A 2D sketches

1 In this first series of exercises, you have to create 2D sketches.

2 These sketches should then be constrained, dimensioned and 'converted' into a part model using one of the 3D part feature tools (e.g. extrude, revolve, sweep, etc.).

3 In each exercise, the procedure for creating the model is:
   (a) Start each exercise with a new metric standard (mm) .ipt file.
   (b) Complete the sketch using the information displayed.
   (c) Fully constrain the sketch.
   (d) Add the dimensions as given (or to your requirement).
   (e) Extrude or revolve the dimensioned and constrained sketch.
   (f) View at a suitable 3D viewpoint.
   (g) Save your completed part model to a suitably named folder.
   (h) Use discretion when creating the model, as appropriate.
   (i) *Note*:
      - Some of the sketches are relatively simple, but I have also included others to make you 'think a bit'.
      - If you are relatively proficient at creating part models, then you may want to miss this chapter, although it is still good practice – this is your decision.
      - In the exercises, I will display the 2D sketch with dimensions and the created part model from the sketch.

Terminology

Sketch

1 A sketch is a plane on which 2D objects are drawn (i.e. sketched).
2 The plane can be XY, YZ, XZ or a user-positioned work plane.

Constraints

1 Geometric constraints apply behaviour to a specific object or create a relationship between two objects.
2 For example, a line may be constrained to be horizontal or two lines may be constrained to be equal in length.

Dimensions

1 The dimensions 'added' to a sketch control the size of the sketch and will be displayed in the drawing view when generated.
2 Inventor dimensions are *parametric* (i.e. if a dimension value is altered, the shape of the object (model) will also be altered).

Base feature

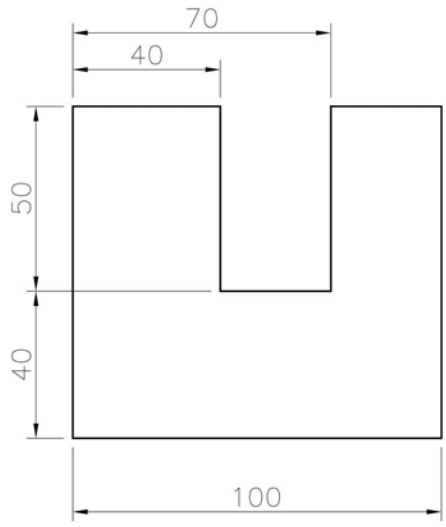
The first sketch of a part that is to be used to create a 3D feature is referred to as the base feature.

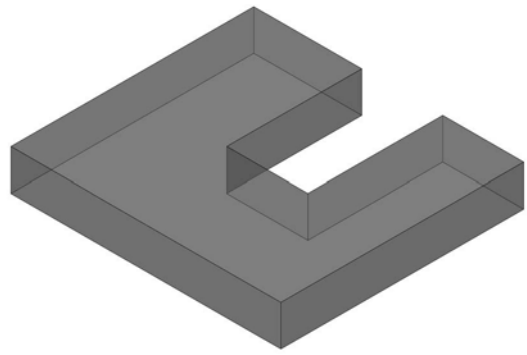
3D feature

When a sketch has been extruded or revolved etc., the result is termed a 3D or part feature.

Exercise A1: shim

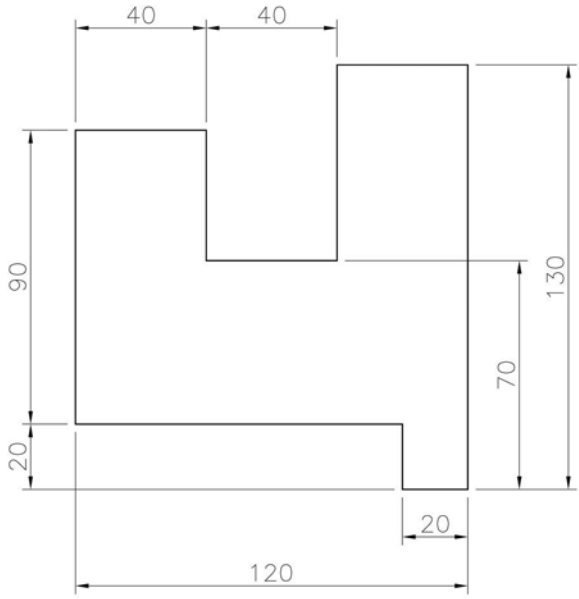
The sketched profile has to be extruded for a distance of 15mm.

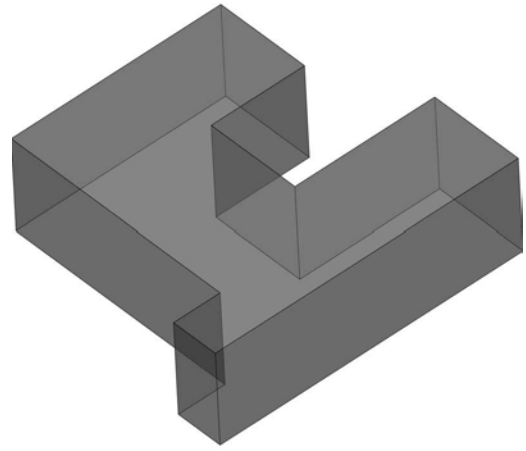




Exercise A2: spacer

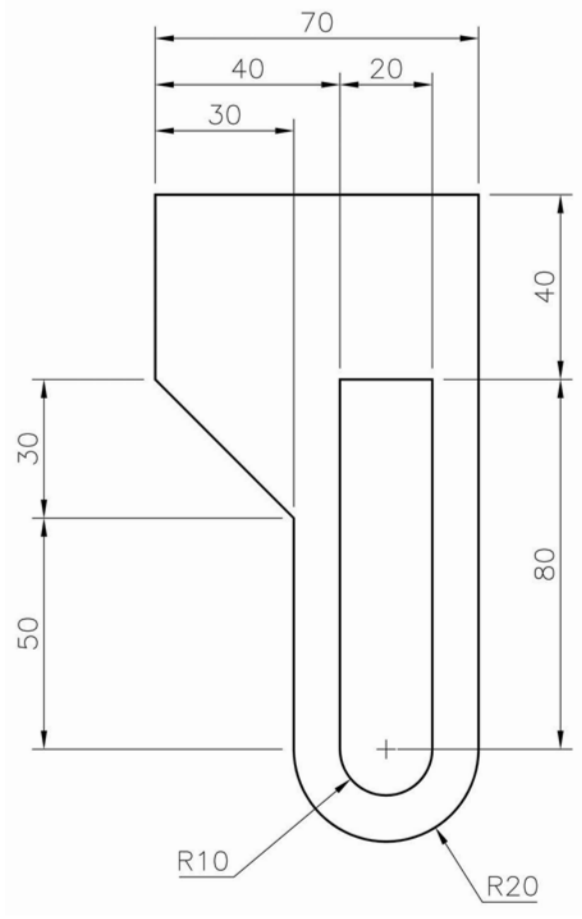
The dimensioned sketch has to be extruded for a distance of 20mm.

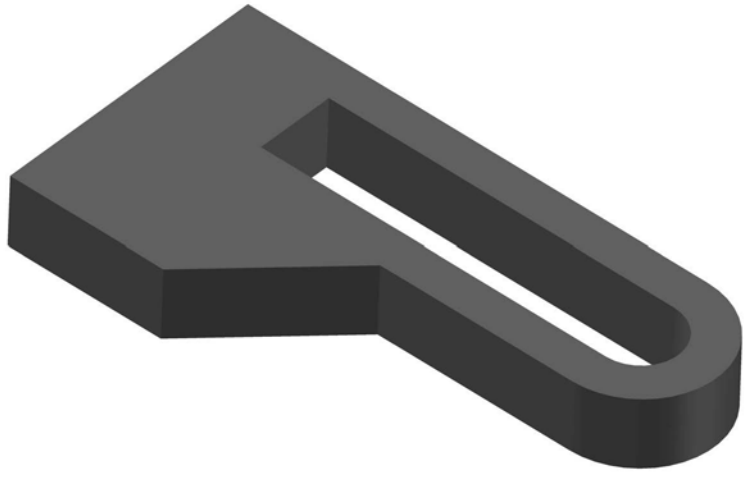




Exercise A3: lock guide

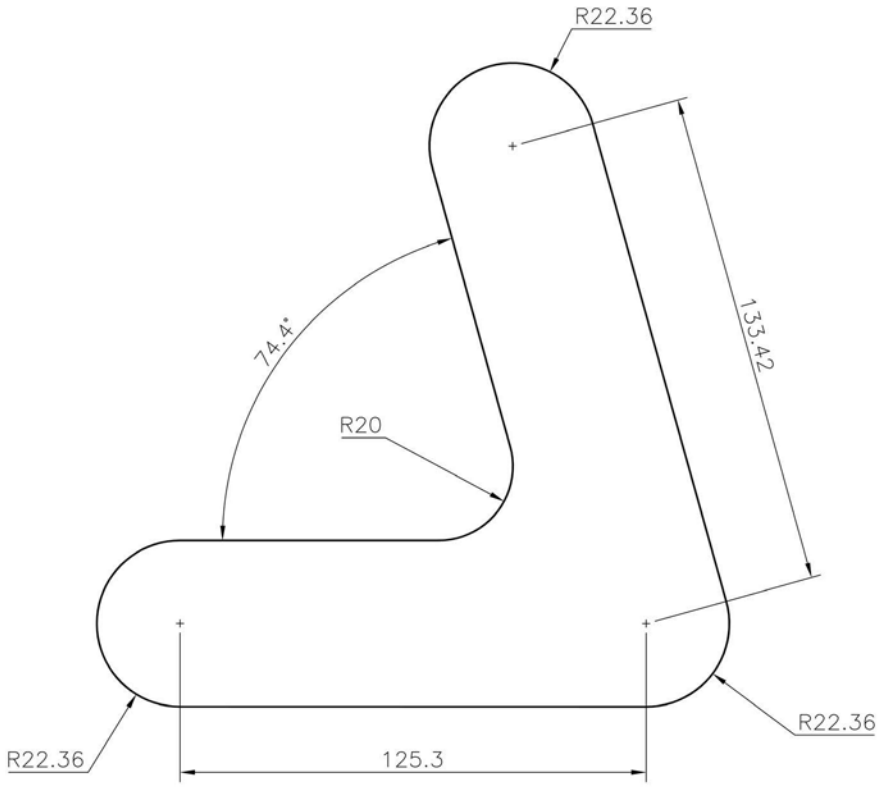
Extrude the sketch for a distance of 18mm.

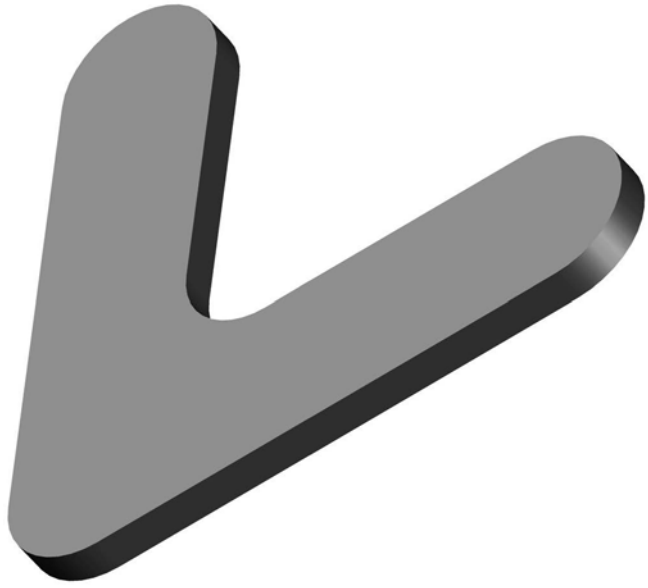




Exercise A4: rocker arm

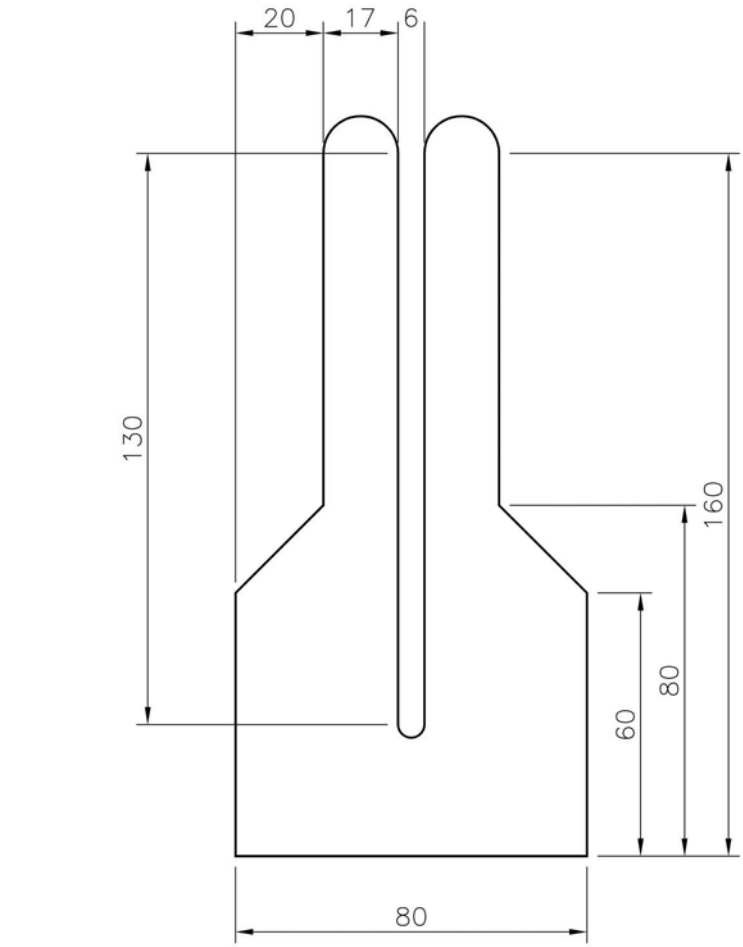
Extrude the sketch for a distance of 10mm.

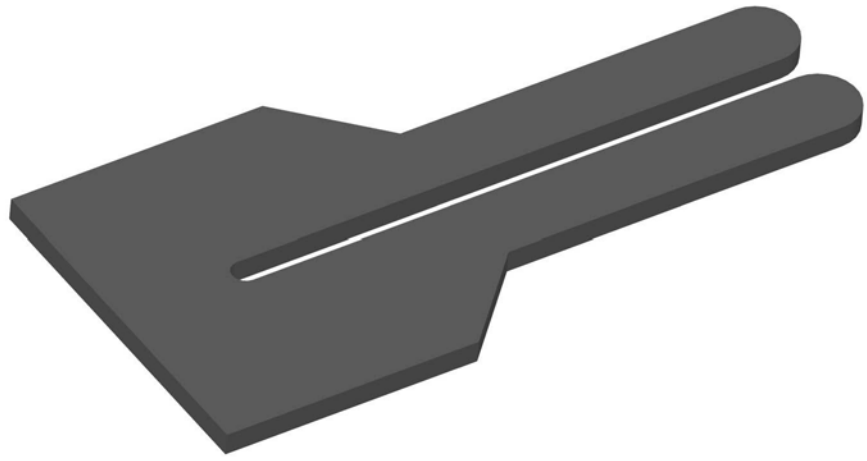




Exercise A5: clip

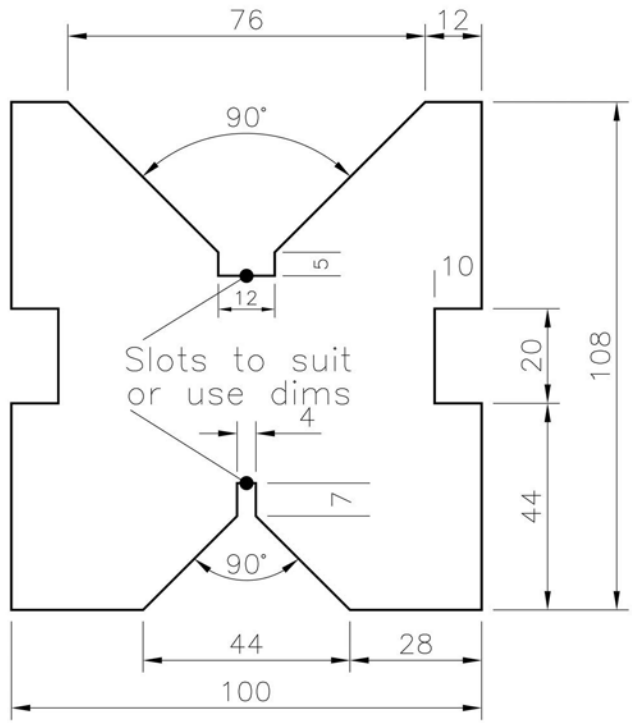
Extrude the sketch for a distance of 5mm.

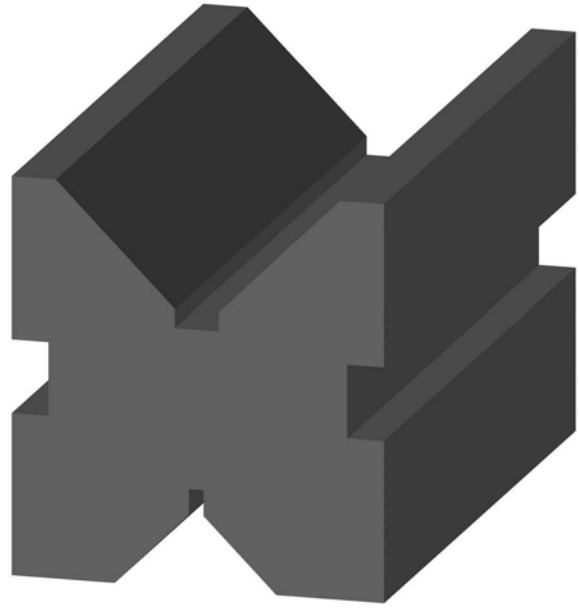




Exercise A6: V block

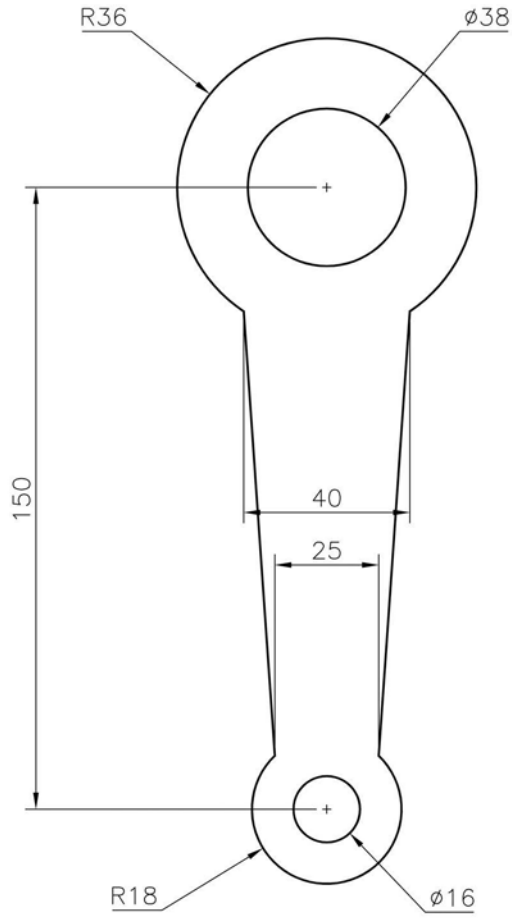
Extrude the sketch for a distance of 180mm.





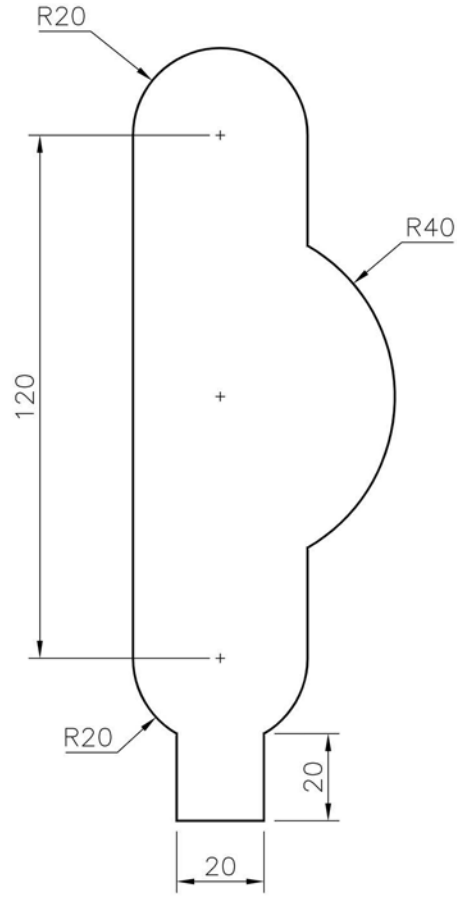
Exercise A7: coupling link

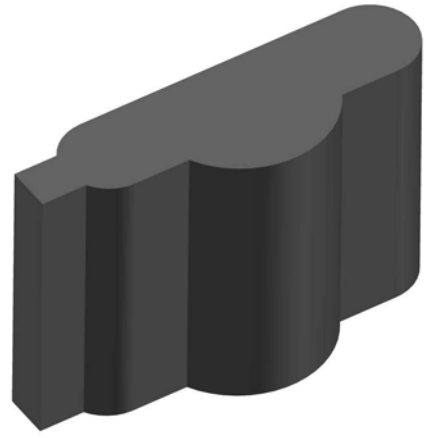
Extrude the sketch for a distance of 11mm.



Exercise A8: container

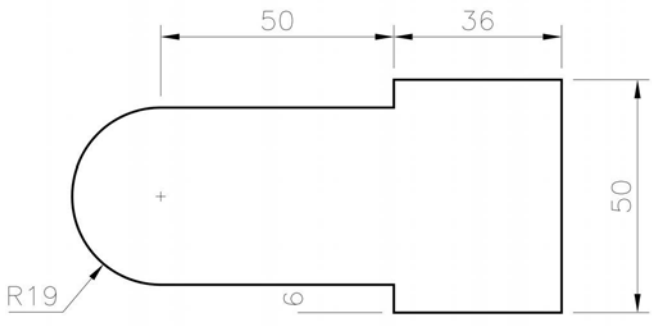
Extrude the sketch for a distance of 120mm.

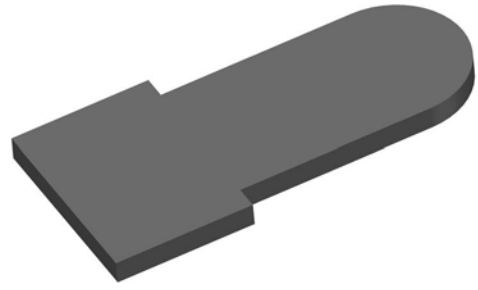




Exercise A9: support

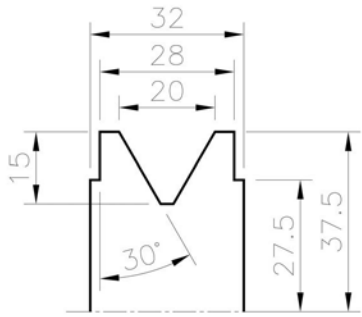
Extrude the sketched feature for a distance of 6mm.





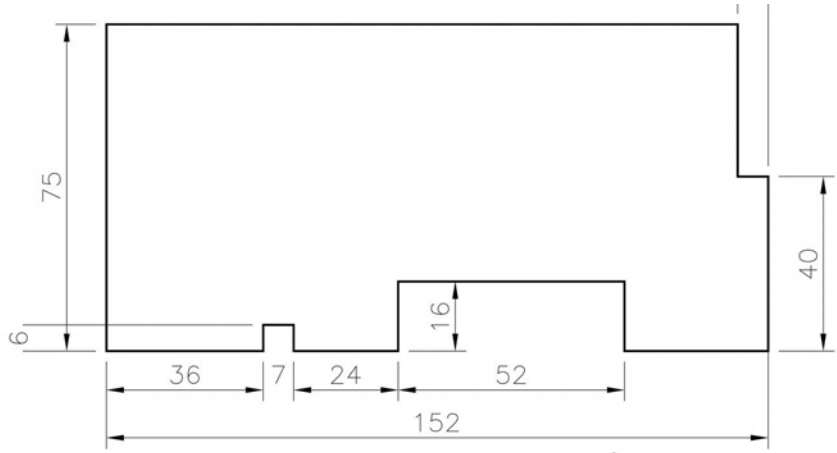
Exercise A10: pulley wheel

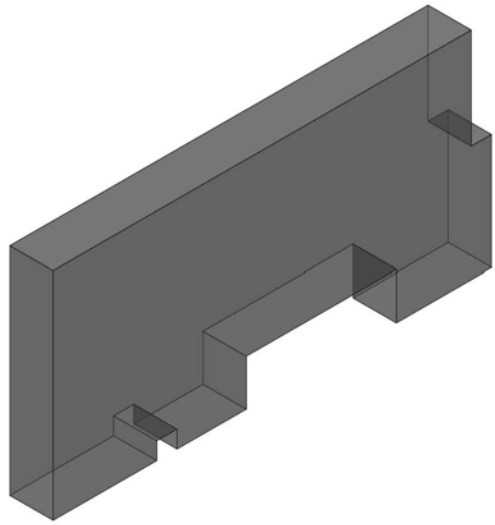
Revolve the sketched feature for 270 degrees then rotate to suit.



Exercise A11: template

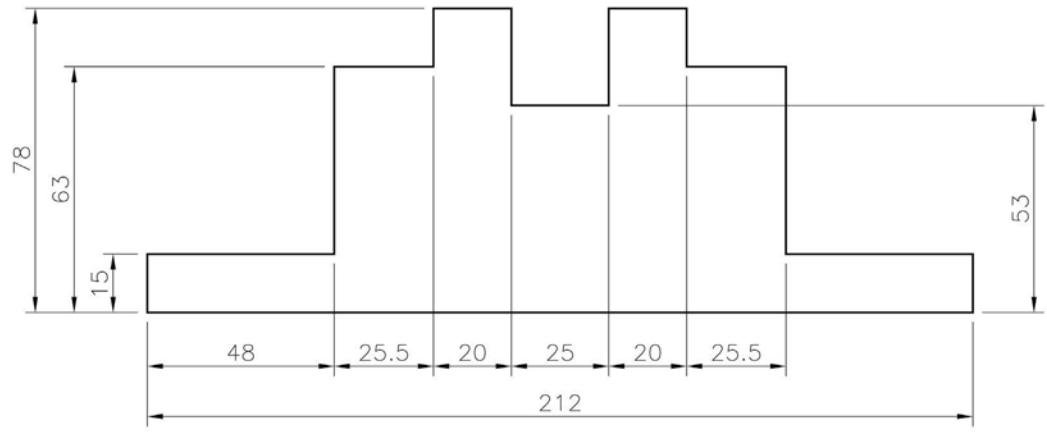
Extrude the sketch 15mm.

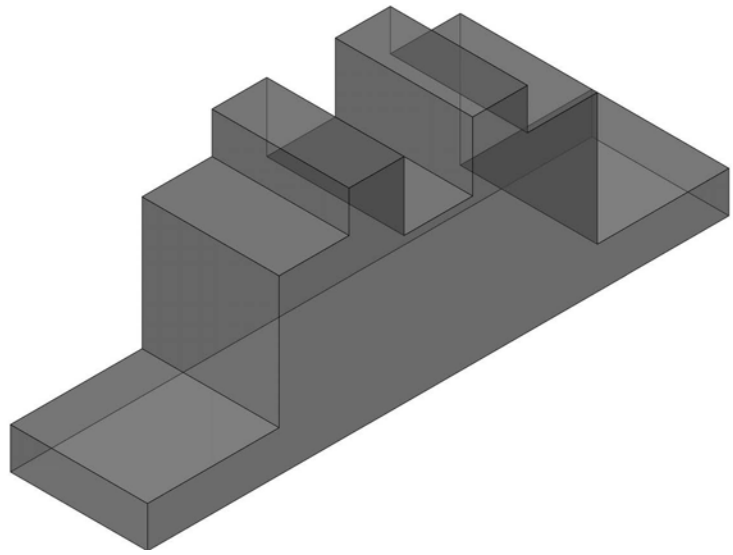




Exercise A12: block

Extrude the block sketch for 50mm.



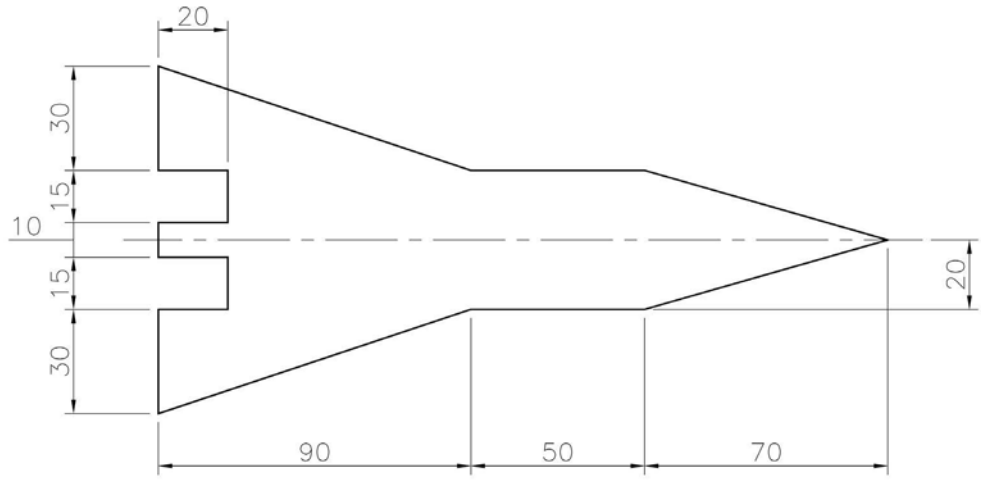


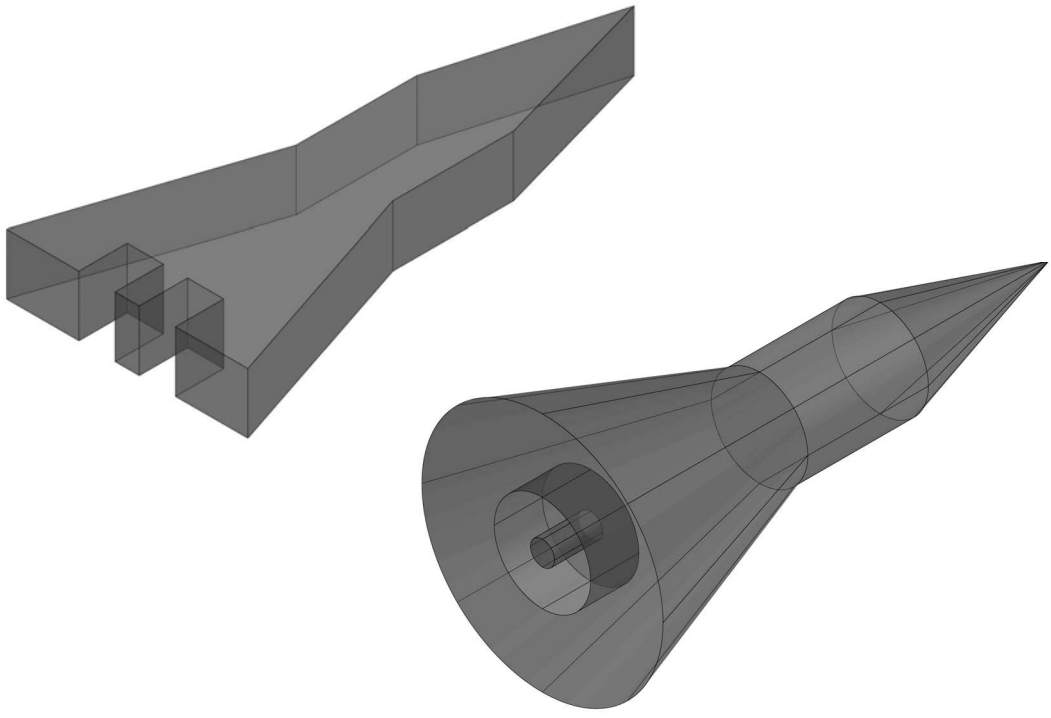
Exercise A13: rocket

Two part models to be created from the one sketch:

1 extrude for 25mm; and

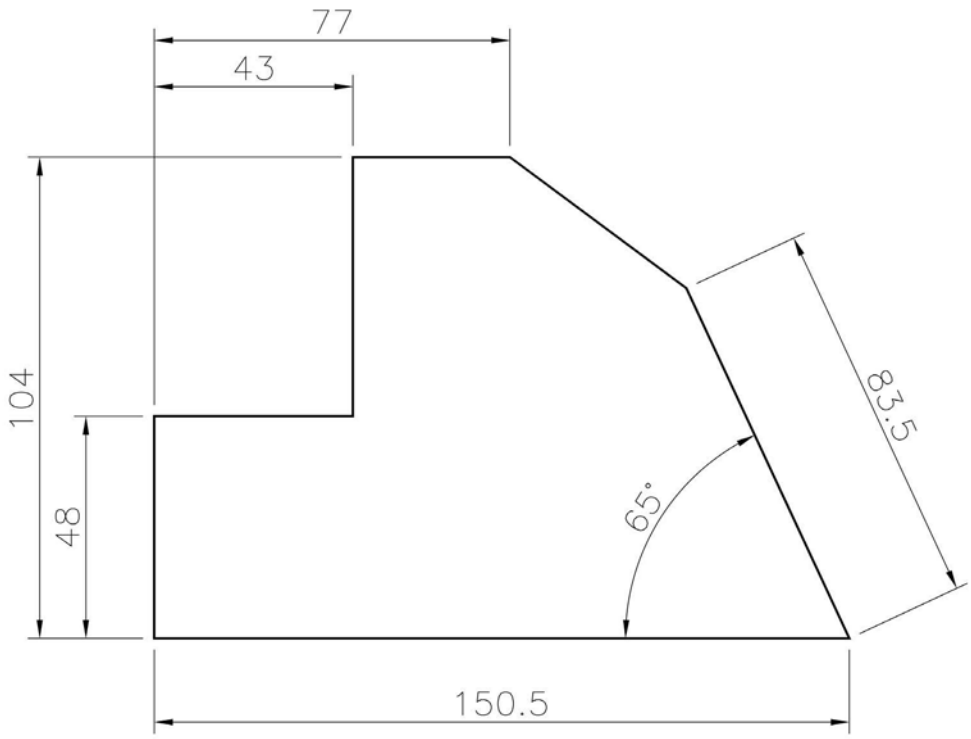
2 revolve for a complete revolution.

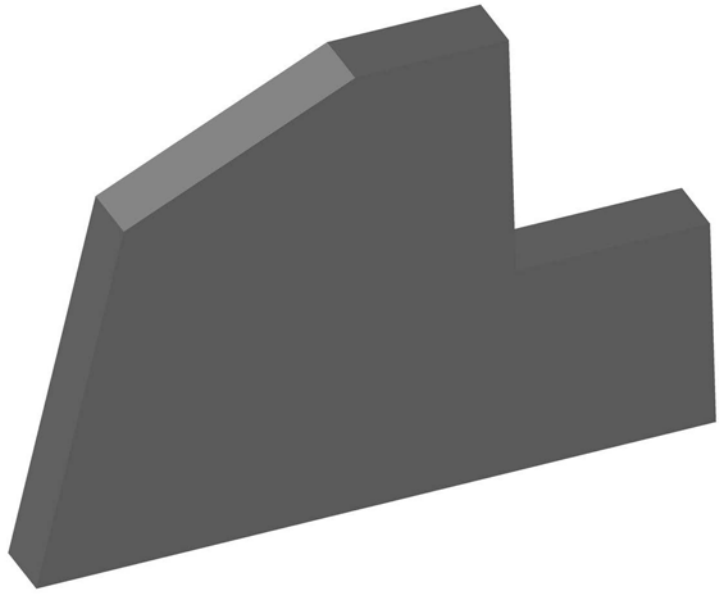




Exercise A14: component

Extrude the dimensioned sketch for 15mm.



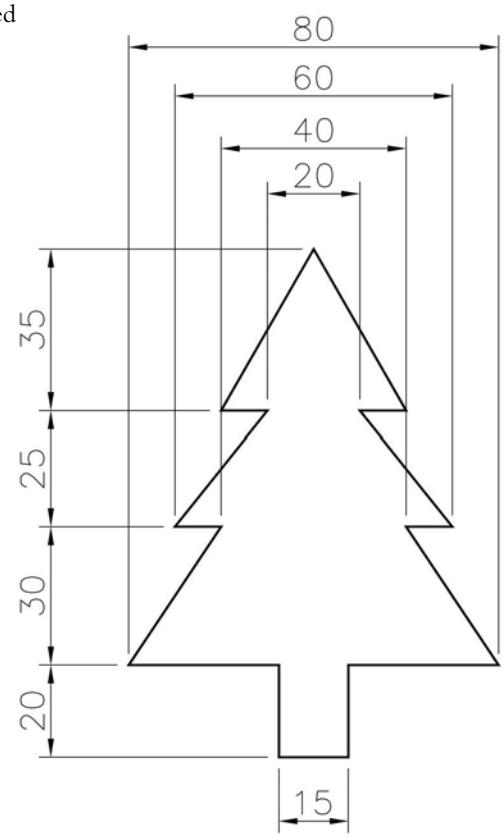


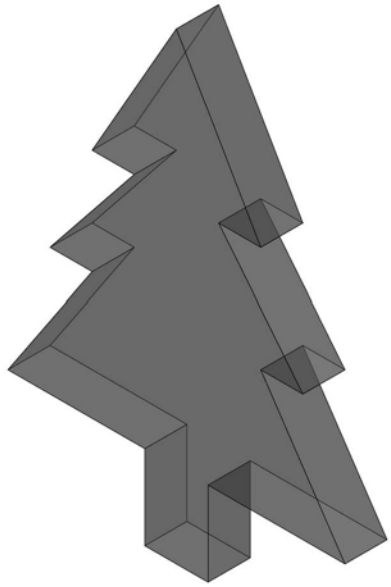
Exercise A15: tree

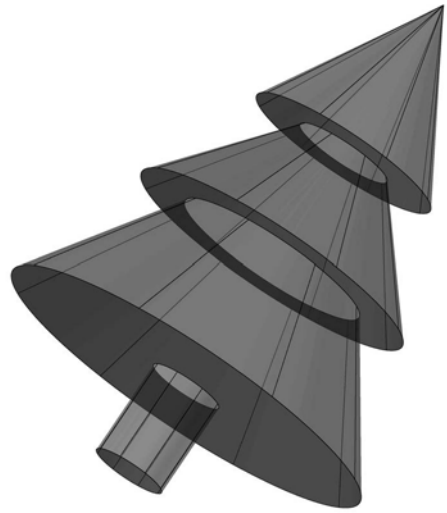
Two part models to be created from one sketch:

1 extrude for 10mm; and

2 revolve for a complete revolution.

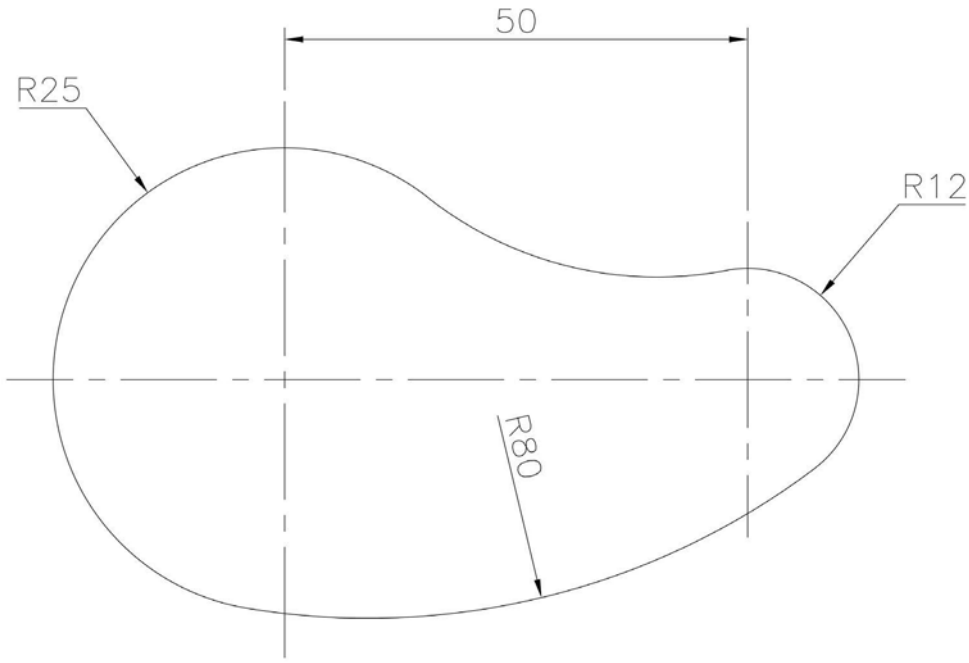


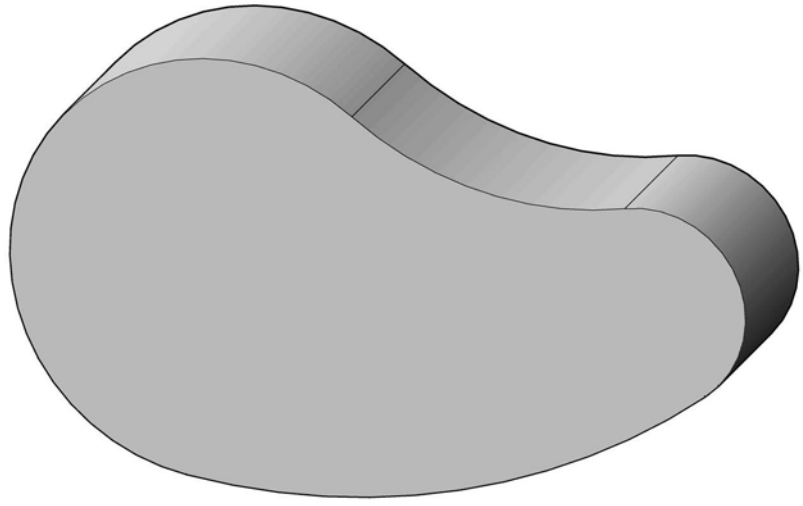




Exercise A16: cam

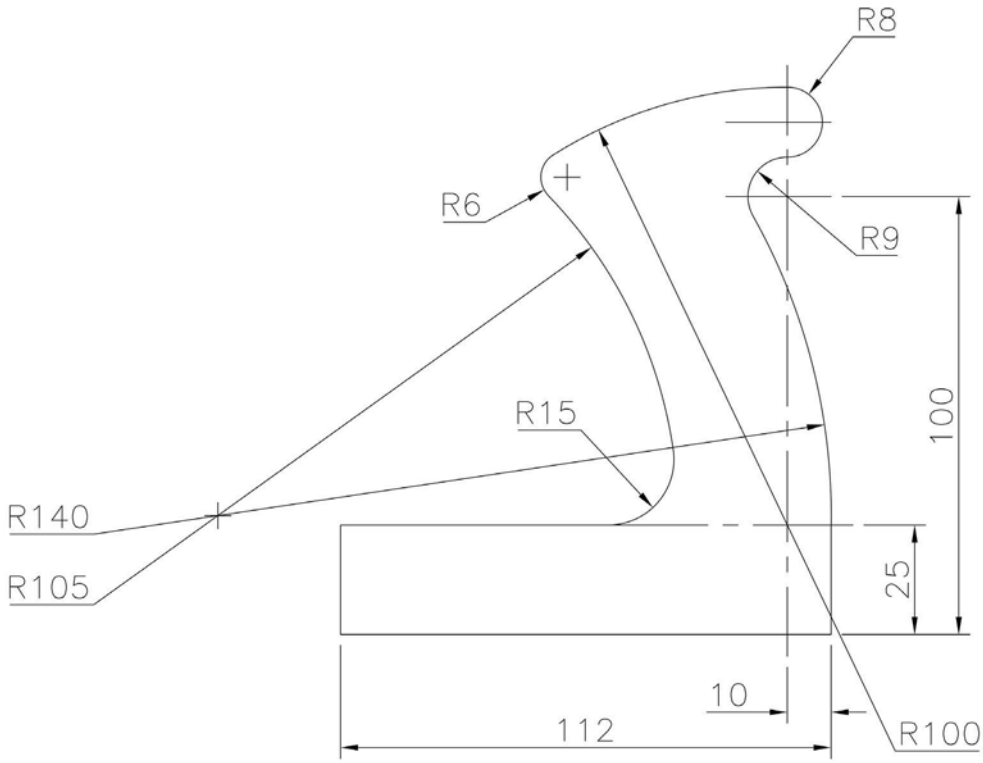
The dimensioned shape is to be extruded for 20mm.

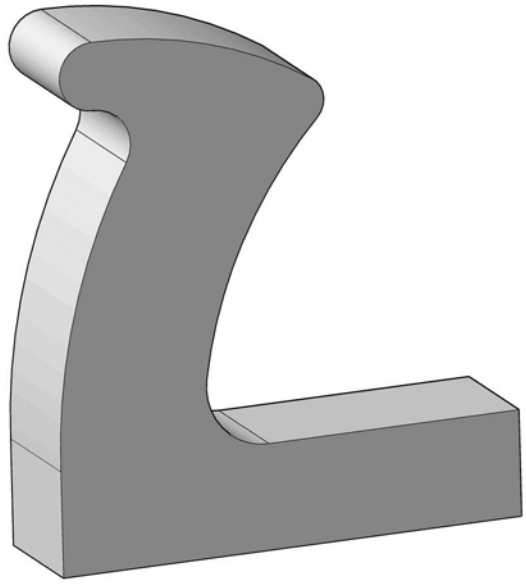




Exercise A17: plane handle

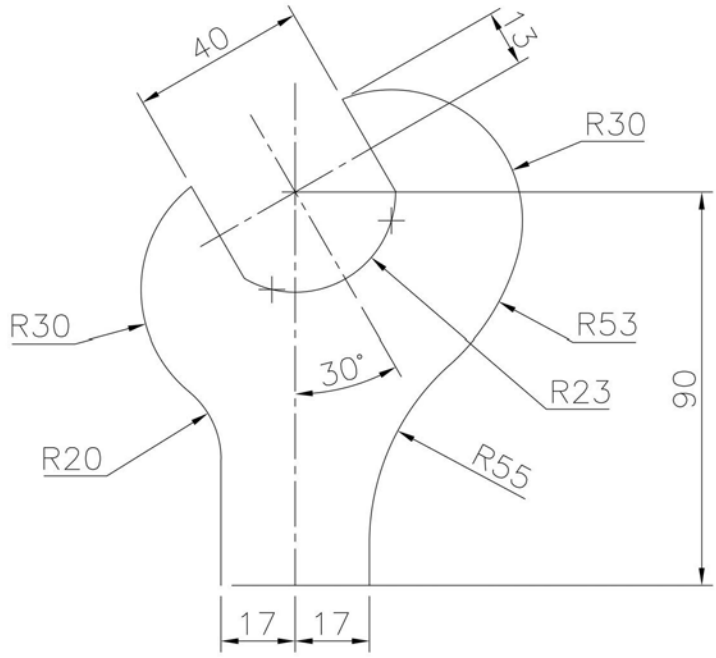
Some interesting work to complete the sketch, which is to be extruded for 30mm.

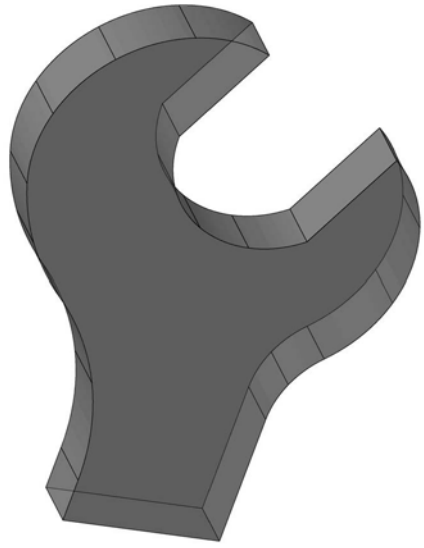




Exercise A18: spanner end

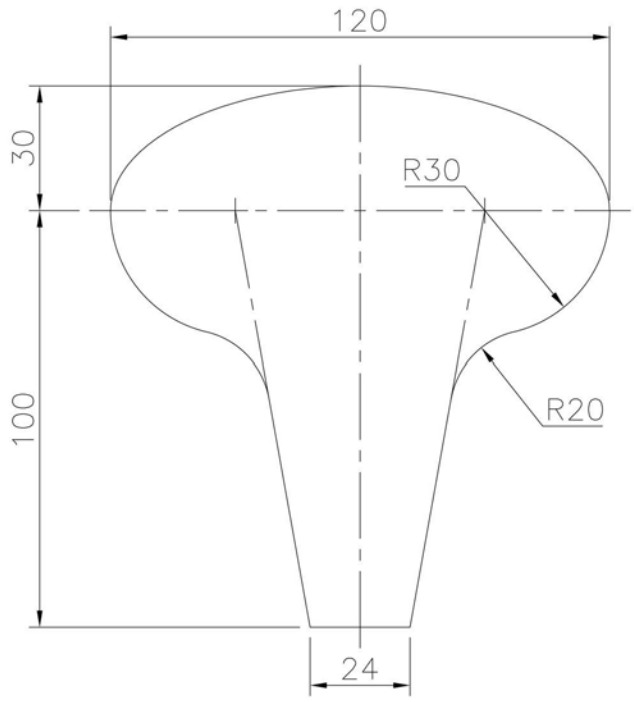
Extrude the sketch for a distance of 12mm.

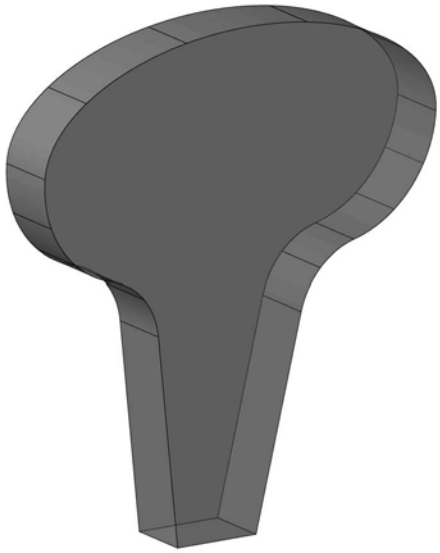


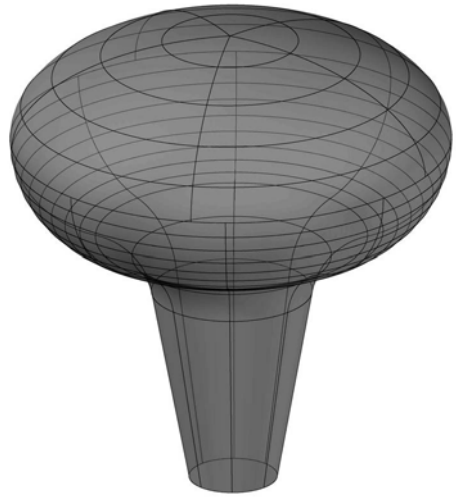


Exercise A19: metalwork dolly

Extrude the sketch (which requires an ellipse) for a distance of 15mm and revolve for a complete revolution.

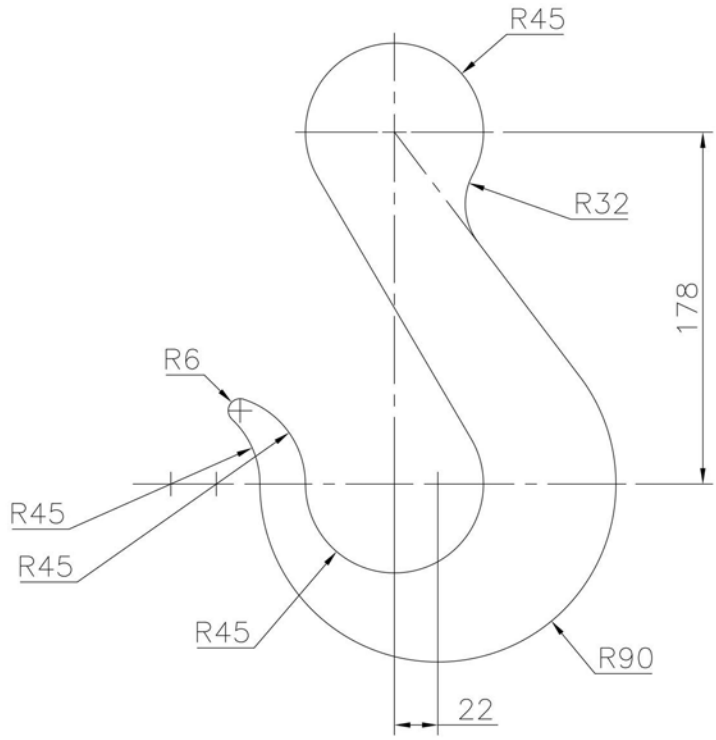


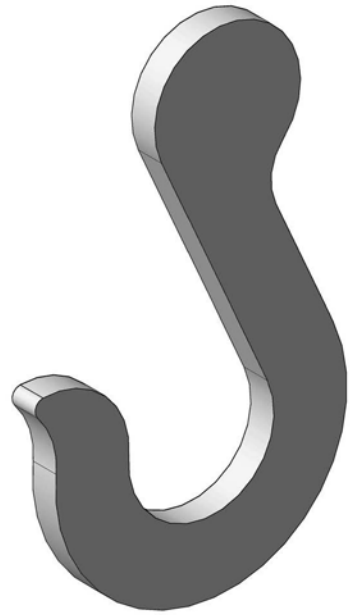




Exercise A20: crane hook

Extrude the dimensioned sketch of the crane hook, selecting a suitable extruded distance (this is really a tangency problem).





J Sheet metal design

1 There are many processes in sheet metal design (fabrication), including:
   (a) stamping;
   (b) drawing;
   (c) punching;
   (d) cutting;
   (e) rolling; and
   (f) other complex operations.

2 Sheet metal work can be considered as:
   (a) a metal blank folded into a finished shape; and
   (b) 'thin plate' work (i.e. less than 1 inch thick).

3 With Inventor®, the general procedure with sheet metal work is to:
   (a) create a sheet metal part using faces;
   (b) create key features of the sheet metal part;
   (c) add/create corner seams;
   (d) cut shapes from the faces;
   (e) add standard features (e.g. chamfers and fillets);
   (f) create a flat pattern model of the part; and
   (g) create a drawing layout of the folded part and flat pattern.

4 The procedure for the sheet metal exercises is:
   (a) Open *a new metric sheet metal (mm) .ipt* file.
   (b) Using Applications, check that the sheet metal environment is active.
   (c) Set Styles (and Save) as required for the Sheet, Bends, Rounds and Corners.
   (d) Use the information given in each exercise to create the part model and/or layout as required.
   (e) Save the completed models/layouts.

Terminology

Thickness

This is the thickness of the flat stock used to create the sheet metal part.

Unfold method

This is the method used to calculate bend allowance – material stretching during bending.

Transition

1 This is the control of the intersection of edges across a bend in the flattened sheet.

2 The transitions available with Inventor include:
   (a) none;
   (b) intersection;
   (c) straight line; and
   (d) arc.

Relief

1 This is a small notch cut next to the end of a bend if the bend does not extend the full width of an edge.

2 The notch can have a straight or round end.

Corner

1 Occurs where three faces meet and corner relief is added to the flat sheet before bending.

2 There are different types of corner relief, including:
   (a) round;
   (b) square;
   (c) tear; and
   (d) trim to bend.

Flange

A sheet metal flange is a simple rectangular face created from an existing face.

Hem

1 Used to eliminate sharp edges or strengthen an open edge of a face.

2 Hem types include:
   (a) single;
   (b) double;
   (c) teardrop; and
   (d) rolled.

Bend

Connects two faces with a straight, angled or curved face.

Fold

Allows flat faces to be folded along sketched lines on a face.

Cut

1 Used to 'extrude' a cut shape usually created on a sketched face.

2 The cut can 'cross' bends and folds.

Punch

Used to create cut-out shapes in flat faces but requires specifically designed iFeatures on sketches.

Exercise J1: inspection cover

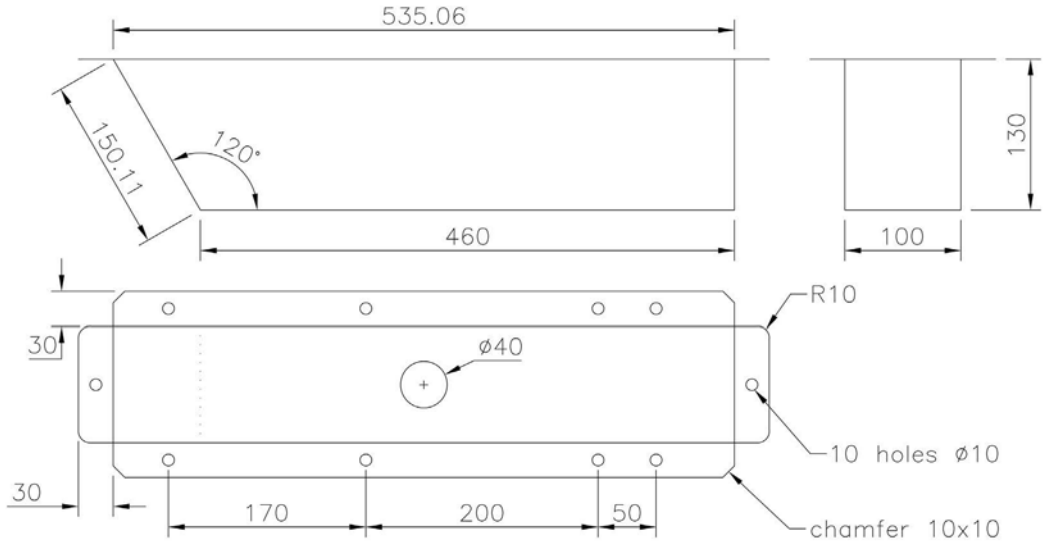
1 With *a new metric sheet metal (mm) .ipt file*, refer to the orthographic layout below and create an open-topped inspection cover (made for 1.6mm aluminium) using the 'overall' dimension given (remember that this diagram does not take any account of bends, tabs, etc.).

2 Each side is to have a 30mm wide mounting flange at 90 degrees to the side.

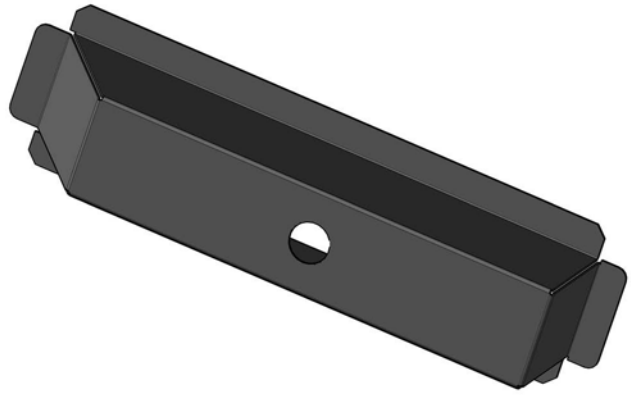
3 The following features are also required:
   (a) 10 ∅10mm holes positioned to your specification on the flanges;
   (b) a single ∅40mm inspection hole in the 'centre' of the top face;
   (c) the long flanges to be chamfered 10×10 at the corners;
   (d) the shorter flanges to have an R10 corner fillet; and
   (e) four corner seams to be added.

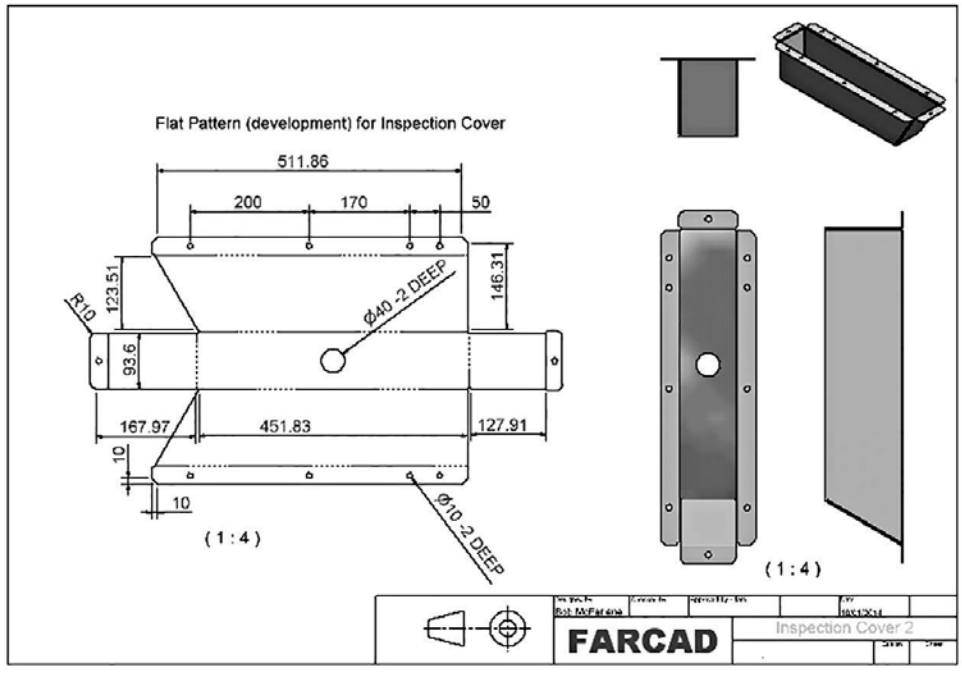
4 Save the model when complete, still with the gaps between sides – we may refer to it in another section.

5 When the model of the inspection cover model is complete:
   (a) Create a flat pattern of the enclosure and save.
   (b) Create a drawing layout (either in First or Third Angle projection) to display:
      - three 'basic' orthographic views;
      - a 3D view of the enclosure; and
      - the flat pattern with dimensions.



Note: Dimensions do not take account of bends/folds, etc.



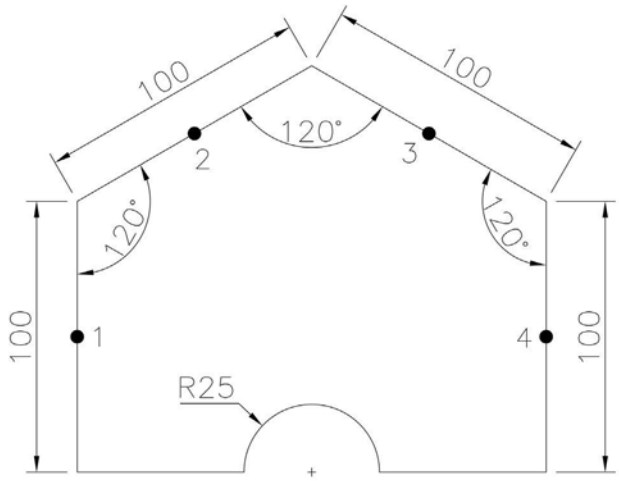

Flat Pattern (development) for Inspection Cover
511.86
200
170
50
123.51
146.31
R10
Ø40 -2 DEEP
93.6
167.97
451.83
127.91
10
10
Ø10 -2 DEEP
(1 : 4)
(1 : 4)
FARCAD
Inspection Cover 2

Exercise J2: sheet metal FOLLY

1 A sheet metal 'FOLLY' model has to be created from 1.6mm brass using the basic drawing information given below.

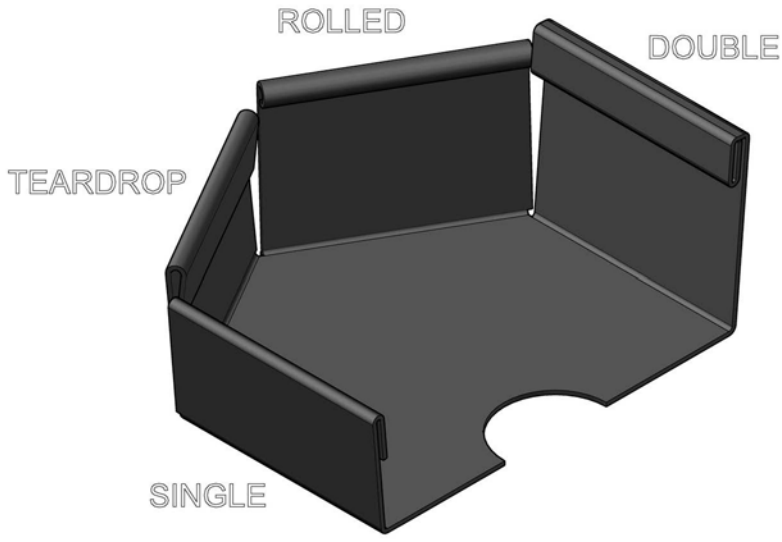
2 The 'free ends' of the four sides of the folly have to be 'hemmed' as follows:
   (a) single, length 20, gap 2;
   (b) teardrop, radius 2, angle 190;
   (c) rolled, radius 3, angle 290; and
   (d) double, length 20, gap 2.

3 Create and save the FOLLY sheet metal part model.



Side 1 Height 50, Single hem
Side 2 Height 60, Teardrop hem
Side 3 Height 70, Rolled hem
Side 4 Height 80, Double hem

All sides are 'inclined' at 95° to the horizontal



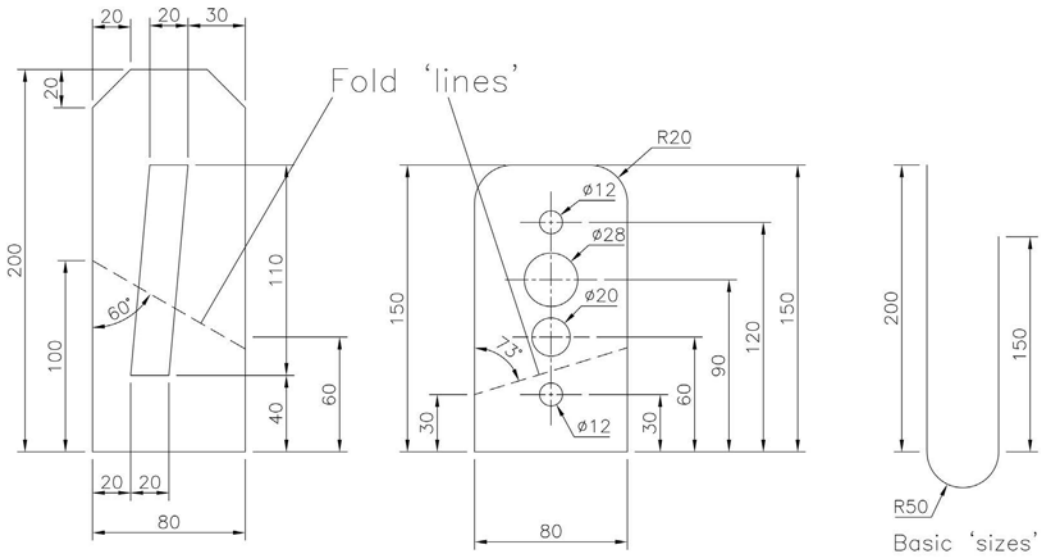
Exercise J3: cable grip

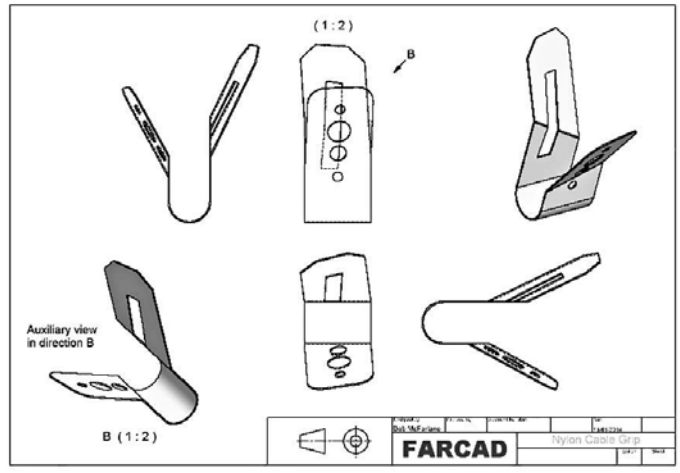
1 A nylon cable grip has to be created from the following information:
   (a) Two vertical 'plates' with sizes $200 \times 80 \times 1.6$ and $150 \times 80 \times 1.6$.
   (b) The plates are 50mm apart at the 80mm edge.
   (c) The longer plate has a 110×20 cut slot.
   (d) The shorter plate has four holes cut in it.
   (e) Each plate has a fold along a given line for 30 degrees.

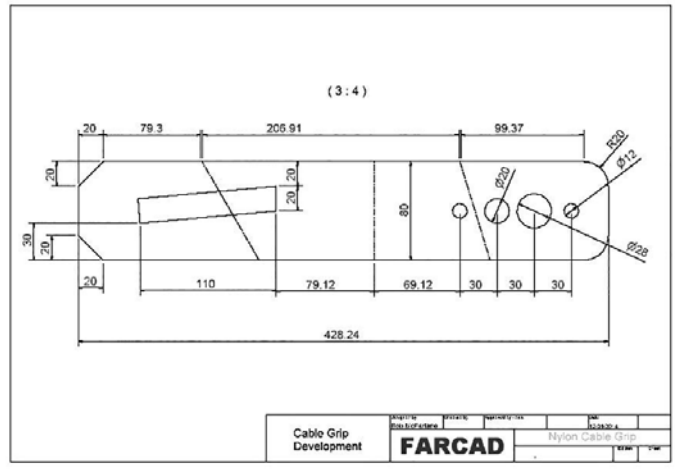
2 Using the information given above and the drawing dimensions below:
   (a) Create a sheet metal model of the cable grip.
   (b) Create a drawing layout to include:
      - the three traditional orthographic views at either First or Third Angle;
      - a 3D view of the model; and
      - any auxiliary view.

3 Using a new drawing layout, produce the flat pattern development with dimensions that will assist manufacture.

4 Remember to save all work.




(1:2)
B
Auxiliary view
in direction B
B (1:2)
FARCAD
Nylon Cable Grip

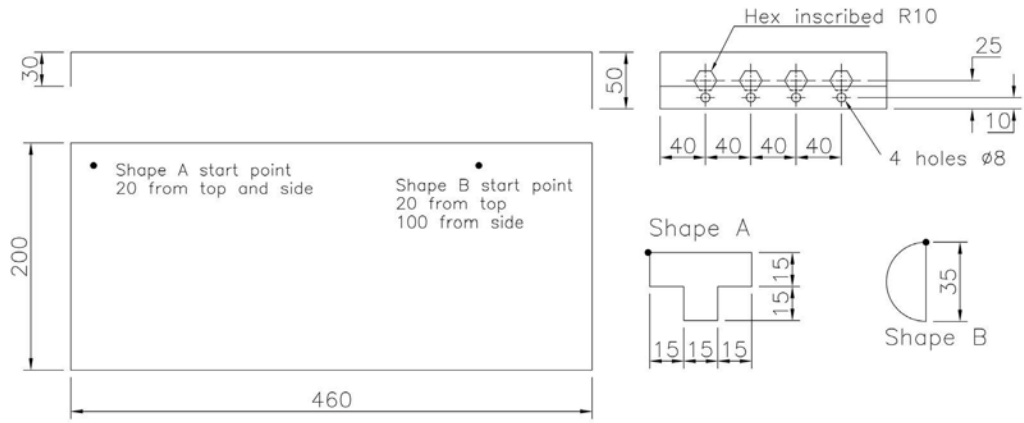

(3:4)
20
79.3
206.91
99.37
R20
Ø12
Ø20
Ø28
80
20
20
30
20
110
79.12
69.12
30
30
30
428.24
Cable Grip
Development
FARCAD
Nylon Cable Grip

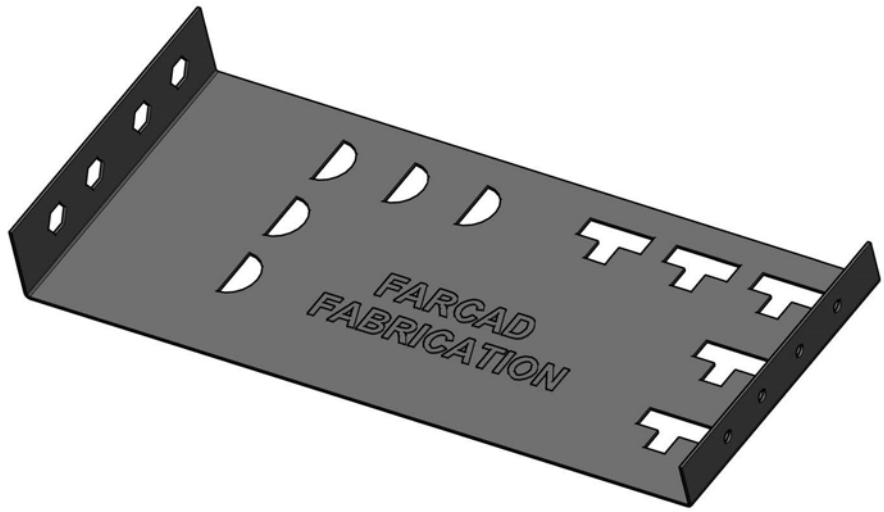
Exercise J4: patterned panel

1 Information:
   (a) Aluminium plate 540 × 200 and 1.6mm thick.
   (b) 30mm and 50mm folds at the ends.
   (c) Various shapes cut out of the flat and ends.

2 Using the information given above and the dimensions in the sketch below, create a sheet metal model of the panel, and:
   (a) shape A pattern distances are 70 horizontal and 60 vertical; and
   (b) shape B pattern distances are 50, both horizontal and vertical.

3 Save your model when complete.



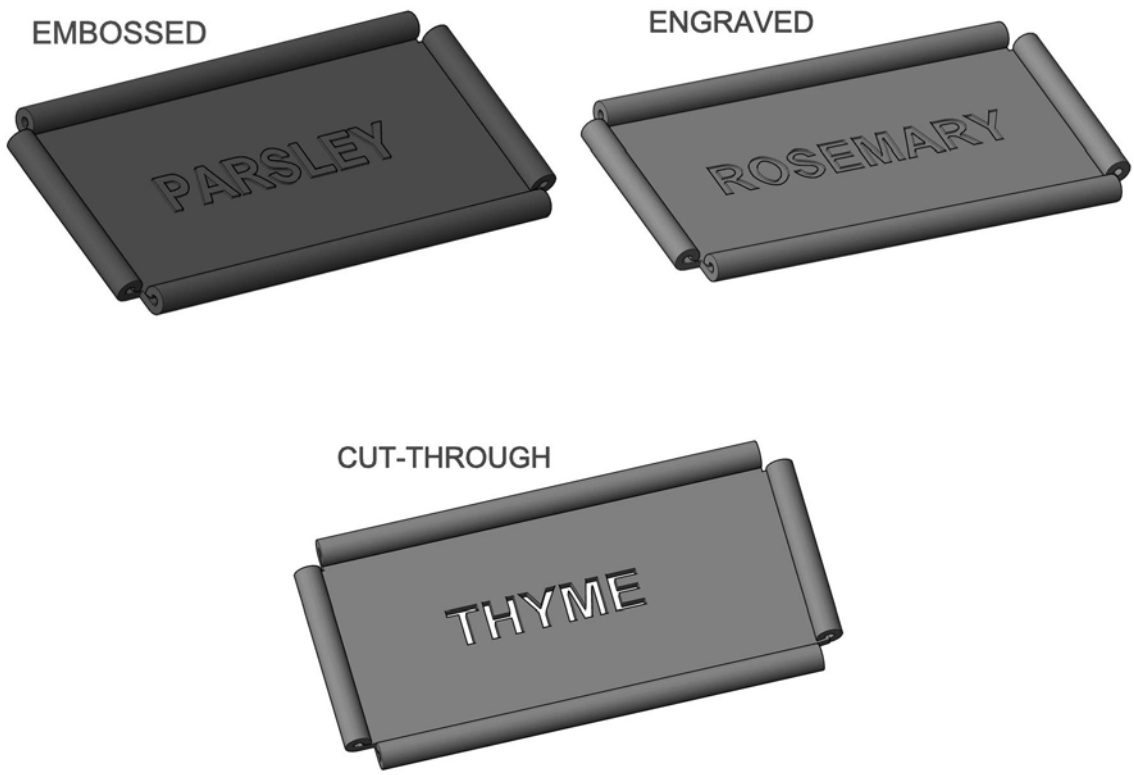


Exercise J5: herb labels

1 A company produces kitchen utensils and one of its products is a label with the name of a herb, made from different materials.

2 Information:
   (a) The plate is $100 \times 50 \times 1.6$.
   (b) The plate has:
      - a 6×6 square cut-out at each corner; and
      - a rolled hem (R1mm and angle 290 degrees on the four sides).
   (c) The herb name can be embossed, engraved or 'cut through'.

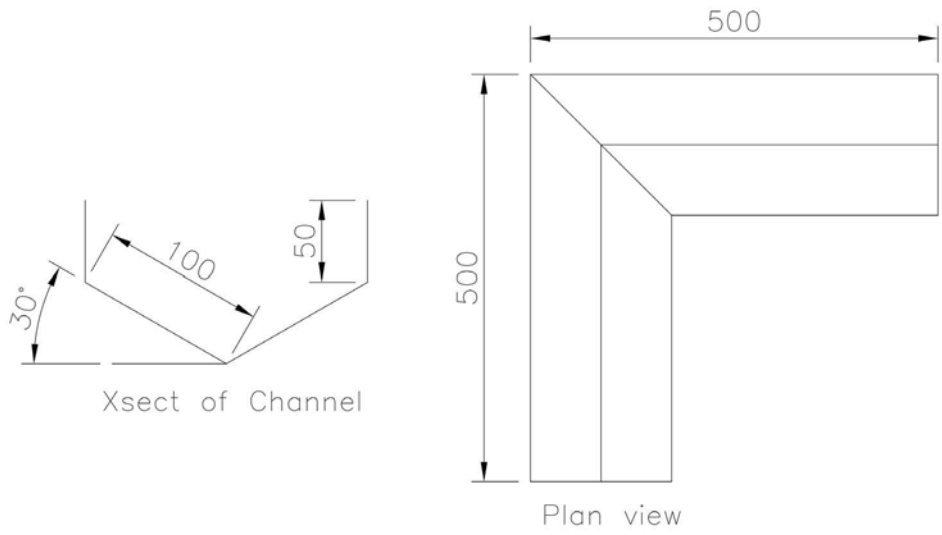
3 Design a herb label for the three (c) options.

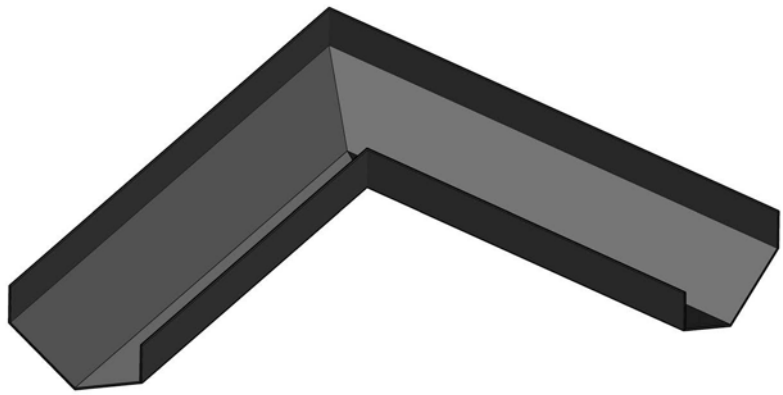


Exercise J6: 90-degree corner V channel

1 A channel system is made from 1.6mm thick aluminium, and the X-section and plan view of part of the channel are displayed below.

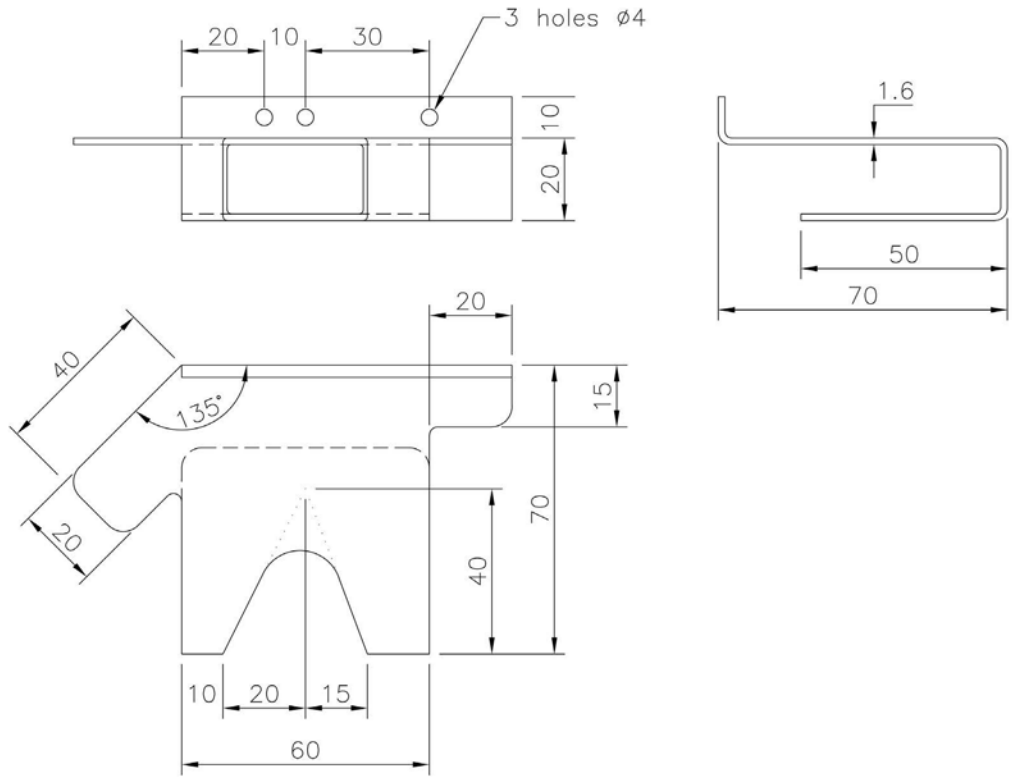
2 Create a sheet metal model for this component, which is open at both ends.





Exercise J7: bracket

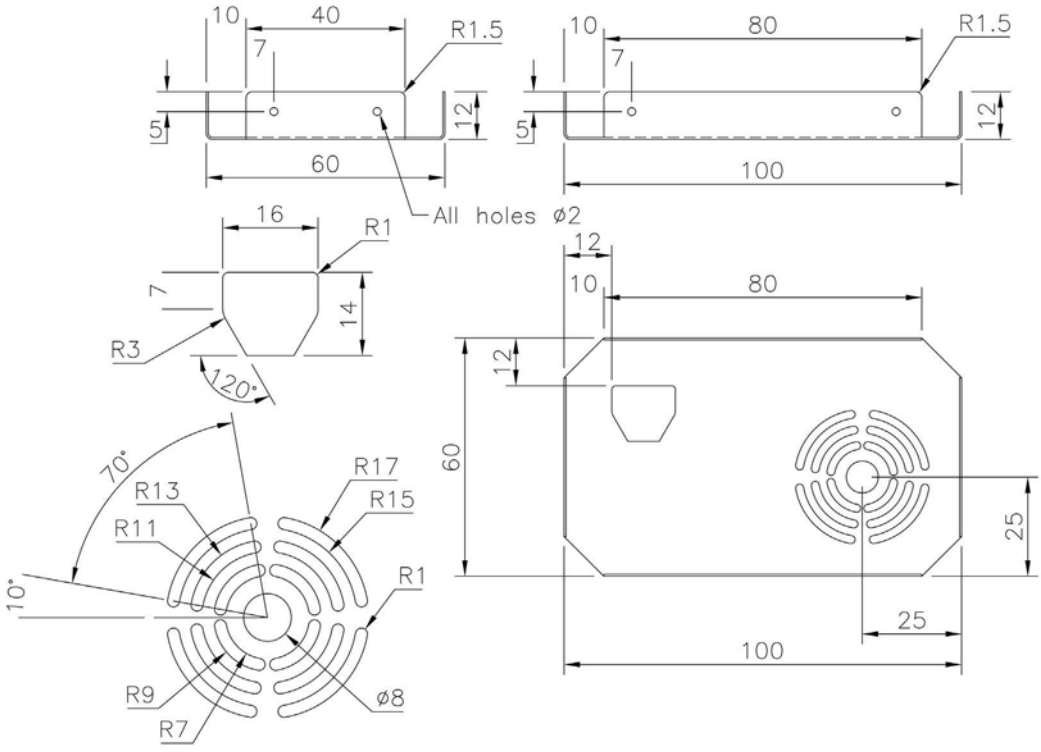
Create a sheet metal bracket from 1.6mm material using the drawing information given in the First Angle orthographic views below.

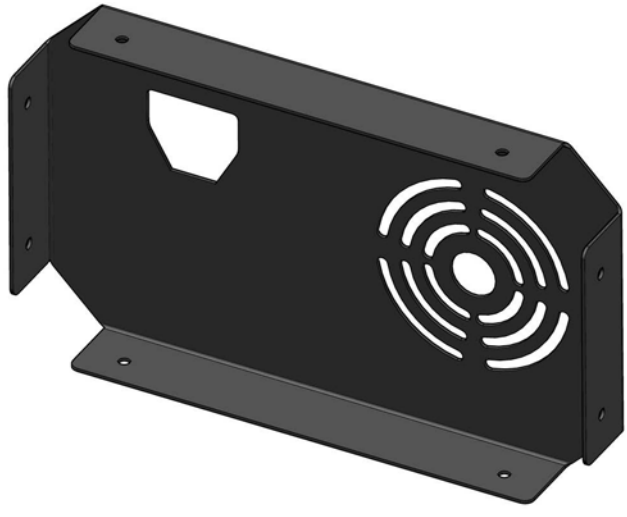


Exercise J8: vent panel

1 A computer vent panel is made from 0.5mm mild steel plate.

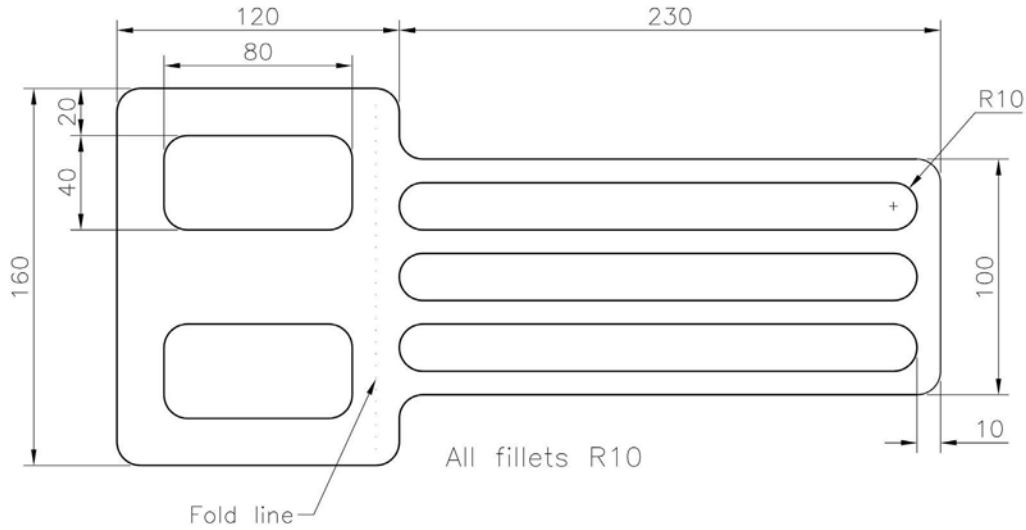
2 Using the orthographic drawings below, create the vent panel.

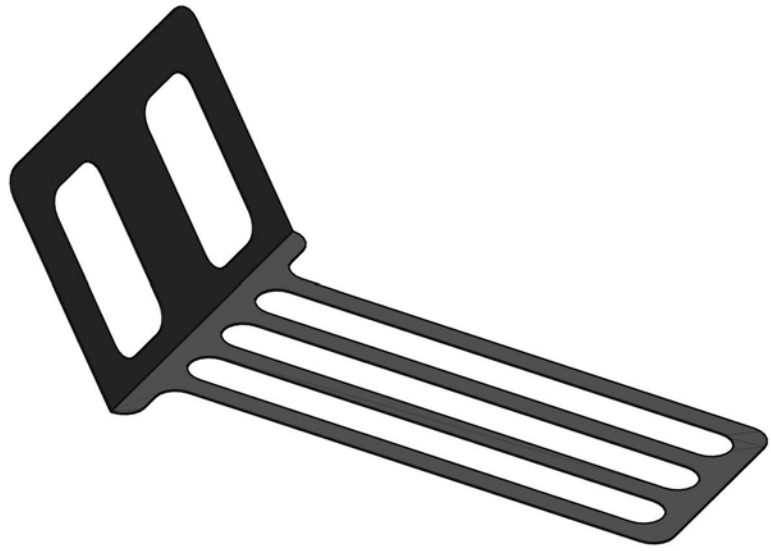




Exercise J9: brass gasket

Create a sheet metal model of the unusual shaped 0.5mm brass gasket using the dimensioned drawing below. The gasket has a 60-degree fold, as indicated.

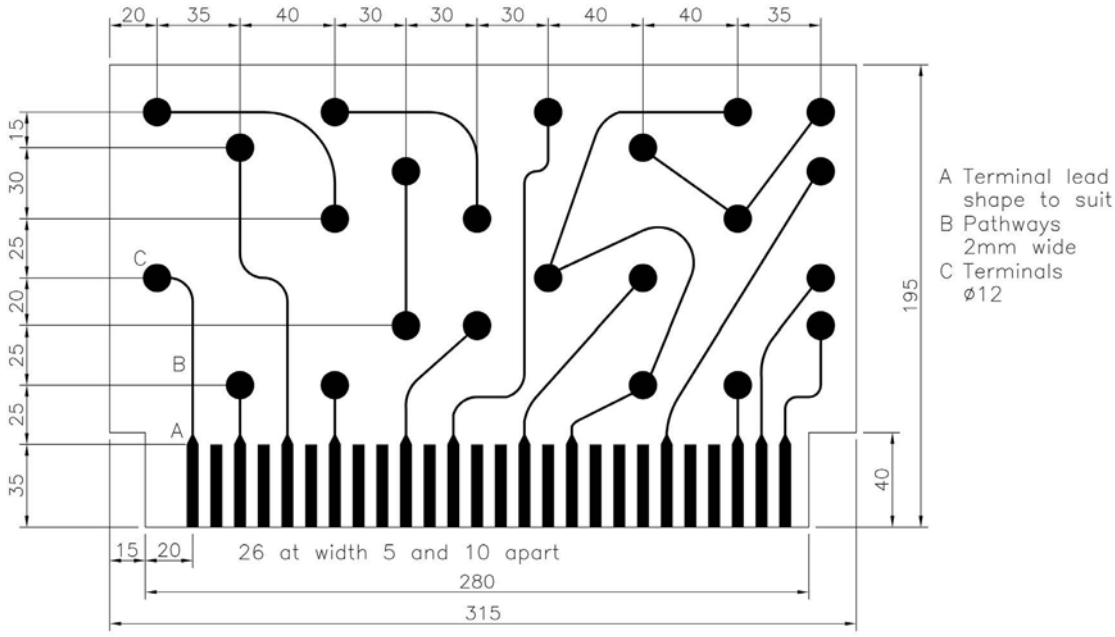


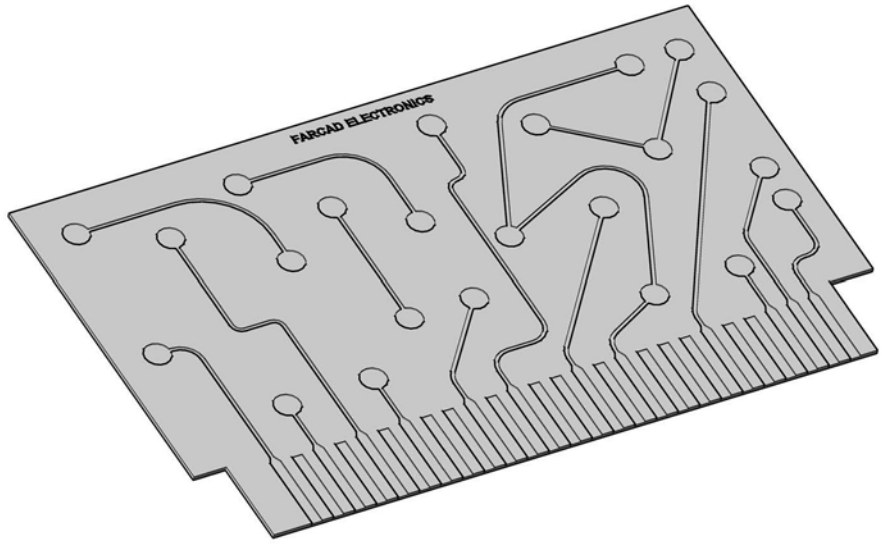


Exercise J10: printed circuit board

1 A printed circuit board has to be created from 1.6mm nylon using the drawing below for the various pathways.

2 Using the information from the drawing (and your discretion for radii, etc.), create the component, noting that all pathways, terminals, etc. are etched 0.5mm into the board.



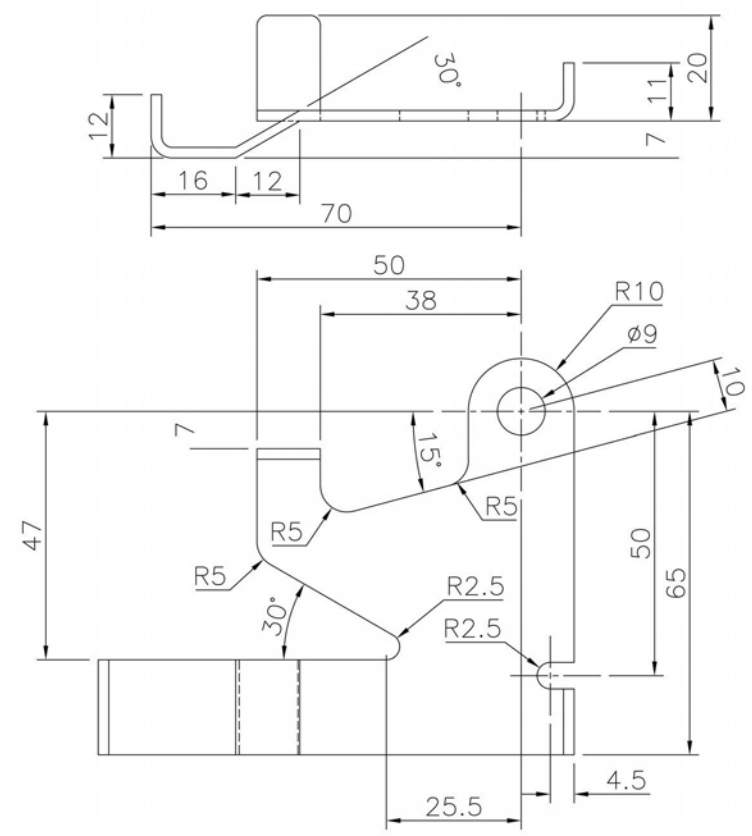


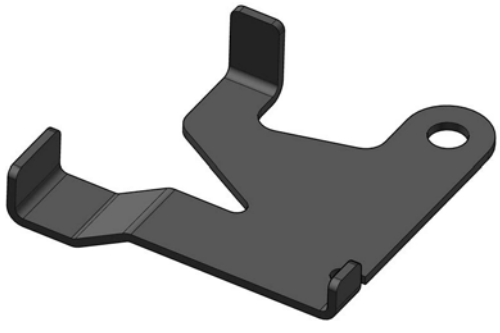
Exercise J11: connecting bracket

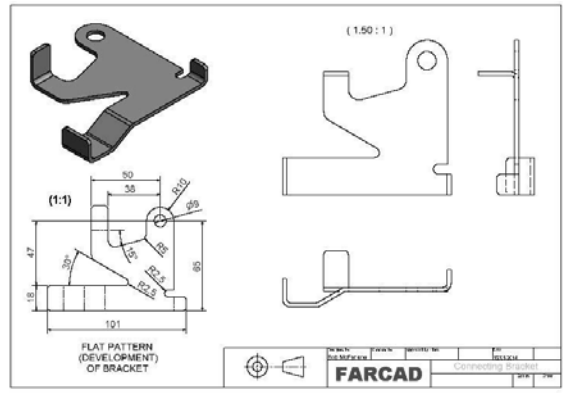
1 Use the two orthographic drawings to create a sheet metal model of the connecting bracket.

2 When the model has been completed, create a drawing layout in Third Angle projection with the flat pattern displayed (with dimensions) full size.

3 The material is 2.0mm thick mild steel with 2mm rounds and a 2mm inside bend radius.

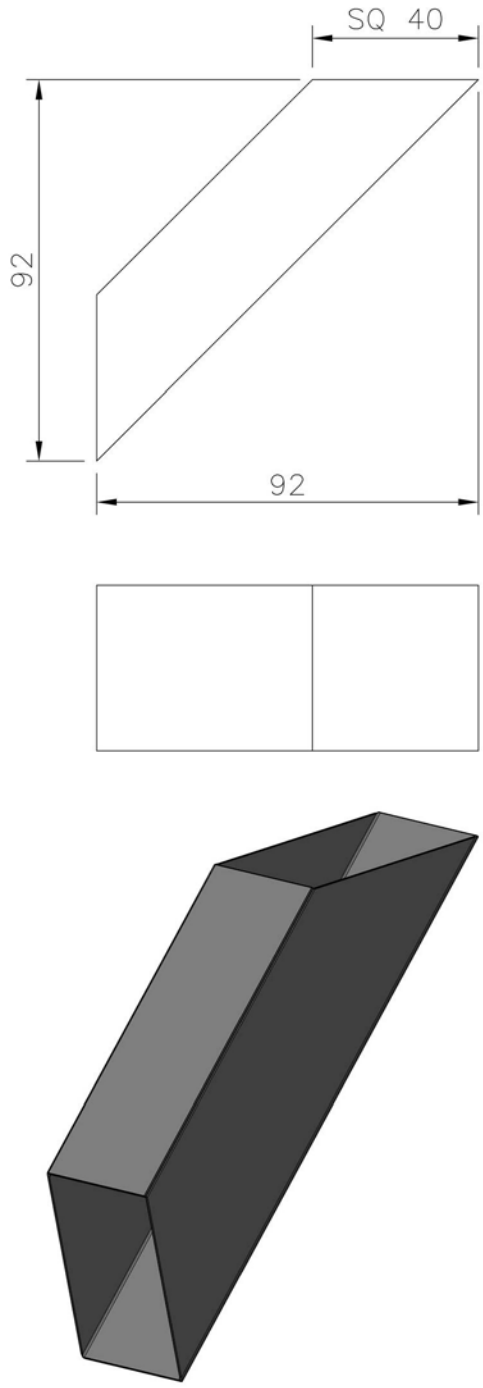




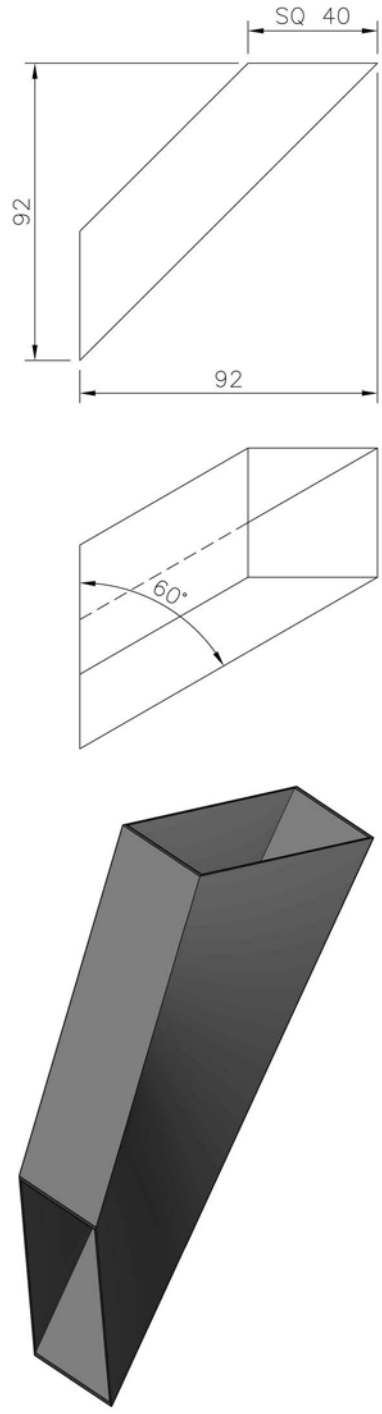


Exercise J12: square chute

1 Create the Mild Steel 0.5mm thick sheet metal chute using the drawing information given below.



2 With a slight variation in the plan view (60-degree angle included as below), create the 'new' model.



K Additionals

1 This chapter is intended to allow the user additional practice with a variety of exercises.

2 The user will be creating additional part models, assemblies and drawing layouts.

3 There is no order to the exercises (i.e. the last is not necessarily harder than the first).

4 The procedure for the exercises should (by now) be obvious to the user, so I have not included any steps – it is now your choice as to how each exercise is completed.