

IntelliShape™ Anatomy

Sizebox Handles
Controls overall size of component

Anchor
Short Whisker is Length and Long Whisker is Height

Handle Toggle
Used to toggle between the Shape and Sizebox handles

Behavior Push Pin
Indicates Shape attaching behavior. Right-click to change options

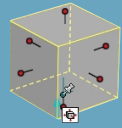
Shape Handles
Used to manipulate each 3D segment in a profile independently.

Extrude Direction Arrow
Blue Arrow Indicates Extrude Direction

Cross-Section (Profile) Indication
Underlying Cross-section (profile) highlights cyan

Shape Resizing

There are (3) three ways to resize a shape. 1.) Push/Pull a Handle conceptually, 2.) Right click on Handle and select "Edit Sizebox", and 3.) Right-click on shape, access IntelliShape Properties, access Sizebox Tab. IMPORTANT: IronCAD, LLC are the developers of 3D Handle technology and hence there are numerous types of handles for reshaping. This paragraph only refers to the sizebox handles which are indicated by Round Red Spheres (handles).



The 3D Fluent User Interface

Right Click Scene Background Menu
This menu provides access to IRONCAD's integrated Rendering Engine, Parametrics, Scene Window splitting, Hide functions, and more.

"S" Hotkey
Users can activate common commands based on the current selection level by selecting "S" on the keyboard.

Menu Button
Allows for the ability to group many of the common actions in a single menu item making it easier to access the necessary commands for the desired action

Quick Access Toolbar
Provides a single location for quick access to common commands.

Fluent Ribbon Bar
Provides a central location to access commands and options that pertain to a specific area of the design process.

3D Scene
This area is the 3D space in which you create your designs.

Scene Browser
This area is used to track all the parts and assemblies in the scene. It also maintains a parts build structure.

Quick Access Property Browser
Provides a single location for common actions and properties at the users current selection level.

Search Browser
Allows for the ability to find data within the current open scene file using various search filters.

World Coordinate Axis Indicator
A visual reference for the user in the scene for determining the global orientation in 3D. Right-click to resize.

Status Bar
This space previews informative messages and tips. Also indicates the current unit of measurement and view size, incorporates common CAMERA tools, and SELECTION tool with integrated CONFIGURATION controls.

Menu Bar
The area where all commands and functionality of IRONCAD can be accessed.

Catalogs/Catalog Browser
A Catalog is a group of related resources such as Intellishapes (features), Parts, Assemblies, Colors, Images, Tools, Documents, and more. To Use catalog items in your designs, simply drag their icons into the scene or on a part desired. Catalogs are used as Component Libraries, Tools, Project Folders, Sketch Pads, and more. The Browser stores the related information of a Catalog topic and you can use the scroll buttons to find desired components.

Catalog Tabs
Allows you to toggle between various catalogs that are active. Users can create their own catalogs with information specific to their needs.

Part Type Toggle
Toggle switch between Innovative and Structured part design modes. See back for more explanation.

Ribbon Contextual Tabs
The contextual tabs at the top of the Ribbon are broken out in the common areas of design.

Ribbon Groups/Galleries
Within the Ribbon Tabs, Groups or Galleries contain common operation functions.

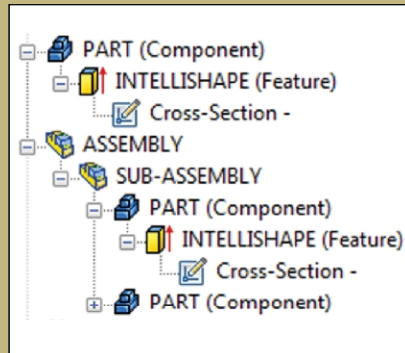
Assembly Modeling

IRONCAD performs all 3D modeling in a single scene environment which negates the need for process-specific design environments. So that being said it is important to understand how IRONCAD has eliminated the need for these separate environments without compromise and so understand the part/assembly hierarchy. The part/assembly diagram provides a simplified breakdown of IRONCAD's part/assy structure and coloring. The colors indicated in flow chart also match the default color settings used in IRONCAD to visually detect our selections. The actual Scene Browser image shows that you actually see in IRONCAD. Note that surfaces a part are not indicated in the Scene Browser. Only standalone surfaces will appear in the Browser.

Selecting Components that are NOT assembled.
1st Left Mouse Click = Part (Cyan highlight)
2nd Left Mouse Click = IntelliShape (Yellow highlight with red handles)
3rd Left Mouse Click = Face (surface) of Part. (Green highlight)

Selecting Components that ARE assembled (Assembly).
1st Left Mouse Click = Assembly (Yellow highlight)
2nd Left Mouse Click = Part (Cyan highlight)
3rd Left Mouse Click = IntelliShape (Yellow highlight with red handles)
4th Left Mouse Click = Face (surface) of Part. (Green highlight)

TIP: To reduce the amount of mouse clicks need to "drill-down" to desired element level. Use the Selection Filter options located on the Status Bar. This allows you to determine in advanced the components you wish to pick with only one mouse click.



Create Empty Part.
This command is useful in creating a top-down assembly structure. It allows the user to build the entire structure from the start, then complete the design information for the parts at a later stage.

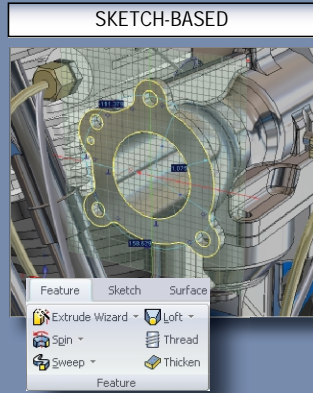
Creating an Assembly (Sub-Assembly).
Select the desired parts and/or assemblies from within the scene by holding down the Shift key and selecting all the parts you wish to assemble. Then depress the "Assemble" icon on the Assembly toolbar.

Re-Structuring and Assembly.
IRONCAD allows you to restructure parts to and from anywhere in the scene. From within the Scene Browser, simply drag a part/assy's icon over the target assembly (sub-assembly), when you see a "boxed plus symbol" appear release the mouse. The process performs bidirectionally. In other words you can drag parts/assys in and out of various assemblies as needed. To drag a part entirely out of any assembly simply drag it onto the scene icon on the very top of the Scene Browser.

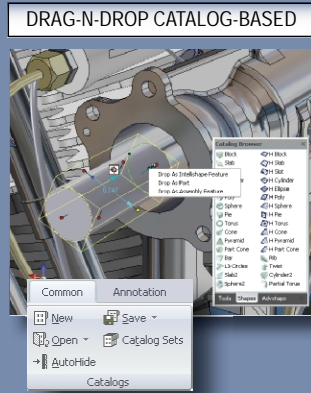
3D Modeling Methods

Creating a solid model part.

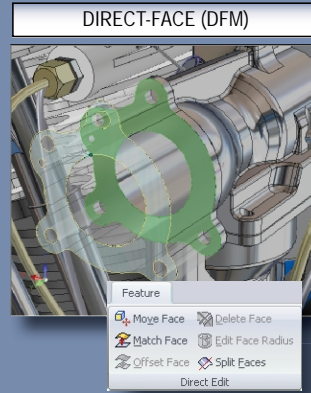
Models are created by combining existing or custom built models/Intellishapes within the scene. IRONCAD has 4 distinct modeling methods and they are: Drag and Drop, Sketching, Direct Face, and Surfacing. The IRONCAD power user will quite often use all 4 modeling approaches concurrently.



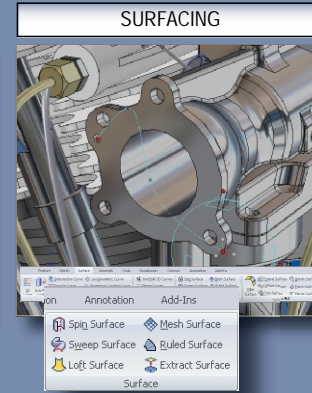
The traditional approach of 3D modeling found in virtually every application to date. Select the type of modeling function you wish to achieve, from the "Intellishape Creation" toolbar. Then select the placement for the drawing grid (workplane) to be positioned by clicking on a surface of a part or the background (creates new part). The TriBall can be used to reposition workplane if the plane does not orientate to your desired "angle".



Drag and Drop a preexisting Intellishape from the desired catalog on the right of interface. Select the desired component from a catalog and drop it into the background of scene to create a new part. Alternately, drag onto a part to become an intellishape of that part. In addition, if you right click and drag from a catalog onto a part/assy you'll have the option to choose these methods.



DFM performs modeling at the face (surface) level of parts thru the use of handles and actual surface manipulation using more "boolean" type functions. Select a surface (face) of a part then right click to select a face manipulation option such as Move, Taper, Offset, Match, etc. Then use the TriBall to re-position the surface to desired target location. Alternately use the "Face/Edge Edit" toolbar to use said commands.



Performs modeling using more "wireframe" type geometry to generate Class "A" type surfaces. Involves the use of 3D Curves and Surface Toolbar. Select the "3D Curve" tool and generate a Curve in 3D Space. Generate more to create various Surfacing types. Note that Surfaces can be standalone "parts" or "Intellishapes of a Surface Part. If created as a Intellishape of a surface, when a closed boundary is formed IRONCAD will automatically make a solid part of it without loosing the Surface geometry.

Part Type Toggle



Located on the status bar, users can designate the part type that will be created when creating new features through drag & drop from catalogs or when starting new features/sketches. This approach is used primarily when designing in a bottom-up design manner to build the parts and assemblies of the design.

Innovative Part Design - Innovative Part Design allows users to build a non-restrictive history of features that make up the design aspects of the part. Although it has a history of features, the system allows the user the ability to dynamically change the history order without adverse effects of feature dependency rules and restrictions.

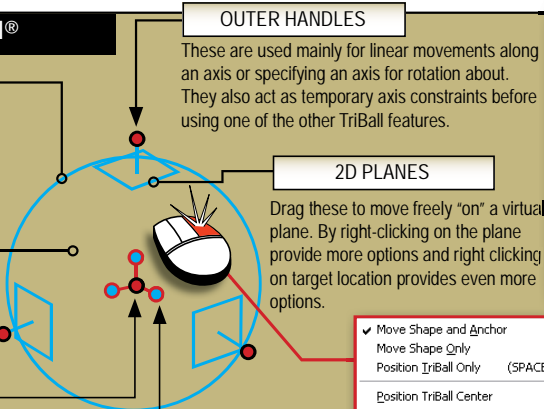
Structured Part Design - Structured Part Design is a history-based structure of features that build the design following an intended design order determined by the user. This part design allows the user to build a rigid sequence to their design that can predictably changed based on the intent provided by the user.

The Patented TriBall®

CIRCUMFERENCE
Drag this to rotate about the virtual axis extending from your "viewpoint" of the TriBall Center. TIP: Really only useful when orientating decals on surfaces.

INTERIOR
Drag inside this blank area to rotate. Also right-click here and set various TriBall options such as snap settings.

CENTER HANDLE
This handle is used primarily to perform point to point re-positioning. It is used by dragging it directly to a target point. Or if you right-click on it provide more specific point options. In addition, often used when an outer handle has been constrained first.



ORIENTATION HANDLES
These are used for orientating objects with the center of the TriBall as the pivot point. There are two ways to use these handles; 1) Drag a handle to "point" the axis at another location, or 2) Right-click and choose an item from the pop-up menu. TIP: These handles are most often used when an outer handle has been first constrained.

OUTER HANDLES
These are used mainly for linear movements along an axis or specifying an axis for rotation about. They also act as temporary axis constraints before using one of the other TriBall features.

2D PLANES
Drag these to move freely "on" a virtual plane. By right-clicking on the plane provide more options and right clicking on target location provides even more options.

TriBall Hotkeys

- F10 = Toggles TriBall On/Off
- CTRL + Drag = Incremental Linear/Angular Snap
- SHIFT + Drag = Invoke SmartSnap to Center Handle
- SpaceBar = Toggle to Move TriBall Only.

- ✓ Move Shape and Anchor
- Move Shape Only
- Position TriBall Only (SPACE key)
- Position TriBall Center
- Reset TriBall to Anchor
- Orient TriBall to Global
- Orient TriBall to Active Coordinate
- Create Attachment Point Based on TriBall Orientation
- ✓ Show Planes
- ✓ Show Constrained Dimensions
- ✓ Show Orientation Handles
- Show All Handles
- Allow Unconstrained Rotation
- Change Snap Increments...

TriBall Interior Right-Click Pop-Up Menu

TriBall® Quick Tips

- LINEAR Movements** - Right-click and drag the Outer Handle. The distance value appears, acting as an odometer to visually gauge your movement distance. As you release, select move from the resulting dialog menu and enter a value.
- ROTATIONAL Movements** - Left-click the desired Outer Handle. This will act as the rotational axis. Then position your cursor inside the blank area of the TriBall until you see an icon of a hand with a double ended arrow under it. Right-click and drag in the direction you wish to rotate and release. Select Move from pop-up menu and enter desired angular value.
- LINEAR PATTERNS** - Right-click on the desired Outer Handle which is "pointing" in the direction in which you wish the pattern to extend. Select "Create Linear Pattern" from pop-up menu.
- RECTANGULAR PATTERNS** - Right-click and drag the desired 2D Plane in the direction you wish to pattern object. As you drag two distance values appear indicating the distance moved from original location. When you release cursor, select "Create Rectangular Pattern" from pop-up menu. Enter the values as needed using the visual dimensions as reference.
- RADIAL PATTERNS** - Left-click an Outer Handle to be used as a Rotational Axis (highlights yellow indication axis is temporarily constrained). Position your cursor inside the blank area of TriBall under hand/arrow icon appears. Right-click and drag in the direction you wish pattern to extend upon release of mouse select "Create Radial Pattern" from pop-up menu and enter appropriate values.

Precise Positioning Tools

Positioning of all components in IRONCAD use the same tools and processes. You can position shapes graphically by dragging and sliding them on the part or scene. "Cruising" is the term coined for this technique. This is extremely useful in conceptual design. Precise positioning of elements in your scene can be accomplished in several ways: 1) TriBall, 2) SmartDimensions, 3) SmartSnap, and 4) Position Dimensions.
TriBall - See The TriBall Tips on this sheet.
SmartDimensions - Select the shape and then select the type of SmartDimension you wish, typically Linear is used in 3D scene. Next select the intellishape at a key reference point and then a target point, edge, or surface to use as a basis for measurement.
SmartSnap - Select Shape and drag while holding down SHIFT key to align with key points and edges on the part. Also use SmartSnap (holding shift key) while pulling on Handle to snap to key points, edges, and surfaces of any part/assy in your scene.
Position Dimensions - Simply position a Part/Assy according the Global Origin Point. To see position dimensions, right click on scene background, select Show and choose Position dimensions from pop-up menu. Then select the Part/Assy and they will appear.