Part Management

This document has been provided to describe in detail the new unfolding routines and interfaces for VISI-Progress that have been introduced from version 11.2 onwards.

File > Open > PDemo2.wkf

Progress > Part Study

Selecting the ‘Part Study’ menu will now display the new Progress interface.
Progress Part Manager Icon Description:

Please take some time to study the following interface and icon descriptions for each phase of the part management.

Part Analysis Menu
1. Part Analysis Menu
2. Part Analysis Build
3. Part Analysis Interactive Build
4. Show Part Analysis Tolerances

Unfolding Menu
1. Unfolding
2. Automatic Unfolding Optimised
3. Automatic Unfolding On Tree
4. Manual Unfolding
5. Delete one un-folded surface
6. Delete all un-folded surfaces

Step By Step Menu
1. Step by Step
2. Add a Step
3. Delete a Step
4. Select Step Number
5. Step Forward / Backwards
6. Part Study Direction
7. Rebuild 3D Parts

Solids
1. Rebuild 3D Parts
Performing the Part Analysis

From the Progress Part Manager, ensure the ‘Part Analysis’ icon is activated and select the Part Analysis Build icon.

1. Part Analysis Build

Select the part to analyse: Select part 1 (on Top Face).

Select the Face linked with the Origin part: Select Face 1 (Top Face).

Select the origin point: Select the TOP centre point of the circle.

Selecting a face from a Solid, the system loads the Extract Skin function, showing this mask:

Here is possible set different options to obtain the skin of the Part, as create a Copy, Delete or not the original part, Search or not the Sharp Edges and then the distance between the two parts.
Keeping the default settings, the system extracts the selected skin, asking to select the face linked with origin point and then the Reference Point:

The selection of the origin point is very important because this point will be used as reference point to following operations.

After the point selection the system shows the mask where set some parameters as Material, Thickness and Neutral Fibre:

Selecting a surface the system asks to set the Material Side, but after that the procedure is exactly the same:
Neutral Fibre Definition

During the Part selection the following dialogue box will be displayed.

Within this dialogue box it is possible to define the neutral fibre either as a constant value or using one of two automatic neutral fibre algorithms:

(i) ROMANOSKI
(ii) OEHLER-KAISER

It is also possible to define which unfolding algorithm is to be used for the step-by-step unfolding.

Compute the Neutral fibre
Follow the next steps to calculate the neutral fibre for the example component.

Select a Neutral Fibre value of 1/3
Displaying and changing Neutral Fibre values for Bends
At this stage the tree will again show all the bending stages, but also the neutral fibre at each bend. Selecting a bend using the LHM button it is possible to change the neutral fibre for a single bend using the bend property window. It is often necessary to use a different neutral fibre when there is a big difference between the bend radii on the part.

Note: - In our example we will not be modifying any of the values that appear.
**Note** :- The previous operation can also be performed by right clicking the bend in the graphics area.

**Dictating the Unfolding Type**

When performing the step by step unfolding it is possible to choose how the bend is developed, there are actually three types that can be selected with the icon shown below. When each one is selected the result will be updated dynamically on the screen.

The following example will explain the differences between the various choices using our example part.

**Constant Length Unfolding:**

Constant length unfolding uses an unfolding algorithm that forces the length of the radius to always be constant. This means that when a bend is folded, the radius and the centre point must change to maintain the total length.
Looking at the 45 degree bend from our example this is what you should expect to see.

**Constant Radius Unfolding:**

Constant radius unfolding uses an unfolding algorithm that forces the radius and the bend centre points to be constant. This will shorten the overall length or the radius and therefore to obtain the correct length an extra flat surface is added to the connected planar face.

**Note** - If constant Radius Unfolding is used it is now possible to apply the flat portion of developed length before or after the bend radii by toggling the respective icon that appears.

Try selecting **Constant Radius, Extension After The Bend**
Now lets try the option **Constant Radius, Extension Before The Bend**

**Extra Flat Surface Added Before the Bend**

**Note:** Having explored the various options please ensure that Step 2 is reset back to the default option, Constant Length Unfolding

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**Set Part Analysis**

After selecting the part to study the result of the analysis will be shown in the main window and the unfolding status will be shown in the Progress tree structure.

Any face/bend selected on the tree will be displayed in the main window.
In order to have a better tree organization we can make some different operations:

1. If you want you can move the branches on tree: select the part and move it on tree, keeping pressed the left key’s mouse.
2. You can select the second icon **Part Analysis Interactive Build**: this function allows to build the tree manually; the tree will be composed, hand by hand, after each face selection.
3. Using the icon **Group Analyzed Faces**, the system permits to group some faces in a unique feature:

If working with an imported solid model, it is very important to choose the right side (internal or external) because the system only works on the surfaces extracted by the part analysis. This could lead to later "non constant thickness" or "bend radius" problems.
Just to know it’s possible also manage the parts of the model in another way as explained below:

**Showing and Hiding Faces / Bends**

It is now also possible to simplify the display of the Part Analysis in the graphics area by hiding and un-hiding the nodes in the tree, this is also useful if you require to remove features unsuitable for the unfolding process.

In our example it would be desirable to disable the formed part of the component, as this is not suitable for the unfolding routines of VISI-Progress. The next sequence will show you how. **Please follow along with the example below:**

This formed feature is not suitable for unfolding using VISI-Progress

**Identifying the position of the formed feature in the tree**

Before we can disable the relevant nodes in the part tree we need to establish which ones they are:

Pick the planar face as shown opposite, this is the parent face to which the form feature is connected.

Plane 8 is highlighted on the tree and connected below this you can see **Bends 10 – 13** that form the outer ring of the formed feature.
Expanding the tree hierarchy
In our example we can see that Bends 10-13 also contain child nodes denoted by the + symbol, these can all be expanded in one click:

Press the right hand mouse button on Plane 8 and select the option Expand All. This shows all the nodes that make up the complete formed feature.

Disabling the nodes of the form feature
Uncheck bends 10, 11, 12 and 13 from the tree, each respective child node will be unchecked automatically.

Form feature removed from the part analysis display.

Extra Information
From within the ‘Part Analysis’ tree, it is possible to drag and drop the positions of the bends. This is very important as this can affect the result of the automatic unfolding. On previous versions of VISI-Progress, it was possible to use ‘Semi-Automatic Unfolding’ to unfold a bend and select an edge as a reference for the unfolding. This option is no longer necessary as the unfolding path can be defined by moving the faces/bends inside the tree. When dragging a face/bend from one position to another, it can only be positioned on a face that shares a common edge.

Selecting the Part using the RHM button will display another menu that will allow the user to change the part thickness, change the origin position or reset all the part attributes.
Unfolding the Part

After viewing the neutral fibre settings from the Progress Part Manager, select the ‘Unfolding’ icon from the top row of icons.

Now take a look at the progress unfolding tree on the right hand side

After the Neutral Fibre definition we need to create the unfolded surface; we have three methods:

- Automatic Unfolding Optimize
- Automatic Unfolding On Tree
- Manual Unfolding

Clicking on the first icon **Automatic Unfolding Optimize**, the system creates the blank surface in automatic way.

The planar parts computation it's completely automatic; the algorithm starts to unfold all the face in a radial way. Starting from that reference face and gradually working outwards.

The user does not have the possibility to control the ‘Automatic unfolding’. The system decides the best way to unfold the part is.

Clicking the second icon **Automatic Unfolding on Tree**, the system creates the blank surface following the surfaces sequence defined on tree.
The third choice is the **Manual Unfolding**: the user can set the unfolding sequence selecting one by one the surface on the model.

In some case the system needs to know the reference edge for the next surface and so the user must be select it.

At the end of these processes, the result is a flattened surface model suitable for the step and strip definition.

As you can see on the screen, when the user has defined some surfaces as Feature, the system creates automatically a surface to close the hole.
The system sets as “Automatic Unfolding” all the surfaces of the Part, but you can change it:

1. Clicking the right key’s mouse, the system shows a mask with Unfolding Surface Menu.

   **Set to Manual:** You can set each part to Manual Unfolding and, during the unfolding process, automatically, when the system arrives to this surface, it asks you the reference edge.

2. You can change also the unfolding types on property tree.

   If you set to Manual Unfolding a part on the node, the system sets automatically all part under the node.

   It’s also possible disable some surfaces from the unfolding process; to do that is necessary de-check the relative node on the tree.

   In case of some errors occurs during the unfolding process, the system gives the possibility to delete a single surface or all the unfolded part.

   Selecting **Delete One Unfolded Surface** the system asks to pick the unfolded surface to delete, updating the Part and the tree.
Delete All Unfolded Surfaces the system delete the unfolded surface, and on tree you can see (to be rebuilt).

In case the user does some error during Cad operation, selecting the last icon Exec Auto Rebuild, the system recreates the defined unfolded surface.

If you have deleted the parts of the model, you have this result:

As you can see from the tree shown opposite the selected bends that make up the formed feature are already disabled from the unfolding sequence. They are displayed in Red to show they are disabled.

It is possible to enable and disable any feature from the unfolding sequence using the unfolding properties panel.

If all the stages have been followed correctly the flat blank should look like the one displayed below:

The hole here represents the faces that were disabled before the automatic unfolding. This will not affect the step-by-step unfolding.
Unfolding the Part Step by Step

After creating the flat blank, select the ‘Step by Step’ icon from the Progress Part Manager.

The ‘Step by Step’ unfolding tree will now display each of the bend stages and the relevant bend angles at each stage.

Before any of the bends can be unfolded it is necessary to create a new stage.

Select the ‘Add a step’ icon as shown below.

NOTE: It is recommended that you always start by adding two stages. The reason is to start the unfolding process on stage 2 and not stage 1. By leaving stage 1 blank, it will always be possible to add a new stage before the start of the unfolding process.

Select the ‘Add a step’ icon twice. Step 1 will be used as a blank stage and step 2 will be used as the first unfolding stage.

Stages as shown on screen
The Unfolding Sequence

Make sure you are at step 2 and select the **Bend 4** from the tree view. This will reveal the Bend Properties panel. The bend is currently set at **90°**.

Now change the bend angle from **90°** to **45°**.

The unfolding sequence will be shown dynamically in the main screen interface.

**Bend 4** should now be coloured blue to indicate it has been modified and its value should now be **45°**.
Note: It is also possible to change the bend angles of any bend by using the RHM button on either the bend on the model or on the relevant node in the tree. From the context sensitive menu that appears, just select ‘change unfold angle value’.

You can try any of the methods discussed for the rest of the bending sequence.

From Step 2 Select Bend 3 and change the bend angle from 90º to 45º. This completes the bending stages for **step 2**.

Select the ‘Add a step’ icon to move to **step 3**.
From **Step 3** select **Bend 4** and change the bend angle from $45^\circ$ to $0^\circ$.

Select **Bend 3** and change the bend angle from $45^\circ$ to $0^\circ$. 
Select **Bend 9** and change the bend angle from **90°** to **0°**.

This completes the bending stages for **step 3**. Select the ‘**Add a step**’ icon to move to **step 4**

From **Step 4** Select **Bend 2** and **Bend 5**, then change the bend angle from **15°** to **0°**. Use the **CTRL** key to select both bends simultaneously.

It is also possible to adjust the bend value to account for any ‘spring back’. All bends can be adjusted within any stage of the sequence, even bends that have been set to 0 degrees.

This completes all of the bending stages for our unfolding sequence. All the unfolding stages are performed on the neutral fibre surfaces. The solid model of each unfolding stage is not automatically created. The reason is that it is possible to create many unfolding scenarios, and only build the solid model of the stages required.
You should now have your original part, plus five stages numbered 0 – 4 and a flat development.

Let's continue and create some solid models from the bend stages we have made.
Continued overleaf.

**Reversing the Part Study direction**

By default you can see that all the bend stages are created from left to right, starting with the final part, progressing in sequence through the bend stages created by the designer. It is possible if preferred to make the parts run left to right but with the finished part as the last stage.

The parts will now dynamically transfer to the opposite side. This operation can be performed at any stage of the unfolding process.
Creation Of Solid Models From Bend Stages

Now we have created our desirable unfolding sequence for the progression it is time to convert the stages into solid model form.

The functions that are available on this page are:
  - Rebuild 3D Parts
  - View Mode
  - Select Some Part Entity
  - Exec Auto-Rebuild

Clicking the icon **Rebuild 3D Parts** the system creates the solid part for each defined step.
Selecting the icon **Select some Part Entity**, the system gives the possibility to select a 3D Part created, showing the relative menu:

Here it’s possible **Detach the Part** or **Change Origin** point.

After the Part detaching the user can edit the solid using the Cad commands and then reattach it as Interactive Part.

Clicking M2 on the relative step on tree:

- **Link Auto-Build Part**: Rebuild the solid automatically. (The system shows the solid, if you want see the surfaces you must select the Step page.)
- **Link Interactive Part**: If you want remodel the solid; you can rebuild the solid manually.
- **Change Origin**: You can change the Origin (Reference Point).

It’s important note that in some cases the 3D Solid Part creation could fail, and so in this cases it’s necessary modify the part using the cad commands and then reattach it as Interactive Part.
Moreover to each step it’s possible define some parameters to the solid part:

- Part Name
- Origin Point
- Part Colour
- Part Transparency

Rebuilt the solid is very important for the following operation to pass the solid part on the strip.

**Adding an Extra Stage into the Step by Step Sequence**

We have now constructed a solid model for each of the unfolding stages. However, if we view the solid stages from the right, it is possible to see that one of the bending stages is too high above the top of the die block. This means that to use these stages it would be necessary to have a cavity in the stripper plate during the lifting stages.

Even at this stage, it is possible to add a new stage into the unfolding process.

**NOTE:** To reactivate the complete VISI-Progress unfolding process, it is only necessary to select ‘Part Study’. All of the unfolding stages will automatically be re-calculated.
From the ‘Step by Step’ unfolding menu select stage 2 from the pull down menu and then select the ‘Add a step’.

This will create a new stage 3 and all of the stages will automatically be shifted one stage.

Select Bend 3 and change the bend angle from 45º to 60º.

Selecting again the Solids page, the system shows that some rebuilt solids are inconsistent, and so needs to select the Exec Auto Rebuild icon to have the new correct result.
**NOTE:** We have now inserted an extra stage, so it is possible to decide whether to use either the stage with the 45° bend or the 60° bending stage. If the 45° stage was definitely not to be used, it wasn’t necessary to create an extra stage, it would have been possible just to edit the bend value and re-build the solid part.

If all the unfolding sequences have been followed correctly, you should have something similar to the bending stages shown above.

Congratulations you have now completed the basic automatic and step by step unfolding tutorial.
VISI Progress
Strip Manager
Part Management

This document has been provided to describe in detail the new strip management routines and interfaces for VISI-Progress that have been introduced from version 11.x onwards.

File > Open > Progress Parts Unfolded.wkf

Progress > Strip Study

Selecting the strip study menu and mask will display on the right of screen

Progress Strip Manager Icon Description:

Please take some time to study the following interface and icon descriptions for each phase of the part management.

Strip Manager Menu

1. Load one strip from Database.
2. Add Internal Punches
3. Add Punches
4. Create Maximum Sheared Punch
5. Cut a Punch
6. Notches
7. Bend Punches
8. Attach Parts to Steps
9. Rebuild the Strip
10. Cfg Editor
11. Exec Auto Rebuild
12. Create Report
13. Automatic Dynamic Strip

Creating your first strip
From the Strip Manager select the Load Strip Icon. You will be prompted to select the unfolded part. Then select an origin point on the unfolded part. You are then able to pick an application point on the screen. Place the part at \( X_0, Y-400, Z_0 \).

Hit the “ESC” once to initiate your strip definition. You will see 2 unfolded parts on the screen side by side to help define the position your parts relative to each other.

Also the dialogue on the right will change to show the Strip Definition Parameters. See overleaf.

Please Enter the parameters as shown in the panel below

**Parts Parameters**
These parameters are for reference only
- **Strip Thickness**: has been taken from the initial part analysis
- **Material**: is taken from the initial part analysis
- **Specific Weight**: is taken from the initial part analysis
- **Breaking Load**: is taken from the initial part analysis

**Strip Parameters**.
These parameters are used when the check icon is used.
- **Strip Name**: can be set by the user
- **Steps number**: is the number of steps you wish to start with
- **Strip to right**: check box turned off will build the strip to the left
- **Length Before Origin**: allows the user to determine extra strip length before the part

**Step Data**
These parameters are used to visualize the layout of the parts in relationship to each other and the strip on the screen

- **Steps Number for Study**: allows the user to set the number of steps shown on the screen to help determine the various parameters
- **Step**: is a distance to set the progression of the strip
- **Strip Width**: determines the width of the strip
- **Upper Discard**: is used to set the distance above the part to the upper side of the strip
- **Lower Discard**: is used to set the distance below the part to the lower side of the strip
- **Minimum Distance Between Parts**: sets the smallest distance allowed between parts
- **Rotation Angle**: is used to rotate the parts on the strip to help while nesting parts of unusual shapes allowing possible overlaps to reduce progression
- **Step Origin**: display the current origin and allow the user to change its location

**Parts in Step**

Selecting two parts to manage in the strip, the system abilities a property tree on the mask to help the user to obtain a good positioning of the parts:

If, during the moving of the parts, they are in collision, the system shows this mask:

Starting from the unfolded part you can select the part and then the position; a menu helps you to insert the part as you want:

It’s possible **Mirror the Part** along the selected Axis, or **Rotate the Part**, or to **Align** it. It is possible makes some modification on property tree to obtain the desired result: start modifying the Step and the Strip Width.
Then we can start to move the parts using the option:

Setting the Minimum Distance value and moving the part on the left-right, the system automatically finds the best position of the parts, changing the up-down value.

It is also possible set the optimisation side; in case you have Up-Down, the system checks the minimum distance moves the parts in that direction, otherwise in the other direction.

We can manage the rotation angle of the parts in three different ways:

- **Free Angle**: you can choose two completely different angles for the parts:
- **Same Angle**: the system sets the same angle for the parts:
- **Opposite Angle**: the system sets the opposite angle for the parts:

Another option that can help the user to position the parts is the possibility to change the reference point of the parts that are showed on the screen with blue circle.

**Calculated Parameters**

These parameters are for information only

- **Parts Weight**: displays the weight of the flat blank
- **Discard Weight**: displays the weight of the waste per progression
- **Discard Percentage**: displays the percentage of waste per progression

When you have already created a strip, the system saves it in a database with the strip name that you have given before.
You can choose the strip from the list and the system automatically shows all the trees with parts and punches.

By selecting this icon you can also create a new strip: you can select the Blanked Part, an origin point on the unfolded part and then to pick an application point on the screen.
Selecting the relative icon, the system loads the relative mask; setting some parameters as input, the system calculates all the possible positions of the part, showing the grid with the results ordered in base of minimum discard %, and clicking on each row you can see the preview result. Clicking on Apply, the system will use the selected parameters to define the strip.

As you can see the system loads, as default, the parameters previous set:

- Upper Discard
- Lower Discard
- Minimum Distance Between Parts

You can use them or modify with new values. The Angular Precision value is referred to the nesting calculation as means the degree value that the system considers to calculate the better position; bigger value means minor precision, but faster calculation.

Clicking on first icon “Recalculate Nesting”, the system starts the calculation showing the relative results on the right grid; as explained before selecting a result the system shows the preview on screen.

There is a further option that is the Fastening: enabling this option the system makes the calculation considering just the case where the set option is possible.

Selecting an Upper Fastening as Option the system needs to know the Faces that you want consider for the fastening (select Pick Fastening icon and then the face), and also the Fastening Width value.

When you have the desired result you must click on Apply icon to have the selected setting on Strip management.

The nesting management is available also in case of Double Parts on Strip.

If you don’t want consider the Nesting calculation, you can close just the mask.

If the calculation is too long, just keeping pressed Esc button, the system stops the calculation showing this mask:

Experiment with the Step data parameters to see the effect on the 2 pieces shown on the screen, and then input...
When you are ready to move on you can use left mouse button (LMB) on the “Tick” to accept your settings.

The screen display will now change to show all 5 steps with the 2D Geometry displayed and a new window on the right will appear showing a tree configuration of your strip.

Note: - To go back to the strip configuration menu you can LMB on the “Load One Strip from Database” Icon and change any parameters you wish.

Change the Steps Number
By clicking on the strip name at the head of the tree you can reveal the strip parameters panel in the lower half of the interface.

Click LMB at the top of the tree on the strip name.
Click into the box next to Steps Number and adjust the number to 10 steps.
Punch Creation

Creating the Piercing Punches

Change to the Isometric view (F4) and zoom in to see steps 1 and 2 visible on the screen.

Left click on the “Add Internal Punches” icon to add Piercing punches. You will see 4 starter solids for the internal punches. The oval shaped punch is not needed and will have to be removed.

In order to remove this punch there are 2 methods available.

- You can find the punch in the tree at step one by clicking the LMB on each punch which in turn will highlight the punch graphically on the screen. Once you find the oval punch (3), use the Right Mouse Button (RMB) and select Delete Punch. Be aware that this method only removes it from the tree and will not remove the punch from the screen. This can be used to temporarily remove a punch from the tree.

  OR

- You can totally remove the punch from the graphics area using the regular delete icon. **For this tutorial this is the preferred method as this punch is not necessary for the strip design.**

Now we are ready to place these punches in the desired positions on the strip.

Using the LMB drag and drop the punches from step 1 to step 3. All of the punches will now move down the strip to the new step in the tree and will be displayed in the new position on the screen.

For the remainder of the punches we will use different functions.
Create Maximum Sheared Punch

The purpose of this function is to create as quickly as possible all the necessary punches to crop the component shape from the strip. Initially the system creates one large punch of the total sheared area, this will then need cutting and refining to make several cropping punches. The following chapter is a basic example of how it works:

Select “Create Maximum Shear Punch”. Using the LMB to execute the command.

Use the values shown below to define your punch and click OK.

- **Step**: refers to the step location on the strip to place the punch.
- **Rotation Angle**: allows you to rotate the punch in the event that you have parts rotated during the layout of the strip.
- **Shift Value**: allows you to move the punch left or right to align with the step.
- **Upper Gap**: gives you the ability to add extra material to the top of the punch allowing it to be outside the strip.
- **Lower Gap**: gives you the ability to add extra material to the top of the punch allowing it to be outside the strip.
- **Gap in Strip Direction**: will add material to the left of the punch to create an overlap.
- **Gap in Opposite Strip Direction**: will add material to the right of the punch to create an overlap.

The Result

If you have followed the steps correctly you should end up with a single punch at stage 4 that resembles the punch opposite.

The next steps will show you how to cut this single punch into multiple punches.
Cutting the Punches

Select the “Cut Punch” icon from the main Progress toolbar to begin separating the punch into smaller ones.

You will be prompted to pick a punch. Select the new punch.

Then select a point using the Parametric point Icon.

Next select the cutting plane normal direction to the ‘Y’ axis.

A graphic representation of the location of the overlap will be shown. Set values in the “Cut a Punch” Dialogue as shown below and click OK.

Overlap area between the newly created punches shown graphically. Select ‘OK’ to accept these settings.
Repeat the process again except using the cutting point as shown below.

Select the point on the edge as shown using parametric point input and use the settings shown opposite.

Again, cut with a normal direction using the ‘Y’ axis.

Now drag the “U-Shaped” punch (2) to step 5.

U shaped punch (number 2) shown in position at Step 5 after dragging.
Your tree should now look like the one on the right. We are going to introduce another method of creating punches for the last 3 punches. First of all delete punch 1 in step 5 using the method we used earlier for the oval punch.

Now using the “Set Drawing Filters” icon, turn on the layer “Punch Profiles”. This will display some predefined profiles in new punch locations.

Click on the “Add punches” icon and when prompted select the profiles 1, 2 and 3 as shown below. After selecting the last profile hit ESC to exit the command.

Now move Punch 1 & 2 from step 5 to step 9.
Move punch 1 from step 3 to step 4.

This is how your punches should be arranged after the dragging and dropping has taken place.

Performing a Shearing Check

It is possible for the designer to check that the cutting punch arrangement will produce the required blank before proceeding to the next stage.

Pick the head of the tree and press the RMB

Now click the option ‘Show Shearing Check’ the left mouse button

The shearing check solid is now displayed underneath the solid strip.

You can delete this afterwards
Strip Simulation

Now it is time to run the strip through the punches and view the results. Using the “Rebuild the Strip” icon brings up the “Strip” dialogue.

1. **Restart**: will rewind the strip to the beginning
2. **Automatic**: will advance the strip through the punches and then display
3. **Strip Simulation**: will advance the strip through the punches and display after each step
4. **Back one Step**: will move the strip back one progression
5. **Advance a Step**: will move the strip forward one progression
6. **Only Sheared Strip**: will advance the strip without showing the bend stages
7. **Check Problems**: will display any problem areas
8. **OK**: Confirms your action, and closes the dialogue
9. **Build not unite strip**: The resulting strip is not united together but is made as a un-united solid group. Not uniting a strip is a good choice for complex strips.

Using the Strip simulation icon move the strip through the part to see a 3D representation of the strip. Rewind the strip to the beginning and click on OK.

Attaching Parts to Strip

There are 2 methods available to attach unfolded parts to a strip.

1st Method of Attachment

Select the Part branch of the Step you want to start with and RMB to display the “Parts Menu”. Select “Attach a Part” to show the Part selection sub-menu and pick from the list available. This will then attach the selected unfolded part on your strip and to all subsequent steps until the end of the strip. **Let's try this method now**: -
• **Select Step 2 > Part1** and attach "**Part Study – step:4**"
• **Select Step 6 > Part1** and attach "**Part Study – step:3**"
• **Select Step 7 > Part1** and attach "**Part Study – step:2**"

**2nd Method of Attachment**

Now for the other method use the "Attach Parts to Steps" icon from the head of the Strip Manager.

You will be prompted to pick the unfolded part from the screen then a menu will appear asking for the step to apply the part.

Select **Part Study – Step 1** from the screen area.
Now we will apply **Part Study – Step 1** to **Step n:8** in the strip using the drop down menu as shown below.

Another possible option on this menu is select: **Drag To Step**.
This function gives the possibility to select the solid and drag it in the relative step; it’s necessary just pick a point included in the step, and the system automatically attach the selected part to the strip, considering the set Origin Point.

You should notice that the from Step n8 through to Step n10 our Part Study step1 has been applied to these positions automatically.

Now go back to the Rebuild the Strip and Select the Strip Simulation Icon from the Dialogue to see your strip build with all the unfolded parts in place. Your finished strip Should look like this.

Congratulations you have now completed the basic Strip Management tutorial
Moreover we can explain in detail other options that you find on Strip Study mask:

**Notches**

In some cases on the punches is necessary add the Notch; to do that, clicking on the relative icon, the system asks to select the edge and the relative face, where you want create the Notch, and shows the mask on screen:

Three notch types are available and you can select which one from the first list and, on the right of the mask, you see the picture type.

In reference to type selected the system gives the possibility to set different values; the preview is available and so after any modification you can see the real effect on screen.

If the selected face isn’t planar you can set the extrusion direction of your notches clicking on "Pick Direction" icon.

The tree types available are:

1. **Rectangular**
   
   You can set the Width and the Length, and also if you need the Distance value, that means the distance from the selected edge.

2. **Cylindrical**
   
   You can set the Width, the Length, and also the two different Blends.
3. Inclined

You can set the Length, the two different Blends and the relative Angle.

In case the selected surface isn’t planar, the system gives you the possibility to select the reference direction, clicking on the relative icon.

Clicking on OK the system will confirm the Notch creation, and so the Sheared Strip will be updated using the new Punch geometry.

Bend Punches

Selecting the relative icon on Strip mask, the system asks to select the faces of the solid that you want to use to create the Bend Punch.

During the faces selection, the system checks if is possible unite the selected faces; if no possible shows an advice on the screen.

Selecting the contiguous faces the system will show the mask, that allows to see the Bend Punch preview and to decide if confirm it or not:

Just in case the system finds a unique Bend axis, gives the possibility to set Face Offset values; another option of this functionality is that the system automatically doesn’t consider the Internal Edges during the punches extrusion, and selecting the relative icon, we can deselect the edges that must be no considered.
Here we can see another example on a Bend Punch creation, and after confirmation, the Strip tree updated with new Punches in the relative steps:

Also after some CAD modifications, the system is able to recognize the Bend Punch:
Cfg Editor

To define the strip is necessary set some parameters; some of these can be edited, changing the values and set these again as default. This is very useful because each customer can set these parameters as wants, to have a better and faster Strip Definition in reference to his settings:

Clicking on OK, the settings are saved and the modifications are available immediately.
**Exec Auto Rebuild**

After some wrong operation (i.e. the user have deleted the strip), could be necessary rebuild the strip; this icon gives this possibility reapplying the set parameters to the complete strip.

This command is also useful in case of the “**Dynamic Strip**” option is disabled, to update the Strip.
Create Report

Selecting this icon the system creates a Report page, with all the parameters set to define the strip and the screenshot.

Dynamic Strip

This option allows to have the automatic rebuilding of the Dynamic Strip, after each modifications, during the strip definition.

If this option is disabled, to have the correct strip on the screen, in reference to the modifications, is necessary select the "Exec Auto Rebuild" command to update the strip.
VISI Progress
Tool Assembly and Standard Elements
Introduction

This training exercise is designed to show the user how to create a progressive die tool based around an existing strip layout. The strip layout does not necessarily have to have been designed using VISI-Progress but could have come from a variety of solid modelling sources.

The intention here is to guide a new user through the various stages required to make a tool, each stage will be dealt with separately, the main stages will be as follows:

- Plate / Bolster creation and modification through Tool Manager
- Custom Plates such as Die Inserts, Lifters etc
- Insertion of Standard Components from supplier catalogues
- Creation of layouts and sections
- Producing a Bill Of Materials and Balloon references

For these tasks it may also be necessary to refer to the Progress Tool Assembly Management reference guide for useful information not covered by this tutorial.

THIS TUTORIAL ASSUMES / REQUIRES GOOD KNOWLEDGE OF VISI-MODELLING

Lets get started!

Reference Diagram of the proposed “Basic Layout”
Section 1 – Building The Plates

Although it is not entirely necessary to have a strip to create some plates in VISI-Progress, it is generally the most common way of beginning to design the tooling.

First of all we need to load a previously designed and saved strip into a new session of VISI, please ensure that you use the file provided as shown below.

File > Open > Solid_Strip.wkf

Stage 1 – Setting the Tooling Parameters

The first thing we need to do is set-up some provisional tooling parameters for the project, this will be explained in the following chapter.

Start the Tool Assembly module from the Progress drop down menu.

You should now be presented with an empty Project tree on the right hand side of the screen. (Shown below).

Creating a new subtool.
A subtool is the very first thing required in a project. It will contain all the plate groups and is basically a container for the tool assembly.

1. LMB click onto the project item. The new subtool icon will appear.

2. LMB click on to “New Subtool” icon.

3. Type in a name for the new subtool of your own choice or use something similar to that opposite.

Type in Cable Clip as the tool name. Click OK.
Now we have created the new Subtool, we need to give some specific parameters to the tool to help define the assembly.

First, let’s consider the next 3 parameters.

It is a relatively simple task to give data to these 4 fields and we can do this automatically from the solid strip as follows:

- LMB click on to “Cable clip” subtool icon to reveal the properties panel for this subtool.
- The properties panel is not yet populated with any useful data.
- Using Block Control select the Strip 1, and the cut off part 2. RMB click to confirm the selections.
If you have selected everything correctly then you should have the following values for the 4 chosen parameters:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Press stroke</td>
<td>0.3</td>
</tr>
<tr>
<td>Strip length</td>
<td>162</td>
</tr>
<tr>
<td>Strip width</td>
<td>70</td>
</tr>
<tr>
<td>Strip thickness</td>
<td>0.9</td>
</tr>
</tbody>
</table>

The punch penetration is assigned automatically and the default value is the strip thickness.

Now lets fill in the remaining necessary empty fields:

<table>
<thead>
<tr>
<th>Properties</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Tool name</td>
<td>Cable-Clip</td>
</tr>
<tr>
<td>Press stroke</td>
<td>50</td>
</tr>
<tr>
<td>Pressure pad stroke</td>
<td>5</td>
</tr>
<tr>
<td>Strip stroke</td>
<td>1</td>
</tr>
<tr>
<td>Punch Height</td>
<td>70</td>
</tr>
<tr>
<td>Punch penetration</td>
<td>0.9</td>
</tr>
<tr>
<td>Strip length</td>
<td>162</td>
</tr>
<tr>
<td>Strip width</td>
<td>70</td>
</tr>
<tr>
<td>Strip thickness</td>
<td>0.9</td>
</tr>
<tr>
<td>Sharpening</td>
<td>0</td>
</tr>
<tr>
<td>Weight</td>
<td>0</td>
</tr>
<tr>
<td>Tool height</td>
<td>0</td>
</tr>
</tbody>
</table>

For more detailed descriptions of the parameters please see the info boxes below.

**Press stroke**

Is the stroke of the ram of the press, starting from its top point to the complete tool closure. This value is useful to determine the tool vertical limits and the gap required between ram and support plate of the press tool. In the software it is used only for the tool visualization when using vertical opening.

**Pressure pad stroke**

This is the stroke that the press must perform after the pressure pad has clamped the strip, it is the stroke that performs the cutting and bending operations. Springs or nitrogen cylinders put between the upper and the central part will create the holding pressure between the plate and the strip. This value can be changed during the tool development, but remember that this value is also used as a stroke for the bolt screws and springs therefore any change means different values for these standards elements. This value is visualised in the complete vertical opening.

**Strip stroke**

This is the stroke required for lifting the strip from the die. This stroke is necessary to simplify the advancing process of the strip in the tool, so that the tool can advance correctly when features such as deep draws and forms are present. This value is visualised in the complete vertical opening.

**Punch height**

It is the total punch height value, that determines the correct position of the upper part of the tool, and it’s a parameter that is used to calculate the SHARPENING value.

When the plates are already defined, any change in this value will automatically reset all the upper part of the tool.

That’s it! We have now set up the required parameters for this tool, next it is time to start adding some plates.
Stage 2 - Adding a Die Plate

All the plates that we initially add at the early stages of the process can be considered as the basic building blocks of the tool and can be subdivided and be given more detail later on in the project.

The Die plate is always considered as the reference plate in VISI-Progress tool build and therefore must be added to the project first.

Initially we must create a group, which is a container for the plates. Follow the next sequence to create a group for the Die:

1. LMB click onto the Cable Clip sub tool.
2. LMB click onto the Group Definition icon that now appears.
3. LMB click onto the Die group.
4. Click ‘OK’ to confirm.

A Die Group has been added to the project.

Each Group has a Properties panel containing useful controls.

**Tool Side**

This parameter determines which side of the tool assembly the group is situated. It can be Upper, Middle or Lower and is a value that must be respected if punch creation and standard elements are to be used in the tool and also if the tool is to be opened horizontally or vertically.
Now we have added a Die Group it is time to add an actual plate into the Group. Follow the next steps to create the initial die plate solid:

1. Make sure the Die group is selected in the project tree.

2. LMB click onto the Add Plate icon.

Now select the type of plate, reference point and its co-ordinate position.

- Select the rectangular block.
- Use "Y axis edge centre as the reference position type.
- Select 'Co-ordinate' input and enter the co-ordinates as shown to the right. Press OK to confirm.

At this stage the system requires a second co-ordinate input of some kind to fix the plate dimensions. However you will notice a new icon appear on the left hand toolbar, this is the 'dialog box data input', here you can fix the plate size in an automatic way. See below:

2. LMB click onto the 'dialog box data input' icon.

Notice the length and width of the plate have automatically been assigned some values.

- **Length** = The total length of the solid strip that was selected at the beginning of the project.
- **Width** = The total width of the solid strip plus an extra 80mm. The extra 80mm is determined by the settings inside the Progress_Tool.cfg
- **Height** = User definable value for plate thickness.

Enter a new plate thickness as shown and click ‘OK’.
The die plate should now appear in the graphics area of the screen as shown below.

Adjusting the properties of the Die Plate

The Die plate is not yet the correct size and needs reducing in width (Y) slightly. To do this it is simply a case of adjusting the relevant property of the die plate. Let’s do this now and also change the plate’s colour:

Focus in onto the y dimension property

<table>
<thead>
<tr>
<th>Y dimension</th>
</tr>
</thead>
<tbody>
<tr>
<td>150</td>
</tr>
</tbody>
</table>

Now change the plate colour:

<table>
<thead>
<tr>
<th>Plate colour</th>
</tr>
</thead>
<tbody>
<tr>
<td>R: 192 G: 192 B: 192</td>
</tr>
</tbody>
</table>

Building direction

This is the "logical" direction in the Z axis of the plate; the value can be "built up" or "built down": for example if the plate is built with a "built down" attribute, when the user modifies the height of the plate (relative to Z axis), the modification will be made in the downward direction and vice versa if the building direction is "built up".

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The result of modifying the Die plate properties.

Stage 3 – Adding A Base Plate

The process of adding another plate to the tool assembly is identical to the one described for the die plate, only the location and sizes of the plate are different.

Firstly we will need to add a group to contain the plate(s). This time we will do this using the context sensitive menu’s.

RMB click onto the “Cable Clip” Sub Tool in the project window and then use the menu sequence as shown below.

The group definition box should appear, here you can choose to type in an alternative name for your group if you want, otherwise accept the default.

LMB click on OK to create the group.
Now we have added a Base Group it is time to add an actual plate into the Group. We have shown how to do this in detail in the definition of the die. Follow the same steps performed to create the base plate solid using the specific details below:

The base plate is now created in the exact position we require as shown below.

Global Modification
If Global modification is set to 'Yes' any dimensional change you make to a plate or group will also apply to the other groups and entities in the tool. For example extending a plate 10mm in Y direction and Global modification set to 'Yes', all other plates will be extended by 10mm.

Lock Entity
If the Lock Entity checkbox is ticked this means that the plate will not be affected by any dimensional changes made by Global Modifications.
Stage 4 – Adding Left and Right Guide Rails

Begin by adding the Left Rail Group into the tool assembly. Rails are added into separate groups to ensure that a modification to one of the rails does not have effect on the other. Create the group definition in exactly the same way as previous examples.

Create the plate as specified below:

Application point for “Left Rail” Snap using intersection filter.

Now input the parameters for the rail dimensions.

Left Guide Rail Positioned on top of the Die
Adding the Right Guide Rail

Again we need to create a separate group for the Right Rail in the same way as the left.

Application point for “Right Rail”. Snap using intersection filter.

Now add the parameters for the right rail dimensions.

Right and left guide rails shown in position.
Stage 4 – Adding A Pressure Pad
In this design we will be adding a “T shape” pressure pad to fit between the guide rails placed in the previous section.

Add the pressure pad group.

Now select the T Shape plate and the application point as shown below.

Pick the midpoint on the **Top** edge of the strip.

Enter the dimension parameters for the Pressure Pad.
Pressure Pad shown in position.

Now we shall elongate the left hand end of the pressure pad plate.

**Elongating the pressure pad by 10mm.**

Pick the pressure pad plate from the tree.

From the top toolbar, select the "Plate Control icon.

From the Plate control panel we can change the plate application / control point. Here we will fix the control point to the mid right hand end of the plate.

The grid represents the preset application points that can be selected on a plate. Here we are selecting the Right hand mid point on the plate. It means any modifications applied to a plate will take effect from this point as reference.

Now make the plate 10mm longer in the X dimension, originally 162mm

Change the X dimension to 172mm. (Originally 162mm)
Now look at the effect on the pressure pad plate.

Stage 5 Adding the Punch Plate

Start by adding the Punch Plate Group to contain the punch plate geometry.

Choose the plate type and dimensions as shown.

Pick the midpoint on the Top edge of the pressure pad as application point.
You should now receive a message warning that the plate is not in the correct position. This is true because the Punch plate should be located with reference to the Punch Height parameter (70) in the tool properties.

Stage 6 - Adding the Punch Backing Plate

Create the Group “Top Backing Plate” in the normal way as shown in all previous examples.

Now add the plate details and dimensions as shown.
Now add the height of the backing plate in a positive Z direction as shown. The X and Y dimensions should be defined by the points you have picked.

Stage 7 – Add the Top Plate to the tool assembly to complete the basic plate layout.

Add the Group “Top Plate” to the project tree in the previously described manner.

Now drag and drop the base plate from the Base group into the Top Plate group. The base and top plate share exact dimensions and properties.

Drag and drop the base plate into the Top Plate group by clicking and holding the left mouse button.

At the pop up menu choose the “Copy” command. This will duplicate the plate into the Top Plate group.
Insert copy co-ordinates as shown below, this will translate the copy into the correct position.

You should now have a top plate located in the Top Plate group of the project tree.

The Result of Stages 1-7

If you have completed the stages successfully, you will have a tool and project tree that resembles the one below.

In the next section we will cover the creation of the Die Forms and the form punches.
Stage 8 – Creating Die Forms and Form Punches.

In this section we will create all the necessary pockets and form inserts in the die to produce the strip.

**Design Intent**

Here is an image of the design intent for the die, this will help give an idea of what is going to be achieved whilst performing the next steps.

Now isolate only the die plate, as we will be creating a “C” shape pocket to house a die insert and provide relief for the formed parts on the strip.

<table>
<thead>
<tr>
<th>Index</th>
<th>Layer name</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>DIE</td>
</tr>
<tr>
<td>1</td>
<td>BASE</td>
</tr>
<tr>
<td>2</td>
<td>PRESSURE_PAD</td>
</tr>
<tr>
<td>3</td>
<td>LRT Rail</td>
</tr>
</tbody>
</table>

Draw the closed “C” shape profile at the right hand end of the plate. Use the dimensions shown below.
Use any modelling technique that you know to create a pocket like the one shown below. It is assumed that you already know the basics of VISI-Modelling, if you are unsure how to create the pocket, ask your tutor.

**Tip** – Pocket From Profile or Extrude and Subtract

Pocket Depth = 12mm

Create the Die Insert into the pocket

<table>
<thead>
<tr>
<th></th>
<th>STRIP/CUT</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>STRIP</td>
<td></td>
<td></td>
</tr>
<tr>
<td>10</td>
<td>TOP_BACK_PLATE</td>
<td></td>
<td></td>
</tr>
<tr>
<td>11</td>
<td>TOP_PLATE</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12</td>
<td>DIE_BLOCK</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Turn on only the STRIP layer and make it the Current Layer.

Create a profile as shown opposite from the underside surface of the Strip. The profile should be positioned at the midpoint of the component.

**Tip:** - Modelling > Sectional Elements.
If you are unsure how to perform this modelling task please consult your tutor.

In side view your section should look like the one below. Extend your section 2mm past the end of the part and make it 25mm deep from the underside of the strip.

Extend the section past the component by an extra 2mm each end.
Now extrude the profile to create the solid die insert.

**Modelling > Extrude elements**
Extrude the profile along the Absolute X axis, 18mm in total which is 9mm in both positive and negative directions as shown.

Now place the die insert on to the DIE layer

Select **Change Attribute**

Pick the Die Insert and place it onto the DIE layer.

Now turn on the Die Layer to display the insert and die block. Make DIE the current layer.
Make a cavity in the main die block to house the DIE insert.

Operation >

1. Pick the Die Block as the target body.
2. Pick the Die Insert as the cavity body.

The Result

Insert now housed in a through pocket as shown.

Add a chamfer to the end of the main die block to act as a chute for the final detached part.

Operation >

Change the distances as shown. All other parameters can stay as default.

Pick the edge shown.

The resulting chamfer.
Add the die insert to the Tool Assembly manager.

Progress >

Add plate
- Add Custom plate
- Move the group
- Translate and copy
- Delete group
- Expand/Collapse Branch
- Unlock Plates

Right mouse click on the Die group and select Add Custom Plate.

Select the Die Insert from the screen area. The die insert is now added to the Die group.

Click on the new die in the group and change the name of the plate from the properties panel.

Change the name of the plate here to Form Insert.

The name change is now reflected in the tree.
**Exercise - Create the Form Punch**
The form punch can be created in exactly the same way as the die insert we previously created. This time use the top surface face of the strip as the reference section.

### Table

<table>
<thead>
<tr>
<th>Layer</th>
<th>Status</th>
</tr>
</thead>
<tbody>
<tr>
<td>STRIPCUT</td>
<td>✔️</td>
</tr>
<tr>
<td>STRIP</td>
<td>✔️</td>
</tr>
<tr>
<td>TOP_BACK_PLATE</td>
<td>✔️</td>
</tr>
<tr>
<td>TOP_PLATE</td>
<td>✔️</td>
</tr>
<tr>
<td>DIE_BLOCK</td>
<td>✔️</td>
</tr>
</tbody>
</table>

Turn on only the STRIP layer and make it the Current Layer.

Create a profile as shown opposite from the topside surface of the Strip. The profile should be positioned at the midpoint of the component.

**Tip:** - Modelling > Sectional Elements.

If you are unsure how to perform this modelling task please consult your tutor.

In side view your section should look like the one below.

- The profile should be the full length of the part in Absolute Y which is 58mm.
- The height doesn’t matter, because later the system will automatically extrude the punch for the correct height.
- The form punch is extruded a total of 18mm with a distance of 9mm positive and negative either side of the profile.

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Stage 9 – Extruding the cutting punches

Now that we have added the forming punch it is time to create and manage the cutting punches.

Note :- Before starting this section make sure that your Workplane is set to ABSOLUTE Top.

Activate all the layers from the layer manager and make sure PUNCHES is the current layer.

Start the punch manager from the Progress menu.

From the punch manager interface, choose the “Add / Select Punches” icon.

Select the 3 punches as shown below. These are considered as “Peripheral” punches as they cut the periphery of the strip.

Click ‘OK’ three times.
The punches are now added to the Punch Management Tree and ordered as shown below.

Add/Select Punches
From the Punches Management select the Add/Select Icon. You will be prompted to select the solid part as punches. The multi-selection is available and you can select all the punches that you have already defined in the Strip Management.

Selecting the solids, the system recognizes the type of punches and creates the relative folders.
It is possible to select a solid that you have not defined as punch in the Strip Management, in this case the system asks what type of punches you want, i.e. peripheral, inner to the piece etc.
Use the punch management to extrude the punches and add clearances.

Click the Extrude Punches icon to extrude the punches in the Peripheral group.

Select the Peripheral group in the punch management tree.

The punches will now be extruded up to the top of the punch plate and clearance cavities will be created around the punches and in the die plates.

The 3 punches are extruded to the back of the Punch plate with clearance cavities.

Expand the punch in the tree and you will now find you can adjust the clearance cavities through the top of the tool and the die plates.

Next we will adjust the clearance cavities through the tool.
Adjusting the Punch Clearance Cavities
The punch management allows us to use a variety of options for adding clearance around the punches and through the corresponding plates. We will deal with each punch in turn.

Punch 1 – The “T” shaped punch.

First we will deal with the Top Cavity parameters. This will deal with the clearance through the punch plate and pressure pad.

Add the parameters as shown. We will not have any clearance through punch plate. But will have constant 0.05 through the pressure pad plate.

No clearance added through punch plate.

Enlarged detail showing clearance through the pressure pad.

Now let’s adjust the Die Cavity

Adjust the parameters as shown for the Die Cavity

An explanation of the parameters used is shown on the next page.
The cavity through the Die and the Base explained.

Die Punch Clearance (0.045)
Die Draft Clearance (0)
Base Clearance (1)
Cutting Height (3)
Die Draft Angle (1)
Base Draft Angle (1)

Now adjust the cavities for the other three punches.

Punch 2

The resulting clearances for Punch 2.
The resulting clearances for Punch 3.

**Differential Die Clearance**

Clicking on this button, the system shows the profile of the punch and asks to select the sides to apply a different gap. If you have selected a side with a different Gap value you can decide whether or not to apply draft; selecting "With Draft", the system applies draft on all sides, included the side with differential gap, alternatively if you don’t want the draft on the selected side you must select: "Drafted only on sides with general gap"
Adding the Final Cropping Punch
The fourth punch to be added is the detaching punch at the final stage.

From the punch manager interface, choose the “Add / Select Punches” icon.

Now extrude the punch contained in the “Inner to the Strip” group.

2) Click the Extrude Punches icon to extrude the punches in the selected group.

1) Select the Inner to the Strip group in the punch management tree.

Side view of extruded punch
Enter the details for the Top and Die Cavities as shown below.

View of extruded punch with Top and Die cavities.

Here is a Tip that will make it easier to find and display punches in the punch manager tree.

**Cad view mode**

This function allows the user to select different graphical representations for the display of punches when selected from the tree, a list of different modes will appear when clicking on the arrow.

Highlight Punch
- Display the Punch only
- Display and Zoom the Punch only

For example, using the "Display and Zoom the Punch only" mode, the icon will change to the following:

Every time a user selects an entity in the tree, only this entity will be displayed on the screen with automatic include all (the rest will be hidden).
Add an offset head fixing to Punches 2 and 4.
To assist the positioning and strength of punch 2, we will add an offset fixing head to the punch. This is done as follows:

1. Click the Add Fixing Icon from the Punch manager.
2. Select the drop down option for Offset Fixing.
3. When prompted pick Punch 2 and 4 as the designated punches and then adjust the fixing properties as shown:

<table>
<thead>
<tr>
<th>Punch 4</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Fixing Properties</strong></td>
</tr>
<tr>
<td>Height</td>
</tr>
<tr>
<td>Offset</td>
</tr>
<tr>
<td>Head Body Clearance</td>
</tr>
<tr>
<td>Clearance Under Head</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Punch 2</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Fixing Properties</strong></td>
</tr>
<tr>
<td>Height</td>
</tr>
<tr>
<td>Offset</td>
</tr>
<tr>
<td>Head Body Clearance</td>
</tr>
<tr>
<td>Clearance Under Head</td>
</tr>
</tbody>
</table>

General Information on other types of fixing available.

- Selecting Hook Fixing, the system asks to pick an edge where you want create the hook to fix the punch:

   - In the property tree of selected punch, you can also see the Fixing Property:

   | **Fixing Properties** |
   | Height | 5 |
   | Width | 5 |
   | Left Offset | 0 |
   | Right Offset | 0 |
   | Head Body Clearance | 0.05 |
   | Clearance Under Head | 0 |

To delete a hook fixing on a punch is necessary to click on the items property tree, then M2 and select: Delete Head Fixing.
**PLATE FIXING**

- Selecting **Plate Fixing**, the system asks to pick the punch where you want create the plate to fix the punch:

In the property tree of the selected punch, you can also see the Fixing Properties:

To delete a Plate fixing on a punch is necessary to click on the referred property tree, then M2 and select:

Select a punch

The system asks to select X axis as referred axis to build the plate

Obtain this plate

In the property tree of the selected punch, you can also see the Fixing Properties:
Add Stalk

Let's continue with our example tutorial!!

**Add Stalk**

Click on the fourth icon and the system asks to pick the punch where the stalk will be inserted.

In the property tree of the selected punch, you can see and change the Stalk Properties.

<table>
<thead>
<tr>
<th>Stalk</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>Rectangular</td>
</tr>
<tr>
<td>Length</td>
<td>0.018</td>
</tr>
<tr>
<td>Width</td>
<td>10</td>
</tr>
<tr>
<td>Punch Length</td>
<td>10</td>
</tr>
<tr>
<td>Punch Width</td>
<td>10</td>
</tr>
<tr>
<td>Punch Height</td>
<td>30</td>
</tr>
<tr>
<td>Blend Radius</td>
<td>3</td>
</tr>
<tr>
<td>X Alignment</td>
<td>Centre</td>
</tr>
<tr>
<td>Y Alignment</td>
<td>Centre</td>
</tr>
<tr>
<td>X Displacement</td>
<td>0</td>
</tr>
<tr>
<td>Y Displacement</td>
<td>0</td>
</tr>
</tbody>
</table>

To delete a stalk on a punch is necessary to click on the referred property tree, then M2 and select delete:

Let's continue with our example tutorial!!

Next we will add some heels to the punches.
Add a Heel to Punch 1 and 3.
To help balance the punch and resist lateral displacement from the front cutting edge of
the punch we will add a heel to punches 1 and 3. Follow the example shown:

1. Click the Add Heels Icon from the Punch manager.
2. Pick the single straight back edges on the punches.
3. On the property tree of the selected punch, the system shows the heel parameters that you
can dynamically change:

   1. **Height**: the height of the heel.
   2. **Width**: the width of the heel.
   3. **Edge Length**: this is the length of the selected edge (it is not modifiable).
   4. **Length**: the length of the heel (it is not modifiable).
   5. **Left Offset**: it is possible create an offset on the heel length (on the left).
   6. **Right Offset**: it is possible create an offset on the heel length (on the right).
   7. **Angle**: this is the value of the heel internal angle.
   8. **Invite Blend**: this the invite blend value.
   9. **Lower Blend**: this the lower blend value.
   10. **Upper Blend**: this the upper blend value.

4. To delete a heel on a punch is necessary to click on the relative property tree, then M2 and
    select:

   ![Heel Properties Table]

Vcamtech Co., Ltd
Adding the Bending Punch

The last punch to be added is the bending punch.

From the punch manager interface, choose the “Add / Select Punches” icon.

Now extrude the punch contained in the “Bend” group.

1) Select Punch 5 in the punch management tree.

2) Click the Extrude Punches icon to extrude the punches in the selected group.

Enter the details for the Top Cavity as shown below.

View of extruded punch with Top cavity.
Stage 10 - Inserting Guide Pillars
Now we have defined the punch and die details the next stage of the tool design is to insert the standard components, beginning with the guide pillars.

Progress > Insert elements

For this example we will use the Fibro parts database.

Select the Cylindrical Column 2021_46 from the columns menu.

1) Pick the lower plate as start plate.

2) Pick the upper plate as end plate.

Choose the application point shown below

The pillar is inserted as shown opposite and below.
Insert a bush around the column into the top plate.

1) Pick the pillar to add the bush

2) Pick the upper plate as the insertion plate.

Select the Stepped Bush 2081_35 from the bushes element selection menu.

Insert the bush parameters as illustrated below, make sure that the Reverse Assembly flag is checked to invert the bush.

Make sure the “Reverse Assembly” is checked.

Bush shown in position around pillar.
Mirror the complete Column and Bush assembly to the opposite side.

Select the “Mirror 2D” option from the edit tool bar and make sure Copy is on.

Pick the **column and bush** to be mirrored.

Now select the mirror options, select point and copy. **Use a Y mirror.** After confirming the Y mirror, select **No** at the Rebuild Transformed Standard Element window. This is because our plates have not changed in any way.

Pick the mid point of the top edge of the top plate as the mirror position.

---

Column and bush assembly mirrored
Stage 11 – Adding the Piercing and Pilot Punches

First we will add the piercing punch that will create the pilot hole in the strip.

Select the Punch 222 from the element menu.

Pick the Punch Plate as the start plate.

Pick the Die Plate as the end plate.

The application point is the centre of the first hole in the Strip. Use centre snap option.
Now insert the punch clearance parameters shown in the diagram below.

The diagram shows the punch in position with the clearance holes.

Placing the Pilot punches
To help with the positioning of the strip we will add 4 pilot punches to the design.

Select the Pilot P220 from the drop down list.
Now choose the start and end plates for the pilot punch.

Pick the Punch Plate as the start plate.

Pick the Die Plate as the end plate.

Now insert the punch clearance parameters as shown below.

The application point is the centre of the second hole in the strip.

<table>
<thead>
<tr>
<th>fibro - P220_1 (ppil2) / Rule: PPIL2</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Parameters</strong></td>
</tr>
<tr>
<td>Diameter</td>
</tr>
<tr>
<td>Length</td>
</tr>
<tr>
<td><strong>Database Values</strong></td>
</tr>
<tr>
<td><strong>Rules</strong></td>
</tr>
<tr>
<td>Clearance Under Head</td>
</tr>
<tr>
<td>Clearance Head Diameter</td>
</tr>
<tr>
<td>Pilot Penetration</td>
</tr>
<tr>
<td>Build Bottom Cavity</td>
</tr>
<tr>
<td>Punch Plate</td>
</tr>
<tr>
<td>Clearance</td>
</tr>
<tr>
<td>Bottom Clearance</td>
</tr>
<tr>
<td>Height</td>
</tr>
<tr>
<td>Guide Plate</td>
</tr>
<tr>
<td>Clearance</td>
</tr>
<tr>
<td>Bottom Clearance</td>
</tr>
<tr>
<td>Height</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>fibro - RISER (riser)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Parameters</strong></td>
</tr>
<tr>
<td>Diameter</td>
</tr>
<tr>
<td>Height</td>
</tr>
<tr>
<td>Chamfer Value</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>fibro - P220 (a_ppil2)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Parameters</strong></td>
</tr>
<tr>
<td>Without Distance Ring</td>
</tr>
</tbody>
</table>
Note: It is now possible to insert the punches into the remaining 3 positions because the *Apply* checkbox for this punch is still enabled allowing you to add more punches as required.

**STAGE 12 – Adding Stripper Bolts**
Having added the standard punches to the design we will now add the stripper bolts to connect the Top Plate to the Stripper Plate.

1. Select the Shoulder Bolt 244_17 from the element menu.
2. Pick the Top Plate as the start plate.
3. Pick the Pressure Pad as the end plate.
4. Now place the remaining 3 pilot punches into the tool as shown. The origin points are at the centres of the strip holes.
Now insert the co-ordinate position for the shoulder bolt and then the parameters for the bolt.

Let's now add a Spring around the Stripper bolt between the Punch Plate and the Pressure Pad plate.

Select the Cylindrical Section Spring 241_25 from the drop down list.
Pick the plates as stated by the selection tree parameters, illustrated below and insert the co-ordinate position for the spring (same as the previously inserted bolt).

Pick the Punch Plate as the start plate.

Pick the Pressure Pad as the end plate.

Enter the parameters for the Spring as shown below highlighted in the boxed area.

16mm Spring Diameter

6mm of required stroke

6mm of required preload

Adjusting these parameters will ultimately dictate the length of the spring. In this case we begin with a 38mm length and after adjusting the parameters the length will adjust to 44mm.

Springs shown in position around bolt screw.

Note: As an alternative this assembly could have been placed as a Group element type.
Some Useful Notes About Spring Parameters

- **Total Length** = gap between plates + preload + stroke
- **Load %**: - This is the target load percentage
- **Real Load %**: - This is the actual load percentage calculated from \( \frac{\text{STROKE}}{\text{TOTAL_LENGTH}} \times 100 \)
- **Calc Stroke**: - This is the actual resulting stroke(mm)
  \[ \text{TOTAL_LENGTH} \times \left( \frac{\text{LOAD\_PERCENTAGE}}{100} \right) \times \text{STRENGTH} \]
- **Real Stroke**: - This is the input value for the stroke you require.
- **Calc Load**: - This is the actual load achieved in (N).
  \[ \text{TOTAL_LENGTH} \times \left( \frac{\text{LOAD\_PERCENTAGE}}{100} \right) \times \text{STRENGTH} \]
- **Real Load(N)**: - This is used for information only and is calculated from STROKE * STRENGTH
- **PreLoad(mm)**: - This is the target load percentage

The parameters **Housing On Start** and **Housing On End** are now driven by the Spring Length. The Spring Length from the supplier database is used and if the spring length exceeds the actual Gap between the plates then the extra distance is passed automatically onto the Housing On Start parameter. This value can be divided between the 2 parameters Housing on Start and Housing on End.

In our example: - Gap Between Plates = 28.2 Stroke = 6 and Preload = 6 therfore we have a total length of 40.2mm.

The next Spring size in the database = 44mm.

Therefore 3.8mm extra length is passed to the **Housing On Start**
Make 4 copies of the Bolt and Spring assembly in 4 new locations.

Select both the Bolt and Spring from the toolbar.

Select the reference position for the copy. This should be the Centre of the bolt.

Insert the 1st copy co-ordinate position.

Now Repeat the same process for the remaining 3 bolt and spring assemblies. At the REBUILD dialogue- select ‘YES’

Copy 2

Copy 3

Copy 4

All the Bolt / Springs shown in position.
Stage 13 – Fixing the Top Plate to the Punch Plate using Cap Head Screws.
The next stage requires us to bolt together the Punch Plate and Top plate by use of cap head screws and some location dowels. The following details describe how.

Enter the co-ordinates for the screw

Select the Top Plate as the start plate.

Select the Punch Plate as the end plate.

Insert the 6 x 40 screw with the parameters shown opposite.
The co-ordinate input box should re-appear allowing you to input another position for a screw to be inserted. Insert screws in the next 3 positions.

Below is what you should expect to see when all four screws are in position. (Plan View)

Next we will insert the location dowels

Select the Dowel 235.1 from the dowel element list.

Select the Top Plate as the start plate.

Select the Punch Plate as the end plate.
Now insert the co-ordinate position for the dowel pin and adjust the parameters to match those shown.

As is consistent with all insertions of standard elements, the co-ordinate input box should re-appear allowing you to input another position for a dowel. **Remember this is done with the APPLY Check box flagged.** Here is the position for dowel 2.
Exercise 1 – Fixing Punches To the Punch Plate
In this exercise you will use cap head screws to fix the punches to the Top Plate. Use the same technique as previously described to insert screws. All the screws co-ordinates and parameters will be provided for you. All co-ordinates are absolute.

<table>
<thead>
<tr>
<th>X</th>
<th>Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>49.7</td>
<td>-36.0</td>
</tr>
<tr>
<td>64.2</td>
<td>-36.0</td>
</tr>
<tr>
<td>62.0</td>
<td>36.3</td>
</tr>
<tr>
<td>76.0</td>
<td>-36.3</td>
</tr>
<tr>
<td>105</td>
<td>21.75</td>
</tr>
<tr>
<td>105</td>
<td>-16.2</td>
</tr>
</tbody>
</table>

Start Plate = Top Plate
End Plate = Punch
Exercise 2 – Fixing Guide Rails
The first task is to fix the guide rails to the die block. Use the same technique as previously described to insert screws and dowels. All the screws co-ordinates and parameters will be provided for you. All co-ordinates are absolute.
Exercise 3 – Fixing Die Insert To Lower Plate
The next task is to fix the guide rails to the die block. Use the same technique as described earlier on to insert screws. All the screws co-ordinates and parameters will be provided for you. **All co-ordinates are absolute.**

Exercise 4 – Fixing the Die Plate To Lower Plate

Start Plate = Lower Plate
End Plate = Die Insert

Start Plate = Die Plate
End Plate = Lower Plate
Finally, add the last dowel for extra location of the die plate.

```
X=134.0
Y=-49
```

Start Plate = Die Plate
End Plate = Lower Plate

Activate all the layers to ensure the whole assembly is visible.
Now subtract all the element cavities from the tool assembly to leave behind the actual holes / cavities in the plates.

Suggested Further Study

Organise the Tool – Placing plates onto the correct side, linking elements etc.
Make Plot Views Of Assembly and Individual Plates
Hole Chart with Extended Dimensions
Assembly Manager To Create Parts List.