as a collection of surfaces. If the model becomes fully bounded i.e. there are no holes, IRONCAD will recognize the model as a solid. Repeat the Sweep process for the other side of the part.

8.3.2. Mesh Surface

A Mesh Surface, as its name suggests creates a surface from a network (or mesh) of touching curves running in two directions. The curves forming the surface are identified in two groups namely ‘U Curves’ and ‘V Curves’. The terminology is coming from long standing surface modeling terms.
To create the upper main body surface, select the Mesh Surface icon. Referring to the image below select the long, top edge of the open slot shape (1) followed by the top edge of the 2D rectangle (2). In the IRONCAD Property Panel click in the’ V Curve(s)’ box and select the three 3D curves 3, 4, and 5. IRONCAD will display the U Curves in bright blue and the V Curves in magenta and provide a shaded preview.

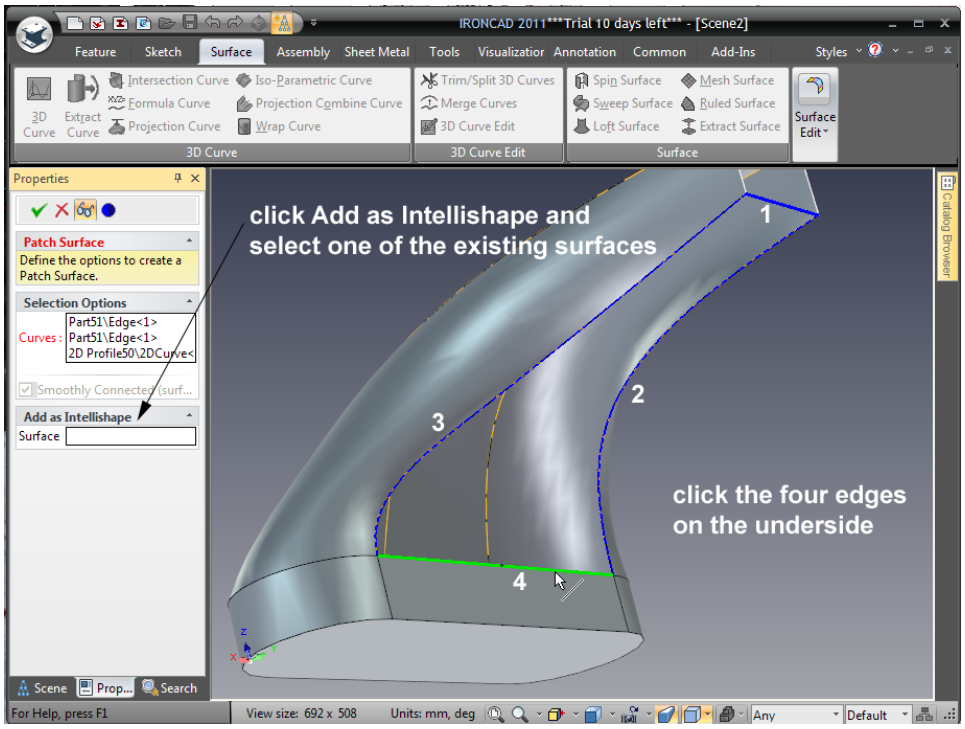


Finish the surface by clicking in the ‘Add as IntelliShape’ box and selecting an existing surface from the model and then clicking OK.

8.3.3. Patch Surface

The Patch Surface is used to create a “patch” connection between 3 or more edges. Click the Patch Surface icon found in the Surface Edit panel of the Windows Ribbon bar. Rotating the model to view the open side, select the four edges bounding the hole.

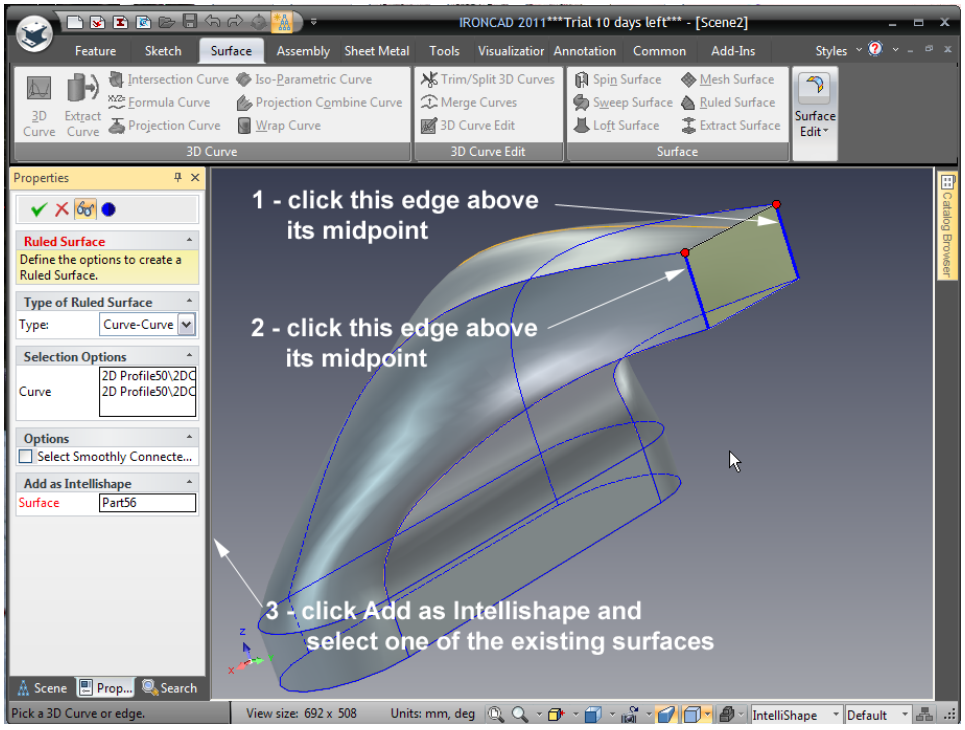
Click in the box for Add IntelliShape and select one of the existing surfaces followed by clicking on OK to accept the surface.



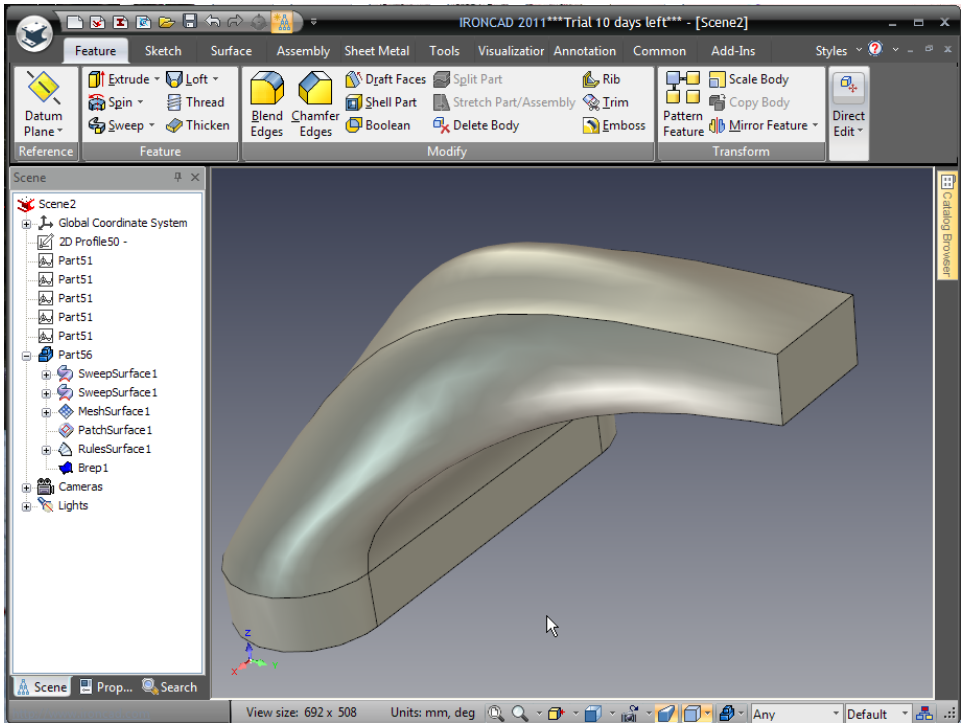
8.3.4. Ruled Surface

A ruled surface can be considered as a ribbon being pulled taught between two curves. The sense in which the curves are selected is important otherwise the surface will twist i.e. for two straight curves a twisted surface would end up looking like a bow-tie.

A Ruled Surface is usually used to create a straight transition between two edges. Click the Ruled Surface icon and select one of the edges of the front rectangle. IRONCAD will highlight the edge and draw a red dot at one end closest to the mouse pick. Select the opposing edge of the rectangle in a similar position. IRONCAD will highlight the curve and draw a red dot to indicate the start point for the ruled surface. If the red dot appears at the opposite end of the curve IRONCAD will draw a bow-tie shaped surface highlight simply click on the curve again to deselect it and reselect the curve closer to the desired end.



Click in the 'Add as IntelliShape' box and then select any of the existing surfaces then accept the surface by clicking OK.



The model is now completely closed and automatically converts to a solid model. Experiment by adding more features to finish the design or by editing a surface such as the top mesh surface. Right-clicking on the central, long curve and selecting edit curve will provide access to the points defining the curve.

This chapter introduced some of IRONCAD's Surface Modeling functionality and 3D curve creation. The chapter covered the use of surface modeling to add a degree of control and form to a simple component resulting in the creation of an editable Solid Model of a faucet, the topics covered are:

- Creation of 3D Curves from points
- Creating Surfaces from Solids
- Create Sweep, Mesh, Edge and Ruled Surfaces to curves and surface edges

The next chapter will provide insight into the photo-realistic rendering aspects of IRONCAD.

9. PHOTOREALISTIC RENDERING

Communicating a design with finished 3D models and detail drawings is only part of the process for IRONCAD users.
Prior to finishing a design there are many steps in the communication chain to expose the design to a wider audience to enable this audience to become stakeholders in the design. Photorealistic rendering opens up other opportunities such as providing early product visualizations for marketing materials, animations to help with production processes or even just having some fun with the design.
IRONCAD is provided with integrated photorealistic rendering and animation tools and just like the rest of IRONCAD design tools these can be simply dragged from the catalog onto the part enabling very rapid results.
This section will:

- Introduce simple drag and drop rendering techniques
- Creating screen based images for use with standard screen capture tools
- Creation of high quality renderings for everyday use to large format poster style renderings. Animation will be covered in a separate section

This section will be based on the design created in the Quick Start section of the manual.

9.1. Adding realism to the screen display

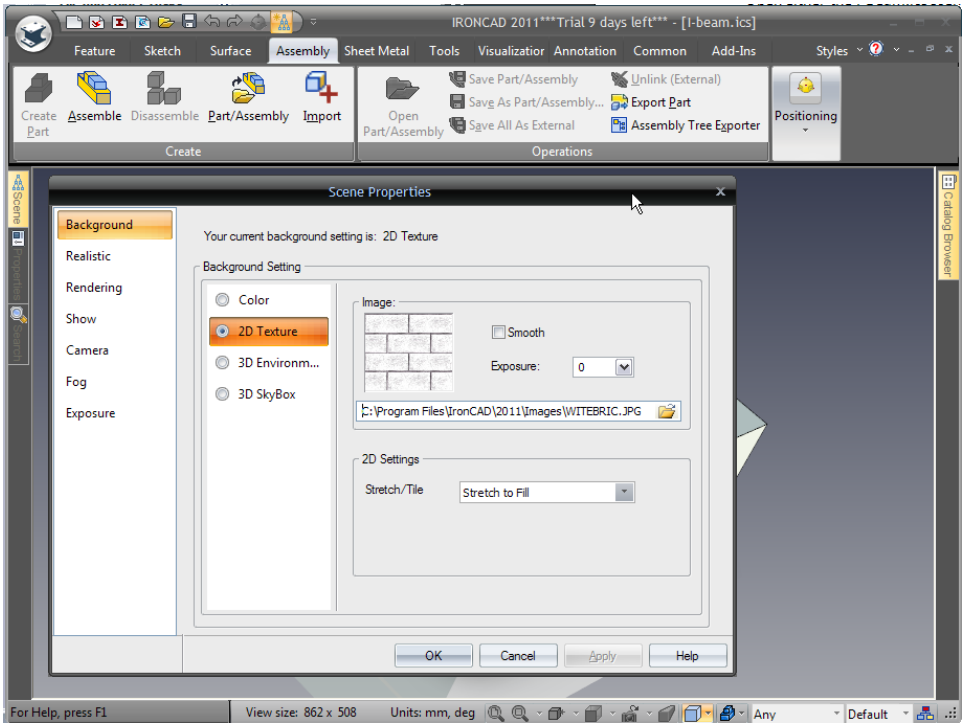
IRONCAD was one of the first products to use a clean working environment, reducing clutter on the screen to help every user to concentrate on the job at hand and not navigate around a multitude of menus. The display is so clean that screen captures often suffice for discussion documents or quick idea communications. IRONCAD has added a number of features to lift that quality to a higher level. Not only does this add a touch of realism, it adds professionalism to these quick captures helping to instill a culture of quality.

All aspects of the screen display can be set. This section will introduce changes to the scene background and cover adding shadows and reflections to the scene background and the part display.

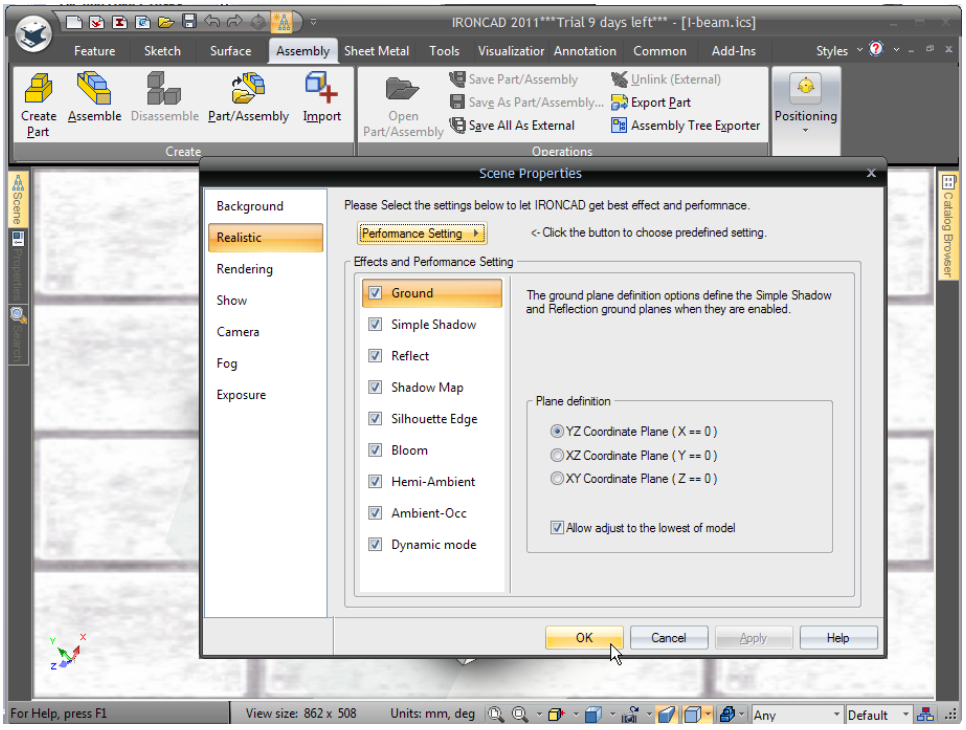
9.1.1. The Scene Background

The IRONCAD scene background is based on an image either repeated or not. For example if the object being designed can be shown in the context of an existing photograph or product screenshot – this can be used as the scene background.

Open either the I-Beam.ics scene file created in the quick start guide or the I-Beam.ics file provided in the tutorials folder. The I-beam.ics file provided has a number of predefined surface textures added so this may be preferable.
Right-click on the scene background and on the resulting drop down menu select Background. IRONCAD will display the Scene Properties dialog box open on the Background tab.

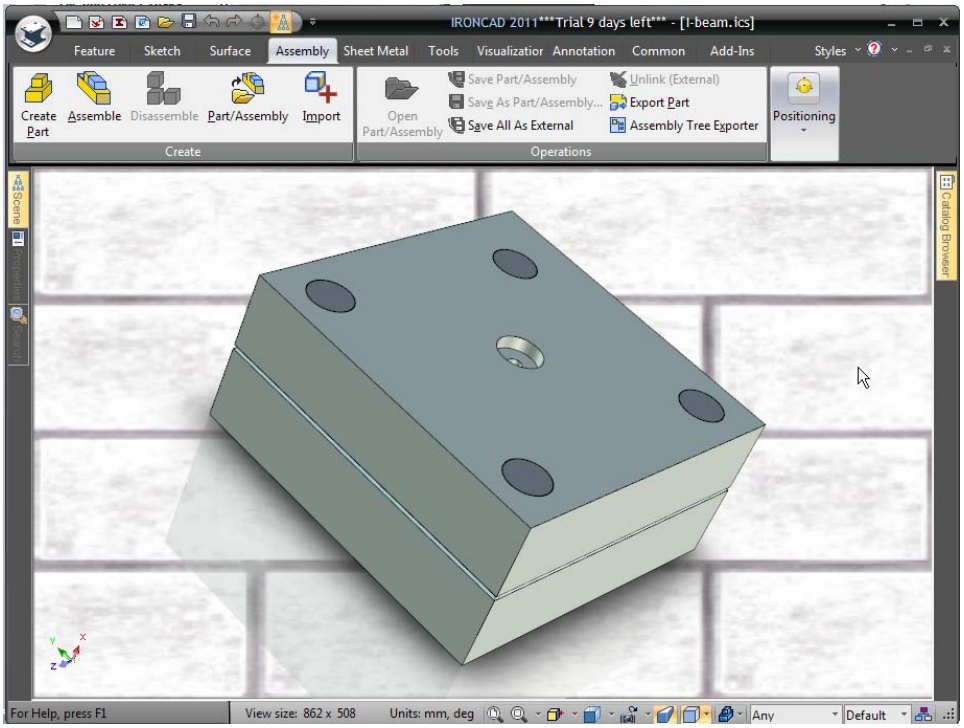


Click on the 2D Texture radio button, and browse for Witebric.jpg (IRONCAD will open the correct folder, however if it doesn't browse for the IRONCAD images folder created at install).
Ensure the option 'Stretch to Fit' is active.
Now select the Realistic tab and set the level of realism to best as shown in the following image.



For the shadow and reflection maps check the XYZ icon in the bottom left corner of the IRONCAD scene and set the appropriate position for the shadow map.

IRONCAD will change the background displaying a brick surface, adding shadow and reflection below the part. The quality of the display is determined by the graphics card and screen resolution of the computer being used. Standard window capture tools can be employed to provide a very quick output from the screen rendered image.



9.2. Basic Rendering

Using the scene settings of the previous section, right-click on the Top Plate component (either on the model or in the Scene Browser) and from the resulting drop down menu select 'Hide Selected'.

On the Windows Ribbon bar, open the Visualization tab, click the Render Now icon (or press CTRL-R). IRONCAD will open a window and generate a rendered image showing the textures and finishes currently applied to the part. The I-beam itself has a plastic spark like finish, the tooling plate a dull sheen and a higher level of reflectance on the guide pins. The image will also be a little washed out due to the predominantly white background of the scene. Once the rendering has completed, right-click in the render window and set the Exposure to approximately -0.3, and the Gamma to 0.7, and click apply. The contrast and lighting of the rendered image will be dynamically changed without having to reprocess. To save the image right-click on the image and select Save As and choose a favored image format.

9.2.1. Scene Exposure

Altering the Exposure and Gamma settings provides dynamic image modification tools, however a similar effect can be provided by changing the amount of light generated by the scene background, normally termed exposure.

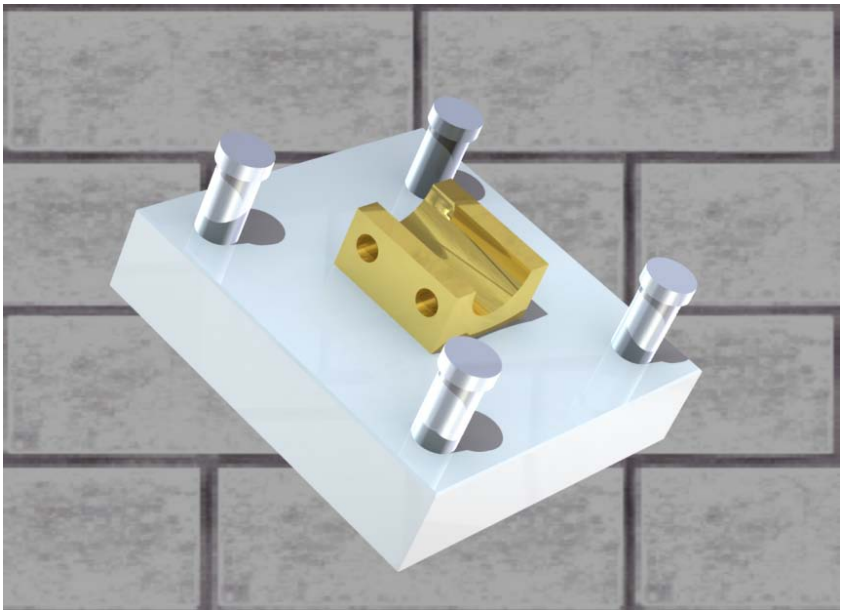
Right-click on the scene background, and select background. In the Image panel set the exposure to -0.6. The render output will now be less washed out.

9.3. Applying a Surface from a catalog

Open the Surfaces catalog, and just like dragging an IntelliShape from a catalog, locate the Gold surface and drag and drop it onto the I-Beam component. Depending on the graphics card used, the surface color of the I-Beam will change to a pale yellow color. Next hit the 'Render Now' button again, the I-Beam will change to a dullish gold appearance.

9.4. Modifying a surface texture

The catalog surfaces provide a great starting point for rendering, being perfect for some instances and not for others. For example, gold is usually thought of as a shiny reflective material so to polish the gold applied to the I-Beam, right-click on the I-Beam component and select 'SmartPaint' from the drop down menu. Navigate to the Reflection tab and set the reflection intensity to approximately 89 and the Fresnel value to 0.3. Now hit the Render Now icon, or press CTRL-R.



The gold surface should now start to look a little more like gold and reflect the background. However you will see one important point in rendering surfaces come to life if they have shape and something to reflect. The surfaces that can reflect the background or other features look nice and shiny. The flat ones with nothing to reflect look, well, flat.

To add a little ‘life’ to the I-Beam, right-click on the I-Beam, select ‘SmartPaint’ and open the Finish tab.
Select the lower left red sphere icon in the Preferred Finishes section, in the Diffuse BRDF drop down select ‘Ashikhmin’ and set the shine value to 1. In the Specular BRDF section, again use ‘Ashikhmin’ setting the ‘Aniso U’ and ‘Aniso V’ values to 0.2 leaving the ‘Rotate’ value to its currently set value.
(BRDF is short for Bidirectional Reflectance Distribution Function and is a method to calculate how light is reflected from an opaque surface.)
Render the scene by pressing CTRL-R the resulting image will show the gold component to be a lot more lifelike. If you are not convinced add some blends to the crisp edges of the I-Beam and re render the scene.

9.5. Saving a Surface Texture to a catalog

To use the surface textures again, simply add them to the Surfaces catalog or a new catalog.
At the left hand end of the Visualization Windows menu tab is the Fill Eye Dropper, select the Eye Dropper, click on the gold I-Beam and left click on the catalog. IRONCAD will create an icon representing the surface texture. To set a name, click on the icon twice (not a double click) to highlight the icons current name. A new name can now be entered.

9.6. Setting a material property with a surface texture

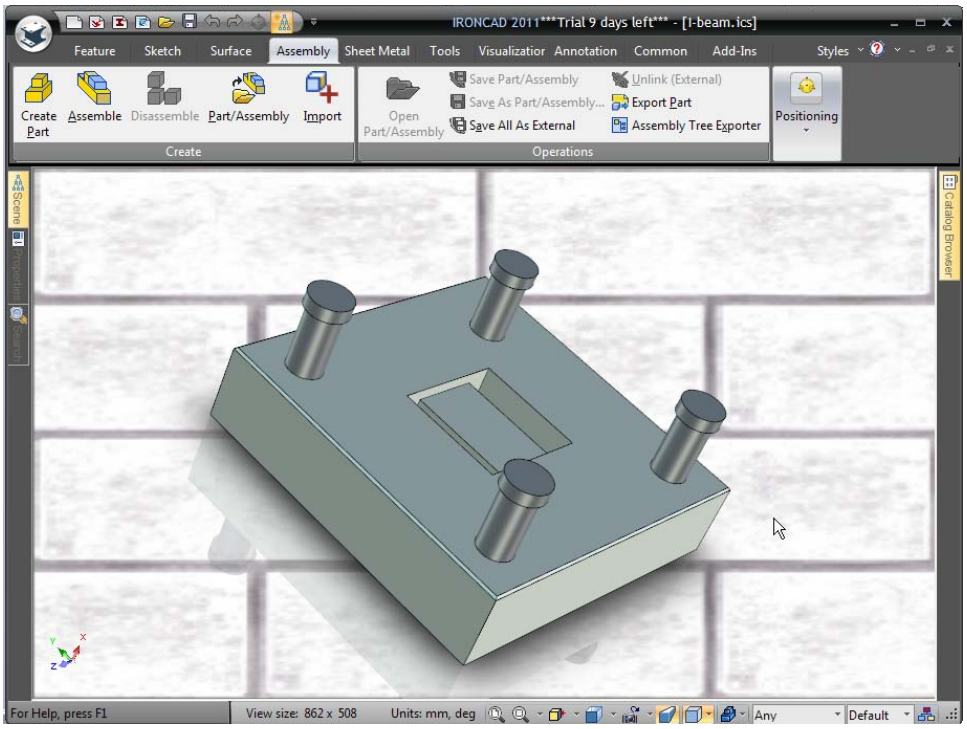
The surface texture may be aligned to the physical properties of a material and not just to its surface treatment. To those ends, setting the material of a part can also be carried over to a catalog surface texture.
Right-click the I-Beam component and from the drop down menu select Part Properties. Open the Material tab and in the Mass Density box enter 19300 – if the units are kg/m^3 or 19.3 if the units are g/cm^3. Enter Gold in the Material box, and click OK.
Repeating exactly the same procedure to add the surface texture to the catalog, click on the Fill Eye Dropper, and select the gold I-Beam and then this time use the right mouse button to drop the surface texture onto the catalog. IRONCAD will respond by prompting SmartPaint or SmartPaint and Material. This time select SmartPaint and Material. Each time this surface texture is added to a part its material will be changed to gold as well as its finish.

9.7. Changing the finish of individual surfaces

Parts may have different finishes on individual faces. So far the rendering section has been concerned with all the faces for a part. Changing the surface texture for a single or group of surfaces on a part is equally straightforward.

Referring back to the current model file I-Beam.ics, right-click on the I-Beam part and from the resulting menu select Hide Selected. The display will show the Bottom Plate and the Guide Pins.

Arrange the view so that the cavity is being presented as shown below and do a quick render to see what the cavity looks like the following image.



On the lower toolbar, click the Save Camera icon so that this view can be returned to quickly.

Zoom into the cavity and click onto one of the faces of the cavity until it is highlighted in green i.e. in face mode. When highlighted in green, right-click and select 'SmartPaint' from the drop down menu.

On the Finish tab select the lower left red sphere icon in the Preferred Finishes section, in the Diffuse BRDF drop down select 'Ashikhmin' and set the shine value to 1. In the Specular BRDF section, again use 'Ashikhmin' setting the 'Aniso U' and 'Aniso V' values to 0.2 leaving the 'Rotate' value to its currently set value. Under the Reflection tab set the Reflection intensity to 80 and the Fresnel Amount to 0 and click ok.

With the surface still highlighted in green, select the Fill Eye Dropper and click on the surface. IRONCAD will change the Fill Eye Dropper to the Apply Eye Dropper. Touch on all the surfaces comprising the cavity being careful to watch the surface highlight before clicking the mouse button. Press the Esc key to cancel the operation when completed. On the camera controls tool panel, click 'Restore Camera' to reset the view to the previously saved view and render the scene.
The cavity will appear polished. To see this even more add a 3mm blend to all the internal cavity edges and render the scene.

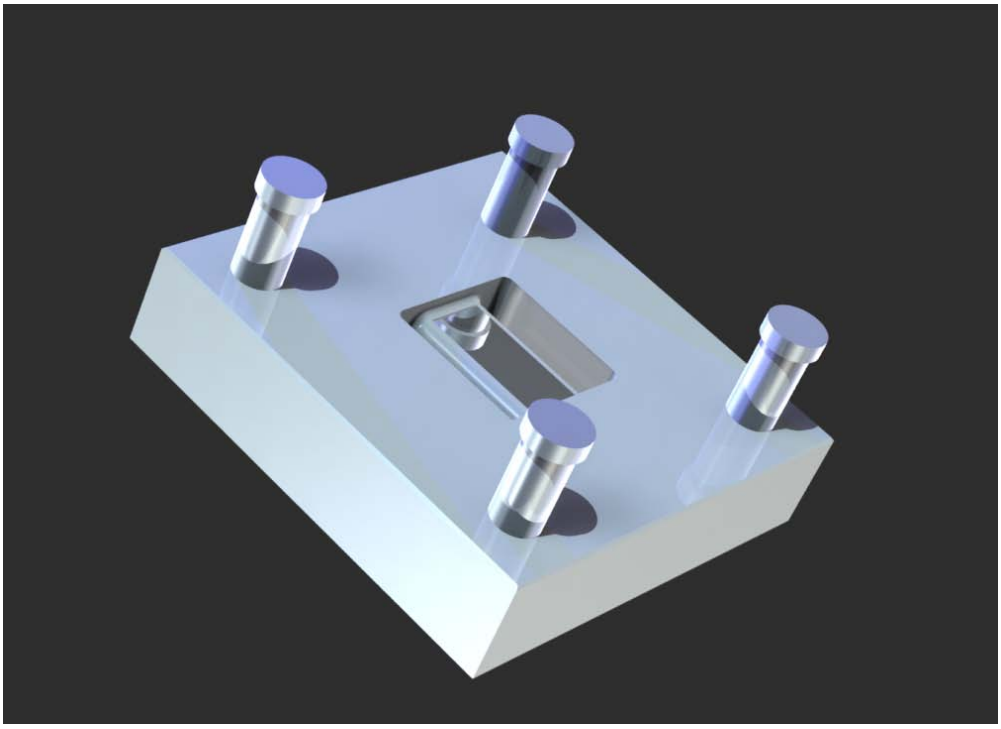
9.8. Adding more realism with HDRI background lighting

The scene so far has been lit with the standard scene lights and with the background image. This limits the realism that can be achieved. IRONCAD can use High Dynamic Range Images (HDRI). These images convey not just the color of each pixel, but also the light intensity. Using HDRI enables the scene to be rendered more realistically.
The first part of this final process is to change the background. IRONCAD is provided with a range of HDRI files. To use them, right-click on the scene background and select the Background tab. Next, set the '3D Environ. . .' radio button. In the Image panel, browse to the Images folder and open a folder called 'EnvironmentImages'. Within that folder select 'StudioEnvironments' and select 'ThreeLightSoft.hdr' and click Open. Back in the Image panel check the Smooth option and set the Exposure to -0.6 leaving everything else untouched. Render the scene to see the progress being made.

9.9. Adding discrete lights

The IRONCAD scene can be lit by a multitude of light types all delivering different results. Lights can be colored, can be different intensities and can have properties set controlling the fall off and the type of shadows they cast. Arguably the most popular is the Spot Light. However in this example the Point Source light will be used. A Spot Light creates a directional light whereas a point source light creates a light spreading in all directions.
To create a point source light first click on Insert Light on the Visualization tab, and then click on the center of one of the Guide Pins. Choose Point Light from the resulting menu.
IRONCAD will report that lights are currently not shown and ask if you want to make them visible for editing. For this example say no. The resulting displayed scene will be brightened.
In the Scene Browser expand the Lights group and right-click on the 'PtLight' entry and select Light Properties... from the menu.

When the light is selected IRONCAD will display a ghost image of the light.
Furthermore, when selected the light can be positioned using the TriBall even if it's a ghosted image.
Click on the Light tab and set the intensity to 0.4, and set its color to the bright blue on the bottom left corner of the color panel.
Ensure Cast Shadows is checked, and click on Advanced Settings and set the option for Soft Ray Traced setting the radius to 4 (in essence this is the size of the source creating the shadow, the larger the source the softer the shadow).
Finally set the position of the light under the Position tab to X:50, Y:200 and Z:50. As mentioned previously the light can be positioned by the TriBall, however these values help to show the impact of the point light source. When the scene is rendered the effect of the blue light can be clearly seen as well as the soft shadows cast by this new light.



9.10. Finishing the scene

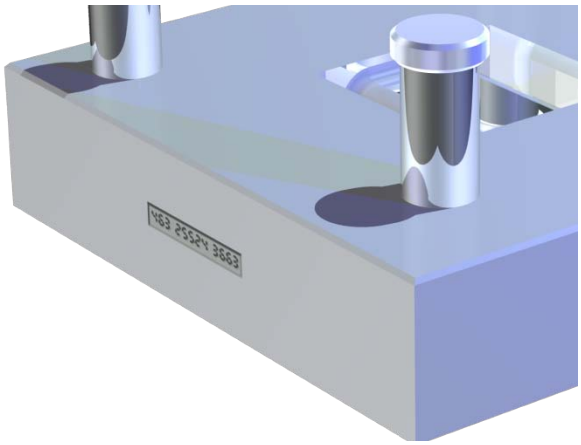
To improve the scene, the polished surface texture from the cavity will be added to the body of the Guide Pins and small chamfers added to the heads of the Guide Pins for added realism. It is detail features like this that can make a significant difference to the resulting image.
Using the same technique as before, select one of the cavity surfaces so that it highlights in green and using the Fill Eye Dropper icon, grab the shiny surface texture

and drop it onto the body section of one of the Guide Pins. The body section will be updated as IRONCAD is in 'face mode' from the selection of the cavity surface. Press the Esc key to cancel the Apply Eye Dropper command.
To add the chamfers to the head of the Guide Pin move the mouse pointer over the cylindrical surface of one of the Guide Pins heads. The icon alongside the cursor will change to a diamond shape indicating that the selection, if made at that point would be a surface. Select the surface. IRONCAD will highlight the surface green with brighter green edges. Right-click and select Chamfer Edges from the resulting menu. Set the value to 1mm and click OK. IRONCAD will add chamfers to all the Guide Pin heads and change the body surface texture as the Guide Pins are linked.

9.11. Adding a decal

Select one of the side faces of the Bottom Plate at surface level (green highlight), right-click and select SmartPaint. Open the Decal tab and set the Decal from selected image radio button. Browse to the Images folder of the IRONCAD installation (i.e. Program Files/IRONCAD/IRONCAD/2011/Images) and select lcd5.tif.

The image will represent a component identity tag.
The Decal icon will activate on the Visualization tab. Select the icon and IRONCAD to display a white projection box for the decal. Right-click and say 'Fit to model' IRONCAD will reduce the size of the projection guide to a more manageable level. Now open the Settings option and using the rotation set the rotation to 90 degrees and the width to 50 and click OK. Click on the Decal icon to turn the positioning tool off and render the scene.
The decal should appear as shown in the partial image below.



If it doesn't investigate positioning the decal using the TriBall

9.12. Completing the rendering

To complete the render setup and take advantage of IRONCAD's power, select the background option on the Visualization tab and set Exposure to -2. Before hitting Render Now, click on the drop down arrow on the Render Icon and set the Render Options as follows:

Super Sampling –		High
Global Illumination –	Type:	Full Illumination
	Quality:	Higher
	Power:	0.4
	Enable Photons:	Check

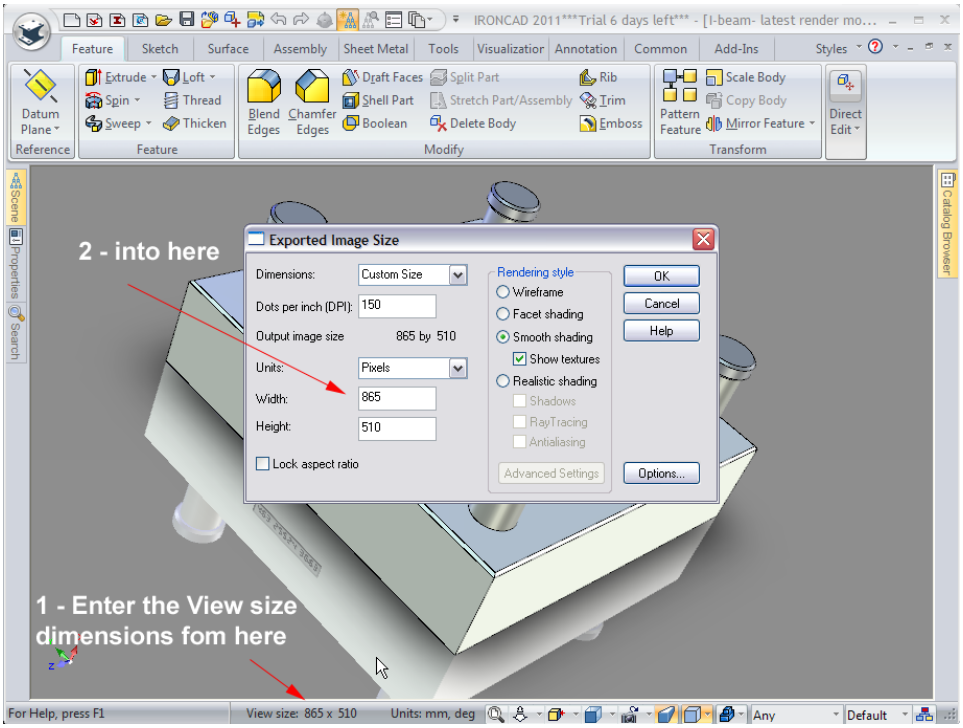
Now set the render off, it will take a little longer with these settings as IRONCAD has to do a lot more work. The image below has an added black base plate with low reflection intensity and a small amount of reflection blur added.

9.13. Poster sized images

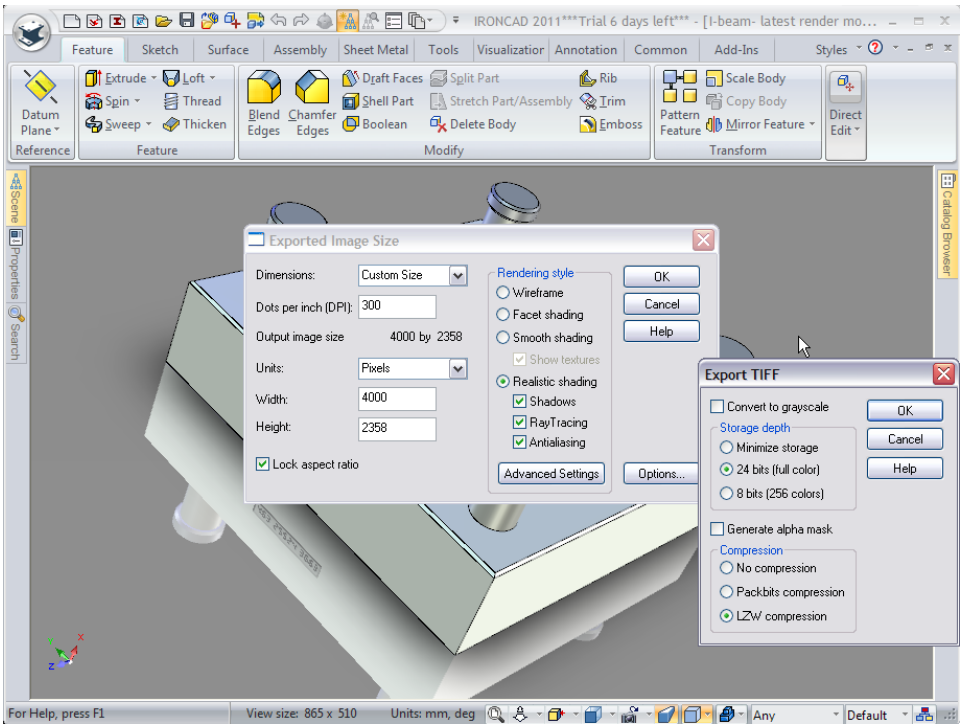
The renderings set so far use the screen size to set the rendering output. If the requirements are for poster sized images, IRONCAD has a specific tool within the Export group of the File menu.
It is useful, but not essential, to test the render output by following the previous steps. After selecting the Export Image option, IRONCAD will ask for the filename and present a list of types. For poster sized images TIF tends to be the most popular.

9.13.1. Ensure what you see is what you get

To ensure the aspect ratio and image contents reflect the current scene, in the 'Export Image Size' dialog box set the Width and Height to match the View size shown on the IRONCAD status bar, as shown in the image below.



Next, ensure the 'Lock aspect ratio' option is checked and replace the Width (or Height) value copied from the status bar with the required Width (or Height) for printing purposes, i.e. 4000. Enter the resolution of the required image in the Dots per inch field and turn on the Shadows, Ray Tracing and Antialiasing options.



The Advanced Settings will be the same as the previous screen based render export. Finally set the desired Options such as color mode (8 or 24 bits) and set the Alpha mask option if the output required just contains the model and not the model and background.

Clicking OK will export the image. Be aware that this process does not use multi-threading so IRONCAD will be un-available during image export.

This completes the section introducing IRONCAD's photorealistic rendering capabilities. It is well worth experimenting with surfaces, textures and materials to build custom libraries of parts. Using the assembly created early on in the Getting Started Guide this section introduced the basics of:

- Simple drag and drop rendering techniques
- Creating screen based images for use with standard screen capture tools
- Creation of high quality renderings for everyday use to large format poster style renderings

The next chapter will complete the Getting Started Guide by introducing the techniques for creating an animation of the I-Beam assembly used in this section.

10. ANIMATING WITH IRONCAD

Extending the built in communication capabilities, IRONCAD has a wealth of tools to create animations to communicate the design.
Animation can be great fun. This section introduces some of the core techniques and terms to provide a foundation on which to build. The chapter will show how to:

- Apply a SmartMotion directly from the catalog
- Edit the dropped SmartMotion so that it performs the relevant moves
- Add an addition segment to an existing path
- Change the start point of an animation segment
- Produce a dynamic, cut-away section in a model

For simplicity, the I-beam assembly used in the rendering section will again be used. Start by opening I-beam.ics saved previously or found in the tutorials section and in addition open the animation catalog.
In IRONCAD animations are broken down into segments and combined in a sequence using IRONCAD's segment editor called the 'SmartMotions editor'.

10.1. Fundamental control of animation segments

10.1.1. Length, Height and Width

IRONCAD uses the Length, Height and Width notation to apply and control the animation. So before applying an animation segment check the orientation of the part, or feature anchor.

10.1.2. Key

A simple animation such as a linear move has a start point and a stop point. These two control points are known as keys. The orientation of the part, assembly or feature at each key can be different with IRONCAD performing the calculations for the position of the model as it moves from one key position to the other. Keys can be added and manipulated to obtain some great results.

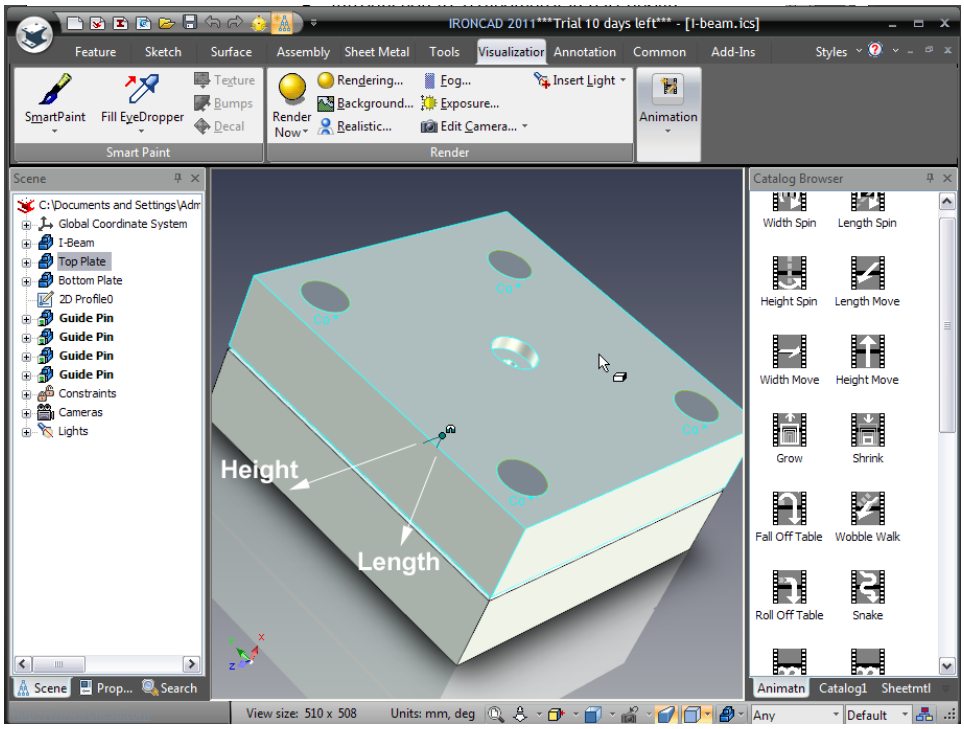
10.1.3. TriBall

The TriBall can be used to position a key in the sequence.

10.2. Animating the I-beam tool arrangement

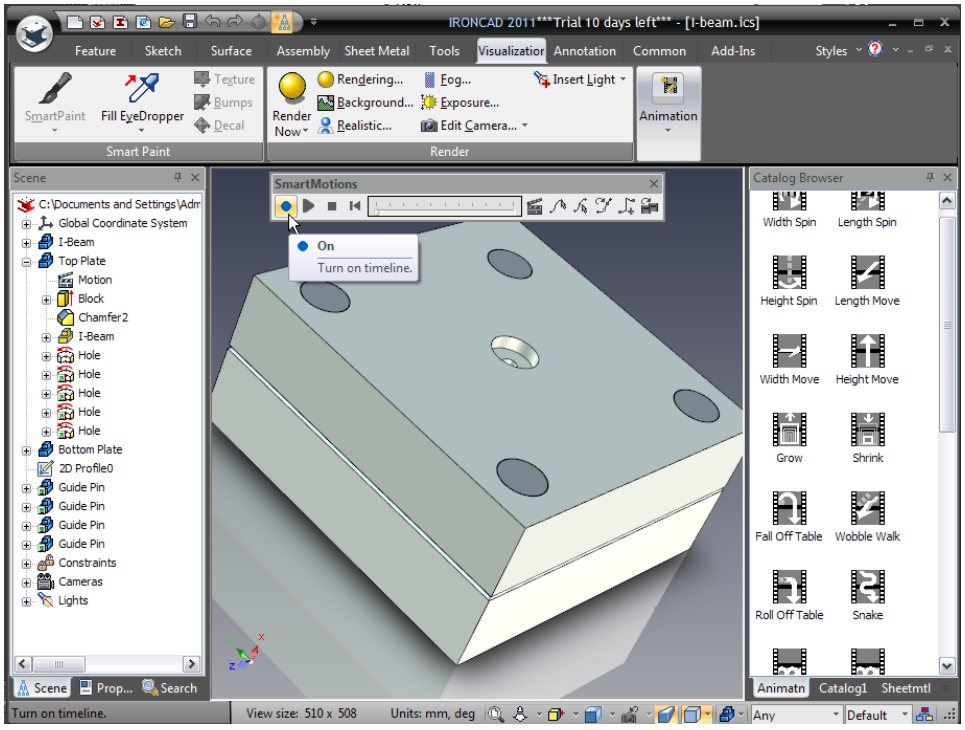
10.2.1. Opening the Top Plate

Select the Top Plate and look at the parts anchor as illustrated below.



To open the mold the Top Plate has to have a Length Move applied in the negative direction. From the Animation catalog drag a 'Length Move' segment onto the Top Plate. IRONCAD will draw a white line to represent the motion. In the scene browser, expanding the Top Plate entry will now reveal a Motion entry.

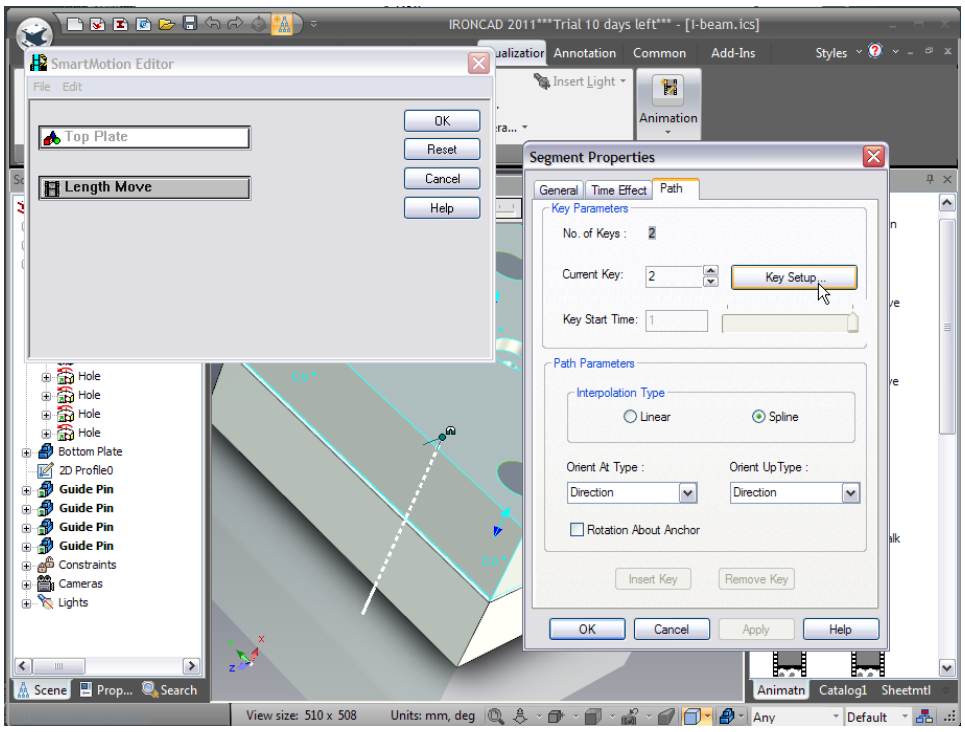
From the representation of the applied animation segment, the Top Plate will move in the opposite direction than that required. This can be confirmed by previewing the animation. Right-click in a free area of the Windows Ribbon bar and select SmartMotions. IRONCAD will display a convenient tool bar for controlling the SmartMotions applied to the scene as shown in the following image.



Click on the Blue ‘on’ button and press the play button. The Top Plate will move through the Bottom Plate.

In the scene browser, right-click on the Motion entry at the top of the Top Plate entry and select Edit. IRONCAD will display the SmartMotion Editor displaying a Length Move entry, right-click ‘Length Move’ and select Properties. In the displayed Segment Properties panel the length of time can be set, however leave it at 2 seconds for now and open the Path tab.

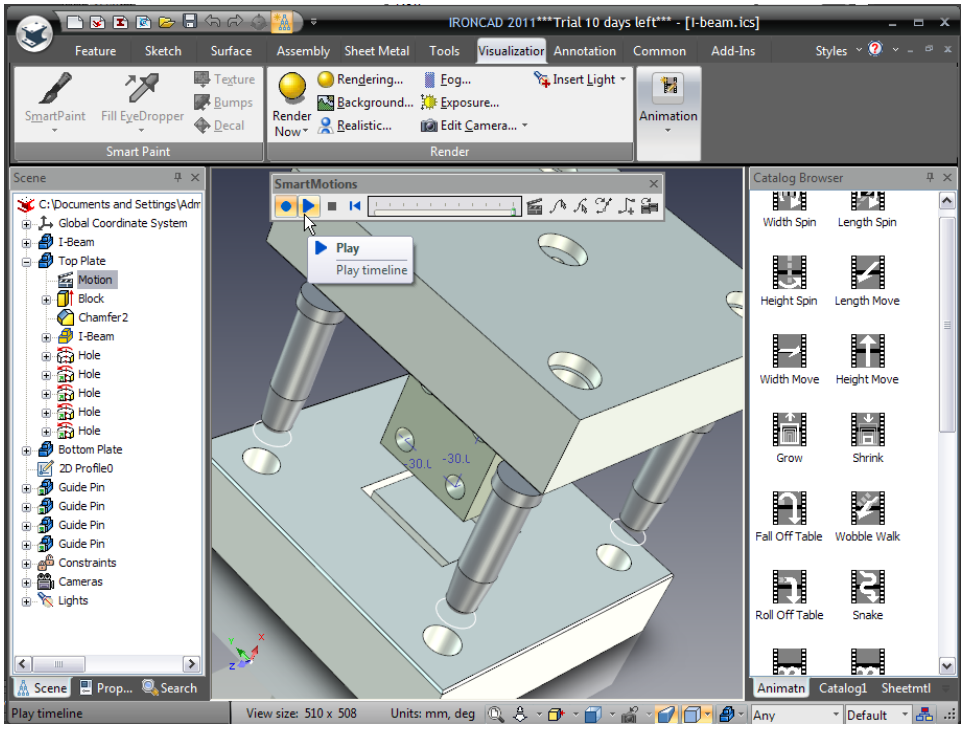
In the Key Parameters pane of the Path tab, IRONCAD reports the number of keys comprising the animation. Using the arrows change the Current Key to 2 and click on the Key Setup button.



IRONCAD will display the spatial position of this key, edit the L (length) value to be minus 50 (-50) and click OK on the three option panels to accept the change. Turn the animation preview on and click on play. The Top Plate will move in the right direction.

10.2.2. Opening the Bottom Plate

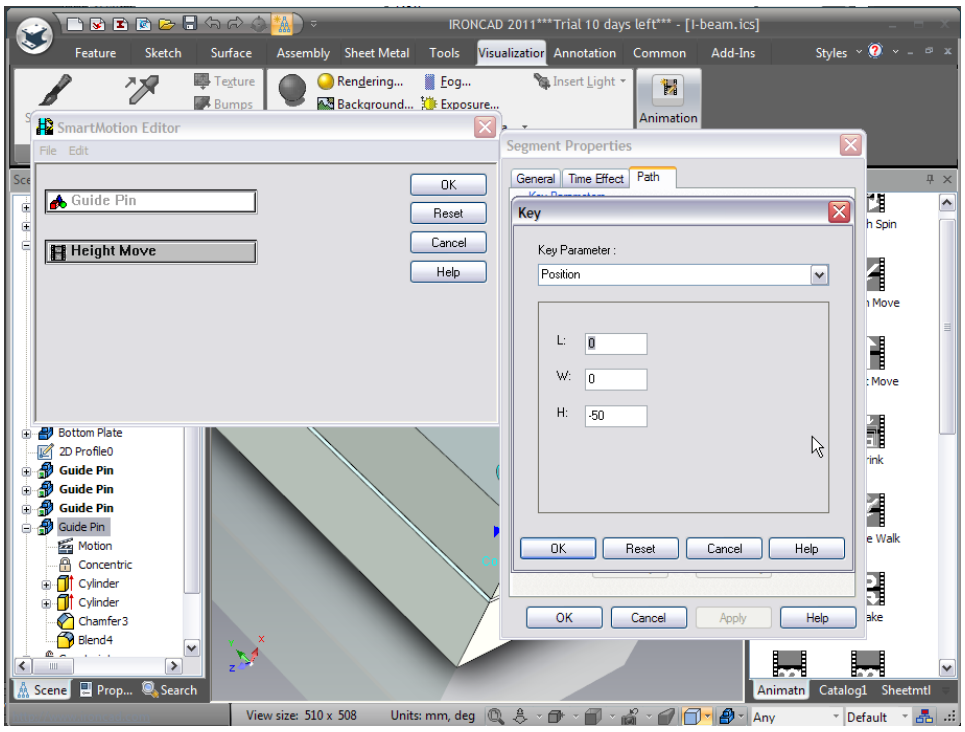
Right-click on the Motion segment just edited in the Top Plate feature tree of the scene browser and select ‘Copy SmartMotion’.
Now right-click on the Bottom Plate and select Paste. IRONCAD will apply the same animation segment setting to the Bottom Plate.
Test the animation by switching the preview on and selecting Play.
This animation segment will run at the same time as the one for the Top Plate and will also run in the correct direction as the Bottom Plate anchor orientation is a mirror of the Top Plates anchor.
The scene should resemble the following screen image at the end of the animation.



10.2.3. Animating the Guide Pins

Turn off the animation preview and select one of the Guide Pins to review its anchor orientation. The Height direction is running into the Guide Pin along the axis so add a Height Move to the Guide Pin from the catalog.

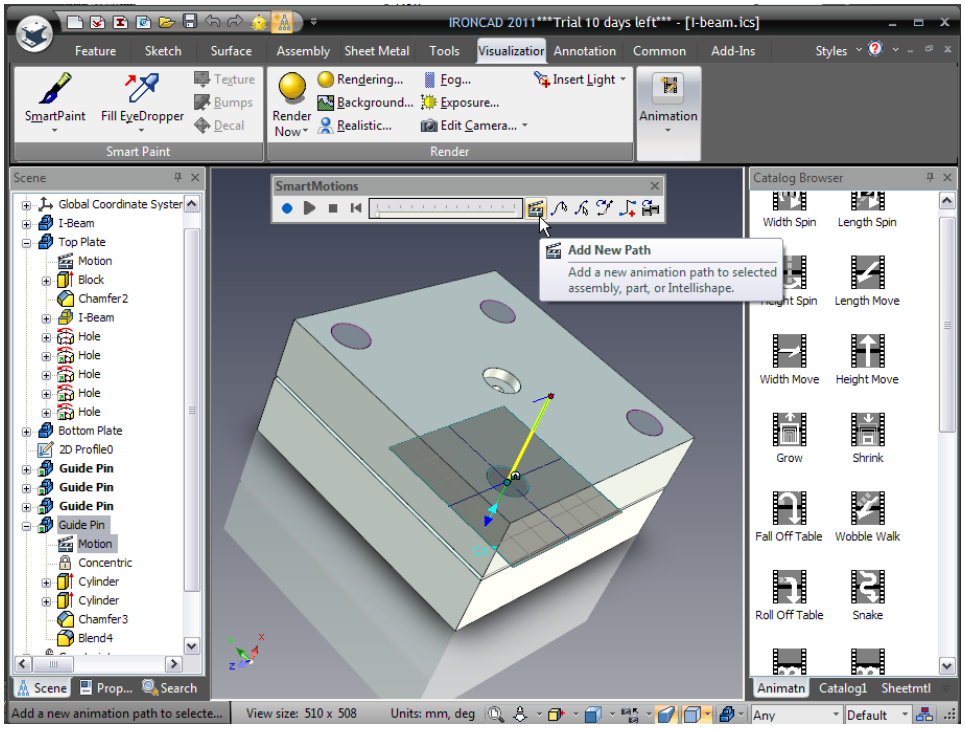
The animation segment is not copied to the linked instances as animations. As with the Top Plate, right-click on the Motion entry in the Guide Pin feature tree and select Height Move, Segment Properties and then set the Height to minus 50 (-50) for Key 2 in the Path tab.



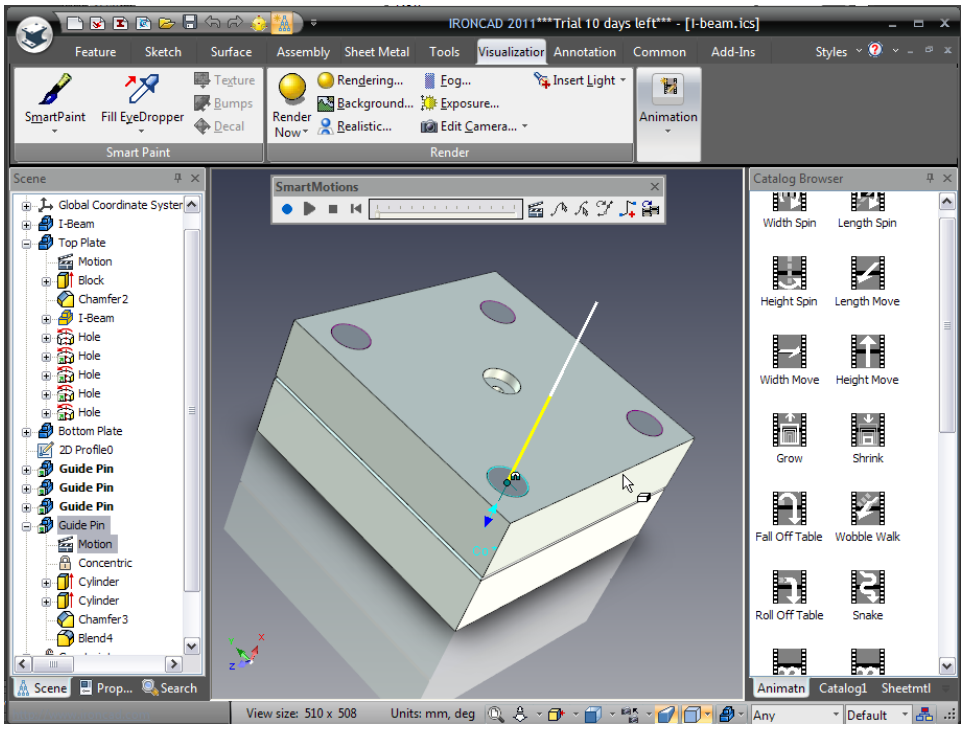
Previewing the animation will show the Guide Pin keeping in step with the Top Plate.

10.2.4. Adding a new animation segment to an existing animation path

Once the Top Plate has opened, the Guide Pin should move with the Top Plate and then continue moving away from the Top Plate. To add a new path to the existing Guide Pin animation path, first select the Guide Pin with the animation segment in place (ensure the SmartMotion preview is turned off) and then left click on the white animation path. IRONCAD will change the path to yellow and show a grid with an anchor at the starting key.

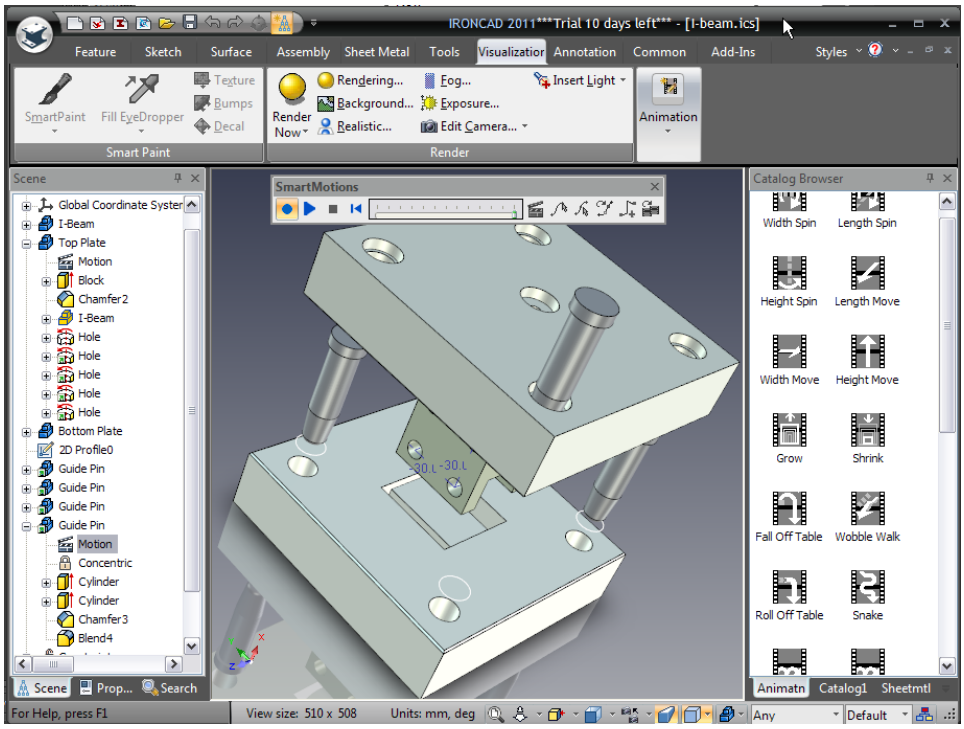


Select Add New Path. IRONCAD will display a wizard to help add the features of the new path. Select Height Move and set the value to -50 and click OK (the next screen is for setting the duration, 2 seconds is fine hence clicking OK directly). IRONCAD will add a new segment onto the first segment.



10.3. The SmartMotion Editor

Click on the IRONCAD home button and select SmartMotion Editor from the Tools section. A timeframe panel will be displayed showing each part being animated. The Guide Pin will have a longer bar than the others, Right-click on the Guide Pin entry and select Properties to see that its duration is 4 seconds.
Previewing the animation will show the Guide Pin staying with the Top Plate until the Top Plate reaches its extents after which the Guide Pin withdraws from the Top Plate.



Turn off the SmartMotion Editor and right-click on the Motion feature of the Guide Pin and select Copy SmartMotion. Right-click on the remaining Guide Pins and select Paste to apply the animation.

10.4. Changing the start point of an animation segment

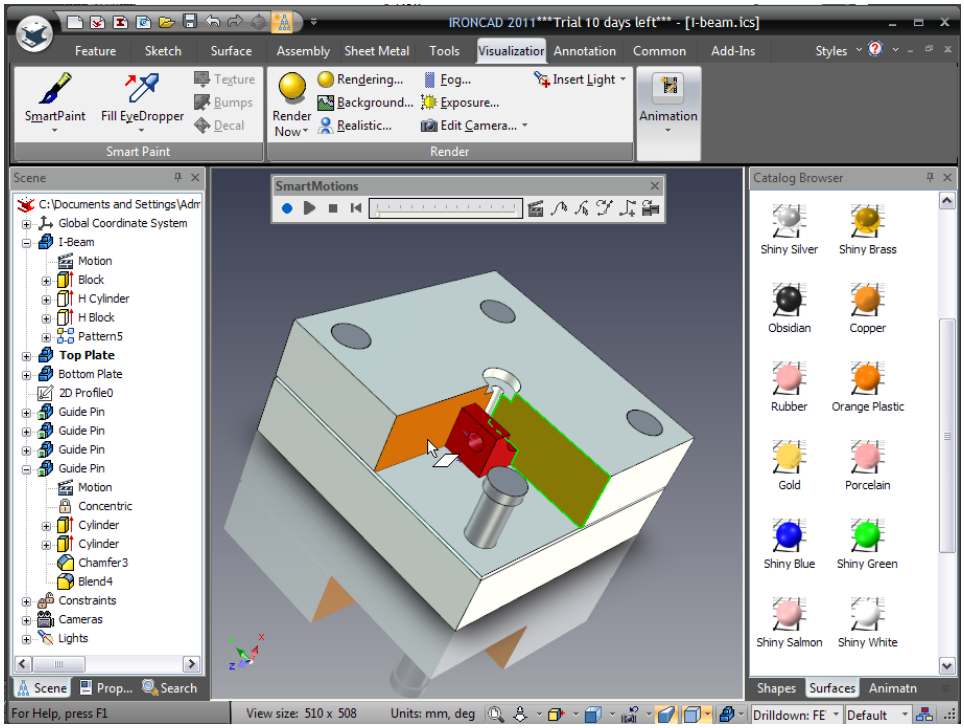
Not all animation segments need to start at the same point in time. For example, the I-Beam part itself should stay in place until the tool plates are open. To change the start point for the I-Beam animation first, hide the Top Plate by right-clicking on it and select Hide Selected from the menu. Next drop a height spin onto the I-Beam component itself and finally open the SmartMotion Editor again and right-click on the I-Beam entry. To change its start point, select Properties and change the Track Start Time to 2.

At this point to make it a little more interesting, open the Surfaces catalog and drop the Shiny Red surface onto the I-Beam.
Now right-click in the scene and click on Show All to re-display the Top Plate before previewing the animation.

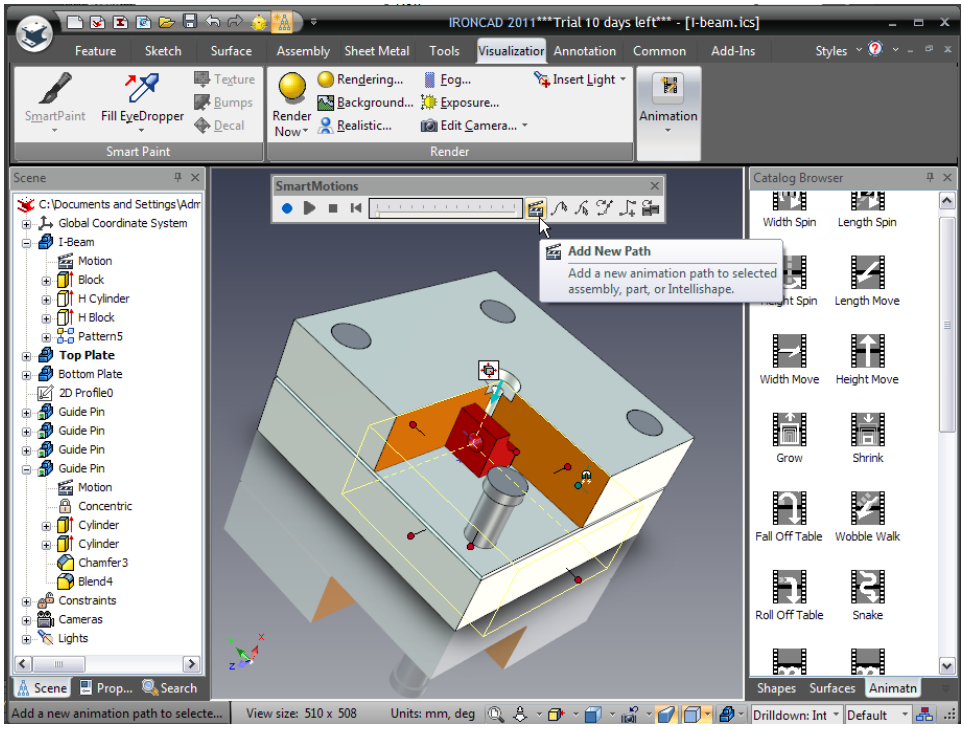
10.5. Creating a dynamic cut-away section.

IRONCAD can animate IntelliShapes as well as 3D models and assemblies. This ability can be used to show viewers what happens inside an object, or what the impact of a machining process might be.

To perform a dynamic cut-away section of a model, open the Shapes catalog and drop an H Block onto the Top Plate aligning the cut faces with the center of the part.
Select the one of the cut faces at the Surface level and drop the surface Orange Plastic onto it from the Surfaces catalog. Drag the same color onto the other cut face as shown in the image.

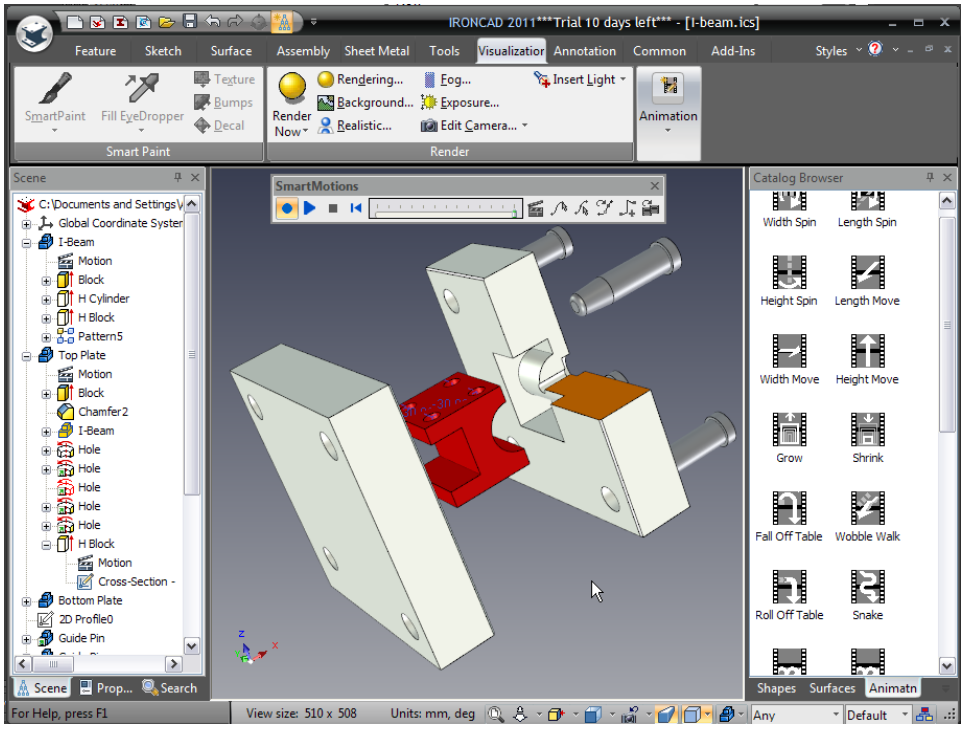


Select the H Block at the IntelliShape level and from the SmartMotions toolbar click on ‘Add New Path’



In the resulting wizard, select Move and then using the pull down select ‘along length direction’ and enter -76 for the distance. IRONCAD will add a new animation path for the IntelliShape showing the path. With the H Block highlighted activate the TriBall and move the H Block away from the Top Plate in the direction of the H Blocks animation path by a distance of 76mm.

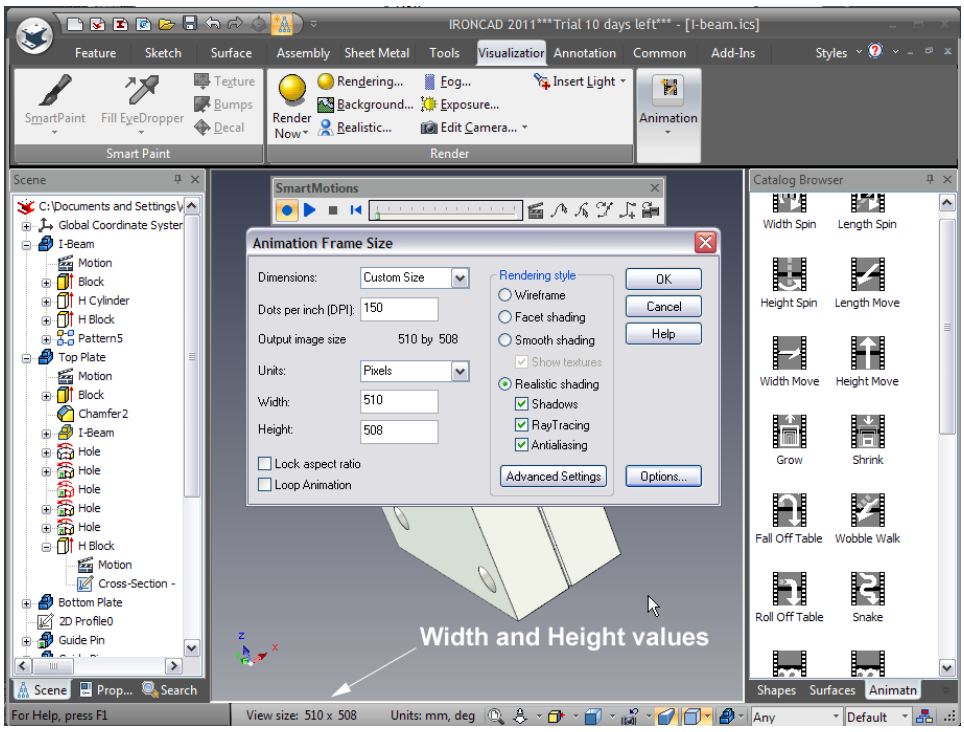
Preview the animation and when the animation is at its end rotate the view so that the view resembles the following image.



10.6. Exporting the animation

From the IRONCAD home button, select File followed by Export and then animation. Enter a name for the animation such as I-Beam Toolset, saving as type AVI. IRONCAD will display an Animation Frame Size dialog box. To maintain the same aspect ratio, in the Width and Height, enter the values of the current View size from the status bar as shown in the following image.

Set the Realistic shading options on for Shadows, Ray Tracing and Antialiasing and select OK to accept the default export format It is worth experimenting with the other options to see which is the most suitable for the intended use of the animation e.g. exporting individual frames can be useful for importing into Windows Movie Maker to introduce pauses and notes. After clicking OK, IRONCAD prompts with an Exporting Animation progress bar, click Begin. Be aware that depending on the animation length, view size and render quality – animations can take some time to create.



This ends the section on animation. The following topics were covered in this section:

- Adding a SmartMotion directly from the catalog
- Editing of the dropped SmartMotion so that it performed the required moves
- Addition of a segment to an existing path
- Changing the start point of an animation segment
- Production of a dynamic, cut-away section in a model

for more product info and support

www.ironcad.com

or 1.800.339.7304

Int'l: +1.770.240.3008