

1B Integrated Design Project

Hints on the use of Creo for Sheet Metal Design

1 Philosophy

As was demonstrated in the final CAD sessions of part 1A, Creo has an application dedicated to sheet metal components, with the ability to flip between the folded component and the flat pattern (usually referred to as the development) as cut out prior to bending. Most modern cutting machines, whether using punches, laser beams or water jets, can accept DXF files, as produced by many CAD packages, as the main instruction input, thus making savings of time and effort while decreasing opportunities for error.

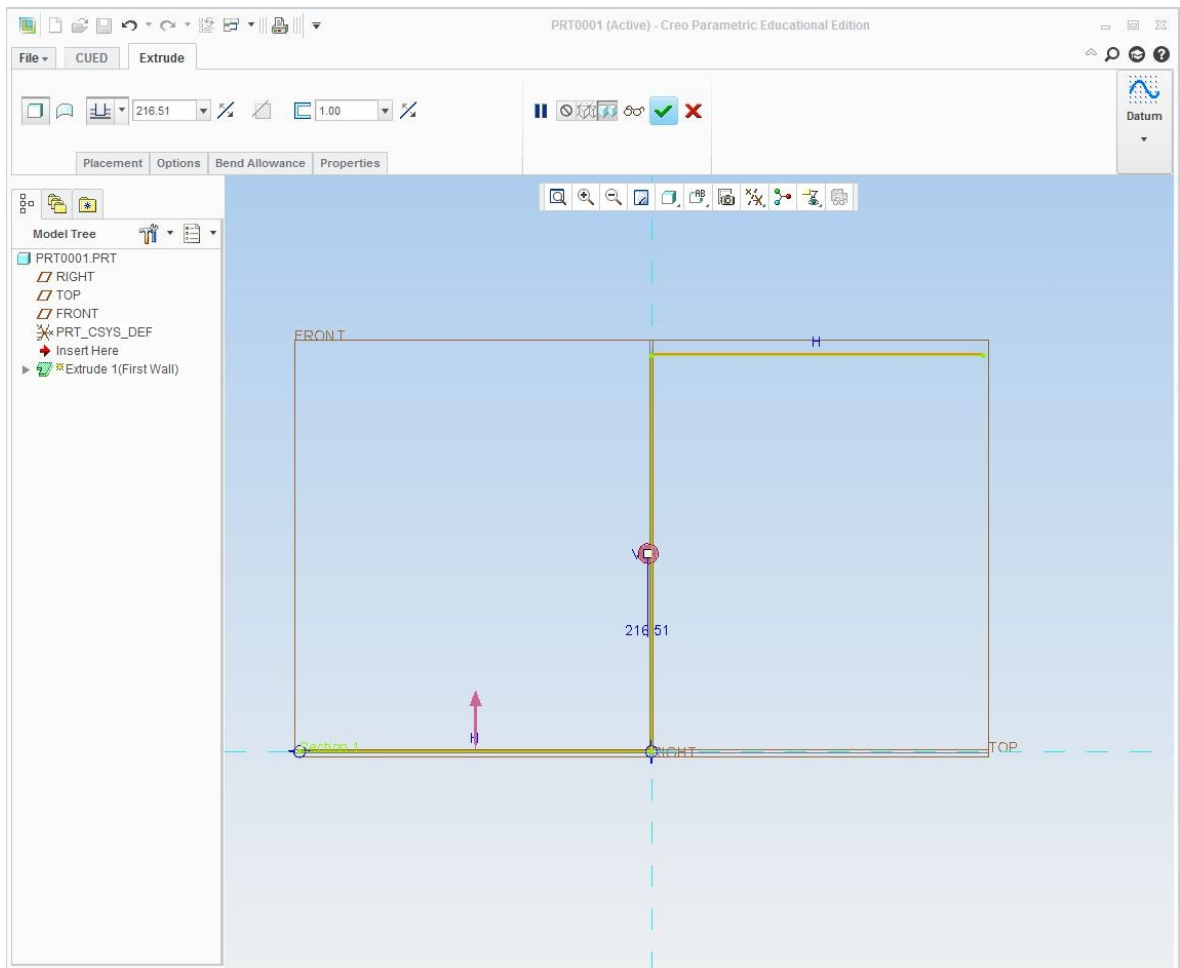
The following notes are a guide to the use of Creo in the IDP. Whilst modelling your design does require some investment of time, it will more than repay this when cutting out and modifying your chassis and other components as well as providing the power to check for interferences and to animate mechanisms.

As designers, it is important to remember that the primary purpose of producing a drawing or a model is to provide a statement of your requirement. Whilst a good designer will design for a method of manufacture, the role of a drawing as a set of manufacturing instructions is secondary, in that manufacturing processes evolve and skilled technicians will have their own techniques. When modelling a folded sheet metal component, your requirement is for the fully folded item and so this is what should be modelled. The power of Creo can then be employed to derive the flat development for cutting out and subsequent bending.

Example - Production of a simple bracket.

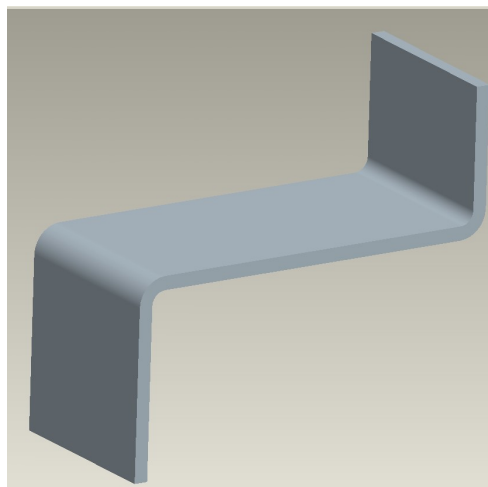
Launch Creo in the normal way, start a new model and immediately change the “Sub-Type” to **Sheetmetal**. Note that the ribbon at the top of the screen has different icons to the solid modeller. Until a first wall is present the only icons highlighted as available are labelled **“Extrude”** and **“Planar”**. Planar allows the modelling of a flat panel, of any shape, which may be subsequently bent up. This would, however, be contrary to the stated principle of “draw the requirement” as it would not afford control of the final dimensions of the folded component. Any changes to the material thickness or bend allowance would cause variations to overall dimensions.

Using the option of **“Extrude”** allows the bent component to be drawn and final, overall dimensions to be fixed. Select this icon and remember that it is the edge view of the component being drawn, complete with bends of suitable radii. The following diagram shows the creation of a “Z” bracket. Note that the use of “internal” sketches is recommended – just press RMB and select from the menu that appears. Or simply select a sketch plane.



Note that the sketch consists of a single line which is **“Thickened”** on the dashboard after leaving the sketcher. Direction of thickening and material thickness and bend radii can be selected on the dashboard. Note that bend radii do not, now, need to be sketched but can be specified under the **“Options”** menu.

The sketch can now be accepted by clicking on the tick icon and with a blind extrusion depth specified the folded bracket produced as shown below.



2 Starting the Chassis

Whilst complex parts can be built up by creating an unattached wall and subsequently attaching extra features to it, it is usually quicker and simpler to start off with a solid extrusion when designing something like a chassis with multiple bends.

The recommended procedure is described below, using an example to illustrate the various stages.

2.1 Shelling from solid model

Draw your chassis design in the standard Creo modeller as a solid “brick” – see Figure 1. Remember to make sensible choices of datum plane positions – you will be pleased you did later. Select, under “OPERATIONS”, the **convert to sheet metal** option and from the menu that appears choose **shell**.



It is now necessary to select which faces of the solid will be removed, bearing in mind that one piece of material from the flat pattern cannot be in two places at once. Once surface choices are complete you will be asked for the material thickness. The sheet steel used for IDP construction is 0.7mm. Figure 2 shows the resulting shell.

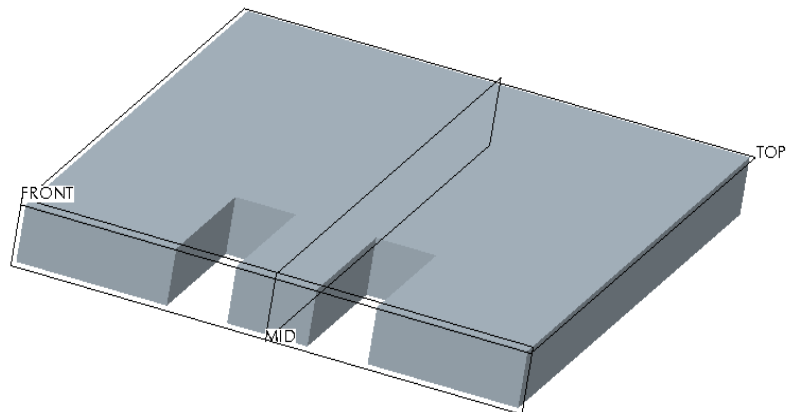


Figure 1

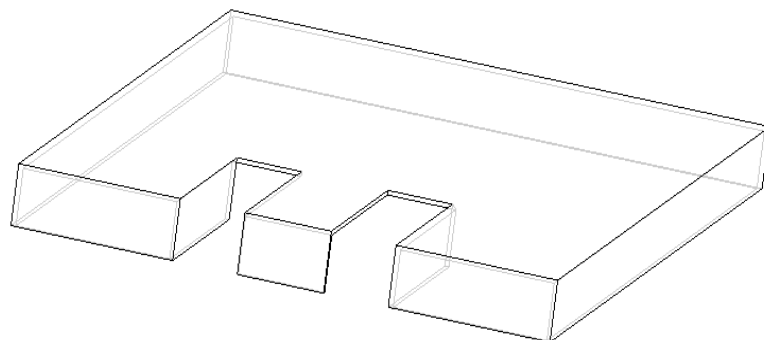


Figure 2

2.2 Sheet metal conversion

2.2.1 Ripping

The shell in Figure 2 does not yet represent a piece of bent sheet metal, as it is not yet obvious which edges are folded and which are not continuous (“ripped”). Select the “**conversion**” option (from the OPERATIONS menu) and note that the “dashboard” becomes :-



Figure 3

The left hand icon represents edge rips. Select this and choose which edges are ripped.

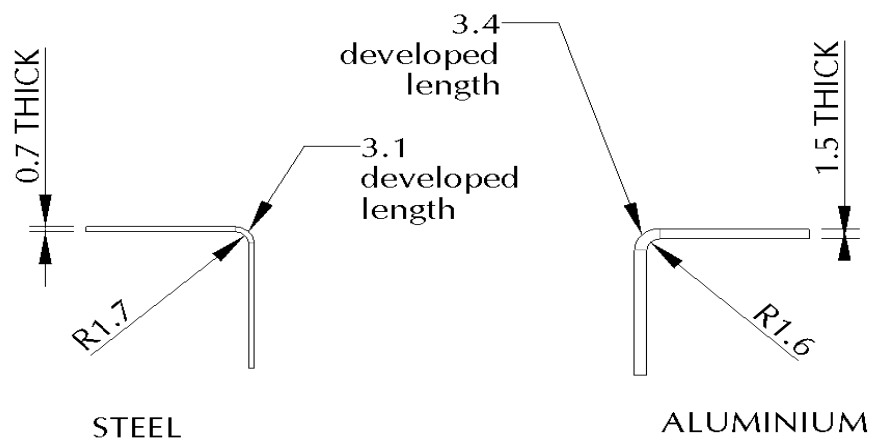
2.2.2 Bend Definition

Next select the third icon and define the bends.

The actual bend radius achieved on the hand bender in the workshop, with this material, is 1.7mm (internal).


The bend properties of the IDP materials are shown in the following diagram. Note that the Aluminium alloy is 1.5mm thick and that an internal bend radius of 1.6 is achieved.

Figure 4



BEND CHARACTERISTICS OF IDP MATERIALS

2.2.3 Flat Pattern

It should now be possible to produce a development of the model by selecting “**Create Flat Pattern**” by selecting the icon - 

Check that the model is now flat, by selecting any saved view other than Top or Bottom. Any edges still folded indicate that not enough edge rips were selected. Check also that a consistent bend allowance of 3.1 has been allocated.

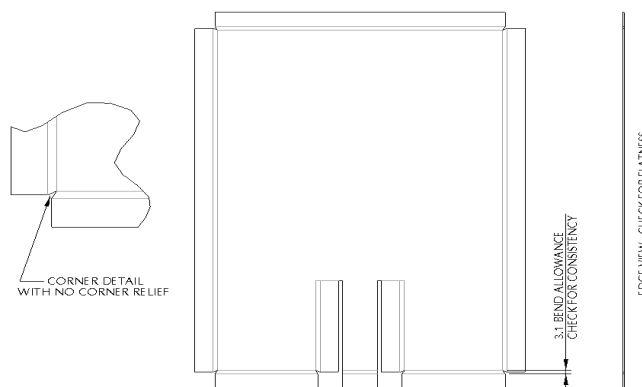


Figure 5

2.2.4 Relief

In Figure 5 a typical flat pattern is shown. Note that as no corner relief was selected then sharp darts are cut into the start of the bend, resulting in a high rate of material deformation in this corner. Although, for the thin material being used, this is acceptable, the ease of bending will be improved by removing some material from this area.

Choose the **Smt Conversion** from the family tree and, on the RMB menu select **“Edit Definition”** to redisplay the dialogue box shown above in Figure 3. Highlight the right-hand icon and see the effect of choosing one of the options shown in Figure 6. Recommended dimensions are shown.

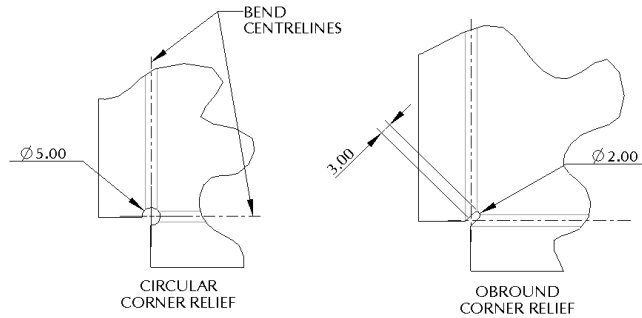


Figure 6

Another nicety is illustrated in Figure 7 where part of a sheet is to be bent down and part not. Although Pro-E does offer bend relief for such instances, cutting machines cannot produce slits of zero width. It is, therefore, suggested that a cut is inserted which goes just beyond the bend area, terminating in a full radius, as shown.

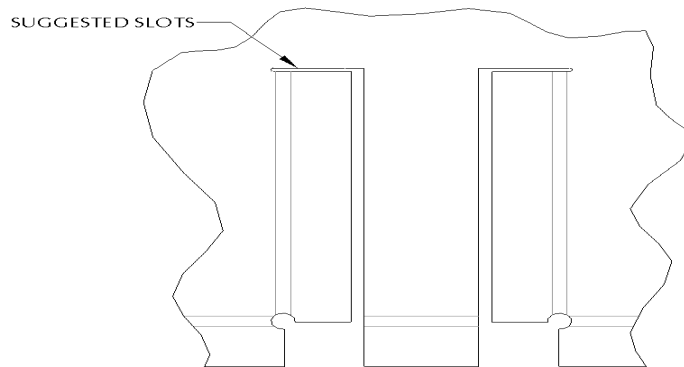
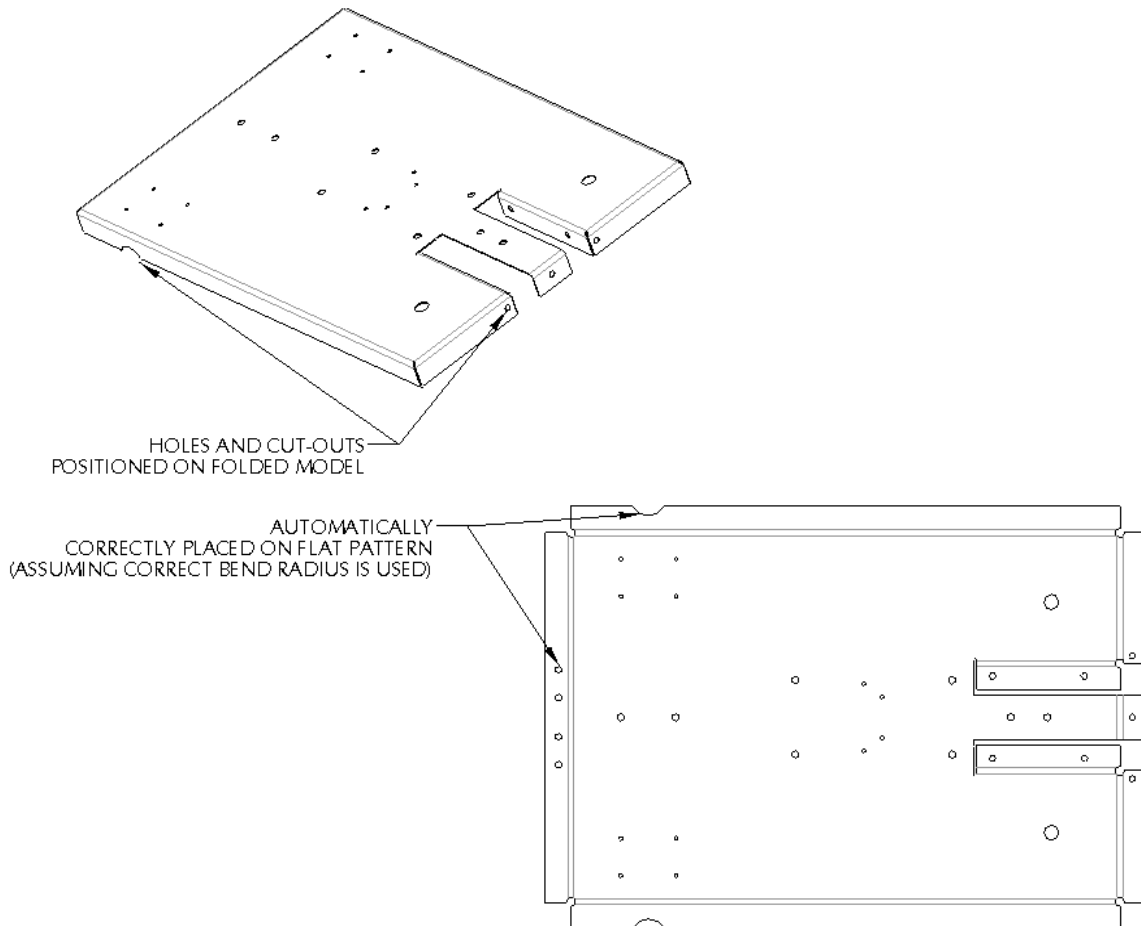


Figure 7

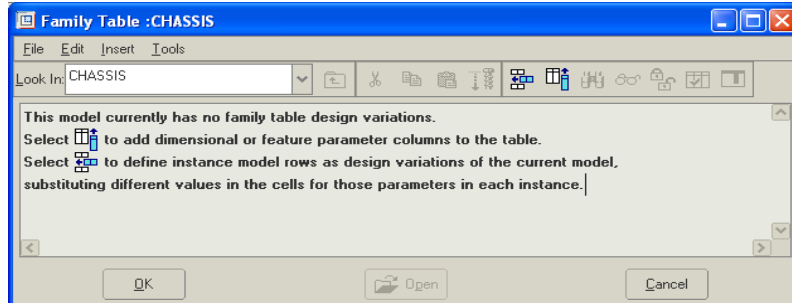
2.3 Adding Cuts



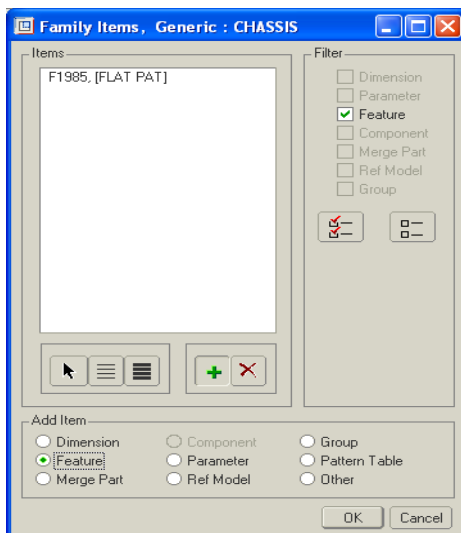
To save time it is strongly recommended that all mounting holes and cut-outs are included in the initial cutting stage to avoid the need for drilling later. The “Flat Pattern” feature, like any other in the model tree, can be suppressed and resumed at will. This allows you to toggle between the development and the folded up form of the model. However this should only be for viewing – as soon as you threaten to make a new feature the flat pattern is automatically suppressed and the new feature is inserted before it in the table. Mounting holes and other features are added to the folded form, and will automatically be placed correctly on the flat pattern. Once a feature has been made it is recommended that you call it something sensible by highlighting it in the model tree with the left mouse button and then, from the RMB menu, selecting **Rename**. There are 2 methods of making holes and cuts, the **Extrude** icon to the right of the screen and from the INSERT menu either **Extrude** or **Hole** (although “hole” will only do round ones).

2.4 Family Table

It is vital to define a Family Table for all folded components as they will be required in both bent and flat forms in drawings, and in the folded form in assembly models. The family table should be started as soon as possible so that the correct instance can be inserted. From the TOOLS menu select **Family Table** and the following box appears -



Select the “Add a Column” icon and the dialogue box shown appears:



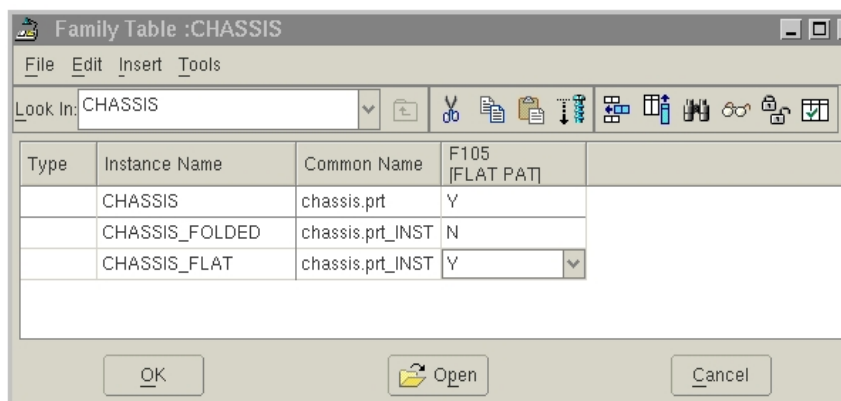
Highlight **Feature** under “Add Item” and select the Flat Pattern so that the form looks as the one depicted.

Accept selections with OK.

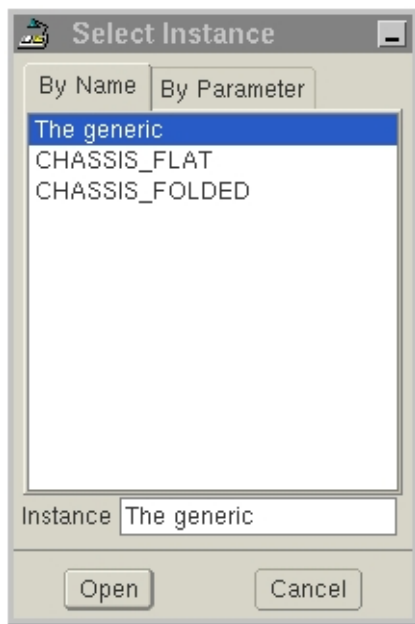
Returning to the Family Table window will show it to be partially filled in.

Selecting the “Add a row” icon  allows a further instance to be added.

Suggest it be called “chassis_folded”. Note that the generic will have the flat pattern turned on and the folded instance must have it turned off. It is also useful to have a further



instance of your chassis in the flattened state so adding a further instance will result in the Family Table shown below.



From now on, whenever this model is inserted into an assembly or a drawing you will be asked which instance you require. (Always choose the generic from the list of files in the **working directory** and choose the instance from the secondary dialogue box that appears, as shown opposite). Never put the generic model away with the Flat Pattern feature suppressed as this causes trouble with references in assemblies and drawings.

N.B. Do a Family Table before inserting the model into either an assembly or a drawing!

3 Evolving the Design

Should the structure of the chassis need modifying, during the design process, then there are several techniques to choose from.

3.1 Simple Modification of Dimensions

To modify the overall size of the chassis one can return to the original solid extrusion. To increase, for example, the overall length of the chassis is straightforward. Select the original extrusion from the Model Tree and on the RMB menu select **Edit Definition**. The relevant dimensions can now be simply altered, although care must be taken that other features such as holes are still in the desired position. See below.

However the same approach cannot be used if it is required to increase the depth of the skirt. This cannot be achieved all round the chassis, as the tabs bent down in the central portion must remain narrower than the slots from which they are cut. Therefore the original extrusion cannot simply undergo a dimension modification but must be Redefined to include a cut giving a step in depth around the problem area. This creates more surfaces and so the First Wall feature must be redefined to produce the desired shell. This, in turn, means that the SMT Conversion feature may need redefinition to re-establish the rips and bends and to achieve this any corner reliefs must be temporarily removed as their references will go missing in the process. This is, clearly, too complicated and so it is recommended that Redefinition of the original feature is avoided and simpler techniques applied.

3.2 Modification of walls

The most useful tools are given by the two icons below.

Create Flat Wall



Create Flange Wall



“Flat” wall simply means that the shape is defined by sketching on the flat face, it may still be attached to the existing wall via a single bend of any angle.

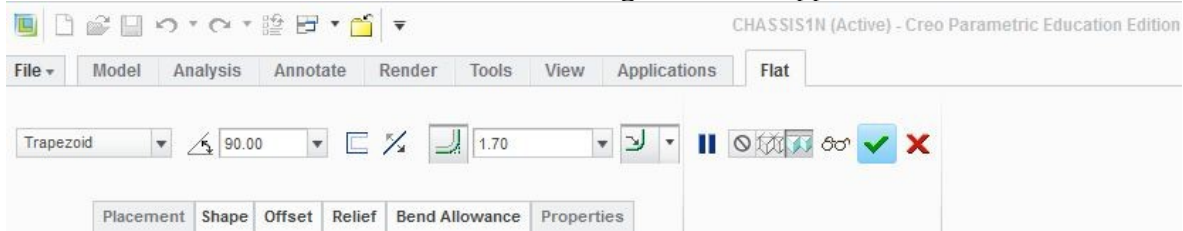
“Flange” wall is effectively the same as “extruded” and means that the sketch is edgewise on, therefore complex bend profiles may be defined although they must be constant over the length of attachment. This command allows attachment to curved edges.

Both of these commands operate with a dashboard type display of parameters to be defined, with the provisional wall shown in lime-green so that the effect of any alterations can be seen graphically before being committed to.

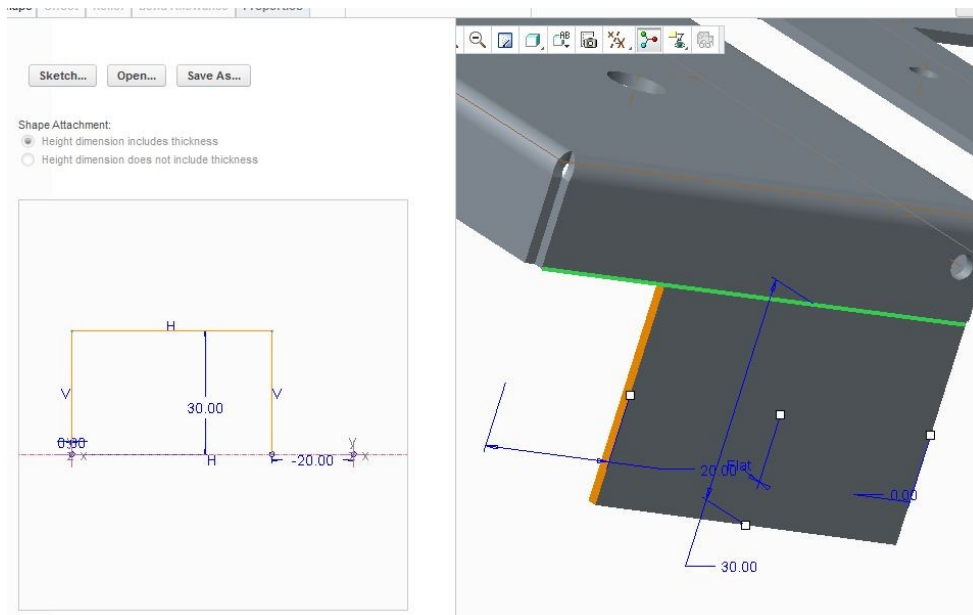
Both have the facility for calling up standard shapes whose dimensions can be edited on a simple graphic menu, or to be user-defined by entering the sketcher.

3.3 Non-uniform Extension of walls

To extend walls by an amount varying over the length of the edge in question, choose the **Create Flat Wall** icon. The following dashboard appears.



Pick the edge to attach to and set the shape to **Rectangle** as well as the angle to **Flat**. Under the SHAPE tab a simplified sketch can be displayed and edited as shown below :-



Note that the effect of changes can be seen “live” in the yellow rendered wall in the main window. The example shown is a simple rectangular addition over the length of the attachment edge minus 16mm; observe that the first time the number 16 is entered a minus sign must be used. Ready-made shapes available for this feature include “**Trapezoid**”, “**L**” and “**T**” while any flat shape can be defined by entering the sketcher with the “**Sketch...**” button.

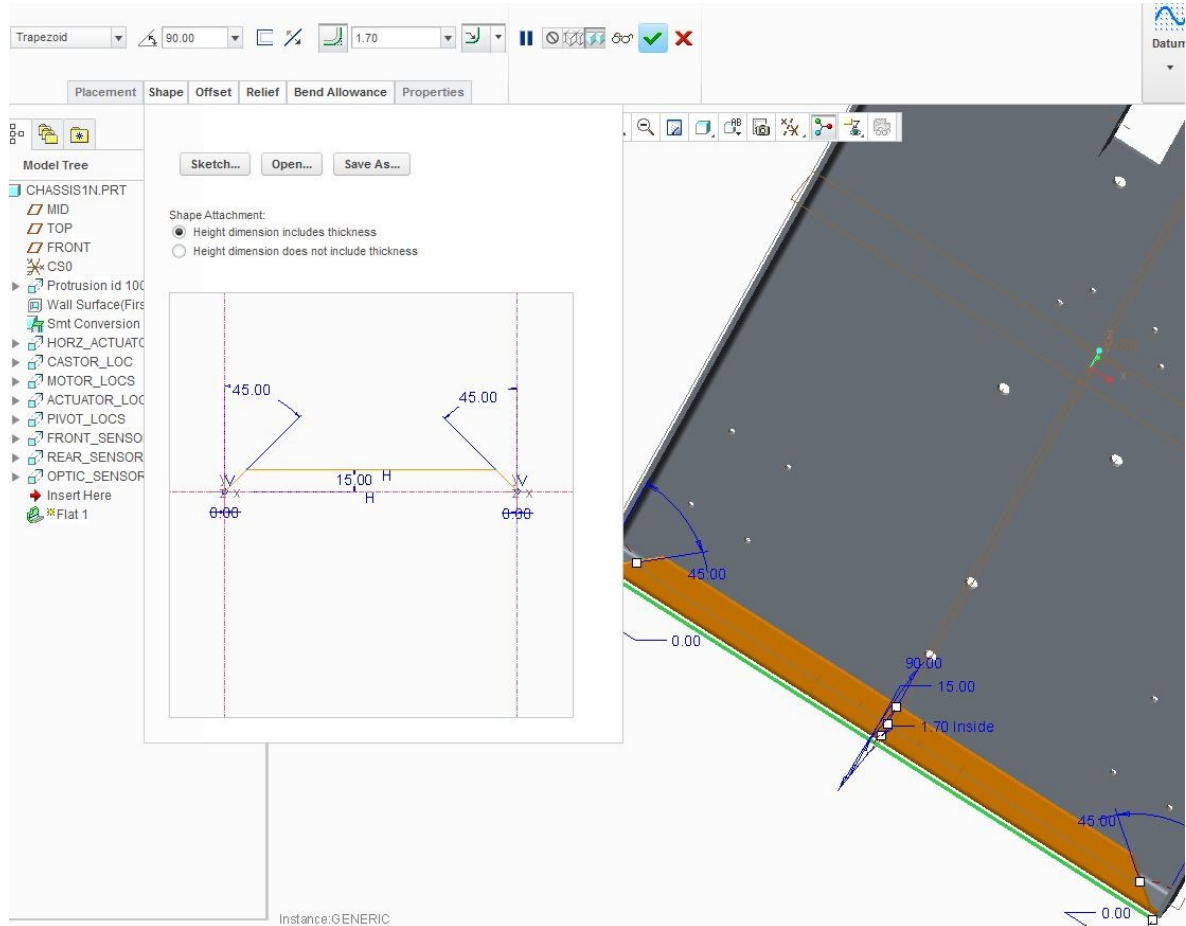
3.4 An Alternative Approach

To change the shape of a wall the alternative would be to cut a larger than needed wall down to that required. To effect this, select the “**Extrude**” icon and choose the **Mid Plane** as the Sketch Plane. Remember that the removed material must be defined by a “closed section” in other words the sketch must be a joined up shape. Once the material to be removed is defined and the sketcher exited then the extrusion depth should be defined as “Through All” and under the “Options” pull-down Side 2 should also be set to “Through All”.

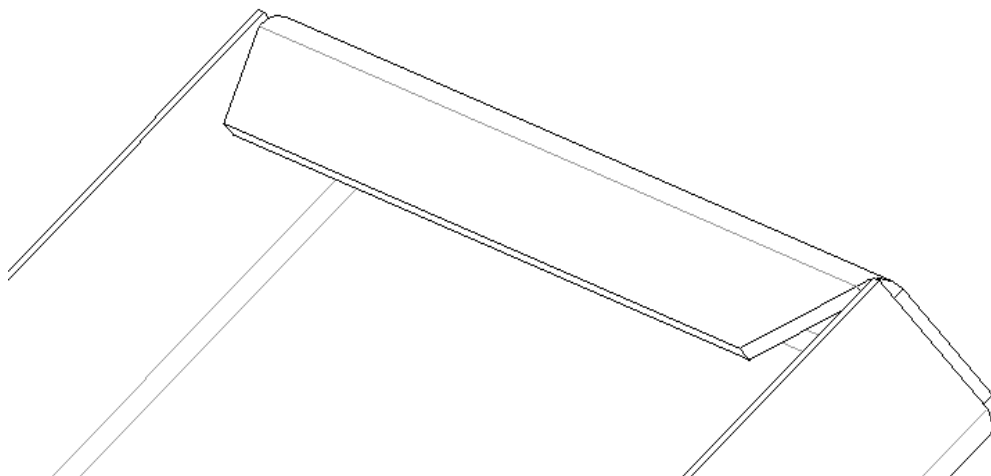
3.5 Adding Walls with Bends

This command also allows walls to be attached via a bend, by selecting the required angle in the field that had been set to **Flat** above.

In the example below, a box with 4 sides bent up has a further return added. As we require to make this addition to all 4 sides, the shape will be trapezoidal, with 45° ends, to avoid interference. Note the effect of the various flip arrows – modifications can be seen “live” in the model window.



Accepting the options above gives the result shown:



3.6 Copying of Features

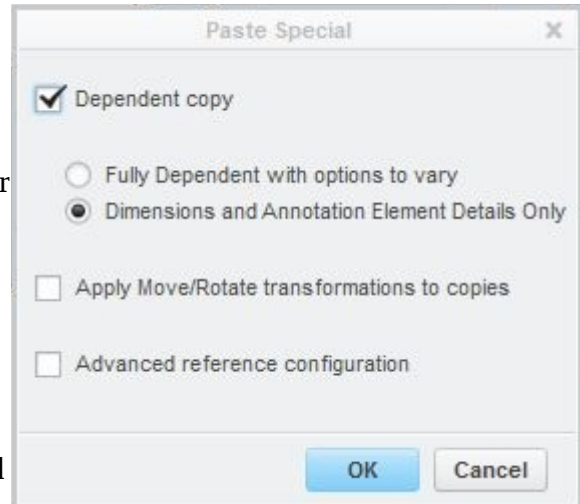
To repeat the feature on all 4 walls the “copy and paste” method can be used. Highlight the feature to be copied and under the EDIT menu select **Copy**. The **Paste** and **Paste Special** commands are now available.

Paste allows an independent copy to be created on any selected edge. However, in this case it would be preferable to have dependent copies, i.e. copies which change to match alterations made to the original feature.

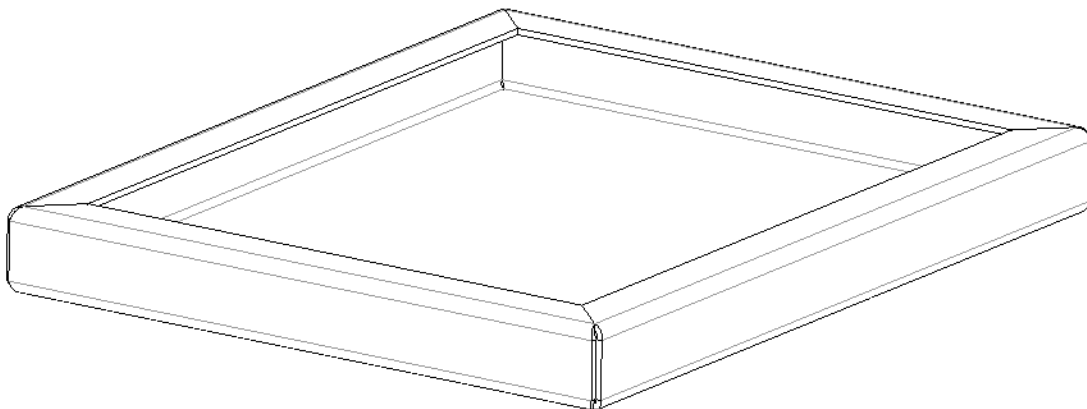


Therefore select **Paste Special** and in the dialogue box which appears (see opposite) tick the box for dependency as shown.

OK-ing this causes the dashboard to appear and it simply remains to select the attachment edge required for the copy. Ensure that if the external edge was selected for the original then the same is selected here and vice-versa.



Note that the “clipboard” is now empty and that further pasting operations require the master feature to be highlighted and copied each time. Repeating the procedure for each wall produces the structure shown below:

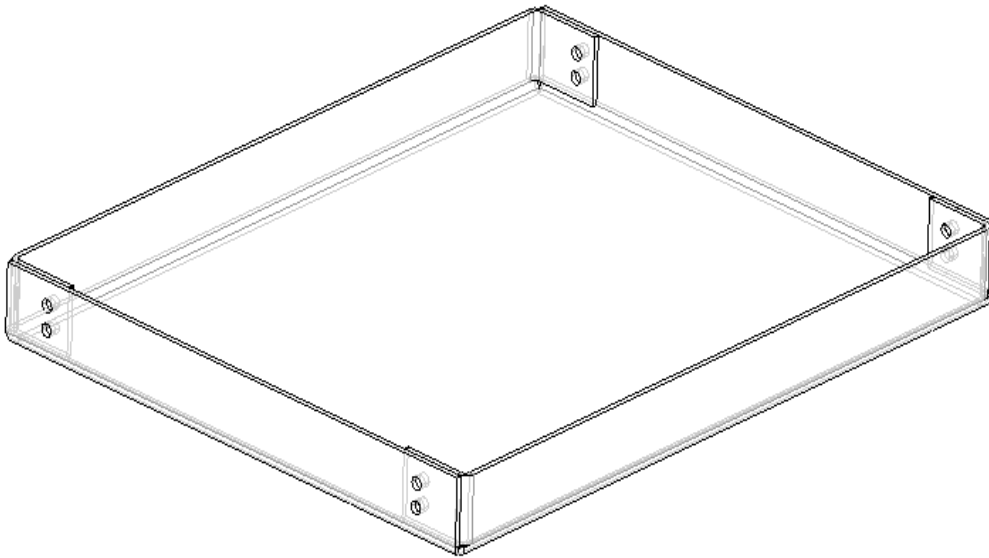


If the dimensions of the original trapezoidal wall are now varied then the copies will change as well. If, however, a difference was required then any dependent feature can always be made retrospectively independent by highlighting it and using the **Make Sec Indep** option from the RMB menu.

Note that, in this case, the new walls all extend over the entire length of their attachment edges despite being of differing lengths. This is because they are defined as doing this. If the original feature had been sketched and had been defined with all finite dimensions rather than attached to ends of existing entities, then some modification of copies might have been necessary to achieve the desired result.

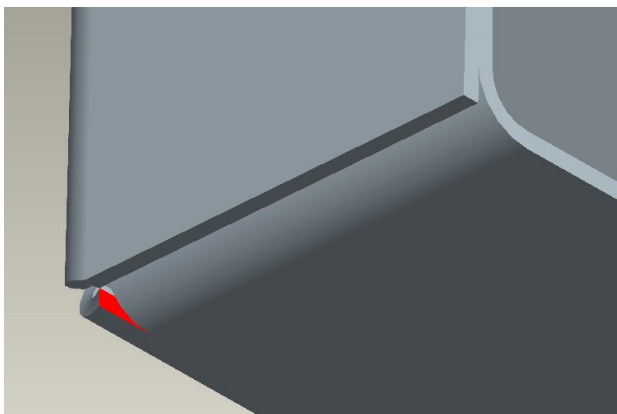
3.7 Tabs Requiring Bend Relief

In our example, it is desired to stiffen the chassis by extending the side walls into tabs to be bent round and pop-riveted to the end walls, as below.



The neatest result is obtained by folding the tabs behind the adjacent end wall. This can be achieved with the same **“Create Flat Wall”** command which will ensure correct, flush alignment of surfaces automatically.

Enter the dashboard for this command as before, highlight the attachment edge (it is suggested that a consistent choice of outside edge is chosen) and define the shape. A nice little wrinkle is to use a trapezoid with a very slight angle (e.g. 2 degrees) to provide clearance from the top edge and from the bend at the base, in case the bending is not quite 100% accurate.



At the bottom corner of the tab, where its bend meets the bend between base and side wall, there is a region of infinite deformation. This must be resolved by some sort of relief and the default (set under the dashboard heading RELIEF) is a simple rip, giving the effect shown highlighted opposite.

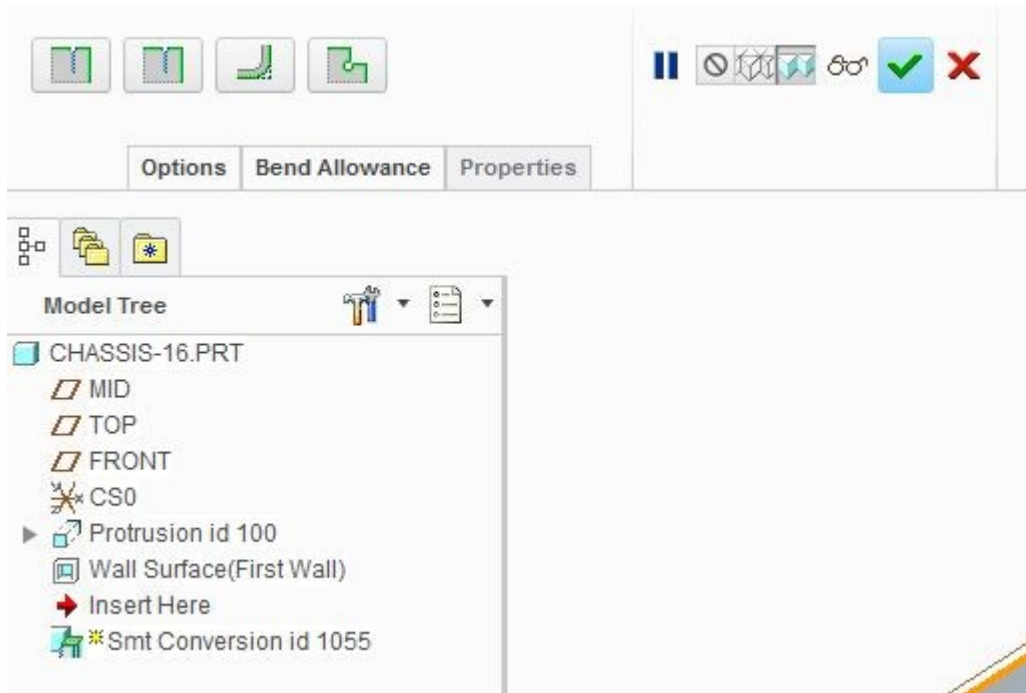
This is actually undesirable, as it would be both difficult to bend and would leave a dangerous sharp corner.

Instead some material would be removed from this area to avoid the situation altogether. Although there are bend reliefs available from the dashboard defining the wall in question, these do not work very well when adjacent to other bends (their purpose is to lie between bent material and flat material, as shown later).

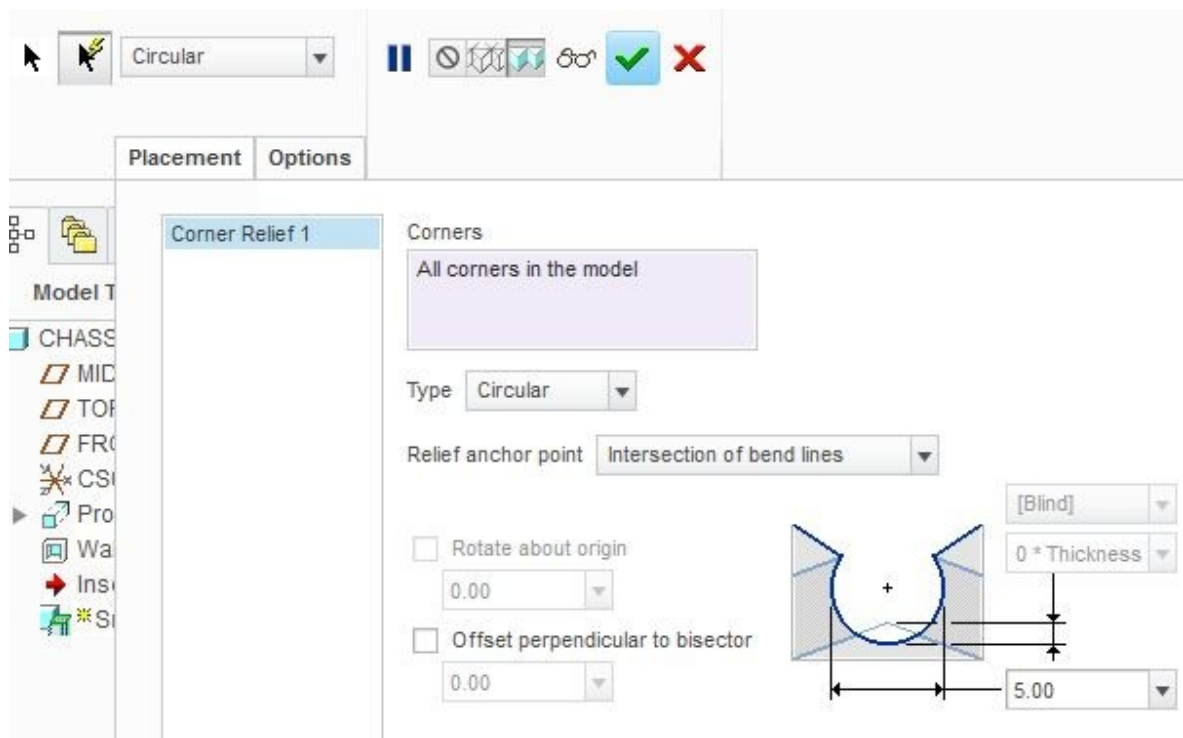
There are 2 recommended methods to resolve this:

1 - Using original Corner Relief

Return to the SMT Conversion feature and edit its definition.



Select **Corner Reliefs** from the SMT Conversion definition and then select **Circular**. Choose **Enter Value** and set diameter to 5mm (This is the value required to avoid all traces of a rip relief).



Accept by selecting the tick, twice. This results in the Flat Pattern now showing large circular cut-outs at the corners. However, if the Flat Pattern is suppressed, it will be observed that the corners look the same as before on the folded-up version (ie with no relief)

). This is because the cuts cannot be computed until the fabrication is flat and, at the moment, the Flat Pattern feature is the first time that the sheet has been flat.

To observe the effect in the folded-up model, it is necessary to insert two features into the model tree.

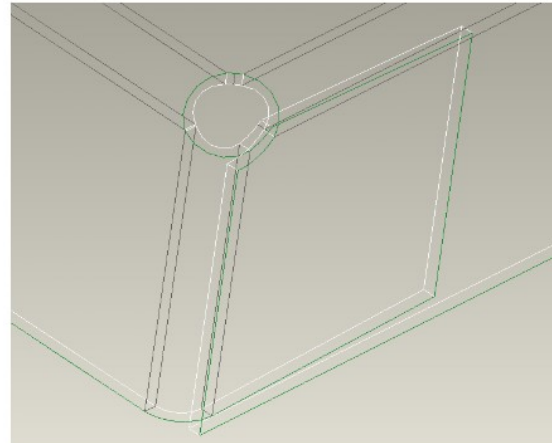
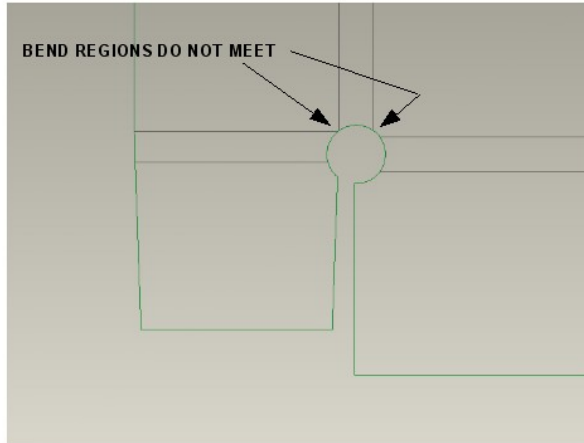
First the **Unbend** feature, using the icon shown.



Follow this with the **Bend Back** feature, using this icon.



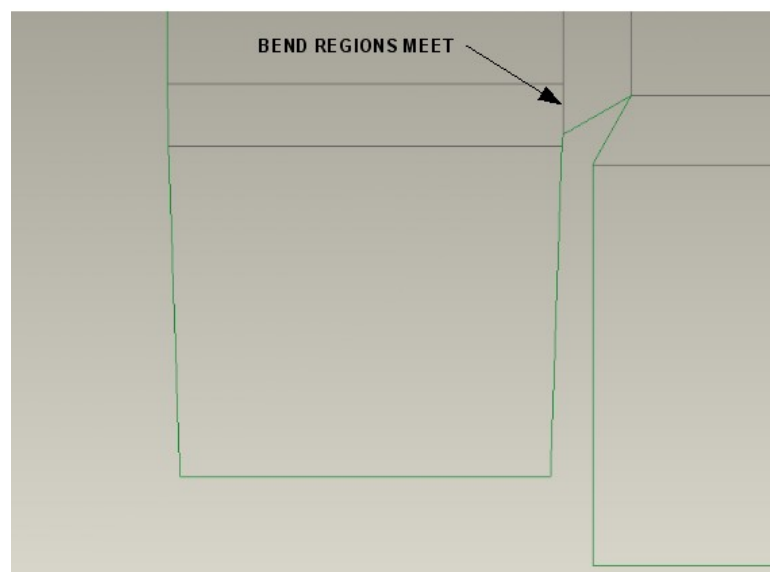
For both of these features a surface or edge must be selected to stay fixed, (make it the same reference for both features) and the options: **Unbend All** and **Bend Back All** should be chosen. Now, if the Flat Pattern is suppressed, the folded corner looks like this:-



Note that the regions of bent metal are separated by the circular relief.

2 - Using relief added by hand in the unbent state

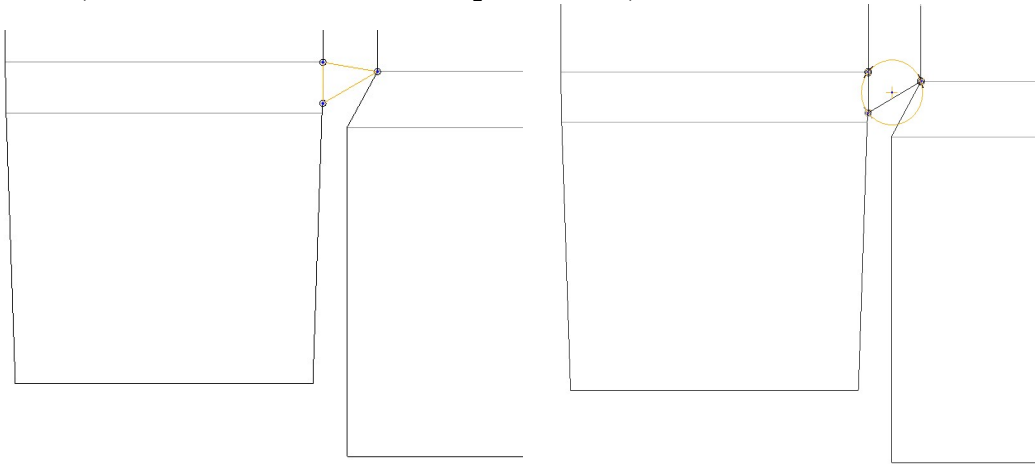
The method above does result in quite a large hole in the corner but it is possible to remove the minimum amount of material if done manually. Add an **Unbend** feature as above and examine the flattened corner detail as example below.



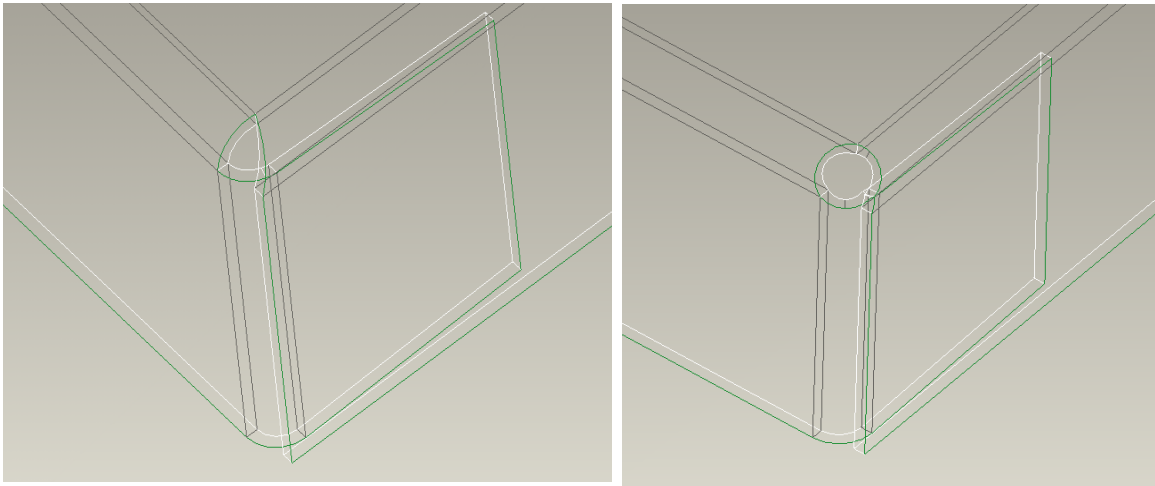
A sheet metal cut can be inserted into the flattened material to remove the line shown, where two bent regions meet.

The first sketch shown below removes a triangle, whereas the second removes a circle attached to points at the junctions of the bend zones (diameter comes out at about 3.4

). Note that in both cases there are no dimensions because existing features are used for all definitions. **(Hint: In the sketcher, add 3 points first.)**



Their effects in the folded state are shown below.



The cut feature can be mirrored onto the opposite corner by selecting it (either from the model or the tree) and choose **Mirror** .

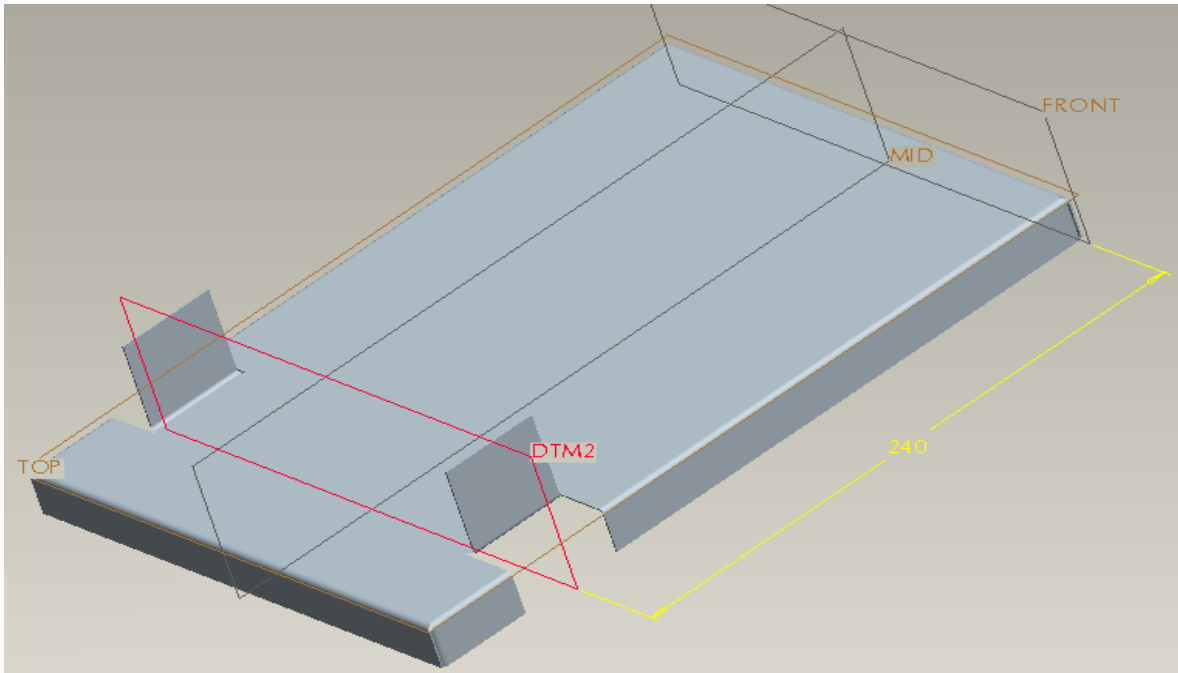


A mini dashboard appears, on which the only item which must be filled in is the mirror plane selection.

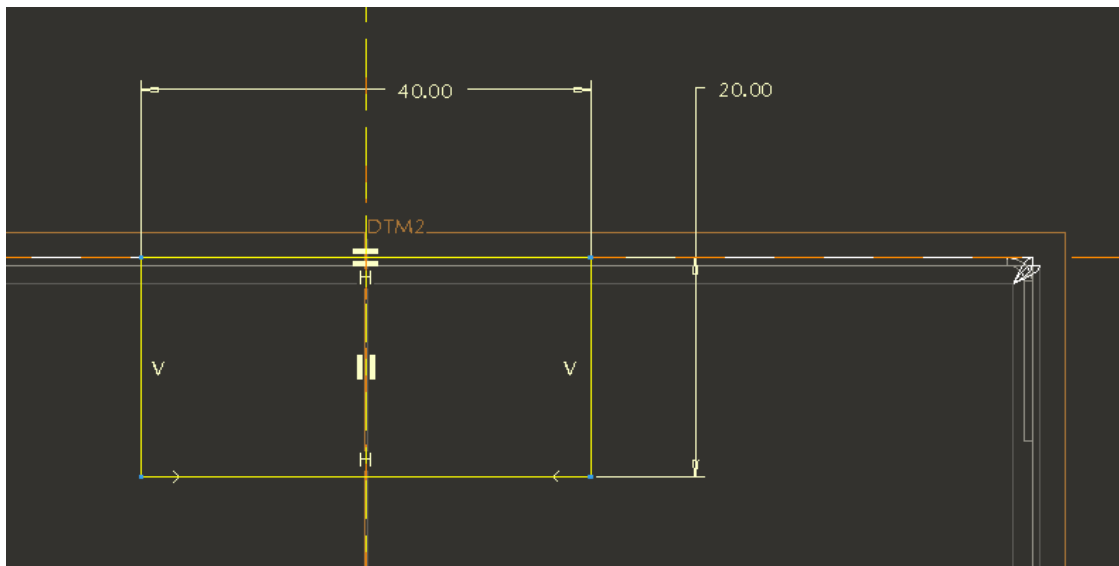
To mirror onto the back corners a datum plane could be inserted halfway down the chassis. Alternatively either 2 or all 4 corners could be cut on the same sketch.

3.8 Adding Bent Tabs from a Flat Background

The same “**Create Flat Wall**” command allows tabs to be bent up from surrounding flat material in the manner shown below. In the example being used it is first necessary to cut away the side walls where the tabs are required. It is also desirable to be able to alter the positions of these features by editing one single dimension and so an auxiliary datum plane may be inserted at an offset from the front plane.



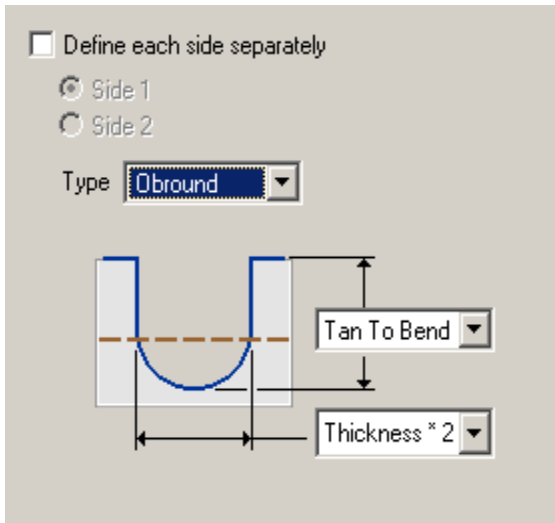
Use **Extrude** to remove a rectangle of material symmetrical to auxiliary datum plane(DTM2) and attached to outer surface of side wall. Only two dimensions will be shown on the sketch, width and depth as seen below.



This cut may be mirrored about the MID datum plane, using “**options**” on the dashboard to make the copy dependant, (thus ensuring ease of subsequent modification while maintaining symmetry of chassis).

Select **Create Flat Wall** icon and pick edge of cut for attachment as shown. Define shape, angle and bend radius.

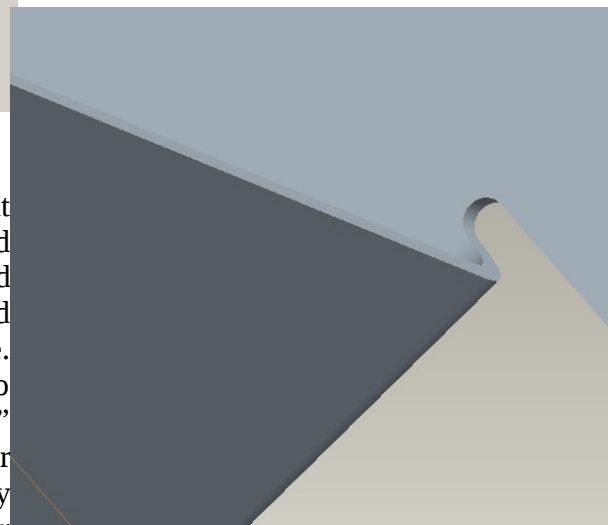
This case is where the inbuilt relief options are particularly useful. Open the RELIEF menu from the dashboard:-



It is recommended that the options shown opposite are used -

- ◆ Leave both sides the same
- ◆ Choose **Obround**.
- ◆ Select “**Tan to Bend**” as the length.
- ◆ Select “**Thickness x 2**” as the width.

Note that the material removed is from the surrounding flat area – the tab width is unaffected. Therefore the width of the original cut must be the tab plus two desired reliefs.

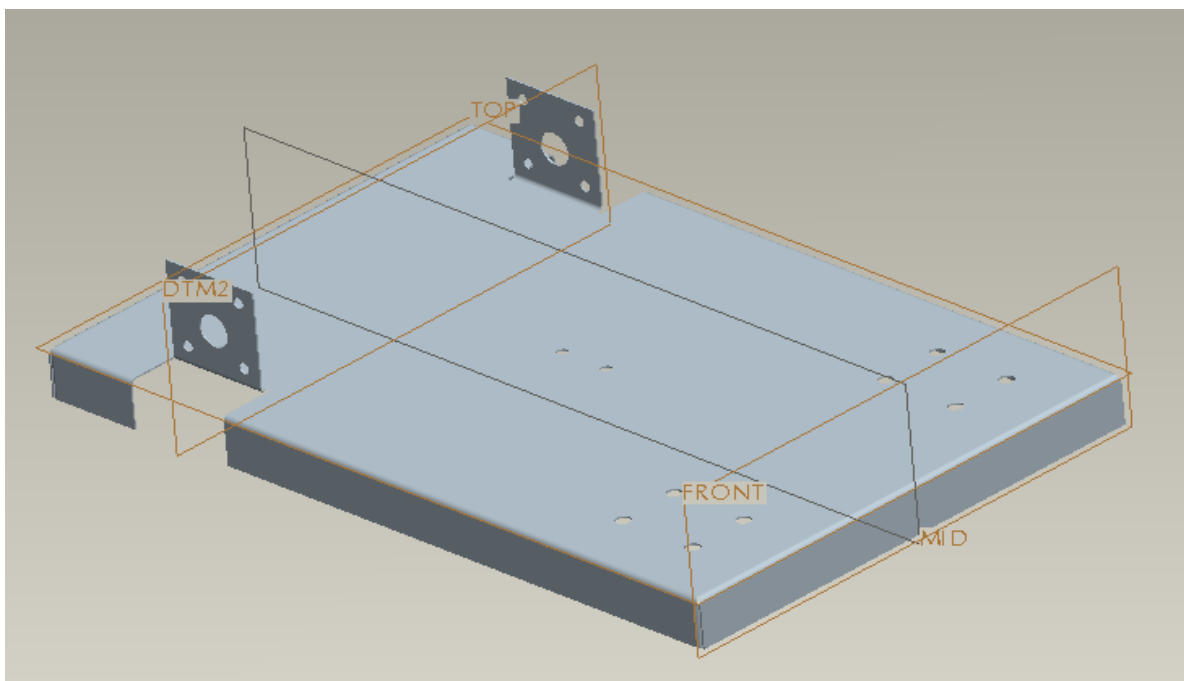


These choices result in a flat pattern that is both easy to cut and to bend and which produces the result pictured opposite.

This feature can now be copied to the other cut using the “Copy & Paste” method previously described. Remember use “**Paste Special**” to make the copy dependant. Note that using the mirror function on this feature does not produce the simple result expected and is unnecessary if

the tab is, itself, symmetrical.

Cuts can be added to these walls, noting that if MID datum plane is selected as the sketching plane then “**Through All**” produces the result below. Reference the cuts to DTM2 horizontally and surface of chassis vertically.




Should the model need modifying then the position of the whole feature can be controlled with the dimension to DTM2 and the width of spacing by the depth of original cuts. The width of brackets is controlled by the width of original cuts and the height of brackets by the bracket definition. The cuts and the holes should always remain symmetrical to DTM2 as this is how they were originally defined..

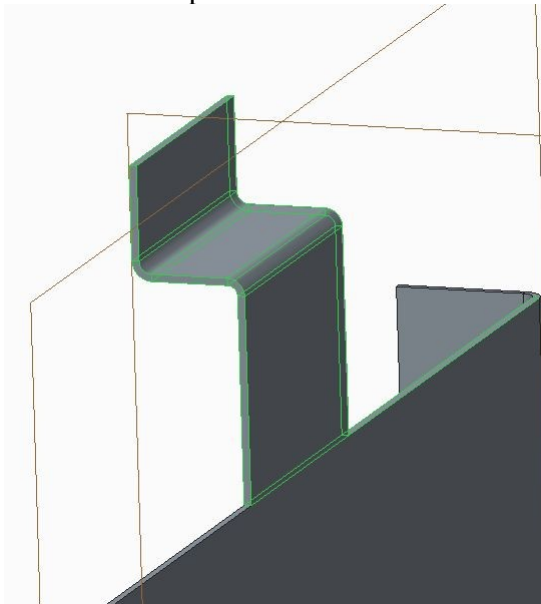
It is always a good idea to spend a little time considering how to define positions and what datums to refer dimensions to in order to simplify subsequent modification.

3.9 Create Flange Wall

All the above modifications and additions have used the **Create Flat Wall** function. However, occasionally, it is useful to be able to create an extra wall by sketching on its edge and then extruding the profile, as, for example, when multiple bends are required.

To create an integral sensor bracket, for example, the dashboard shown below is entered by selecting the icon - 

Select the desired attachment edge for **Placement**, under **Profile** go to the **User Defined** option and enter the sketcher (sketching on the MID datum plane), specify the length as symmetrical about the sketch plane.



Other possibilities offered by this command include the ability to attach to a curved edge but to manufacture this type of feature would require more sophisticated forming equipment than is available. Also, under the standard profiles are included various types of “safety edge” where material is folded to avoid sharp edges, but these are not considered relevant to the IDP.

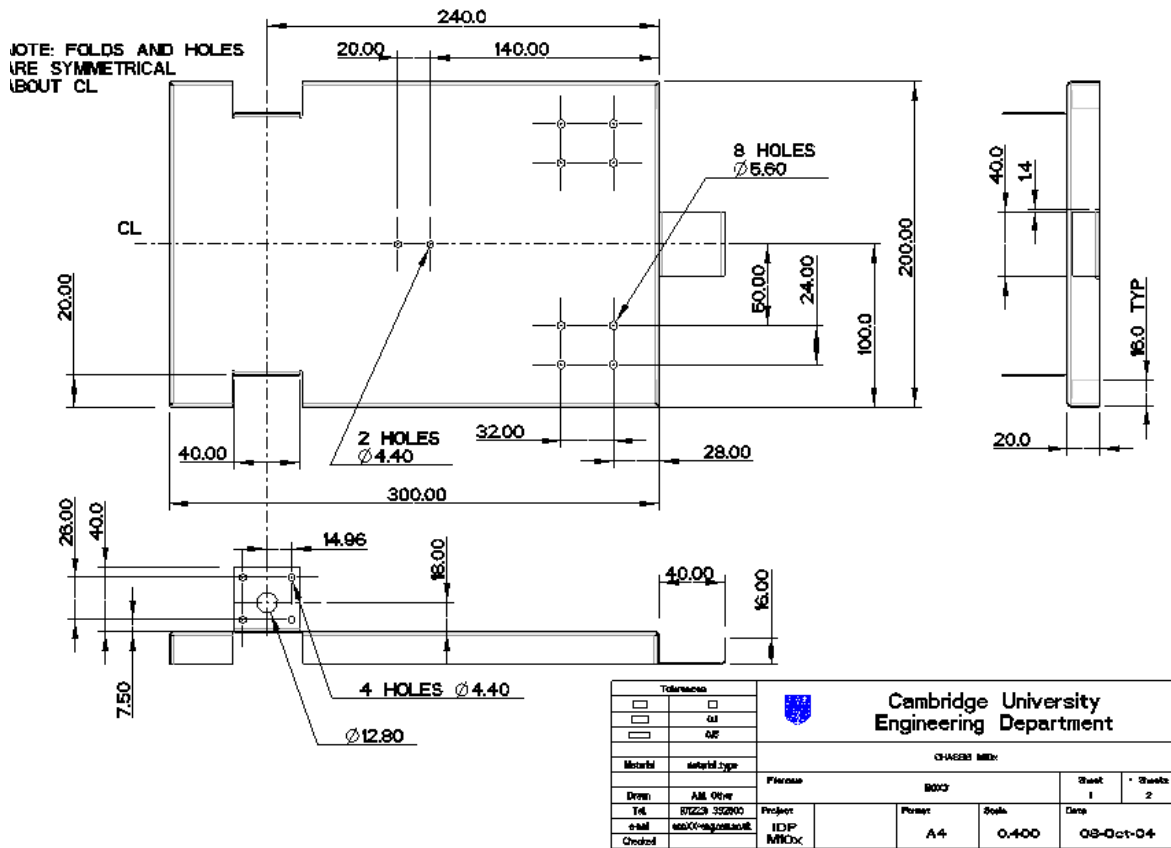
Part 4 - Output – Drawings and DXF's

Drawings must be produced to gain Design Approval. The requirements are:

1. A general assembly drawing together with enough prints of rendered models to convey the operating principles of the design.
2. Dimensioned drawings for all components to be manufactured. In the case of the chassis and other sheet metal parts these should comprise views of both the folded form and the flat pattern.

4.1 Drawings

Start a new drawing and select “**Empty with Format**”, browsing for a suitable CUED border. When in the drawing, add both “generic” and “folded” instances of the model – from RMB menu choose **Properties** and then **Drawing Models**. Select **Add Model** from the DRAWING MODELS menu and insert chassis model twice, once in each instance. Use **Set Model** from the DRAWING MODELS menu to determine which instance views are added from.



As most features have been defined in the folded state, it is best to dimension these in the folded views.

Overall dimensions and the positions of the main groups of holes should be shown. It is necessary to display centre-lines by selecting them from the dialogue box accessed via the icon shown.



The dialogue shown now appears and the right-hand tab will display centre-lines once the view(s) required have been highlighted.

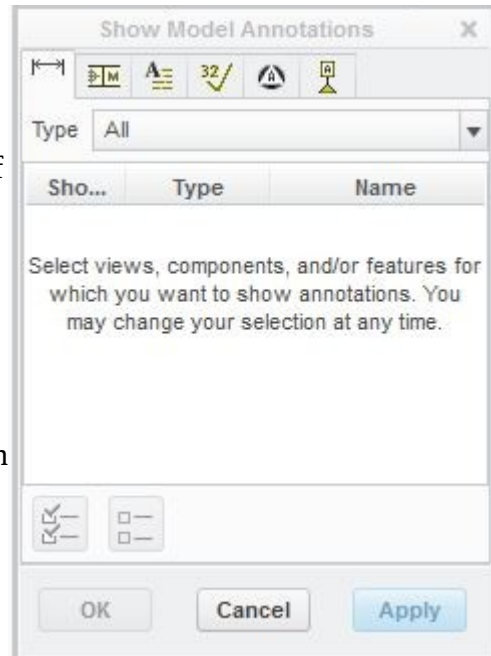
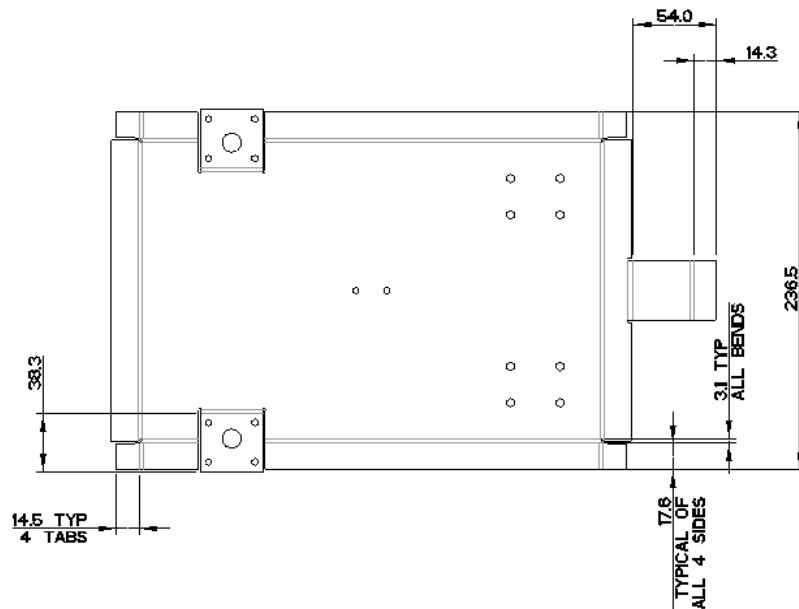
Once the centre-lines are displayed manual dimensioning is much simpler as the centre-line itself may be used as a reference. Automatic dimensioning does not really work with sheet metal, so it is recommended that dimensions are inserted manually by selecting the icon shown.



The box shown is displayed at this point and as a default the option “On Entity” should be selected, although other options allow for the centre of circles or radii to be used etc.

It is not necessary to dimension every last hole as this information is contained in the DXF file. It is permissible to use a note about symmetry to minimize the number of dimensions, provided the intention is unambiguous. Note that, with sheet metal, it is not possible to rely upon the automatic dimensioning provided by Pro-E and it is recommended to insert dimensions manually.

It is still necessary to produce flat views (from the flat instance of the model) but the only features which **must** be dimensioned on these are the bend lines (which must be scribed by hand in the workshop later). It is acceptable to dimension to one side or the other of bends or even to their centre line, provided that dimensions appear stating the relationship between these. Such dimensions may be shown once as “TYPICAL” as in the example below.

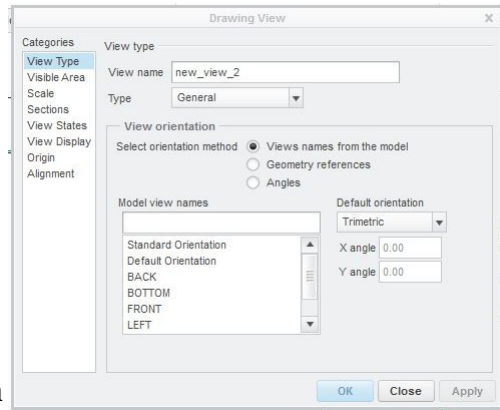


<input type="checkbox"/> Tolerances <input type="checkbox"/> ϕ <input type="checkbox"/> \pm					
<input type="checkbox"/> Materials <input type="checkbox"/> Material Type		Cambridge University Engineering Department CH0286			
Drawn	AM Otter	Project	8003	Sheet	2
Tel	01223 352800	Project	IDP	Scale	2
eMail	amotter@cam.ac.uk	Project	MFCx	Format	A4
Checked		Project		Scale	0.400
		Project		Date	08-Oct-04

Remember to alter any of the properties of a view, Highlight that view and on the RMB menu select “Properties”.

This will bring the dialogue box shown up.

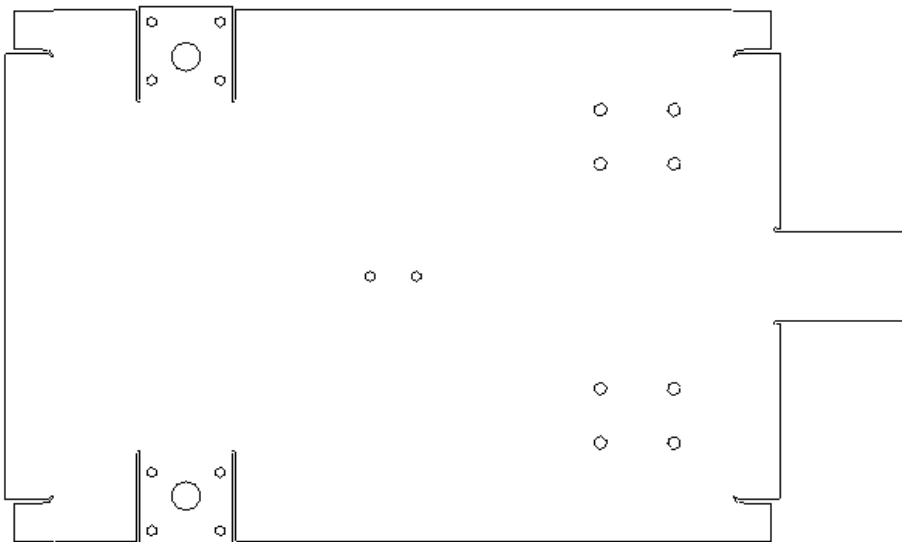
The most useful page being “View Display” which allows the display of a view to be locked into a line drawing mode, either “Hidden” or “No Hidden” line display.



4.2 DXF

The cutting machine requires a DXF file to be produced. Start a new drawing file, choosing **Empty with Format**, and select the format “1b_steel_450x450_dxf.frm” or alalloy_200x150. Attach the flat instance of the model and Add a Top View at a scale of 1:1, ensuring that no notes or dimensions are displayed. Highlight the view and from the RMB menu select properties. Open the **View Display** page of the dialogue box and change **Display Style** to **No Hidden** and change **Tangent Edges Display Style** to **None**. These actions get rid of hidden detail and bend lines so that the view will be left with only cut lines displayed. Choose command “Save a Copy” of the file and in the dialogue box change File Type to DXF. It is helpful to produce one DXF file for all components in 0.7mm steel and one (if applicable) for all components in 1.5mm aluminium. Do this by attaching all relevant models to the drawing and selecting new model for each view. **Take care that all scales are 1:1.**

Submit the DXF files via the bulletin boards accessed from the IDP website under “Project & Team Forums”. Upon logging in to this environment go to the Forum “Student submission of DXF files” and post your DXF files by pressing the button “NEW TOPIC”. Fill out the subject box with your team number and place some text in the main space including team number and material(s) required. Attach your DXF files by pressing the “Upload attachment” button towards the bottom of the screen. Browse for the required file(s) and press the “Add the file” button. When complete press the “Submit” button below the message. Note that there is a limit of 3 attached files per message.



Drawing file appearance prior to creation of DXF file.

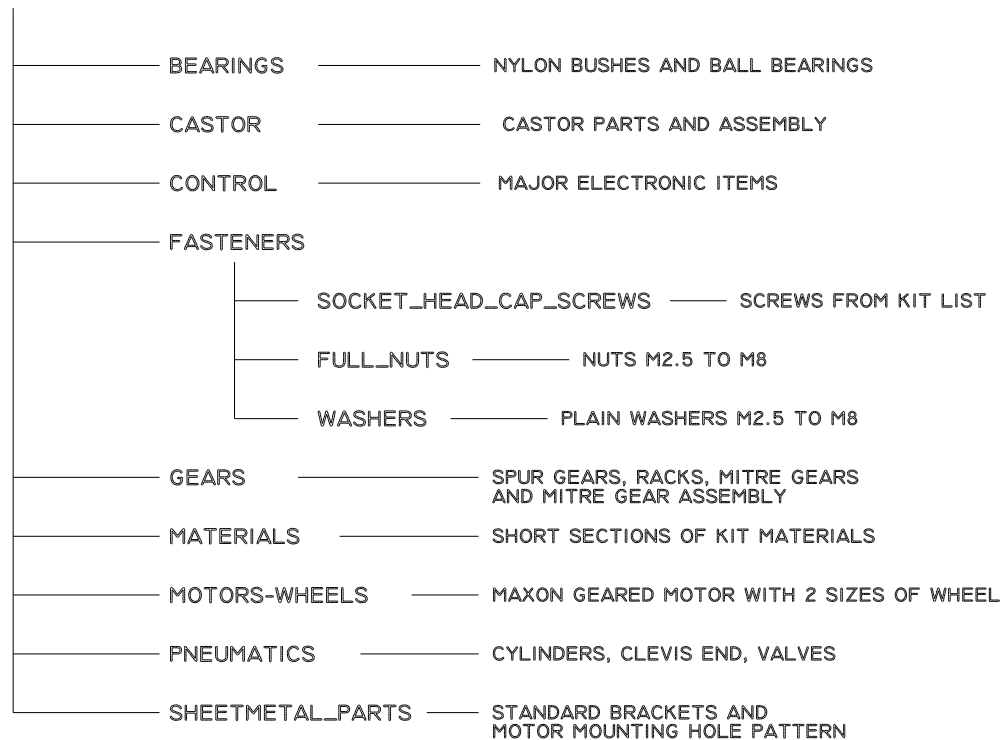
Determine the order in which bends are to be executed – if in doubt seek advice from workshop technicians. The cut sheet should have bend lines scribed upon it to show the start of each bend. Be careful to scribe upon the side of material that will be uppermost (and therefore visible) in the bending machine.

5 Assembly Modelling

5.1 Parts Library

It is strongly recommended that the entire assembly is modelled before gaining design approval and starting manufacture. To help with this, models of the standard component parts have been placed in the MDP library. This is accessed, from the “Open File” dialogue box, by changing directory to CUED Library and then selecting mdp_lib. The library structure is shown below and it will be found that each section has a description of the contents.

MDP LIBRARY

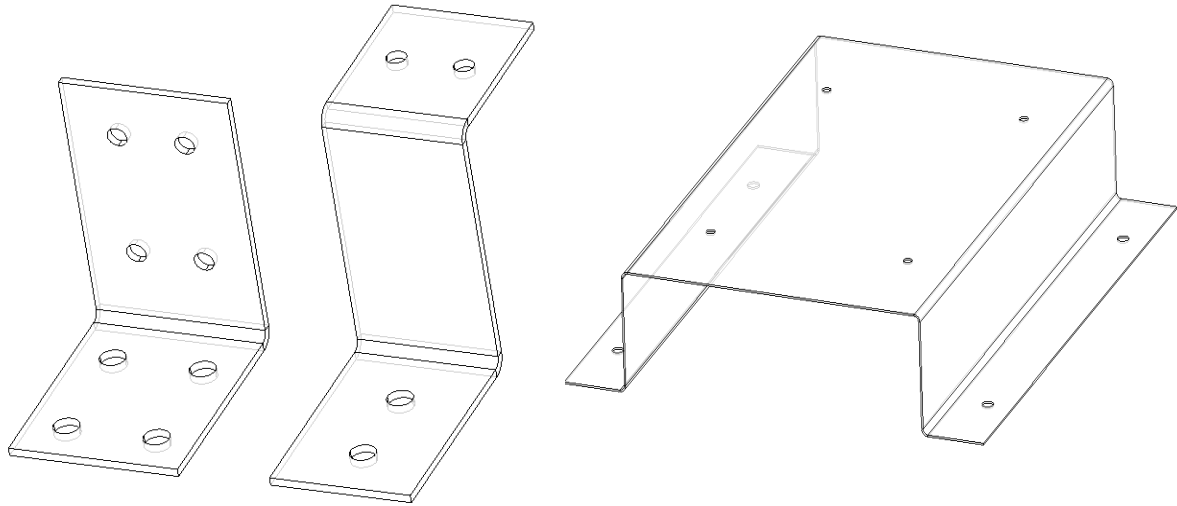


Use the Preview facility offered by Pro-E to see the file before opening it. Note that some of the items have been pre-assembled to save time. Also note that the gears are shown as smooth drums of the pitch-circle-diameter. To mesh the gears ensure that these circles are just touching. However to check the overall space envelope required by a gear, **Suppress** the final cut and a new outer surface (coloured yellow) will be revealed.

When building up a large assembly, it is much easier to use sub-assemblies, rather than bringing loads of individual components into the top-level model. For instance: assemble the motor, shaft extension and wheel (already done in the library), together with a suitable bracket of your own devising, into a drive-unit assembly before inserting this into the overall robot, thus saving having to do everything twice and minimising the number of datum features that need to be displayed when positioning things.

5.2 Standard Brackets and other materials

In addition to the chassis, there will, probably, be a requirement for auxiliary brackets formed from sheet metal. To save time, a small library of standard brackets has been compiled, which can be modified to individual requirements. Note that from wherever library files are obtained, they are kept as read-only. It will be necessary to save a re-named copy to allow modification. The types of brackets offered are shown below.



There are also short lengths of the bar and angle materials which may be used similarly.

5.3 Assembly Tips

When placing parts in an assembly model, remember that when one or two constraints have been made then the part may be moved by hand (in the remaining degrees of freedom only) by holding Ctrl & Alt while using the MMB for rotation and the RMB for translation. There is nothing wrong with leaving a component partially constrained while other parts and features are arranged.

If it is required to add a mounting hole to match a placed component, this can be done from within the assembly model by using the “Activate” facility. A typical procedure might be -

- ◆ Insert a component into the assembly and place known constraints upon it. (e.g.: mounting surfaces and orientation of faces and edges).
- ◆ Position the component by hand to roughly the right position in remaining plane(s).
- ◆ Activate the model to be cut (highlight and RMB menu). Observe little green diamond.
- ◆ Cut a hole to match that in placed component, possibly using concentric circles.
- ◆ Place dimensions on cuts using references in chassis model alone. Resolve redundant dimensions and constraints as prompted (by deleting coincidences).
- ◆ Tweak dimensions to round numbers as required.
- ◆ Accept cut feature if satisfactory.
- ◆ Re-Activate assembly model (highlight top level of model tree and use RMB menu).
- ◆ Return to definition of placement of partially constrained component and add constraint aligning mounting holes.

However, this technique is not suitable for use with instances of family tables. If it is used to place holes in the folded instance of a chassis then all sorts of problems will be experienced when the generic model is examined, from suppression of features to lost references.

A safer method for folded components is to insert the mounting holes into the generic model without worrying about their exact placement and then editing them in the assembly to bring them to the desired position.

The procedure is, therefore, amended as follows:

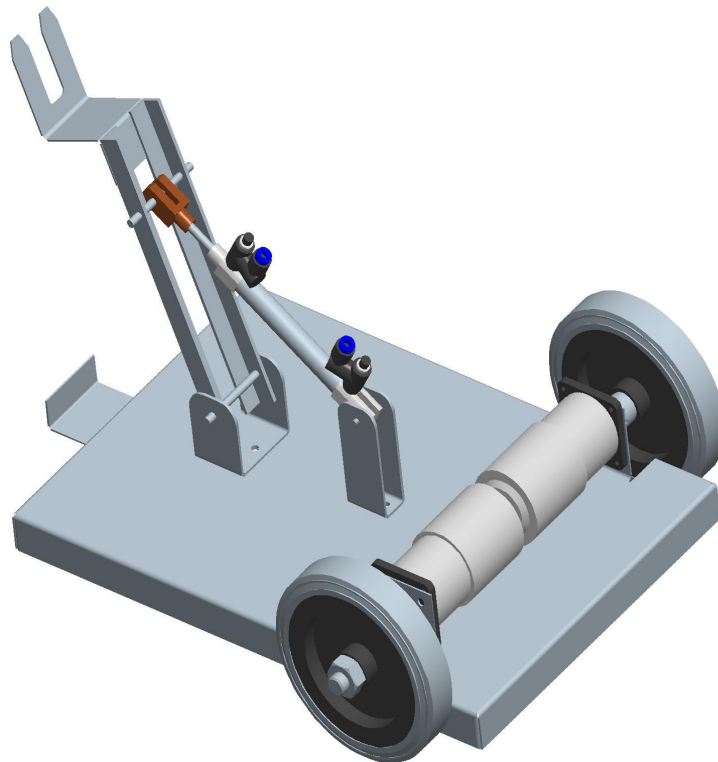
- ◆ Insert approximate mounting features into chassis model.
- ◆ Insert a component into the assembly and place known constraints upon it. (e.g.: mounting surfaces and orientation of faces and edges).
- ◆ Position the component by hand to roughly the right position in remaining plane(s).
- ◆ Edit holes to match those in placed component.
- ◆ Tweak dimensions to round numbers as required.
- ◆ Return to definition of placement of partially constrained component and add constraints aligning mounting holes.

5.4 Mechanisms

When assembling the elements of a mechanism **Connections** dashboard is used, rather than the static **Constraints**. The model can rapidly become over-complicated unless some simple golden rules are observed.

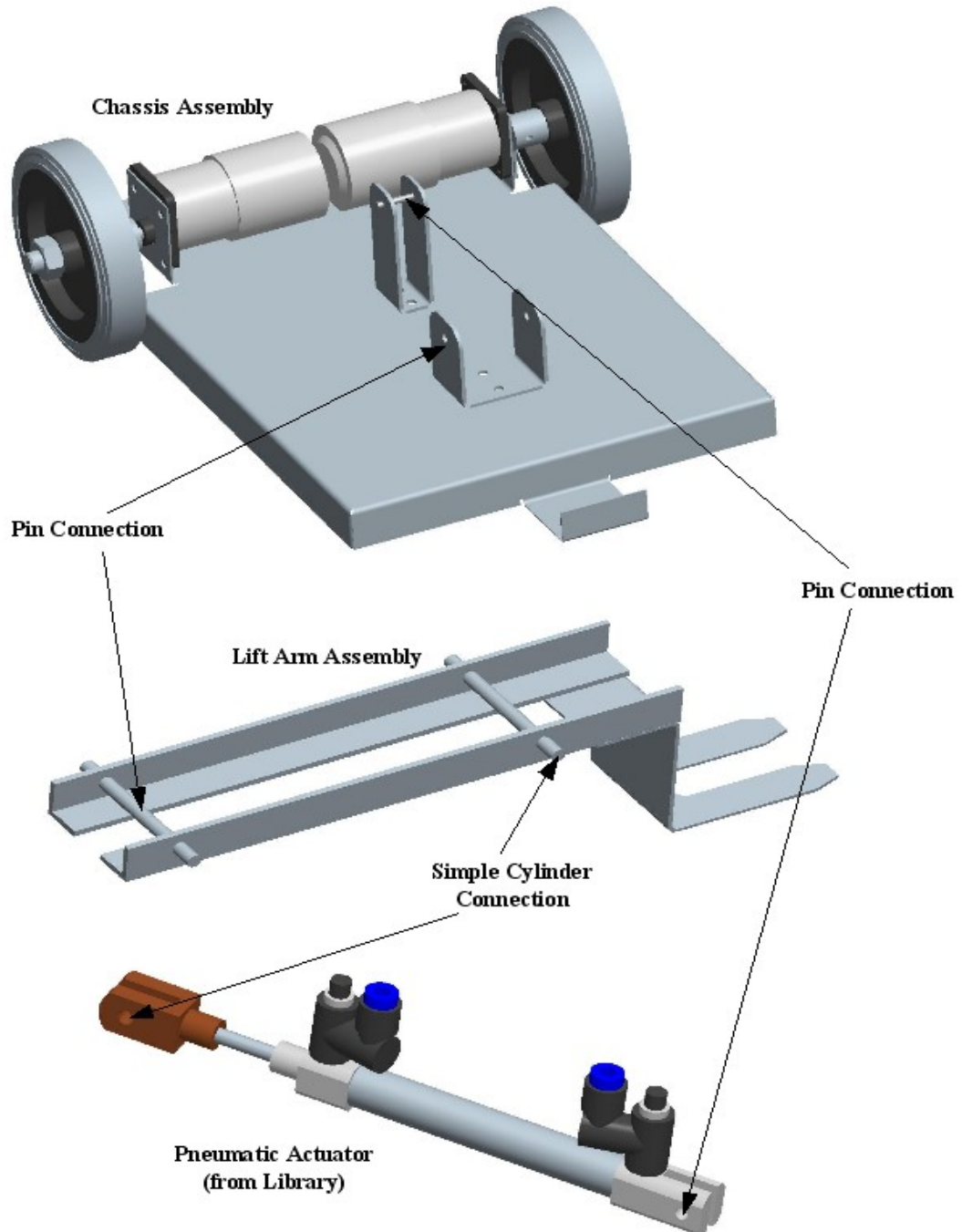
- ◆ When placing an item use either **Constraints** or **Connections** page of dashboard, never both at the same time.
- ◆ Seek to minimize the number of connections used to describe a mechanism.
- ◆ Avoid placing individual components into a dynamic assembly. Build up sub-assemblies of components which are statically fixed to each other and then combine these with dynamic connections in a higher level assembly which defines the mechanism.

In the example shown below, a lifting arm is mounted on the chassis and driven by a pneumatic cylinder.



The recommended approach is -

1. Build up the chassis assembly, with relevant mounting brackets.
2. Create the lifting arm assembly (by attaching component parts with static constraints).
3. Start a new assembly and insert chassis, fixed to main assembly frame of reference by using **default location** button on right hand side of constraints dialogue box.
4. Insert lifting arm assembly by selecting the dynamic connections dialogue and using a **Pin** joint.
5. Make a copy of the pneumatic actuator from CUED Library and insert this copy into the assembly with a pin connection at one end and a simple cylinder at the other (to avoid redundant axial restraint).



Index

FUNCTION	REFERENCE	NOTES
2 - Getting Started		
Starting with Solid Model	2.1	Entering sheet metal application and creating shell
SMT Conversion	2.2	Defining sheet metal features
Ripping	2.2.1	Deciding which edges are discontinuous and which bent
Bend Definition	2.2.2	Values of bend radii for materials used in IDP
Flat Pattern	2.2.3	Development of model as cut from sheet, before being bent
Reliefs	2.2.4	Adding corner reliefs and slots for ease of manufacture
Adding Holes and Cut-outs	2.3	Use of Flat/Folded models
Family Table	2.4	Allowing flat and folded instances to be used independently
3 - Modification Techniques		
Simple Modification of Dimensions	3.1	Modification of original extrusion & pitfalls
Modification of Walls	3.2	Extends wall uniformly over entire length of edge
Non-Uniform extension of Walls	3.3	Allows extension over part of edge and to varying lengths
Adding Walls with Bends	3.4	Attaching new bent wall
An Alternative Approach	3.5	Cutting away unwanted material
Copying	3.6	Adding multiple new features by “copy and paste” method
Tabs Requiring Bend Relief	3.7	Adding extra folded tabs and managing relief of material in corners
Bent Tabs from Flat Background	3.8	Adding bent walls to flat material, including relief
Create Flange Wall	3.9	Extrusion of wall with several integral bends
4 - Output		
Drawings	4.1	Drawings and Dimensioning
DXF	4.2	Producing files for cutting
5 - Assembly		
Component Library	5.1	Location and use of library of standard parts
Brackets	5.2	Notes on use of standard brackets
Assembly Tips	5.3	Some guidance on alignment of components
Mechanisms	5.4	Assembly with dynamic connections