ZW3D
CAD Fundamentals Training Guide

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ZW3D™ V2017 CAD Fundamentals Training Guide

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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 **CAD Fundamentals 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 **CAD Fundamentals 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

Day 1

Support Files and System Requirements

The ZW3D download file

The ZW3D download file contains all of the necessary support files that will be used throughout this training class. The ZW3D support files should automatically copy into the “training” folder located in your user directory. For example you should see the training files in “C:\Program Files\ Program Folder \training”.

This is your own personal copy of all training files. Maintain the copy in your Program Files directory as a backup of your original unmodified training data for future use. See figure below for reference.

If you browse the Program Files\ Program Folder location you will also find a PDF directory that contains the electronic copy of basic training and installation manuals.

System Requirements

ZWCAD Software Co., Ltd. recommends the following configuration for operation of ZW3D.

Recommended configuration

- Intel Pentium® IV（2GHz or up）, Intel Xeon®, Intel Core® or equivalent AMD® processor
- 2GB 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-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 8.1（32 bit or 64 bit）
- Microsoft® Windows 10（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

Chapter 1

The ZW3D User Interface

Starting a Basic ZW3D Session

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. See the ZW3D Objects form on the following page for reference.

Once you have launched your ZW3D Software, 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.
See the **HELP Manual** for information on the “search path.”

OPEN a File

Use the Create a New File form to specify what type of ZW3D object you are going to create. You can choose from a template displayed at the lower left corner when creating new ZW3D objects. Templates are standard ZW3D objects contained in the files “*Templates_MM.Z3*" and *“Templates_IN.Z3”*

These files are typical ZW3D CAD/CAM files 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.

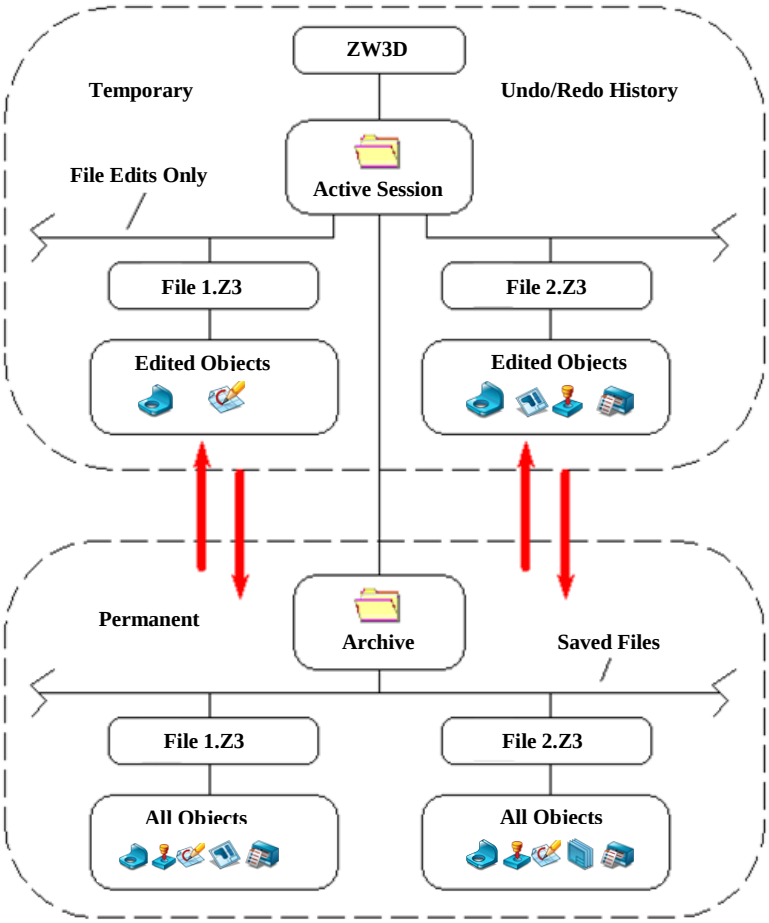
The unique name and description could be also added in the form.

When using the ‘Create a file’ command inside of ZW3D, after exiting to the root level, the ZW3D Objects manager will be displayed. Several commands such as “Edit a object”  and “delete objects”  are displayed on the right-click menu. Use this form to manger a variety of objects or edit existing objects.

Basic Work Flow

The ZW3D data hierarchy allows root objects such as parts, assemblies, sketches, drawing packets, and CAM plans associated with a design to be stored in a single manageable ZW3D object file. When you are running ZW3D it is referred to as the active session. The active session is a temporary work area into which portions of ZW3D files are loaded as needed during a design or revision.

The active session stores only the modified data. The session concept allows large complex parts and assemblies to be worked on as efficiently as the simplest part. You can view and manipulate the entire design while only the objects and entities being edited are retrieved thus requiring less system resources. The figure below illustrates the basic workflow during the active session.



Basic Workflow

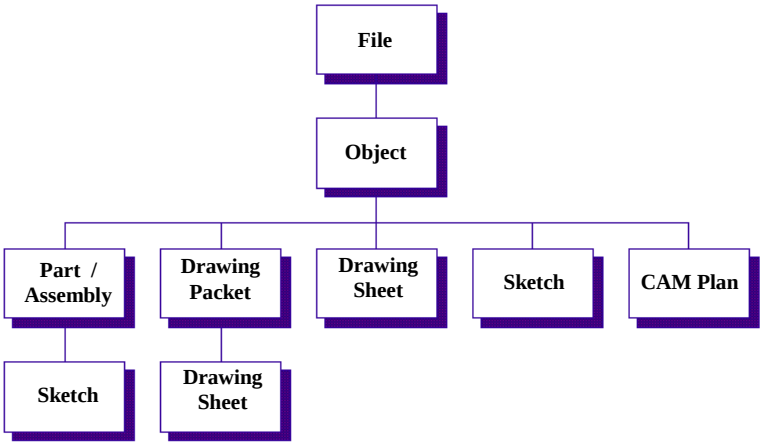
The Active Session

When you are running ZW3D it is referred to as the **Active Session**. When you create a new file it resides in the active session and contains the initial root objects that you create. When you save a file that is new and has not been saved yet, the File Browser is displayed and defaults to the "**File Save**" directory defined on the Files tab of the ZW3D Configuration Form or the last directory specified by this command or the Save File As command. If you save all files and one of the files is new, it is saved from the active session to the "**File Save**" directory mentioned above.

When you open a file, the active session only retrieves the root objects and sub-objects needed. Also, display information is loaded on-demand. Display information that lies outside the initial view window, that is blanked, or that isn't needed for the active display mode (e.g., facets for shading) is not loaded. This is all part of the ZW3D efficient object manager.

A file opened during the active session does not represent the entire file - only what you have edited. When you save a file that was opened, the original file is updated and the session is cleared of the file. You can think of the active session as an efficient work area where multiple files can be created, opened and then saved as needed.

When you save a file or save all files that are new to the active session, they are copied to a permanent (archive) directory that you specify. At that time they become archived files. When you open an archived file, only the root objects (and sub-objects) that you edit are retrieved. These are referred to as file edits. When you save the session it only contains the file edits. It does not contain the complete archived file. When you save a file or save all files (that were archived files) the archived files are updated with the new file edits. Once this occurs, the file edits are cleared from the active session.



Note:
It is important to remember that the UNDO and REDO history is maintained only during each session and that every time a file in the session is SAVED, the UNDO and REDO history is reset.

Session Management

The use of session management starts in the configuration file.

Click on Configuration > General tab.

See the **Session** part.

By default all the options in this section are disabled.
The first option, **Enable session management** is the key. This turns on or off the use of session management.

Session management can be thought of as having two definitions.

1. The first is simply a quick way of saving data during your work day.
2. The second is more sophisticated. I like to think of it as "What if Analysis".

3. As described above, the active session only retrieves the root objects and sub-objects as needed. When you activate a previously archived part and start making changes, only those changes are stored in the session. So it is logical to conclude that when you save the session using " **File > Manage Session >Backup Session**", it will be fast. After using "**Backup session**" you could exit out of ZW3D without Saving the File and the next time you enter ZW3D you will be returned to the exact spot you left your part.

   Click on the button of the graph to show hidden menu items.
4. "What if Analysis," This is the real power of Session Management.

You start with a part that has already been archived or saved. Engineering comes in and says,
"I would like to try two different scenarios. We will call them, Test-A and Test-B.

Pick **Open Session** and enter **Test-A**
Open the File/part
Make the changes.
Pick **Backup session**

Pick **Open Session** and **enter Test-B**
Open the File/part again.
Make the changes.
Pick **Backup Session**.

Now engineering has the choice of going with Test-A or Test-B.

Pick **Open Session** and **pick Test-A** then **Test-B** to switch between the two. When a decision has been made Pick **File > Save**

The changes made in that session will now be saved to the archive file.

There are two ways to clean up the out-of-date sessions.

1. Click **File > Close All** then **File > Manage session > delete session**
2. Go to your User directory and delete the .Session directory.

The last feature concerning sessions we will cover here is the **Emergency session backup.**(See the Session part in Configuration> General tab)

If you have selected this option in the configuration, there is a good chance that your session will be backed up in the unfortunate circumstance of a crash. This session is written to a separate location from the standard Backup session(see File > Manage Session > Open Session). Session was our active session and EmergencyBackup001 was saved when ZW3D crashed.

This is done in case the crash is not able to create a clean save. We didn't want to overwrite your last legitimate backup.

Open ZW3D and you will be returned to where you last did a **Backup session** command.

Open the EmergencyBackup and compare the results to your previous backup. I suggest that you do a regen of the history to make sure that the file is not corrupt.

ZW3D puts the backup in a separate "emergency" directory so the user can check it out and use it if desired -- otherwise, he still has his last good backup.

The emergency session backup is only triggered by certain signals that we are able to intercept. Killing the process or pulling the plug doesn't get through to our signal handler.

Key Mouse terms used in this training manual

In an attempt to keep things simple and follow a standard format in this training manual we have implemented the use of a few abbreviations. As you go through the exercises that were designed for you to practice you will see abbreviations as described below.

<Left-click> means to press the left mouse button. (LMB)

<Middle-click> means to press the middle mouse button. (MMB)

<Right-click> means to press the right mouse button. (RMB)

<Right-click> Draw means to press the right mouse button and pick the Draw command. A variety of commands will be shown to you throughout this training manual to help locate commands. We will use the right click menus as much as possible since they give quick access to the available commands or options for the task at hand.

<Shift-click> means to hold down the <Shift> key and press the left mouse button. This will chain pick connected entities.

<Ctrl-click> means to hold the <Ctrl> key and press the left mouse button. This will unselect a highlighted entity.

Other conventions used in the training manuals

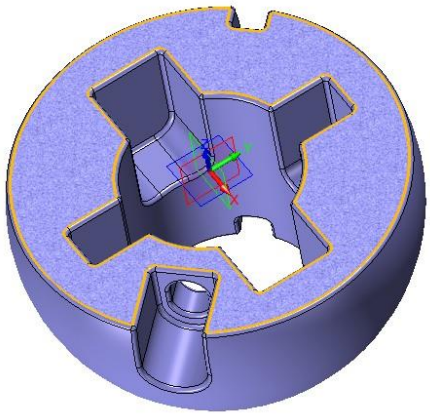
Edit>Scale This means select the Edit pulldown menu and select the Scale command.

The Object Editor

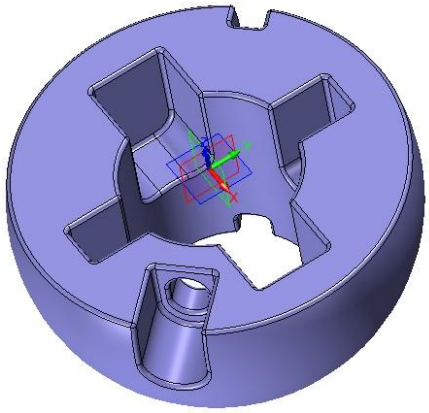
The **Object Editor** provides a quick and easy way to perform many frequently used editing commands. Simply move the cursor over an object such as a curve, text, or feature. That object will highlight. Right-click the mouse to display a set of editing commands tailored to that specific object type. You can also double-left-click on an object such as a sketch to activate it for editing or a curve to activate the curve editor.

These are quick response requests that require no additional selections or input. The table below describes the Object Editor functionality of each mouse button.

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.
 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.



Place your cursor over an object and it will highlight. Press the <Right-click> to display an object oriented menu of options for processing the highlighted object.
With no entity highlighted, pressing the <Right-click> will display a menu of ZW3D commands.



Pick multiple ZW3D objects with the LMB to highlight them. Press the <Right-click> to display a menu of options for processing all of the highlighted objects.

Undo / Redo

If you make a mistake or change your mind, ZW3D allows you to undo (Ctrl+Z) the last executed command. If you change your mind about undoing a command you can redo (Ctrl+Y) it. You can move back and forward through the undo/redo steps of your design. Only a few viewing related functions are not logged as an undo redo step.

"Ctrl + Z" is the keyboard macro for Undo.
"Ctrl + Y" is the keyboard macro for Redo.

Keyboard macros are displayed in Tool tips.

Control of the amount of memory allocated to storing the Undo and REDO data per ZW3D session is defined in the configuration.

Go to the **Utilities** pull down menu. Pick the **Configuration** menu. Then Pick the **General tab** and make your adjustments.

The ZW3D Data Manager

The **ZW3D Data Manager** is a tool that you can use to control many aspects of your part, lights, drawing or CAM plan. It contains the **History Manager**, **Assembly Manager**, **Layer Manager**, **View Manager**, the **Input Manager** and **Visual Manager**, each on

separate tabs. Not all of the managers will be available at any given time. Refer to the **HELP Manual** for each of the managers to understand how they operate.

Refer to the HELP Manual

Also refer to the **HELP Manual** for the following basic functionality.

- **Hints - Show Hints**
- **Part Level - About the Part Level**
- **Sketch Level - About the Sketch Level**
- **Drawing Packet Level - About the Drawing Packet Level**
- **Drawing Sheet Level - About the Drawing Sheet Level**
- **ZW3D - Configuration Form**
- **Function Keys - About**
- **Tool Bars – Utility - Tool Bars and Control Keys**
- **Other Keys**
- **Mouse - for Accepting Default Values**
- **File Level**
- **Blank Entities** (Blank, Unblank, Swap)

Lesson 1

Managing ZW3D data at the Object Level

Launch **ZW3D** and OPEN the file "**Fundamentals.Z3**". See the \ Program Folder \training directory

You will see several **ZW3D** objects in this file at the lower right corner of the form. You may double-click on a object to open it or click the **Open** button to reach to object browser .We will be working with many of these objects throughout this training course.

At the **ZW3D** Object level we can perform a variety of operations in the currently active ZW3D file. We can perform file maintenance, **Copy** objects, **Delete** Objects, **Move** objects, **Rename** objects and **Preview** objects. We can even copy objects to other **ZW3D** files.

Let's practice some of these operations.

Click the **Open** button to reach to object browser

1. Pick the **"Part/Assembly"** icon  from the **ZW3D** root. A form will appear. We can use this form to create a new ZW3D object file and activate it for editing. A ZW3D file can contain any number of root objects objects (e.g., parts, assemblies, drawings, sketches, CAM process plans, etc.) related to a design.

   The Part/Assembly icon should be active  .

   Notice the **Template** window displays "**[Default]**", "**PartTemplate(MM) or PartTemplate(IN)**". Templates are a method for building your design, drafting and manufacturing standards into **ZW3D**. We will cover customizing templates to meet all of your corporate, department or personal requirements later in this manual.

   The Name defaults to "**Part001."**

   Pick **OK** to accept the default part object name.

2. **ZW3D** created a Part/Assembly object named "**Part001"** as indicated in the title bar

3. We won't be creating any geometry in these objects yet.

Exit Part  when you are finished. This command is also conveniently located on the RMB menu

4. Notice "**Part001"** has been added to the current ZW3D files list of objects.

5. Pick the **Sketch** icon from the **ZW3D** root  .

6. Notice the **Templates** window changes. We currently have no sketch templates defined per our standards.

7. Pick **OK** to accept the default sketch name.

8. **ZW3D** creates a **Sketch** object named "**Sketch001"** as indicated in the title bar.

The XY icon in the middle of the screen indicates 0,0. We also create a point and an anchor constraint at this location for your convenience.

9. Exit Sketch when you are finished.

10. Notice "**Sketch001"** has been added to the current **ZW3D** files list of objects.

11. So far we have added 2 blank objects to our **ZW3D** file. Let's delete them now since we don't really need them.

    In the **ZW3D** Objects form, **<Right-click>** over the "**Part001"** object

    The menu displayed offers a variety of functions that can be performed on this object. Pick **DELETE**. 

    <Right-click> over the "**Sketch001**" object and pick **DELETE.**

12. Let's review the **Preview** section in the top of the **ZW3D Objects Manager** form.

    Select the "**Graphics**" option from Preview pull-down list.

    <Left-click> over a couple of the Part objects in this form. A graphic preview of each object will appear in the upper left corner of the screen.

    The preview can be displayed in wireframe or shaded mode. To change the display mode of the preview use the keyboard macro **<CTRL-F>** to toggle the display.

13. Select the **Attributes** option from Preview pull-down list.

    <Left- Click> over one or two of the part objects to display the **Part attributes** form. These forms contain **Bill of Material** information, parametric relations for deriving families of parts and user defined attribute information. This information is also used to semi-automatically fill in the title block on a drawing. (See **Edit > Regen Text** in the sheet)

14. Dismiss the **Attributes** form if it is still displayed on your screen.

15. Set the **Preview** to **OFF**. Next we will review the **Copy** option.

16. **<Right-click>** over the "**Blow Mold Completed**" object and choose **Copy**  from the shortcut menu. This **copies the object to the clipboard**.

17. **<Right-click>** and choose **Paste** from the shortcut menu. A new object named Blow Mold Completed_1 is created.

    You will see the additional copy appear in the **ZW3D** Objects form.

18. You can also open another file and pick **Paste** to copy the Blow Mold Completed part object. The two parts are not linked in any way but the history is maintained.

19. Let's talk about **Rename**  . This option is found on the shortcut menu when you **<Right-click>** an object. **Rename** allows us to change the name of an existing **ZW3D** object.

Try renaming our new copied part.

20. Let's edit **Blow Mold Completed**. To do this double click on its name or **<Right-click>** and choose the Edit.

21. Let's review some of the display manipulation commands.

    Use the keyboard macros or pick the icon as shown in the following steps.

     Set Display to **Shade**.

     **Zoom Window**.

     **Zoom All**.

     **Top View**.

     **Front View**.

     **Left View**.

     **Right View**.

22. These next few steps will help us understand how to use the **Dynamic Rotate, Pan** and **Zoom** commands in **ZW3D**.

    < Middle -click>and drag will allow you to **PAN** the display. Try this.

    < Rolling the mouse > will allow you to **ZOOM** the display. Try this.

    <Right-click>and drag will allow you to **ROTATE** the display. Try this.

    Dynamic rotate rotates about the view origin.

you can simply RMB - Drag to dynamically rotate, LMB - Drag to Pan.

23. Use the 

key to rotate your model. While the F3 key is pressed, **Dynamic Rotate** will be constrained about the **Z-Axis.** **Pan** and **Zoom** are not affected. Try this.

24. Use the 

key to rotate your model. While the F4 key is pressed, **Dynamic Rotate** will be constrained about an imaginary axis starting at the View Origin and projecting perpendicular to the operator. This is referred to as the display axis. **Pan** and **Zoom** are not affected. Try this.

25. Right-click in the **Tools** pull down menu >**Customize**. You can press a key or key combination to assign hotkey.

Chapter 2
Understanding the Right Click Menu

Mouse Input Options

When you **<Right-click>** during many commands that prompt you for input **ZW3D** will display a menu of the input options for that command prompt. The options that appear are command sensitive and will vary depending on the situation. The input options index lists all of the available input options.

Following are a few options that may be available in the right-click menu.

Grid - Use this input option to constrain the cursor to select points on the active sketch grid.

Absolute - Use this input option to specify absolute X,Y,Z coordinates as point. The coordinates are measured from the world coordinate frame of the active part.

Relative - Use this input option to select a point by entering an X,Y,Z coordinate value that is relative to the last point selected.

Critical - Use this input option to have the cursor highlight critical points (endpoints, midpoints, datum plane origins, etc.) of lines, edges, curves, and datum planes during point input.

On Curve - Use this input option to constrain the selection to only those points that lie on the highlighted curve. *As you move the cursor over a curve only points on the curve will be highlighted.*

Tangent - Use this input option to select an endpoint that is tangent to a line, arc, circle, or curve.
Use this input option to switch back to the **Default** point selection mode from other modes such as

Default – **Relative, Critical, or Tangent**. **Default** mode is the absolute point you pick or the coordinates you enter. Also use Default mode to snap to grid points that are in close proximity with other Critical points.

Entity - Use this input option to select an entity as input.

Lesson 2

Sketching with the Right Click Menu

Sketching with the Right Click Menu

1. Launch **ZW3D** and **OPEN** the file "**Fundamentals.Z3**".

See the \training directory

You will see that there are several ZW3D objects in this file.

2. Pick the **Sketch** icon from the **ZW3D** root.

3. Name the sketch **Training**.

   You will be placed in the **Sketch** mode and ready to start creating your first sketch. The sketch Tabs will be displayed automatically.

   The XY icon on the screen indicates (0, 0). We also put a point and an anchor constraint at the origin by default.

4. In this first lesson we are going to get familiar with the **ZW3D Draw** command by creating the three simple profiles shown on the following page. We will approximate the size of the profiles by free hand picking. We do not care about dimensional accuracy for now. That will come in a little while. Pay close attention to the active tangent condition before picking each point.

   ZW3D is an intelligent sketching tool that generates implied geometric constraints based on points you pick. As we become familiar with the sketcher, we will learn how to control these constraints. For now, let's accept the defaults and move on to the next step and begin sketching.

5. **<Right-click> Draw**  . Start with the left most profile shown in the figure below.

   Start on the left side of the screen and pick point 1.

   Don't push any more buttons and don't move your cursor.

   Let's first talk about what is happening with **Draw** after picking the first point.

   Move your cursor to the right of the point you picked until the line dragging is horizontal. Notice ZW3D is attempting to imply a Horizontal constraint. Don't push any buttons yet.

   Move your cursor up or down from the point you picked until the dragging line is vertical. Notice ZW3D is attempting to imply a Vertical constraint. Again, don't push any buttons yet. We're getting there.

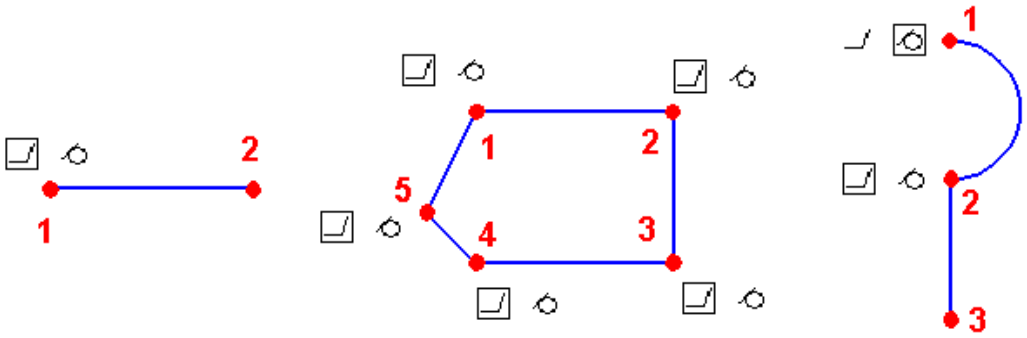
   Now, notice the two symbols  **(C0)** and  **(tangent)** are displayed next to the selected point.

   These symbols indicate the tangency condition at this point. The symbol in the green box is currently the active condition. By default the  (C0) tangency condition is active. We can draw lines when this condition is active as you saw when moving your cursor earlier. Selecting point 1 again will toggle the active tangency condition to  . We can draw arcs when this is the active condition. Move the cursor to see what we mean, but don't push any buttons yet.

With your cursor over the first point you picked, press the **LMB**. Notice the tangency condition changes to . An arc should drag on the screen.

With your cursor over the first point you picked, press the **LMB** again. Notice the tangency condition changes back to . Pick point 2 as shown in the sketch to create the horizontal line.

Press the **<Middle-click>** to finish the **Draw** command.



6. **<Right-click> Draw**, create the other two (2) profiles. Remember what we talked about above to control how the **Draw** command acts.
7. Press the **<Middle-click>** to terminate **Draw** when you are finished and **Save** your work.

Refer to the HELP Manual

Refer to the **HELP Manual** for more Quick Draw exercises.

- Quick Draw – Lines
- - Tangent Arcs
- - 3 Point Arcs
- - Radius Arcs
- - Circles
- - Curve through Points

Managing Implied Constraints within a Sketch

1. In this next example of sketching we are going to show you how the implied constraints can be controlled as you create the sketch. ZW3D offers two options in the sketch mode that will allow you to control how implied constraints are created. They are "**Focus**" and "**Smart Pick**". Both of these options are available from the **<Right-click>** while using the various sketching commands.

2. Pick the **Sketch** icon from the **ZW3D** root.

3. Name the sketch **Focus**.

4. We are going to draw the rectangular shapes shown in the figures below. Notice that each figure has the bottom and right edges of the rectangle drawn. We want the top line to be drawn so it is perpendicular to the right line. If we allow ZW3D to generate its implied constraint, it will place a parallel constraint between the bottom horizontal line and the top horizontal line as shown in the left figure below. We want a perpendicular constraint generated.

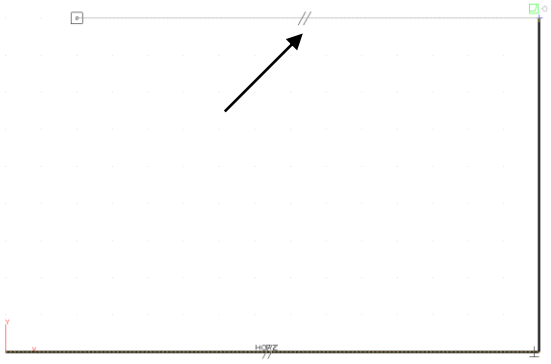
   <Right-click> Draw and create the 2 lines shown in the figures below. Stay in the command.

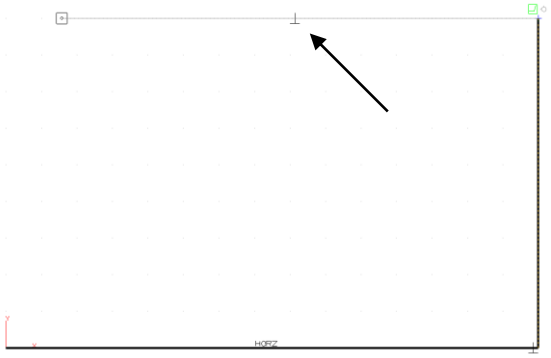
   Move your cursor to the left as if you are going to draw the top horizontal line. Notice as you move it, the constraint changes from perpendicular to parallel.

   <Right-click> and choose **Focus** and pick the right vertical line.

   Move your cursor to the left. Notice as you move it the constraint changes to perpendicular no matter how far to the left you move it. The focus has been changed so the implied constraint will be associated to the line you just picked.

   Complete the rectangle.

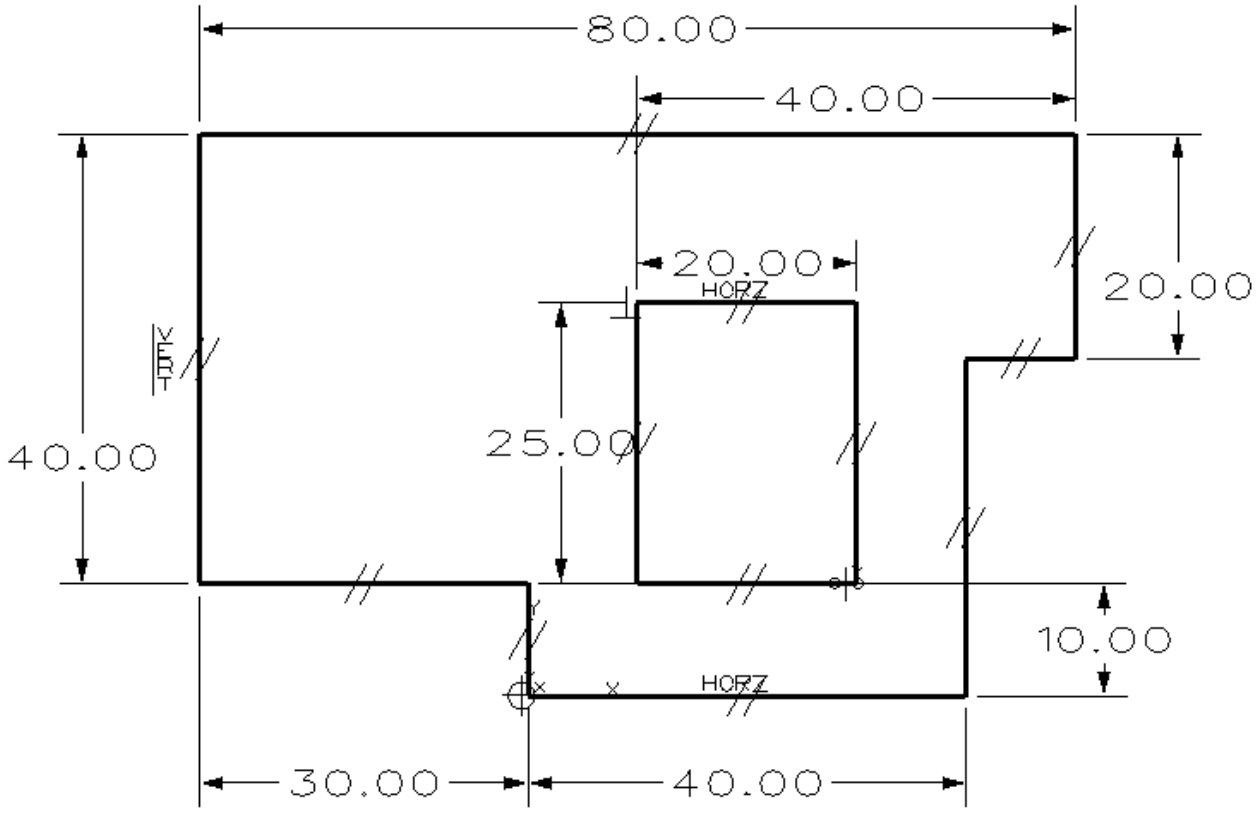




5. **Exit** the sketch when you are finished. 

Using ZW3D Coordinate Input Methods

1. In these next few steps we will work with the various coordinate input methods available in **ZW3D** to create the sketch shown in the following figure. This sketch consists of an inside and outside shape that will be part of the same sketch.



2. Pick the **Sketch** icon from the **ZW3D** root.

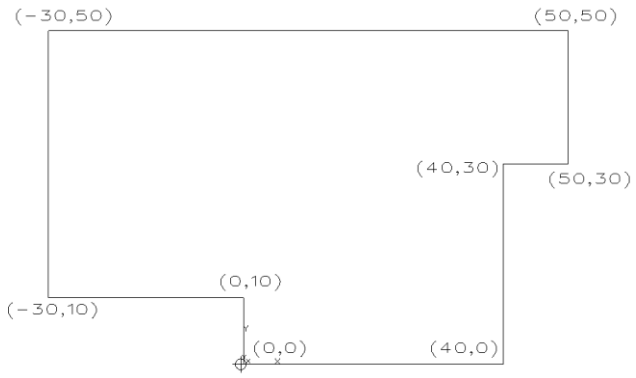
3. Name the sketch **Coordinate_Input**.

4. **<Right-click> Draw** to create the outside shape shown in the figure below.

5. **<Right-click> Default** (tells coordinate input to reference the 0,0,0 World Coordinate System.)

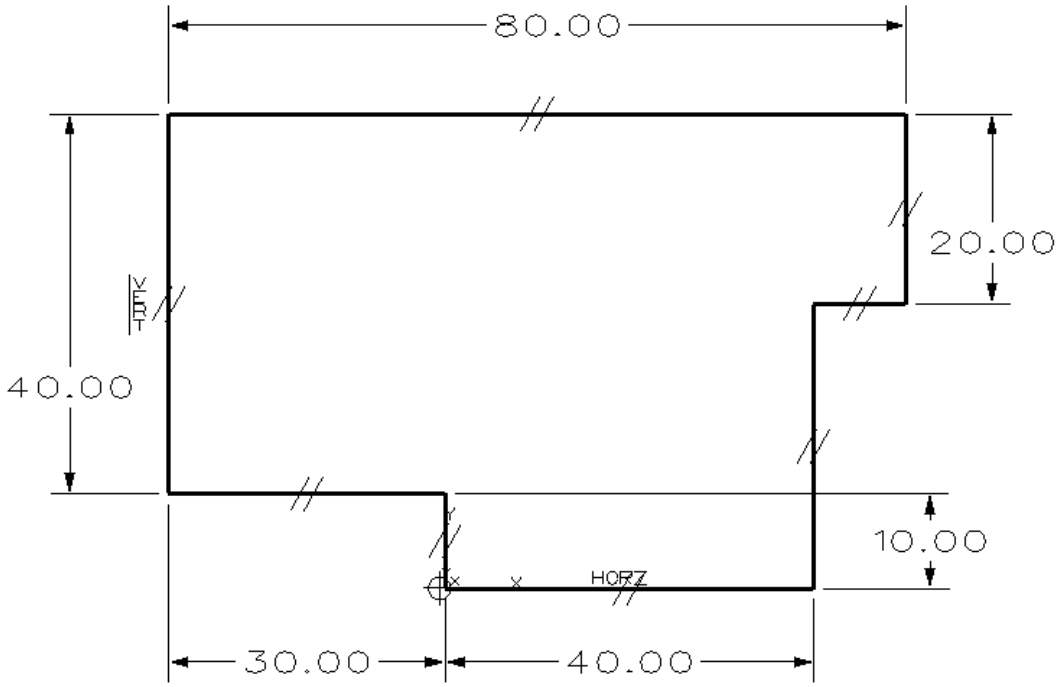
Type the following coordinates and hit ENTER on the keyboard.

Start point: 0,0
Specify point: 40,0
Specify point: 40,30
Specify point: 50,30
Specify point: 50,50
Specify point: -30,50
Specify point: -30,10
Specify point: 0,10
Specify point: 0,0



6. **<Right-click> Dimension** to add the dimensions shown below.

7. **<Right-click> Auto Constrain** and pick the (0,0) point. This command will automatically add all of the "Horiz", "Vert" and "Parallel" constraints you see below. The sketch is now **well defined**. The sketch is anchored at (0,0) and any changes you make to the dimension will update the sketch correctly.



8. Next we are going to create the inside shape. We will do this using a few of the input options available from the **<Right-click>** menu. We want to start the sketch so the left vertical line of the inside profile is precisely half way between the left and right most vertical lines.

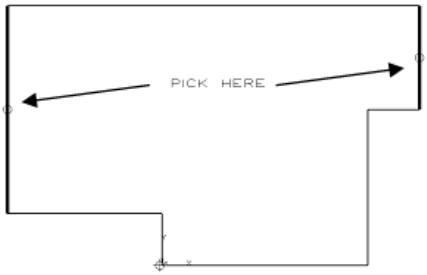
9. **<Left-click> Rectangle**  **,** and select the **corner method**  .

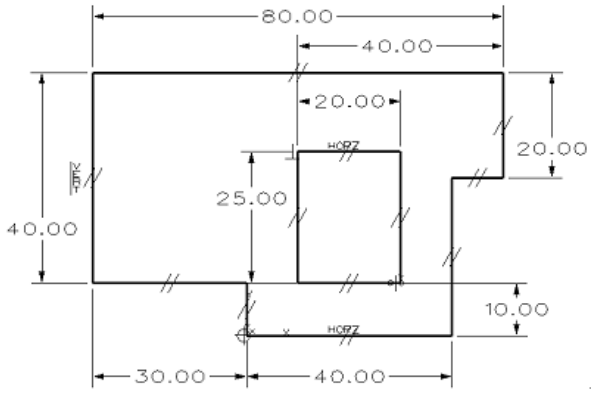
10. **<Right-click> Between.**

11. **<Right-click> Critical.**

12. Start by picking the 2 points indicated in the left figure below. Then **<Middle-click>** .

13. Complete the internal rectangle by dragging the lower right corner down until it snaps to the top of the 10.00 vertical line.





14. Complete the sketch by adding three more dimensions as shown.

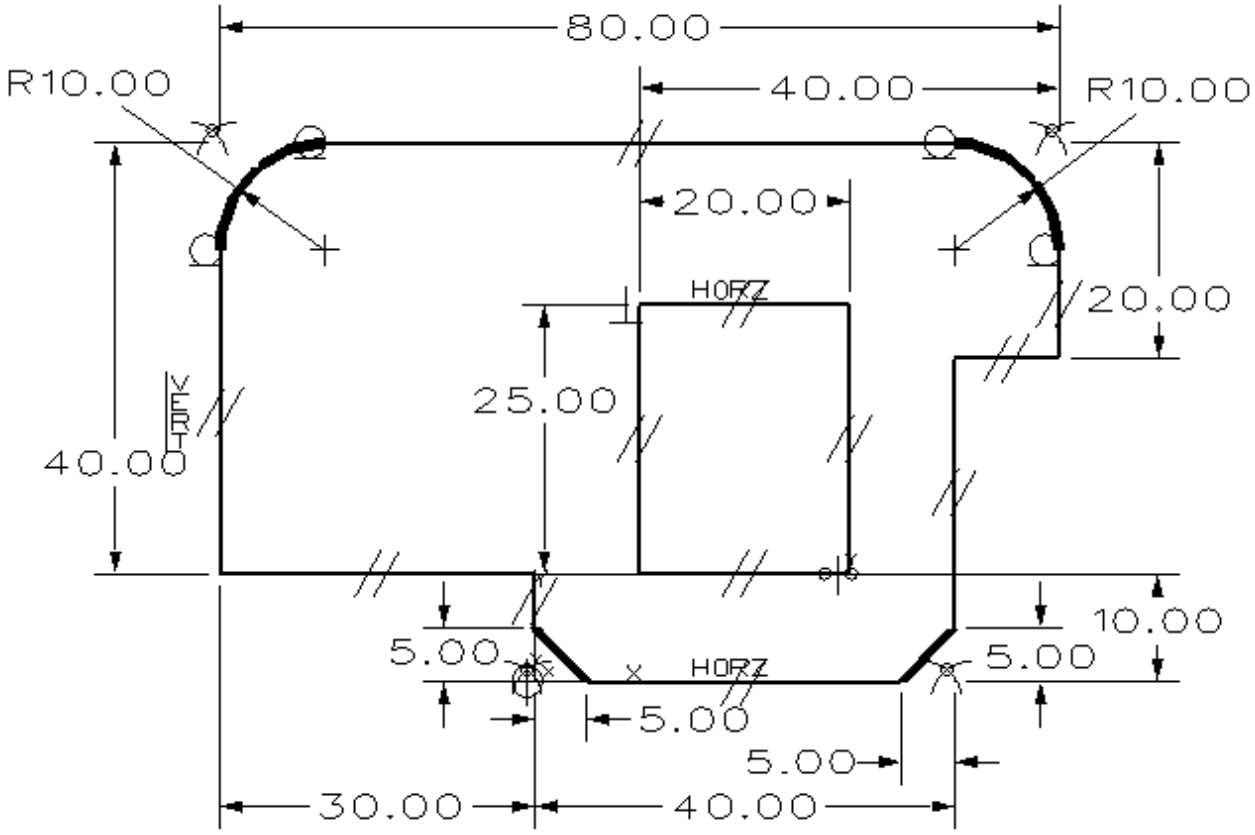
15. The coordinate input options we used to draw the sketches above can also be used with a wide variety of commands available while in a sketch. Simply **<Right-click>** to see what your options are when in any **ZW3D** command.

16. Use the "**F2**", **PAN, ZOOM** and **ROTATE** commands to adjust your display as required. Remember, while holding down the **F2** key, **PAN** is **<Middle-click>and drag**, **ZOOM** is the **<Middle-click>** and **ROTATE** is the **<Right-click>and drag.**

17. **<Right-click> Fillet** and add a couple fillets to the sketch as shown below.

18. **<Right-click> Chamfer** and add a couple chamfers to the sketch as shown below.

19. Double click on a couple dimensions and change their values slightly. **ZW3D** sketches can be dimension driven.


80.00
R10.00
40.00
R10.00
20.00
HORZ
20.00
25.00
40.00
VERT
10.00
5.00
HORZ
5.00
5.00
5.00
30.00
40.00

Chapter 3
Understanding Sketch Tab

Sketch Tab - Sketch Level

Let's begin by reviewing the Sketch Tab that are available while in the sketch mode. As you change from **Part** mode to **Sketch** mode you will notice the Sketch tab icons change to reflect the mode you are in. This style of interface will prevent your graphic screen from becoming cluttered with Sketch tab that will not be getting used. The **ZW3D** interface stays focused on the task at hand.

The Sketch Tab contain the most used sketch level commands. Please refer to the **HELP Manual** for more about each command on the sketch level Sketch Tab.

There are several tabs in the sketch level including **Sketch** tab, **Constraint** tab, **Tools** tab and **Inquire** tab.

If for some reason you want to **hide** one of the tabs, Right-click in the Ribbon border and select or clear Ribbon Tab.

Right-click and select Hide， you can hide Ribbon button.

The Ribbon support large ICONS and small ICONS mixed arrangement. Right-click and select **Large Icon** or **Small Icon**， you can modify Ribbon Button size. Size is automatically adjusted by default.

Lesson 3

Working with ZW3D Sketching Tools

1. Launch **ZW3D** and **OPEN** " **Fundamentals.Z3**".

2. Pick the **Part/Assembly** icon from the **ZW3D** root. Name it **Part_Lesson_3a**.

3. Let's begin this lesson by creating some layers to place our shapes onto. ZW3D allows you to place shape geometry on various layers as required.

4. Now select the **Layer Manager**. 

   Only the Default layer should appear in the Layer Manager. We are going to create the others.

5. Create each of the following layers one at a time:

   "sh1", "sh2" and "sh3"

   Pick **New Layer** from the **Layer Manager,** specify "**sh1**" in the name field of the **Layer Form**, and then pick **OK**. The layer description is optional. Enter a description if you would like.

6. Repeat the procedure to create layer **sh2** and **sh3**.

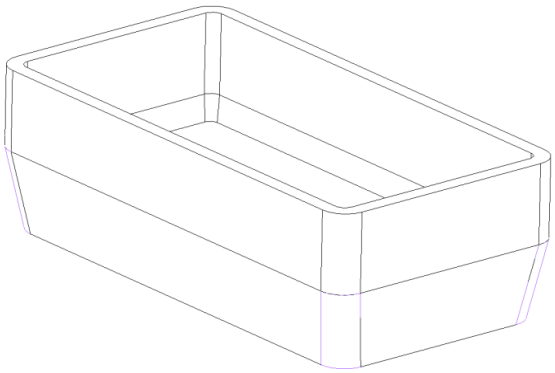
   Pick **OK**, save settings and exit the **Layer Manager**.

   We will discuss how to add these layers to your Templates file later in this manual.

   After you have created the layers, right click over layer "**sh1**" in the **Layer Manager** to reactivate it.

   Leave the **Layer Manager** (and the ZW3D Data Manager) displayed as you continue with this lesson.

7. Let's begin building a simple part using some basic **ZW3D** commands. These commands may seem simple and basic upfront, but when combined with various options, they can put a lot of modeling power at your finger tips. The part we are going to model is shown in the following figure. It was designed to help you understand some basic features.



8. From the Shape Tab pick the **Extrude** command  .

<Middle-click> to accept the defaults, (twice). The XY plane will become the active sketch plane and ZW3D will move into Sketch mode.

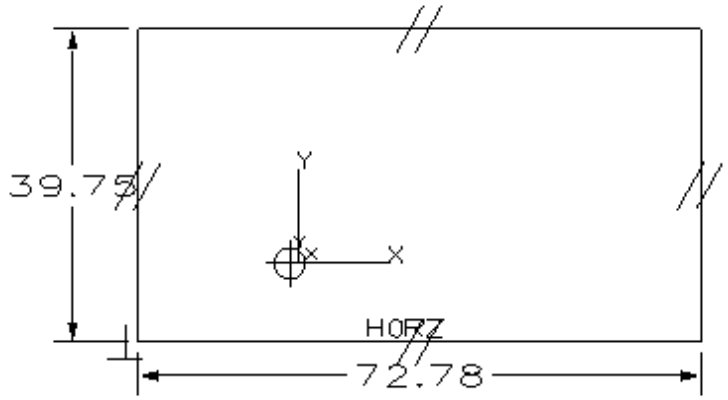
9. From the **Sketch Tab**, use the **Plane and 2 Points Rectangle** method to draw the rectangle shown in the figure below. Do not place one of the corners on (0,0). Your dimensions may vary. That's OK.

   Notice the rectangle is automatically dimensioned and constrained. At first glance one would think the sketch is well defined. In some respects it is. The size of the rectangle is fully defined. **ZW3D** does not require a sketch to have any constraints to be considered valid. If you are at the concept stage of a design, just sketching general shapes may be enough for you to complete your job. You can apply the geometric constraints at a later time when you have determined the job is going to be taken to completion.

10. Geometric and dimensional constraints can be erased as required using the **Erase** command.

    Simply highlight a dimension or geometric constraint > **<Right-click> Erase**. You could also pick multiple constraints and **<Right-click>** Erase them.

    Let's leave the constraints as is for now though.



11. <Right-click> **Exit Sketch.**

    OR pick the **Exit Sketch** icon from the Sketch Tab.

12. After we exit the sketch, **ZW3D** remembers that we started to create an Extruded Solid and continues asking the appropriate questions to complete the shape. There are two methods of providing the input:

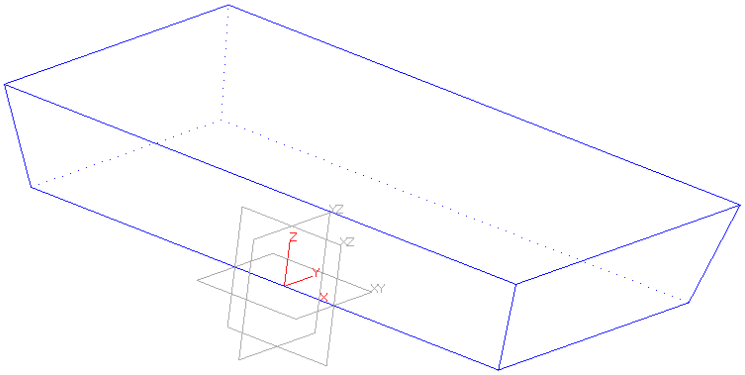
- You can fill out the options form completely then click OK.
- You can respond to the command prompts. However, you are only prompted for required input. Options such as Draft Angle would still have to come from the options form.

  You could dynamically adjust the numeric values by picking the up or down arrows next to each numeric field. Try this and adjust the **Start, End** and **Draft Angle** values as shown in the **Options** form.

Remember, when prompted for the Start, if you are using the value of zero, you can simply <middle-click>. For now, <middle-click> for the **Start**, enter 15 for the **End** and 15 again for the **Draft Angle** (on the Options tab of the form).

Notice the green 'Define' buttons on the form.  Green indicates that these values have been provided. Yellow means no value has been given but it is not required either. Red needs to be defined. If we had created a separate sketch and then picked the Extrude, the profile button would have been red. You could still fill out the values like we just did, then click on the red define button and pick the profile sketch. They do not have to be done in any particular order.

Pick OK to complete this step.



13. We now have a base solid (1st solid feature in this part).

    All of the dimensional information that was in the sketch is still there.

    Click on t sketch from history record and associated dimensions. You may need to change to the wireframe display to see the sketch.

    Select **Sketch** from the **Pick Filter** and double click on the sketch to edit it. **ZW3D** should be in the Sketch mode now noted in the title bar.

    Now, double click on the dimensions to change the width to 45 and the length to 90.

    You could make changes to any of the dimensions displayed. You are actually editing the history of the model. When you are done making changes and exit the sketch, the 'extrude' will replay and the new model will be displayed.

14. You should still be in the sketch mode and editing the rectangular sketch you created earlier.

    From the Constraints Tab pick the **Constraint Status**  command.

    Your sketch should not be well defined yet. If it were well defined it would be GREEN.

    The constraint status should be similar to the following:

2 DOF left/Geometry 4Under constrained 1 Well defined/Dimensions 2 Solved/Constraints 5 Solvd.

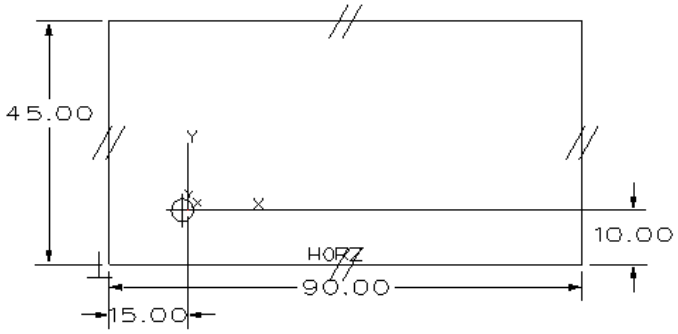
15. Let's fully constrain it now. **<Right-click> Constrain.**

    The first thing to notice is that some of the geometry is **Under constrained**. We are in fact, missing two dimensions or constraints. Let's let the software finish the job.

    For the Base point, enter **0,0**.

    We will talk about how to apply some control to the **Constrain** command later in this course.

    The entire sketch should turn GREEN and the Geometry status should all be **Well defined**.



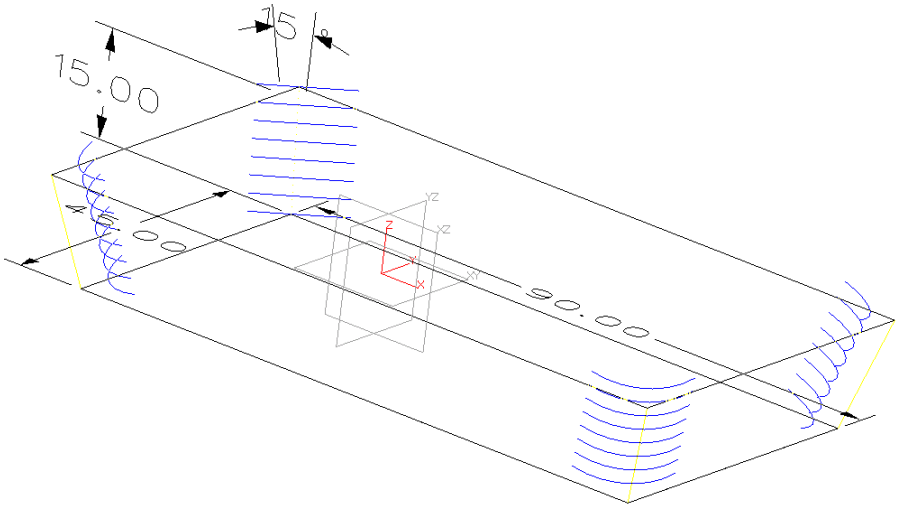
    This may or may not be constrained according to your design intent. We could fix this in a variety of ways. You could erase the new dimension on the right and add one to the left edge of the sketch. Or we could erase the dimension and **move** the rectangle so the corner is at (0,0). This would probably be best. You can try this if you want. (Pick the 4 lines, then <right-click> Move.

    Dismiss the **Inquire Constraints** Menu.

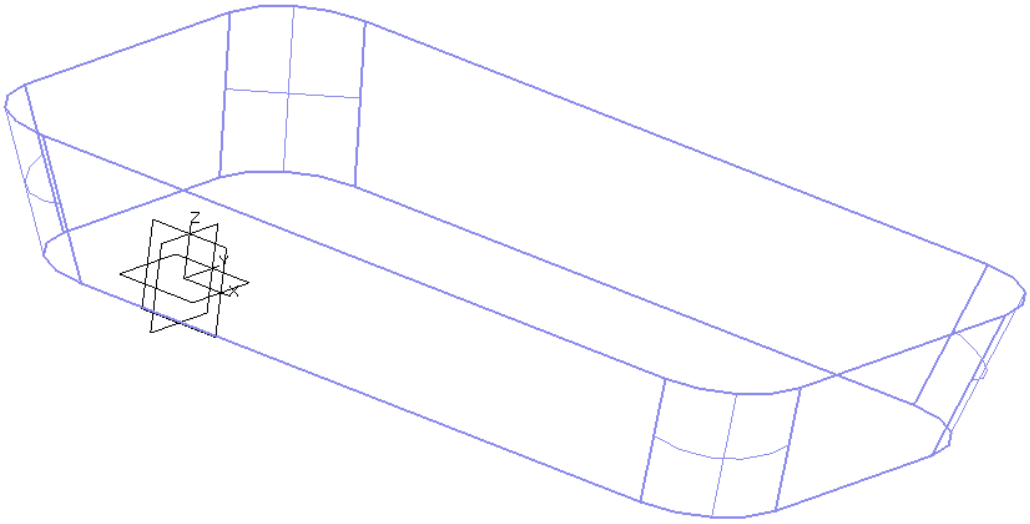
16. <**Right-click> Exit Sketch.**

The extrude command has now automatically replayed.

17. Pick the **Fillet** command  from the **Shape Tab**. Add **10mm** fillets to the four corners as shown in the figure below. **ZW3D** will highlight the fillet size for visual verification.



Your model should appear as shown in the following figure.



18. In a previous step we made some dimensional changes to our rectangular feature by double clicking on an edge of the model to display the sketch and its associated dimensions. Another method we could use to edit feature dimensions is to pick on the "**Dimensions**" icon located in the toolbar. See the figure below for reference.

    Pick **Dimensions**  to display the model dimensions. This is a toggle command. If the dimensions are already displayed they will turn off. Pick it a second time if required to display the dimensions.

19. Pick **Wireframe mode**  . Double click on the **Radius** dimension and change them to **5mm**. Remember to set the necessary **Pick Filter**, in this case **Dimensions**.

20. Double click on the **Draft Angle** dimension and change it to **10** degrees.

21. The history will automatically **Regen**  because **Auto regen on edit** is set by default in the **Configuration>Part** tab.

22. We have covered a lot of functionality in **ZW3D** so far using simple shapes. The simple shapes allowed you to focus on the type of functionality we were trying to cover instead of getting lost in complex shapes. All of the editing functionality we covered so far applies to Solids, Surfaces and Wireframe modeling techniques (hybrid modeling).

23. Next, let's talk about the **History Manager**. 
    You can use the **History Manager** to view the active part's history and as a selection list for some commands. You can right-click on a listed item to display the **Object Editor** Menu and select a command to act upon the selected item. The grayed list indicates that the operations which are not executed during a history replay.

    Edit > History Operation > History Play  , or <Right-click> in the space , select History Play

The **History Manager** will change when you pick the **Replay next operation** command  . All of the features created will appear in the lower list, and the model should disappear from the graphics screen.

Notes: History Manager contains a upper list and the lower list. Through the indicator shown in blow to differentiate.

The upper list shows all history-based operations within the active part/assembly. The lower list shows the operations which are not executed during a history replay.

A few commands such as **Continue to End** and **Delete Unplayed Features** are shown in the right menu of indicator. **Delete Unplayed Features** will remove the remaining history that has not been replayed.

From the **History** Window, **<Right-click>** over "**Sketch1**" and pick "**Continue To.**"

ZW3D will replay all of the features above this sketch. There should be the 3 initial datum planes in your model appearing on your screen now.

Pick the **End Replay**  icon in the **History** form and pick "**Yes**" to terminate.

You just erased all of the features created in this model. There are a variety of ways we could have erased the model. This method works out to be the cleanest as far as keeping the model history small. This is not always the best method to use though.

If we used the "**ERASE**" command to erase the entire model, the **History Manager** would keep a record of that operation. The advantages are that we could recover the erased features simply by stepping through the history and removing the **ERASE** operation.

We will talk more about how the **History Manager** can be used to assist in editing models later in this manual.

24. Pick the **UNDO** command repeatedly to bring back your model. **ZW3D** has unlimited **UNDO** and **REDO** potential.

25. We will leave the ZW3D Manager window open.

26. Turn the **Dimension display off** if necessary. 

27. Let's create a cut feature in our model. Pick the **Extrude** command again  .

    <Middle-click> to create the new sketch.

    Turn **Reference face edges ON** in the **Optional Inputs** section of the form.

    Pick the top of the model for the Insertion Plane.

28. The reference edges will appear as dotted or dashed lines around the model. We are going to use these edges for our extrusion sketch. They are associated to the model. If changes are made to the model these edges will update accordingly.

    Highlight any of the reference edges and **<Shift> Pick**. This will chain pick all of the edges.

    <Right-click> Toggle Type 

    ZW3D will change the line type of the edges to a recognizable font for extruding, however they are still reference lines so if our initial sketch updates this one will follow.

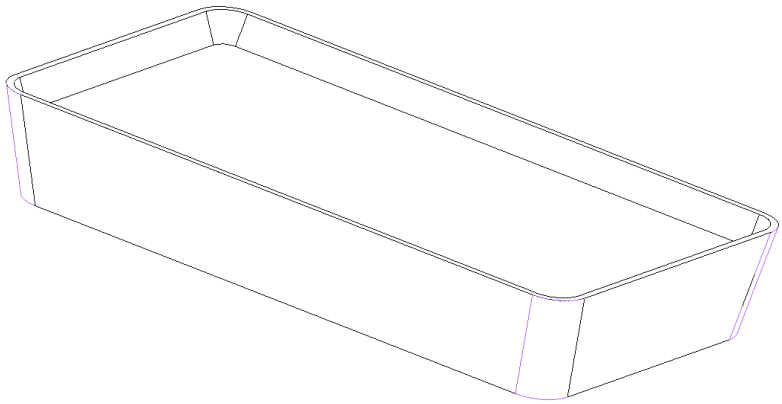
29. **<Right-click> Exit Sketch.**

    After we exit the sketch, **ZW3D** remembers that we started to create an Extruded Solid and continues asking the appropriate questions to complete the solid.

    Choose **Remove** method, 
    Set **Start** to 0, **End** to -4.0, **Draft Angle** to 25,
    Set **Offset** to Shrink/Expand, and **1st offset** to -1.

    Notes: Shrink/Expand creates an offset by shrinking or expanding the profile. The shrink/expand direction is determined by the distance value (1st offset). A negative (-) value expands to the outer side and a positive (+) value shrinks to the inner side of the profile.

30. Pick **OK** to complete the command. Your model should appear as shown in the figure below.



31. Let's add another feature to complete our design. Pick the **Extrude** command.

    <Middle-click> for a new sketch

    Reference face edges should still be **ON** .

    Pick the thin top face of the model created in the previous step as the Insertion Plane.

32. The reference edges will appear as dotted or dashed lines around the model. We are going to use these edges for our extrusion sketch. They are associated to the model. If changes are made to the model these edges will update accordingly.

Highlight any of the reference edges and **<Shift> Pick**. This will chain pick all of the edges. There are two of them in this case.

<Right-click> Toggle Type

ZW3D will change the line type of the edges to a recognizable font for extruding.

33. **<Right-click> Exit Sketch**

34. Again, **ZW3D** remembers that we started to create an Extruded Solid and continues asking the appropriate questions to complete the solid. Again, adjust the form as indicated below.

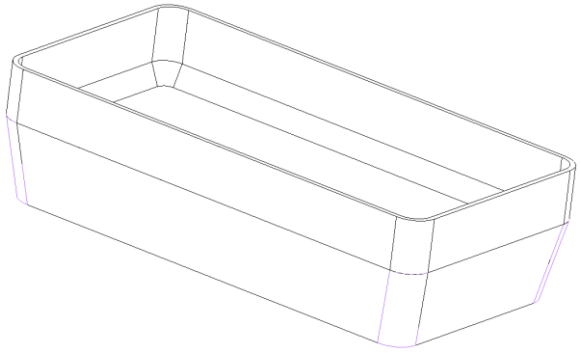
Choose **Base** method, 
Start = 0, **End** = 10,
Draft Angle = 0.

Make sure the Draft Angle is set back to "0" to avoid the following error message.

WARNING: Offset curve intersections, possible bad geometry.

Note: An alternative method would be to use "Face Offset" (1 command instead of 2)

Your model should appear as shown in the figure at the right.



35. **<Right-click> Redefine Last** 

ZW3D will display the Options form for the last feature you created, the extrusion. Make any reasonable changes in the form and pick **OK** to process.

Change the operation from a BASE feature to **ADD**.

Change the "**End**" value to **15** and pick **OK**.

36. Let's make a few dimensional changes to our part.

Step Thru History again .

<Right-click> over "**Extrude2_Cut**" and pick "**Rollback**"

ZW3D will replay the model up to that feature.

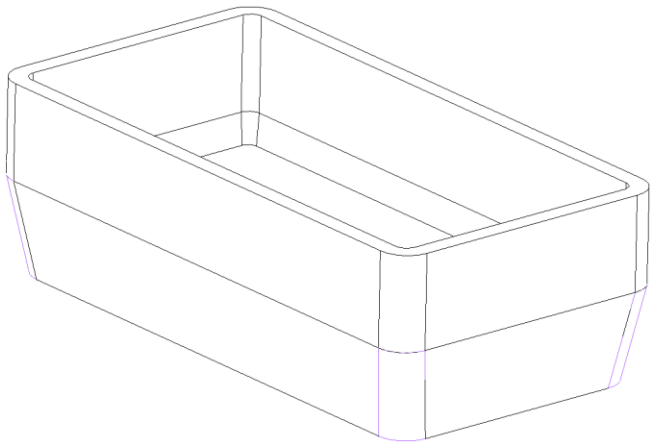
Pick the pencil icon at the bottom of **the History Manager** to edit the next operation. **ZW3D** will display the **Options** form for that feature.

Change the End to **-5**, the **Draft angle** to **5** and the 1st **offset value** to **–3**.

Pick **OK** to process the changes.

Pick the **Replay next** operation icon located at the bottom of the **History Manager** form until the remainder of the model is updated.

Notice the final extrusion reflects the change from **1mm** to **3mm** for the wall thickness.



Pick **Exit Part** when you are finished.

The **ZW3D Objects** window will appear.

37. Let's continue practicing working with sketches.

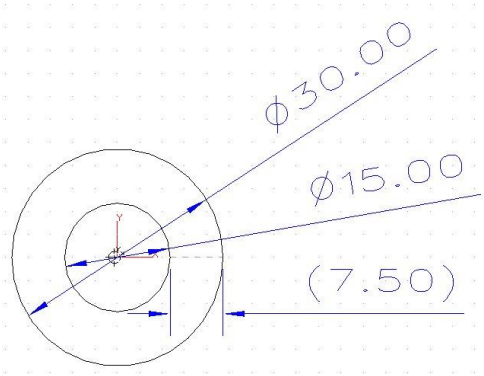
    Pick the **Part/Assembly** command from the **ZW3D** root.

38. Pick **Standalone Sketch** from the menu. Name the sketch **Lesson_3b**. Pick **OK**.

39. Go to the **Edit > Preferences** pull down menu and verify the **Units are set** to **MM** and the grid **Spacing** to **5** as shown at the right.

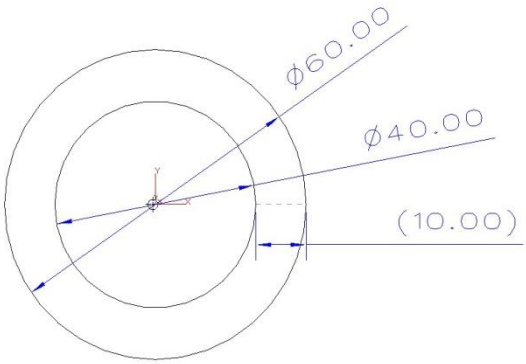
40. Select the **Concentric circles** command from the **Ready Sketch** panel of Sketch tab.

Place it at (0,0).



Double-click in the outside diameter and change it to 60.

Make the inside 40. The 10.00 dimension is a reference dimension and will update automatically.



41. Select the **Show All Constraints** command.

Since we used **Ready Sketch** in the construction of these circles, some constraints are blanked and need to be visible now.

42. Next we will make a copy of these two circles and place them to the right.

43. Make room for this step by zooming out and panning the circle to the left side.

44. Pick the **Edit > Copy**  (from the pull down)

For **Entities to copy <Right-click> Pick All**.
Adjust the **Optional Inputs** in the options form as indicated below.
Set **Direction** to Horizontal,
Angle = 0, **Scale** = 1.25.

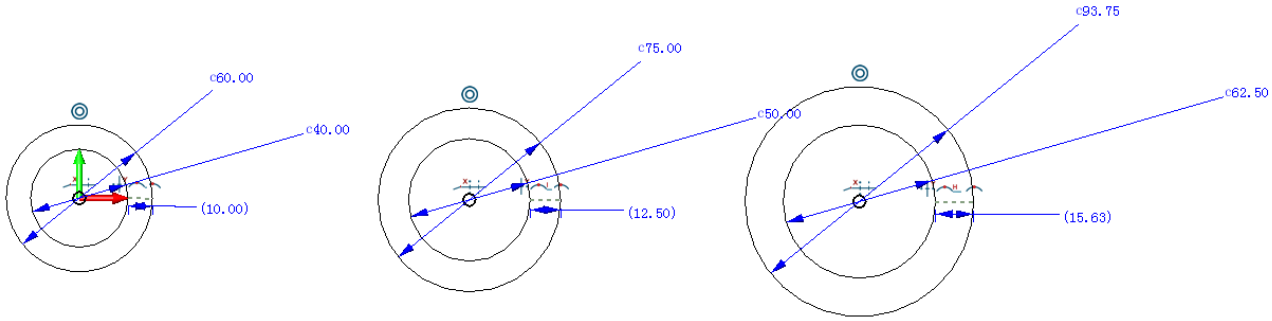
For **From point**, pick at the center of the circles.

After picking this point move your cursor around the display. You should notice that ZW3D is restricting movement to a horizontal direction as indicated in the options form. Play with the **Direction** option if you would like but be sure to set it back to **Horizontal** before proceeding.

Hit **<CTRL>-K** to turn coordinate display **ON** near the lower right side of the display

<Right-click> Critical and enter the coordinates **160,0** as **To point.**

Make 2 **Copies** and then hit **OK.**



45. Window pick the right most copy of the circles you just created.

<Right-click> Erase leaving you with 2 sets of circles.

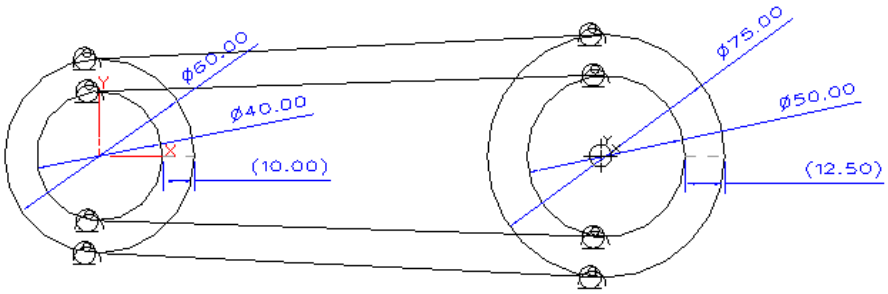
<Right-click> Draw.

<Right-click> Tangent.

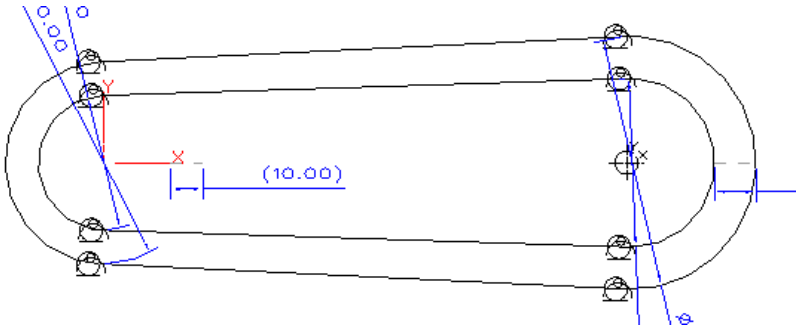
Pick the circles to create tangent lines as shown below. Repeat this command 4 times.

This will instruct **ZW3D** to create the lines tangent to the circle based on your pick points. You will see a tangent symbol appear on the screen for visual verification.

Notice geometric constraints are added forcing the lines to remain tangent to the circles until you instruct otherwise.



46. Select **Trace Profile** from the Sketch Tab and pick the closed boundaries shown below. You will need to do this twice. This keeps your geometry and constraints intact while selecting the portion of the sketch that you will ultimately want to extrude in the model.



47. We could have drawn this sketch using one command; **Draw**, which is accessible from the **<Right-click>** menu. We chose this method to introduce you to more of the **ZW3D** sketching tools. In practically every task you attempt with **ZW3D**, you will find there is more than one method. You will become familiar with the methods you prefer most as you learn more about **ZW3D**.

48. Pick **Exit sketch** when you are finished.

49. You should be at the **ZW3D Objects** level of "**Fundamentals.Z3**".

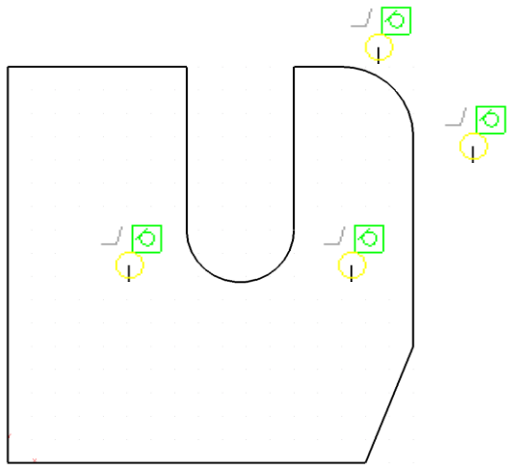
50. Pick **Sketch** from the **ZW3D** root and name the sketch **Lesson_3c**.

51. Go to the **Edit > Preferences** pull down menu and set the **Unit**s to **IN** and the **Grid Spacing** to **.25** as shown in the lower left figure below.

52. We are going to use the **Draw** and **Dimension** commands in this lesson.

Just so we sketch this part fairly close to the scale we want, Rolling the mouse and zoom the screen. We want the number in the lower left of the screen to be about 4.0 inches. This is the vertical height of the screen and that is about the height we want the sketch.

<Right-click> Draw.



Draw the profile shown in the following figure. Don't worry about the exact size of the sketch in this step. We will adjust it later.

Hint:When using the Draw command for the two arcs, make sure you are in the tangent mode  on both sides of the arc. This will ensure that you get a tangent constraint at both sides of the arcs.

53. Let's check to see if the sketch is fully constrained **or well defined**.

Pick the **Constraint Status** command  . **ZW3D** will display a form similar to the figure below. Green geometry is well defined. Your sketch is probably not well defined.

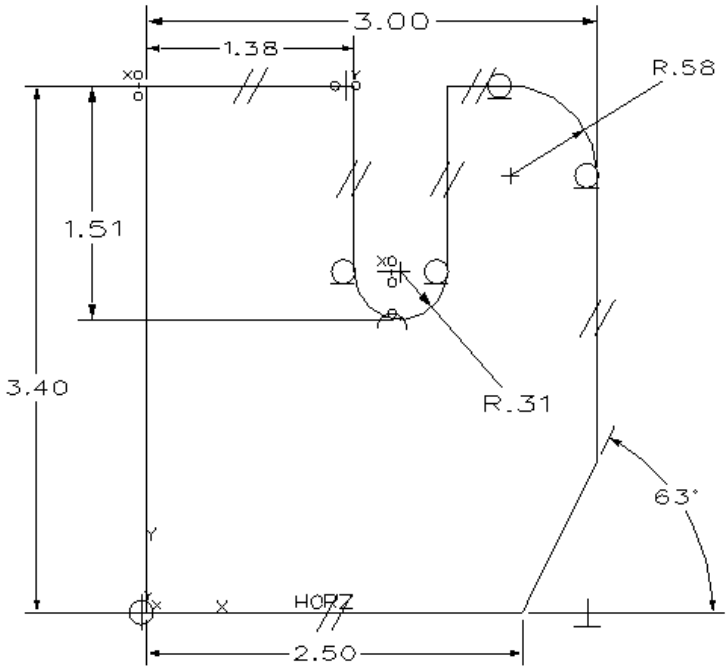
You result should be similar to the following:

Geometry
10 Under constrained
1 Well defined
Constraints
11 Solved

Our sketch is Under Constrained. We could, however, exit the sketch and continue modeling, but a good technique to remember here is to test the sketch to make sure that any anticipated changes will update the model according to your design intent. We need dimensions to define our intent. To do this let's move on.

Leave this form on the screen as you go through the next few steps. It will update as your sketch changes.

54. **<Right-click> Dimension** as shown in the following figure. Your dimension values may vary. That's OK. We will adjust them later.



Suggestions:

<Right-click> Dimension and add all of the required dimensions to your sketch.

If your sketch is not well defined yet, look for missing dimensions and geometric constraints and add them. HINT: You may need to add two parallel constraints.

If you cannot see anything missing, use the <Right-click> Constrain command and pick the end of one of the two lines near (0,0). Let ZW3D determine what is missing for your sketch to be well defined.

The **Dimension** command will generate a dimension based on where you select an object, and the order in which you select them. For example, the angular dimension in the figure above could have resulted in an obtuse angular dimension instead of the acute angular dimension.

55. All of the dimension values in your sketch are currently in inches. Let's change them to millimeters.

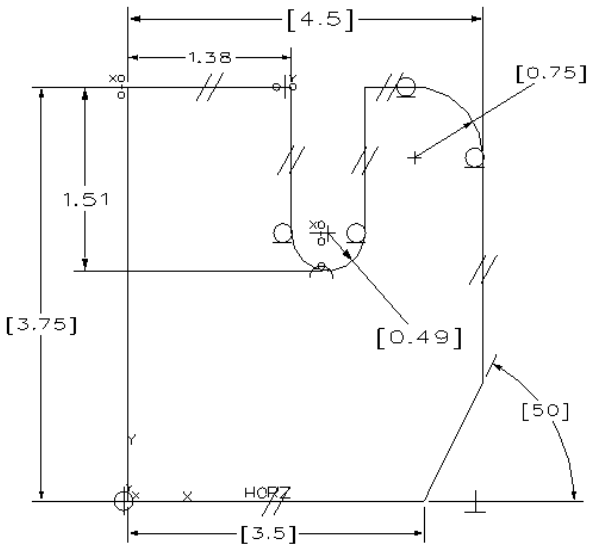
    Go to **Edit > Preferences** pull down menu.

    Change the **Units** to **MM**. All of the dimensions will change upon successfully completing this step.

    ZW3D allows you to change the units you are working in anytime during a design.

56. Change them back to inches using the same command. We will continue this lesson using the inch units. **Since this command is not history based and therefore not stored in the undo/redo buffer, an undo will not do!**

57. Now that we have a well defined sketch, let's change the values of the dimensions to match those shown in the following figure.



Double click on each dimension and change all of the values through the **Input Dimension Value** form to match the figure.

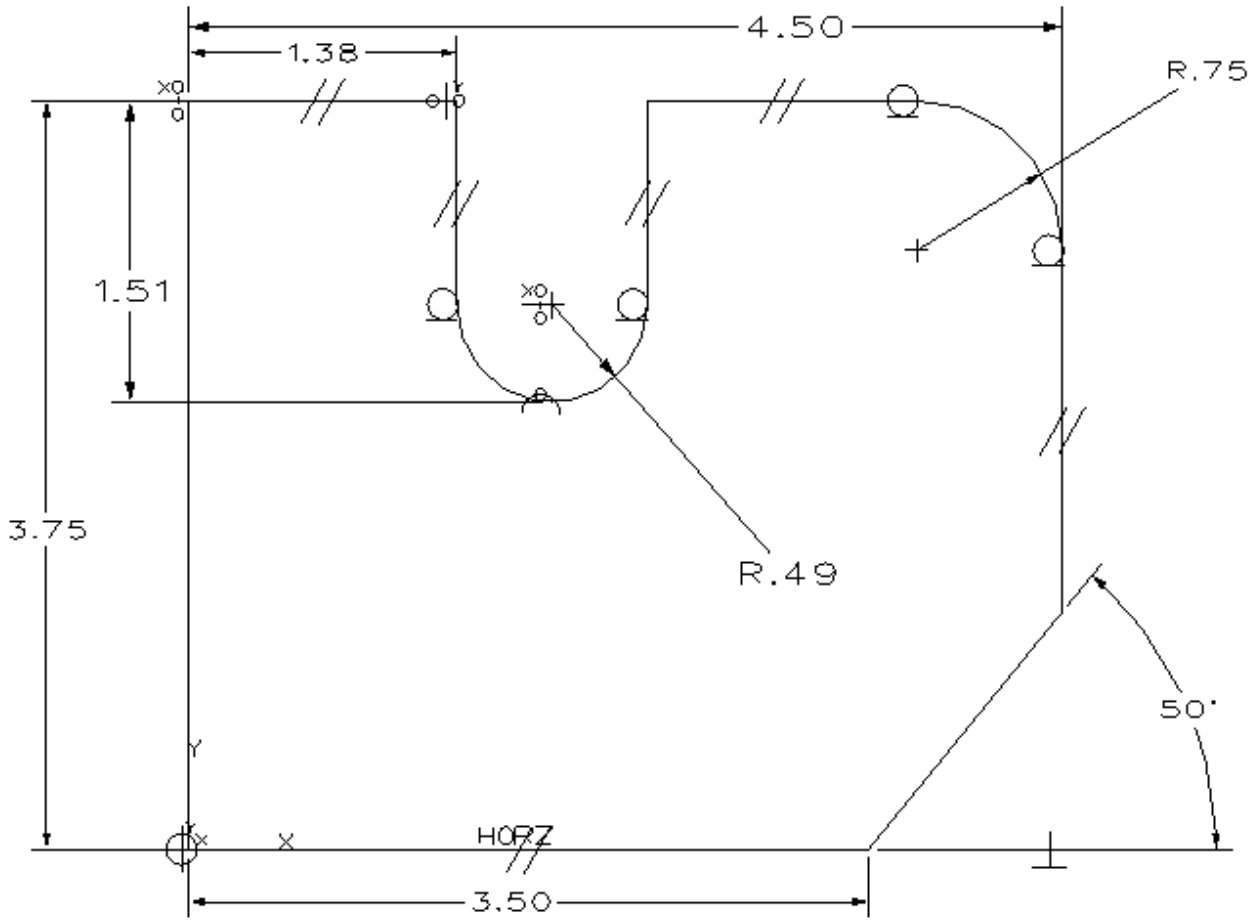
Check the box to "**Delay solve**".

This prevents the sketch from making gross changes based on partial information by considering all new dimension values in a single solve.

Example: Change the 2.5 to 3.5 without the Delay solve box checked and the angled line will flip.

58. Pick the **Solve current sketch manually**  command in the Quick Access toolbar to apply all dimensional changes once you are finished.

    See figure below for results.



The delay mode will stay in effect throughout the active session until it is turned off.

Click on **Solve current sketch automatically** to get out of the Delay mode.

59. Pick **Exit Sketch** when you are finished.

60. Pick **Sketch** from the **ZW3D** root. Name the sketch **Lesson 3d**.

61. We are going to continue focusing on the use of the **<Right-click> Draw** and **<Right-click> Dimension** commands in this task.

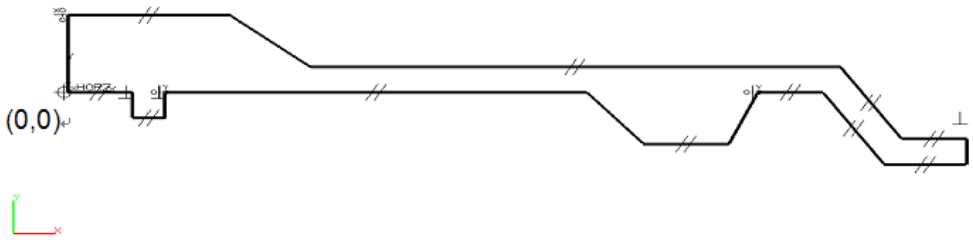
    <Right-click> Draw.

    Draw the profile shown in the figure below. Dynamically drag the origin to the far left.

    HINT: The part can be drawn on a 6.5 grid. Try drawing fairly close to scale.

    Use CTRL-K to display coordinates as you sketch.

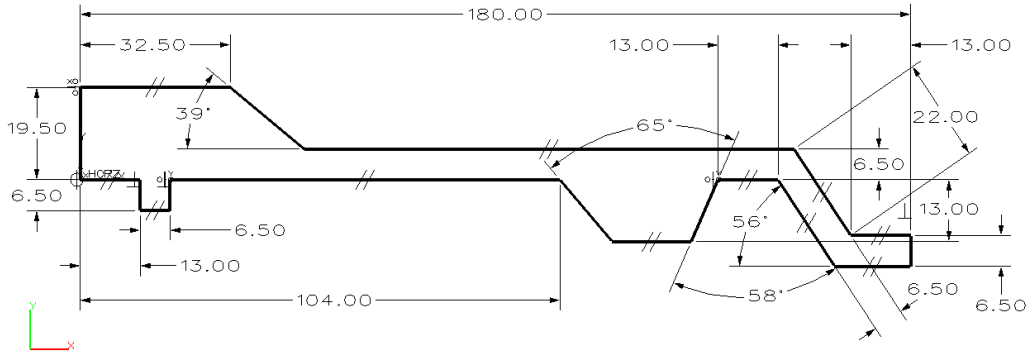
    Remember to **<Right-click> Smart Pick Off** to avoid creating unnecessary implied constraints.



We would like to create as many implied constraints as possible while we draw the initial sketch. However, you might have to delete some of them in order to achieve the dimension values specified in the figure at the end of this lesson. There are both advantages and disadvantages to implied constraints as you may find out in this lesson.

Apply all of the dimensions using <Right-click> Dimension as shown in the following figure. Your dimension values may vary. That's OK. We will adjust them later.

After you have added all of the necessary dimensions, check the status of the sketch.



Pick the **Constraint Status** command.

Leave the form on the screen while you complete the constraining process. Remember GREEN geometry is well defined.

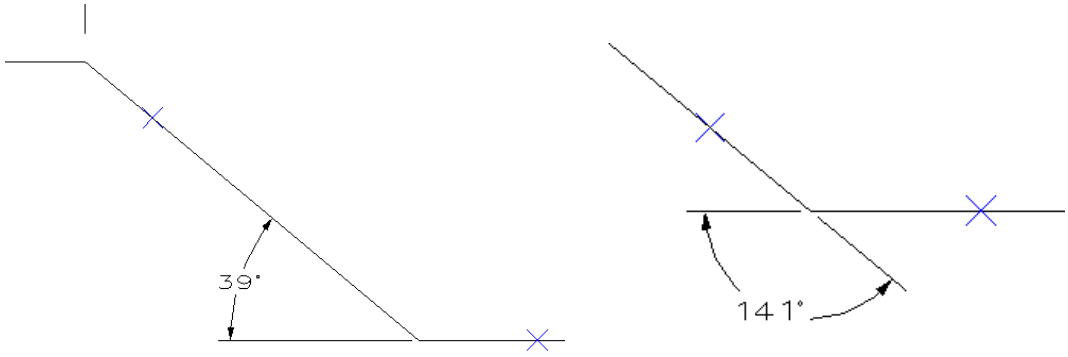
<Right-click> Constrain if necessary.

Pick the lower left corner of the sketch for the base point or Middle-click because this point is (0,0)

Your sketch should be **Well defined** now indicated by the GREEN lines.

HINT 1:

When inserting an angular dimension like the one shown below left, pick the two lines in the approximate positions shown. Move the cursor around and try different locations to see what angular dimension you get. The key here to pick the end of the line where you would want the extension lines to come off.



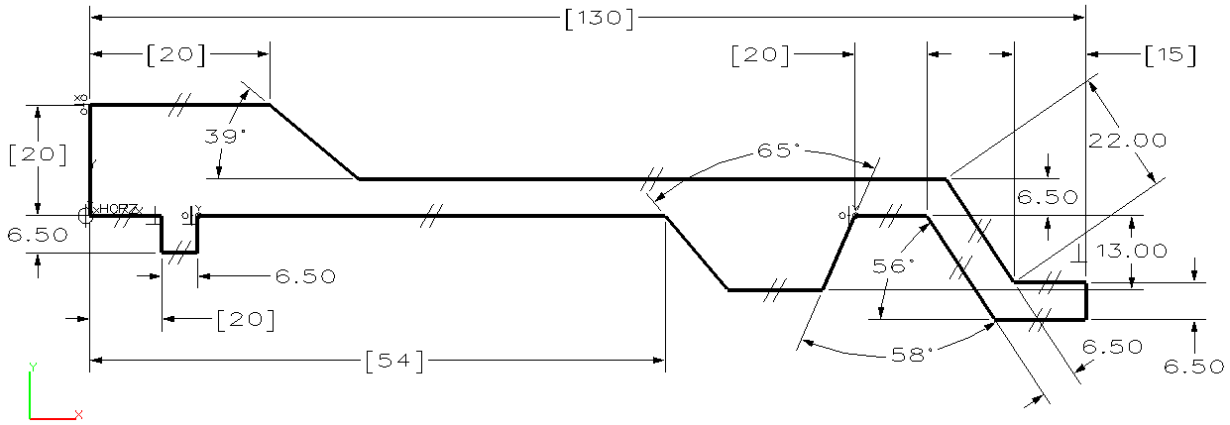
62. Now that we have a well defined sketch, let's change the values of the dimensions to match those shown in the following figure.

    Double click on each dimension or Right-click - **Modify Value** and change all of the dimensions

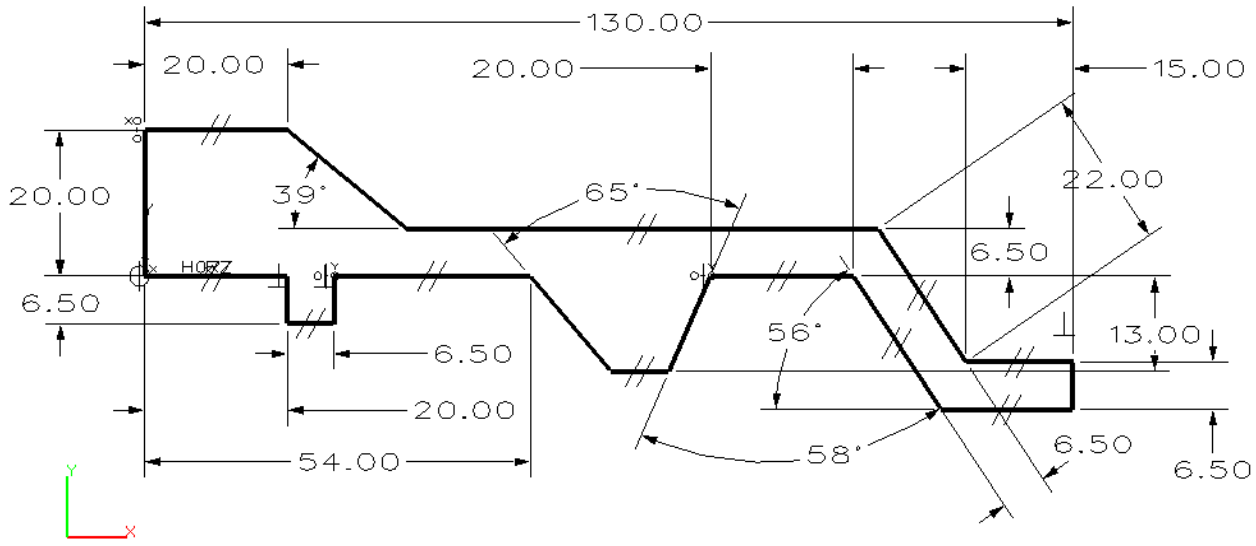
63. Adjust the **Input Dimension Value** form to "**Delay solve**".

    This prevents the sketch from making gross changes based on partial information by considering all new dimension values in a single solve.

    An example would be changing the 180 to 130 before changing the 104 to 54. The geometry will flip on itself. If you try this, UNDO before continuing.



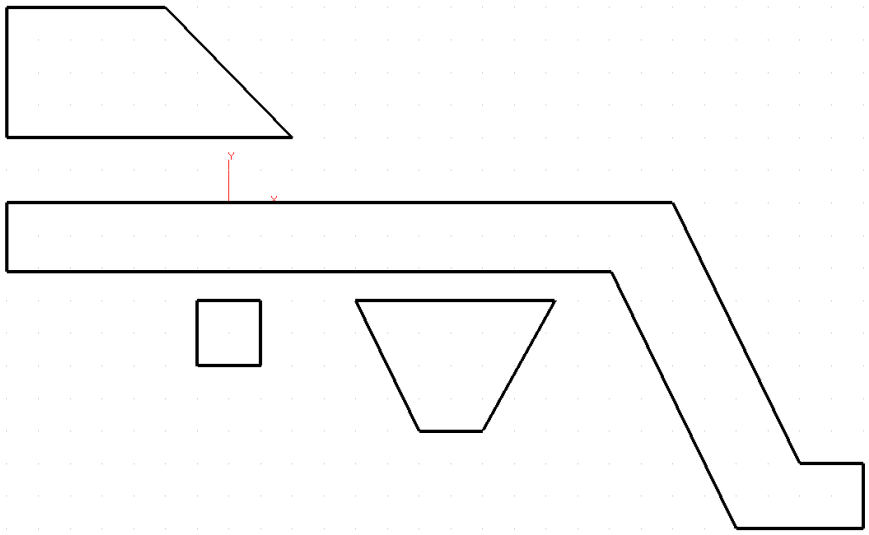
64. Pick the **Solve current sketch manually** command in the **Quick Access** toolbar to apply all dimensional changes once you are finished.



65. Save your work  .

66. Pick **Exit Sketch** when you are finished  .

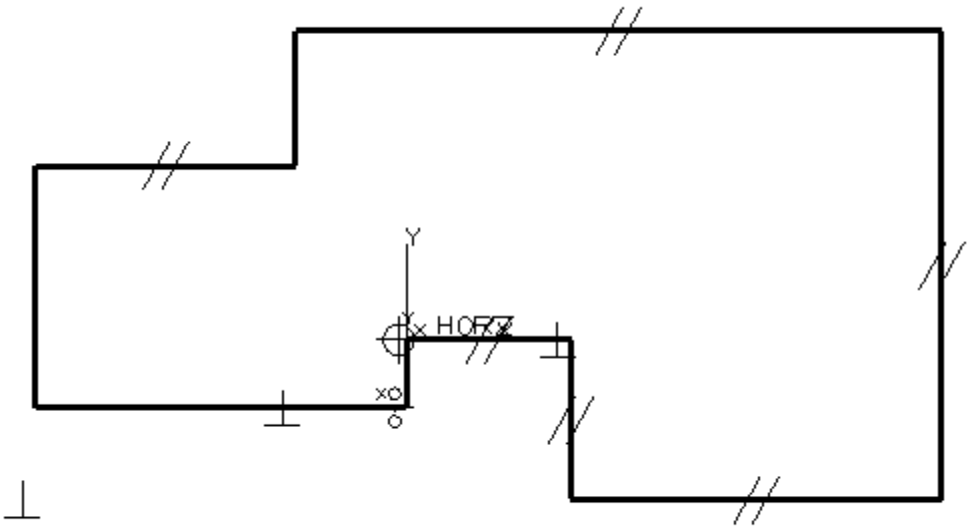
Here is something to keep in mind now that you have successfully created the above sketch. Keep sketches as simple as possible. As you design products or perform other engineering tasks, you make every attempt to keep your tasks as simple as possible. That same concept applies while working inside of ZW3D. This sketch could have been broken down into multiple and more simple sketches as shown below. Follow this guideline whenever possible. Why make your job more difficult than it needs to be?



A new command in ZW3D version 14 is the **DRAG** command  .

1. Pick the **Sketch** icon from the **ZW3D** root Name the sketch **Lesson_3e.**

2. <Right-Click> **DRAW.**
Draw the profile as shown below. Size doesn't matter.



Drag is on the **Edit** pull down menu.

There are essentially two ways it can be used.

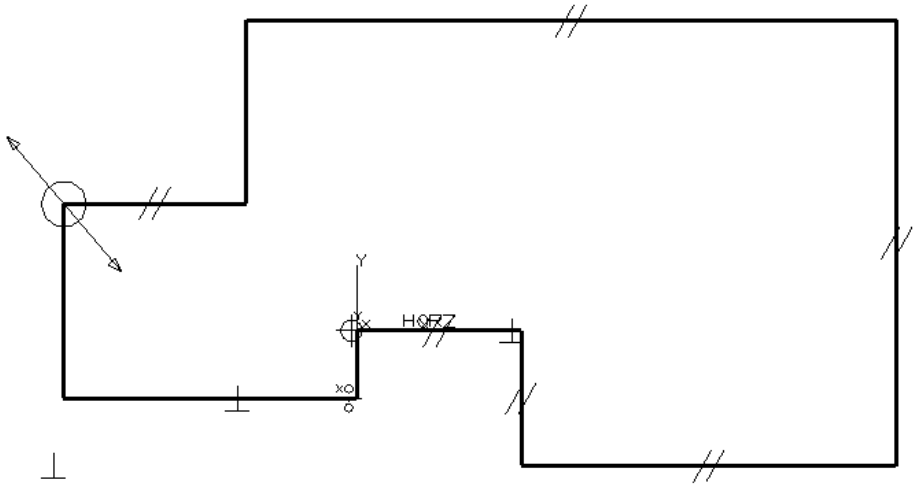
a. Pick on a critical point or corner of the sketch
b. Pick on a non-critical point

We will show you both.

One of the rules for sketching is to try to get the sketch geometrically correct before adding dimensions.

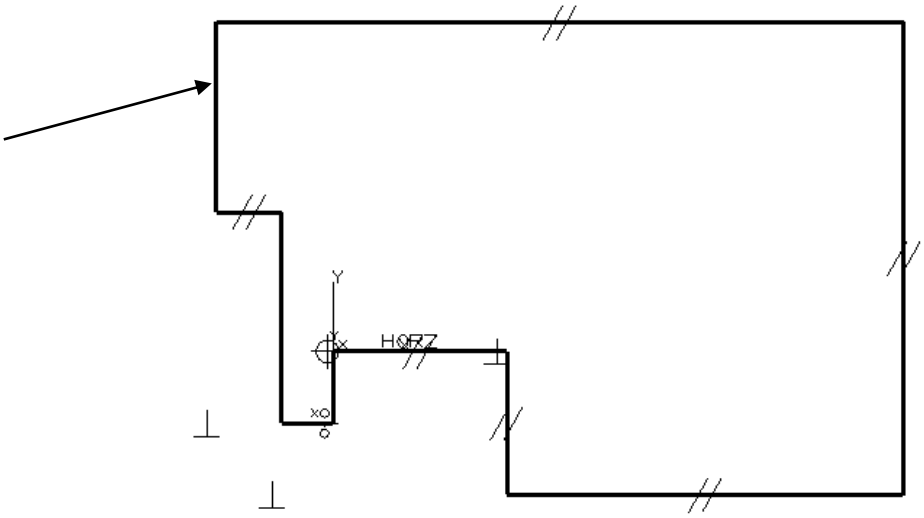
3. Pick the **"Drag"** command.

   a. Pick on the critical point shown below for the **From point**

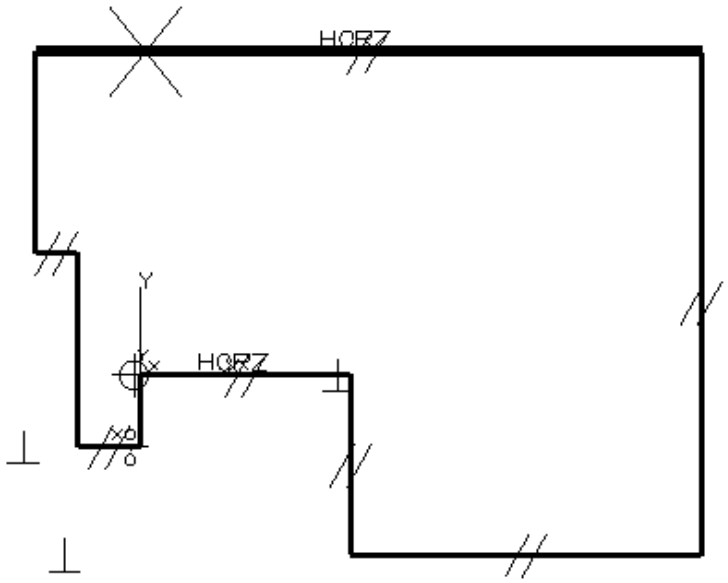


   b. For the **To point** move the cursor around to see the results. Pick the final location as shown below.

Notice how the geometry changes as you drag. The two lines attached to the critical point move while the next piece of geometry trims or extends and the constraints stay intact.

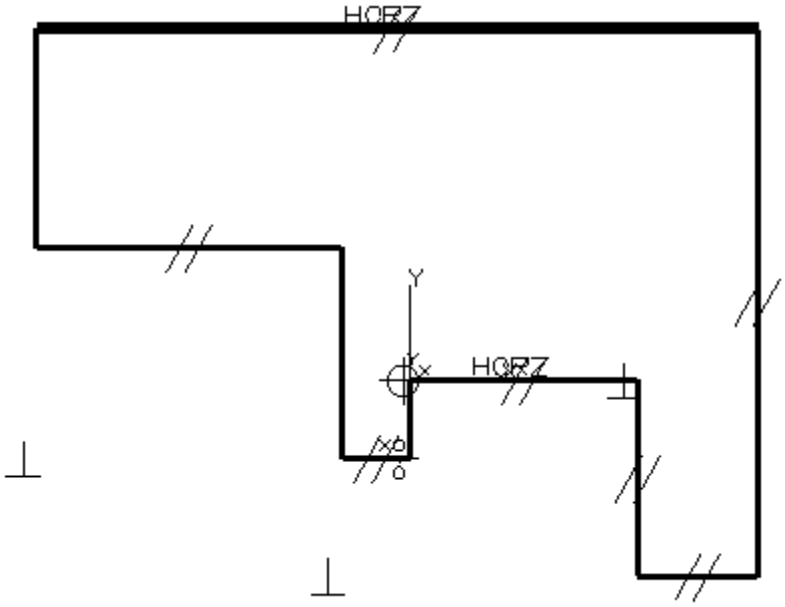


4. Let's DRAG again. Pick the "Drag" command.



This time pick on the top horizontal line indicated above. **Do not pick the critical point.**

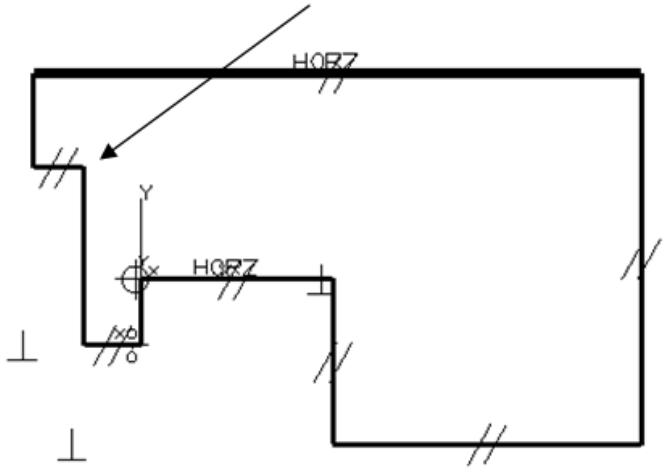
As you drag this geometry around the screen, notice that the horizontal line does not change length. All other constraints are adhered to.



5. Pick the "**Drag**" command.

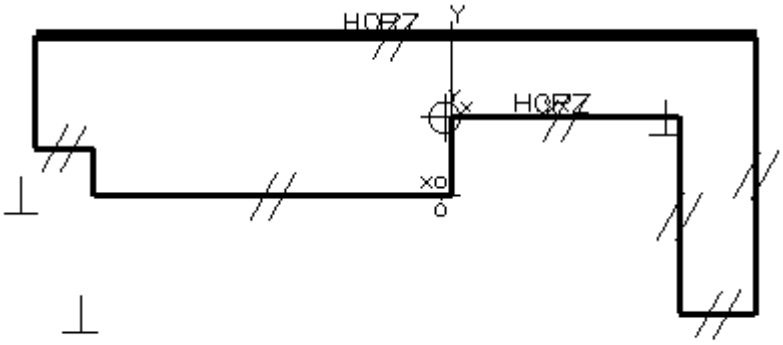
For **Geometry** option, this time pick the top horizontal line as **Additional Drag Geometry**.

Drag the original **critical point** on the model and notice the difference. (**Right click > Critical** to pick the point)



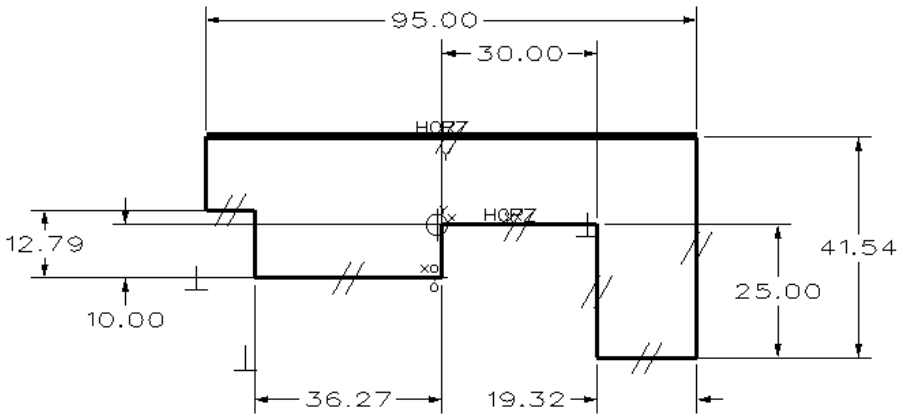
The **additional geometry** acts as if its length is dimensioned. It does not vary during the DRAG.

If the sketch were 'well defined', **DRAG** would have no effect.



Dimension the part. Don't worry about the exact values.

6. Easiest way is the **<Right-click> Auto Constrain** and pick (0,0) as the **Base Point**.



7. Pick the "**Drag**" command.

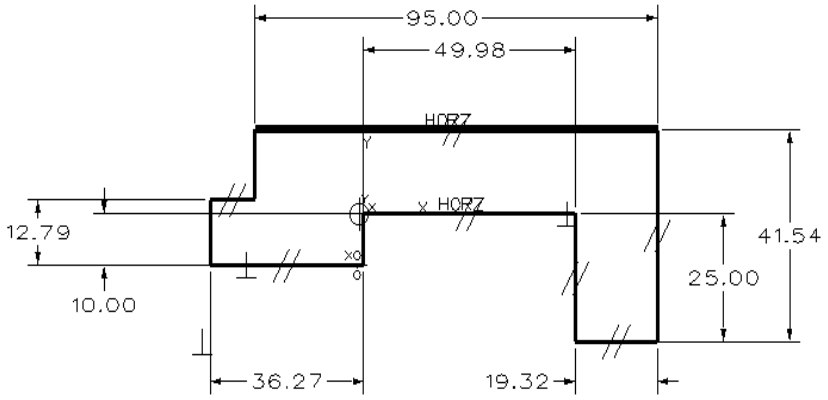
Pick the right critical corner of the 30.00 line.

Can't do it. So let's use the other optional input.

8. Pick the "**Drag**" command.

Pick the **Dimension** option and select the **30.00** dimension.

Drag the same corner to the right as shown below.



The **Dimension** option is like erasing the dimension. It allows that value to vary during the drag.

The **Geometry** option works the opposite way. It locks additional geometry during the drag.

The **Relaxed solution** option will possibly allow a solution to be found when one can't be determined under normal circumstances.

Lesson 4

Applying 2D Constraints

Constraint Tab

Geometric constraints allow designers to build design intent into 2D geometry. These constraints are added to geometry in a sketch to control geometric relationships. Constraining sketch geometry enforces design intent even if the geometry changes.

ZW3D lends itself to conceptual design in that sketches do not need to be well defined. An engineer can generate a conceptual model without constraints, and apply the constraints at a later time.

The pull down menu and Tab appearing above contain all of the commands to add geometric constraints

Geometric constraints can be added manually using the individual commands shown in the menus below. We can automatically add constraints using the **Auto Constrain**  command. We can also auto constrain a sketch by setting the **Sketch Settings Form** to **Auto Constrain** before creating sketch geometry. (See **Edit > Preferences**)

There are several commands that can assist you in clearing up the appearance of sketches during the constraining process. Sometimes a sketch can become so filled with information, they become difficult to visualize. To help with this problem use any of the following commands:

- Use the ERASE Entities  command <Left-click>Pick, <Middle-click> Erase to erase geometric constraints.
- Use **Toggle Constraints On/Off** and **Toggle Dimensions On/Off** . See figure below for reference.

Refer to the HELP Manual

Refer to the **HELP Manual** for information about each command on the Constraint Tab. The table below lists each command, icon, constraint symbol.

Icon	Command Name	Symbol
	Auto Constrain	-
	Add Constrain	-
	Constraint (Anchor)	
	Constraint (Point Horizontal)	

	Constraint (Point Vertical)	
	Constraint (Point to Midpoint)	
	Constraint (Horizontal)	
	Constraint (Vertical)	
	Constraint (Point to Line)	
	Base Point Command Option	
	Constraint (Point to Curve)	
	Constraint (Point to Intersection)	
	Constraint (Parallel)	
	Constraint (Perpendicular)	
	Constraint (Tangent)	
	Constraint (Point to Center)	
	Constraint (Symmetrical)	
	Constraint (Equal Length)	
	Constraint Status	-

Lesson 4a

Working with ZW3D Constraints Basics

Constrain Lines Horizontal and Vertical

Use these commands to create a horizontal or vertical constraint, and attach it to a line(s) so that it remains horizontal or vertical. The constrained line(s) will remain horizontal or vertical until the constraint is deleted.

We have seen that when you sketch a rectangle, the Horizontal constraint is automatically inserted. But we also know that changes can and will happen that affect our design intent. Let's sketch a shape and put our own constraints on to demonstrate this.

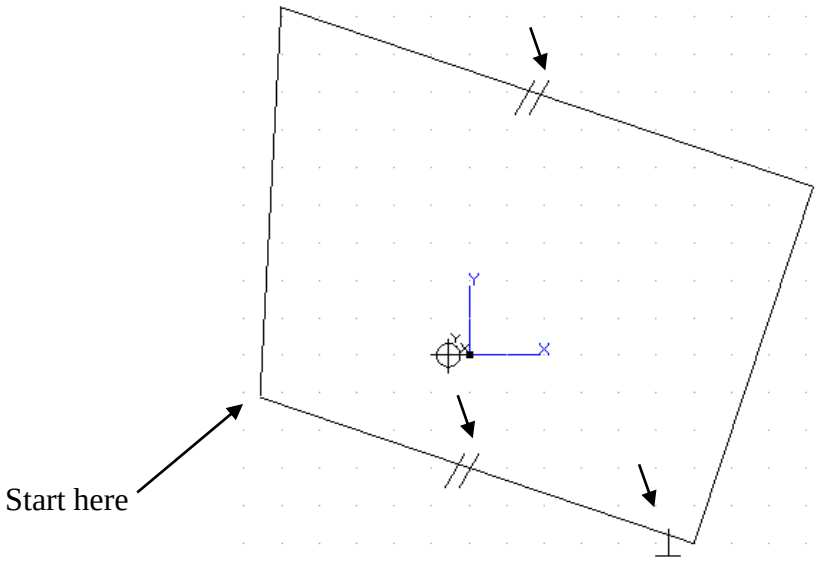
1. Pick the **Sketch** icon from the **ZW3D** root. Name the sketch **Lesson_4a**

2. **<Right-click> Draw.**

3. Go to the **Edit > Preferences** pull down menu to verify the units are **MM** and the grid is **5**.

4. Create the sketch in the figure below. As you create the sketch make sure to imply a parallel constraint between the two nearly horizontal lines and a perpendicular constraint between the right line and the bottom line.

   See figure below for reference. (Start in the lower left corner and go counter clockwise.)

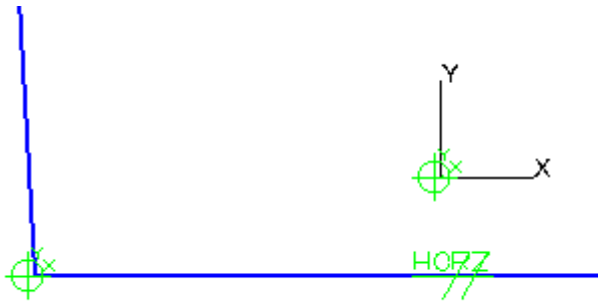


5. Open the **Constraints** Tab.

6. Add a **horizontal** constraint to the first line.

   The other line should go Horizontal also since we created them as parallel.

7. Add an **Anchor Constraint** to the lower left corner of the sketch. Set the coordinate option to both.



   Your sketch should appear similar to the figure above.

8. Pick the **Constraint Status** command to verify the status of your sketch. Your results should match those shown below.

3 DOF left

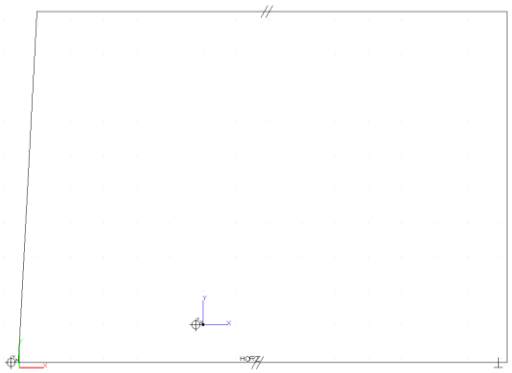
Geometry

4 Under constrained

1 Well defined

Constraints

5 Solved



Leave this form on the screen as you complete this task. The results in it should match ours.

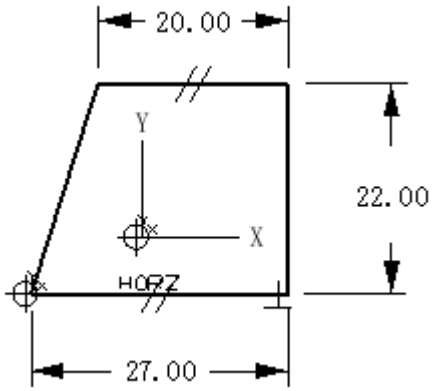
We are missing 3 Dimensions or Constraints. Your design intent will determine what you add. DOF = Degrees of Freedom

9. **<Right-click> Dimension** and dimension the sketch as shown below.

10. Pick the **Solve Current Sketch** command to verify the dimensional constraints we just added. See the message window. Your dimension values may vary. That's OK.

11. **Double click on dimension** and adjust its value to match the figure below. Your **Constraint Status** form should appear similar to ours.

    All constrained geometry is well defined
    Geometry
    5 Well defined
    Dimensions
    3 Solved
    Constraints
    5 Solved



12. Do not erase your sketch. We are going to use it for the next several steps.

Constrain Point to Curve

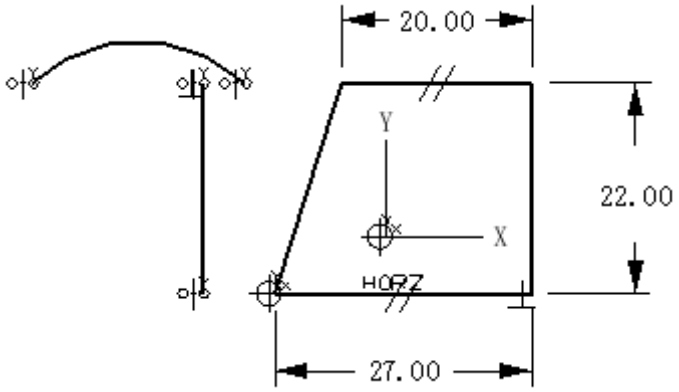
Use this command to create a **points on curve constraint**, and attach it to a point(s) so that it remains on a base curve. The base curve can be an arc, circle, or curve. If the base curve is modified, the constrained point(s) will remain on it until the constraint is deleted

13. <Right-click> **Draw.**
    Sketch the vertical line to the left our previous sketch as shown in the figure below. Start the line about 8 mm from the lower left corner. It should be the same height.

    <Right-click> **Draw.**
    Sketch the arc above the line you just created. Refer to the suggestion below if required. See figure below for reference.

    Here are a few suggestions for drawing the arc, if you need them:

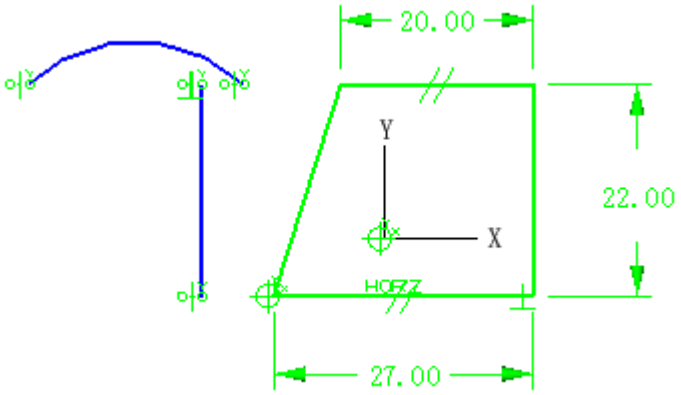
    - **Pick the right side of the arc so it lines up with the top of the vertical line.**
    - **Hold the <ALT> key down and pick the left side of the arc so its endpoint also lines up with the top of the vertical line, but far enough to the left that the center is obviously to the left of the vertical line.**
    - **Release the <ALT> key.**
    - **Move your cursor up and pick the 3rd point of the arc. Don't pick up any unwanted constraints with this pick.**



14. Pick the **Constraint Status** command to verify the status of your sketch. Your numbers should match those shown below.

    Leave this form on the screen as you complete this task. Your results should match ours.

    4 DOF left / Geometry 2 Under constrained 5 Well defined / Dimensions 3 Solved / Constraints 10 Solved.



15. Let's fully constrain the new geometry as shown in the figure below.

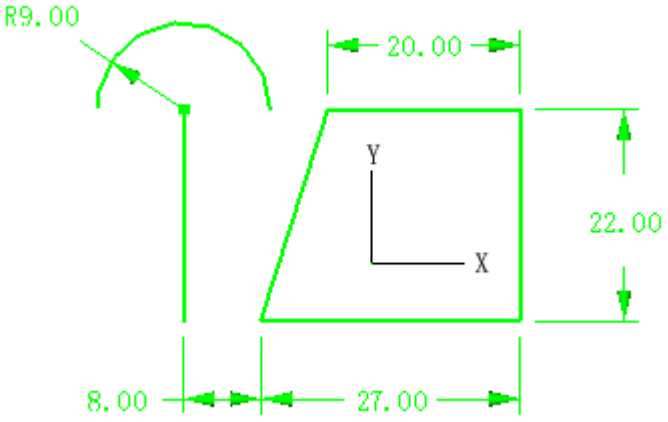
    <Right-click> Dimension and add the 8mm horizontal dimension. Use the **From** and **To** options. Your dimension value may vary. That's OK for now.

    <Right-click> Dimension and add the 9mm radial dimension.

16. Add a **Concentric Constraint** between the arc center and the top endpoint of the vertical line. Pick the arc (center) as the base and the line as the point to constrain concentric to base.

17. Again your dimension values may vary. That's OK for now.

18. Double click on the required dimensions and change their values to match the figure below.



Well defined Sketch
Constraints have been moved for clarity

19. Erase the 8mm dimension

20. Pick the **Point to curve constraint** command to connect the right end of the arc to the upper left corner as shown in the figure below.

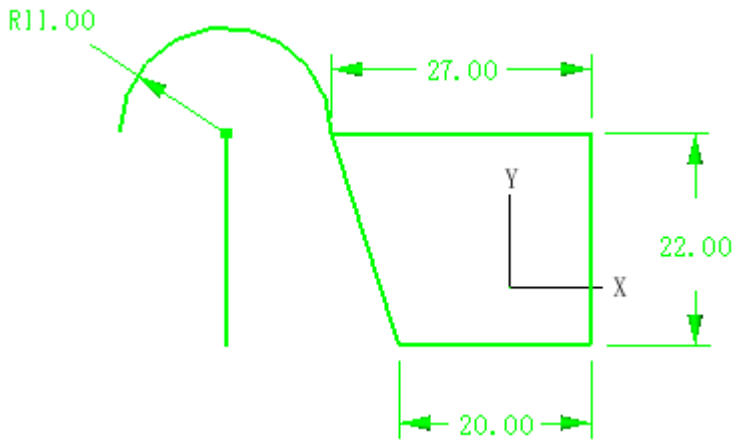
Pick the arc as the ‘curve to constrain points to’.
Pick the corner of the right geometry as the ‘points to constrain on curve.’

21. The sketch will update automatically.

22. Change the **20 to 27** .

23. Change the **27 to 20**.

24. Change the **R 9.00 (Roll the wheel)** to **R 11** as a test.



Well defined Sketch
Constraints have been moved for clarity

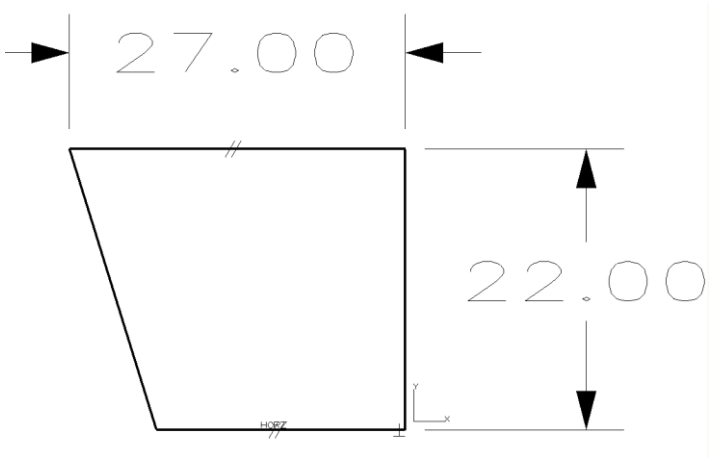
Constrain Point to Line


Use this command to create a point to line constraint and attach it to a point(s) so that it remains co-linear with a base line. If the base line is modified, the constrained point(s) will remain co-linear with it until the constraint is deleted

1. Let's make a couple changes to our existing sketch.

2. Pick the arc, the left vertical line and the 20mm dimension.

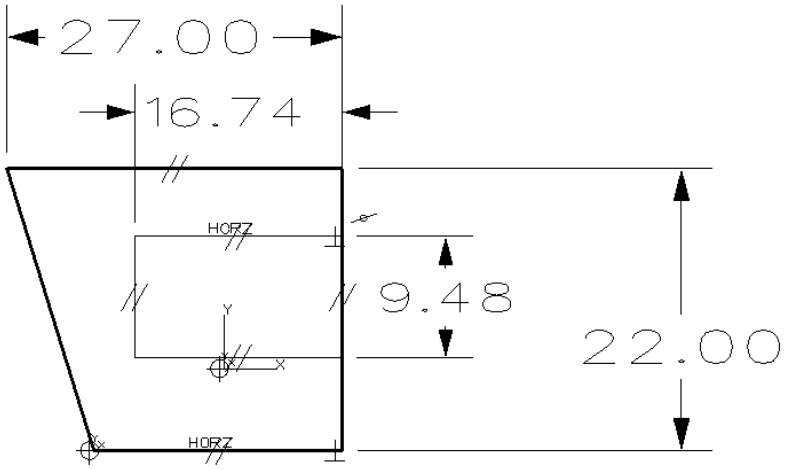
   <Right-click> Erase.



3. Pick the **Rectangle (corner)** command  .

4. **<Right-click> Default** so **ZW3D** does not snap to an endpoint.

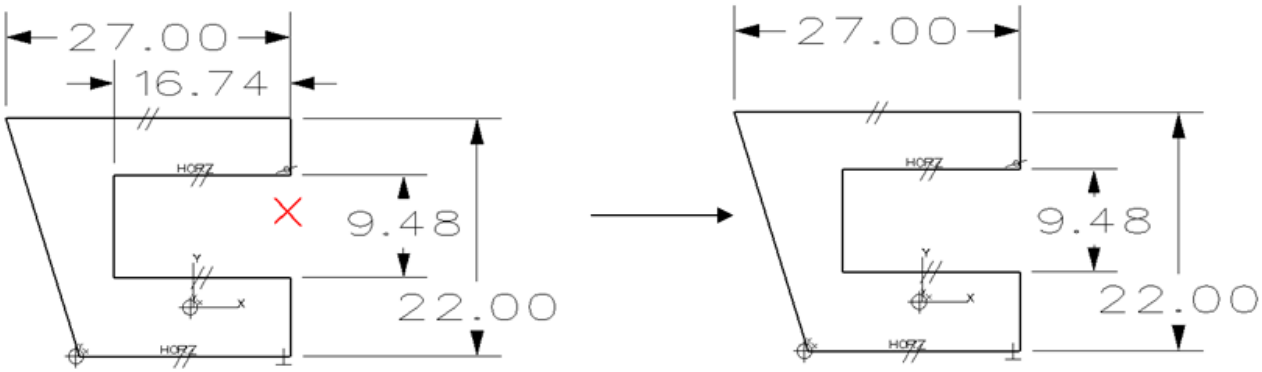
Add the internal shape shown in the figure below. Your dimensions may vary. That's OK.



Notice the constraint that has been automatically attached in the above figure. This constraint will maintain a collinear relationship between the 2 vertical lines.

5. <Right-click> **Trim.**

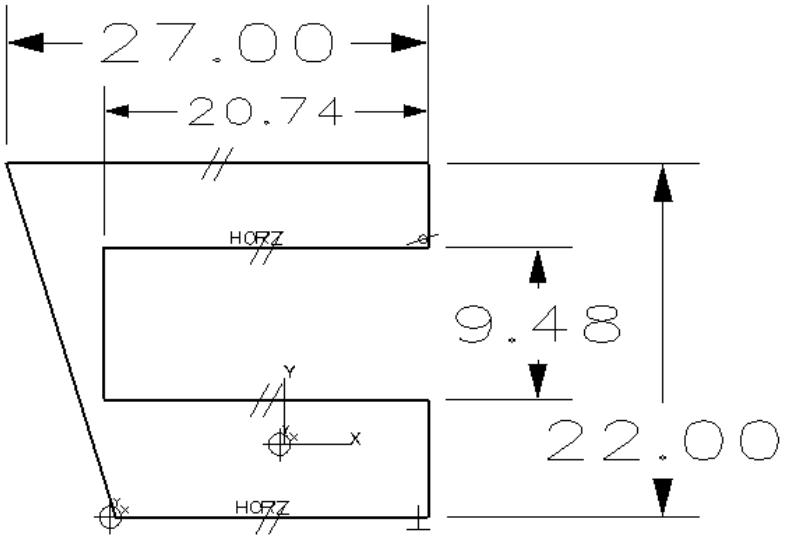
   Pick the portion of the line(s) to be removed as shown in the figure below. Remember there are two lines in this area due to the internal shape we created.



   Notice we lost the inside horizontal dimension. Add it back in. Generally speaking, trimming is not what you want to use because dimension and or constraints that are attached to those entities will be lost. We will talk about using a TRACE PROFILE later.

6. Change the **16.74 mm** dimension to **20.74 mm**.

7. The right side of the rectangle will still be attached after modifying the dimension value because of the Point to Line constraint.



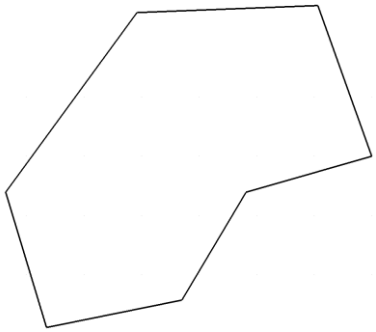
8. Window the above sketch and **<Right-click> Erase.**

Constrain Lines Parallel

Use this command to create a parallel constraint and attach it to a line(s) so that it remains parallel to a base line. If the base line is modified, the constrained line(s) will remain parallel until the constraint is deleted

1. Erase the sketch you made in the previous example.

2. **<Right-click> Draw** and create the sketch similar to the figure below. Try not to pick up any constraints. If you do just erase them. (**Right-click – Erase**)

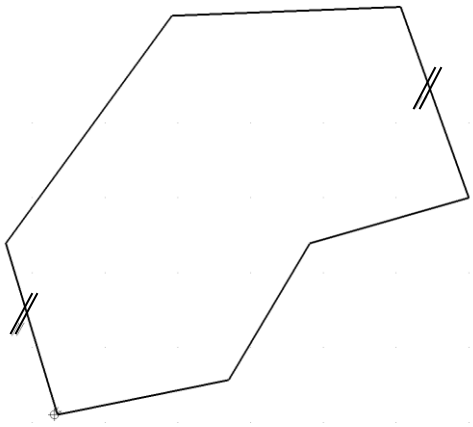


3. Add the **Parallel** constraint and the **Anchor** constraint in the lower left corner.

See below.

The base line for the parallel constraint can be either line you want. In this situation it does not matter. Be aware however that the base line can make a difference in how your sketch solves. The base is the line that remains in the current position.

If you didn't see the part update, hit **Undo** then **Redo.**

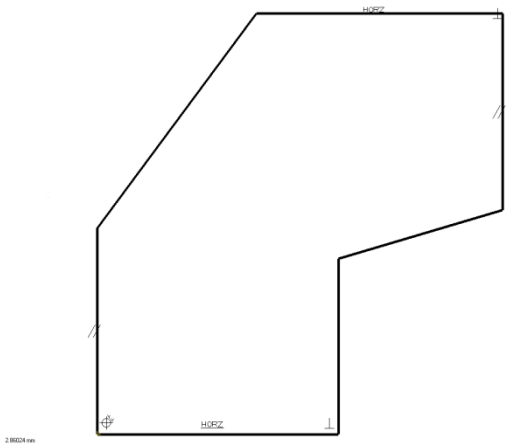


Constrain Lines Perpendicular

Use this command to create a perpendicular constraint and attach it to a line(s) so that it remains perpendicular to a base line. If the base line is modified, the constrained line(s) will remain perpendicular until the constraint is deleted.

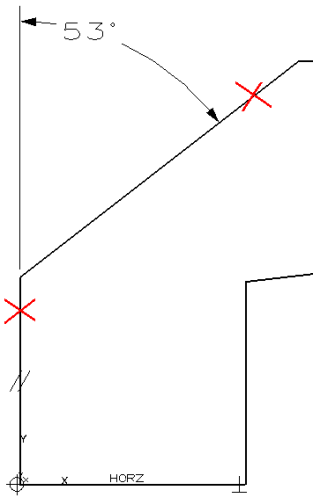
4. Let's continue by adding 2 **Horizontal** **Constraints** as shown in the figure below.

5. Add 2 **Perpendicular** **Constraints** as shown in the figure below. When adding the 2 perpendicular constraints, pick the horizontal lines as the base lines to achieve the results we require. The base lines will remain in their current position.



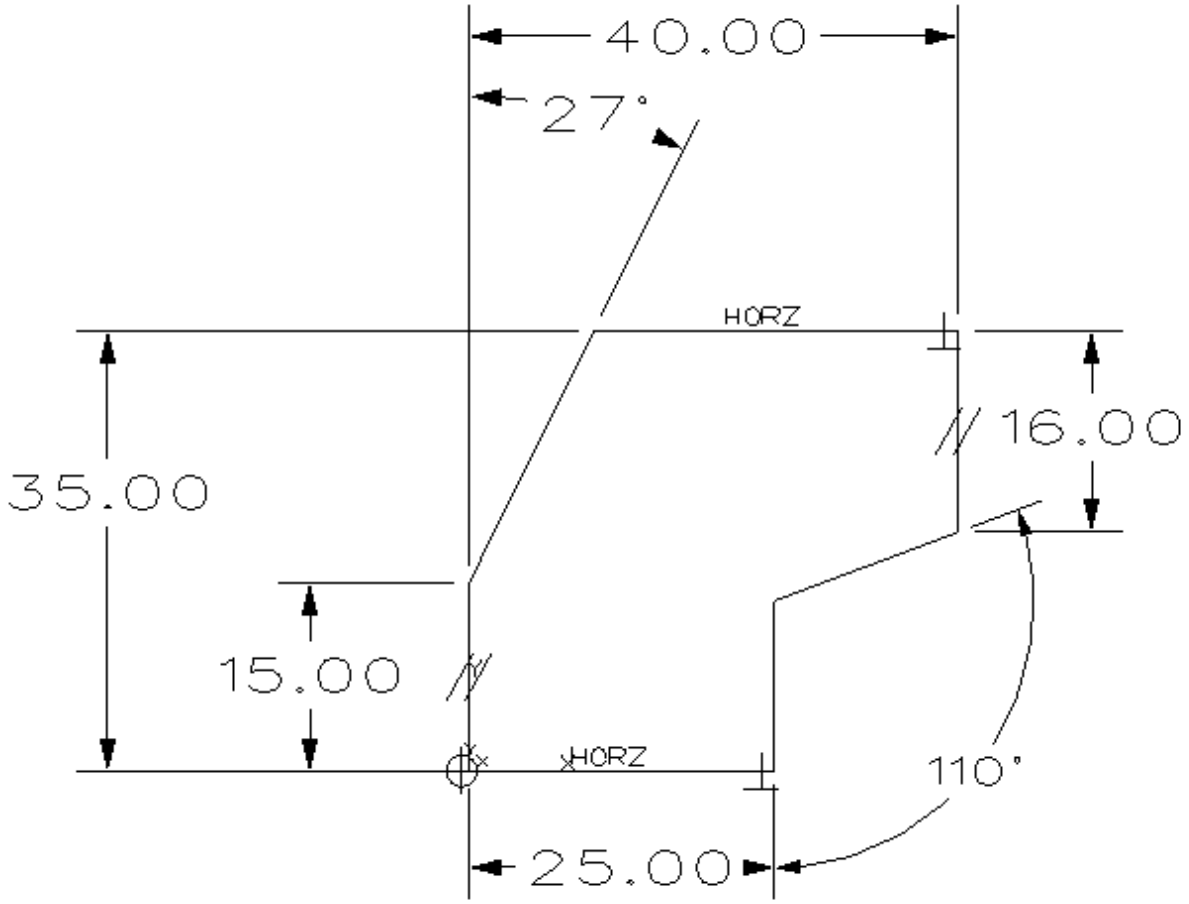
6. **<Right-click> Dimension** and add the dimensions as shown in the figure below.

   Even the angular dimensions can be inserted using the Quick Dimension Utility. For example: To get the 53 degree angle below, <Right-click> Dimension. Pick the vertical line near the top but not the critical end point. Slide the cursor along the angled line until you see the dimension you want. Now pick the text location. Change this to 27. Try the 110 degree angle by yourself.



7. Double click on each dimension and change its value to match the figure below. Pick **Delay solve** of sketch in the **Input Dimension Value** form.

8. Pick the **Solve Current Sketch manually** command to update the sketch. All the dimensions will be updated simultaneously.



The 15.00, 16.00 and the 25.00 dimensions can be done with a single pick.

The 40.00 and 35.00 you will need to pick the two end points then position the text.

Constrain Entities Tangent

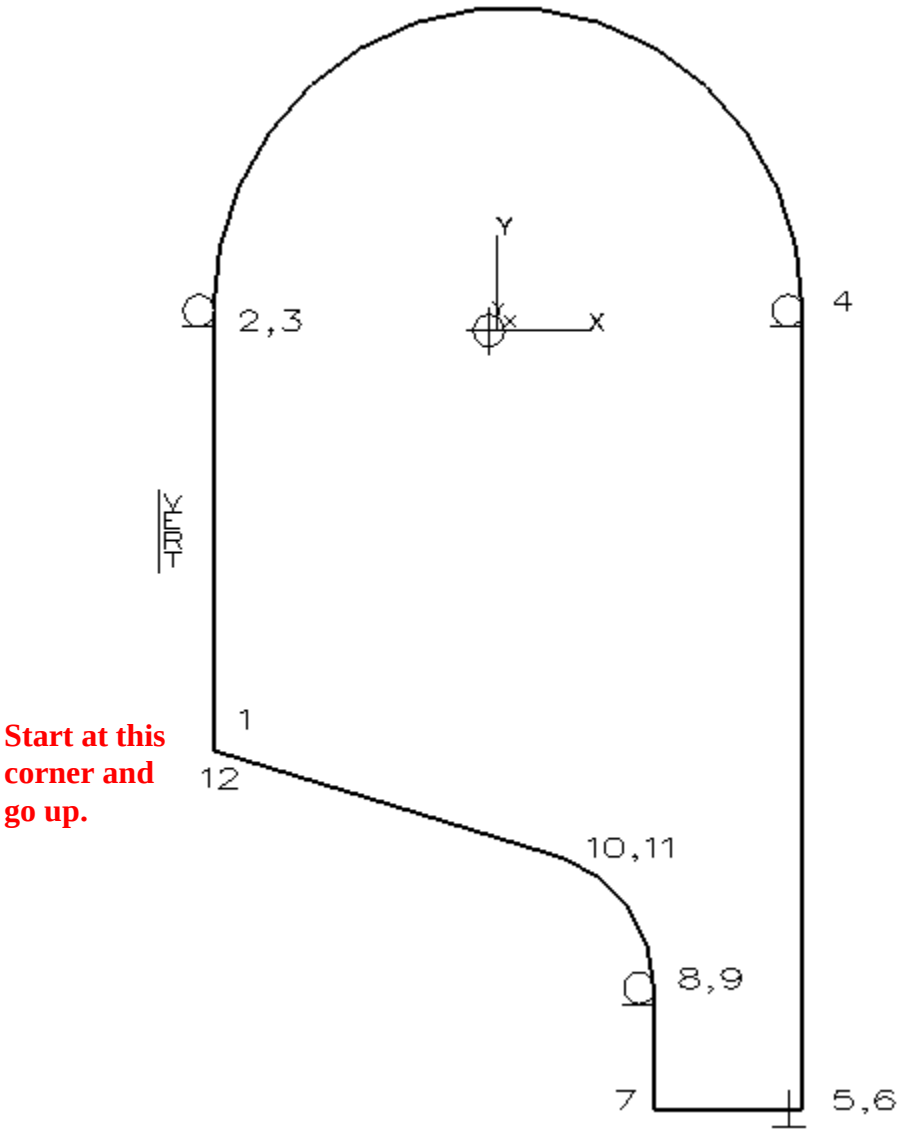
Use this command to create a tangent constraint and attach it to two lines, arcs, circles, or curves so that they remain tangent. If either is modified, the other will remain tangent to it until the constraint is deleted.

1. Erase the sketch you made in the previous example.

2. **<Right-click> Draw.**

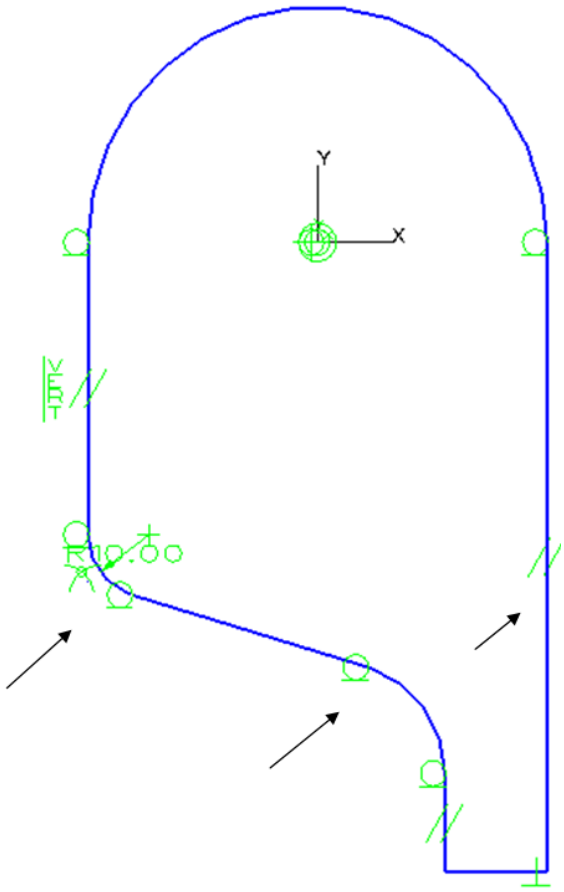
   Create the sketch similar to the figure below.

   Try to position the center of the arc close, not exactly at (0,0).



3. Add an **Concentric Constraint** between the center of the large arc and (0,0).
4. Add a **Parallel Constraint** between the three vertical lines.
5. Add any missing tangent constraints between lines and arcs as shown below.
6. Add a fillet at the start corner.

Your sketch should appear similar to the following figure.



7. Turn on the **Inquire status of constraints.**

The constraint status should be:
6 DOF left / Geometry 9 Under constrained 1 Well defined / Dimensions 1 Solved / Constraints 13 Solved.

8. < Right-click> Dimension.

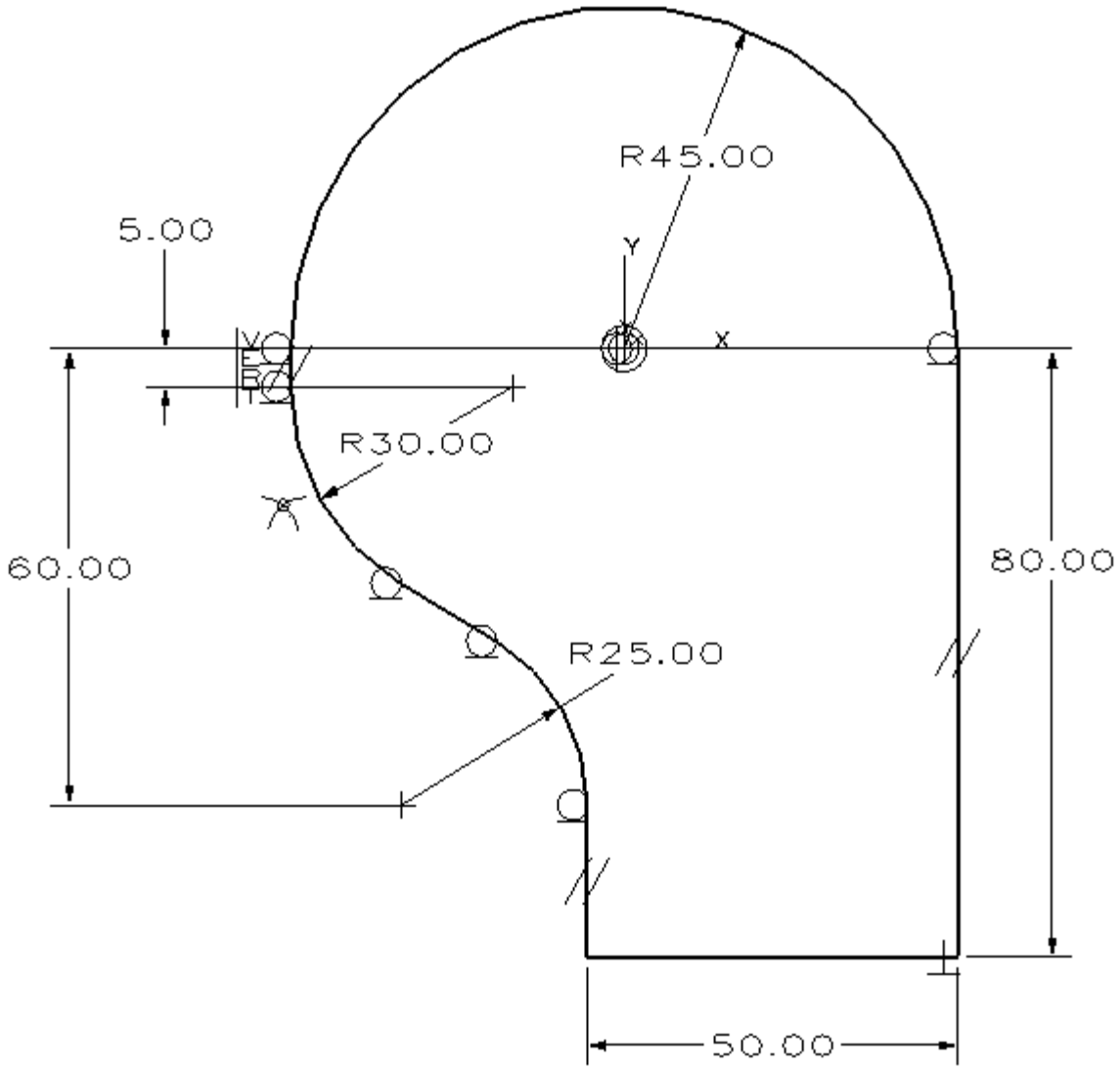
Add the six dimensions as shown in the following figure.

You will need to dimension the entire sketch and then have all the dimensions solve or update simultaneously. There are a couple of ways to do this.
Before you start dimensioning, pick the **Solve current sketch manually** button.

The other way is to check the **Delay solve** option when changing the first dimension.
The need for this will depend on how close to scale your sketch was created.
Try editing without this option until the part changes shape incorrectly.

9. Double-click on each dimension and change the value to match the figure below.

10. When all the dimension have been changed, **Solve current sketch manually**. To switch out of the delay mode pick, choose **Solve current sketch automatically.**



Constrain Point to Center

 Use this command to create a concentric constraint and attach it to a point(s) so that it remains concentric to a base arc or circle. If the base arc or circle is modified, the constrained point(s) will remain concentric to it until the constraint is deleted.

11. **<Right-click> Draw** and sketch a circle as shown in the figure below left.

    Avoid creating any constraints.

    Before you pick the center of the circle, hold the **<ALT>** key down as indicated in the command prompt so the **DRAW** command will create a circle. Remember, DRAW can create lines, arcs, circles and splines, all controlled by pressing the **<ALT>** key at the proper command prompt.

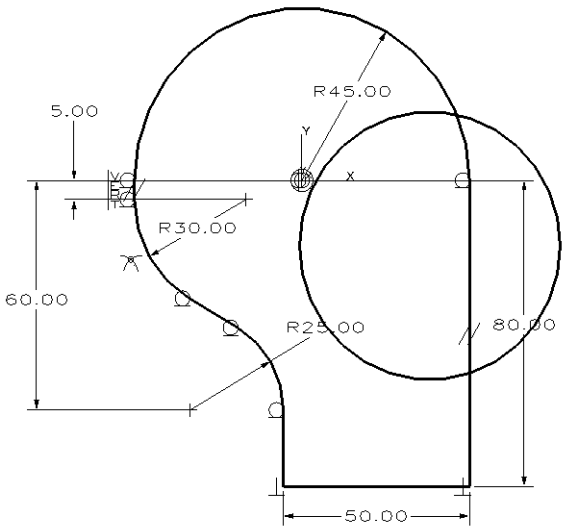
    Also if you pick the center point on the center of the large arc, **ZW3D** will add a concentric constraint automatically. If you do not pick at the center of the large arc, **ZW3D** will simply create the circle at the location you pick.

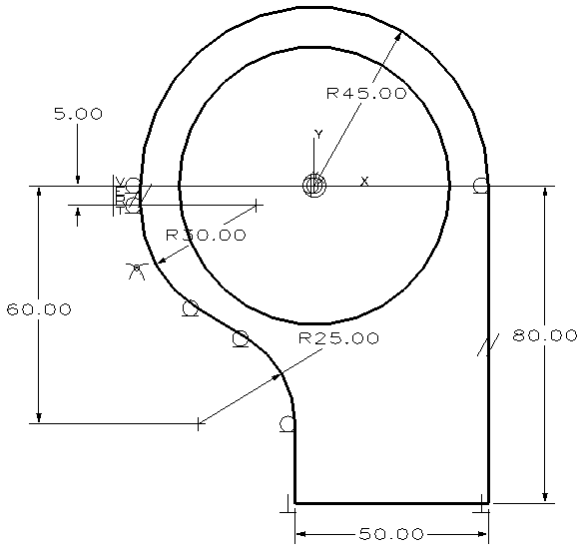
12. Since you placed the circle so it is not automatically concentric to the large arc, pick the **Point to Center (Concentric)** command and add a  constraint between the arc and circle.

    Pick the large arc as the base and the circle we just created as the point to constrain as concentric.

    Remember the large arc already has a fixed X and Y constraint attached to it and will not move. See how the smaller circle snaps to the center of the large arc.

Add a diameter dimension if you want.





Constrain Symmetrical

Use this command to create a symmetrical constraint and attach it to pairs of points to locate them symmetrically about a base line. The constrained pairs of points will remain symmetrical about the base line until the constraint is deleted. Symmetry constraints are useful in eliminating the need for a lot of equations.

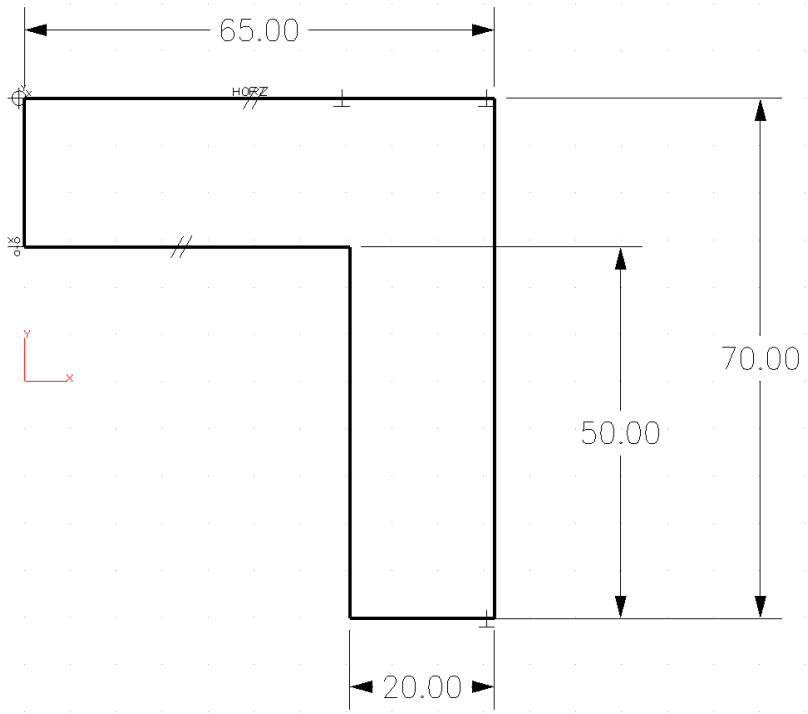
1. Erase the sketch you made in the previous example.

2. **<Right-click> Draw** and create the sketch similar to the figure below.

   The automatic/implied constraints may vary on your sketch. That's OK.

3. **<Right-click> Auto Constrain** to fully constrain the sketch automatically.

   Pick the upper left corner of your sketch as the base point.



4. Use the **Constraint Status** command to verify you have a **well defined** sketch.

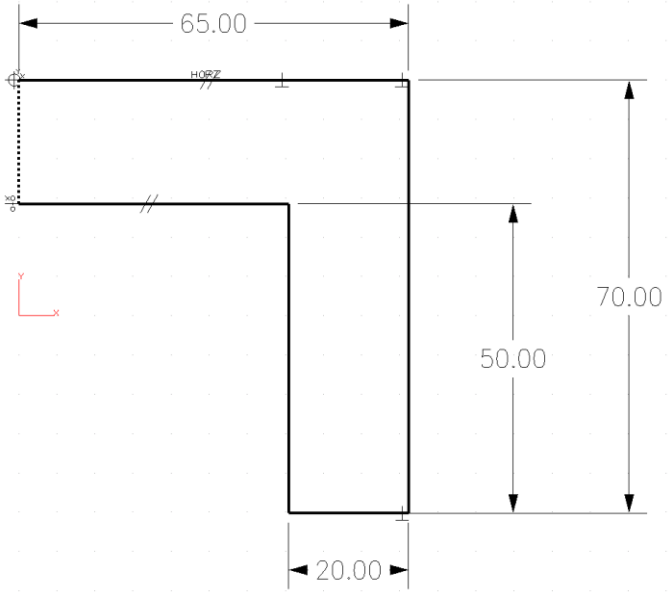
   Remember, the sketch geometry will turn GREEN if it is well defined. Leave this form on the screen as you complete this lesson.

5. The next thing we need to do is mirror the sketch about the short vertical line on the left side of our sketch. But first we need to change that line to construction geometry so it can be used for mirroring but not become part of the solid feature.

Highlight the short vertical line on the left side of our sketch.

<Right-click> Toggle Type  **.**

The line is changed to a construction line. It can be used as a mirror axis for our sketch but will not be used to define our feature.



6. Pick the **MIRROR** command  from the **Sketch** Tab > Basic Editing.

   Use a window to pick all the geometry in the sketch except the construction line.

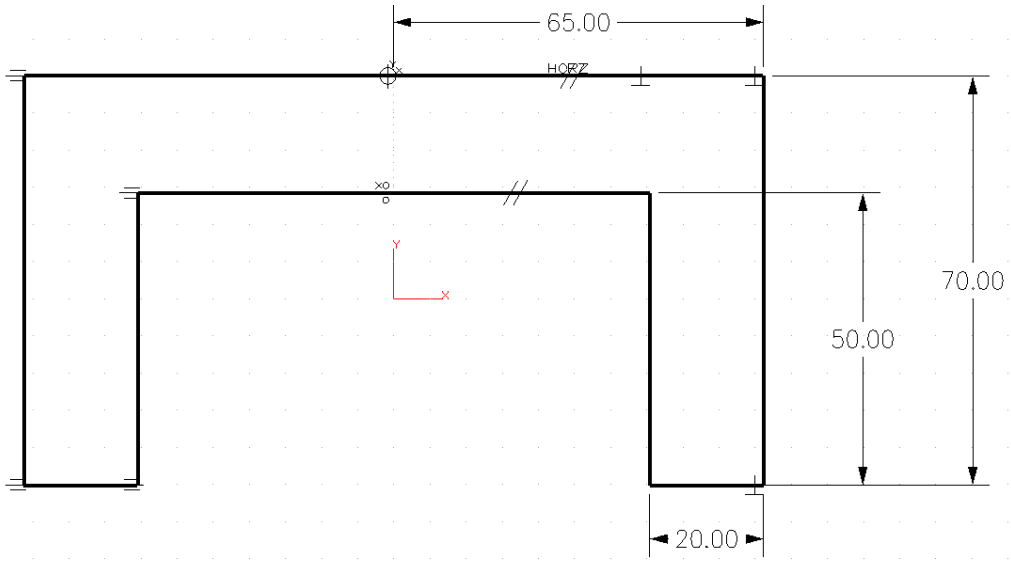
   ZW3D filters out dimensions and geometric constraints so they do not get mirrored.

   If you pick the construction line, we will not be able to use it as a symmetry line to mirror about.

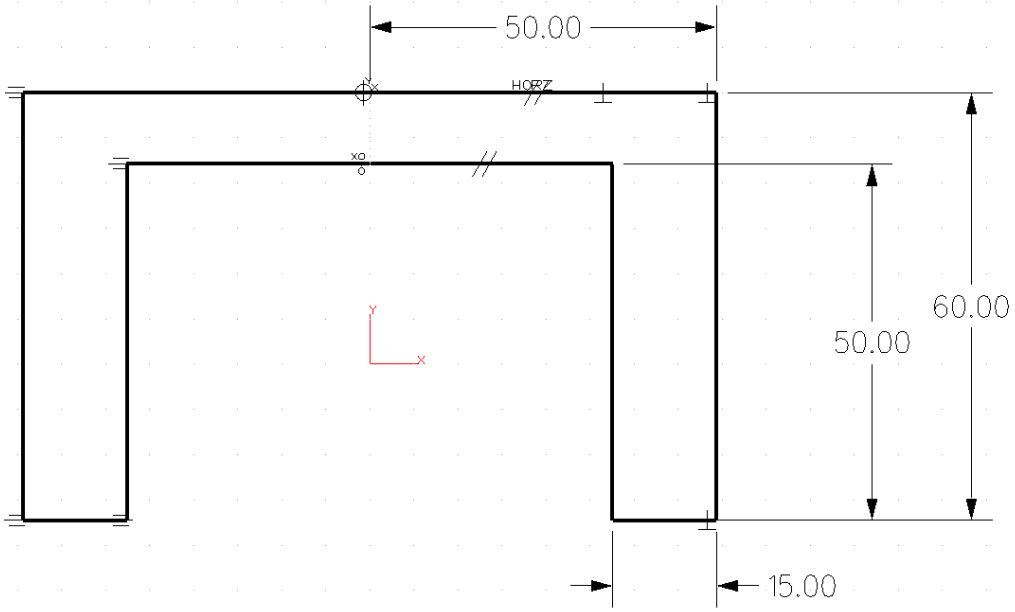
   Pick the construction line as the symmetry line.

   Notice the only constraints added are  **Symmetry Constraints**. The sketch is still well defined.

   Another picking technique that I like is to chain pick or <shift-pick> all 6 lines, then <ctrl-pick> the line of symmetry. Watch the text input line to see the number of entities picked.



7. Verify the design intent by changing the dimensions as shown in the figure below. Notice how the sketch updates symmetrically on both sides.



Constrain Equal Distance

Use this command to create an equal distance constraint and attach it to an entity so that it remains an equal distance from another entity. For example, the length of one line may be constrained to equal the length of another. This constraint may also be used with radii of circles. Two circles may be constrained to have the same radius (this is done automatically in the sketcher when multiple circles are created with a single radii), or a line may be constrained to have a length equal to the radius of a circle.

8. **<Right-click> Fillet** and add an 8mm fillet to the upper right and left corners of the sketch. Both fillets get dimensioned automatically.

9. Highlight the radial dimension on the left fillet.

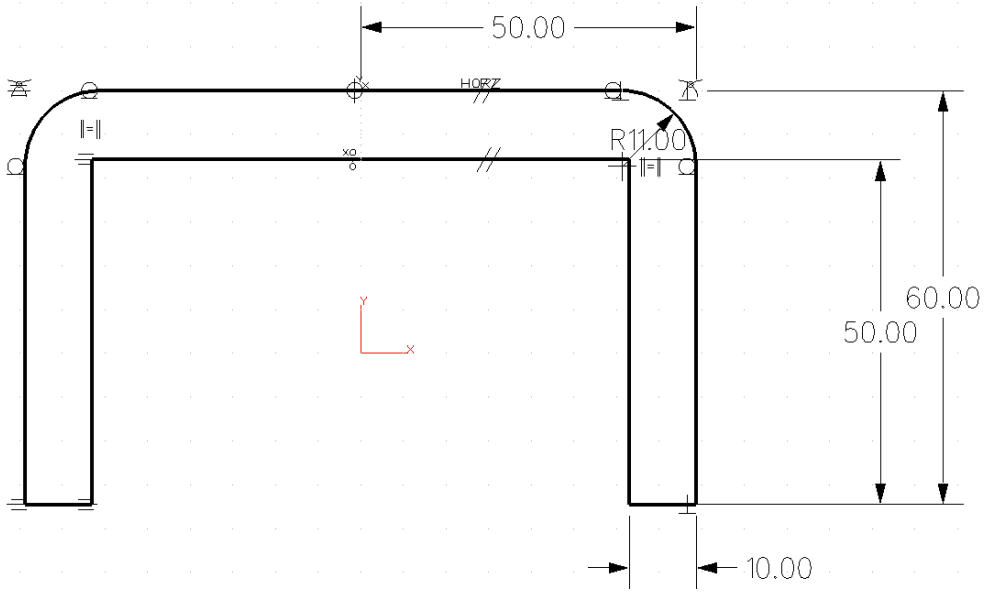
   <Right-click> Erase.

10. Pick the **Equal Distance** command from the **Constraints** Tab.

    Pick the right fillet as the base.

    Pick the left fillet to distances as equal.

11. Double click on the dimension for the right fillet and change its value to **11mm**. Notice the left fillets updates to reflect the change since we added the equal distance constraint. See figure below for reference.



Lesson 4b

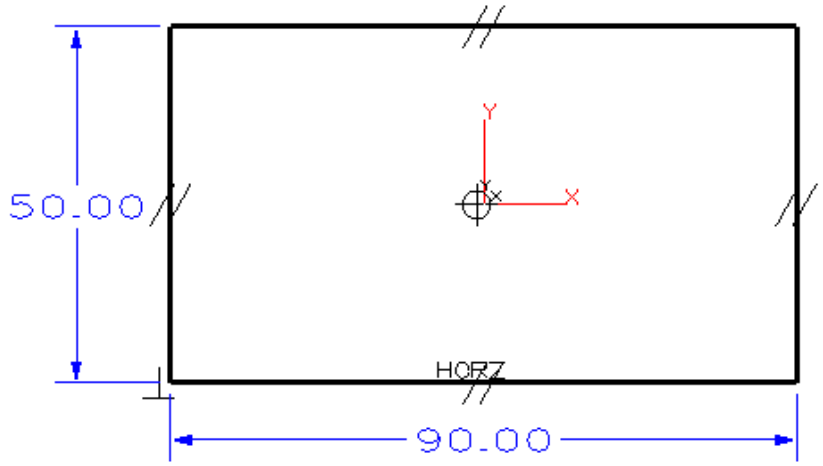
Working with ZW3D Constraints Advanced

1. Pick the **Sketch** icon from the **ZW3D** root. Name it **Lesson_4b**. Open the **Sketch Tab.**

2. Create a rectangle using the **Center, Corner Rectangle** command.

   <Right-click>Critical, set the center of the rectangle to **0,0.**

   Your dimensions may vary from those shown below, that's OK. It is more important that the center of the rectangle be at **0,0**.



   In this example we have a rectangle centered about **(0,0). Looks complete, right?**
   Not so fast! There are no dimensions or constraints to keep it at (0,0) if we make a change to the dimensions. Try changing one, then **UNDO**.

3. There are a couple of different ways to Fully Constrain our example and if you consider the different ways you can dimension the rectangle there are at least 5 solutions. Our design intent is to simply keep the rectangle centered on (0,0) no matter how the dimensions change.

   A. The first way is to add a **Mid Point Constraint**. Let's try this, then **UNDO.**

   Pick the upper left corner of the rectangle.
   Pick the lower right corner.
   And finally pick the point at (0,0)
   Make a dimensional change to test our constraint system. Cool!

   Now **UNDO** the dimension change.

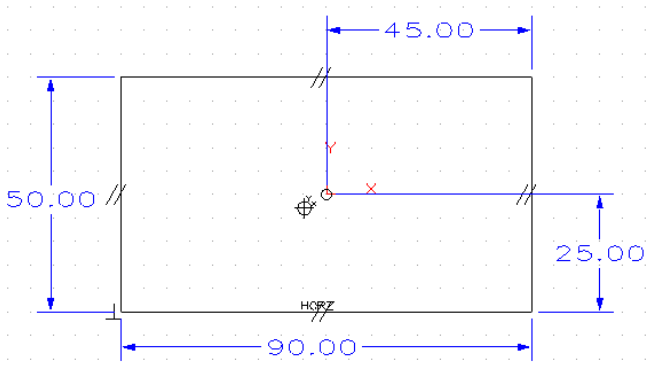
4. Now let's play, "What if?" What if we find out that our original design intent has changed and there is a possibility that this feature of our model may not be centered on (0,0).

5. **UNDO** the midpoint constraint or erase the constraint symbol.

6. **<Right-click> Dimension** and add the two dimensions as shown in the following figure. Again your dimension values may vary. That's OK. Now when the actual change is announced you can edit the 45.00 and/or 25.00.

   Try it! You'll like it!

7. **Save** your file.

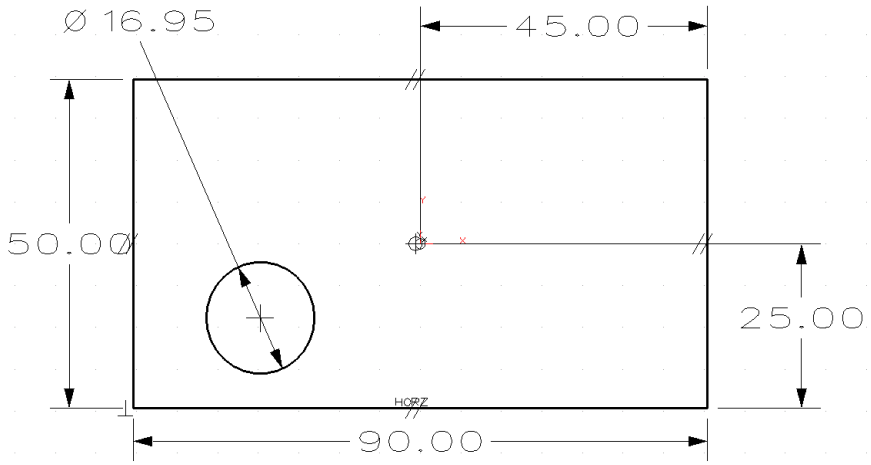


8. **<Right-click> Draw.**

   Hold the **<ALT>** key down as indicated in the command prompt.

   Add the circle and dimension it as shown in the following figure.

   The circle is not well defined yet because its location is not defined. Let's finish constraining the circle so we can control its location.



9. Add a **Horizontal Dimension**  from the center of the circle to the point as shown in the following figure.

   Now the design intent is such that we want the circle constrained horizontally to the center of the part. In other words, we need the center of the circle and the center of the rectangle to share the same Y value. Instead of adding a dimension and writing an equation, we can simply add a constraint to **Constrain Horizontal**.

10. Click on **Points Horizontal** in the **Constraints** Tab.

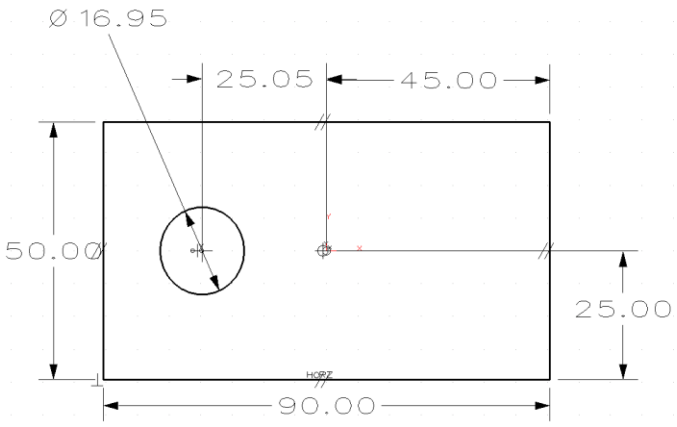
11. Pick the (0,0) point as the base point, and then pick the circle. Carefully follow the command prompt.

    The circle will move to its new position after the constraint has been successfully added.

    You should see the symbol  added to your sketch.

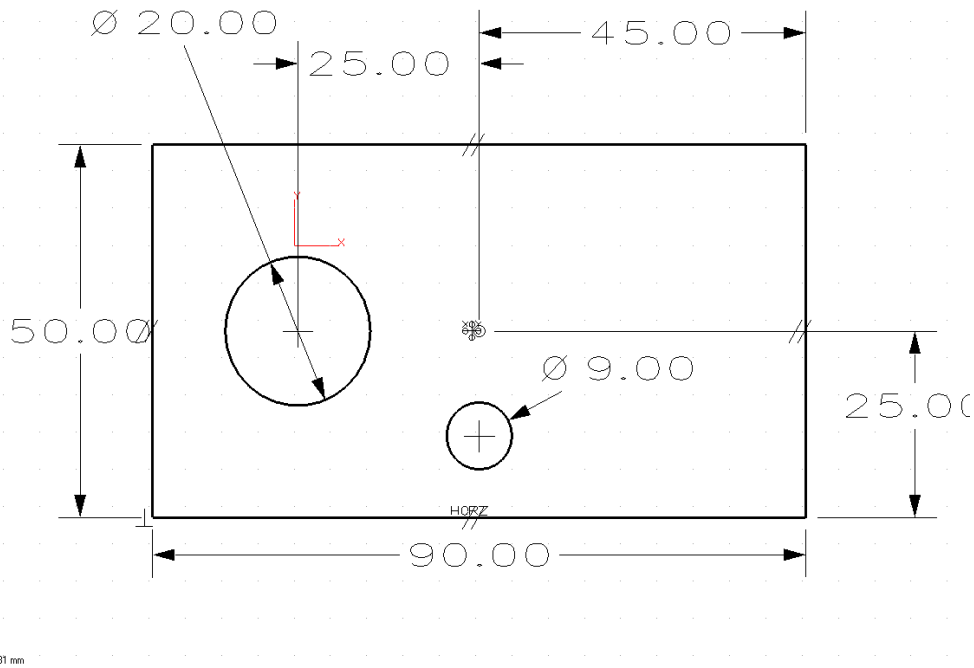
    Your screen should appear as shown in the figure below.

    Now, no matter how you change the dimensions, the circle will always lie on the same X-axis as the center of the part.



12. Try on your own adding another circle and constraining it vertically to the center of the part as shown in the following figure.

    Hint: At bottom of the page if you need it.



13. **Save** your session.

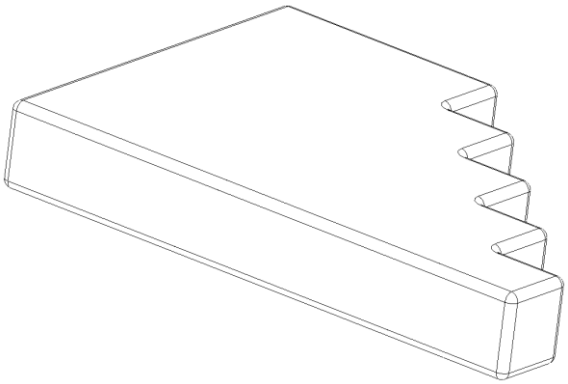
14. Pick **Exit Sketch** when you are finished.

Let's practice some of the commands we have not used very much yet. We won't give you specific details for each of these commands. Just create the following sketches to help familiarize yourself with the various **LINE** commands, **MOVE, ROTATE, STRETCH, SCALE, TRIM** and **MIRROR** commands.

Hint from above. You will need to add a "**Constrain Vertical**" between the circle center and the point at the center of the rectangle.

The geometry we are going to create in this exercise is a very simple shape, similar to stair steps. Our objective is to show you how to associate one dimension to another using ZW3D equations.

1. Pick the **Part/Assembly** icon from the ZW3D root. Name the new object **Stair_Steps**. Pick New Assembly Mode.



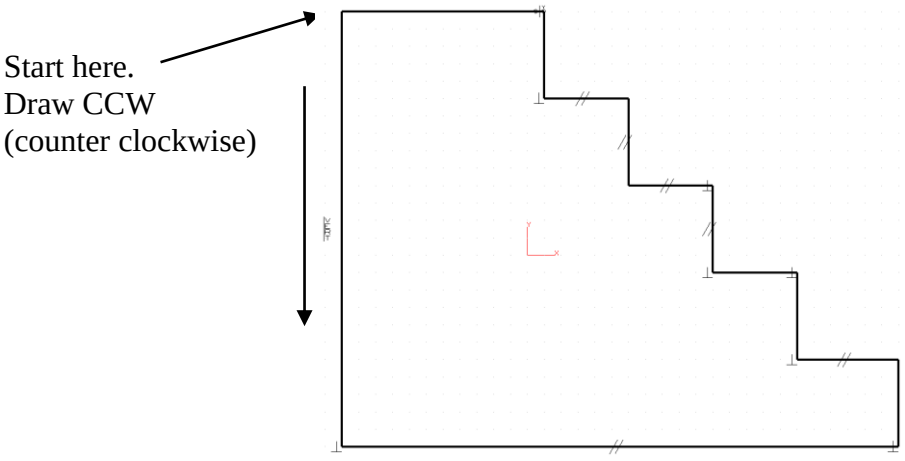
2. Pick the **Extrude** command from the **Shape Tab**. 

   <Middle-click> twice to start a new sketch on the default XY Plane.

5. **<Right-click> Draw.**

   Create the stair step sketch shown in the following figure.

   The dimension values may vary in your sketch. That's OK.



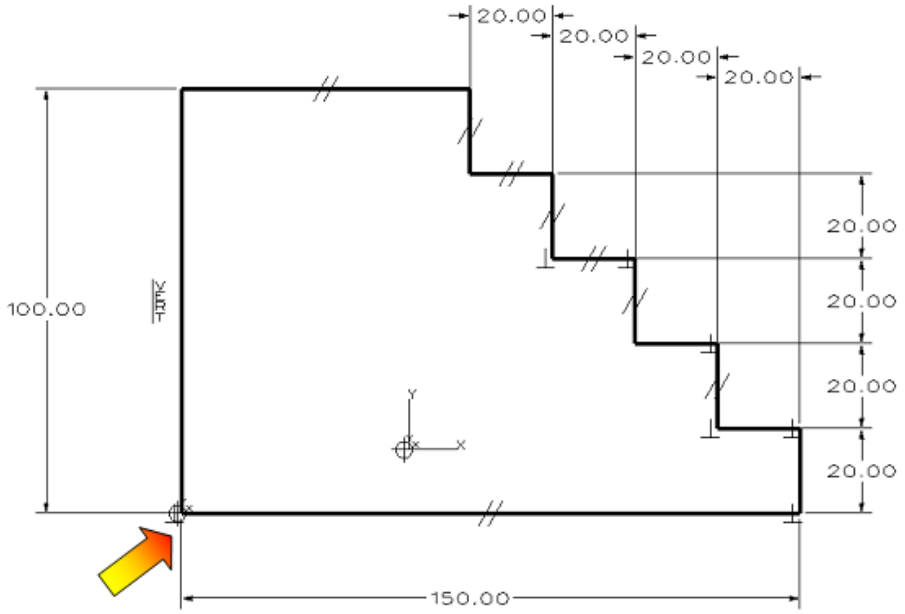
6. Next, let's use the **Auto Constrain** command to fully constrain our sketch.

   <Right-click> Auto Constrain.

   Pick the lower left corner of the sketch.

   Your well defined sketch should appear similar to the figure below.

Your dimension values may vary. That's OK. We will adjust them later in this exercise.



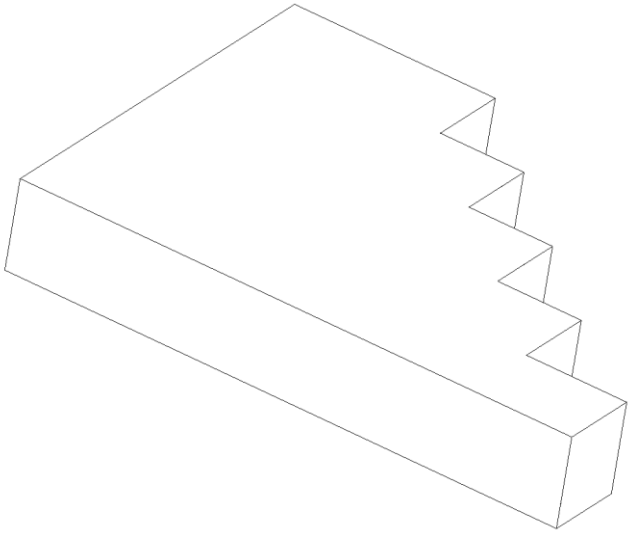
Pick the corner as indicated above.

7. Your sketch should display all geometric and dimensional constraints that have been applied so far. There are times when you need to turn them off or back on.
   Right-click and pick the **Hide ALL Constraints** icon.
   This will turn all constraints in the sketch part off.
   Right-click and pick the **Show ALL Constraints** icon to turn the **Constraints ON**.
   Pick the **Dimensions** icon from the **Document Awared Toolbar**.
   This will turn the appearance of all dimensions OFF.
   Repeat the icon pick to turn the dimensions back ON.

8. Now that we have finished working on the sketch lets extrude it into a solid.

9. **<Right-click> Exit Sketch.**

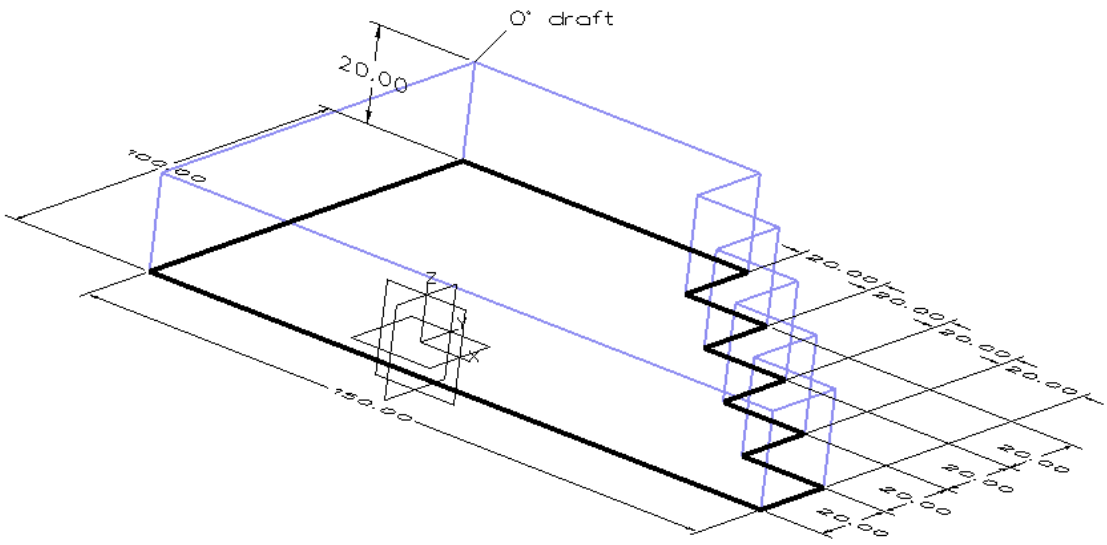
10. **ZW3D** remembers that we started this command with **Extrude** and redisplays the **Options** form. Adjust the **Start** and **End** of the extrusion as shown in the form below.
    Your solid model should appear as shown in the figure below.



11. Now that we have completed the basic design of our model, let's associate some of the dimensions by making all stair steps equal in size and controlled by only one dimension.

    Use the **<SHIFT> <Right-click>** keyboard macro to verify your pick **Filter** is set to **ALL**.

    Pick **Dimensions** icon from the **Quick Access** toolbar.



    There are a couple methods we could use to make these changes.

    Place your cursor over the overall length dimension to highlight it.
    <Right-click> Modify Value and change the overall length dimension to **170**.

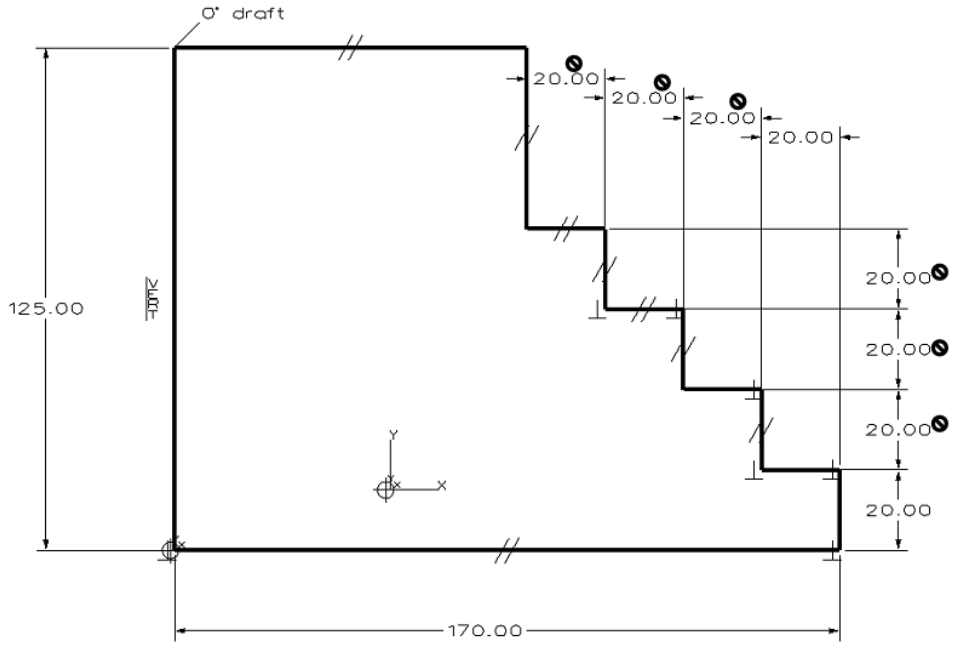
    Place your cursor over the overall height dimension to highlight it.
    <Right-click> Modify Value and change the overall height dimension to **125**.

    Or

Double **<Left- click>** on the overall length dimension and change its value to 1**70**.
Double **<Left- click>** on the overall height dimension and change its value to **125**.

Notice the **history** will automatically regen because **Auto regen on edit** is enabled on the **Part tab of the configuration.** If this were turned off, the **Auto regen** button on the Utility tool bar would turn green requiring you to manually click the button to regen the history.

12. Highlight the sketch and double **<Left- click>** on it. This will take you into the sketch mode where you can edit dimensions on the current sketch and change the shape of the sketch by erasing and replacing some of its objects. The sketch must be valid before ZW3D will permit you to update the model.



13. In these next few steps, we are going to associate all of the stair steps to be equal in size and controlled by 2 dimensions.

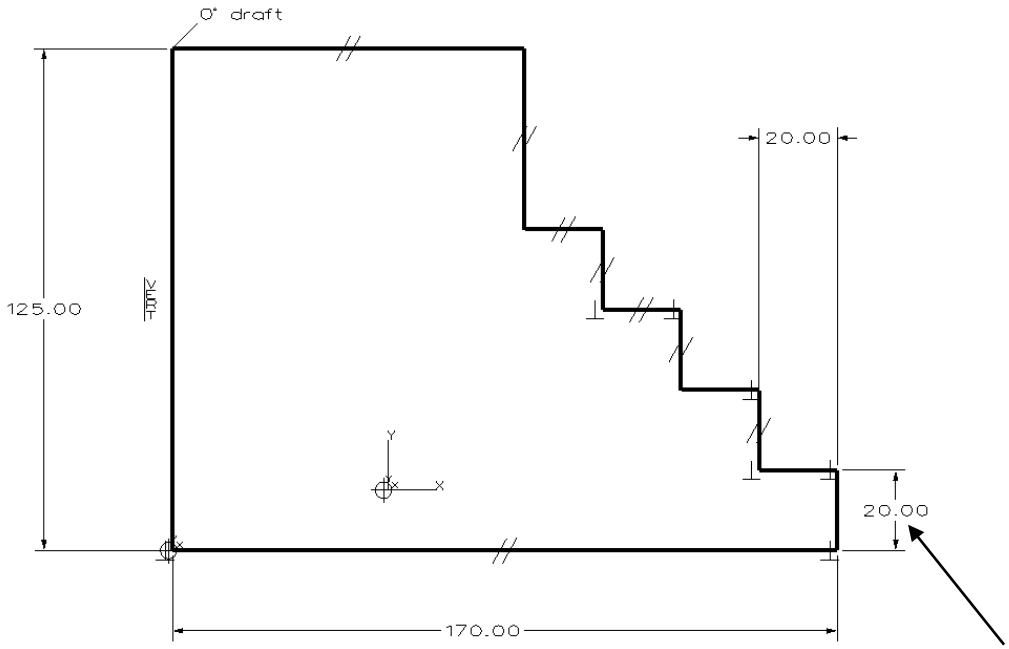
    Erase only the dimensions shown above with the symbol by them. There are a few ways we can do this while in the sketch mode.

    Highlight a single dimension by moving your cursor over it; don't pick it, and **<Right-click> Erase.**

    Or

    Pick all of the dimensions to erase and **<Right-click> Erase.**

    Your current sketch should appear as shown in the figure below.



14. Highlight the Vertical Dimension defining the height of the stair step. **<Right-click> Modify Value.**

    The following form will be displayed.

    In the **Input Dimension Value**, Pick **Select Dimension**  and pick the Horizontal dimension defining the length of the stair step.

    ZW3D will not allow you to pick the vertical dimension again by mistake.

    Your **Input Dimension Value** form should display “Sketch1_d?”. Your numbers may vary. That’s OK.

    Pick **OK** in the form.

15. Double **<Left- click>** on the **Horizontal Dimension** and change its value to **25.00**.

    The **Vertical dimension** value will change to match since they are associated to one another.

    Let’s check to see which sketch entities have yet to be constrained. Pick the **Constraint Status** icon from the **Constraint** Tab.

    You result should be similar to the following:
    7 DOF left / Geometry 8 Under constrained 5 Well defined / Dimensions 4 Solved / Constraints 13 Solved.

    - GREEN entities are fully constrained or well defined.
    - Blue entities are partially/under constrained.

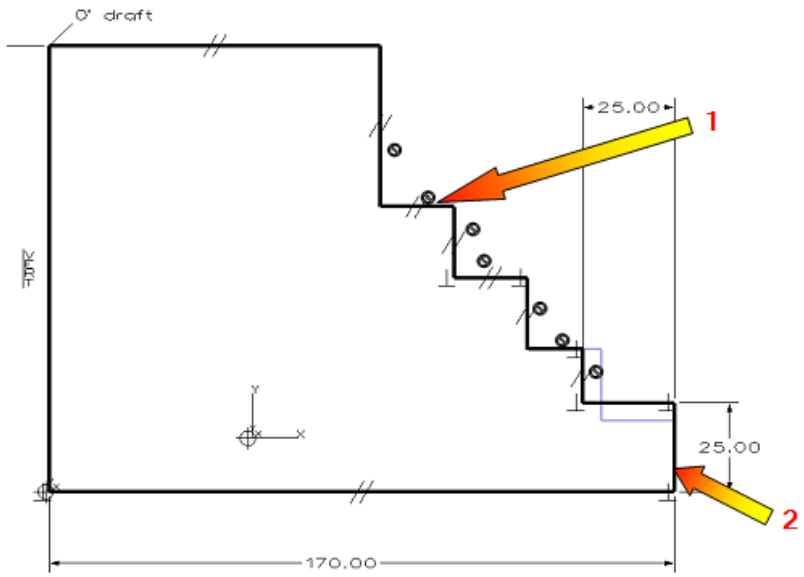
- Red entities have redundant or inconsistent constraints. (Dimensions and constraints that appear in red are ignored during the solve process).

16. Leave the **Constraint Status** form on the screen when performing this next step.

17. Erase the vertical dimension controlling the overall height of the model.

18. Pick the **Constrain Distance to be Equal** icon from the **Constraints** Tab.

Complete the constraining process until the entire sketch is GREEN. Apply this constraint to each line defining the stair steps as shown in the following figure.



1:Pick these 7 lines to constraint the distance equal to the base entity. 2: Base entity to fix.

The **Constraint Status** form should indicate the sketch is Well Defined.

Your sketch should update to reflect the changes.

19. **<Right-click> Exit Sketch.**

    You might receive the following error from ZW3D when trying to exit a sketch, which says "Some constraints for [sketch1] are not solved. Regeneration could change the sketch. Regeneration the sketch before exiting?".

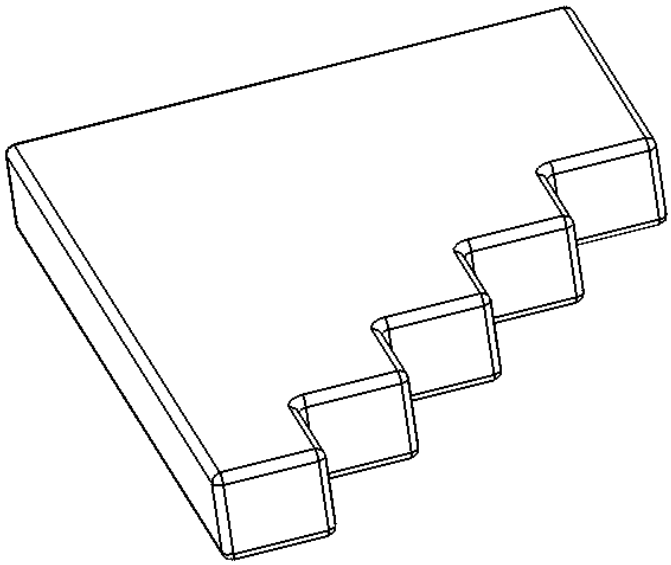
    This message indicates you have not regenerated(solved) the sketch after applying the final constraints while the **Delay Solve** was on.

    If this message appears, pick the "**Yes**" button in the form to regenerate the sketch constraints before exiting the sketch mode.

    ZW3D has automatically updated the model based on the final constraints you just applied.

20. Pick the **Fillet** command from the **Shape Tab**  . Add 1.0mm fillets to the entire model as shown in the figure below. **ZW3D** will highlight the fillet size for visual verification.

    <Right-click> Pick All to select all edges in the active part object.



Completed Model

21. Finished.
    Save your model and **Exit part.**

Rules for Sketching

First rule of sketching: “sketch pretty close to scale.” Use the zoom scale number in the lower left of the screen as a guide. The value is roughly the vertical dimension of the current screen.

Second rule of sketching: “pay attention to the first rule.” If you have sketched to scale you probably shouldn't need to make large changes in values. Set the Step Size to a small number and use the arrow keys to make small adjustments.

Third rule of sketching: "don't draw the entire sketch, then come back and try to dimension and constrain." You have heard the expression, "pay as you go"? Constrain as you go. Draw an area and constrain it. If a sketch is really complicated, break it into smaller sketches.

Fourth rule of sketching: “If you have started constraining a sketch, especially if you have drawn circles, don't trim.” Trimming will destroy constraints. Use a trace profile.

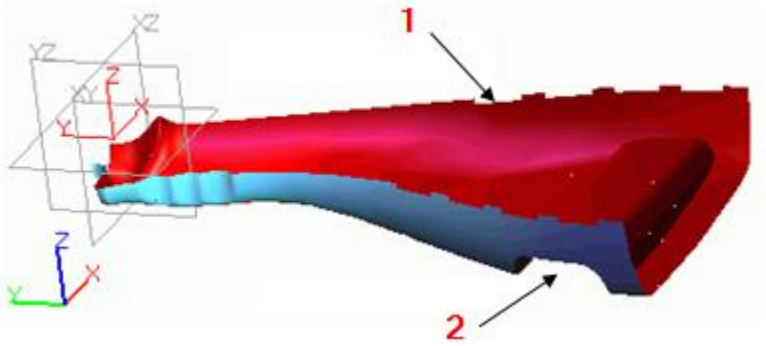
Fifth rule of sketching: “Create the required geometric and dimensional constraints first.” If the sketch still requires some constraints, use the Auto Constrain command to finish the process by <Right-clicking> Constrain.

Chapter 5

Basic Solids Design and Editing with ZW3D

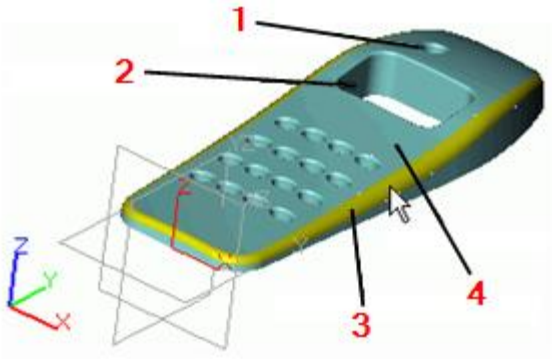
Solid Features in ZW3D

Solid feature commands (e.g., **extrude, revolve, sweep, loft, fillets, chamfers, ribs,** etc.) can be performed on **open** or **closed shapes.** While a solid has been traditionally defined as a closed shape, it is important to remember that **ZW3D** generally allows you to create these features even if the shape is open (i.e., one or more faces of the solid are missing). Only the faces that interact with the feature are modified.



Solid Features on Open Shapes(1: Open Shape Feature 2: Solid Cut Feature)

There is a special type of solid feature called a base feature. Base features are used to begin the construction of a new part by defining its basic shape. Other features are then applied to the basic shape until the part is complete. A part can be more than one base feature. Many solid features use one or more sketches as input (e.g., **Bosses** and **Cuts**) while others do not require a sketch (e.g., **Fillets** and **Chamfers**).



1: Hole Feature 2: Extrude Cut Feature 3: Variable Fillet Feature 4: Base Feature

Solid Feature Examples

In this discussion you have learned that:

- Solid Features can be applied to open or closed shapes.
- Solid Base features are used to begin a new part by defining its basic shape.
- Some solid features (e.g., Bosses and Cuts) defined by one or more sketches.
- Other solid features (e.g., Fillets and Chamfers) do not require sketches.

The ZW3D Command Options Forms

By default you are sequentially prompted for the **Required inputs** during command execution. ZW3D contains additional functionality found under the **Options** and **Advanced** options tabs. The **Options** Form can automatically be displayed in either its short or long form for all commands. The behavior of the **Options** Form is controlled by the **Show option form** from the General tab of the ZW3D Configuration Form.

- **Short** - Only the **Required Inputs** portion of the form is displayed.
- **Long** - Both the **Required Inputs** and the **Optional Inputs** portions of the form are displayed.

The short version of the form displays only the **Required Inputs** for a command while the long version displays both the **Required Inputs** and the **Optional Inputs**. Switching between the long and short forms is accomplished with the double arrow icon  at the bottom left of the form.

The question mark  icon displays the documentation for the current command.

With the **Options Form** displayed, the required and optional inputs can be entered in any order at any time. It is very easy to go back and change an input that was previously entered or to skip ahead and supply an input that will make the preview echo look better. If a **Required Input** is not entered, you are reminded to supply the input when the "**OK**" button is pressed.

Notes:

When inputting values in an option form, be aware if you move your cursor off the form.

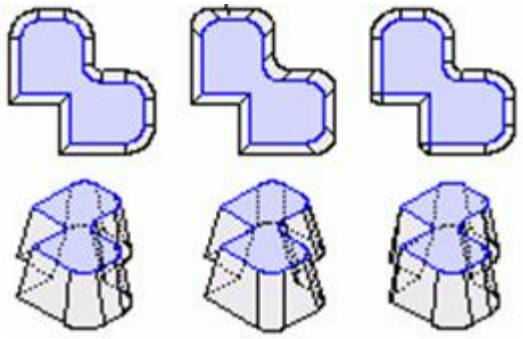
Moving the cursor off a form changes ZW3D to the dynamic input mode. You will see the numbers in the active field change relative to the current cursor position.

Direct form input or dynamic input is fully supported in ZW3D. Be aware of your input method.

IMPORTANT REMINDER: If you type a value, hit ENTER on the keyboard.

Optional Inputs Common to Basic solid Feature Commands

Extruded Profile with Draft



Variable Constant Round

Draft angle
Enter the draft angle if desired. Positive and negative values are acceptable.

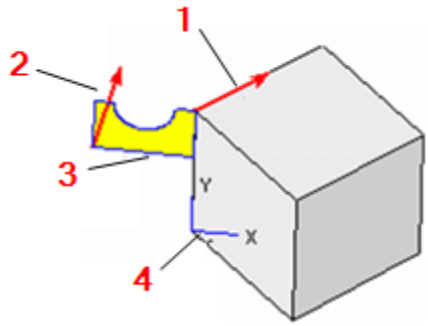
Blend
Use this option to select the corner blend method. Refer to the figure below.

- **Variable** - Corners are not modified. They maintain the same convexity/concavity (default)
- **Constant** - Filleted/rounded corners will maintain the same radius.
- **Round** - Convex corners are rounded. Concave corners are not rounded.

Draft in extrude direction
Check the box to apply draft in the extrude direction. Otherwise, draft will be applied in the direction normal to the profile or sketch plane.

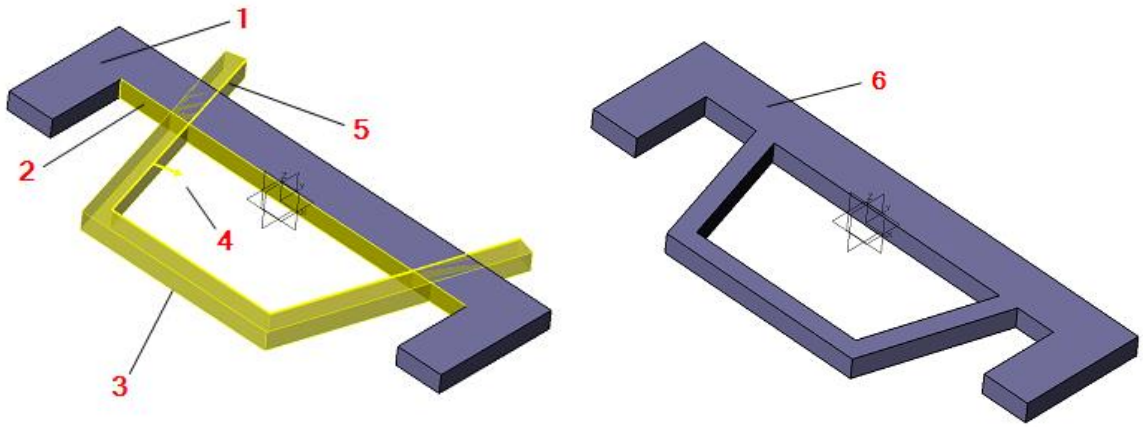
Direction
Specify the direction to extrude. This option will override the default extrude direction that is normal to the sketch plane.

Profile Cap
If the selected profile is open (i.e., it does not form a closed loop), you can use this option to specify the boundary face. This face will serve to close the open profile. (Refer to the HELP Manual Tips & Techniques.)



1:Extrude Direction 2: Normal Direction 3: Profile 4: Sketch Plane

Refer to the illustration below.



1:Starting Shape 2:Profile Cap 3:Thickness 4: Interior 5: Open Profile 6: Final Shape

Offset, 1st offset, 2nd offset
Use this option to specify an offset method and distance(s) to be applied to curves, curve lists, or open or closed sketched profiles. This option adds thickness to the feature automatically. Follow the steps below for each method.

- **None** - No offset

- **Shrink/Expand** - Creates an offset by shrinking or expanding the profile. The shrink/expand direction is determined by the distance value. A negative (-) value expands to the outer side and a positive (+) value shrinks to the inner side of the profile.

  Enter the offset distance (1st offset).

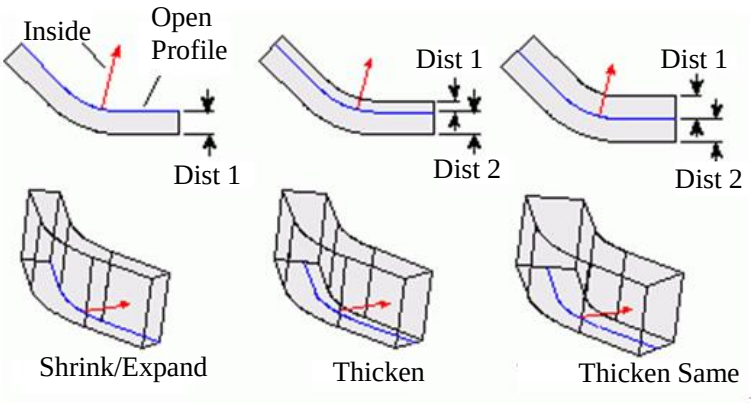
- **Thicken** - Creates a thickness from the profile. The thickness directions are determined by two distance values. The first distance shrinks the profile toward the inner side. The second distance expands the profile toward the outer side.

  Enter the inner offset distance (1st offset) and then the outer offset distance (2nd offset).

- **Thicken Same** - Create a symmetrical thickness about the profile. The thickness on both sides of the profile is determined by a distance value. The total thickness equals 2 times the distance.

  Enter the offset distance to be applied to both sides (1st offset).

- Connected perimeter faces are created by this option. The ends of the faces will remain open unless the **End caps** option is used (see below). If both end caps are specified, a solid is created.



Use these options to control the placement of end cap faces on the start and end of shapes. This can automatically form closed volumes when a closed profile is used or when an open profile with the **Profile Cap** option is used. Select the icon to apply.

Refer to the HELP Manual

Refer to the **HELP Manual** and review the following commands.

Extrude Solid
Revolve Solid
Dynamic View On/Off (F2, F3 and F4)
Rotate View about a Line or Axis
Set View Axis
Set View Origin (Ctrl+`)
Standard Views
Insert Sketch
Sketch Settings Form
Fillet Feature
Chamfer Feature
Hole Feature
Pattern Features (Circular)
Insert Datum Plane
Reference Curves
Curve from Edge
Trim / Extend Wireframe Curve
Mirror Features
Turn Dimensions On/Off "Ctrl+D"
Turn Constraints On/Off
Show All
Show Target

Lesson 5

Basic Solid Modeling & Editing Techniques

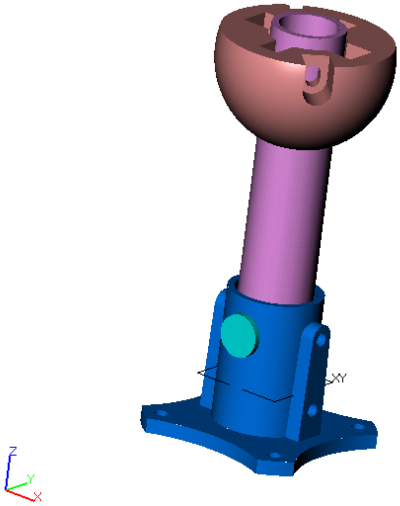
Designing a Ceiling Fan using the Bottom Up Design Method

In this next lesson we are going to model several components that make up a ceiling fan assembly using the **Bottom Up** design method. Each part will be built independent of the next.

In practice, the **Top Down** method is probably more efficient and lets you reference existing pieces of the model and preserve design intent. Take a look at the simple assembly below. In the Top Down approach, if the tube were to shrink or expand you would want the flange and the ball to update accordingly. We will show an example of this at the end of the chapter.

1. Let's begin by modeling the **Support Pin** shown in the figure right. The pin goes through the top of the **Tube** and is buried inside the **Ball.**

2. Launch ZW3D and create a **New Multi-Object File** with the name "**My_Fan.Z3**". 



3. Create a new Part / Assembly object and name it "**Support_Pin**". Pick New Assembly Mode. You will be placed at the Part level of ZW3D

   The **Shape** and other Tabs will appear at the top of the graphics screen as shown in the figure below. If you don't have a valid license for any of the software modules that Tab will not appear.

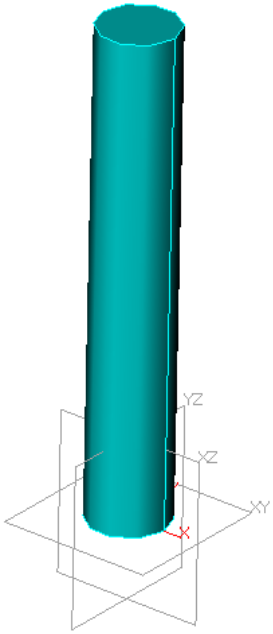
4. Go to the **Edit pull down > Preferences** and verify your units are set to **MM**.
   If your units are set to something else you may want to make a system wide change because we will be creating several mm parts. Change the units here because this part is already open. Also click on the Utilities pulldown menu. Click on Configuration. Click on Default linear units and change to mm.

5. Pick the **Cylinder** command from the **Shape Tab**. 

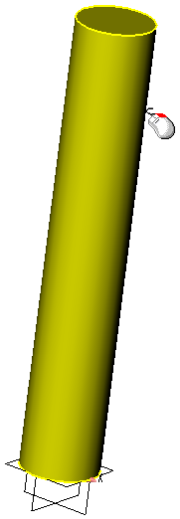
6. When prompted to enter the **Center Point** on the command line type in **0,0,0** and hit <**ENTER**> Get in the habit of typing and hit <Enter>

Then fill in the radius at **3mm** and the length at **35mm** as shown below and pick **OK**.

The default align plane is the XY Datum, which is the desired Align Plane, so no action is needed.



7. Pick the **Zoom All <CTRL + A>**  command from Document Award Toolbar to view your entire model.
8. Pick the **Display Shaded <CTRL + F>**  command from Document Award Toolbar to shade your model.
9. Pick the **Entity Filter** and set it to **Shape**.
10. **Highlight your model > <Right-click> Attributes**.



Change the Face Color to **Dark Purple and pick OK**. See figure below for reference.

ZW3D allows you to change the characteristics of an entity by <Right-clicking> over it and picking the attributes command.

For a face, the Face Attributes dialog box would appear.

For a dimension, the Dimension Attributes dialog box would appear.

For text, the Text Attributes dialog box would be displayed.

For Hatch in a drawing, the Hatch Attributes dialog box would appear and so on.

The Key is the Filter Icon.

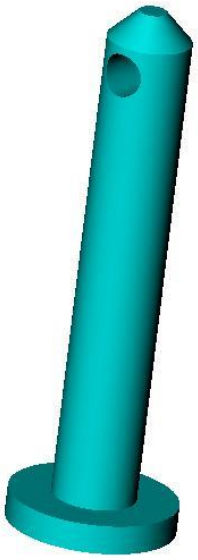
The filter is what determines what is available for selection and the shortcut or right-click menu is directly related to the pre-selected item(s).

11. Save.

12. Pick **Exit part**.

Completed Support Pin

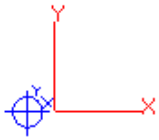
The next part we are going to design is the **Flange Pin** shown in the figure below.



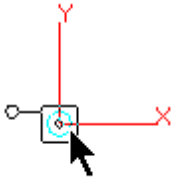
1. Create a new Part / Assembly object and name it "**Flange_Pin**". Pick New Assembly Mode.

   You will be placed at the Part level of ZW3D.

2. Go to the **Edit pull down > Preferences** and verify your units are set to **MM**.



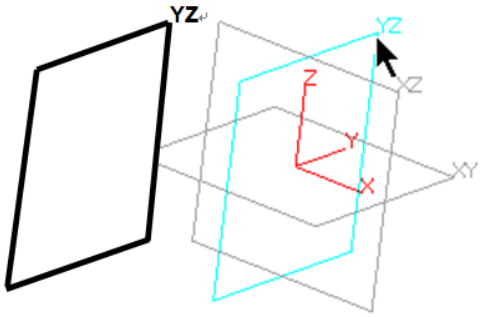
Anchor Constraint



Anchor Constraint was hidden to show proper Critical **Snap to Zero Point**

3. Right-Click in the graphics window and Pick the **Insert Sketch** command from the shortcut menu.

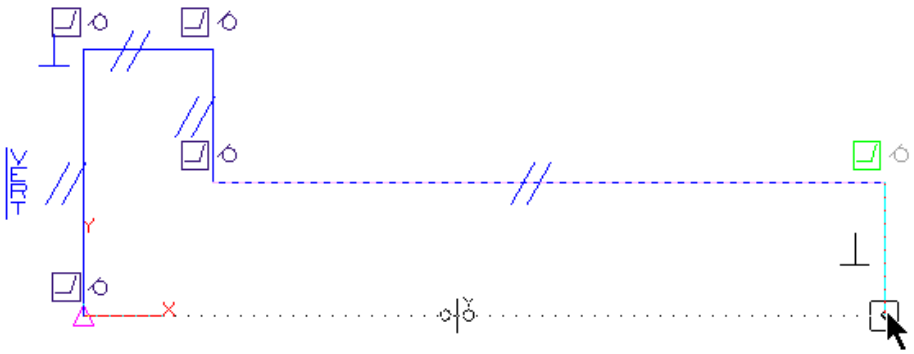
4. Choose the YZ Datum for the Insertion Plane.



<Middle-click> or Pick OK from the Options box to accept the remaining defaults.

You will be placed at the sketch level and on the YZ datum plane.
The display has been adjusted specific for sketching. All 3D modeling commands have been turned off since they do not apply in a sketch.

5. Let's begin by drawing the sketch shown in the figure below. This will make the basic shape of the Flange Pin with the exception of the hole. Make sure you follow the command prompt.

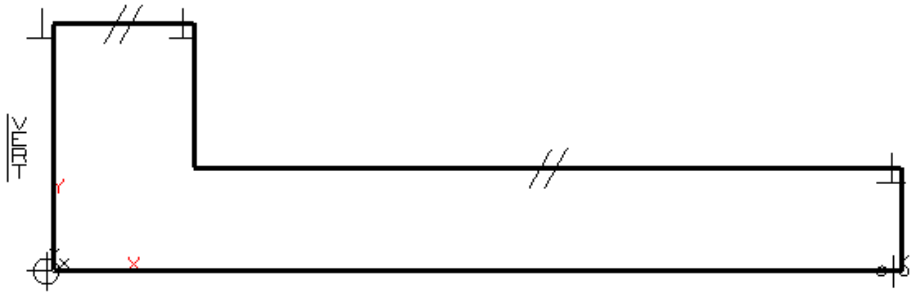


Pick the **Draw Tool**  command from the **Sketch** Tab. This command is also found with a **Right Click** in the Graphics area of the screen.

With the Draw Tool you will default to creating connected lines. Your starting point of your first line should be at Zero, Zero. You can move your cursor to Zero and Pick.

If necessary, you can hide your constraints with the **Hide All Constraints** command.

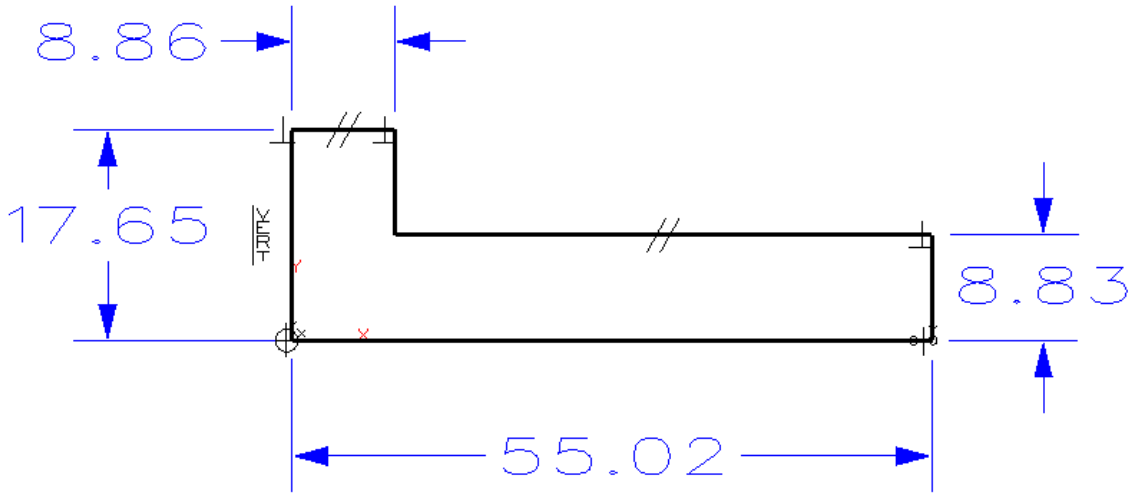
Your final sketch should look like the following.



Next we will apply all of the dimensions and edit the dimensions.

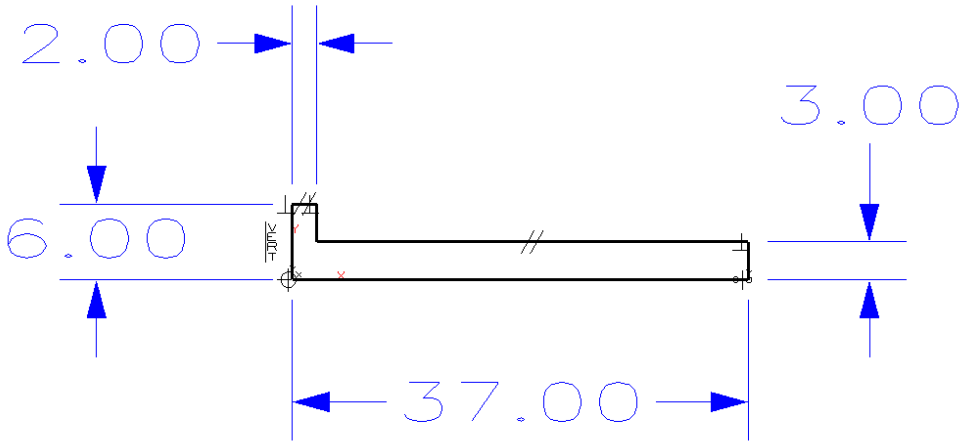
Using the smart **Dimension** Tool or the **Dimension** command (**Right Click** in the Graphics windows)  , you can quick dimension your sketch. Simply pick the line that you want to dimension the length of and place your dimension. Don't worry about values now; simply get the dimensions on your sketch to constrain it.

HINT: Use Middle Click to quickly re-execute the Dimension Command.



After you have your dimensions on the sketch, edit the values of each dimension by **Right Clicking** on a dimension and choosing **Modify Value** or **Double Clicking** a dimension.

When you are done editing dimensions your sketch should look like this.



Solve current sketch and look for this in the message window:

MESSAGE: Profile constraint system successfully solved.

HINT: Make this a Best Practices habit.

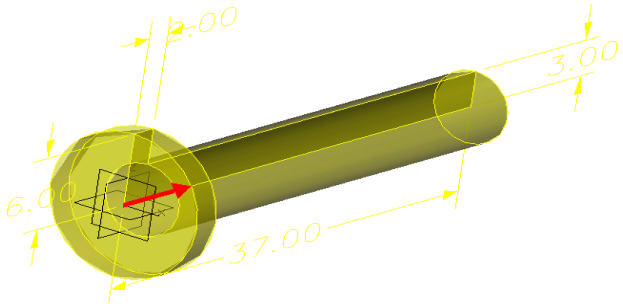
6. **<Right-click> Exit Sketch.**

7. Choose **Revolve** from the **Shape** Tab. 

8. The options form for the Revolve command should appear on the screen

You are prompted to select the Profile. We will use the sketch we created in the previous step. You can simply move your cursor over the sketch and **Left-Pick** when you see it highlight.

Alternately, you can click on the **History Manager** tab, pick the sketch from the history list, then return to the **Input Manager**.



You are then immediately prompted for an Axis of Rotation.

NOTE: Anytime ZW3D wants you to provide an axis you can do one of four things.

1) Select an existing line from a sketch, edge or 3D Line
2) Right Click in the Graphics window for the Direction/Axis Menu.
3) Pick on the X, Y or Z axis
4) Pick on the axis in the lower left corner of the graphics window.

For this part our axis of revolution is the **Y axis**. Simply **Right-Click** in the Graphics window and choose **Y axis** from the Direction/Axis shortcut menu.

For **Start angle**: Middle Click to get the default value of Zero.

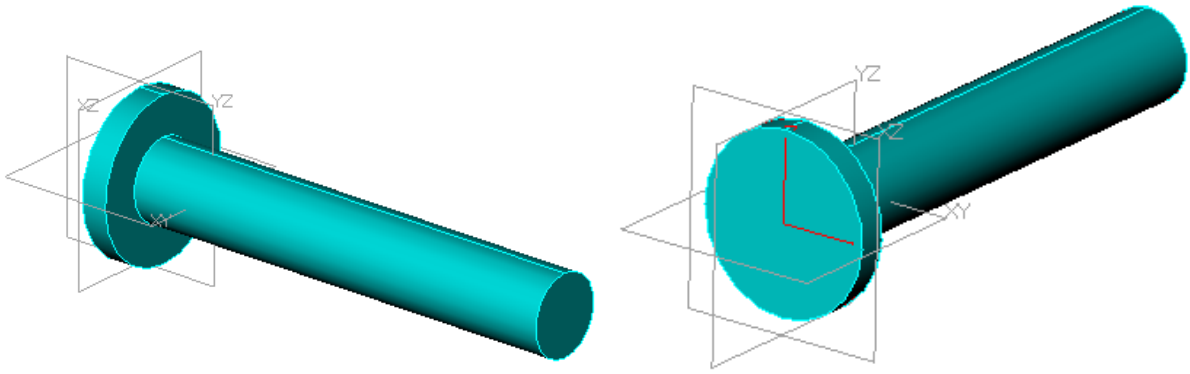
For **End angle**: Middle Click to get the default value of 360.

Pick **OK**.

NOTE: Revolve supports optional inputs that allow you to offset, shrink, expand, and thicken the sketch profile used to produce the Revolve.

Choose OK or Middle Click when the required options are filled in.

Your solid model should appear as shown in the following figures.



9. Let's create the Hole near the tip of the Flange pin by Extruding a sketch of a circle (we will introduce the Hole Feature later).

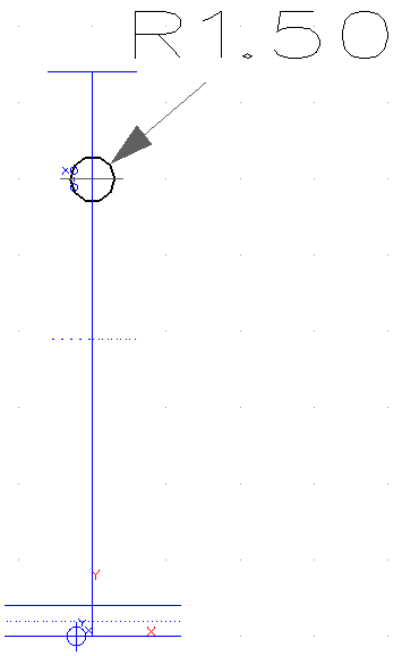
10. Insert a sketch on the XY Datum by doing the following.

    Right-Click > Insert Sketch
    Choose the XY Datum or **Middle Click** (Default datum for inserting sketches)
    Middle-Click or **Choose OK** in the Options form

    The title bar in ZW3D should display "Sketch[Sketch*]",the number may vary, that is OK.

    "Sketch[Sketch*]" indicates that you have successfully entered the sketch mode.

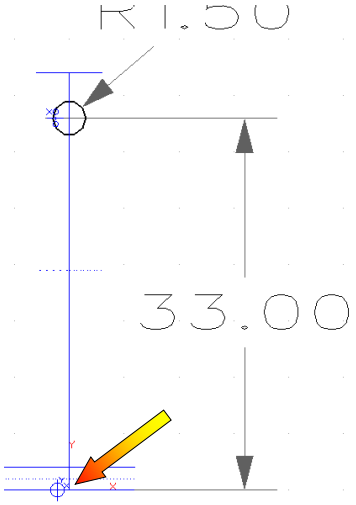
    If you have a shaded view of your part while in sketch, you can **un-shade with CTRL-F.**



Create a **Circle by Radius**  **and Center**  and specify a **1.5mm Radius** on the command line and position the circle as shown in the figure to the right (aligned with Zero).

Dimension the circle to be 33mm from Zero

Exit the Sketch or Right-Click **Exit Sketch** from the shortcut menu.

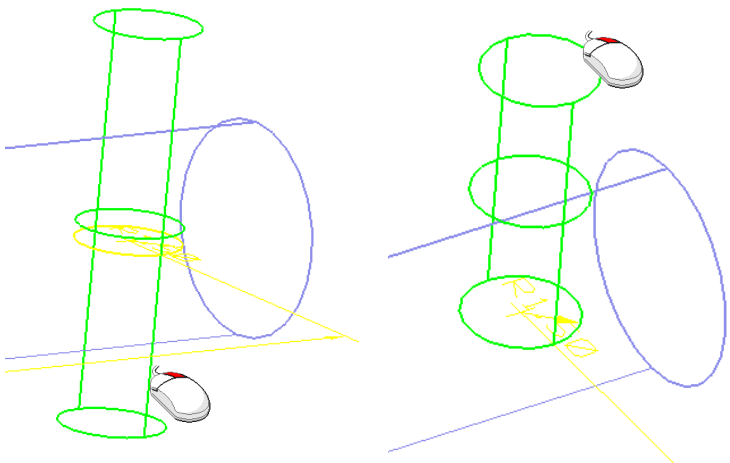


Note: the point indicated above is zero point.

12. Pick the **Extrude**  command.
Respond to the command prompts as follows:

Choose **REMOVE**  as the Boolean Option, this will perform an extrude cut or hole in the model (see the image to the right).

Profile – **Select the Sketch** that we just created.



For the Start Value – Move your cursor into the Graphics area. You will see the circle moving up and down with your cursor. Move your cursor above the sketch, **Right-Click** and choose **Through All** from the shortcut menu (See Image to the left).

For the End Value – do the same thing, except move your cursor in the opposite direction, **Right-Click** and choose **Through All** (See the Image to the right).

Pick OK or Middle-Click to complete the Extrude remove feature.

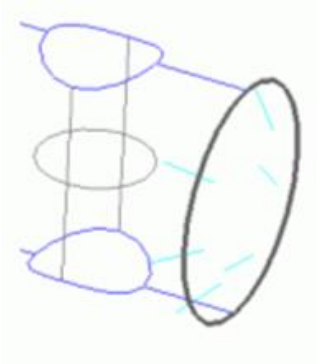
Note: There are other options like, To Point, which allows you to have an extrude position with an associative relationship to a point.

13. The next thing we want to do is put a chamfer on the end of the Flange Pin. Pick the Chamfer command.

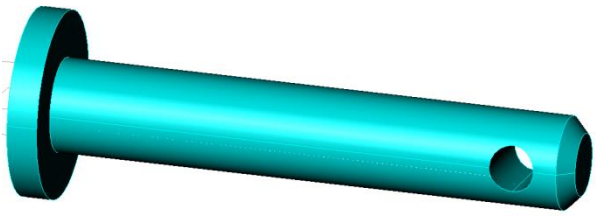
    Pick the small diameter edge of the Flange Pin at the end of the part where we just built the hole. (See the image to the right).

    Middle-Click to accept your selection.

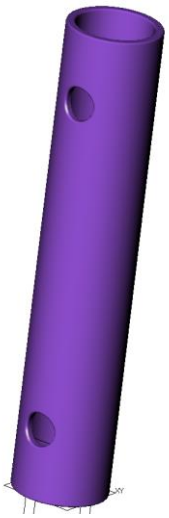
    Set the Chamfer **Set Back to 1mm**.
    Pick OK or Middle-Click to complete the Chamfer feature.



14. Your completed part should look like the following figure. Save and Exit.



The next part we are going to create is the Tube.



1. Launch ZW3D and OPEN the file "**My_Fan**" **.**(if it's not already open)

2. Create a new Part / Assembly object and name it "**Tube**". Pick New Assembly Mode. You will be placed at the part level of ZW3D.

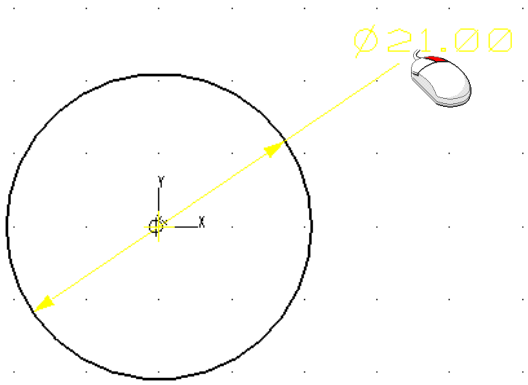
3. **Right-Click** in the graphics window and pick **Insert Sketch** from the shortcut menu.

   <Middle-click> two (2) times to accept all of the defaults.
   You will be placed at the sketch level and on the XY datum plane.

   Again, the display has been adjusted specific for sketching. All 3D modeling commands are not available while creating a sketch feature.

4. **<Right-click> Draw** or Pick the **Center**  **, boundary circle**  command from the Sketch Tab.

   If you choose to use the Draw command from the <Right-click> shortcut menu, press and hold the <Alt> key when picking the center of the circle.
   The Center, boundary Circle will allow you to create consecutive concentric circles. Simply **Middle-Click or hit <ESC> to end** the creation after you have place one circle.



Make the Diameter 21mm by placing a dimension with the **Dimension** tab, which can also be found on the **Right-Click Shortcut** menu and change its value to 21. If you failed to change the value when you placed the dimension simply right click on the dimension and choose **Modify Value** from the shortcut menu (Shown in the figure to the right).

5. **<Right-click> Exit Sketch.**

6. Choose the **Extrude** Command.

   Fill in the options form for the Extrude as follows:
   Start: **0 or <Middle-click> to accept the default of 0**.
   End: **100**
   <Middle-click> or Pick OK to Finish.

7. Next we want to create the Inside Diameter of the Tube. There are several ways to accomplish this (Shell, Extrude cut another circle, we could have drawn the OD and the ID in our sketch). What we will do here is introduce some advanced extrude options as well as show how to edit or redefine the Extrude feature that we just created.

   Simply **Right-Click** in your graphics window and choose **Redefine Last** from your command shortcut menu.
   Note: Another way of doing this is from your **History Manager** Right-Click the Extrude Base feature and choose redefine.

   Redefine the Extrude Options as Follows.
   Change the **Offset to Thicken**
   Change the **Inner Offset to 2**. This will give a 2mm inside wall thickness to our Extrusion feature.

   Note: If you set the Outer offset to 2mm it would have made our OD 2mm larger than our original sketch.

   Your Tube should now look like the figure below.

8. Next, we want to create two cross-holes in the tube that will get used during the assembly process. We need to create a sketch on the XZ or YZ work plane. You should already see these planes in your model.

   Pick the **Extrude** command.

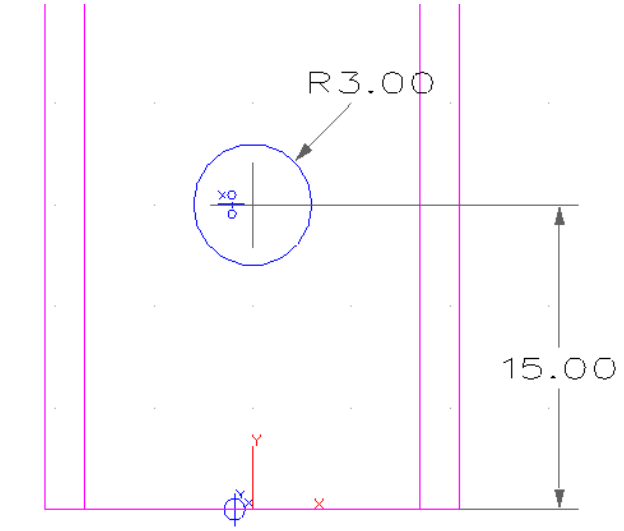
   Pick the **Remove** operator option

   Respond to the command prompts as follows:

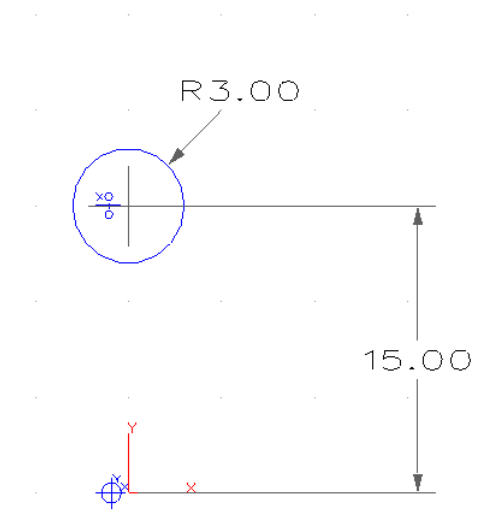
   <Middle-click> to start a new sketch.

   For the insertion plane, pick the **XZ** datum plane or type **XZ** on the Command Line.

9. Create and fully constrain the sketch shown in the figure below.

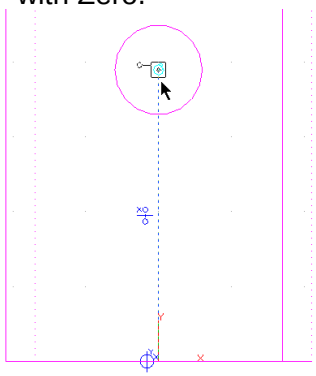


Sketch with Show All turned ON



Sketch with Show Target turned ON

TIP: You can align the center of your circle with Zero.



TIP: You can Quickly Apply missing constraints and dimensions with the **Auto Constrain** Command which is also available on the command shortcut menu by using **Right-Click**.

After placing the circle in the sketch above you can Auto Constrain with Zero as your base point to get the remaining dimensions.

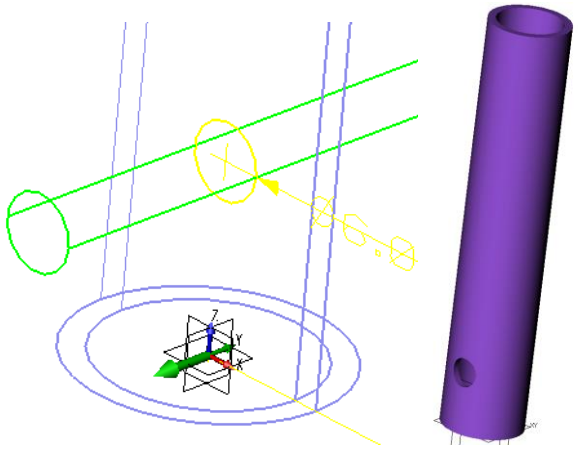
10. <Right-click> Exit Sketch

Specify start: **Right-Click > Thru All** in one direction.

Specify end: **Right-Click > Thru All** in the other direction

No Draft Angle.

Your solid model should appear as shown in the figure below.



11. In these next few steps we are going to copy the cross hole up **70**mm. There are a few techniques we can use to accomplish this task. We could simply copy the hole up a specified distance using the Pattern command; we could have created 2 circles in the previous sketch we just extruded or we could mirror the hole about a plane. Let's try using the mirror method since we have not used that command yet.

12. Let's begin by creating a Datum plane in the middle of the tube. We will use this plane to mirror the cross hole about.

<Right-click> Insert Datum. Pick the XY icon in the Options menu. 

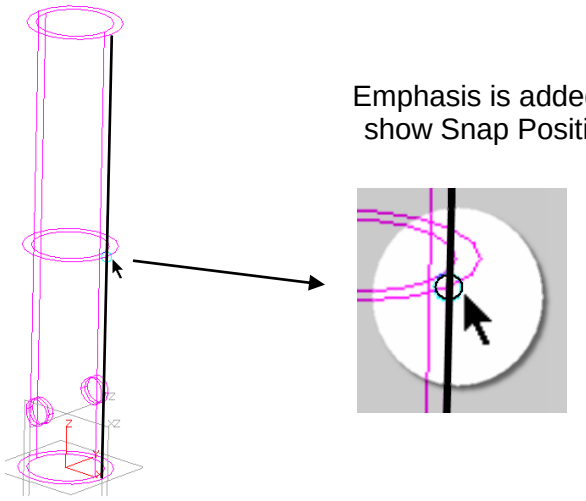
At the prompt, Specify Offset: **<Right-click>** for the Position Options and choose **To Point** as shown in the figure above right.

At the prompt, **<Right-click>** and choose **Critical** from the Position/Snap Menu as shown in the figure below left.

Pick the edge as shown in the figure to the middle. You will see your cursor snap to the center of the edge. **Middle-Click or choose OK to Finish**

Note: This will establish the datum to always be positioned at the center even if the length of the tube would change.

Note: If we wanted to specify a percentage of the total height of the tube we could have used the "Along" option from our Position Menu (figure to left).



13. Controlling the **ZW3D Manager** Window. This on the Tools pull-down menu turns on/off the menu.

    a. Tab on the ZW3D Manager displays the **History**.

    b. **Right-Click** in the graphics window and choose **ZW3D Manager** from the command shortcuts.

    c. **Do it again if necessary to turn it back on.**

    IMPORTANT: Feature and Datum Selection can be made from the History Manager when a command prompts for input. We can use this when creating a mirror feature.

14. Pick the **Mirror** command from the **Shape** Tab.

    Select the cross hole (labeled as **Extrude2_Cut** in the History Manager) as the feature to mirror.

    Middle-Click to Confirm your Selection

    Select the datum plane (labeled as Plane1 in the History Manager) that we created in the previous step as the mirror plane.

    NOTE: You should see the Features in the History Highlight in **BOLD** as you select them from the History Manager.

    <Middle-click> to complete the command.

15. Pick the **Auxiliary View** command from the Document Awared toolbar.

16. Change the display to **Shaded** if it is not.

17. In this final step for the Tube design we are going to round off all edges of the tube.

Pick the **Fillet** command from the **Shape** Tab.

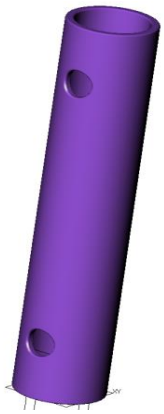
For edges to select, **<Right-click>** for the Selection Options Menu and Choose **Pick All**

Middle-Click to confirm your selection set.

Specify the fillet Radius to be **0.5mm** in the options form.

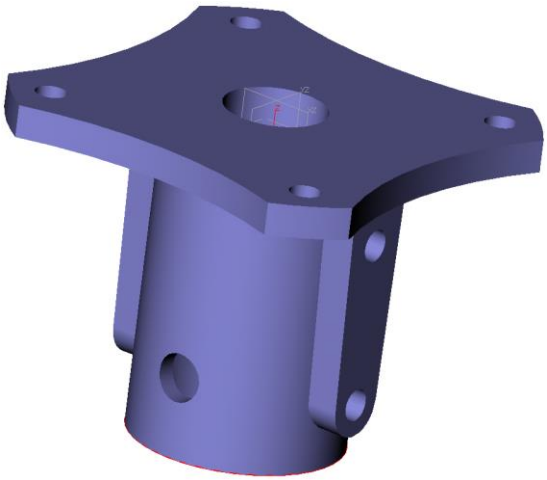
<Middle-click> or Pick **OK** to complete the Fillet Feature.

18. Save your File – This saves all of the objects in the file.
19. Exit Part when you are finished.



Your completed Tube should appear similar to the figure above.

The next part we are going to create is the Flange portion of our Ceiling Fan.



1. Create a new Part / Assembly object and name it "**Flange**". Pick New Assembly Mode.

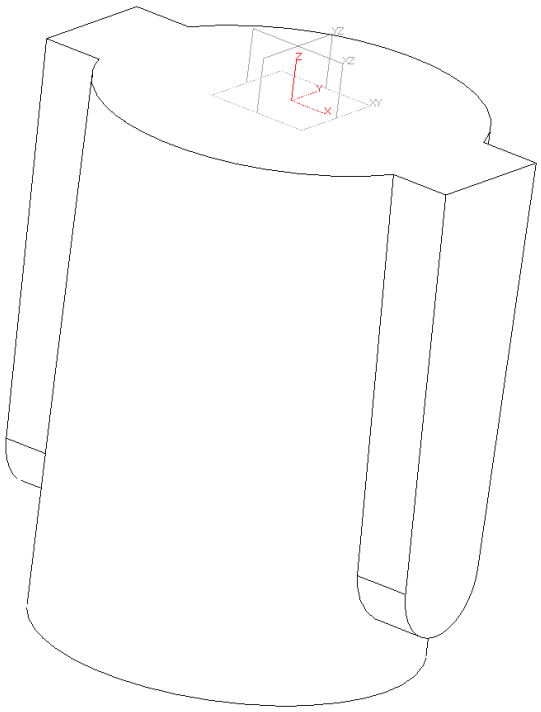
   You will be placed at the Part level of ZW3D.

2. You can quickly verify that you are indeed in a mm unit part by looking at the **screen zoom size** in the lower left of the display. If it says 200mm that is a big clue.

3. Pick the **Cylinder** command found on the Shape Tab.

   Fill in the required inputs in the Options Form as below:
   Center = 0,0,0;
   Diameter = 27;
   Length = -37.

Now let's create the 'ears' on the sides of the Flange.



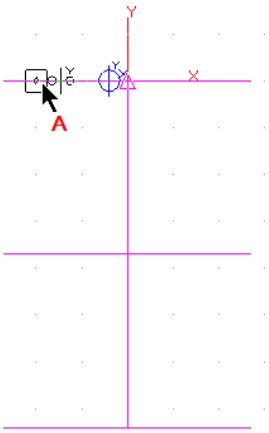
4. Right-Click and choose **Insert Sketch**.

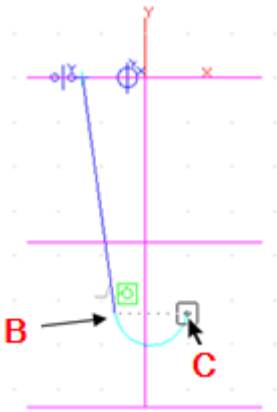
   Insert it on the YZ plane. You could pick the plane or type "YZ" at the Insertion Plane prompt.

5. Refer to the figure below for the sketch we'll be creating.

6. Pick the **Draw Tool .** Also found on the **Right-Click** Shortcut Menu as **Draw.**

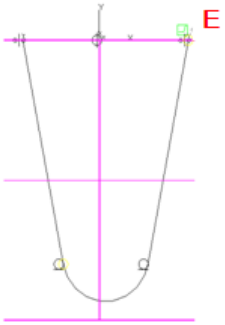
   <Right-click> and verify that the position/snap setting is set to **Default or Critical**. Relative or Absolute will give an undesired result.

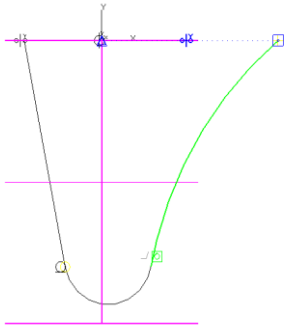


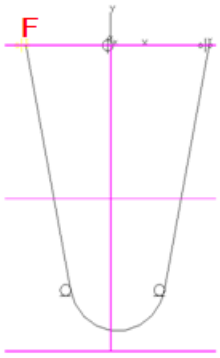


A: Pick 1 B: Pick 2 and 3(see note below) C: Pick 4

Note :ZW3D will prompt you that "Specify a point (<Alt> for curve) or pick start point to toggle mode." Toggles from connected line to Tangent arc and back to connected line.







E:Pick 5 and 6(see note below)

E: Pick 7, Close the profile

Note : Make sure when you choose Pick 5 that your curve is snapping to form a straight line. In the choice of 5, don't pick in the cylinder top place. When the line with dotted line shows that says it is a straight line. After finish sketch, the top left corner and top right corner to parallel constraint.

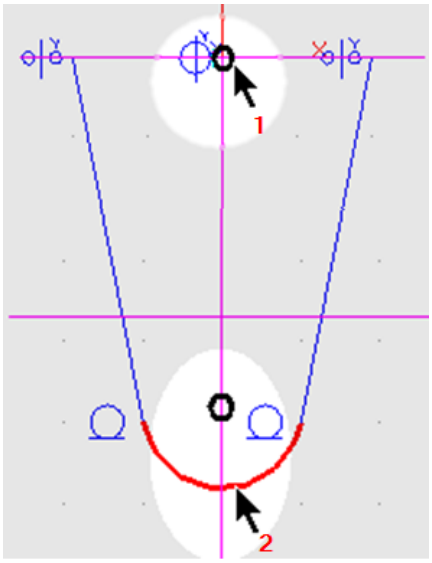
7. Let's continue constraining our sketch. Select the **Constraint** Tab.

   Each sketch contains a point locked or Anchored at Zero. We want the center of our arc to be aligned on the Y axis or in other word we want to constrain its motion along the X axis to be the same as the Point anchored at Zero.

   Choose the **Points Vertical Constraint** command.

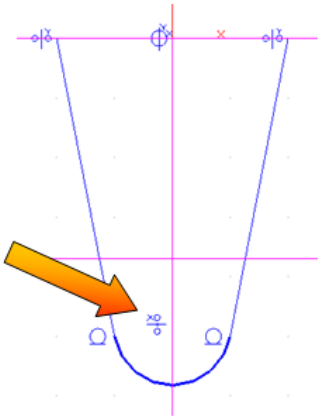
   Pick the base point (see Pick 1).

   Pick the Align Point (see Pick 2).



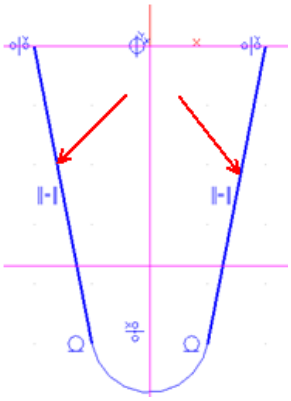
1: Pick 1, Base Point for Aligning X 2: Pick 2 Points to Constrain Vertical to Base Point

Result is the following.



Next we want to apply an **equal length constraint** between the two diagonal lines.

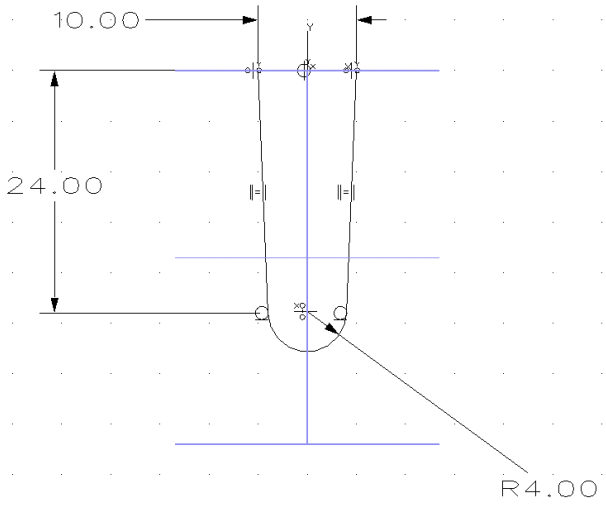
Pick Bold Line.



Solve Current Sketch.

Finally Dimension as shown below.

Consider using the **Auto Constrain** Tool and the **Dimension Tool**.



Solve Current sketch.

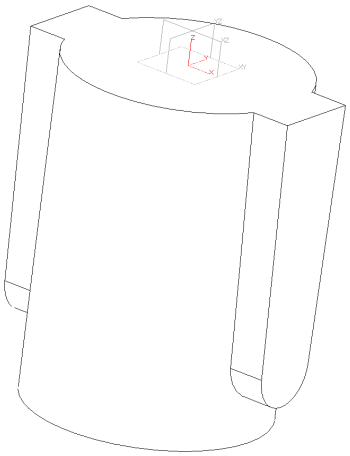
8. **<Right-click>** Exit Sketch.

9. Choose the **Extrude** Command select the sketch you just created for the profile and adjust the values as follows:

   Add Option 

   Specify Start: **17**
   Specify End: **-17**

   Your solid model should appear as shown in the figure below.

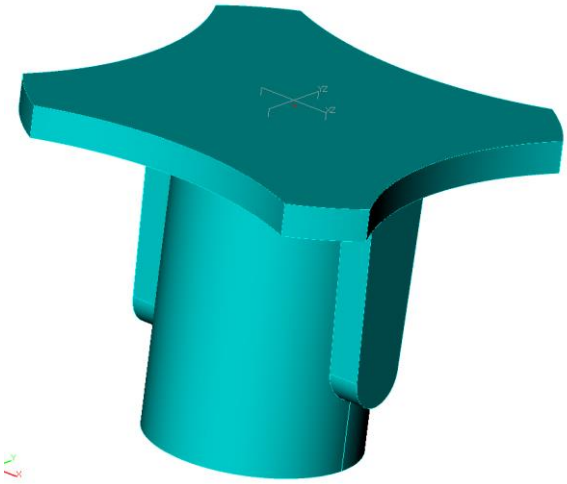


10. Let's start creating the collar on the top of our model.

    Pick the **Extrude** command.

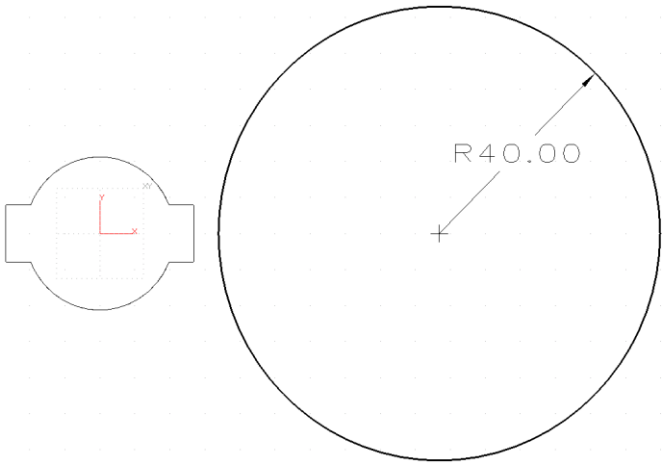
    <Middle-click> two (2) times to accept all of the defaults.

    You will be placed at the sketch level and on the XY datum plane.



11. Pick the **Center, boundary circle** option of the **Circle** command located on the **Sketch Tab**.

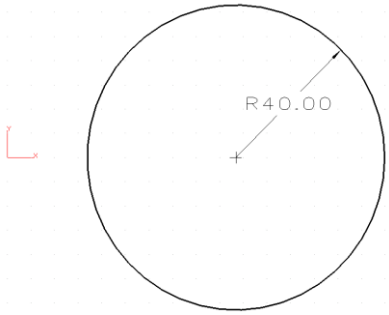
    Create a circle located at **61.5,0** with a radius of **40**



12. Notice in the above figure that the extrusion of the body appears along with the sketch. If our design contained a variety of components and shapes, we might have a difficult time visualizing our simple sketch to work on.

    From the View pull down, select the **Show Scope > Show Target** command to display only the active sketch.

    The part/assembly will disappear and only the currently active sketch indicated in the title bar should appear.



13. L et's make three copies of the circle.

    Pick the **Rotate** command from the Sketch Tab  .

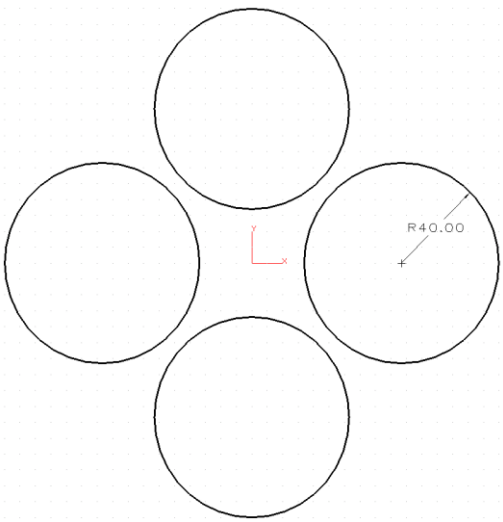
    For the entities to rotate pick the circle.

    Pick the **Copy** command input option.

    Enter **3** for the number of copies.

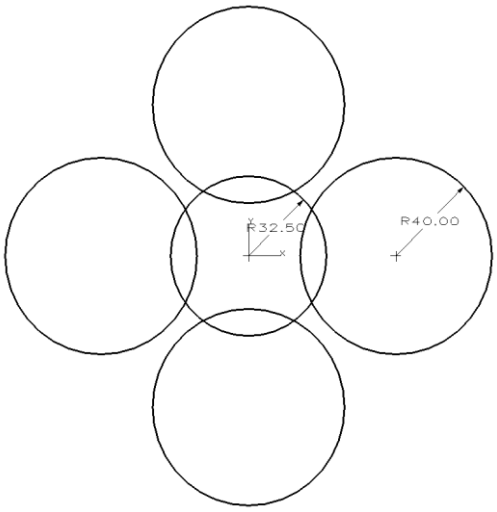
    Use **0,0** for the base rotation point

    Enter **90** for the rotation angle.



14. Pick the **Center, boundary circle** option of the **Circle** command located on the **Sketch Tab** again.

    Create a circle located at **0,0** (comma) and with a radius of **32.5**.



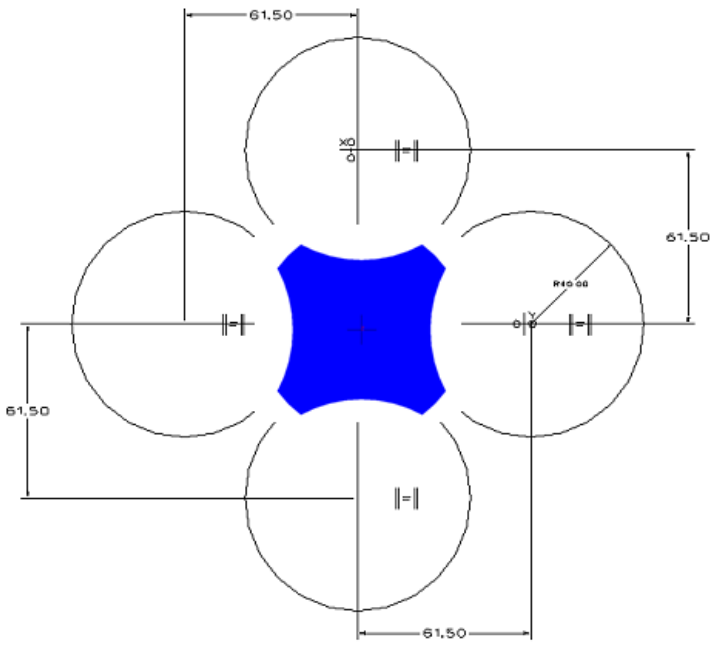
15. Make the 4 large outside circle equal to the base circle that has the 40mm Rad Dimension.

16. Select the Auto Constrain Command and Pick the Zero Point from the Sketch for the base point.

17. Finish this sketch by tracing  the portion of the circles we want as our final sketch rather than trimming away what we don't want.

    Pick the Trace Tool from the **Sketch Tab**.

    Pick on the portion of the circles to keep until your sketch is similar to the figure below.



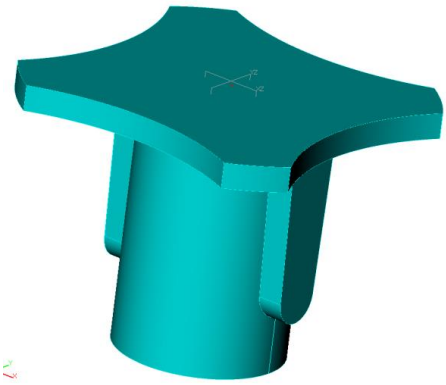
18. Verify your sketch is well defined by using the **Constraint Status** command.

19. Remember ZW3D can work with under over and well defined sketches.

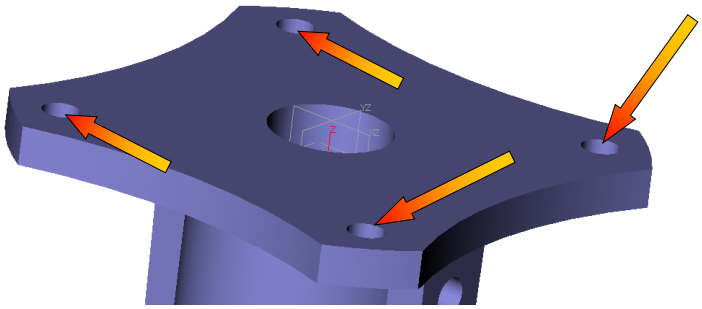
20. To inquire individual constraints simply double click the constraint or right-click it and pick inquire from the shortcut menu.

21. **Solve the current sketch** before we exit.

22. **<Right-click> Exit Sketch**. Make the extrusion 4 units long and ADD it as shown in the figure below.



23. Now we will create four mounting holes indicated in the image below.



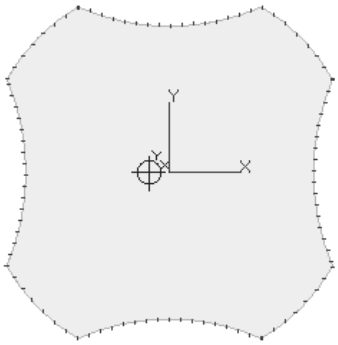
In order to easily establish the hole locations, we simply need to create a sketch that contains a line drawn from the center at the distance and angle for one of the holes. We will then create a circular pattern from the first hole.

Create a new sketch. **Right-Click > Insert Sketch.**

For your Sketch options turn **Reference edges** on as shown in the following figure.

Pick the top face of the Flange boss feature we just created for the sketch plane of the new profile.

You should notice red dotted lines on all edges of the flange boss feature. The dotted edges in the figure below are the reference edges.



24. Select **Show Scope > Show Target** from the **View** pull-down menu to get a clearer look at the reference edges that were generated from that face.

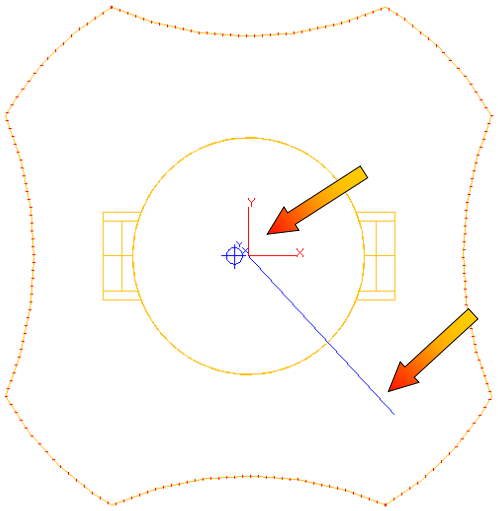
    Note: You can retrieve reference edges with the Reference geometry found on the Sketch Tab.

25. Pick the **Create line** command  .

    For the first point, pick the center of the circle or Pick the pre-established Zero Point.

    For the second point, pick close to where we want the first circle as shown in the figure below.

    Do not acquire any implied constraints as you pick the second point of the line. If you do, make sure you erase them. To erase constraints, simply highlight them and <Right-click> Erase.

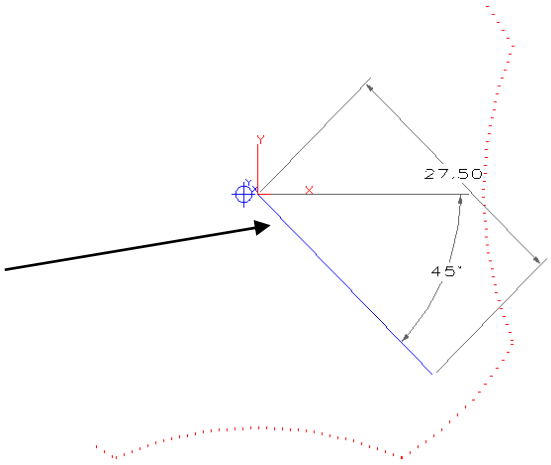


26. **Dimension** the entire sketch as shown below (the Line's Length is 27.5 and the angle is 45º)

    The **Angular Dimension** tool can be found on the **Constraint** Tab.

    Double click on each dimension and change their values to match the sketch below.

    Optional: You can verify if your sketch is well defined by using the **Constraint Status** command.

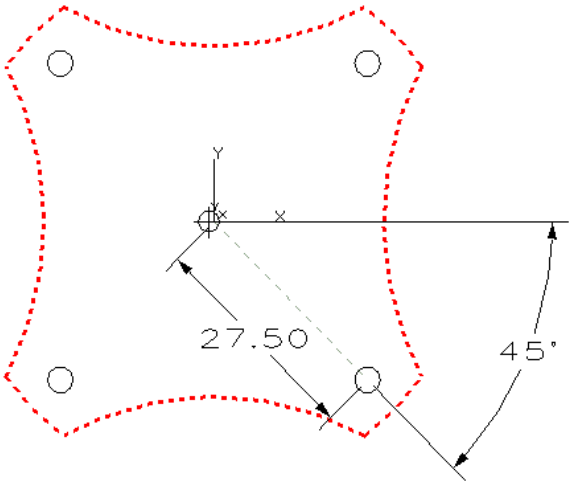


    Note: Pick near the end indicated in the figure for the angular dimension.

    Right click on the line and pick **toggle type**.

    From the **Geometry** tool bar, **insert a point** at the end of the line.

    From the **Tools** tool bar, rotate the point about the (0,0) point and make 3 copies.



Solve the current sketch.

27. <Right-click> Exit sketch.

28. Choose the **Hole Feature** command from the Shape Tab.

The Hole Options form allows you to create various types of Holes (Simple, Tapped, CBored, CSunk, Tapered, Spot Face), if you use ZW3D CAM it will automatically machine these holes according to the rules you establish.

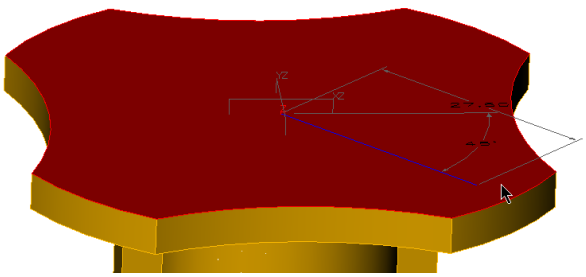
Note: One of the benefits of the Options form is that you can work thru it in any order simply go the option that you want to set and set it.

Set your hole options as follows.

Hole Type:	**Simple**
Dia (D1):	**4.0**
End:	**Thru-All**

Pick the Red button to the right of **Face** in the Options form. This will take us to the step of picking the part face to receive the hole (**Note:** ZW3D does not require that the face be planar).

Pick the Top Face of the Flange to be the face to receive the hole. See figure below for the face to select.



You are immediately prompted to "**Pick your hole locations**."

Right-click and pick **sketch** from near the bottom of the menu.

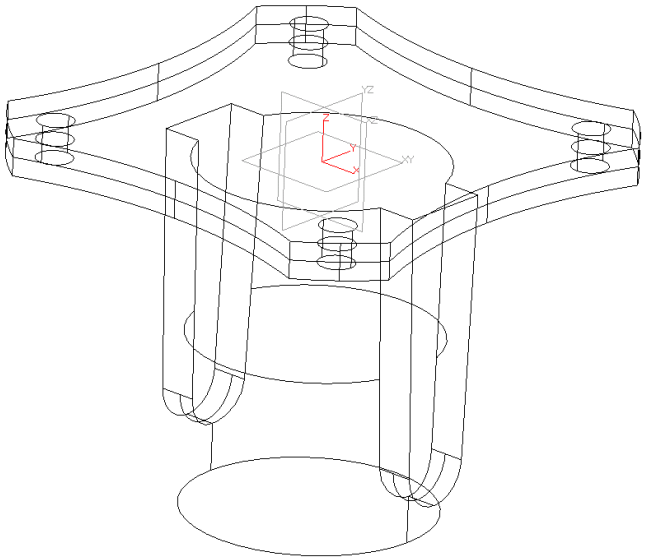
This is a special function that is only looking for points. Since we currently have four points in the sketch, we will get four holes.

Important: Your position/snap **<Right-Click>** menu will determine what geometry positions you can connect to  .

The manager window switches out of the command and shows your history. **Pick the last sketch**. The manager window will again switch back to the input manager. **Pick OK**

Blank the sketch  .

Your result should look like the following figure.



This is the BEST method for locating holes because it is the most flexible. You can edit the sketch to add or delete points/holes. **Regen** the history will update the model with the new number of holes.

Let's show you another technique…just because we can.

Step through the history.

Edit Sketch3  .

Erase the 3 additional points  .

Play the history to the end. You now have only one hole

Let's make the additional copies of the mounting hole again.

Pick the **Pattern** command from the **Shape Tab**.

Pick the **Circular Type** of Pattern.

For the entities to pattern or **Base**, pick the hole we just created (consider picking the hole command from the History Manager or set your pick filter to **Feature**)

For the **Direction** or axis of revolution, **<Right-click> Z axis.**

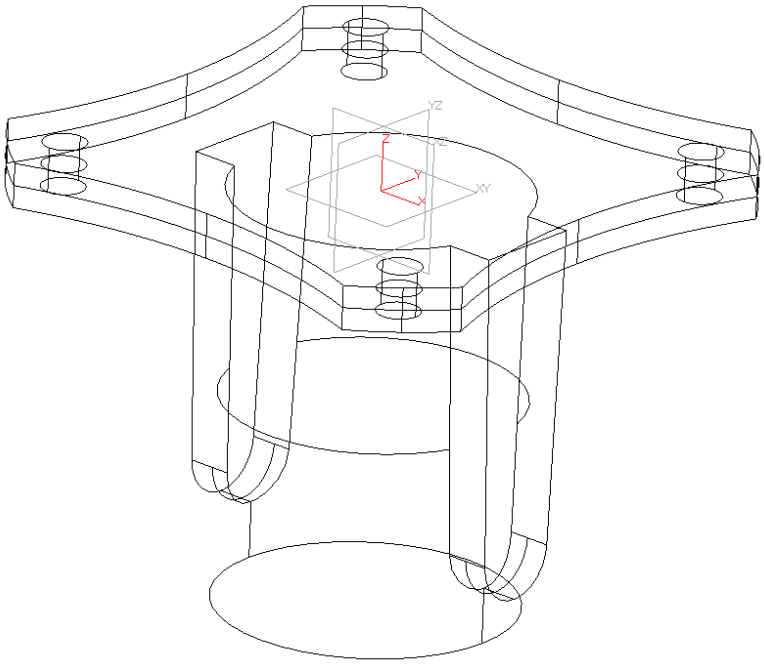
Remember: When prompted for direction you can choose existing linear elements or right-click to get the direction menu or click directly on the axes located at (0,0,0) or the axes in the lower left corner of the screen.

Enter **4** for the **Number** of features.

Since we picked a positive Z-axis lets enter **90** for the **Angle** between features.

NOTE: The additional prompts for Number and Spacing would produce additional copies along the direction axis. The Advanced Tab provides options for toggling off/on individual items from the pattern.

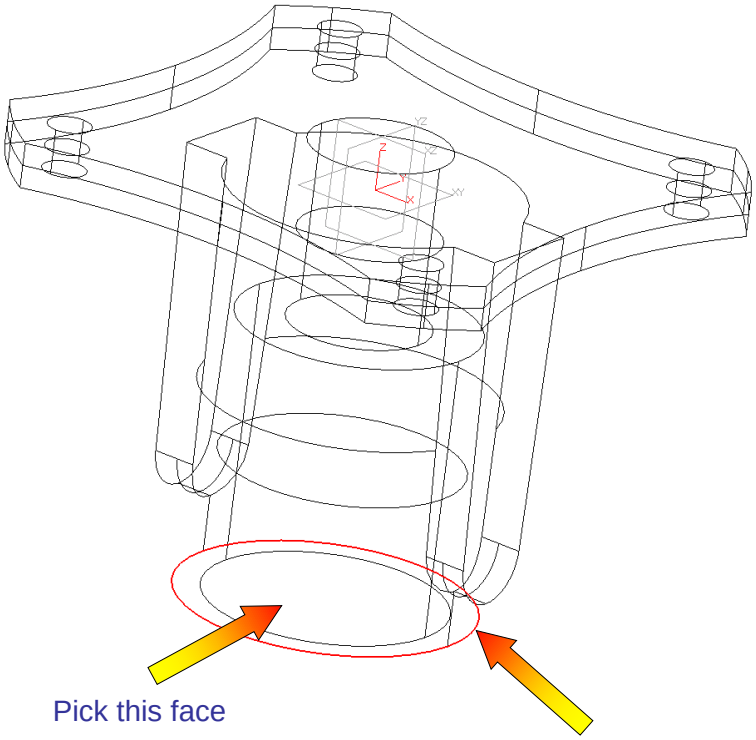
Pick **OK. See below.**



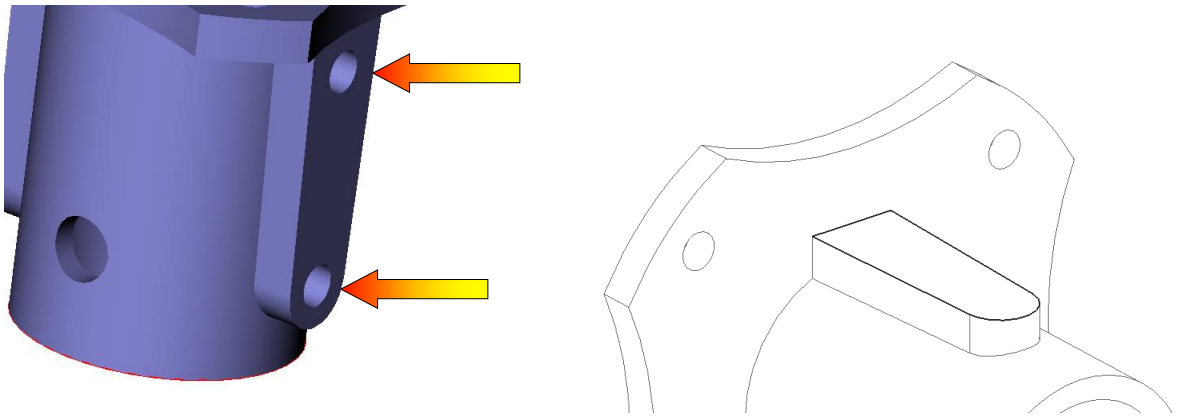
29. Let's create the thru hole and counter bore using the **Hole** command.

30. Pick the **Hole** command from the **Shape** Tab.

Hole type=Counter-Bore,
Pick the **Face** as indicated below,
For the **Location** to place holes, pick the edge as shown in the figure below. Make sure the **Critical** input option is set using the right-click menu.
Dia=13, **D2**=22,**H2**=26.
End= Thru-All.
Pick **OK** to complete the command.



Next we are going to create a pair of holes through the sides of the flanges as shown in a figure below.



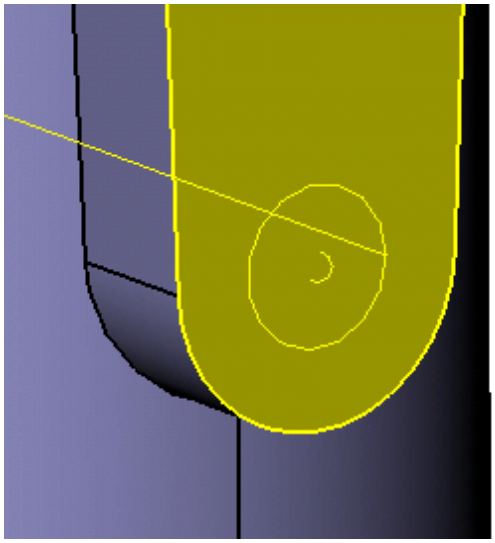
Rather than creating a sketch on this face to locate the holes, which you should already be able to do. We will introduce another technique.

31. Pick the **Hole** command from the **Shape** Tab.

Hole Type: **Simple** ~ Dia (D1): **4.0** ~ End **Thru-All.**

Pick the **Face** shown in yellow.

For the first hole **Location**, pick the **Critical** center of the arc as shown below. (Make sure the Critical input option is set using the right-click menu, and then pick the arc to select its center)



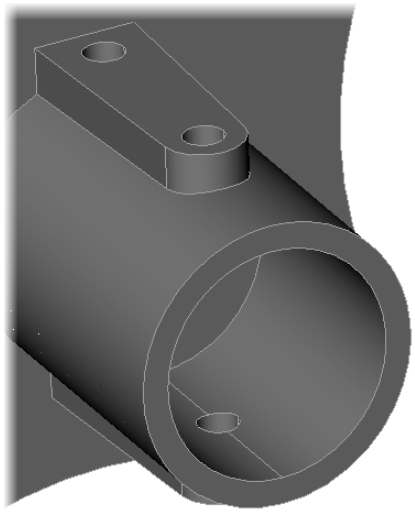
The next hole we want to position **20**mm **Offset** from the first in the **Z** direction.

The **Offset** position is found on the **<Right Click>** Input Options menu.

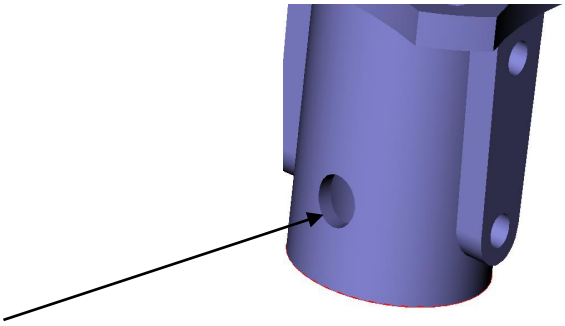
After selecting **Offset**, pick the **Critical** center of the arc again as the **Ref Point**.

Choose the XY plane for **Plane,** and .input 20 for **Z Offset**.

Pick **OK.** Results.



32. Next we are going to create a cross hole in the cylinder portion of the flange.



Try creating this hole without using the instructions simply look at the picture for the dimensional data and orientation.

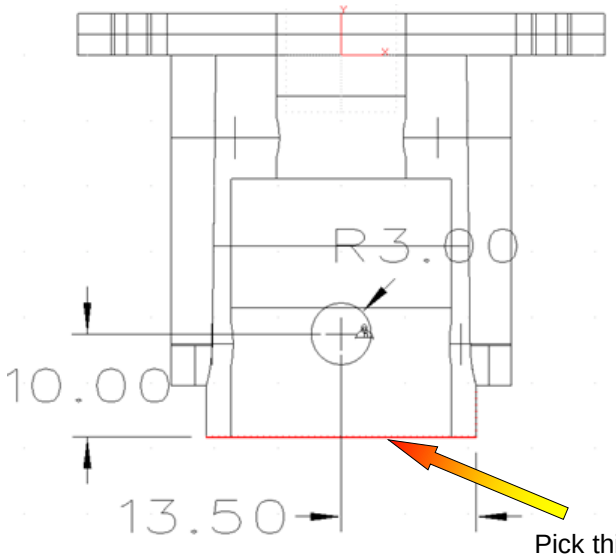
33. <Right-click> Insert Sketch

Pick the XZ Plane or type “XZ” at the **Insertion Plane** prompt. If you cannot select the XZ plane, make sure the pick filter, located on the Document Awared toolbar, is set to **ALL**.

Accept all defaults and complete the command.

34. Pick the **Reference geometry** (Curve) command located in the **Sketch Tab**.

Pick the 2 edges of the flange as shown in the following figure.



Pick this edges to project reference geometry for dimensioning the sketch.

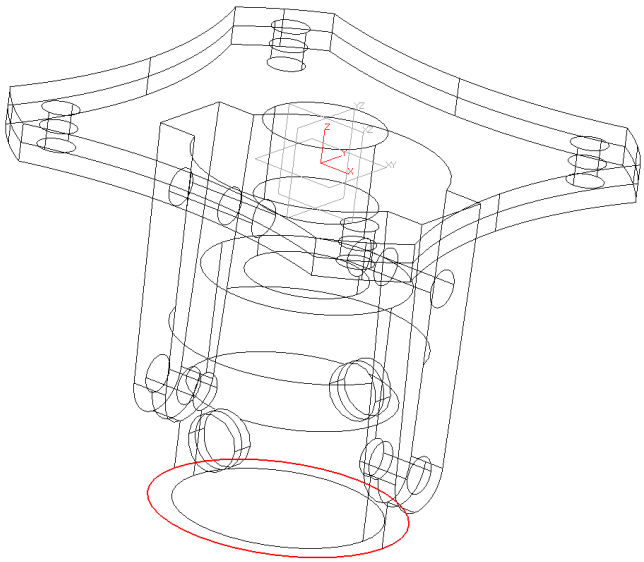
35. Create the circle as shown in the figure above.

36. Fully constrain the sketch as shown in the figure above.

*Note: Try another technique for constraining the circle to the center of the part. Erase the 13.5 Dimension and the vertical reference line. Apply a midpoint constraint in the **'X' direction only**. (Look under the Anchor constraint icon.) Using this technique, if the diameter of the flange changes, we don't have to remember to edit this sketch also.*

37. **Solve the current sketch and then <Right-click> Exit Sketch** after it is well defined.

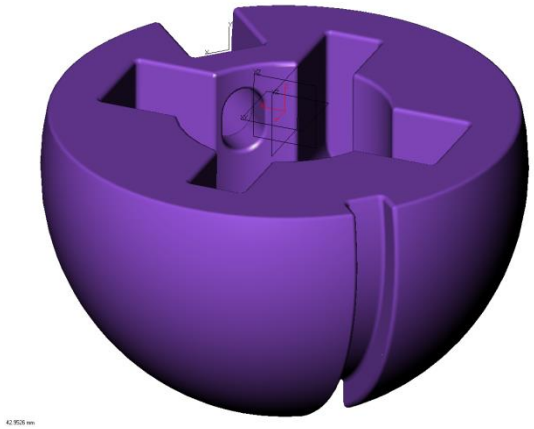
38. **Extrude and Remove** the circle through both sides of the cylinder as shown in the figure below.



39. Pick the **File/Save** command.

40. Use the **Exit part** command

Let's build our next part for the Ceiling Fan, "The Support Ball".



1. From in **My_Fan.Z3**, create a new Part / Assembly object and name it "**Support_Ball**"**.** Pick New Assembly Mode.

   You will be placed at the Part level of ZW3D.

2. Pick the **Sphere** command from the **Shape** Tab.
   For the Sphere center point pick at **0,0,0** or type ",," or just "0"".
   For the Sphere Diameter type **50 <ENTER>**.

3. Pick **OK**

4. Let's move the sphere down 6 mm so we can slice the top of it off.
   Pick the **Move entities along direction** command within the **Move** command under the **Shape** Tab.
   Pick the sphere for the entities to Move.
   <Right-click> **-Z Axis** for the move direction. (Notice the sphere begins to drag on the screen.)
   Specify **6** for the distance to move and complete the command

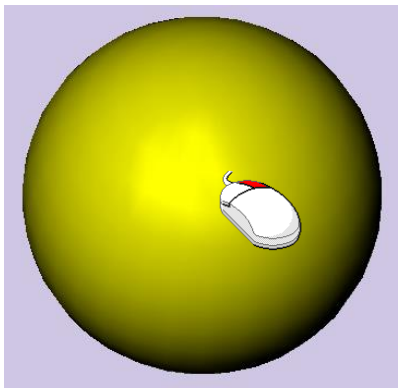
   Important: There is an input for Angle. This allows you to rotate around the Direction Axis. If you simply want to rotate a shape around an axis, simply use Move Along Direction and enter Zero for the distance and specify your rotation angle.

5. Next we have to cut away the top portion of the sphere.

   Set your **Pick Filter** to **Shape**.

   Hightlight the Sphere and <Right-Click>. The shortcut command that are specific to sphere will appear.

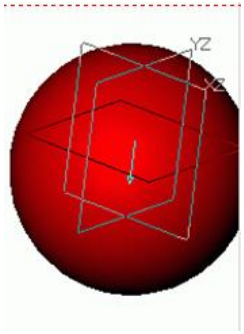
   Choose **Trim**



Select the XY Datum for the Trimming

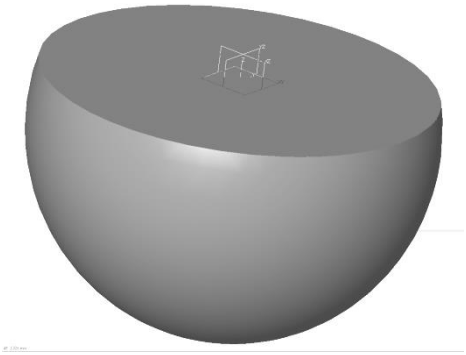
Remember: it is often easier to pick the datum from the **History Manager** (Ctrl + H).

Check **Flip side to keep** in the options form. You will see the datum normal arrow flip to the down direction. The arrow points at the part to keep.



Pick **OK**.

Your model should appear as shown in the figure to the right.



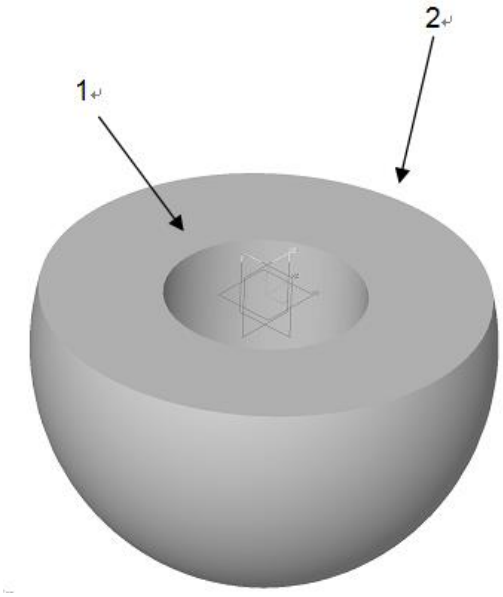
6. In this next step we are going to create a clearance hole through the sphere as shown in the figure below. The hole will be used later to assemble the **Tube** created earlier.

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

   Change the settings in the form as shown below.

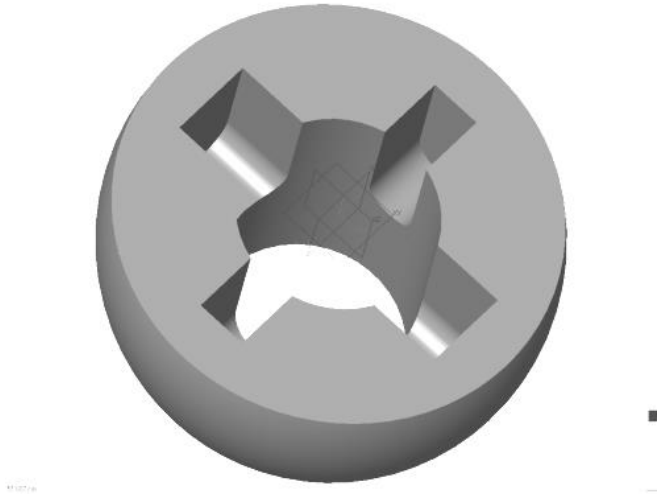
Hole type: Simple, **Dia**=22, **End**= Thru-All.

Refer to the figure below for picking geometry.



1: Pick this face as the face to place the hole on. 2: RMB > Critical and pick this edge to select the center of the face.

7. Next we are going to create a cross-shaped feature as shown below.



Pick the **Extrude** command.

Pick the **Remove** Option.

<Middle-click> two (2) times to accept all of the defaults.
This will put you in sketch on the XY datum.

8. Pick the **rectangle** command from the **Sketch** Tab  .

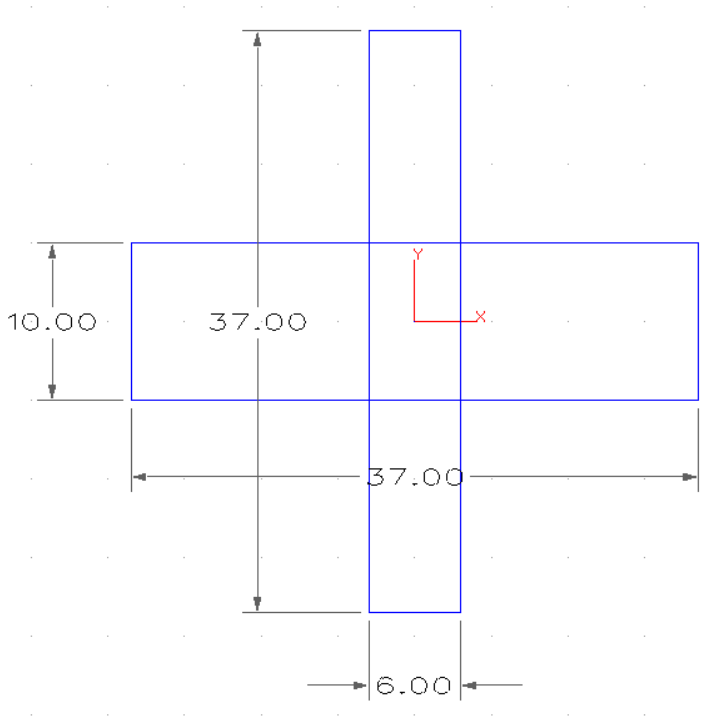
   Left click the Origin point to place it at (0,0).

   See figure for reference.

   Edit the dimensions to match one of the rectangles shown.

   Repeat the command, middle click for the placement and edit the dimensions to match the second rectangle.

   Note: Ready sketches are well defined. Modify a dimension value to test that rectangles stay centered.



9. Pick the **Show Target** command. Only the rectangles on the active sketch plane should appear on the screen now.

10. Pick the **Hide All Constraints** to hide the current geometric constrains.

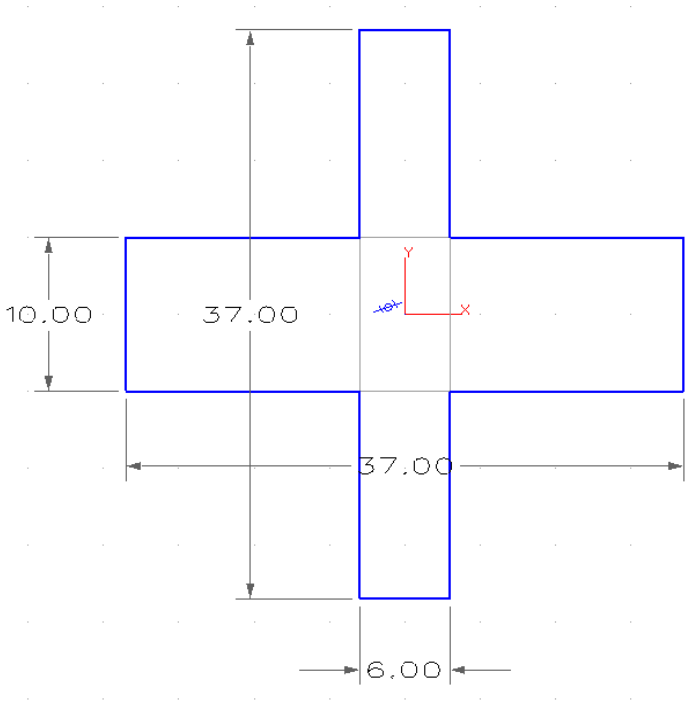
11. Pick **Solve Current Sketch**.

In our next step we are going to trace the areas of the sketch we want; rather than trimming away what we don't want. This will allow the current set of dimensions and geometric constraints to stay intact. If we were to trim this shape we would likely need to apply additional constraints to re-establish this design intent.

12. Choose **Trace Profile** from the **Sketch** Tab.

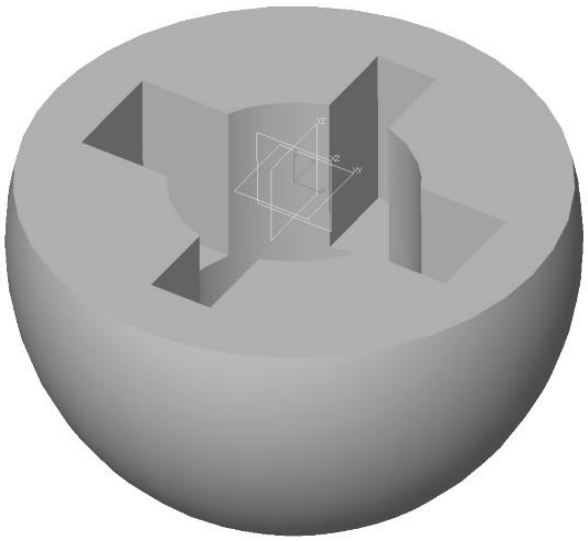
    Pick portions of the sketch that you want to trace or keep for your extrude remove.

    Your sketch should now appear as shown in the figure below.



13. <Right-click> **Exit Sketch.**

    Make sure that the **Remove** option is picked.
    Specify the **Start as 0**
    Specify the **End as 15**
    No Draft Angle.
    Your part should appear as shown in the figure below.



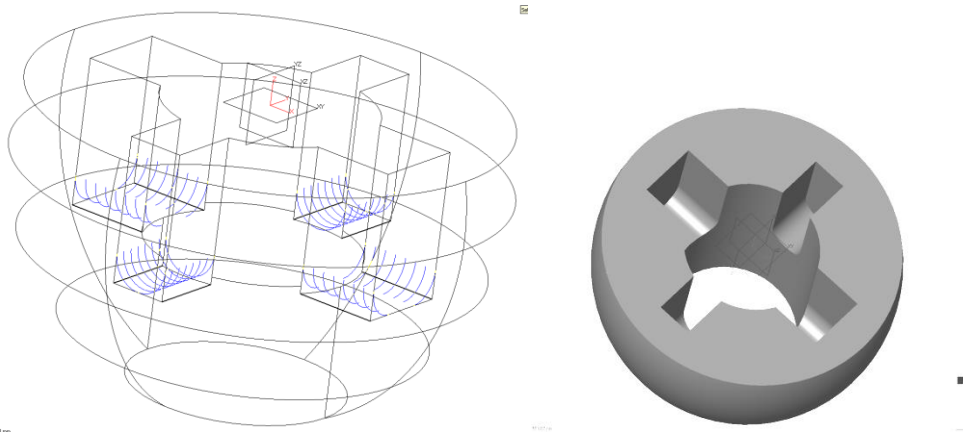
14. The next thing we need to do is round off some of the edges on the inside of the extrusion we just created.

    Pick the **Fillet** command from the **Shape** Tab.

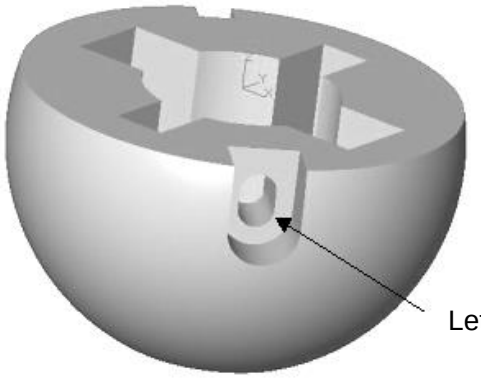
    Select the 8 inside edges of the cross-slot feature as shown in the left figure below.

    Specify a radius value of **3**

    Your model should appear as shown in the right figure below.

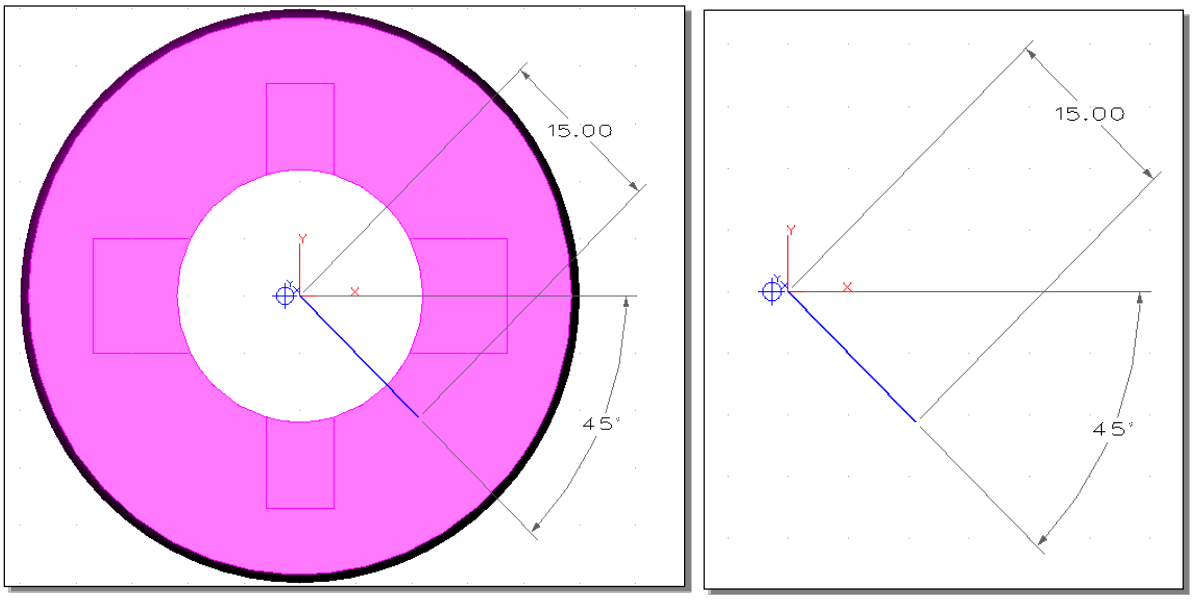


15. The next set of features we are going to design are shown in the figure below. They are a cutout and a slot for a setscrew. Since these features are oriented at a user-defined angle, we will have to create a datum plane to place the sketch on.



16. Let's start by creating a sketch. We will construct a datum plane normal or perpendicular to the sketch. This is required to build these features at their correct orientation.

    Create the sketch in the figure below on the XY Plane.



Show All.

Show Target .

Exit Sketch

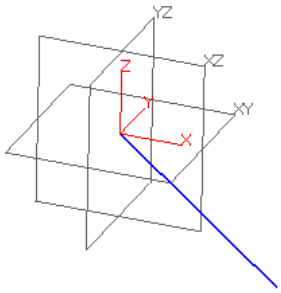
17. Blank the Support Ball shape so that only the sketch is showing.

    Pick the **Blank** command  .

    Set your **Pick Filter** to **Shape**.

    <Right-Click> **Pick All**.

    Choose **OK** or <Middle-Click>. (You should only see your sketch on the screen).



18. Turn the Dimensions off with the **Dimension**  **.**

    You will now only see the line from the sketch on the screen

19. **<Right-click> Insert Datum.**

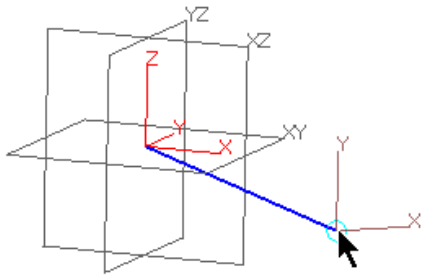
We will use the default intelligent datum option to create a datum normal to the sketch line.

<Right-click> and select **Critical.**

Then pick the end of the line as shown in the figure below and pick **OK** from the Options form.

Note: There are optional inputs for angles, offset and alternate orientation of X Y and Z axis.

You can now **Unblank** the shape using the appropriate icon on the toolbar.



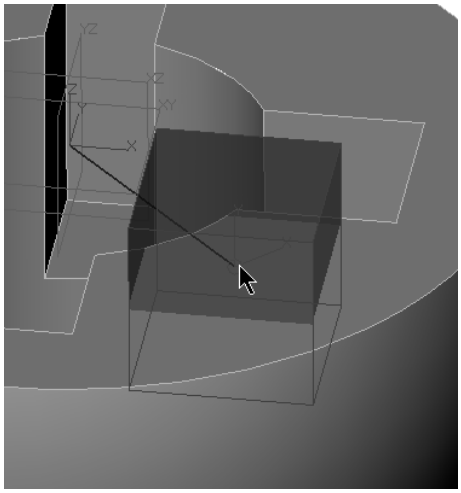
20. We will begin by creating another solid shape that we will later remove from the current shape.

Pick the **Block** command. Use the **Base** option.

1st Point: Place the Center of the Block at the end of the sketch line (Zero of our new datum) see the following figure.

2nd Point: Just pick somewhere on the screen.
Skip down to the Length, Width, Height inputs and type in the following values: **X = 8, Y=22, Z=30**

Then Pick the **Optional input for the Align Plane** and Pick the Datum you created in the previous step. (Reminder: it may be easier to pick this datum from the History Manager).



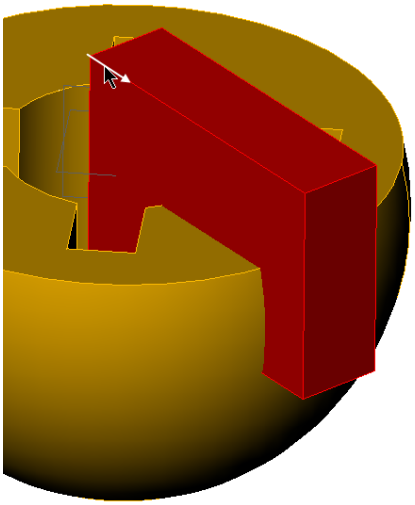
21. Next we will move this block into position using the **Move Along a Direction** Command within the **Move geometry** command under the Shape Tab.

    Pick the Block as the Entity to Move.

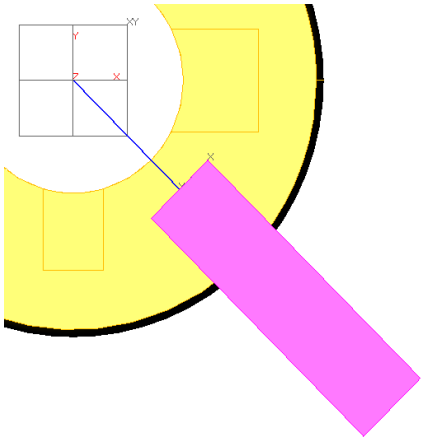
    For direction pick the edge indicated in the following image.

    The Distance is **15**.

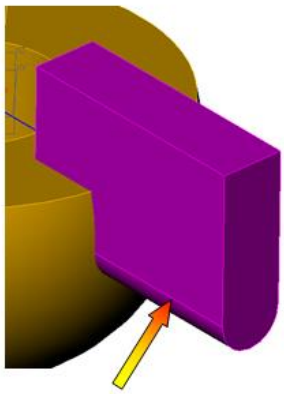
    Pick **OK** when finished.



Your results from a Top View  should look like the following figure.



22. Pick the **Fillet** command from the **Shape** Tab. Fillet the bottom edges of the block we just created as shown in the figure below. Specify a Radius of **4.**



23. Let's remove the block with the fillets from the Ball to create a cutout for the setscrew.

    Set your Entity Filter (Shift-Right-Mouse) to Shape.

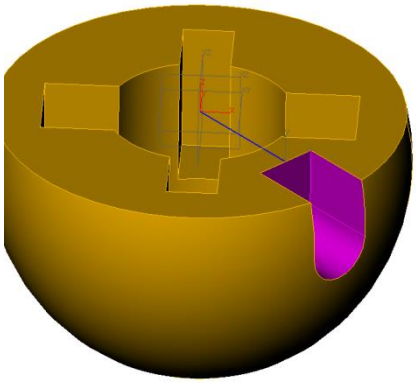
    Select the **Ball Shape and <Right-Click>** combination

    Choose **Remove** from the Shape shortcut menu.

    Then Pick the Block with the fillets as the shape to be removed

    Pick **OK** from the Options form.

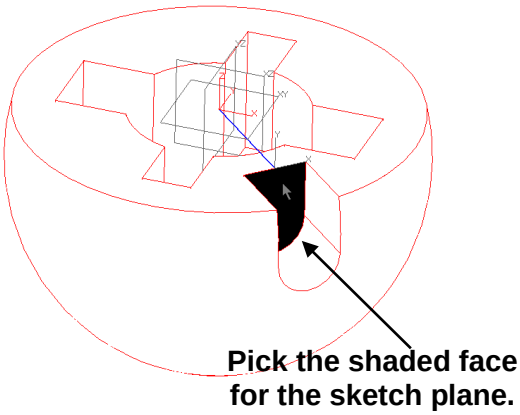
    Results.



24. Let's create the slot cutout through the back of the feature we just created.

    Pick the **Extrude** command from the **Shape** Tab.
    Middle-Click to indicate that you want to create a sketch.
    Check the **Reference Face Edges** in the Options form.
    Pick the back face from the cutout you just made as the insertion plane.

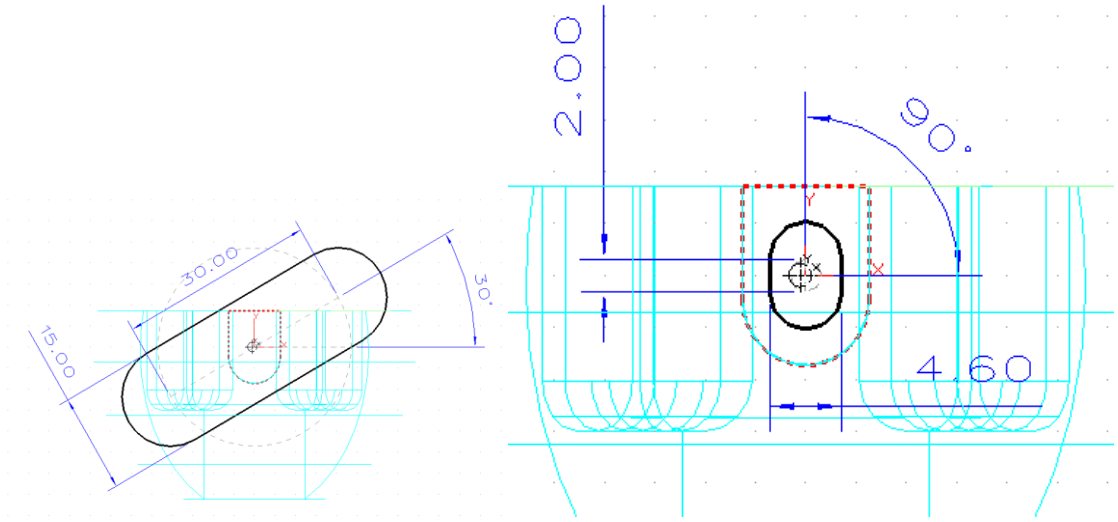
    See the figure below for reference.



25. Pick the **Rotated Slot** command from the ReadySketch panel of **Sketch** Tab  .

    Left-click to locate it at (0,0). See the figure left below.

26. Edit the dimensions as shown below. See the figure right below.



27. <Right-click> **Exit Sketch.**

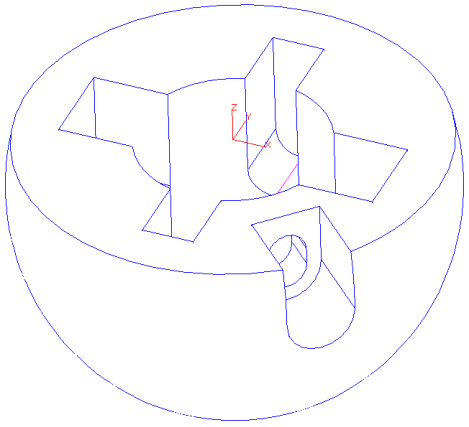
28. ZW3D remembers that you started this step with the Extrude command and will respond with prompts accordingly.

    Pick the **Remove** option.

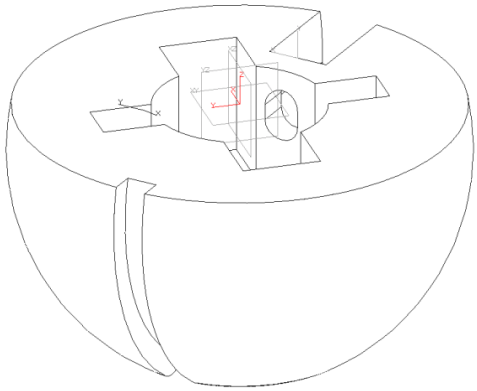
    Specify Zero for the Start.

    For the end, **<Right-click> To Point** and pick the origin at 0,0,0.

    No Draft Angle.



29. In these final few steps for the Support Ball design we need to cut a groove that will be used as a guide after the Fan has been assembled. We will work with existing geometry in the model to generate a wireframe curve that we'll use as the sweep path for our groove. Remember, ZW3D is a hybrid modeling system, which means we can use wireframe geometry, surfaces and solids, in one model to build the necessary features.



Let's begin by extracting a curve from the Support Ball to use as a sweep path for the groove, and rotate it into position.

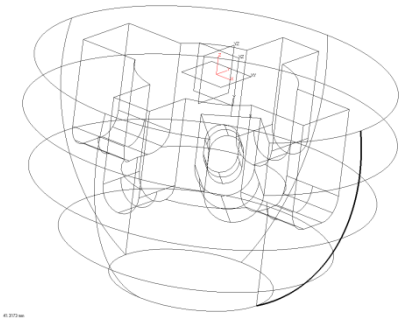
Go to **Attributes > Line** pull down menu and set the line color to **green** and the width to the **4th choice down** the list. See figure below for reference.

Pick **OK** to complete the command.

Nothing will change on the screen from this command. The affects from this command will appear with the following commands.

30. Pick the **curve from edge** command from the Wireframe Tab  .

    Pick the edge of the sphere shown in the figure below to extract the wireframe.



31. Let's rotate the curve into position.

    Pick the **Move along direction** command within the **Move** command under the **Shape** Tab.

    Pick the curve you created in the previous step.

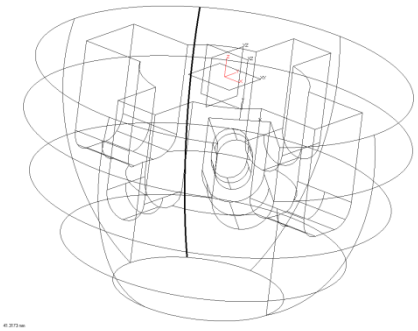
    Optionally, you can set the **Pick Filter** to "**Curve**" if it helps.

    Move direction: **<Right-click> Z axis**

    Move Distance: **0**

Rotation angle: **135**

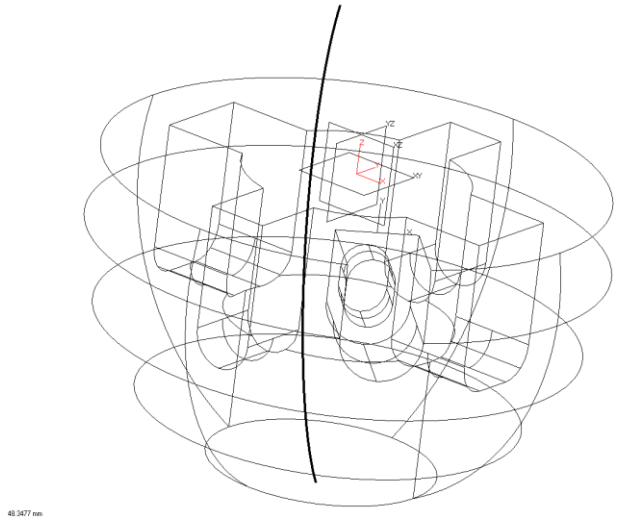
OK.



32. Next we need to make the curve a little longer in both directions so the profile we sweep along it will pass through the top and bottom of the Support Ball.

    Pick the **Trim/Extend Curve** command from the **Wireframe** Tab  .
    Pick the curve we extracted in the previous step.
    Check the Extend both ends option in the form.
    Drag your cursor until the length is near 10 or type 10 into the form.

    Your solid model should appear as shown in the following figure.



33. Now that we have the curve to use as a sweep path, we need to create a datum plane at the end of that curve so we can create the sketch to sweep along it.

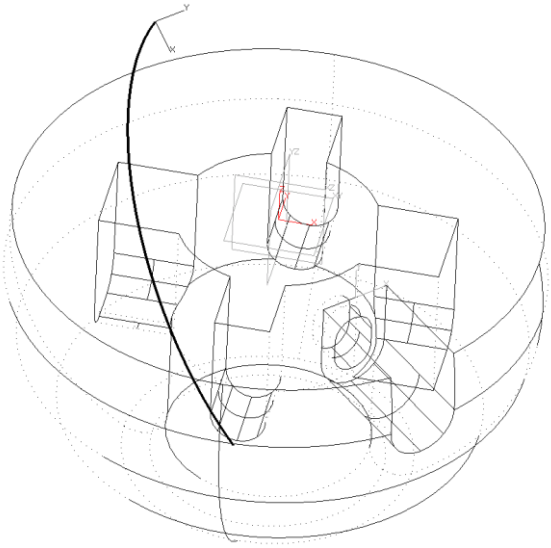
    <Right-click>**Insert Datum.**

    For reference geometry, <Right-click> **Critical** and pick the top end of the curve we just created relative to the figure below.

    Pick the "**X Point**" button in the Options form.

Pick (0,0,0) to force the X axis of your new datum to point radially from the center of the sphere.

Your solid model should appear as shown in the figure below.



34. <Right-click> **Insert Sketch.**

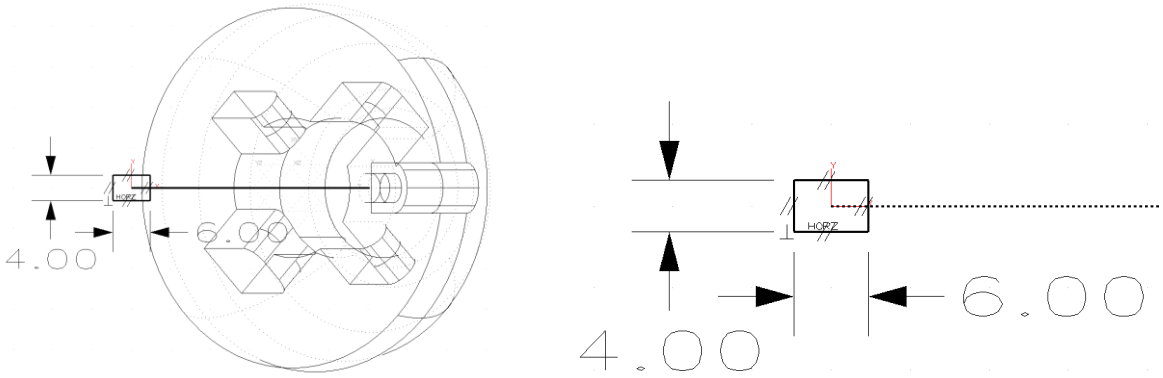
    Pick the datum plane you created in the previous step and accept the defaults.

35. Draw and fully constrain the sketch shown in the figure below.

    Pick the **rectangle** command from the **Sketch** Tab.

    Specify the center of the rectangle to be (0,0).

36. Pick **View > Show Scope > Show Target** from the **View** pull down menu .



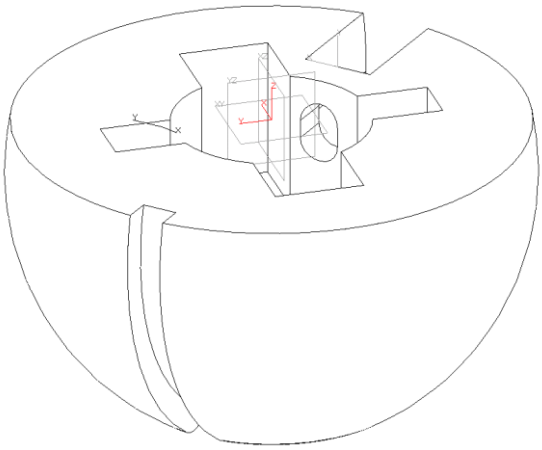
37. <Right-click> **Exit Sketch.**

38. Pick the **Sweep** command from the **Shape** Tab  .

Choose the **Remove** Option  .

For the profile, pick the rectangular sketch.

For the path, pick the curve.



39. Erase the sweep curve and the datum plane located at the end of the curve. Your model should appear as shown in the figure below.

40. Let's round off all of the edges on the Support Ball. Pick the **Fillet** command.

    For the edges to fillet, <Right-click> **Pick All.**

    Make the fillet radius **.5** mm.

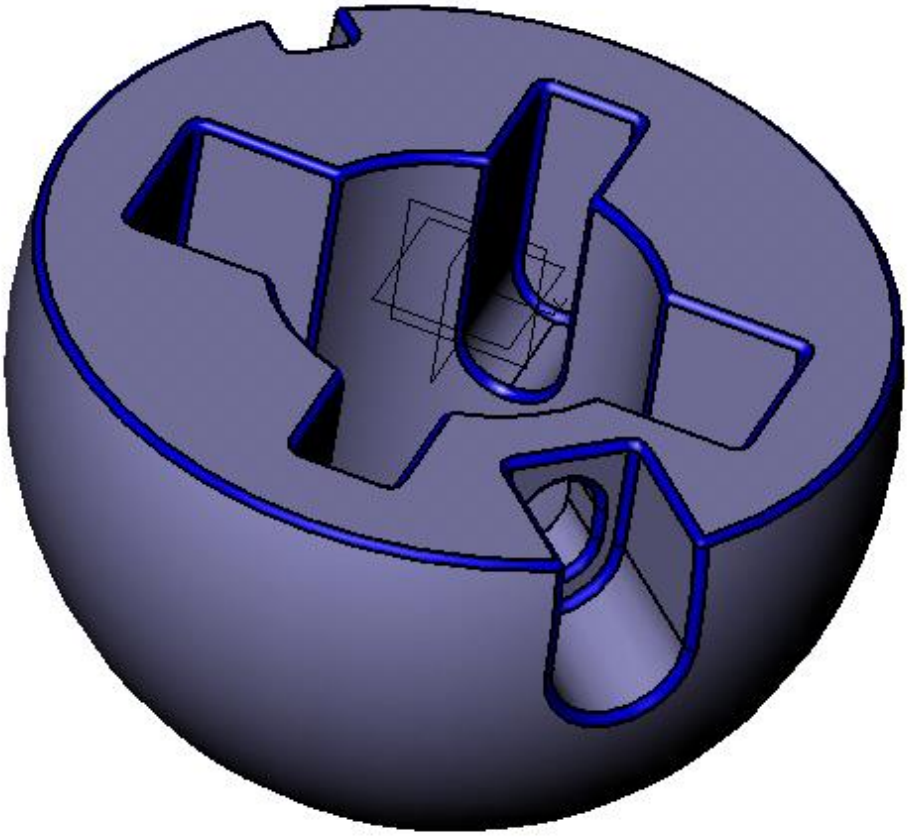
41. Change your **Pick Filter** to **Shape**. <Right-click> over the object and pick **Attribute**. Change the color of the Support Ball to ZW3D Purple.

42. Hold down the F3 key. Hold Right Mouse Button, RMB, down and slide side to side to spin your part about the Z-axis.

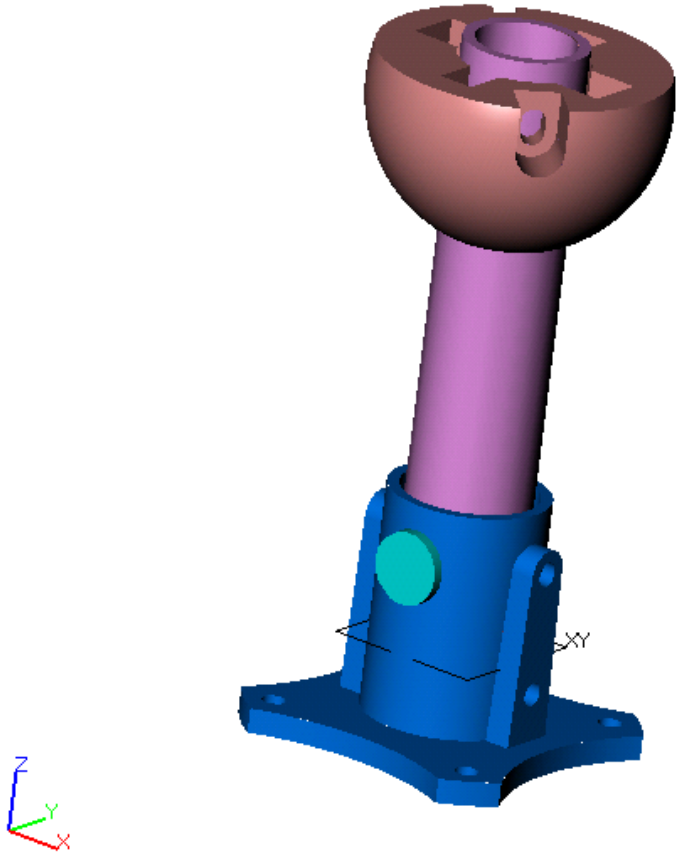
43. Save your model.

    Finished.

    Your completed model should appear as shown in the figure below.



ASSEMBLY - Bottom up approach



Since all the individual parts for our fan are already constructed, we are going to show you the **'Bottom Up'** approach of assembling components. An example of the '**Top Down'** approach will be shown later.

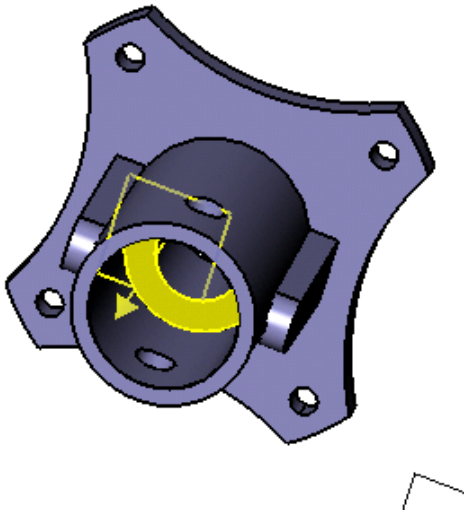
1. Create a new Part / Assembly object and name it "**Fan_Assembly**". Pick New Assembly Mode.

2. Insert the first component into the assembly.

- Use the RMB (Right Mouse Button), and Select the Insert Component command.
- Place it anywhere on the screen. **(NOT 0,0,0).**
- Select the 'Flange' component from the list of objects.

3. We are now going to align the flange with the world XYZ Datum planes.

Key to successful aligning: Put your **filter on face** and **RMB on-entity.**



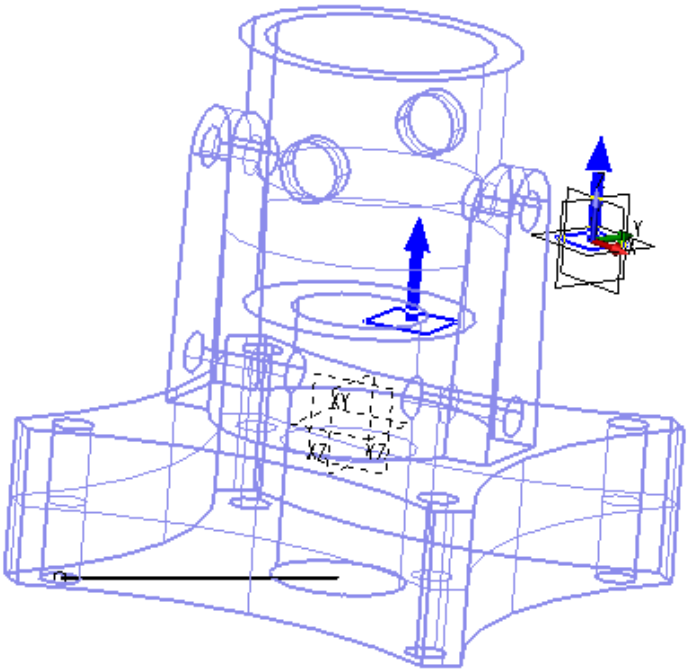
- It is not necessary to pick an alignment icon from the optional inputs section of the form. If you pick a cylindrical surface, ZW3D assumes a **concentric** constraint;
  a flat face, we assume **coincident.**
- Select the bottom face of the counter bore as shown.
- Look for the rectangular constraint icon before picking.
- Select the XY plane. Notice the component moves.
- Select the **same facing** option. Notice the component reverses its direction.

Don't hit OK yet. You will be picking 3 pairs of faces or planes to fully constrain most objects.

If you do hit OK prior to completing the alignments, just **Right-click > Align**.

You can switch between opposite and same facing until you pick the next face.

Notice that the centerline of the component is not aligned with the Z axis of the world plane. Also notice that there is now a Planar constraint shown.

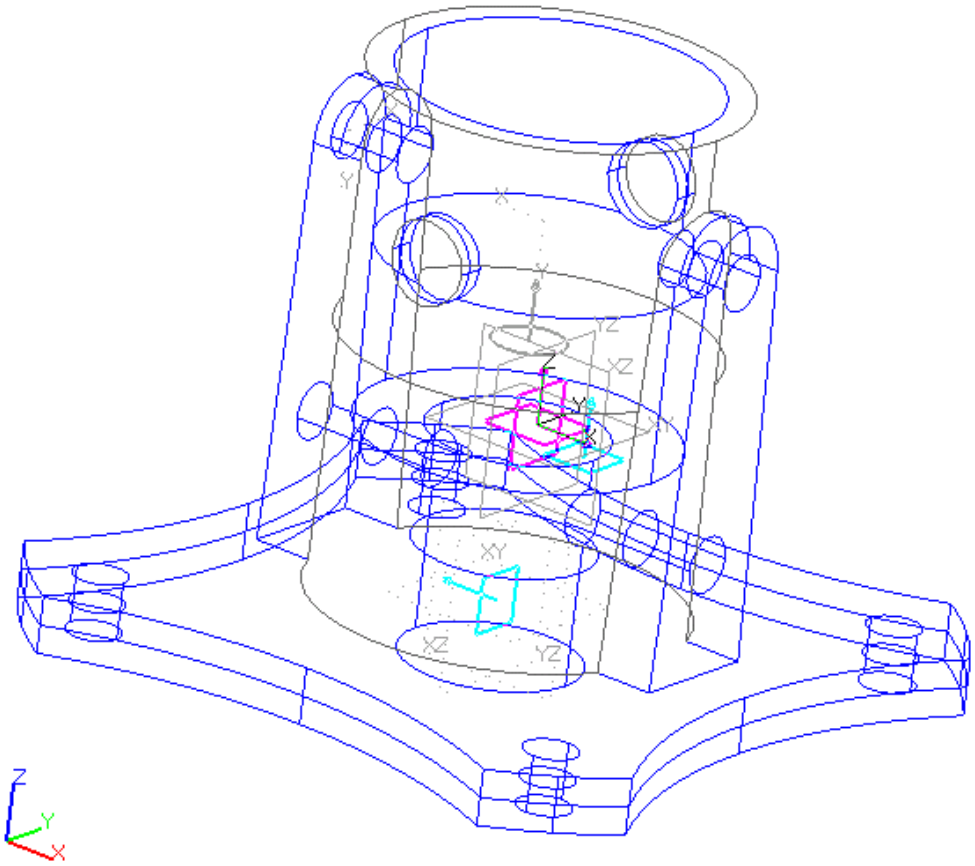


We now need to be able to see the datum planes of the component.

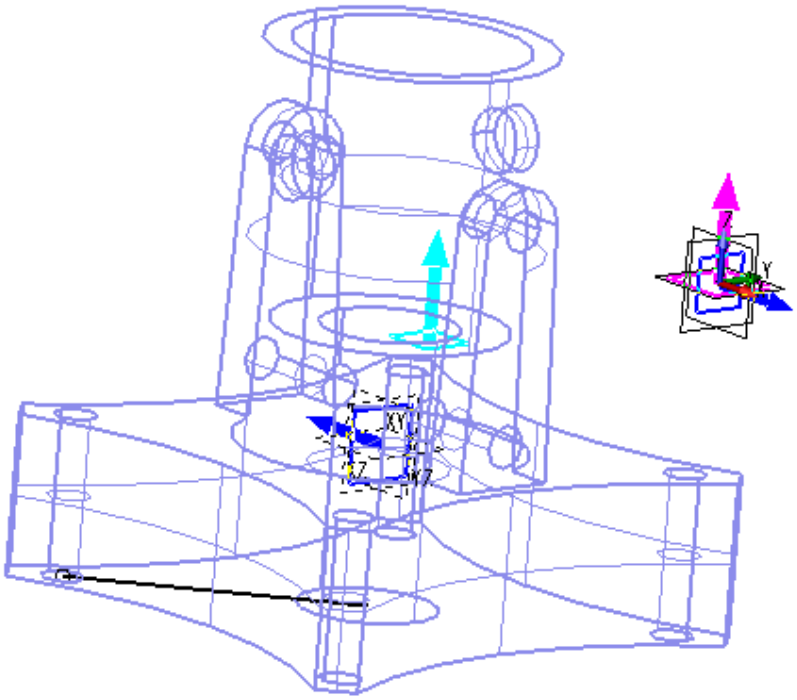
4. Pick the **View > External Datums** command.

5. Let's create our second alignment constraint.

- Select the component's **YZ** plane.
- Select the **Opposite facing** option.
- Select the world **YZ** plane.



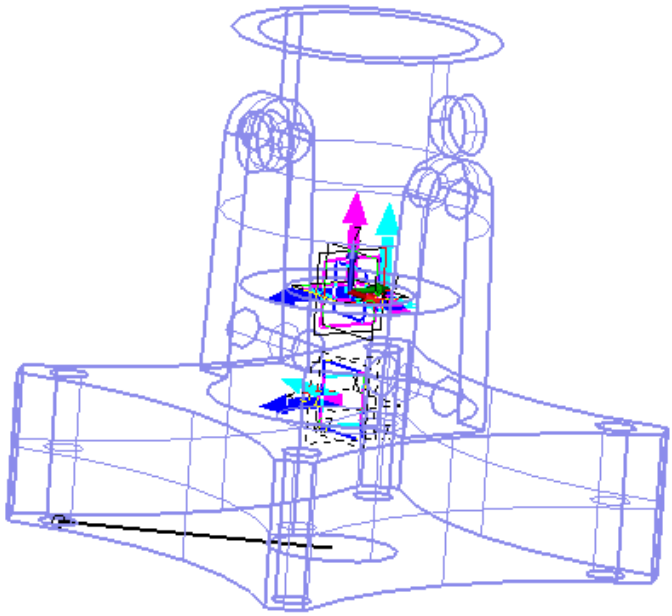
Your component should now look like the training image.



6. Third and final alignment constraint.

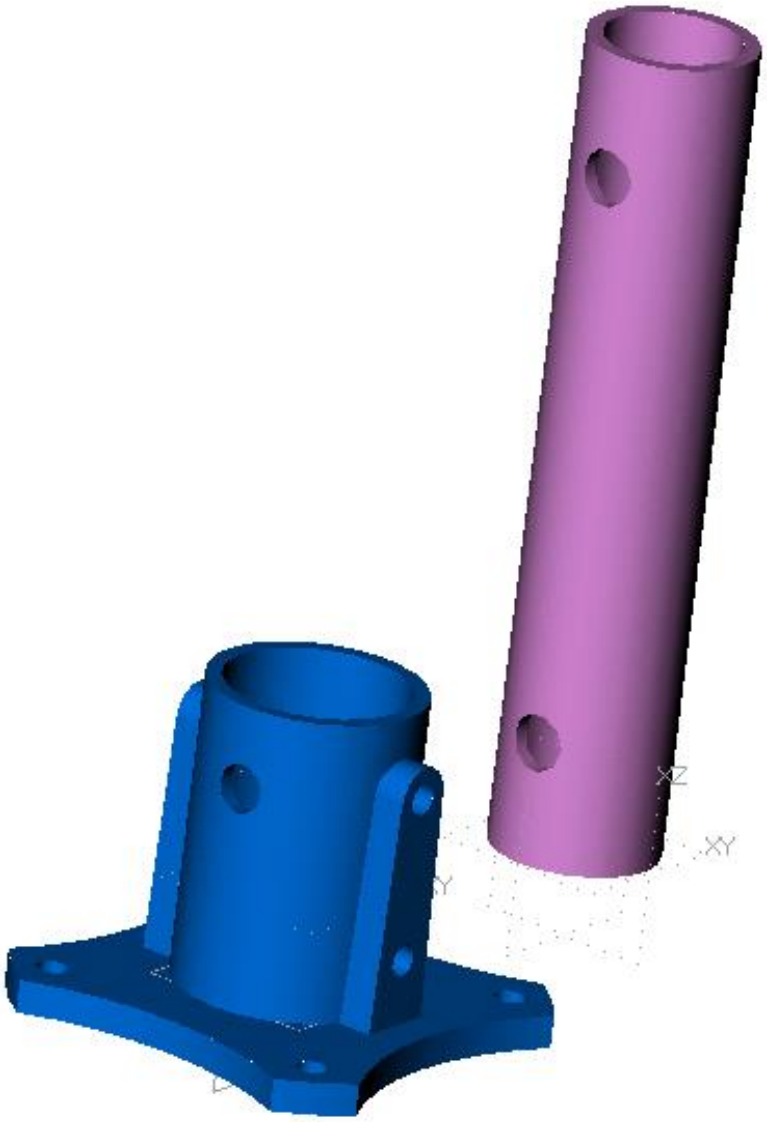
- Select the **XZ** plane.
- Select the **same facing** option.
- Select the world **XZ** plane.

Select **OK.**



7. The component should look like the training image.

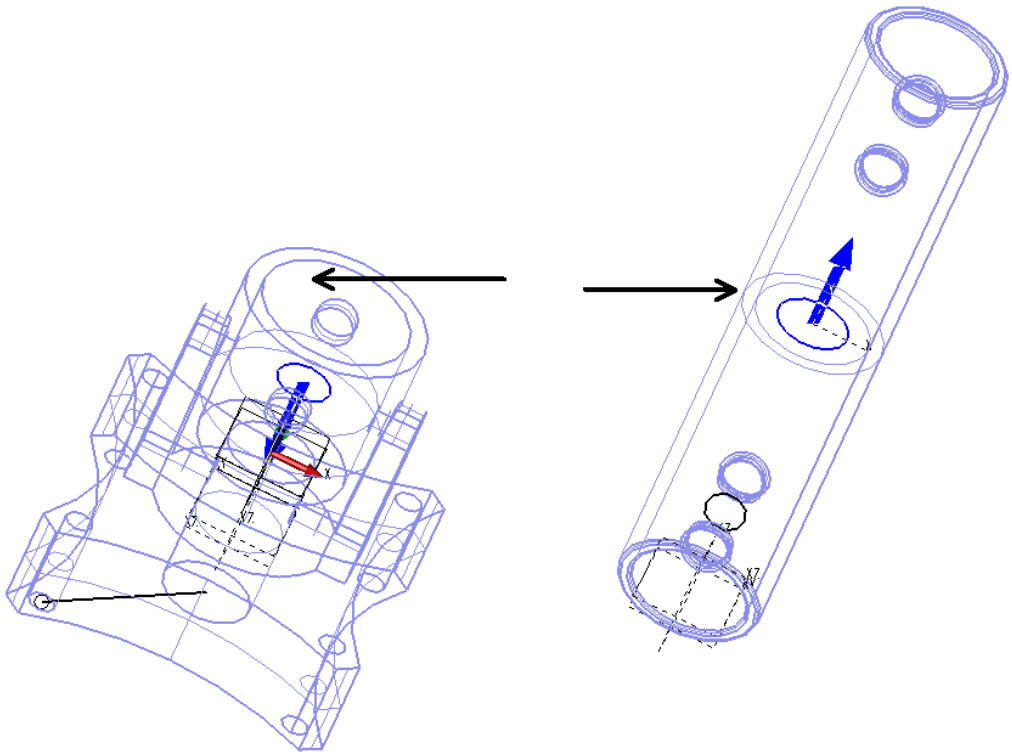
- Use the **Inquire Alignments** command  on the **Assembly Tab** to examine the constraint system.
- Select the **Cancel** button from the Show alignment status form.



8. Let’s insert a second component.

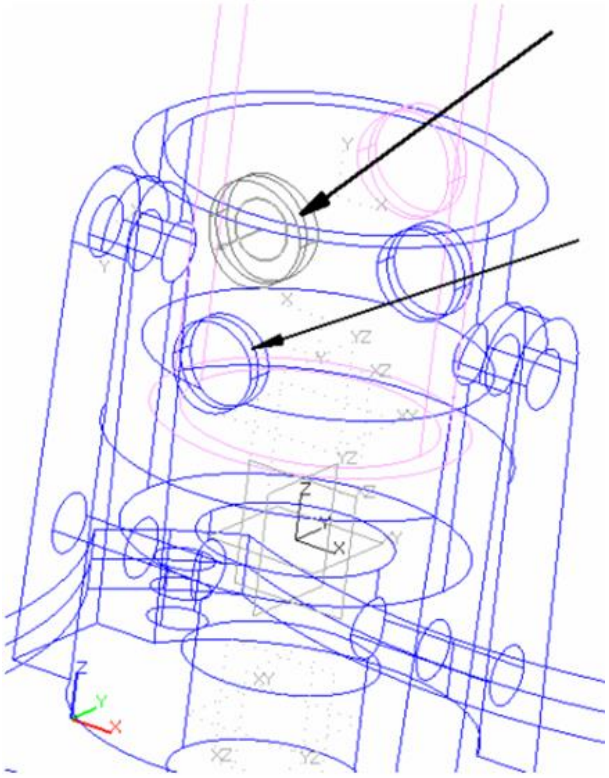
- Select the **Insert Component** command.(Right click> Insert Component)
- Select the ‘Tube’ component from the list of objects.
- For the origin, pick a location away from the Flange.

Zoom All.



9. We will now align the ‘Tube’ with the ‘Flange’.

- Select the outer cylindrical face of the ‘Tube’.
- Select the **Opposite** option.
- Select the inner cylindrical face of the ‘Flange’ component.



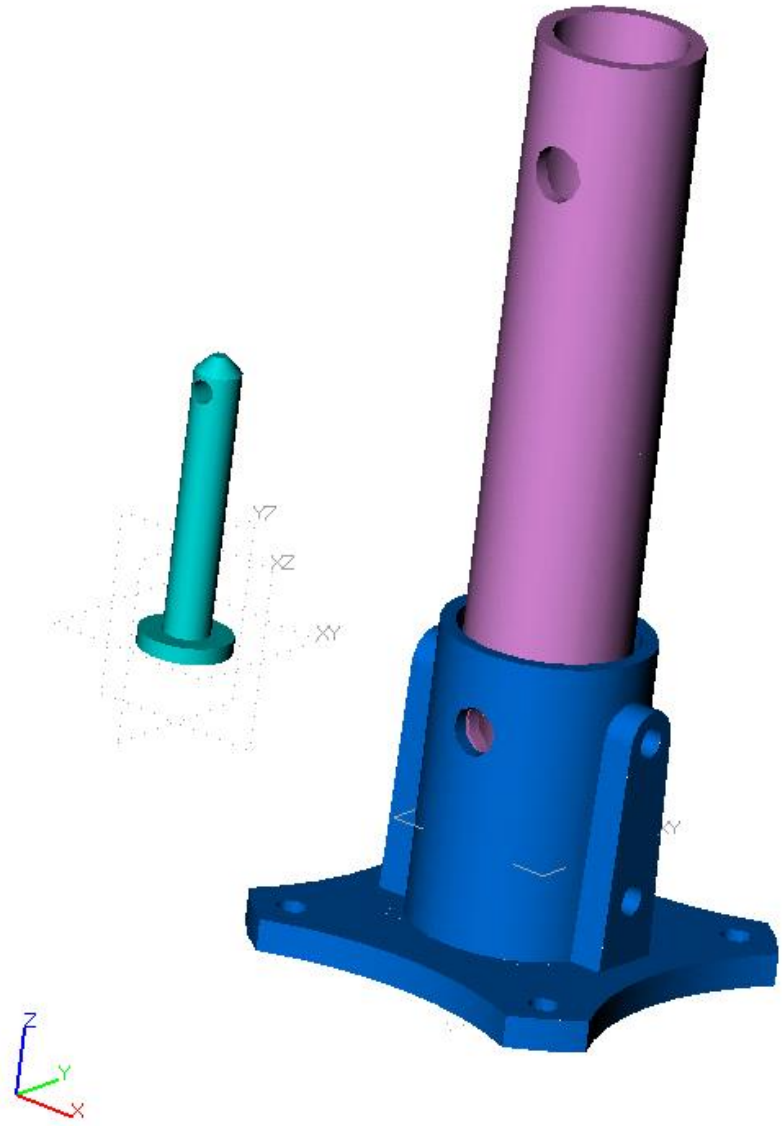
10. Select the face from the cross hole in the ‘Tube’ component (filters may be necessary).

- Select the **Opposite facing** option.
- Select the face from the cross hole in the ‘Flange’ component.

You have now eliminated all 6 degrees of freedom. The part can’t move in X, Y or Z and can’t rotate about X, Y or Z either.

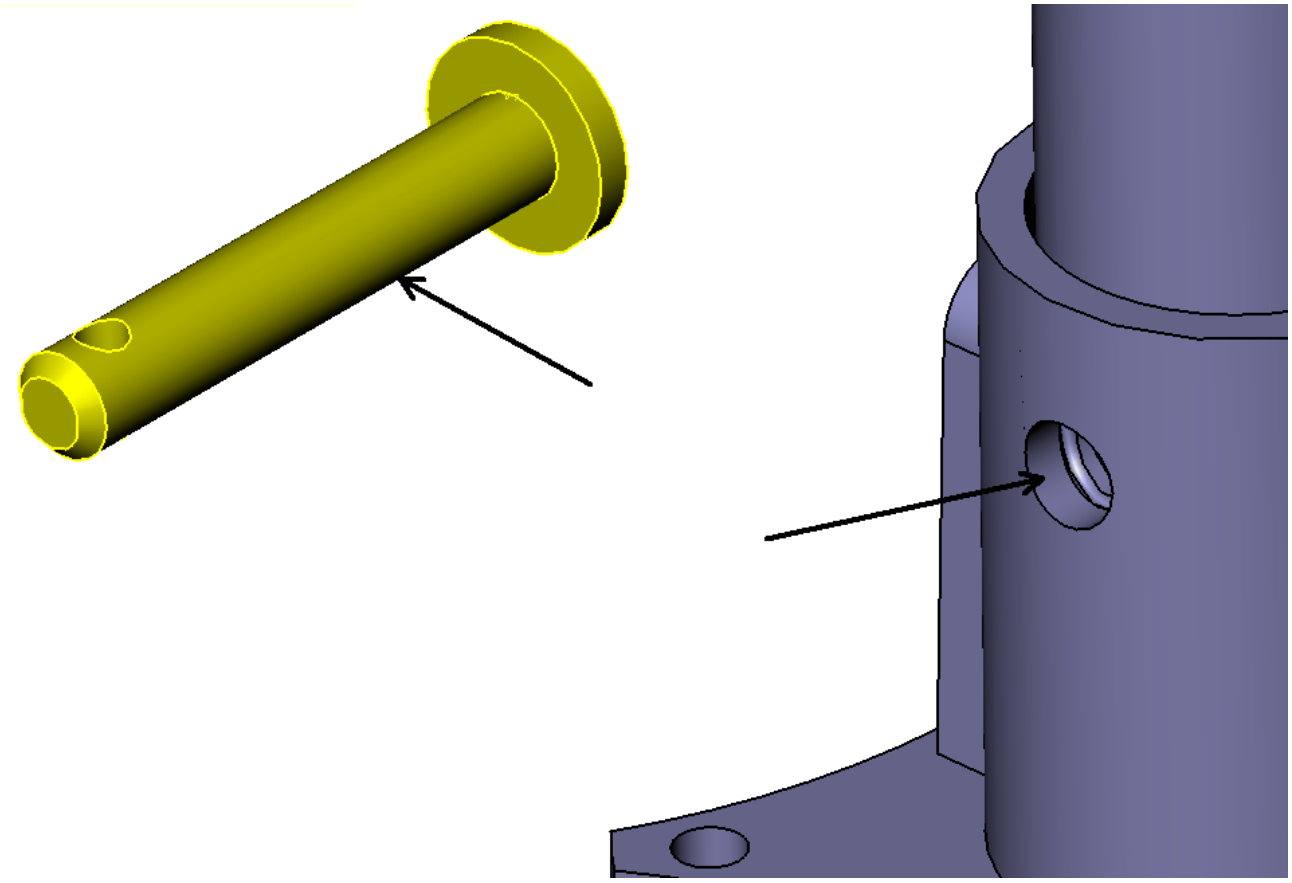
Select **OK.**

11. Turn the shading on. Your display should now look like the training image.



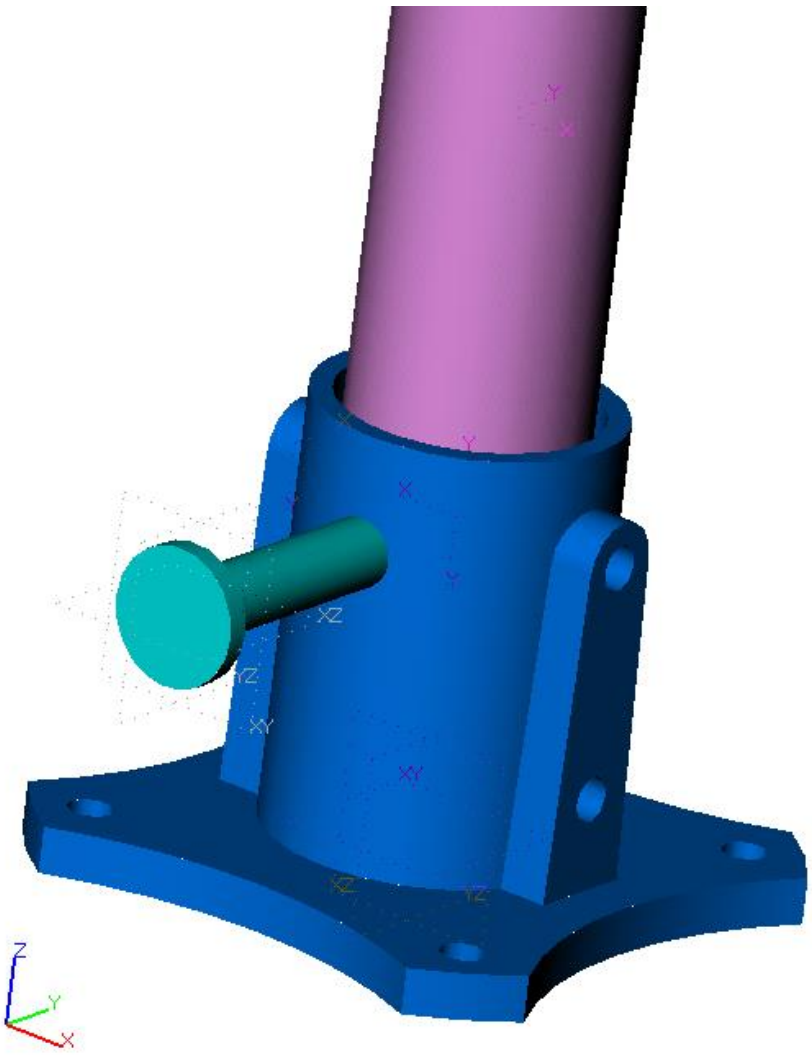
12. Now we will insert another component.

- Select the **Insert Component** command.
- Select the ‘Flange Pin’ component from the list of objects.
- For the origin, pick a location away from the assembly.

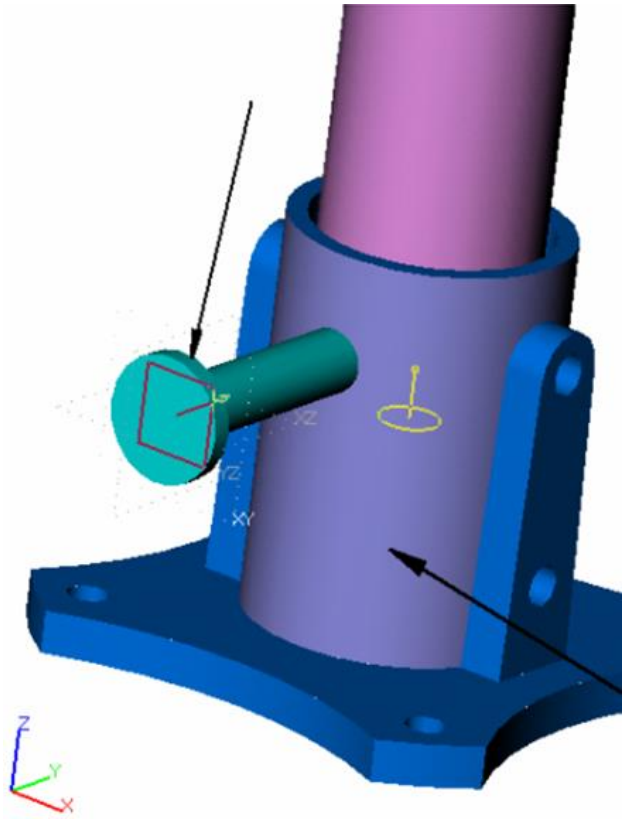


13. Again, this component needs to be aligned.

- Select the cylindrical face on the ‘Flange Pin’ component.
- Select the **Facing** option that is appropriate for this component.
- Select the inside hole face from the ‘Flange’ component.

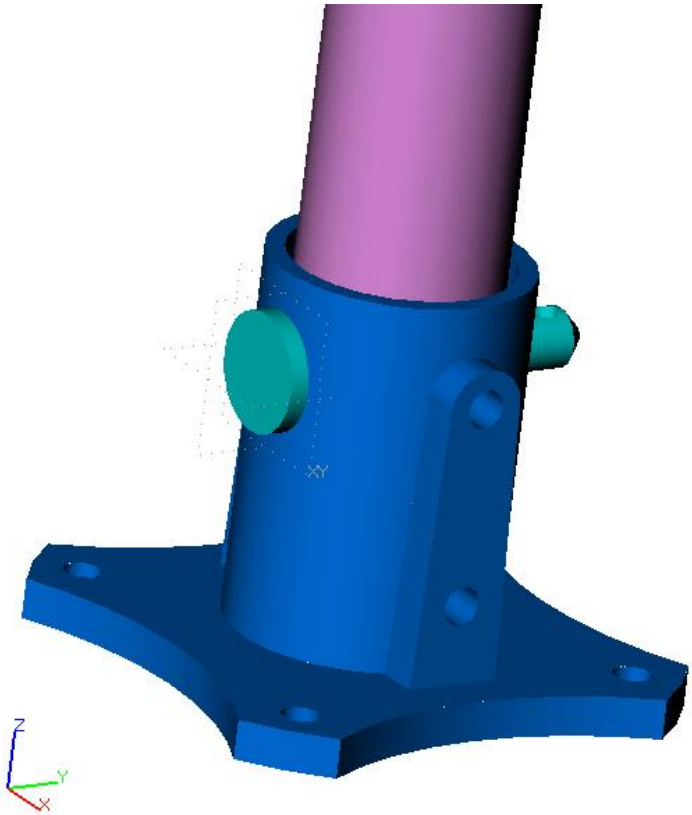


Your assembly should now look like the training image.



14. Now we will make the head of the pin tangent to the flange.

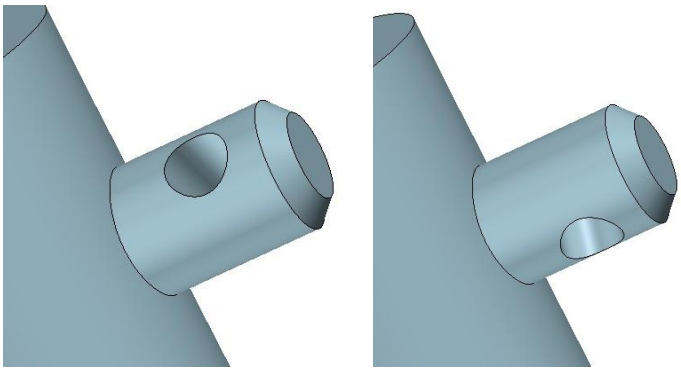
- Select the mating face from the head of the 'Flange Pin' component (filters may be necessary).
- Select the **Opposite facing** option.
- Select the **Tangent** icon from the Optional Inputs. See the figure above.
- Select the outer cylindrical face from the 'Flange' component.

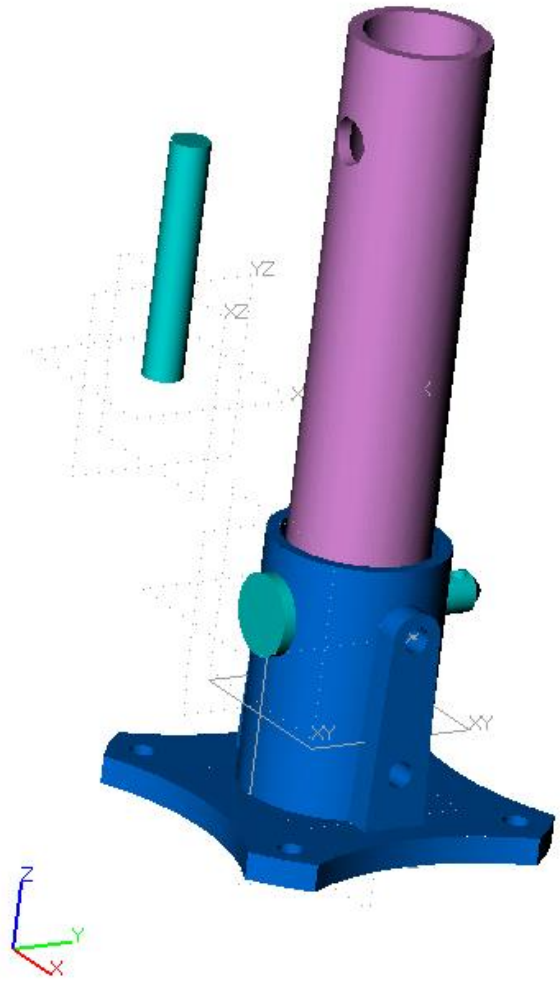


15. Turn the shading on. Your display should now look like the training image.

Try to define a rotational constraint on the Flange Pin.

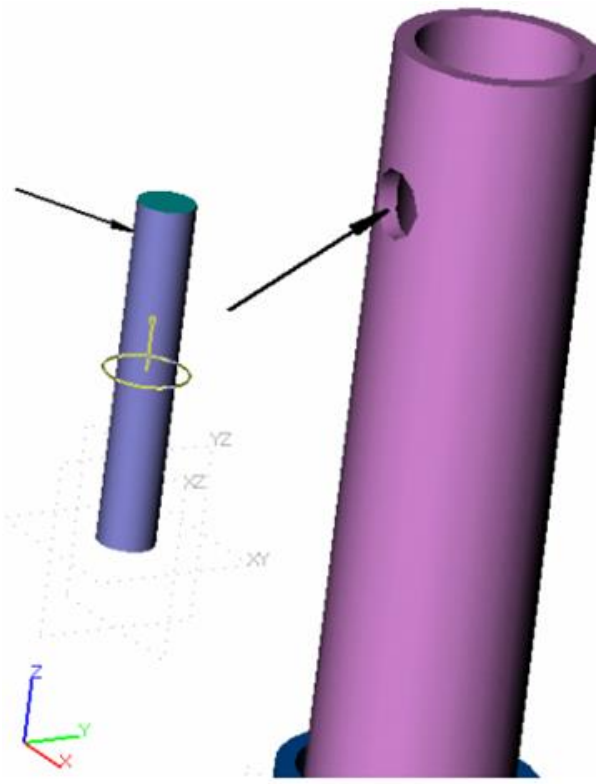
Note: Pick the hole in the flange pin as the first entity, pick the angle icon  (input 45 degrees), and pick the YZ plane as the second entity.





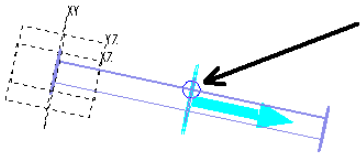
16. Now let’s insert the ‘Pin’.

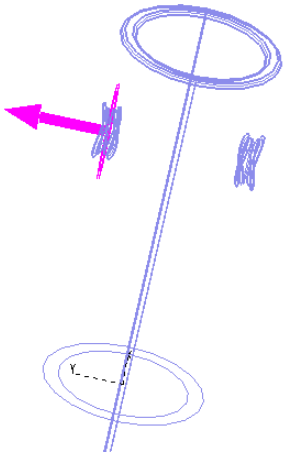
- Select the **Insert Component** command.
- Select the ‘Pin’ component from the list of objects.
- Select an origin away from the assembly.



17. We need to align this component also.

- Select the cylindrical face on the ‘Pin’ component.
- Select the inside hole face from the ‘Tube’ component.
- Select the **Opposite facing** option.

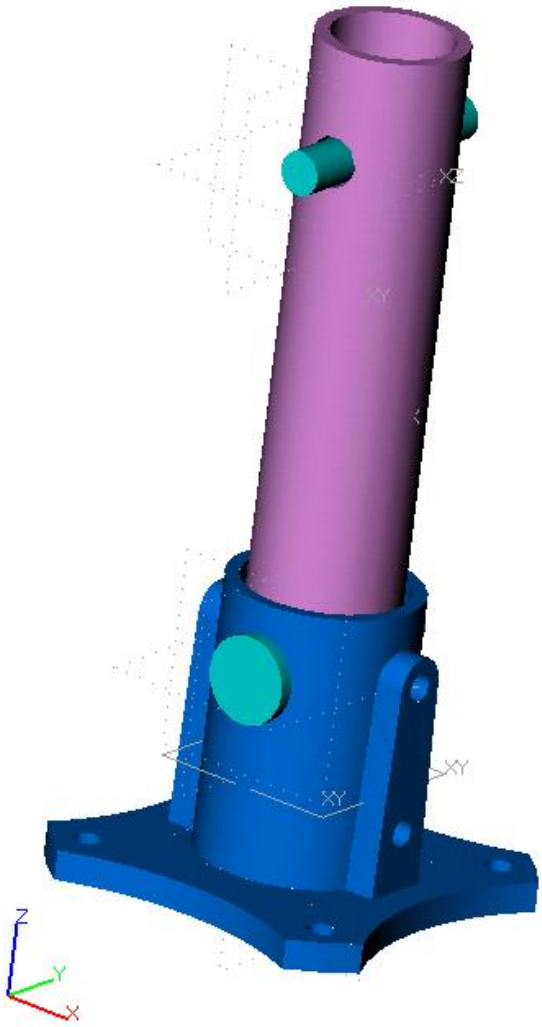




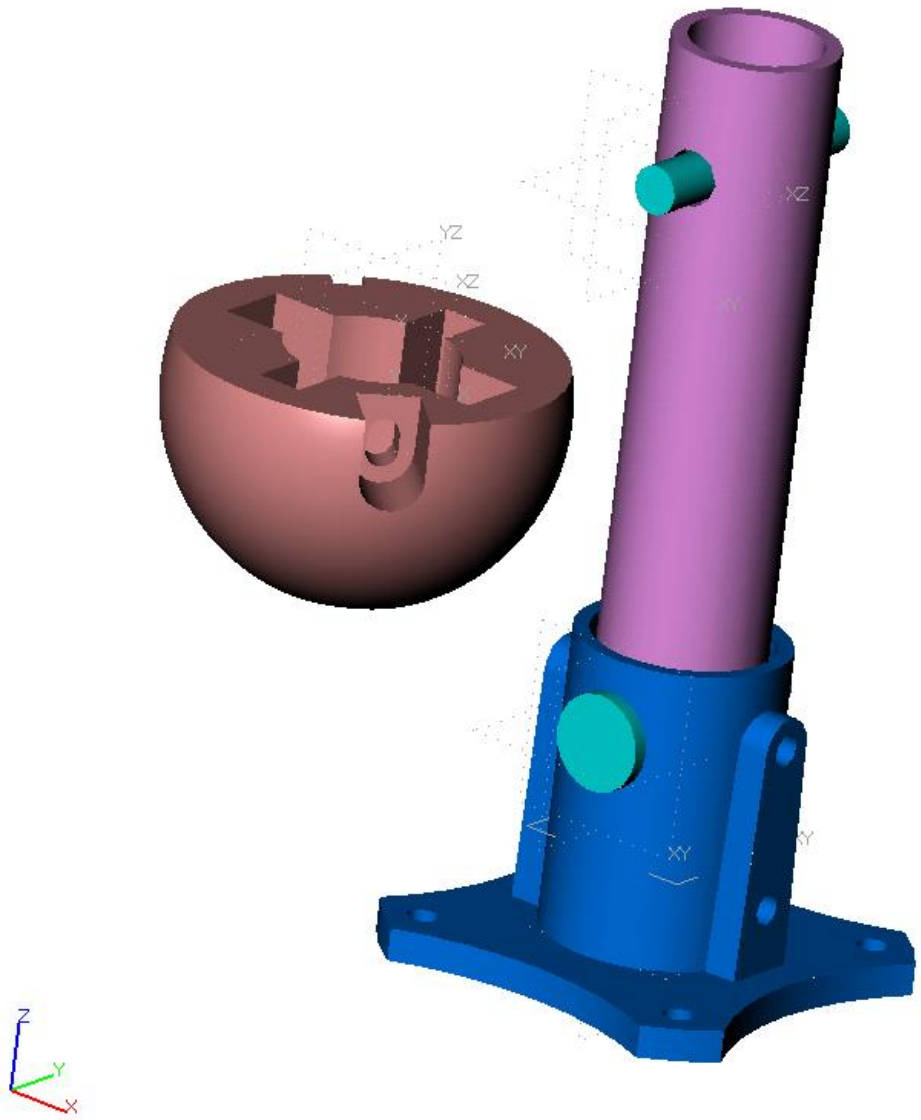
18. Now we will use a point constraint to determine the insertion depth of the ‘Pin’.

- Select the right face of the cylindrical face on the ‘Pin’ component as shown(filter on **Face**).
- Select **XZ** plane on the ‘Flange_Pin’ component. (filter on **All**)
- Select Same facing, and type 17.5mm on Offset.

Select **OK.**

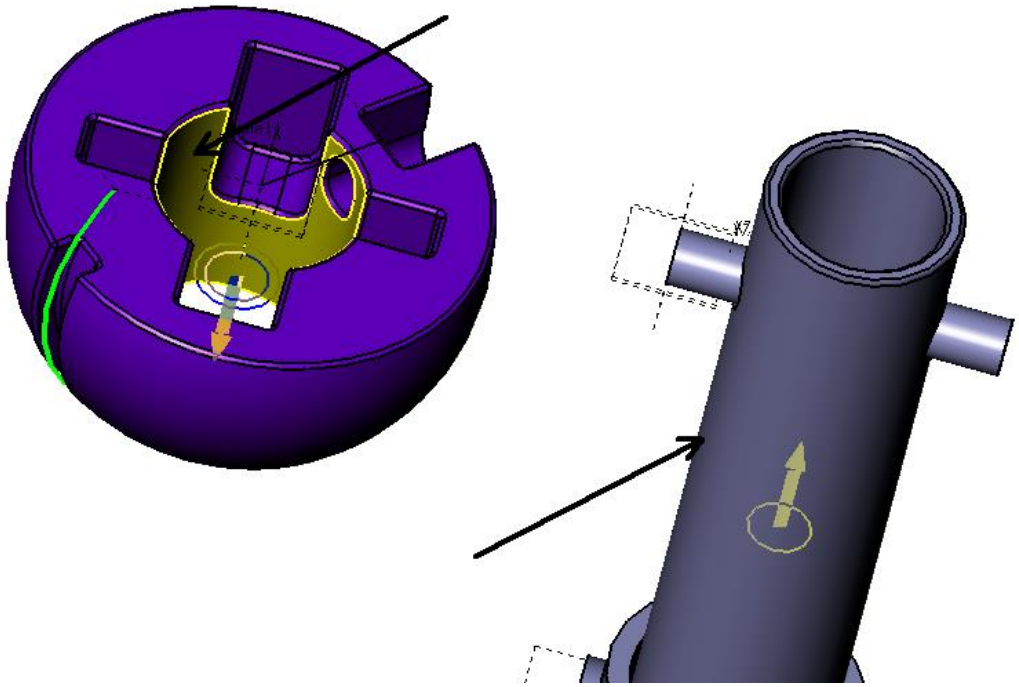


Your assembly should now look like the training image.



19. Let's insert one more component and align it.

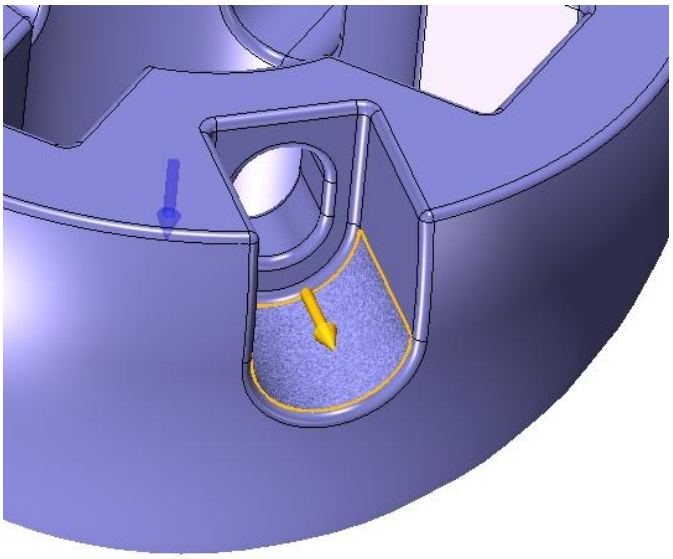
- Select the **Insert Component** command.
- Select the 'Ball' component from the list of objects.
- Select an origin away from the assembly.



20. To align:

- Select the inner cylindrical face on the ‘Ball’ component as shown.
- Select the **Opposite** option.
- Select the outer face of the ‘Tube’ component

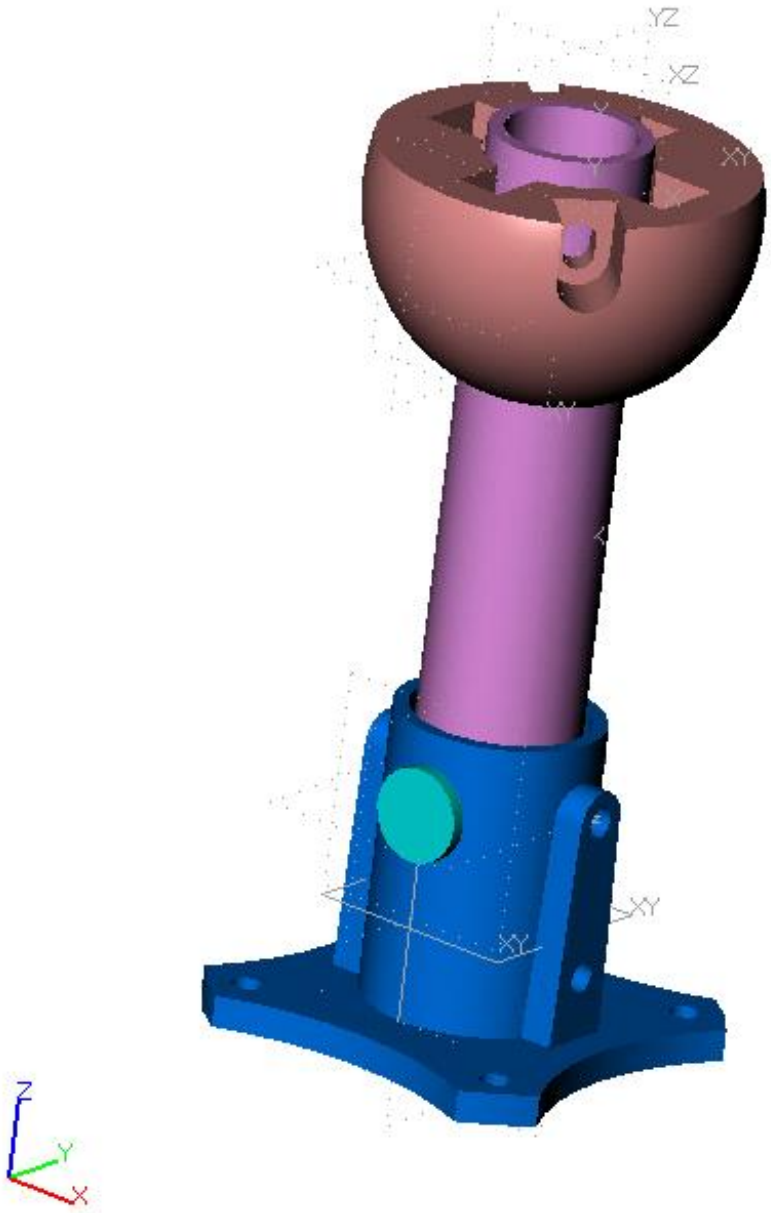
Pick **OK.**



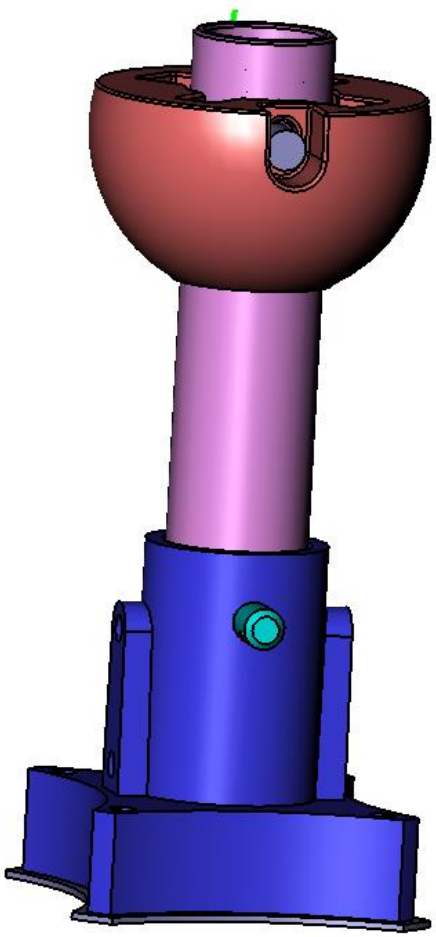
21. To align:

- Select the cylindrical face on the ‘Ball’ component as shown.
- Select the **outer cylindrical face** of the ‘Pin’ component.
- Select the **Facing** option that is appropriate for this component.

The slot in the Ball is now concentric with the Pin.



Your assembly should now look like the training image.



22. Use the **View/External Datums** button to remove the Datum Planes from the display.

23. Turn the shading on.

24. Select the **File/Save File** command, and save your assembly.

25. Let's check alignments before going on.

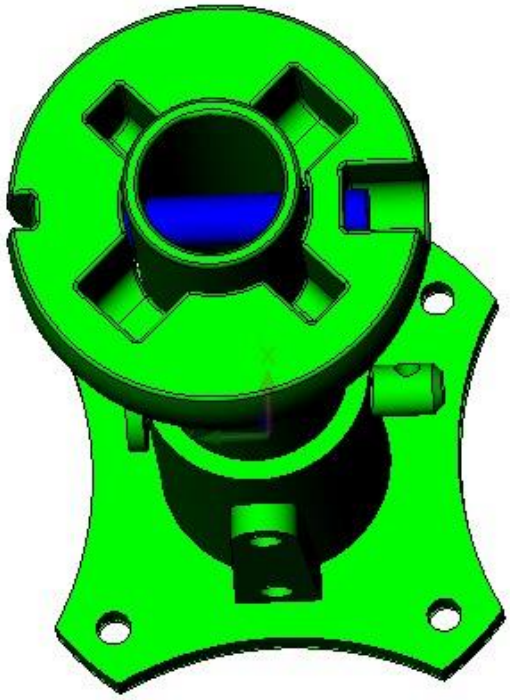
Use the **Inquire Alignments** command again.

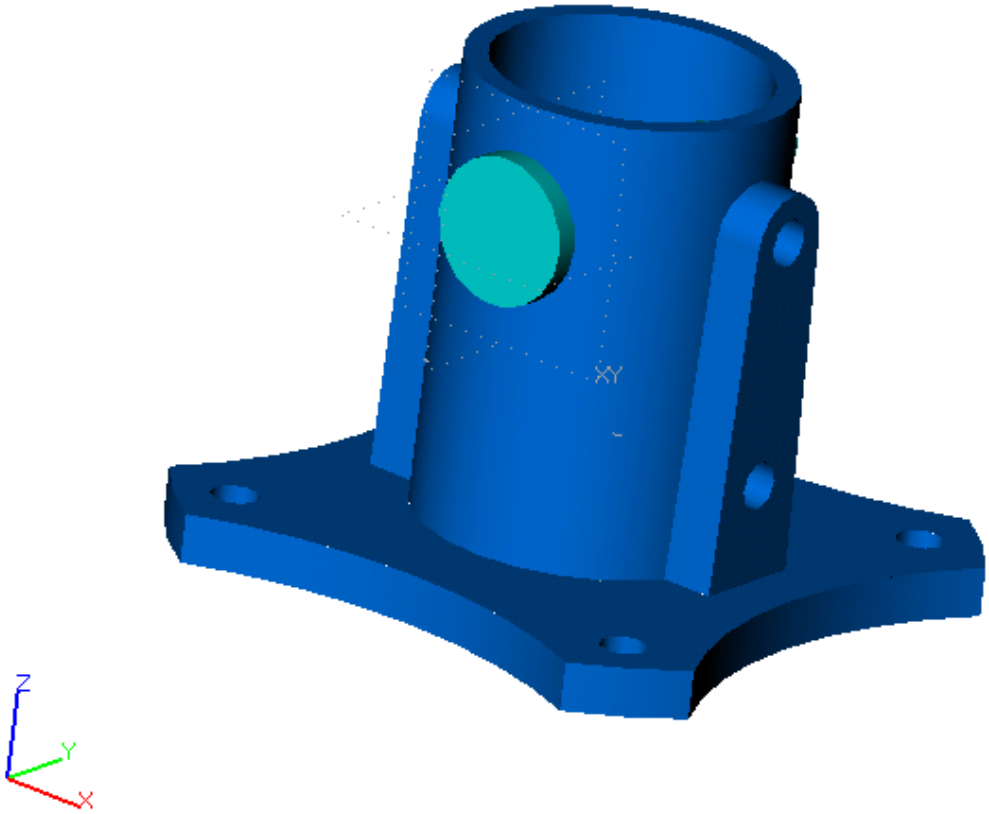
Look at the color coded graph to help interpret the results. You will see that all the components are **Well Defined** except the **Support Pin**. It has **1 rotational dof** (degree of freedom) .

Click **Next** to cycle through the individual components.

Think about what the pin does. We really don't care that it can rotate so leave it alone.

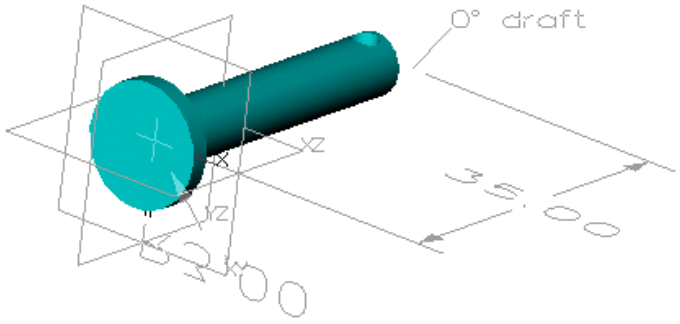
The flange pin would also be able to rotate except we gave it a specific angle alignment.





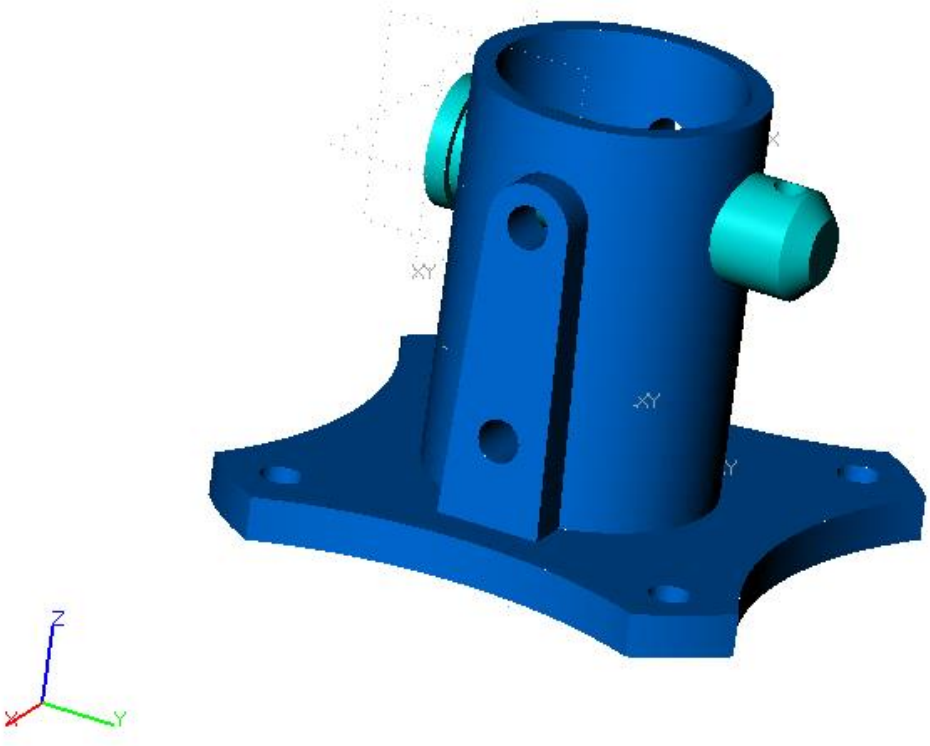
26. Let’s create an intentional interference condition and have the system check it.

- Use the **RMB** and select ZW3D Manager (if it’s not already displayed).
- Now select the **Assembly Manager** icon tab.
- select the ‘**Ball’** component from the tree, right click on it and select the **blank** option.
- Blank the ‘**Support Pin’** and ‘**Tube’** components.



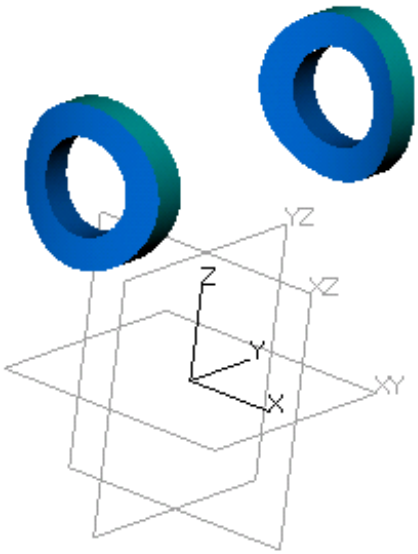
27. Modify the ‘Flange Pin’ geometry.

- Double click on the ‘Flange Pin’ Component. (Pick filter set to **all**).
- You are now in the ‘Flange Pin’ object.
- Select **View > Show Scope > Show Target** to only display geometry in the active object.
- Double click on the long cylindrical surface of the ‘Pin’.
- Double click on the Dia **6.0** dimension and change the value to **10**.
- The part history will automatically regenerate.



28. Going back to the Assembly.

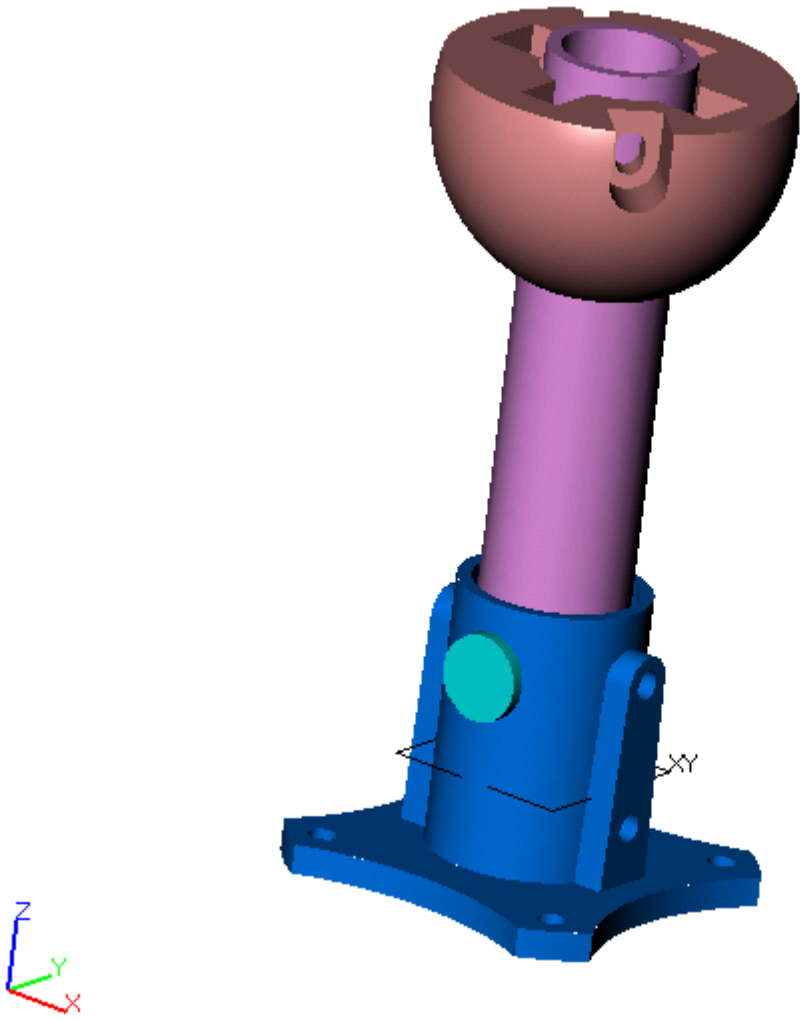
- Select the **Show All** icon from the Document Awared toolbar to display all of the geometry.
- Select the '**Exit part'** button to return to the 'Assembly'.



29. Select the **Interference Check** icon from the **Assembly** Tab  .

- **Base** select the **Flange**.
- **Check** select the **Flange Pin**
- Check the message window for interference message.
- Go to the **Assembly Manager** again and blank all components.

30. Select the **Undo** command to undo the history until you are back to the Assembly (see title bar) with all components visible.



31. Select the **Unblank All** command.

Your assembly should now look like the training image.

32. Select the **Exploded Config** icon from the **Assembly** Tab .

Use the **Middle Click** to accept the command.
Your assembly should now look like the training image. You may need to use the **Move along direction** command on some components to adjust their spacing.

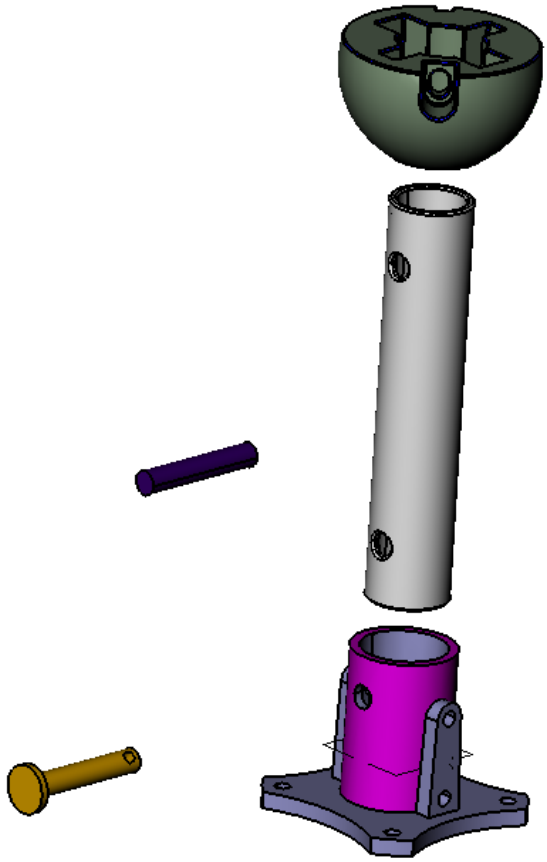
In order to manage the original and the Explode assembly, ZW3D automatically creates a new Assembly Configuration.

See the Title bar at the top of the screen, it shows [Explode_Asm].

Use the **Activate Configuration** command shown below .

You can select between the **original** and the **exploded** configurations.

Using the configuration technology, ZW3D can have multiple exploded configurations.



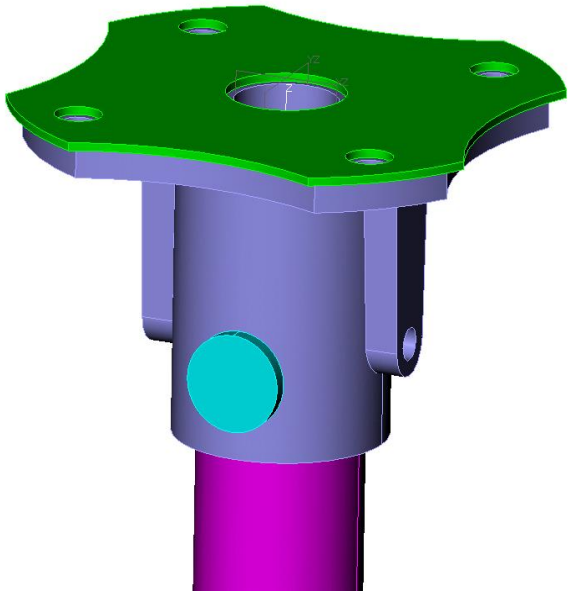
On a drawing sheet, when laying out a standard view, click on the Advanced tab > config and pick either the default or the exploded configuration.

ASSEMBLY - Top Down approach

Our Next Project with this assembly will introduce the 'Top Down' method of creating a part. We will create a new empty part object within the context of the assembly and we will copy geometry from other parts in the assembly to begin the construction of our new part.

For the purpose of demonstrating this approach we will construct a simple gasket that would sit on the bottom of the flange as shown to the right.

(Hopefully you have already been exposed to this concept in Chapter 3 of the Innovator Training manual.)

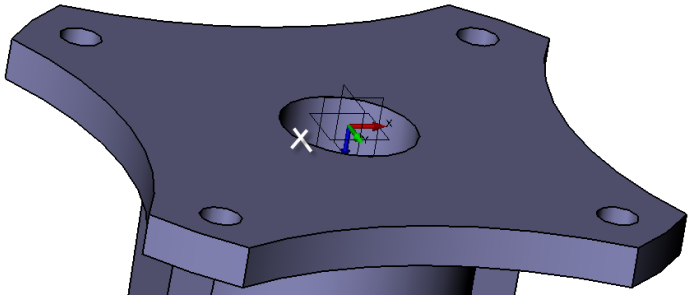


1. **Insert** pull down menu > **Component** (or Right Click – Insert Component).

Type in the name of the new empty Part (Flange_Gasket) on the command line as shown above and Hit <ENTER>.

Then check the **Anchor** option from the Insert Component Options Window.

You will be prompted on the command line for the insertion point.
Right-click > Critical and pick the center of the hole at the top of the Flange.

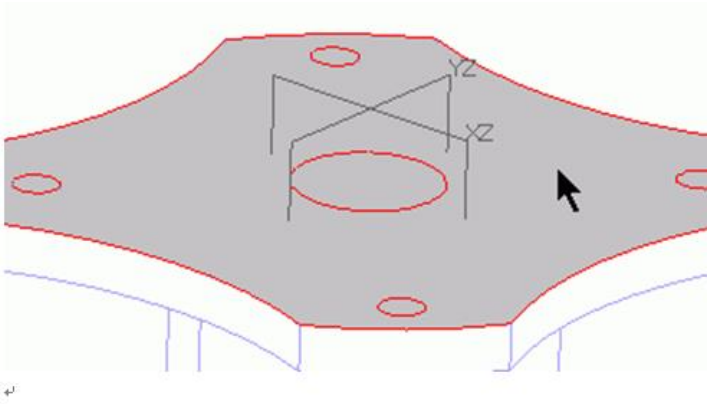


2. Your Title bar should show you that you are in the new **Flange_Gasket** part object that we just created. See the title bar, it shows "[Part[Flange_Gasket]".

   You can also pick **View > Show Scope > Show Target** and you will see only the three base datum planes on the screen.

3. Next we want to copy the bottom planar face of the flange into our Flange_Gasket to use for the creation of the gasket. Pick the **Show All** icon from the top toolbar.

4. Insert Component > Pick **Flange** > **, select Assembly pull down menu** > **pick Reference Face**  > **set your Pick Filter to Face** > **select the bottom face of the flange as shown below and Middle Click or hit <Enter>.**

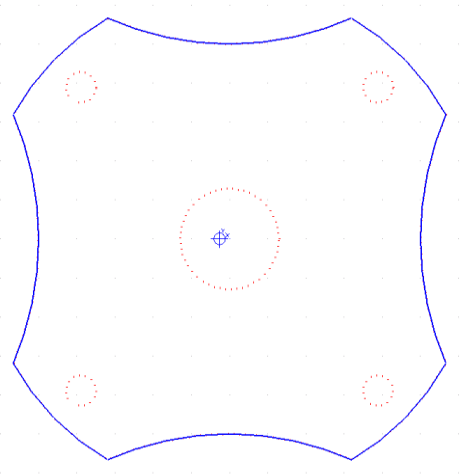


Select the **Show Target** icon from the top toolbar and you will see the Face you just copied and the three base datum planes on the screen. In Wireframe display, the face will have blue dotted edges representing the open edges of the face.

5. Now we need to build the gasket. We will build two separate sketches. One for the main body or outside shape of the gasket and the other for the holes in the gasket.

Insert a sketch and select the face we just copied as the planar face or reference plane of the sketch. Be sure that the copy **Reference face edges in the Sketch Options form is checked.** Then pick **OK**

Use the **Trace command** to trace the outside reference edges for your sketch

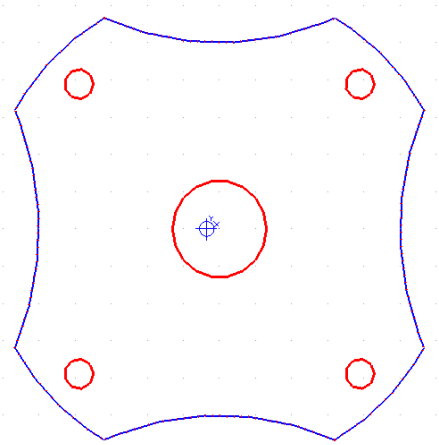


Exit Sketch.

Insert Sketch Again, and pick the same face for your sketch with the **reference face edges** checked.

When in the sketch **select the 5 red dotted circles** of the holes and **Right Click** and Pick **Toggle Type** from the Right-Click Shortcut menu. Your circles will become red solid circles.

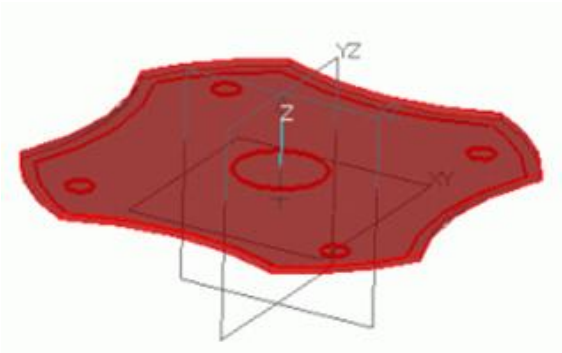
This subtle change is significant because the red circles still reference the model geometry but we can now use them when we leave the sketch.
Your sketch should look like the following figure.



Exit Sketch.

6. Next, **Erase the Face** that you used to generate these sketches

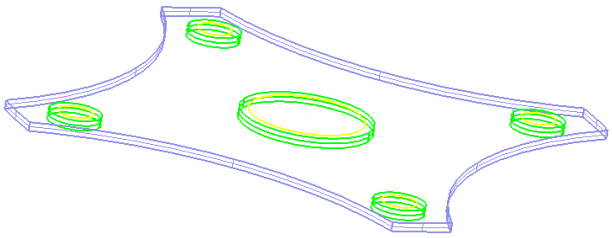
7. **Extrude** the first sketch that you made for the **Base** shape of the gasket.
Fill in the option form as shown.
Start = 0
End = 1
Offset = Shrink/Expand -> 2.00



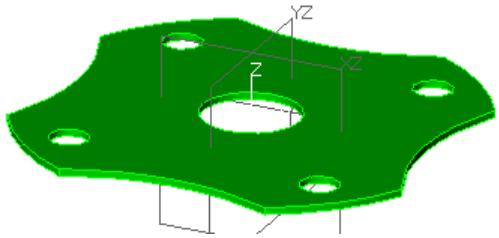
8. **Extrude Remove** the second sketch to create the holes in the gasket.
Fill in the Options as shown.
Start=0
End = Through All
Offset = Shrink/Expand -> 1

Pick OK to complete the Extrude Cut.

As shown below.

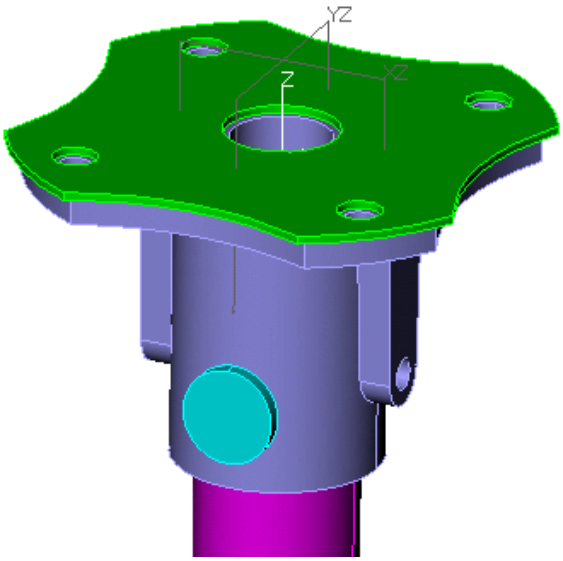


9. Your result should look like the following. Pick **Show All** from the Document Awared toolbar.



Right-Click > Exit Part.

And you will be returned to the main Fan Assembly Object.



10. Now let's edit the Flange to see changes update in the Flange Gasket.

Set your entity filter to Component and double click the Flange.
Your title bar should show that you are in the Flange Part and if you pick the Show Target you will only see the flange.

Open the **History** Window of the Flange (CTRL-H).

Right Click on **Sketch2** and choose **edit.**

Change the 40mm Radius Dimensions to 43.

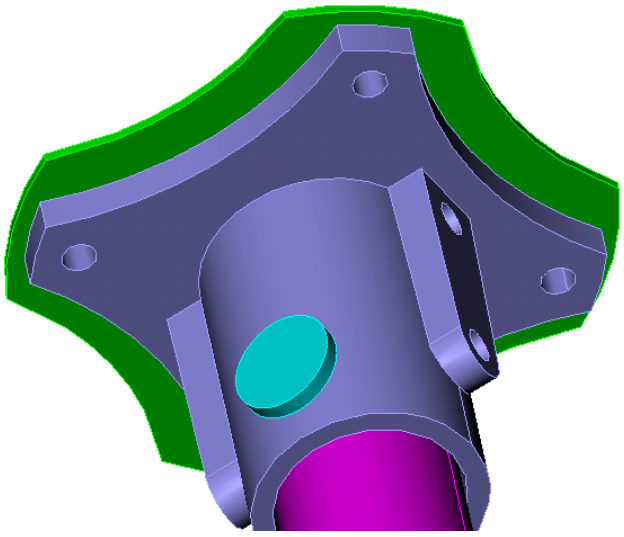
Exit Sketch.

The history will automatically regen.

Then **Exit part.**

Check your title bar to see if that you are at the parent assembly.

Your assembly should look like the following figure.



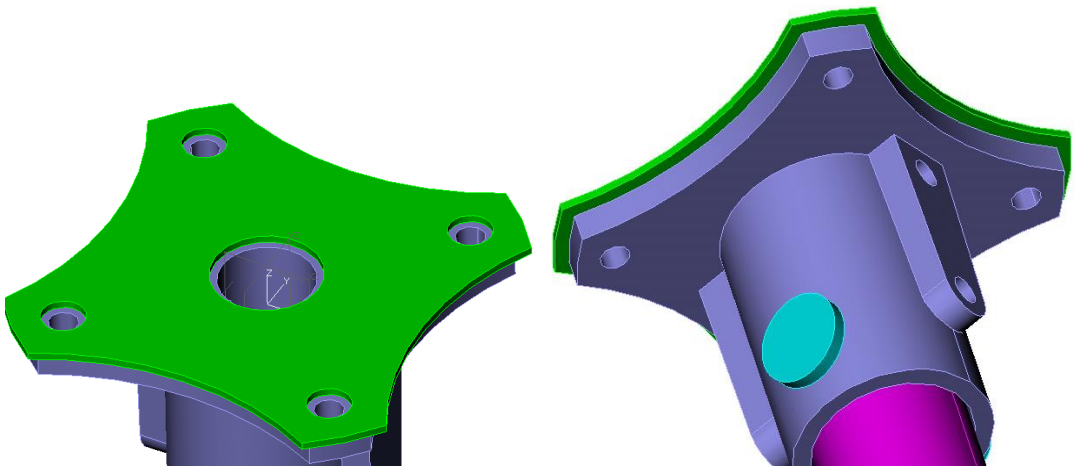
The Flange Gasket has not updated, yet.

Set your entity filter to Component and Double click on the Flange Gasket.

You are now editing the flange gasket.

Pick the **Regen** Button.

Your Flange Gasket should have updated and looks like the following image.



Exit part.

As an additional exercise, let's create the flange gasket a different way.

1. Make sure you are at the Assembly level and not in a component part. (see title bar)

2. **Blank** the Flange Gasket component from the assembly.

3. **Insert Component** – Name it **Flange_Gasket-2** – anchor and position at zero.

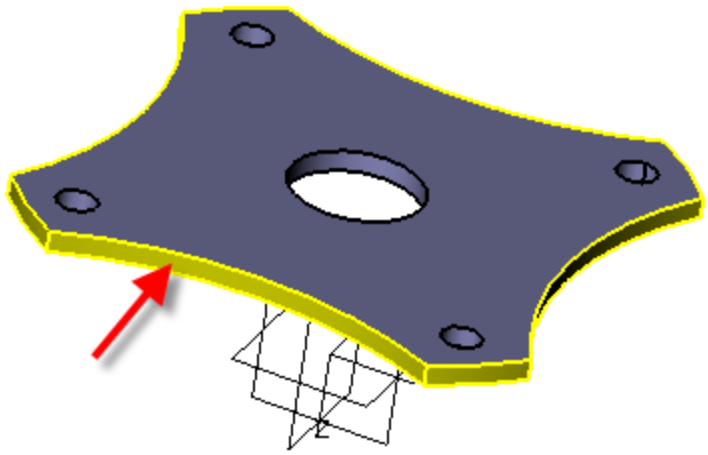
4. Insert Component > Pick **Flange** > **select Assembly , pull down menu** > **pick Reference Face** > **set your Pick Filter to Face** > **select the bottom face of the flange as shown below and Middle Click or hit <Enter>.**

5. This time we will use the **Shell** command from the Shape Tab.

Pick the open surface. Make the thickness -2.0

Since we want to add material to the blue side of the surface it will be a negative thickness

6. Use the **Face Off** command and add **2mm** to the 8 outside surfaces shown below.



7. Repeat this command and add a negative 1.0 to the 5 holes. This will make the holes larger.

8. Select **Show All** to view the entire assembly.

9. You may want to change the **face attribute** of your new shape.

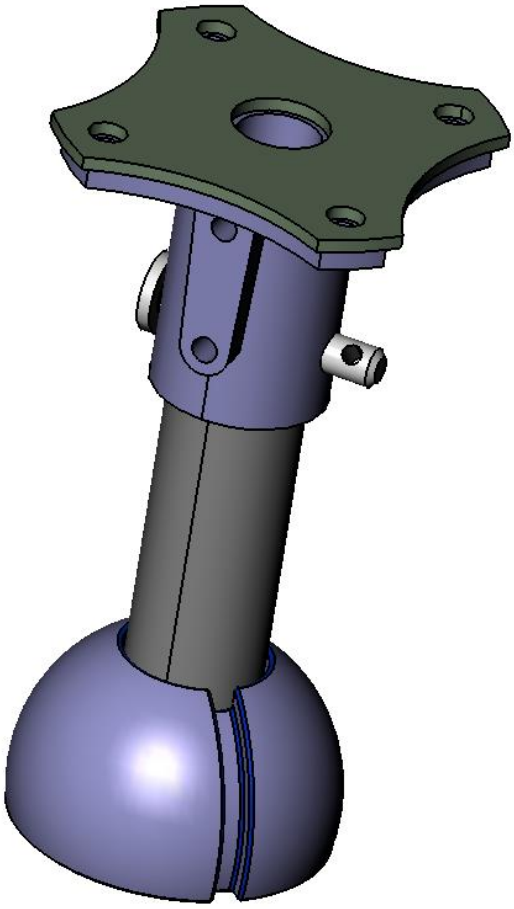
10. Now, let's go back and **change the thickness** of the gasket to one.

Right-click on Shell and select **Open/Close.**

Right-click on Thickness and select **Edit.**

Change this to 1.0.

The History will automatically regen.



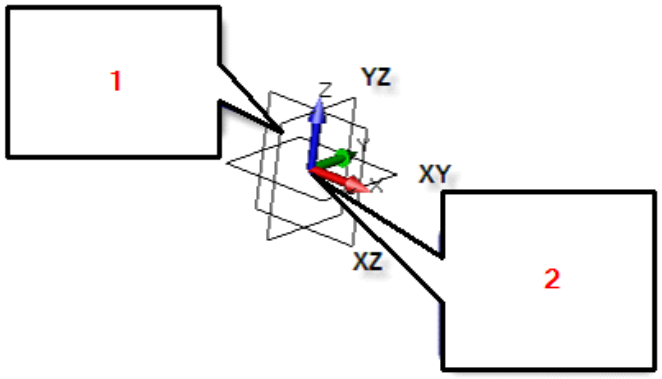
Day 2

Chapter 6

Datum Planes and Reference Geometry

Datums

Datums are used to establish a plane of reference. A datum plane or a planar part face can define them. When a sketch is created, you are prompted to select an insertion plane. When you select a datum plane or planar face, the sketch is aligned with the plane and sketch geometry is created on the plane. The sketch is parametrically constrained to the plane on which it is created. If the plane moves, the sketch will move with it during the next history replay. There are three default datum planes as shown below.

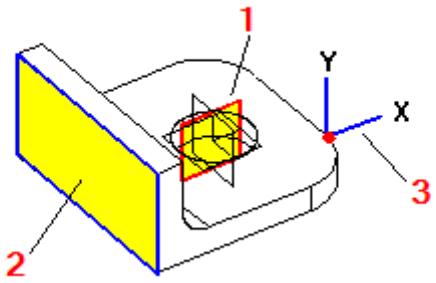


1: Default Datum Planes 2: World Origin and Traid

Datum planes are also used to create certain features that do not require the use of a sketch such as parting lines and section curves. The Insert Datum Plane command creates datum planes using a variety of methods.

In this discussion you have learned that:

Datums are planes of reference used to locate sketches and create features.
Datums can be datum planes and planar part faces.
There are three default datum planes located at the world origin.
Datum planes can be created using the Insert Datum Plane command.



1:Default Datum Plane 2:Planar Part Face 3: New Datum Plane

Refer to the HELP Manual

Refer to the and review the following commands.

Datum Plane - About
Datum Plane - Insert
Datum Plane – Make Local
Datum Plane - Mirror
Datum Plane – Toggle External

Create External Reference Geometry Menu - Sketch Level(Insert> Reference Geometry)
This menu contains commands for creating external reference geometry in a sketch. External refers to geometry that is referenced from outside the active sketch plane. Using external references will allow sketch geometry to adapt to changes made to the active part or other components in the case of an assembly.
ZW3D uses several terms that that are related but do have separate meangs.
"Datum Plane" refers to a database entity that defines a plane.
"Plane" refers to any specification of a plane in space, whether or not it is a datum entity (i.e. a planar face pick, +X, -X, +Y,...).
"Frame" is used when 3D coordinates are of being talked about as opposed to something happening on a "plane".

Refer to the HELP Manual

Refer to the **HELP Manual** and review the following commands.

Project - a Reference Curve
Project - a Reference Face
Project – a Reference Point "F7"
Toggles - Curve Type
Toggles - Display
Toggles - Perspective View
Toggles – (View) Labels
Toggles - Display (On by default)
Toggles - Dimensions (On/Off) (Ctrl+D)
Toggles - External Datums (Off by default)
Toggles - Grid On/Off (Ctrl+G)
Toggles - Line Limits (On/Off)
Toggles - Construction Geometry (On/Off)
Toggles - Sketch Constraints
Input Options (Index)
Input – Options Menu

Chapter 7

Creating Holes, Fillets and Chamfers

Refer to the HELP Manual

Refer to the **HELP Manual** and review the following commands.

Hole - Simple
Fillet - Constant Radius
Fillet - Elliptical
Fillet - Face
Chamfer - with 1 setback
Chamfer - with 2 setbacks
Chamfer - at Vertex
Shell
Offset - Volume
Offset – Face of a shell

Lesson 7

Creating Placed Features

OPEN the file “**Fundamentals.Z3**”

Follow the instructions supplied in each object to create the required fillet.

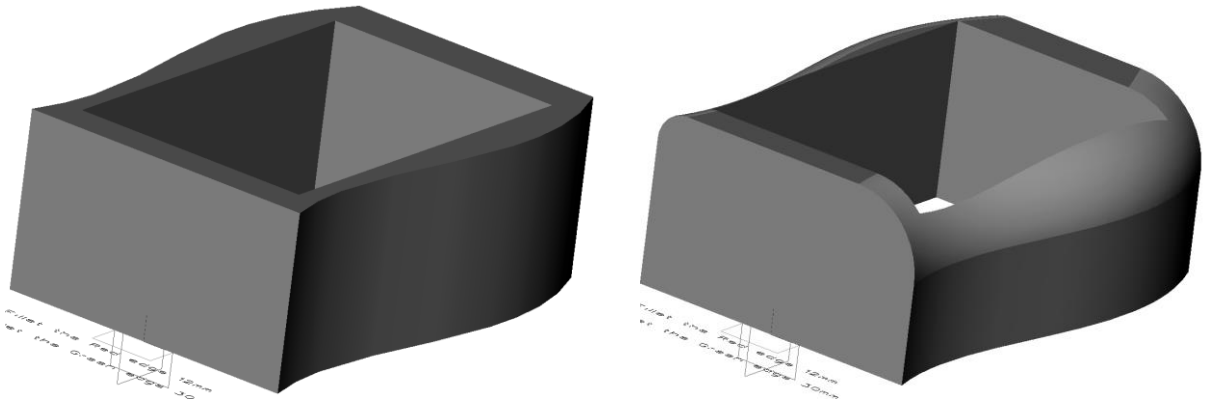
Edit the object “**Fil_Box_Shape_with_Cutout**”.

Change the display to wireframe if it is not already. This will allow you to see the edge geometry that the instructions will refer to. After completing the fillets on this part change it back to the shaded mode to get a better look at your results.

In this example we will show how ZW3D can generate fillets that will extend beyond a selected face.

Fillet the RED edge **12mm**.

Fillet the GREEN edge **30mm**.



Exit Part.

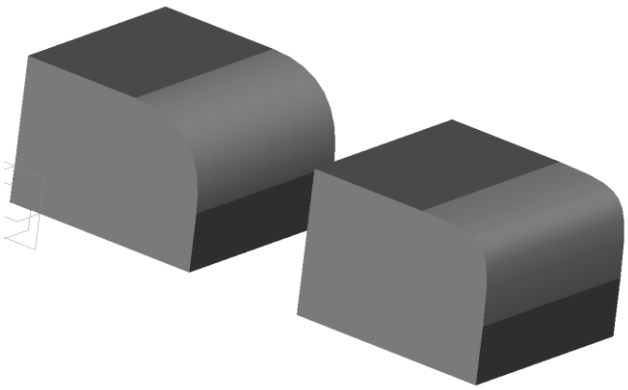
Edit Fil_Conic.

Set the display to **wireframe**.

Fillet the RED edge **30mm** with a conic ratio of **0.5**. Change ARCTYPE to **Conic**.

Fillet the GREEN edge **30mm** with a conic ratio of **0.7**. Pick ARCTYPE to change its value.

<Right-click> **Redefine Last** and change the conic value to **0.9**. Notice the increase in the sharpness of the fillet.

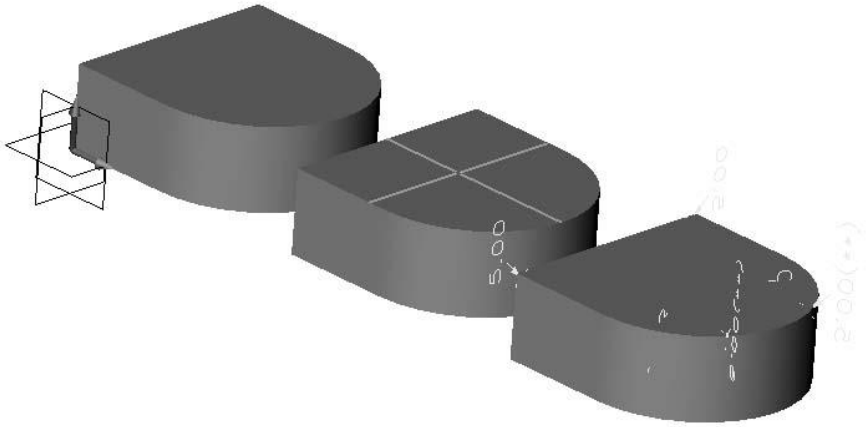


Exit Part.

Edit **Fil_D_Shape.**

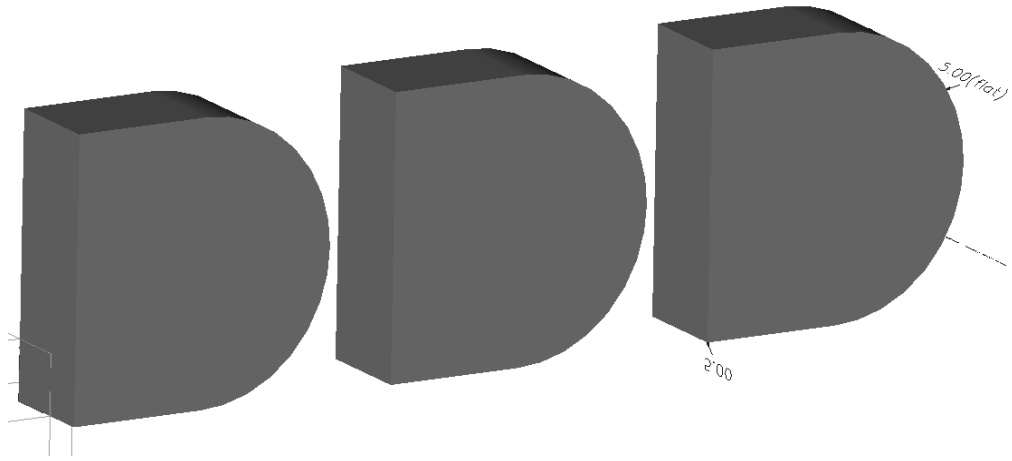
Pick the **Set Origin** command.

<Right-click> **On Face** and pick the top face of the middle object as shown in the figure below.



RMB-Drag to rotate and MMB-Drag to pan the view of your model as shown in the figure below. Notice as you rotated the view that the rotation point was about the point you defined in the previous step.

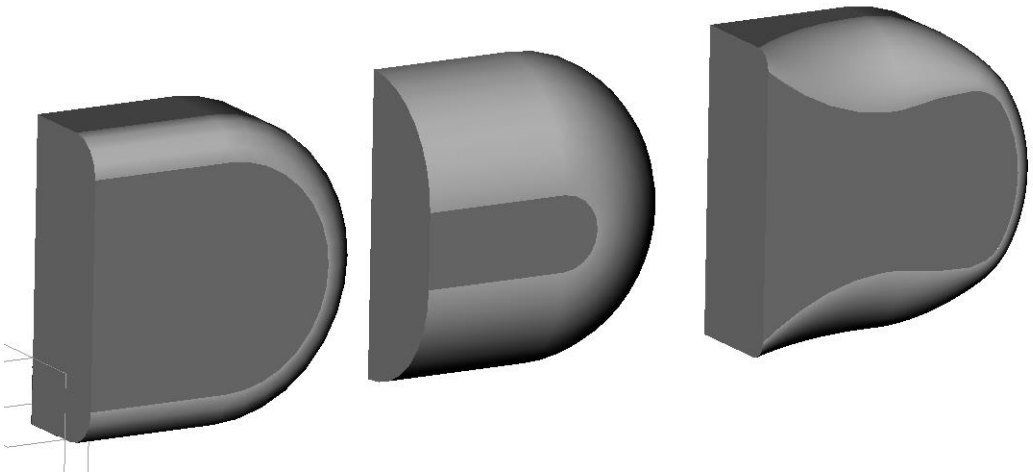
This view orientation will make reading the directions for each fillet example easier.



Fillet the RED edge **10mm**.

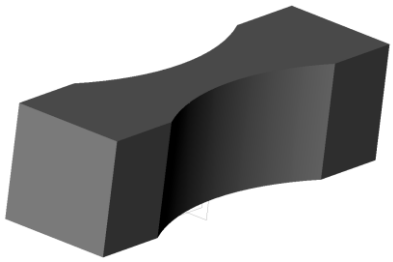
Fillet the GREEN edge **30mm**. Notice the vertical faces disappear.

For the 3rd shape, use Fillet with any size radius and the fillet command will pickup on the existing variable fillet attributes.



Exit Part.

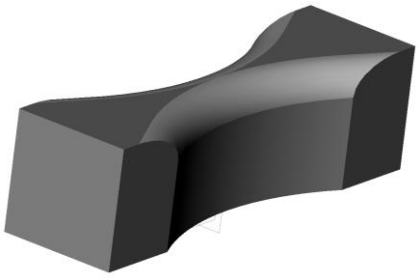
Edit **Fil_Dog_bone_shape**.

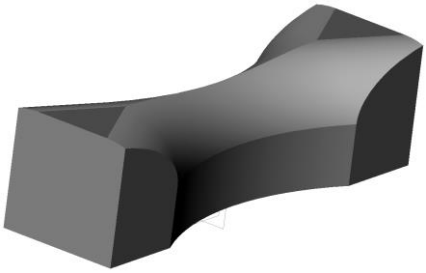


Fillet the RED edges **10mm**. This will generate a tangent condition between the fillet and the faces it is attached to as shown in the left figure below.

UNDO the last command after you have evaluated the results.

This time Fillet the RED edges to **20mm**. This will generate a non-tangent condition between the fillets and the faces they are attached to as shown in the right figure below.

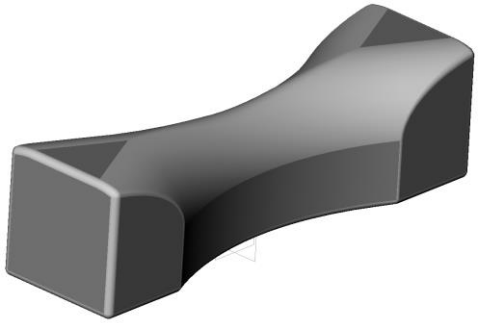




The purpose of this step is to show how ZW3D can create overlapping fillets and adjust to the condition. Notice in the figures above how the fillets in the left image are tangent to their adjacent faces, and the fillets in the right image are tangent to the vertical faces and intersecting each other at the top.

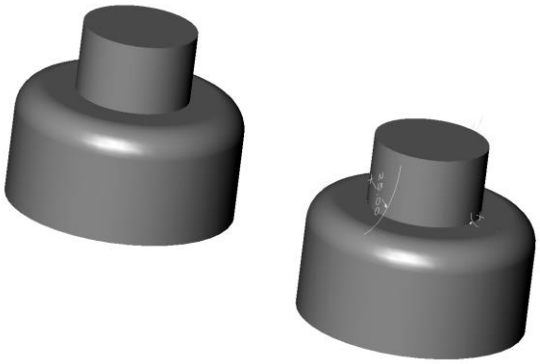
Create a **5mm** fillet along the top edge where the two fillets intersect. Fillet will fail if you try to make this one too large. Play around a bit with this thought. Make sure when you finish playing that you have a 5 mm fillet along the top.

Pick the **Fillet** command again and **<Right-click> Pick All** to pick the entire model. Create a **2mm** fillet on all remaining edges. Your model should appear as shown in the figure below.



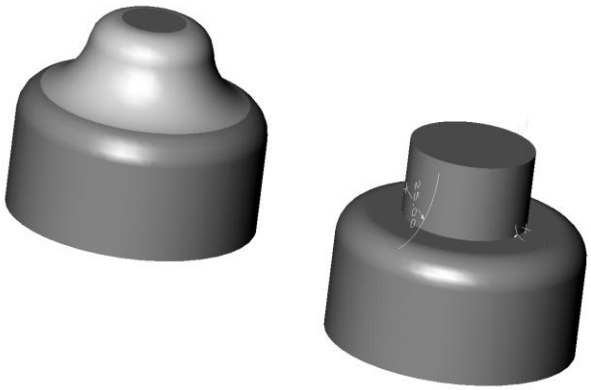
Exit Part.

Edit **Fil_Face_Remove.**



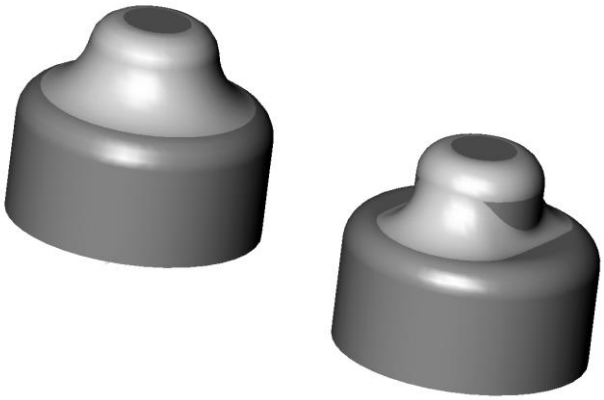
Fillet the RED edge **25mm.**

Fillet the GREEN edge **12mm**. See following figure for results.



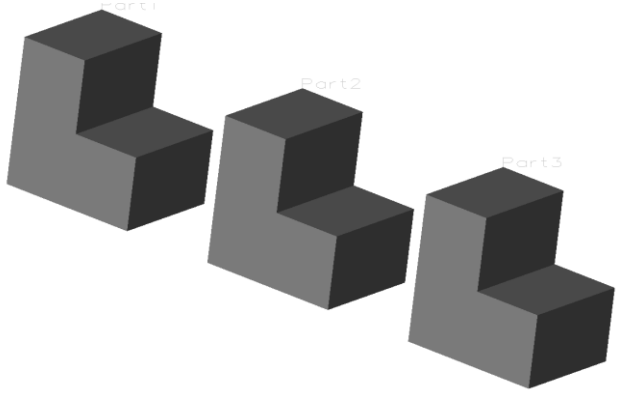
Create a **variable fillet** with the fillet attributes shown.

Fillet the WHITE edge **12mm**. See following figure for results.



Exit Part.

Edit **Fil_L_Shape.**



For PARTS 1 through 3 apply the fillets in the following order. In this step we are going to demonstrate how ZW3D generates fillets based on the order and method you attempt to apply a fillet.

Fillet the RED edge in all 3 parts **10mm**.

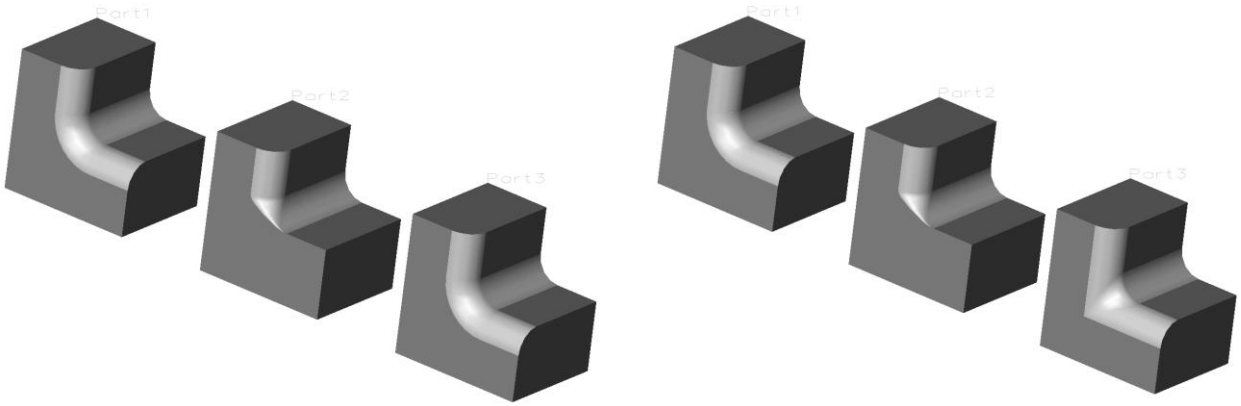
Fillet the GREEN edge in all 3 parts **10mm**.

Fillet the WHITE edge in part 1 **10mm**.

Fillet the WHITE edge in part 3 **10mm**.

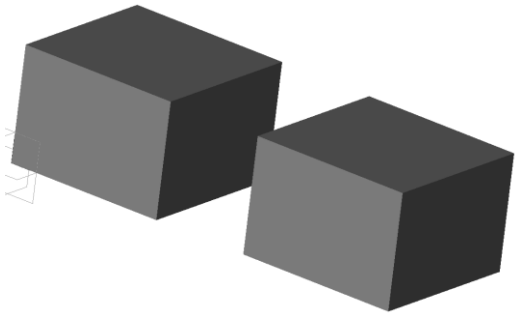
UNDO the last step.

Fillet the white edge in part 3 **10mm** using the chain pick method this time. Hold your <SHIFT> key down to pick the white edge. Notice the difference in the appearance of the fillet by using the chain pick method in part 3 compared to the method used in part 1.



Exit Part.

Edit **Fil_Setback.**



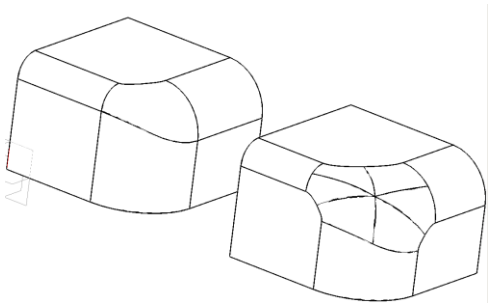
Apply next few steps to the left part.

Fillet the RED edge **10mm**.

Fillet the GREEN edge **20mm**.

Fillet the WHITE edge **30mm**.

See left figure below for results. Pay attention to the result of the corner fillet. In the following step we are going to apply some control to that corner using the RELIEF option.



Apply next few steps to the middle part.

Fillet the RED edge **10mm**.

Fillet the GREEN edge **20mm**.

Now we are going to apply a fillet to the WHITE edge using the RELIEF option. Fillet the WHITE edge **30mm** with a **RELIEF of 10**. See right figure above for results. The higher the setback (relief) value, the more smoothing you will see at corner patches.

<Right-click> Redefine Last and change the **RELIEF to 20** and again to **25** to better understand this option.
If the Redefine Last command responds with "*The last operation did not create a feature",* highlight the last fillet you created and <Right-click> Redefine.

Exit Part.

Begin a new part and name it **FILLETING**.

Pick the **Block** command from the **Shape** Tab. 

Pick **(0,0,0)** for the **1st point**.

Pick any other point on the screen for the **2nd point.**

Change the **Length, Width and Height** values to **90**.

Pick **OK.**

Your current model should be a simple cube, 90mm x 90mm x 90mm.

We are using a very simple part in the example so we can focus on the filleting options we are about to learn instead of getting bogged down with complex geometry.

Let's begin applying some fillets to the corners.

Pick the **Fillet** command from the Shape Tab.

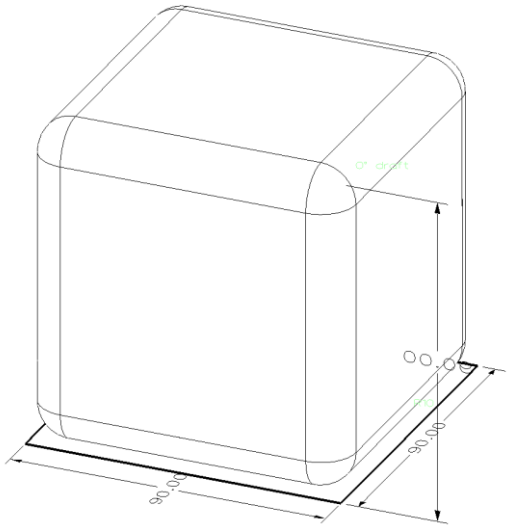
ZW3D can fillet all edges of a model in one command using any of the various entity selection methods.

<Right-click> Pick All.

Dynamically adjust the fillet radius to a reasonable size by picking the up or down arrows in the options form. ZW3D will verify that all edges can accept the fillet size you attempt to create. If the fillet fails when attempting to fillet all edges, try a smaller fillet.

Your model should appear similar this.

Pick the **Turn dimensions ON/OFF** command from the Document Awared toolbar. The dimensions should appear in your model as shown in the figure below.



Go to the **Edit pull down > Preferences.**

Change the units to **inches.**

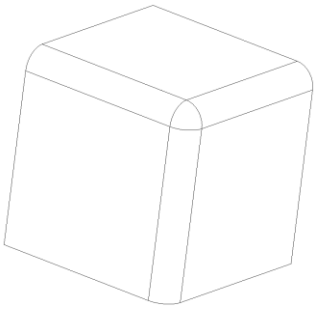
The dimensions on your screen should update automatically to reflect inches. If they do not, try a redraw **“F5”.**

Pick **UNDO** from the utilities toolbar until the fillets you created are gone. This will also set the units back to millimeters.

Pick the **Fillet** command.

Pick 3 edges to create the fillets as shown in the figure below.

Specify the Radius to be **10mm.**



Corner Radius without Relief

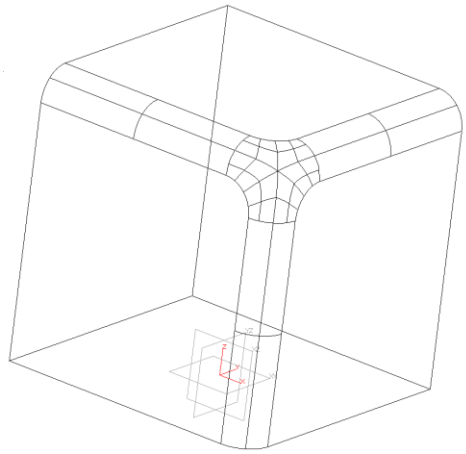
Let’s make a change to the fillets we just created.

<Right-click> Redefine Last.

Set the Relief option **10**.

The Relief option is used to control the amount of smoothing at a corner patch. With the relief set to zero (its default value), the filleting algorithm decides where to trim the rails of the fillets meeting at a corner.

Compare these results with the figure above.



Corner Radius with Relief = 10

In this next example we are going to use options that allow us to control the fillet in a free form manner of speaking.

Pick **UNDO** to remove all fillets from your model.

Pick the **Fillet** command again. DO NOT pick the edges.

Change the radius to 15 and hit <ENTER> on the keyboard.

Pick the **Vertex define** button in the **Advanced** options form.

Pick the vertex shown in the left figure below as the Vertex to Fillet.

Vertex allows you to pick a corner to be filleted instead of having to pick three edges.

Note: The edge radius displayed in the options form works in conjunction with the vertex fillet radius that will be displayed in the next form.

The value of the vertex fillet is 1.5 X the edge radius. You can go higher for more smoothing. A lower value may give undesirable results.

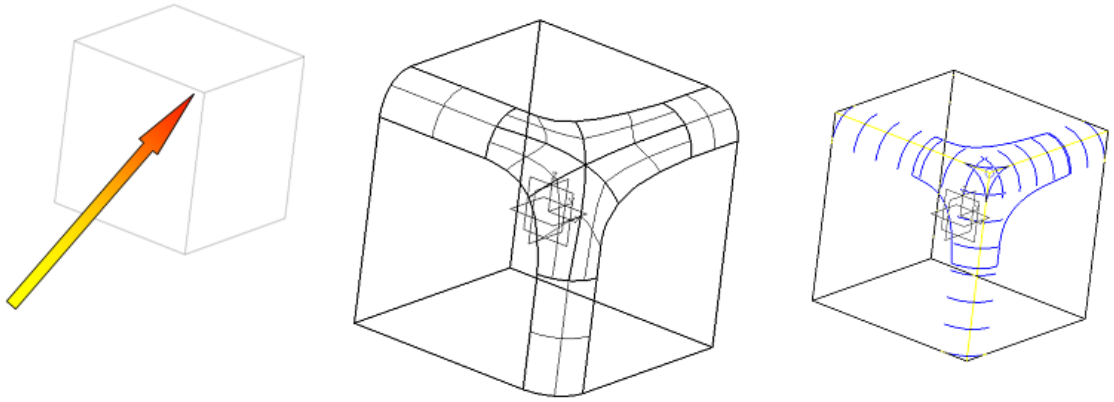
Specify a value and hit **OK** or use the up and down arrows.

Pick the RIGHT edge and specify a **relief** value by adjusting the Up and Down arrows.

Pick the LEFT edge and specify a **relief** value by adjusting the Up and Down arrows.

Pick the MIDDLE (Vertical) edge and specify a relief value by adjusting the Up and Down arrows.

Complete the command to review your results.



Completed fillet with setback (relief) values of 55, 35 and 40

In the final step of this lesson we are going to modify the face setback (relief) of the fillet just created.

The FACE option allows for explicit control over the face setback at the spherical radius of corners. This control is independent of the fillet radius. This option allows you to design a corner patch and see it echoed on the part before the fillet operation is complete.

The corner patch can be edited before the fillet is complete by selecting the Vertex option again and then selecting this Face option again. It can be edited afterwards with the Redefine Feature command.

Highlight the fillet just created.

<Right-click> Redefine Last.

Pick the **Vertex Define** button in the form.

Pick the same vertex.

The fillet that you are in the process of redefining should preview on the model.

<Middle-click> at the radius option to display the face options form shown below.

Pick the **Face Define** button from the form.

Pick the TOP face and specify a face relief value of **25**.

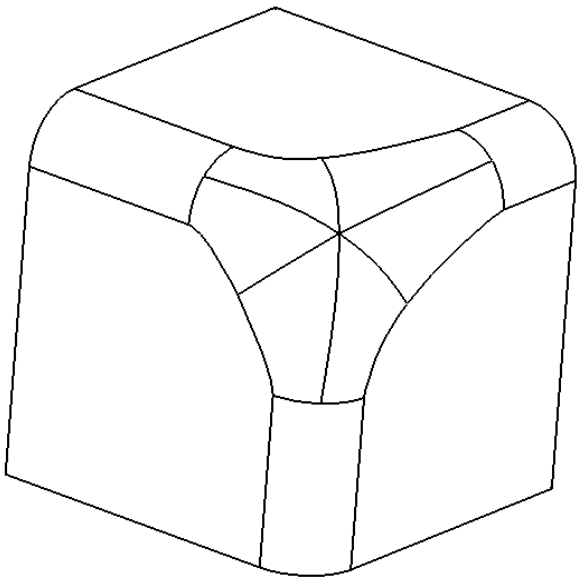
Pick the **Face Define** button from the form again.

Pick the RIGHT face and specify a face relief value of **30**.

Pick the Face Define button from the form again.

Pick the LEFT face and specify a face relief value of **28**.

Complete the command and review your results with the following figure.



Completed Model

Finished.

Exit Part.

Begin a new part and name it **WHEEL.**

Pick the **Revolve** command.

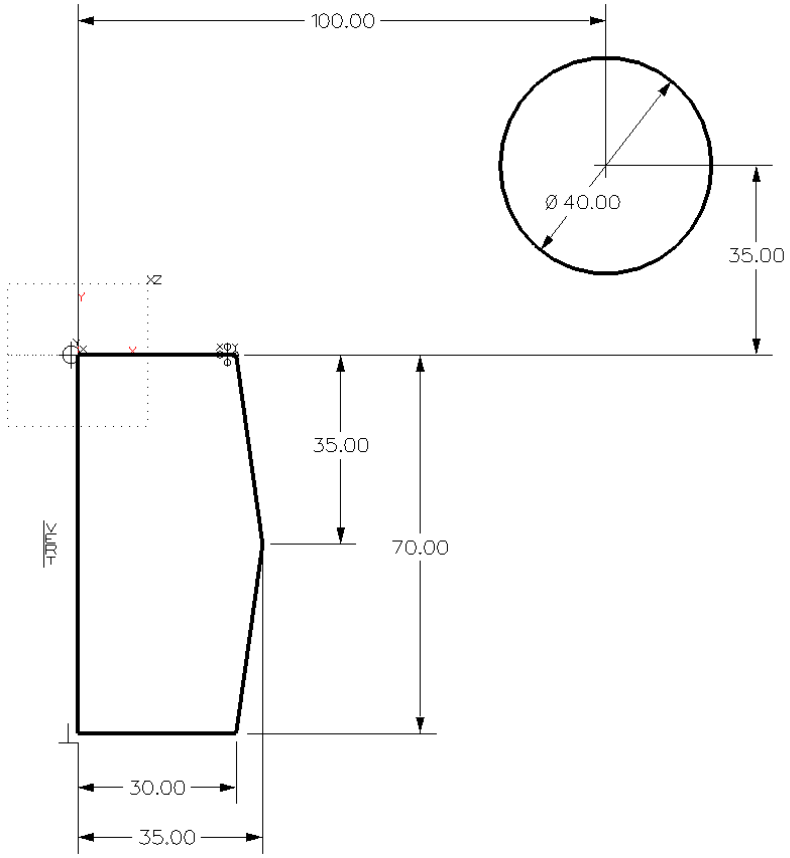
Pick the **XZ Plane** for the new sketch.

Create the sketch shown in the figure below.

Since we have done a lot of sketching already, we will leave the method for creating this sketch up to you. Just three points we would like you to adhere to.

Add the dimensions as shown in the sketch.
Match all dimension values.

Create an **Anchor constraint** at 0,0.



Use the **Constraint Status** command to verify your sketch is “well defined”.

<Right-click> Exit Sketch.

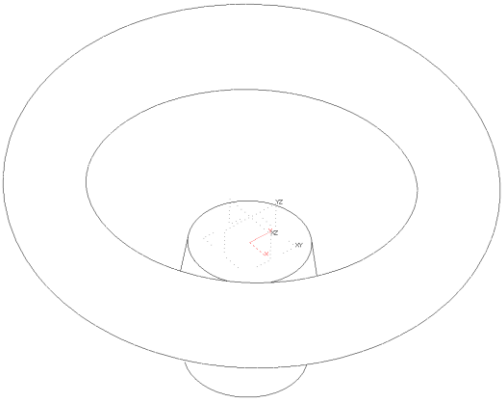
The values for the revolution are:

Method: Base.

<Right-click> Z-Axis for the revolve axis.

Start = **0,** this is the default value, so you can **<Middle-click>.**

End = **360,** this is the default value, so you can **<Middle-click>.**



Let’s create the mounting hole in the center portion of the wheel as shown in the following figure.

Pick the **Extrude** command and create a new sketch.

Turn **Reference face edges** on in the options form.

Pick the **TOP face parallel to the XY Plane** for our new sketch. Do not pick the XY plane.

Create the sketch shown in the following figure.

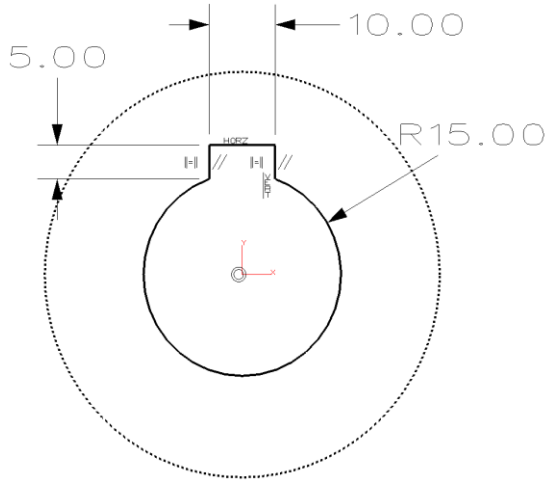
Again, since we have done a lot of sketching already, we will leave the method for creating this sketch up to you. Here are a few helpful hints though:

Our sketch was created using 3 commands, **Circle** - **boundary method**  **, corner rectangle**  and **Trim**  .

Add the necessary geometric constraints and dimensions.

Modify all dimension values to match.

Pick the **Show Target** command from the Document Awared toolbar to display your sketch.



<Right-click> Exit Sketch.

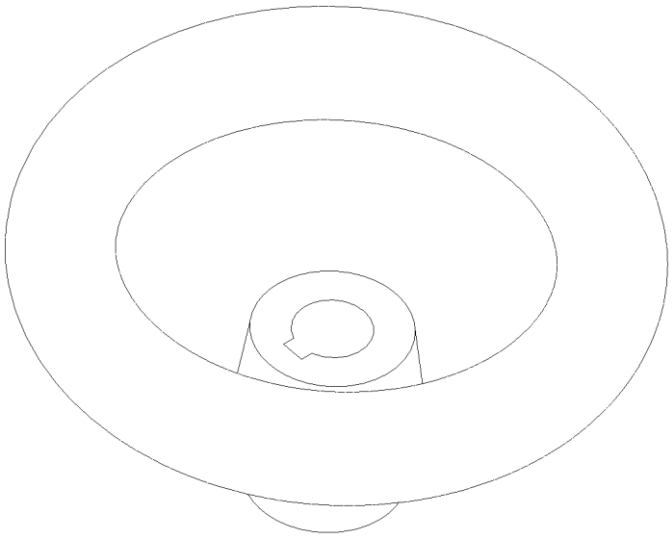
The values for the extrude are:

Method: **Remove.**

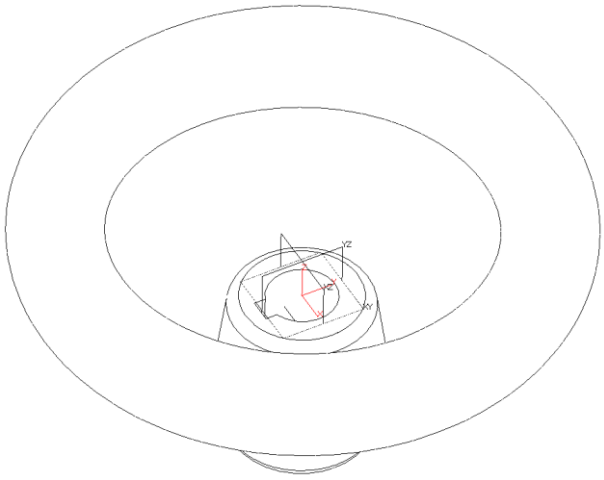
Start = **0.**

End = **<Right-click> Thru All.** (We could also use Boundary Face and pick the bottom face.)

No Draft Angle.



Create a **5mm fillet** on the top and bottom edges of the mounting hub as shown in the figure below.



Let's create a sweep path for the first wheel spoke.

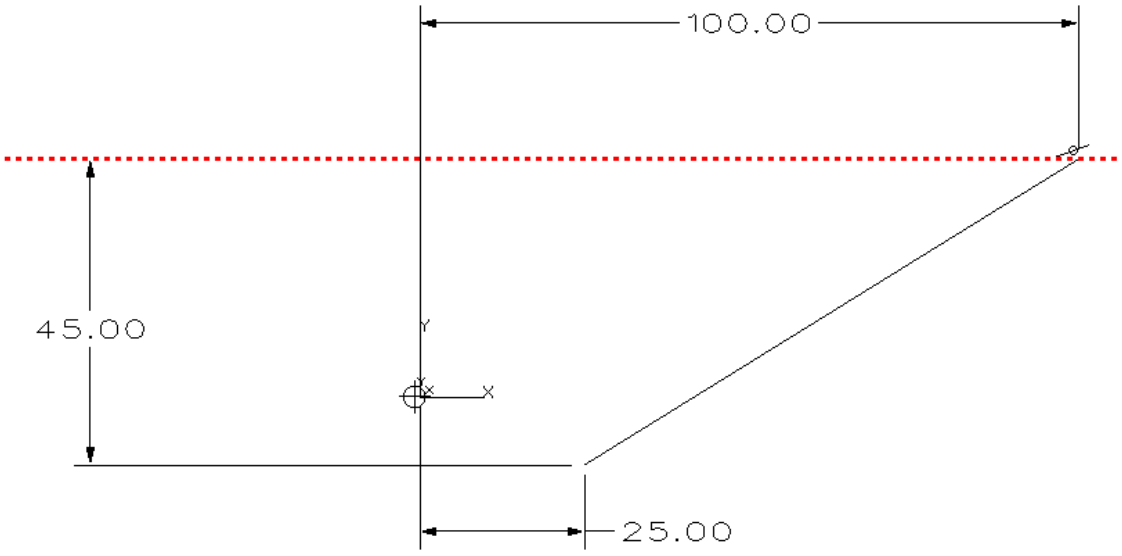
<Right-click> Insert Sketch.

Pick the YZ Plane or type YZ in the command prompt for the new sketch.

<Right-click> Draw and create the line shown in the following figure.

<Right-click> Dimension the entire sketch as shown in the figure below. Match all dimensions.

<Right-click> Exit Sketch.



To complete the first spoke we need to create the sketch to sweep through the line we created earlier.

Pick the **Sweep Shape** command.

Middle-click to create a new sketch.

Pick the XY Plane for the new sketch.

Create a circle with a Radius of **10**. Place the circles center at 0,0.

<Right-click> Exit Sketch.

The Sweep command continues asking the necessary questions to finish up. Notice the circle we just created is already highlighted without picking it. ZW3D automatically uses this sketch as the cross section for our swept feature.

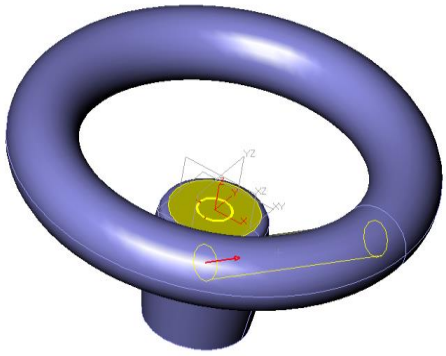
Pick the line we created earlier for the sweep path.

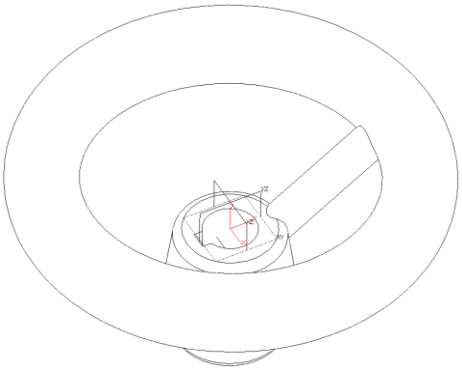
Change the **Frame to Selected** in the Advanced Options form.

Select the top face of the wheel hub.

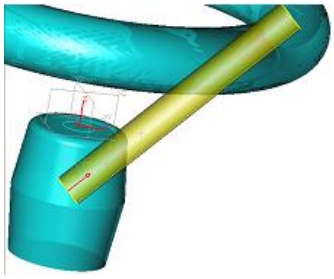
Change the Sweep type to **Add.**

Pick **OK** to complete the command.

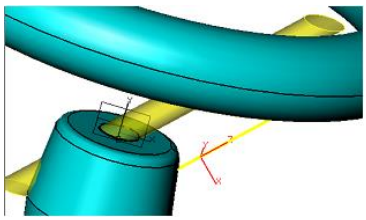




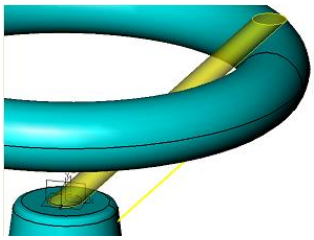
Natural - The default reference frame of the profile or entity is used to control the sweep along the curve. The sweep begins at the start of the curve path.



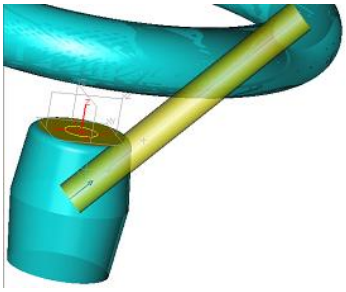
At Profile - The frame is built on the point on the sweep path where the profile is. This will make the surface interpolate the profile but the profile will not be at the beginning of the surface.



At Path - The frame is built at the start of the sweep path.



Selected - You are prompted to select a datum plane or part face whose default reference frame will be used to control the sweep. The reference frame and the entities being swept are theoretically joined as a rigid body. The reference frame is then moved to the start of the curve path.



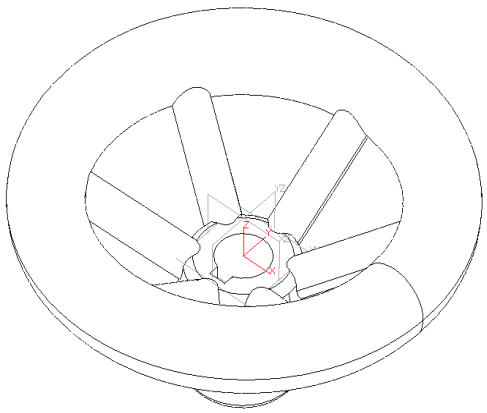
Pick the **Pattern** command from the **Shape** Tab and then select the **Circular** method from the Required Inputs section of the Options Form. (see form below).

Pick the spoke as the feature to pattern. (set filter to feature)

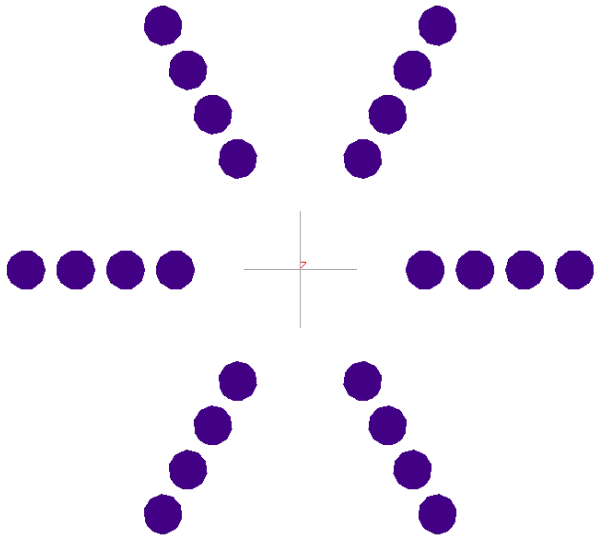
<Right-click> Z-Axis for the axis of rotation.

Fill in the **Number** and **Angle** as shown at right.

Pick **OK.**



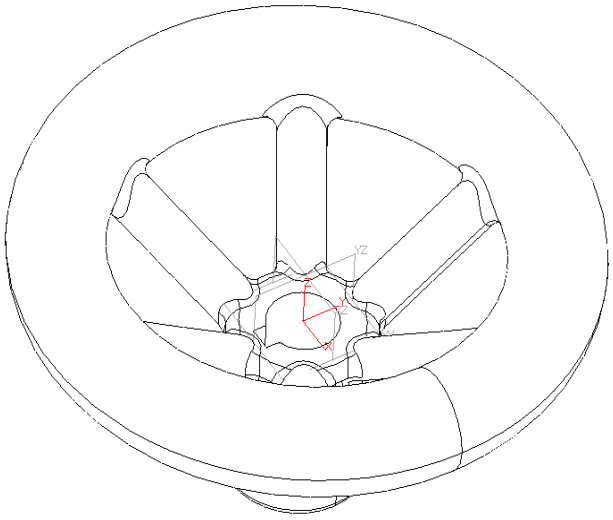
The following is an example of a circular pattern using the second number and spacing fields.



Create a fillet, R=5 at both ends of all the spokes.

Press and hold the <SHIFT> key while you pick to chain pick all tangential edges.

If you need to unpick an entity, use <CTRL> Pick and pick that entity.



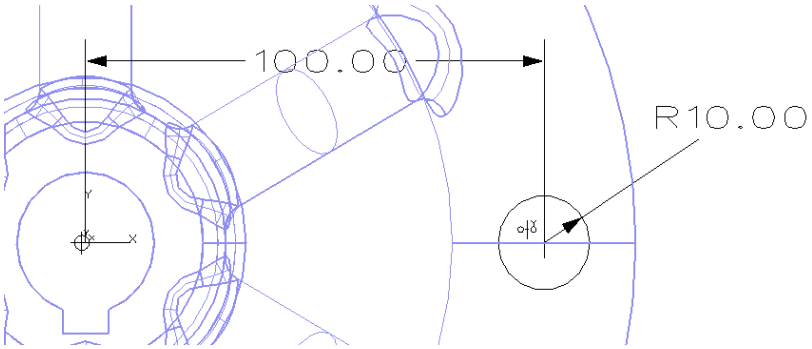
The final operation for our wheel design is to cut a hole through the handle portion as shown in the figure below.

Pick the **Extrude** command from the Shape Tab.

Activate the XY Plane for the new sketch.

Create a **circle**.

Radius **10mm.**
Center point at **100,0.**



Or add "**Constrain points horizontal"**  **.**

<Right-click> Dimension.

Add the 100mm horizontal dimension.

<Right-click> Exit Sketch.

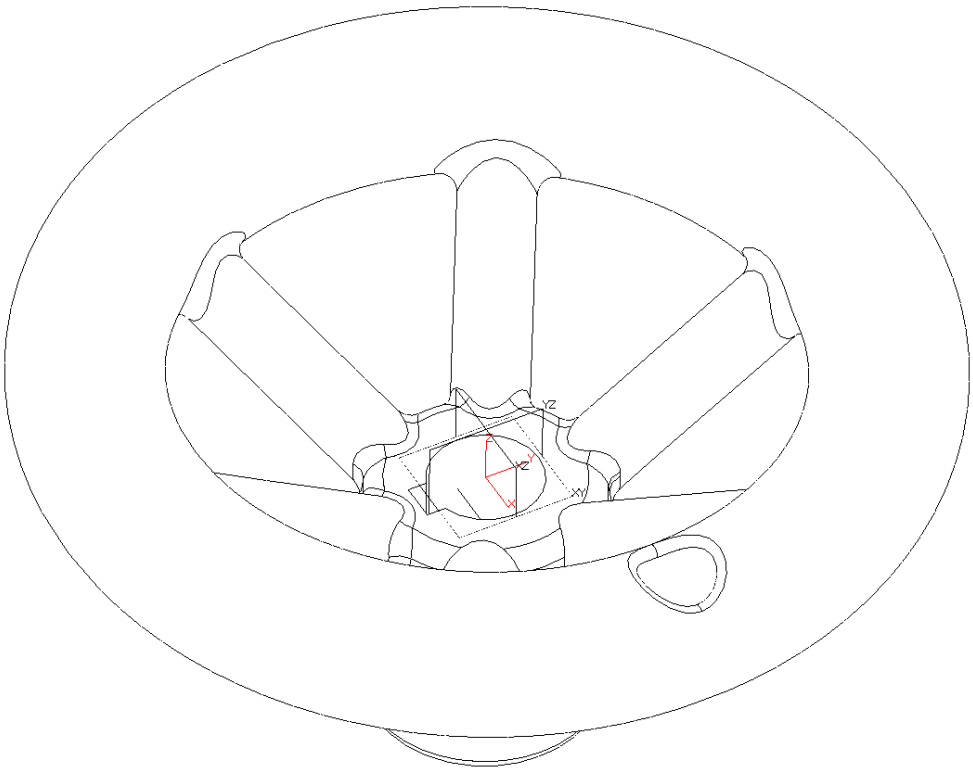
The extrude option form should reappear.

Pick the **Remove** option.

Specify the **Start** to be **65.**

Specify the **End** to be **0.**

Create **2mm fillets** at the edges of the hole.



Save and **exit** the part.

Chapter 8

Working with Multiple Components

Refer to the HELP Manual and review the following commands.

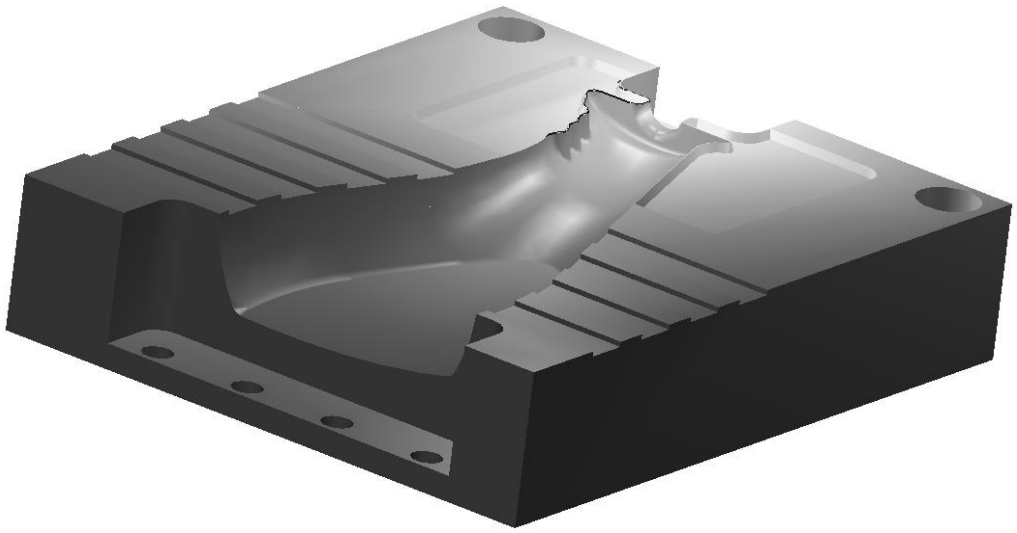
Insert Component
Component – Merge into Active Part
Combine (Add, Remove, Intersect) Shapes
Trim Shapes
Divide Shapes
Replace Faces
Tie Off Self-intersections
Sweep – Simple Shape
To Create a Rod
Variable Shape
Variational shape
Pattern (3D)
Mirror (3D)
Move
Along a direction
Point to point
Punch

Lesson 8

Working with Multiple Components

Designing a Blow Mold

This exercise will take you through several ZW3D commands to design a blow mold as shown in the figure below. The bottle we will use for this example is a fully parametric model created in ZW3D. It could also be an IGES file or Point Cloud file you might receive from your customer.



OPEN the ZW3D file "**Fundamentals.Z3**".

Pick the **Part/Assembly** icon from the **ZW3D** root to create a **Part** object.

Name it "**Blow_Mold**".

We will begin by inserting the component called "**Bottle**". This is the bottle that the blow mold will be designed for. It is an existing ZW3D object. If you would like to view how it was designed in ZW3D, **OPEN** this object and **Step Thru its History**.

<Right-click> Insert Component.

Pick **Bottle** from the **File/Part** list of the Options Form.

From the keyboard, type ",," to insert the bottle at 0,0,0 or <Left-click> at 0,0,0.

The 0,0 origin in the source object will be placed at the 0,0 origin in the destination object.

Uncheck **Align after insertion** from the **Optional Inputs** form.

Pick **OK** to accept the settings.

Your screen should display the bottle as shown in the following right figure.

The next thing we want to do is adjust the orientation of the bottle to suit our model designers' requirements relative to machining. We want to rotate the bottle about the X and Y-axes.

Pick the **Move entities along a direction** command within the **Move** command under the **Assembly** Tab.

Verify that your pick filter is set to **Component** or **All**. <Shift><Right-click> to verify.

For Entities to Move, pick the bottle. This can be done with a single pick due to your pick filter.

For the Move Direction, **<Right-click> X-Axis.**

Distance is 0. Remember; you can <Middle-click>.

Angle is -90. Pick the increment or decrement arrows to view the rotation dynamically.

Remember positive rotation is CCW and we are rotating about the positive X-Axis.

For the Rotation Direction, pick the appropriate increment or decrement button until the value is –90 degrees. Remember positive rotation is CCW about the axis.

<Middle-click> to restart the **Move entities along a direction** command.

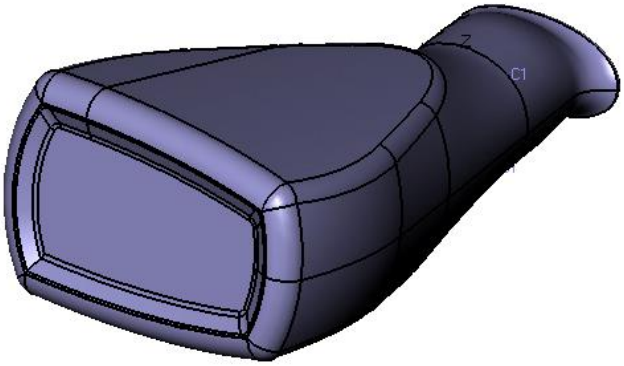
Pick the bottle shape again.

Set the **Distance to 0.**

Set the angle is **-90 degrees** about the **Y-Axis**.

Complete the command.

See figure below for final results.



The next step is to begin modeling the stock from which we will cut the cavity.

<Right-click> Insert Sketch.

<Middle-click> to accept all of the defaults.

Pick the **Rectangle - Corner** command from the **Sketch Tab**.

Draw and **constrain** the rectangular sketch as shown in the figure below.

The grid can be turned off if you prefer or left on. You decide.

Some suggestions that will help in the constraining process are:

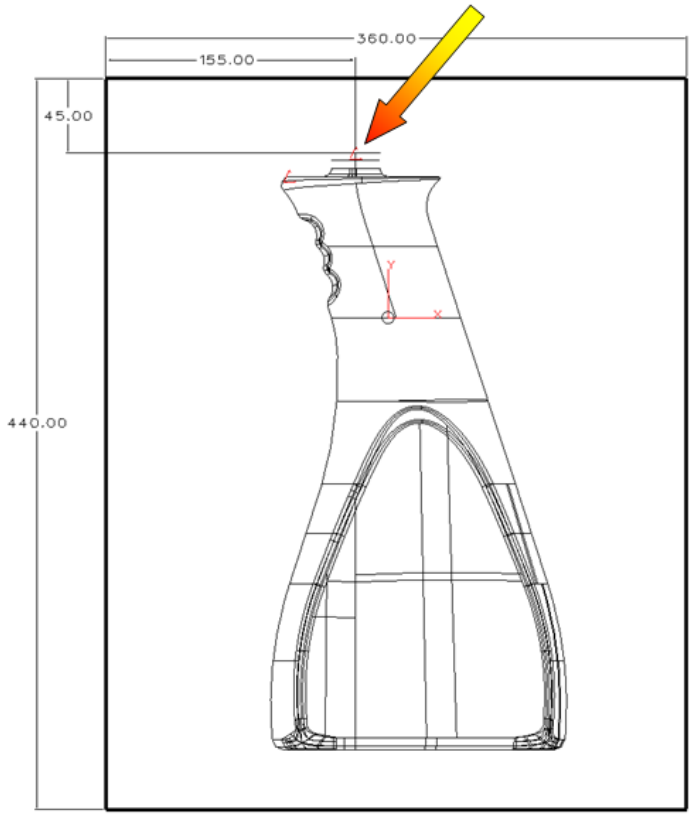
Apply the necessary dimensional constraints by using the <Right-click> **Dimension** or  .

Press the "**F7**" function key to attach dimensions to model edges. "F7" instructs ZW3D to snap to critical points anywhere in the active part or assembly.

Do not use the **Auto-Constrain** command as your first choice for constraining. This will create constraints that might be undesirable for this particular design.

A good rule of thumb to follow with **Auto Constrain** is to manually create the required geometric and dimensional constraints first. If the sketch still requires some constraints, use the **Auto Constrain** command to finish the process by <Right-clicking> **Constrain.**

Press "F7" to pick the top edge of the model when dimensioning to it. As indicated below.



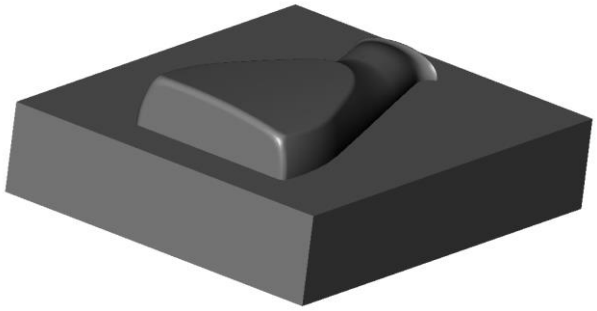
Pick the **Constraint Status** command to verify the sketch is **well defined**.

When finished constraining the sketch, <Right-click> **Exit Sketch** to return to the model mode.

Pick the **Extrude** command from the **Shape** Tab to extrude the rectangular sketch.
Set the **start to be 0** and the **end to be –91** and **no Draft angle**.

<Middle-click> or pick **OK** to complete the command.

Your model should appear as shown in the following figure.

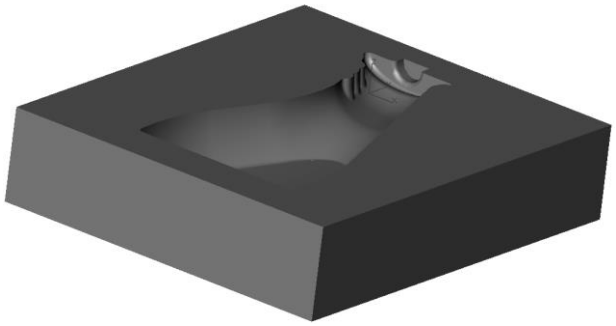


The bottle component we inserted earlier is still associated to its source (parent) object. If that source object is changed, this model will update to reflect those changes. Merging allows us to edit this component in the current part.

Place your cursor over the bottle to highlight it.

<Right-click> Merge . **Method: Remove** . Pick the bottle as **component**.

<Middle-click> or pick OK to complete the command.



Let's review what we have done so far. **History Play**.

Pick the **Replay next operation** button located in the bottom of the History Manager. Replay each operation one at a time as a review of what you we have accomplished so far.

Each time you pick this button, you will see a feature taken from the pending window and moved up to the features window.

Keep in mind that at any time during this history replay you can make changes to the design by adding, removing or editing features. We will talk more in depth about the stepping thru history later.

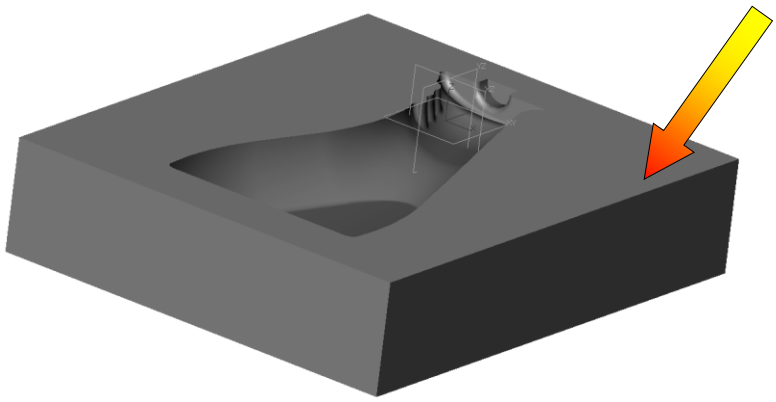
Next we are going to design the push up area and the bottle top area of the blow mold.

<Right-click> Insert Sketch.

Adjust the options form and pick the top face of your design as shown in the figure below.

The Reference face edges option tells ZW3D to extract only model edges that are coplanar to the face you are picking. The model edges will appear as hidden or dotted lines on your model and can be used to assist in building other sketches or can actually become part of another sketch.

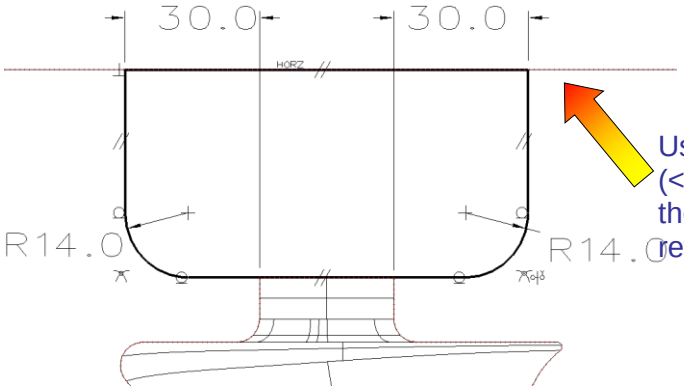
A similar command to the Edges option is Project Reference Curves. This command creates Construction Geometry curves on the active sketch plane by projecting 3D curves. 3D curves could be lines, arcs, curves, or edges. If an edge is modified, all projected construction geometry from that edge will update the next time the active part or sketch is updated.



Pick **View > Show Scope > Show Target** from the top pull down menu to simplify the display while constraining the sketch.

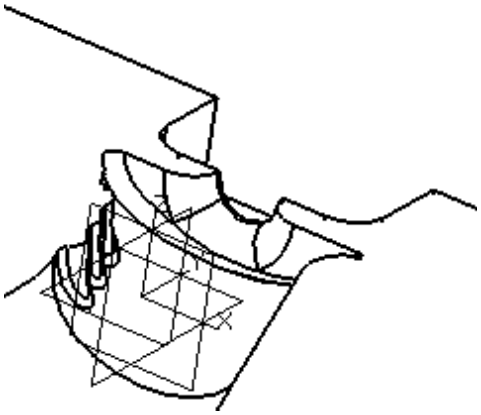
Create and fully constrain the sketch as shown in the following figure. Keep in mind that you might have to erase some dimensions and geometric constraints to achieve this.

These are some of the commands you might consider using.  Pick the icon or use the <Right-click> menu to access these commands.



Use the Default snap option (<Right-click> Default) to snap the top horizontal line to the top reference edge.

Well defined sketch



Pick **Constraint Status** to verify the sketch is well defined.

<Right-click> Exit Sketch.

Pick the **Extrude** command.

Pick the **Remove** icon in the options form.

Pick the sketch you created in the previous step.

For the Start specify **0.**

For the End specify **–32.**

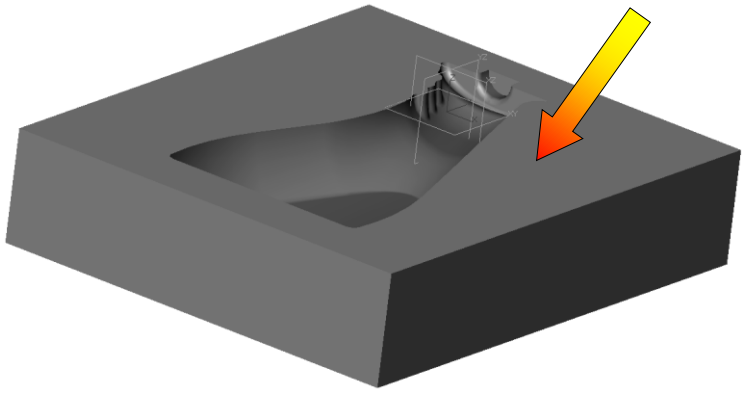
Next we are going to design the push up area and the bottle bottom area of the blow mold.

<Right-click> Insert Sketch.

Adjust the options form and pick the top face of your design as shown in the figure below.

The Reference face edges option tells ZW3D to extract only model edges that are coplanar to the face you are picking. The model edges will appear as dotted lines on your model and can be used to assist in building other sketches or can actually become part of another sketch.

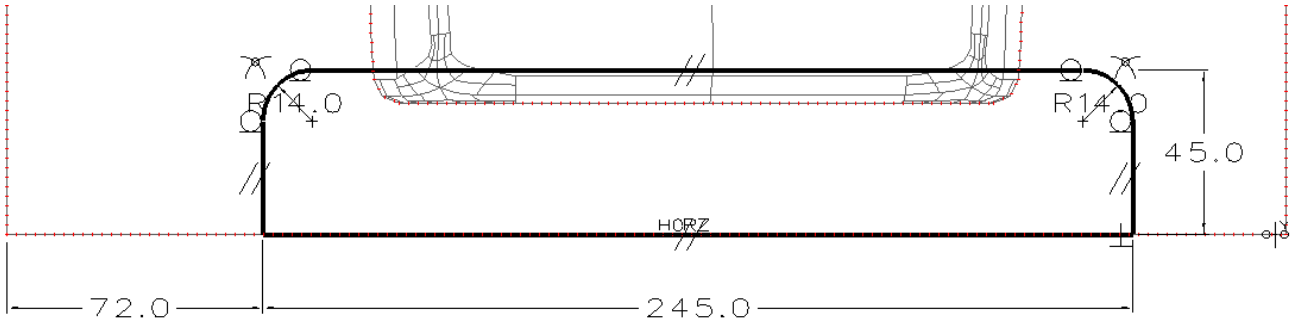
A similar command to the Edges option is Project Reference Curves. This command creates Construction Geometry curves on the active sketch plane by projecting 3D curves. 3D curves could be lines, arcs, curves, or edges. If an edge is modified, all projected construction geometry from that edge will update the next time the active part or sketch is updated.



Pick **View > Show Scope > Show Target** from the top pull down menu to simplify the display while constraining the sketch.

Create and fully constrain the sketch as shown in the figure below. Keep in mind that you might have to erase some dimensions and geometric constraints to achieve this.

These are some of the commands you might consider using.  Pick the icon or use the <Right-click> menu to access these commands.



Well defined sketch

<Right-click> Exit Sketch.

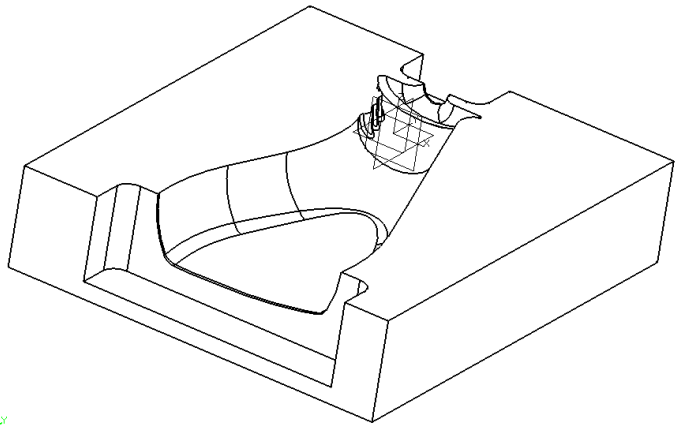
Pick the **Extrude** command.

Pick the **Remove** icon in the options form.

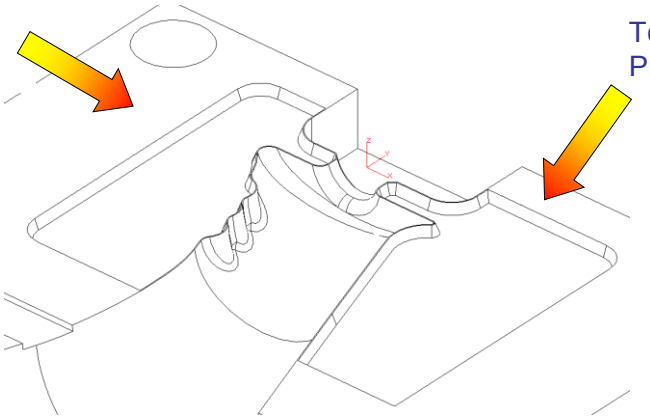
Pick the sketch you created in the previous step.

Set the Start **0.**

Set the End **–76.2** with **no draft angle**.



In these next few steps we are going to design a left and right pinch off for the mold as shown in the following figure. To do this we will borrow from some existing geometry and build the rest.



Designing a Pinch Off

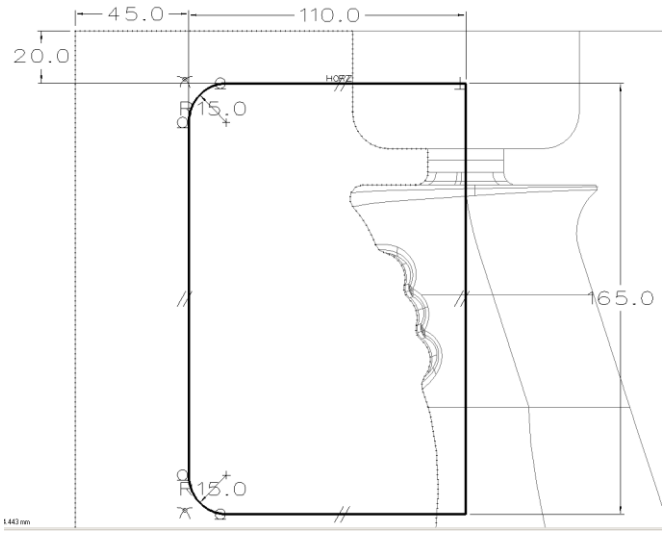
<Right-click> Insert Sketch. Turn **Reference face edges on**.

Pick the top left face of your design. Refer to the figure above to determine the appropriate face.

Create and fully constrain the rectangular sketch shown in the figure below.

These are some of the commands you might consider using.  Pick the icon or use the <Right-click> menu to access these commands.

Notice in this figure the hidden or dotted lines. These are the edges extracted from all model edges coplanar to the sketch plane (face) you picked. We are going to use these edges in combination with the rectangular sketch to build the final sketch for extruding.



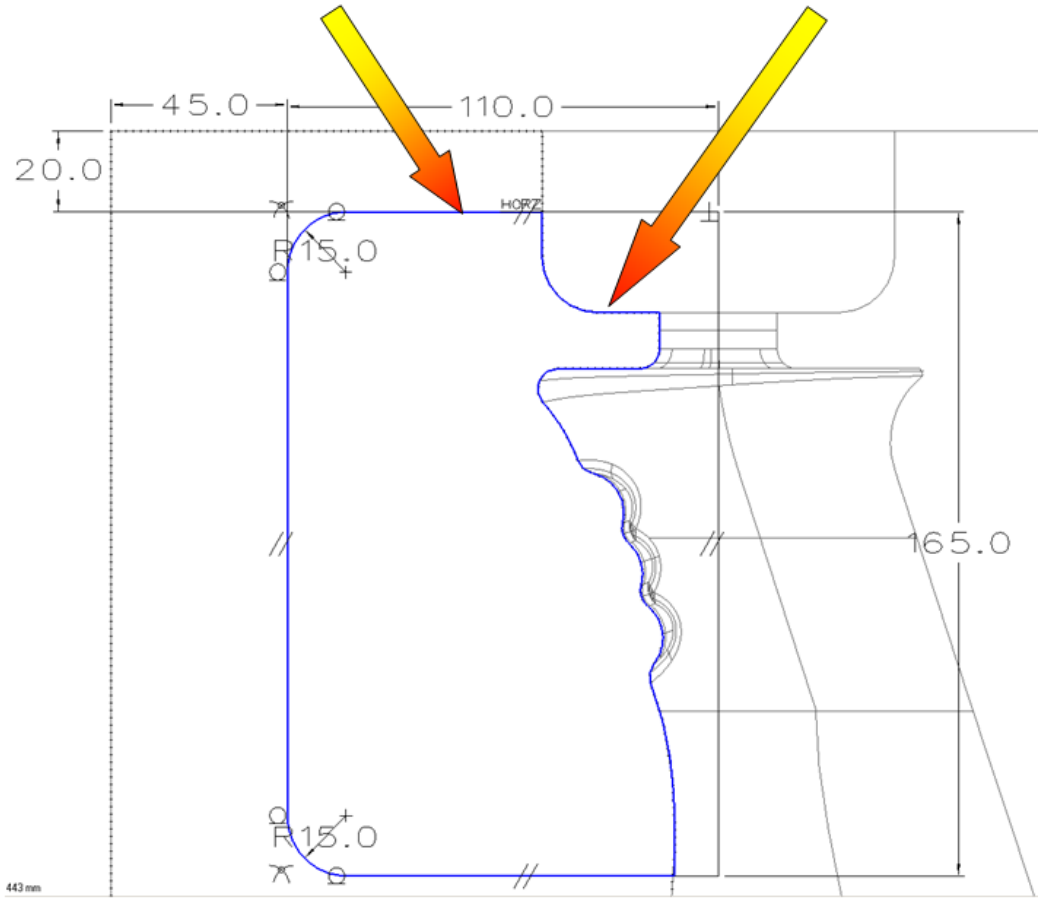
Pick **View > Show Scope > Show Target** from the top pull down menu to simplify the display while constraining the sketch.

Next we are going to use some of the model edges and the rectangular sketch to determine our final sketch.

Pick the **Create Trace Profile** command from the **Sketch** Tab.

Pick near the 2 points as shown in the figure below to create the trace profile. Your final sketch should appear as shown by the thicker lines in the figure below.

Trace profiles are parametric. They are associated to the model edges. As these model edges change, trace profiles update to reflect thee changes.



<Right-click> Exit Sketch.

Pick the **Extrude** command.

Pick the **Remove** icon in the options form.

Pick the trace outline you just created.

Set the **Start** to **0.**

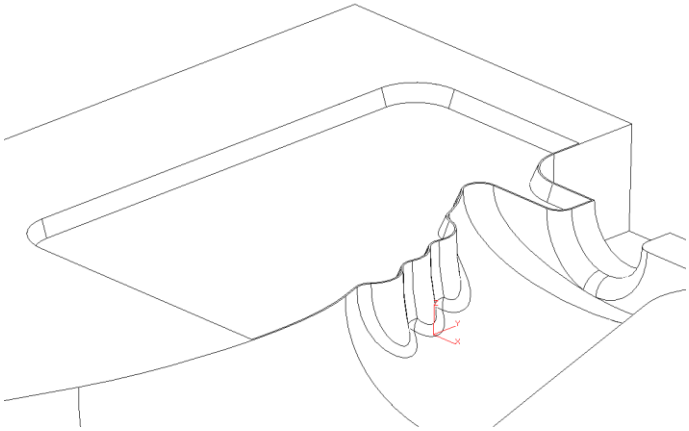
Set the **End** to **–3** (may be +3).

Set the **Draft Angle** to **16 degrees.**

Pick the **Offset** option and set it to **Shrink/Expand.**

Set the Offset distance specify **-.4.**

Your model should appear as shown in the figure below.



Let's create the Pinch Off for the other side.

<Right-click> Insert Sketch.

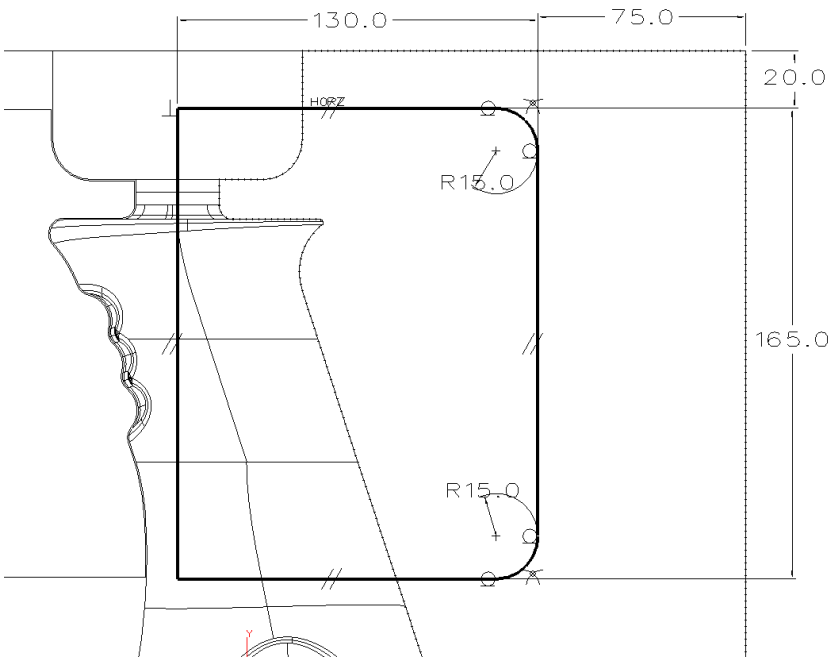
Turn **Reference face edges on**.

Pick the top right face of your design.

Create and fully constrain the rectangular sketch shown in the figure below.

These are some of the commands you might consider using.  Pick the icon or use the <Right-click> menu to access these commands.

Notice in this figure the hidden or dotted lines. These are the edges extracted from all model edges coplanar to the sketch plane (face) you picked. We are going to use these edges in combination with the rectangular sketch to build the final sketch for extruding.



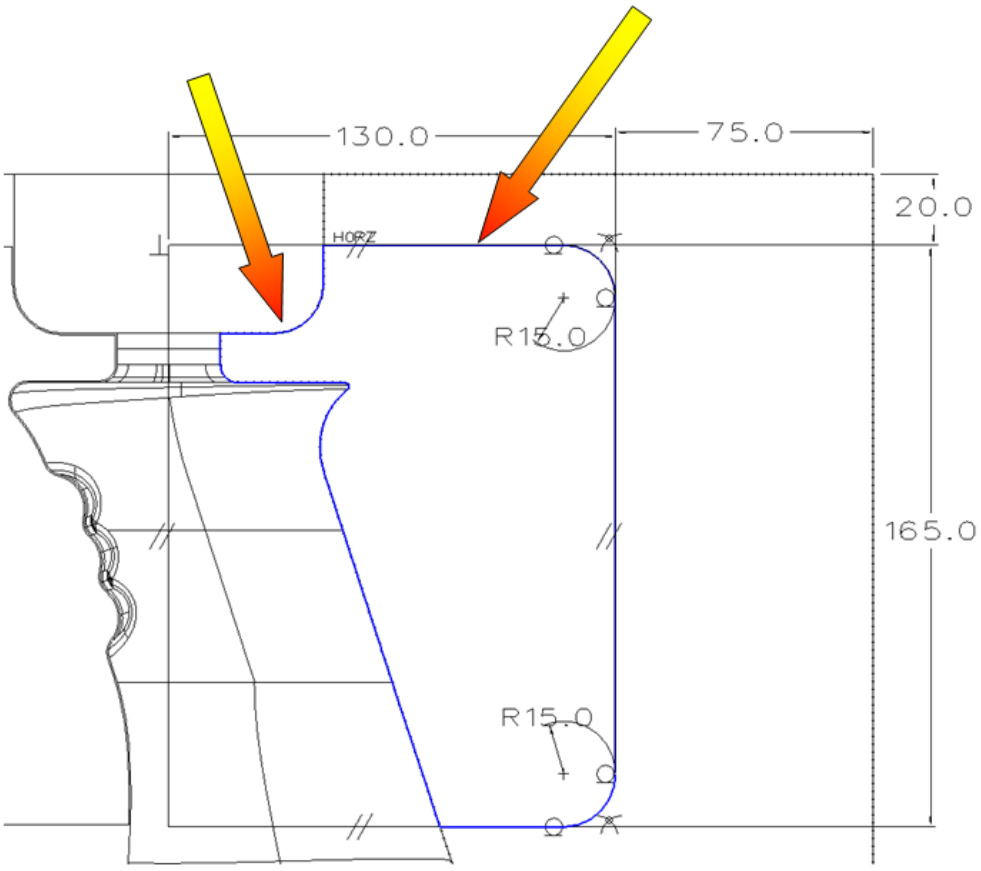
Pick **View > Show Scope > Show Target** from the top pull down menu to simplify the display while constraining the sketch.

Next we are going to use some of the model edges and the rectangular sketch to determine our final sketch.

Pick the **Create Trace** Profile command from the **Sketch** Tab.

Pick near the 2 points as shown in the figure below to create the trace profile. Your final sketch should appear as shown by the thicker lines in the figure below.

Trace profiles are parametric. They are associated to the model edges. As these model edges change, trace profiles update to reflect thee changes.



<Right-click> Exit Sketch.

Pick the **Extrude** command.

Pick the **Remove** icon in the options form.

Pick the trace outline you just created.

Set the Start to **0.**

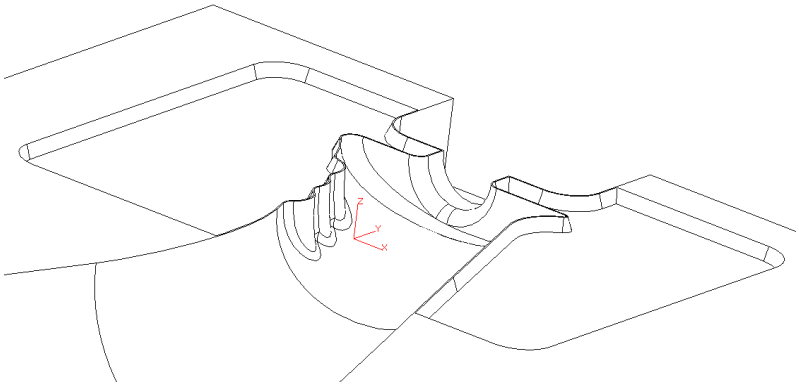
Set the End to **–3.**

Set the Draft Angle to **16 degrees.**

Pick the Offset option and set it to **Shrink/Expand.**

Set the Offset distance specify **-.4.**

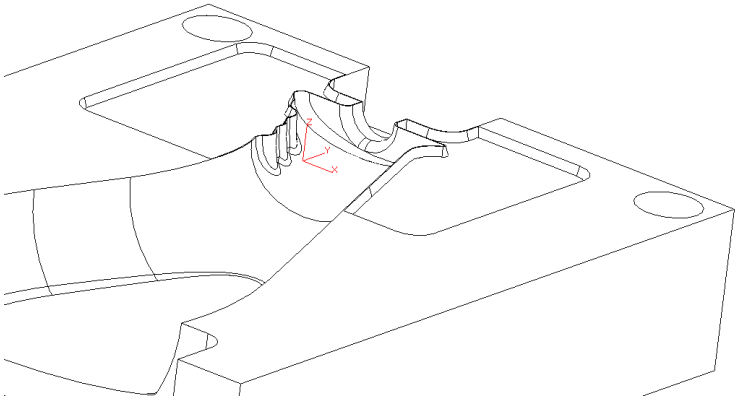
Your model should appear as shown in the figure below.



Let's create the 2 holes as shown in the following figure.

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

Hole type = Simple; **Dia** = 38.1; **End** = Thru-All. See other fields in next step.



Place the Holes on the top **face** of the Blow Mold as shown in the following figure.

For the Hole Locations **<Right-click> From 2 Lines.**

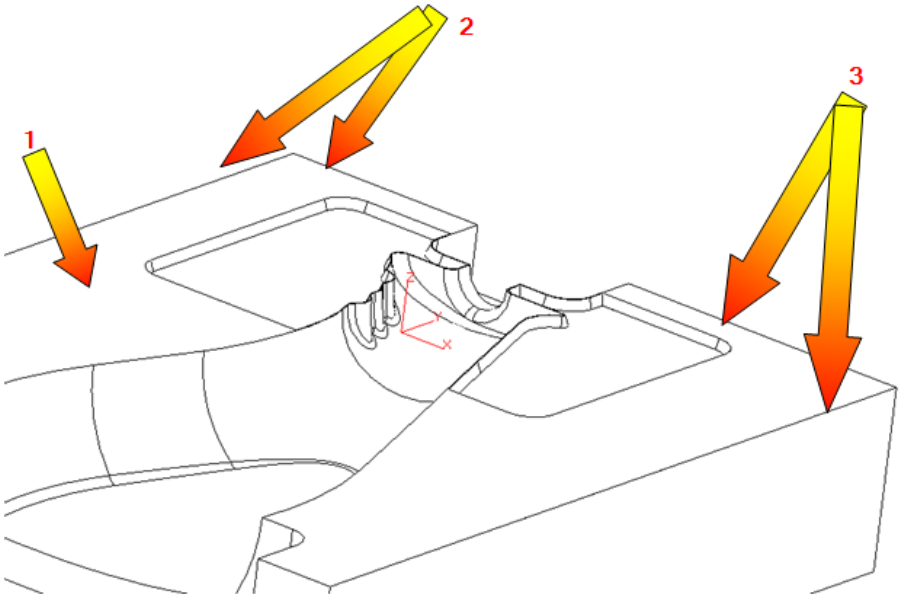
Pick the outer 2 edges to align the 1st hole from as indicated in the following figure.

The holes will be **25.4mm from both edges**. Carefully follow the command prompts.

Pick the outer 2 edges on the other side of the blow mold to align the 2nd hole from as indicated in the following figure.

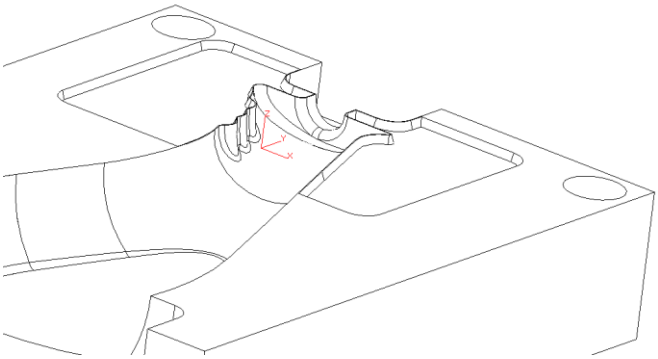
The holes will be **25.4mm from both edges**. Carefully follow the command prompts.

<Middle-click> to accept the remaining default prompts.



1: Pick this Face to place the holes on. 2: Pick these 2 Edges to place the 1st hole.
3: Pick these 2 Edges to place the 2nd hole.

See the result below.

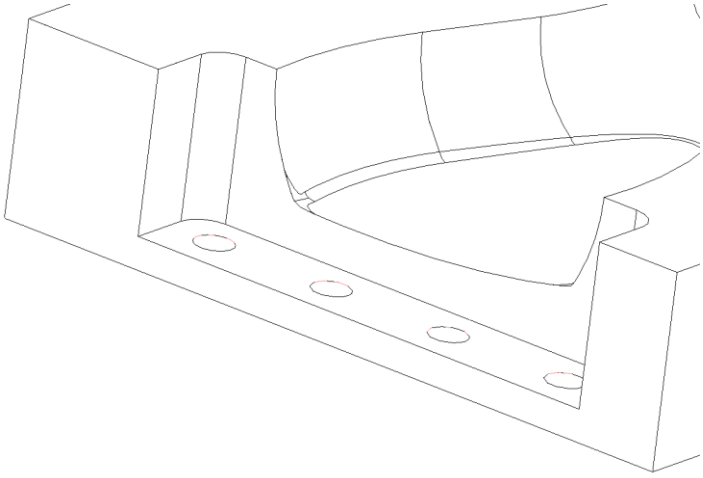


Let's create the mounting holes for the push up as shown in the following figure.

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

Hole type = Simple; **Dia** = 19.1; **End** = Thru-All. See other fields in next step.

See the result below.

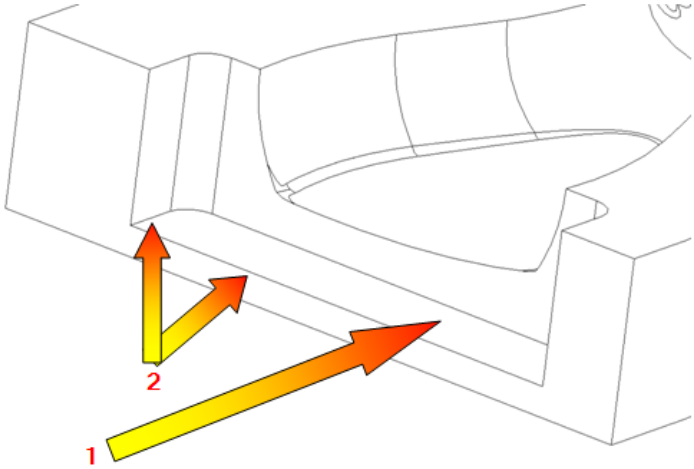


Place the left most hole on the face indicated in the following figure.

For the Hole Locations **<Right-click> From 2 Lines.**

Pick the 2 edges to align the 1st hole from as shown in the following figure.

Place the hole **25mm from the left edge** and **22.5 from the front edge**.



1: Pick this Face 2: Pick these edges

Let's create copies of the hole for a total of four. Pick the **Pattern** command from the **Shape** Tab and then select the **Linear** method icon from the **Required Inputs** section of the Options Form.

For the **Features** to pattern, pick the hole you just created.

For the direction of the Pattern you have a couple choices. You could <Right-click> **X-Axis** or you could pick the linear edge that defines the length of the opening the hole is in. Pay attention to the direction vector that appears when picking an edge.

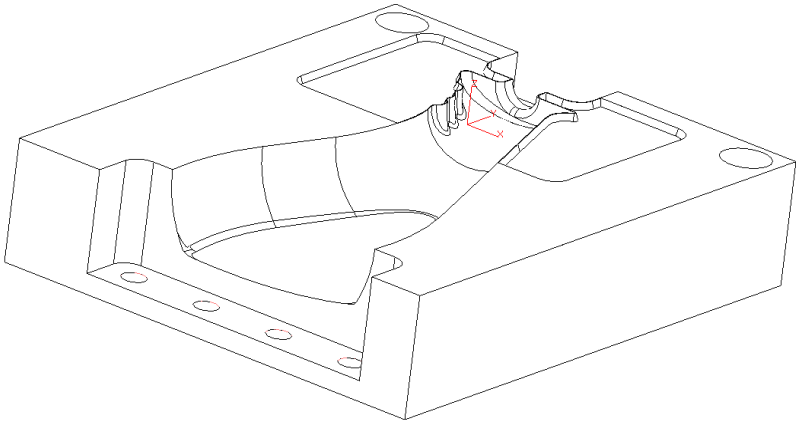
If you pick the linear edge, make sure you pick it at the correct end. A direction vector will appear indicating which way the pattern will be placed.

The Total number of features will be **4.**

The Spacing between each feature is to be **65mm.**

<Middle-click> to accept the remaining defaults.

Your model should appear as shown in the figure below.



Let's verify the location of the holes with one of the measuring commands available in ZW3D.

Select **Inquire > Distance**  **> Point – Point**  **.**

Measure the distances between the Hole centers and the edges. Measure a few areas on the model to familiarize yourself with this command and its result form.

After creating the alignment holes we can notice a slight error in our design. Look at the figure of the blow mold above to see what we are talking about. The left alignment hole is too close to the cutout defining the left pinch off. Let's adjust the sketch for the left pinch off so there is more clearance between the hole and the cutout.

<Shift><Right-click> and make sure your pick filter is set to **ALL**.

Right-click Edit on Sketch4. This command does not step through the history but does open the sketch for edit. You can see the entire part.

We need to adjust the **45mm** dimension to **65mm** and the **110mm** dimension to **90mm**.

There are several ways we can go about fixing this design problem.

We could **step through the history** and **Edit** the sketch for the left side pinch off.
Right-click Redefine on Sketch4 in the History list. This will step through the history automatically and stop with sketch4 open for you to edit. Then exit.

We could leave our pick filter set to **ALL**, highlight the entire sketch and double click on it. This will activate the sketch. Then we double click on the dimensions to change their values.

Using this technique, when you edit the first value the entire history will automatically regen. You will have to repeat the procedure for the second value.

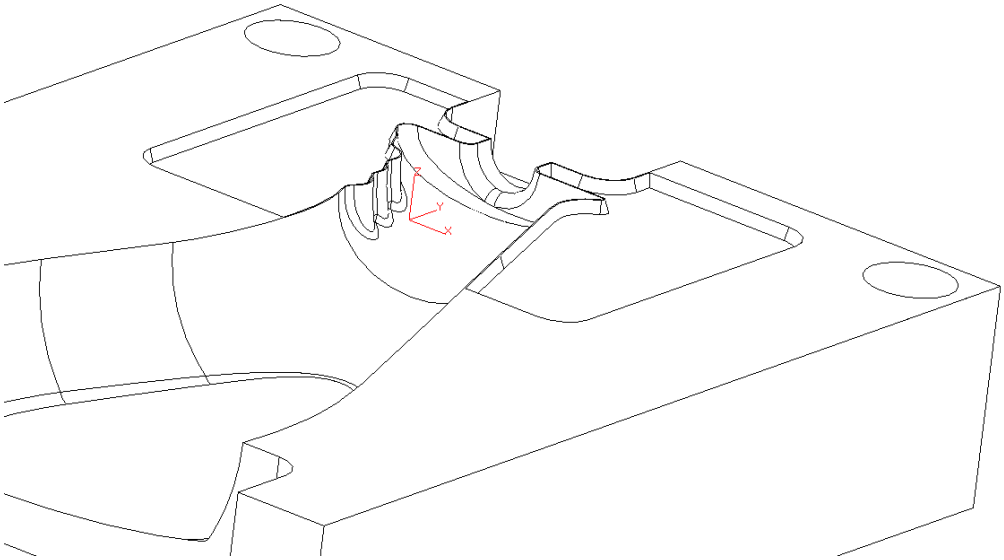
Or

Start by using technique 4 above but after you double click on one of the dimensions, click on the "**Delay Solve**" button.

Make the first edit.

Double click on the second dimension; now turn the "Delay Solve" off. The history will now regen with both changes.

Your model should appear as shown in the following figure.



Let's finish our blow mold design by adding some alignment grooves.

<Right-click> Insert Sketch.

Pick the top face of the blow mold for the new sketch.

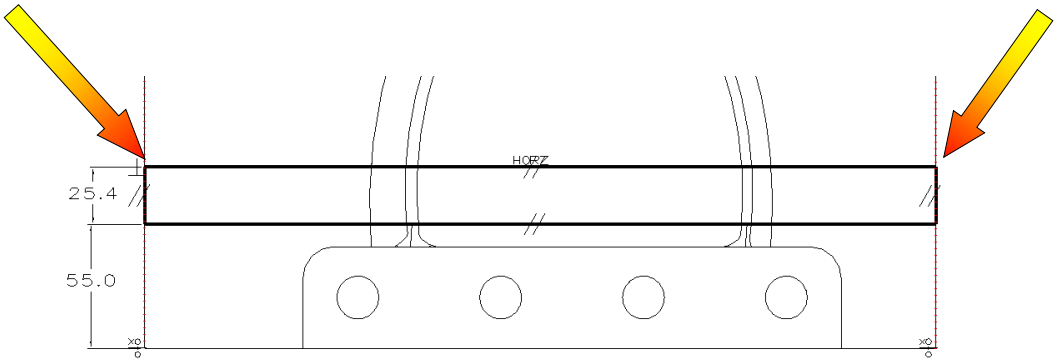
Turn Reference face edges off.

Pick the **Curve Reference** command from the Tools Tab.

Pick the left and right edges of the blow mold. See figure below for reference. You will see the hidden or dotted lines on the left and right of the blow mold. **Curve Reference** projects reference geometry and extracts those edges so we can use them during the constraining process of our sketch.

Create and fully constrain the rectangular sketch as shown in the following figure.

These are some of the commands you might consider using.  Pick the icon or use the <Right-click> menu to access these commands.



<Right-click> Exit Sketch.

Pick the **Extrude** command.

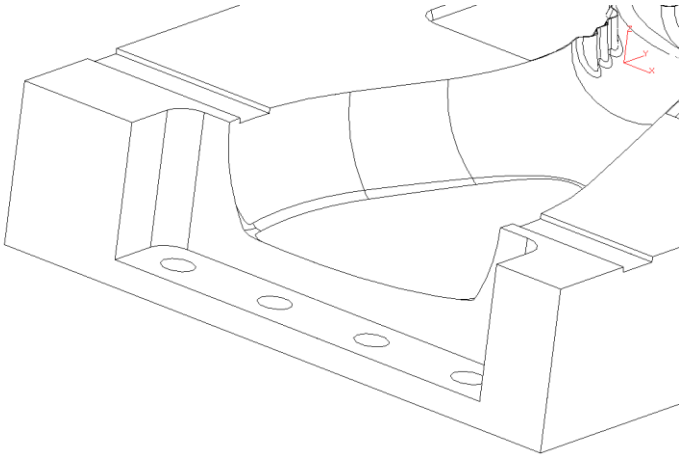
Pick the **Remove** icon in the options form.

Pick the sketch you just created.

Set the Start to **0.**

Set the End to **–3** with **no draft angle**.

Complete the command.



Let's complete our design by creating the other 3 grooves.

Pick the **Pattern** command from the **Shape** Tab and then select the **Linear** method  icon from the **Required Inputs** section of the Options Form.

Pick the groove you just created.

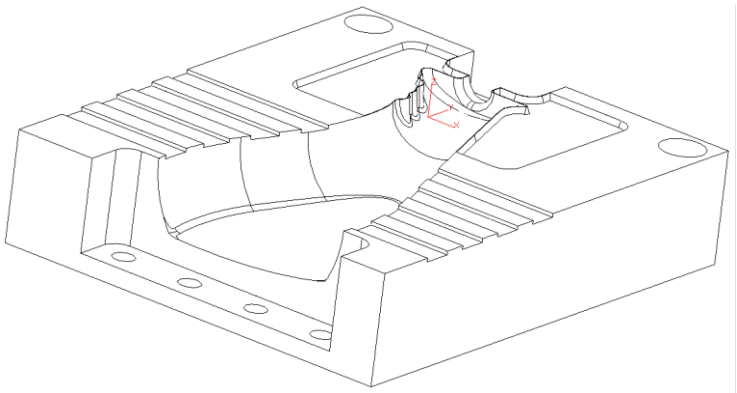
For the direction of the Pattern you have a couple choices. You could <Right-click> **Y-Axis** or you could pick the linear edge that is parallel to the **Y-Axis**. You decide.

If you pick the linear edge, make sure you pick it at the correct end. A direction vector will appear indicating which way the pattern will be placed.

The Total number of features will be **4.**

The Spacing between each feature is to be **50mm.**

Your model should appear as shown in the figure below.



Completed Blow Mold

Designing a Telephone

This exercise will take you through several ZW3D commands to design a telephone as shown in the figure below. You should be familiar now with where most of the commands are located.

OPEN the ZW3D file "**Fundamentals.Z3**".

Pick the **Part/Assembly** icon from the **ZW3D** root to create a Part object.

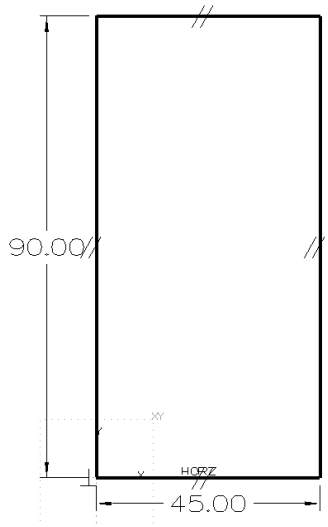
Name it "**Telephone**".

Pick the **Extrude** command and create a default sketch.

Pick the **Rectangle-Corner** command. 

Create the sketch shown in the following figure.

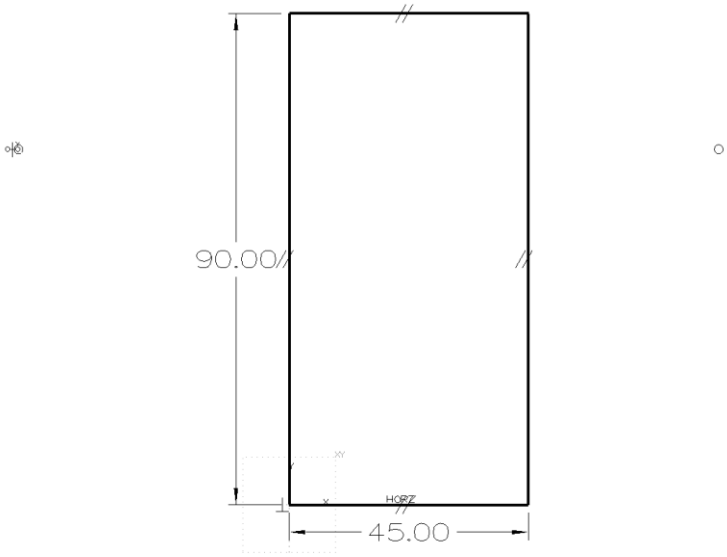
Notice the geometric constraints that were automatically added to the sketch.



<Right-click> Points

Create 2 points similar to those shown in the following figure.

Notice the point's line up in the y direction. When creating the second point, guide ZW3D to automatically create a Points Horizontal constraint.



<Right-click> **Auto Constrain.**

Pick the lower left corner of the rectangle as the base point to fully constrain our sketch. Your dimension values may vary. That's OK. We will adjust them in a few minutes.

Verify your sketch is well defined using **Constraint Status.**

Change the dimension values as shown in the figure below.

Delay solving the sketch if your dimensions are not close to being to scale.

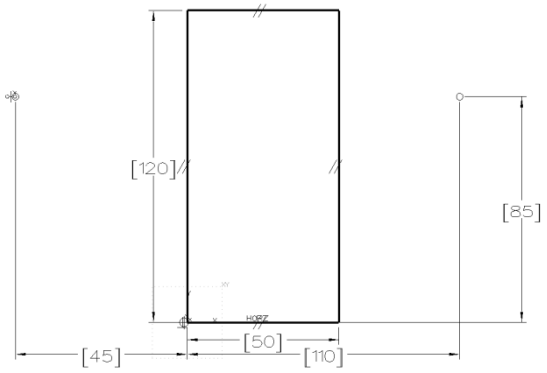
Brackets **[ ]** will appear around each dimension to indicate that dimension has a pending change. One dimensional change may not be possible without another. Delaying the sketch solve allows ZW3D to consider all dimensional changes before updating the sketch.

Remember there are a couple ways we can change dimensions in a ZW3D sketch and part.

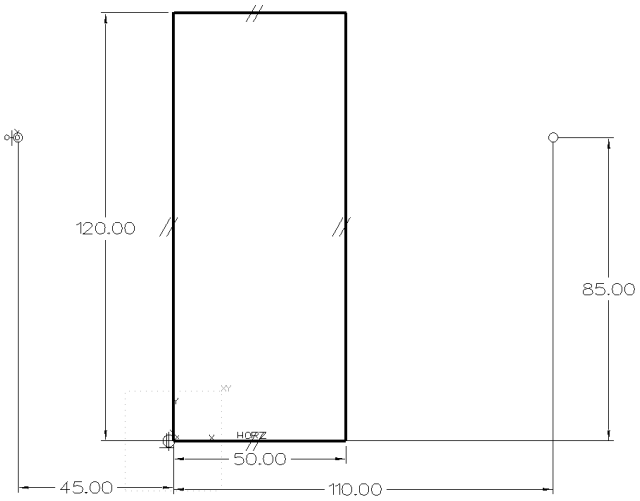
Highlight each dimension, <Right-click> Modify Value.

Or

Double click on each dimension to use **Input Dimension Value** form.



Pick the **Solve current sketch automatically** command from the Utilities toolbar to apply all dimensional changes to the sketch. Typically this icon appears as GREEN when there are pending changes. This rule does not apply when in a sketch.



Create two **Circles** tangent to the vertical lines on the rectangle as shown below.

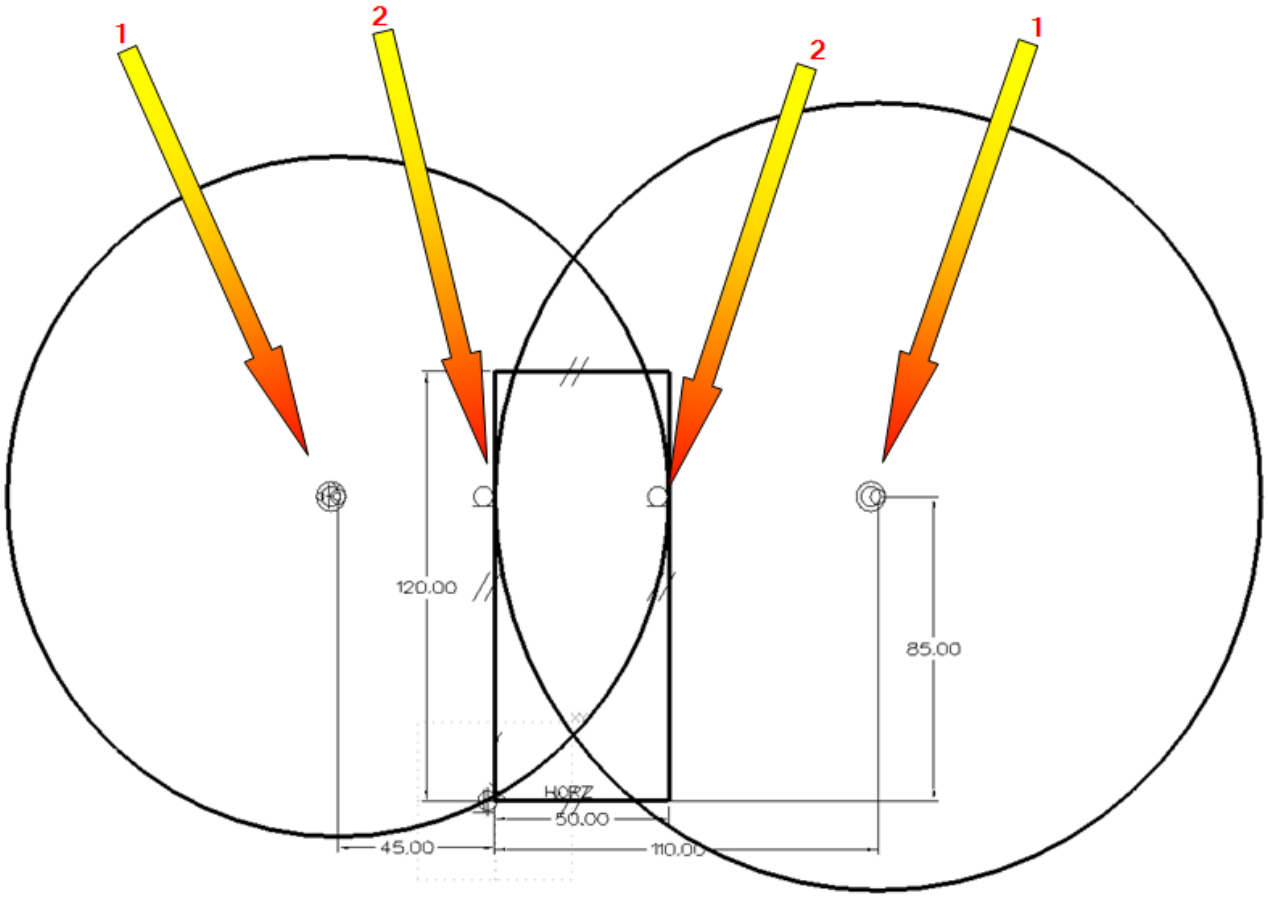
<Right-click> **Critical** and pick one of the points for the center of the first circle.

<Right-click> **Tangent** and pick the appropriate vertical line.

The resulting circles should be constrained as concentric to the points.

Review the figure below to be sure your sketches are similar.

The geometric constraints may vary. That's OK.



1: Critical 2: Tangent

<Right-click> Trim.

Trim away the portion of the sketch we no longer need as shown in the following figure. Remember, trimming requires that you simply pick on the part of the geometry to be removed.

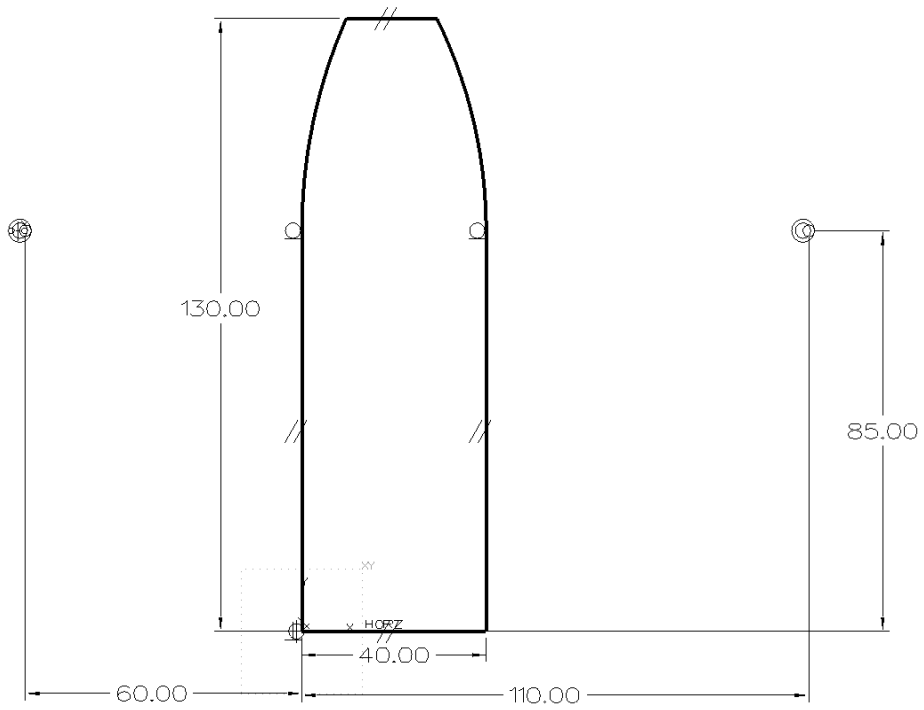
Don’t worry about the dimension values in this step.

Dimensions may have changed from vertical to horizontal to aligned. Replace any dimensions as required so your sketch appears similar to the sketch figure below.

Again, don’t worry about the dimension values in this step.

Turn the **Constraint Status** command on to verify the status of your sketch. Leave it on the screen for the next several steps. Watch how ZW3D updates the sketch status as you add and remove constraints.

Modify the dimensions to match the sketch shown below. Delay the sketch solve if required.



Notice the vertical lines are parallel to each other. In our sketch they have parallel and perpendicular constraints associated to them. Your sketch may vary. That’s OK.

Erase any parallel and perpendicular constraints associated to the vertical lines.

Highlight the constraint <Right-click> **Erase**. Use the <Alt> + pick method if required. Review the notes below if you are not familiar with the **<Alt> + pick** method.

Note:
Occasionally multiple constraints, or any entity for that matter, are located on top of one another. When the user tries to pick an entity, sometimes the wrong one is highlighted.

In order to correctly select the appropriate entity, the user can "filter" the pick using the <Alt> key.

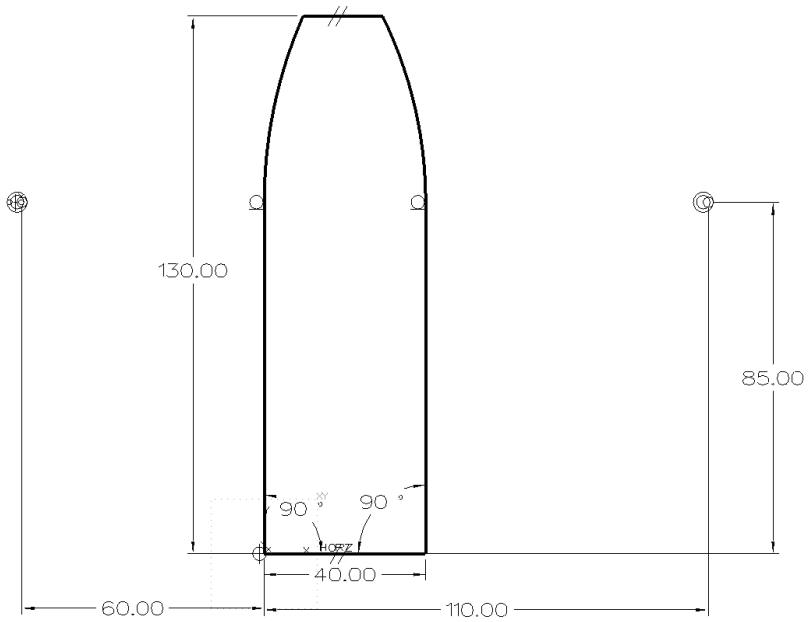
The <Alt + Pick>, picks the second entity. While selecting entities, the first valid entity pick is ignored and the second one is used.

The <Alt + Pick> method can be extended by using combinations of the <Ctrl> and <Shift> keys together with the <Alt> key.

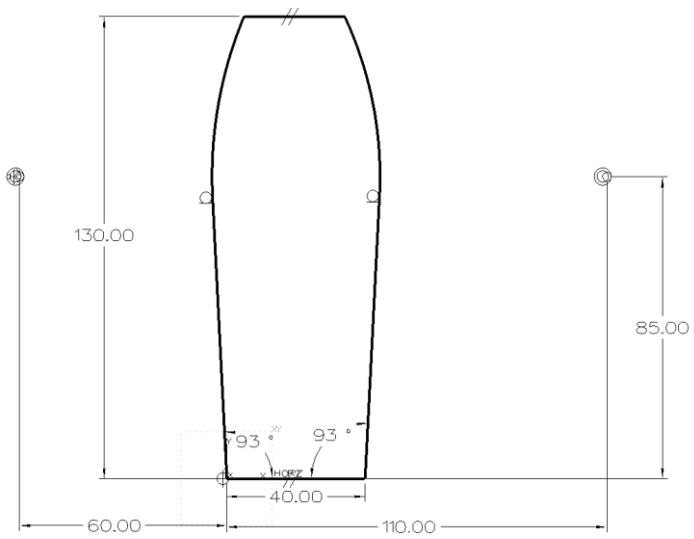
If parallel constraints were applied in your sketch, you will only need to erase one of them. Notice when you highlight a parallel constraint, its associated partner constraint also highlights. Erasing one parallel constraint automatically erases its associated partner.

Add the angular dimensions to our sketch as shown in the following figure.

<Right-click> Dimension.



Change both angular dimensions to **93**°.



Create **10mm fillets** on the four corners of our sketch.

Use the **Chain of curves fillet** command.

Or

<Right-click> Fillet and create four individual fillets. You decide.

Note:

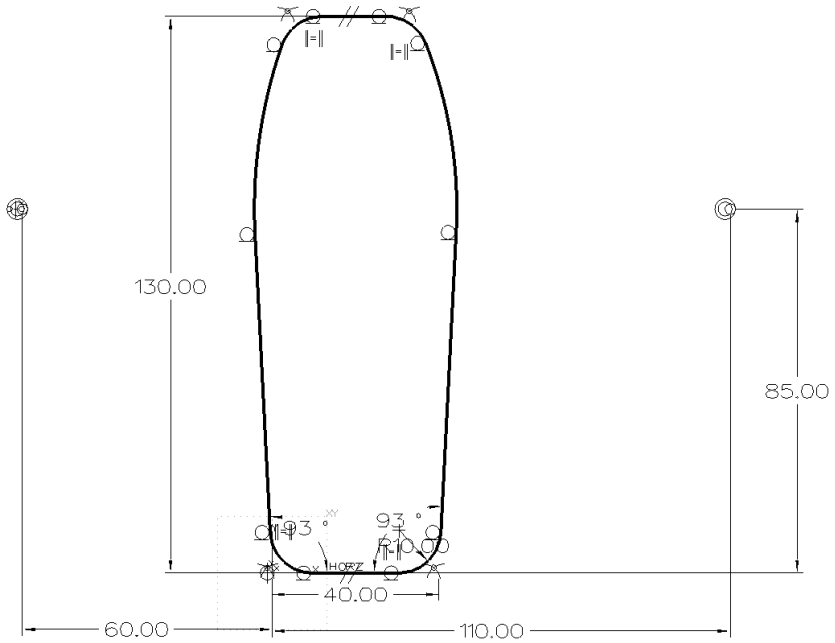
If you use the Chain of curves fillet command, you will have a few methods for picking the available geometry to fillet. Use <Shift> pick to chain pick the entire sketch, window pick the entire sketch or <Right-click> Pick All. The fillets created using this method will be associated to one another. A single dimension will be added to the sketch that will control the size of all four fillets.

Ignore the error message. It indicates that a fillet can't be added at the existing tangent points.

If you use the <Right-click> Fillet method ZW3D will create four individual fillets with individual constraints. ZW3D offers two methods for picking the geometry to fillet. Pick two adjacent edges to preview the fillet or pick at the corner.

Notice how your design intent is maintained. ZW3D adds the necessary geometric constraints to maintain the 40mm and 130mm dimensions.

Your sketch should appear similar to the following figure.

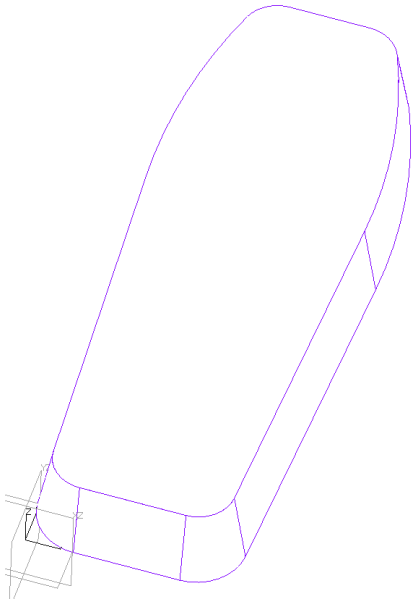


Regen the sketch.

<Right-click> Exit Sketch.

ZW3D remembers you started this operation with the **Extrude** command and displays the options form for extrude. Adjust the form as shown in the left figure below to complete this operation.

Start = 0, **End** = 15, **Draft angle**.= -10.



When draft angle is applied to extrude type commands, ZW3D offers three Blend options that determine how fillets will be handled. Experiment with the telephone to see the differences.

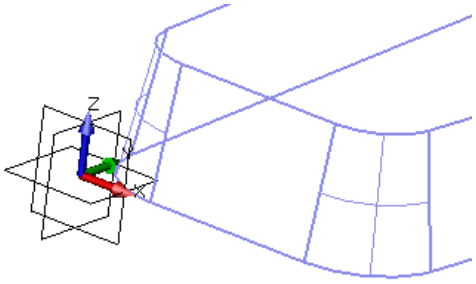
<Right-click> Redefine last.

Review the 3 blend options shown below using the options form.

If **Redefine** last is not available, highlight the model and <Right-click> **Redefine.**

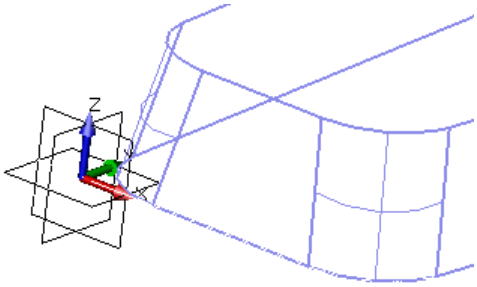
Variable:

Corners are not modified. They maintain the same convexity/concavity (default).



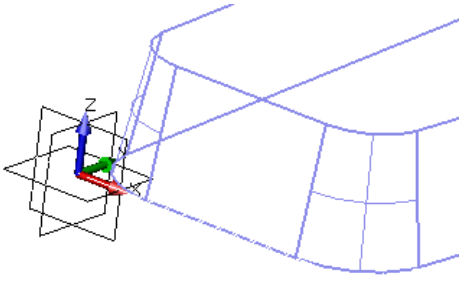
Constant:

Filleted/rounded corners will maintain the same radius.



Round:

Convex corners are rounded. Concave corners are not rounded.



Let's move our model to the 0,0,0 origin in the database.

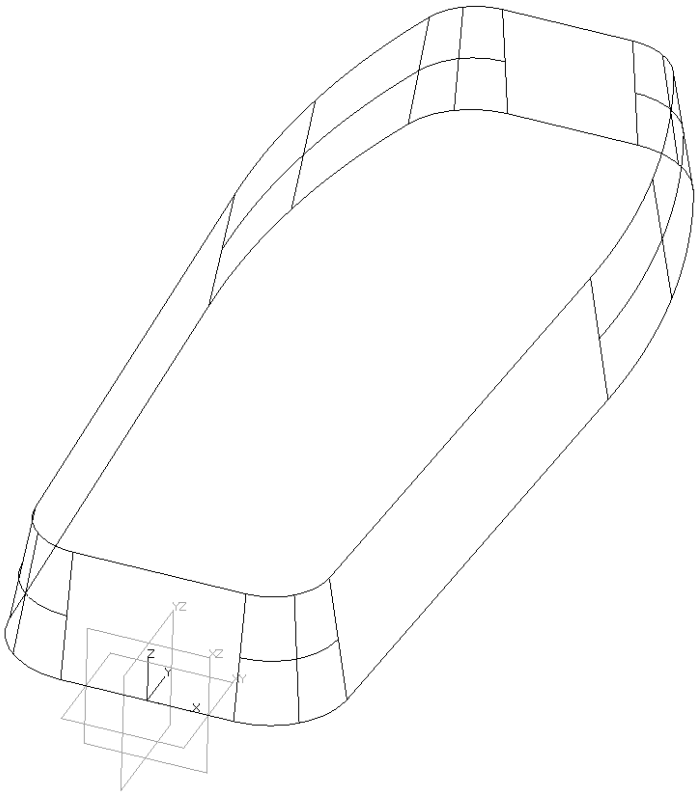
Pick the **Move entities from point to point**  command within the **Move** command under the **Shape** Tab.

Pick the telephone shape to move.

For the "**From**" point, <Right-click> **Critical** and pick the middle point of the lower horizontal line lying on the **XY-Plane**.

For the "**To point**", type or pick **0,0,0.**

See figure below for reference.



In the next part of this lesson we are going to create a swept surface to use for shaping the top of our telephone.

<Right-click> Insert sketch.

Pick the **YZ – Plane** to create the new sketch on.

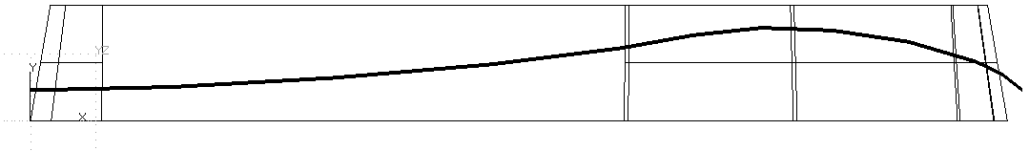
We will begin by creating a curve through specified points. This curve will be used later as the path curve for the swept surface.

Pick the **Through point curve**  command from the Sketch Tab.

The start and end tangencies will remain undefined as shown in the options form below. Start and end tangencies can have optional direction vectors applied as required, but we will disregard this option for now.

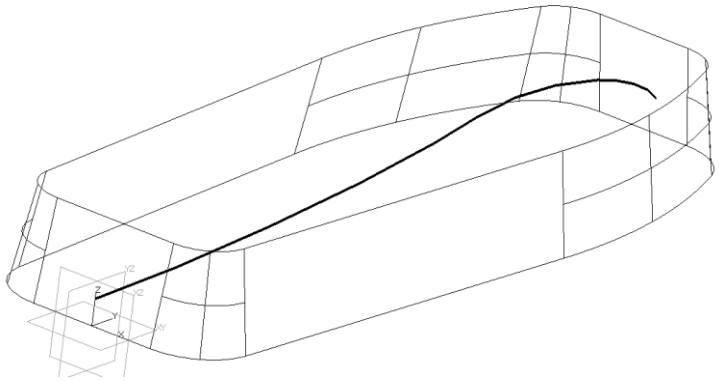
Specify the following co-ordinates to pass the curve through.

1. Point	**0,4.**
2. Point:	**46,6**
3. Point:	**82,10**
4. Point:	**96,12**
5. Point:	**118,10**
6. Point:	**132,4**



The curve can also be controlled by dimensions using the <Right-click> Constrain command.

<Right-click> Exit sketch.



Let's create a new sketch on the XZ-Plane.

<Right-click> Insert sketch.

Pick the **XZ-Plane** for our new sketch.

Let's create a **3-Point Arc** using the previous curves start point as the middle point of the arc.

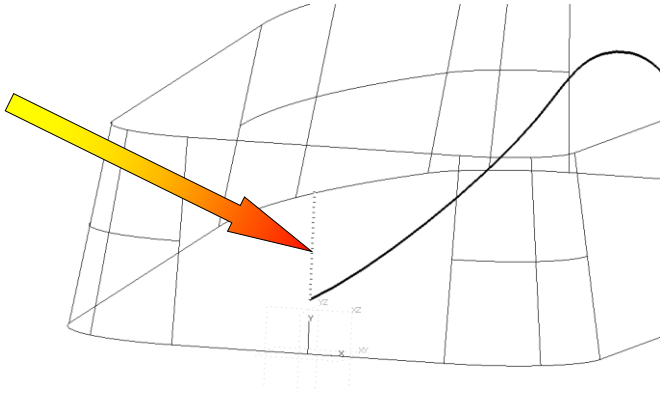
First we have to create a reference curve from the previous sketch to attach our arc to.

Pick the **Curve Reference** command from the Sketch Tab.

Pick the previous sketch.

ZW3D will create a reference curve that is associated to the sketch and project it to the XZ-Plane. As changes are made to the original sketch, the reference curve will update. Remember reference curves are red dotted or dashed curves. See figure below for reference.

Verify your model to see this reference curve was successfully created.



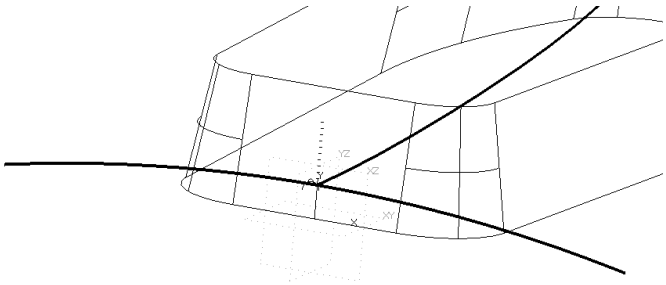
Now let's create the **3-Point Arc**.

Pick the **Arc** command and then select the **3-Point** method from the **Required Options** section of the form to create an arc through 3 points.

Specify the Start point: **-40,0**

Specify the Endpoint: **40,0**

For the point on the arc, <Right-click> **Critical** and pick the lower end point of the previously created reference curve as shown below.

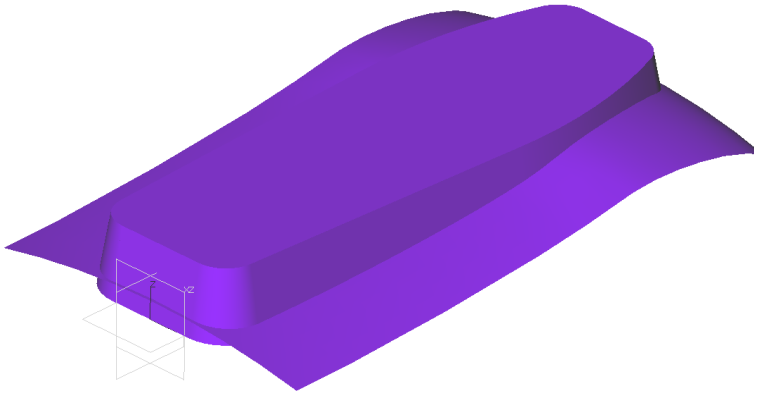


<Right-click> Exit sketch

Let's create the swept face. ZW3D is a hybrid modeler, which allows the user to work with surfaces as if they are solids. We can use the same sweep command to create a swept solid or surface. ZW3D will determine that the geometry used does not form a solid and will create a surface instead.

ZW3D offers a few methods for trimming the original model to the surface we are about to create. We will practice a few of these methods, but first let's create the swept surface.

Pick the **Sweep** command from the **Shape** Tab and define the profile and path.

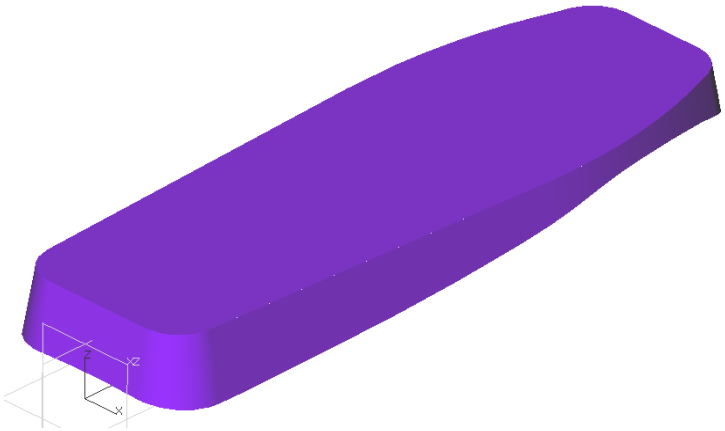


Notice that the trimming face is colored Pink (default) on one side and your chosen color on the other side. This signifies the direction of the surface normal (Pink is the negative side).

Let's try this step again and adjust the result by removing material during the sweep operation as shown in the following figure.

<Right-click> Redefine last.

Adjust the required input to **Remove** and pick OK. See following figure for reference.



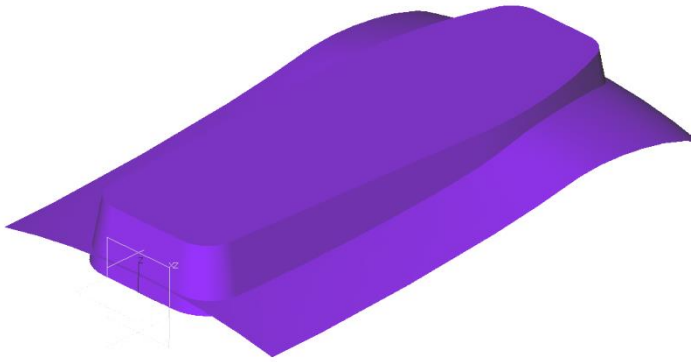
The result is that ZW3D removes material from the pink, negative, side of the swept surface.

Let's try this step again by resetting the required input back to base. This way we can continue working with other methods for trimming 3D geometry.

<Right-click> Redefine last.

Don't use UNDO since the last operation performed does not exist in the history tree yet since it is getting redefined, not modified. **UNDO** will affects the last completed operation in the history.

Change the required input back to **Base** and pick **OK**. See following figure for reference.



This time we are going to use the **Trim Shapes** command. Trim Shapes trims a shape with one or more faces and removes material in the specified direction.

Pick the **Trim Shapes** command from the **Shape** Tab.

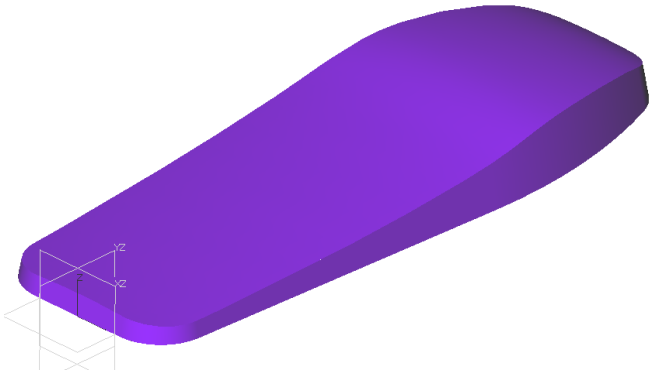
For the **Base B** pick the telephone. Multiple shapes can be picked for trimming in a single operation if required.

For the **Trimming T** shapes pick the swept surface.

We want to keep the bottom portion of our telephone. ZW3D will display direction vectors pointing towards the side to keep.

Pick the **Flip side (S) to keep** option if required to keep the bottom portion. See figure below for reference.

Try this with and without the **Cap trimmed region** option checked.



UNDO the last trimming operation.

Next we will use the **Divide Shapes** command. Divide Shapes divides the telephone shape at the specified cutting plane or face without removing any portion of it.

Pick **Divide Shapes** command from the **Shape** Tab.

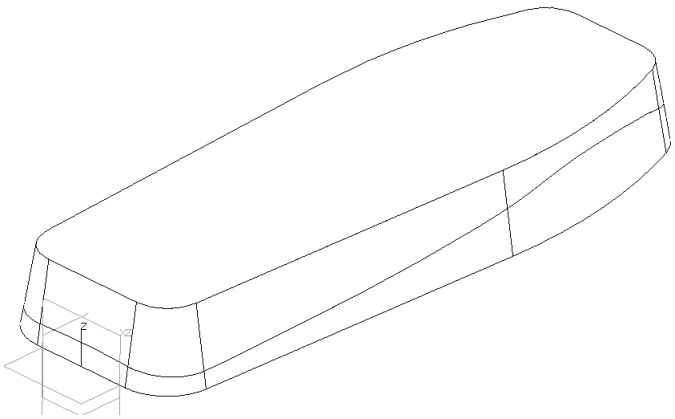
For Shapes to be divided pick the telephone. Multiple shapes can be picked for trimming in a single operation if required.

For dividing shapes pick the swept surface.

Adjust the options form as shown in the figure below. We want to divide the telephone and delete the swept surface during the operation by setting the cutter option to DELETE. The KEEP option will retain the dividing surface. The DIVIDE option will divide the cutting surface also.

Review the HELP Manual for this command.

Your model should be displayed as two individual pieces and the swept surface we used as the cutter should be removed. See figure below for reference.



UNDO the last dividing operation.

The final operation we will use and retain is the replace face command. Use this command to replace one or more faces of a solid or shape with another face.

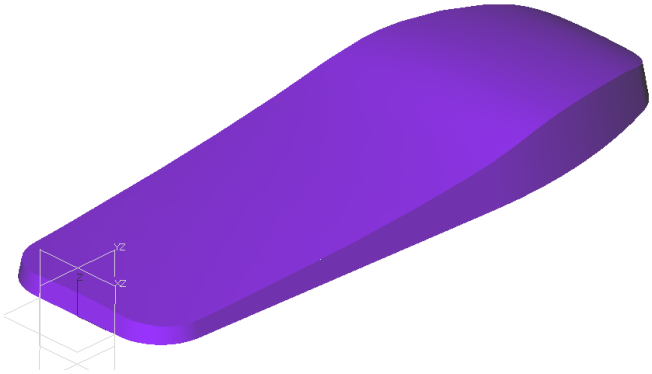
Pick the **Replace**  command from the **Shape** Tab.

The option Keep replace face will retain the original face. We want to erase the replaced face.

For Faces to replace(Base) pick the top face of the telephone.

For the replacing face pick the swept surface.

ZW3D will add and remove as required to generate the result. Your telephone should appear as shown in the following figure.



Which method you use for trimming or replacing faces will vary based on the design results you need to obtain. These last several steps should have familiarized you with the various methods offered by ZW3D for trimming, dividing and replacing faces in a model.

Next we will add a variable fillet along the top edge of our model.

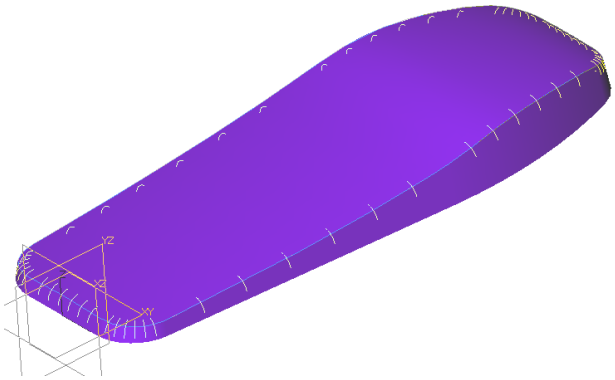
Pick the **Fillet** command from the **Shape** Tab. Fillet creates constant and variable fillets as specified in the options form.

Set the **Radius** to **2mm** in the options form.

Chain pick the top edges as shown in the following figure.

Highlight part of the edge and <Shift><Left-click>.

See figure for reference. Do not apply the fillet yet.



So far we told ZW3D what edge we intend to fillet. Currently, ZW3D thinks we want to create a constant fillet. We need to add the variable attributes next.

Pick the **Add** button in the Variable radius form so we can add the required variable attributes.

We are going to attach 4 variable fillet attributes along the highlighted edge.

Zoom window to the top of the phone.

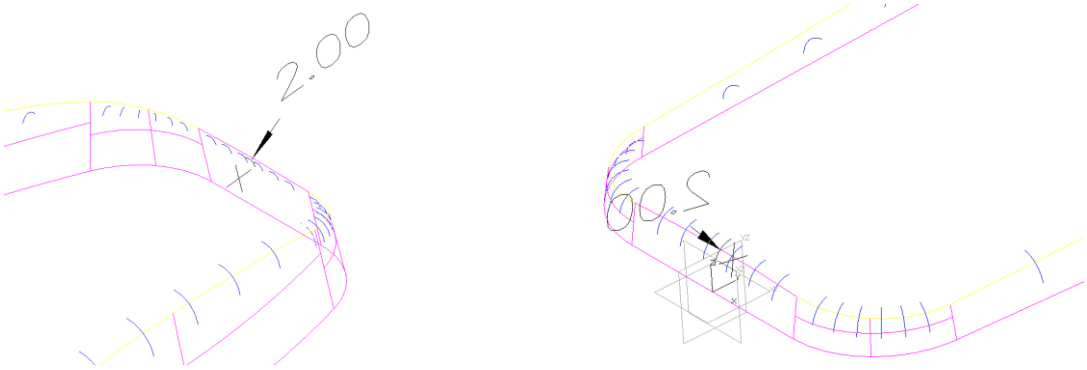
<Right-click> along and pick the upper edge as shown in the left figure below.

Specify the percentage along the edge to be **50**% and the radius to be **2**mm.

Press and hold the **middle mouse button** to Pan towards the bottom of the phone.

<Right-click> along and pick the upper edge as shown in the right figure below.

Specify the percentage along the edge to be **50**% and the radius to be **2**mm.



Press and hold the **middle mouse button** to Pan towards the middle of the phone.

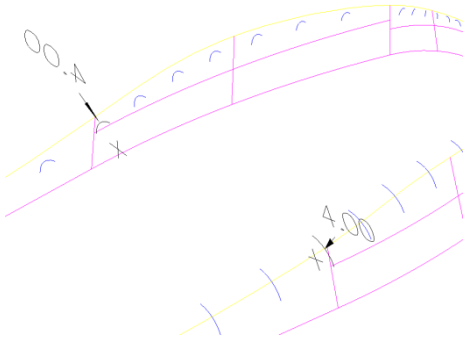
<Right-click> **Critical** and pick on the upper edge as shown in the left figure below.

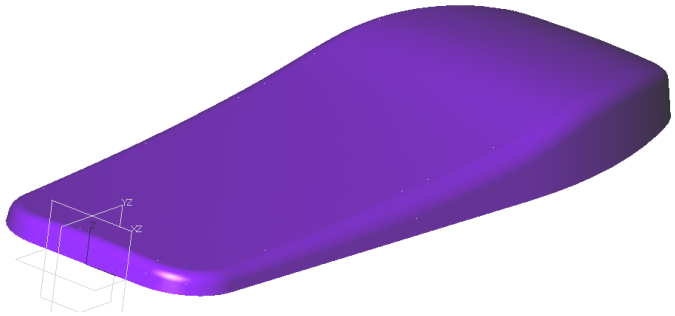
Specify the radius to be **4**mm.

<Right-click> **Critical** and pick on the other side of the upper edge as shown in the left figure below.

Specify the radius to be **4**mm.

Complete the command.

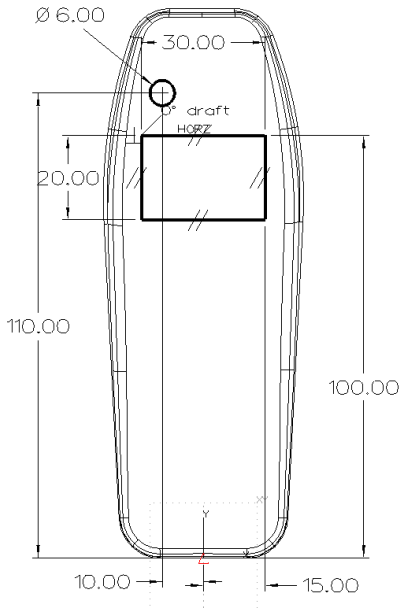




Next we will create the LED and ON/OFF switch cutouts.

Pick the **Extrude** – **remove** command and activate the **XY-Plane** for the following sketch.

Create and fully constrain the sketch as shown in the following figure.



Verify your sketch is well defined using **Constraint Status**.

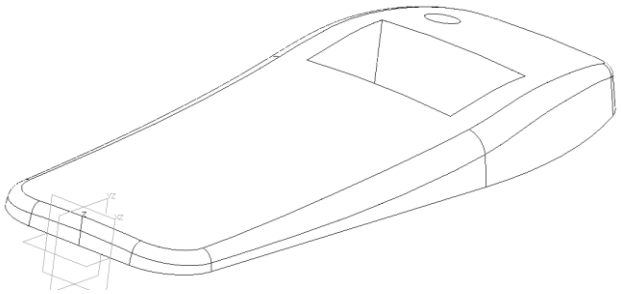
<Right-click> Exit sketch.

ZW3D continues with the extrude command by displaying its options form.

For the **Start** <Right-click> **Boundary Face** and pick the top face of the telephone.

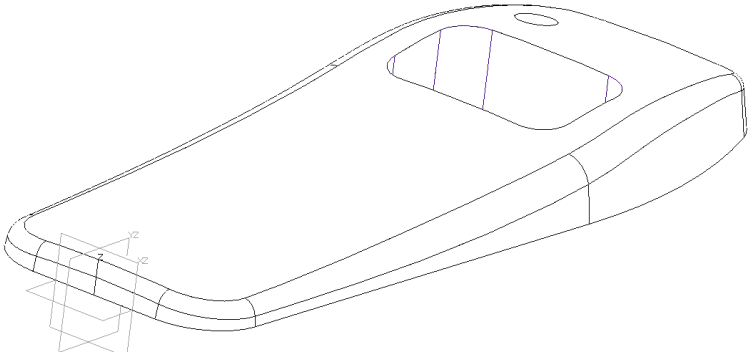
For the **End** <Right-click> **Boundary Face** and pick the bottom face of the telephone.

No Draft Angle.



The method we used for defining the start and ends for our extrusion are parametric. If the telephone becomes thicker, these boundary definitions and associated features will update automatically.

Add **5mm** constant **fillets** to the inside corners of the LED cutout.



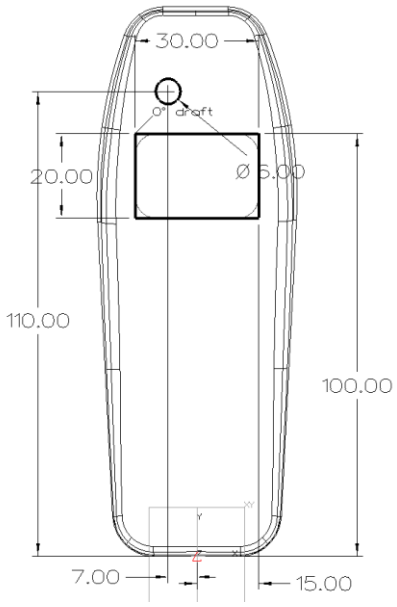
Change the position of the 6mm diameter hole.

Set the **Pick Filter** to **Feature**.

Highlight the 6mm diameter hole feature and double click on it to display all of its dimensions. Remember the LED window is part of the hole feature so it will also highlight. That's OK.

If you have problems making just the hole and LED features highlight, hold the **<Alt>** key down. **<Alt>** is an extended entity pick method we used earlier.

Double click on the **10**mm dimension and change it to **7**mm. This is our only change so it is not necessary to select "Delay Solve".



The history will automatically regenerate.

Let's create the first button hole for the keypad. There are a variety of methods available in ZW3D to create this hole. Let's try a method we haven't worked with yet. We will activate a new sketch and define precisely where we would like 0,0 origin to be instead of accepting some default location.

Pick the **Extrude** command. Middle-click to insert a sketch.

Pick the Origin Define button from the options form and specify **-12,60** for the origin. This coordinate is relative to the world **XY Plane** in our model.

For the insertion plane pick the **XY-Plane**.

Complete the command. ZW3D will create a new sketch plane and place its origin at the coordinates specified above relative to the world XY Plane. See figures below for reference.

We could insert an ellipse from the Sketch Tab then make sure the constraints are complete to allow for easy changes OR we could insert predefined geometry that is already constrained. You just have to edit two dimensions.

Open the Sketch Tab.

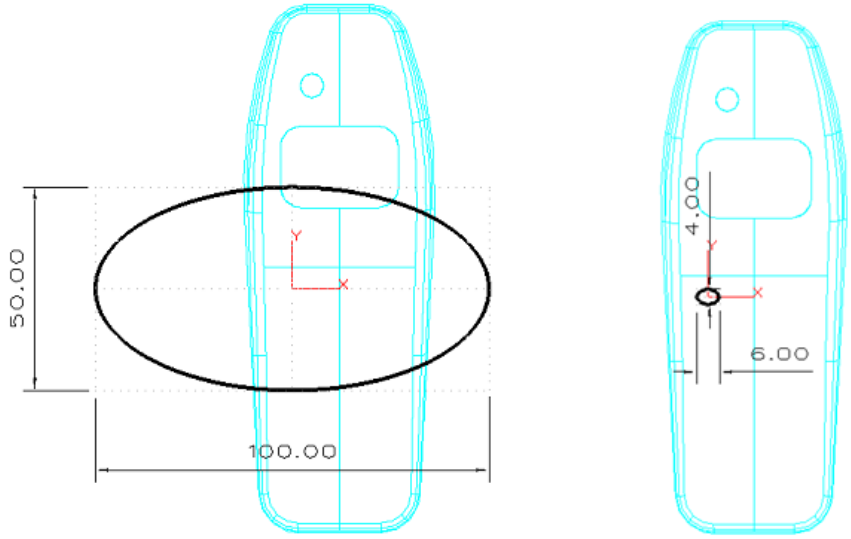
Spend some time familiarizing yourself with all the option.

Pick the **Ellipse** command  .

Left click to set the location on (0,0).

Change 50 to **4.**

Change 100 to **6.**

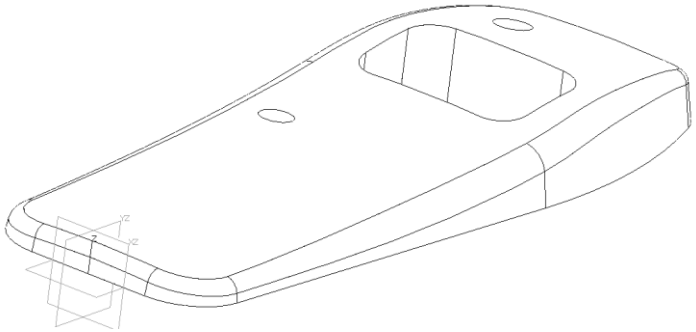


<Right-click> Exit sketch.

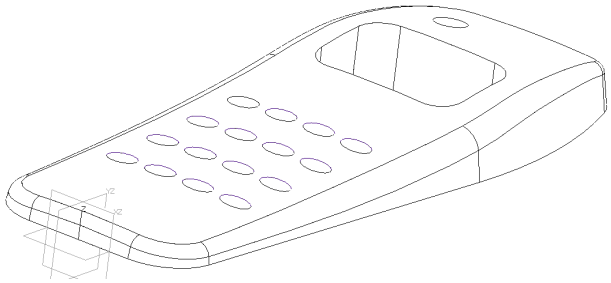
ZW3D continues with the **Extrude** command by displaying its options form.

For the **Start** <Right-click> **Boundary Face** and pick the top face of the telephone.

For the **End** <Right-click> **Boundary Face** and pick the bottom face of the telephone.



Create a 4 x 4 linear pattern of the elliptical hole we just created.



Pick the **Pattern** command from the **Tools** Tab and then select the **Linear** method icon from the **Required Inputs** section of the Options Form.

Use the **Filter – Feature** and pick the hole.

First direction <Right-click> **X Axis.**

Number of features in the first direction = **4.**

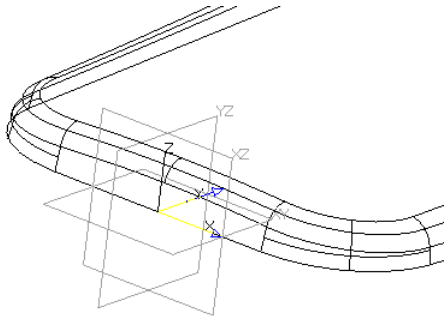
Spacing in the first direction = **8** mm.

Second direction <Right-click> **Y Axis.**

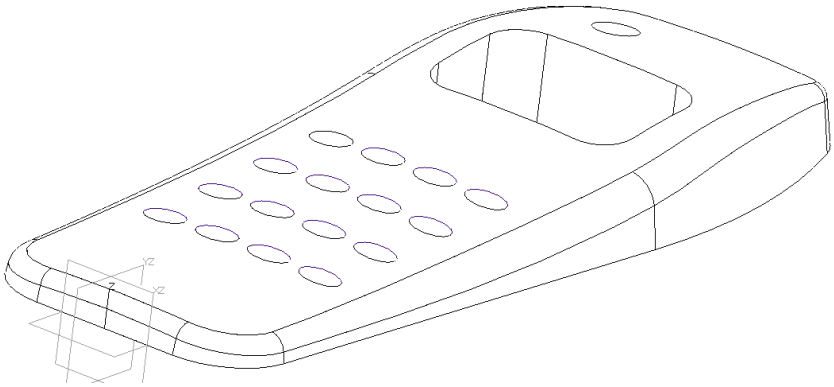
Number of features in the second direction = **4.**

Spacing in the second direction = **-11** mm.

ZW3D displays direction vectors based on the direction of the axes you indicated in the Linear Feature command. Specify a negative or positive value for spacing based on these vectors.



See the result below.



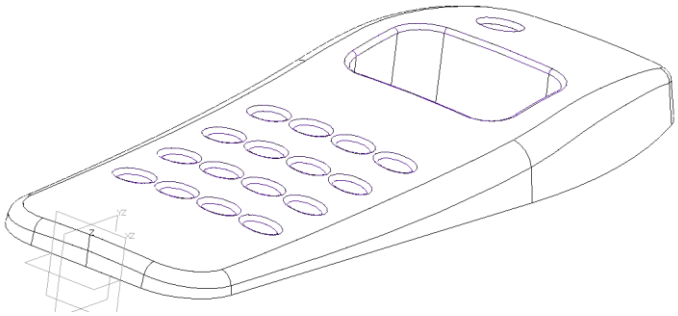
Create a constant Loop fillet of radius .7mm on the top face.

Pick the **Fillet** command from the **Shape** Tab.

Pick the **Loop Fillet**  of Fillet.

As you place your cursor over the top face all edges will highlight. Pick the top face and complete the process based on the options form shown below.

Use this command to create a constant radius fillet along the edge loop(s) of a face. There are advantages to using this command over filleting each edge individually. You can select all of the edges by selecting the face. This command also creates a more stable part. If the face is later modified, the fillets will adapt accordingly during the next regeneration. Changes can sometimes affect edges, and due to your change, the edge fillet may no longer work because either there are a different number of edges, or maybe the edge(s) no longer exists.



The loop option we used in this step was “All”. If the variable fillet did not exist along the outer edge of our telephone, we would have to use “Inner”. Inner would instruct ZW3D to apply fillets to the inside edges only, avoiding all outside edges. Other options are “Outer” and “Specified”.

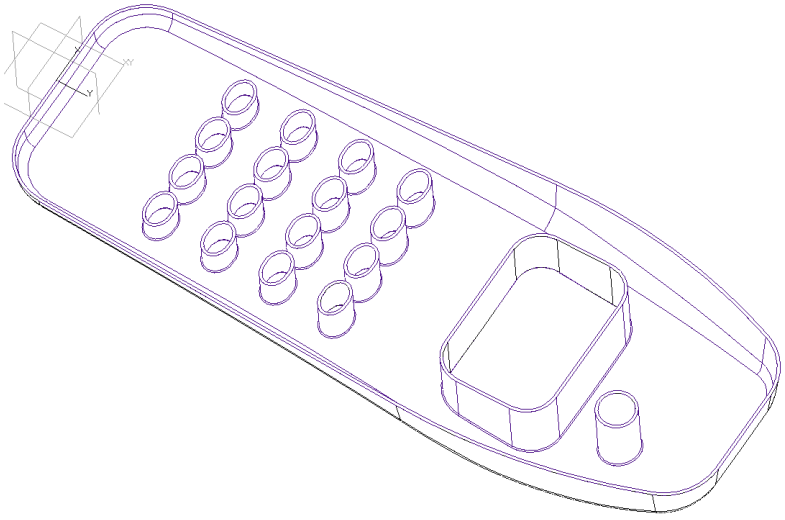
To complete our telephone design, let’s hollow out the inside.

Pick the **Shell**  command.

Pick the telephone as the shape to shell.

Specify wall thickness **–0.5** mm. (A minus value shells inwards, a plus value outward).

Pick the bottom face as the face to be left open.



Finished!

Chapter 9

Creating Face Draft and Copying Features

Refer to the HELP Manual

Refer to the **HELP Manual** and review the following commands.

Draft (Mold Design)
Draft Feature
Inquire Face Draft Angle Violations
Mirror (3D)
Scale – Part Entity

Lesson 9

Creating Face Draft

Let's begin this lesson by working on a primitive shape that will clearly illustrate the fundamental features of the ZW3D face draft tools. During this lesson you will learn to apply draft to faces.

Open the file "**Fundamentals.Z3**".

Create a new part named "**Primitive**".

<Right-click> Insert sketch.

<Middle-click>twice to accept default plane (**XY Plane**).

Draw a radius. Pick the **Arc** command and then select the **Center** method from the **Required Inputs** section of the form to create an arc as shown below.

Locate the center at the 0,0 point.
Enter a start point of (0,100).
Enter an end point of (-100,0).

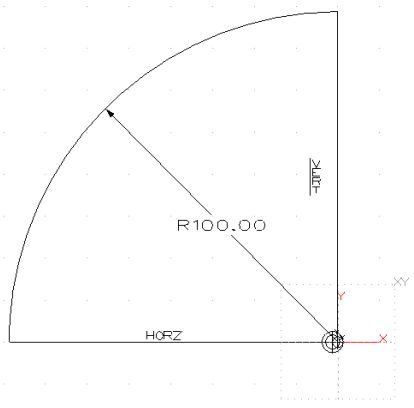
<CTRL+A> to Zoom All.

<Right-click> Draw. Create two lines to close the profile as shown in the figure below.

<Right-click> Auto Constrain.

Pick the (0,0) point again to fully constrain your sketch.

Your sketch should match the figure below.



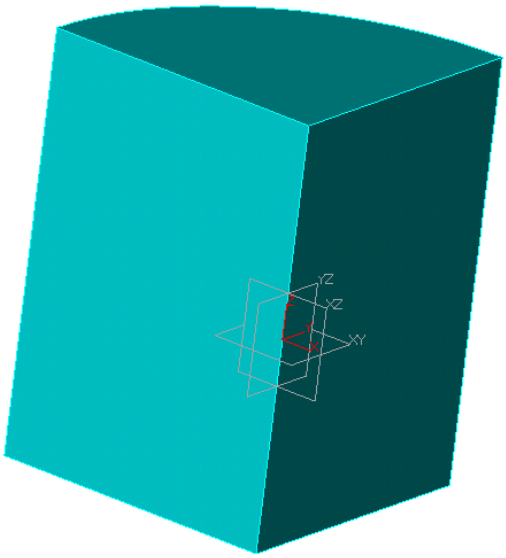
Regen current sketch.

<Right-click> Exit sketch.

Pick the **Extrude** command  from the **Shape** Tab. Specify a start value of **–70** and an end value of **70**.

Optional Input values: ***End caps*** set to “Both”, and ***Draft angle*** set to zero.

Click **OK** to finish extrusion command and **<CTRL+A>** to Zoom All.



Save your part.

Pick the **Display Shaded** command from the top toolbar to shade your model for the following steps.

Pick the **Draft**  command from the **Shape** Tab.

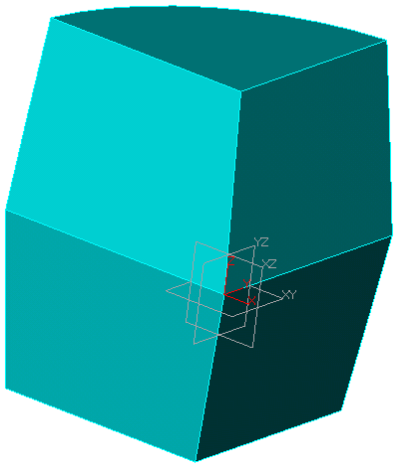
Pick the **XY Plane** for the plane to **draft about.** This is the plane from which the draft will start.

Set the **Draft angle to 0** in the options form.

Select the **Variable Draft** tab and then pick the **Add Draft** button in the optional input section to add a **10 degree** draft attribute to both flat faces on our model.

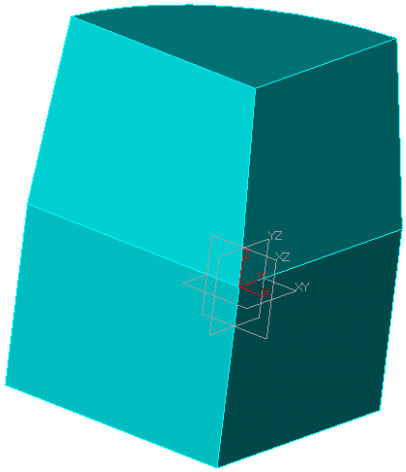
Click **OK** to finish.

See the figure for results.
Draft should be added to the flat faces only and split about the XY Plane.



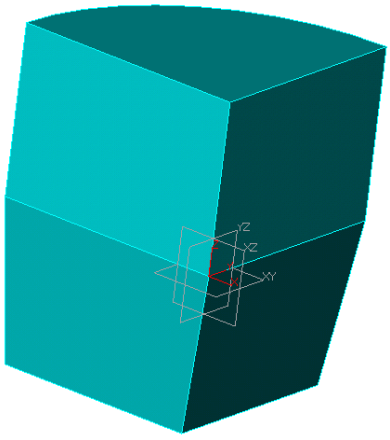
<Right-click> Redefine last.

Change the Draft side to **TOP**. See following figure for results.



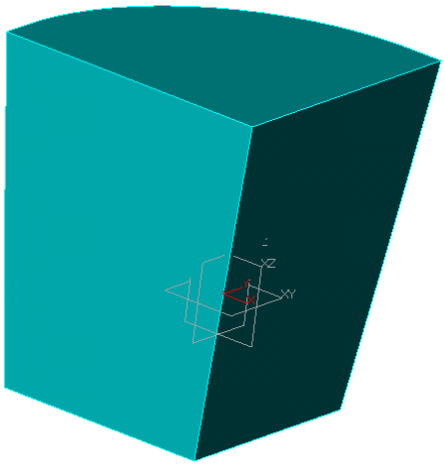
<Right-click> Redefine last.

Change the Draft side to **BOTTOM**. See following figure for results.



<Right-click> Redefine last

Change the Draft side to **NEUTRAL**. The direction of the draft will depend on a positive or negative draft angle specified earlier in this step. Draft is still about the XY Plane.

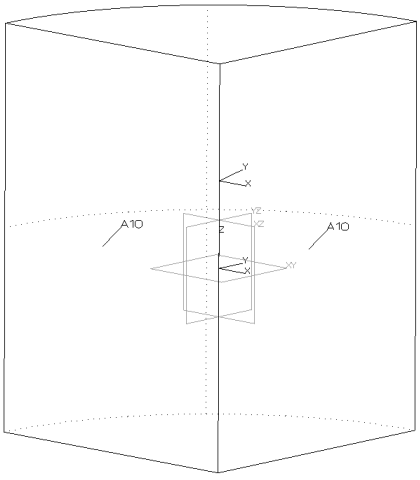


UNDO the previous draft operation. Leave the draft attributes. See right figure below for reference.

Let's create a new plane so we can attempt to draft about a non-standard plane.

<Right-click> Insert Datum.

Adjust the form so the method is **XY**  , Offset is **30** and the X Angle is **10**.



Pick the **Draft** command from the **Shape** Tab again.

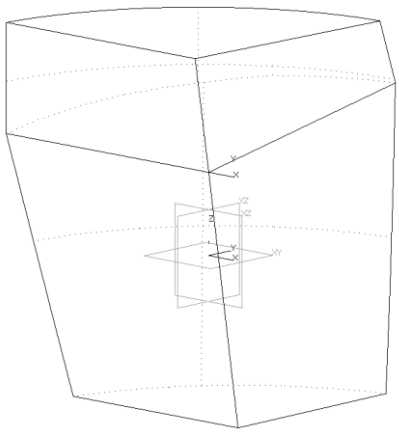
In this example you must specify which faces that are going to be drafted first because the top and bottom faces now intersect the draft plane (theoretically). **Pick** the two faces with the draft attributes assigned for the **Face** option.

Pick the **Datum plane** created in the previous step as the plane to **draft about**. This is the plane from which the draft will start from.

Set the **Draft angle to 0** in the options form. Remember we are using the **10** degree draft attributes created earlier on the 2 flat faces.

Click **OK** to finish.

The curved face should not get any draft assigned to it, only the flat side faces as shown in the figure below.



<Right-click> Redefine last.

Play with the **Draft side** options to better understand this command.

Finished!

Chapter 10

Editing and Managing ZW3D Objects

Refer to the HELP Manual

Refer to the **HELP Manual** and review the following commands.

History - Redefine Feature
Step through Part History
Redefine
Rollback
Replace
Rename
Encapsulate
Cut (to Clipboard) (Ctrl+X)
Copy (to Clipboard) (Ctrl+C)
Paste (from Clipboard) (Ctrl+V)
Delete
History – Set Feature Tolerance
History – Set Feature Suppression
Unsuppress All
Delete Suppressed
Conditional

Chapter 11

Customizing ZW3D CAD Templates

Customizing ZW3D CAD Templates

The purpose of this chapter is to help our customers understand how to build their design and drafting standards into the ZW3D CAD system. To accomplish this task we will modify the file “Templates_IN.Z3” and “Templates_MM.Z3”

The Templates file is a unique file recognized by ZW3D as containing root object templates. These are root objects used to start a new part, component, drawing sheet, or CAM process plan. Using the Template option from the New File/Object Form will allow you to select a template from the “Templates_IN.Z3” or “Templates_MM.Z3”file. Doing so will copy the template and use it as the basis of the new root object.

This saves time by allowing you to set the working environment of the templates once and automatically apply them to new root objects. You might have part templates for each customer containing specific layers, attributes, and preferences or drawing sheet templates for standard drawing sheet sizes that are pre-formatted with title block information, dimension standards, etc.

When you begin **creating a new ZW3D Object**  , you will be presented with a window displaying all of the available template objects for the type of object you will be designing. Part templates, Drawing templates and CAM Process Plan templates are displayed in separate windows as shown in the figures below. This saves time by allowing you to define your own personal or corporate working environment (corporate or department standards) once, and automatically merge those standards into any new ZW3D CAD or CAM design you develop. You can even create your own version of Templates.ZW3D and give it any name you desire. For example, use YOURNAME.ZW3D or COMPANYNAME.ZW3D, or any name that makes sense to use and set it as the default inside of ZW3D.

Template “PartTemplate(MM)”/“PartTemplate(IN)” is provided for Part/Assembly  .

Templates such as A0(H)、A1(H) are provided for Drawing Sheet  .

We have talked a lot about objects in ZW3D, but what are objects? Objects in ZW3D are the 2D sketches and 3D Shapes you design. They are the detail drawings you create and the CAM plans you will create to manufacture your designs. A ZW3D file can contain several objects of multiple types. See the figure below for reference.

Lesson 11

Customizing ZW3D CAD Templates

OPEN the file, "**Templates_IN.Z3** or **"Templates_MM.Z3**". (Whichever units you are using.)

It is located in the "*Program Folder \resource"*. The ZW3D Objects window will appear as shown in the following figure. You will see a Part Template (Part) and several Drawing Sheets (Sheet) that can be used when creating parts, assemblies and drawings. We are going to go through the steps to modify the Part templates and Drawing Sheets.

A shortcut to access your active template file is found under the **File pull down menu > Templates**.

There is a copy of the template file in the Program Files Program Folder \resource folder so don't worry about making changes.

Start by double clicking on the Part Template.

You can make any changes you want from the **Attributes** pull down. You could change the **Point** attributes. You decide. You can add predefined layers. Select the **Layer Manager** tab.

Choose the **Dimension Attributes**.

Do you normally build parts with one, two or three place decimal precision?

Press F1 key ,it will display HELP Manual about each command you pick. This will make it easy for you to read about each of the CAD settings you are about to adjust in your template object.

Select **Edit > Preferences** from the top pull down menu.

Make sure the units and tolerance are what you want.

Maybe you like to see the hidden lines (back facing edges) dashed.

Select **Attributes > Face…** from the top pull down menu. Change the **Back Color** to red. Click 'OK' to save this change as the default back color for each solid or surface model you design when using this template object.

If you create both inch and metric parts from time to time maybe choose a different face color for the two templates as a reminder what units you are in.

Pick the **SAVE** command located in the upper left toolbar to save the adjustments you have made so far.

Select **Attributes > Line…** from the top pull down menu.

Change the default color you would like wireframe geometry to appear as and select a wider width. We selected the 2nd widest line width available in ZW3D. This will make wireframe geometry easier to see on the screen during the shaded display and when there is a lot of geometry on the screen.

See following figure for reference.

Pick '**OK'** to save your line attribute adjustments.

Select **Attributes > Dimension** from the top pull down menu.

Use this form to set a wide variety of dimension and text attributes. You can automatically make all attributes comply with one of five dimension standards (e.g., ANSI, ISO, DIN, JIS or GB) by selecting the desired standard, Click '**OK'** button. Use the **ZW3D Configuration Form** (See the **Utilities** pull down) to set the dimension standard for new parts, sketches and drawing sheets.

Go through this form and make the necessary adjustments you require. This form contains many settings that you may not be very familiar with yet. To review what these settings are, review the HELP Manual for this form.

Go to **Attributes > Dimension**. Click on the help icon at the top right of the form. ZW3D will display all HELP Manual for this command. All Options Forms have this help icon. Review this information as you adjust your dimension settings to understand what each of the various settings is used for.

When you are finished making adjustments for dimensions, Pick **OK** to save them.

Save your changes.

We are finished adjusting the **MM part template**. Let's move on to the sheet templates.

Select the **Exit part** icon from the top toolbar. The **ZW3D Objects** form will appear.

Let's make a few changes to one of the Drawing Sheet objects. These changes will affect the appearance of a drawing you create when you use this drawing sheet object as a template.

From the **Preview** section of the Objects window, pick "**Graphics**". This will give you a graphic preview of all objects you select to open. Pick one at a time to see the preview.

Choose the "**A0(H) (Sheet)**" drawing object template and **Right click > Edit** to activate it.

Once you are inside the drawing sheet template object you can double click over the various graphic objects to move them. If you would like to edit them, pick the appropriate command from the Edit pull down menu and follow the command prompt or <right-click> over an entity for options. This would be the appropriate place to enter you company name in the title box.

To adjust the appearance of drawing objects as you create and annotate drawing views let's go through the commands that are available from the **Attributes** pull down menu. From this menu we can adjust the default settings for Lines and Dimensions. These are the same adjustments we explored earlier when we adjusting our part template. Additionally, we can adjust Text,

Since we are in a drawing sheet, let's go through the **Hatch** and the appearance of **View Layouts** attributes forms and make some adjustments.

Go to **Attributes > Hatch.**

The form in the figure at the right will appear. Go through the settings in this form to familiarize yourself with them and make adjustments as required.

Note: the special formatted text in the title block. Not now, but when you use the templates, Select **Edit > Regen Text** and any information that has been added to the **Part Attributes** of the part you are laying out will be added to the title block.

When you are finished, pick **OK** to save any adjustments you made.

Go to **Attributes > View…**

The form in the figure at the near right will appear. Go through the settings in this form to familiarize yourself with them and make any necessary adjustment.

Apply any changes you made.

Select the **Exit part** icon to display the **ZW3D Objects** window.

SAVE your changes.

Let's talk some about what we should have completed so far.

We adjusted a Part object template for the attributes we use most often.

We also adjusted a B size-drawing format that can be used to create detail drawings when we are ready.

You might also consider importing a DXF or IGES file of your existing title blocks so you do not have to redraw them.

If you use AutoCAD, you could simply import the DWG files of your existing title blocks directly into a ZW3D drawing sheet.

Finished!

The one thing that is not in there is a new suggestion I have if you have to export your drawings to AutoCad format.

When setting the sheet template attributes you may want to set the text font and dimension font to a TTF like Arial. AutoCad does not recognize our default font so in AutoCad our text gets converted to their default font which shifts all the text and dimensions on a drawing.

The procedure when changing attributes of all the existing text on the sheet:

You do not want to simply window pick all the text and change the Font because you will change every attribute to be the same.
Windows pick all the text on the sheet.
Right-click attributes
Click "All Off" (near the bottom)
Click "Toggle" and pick Font
Double click the font you want
This will change only the font and none of the other attributes like height, width, justification.

Appendix A

Best practice settings for the ZW3D Configuration Form

Best practice settings for the ZW3D Configuration Form

Configuration 

General tab

Automatic file locking is only suggested if you are storing files on a File server.

Information on Session Management can be found in the help documents.

The **Automatic file locking** is added to the out of the box settings.

Part tab

The project to expand the capabilities of our Filleting is on-going. You can choose to run the NEW or the OLD engine first.

Background tab

If you change the Background color check **Apply to active object** to apply the color to the active object.

Display tab

Change **2D line thickness** option to a medium thickness for sketch and drawing sheet lines.

If you do not have a Spaceball device, change **3D input device** option to None. ZW3D will start faster if it doesn't have to search for a device that is not there.

Files tab

Object template file, Default Part template, Default Sheet template and Default CAM template are the default templates that will be used if you do not select one when opening a new object.

Templates_MM.Z3 is the default template if you elect to work in mm units. If you import an old template file from V12 (see Utilities > Import Configuration) you will need to edit this name.

ZW3D Files Directory is also new (Version 13). Having this directory outside of the ZW3D file structure is preferred. Make sure that this directory is backed up daily.

User tab

ZW3D will auto-fills these fields, but they are for reference only. User name comes in handy when you are storing on a file server. See the help documents.

Click on the OK button and let ZW3D restart.

Appendix B

ZW3D PDM

Data Management

Efficient product data management is essential to the success of any organization. ZW3D PDM streamlines the flow of documents through an engineering process, thereby fostering communication, cooperation and teamwork. ZW3D PDM's approach enables the key members of an engineering group to work together as a team throughout a product's life cycle.

ZW3D aims to create a management system that ensures that all members of an engineering group, such as design and manufacturing, can work together in a collaborative effort to impose their requirements upon a product's design. This approach provides an environment that encourages and facilitates success and in which members of a team can spend their time producing instead of searching for documents and information.

How Can ZW3D PDM Help You

In order to succeed, an organization must efficiently manage the thousands of documents created per year. Finding the right document with the right revision is an extremely critical issue, since errors due to poor data management may be damaging and costly. ZW3D PDM fulfills this essential requirement by organizing the storage of and access to data and document files, as well as facilitating their flow throughout the organization.

The ZW3D approach reduces the cost of errors at every stage of development. It is well known that errors detected in the prototype and manufacturing stages are costly and time consuming to correct, while errors detected at the earlier development stages can be corrected easily. ZW3D PDM's revision-management feature and electronic vault enable greater control over all the data that is created in ZW3D.

Rapid Product Data Management (PDM) implementation

ZW3D PDM is easy to install, can be implemented rapidly. ZW3D PDM does not require lengthy consulting time and there is no need to change your business structure.

Revision control and life cycle management: ZW3D PDM's revision management options enable you to control the flow of revisions through an engineering group.

ZW3D PDM Life Cycle Management

ZW3D PDM enables users to maintain CAD/CAM information related to their products throughout a product's life cycle. By mirroring the physical process of document management, ZW3D PDM uses check in, check out, release and obsolete functions to manage the life cycle of your documents. ZW3D PDM will create new versions of a CAD/CAM file protecting it from unauthorized modifications.

Definition of Functions

Check In Active

When you add a document to the PDM system to be managed and revision controlled in the PDM system and Vault.

Check In External

When you add a document that is not a native CAD file to the PDM system to be managed and revision controlled in the PDM system and Vault.

Check Out

When you get a document from the PDM system, which is now changed to a new revision that you are working on, in turn the file in the PDM system of the previous version is now locked.

Check Out and Edit

Same as Check Out but opens the file to be edited.

Release Active

When you Release a document you are changing it to read only.

Release External

When you Release a document that is not a native CAD file you are changing it to read only.

New Release and Edit

Creates a new release of the file and opens the file to be edited.

New Release and Insert

Creates a new release of the file and inserts it into the current file.

Obsolete Active

Use this operation on documents that are no longer current versions of your documents and you don't want other users to use them anymore.

Obsolete External

Use this operation on documents that are not native CAD files, are no longer current versions of your documents and you don't want other users to use them anymore.

Setting up ZW3D PDM

All that is required to set up the ZW3D PDM system is a few changes in the ZW3D Configuration dialog box.

One Object per file

When using PDM you will want only one ZW3D Object per file. Check **One object per file** on the General tab in the Configuration form.

User Name

This should be set for each computer containing ZW3D CAD/CAM software to match the current user's name. This setting is on the User tab in the Configuration form.

Vault Locations

This should be set for each computer containing ZW3D to match the specific directory that an administrator would create to be used for the Vault location and the User's work directory. You may set the directory through **Vault directory** and **Work directory** fields on the **PDM** tab in the Configuration form. These locations can be on the local computer or a network drive.

Using the ZW3D PDM File Functions

There are a few different ways to access the ZW3D PDM functions. In this section you will see how to access them.

From the **File > Open** Dialog Box

The image below shows the File Open Dialog box where you can access some of the PDM functions. These functions are:

Change To Work Directory

This will change the directory from where you currently are in your file system to your PDM work folder.

Copy File from Vault

This will copy a read only version from the Vault to the user's work directory for reference.

Check Out

This will copy the file from the vault to the user's work folder in a read/write state.

This will give the user complete control of the file. Also the revision of the file has been incremented by one and the file that is in the vault is locked so that other users can't use an out of version file.

New Release

This is used to take a Released file to be used for editing

Example: If you had Released a file into the PDM system it would be at a specific revision level. We will use "1". When you create a new release you will be changing the revision level to "2.1" this is the first version of the new release. (The revision scheme will be covered later in this document)

This will copy the file from the vault to the user's work folder in a read/write state.

This will give the user complete control of the file.

From the File > Save or Save As Dialog Box

The image above shows the **File Save** or **File Save As** dialog box where you can also access some of the PDM functions. These functions are:

Change To Work Directory

This will change the directory from where you currently are in your file system to your PDM work folder.

Check In

This will take a new file or a file that has been previously checked out and place it into the PDM system.
If this is the first time the file has been checked into the PDM system, the revision level will start at "1.1"
If this is a file that has been previously checked out, the revision level will be incremented to the next number.

Release

This will take a new file or a file that has been previously checked out and place it into the PDM system.
If this is the first time the file has been put into the PDM system, the revision level will start at "1"
If this is a file that has been previously checked out, the revision level will be the current major revision level.

Obsolete

This will take a file and place it in an obsolete state in the PDM system.

From the "File > Manage Vault" Menu

The "**File > Manage Vault**" shows all the options available for ZW3D PDM.

ZW3D provides application menus for interfacing with the ZW3D PDM functions. You can get to these functions by selecting **File > Manage Vault** from any ZW3D level. The table below lists the current ZW3D PDM functions and the menu paths to locate them.

ZW3D PDM Menu Functions	
Select: File > Manage Vault	
Activate	Logging in with your user name and password activates ZW3D PDM. See **User and group Manager** for more information.
Deactivate	This deactivates ZW3D PDM.
Manage Users	Use this to manage (Add/Edit/Delete) ZW3D PDM users and groups. This displays the **User and group Manager**.
Manage Files	This displays the **ZW3D PDM Manager**.
View File	This displays the **ZW3D PDM Manager** and opens a selected file.
Insert Component	This displays the **ZW3D PDM Manager** and inserts a selected file as a component.
Copy File from Vault	This displays the **ZW3D PDM Manager** and copies a selected file to the work directory.
Manage File Network	This applies PDM functions to all files referenced by the active part or assembly. (See **Manage File Network** below).
Select: File > Manage Vault > Life Cycle	
Check In External	Check any external file into the vault.
Check In Active	Check the currently active file into the vault.
Check Out	Check a file out and copy it to the work directory.
Check Out And Edit	Check a file out to the work directory and open it.
Check Out And Insert	Check a file out to the work directory and insert it as a component.
Release External	Release an external file.
Release Active	Release the currently active file.
New Release	Create a new release of the selected file and copy it to the work directory.
New Release And Edit	Create a new release of the selected file and open it.
New Release And Insert	Create a new release of the selected file and insert it as a component.
Obsolete External	Obsolete an external file.
Obsolete Active	Obsolete the currently active file.


Once you set the vault and work directories using the **PDM** tab of the **ZW3D Configuration Form**, the **File Browser PDM** drop down list activates to allow some of the PDM commands to be executed directly. The drop down list is specific for the Save and Open PDM functions.

The ZW3D PDM Manager

Invoked By: **File > Manage Vault > Manage Files**

With the **ZW3D PDM Manager** you can search for and manage ZW3D object files and other non-ZW3D files such as word processor, database and spreadsheet tables associated with your design projects throughout their life cycle. You can also perform product life cycle functions (e.g., Check In, Check Out, Release, etc.) and set user and group permissions using the ZW3D PDM Manager.

Current Directory

Use the drop-down list to switch between the vault directory and your user directory. File searches will be performed on the current directory.

File Filter

To filter the current vault directory (or your user directory); enter the filter criteria in the **File Filter Form** (see below). The **ZW3D PDM Manager** will locate and list those files that match all of the fields provided.

File List

The results of a file search are listed here. All files are listed if no search criteria are entered.

Revision List

The revision history of the selected file is listed here. Each revision represents a file that is listed in the **File List**. You can select a file from the File List by selecting its revision level from the **Revision List**. You can also review a files revision history using this list. You can perform **Object Editing** (RMB Functions) on selected files. See **Mouse Button Functionality** for more information.

State List

This list shows the current state of the selected file (e.g., New, Checked In, Checked Out, Released or Obsolete). The user and group name and their access level to the file are also

listed. See also **Changing a File's Release State** and **Changing ZW3D User and Group Permissions** for more information.

Preview

The preview of the file is shown here. You must use the **Capture Preview** command while the file is activated prior to checking it into the ZW3D PDM vault. You will be reminded to do so first if no preview is available for the file.

Mouse Button Functionality

File List

Right-click on a file to display the following **RMB** functions. Some functions may be disabled depending on the current state of the file.

Check In
Check Out
Release
New Release
Obsolete
View File
Copy File
Delete File

State List

Right-click on an item and select "**Change State**" to display the **Change File State Form**. You can change the state of the file or the user and group permission for the file. See also **Changing a File's Release State** and **Changing ZW3D User and Group Permissions** for more information.

File Filter Form

Use this form to filter the **Files List** (see above). If you are just looking a file and you know the file name, complete only the **File Name** field. If you know the revision level or the user name, complete those fields as well to limit the filtered list. If you're not sure, leave the fields blank. Otherwise, the file may not be located.

When you are done entering the search criteria, put a check in the box next to the fields that you want to search by or press <**ENTER**> in any of the input fields. You can enable or disable fields to modify the search results. If no search criteria are entered, all files in the current directory are listed.

The File Filter is only active as long as the menu is visible. If the user closes the File Filter menu then the values reset to defaults (= show everything).

Keyword - Finds all files that contain the specified pattern (keyword)
Hide inaccessible files - Check this box to hide all files which are inaccessible to the current user because of their permission settings.

Changing a File's Release State

Invoked By: **File > Manage Vault > Manage Files**

 You can change a file's release state (e.g., New, Checked In, Checked Out, Released or Obsolete) by right-clicking on the release flag and select **Change State**. This is done in the State list of the **ZW3D PDM Manager** while the file is selected from the File list of the manager. See **Mouse Button Functionality** and the **Change File State Form** for more information.

Changing ZW3D User and Group Permissions

Invoked By: **File > Manage Vault > Manage Files**

 You can change a file's user and group name and permissions (e.g., read or write) by right-clicking on the USER or GROUP flag and select **Change State**. This is done in the State list of the ZW3D PDM Manager while the file is selected from the File list of the manager. See **Mouse Button Functionality** and the **Change File State Form** for more information.

Change File State Form

Invoked By: **File > Manage Vault > Manage Files > Change State**

Use this form to manually change a file's release state, user and/or group name and file permissions. It should not be used during regular use of the ZW3D PDM system.

The form will default to the current state of the selected file. Use the **State** section to change the file's current state (e.g., New, Checked In, Checked Out, Released or Obsolete). Use the **User** section to change the User and Group name and assign read/write permission to the assigned user or group. Use the Description to modify the description assigned to the file.

User Permissions Form

Invoked By: **File > Manage Vault > Manage Files > Change State > User**

Use this form to assign user permissions for the file. First select, **File > Manage Vault > Manager Users** to set up your users using the **User and Group Manager** (see below). Then using the **User Permissions Form** shown below, select the User from the first drop down list and then check the box next to **Read** and/or **Write** to assign permissions to that user. Repeat the process for up to ten users.

Group Permissions Form

Invoked By: **File > Manage Vault > Manage Files > Change State > Group**

Use this form to assign group permissions for the file. First select, **File > Manage Vault > Manager Users** to set up your groups using the **User and Group Manager** (see below). Then using the **Group Permissions Form** shown below, select the Group from the first drop down list and then check the box next to **Read** and/or **Write** to assign permissions to that group. Repeat the process for up to ten groups.

User and Group Manager

Invoked By: **File > Manage Vault > Manage Users**

Use this form to create and manager users and groups. You can assign names and passwords for each user. You can also assign administrator privileges to a user. Refer to the list of options below.

User list - The upper portion of the manager lists the current users as they are created. Each user is assigned either an Administrator or a User privilege as indicated next to the user's name. Pick **Yes** or **No** next to **Administrator** to assign this privilege to a user.

User name, Password - This is the login name and password that users will enter when they activate the **ZW3D PDM Manager**.

Groups - If the user is assigned to one or more groups, they are shown here. See Select Groups to assign a new user to a group.

Administrator - Pick **Yes** or **No** next to assign Administrator privileges to a user.

Select Groups - When creating a new user, pick this option to select one or more groups that the new user will be assigned to. Read and Write privileges to files can be assigned by user or group. Refer to the **Change File State Form.** Also refer to the **User and Group Manager**.

Clear Input - This clears the values in the User name, Password and Groups fields.

Add User - This adds a new user to the User list based on the User name, Password and Group fields.

Delete User - Select a user from the **User list** and then pick this option to delete them.

Show Groups - This lists the groups that are currently defined. Refer to the **User and Group Manager**.
Add Group - Pick this option, enter a name for the group in the popup form and then pick OK to create the group.

Delete Group - Pick this option, select a group from the list and then pick OK to delete the group.

Manage File Network

Invoked By: **File > Manage Vault > Manage File Network**

Use this command to apply the **Check Out**, **New Release**, **Check In**, **Release** or **Obsolete** PDM functions (see **The ZW3D PDM Interface** above) to all the files referenced by a selected component or the active part/assembly. See below for additional options.

Required Inputs

Component - Select the component to apply PDM functions to or middle-click for the active part or assembly.

Optional Inputs

Option - Select the PDM function to apply.

Include components - Check this box to include child components of the selected component or of the active part/assembly.

Include hard-linked objects - Check this box to include hard-linked objects. See Tools > Link Manager for more about hard-linked objects.

Include imported files - Check this box to include imported files referenced by the part history of the selected component or of the active part/assembly (e.g. IGES or point cloud files).