ZW3D
Progressive Die Training Guide

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ZW3D™ V2014 Progressive Die 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 **Progressive Die 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 **Progressive Die 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

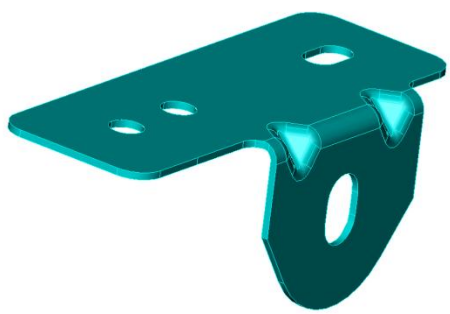
Content

ZW3D Progressive Die Training Guide

This training guide was put together for the purpose of covering all of the fundamental aspects of creating a progressive die assembly in ZW3D which is associative to the Strip. We will make changes to the customer part and strip to show this associativity in action. We will begin by importing a customer part and producing an accurate strip layout for the part. We will also work with standard punches and pilots and you can optionally work through a standard part creation exercise in the Appendix section. We will not create the entire assembly, but we will leave this course with the necessary tools to complete the entire die assembly. We will also cover topics important to detailing and communicating your engineered models to manufacturing.

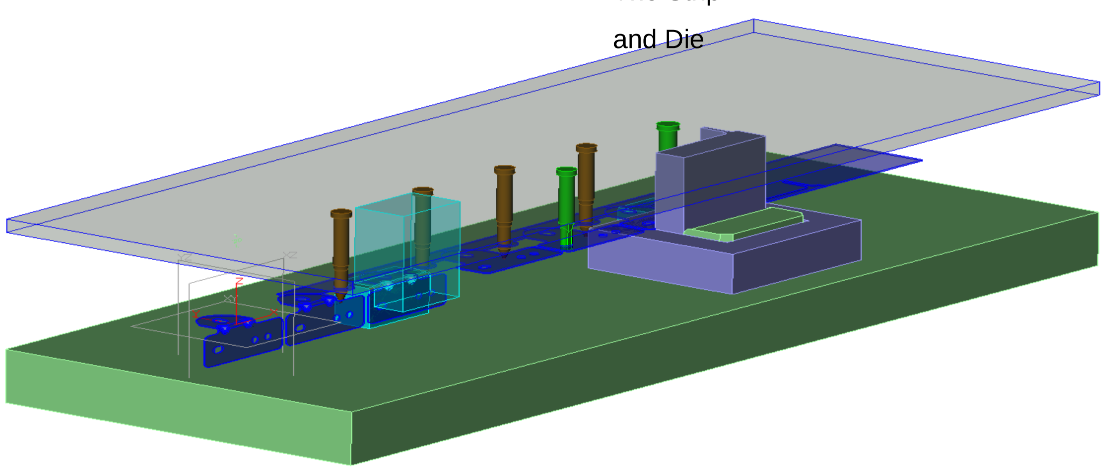
Customer Part

Part5361.X_T



The Strip

and Die



Lesson 1 – Importing the Customer Part

Let's get started.

1) We will start by creating a **Multi-Object** file. We will name it according to the part we will be importing. Let's call it **Job_5361.Z3.**

2) Next we will create a **new part object** to import our Customer's Parasolid file into.

NOTE: I like to leave this part imported as the customer sent it. If it needs some repair it could be done here or in an intermediate part object.

3) Pick **Part/Assembly** and choose the Inch Part Template and name the New Part Object "**001_Customer_Part_5361**".

IMPORTANT: Object name should remain unique, so if you like to use generic names from job to job, simply add the job number into the name.
NOTE: If you normally work in mm.

Since this lesson is done in INCH units you may want to delete or rename the ZW3D Config file in your USER directory and restart ZW3D, pick INCH units when prompted. The other option is to use the EDIT > Preference > Units > IN in every part object you open.

4) Check your part preferences as shown to the left. These are the defaults, make sure yours match. You may want to edit your Template for your Inch Parts.Now we are ready to import the customer's part.

5) Choose **File Import.**

Check off **Parasolid** – check your import options.

Use the **HISTORY –No Backup** option

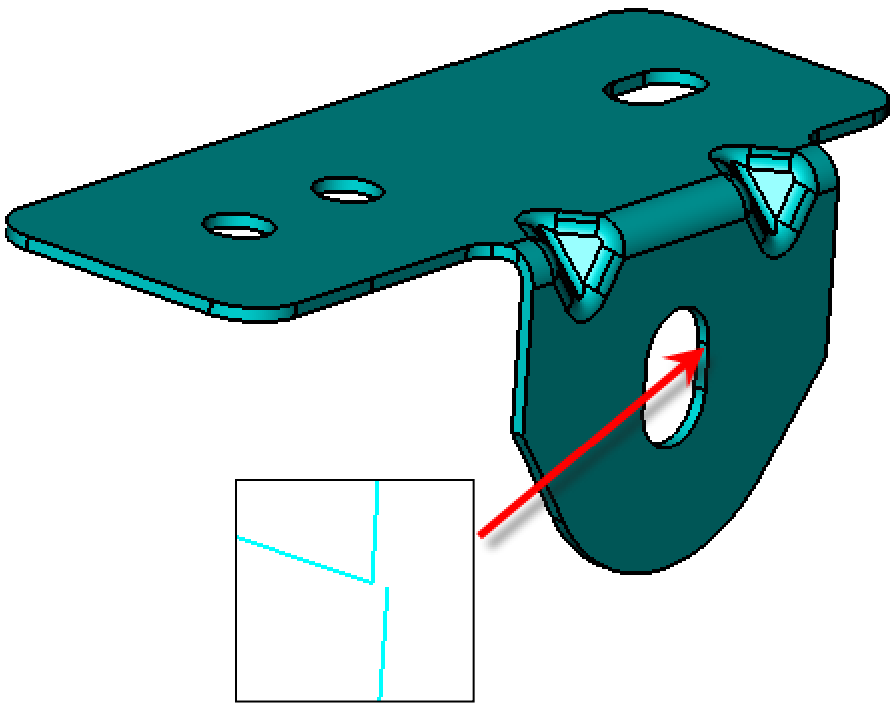
The File Part5361.X_T will be found on the training folder under the program directory.

The image to the right will be your result.

6) **Change your face attributes** and set the front face color to cyan.

7) Your import requires **HEALING**. See the area of the part indicated below. Pick the **Heal/Analyze** command from the far right of the Heal Tab. 

8) Set the tolerance to .0003 and hit OK.

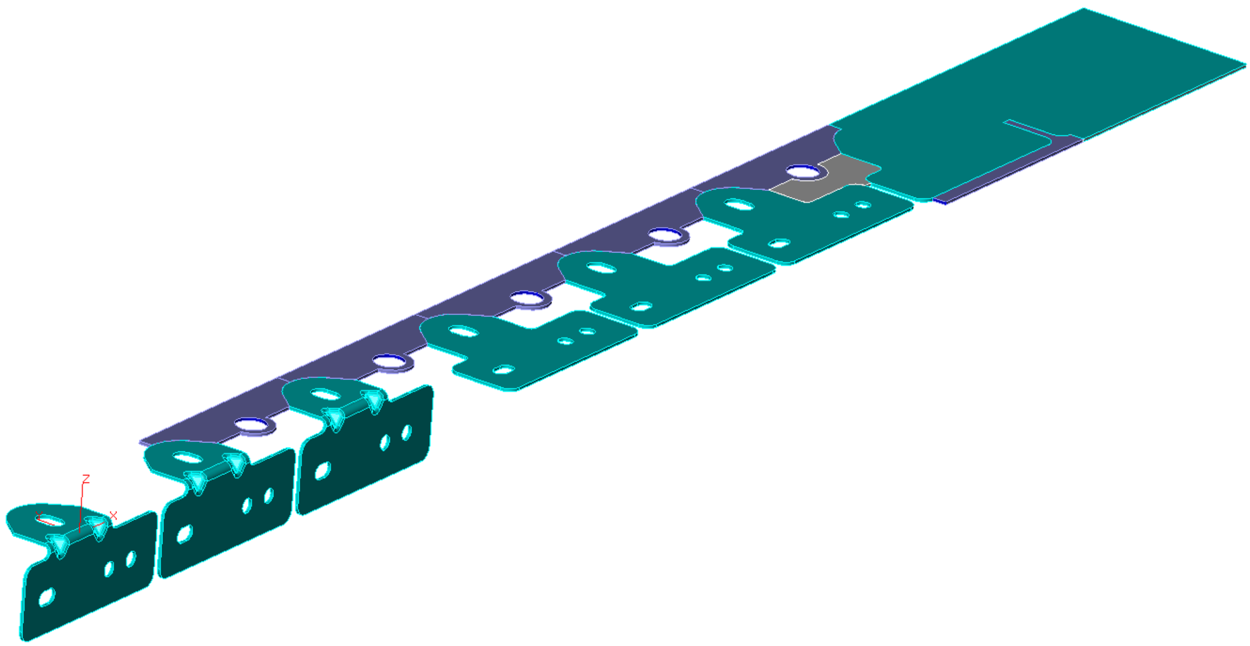


Notice that the gap has been fixed.

Save. 

Lesson 2 – Strip Development

In this exercise we will use an associative copy of the customer part to develop the strip. And we will look at design methods to establish geometry that can be used later in the development of custom punches, dies and blocks to best maintain associativity to the strip.



Positioning the Customer Part

1) **Exit** Part to get to your object manager.

2) Create a **New Part Object**. Name it: **002_Strip_5361.**

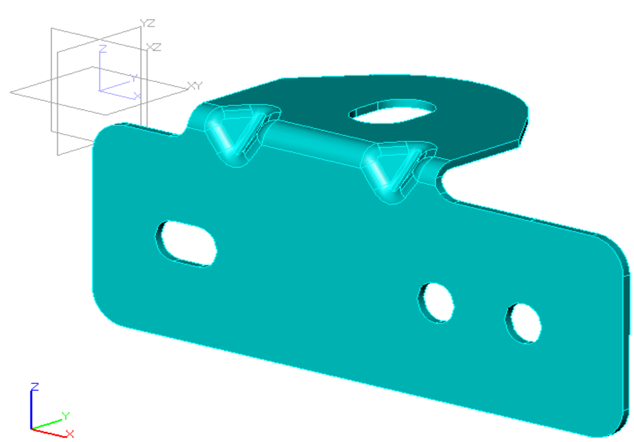
IMPORTANT: You should still be in the same file. Verify this by looking at your title bar.

Next we will want to create an associative copy of the customer part and position it as the final cut off stage in our strip.

3) Right-Click> **Insert Component.**

Choose the **Customer_Part** from the list of parts and position it in space. In the following steps we will use the assembly alignment tools to position the part relative to zero, so this initial placement of the copy is not critical.

Desired Location



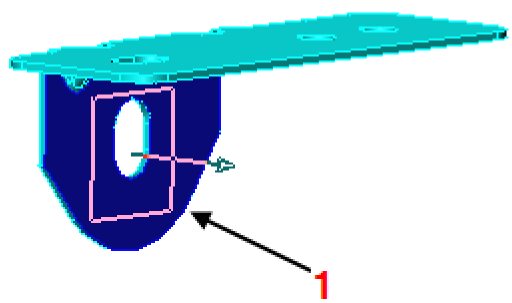
The image below shows how we will be positioning our part.

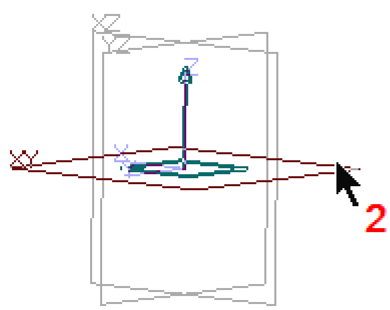
4) Using the Align command, position the part as shown in the image on the previous page. With the longest part of the bent tab pointing in the Positive X direction. The Align Command can also be access by Right-Clicking on the screen and picking Align.

HINT:
Set your **Entity Filter** to **Face/Datum** and your ‘**Snap to’ or ‘pick filter’ to On Entity.**

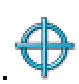
Firstly, align the part face indicated below(1) with the **XY** datum(2). 

Check the **opposite** option.

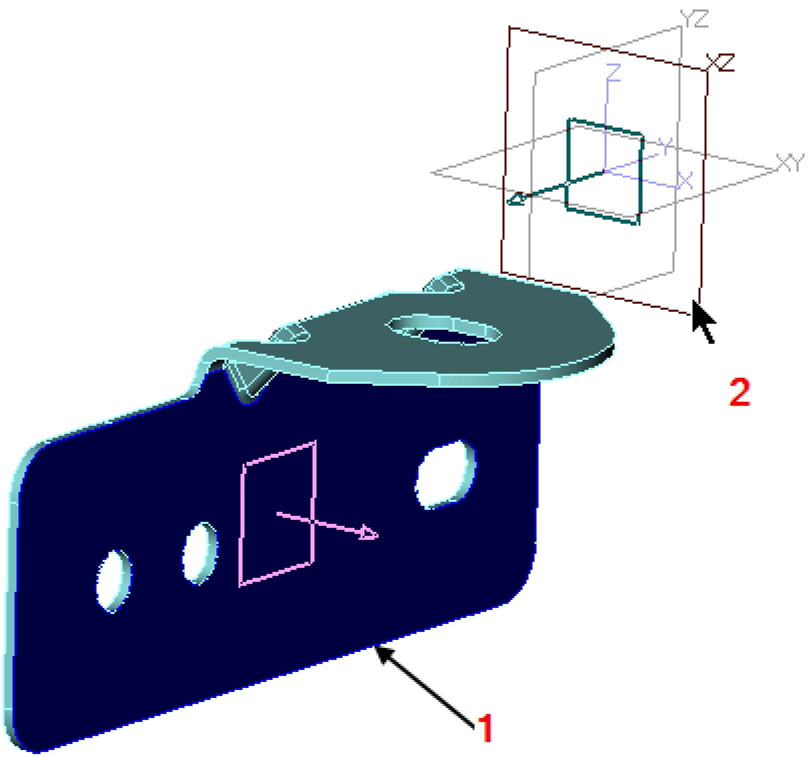




1: 1st entity 2:2nd entity

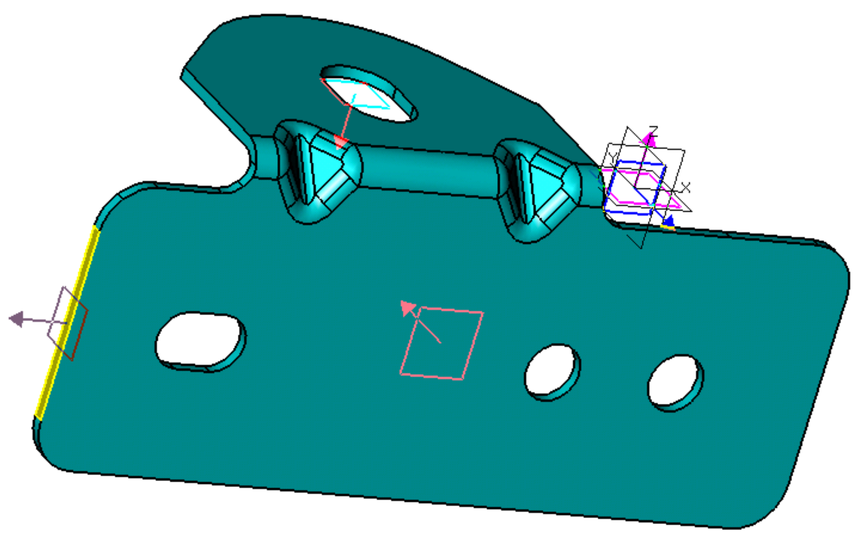
Then, align the part face indicated below(1) with the **XZ** datum(2). 

Check the **opposite** option again.

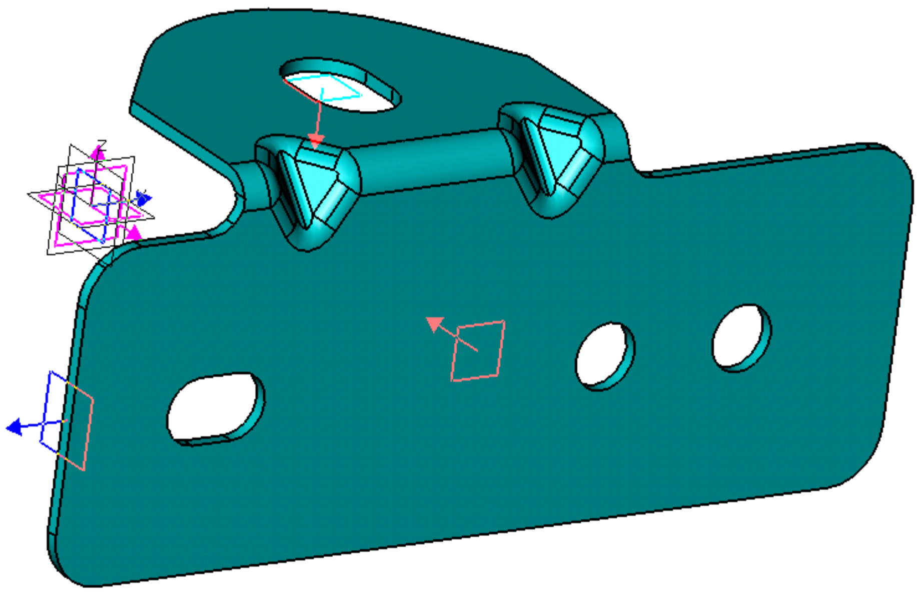


1：1st entity 2:2nd entity

For this last alignment we want the left side of the part to align with the YZ Datum.



Resulting alignment should be consistent with the following image on this page.



You should now **Merge** the component into the active part. (**Merge** command from the shape Tab )

Sheet Metal Attributes

We will want to establish our Sheet Metal Attributes for our part, so that our unbending is accurate.

5) Using **Inquire commands** verify the **inside bend radius** of your part.

6) Then Set your Sheet Metal Attributes.

Bend Radius：0.07
K-Factor:0.45
Flat Tolerance：0.004
Ratio：1.5

You will see an item in your **History Manager**  that shows the setting of your Sheet Metal Attributes.

Establishing a Variable to Drive Pitch

We will now want to make several copies of this part along the X Axis to begin designing the strip at each stage in the die. ZW3D allows us to create a driving parameters or variables such as pitch to offset each copy, so if a change would need to be made to the pitch in order to accommodate tooling we could change the offset of any parts that would be driven by our pitch variable.

7) Using an Inquire command determine the length of your part. Our pitch value must exceed the length of the part.

You get the same results as following: Distance=3.346457/ Dist-X=3.346457/ Dist-Y=0/ Dist-Z=0.

8) Choose **Insert Variable** from the Pull Down menus.

9) Fill in the form as following: Variable=Pitch, .Expression=3.5.in and check the **Distance** box.

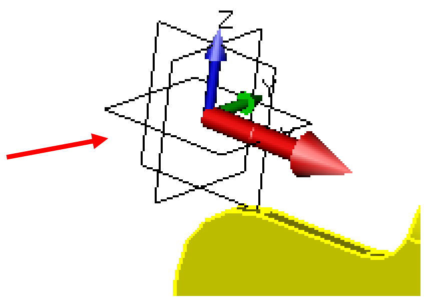
NOTE: ZW3D Variables are case sensitive and it is good practice to specify units.

Creating the Stages on the Strip

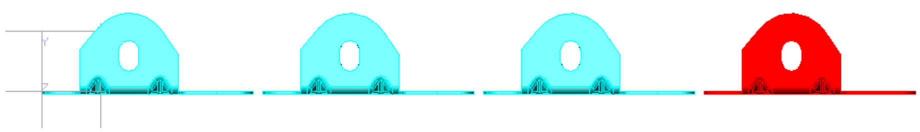
The fastest way to build previous stages in your Strip progression is to simply copy the previous ones until a significant change occurs. Make the changes then copy that stage for the next set of previous progressions that are similar and repeat. In this case we will make two sets of copies and in a later exercise come back and insert a pre-strike stage.

10)Choose **Edit >Copy>Copy Entities,** and pick the **Copy along a direction** from the pull down menus or choose the **Copy Along a Direction** command  within the **Copy** command under the Shape Tab.

Entity is the Main Shape
Direction: X Axis (choose this option from the Right-click menu)
Distance: Type in <u>**Pitch**</u>
Hit <ENTER>
Angle: Zero
Copies: 3



11)We need to unfold the last copy to create the first stages of our Strip Progression. We will need to remove the strengthening darts/gussets in order to be able to unfold the part.

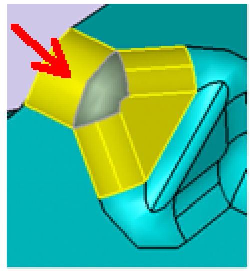


Top View We will unfold this one.

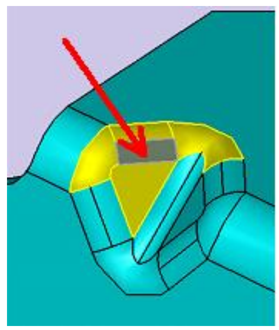
12)Let's isolate the shape/part for editing by **Blanking**  it then swapping visibility  of blanked entities.

13)**Erase** the Front and Back faces of the strengthening darts/gussets.

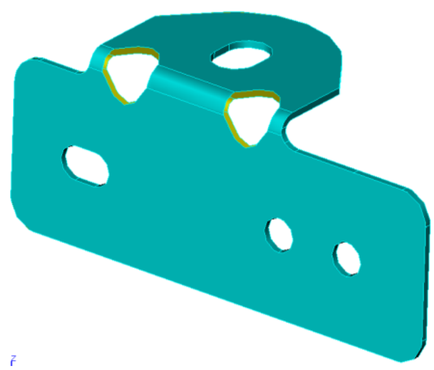
Don't pick the place as indicated in the right figure below.



For the place in the figure below, Try the Right-click Face region option.

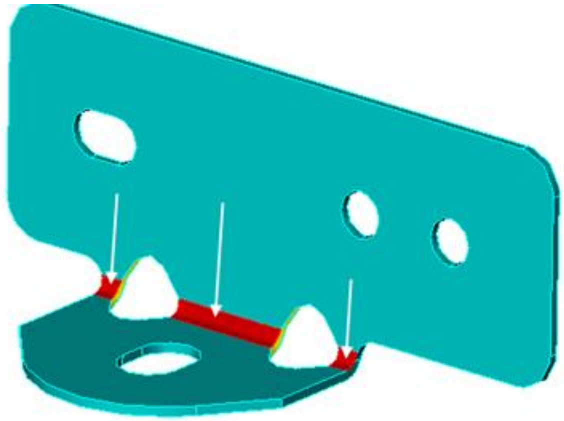


Your part should look like the images below.



14)Use the **Mark Bend** command to teach ZW3D where the bend is on this part we imported from Parasolid. 

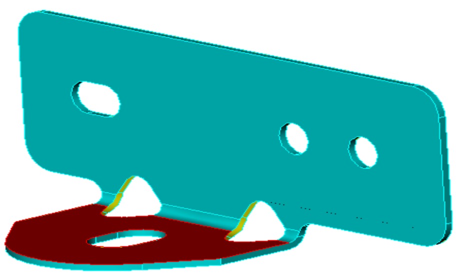
--Pick the three inside faces of the bend as highlighted and indicated in the image below.
--K factor = 0.45.

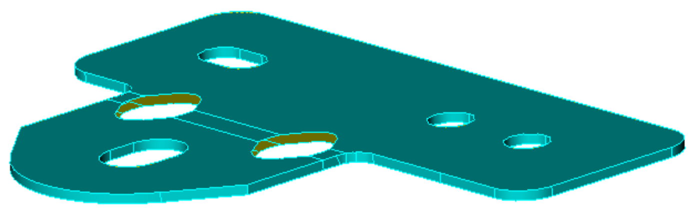


Make note that this is an open shape – faces are missing, yet ZW3D will let us flatten it.

15)Choose **Unfold** Command. 

Set the Optional input for a Stationary Face.

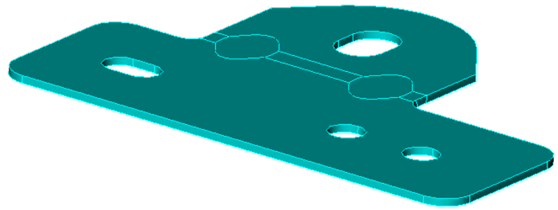




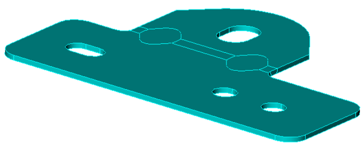
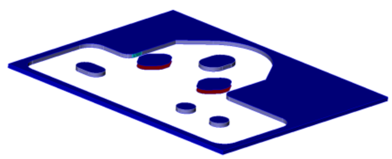
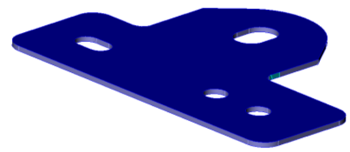
Pick the Face Highlighted in the image to the right below.The Image to the right above should be your result.

16)Use the **Fill Gaps** command from the Heal Tab to fill in the openings in the part. 

No need to pick; just hit **OK**.



Next we will simplify the flattened stage to remove the various faces that represent the bend lines and the filled gaps for the purpose of making the flattened stage have a cleaner appearance. The following images show the progression.

 to  to 

17) Choose **Insert -> Features -> Stock** from the pull down menus or the Mold and Die Tab.

18) Pick the Flattened Stage as the Shape for the stock and specify the XY datum as the plane.

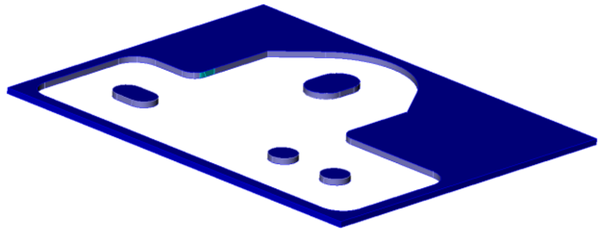
--Since the datums are blanked you can select them from the History Manager or simply type in **XY** in the options form. XY is also the default, so you could MMB.

--Specify an X and Y offset of **0.1** to make the stock slightly larger than the part.

Notice the stock is created with a transparency attribute so you can see your part through it

19) Choose **Shape** from your selection filter. Right click on the stock and choose Remove. You will be prompted to select a shape to remove. Right-Click and choose Pick-All, then click OK to finish.

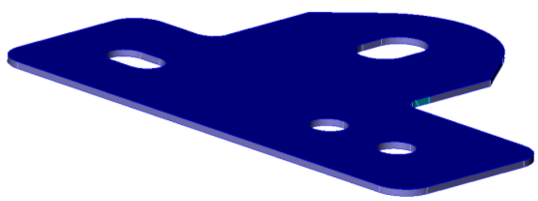
Your result will be as follows…



20) Insert Stock one more time and pick the large shape to drive the stock size and pick OK No offsets are needed for X or Y Lengths.

21) Right click on the stock and choose Remove. You will be prompted to select a shape to remove. Right-Click and choose Pick-All, then click OK to finish. Your result will be as follows.

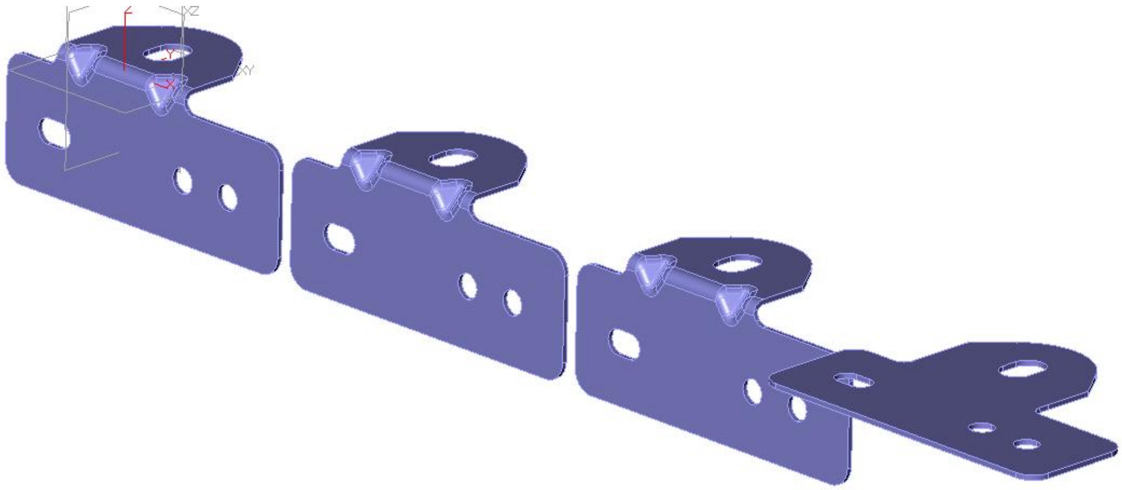
Note : This time when picking the Base, pick the stock near the middle to get the correct shape.



We no longer want the transparent attribute so right-click on the Shape and pick **Attribute**.

Adjust the **transparency** to zero and pick OK.

22) Pick the **Unblank All** command . Your results should be as shown below,

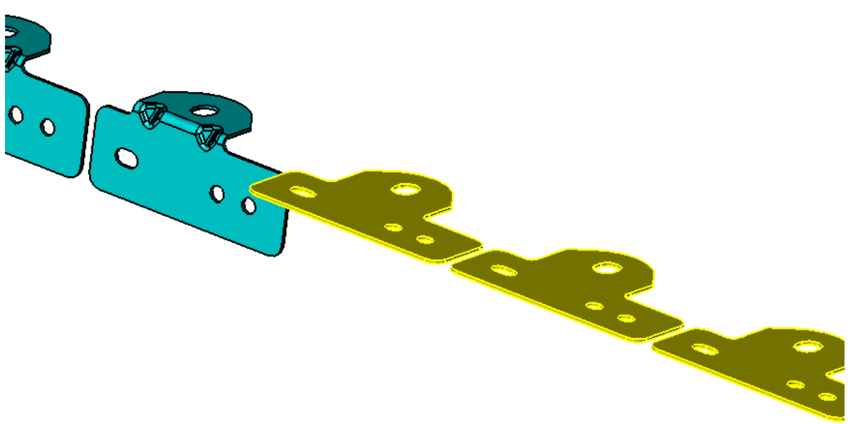


Now we need to make four copies of the flattened stage to run out the rest of the strip.

23) Choose **Edit >Copy>Copy Entities,** and pick the **Copy along a direction** from the pull down menus**.**

Direction: **X Axis**
Distance: Type in **<u>Pitch</u>** and Hit <ENTER>
Copies: **4**

4 Copies total only 2 are shown in the figure below.



Developing the Blank, Carrier Pilot and Pitch Notch

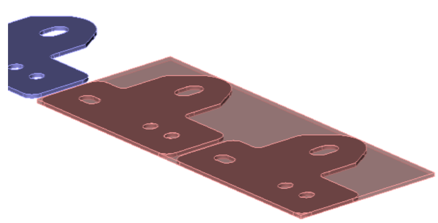
We will develop the blank area of the strip along with the carrier and pilot hole. We will develop these shapes with the intent of maintaining maximum associativity through the design process. In the end, our strip will be made up of many shapes rather than one shape with all of the pieces united. This will allow us to make associative copies of any individual piece(s) for the development of other associative parts/assemblies. Also if multiple shapes are needed in the creation of another part these can be united at that time. It is easier to divide them up now and unite them later as needed.

THE BLANK

Development of the Blank is easiest to create when done with a stock command whenever possible. The Stock Command allows you to create a shape that envelopes other shapes or faces with a minimum usage based on a datum orientation. You can incrementally or nominally set values for its size.

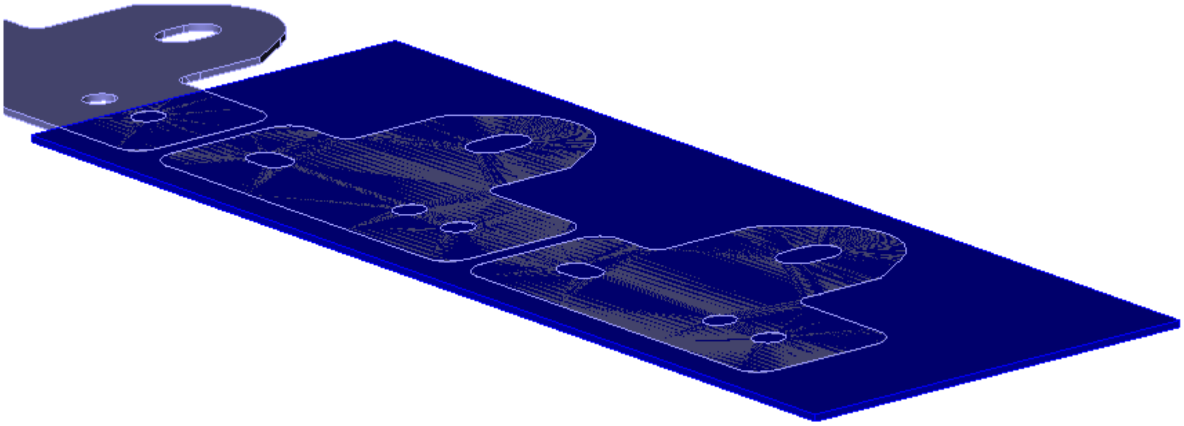
24) Begin by Selecting the **Stock Command** found under the **Insert - Feature** pull down menu. It is also found on the Mold and Die Tab. 

- Set your filter to Shape and pick the first stages in your strip to define the blank area of your strip. See the image below.



- Then fill in the Stock form as shown below.
- Plane is XY.
- Length X has a side .
- increment of 1 (longer).
- Width Y is set to a nominal value of 3.

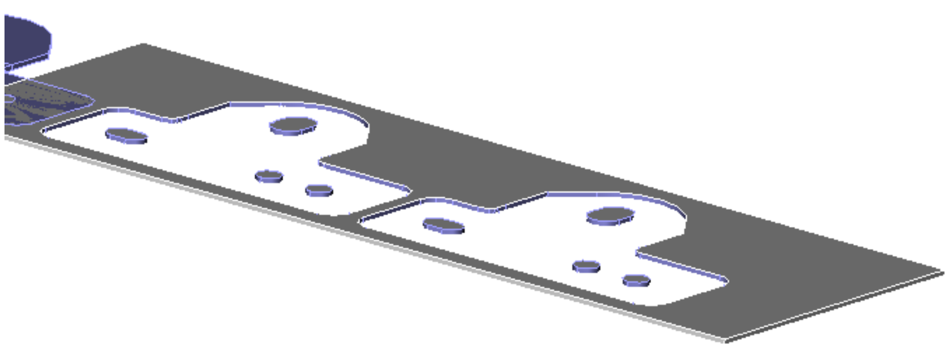
The final results are as shown in the image below



25)**Combine Shape**  > **Remove** . The next step will be to remove the first two stages of the progression from the stock we created. Be sure to check the box to **Keep Removed Shapes**. This will give us a strip blank made up of all of the pieces needed to create custom punches and die blocks.

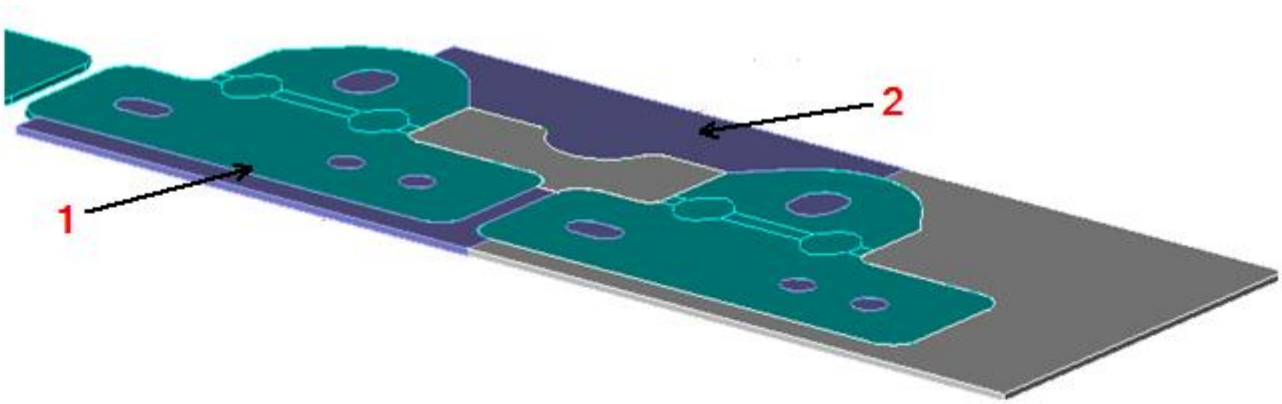
26)**Blank** the two stages you used as tool bodies in the removal operation of the previous step to verify your results.

You should see the following results…



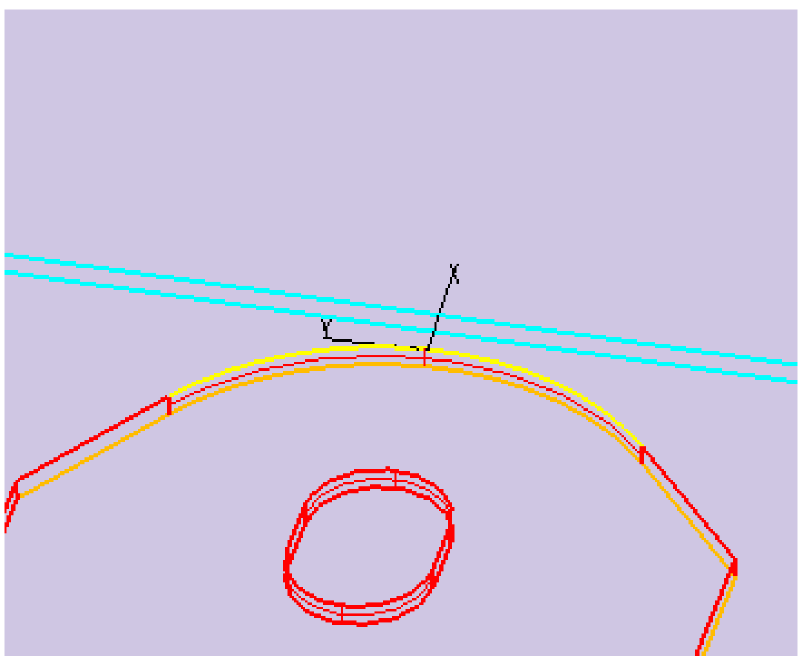
27) **Unblank All**. 

28)We need to establish the blank area as it relates to the various punching operations, so we will split up this shape with a few datums and sketches to establish the pitch notch and carrier/pilot pierce as shown in the image below…

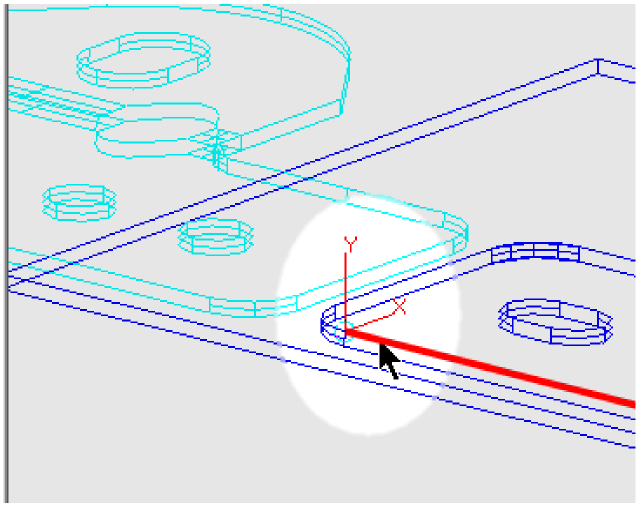


1: Pitch Notch 2: Carrier Shape

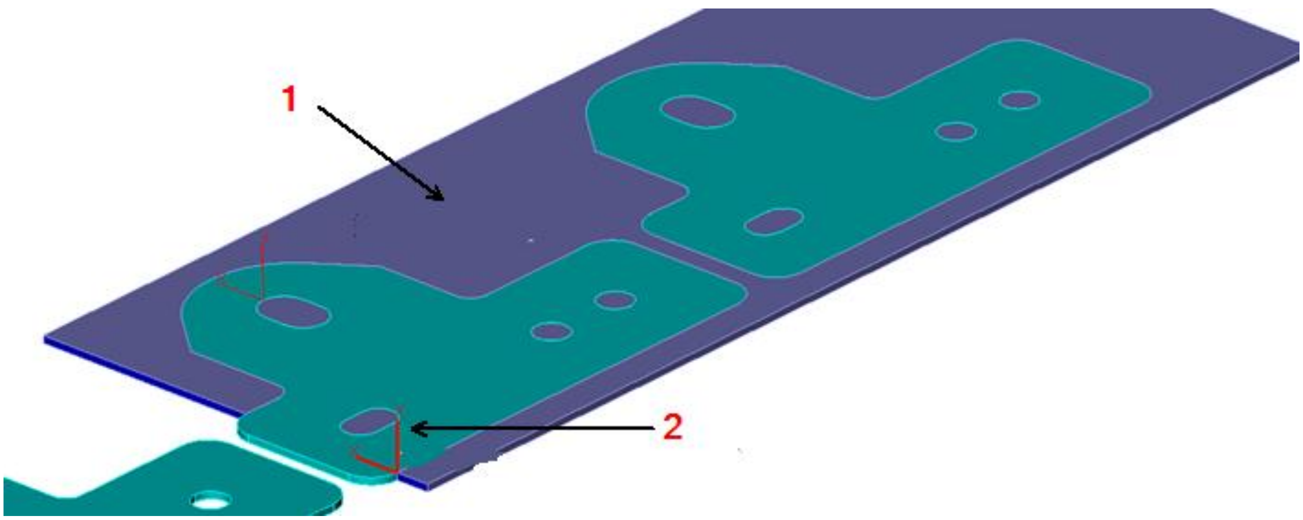
29) Insert a **YZ Datum** establishing its Origin at the **middle** of the **edge** shown in the image below. This datum will be used to split up the Blank area to help create the carrier portion



30) Insert a **YZ Datum** establishing its Origin at the Critical endpoint of the edge as shown in the image below. This datum will be used to cut off the main blank area to establish the pitch notch.

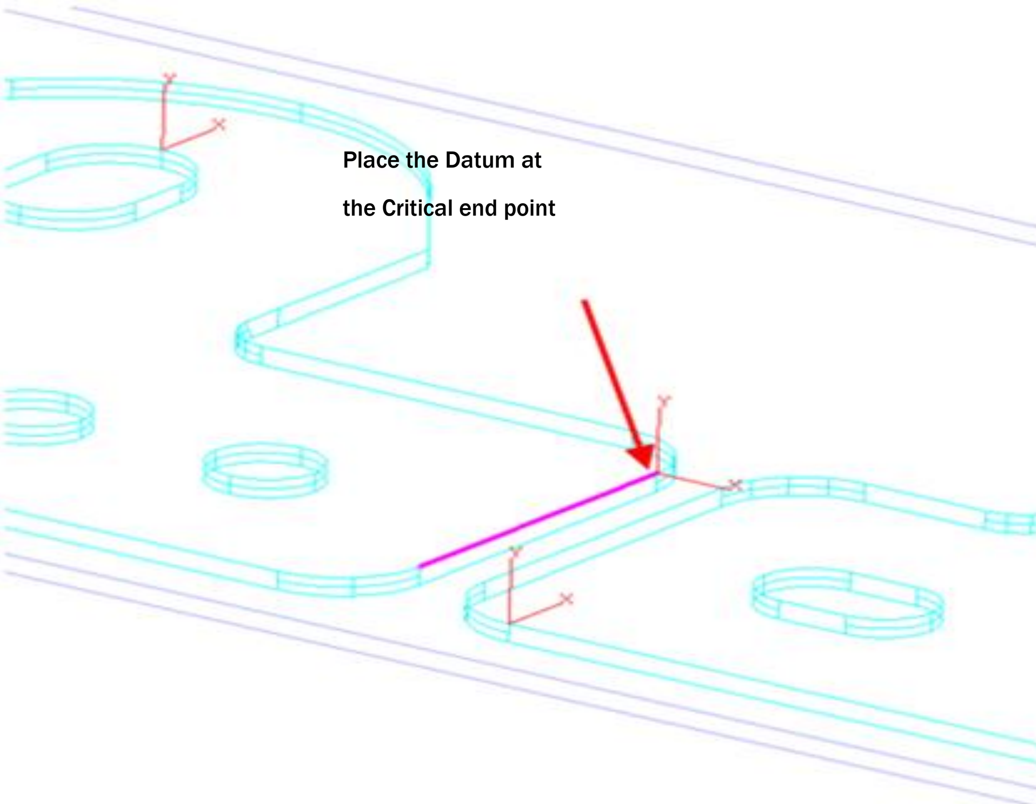


31) Using the **Trim Shapes** command then **Trim** the main blank shape(number 1 arrow) **with the datum**(number 2 arrow) you just created.



Your results should match that shown in the **image below**…

32) The two datums we created need to be copied along the X Axis at a distance of Pitch to establish the other end of the Pitch Notch and Carrier portion.



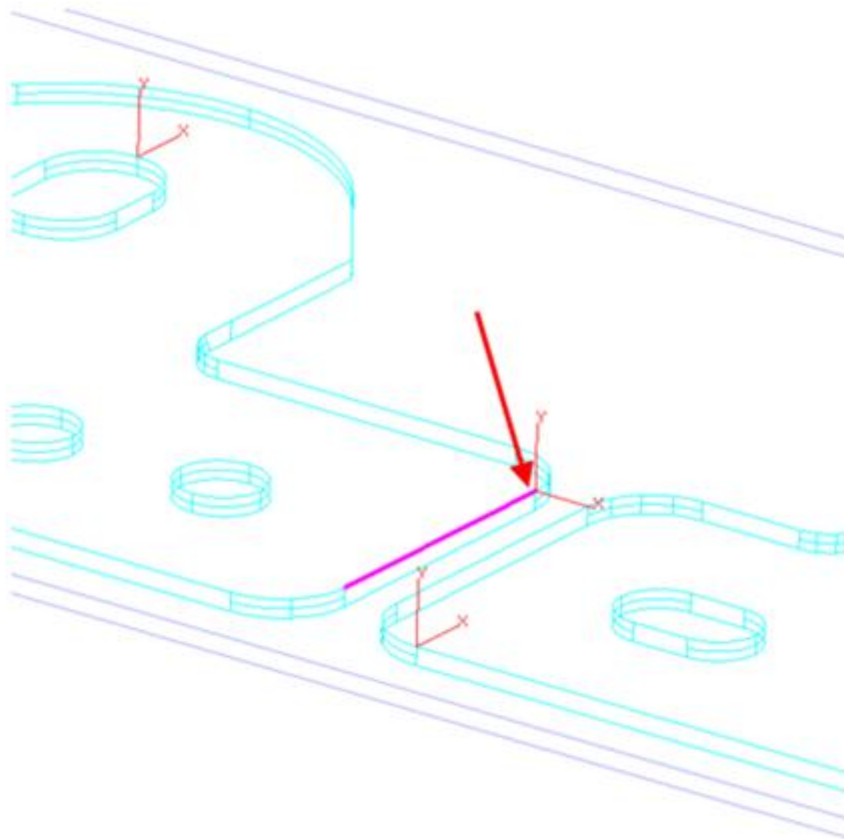
Using the **Copy Along a Direction** command make one copy of both Datums along X at a Distance of **Pitch.**

33)We need to create one more datum to split out the shape for the pitch notch.

The fifth datum is a **XZ** Datum with an origin at the end of the edge shown below…

The resulting Datum is shown for clarity.

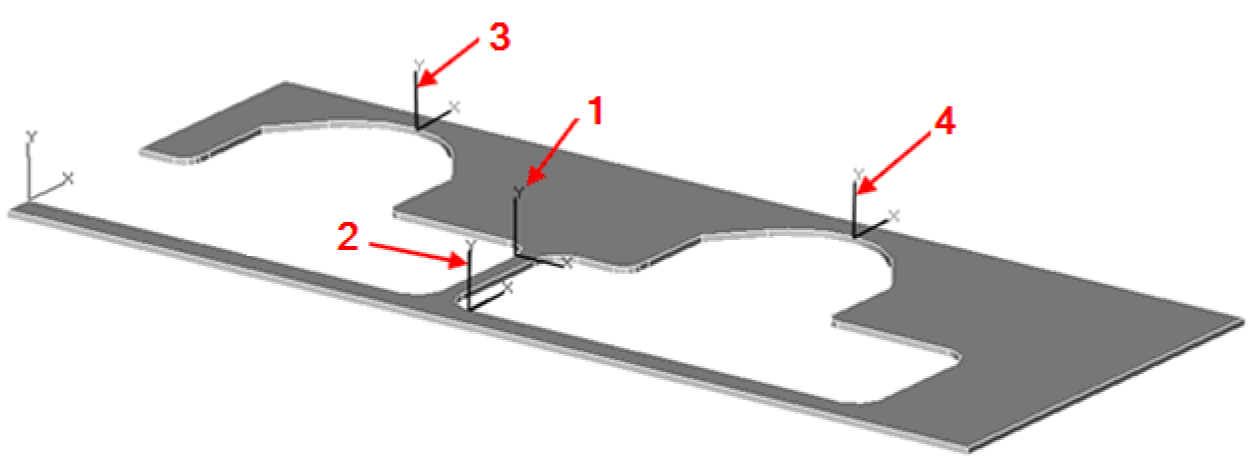
Note: place the datum at the critical end point of this edge.



34) Now let's start splitting up this shape to breakout the Pitch Notch and Carrier Shape.

35) The following image will help you understand the next set of instructions.

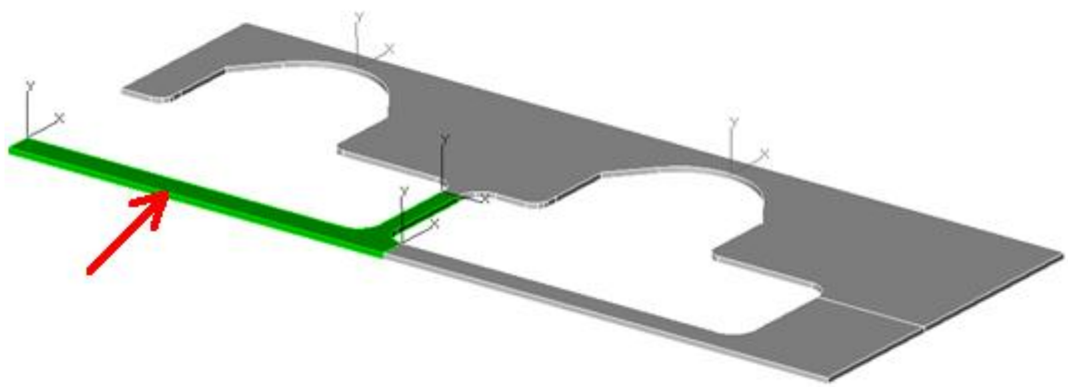
36) **Blank** everything except what you see below.



1: Datum1 2: Datum2 3: Datum3 4: Datum4

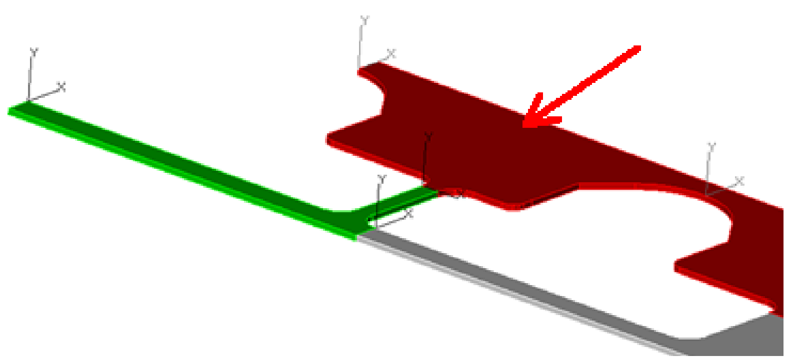
37) Using the **Divide Shape** Command  (shape Tab) Divide the shape shown above with Datum 1.

New Shape For Pitch Notch is indicated in the figure below.



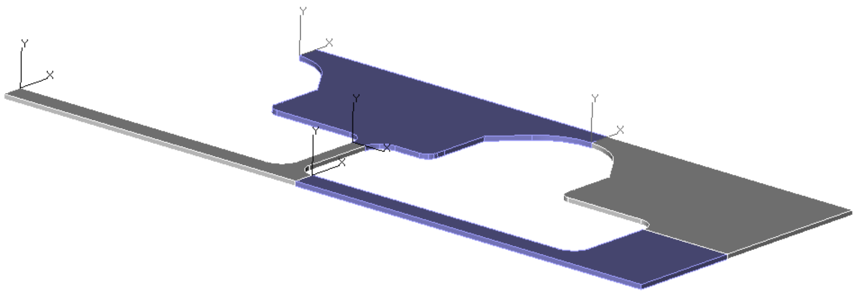
38) Repeat the command and divide the resulting shape with datum 2.

39) Using the **Trim Shape** command, trim the largest remaining shape with Datum 3 keeping the right side. (as indicated below)



The remaining shape from the previous trim needs to be split using the **Divide Shape** command and dividing it with Datum 4.

Results so far…

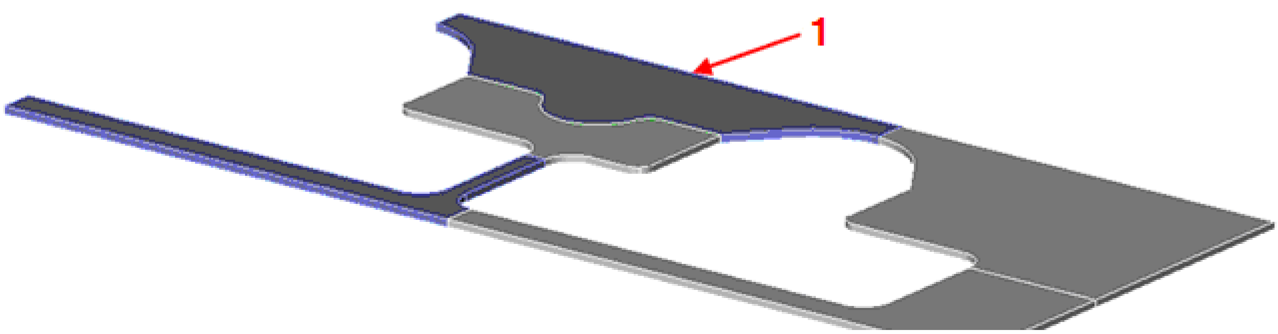


You may be noticing some transparency has snuck back in because of our use of the Stock command.

Put you filter on shape – window pick the 4 shapes – middle-click - Attributes and Pick OK

Next we need to complete the Carrier shape, but this split requires a surface rather than a datum to split the remaining shape. The following image shows the carrier shape we will be creating…

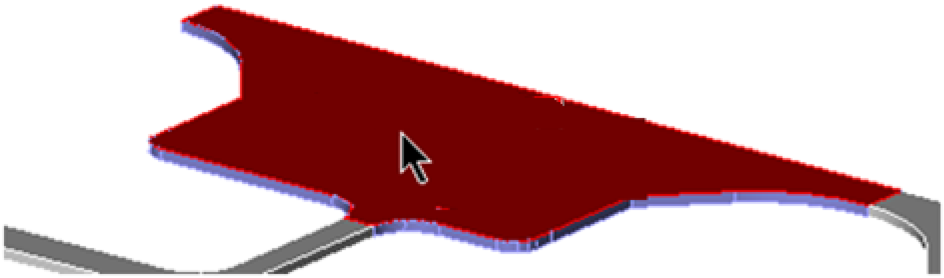
1:Carrier Shape



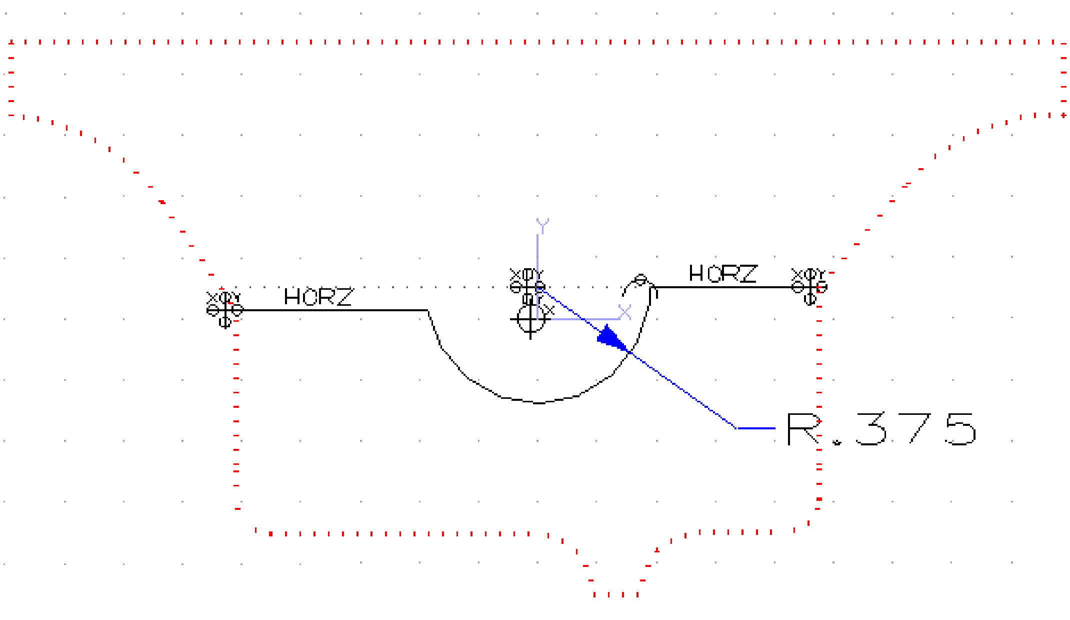
1:Carrier Shape

41) Create a sketch on the face shown in the image below.

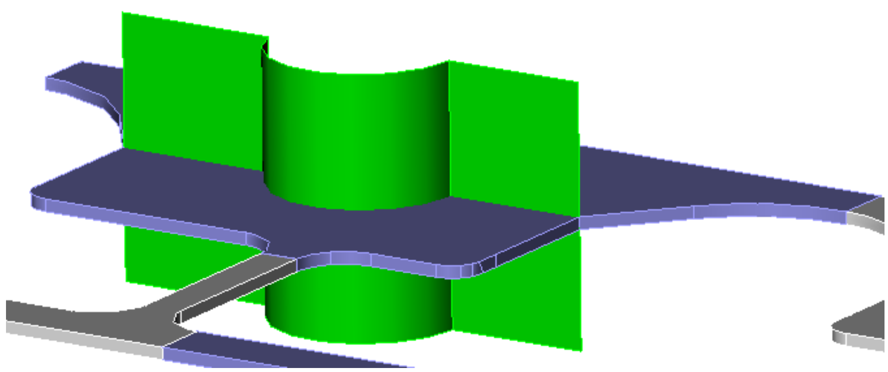
Be sure to check the **Reference Face Edges** Box in the Sketch Options form.



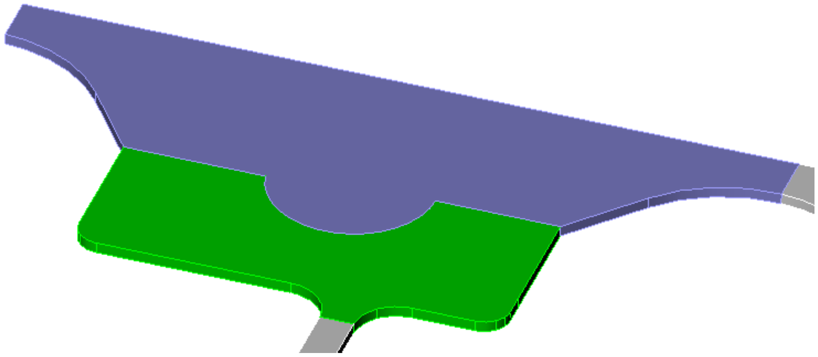
Construct the sketch as shown below. Keep your sketches simple. We can put fillets on the shapes rather than in the sketch if you so desire.



42) Extrude the Sketch as a Base Shape to an extent that exceeds the thickness of the part.

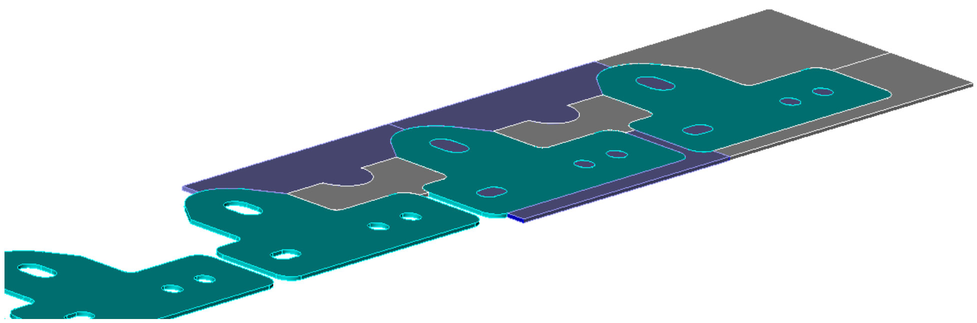


43) Then **Divide** the intersecting shape with the surfaces you extruded.



44) Now we will make one copy of the two shapes we just created. Copy along the Negative X direction a distance of Pitch.

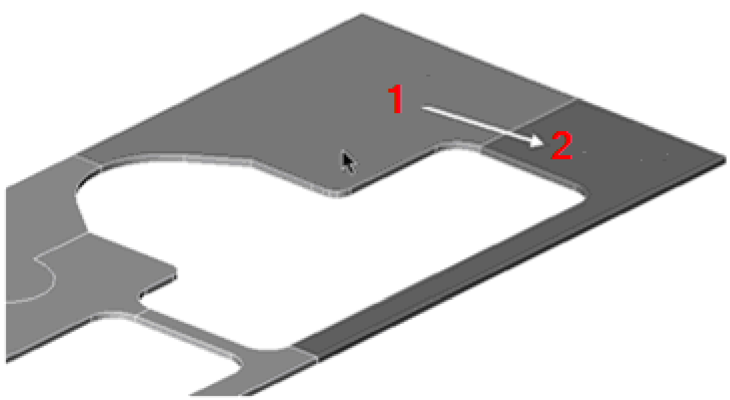
45) **Blank** or **erase**  the five datums.



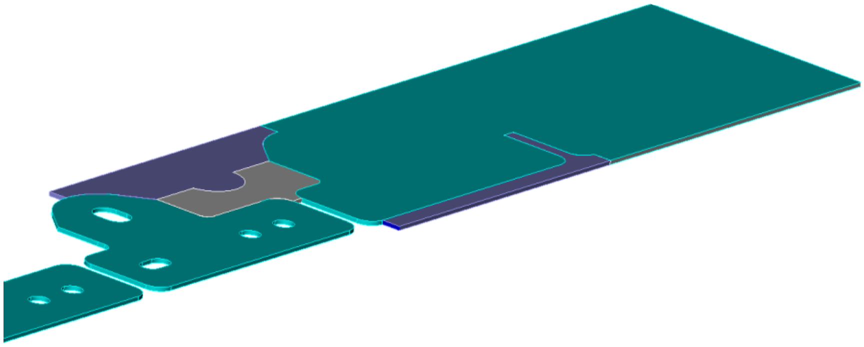
46) To tidy up the blank area of the strip, we should add the shapes shown below into one.

Use the **Combine Shape** Command and the **Add** Option. 

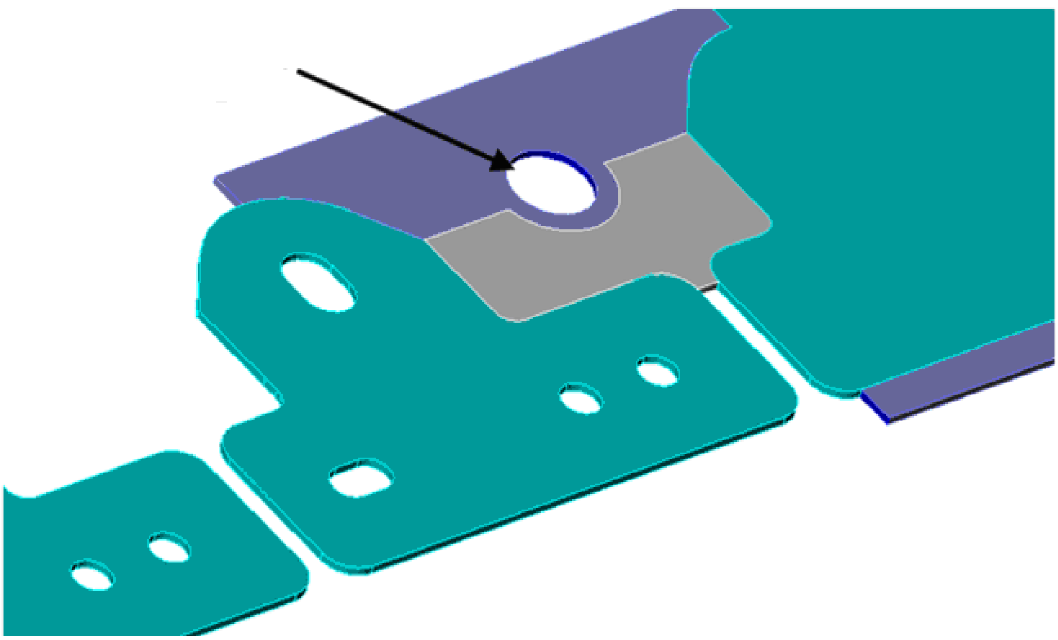
Add shape 1 to shape 2.



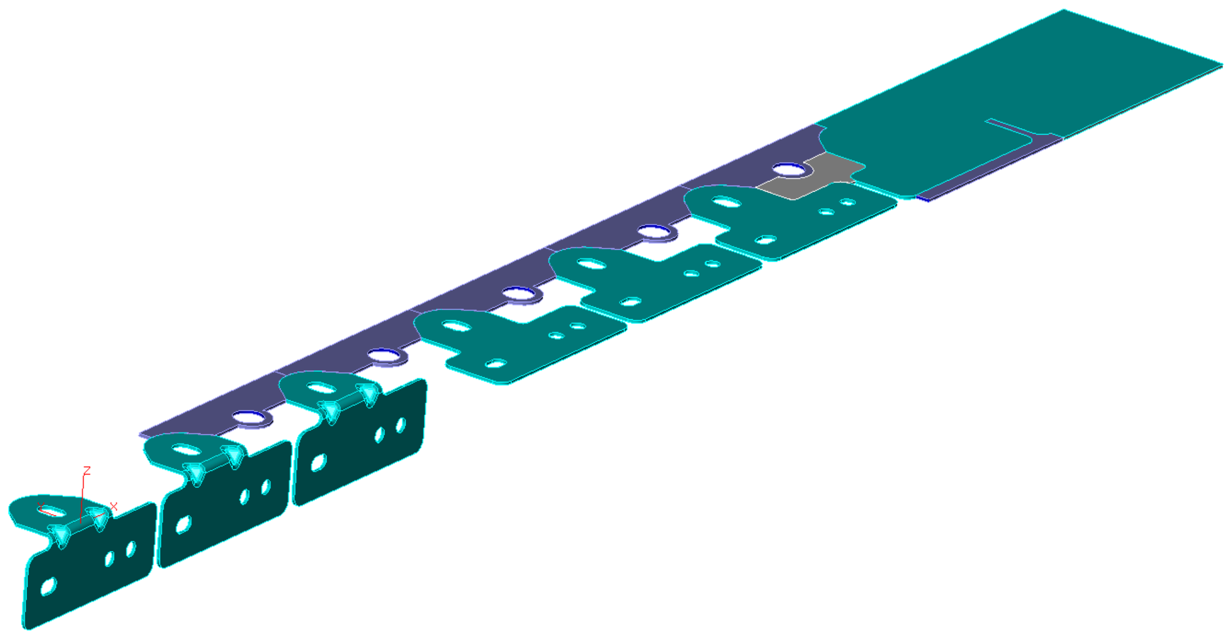
47) Continue to add shapes until you have the results below. You can join more than two at a time.



48) To finish the Blank area of the strip we need to put a Pilot Hole in the strip. Put a 1/2" Diameter Pilot hole in the center of the radius as shown below.



49) Make 4 Copies the Carrier shape that contains the hole along the Negative X axis by an increment of Pitch to complete the strip layout.



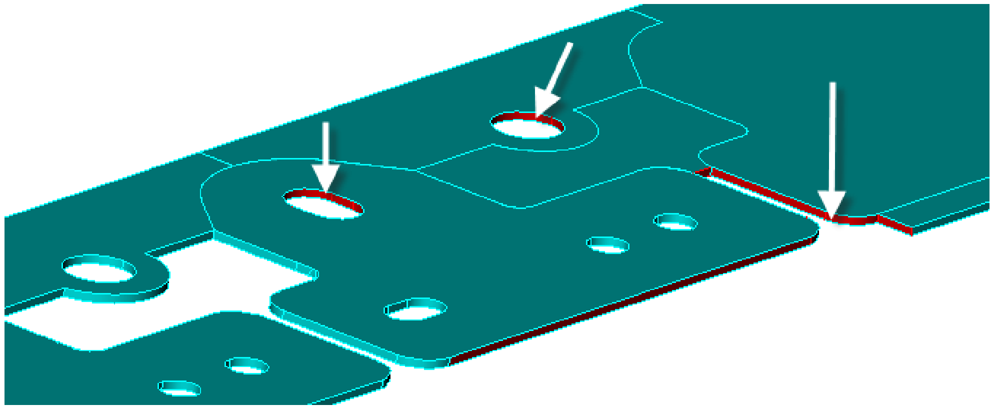
50) To test the integrity of the associativity of the strip, change the **Pitch** variable value to 3.75 inches. Use the **Redefine** option when you right-click on the Pitch command in the history.

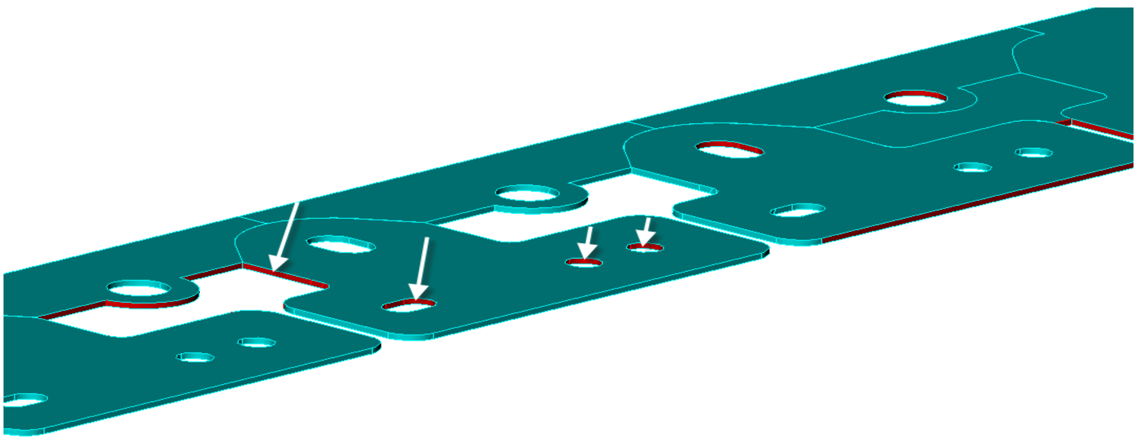
Notice the spacing has increased.

51) Then change Pitch back to 3.5 inches.

52) The final step of the strip development is to identify the thickness faces where punching action will be occurring. This is a helpful visual cue, but this is certainly optional. Set the thickness face to a color that you have not been using that stands out. I used Red.

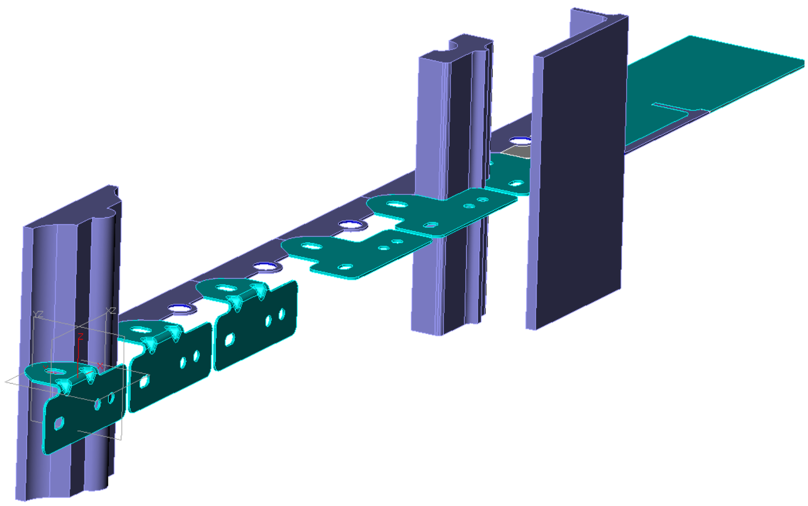
HINT: Most of the holes have two faces; the slots each have four.



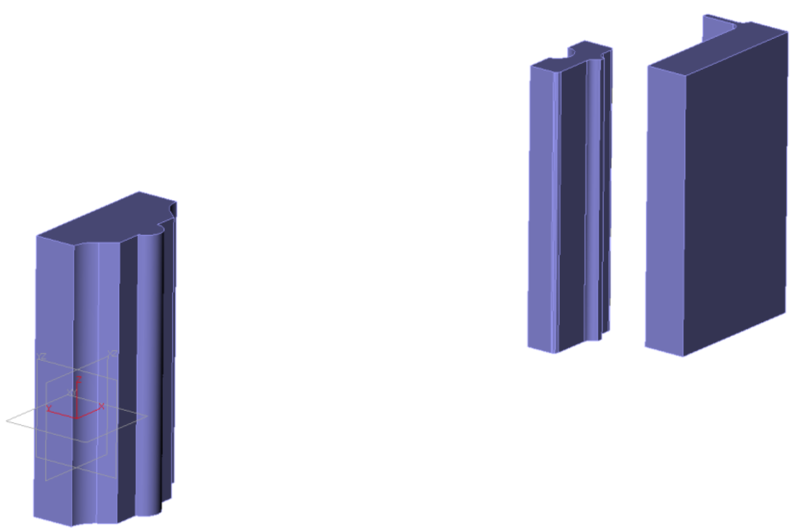


Lesson 3 – Master Punch Reference Geometry

In this lesson we will create reference geometry that will be used in the creation of our punches and dies. We will not create any reference geometry where a purchased punch would be used. This reference geometry will be associative to the strip.



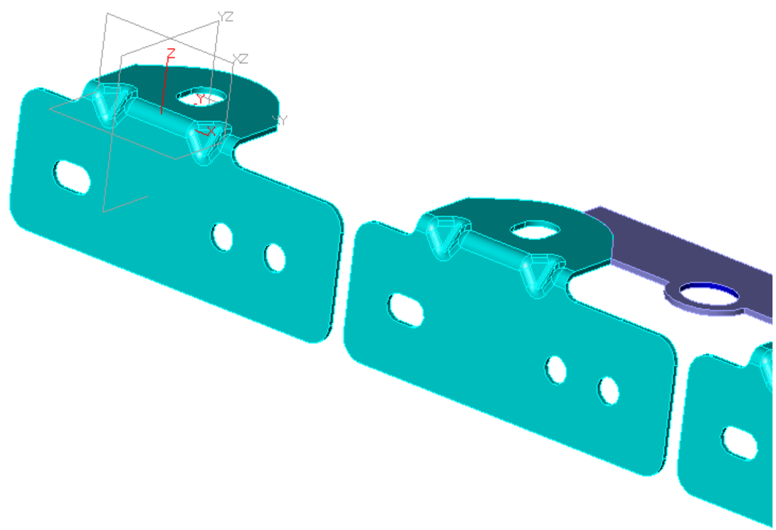
To give you a conceptual overview, we will insert the strip as a component then copy in only the faces we will need for creating the master punch geometry. We will create the punches by offsetting the face we copied from the strip. If required, we will then position them in the strip by moving them the distance of the Pitch variable, we established in the strip object.



Create a new object in the **Job_5361.Z3** file called **003_Ref_Punch_Master_5361**.

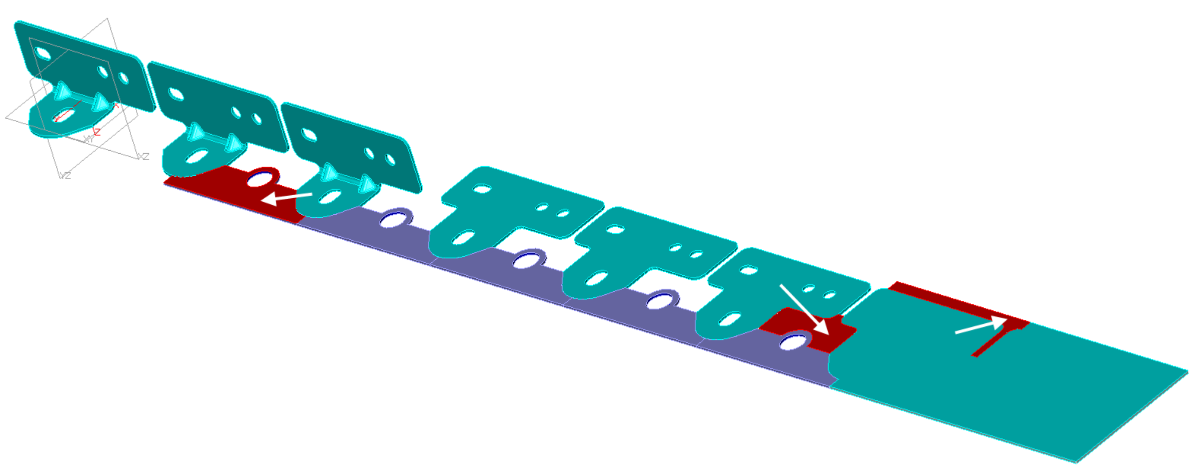
1) Insert Component –> **002_Strip_5361**.Anchor it.The insertion point is 0,0,0.

2) You should now see your strip component with the same relative position to Zero as it had when it was only a part.



3) Using the **EDIT** pull-down menu **–> Copy –> Assembly Geometry** command, we will copy select faces from the strip where a custom punch and die will be required.Set your filter to face.

To identify the faces to be copied from the strip.



After completing the copy command, **Blank** the strip component (you may have to set your filter to all or component to be able to select the strip).

4) We need to move the faces into the position where the actual punching action will take place. Move all of the faces in the Negative X direction a distance of Pitch. The pitch variable was not established in this part object, but rather in the strip object, I will show you how to get it from the strip in these next few instructions.

   Choose **Edit >Copy>Copy Entities,** and pick the **Copy along a direction** from the pull down menus**.**
   Select the three faces.
   Direction: **–X axis.**
   Distance: **Right Click** and choose **Expression.**
   Pick **Select Variable** from the Input Expression window. You will see the variable browser window popup. Choose the **<========** as shown above.
   Then Choose the **002_Strip_5361** then the **Pitch variable.**
   Your input expressions window should then look like the following [002_Strip_5361:Pitch].
   Choose OK on the Input Expressions popup window.
   Choose Cancel on the Variable Browser Window .
   Choose OK to complete the MOVE Command.
   NOTE: the variable browser allows you to navigate to other files and objects for variable input into this part or object. This is one of the reasons that object names should remain unique.

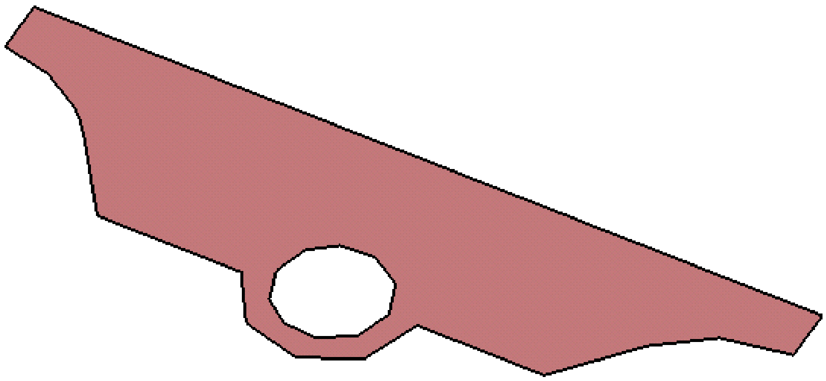
5) **Unblank** all entities and you will see that the faces we copied from the strip are now in punch position.

6) Undo the **Unblank**.

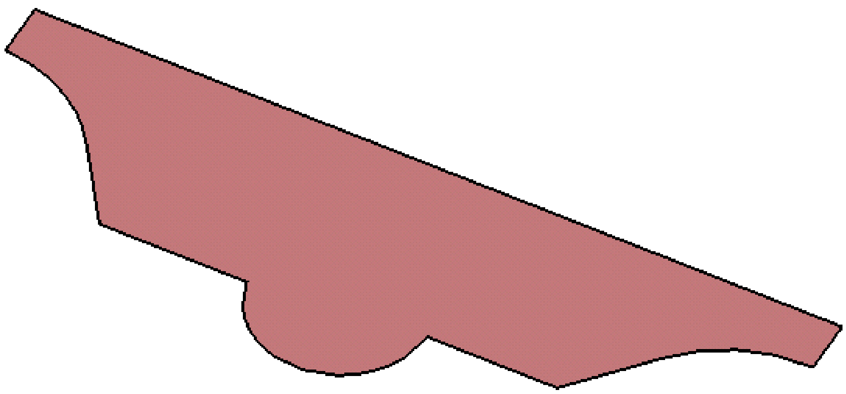
The last punch which will cut off the final part in our strip, does not need the pilot hole face.

7) Pick the **Delete Loop** command  from the **Free Form Tab**.

Select the Face and set the Loop option to Inner as shown in the image below. Then pick OK.



8) Your results should be as follows…



We need to thicken all of these faces so that they could be modified to be a punch and so that they could be used as tool bodies against a die block.

9) Choose the **Shell** Command from the Shape Tab. 

Shelling a face will thicken it.

Middle click for the default selection

Specify an offset of -2.875 inches (This will put the tops at the typical Punch height if using a 1/8" engagement when the die is closed)

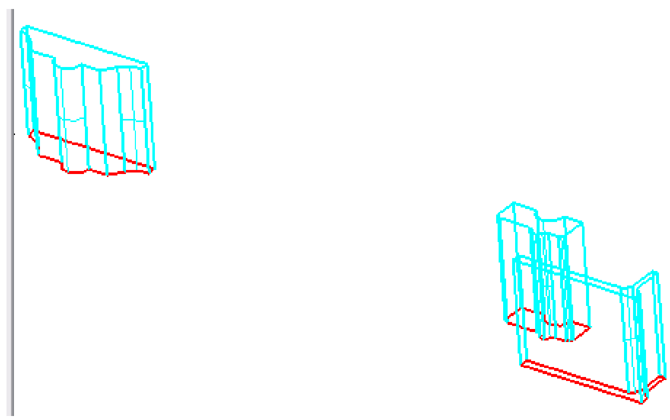
Do NOT specify an open face

Be sure that Create Side faces is checked

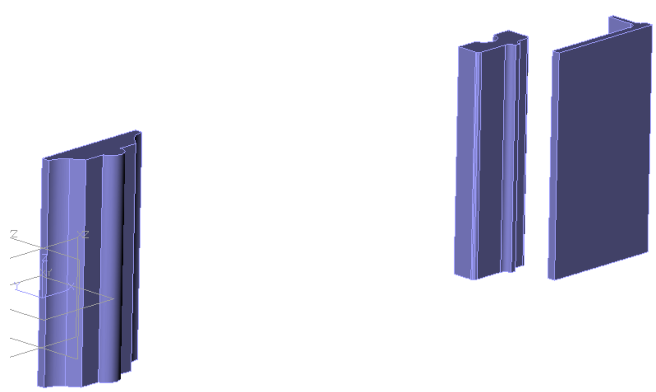
Results in the following Image…

10)Choose the **Offset Face** command  and pick the bottom faces of these shapes (see image below for face selection), so that these shapes thicken with material added in the negative Z direction.

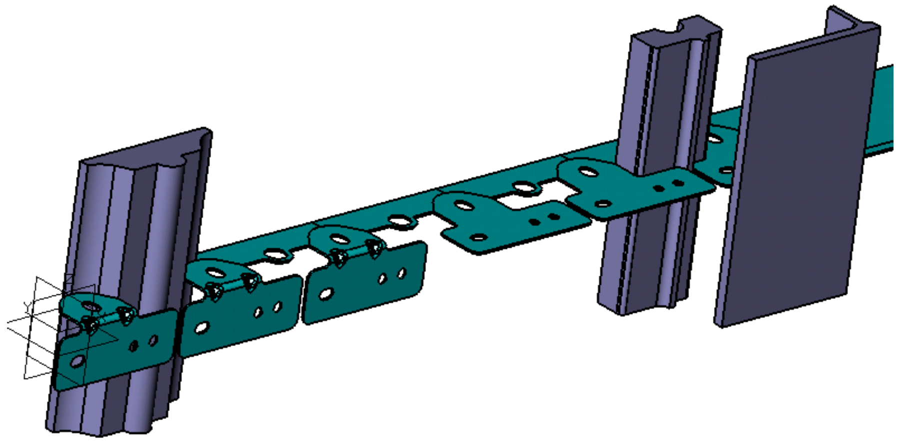
Fill in the Offiset T :3



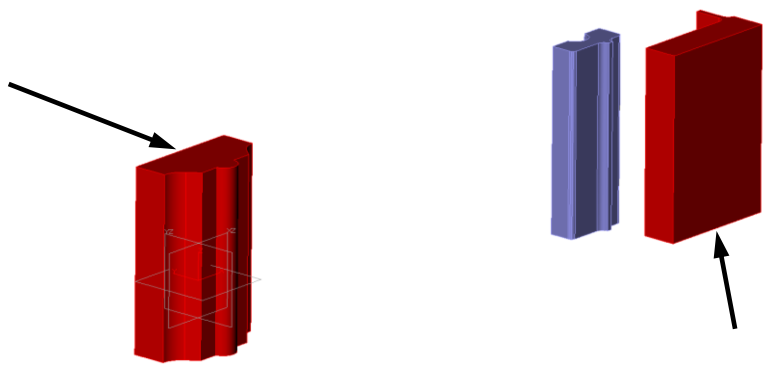
Results …



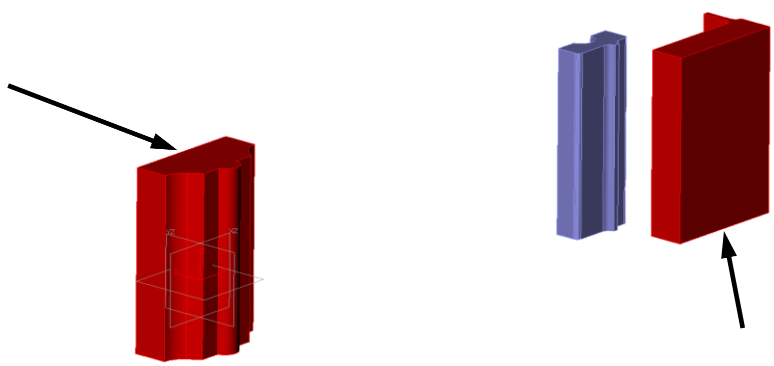
11)**Unblank** all entities and you will see the reference punch shapes still in punch position.



12) **Erase** the **002_Strip_5361** component. We no longer need to reference the strip.



13) Now, we will offset the flat faces of the shape for the Punch notch and the final cutoff punch by ¾ inches.

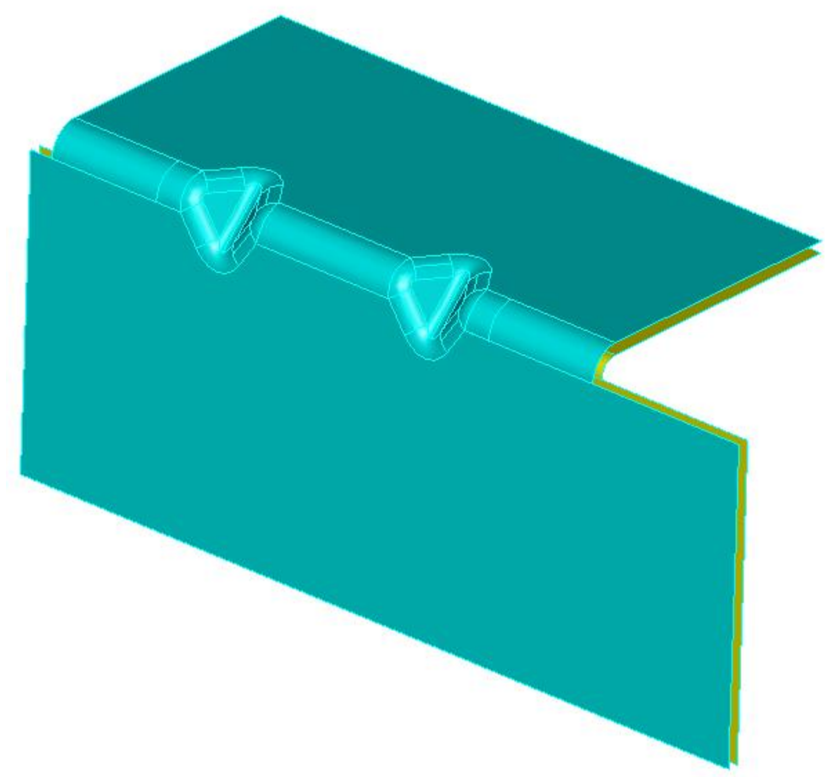


These shapes will be used in the creation of the punches as well as the Die Blocks that each custom punch engages.

Lesson 4 – Master Form Trim Surfaces

In this lesson we will create reference geometry that will be used in the creation forming blocks. Again, this is master reference geometry that will be used to facilitate the creation of an associative form block.

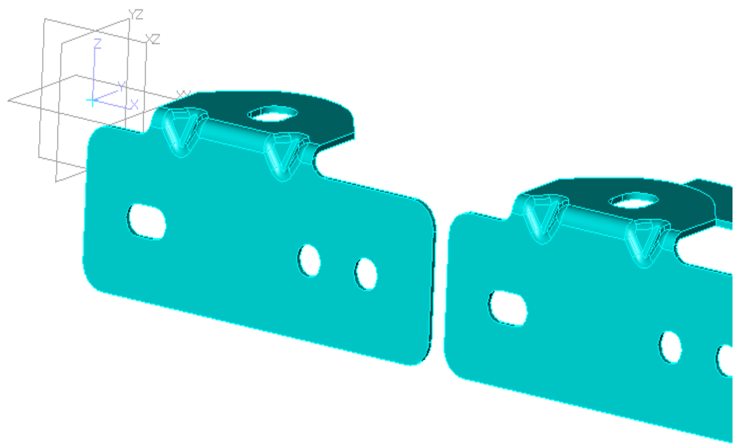
To give you a conceptual overview of this lesson we will insert the strip as a component then copy in only the shapes/faces we will need for creating the master form trim surface geometry.   If required, we will then position them in the strip by moving them the distance of the Pitch variable, we established in the strip object.



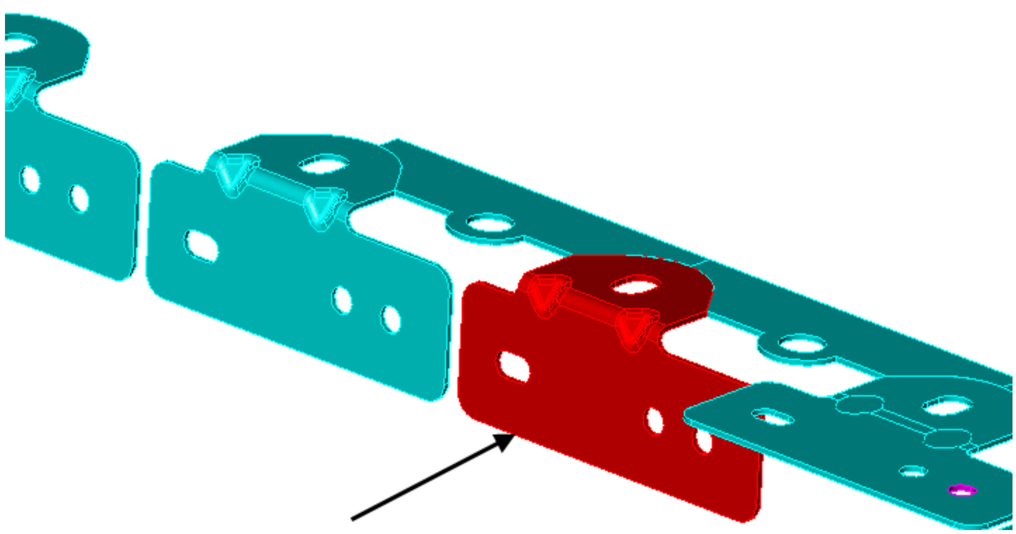
Create a new object in the **Job 5361.Z3** file called **003_Ref_Form_Master_5361**.

1) Insert Component –> 002_Strip_5361.
   - Anchor it.
   - The insertion point is 0,0,0.

2) You should now see your strip component with the same relative position to Zero as it had when it was only a part.

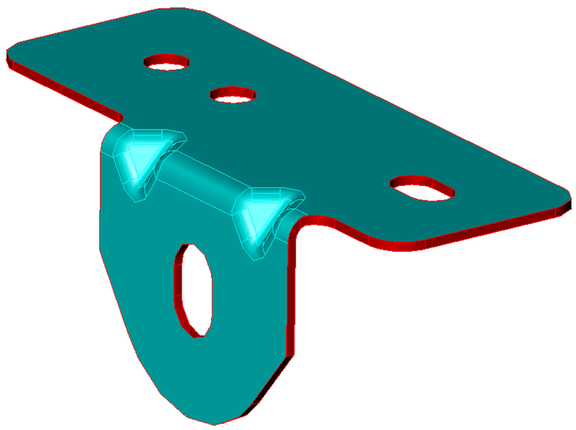


3) Using the **EDIT** pull-down menu**– Copy – Assembly Geometry** command, we will copy select Shape from the strip where forming occurs.
   - Set your filter to **Shape**
   - See the following image below to identify the Shape to be copied from the strip.

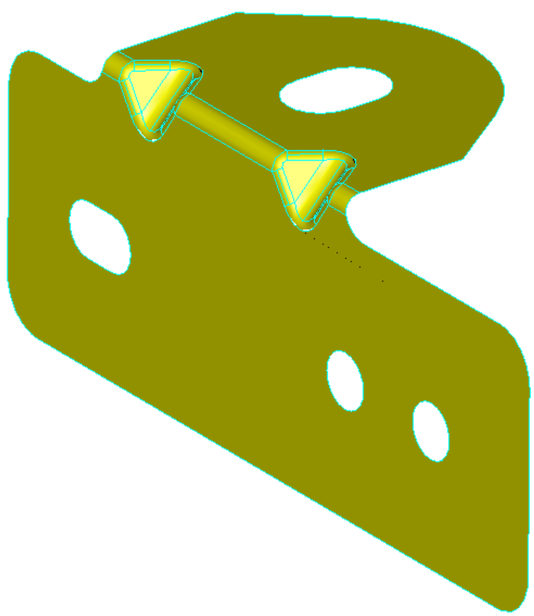


4) After completing the copy command, **erase** the strip component (you may have to set your filter to **all** or **component** to be able to select the strip).

5) To create the trim surfaces for the forming blocks we need to **erase** the thickness faces of the shape.



6) The result after erasing the thickness faces will be two independent shapes; one representing the bottom skin of the part, the other the top skin of the part.

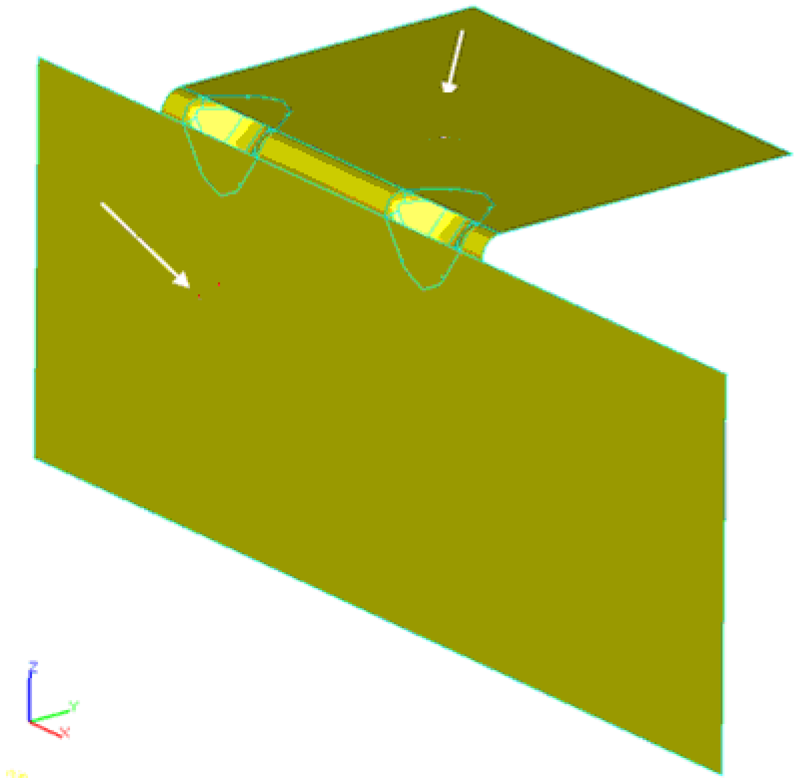


7) **Blank** the shape of the top skin.

8) We need to remove the openings in the faces of the surfaces because they are not needed in creating a form block. We will use the delete loop command to delete the

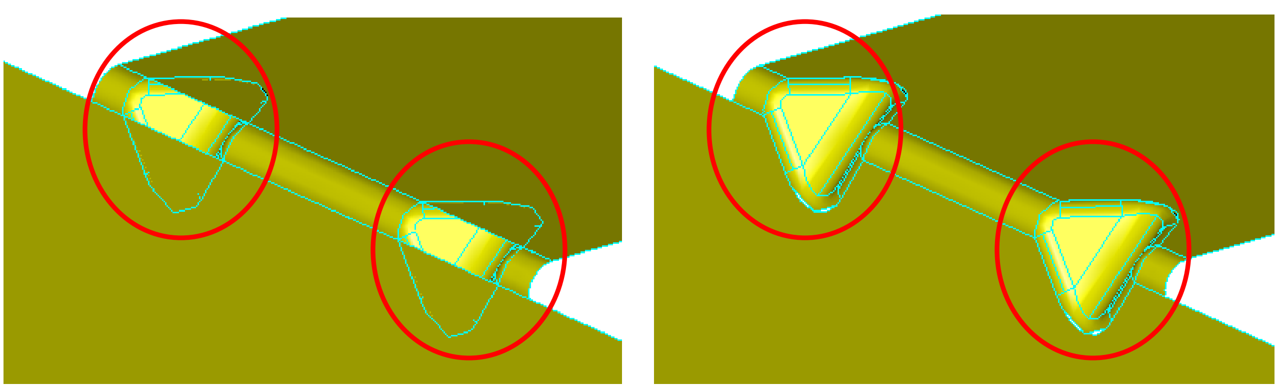
inner and outer loops and completely untrim the two faces with the slot and hole cutout in them. The **Delete Loop** command is found on the **Freeform Tab**.

9) Results…(delete loops as indicated in the figure below)



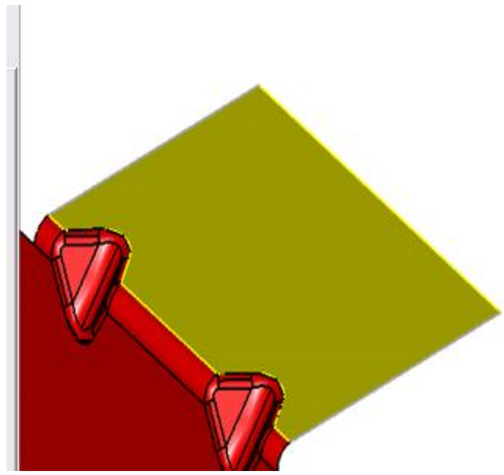
10) Next we need to retrim the surface back to the gussets or strengthening darts.

– Use the **Trim to Curves** Command found on the Freeform Tab. 

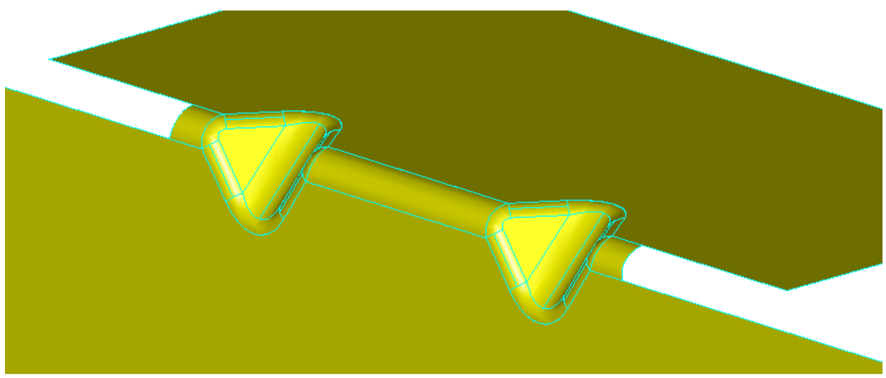


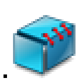
11)Let's extend the smaller top surface to make sure that our surfaces for trimming a form block are large enough to extend past the edges of that block.

- Use the Surface Extend  command from the Freeform Tab.
- Select the edges and fill in the Options Form as shown below.
- Type 0.5 for the Dislance.



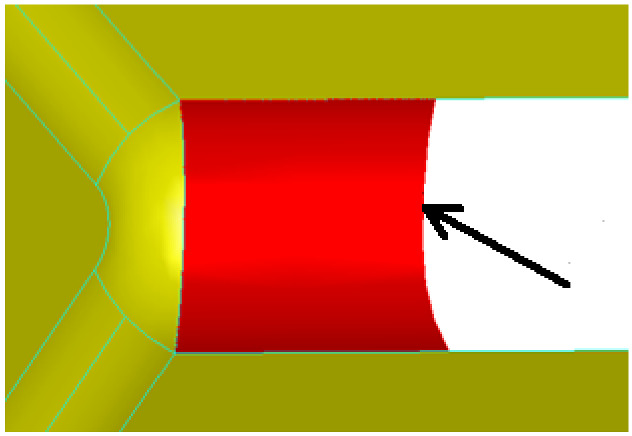
Results…



12)Many of the faces become unsewn in this process of deleting loops and extending face edges, so we should take the time to sew the part. Run the **Sew** Command from the Heal or Freeform Tab.

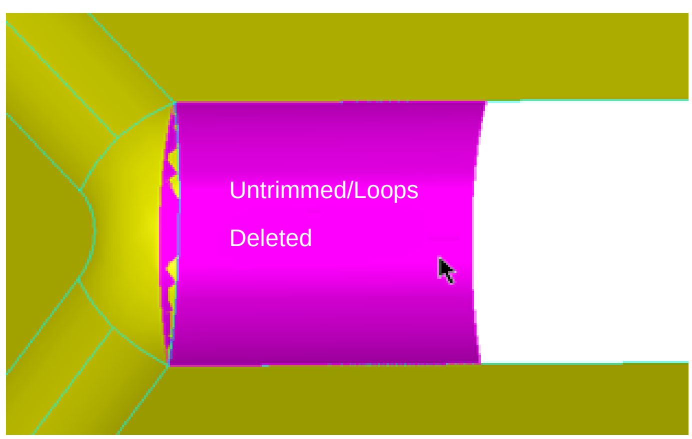
13)We can now fill in the missing fillet faces. If you look closely at the existing fillet you will see that it was trimmed. The exposed edge of the face (shown to the right) is actually made up of two edges – the edge is split – caused when the original model was filleted or bent. We should reduce the complexity of this face either by untrimming / deleting face loops or by concatenating edges.

The edge indicated below is split in two.



14) I typically try to simplify the face by deleting loops before using the edge concatenation, because the concatenate command can slightly modify the geometry within an angular tolerance.

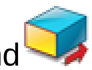
15) Use the Delete Loops command from the Freeform Tab to completely untrim the face shown to the right.
Results shown to the right…

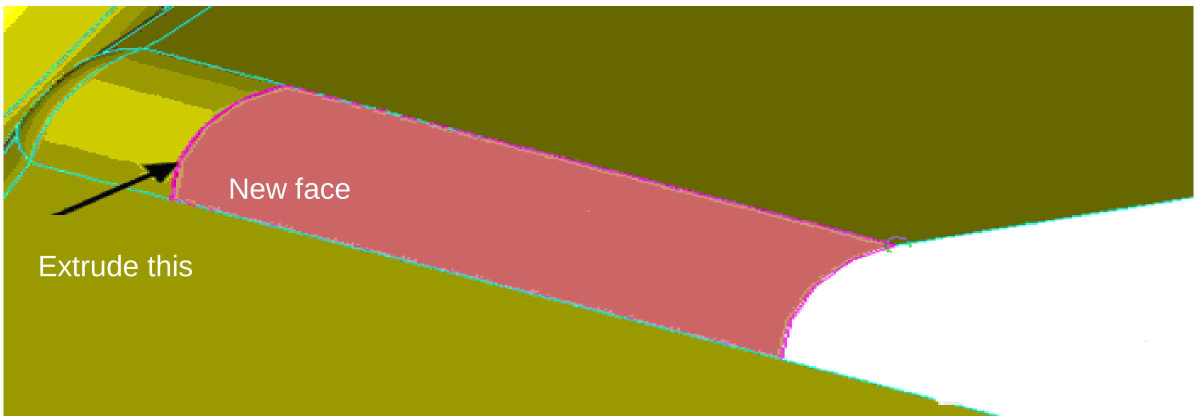


16) Next we need to retrim the surface back to the gussets or strengthening darts.

- Use the **Trim to Curves** Command found on the Freeform Tab. 

17) Run the Sew Command again to sew the face back in. 

18) Use the **extrude** command  with the **add**  option to extrude the edge indicated below. This will create the new face shown below.



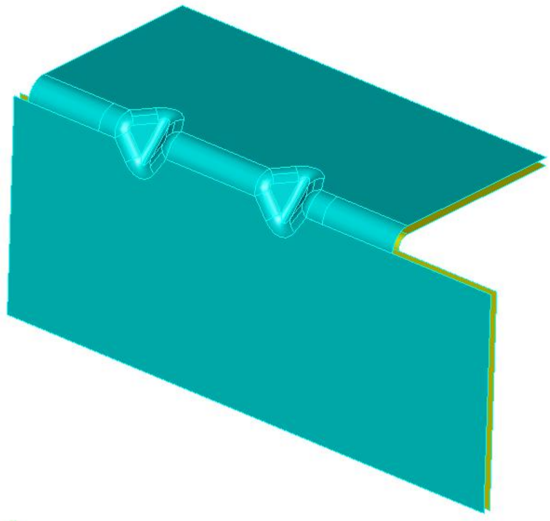
19) Repeat steps 14-20 for the other fillet on this skin of the part.

20) **Sew** any all faces to close any unsewn edges.

21) Remember we blanked the outer skin of surfaces for this part.

22) We need to swap our blanked entities with our visible ones, so REPEAT this lab for the outer skin.

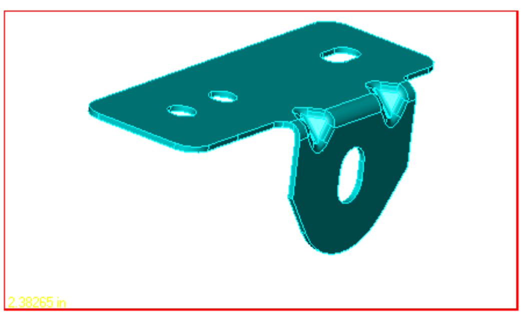
23) The final result will look like the following…

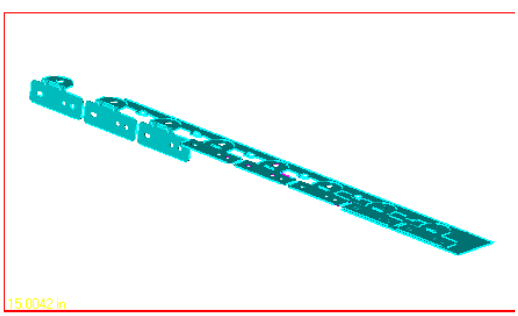


Summary

Your ZW3D Object Manager should look like the following images…

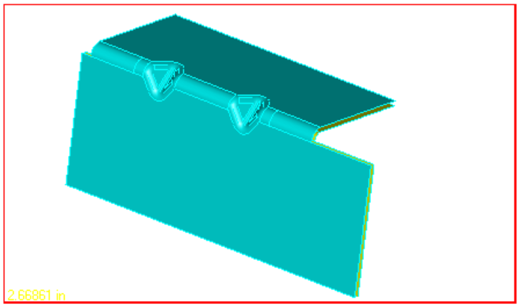
(Each image on the left shows the graphics preview of the objects as they are listed in the object manager)

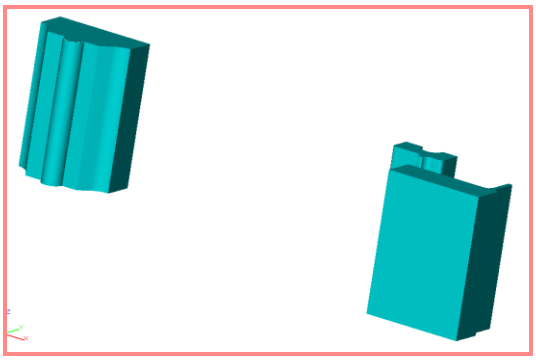




In the ZW3D Object Manager , you can see these messages look like the following table:

001_Customer_Part_5361
002_Strip_5361
003_Ref_Form_Master_5361
003_Ref_Punch_Master_5361





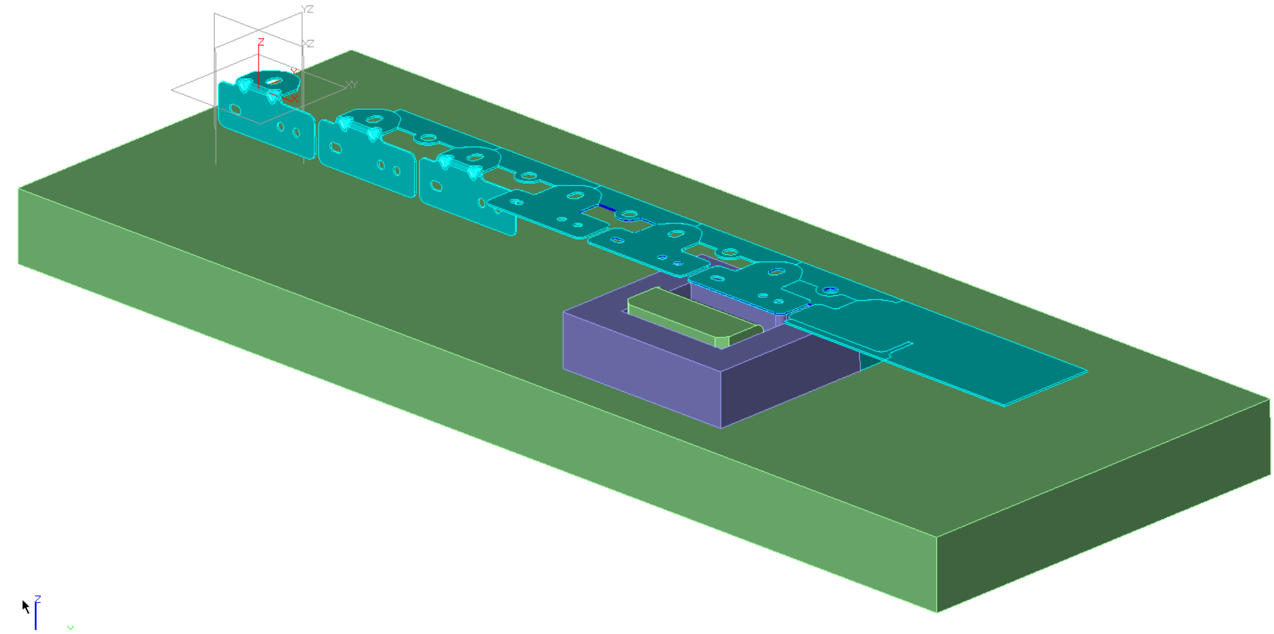
Lesson 5 - Creating the Die Subassembly

in the context of the Main Assembly

(Alternate to Appendix C)

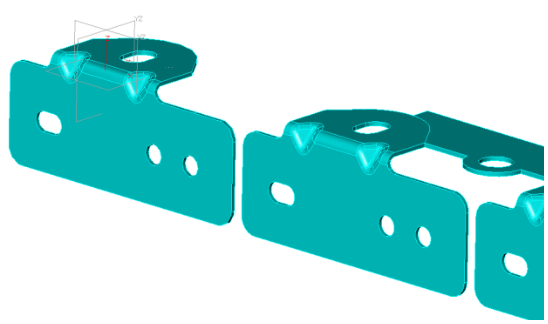
This lab will focus on the creation of the Die Subassembly within the context of the main assembly. We will be implementing a method where we are not concerned about individual parts until the end of the design. Rather than creating parts we will create one part with many shapes. At the end each shape will become its own part.

In this lab you will create a single part object that will have shapes that will eventually make up the die subassembly. We will not create a complete subassembly, but we will give you a foundation to work from. The actual Die Subassembly object will not be created until we create the individual part objects.



Create a new object in the **Job_5361.Z3** file called **010_Main_Assembly_5361**.

1) **Insert Component** –> 002_Strip_5361.Anchor it.The insertion point is 0,0,0.



2) You should now see your strip component with the same relative position to Zero as it had when it was only a part.

3) We will create the Die Reference Object.

- Right Click in the graphics area (be sure you have not selected anything) and choose **Insert Component**
- The prompt asks for a name for a new object.
- Type in **100_Die_SubAsm_5361** and hit <ENTER>
- Check the **Anchor Component** Box,
- then Specify 0,0,0 (comma comma) for the location and hit <ENTER>

4) If you look at the title bar, you will see that ZW3D has placed you in the new object that you created in the previous step…

5) You can still see the strip from the main assembly. Pick the **show target** button  from the Document Awared Toolbar to see what is only contained in the Die_SubAsm part. (3 planes)

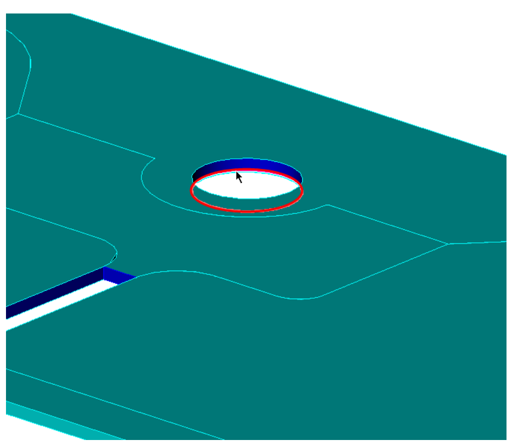
6) Pick Show All  to see the higher level assembly components.

NOTE: The Die_SubAsm object will contain all of the shapes (solids/surfaces) of the die steel, but will not be a true assembly; it will be only one part object.

7. Next we will create a Die Block for the Pilot Hole.

REMEMBER: We positioned the bottom of our strip on the XY Datum, so the XY datum and any reference geometry from the bottom of the strip can be used to establish the top of our die steel.

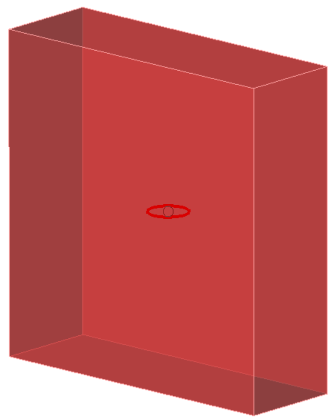
8. Choose EDIT-COPY-Assembly Geometry.
- Choose the bottom edge of the first Pilot hole and Hit <ENTER>. See image below.
- **Remember** set your filter to edge to make your selection.
- **NOTE:** If it does not highlight, turn off shading.



9. **Show Target** . You will see a circle.

10. Create a Primitive Block as a base shape with the Block center at the center of the circle and the second point can be anywhere. Then specify the following in the block options form…

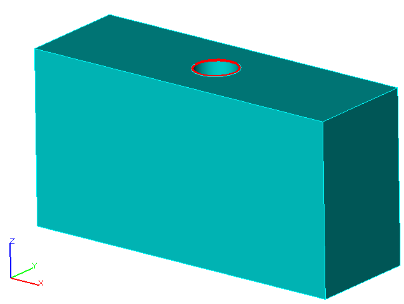
- X = 3.0
- Y = 1.0
- Z = -3 (our die steel will be 1.5 tall, so with a primitive block by center and corner -3 will put 1.5" below our XY Datum.



11. Next trim the new block with the XY Plane.
12. Use the curve that we copied from the strip to extrude remove a hole in the block with an offset of your desired clearance. In this case I will use a .010 inch clearance.

Your final block should look like the following image…

You can erase the red circle.



14. The next shape we will make is the die block for the punch notch.

15. The creation of this shape part will be similar to the last one, but we will use the master punch reference object we created earlier to establish the cut in the block.

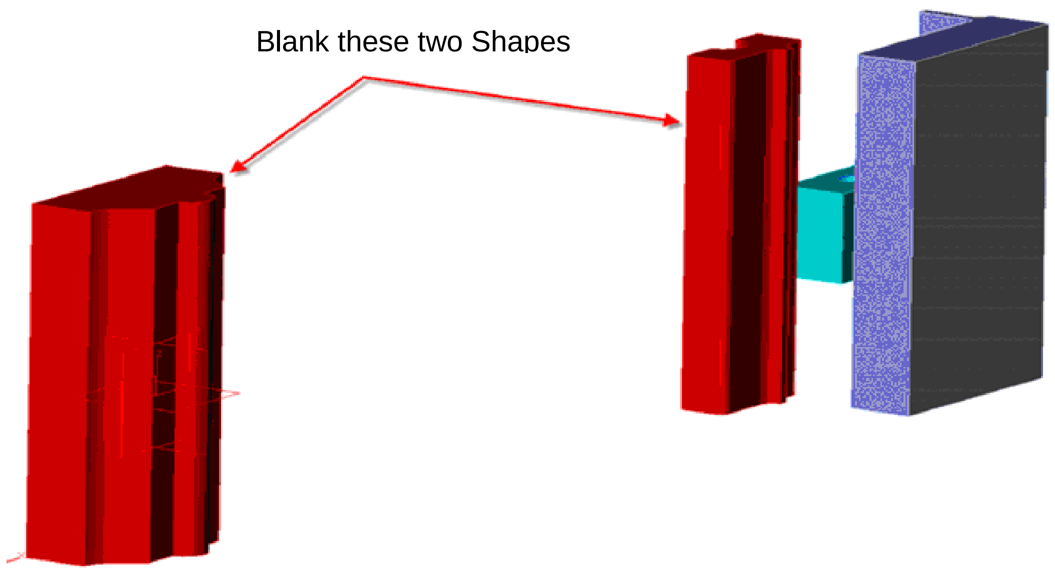
16. **Show Target**. You will see the previous block you created.

17. Choose EDIT -> COPY -> External Part from the Pull down menus and choose .

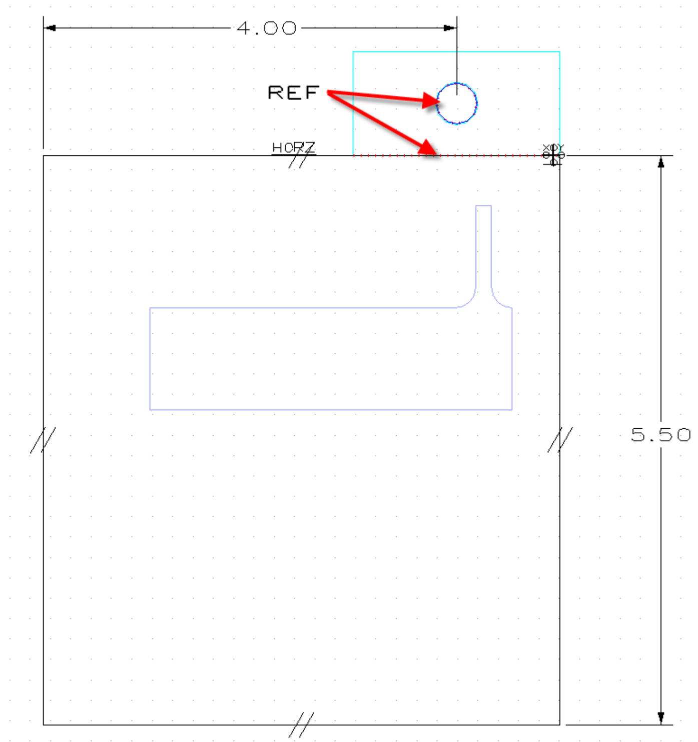
    Choose **003_Ref_Punch_Master** from the object list.

Type in 0,0,0 for the location.

See the image below for your results.



18. **Blank** the two shapes indicated in the image above.
19. Create a sketch on the XY datum that looks like the following image. I used 2 Point Rectangle to draw the rectangle avoiding references to the Zero Point. The Ref note on the image to help show which elements were referenced into the sketch from the Model (the Pilot Hole Curve and the bottom edge of the previous block.



20. We locate the upper right corner of this rectangle at the end of the horizontal reference line.

21. Extrude the sketch you just created as a base shape. 

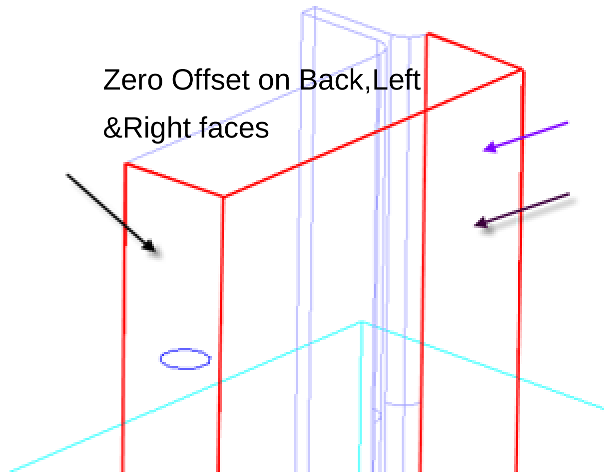
Start is 0

End -1.5

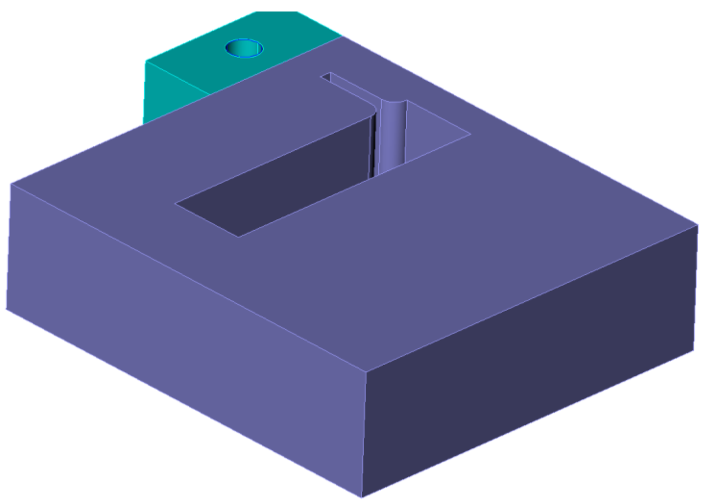
22. Use the **volume offset** command  found on the shape Tab to increase the size of the remaining shape from the reference master punch by the clearance needed to cut the die block. Use .01, so that you can see the change.

- Select the punch shape
- Specify a .01 offset
- Be sure to uncheck the Keep Original Option and do not specify stationary.
- Then choose the Advanced Tab and pick the **Add Offset** button.
- Specify a Zero Offset on the Back, Left and Right Faces indicated in the image below. The command only allows you to pick one face at a time, so you will have to Add Offset

3 times. Alternately you could volume offset all faces then offset those that should have remained at zero -.010.



23. You should see the master punch take on the default color and increase in size slightly.
24. **Combine Shape > Remove**. The next step will be to remove the Ref Punch Master shape from the Extruded block we just created and your final result will look like the following image…

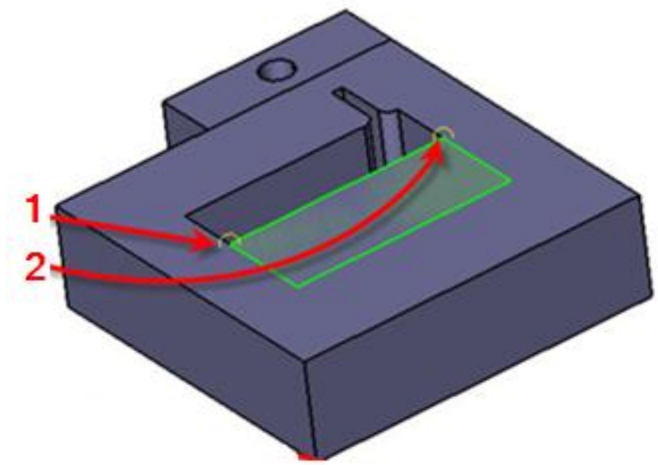


25. Let's create the Heel Insert shape and use it to remove additional material from the block.

Create a Primitive Block.

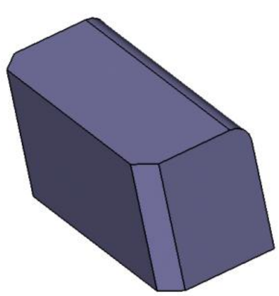
- Base Shape
- Opposite Corners
- Pick the First corner and Second corner as shown in the following image…

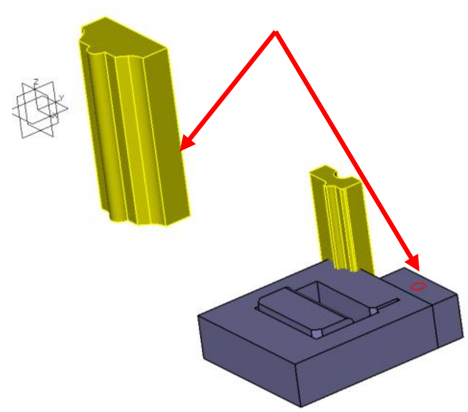
- Y length =-1.5
- Y length =-1.5
- For the **X length**, right-click "**Expression**" and pick the **Variable** icon browse for the Pitch variable in the Strip.



1: 1st point   2: 2nd Point

26. Offset the top face of your new Heel Insert block by +.25 inches.
27. The next step will be to remove  the Heel Insert shape from the die block for the pitch notch, but check the option to **Keep Removed Shapes.**





28. You will now have the proper opening in the die block.
29. You can finish modeling the Heel Insert by adding the fillets and chamfers to the block.

30. Unblank.

31. We will not create any more die blocks, so we will erase the two shapes from the Ref Punch master that are remaining. If you were to continue to layout die steel you would want to take these into consideration.

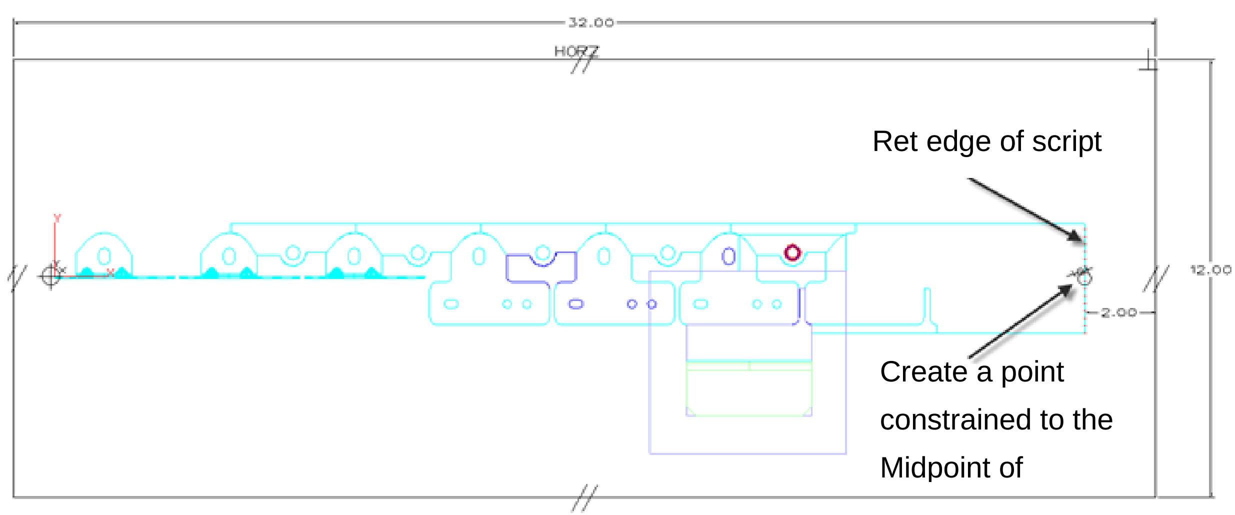
The last Block we will create is the Die Plate.

32. Show All.

33. Create a Datum parallel to XY that is 1.5 inches in the negative Z direction

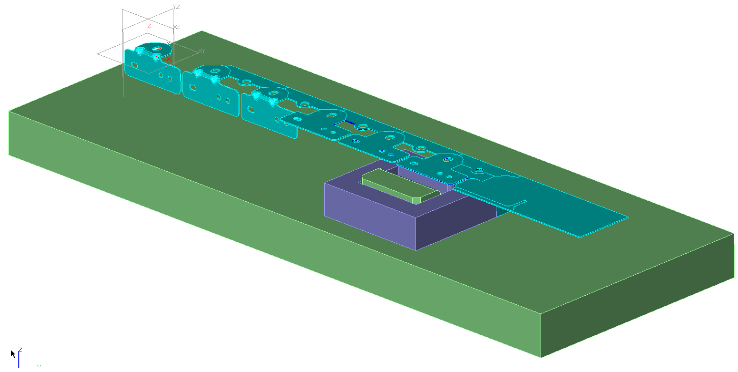
34. Create a sketch of a rectangle on this new datum that is centered on the strip. This sketch will be the perimeter of the Main Die Plate. The image below contains the sketch you should create.

- Constrain a Point to the midpoint of the vertical reference line.
- Midpoint Constrain the right hand vertical object line to the point. Be sure to only constrain on the Y Axis.
- Change dimensions as shown below.



35. Extrude (Base option) the rectangle 2 inches in the negative Z direction.

36. Your Die Subassembly should appear as follows. (You may need to navigate back to parent object or pick the Show All button to see the entire subassembly.



37. When you have finished your preliminary design and it has been approved you can quickly create a part from each shape for individual detailing and put these parts back together in a true assembly. The shapes that we have here cannot be put on a BOM, so an assembly would need to be made.

38. The command for pushing shapes out to parts is Assembly Tab -> **Extract shape.**

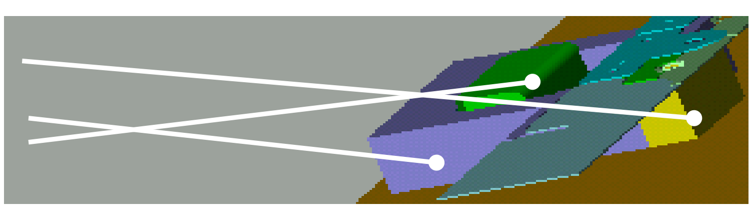
39. Before you pick this command you will want to name the shapes using the **Attach Name Tags** Command found on the Attributes pulldown menu.

- **Attributes -> Name -> Attach Name Tag**
- You will be prompted to select an entity and give it a name. You will need to name the Shapes.**Important**: Set your filter for this command to Shape.

40. **Name the Plate – 101_Die_Plate_5361.**

41. Repeat this for the two die blocks and the punch retainer we just created. Name them as follows…

- 102_Die_Block_5361
- 103_Die_Block_5361
- 104_Punch_Retainer_5361



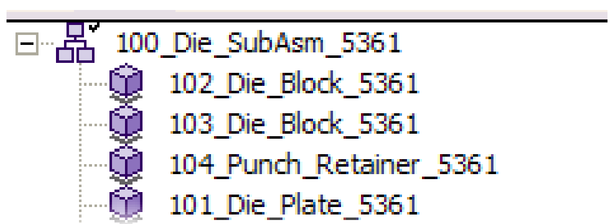
42. Now we can run the **Extract Shapes** command referred to in Step 40 above.

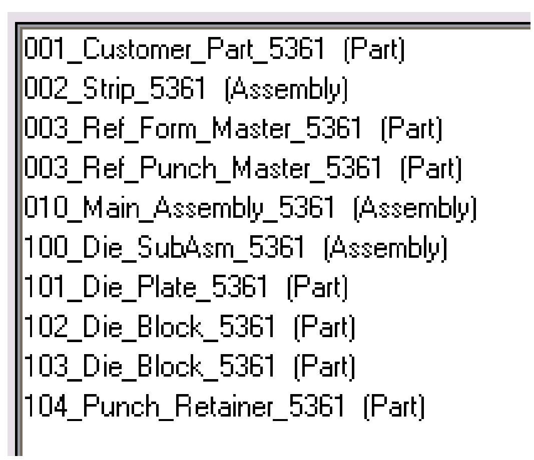
- You will be prompted to select shapes (simply middle click for all)
- Notice that a seed part/template is automatically specified if you have this set in your configuration. This is useful for establishing Part Attribute defaults and default units for these new parts.
- Answer YES, when prompted to extract shapes that are not imported from components.

NOTE:

i. Future History Regens will prompt you to overwrite existing objects.
ii. Any edits to these parts should be made in this assembly of shapes prior to the Extract in the History Tree.
iii. You can easily create spaces between these blocks by moving or offsetting faces on these blocks after creation.

43. If you look at your assembly manager, you will see that your Part is now a subassembly containing the parts (named shapes) you labeled in Steps 42-43.



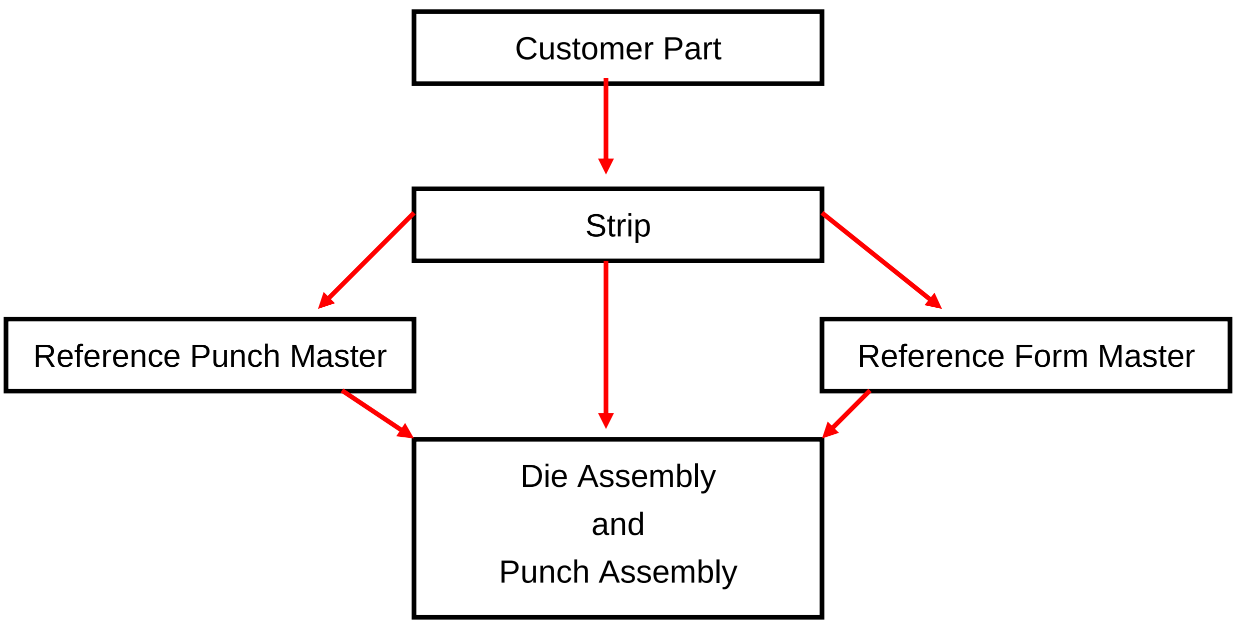


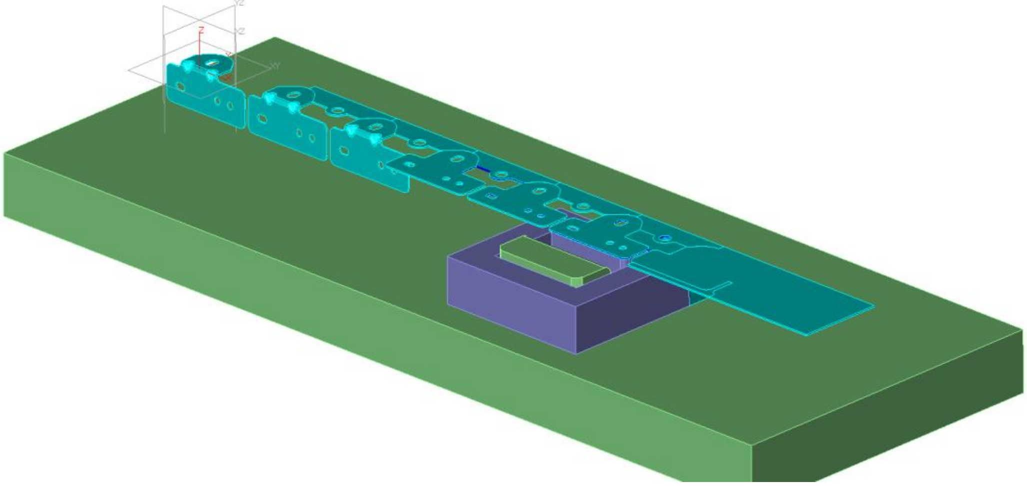
Lesson 6 - Test the Associativity and Establish Automated Updates

This lab will examine the associativity/dependency between objects and establish automated updates when changes are made to the strip. We will also test our design to make certain that the individual components will respond to various changes.

Let’s Review the dependency structure.

The customer part drives the strip which drives the Reference Masters and the Die Assembly components. The Reference Masters may also be used to drive the individual die block details.





1. Let’s start by making a small change to the pitch variable in the Strip.

2. Navigate to the Main Assembly.
From the Object Manager Edit the object 010_Main_Assembly_5361

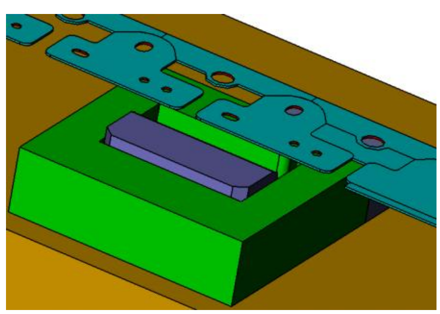
3. Open the **Assembly Manager**.

4. Right-Click on the 02 Strip 5361.1 and choose **Edit Part**. See the Title Bar.

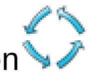
5. Open the History Manager.

6. Right-Click on the Pitch.1 Variable and choose **Open/Close**. See the image to the right.

7. Edit the Value of the Pitch from 3.5.in to 3.6.in



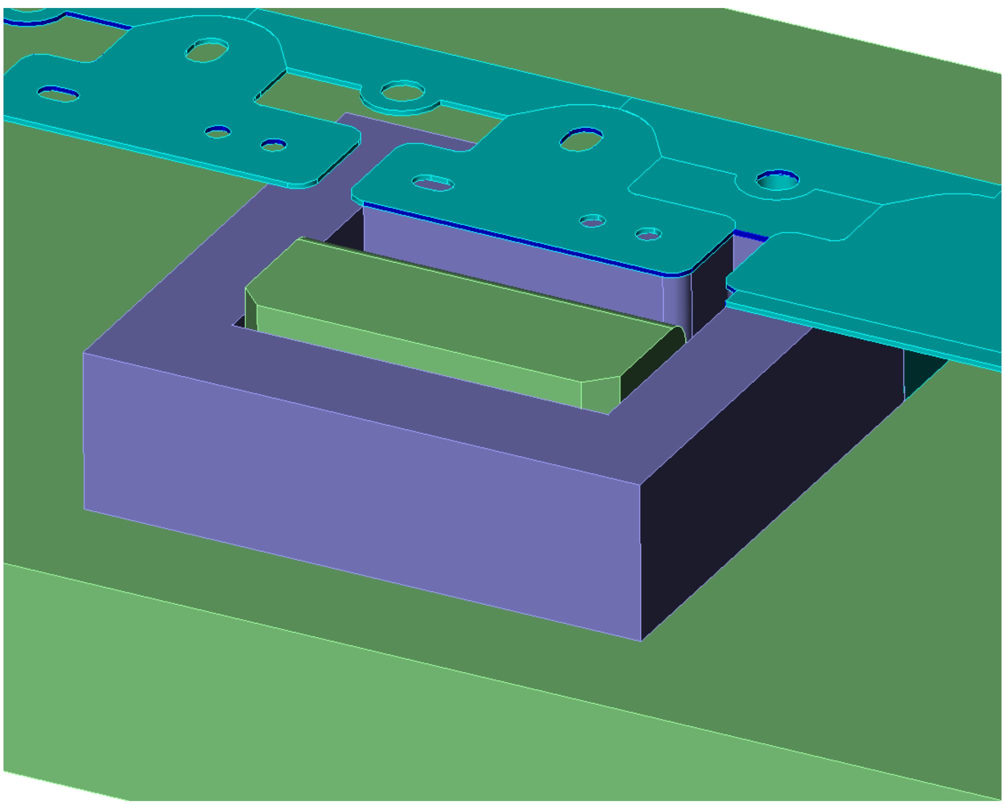
8. Your strip should **regen**, but the other parts in the assembly will not update automatically.

9. Go up to the Object Manager and Choose the **Regen** Button , then pick the **Ref Form, Ref Punch** and **Die SubAsm** and hit <ENTER> or middle click.

10. The status line's message window will report any errors in the regen. Optionally, you can edit each object and pick the **Regen** Button.

11. **Edit** the Main Assembly

12. Your results should look similar to the image below.



13. If all updated correctly, you would now want to establish an automated link between the strip and the objects it drives.

14. Navigate from the Main Assembly and Edit the Strip.

15. From the Pull down menus pick **Tools –> Link Manager**.

16. From the Link Manager window choose the first **Insert button** from the upper right area of the form. You will see an objects list appear.

- Pick **003_Ref_Form_Master**
- Pick **003_Ref_Punch_Master**
- Pick **100_Die_SubAsm**

17. These objects will appear in the Link Manager as shown below…

* Job_5361.Z3,003_Ref_Form_Master_5361
* Job_5361.Z3,003_Ref_Punch_Master_5361
* Job_5361.Z3,100_Die_SubAsm_5361

18. Be sure to **check the Auto Regen box**. When this is checked the listed objects will regen every time that you pick the Regen button in the host part, which is the **Strip** in this case.

19. Dismiss the menu.

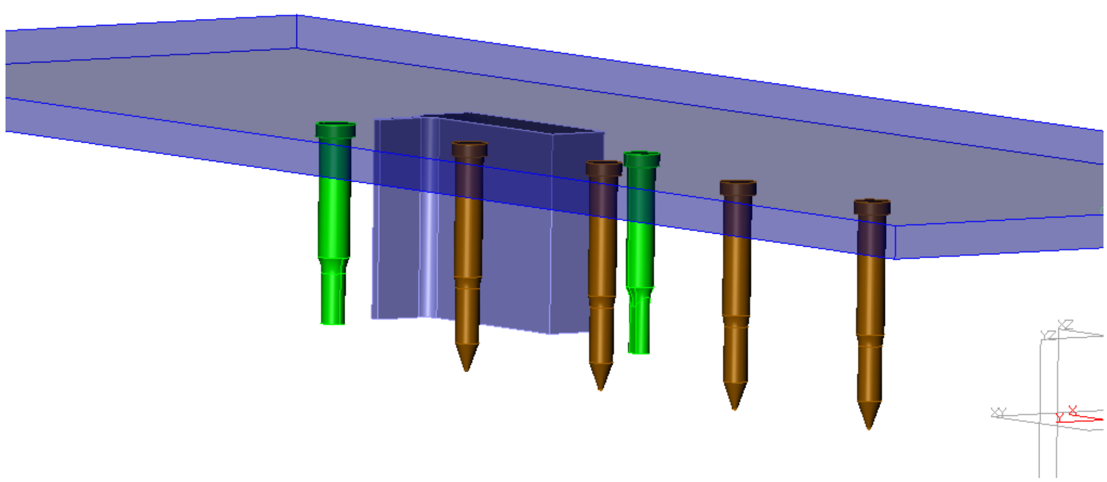
20. Now, change the **Pitch** variable in the strip back to **3.5.in**. Your strip should update along with all the parts that the strip drives.

Lesson 7 - Punch Subassembly

in the context of the Main Assembly

This lab will focus on the creation of the Punch Subassembly within the context of the main assembly. We will be implementing best practice methods for maintaining associativity to the strip and the reference masters that we created in the two previous lessons.

In this lab you will create the Punch Subassembly. We will use Standard Punch and Pilots as well as create a custom punch. We will not create a complete subassembly, but we will give you a firm foundation to work from. The Punch Subassembly is an object that will in itself not contain any sketches or shapes, but rather it will contain parts and perhaps other assemblies, much like the main assembly which will contain parts and subassemblies of parts.



We will start this lesson from the Main Assembly.

1)We will create the Punch Subassembly Object.

- Right Click in the graphics area (be sure you have not selected anything) and choose **Insert Component.**
- Type in **200_Punch_SubAsm_5361.**
- Check the **Anchor** Component Box.
- then Specify **0,0,0** (comma comma) for the location and hit OK.

If you look at the title bar, you will see that ZW3D has placed you in the new object that you created.

2) You can still see the main assembly. Pick the **show target** button from the Document Awared Toolbar to see what is only contained in the Punch Sub Assembly, which is nothing currently.

3) Pick **Show All** to see the higher level assembly components.

NOTE: The 200_Punch_SubAsm object, as the name indicates will only contain other components, not geometry, so we need to create other new parts in this assembly object.

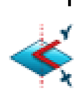
4) The first part we will create in the Punch SubAsm is the main Punch Plate
   - **Insert Component.**
   - Type the new name of the part – **201_Punch_Plate_5361.**
   - Choose the **Anchor** Component check box.
   - Type in the location as **0** and hit **OK.**

Again, if you look at the title bar, you will see that ZW3D has placed you in the 201_Punch_Plate object.

5) Make sure that the part preferences are set to your liking. For this exercise you will want to set your preferences as follows…
   - Choose **EDIT -> PREFERENCES.**
   - Set units to **inch.**
   - Set tolerance to **.0005.**

6) We will make the Punch Plate the same size as the Die Plate.

7) Choose **EDIT -> COPY -> Assembly Geometry.**
   - Set your **Filter to Face.**
   - An Pick the Top Face of the Die Plate.

8) We will then **create a datum** parallel to the absolute XY Datum at a distance of 2.875 inches in the Z direction. 

9) **Create one more datum** parallel to the previous datum at a distance of 0.5 in the Z direction.

10) Now **insert a sketch** on the face that you copied from the Die Plate. Be sure that the **reference face edges option is checked** on the insert sketch options box.

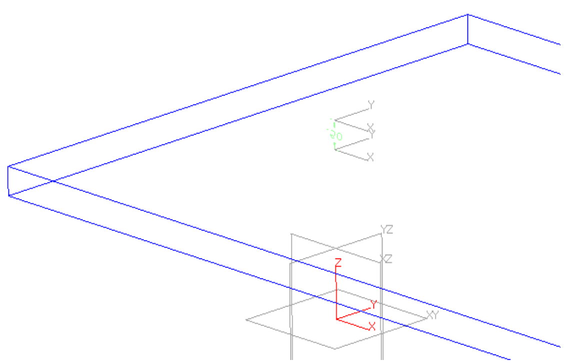
11) When in the sketch choose the **Trace profile** tool from the Sketch Tab and trace the rectangular outside reference edges.

12) **Exit the sketch** and **extrude** the sketch from the **Plane 1** Datum to the **Plane 2** Datum you just created.

   – NOTE: This allows for the dependencies to receive changes. The Datum 1 location is relative to the bottom of the strip and the datum 2 location is relative to datum one, so if the punch plate needs to be move you can edit the datum 1 offset and datum 2 will still maintain a distance of 1, so the actual thickness won't change. Using these datums allows us to sketch on the reference face which also improves the associativity between the sketch and the face.

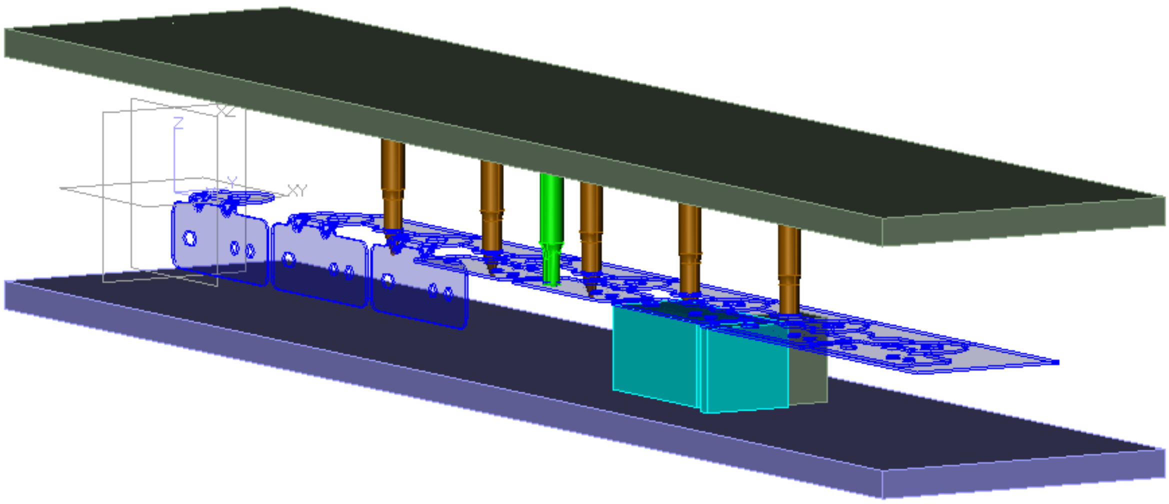
13) **Erase** the face we copied in from the assembly. You can optionally erase the 2 datums.

14) **Exit Part** until you are at the Object level.



Standard Punches and Pilots

Now that we have our Punch Plate established we can insert some standard parts like Pilots and Punches. Appendix A demonstrates the creation of these components. We will simply use a standard part from the existing library. Since these parts have an infinite number of options, we cannot create these from pre-determined design table, but we can work with a parametric part by copying it and making it to our specification.



Leave your current file open.

1. **Open the file** called "**standard-parts(ProgDie).Z3**" . 

We need a standard punch for the Pilot hole in our strip.

2. In the objects manager **right-click** the **Dayton Punch KJX (Part)** and pick **Copy**

3. Then choose the **Paste** button from the column of buttons on the right.

You will see a new object appear in your object list called, **Dayton Punch KJX1 (part)**

4. **Right-Click** the **Dayton Punch KJX1 (part)** and choose **Cut**.

5. Navigate to the object manager of your **Job_5361.Z3** file and choose the **Paste** button. You will see the object appear in the Object list along with the rest of the parts and assemblies we have been creating.

6. **Right-Click** the **Dayton Punch KJX1** and **Rename** it to**:**

Dayton Punch KJX_5361

7. This procedure will prevent you from creating a Punch with the same name as your standard-part Pilot. Keeping object names unique is important to ZW3D's internal file/object management. Go ahead and **close the standard-parts.Z3** file without saving.

Now you can edit the parameters of Dayton Punch KJX_5361

8. **Edit** the Dayton… part object and choose **Attributes -> Part** then pick on the **Driving Parameters** Tab.

Appendix A is a lab of the creation of this part and setting up the driving parameters.

Set the Parameters as follows…

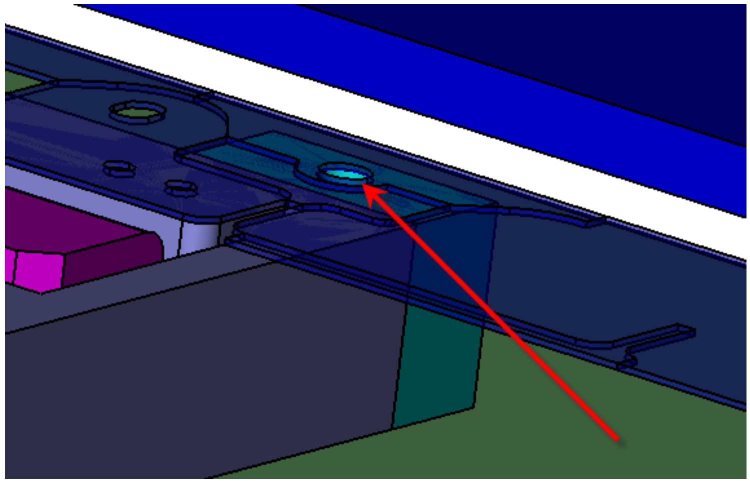
9. Choose **Apply**. Your part should update.

10. **Navigate to your Main Assembly** using the object manager.

11. **Navigate to your Punch Subassembly** using the Assembly Manager.

Our Punch needs to be in the Punch Subassembly, but we need it to be in a location that is in reference to the strip. We can either insert the strip as a component into our Punch Subassembly or we can copy the faces from our strip that we need for inserting our punch. Either method is legitimate. The method we will use will be to copy the face(s) that we need to locate the Punch.

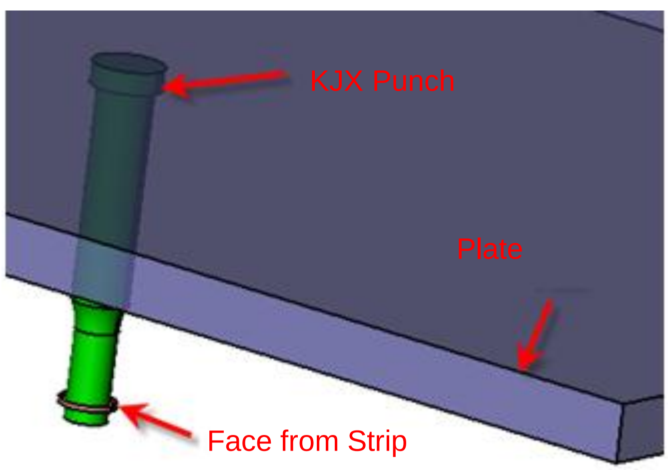
12. Using the **EDIT -> COPY ->Assembly Geometry** command, copy the face of the first pilot hole where this punch will be engaging.



13. **Show Target**

14. From **the** Assembly Tab choose the **anchor** command  and Anchor the face you just copied from the strip.

15. **Insert the Dayton Punch KJX_5361** with the head mated (opposite facing) to the bottom of the Main Punch Plate and the axis of the punch concentric with the face we just anchored.



Now we will repeat this process for the Dayton Pilot KPA.

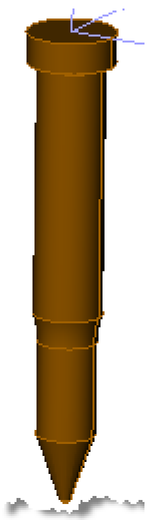
Get the Standard-Part from the Library file to the Job File.

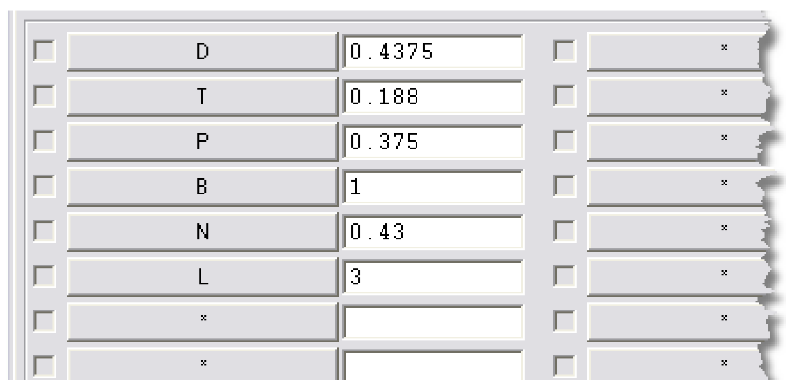
16. **Open Standard-Parts(ProgDie).Z3** using the File pulldown.

- In the object manager **right-click** the **Dayton Pilot KPA (part)** and pick **Copy**
- Pick the **Paste** Button.
- **Cut** the new copy of the Dayton Pilot KPA.1 from the Standard-Parts file
- **Paste** it in Job_5361.Z3's Object manager.
- **Rename** the Dayton Pilot KPA.1 to a name suitable for your job. Let's call it, Dayton Pilot KPA_5361.

17. Modify the Pilot to fit the Strip
    - Edit the Dayton… part object and choose **Attributes -> Part** then pick on the **Driving Parameters** Tab.
    - Appendix A is a lab of the creation of this part and setting up the driving parameters.

Set the Driving Parameters as follows…





☐	D	0.4375	☐	×
☐	T	0.188	☐	×
☐	P	0.375	☐	×
☐	B	1	☐	×
☐	N	0.43	☐	×
☐	L	3	☐	×
☐	×		☐	×
☐	×		☐	×

18. Choose **Apply**. Your part should update.

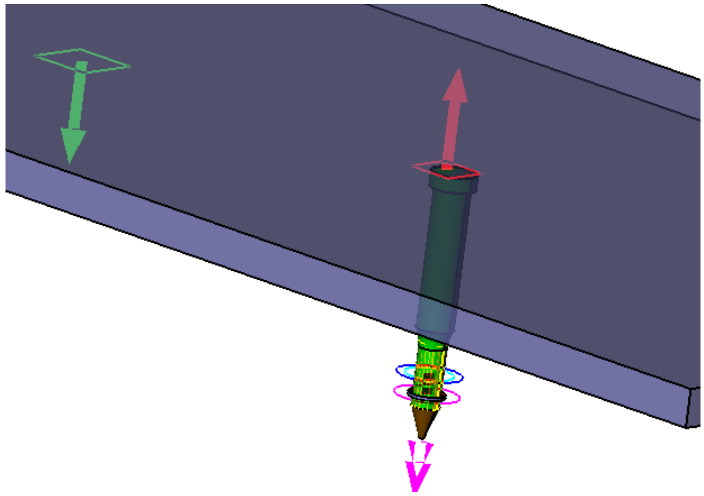
19. **Open the Main Assembly** Object (010_Main_Assembly)

20. **Navigate to the 200_Punch_SubAsm**.

21. If you click on the **Show Target** all you will see at this point is the Punch Plate, the Punch_KJX and the open cylindrical face from the pilot hole.

22. Now, **Insert the Component**, **Dayton Pilot KPA_5361 …from your Job file.**

- Mate the top face of the Pilot to the bottom Face of the Plate.
- Align the Pilot with the face we copied earlier from the strip.



Let's put in 4 more for the other stages in the progression by using the Pattern Command.

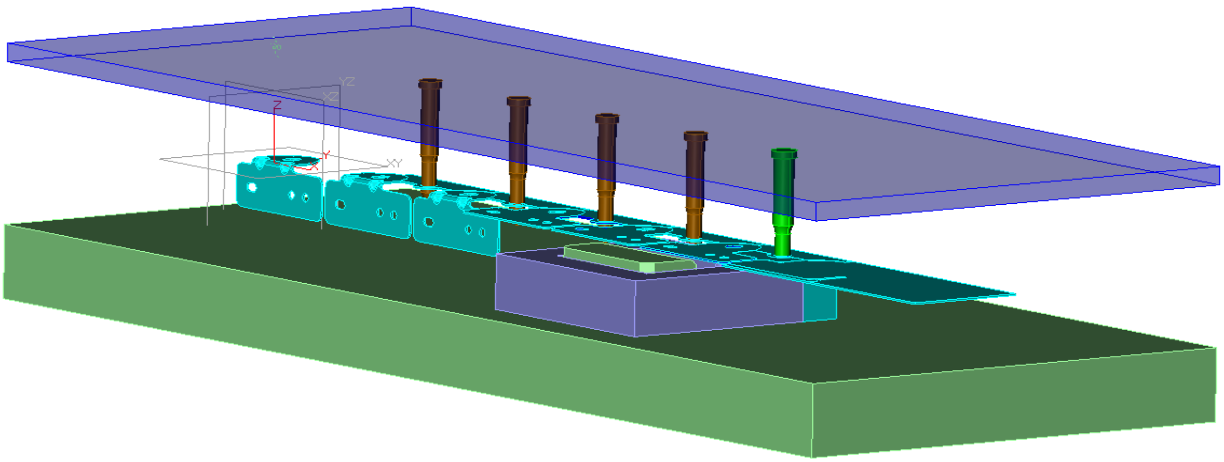
23. Choose the **Pattern** Command from the Shape Tab.

- **Pick the Pilot** as the base component to pattern.
- Direction is a Negative X **(-X).**
- Quantity is **5.**
- Spacing is the **Pitch variable** from the Strip component.
    i. Right Click and choose expression.
    ii. Then Pick the Get Variable button from the Input Expression form.
    iii. Navigate to the Strip and choose the Pitch Variable.
    iv. The Input expressions form should look like [002_Strip_5361:Pitch].
    v. Then Choose **OK** in the Pattern Options Form.
    vi. You can also close the Variable Browser dialog box.

24. You can see that the BOM is accurate by selecting the **BOM Tool** from the Assembly Tab. Hit **OK**. (Remember we are still in the Punch Subassembly).

25. **Erase** the Face that we copied from the Strip, since we no longer need it.

26. Choose the **Show All** button from the Document Awared toolbar and you will see the entire assembly.



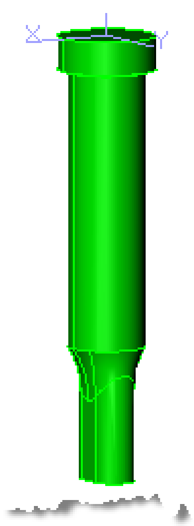
27. **Save** your File.

28. Repeat this procedure using the Dayton Punch KJO from the Standard-Parts.Z3 file.

- **Copy** the object and **Paste** it into the Standard Parts file.

- **Cut** the new Punch and **Paste** it into the Job 5361 object list.

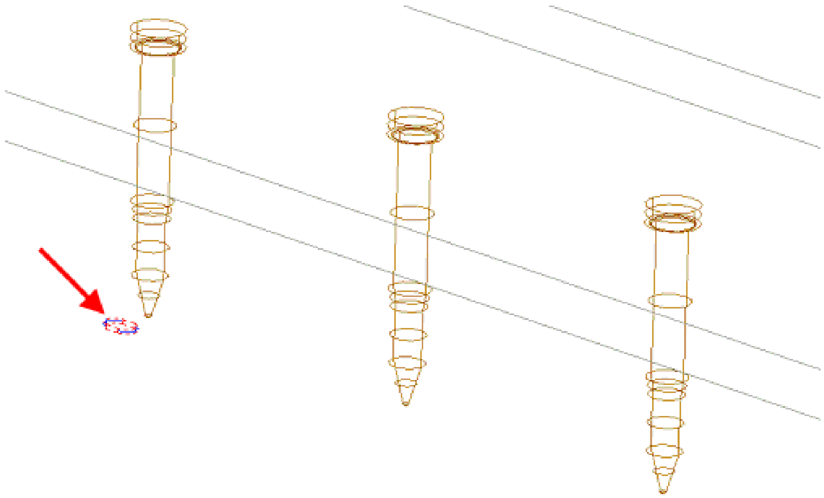
- Edit the **Driving Parameters** of the Part Attributes to be as follows…



D	0.4375
T	0.188
B	1
L	3
P	0.3937
PR	0.11811
×	
×	

- **Navigate to the 200_Punch_SubAsm.**

- Copy reference Geometry from the strip using the **EDIT - > COPY -> Assembly Geometry** command. (Slot shaped feature)



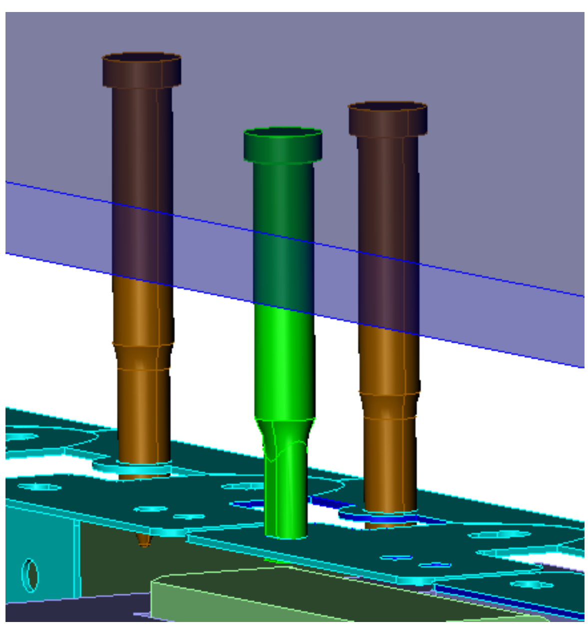
- **Anchor** the faces that you copied into the assembly.

Tip: You can **sew** the faces into one shape before anchoring.

- **Insert the Punch** into the Punch Assembly.

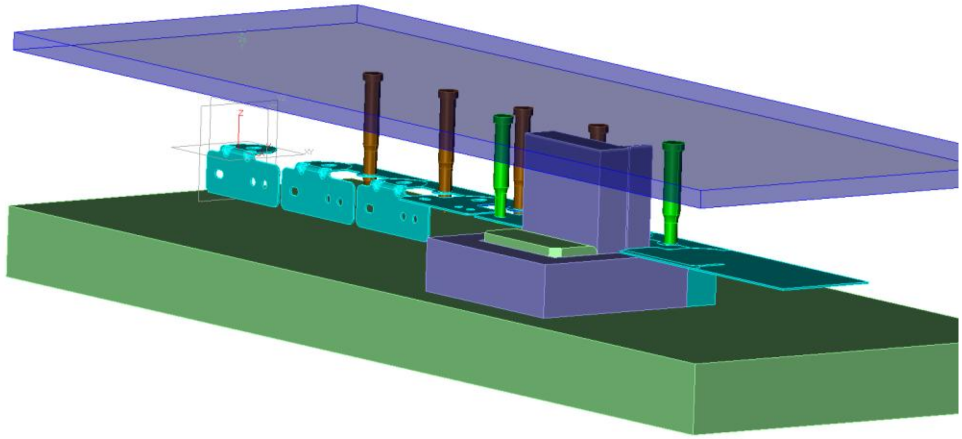
Hint: Mate the planar faces and connect edge to an edge.

See the following image for your final results



Custom Punches

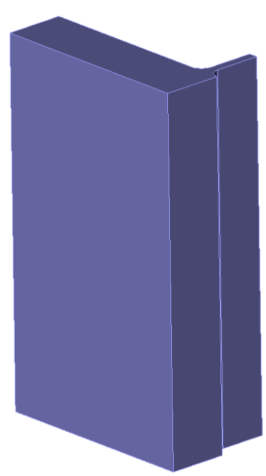
Now that the punch plate is established we can create some custom punches. The technique is nearly identical to that of the custom die blocks. We will use the reference master punch and we will use the copy assembly geometry commands to acquire the faces that we need to create an associative model of our punch.



1. **Navigate to the Punch Subassembly**.

2. **Insert a new component** called, 202_Punch_5361.
   - Be sure to anchor it and place it at Zero.

3. Use **EDIT -> COPY -> External Part** to copy in the part **003_Ref_Punch_Master_5361**



4. **Erase** all of the shapes leaving only the portion that would be used to create the punch for the Punch Notch – see the image to the right for the shape to keep (not erase).

The height from Zero was established in the Ref Punch, so the top is at the correct height.

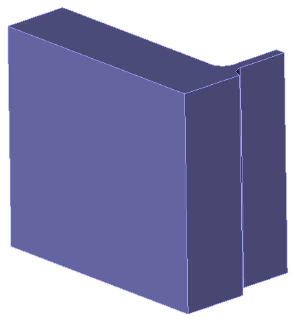
5. Pick the **Show Target** command.

The punch length will run from the Punch Plate to .125 below the XY Plane.

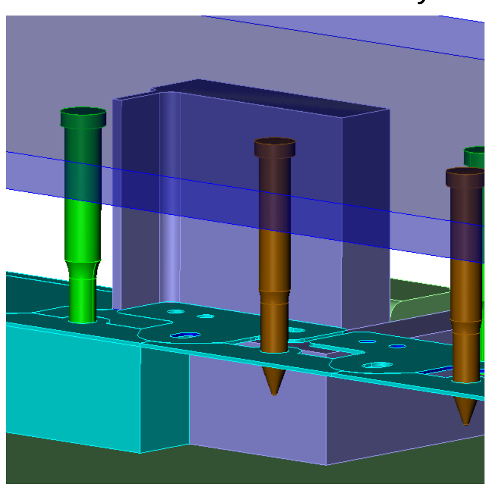
6. **Create a datum** offset from the XY Plane -.125.

7. **Trim** the shape for the pitch notch punch with the new datum.

Your shape should look like the following image…



8. **Show All** to see the part in the context of the assembly



9. **Save** your file.

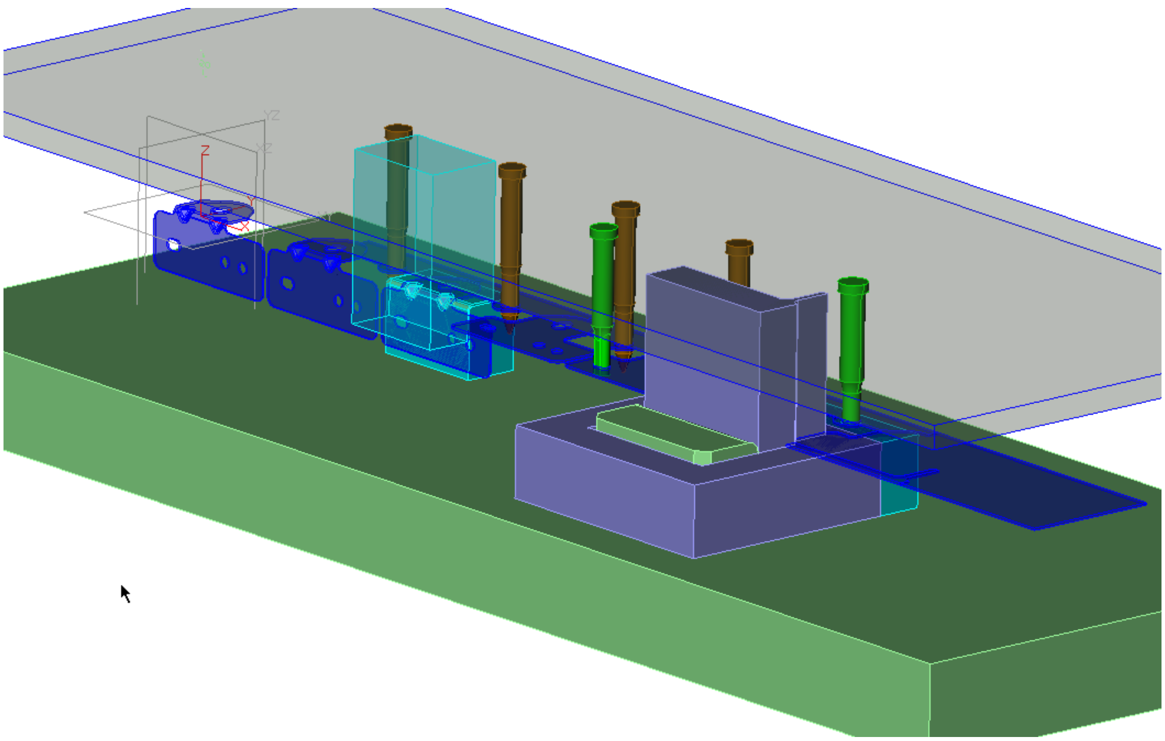
10. go to the Punch Subassembly.

11. go to the Main Assembly

Lesson 8 - Creation of Form Blocks

This lab will focus on the creation form blocks using the Ref_Form Surfaces within the context of the main assembly. We will be implementing best practice methods for maintaining associativity.

In this lab you will create the form blocks for the lower Die Subassembly and the upper Punch Subassembly. We will use the reference trim surfaces we created in an earlier lab.



Using the Assembly Manager , **right-click** the **Die SubAssembly** and choose **Edit Part**.

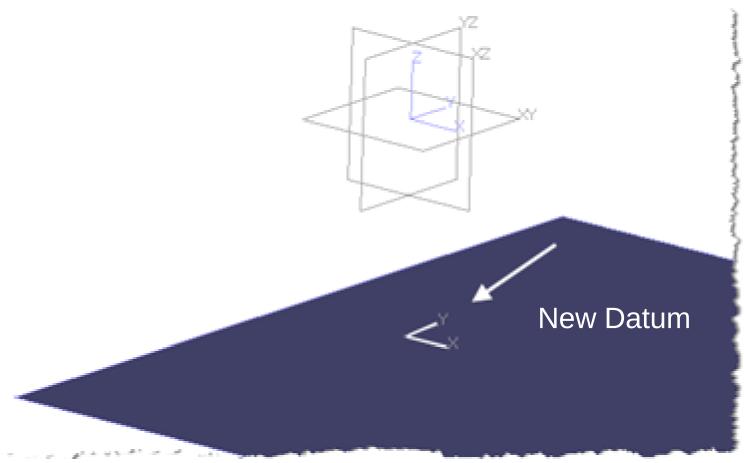
1. **Insert a new component** with the following options…
   - Name: **105_FormBlock_5361.**
   - Anchor: **Checked.**
   - Location: **0,0,0 (remember to hit Enter after typing in this value).**

The Geometry needed to create the block for the lower die is the top face of the die plate and the reference master trim surfaces.

2. Choose the **following** commands from the pulldown menus: **Edit -> Copy -> Assembly Geometry**. Choose the **top face of the die plate**.

3. **Create a Datum** Parallel to XY that is at the Z depth of the Die Plate by establishing the offset of the datum to be positioned to a point on that face. This datum will be used to trim off any excess material that might interfere with the die plate.

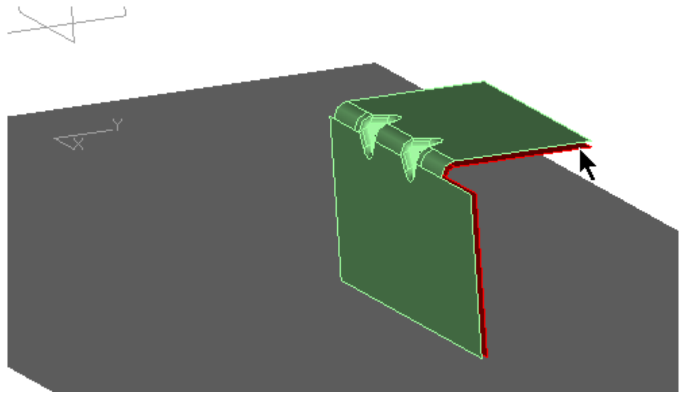
4. **Show Target** to review your results .



5. We can **now** copy the geometry from the Reference Master Trim Surfaces by using the following commands from the Pulldown Menus: **EDIT -> COPY -> External Part**.

- Select **003_Ref_Form_Master_5361**
- You will be prompted for a Position. Position it at **0,0,0**. (Be sure to hit enter after typing in the position value.

You should see the part object that we modeled on page 37.

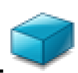


6. To verify that it is in the correct position click on the **Show All** command to see the part in the context of the main assembly.

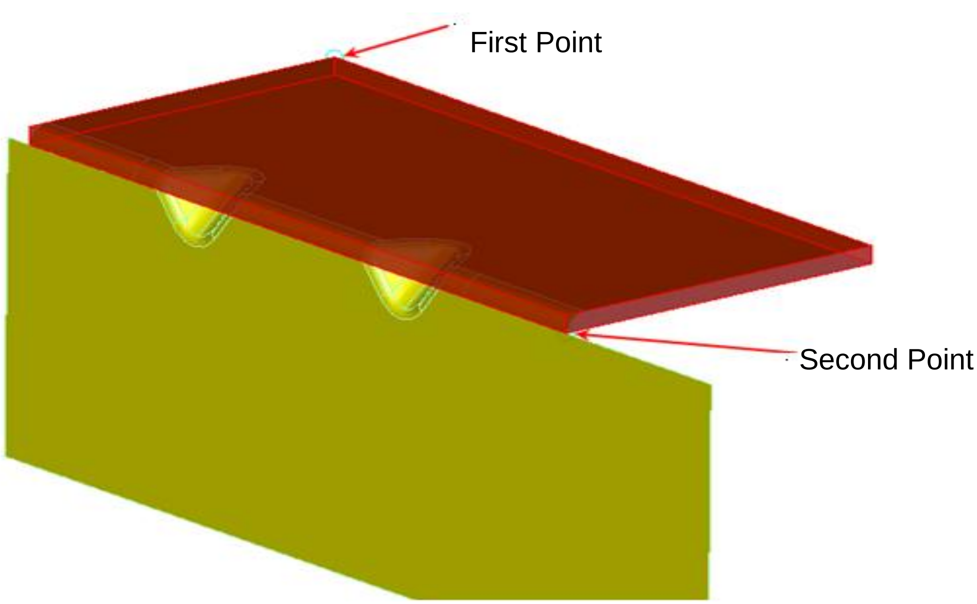
7. Click on **Show Target**

8. **Erase** the outer (shape) that would be used on the upper form block.

9. **Erase** the face we copied from the Die Plate.

10. **Create a primitive <u>Base</u> Block** as follows… 

- Use the Block by Corners Option Pick the corners as shown… 



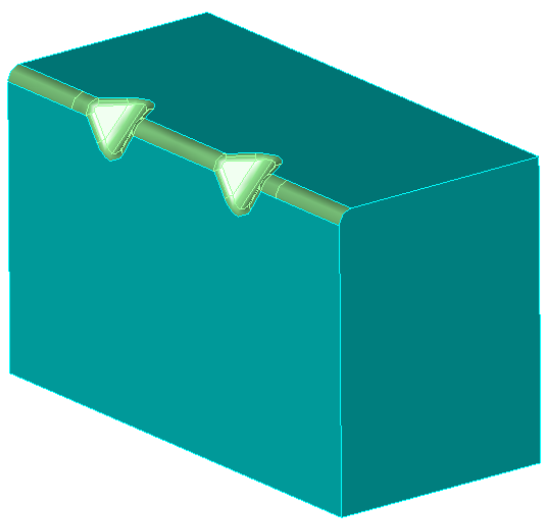
- Click in the box indicated below to the right of Z Length from the Options form. When you move your cursor out into the modeling area you will notice that you can dynamically change the height of the block.
- **Right-Click** on the screen for position options and choose **To Point…**
- Snap to the datum that you created in step 5.
- Complete the Block Feature.

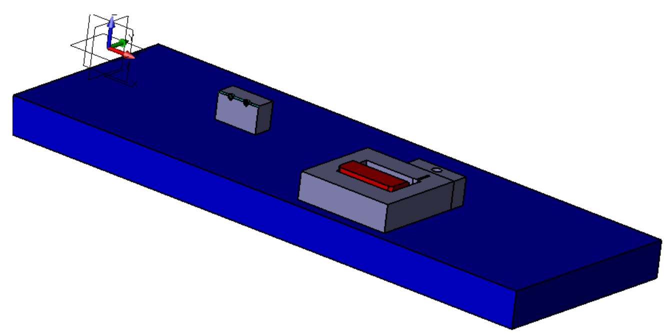
11. If you **desire** the block to be a nominal size, you can resize the block using **face offset commands**.

12. Set your **filter to shape**.

13. **Right-Click the Block and choose remove**. Select the Reference Form Faces as the shape to be removed.

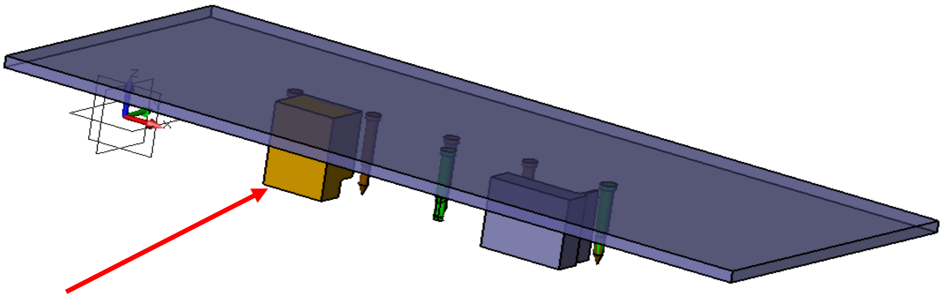
Your results will be as follows…

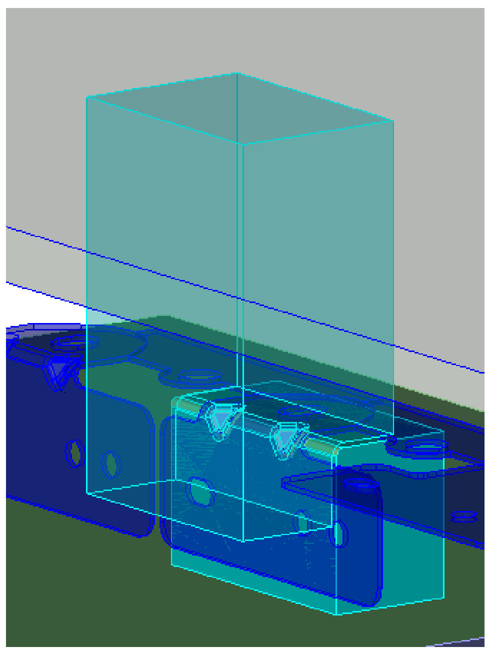


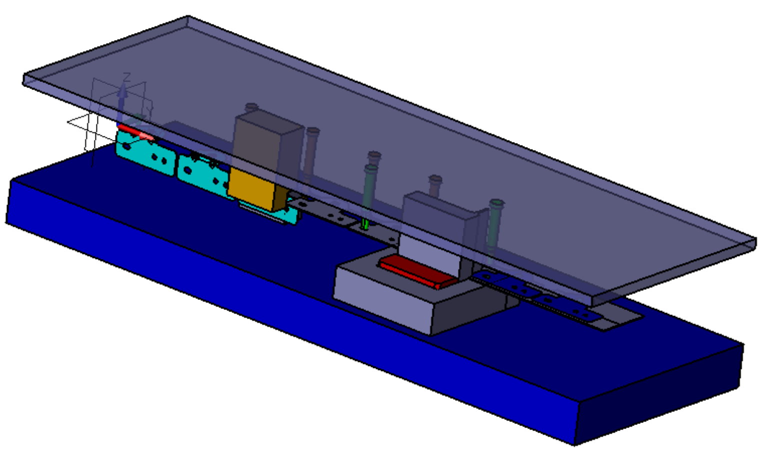


Note: You could also create a base block using the Stock Command.

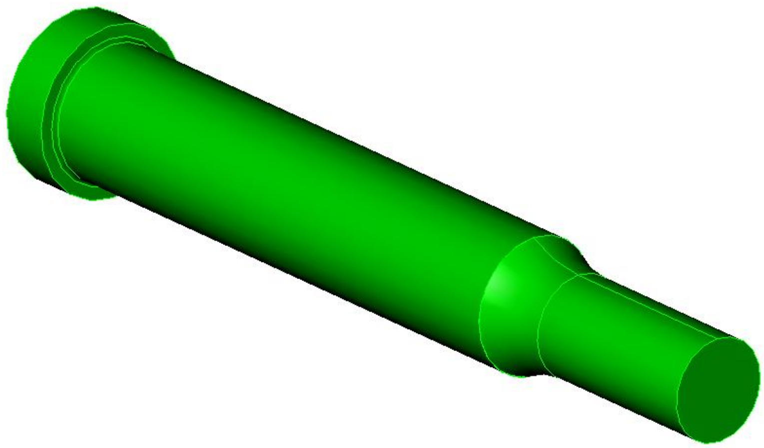
Now you should be able to repeat this process for the Upper Block.

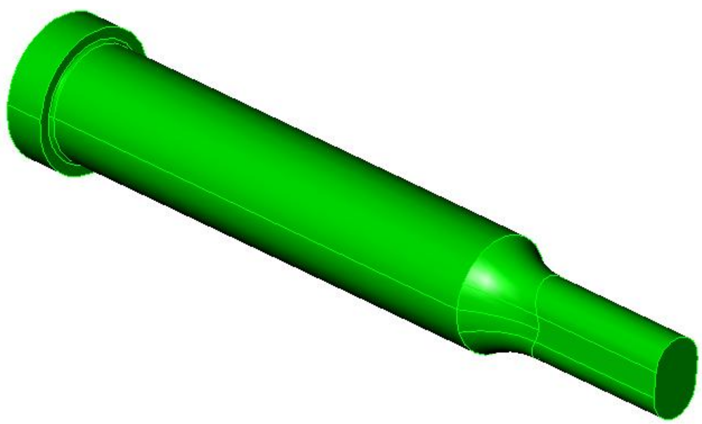


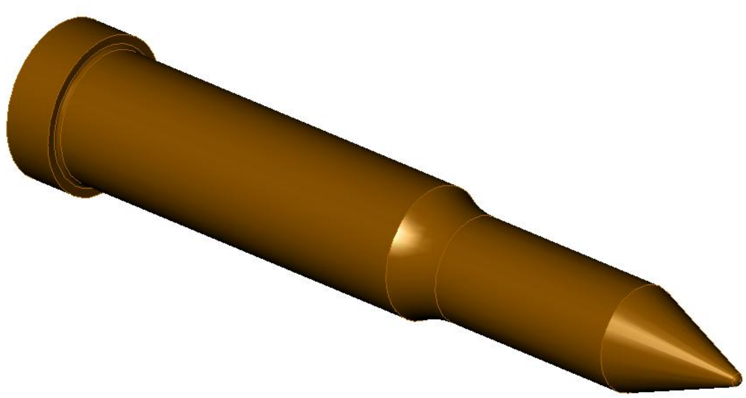




Appendix A - Standard Parts Creation

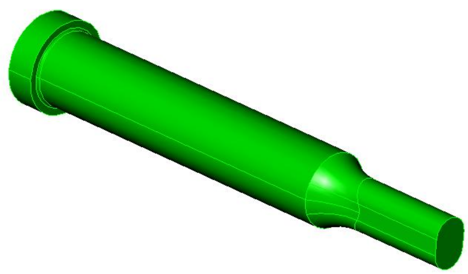






Punches and Pilots have nearly an infinite number of sizes and lengths, so we will not be employing the part tables' functionality in this lab. Rather, we will be designing a Punch and naming key dimensions and variables so that you can easily create a new one by editing these parameters.

The Punch we will make will be a Dayton KJO and you will be able to apply these principles to the creation of other similar punches.
I often find it easier to create standard parts using multiple features rather than a complex sketch that is revolved, so that is the approach we will use in this lab.



1. Begin by **creating a new File called MY_Standard_Parts.**

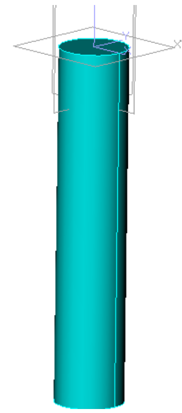
2. **Create a new part object called Dayton_Punch_KJO**.

3. **Create sketch** of a circle centered on Zero with a Diameter of 0.50. This will be for the Shank of the punch.

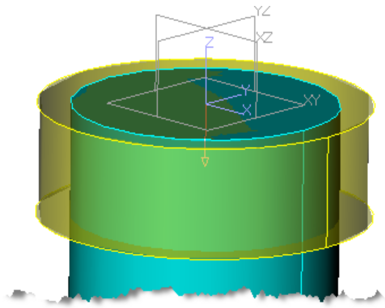
4. Tag the dimension with a Name of **D.**

   - Choose **Attribute – >Tags –> Attach name tag to entity.**
   - **Pick** the **dimension.**
   - **Type in D on the prompt line and hit enter.**

5. **Extrude the circle** as a base shape in the Negative Z direction a distance of 2.5. The 2.5 value will later be tagged as a driving parameter for Length, so it is best if it is a positive value. The extrude function lets you set the direction as an optional parameter, so setting it to negative Z allows us to use a positive end value.

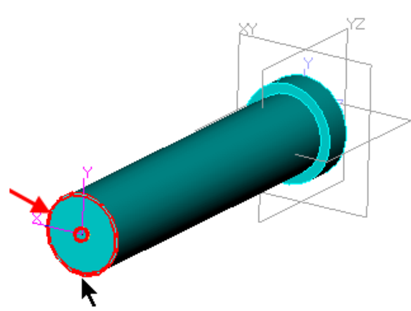


6. We can quickly create the head of the punch by **extruding as an added shape**. Use the sketch we created in step 3 with an offset of 0.0625. Also, extrude in the negative Z direction, so that your head dimension (T) value will be a positive number. (.188)

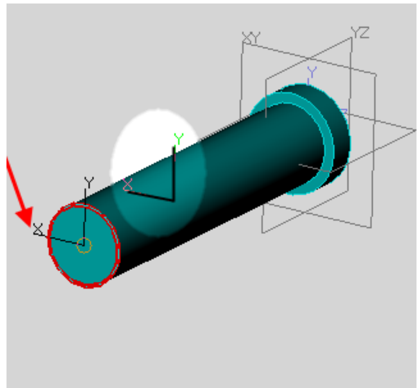


7. We need to cut off the base length of the cylinder by the B dimension of the punch, so we will create a datum offset by 1” from the bottom of the cylinder.

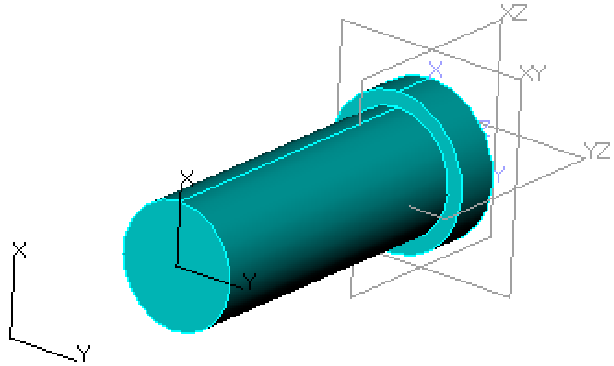
   - **Insert Datum**
   - Pick XY from the options form
   - Offset = 0
   - Then specify the optional input for Origin as the center of the bottom edge of our base extrusion(as indicated in red in the figure below).



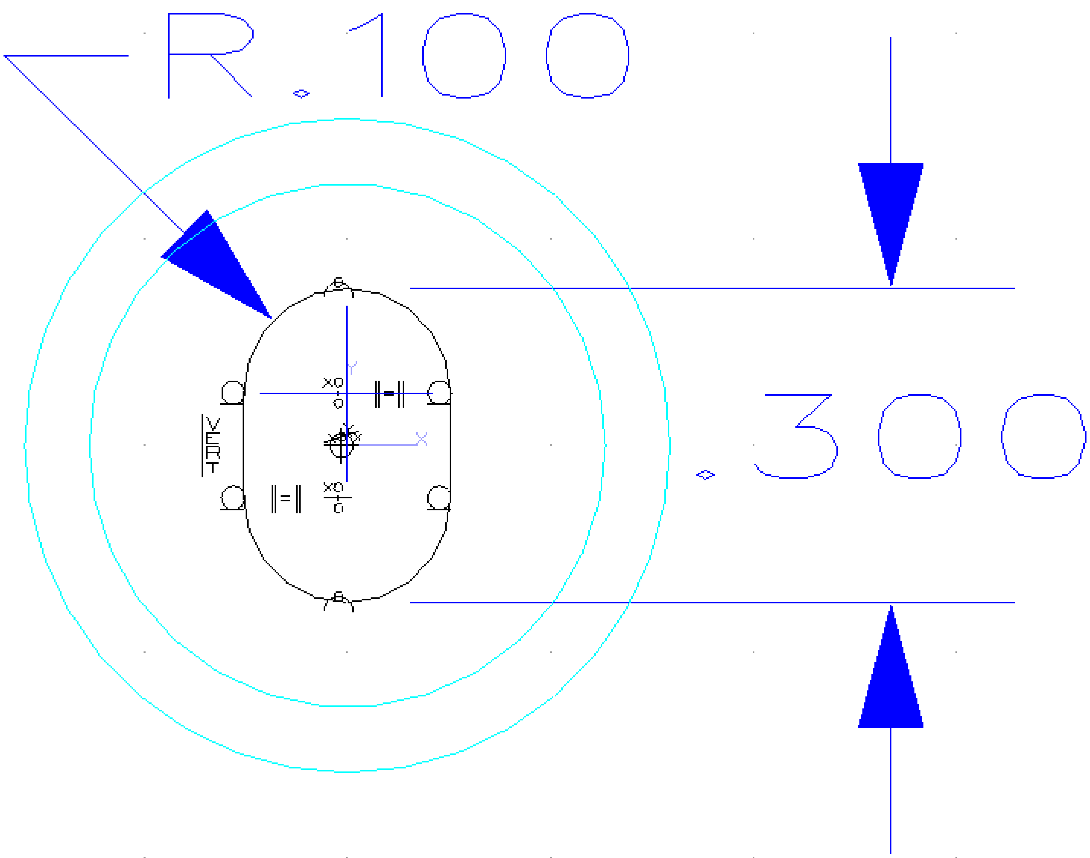
8. **Insert another datum** (as highlighted in the figure below) parallel to the previous datum (as indicated in red in the figure below) with an offset of 1.



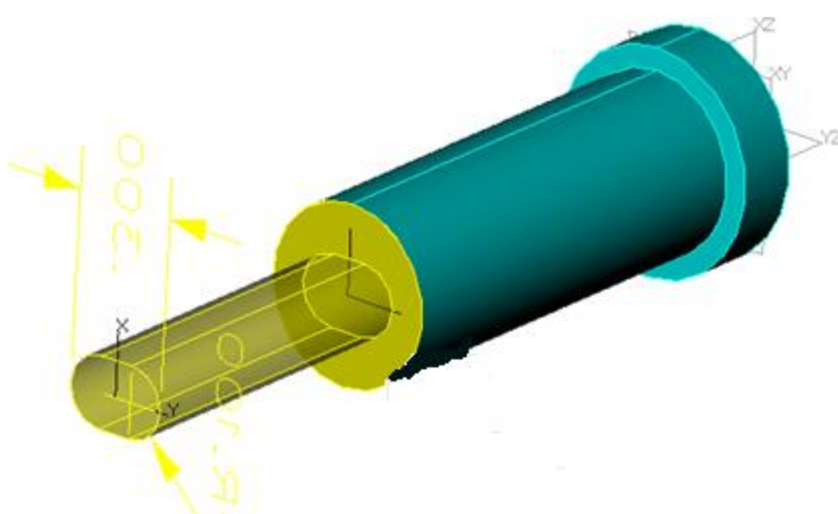
9. **Trim** the shape to the second datum we created keeping the side in the positive Z.



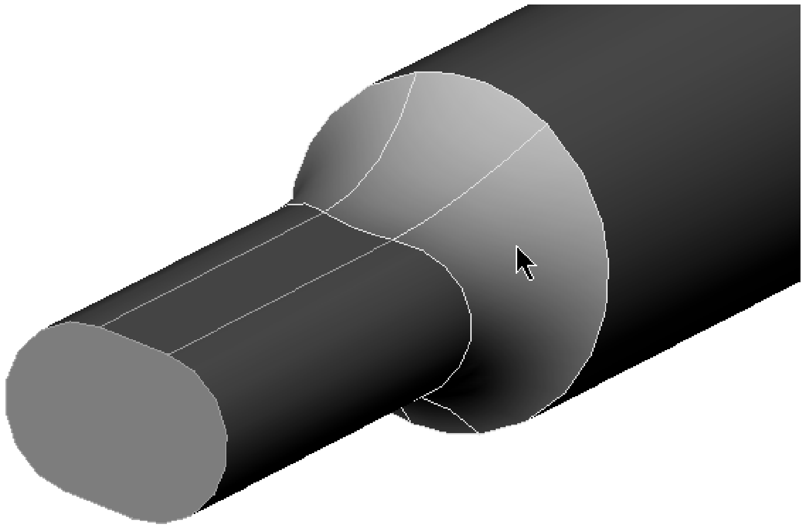
10. **Insert the sketch** shown in the following images. Put the sketch on the first datum we created.
    - Be certain that your profile is centered on Zero
    - Change dimensions to verify the integrity of your constraints.



11. **Extrude** this sketch with a start of Zero and end it at the face that was the result of the previous trimmed shape. Be sure to select the Add option.

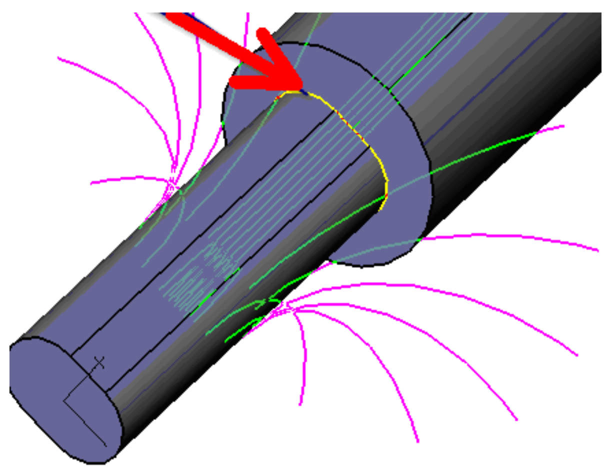


Let's put the **0.50 radius fillet** on the part between the shank and previous extrusion. See the image below for the results we will achieve in then next steps.

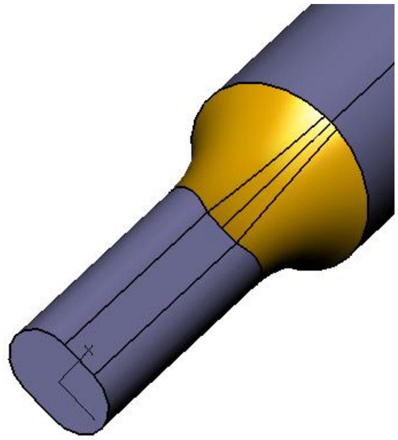


12. **Pick** the **fillet**  command from the Shape Tab.

13. Use a **radius of 0.5**, Check the option "**Hold fillet to edge**". Shift pick the edge as indicated in the figure below. Then pick **OK**.



Your results should look like this:

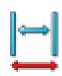


14. Save your file.

Test the integrity of the associativity of your part by changing dimensions within a reasonable context.

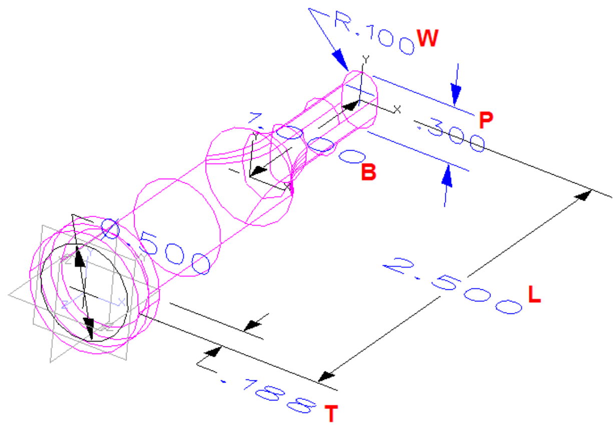
- Your model should not have any trouble regenerating changes.

Now we need to make it easier to change the key parameters of this part. This is done by name tagging the remaining key dimensions and putting those key dimensions in a form in a sequence that is logical.

15. **Turn Dimensions On/Off** button  from the Document Awared toolbar, until all the dimensions are showing on your part.

16. Choose **Attribute – Tags – Attach name tag to entity.** And name tag the dimensions as shown in the image below.

    T = .188; B = 1.00
    L = 2.50
    P = .300
    W = .100



Now we will put these Driving Dimensions into a table to make it convenient to derive new parts by editing the parameters of this part.

17. Pick **Attributes –> Part** from the Pull down menus. **Select the Driving Parameters Tab**.

18. **Pick the first button with the Star *** and then **select your D dimension** from the base sketch.

    Your Driving Parameters should look like the image below when you are done.

D	0.5	*	
T	0.188	*	
B	1	*	
L	2.5	*	
P	0.3	*	
W	0.1	*	
*		*	
*		*	
*		*	
*		*	

19. **Edit a parameter** like L (Just change 2.5 to 3) and pick **Apply** then **OK**.
    - You will see your part change.
    - **Change it back** to its original value using the Driving Parameters table found under Attributes – Part.

If you select the Derive new Part Button at the bottom of the Table (shown in the image above), you will be prompted for an object name and the option to create a new file. This is one way to generate a new Punch.

You can also open your Standard Punches file and copy the Punch object that you want and Paste it into your current project file as its own object then edit the Driving Parameters table to change the copied object to create a punch for a particular progressive die.

Appendix B - Table / Library Parts Creation

There are many parts like fasteners, springs and spools that are catalog parts that only have certain sizes, unlike punches and pilots. These parts can be driven from a design table. ZW3D has excellent support for design tables. In this lab we will work thru the process creating a threaded spool and we will show the various methods of inserting these types of components into an assembly.

There are 4 Basic Steps to creating a Part Table/Library

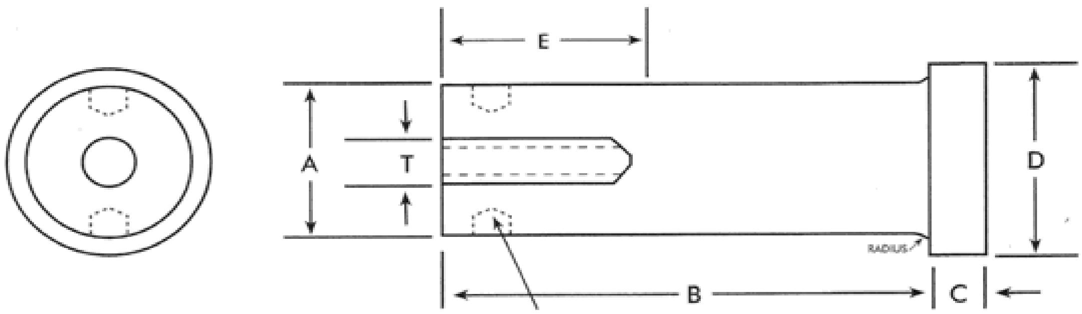
1. Create a parametric part that can be edited either with Variables and/or critical Dimensions
2. Test the model to be sure that reasonable edits will regen properly.
3. Name Key Dimensions with Attribute Tags and Assign your driving Variables and Dimensions to the Driving Parameter attributes of the Part
4. Create and Populate the Part Table from the Driving parameters.

NOTE: Part Tables can be edited long after creation.

IMPORTANT: The library parts generated by the part table are, by default, not linked to nor dependant on the table after creation. These parts can be individually edited or modified and they have their own stand alone history tree. At the end of the exercise we will discuss some methods of using a part table.

1. Begin by creating a new **Multi-Object File** called **Table_Parts.**

2. Create a **new part object** called **Threaded_Spool [use the PartTemplate(IN)]**. We will be building the following part table.



1/4″ ×1/4″ HOLE PROVIDED TO ASSIST IN INSTALLATION AND REMOVAL

PART No.	B±0.05	PART No.	B±0.05	PART No.	B±0.05	PART No.	B±0.05
T-0710	1						
T-0711	1-1/4	T-1011	1-1/4	T-1111	1-1/4		
T-0712	1-1/2	T-1012	1-1/2	T-1112	1-1/2	T-1212	1-1/2
T-0713	1-3/4	T-1013	1-3/4	T-1113	1-3/4	T-1213	1-3/4
T-0720	2	T-1020	2	T-1120	2	T-1220	2
T-0721	2-1/4	T-1021	2-1/4	T-1121	2-1/4	T-1221	2-1/4
T-0722	2-1/2	T-1022	2-1/2	T-1122	2-1/2	T-1222	2-1/2
T-0723	2-3/4	T-1023	2-3/4	T-1123	2-3/4	T-1223	2-3/4
T-0730	3	T-1030	3	T-1130	3	T-1230	3
T-0731	3-1/4	T-1031	3-1/4	T-1131	3-1/4	T-1231	3-1/4
T-0732	3-1/2	T-1032	3-1/2	T-1132	3-1/2	T-1232	3-1/2
T-0733	3-3/4	T-1033	3-3/4	T-1133	3-3/4	T-1233	3-3/4
T-0740	4	T-1040	4	T-1140	4	T-1240	4
T-0742	4-1/2	T-1042	4-1/2	T-1142	4-1/2	T-1242	4-1/2
T-0750	5	T-1050	5	T-1150	5	T-1250	5
T-0752	5-1/2	T-1052	5-1/2	T-1152	5-1/2	T-1252	5-1/2
T-0760	6	T-1060	6	T-1160	6	T-1260	6

$A = .750 \pm ^{.000}_{.010}$	$A = 1.000 \pm ^{.000}_{.010}$	$A = 1.250 \pm ^{.000}_{.010}$	$A = 1.500 \pm ^{.000}_{.010}$
C = 1/4″	C = 1/4″	C = 1/2″	C = 1/2″
D = 1″	D = 1-1/4″	D = 1-5/8″	D = 1-7/8″
E = 1″	E = 1-1/4″	E = 1-1/4″	E = 1-1/4″
$T = 5/16^{-18}$	$T = 3/8^{-16}$	$T = 1/2^{-13}$	$T = 5/8^{-11}$

3. We will start by defining all of the variables for this part. The two key variables are A (spool diameter) and B (spool length), since they determine the other variables.
   - If… then… logic can be built into the equation editor that can cause A to drive C, D, E, and T, but for simplicity sake we will simply have the table drive these

values. The benefit to the user is that they will be seen when inserting a part rather than being calculated in the background.

4. Select **Variable** from the **Insert** pull down menu.

5. Fill out the dialog box using as shown to the right.
   - Variable: **A**
   - Expression: **.75.in**
   - Distance: **CHECKED**

6. Repeat this for each variable B, C, D, E, and T.
   - Expression for B: **1.in**
   - Expression for C: **.25.in**
   - Expression for D: **1.in**
   - Expression for E: **1.in**
   - Expression for T: **.3125.in**

NOTE: It is best to start at the minimum value for a variable like B because the Table editor allows us to populate the table by incrementing various values.

7. The table will also fill in Part Number information, so it's good to have a default part number established in your reference part.
   - Choose **Part** from the **Attributes** pull down menu.
   - From the Standard Attributes Tab fill in the Number as **T-0710.**

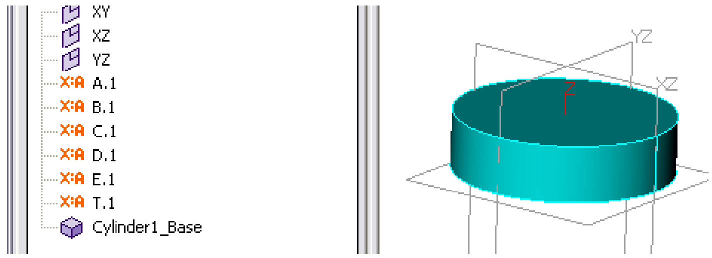
Now we can begin modeling this part using the variable**s we establis**hed to drive the parameters of the model. Then we will put these variables into a table and populate the table with the chart shown at the beginning of this appendix.

8. Let's start by creating a Primitive Cylinder feature that represents the head of the spool.

   This command is found on the Shape Tab. 

   - Center: 0,0,0
   - Radius: D/2
   - Length: C

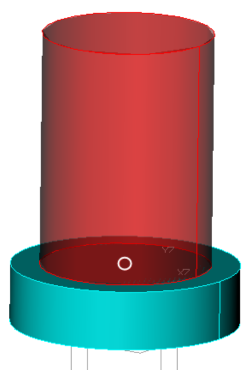
9. Your results with your history tree should look like the following image.



10. Create another cylinder centered on the top of the first cylinder.

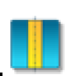
- Choose the **Add** option from the top of the options form. 

- To snap the center to be concentric with the other cylinder be sure to right click and choose **Critical** from the **Snap Menu** and Make sure your **Filter** is set to **All or Edge**

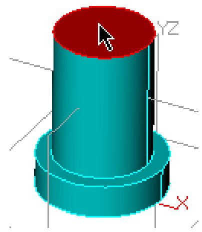


- Radius: **A/2** (hit <ENTER>).
- Length: **B** (hit <ENTER>)

11. Next we will put the hole in the end of the part. Since this is a standard purchase part and there is really no need to detail or manufacture this individual part, we will make the hole a simple hole rather than a tapped hole to simplify things.

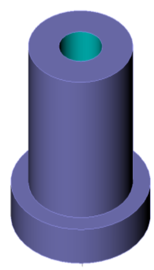
12. Pick the **Hole** feature from the Shape Tab. 

- When prompted for the face to place holes, pick the top face of the spool.



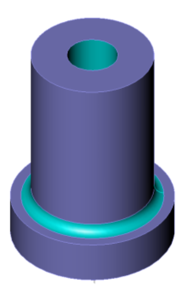
- When prompted for the location pick the edge of the top face and you should see your cursor snap to the center of the face. Left click one time to lock in the snap point and then Middle Click to accept because we are only putting in one hole.
- When prompted for Diameter, type in ***T*** on the prompt line and hit <ENTER>
- When prompted for Hold Depth type in ***E*** on the prompt line and hit <ENTER>
- Pick OK on the bottom of the Hole options form to complete the Hole feature.

Your results should look like the following image…



13. Now we need to put the .06 radius on the edge between the two cylinders. I typically put the non essential features of the part on last so that the main parameters are unaffected by them.
    - We are not going to take the time to put the ¼ x ¼ installation hole in. It would simply add more unnecessary complexity.

Your final model should look like the following image…



14. We should test our associativity between features by making reasonable changes to the variables.

    - From your History Manager/Tree **right click** on the **B** variable and choose **Open/Close.**
    - **Right Click** on **Expression** and choose **Edit.**
    - A dialog box will appear. Change the value to **4.in** and pick **OK.**
    - The history will automatically regen with the new value.
    - Repeat this step setting B back to **1.in** rather than using undo to test the parametrics when the part is getting smaller.
    - Make reasonable changes to other variables to see the model recomputed.
    - Since everything is working properly, we can add our variable to the Driving parameters of our Part Attributes.

15. Choose **Part** from the **Attributes** pull down. And select the **Driving Parameters** tab.

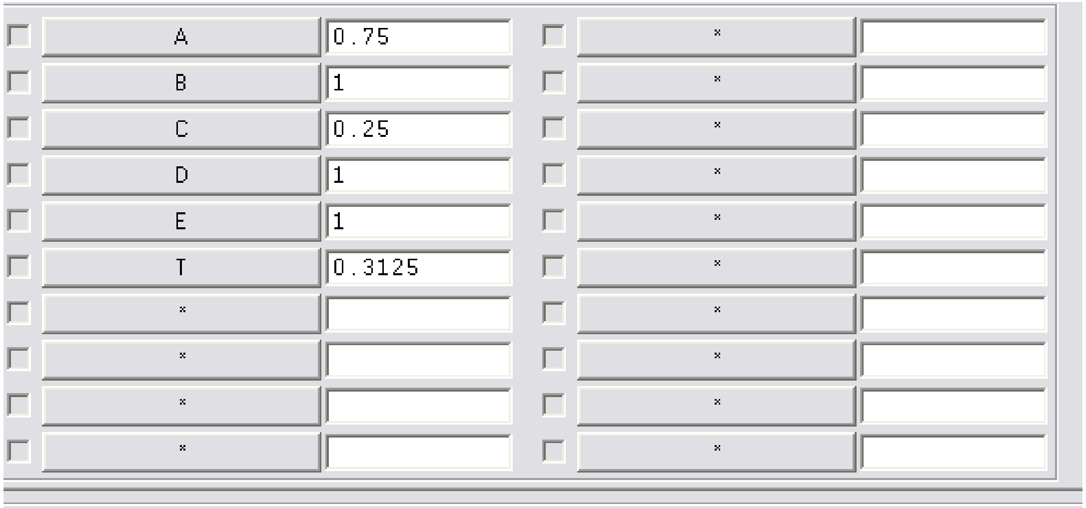
16. We add variables to the parameters by picking a button.

then **Right Click** and choose **Browse Variables**

17. The **Variable Browser** will appear. Simply **Left Pick** the **A** variable and it will appear as the first item in your driving parameters

18. Then **pick the next button** from the driving parameters form and pick the next Variable listed in the Variable Browser until all of the Variables we created are in the list. If you were also using dimensions you can select those by Showing Feature Dimen or Showing All Dimen (see the right click options in step 20 above).

When you are finished adding your variables your part attributes form should look like the following image…



19. You can choose the **OK** button at the bottom of the Variable Browser form to close it out.This is a great time to save your file.We can now setup the Part Table.

20. Choose **Part Table from the Utilities** pull down menu.

21. Choose **No** from the Dialog Box that pops up to create a New Table.

You will see the table editor appear…

```
# Refer to online help for more information.
#
# description
# file = Table_Parts.VX
# units = inch
# simplify = not available
# input = Material, Material, Aluminum
# input = A, A@, 0.75
# input = B, B@, 1
# input = C, C@, 0.25
# input = D, D@, 1
# input = E, E@, 1
# input = T, T@, 0.3125
# output = Name, Name
# output = Number, Number
#
Threaded_Spool,Material,Number,Density,Color,A@,B@,C@,D@,E@,T@
#
Threaded_Spool.1,Aluminum,0001,1,140.140.235,0.75,1,0.25,1,1,0.3125
```

The first entries in the table are derived from our current part with the exception of the part number attribute.

- We should edit the default part number now to make easier to edit the part numbers later.

- **Right-Click** on the first table entry and choose **Edit**
- **Type in the Value T-0711** for the Number and Pick **Apply**.
- **Choose OK**. This will close the editor pop up and return you to the table editor.

Now we **n**eed to begin populating the Table. If you refer back to the beginning of the Appendix you will see that all of the Parts with an ***A*** value of .75 inches increment ***B*** by .25 inches 13 times then ***B*** increments by .5 inches 4 times. ZW3D's Table editor allows us to pattern entries to create additional entries.

- **Right-Click** on the first entry in the table and choose **Pattern**
- Fill in the Dialog Box as shown below…

1st Parameter: B@	Quantity: 12	Increment: 0.25

- Pick **OK**. And you should see a 12 more entries added to your table.
- To get the remaining 5 entries **Right-Click** the Last Entry in the Table and Pick **Pattern.**
- Fill in the Dialog Box as shown below…

1st Parameter: B@	Quantity: 5	Increment: 0.5

Your table should look like the following image…

```
#
Threaded_Spool.1,Aluminum,T-0711,1,140.140.235,0.75,1,0.25,1,1,0.3125
Threaded_Spool_2,Aluminum,T-0711,1,140.140.235,0.75,1.25,0.25,1,1,0.3125
Threaded_Spool_3,Aluminum,T-0711,1,140.140.235,0.75,1.5,0.25,1,1,0.3125
Threaded_Spool_4,Aluminum,T-0711,1,140.140.235,0.75,1.75,0.25,1,1,0.3125
Threaded_Spool_5,Aluminum,T-0711,1,140.140.235,0.75,2.25,0.25,1,1,0.3125
Threaded_Spool_6,Aluminum,T-0711,1,140.140.235,0.75,2.75,0.25,1,1,0.3125
Threaded_Spool_7,Aluminum,T-0711,1,140.140.235,0.75,3.25,0.25,1,1,0.3125
Threaded_Spool_8,Aluminum,T-0711,1,140.140.235,0.75,3.75,0.25,1,1,0.3125
Threaded_Spool_5,Aluminum,T-0711,1,140.140.235,0.75,2,0.25,1,1,0.3125
Threaded_Spool_6,Aluminum,T-0711,1,140.140.235,0.75,2.25,0.25,1,1,0.3125
Threaded_Spool_7,Aluminum,T-0711,1,140.140.235,0.75,2.5,0.25,1,1,0.3125
Threaded_Spool_8,Aluminum,T-0711,1,140.140.235,0.75,2.75,0.25,1,1,0.3125
Threaded_Spool_9,Aluminum,T-0711,1,140.140.235,0.75,3,0.25,1,1,0.3125
Threaded_Spool_10,Aluminum,T-0711,1,140.140.235,0.75,3.25,0.25,1,1,0.3125
Threaded_Spool_11,Aluminum,T-0711,1,140.140.235,0.75,3.5,0.25,1,1,0.3125
Threaded_Spool_12,Aluminum,T-0711,1,140.140.235,0.75,3.75,0.25,1,1,0.3125
Threaded_Spool_13,Aluminum,T-0711,1,140.140.235,0.75,4,0.25,1,1,0.3125
Threaded_Spool_14,Aluminum,T-0711,1,140.140.235,0.75,4.5,0.25,1,1,0.3125
Threaded_Spool_15,Aluminum,T-0711,1,140.140.235,0.75,5,0.25,1,1,0.3125
Threaded_Spool_16,Aluminum,T-0711,1,140.140.235,0.75,5.5,0.25,1,1,0.3125
Threaded_Spool_17,Aluminum,T-0711,1,140.140.235,0.75,6,0.25,1,1,0.3125
```

22. The only items to be manually edited are the Numbers (part number).

- You simply right click on each entry and choose Edit (or Double Click)
- Edit Number field and pick OK.
- Your final results should look like the following image…

```
#
Threaded Spool.1,Aluminum,T-0710,1,140.140.235,0.75,1,0.25,1,1,0.3125
Threaded Spool_2,Aluminum,T-0711,1,140.140.235,0.75,1.25,0.25,1,1,0.3125
Threaded Spool_3,Aluminum,T-0712,1,140.140.235,0.75,1.5,0.25,1,1,0.3125
Threaded Spool_4,Aluminum,T-0713,1,140.140.235,0.75,1.75,0.25,1,1,0.3125
Threaded Spool_5,Aluminum,T-0720,1,140.140.235,0.75,2,0.25,1,1,0.3125
Threaded Spool_6,Aluminum,T-0721,1,140.140.235,0.75,2.25,0.25,1,1,0.3125
Threaded Spool_7,Aluminum,T-0722,1,140.140.235,0.75,2.5,0.25,1,1,0.3125
Threaded Spool_8,Aluminum,T-0723,1,140.140.235,0.75,2.75,0.25,1,1,0.3125
Threaded Spool_9,Aluminum,T-0730,1,140.140.235,0.75,3,0.25,1,1,0.3125
Threaded Spool_10,Aluminum,T-0731,1,140.140.235,0.75,3.25,0.25,1,1,0.3125
Threaded Spool_11,Aluminum,T-0732,1,140.140.235,0.75,3.5,0.25,1,1,0.3125
Threaded Spool_12,Aluminum,T-0733,1,140.140.235,0.75,3.75,0.25,1,1,0.3125
Threaded Spool_13,Aluminum,T-0740,1,140.140.235,0.75,4,0.25,1,1,0.3125
Threaded Spool_14,Aluminum,T-0742,1,140.140.235,0.75,4.5,0.25,1,1,0.3125
Threaded Spool_15,Aluminum,T-0750,1,140.140.235,0.75,5,0.25,1,1,0.3125
Threaded Spool_16,Aluminum,T-0752,1,140.140.235,0.75,5.5,0.25,1,1,0.3125
Threaded Spool_17,Aluminum,T-0760,1,140.140.235,0.75,6,0.25,1,1,0.3125
```

23. To create the next column of Entries we can copy on from our current set that requires the least editing, like T-0711.

- **Right Click** on the row for the entry of T-0711and Pick **Copy**.
- **Right-Click** on the last row and Choose **Paste**. The new row will be pasted in after the row you Right-Click on. It will not replace the last row.

24. **Edit** the row you pasted in to **change the default Number (part number) to T-1011** as well as the rest of values shown in the following table…

Threaded_Spool	Threaded_Spool_2
Material	Aluminum
Number	T-0711
Density	1
Color	140.140.235
A@	1
B@	1.25
C@	0.25
D@	1.25
E@	1.25
T@	0.375

Pick **Apply**, then **OK** when completed.

25. Let’s Pattern this last entry to fill in a majority of the next column in our chart.

- **Right-Click** the last entry in the table and Choose **Pattern**
- Fill out the pattern dialog box as follows…

1st Parameter： B@	Quantity： 12	Increment： 0.25

- Manually edit the part number fields
- Pattern the last entry of the table to create the remaining 4 entries.

Your final result should appear as follows…

```
#
Threaded_Spool,Material,Number,Density,Color,A@,B@,C@,D@,E@,T@
#
Threaded_Spool.1,Aluminum,T-0710,1,140.140.235,0.75,1,0.25,1,1,0.3125
Threaded_Spool_2,Aluminum,T-0711,1,140.140.235,0.75,1.25,0.25,1,1,0.3125
Threaded_Spool_3,Aluminum,T-0712,1,140.140.235,0.75,1.5,0.25,1,1,0.3125
Threaded_Spool_4,Aluminum,T-0713,1,140.140.235,0.75,1.75,0.25,1,1,0.3125
Threaded_Spool_5,Aluminum,T-0720,1,140.140.235,0.75,2,0.25,1,1,0.3125
Threaded_Spool_6,Aluminum,T-0721,1,140.140.235,0.75,2.25,0.25,1,1,0.3125
Threaded_Spool_7,Aluminum,T-0722,1,140.140.235,0.75,2.5,0.25,1,1,0.3125
Threaded_Spool_8,Aluminum,T-0723,1,140.140.235,0.75,2.75,0.25,1,1,0.3125
Threaded_Spool_9,Aluminum,T-0730,1,140.140.235,0.75,3,0.25,1,1,0.3125
Threaded_Spool_10,Aluminum,T-0731,1,140.140.235,0.75,3.25,0.25,1,1,0.3125
Threaded_Spool_11,Aluminum,T-0732,1,140.140.235,0.75,3.5,0.25,1,1,0.3125
Threaded_Spool_12,Aluminum,T-0733,1,140.140.235,0.75,3.75,0.25,1,1,0.3125
Threaded_Spool_13,Aluminum,T-0740,1,140.140.235,0.75,4,0.25,1,1,0.3125
Threaded_Spool_14,Aluminum,T-0742,1,140.140.235,0.75,4.5,0.25,1,1,0.3125
Threaded_Spool_15,Aluminum,T-0750,1,140.140.235,0.75,5,0.25,1,1,0.3125
Threaded_Spool_16,Aluminum,T-0752,1,140.140.235,0.75,5.5,0.25,1,1,0.3125
Threaded_Spool_17,Aluminum,T-0760,1,140.140.235,0.75,6,0.25,1,1,0.3125
Threaded Spool 2,Aluminum,T-1011,1,140.140.235,1,1.25,0.25,1.25,1.25,0.375
```

26. You can continue to add the other columns repeating the process laid out in the previous steps.

27. Choose the **Save** button from the lower right corner of the **Table** Editor to save the table.

28. Accept the default file name and location. It will place the table by default in the location of your file, so it would be best to save the file and the table to a network resource that others can access.

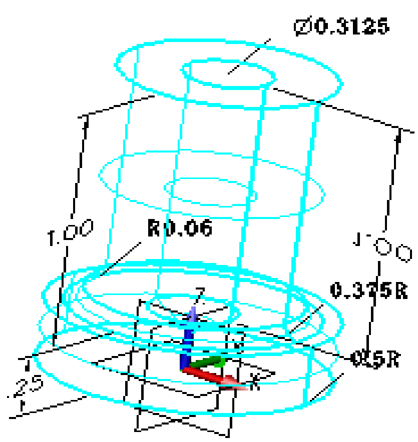
Now that the table is populated we need to look at the various ways of using the table to generate parts.

There are basically 2 places where you can create table parts.

- When creating a New Part/Assembly
- In an existing part / assembly.

29. Creating a new Library Part from the File **New** Menu

- Pick **File New**  Multi-object , Name it: Table_Parts_1.Z3.
- Pick the **Part Library** button. 
- Browse for the table file, open the file: Table_Parts
- Selece desired size as following…
  - Material：Aluminum
  - A:0.75
  - B:1
  - C:025
  - D:1
  - E:1



-

30. If you edit the part attributes of the Library Part you just created you will see that all of the Table entries are accurate including the part Number, so your BOM's will be accurate.

31. You can also create a Library Part right from the Object Manager of an existing file. This part object will be created in the current file.

    - From the Object Manager of an existing file choose the **Part Library** button.
    - You will see the same Library Form as the one that appeared in Step 35 on the previous page.
    - Fill it out and Pick OK and your new Library part will be listed in your objects manager. You can then use it like any other part object.

32. If you are in an existing assembly you can build a Library part that will be put in your current file's object list and you will also be prompted for insertion points and mating conditions as if you were adding the component.

    - Go into an assembly or edit any part object and Right-Click on a blank area of the screen. Pick Part Library from the shortcut menu.
    - You will see the same Library Form as the one that appeared in Step 35.
    - Fill it out and Pick OK and your new Library part will be added to your objects list and you will be prompted to locate your new library part into your current model or assembly.

33. Some customers are populating a single file with all of the library parts after they have created the table. When they need a Spool, for example, they open the Spool.Z3 file and pick the one that they want. Others are creating them on the fly. Since ZW3D likes unique name for files, if you create a duplicate, ZW3D will automatically find the one that was already created and put in into your assembly, provided that it can be found in your search path. (See the File pulldown)

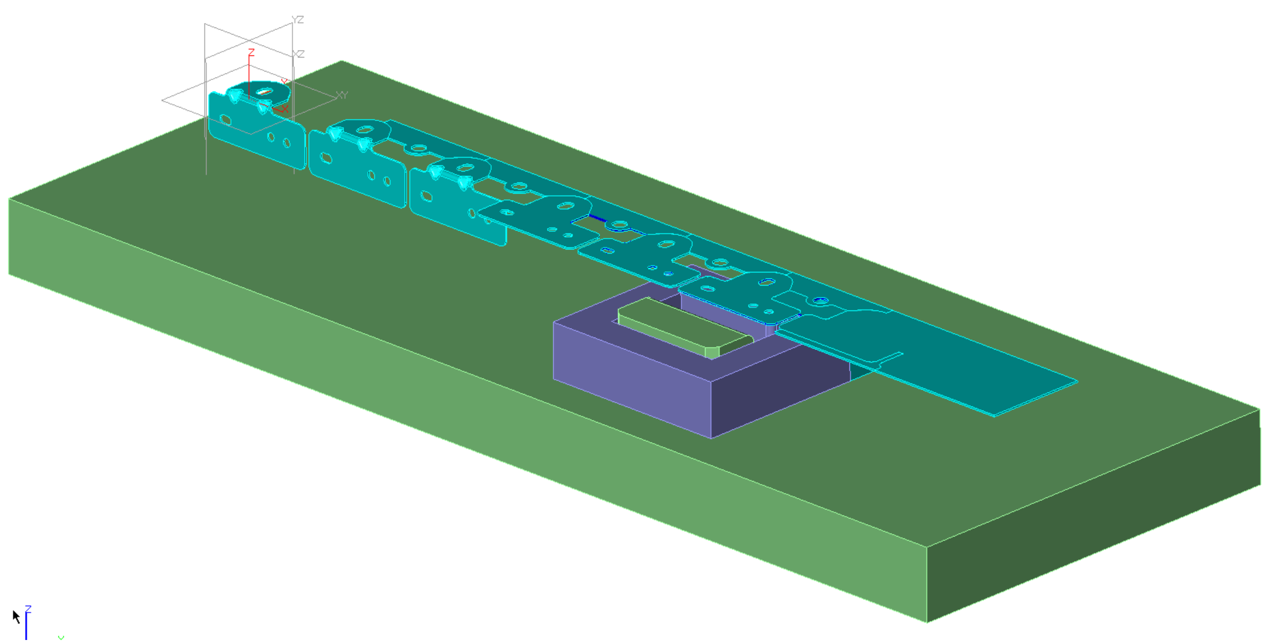
Appendix C - Die Subassembly

in the context of the Main Assembly

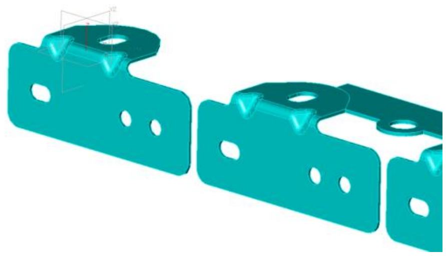
(Also See Chapter 5 for an alternate method)

This lab will focus on the creation of the Die Subassembly within the context of the main assembly. We will be implementing best practice methods for best maintaining associativity to the strip and the reference masters that we created in the two previous lessons.

In this lab you will create the Main Assembly and the Die Subassembly along with several of the die blocks. We will not create a complete subassembly, but we will give you a firm foundation to work from. The Die Subassembly is an object that will in itself not contain any sketches or shapes, but rather it will contain parts and perhaps other assemblies, much like the main assembly which will contain parts and subassemblies of parts.



Create a new object in the **Job_5361.Z3** file called **310_Main_Assembly_5361**.



1)

- **Insert Component** –> **002_Strip_5361**.
- **Anchor** it.
- The insertion point is **0,0,0**.

You should now see your strip component with the same relative position to Zero as it had when it was only a part.

NOTE: There are two ways to insert a new part/assembly object into this assembly that has not yet been created. The following is given as an overview for you to read thru, but not execute. We will use method 2 in these lessons.

- **Method 1 (quicker, cannot choose template)**
  - i. While in your Assembly object, choose Insert Component.
    1. Type in a new name for your part/asm object on the prompt line and hit enter.
    2. Anchor it.
    3. Insert it at 0,0,0
  - ii. You are automatically placed in this new part/asm for editing within the context of the assembly
- **Method 2 (longer, but you can choose a template)**
  - i. From the Object Manager create a new part object, choosing the template and the name for the part that you desire.
  - ii. We need to make this empty part in the context of the assembly, so you will need to navigate back to the object manager then into the Assembly where you can insert this empty part/asm as a component, anchoring it, and placing it at 0,0,0.
  - iii. You can then open the Assembly manager and double click on the component to edit it.

2) We will create the Die Subassembly Object using method 1 as described above.

- Right Click in the graphics area (be sure you have not selected anything) and choose **Insert Component**
- The prompt asks: Specify part to instance.
- Type in the new name: **400_Die_SubAsm_5361** and hit <ENTER>.
- Check the **Anchor** Component Box.
- then Specify **0,0,0** (comma comma) for the location and hit <ENTER>.

If you look at the title bar, you will see that ZW3D has placed you in the new object that you created in the previous step…

3) You can still see the strip from the main assembly. Pick the **show target** button from the Document Awared Toolbar to see what is only contained in the Die Sub Assembly.

4) Pick **Show All** to see the higher level assembly components.
NOTE: The Die_SubAsm object will only contain other parts, so we need to create other new parts in this assembly object using one of two methods as described on Page 106.

5) Next we will create a Die Block for the Pilot Hole.
REMEMBER: We positioned the bottom of our strip on the XY Datum, so the XY datum and any reference geometry from the bottom of the strip can be used to establish the top of our die steel.

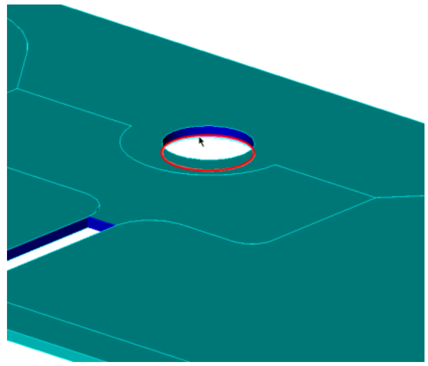
6)

- Right Click in the graphics area (be sure you have not selected anything) and choose **Insert Component**.
- Type in **402_DieBlock_5361** and Hit <ENTER>.
- Choose the **Anchor** option from the insert component options form.
- For the location type be in **0,0,0** for the location and Hit <ENTER>.

Take a look at your title bar and it should show that you are now editing 402_DieBlock_5361.

7) Choose **EDIT** pull-down menu **-COPY-Assembly Geometry**.
Choose the **bottom edge** of the first Pilot hole and Hit <ENTER>. See image at the right.

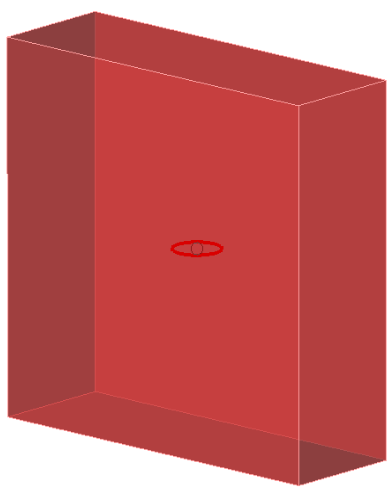
Remember set your filter to edge to make your selection.
NOTE: If it does not highlight, turn off shading.



8) Show Target. You will see a circle.

9) Create a Primitive Block as a base shape with the Block center at the center of the circle and the second point can be anywhere. Then specify the following in the block options form…

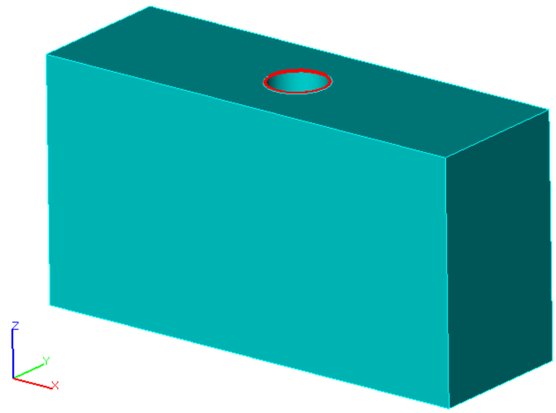
- X = 3.0
- Y = 1.0
- Z = -3 (our die steel will be 1.5 tall, so with a primitive block by center and corner -3 will put 1.5" below our XY Datum).



10) Next **trim** the new block with the **XY Plane**

11) And use the curve that we copied from the strip to **extrude remove** a hole in the block with an offset of your desired clearance. In this case I will use a .010 inch clearance.

Your final block should look like the following image…



12) Optionally, you can **erase the curve** that you used to create the extrude_cut.

The next component we will make is the die block for the punch notch.

The creation of this part will be similar to the last one, but we will use the master punch reference object we created earlier to establish the cut in the block.

13) From the Assembly Manager **right-click** on the **400_Die_SubAsm_5361** and pick **edit part.**
This will take us to the correct sub assembly.

14) Insert Component.
Type in **403_DieBlock_5361** and Hit <ENTER>
Choose the **Anchor** option from the insert component options form.
For the location type be in **0,0,0** for the location and Hit <ENTER>

15) Take a look at your title bar and it should show that you are now editing 403_DieBlock_5361.

16) Choose **EDIT-COPY-Assembly Geometry**. We are copying the same geometry as we did in Step 7 of this Lab.
Choose the **bottom edge** of the first Pilot hole and Hit <ENTER>.

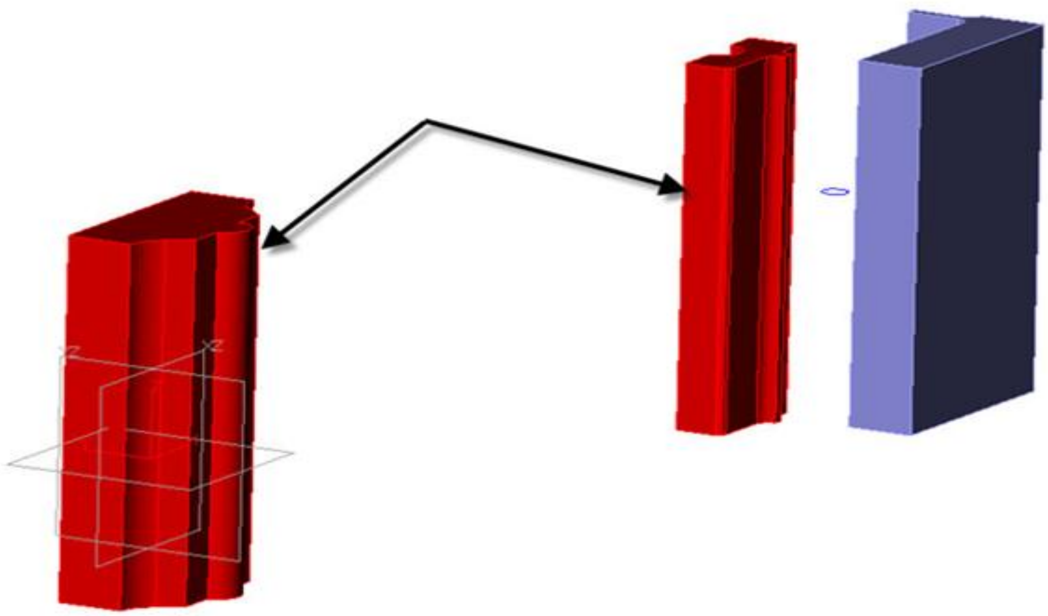
17) Show Target. You will see a circle

18) Choose **EDIT – >COPY –> External Part** from the Pull down menus and choose

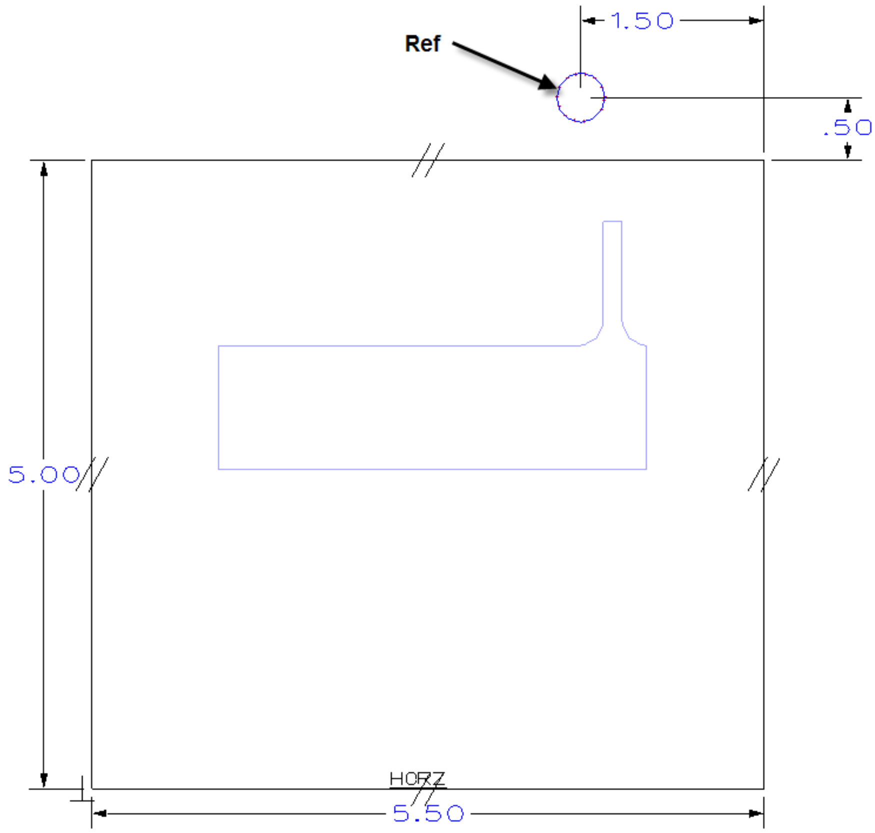
- Choose **003_Ref_Punch_Master_5361** from the object list.
- Type in **0,0,0** for the location.

See the image below for your results.

19) Erase the two shapes indicated in the image above.



20) Create a sketch on the XY datum that looks like the following image. I used 2 Point Rectangle to draw the rectangle avoiding references to the Zero Point. The Ref note on the image to help show which elements were referenced into the sketch from the Model.



We placed the .50 dimension because we remembered the size of the die steel used for the pilot hole. We could have also copied a face from the 402_DieBlock to constrain our sketch.

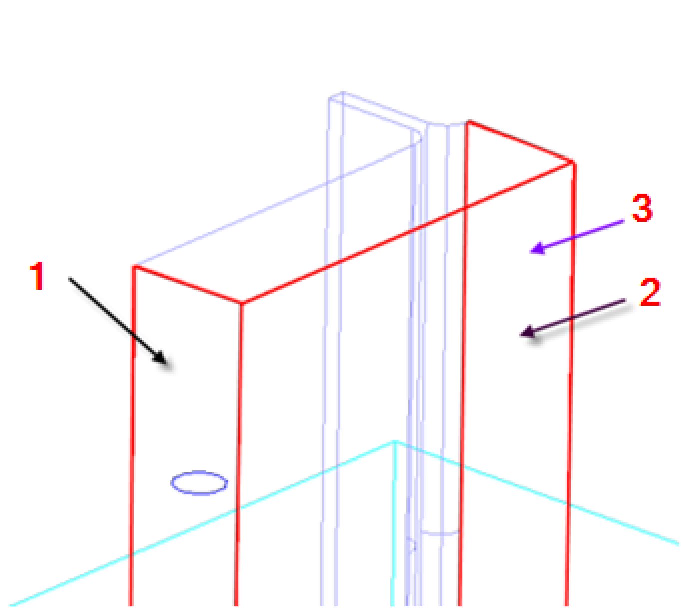
21) Extrude the sketch you just created as a base shape.

- Start is 0
- End -1.5

22) Use the **volume offset**  command found on the shape Tab to increase the size of the remaining shape from the reference master punch by the clearance needed to cut the die block. Use **.01**, so that you can see the change.

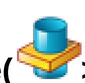
- Select the punch shape.
- Specify a .01 offset.

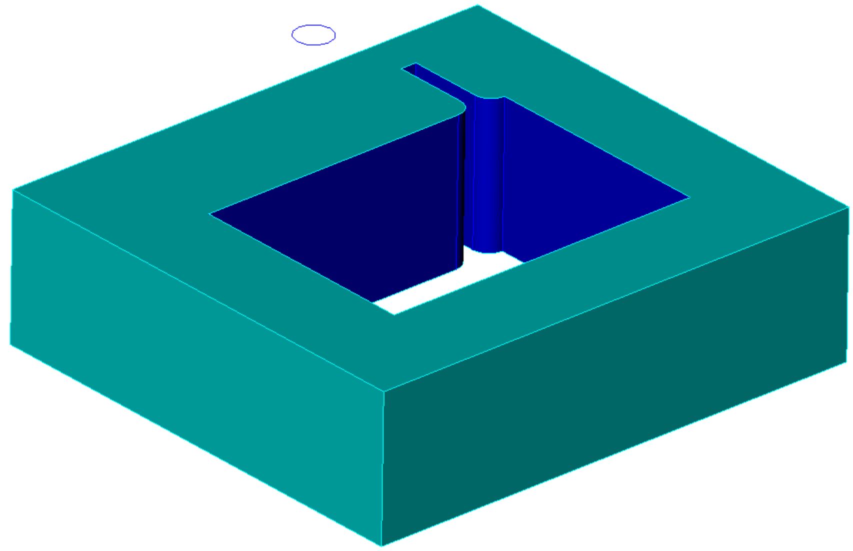
- Be sure to uncheck the Keep Original Option and do not specify openings.
- Then choose the **Advanced Tab** and pick the **Add Offset button**.
- Specify a Zero Offset on the Back(2), Left(1) and Right(3) faces indicated in the image below. The command only allows you to pick one face at a time, so you will have to Add Offset 3 times.



You should see the master punch take on the default color and increase in size slightly

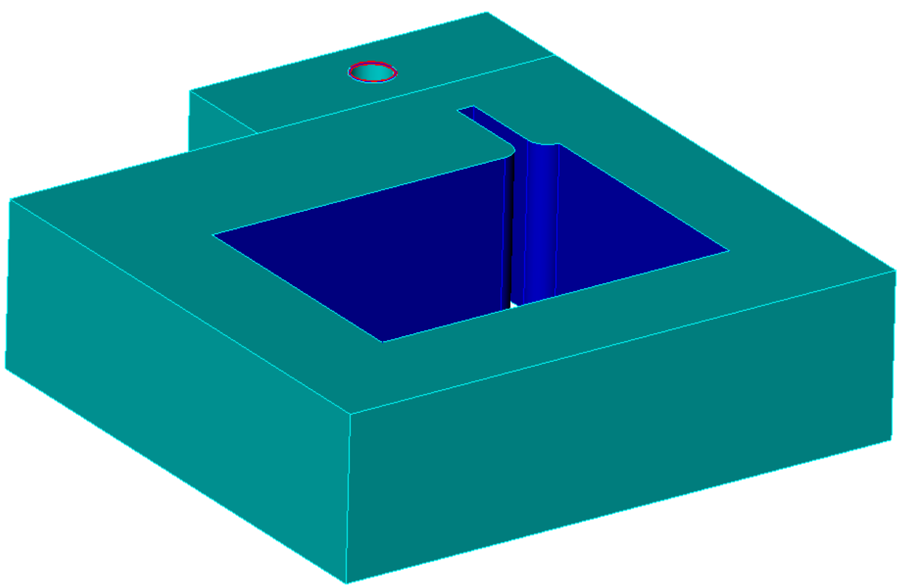
We can also use this shape to make room for the retainer. Simply **offset the back face** of the ref punch geometry by 1.5 inches.

24) Finally **remove(** **>**  **)** the Ref Punch Master shape from the Extruded block we just created and your final result will look like the following image…



25) Exit part to navigate back to the Die Subassembly

26) Your Die Subassembly should look like the following image…



27) Let's create one more part in our Die Subassembly – the Heel Insert

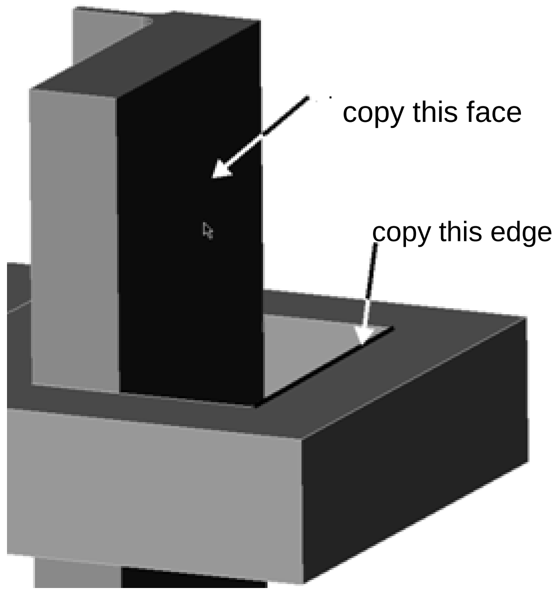
28) Make certain that you are at the Die Subassembly by reading your ZW3D Titlebar

29) Insert a new component named, **404_Heel_Insert_5361**. (104_Punch_Retainer)

- Anchor it.
- Locate it at 0,0,0.

30) Insert as a component the **003_Ref_Punch_Master_5361** Part
- Anchor it
- Locate it at 0,0,0.

31) Using **Edit -> Copy -> Assembly Geometry**, Copy the back face from the shape for the pitch notch punch. Also copy the back edge of the cutout from the 402_DieBlock.



32) Use the **Erase** Command to erase the 003_Ref_Punch_Master that you just inserted.

33) Show Target to see only the edge (curve) and the face.

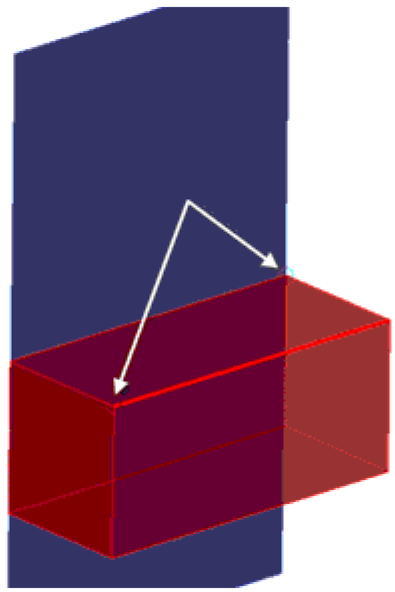
34) In order to establish corner points for an opposite corner rectangle we will use the **Project Curve** command from the Wireframe Tab to project the edge curve onto the face.

- Select the curve when prompted to select curves to project.
- Select the face when prompted for the face to receive the curve.
- The default direction will work in this case or you could optionally specify the Y Axis.

- Pick **OK**.
- You should see a new curve projected onto the face.

35) Create a primitive Block by opposite corners and specify a Z value of -1.5 inches. The corners should be at opposite ends of each curve.

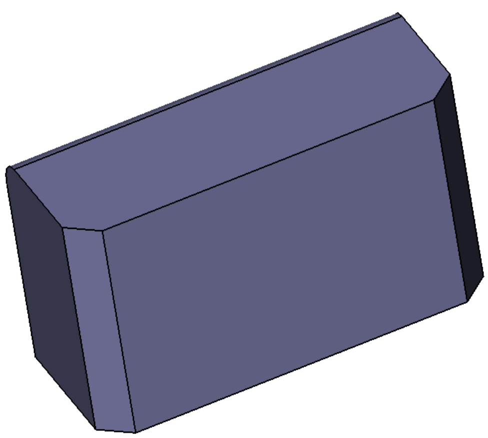
Pick the opposite corners as indicated below.

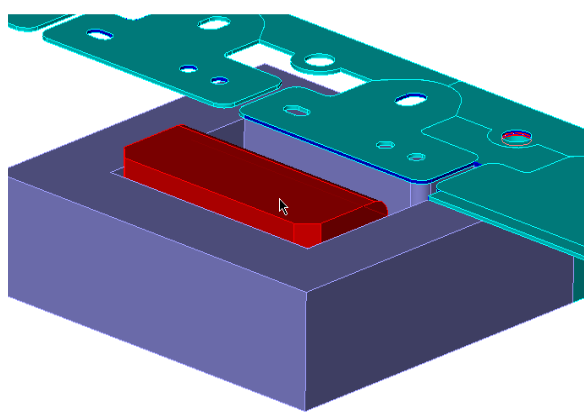


36) Then offset the top face by ¼ inch. And add Chamfers and fillets.

37) Erase the face and curve.

38) Your final part should look like the following images…





39) The last part we will create is the Die Plate.

40) Insert a new component named, **401_Die_Plate_5361**.

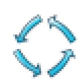
- Anchor it.
- Locate it at 0,0,0.

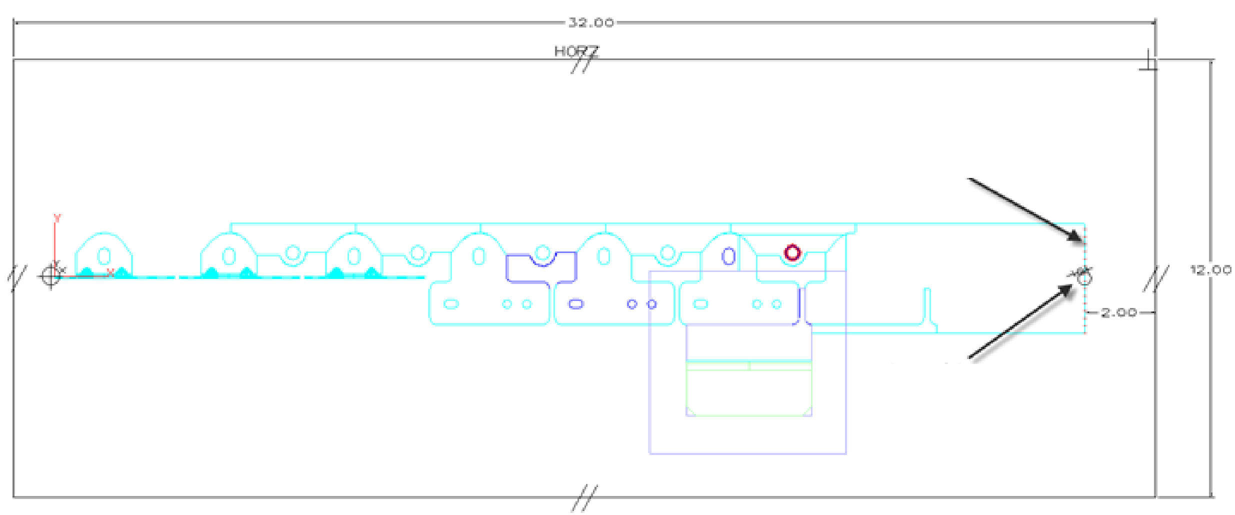
41) Insert as a component the **003_Ref_Punch_Master_5361** Part

- Anchor it.
- Locate it at 0,0,0.

42) Create a Datum parallel to XY that is 1.5 inches in the negative Z direction.

43) Create a sketch of a rectangle on this new datum that is centered on the strip. This sketch will be the perimeter of the Main Die Plate. The image below contains the sketch you should create.

- Constrain the Point to the midpoint of the vertical reference line.
- Midpoint Constrain the right hand vertical object line to the point. Be sure to only constrain on the Y Axis.
- Pick the **Solve current sketch automatically** button  to see the constraints applied.
- Change dimensions as shown below.



44) Extrude the rectangle 2 inches in the negative Z direction.

45) Erase the components that were inserted with the Punch Master.

46) Your Die Subassembly should appear as follows. (You may need to navigate back to parent object or pick the Show All button to see the entire subassembly.