ZW3D
Innovator Training Guide

ZW3D™ V2014 Innovator Training Guide

Forward

The full version of the ZW3D documentation is available by selecting **Help Browser** from the ZW3D **Help Menu**. Go to the **"Program Folder\doc"** directory on the ZW3D program directory and open the file **"ZW3D.chm".**

This **Innovator Training Guide** is available as a printed manual and in PDF format. To view or print the PDF version, install the Adobe Acrobat Reader and open the corresponding files in the **" Program Folder\PDF"** folder in the program directory. Refer to the **HELP Manual** for information relating to specific tasks.

NOTES:

1. For best results, view and print the PDF version of the documentation using the Adobe Acrobat Reader.
2. Since the **Innovator Training Guide** is only a portion of the complete documentation, it is highly recommended that you utilize the **HELP Manual**. You will find additional topics, more information on command options, input options, additional command notes, and cross-links to related tasks and topics.
3. It should be noted that the HELP Manual will always be more up-to-date than any printed or PDF material. If you find any discrepancies between the two, always refer to the HELP Manual.
4. While you are working with ZW3D, simply press **F1** to view the documentation about the command you select.

To help you customize your CAD/CAM requirements, please contact our Customer Support department on our website http://www.zwsoft.com/ or call at (8620) 38289780 ext 565. You can also contact your local sales representative or application engineer to discuss your requirements.

Thanks for being our customer!
Team ZW3D

Table of Contents

System Requirements

ZWCAD Corp. recommends the following configuration for operation of ZW3D CAD/CAM Software.

Recommended configuration

- Intel Pentium® IV（2GHz or up）, Intel Xeon®, Intel Core® or equivalent AMD® processor
- 4GB RAM and up
- 8GB Hard Disk space for installation and up
- 2GB swap space and up
- 3 Button Mouse with scroll wheel
- Microsoft DirectX® 9 -capable graphics card and up or OpenGL-capable graphics card
- 1280×1024, 1680×1050, 1920×1080 32 bit Color Video resolution

Minimum configuration

- Intel Pentium® IV（2GHz or up）, Intel Xeon®, Intel Core® or equivalent AMD® processor
- 1GB RAM
- 1GB swap space
- 1GB Hard Disk space for installation
- 3 button mouse
- Microsoft DirectX® 9 -capable graphics card and up or OpenGL 2.1 and up -capable graphics card
- 1024 x 768　32 bit　Color Video Resolution

Operating Systems Supported

- Microsoft® Windows XP SP3（32 bit）
- Microsoft® Windows 2003 Server（32 bit）
- Microsoft® Windows 2008 Server（32 bit）
- Microsoft® Windows Vista（32 bit or 64 bit）
- Microsoft® Windows 7（32 bit or 64 bit）
- Microsoft® Windows 8（32 bit or 64 bit）

Required for licensing

- Ethernet card or Hardware key

Required for Floating License

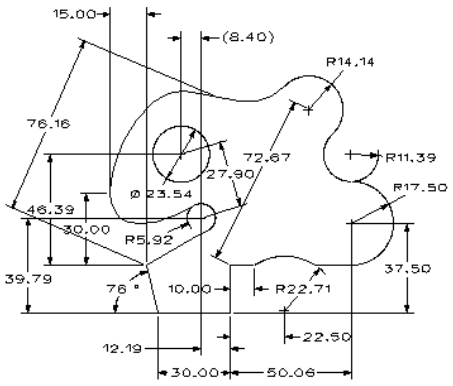
- Installation of TCP/IP Network Protocol

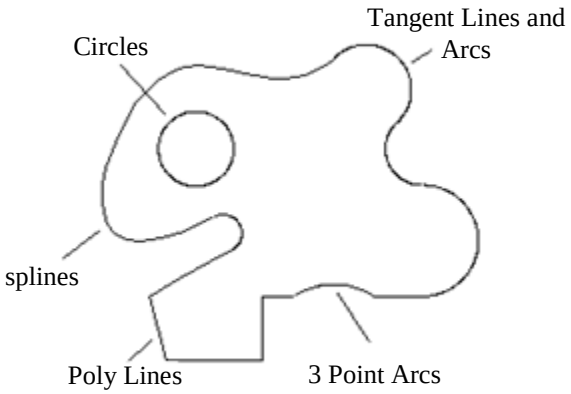
Recommended for training and support

- Speakers or head phones
- Space Ball devices are supported

Review of ZW3D Products

ZW3D Sketcher





ZW3D features an easy to use sketcher for quickly defining 2D feature profiles and path curves.

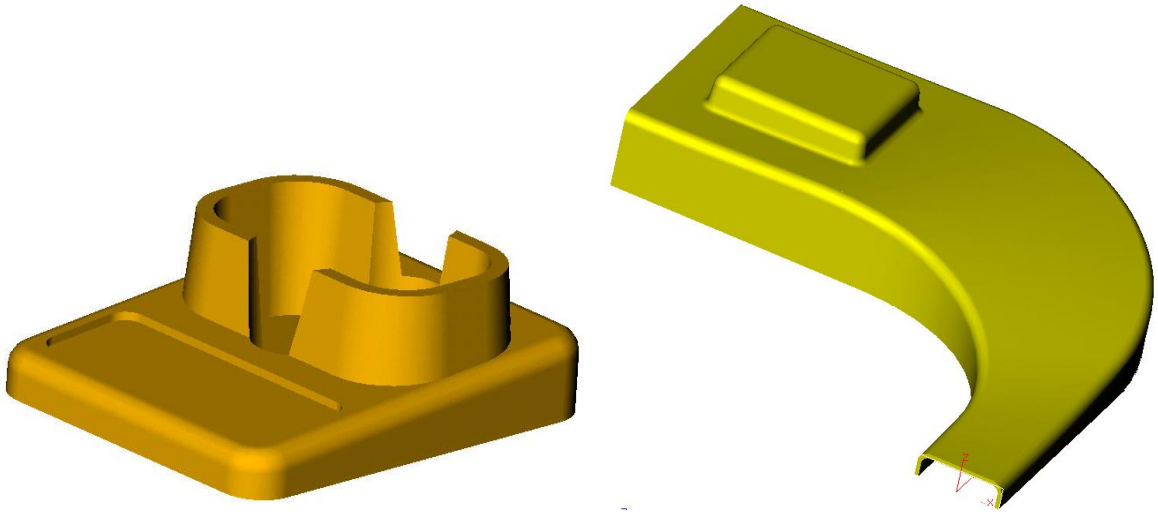
A principle highlight is our Quick Draw (Right-click > Draw) function that simplifies creation of all common sketch geometry with a single command. An intelligent, highly interactive cursor helps define and capture design intent. Geometry can be automatically dimensioned and constrained as it is created. 3D Part data can be easily referenced for constructing and locating sketch geometry.

Because ZW3D is tolerant of over and under-constrained sketches, users can create and modify dimensional and geometric constraints in whatever fashion is convenient --- or not at all!

ZW3D now uses the D-Cubed™ Constraint Manager to help create correctly constrained geometry. A powerful Smart Solve function is provided to automatically suggest and generate any missing dimensions and constraints needed to define a fully constrained sketch.

In ZW3D, object-oriented tools ensure that edits are easy and intuitive. For example, a simple double-click is all that's needed to modify dimension driven geometry. Double-click a curve to access the powerful constraint based spine editor. Select an entity and hold down the middle mouse button to access a complete list of its edit methods. Using the powerful filtering tool, pick a list of entities, and then select a method to perform mass edits. Cut and paste entities between sketches. It's all so easy.

ZW3D Modeling



The foundation of a CAD/CAM system is its geometric modeling kernel.

At the heart of ZW3D is the second generation of a proprietary technology called ZW3D OVERDRIVE™. The ZW3D OVERDRIVE kernel and database are truly high performance in every sense of the word. In independent benchmark tests of functionality, speed and robustness, ZW3D compares favorably against every CAD/CAM product available today --- regardless of price.

One key to the profound stability of ZW3D OVERDRIVE is the methodology we call Proximity Compliance Tolerancing. This unique technology is a substantial improvement over other kernels that rely on a fixed, relative or "adaptive" tolerancing scheme, and has the potential to eliminate tolerance problems entirely. While obviously a significant advance, this is just one example of the many groundbreaking technologies that characterize the ZW3D OVERDRIVE modeling kernel.

The desire to address the full range of 3D modeling tasks, from industrial design through mechanical engineering and on to mold and tool design, has led experts to conclude that a comprehensive CAD/CAM system must support a combination of solid, surface and wireframe modeling techniques. This approach is known as hybrid modeling, a concept pioneered by ZW3D Corporation. No other company has more experience with this technology than ZW3D Corporation, and no other modeler has integrated solid, surface, and wire frame geometry as seamlessly.

ZW3D Drafting

ZW3D Drafting dramatically simplifies the process of creating production drawings and details.

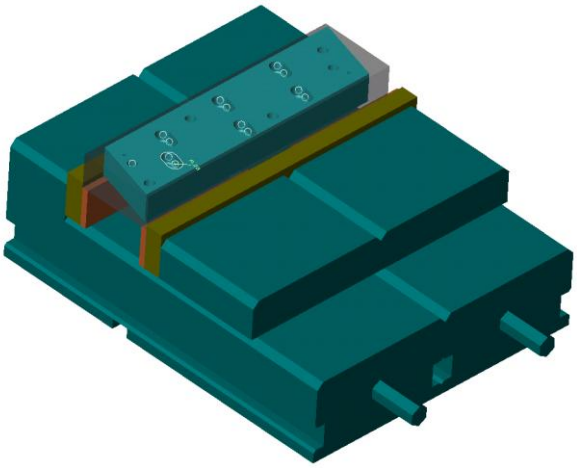
If you are designing 3D models or using traditional 2D CAD techniques, ZW3D makes it easy. Creating views of 3D models is a snap, and of course all views are fully associative and synchronized with the master model from which they are created. The unique object server architecture of ZW3D allows the user to decide whether drawings are to be saved in the same file as the 3D data or in separate files.

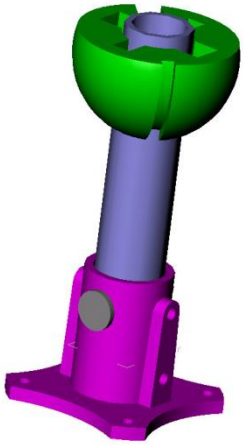
When there is a need to use traditional 2D drafting techniques, ZW3D again rises to the occasion. Using our incredibly intelligent Quick Draw and Quick Dimension tools, users can perform almost all standard geometry creation and annotation using just two commands! And of course, our intuitive object oriented editing commands make changes a snap.

All the functionality needed to generate drawings conforming to ANSI, ISO, DIN, and JIS standards is included, as well as productivity tools like symbol libraries, region Booleans, and boundary clipping for quick detail creation.

The ZW3D Table Manager aids in creating Bill of Material (BOM), Hole and Electrode tables.

ZW3D Assembly

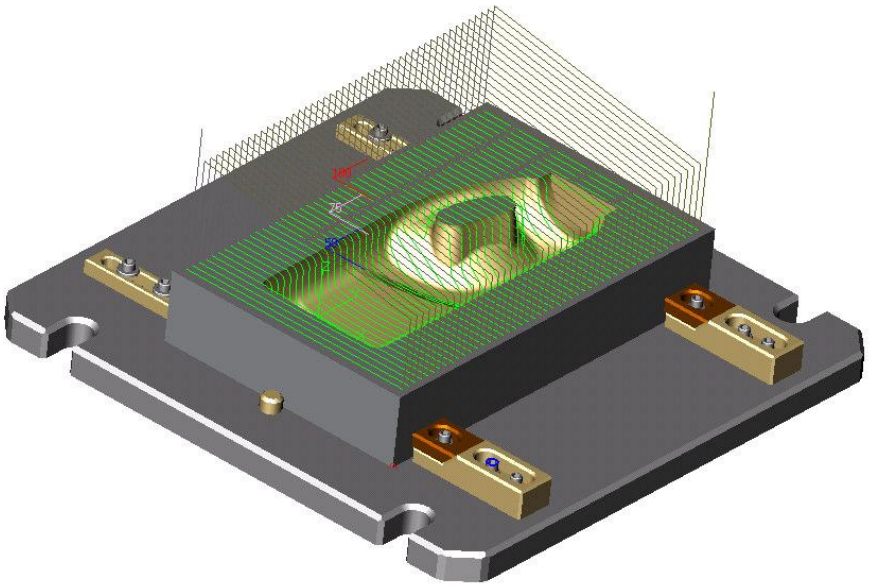




Assembly modeling is not considered an add-on or adjunct application.
Support for efficient definition, manipulation and management of assemblies was a fundamental consideration in the design of the ZW3D architecture. For example, the ZW3D OVERDRIVE Object Manager loads only the display data for an object into memory unless the object is activated for edit. This minimizes the memory footprint for a given assembly and maximizes the assembly size that can be effectively worked with. The ZW3D OVERDRIVE Object Manager also allows the user to decide how the assembly is to be distributed into files --- each component in a separate file, the entire assembly in one file, or anywhere in between.

With ZW3D, true in-context assembly design is now a reality. Selecting a component for edit is a simple double-click. Of course, ZW3D provides tools to manage the scope of the assembly display and a browser for viewing and manipulating the assembly using its tree graph. Simplified and alternate components can be defined. Support for creation and recall of alternate assembly configurations is integrated into the core ZW3D modeling system. Controls for object sign-out/sign-in and management of revisions are built in to the ZW3D OVERDRIVE (TM) kernel of ZW3D, as are assembly level features and the ability to easily create part variations and derivations. Assembly animations and output of AVI files is also supported.

ZW3D Manufacturing

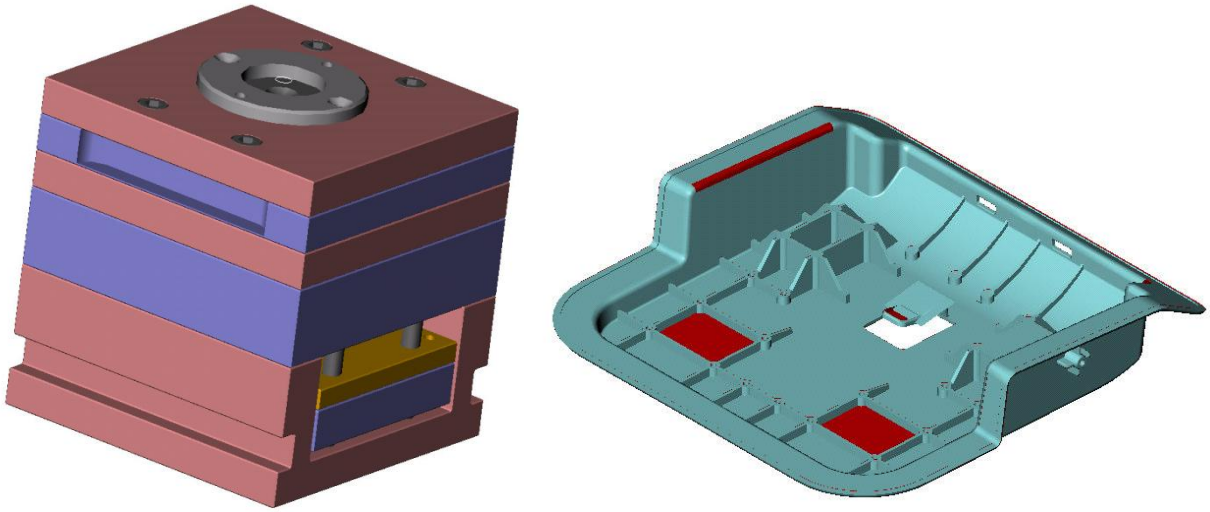


CAM modules further advance the concepts of intelligent process-oriented manufacturing software. These concepts were pioneered by ZWCAD Software Co., Ltd. almost 25 years ago. From basic drilling through multi-axis milling, a single concept unifies ZW3D. CAM utilizes a manufacturing knowledge base and leverages the information stored in the digital CAD model. ZW3D CAM provides easy-to-use tools for creating, accessing and managing a database of manufacturing resources, 'machineable' data and standard processes. The result is increased productivity and improved quality.

Of course, when it comes to NC machining the tool path is paramount. ZW3D provides a comprehensive suite of tools for calculating tool motion; 2, 3 & 5 axis milling and drilling. Almost any NC machine can be programmed with ZW3D. True 3D-workpiece modeling allows the system to simulate material removal and uses this information to intelligently machine the
remaining stock. Whenever possible, design feature information is utilized to streamline the NC
programming task.

While ZW3D tool path generation is among the very best in the industry, many will feel that the most significant advantage to ZW3D CAM is the ability to maintain complete associativity and synchronization with the 3D-product model. Because the manufacturing information can be updated directly from the design data, process definition and NC programming can begin sooner and design changes are much easier to accommodate. No system, which relies on third party add-ons for CAM, can offer the level of integration and associativity provided by ZW3D.

ZW3D Mold & Die

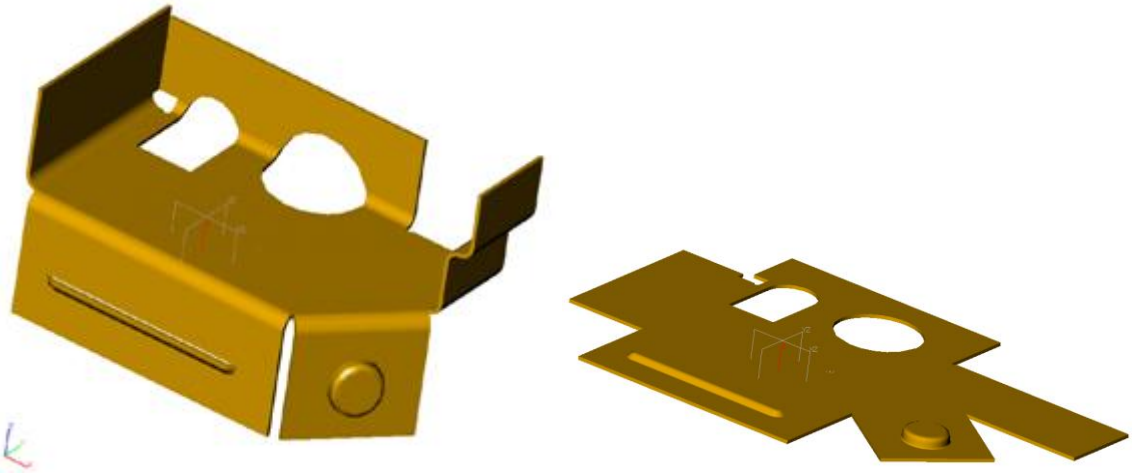


Mold & Die is a specially bundled and configured set of tools optimized for the design and documentation of plastic injection, compression and blow molds. Built upon, and fully leveraging the ZW3D unified modeling kernel and assembly-centric database, Mold & Die includes a number of specialized tools for quickly and easily performing common mold design functions while also solving the most difficult mold design problems.

Mold & Die includes powerful functions for draft creation and analysis, core and cavity separation, mold base definition, shutoff creation, generation of parting line and parting surface, and slide and insert design among others. These tools are organized into a logical, procedural interface to help the designer organize and manage the entire mold design process, from the initial referencing or import of part geometry all the way through generation of the layout and detail drawing package. When completed, the 3D mold design is accessed directly by the 3-axis CAM package for machining the mold further improving productivity.

ZWCAD Software Co., Ltd. worked closely with an international team of experienced mold & die makers during the design and development of Mold & Die. The functionality and expertise captured represent the cutting edge of plastics engineering and a genuine understanding of the mold making process.

ZW3D FabDesign (Sheet Metal)



Offering a comprehensive set of tools for creating standard **sheet metal** features, FabDesign is an extremely interactive and easy to use solution for the design and documentation of chassis, brackets, housings and other fabricated sheet metal components and assemblies containing sheet metal parts.

With ZW3D FabDesign, a designer can create a feature-based solid model of a sheet metal part in the context of a full 3D assembly, folding and unfolding the entire part or individual bends at will and adding features to the part in the folded or unfolded state. As the part is designed,
manufacturability is ensured through automatic creation of bend allowances and relief. As a fully integrated ZW3D application, the FabDesign user leverages all the benefit of the ZW3D powerful assembly modeling, including interference checking and configuration management.

Although the FabDesign environment is optimized for sheet metal design, users are not restricted, as with some systems, to only sheet metal specific tools. The full complement of parametric hybrid modeling functionality offered by ZW3D is always available for difficult design tasks.

Process plans can be defined in the layout by showing views of the part at various states of bending.

Chapter 1 Starting a Basic ZW3D Session

If this is the first time you have started the ZW3D program you will be asked which units you want to work in. For right now let's answer "Millimeters" because all the training lessons are in mm units.

This not only sets your default units but adjusts a number of default menu settings.

When you have completed these lessons, if you would rather work in inches go to the windows explorer, find your user directory and delete the _config file. The next time you start ZW3D you will again be asked this question about units. See the ZW3D Install Guide for the location of your User directory depending on which OS you are running.

The first time you enter ZW3D, a slide show of **Quick Tips** is displayed. This will give the first time user all the basic tools he/she will need to start building geometry.

You can review the "Quick Tips" at any time by clicking on "Quick Tips" to the left of the HELP
pulldown. You can click the button which is highlighted from picture below to get the hided menu.

There are other options that should be reviewed before proceeding with this training, such as QuickTips and Show-n-Tell.

In addition to the Quick Tips, there are also Highlights of the current version and several tutorials under Show-n-Tell™. All 3 are available in several native languages. If yours is not available, contact your local reseller.

Show-n-Tell™ tutorials are self paced training aids that should be reviewed at this time.

If you want to run the tutorials more than once, DO NOT SAVE the Show & Tell Files.

Show-n-Tell™ is built into ZW3D CAD/CAM for on screen step-by-step lessons that work like a 3D book. Users can step back and forth through tutorial steps with a click of a button. All instructions are on screen eliminating the need to flip back to a video or hold a book open. 3D models, showing the expected result, can be rotated and zoomed, giving the user better feedback and understanding. Users work at their own pace, not that of an instructor, so no one gets left behind.

There are 4 built in lessons. The **Open** option above right lets you browse your computer for addition lessons that you previously downloaded from the ZWCAD Software Co., Ltd. web site.

Use the **NEW** or **OPEN** commands to begin creating a new ZW3D project file. ZW3D Project files are given the file extension ".Z3". A project file can contain multiple ZW3D objects such as Parts, Assemblies, Drawings, 2D Sketches, Equation Sets or CAM Process Plans.

Once you have launched your ZW3D Software, (double-click the icon on the desktop) use the **OPEN** form shown below to specify the name of a new or existing project file you would like to work on. ZW3D will create the project file if it does not exist or if it is not in the ZW3D search path. The search path can be adjusted to meet your requirements. You can find more information in the HELP Manual.

Use the Create New File form to specify a unique name for the type of ZW3D object you are going to create. You can choose from a template when creating new ZW3D objects. Templates are standard ZW3D objects contained in the file "*Templates.Z3*" by default.

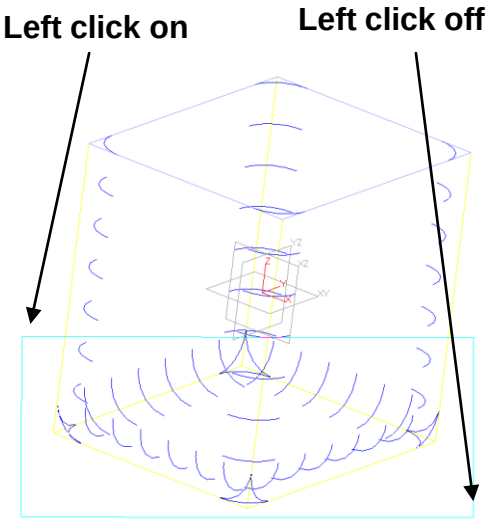
Templates.Z3 is a typical ZW3D CAD/CAM file with root level objects containing your company's design and drafting standards, or manufacturing strategies. Microsoft WORD uses "dot" files as templates. We will talk about adjusting this template file to meet your requirements later in this manual.

When using the 'Open a file' command inside of ZW3D, after browsing for the file, the ZW3D Objects form if the file has more than one root object. Use this form to create a variety of new objects or edit existing objects.

Mouse Button Functionality for Object Editing	
 Left	**Single click** - picks entities and adds them to an active list. **Double click** - selects an entity and automatically invokes a default command to process the entity.
 Middle	**Single click** - Accepts the active list (from the single-left-click) and displays a menu of options for processing the list. **Roll the wheel** - to Zoom **Roll the wheel** - to change a value such as the radius from 5 to 15. **Click - Hold and Drag** – will dynamically pan/scroll the current view.
 Right	**Single click (With an entity highlighted)** - picks the entity and displays a menu of options for processing it. **Single click (With no entity highlighted)** - displays a default menu for inserting/creating objects and various other options. **Click - Hold and Drag** – will dynamically rotate the current view.

Definitions

Window pick: takes 2 left clicks defining the diagonals of a window. It is used to pick many entities with a minimum of clicks.



Entity Filter: helps you select the correct entity type. An example would be if you want to erase a face from a solid shape. You would need to set the filter to face so you don't erase the entire shape.

RMB = Right Mouse Button.

RMB Filter = Context sensitive menu.
With the cursor over an entity this filter gives you a list of commands or options that can be used on that entity.

Chapter 2 The First 5 Minutes

Let's take ZW3D for a quick test drive. We'll create a new ZW3D file and a new part at the same time. Then we'll use a sketch to create a simple extrude shape. After viewing the part in different display modes and learning about Dynamic View, we'll save the file. You now have the keys. Let's go for a ride.

1. From your desktop (or from your user directory using Windows Explorer) select the **ZW3D.exe** icon  to launch ZW3D.

The ZW3D splash screen will appear while the program loads. Now you need to create a file to work in.

2. Pick the **Create a New File** icon  or press (Ctrl+N) to display New File/Object Form shown below. Pick New Assembly Mode.

3. This form allows you to create a new file and a root object (e.g., part/assembly, drawing sheet, etc.) at the same time. The option available will depend on the software bundle you have purchased.

As you can see, the **Part/Assembly** icon  is selected by default.

Users can choose whether use a template or not.

Type **TEST_DRIVE.Z3** in the Unique Name field and optionally give it a Description.

This form may look different from yours. Configuration settings and available licenses will affect which icons can be selected.

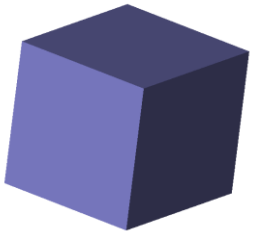
Pick **OK.**

Now we can create geometry.

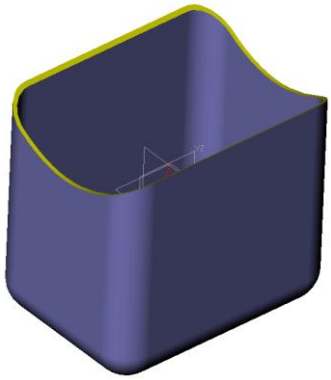
The three default planes that appear in the center of the screen give you a reference to the world coordinate system. The center is defined as (0,0,0).

We will start by using a solid primitive entity.

4. Pick the **Block** icon from the **Shape tab**. 
For the first point, pick the origin at (0,0,0),
…..Pick the second point so the Block almost fills the screen.
Middle-Click or pick **OK.**



Block



The final result

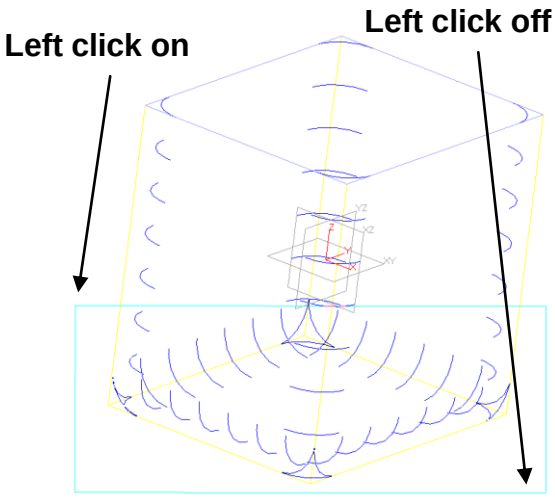
If you want to view your part in **shaded** mode click this icon. 

It is a toggle; so click again to switch back to wireframe mode. 

5. Add a **15mm fillet** to all edges, except the top 4. 

Pick the edges as shown in the image at the right. Use a window pick for speed.
*****NOTE:** Window pick in ZW3D takes 2 clicks. Click the MMB (Middle Mouse Button) when done and roll the wheel on your mouse to increase the **Radius from 5 to 15.**

6. Pick **OK.**



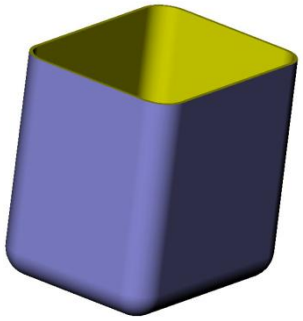
7. We are now going to hollow out the shape and leave the top open as shown below. Pick the **Shell** command also on the **Shape tab** to create a 2mm wall. 

Pick the shape.

Move the mouse to the text box of "Thick". **Roll the mouse** until you have defined a thickness of **-2.0** on the menu.

(-2.0 creates the wall on the inside of the shape. A positive number would make the exterior of the shape grow.)

Middle click to advance to the next prompt.
Pick the top face to leave open.
Middle click to end that selection.
Middle click or pick **OK** to end the command.



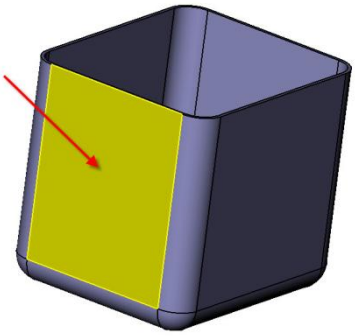
If you look at the message window you will see the following:
WARNING: Operation may fail if thickness is greater than fillet radii.
It is just a warning but pay attention to all messages in this window.

8. Now we are going to contour the open end of the shape with a surface created by extruding a freeform sketch curve.

Pick **Extrude** from the **Shape tab**. 

Middle click to create a new sketch.

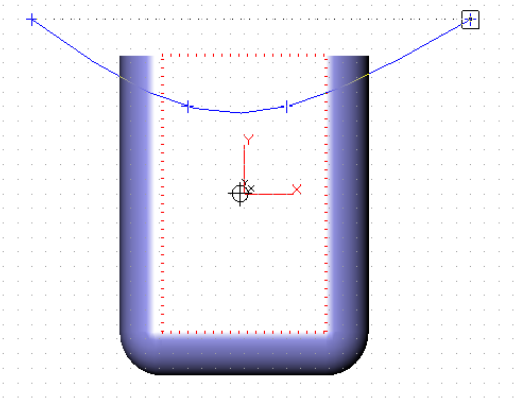
Pick the face indicated in the image below.



9. Pick the **Through point curve** command from the **Wireframe tab**. 

Pick 4 points similar to the image below.
MMB to end the point selection.

MMB to end the command.



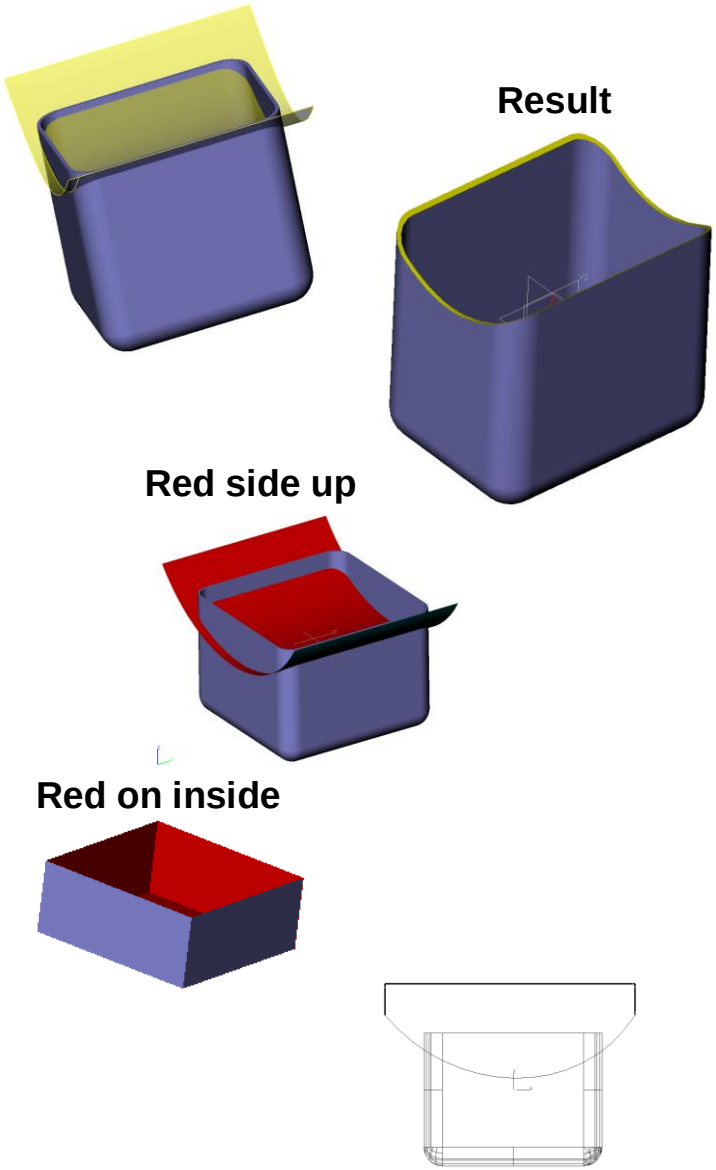
Right click and pick **Exit Sketch** or… 

10. You are returned to the **Extrude** command.

Pick the **Remove** option at the medium of the menu. 

Drag your cursor and pick the **Start** and **End** points so that they completely cut through the shape as shown below.

Hold down the right mouse button and drag to rotate the model. This may help you see if you are going to cut the entire part



To explain briefly why we chose the remove option, **Right-click** and pick **Redefine last**.

Pick the **Base** option and **OK.** 

Hold down the right mouse button and drag to rotate the part again and notice one side is red/pink and the other is the solid face color. ZW3D uses red/pink as the default color for the back side of a face or the inside of a solid. This can be easily demonstrated by inserting a block out in space then erase the top face.

If you think of our surface as being the bottom face of a solid, we would do a remove operation to cut off the top of our shelled part.

File > Save/Close (from the pulldown menu).

NOTES:

In the next exercise we are going to create a wheel

- Create a sketch
- Dimension and Constrain
- Revolve Base Shape
- Use primitive geometry
- Pattern
- Extrude Remove
- Fillet

Chapter 3 Wheel Assembly

1. **Begin a new File/part** and name it **Wheel-Assembly.**

This part will be built in mm units. If you needed to build a part in inches you would go to the EDIT pulldown > Preferences > change the units for that part to inch. All other parts would still be created in mm.

This design is actually a simple assembly of a steering wheel and a handle. We will start by designing the wheel using the "Top Down" design technique.

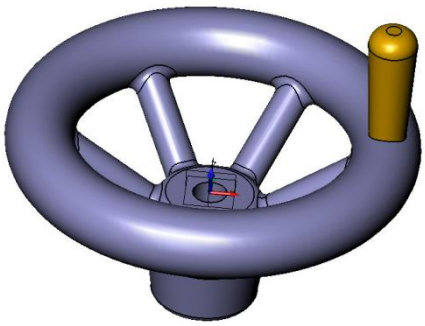
2. **Right-click > Insert Component**

(1. Type the part name in the input field under the folder icon – Wheel,

(2. Check – Anchor,

(3.Type 0 ,

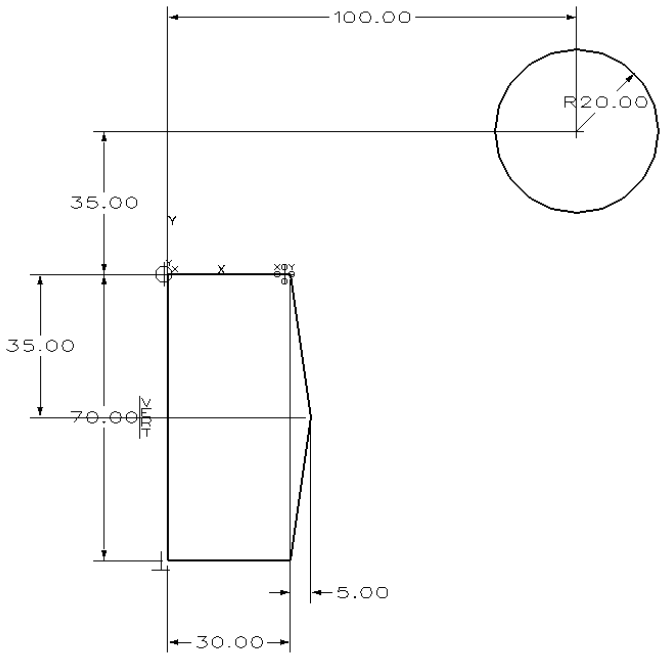
(4. Pick OK.



Notice the title bar at the top of the screen. You are now in a part called Wheel.

3. Pick the **Revolve** command .
Middle-click to create a new sketch.
Pick the XZ Plane for the new sketch.

4. Create the sketch shown in the figure below left.
The dimensions are for reference only.



Draw the circle first. Use the **circle** command from the **Sketch tab** and pick the **Radius** option.

As an aid to locating the center,Hit <ctrl + k> on the keyboard. This will display the coordinates at the bottom of the graphics window. They just need to be relatively close. When you add the dimensions you will make it exact.

Right-click > Draw to create the hub cross section. Notice some constraints are automatically added as you draw each line segment. Start the upper left corner at (0,0) and draw the vertical line.

5. **Right-click > Dimension** and add the **30.00 dimension** at the bottom. Pick on the line and place the text.

6. **Right-click > Constrain.**
   Pick an end point near (0,0). Your screen should look like the above image.

   90% of the commands you will use in the sketcher are found on the **Right-click menu**.

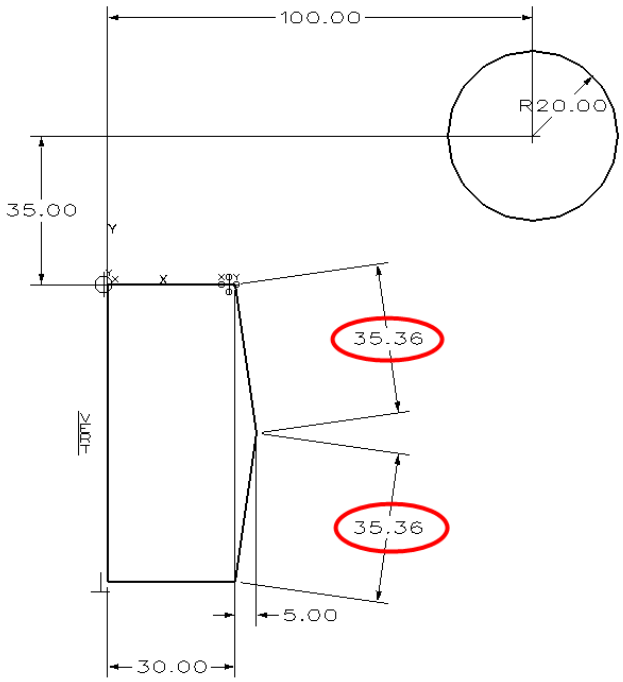
7. Our design intent dictates that we change the dimension scheme a little. Erase the 70.00 and 35.00 dimensions from the hub.

   (You can also **Right-click** on each dimension and pick **Erase.**)

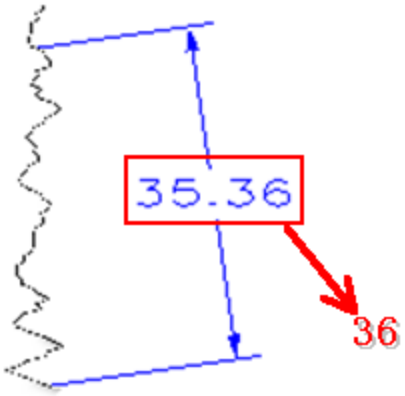
   Open the Constraint tab. Use the **Linear dimension** command.

Pick the third option.

Add the two dimensions shown below.



Double-click on the upper dimension and change the value to **36.00**.



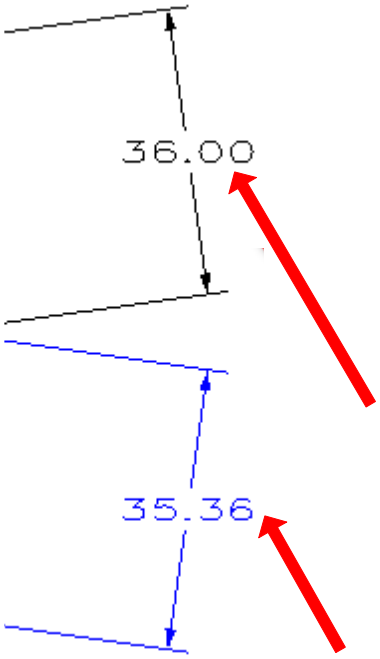
They still need to be equal…

Double click on the lower dimension;

Pick the **Select Dimension** button as shown below. 

Pick the first 36.00 dimension.

Pick **OK.**



Both numbers should be the same. You just wrote a simple equation setting the two dimensions equal to each other.

To see the equation, double-click the dimension again.

We are not going to teach equations at this point in your training this is just for your reference. Pick **Cancel.**

8. **Right-click** and pick **Exit Sketch** or .

9. You are returned to the part level and are still in the **Revolve** command.

Look ahead for a second. The profile, start and end angles have default values so you just need pick the axis then pick OK.

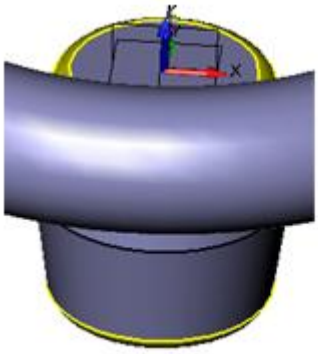
Right-click and Pick **Z-Axis** for the revolve axis,

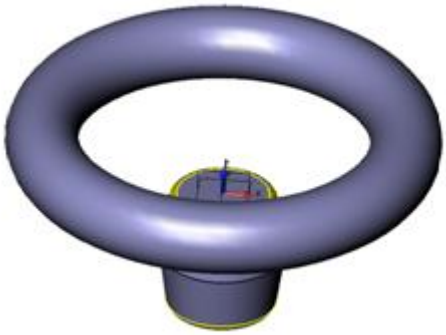
Pick **OK.**

10. Add a **5mm fillet** to the top and bottom edges of the hub.

If on a rare occasion you needed to enter a value in other units, type for example .125.in.

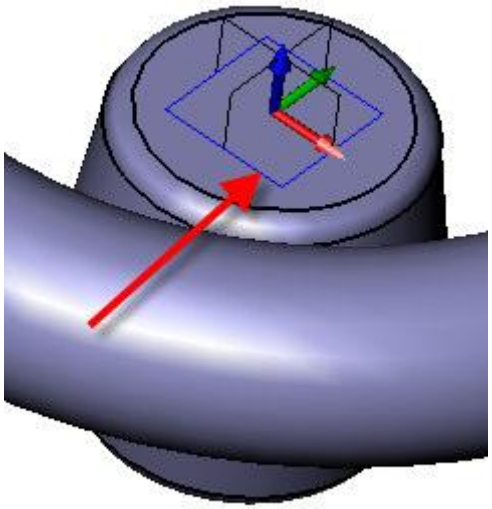
If you anticipate that these edges may have different values in the future, use two separate commands for ease of editing.





11. Let's create a **20mm** diameter **circle** on the top face of the hub as shown below to be used for the mounting hole in the hub in a later step.

Create a sketch on the XY plane. Right-click > Insert Sketch or the Insert pulldown.



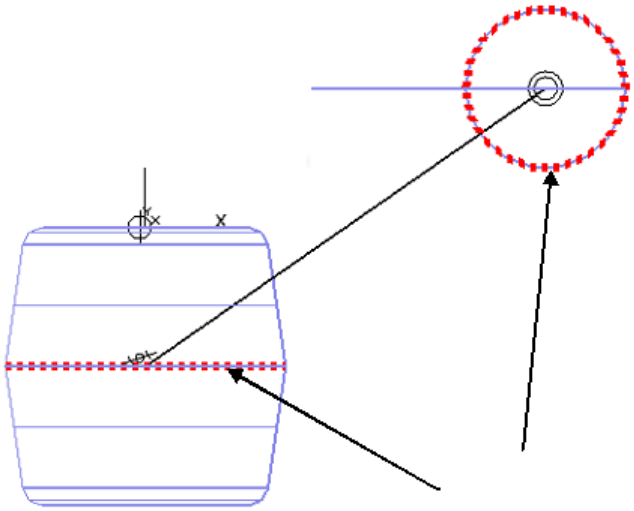
Use the Circle command in the **Sketch tab**, choose **Radius** method and set **Diameter to 20.**

Right-click and pick **Exit Sketch**.

Now for the first spoke.

12. **Insert a sketch** on the **XZ** Plane.

From the **Sketch** tab**, draw 2 reference curves** picking the center edge of the hub and the circle shown below. Left click and choose .



Pick the **Show Target** icon. 

This is a toggle. You can click again to see the model.

Right-click > Draw.

Right-click > Critical.

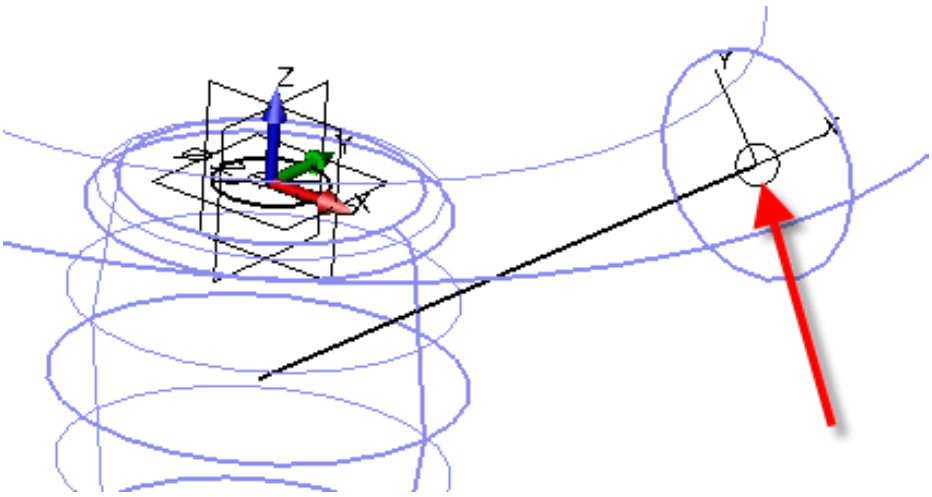
Create the line shown. **Middle-click** to end the command after the second point.

Right-click > Exit Sketch.

We need a datum plane to orient a cylinder that we will use for the spoke.

13. **Right-click** and pick **Insert Datum.**

Pick the end point of the new line as shown below. (**Critical** pick) Pick **OK.**



We are going to draw a cylinder and we want it to be the same length as our line and lined up with the new datum.

14. From the **Shape tab** pick the **Cylinder.** 

Use **Add** method. 

For the **Center**, pick the lower end of the line(at the center of the hub).

Choose and set **Diameter** to 20.

For the **Length**, Right-click >Length and pick the two ends of the line starting at the center of the hub.

For the **Align plane**, pick the datum we just created. Notice the cylinder orients itself to the datum.

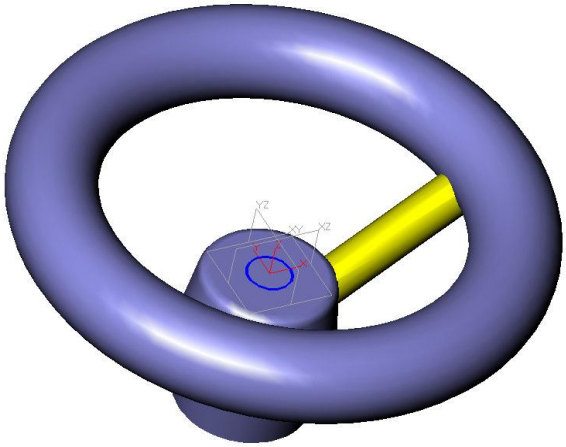
Pick **OK.**

15. This would be satisfactory in some engineering departments…..but none that I have worked in.

We need more spokes…

On the **Shape tab** select the **Pattern** command. 

Then pick the **Circular** option. 



Change the **filter** to **Feature,**

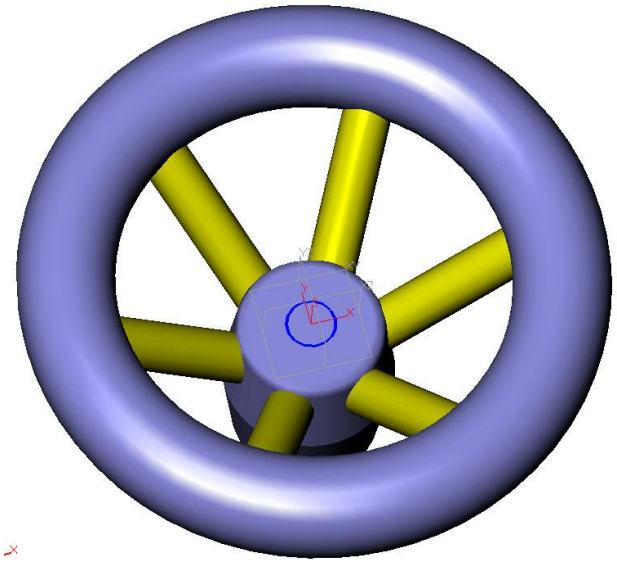
Pick the spoke (Cylinder) as Base,

Right-click and pick the **Z-Axis as Direction,**

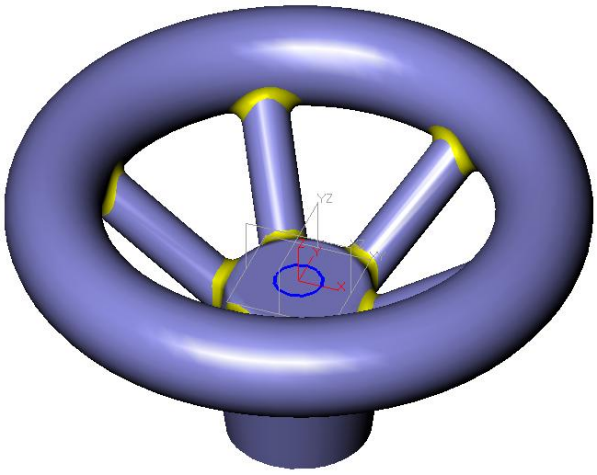
Make **6** spokes(**Number)** at a spacing of **60** (**Angle**)degrees,

The last two options are not needed here; just ignore them.

Pick **OK.**



16. **Add a 5mm fillet** to all edges using **right-click > Pick All**.



17. Let's use the circle on the top of the hub and cut a hole out of the center.

Pick the **Extrude** command from the **Shape tab**. Pick the **Remove** option.

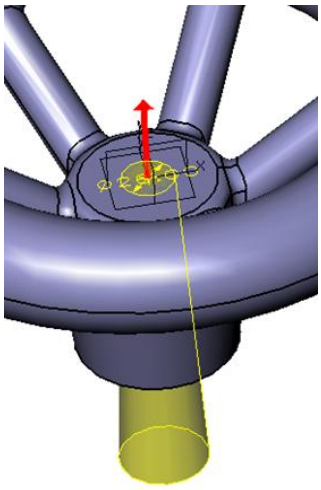
Select the circle created in step 10 as the profile.

You can **middle-click** to accept **0** as the start point.

Drag your cursor below the hub, **right-click** and pick **Through All.**

Add a **Draft angle of -3.**

Pick **OK.**



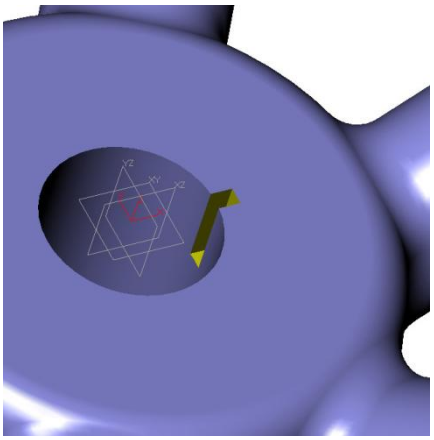
18. A keyway is required to keep the hub from spinning on the shaft.

Let's use a primitive block. Pick the **Block** – **Remove** command. 

(Right-click) **Critical** pick the point on the edge shown adjacent to the X-axis. Pick anywhere for the second point then edit the form as indicated below.

X length =5, Y length = 2, Z length = 20.

Result:



Hold down the F3 key and the right mouse button to rotate the view about the centerline of the hub.

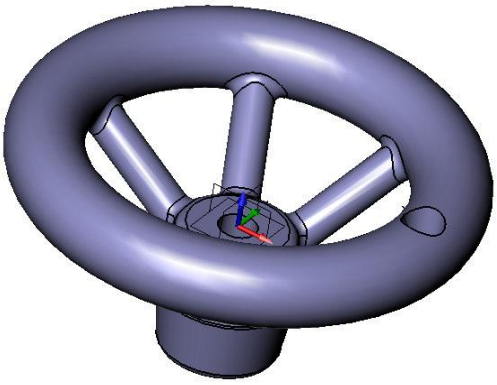
19. The final operation for our wheel design is to make room for a crank handle.

Use the primitive **Cone**  as a **Remove** feature. 

Critical pick the end of the sketched line which should still be visible.

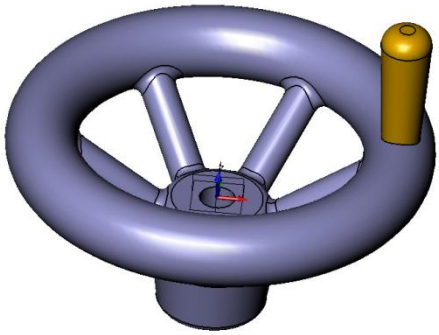
Select the **Diameter** button, set **1st Diameter** to 20, **Length** to 90 and **2nd Diameter** to 30. Pick **OK.**

Result :



Save your file. 

The Wheel is complete. We will now build a handle as shown below.



20. **Right-click > Exit part.** You are now back in the **Wheel-Assembly**.
You need another component.

21. **Right-click > Insert component.**

(1. Type – Handle,

(2. Check – Anchor,

(3. Type 100,0,35 or pick the far end of the spoke centerline.

Notice the title bar at the top of the screen. You are now in a part called **Handle**. Do you know why you can still see the Wheel?

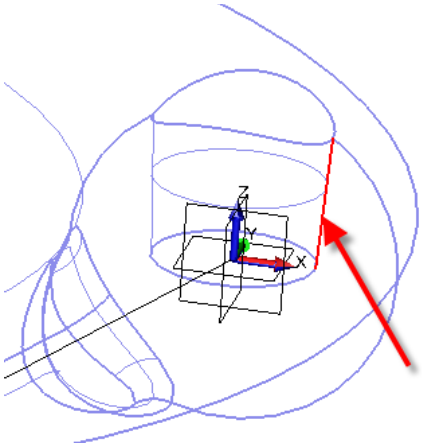
22. Pick **Show Target.**  You are currently in the Handle but since we are building this in the context of an assembly you can see the entire assembly and can use any geometry in the wheel as a reference for the handle.

23. Pick **Show All**. 

We will reference the edge of the hole that we created in the wheel to start the handle.

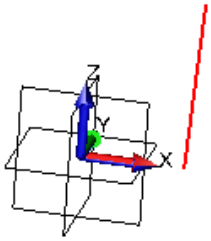
24. Pick the **Reference Geometry** command from the **Assembly tab.**

25. Pick the **Curve** option and select the edge shown at the right.



By default，the option of **Record State** has been picked. Make sure that it's not been picked.

26. Pick **Show Target** again.



You are looking at the reference curve that now exists in the Handle. Since it is a reference curves, if the wheel dimensions change the handle will update accordingly.

The handle is going to be 90mm long so we will create a datum plane offset from XY.

27. **Right-click > Insert Datum,**

Pick the **XY** plane,
Specify an offset of **90.**

28. Pick the **Reference Geometry** command from the **Assembly tab.**  **Click**  **.**

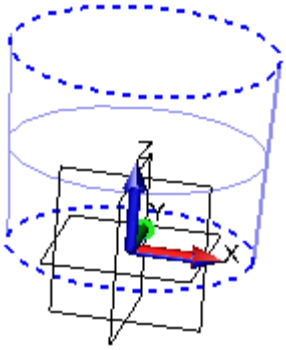
Pick the bottom of the cone that you have created on step 19 and unpick the State Record.

29. Revolve the reference curve about the Z axis of you lastest created,

Open the Shape tab,

Pick **Revolve,**
Pick the reference curve,
Pick the **Z axis** of you lastest created,

Right-click > Plane Axis > Pick the reference you lastest created > Z axis;
Pick **OK.**



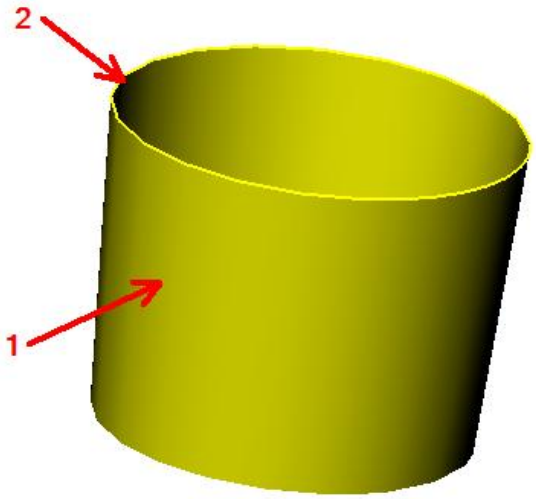
In the wireframe mode you can see that you have one revolved surface open at the top and bottom. Open surfaces are indicated by the dashed font.

We need to extend the top edge.

30. Open the FreeForm tab.

31. Pick **Extend Face**. 

Pick the face with number 1 arrow,
Pick the edge with number 2 arrow,
Right-click > pick **distance,**
Right-click > **Critical**,
Pick the **center of the top edge(with number 2 arrow) and the new datum,**
Middle-click,
Hit OK.

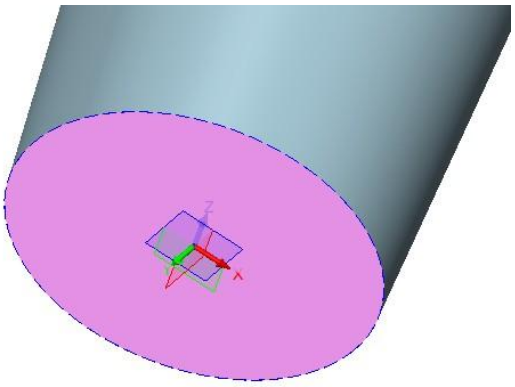


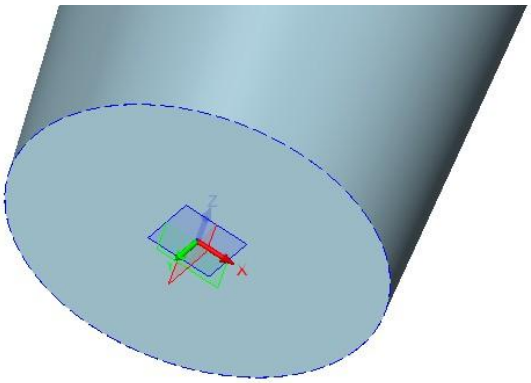
Let's close the shape into a solid.

32. Open the **Heal tab,**

33. Pick **Fill Gap**, select the top edge and hit **OK.** 

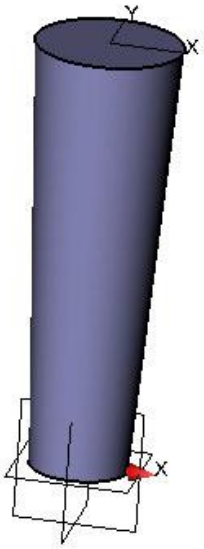
Notice that the border of the referenced bottom face is blue dotted line. It shows the open edge, in other words, the bottom face dose not connect to the side face. We also notice that bottom face orientating outside is red. We can change the surface direction by using the **Reverse Direction** command  in the Edit Face Tab . As we can see below.





Then select Heal Tab and pick the **Sew** command. We can notice that blue dotted line changes to black.

You now have a solid and the correct length.



Add a fillet to the top edge.

34. Open the **Shape tab,**

    Pick **Fillet,**
    Pick the top edge,
    Set the **radius** to about **10.**

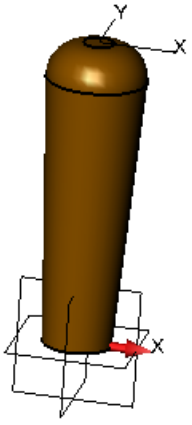
    Pick **OK.**

35. Change the color of this shape,

    Put the **entity filter** on **Shape**,

    Right-click > Face Attribute on the geometry,

    Pick yellow or brown.

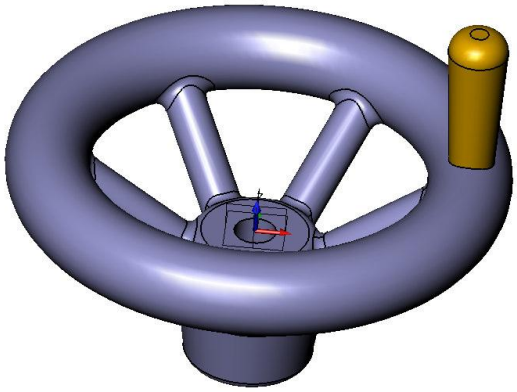


36. **Save** your file. **Exit** part.

Notice the title bar at the top of the screen. You are now back in the Assembly.

After each command below, check the title bar to verify your active part.

The **Assembly Manager**  shows that you have inserted 2 components.



37.Right-click on either component. The context sensitive menu shows the commands that you can issue on an assembly component.

The first one in the list is the default command. That means if you double-click a component instead of right-click, this is the command that will be issued.

-Pick **Edit Part.**

-Use **Show Target** to see the target part.
May have to click twice.

-Open the **History manager**. 

38.-Open the **Assembly Manager** again.

-**Double-click on the Handle.**

You can navigate directly from one component to the other without going back to the Assembly.

This last sequence of commands is the common way you will use to navigate from the assembly to the piece parts and back to the Assembly.

Let's look at one more option before we make a change to the design.

The other one I will talk about right now is **Auto Regen**.

39. **Right-click on the Handle.**

Pick **Auto Regen.**

Use this command to set the **Auto Regen** status of a component. Select one of the following options shown below. The checkmark symbol  will appear next to the child component node symbol in the **Assembly Manager** for the component.

- **None**
  The component is not regenerated when its parent is regenerated.
- **At insertion**
  The component is regenerated when the operation that inserts it is regenerated. Components will be regenerated sequentially according to their order of insertion. Use this option if the component pulls geometry from the Parent part into the Child. This is the typical way an assembly should be structured; sequentially from top to bottom of the history.
- **At end**
  The component is regenerated at the end of the regen of its parent part/assembly. Use this option if the component is pulling geometry from a Child Component into the Parent.

40. Pick **At Insertion.**

Let's now make a change to the wheel and see how it is propagated to the handle design.

41. Double-click on the Wheel from the **Assembly Manager.**

Use **Show Target** to see the target part.

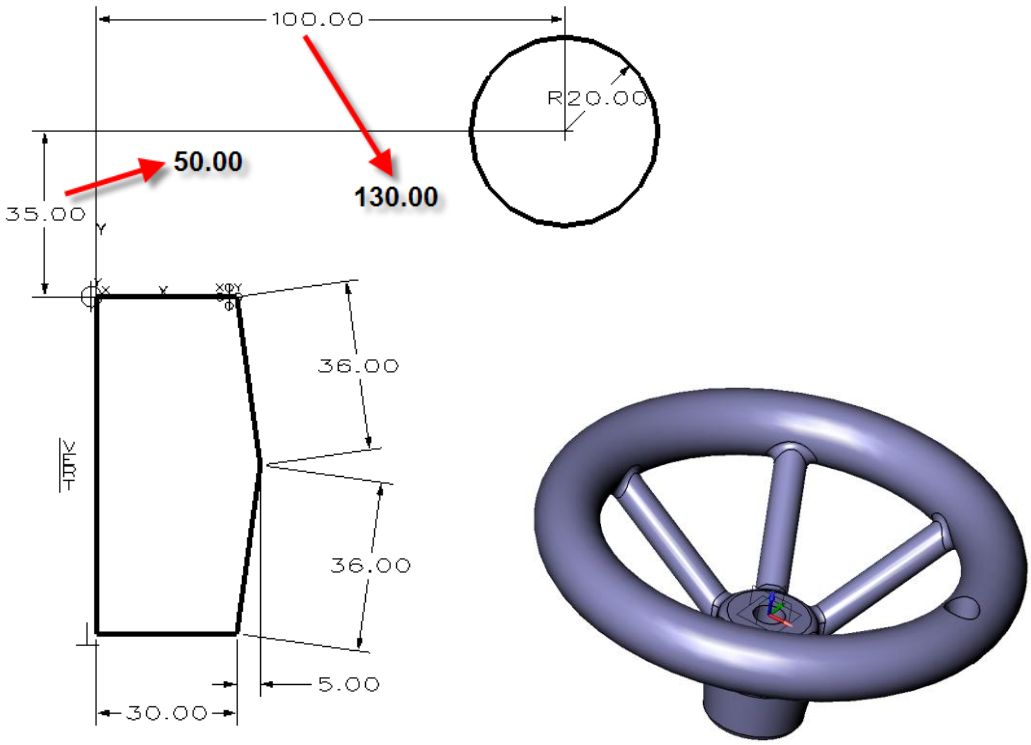
Open the **History manager**.

Redefine the Initial sketch.

42. **Right-click on Sketch1** in the history and select **Redefine.**

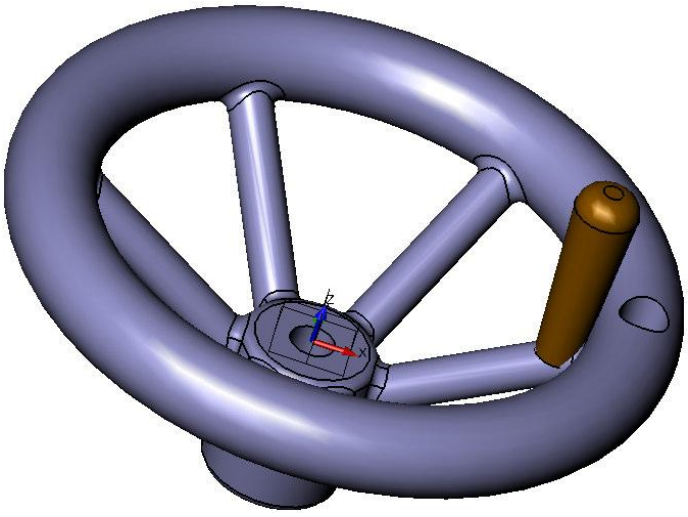
If all of the dimensions are not visible, unblank them. 

Make the two changes shown below by double-clicking on the dimensions.

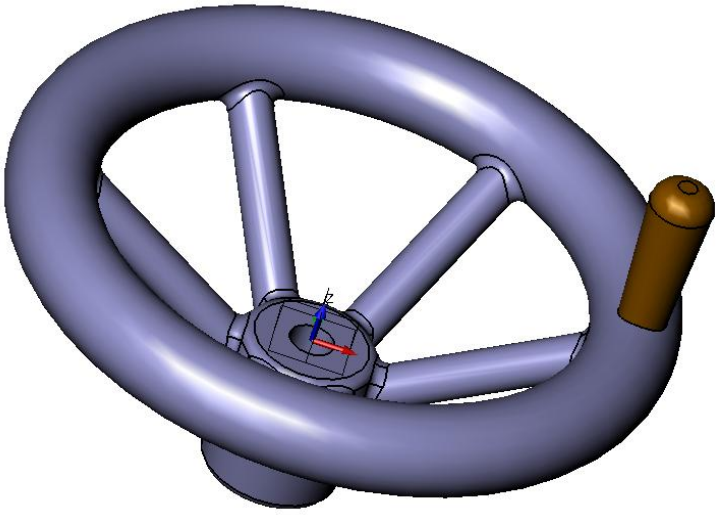


Exit the sketch and the history will finish regenerating.

Exit the Wheel and you will be back in the Assembly.



Regen the history of the Assembly and the **Auto regen** of the handle will automatically rebuild the handle with the new reference geometry from the wheel.



Save your file.

If you have the **ZW3D Designer Software Bundle** or better, you can use the **Link Manager** under the **Tools pulldown** to extend the **Auto Regen** capability to other objects in your file such as 2D Sheets and other external files where you may have used these components. See the CAD Fundamentals Training Manual and the HELP DOCUMENT for more information.

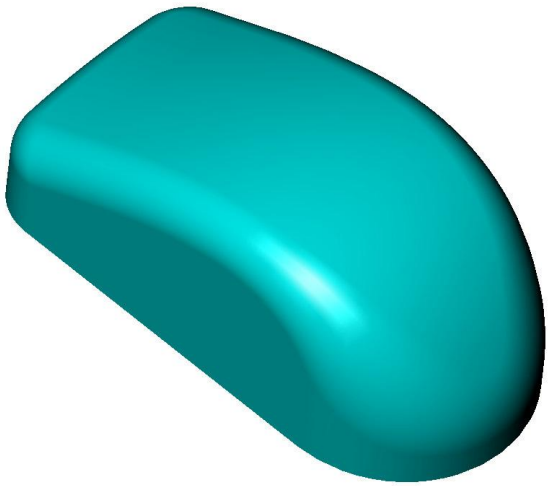
NOTES:

In the next exercise we are going to create a Mouse using some hybrid modeling techniques.

- Sketch
- Extrude
- Sweep
- Surface
- Combine shape > Intersect
- Draft Fillet Shell
- Combine shape and divide shape
- Extract geometry to component
- Creating a drawing sheet

Chapter 4 Hybrid Mouse Exercise

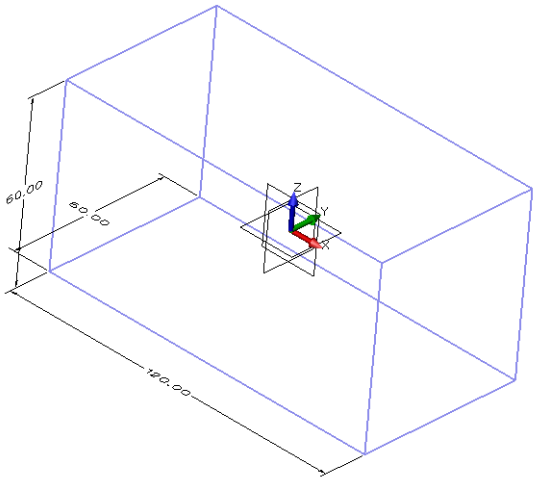
This is another assembly. We are going to take a different approach. It is still considered a Top Down approach but we will just build individual shapes and separate them at the very end.



1. **Begin a new part** and name it **MOUSE.** Notice: Not allow new part at New Assembly Mode**.**

2..**Pick the Block** command,

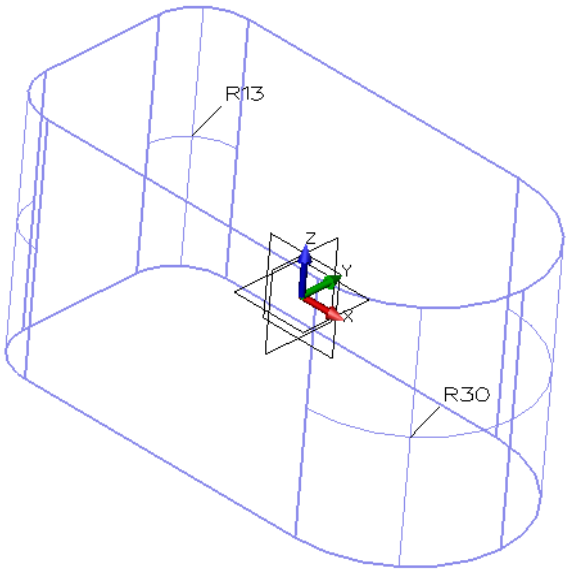
Center it on **0,0,0,**
Pick the 2nd point anywhere then
change the
X length = 120,
Y length = 60,
Z length = 60.



3. **Fillet** the four vertical edges as shown.

The block is 60 mm wide so a 30mm Radius on the two edges closest to you creates a full radius.

HINT: I always change the radius value before picking the edges if I know the exact value I want to use so the preview is correct.



We are going to sculpt the top surface. To accomplish this we will create 2 sketches and sweep one along the other.

4. **Create a sketch** in the **XZ** plane.

From the **Sketch tab** select the **point curve** command,

Type the following coordinates, each followed by **<ENTER>,**

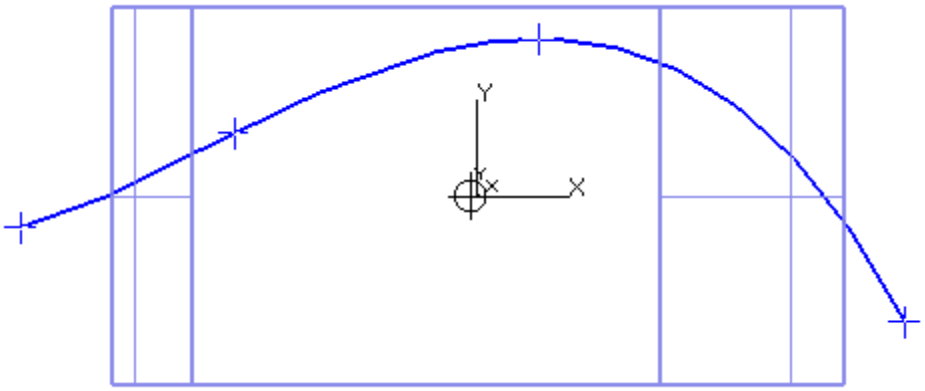
-75, -5

-40, 10

10, 25

70, -20

Pick **OK.**



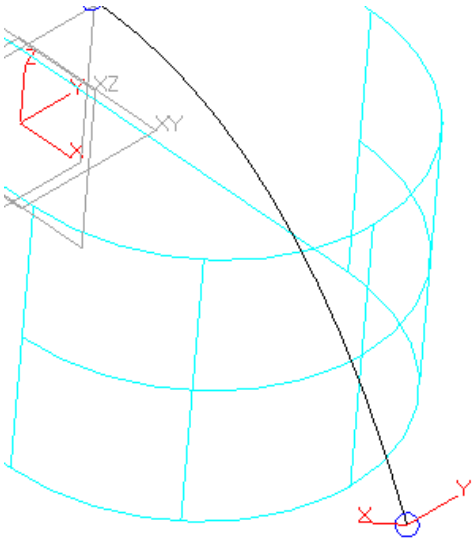
You should have a curve that looks like the above image.

5. **Exit the sketch.**

6. **Right-click** > **Insert Datum.**

Select the critical point at the left end of the previous sketch.

Pick **OK.**



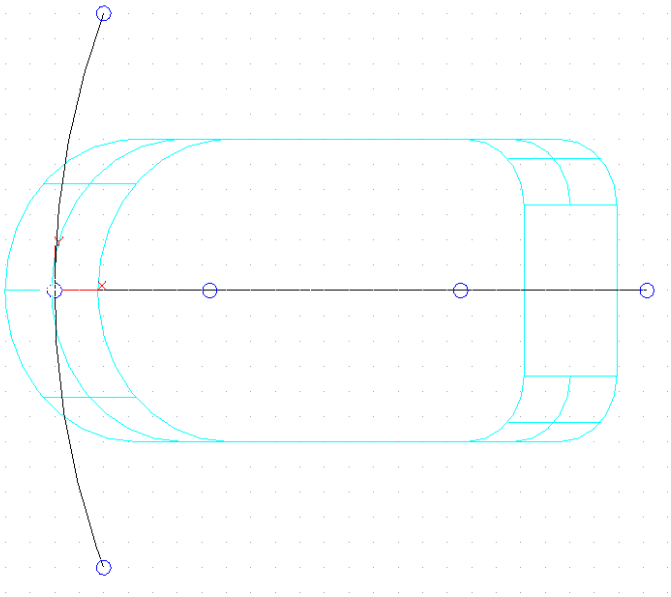
Insert a new sketch on the new Datum.

7. Create a **3 point arc** picking the Start, End, then the mid point.

Use the following points.

10, 55
10, -55
0,0

You should have a curve that looks like the image at the right.

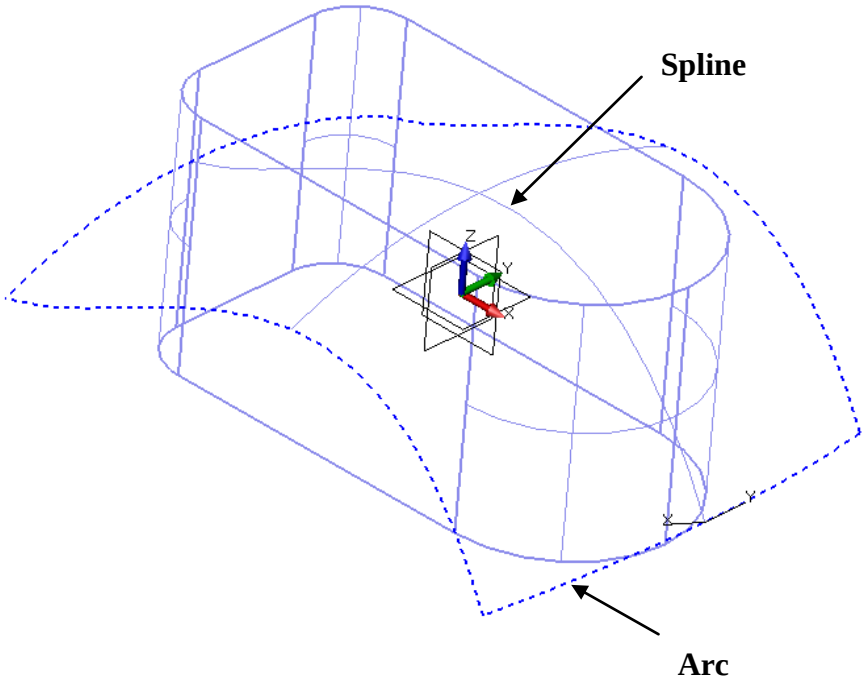


8. **Exit** the sketch.

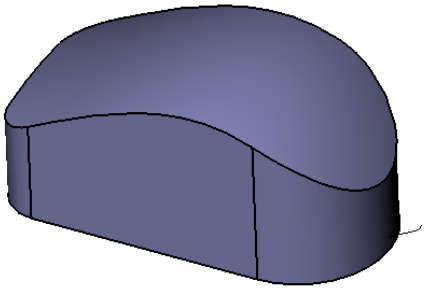
9. From the **Shape tab**, pick the **Sweep** command. 

Use the **Base** option. The profile is arc and the path is the spline. 

Pick OK.



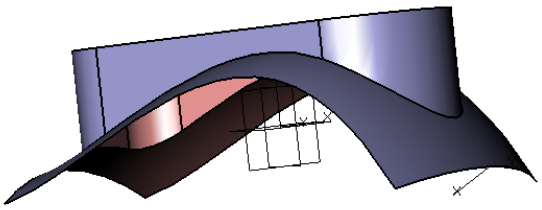
Now that you have created the sculpting tool there are a few commands we could choose from, only one is going to give us the desired result.



Let's start with **Combine Shape** > **Add.**

Combine shapes icon on the **Shape tab.**

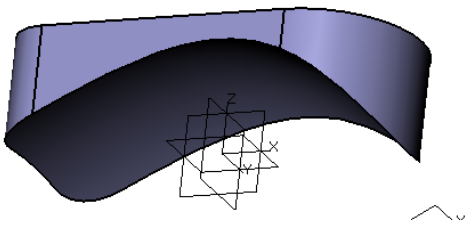
Make sure you uncheck the **Keep Shapes** option.



Undo.

Combine Shape > **Remove** **.** (Slices off the part of the Base shape facing the pink side of the surface)

Pick the block first then the surface. You can also try picking the surface first then the block.



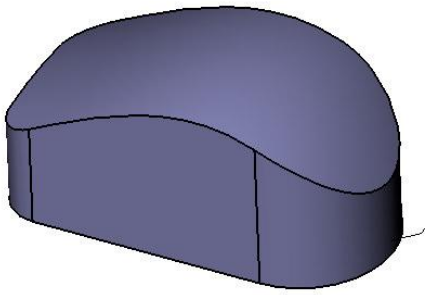
Undo.

None of these operations is what we want.

10. **Combine Shape** > **Intersect**. This is what we will use.

The pick order is not important for this command.

Pick **OK.**



In the last few steps we are going to:

Draft the part about a plane,
Fillet the top edge,
Shell the housing,
Split part into two,
Make a separate object for each of the new shapes.

We need to define a datum plane before we can add the draft. Our design intent calls for the parting line of our model to be 13 mm below the XY plane.

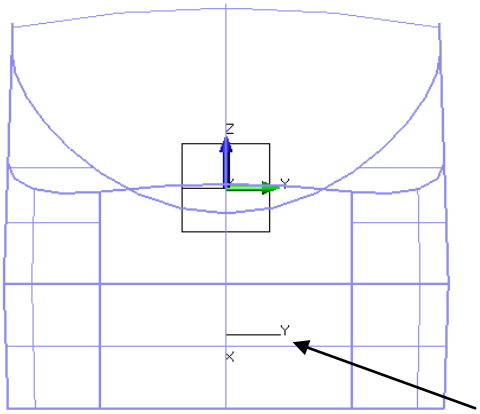
11. **Right-click** - **Insert Datum.**

Pick **XY** plane, and set **Offset** to -13. 

Now we need to add draft.

12. Select **Draft** from the **Shape tab**  , then select **Shape**  **.**

Pick the new datum to Draft about… (may need to change the filter)



Use a **Draft angle of 2.**

Make sure not picking the **Recompute fillets** option.

The **Side S option** should be set to **split.**

You can try the other options to see their affect.

Try a negative angle. Use a larger value when testing so the angle can be more easily seen.

Pick **OK.**

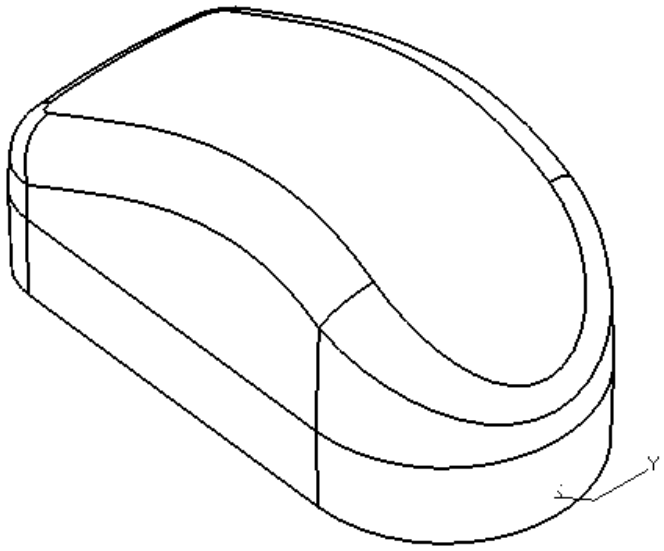
Now we are going to run a fillet around the top edge of what is starting to look like a mouse.

13. Pick the **Fillet** command.

Put a **10mm constant radius fillet** all the way around the top edge.

Hold down the <**Shift**> key and **pick** an edge on the top surface. This will chain pick all tangent edges.

Pick **OK.**



Let's Shell this part then split it into separate top and bottom pieces.

14. From the **Shape tab** pick **Shell**.

Use a **thickness of -3.**

There will be no Open face on this one.

15. Select **Divide shapes** under the **Shape tab.** 

Pick your **Part** then the **Datum** you used for the Draft command.

You now have two separate but closed shapes. In other words, you have two solids.

There are several ways to verify this. Put your **filter on Shapes** and window pick the part.

The **Text Input** window will show **2 picked**. Or you could temporarily blank one piece. Or temporarily move one.

16. Change the color of one or both pieces.

With the **filter on Shape**. **Right-Click > Face Attributes**.

You now have two shapes that are obviously associated to each other. For purposes of this exercise we will assume the design of the insides is complete.

If you were going to make detail drawings you want these two shapes to be separate ZW3D objects like we did in the last lesson.

17. My design intent calls for 3 holes in the base and a slight change to the part thickness.

At this point we will create a sketch on the bottom face of the base and insert three points for the hole locations.

Hold down the Right Mouse Button and rotate the part so you can see the bottom face.

Right-Click > Insert Sketch.

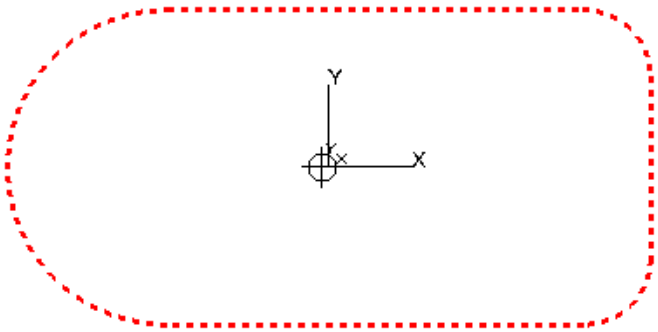
Pick the bottom face and Reference face edges.

Pick **OK.**

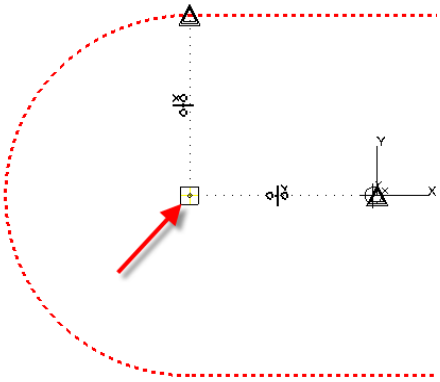
Creating sketch to locate multiple holes

We just need 3 points. Create them at the critical center of the 3 arcs.

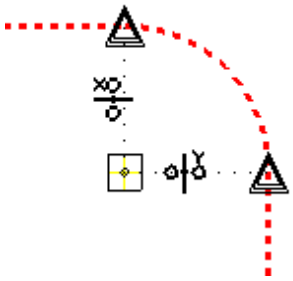
Click on **Show Target** so we can see just the reference curves.



The first point should be located as shown right. Let the automatic constraints line up the point.



The other two points can be located in a similar manner.



Pick **OK.**

Exit Sketch.

Blank the **Cover** of the mouse by using the **blank** entities command  . Set **filter** to Shape and pick the **Cover** as the entity to blank.

Pick the **Hole** command from the **Shape tab**. 

Pick the face, pick the bottom face of the mouse.

For the location, **Right-Click** in the back ground and pick **Sketch** from the menu.

Pick the Sketch just created from the History List. You are returned to the command. Change **Blind** to **Thru-All**.

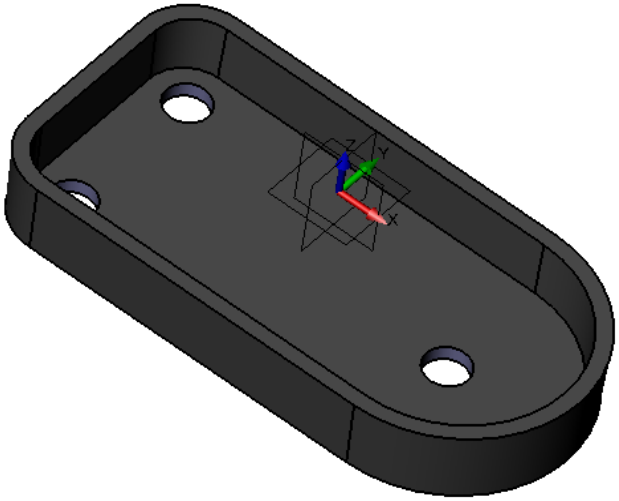
Pick OK to accept the rest of the default settings.

You can finish replaying the history.

Pick **Replay all operation.** 

The two extract commands will re-extract the two revised parts and all downstream applications will also be updated. We will see our drawing in a minute. If you had a CAM Plan for either of these parts it would also update.

Exit Part.
Double click on the Base to verify that we indeed have holes in the bottom face.
You can check the Material and Part Attributes to see that they have not changed.



18. Adding Material Attributes

Pick Tools > Material Attributes. 

The Material needs to be specified in the Title Block and is also used in the Mass Property calculation.

First select Bundles.Z3 from **File/Bundles**

Select **Thermoplastic** from our predefined **Material Attribute Bundle**s.

select the shape then click **Apply**.

Click **OK.**

With your **filter on Shape**, **Right-click** on the shape and pick **Entity Infor** to verify.

User should be able to see that “material: Thermoplastic” and “density(kg/m3): 1210” are displayed.

Calculating Mass Properties

Now that the shape has material attributes assigned you can calculate the Mass Properties.

Pick **Inquire** > **Mass Properties** from the pulldown menu.

If the **Mass Units** are not what you want, change them, then pick the shape.

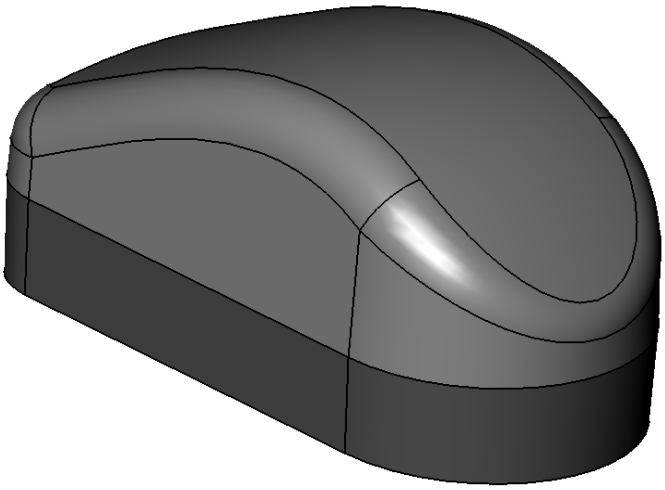
Pick the **Record Inquired Information**

You can see them by clicking on **TOOLS > Variable Browser**.

19. Open the **Assembly tab** and select:

Extract shape 

Pick the **top of the part** and name it **Cover**. Pick **OK.**



20. Repeat this command for the bottom piece and name it **Base**.

Pick **OK.**

Up until now the geometry on the screen has been **SHAPES** either open or closed (solids). You now have **2 Components.**

Put your **filter** on **component** and window pick the geometry. Check **Text Input** window .

It may be a good idea to insert a comment, then rename that history item to: "**Make all edits above this line**"

You can click the button which is highlighted from picture below to get the hided menu.

Insert > Comment.

Right-Click (on the "Comment.1")> Open/Close to show the details.

Exit the part.

Let's look at the object list.

NOTES:

In the next exercise we are going to work with 2D sheets

- Create a 4 view layout
- Change view attributes like line visibility
- Add standard dimension Edit
- Dimensions attributes Add a
- simple section view
- Cut/Paste the section to sheet 2
- Add notes

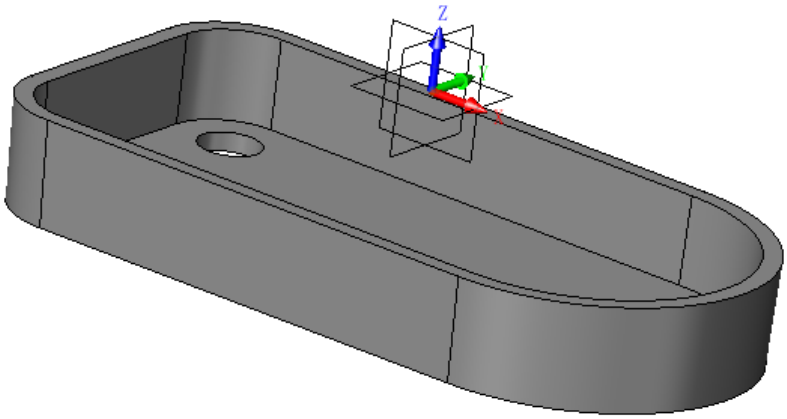
Chapter 5 Making Detail Drawings

You have just created two new objects and they were inserted into our original part as components.

Create a 4 view layout

Double click on the **Base** to open it.

Notice the history has been compressed into one line. Any changes to the individual parts after this must be done in the original part **Mouse** and must be done before the two Extract commands.



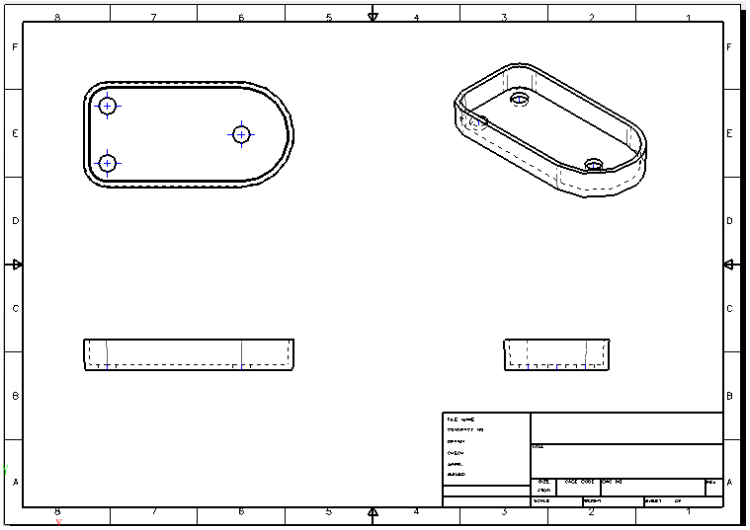
Let's make a drawing with a 4 view layout of the **Base**.

Right-click - 2D Sheet.

Pick a sheet template or hit OK for the default. The default template is defined on the **File** tab of the configuration. （Pick A3 here.）

Click Layout  > Select the **base** component > Pick Iso View  > click OK.

Do not forget to select **Convert to arcs/lines**  option (Edit Curve tab) when laying out a view of a part with draft.



Dimensioning

Before we start dimensioning let me point out a configuration setting you may want to set.

Click on **Utilities > Configuration > 2D tab.**
Check the box of **Highlight modified dimensions** and we will see how it works later.
Open the **Dimension tab.**

The first icon  is the same as the Right-click **Dimension** command.

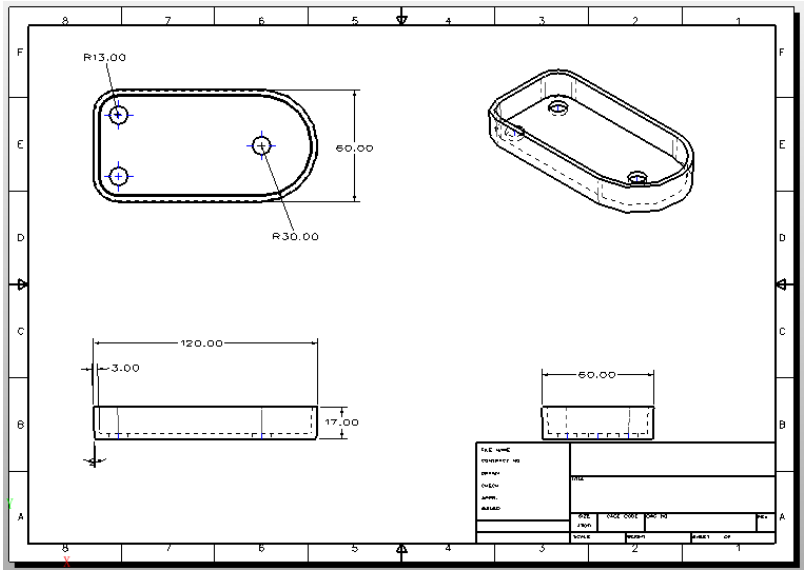
Use this command to create a dimension by selecting an entity or picking dimension points. Six different dimension types can be created with this command depending on the entities, points, and command options selected.

You can use this command to quickly add simple dimensions. You can always go back and edit the dimension attributes or use the dimension commands on the tab to add all dimension types.

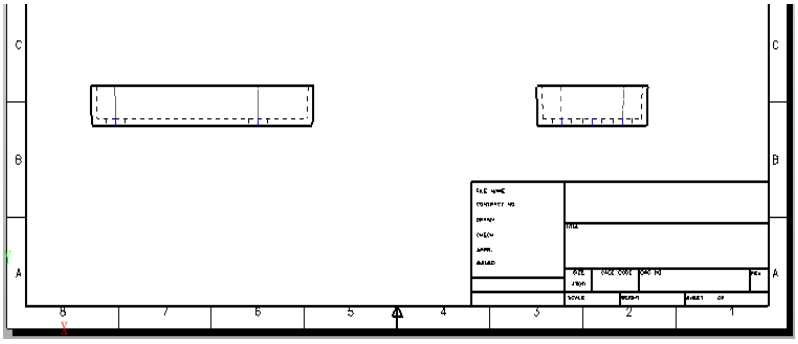
This is a simple drawing but I think it will highlight some of the dimensioning techniques you will need to learn.

Save your file.

Before turning the page, try the **Right- click – Dimension** command and see if you can insert the basic dimensions shown here. Adding the Angular dimension, appended text and the parenthesis will be covered next.



Let's start by moving the two lower views up a little.



Click on both views, **Select Edit Form > Move.**

Pick a **from point** and a **to point** for the move.

You probably won't need this but to guarantee that the move is vertical you can pick the **Vertical Direction** option in Optional Inputs.

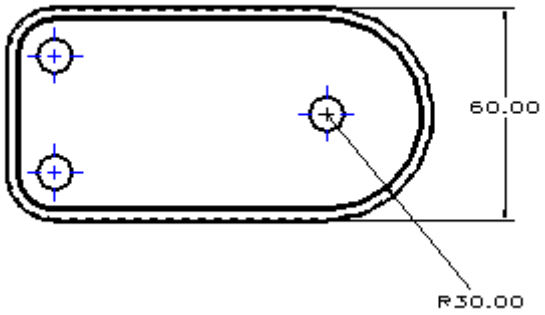
Pick **OK.**

Let's add the two dimensions in the top view.

Right-click – Dimension pick the large outside radius and position the text as shown.

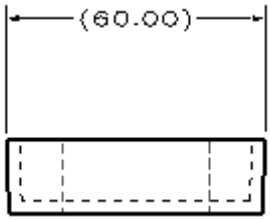
Middle-click to continue.

Pick the two end points and position the vertical dimension.



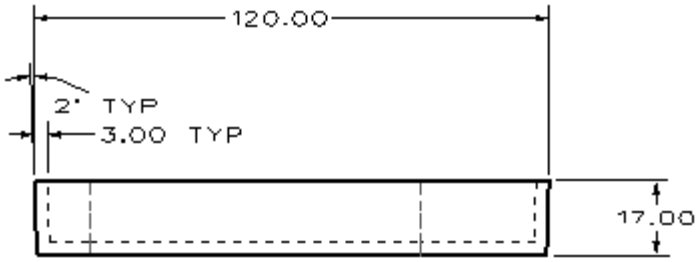
Middle-click.
See the right side view.

Pick the two end points and position the text.



You can continue picking end points for the **120** and **17** dimensions.

If you have left the command, you can middle-click to get back into the **Dimension** command.



Before we turn to part Linear Offset Dimension， maybe we need to make all hidden lines visible.

Right-click on the front view and select **Attributes**.

Choose **Lines** tab, pick the **Hidden** option below and set **Style** to **dashed font**. 

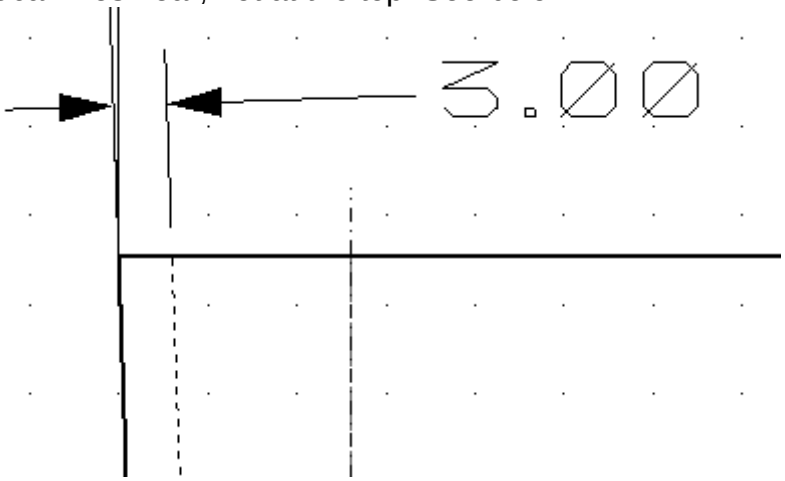
Repeat all this commands for the **right view**.

Linear Offset Dimension

You may want to zoom in to do the **3.00** dimension.

Use the **Linear Offset** command from the tab for the 3.00 dimension because it is being measured at an angle.

Pick the two near vertical lines near, not at the top. See below



Angular Dimension

Use the **Angular Dimension** command from the **Dimension tab** for the 2 degree.

Normally, an angular dimension requires you to pick two lines. In this case there is no vertical edge to dimension the draft angle so we need to pick the option shown above.

Pick the angled line as shown at right; near, not at the top.

Position the text as shown. The dimension will be from an imaginary vertical line.

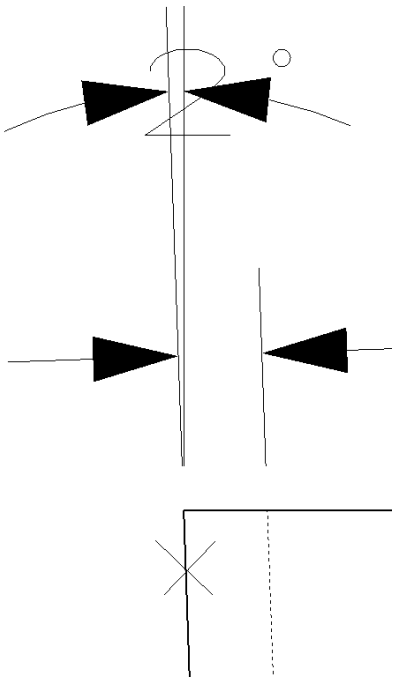
There are several choices based on design intent. Here it doesn't make sense to dimension the angle with 178°.

So the text position is important.

Don't worry about the exact placement until you have finished modifying all the dimensions.

Now that you have inserted the basic dimensions, you need to change/add a few things.

As we make the specific changes needed to match the example drawing, feel free to look at other options or dimension the part as you see fit.



Adding Appended Text

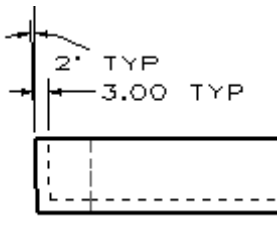
Let's add the word "TYP" (Typical) to the end of the two dimensions below. This indicates that this dimension controls the entire part not just this edge.

Click on both dimensions, **Right-click – Attributes**,

See the **Text** tab.

Input "TYP"(with a blank) into the **"XXAA"** field.

Pick **OK.**

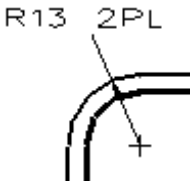


You should now be able to add the text "2PL" (2 places) after the R13.00 dimension.

Also change the precision to zero decimal places. Right-click on the dimension "R13.00" and pick **Attributes** to open the Dimension Attributes form.

See the **General** tab, click on the **decimal button**, and set decimal number to 0.

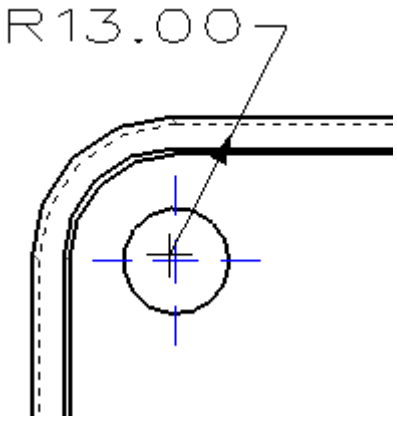
See the image below.



Adding a Dimension Tail

If you prefer to see the dimension leader with a tail,

click Radial/Diametric button  .Pick the arc which you want to dimension and pick the position of the text. Click Dim Attribute under the Setting > change  to  > click OK.



Reference Dimension

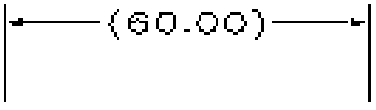
Both 60.00 dimensions are for reference only. ANSI dimensioning standard calls for parenthesis around a **reference dimension.**

Click on both 60.00 dimensions, **Right-click > Attributes** to open Dimension Atrributes form.

In the **General** tab, click on the **tolerance** icon, and set it to Reference value  .

Pick **OK.**

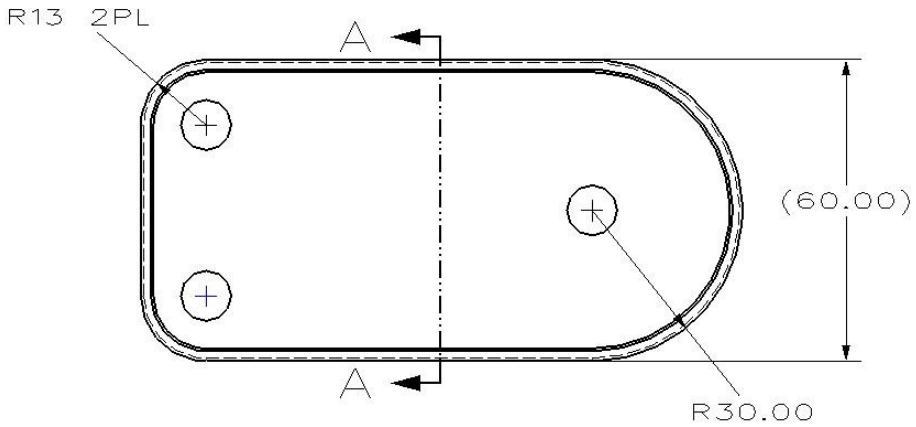
While adding this, try the other options for adding tolerance.



Section View

Let's create a section view from the top view.

Open the **Layout tab**. Click on the **Section** command. 



Pick anywhere on the geometry in the top view.

When prompted for the point picked indicated above.

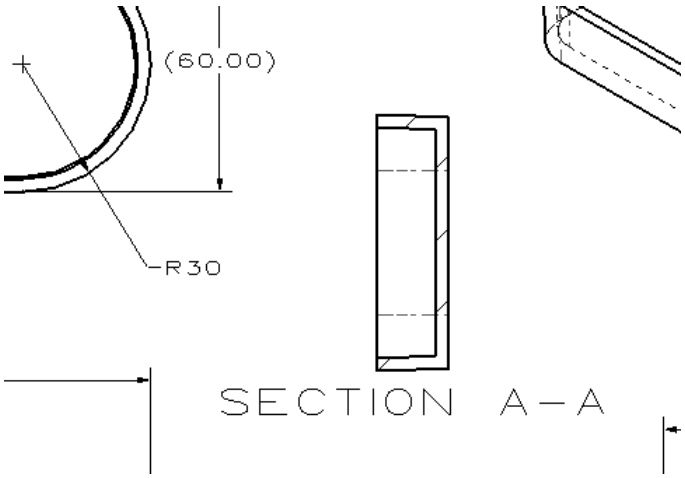
Drag the cursor until the section arrow is vertical.

The default option for the location is **Orthogonal**. This will automatically line up the section view.

If there is plenty of room this is preferable but let's say we don't have enough room and would like to place this section in the middle of the sheet.

Change the **Location** option to **None**.

Left-click where you want the view located.



You can still change other options if you like.

Pick **OK.**

Adding Text Notes

To add notes on your drawing, open the **Drawing tab**.

Click on **text.** 

Activate the text input box.

Right-click in the field of the drawing and Pick **Editor**.

Input the following information into the editor:

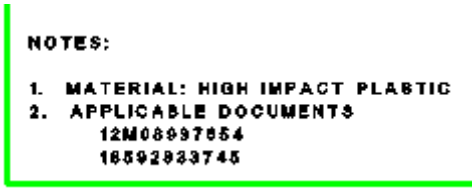
NOTES:
1.MATERAIL:HIGH IMPACT PLASTIC
2.APPLICABLE DOCUMENTS:
12M08997654
18S92833745

See the **Option** pulldown for special characters and font selection.

If you are required to export drawing to AutoCAD formats you may want to consider setting your text and dimension font to a TTF. (True Type Font).

The correct place to do this is in your templates.

Result :



Adding Text to Second Line of a Dimension

Earlier we **added the word** "**TYP**" next to a dimension.

What if you don't have room next to a dimension or you prefer to **add it on a second line**? (see the figure in this section)

Instead of Right-click > Attributes. You could use **Right Click > Modify Text.**

Optional inputs is looking for **User Text.**

Or, in this case we haven't created the dimension yet so let's do both at the same time.

Pick **Linear Dimension** in the Dimension tab, and use **Horizontal** method**.**

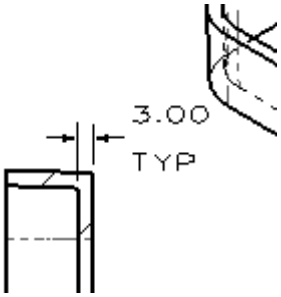
Pick the end points shown below and position the text.

We are going to add user text after the automatic value.

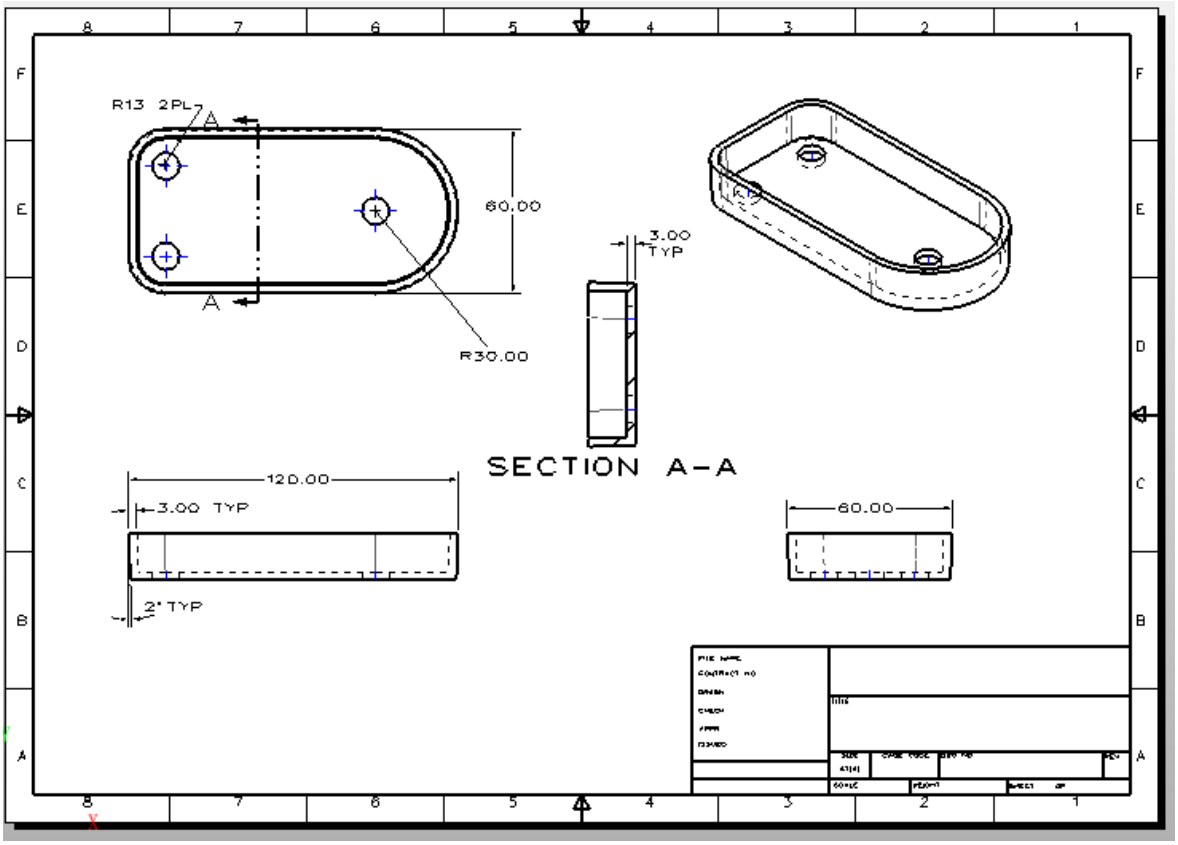
In the background, Right-click, and select Editor.

Hit **<Enter>** on the keyboard and **type TYP.**

Hit **OK.**

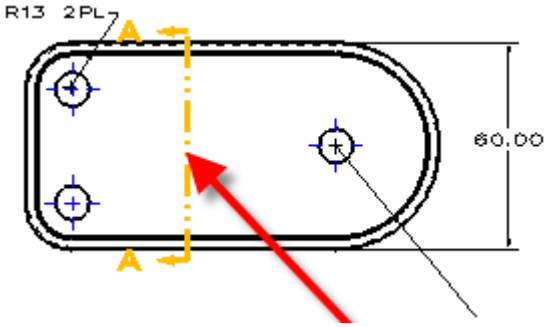


Redefine Section



The lower arrow of Section A-A is too close to the part. How do we move it down?

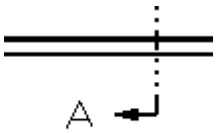
To move the “A” and the arrow, Left-click on the section line and drag it**.**



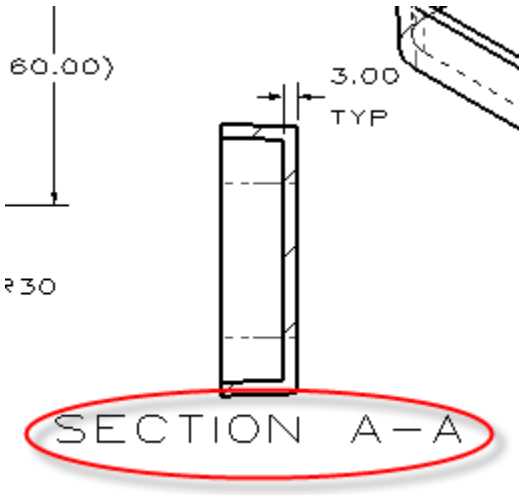
Note: You can also right-click on the section view itself.

Pick **Base Arrow** and pick the new location.

Pick **Align Arrow** and pick the new location.



Moving Section A–A label



Right-click on the text and pick **Move.**

SECTION A–A

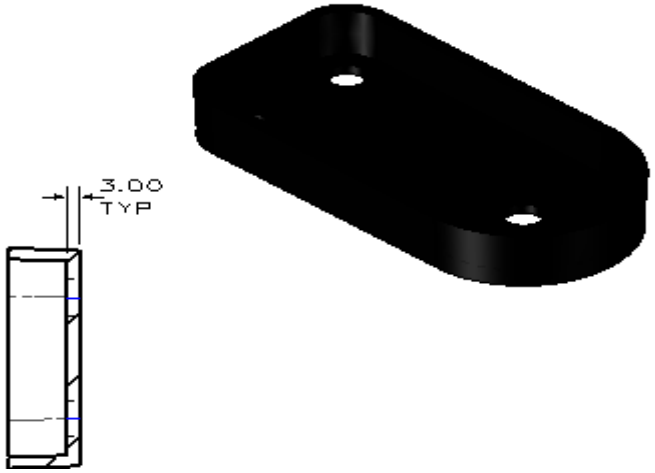
Change the isometric view to be shaded

This is something you never would have done on your old drafting board. I am talking about the isometric view much less a shaded view.

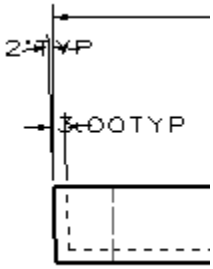
Right-click on the isometric view and pick **Attributes** to open the Layout Attributes form.

Pick **Shaded** under the **General** tab. 

In addition to shading the view you could also change the view scale if you would like.

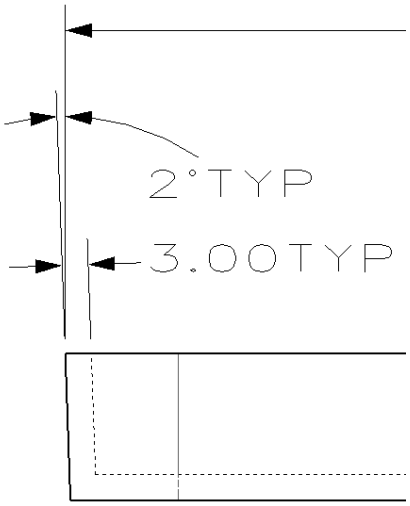


Fix the text location of dimensions



We left the location of some dimensions until we were finished making changes.

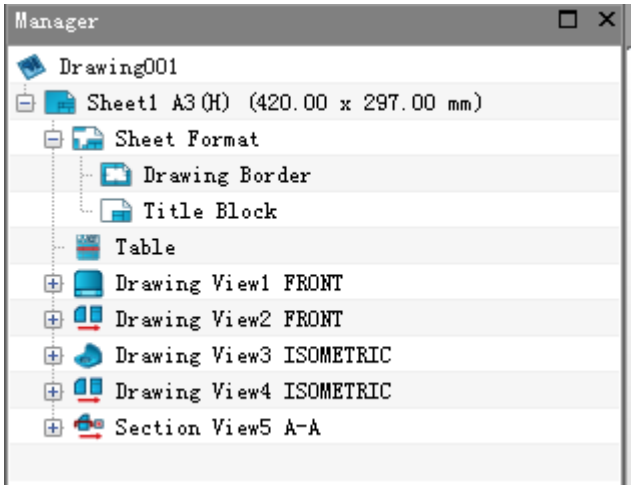
Right-click > Change **Text Point**



Add Title Block information

If you are not using the ZW3D default Templates your title block should be located at the lower left corner of the drawing sheet.

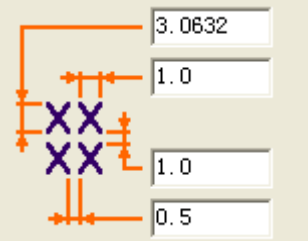
Please double click Tittle Block from the History Management.



User's company name may not fit properly. Your company name may be longer or shorter. Follow along regardless.

To adjust the width, **Right-click > Attributes** on the text to open the **Text Attributes** form.

Adjust the following form.



All the numbers below the height are percentages of the height. Change the width to 0.20.

Ultimately, this change should be done in your Templates so you don't have to keep duplicating your effort.

Add Title Block information(continued)

Some of the text found in the Title Block is specially formatted.

[$part_name]
[$Part_number] etc.

These are **embedded text variables**. These variables get linked to the first view created on the sheet.

You will go back to the part level of the Base and add the information that we need for the Title Block and possibly a Bill of Materials that we may create later.

Right-click on any of the views and pick **Edit Part**.

Now we are in the Base part (See the Title bar at the top of the screen)

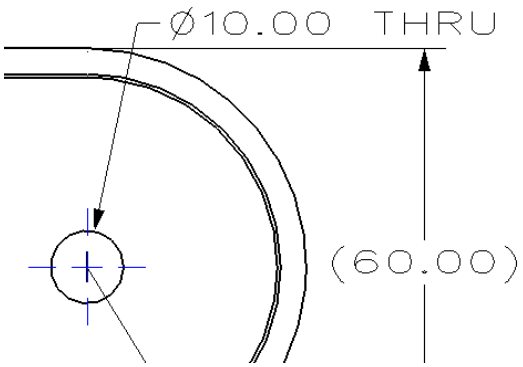
on the location of the holes now.

Hole Callout

Let me show you how to create a hole callout for the size. See the **Insert** pull down menu.

Select **Dimension,** and use **Hole Callout** at the bottom of the menu.

Insert >Dimension> Hole Callout works on holes created with the **Hole** command in ZW3D or Imported parts where you have used the **Utilities > Flag Hole Feature** command.



Pick the Top View. Pick one of the holes. See below. Pick **OK**.

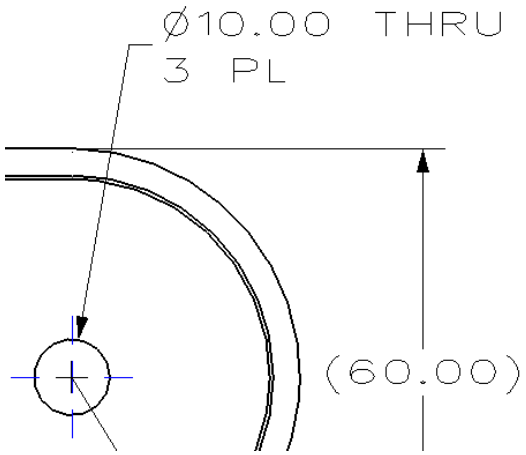
Right-Click > (on the dimension) Change **Text Point**. Move it up a little so we can add a second line of text.

Right-Click > **Modify Text**. Add **3 PL.**

[VxX]c10.00 THRU
3 PL

See the result below.

Pick **OK**.



Feature Control and Datum Symbols

If you don't use these, skip to the next topic.

Feature Control Symbols(FCS) 

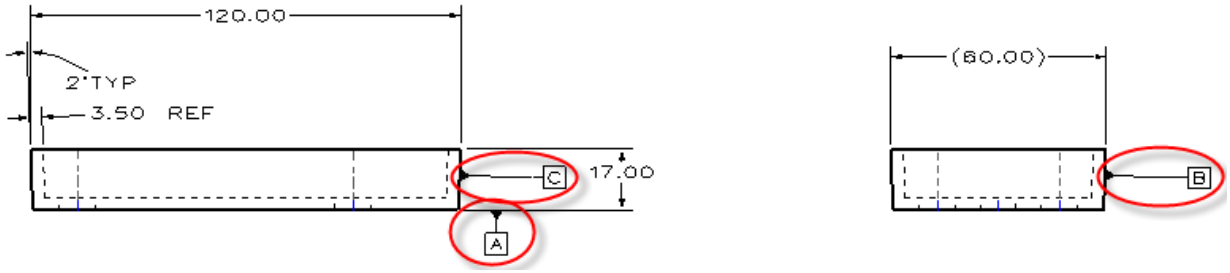
Datum Symbols 

Datum Target point 

Datum Target Circular symbol 

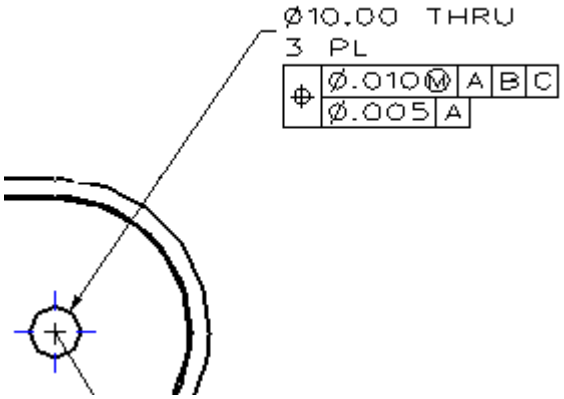
These tools are part of **Geometric Dimensioning and Tolerancing. GD&T**

Datum Symbols

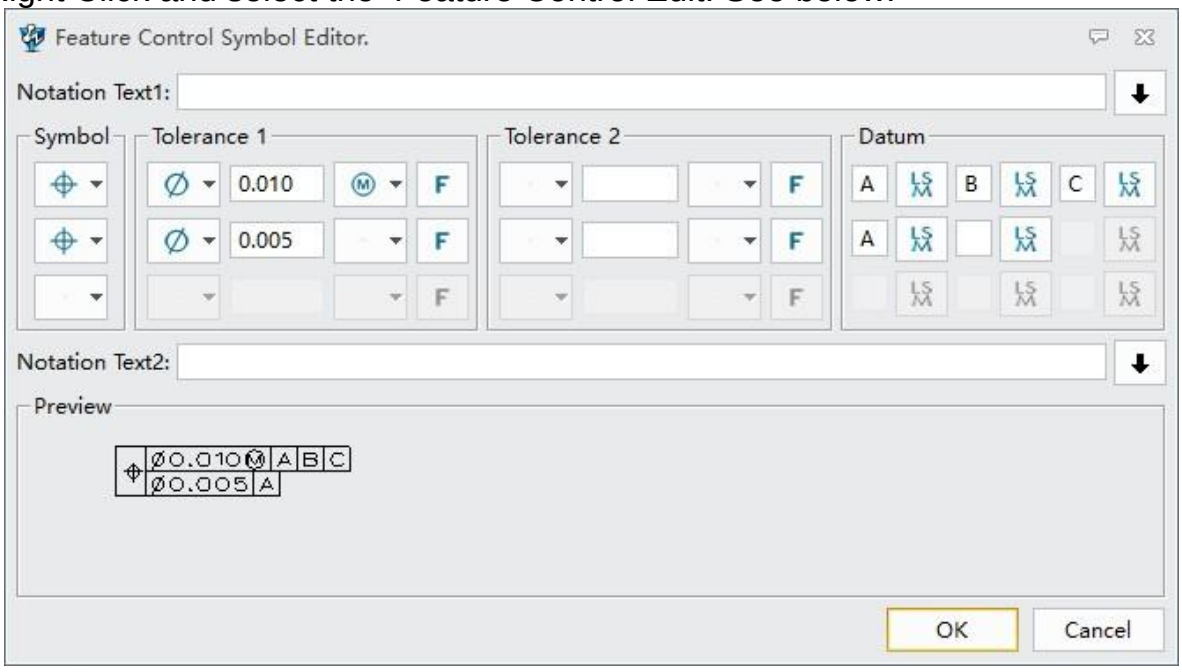


The Datum tag is just the Letter A, B, C, AA, ZZ or whatever is required. Pick the part geometry. Position the text.

Feature Control Symbols



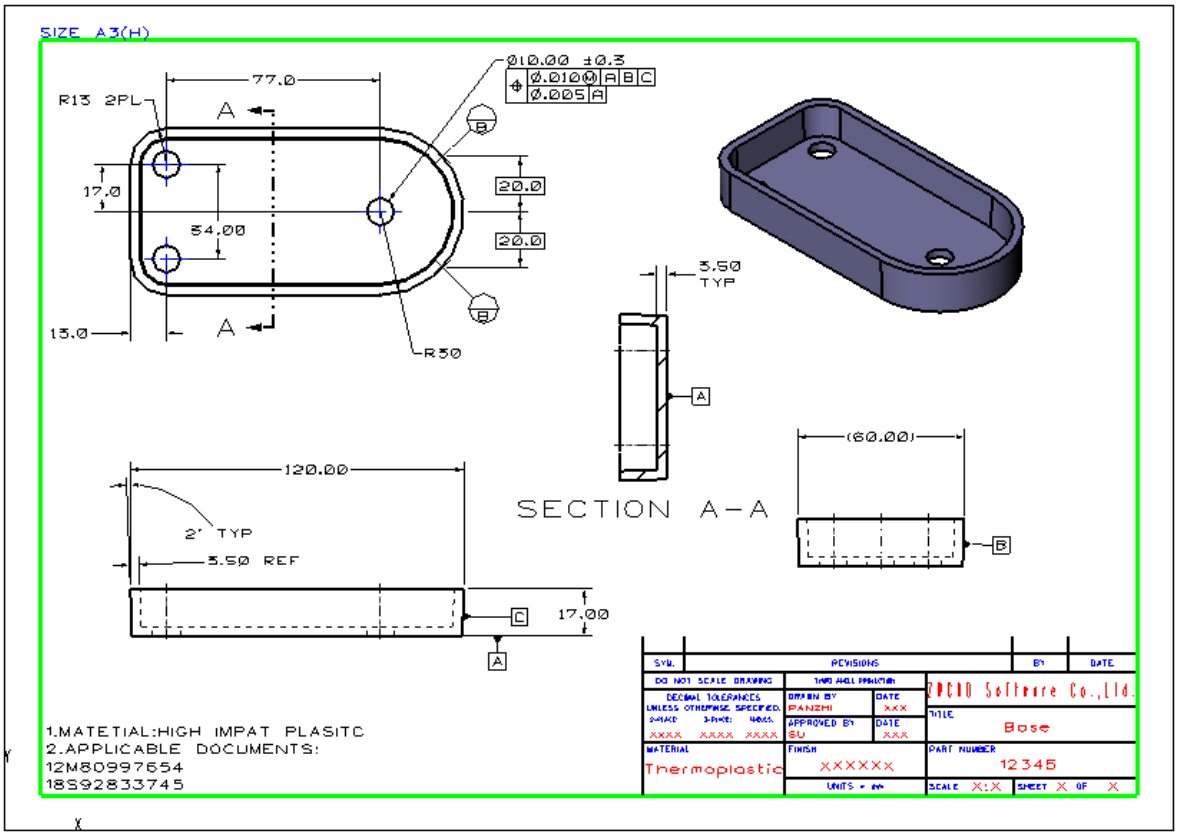
The Composite FCS shown above is constructed thus:
Right Click and select the Feature Control Edit. See below.

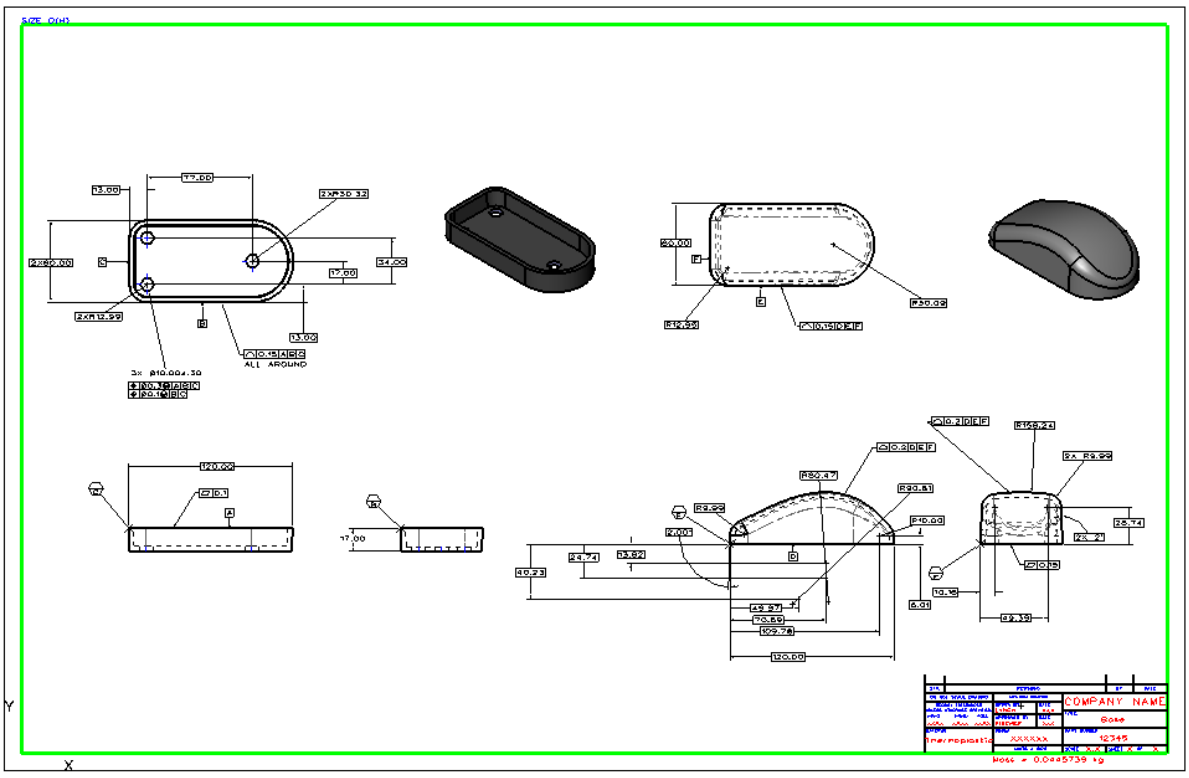


Result:

```
[Box][CPos][VxX]c.010[VxX]t[VxX]xA[VxX]xB[VxX]xC
[VxX]c.005[VxX]xA
```

Alternate drawing techniques





Other Topics

 Image symbol

 Surface and texture symbol

 Weld symbol

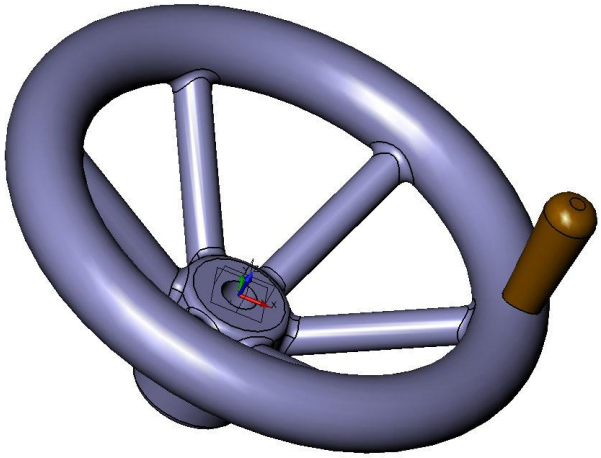
 Insert symbol

Look over the other tabs. Here are just a few things to look at. More information can be obtained in the HELP Manual.

Save your file. **Exit** part.

Assembly drawing of the Wheel

Open the **Wheel-Assembly.Z3** file we created earlier.



Open the **Wheel_Assembly** part.

Before we make the assembly drawing layout we need one more component but it will not require any geometry.

GLUE

I'm sure you are thinking of other ways of attaching these two parts but this is my design so "Go with the flow".

Right-click > Insert Component.

Call it Glue > Anchor > and locate at zero.

There will be no geometry but for our last step we need to define **Part attributes**(from **Attributes** pull down menu).

From the Assembly Manager **double-click on the Wheel and Handle** and define your own **Material and Part Attributes**.

When you are finished, **double click** on the **Wheel-Assembly**.

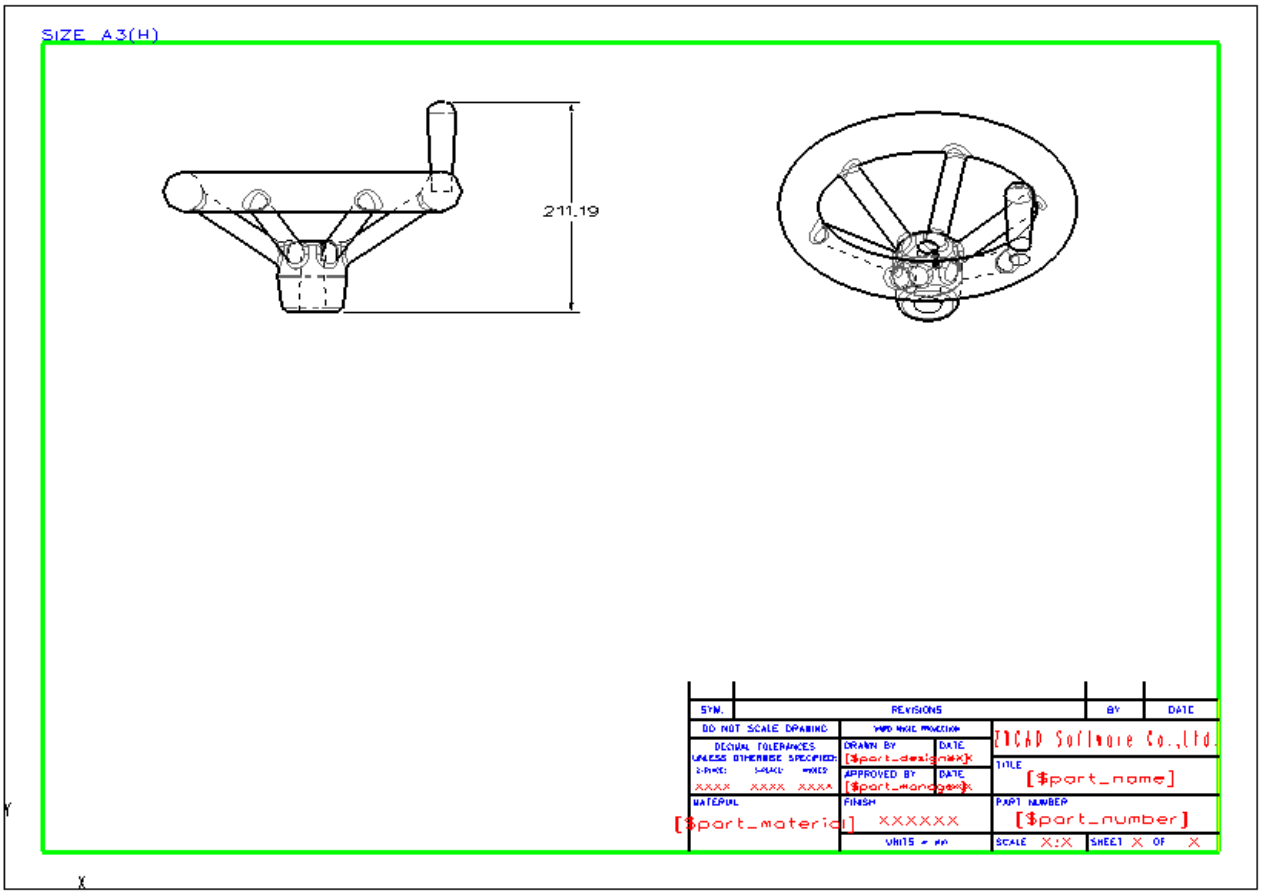
Add Part Attributes here also.

Right-click > 2D Sheet.

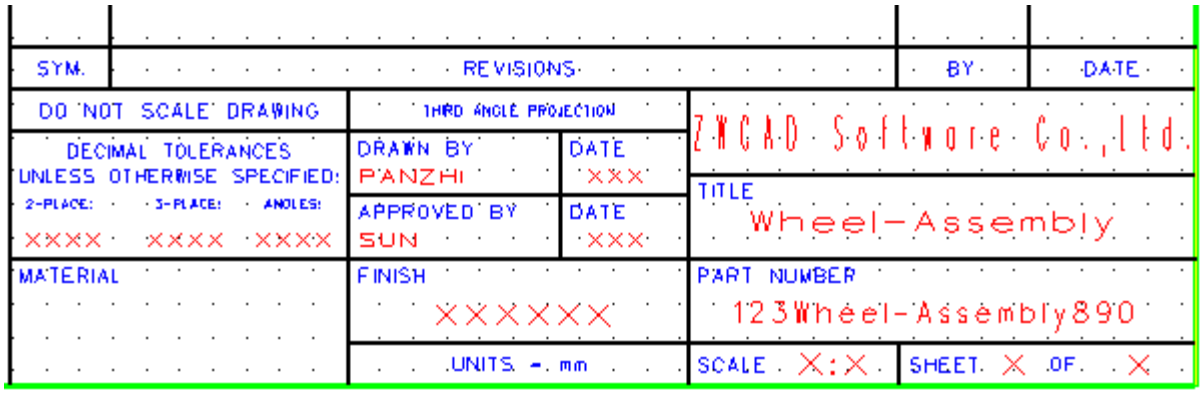
Erase the Front and Right side views.

Move the remaining views to a convenient position.

Add a dimension as shown below.



Pick **Edit > Regen Text** from the pulldown menu. This will fill in the Title Block with the part attribute information you provided in the last step.



BOM table-Bill of Materials

Erase the text in the **Material** box of the title block. This is the default material and doesn't apply to an assembly drawing.

The mass property value is correct so don't erase that…unless you don't want to see it. Insert a Bill of Material (**BOM table**), using BOM table command under **Layout** tab or Table Manager. 

Pick the **Front View.**

You can select any or all of the attributes from the **Available** column by **double clicking** or click once and pick the arrow pointing to the **right**.

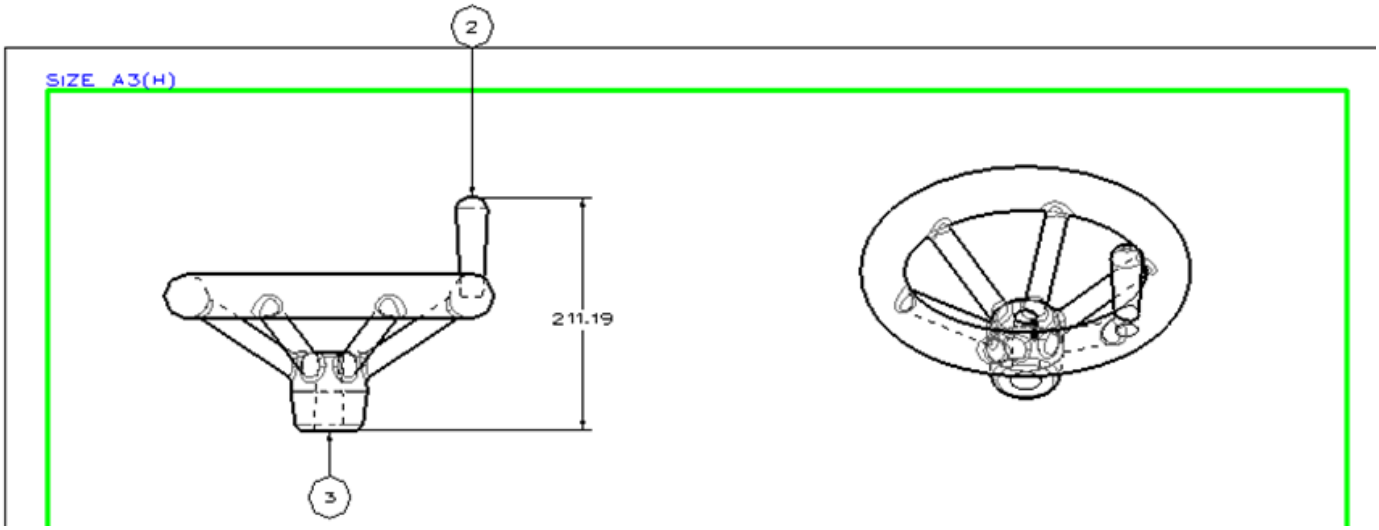
You can also **double-click** on attributes in the **Selected** column or click once and pick the arrow pointing to the **left** and they will be removed.

When you have all the attributes in the selected column, use the **Up/Down arrows** to rearrange their order.

Unless you are going to create more than one table on this sheet it is not necessary to name it.

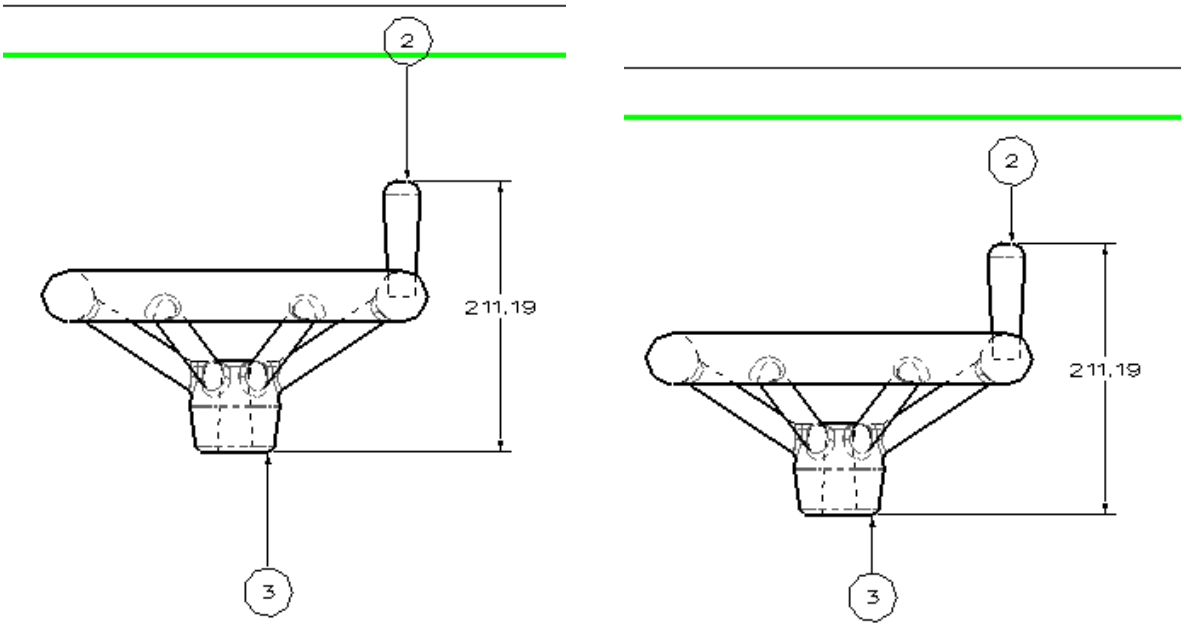
Pick **OK.**

Automatically **Insert**  this table on your drawing.



Pick **Clean-up balloon locations.**

Select the Front View , Decrease the offset value so that all balloons are inside the border.



You will notice that there is no balloon for the glue. Typically, I would want that to be the last item on the BOM.

Right-click on the Table > **Edit.**

Click on the **Open Editor** button located under the **Columns** tab from the Options tab.

Just click on ID number 1 and type 3.

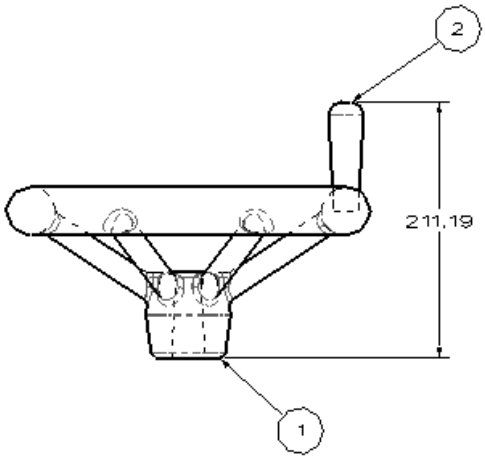
Click on 3 and type 1.

Pick **OK**.

Pick **OK**.

You can also edit any other value like the quantity of glue.

You can also **Right-click** on each balloon and **Change Text Point** as shown below.



In version 15 you will be able to **Modify table balloon location.** This will move the arrow not only to a better location on the component but to another view if that component can be seen more easily.

There will also be a **Sync BOM Table with Part Attributes** command so if you want to input the attributes directly on the table, ZW3D can update all the Parts.

NOTES:

In the next exercise we are going to import an images file and look at just some of the techniques to Heal/Repair the data.

- Import
- Sew
- Trimmed Plane
- Fill Gap
- Trim to curve
- Ruled Surface
- Extend Face to close gap between edges

Chapter 6 Importing and Creating Geometry

Let's create a **Multi-Object file** called **"Project_554.Z3"**.

Create an **Part** object called **"01_IGES_001"**. Pick New Assembly Mode.

You can optionally give either the File or the Part a description.

Now that we are in the file, we can verify what units we will be working in. To do that, we will select **Edit** pulldown menu and then the **Preferences** option.

The default system tolerance is 0.01 and we will keep that value for this exercise.

The Part settings menu allows us to configure:

Units – Option that allows the user to define work units.

Tolerance - This setting refers to the default part geometry tolerance measured in the default units. The geometry tolerance is feature dependent. If the geometry tolerance is modified, only newly created features are affected. You can adjust the geometry tolerance for individual features using the Set Feature Tolerance command.
If you start ZW3D using INCH units, the default tolerance is set to 0.0005. If you start ZW3D using MM units, the default tolerance is set to 0.01.

Alternate display of free edges – Option that allows visualization of openings and "**Gaps**" in the 3D model.

Auto sew on face creation– Option that indicates if the new faces created by the user are automatically sewn to the active model.

Merge shape before cut(remove) – Option that indicates wwhenever you have a Boolean Operation like, holes on surfaces or openings across two objects, they can be joined/merged as one or kept separate.

Use component light sources – Enable this option to use light sources associated with the first level of components in the active part/assembly in addition to any locally defined light sources. When disabled, only local light sources (or default lighting) are used.

************NOTE: On the Part Settings menu, it is recommended to turn "Auto Sew" off** allowing the user to analyze the geometry after importing.

Now, let's import an IGES file. To do that, select, **File** and go to **Import**.

File Type select IGES File.

Open the file "Backcover_554_Rev_04.igs" which is located in" installation directory/training"

This opens the **IGES Import** dialog so that you can define settings for the file that you are importing.

We will select the IGES option for importing.

Auto Sew Geometry – indicates if the geometry is to be automatically united.

Auto activate part – Indicates that after importing, the geometry is automatically activated. This command is only necessary if the importing is done at the object level of the work session.

Rename layer collisions
– Renames the layers if they already exist with the same name.

Remove duplicate surfaces – If there are duplicate trimmed surfaces, the duplicate is thrown away.

Break trim edges - Will analyze the edges of the surface and break the edges on the tangent points if needed.

Reset Dependencies - It is not uncommon for third party IGES files to contain incorrect dependency flags causing entities to not import, or to import more than once. Use this option to flushes the dependencies reported in the file, and recreate them based on the entity references from the parameter section of the IGES file.

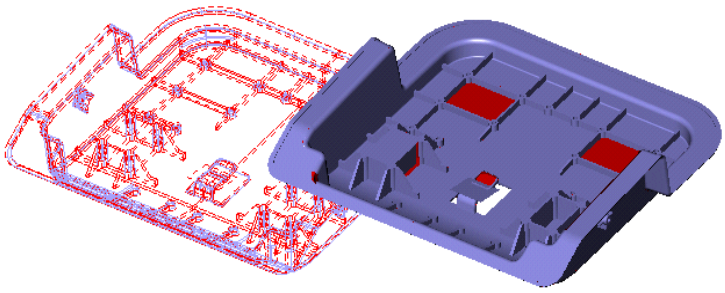
Drawings – Import the 2D lay-outs included in the file for the 3D environment.

Bounded planes – Import planes with limits.

Unbounded planes – Import planes without limits.

*********NOTE:** **For this exercise**, deactivated the **Auto Sew Geometry** option and select '**OK**'.

After you have finished importing the IGES file, the object should look like this:



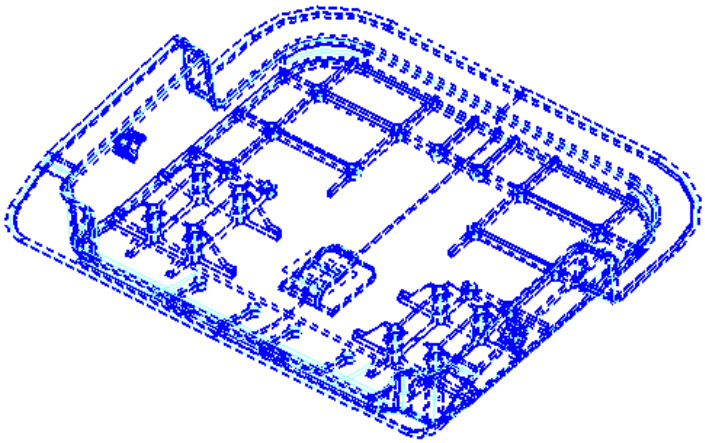
Your model has all the edges of the surfaces dashed, because they were not sewn during the import.

The quality of the imported geometry can be improved by “healing”.

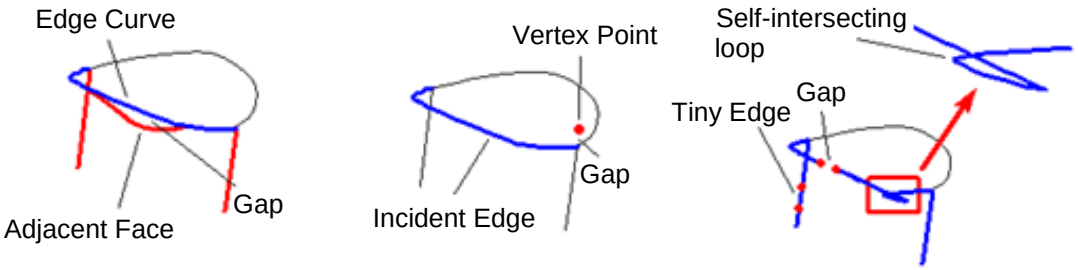
Select the **Heal tab** from the menu area. Select the command **Heal Part Topology**  .

This command defines the tolerance for correction. For this exercise keep the tolerance at **0.01**.

See the result below:



The image below shows some examples of problem geometry.



During the import we deactivated the auto sew command. Now let’s join the faces using **Sew**. To do this we need to display the **Free Form tab** as shown below. The icons on this tab are the tools for the building and manipulation of individual surfaces. Refer to the help document for more about each command.

A few commands are described below.

 **N-Sided Patch**: Use this command to create a face by patching across 3 or more profiles.
The profiles can be wireframe geometry, sketches, or face edges. Required inputs include the profiles to define the patch.

 **Intelligent Blend**: Use this command to create an intelligent blend face. Required inputs include the curves, edges, or faces that the new face will start from, go to and pass through. Optional inputs include using curves, faces or a datum for spine control, circular and conic cross-sections, sewing, capping and the ability to give the face a unique name if desired.

 **Modify number of Isolines**: Use this command to set the number of isolines displayed in the U and V directions of a face. You can also set isolines with the Modify Face command.

 **Explode Faces**: Use this command to separate faces from a shape. Use the **Group** option to maintain the connectivity of the explode faces (i.e., they become a separate shape). Use the **Sew** command to join faces back into a shape.

 **Trim to Faces**: Use this command to trim away part of a face or shape where it intersects with other faces, shapes, and/or datum planes.

 **Extend a Face**: Use this command to extend one or more edges of a face. First select the face. Then select the edges to extend and enter the distance to extend.

Select the **Sew** command. 

Select **OK** to select all faces. You can change the value of the tolerance to be different from that of the Part Tolerance. **Don't change it at this time.**

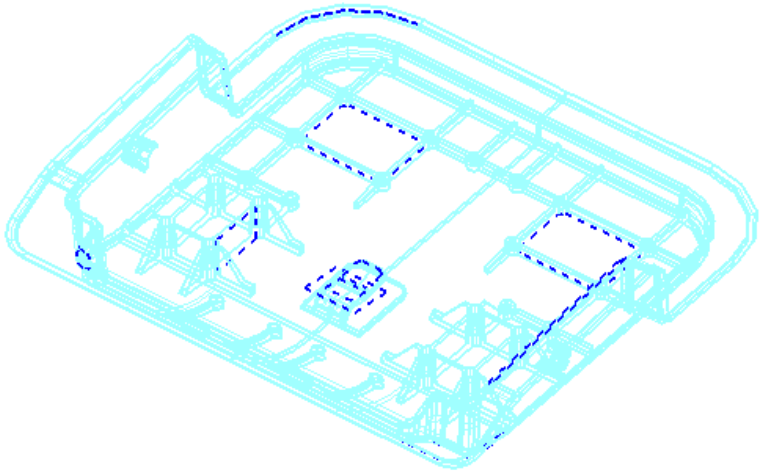
Suggestions to help determine the scope of the needed repairs:

1. Put your **Pick Filter** on **Shape**. Window pick the entire screen. The message window will report how many separate **shapes** have been imported.

2. Cancel select. Right-click on the biggest shape that highlights and pick **Face Attributes.** Change the **face color** to something other than the default blue. Do not pick pink; this is the default color of the back side of a face. This way you can visually see the other shapes and it also may give you an immediate idea of how to approach the repairs.

Your model should have most of the edges sewn. Do you notice the difference in appearance between the sewn edges and the open edges?

Whenever the surfaces are missing or have "Gaps", the edges will be blue and dashed.



The sewing result should be “Number of unmatched edge is [61]. Maximum gap between unmatched edges is [0.011831mm]”.

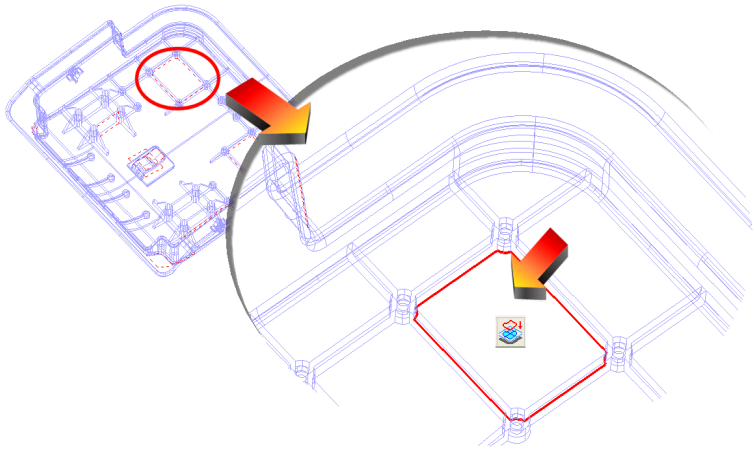
After sewing, the system will give you information about the largest **Gap** existing on the geometry. The user can refer to that value to redefine the **Sew** command using it as the new tolerance. You can use the **Heal** command again to correct the geometry with the intended tolerance. We will do this after we are finished repairing the model.

*************NOTE:** Since the majority of faces have been joined by sewing, **turn the ‘Auto Sew’ option back on under ‘Edit > Preferences’.**

Creating the missing surfaces is the next step.

For this exercise let’s use the “**Trimmed Plane**” command  . This command permits the construction of surfaces from 2 or more edges and/or curves.

Select edges shown in the image(Look closely, there are 9 edges). <**middle-click**> to “accept” the edge selections. <**middle-click**> again to “finish” the command.

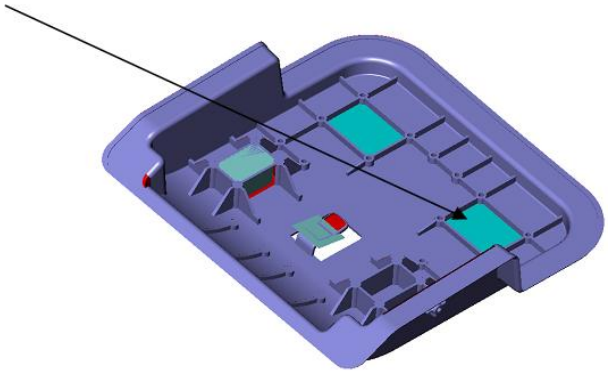


The bottom faces of the pockets are missing. When the part is shaded, the red face you see is actually the inside of the back face of the part. The default color scheme in ZW3D defines the inside or the back side of a face as pink. However, an import retains its colors.

To verify that a face is missing as opposed to a face that has its normal reversed, zoom in close to one of the dashed edges and window pick across the edge and look at the message window. Your **Filter** must be set to **Edge.** This will give you a count of the edges found. One edge means a face is missing.

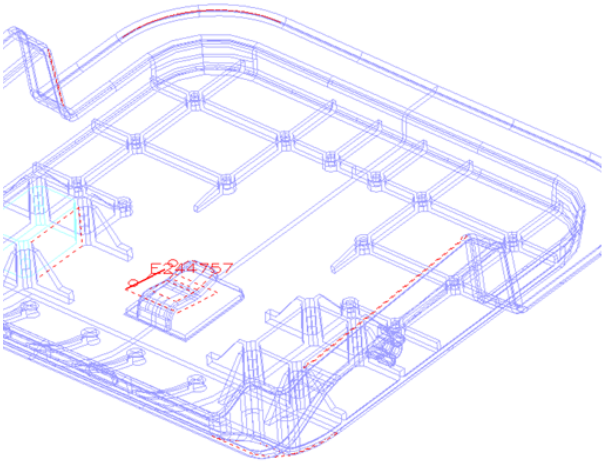
For the next surface we will use another command which will also allow us to create a surface on the free edges. These tools are found on the **Heal tab.** 

Select the "**Fill Gap**" command. Select an open edge indicated below and then <**middle-click**> to fill and confirm.

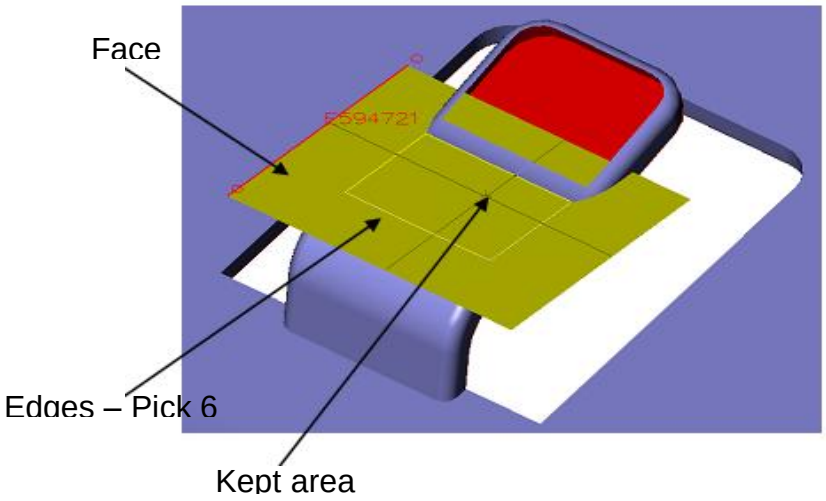


This menu also contains a command which helps us detect problems with the geometry.

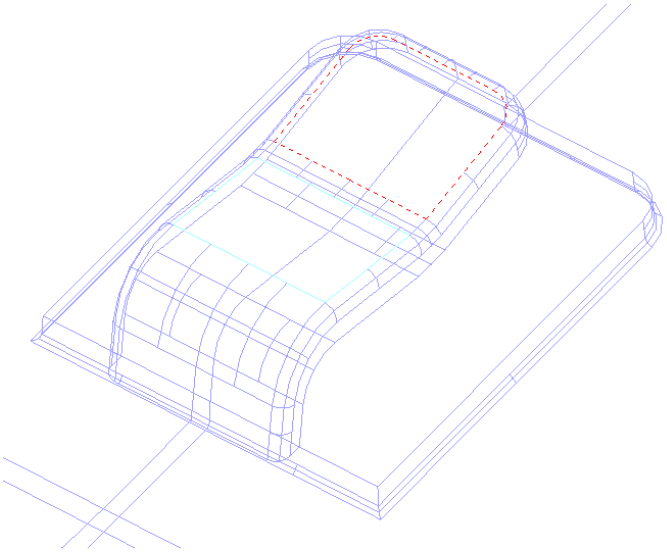
Select the command "**Show Open Edges**"  . This command lists the number of free edges. It also displays a form, which allows you to "**Zoom To**" to better identify the problem area.



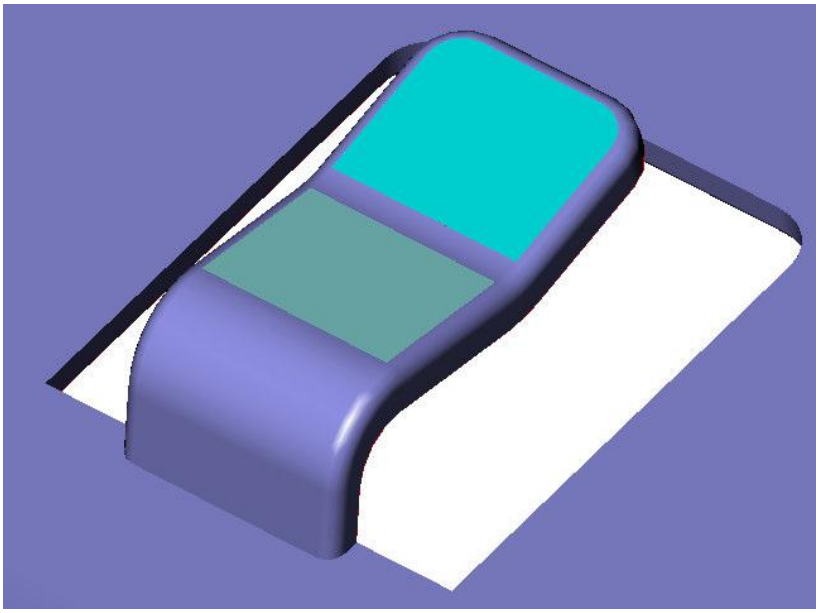
The problem here is a face was imported but not trimmed to its boundary. In this situation we can use the command "**Trim to curves**"  . To use this command, first select the surface to be trimmed. Next select the edges/curves for that trim. (Use the edges shown in the image). Finally, indicate the side you wish to keep.



Trimming a surface does not automatically sew. Select the **Sew** command  , and select **OK**.

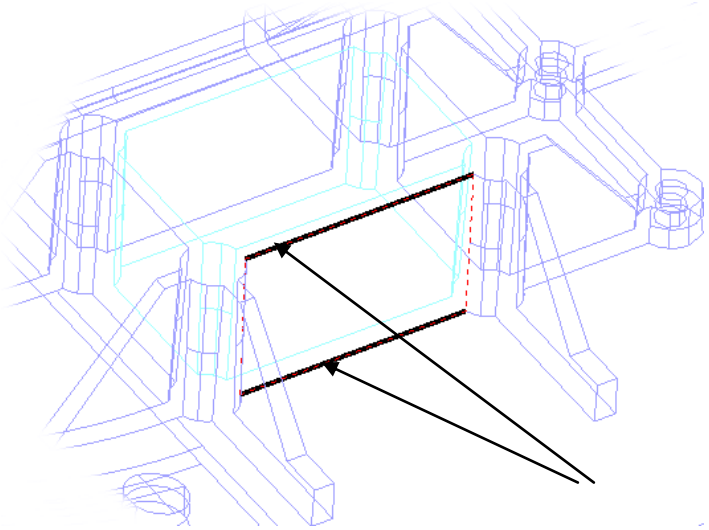


This next area is missing a face. Use the **Fill Gap** command  to create the next surface and select an edge. This is shown in the image below.



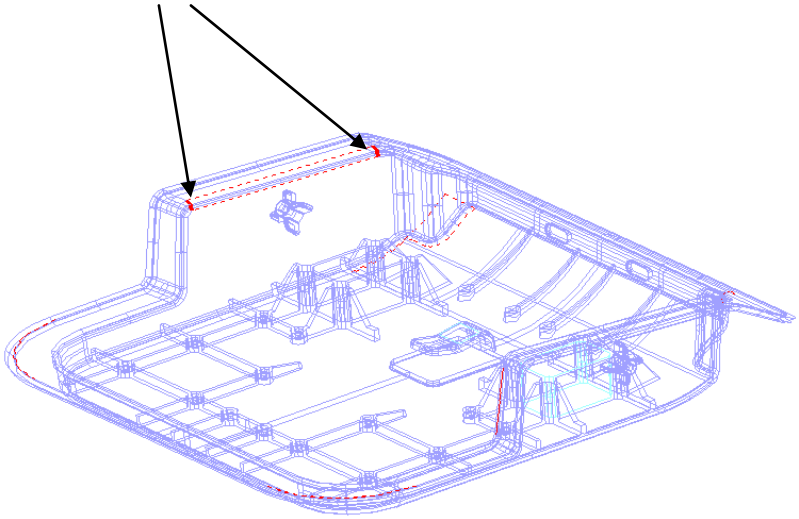
A new surface should be created and automatically sewn per the **Preferences** setting.

Find the problem area shown below. Again we have a surface missing. Select the **Ruled** command  , for the construction of this surface. Just select the two edges as shown in the image. Select both edges on the same end to avoid creating a twisted surface. (generally referred to as a “bow-tie” effect)

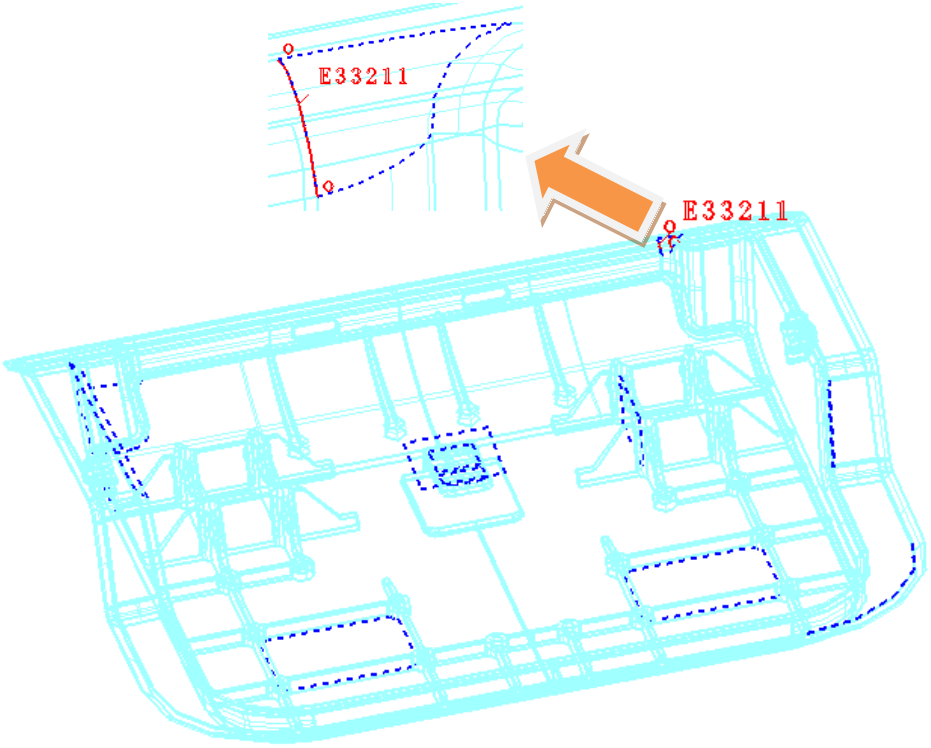


The **Ruled** command will not only create flat surfaces, but will also create surfaces supported by two curved edges.

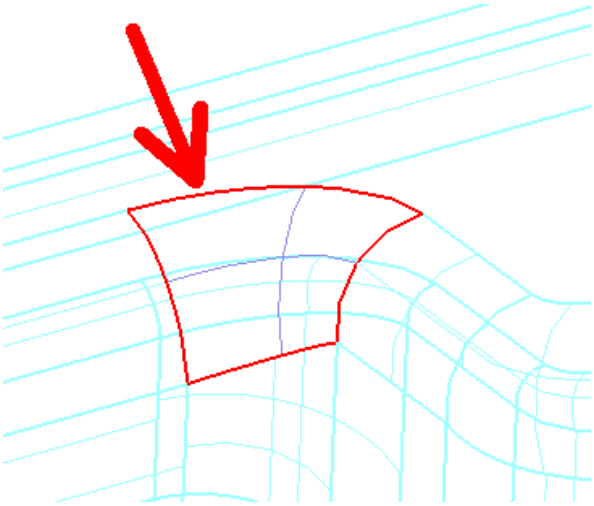
Select the arcs as shown in the image below. This type of surface creates straight line rulings between each point on the two curves selected.



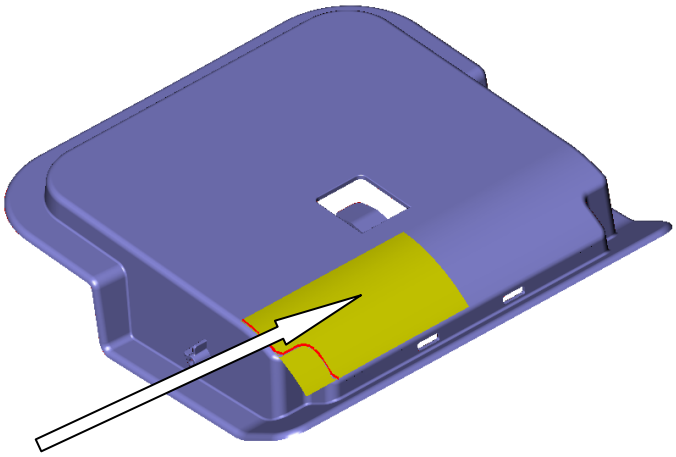
Once again use the command **Show Open Edges**  to locate the open edge shown on this image.



After locating this missing face, use the **Fill Gap** command. Activate **tangency** options . Select one of the edges. You will now see a new surface that is tangent to the surfaces from which it originated.



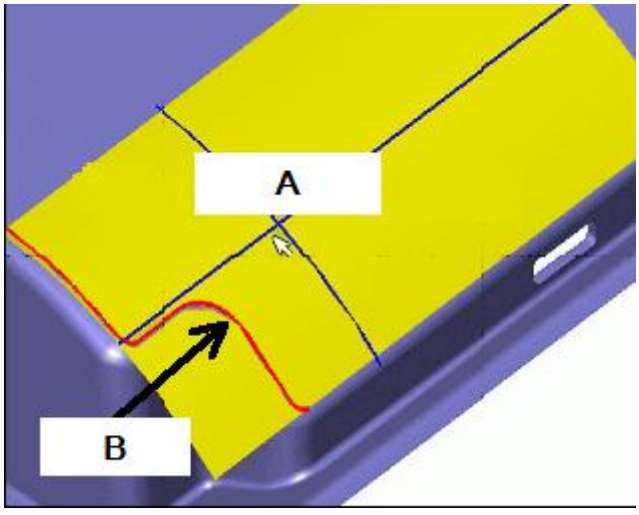
Now we are going to repair the surface shown in this image by using **Trim to Curves.**



Select the surface to be trimmed.(You may have to change the filter to FACE)

See the figure below, select the boundary edge(B) (use <shift- pick> to chain select the edges).

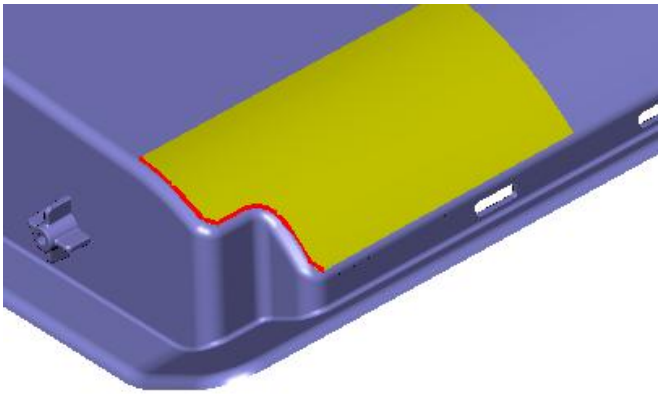
Finally select the side of the face to keep(B).



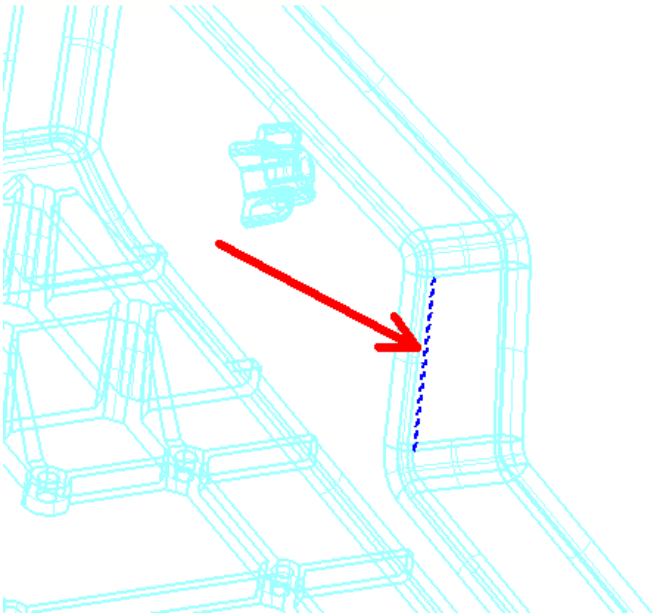
Again this surface was trimmed.

Sew and select **OK.**

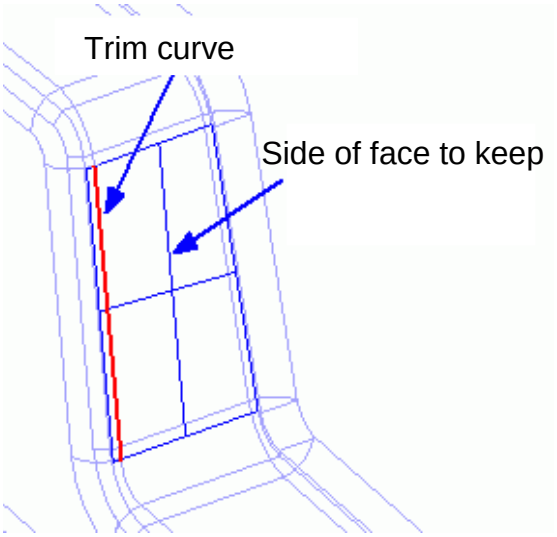
See the result below:



The next surface will be repaired by extending and trimming. Use the **Extend** command on the **Free Form tab**. Select the open edge of the planar surface (on the right) to extend. Select an extension value of "**0.5**".



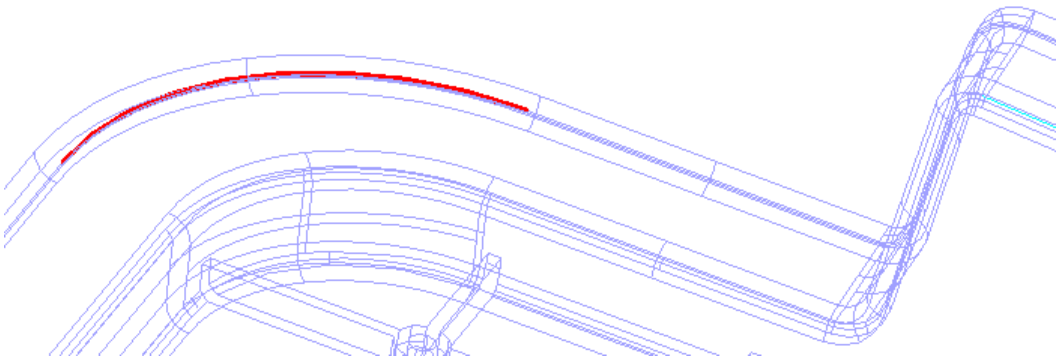
Now **trim the surface to the edge** which will be the boundary.



Use **Sew command again for that face.**

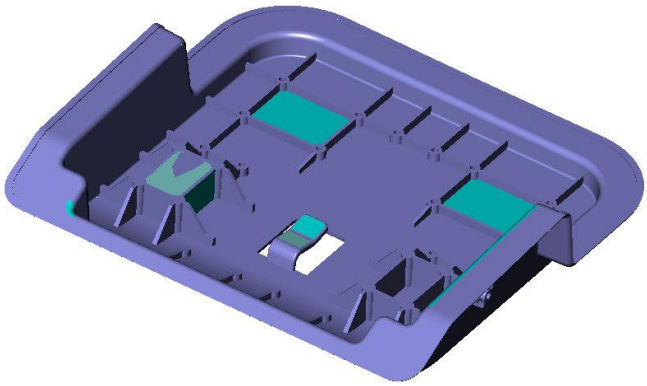
If you verify the openings on the model again, you will see one of two possible results. If after the initial sew command you sewed again with the tolerance equal to the largest gap, you will have a solid with zero open edges.....you're done.

If you did not select that command you will have two edges left that do not match.



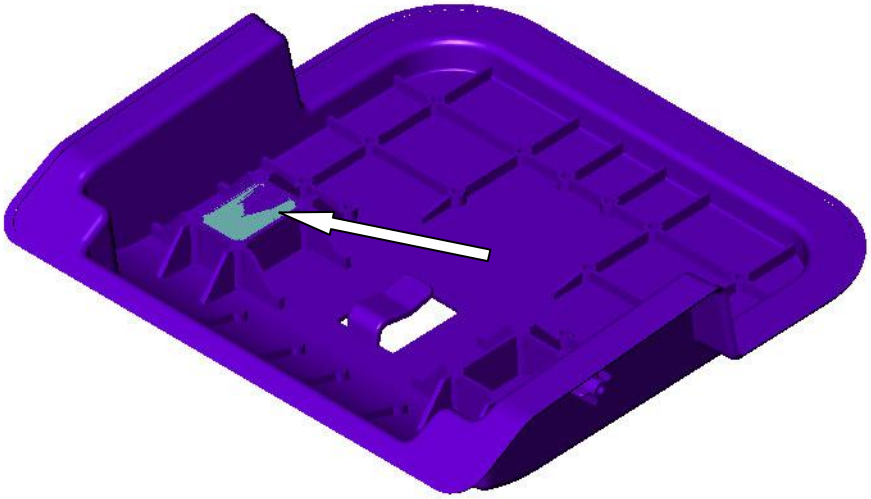
To easily repair the fault in the geometry use the **Close edge's gap** command  . Just select two of the edges that need repair. The software will only select open edges so you don't have to be real accurate picking.

If you analyze the model again, you can verify that it is completely closed(use **Show open edges** command). However, the model does not have to be completely closed to continue the project in ZW3D.



Turn on **Shade**. Change the pick filter to **Shape**. Run the cursor over the edge of the part. **Right-Click** and select **Attributes**. Change the **Face color** or select **OK** for the default. Run the cursor over the edge again. This will show us that we are still missing something.

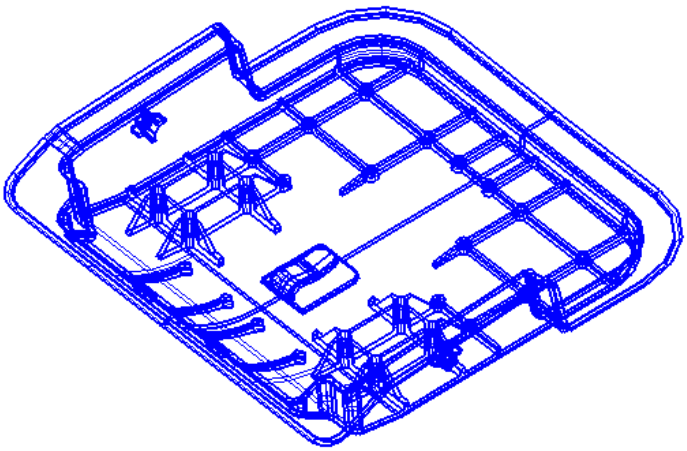
Using **Blank**, **Unblank** and **Swap entity visibility** you will see that a solid was left in the model that needs to be subtracted.



We can use **Combine Shapes** **with the Remove option** as suggested in this image.

Select the mold part as your base. Select the object shown in the image as the object to be removed.

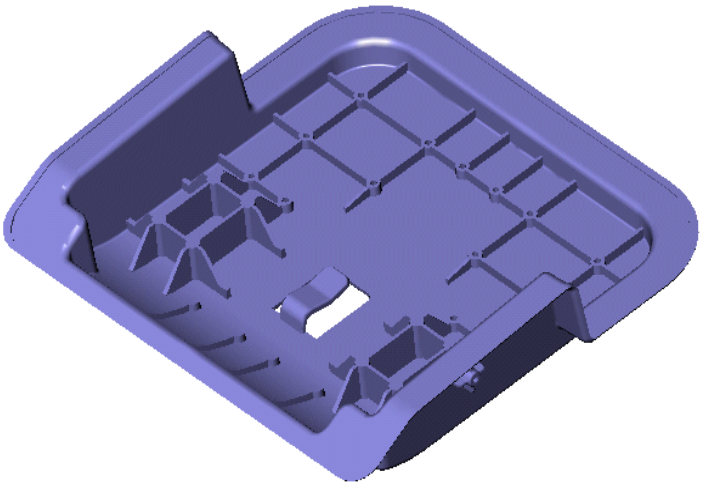
To complete the repair on the model we just need to heal by selecting **Heal part topology** with a tolerance of **0.005**.



This will complete the import and repair of your model.

NOTE: This is an excellent command to run before you export a part object. It tightens up the tolerance which usually yield a more successful import.

Save your file.



NOTES: