Theoretical and practical content for modelling with inventor and drawing with Autocad

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Theoretical and practical content for modelling with Inventor and drawing with Autocad

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If the reader detects a mistake in the book or wishes to contact the authors, he can send an email to edicion@editorial.upv.es

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1. Introduction

Graphical Engineering is a fundamental discipline that trains engineers in various fields, such as industrial engineering and organisation, chemical engineering, and energy engineering. In this context, spatial vision, and graphical representation software such as Autodesk AutoCAD© and Autodesk Inventor© play a crucial role. This book is designed to provide a deep and practical understanding of these fundamental elements.

Objectives

The main objective of this book is to train future engineers in the knowledge and use of techniques and tools that allow the graphical representation and modelling of elements in the engineering field. This includes studying representation systems, training students to interpret and create engineering drawings, and using the computer to develop industrial objects and parts. This comprehensive approach ensures that students acquire fundamental skills for their professional development.

Contextualisation of the environment

In the modern industrial environment, it is essential to know, understand and manage the language and techniques of graphic representation to solve various problems and perform design activities. Knowledge of graphic resources is necessary to transmit ideas and proposals, thus facilitating technical communication. The widespread implementation of computer-aided design systems in industry underlines the importance of this knowledge. Therefore, this book incorporates the learning of 2D drafting and 3D modelling tools, preparing students to face challenges in the professional field.

The importance of spatial vision

Spatial vision is a cognitive ability that allows us to understand and manipulate objects in three dimensions. This ability is essential as it facilitates the interpretation and creation of three-dimensional planes and models, improving accuracy and efficiency in product design and manufacturing. Spatial vision allows us to visualise complex components and assemblies and is essential for problem-solving and design innovation.

In engineering, spatial vision is developed through constant practice and advanced graphical tools. Modelling and technical drawing exercises help students improve this skill, preparing them for the future. In addition, visualising in 3D is indispensable for understanding how distinct parts of a system interact, which is essential for process engineering and the optimisation of complex systems.

Practice with representation software

Using graphical representation software is an integral part of Engineering Graphics training. These programs allow students to create accurate and detailed models of components and systems, facilitating understanding and analysis of complex designs. Among the most widely used programs in this field are Autodesk AutoCAD© and Autodesk Inventor©, each with specific features and applications that complement student learning.

AutoCAD© is a computer-aided design (CAD) program developed by Autodesk©, renowned for its versatility in creating 2D and 3D technical drawings. Its intuitive interface and editing features make it essential for engineers, allowing them to draw and modify draws and better understand the geometry of objects.

Inventor©, also developed by Autodesk©, focuses on 3D modelling for product design, rendering, and simulation. It is ideal for creating complex three-dimensional models, visualising, and analysing designs, and performing digital prototypes and virtual testing, reducing time and costs in product development.

2. Specific Objectives

This book aims to provide students with a comprehensive, direct learning experience using AutoCAD© and Inventor©. Through a series of exercises and projects, students will develop the skills necessary to tackle design challenges in the professional field.

Specific objectives of the book include:

1. Develop spatial vision: Through 3D part modelling exercises, students will improve their ability to visualise and manipulate objects in three dimensions.
2. Become familiar with Autodesk AutoCAD©: Students will learn to use basic and advanced AutoCAD© features to create and edit technical drawings.
3. Master Autodesk Inventor©: Students will learn to use Inventor© to design complex products and systems through modelling and simulation practices.
4. Integrate graphic knowledge: The exercises are designed to integrate knowledge of geometry, mathematics, and physics with graphic skills, providing a comprehensive understanding of engineering design.
5. Prepare for the professional environment: By doing the exercises, students will be better prepared to use CAD tools in their future careers, applying the knowledge acquired in real projects.

Distribution of lab sessions

3D CAD sessions are performed using Inventor©, and 2D CAD sessions using AutoCAD©. This structure allows students to familiarise themselves with each tool before deep into their advanced capabilities and applying them in integrative projects.

The lab sessions are distributed as follows:

- Lab session 1: 3D CAD - Environment and handling of Inventor©
- Lab session 2: 3D CAD - Constraints
- Lab session 3: 3D CAD - Auxiliary planes and holes
- Lab session 4: 2D CAD - Environment and handling of AutoCAD©
- Lab session 5: 2D CAD - Tangencies, layers, and PDF printing
- Lab session 6: 2D CAD - Matrices, splines, and symmetry
- Lab session 7: 3D CAD - Review exercises
- Lab session 8: 3D CAD - Revolutions and advanced dimensioning of sketches

- Lab session 9: 2D CAD - Patterns
- Lab session 10: 2D CAD - Standardized representation of engineering parts
- Lab session 11: 2D CAD - Standardized representation of engineering parts II
- Lab session 12: 2D CAD - Standardized representation of engineering parts III

Structure of the book

The book is divided into sections focusing on graphic design and CAD software use. Below is an overview of the sections:

1. **Autodesk Inventor© Practices**: This section focuses on parametric modelling. Students will learn to create 3-dimensional parts using sketches, constraints, dimensioning, extrusions, holes, symmetry, patterns, and revolutions.
2. **Autodesk AutoCAD© Practices**: The exercises in this section are designed to familiarise students with AutoCAD© features. They primarily include 2D drawing tasks, printing plans, and setting up title blocks for printing to PDF.
3. **Self-Assessment and Reflection**: The last section of the book focuses on self-assessment of the exercises completed, which allows students to reflect on the knowledge they have acquired.

Conclusion

Engineering Graphics is a fundamental competency for engineers of all disciplines. This practical book is designed to help students develop skills in spatial vision and use 2D and 3D CAD software, preparing them to face the professional world. Upon completing this course, students will have acquired technical knowledge and will have improved their visual space. They will be able to solve complex problems and work efficiently with advanced computer-aided design tools.

3. SESSION 1: 3D CAD- Environment and handling of Inventor©

In this first lab session of the 3D modelling course, we will focus on the fundamental aspects of working in a three-dimensional environment. Next, we will explore the following topics:

1. **Application environment**
    - **New file (.ipt)**: we will learn how to create a new 3D model file in the correct format.
    - **Ribbons**: we will practise using the ribbon tools, focusing on the 3D Modelling and Sketch sections.
    - **Display Menu**: we will use the display menu to adjust and optimise the display of the model.
2. **Introduction to Basic Modelling I**: we will learn the essential elements to create the first sketches and parts.
    - **Simple Sketches**: we will start with the creation of basic flat sketches that will serve as a basis for our models.
    - **Simple Shapes**: we will develop skills to create and manipulate simple geometric shapes.
    - **Orientation in space**: we will navigate within three-dimensional space.
    - **Basic extrusions**: we will apply extrusion techniques to generate 3D geometry from flat sketches, adding depth and details to our models and applying both positive and negative (or cutting) extrusions.
    - **Multiple extrusions**: we will experiment with multiple extrusions to build more complex shapes.
    - **Redefine the sketch plane**: we will move the parts to other work planes.

This session will provide the foundations for effective and accurate 3D modelling.

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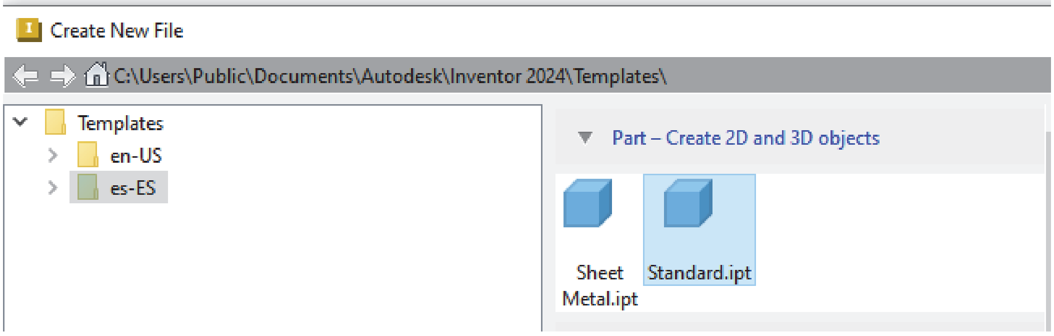
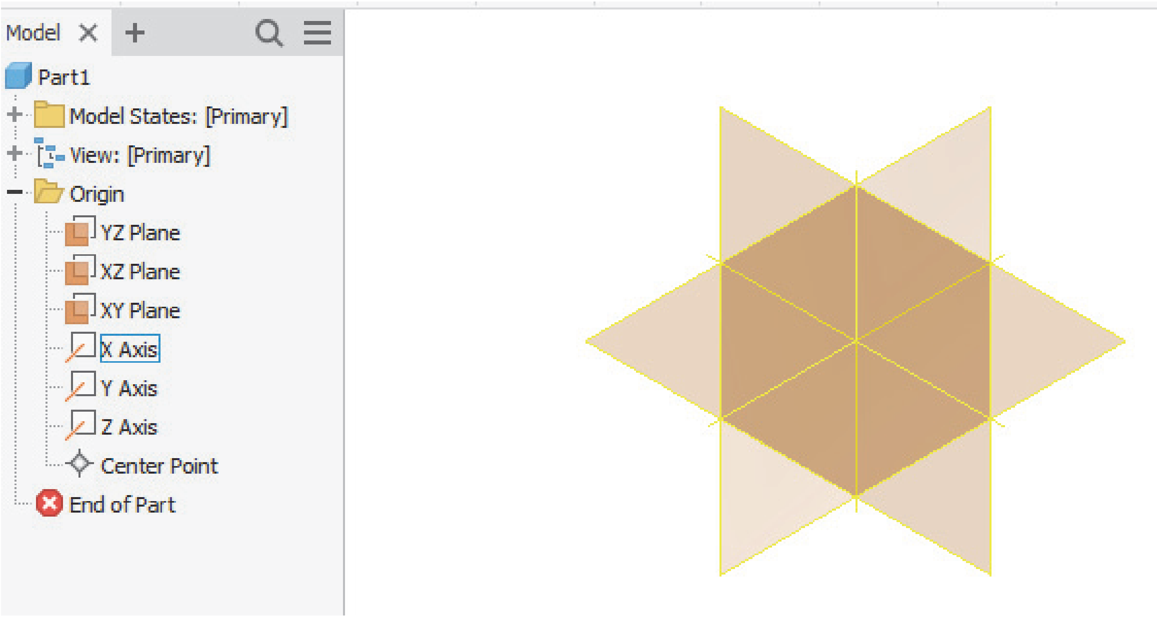
Application environment

EXERCISE 1

Step 1. Select from the NEW FILE menu: template, graphics window, movement tools, model tree (sequence of modelling operations), source elements (planes, axes, and centre).



Step 2. View the different planes.



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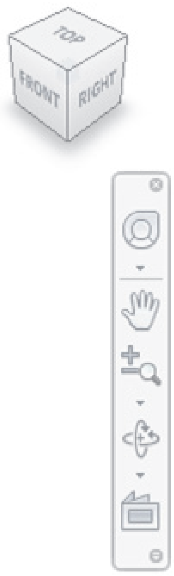
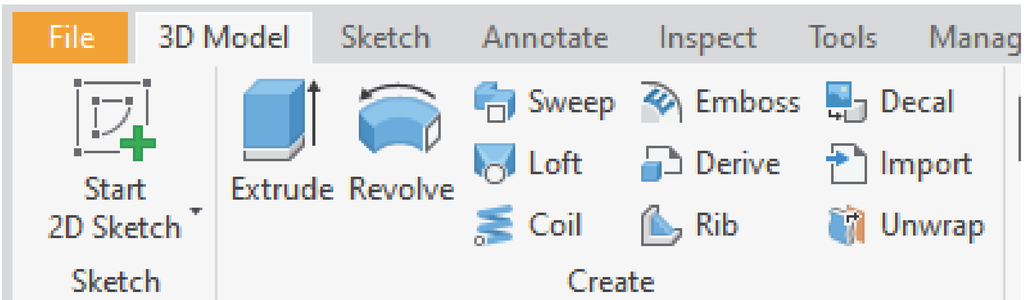
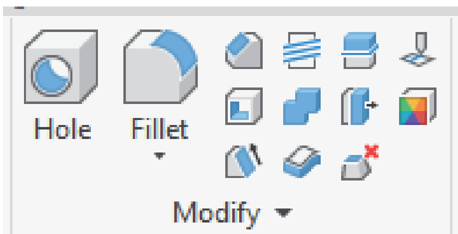
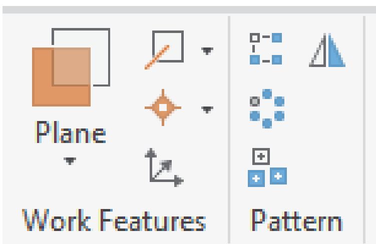


Step 3. View the different MODELLING OPERATIONS: solid modelling creation and modification operations.







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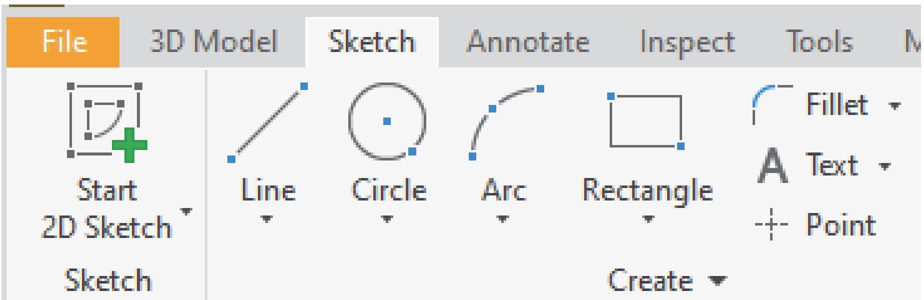
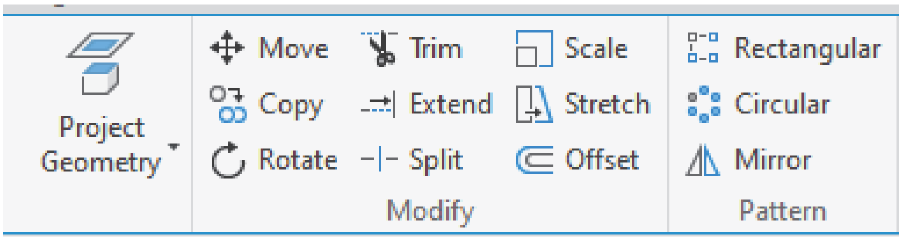
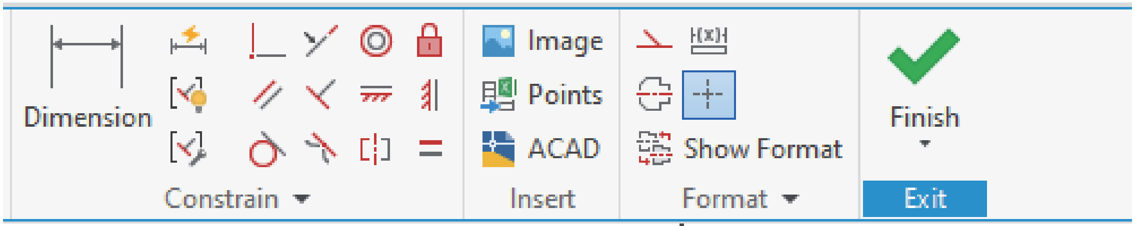
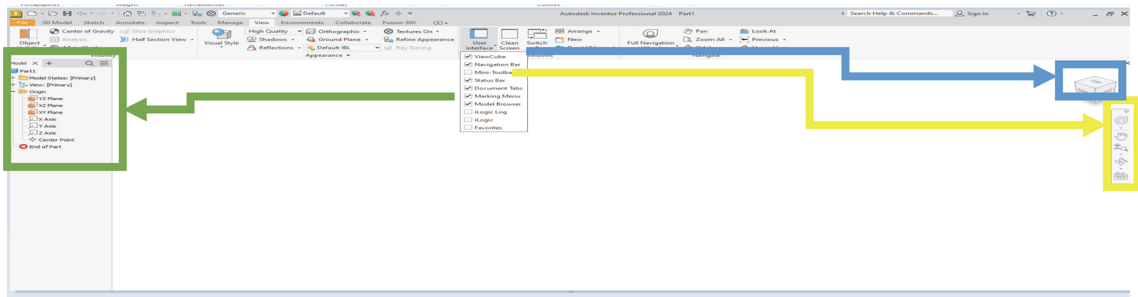
Step 4. Select the SKETCH MODE menu: 2D drawing operations for creating profiles (flat shapes).







Step 5. Select **View Menu.**



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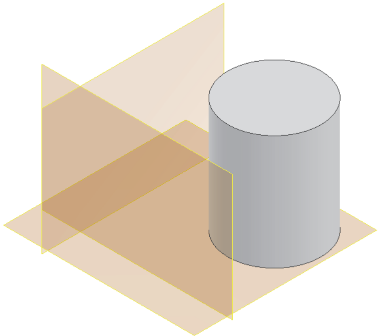
Introduction to Basic Modelling

EXERCISE 2. CYLINDER

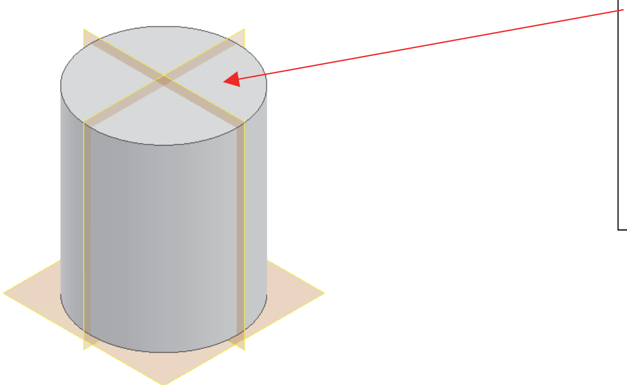
Step 1. Centre at any point.



Learning objectives:

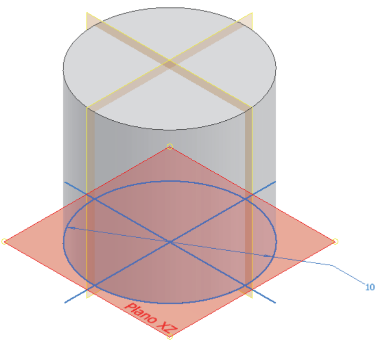
- To navigate in space
- Understand how modelling applications work
- Handling sketches and space management
- Creating simple shapes

Step 2. Centred on source planes.



- Coincidence of the centre of the circle with the centre of the origin:

Step 3. With diameter dimension (add it to the sketch already created).



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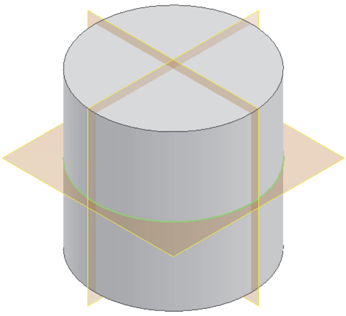
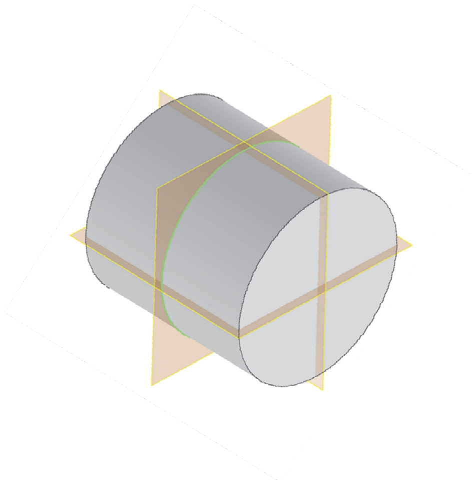
Step 4. Modify the extrusion feature to be symmetrical on both sides.

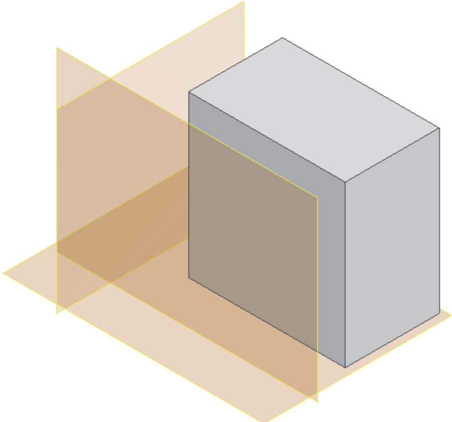


Step 5. Redefine the sketch plane (allows you to choose another sketch plane without having to draw it again).



EXERCISE 3. BOX

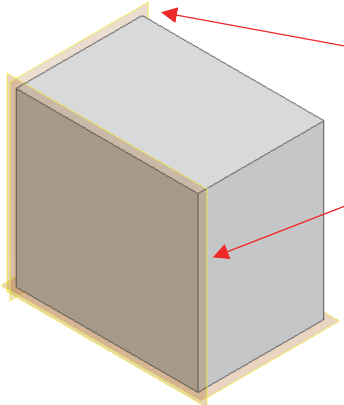
Step 1. Vertex at any point.



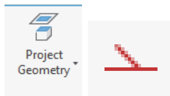
Learning objectives:

- To navigate in space
- Understand how modelling applications work
- Handling sketches and space management
- Creating simple shapes

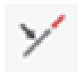
Step 2. Vertex at Origin.



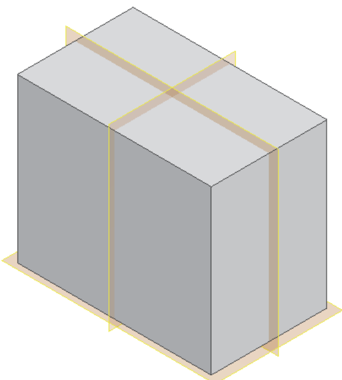
- Projection of planes in construction mode



- Line collinearity constraint



Step 3. Centred on the planes of origin (centre of two points).



- Drawing the rectangle in the sketch with the option:

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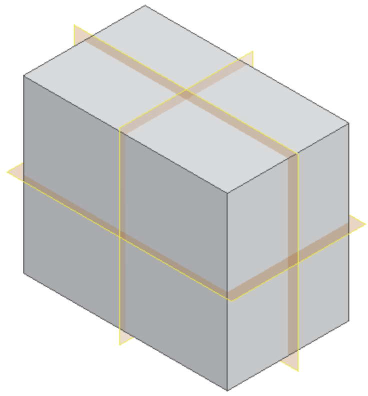
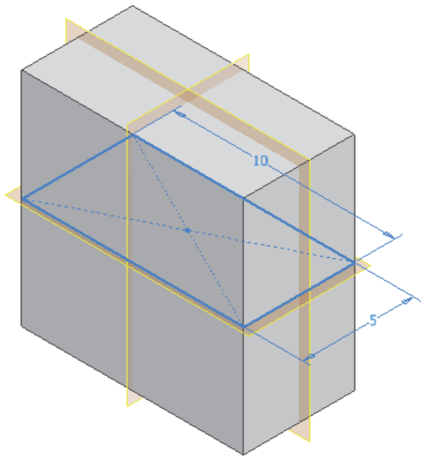
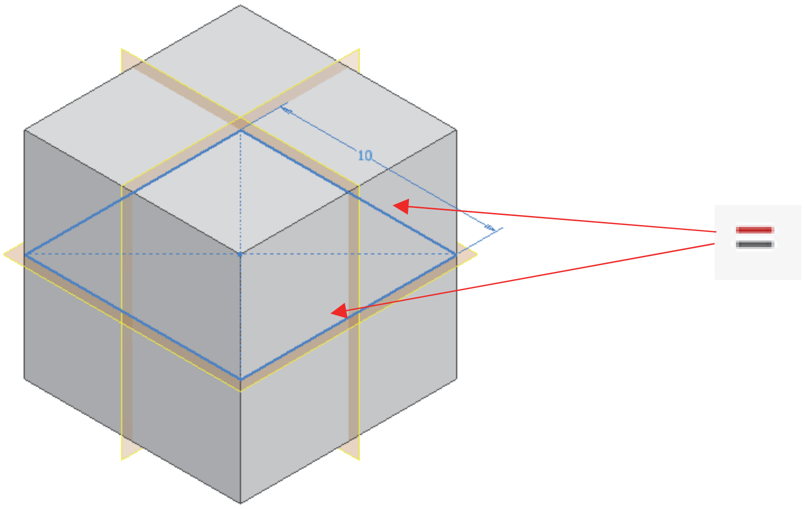
Step 4. Modify the extrusion feature to be symmetrical on both sides.



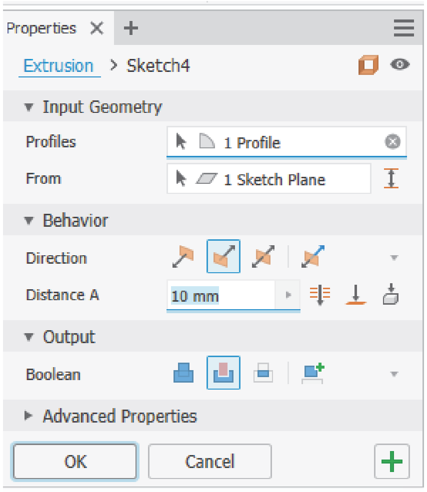
Step 5. A dimension for each side (add to the sketch).

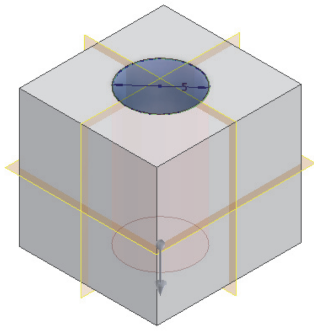


Step 6. A dimension for one side only and equality constraint.



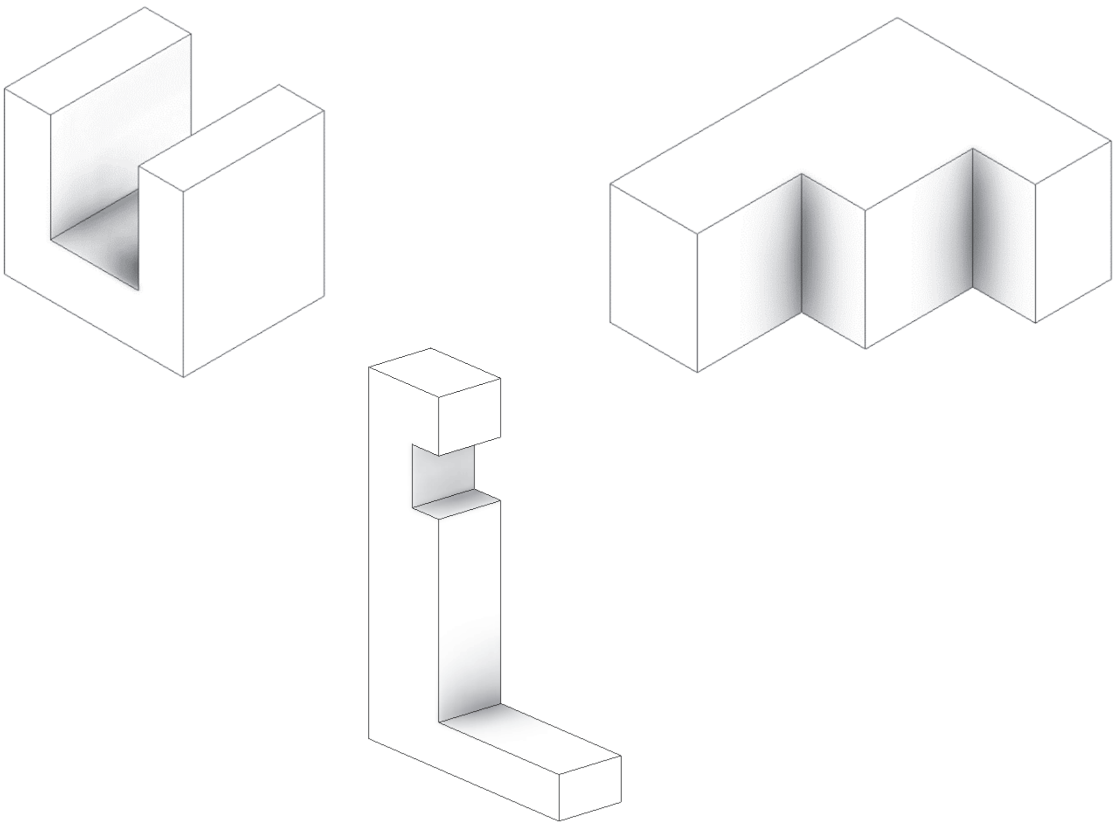
Step 7. Create a sketch (circle) on the top face and perform a negative Extrusion (BLIND and THROUGH).





EXERCISE 4: Simple Parts

Step 1. Model the following parts as simple extrusions and use the constraints. Redefine the sketch planes of the three parts once they have been created.



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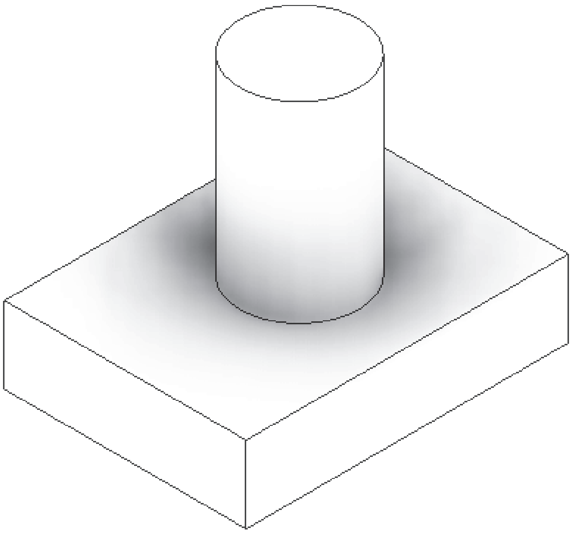
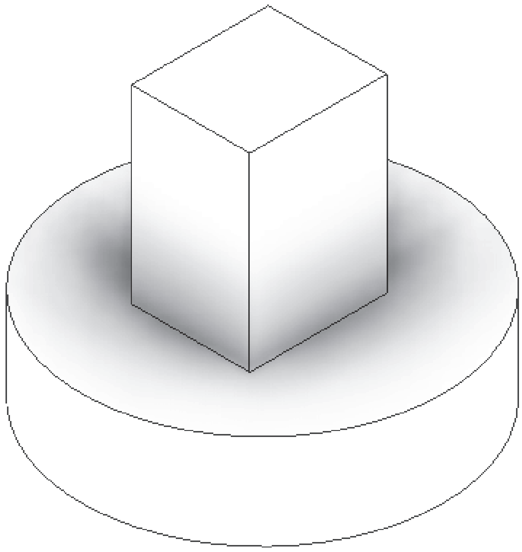
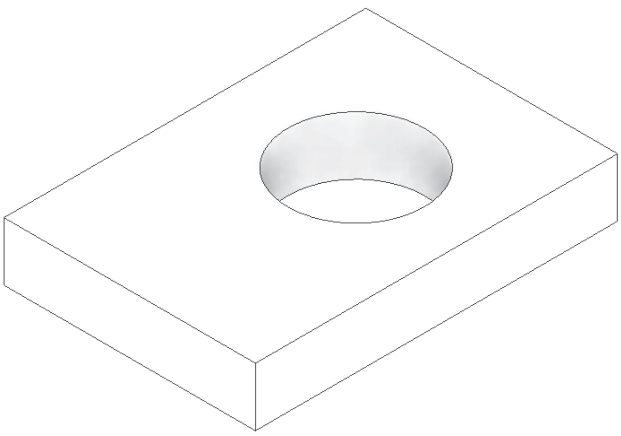
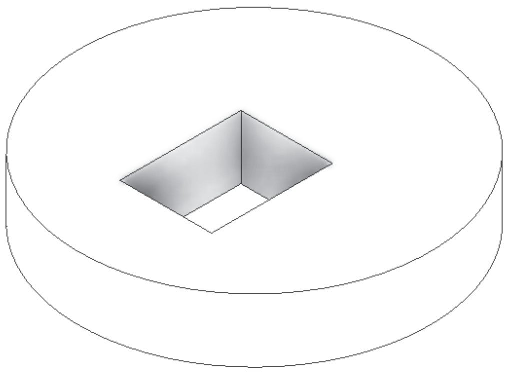
EXERCISE 5. Parts with multiple extrusions

Step 1. Model the following parts with multiple positive and negative extrusions.









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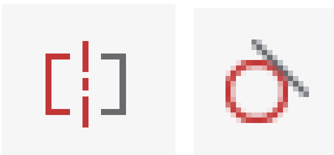
4. SESSION 2: CAD 3D- Constraints

In this session, we will study fundamental concepts of basic modelling. We will look at the following topics:

1. **Introduction to Basic Modelling II**:
    - **Simple Sketches**: we will create basic sketches to become familiar with the drawing and editing tools in the 3D environment.
    - **Geometric constraints**: we will apply geometric constraints to define the relationships and behaviours between the sketch entities, ensuring that they maintain their desired shape and position.

    - **Symmetry and tangency constraints:** the symmetry constraint ensures that two geometric entities (lines or arcs) are symmetrical around a centre line. In addition, the tangency constraint ensures that two entities, such as a circle and a line, touch at a single point without crossing each other.



    - **Dimensional constraints (Dimensions)**: We will use parametric dimensions to control the size and position of elements in the sketch, allowing for exact precision and adjustment of dimensions.

These concepts are essential for developing accurate and functional 3D models.

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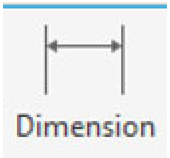
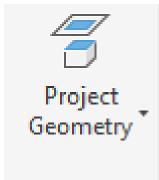
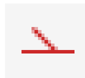
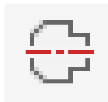
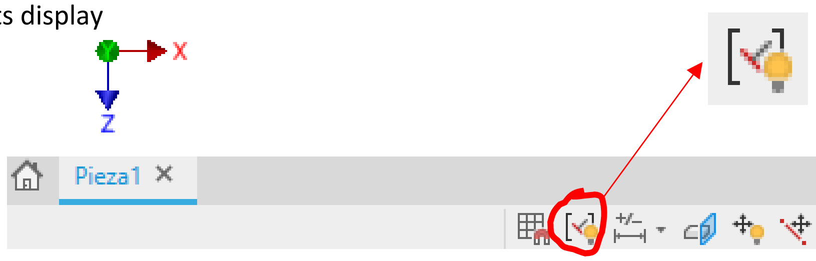
SUMMARY

SKETCH MODE			
Constraints	**Dimensions**	**Projections**	**Lines in sketches**
Equality dimension Parallel Perpendicular Point-point / line-point coincidence Straight-straight matching Horizontal Vertical	• Lineal • Aligned • Angle  	• Projection of axes and planes • Geometry projection  	• Construction Centreline - - - - -  • Axis Lines _._._. _ 

F8: activates the display of geometric constraints of the sketch
F9: deactivates its display



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Introduction to Basic Modelling II

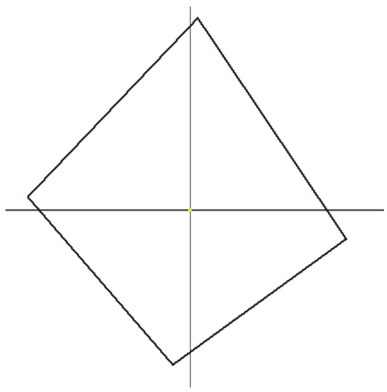
Modify QuadrilateralIrregularA.ipt and QuadrilateralIrregularB.ipt to obtain the desired geometry in Exercise 1, Exercise 2, and Exercise 4, respectively.

EXERCISE 1. Rhombus

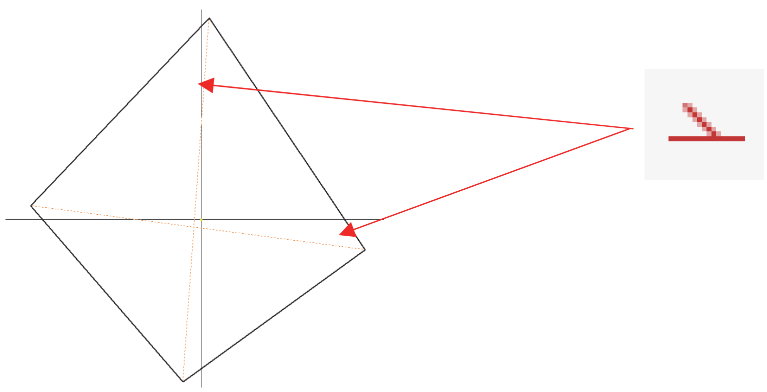
<u>Step 1</u>. Open the file **QuadrilateralIrregularA.ipt**

http://tiny.cc/0138_CI_A

<u>Step 2</u>. Irregular Quadrilateral drawn with the Line tool.



<u>Step 3</u>. Draw the diagonals with the line tool and convert them to construction centrelines.



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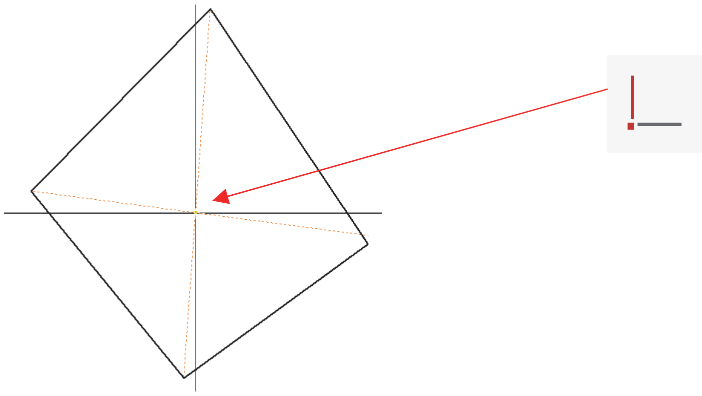
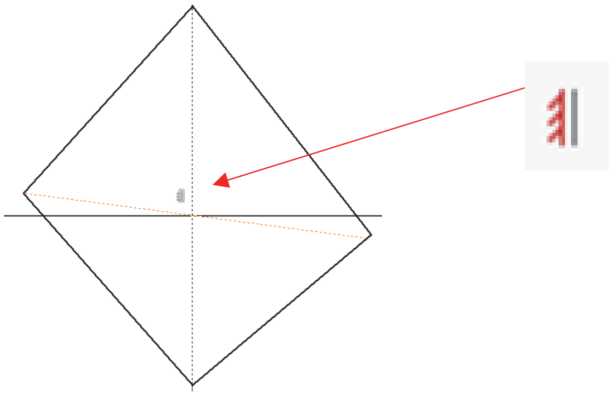
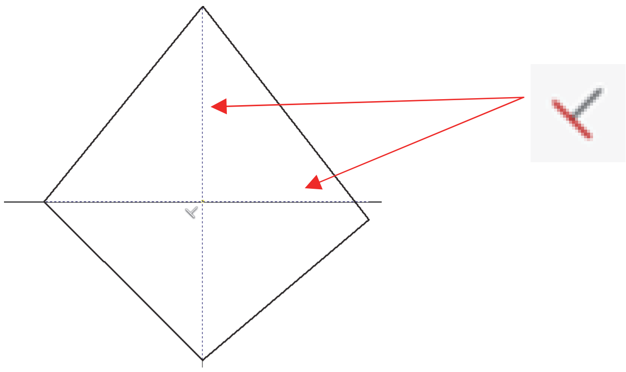
Step 4. Use the constraint of coincidence of the two diagonals about the origin (point-line).



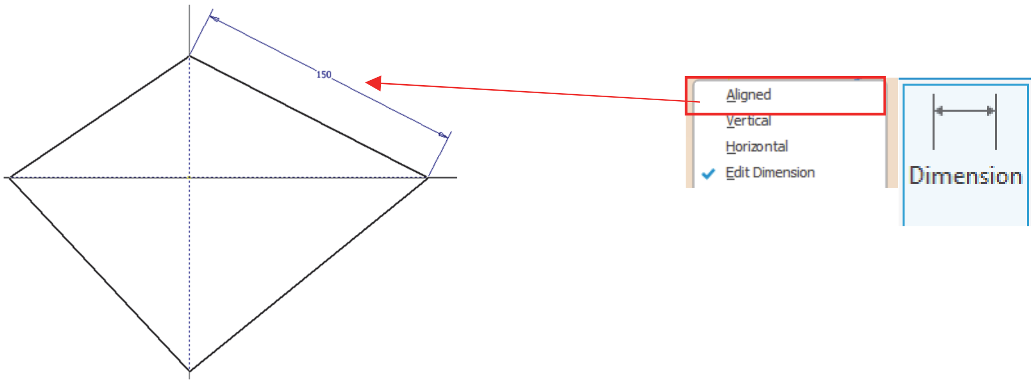
Step 5. Apply to a diagonal the vertical constraint.



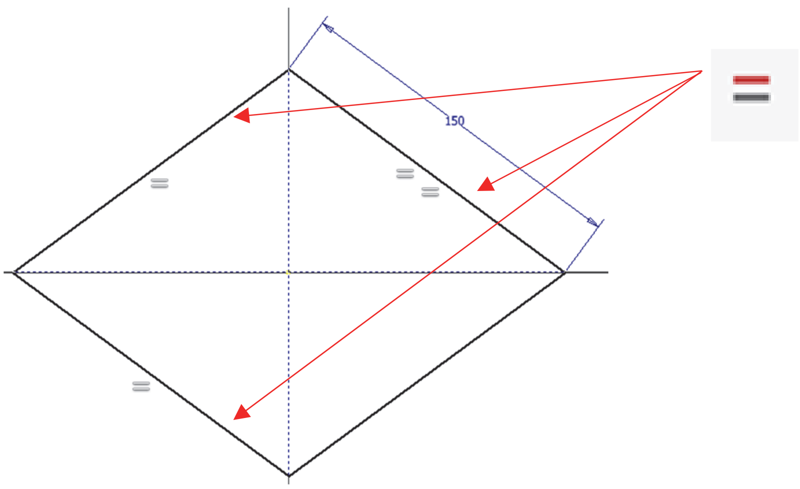
Step 6. Apply the perpendicular constraint to the two diagonals.



Step 7. Dimensioning the length of one side with an aligned dimension.



Step 8. Equality constraint applied to 3 sides (the other side will automatically also be equal) (RHOMBUS).



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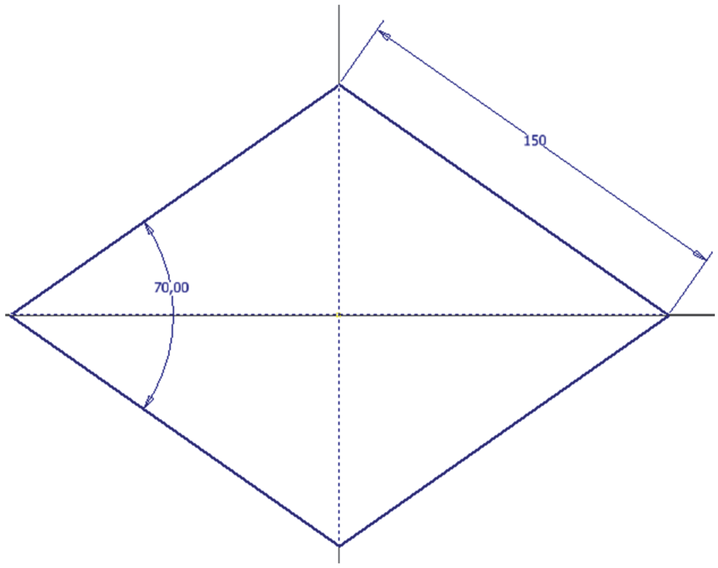
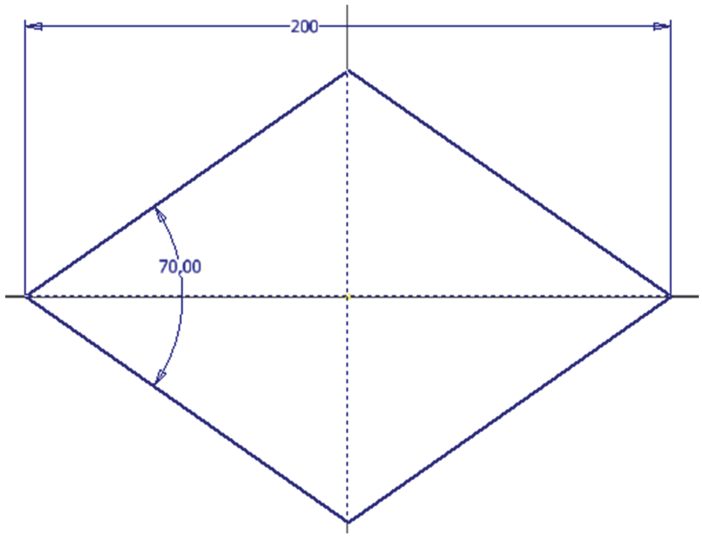
Step 9. Angle dimension: the rhombus is fully constrained.

Modify the dimensions and check that the rhombus is perfectly parameterised (defined).



Step 10. Change the side length dimension to the diagonal length.

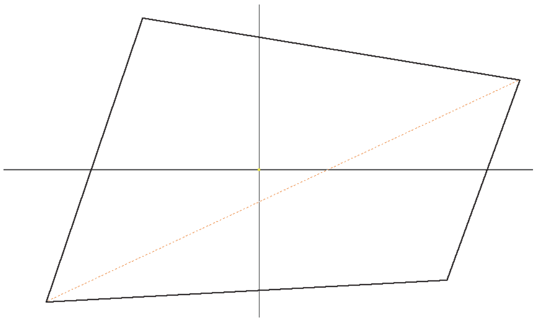


EXERCISE 2. Parallelogram / Rectangle / Square

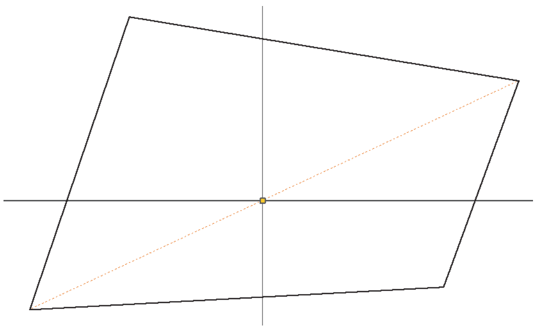
Step 1. Open the file **QuadrilateroIrregularB.ipt**

http://tiny.cc/0138_CI_B

Step 2. Draw a diagonal with the Line tool and convert it to a construction centreline.



Step 3. Constraint of coincidence of the midpoint of the diagonal with the origin (point-point).



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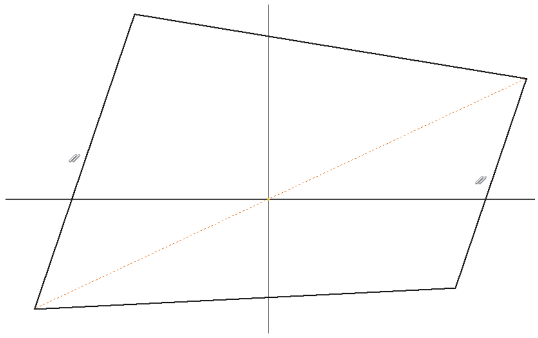
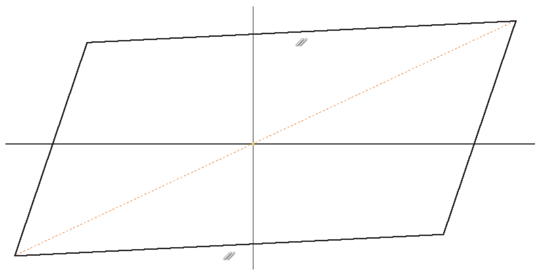
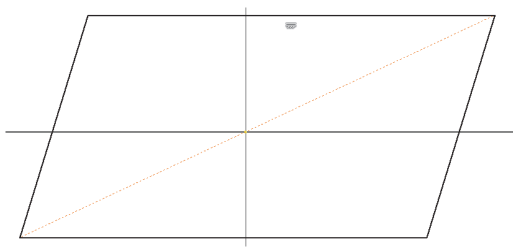
Step 4. Parallelism constraint of 2 opposite sides.



Step 5. Parallelism constraint of the other pair of opposite sides (PARALLELOGRAM).

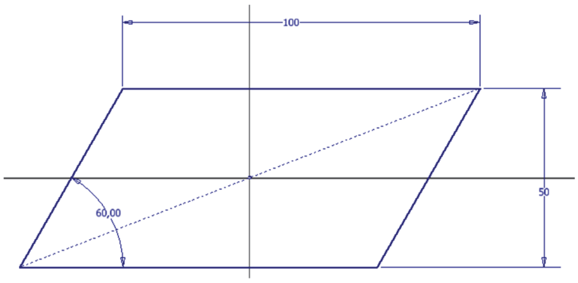


Step 6. Horizontal constraint applied to one side.



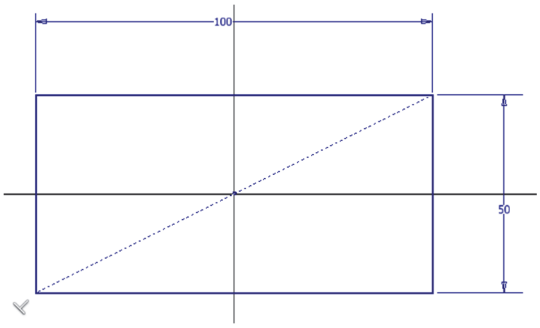
Step 7. Dimension of the parallelogram: 1 angle and two non-parallel sides (or vice versa). The parallelogram is constrained.

Modify the dimensions and check that the parallelogram is perfectly parameterised.



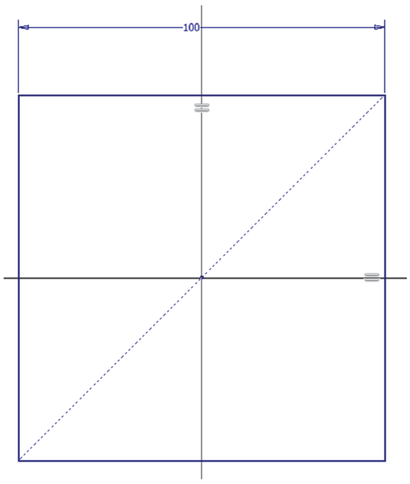
Step 8. Change the angle dimension to a perpendicular constraint (RECTANGLE). The rectangle remains fully constrained.

Modify the dimensions and check that the rectangle is perfectly parameterised (defined).



Step 9. Change a side dimension to an equality constraint (SQUARE). The square is still fully constrained.

Modify the dimension and check that the square is perfectly parameterised.



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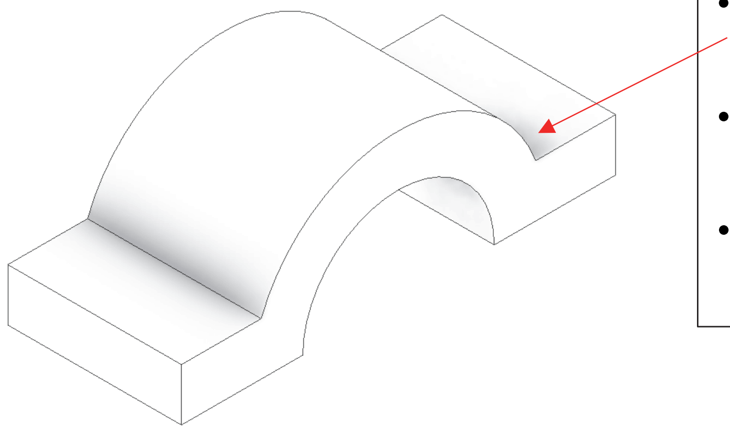
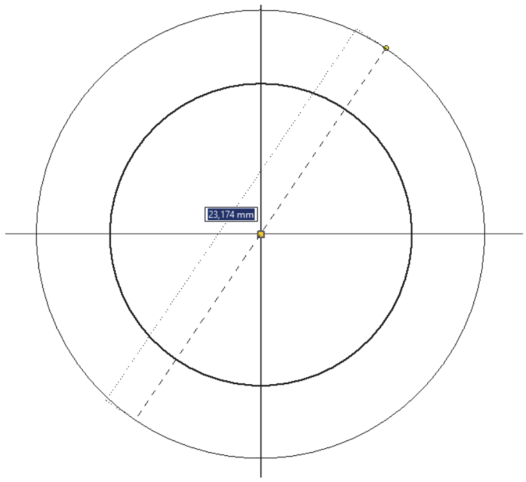
EXERCISE 3. Concentric Parts



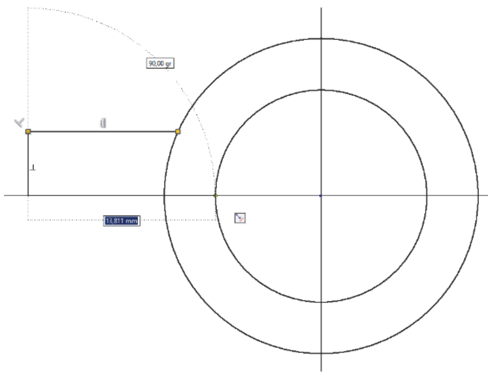
Learning objectives:

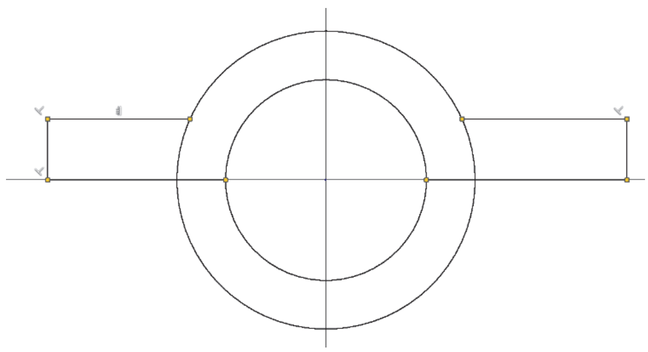
- To understand the operation of geometric constraints in the sketch.
- To deal with the dimensional restrictions in the sketch (dimensions).
- To learn how to modify the dimensions of models.

Step 1. We will draw two concentric circles.



Step 2. We will then draw the lines on the left and right sides.





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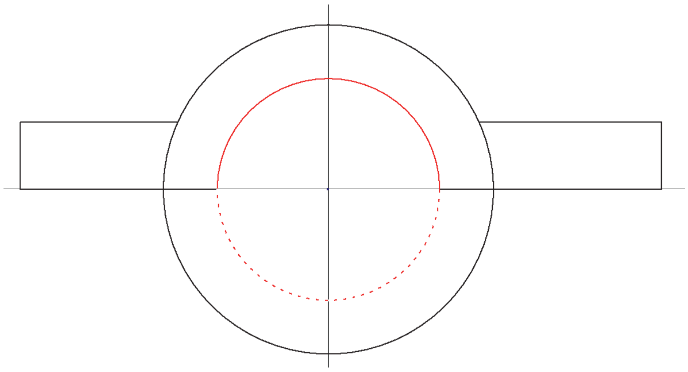
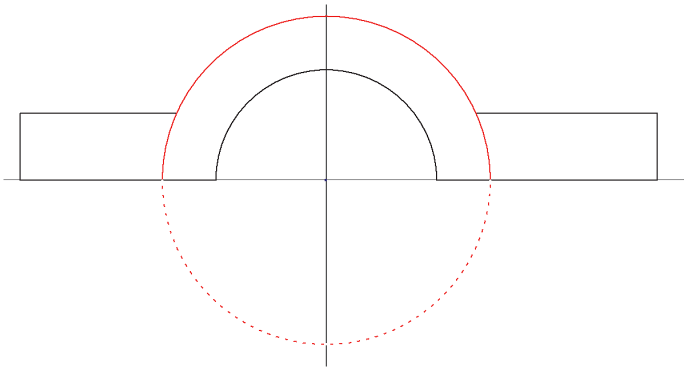
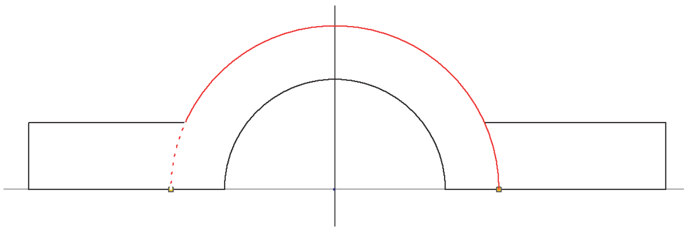
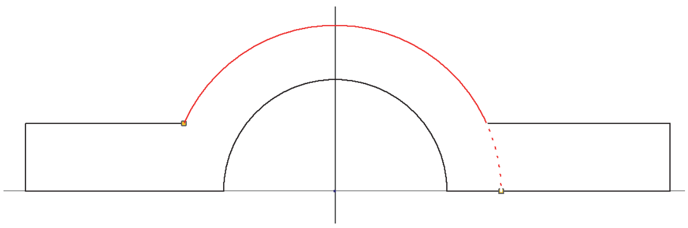
Step 3. We will trim the excess with 









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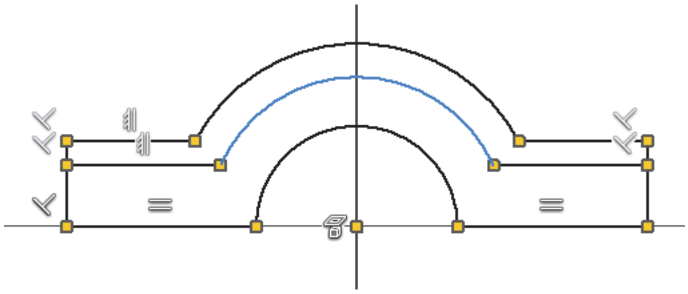
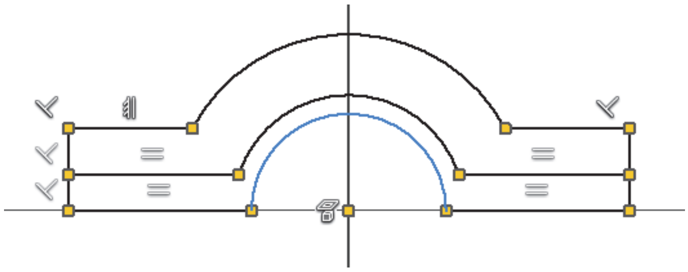
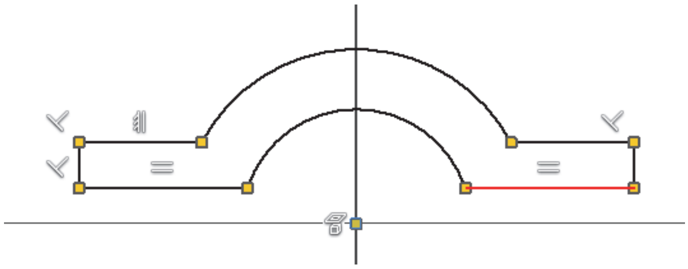
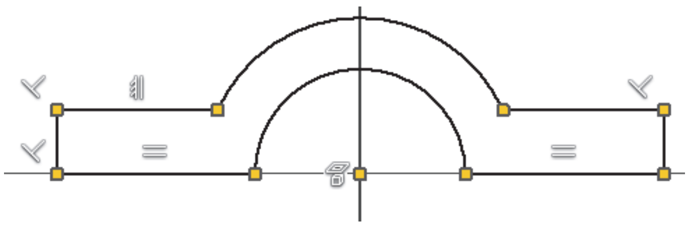
Step 4. Try moving the remaining arc entities to see how the sketch deforms or varies (press F8 to display all sketch constraints and F9 to hide).



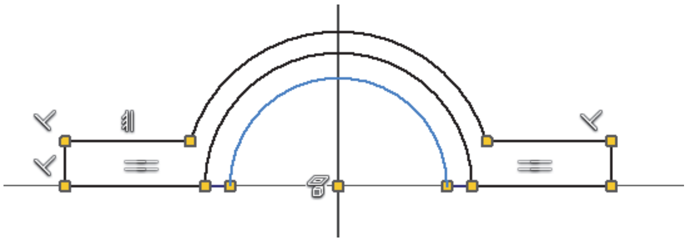


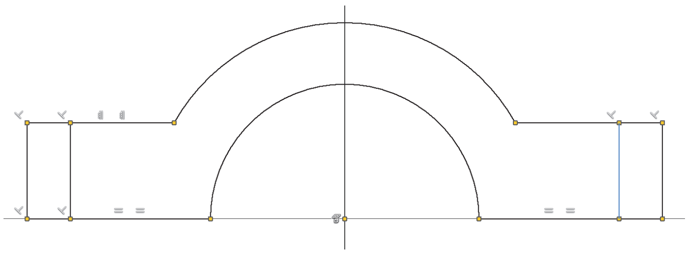
Step 5. Apply the horizontality constraint of the centre point with the lower right side (in red) with 





Step 6. Now check how it is deformed.



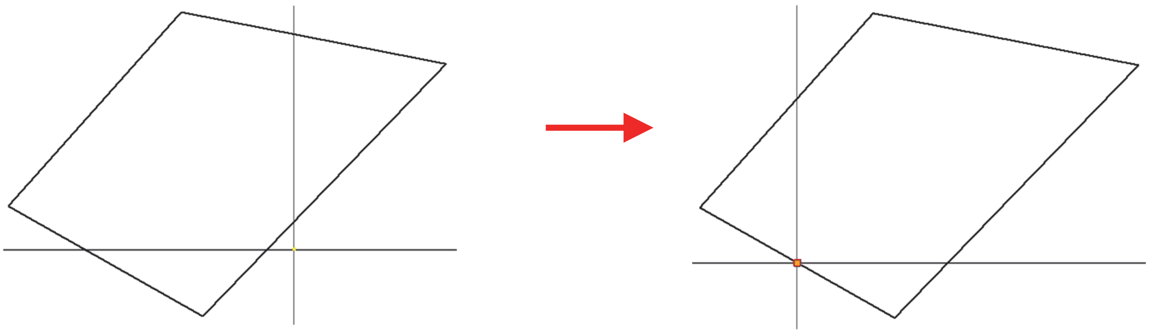


EXERCISE 4. Trapezoid with Symmetry and Equality constraints

Step 1. Open the File QuadrilateralIrregularC.ipt and convert the sketch to an isosceles triangle.

http://tiny.cc/0138_CI_C

Step 2. Apply a constraint of coincidence of the midpoint of the lower side with the origin (point-to-point).



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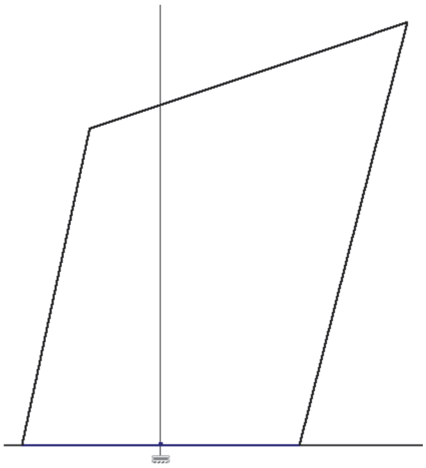
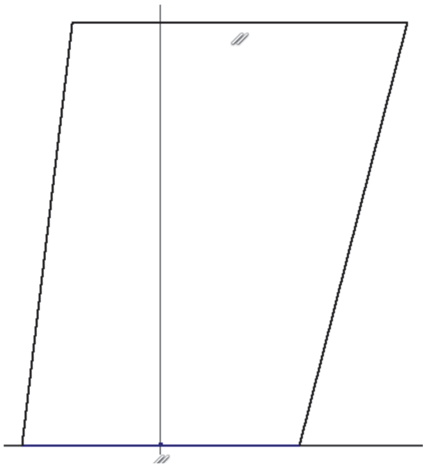
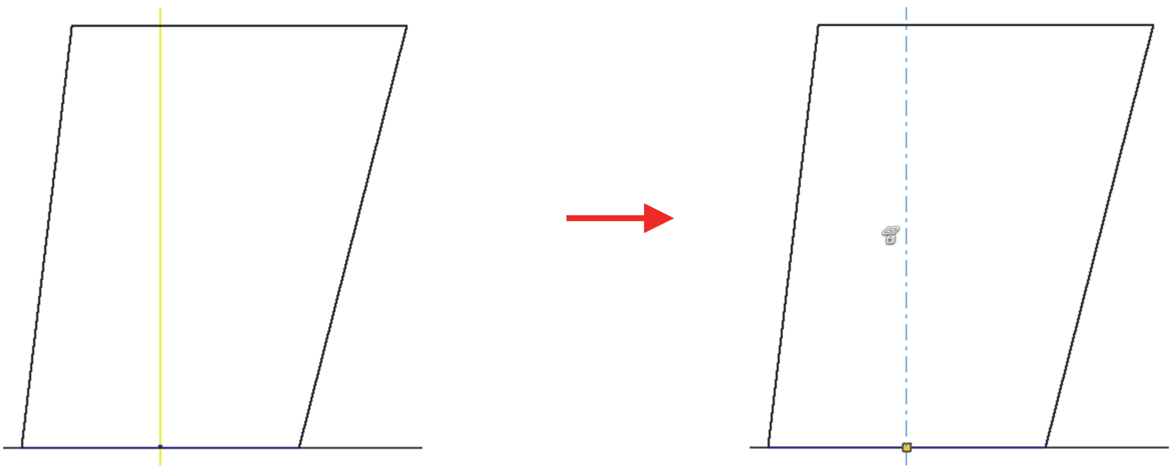
Step 3. Apply a horizontal constraint to the bottom side.



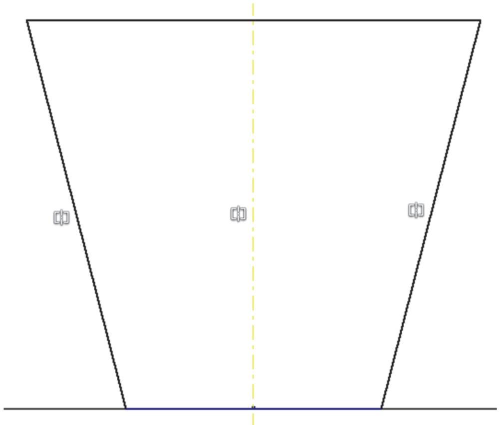
Step 4. Apply a parallelism constraint between the top and bottom sides.



Step 5. Project the Z-axis and convert it from the projected line to the construction axis.

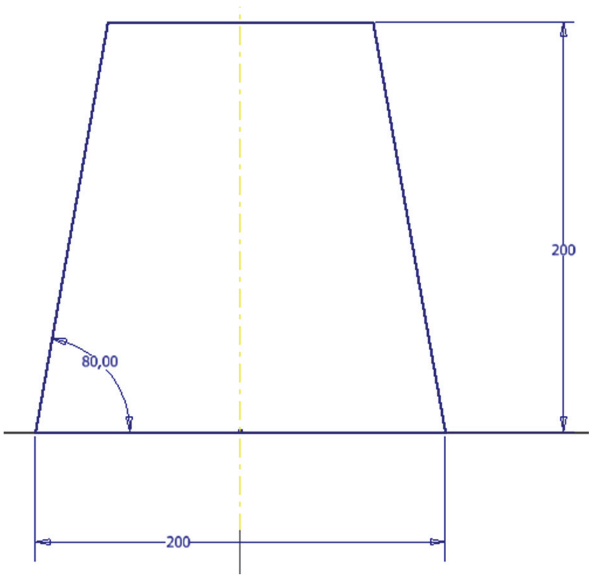


Step 6. Apply Symmetry constraint (mirror) of the lateral sides in relation to the central axis (ISOSSCEL TRAPECIUM).



Step 7. Dimension of the isosceles trapezium: 1 angle, the base, and the height. The isosceles trapezium is completely constrained in this way.

Step 8. Modify the dimensions and check that the isosceles trapezium is perfectly parameterised (defined).



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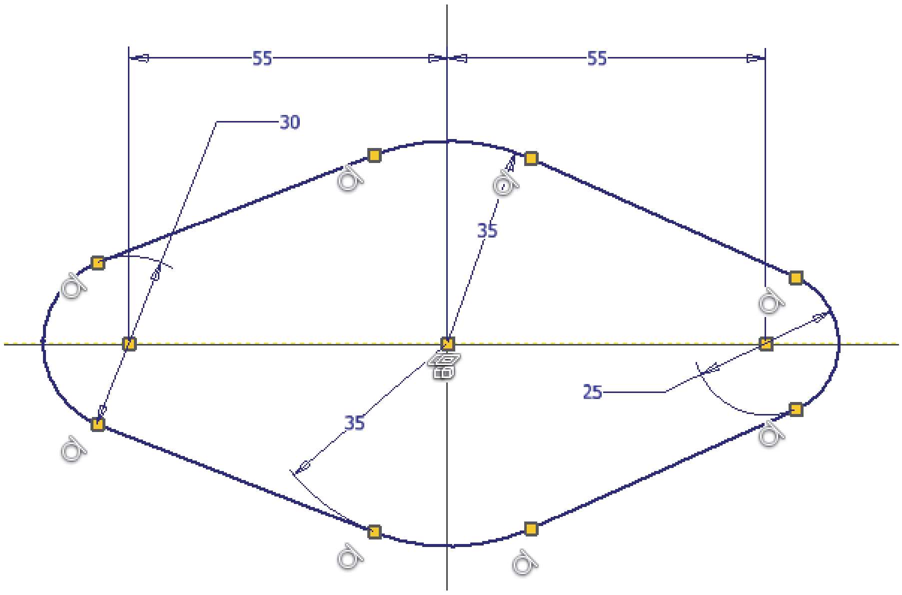
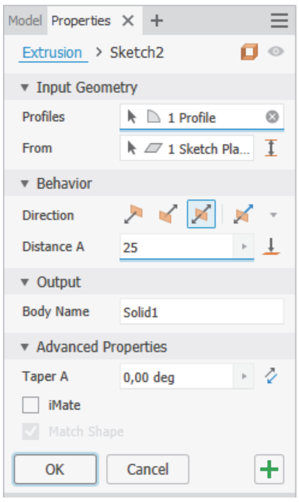
EXERCISE 5. Sketch with Tangency

Step 1. Create the following dimensioned sketch by applying tangency constraints.



Step 2. Apply the rest of the restrictions that are necessary so that the sketch is completely constrained.

Step 3. Create a 25mm symmetrical extrusion.



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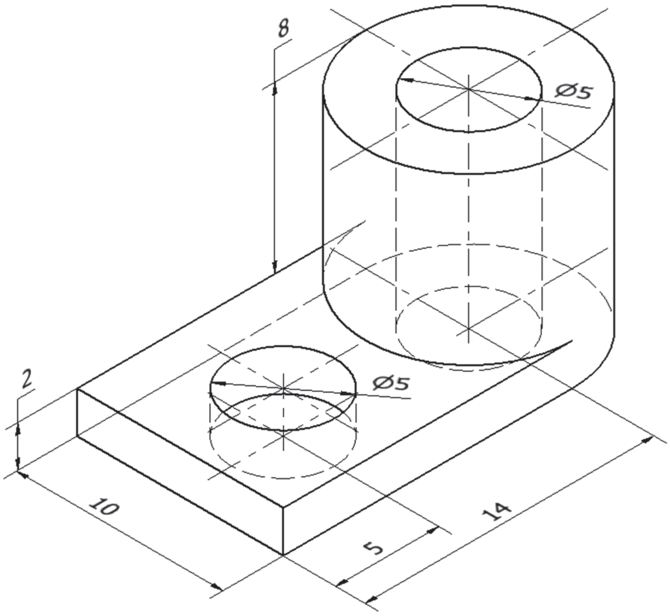
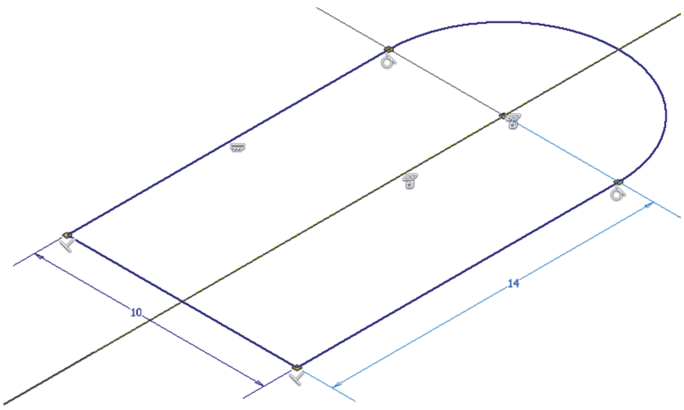
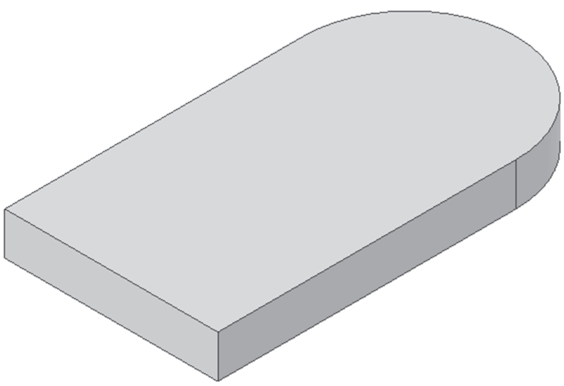
EXERCISE 6. Session summary exercise

Make the following part using extrusions and the corresponding restrictions.



Step 1. EXTRUSION. Sketch placed on the horizontal plane. Place the centre of the arc at the centre of the planes of origin.





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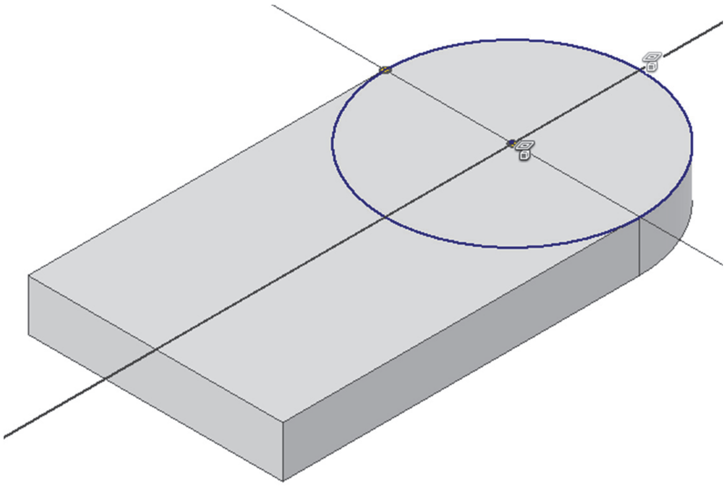
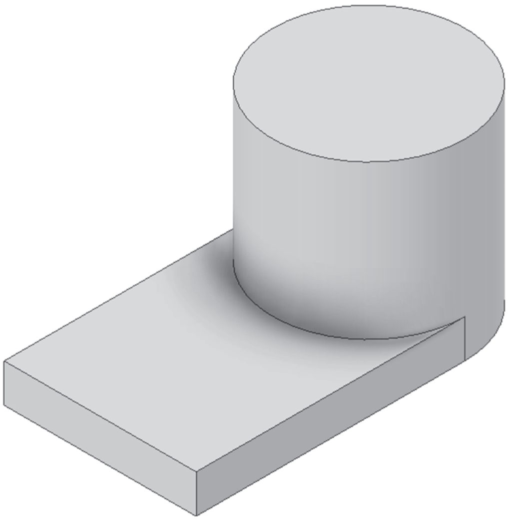
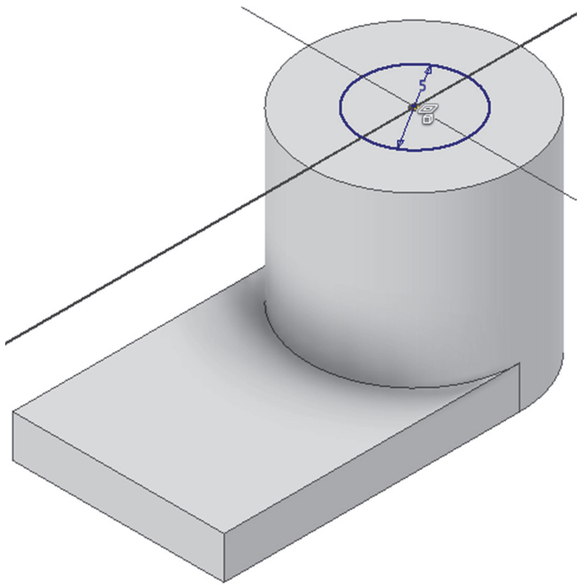
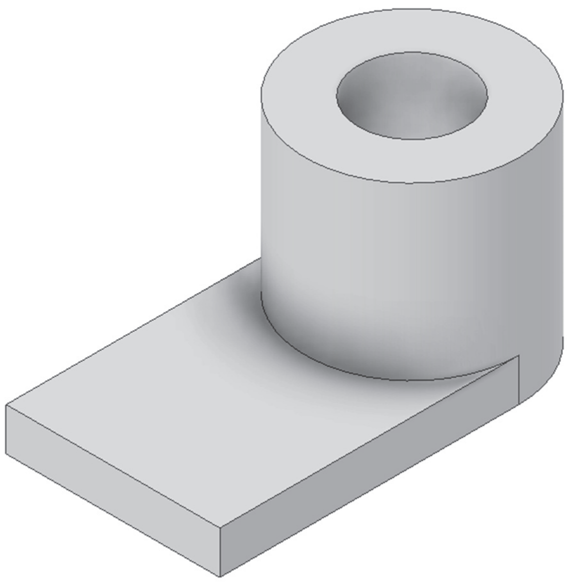
Step 2. EXTRUSION. The sketch is located on the upper face (the diameter is not re-dimensioned, but matching constraints are used).





Step 3. CUTTING EXTRUSION. The sketch is located at the upper base of the cylindrical part. Sketch with circumference with centre coincident (concentric) with the upper base of the cylinder.





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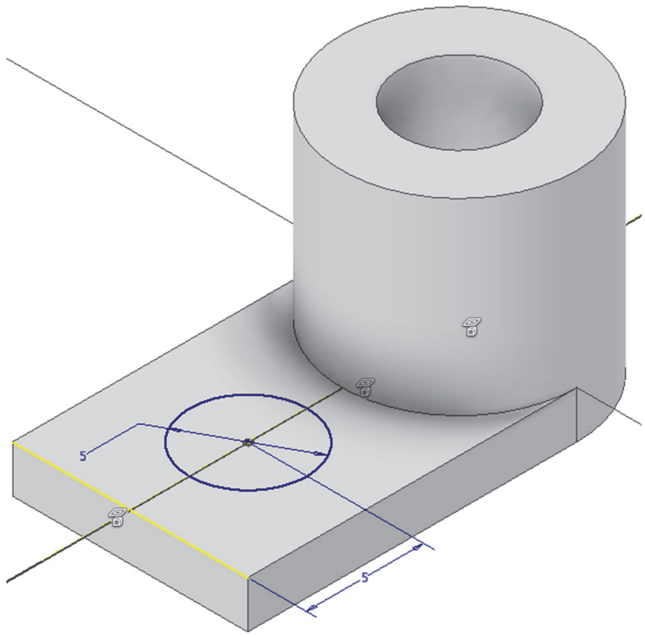
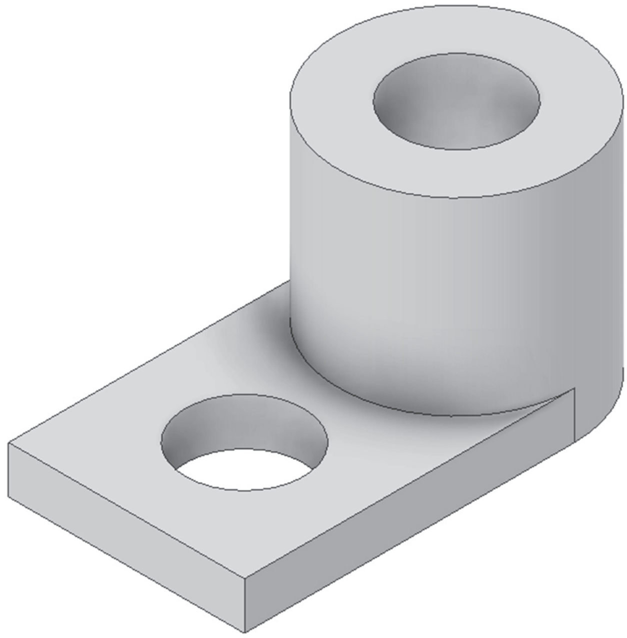
Step 4. CUTTING EXTRUSION. The sketch is located on the upper face of the base. Sketch with circumference with centre on the symmetry axis (command mirror in Inventor).





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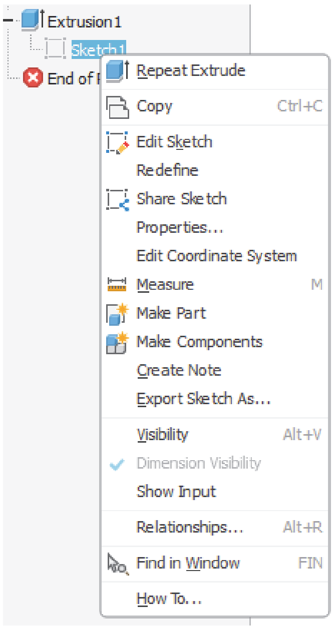
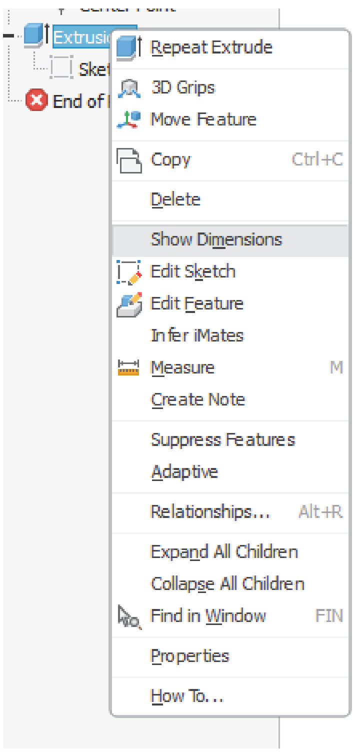
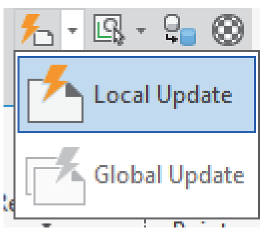
Step 5. Modify the dimensions in two ways:

From sketch mode: "Edit sketch"



From the model tree: Select an operation and "Show dimensions" and "Refresh" the tree with





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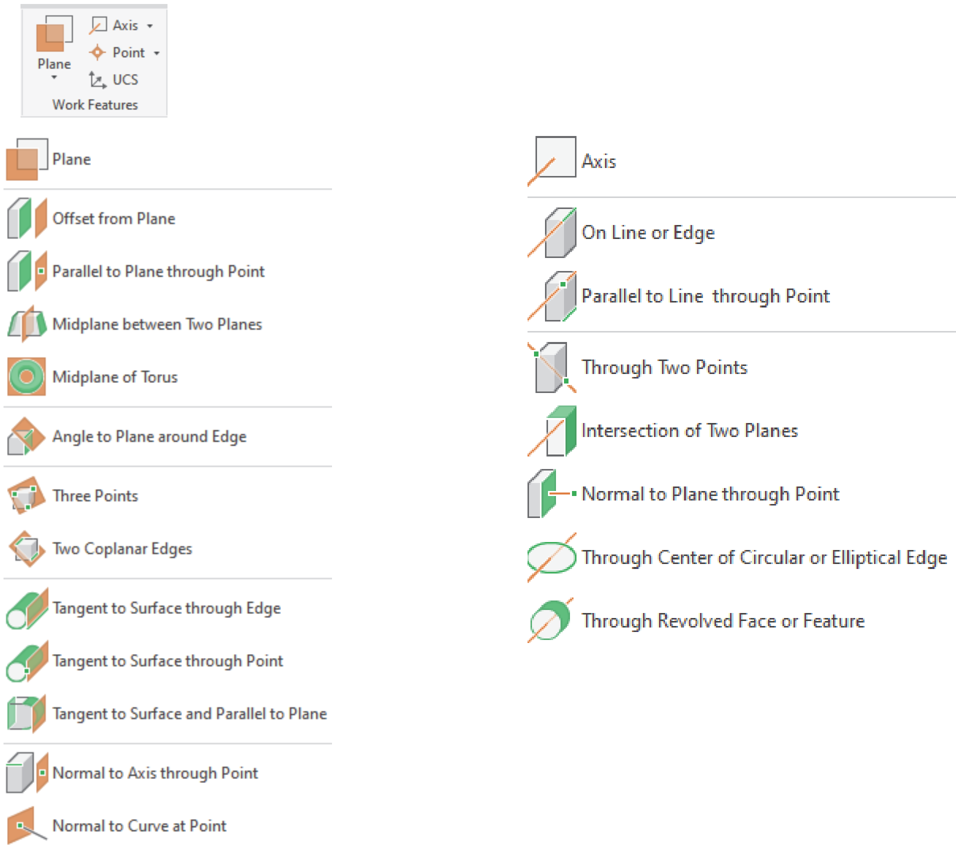
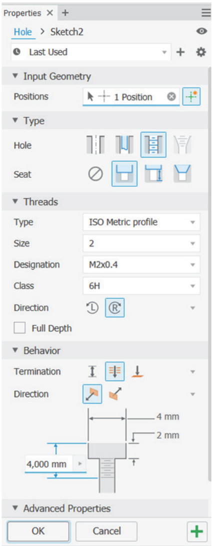
5. SESSION 3: CAD 3D- Auxiliary planes and Holes

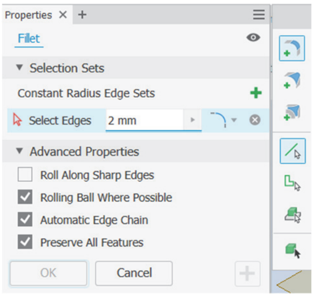
In this practical session, we will start using **working planes and axes** necessary to define the geometric references in 3D modelling.

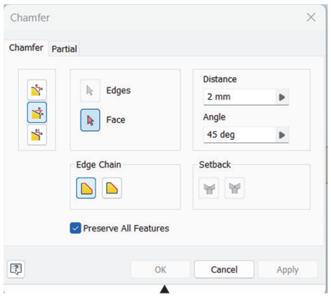


We will then look at the **creation of holes**, including how to create them based on points and concentricity constraints, which ensure perfect alignment in cylindrical components. We will learn to implement them using sketches with points for accurate placement.

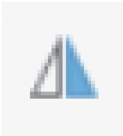


We will then explore **rounding and chamfering**. They are basic industrial techniques for softening edges and corners.





Finally, we will recall the importance of symmetries in design, which help to maintain consistency by applying changes uniformly across a symmetrical plane (command mirror in Inventor).



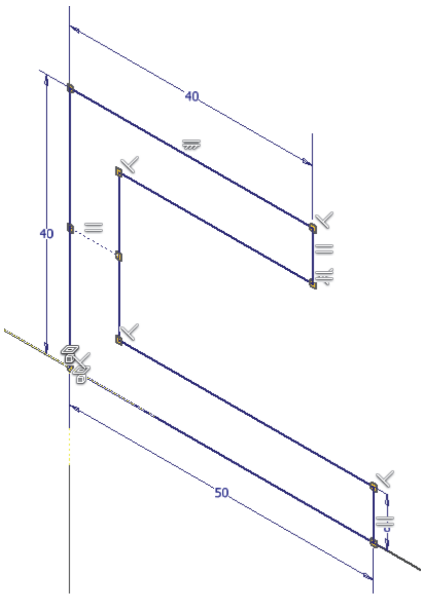
Introduction to Basic Modelling II

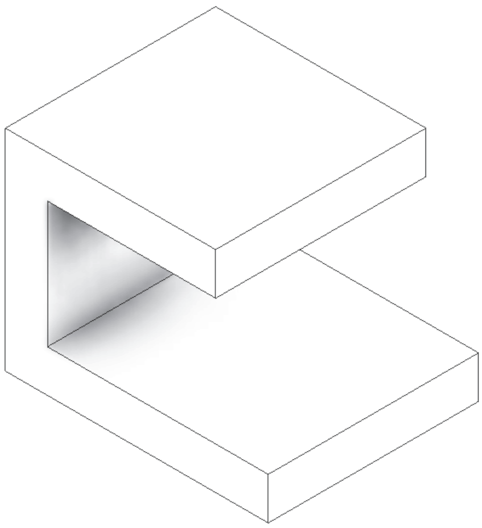
EXERCISE 1. Exercise Planes

Step 1. Create the following sketch.

Learning objectives

- Understand and learn to use the working operations: Planes and Axes
- Create symmetrical copy operations
- Fully constrain a sketch





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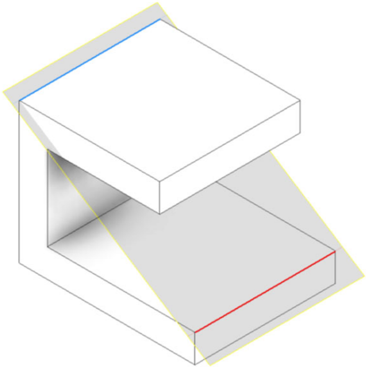
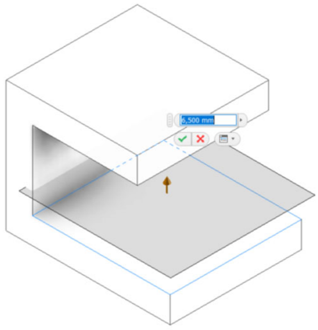
Step 2. Create the following planes.

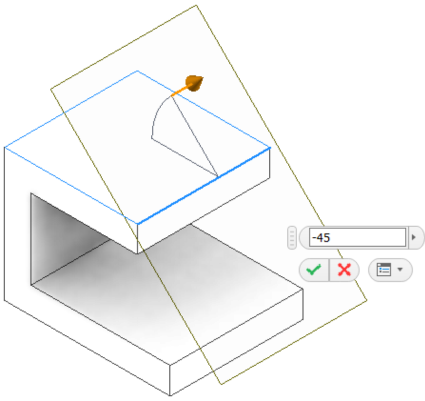
Plane passing through two coplanar edges



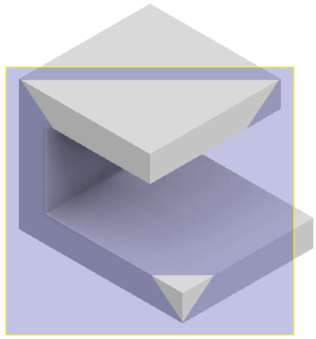
Due to a phase shift from another plane



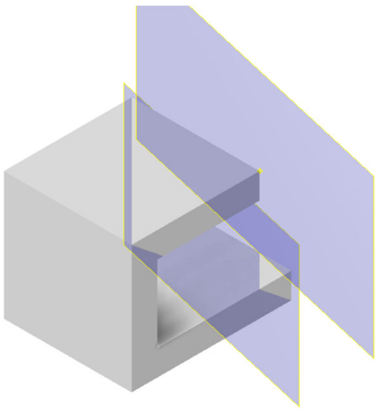
Plane passing through an edge and with an angle on another plane



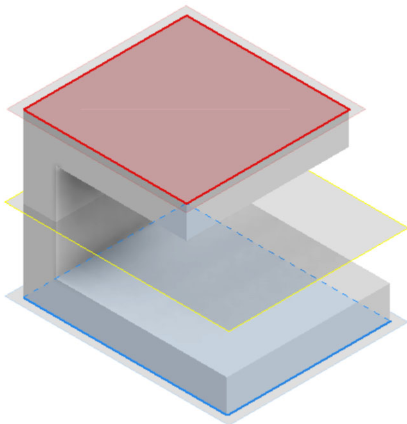
Plane passing through three points



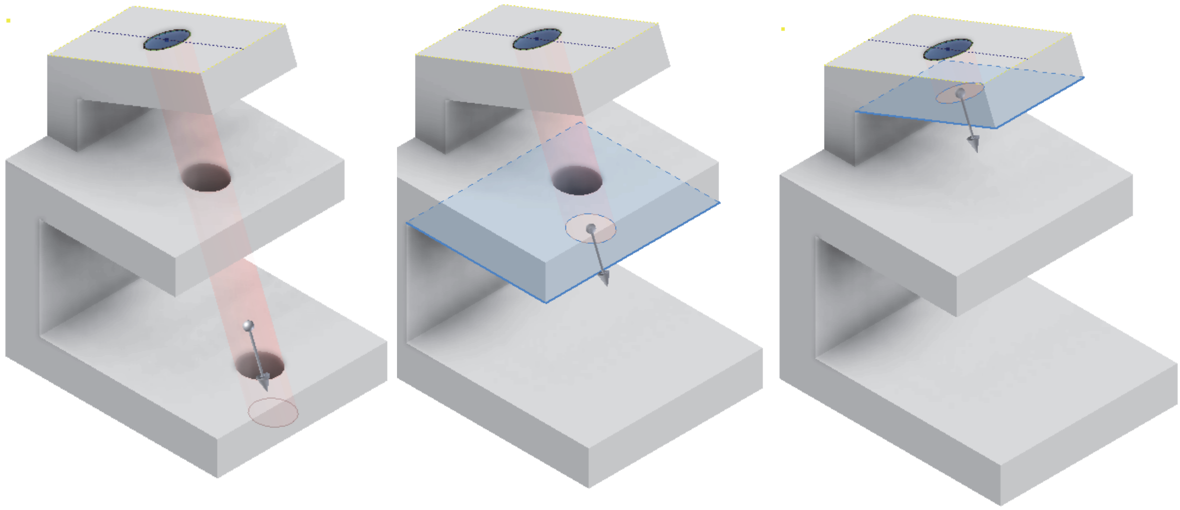
Plane through point (created in the model) and parallel to plane



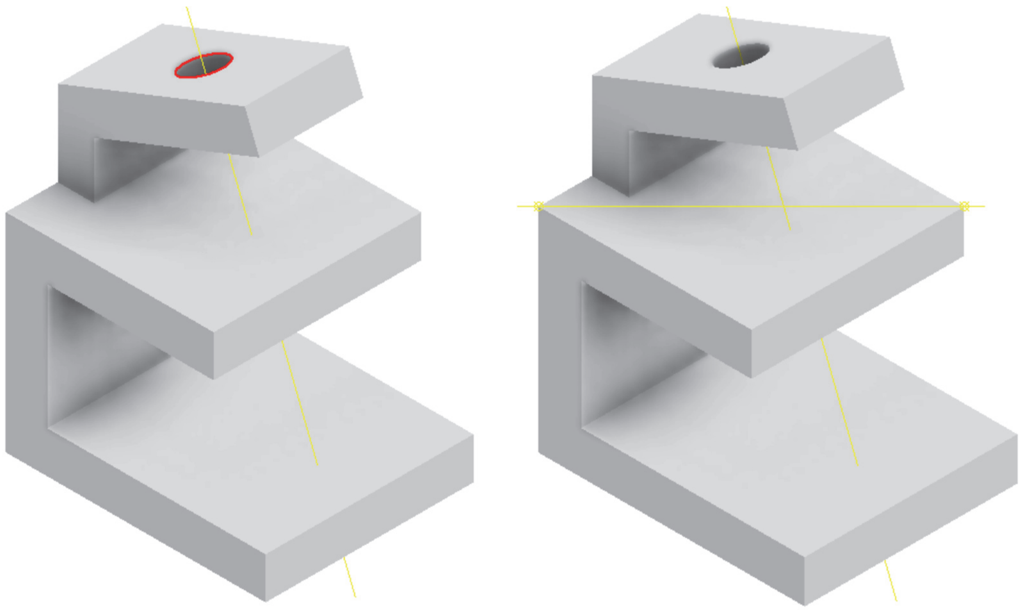
Middle plane between two planes



Step 3. Create the top fin in the vertical mid-plane and then a negative extrusion of a circle sketch that goes through the whole part, only two walls and one wall.



Step 4. Create an axis coaxial to the cut made (left) and an axis passing through two points (right).



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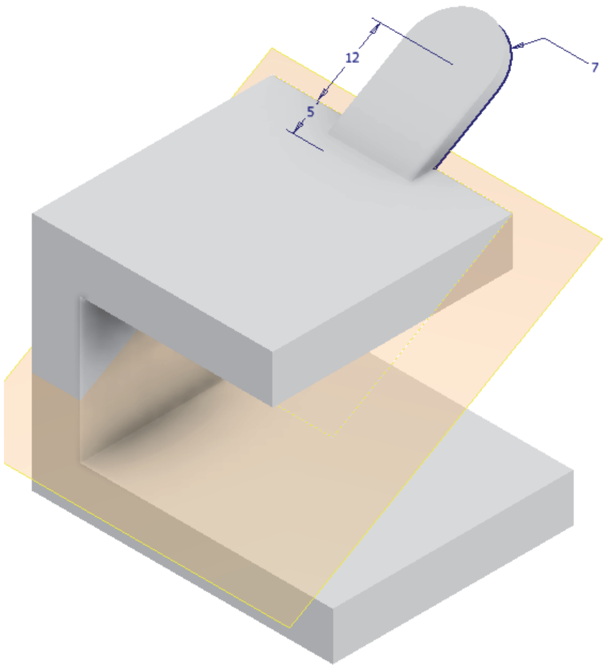
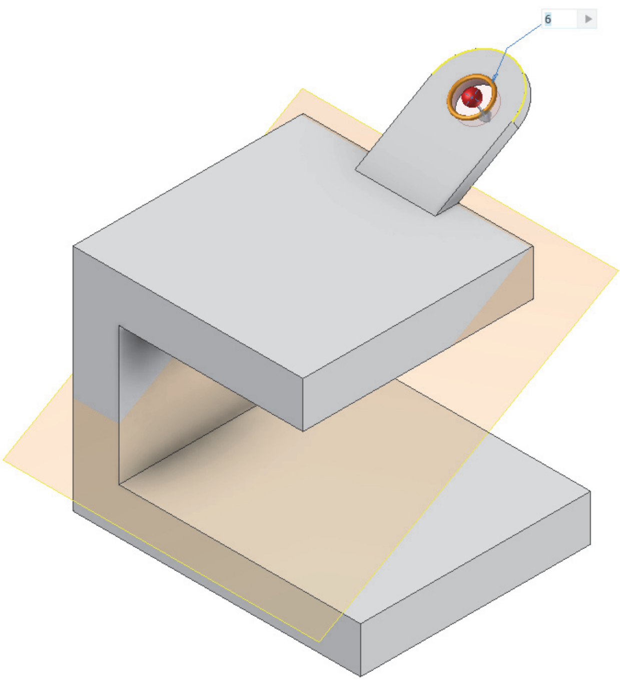
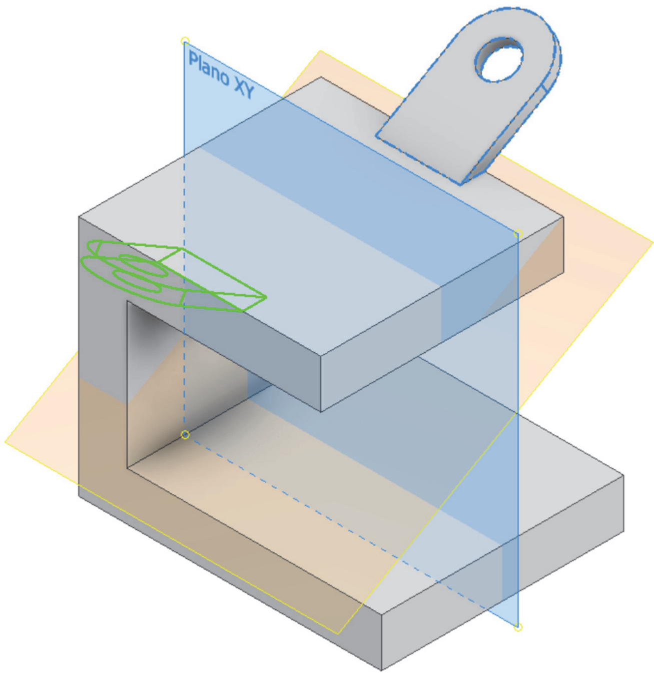
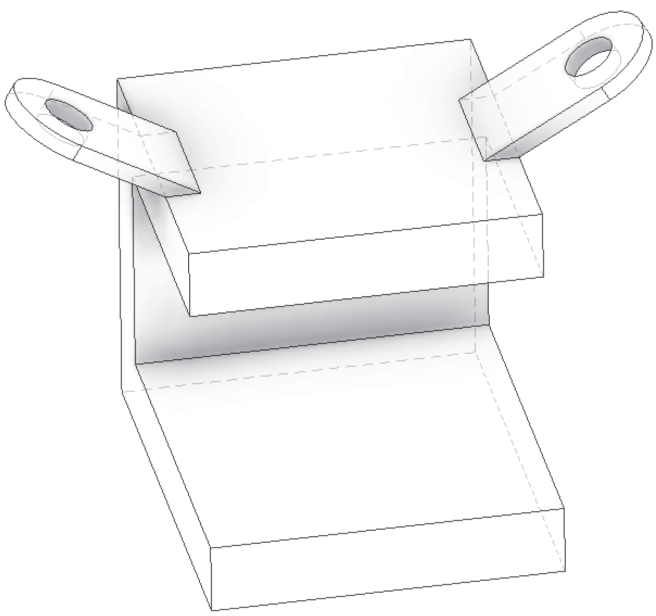
Step 5. Create a lateral tab (lug) on a plane inclined 30° above the upper plane and a concentric straight-through hole of the dimensions shown.





Step 6. Finally, the symmetry of both operations, the ear (extrusion) and the hole are made on the median plane of the part.





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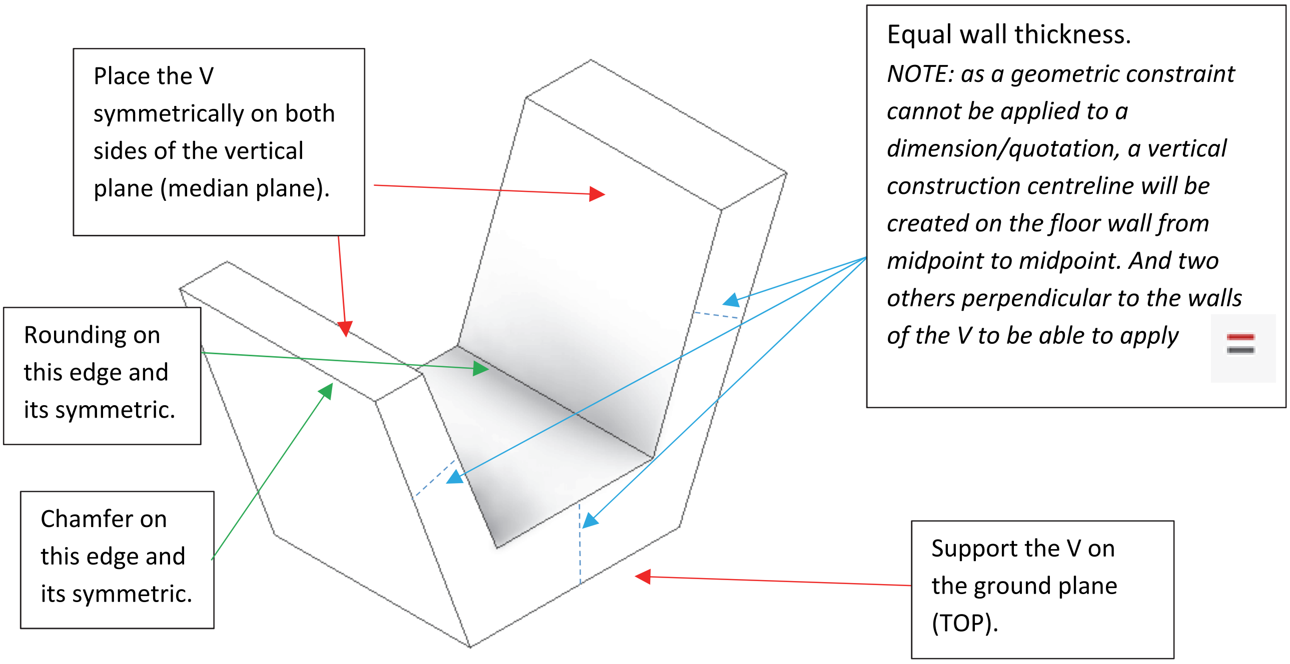
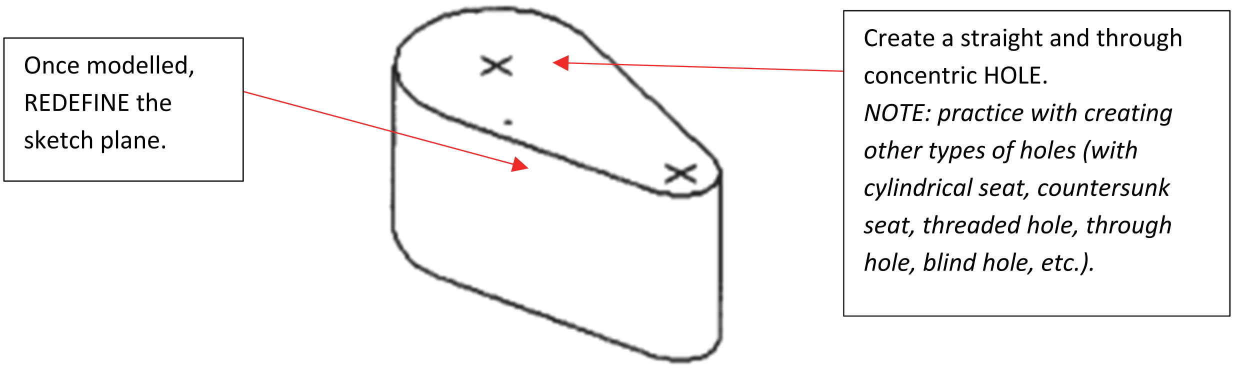
EXERCISE 2. V paperweight

Play with the restrictions: apply, eliminate, modify... so that the V is centred on a medium plane, resting on the ground, the thicknesses of its sides and the side resting on the ground are equal (the opening angle must be 45º).

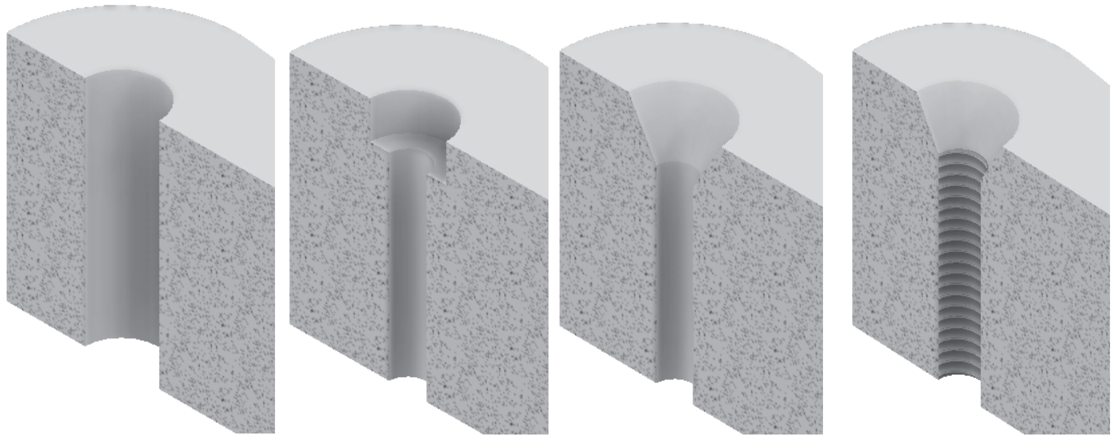


EXERCISE 3. Pinball flippers

Create different restrictions: apply, delete, modify... Redefine the plane of the sketch: make the plane vertical instead of horizontal (without redrawing the sketch). Create a concentric hole in the thickest area to anchor it and make it pivot.



Examples of the distinct types of holes are shown below:

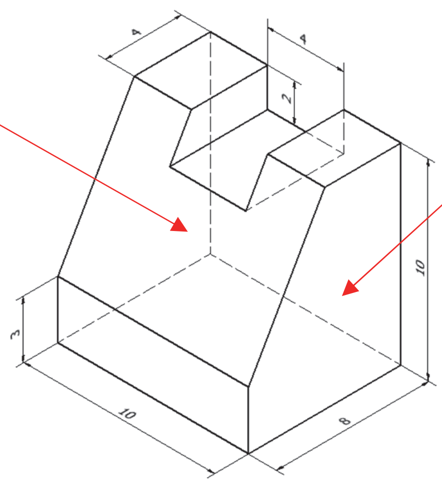


EXERCISE 4. Part with Rounding and Chamfers

Model the following part to meet the following, using the corresponding geometric constraints:

1) Supported in the horizontal plane.
2) Centred in a vertical plane.
3) The upper groove must be centred so that when the dimensioning is changed, the result maintains the symmetry of the part (mirror).
4) Create a straight-through cylindrical hole in the centre of the inclined face using a sketch with dots.
5) Pour the part from the sides using an "offset" from the sketch of the first extrusion.
6) Finally, ROUND the upper edges of the part that are perpendicular to the back face.

Create a straight through HOLE in the centre of the face and on the middle plane.
NOTE: create a sketch with a point at this position.



Create a negative EXTRUSION with a sketch to be drawn using the OFFSET command
NOTE: the offset dimension shall be 3 mm.

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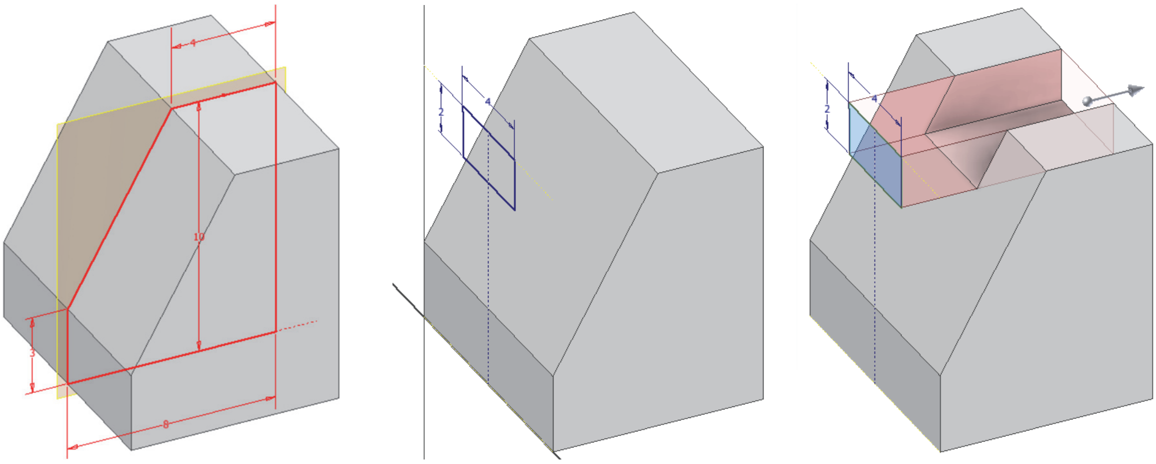
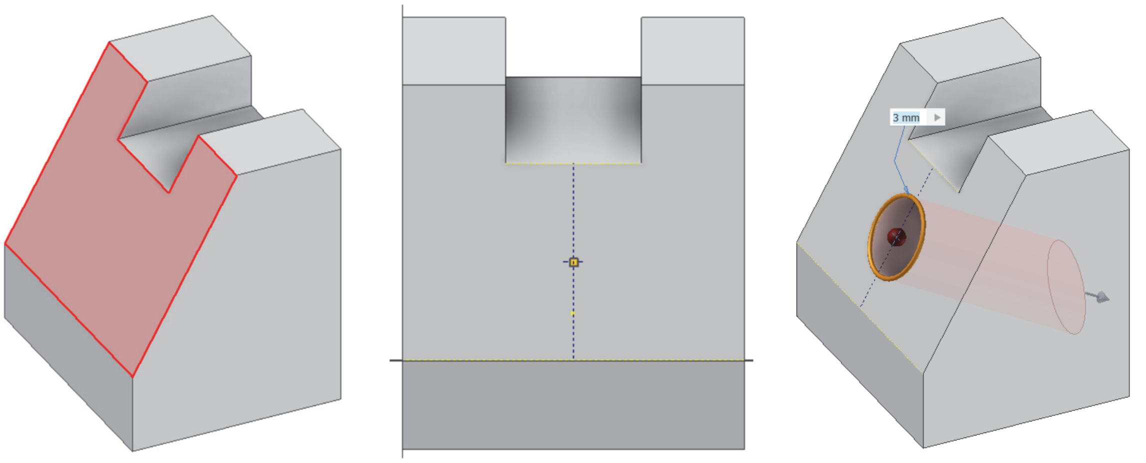
Steps 1, 2 and 3

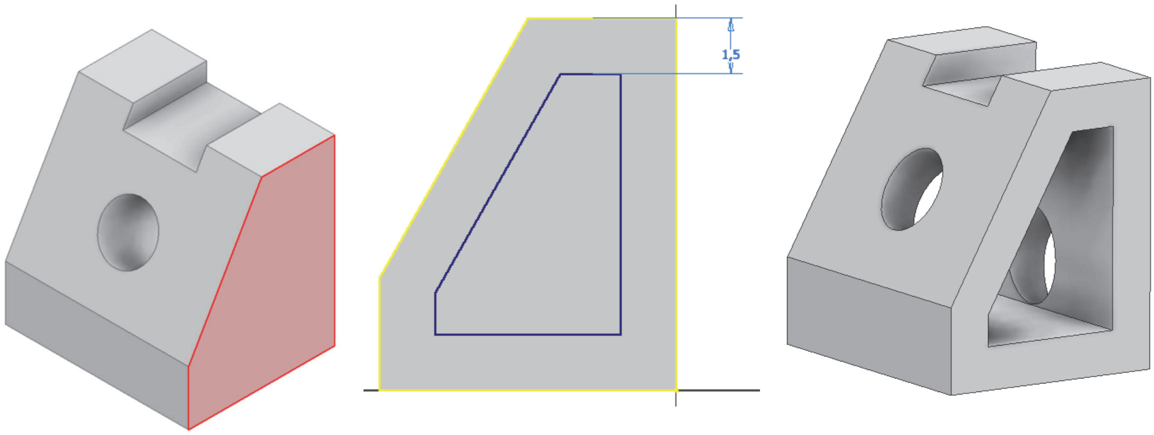


Step 4

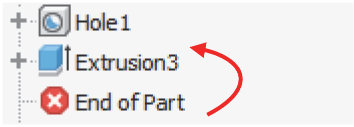


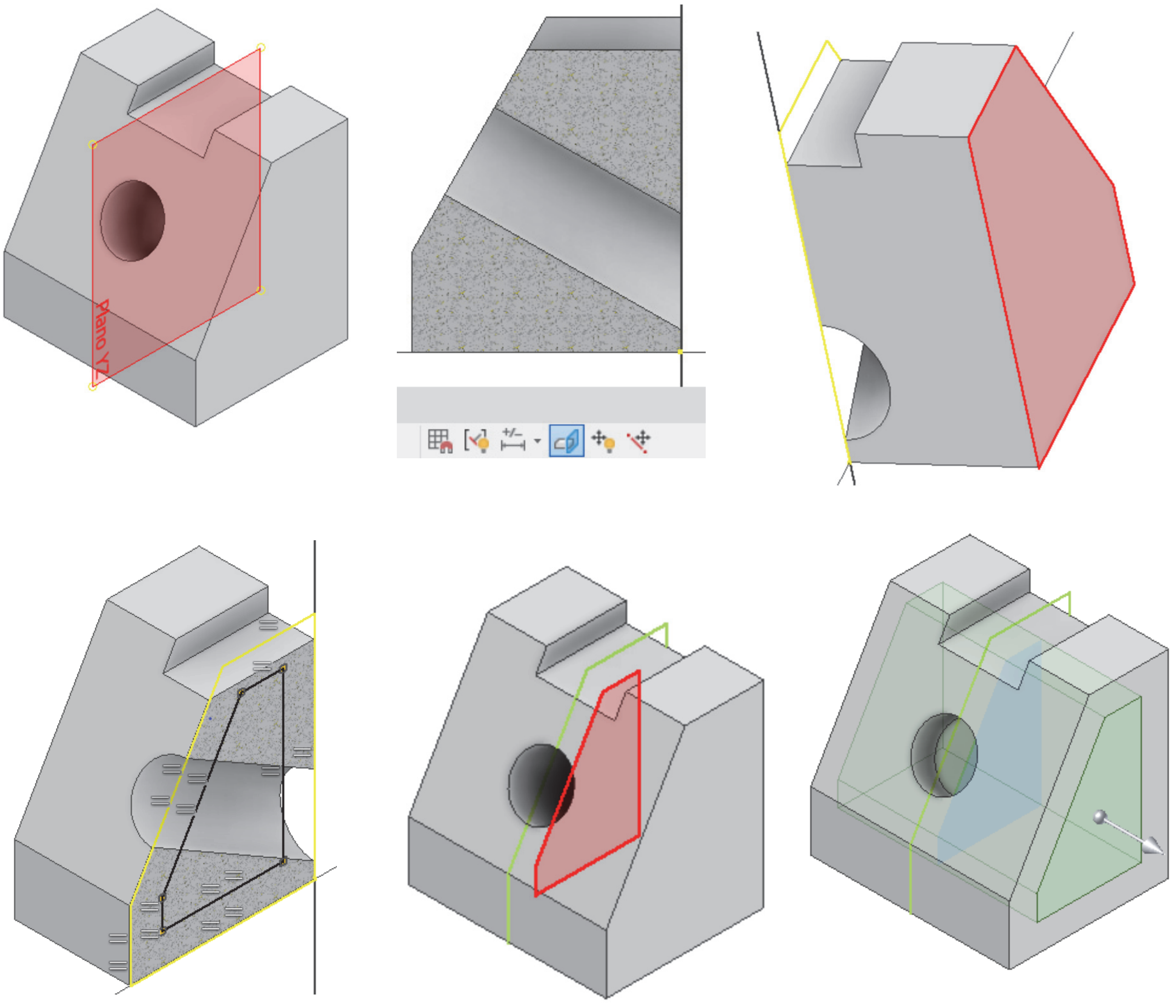
Step 5. Modify the height of the groove to 1 mm beforehand and sketch using the OFFSET.

Mode 1: Take a side face of the part as a sketch plane.

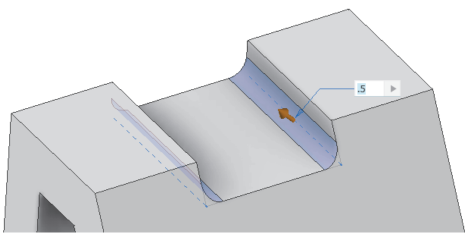


Mode 2: Take the sketch plane as the central origin plane (the middle plane of the part). To do this, we suppress the last extrusion or move the end of the part up in the model tree:





Step 6. Implement a rounding.



Step 7. Change rounding to chamfering (1 mm x 45º).

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6. SESSION 4: CAD 2D – Environment and use of AutoCAD©

This session is designed to provide a basic understanding of the fundamental functions of AutoCAD©. We will start studying the program through the various tools it offers us: the drawing area, the command line and the status bar with its tabs and activation buttons.

During this session, we will address the following topics:

1. **Environment of AutoCAD©**
    - Main tools
    - Drawing area
    - Command line
    - Status bar
2. **Zoom and framing:** we will learn how to adjust our view to work accurately at different scales.
3. **Layers:** we will discover how to organise and manage our layers to define line types, thicknesses, etc.
4. **Drawing:** we will practice creating lines and circles, which are fundamental for any geometric design.
5. **Selection and Editing Tools:** we will use these tools to define and optimise our drawings
    - **Delete:** this tool allows us to remove selected objects from the drawing. It is essential to correct errors and clean the workspace of unnecessary elements. You cannot have non-useful elements in the drawings you deliver.
    - **Trim:** with this function, you can cut parts of the objects that exceed a defined limit.
    - **Extend:** allows you to stretch objects to the specified limits.
    - **Offset:** this function creates a parallel copy at a specified distance from the original object. It is a useful tool for the parts we will work with.
    - **Fillet:** with this tool, you can join two objects with a defined radius arc.
6. **Geometric transformations**
    - **Rotate:** allows you to rotate objects around a specified base point.
    - **Move:** this function moves the selected objects to a new location within the drawing space.
    - **Copy:** allows you to duplicate objects and place them in a new position. Multiple copies can be made.

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7. **Drawing aids**
    - **Ortho mode:** this mode restricts cursor movement to right angles (0, 90, 180, 270 degrees), facilitating the creation of precise orthogonal drawings. It is a tool to be used so that they are perfectly aligned when representing different views.
    - **References (CTRL + Right-click):** these references allow you to adjust the cursor to specific points, such as endpoints, tangents, perpendiculars, midpoints, centres, etc.
8. **PDF printing:** we will see how to prepare and print our projects professionally.

To reinforce these concepts, we will perform some practical exercises that will include:

- Line and angle figures to practice precision.
- Figures with tangents, offsets, auxiliary lines, etc.

This session will provide a solid foundation for future AutoCAD© exercises.

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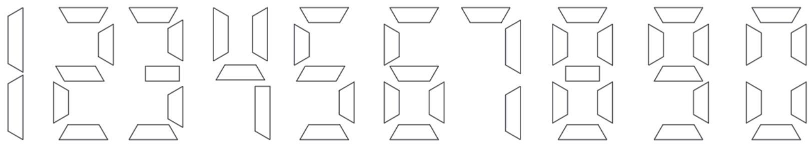
EXERCISE 1

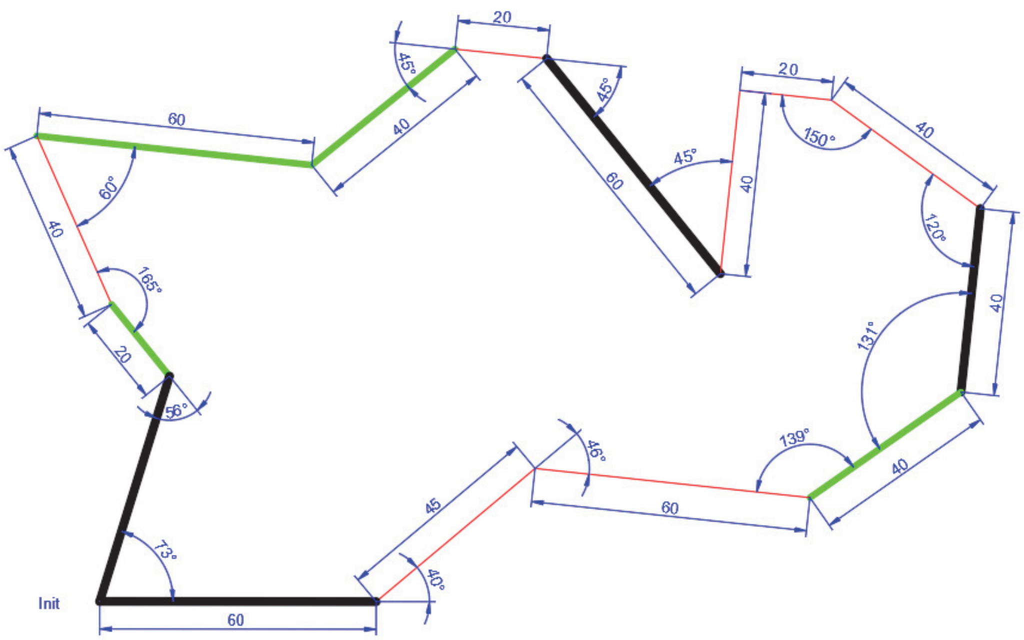
Draw the following graphic representation: the first three digits of your ID or Passport.



EXERCISE 2

Create the following layers and render the following image considering the thickness of the lines.

NAME	LINE TYPE	THICKNESS	COLOR
BASIC 1	CONTINUOUS	0,50	GREEN
BASIC 2	CONTINUOUS	0,70	AUTOMATIC (BLACK/WHITE)
BASIC 3	CONTINUOUS	0,18	RED



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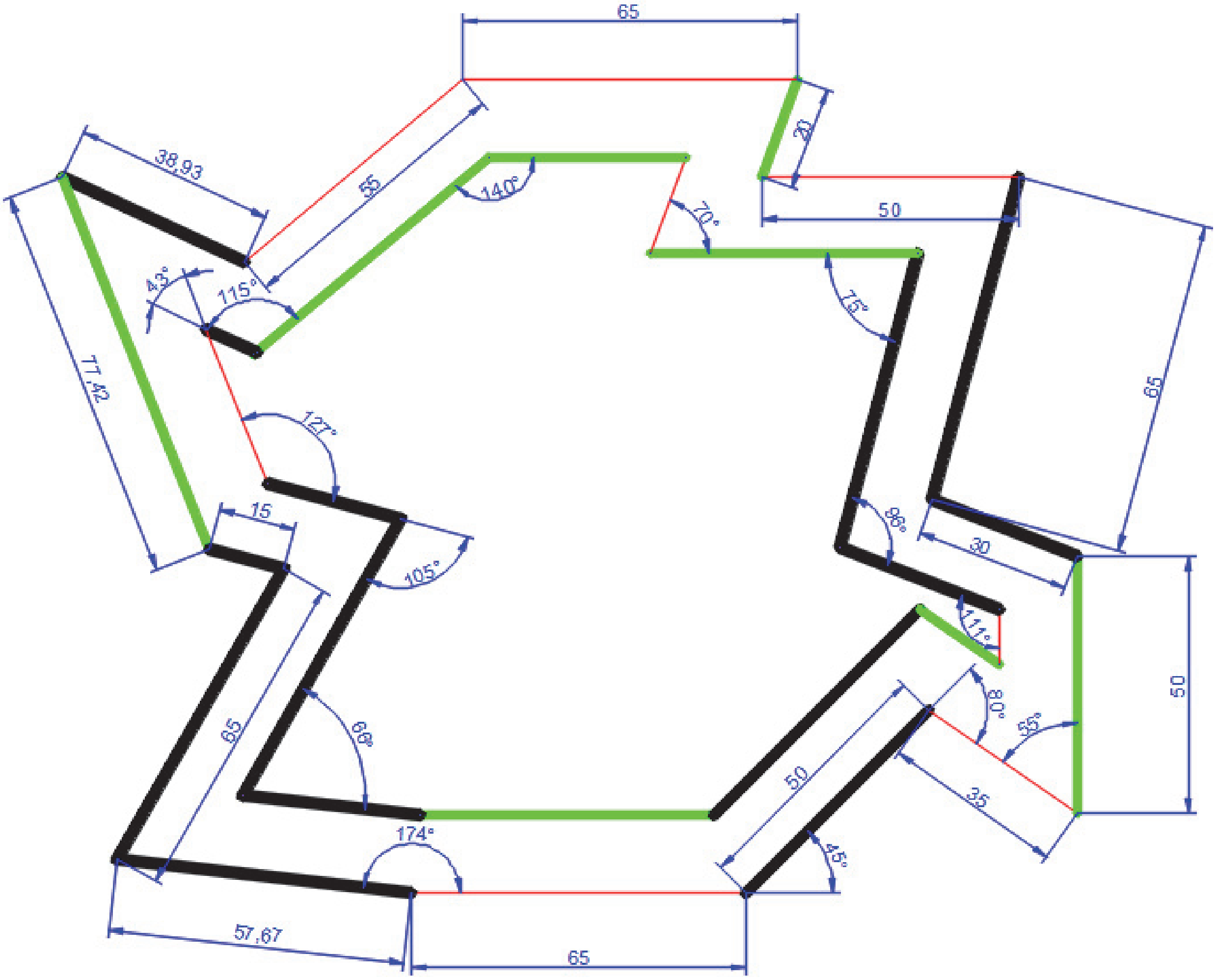
EXERCISE 3

Draw the geometric shape represented in the figure, considering the layers defined below. The inner figure is out of phase with the outer one by **15 units**.

NAME	LINE TYPE	THICKNESS	COLOR
BASIC 1	CONTINUOUS	0,50	GREEN
BASIC 2	CONTINUOUS	0,70	AUTOMATIC (BLACK/WHITE)
BASIC 3	CONTINUOUS	0,18	RED



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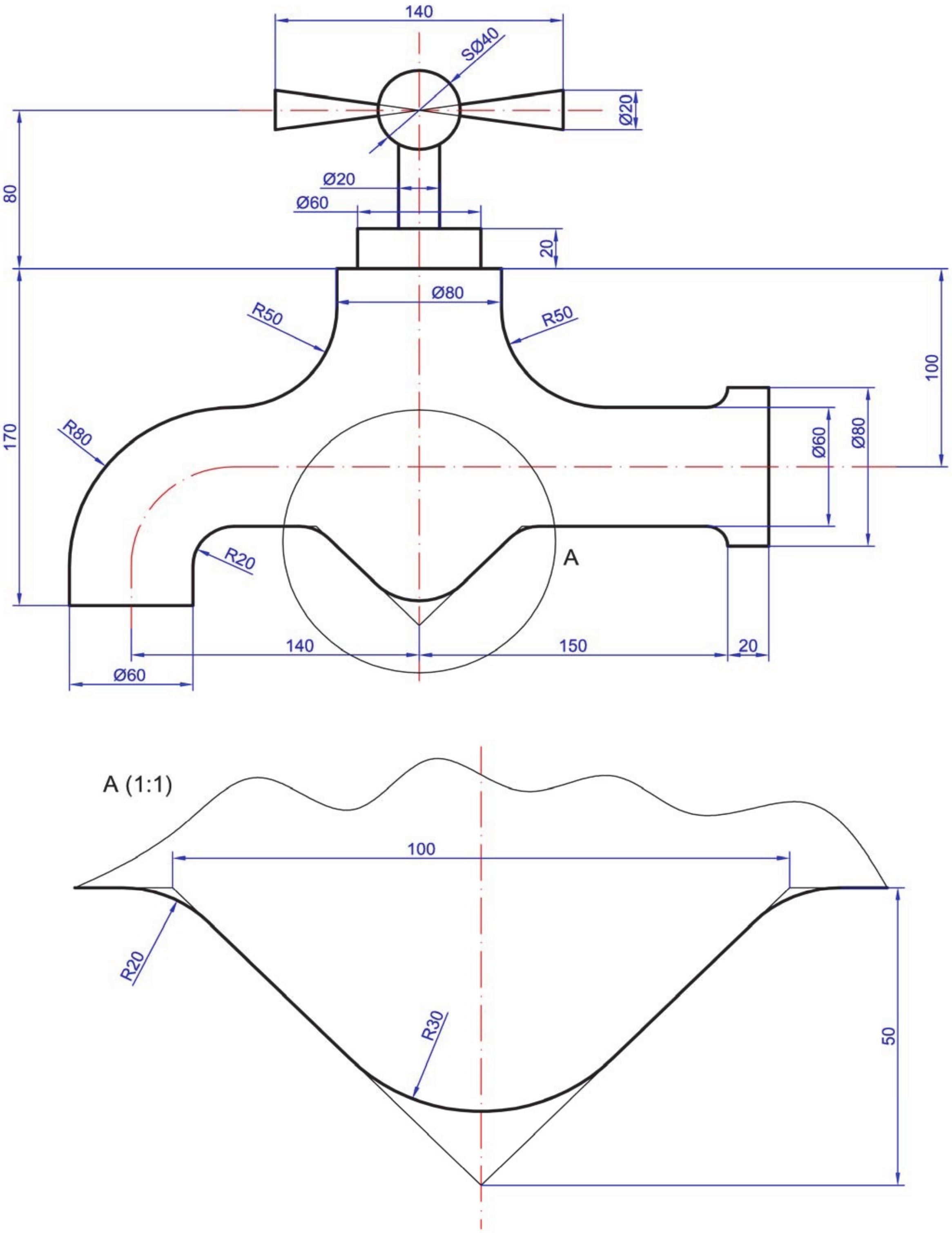
EXERCISE 4

Draw the geometric shape represented in the figure without delimiting it. Prepare this figure and try to print it in PDF. In the following session, printing will be done directly using this exercise.

The layers you will have to use for the graphic representation are.

NAME	LINE TYPE	THICKNESS	COLOR	USE
AXES	CONTINUOUS	0,18	RED	Axes, circumf. of construction, plan traces
DIMENSIONS	CONTINUOUS	0,18	BLUE	Dimensioning elements
VIEWS	CONTINUOUS	0,50	BLACK	Edges and contours: thickening of cutting planes
AUXILIARY	CONTINUOUS	0,18	MAGENTA	Auxiliary lines and curves
GRAPHIC WINDOW	CONTINUOUS	0,18	MAGENTA	Graphic window



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7. SESSION 5: CAD 2D- Tangencies, Layers and PDF Printing

In this session, we will learn some fundamental aspects of AutoCAD© that will help you to improve your drawing and printer skills. The following are the contents we will study:

1. **Drawing: review of tangencies**

   We will begin with a thorough review of tangencies. Tangencies are essential for creating smooth and precise connections between curves and straight lines. We will learn how to:

   - **Draw tangent** lines to circles and arcs.
   - **Use commands** to create tangents between multiple elements.

2. **Drawing Aids Review**

   We will review two essential tools that facilitate precision in drawing:

   - **Ortho mode**: this mode restricts cursor movement to angles of 0º, 90º, 180º, and 270º, allowing precise vertical and horizontal strokes.
   - **References** (CTRL + Right Button): we will use the references to adjust the cursor to specific points, such as ends, means and centres of objects.

3. **Layers: review**

   In this section, we will focus on layer management, which is crucial for organising drawings and specifying line thicknesses, line types, etc.:

   - **Definition of TLScale and ISO dashed lines:** we will learn how to define and apply line-type scales and understand the use of dashed lines in ISO standards.
   - **Importance of correct axis definition:** we will see why it is vital to define axes correctly.
   - **Layer organisation:** we will review how to organise elements using layers, assigning them specific properties such as colours, line types and thicknesses.

4. **PDF printing review**

   Finally, we will review the PDF printing process, ensuring that our documents print correctly:

   - **Graphic window:** we will explore the print window settings, selecting the appropriate areas to print and adjusting the parameters for optimal results.

EXERCISE 1

Open the exercise you have prepared at home from the previous session, modify the line type of the “AXES” layer and set the linotype scale (ESCALATL) to 0.3.

NAME	LINE TYPE	THICKNESS	COLOR	USE
AXES	ISO 10W100	0,18	RED	Axes, circumf. of construction, plan traces
DIMENSIONS	CONTINUOUS	0,18	BLUE	Dimensioning elements
VIEWS	CONTINUOUS	0,50	BLACK	Edges and contours: thickening of cutting planes
AUXILIARY	CONTINUOUS	0,18	MAGENTA	Auxiliary lines and curves
GRAPHIC WINDOW	CONTINUOUS	0,18	MAGENTA	Graphic window

With PDF printing, you must generate the graphic representation, including the updated date in the box, your name, and the scale of representation.

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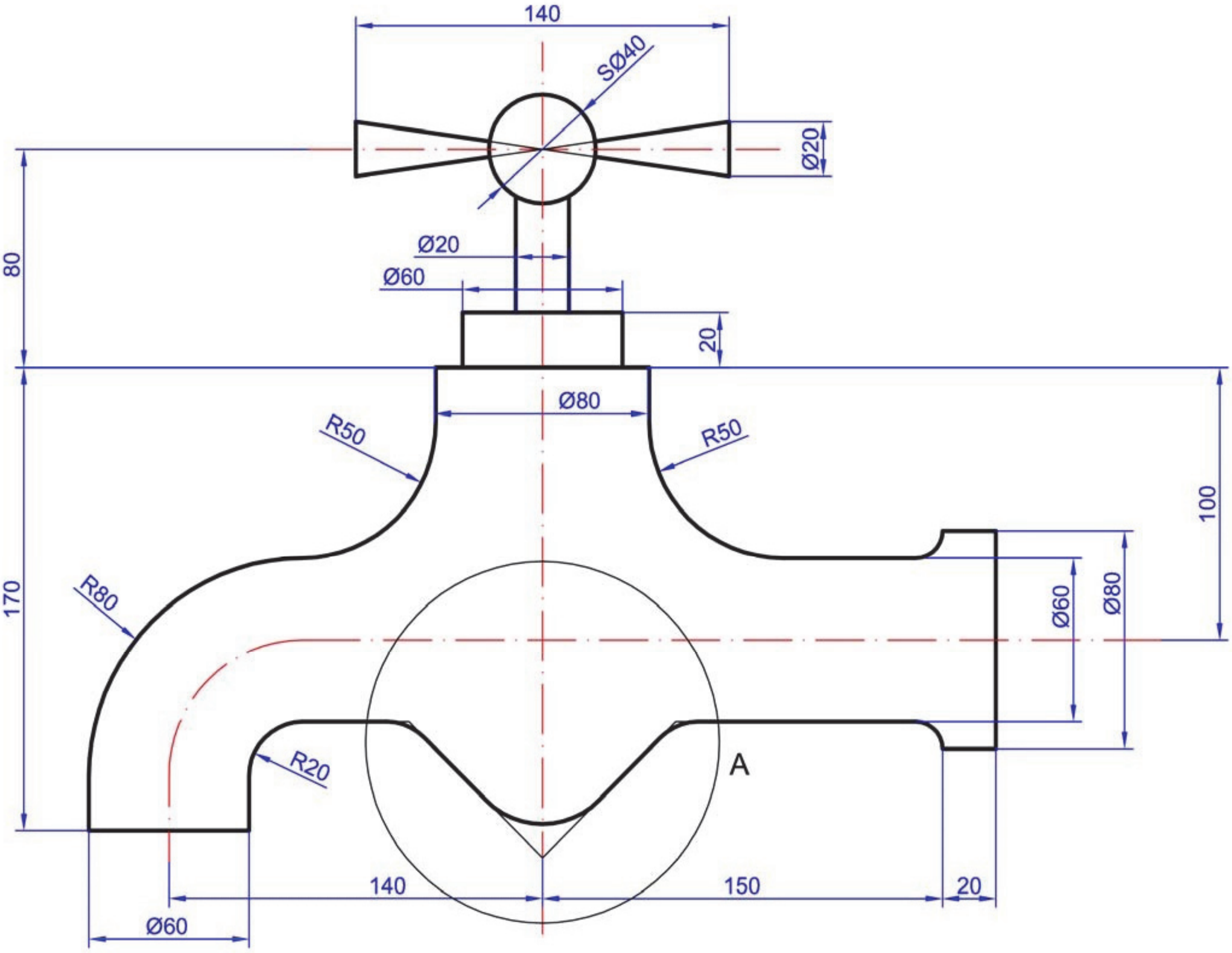

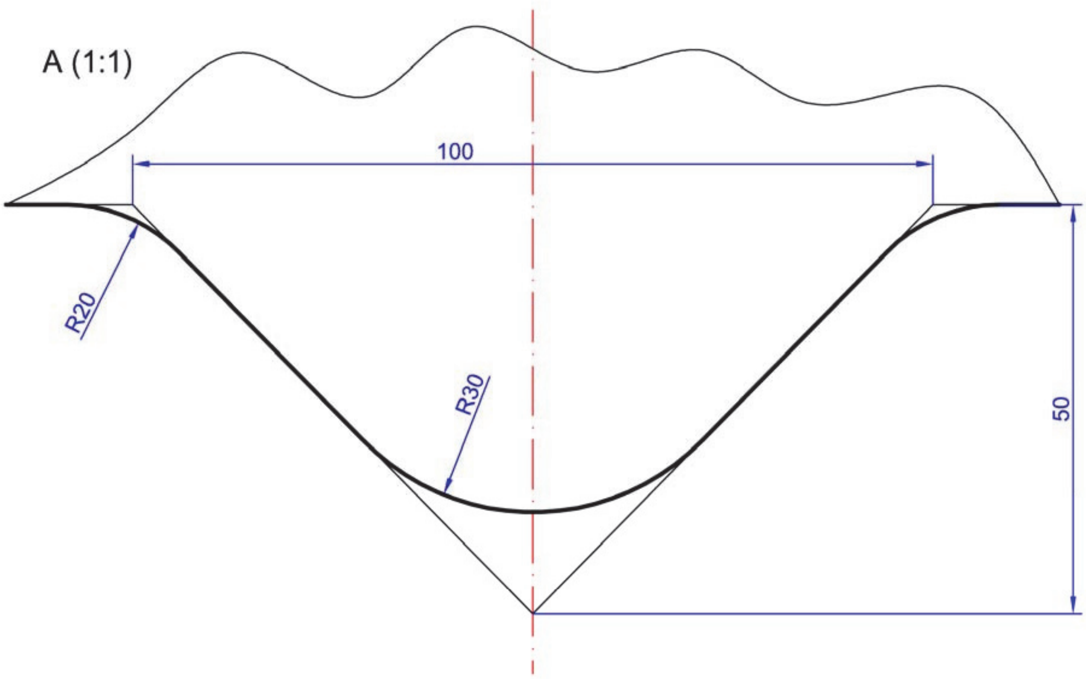

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EXERCISE 2

Draw the geometric shape represented in the figure using appropriately each layer in the different lines.

NAME	LINE TYPE	THICKNESS	COLOR	USE
AXES	ISO 10W100	0,18	RED	Axes, circumf. of construction, plan traces
DIMENSIONS	CONTINUOUS	0,18	BLUE	
VIEWS	CONTINUOUS	0,50	BLACK	Edges and contours: thickening of cutting planes
GRAPHIC WINDOW	CONTINUOUS	0,18	MAGENTA	Auxiliary lines and curves. Graphic window

Line type scale 0.3

With PDF printing, you must generate the graphic representation, including the updated date in the box, your name, and the scale of representation.

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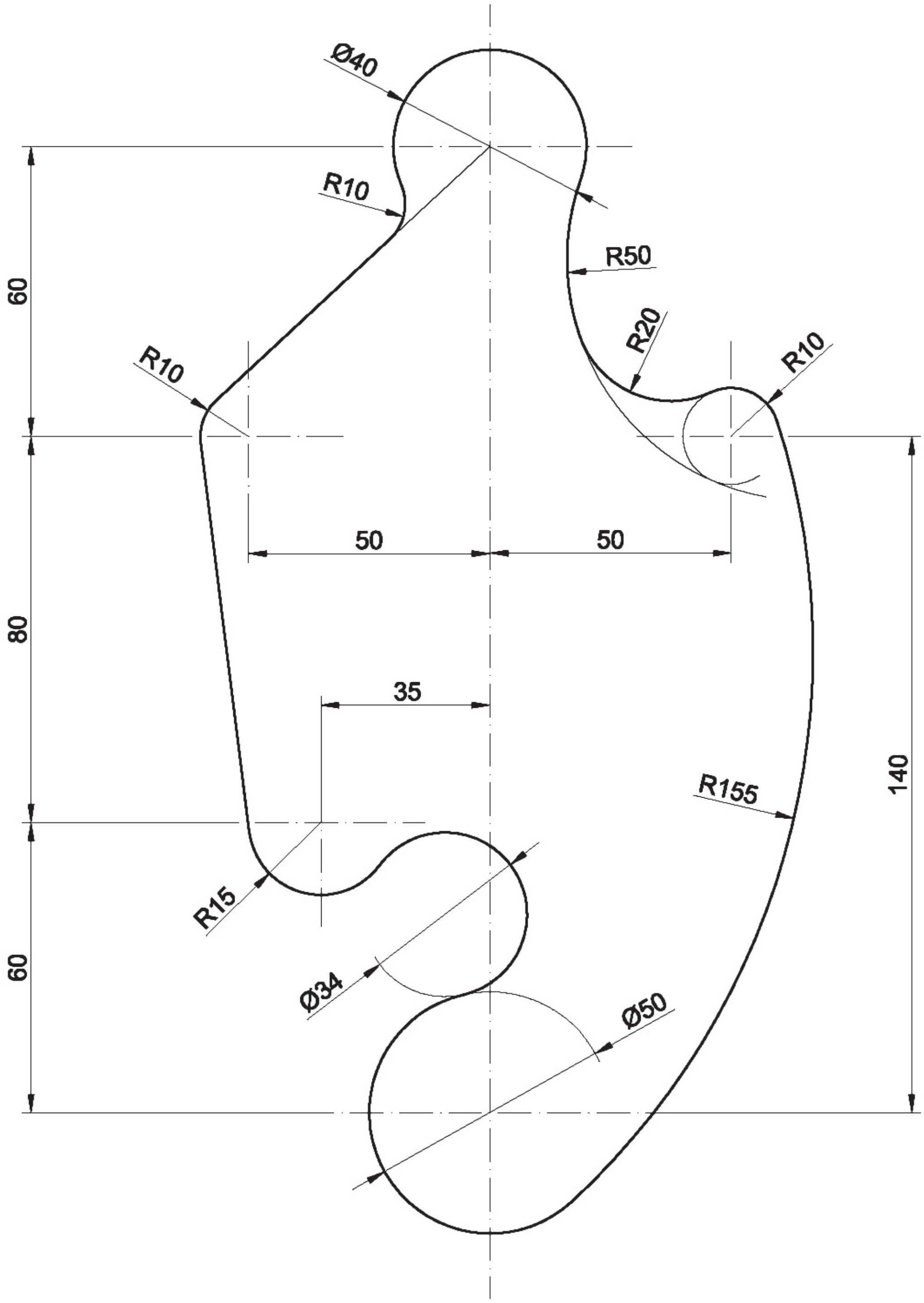

Ø40
R10
R50
60
R20
R10
R10
50
50
80
35
140
R155
R15
60
Ø34
Ø50

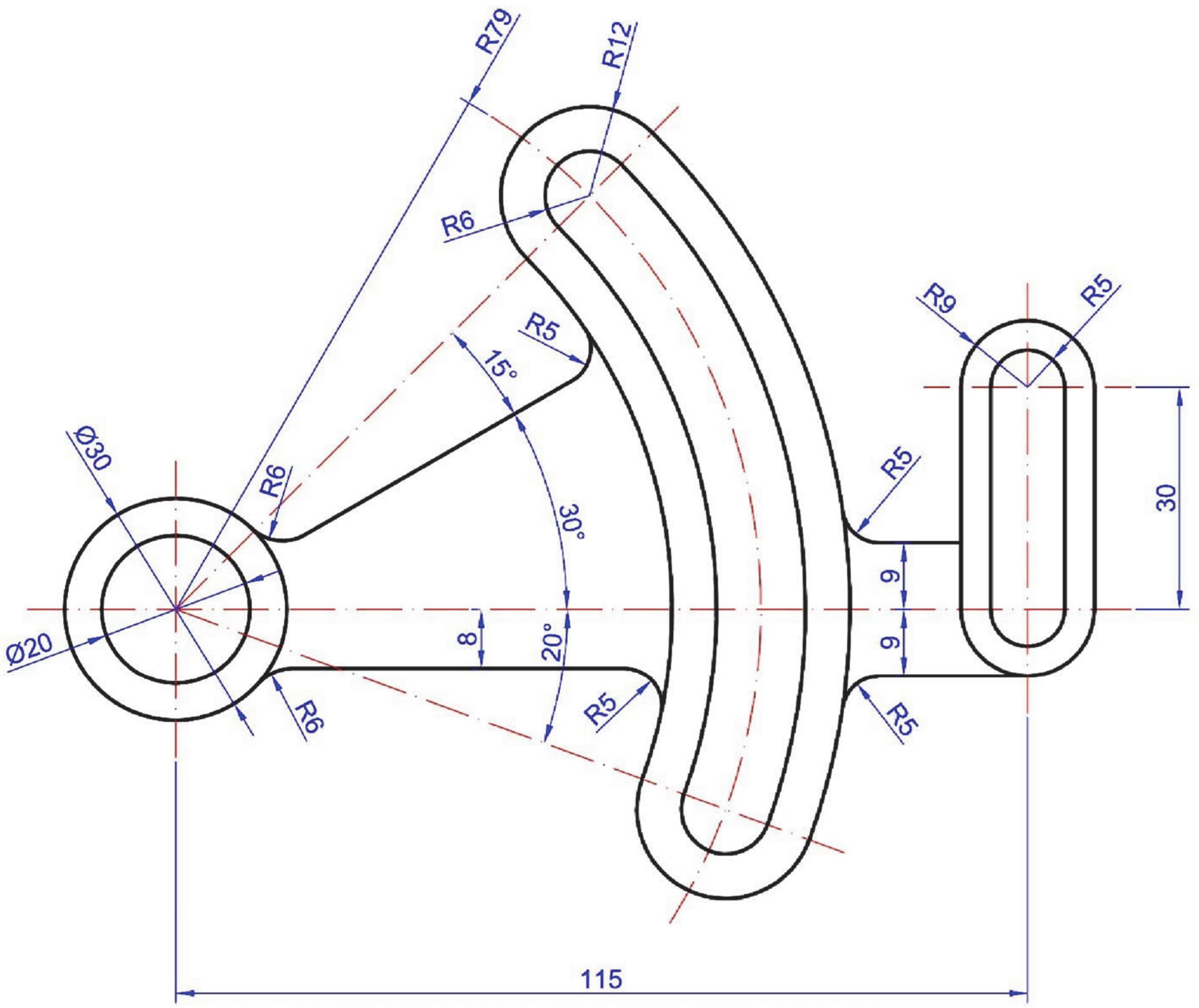
EXERCISE 3

Draw the geometric shape represented in the figure using appropriately each layer in the different lines:

NAME	LINE TYPE	THICKNESS	COLOR	USE
AXES	ISO 10W100	0,18	RED	Axes, circumf. of construction plan traces
DIMENSIONS	CONTINUOUS	0,18	BLUE	
VIEWS	CONTINUOUS	0,50	BLACK	Edges and contours
GRAPHIC WINDOW	CONTINUOUS	0,18	MAGENTA	Auxiliary lines and curves. Graphic window

Line type scale 0.3

With PDF printing, you must generate the graphic representation, including the updated date in the box, your name, and the scale of representation.



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8. SESSION 6: CAD 2D- Arrays, Splines and Symmetry

This session explores advanced AutoCAD© techniques, ranging from spline curves to geometric transformations and matrices.

During this session, we will address the following topics:

1. **Drawing: Spline Curve**
    - **Creating Spline curves**
    - **Hatching areas** with Spline
2. **References (activation button)**
    - Using **references** to improve drawing accuracy
    - **Activation** of buttons and tools for shortcuts
3. **Geometric transformations**
    - **Symmetry:** apply symmetry to duplicate and mirror objects
    - **Scaling:** adjusting the size of objects while maintaining their proportions or modifying them as necessary
4. **Matrix**
    - **Rectangular**: create arrays of objects in rows and columns
    - **Polar:** arrange objects around a midway point in a circular shape
5. **Editing**
    - **Grip**: use grips to modify selected objects quickly
    - **Decompose:** break composite objects into their components for detailed editing

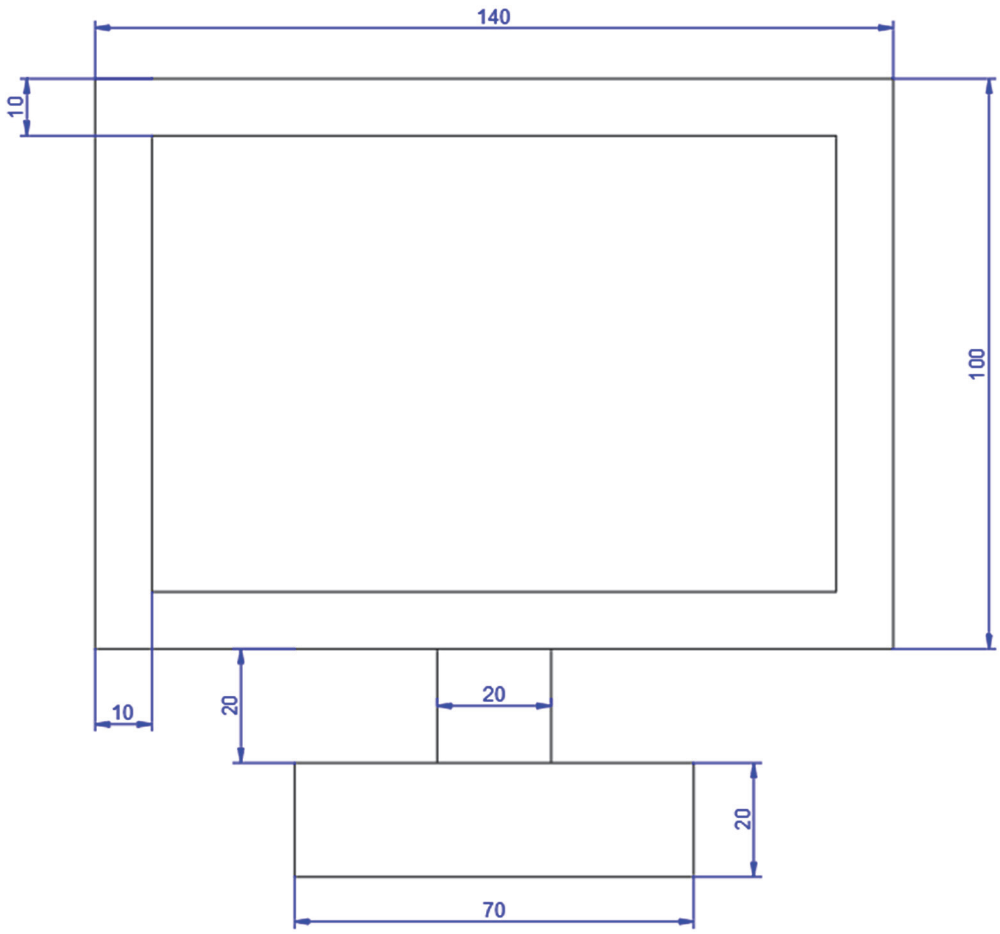
EXERCISE 1

Draw the geometric shape "monitor" represented in the figure. Create a second copy of the monitor with the "copy" command and move it away with the "Move" command. In addition, you must make use of the following layers:

NAME	LINE TYPE	THICKNESS	COLOR	USE
AXES	ISO 10W100	0,18	RED	Axes, circumf. of construction plan traces
DIMENSIONS	CONTINUOUS	0,18	BLUE	
VIEWS	CONTINUOUS	0,50	BLACK	Edges and contours
GRAPHIC WINDOW	CONTINUOUS	0,18	MAGENTA	Auxiliary lines and curves. Graphic window

Line type scale 0.3

With PDF printing, you must generate the graphic representation, including the updated date in the box, your name, and the scale of representation.



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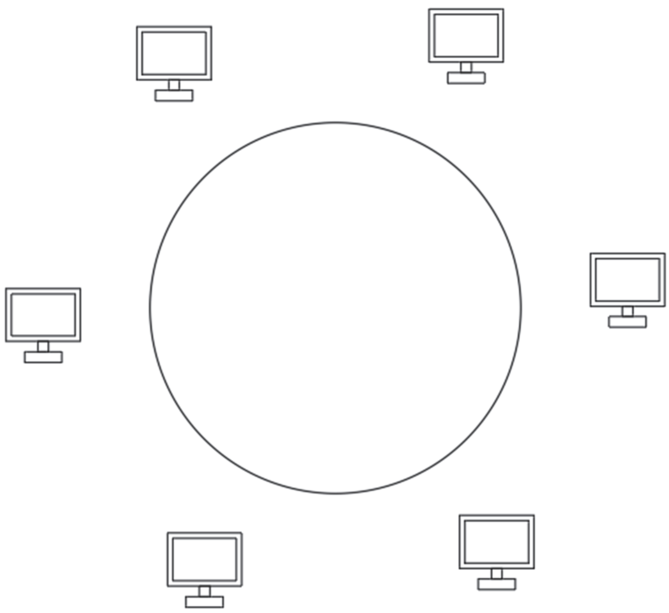
Step 1. Create a rectangular matrix of five columns and four rows from the “monitor” element you created.

Step 2. Using the second monitor, create a circle and a polar matrix around the centre of the circle of the “monitor” element you made at the beginning.

Remember the viewing angle of the objects that you will have to modify so they can be displayed as follows.



Step 3. Use the decompose tool (Explode) to see how a single "monitor" from the previous step can be removed.

Step 4. Make a Spline that joins the centres of the different monitors.

Step 5. Make an ANSI 31 shading inside two of the monitor screens.

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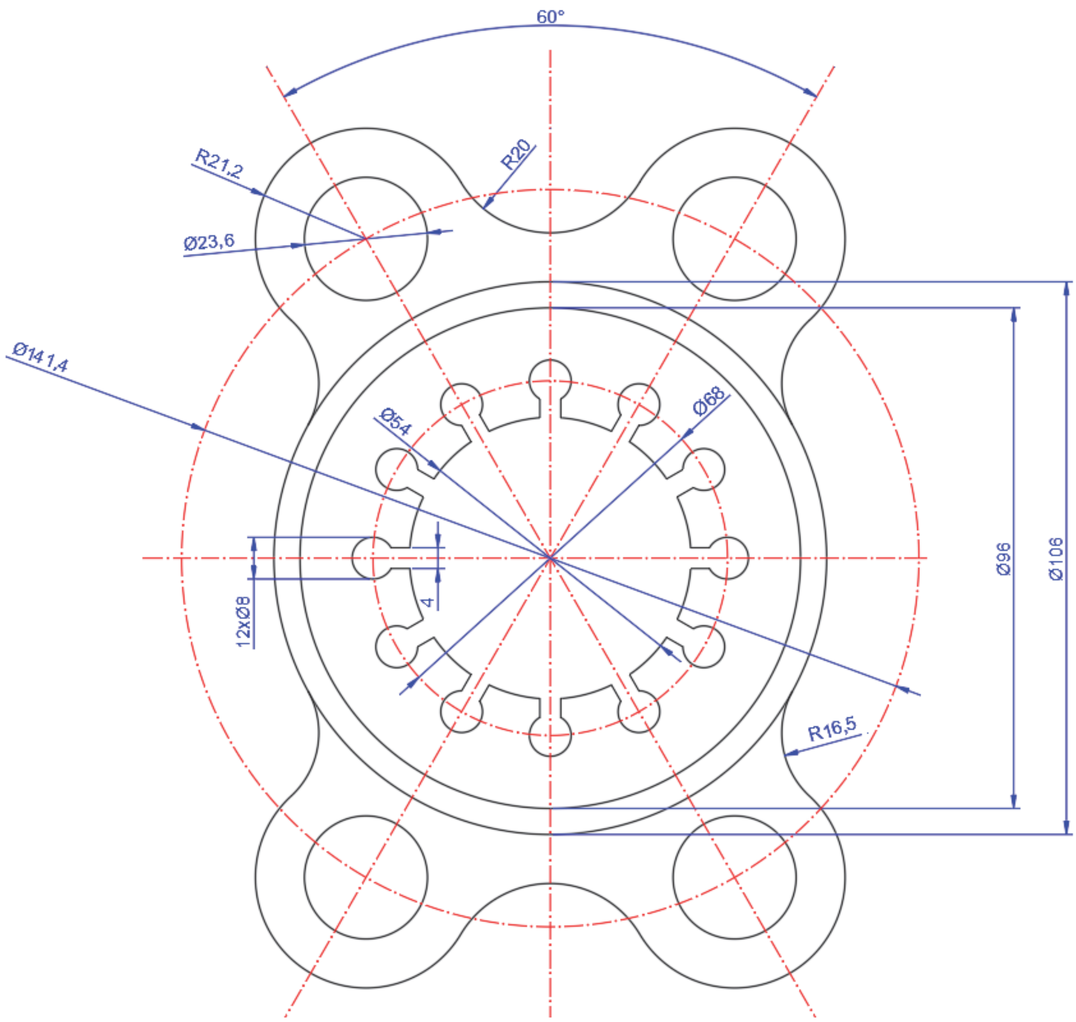
EXERCISE 2

Draw the geometric shape represented in the figure using appropriately each layer in the different lines:

NAME	LINE TYPE	THICKNESS	COLOR	USE
AXES	ISO 10W100	0,18	RED	Axes, circumf. of construction plan traces
DIMENSIONS	CONTINUOUS	0,18	BLUE	
VIEWS	CONTINUOUS	0,50	BLACK	Edges and contours
GRAPHIC WINDOW	CONTINUOUS	0,18	MAGENTA	Auxiliary lines and curves. Graphic window

Line type scale 0.3

With PDF printing, you must generate the graphic representation, including the updated date in the box, your name, and the scale of representation.



EXERCISE 3

Draw the geometric shape represented in the figure using the matrix and symmetry commands correctly. In addition, you must make use of the following layers:

NAME	LINE TYPE	THICKNESS	COLOR	USE
AXES	ISO 10W100	0,18	RED	Axes, circumf. of construction plan traces
DIMENSIONS	CONTINUOUS	0,18	BLUE	
VIEWS	CONTINUOUS	0,50	BLACK	Edges and contours
GRAPHIC WINDOW	CONTINUOUS	0,18	MAGENTA	Auxiliary lines and curves. Graphic window

Line type scale 0.3

With PDF printing, you must generate the graphic representation, including the updated date in the box, your name, and the scale of representation.

The central element that organises this shape is a half-regular polygon.

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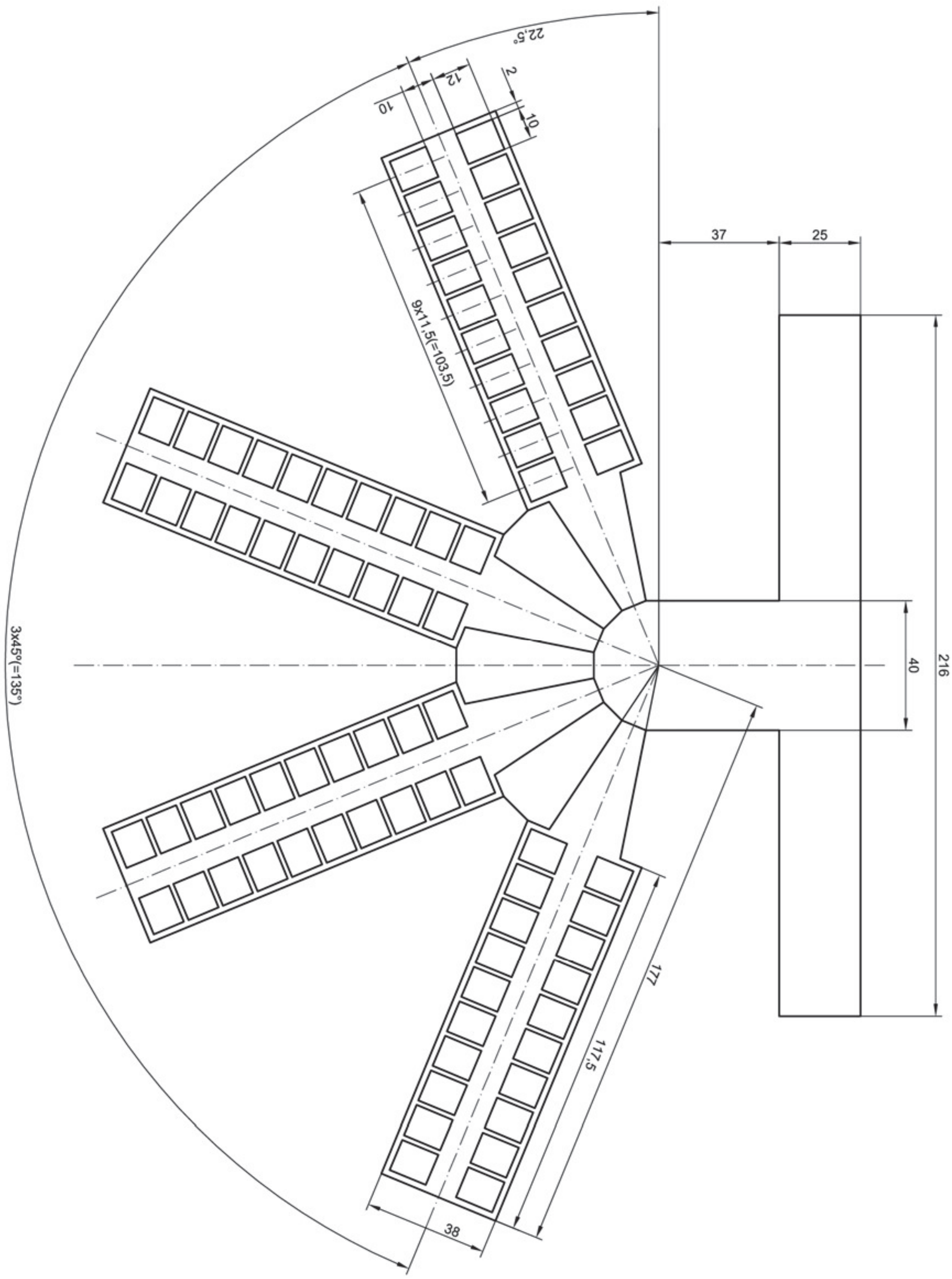
22,5°
2
12
10
10
37
25
9x11,5(=103,5)
216
40
3x45°(=135°)
177
117,5
38

EXERCISE 4

Draw the geometric shape represented in the figure using the matrix and symmetry commands correctly. In addition, you must make use of the following layers:

NAME	LINE TYPE	THICKNESS	COLOR	USE
AXES	ISO 10W100	0,18	RED	Axes, circumf. of construction plan traces
DIMENSIONS	CONTINUOUS	0,18	BLUE	
VIEWS	CONTINUOUS	0,50	BLACK	Edges and contours
GRAPHIC WINDOW	CONTINUOUS	0,18	MAGENTA	Auxiliary lines and curves. Graphic window

Line type scale 0.3

With PDF printing, you must generate the graphic representation, including the updated date in the box, your name, and the scale of representation.

CLARIFICATIONS

- Non-dimensioned rounding radius are 2.5 mm
- Non-dimensioned hole diameters are 5 mm

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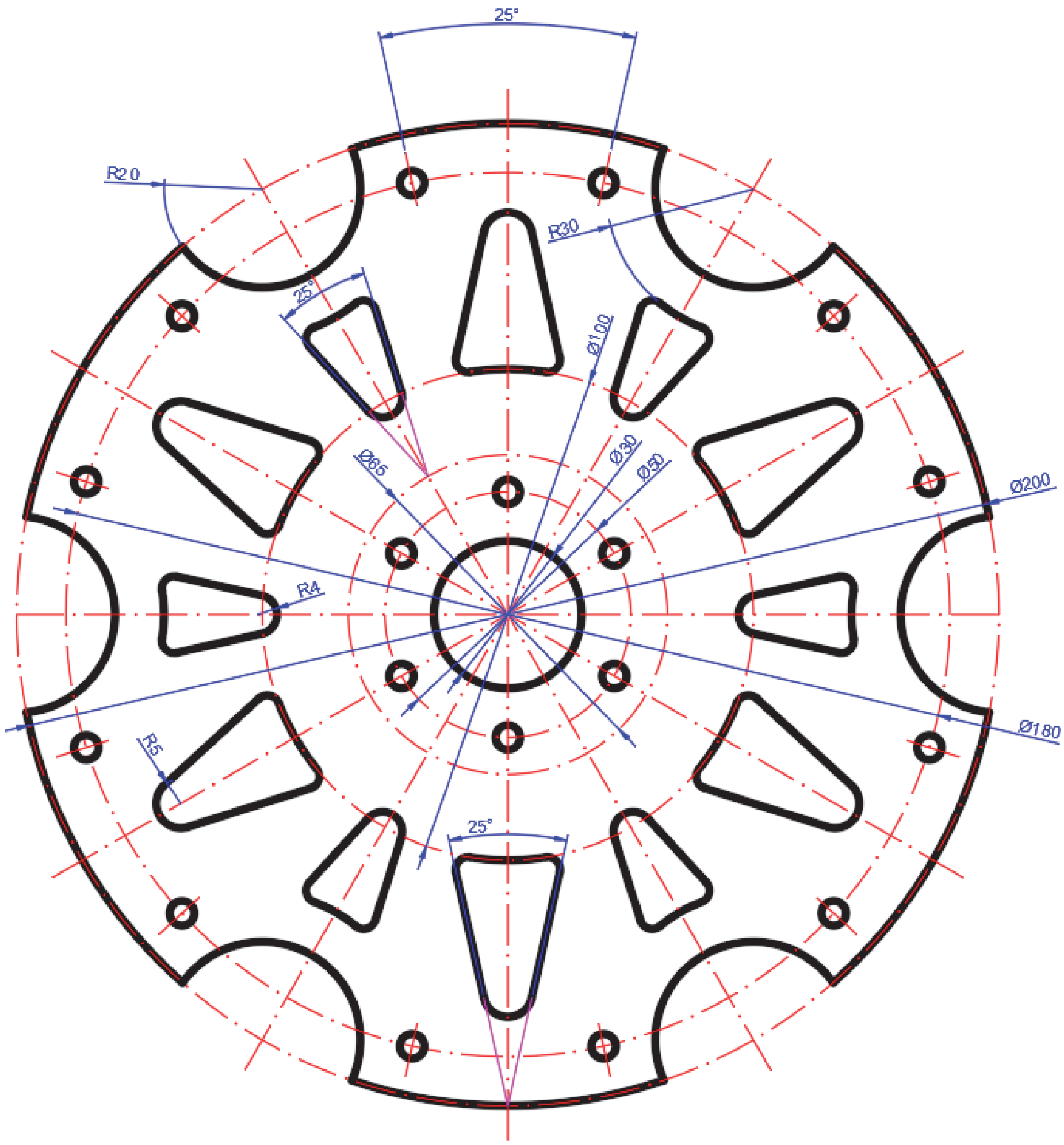
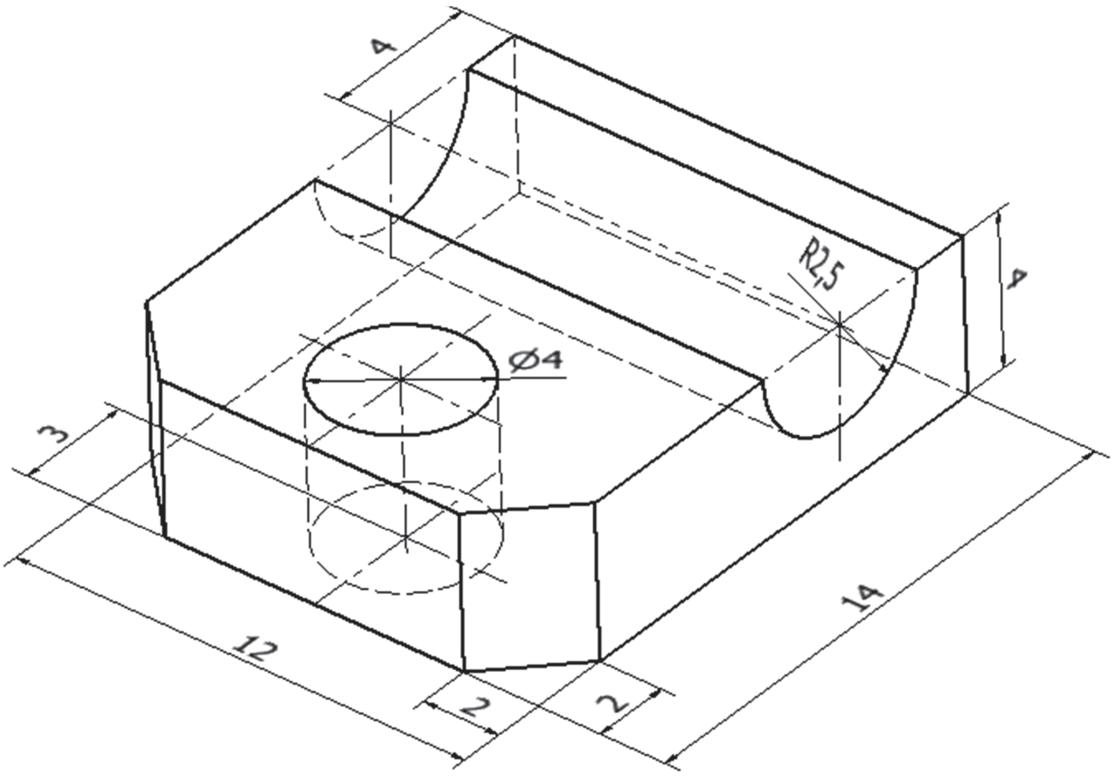

25°
R20
R30
25°
Ø100
Ø65
Ø30
Ø50
Ø200
R4
R5
Ø180
25°

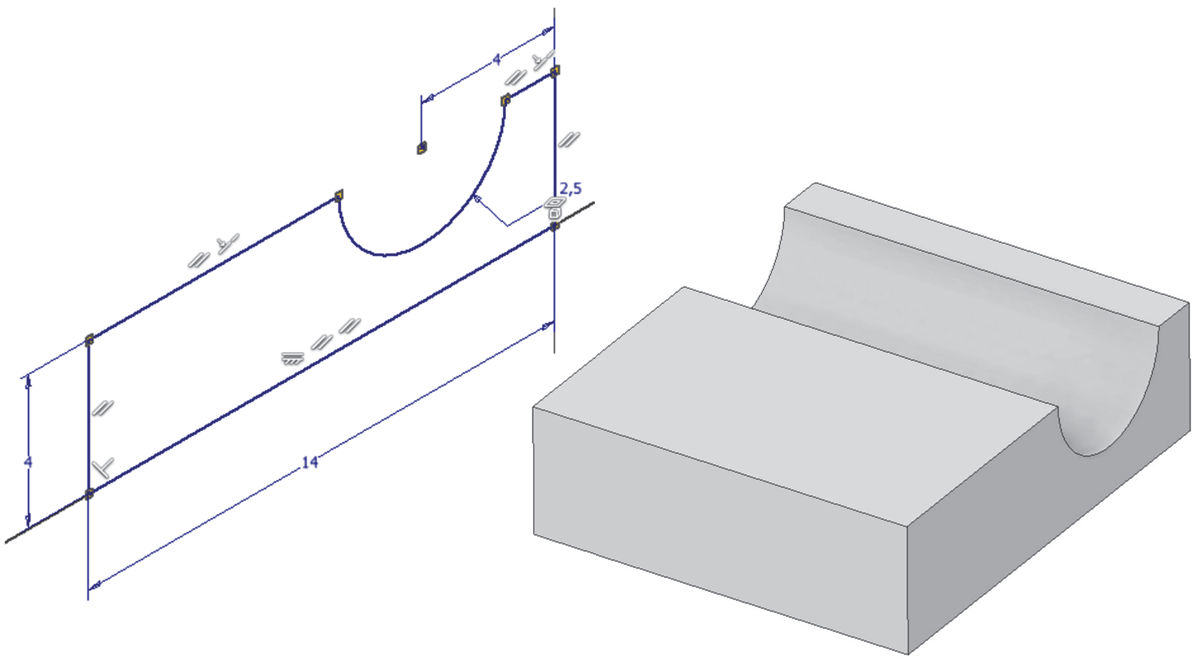
9. SESSION 7: CAD 3D –Review exercises

In this session, we will review the exercises we have learnt in previous sessions in Autodesk Inventor©.

EXERCISE 1



Step 1. Symmetrical extrusion. Sketch placed on the frontal plane.



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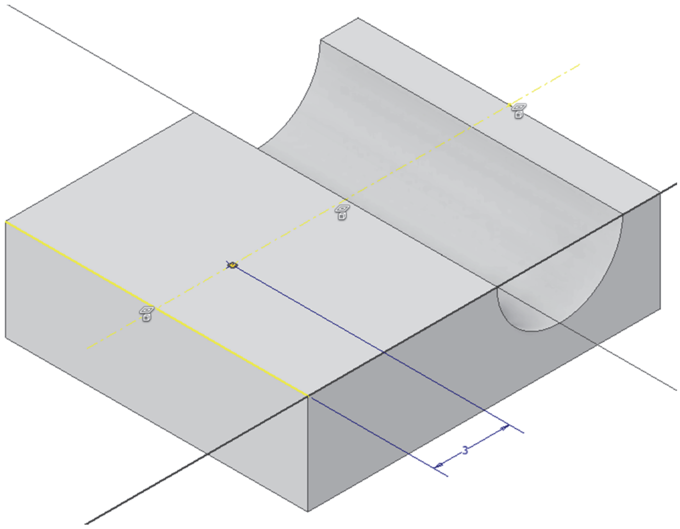
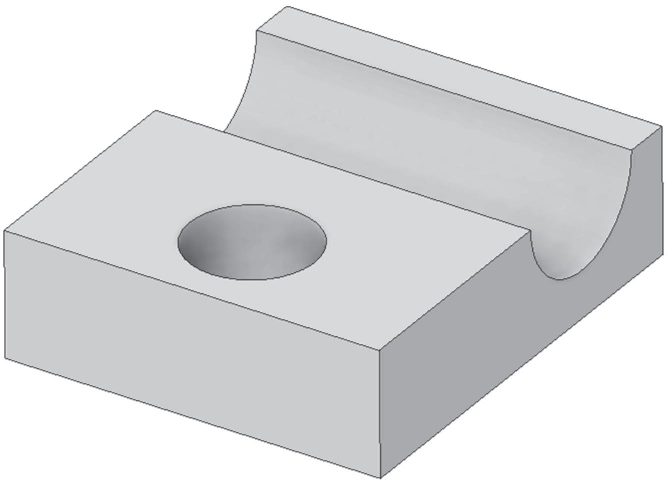
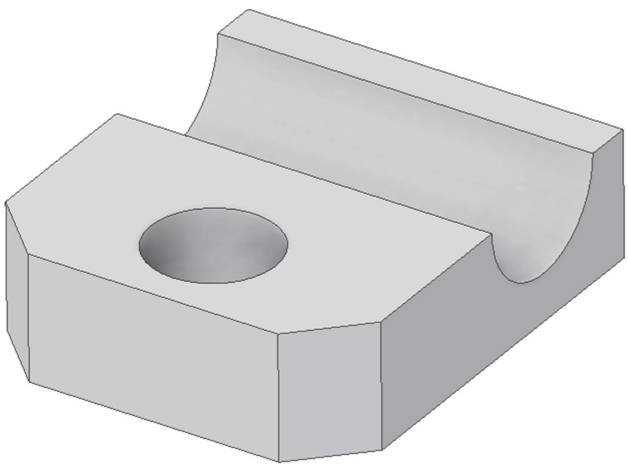
Step 2. Through hole. Sketch with a point located on the symmetry axis of the upper face of the base.





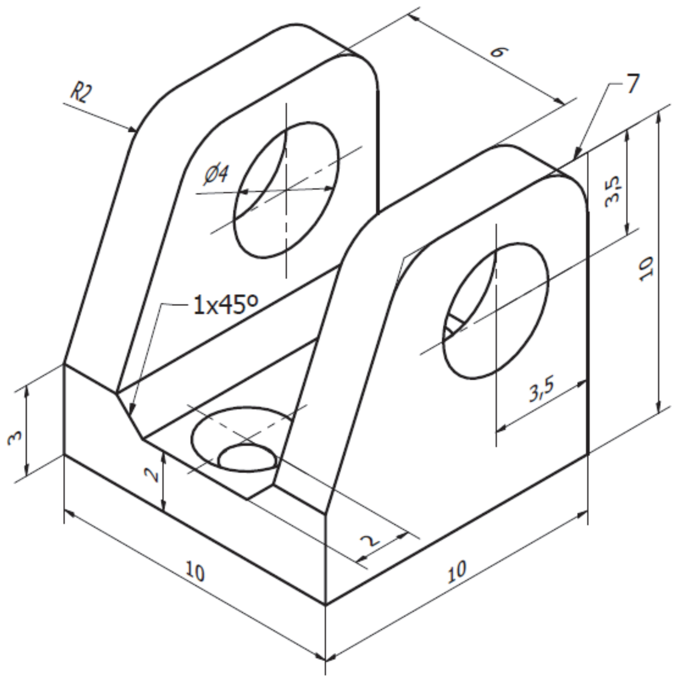
Step 3. Chamfers. No sketch is required.



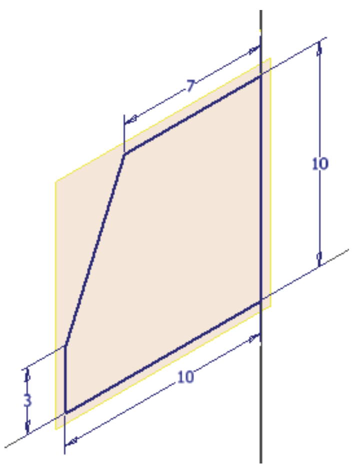
EXERCISE 2

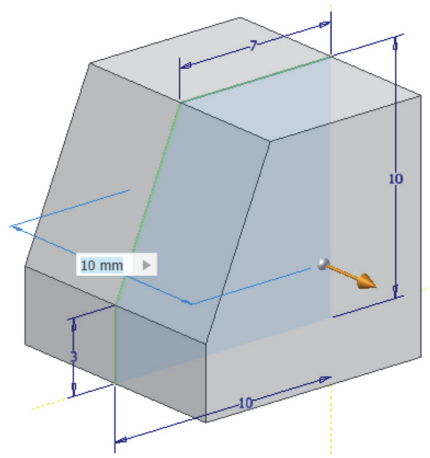
Model the following part so that it meets the next, using the corresponding geometric constraints:

1) Supported in the horizontal plane.
2) Centred in a vertical plane.
3) The side walls must maintain the exact dimensions and be centred so that when the dimensioning is modified, the result retains the symmetry of the part (mirror).
4) With straight-through holes in the vertical walls and centred holes with countersunk and tapped holes in the base.
5) With roundings and chamfers where indicated.



Step 1. Create a sketch in a vertical plane and draw the sketch, which will then be extruded on both sides by 10 mm. The final part should look as shown above.





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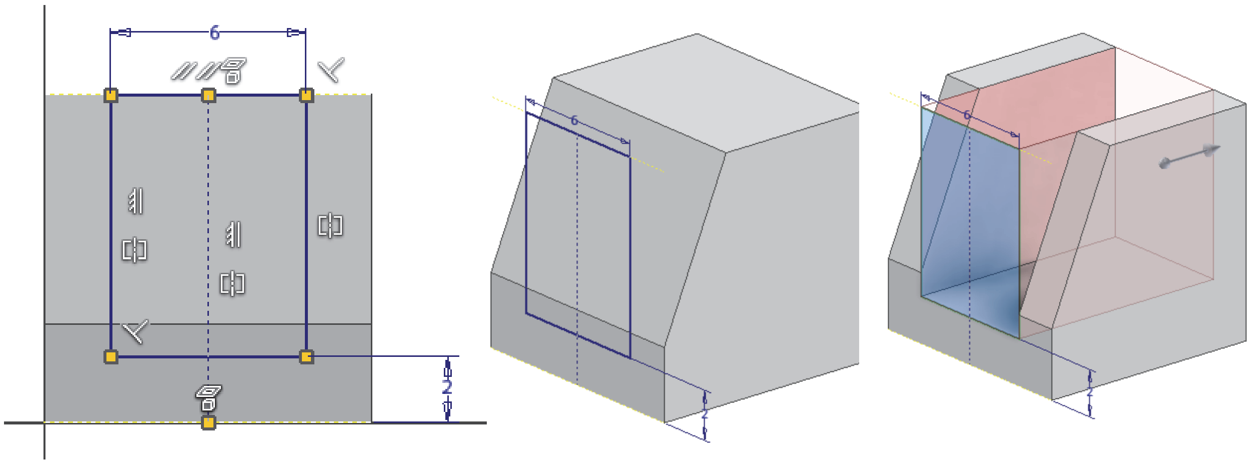
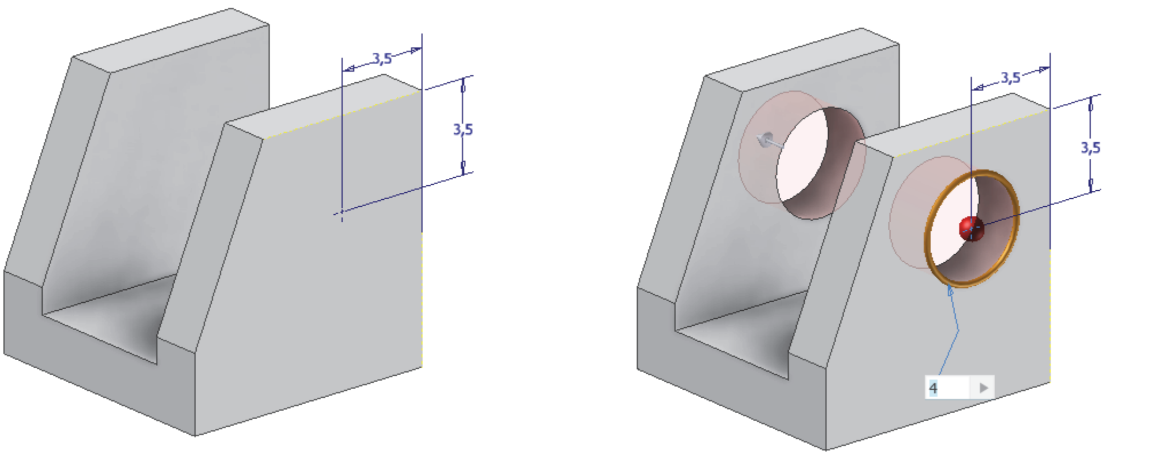
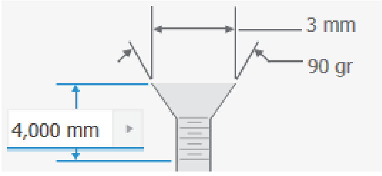
Step 2. Create a sketch on the front or rear face and draw the sketch that will be extruded through. Make sure the symmetry is maintained when modifying the 6 mm dimension.

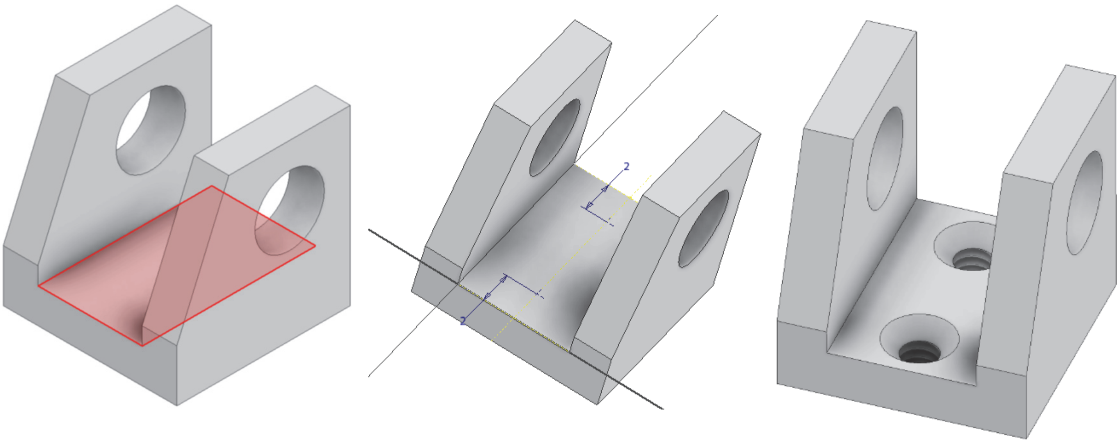


Step 3. Create a sketch on the side face of any wall and place a point 3.5 mm from the references (edges) indicated. Then make a straight through HOLE.

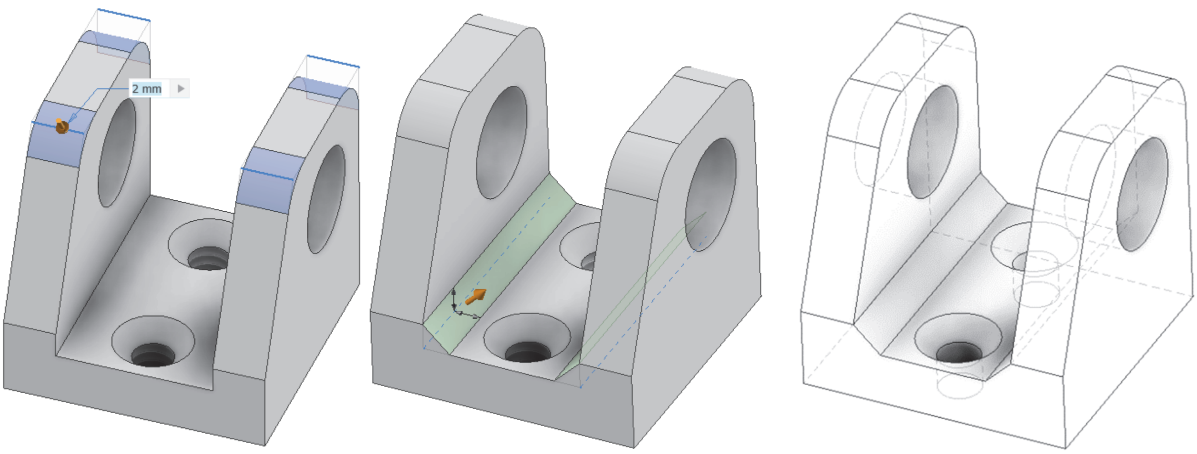


Step 4. Create a sketch on the upper base with two points located on the middle plane at 2 mm from the ends of the part. Then, make a HOLE with a conical countersunk seat and ISO M2 metric thread.





Step 5. Create the roundings (R=2 mm) and chamfers (1 mm x 45º) on the indicated edges. Observe the part using different display modes.



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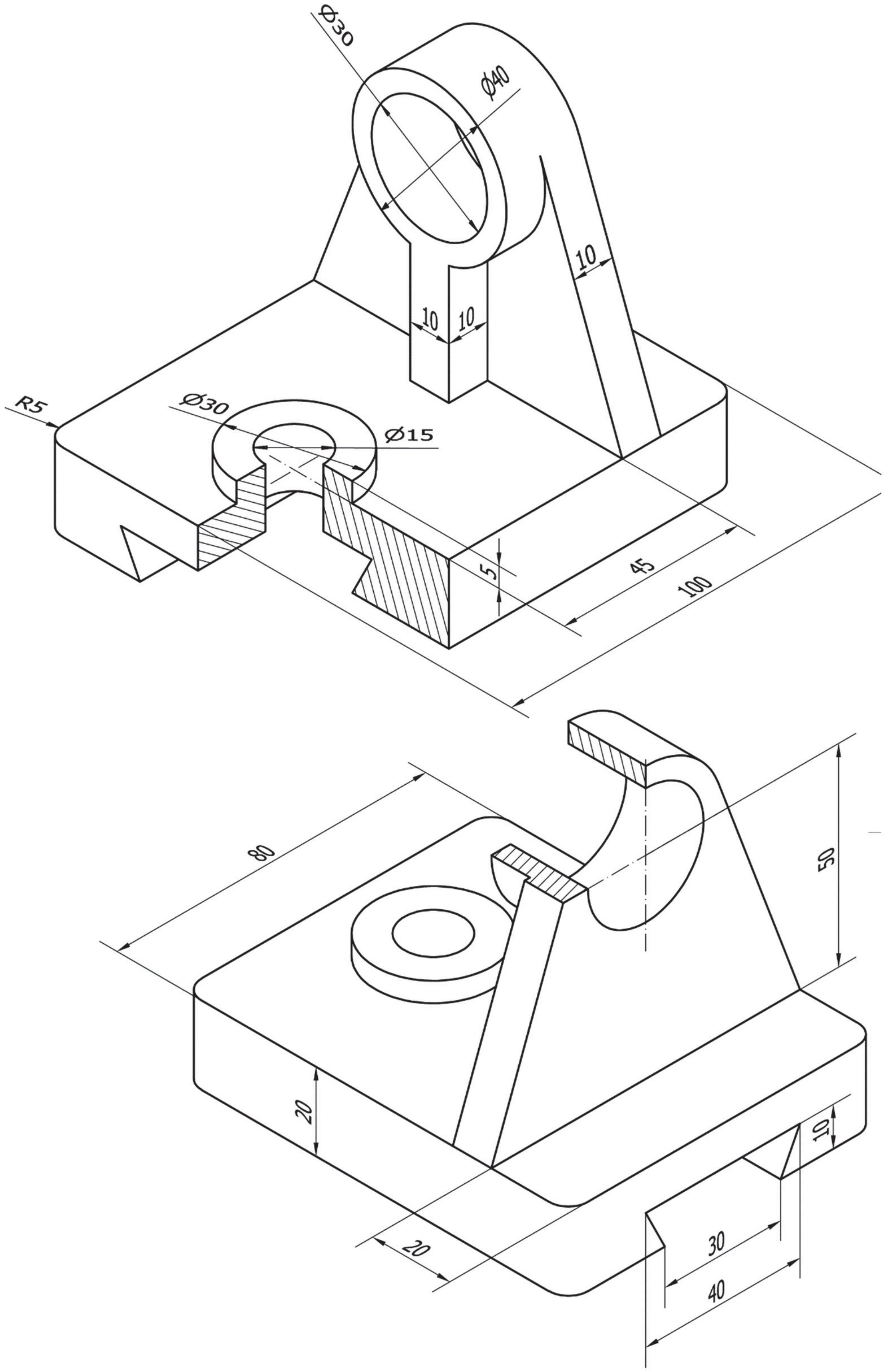
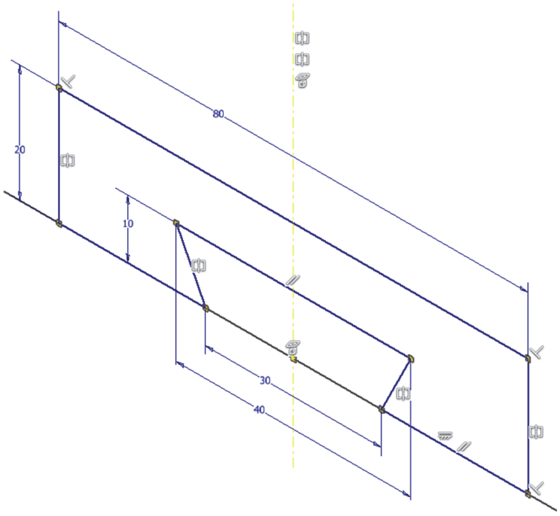
EXERCISE 3

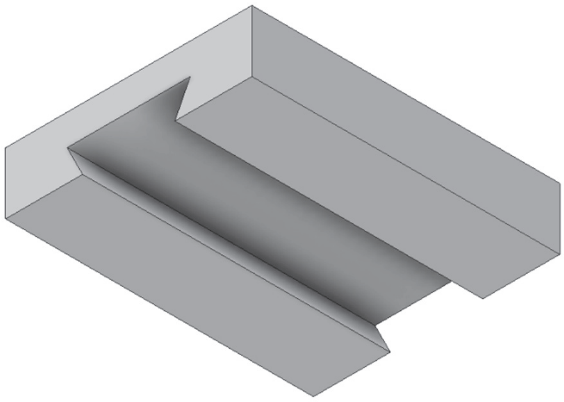
Model the following part.



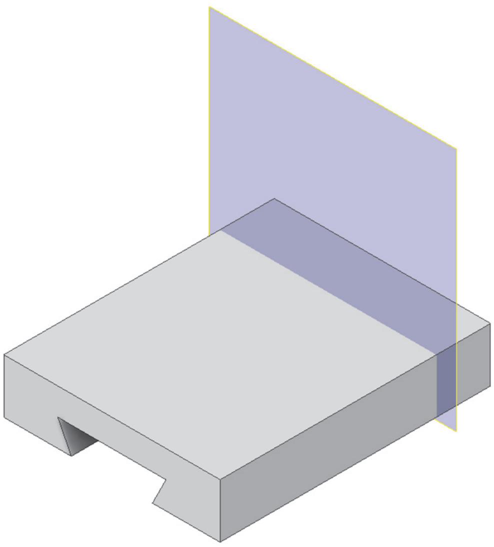
Note: the Ø15 mm hole, modify it to be a M15 thread.

Step 1. Extrusion. Sketch placed on the profile plane. Symmetrical sketch.

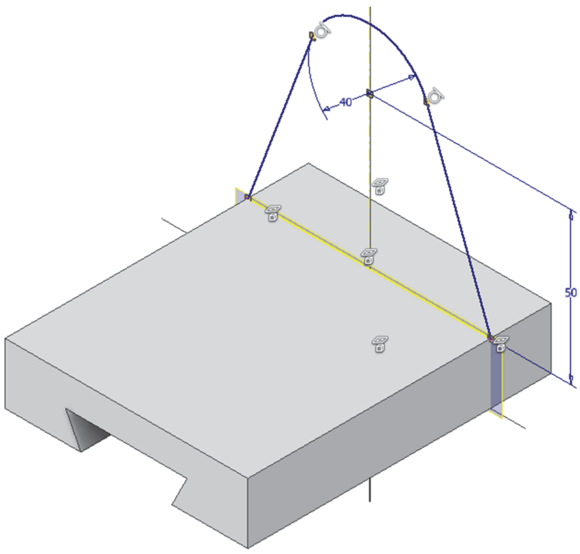


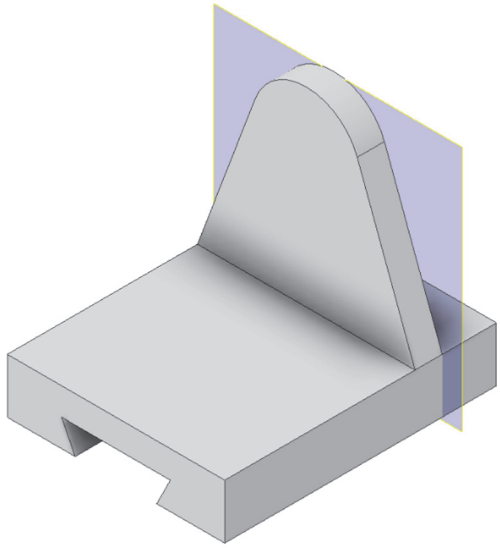


Step 2. Auxiliary plane with offset to right rear lateral face. No sketch is required.

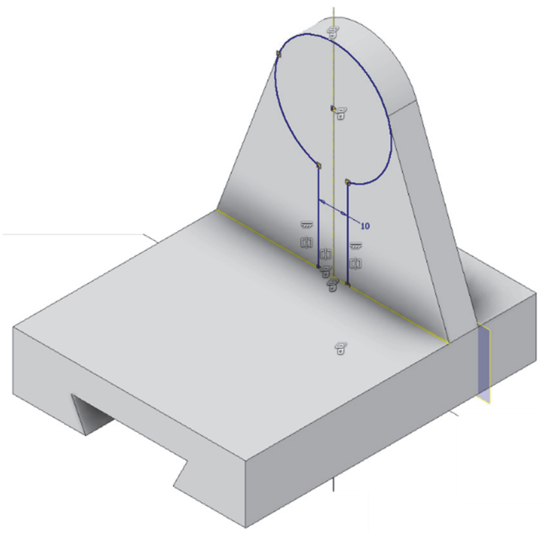


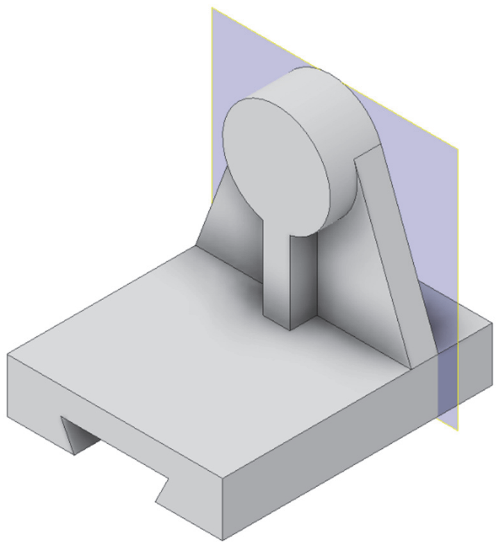
Step 3. Extrusion. Sketch placed on the auxiliary plane. Symmetrical sketch.



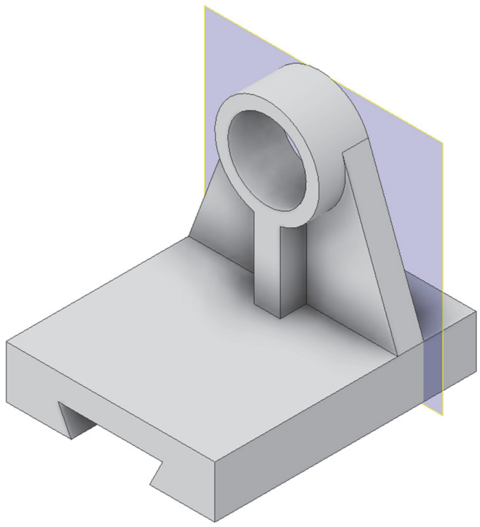


Step 4. Extrusion. Create a sketch placed on the front face of the last part. Symmetrical sketch.

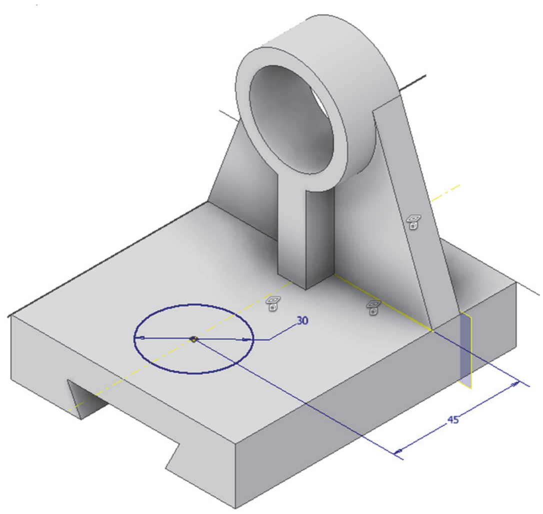


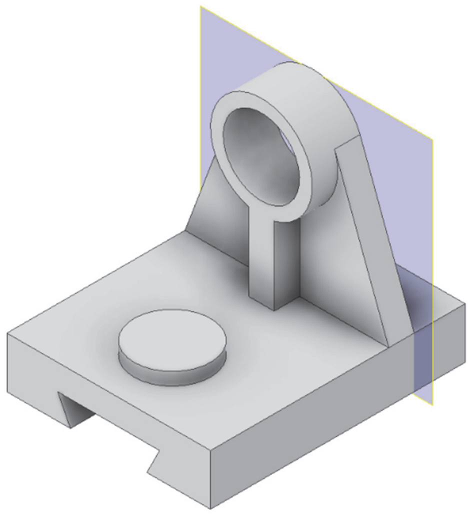


Step 5. Generate a concentric through hole. No sketch is required.



Step 6. Extrusion. Sketch on the upper side of the base. Generate a symmetrical sketch.





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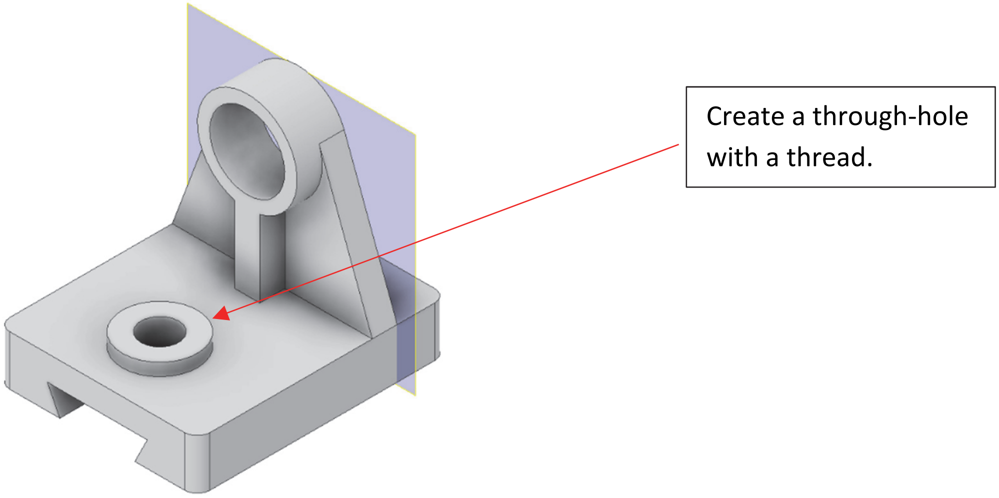
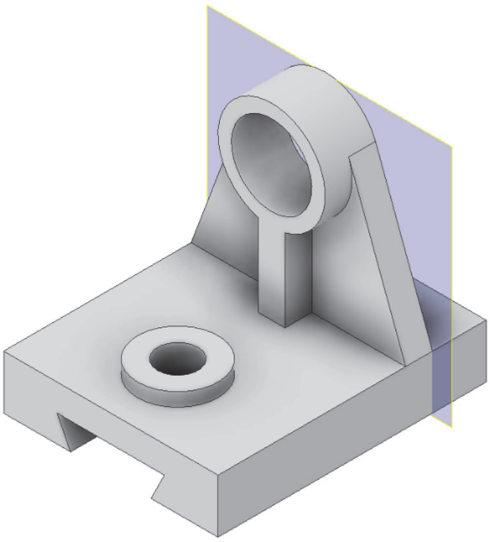
Step 7. Create a concentric through hole. No sketch is required.



Step 8. Rounding. No sketch is required.



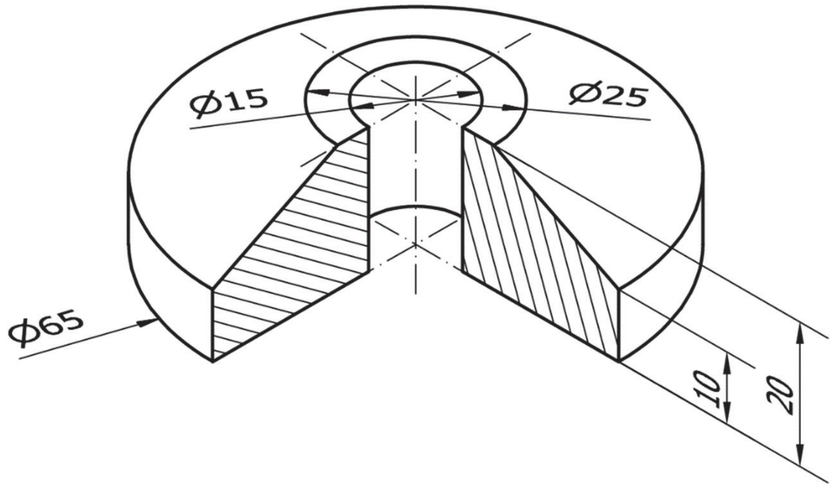
10. SESSION 8: CAD 3D - Revolutions and Advanced Sketch Dimensioning

In this session, we will look in depth at key concepts such as:

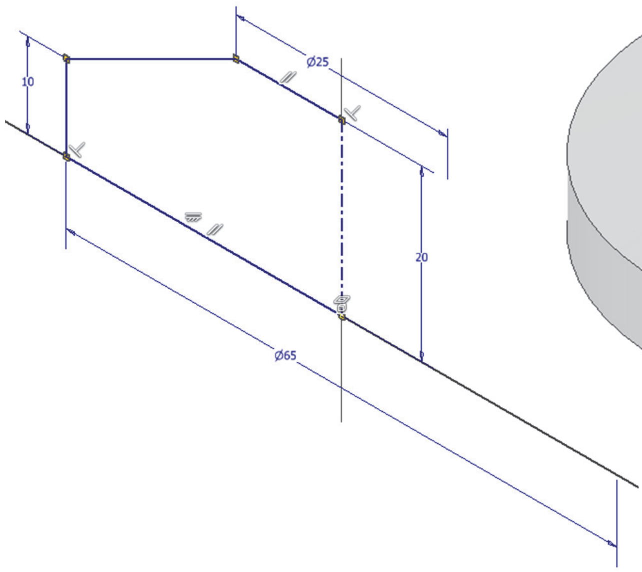
1. **Advanced dimensioning**: we will see more types of dimensioning to provide parts with correct dimensions.
2. **Revolve operations**: we will learn how to create revolve operations, a necessary technique for modelling objects in 3D. By revolutionising sketch profiles around an axis, we can generate complex shapes.

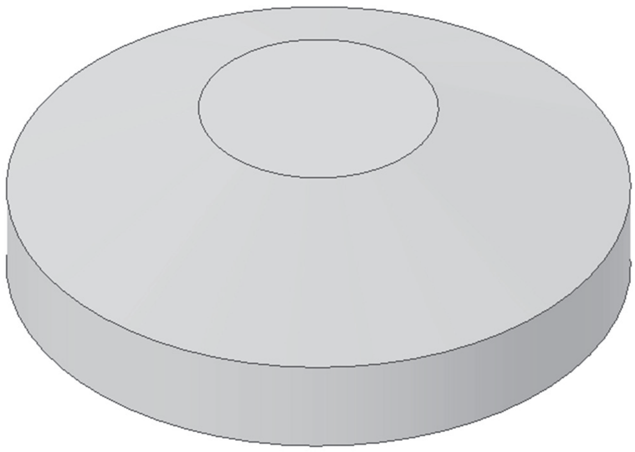
EXERCISE 1

Make the following part using the revolution command.



Step 1. Revolution. Sketch located in frontal or profile plane (indistinctly). Sketch with diameter dimensions to the axis of revolution.





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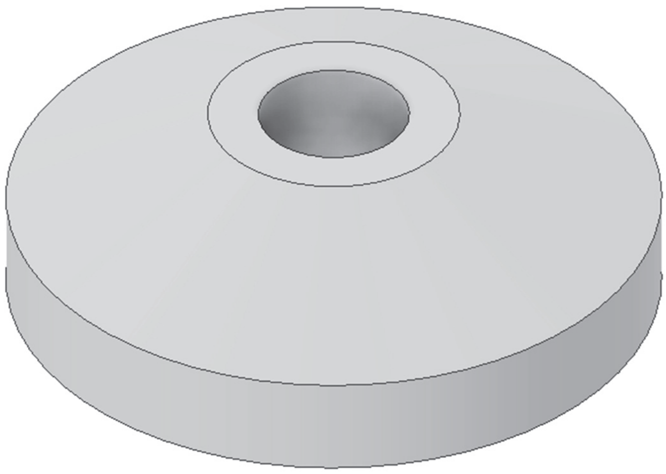
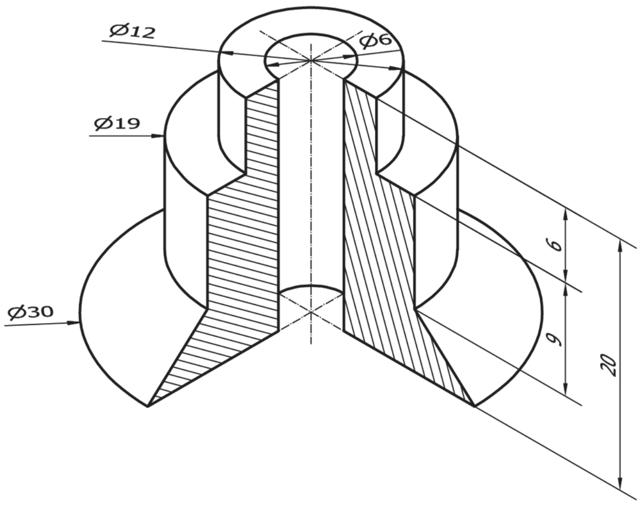
Step 2. Through hole concentric to the circumference. No sketch is required.



EXERCISE 2

Make the following part using an operation of revolution and a hole.



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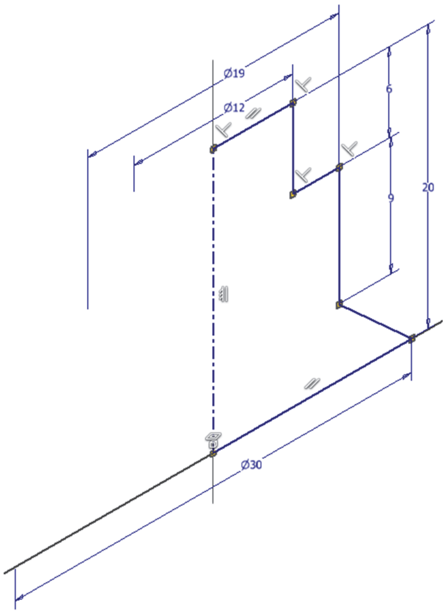
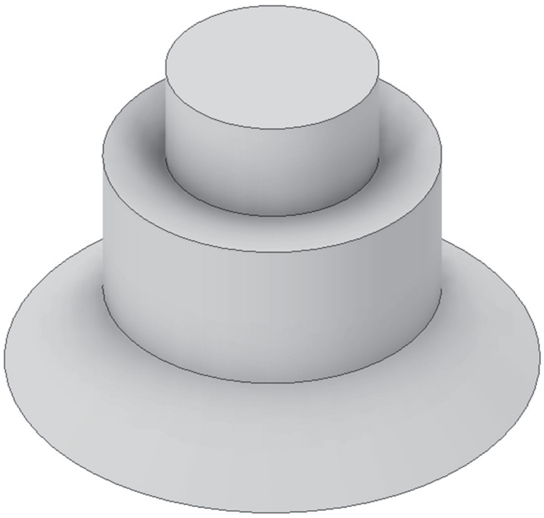
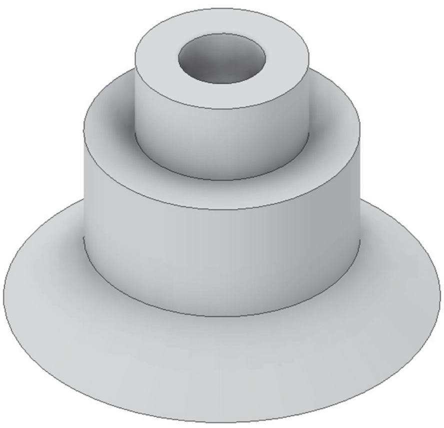
Step 1. Revolution. Sketch located in frontal or profile plane (indistinctly). Sketch with diameter dimensions concerning the axis of revolution.





Step 2. Concentric through-hole. No sketch is required.



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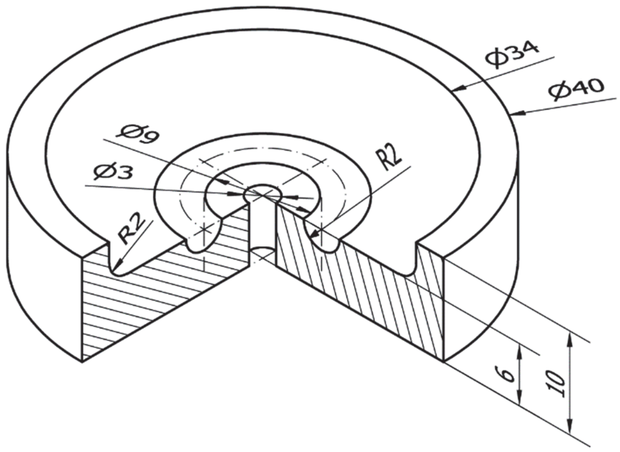
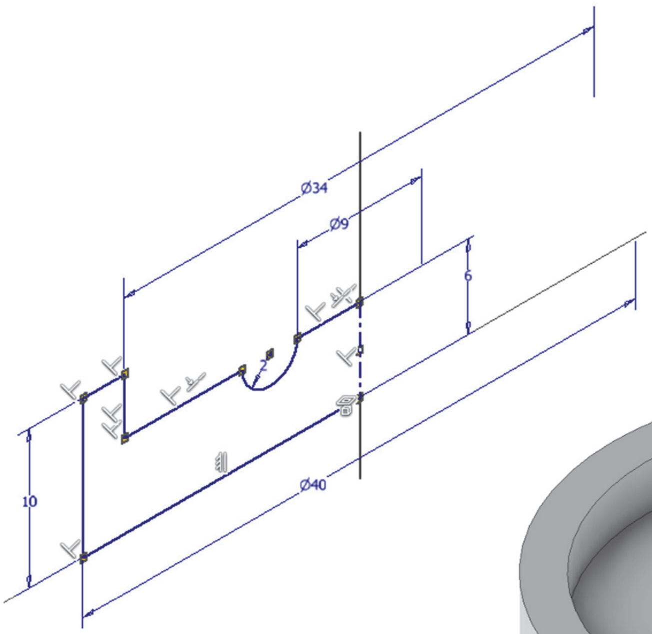
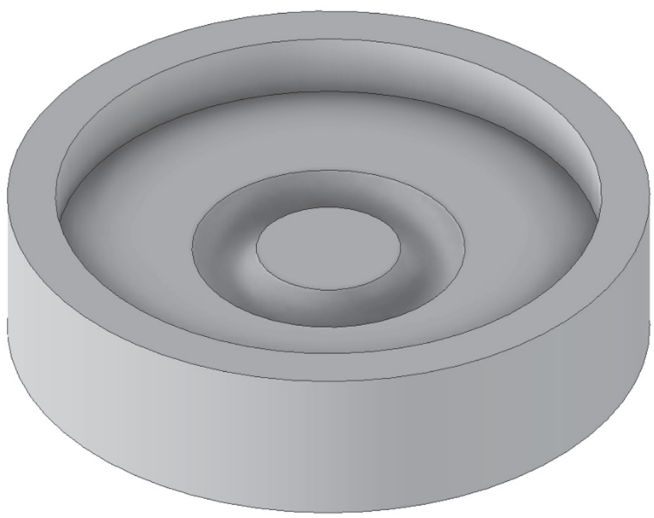
EXERCISE 3

Make the following part using a revolve operation, a hole and a joint.



Step 1. Revolution. Sketch located in frontal or profile plane (indistinctly). Sketch with diameter dimensions to the axis of revolution.





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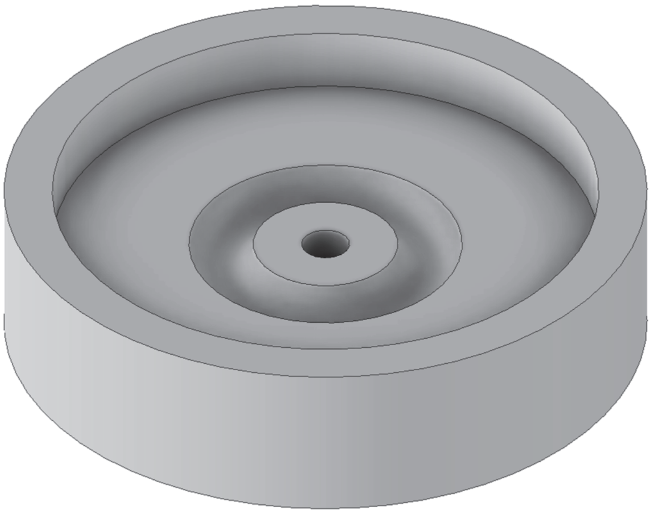
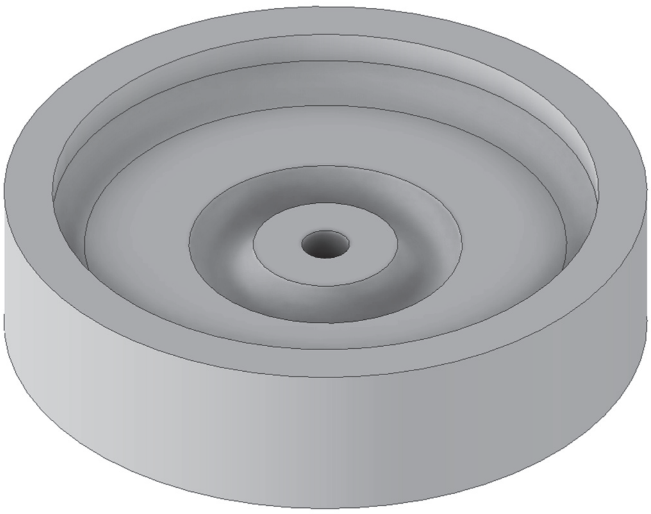
Step 2. Concentric through-hole. No sketch is required.



Step 3. Rounding (splicing). No sketch is required.



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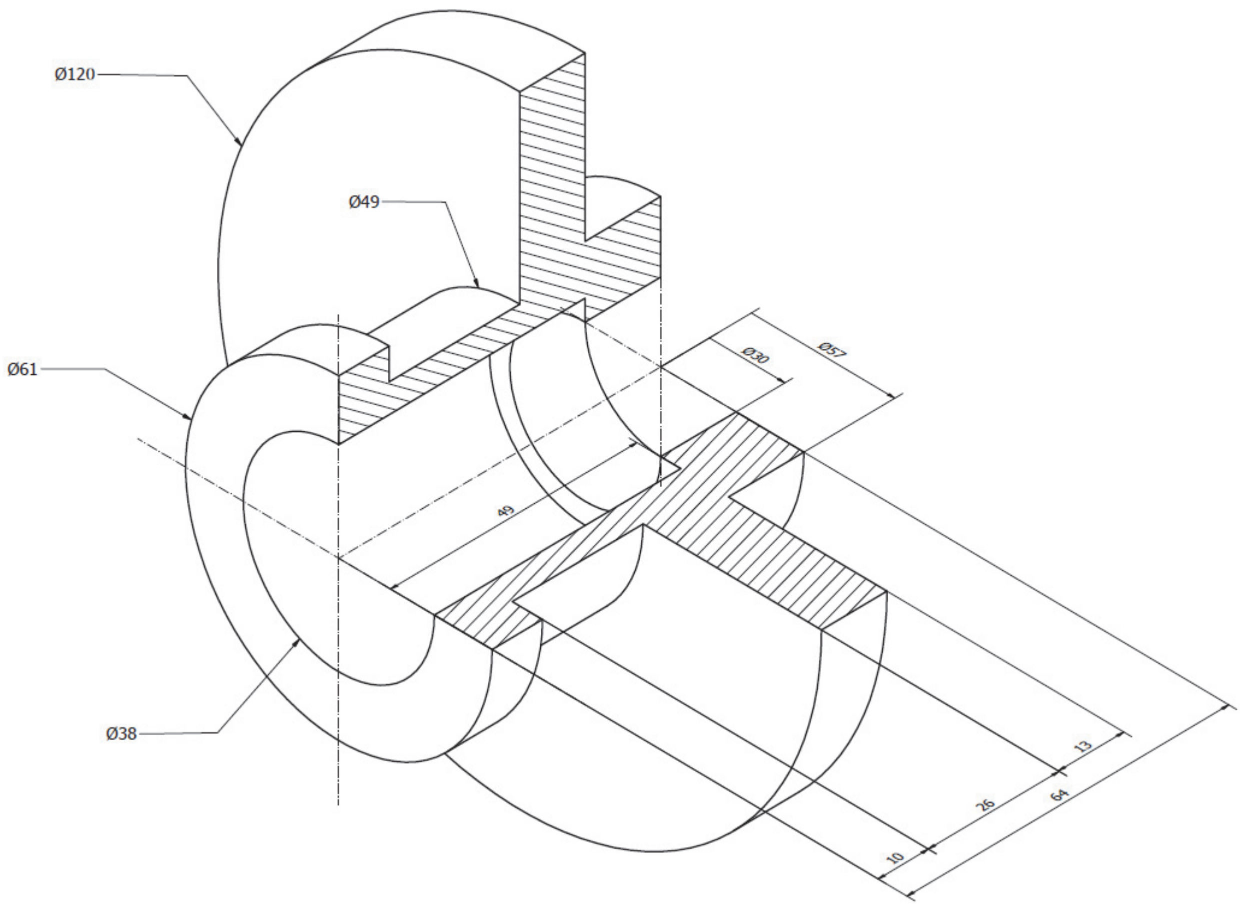
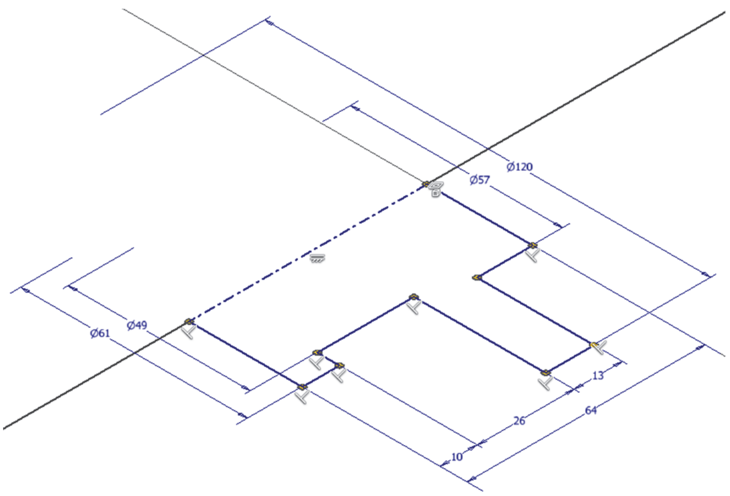
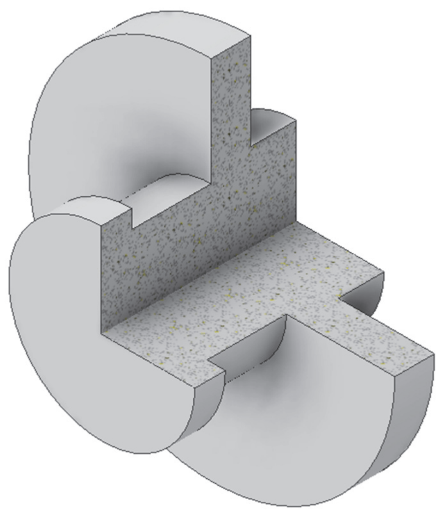
EXERCISE 4

Make the following part using operations of revolution.

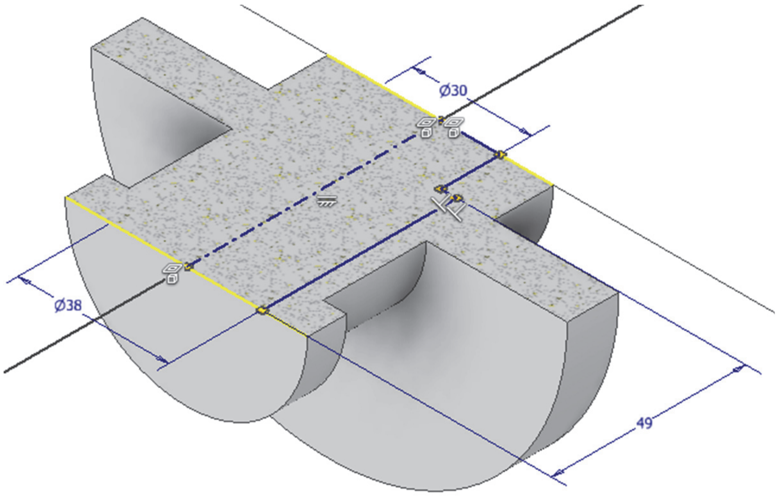


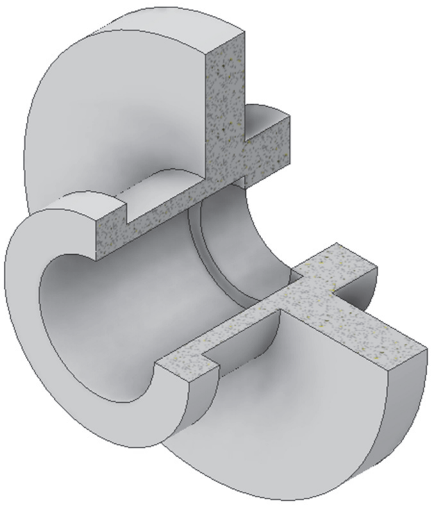
Step 1. Revolution. Sketch located in the horizontal or frontal plane (indistinctly). Sketch with diameter dimensions concerning the axis of revolution.





Step 2. Cutting Revolution. Sketch located in the horizontal or frontal plane (indistinctly). Sketch with diameter dimensions concerning the axis of revolution.





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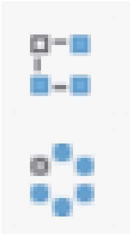
11. SESSION 9: CAD 3D- Patterns

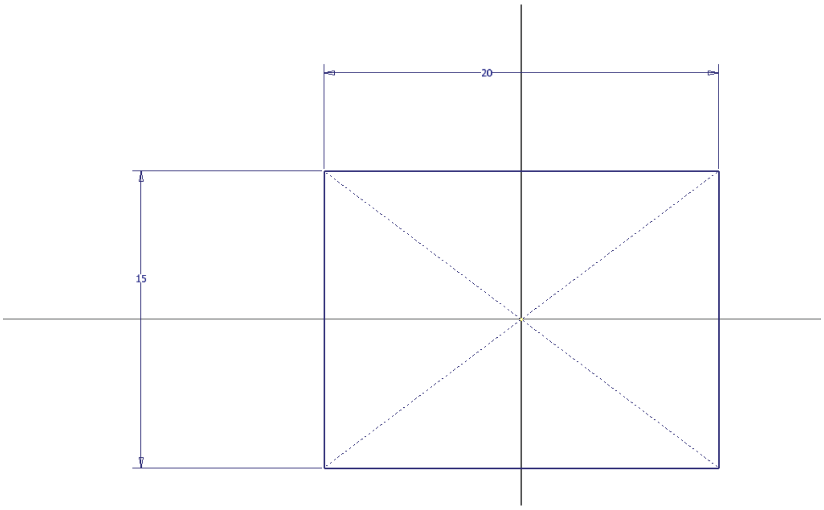
In this session, we will look in depth at:

1. **Operations of revolution**: we will remember how to perform positive and negative revolutions.
2. **Rectangular and fitted patterns**: we will explore the rectangular and circular pattern tools.

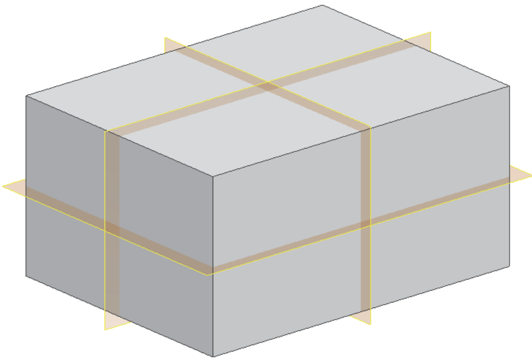


EXERCISE 1. Exercise on rectangular and circular patterns

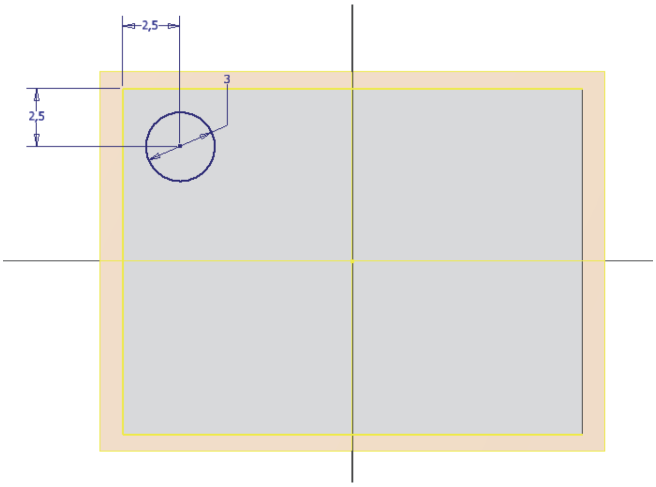
Step 1. Open a new sketch with a rectangle centred on the 20 x 15 mm axes.



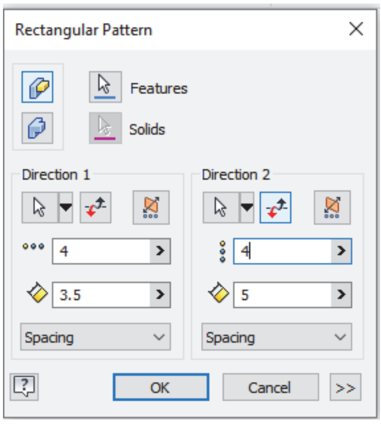
Step 2. Generate a Symmetrical extrusion 10 mm in height.

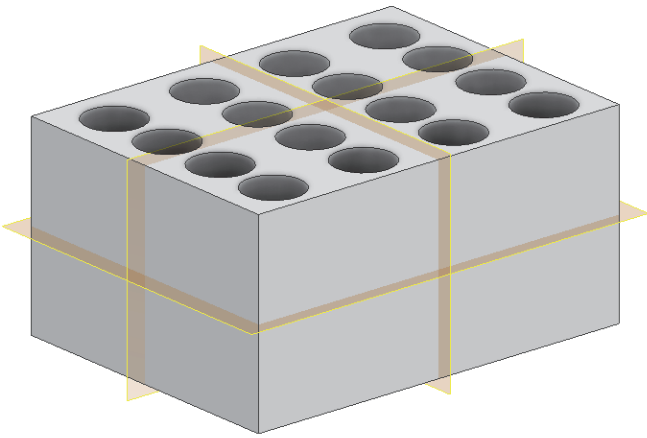


Step 3. Create a through hole with a diameter of 3 mm at 2.5 mm from the edge.



Step 4. Create a rectangular pattern with the following characteristics on different axes. Select the axes appropriately.





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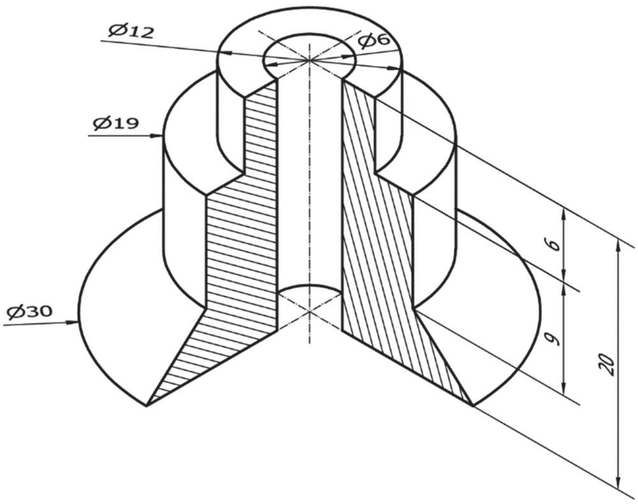
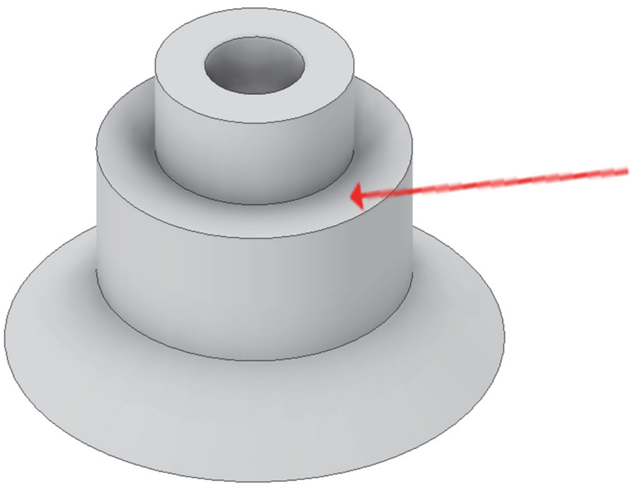
EXERCISE 2. Circular Patterns Exercise

Step 1. Open **exercise 2** from the previous session.



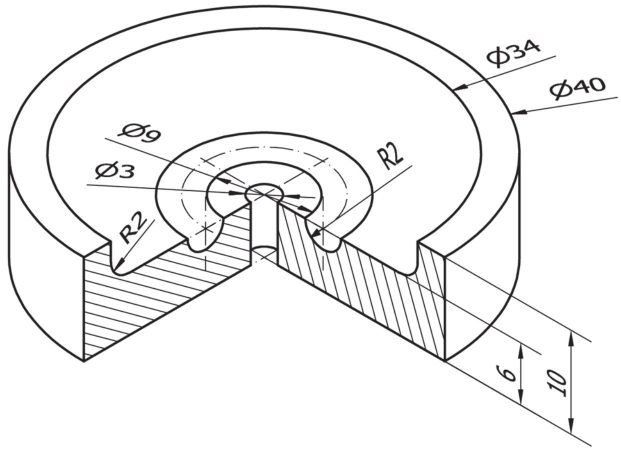
Step 2. Add the following holes using patterns.



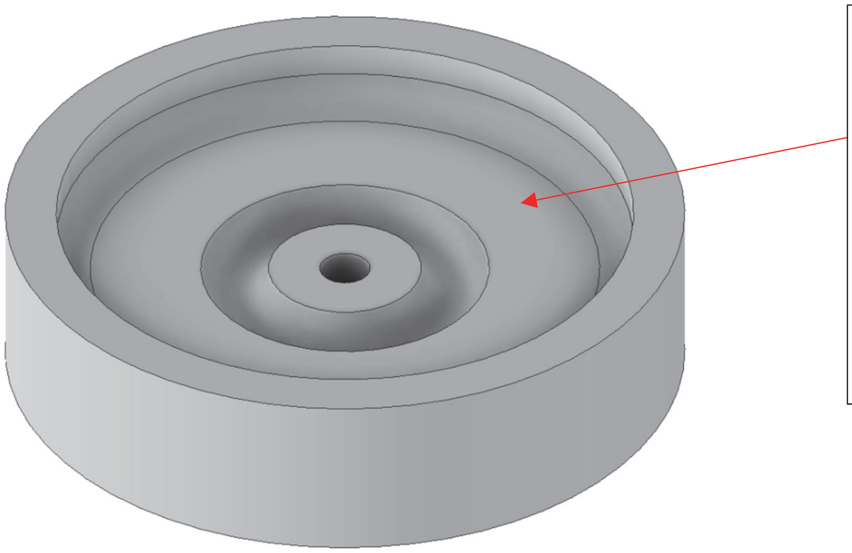
Add to the workpiece 4 ***straight through holes***, to be made with sketch positioning with a point for the initial hole, evenly distributed in a circular (polar) pattern.

EXERCISE 3

Step 1. Open **exercise 3** from the previous session.



Step 2. Includes a circular pattern of 8 ***through threaded holes in the indicated surface of the part.***



Add to the part 8 ***through threaded holes,*** to be made with sketch positioning with a point for the starting hole, evenly distributed in a circular (polar) pattern.

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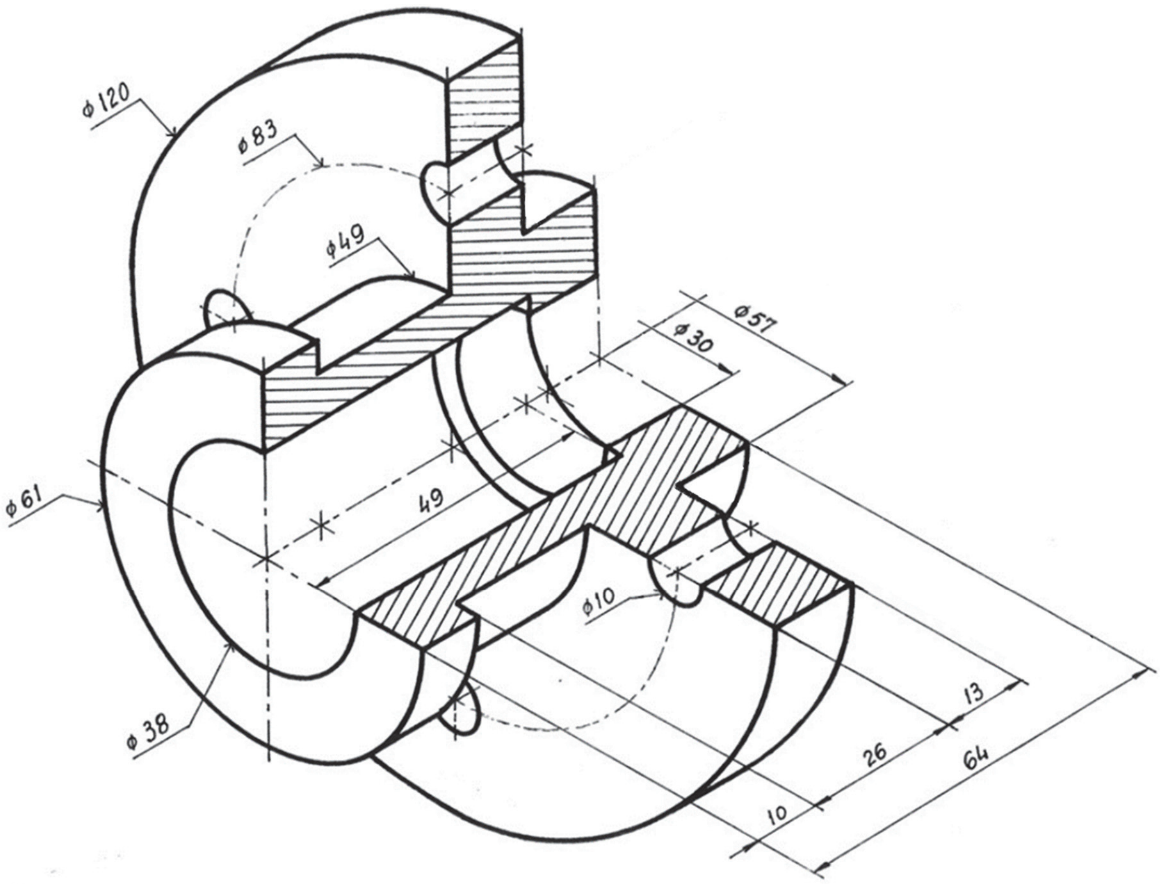
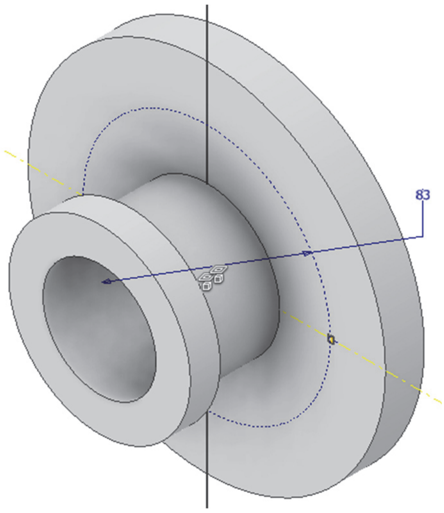
EXERCISE 4

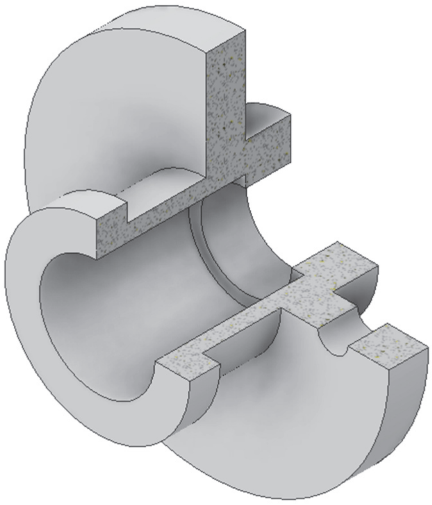
Extend exercise 4 from the previous session by adding the four holes with a diameter of Ø10 mm.



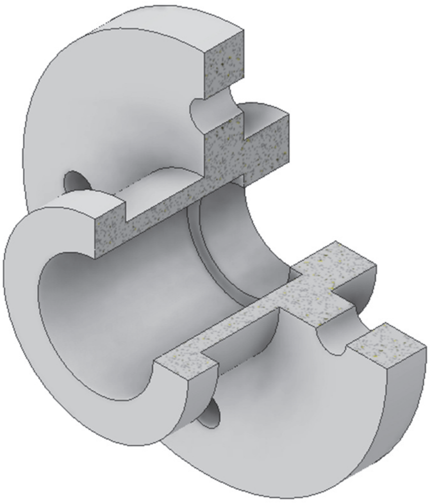
Step 1. Open exercise 4 from the previous session.

Step 2. Create a through hole. Sketch on the larger cylindrical part. Sketch with a point located on the symmetry axis and the auxiliary circumference.





Step 3. Circular pattern with 4 holes about the axis of revolution of the part. No sketch is required.



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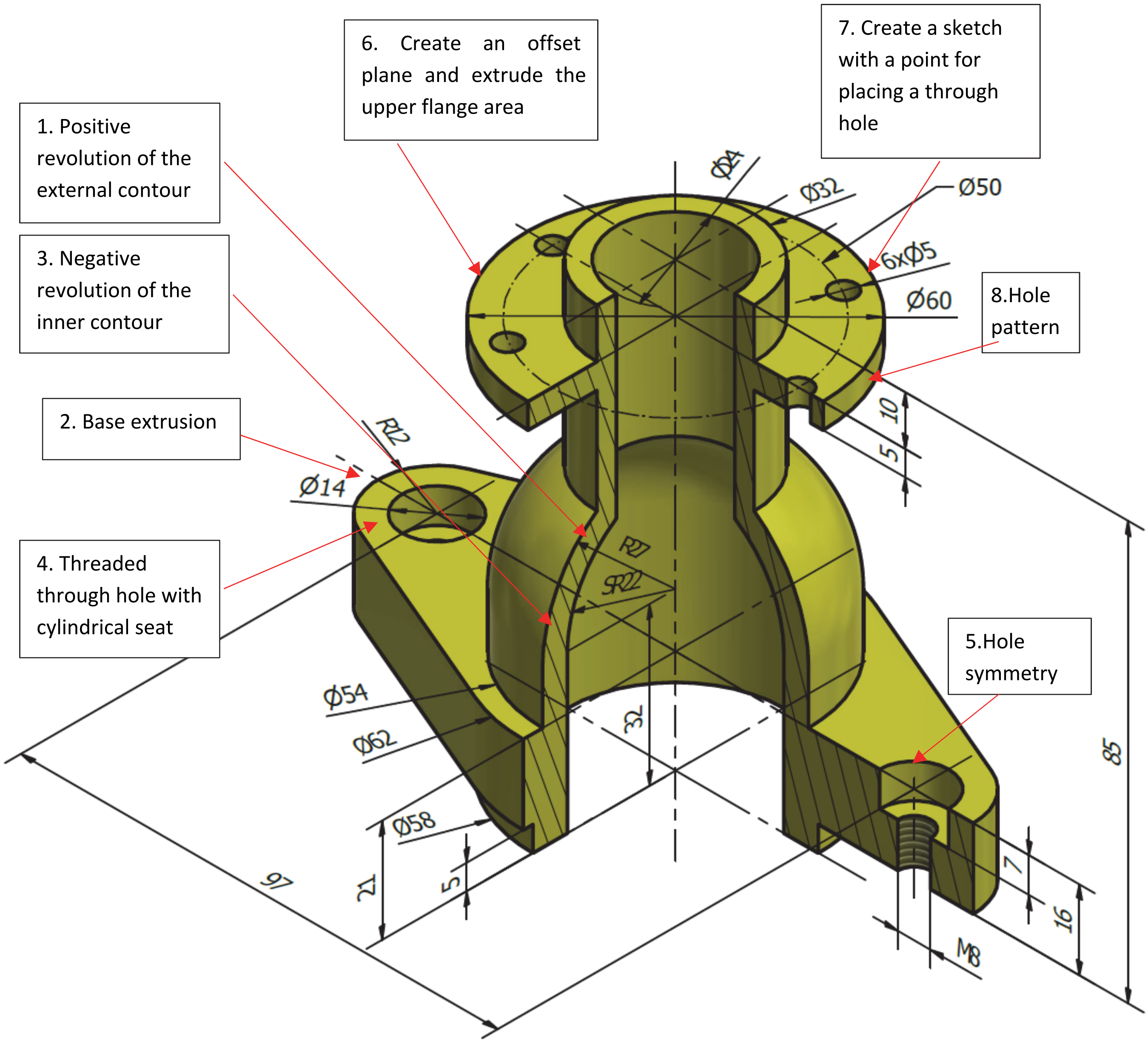
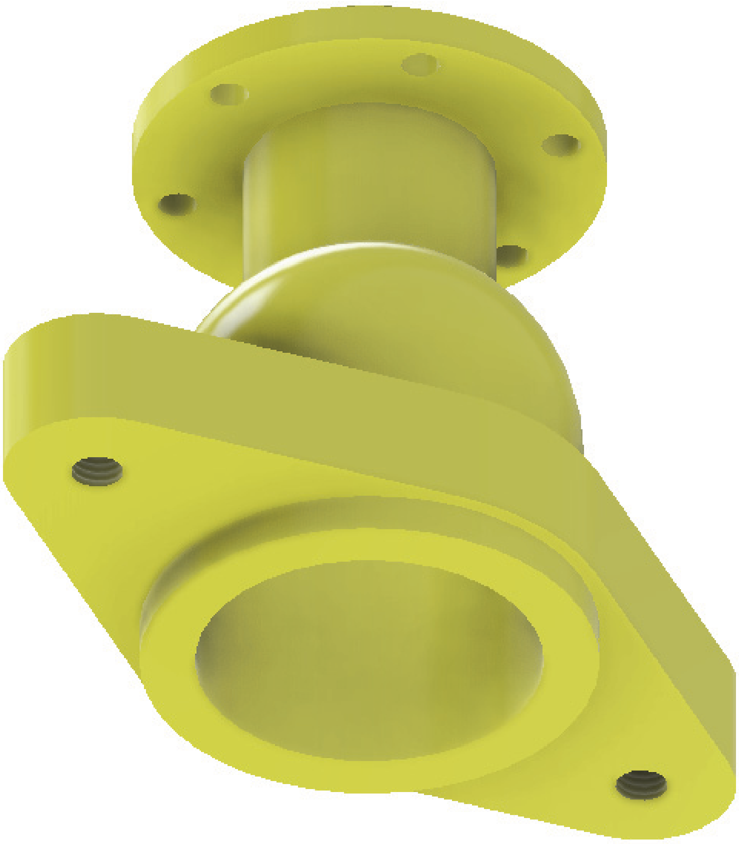
EXERCISE 5

Model the following part represented by the dimensions of the perspectives following the instructions in the note boxes.



NOTE: The "M" in place of the "Ø" symbol means that the hole of the indicated diameter (figure accompanying the M) is threaded with a "Metric" thread profile.



12. SESSION 10: CAD 2D – Standardized representation of engineering parts

This session will cover the following contents, allowing us to obtain views on distinct parts.

1. Drawing: Spline curve
2. Shading
3. Dimensioning
   - Types of dimensioning: linear, aligned, radius, diameter, angle
   - Standards and good practices for correct dimensioning in technical drawings
4. Cuts
   - Types of cuts (total, partial, local)
5. Guidelines and arrows
   - Use of guidelines to accurately guide and define cuts
   - Role and application of guidelines and arrows in technical drawing annotation
   - Standards and conventions of use
6. Styles
   - Text styles in technical drawings
   - Different dimensioning styles and guidelines and their application in technical documentation
7. Editing dimensions
   - Use of grips and properties for precise dimension editing
   - Tools and techniques to adjust and modify dimensions according to design needs
   - Inclusion of dimensioning symbols
8. Practical exercises
   - Representation of engineering parts using views, cuts, auxiliary lines, and dimensioning
   - Practical exercises to apply and reinforce the concepts learned

This chapter will provide a detailed and practical guide to mastering technical drawing techniques, ensuring that engineering parts are represented accurately and clearly, and facilitating their interpretation and manufacture.

EXERCISE 1 Review modification of layers and printing documents (creating files)

With the following layers, modify the exercises of session 4, 5 and 6 by modifying layers and dimensioning.

NAME	LINE TYPE	THICKNESS	COLOR	CHARACTERISTICS	USE
AXES	ISO 10W100	0,18	RED		Axes, circumf. Of construction, plane traces
EXTREMES	CONTINUOUS	0,50	RED		Extremes and changes of direction of cutting planes
DIMENSIONS	CONTINUOUS	0,18	BLUE	ISO-25 dimensioning style (Offset from origin 0, General scale 1.5)	Dimensioning elements
HATCHES	CONTINUOUS	0,18	GREEN	Pattern: ANSI 31 Scale: 0.9	Section hatching
VIEWS	CONTINUOUS	0,50	BLACK	Text Style: Arial Font Text Size: 2.5 Guideline Style: Text Size and arrow: 6	Edges and contours: thickening cutting planes
GRAPHIC WINDOW	CONTINUOUS	0,18	MAGENTA	Optional	Auxiliary lines and curves

Line type scale 1.2

You must update the pdf printout of the three exercises and update the scale in the box so that the entire part, including the dimensions, can be displayed.

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EXERCISE 2

SURNAME, NAME	ID. PC

TO CORRECT YOUR EXAM, IT MUST BE SUBMITTED WITH THE PROPER IDENTIFICATION: SURNAME, NAME, ID PC.

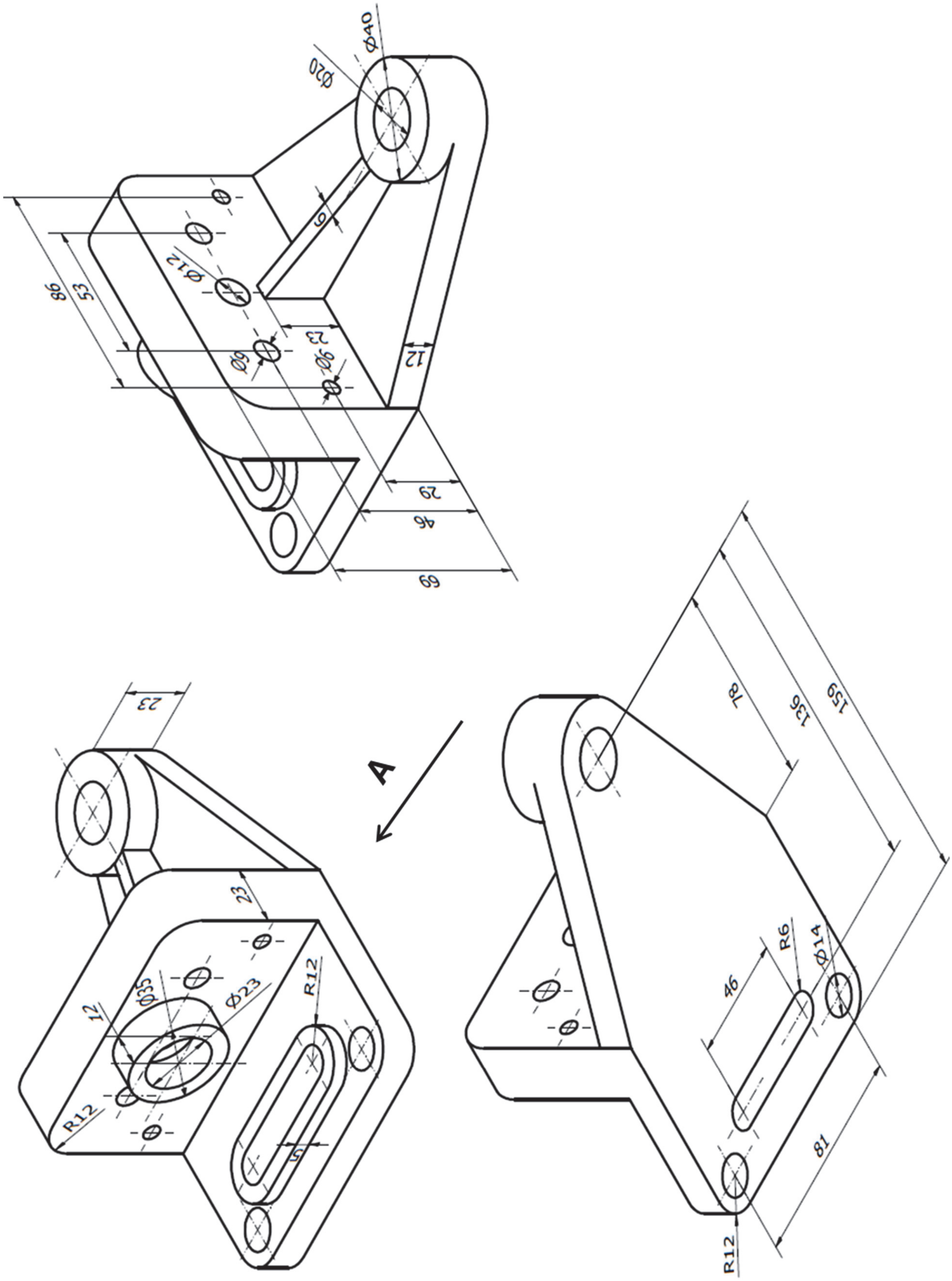
It is the student's responsibility to save the exam in the grade folder. Exams saved after the due date will be considered NOT SUBMITTED and will NOT BE GRADED.

Exercise: Draw the following part in AutoCAD© using 3 dimensioned views (you will be penalised if you do not follow the norm) using the European System and without hidden lines, using the AutoCAD© layers functionality.

1. Views
   - ❑ Draw the front view according to direction A. The front view should be cut by the symmetry plane.
   - ❑ Draw the top view. The top view should be cut by parallel planes (which allows you to define all the holes with de different diameters Ø6, Ø9, Ø23, and Ø12 mm).
   - ❑ Draw the left view. The left view should be cut by a local cut (which allows you to see the interior of the through-holes with a diameter of Ø14 mm).

2. Clarifications
   - ❑ Create a presentation in A4 format (horizontal) at a 2:3 scale (use the title block provided)

3. Files should be submitted in P:\GITI or P:\GIE\Folder_Student
   - ❑ PDF File (1stSurname_2ndSurname_Name.pdf)
   - ❑ DWG File (1stSurname_2ndSurname_Name.dwg)



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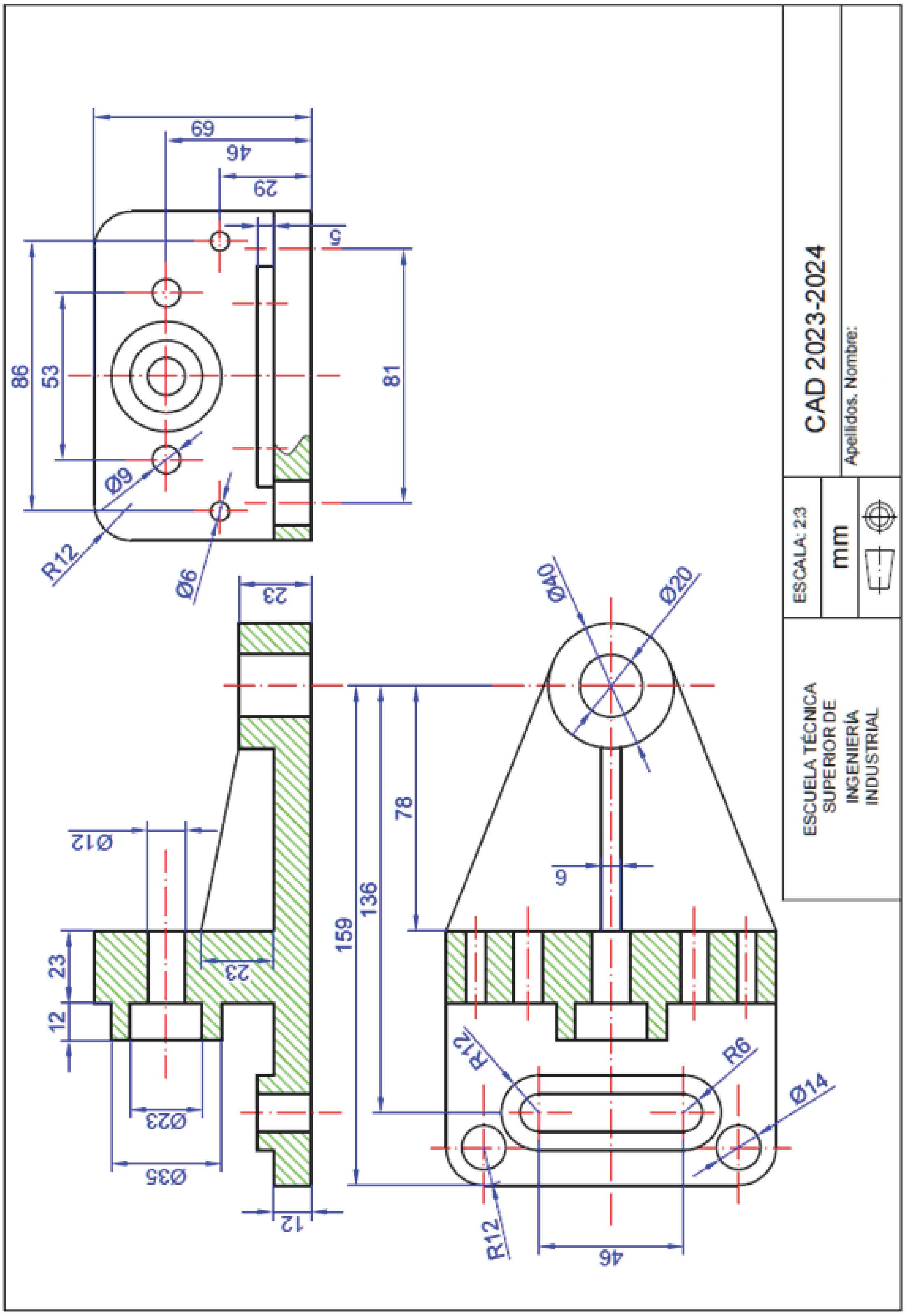

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EXERCISE 3

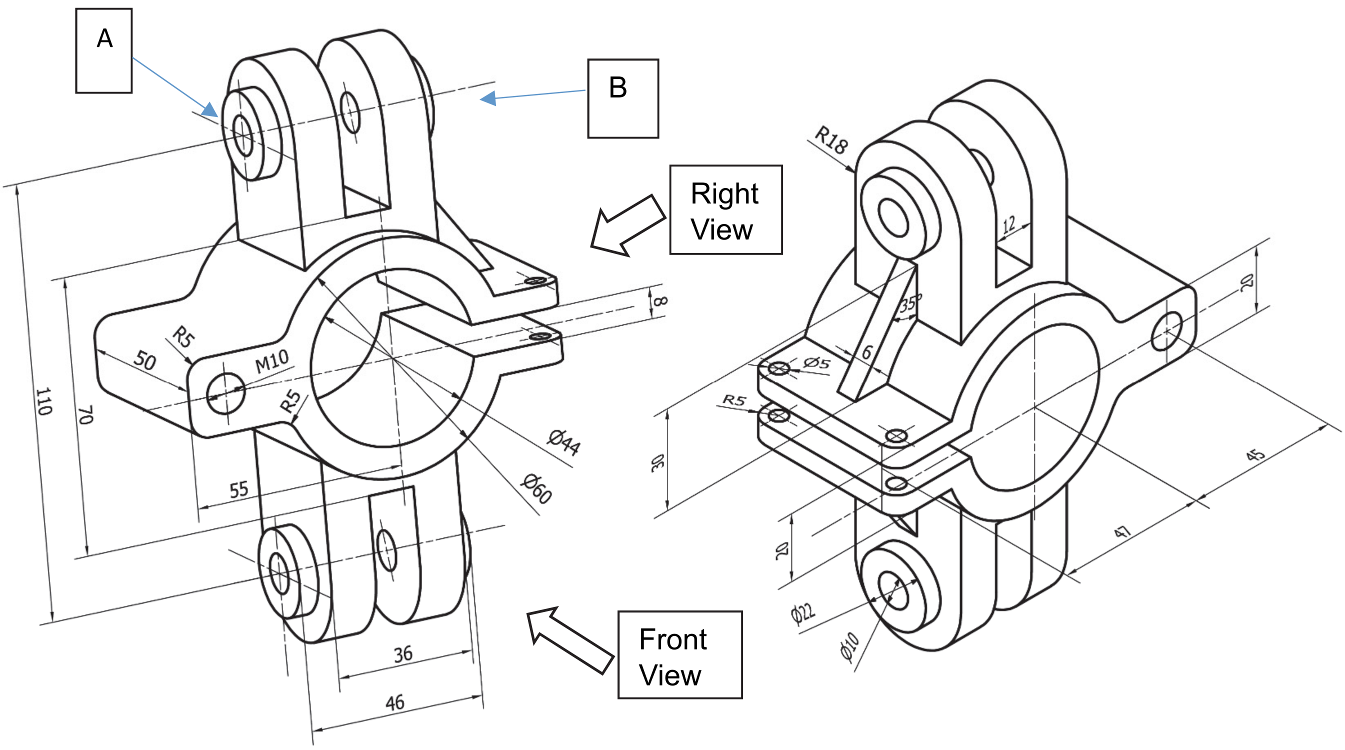
SURNAME, NAME	ID. PC

TO CORRECT YOUR EXAM, IT MUST BE SUBMITTED WITH THE PROPER IDENTIFICATION: SURNAME, NAME, ID PC.

It is the student's responsibility to save the exam in the grade folder. Exams saved after the due date will be considered NOT SUBMITTED and will NOT BE GRADED.

Exercise: Draw the following part in AutoCAD© using 3 dimensioned views (you will be penalised if you do not follow the norm) using the European System and without hidden lines, using the AutoCAD© layers functionality.

1. Views
   - ❑ Draw the front view according to the direction "Front View," made without cuts.
   - ❑ Draw the top view. The top view should be cut by parallel planes (which allows you to see a complete view of the M10 thread, as well as holes "A" and "B").
   - ❑ Draw the right view. The right view should be cut by a local cut (which allows you to see the interior of the through-holes Ø5 mm).
2. Clarifications
   - ❑ All the holes are through holes.
   - ❑ Create a presentation in A4 format (horizontal) at a 3:5 scale (use the title block provided).
3. Files should be submitted in P:\GITI or P:\GIE\Folder_Student
   - ❑ PDF File (1stSurname_2ndSurname_Name.pdf)
   - ❑ DWG File (1stSurname_2ndSurname_Name.dwg)



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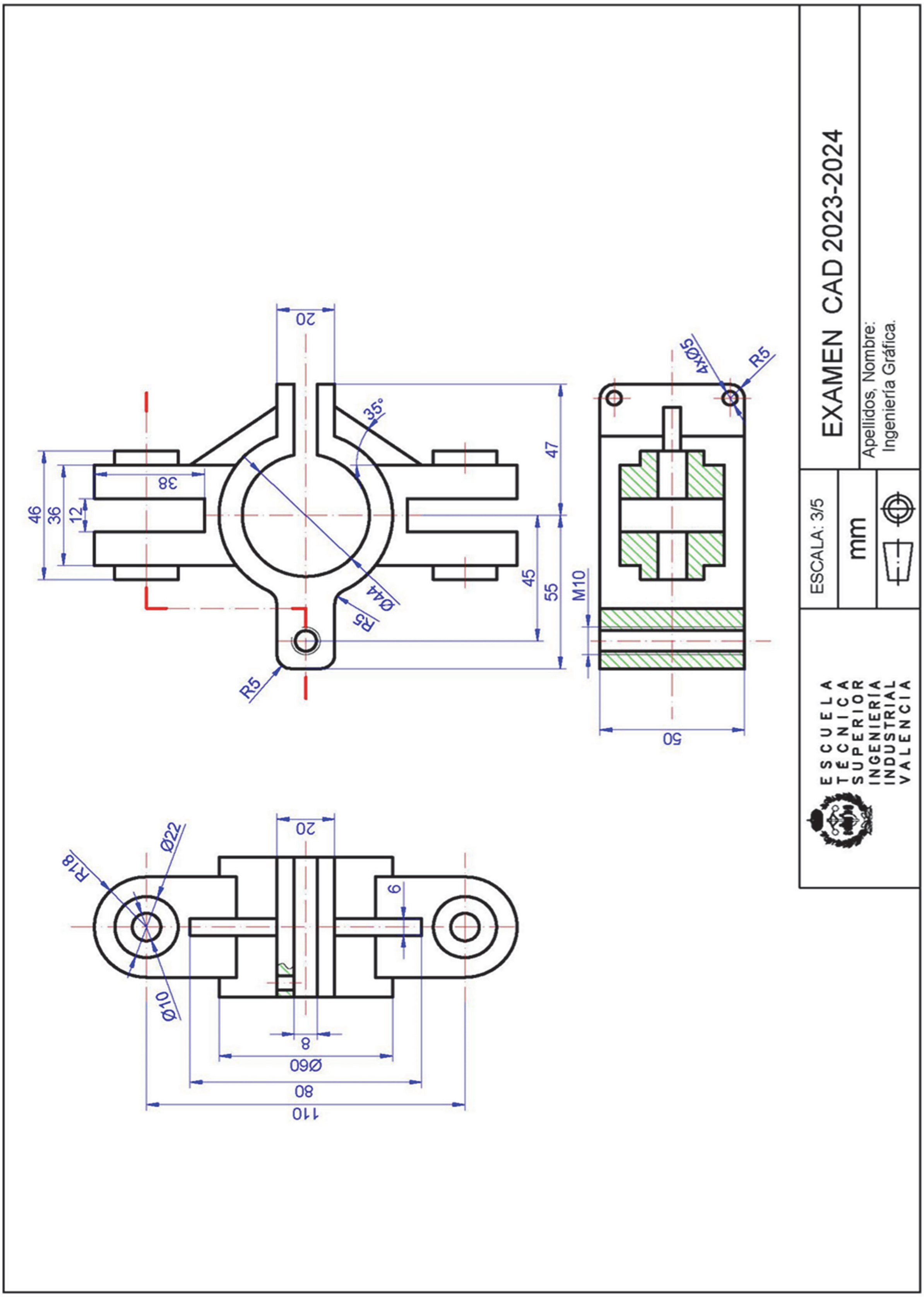

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13. SESSION 11: CAD 2D - Standardised representation of engineering parts II

EXERCISE 1

Exercise: Draw the following part in AutoCAD© using 3 dimensioned views (you will be penalised if you do not follow the norm) using the European System and without hidden lines, using the AutoCAD© layers functionality.

1. Views
   - ❑ Draw the front view according to direction A. The front view should be cut by the symmetry plane of the part.
   - ❑ Draw the top view. The top view should be done with a local cut (which allows you to define the holes with Ø10 mm diameter).
   - ❑ Draw the left view without cutting.
2. Clarifications
   - ❑ All the holes are through holes.
   - ❑ Create a presentation in A4 format (horizontal) at a 2:3 scale (use the title block provided).
3. Files should be submitted in P:\GITI or P:\GIE\Folder_Student
   - ❑ PDF File (1stSurname_2ndSurname_Name.pdf)
   - ❑ DWG File (1stSurname_2ndSurname_Name.dwg)

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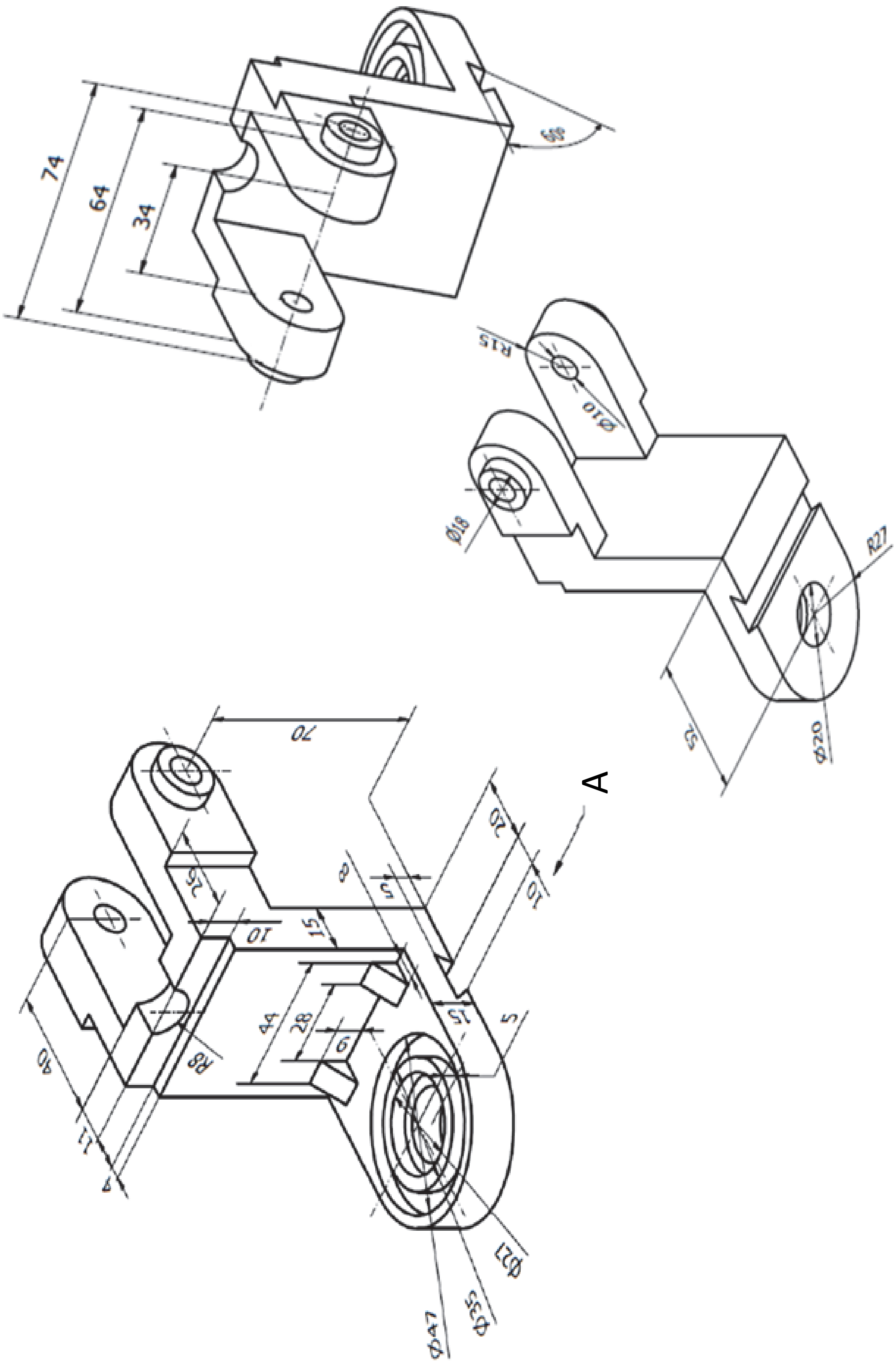
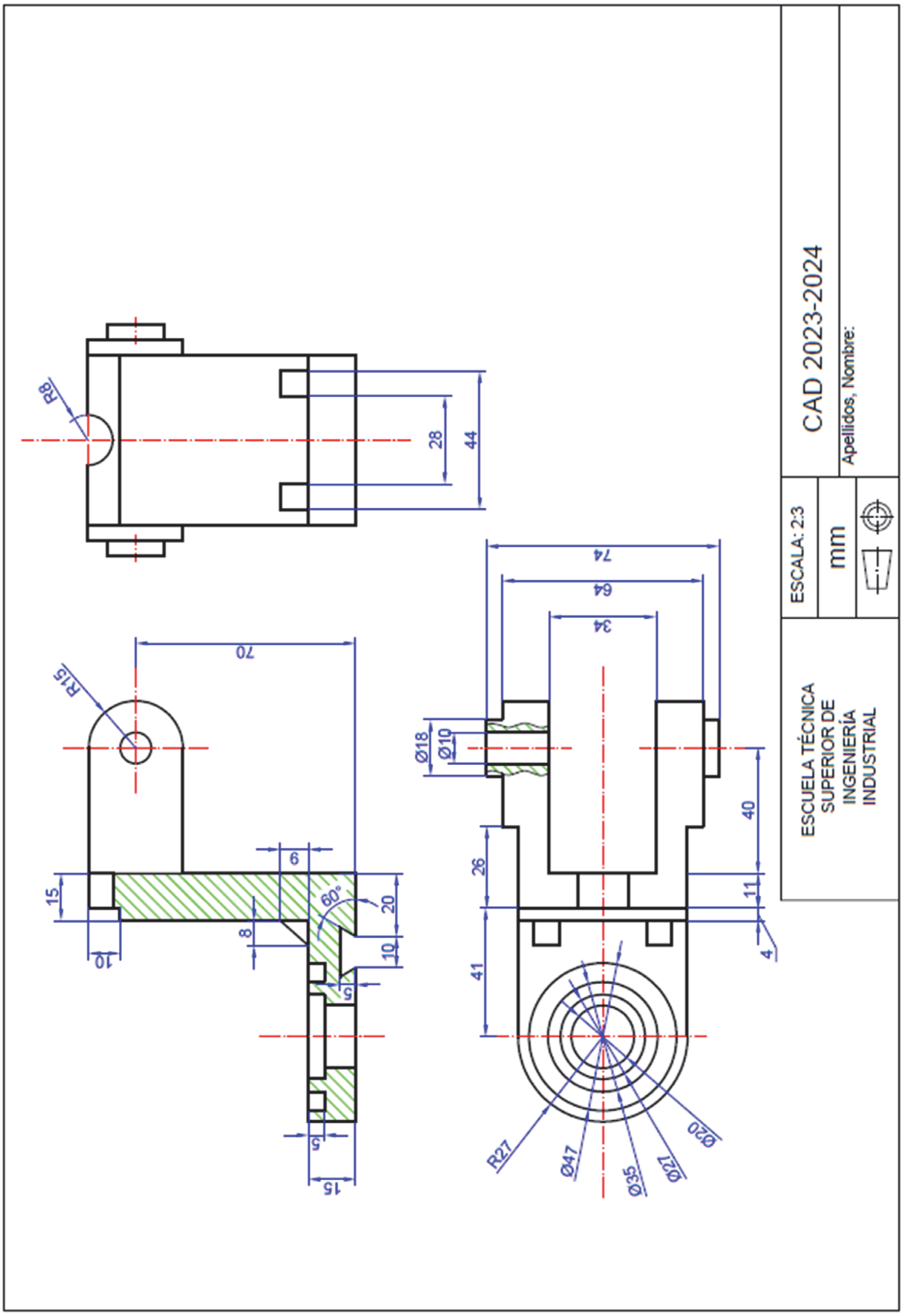

74
64
34
R15
Ø10
Ø18
R27
Ø20
A
70
Ø47
Ø35
Ø27



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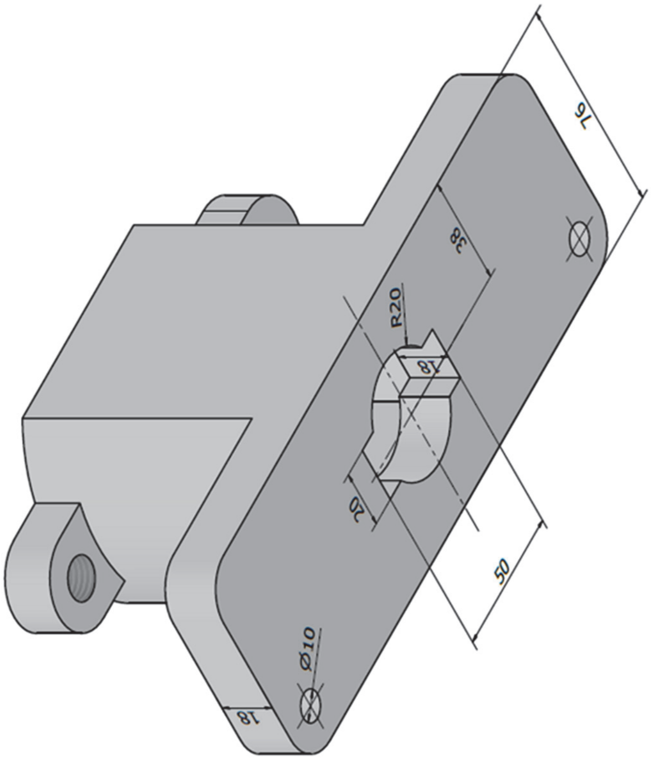
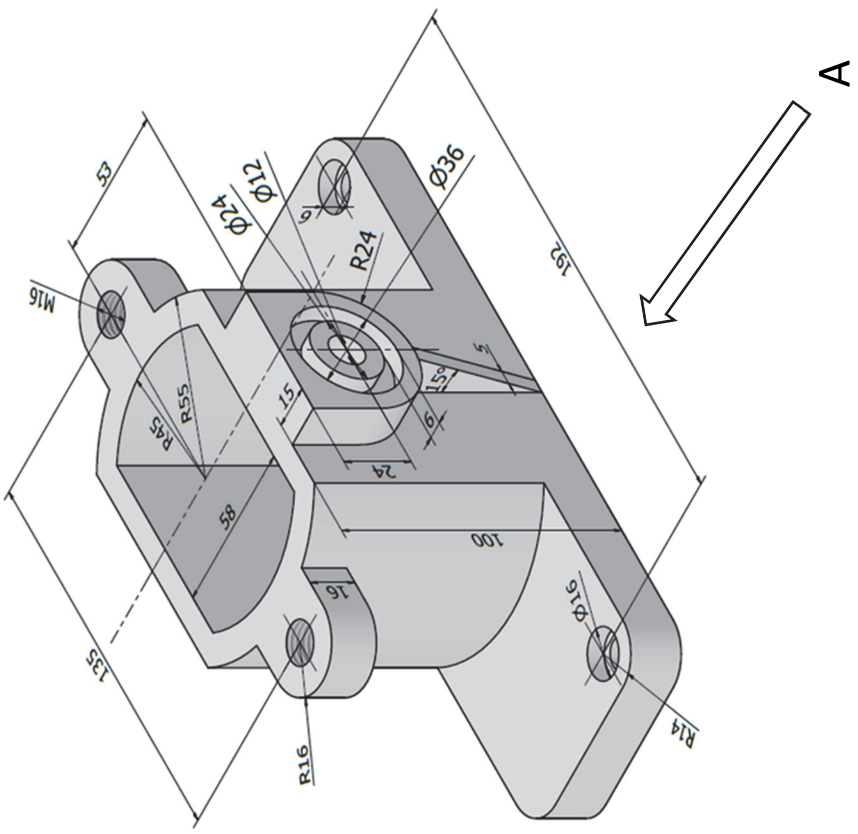
EXERCISE 2

Exercise: Draw the following part in AutoCAD© using 3 dimensioned views (you will be penalised if you do not follow the norm) using the European System and without hidden lines, using the AutoCAD© layers functionality.

1. Views
   - ❑ Draw the front view according to direction A, made with a half-cut. The front view should have a local cut (which allows you to see the interior of the through hole of Ø16 mm).
   - ❑ Draw the top view without cuts.
   - ❑ Draw the left view. The left view should be cut by a total cut using the plane of symmetry.
2. Clarifications
   - ❑ The Ø12 mm hole reaches the interior surface of the part.
   - ❑ All the other holes are through holes.
   - ❑ Create a presentation in A4 format (horizontal) at a 2:5 scale (use the title block provided).

4. Files should be submitted in P:\GITI or P:\GIE\Folder_Student
   - ❑ PDF File (1stSurname_2ndSurname_Name.pdf)
   - ❑ DWG File (1stSurname_2ndSurname_Name.dwg)





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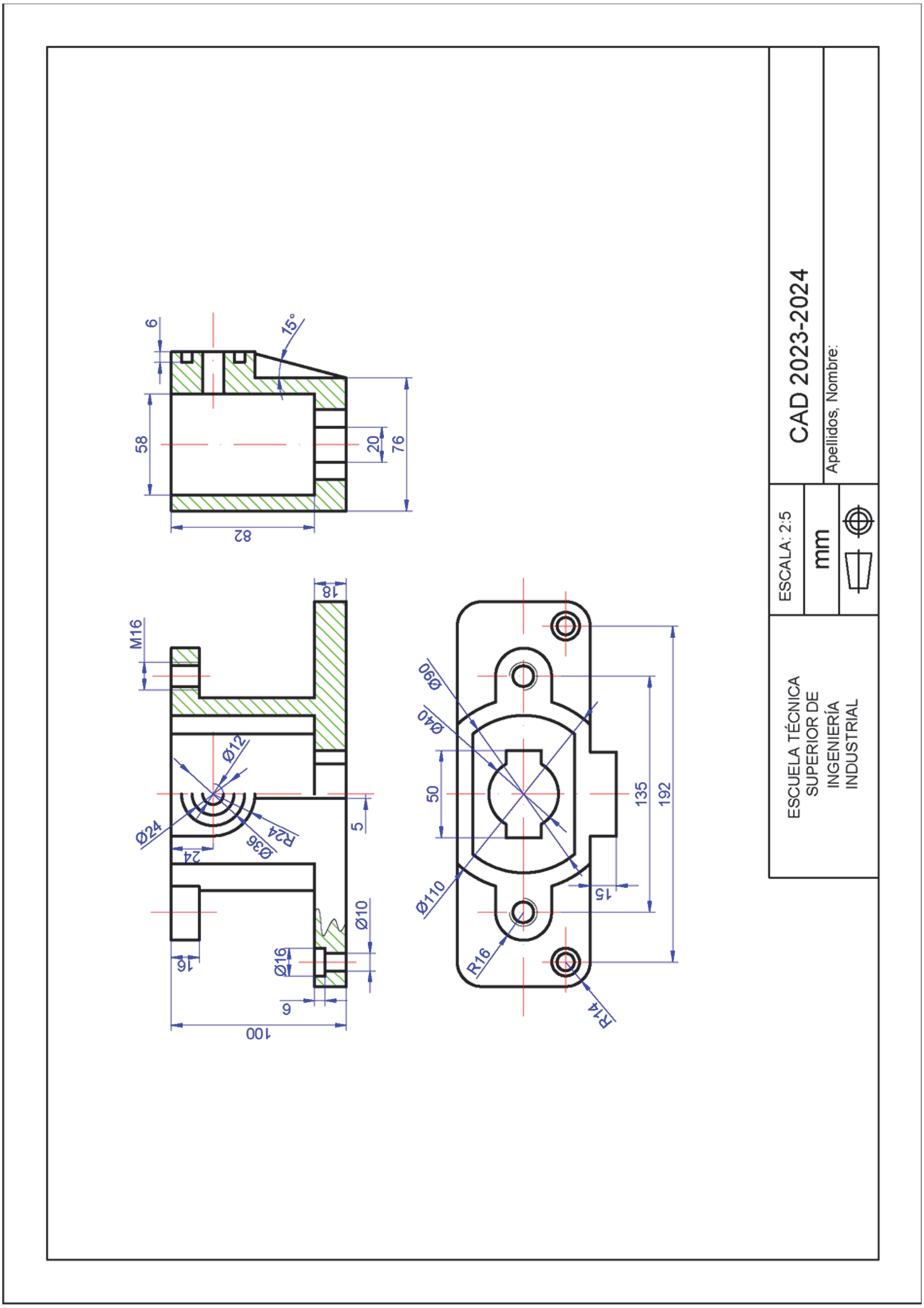
CAD 2023-2024
Apellidos, Nombre:
ESCALA: 2:5
mm
ESCUELA TÉCNICA
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INGENIERÍA
INDUSTRIAL
6
15°
58
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76
82
18
M16
Ø12
Ø24
Ø36
R24
24
5
Ø10
Ø16
16
6
100
Ø90
Ø40
50
Ø110
R16
R14
15
135
192

EXERCISE 3

Exercise: Draw the following part in AutoCAD© using 3 dimensioned views (you will be penalised if you do not follow the norm) using the European System and without hidden lines, using the AutoCAD© layers functionality.

1. View
   - ❑ Draw the front view according to direction A. A half-cut should cut the front view. The front view also should have a local cut (which allows you to see the through-hole character of the Ø20 mm hole.
   - ❑ Draw the top view. The top view should be done with a local cut to show the through character of the rear grooves.
   - ❑ Draw the right view. The right view should be cut by the symmetry plane.
2. Clarifications
   - ❑ All the holes are through holes except the threaded hole.
   - ❑ The thread depth is 30 mm with full thread.
   - ❑ Create a presentation in A4 format (horizontal), at a 1:3 scale (use the title block provided).
5. Files should be submitted in P:\GITI or P:\GIE\Folder_Student
   - ❑ PDF File (1stSurname_2ndSurname_Name.pdf)
   - ❑ DWG File (1stSurname_2ndSurname_Name.dwg)

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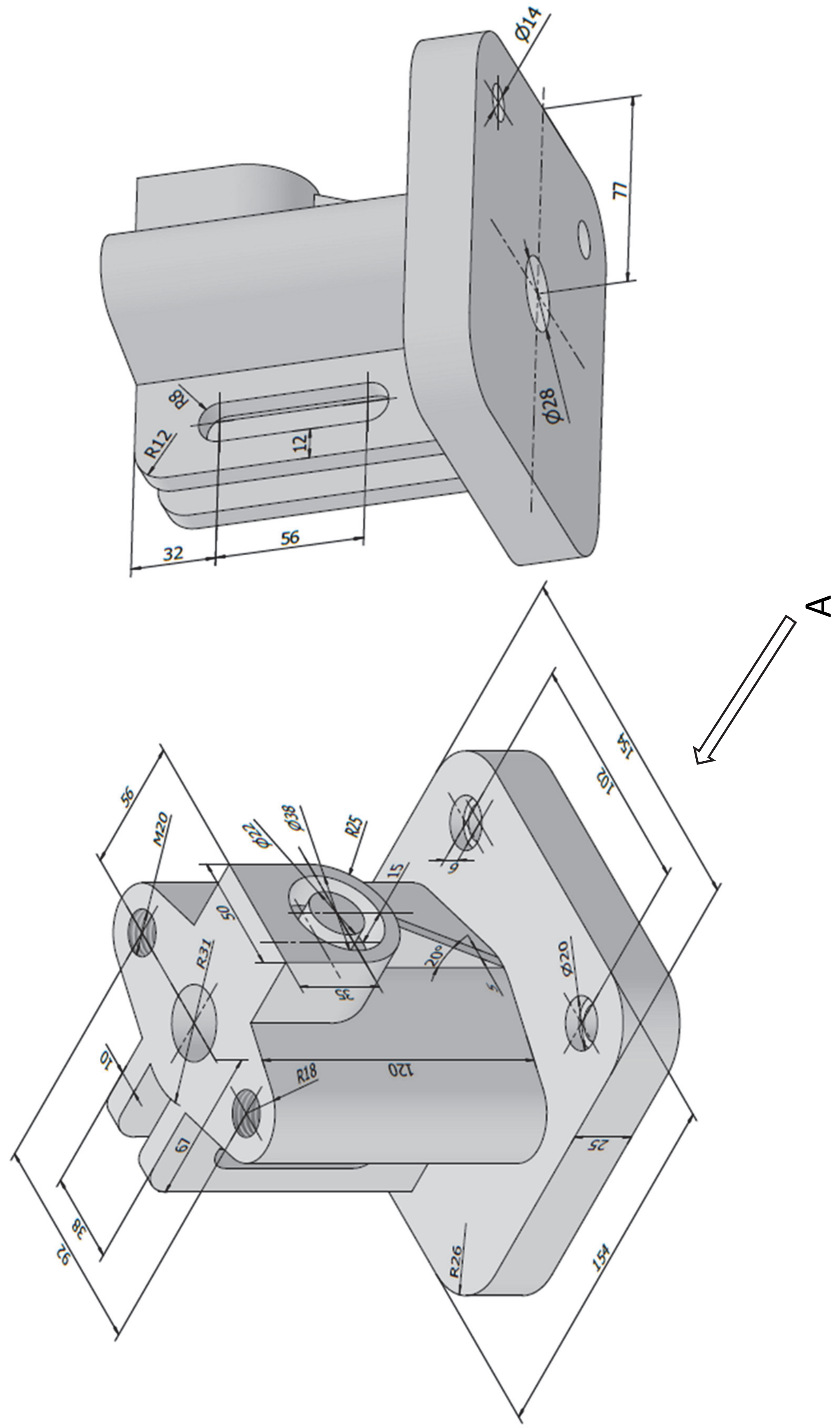
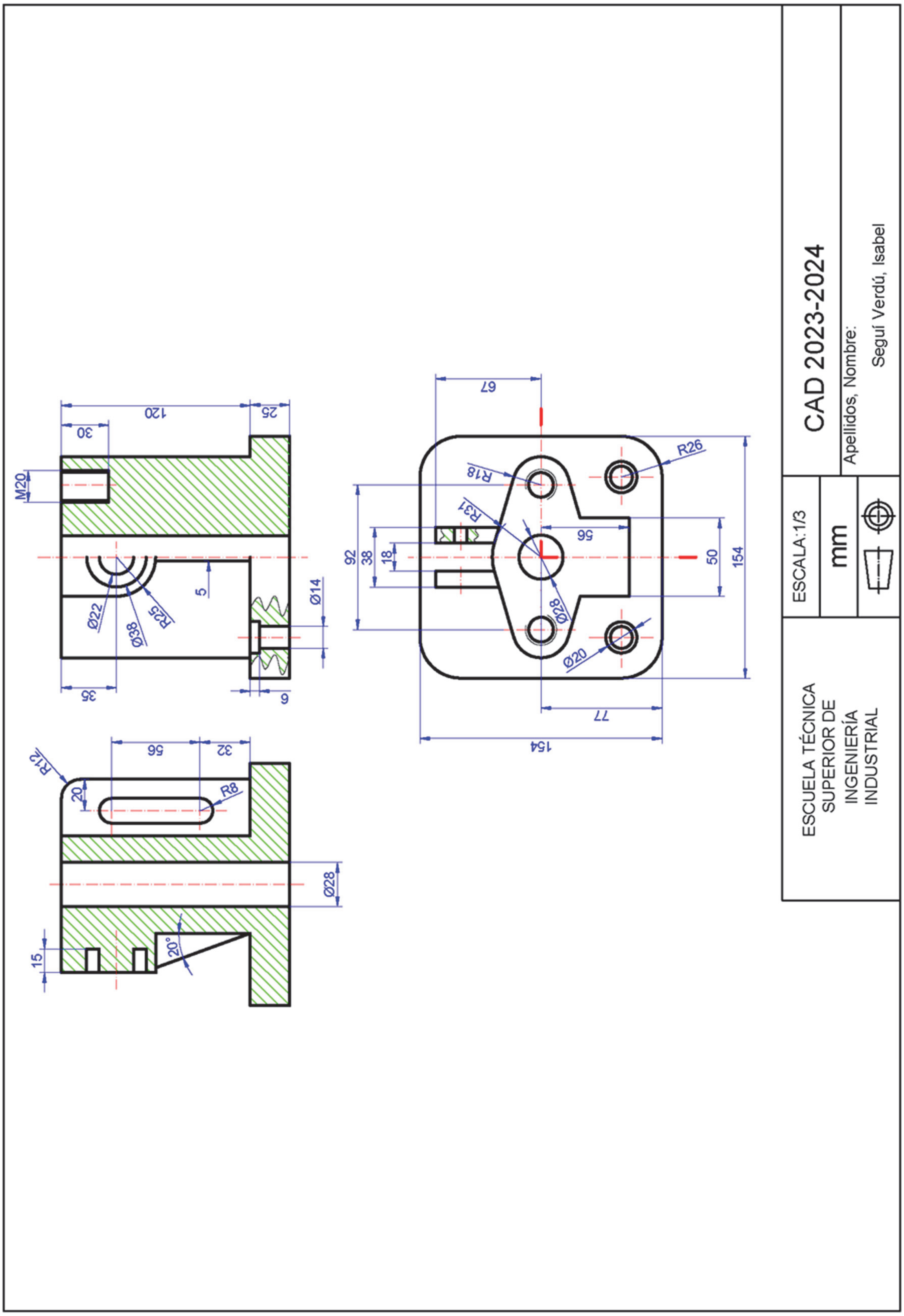

Ø14
77
Ø28
R8
R12
12
32
56
A
56
M20
Ø22
Ø38
R25
15
50
R31
20°
Ø20
R18
67
R26
154



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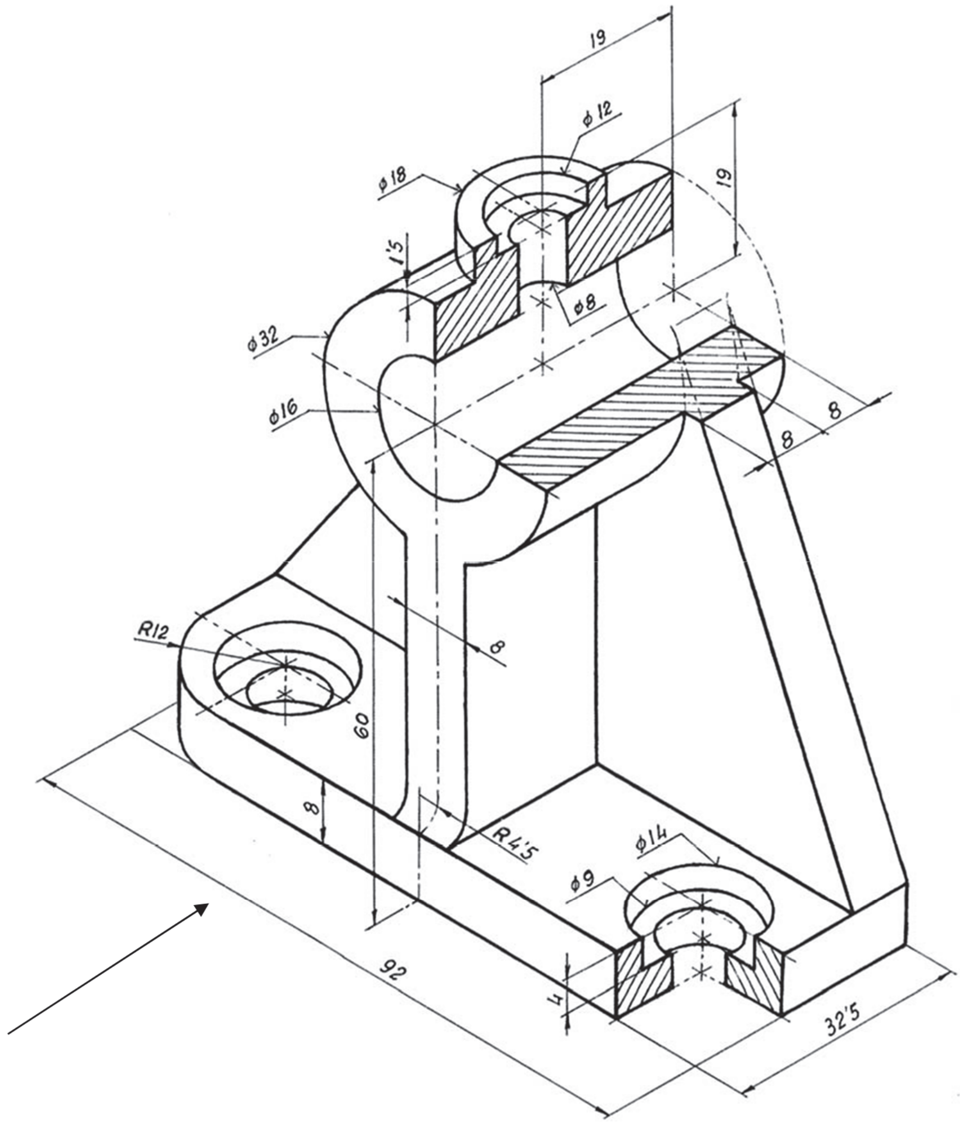
14. SESSION 12: CAD 2D - Standardised Representation of Engineering Parts III

EXERCISE 1

Exercise: Draw the following part in AutoCAD© using 3 dimensioned views (you will be penalised if you do not follow the norm) using the European System and without hidden lines, using the AutoCAD© layers functionality.

1. Views
   - ❑ Draw the front view according to direction A. The front view should be done with a local cut (which allows you to see the holes in the base).
   - ❑ Draw the top view. The top view should be cut by a plane parallel to the base and at a height of 30 mm from the ground.
   - ❑ Draw the left view. The left view should have a half cut (which allows you to see the inside of the two upper cylindrical elements).
2. Clarifications
   - ❑ Create a presentation in A4 format (horizontal), at a 1:1 scale (use the title block provided).
3. Files should be submitted in P:\GITI or P:\GIE\Folder_Student
   - ❑ PDF File (1stSurname_2ndSurname_Name.pdf)
   - ❑ DWG File (1stSurname_2ndSurname_Name.dwg)



A

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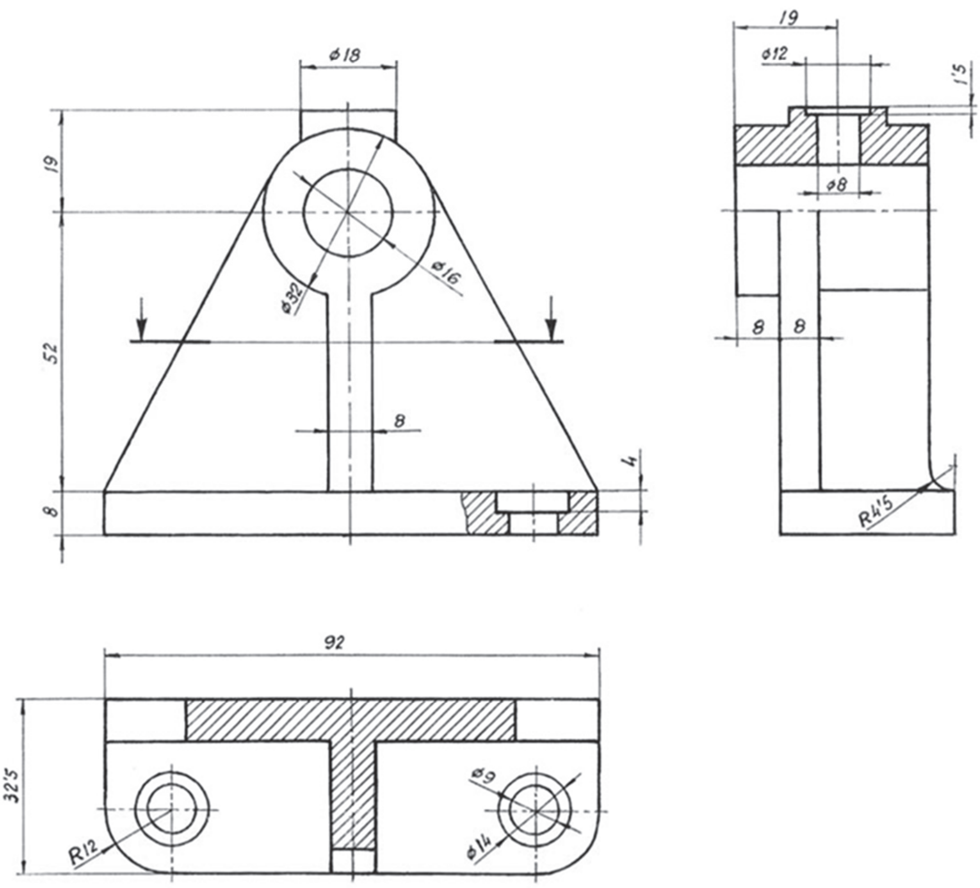

ESCUELA TÉCNICA SUPERIOR DE INGENIERÍA INDUSTRIAL	ESCALA	CAD 2021-2022
	mm	Apellidos, Nombre:

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EXERCISE 2

Exercise: Draw the following part in AutoCAD© using 3 dimensioned views (you will be penalized if you do not follow the norm) using the European System and without hidden lines, using the AutoCAD© layers functionality.

1. Views
   - ❑ Draw the front view. The front view should be cut by the symmetry plane of the part.
   - ❑ Draw the top view. The top view should be done with a local cut (which allows you to define the hole with the axis e2).
   - ❑ Draw the right view. The right view should be cut by a vertical plane (which allows you to see the rectangular holes).
2. Clarifications
   - ❑ The part has a plane of symmetry, and all the holes are through holes.
   - ❑ Create a presentation in A4 format (horizontal), at a 1:2 scale (use the title block provided).
3. Files should be submitted in P:\GITI or P:\GIE\Folder_Student
   - ❑ PDF File (1stSurname_2ndSurname_Name.pdf)
   - ❑ DWG File (1stSurname_2ndSurname_Name.dwg)

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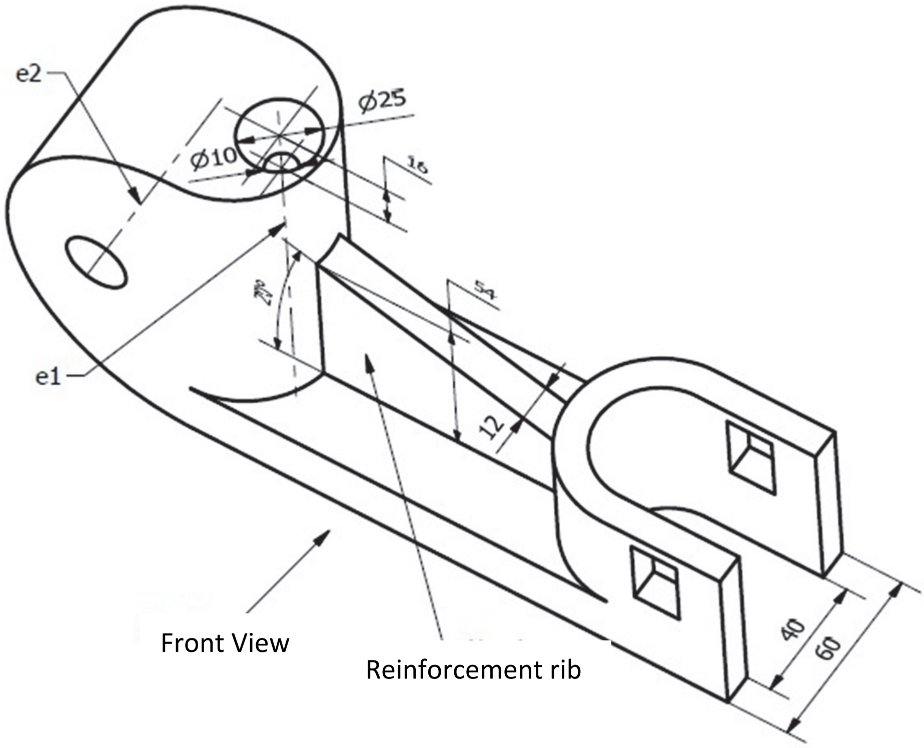
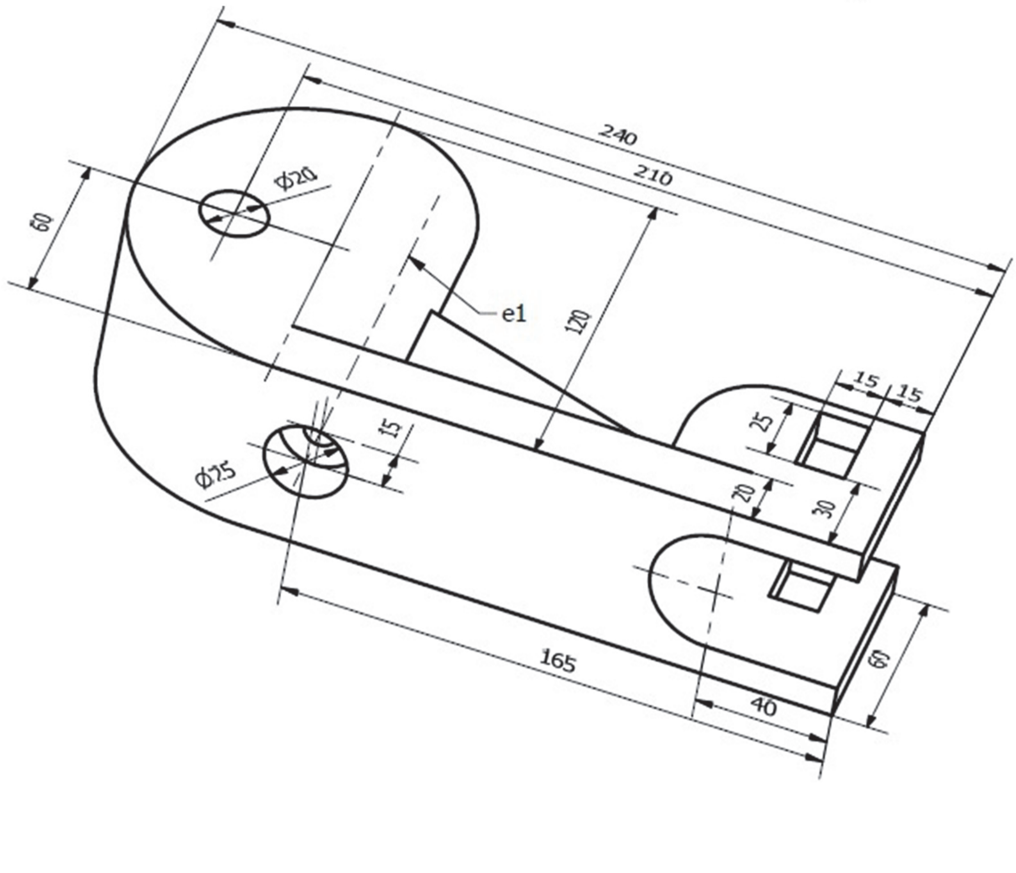
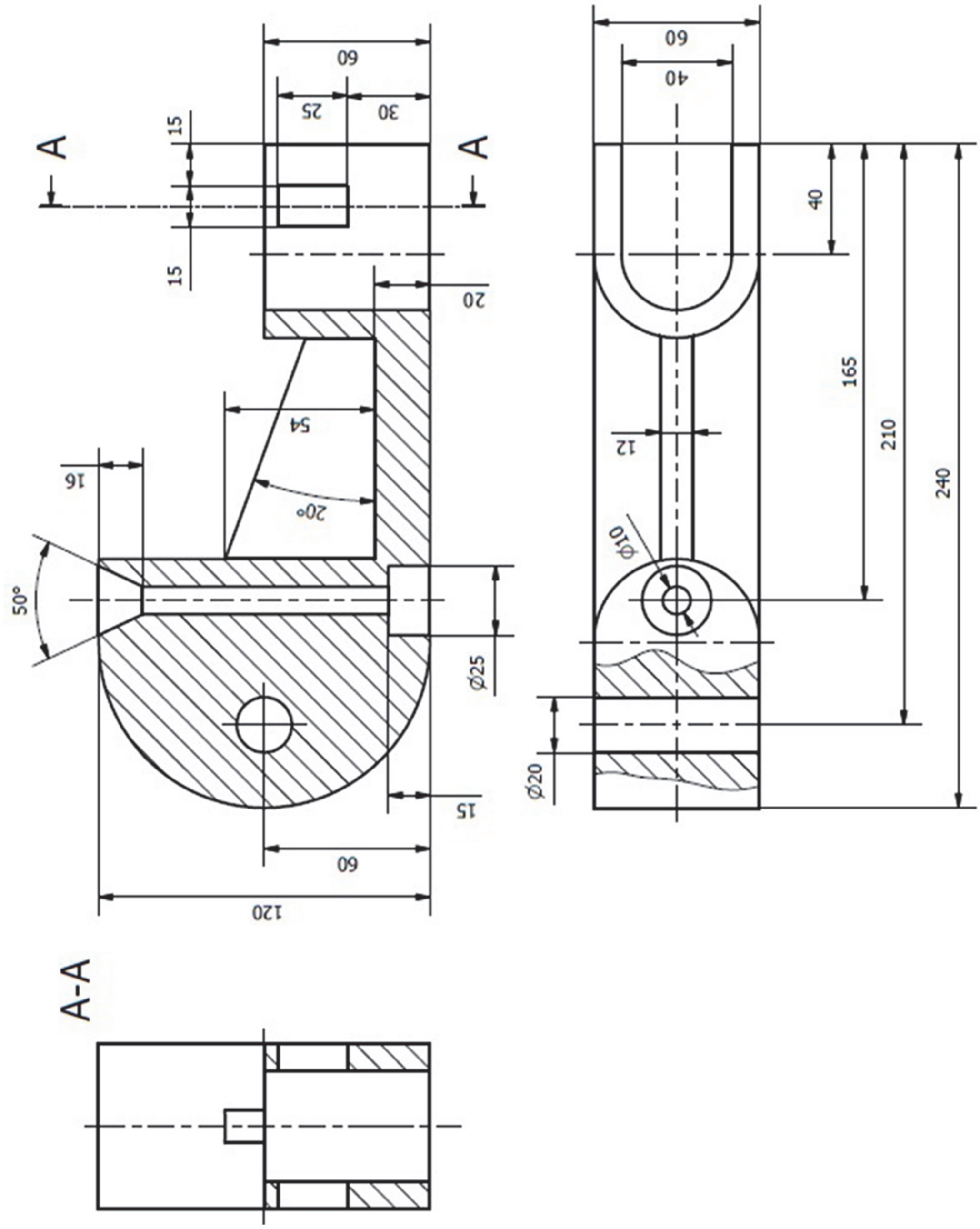

e2
Ø25
Ø10
16
54
12
e1
40
60
Front View
Reinforcement rib


240
210
Ø20
60
e1
120
15
15
15
25
20
30
Ø25
165
40
60



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15. Self-assessment- Rubric

Below is a rubric to help you assess your readiness for the Inventor© exam

This template has been prepared as a simplified version of how 3D CAD models are evaluated.

Minimum criteria to know if I model well. The criteria are organised into blocks: a block associated to the basic criteria, a block associated to the modeling operations, and a block associated to the quality of the sketches. The criteria are weighted differently . If you fill in the salmon-colored boxes in this template per exercise (YES: meets the criteria; NO: does not meet the criteria) and the TOTAL score is >= 8, you should be ready to pass the CAD3D part.

	BASIC CRITERIA		OPERATIONS	SKETCHES			
	Criteria 1 (10%)	Criteria 2 (10%)	Criteria 3 (20%)	Criteria 4 (20%)	Criteria 5 (20%)	Criteria 6 (20%)	
	Orientation and position of the part	Full 3D geometry / Organisation of the model tree	Operations/Functions well performed	Sketch geometry	Sketch dimensioning	Sketch constraints	TOTAL
I know how to model?							0
YES>= 8							
	Description:	**Description:**	**Description:**	**Description:**	**Description:**	**Description:**	
	The part has been positioned in the according to the directions given (on the correct plane, centred in the origin planes, ...).	The part is completely modelled. In addition, the model tree has the main operations at the beginning and the finishing operations at the end (roundings, chamfers, ribs, ...).	Extent of operations (up to surface, next pass-through, ...), Nerves, holes (with seats, with metric ISO thread, ...)	Entities projected as construction. Only those belonging to the final geometry are solid.	Must be the same as in the directions of the perspective provided.	Modify some dimensions slightly and check if it maintains symmetry or tangencies or ...	

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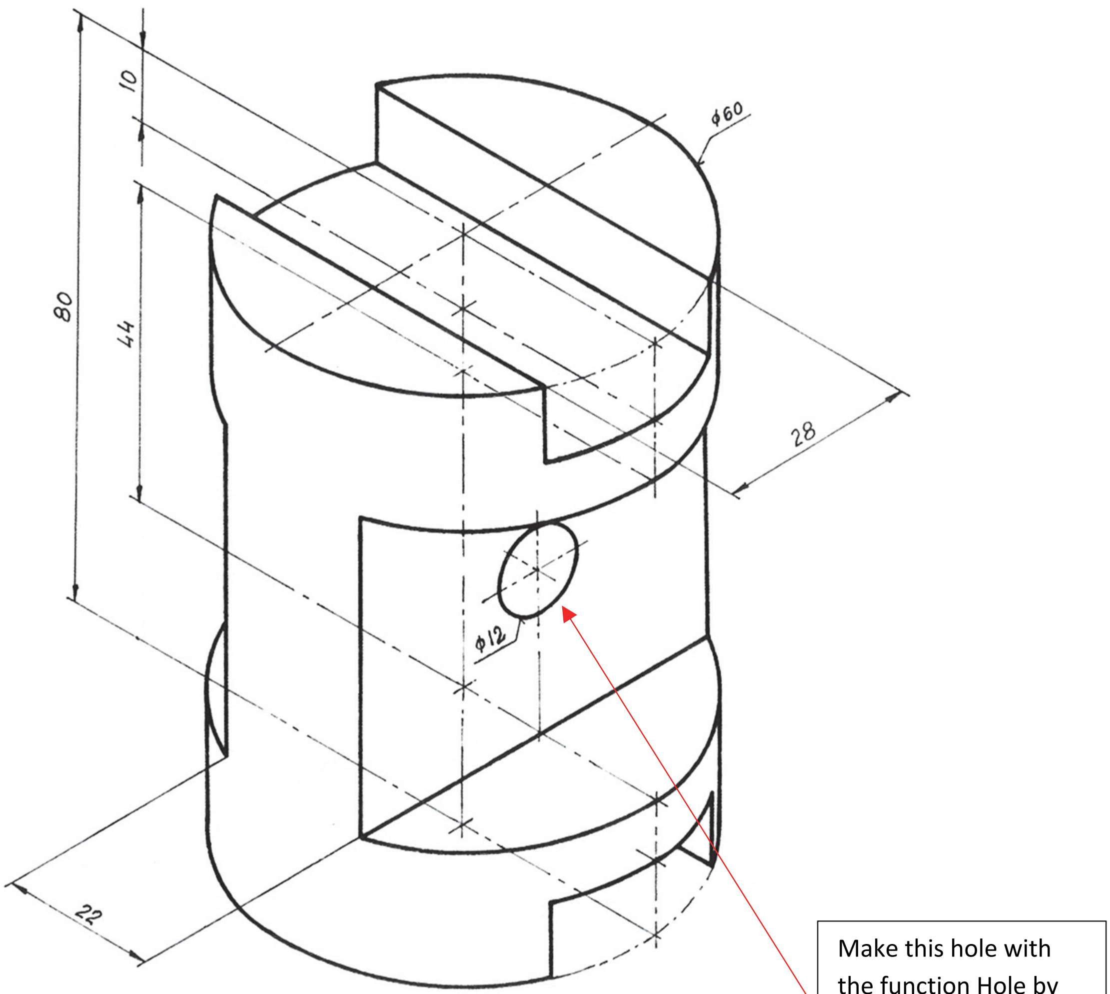
16. Annex I: Other exercises with Inventor ©

This appendix offers practical exercises to practice in Autodesk Inventor©. It includes sketch creation, geometric constraints, and advanced part modelling activities.

EXERCISE 1



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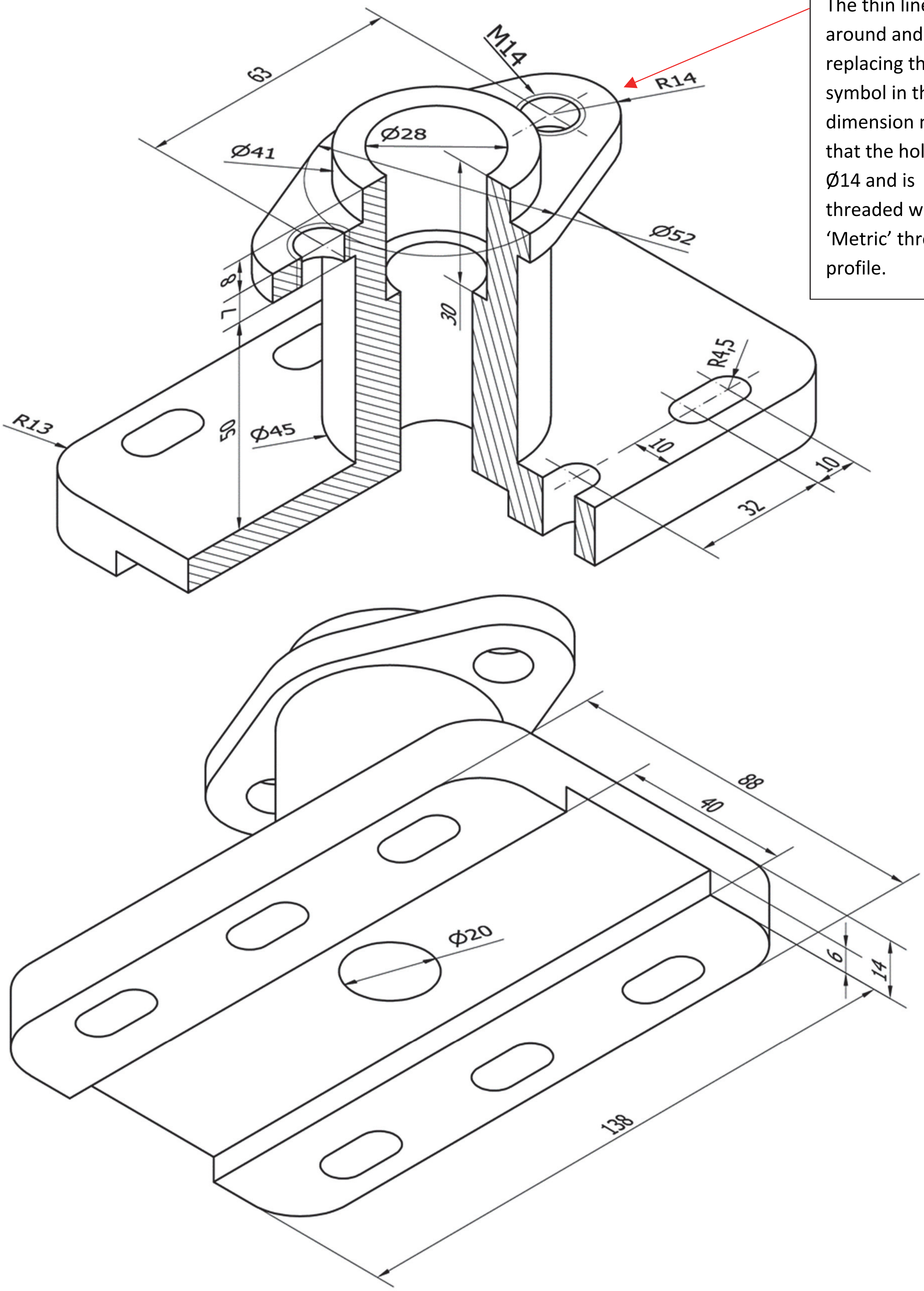
EXERCISE 2

The thin line around and the 'M' replacing the 'Ø' symbol in the dimension means that the hole is Ø14 and is threaded with a 'Metric' thread profile.

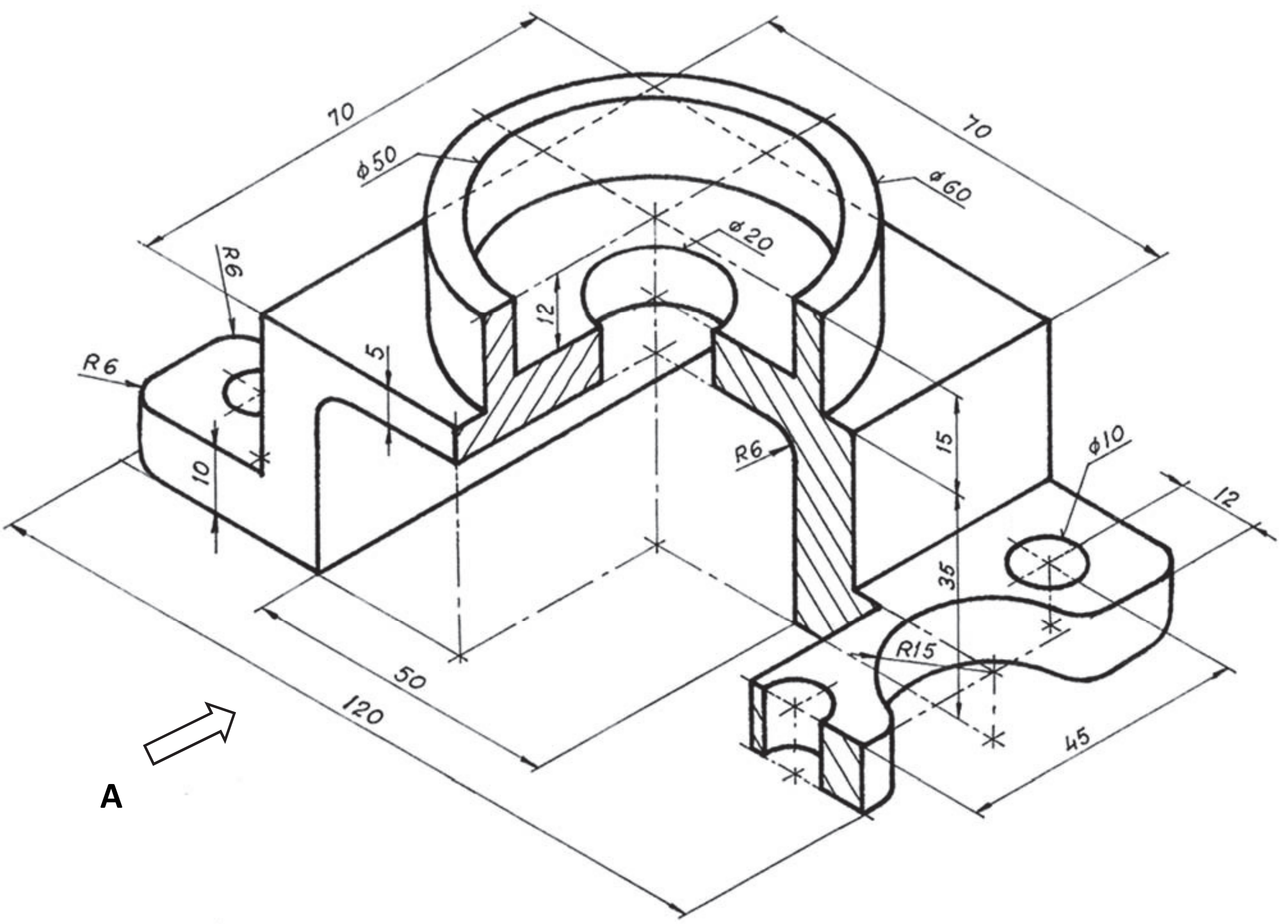


17. Annex II: Other exercises with AutoCAD©

EXERCISE 1

Exercise: Draw the following part in AutoCAD© using two-dimensioned views (you will be penalised if you do not follow the norm) using the European System and without hidden lines, using the AutoCAD© layers functionality.

1. Views
   - ❑ Draw the front view according to direction A, made with a half cut (hatching on the right side) and with a local cut (which allows you to see the other holes of the part).
   - ❑ The top view should have a local cut for the holes. (in the left half of the Front View).
   - ❑ Draw the top view.
2. Clarifications
   - ❑ The part has a plane of symmetry, and all the holes are through holes.
   - ❑ Create a presentation in A4 format (vertical), at a 1:1 scale ((use the title block provided).
3. Files should be submitted in P:\GITI or P:\GIE\Folder_Student
   - ❑ PDF File (1stSurname_2ndSurname_Name.pdf)
   - ❑ DWG File (1stSurname_2ndSurname_Name.dwg)



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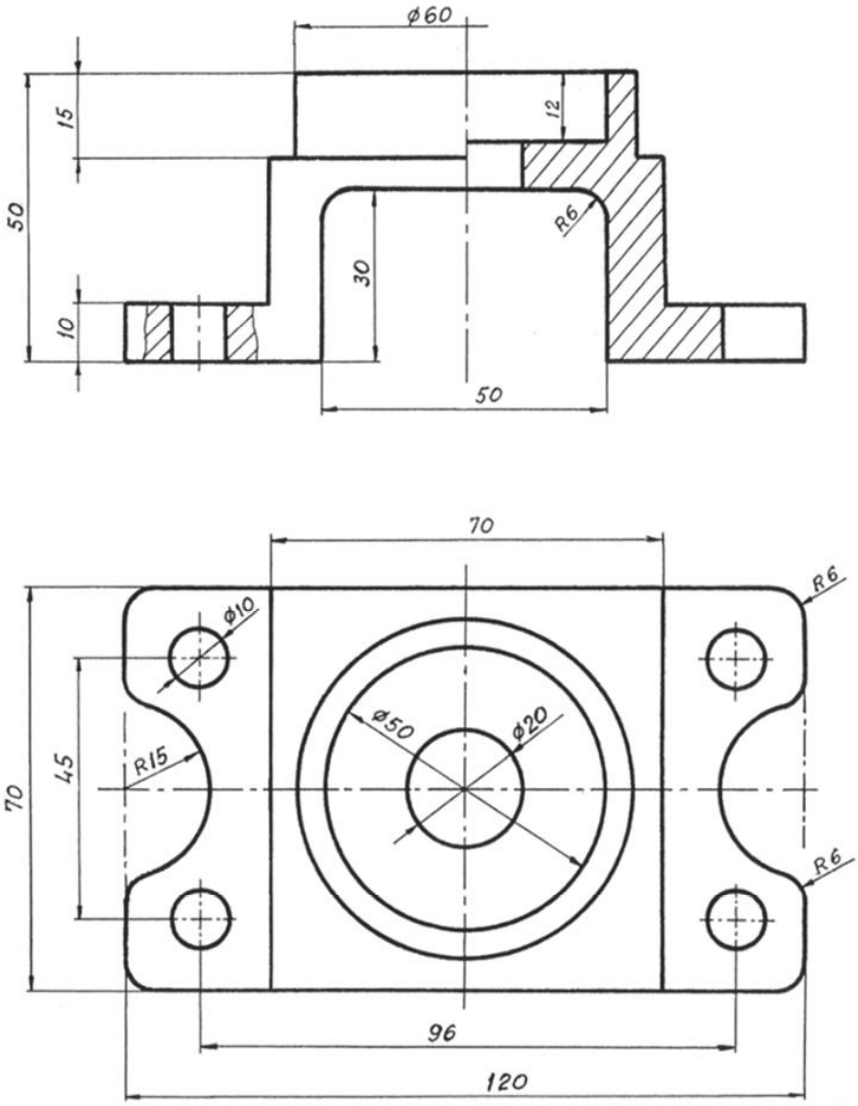

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	mm	Apellidos, Nombre:

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EXERCISE 2

Exercise: Draw the following part in AutoCAD© using 3 dimensioned views (you will be penalised if you do not follow the norm) using the European System and without hidden lines, using the AutoCAD© layers functionality.

1. Views
   - ❑ Draw the front view according to direction A, made with a local cut (which allows you to see the interior of the hole called "B").
   - ❑ Draw the top view. The top view should be cut by a total plane parallel to the projection that passes through the axis of the hole called "C".
   - ❑ Draw the left view. The left view should have a total cut to define the elements called "D". The plane should pass through the axes of the two holes with cylindrical countersinks with Ø7 and Ø10 mm diameters.
2. Clarifications
   - ❑ The part has a plane of symmetry, and all the holes are through holes.
   - ❑ Create a presentation in A4 format (horizontal) at a 2:3 scale (use the title block provided).
3. Files should be submitted in P:\GITI or P:\GIE\Folder_Student
   - ❑ PDF File (1stSurname_2ndSurname_Name.pdf)
   - ❑ DWG File (1stSurname_2ndSurname_Name.dwg)

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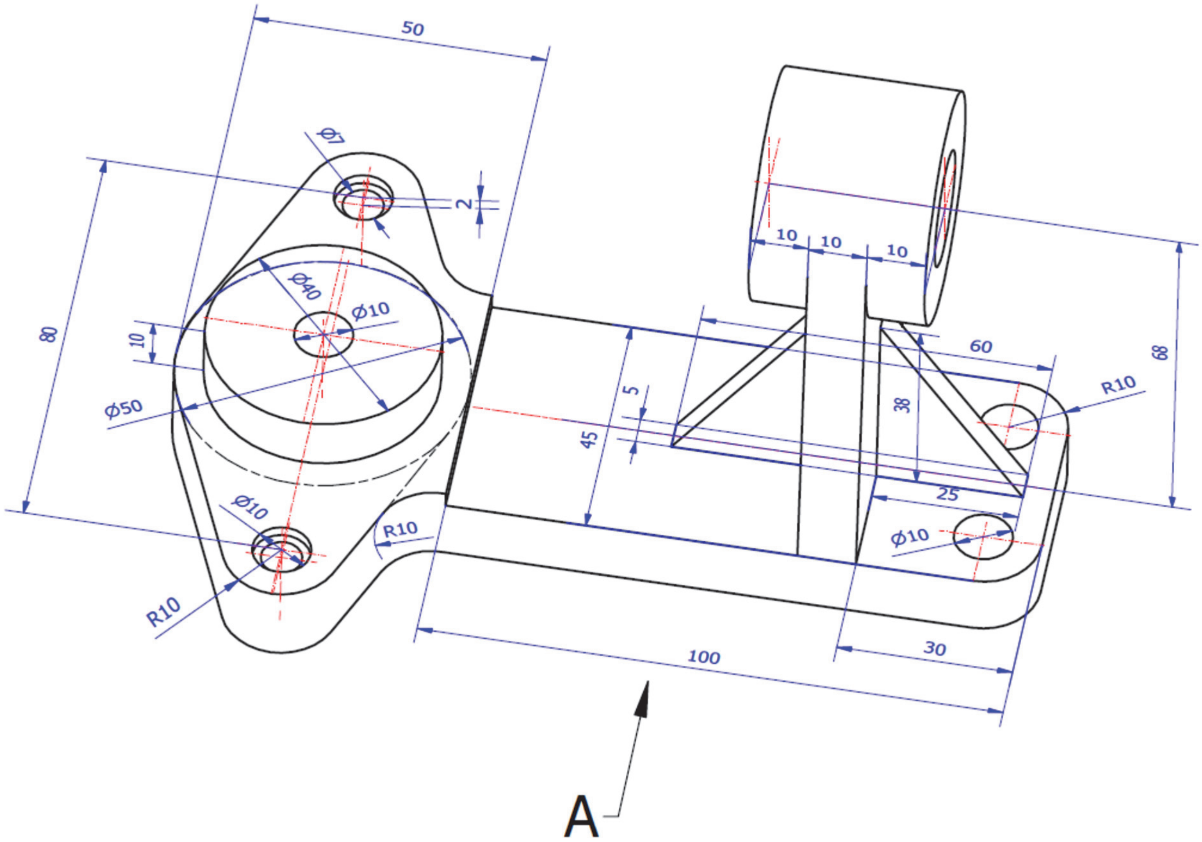
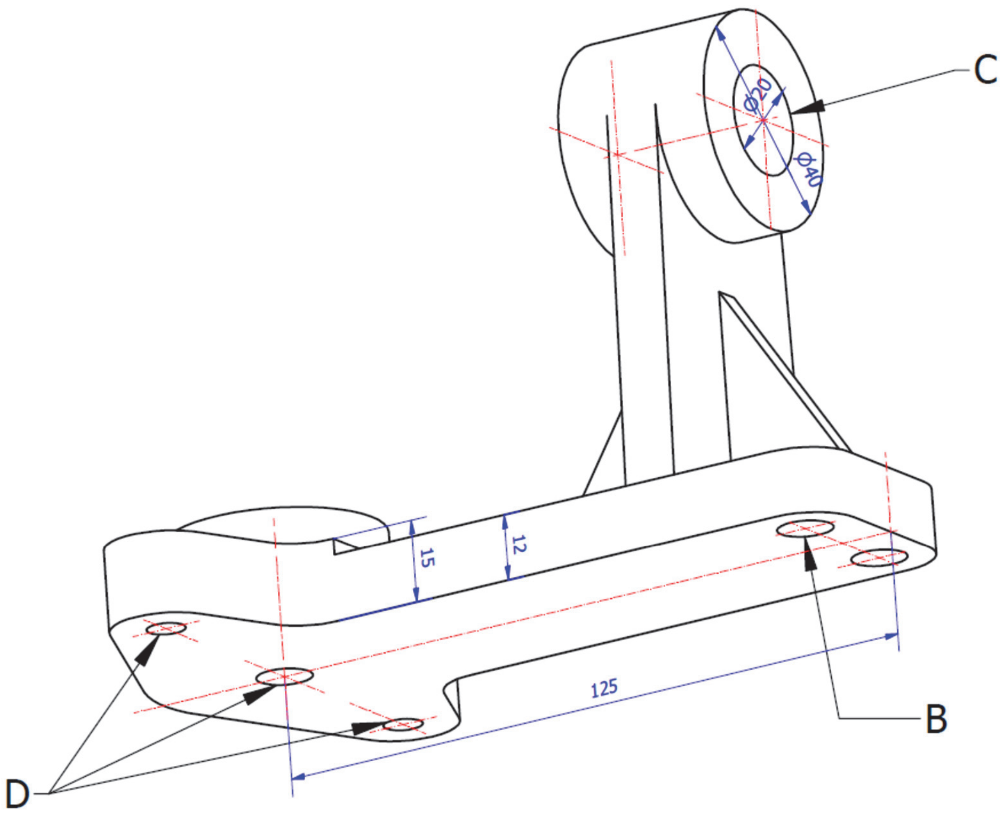
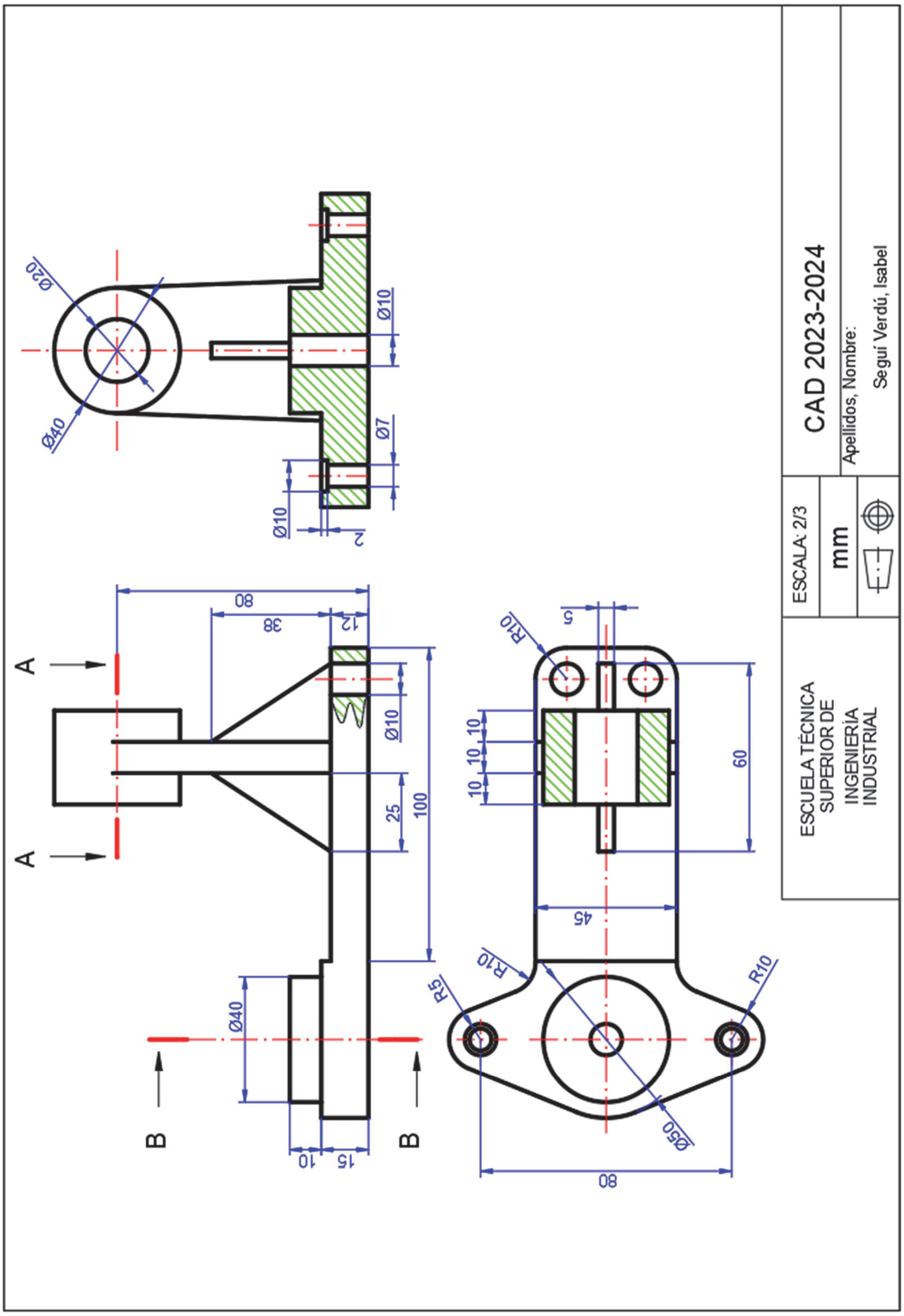

50
Ø7
2
80
10
Ø40
Ø10
Ø50
Ø10
R10
R10
10
10
10
60
R10
68
5
45
38
25
Ø10
100
30
A


C
Ø20
Ø40
15
12
125
B
D



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