CATIA V5R16

- Welcome to **CATIA** (**Computer Aided Three Dimensional Interactive Application**).
- As a new user of this software package, you will be joining hands with thousands of users of this high-end CAD/CAM/CAE tool worldwide.
- If you are already familiar with the previous releases, you can upgrade your designing skills with the tremendous improvement in this latest release.
- CATIA V5, developed by Dassault Systemes, France, is a completely re-engineered, next-generation family of CAD/CAM/CAE software solutions for Product Lifecycle Management.
- Through its exceptionally easy-to-use state of the art user interface, CATIA V5 delivers innovative technologies, for maximum productivity and creativity from the concept to the final product.
- CATIA V5 reduces the learning curve for the user, as it allows the flexibility of using | feature based and parametric designs.

- CATIA V5 serves the basic design tasks by providing different workbenches.
- A workbench is defined as a specified environment consisting of a set of tools, which allow the user to perform the specific design tasks in a particular area.
- The basic workbenches available in CATIA V5 are :

• Part Design Workbench

The **Part Design** workbench is a parametric and feature-based environment, in which you can create solid models.

• Wireframe and Surface Design Workbench

- The **Wireframe and Surface Design** workbench is also a parametric and feature-based environment, in which you can create wireframe or surface models.
- The tools in this workbench are similar to those in the **Part Design** workbench, with the only difference that the tools in this environment are used to create basic and advanced surfaces.

- **Assembly Design Workbench**
  - The **Assembly Design** workbench is used to assemble the components using the assembly constraints available in this workbench.
  - There are two types of assembly design approaches:
- Bottom-up
- Top-down
- **Drafting Workbench**
  - The **Drafting** workbench is used for the documentation of the parts or the assemblies created earlier in the form of drawing views and their detailing.
  - There are two types of drafting techniques:
- Generative drafting
- Interactive drafting

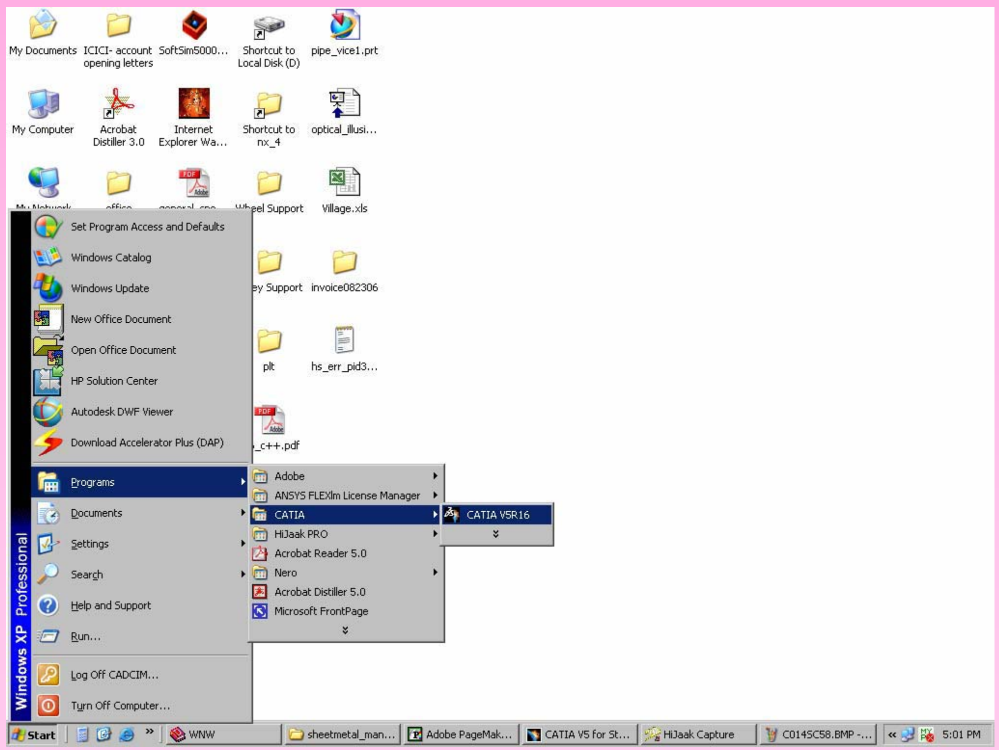
SYSTEM REQUIREMENTS

The following are the system requirements to ensure smooth running of CATIA V5R16 on your system:

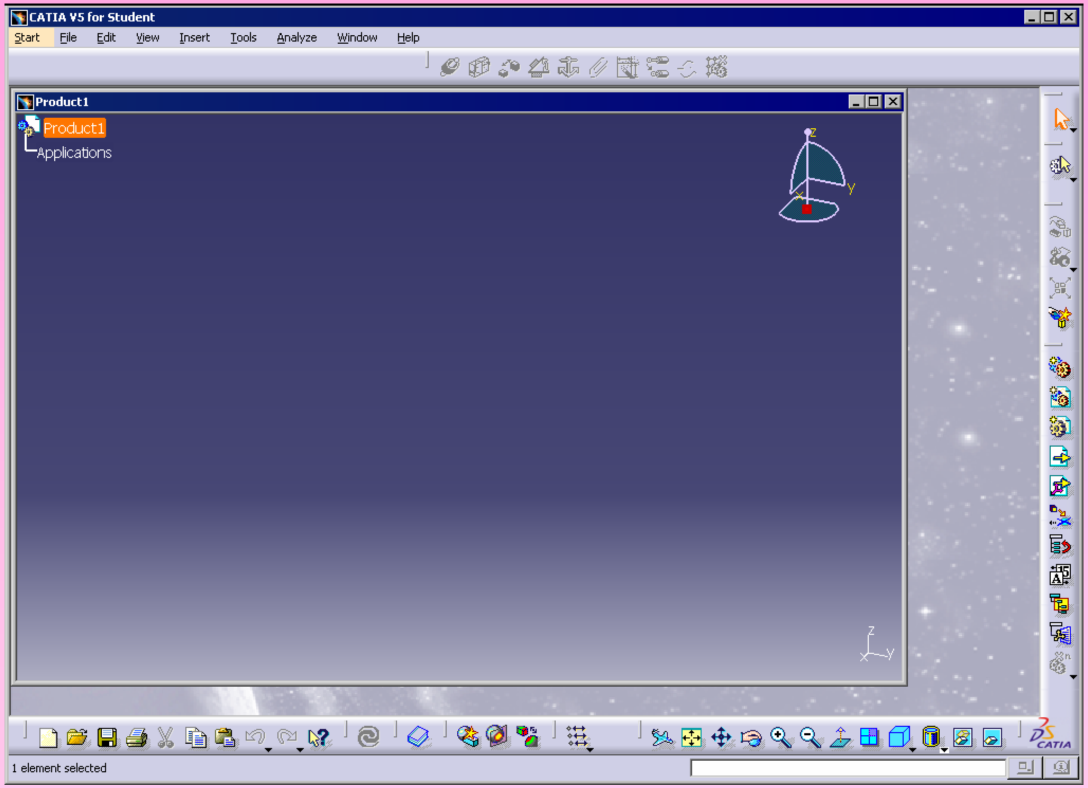
- System unit: An Intel Pentium III or Pentium 4 based workstation running Microsoft 2000 Professional Edition or Windows XP Professional Edition.
- Memory: 256 MB of RAM is the minimum recommended for all applications. 512 MB of RAM is recommended for DMU applications.
- Disk drive: 4 GB Disk Drive space (Minimum recommended size)
- Internal/External drives: A CD-ROM drive is required for program installation.
- Display: A graphic color display compatible with the selected platform-specific graphic adapter. The minimum recommended monitor size is 17 inches.
- Graphics adapter: A graphics adapter with a 3D OpenGL accelerator is required with minimum resolution of 1024x768 for Microsoft Windows workstations and 1280x1024 for UNIX workstations.

GETTING STARTED WITH CATIA V5R16

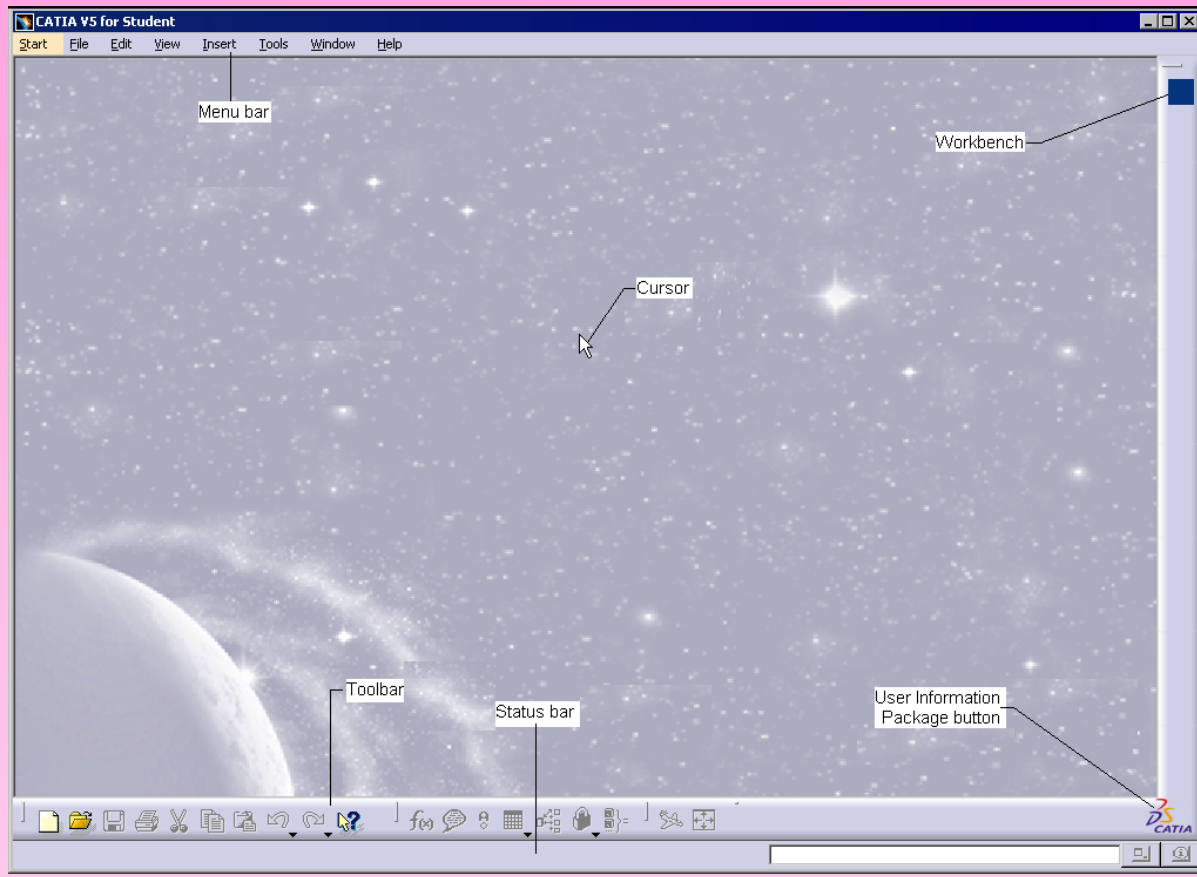
- Install CATIA V5R16 on your system and then start it by double-clicking on the shortcut icon of **CATIA V5R16** on the desktop of your computer.
- You can also choose **Start > Programs > CATIA > CATIA V5R16** from the taskbar menu, as shown in the figure.



Starting CATIA V5R16 using the taskbar shortcuts



The initial screen that appears after starting CATIA V5R16



The screen that appears after closing the initial product file

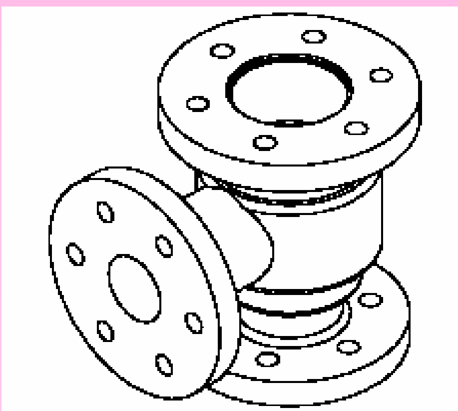
IMPORTANT TERMS AND DEFINITIONS

Feature-based Modeling

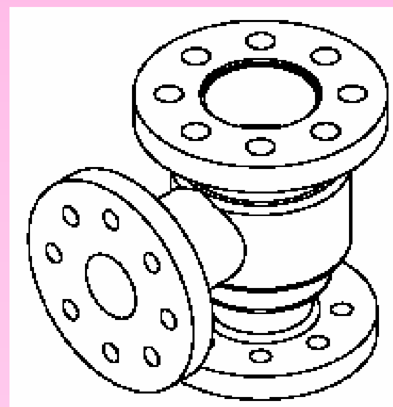
- A feature is defined as the smallest building block that can be modified individually.
- A model created in CATIA V5 is a combination of a number of individual features and each feature is related to the other directly or indirectly.

Parametric Modeling

The parametric nature of a software package is defined as its ability to use the standard properties or parameters in defining the shape and size of a geometry.



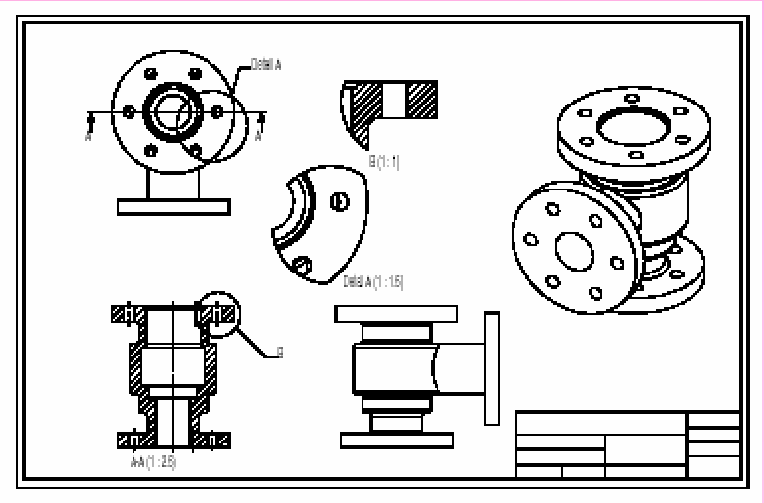
Body of a pipe housing



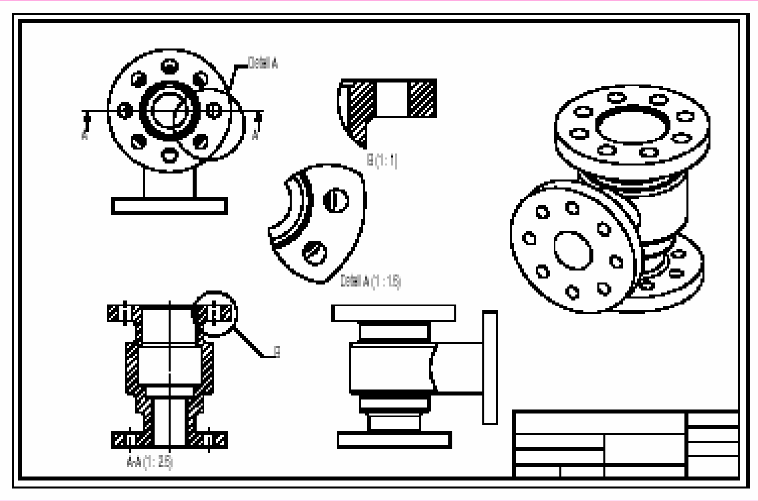
Modified body of pipe housing

• Bidirectional Associativity

The bidirectional associativity ensures that if any modification is made in the model in any one of the workbenches of CATIA V5, it is automatically reflected in the other workbenches immediately.



The drawing views of the body part before making the modifications



The drawing views, after modifications

- **CATPart**

CATPart is a file extension associated with all the files that are created in **Sketcher**, **Part Design**, and **Wireframe** and **Surface Design** workbenches of CATIA V5.

- **CATProduct**

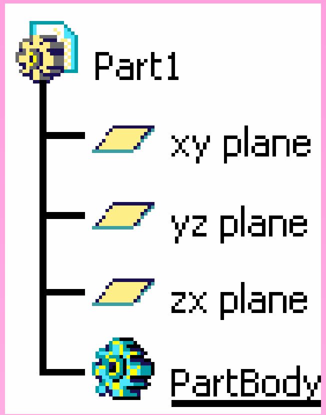
CATProduct is a file extension associated with all the files that are created in **Assembly Design** workbench of CATIA V5.

- **CATDrawing**

CATDrawing is a file extension associated with all the files that are created in **Drafting** workbench of CATIA V5.

Specification Tree

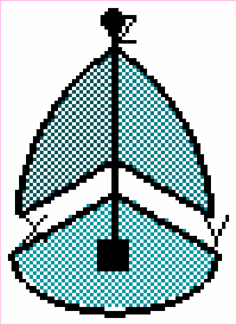
- The specification tree keeps a track of all the operations that are carried on the part, as shown in the figure.
- The specification tree that appears when you start a new file under the **Part Design** workbench, is as shown in the figure.



The specification tree that appears on starting a new CATPart file

Compass

- It is a tool that is used to manipulate the orientation of parts, assemblies, or sketches.
- You can also orient the view of the parts and assemblies.
- By default, it appears on the top right corner of the geometry area.



The Compass

- **Constraints**
  - Constraints are logical operations that are performed on the selected element to define its size and location with respect to other elements or reference geometries.
  - The constraints in **Sketcher** workbench are called geometric constraints and the constraints available in the **Assembly Design** workbench are called assembly constraints.
- **Geometric Constraints**

  These are the logical operations performed on sketched elements to define their size and position with respect to other elements.

The constraints available in the **Sketcher** workbench are:

- **Distance**
- **Length**
- **Angle**
- **Radius / Diameter**
- **Semimajor axis**
- **Semiminor axis**
- **Symmetry**
- **Midpoint**
- **Equidistant point**
- **Fix**
- **Coincident**
- **Concentricity**
- **Tangency**
- **Parallelism**
- **Perpendicular**
- **Horizontal**
- **Vertical**

- **Assembly Constraints**

  Constraints available in the **Assembly Design** workbench are logical operations performed to restrict the degree of freedom of the component and to precisely define their location and position with respect to other components of the assembly.

- **Coincidence Constraint**
- **Contact Constraint**
- **Offset Constraint**
- **Angle Constraint**
- **Fix Component**
- **Fix Together**
- **Quick Constraint**

- **PartBody**
  - It is the default body available under **Part Design** workbench.
  - All the solid related features, such as pad, pocket, shaft, and so on are placed inside it.

- **Geometrical Set**

  The geometrical set is defined as a body that includes newly created planes, surfaces, wireframe elements, and reference elements.

- **Wireframe**

  The wireframe construction elements aid in creating surfaces and are used as a substitute to entities drawn in the **Sketcher** workbench.

- **Surface**
  - Surface are geometric feature which have no thickness.
  - They are generally used to create complex shapes that are difficult to create using the solid feature.

- **Feature**
  - A features is defined as a basic building block of a solid model.
  - The combination of various features results in a complete model.

- **Reframe on**

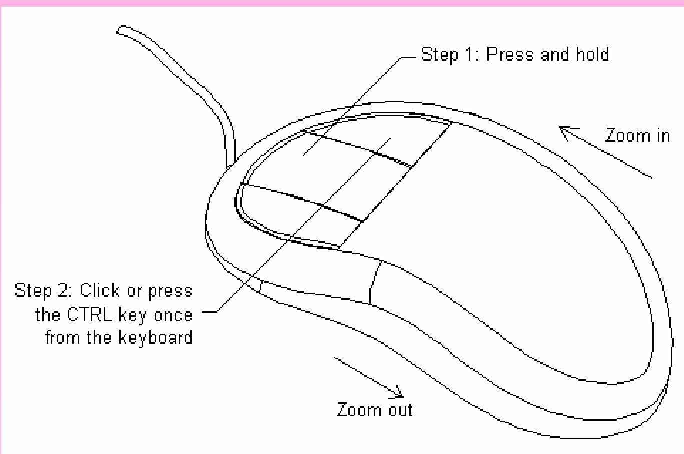
  Sometimes a feature, body, or a sketch may not be visible in the available space in the geometry area, this operation is used to view that particular selection in the available display space.

- **Center Graph**

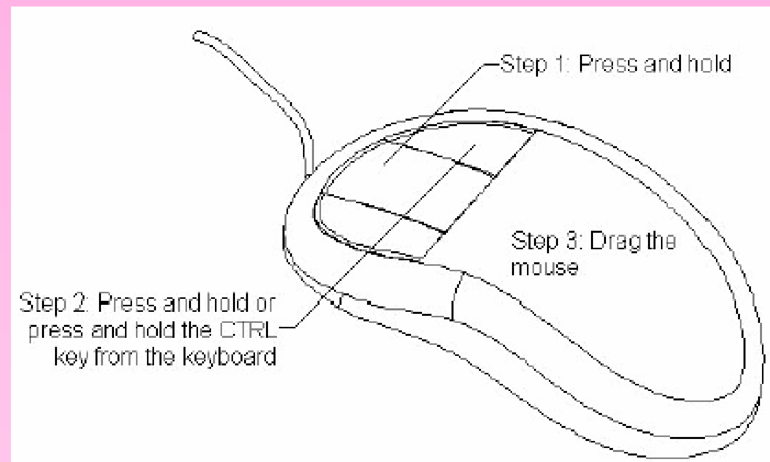
  This option brings the selected feature, body, or sketch in the specification tree to the middle left portion of the geometry area.

➢ UNDERSTANDING THE FUNCTIONS OF THE MOUSE BUTTONS

- To work with CATIA V5 design workbenches, it is necessary that you understand the functions of the mouse buttons.
- The efficient usage of these three buttons, along with the CTRL key on the keyboard, can reduce the time involved in completing the design task.



Using the three button mouse to perform the zoom in and zoom out operations



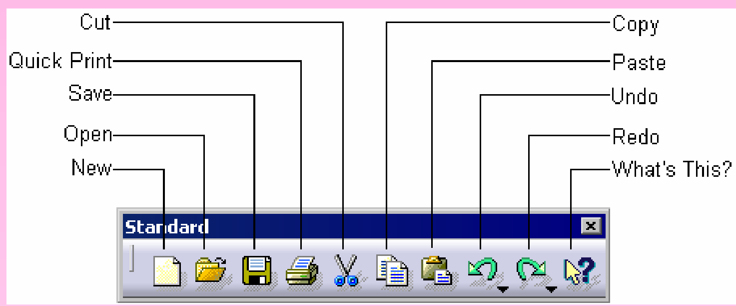
Using the three button mouse to perform the rotate operation

TOOLBARS

- CATIA V5 offers a user-friendly design environment by providing specific toolbars to each workbench.
- The toolbars that appear in various workbenches are:

Standard Toolbar

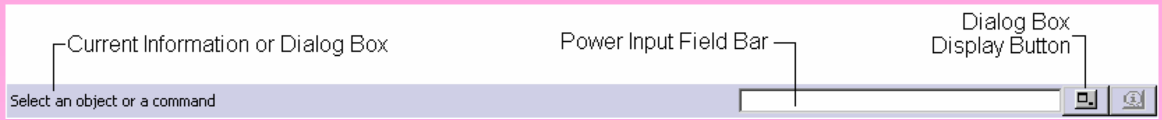
- This toolbar is common to all workbenches of CATIA V5.
- The following figure shows the Standard toolbar.



The Standard toolbar

• Status Bar

The **Status** bar, which appears at the bottom of the CATIA V5 window, comprises of three areas, as shown in the figure.



The status Bar

• Current Information or Dialog Box

The **Current Information or Dialog Box** area displays the current information about the selected feature or current tool.

• Power Input Field Bar

The **Power Input Field** bar lets you invoke the commands and enter the data or value that can be directly associated with the feature.

- **Dialog Box Display Button**

Choosing the **Dialog Box Display** button will turn on and turn off the display of the current dialog box.

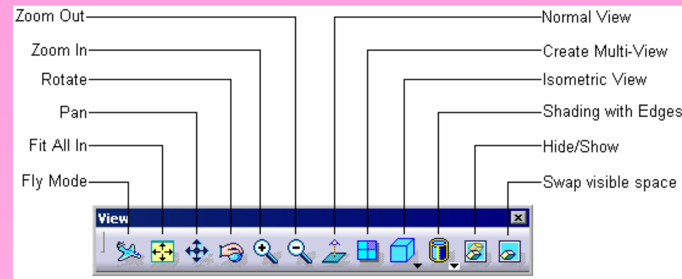
- **User Information Package Button**

Choosing this button will open a window with a default link that is C:\Program Files\Dassault Systemes\B16\intel_a\resources\galaxy\default.htm.

- **Part Design Workbench Toolbars**

You can invoke the **Part Design** workbench by choosing the **New** button from the **Standard** toolbar and selecting **Part** from the **New** dialog box.

- View Toolbar

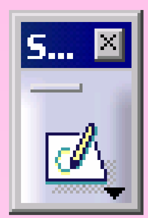


The View toolbar

- Select Toolbar

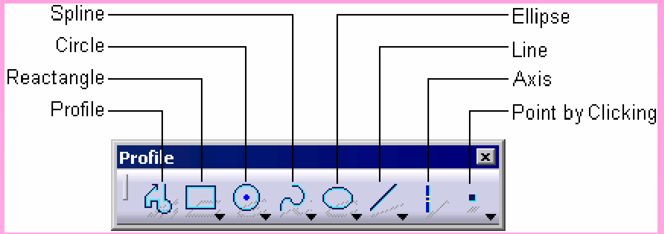
The Select toolbar

- Sketcher Toolbar



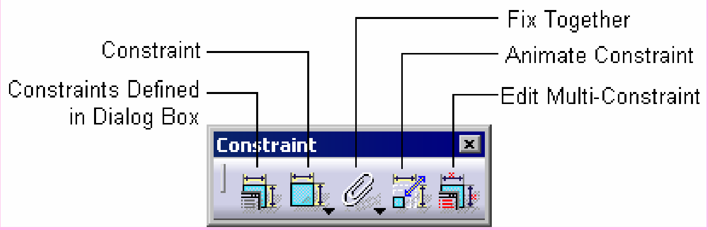
The Sketcher toolbar

- Profile Toolbar



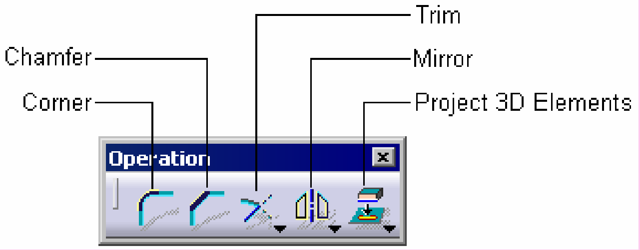
The Profile toolbar

- Constraint Toolbar



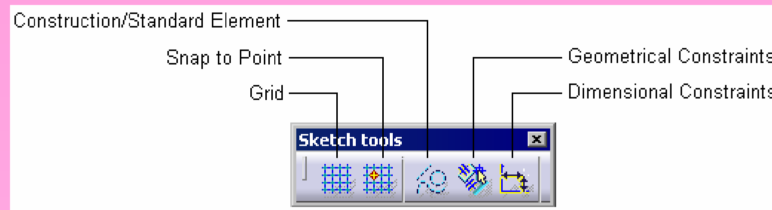
The Constraint toolbar

- Operation Toolbar



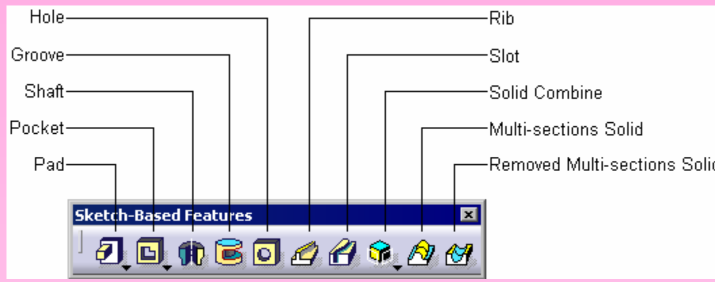
The Operation toolbar

- **Sketch tools Toolbar**



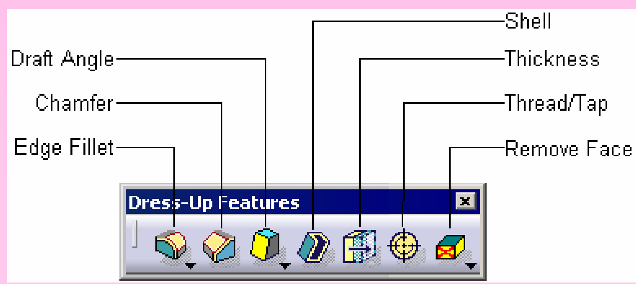
The Sketch tools toolbar

- **Sketch-Based Features Toolbar**



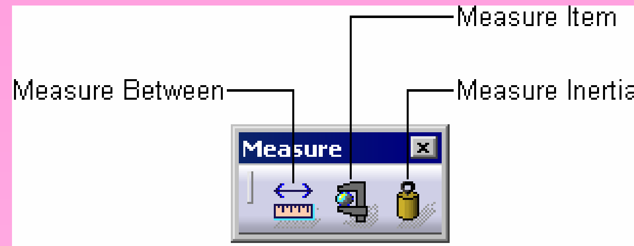
The Sketch-Based Features toolbar

- **Dress-Up Features Toolbar**



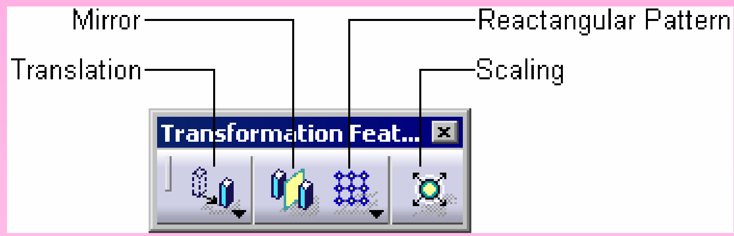
The Dress-Up Features toolbar

- Measure Toolbar



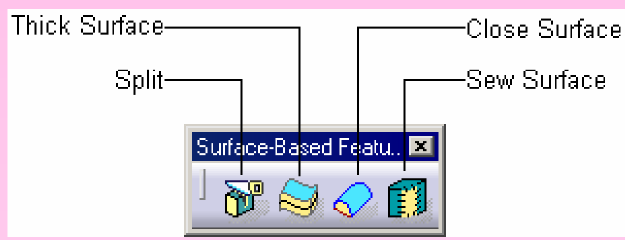
The Measure toolbar

- Transformation Features Toolbar



The Transformation Features toolbar

- Surface-Based Features Toolbar



The Surface-Based Features toolbar

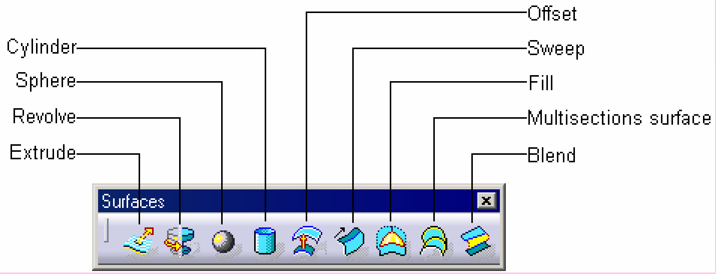
- **Apply Material Toolbar**

The Apply Material toolbar

- **Wireframe and Surface Design Workbench Toolbars**

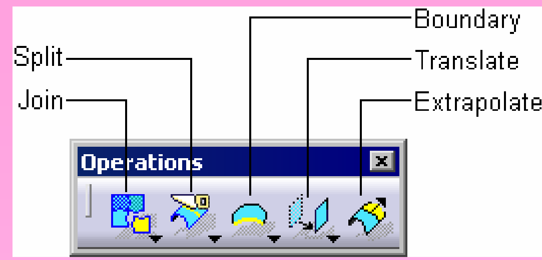
You can invoke the **Wireframe and Surface Design** workbench from the main menu bar by choosing **Start > Mechanical Design > Wireframe and Surface Design**.

- **Surfaces Toolbar**



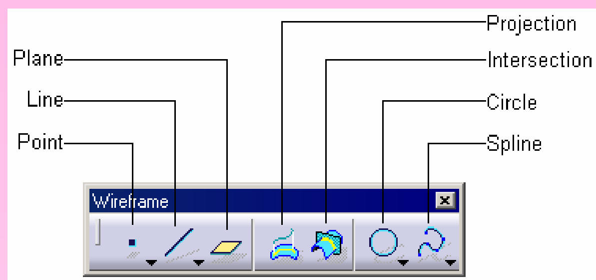
The Surfaces toolbar

- Operations Toolbar



The Operations toolbar

- Wireframe Toolbar

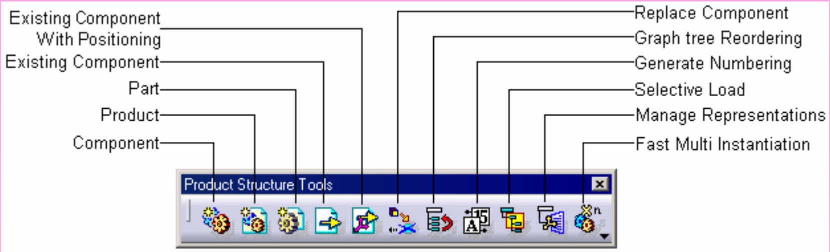


The Wireframe toolbar

- **Assembly Design Workbench Toolbars**

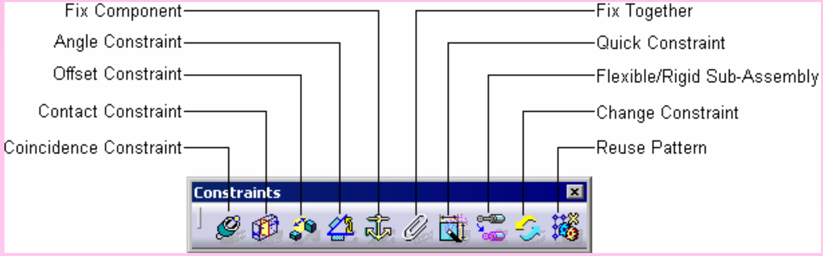
  You can invoke the **Assembly Design** workbench by choosing the **New** button from the **Standard** toolbar and selecting **Product** from the **New** dialog box.

- **Product Structure Tools Toolbar**



The Product Structure Tools toolbar

- **Constraints Toolbar**

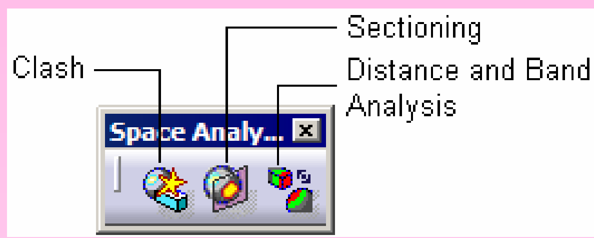


The Constraints toolbar

- Move Toolbar

The Move toolbar

- Space Analysis Toolbar

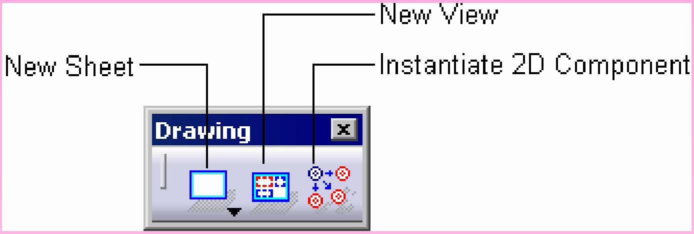


The Space Analysis toolbar

- **Drafting Workbench Toolbars**

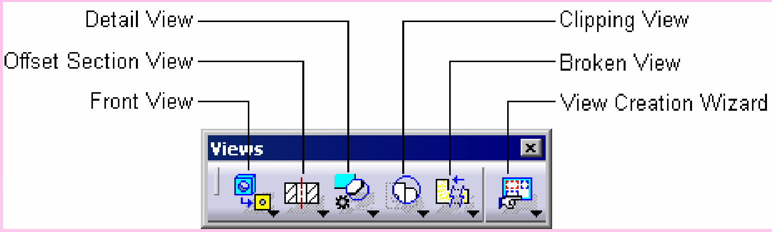
To invoke the **Drafting** workbench, choose the **New** button from the **Standard** toolbar and select **Drawing** from the **New** dialog box.

- **Drawing Toolbar**



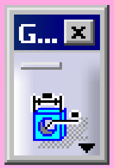
The Drawing toolbar

- **Views Toolbar**

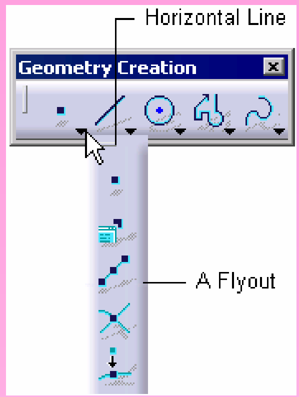


The Views toolbar

- Generation Toolbar



The Generation toolbar



The flyout that appears when a down arrow is chosen

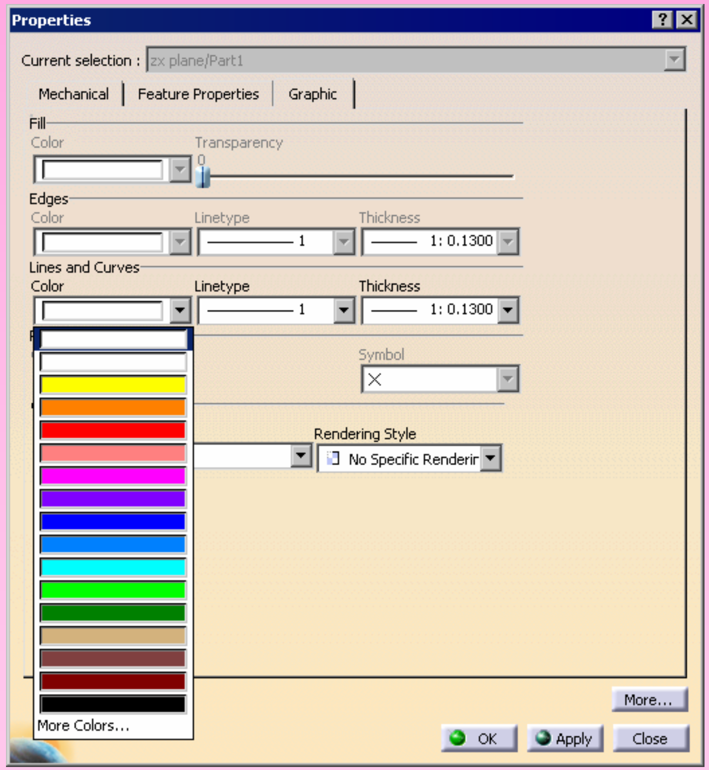
HOT KEYS

- CATIA V5 is more popularly known for its icon driven structure.
- The hot keys, along with their functions, are listed in the table shown.

Hot Keys	Function
CTRL+Z	Invokes the **Undo** tool
CTRL+Y	Invokes the **Repeat** tool
CTRL+S	Saves the current document
ALT+ENTER	Invokes the **Properties** tool
CTRL+F	Invokes the **Search** tool
CTRL+U	Invokes the **Update** tool
SHIFT+F2	Invokes the **Specification Overview** tool
F3	Toggles the display of the specification tree
SHIFT+F1	Invokes the **What's This?** tool
F1	Invokes the CATIA V5 **Help** tool
CTRL+D	Invokes the **Fast MultiInstantiation** tool in the **Assembly Design** workbench
CTRL+E	Invokes the **Define MultiInstantiation** tool in the **Assembly Design** workbench

➢ COLOR SCHEME

CATIA allows you to use various color schemes as the background screen color, and also for displaying the entities on the screen.



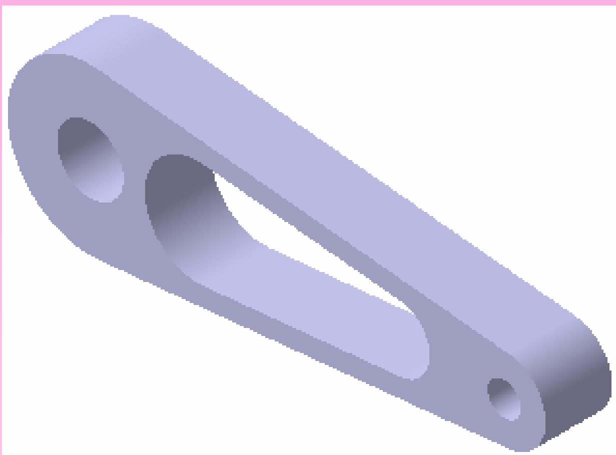
The **Properties** dialog box

Learning Objectives:

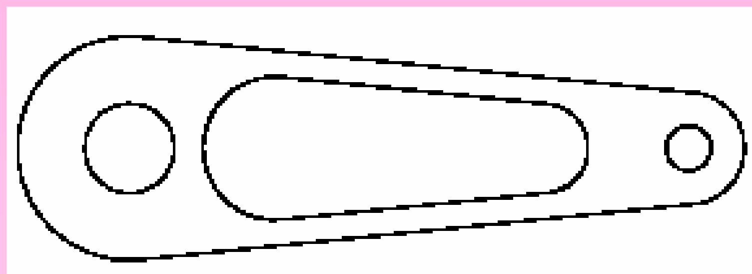
- *Understand the Sketcher workbench of CATIA V5.*
- *Start a new file in the Part workbench and invoke the Sketcher . workbench within it.*
- *Set up the Sketcher workbench.*
- *Understand some important Sketcher terms.*
- *Draw sketches using some of the tools in the Sketcher workbench.*
- *Use some of the drawing display tools.*

THE SKETCHER WORKBENCH

- Most of the components designed using CATIA V5 are a combination of sketched features, placed features, and derived features.
- The placed features are created without drawing a sketch, whereas the sketched features require a sketch that defines the shape of the sketched feature.



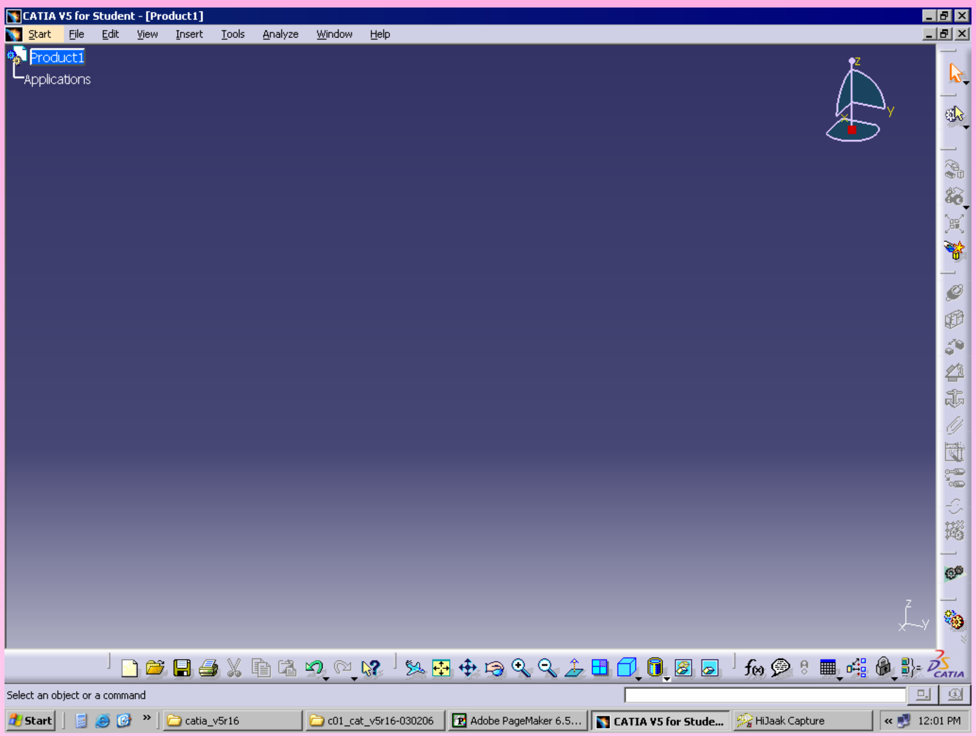
Solid Model of a Link



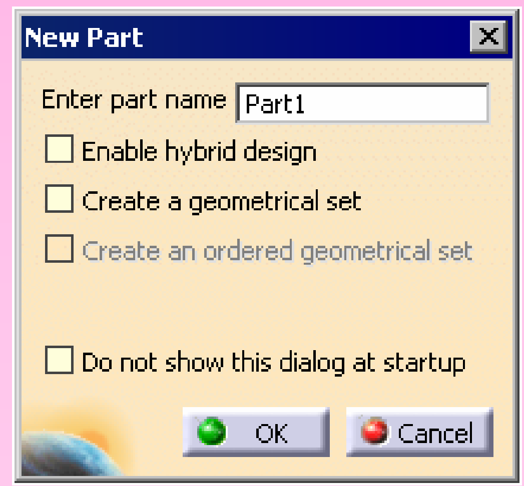
Base sketch for the solid model

STARTING A NEW FILE

- When you start CATIA V5R16, a new **Product** file with the name **Product1** is displayed on the screen, as shown in the figure.
- Choose **Start > Mechanical Design > Part Design** to make sure that you are in the **Part Design** workbench.
- The **New Part** dialog box is displayed, as shown in the figure.

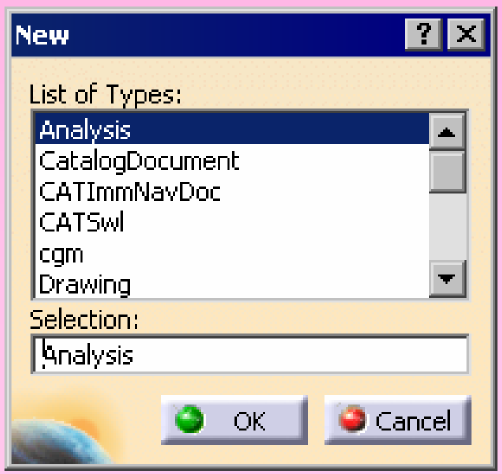


Initial screen that appears after starting CATIA V5R16

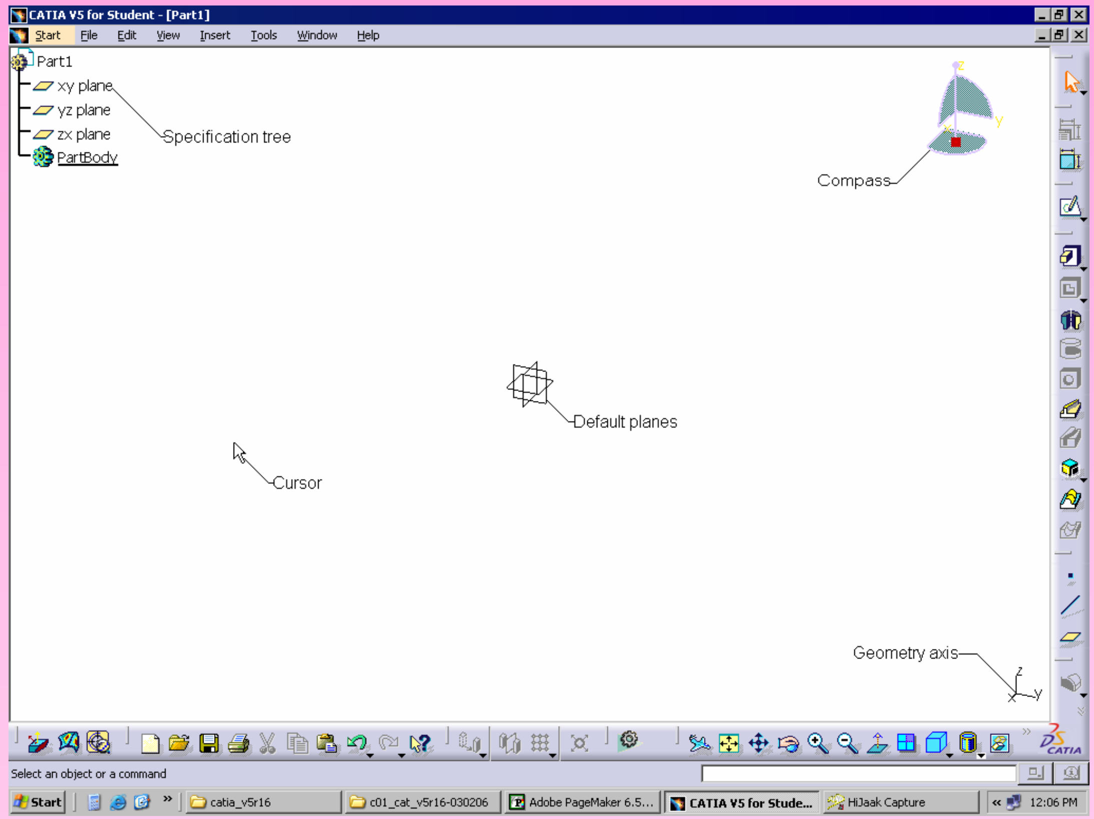


The **New Part** dialog box

- Alternatively, you can choose the **File > New** option from the menu bar; the **New** dialog box is displayed, as shown in the figure.
- A new file in the **Part Design** workbench is displayed on the screen, as shown in the figure.



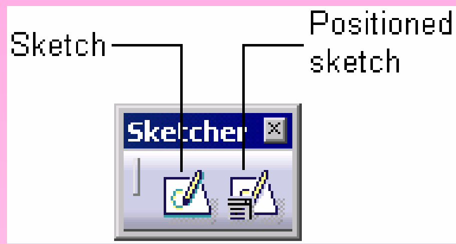
The New dialog box



A new Part Design workbench file

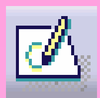
INVOKING THE SKETCHER WORKBENCH

- To invoke the **Sketcher** workbench, choose the down arrow on the right of the **Sketch** button in the **Sketcher** toolbar; a flyout will appear.
- The **Sketcher** flyout as an independent toolbar, is as shown in the figure.

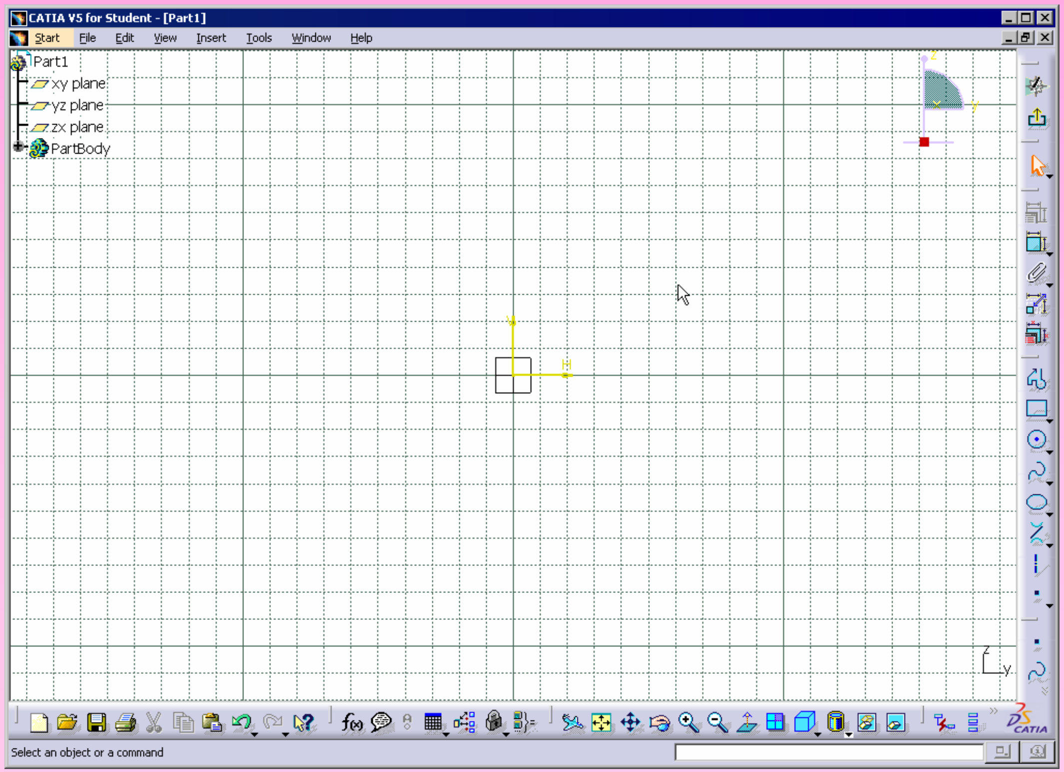


The Sketcher toolbar

Invoking the Sketcher Workbench using the Sketch Button



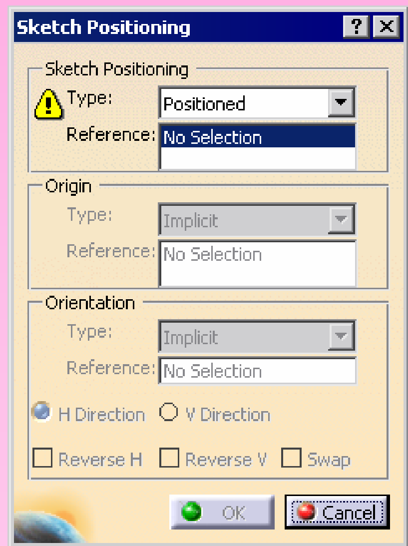
- To invoke the **Sketcher** workbench using is method, choose the **Sketch** button from the **Sketcher** toolbar.
- The **Sketcher** workbench that appears after on selecting the yz plane as the sketching plane, is shown in the figure.



The Sketcher workbench invoked using the yz plane as the sketching plane

• Invoking the Sketcher Workbench using the Positioned Sketch Button

- To invoke the **Sketcher** workbench using this option, choose the **Positioned Sketch** button from the **Sketcher** toolbar.
- The **Sketch Positioning** dialog box will be displayed, as shown in the figure.



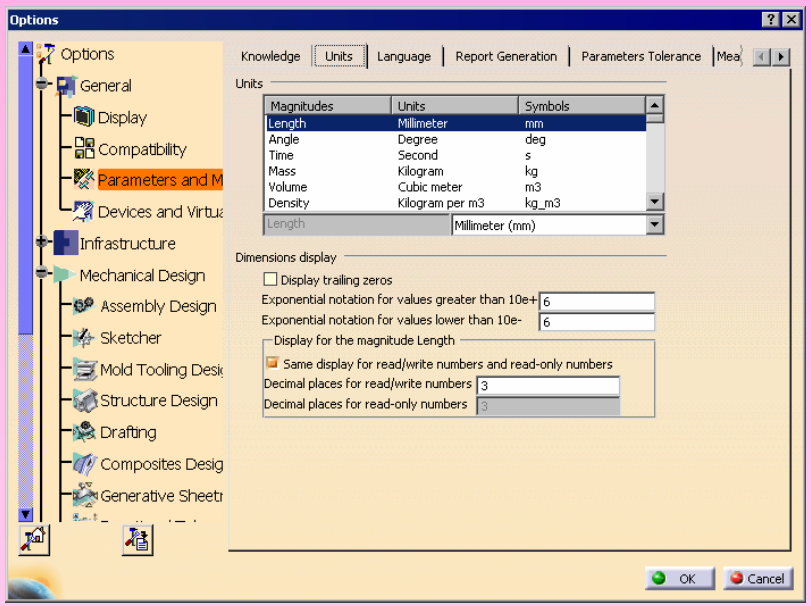
The Sketch Positioning dialog box

SETTING UP THE SKETCHER WORKBENCH

- After invoking the **Sketcher** workbench, you need to set the workbench as per the sketching or drawing requirements.
- These requirements include modifying units, grid settings, and so on.

Modifying Units

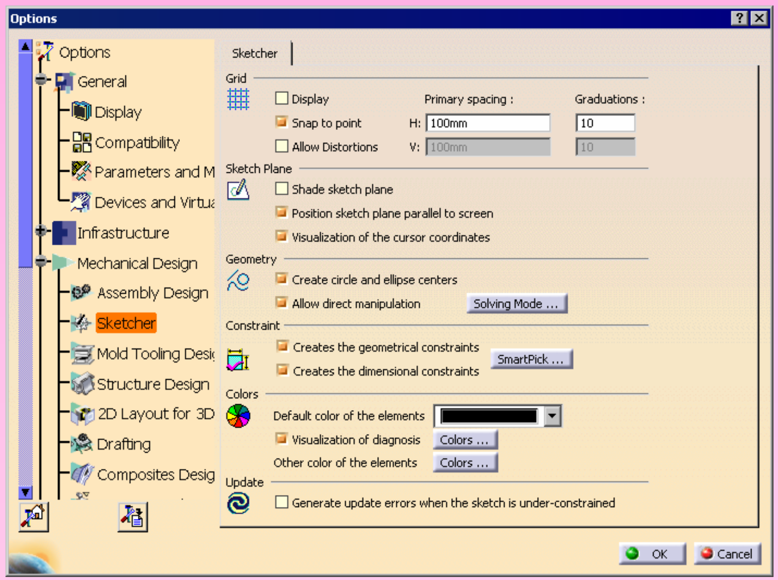
- The **Options** dialog box after invoking the **Units** tab, is shown in the figure.



The Options dialog box with the Units tab chosen

• Modifying the Grid Settings

- To change the values of **Primary Spacing** and **Graduations**, choose **Tools > Options** from the menu bar; the **Options** dialog box will be displayed.
- Choose the **Mechanical Design** option from the tree on the left of the dialog box.
- Next, choose the **Sketcher** option to display the **Sketcher** tab on the right of the **Options** dialog box, as shown in the figure.



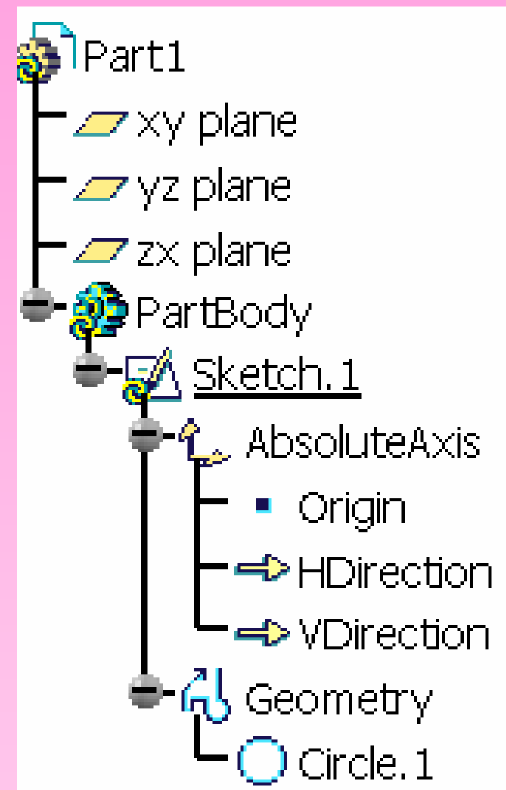
The Options dialog box with the Sketcher option selected

UNDERSTANDING THE SKETCHER TERMS

- **Specification Tree**

  The various levels under **Sketch.1** in the specification tree are:

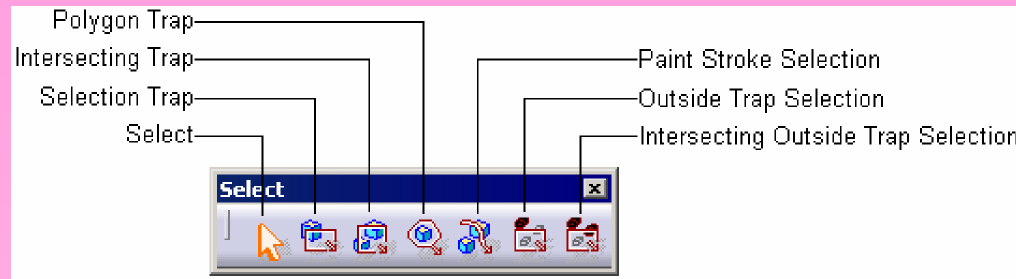
- **AbsoluteAxis**
  - **Origin**
  - **Hdirection**
  - **VDirection**
- **Snap to Point**
- **Construction/Standard Element**



The expanded form of the specification tree

- **Select Toolbar**

The Select toolbar

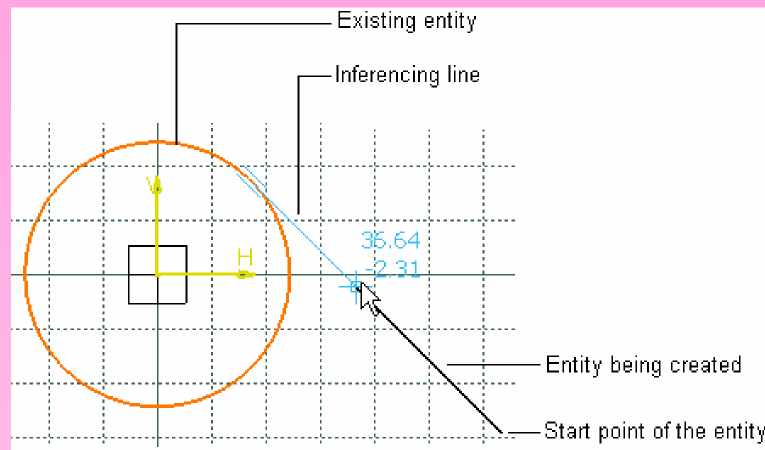


The Select toolbar

The tools available in **Select** toolbar are:

- **Select**
- **Selection Trap**
- **Intersecting Trap**
- **Polygon Trap**
- **Paint Stroke Selection**
- **Outside Trap Selection**
- **Intersecting Outside Trap Selection**

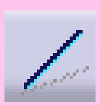
- **Inferencing Lines**
  - The inferencing lines are temporary lines that are used to track a particular point on the screen.



An example of an inferencing line

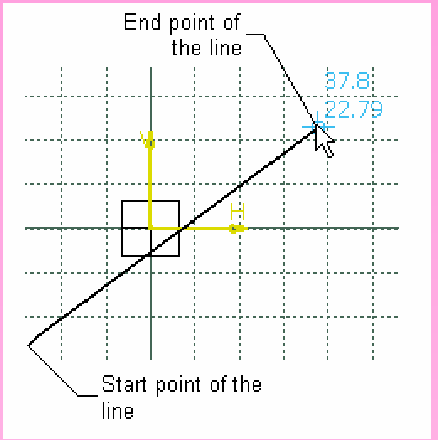
DRAWING SKETCHES USING THE SKETCHER TOOLS

- **Drawing Lines**



CATIA being parametric in nature, allows the user to draw a line of any length and at any angle, and then change it to the desired length and angle.

- **Drawing Lines by Specifying Points in the Geometry Area**

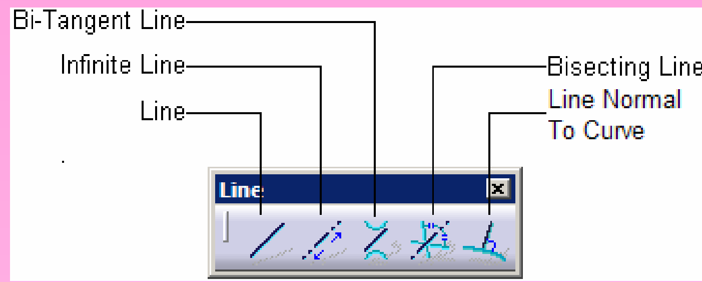


A line drawn by selecting the start and endpoints from the geometry area

- **Drawing Lines Using the Sketch tools Toolbar**

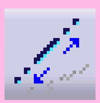
The expanded form of the Sketch tools toolbar after invoking the Line tool

- **Drawing Lines by entering the Start and End point values**
- **Drawing Lines with a Symmetrical Extension**



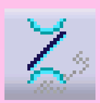
The Line toolbar

- **Drawing Infinite Lines**



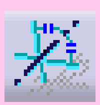
To draw an infinite line, invoke the **Infinite Line** tool from the **Line** toolbar.

- **Drawing Bi-Tangent Lines**



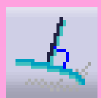
Bi-Tangent lines are the lines that are tangent to two circles, arcs, ellipses, conics, or any curved geometry.

- **Drawing Bisecting Lines**



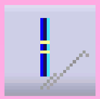
Bisecting lines are the lines that pass through two intersecting lines such that the angle formed between them is divided equally.

- **Drawing Lines Normal To Curve**

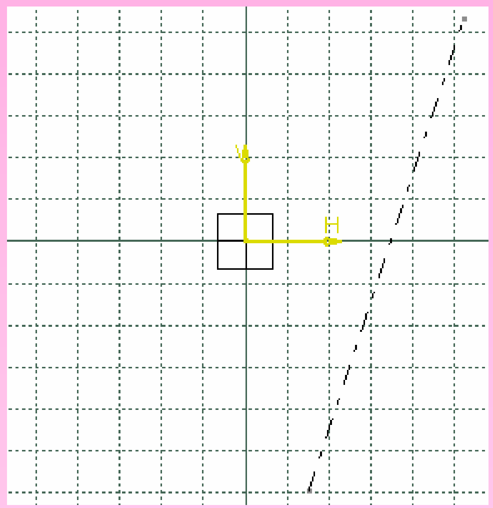


To draw a line normal to a curve, choose the **Line Normal To Curve** button from the **Line** toolbar.

- **Drawing Center Lines**

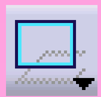


- You can draw a center line in CATIA using the **Axis** tool.
- Generally, this tool is used to create the axis for the revolved feature.



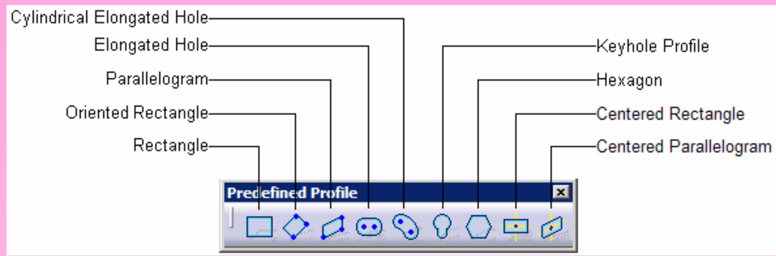
An axis drawn in the geometry area

- **Drawing Rectangles, Oriented Rectangles, and Parallelograms**



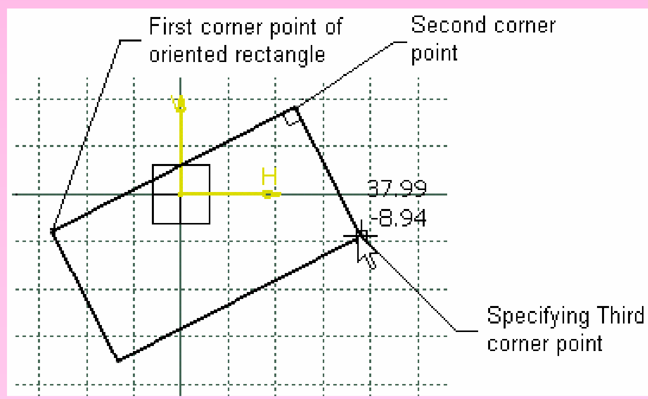
Rectangle is a basic geometry that comprises of four sides.

- **Drawing Rectangles**



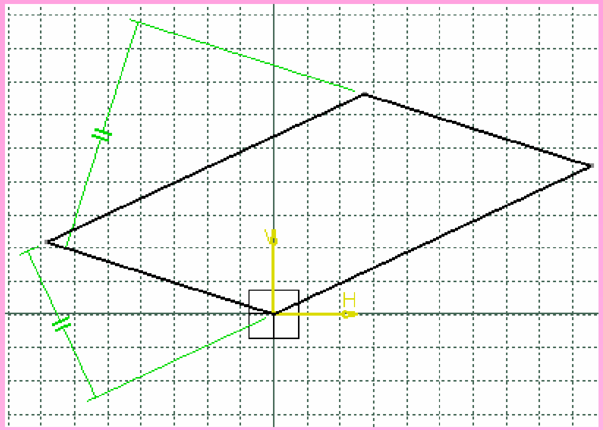
The Predefined Profile toolbar

- **Drawing Oriented Rectangles**



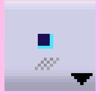
Selecting the corner points to draw an oriented rectangle

- **Drawing Parallelograms**

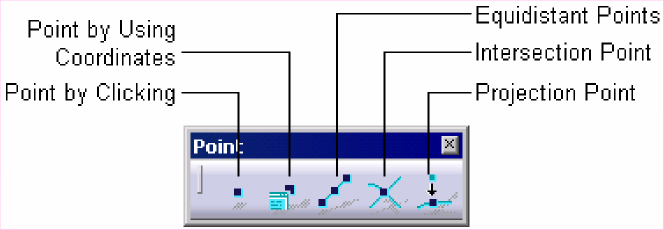


A parallelogram drawn by specifying the corner points

- **Drawing Points**

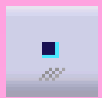


A point is defined as the geometrical element that has no magnitude, length, width, or thickness.



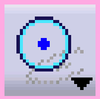
The Point toolbar

- **Drawing Points by Clicking**

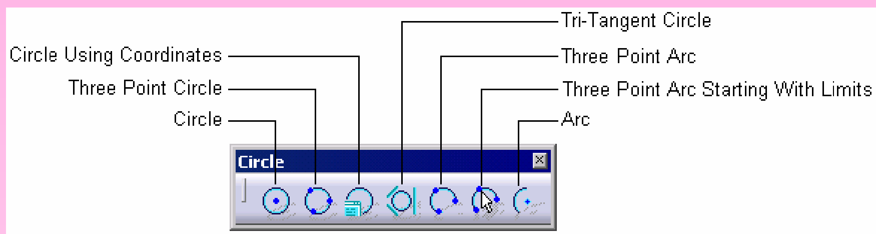


To draw points by clicking, invoke the **Point by Clicking** tool from the **Point** toolbar; the **Sketch** tools toolbar expands and you will be prompted to click to create the point.

- **Drawing Circles**

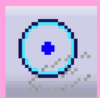


- To draw a circle, choose the down arrow on the right of the **Circle** button in the **Profile** toolbar.
- The **Circle** toolbar is displayed, as shown in the figure.



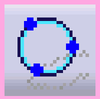
The Circle toolbar

- **Drawing Circles Using the Circle Tool**



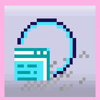
To draw a circle, invoke the **Circle** tool from the **Circle** toolbar.

- **Drawing a Three Point Circle**



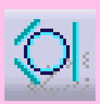
To draw a three point circle, invoke the **Three Point Circle** tool from the **Circle** toolbar; the **Sketch tools** toolbar expands.

- **Drawing Circles using Coordinates**



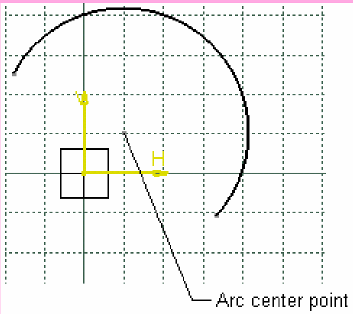
  - Invoke the **Circle Using Coordinates** tool from the **Circle** toolbar; the **Circle Definition** dialog box will be displayed.
  - You can specify the coordinate values of the center point and radius, using the options in this dialog box.

- **Drawing Tri-Tangent Circles**



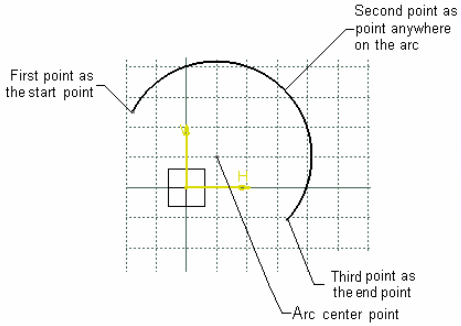
To draw a tri-tangent circle, invoke the **Tri-Tangent Circle** tool from the **Circle** toolbar.

- **Drawing Arcs**
  - An arc is a geometric element that forms a sector of a circle or ellipse.
  - The tools to draw arcs are available in the **Circle** toolbar.
- **Drawing Arcs by Defining the Center Point**



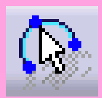
An arc

- **Drawing Three Point Arcs**

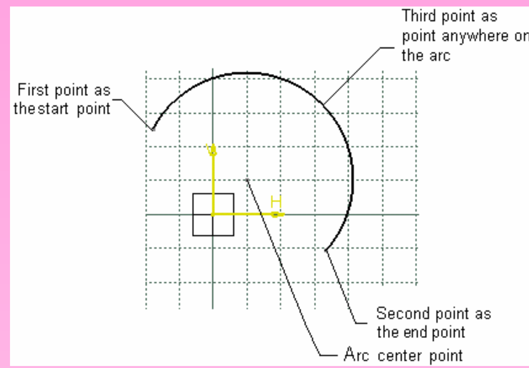


Selecting the points to draw a three point arc

- **Drawing Three Point Arcs Starting With Limits**

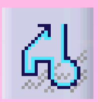


To draw a three point arc starting with limits type of arc, invoke the **Three Point Arc With Limits** tool from the **Circle** toolbar.

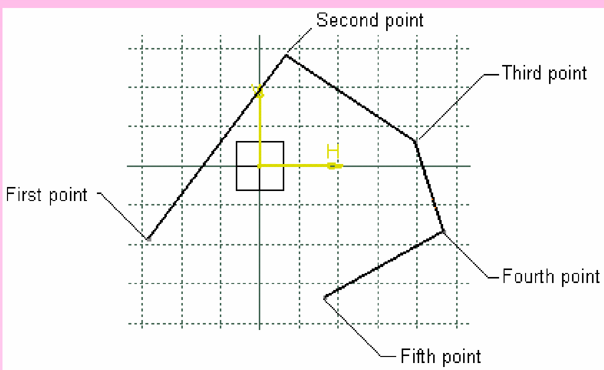


Selecting the points to draw a three point arc starting with limits

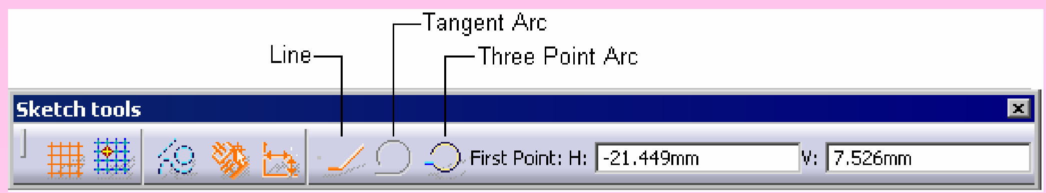
- **Drawing Profiles**



In CATIA, a profile is defined as a combination of continuous lines and arcs.



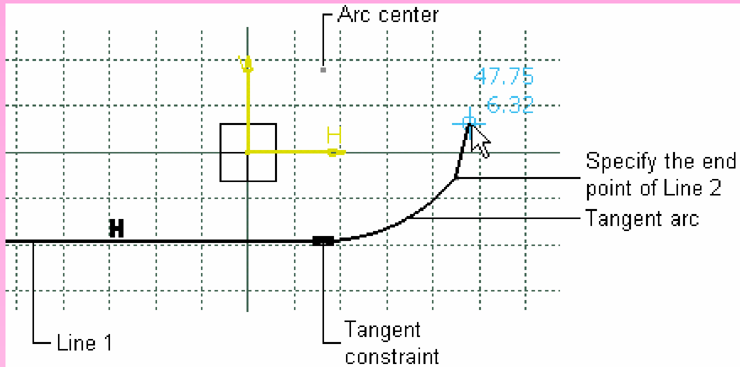
An open profile drawn using the Profile tool



The Sketch tools toolbar

Drawing a Tangent Arc Using the Profile Tool

To draw a tangent arc in continuation with the line, invoke the **Profile** tool from the **Profile** toolbar.



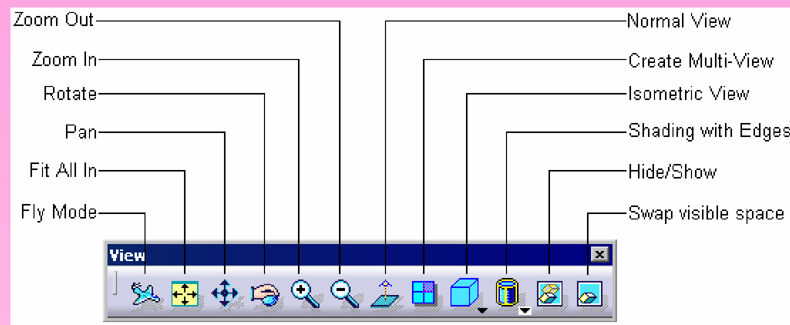
A tangent arc being drawn using the Profile tool

Drawing Three Point Arcs using the Profile Tool

- To draw a three point arc using the **Profile** tool, invoke it from the **Profile** toolbar.
- You will notice the **Three Point Arc** button is available in the **Sketch tools** toolbar.

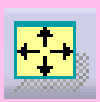
DRAWING DISPLAY TOOLS

The drawing display tools for viewing drawing elements or geometries are available in the **View** toolbar, as shown in the figure.



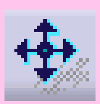
The View toolbar

- ## Fit All In



The **Fit All In** tool is used to increase the geometry area so that all the sketched elements or geometry are included in the visible space.

- ## Pan

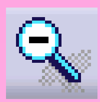


The **Pan** tool is used to drag the current view in the geometry area.

- **Zoom In**

The **Zoom In** tool is used to zoom into the sketches in increments.

- **Zoom Out**

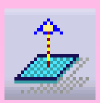


The **Zoom Out** tool is used to zoom out of the sketch in increments.

- **Zoom Area**

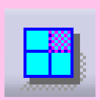
The **Zoom Area** tool is used to define an area, which is to be magnified and viewed in the available geometry area.

- **Normal View**



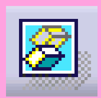
The **Normal View** tool is used to orient the view normal to the sketch plane in the current Sketcher workbench.

- **Splitting the Drawing Area into Multiple Viewports**



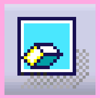
The **Create Multi-View** tool is used to split the drawing area into four viewports.

- **Hiding and Showing Geometric Elements**



To hide a sketcher element, invoke the **Hide/Show** tool by choosing the **Hide/Show** button from the **View** toolbar; you are prompted to select an element.

- **Swap Visible Space**



To view the space where all hidden elements are stored, invoke the **Swap visible space** tool from the **View** toolbar.

Tutorial 1

In this tutorial, you will draw the sketch of the model shown in **Figure A**. The sketch is shown in **Figure B**. You will not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

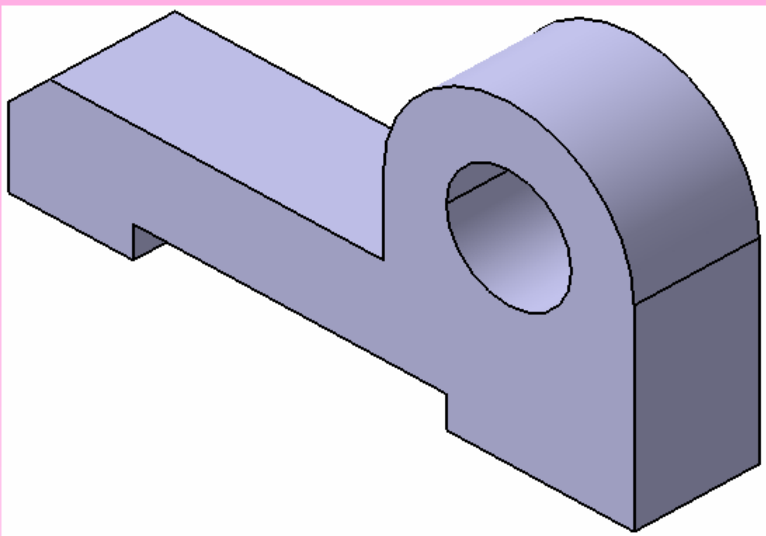


Figure A The solid model for Tutorial 1

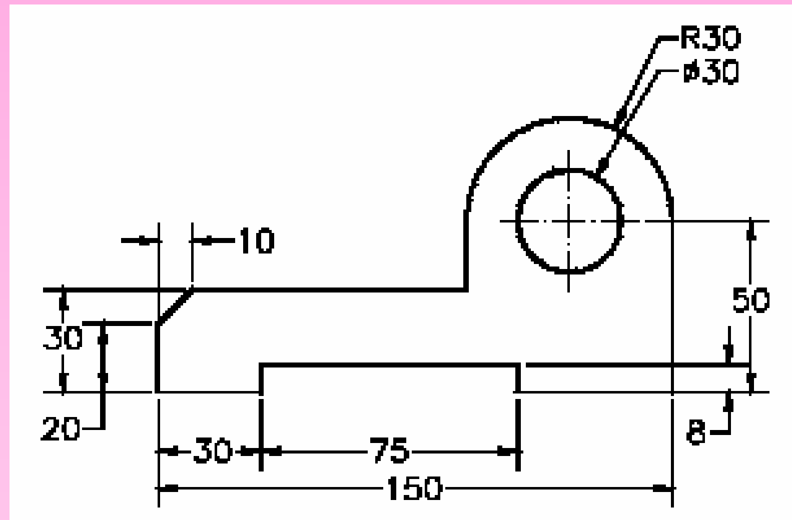


Figure B The sketch of the model

1. Start CATIA V5 and then start a new CAT part file.
2. Draw the sketch of the model using the **Line**, **Arc**, and **Circle** tools, as shown in **Figure C** and **Figure D**.

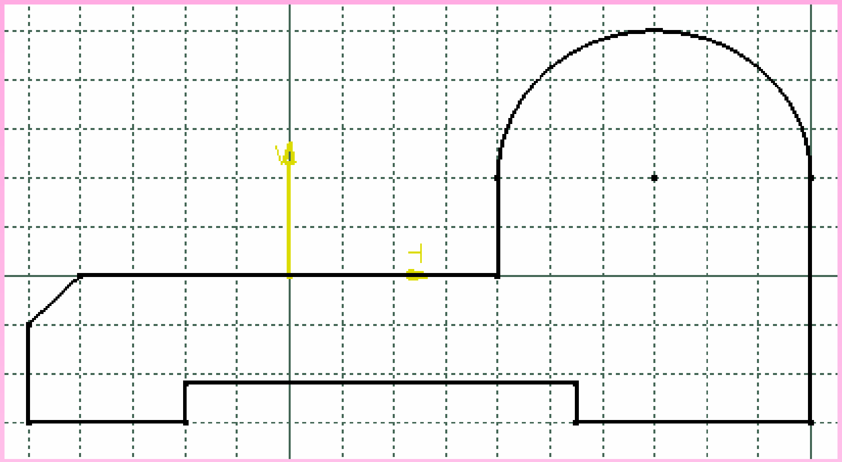


Figure C The outer loop of the sketch

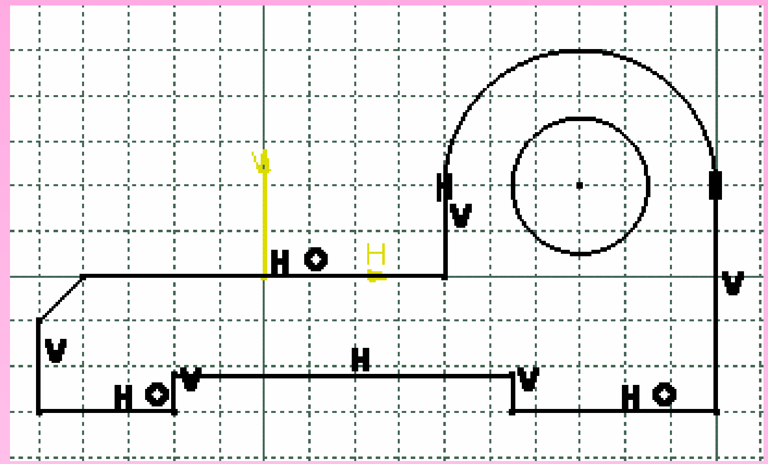


Figure D The final sketch for Tutorial 1

3. Save the file in **\My Documents\CATIA\c01 folder** and then close it.

❑ Tutorial 2

In this tutorial, you will draw the sketch of the model shown in **Figure A**. The sketch is shown in **Figure B**. You will not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

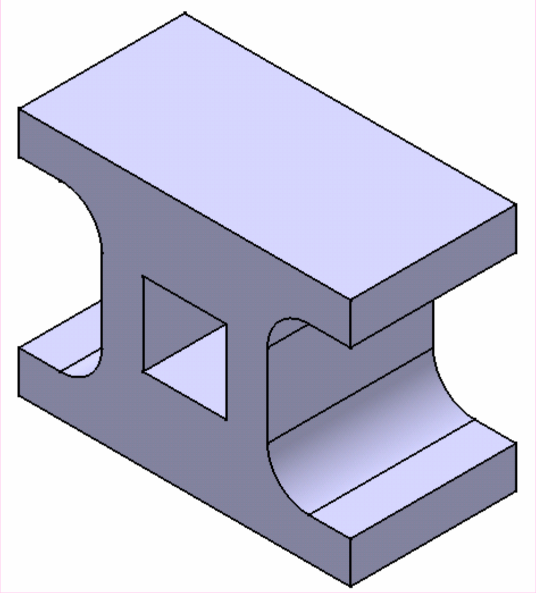


Figure A The solid model for Tutorial 2

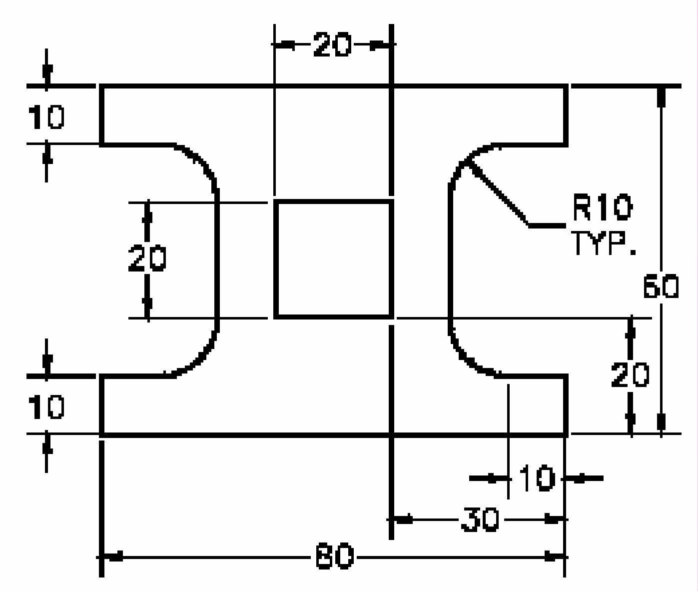


Figure B The sketch of the model

1. Start a new **CATpart** file.
2. Draw the sketch of the model using the **Profile** and **Rectangle** tool, as shown in **Figure C**, **Figure D** and **Figure E**.

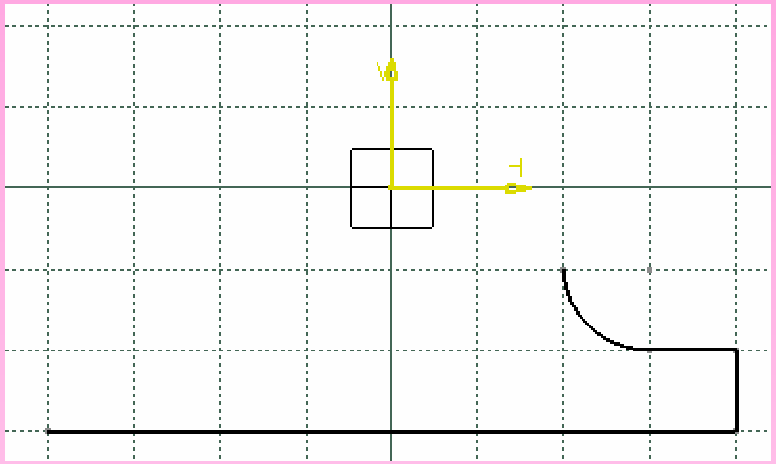


Figure C The sketch after drawing the three lines and a tangent arc

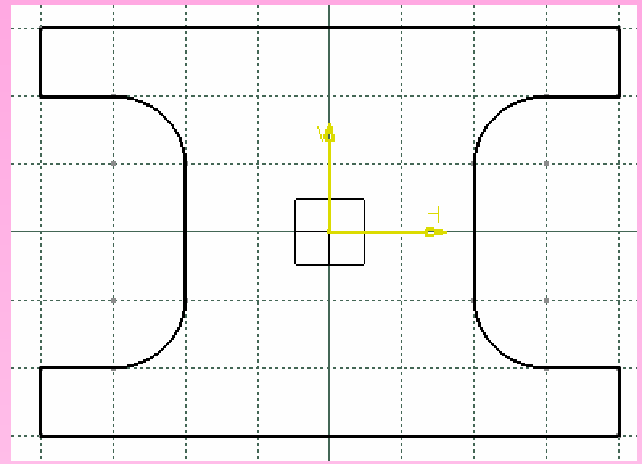


Figure D The sketch after drawing outer loop of the sketch

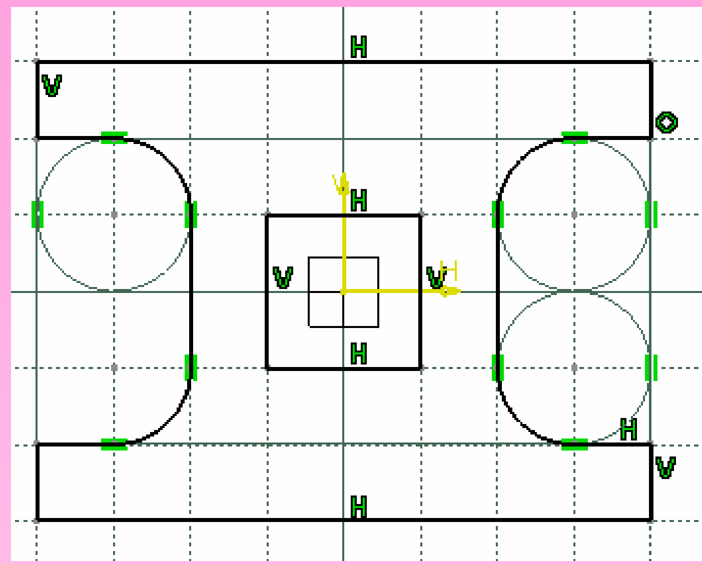


Figure E **The final sketch after drawing the inner loop of the sketch**

3. Save the file in **\My Documents\CATIA\c01 folder** and then close it.

Tutorial 3

In this tutorial, you will draw the sketch of the model, as shown in **Figure A**. The sketch is shown in **Figure B**. You will not dimension the sketch. The solid model and dimensions are given for your reference. **(Expected time: 30 min)**

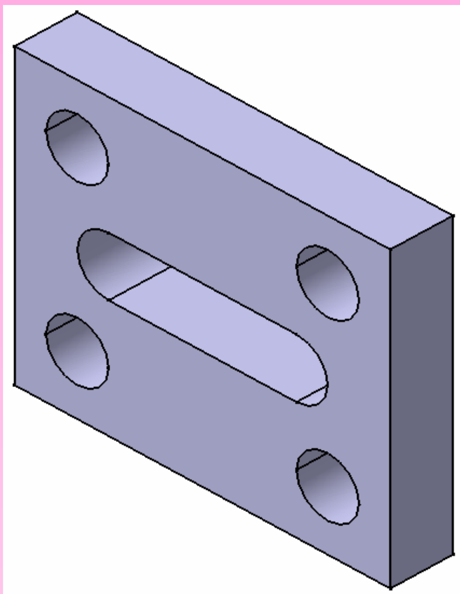


Figure A **The solid model for Tutorial 3**

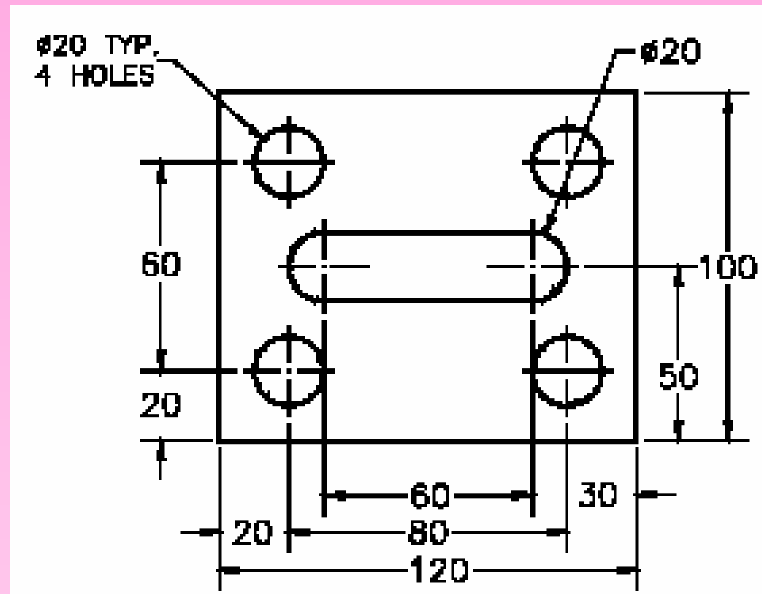


Figure B **The sketch for the solid model**

1. Start a new **CATpart** file.
2. Draw the sketch of the model using the **Rectangle**, **Profile**, and **Circle** tools, as shown in **Figure C**, **Figure D** and **Figure E**.

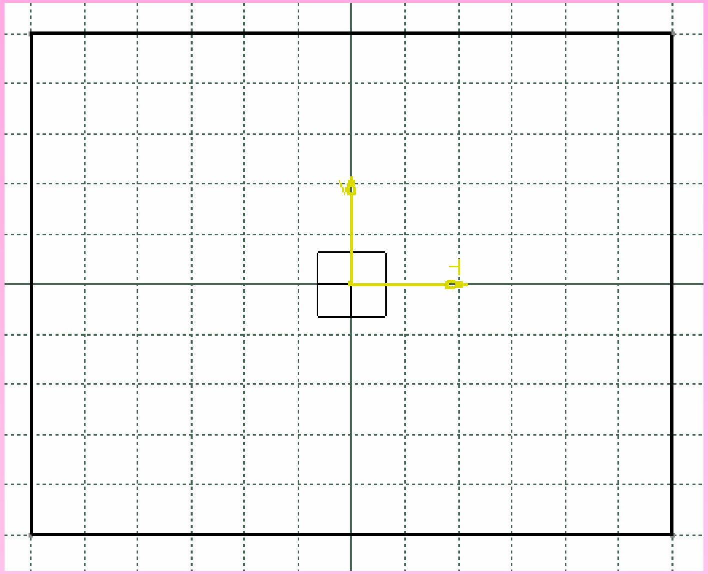


Figure C **The outer loop of the sketch**

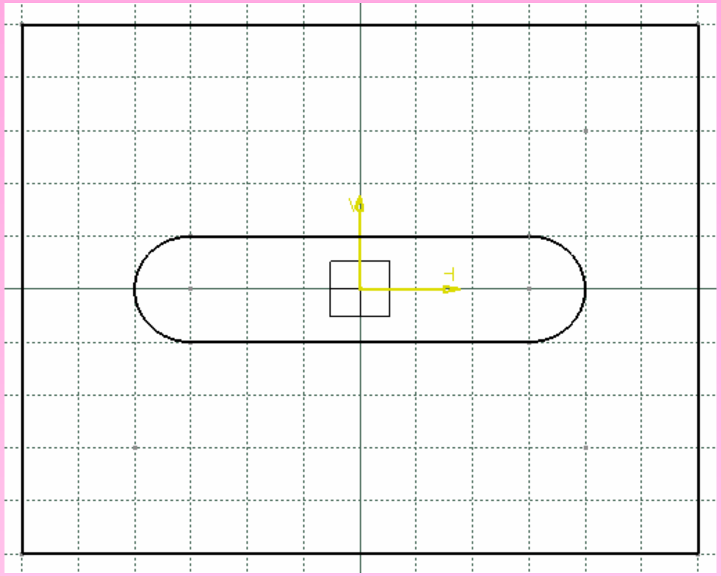


Figure D **The sketch after drawing the elongated hole**

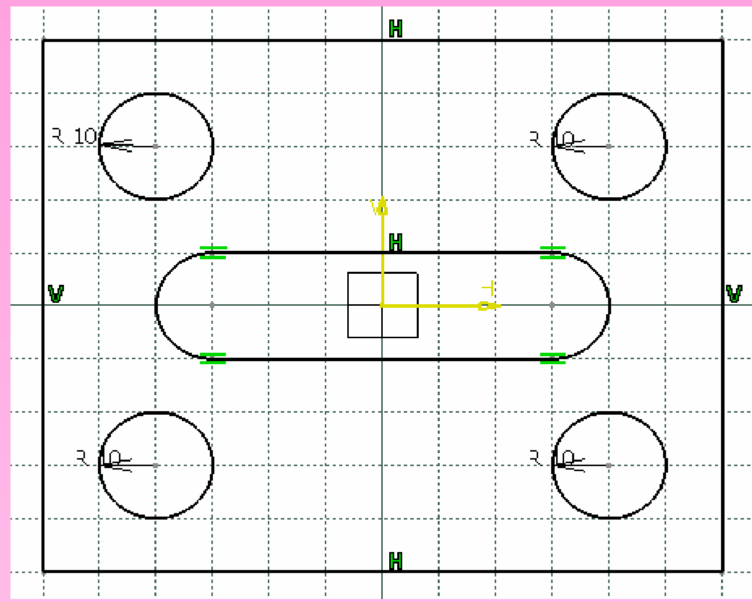


Figure E The final sketch

3. Save the file in **\My Documents\CATIA\c01 folder** and then close it.

Tutorial 4

In this tutorial, you will draw the sketch of the model shown in **Figure A**. The sketch is shown in **Figure B**. Do not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

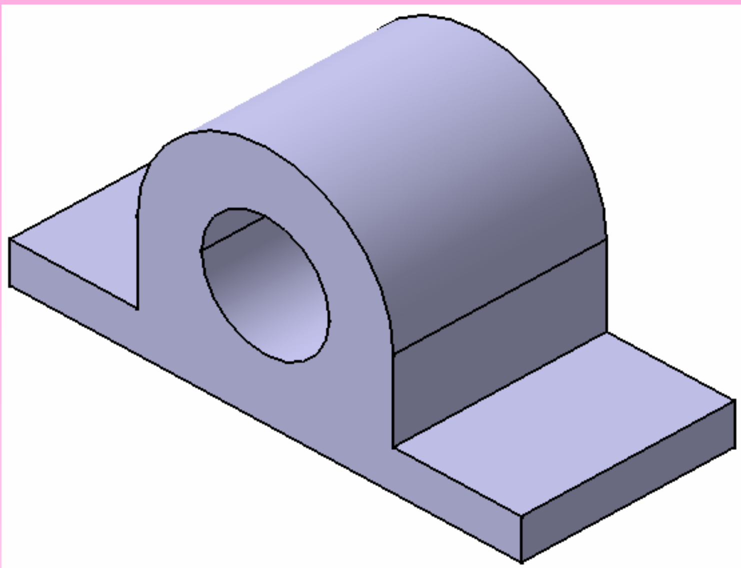


Figure A The solid model for Tutorial 4

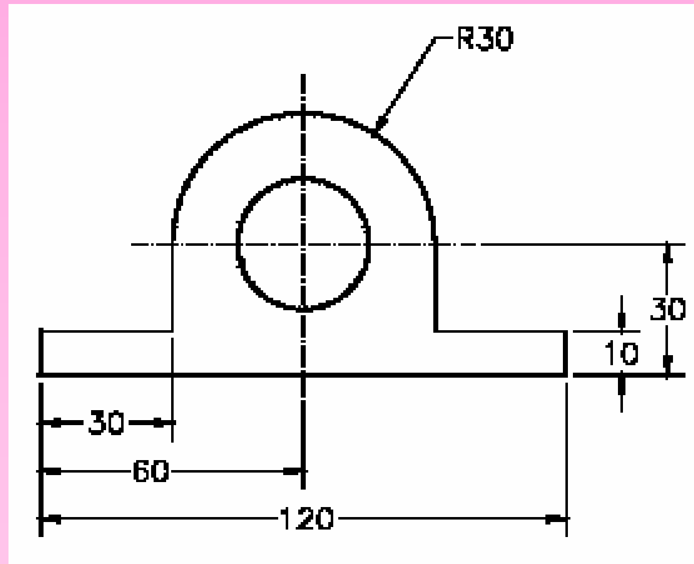


Figure B The sketch for the solid model

1. Start a new **CATpart** file.
2. Draw the sketch of the model using the **Profile** and the **Circle** tools, as shown in **Figure C** and **Figure D**.

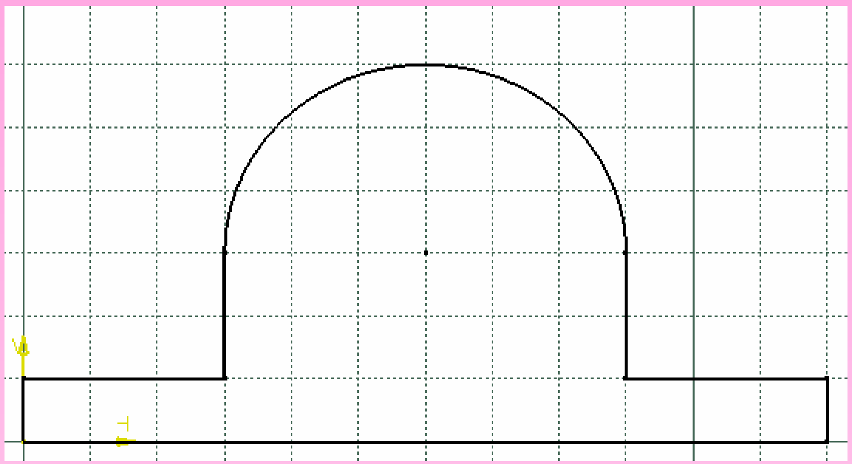


Figure C The sketch after drawing the outer loop

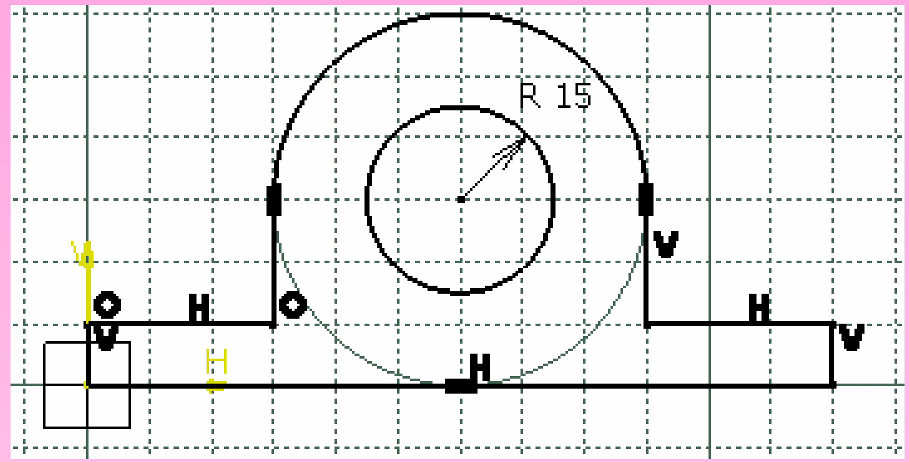


Figure D The final sketch for Tutorial 4

3. Save the file in **\My Documents\CATIA\c01 folder** and then close it.

Exercise 1

Draw the sketch of the model shown in **Figure A**. The sketch to be drawn is shown in **Figure B**. Do not dimension it. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

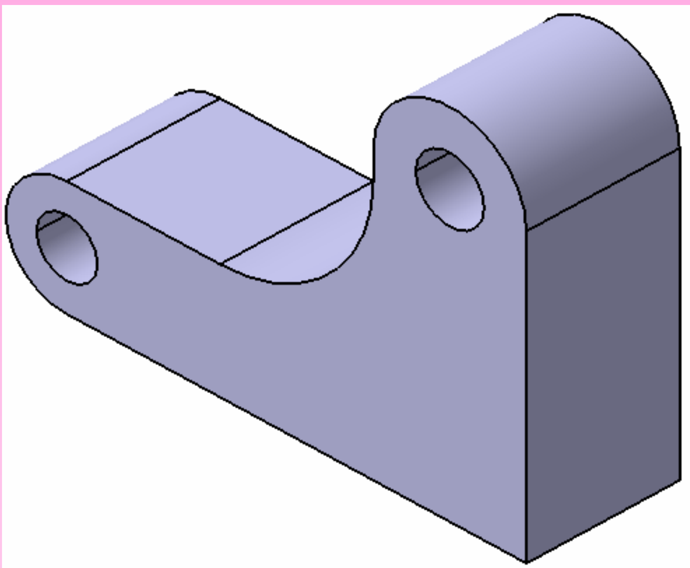


Figure A The solid model for Exercise 1

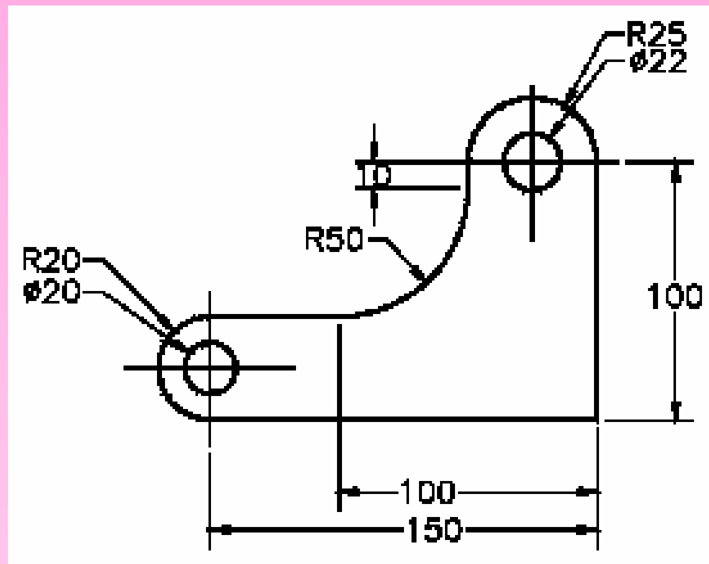


Figure B The sketch of the model

Exercise 2

Draw the sketch of the model shown in **Figure A**. The sketch to be drawn is shown in **Figure B**. Do not dimension it. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

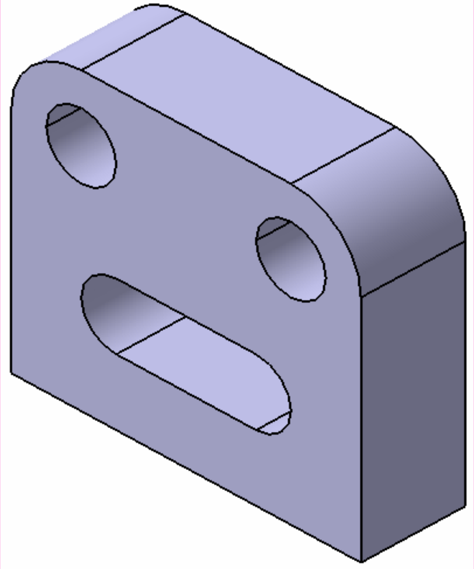


Figure A The solid model for Exercise 2

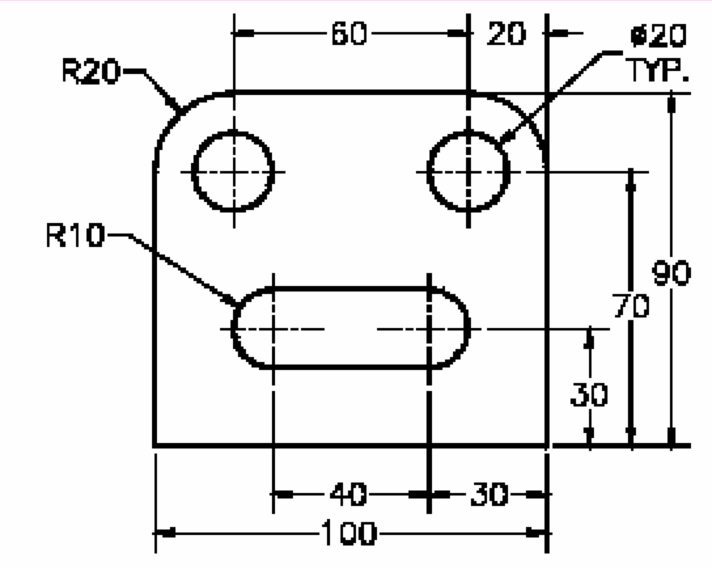


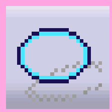
Figure B The sketch of the model

Learning Objectives:

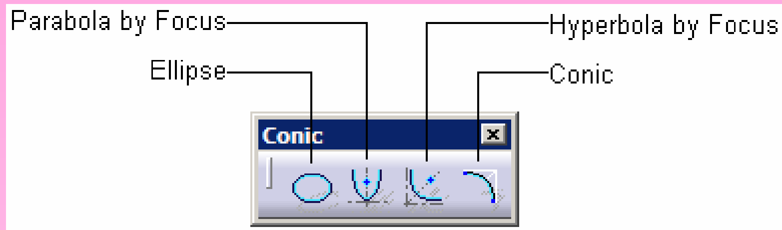
- *Draw ellipses.*
- *Draw splines.*
- *Connect two elements using an arc or a spline.*
- *Draw elongated holes.*
- *Draw cylindrical elongated holes.*
- *Draw key holes.*
- *Draw hexagons.*
- *Draw centered rectangles.*
- *Draw centered parallelograms.*
- *Draw different types of conics.*
- *Edit and modify sketches.*

OTHER SKETCHING TOOLS

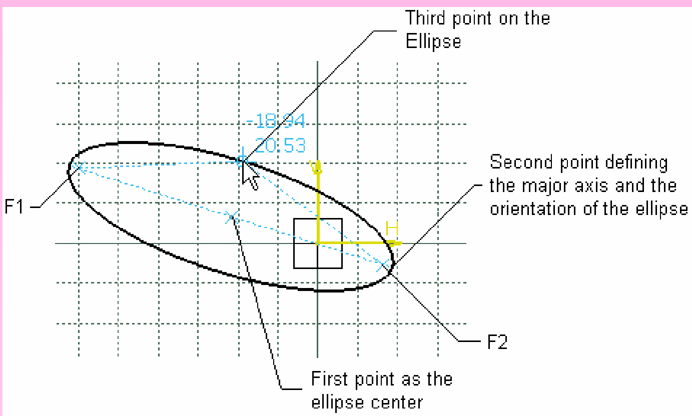
• Drawing Ellipses



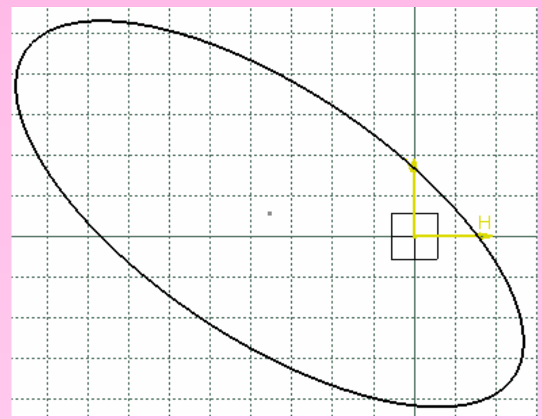
To draw an ellipse, invoke the **Ellipse** tool by choosing the **Ellipse** button from the **Conic** toolbar, as shown in the figure.



The Conic toolbar

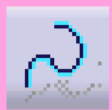


Specifying three points to draw an ellipse

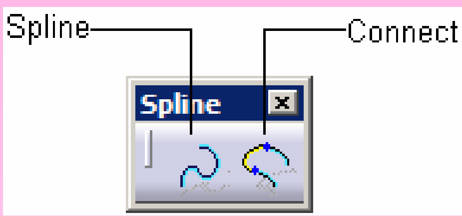


The resulting ellipse

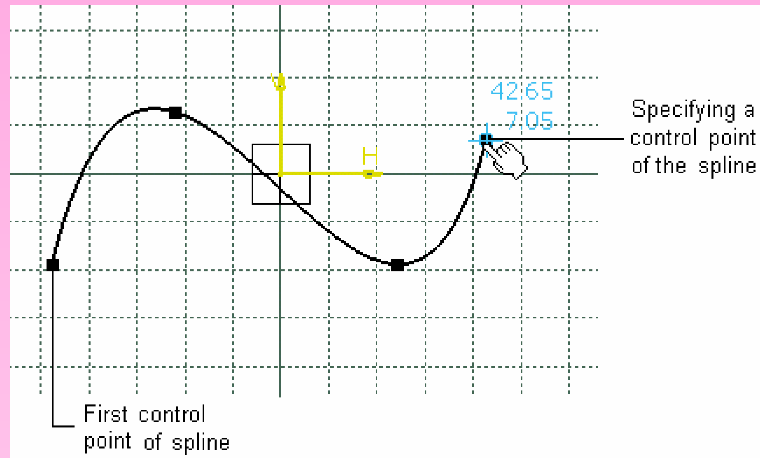
- **Drawing Splines**



To draw a spline, choose the down arrow on the right of the **Spline** button in the **Profile** toolbar and invoke the **Spline** toolbar, as shown in the figure.

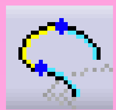


The **Spline** toolbar

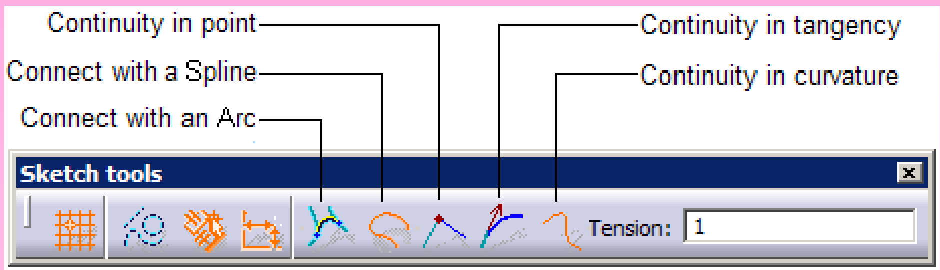


Specifying points for drawing a spline

• Connecting two Elements by a Spline or an Arc



- Two elements such as lines, arcs, ellipses, circles, or splines can be connected together by an arc or spline.
- To do so, invoke the **Connect** tool from the **Spline** toolbar; the **Sketch tools** toolbar expands, as shown in the figure.



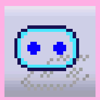
The Sketch tools toolbar after invoking the Connect tool

• Connecting Two Elements with a Spline

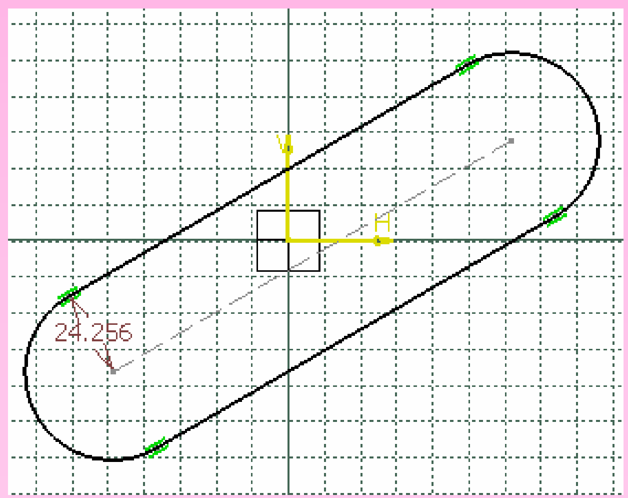
- By default, the **Connects with a Spline** button is chosen in the **Sketch tools** toolbar.
- You are prompted to select the first element to be connected.

- **Connecting Two Elements with an Arc**
  - To connect two selected elements with an arc, choose the **Connect** button from the **Spline** toolbar.
  - Choose the **Connects with an Arc** button from the **Sketch tools** toolbar; you will be prompted to select the first element to be connected.

- **Drawing Elongated Holes**

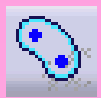


To draw an elongated hole invoke the **Elongated Hole** tool from the **Predefined Profile** toolbar.

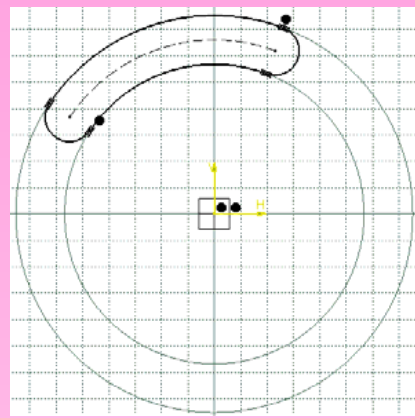


An elongated hole profile

- **Drawing Cylindrical Elongated Holes**

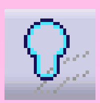


To draw a cylindrical elongated hole, invoke the **Cylindrical Elongated Hole** tool from the **Predefined Profile** toolbar.

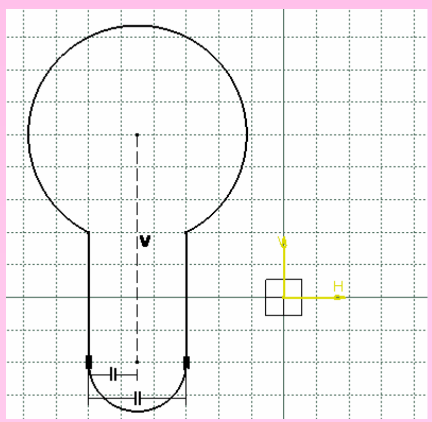


A cylindrical elongated hole

- **Drawing Keyhole Profiles**

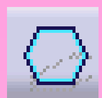


To draw a keyhole profile, invoke the **Keyhole Profile** tool from the **Predefined Profile** toolbar.

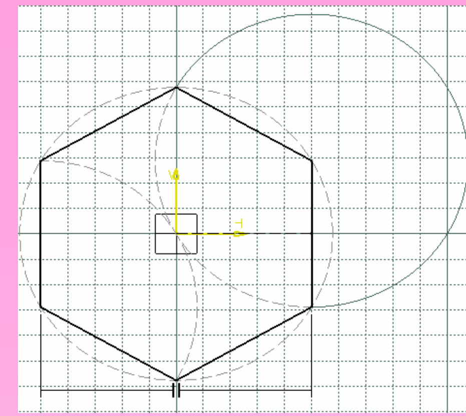


A keyhole profile

- **Drawing Hexagons**

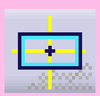


To draw a hexagon choose the **Hexagon** button from the **Predefined Profile** toolbar.

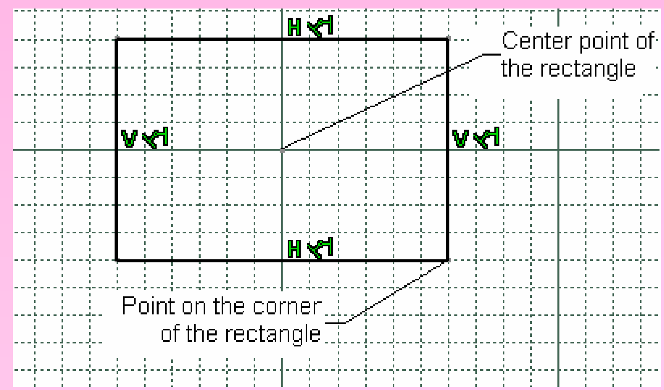


Hexagon drawn using the Hexagon tool

- **Drawing Centered Rectangles**

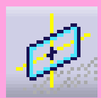


To draw a centered rectangle, choose the **Centered Rectangle** button from the **Predefined Profile** toolbar.

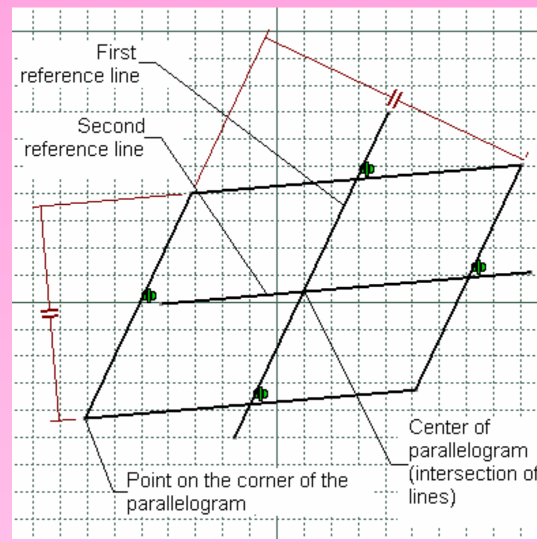


Center rectangle along with the center point and the point on the corner of the rectangle

• Drawing Centered Parallelograms



To draw a centered parallelogram, choose the **Centered Parallelogram** button from the **Predefined Profile** toolbar.

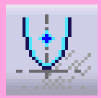


Centered parallelogram with the first line, second line, and the point on parallelogram

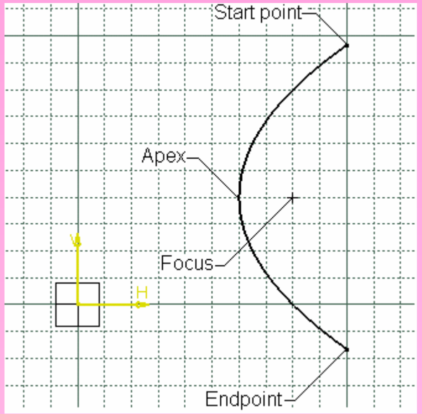
• Drawing Conics

To draw various conics available in CATIA V5R16, choose the down arrow available on the right of the **Ellipse** button in the **Profile** toolbar.

- **Drawing a Parabola by Focus**

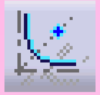


To draw a parabola by focus, invoke the **Parabola by Focus** tool from the **Conic** toolbar.

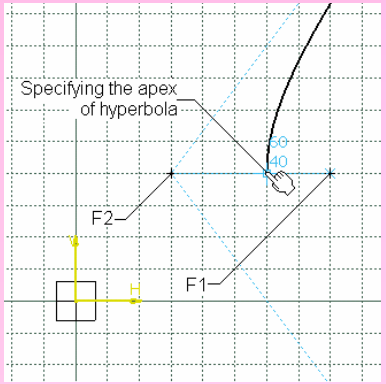


Points used to draw a parabola

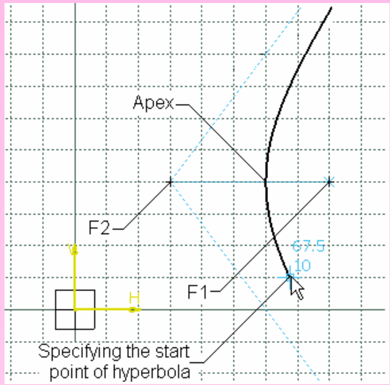
- **Drawing a Hyperbola by Focus**



To draw a hyperbola by focus invoke the **Hyperbola by Focus** tool from the **Conic** toolbar.

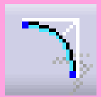


Specifying the apex of the hyperbola

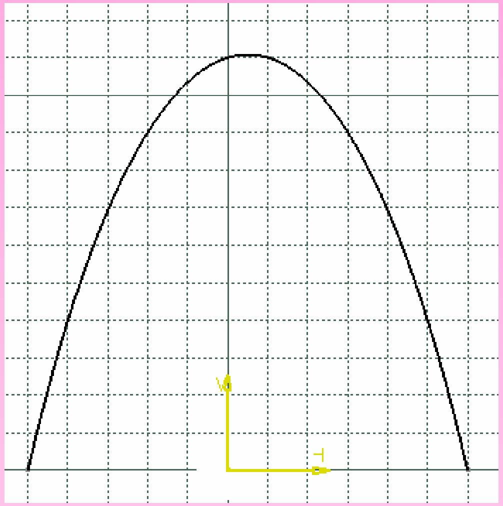


Specifying the start point of the hyperbola

- **Drawing Conics**



To draw a conic, invoke the **Conic** tool from the **Conic** toolbar.

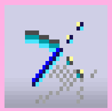


A Conic

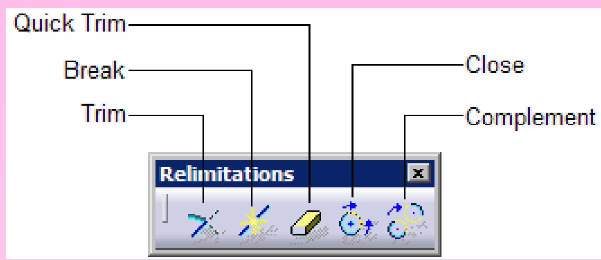
EDITING AND MODIFYING SKETCHES

- CATIA V5 provides you with a number of tools that can be used to edit the sketched elements.
- These include trimming the sketches using the quick trim, breaking a sketched element, filleting the sketches, adding chamfer to the sketches, and so on.

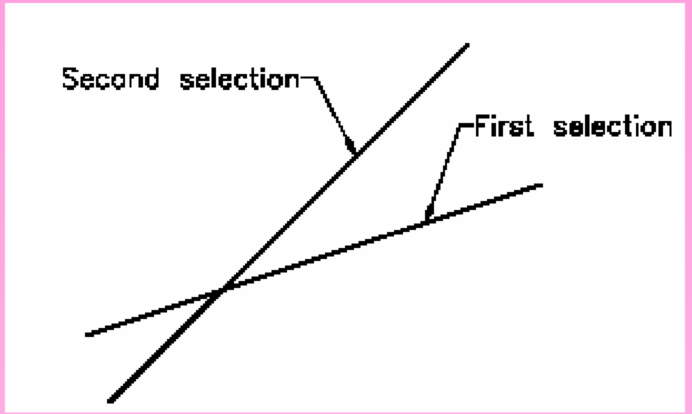
Trimming Unwanted Sketched Elements



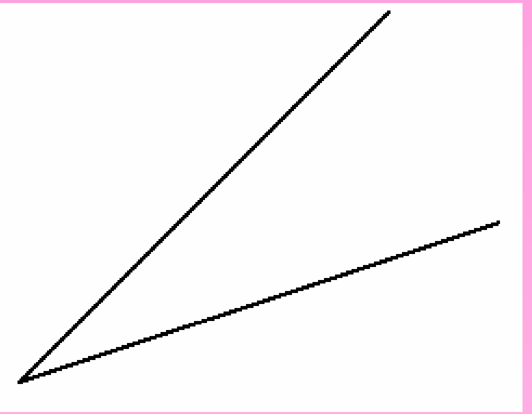
- Invoke the **Relimitations** toolbar by choosing the down arrow provided on the right of the **Trim** button in the **Operation** toolbar.
- The **Relimitations** toolbar is shown in the figure.



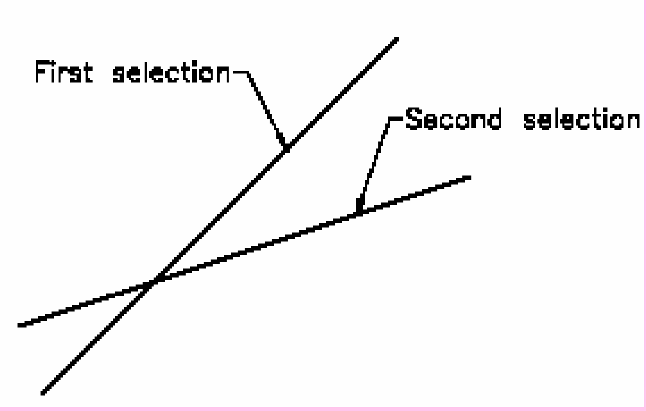
The Relimitations toolbar



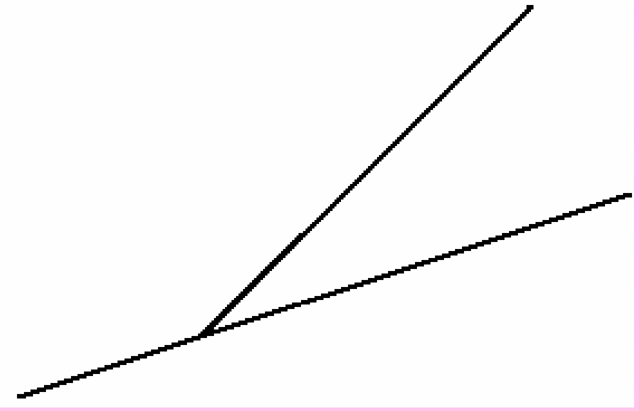
Elements to be selected for trimming



The resulting trimmed elements

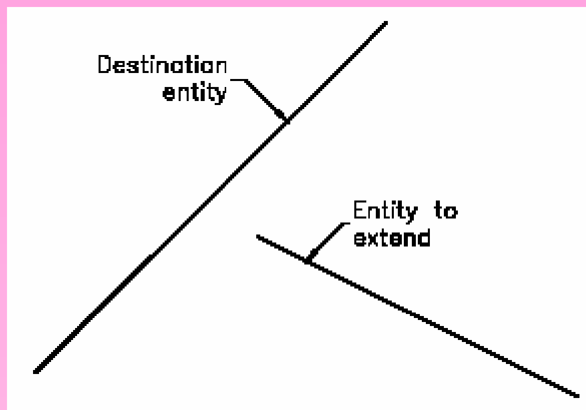


Elements to be selected for trimming

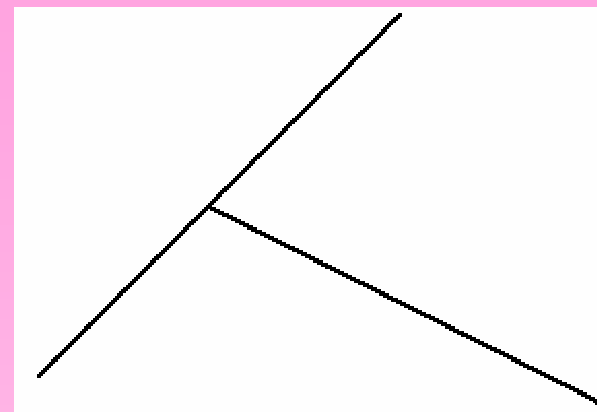


The resulting trimmed elements

- **Extending Sketched Elements**

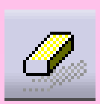


Elements selected to be extended



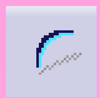
The resulting extended element

- **Trimming by Using the Quick Trim Tool**

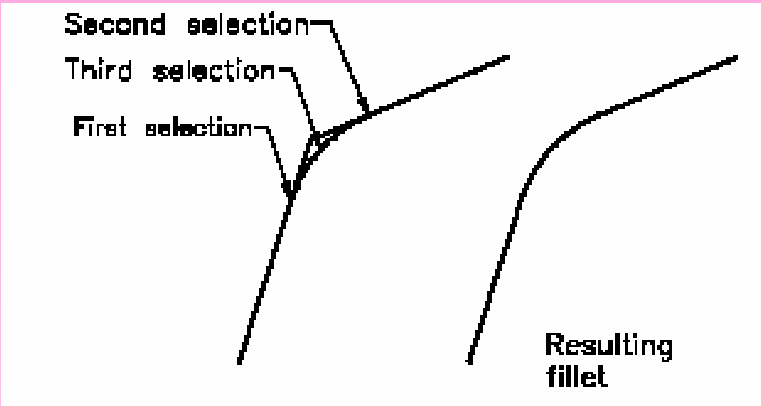


CATIA V5 also provides you with a tool using which you can quickly trim the unwanted sketched elements.

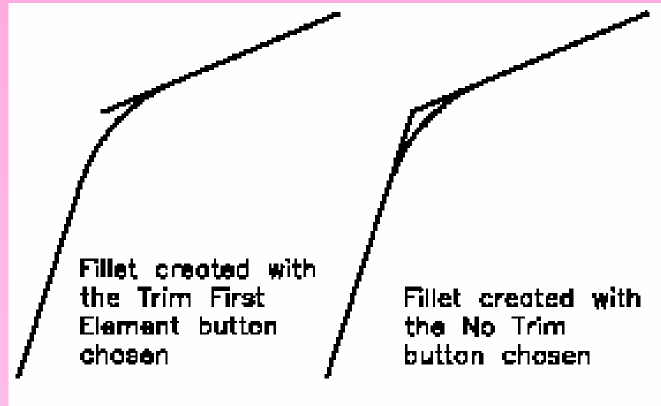
Filleting Sketched Elements



In the **Sketcher** workbench of CATIA V5, you are provided with the **Corner** tool to fillet the sketched elements.

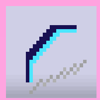


Elements to be selected and the resulting filleted sketched elements

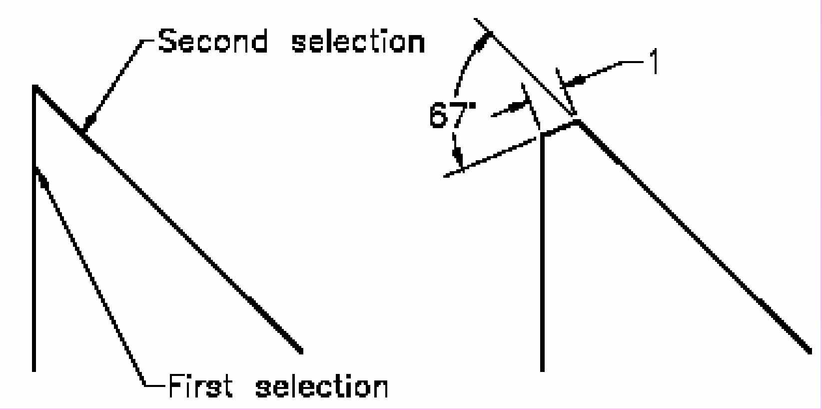


Fillets created using the selected Trim First Element and the No Trim buttons

• Chamfering Sketched Elements

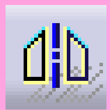


The **Sketcher** workbench of CATIA V5 also provides you with a **Chamfer** tool to chamfer the sketched elements.

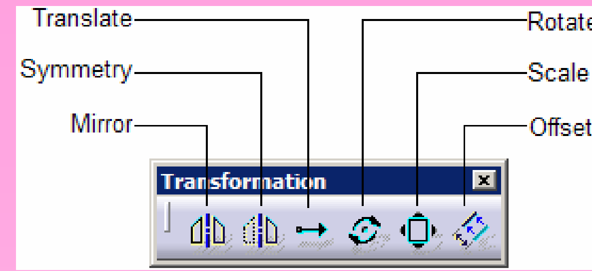


Elements to be selected and the resulting chamfer created

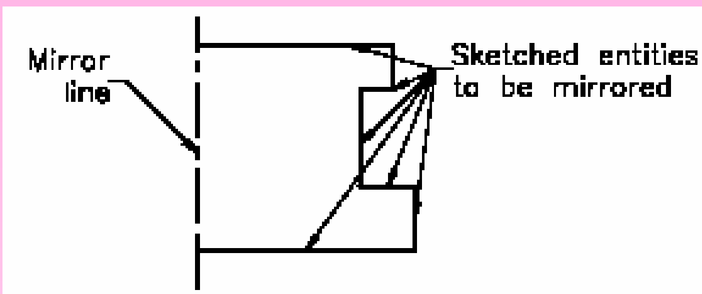
• Mirroring Sketched Elements



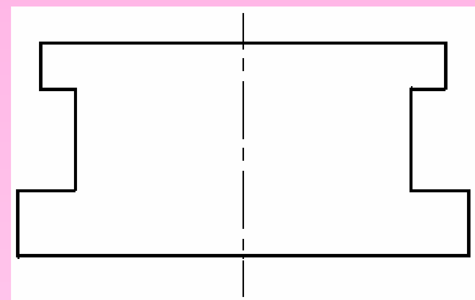
- You can mirror the sketched elements along the mirror line in the **Sketcher** workbench of CATIA V5 using the **Mirror** tool.
- Choose the down arrow on the right of the **Mirror** button provided in the **Operation** toolbar to invoke the **Transformation** toolbar, as shown in the figure.



The **Transformation** toolbar

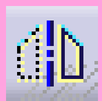


Elements selected to be mirrored and the mirror line to be selected



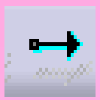
The Resulting mirrored sketch

• Mirroring Elements Without Duplication

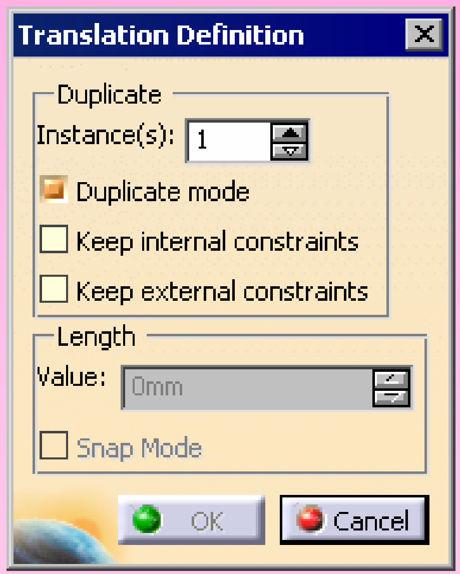


The **Symmetry** tool mirrors the sketched elements about a mirror axis but deletes the original elements.

• Translating Sketched Elements

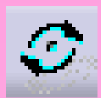


The **Sketcher** workbench provides you the **Translate** tool to move the selected sketched elements from their initial position to the required destination.

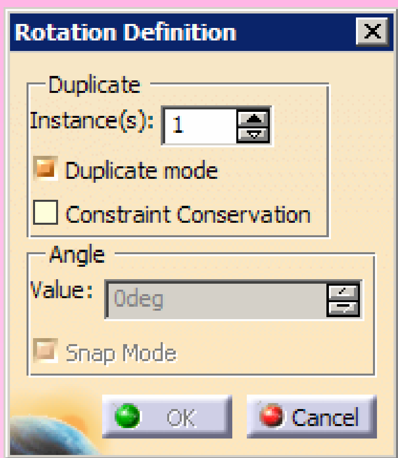


The Translation Definition dialog box

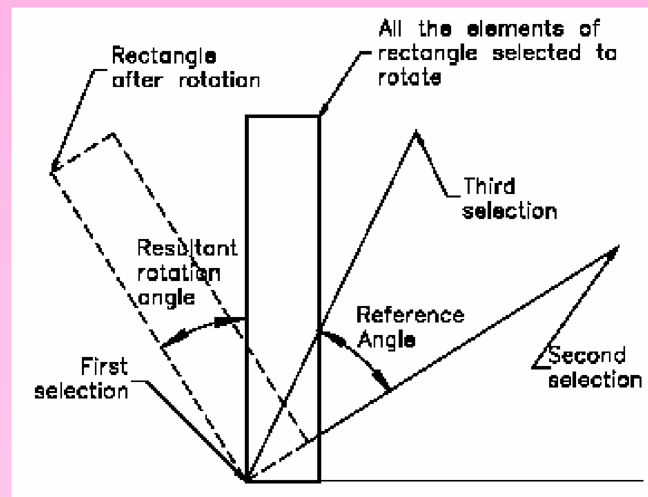
• Rotating Sketched Elements



- The **Rotate** tool is used to rotate the sketched elements around a rotation center point.
- Select the elements by drawing a window around them and then choose the **Rotate** button from the **Transformation** toolbar.
- The cursor is replaced by the point cursor and the **Rotation Definition** dialog box is displayed, as shown in the figure.



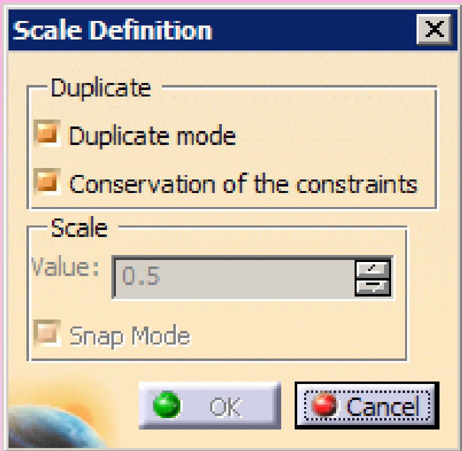
The Rotation Definition dialog box



Points to be selected and the rotated elements

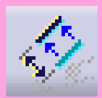
• Scaling Sketched Elements

- To scale the sketched elements, select them and then choose the **Scale** button from the **Transformation** toolbar.
- The **Scale Definition** dialog box is displayed, as shown in the figure.

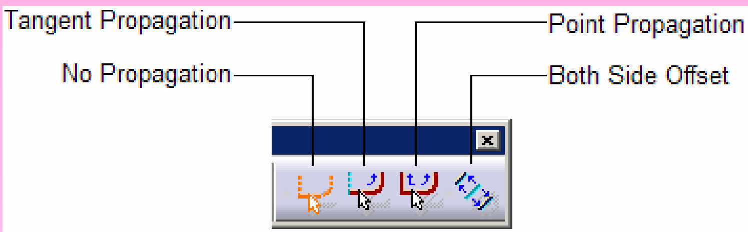


The Scale Definition dialog box

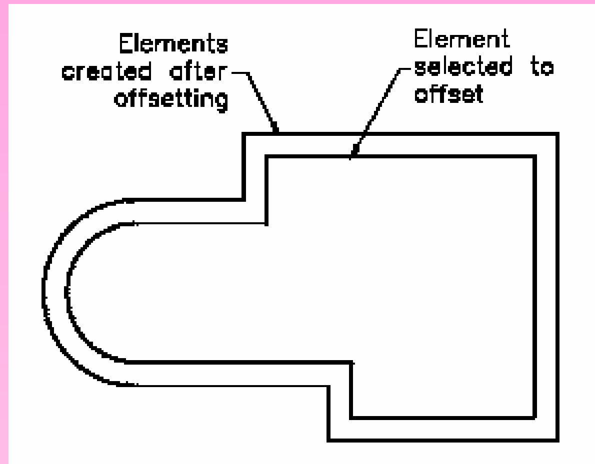
• Offsetting Sketched Elements



To offset the sketched elements, select them and then choose the **Offset** button from the **Transformation** toolbar.



The four additional buttons in the expanded Sketch tools toolbar



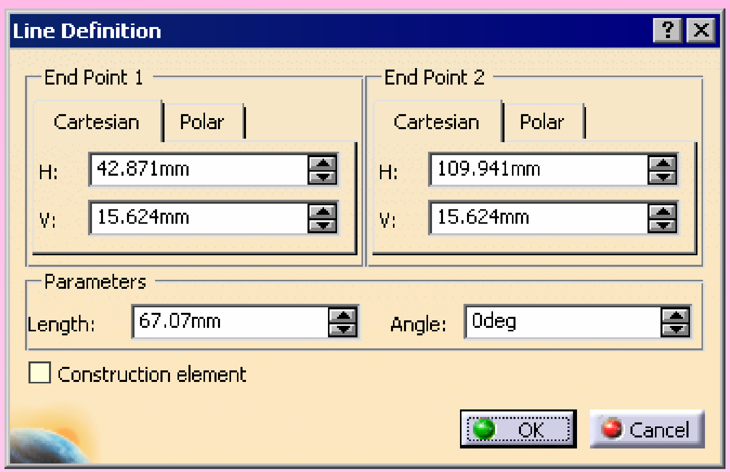
Elements created after offsetting

- ## Modifying Sketched Elements

You can modify the sketched elements, using various tools available in the **Sketcher** environment of CATIA V5.

- ## Modifying the Sketched Line

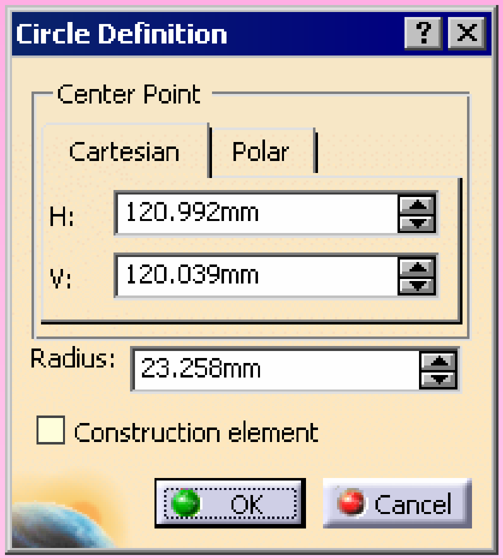
  - You can modify a sketched line using the **Line Definition** dialog box.
  - To invoke it, double-click on the sketched line.
  - The **Line Definition** dialog box is displayed, as shown in the figure.



The Line Definition dialog box

Modifying a Sketched Circle

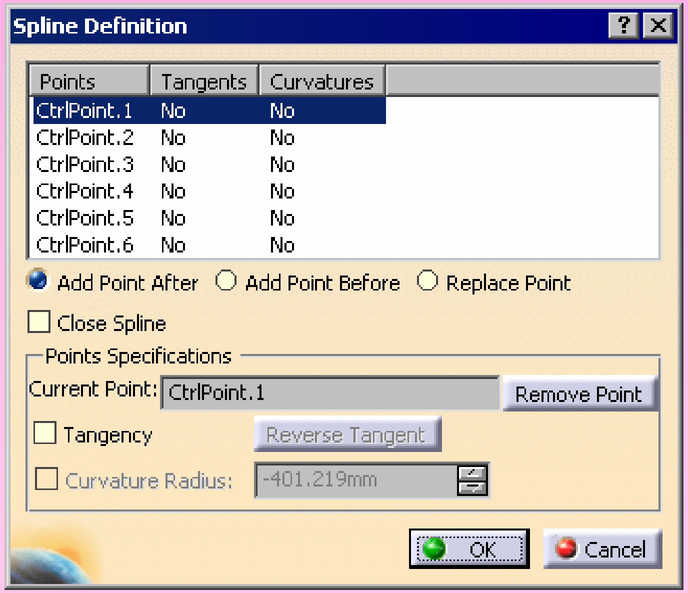
- You can modify a sketched circle using the **Circle Definition** dialog box.
- You can invoke this dialog box by double-clicking on the sketched circle.
- The **Circle Definition** dialog box is displayed as shown in the figure.



The Circle Definition dialog box

- **Modifying the Sketched Arc**
  - The arcs are also modified using the **Circle Definition** dialog box.
  - To invoke it, double-click on the arc to be modified.
- **Modifying the Sketched Spline**

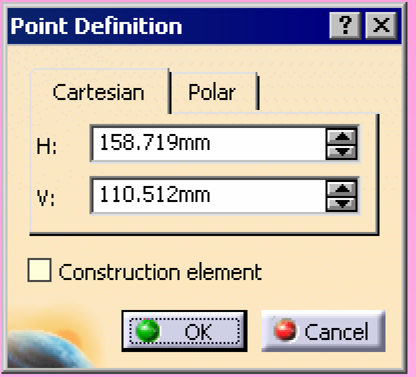
  You can modify a spline using the **Spline Definition** dialog box, which is displayed when you double-click on the spline that needs to be modified.



The Spline Definition dialog box

• Modifying the Sketched Point

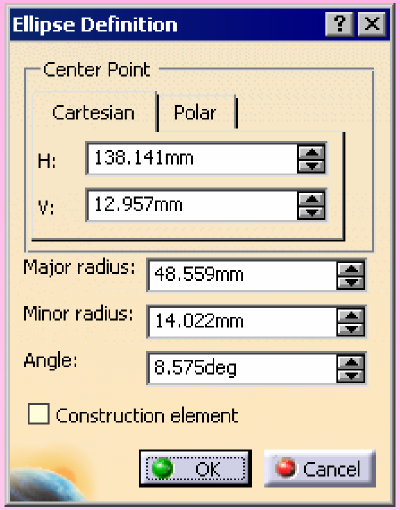
To modify a sketched point, double click on it; the **Point Definition** dialog box is displayed, as shown in the figure.



The **Point Definition** dialog box

• Modifying the Sketched Ellipse

To modify a sketched ellipse, double-click on it; the **Ellipse Definition** dialog box is displayed, as shown in the figure.



The **Ellipse Definition** dialog box

- **Modifying the Sketched Elements by Dragging**

  The modification of the sketched element can be done by dragging its start point, end point, profile, or the control points.

- **Deleting Sketched Elements**

  - To delete the sketched element, select the sketched element and choose the DELETE key.

  - You can also delete the sketched elements by selecting them and then right-clicking to invoke the contextual menu.

Tutorial 1

In this tutorial, you will draw the sketch of the model shown in **Figure A**. The sketch is shown in **Figure B**. You will not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30min)**

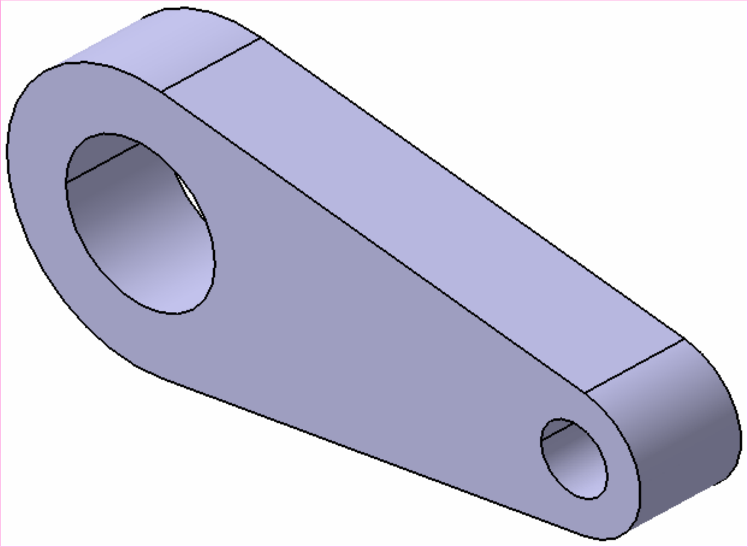


Figure A The Model for Tutorial 1

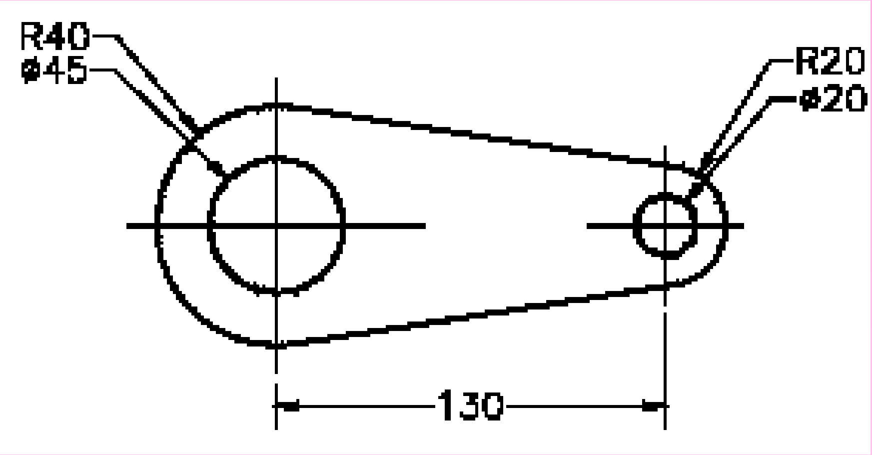


Figure B The sketch for Tutorial 1

1. Start a new file in the **Part** workbench and draw the outer loop of the sketch using the **Circle** and **By-Tangent Line** tool, as shown in **Figure C**, **Figure D**, **Figure E** and **Figure F** .

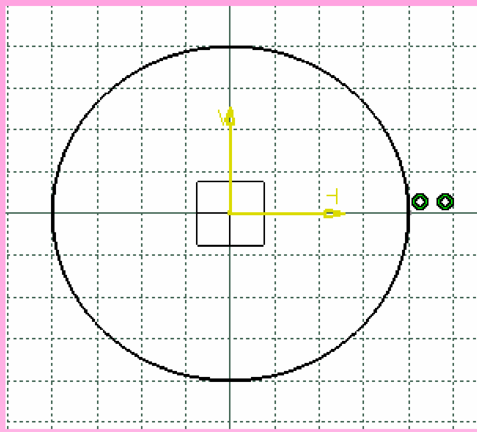


Figure C The Sketch after drawing the first circle

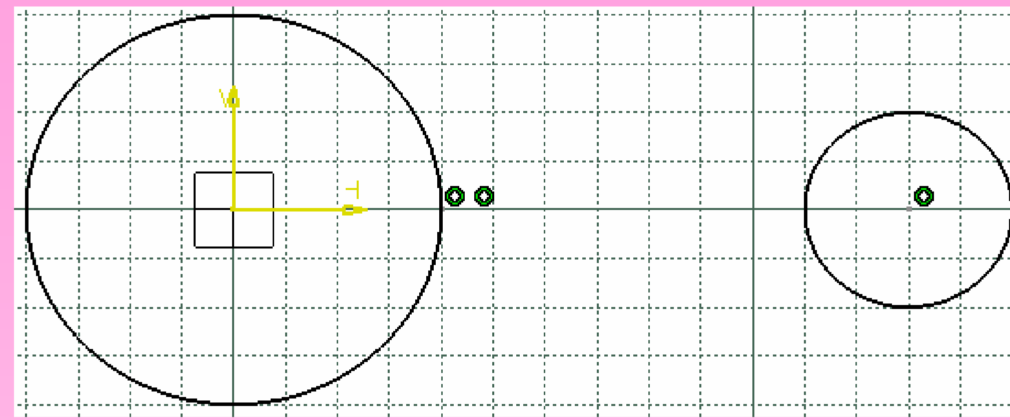


Figure D The Sketch after drawing the second circle

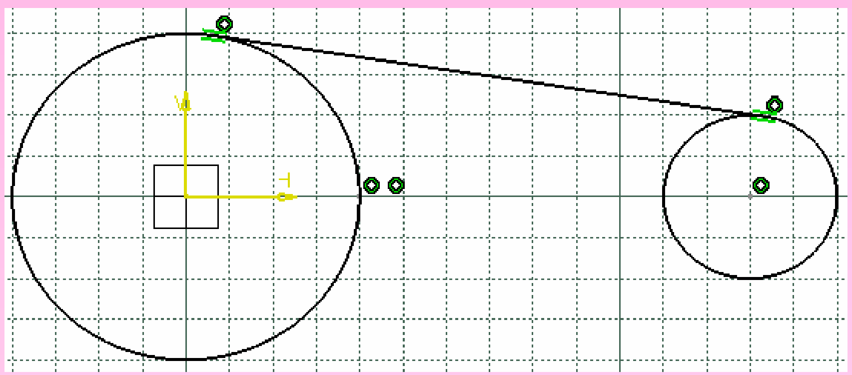


Figure E The sketch after drawing the first tangent line

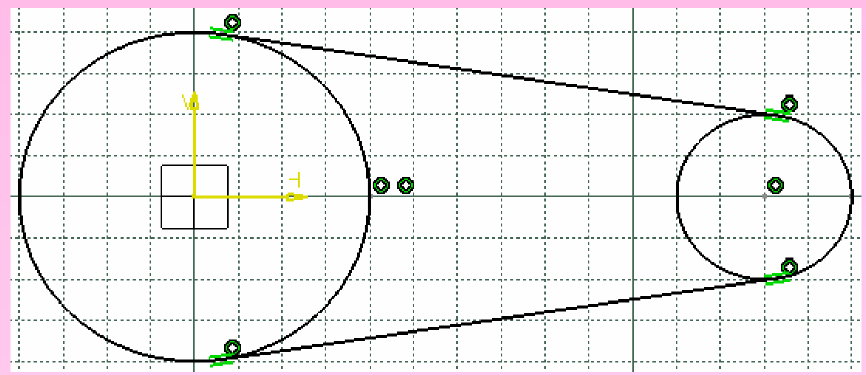


Figure F The sketch, after drawing the second tangent line

2. Trim the unwanted portion of the outer loop of the sketch using the **Quick Trim** tool, as shown in **Figure G** and **Figure H**.

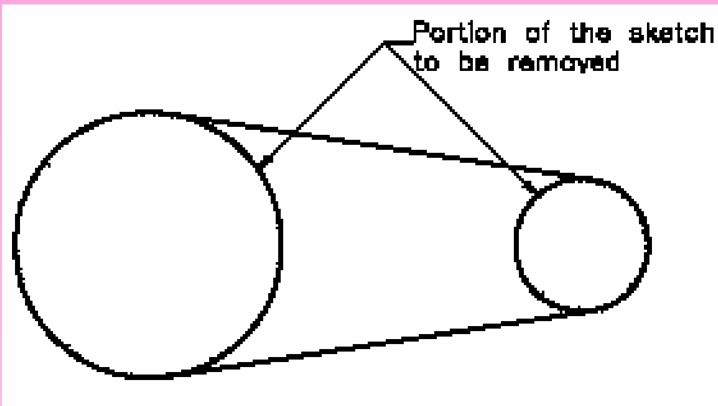


Figure G **The unwanted portion of the sketch to be trimmed**

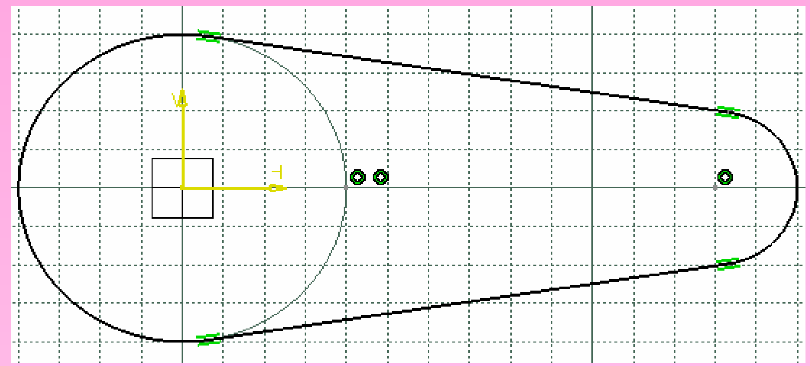


Figure H **The sketch after trimming the unwanted portion**

3. Draw the inner loops of the sketch using the **Circle** tool, as shown in **Figure I**.

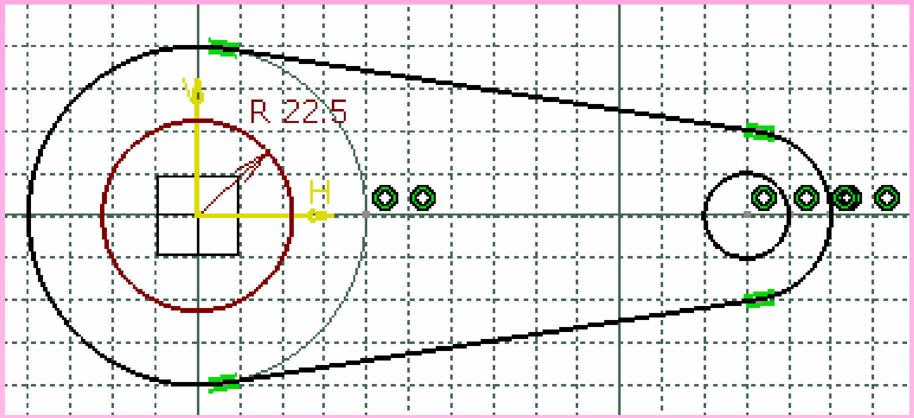


Figure I Final sketch

4. Save the file in **\My Documents\CATIA\c02 folder** and then close it.

Tutorial 2

In this tutorial, you will draw the sketch of the model shown in **Figure A**. The sketch is shown in **Figure B**. You will not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30min)**

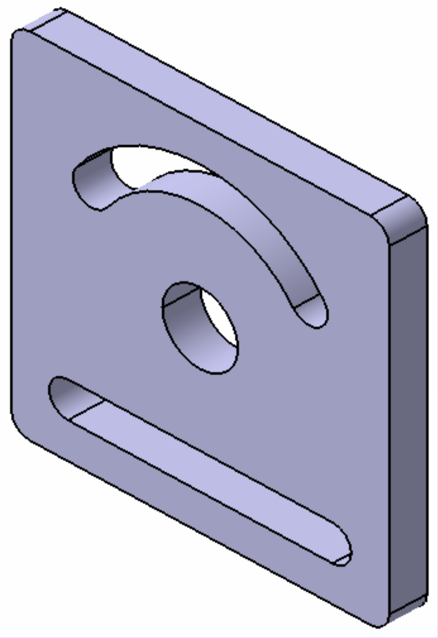


Figure A The model for Tutorial 2

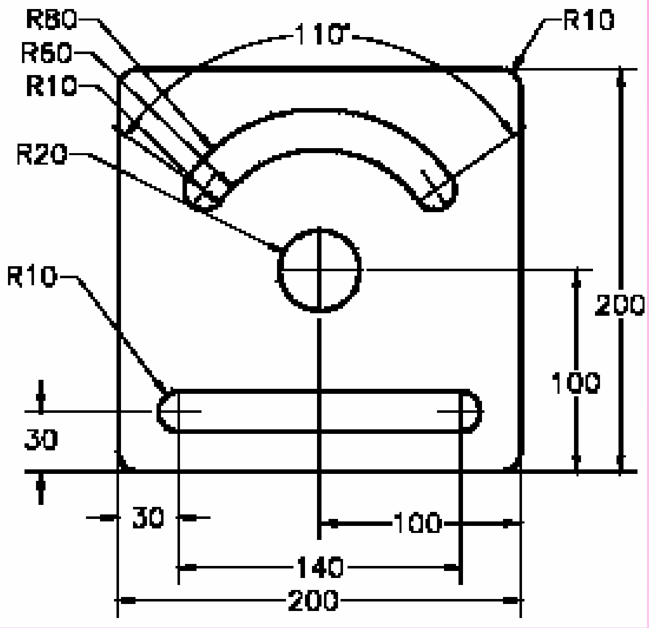


Figure B The Sketch of Tutorial 2

1. Start a new file in the **Part** workbench.

2. Draw the outer loop of the sketch using the **Rectangle** tool and then edit it using the **Corner** tool, as shown in **Figure C**, **Figure D** and **Figure E**.

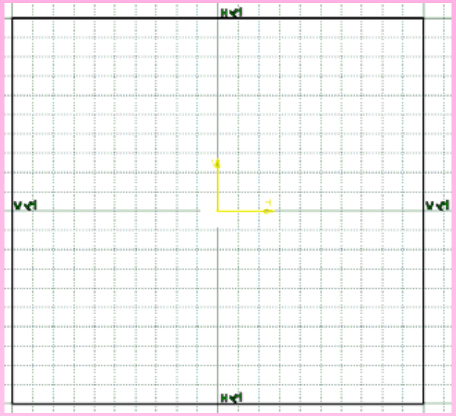


Figure C The sketch after drawing centered rectangle

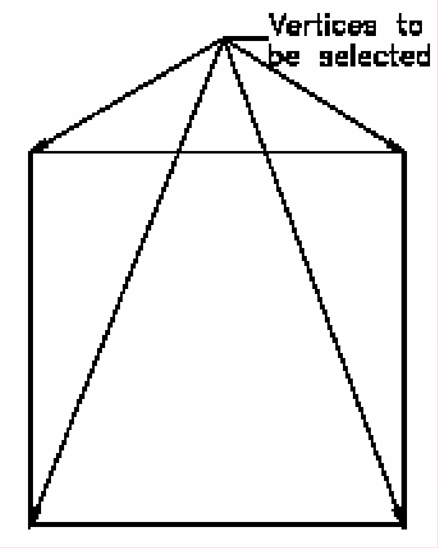


Figure D The vertices to be selected

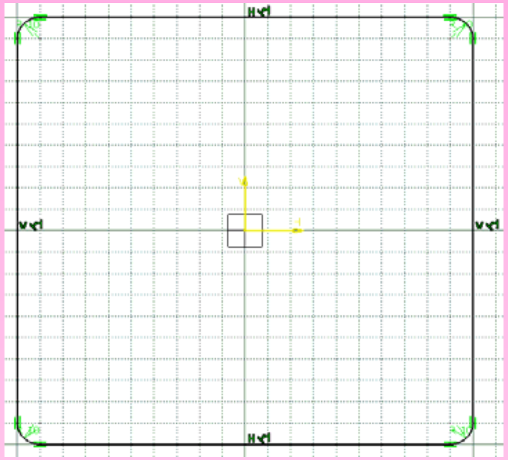


Figure E The final outer loop of the sketch

3. Draw the inner loop of the sketch using the **Circle**, **Elongated Hole**, and **Cylindrical Elongated Hole** tool, as shown in **Figure F**, **Figure G** and **Figure H**.

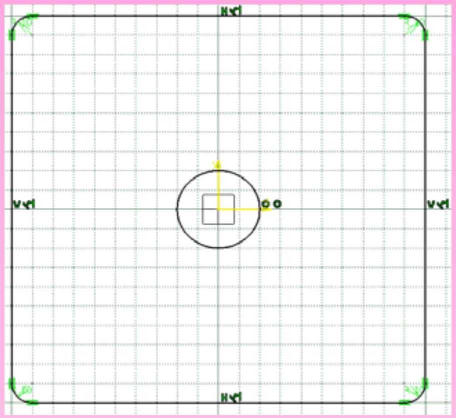


Figure F The sketch after drawing the circle

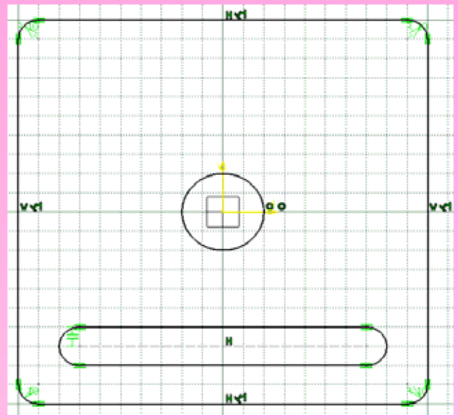


Figure G The sketch after drawing the elongated hole

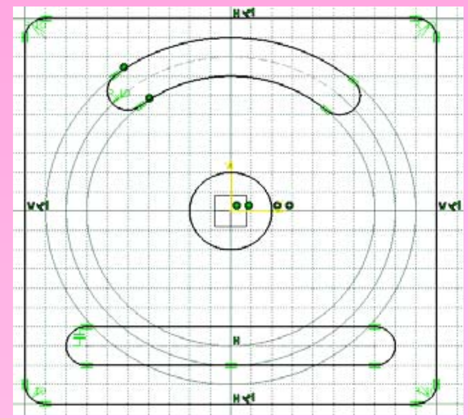


Figure H The final sketch

4. Save the file in **\My Documents\CATIA\c02 folder** and then close it.

Tutorial 3

In this tutorial, you will draw the sketch of the model, as shown in **Figure A**. The sketch is shown in **Figure B**. You will not dimension the sketch. The solid model and the dimensions are given only for your reference. **(Expected time: 30 min)**

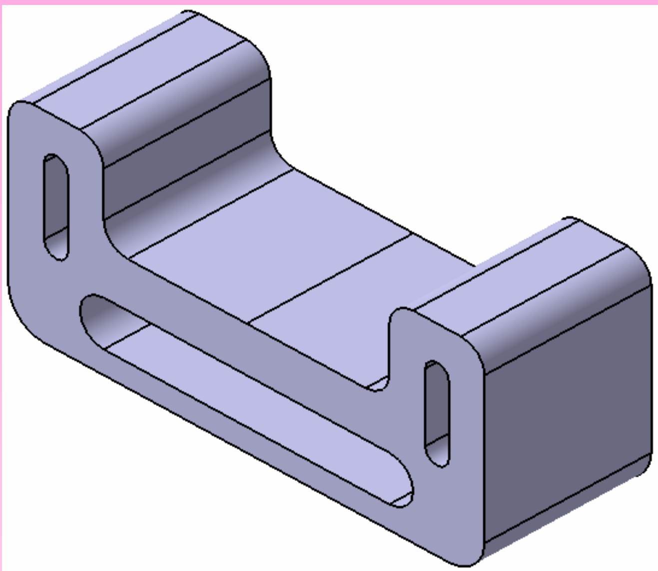


Figure A The model for Tutorial 3

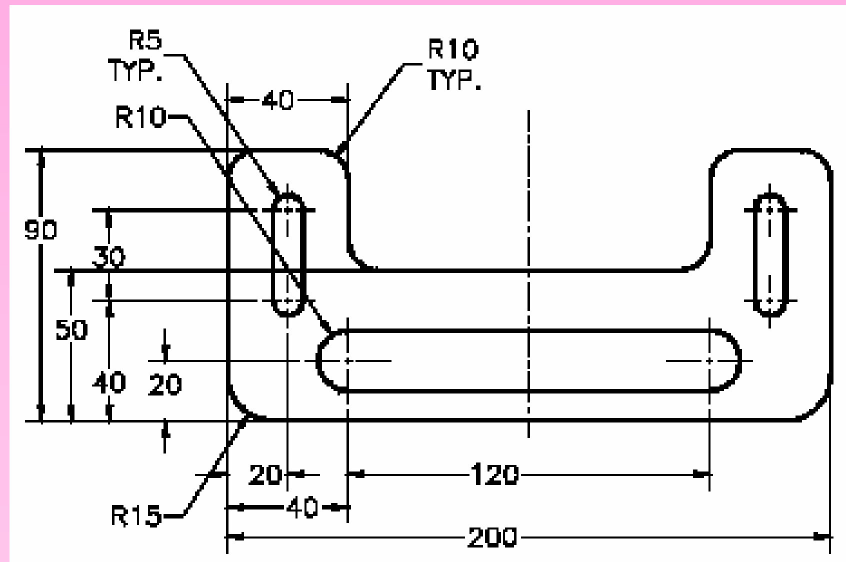


Figure B The sketch for Tutorial 3

1. Draw the right half of the sketch using the **Profile** and **Elongated Hole** tool, as shown in **Figure C**.

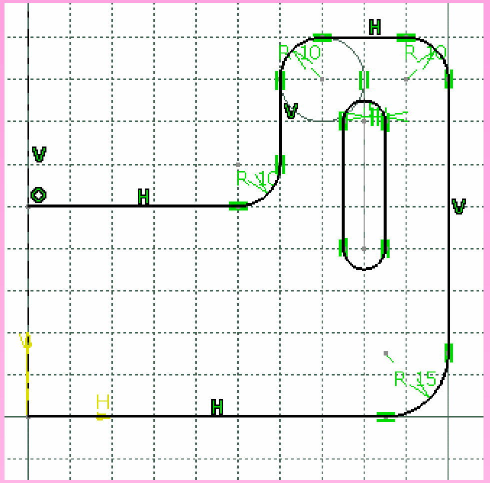


Figure C The sketch after creating the elongated hole

2. Mirror the sketch along the vertical axis of origin, as shown in **Figure D**.

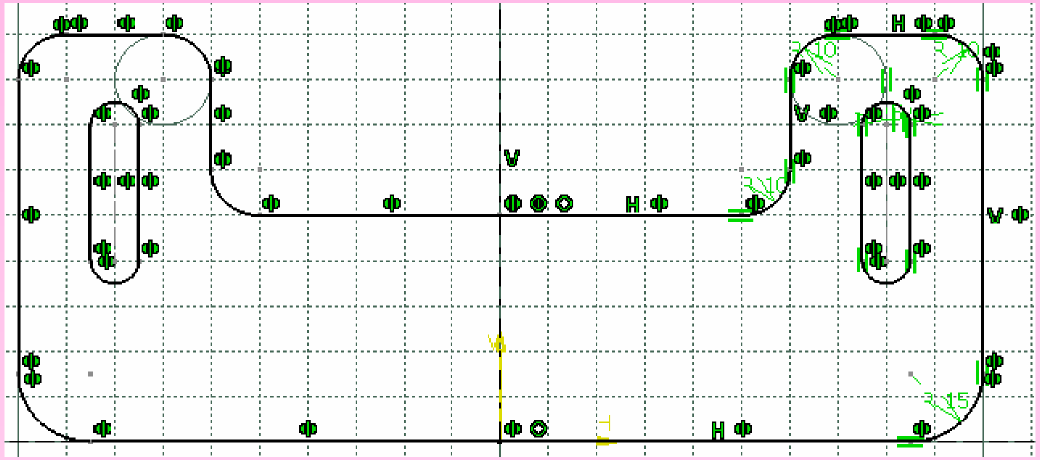


Figure D The sketch after mirroring

3. Draw the elongated hole in the lower portion of the sketch, as shown in **Figure E**.

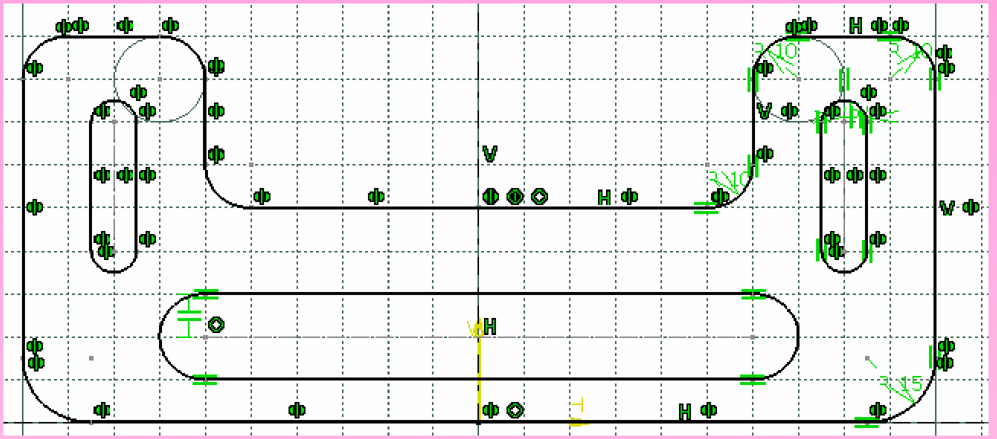


Figure E The sketch after creating the horizontal elongated hole

4. Save the file in **\My Documents\CATIA\c02 folder** and then close it.

Exercise 1

Draw the sketch of the model shown in **Figure A**. The sketch to be drawn is shown in **Figure B**. Do not dimension it. The solid model and dimensions are given for your reference. **(Expected time: 30 min)**

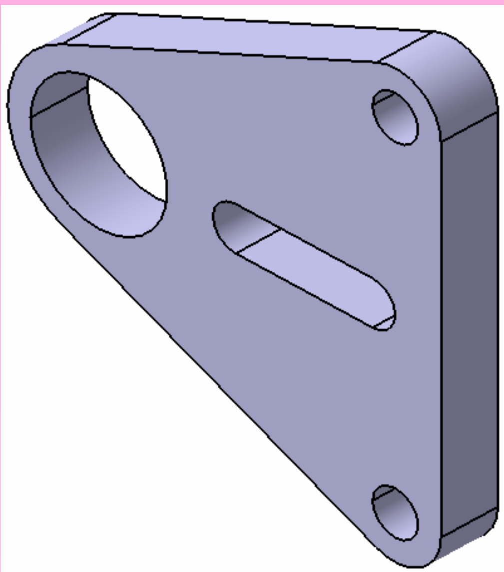


Figure A The model for Exercise 1

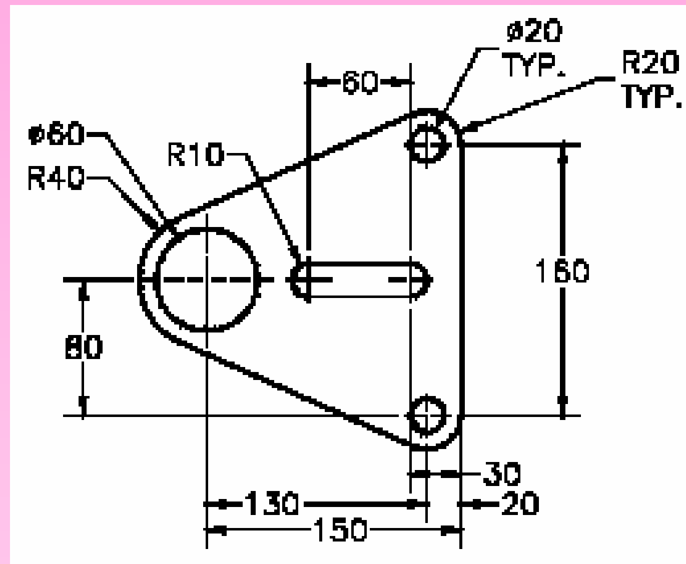


Figure B The sketch for Exercise 1

Exercise 2

Draw the sketch of the model shown in **Figure A**. The sketch to be drawn is shown in **Figure B**. Do not dimension the sketch. The solid model and dimensions are given only for your reference. **(Expected time: 30 min)**

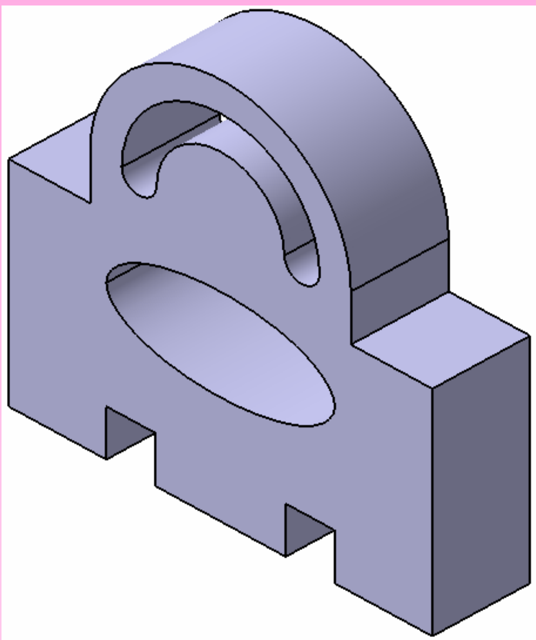


Figure A The model for Exercise 2

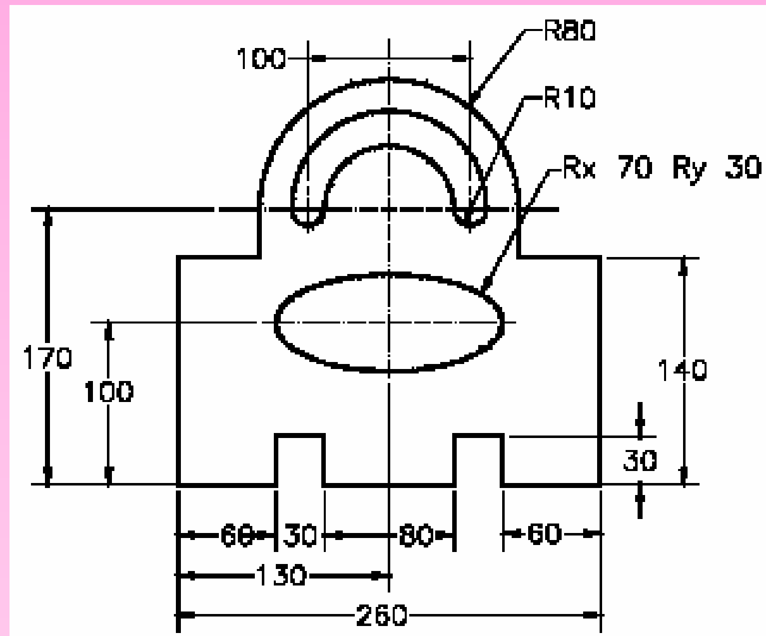


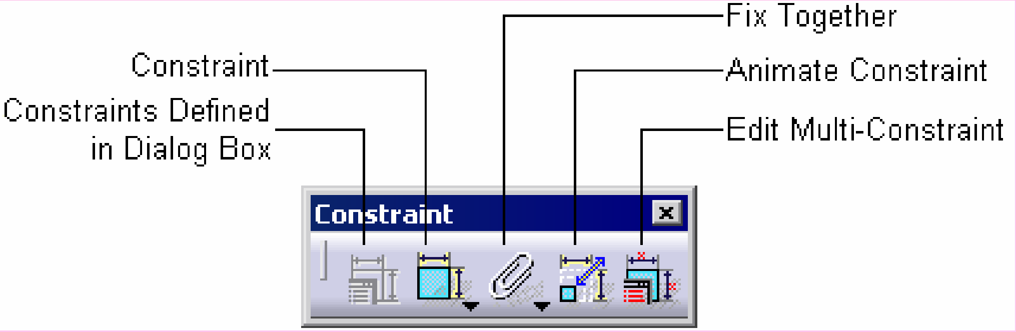
Figure B The sketch for Exercise 2

Learning Objectives:

- *Add geometrical constraints to sketches.*
- *Add dimensional constraints to sketches.*
- *Understand the concept of iso-constraint, under-constraint, and over-constraint sketches.*
- *Analyze and delete over defining constraints.*
- *Create base features by extruding sketches.*
- *Create base features by revolving sketches.*
- *Dynamically rotate the model view .*
- *Modify the view orientation*
- *Switch between various display modes*
- *Assign material to the model*

CONSTRAINING SKETCHES

- You need to constrain the sketches so as to restrict their degrees of freedom and make them stable.
- The first step is to apply the geometrical constraints to the sketch, some of which are automatically applied, while drawing.
- After applying the remaining geometrical constraints, you need to add dimensional constraints, using the tools in the **Constraint** toolbar, as shown in the figure.



Tools in the Constraint toolbar

CONCEPT OF ISO-CONSTRAINED SKETCHES

Generally, after drawing the sketch and applying the constraints, the sketch can exist in any one of the following five stages:

- **Iso-Constrained**

  An Iso-Constraint sketch, also known as fully constraint sketch, is the one in which all degrees of freedom, of each element, are defined using the geometric and dimensional constraints.

- **Under-Constrained**

  An Under-Constrained sketch is one in which all degrees of freedom, of each entity are not completely defined using constraints.

- **Over-Constrained**

  An Over-Constrained sketch is the one in which some extra constraints are applied.

- **Inconsistent**

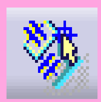
  The Inconsistent stage of the sketch occurs when a change is made to the sketch, but the its current geometry cannot accommodate that change.

- **Not Changed**

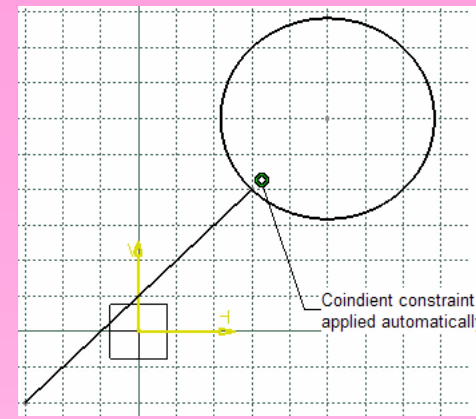
  The Not Changed stage of the sketch occurs when the sketched elements are over defined and you change the value of the dimensional constraints.

ADDING GEOMETRICAL CONSTRAINTS

Adding Automatic Constraints

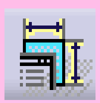


This tool is active by default and therefore, as you draw an element, some constraints are automatically applied to it.

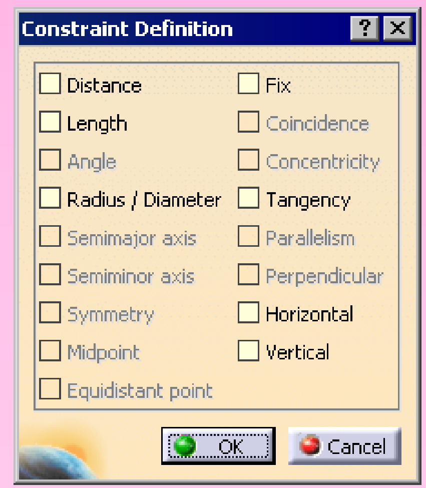


The coincident constraint applied automatically

Applying Additional Constraints to the Sketch



You need to manually apply additional constraints to the sketch by defining them using the **Constraint Definition** dialog box, as shown in the figure.

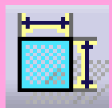


The Constraint Definition dialog box

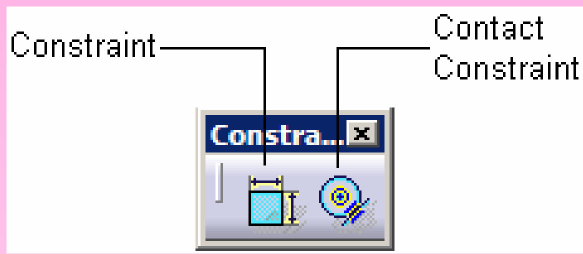
The constraints that are provided in the **Constraint Definition** dialog box are:

- **Distance**
- **Length**
- **Angle**
- **Radius / Diameter**
- **Semimajor axis**
- **Semiminor axis**
- **Symmetry**
- **Midpoint**
- **Equidistant point**
- **Fix**
- **Coincident**
- **Concentricity**
- **Tangency**
- **Parallelism**
- **Perpendicular**
- **Horizontal**
- **Vertical**

• Applying Dimensional Constraints



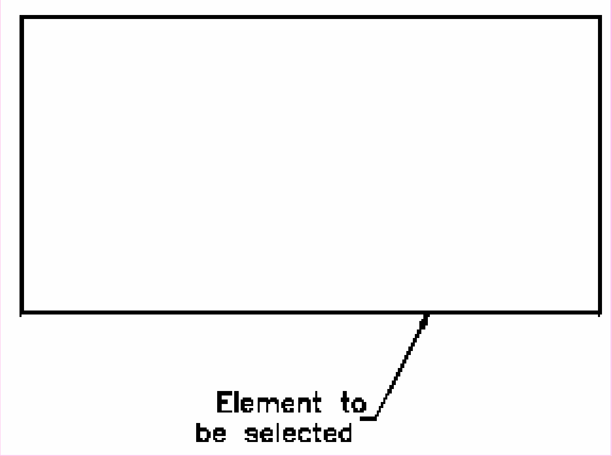
- After applying the geometric constraints, you need to apply the dimensional constraints to fully define the sketches, using the **Constraint** tool.
- Choose the **Constraint** button from the **Constraint** toolbar or the **Constraint Creation** toolbar shown in the figure.



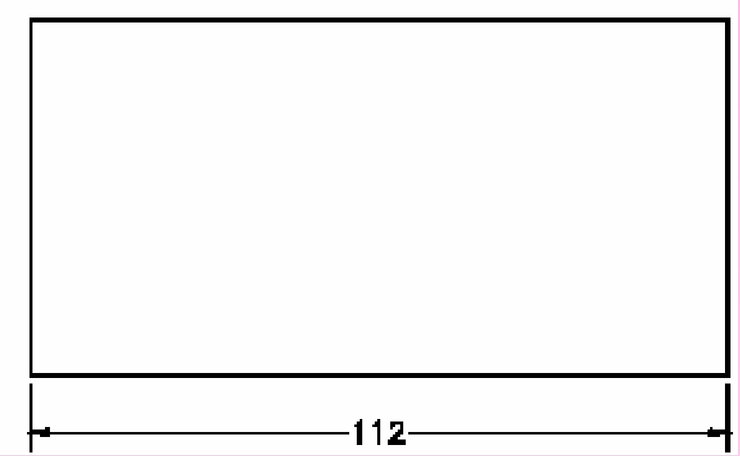
The Constraint Creation toolbar

- **Linear Dimensioning of a Line and Between Two Points**
  - To apply a linear dimension between two points, invoke the **Constraint** tool, and select two points from the geometry area.
  - Next, right-click to display the contextual menu, as shown in the figure.

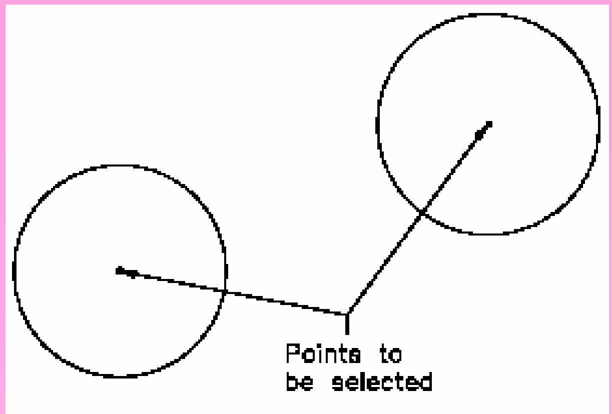
The contextual menu



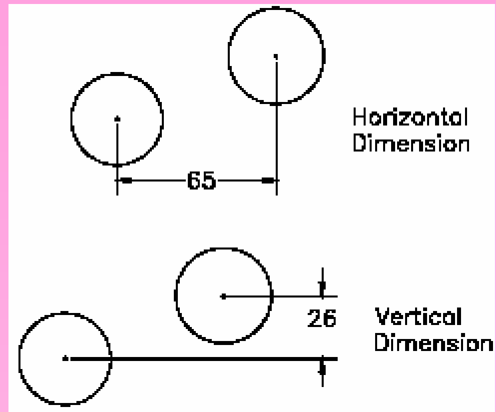
Element selected to be dimensioned



Resulting linear dimension



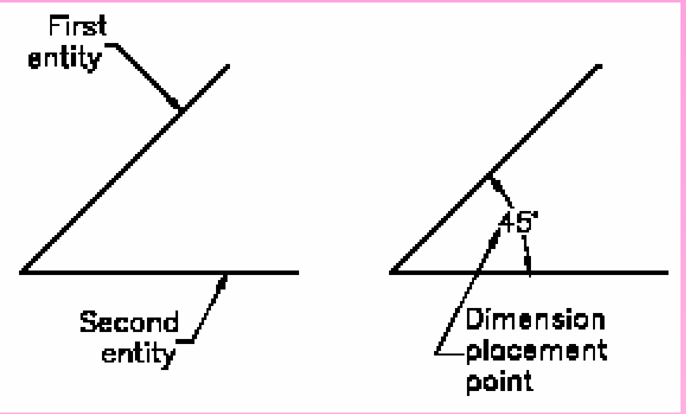
Elements selected to be dimensioned



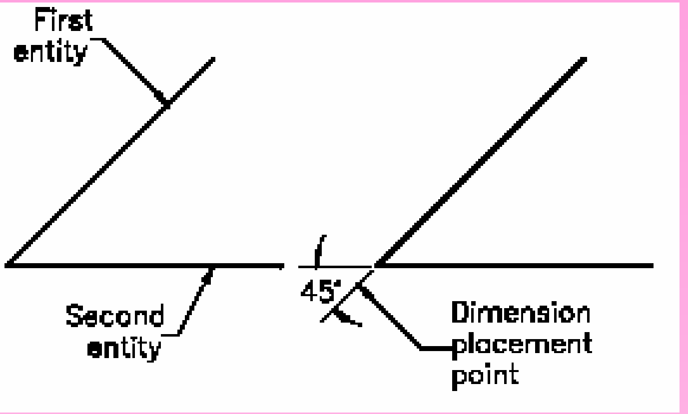
Resulting linear dimension

- **Dimensioning An Inclined Line**
  - By default, whenever you select an inclined line, the aligned dimension is applied to it.
  - You can also apply a horizontal or vertical dimension to it.
- **Dimensioning An Arc and A Circle**
  - By default, the diameter dimension is applied to circles and the radius dimension is applied to arcs.
  - Invoke the **Constraint** tool and select the arc or the circle that you need to dimension.

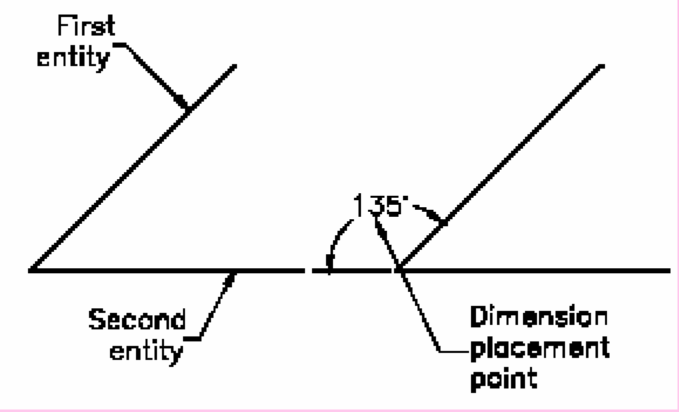
- Applying an Angular Dimension



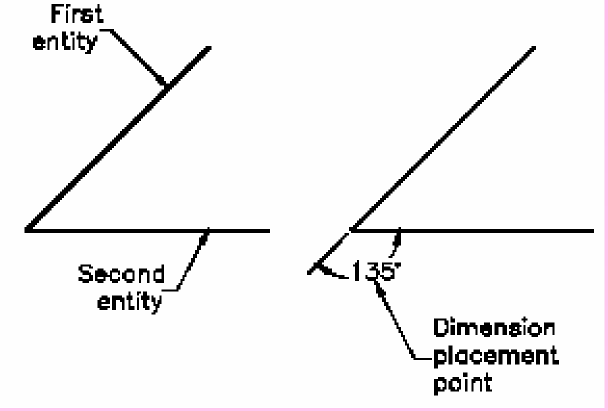
Angular dimension placed according to the placement point



Angular dimension placed according to the placement point

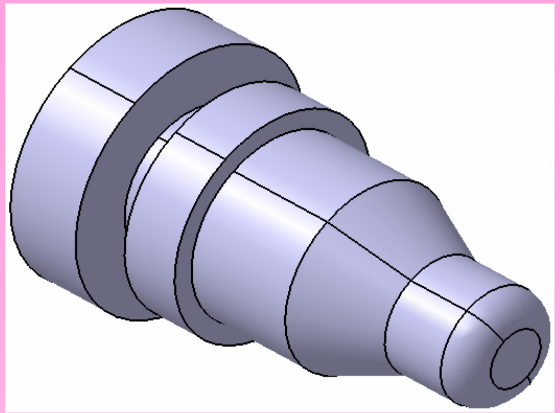


Angular dimension placed according to the placement point

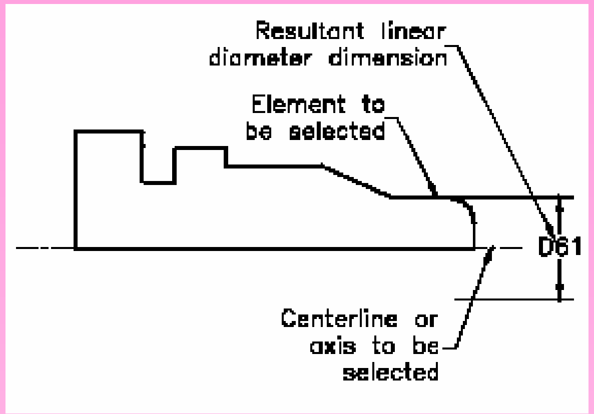


Angular dimension placed according to the placement point

- **Applying Linear Diameter Dimensions**



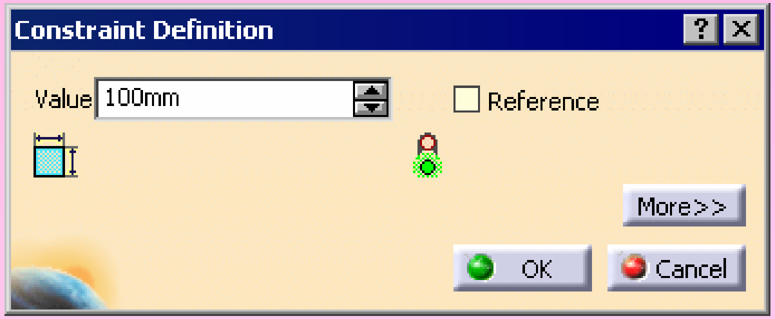
Model created by revolving a sketch around the horizontal center line



Element and center line selected and the resulting linear diameter dimension

- **Modifying Dimensions After Placing**

To modify the dimension value, double-click on it; the **Constraint Definition** dialog box is displayed, as shown in the figure.



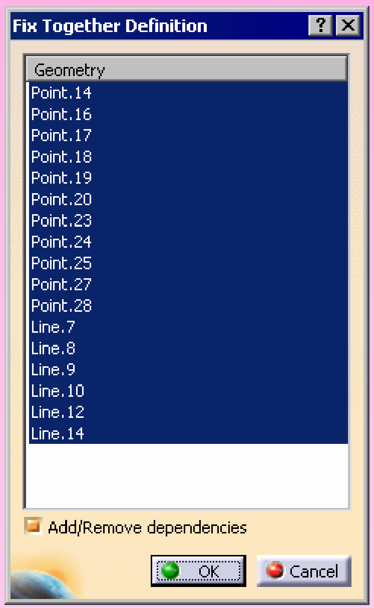
The Constraint Definition dialog box invoked after double-clicking the dimension

• Applying Contact Constraints

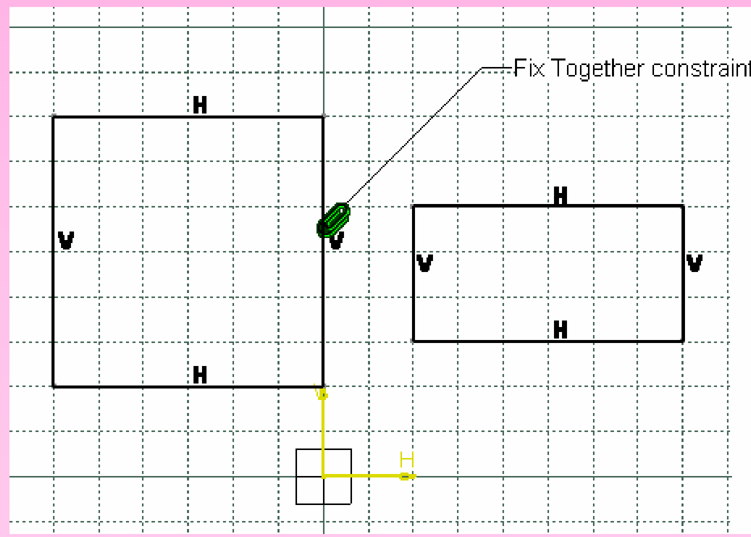
The **Contact Constraint** tool is used to automatically apply geometrical constraints to the selected elements, depending on their position and geometry.

• Applying Fix Together Constraints

To apply this constraint, select all the entities that you need to fix together, and choose the **Fix Together** button from the **Constrained Geometries** toolbar; the **Fix Together Definition** dialog box will be displayed, as shown in the figure.

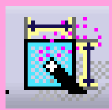


The Fix Together Definition dialog box

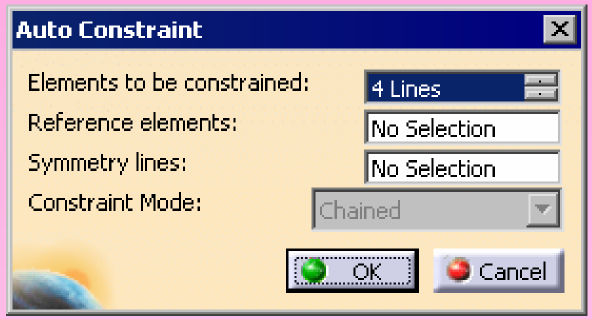


The Fix Together constraint applied to the sketched entities

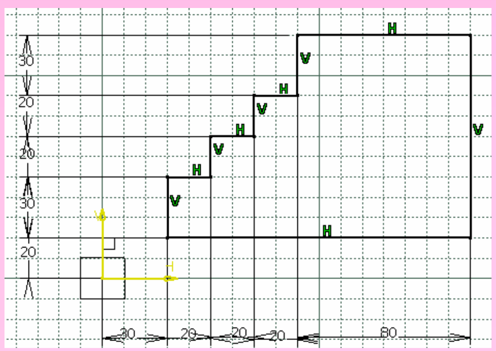
• Applying Automatic Constraints



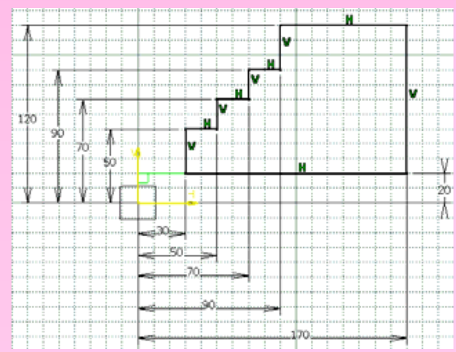
- To apply constraints, select the elements to be constrained, and choose the **Auto Constraint** button in the Fix Together flyout in the **Constraint** toolbar; the **Auto Constraint** dialog box is
- The number of elements selected is displayed in the **Elements to be constrained** display box, as shown in the figure.



The Auto Constraint dialog box

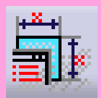


Automatic dimensions applied using the Chained option

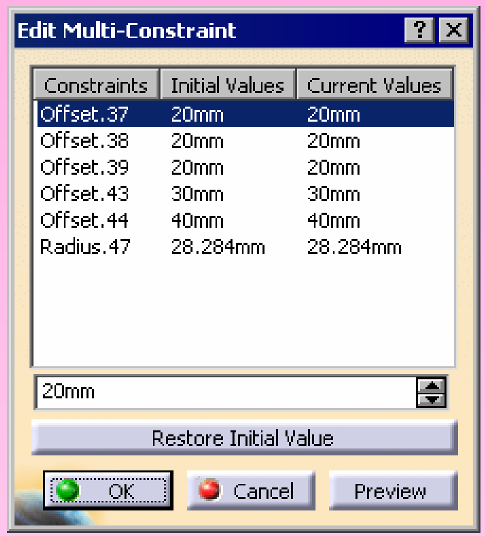


Automatic dimensions applied using the Stacked option

• Editing Multiple Dimensions



- To use the **Edit Multi-Constraint** tool, you first need to apply all the required dimensions to the sketch.
- Next, choose the **Edit Multi-Constraint** button from the **Constraint** toolbar; the **Edit Multi-Constraint** dialog box will be displayed, as shown in the figure.

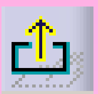


The Edit Multi-Constraint dialog box

ANALYZING AND DELETING OVER DEFINING CONSTRAINTS

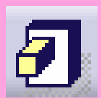
- After applying dimensions to the sketches, if the sketch is over-constrained, you need to delete the over defining constraints, which are displayed in purple.
- Select the overdefining constraint and press the DELETE key.

EXITING THE SKETCHER WORKBENCH



Choose the **Exit workbench** button from the **Workbench** toolbar; you will exit the **Sketcher** workbench and the **Part Design** workbench is invoked.

CREATING BASE FEATURES BY EXTRUDING



- To invoke the **Pads** toolbar, choose the down arrow on the right of the **Pad** button in the **Sketch-Based Features** toolbar.

- The **Pads** toolbar is shown in **Figure A**.

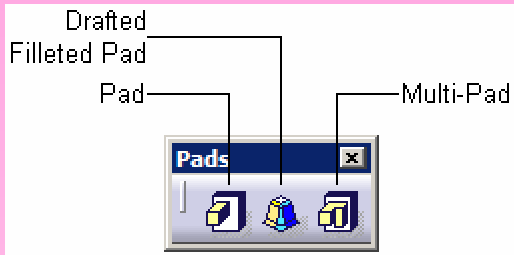


Figure A The Pads toolbar

- Select the sketch and then choose the **Pad** button from the **Pads** toolbar; the **Pad Definition** dialog box is displayed, as shown in **Figure B**.

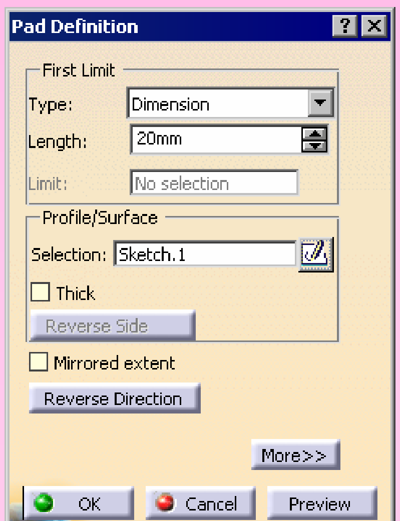
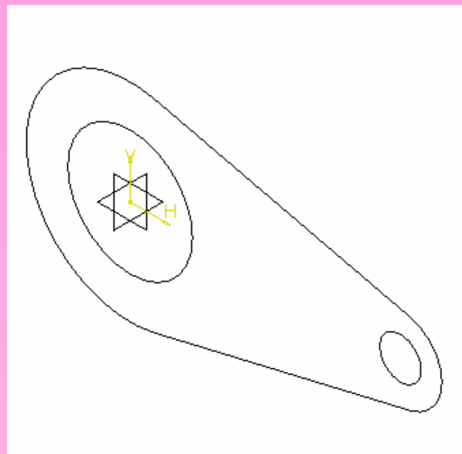
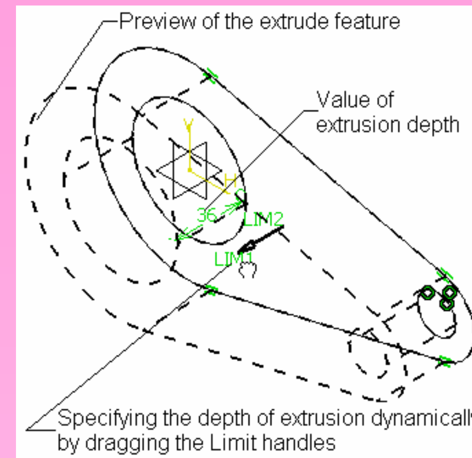


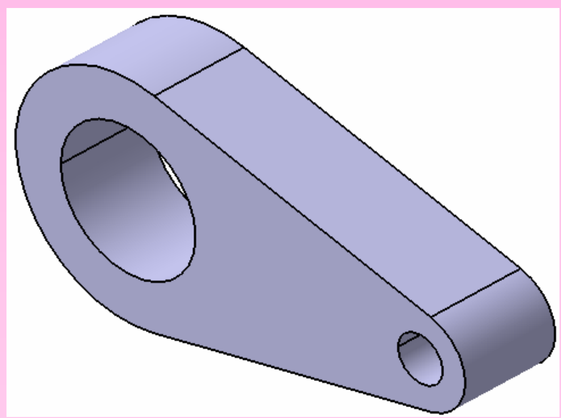
Figure B The Pad Definition dialog box



The sketch after exiting the **Sketcher** workbench



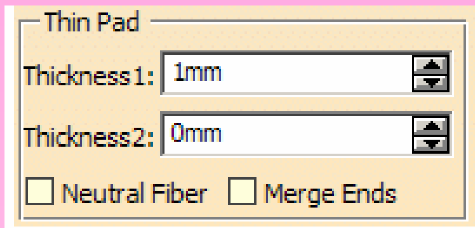
Dynamically dragging the **Limit** handle to specify the depth of extrusion



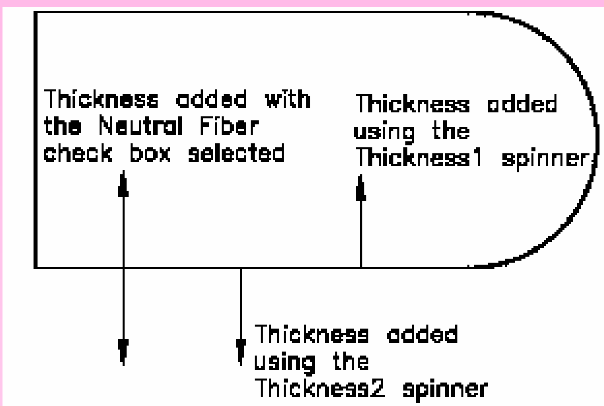
The model after extruding

• Creating a Thin Extruded Feature

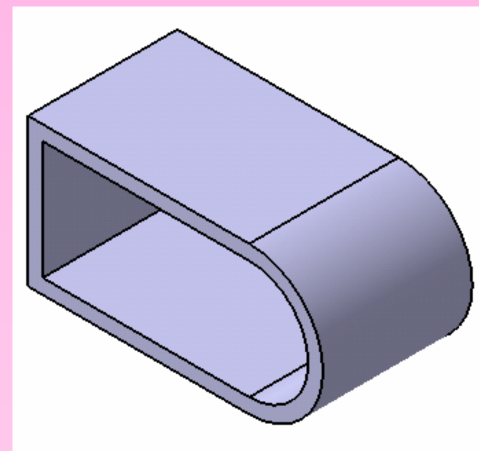
- To create a thin extruded feature, choose the **Thick** check box from the **Pad Definition** dialog box.
- The dialog box expands and the options in the **Thin Pad** area of the **Pad Definition** dialog box are invoked, as shown in the figure.



The Thin Pad area of the Pad Definition dialog box



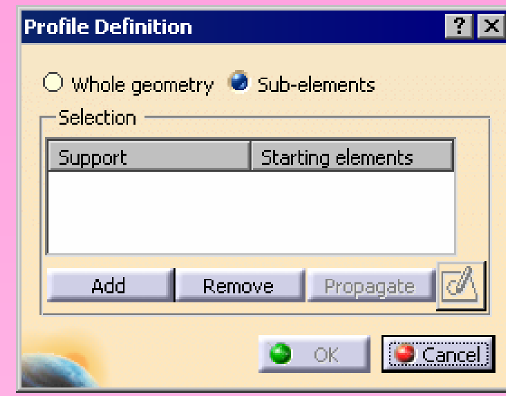
Thickness added using different options



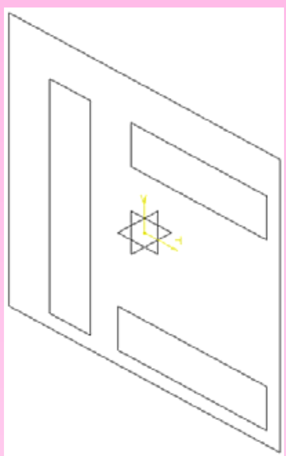
An extruded feature created using the Thin option

• Extruding the Sketch by Selecting the Profile Using the Profile Definition Dialog Box

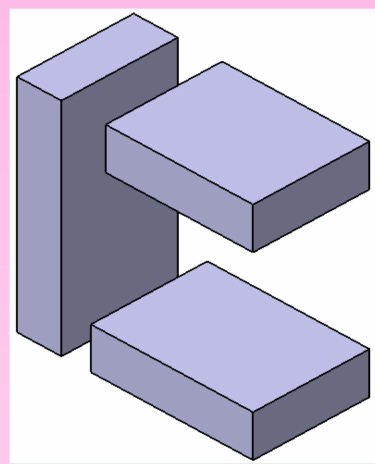
- To create a feature by selecting a particular contour from a multi-contour sketch, draw the sketch and invoke the **Pad Definition** dialog box.
- Right-click on the **Selection** selection area and choose the **Go to profile definition** option.
- The **Profile Definition** dialog box is displayed, as shown in the figure.



The Profile Definition dialog box

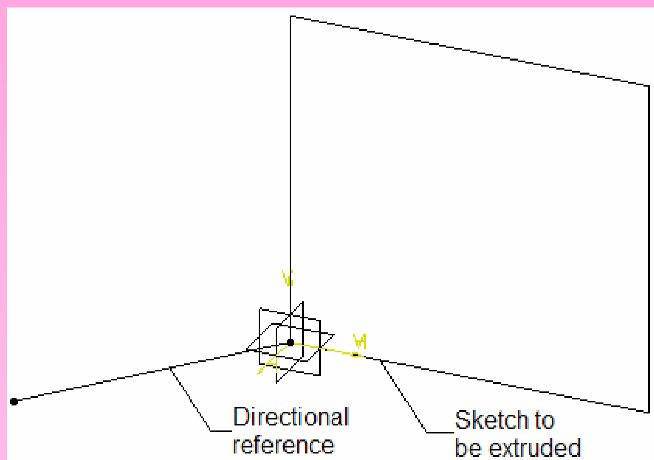


A multi-contoured sketch

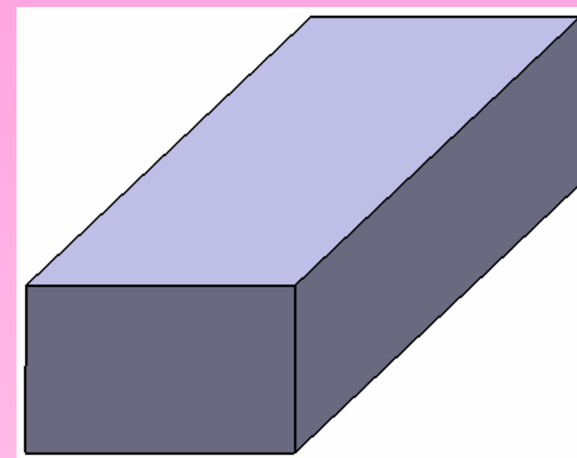


Extruded contours

- **Extruding the Sketch Along a Directional Reference**



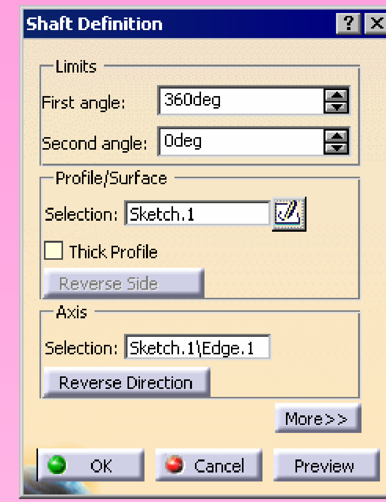
Sketch to be extruded and the directional reference



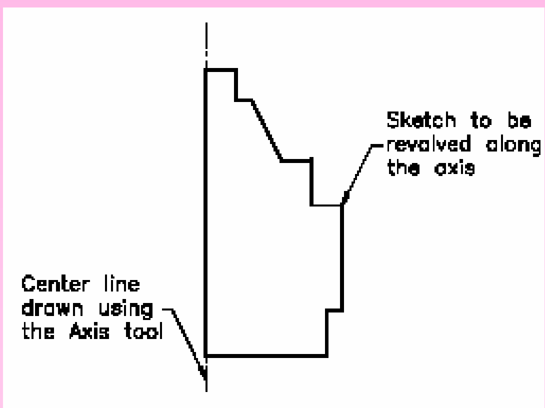
The extrude feature created by extruding a sketch along the directional reference

CREATING BASE FEATURES BY REVOLVING SKETCHES

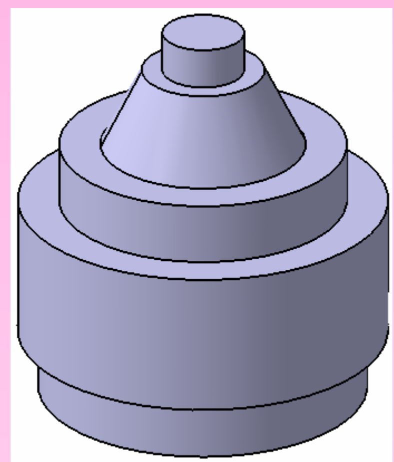
- To create a revolved feature, you first need to draw the sketch that will be revolved around a center line, which is also known as axis.
- Next, exit the **Sketcher** workbench and choose the **Shaft** button from the **Sketch-Based Features** toolbar; the **Shaft Definition** dialog box will be displayed, as shown in the figure.



The **Shaft Definition** dialog box



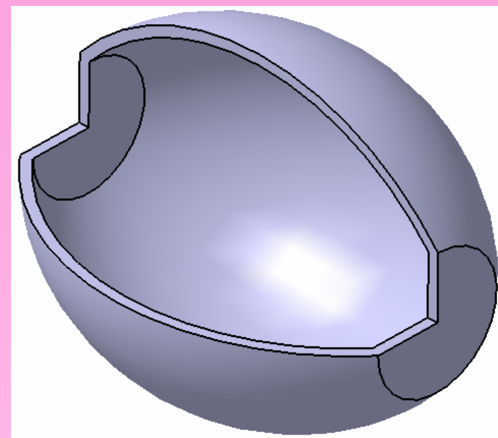
The sketch to be revolved and the axis



The model after creating the shaft feature

- **Creating Thin Shaft Features**

You can also create a thin revolved feature, using the **Shaft** tool.

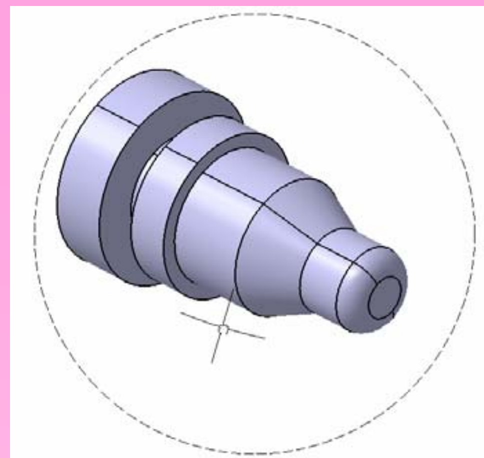


The model after creating the thin shaft feature using an open profile

➢ DYNAMICALLY ROTATING THE VIEW OF THE MODEL

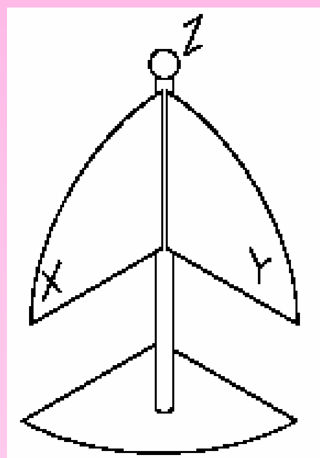
- CATIA V5 allows you to rotate the view of the model dynamically in the 3D space so that the solid model can be viewed from all directions.
- This tool can be invoked even when you are inside some other tool.

- **Rotating the View Using the Rotate Tool**



View of the model being rotated using the **Rotate** tool

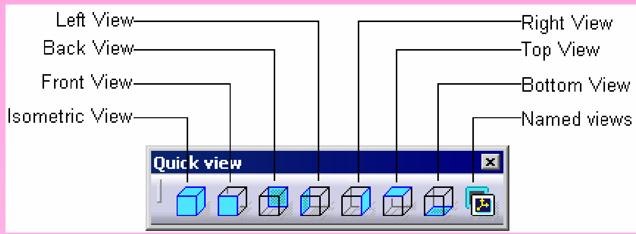
- **Rotating the View Using the Compass**



The Compass

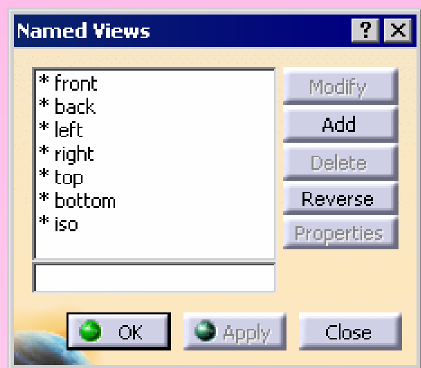
MODIFYING THE VIEW ORIENTATION

- To invoke the **Quick View** toolbar toolbar, choose the down arrow on the right of the **Isometric** View button in the **View** toolbar.
- The **Quick view** toolbar is shown in the figure.



The Quick view toolbar

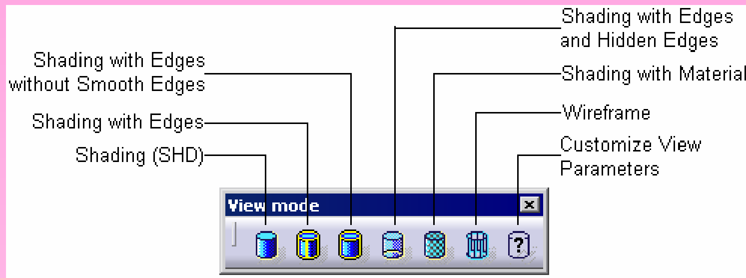
- This release of CATIA V5 includes the **Named views** tools in this toolbar.
- When you choose the **Named views** button, the **Named Views** dialog box will be displayed, as shown in the figure.



The Named Views dialog box

➢ DISPLAY MODES OF THE MODEL

- To invoke the **View** mode toolbar, choose the down arrow on the right of the **Shading with Edges** button from the **View** toolbar.
- The **View mode** toolbar is displayed in the figure.



The View mode toolbar

- The tools available in the **View mode** are discussed next:

• Shading (SHD)

When you choose the **Shading (SHD)** button from the **View mode** toolbar, the model will be displayed in the shaded mode.

• Shading with Edges

The **Shading with Edges** mode is the default mode, in which the model is displayed.

• Shading with Edges without Smooth Edges

When you choose the **Shading with Edges without Smooth Edges** button, the model will be displayed in shading with edges but the display of the smooth edges will be removed.

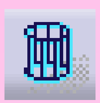
• Shading with Edges and Hidden Edges

When you choose the **Shading with Edges and Hidden Edges** button, the model will be displayed in shading with edges, and hidden edges will also be displayed along with the visible edges.

• Shading with Material

When you choose the **Shading with Material** button, the model will be displayed in the rendered mode.

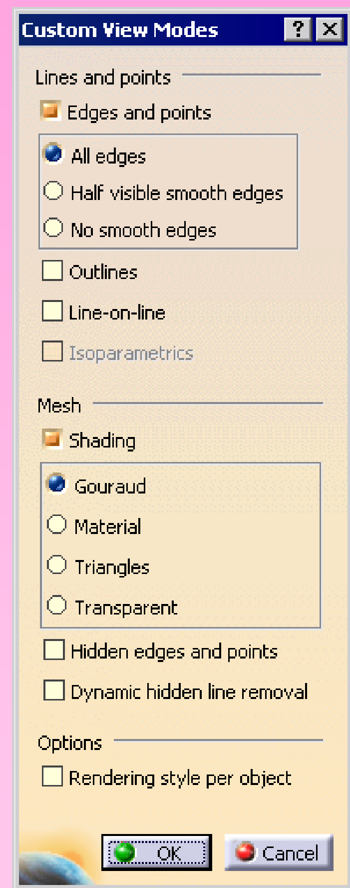
• Wireframe (NHR)



When you choose the **Wireframe(NHR)** button, the model will be displayed in the wireframe mode.

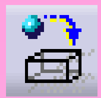
• Customize View Parameters

When you choose the **Customize View Parameters** button, the **Custom View Modes** dialog box will be displayed, as shown in the figure.



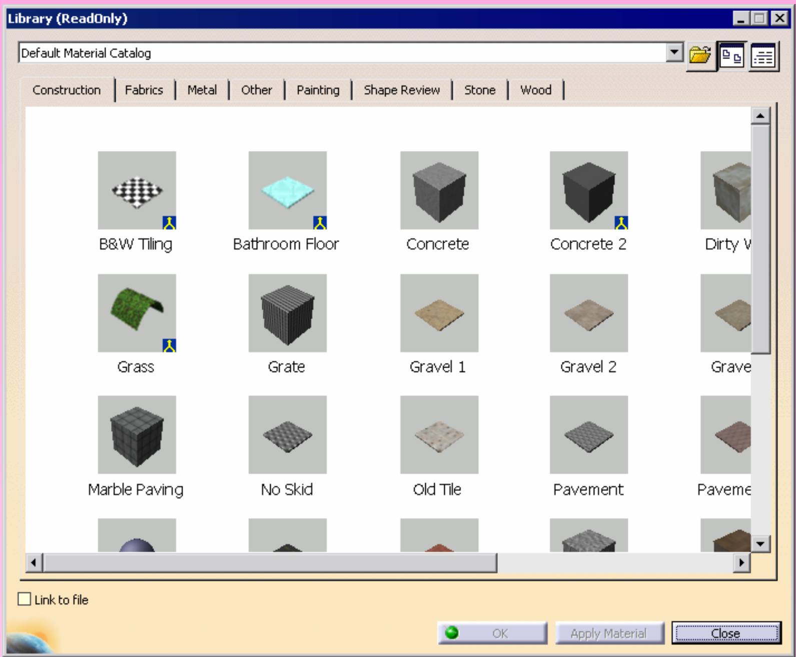
The Custom View Modes dialog box

➢ ASSIGNING MATERIAL TO THE MODEL

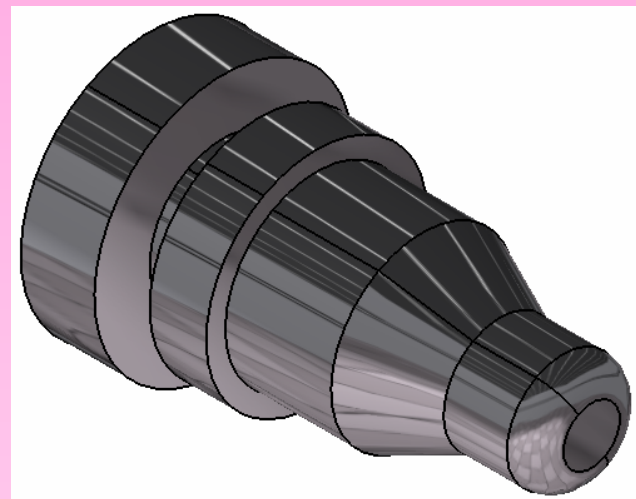


• CATIA V5, also allows you to assign material to the models in the **Part Design** workbench.

• All the physical properties of the material applied to model, are assigned to model.



The **Library (ReadOnly)** dialog box



The model, after assigning Aluminium material

Tutorial 1

In this tutorial, you will create the model shown in **Figure A**. Its dimensions are shown in **Figure B**. After creating this model, apply Copper material and then rotate the view of the model in the 3D space. **(Expected time: 20 min)**

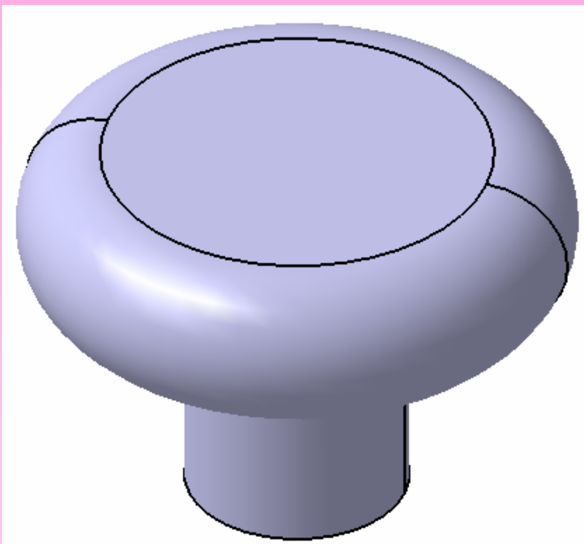


Figure A Solid model for Tutorial 1

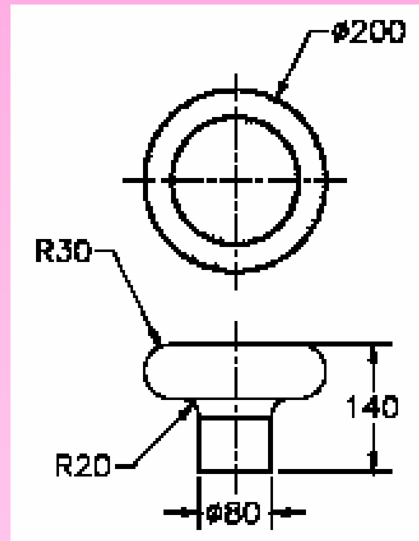


Figure B Views and dimensions for Tutorial 1

1. Start a new file in the **Part Design** workbench of CATIA V5.
2. Invoke the **Sketcher** workbench by selecting the yz plane from the specification tree.
3. Draw the sketch using the tools in the **Sketcher workbench** and then add the required geometrical and dimensional constraints, as shown in **Figure C**, **Figure D** and **Figure E**.

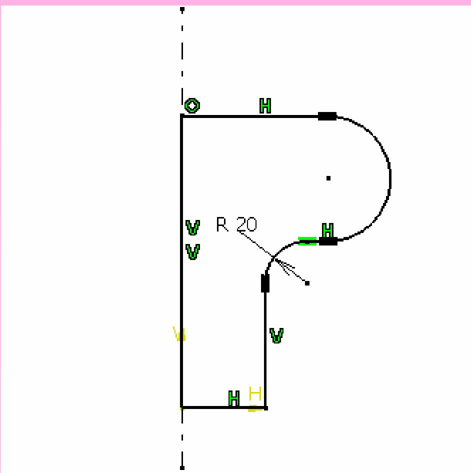


Figure C Sketch drawn using the **Axis** and **Profile** tool

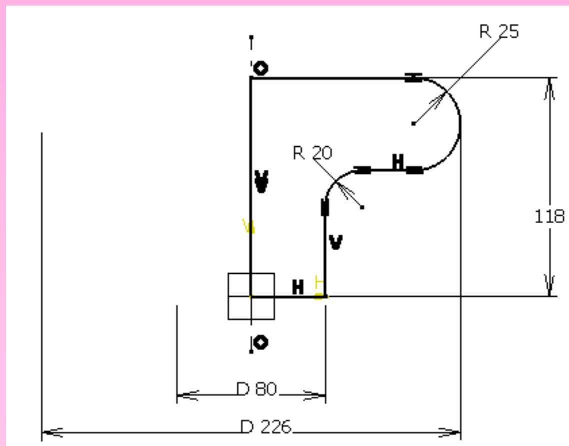


Figure D The sketch after applying all constraints

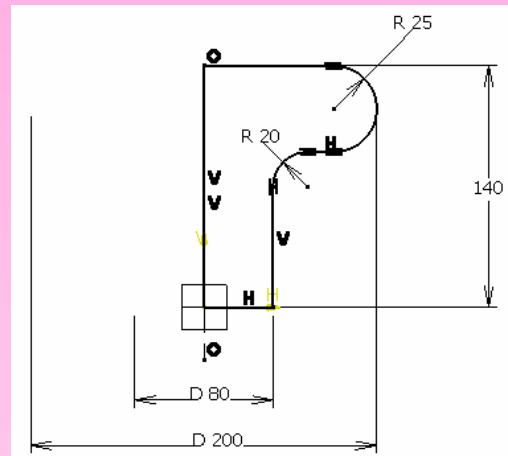


Figure E Final sketch for the base feature

4. Invoke the **Shaft** tool to create the shaft feature, as shown in **Figure F**.

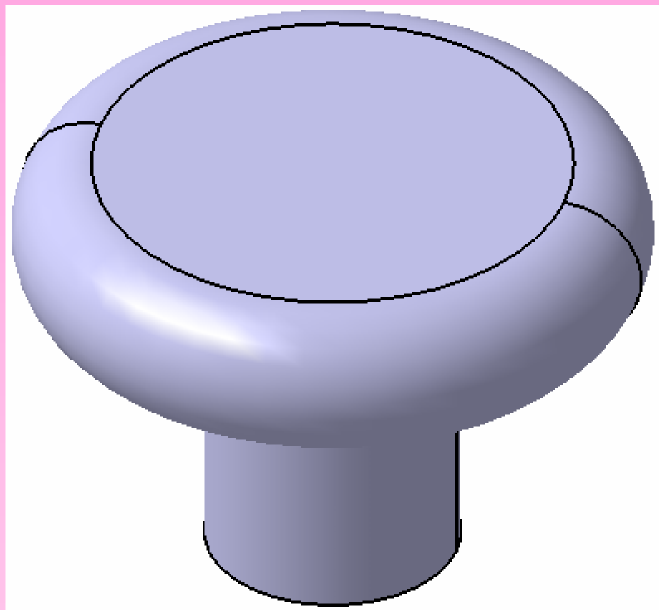


Figure F Final model for Tutorial 1

5. Assign a material to the model and rotate its view in the 3D space, as shown in **Figure G** and **Figure H**.

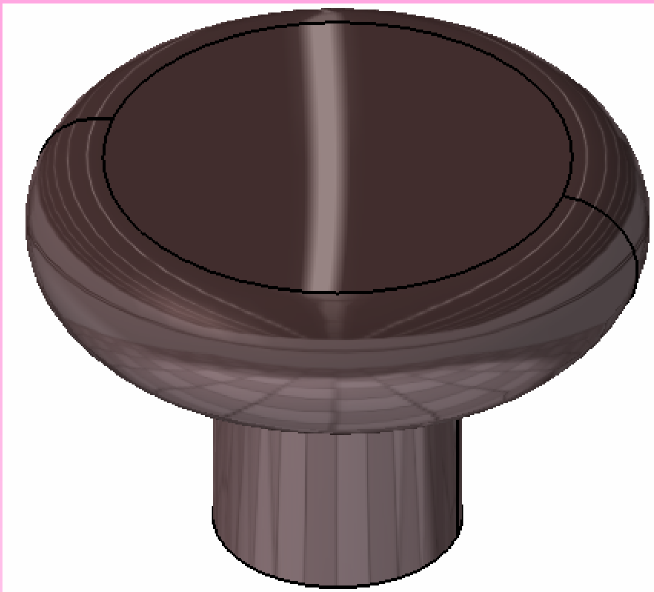


Figure G The model after assigning the Copper material

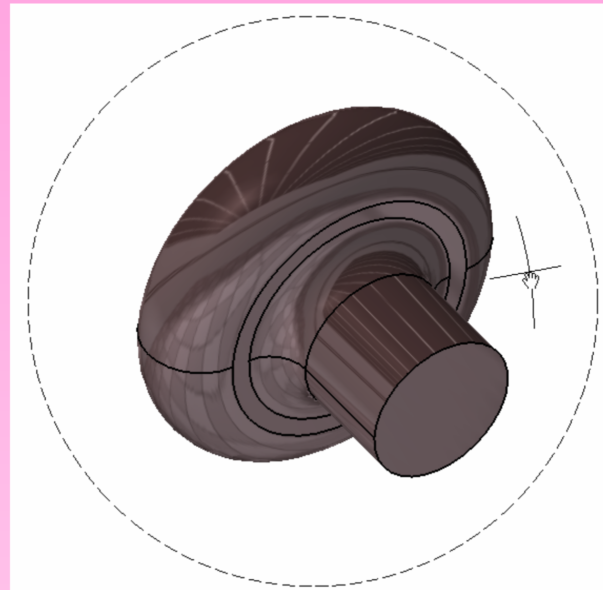


Figure H The view of the model being rotated

6. Save the file in **\My Documents\CATIA\c03** folder and then close it.

Tutorial 2

In this tutorial, you will create the base feature of the model shown in **Figure A** by extruding a sketch drawn on the yz plane. You will then apply the Aluminium material to the model and then rotate its view in the 3D space. The dimensions of the model are shown in **Figure B**. **(Expected time: 20 min)**

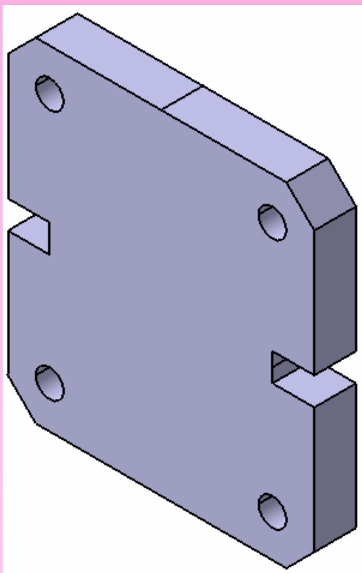


Figure A **Solid model for Tutorial 2**

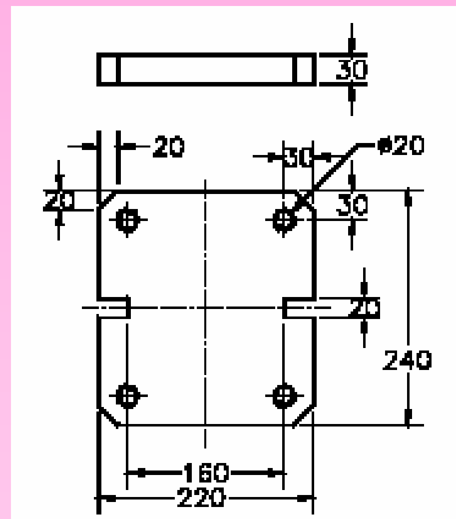


Figure B **Views and dimensions for Tutorial 2**

1. Start a new file in the **Part Design** workbench.

2. Select the yz plane from the geometry area and invoke the **Sketcher** workbench.

3. Draw the sketch of the base feature using the tools in the **Sketcher** workbench, as shown in **Figure C**, **Figure D**, **Figure E** and **Figure F**.

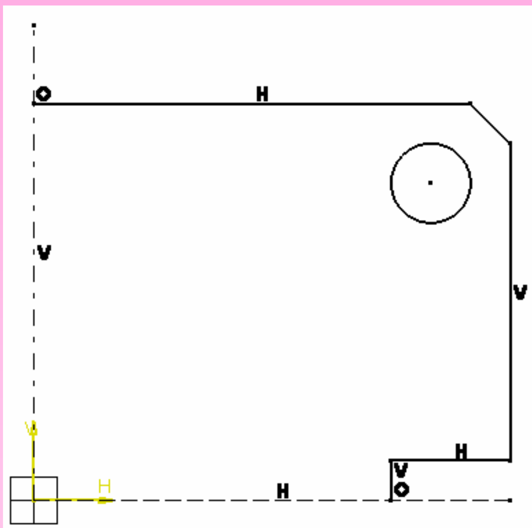


Figure C One quarter of the sketch

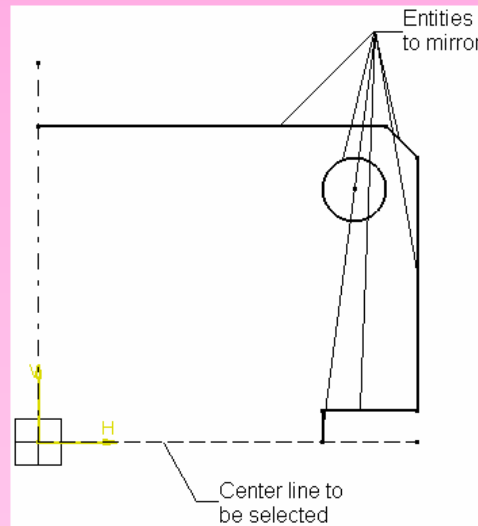


Figure D Elements to be selected

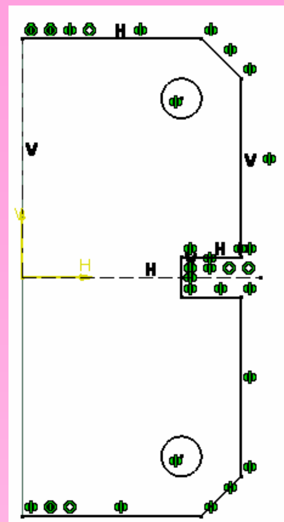


Figure E Resulting mirrored sketch

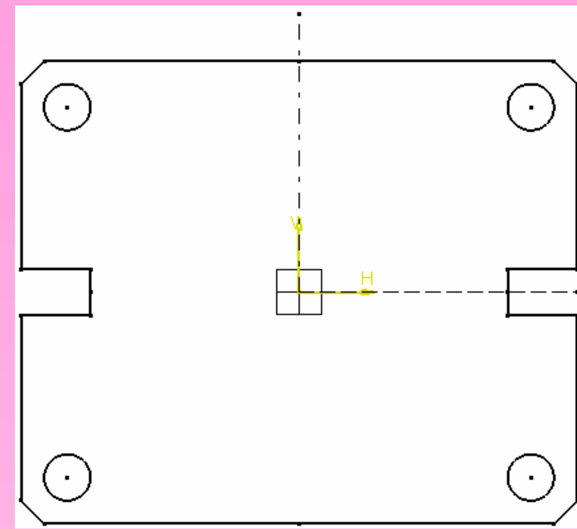


Figure F Resulting mirrored sketch

4. Apply geometrical and dimensional constraints to the sketch to make it iso-Constraint, as shown in **Figure G**.

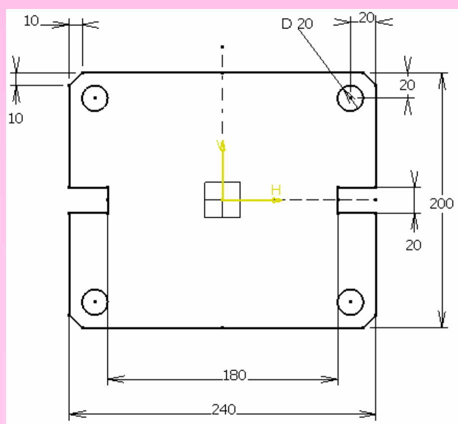


Figure G The Sketch after applying all dimensions

5. Exit the **Sketcher** workbench and invoke the **Pad Definition** dialog box.

6. Extrude the sketch to the given depth, as shown in **Figure H**.

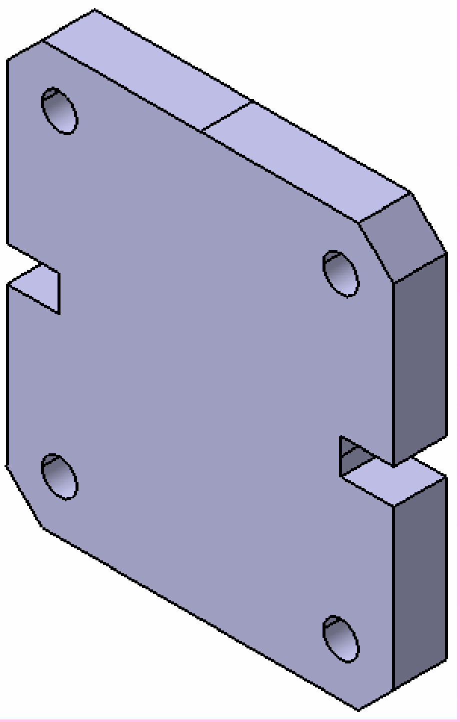


Figure H The model after creating the base feature

7. Assign a material to the model and then rotate it in the 3D space, as shown in **Figure I**.

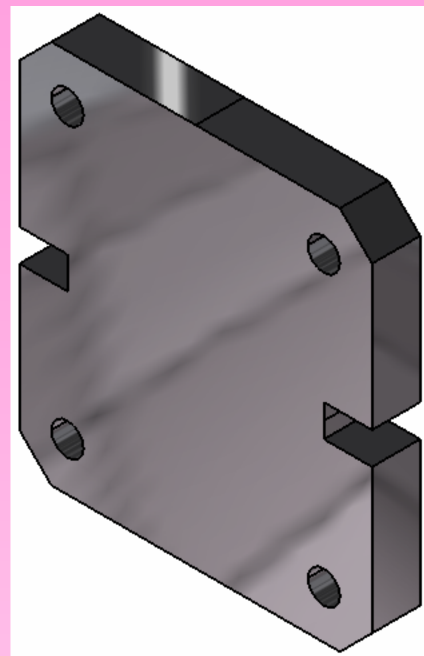


Figure I The model after assigning the Aluminium material

8. Save the file in **\My Documents\CATIA\c03** folder and then close it.

Exercise 1

Create the model shown in **Figure A**. Its dimensions are shown in **Figure B**.

(Expected time: 30 min)

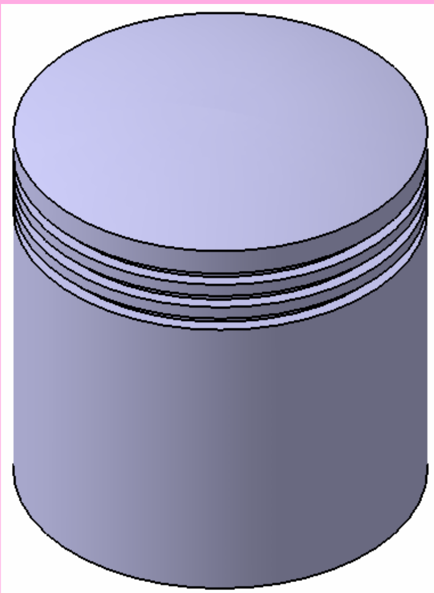


Figure A Solid Model for Exercise 1

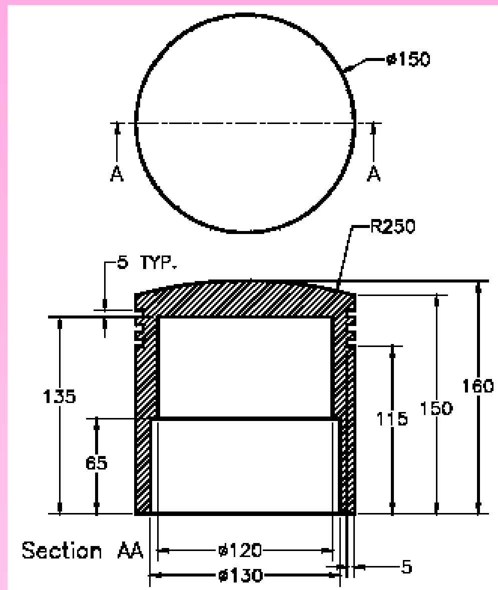


Figure B Views and dimensions for Exercise 1

Exercise 2

Create the model shown in **Figure A**. Its dimensions are shown in **Figure B**.

(Expected time: 30 min)

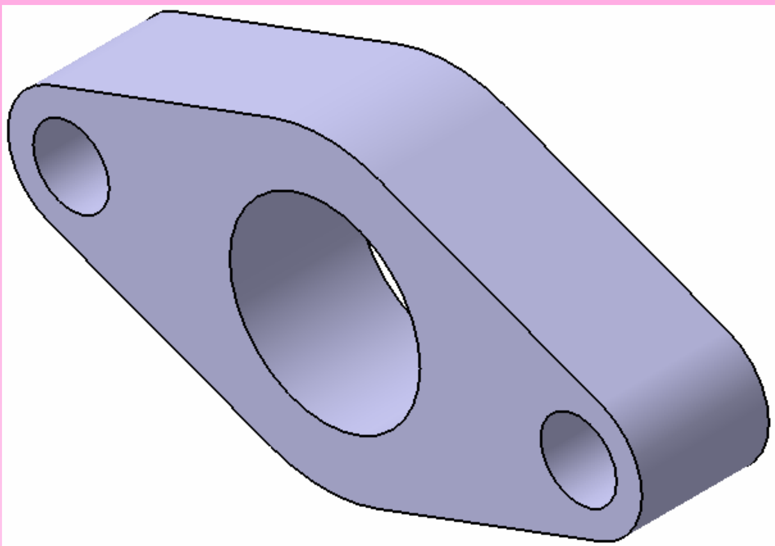


Figure A Solid model for Exercise 2

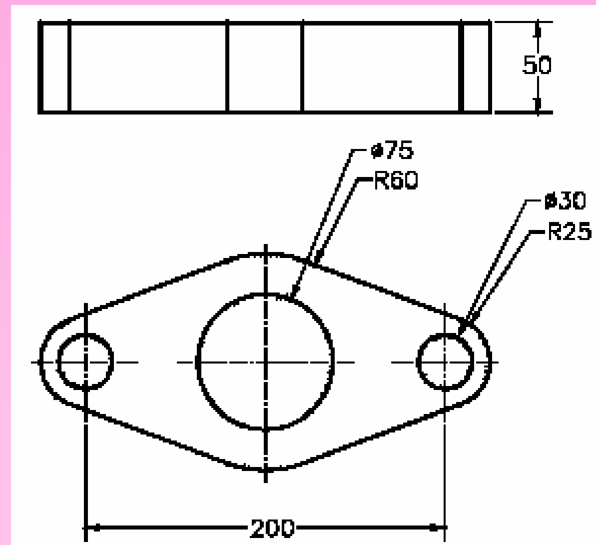


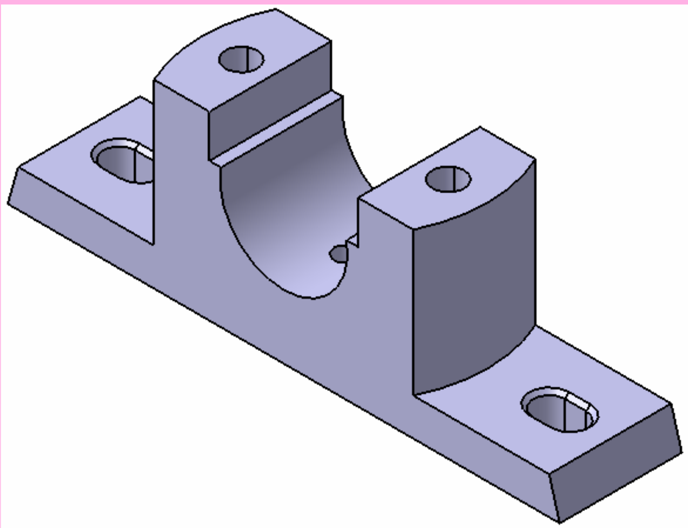
Figure B Views and dimensions for Exercise 2

Learning Objectives:

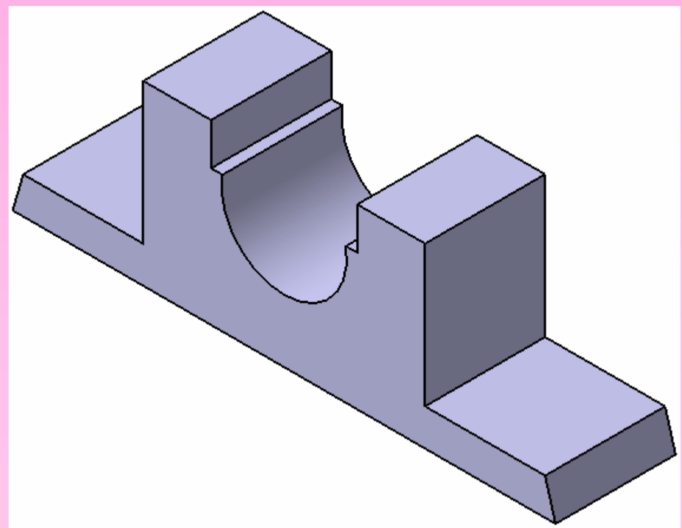
- *Understand the importance of sketching planes.*
- *Create reference elements.*
- *Create drafted filleted pad features.*
- *Create multi-pad features.*
- *Use feature termination options.*
- *Create pocket features.*
- *Create groove features.*
- *Extrude and revolve the planar and curved faces.*

➢ IMPORTANCE OF SKETCHING PLANES

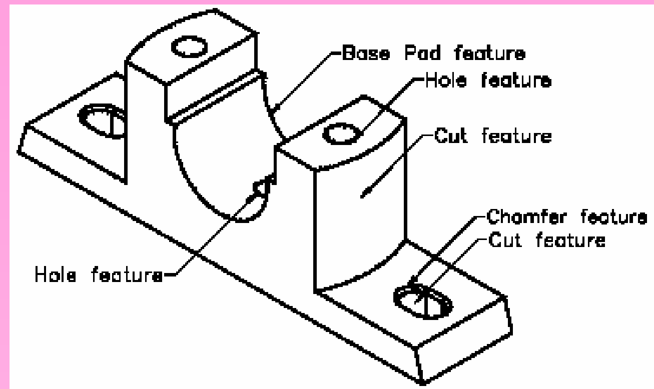
- Most mechanical designs consist of multiple features such as sketched features, referenced elements, and placed features that are integrated to complete a model.
- You can select any one of the default planes as the sketching plane to create the sketch of the base feature.



A multifeatured model



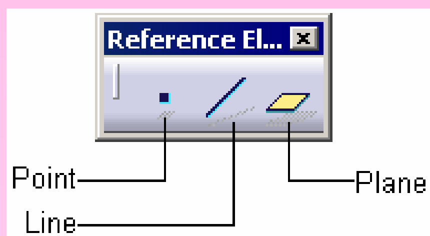
Base feature of the model



Other features of the final model

REFERENCE ELEMENTS

- The reference elements are the features that have no mass and volume and are used only to assist you in the creation of the models.
- You can invoke the tools to create reference elements using the **Reference Elements** toolbar, as shown in the figure.



The Reference Elements toolbar

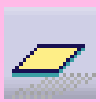
Reference Planes

You need to select other default planes or create new planes to be used as the sketching plane for other features.

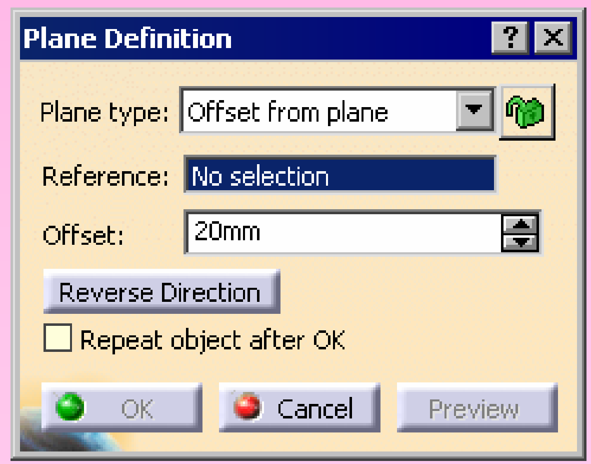
Default Planes

- The three default planes are: xy, yz, and zx plane.
- It is recommended that you carefully select the sketching plane for drawing the sketch of the base feature, which can be drawn on one of the three datum planes provided by default.

Creating New Planes

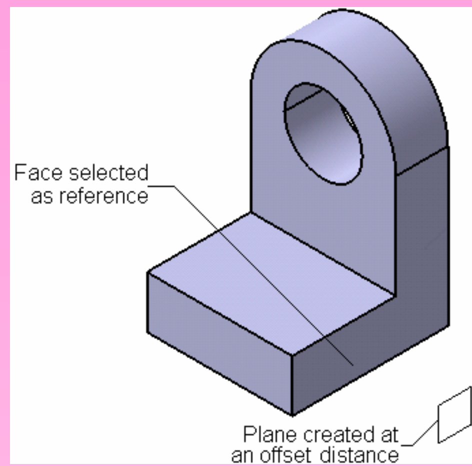


- To create a new plane, choose the **Plane** button from the **Reference Elements (Expanded)** toolbar.
- The **Plane Definition** dialog box is displayed, as shown in the figure.



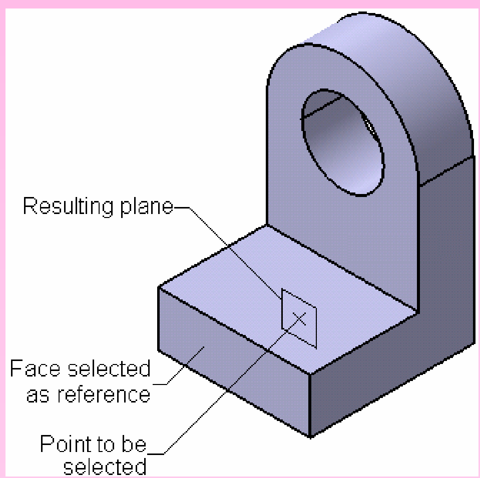
The Plane Definition dialog box

- Creating a Plane at an Offset from Existing Plane/Planar Face



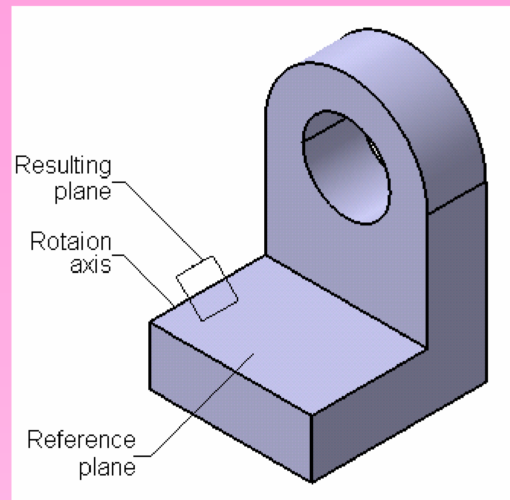
Reference planar face and the resulting plane

- Creating a Plane Parallel to an Existing Plane and Passing Through Point



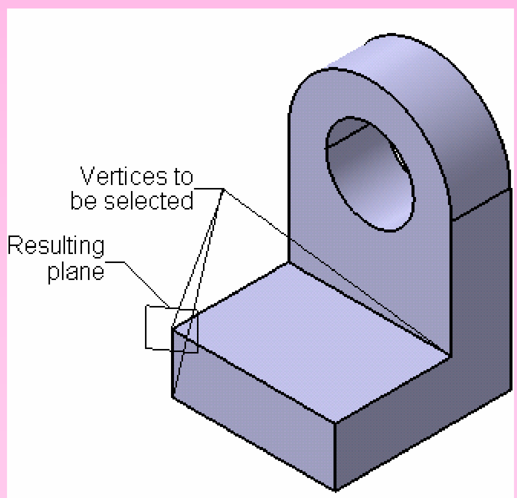
Reference planar face, sketched point, and the resulting plane

- Creating a Plane at an Angle/Normal to Plane



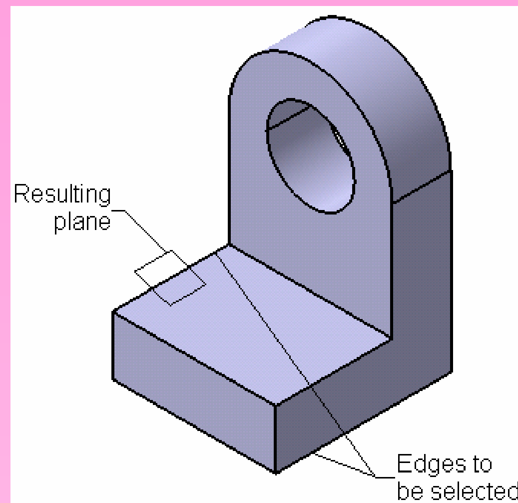
Rotation axis, reference plane, and the resulting plane

- Creating a Plane Through Three Points



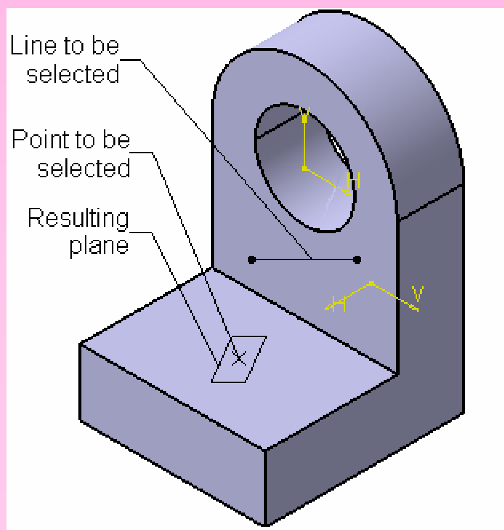
Three vertices to be selected and the resulting plane

- Creating a Plane Through two Lines



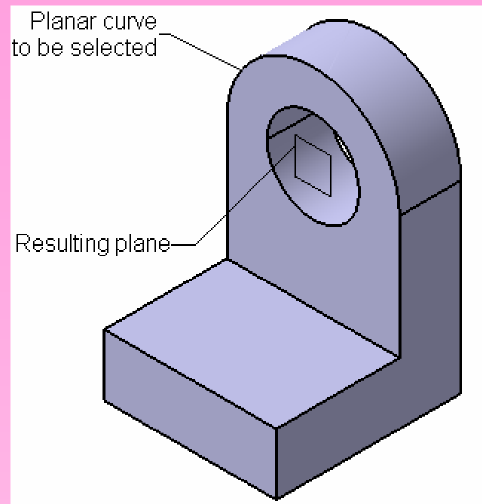
Vertices to be selected and the resulting plane

- Creating a Plane Through Point and Line



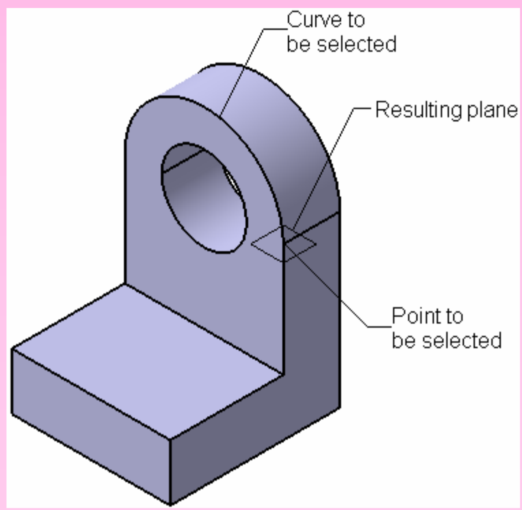
The point and line to be selected, and the resulting plane

- Creating a Plane Through Planar Curve



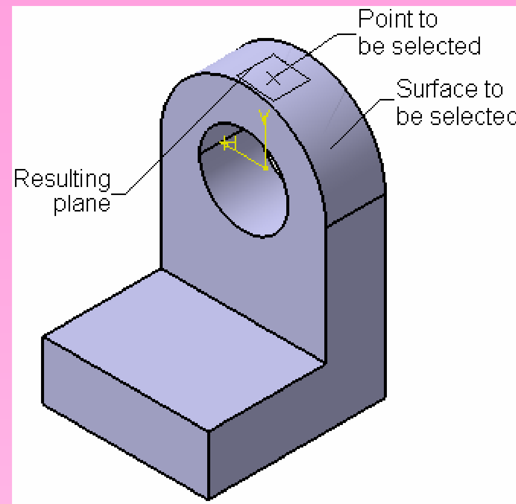
The planar curve to be selected and the resulting plane

- Creating a Plane Normal to Curve



The curve to be selected, the point to be selected, and the resulting plane

- **Creating a Plane Tangent to Surface**



The reference surface to be selected, the point to be selected, and the resulting plane

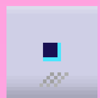
- **Creating a Plane Using Equation**

The **Equation** option is used to create a plane using the equation Ax+By+Cz = D, where the values of A, B, C, and D are variable and can be changed to modify the orientation of the plane.

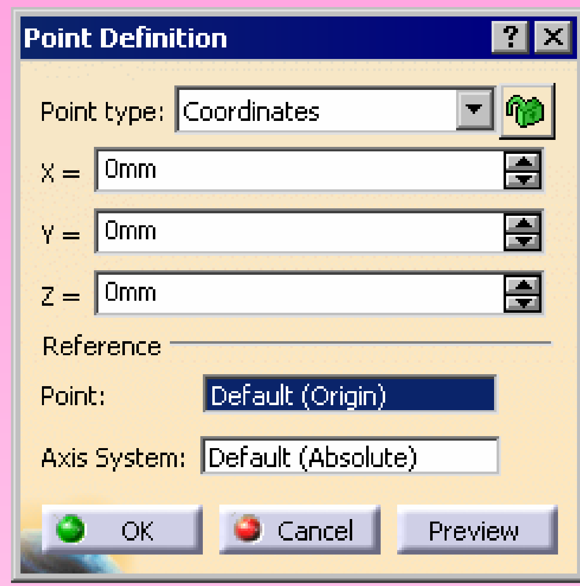
- **Creating a Plane Using a Mean Through Points**

The **Mean through points** option is used to create a plane at an orientation defined by the mean of selected points.

• Creating Points



To create points, choose the **Point** button from the **Reference Elements** toolbar. The **Point Definition** dialog box is displayed, as shown in the figure.



The Point Definition dialog box

• Creating a Point Using Coordinates

The **Coordinates** option is used to create a point by specifying the values of its coordinates.

• Creating a Point On Curve

The **On Curve** option is used to create a point on a selected curve.

- **Creating a Point On Plane**

  The **On plane** option is used to create a point on a selected plane.

- **Creating a Point On Surface**

  The **On Surface** option is used to create a point on a selected surface.

- **Creating a Point at the Circle Center**

  The **Circle center** option is used to create a point at the center of the selected circle.

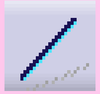
- **Creating a Point Tangent on Curve**

  The **Tangent on curve** option is used to create a point tangent to the selected arc.

- **Creating a Point Between Two Points**

  The **Between** option is used to create a point between two selected points by defining the ratio of the distance from the two points.

- **Creating Reference Lines**

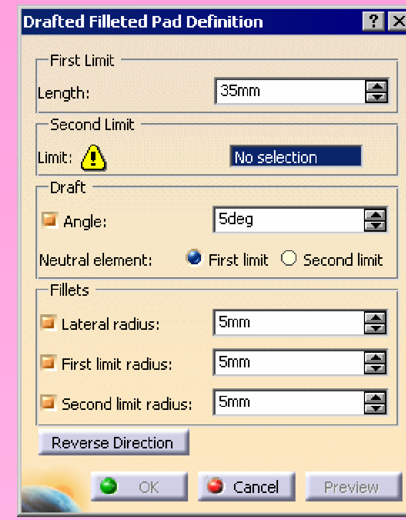


To create reference lines, choose the **Line** button from the **Reference Elements** toolbar; the **Line Definition** dialog box is displayed.

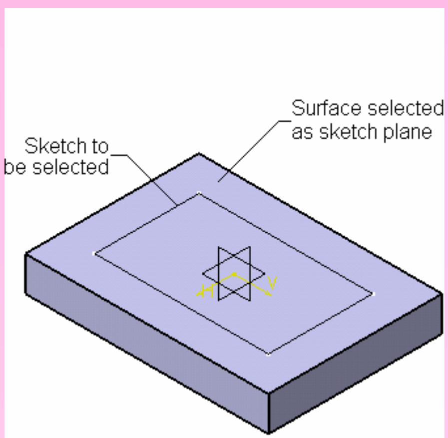
➢ OTHER SKETCH-BASED FEATURES

• Creating Drafted Filleted Pad Features

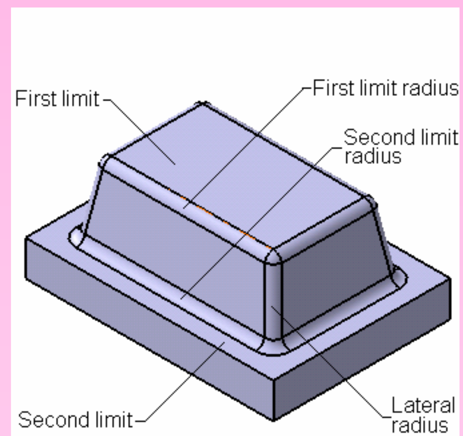
- Choose the **Drafted Filleted Pad** button from the **Pads** toolbar.
- The **Drafted Filleted Pad Definition** dialog box will be displayed, as shown in the figure.



The Drafted Filleted Pad Definition dialog box



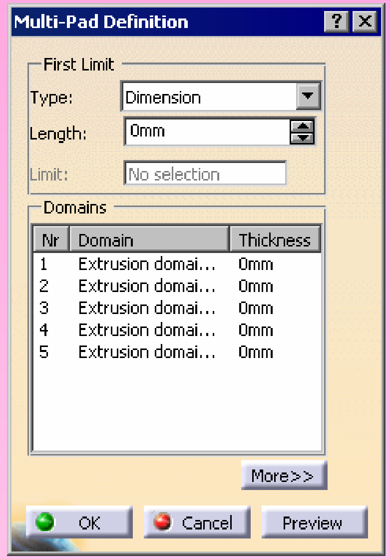
The sketch to be extruded and the plane to be selected as the second limit



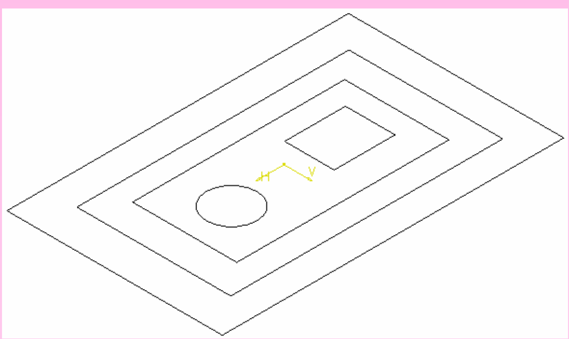
The resulting drafted filleted pad

• Creating Multi-Pad Features

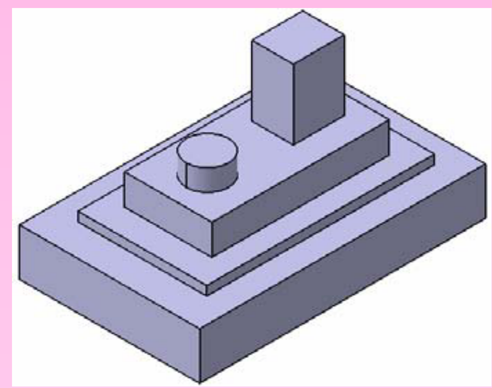
- To create a multi-pad feature, choose the **Multi-Pad** button fromthe **Pads** toolbar and then select the sketch to be extruded.
- The **Multi-Pad Definition** dialog box is displayed,as shown in the figure.



The **Multi-Pad Definition** dialog box



The multi-loop sketch to be extruded



The resulting extruded feature using the **Multi-Pad** tool

- **Other Feature Termination Options**

  The other feature termination options are:

- **Up to next**

  The **Up to next** option is used to extrude the sketch from the sketching plane to the next surface that intersects the feature.

- **Up to last**

  The **Up to last** option is used to extrude the sketch up to the last surface of the model that intersects the feature.

- **Up to plane**

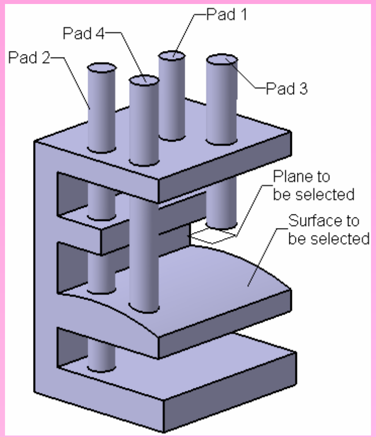
  The **Up to plane** option is used to extrude the sketch from the sketch plane up to the selected plane or the planar face.

- **Up to surface**

  The **Up to surface** option is used to extrude the sketch from the sketch plane to the selected surface or planar face.

- **Feature Termination at an Offset**

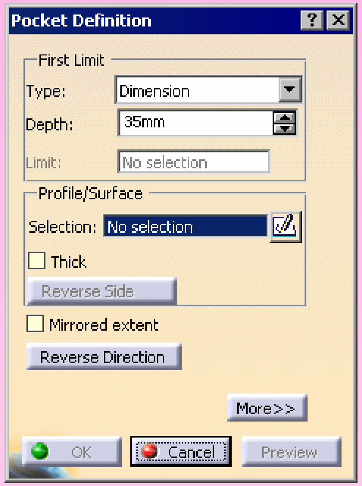
  You can also terminate the features at an offset from the planes or faces selected for feature termination.



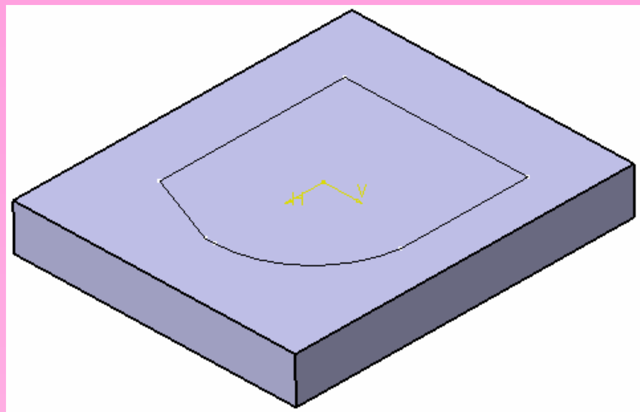
Pad features created using different feature termination options

- **Creating Pocket Features**

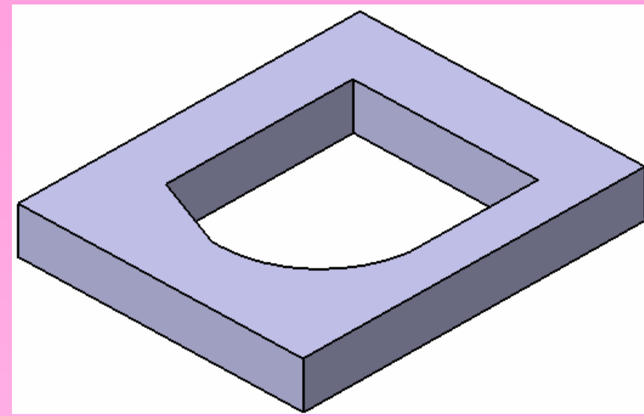
  - To create a pocket feature, draw the sketch and then choose the **Pocket** button from the **Sketch-Based Features** toolbar or from the **Pockets** toolbar.
  - The **Pocket Definition** dialog box is displayed, as shown in the figure.



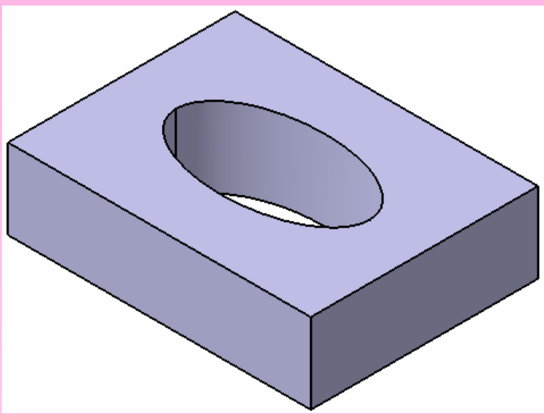
The Pocket Definition dialog box



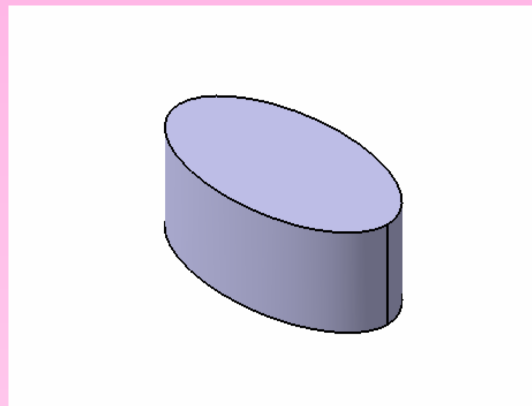
The sketch for the pocket feature



The resulting pocket feature

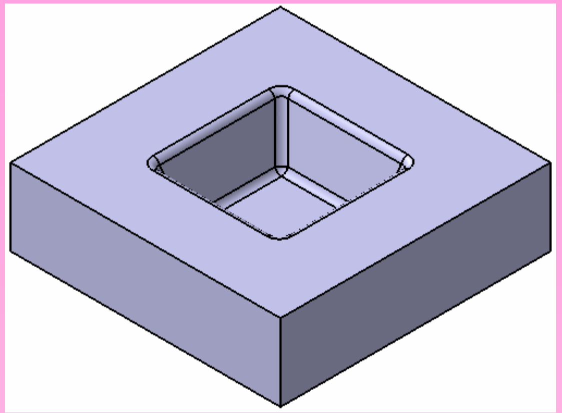


Pocket feature with the default material removal side selected



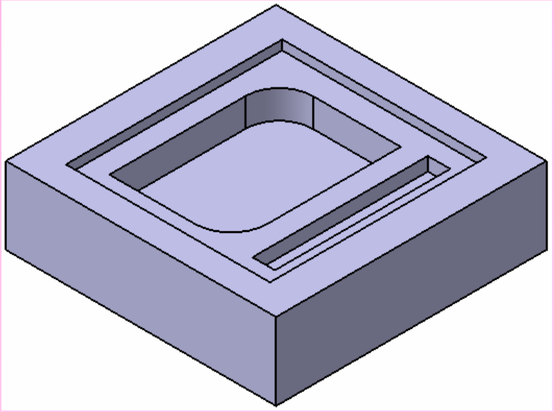
Pocket feature after flipping the material removal direction

- Creating Drafted Filleted Pocket Features



The drafted filleted pocket feature

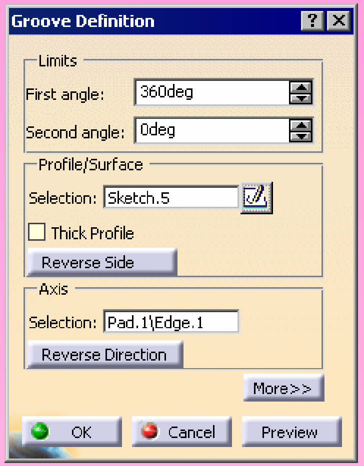
- Creating Multi-Pocket Features



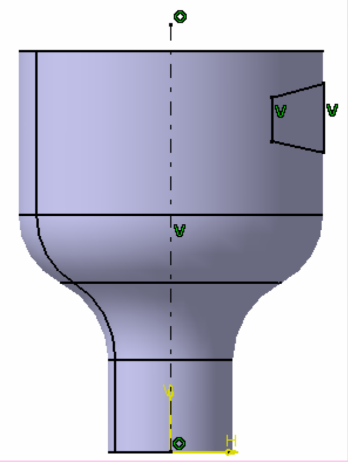
The multi-pocket feature

• Creating Groove Features

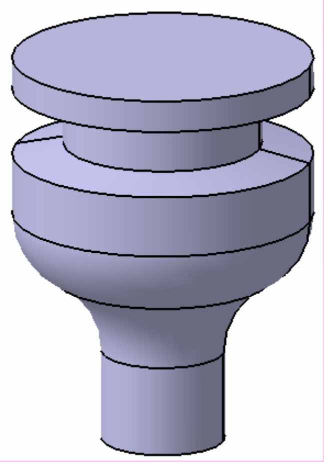
- To create a **Groove** feature, draw the sketch and then choose the **Groove** button from the **Sketch-Based Features** toolbar.
- The **Groove Definition** dialog box is displayed, as shown in the figure.



The Groove Definition dialog box

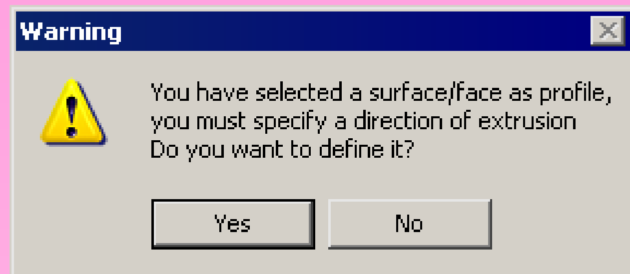


Sketch for creating the groove feature

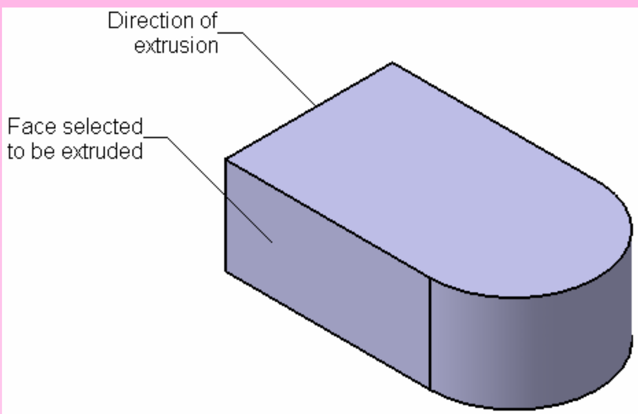


The resulting groove feature

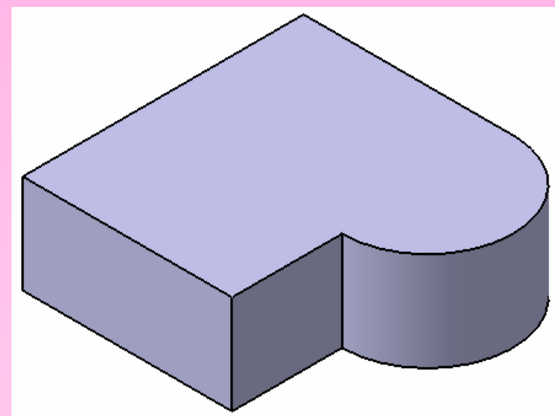
• Extruding and Revolving Planar and Nonplanar Faces



The Warning Message Window

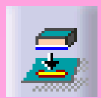


Face selected to extrude and the direction of extrusion

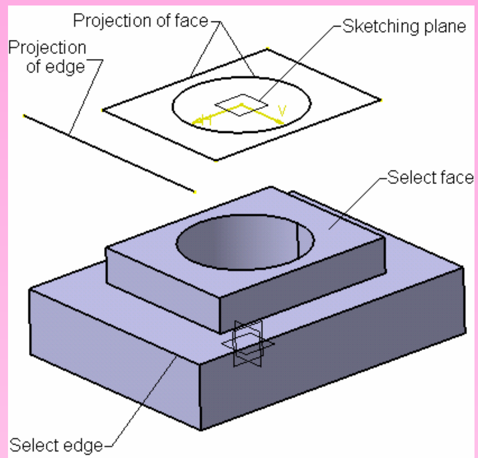


The resulting pad feature

• Projecting 3D Elements



The **Project 3D Element** tool is used to project selected 2D or 3D elements on the current sketch plane.



The projected elements

Tutorial 1

In this tutorial, you will create the model shown in **Figure A**. The views and dimensions of the model are shown in **Figure B**. **(Expected time: 30 min)**

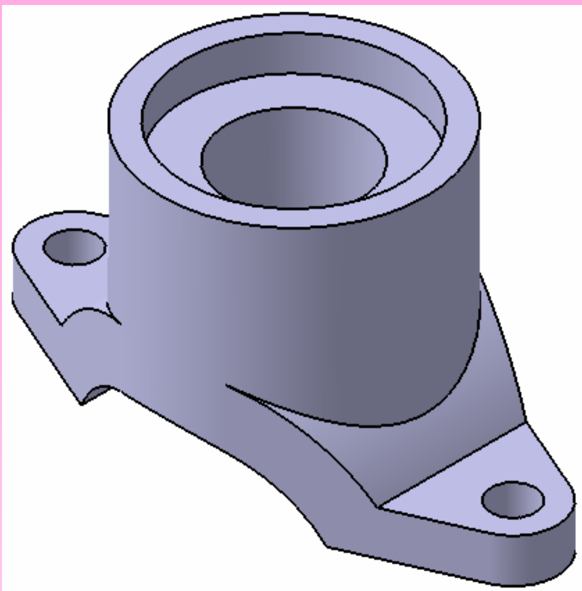


Figure A Solid model for Tutorial 1

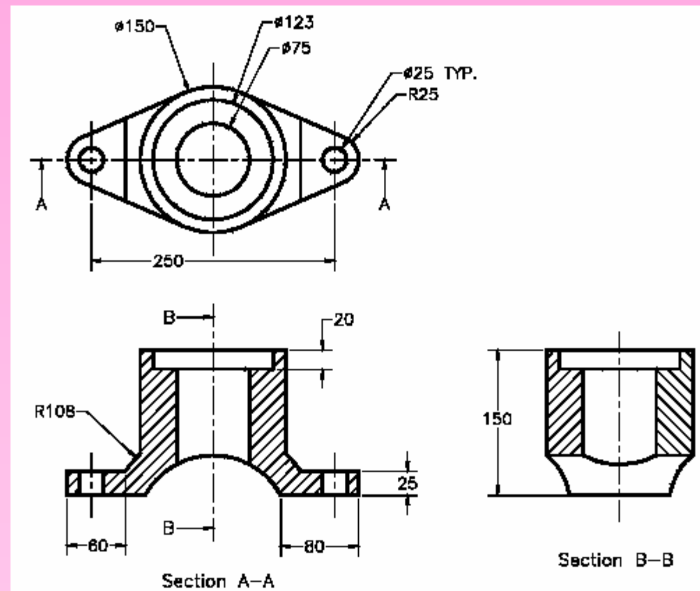


Figure B The views and dimensions for Tutorial 1

1. Start a new session of CATIA V5 and start a new file in the **Part** workbench.
2. Draw the sketch of the base feature on the yz plane, as shown in **Figure C**.

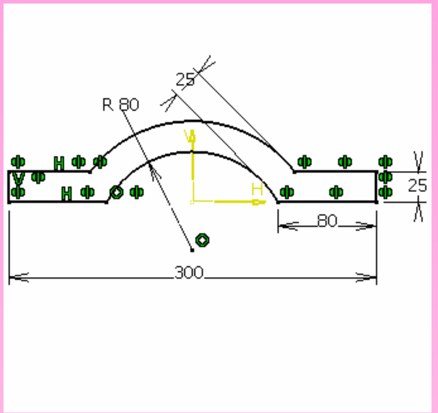


Figure C The sketch of the base feature

3. Extrude the sketch to the required distance using the **Pad** tool, as shown in **Figure D**.

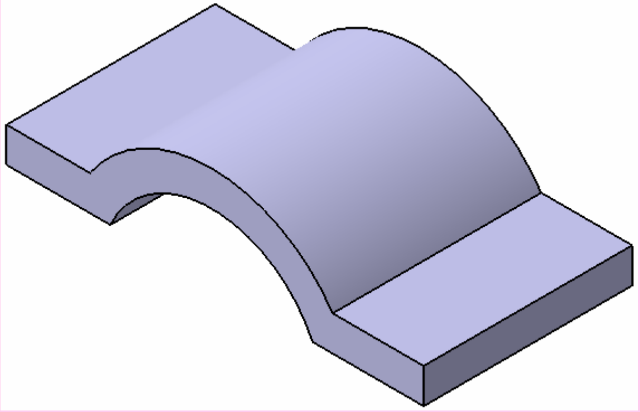


Figure D The model after creating the base feature

4. Create the second feature, which is a **Pocket** feature, as shown in **Figure E** and **Figure F**.

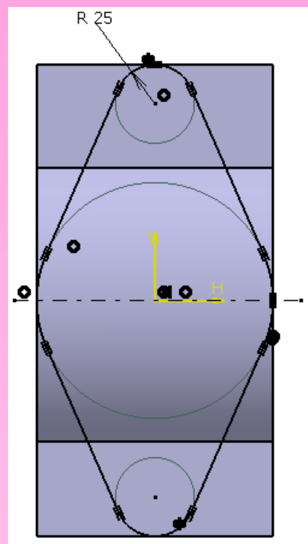


Figure E The sketch of the second feature

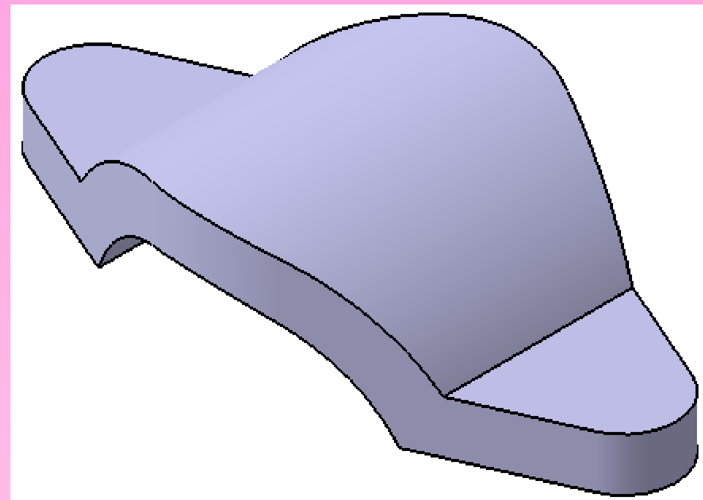


Figure F Model after creating the second feature

5. Create the third feature by extruding the sketch, drawn on a plane at an offset from the xy plane, as shown in **Figure G**, **Figure H** and **Figure I**.

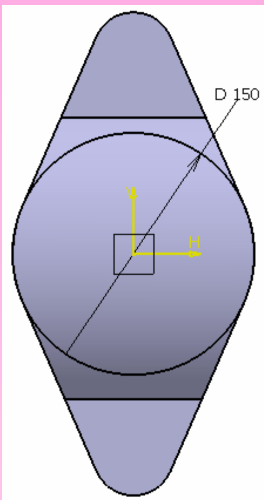


Figure G Sketch of the third feature

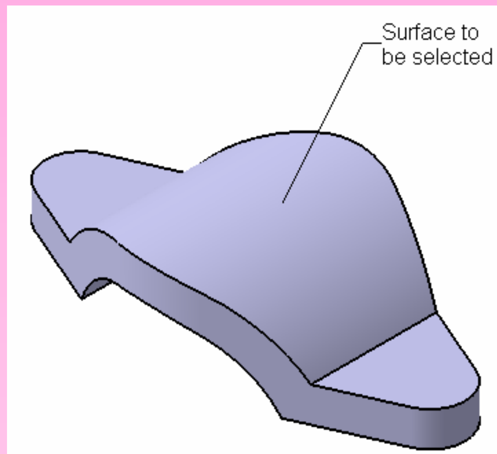


Figure H Surface to be selected

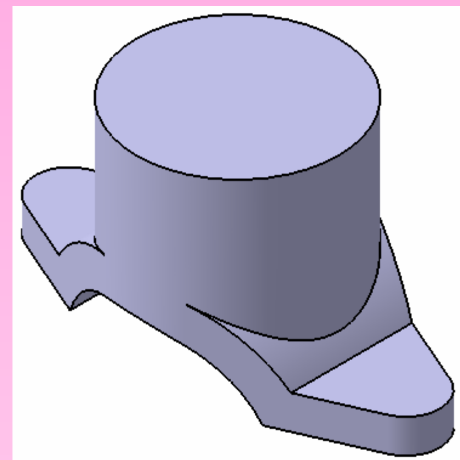


Figure I The model after creating the third feature

6. Create the fourth feature, which is a **Groove** feature, as shown in **Figure J** and **Figure K**.

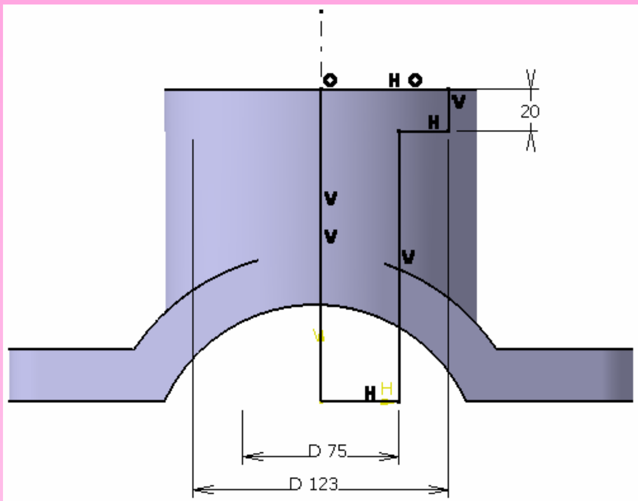


Figure J Sketch of the fourth feature

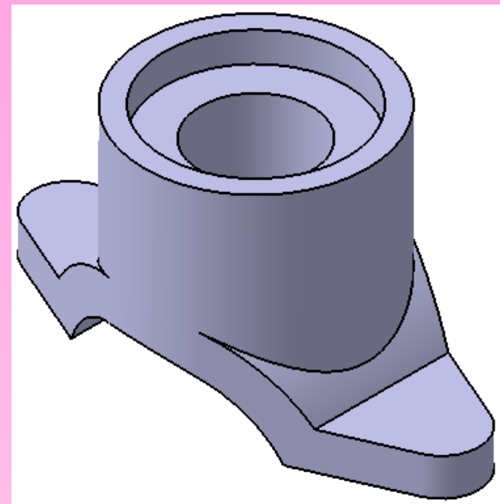


Figure K The model after creating the fourth feature

7. Create the last feature of the model which is the **Pocket** feature, as shown in **Figure L** and **Figure M**.

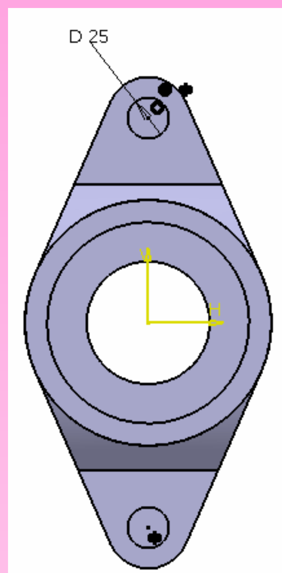


Figure L Sketch of the fifth feature

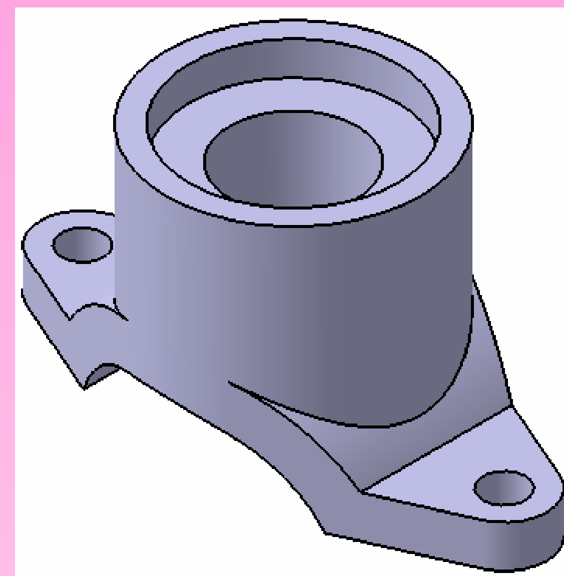


Figure M Final model

8. Save the file in **\My Documents\CATIA\c04** folder and then close it.

Tutorial 2

In this tutorial, you will create the model shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 30 min)**

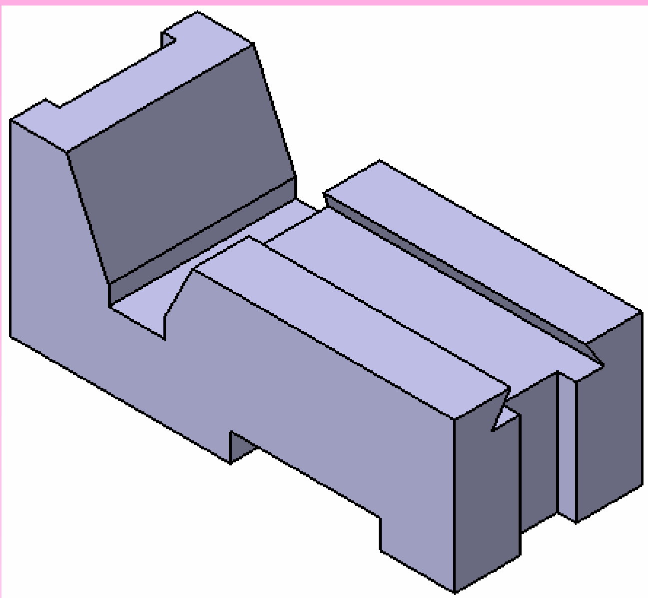


Figure A Model for Tutorial 2

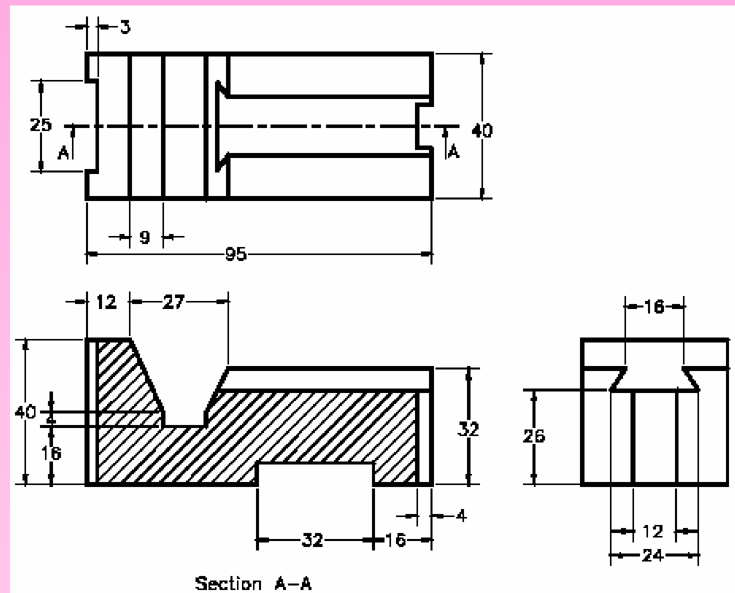


Figure B Views and dimensions for Tutorial 2

1. Start a new file in the **Part** workbench and draw the sketch of the base feature on the yz plane, as shown in **Figure C**.

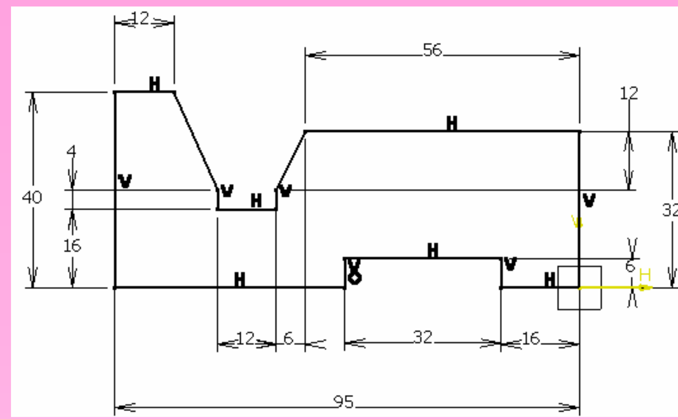


Figure C Sketch of the base feature

2. Extrude the sketch to a distance of 40 using the **Pad** tool, as shown in **Figure D**.

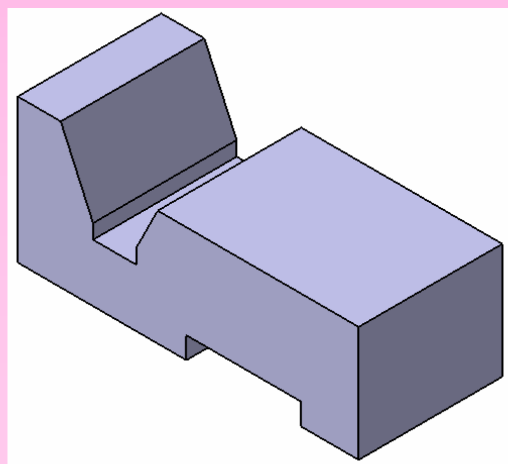


Figure D Base feature of the model

3. Create the second feature, which is a **pocket** feature, as shown in **Figure E** and **Figure F**.

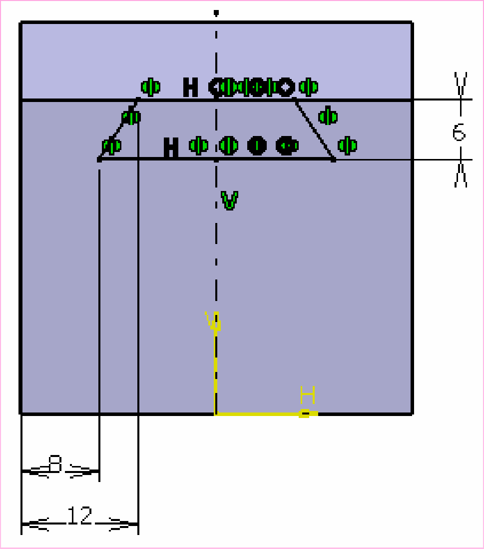


Figure E Sketch of the second feature

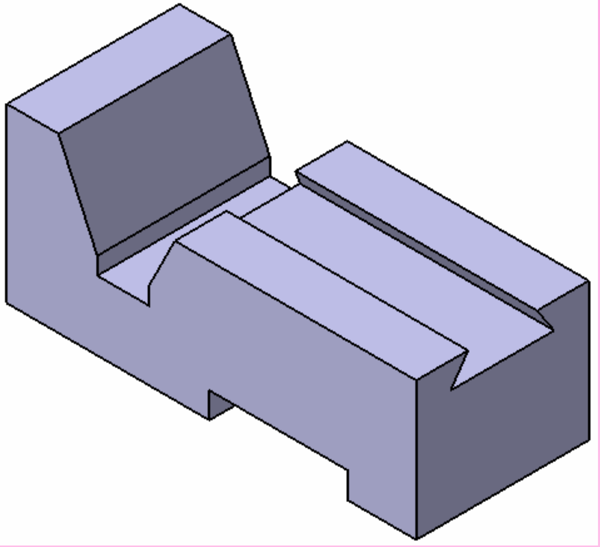


Figure F The model after creating the second feature

4. Create the third feature, by extruding a sketch drawn on the right face of the base feature, using the **Pocket** tool, as shown in **Figure G** and **Figure H**.

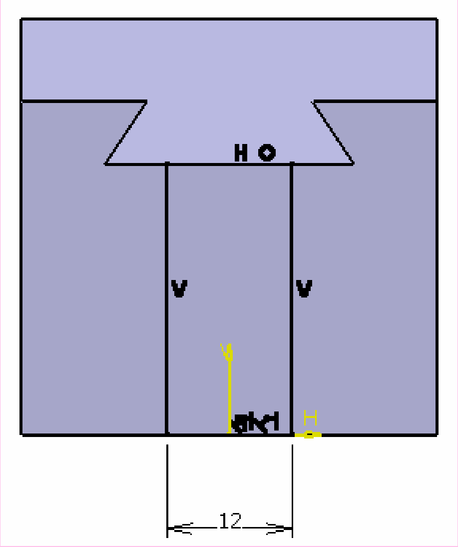


Figure G Sketch of the third feature

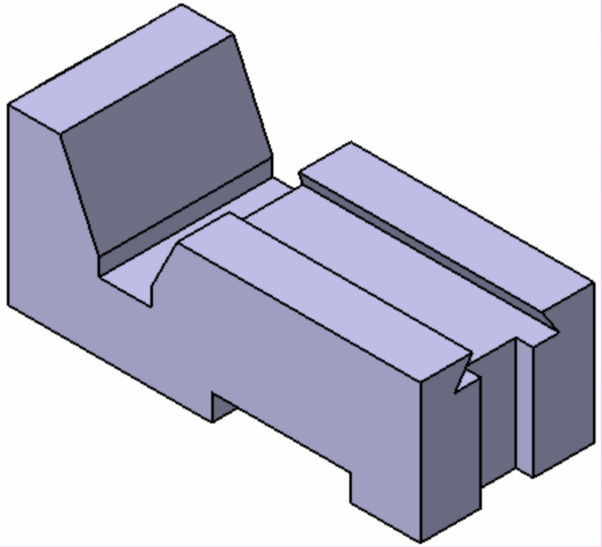


Figure H The model after creating the third feature

5. Create the fourth feature, which is also a **pocket** feature, as shown in **Figure I**.

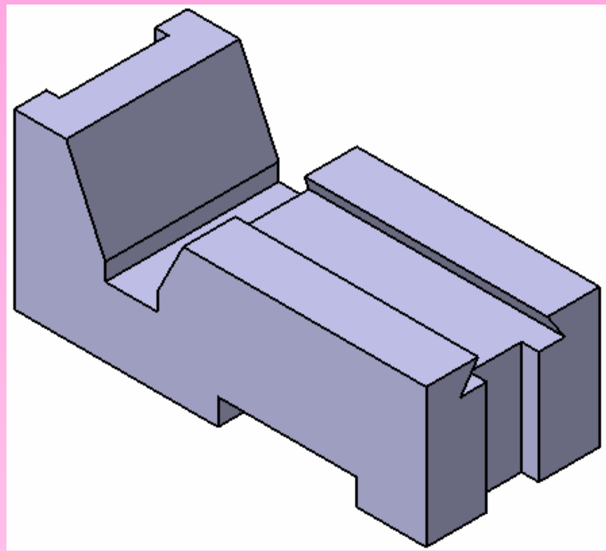


Figure I Final model

6. Save the file in **\My Documents\CATIA\c04** folder and then close it.

❑ Tutorial 3

In this tutorial, you will create the model shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

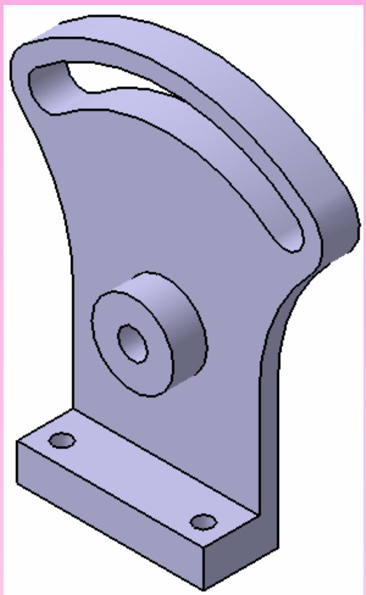


Figure A Model for Tutorial 3

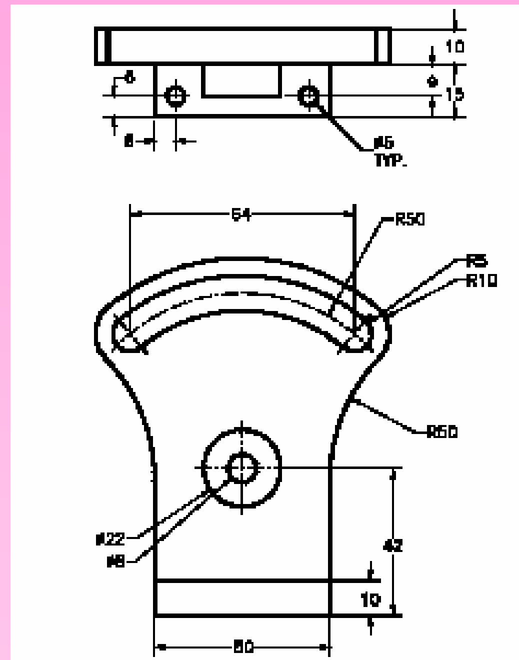


Figure B Views and dimensions for Tutorial 3

1. Start a new file in the **Part** workbench and create the base feature of the model by extruding the sketch drawn on yz plane, as shown in **Figure C** and **Figure D**.

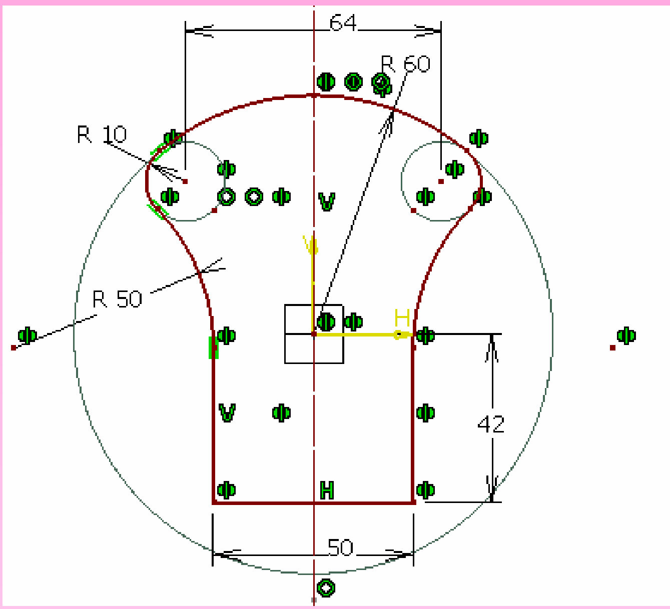


Figure C Sketch of the base feature

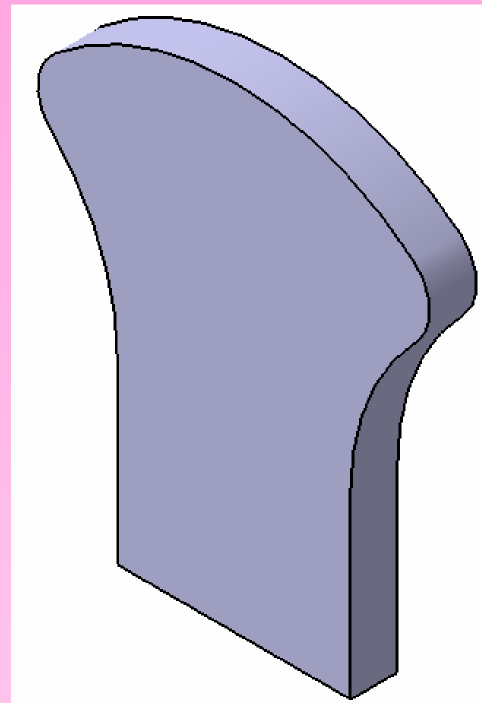


Figure D Base feature of the model

2. Create the second feature by extruding the sketch drawn on the front face of the base feature, as shown in **Figure E** and **Figure F**.

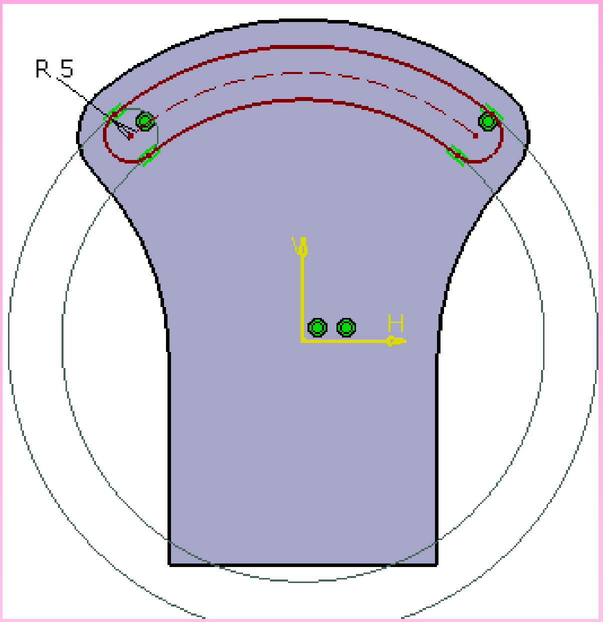


Figure E Sketch of the second feature

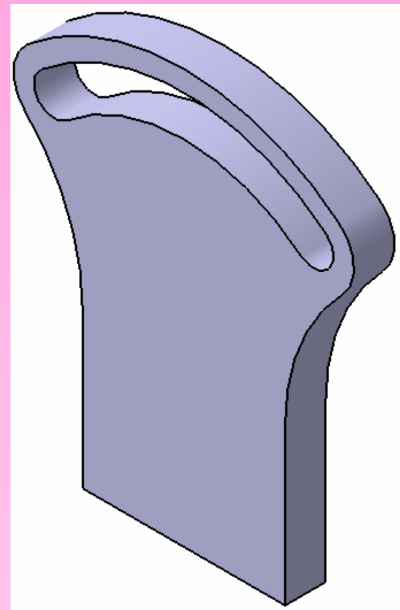


Figure F Model after creating the second feature

3. Create the third feature by extruding a rectangular sketch drawn on the front face of the base feature, as shown in **Figure G** and **Figure H**.

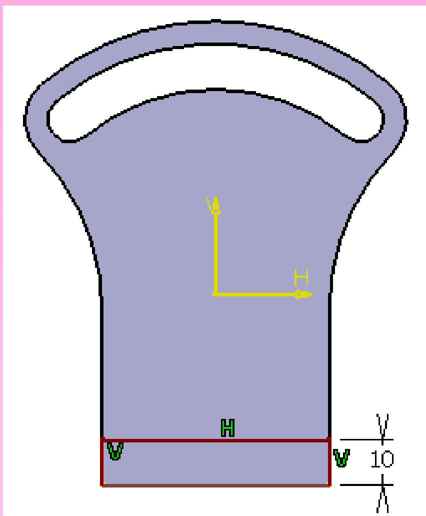


Figure G Sketch of the third feature

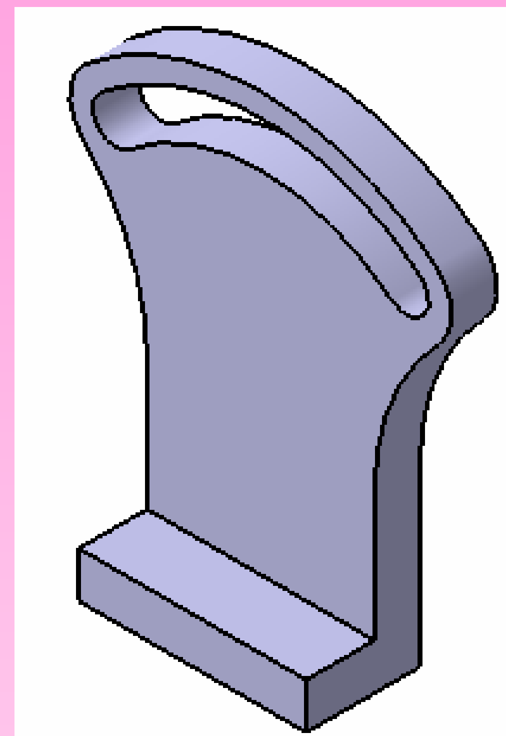


Figure H Model after creating the third feature

4. Create the fourth feature by extruding the circular sketch to both the sides of the sketching plane. The sketch for this feature will be drawn on the front face.
5. Create the fifth feature by extruding the sketch drawn on the front face of the fourth feature using the **Pocket** tool.
6. Create the sixth feature by extruding the sketch drawn on the top face of the third feature using the **Pocket** tool. The final model is shown in **Figure I**.

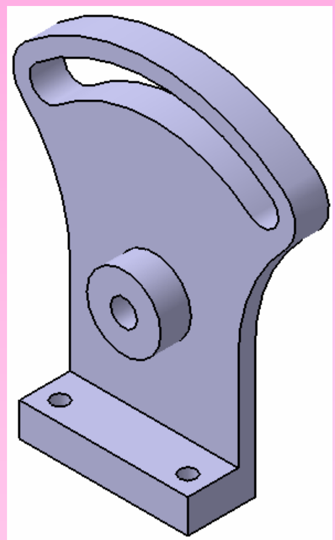


Figure I Final model

7. Save the file in **\My Documents\CATIA\c04** folder and then close it.

Exercise 1

Create the model shown in **Figure A**. Its views and dimension are shown in **Figure B**.

(Expected time: 30 min)

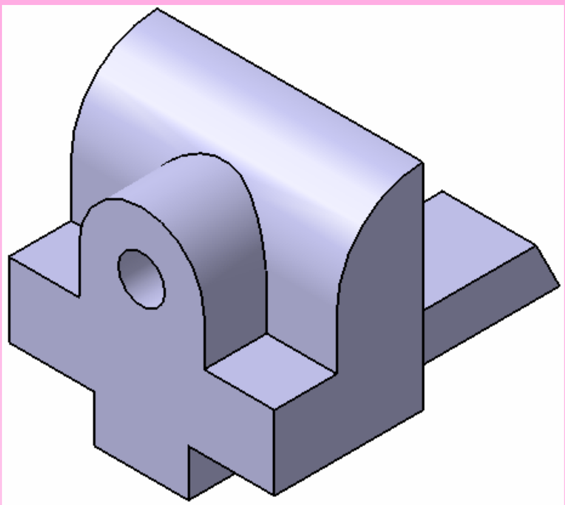


Figure A Model for Exercise 1

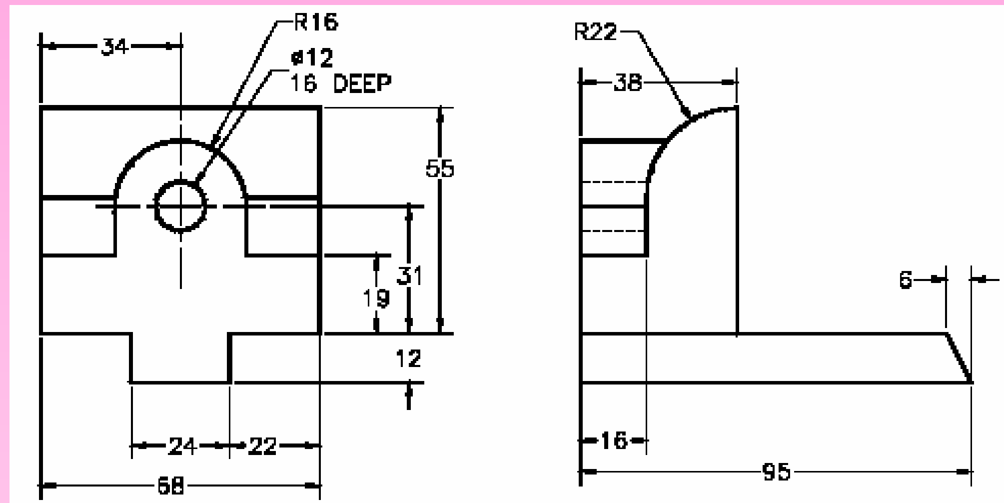


Figure B Views and dimensions for Exercise 1

Exercise 2

Create the model shown in **Figure A**. Its views and dimension are shown in **Figure B**.

(Expected time: 30 min)

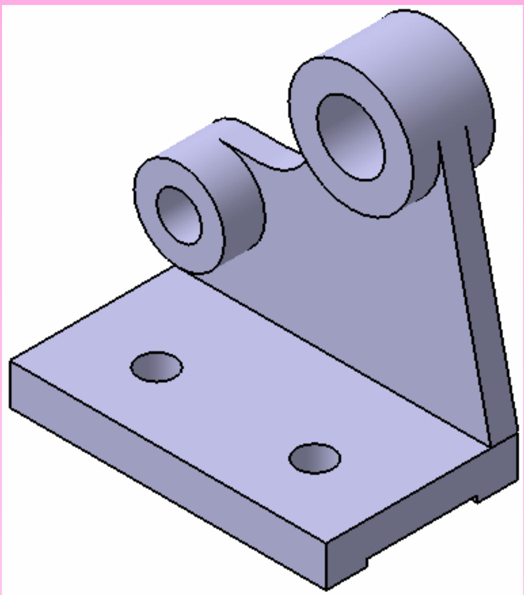


Figure A Model for Exercise 2

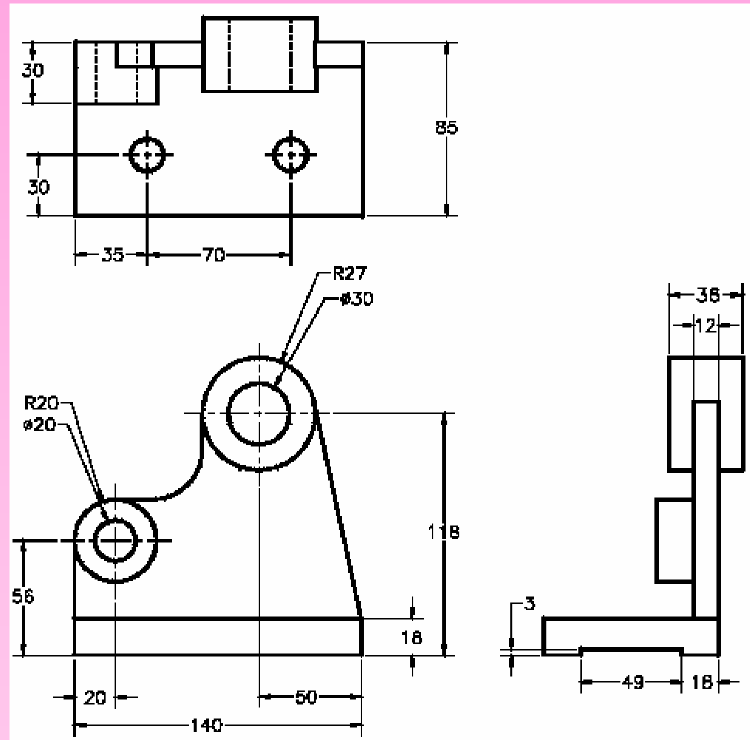


Figure B Views and dimensions for Exercise 2

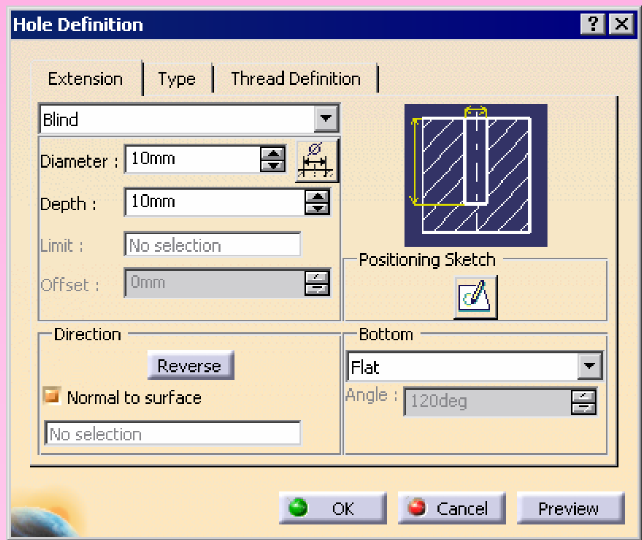
Learning Objectives:

- *Create holes using the Hole tool.*
- *Create fillet features.*
- *Create chamfer features.*
- *Add draft to the faces of the models.*
- *Create a shell feature.*

➢ ADVANCED MODELING TOOLS

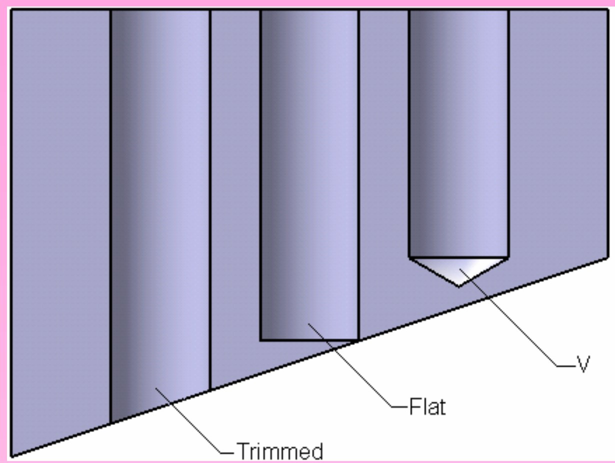
• Creating Hole Features

- The preview of the hole feature is displayed, along with the **Hole Definition** dialog box.
- The **Hole Definition** dialog box is shown in the figure.

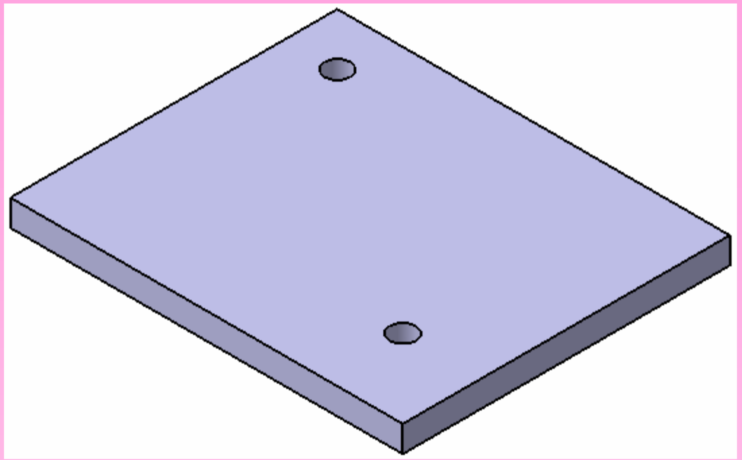


The **Hole Definition** dialog box

- **Creating a Simple Hole**



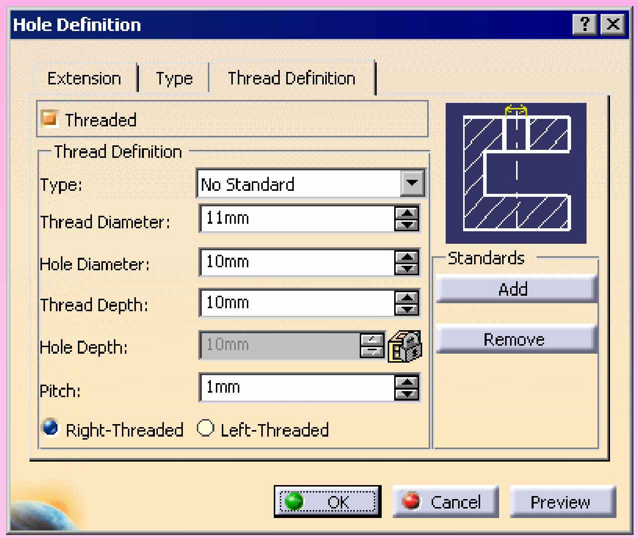
Types of bottom termination options for a hole feature



Base plate with holes created using the **Hole** tool

Creating a Threaded Hole

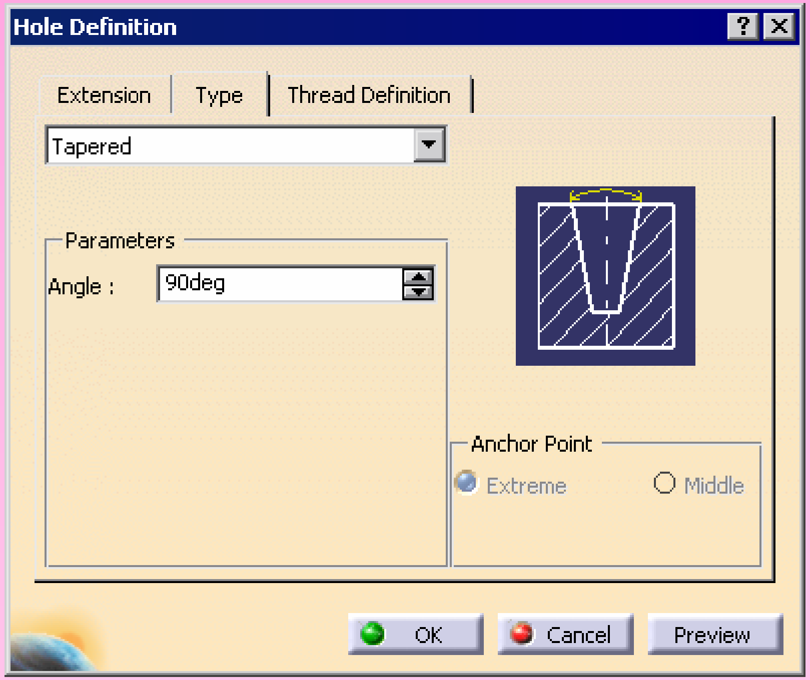
- To create the threaded hole, invoke the **Thread Definition** tab from the **Hole Definition** dialog box.
- By default, the **Threaded** radio button is cleared.
- Select the **Threaded** radio button to invoke the options available in the **Thread Definition** tab.



The Hole Definition dialog box after selecting the Threaded radio button

• Creating a Tapered Hole

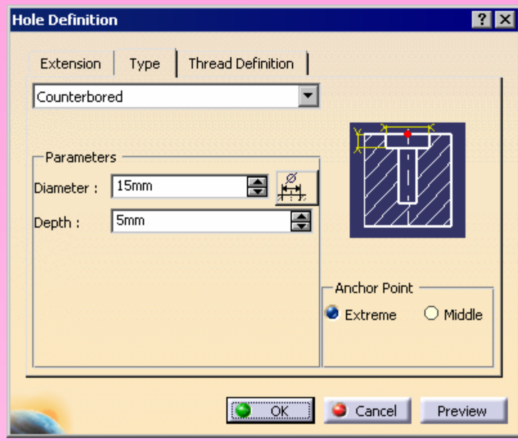
To create a tapered hole, invoke the **Type** tab of the **Hole Definition** dialog box and select the **Tapered** option from the drop-down list, as shown in the figure.



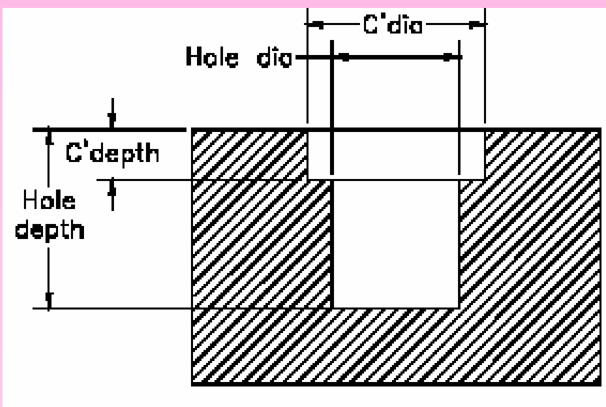
The Hole Definition dialog box after selecting the Tapered option from the drop-down list in the Type tab

• Creating a Counterbored Hole

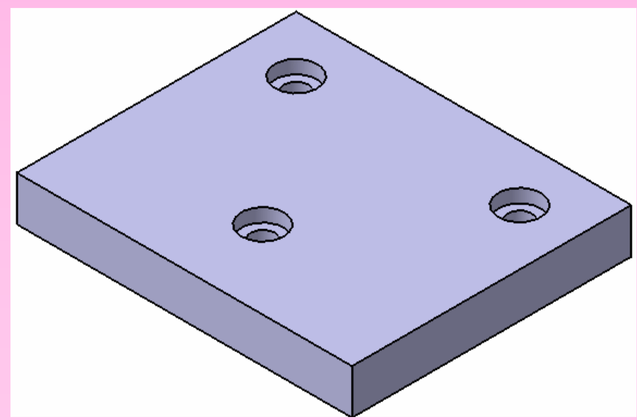
To create a counterbored hole, select the **Counterbored** option from the drop-down list in the **Type** tab of the **Hole Definition** dialog box, as shown in the figure.



The Hole Definition dialog box after selecting the Counterbored option from the drop-down list in the Type tab



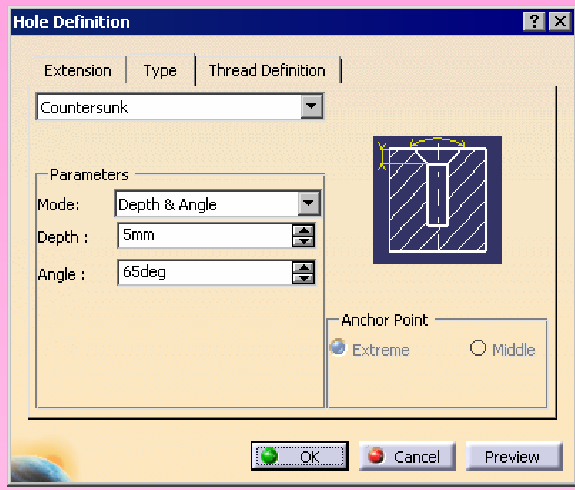
The sectional view of counterbored hole



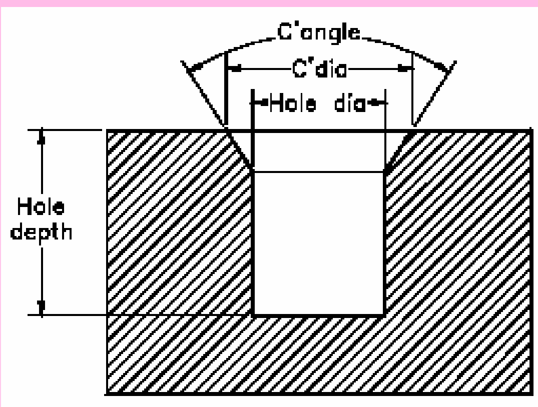
The base plate with counterbored holes

Creating a Countersunk Hole

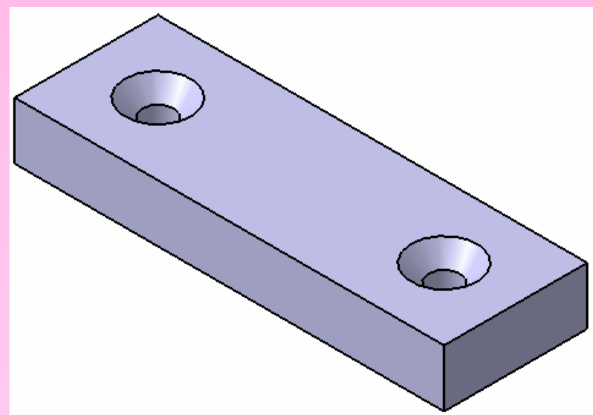
To create a countersunk hole, select the **Countersunk** option from the drop-down list in the **Type** tab, as shown in the figure.



The Hole Definition dialog box, after selecting the Countersunk option from the drop-down list in the Type tab



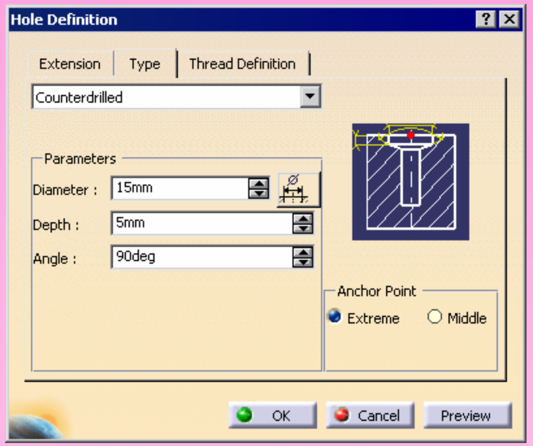
The sectional view of the countersunk hole



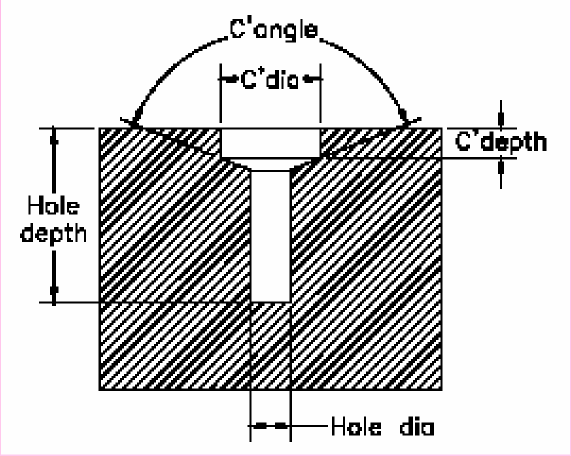
Spacer plate with the countersunk holes

• Creating a Counterdrilled Hole

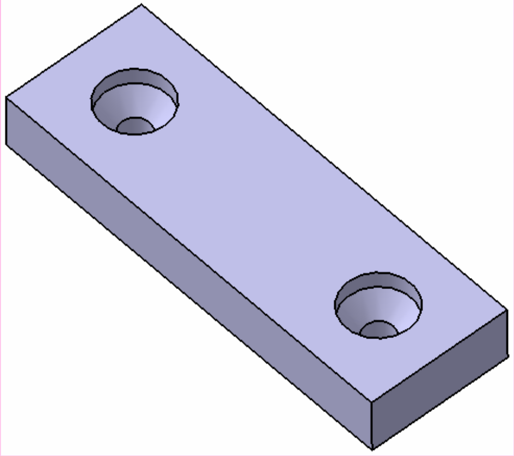
To create a counterdrilled hole, select the **Counterdrilled** option from the drop-down list in the **Type** tab; as shown in the figure.



The Hole Definition dialog box, after selecting the Counterdrilled option from the drop-down list in the Type tab



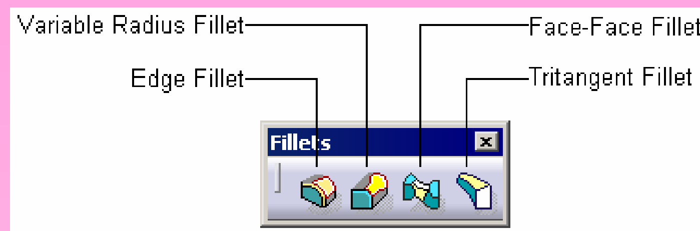
The sectional view of the counterdrilled hole



The spacer plate with the counterdrilled holes

- ## Creating Fillets

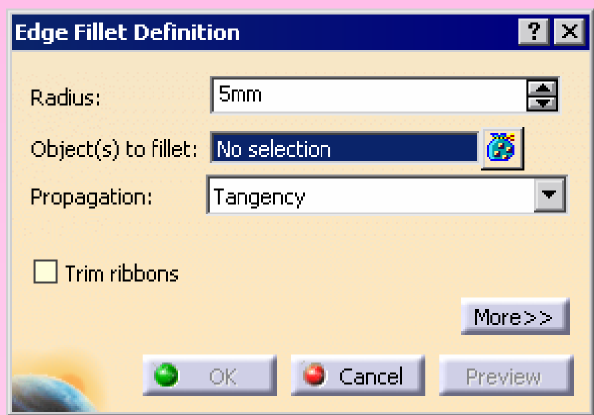
Choose the black arrow on the right of the **Edge Fillet** button in the **Dress-Up Features** toolbar; the **Fillets** toolbar is invoked, as is shown in the figure.



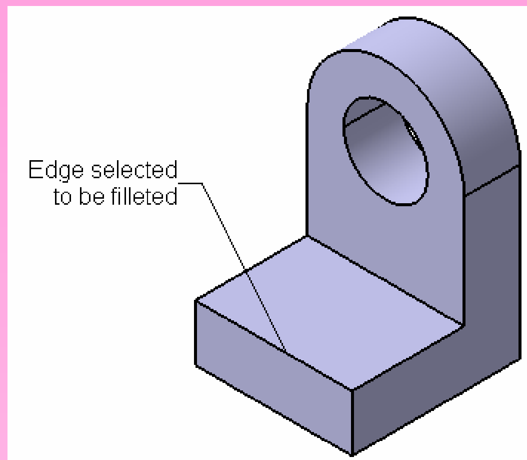
The Fillets toolbar

- ## Creating an Edge Fillet

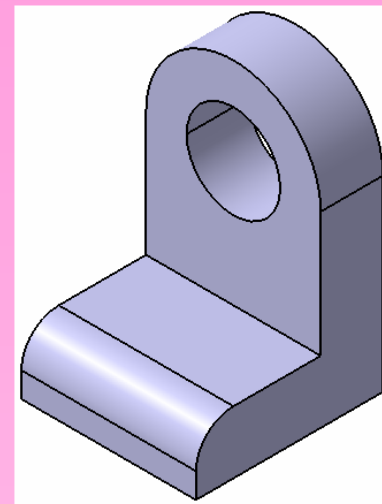
- To create an edge fillet, choose the **Edge Fillet** button from the **Fillets** toolbar.
- The **Edge Fillet Definition** dialog box is displayed, as shown in the figure.



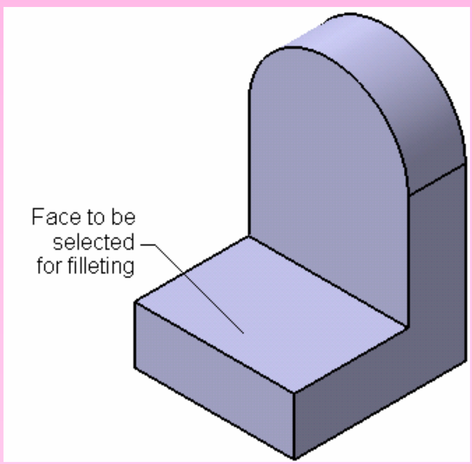
The Edge Fillet Definition dialog box



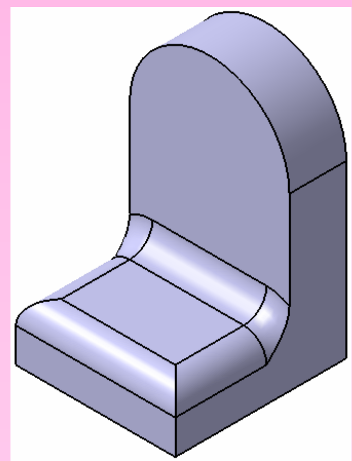
Edge selected to be filleted



Resulting edge fillet



Face to be selected

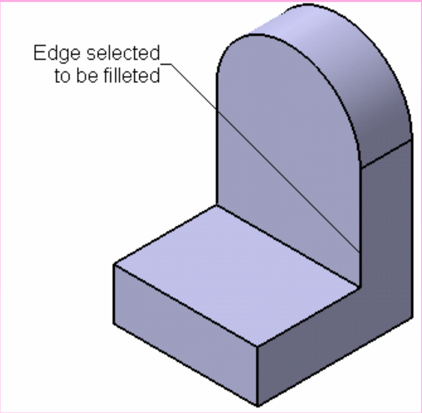


Resulting fillet

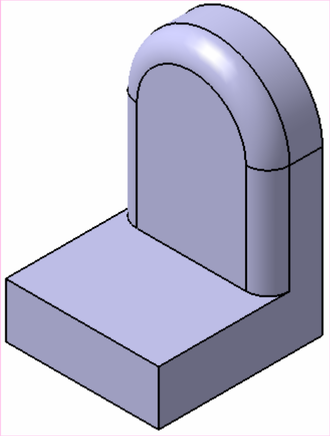
- **Managing Selected Entities**

  To manage the entities in the current selection set, choose the **Selection Filter** button on the right of the **Object(s) to fillet** selection area; the **Fillet** objects dialog box is displayed.

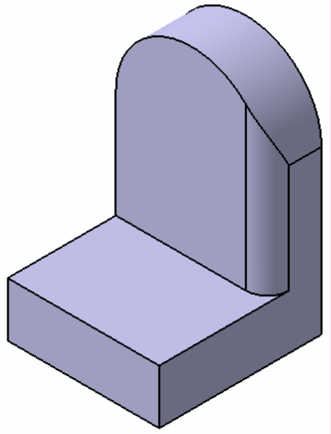
- **Managing the Fillet Propagation**



Edge selected to be filleted

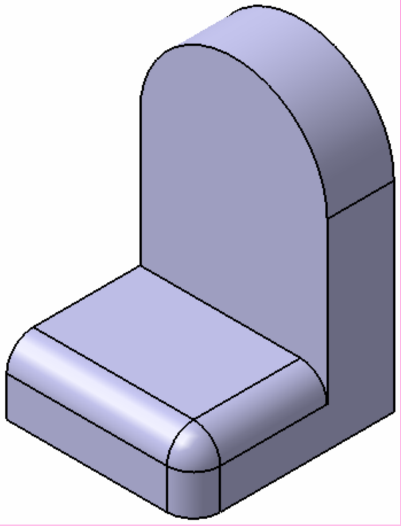


Fillet using the tangent propagation

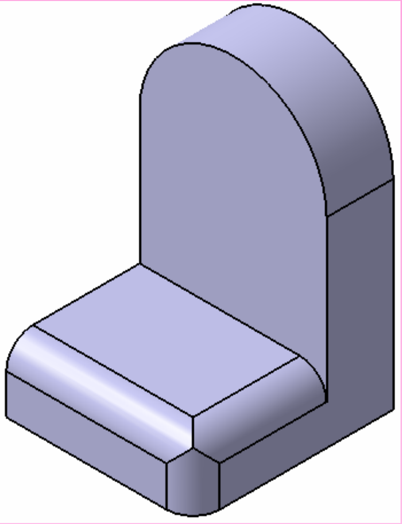


Fillet using the minimal propagation

- Trimming the Overlapping Fillets

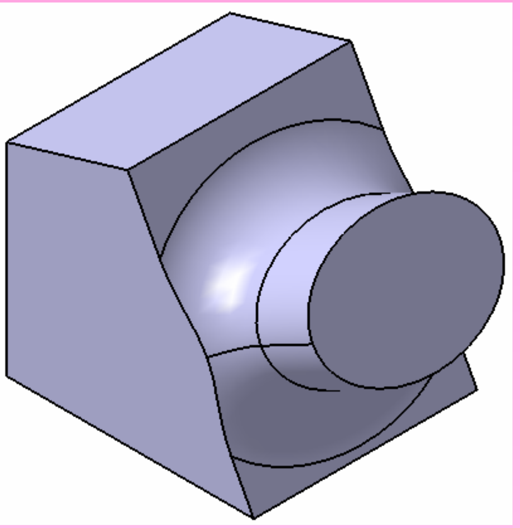


Fillet with the **Trim ribbons** check box cleared

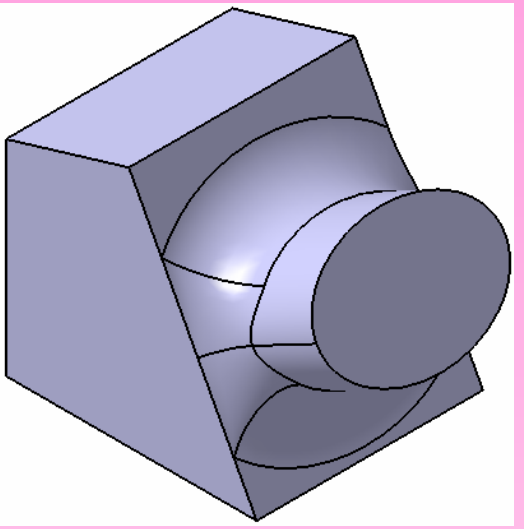


Fillet with the **Trim ribbons** check box selected

- **Selecting the Edges to Keep**

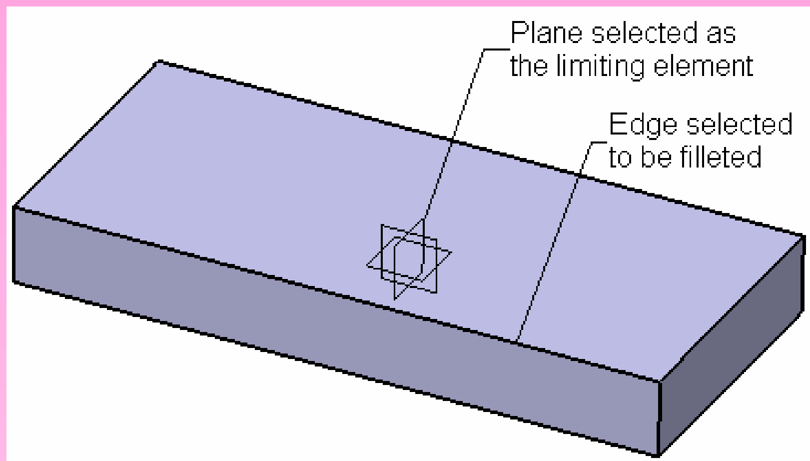


Edges distorted to accommodate the fillet radius

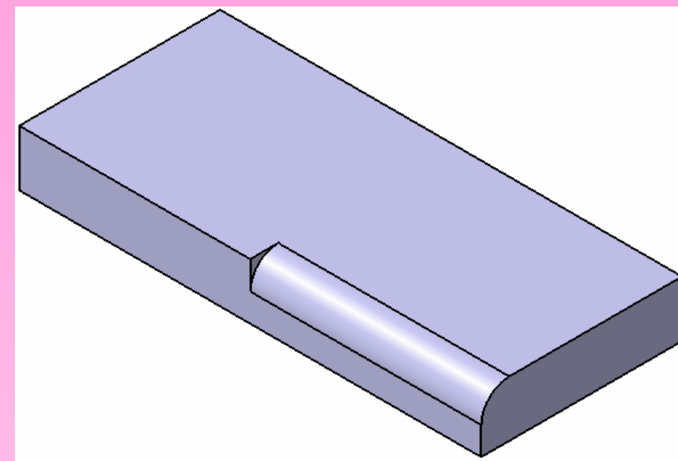


The model after selecting the edges to be keep

- **Setting the Limits of the Fillet**

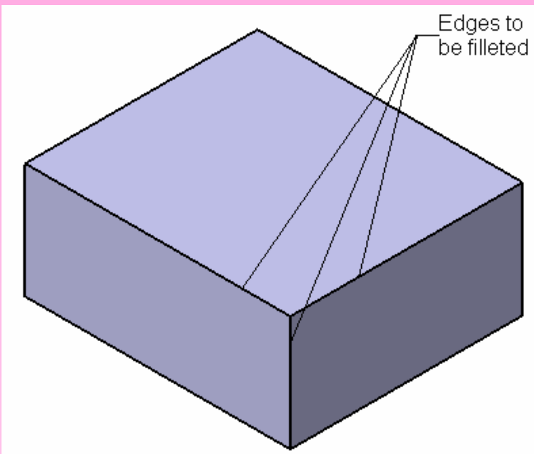


Edge to be filleted and the limiting element

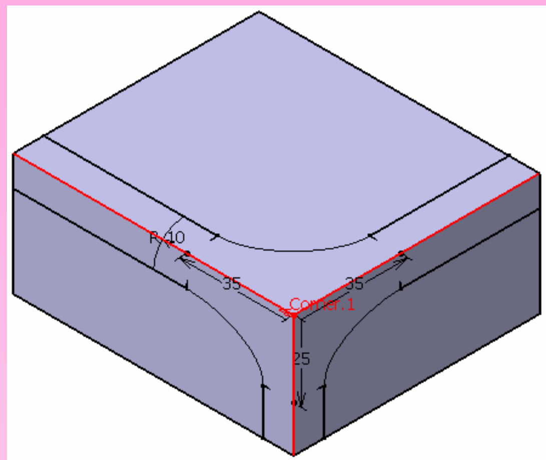


Resulting fillet

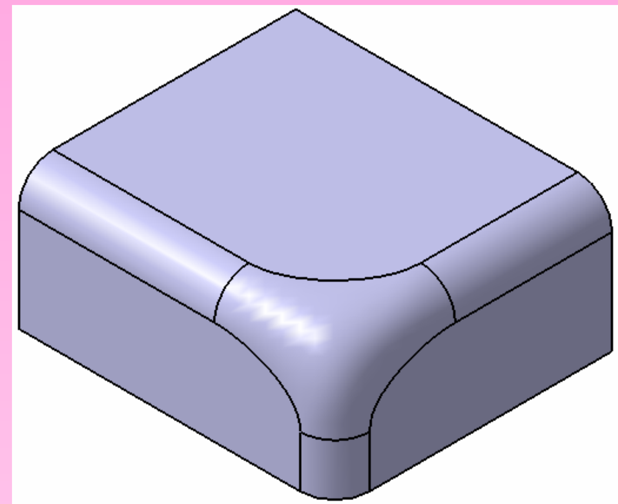
- **Setback Fillet by Blending the Corners**



Edges to be selected

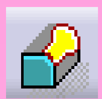


Preview of the setback fillet

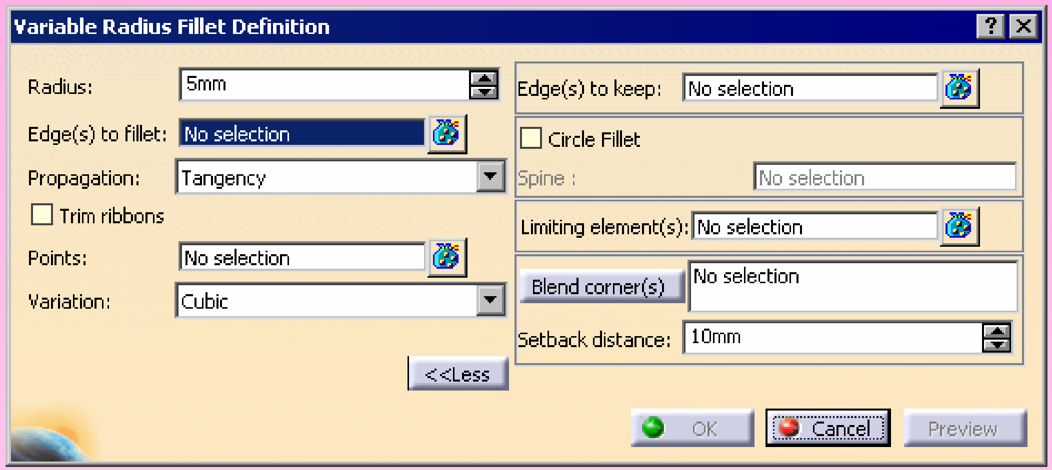


Resulting setback fillet

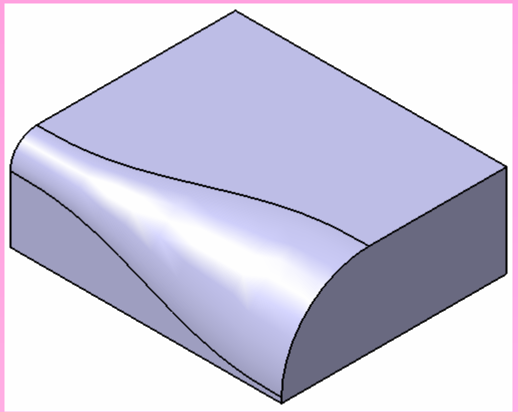
• Creating Variable Radius Fillets



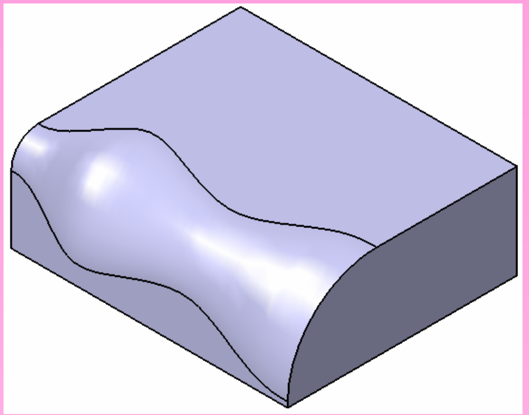
To create variable radius fillet, choose the **Variable Radius Fillet** button from the **Fillets** toolbar; the **Variable Radius Fillet Definition** dialog box is displayed, as shown in the figure.



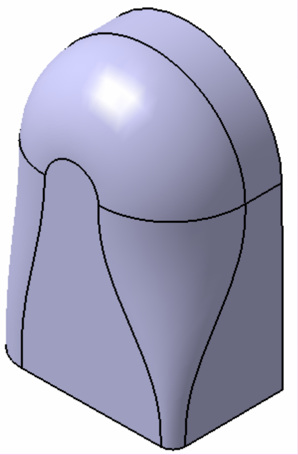
Expanded form of the Variable Radius Fillet Definition dialog box



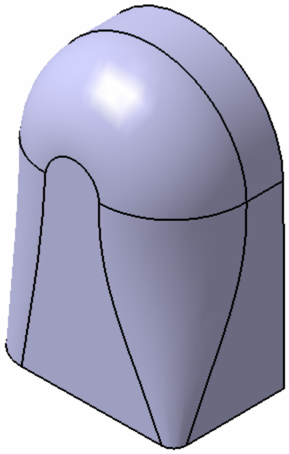
Variable radius fillet created by specifying radii at the two end points of the edge



Variable radius fillet after defining additional control points

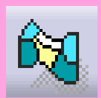


Variable radius fillet with the **Cubic** option selected



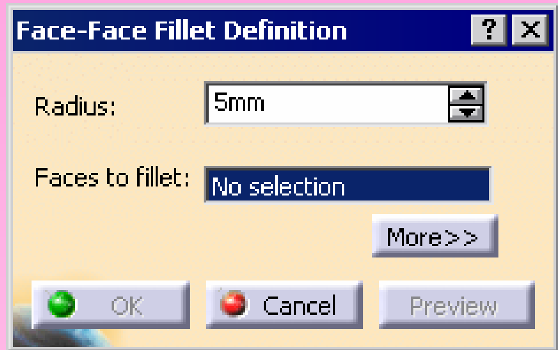
Variable radius fillet with the **Linear** option selected

• Creating Face-Face Fillets

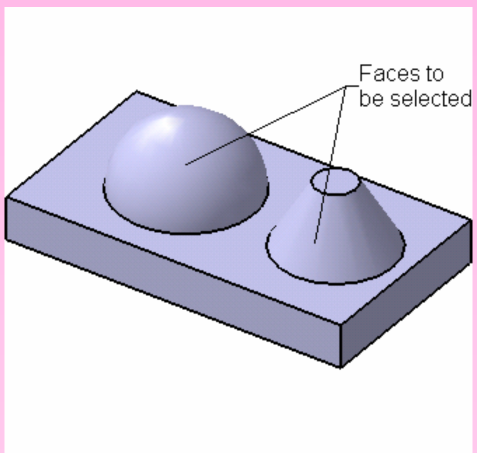


• To create a face fillet, choose the **Face-Face Fillet** button from the **Fillets** toolbar.

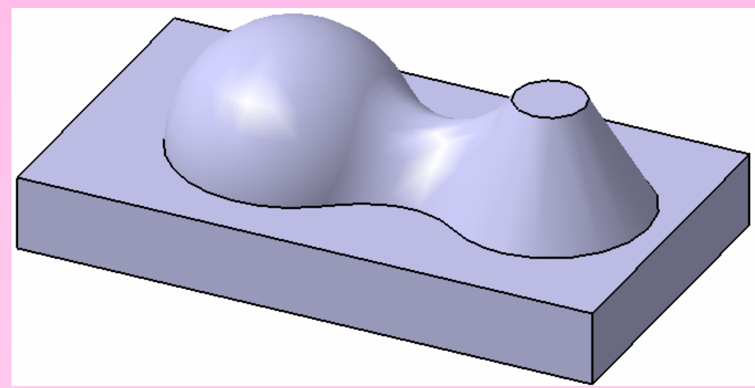
• The **Face-Face Fillet Definition** dialog box is displayed, as shown in the figure.



The Face-Face Fillet Definition dialog box

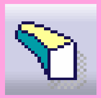


Faces to be selected

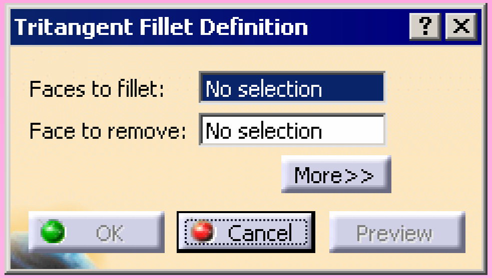


Resulting face-face fillet

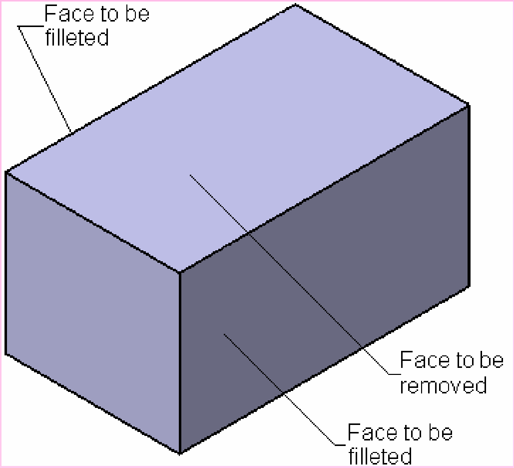
• Creating Tritangent Fillets



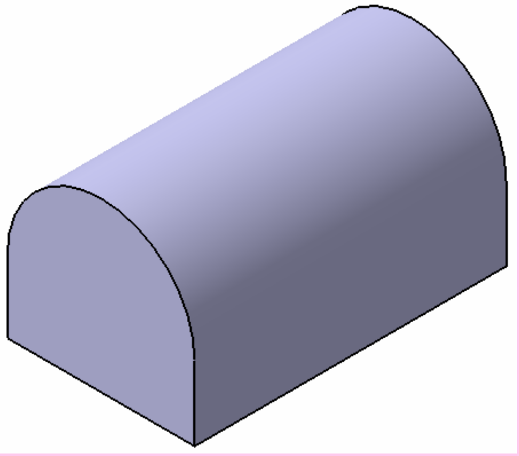
To create a tritangent fillet, choose the **Tritangent Fillet** button from the **Fillets** toolbar, the **Tritangent Fillet Definition** dialog box is displayed, as shown in the figure.



The Tritangent Fillet Definition dialog box

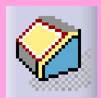


Faces to be selected

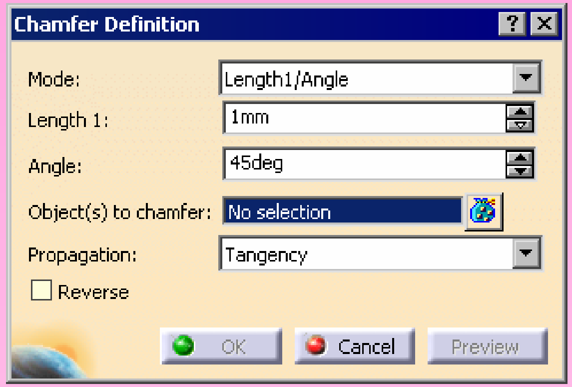


Resulting tritangent fillet

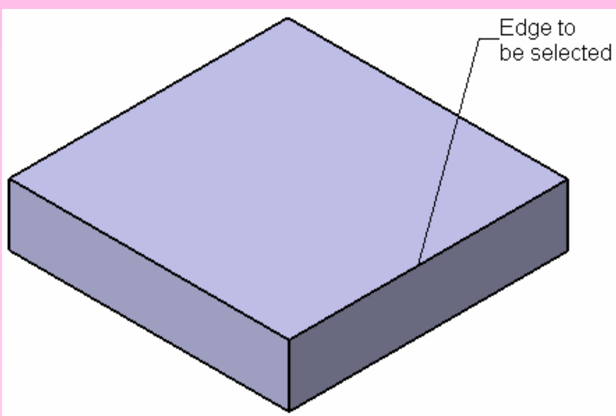
• Creating Chamfers



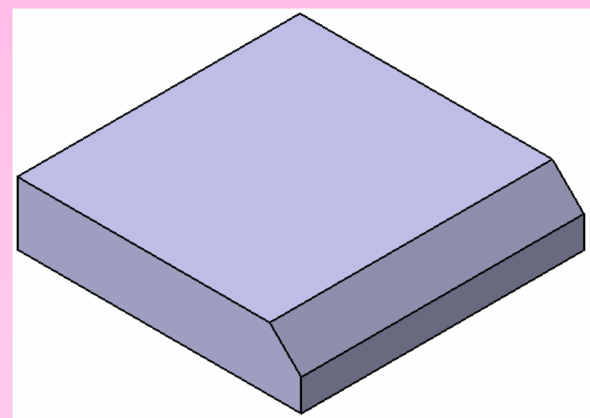
To chamfer the edges of the model, choose the **Chamfer** button from the **Dress-Up Features** toolbar; the **Chamfer Definition** dialog box is displayed, as shown in the figure.



The **Chamfer Definition** dialog box



Edge to be selected



Resulting chamfer

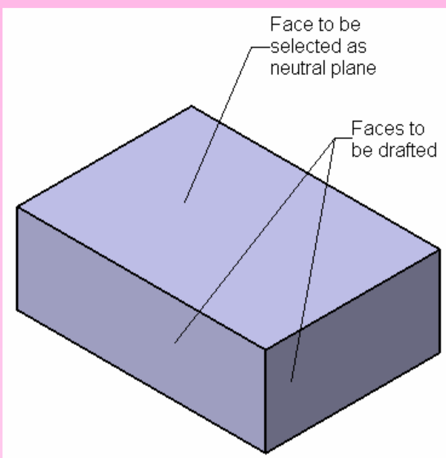
Adding a Draft to the Faces of the Model

- A draft is defined as the process of adding a taper angle to the faces of the model.
- Adding draft to the faces of the model is one of the most important operations, especially, while creating the components that needs to be cast, mold, or formed.
- The **Part** workbench of CATIA V5 provides you with various tools to draft faces of the model.

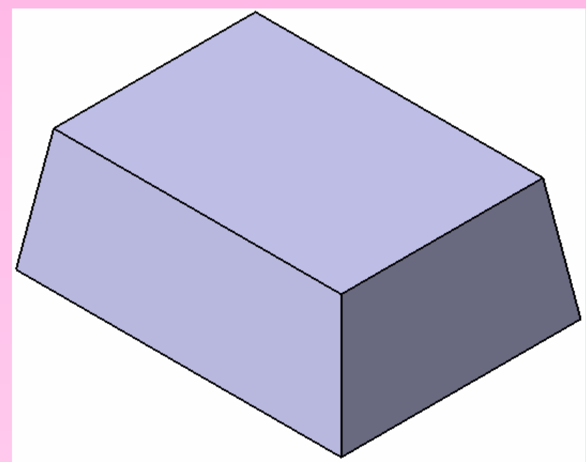
• Adding a Simple Draft

- To add a draft, choose the **Draft Angle** button from the Drafts toolbar.
- The **Draft Definition** dialog box will be displayed, as shown in the figure.

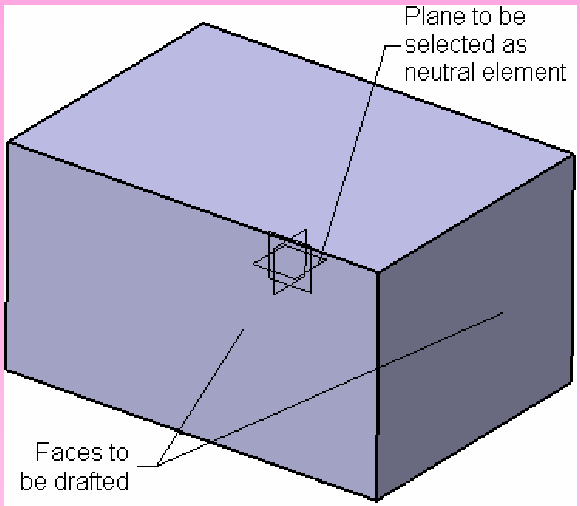
The Draft Definition dialog box



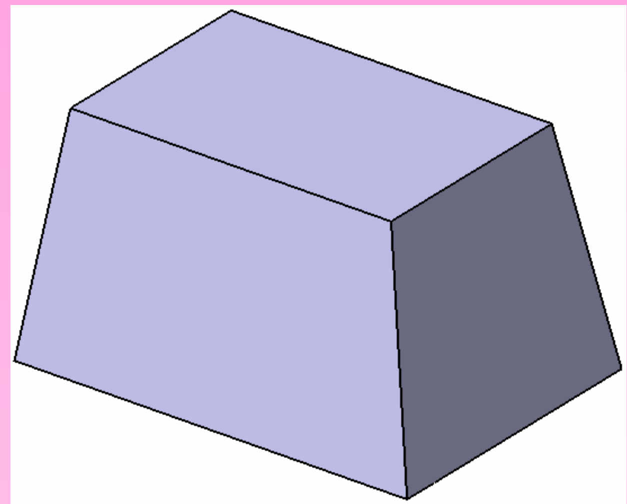
Faces to be selected



Resulting drafted faces

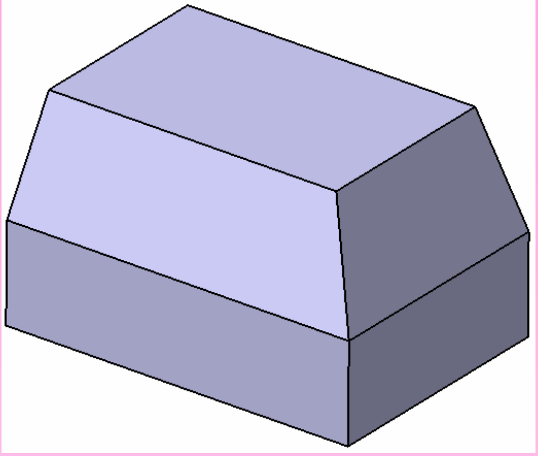


Faces and planes to be selected

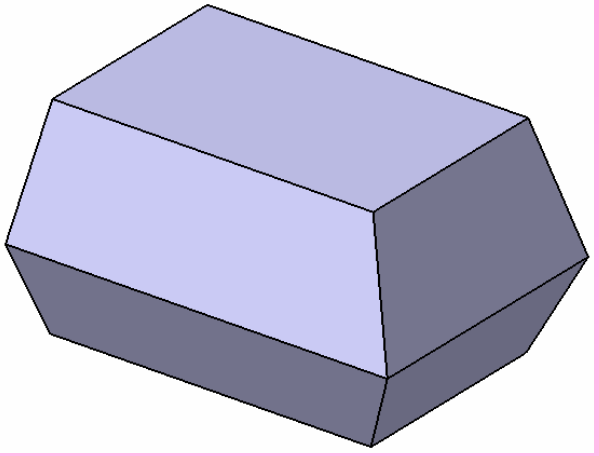


Resulting drafted faces

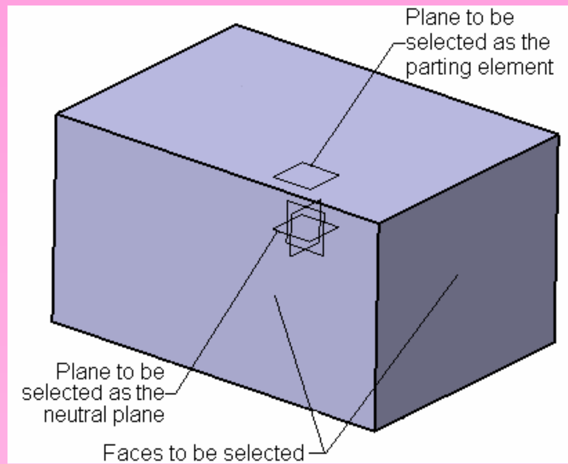
- **Defining the Parting Element While Adding Drafts to the Faces**



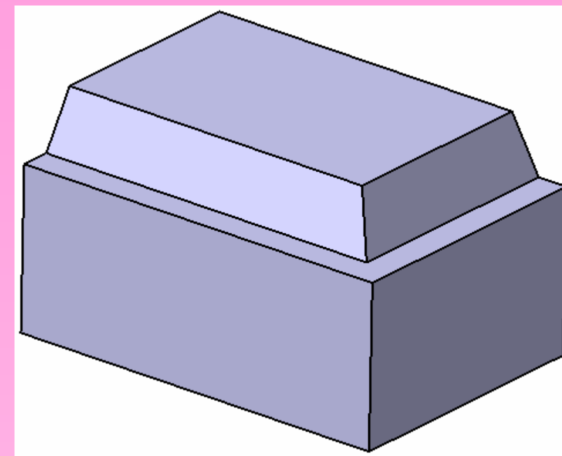
Faces drafted with the Parting = Neutral check box selected.



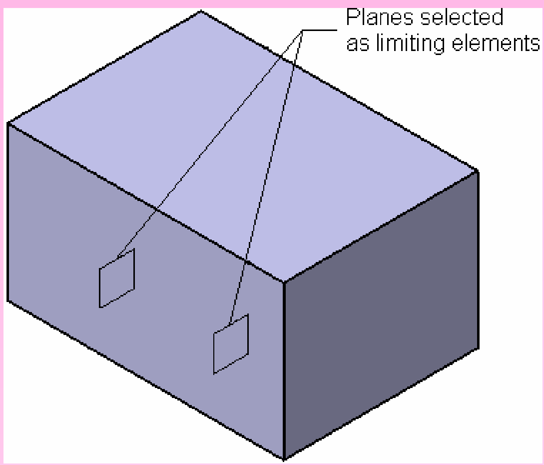
Faces drafted with the Draft both sides check box selected



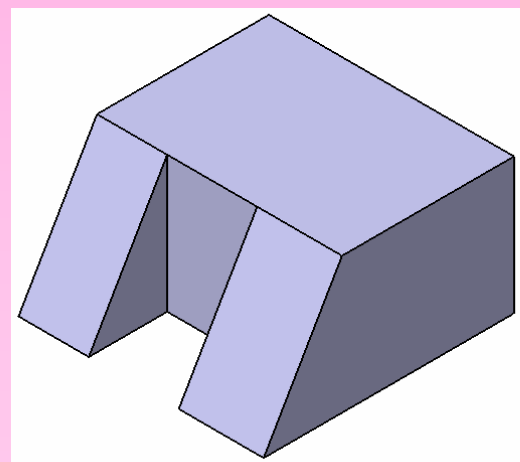
References to be selected



Resulting drafted faces



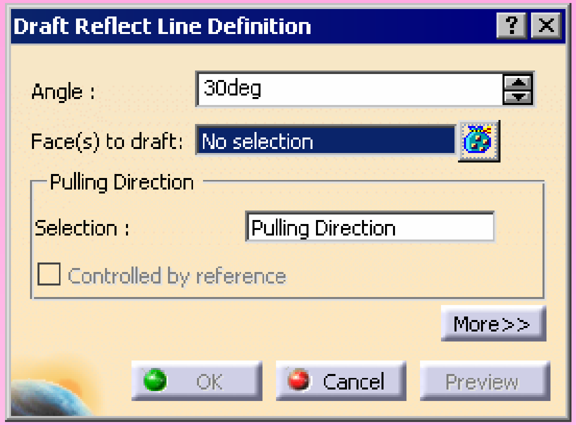
Limiting faces to be selected



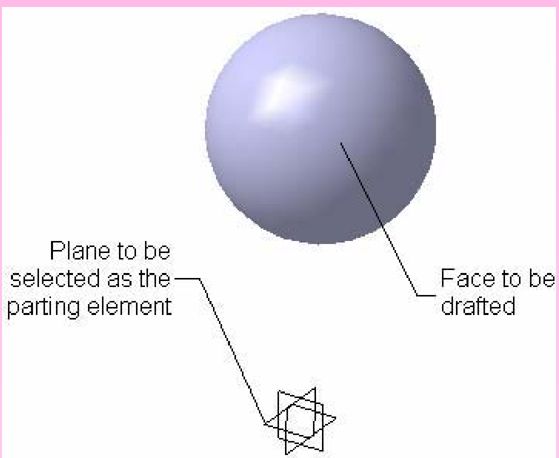
Resulting drafted face

• Adding Drafts using the Reflect Line

- To create this type of draft feature, choose the **Draft Reflect Line** button from the **Drafts** toolbar.
- The **Draft Reflect Line Definition** dialog box is displayed, as shown in the figure.



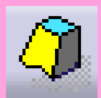
The Draft Reflect Line Definition dialog box



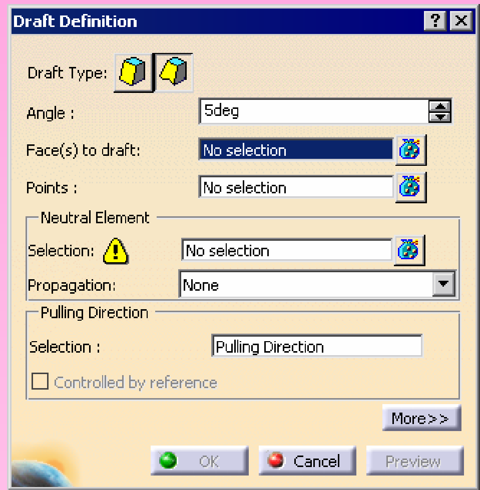
Face and plane to be selected

Resulting draft feature

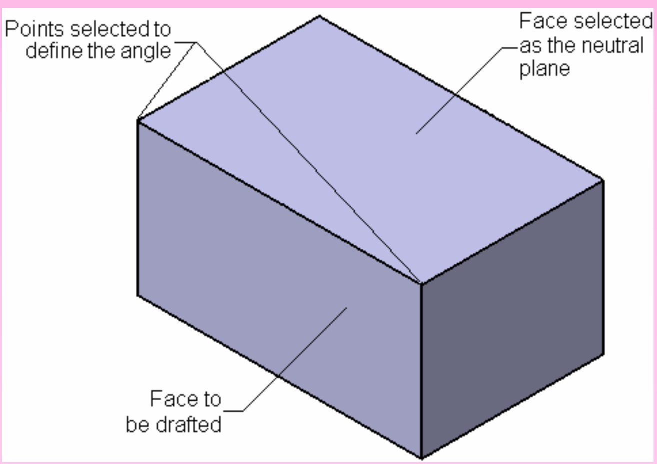
• Adding a Variable Angle Draft



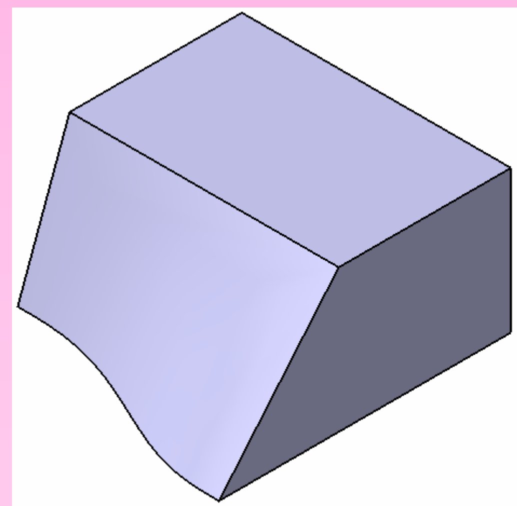
- To create a variable angle draft, choose the **Variable Angle Draft** from the **Drafts** toolbar.
- The **Draft Definition** dialog box is displayed, as shown in the figure.



The Draft Definition dialog box

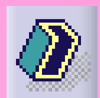


References to be selected

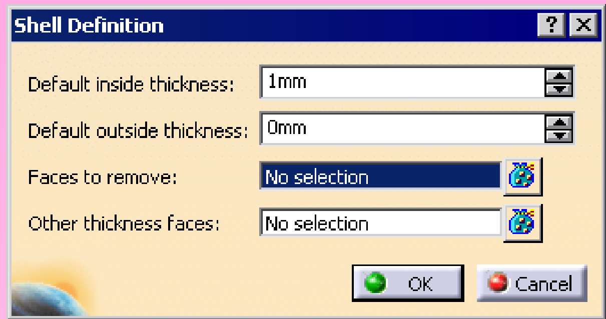


Face after adding draft

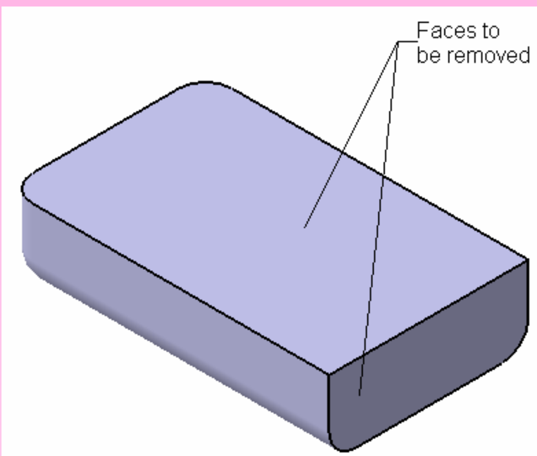
• Creating a Shell Feature



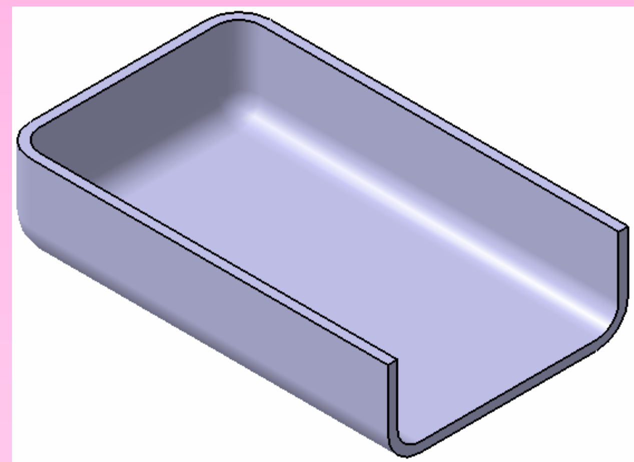
- To create a shell feature, choose the **Shell** button from the **Dress-Up Features** toolbar.
- The **Shell Definition** dialog box is displayed, as shown in the figure.



The Shell Definition dialog box

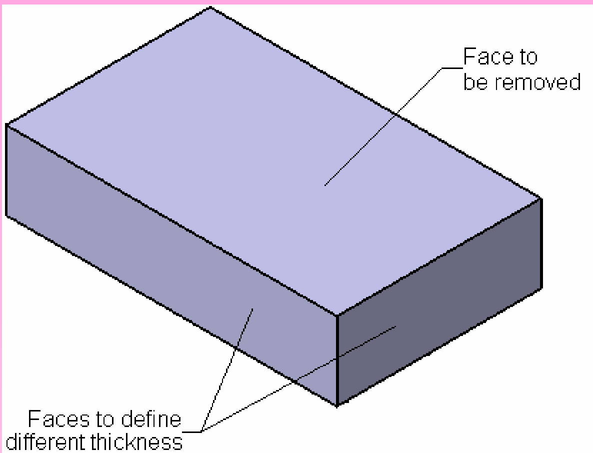


Faces to be selected for removal

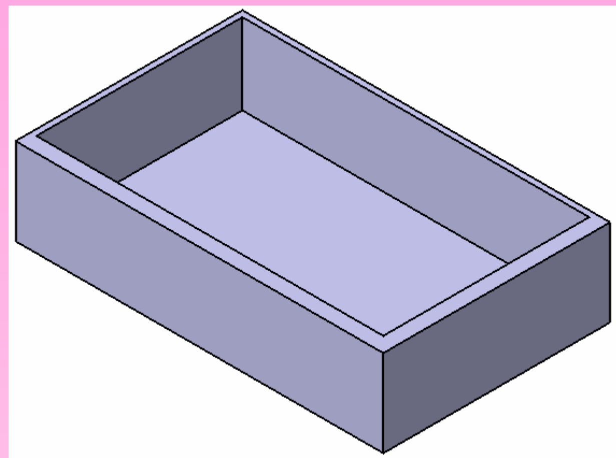


Resulting shelled model

- **Creating a Multithickness Shell**



Faces to be selected



Resulting shelled model

Tutorial 1

In this tutorial, you will create the model of the nozzle of a vacuum cleaner shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

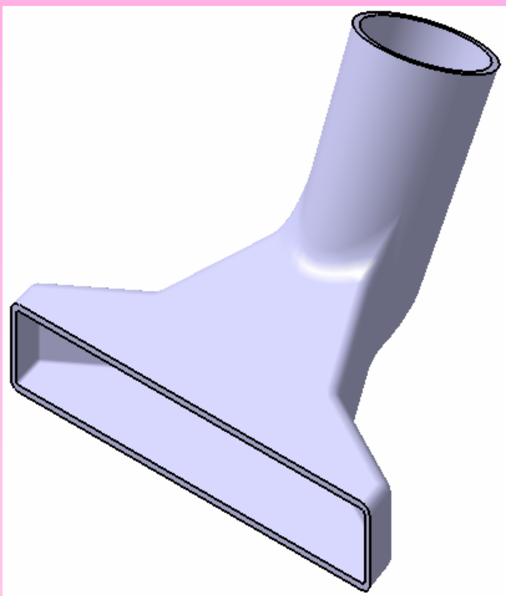


Figure A Model of the Vacuum Cleaner for Tutorial 1

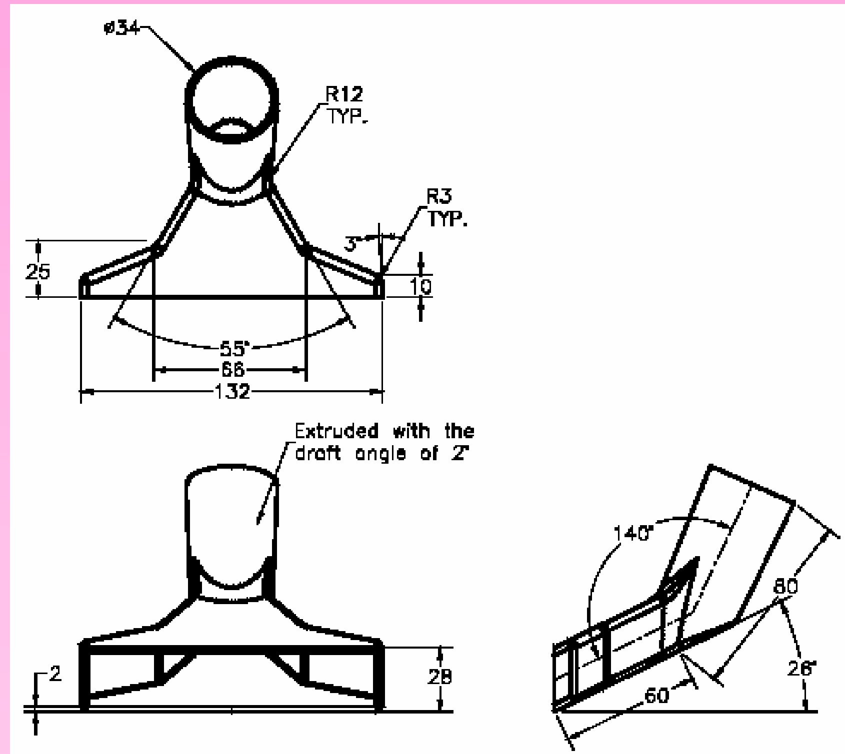


Figure B Views and dimensions of the Vacuum Cleaner for Tutorial 1

1. Start a new file in the **Part** workbench and create the base feature of the model by extruding the sketch along the selected direction, as shown in **Figure C** through **Figure G**.

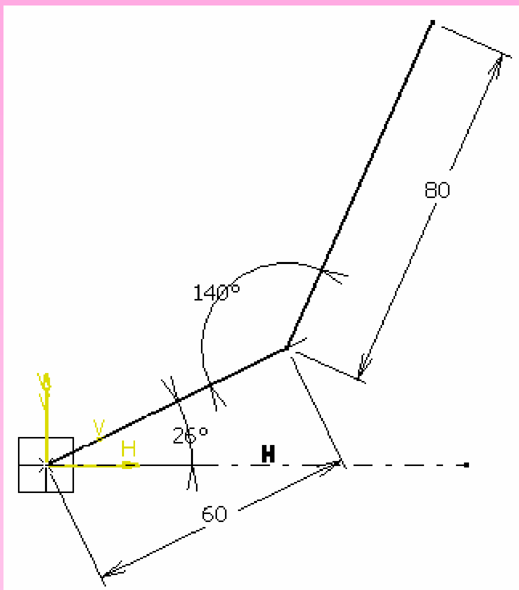


Figure C Reference sketch

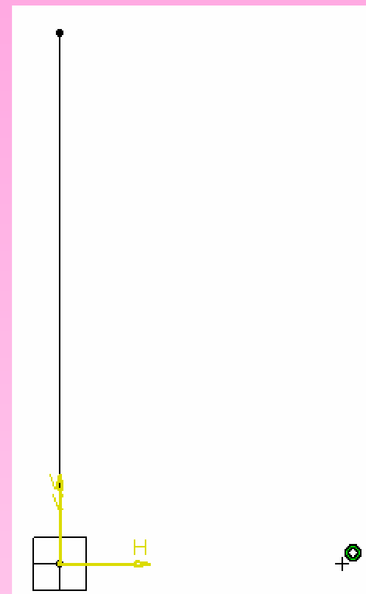


Figure D Point to be placed

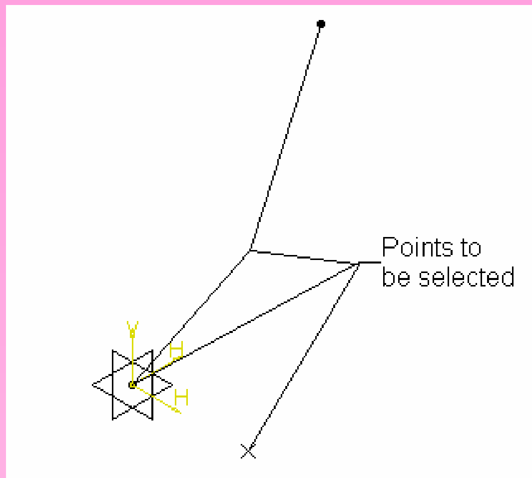


Figure E Points to be selected to create plane

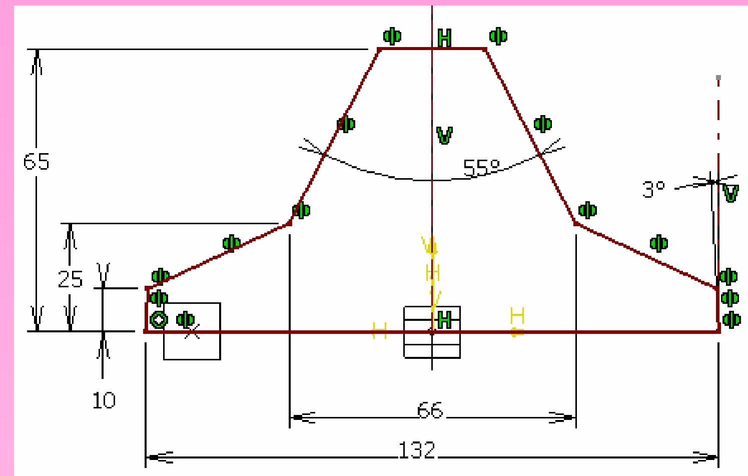


Figure F Sketch of the base feature

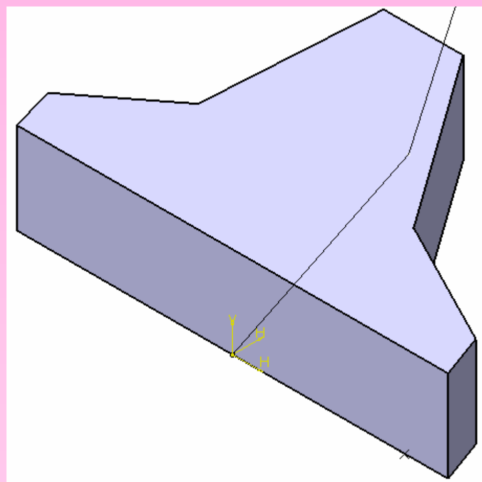


Figure G Model after creating the base feature

2. Create the second feature of the model by extruding a sketch using the **Drafted Fillet Pad** tool, as shown in **Figure H** and **Figure I**.

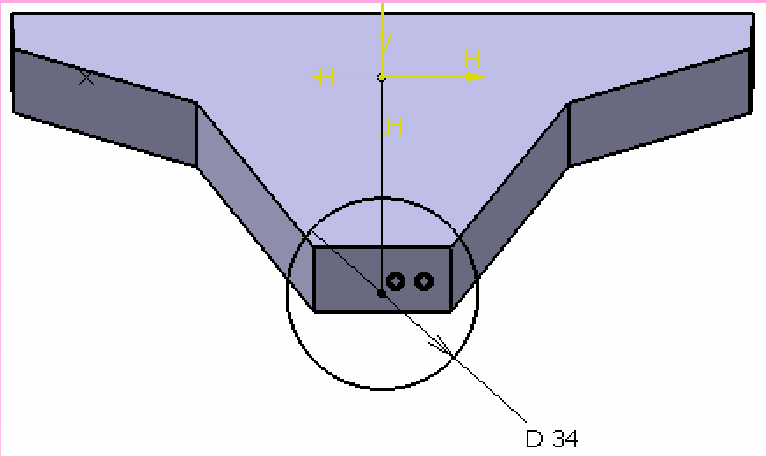


Figure H Sketch for the second feature

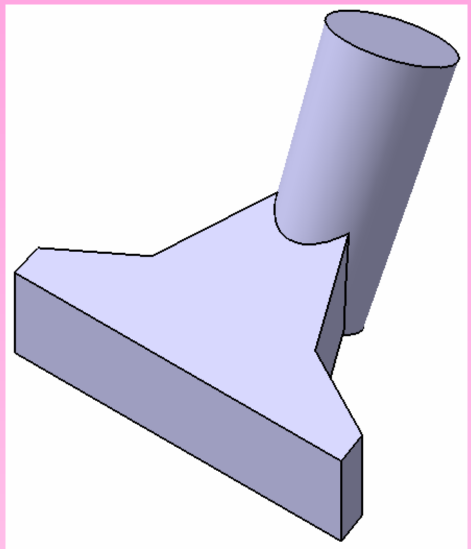


Figure I Resulting second feature

3. Create the third feature of the model, which is a cut feature. It will be used to remove the unwanted portion of the second feature, as shown in **Figure J** and **Figure K**.

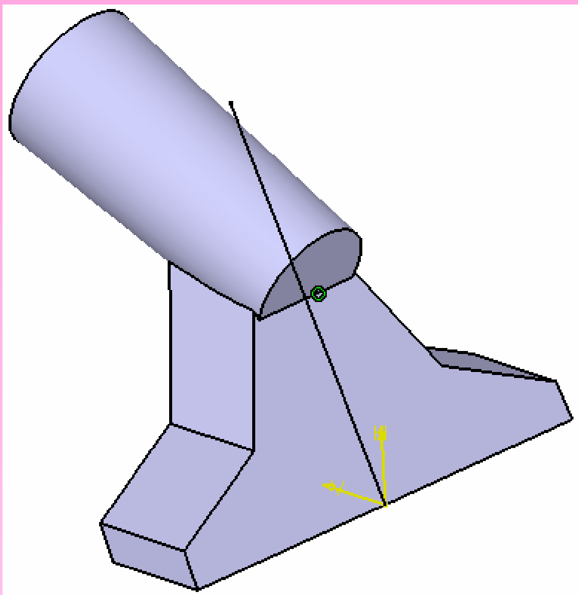


Figure J Sketch for the Pocket feature

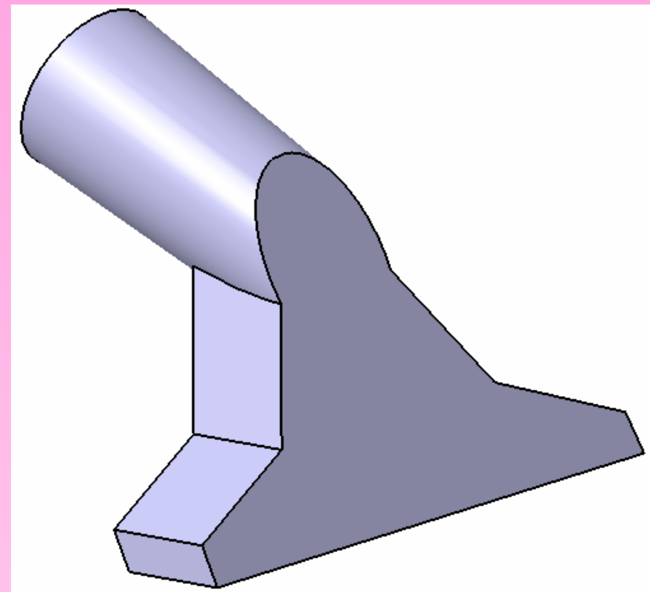


Figure K Model after creating the third feature

4. Apply fillets to all edges of the model, as shown in **Figure L**, **Figure M**, **Figure N** and **Figure O**.

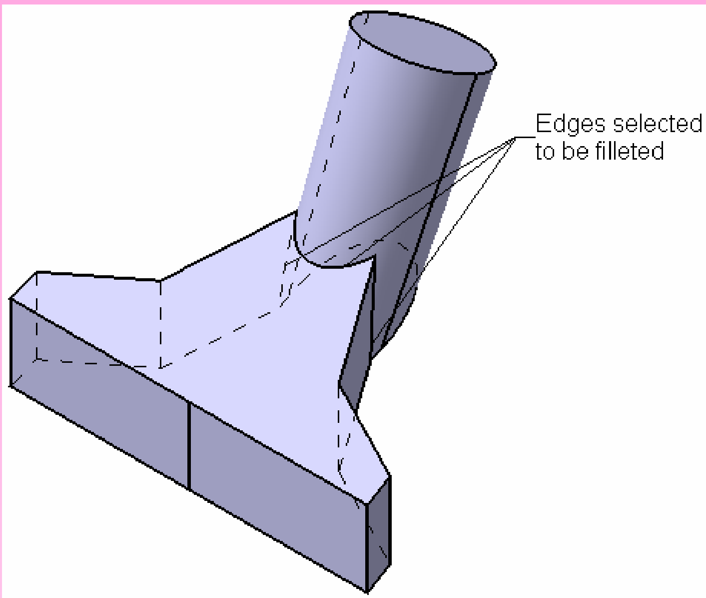


Figure L **Edges to be selected**

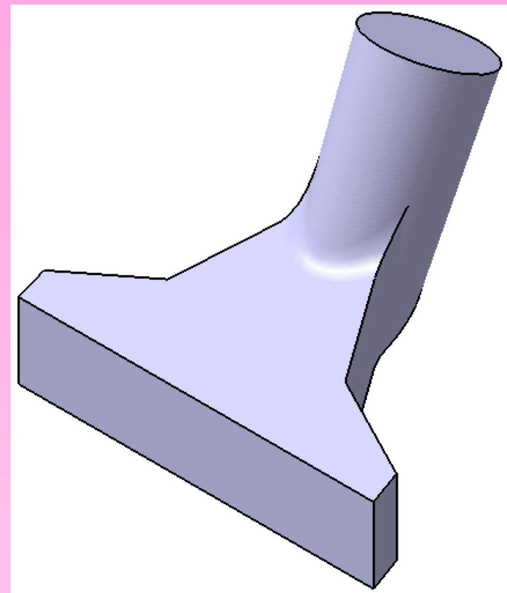


Figure M **The model after creating the fillet**

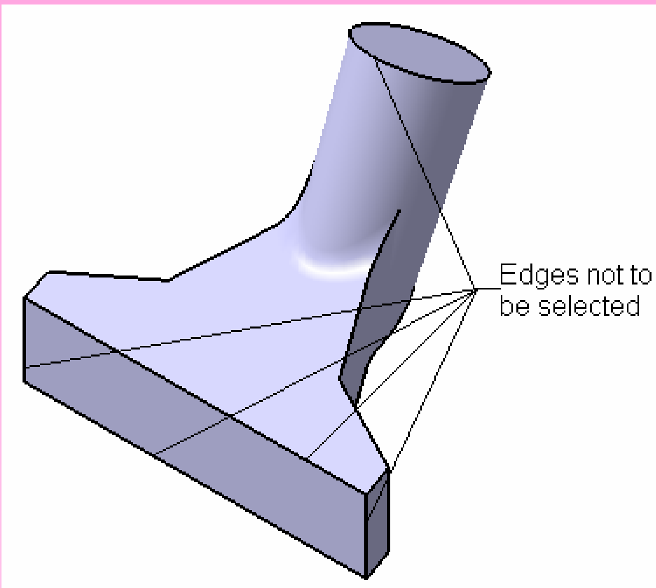


Figure N Edges not to be selected

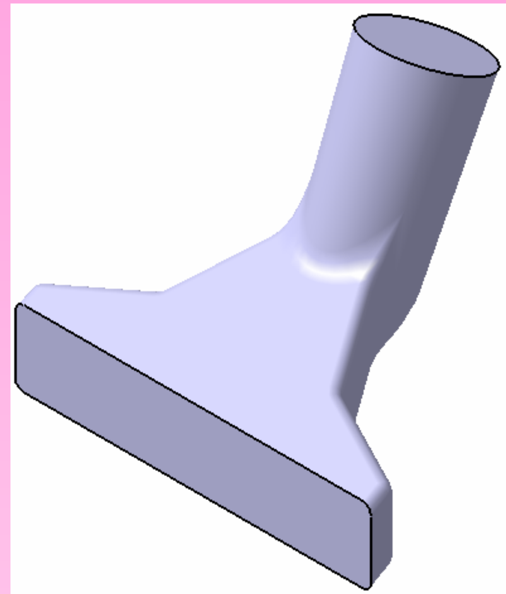


Figure O Model after creating second fillet

4. Shell the model using the **Shell** tool, as shown in **Figure P** and **Figure Q**.

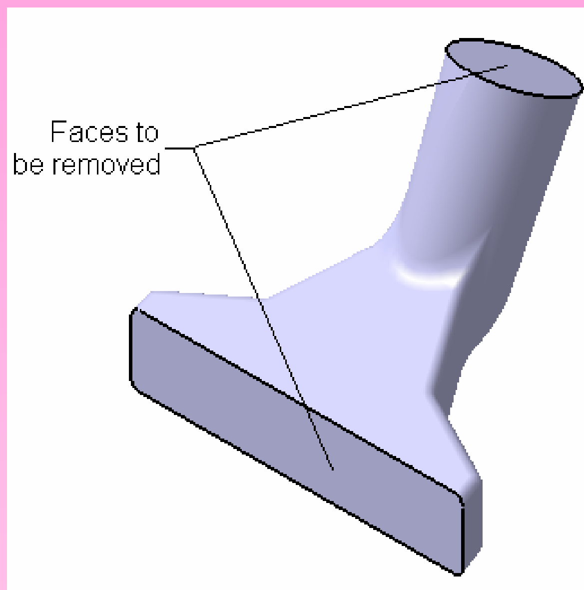


Figure P Faces to be removed

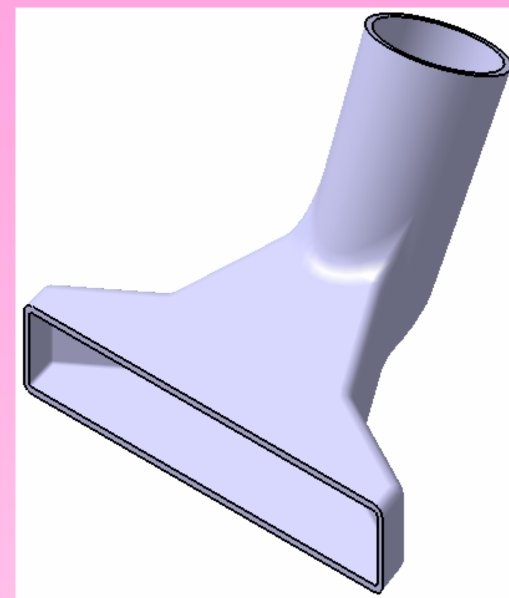


Figure Q Final model after shelling

5. Save the file in **\My Documents\CATIA\c05** folder and then close it.

Tutorial 2

In this tutorial, you will create the model of the plastic cover shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 30 min)**

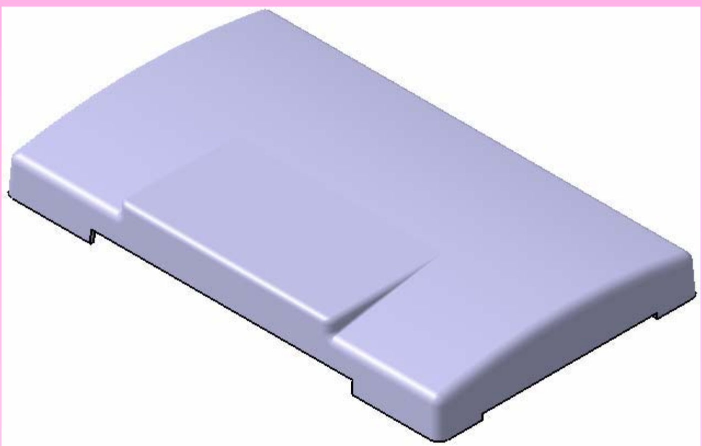


Figure A Model of the Plastic Cover for Tutorial 2

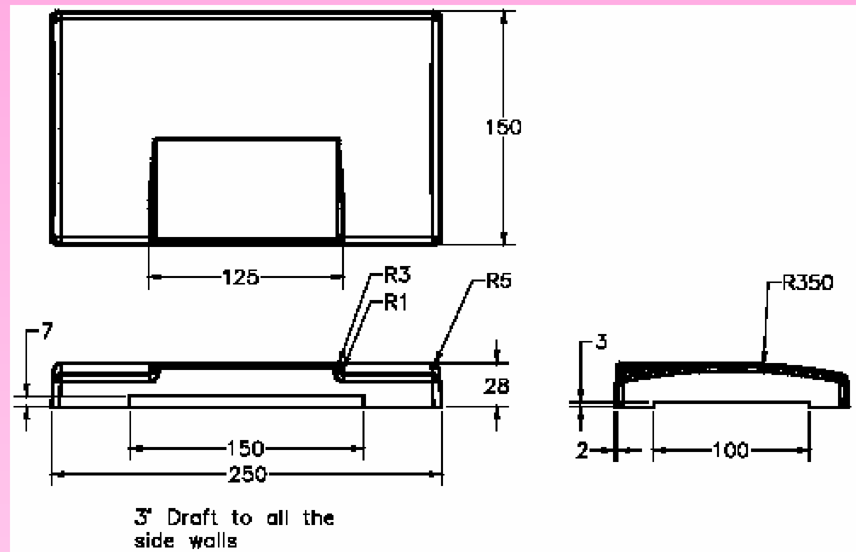


Figure B Views and dimensions of the Plastic Cover for Tutorial 2

1. Create the base feature of the model by extruding the sketch drawn on zx plane equally to both the sides of the sketch plane, as shown in **Figure C** and **Figure D**.

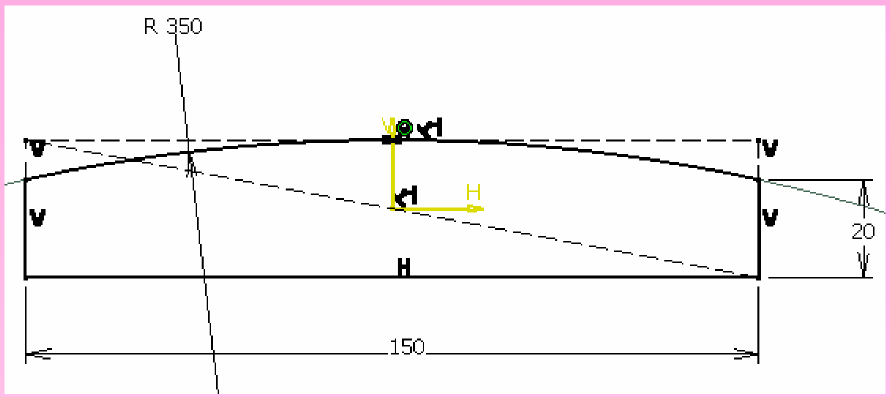


Figure C Sketch of the base feature

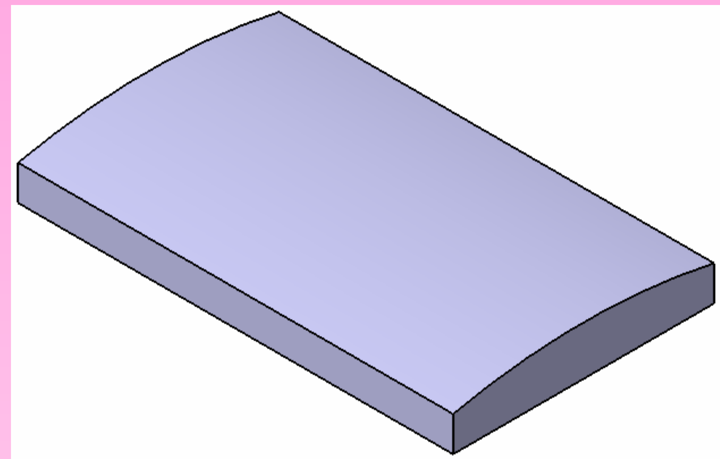


Figure D The model after creating the base feature

2. Create the second feature by extruding the sketch drawn on a plane created at an offset distance from the xy plane, as shown in **Figure E** and **Figure F**.

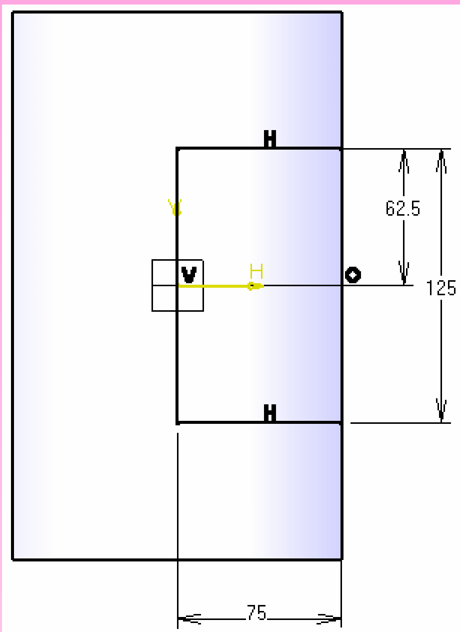


Figure E Sketch of the second feature

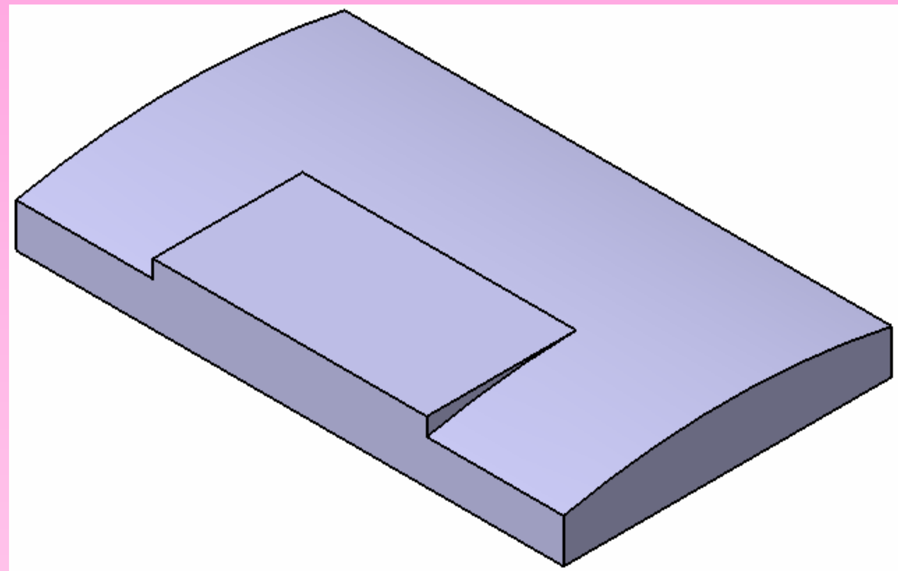


Figure F The mode after creating the second feature

3. Add the draft feature to all faces of the model except the upper and the lower faces, as shown in **Figure G**.

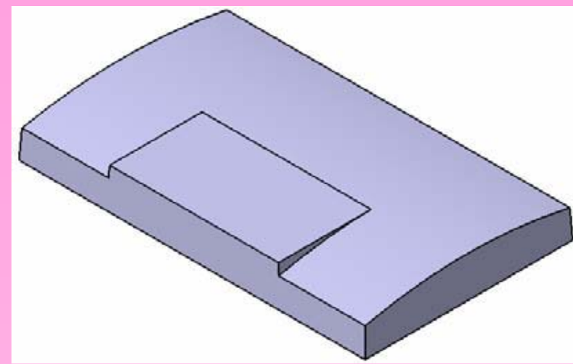


Figure G The model after drafting all the vertical faces

4. Fillet the edges of the model, as shown in **Figure H** through **Figure M**.

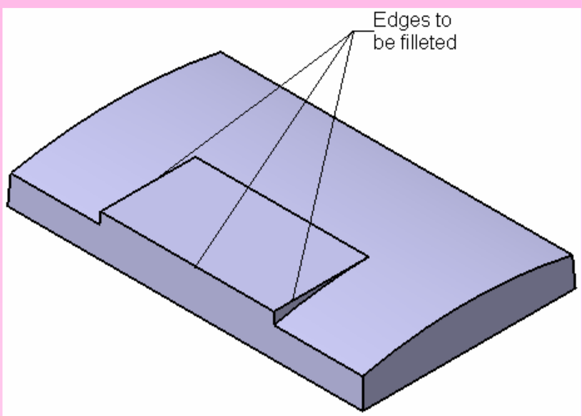


Figure H Edges to be selected

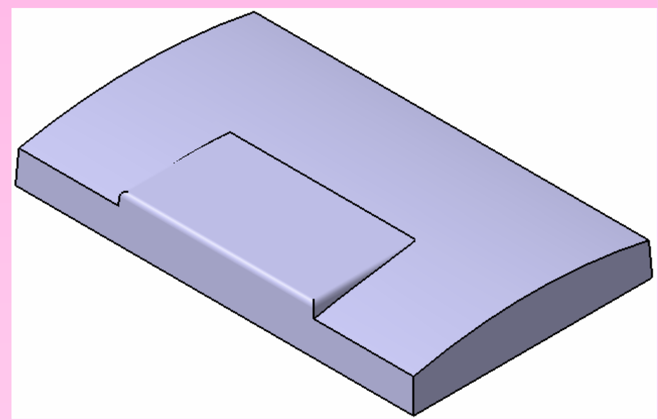


Figure I The model after filleting the first set of edges

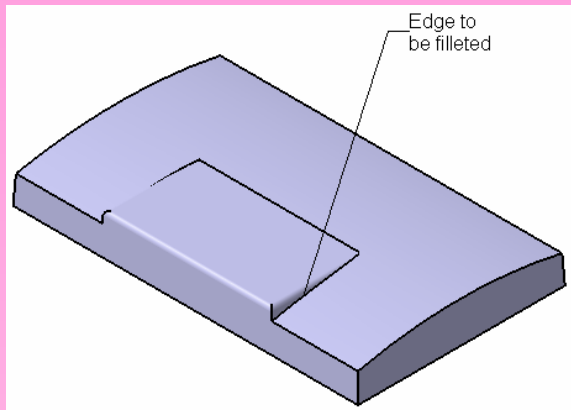


Figure J Edge to be filleted

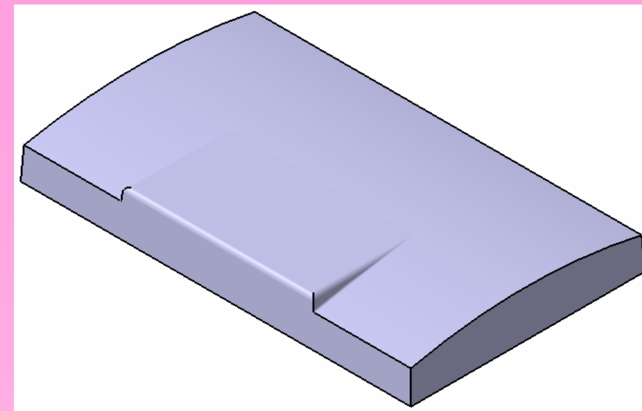


Figure K The model after filleting the second set

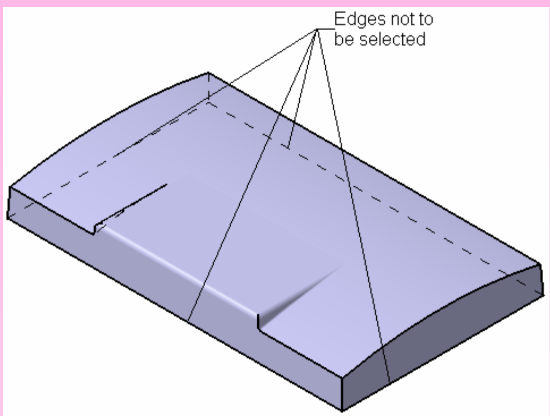


Figure L Edges not to be selected

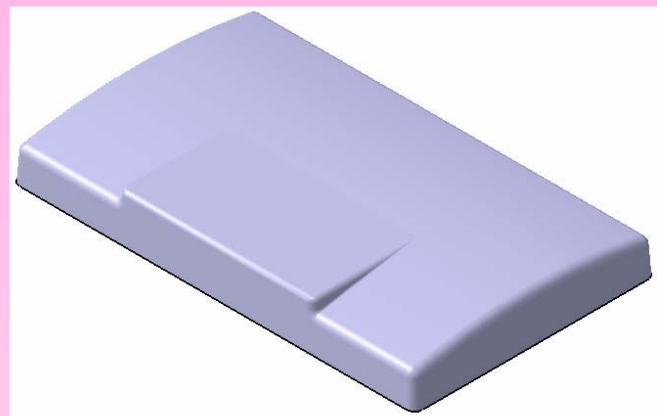


Figure M Resulting filleted model

5. Shell the model using the **Shell** tool by removing the bottom face of the model, as shown in **Figure N** and **Figure O**.

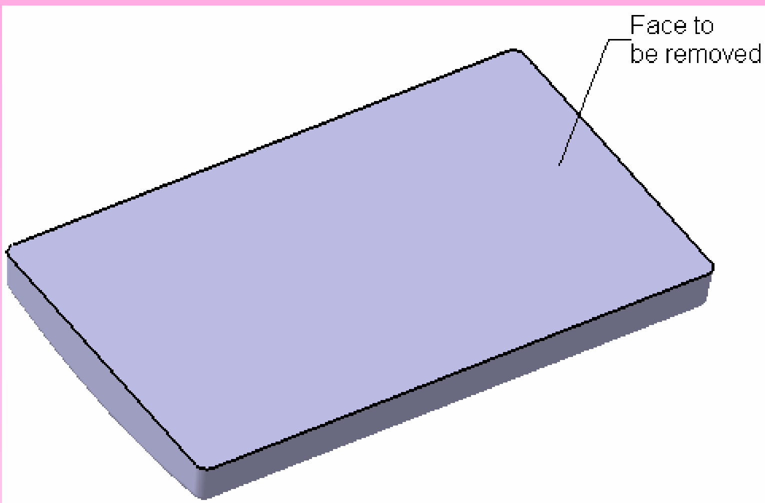


Figure N Face to be removed

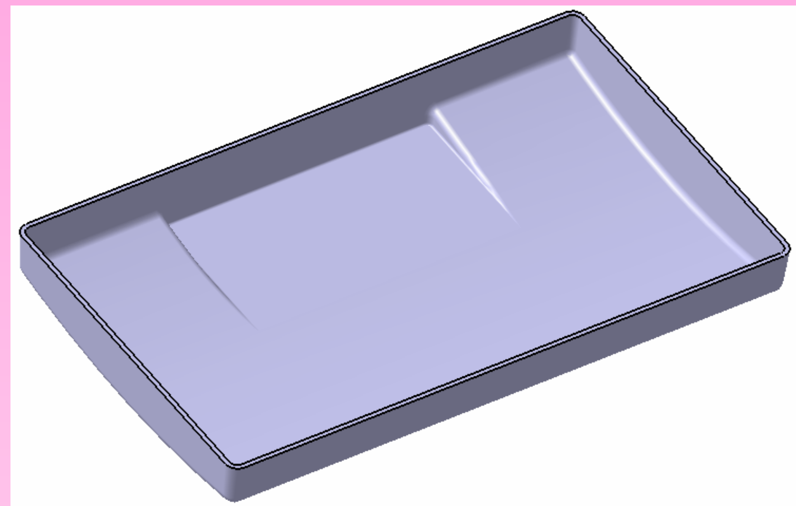


Figure O Resulting shelled model

6. Create two pocket features to complete the model, as shown in **Figure P**.

Figure P Final model after creating the remaining features

7. Save the file in **\My Documents\CATIA\c05** folder and then close it.

Exercise 1

Create the model of the Clutch Lever shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 30 min)**

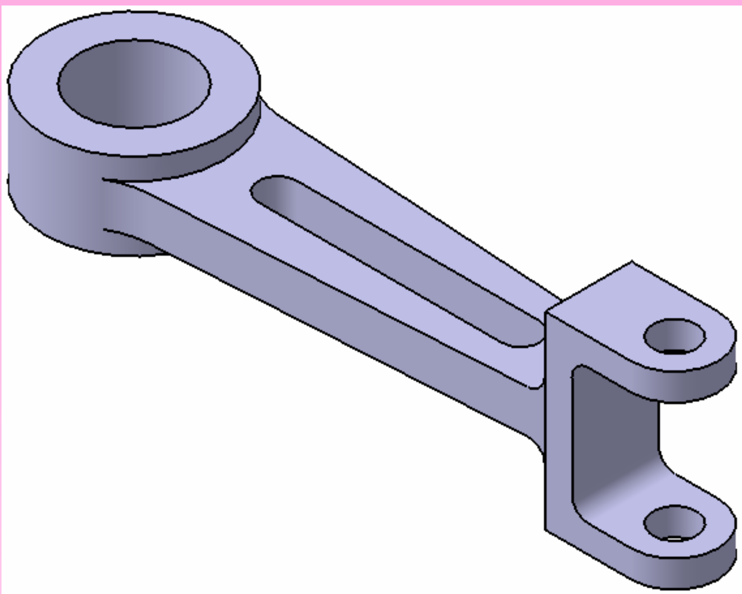


Figure A Model of the Clutch Lever for Exercise 1

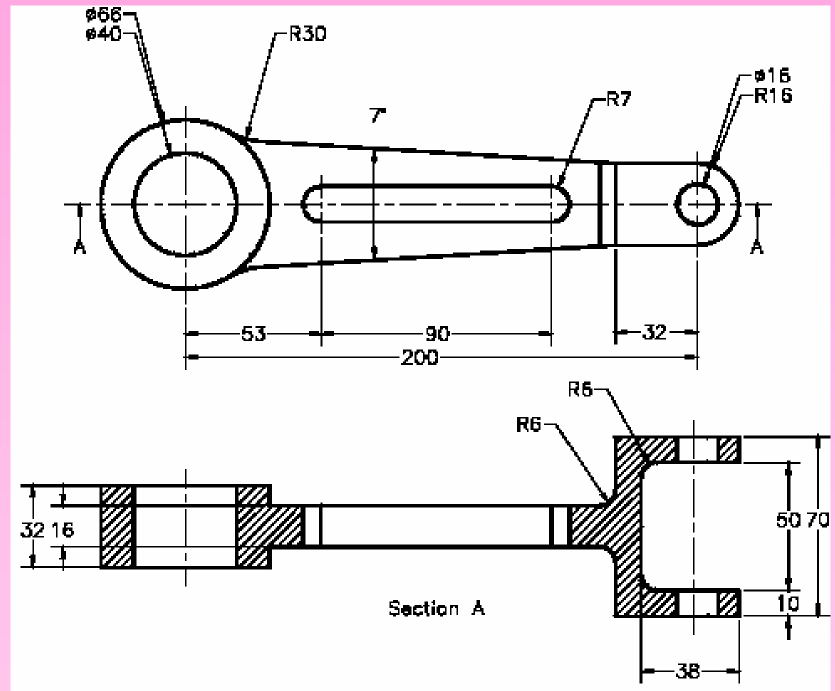


Figure B Views and dimensions of the Clutch Lever for Exercise 1

Exercise 2

Create the model of the Clamp Stop shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 60 min)**

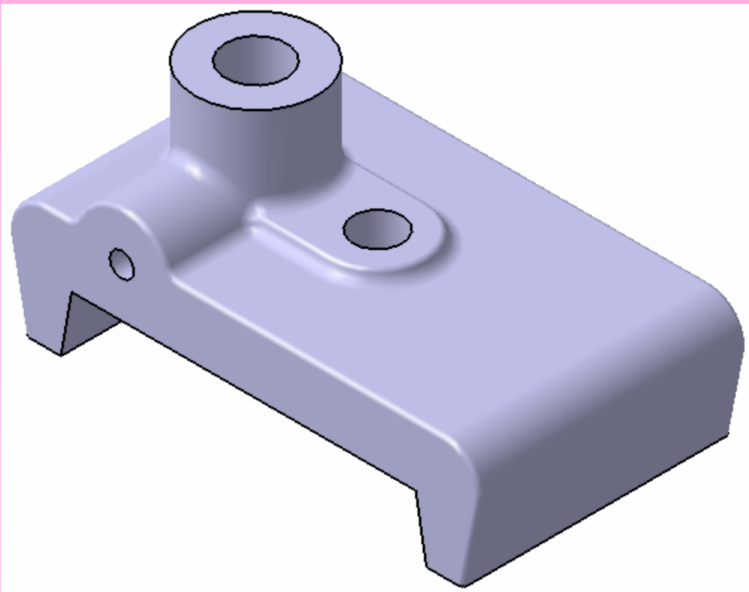


Figure A Model of the Clamp Stop for Exercise 2

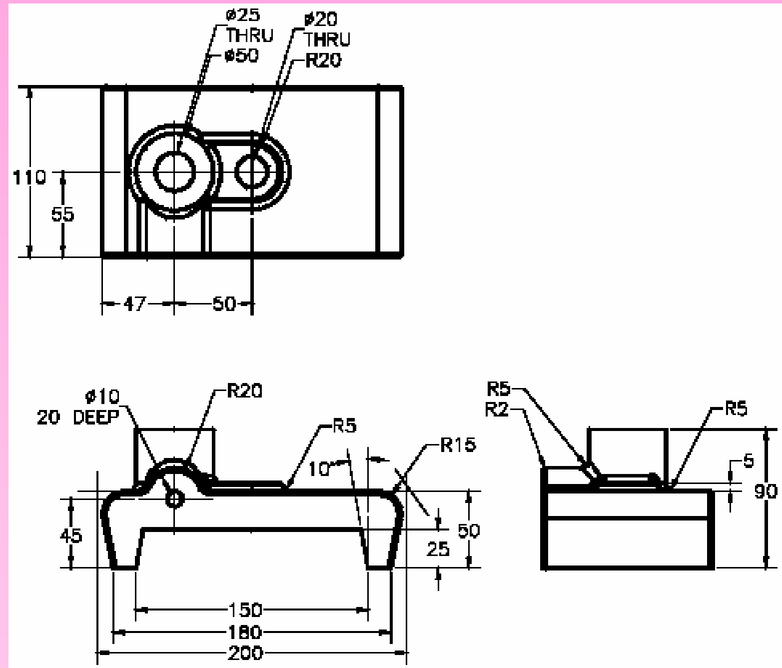


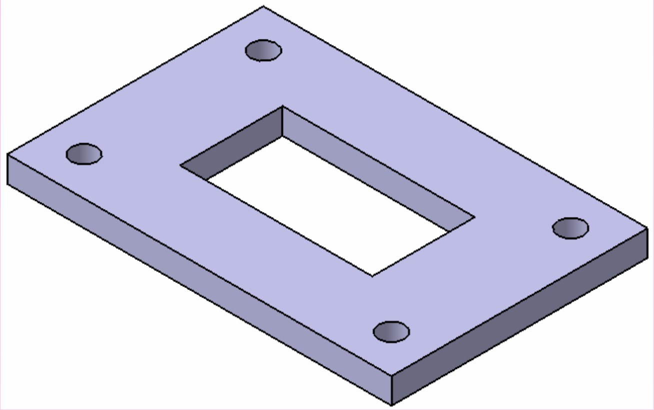
Figure B Views and dimensions of the Clamp Stop for Exercise 2

Learning Objectives:

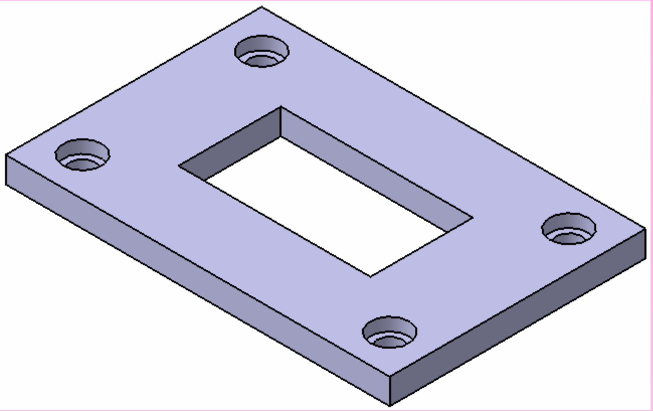
- *Edit features using the Definition option.*
- *Edit features by double-clicking.*
- *Edit sketches of the sketch-based features.*
- *Manage features and sketches by cut, copy, and paste functionality.*
- *Cut, copy, and paste features and sketches from one file to another.*
- *Copy features using drag and drop.*
- *Delete features.*
- *Deactivate features.*
- *Activate the deactivated features.*
- *Define features in the work object.*
- *Reorder features.*
- *Understand parent child relationships.*
- *Understand the concept of update diagnose.*
- *Measure elements.*

EDITING FEATURES OF A MODEL

- Editing is one of the most important aspect of the product design cycle.
- Almost all designs require editing either during their creation or after they are created.



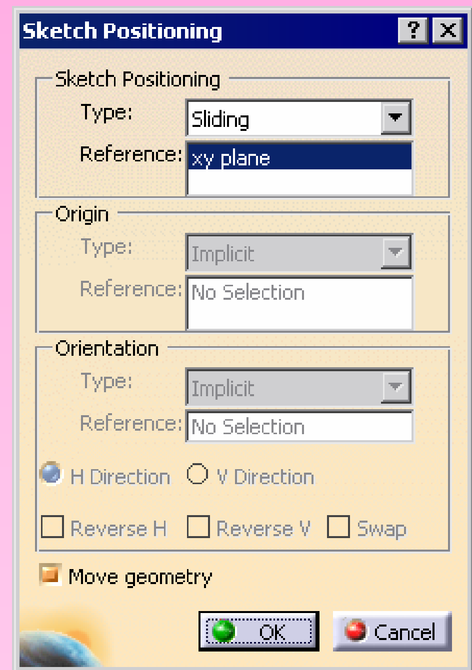
Base Plate with simple holes



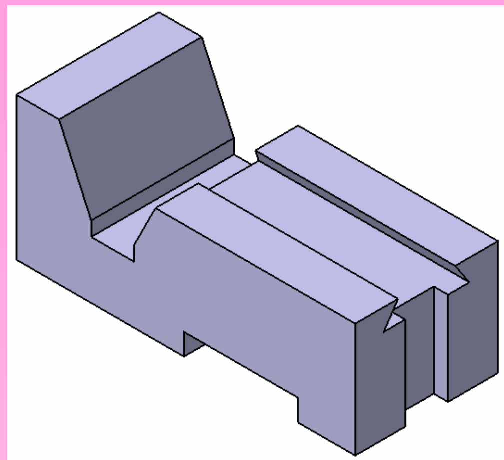
Modified Base Plate

The editing methods in CATIA V5 are listed next:

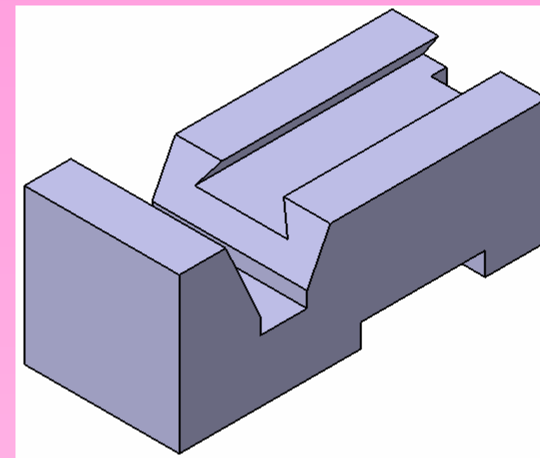
- **Editing Using the Definition Option**
- **Editing by Double-clicking**
- **Editing the Sketch of a Sketch-based Feature**
- **Redefining the Sketch Plane of Sketches**
  - To edit a sketch, select it from the specification tree and invoke the contextual menu.
  - Move the cursor to the name of the sketch at the bottom of the contextual menu and choose the **Change Sketch Support** option from the cascading menu.
  - The **Sketch Positioning** dialog box is displayed, as shown in the figure.



The Sketch Positioning dialog box

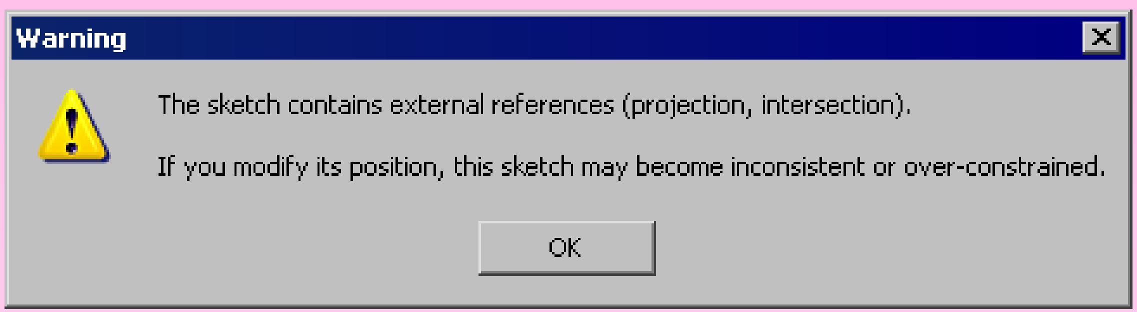


Base feature created on the yz plane



The model after redefining the sketch plane of the base feature to zx plane

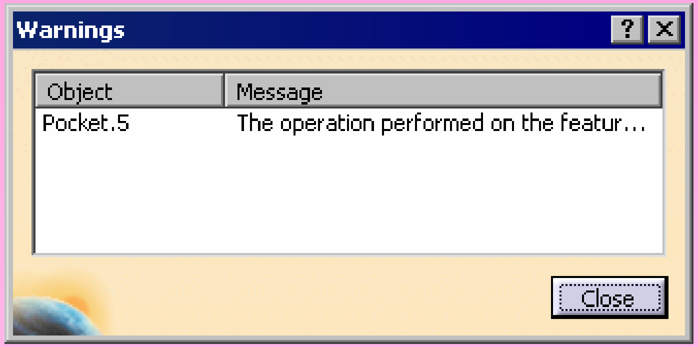
When you redefine the sketch plane of the sketch-based features, sometimes a **Warning** dialog box is displayed, as shown in the figure.



The Warning dialog box

- **Managing Features and Sketches by Cut, Copy, and Paste Functionality**

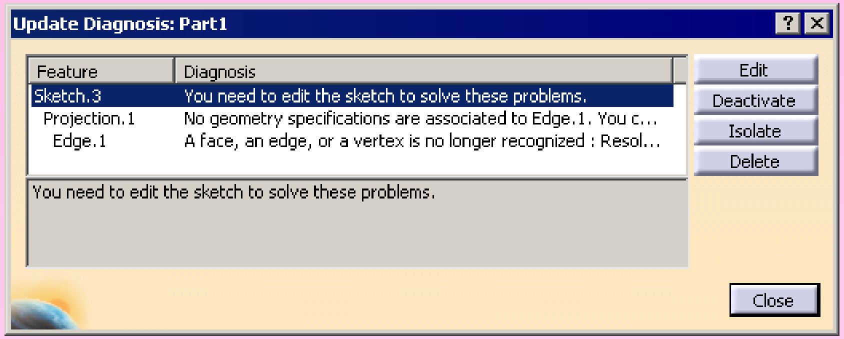
  If the resulting feature merges with the model, the **Warnings** dialog box is displayed, as shown in the figure.



The **Warnings** dialog box

- **Understanding the Concept of Update Diagnose**

  Sometime, after editing or modifying a feature, the **Update Diagnose** dialog box is displayed, as shown in the figure.

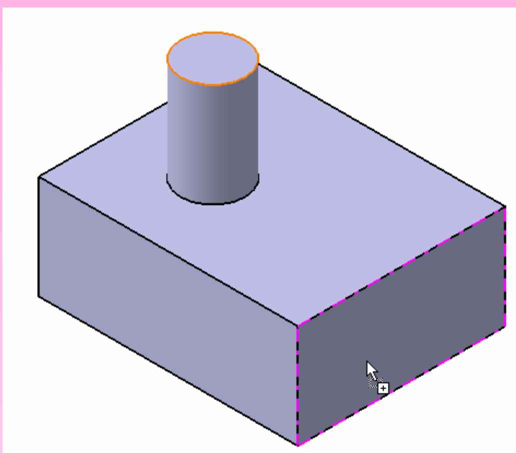


The **Update Diagnosis** dialog box

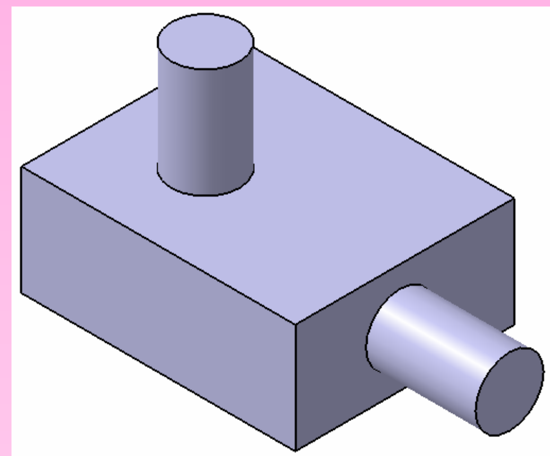
- **Cut, Copy, and Paste Features and Sketches from One File to Another**

  Select the fillet that you need to copy from the geometry area and use the CTRL+C keys to copy the selected feature.

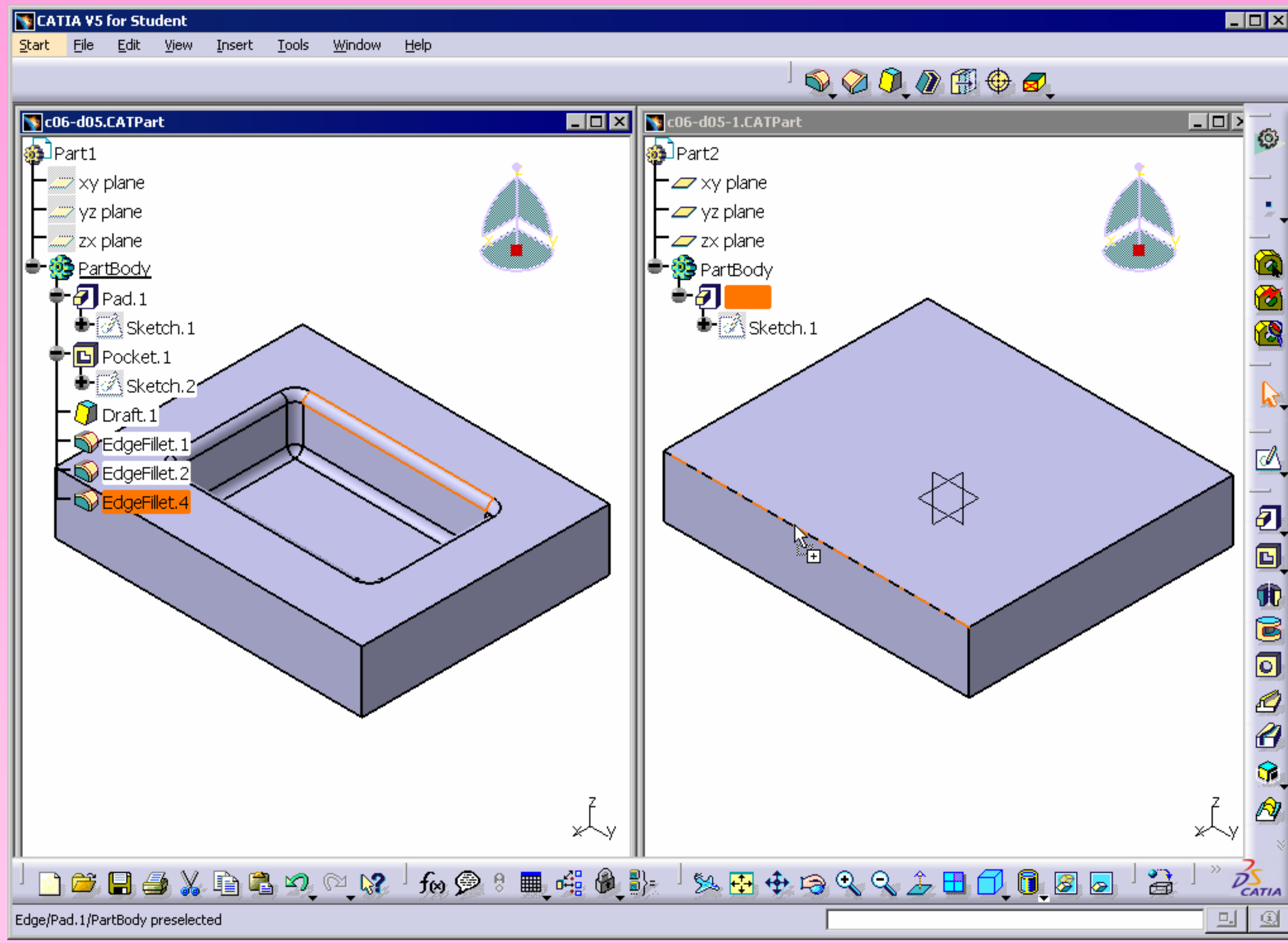
- **Copying Features Using Drag and Drop**



Feature being dragged



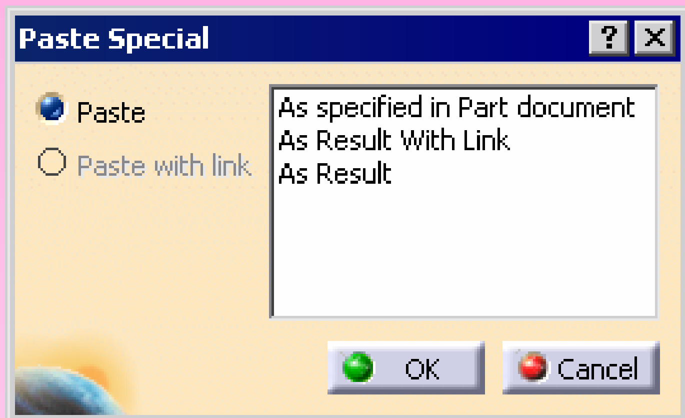
Resulting pasted feature after editing its sketch



Fillet feature being dragged to be pasted on the edge of the model in another file

Copying and Pasting PartBodies

- To use this option, copy the **PartBody** from one of the files and the select the **PartBody** in the other file in which you need to paste it.
- Right-click to invoke the contextual menu and choose the **Paste Special** option.
- The **Paste Special** dialog box is displayed, as shown in the figure.

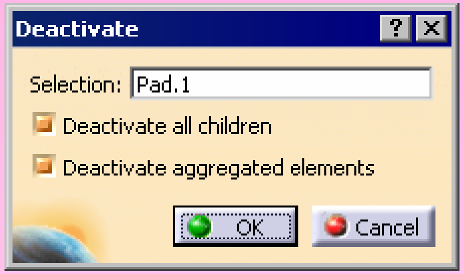


The **Paste Special** dialog box

- Deleting Unwanted Features

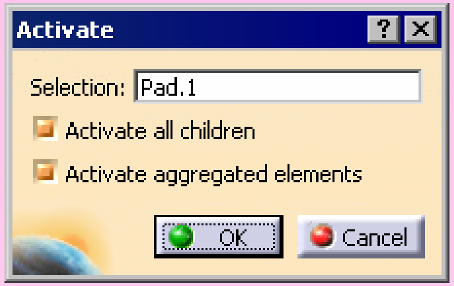
The **Delete** dialog box

- Deactivating Features



The **Deactivate** dialog box

- Activating the Deactivated Features



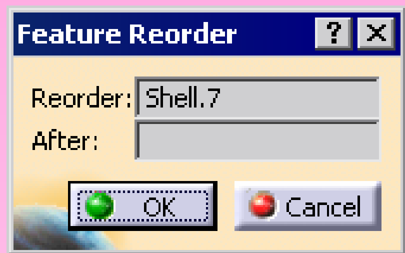
The **Activate** dialog box

- ## Defining Features in Work Object

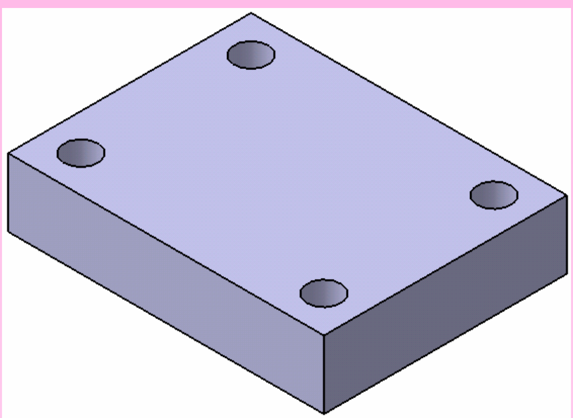
Defining a feature in the work object is as a process in which you rollback the model to an earlier stage.

- ## Reordering the Features

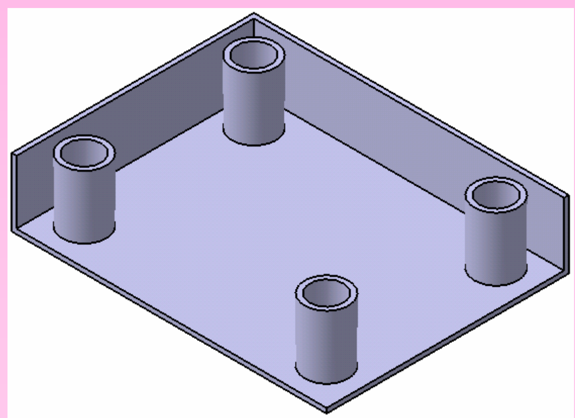
Reordering is defined as a process of changing the order, in which the features were created in the model.



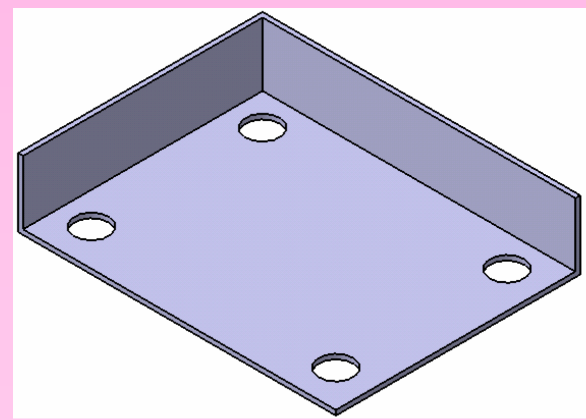
The Feature Reorder dialog box



Model with through holes on the top face



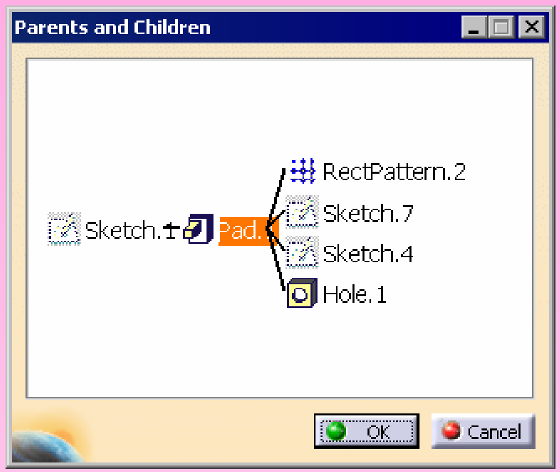
Model after shelling



The model after reordering the Shell feature

• Understanding Parent Child Relationships

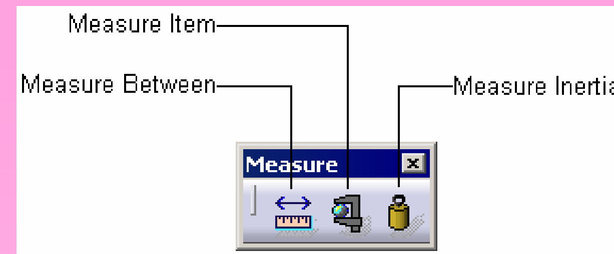
Invoke the contextual menu and choose the **Parents/Children** option; the **Parents and Children** dialog box is displayed, as shown in the figure.



The Parents and Children dialog box

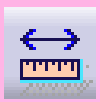
➢ MEASURING ELEMENTS

- The **Part** workbench of CATIA V5 also provides you with the tools to measure distance, angle, radius, area, and inertia.
- To measure these elements, you can use the tools available in the **Measure** toolbar, as shown in the figure.

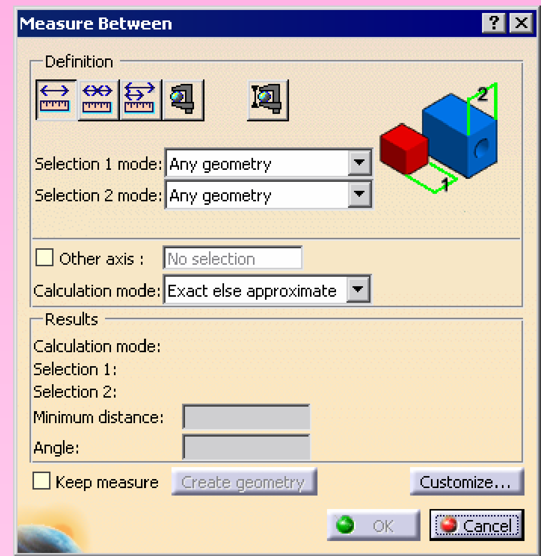


The **Measure** toolbar

• Measuring Between Elements



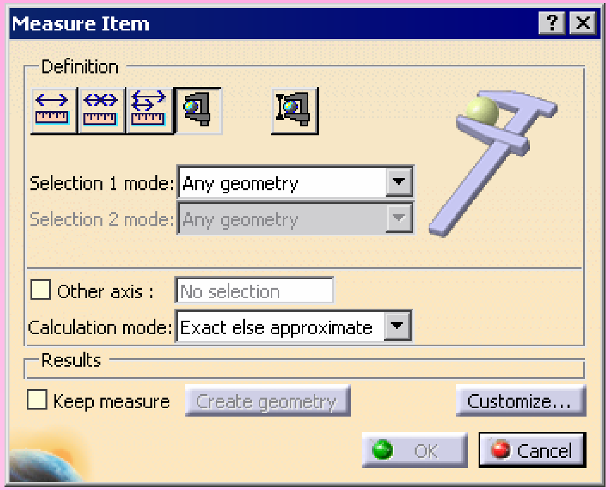
To measure the distance and angle between two elements, choose the **Measure Between** button from the **Measure** toolbar;the **Measure Between** dialog box is displayed, as shown in the figure.



The **Measure Distance** dialog box

• Measuring Items

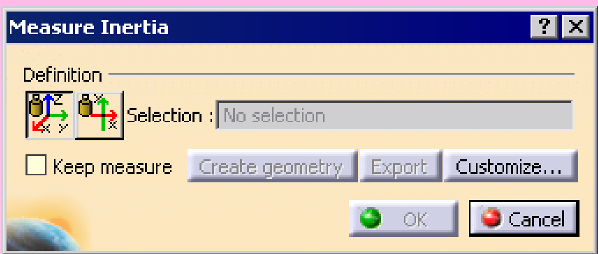
- To measure elements using this tool, choose the **Measure Item** button from the **Measure** toolbar.
- The **Measure Item** dialog box is displayed, as shown in the figure.



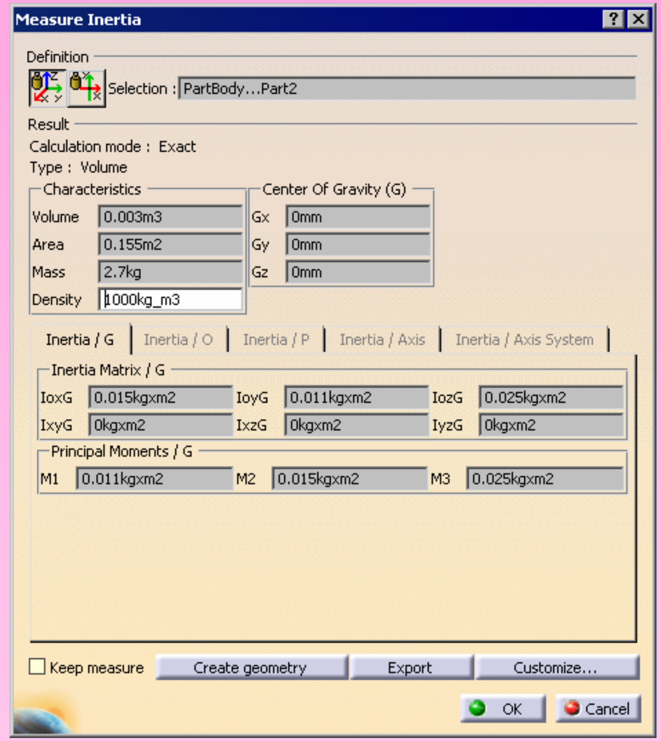
The Measure Item dialog box

• Measuring Inertia

Choose the **Measure Inertia** button from the **Measure** tool; the **Measure Inertia** dialog
box is displayed, as shown in the figure.



The Measure Inertia dialog box



The expanded Measure Inertia dialog box

Tutorial 1

In this tutorial, you will create the model of the Bottom Seat shown in **Figure A**. Its views and dimensions are shown in **Figure B**. After creating this model, you will perform the following modifications.

1. Change the two holes on the front face of the model to countersunk holes.
2. Change the hole on the right face of the model to counterbored hole.
3. Change the curved pocket feature on the upper face of the model to a rectangular slot.

(Expected time: 45 min)

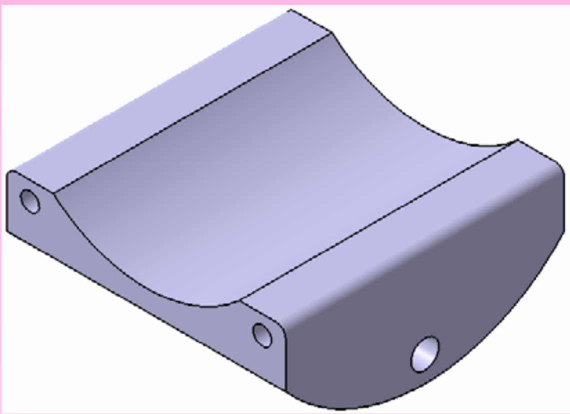


Figure A Model of the Bottom Seat for Tutorial 1

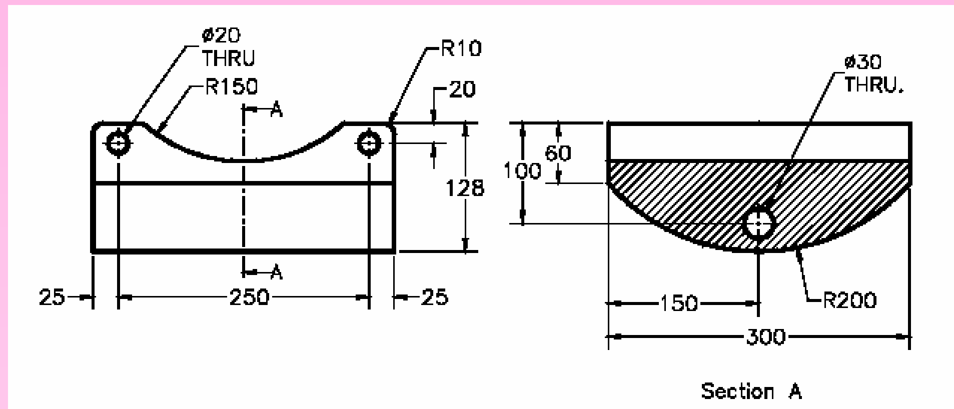


Figure B Views and dimensions of Tutorial 1

The model, after modifications, is as shown in **Figure C**.

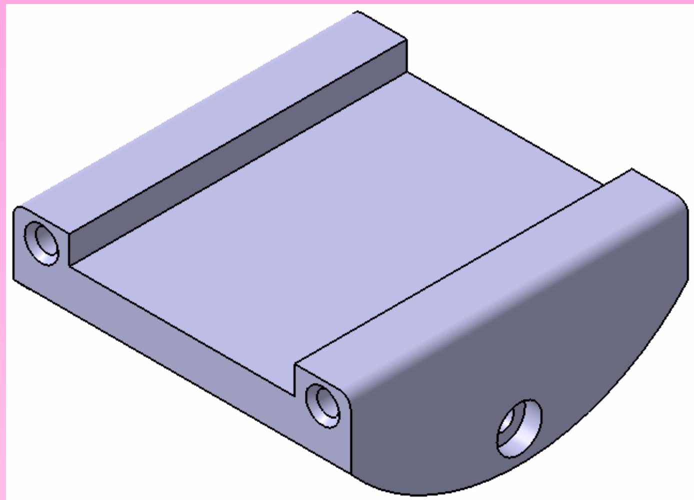


Figure C The model after making the modifications

1. Start a new file in the **Part** workbench of CATIA V5 and create the base feature of the model by extruding the sketch drawn on the zx plane, as shown in **Figure D** and **Figure E**.

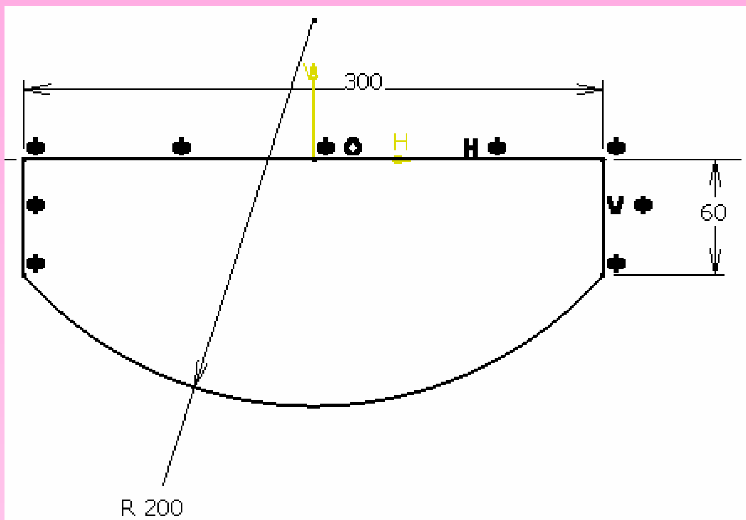


Figure D Sketch of the base feature

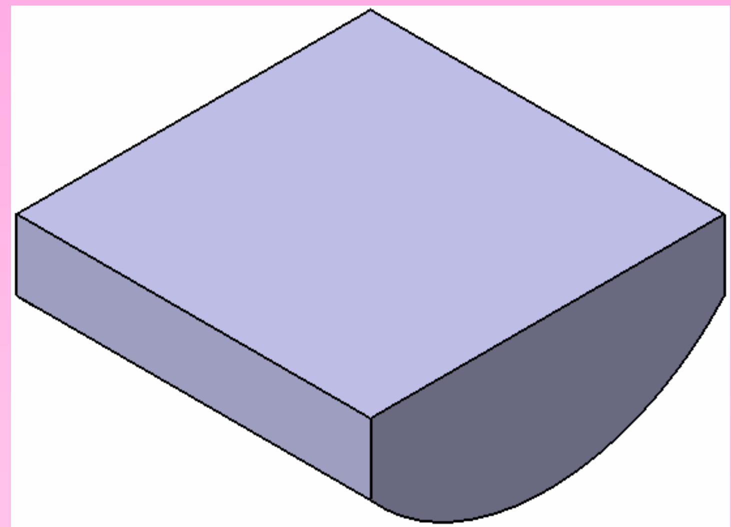


Figure E The model after creating the base feature

2. Create a pocket feature by extruding a sketch drawn on the front face of the base feature, as shown in **Figure F** and **Figure G**.

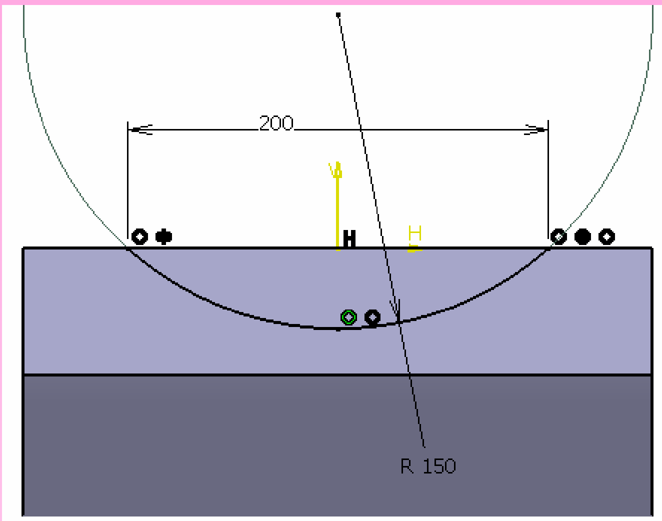


Figure F **Sketch of the second feature**

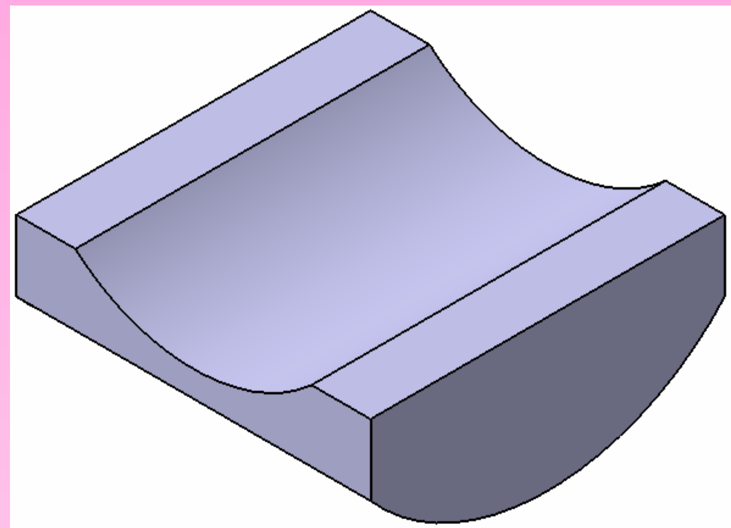


Figure G **The model after creating the second feature**

3. Fillet the right and left edges of the model, as shown in **Figure H**.
4. Create the hole features on the front and right faces of the model, as shown in **Figure H**.

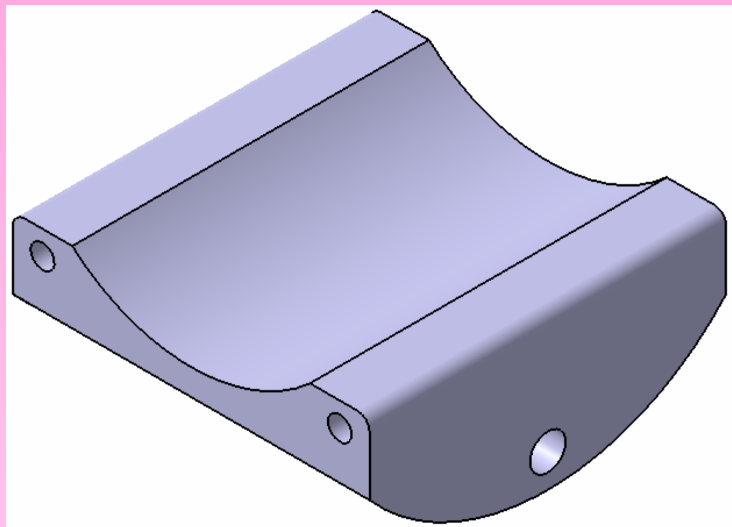


Figure H The model after creating all features

5. Edit the model, as shown in **Figure I** and **Figure J**.

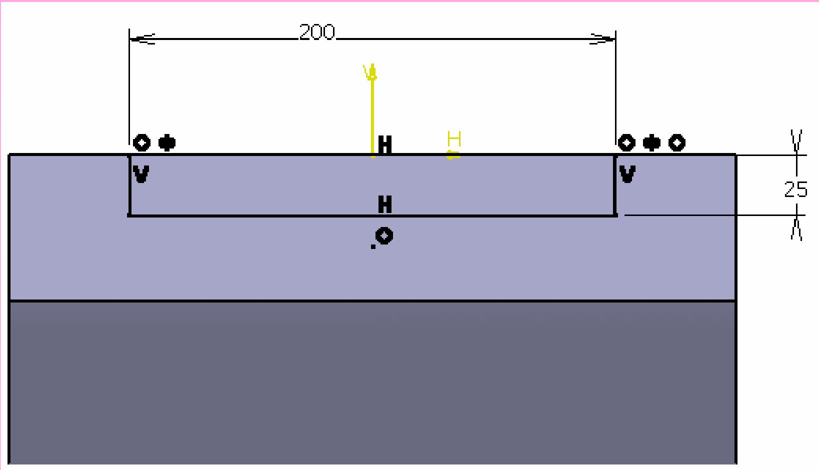


Figure I Sketch after modifications

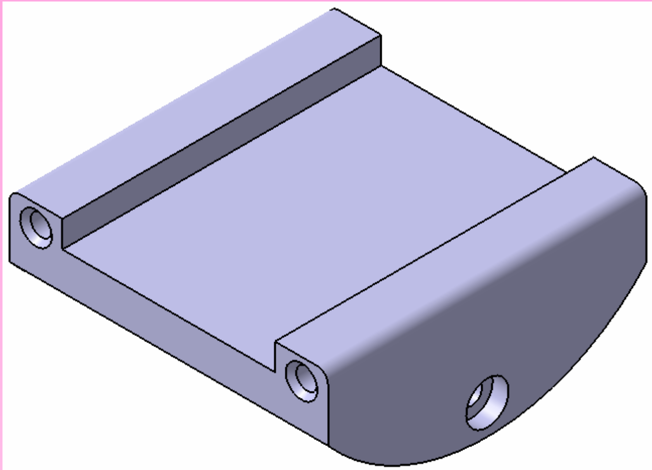


Figure J Model after modifications

6. Save the file with the file name **c06tut1.CATPart** within the c06 folder.

Tutorial 2

In this tutorial, you will create the model of the Vice Jaw shown in **Figure A**. Its views and dimensions are shown in **Figure B**. After creating this model, you will edit some of its dimensions. **Figure C** shows the views and the dimensions that need to be edited.

(Expected time: 30 min)

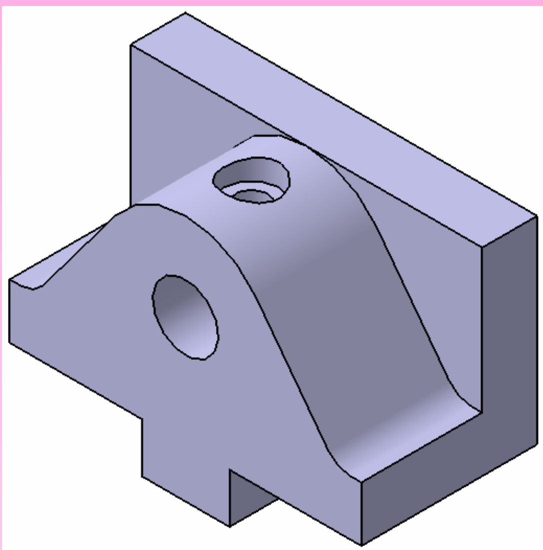


Figure A Model of the Vice Jaw for Tutorial 2

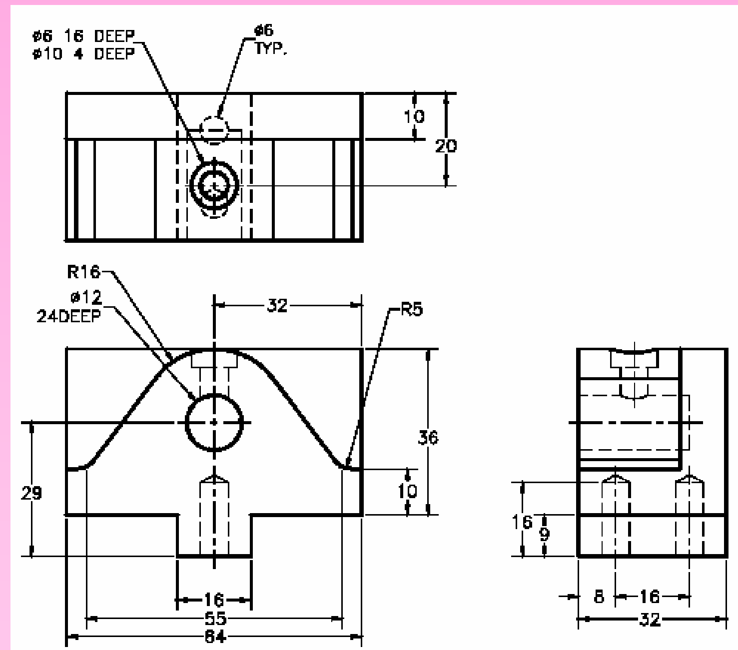


Figure B Views and dimensions of the Vice Jaw for Tutorial 2

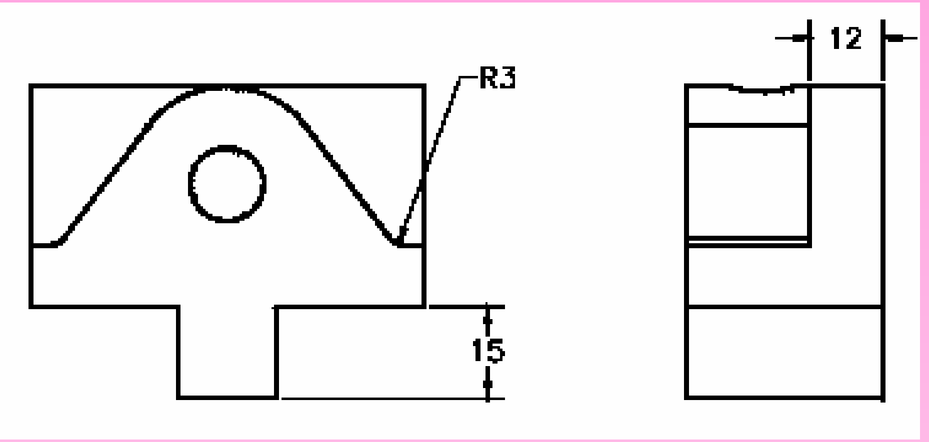


Figure C Dimensions of the Vice Jaw to be modified

1. Create the base feature of the model by extruding the sketch drawn on the zx plane, as shown in **Figure D** and **Figure E**.

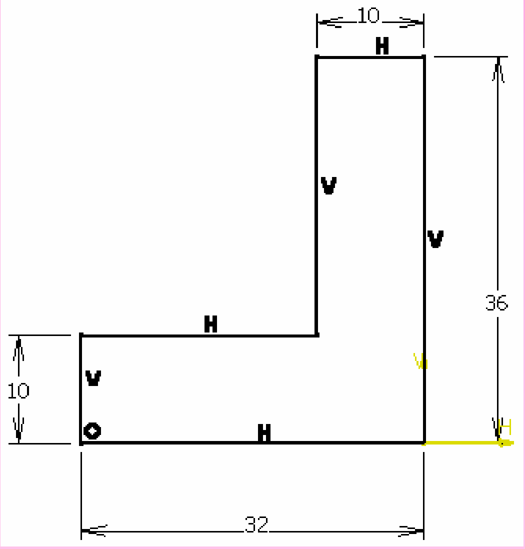


Figure D Sketch of the base feature

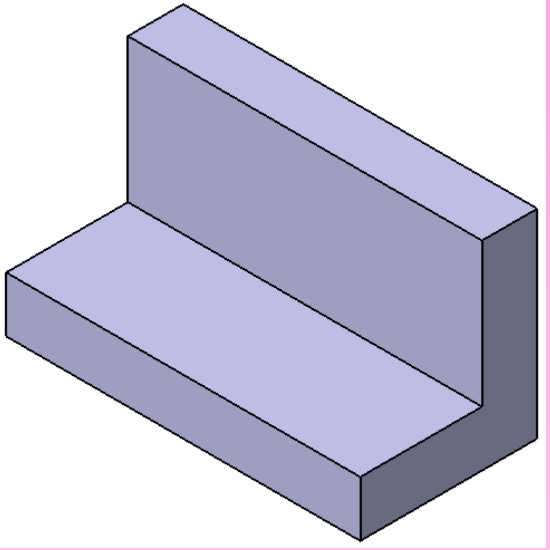


Figure E Model after creating the base feature

2. Create the second feature of the model by extruding a sketch drawn on the front face of the base feature, as shown in **Figure F** and **Figure G**.

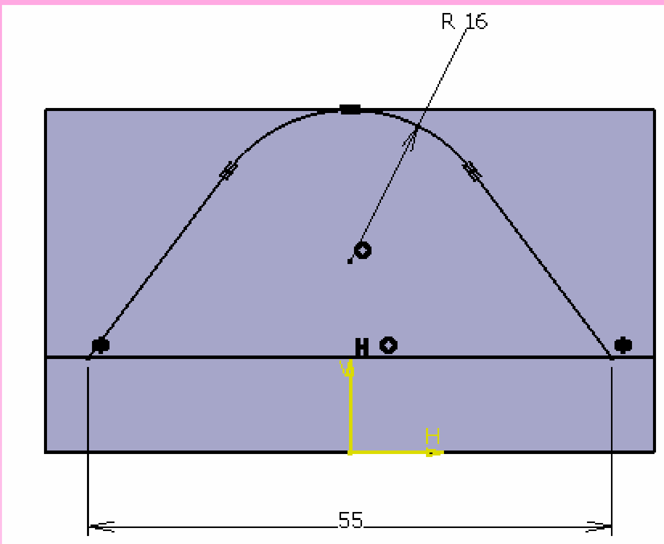


Figure F Sketch of the second feature

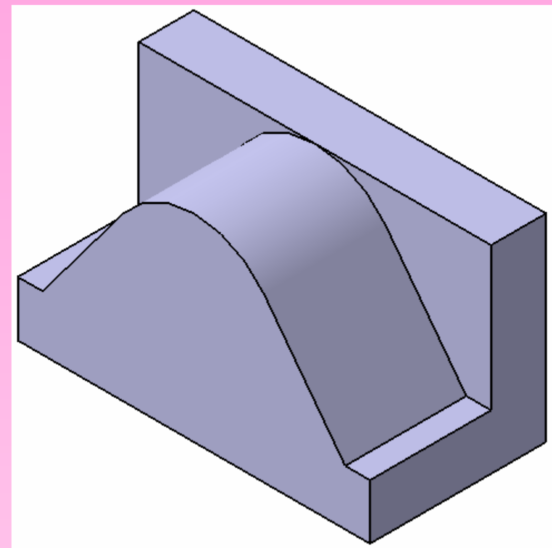


Figure G Model after creating the second feature

3. Create the third feature of the model by extruding a rectangular sketch drawn on the front face of the base feature, as shown in **Figure H** and **Figure I**.

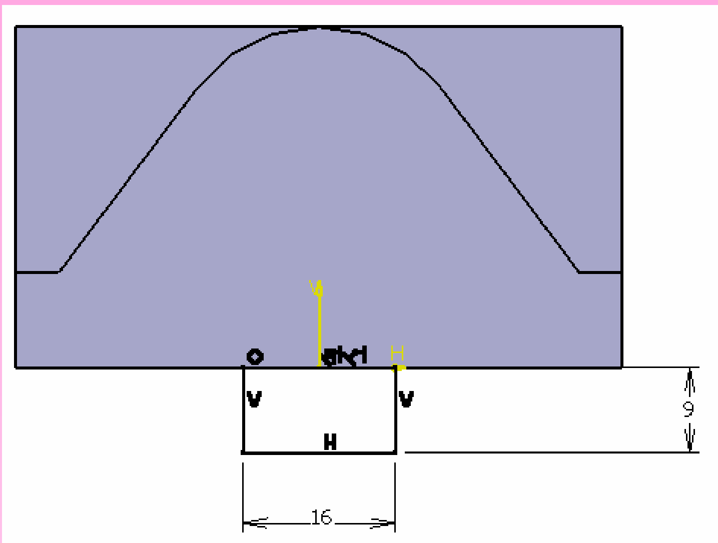


Figure H Sketch of the third feature

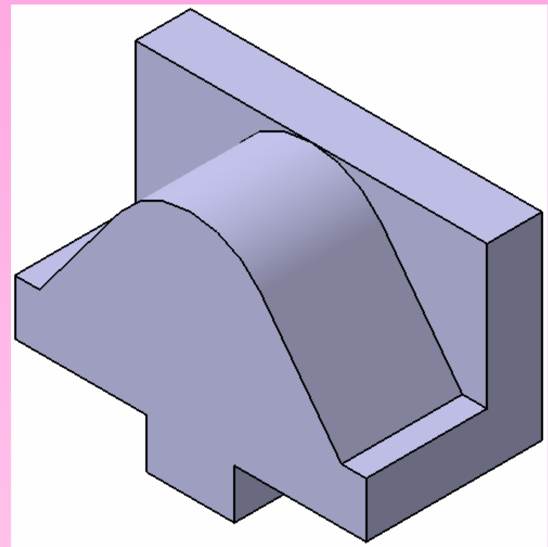


Figure I The model after creating the third feature

4. Create the holes using the **Hole** tool, as shown in **Figure J**.

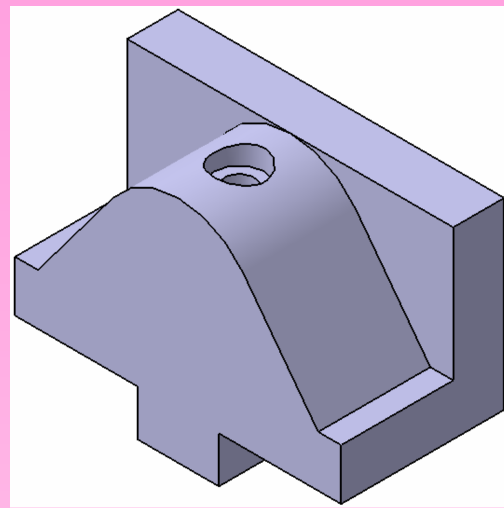


Figure J The model after creating the counterbore hole

5. Apply fillet to the edges of the model, as shown in **Figure K**.

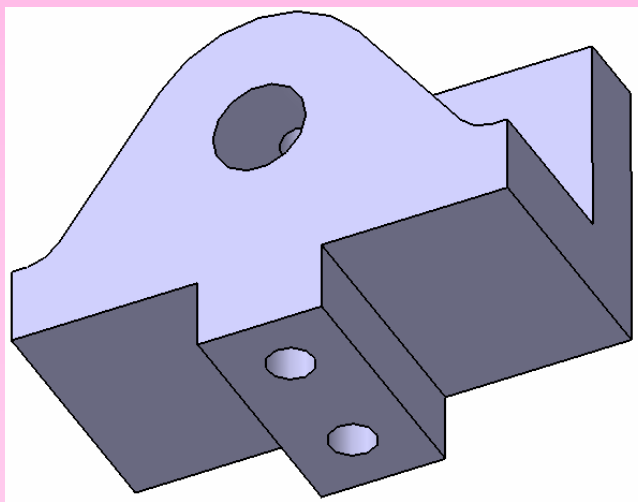


Figure K Final model after creating all the features

6. Modify the model, as shown in **Figure L**.

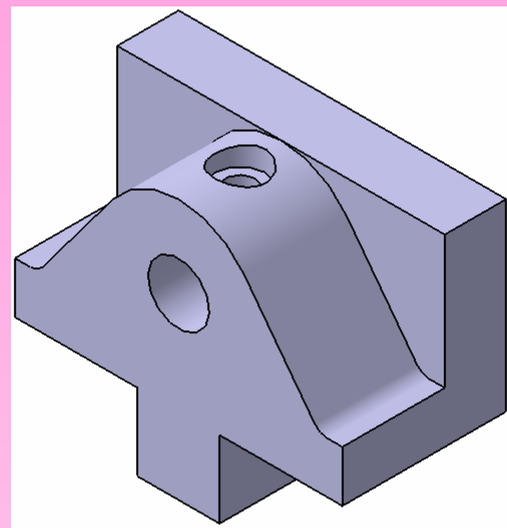


Figure L The model after modifying the design

7. Save the file with the file name **c06tut2.CATPart** within the c06 folder.

❑ Tutorial 3

In this tutorial, you will create the model shown in **Figure A**. Its views and dimensions are shown in **Figure B**. After creating this model, you will edit its design by replacing the counterbored holes with countersunk holes. Also, replace the rectangular slot by an elongated slot. **Figure C** shows the views and dimensions of the model, after editing. **(Expected time: 45 min)**

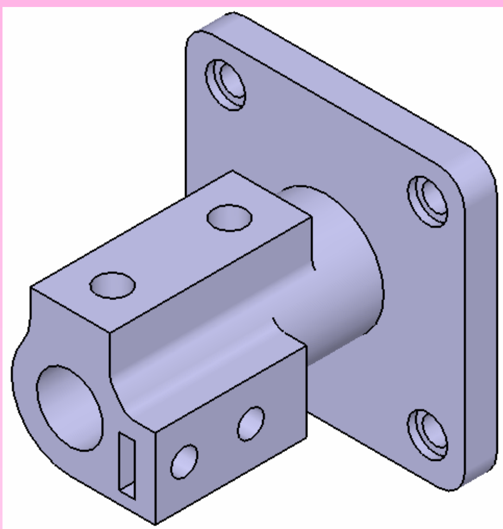


Figure A Model for Tutorial 3

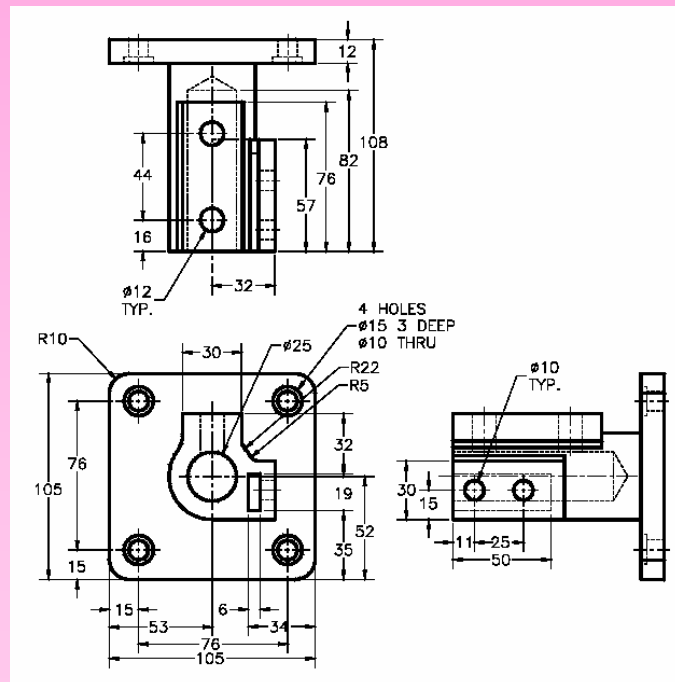


Figure B Views and dimensions for Tutorial 3

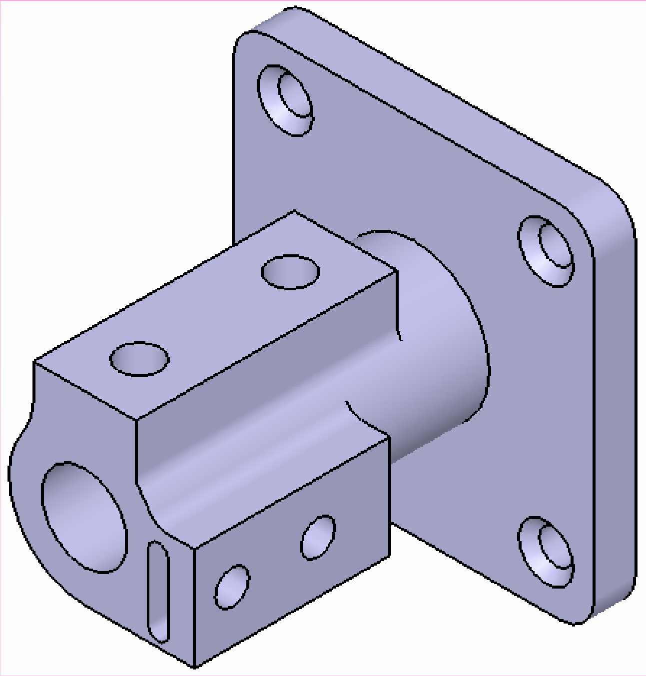


Figure C Final edited model

1. Start a new file in the **Part Design** workbench.
2. Create the base feature of the model by extruding the sketch drawn on the yz plane, as shown in **Figure D**.

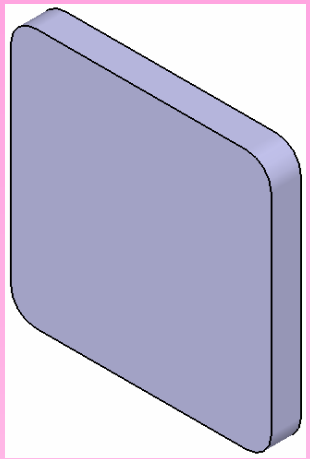


Figure D Base feature of the model

3. Create a pad feature by extruding a sketch drawn on the front face of the base feature, as shown in **Figure E**.

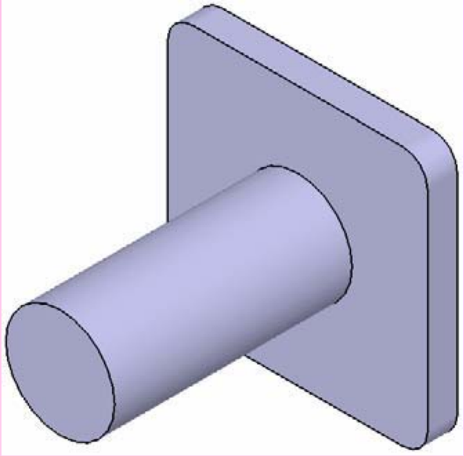


Figure E Second feature of the model

4. Create the third and forth features by extruding the sketch drawn on the front face of the second feature, as shown in **Figure F** and **Figure G**.

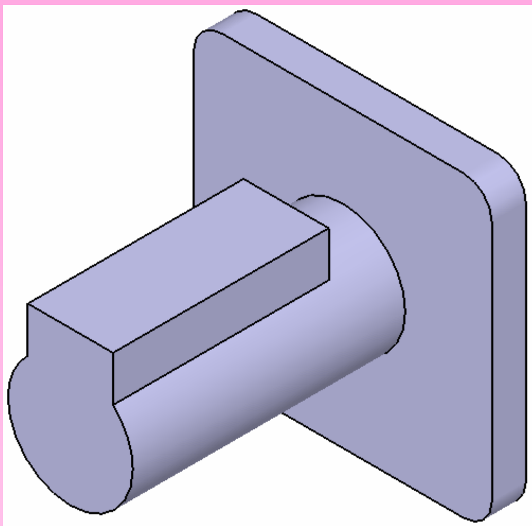


Figure F Third feature of the model

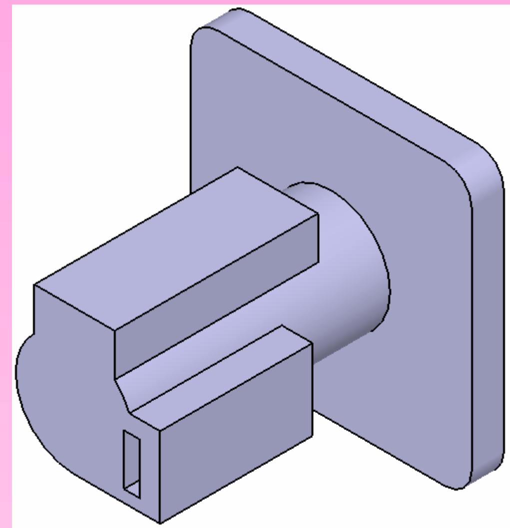


Figure G Model after creating the Pocket feature

5. Create the holes using the **Hole** tool and fillet the required edges, as shown in **Figure H**.

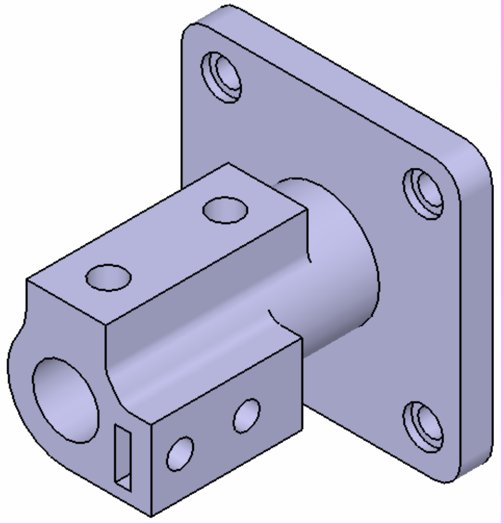


Figure H The final model after creating all the features

6. Edit the design of the model, as shown in **Figure I** and **Figure J**.

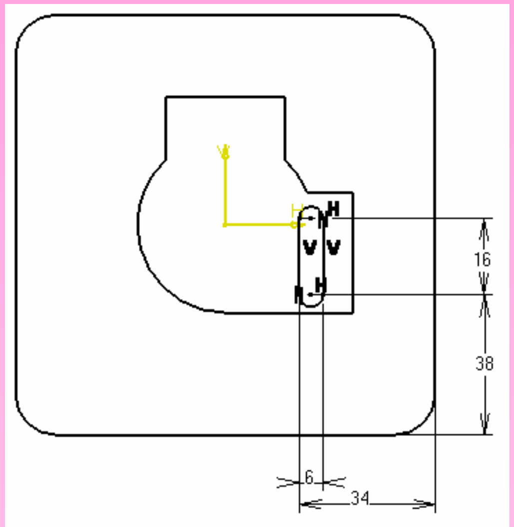


Figure I Edited sketch of the pocket feature

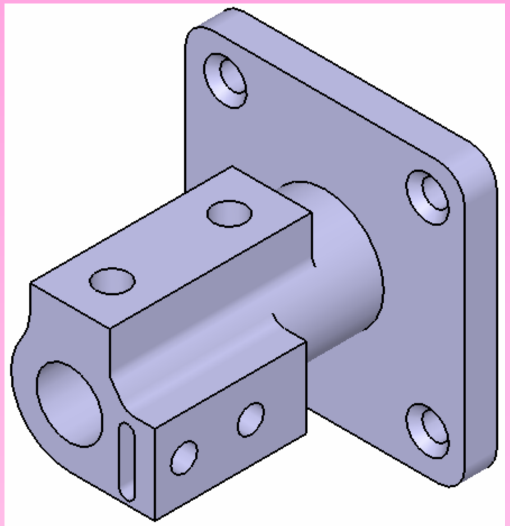


Figure J Final edited model

7. Save the file with the file name **c06tut3.CATPart** within the c06 folder.

❑ Exercise 1

Create the model shown in **Figure A**. Its view and dimensions are shown in **Figure B**.

(Expected time: 45 min)

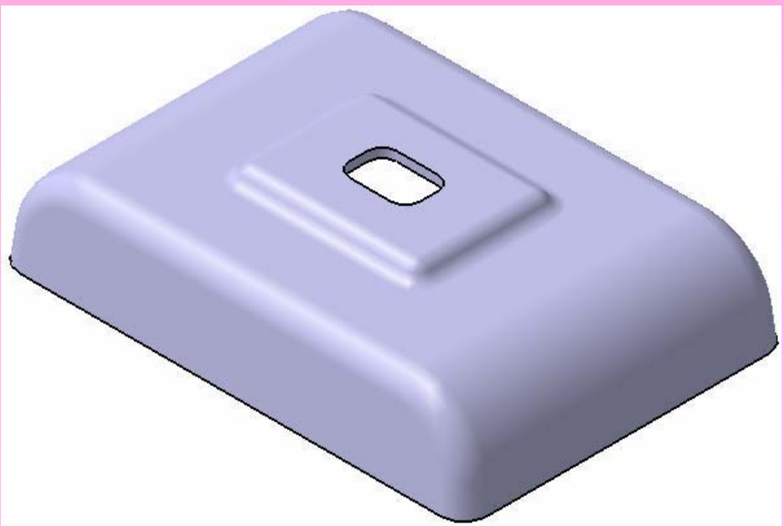


Figure A Model for Exercise 1

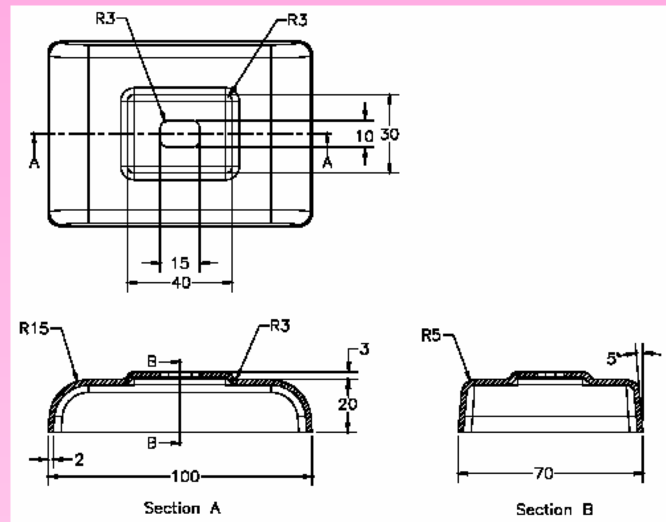


Figure B Views and dimensions for Exercise 1

Exercise 2

Create the model of the Plummer Block Casting shown in **Figure A**. The views and dimensions of the model are shown in **Figure B**. **(Expected time: 30 min)**

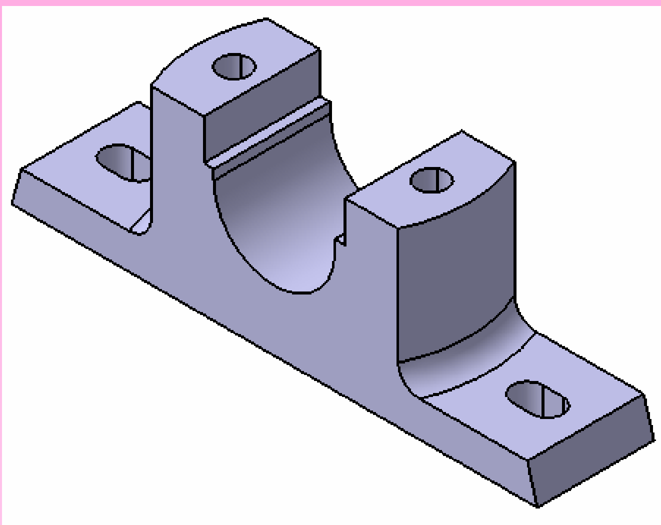


Figure A Model for Exercise 2

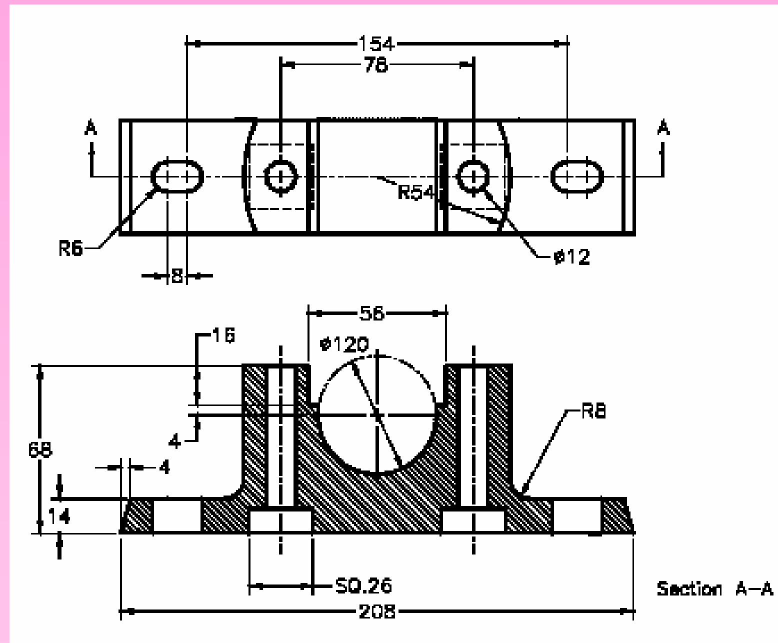


Figure B Views and dimensions for Exercise 2

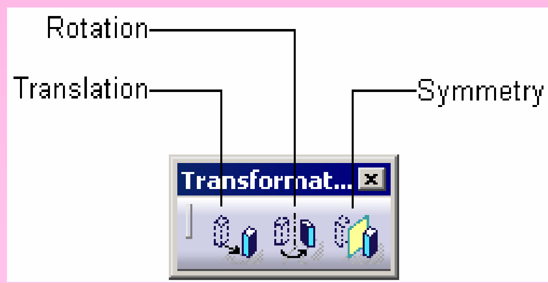
Learning Objectives:

- Translate bodies.
- Rotate bodies.
- Create symmetry features and bodies.
- Mirror features and bodies.
- Create rectangular patterns.
- Create circular patterns.
- Create user patterns.
- Scale models.
- Work with additional bodies.
- Add stiffeners to a model.

TRANSFORMATION FEATURES

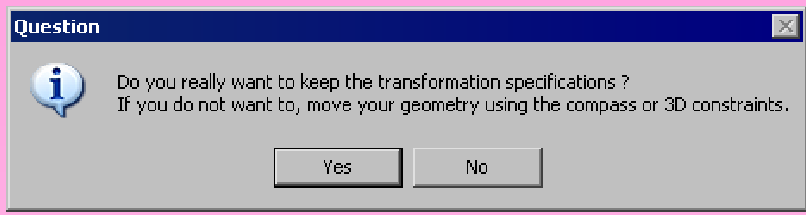
Translating Bodies

- To translate the current body, invoke the **Transformations** toolbar, by choosing the down arrow on the right of the **Translation** button from the **Transformation Features** toolbar.
- The **Transformations** toolbar is shown in the figure.

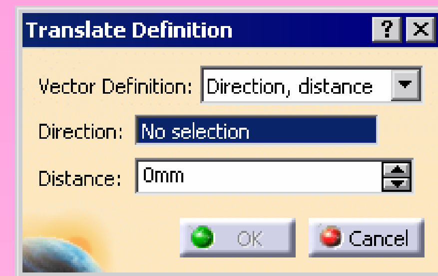


The Transformations toolbar

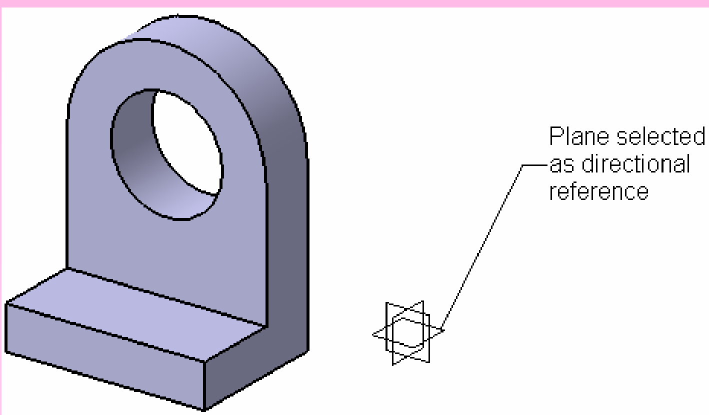
- Choose the **Translation** button from the **Transformations** toolbar. The **Question** dialog box is displayed, as shown in the figure.



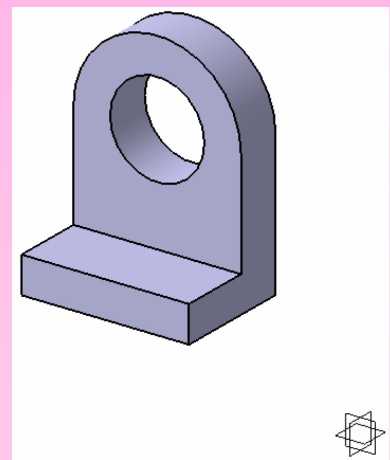
The Question dialog box



The Translate Definition dialog box

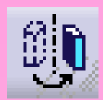


Plane selected as directional reference

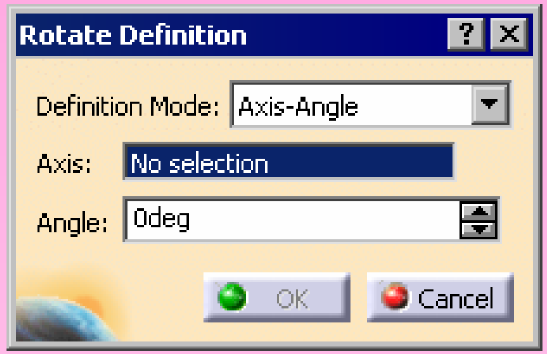


Resulting translated body

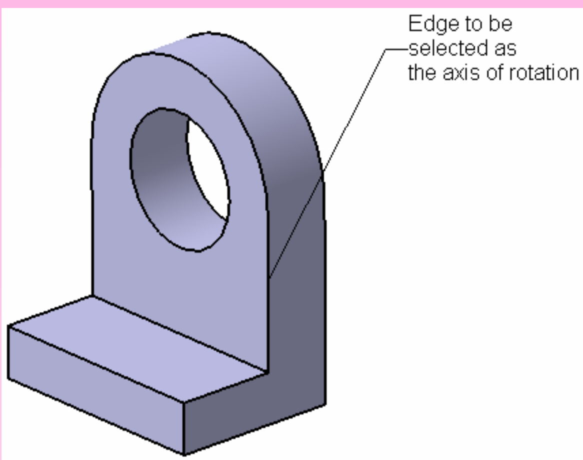
• Rotating Bodies



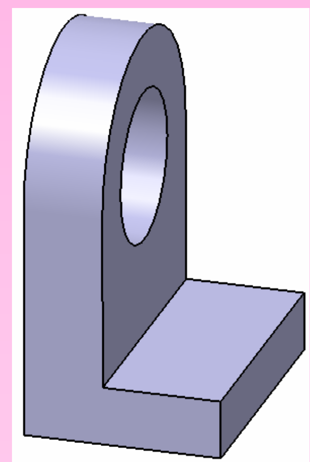
- Choose the **Rotation** button from the **Transformations** toolbar; the **Question** dialog box is displayed.
- Choose the **Yes** button from this dialog box; the **Rotate Definition** dialog box is displayed, as shown in the figure.



The Rotation Definition dialog box



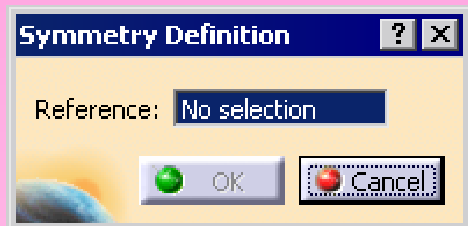
Edge to be selected as rotation axis



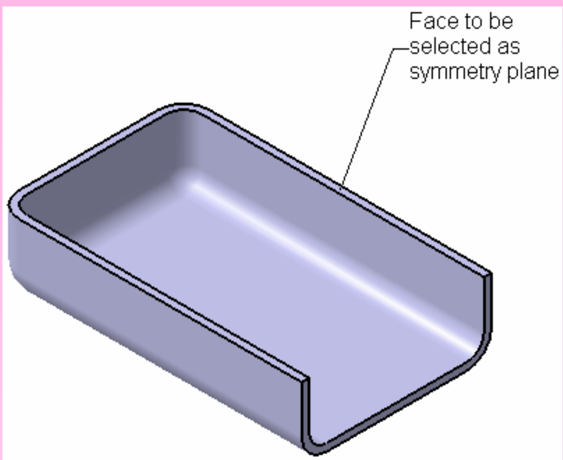
Resulting rotated body

• Creating Symmetry Features

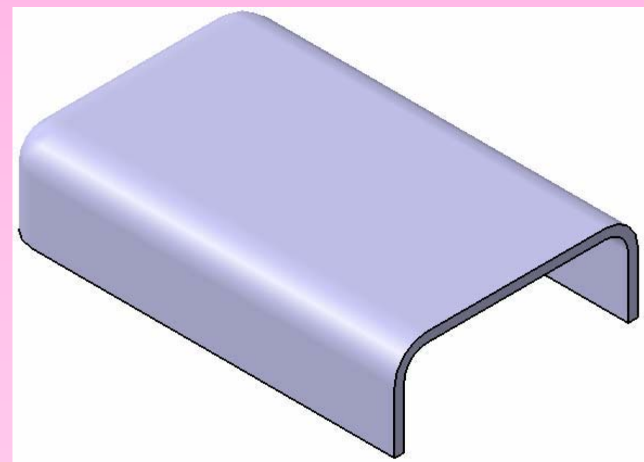
- To invoke this tool, choose the **Symmetry** button from the **Transformations** toolbar and then choose the **Yes** button from the **Question** dialog box.
- The **Symmetry Definition** dialog box is displayed, as shown in the figure.



The Symmetry Definition dialog box



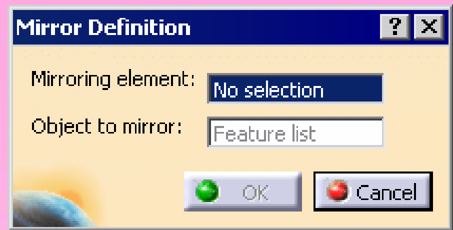
Symmetry reference to be selected



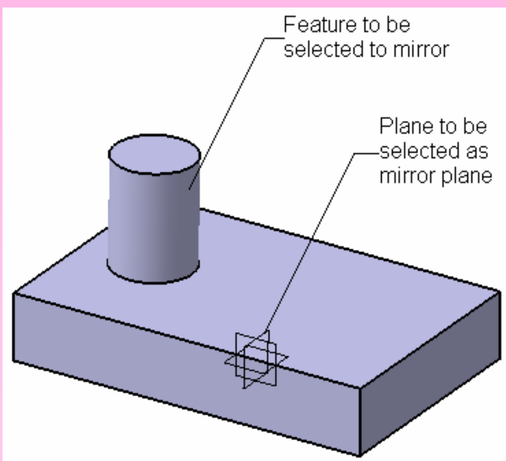
Resulting symmetry feature

• Mirroring Features and Bodies

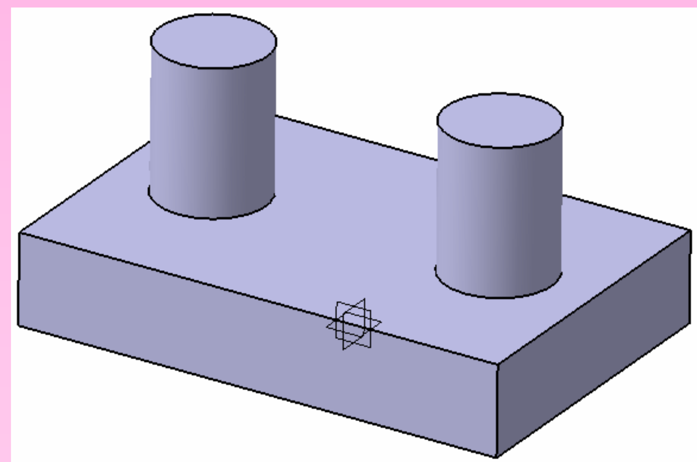
To mirror the selected feature, select it from geometry area or from the specification tree and choose the **Mirror** button from the **Transformation Features** toolbar; the **Mirror Definition** dialog box is displayed, as shown in the figure.



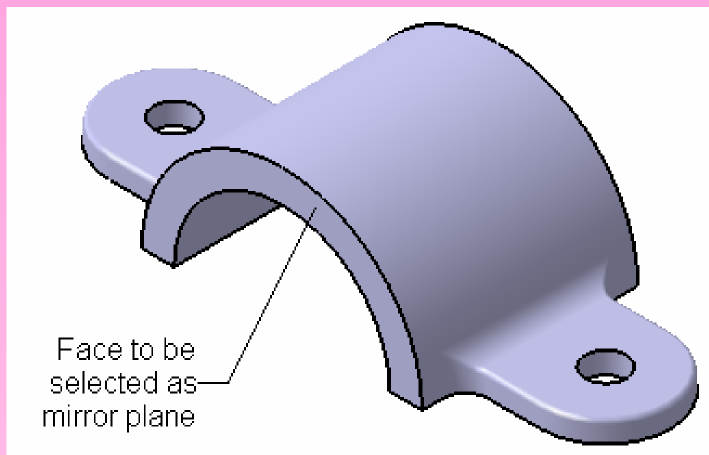
The Mirror Definition dialog box



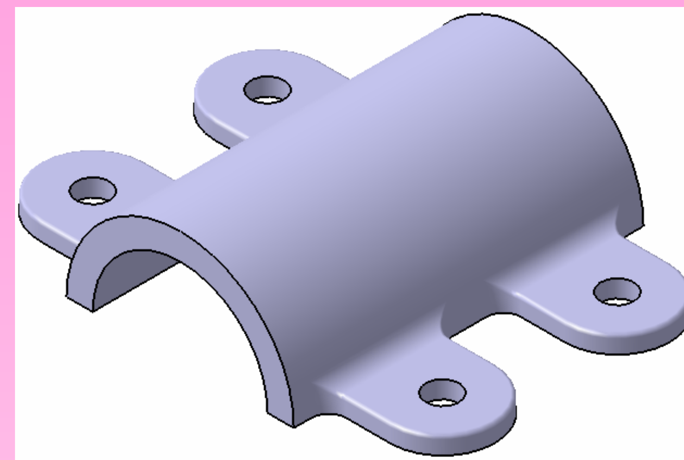
Feature and the mirror plane to be selected



Resulting mirrored feature

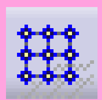


Face to be selected as mirror plane



Resulting mirrored model

• Creating Rectangular Patterns



- To create the rectangular pattern, choose the **Rectangular Pattern** button from the **Patterns** toolbar, as shown in **Figure A**.
- The **Rectangular Pattern Definition** dialog box is displayed, as shown in **Figure B**.

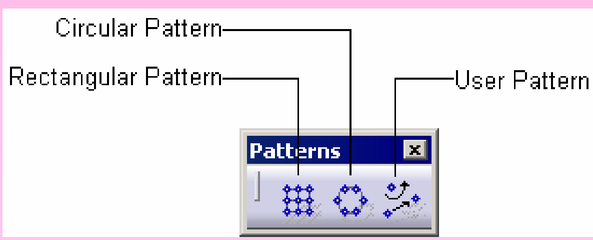


Figure A The Patterns toolbar

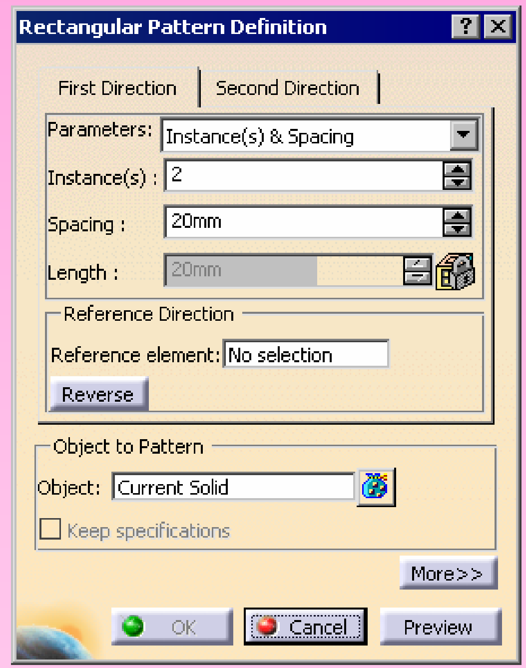
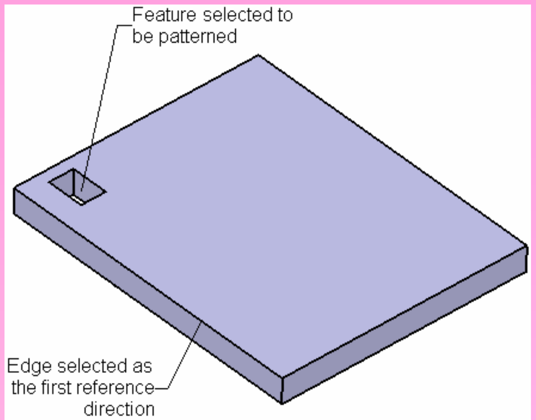
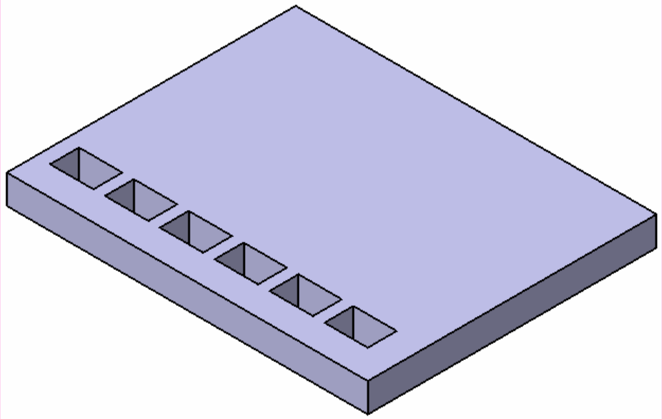


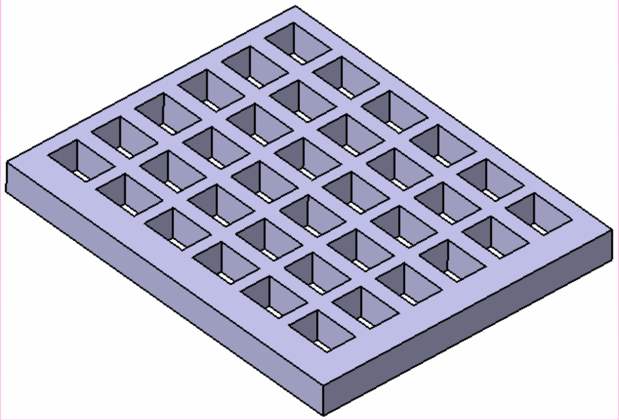
Figure B The Rectangular Pattern Definition dialog box



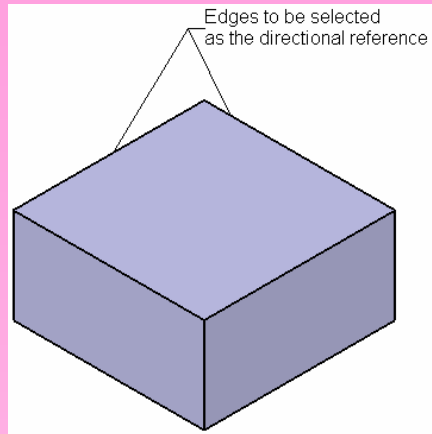
Feature and the edge to be selected



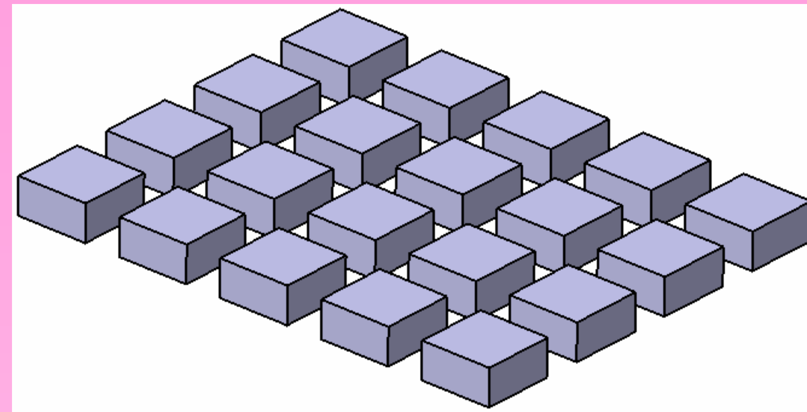
Resulting pattern



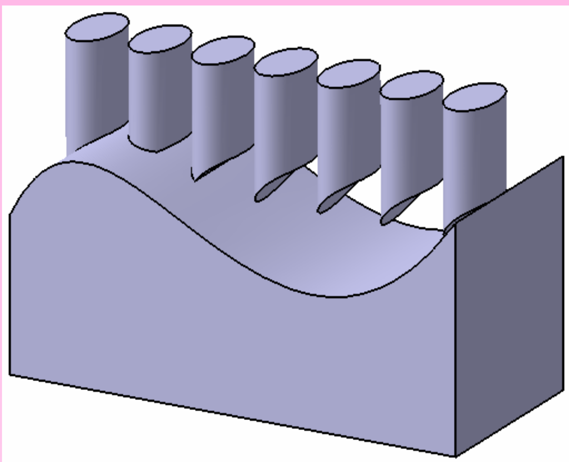
Rectangular pattern in two directions pattern



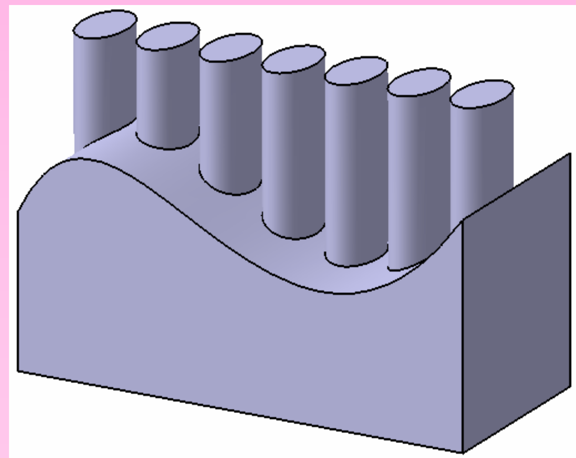
Directional references to be selected



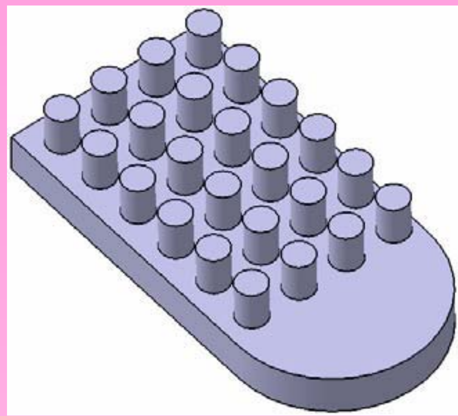
Resulting pattern of the PartBody



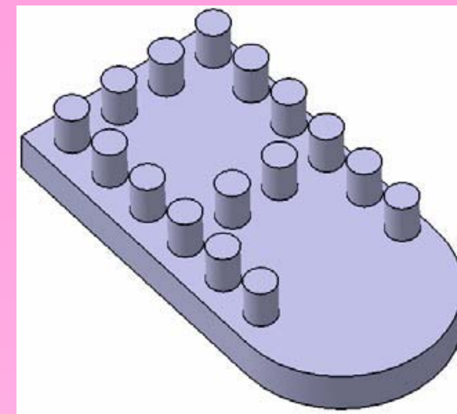
Pattern with the Keep specifications check box cleared



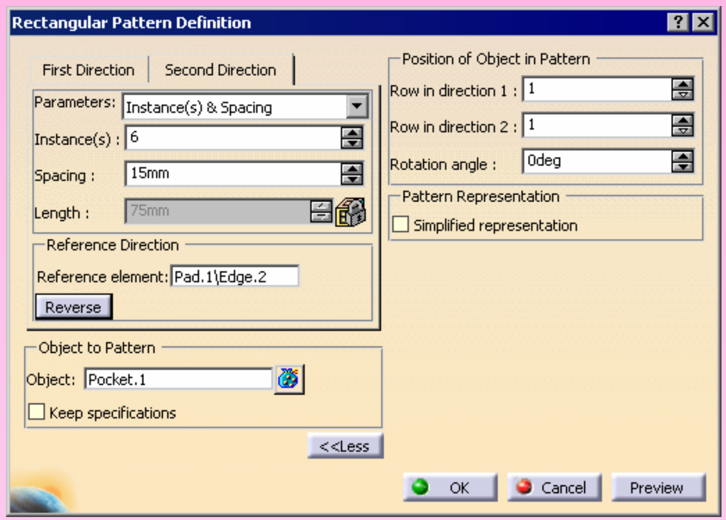
Pattern with the Keep specifications check box selected



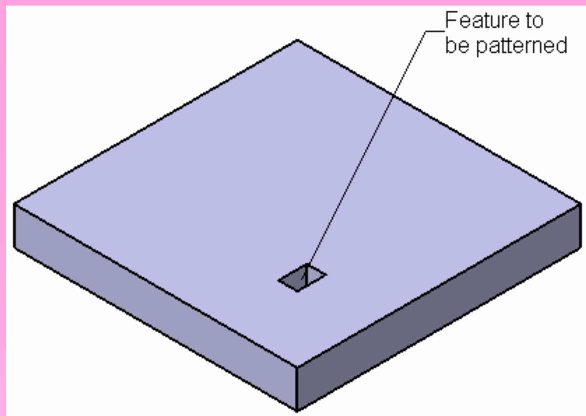
Patterned feature without skipping any instance



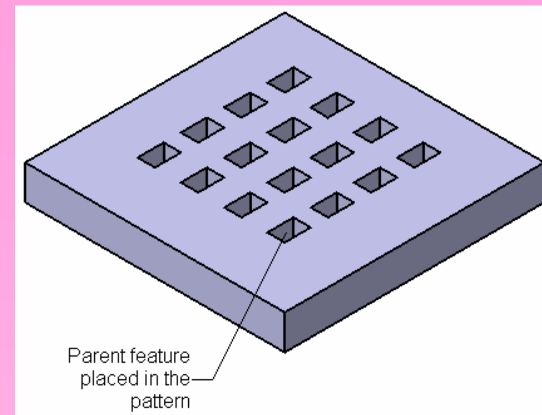
Patterned feature after skipping some of the instances



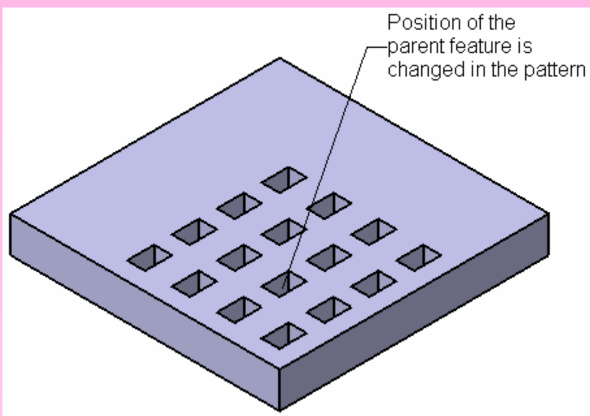
The expanded **Rectangular Pattern Definition** dialog box



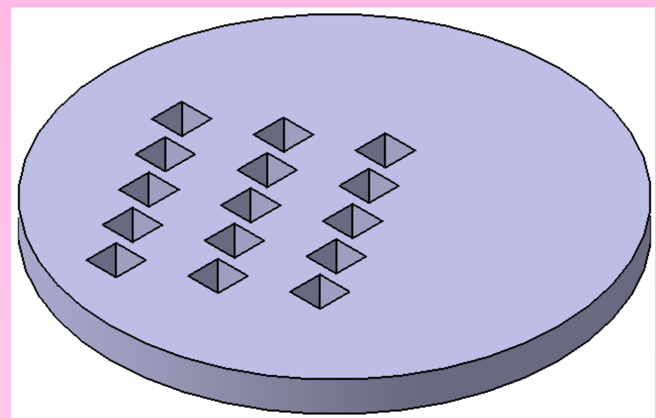
Feature to be pattern



Position of the parent feature in the pattern

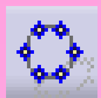


New position of the parent feature

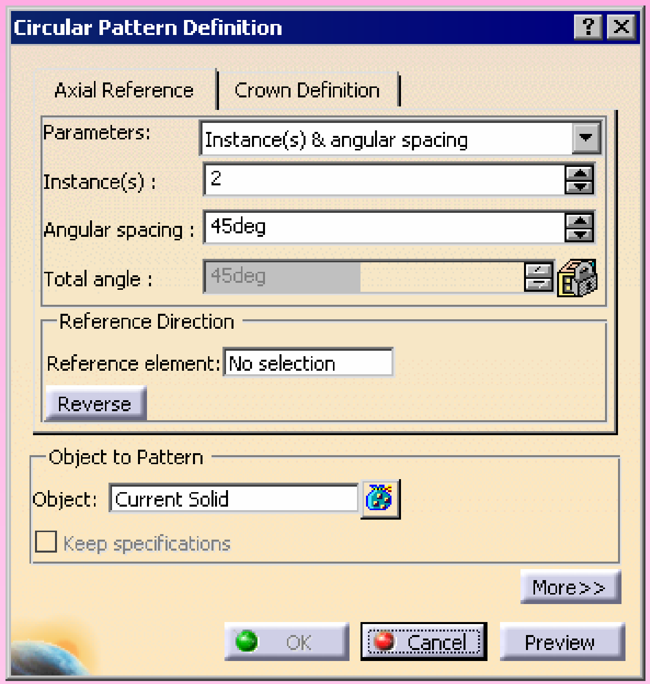


Rotated pattern

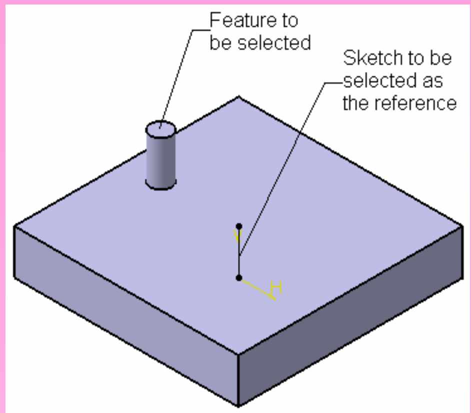
• Creating Circular Patterns



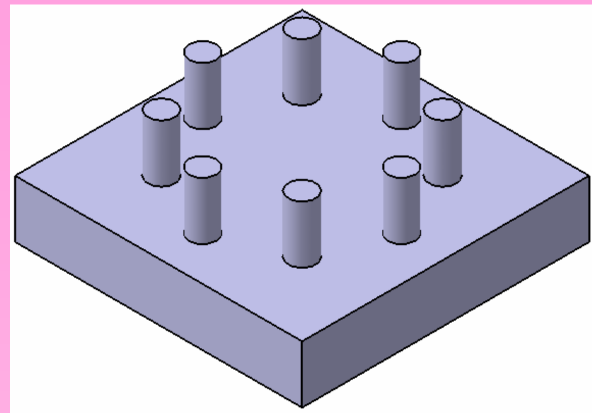
- To arrange features or the current body in a circular manner, choose the **Circular Pattern** button from the **Patterns** toolbar.
- The **Circular Pattern Definition** dialog box is displayed, as shown in the figure.



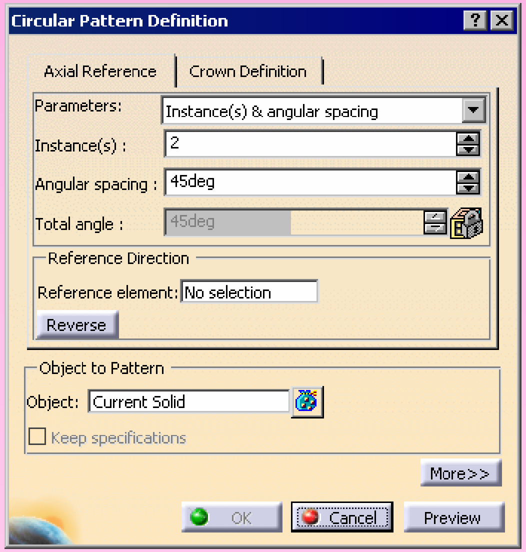
The Circular Pattern Definition dialog box



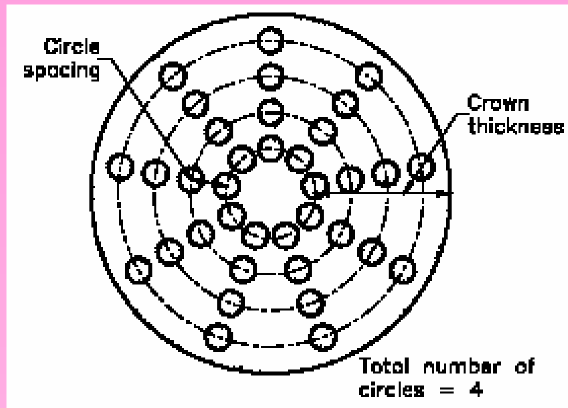
Feature to be patterned and the reference element to be selected



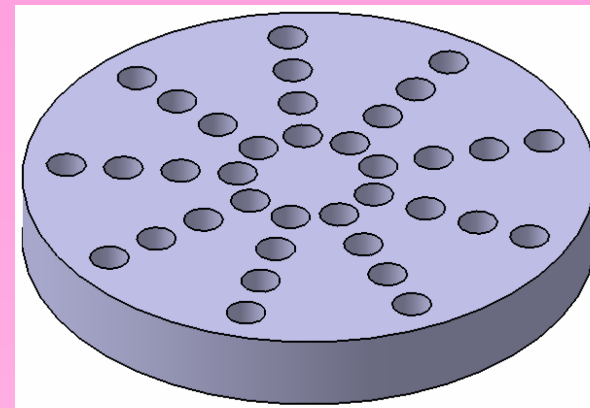
Resulting patterned feature



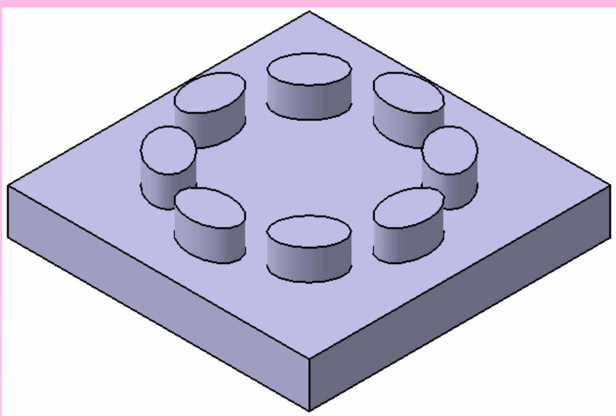
The **Circular Pattern Definition** dialog box after invoking the **Crown Definition** tab



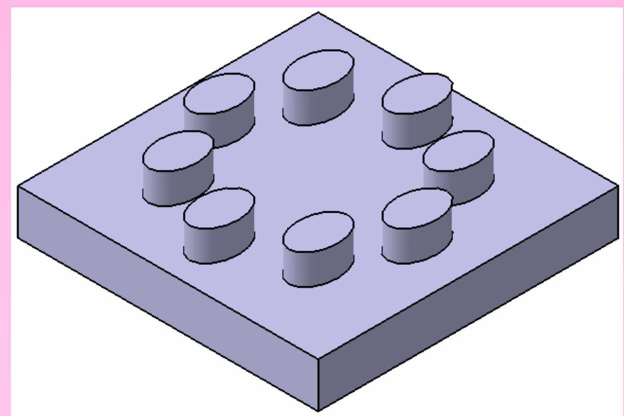
Various parameters used to define the crown in the pattern



Resulting pattern after defining the crown parameters

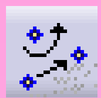


Pattern with the Radial alignment of instance(s) check box selected

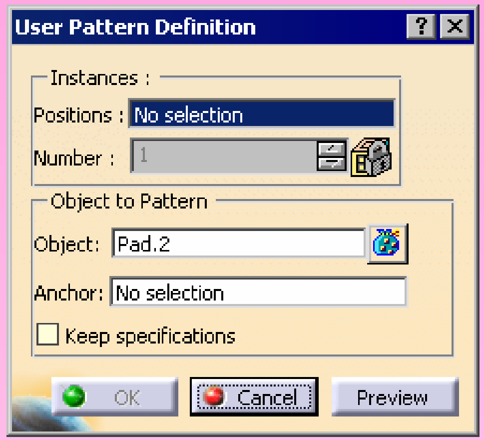


Pattern with the Radial alignment of instance(s) check box cleared

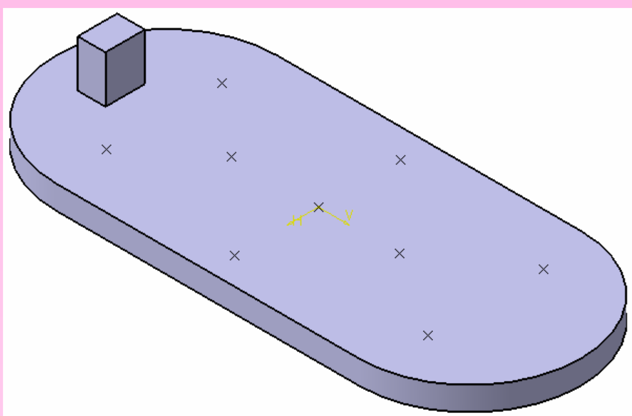
• Creating User Patterns



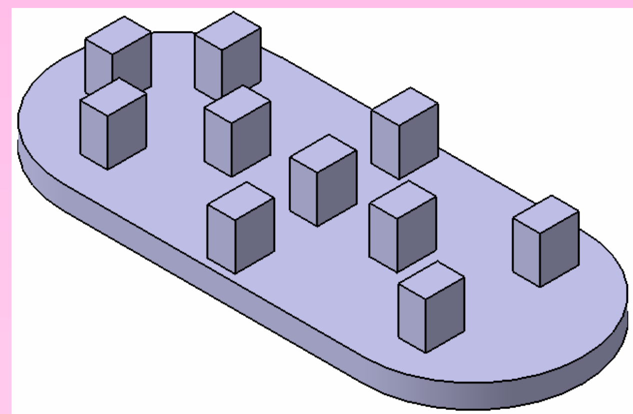
- To create a user pattern, select the feature to be patterned and then choose the **User Pattern** button from the **Patterns** toolbar.
- The **User Pattern Definition** dialog box is displayed, as shown in the figure.



The User Pattern Definition dialog box

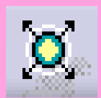


Features and the reference sketch

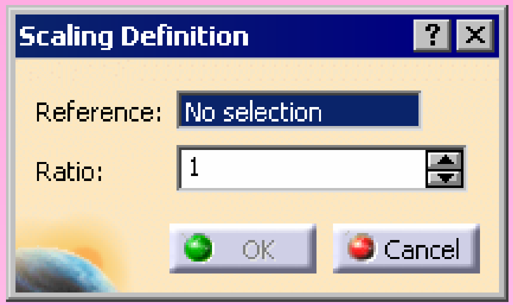


Resulting pattern

• Scaling Models



- To scale a model, choose the **Scaling** button from the **Transformation Features** toolbar.
- The **Scaling Definition** dialog box is displayed, as shown in the figure.

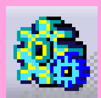


The Scaling Definition dialog box

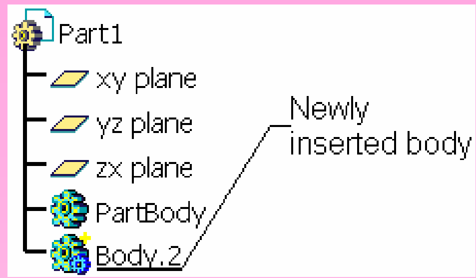
WORKING WITH ADDITIONAL BODIES

- The **Part** workbench of CATIA V5 provides you with a tool to insert new bodies in the current model.
- You can create features in the newly created body and then perform the boolean operations on two or more than two part bodies.

• Inserting a New Body

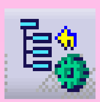


- To insert a new body, choose the Body button from the **Insert** toolbar.
- A new body named **Body.2** is added under the current body in the specification tree, as shown in the figure.

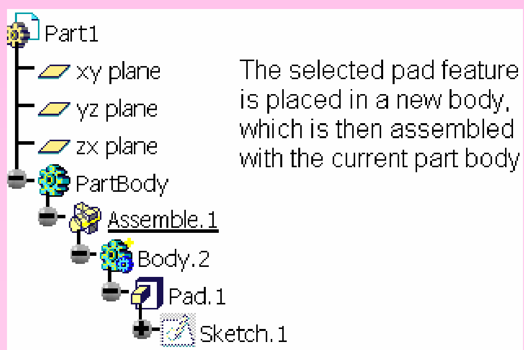


Newly inserted body

• Inserting Features in the Assemble Feature



To assemble the feature, select it and then choose the **Assemble Feature** button from the **Insert** toolbar.



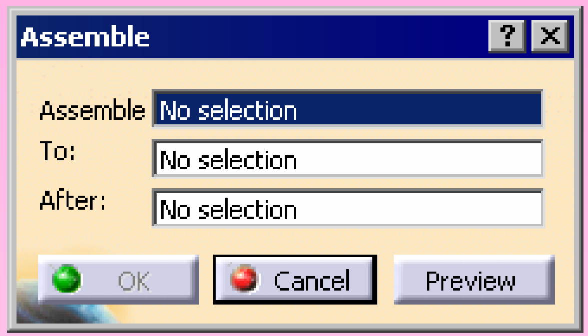
The specification tree

- **Applying Boolean Operations to Bodies**

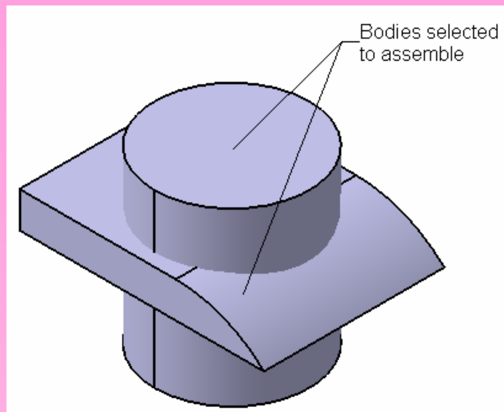
After inserting the bodies, you can apply the boolean operations, such as addition of two bodies, subtraction of one body from the other, retaining the intersection portion of two bodies, and so on.

Assembling Bodies

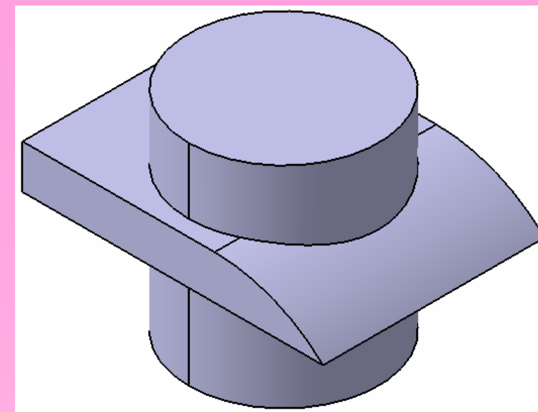
- To assemble the two bodies together, choose the **Assemble** button from the **Boolean Operations** toolbar.
- The **Assemble** dialog box is displayed, as shown in the figure.



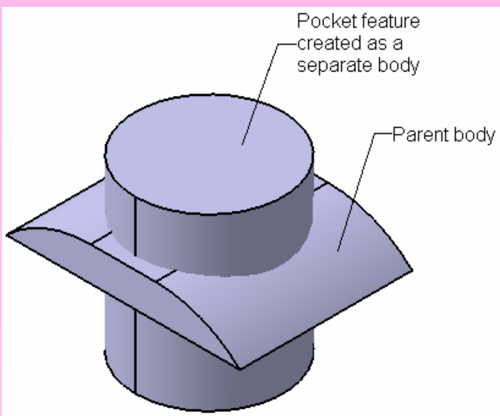
The Assemble dialog box



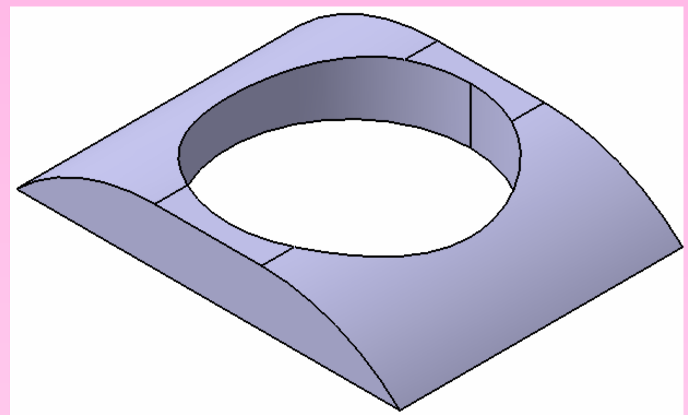
Bodies to be assembled



Resulting assembled body



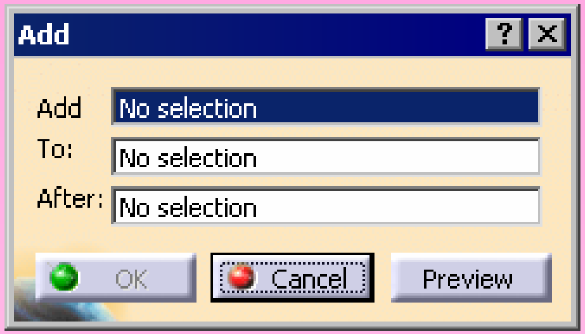
Pocket body and the parent body



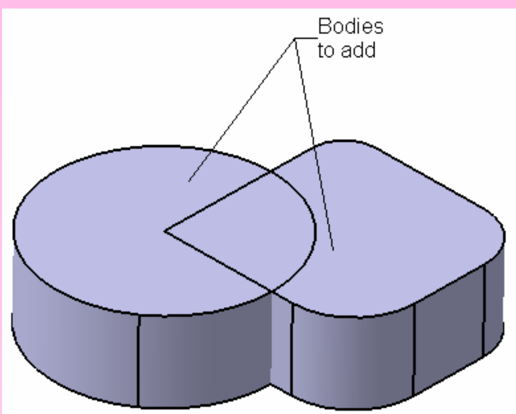
Resulting assembled body

• Adding Bodies

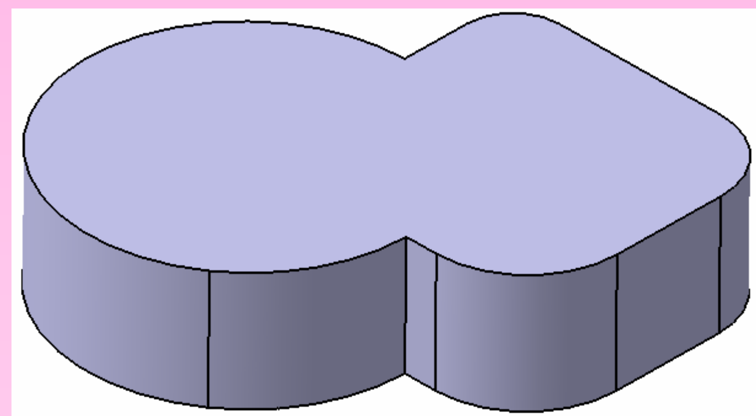
- The **Add** tool is used to add the selected bodies together.
- When you invoke this tool, the **Add** dialog box is displayed, as shown in the figure.



The Add dialog box

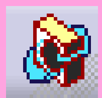


Bodies to be added

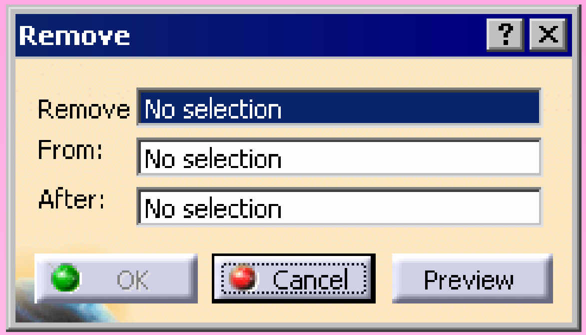


Resulting body

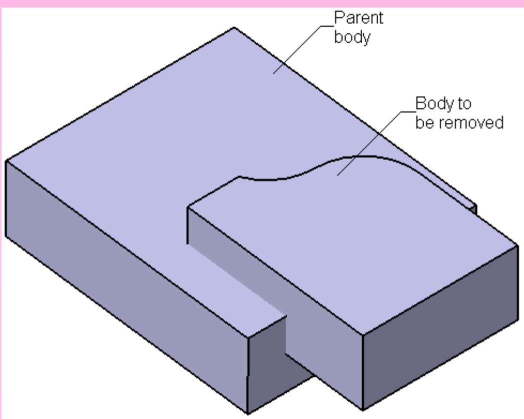
- **Subtracting Bodies**



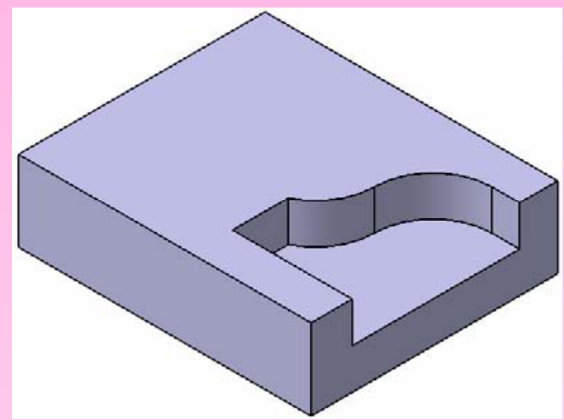
- The **Remove** tool is used to subtract the selected body from another body.
- When you invoke this tool, the **Remove** dialog box is displayed, as shown in the figure.



The Remove dialog box



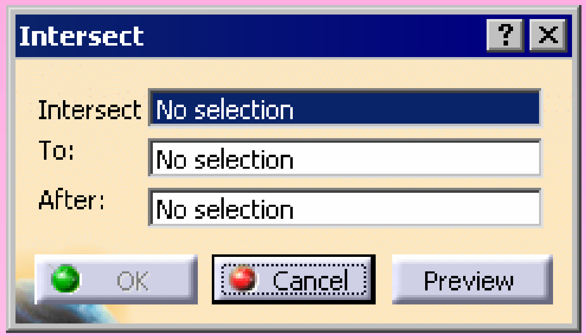
Body to be removed and the parent body



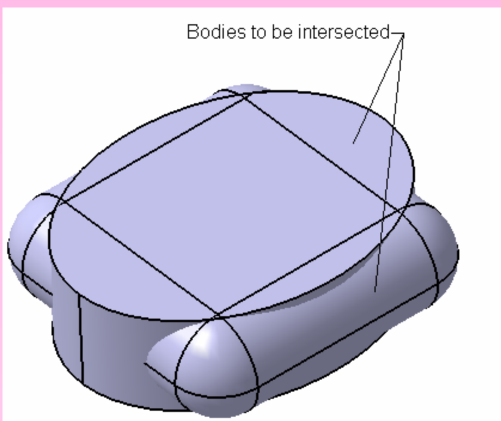
Resulting body

Intersecting Bodies

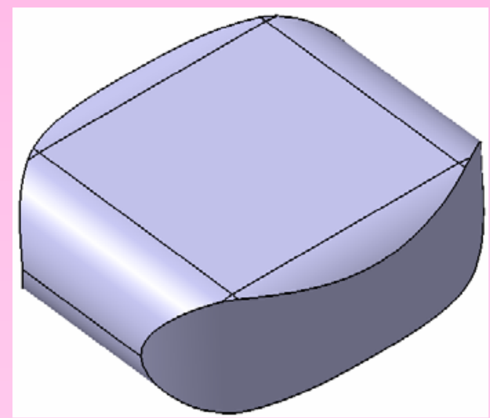
- The **Intersect** tool is used to retain the common portion of two intersecting bodies and remove the other portions of the selected bodies.
- When you invoke this tool, the **Intersect** dialog box is displayed, as shown in the figure.



The Intersect dialog box

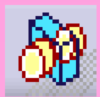


Body to be selected

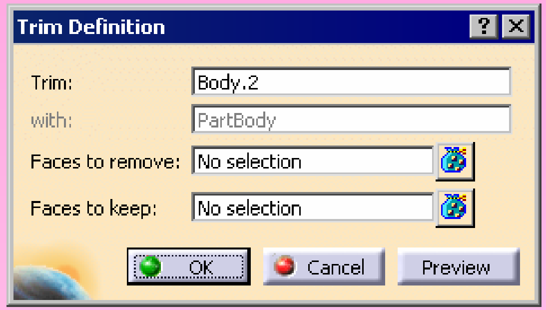


Resulting body

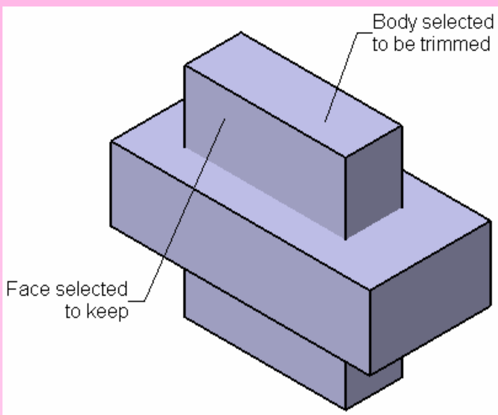
• Trimming Bodies



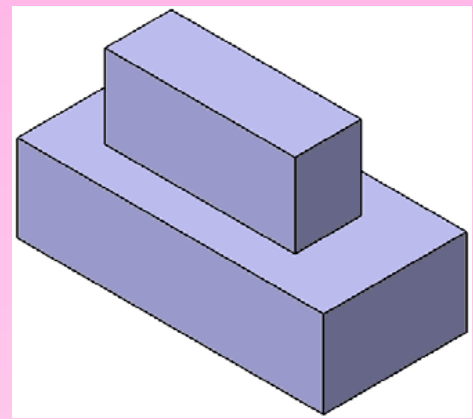
- On invoking this tool, you will be prompted to select the body to trim, therefore do so.
- Note that the parent body can not be trimmed.
- The **Trim Definition** dialog box is displayed, as shown in the figure.



The Trim Definition dialog box

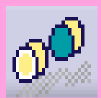


Body to be trimmed and the face selected to specify the portion of the body to be retained

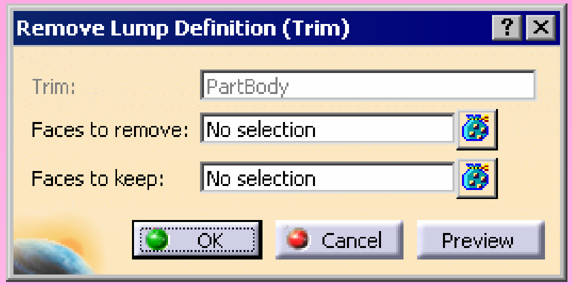


Resulting trimmed body

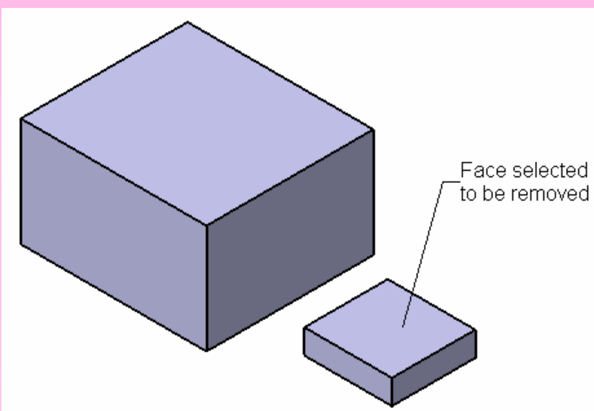
• Removing Lumps



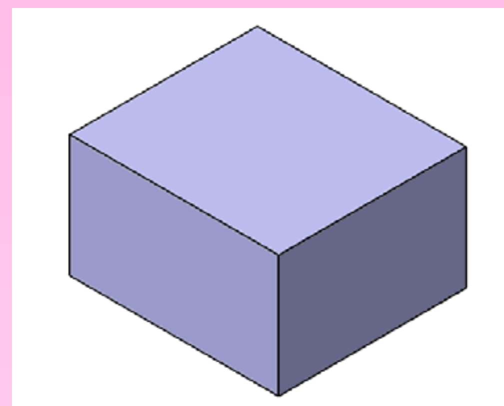
- When you invoke this tool, you are prompted to select the body to be trimmed.
- Select the body from the geometry area; the **Remove Lump Definition (Trim)** dialog box is displayed, as shown in the figure.



The Remove Lump Definition (Trim) dialog box

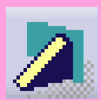


Face selected to be removed

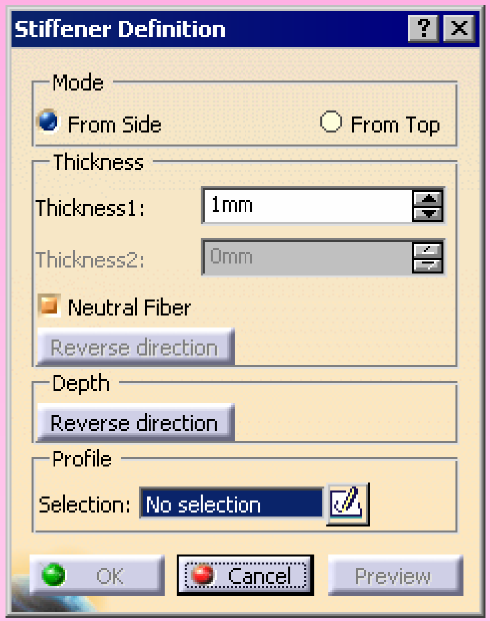


Resulting body after removing the lump

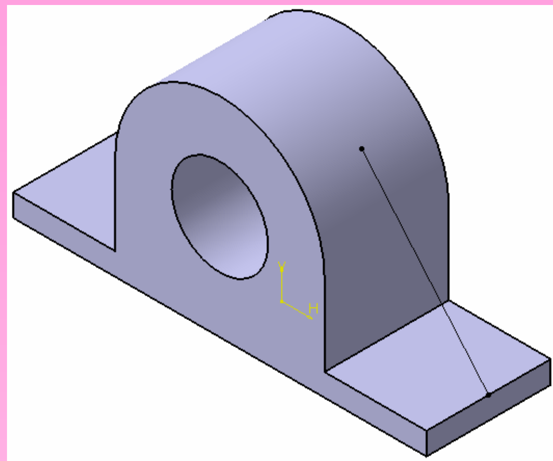
ADDING STIFFENERS TO THE MODEL



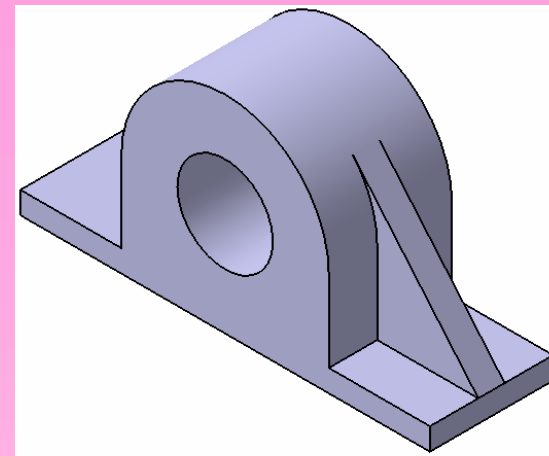
- Stiffeners are generally added to increase the strength of the components under the increased loading conditions.
- To add a stiffener, choose the **Stiffener** button from the **Advanced Extruded Features** toolbar.
- The **Stiffener Definition** dialog box is displayed, as shown in the figure.



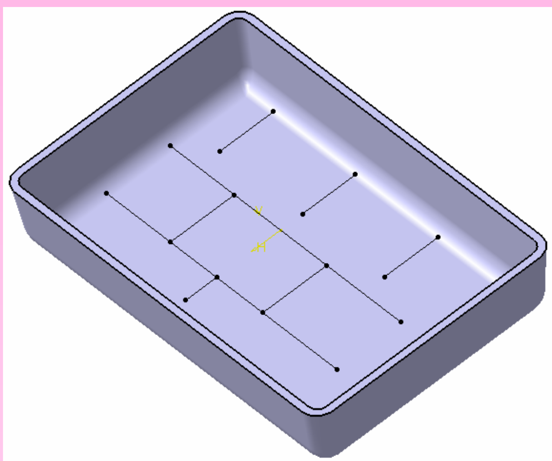
The Stiffener Definition dialog box



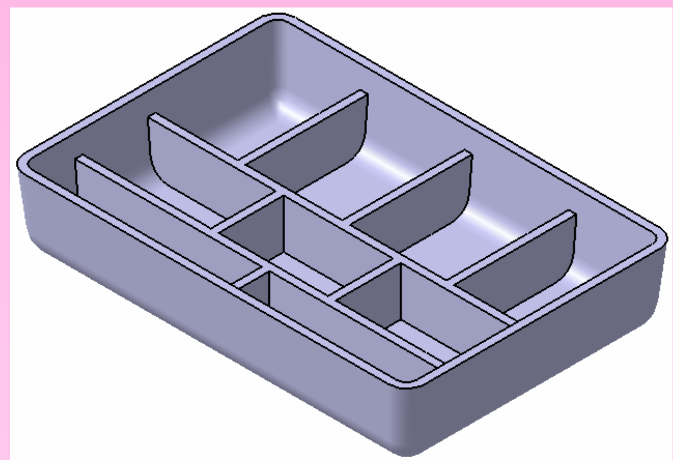
Sketch for the stiffener



Resulting stiffener



Sketch for the stiffener



Resulting stiffener

❑ Tutorial 1

In this tutorial, you will create the model of the Soap Case shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

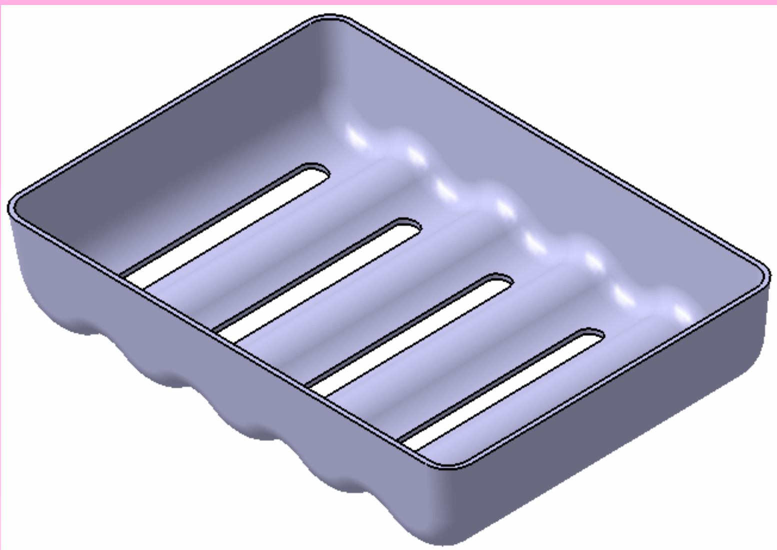


Figure A Model of the Soap Case for Tutorial 1

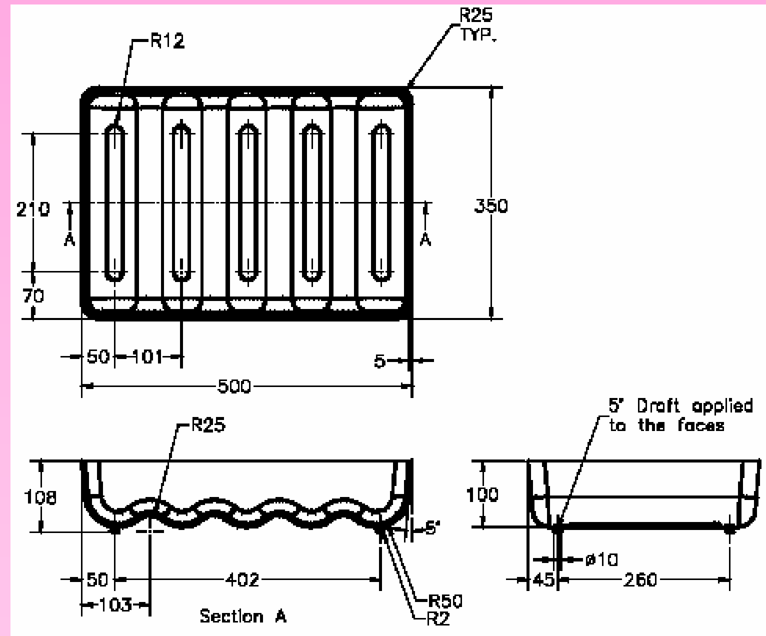


Figure B Views and dimensions of the Soap Case for Tutorial 1

1. Create the base feature of the model, as shown in **Figure C**.

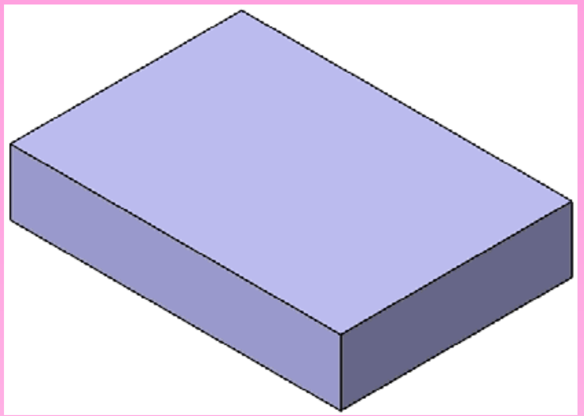


Figure C Base feature of the model

2. Apply draft to faces of the base feature, as shown in **Figure D**.

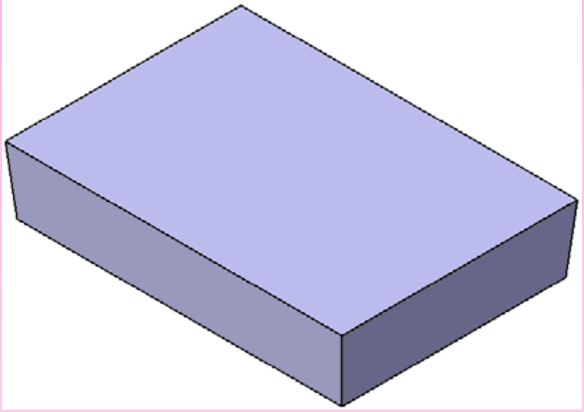


Figure D The model after creating the draft feature

3. Create a pocket feature to shape the lower portion of the model, as shown in **Figure E**.

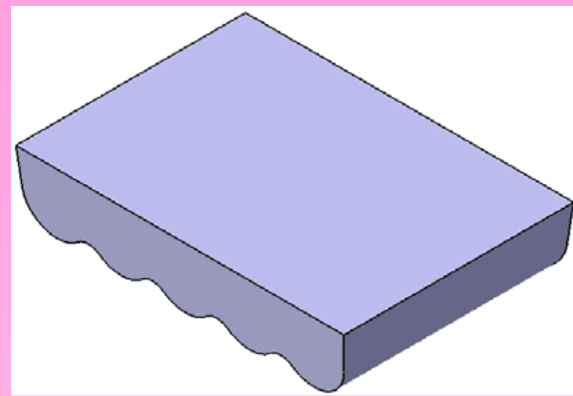


Figure E The model after creating the pocket feature

4. Fillet the edges of the model, as shown in **Figure F** and **Figure G.**

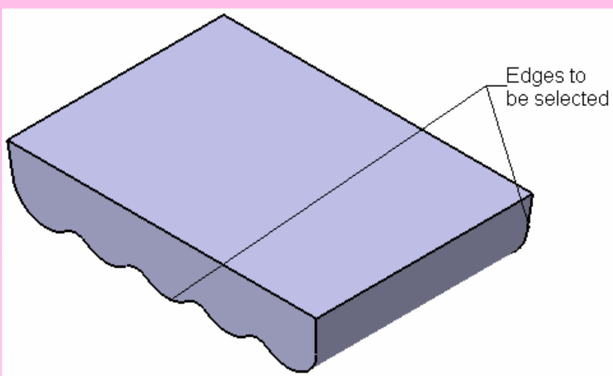


Figure F Edges to be selected

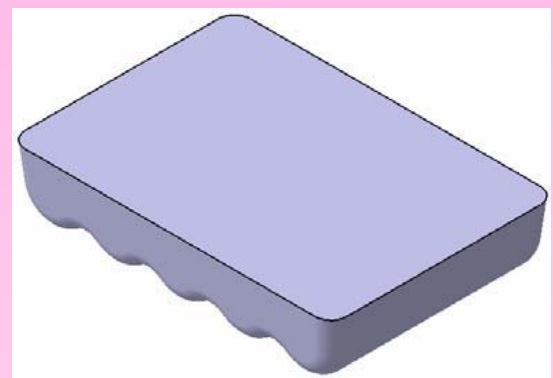


Figure G The model after filleting the edges

5. Shell the model and remove the top face.

6. Create a pocket feature that will be used as a vent for removing water from the Soap Case, as shown in **Figure H**.

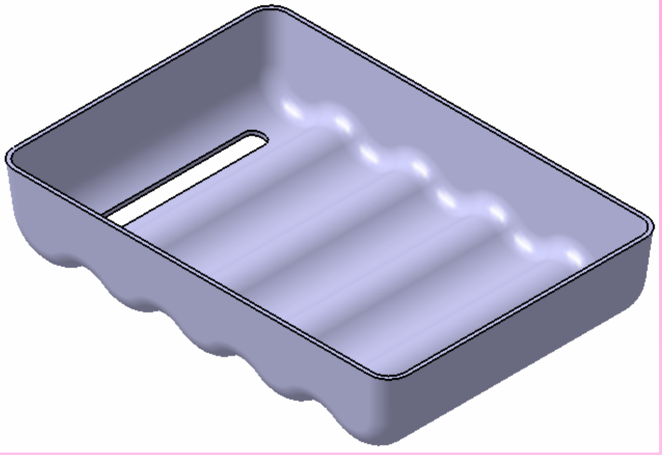


Figure H **The model after creating the pocket feature**

7. Pattern the newly created pocket feature, as shown in **Figure I** and **Figure J**.

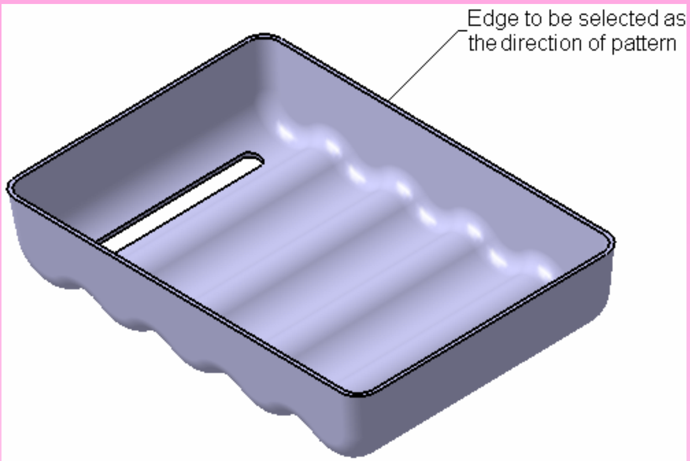


Figure I Edge to be selected as reference

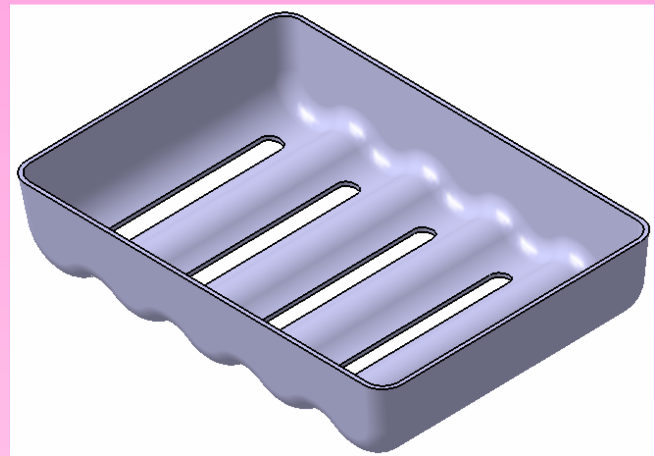


Figure J The model after creating the pattern

8. Create the standoff for the Soap Case using the **Pad**, **Draft**, and the **Edge Fillet** tools, as shown in **Figure K.**

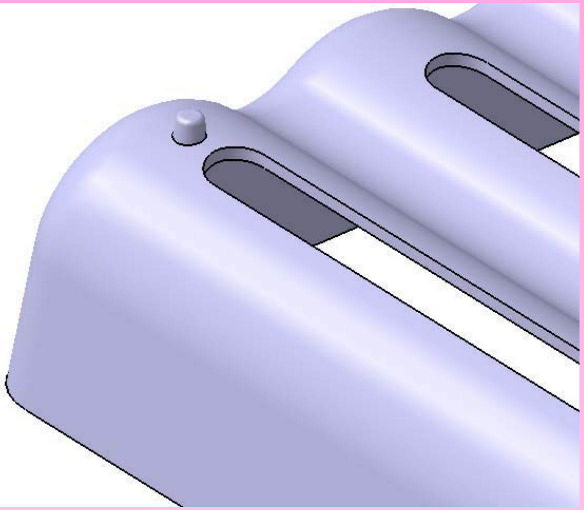


Figure K The model after creating the standoff

9. Pattern the standoff, as shown in **Figure L** and **Figure M**.

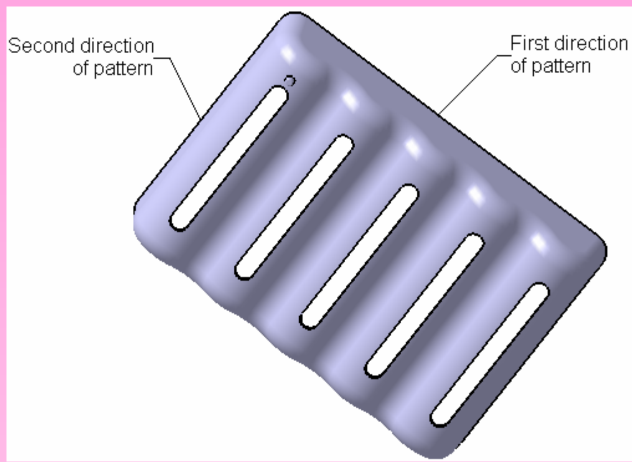


Figure L Edges to be selected as reference

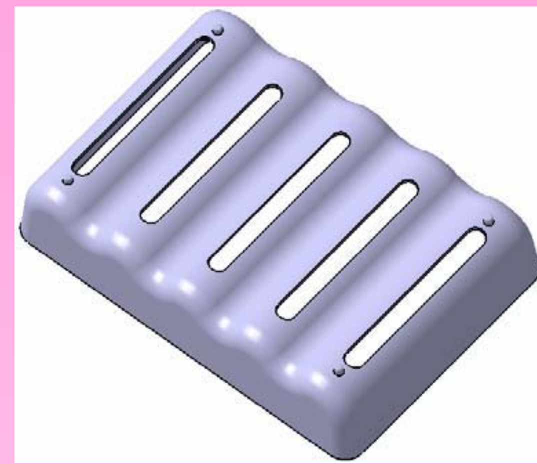


Figure M Final model after patterning

10. Save the file in **\My Documents\CATIA\c07** folder and then close it.

Tutorial 2

In this tutorial, you will create the model of the Motor Cover shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

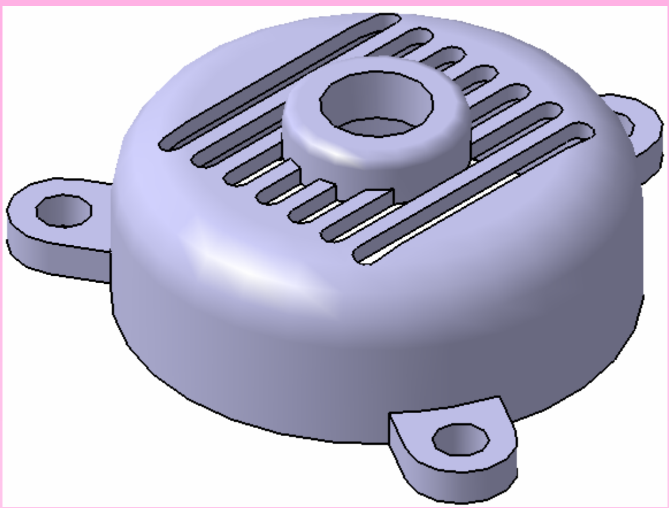


Figure A Model of the Motor Cover for Tutorial 2

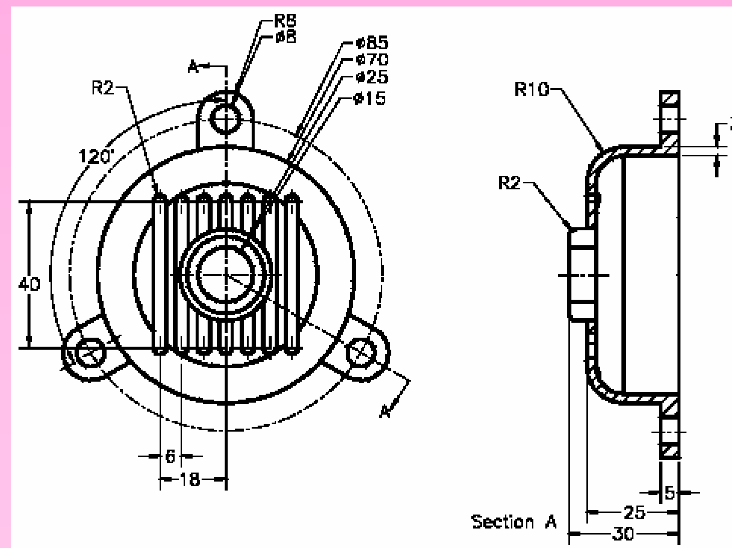


Figure B Views and dimensions of the Motor Cover for Tutorial 2

1. Create the base feature of the model by revolving a sketch drawn on the yz plane, as shown in **Figure C**.

Figure C Base feature of the model

2. Shell the model and remove the bottom face of the model, as shown in **Figure D**.

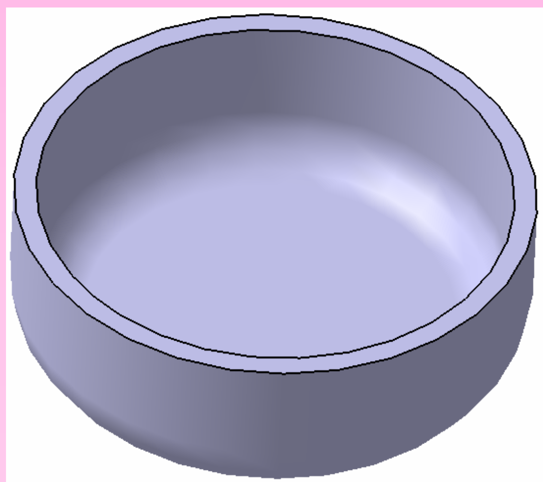


Figure D The model after creating the shell feature

3. Create the pad feature on the outer periphery of the base feature, as shown in **Figure E**.

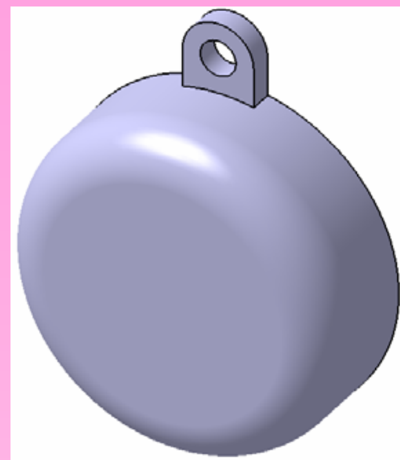


Figure E The model after creating the pad feature

4. Pattern the newly created pad feature using the **Circular Pattern** tool, as shown in **Figure F**.

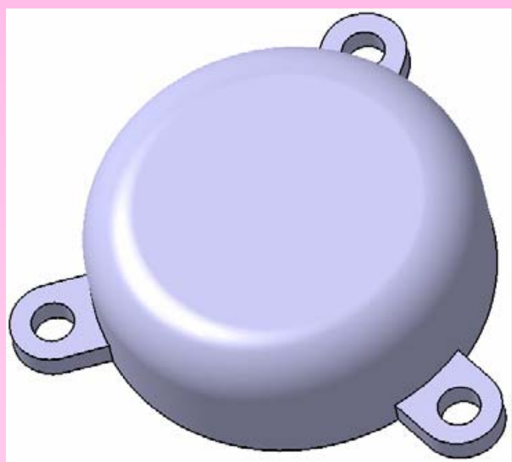


Figure F The model after patterning the pad feature

5. Create the pocket feature on the top face of the base feature, as shown in **Figure G**.

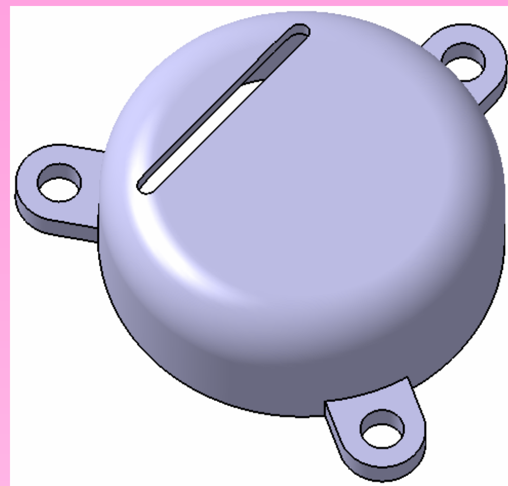


Figure G The model after creating the pocket feature

6. Pattern the pocket feature using the **Rectangular Pattern** tool, as shown in **Figure H**.

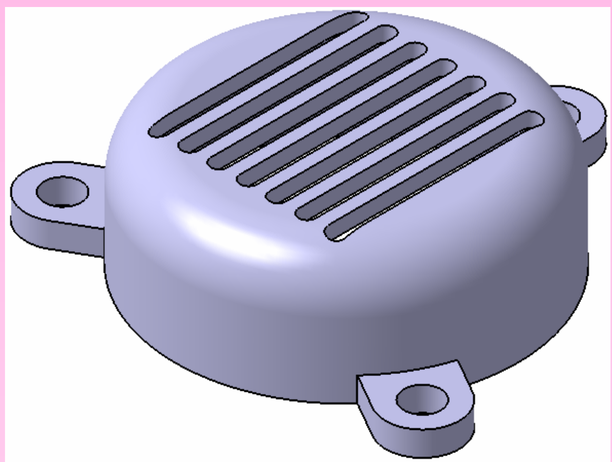


Figure H The model after patterning the pocket feature

7. Create the remaining features of the model, as shown in **Figure I**.

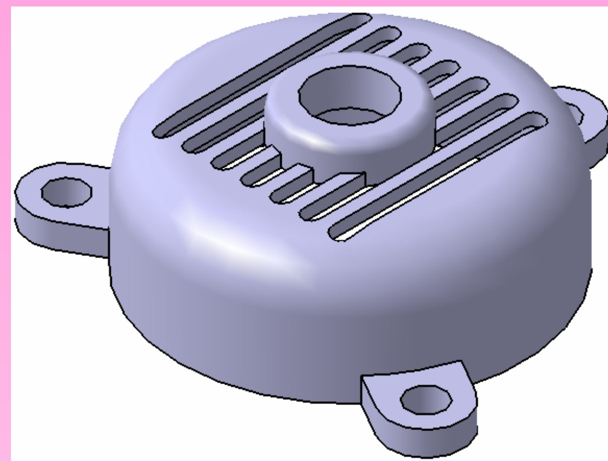


Figure I Final model of the Motor Cover

8. Save the file in **\My Documents\CATIA\c07** folder and then close it.

Exercise 1

Create the model of the Bracket shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

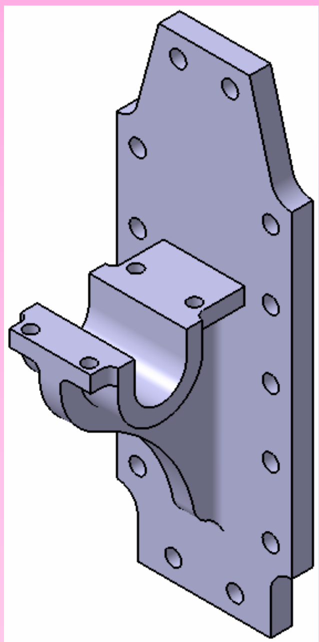


Figure A Model of the Bracket for Exercise 1

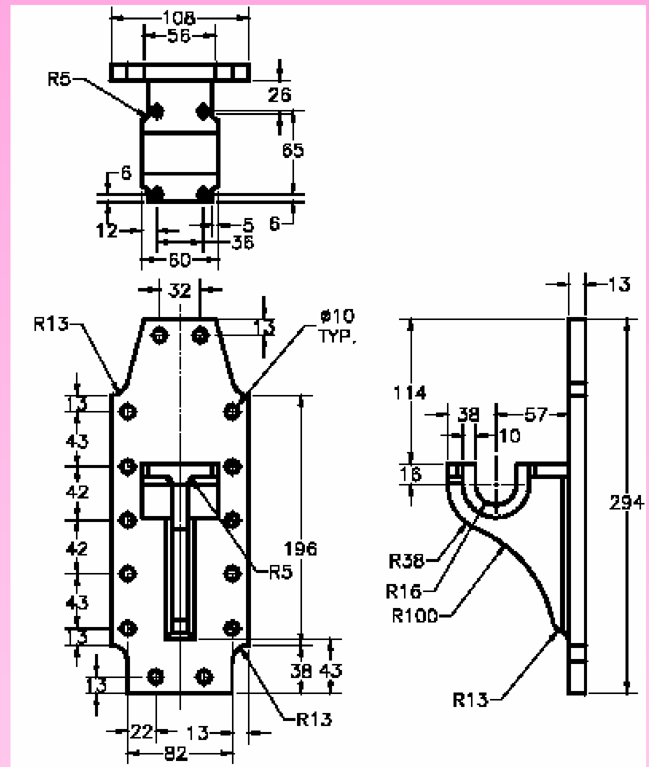


Figure B Views and dimensions of the Bracket for Exercise 1

❑ Exercise 2

Create the model of the Valve Body shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

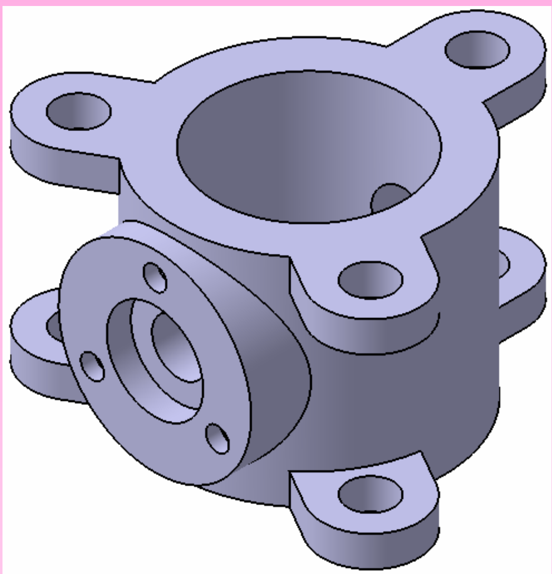


Figure A Model of Valve Body for Exercise 2

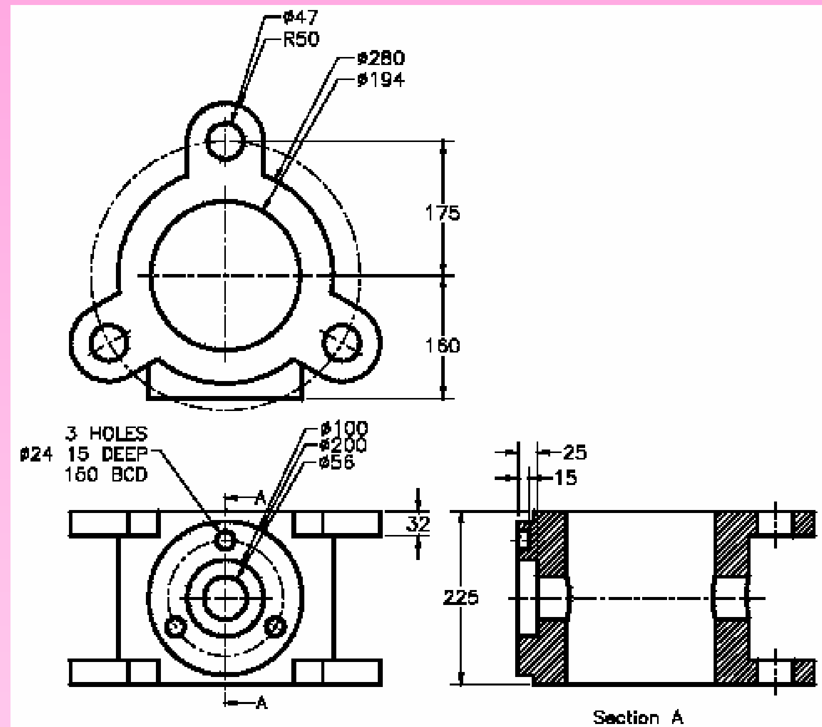


Figure B The views and dimensions of the Valve Body for Exercise 2

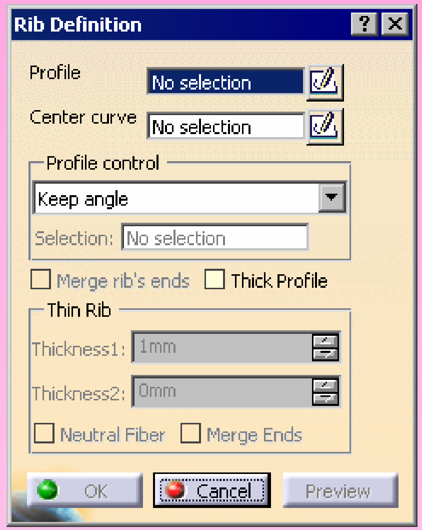
Learning Objectives:

- *Create rib features.*
- *Create slot features.*
- *Create multi-section solid features.*
- *Create removed multi-section solid features.*

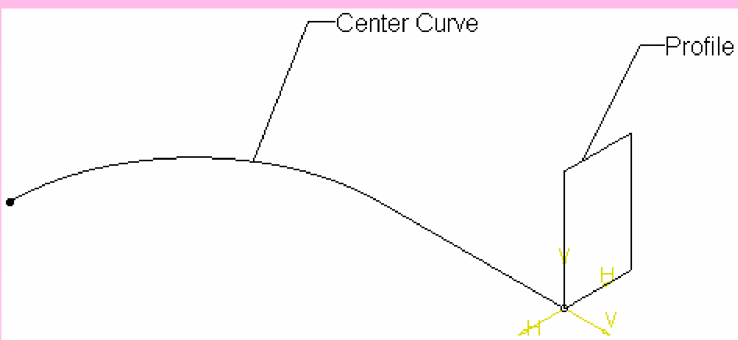
ADVANCED MODELING TOOLS

Creating Rib Features

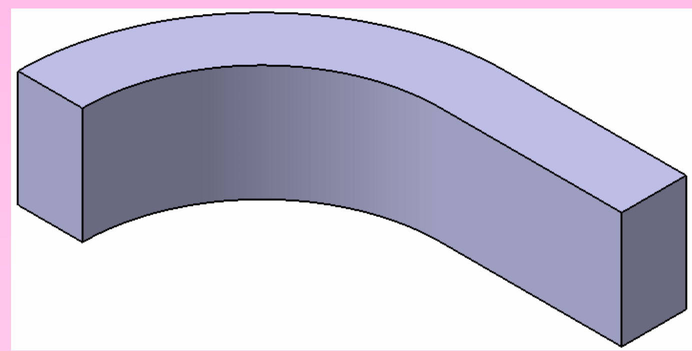
To create a rib feature, choose the **Rib** button from the **Sketch-Based Features** toolbar; the **Rib Definition** dialog box is displayed, as shown in the figure.



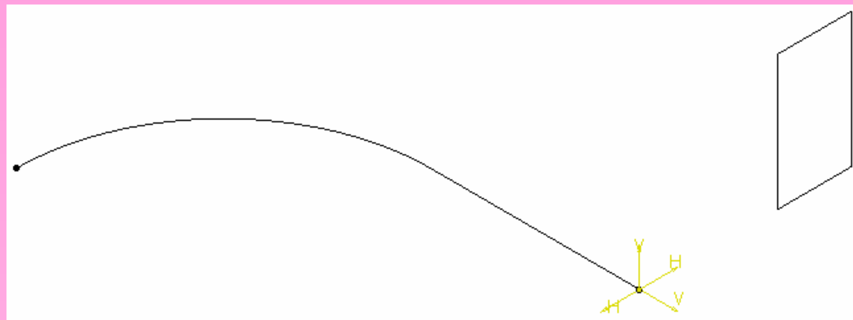
The Rib Definition dialog box



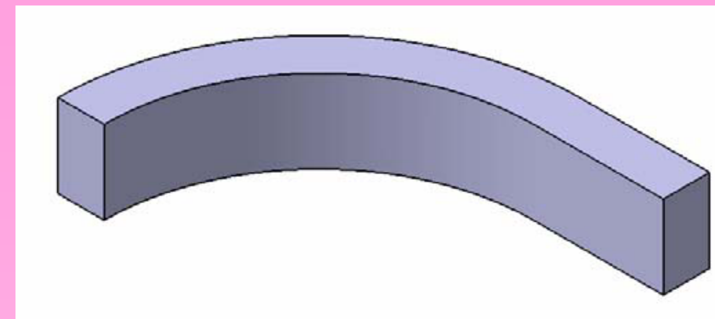
Profile and the center curve to be selected



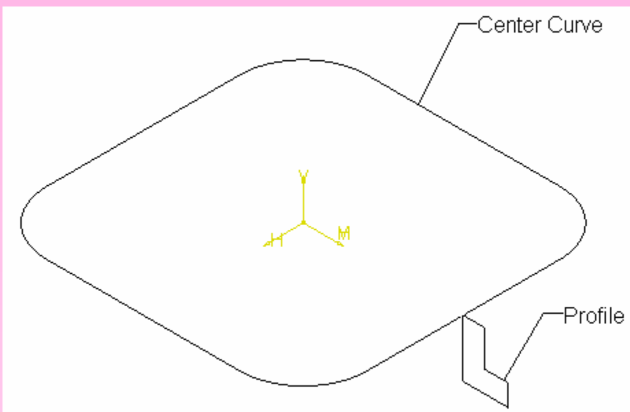
Resulting rib feature



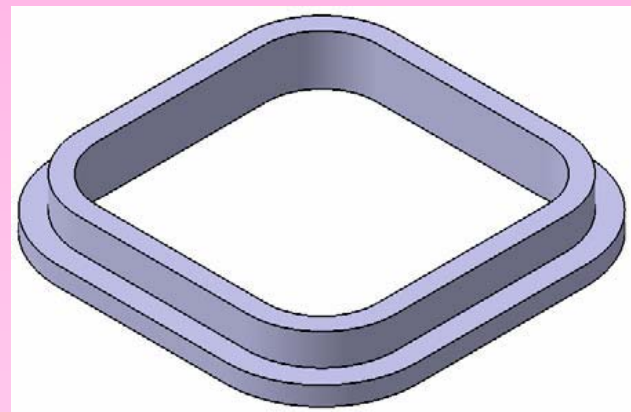
Unattached profile and the center curve



Resulting rib feature

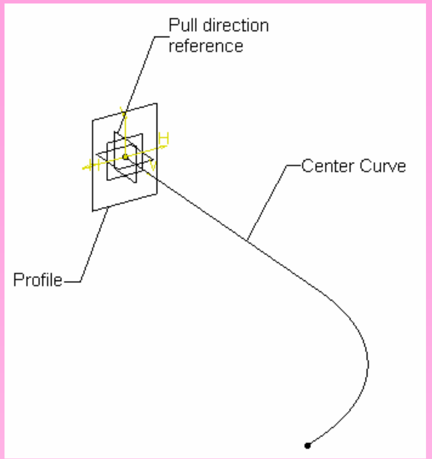


Closed profile and closed center curve

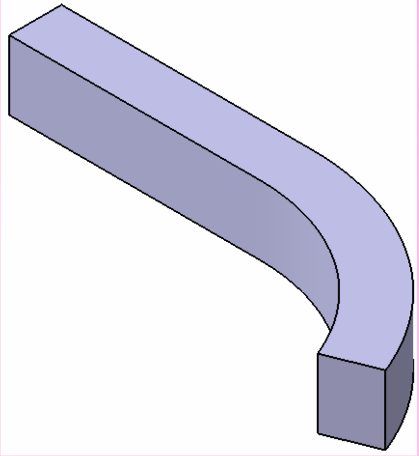


Resulting rib feature

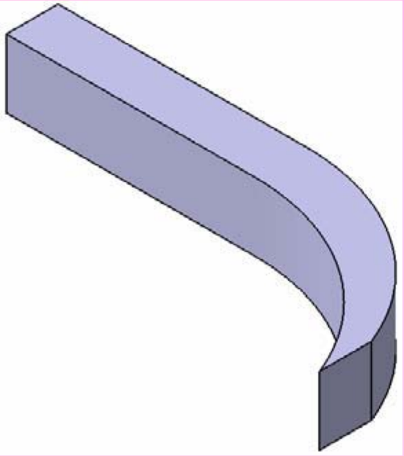
- Defining the Pulling Direction



The profile, center curve, and the pulling direction to be selected

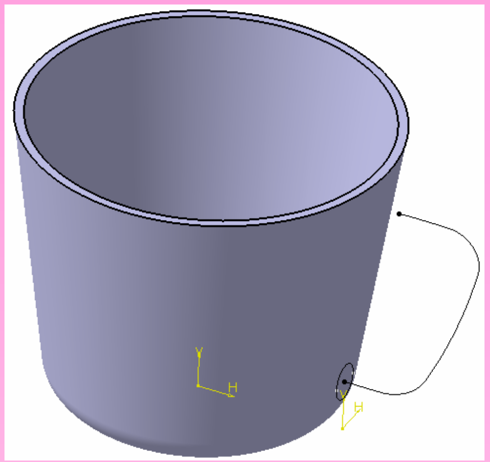


The rib feature without the pulling direction

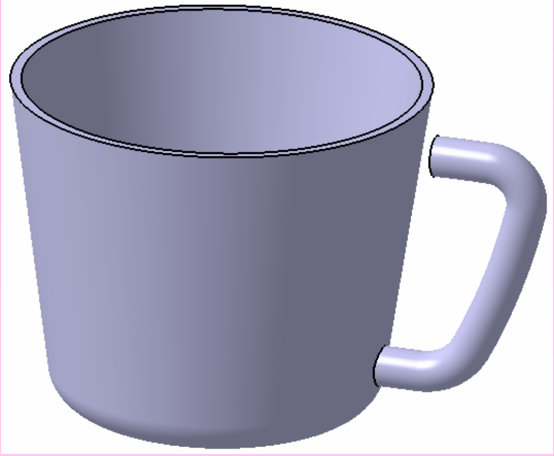


The rib after defining the pulling direction

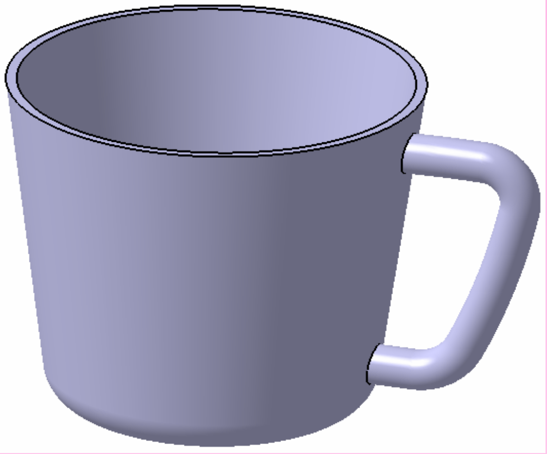
- **Merging the End Faces of the Rib**



The profile and center curve



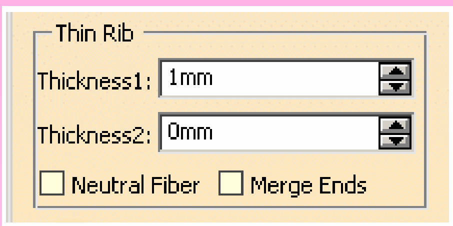
Rib feature with the Merge rib's end check box cleared



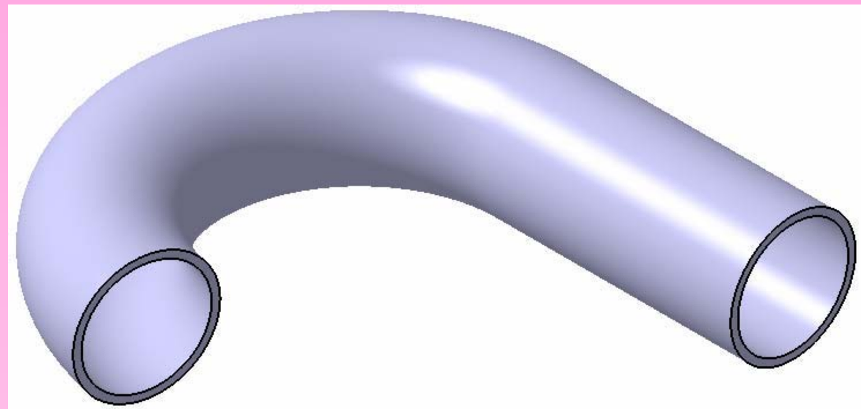
Rib feature with the Merge rib's end check box selected

- **Creating Thin Rib Features**

You can create a thin rib feature by selecting the **Thick Profile** check box from the **Rib Definition** dialog box.



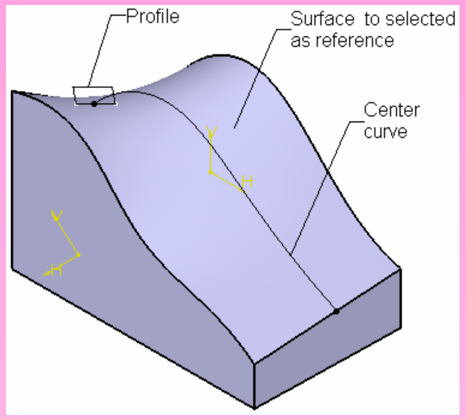
The Thin Rib area of the Rib Definition dialog box



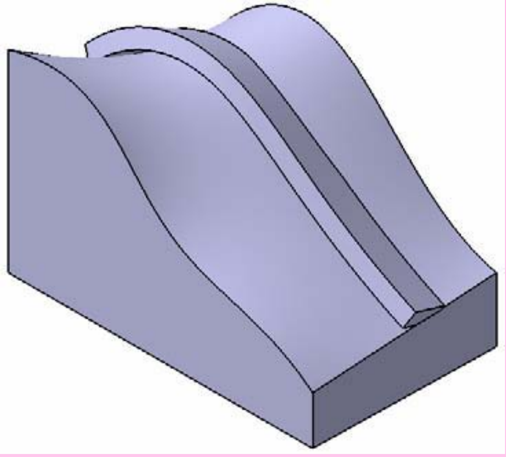
A thin rib feature

- **Defining the Reference Surface**

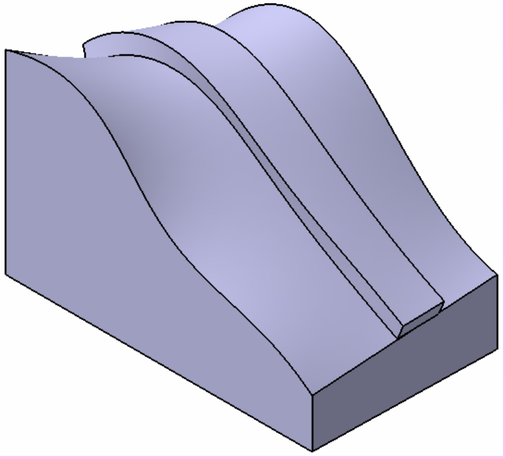
A reference surface can be defined for creating the rib feature.



The profile, center curve, and reference surface to be selected

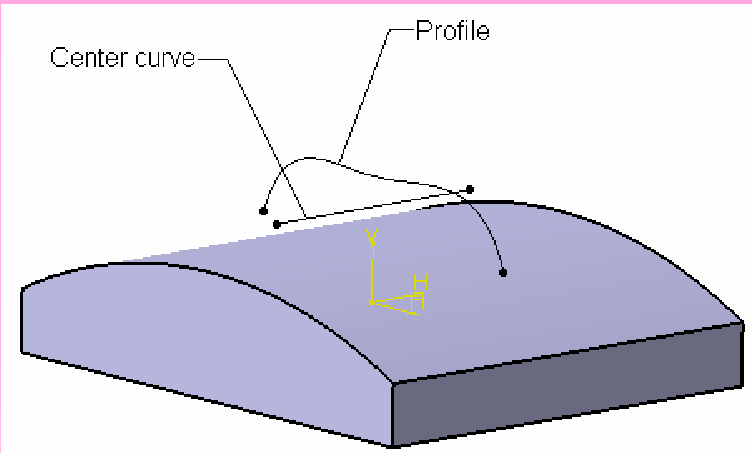


Rib feature with the Keep Angle option selected from the Profile control area

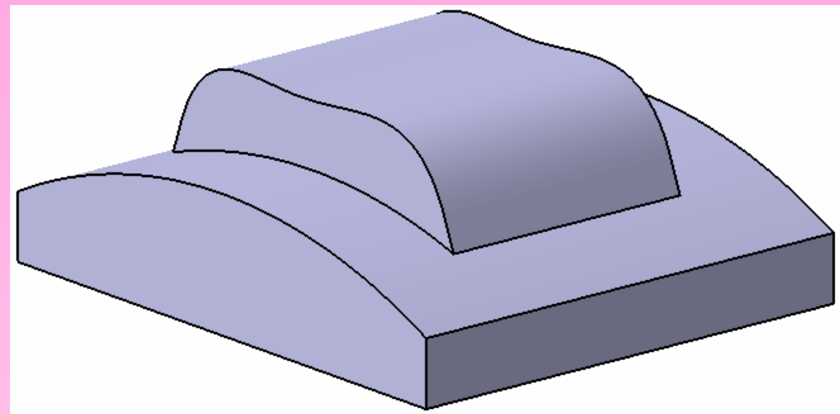


Rib feature after defining the reference surface

- **Creating Rib Features Using Open Profile and Open Center Curve**



Profile and the center curve

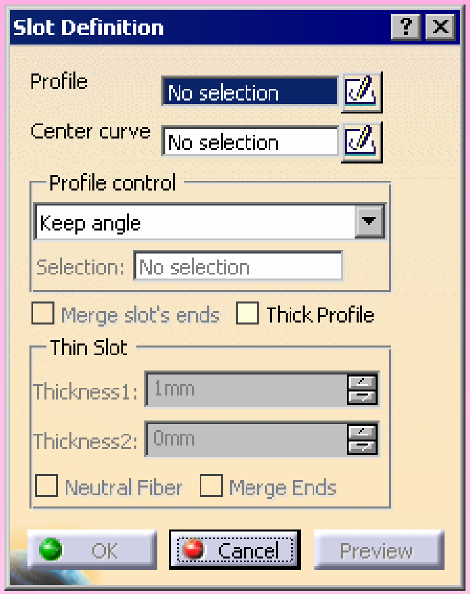


Resulting rib feature

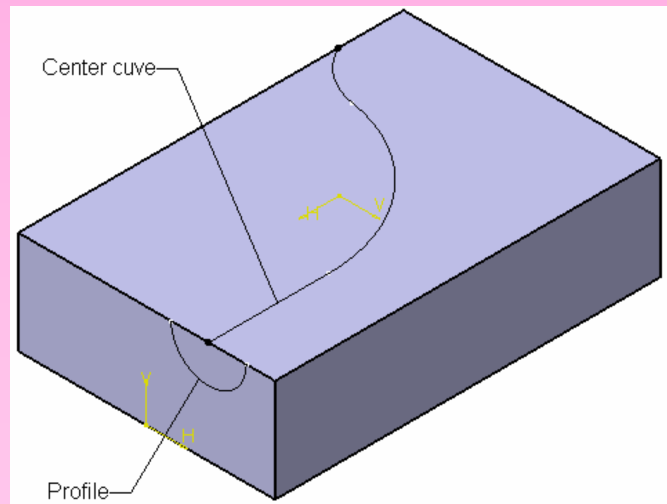
• Creating Slot Features

- The **Slot** tool is used to remove the material by sweeping a profile along a center curve.

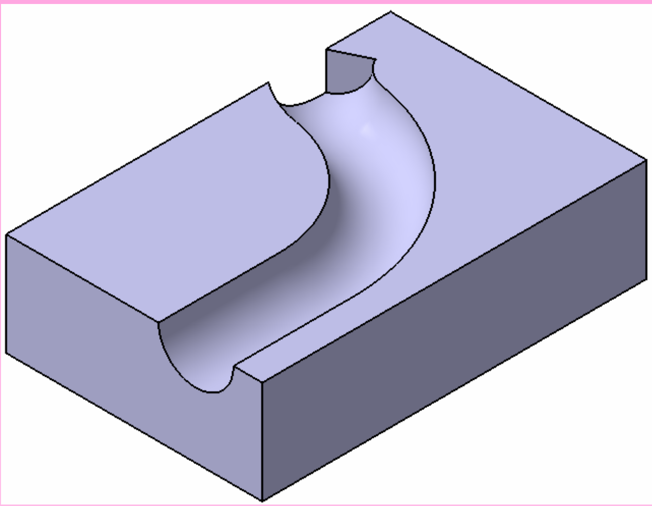
- To create this feature, choose the **Slot** button from the **Sketch-Based Features** toolbar; the **Slot Definition** dialog box is displayed, as shown in the figure.



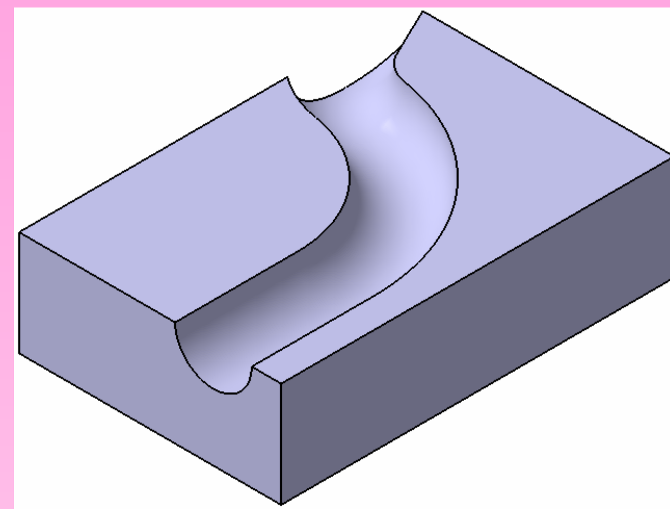
The Slot Definition dialog box



Profile and the center curve

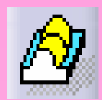


Resulting rib feature

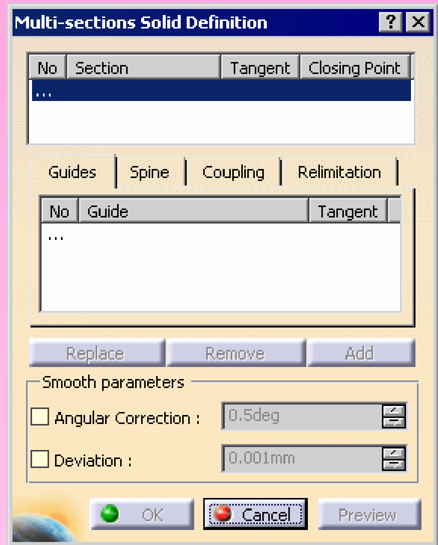


The rib feature, after selecting the **Merge slot's ends** check box

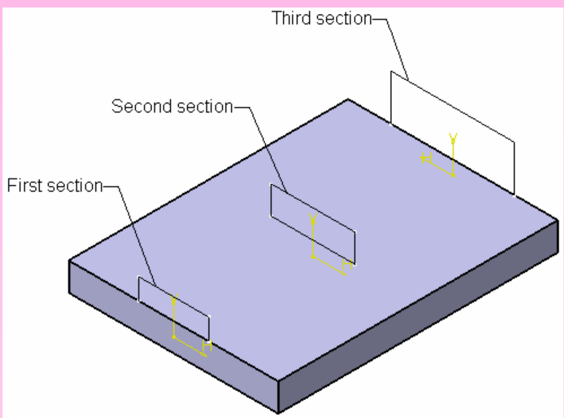
- **Creating Multi-sections Solid Features**



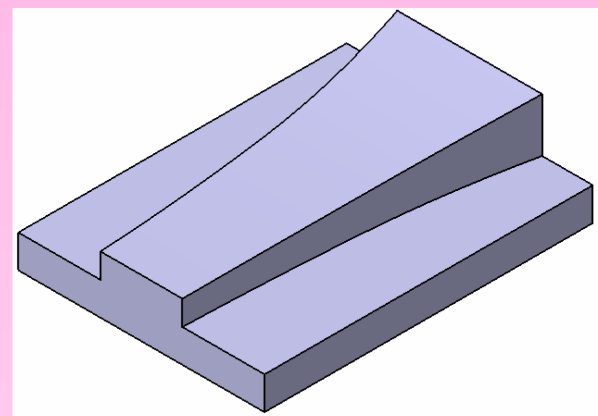
- To create a loft feature, choose the **Loft** button from the **Sketch-Based Features** toolbar.
- The **Multi-sections Solid Definition** dialog box is displayed, as shown in the figure.



The Multi-sections Solid Definition dialog box



Sections selected to create the multi-sections solid



Resulting multi-sections solid feature

Multi-sections Solid Section with Unequal Number of Vertices

- To create this type of feature, invoke the **Multi-sections Definition** dialog box, and select the triangular and the rectangular sections.
- Set the closing points, if required.
- Now, invoke the **Coupling** tab, as shown in **Figure A**.

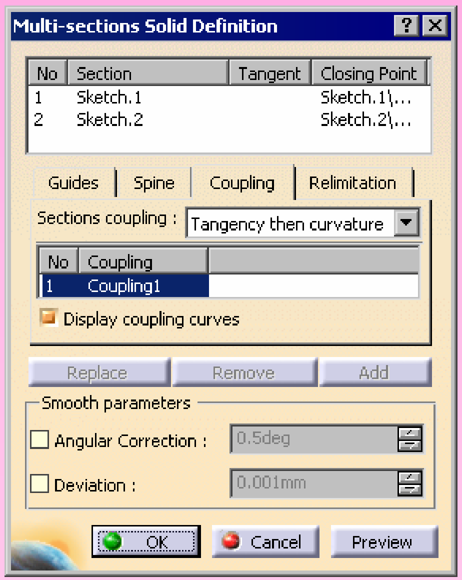
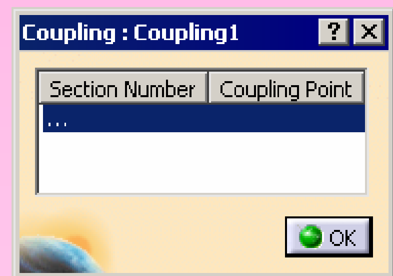
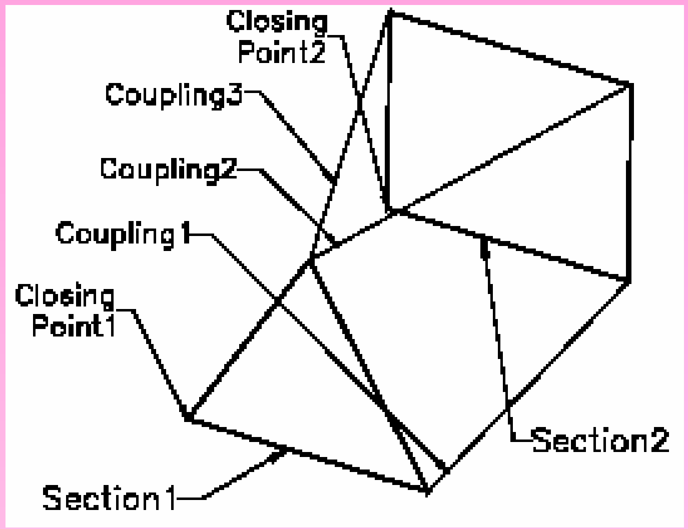


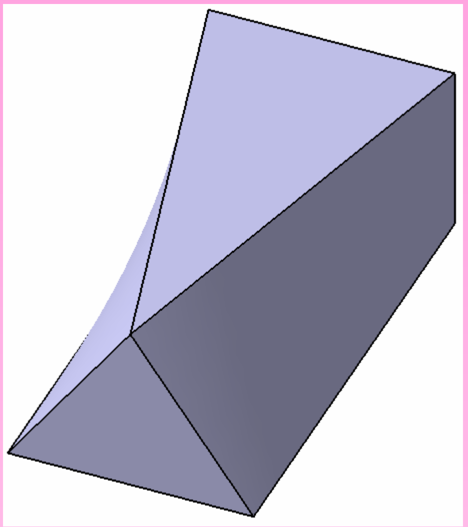
Figure A The **Multi-sections Solid Definition** dialog box after invoking the **Coupling** tab



The **Coupling** dialog box

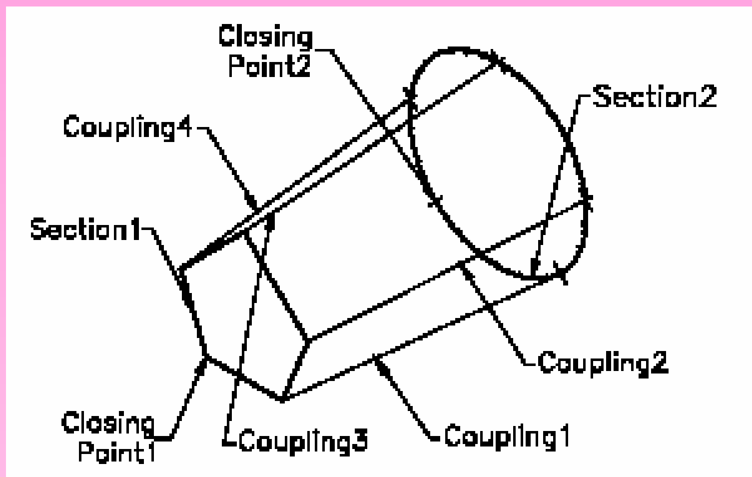


Sections, closing points, and couplings

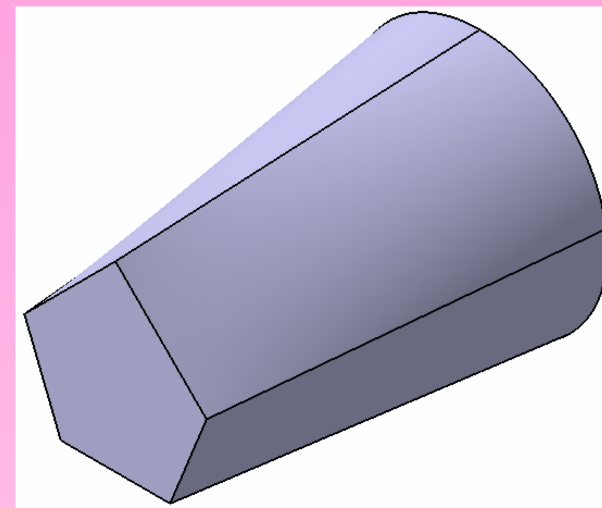


Resulting multi-sections solid feature

- Creating a Multi-sections Solid of Circular Section with a Polygonal



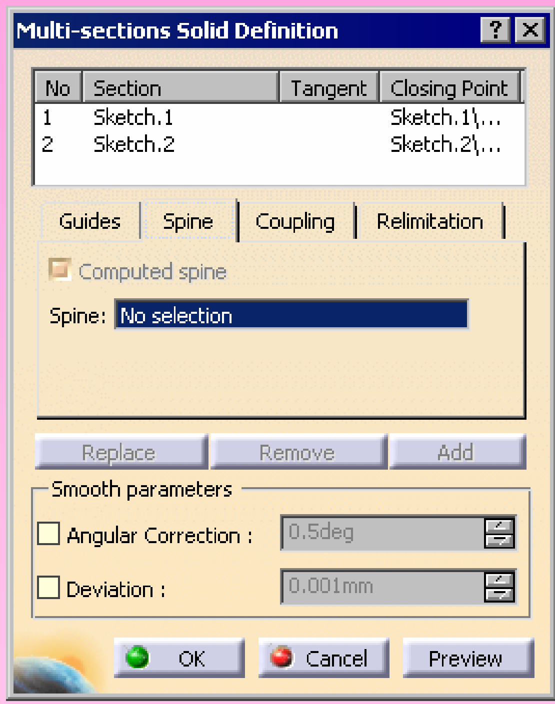
Sections, closing points, and couplings



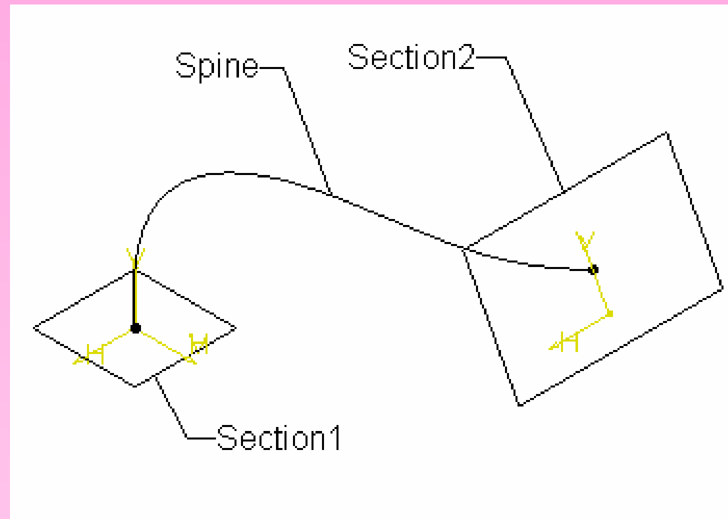
Resulting multi-sections solid feature

• Creating a Multi-sections Solid of the Sections Along a Spine

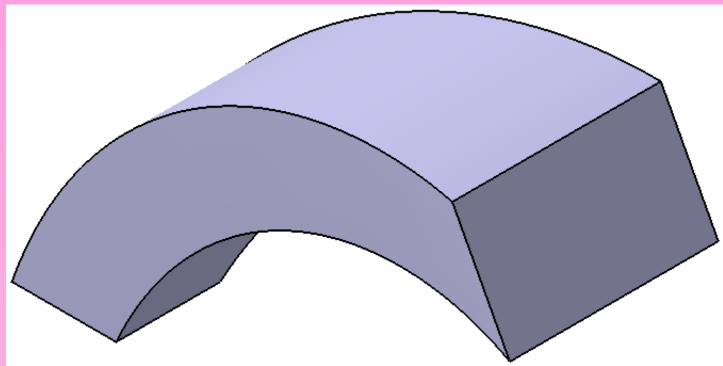
To create a multi-sections solid feature using this option, invoke the **Spine** tab of the **Multi-sections Solid Definition** dialog box, as shown in the figure.



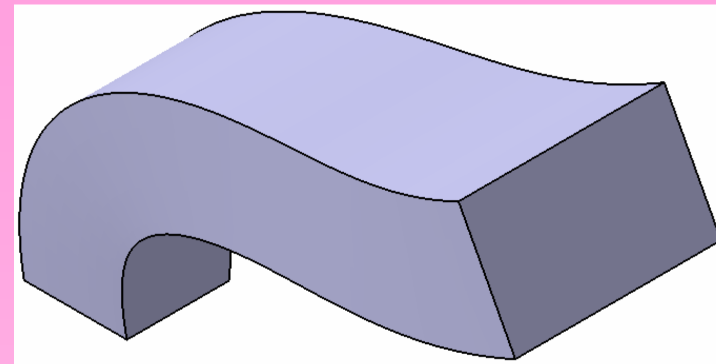
The Multi-sections Solid Definition dialog box after invoking the Spine tab



The sections and the spine to be selected

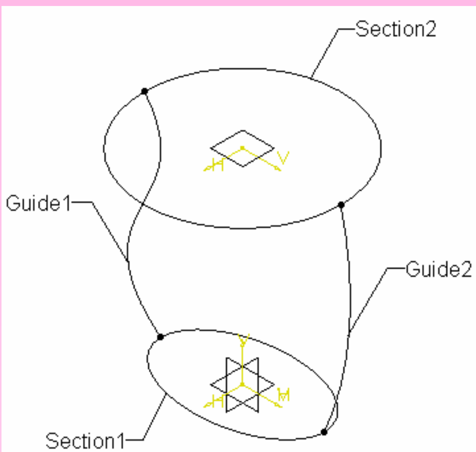


Multi-sections solid feature without selecting the spine

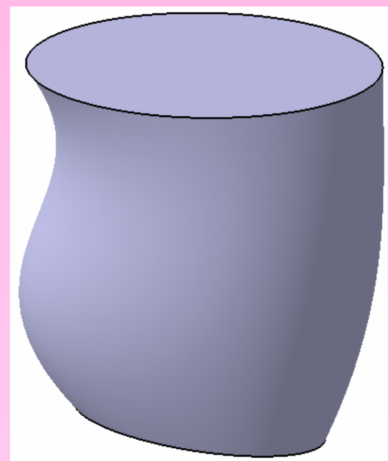


Multi-sections solid feature after selecting the spine

- Creating a Multi-sections Solid Feature with Guides

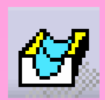


The sections and guides

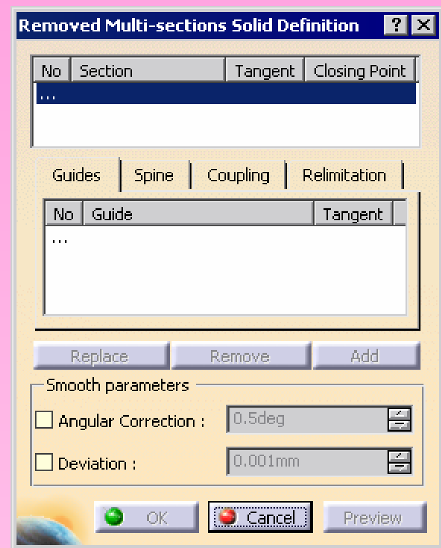


Resulting multi-section solid feature

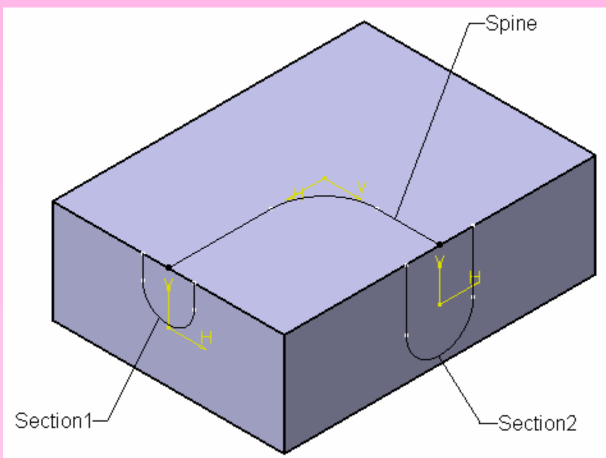
• Creating Removed Multi-sections Solid Features



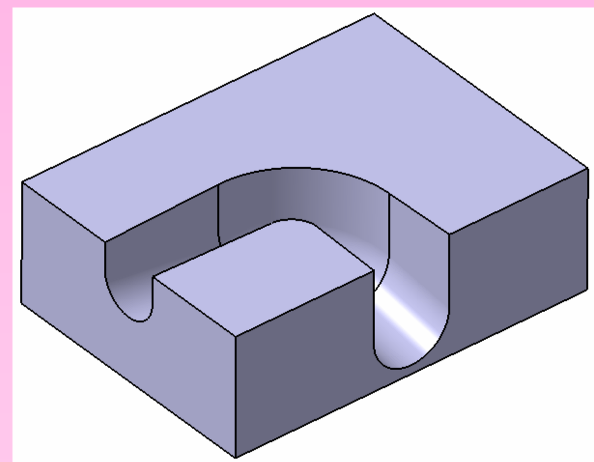
When you invoke this tool, the **Removed Multi-sections Solid Definition** dialog box is displayed, as shown in the figure.



The Removed Multi-sections Solid Definition dialog box



Sections and the spine to be selected



Resulting feature

Tutorial 1

In this tutorial, you will create the model of the Upper Housing as shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 1 hr)**

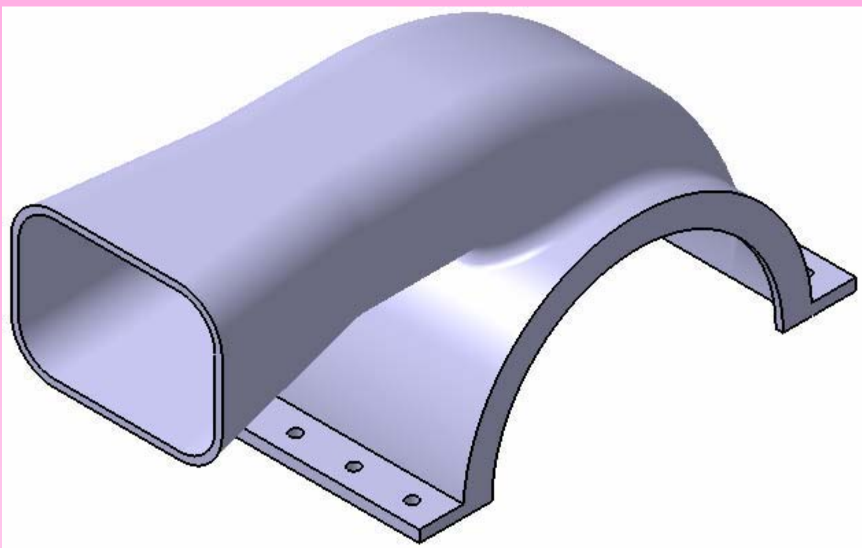


Figure A Model of the Upper Housing for Tutorial 1

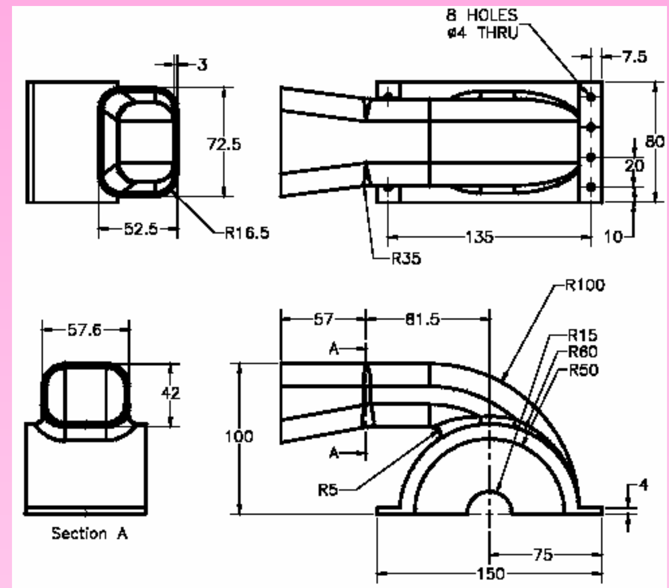


Figure B Views and dimensions of the Upper Housing for Tutorial 1

1. Create the base feature of the model, as shown in **Figure C**.

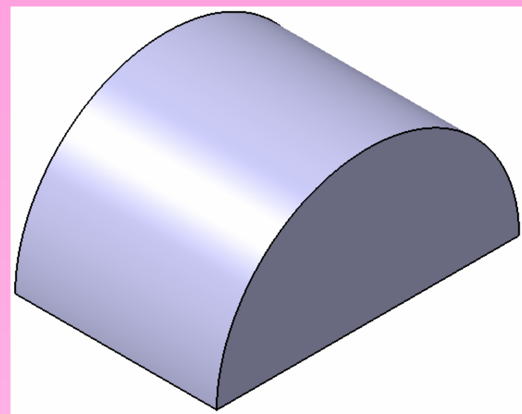


Figure C Base feature of the model

2. Create the rib feature, as shown in **Figure D** through **Figure G**.

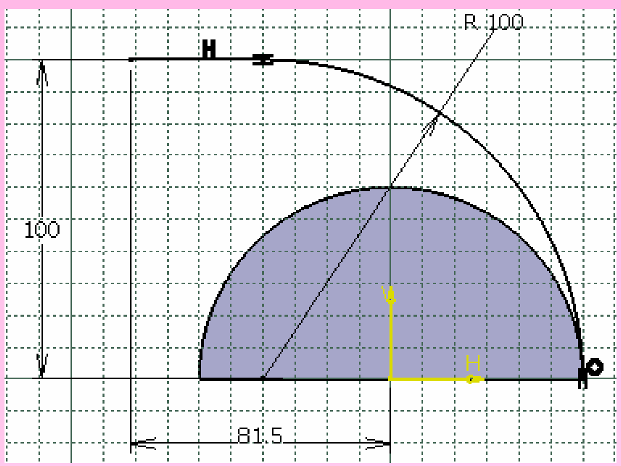


Figure D Sketch of the center curve

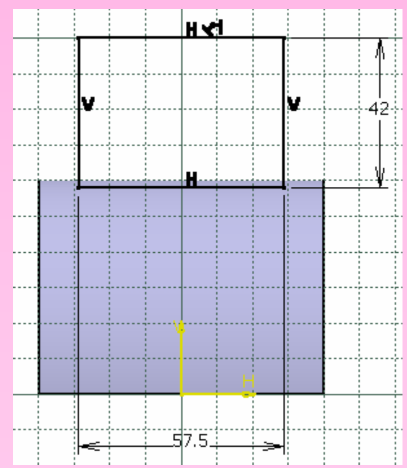


Figure E Sketch of the profile

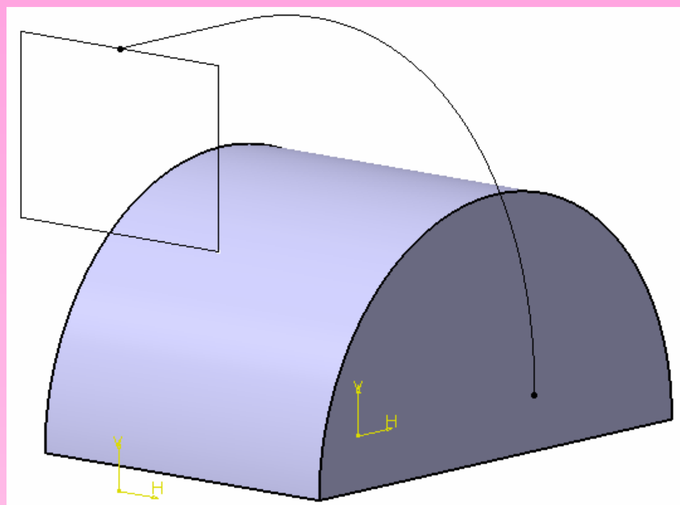


Figure F Sketches for the rib feature

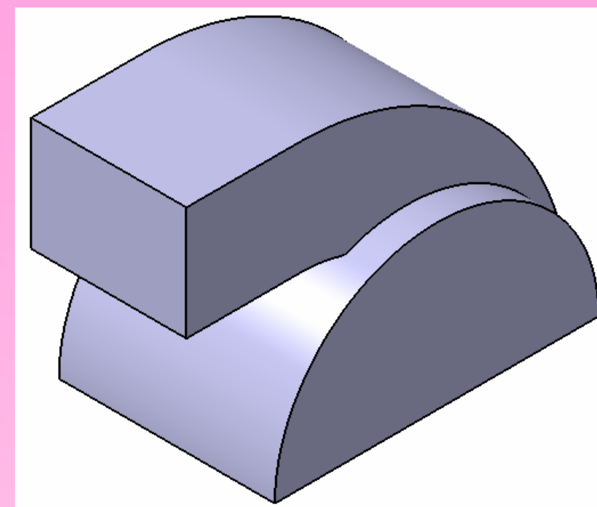


Figure G Model after creating the rib feature

3. Create the multi-sections solid feature, as shown in **Figure H** and **Figure I**.

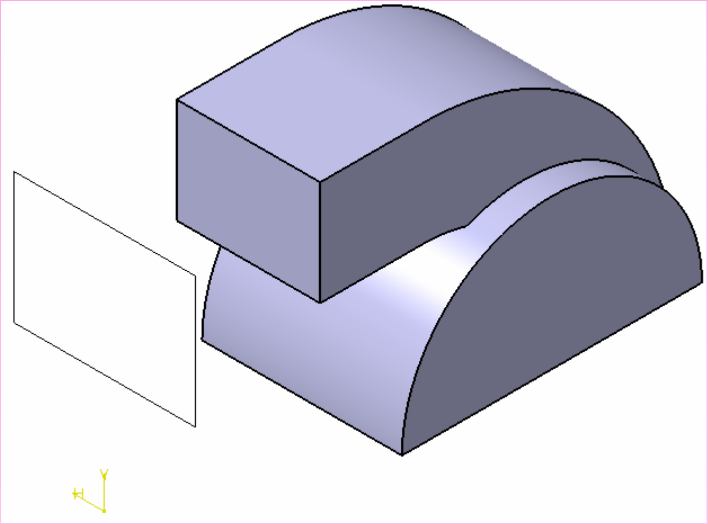


Figure H Second section of the multi-sections solid feature

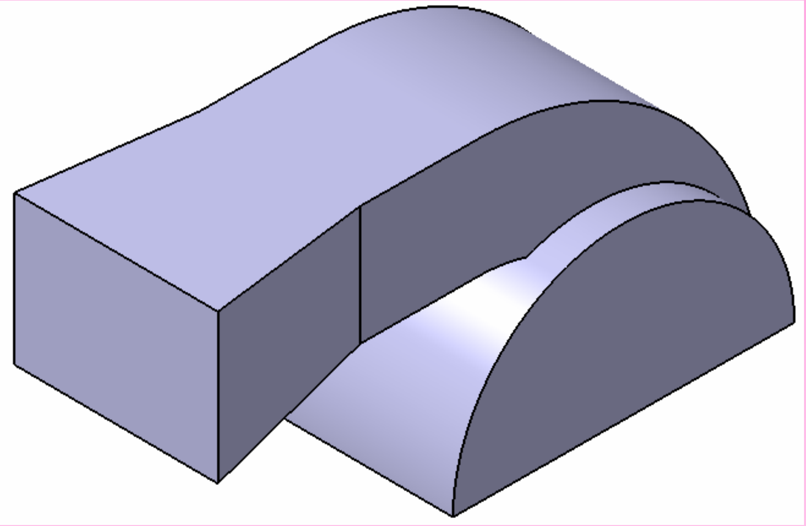


Figure I Model after creating the multi-sections solid feature

4. Fillet the edges of the model, as shown in **Figure J**.

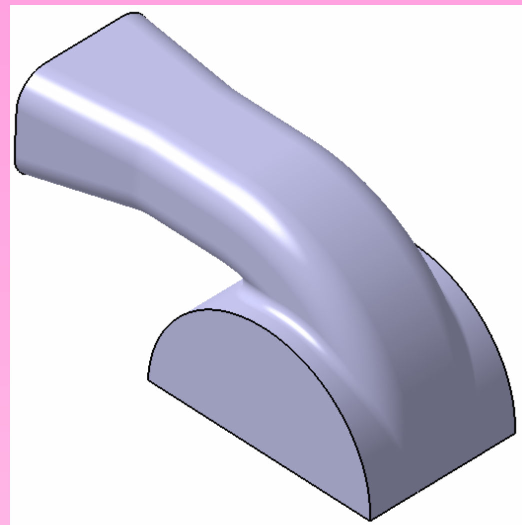


Figure J Model after filleting the edges

5. Shell the model and remove the bottom and the left planar face of the model, as shown in **Figure K**.

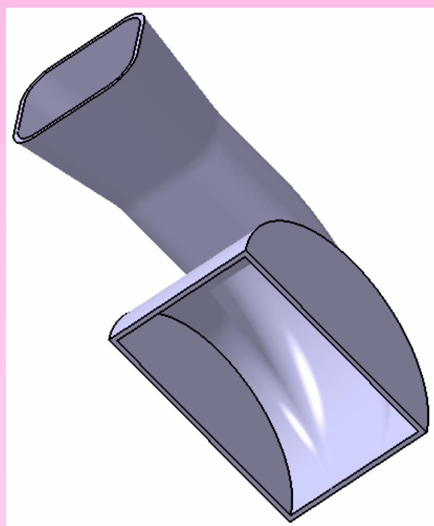


Figure K Model after shelling

6. Create the other features to complete the model, as shown in **Figure L**.

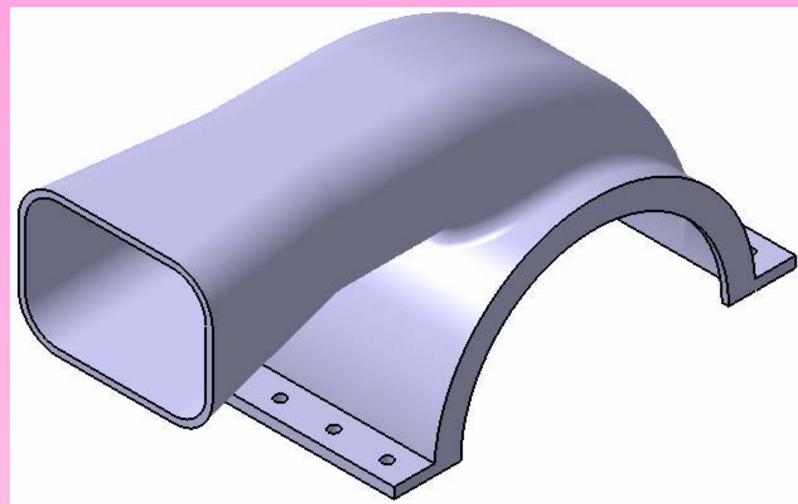


Figure L Final model of the Upper Housing

7. Save the file in **\My Documents\CATIA\c08** folder and then close it.

Tutorial 2

In this tutorial, you will create the model of the Helical Gear shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 1 hr)**

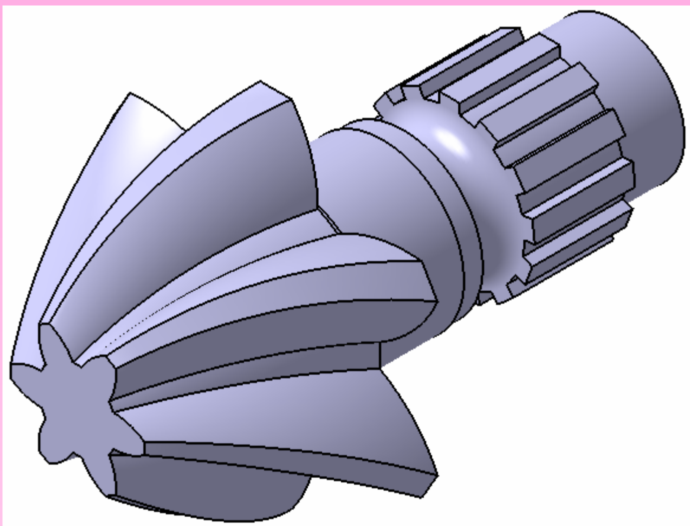


Figure A Model of the Helical Gear for Tutorial 2

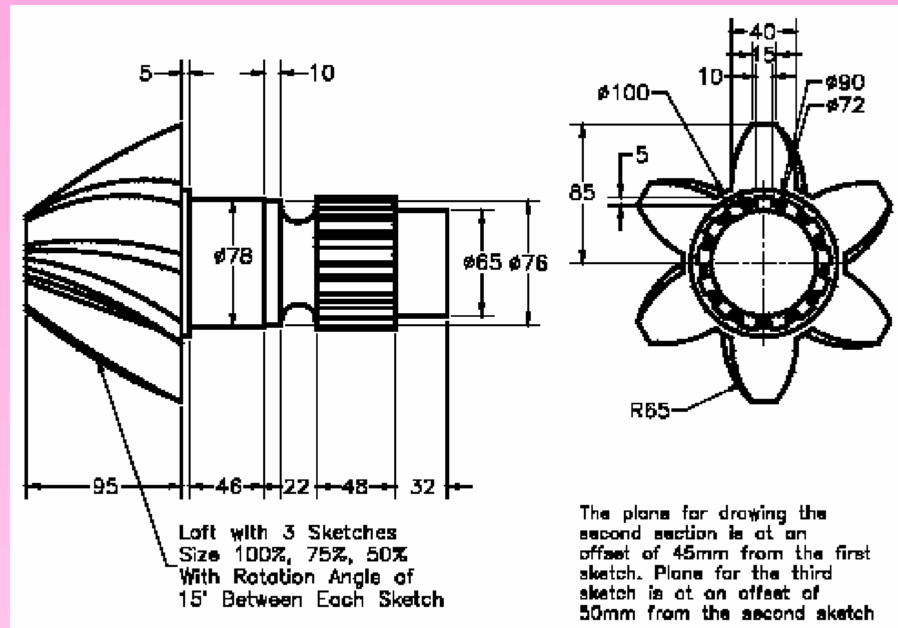


Figure B Views and dimensions for Tutorial 2

1. Create the base feature of the model, as shown in **Figure C**.

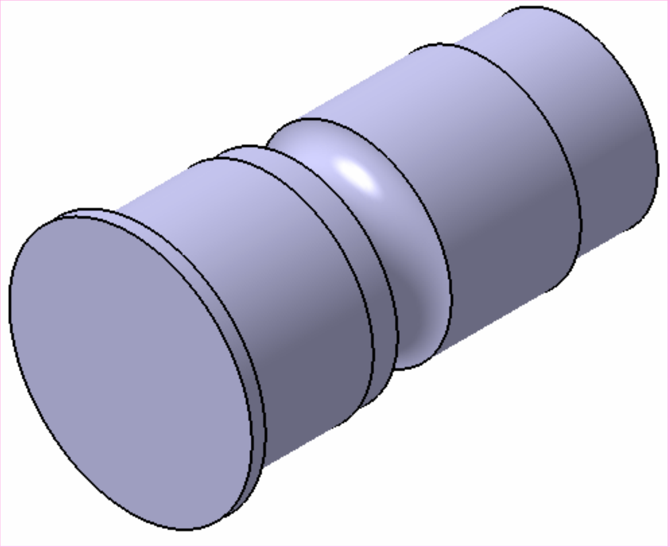


Figure C Base feature of the model

2. Create a pad feature on the right of the base feature and then pattern it using the **Circular Pattern** tool, as shown in **Figure D** and **Figure E**.

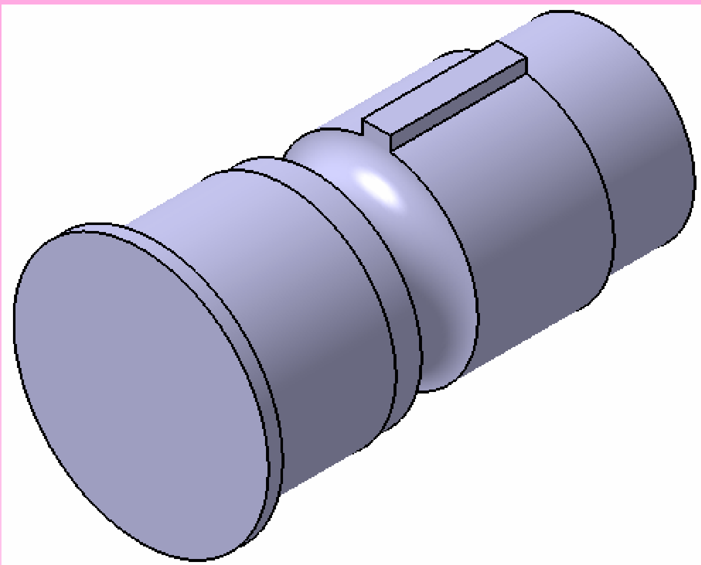


Figure D The model after creating the pad feature

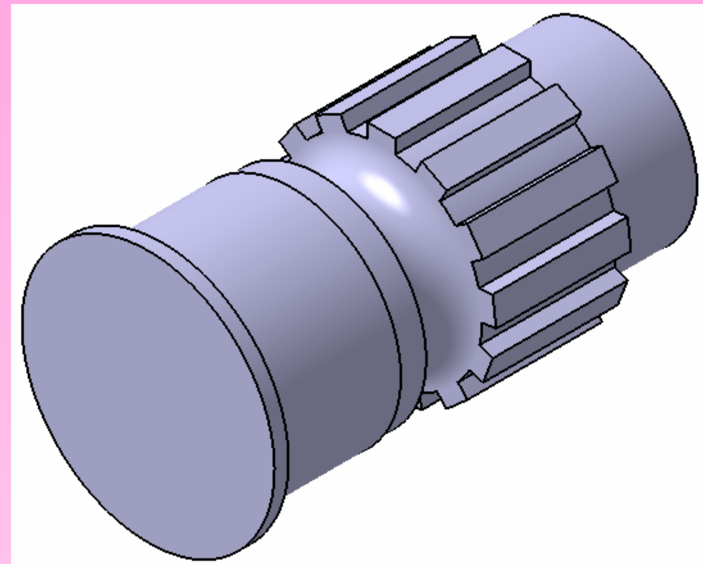


Figure E The model after creating the pattern

3. Draw the sketch of the first section of gear tooth on the front face of the base feature.
4. Draw the sketch of the second section of the gear on a plane that is at an offset distance from the front plane of the base feature. This sketch is the 75% scaled sketch of the first section and is rotated at an angle of 15-degree.
5. Draw the sketch of the third section of the gear on a plane that is at an offset distance from the plane created earlier. This sketch is the 50% scaled sketch of the section and is rotated at an angle of -15-degree, as shown in **Figure F**.

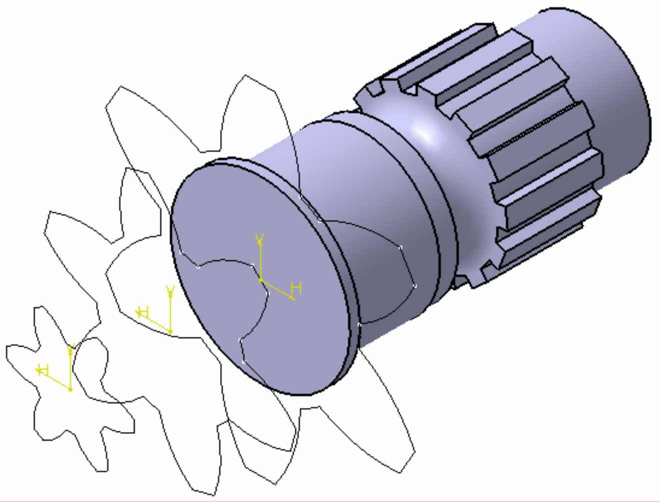


Figure F The model after drawing all the sections

6. Create a multi-sections solid feature using three sections, as shown in **Figure G**.

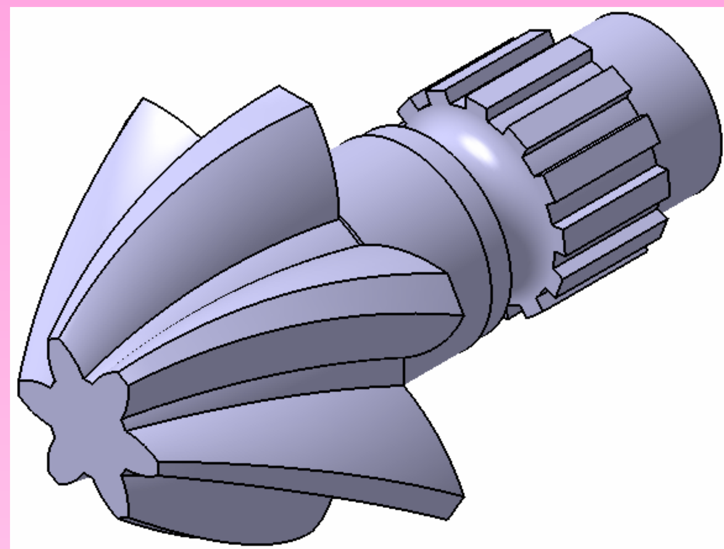


Figure G Final model

7. Save the file in **\My Documents\CATIA\c08** folder and then close it.

❑ Tutorial 3

In this tutorial, you will create the model of the Mouse Cover as shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 30 min)**

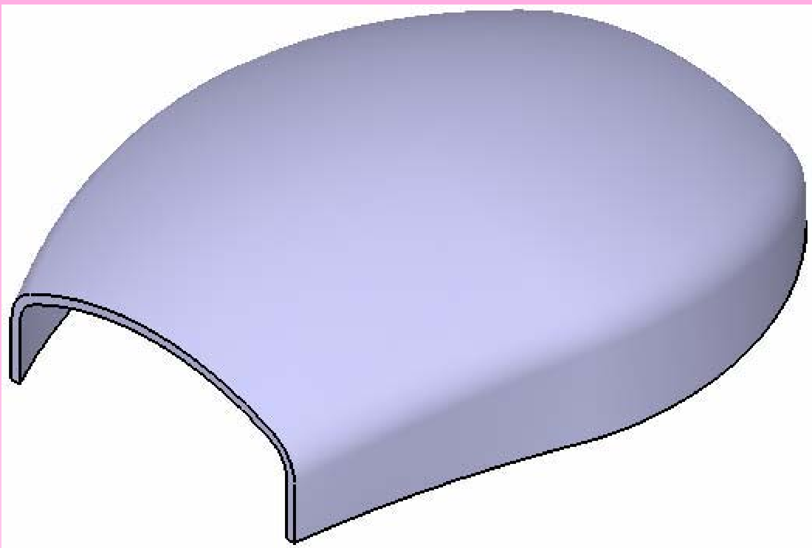


Figure A Model of the Mouse Cover for Tutorial 2

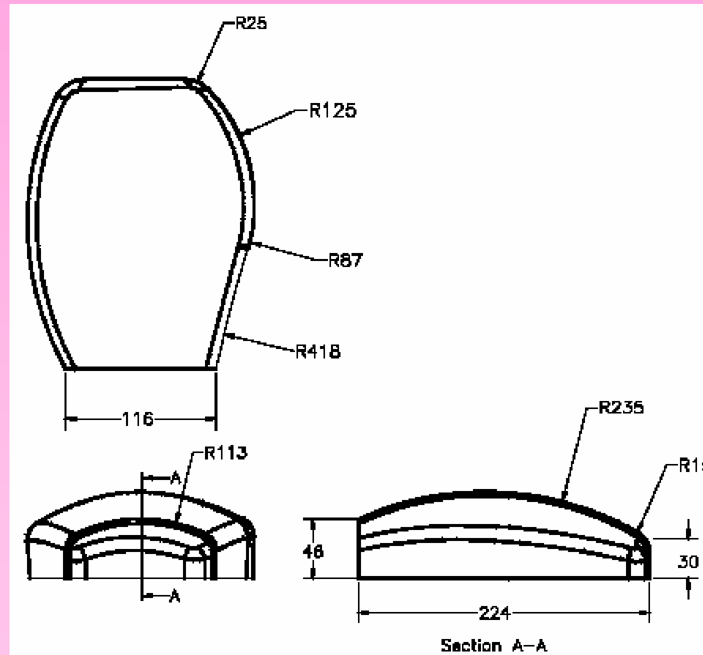


Figure B Drawing views and dimensions for Tutorial 2

1. Draw sketches for creating a multi-sections solid feature with guides. The sketches include two sections and three guides, as shown in **Figure C** through **Figure H**.

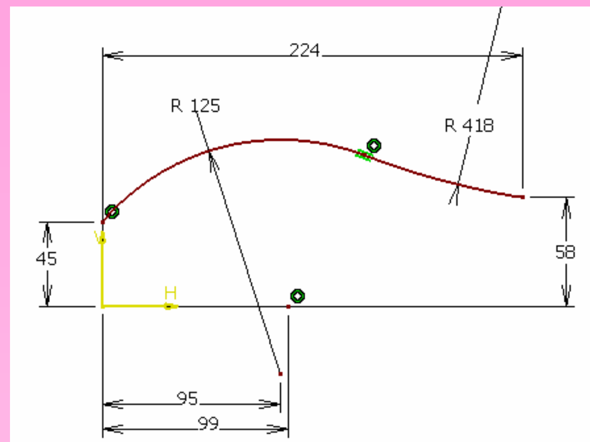


Figure C Sketch of the first guide

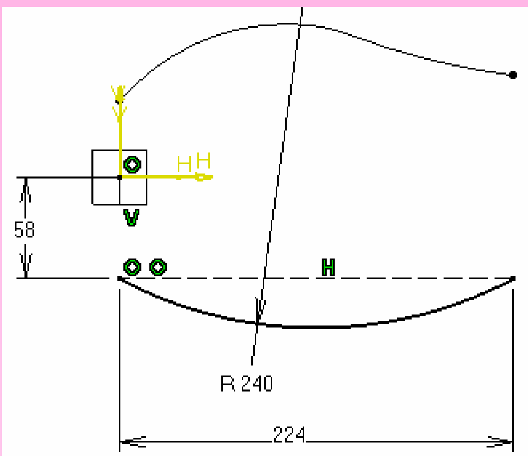


Figure D Sketch of the second guide

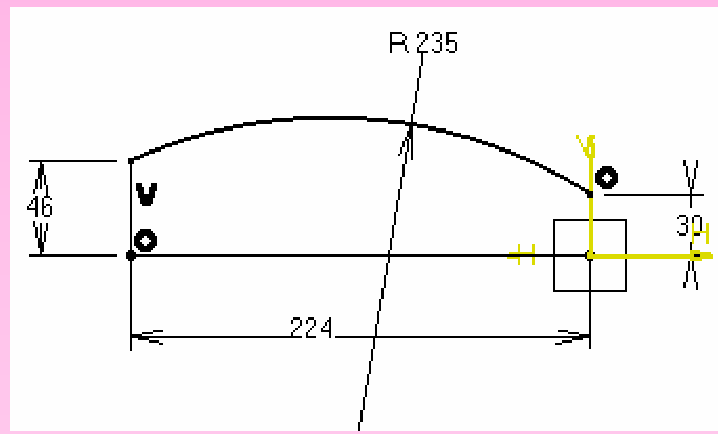


Figure E Sketch of the third guide

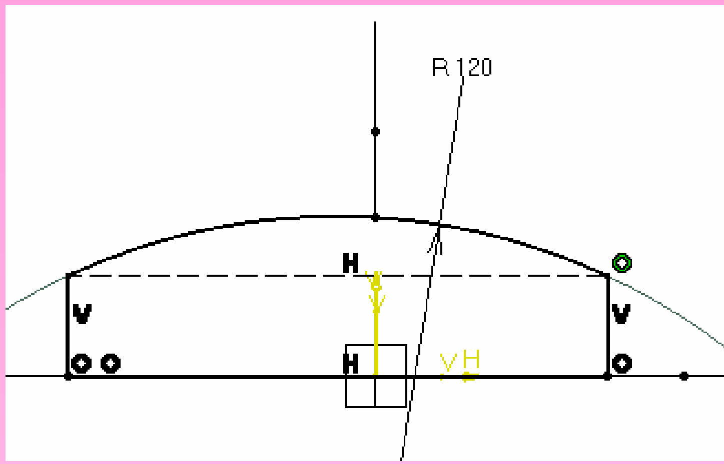


Figure F Sketch of the first section

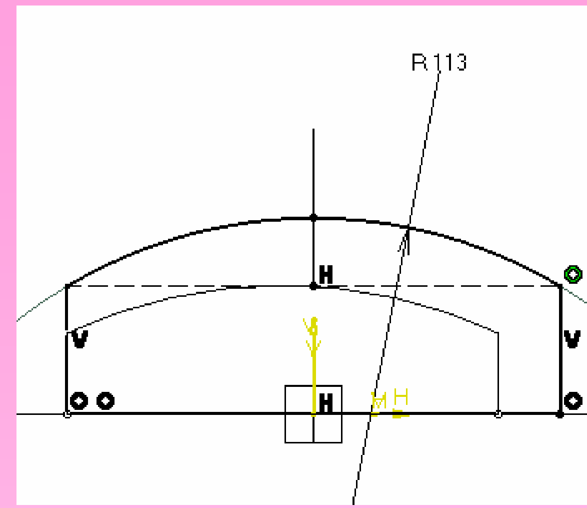


Figure G Sketch of the second section

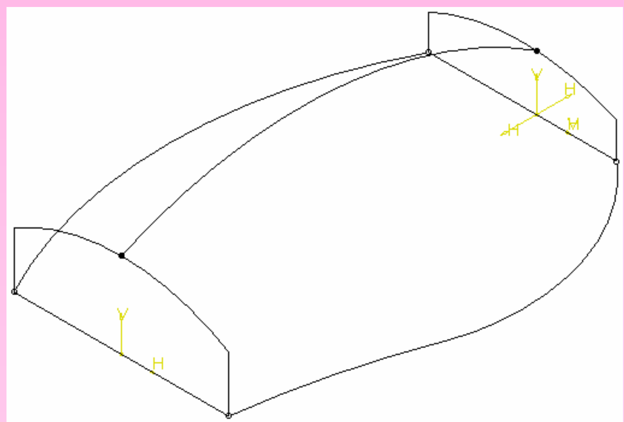


Figure H Model after drawing all sketches

2. Create the base feature by creating multi-section solid with guides, as shown in **Figure I** and **Figure J**.

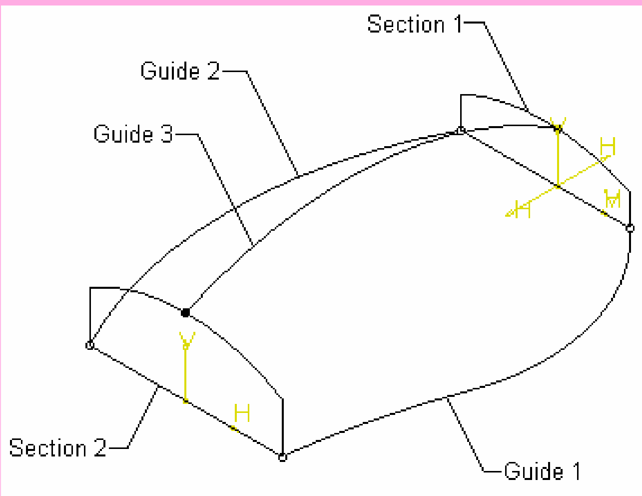


Figure I Sections and guides to be selected

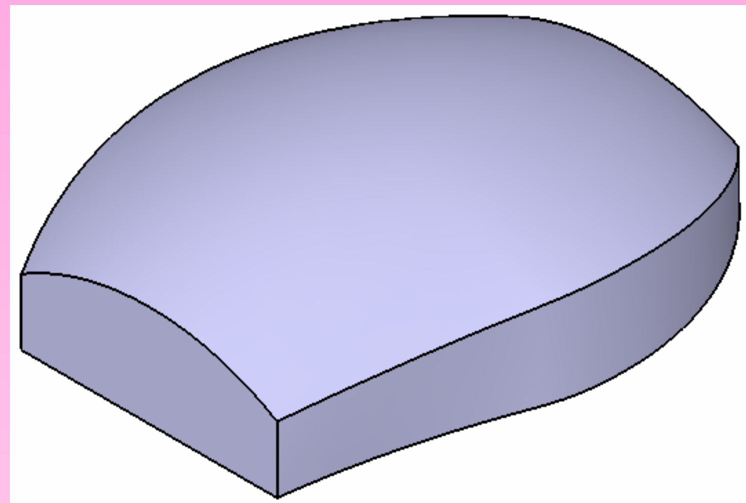


Figure J Base feature of the model

3. Fillet the edges of the base feature, as shown in **Figure K** through **Figure N**.

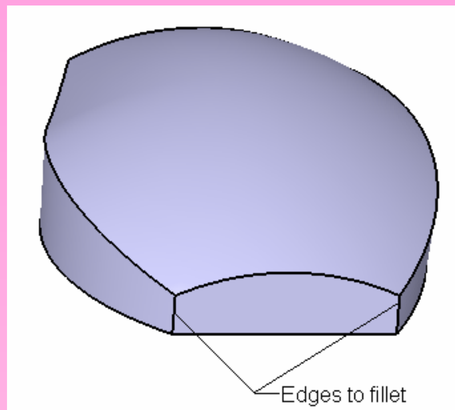


Figure K Edges to be filleted

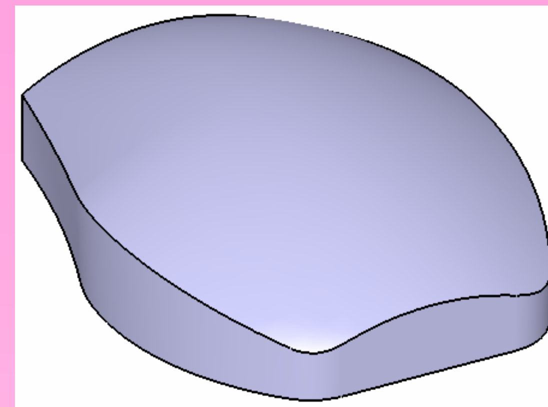


Figure L Model after filleting the edges

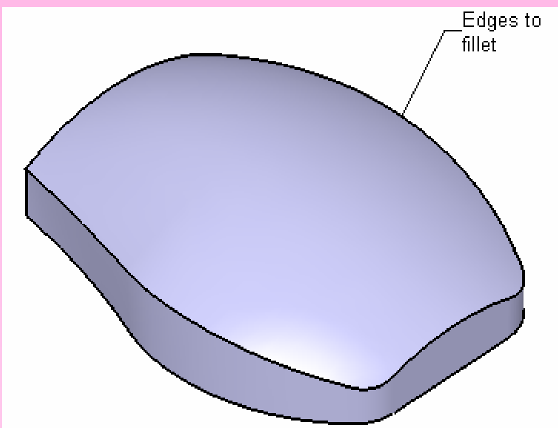


Figure M Edge to be filleted

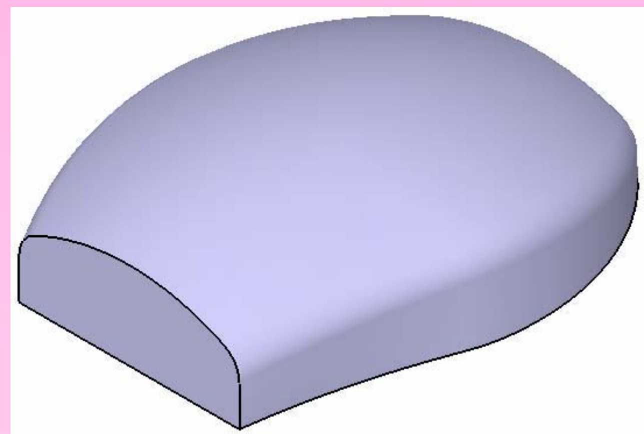


Figure N The model after filleting the selected edge

4. Shell the model by removing the front and the bottom face of the model, as shown in **Figure O**.

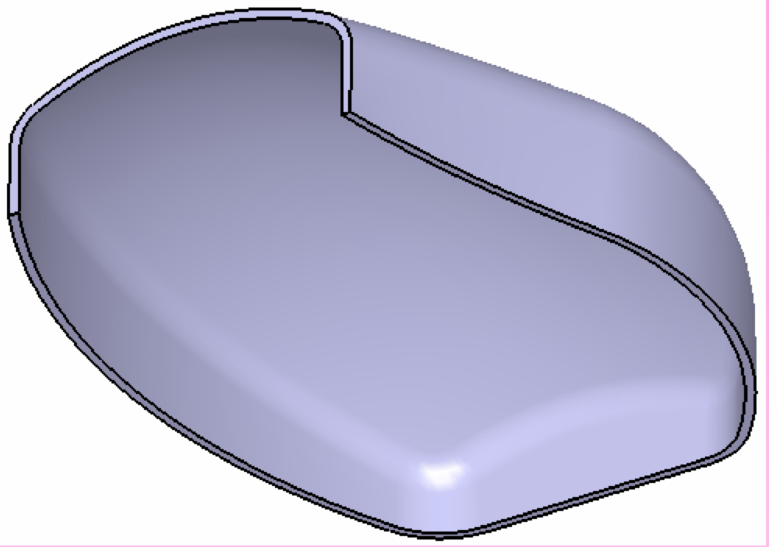


Figure O Final model

5. Save the file in **\My Documents\CATIA\c08** folder and then close it.

Exercise 1

Create the model of the Angle Flange shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

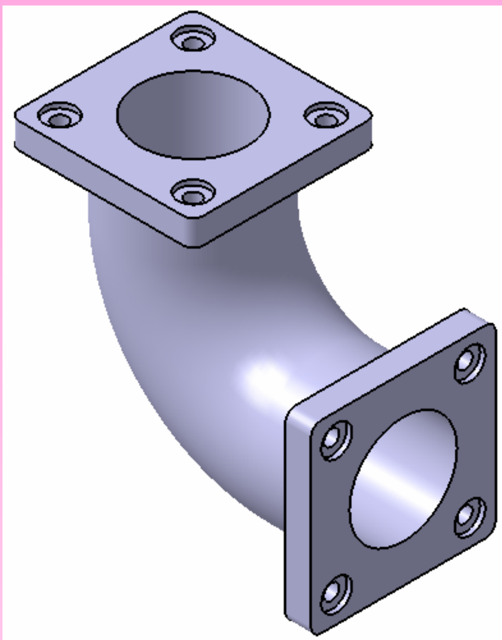


Figure A Model of the Angle Flange for Exercise 1

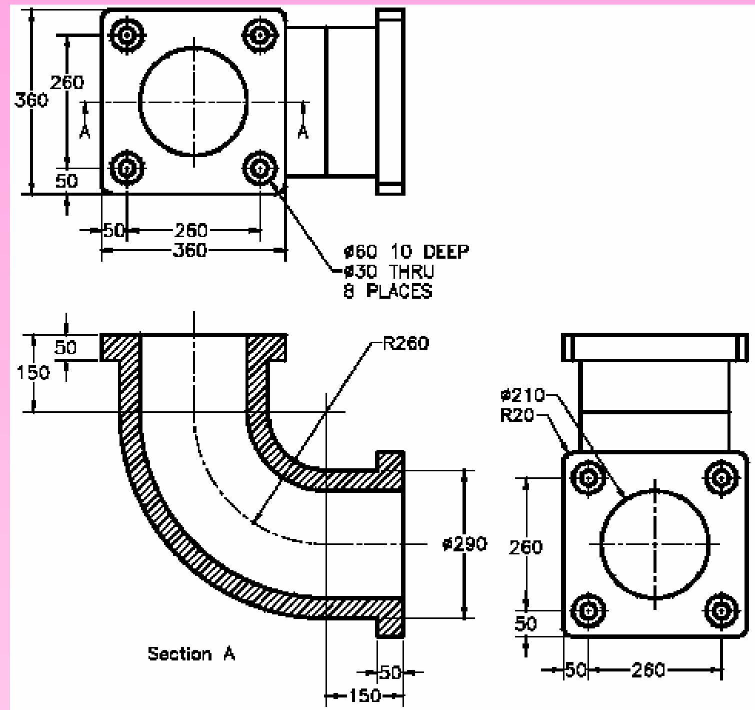


Figure B Views and dimensions for Exercise 1

Exercise 2

Create the model of the Carburetor Cover shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

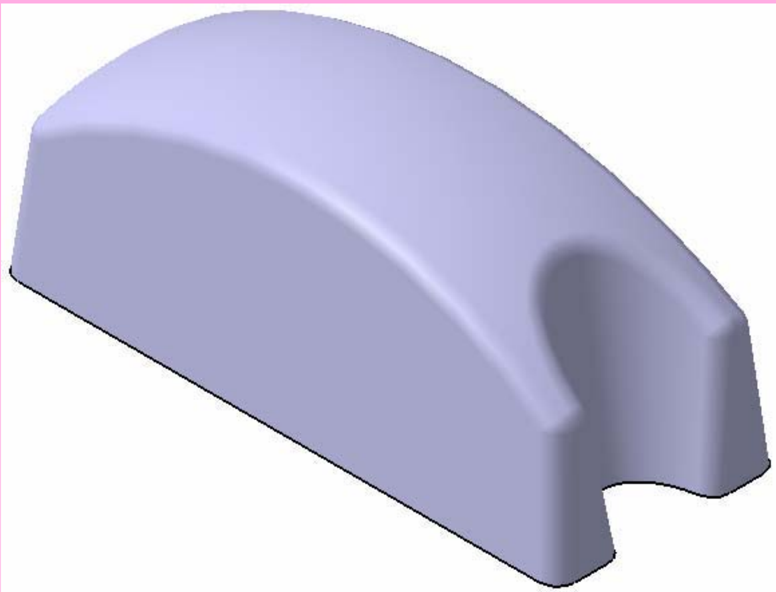


Figure A Model of Carburetor Cover for Exercise 2

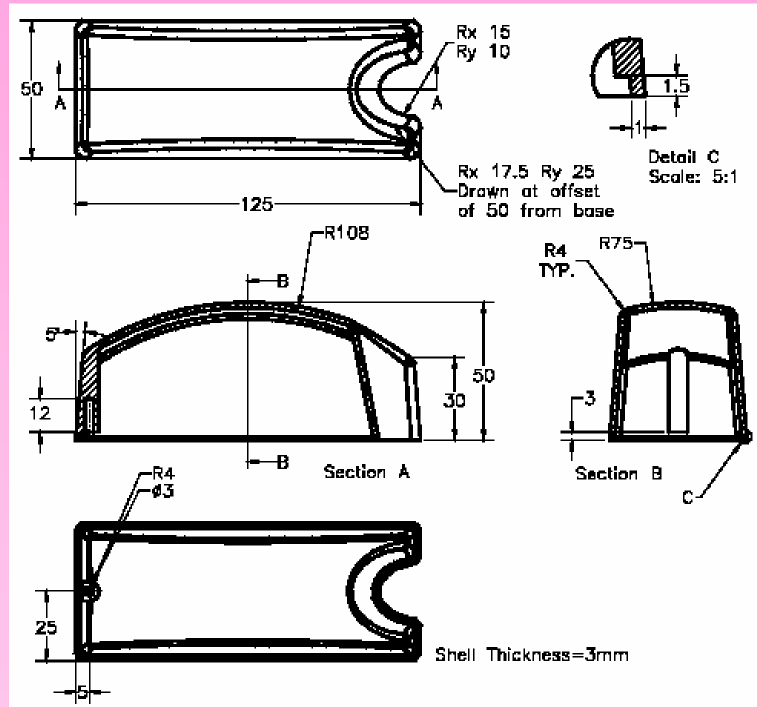


Figure B Views and dimensions for Exercise 2

Learning Objectives:

- *Create wireframe geometry.*
- *Create extruded surfaces.*
- *Create revolved surfaces.*
- *Create spherical surfaces.*
- *Create offset surfaces.*
- *Create swept surfaces.*
- *Create fill surfaces.*
- *Create multisections surfaces.*
- *Create blend surfaces.*
- *Split surfaces.*
- *Trim surfaces.*
- *Join surfaces.*

NEED OF SURFACE MODELING

- The product and industrial designers give importance to product styling and providing a unique shape to components.
- Generally, this is done to make sure that the product looks attractive and presentable.
- Surface models are three-dimensional models with no thickness and unlike solid models, they do not have mass properties.

➢ WIREFRAME AND SURFACE DESIGN WORKBENCH

The **Wireframe and Surface Design** workbench provides the tools to create wireframe construction elements during preliminary design and enrich existing 3D mechanical part design with wireframe and basic surface features.

• Starting Wireframe and Surface Design Workbench

- Start a new session of CATIA V5 and close the new product file, which is opened by default.

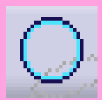
- Next, choose **Start > Mechanical Design > Wireframe and Surface Design** from the menu bar to start a new file in the **Wireframe and Surface Design** workbench.

➢ CREATING WIREFRAME ELEMENTS

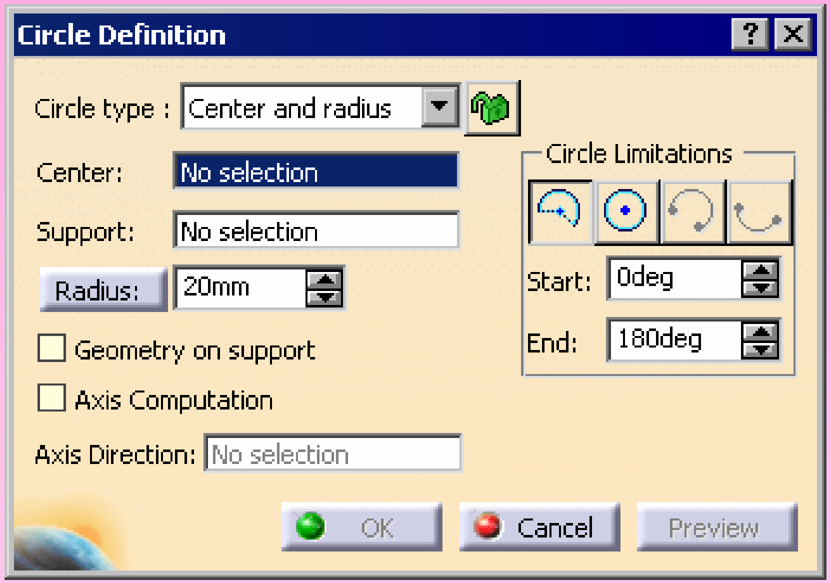
- The wireframe construction elements aid in creating surfaces.

- The sketches drawn in sketcher workbench can also be used to create surfaces.

• Creating Circles

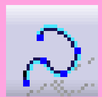


Choose the **Circle** button from the **Wireframe** toolbar; the **Circle Definition** dialog box is displayed, as shown in the figure.



The Circle Definition dialog box

• Creating Splines



- Choose the down arrow on the right of the **Spline** button to invoke the **Curves** toolbar, as shown in **Figure A**, and then choose the spline button.
- The **Spline Definition** dialog box is displayed, as shown in the **Figure B**.

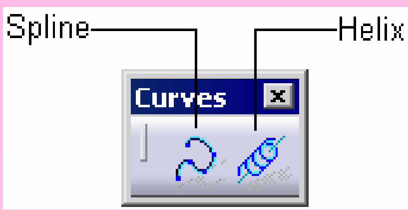


Figure A The Curve toolbar

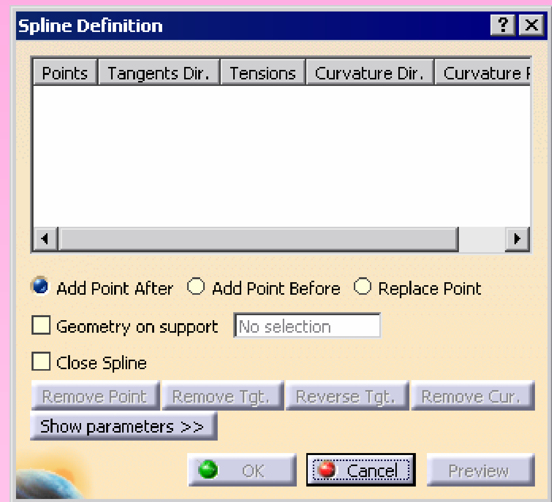
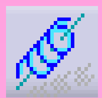
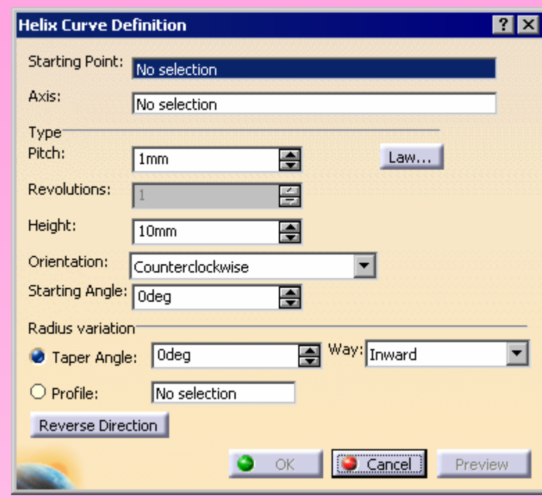


Figure B The Spline Definition dialog box

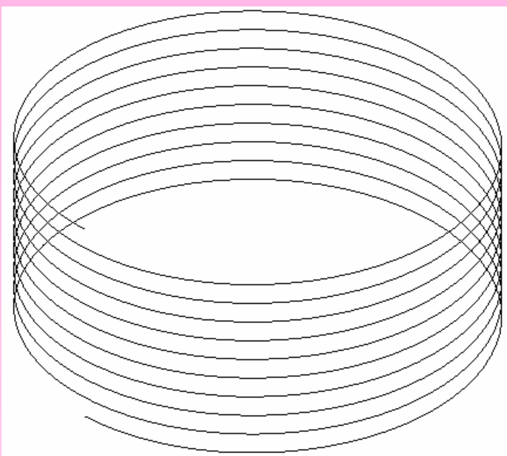
• Creating Helix



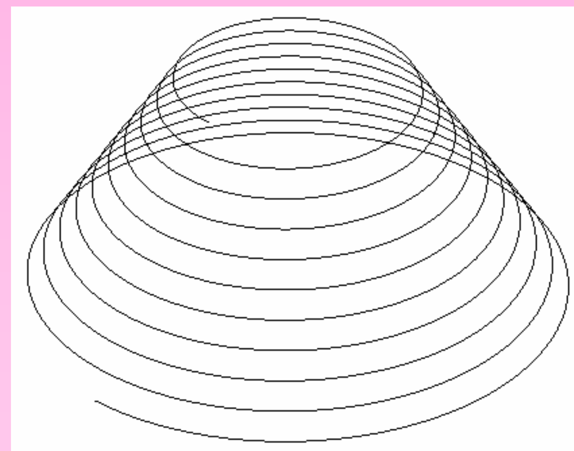
When you invoke the **Helix** tool, the **Helix Curve Definition** dialog box will be displayed, as shown in the figure.



The Helix Curve Definition dialog box



The helix without specifying the taper angle

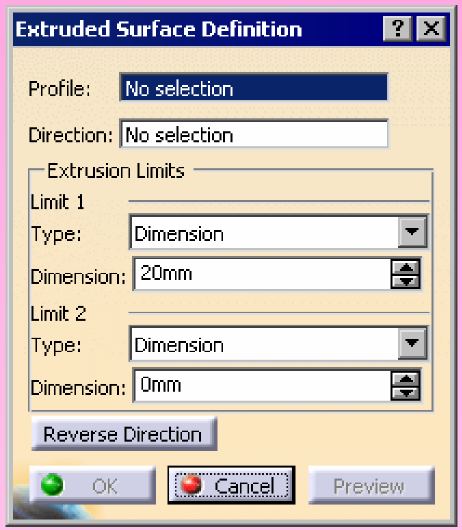


The helix with a specified taper angle

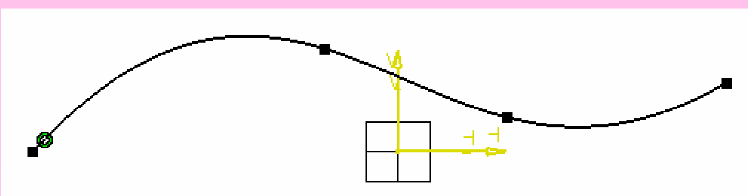
CREATING SURFACES

• Creating Extruded Surfaces

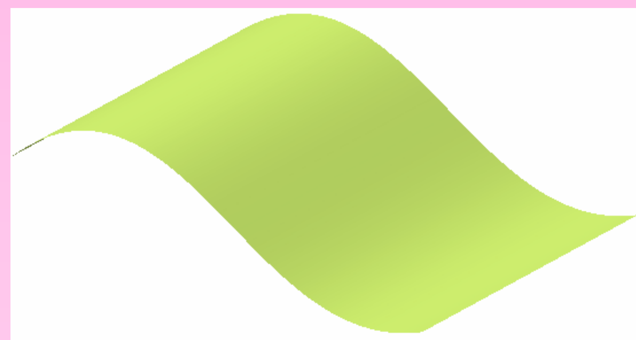
Choose the **Extrude** button from the **Surfaces** toolbar; the **Extrude Surface Definition** dialog box is displayed, as shown in the figure.



The Extruded Surface Definition dialog box

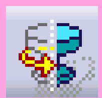


The profile to be extruded

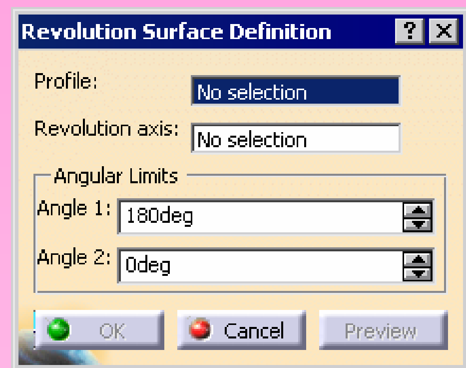


The resulting extruded surface

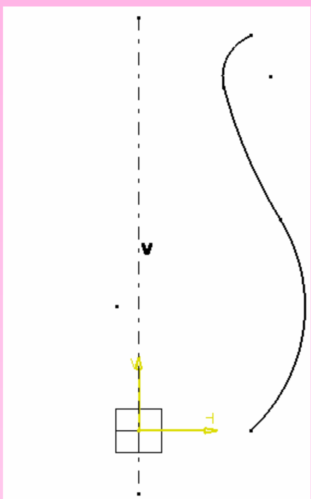
• Creating Revolved Surfaces



Choose the **Revolve** button from the **Surfaces** toolbar; the **Revolution Surface Definition** dialog box is displayed, as shown in the figure.



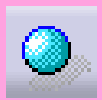
The Revolution Surface Definition dialog box



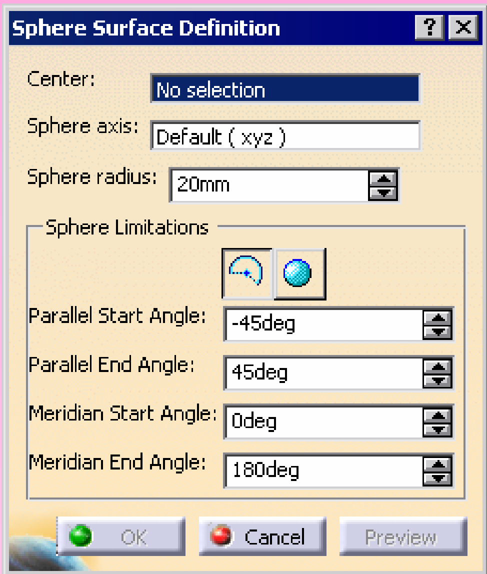
The profile and revolution axis

Surface revolved through an angle of 180-degree

• Creating Spherical Surfaces



When you invoke the **Sphere** tool, the **Sphere Surface Definition** dialog box is displayed, as shown in the figure.



The Sphere Surface Definition dialog box

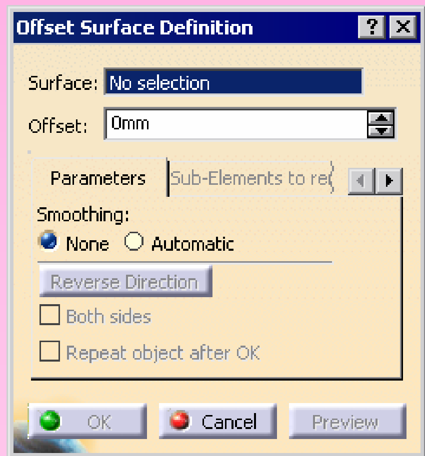
A spherical surface

- ## Creating Cylindrical Surfaces

Choose the **Cylinder** button from the **Surfaces** toolbar; the **Cylinder Surface Definition** dialog box is displayed.

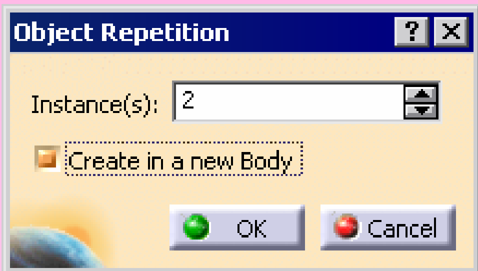
- ## Creating Offset Surfaces

- Choose the **Offset** tool from the **Surfaces** toolbar.
- The **Offset Surface Definition** dialog box is displayed, as shown in the figure.

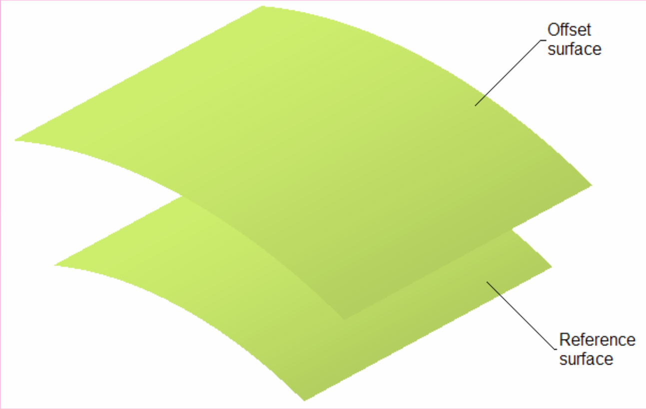


The Offset Surface Definition dialog box

- The **Repeat object after OK** check box is used to create multiple offset surfaces.
- Select the **Repeat object after OK** check box and exit the **Offset Surface Definition** dialog box.
- The **Object Repetition** dialog box is displayed, as shown in the figure.



The Object Repetition dialog box

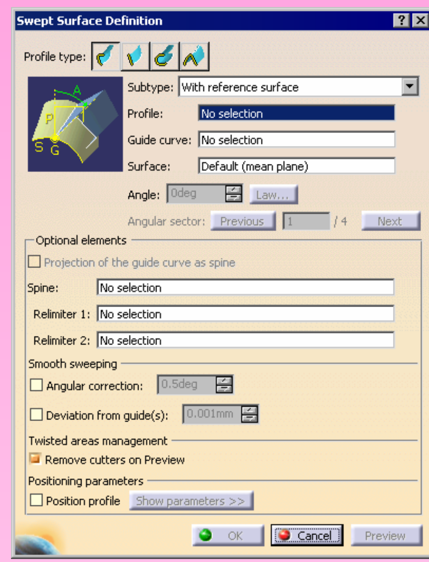


An offset surface

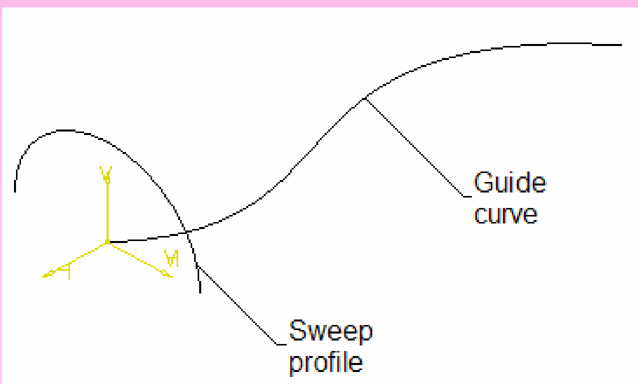
• Creating Swept surfaces

• Choose the **Sweep** button from the **Surfaces** toolbar.

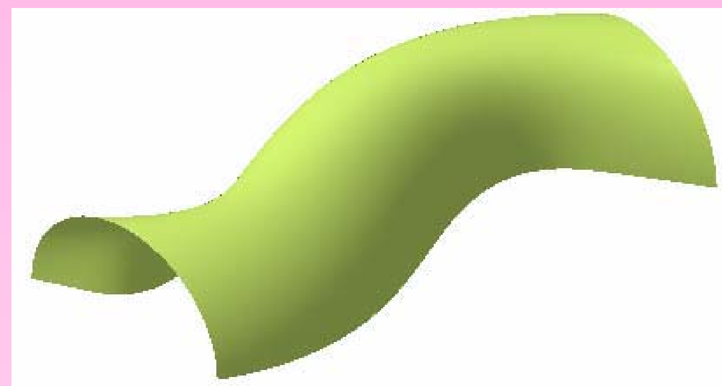
• The **Swept Surface Definition** dialog box is displayed, as shown in the figure.



The Swept Surface Definition dialog box

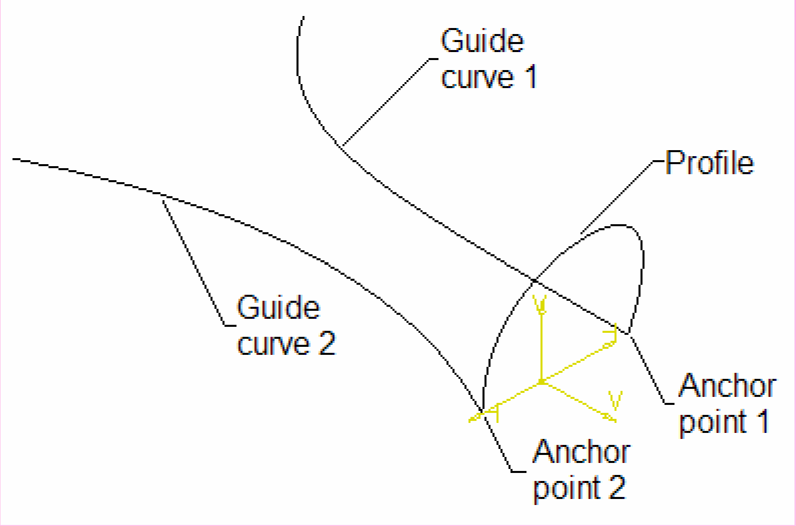


The sweep profile and guide curve

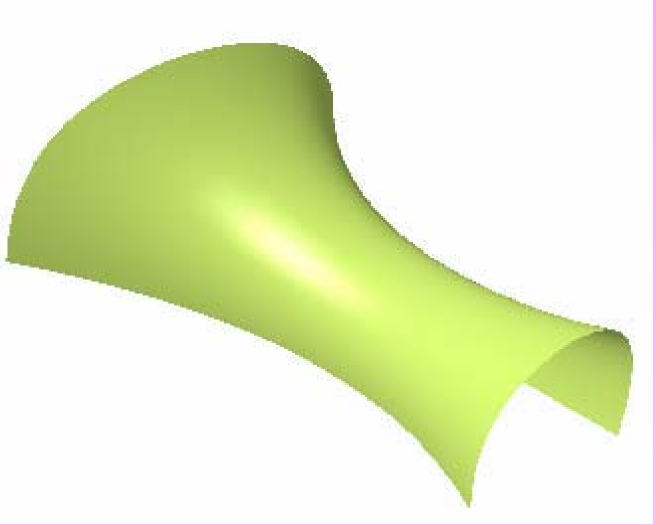


The resulting swept surface

- **Swept Surface with two Guide Curves**



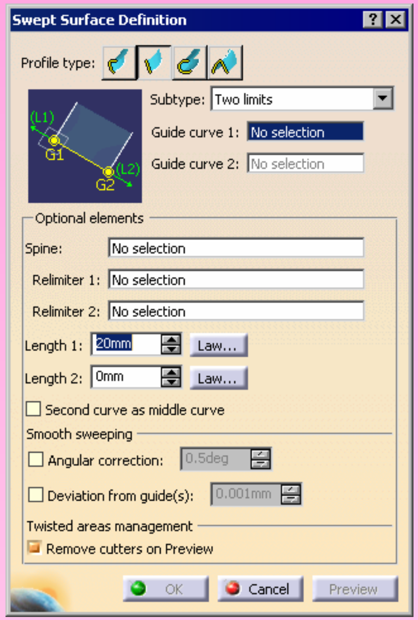
The sweep profile and guide curves



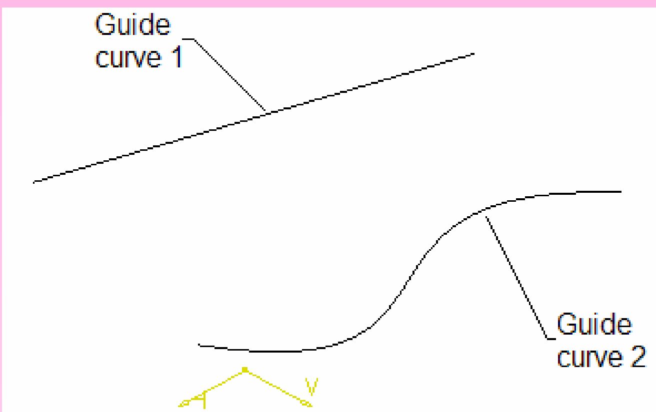
The resulting swept surface

• Swept Surface with Two Limits

- To create a swept surface with two limits, you need to draw two limit curves.
- Next, invoke the **Swept Surface Definition** dialog box, as shown in the figure.



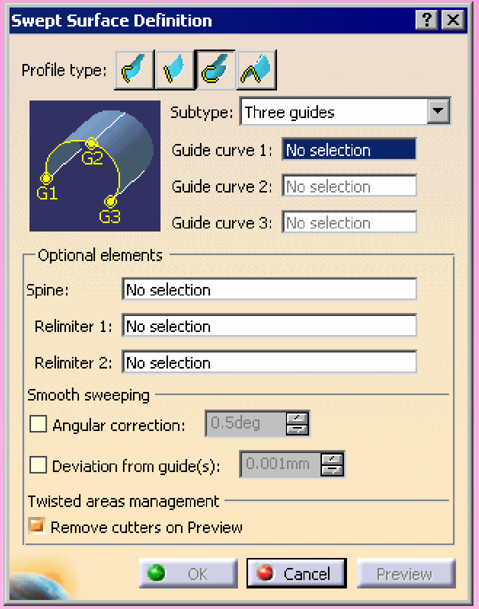
The **Swept Surface Definition** dialog box



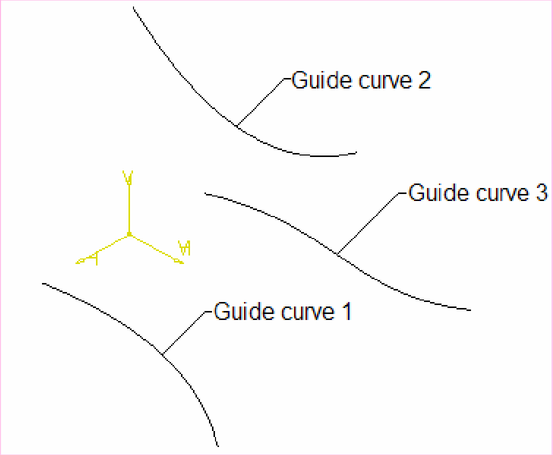
The sweep profile and guide curves

The resulting swept surface

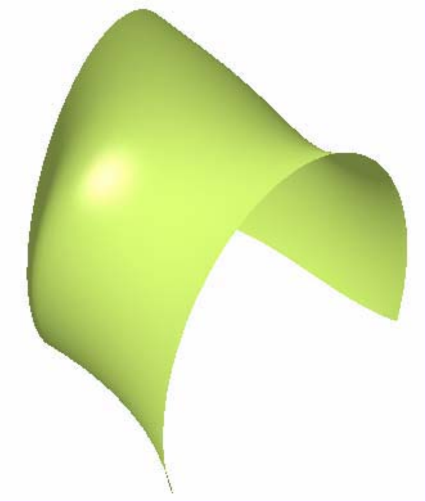
- **Swept Surface with Three Curves**



The **Swept Surface Definition** dialog box



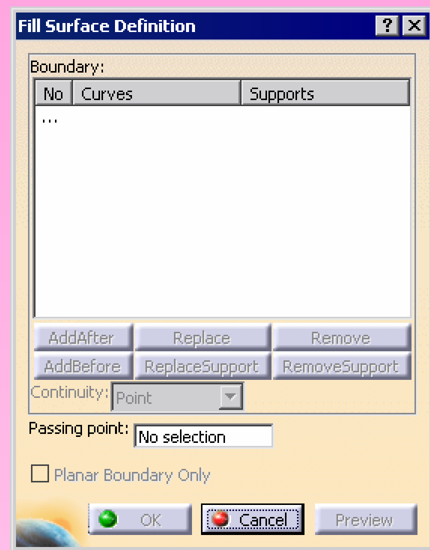
The guide curves



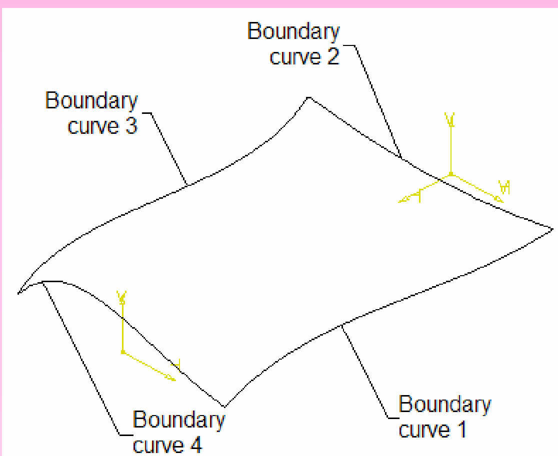
The resulting swept surface

- **Creating Fill Surfaces**

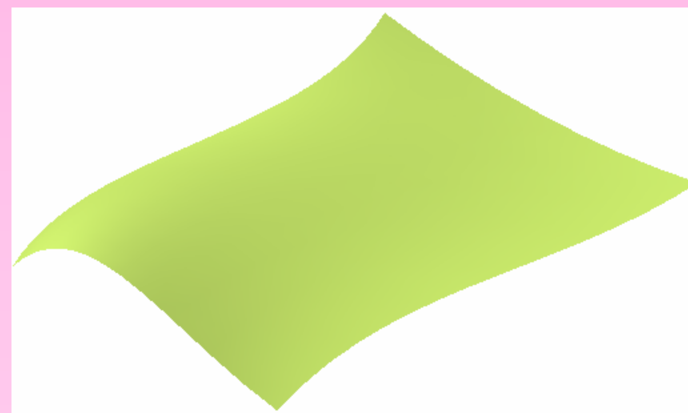
- Choose **Fill** button from the **Surface** toolbar.
- The **Fill Surface Definition** dialog box is displayed, as shown in the figure.



The **Fill Surface Definition** dialog box

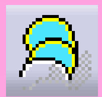


The boundary curves

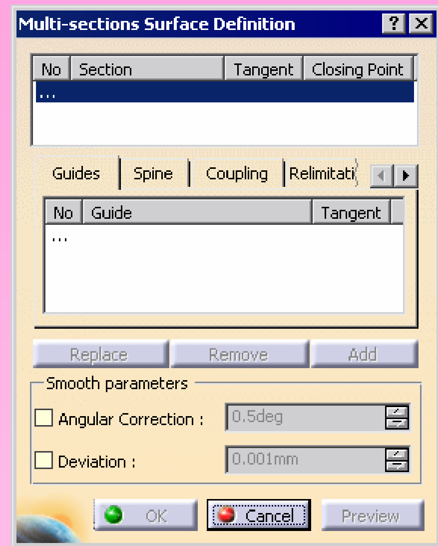


The resulting fill surface

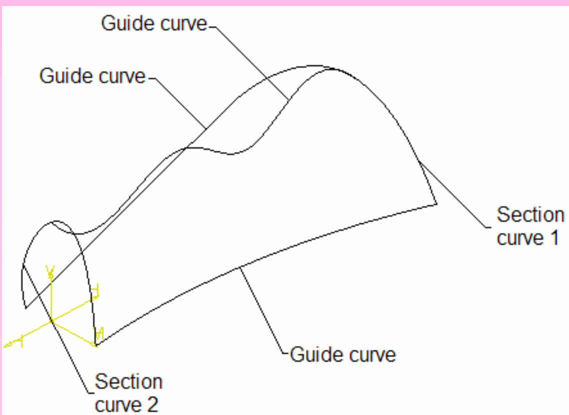
• Creating Multisections Surfaces



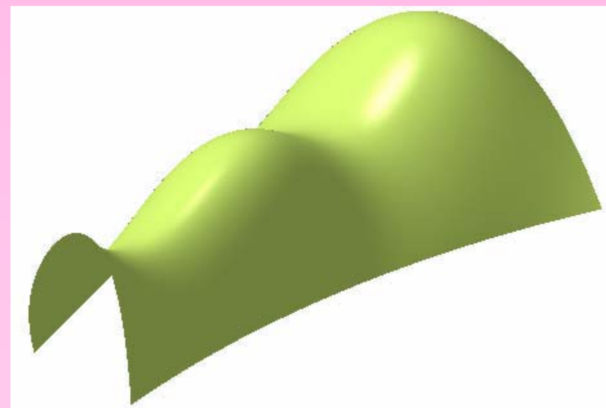
Choose the **Multi-sections surface** button from the **Surface** toolbar; the **Multi-sections Surface Definition** dialog box is displayed, as shown in the figure.



The Multi-sections Surface Definition dialog box



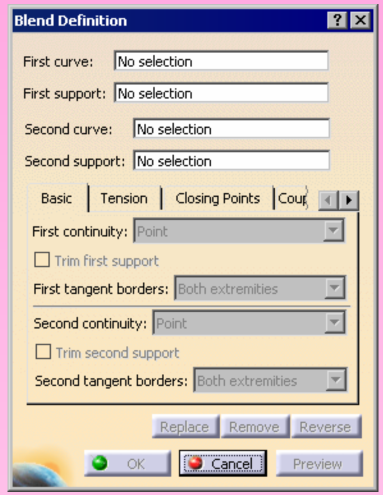
Sections and guide curves



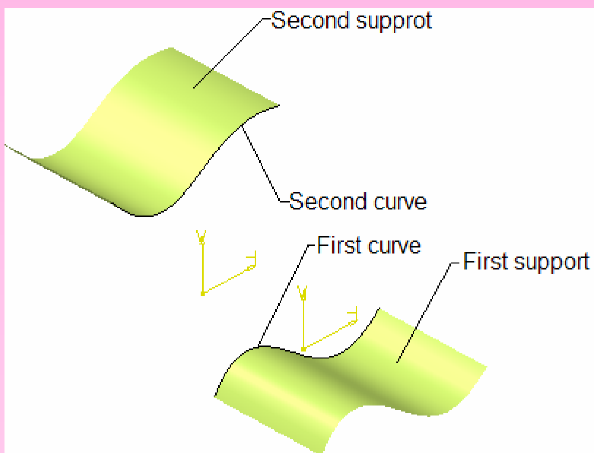
The resulting multisection surface

• Creating Blended Surfaces

- To create a blended surface, draw some curves and create support surfaces.
- Choose the **Blend** button from the **Surfaces** toolbar.
- The **Blend Definition** dialog box is displayed, as shown in the figure.



The Blend Definition dialog box



The curves and support surfaces

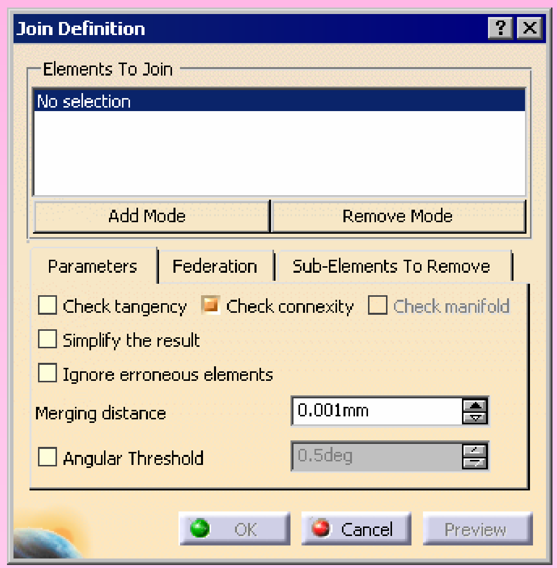
The resulting blend surface

OPERATION ON THE SHAPE GEOMETRY

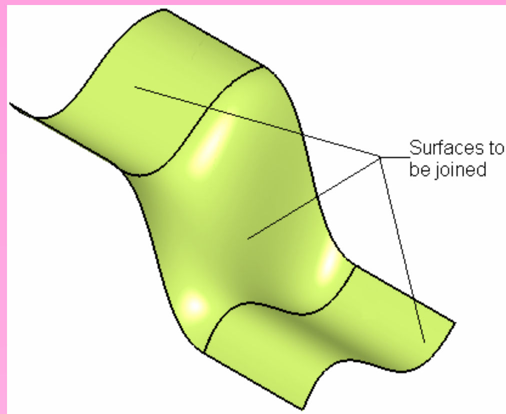
- Generally, the surface models are a combination of various surfaces.
- You need to join, trim, split, or translate the surfaces to manage multiple surfaces.
- CATIA V5 provides a number of such operation tools that can be used on the surfaces created using the tools discussed earlier in this chapter.

Joining Surfaces

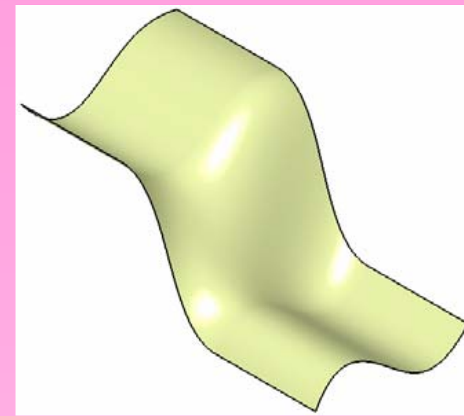
When you invoke the **Join** tool; the **Join Definition** dialog box is displayed, as shown in the figure.



The Join Definition dialog box

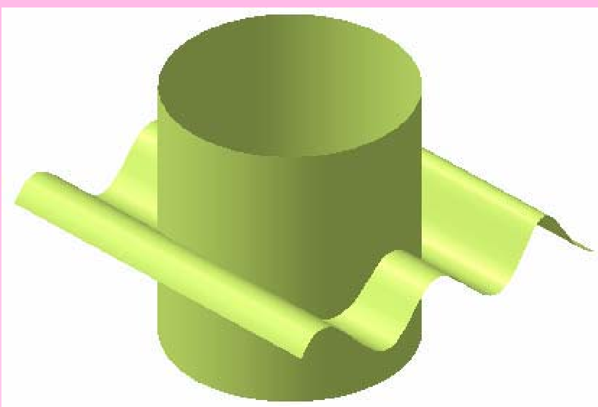


Surfaces to be joined



Resulting joint surface

- **Spliting Surfaces**

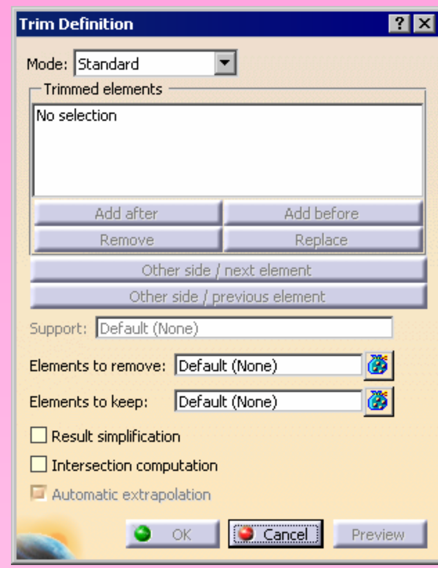


The split surface and the cutting surfaces

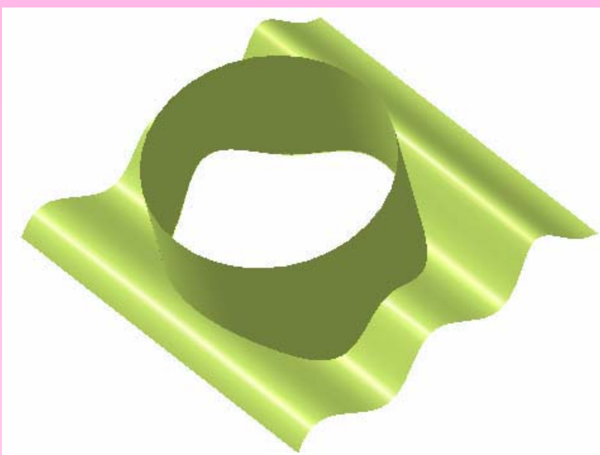
The resulting split surface

• Trimming Surfaces

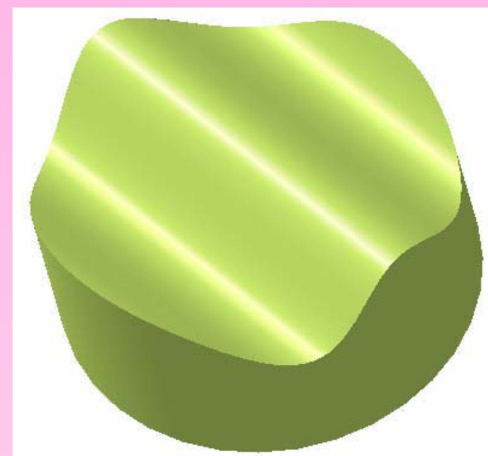
When you invoke the **Trim** tool, the **Trim Definition** dialog box will be displayed, as shown in the figure.



The Trim Definition dialog box



The trimmed surfaces



The trimmed surface with the Other side of element buttons selected

❑ Tutorial 1

In this tutorial, you will create the model shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

Figure A The isometric view of the model

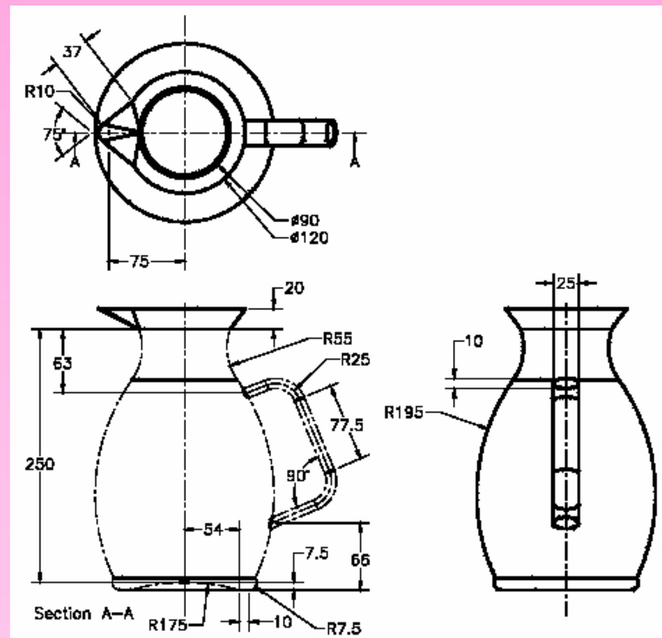


Figure B The views and dimensions of the model

1. Start CATIA V5 and start a new file in the **Wireframe and Surface Design** workbench.
2. Draw the sketches for the multisections surface, as shown in **Figure C**, **Figure D** and **Figure E**.

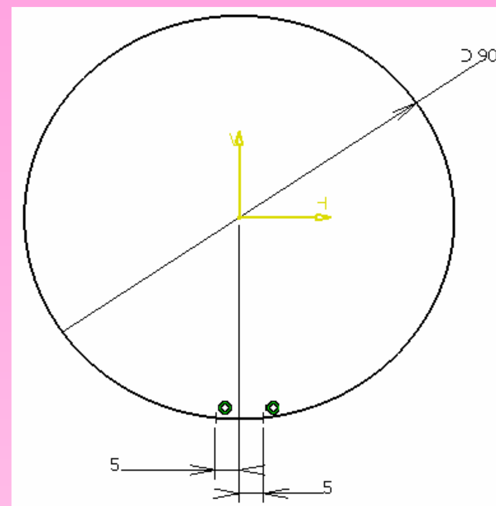


Figure C First section for the base surface

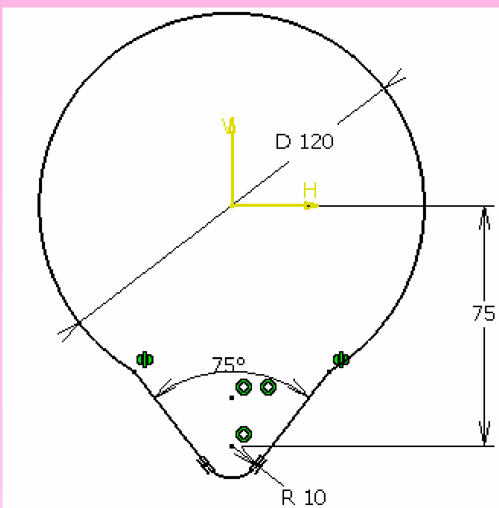


Figure D The second section for base surface

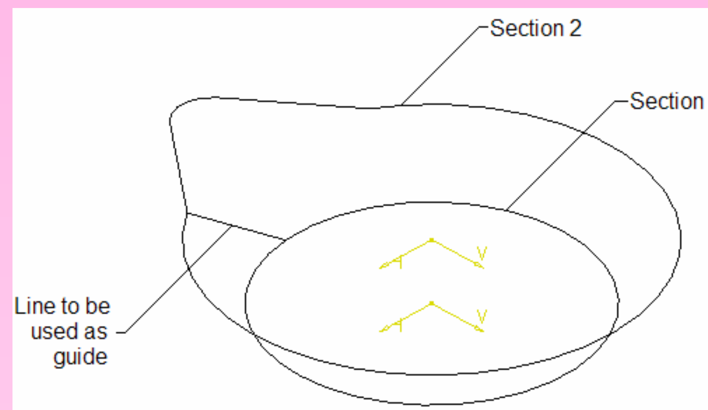


Figure E The line created between two points on the sections

3. Create the multisections surface, as shown in **Figure F** and **Figure G**.

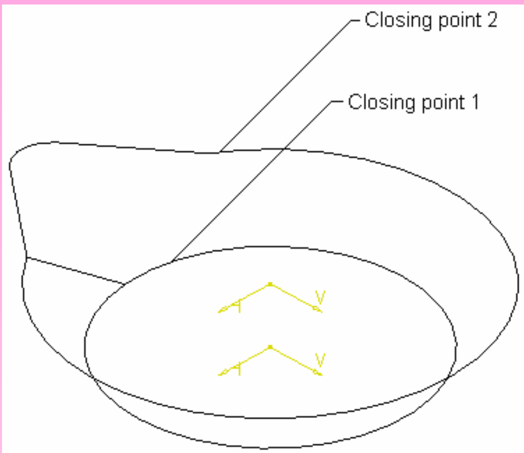


Figure F The sketch showing the position of closing points

Figure G The multisection lofted surface

4. Draw the sketch to create the revolve surface, as shown in **Figure H**.

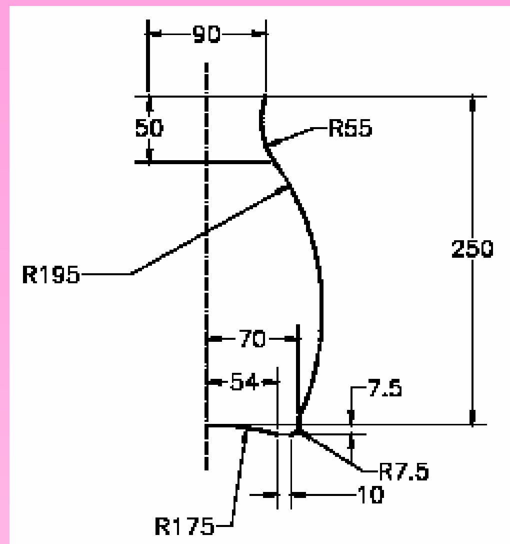


Figure H The axis and profile to be revolved

5. Create the revolved surface, as shown in **Figure I**.

Figure I The model after creating the revolved surface

6. Draw the sketch to create the sweep profile, as shown in **Figure J** and **Figure K**.

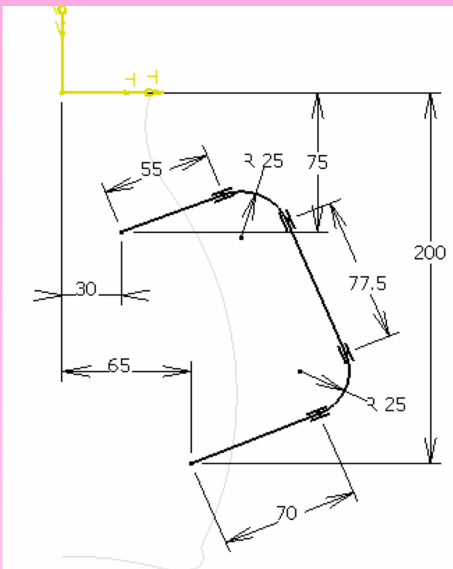


Figure J **The guide for the swept surface**

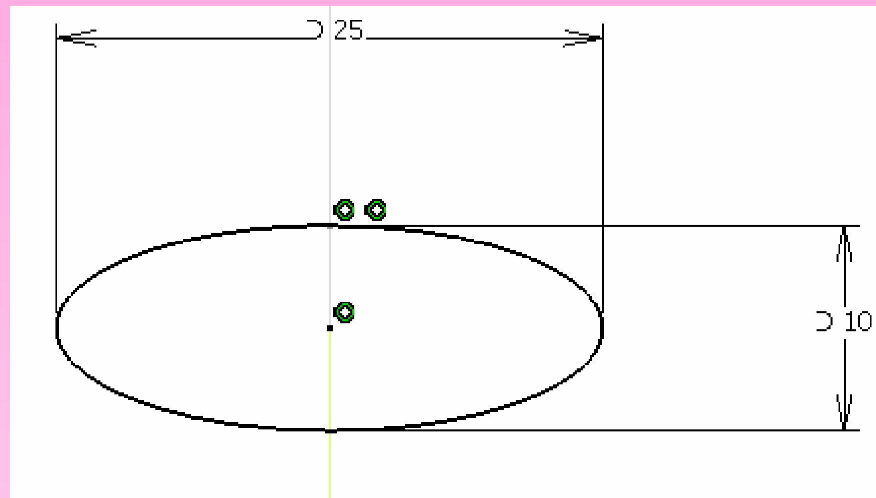


Figure K **The profile for the swept surface**

7. Create the swept surface.

8. Split the swept surface with the revolved surface, as shown in **Figure L**.

Figure L The isometric view of the model after splitting the swept surface

9. Save the file in **\My Documents\CATIA\c09** folder and then close it.

❑ Tutorial 2

In this tutorial, you will create the model shown in **Figure A**. Its views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

Figure A The isometric view of the model

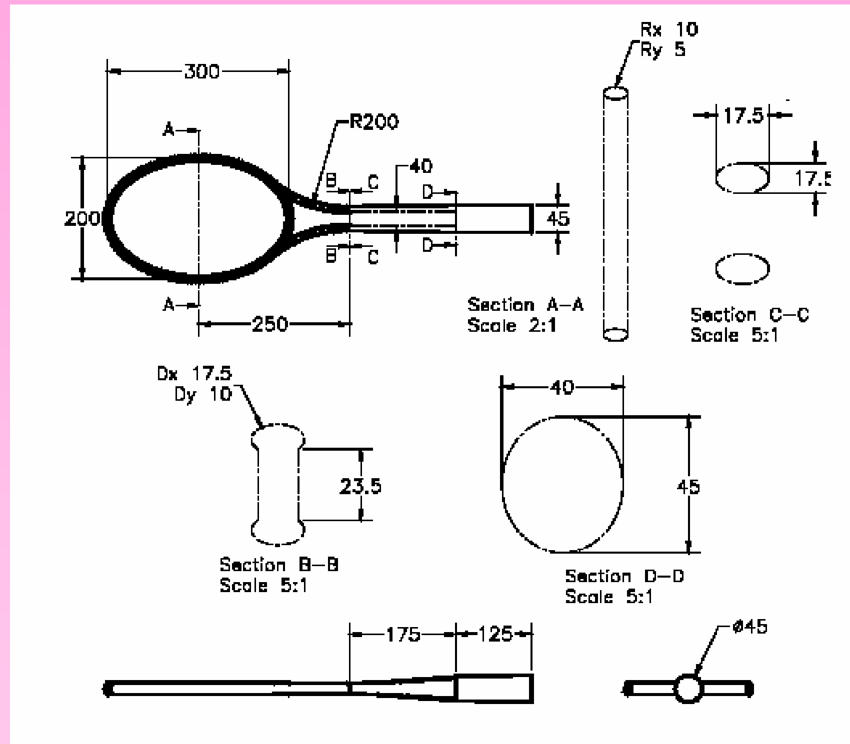


Figure B The views and dimensions of the model

1. Start a new file in the **Wireframe and Surface Design** workbench.
2. Create a swept surface as the base feature, as shown in **Figure C**, **Figure D** and **Figure E**.

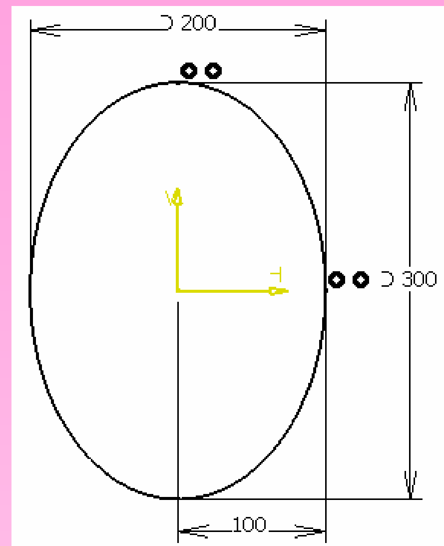


Figure C The guide curve

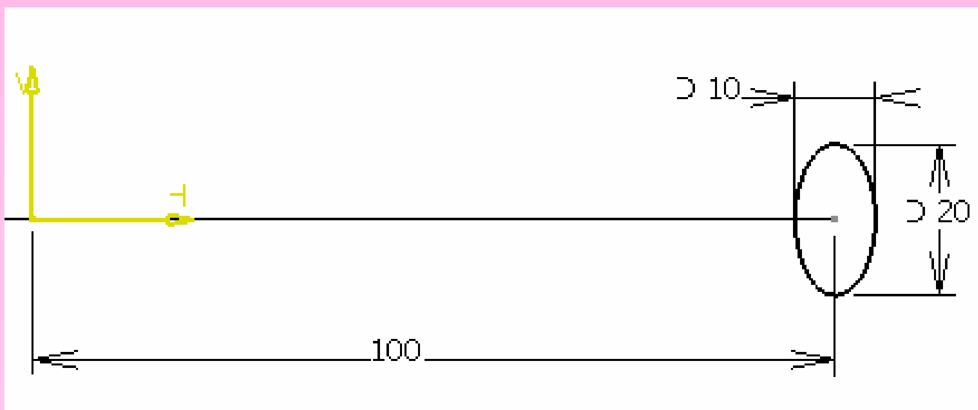


Figure D The sweep profile

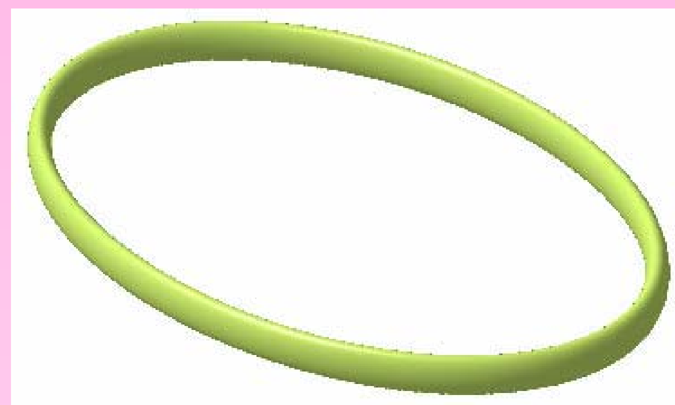


Figure E The resulting swept surface

3. Create the second swept surface, as shown in **Figure F**, **Figure G** and **Figure H**.

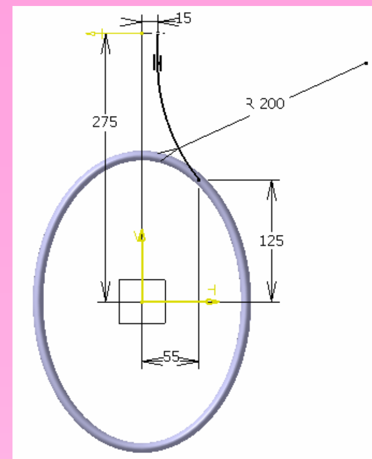


Figure F The sketch of the guide curve for creating the second sweep feature

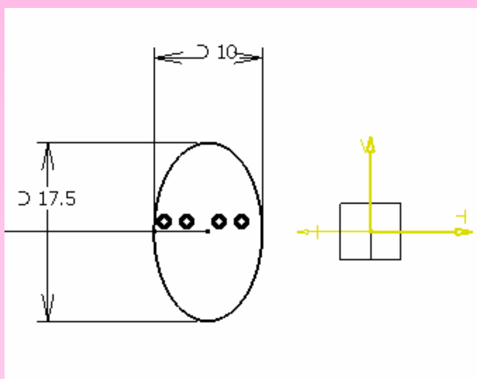


Figure G The sketch of the sweep profile for creating the second sweep feature

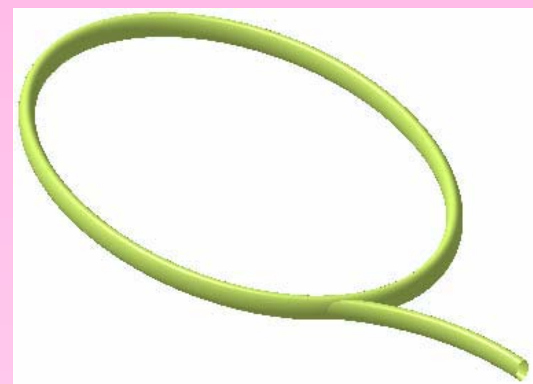


Figure H The model after creating the second sweep feature

4. Create the symmetry of the second swept surface, as shown in **Figure I**.

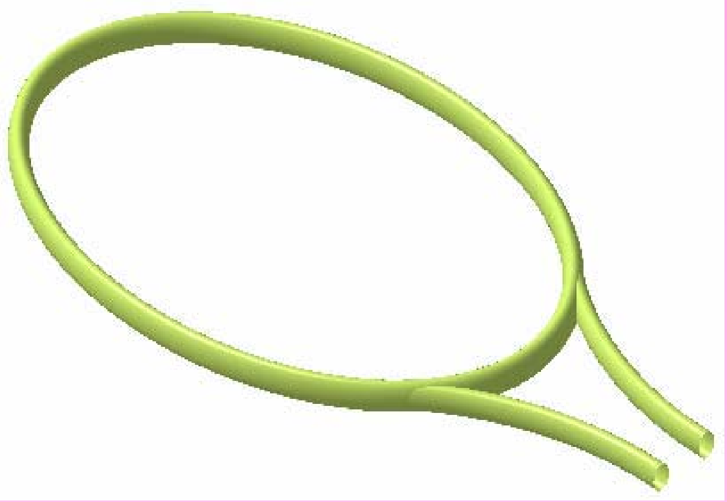


Figure I The model after creating the symmetry feature

5. Create the multisection surface, as shown in **Figure J** through **Figure M**.

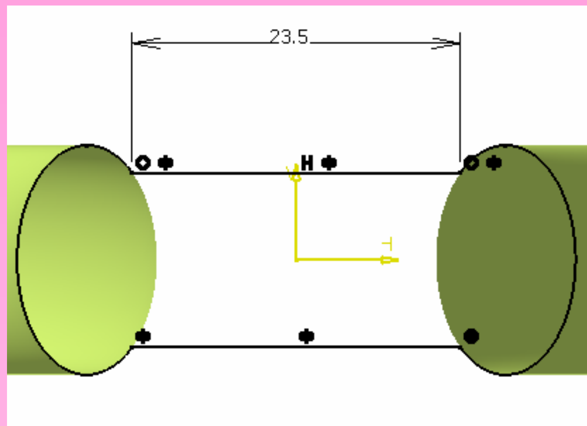


Figure J The first section for creating the multisection surface

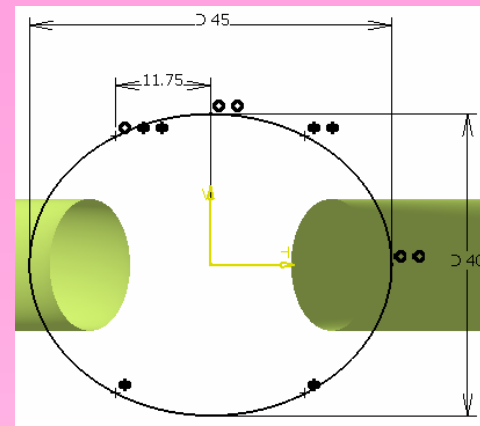


Figure K The second section for creating the multisection surface

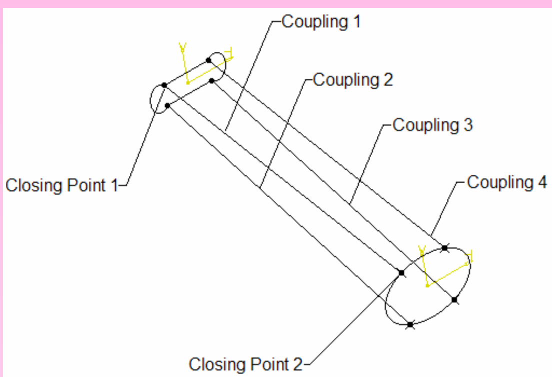


Figure L The couplings and closing points

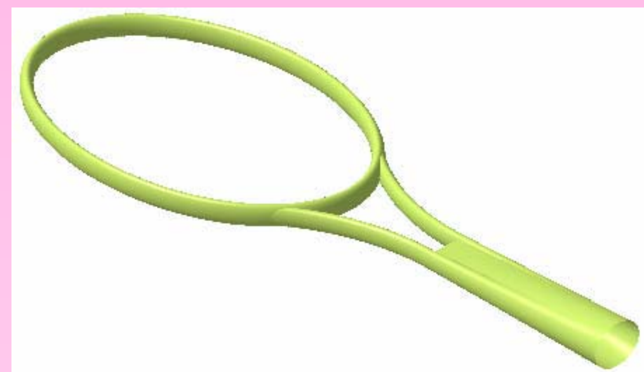


Figure M The model after creating the multisection surface

6. Create the blended surface, as shown in **Figure N** and **Figure O**.

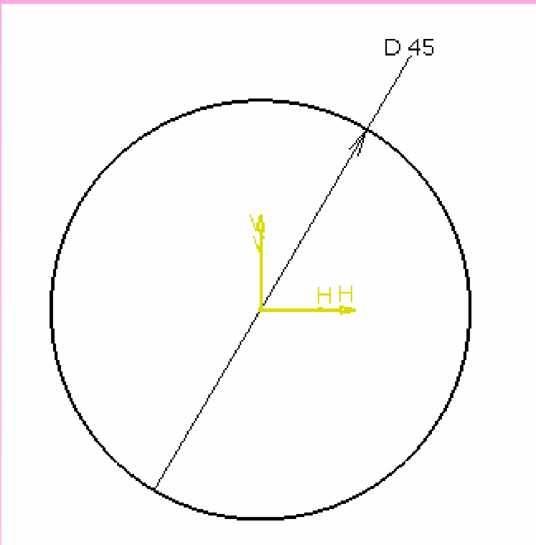


Figure N **The sketch of the blended section**

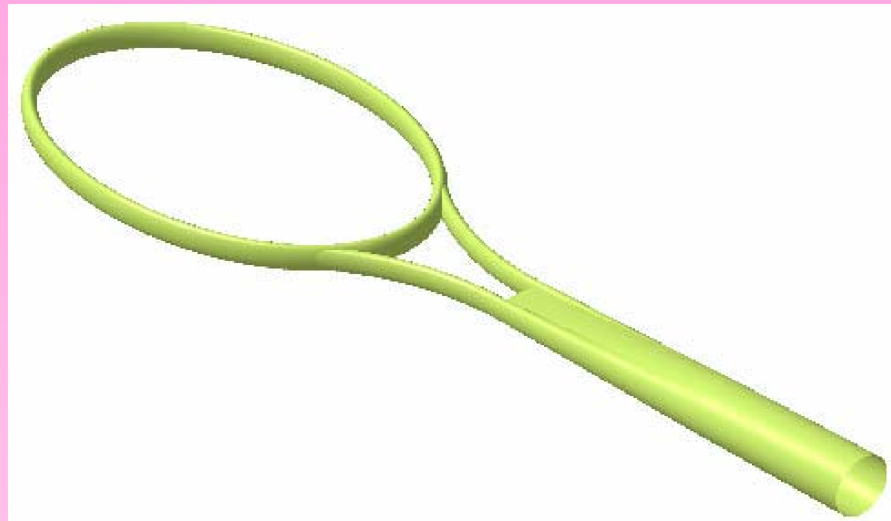


Figure O **The model after creating the blend surface**

7. Create the fill surfaces, as shown in **Figure P**.

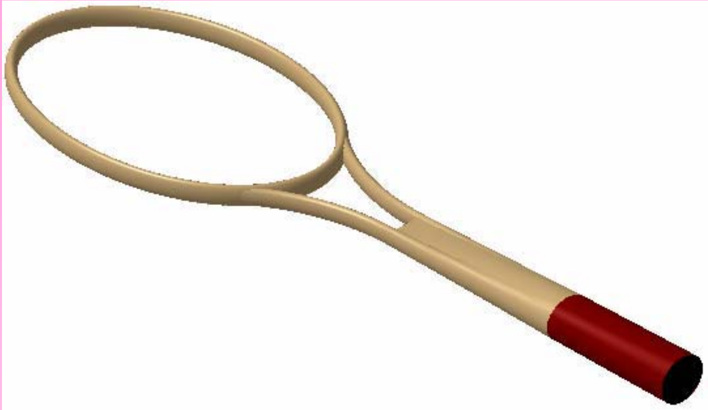


Figure P The final model

8. Save the file in **\My Documents\CATIA\c09** folder and then close it.

Exercise 1

In this exercise, you will create the surface model shown in **Figure A**. Its orthographic views are shown in **Figure B**. **(Expected time: 30 min)**

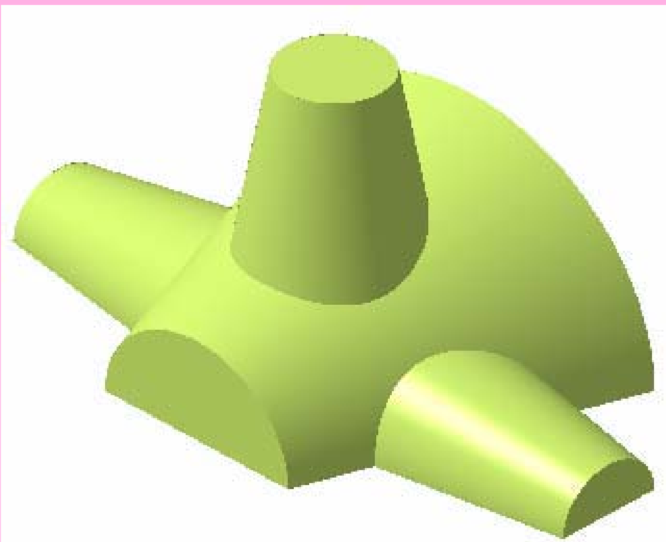


Figure A The isometric view of the model

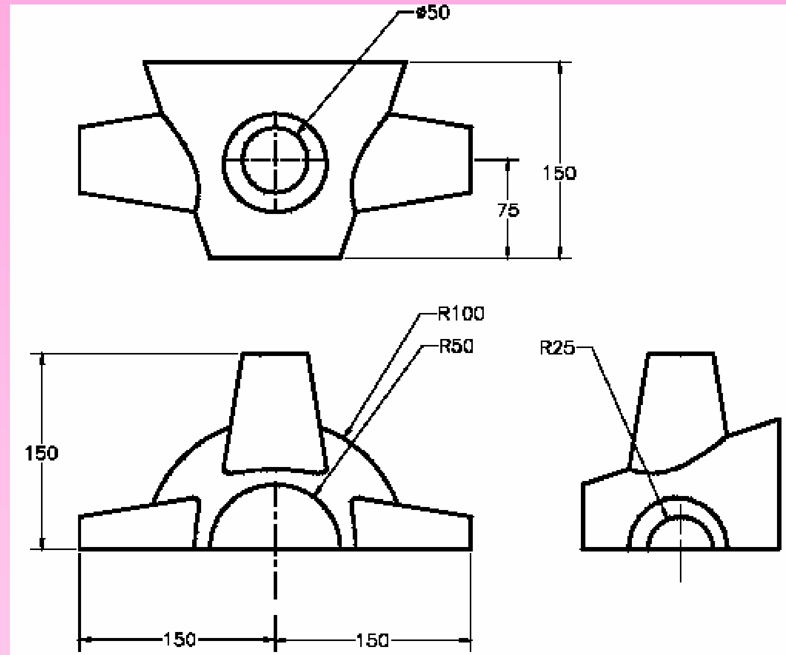


Figure B The views of the model

Exercise 2

In this exercise, you will create the surface model shown in **Figure A**. Its orthographic views are shown in **Figure B**. **(Expected time: 45 min)**

Figure A The isometric view of the model

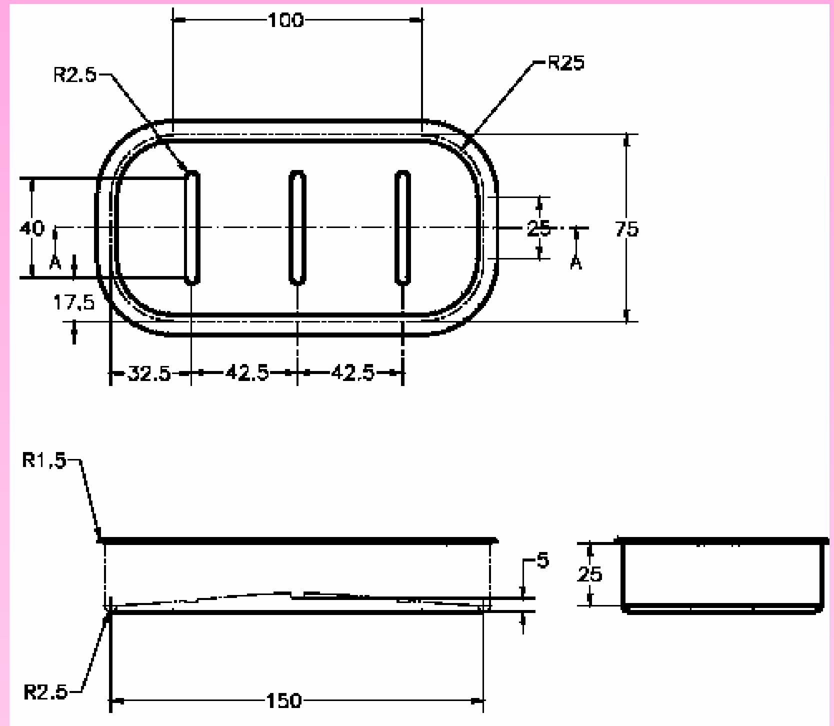


Figure B The views of the model

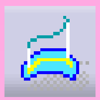
Learning Objectives:

- *Create projection elements.*
- *Create intersections.*
- *Heal geometries.*
- *Restore surfaces.*
- *Extract elements.*
- *Create boundaries.*
- *Transform features.*
- *Create curves or surfaces by extrapolation.*

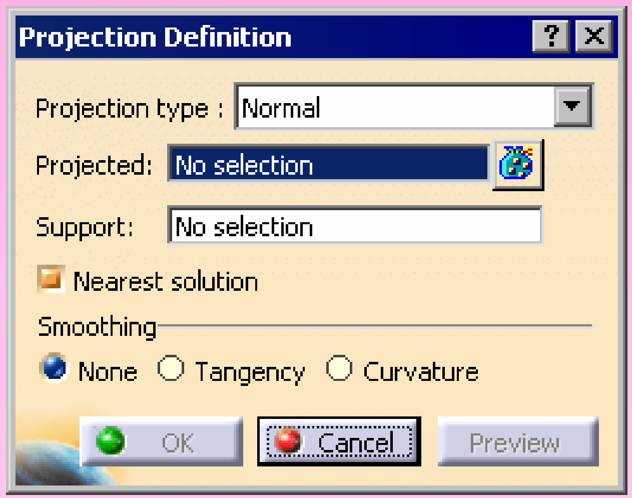
➢ SURFACE OPERATIONS

CATIA V5 also provides you with various tools that are used to modify surfaces. These are known as surface operation tools and are used regularly while creating surface models.

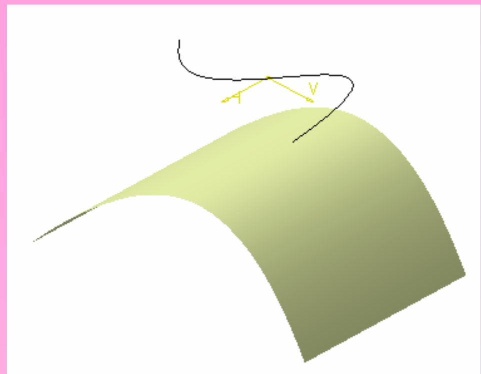
• Creating Projection Curves



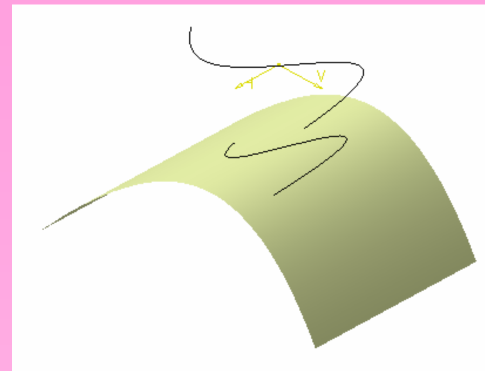
Choose the **Projection** button from the **Wireframe** toolbar; the **Projection Definition** dialog box is displayed, as shown in the figure.



The Projection Definition dialog box



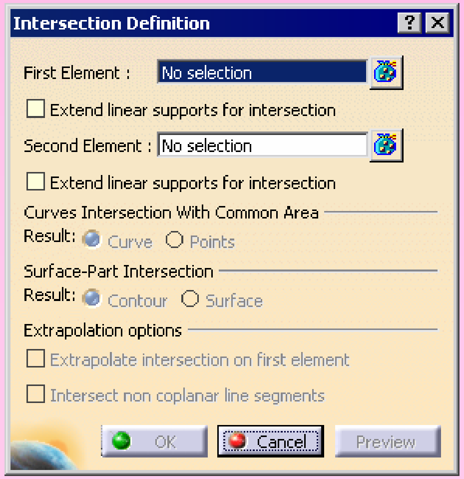
The curve and the support surface



The resulting projected element

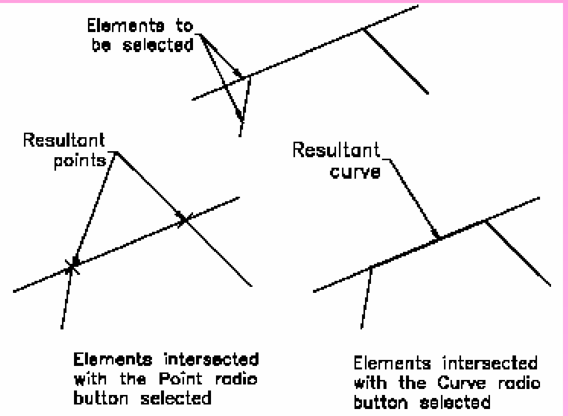
- **Creating Intersection Elements**

When you invoke this tool, the **Intersection Definition** dialog box is displayed, as shown in the figure.



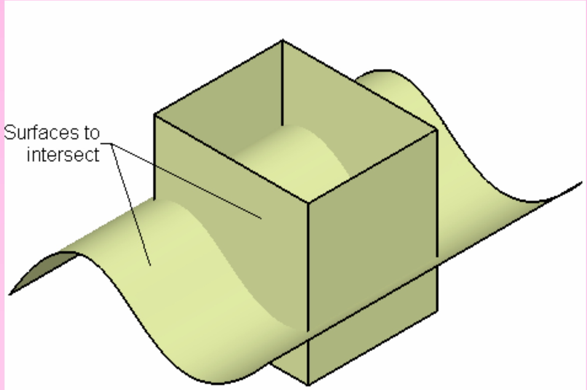
The **Intersection Definition** dialog box

- Intersection of Two Curves

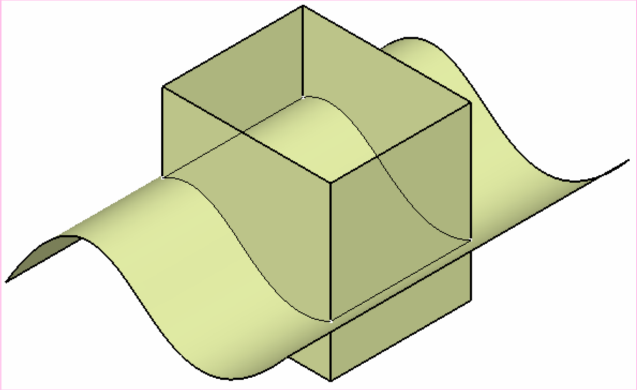


Elements to be intersected and the resulting intersections

- Intersection of Two Surfaces



Surfaces to be selected

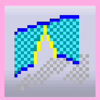


Resulting intersection curve

• Intersection of a Solid and a Surface

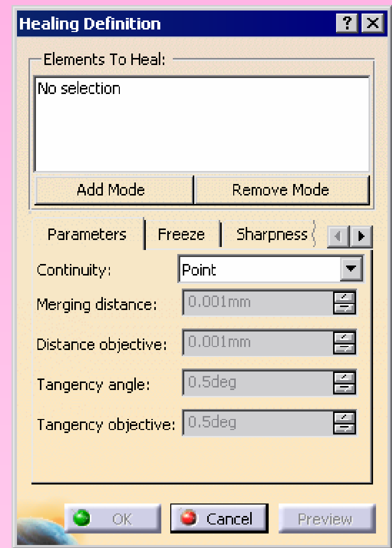
When a solid and a surface are intersected, the result could be a contour or a surface, depending on the option selected in the **Surface-Part Intersection** area in the **Intersection Definition** dialog box.

• Healing Geometries

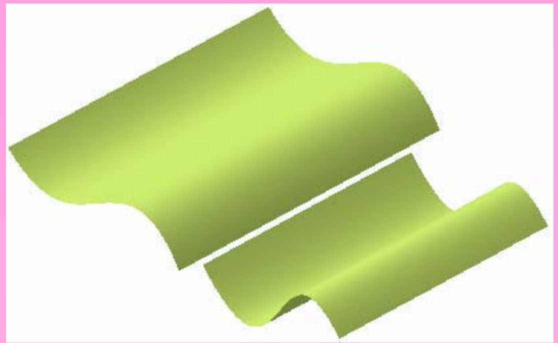


Choose the **Healing** button from the **Join-Healing** toolbar to display the **Healing definition** dialog box, as shown in the figure.

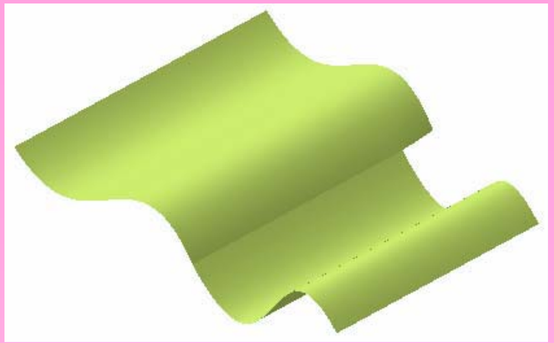
The Join-Healing toolbar



The Healing Definition dialog box

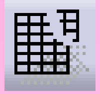


The surface to be healed

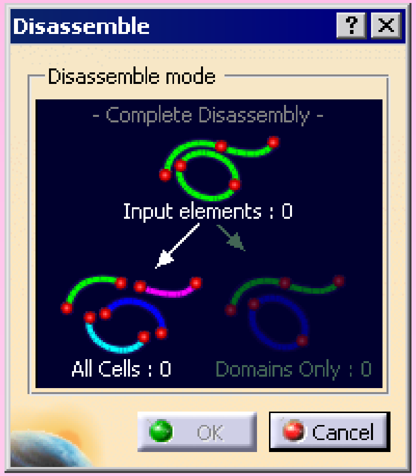


The resulting healed surface

- **Disassembling Elements**



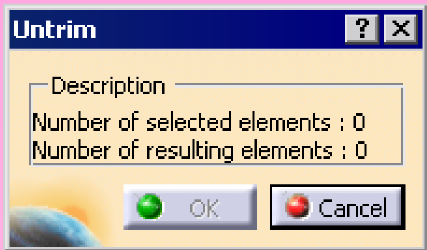
  - To disassemble elements, choose the **Disassemble** button from the **Join-Healing** tool bar.
  - The **Disassemble** dialog box is displayed, as shown in the figure.



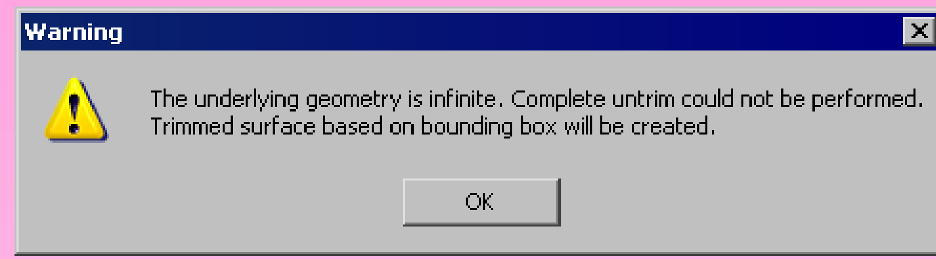
The **Disassemble** dialog box

• Untrimming Surface or Curve

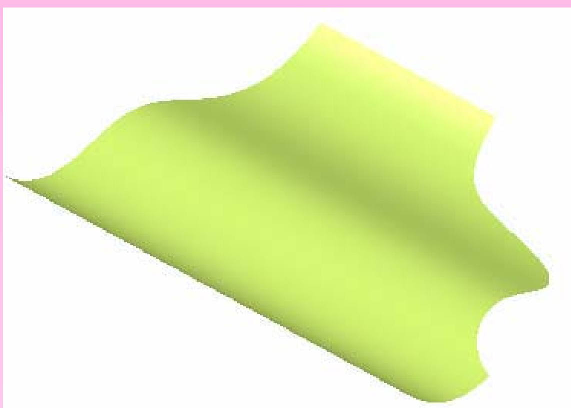
To untrim a surface, choose the **Untrim** tool from the **Join-Healing** toolbar; the **Untrim** dialog box is displayed, as shown in the figure.



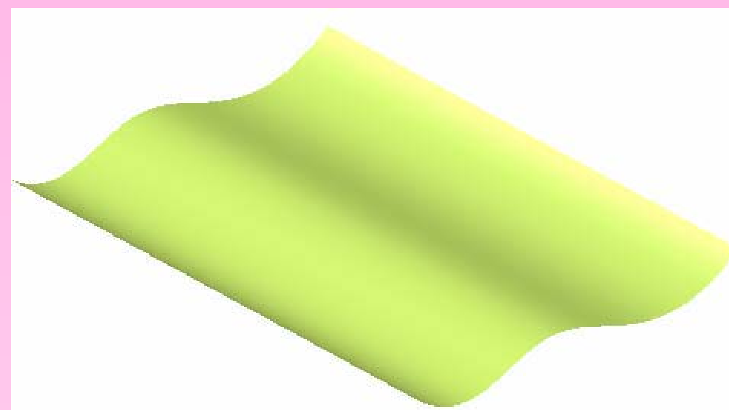
The **Untrim** dialog box



The **Warning** box

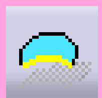


The trimmed surface

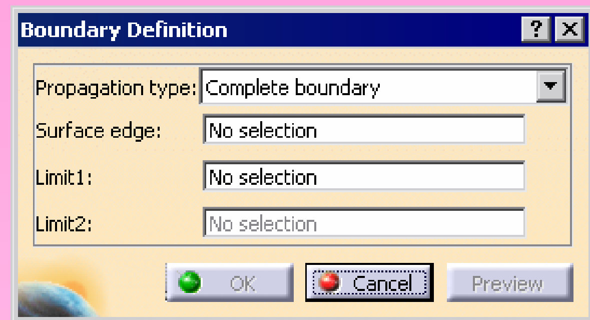


The untrimmed surface

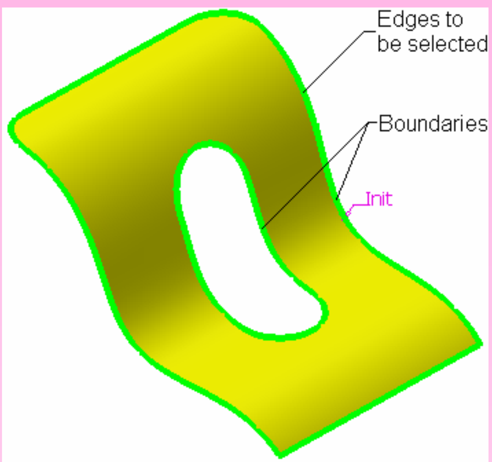
- ## Creating Boundary Curves



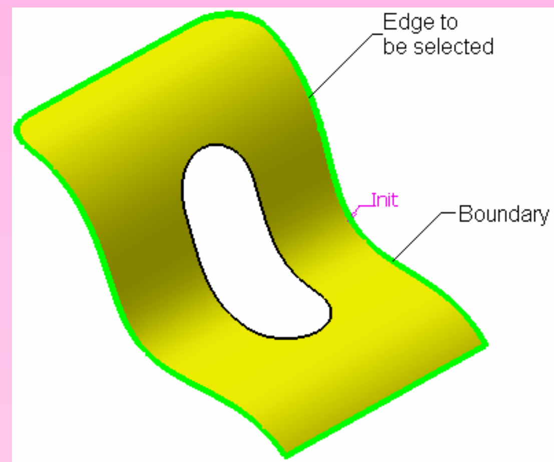
Choose the **Boundary** button from the **Operation** toolbar to invoke the **Boundary Definition** dialog box, as shown in the figure.



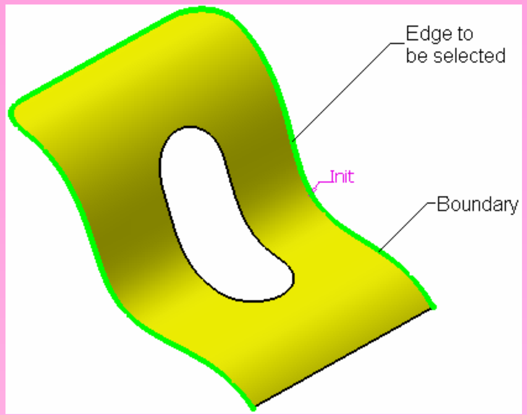
The Boundary Definition dialog box



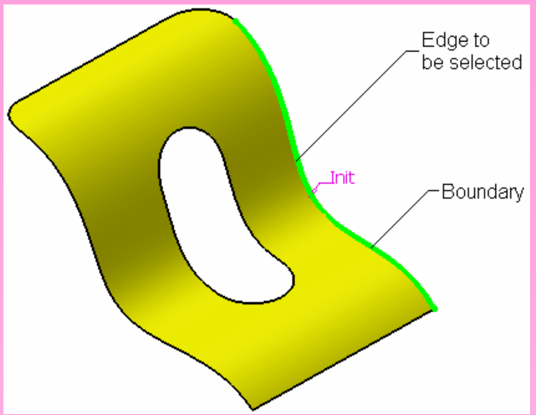
Boundary with a complete continuity



Boundary with a point continuity



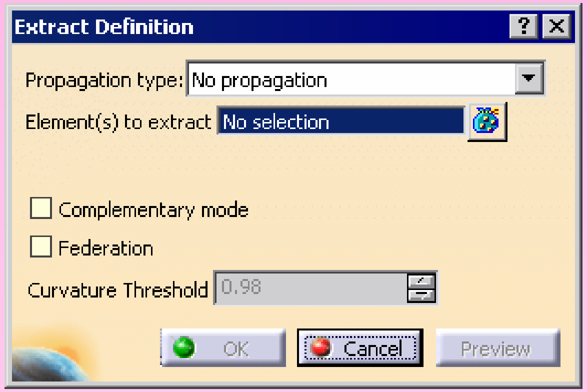
Boundary with a tangent continuity



Boundary with no continuity

- **Extracting Geometry**

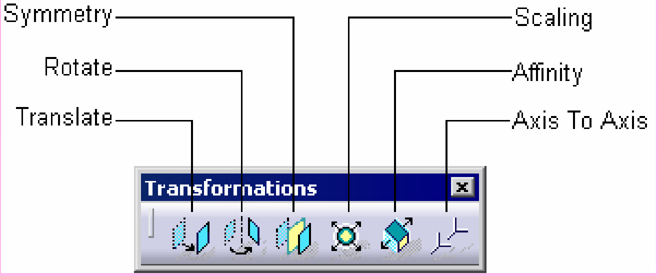
- Choose the **Extract** button from the **Extracts** toolbar, which is invoked when you choose the down arrow on the right of the **Boundary** button in the **Operations** toolbar.
- The **Extract Definition** dialog box is displayed, as shown in the figure.



The **Extract Definition** dialog box

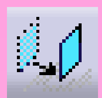
• Transformation Features

Transformation features are the features that are used to change the physical position of the geometry, such as translating, rotating, scaling, and so on.

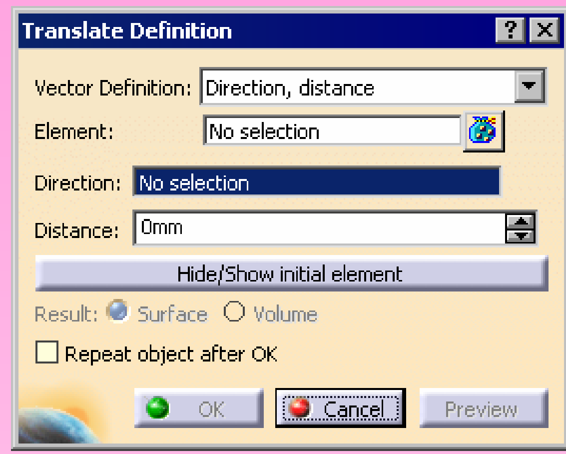


The Transformations toolbar

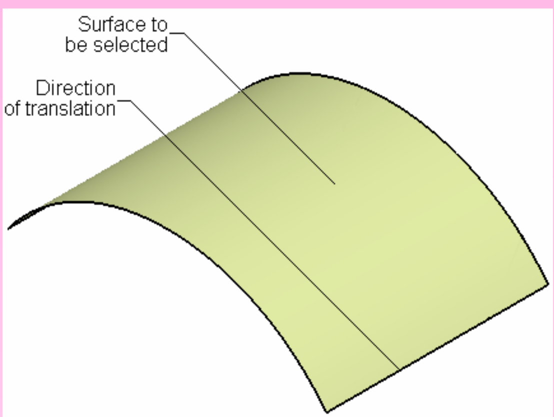
• Translating Elements



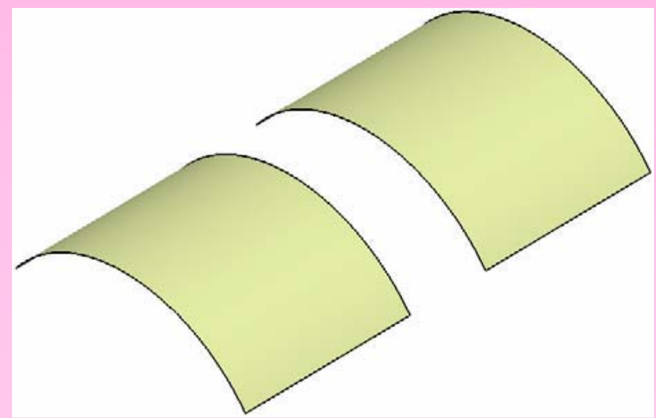
To translate an element, choose the **Translate** button from the **Transformations** toolbar; the **Translate Definition** dialog box is displayed, as shown in the figure.



The Translate Definition dialog box

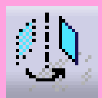


Surface and directional reference

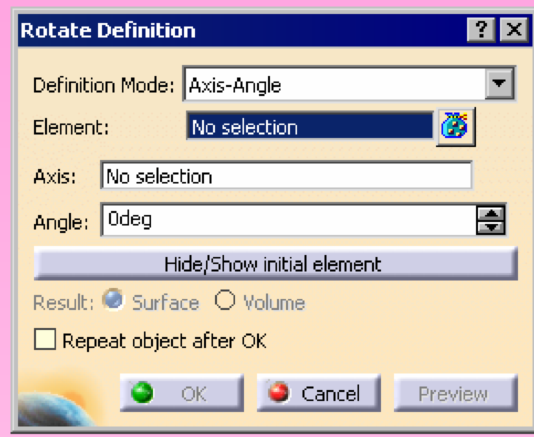


Resulting translated surface

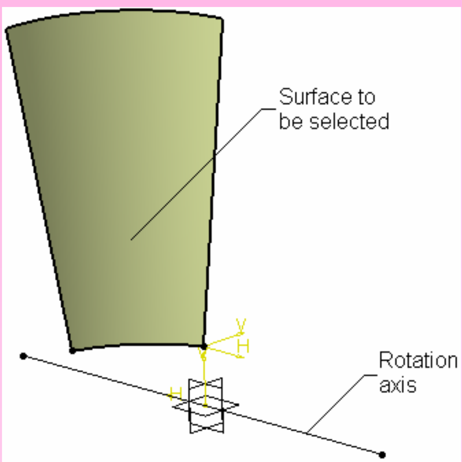
• Rotating Elements



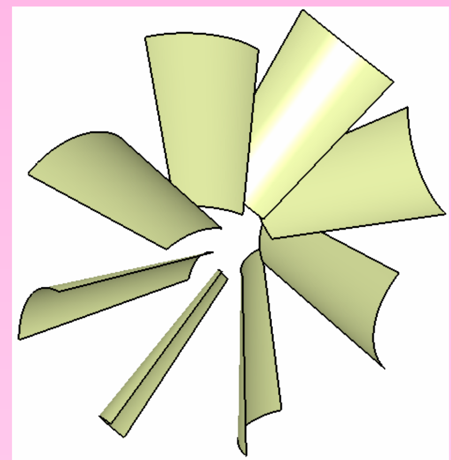
When you choose the **Rotate** button from the **Transformations** toolbar, the **Rotate Definition** dialog box will be displayed, as shown in the figure.



The **Rotate Definition** dialog box

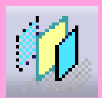


Surface and the axis of rotation

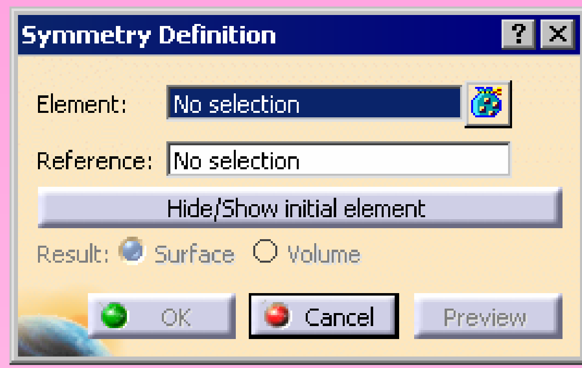


Resulting rotated surface

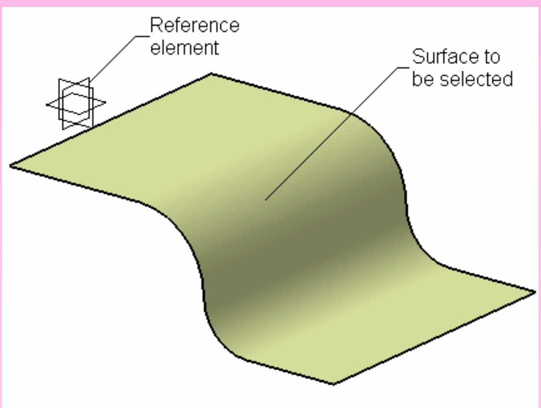
• Creating Symmetry Elements



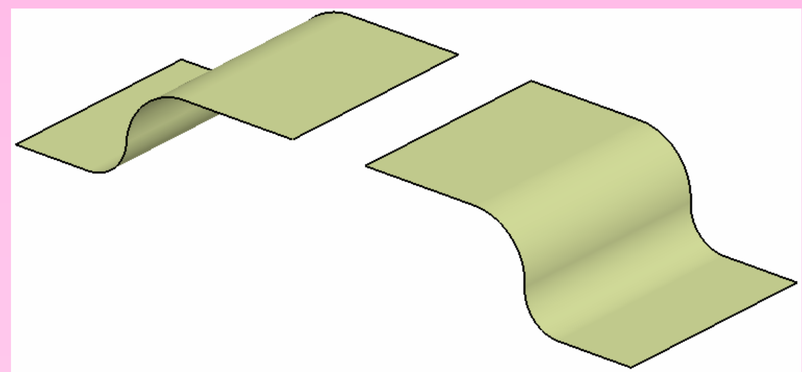
When you invoke the **Symmetry** tool, the **Symmetry Definition** dialog box is displayed, as shown in the figure.



The Symmetry Definition dialog box

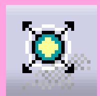


Surface and the reference element

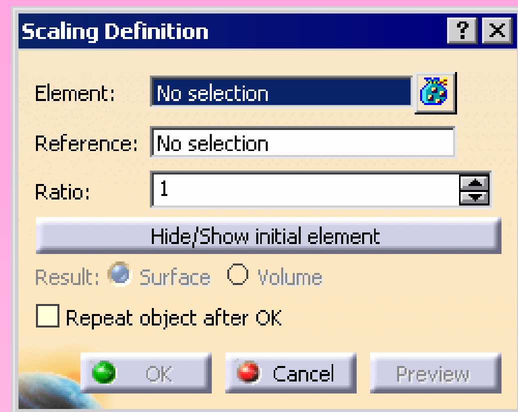


Resulting mirrored surface

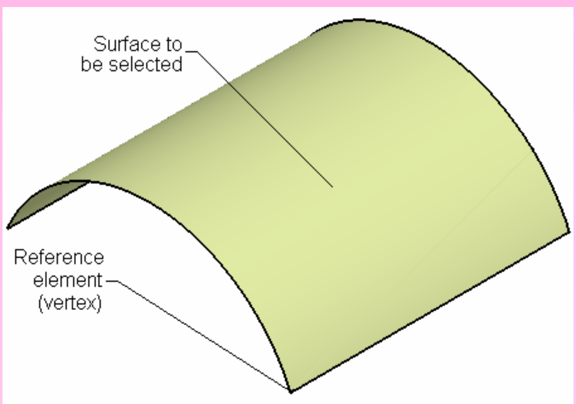
• Scaling Elements



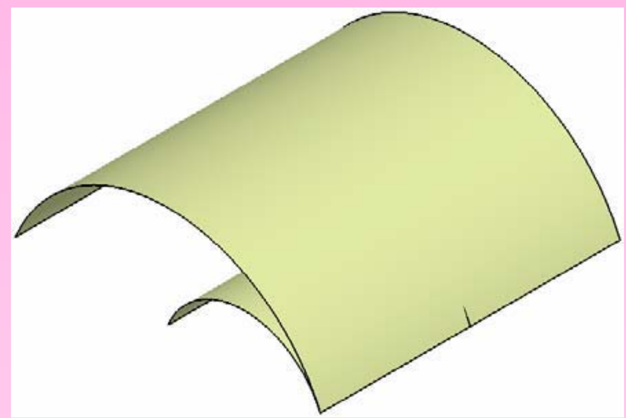
When you invoke this tool, the **Scaling Definition** dialog box is displayed, as shown in the figure.



The Scaling Definition dialog box

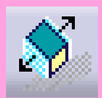


Surface and the reference element

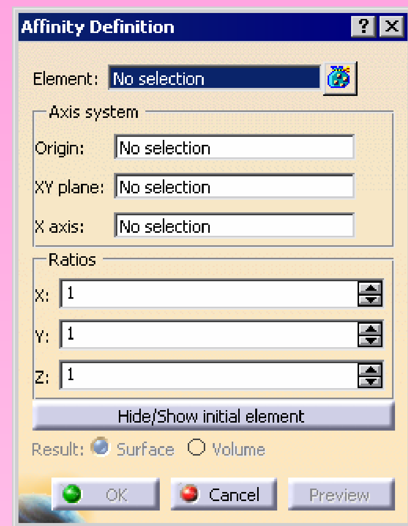


Resulting scaled surface

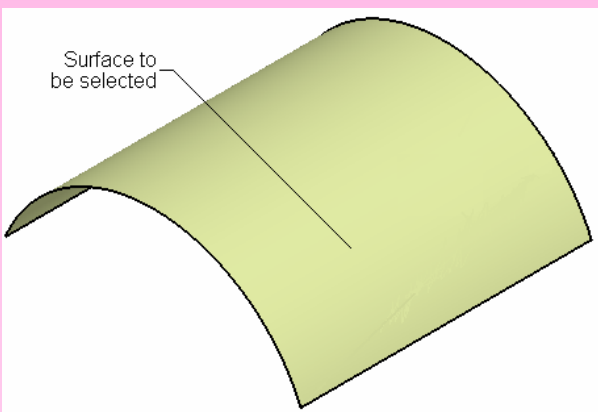
• Nonuniform Scaling of Elements



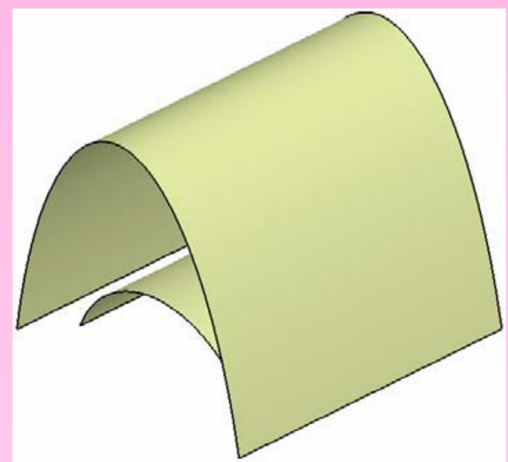
When you invoke this tool, the **Affinity Definition** dialog box is displayed, as shown in the figure.



The Affinity Definition dialog box

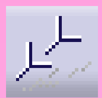


Surface to be selected

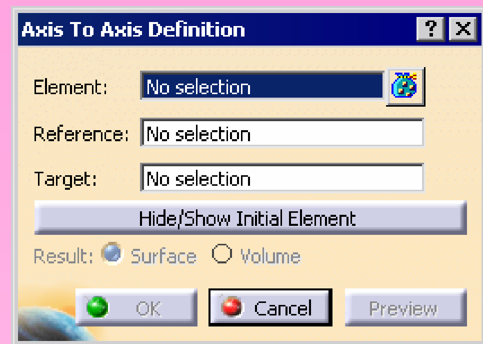


Resulting nonuniform scaling

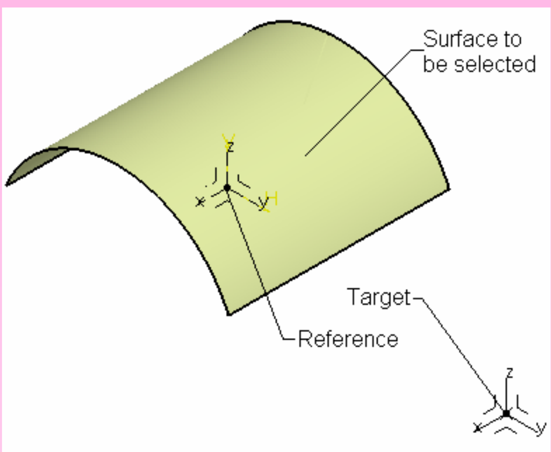
• Transforming an Element From Axis to Axis



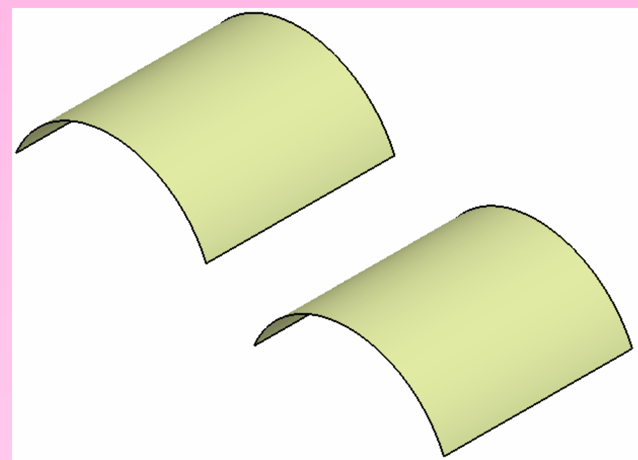
When you invoke this tool, the **Axis To Axis Definition** dialog box is displayed, as shown in the figure.



The Axis To Axis Definition dialog box



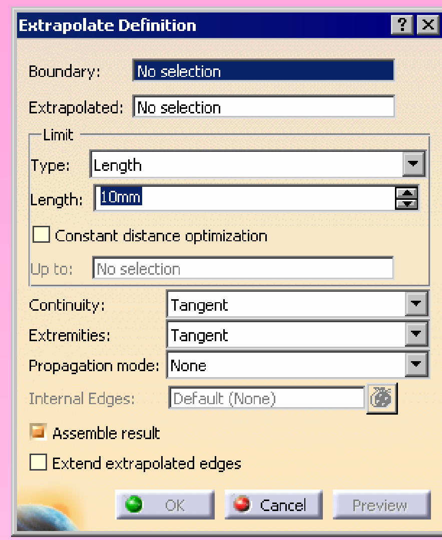
Surface and axes to be selected



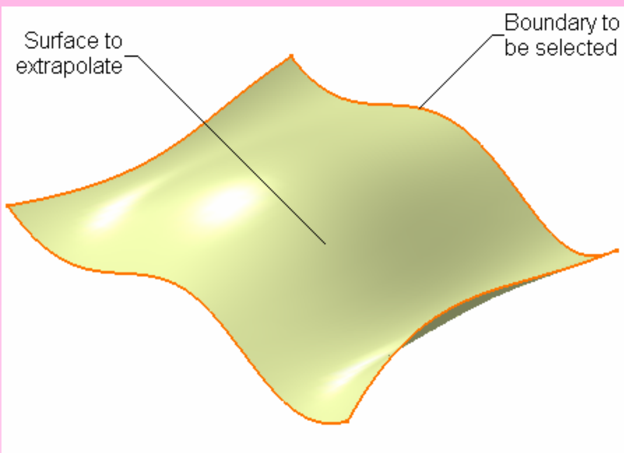
Resulting transformed surface

• Extrapolating Surfaces and Curves

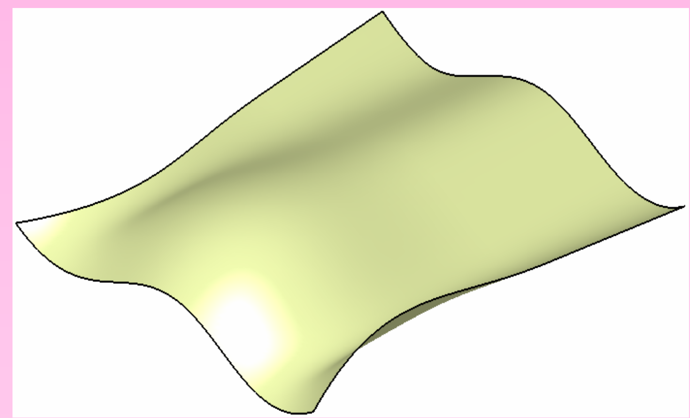
When you invoke the **Extrapolate** tool, the **Extrapolate Definition** dialog box is displayed, as shown in the figure.



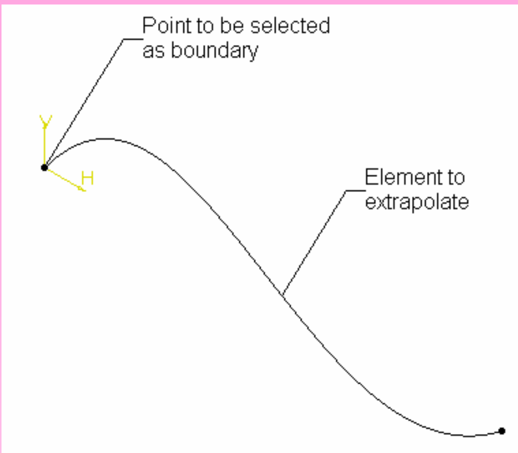
The Extrapolate Definition dialog box



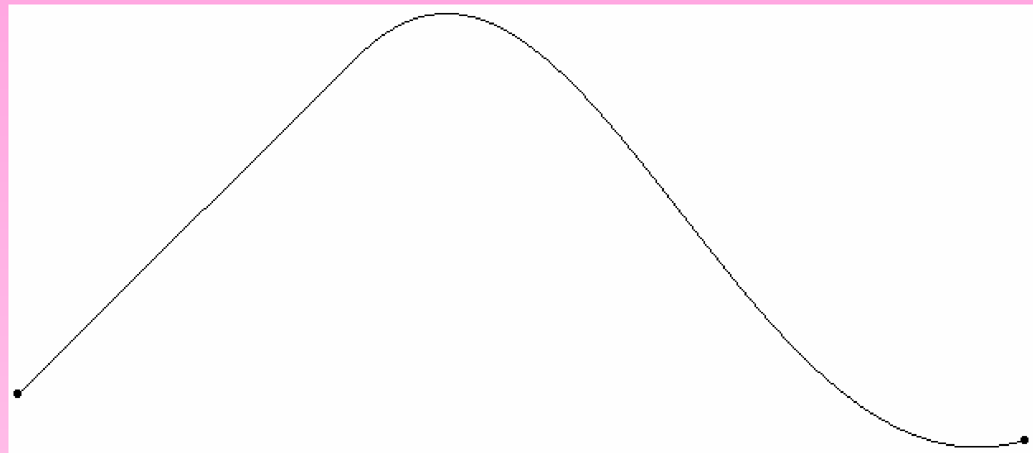
Boundary and surface to be selected



Resulting extrapolated surface



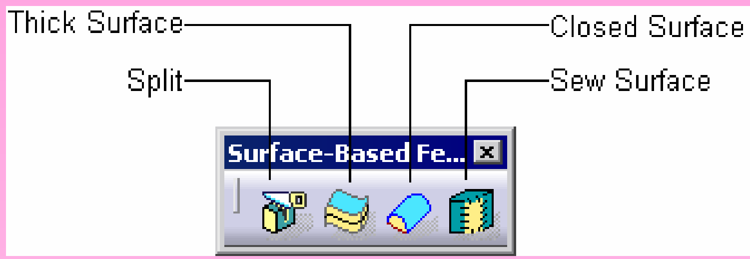
Boundary and curve to be selected



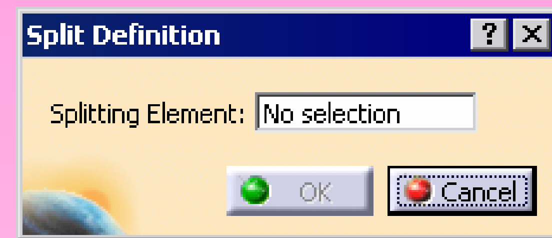
Resulting extrapolated curve

• Splitting a Solid Body With a Surface

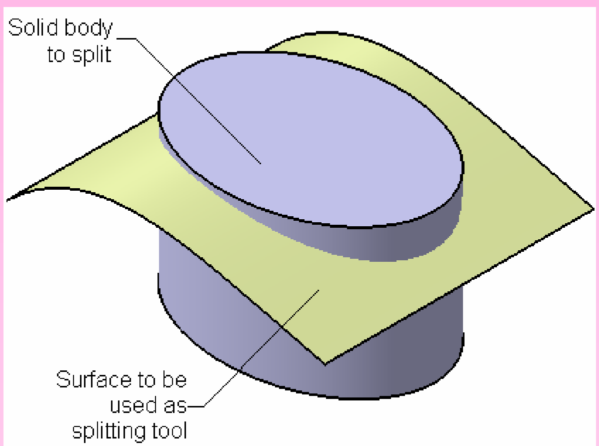
Choose the **Split** button from the **Surface-Based Features (Extended)** toolbar; the **Split Definition** dialog box is displayed, as shown in the figure.



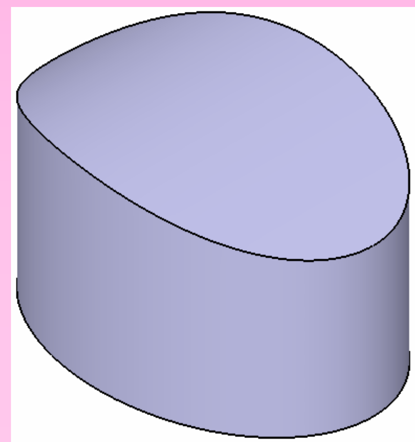
The **Surface-Based Features (Extended)** toolbar



The **Split Definition** dialog box



Body to be split and the splitting surface

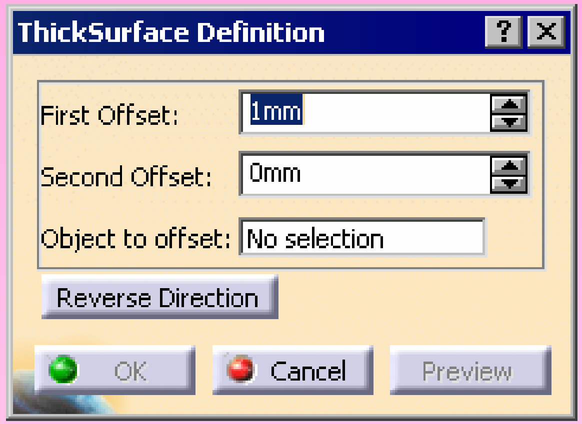


Resulting solid body after hiding the splitting surface

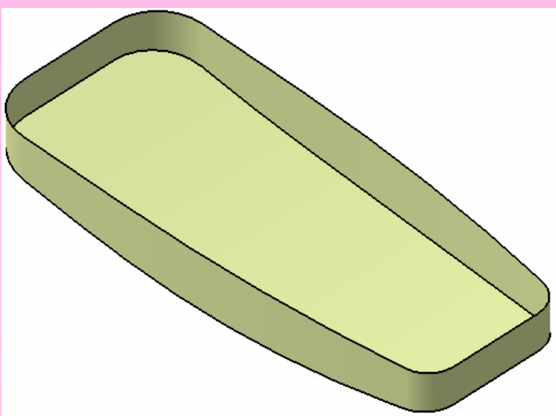
➢ SOLIDIFYING SURFACE MODELS

• Adding Thickness to a Surface

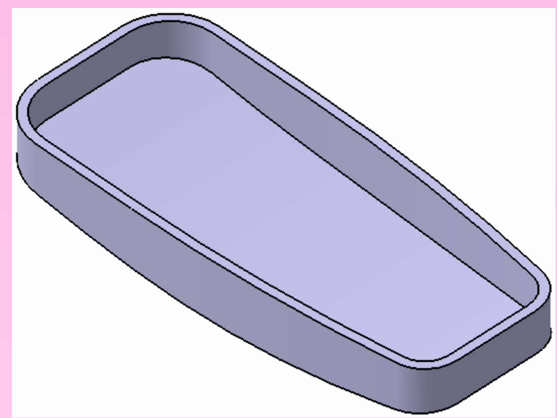
- To add thickness to a surface you first need to switch to the **Part** workbench and then choose the **Thick Surface** button from the **Surface- Based Features (Extended)** toolbar.
- The **ThickSurface Definition** dialog box is displayed, as shown in the figure.



The ThickSurface Definition dialog box

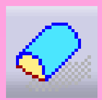


Surface to be thickened

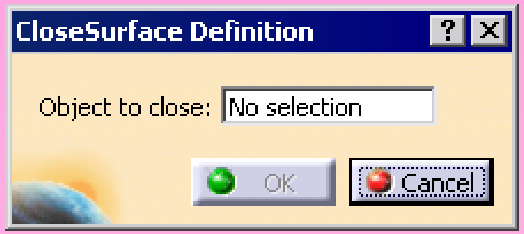


Resulting solid model

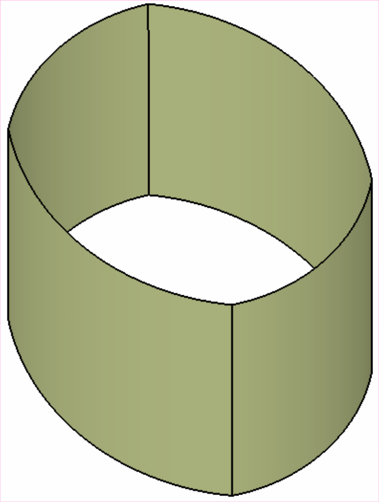
• Creating a Solid Body From a Closed Surface Body



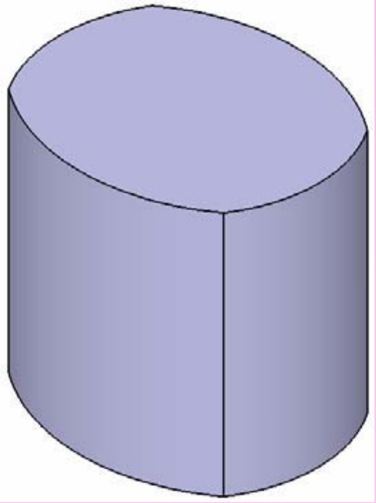
When you invoke the **Close Surface** tool, the **CloseSurface Definition** dialog box is displayed, as shown in the figure.



The CloseSurface Definition dialog box

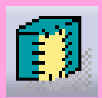


Surface with open ends

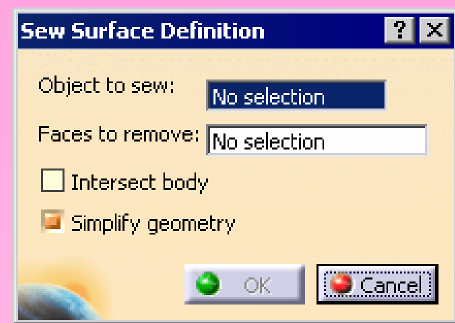


Resulting closed body

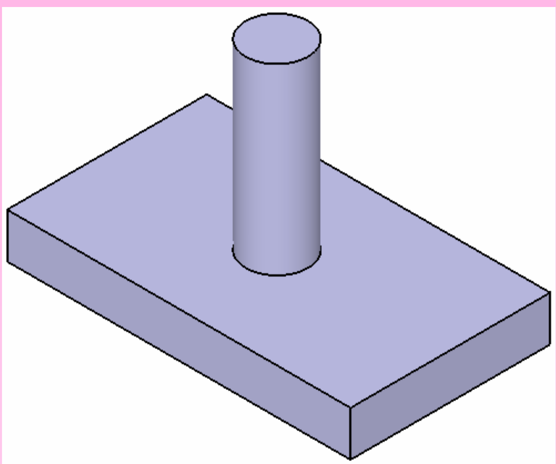
• Sewing a Surface With a Solid Body



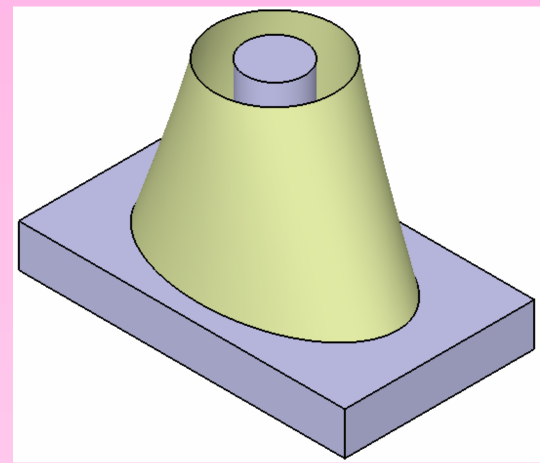
Choose the **Sew Surface** button from the **Surface-Based Features (Expanded)** toolbar; the **Sew Surface Definition** dialog box is displayed, as shown in the figure.



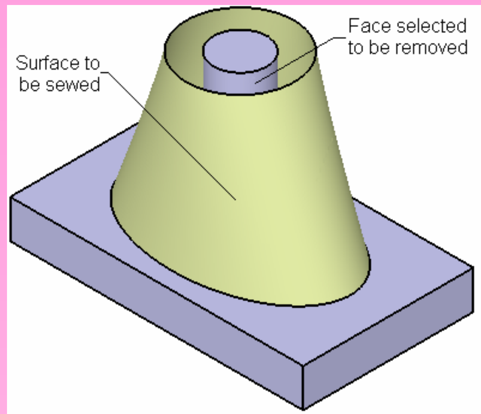
The Sew Surface Definition dialog box



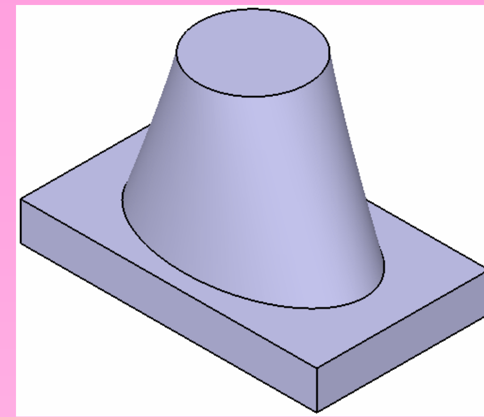
Solid body



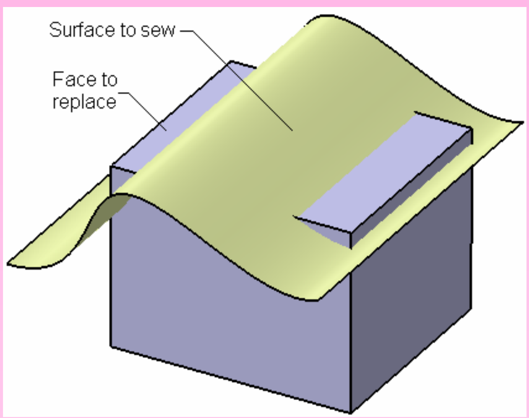
Surface created on the solid body



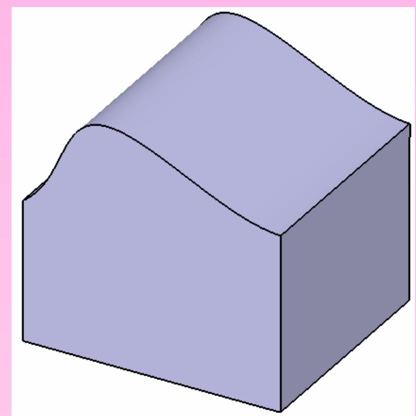
Surface to be sewed and face to be removed



Resulting solid body



Surface to be sewed and face to be removed



Resulting solid body

Tutorial 1

In this tutorial, you will create the model of the back cover of a toy monitor shown in **Figure A**. This model will be created using the tools in the **Wireframe and Surface Design** workbench and the **Part Design** workbench. After creating the surface model,you need to convert it into a solid body. Its views and dimensions are shown in **Figure B**. **(Expected time: 1 hr)**

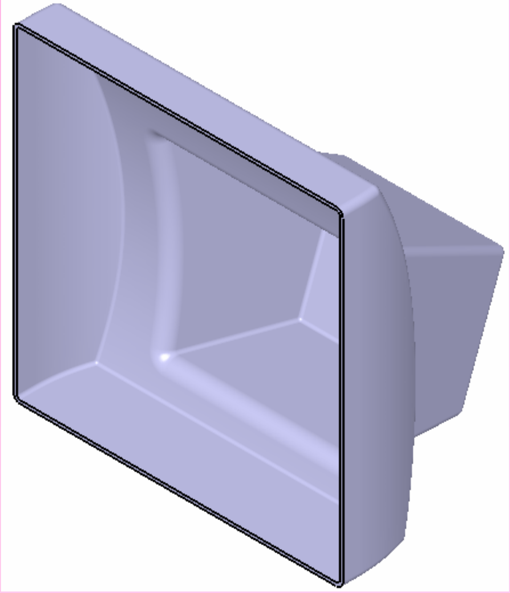


Figure A Back cover of the toy monitor

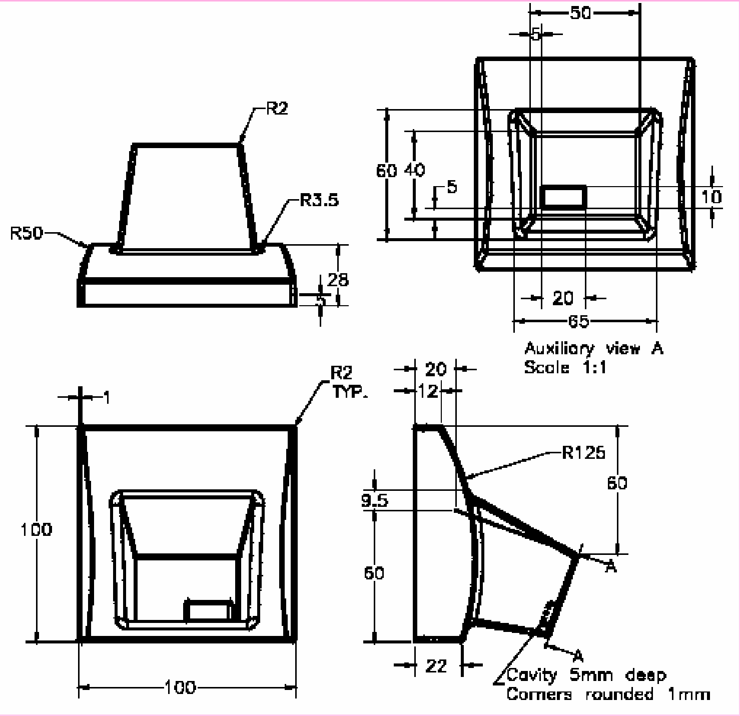


Figure B Views and dimensions for Tutorial 1

1. Start a new file in the **Wireframe and Surface Design** workbench and create the base surface of the model by extruding the sketch drawn on the zx plane, as shown in **Figure C** and **Figure D**.

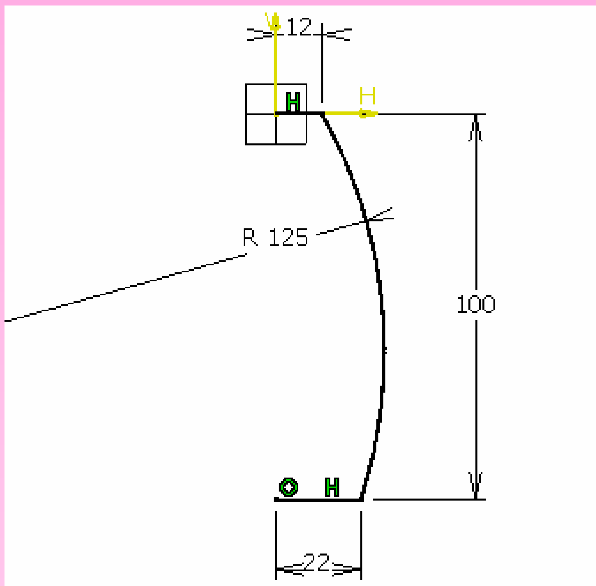


Figure C Sketch of the base feature

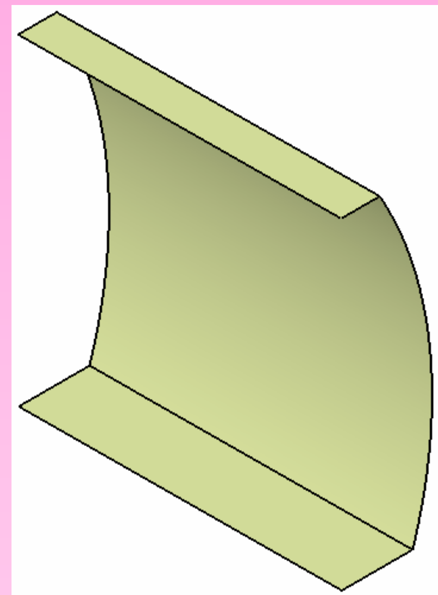


Figure D Resulting base surface

2. Create the other surfaces that are required for the basic structure of the model, as shown in **Figure E** through **Figure N**.

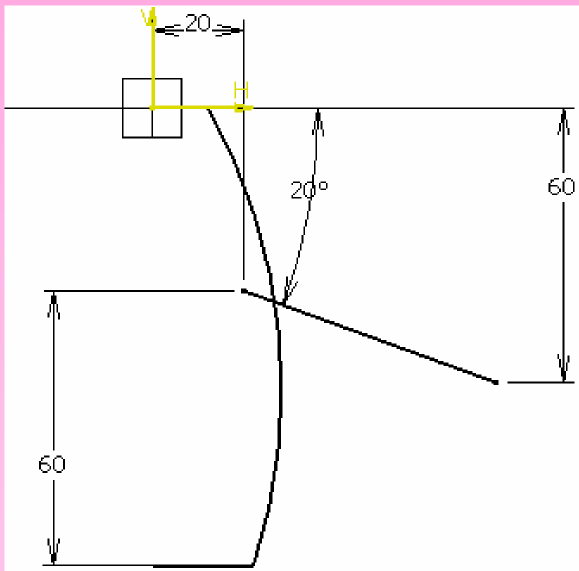


Figure E Reference sketch

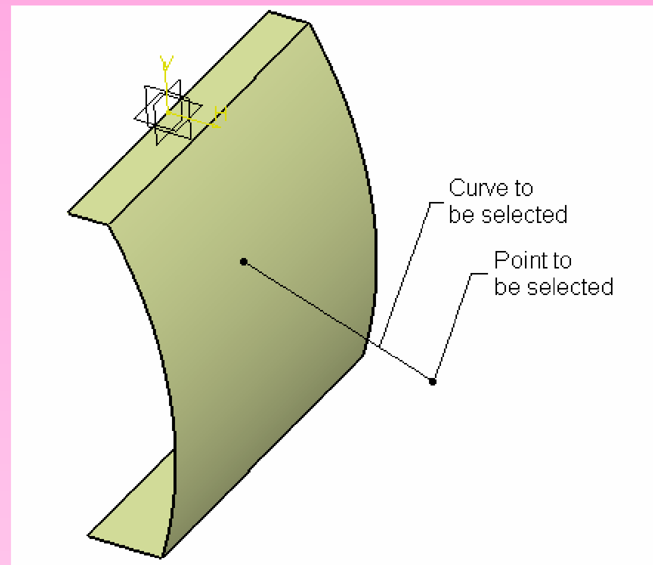


Figure F Point and curve to be selected

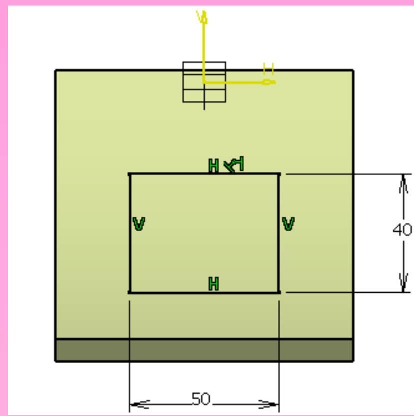


Figure G Sketch to be drawn on the new plane

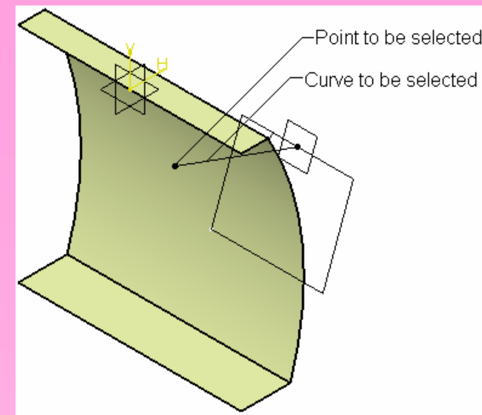


Figure H Point and curve to be selected

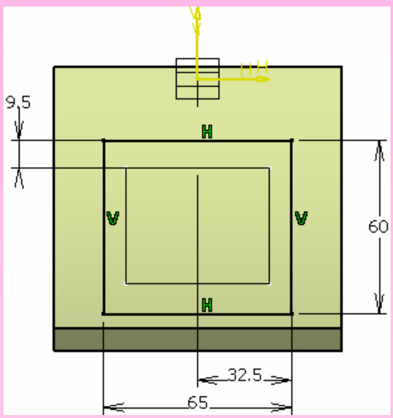


Figure I Sketch to be drawn on the new plane

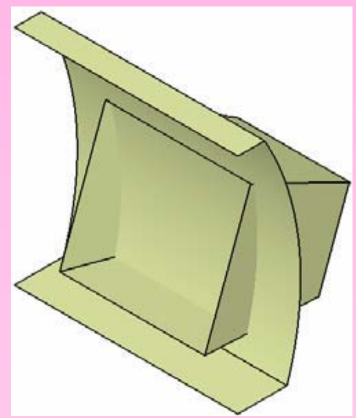


Figure J Resulting lofted surface

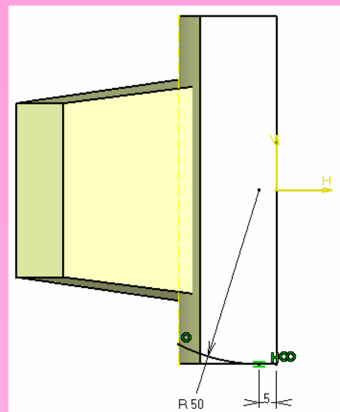


Figure K Sketch to be drawn

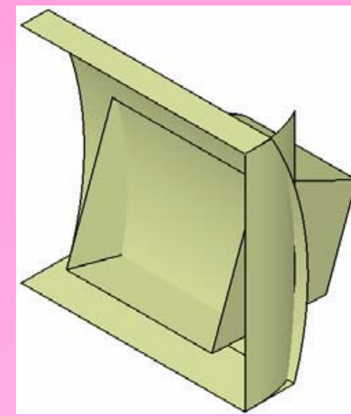


Figure L Resulting extruded surface

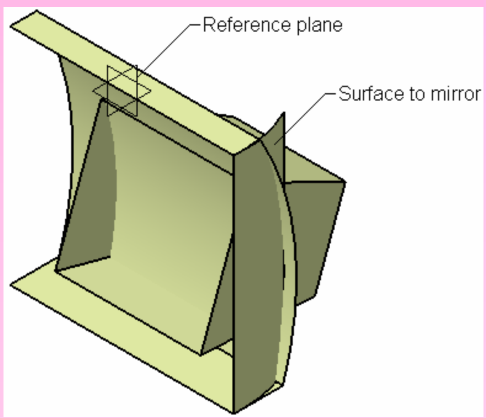


Figure M Surface and reference to be selected

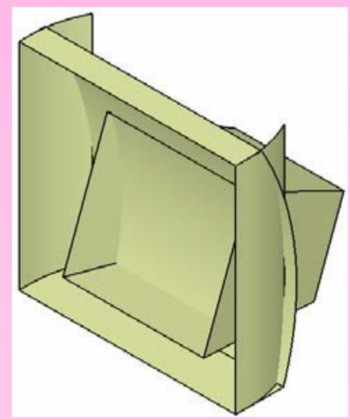


Figure N Resulting mirrored surface

3. Trim the unwanted surfaces, as shown in **Figure O**.

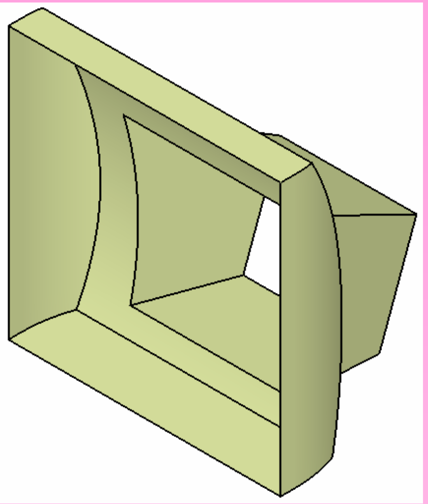


Figure O The model after trimming the surfaces

4. Invoke the **Part** workbench and convert the surface model into a solid model, as shown in **Figure P**.

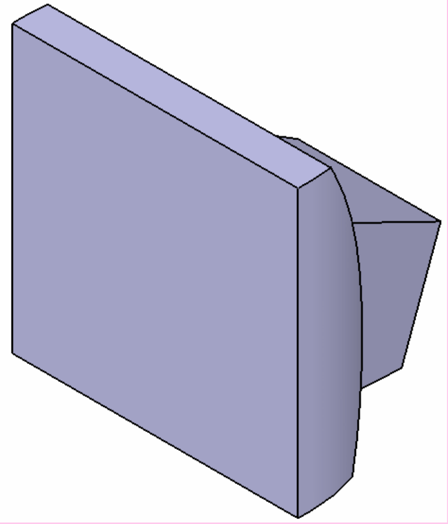


Figure P The model after solidifying the surfaces

5. Fillet the edges and shell the model, as shown in **Figure Q**.

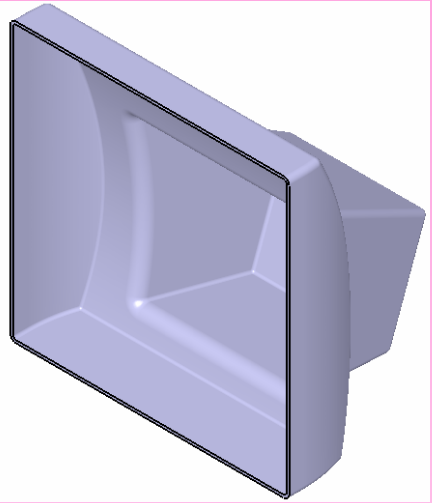


Figure Q Final model for Tutorial 1

6. Save the file in **\My Documents\CATIA\c10** folder and then close it.

Tutorial 2

In this tutorial, you will create the model of the Hair Dryer Cover shown in **Figure A**. It will be created using the tools in the **Wireframe and Surface Design** workbench and the **Part Design** workbench. The views and dimensions are shown in **Figure B**.

(Expected time: 1 hr)

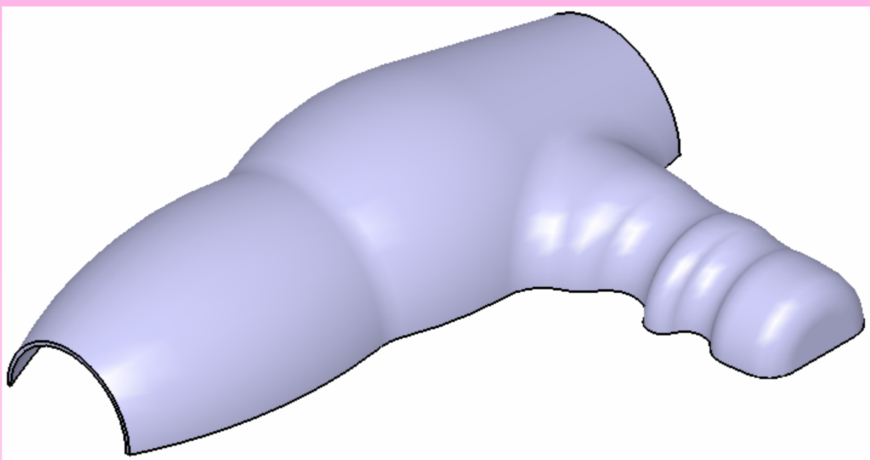


Figure A Model of the Hair Dryer Cover

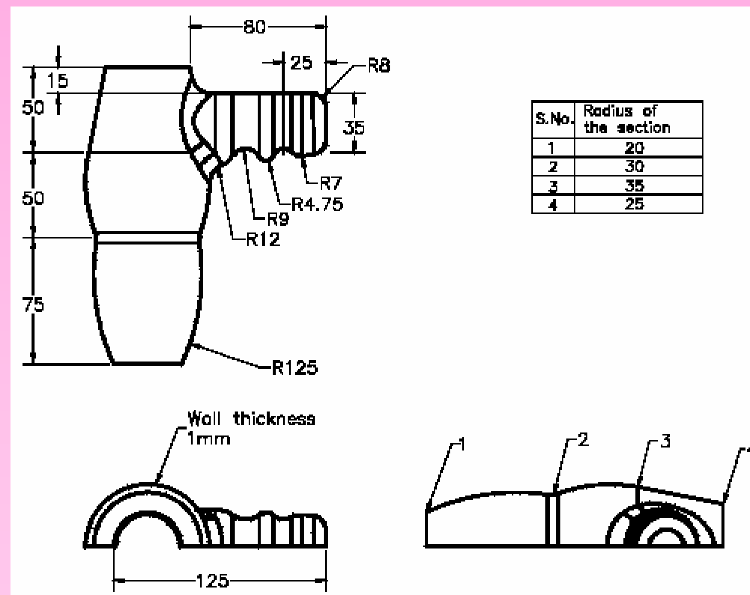


Figure B Views and dimensions for Tutorial 2

1. Start a new file in the **Wireframe and Surface Design** workbench and create the base surface, which forms the body of the Hair Dryer Cover, using **Multisections surface** tool, as shown in **Figure C** and **Figure D**.

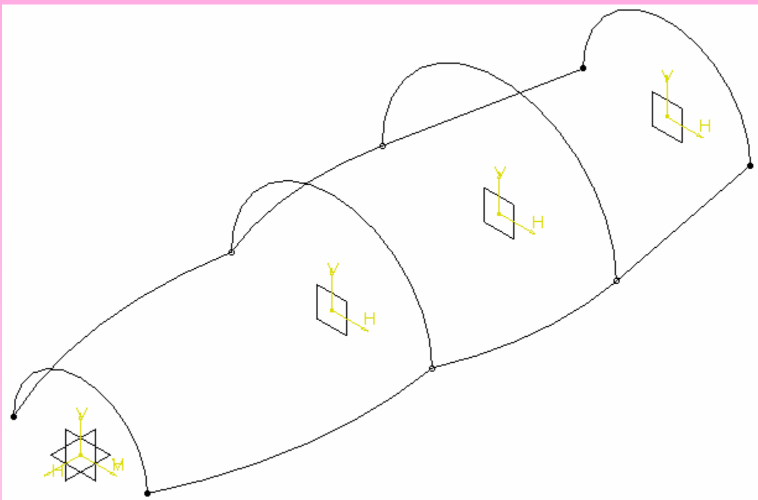


Figure C Sketches of sections and guide curves

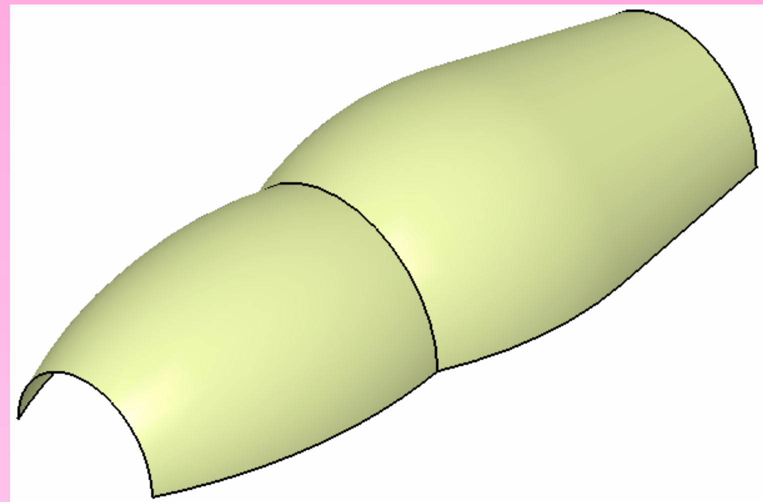


Figure D Resulting base surface

2. Create a swept surface that forms the handle of the cover, as shown in **Figure E**, **Figure F**, **Figure G** and **Figure H**.

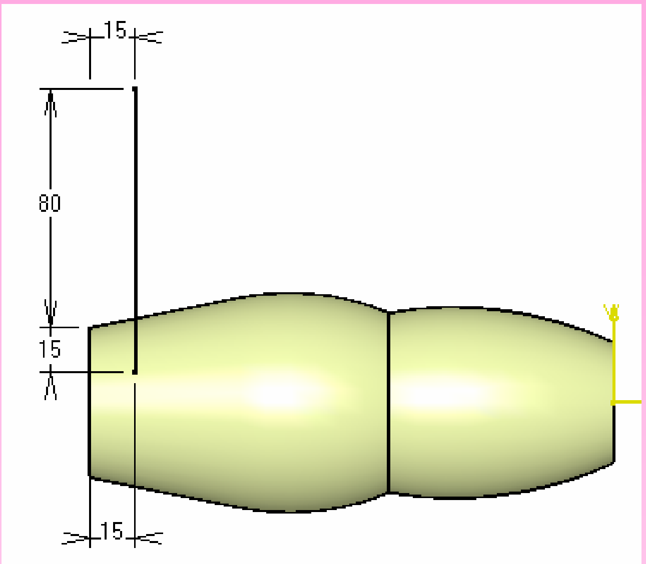


Figure E **Sketch of the first guide curve**

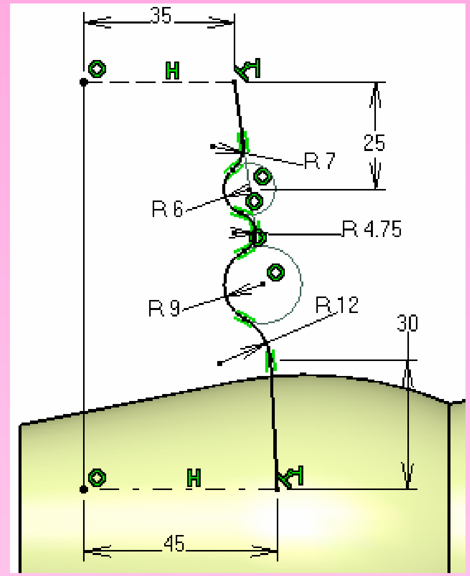


Figure F **Sketch of the second guide curve**

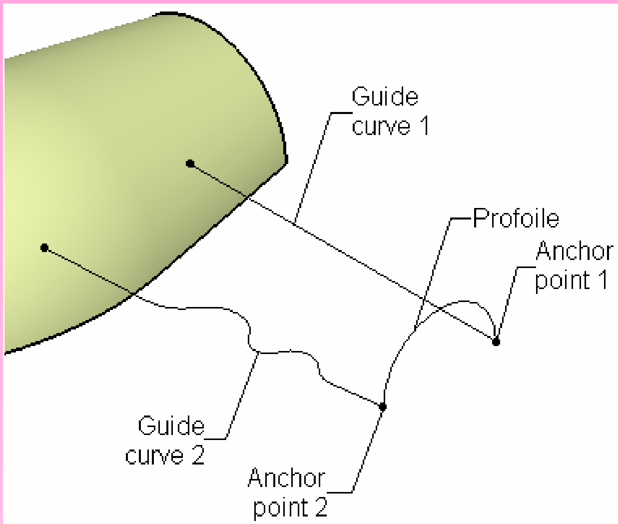


Figure G Profile, guide curves, and anchor points to be selected

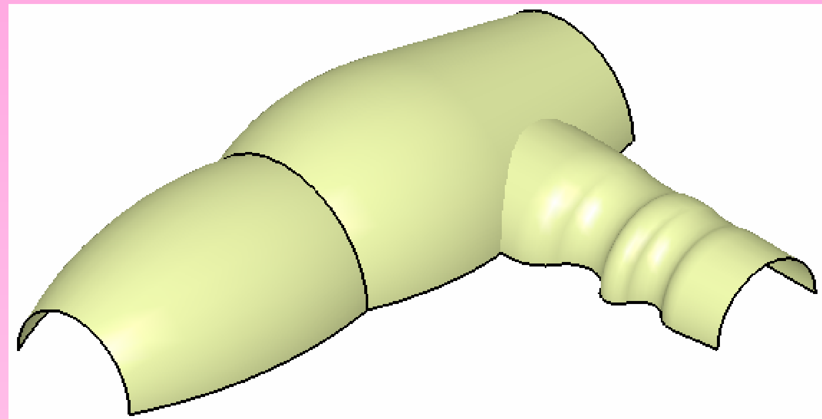


Figure H Resulting swept surface

3. Trim the surfaces and create a flat boundary surface to close the surface body, as shown in **Figure I**, **Figure J**, **Figure K** and **Figure L**.

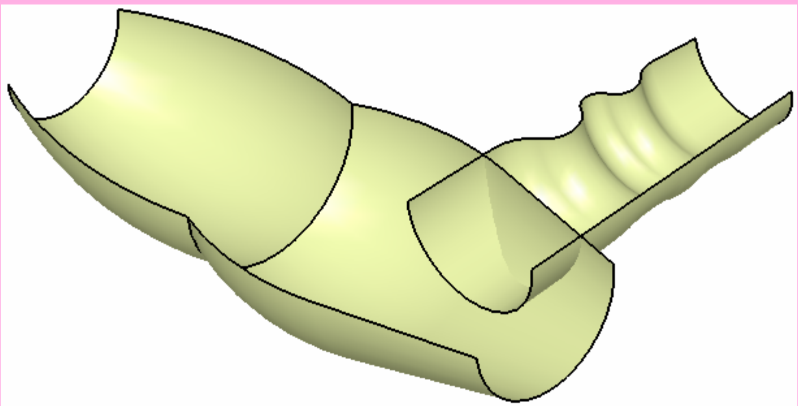


Figure I Intersecting surfaces

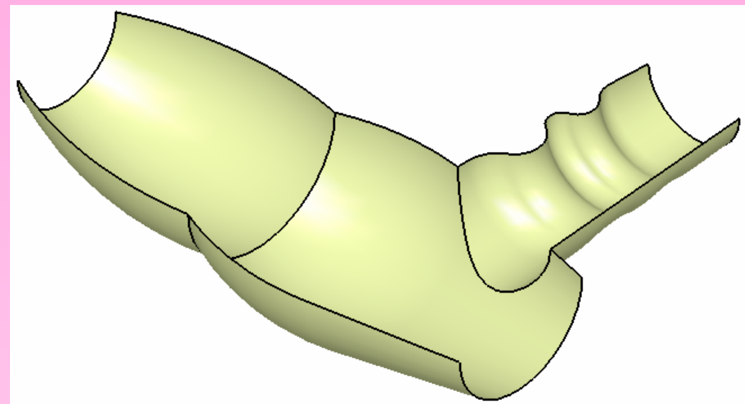


Figure J Resulting trimmed surface

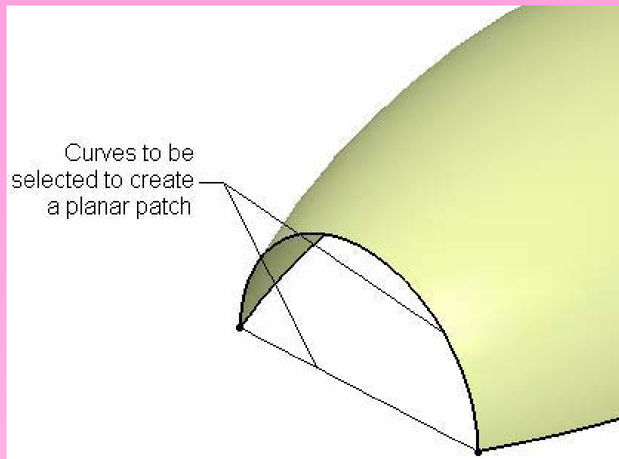


Figure K Curves to be selected

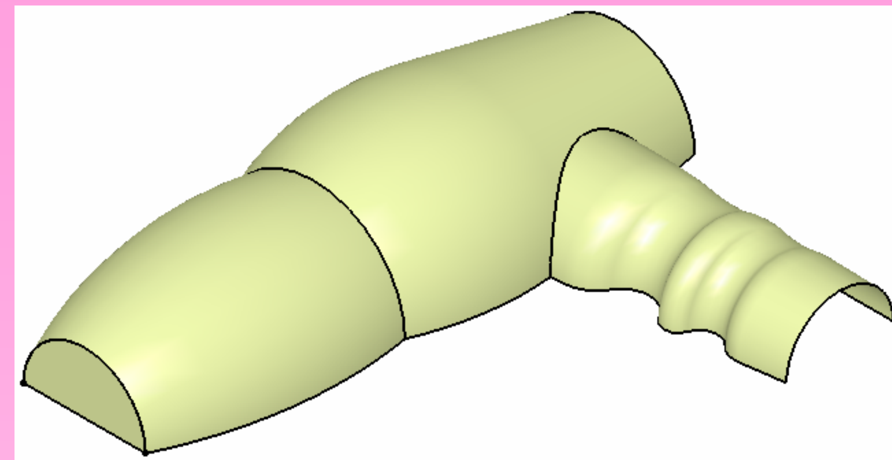


Figure L Resulting filled surface

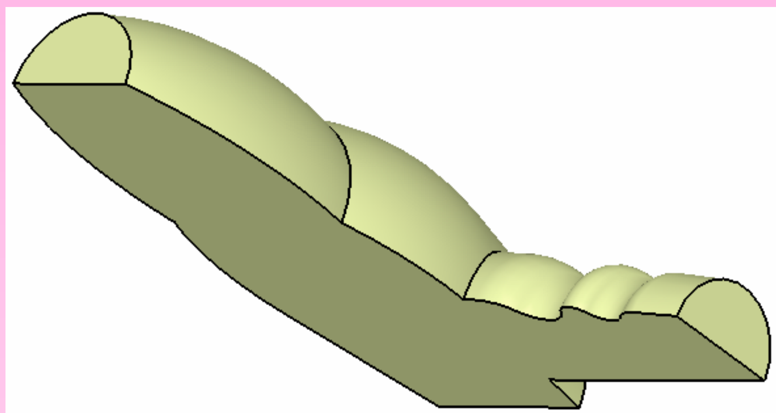


Figure M The model after creating all the flat boundary surfaces

4. Join the surfaces together.
5. Invoke the **Part Design** workbench and solidify the surface body using the **Close Surface** tool, as shown in **Figure N**.

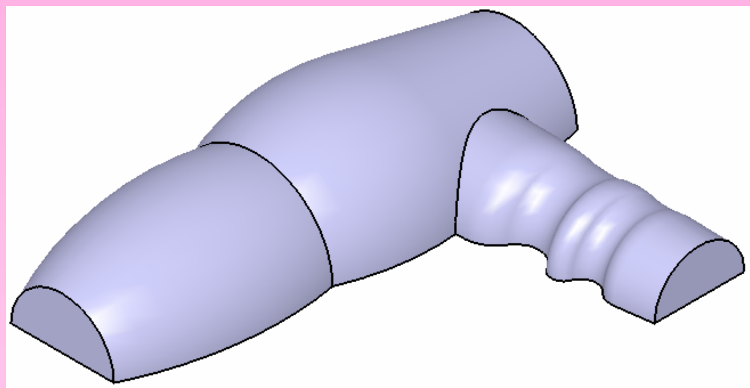


Figure N Resulting solid body

6. Create the remaining features of the model using the **Edge Fillet** and **Shell** tools, as shown in **Figure O**.

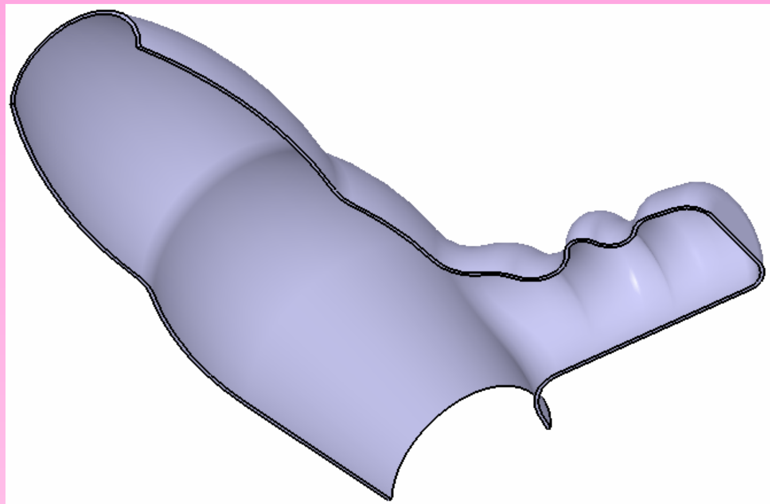


Figure O Final model of the Hair Dryer Cover

7. Save the file in **\My Documents\CATIA\c10** folder and then close it.

Exercise 1

Create the model of the Cover shown in **Figure A**. Its views and dimension are shown in **Figure B**. **(Expected time: 30 min)**

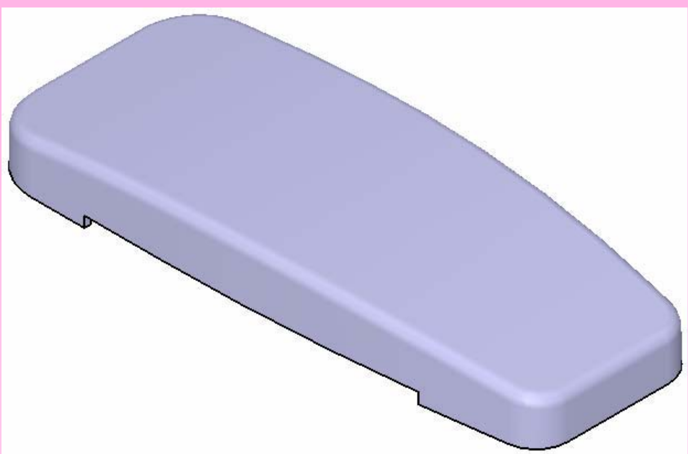


Figure A Model of the cover

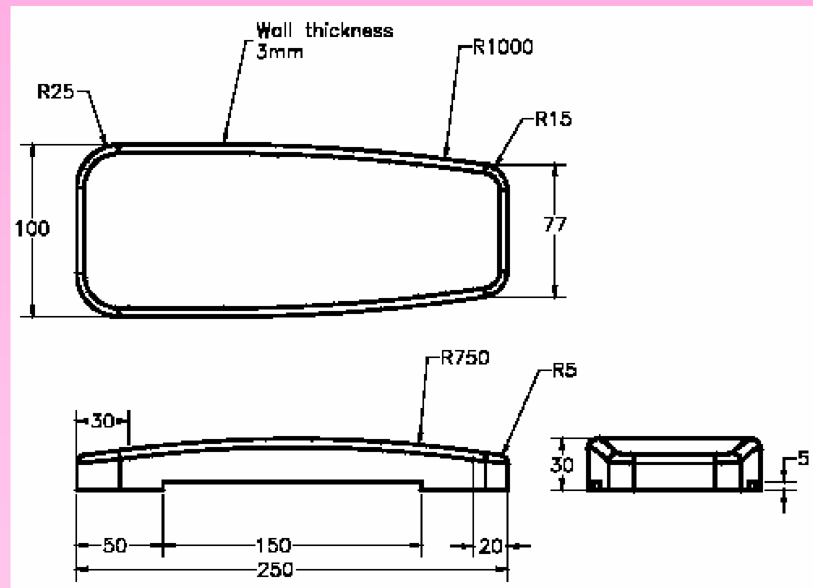


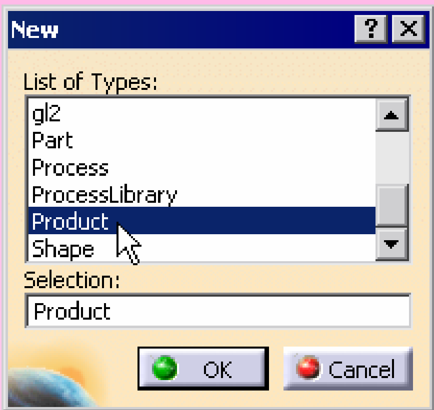
Figure B Views and dimensions of the model

Learning Objectives:

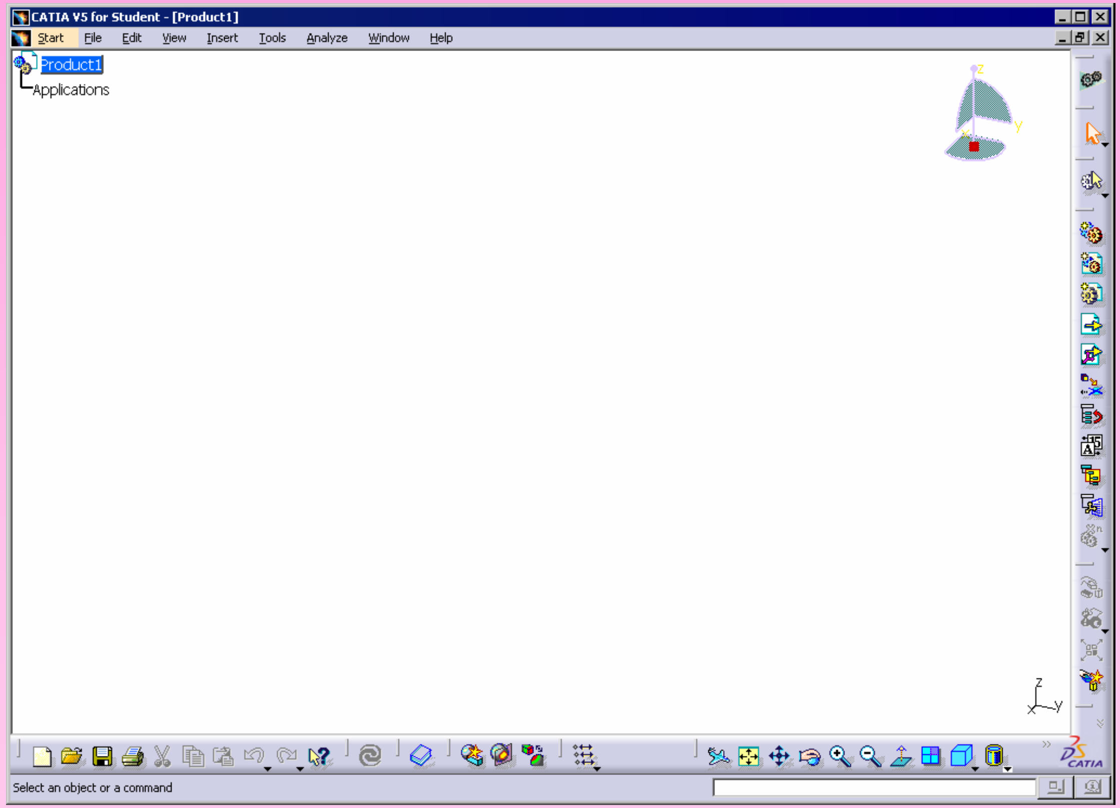
- *Insert components into an assembly file.*
- *Create bottom-up assemblies.*
- *Insert components into a product file.*
- *Move and rotate components inside an assembly.*
- *Add constraints to individual components.*
- *Create top-down assemblies.*
- *Edit assembly designs.*
- *Create the exploded state of an assemblies.*

➢ ASSEMBLY MODELING

- The primary method to start a new product file is by selecting **File > New** from the menu bar to open the **New** dialog box.
- From this dialog box, select **Product**, as shown in **Figure A**.



The Product option selected from the New dialog box



Screen display after starting a new file in the Assembly Design workbench

- **Types of Assembly Design Approach**

  In CATIA V5 you can create assembly models by adopting two types of approaches.

- **Bottom-up Assembly**

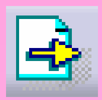
  The bottom-up assembly is the most preferred approach for creating assembly models.

- **Top-down Assembly**

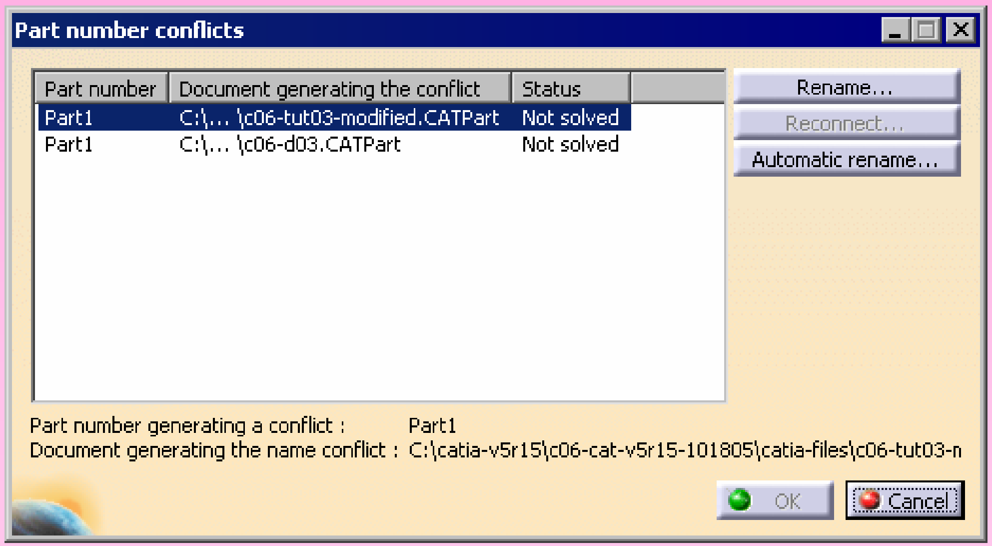
  In the top-down assembly design approach, components are created inside the **Assembly Design** workbench.

CREATING BOTTOM-UP ASSEMBLIES

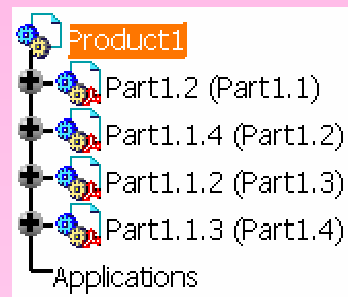
Inserting Components in a Product file



When you insert additional components, the **Part** number conflicts dialog box is displayed, as shown in the figure.

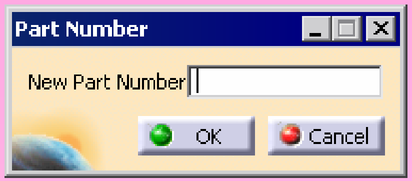


The **Part number conflicts** dialog box

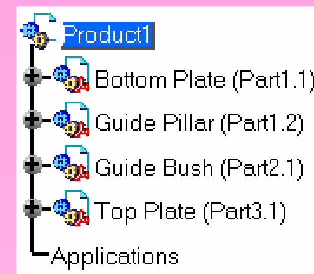


The specification tree showing four components

If you choose the **Rename** button from the **Part number conflicts** dialog box, the **Part Number** dialog box is displayed, as shown in the figure.



The **Part Number** dialog box



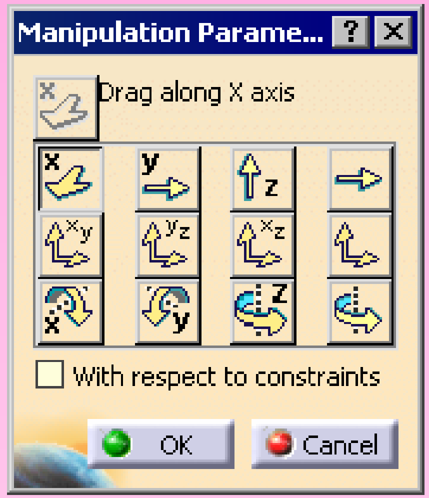
The specification tree showing four components with unique part numbers

- **Moving Individual Components**

CATIA allows you to move and rotate the individual unconstrained components inside the product file without affecting the position and location of the other components.

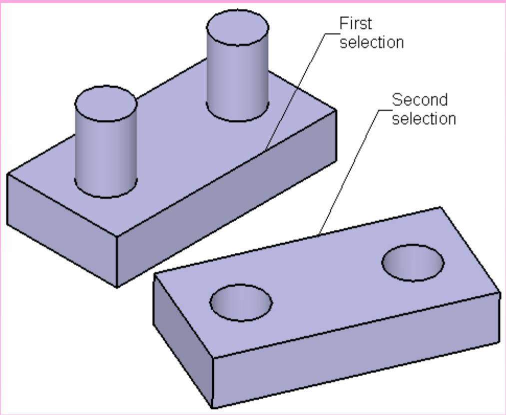
- **Moving and Rotating by Using the Manipulation Tool**

To translate or rotate any component, choose the **Manipulation** button from the **Move** toolbar; the **Manipulation Parameter** dialog box is displayed, as shown in the figure.

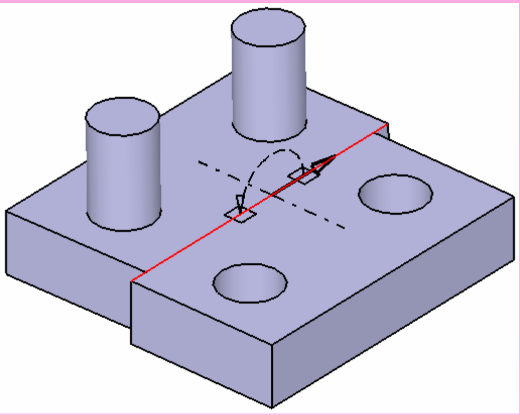


The Manipulation Parameter dialog box

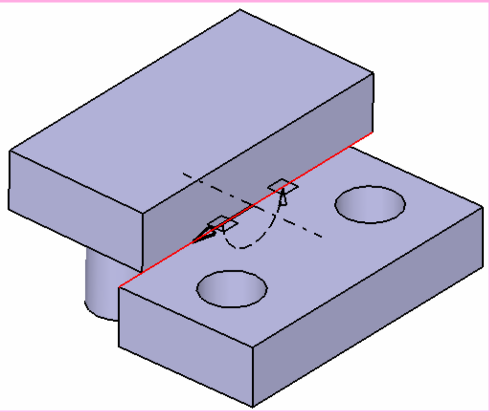
- **Moving Components by Using the Snap Tool**



Geometric elements selected to be snapped



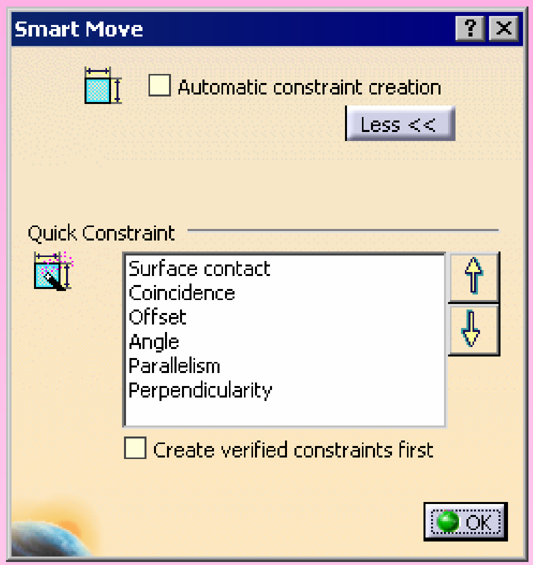
Position of components after snapping



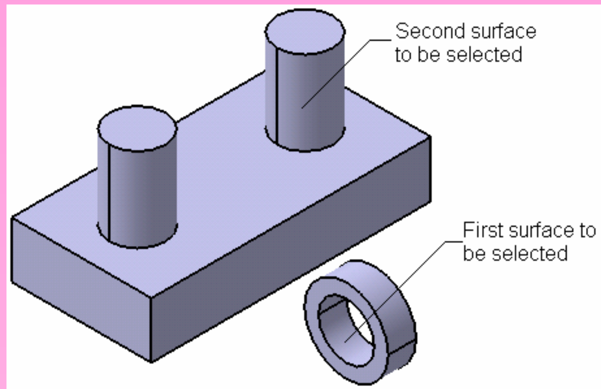
Position of components after the snapping direction is reversed

Moving Components by Using the Smart Move Tool

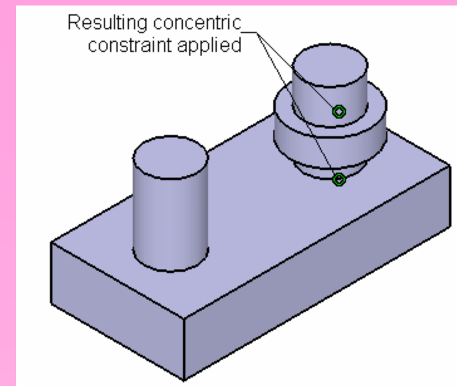
- To invoke this tool, choose the down arrow besides the **Snap** button to invoke the **Snap** toolbar.
- Choose the **Smart Move** button to invoke the **Smart Move** dialog box.
- Choose the **More** button to expand it, as shown in the figure.



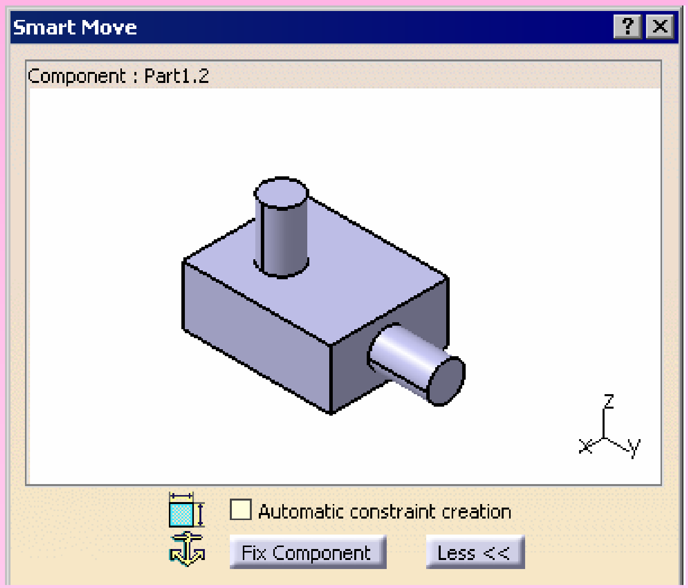
The expanded Smart Move dialog box



Surfaces to be selected



Resulting constraint applied

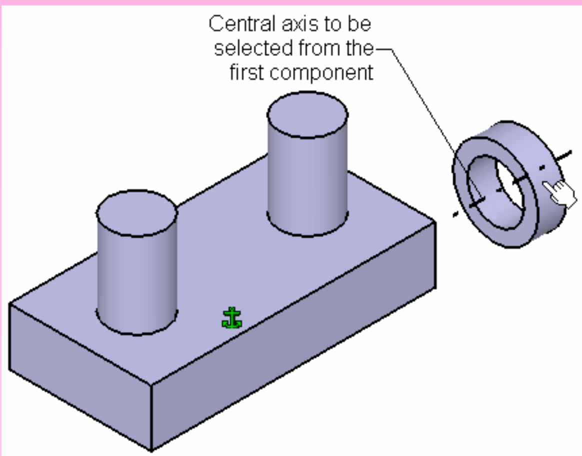


The partial view of the **Smart Move** dialog box with viewer

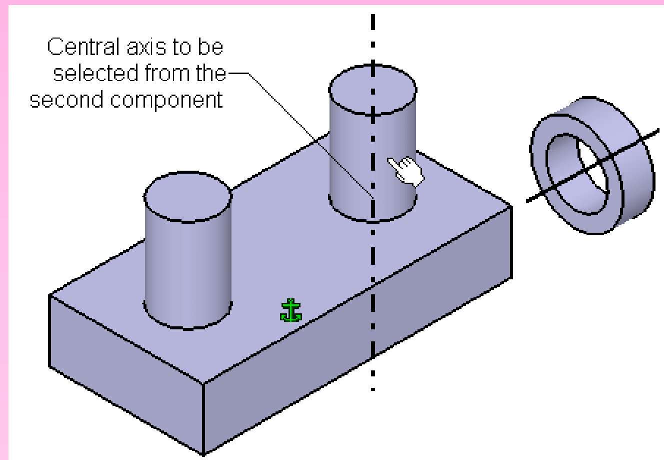
- **Manipulating Components using the Compass**

  The orientation of the components can also be manipulated using the compass on the top right corner of the geometry area.

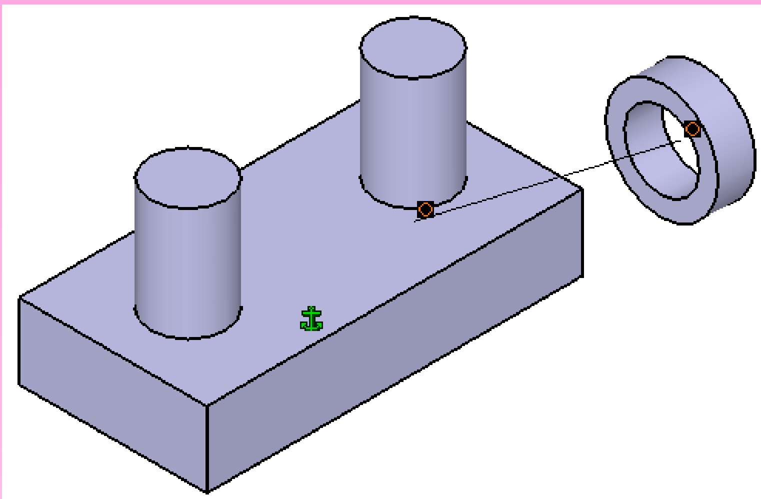
- **Applying Constraints**
  - **Fix Component Constraint**
  - **Coincidence Constraint**



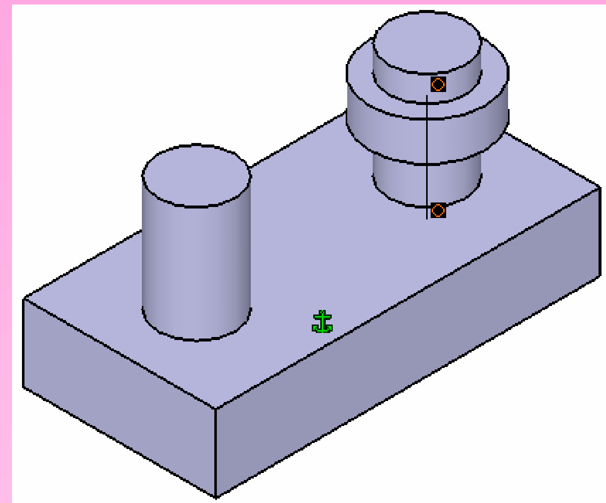
Central axis of the first component to be selected



Central axis of the second component to be selected

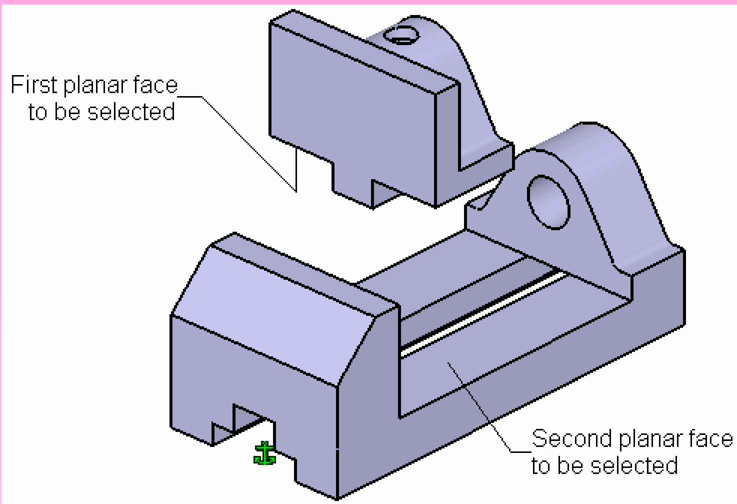


Coincidence constraint applied between two components

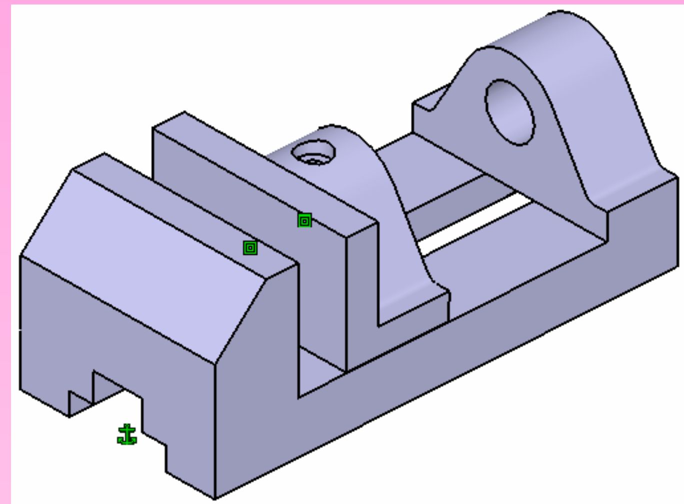


Position of the components after updating

- Contact Constraint

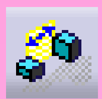


Planar faces to be selected

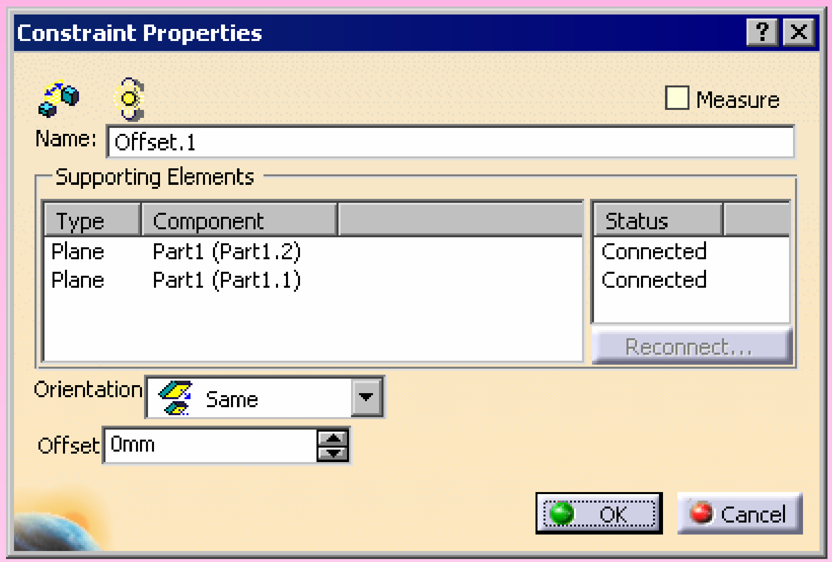


Position of components, after a constraint is applied and updated

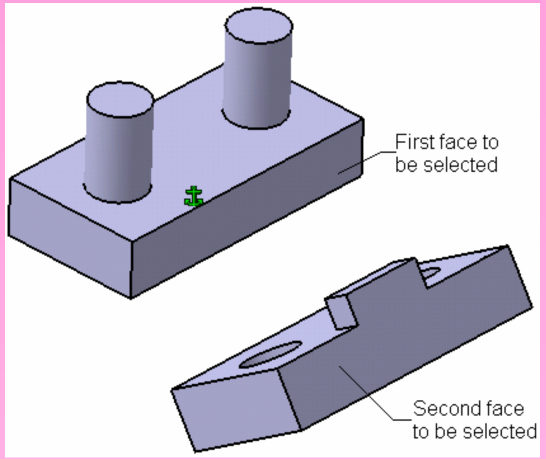
- **Offset Constraint**



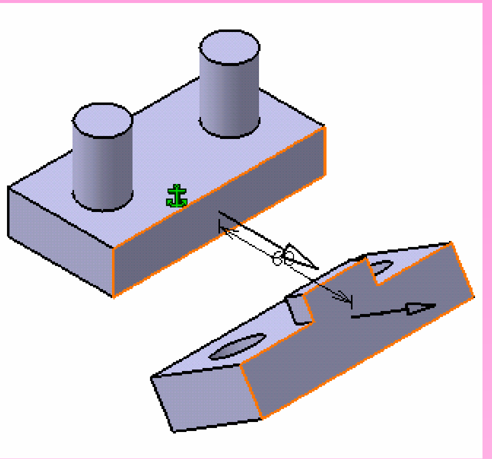
- After invoking the **Offset Constraint** tool, you are prompted to select the first geometric element for the **Offset** constraint.
- Select a planar face, circular face, plane, axis, or a point from the geometry area.
- Select a planar face of another component; the **Constraint Properties** dialog box is displayed, as shown in the figure.



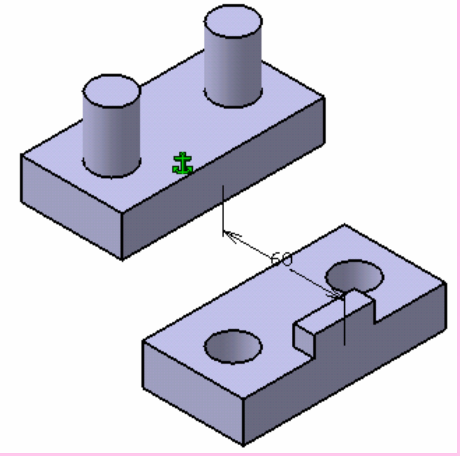
The Constraint Properties dialog box



Faces to be selected



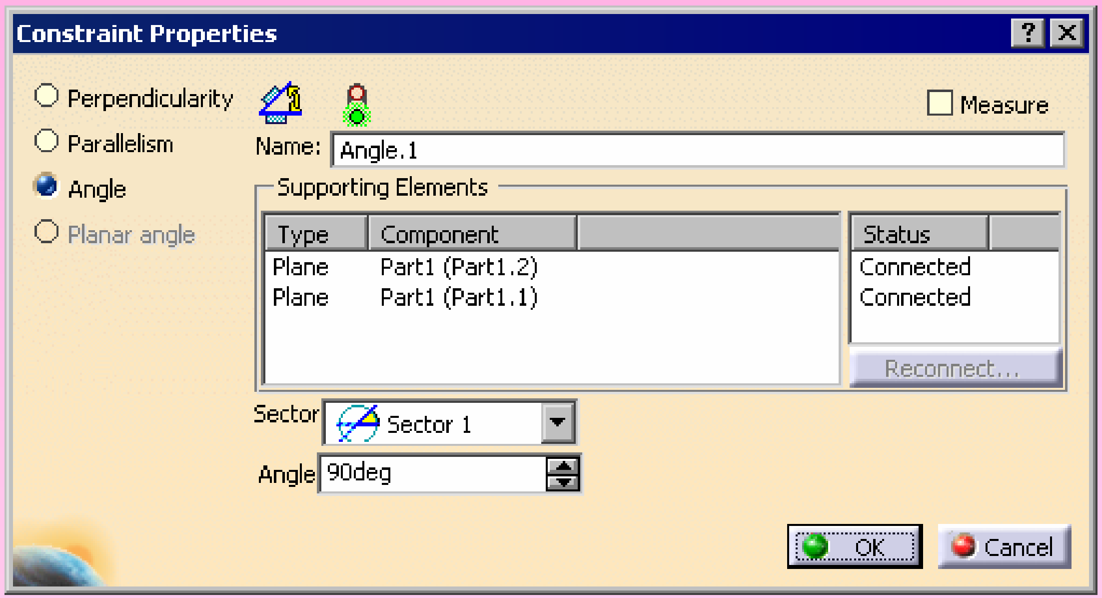
Arrows in the same direction



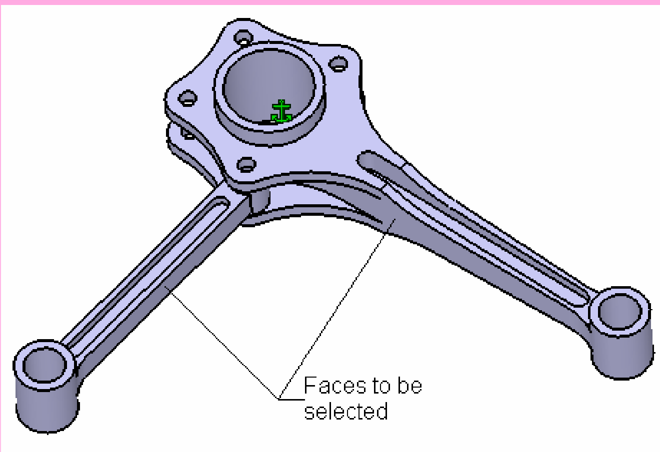
Components after updating

• Angle Constraint

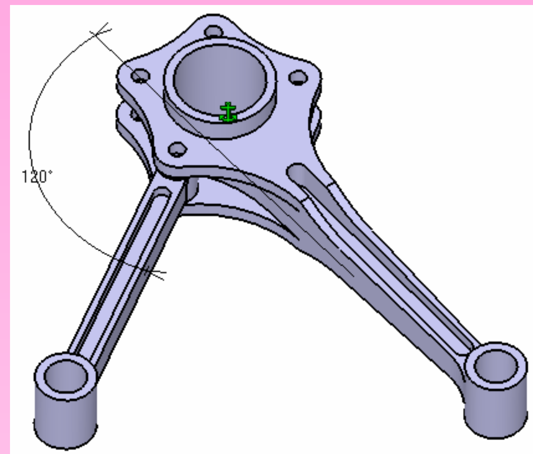
- To invoke this tool, choose the **Angle Constraint** button from the constraint toolbar.
- Now, select the two planar faces from the two different components that you need to place at some angle from each other.
- Once the selection is complete, the **Constraint Properties** dialog box is displayed, as shown in the figure.



The Constraint Properties dialog box for the Angle constraint



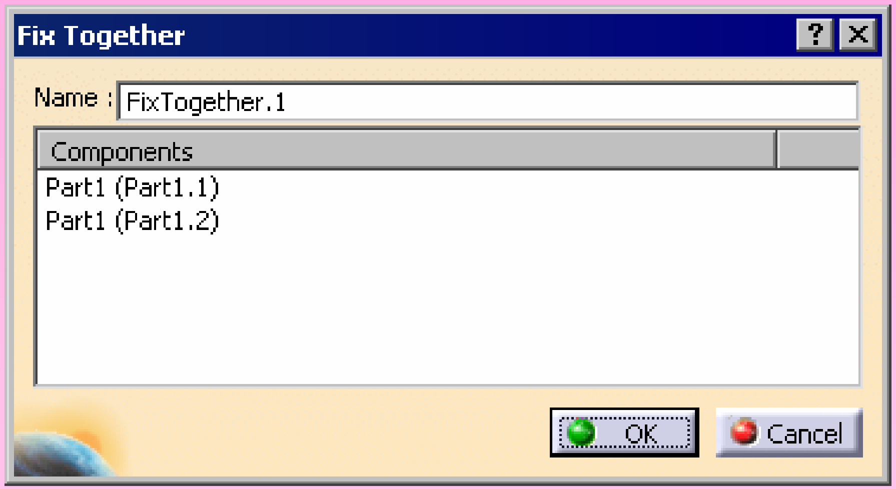
Faces to be selected



The orientation of the faces after applying the **Angle** constraint and updating

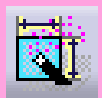
- **Fix Together**

To invoke this tool, choose the **Fix Together** button from the **Constraints** toolbar; the **Fix Together** dialog box is displayed.



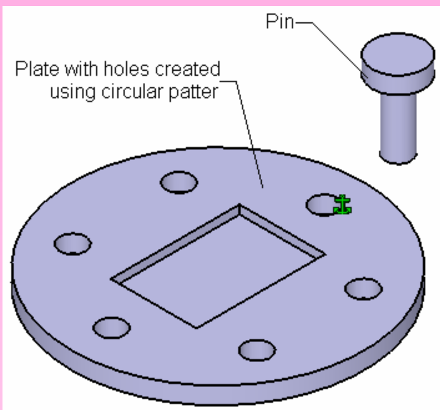
The Fix Together dialog box

- **Quick Constraint**

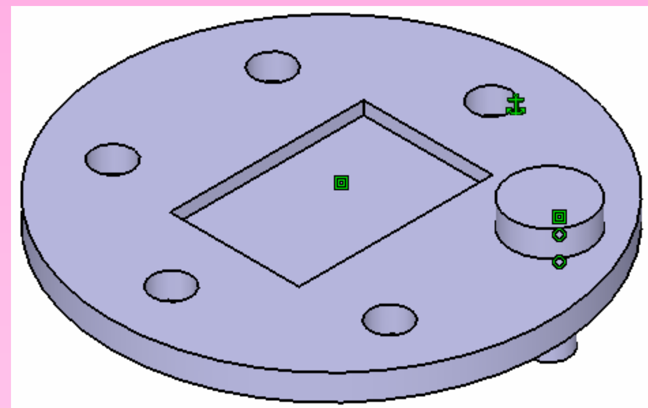


To apply constraints, choose the **Quick Constraint** button from the **Constraints** toolbar.

- **Reuse Pattern**

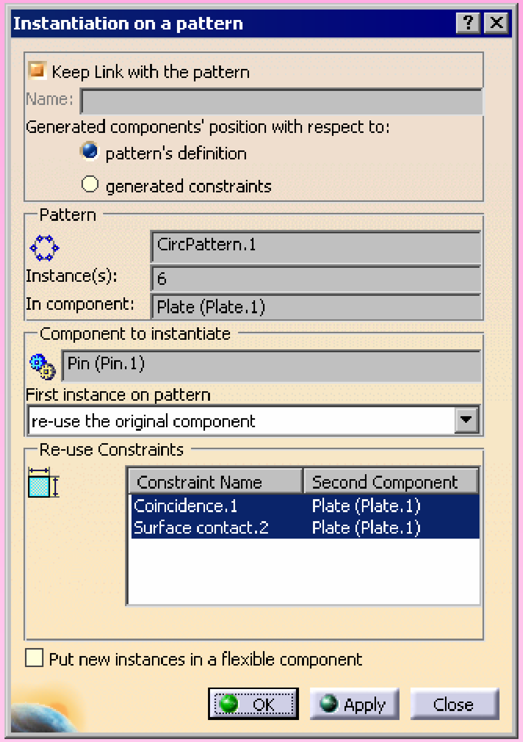


The Pin and the Plate having patterned holes

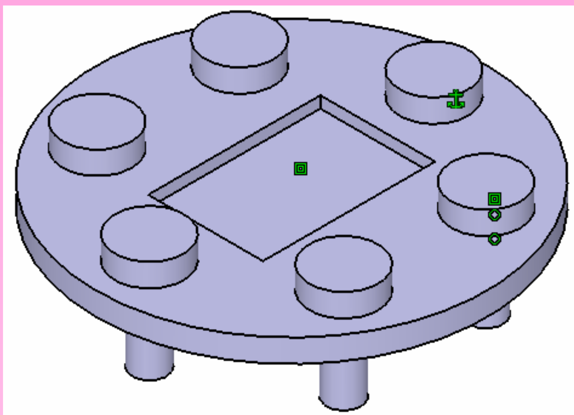


The Pin assembled to one of the instances of the patterned hole

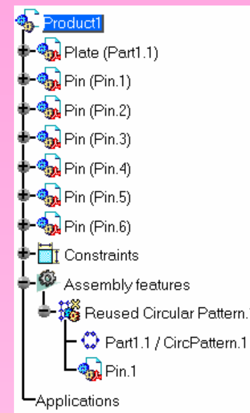
- After selecting the **Coincidence** constraint, choose the **Reuse Pattern** button from the **Constraints** toolbar.
- The preview of Pins assembled with all instances of hole is displayed in the geometry area.
- The **Instantiation on a pattern** dialog box is displayed, as shown in the the figure.



The Instantiation on a pattern dialog box



The assembly after the selected component is patterned



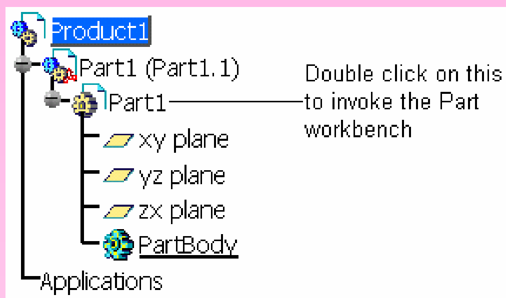
The Specification tree after creating the component pattern

- **Inserting Existing Components With Positioning**

The **Existing Component With Positioning** tool is used to insert, position, and apply constraints to a component in a single operation and is an enhanced form of the **Insert Existing Component** tool.

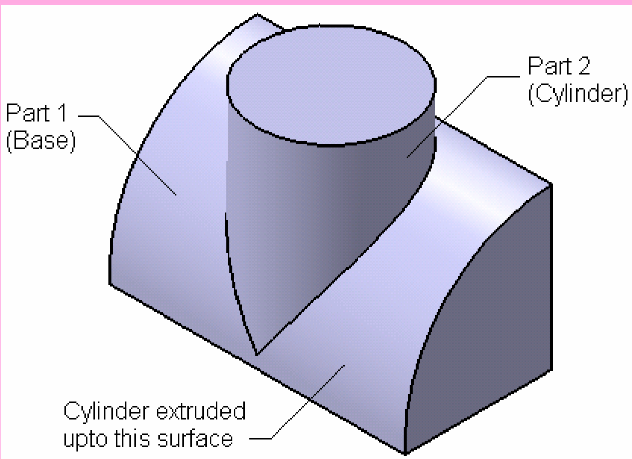
➢ CREATING TOP-DOWN ASSEMBLIES

- **Creating Base Part in Top-Down Assembly**

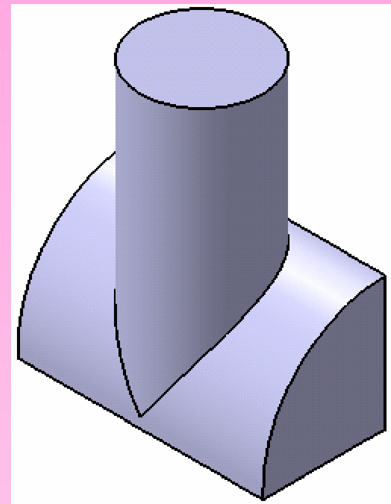


The fully expanded specification tree after inserting a part in the product file

- **Creating Subsequent Components in the Top-down Assembly**



Two different parts created in a product file and the cylinder extruded up to surface



Up to surface relation maintained even after moving the base downward

Creating Subassemblies in Top-down Assembly

- While creating complicated assemblies, you may need to have subassemblies inside an assembly.

- In CATIA V5, there are two types subassemblies that can be created in the **Assembly Design** workbench: **Product** and **Component**.

Product Subassemblies

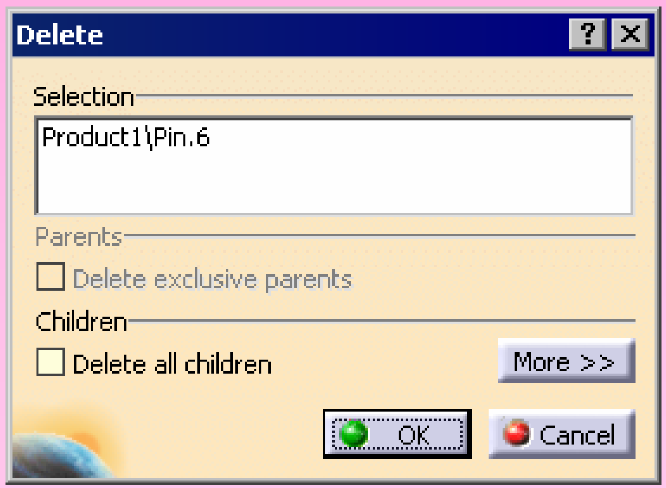
Component Subassemblies

The specification tree having a product and a component within an assembly file.

EDITING ASSEMBLIES

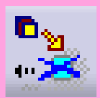
• Deleting Components

While working in the **Assembly Design** workbench, you may need to delete some of the constituent parts and subassemblies.

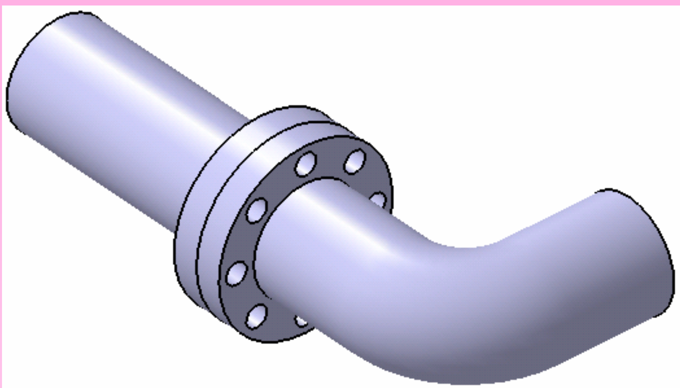


The Delete dialog box

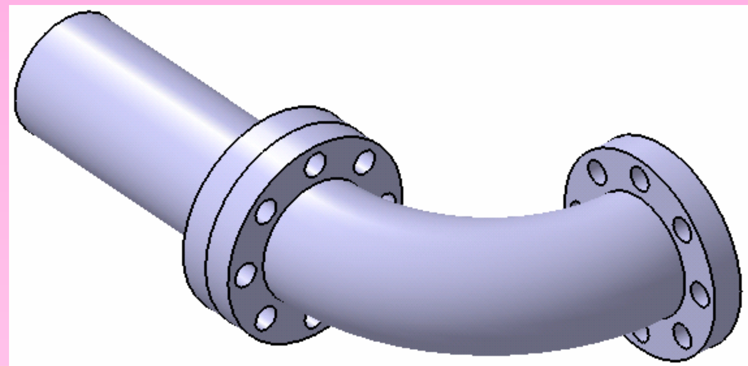
• Replacing Components



In CATIA V5 you can replace an existing component with another component inside an assembly.



The original component

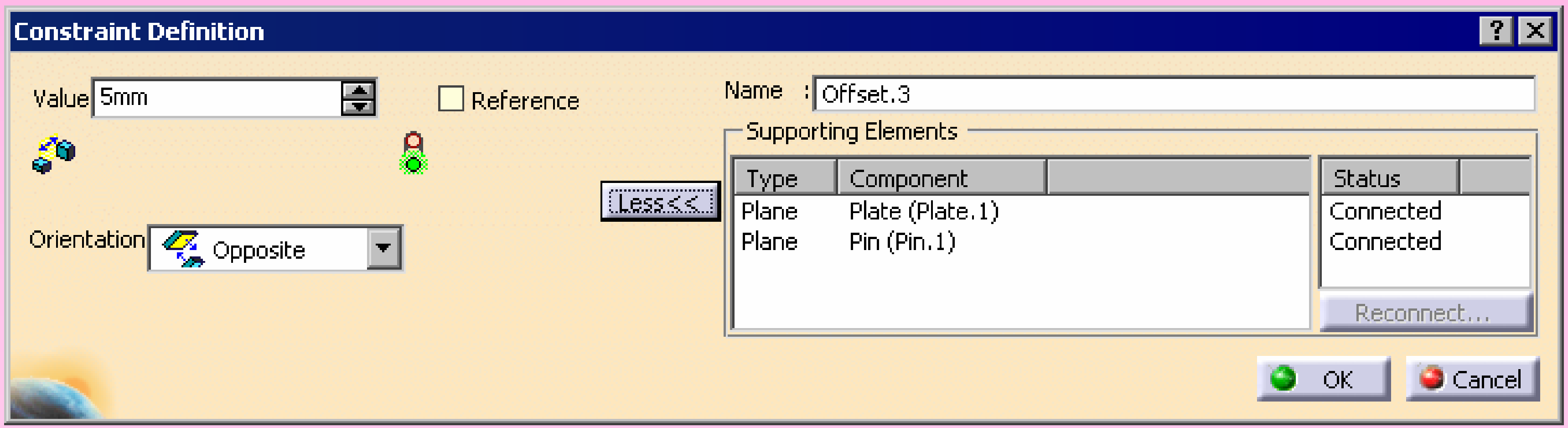


The replaced component

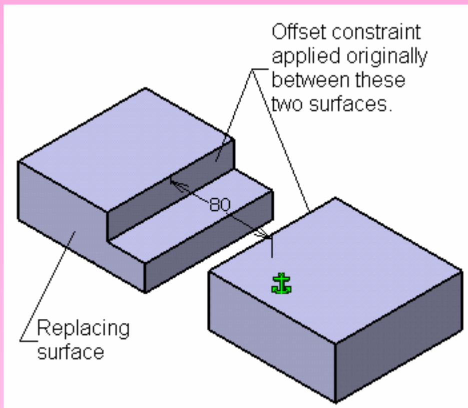
- **Editing Components Inside an Assembly**

  You can also edit the features and modify the sketches of the parts of assembly within the **Assembly Design** workbench.

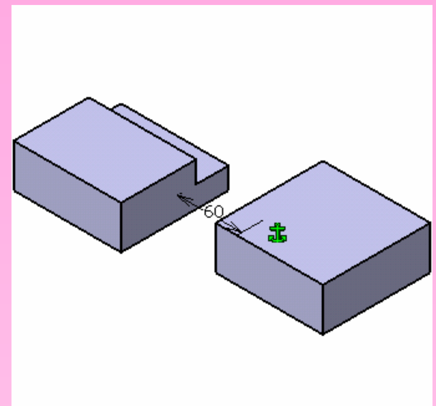
- **Editing Subassemblies Inside an Assembly**
- **Editing the Assembly Constraints**
  - **Editing the Constraint Definition**



The Constraint Definition dialog box

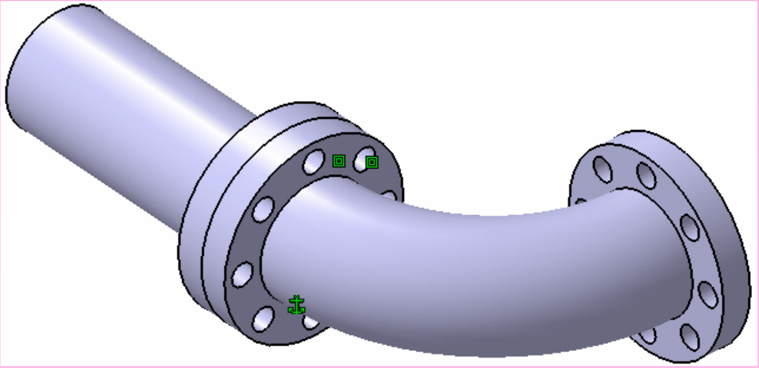


The associated and replacing surface for the **Offset** constraint

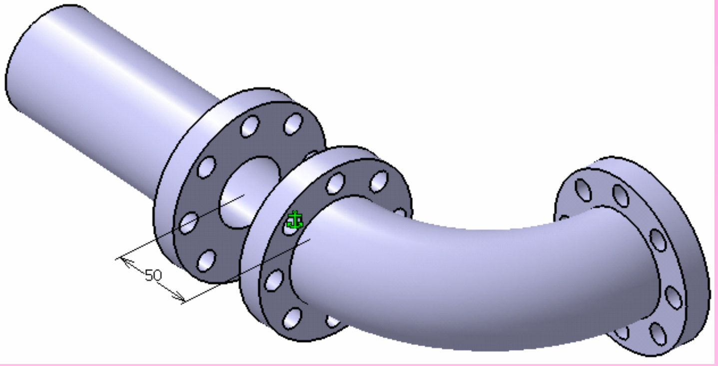


The components after editing the offset constraint and updating it

- Replacing a Constraint



Contact constraint to be replaced by the **Offset** constraint



The components after applying the **Offset** constraint

- **Simplifying the Assembly**

While working on large assemblies consisting of a large number of parts and subassemblies, it is recommended to hide some of the parts to improve the visibility of other parts and to suppress the parts that are not required at that particular stage of design cycle.

- **Hiding a Component**

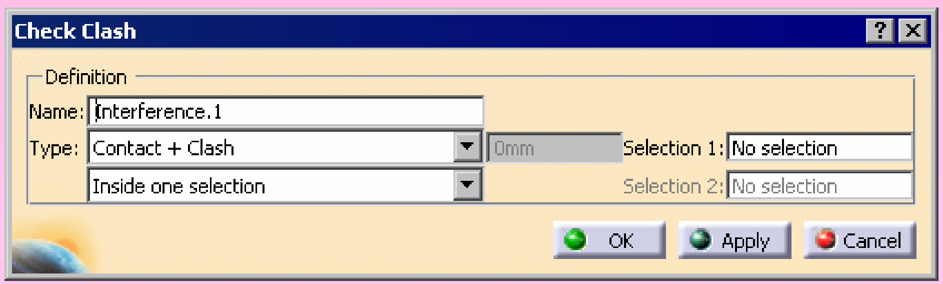
The **Hide**/**Show** tool is used turn off the display of the selected component of the assembly.

- **Deactivating a Component**

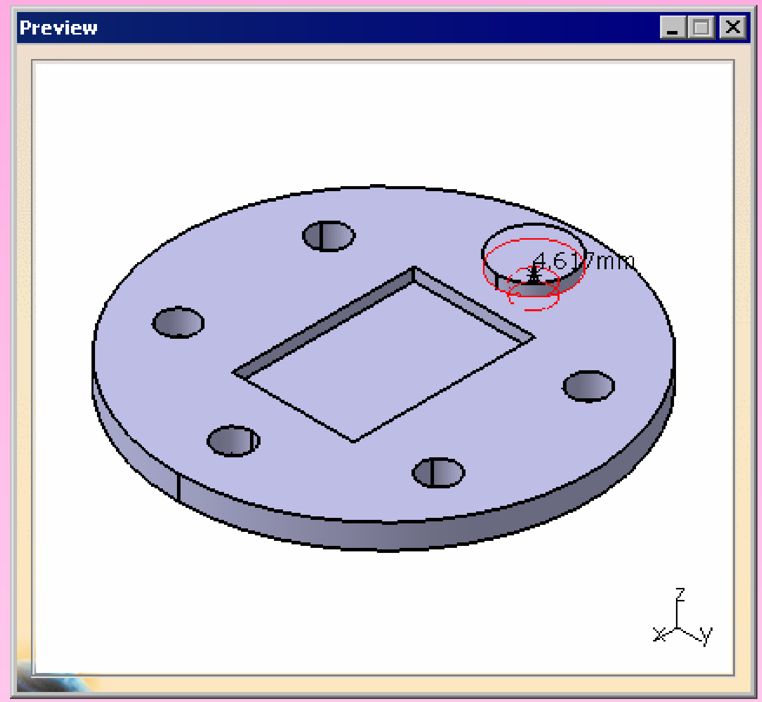
Deactivating the component removes it temporarily from the assembly.

• Interference Detection

The interference is detected using the **Clash** tool, which is invoked by choosing the **Clash** button from the **Space Analysis** toolbar. The **Check Clash** dialog box is displayed, as shown in the figure.



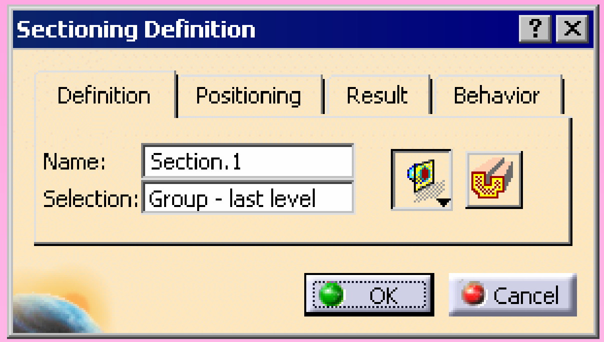
The Check Clash dialog box



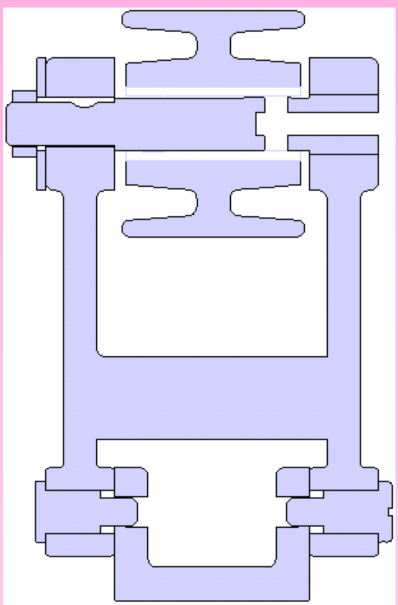
The Preview window

• Sectioning an Assembly

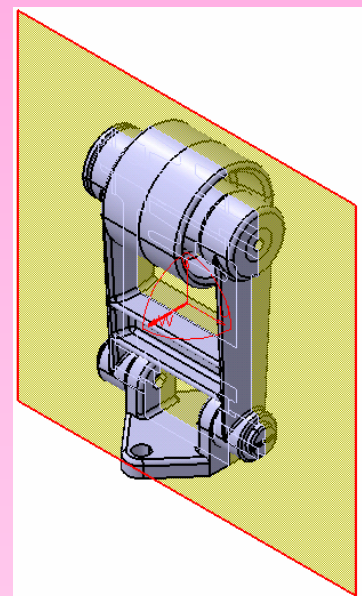
To section an assembly model, choose the **Sectioning** button from the **Space Analysis** toolbar; the **Sectioning Definition** dialog box is displayed, as shown in the figure.



The Sectioning Definition dialog box

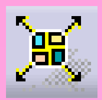


The 2D section view of the complete assembly

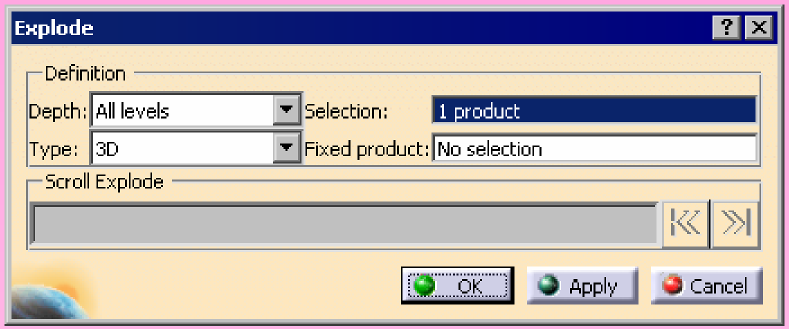


The sectioning plane

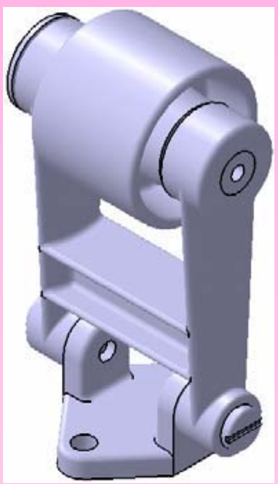
• Exploding an Assembly



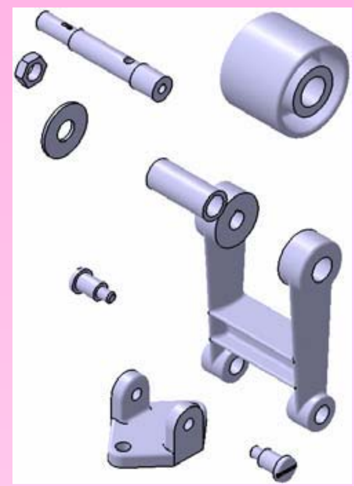
To explode an assembly, choose the **Explode** button from the **Move** toolbar; the **Explode** dialog box will be displayed, as shown in the figure.



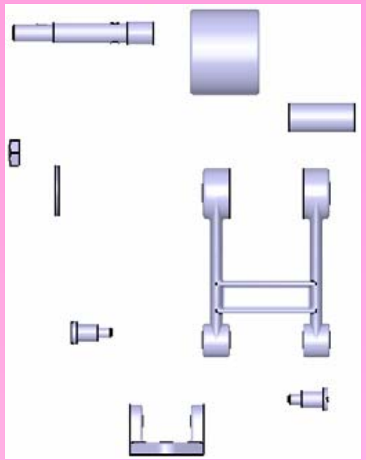
The **Explode** dialog box



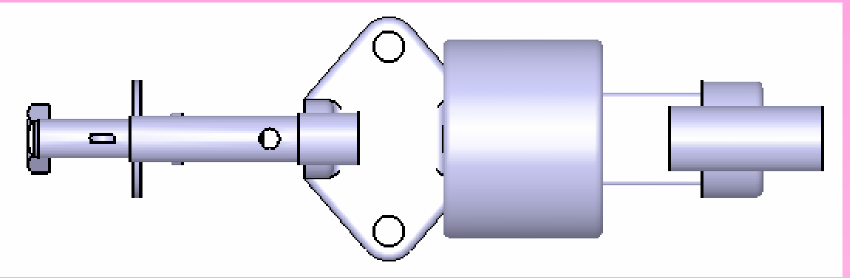
The Belt Tightener in the assembled state



Overlapping components in 3D explosion of the assembly



Front view of the exploded Belt Tightener assembly exploded using the 2D option



Top view of the exploded Belt tightener assembly

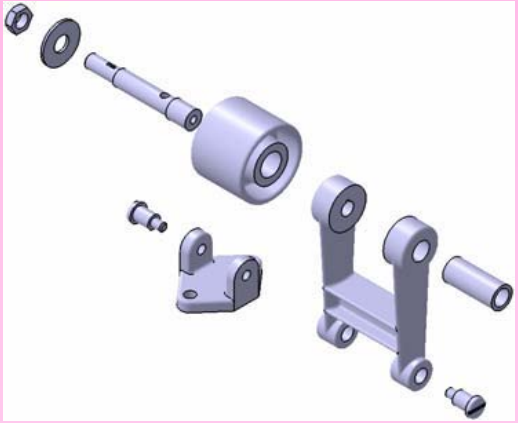


Figure showing the exploded assembly with Constrained selected as the type

Tutorial 1

In this tutorial you will create all the components of the Blower assembly and then assemble them together. The Blower assembly is shown in **Figure A**. After creating it, you will generate the exploded view. The exploded view of the Blower assembly is shown in **Figure B**. The dimensions of all components are given in **Figure C**, **Figure D**, **Figure E**, **Figure F**, **Figure G** and **Figure H**. **(Expected time: 2.5 hrs)**

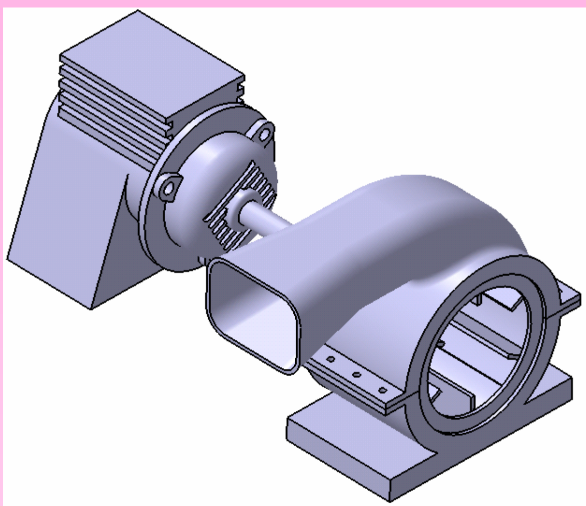


Figure A The Blower Assembly

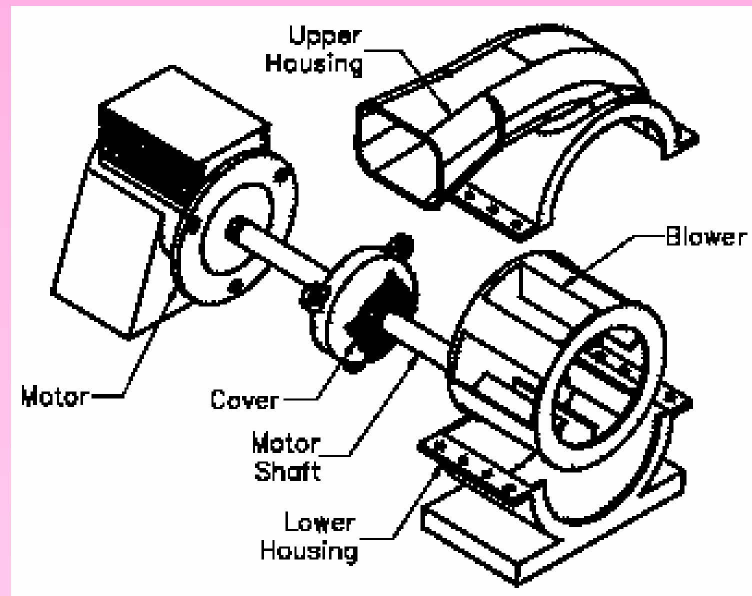
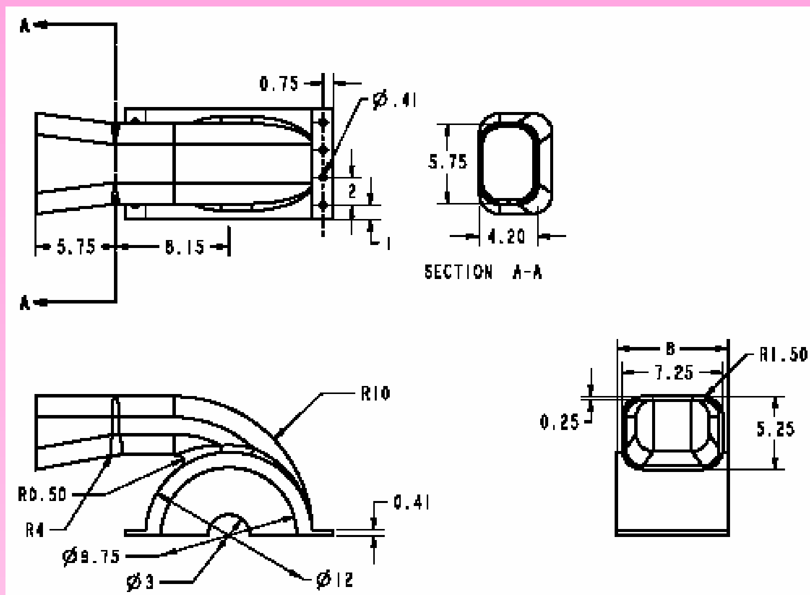


Figure B Exploded view of the Blower assembly



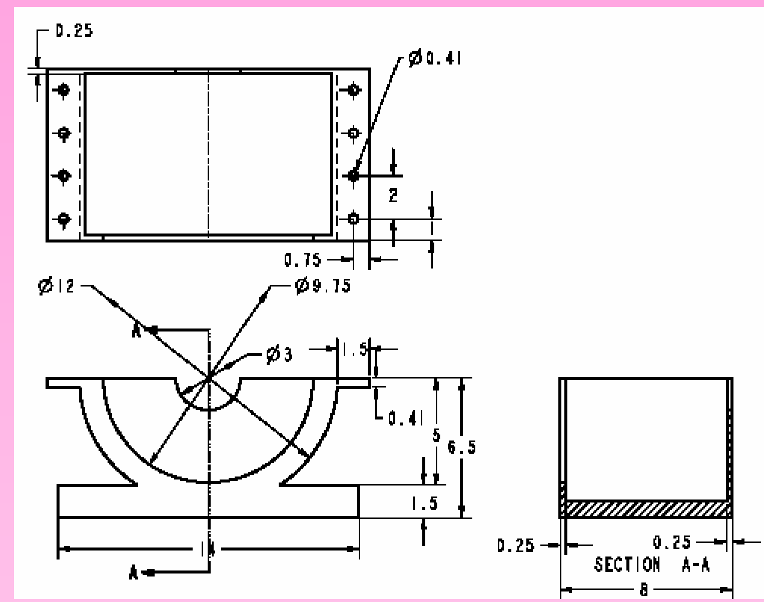


Figure C Views and dimensions of the Upper Housing

Figure D Views and dimensions of the Lower Housing

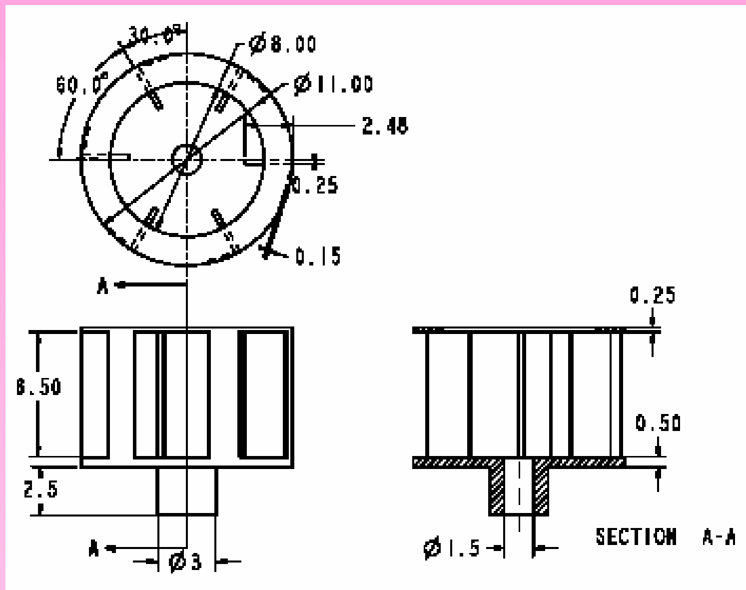


Figure E Views and dimensions of the Blower

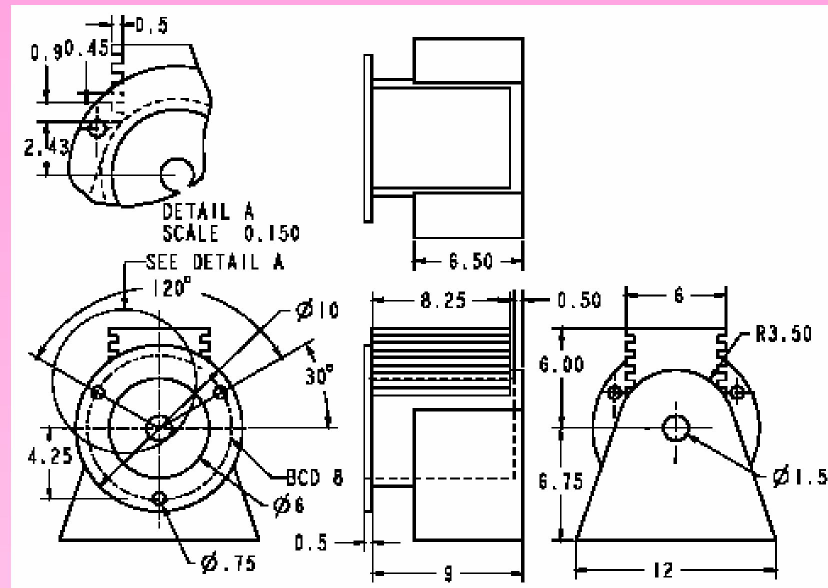


Figure F Views and dimensions of the Motor

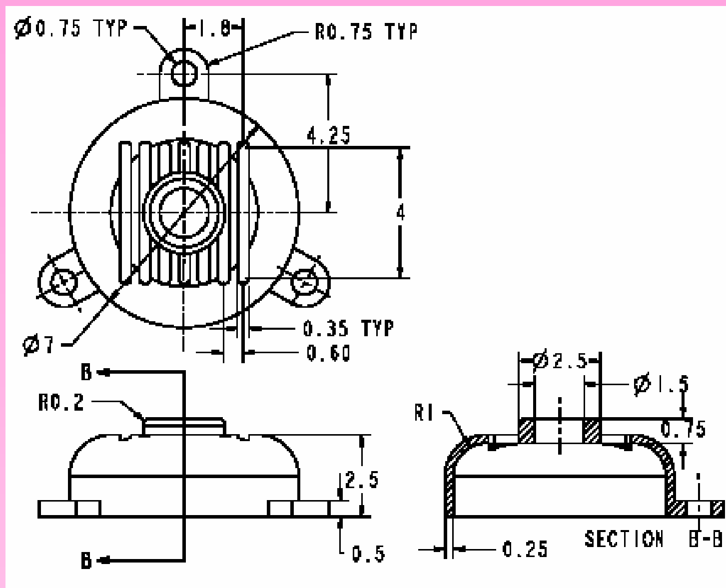


Figure G Views and dimensions of the Cover

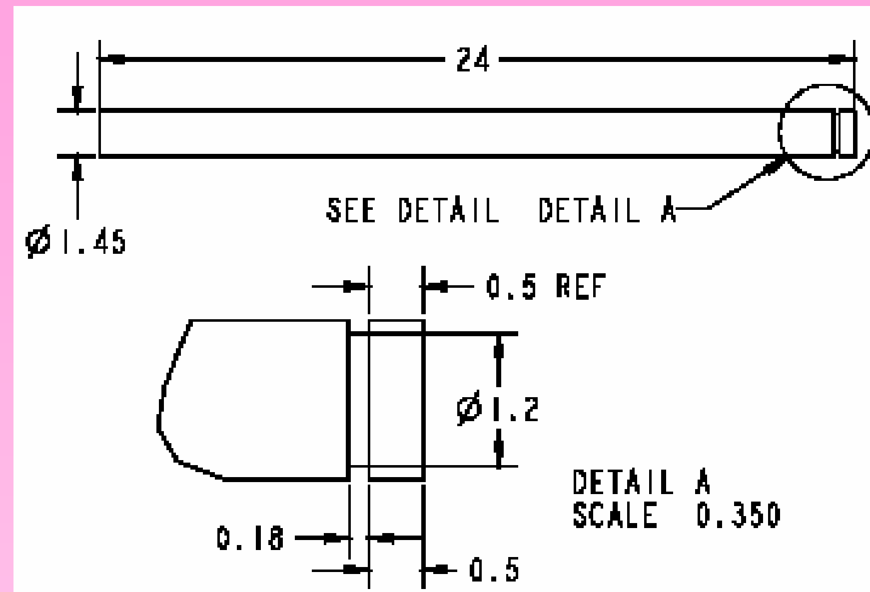


Figure H Views and dimensions of the Motor Shaft

1. Create all components of the assembly as separate part files in the **Part Design** workbench.
2. Start a new file in the **Assembly Design** workbench.
3. Insert the Lower Housing into the assembly as the base component, set its orientation, and apply the **Fix** constraint to it at its default location, as shown in **Figure I**, **Figure J** and **Figure K**.

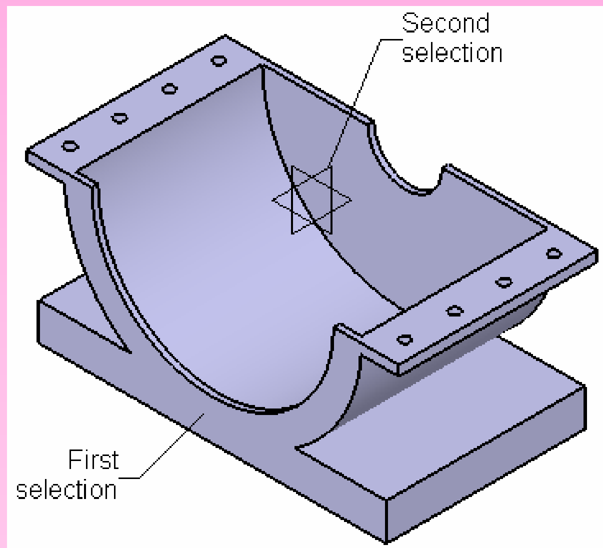


Figure I First and second elements to be selected

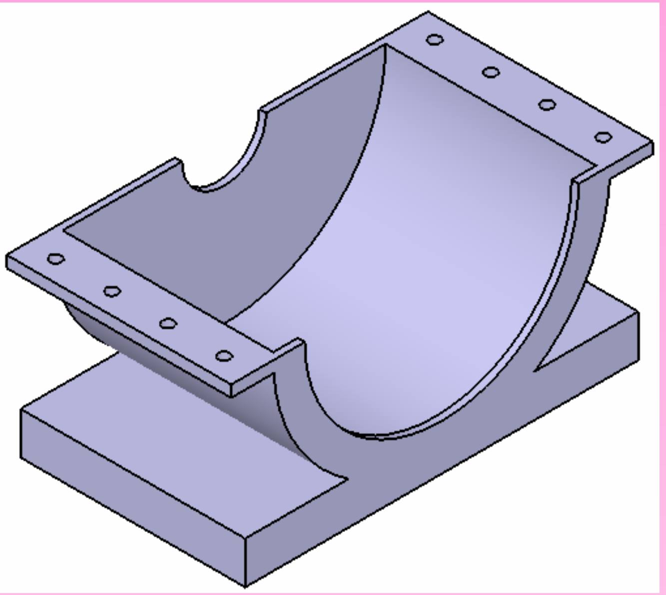


Figure J The Lower Housing after modifying its orientation

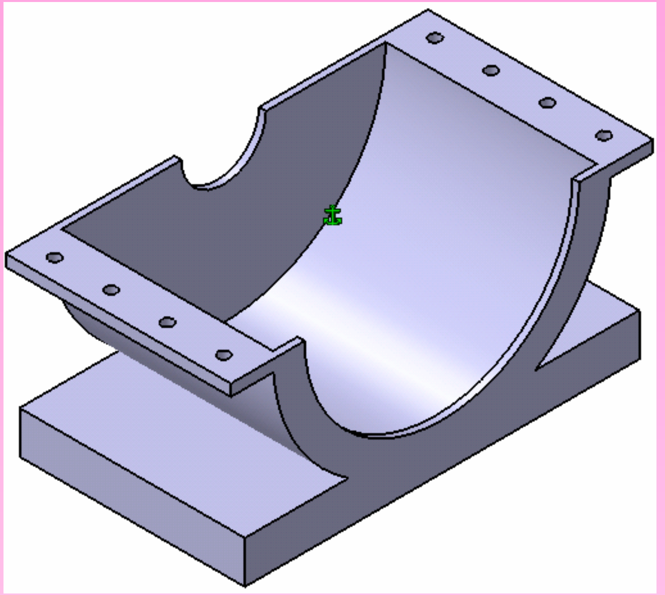


Figure K Lower Housing after it is fixed at its default location

4. Insert the Upper Housing into the assembly and place it over the lower housing by applying proper constraints, as shown in **Figure L**, **Figure M** and **Figure N**.

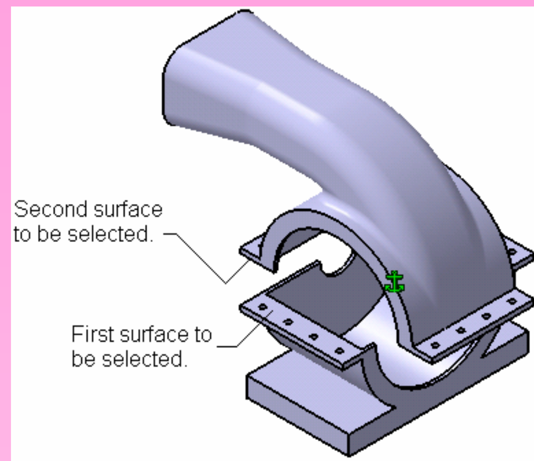


Figure L The surfaces to be selected to apply Contact constraint

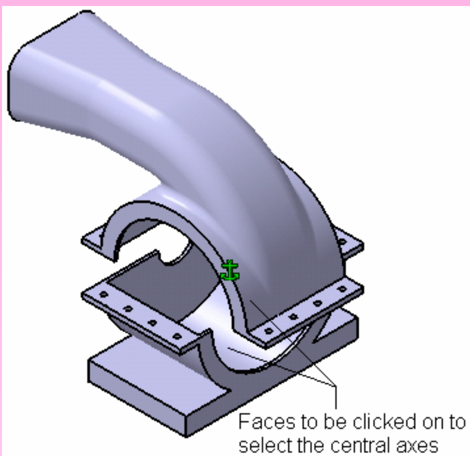


Figure M Surfaces on which you need to click to select the central axes

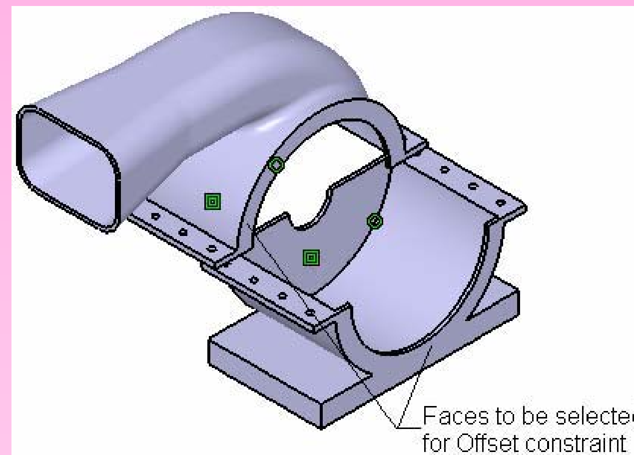


Figure N The surfaces to be selected for applying Offset constraint

5. Hide the Upper Housing. Insert and place the blower inside the lower Housing.
6. Now, insert and constrain the Motor, the Motor Shaft, and the Cover refer to **Figure O**, **Figure P**, **Figure Q**, **Figure R**, **Figure S**, **Figure T** and **Figure U**.

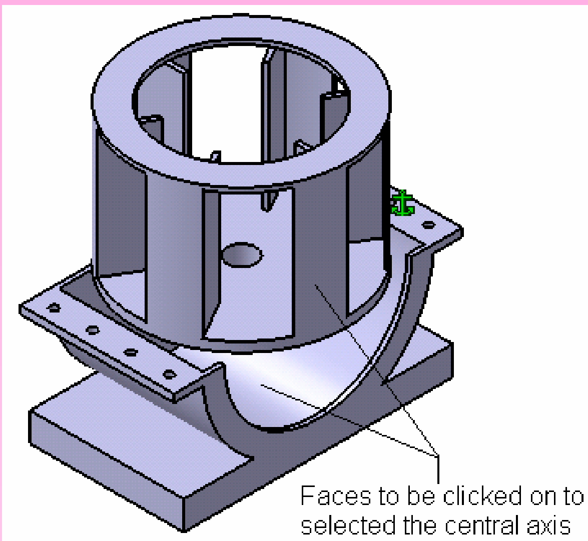


Figure O Faces to be clicked to select the central axes

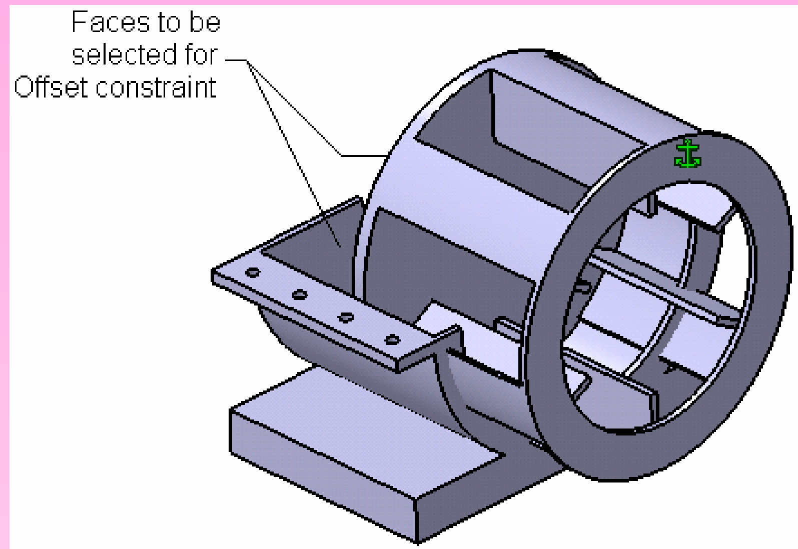


Figure P Faces to be selected for applying Offset constraint

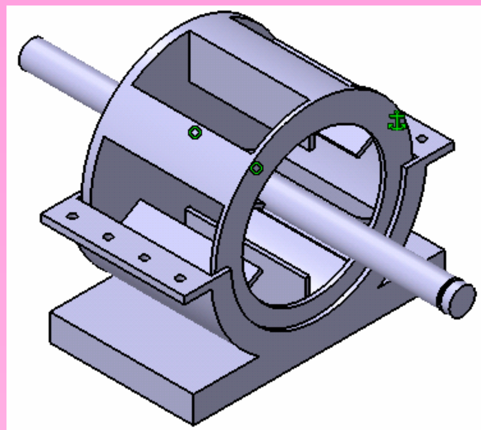


Figure Q Motor Shaft inserted at its default location

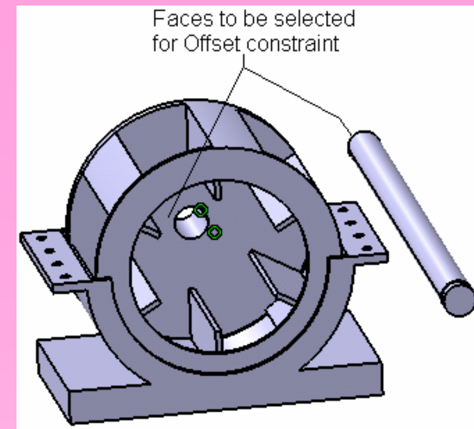


Figure R Faces to be selected for Offset constraint

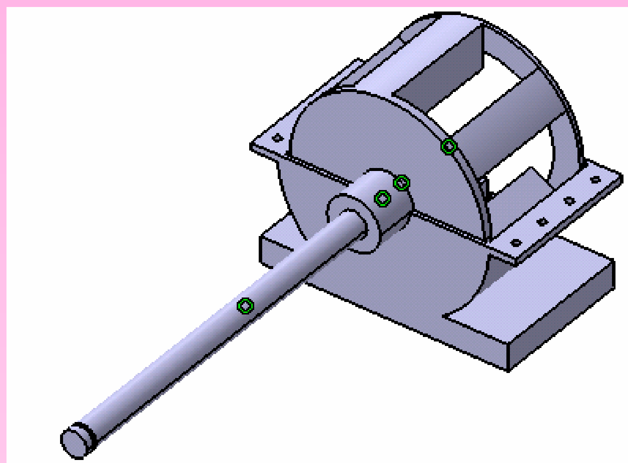


Figure S Position of the Motor Shaft with respect to the Blower shown from back side

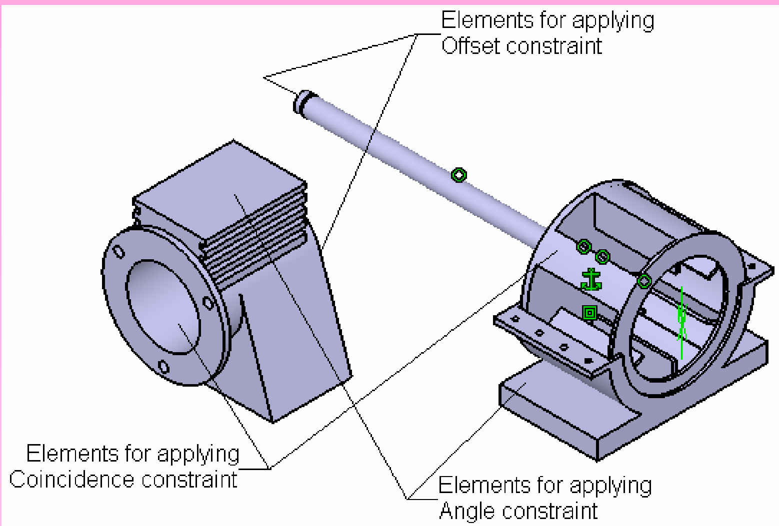


Figure T Elements to be selected for applying various constraints

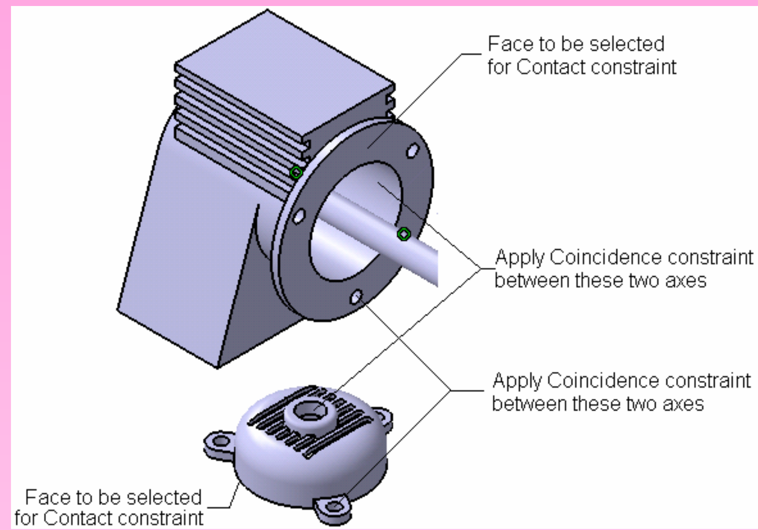


Figure U Various faces to be selected for applying the constraints

7. Turn on the display of the Upper Housing, as shown in **Figure V**.

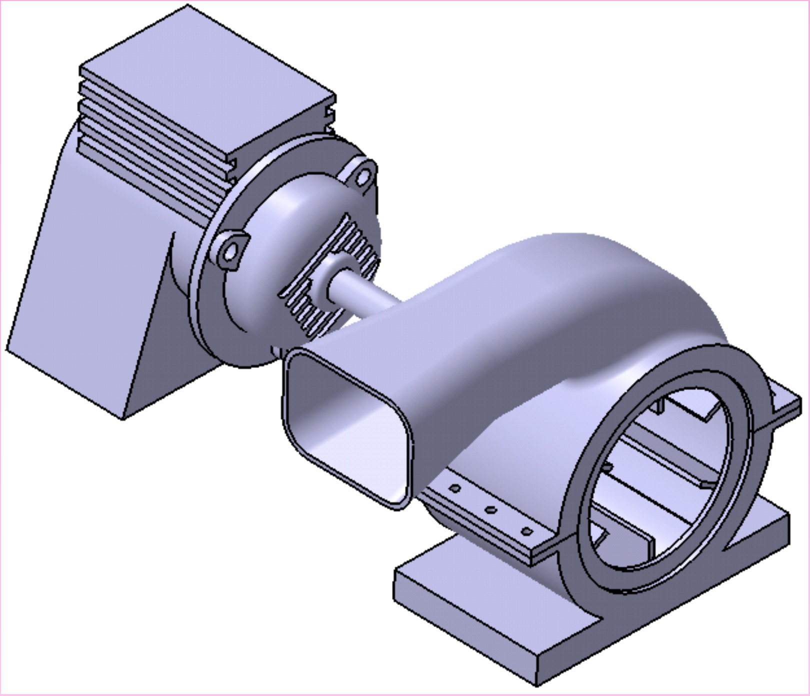


Figure V The final Blower assembly

8. Create the exploded state of the assembly, as shown in **Figure W**.

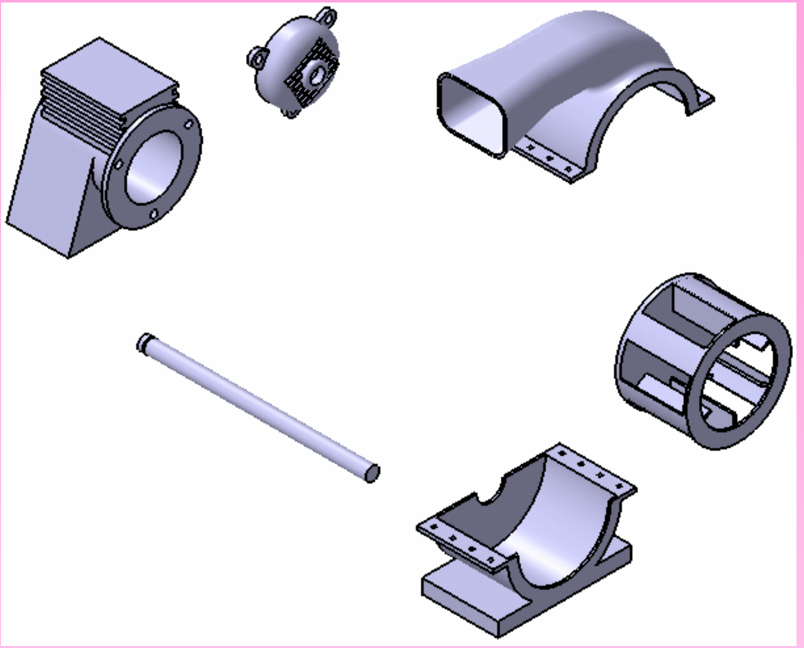


Figure W The exploded view of Blower assembly

9. Save the file in **\My Documents\CATIA\c11** folder and then close it.

Tutorial 2

In this tutorial you will create some components of a Press Tool Base assembly using the top-down assembly approach. The Press Tool Base assembly is shown in **Figure A**. The exploded state of this assembly is shown in the **Figure B**. The dimensions of all components are shown in **Figures C** and **Figure D**. **(Expected time: 45 min)**

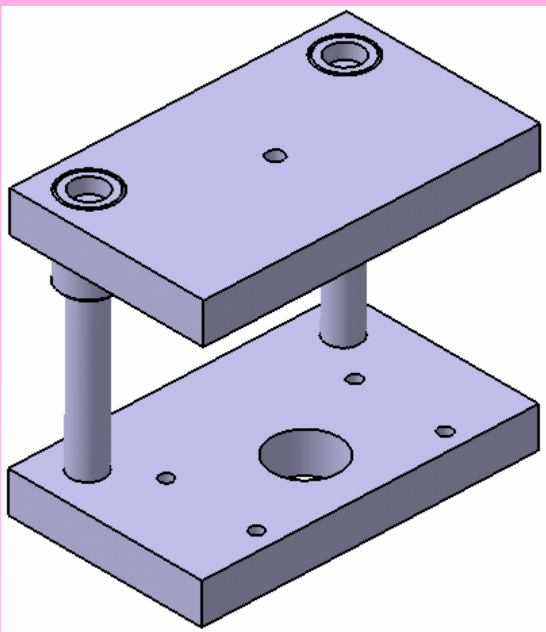


Figure A The Press Tool Base assembly

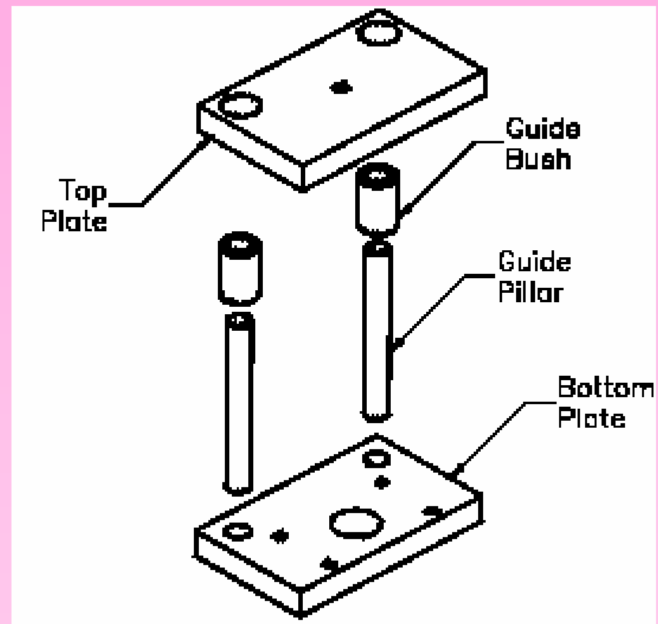


Figure B The exploded state of the Press Tool Base assembly

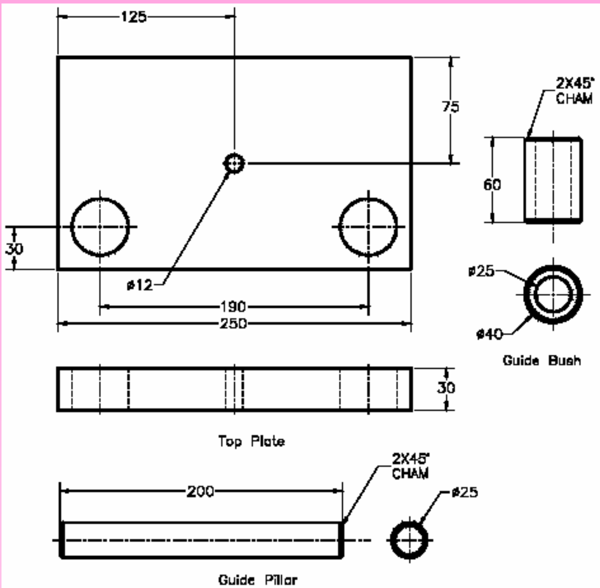


Figure C Views and dimensions of the Top Plate, Guide Pillar, and Guide Bush

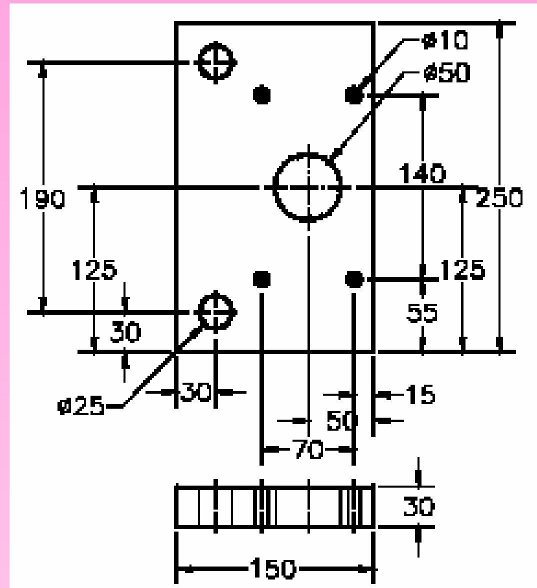


Figure D Views and dimensions of the Bottom Plate

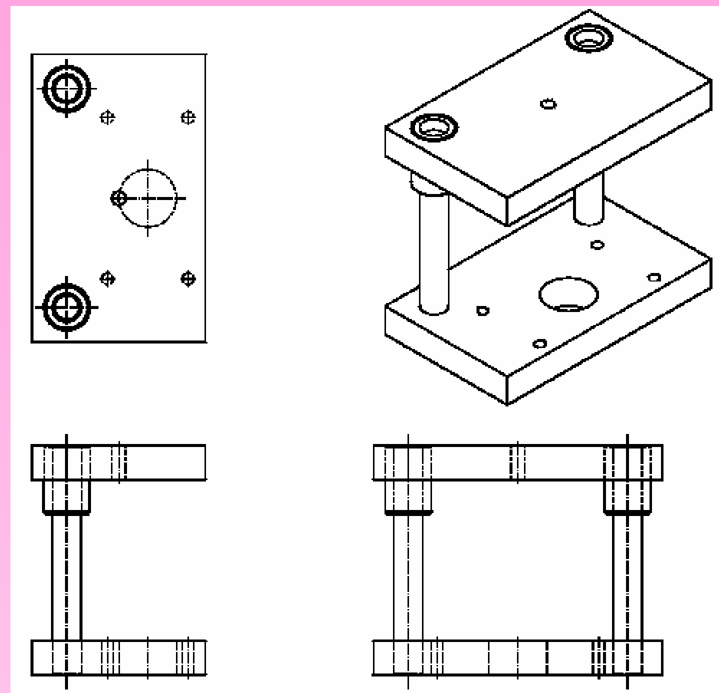


Figure E Drawing views of the Press Tool Base assembly

1. Start a new product file.
2. Create a new part inside the assembly. Modify its name and create features of the base component, as shown in **Figure F**. In this assembly the Bottom Plate will be the base component.

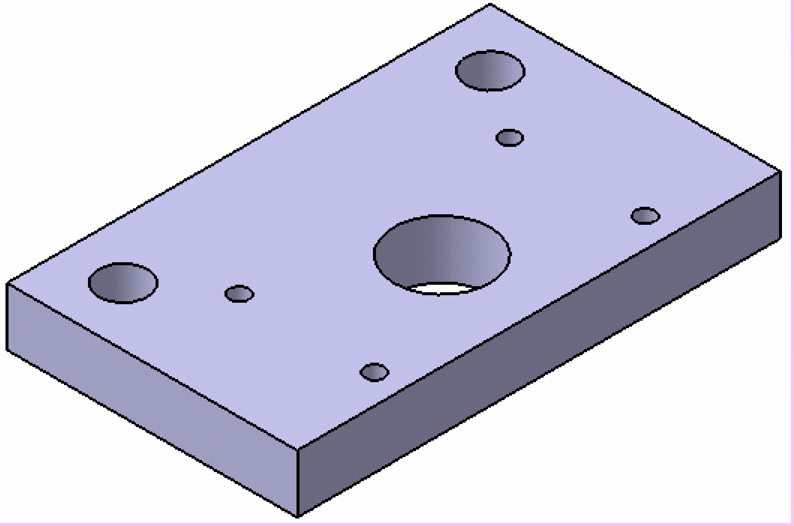


Figure F The final model of the Bottom Plate

3. Create the Guide Bush and Guide Pillar as subsequent components inside the product file, as shown in **Figure G**, **Figure H** and **Figure I**.

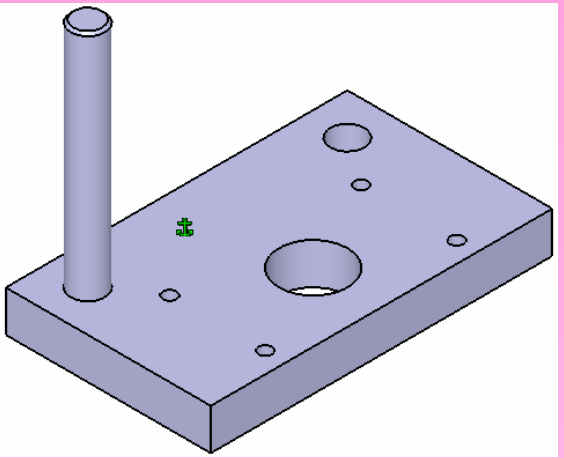


Figure G The final Guide Pillar

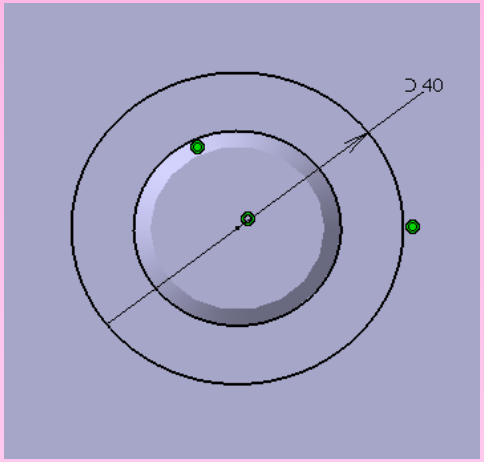


Figure H Sketch of Pad feature for creating Guide Bush

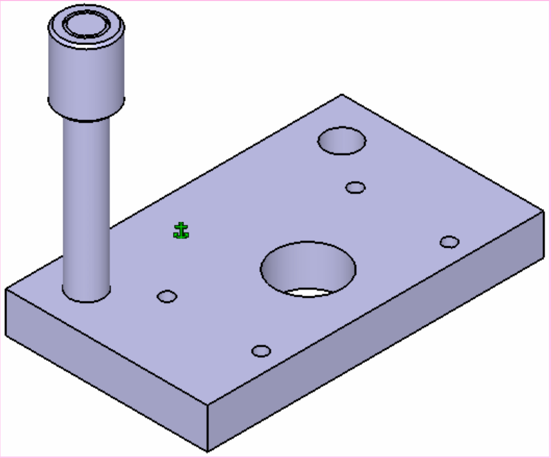


Figure I Final Guide Bush

4. Guide Pillar and Guide Bush are to be duplicated using the **Reuse Pattern** tool, as shown in **Figure K**.

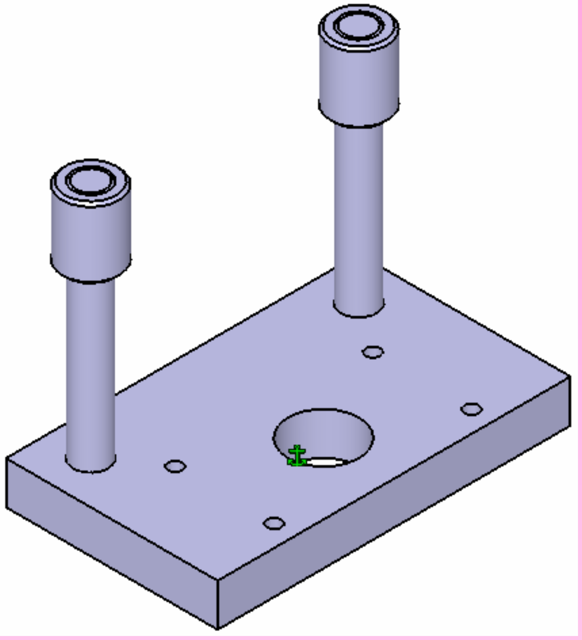


Figure K The assembly after placing the second set of Guide Pillar and Guide Bush

5. Create the Top Plate, as shown in **Figure L**.

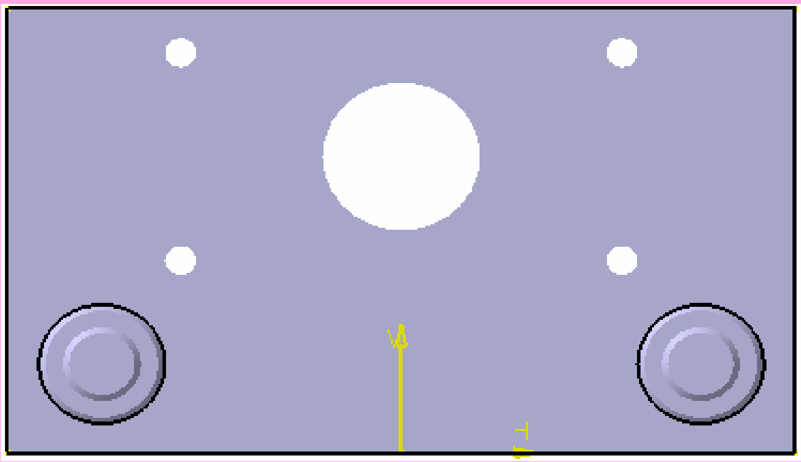


Figure L Sketch of the Pad feature for creating Top Plate

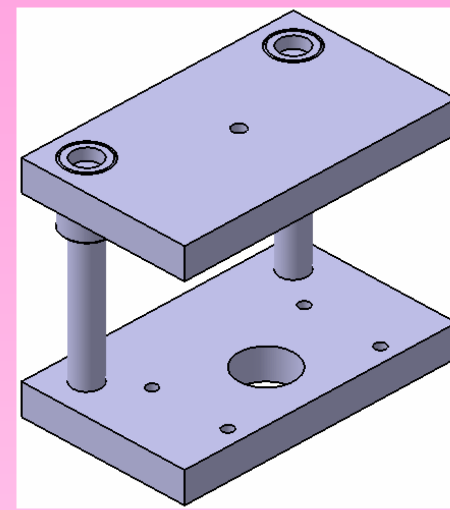


Figure M The final Press Tool Base assembly

6. Save the file in **\My Documents\CATIA\c11** folder and then close it.

Exercise 1

Create the assembly of the Radial Engine shown in **Figure A**. The assembly in the exploded state is shown in **Figure B**. Note that this exploded view is provided only for your understanding and has not been generated using CATIA V5. The dimensions of various parts of this assembly model are given in **Figure C** and **Figure D**.

(Expected time: 3 hr 30 min)

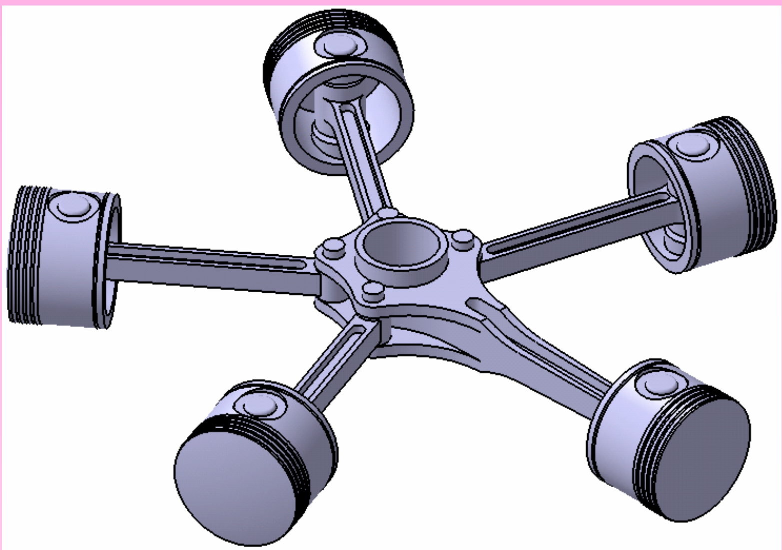


Figure A The Radial Engine assembly

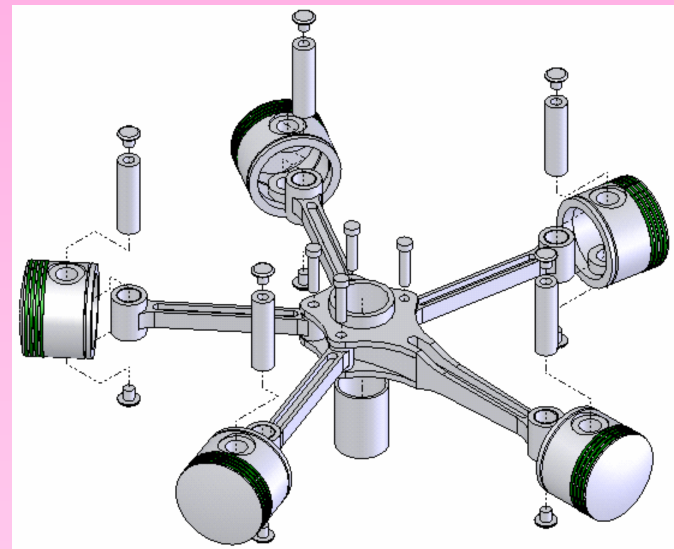


Figure B Exploded view of the Radial Engine assembly

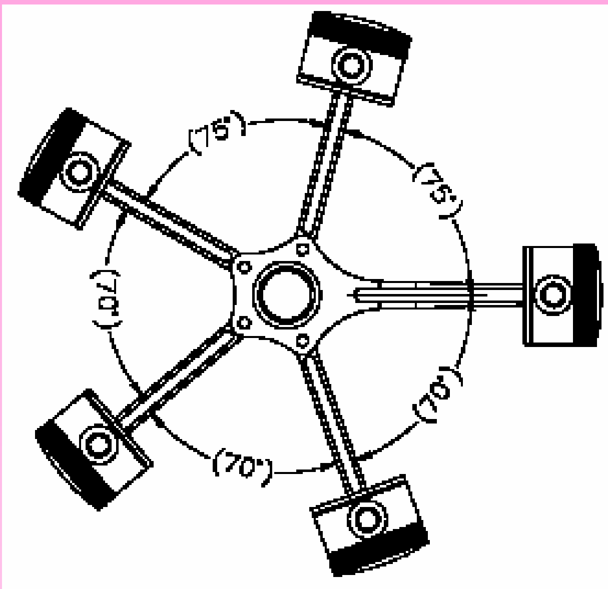


Figure C Positioning of the Articulated Rods

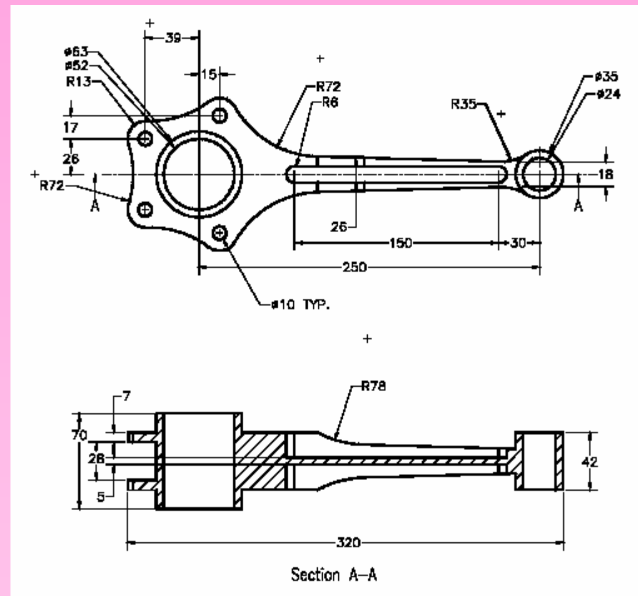


Figure D Views and dimensions of the Master Rod

Learning Objectives:

- Start new files in the Drafting workbench.
- Generate views using the View Creation Wizard.
- Generate front views.
- Generate advanced front views.
- Generate projected views.
- Generate auxiliary views.
- Generate isometric views.
- Generate section views.
- Generate aligned section views.
- Generate section cuts.
- Generate aligned section cuts.
- Generate detail views.
- Generate detail view profiles.
- Generate clipping views.
- Generate clipping view profiles.
- Generate broken views.
- Generate breakout views.
- Edit and modify drawing views.
- Modify hatch patterns of the section views.

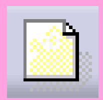
THE DRAFTING WORKBENCH

- After creating parts and assembling them, you need to generate their drawing views.
- A 2D drawing is the life line of all the manufacturing systems because on the shop floor or tool room, the machinist mostly needs the 2D drawings for manufacturing.
- CATIA V5 provides you with the **Drafting** workbench, which is a specialized environment for generating 2D drawing views.
- This workbench provides all tools required to generate drawing views, modify them, and apply dimensions and annotations to them.

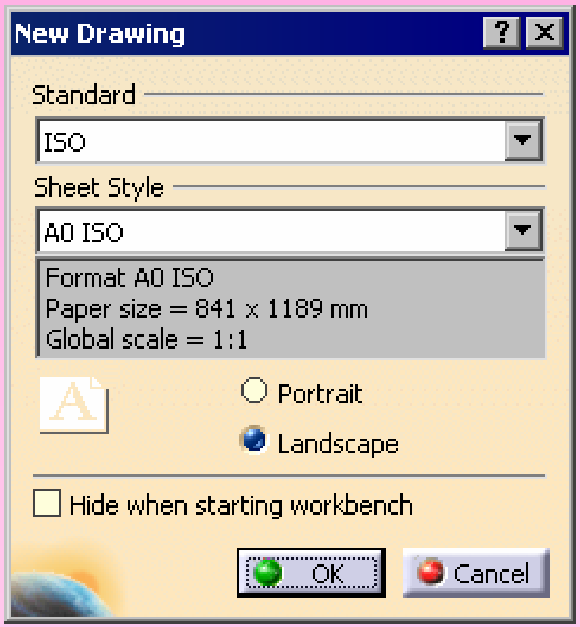
Starting a New File in the Drafting Workbench

To generate drawings views, you first need to start a new file in the **Drafting** workbench.

Starting a New File in the Drafting Workbench Using the New Tool



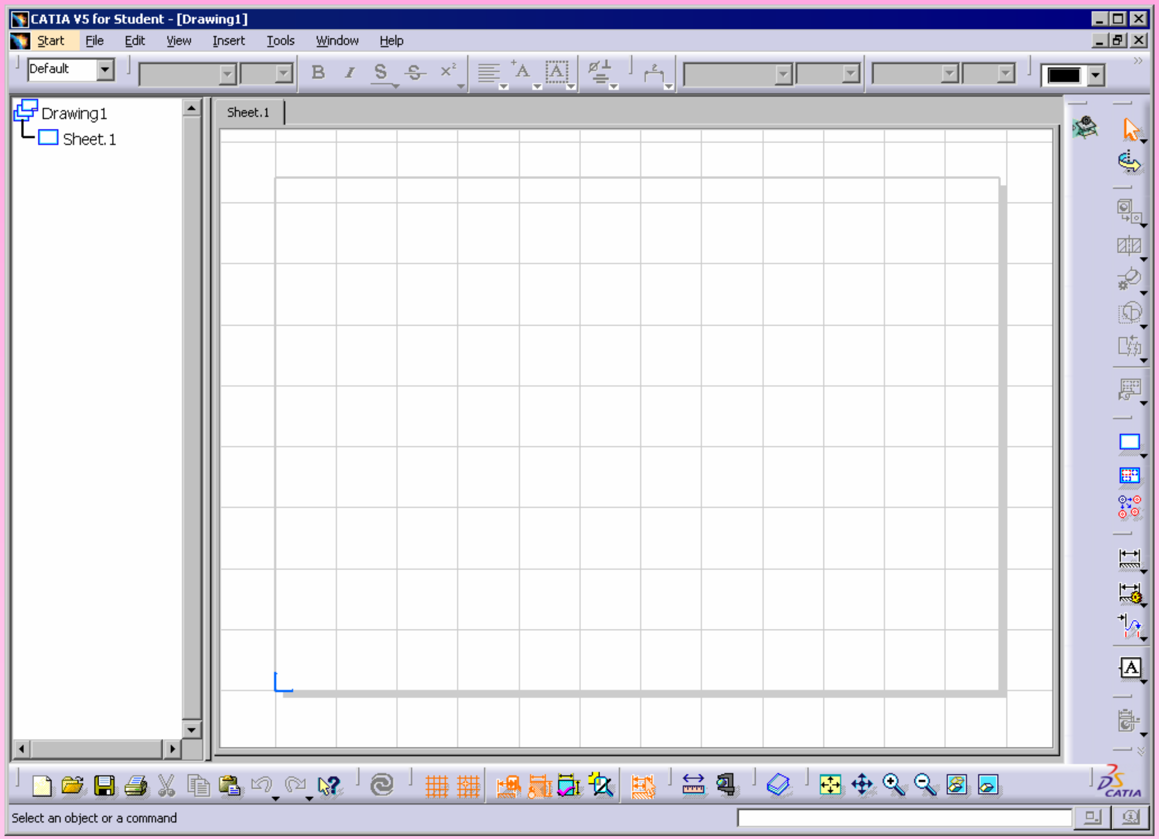
- To start a new file in the **Drafting** workbench using the **New** tool, choose the **New** button from the **Standard** toolbar.
- The **New** dialog box will be displayed. Select **Drawing** and choose the **OK** button.
- The **New Drawing** dialog box is displayed, as shown in the figure.



The New Drawing dialog box

The options available in the **New Drawing** dialog box are:

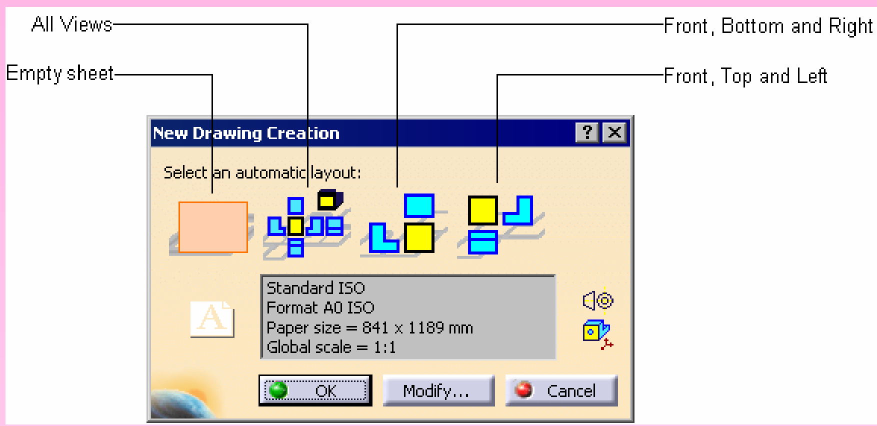
- **Standard**
- **Sheet Style**



The initial screen after starting a new file in the Drafting workbench

Starting a New File in the Drafting Workbench Using the Start Menu

- To start a new file in the **Drawing** workbench using this option, choose **Start > Mechanical Design > Drafting** from the menu bar.
- The **New Drawing Creation** dialog box is displayed, as shown in the figure.



The New Drawing Creation dialog box

The options available in the **New Drawing Creation** dialog box are:

- **Empty Sheet**
- **All Views**
- **Front, Bottom and Right**
- **Front, Top and Left**

TYPE OF VIEWS

- **Front View**
- **Projected View**
- **Section View**
- **Aligned Section View**
- **Auxiliary View**
- **Detail View**
- **Clipping View**
- **Broken View**
- **Breakout View**

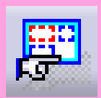
GENERATING THE DRAWING VIEWS

In CATIA V5, there are two methods of generating the views after you start an empty file in the **Drafting** workbench, using **Wizards** and generating the drawing views is generating each view one by one.

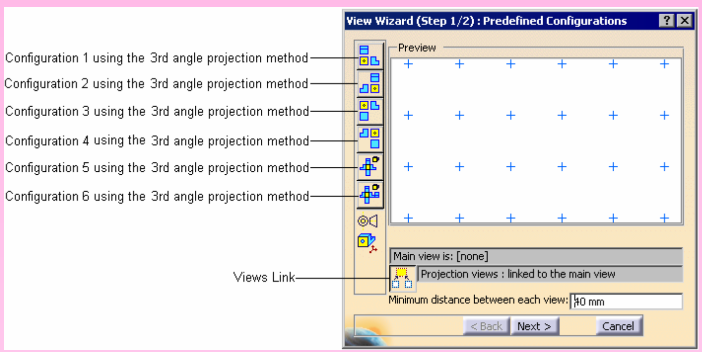
• Automatically Generating the Views

The **Wizard** toolbar

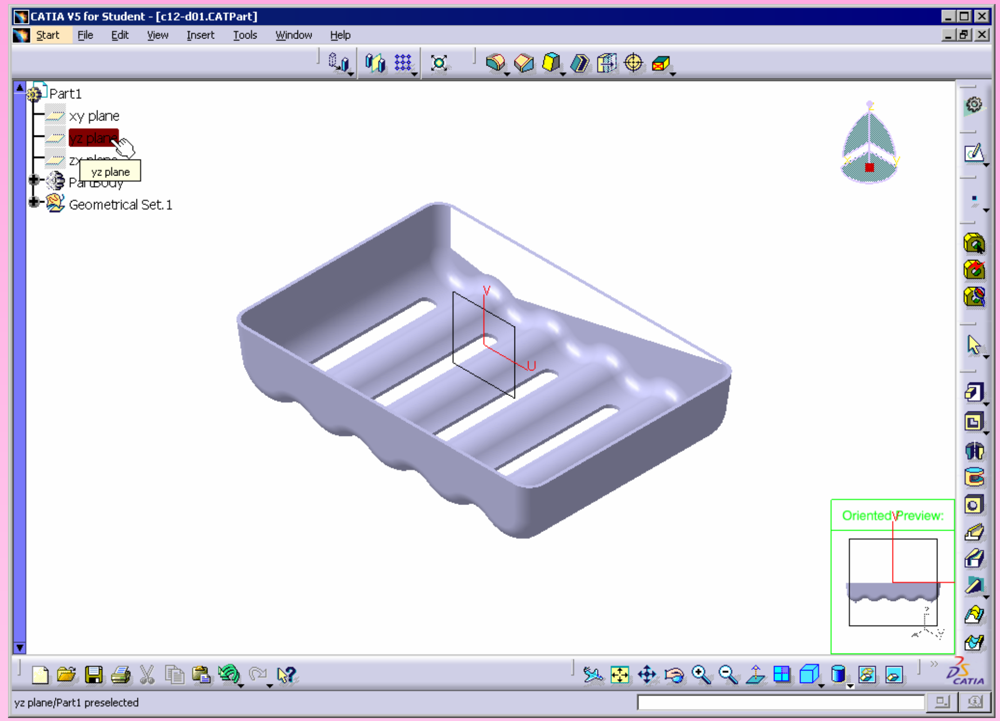
• Generating the Views Using the View Creation Wizard



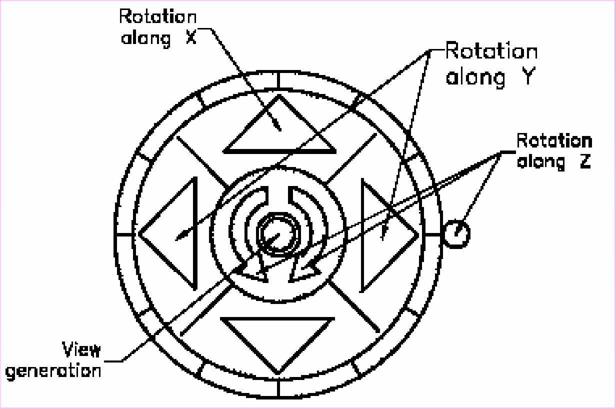
When you invoke the **View Creation Wizard** tool, the **View Wizard** dialog box is displayed, as shown in the figure.



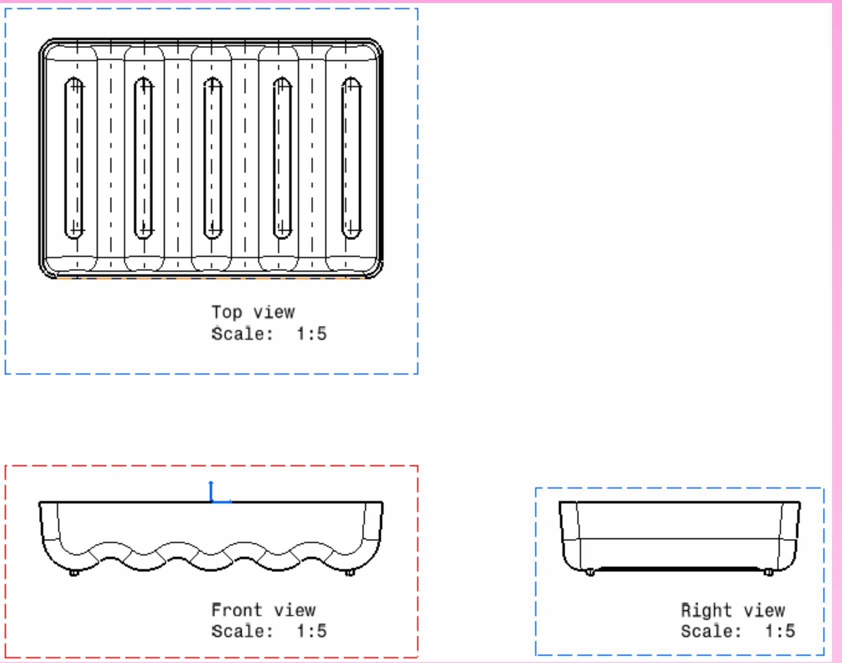
The **View Wizard** dialog box



Preview of the front view on the lower right corner of the geometry area

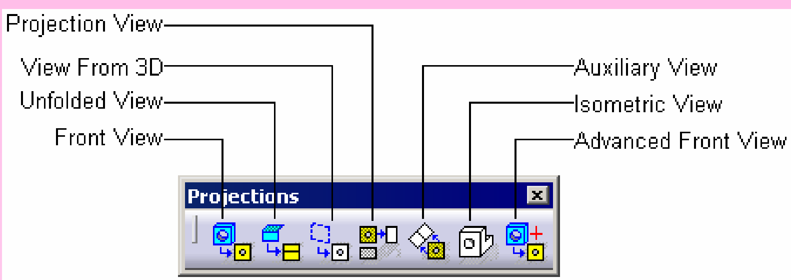


Controls provided on the blue knob

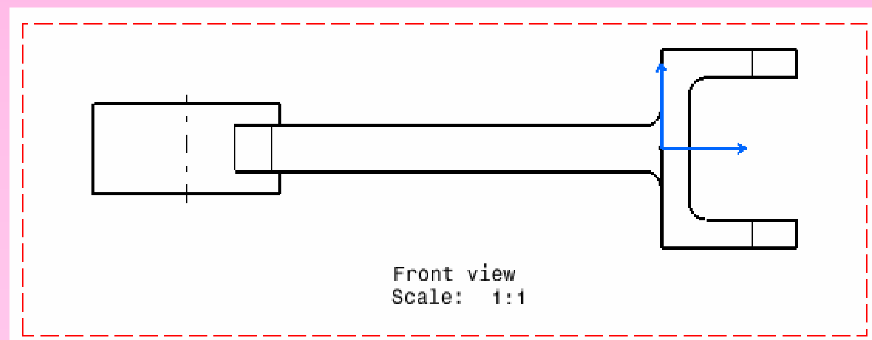


Views generated using the **View Creation Wizard** dialog box

- Generating the Front, Top, and Left Views
- Generating the Front, Bottom, and Right Views
- Generating All the Views
- Generating individual Drawing Views
- Generating the Front View

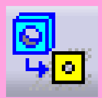


The Projection toolbar

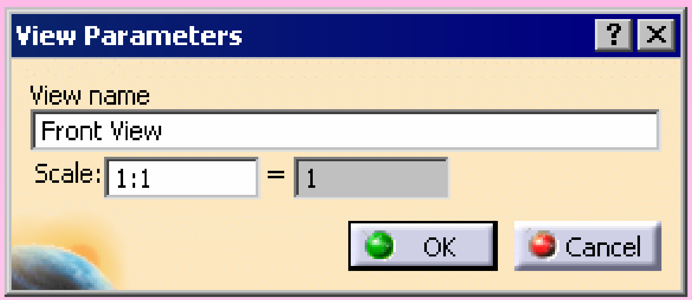


The front view generated using the Front View tool

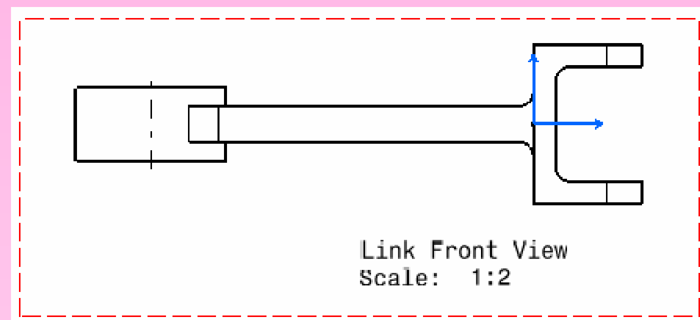
• Generating the Advanced Front View



To generate a front view using this tool, choose the **Advanced Front View** button from the **Projections** toolbar; the **View Parameters** dialog box is displayed, as shown in the figure.

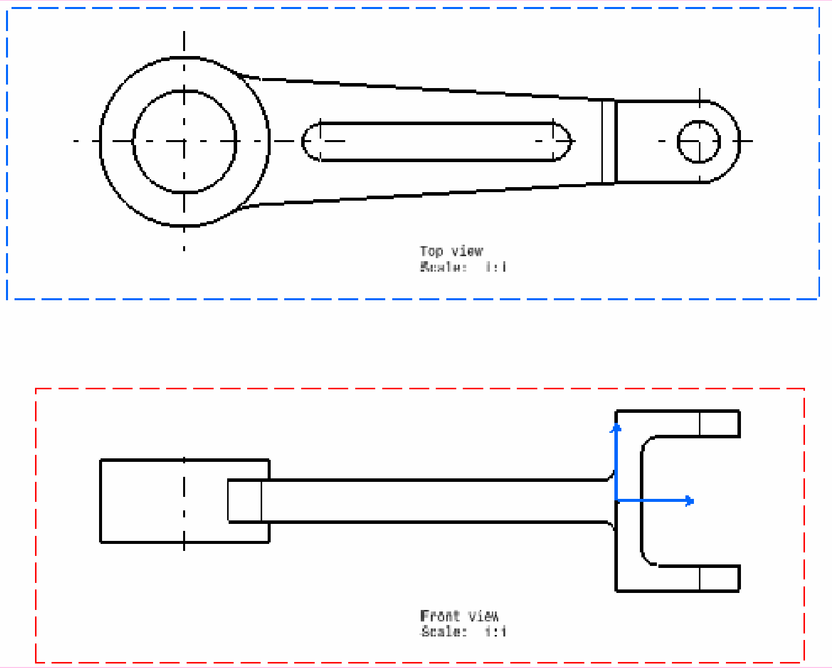


The **View Parameters** dialog box



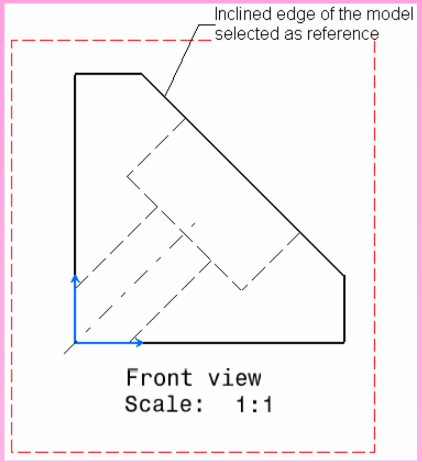
The front view generated using the **Advanced Front View** tool

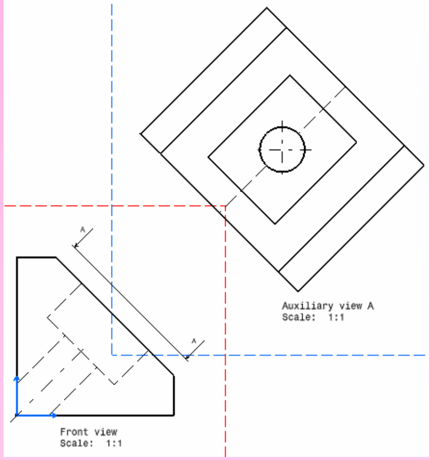
- **Generating Projected Views**



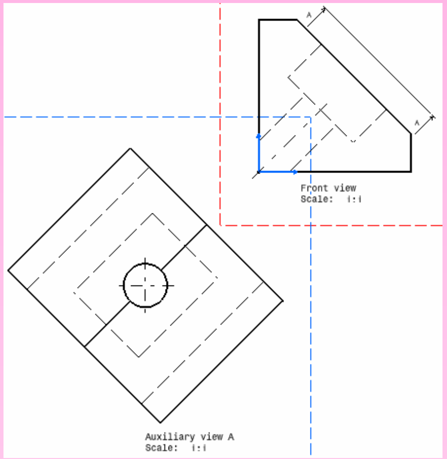
Top view generated from the front view using the **Projection View** tool

- **Generating Auxiliary Views**





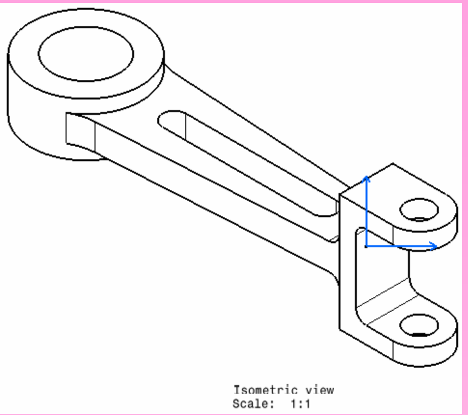
Reference to be selected to generate the auxiliary view



Auxiliary view placed above the parent view

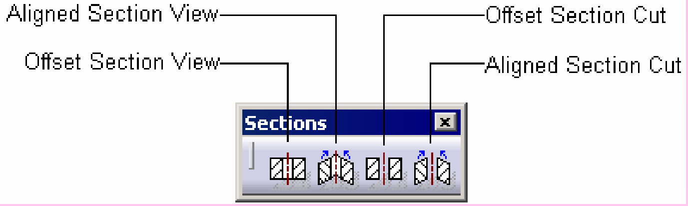
Auxiliary view placed below the parent view

- Generating Isometric Views

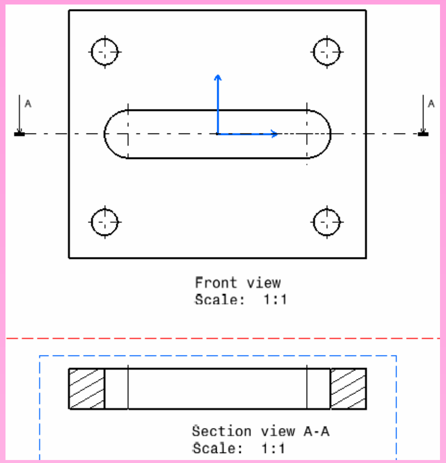


Isometric view generated using the **Isometric View** tool

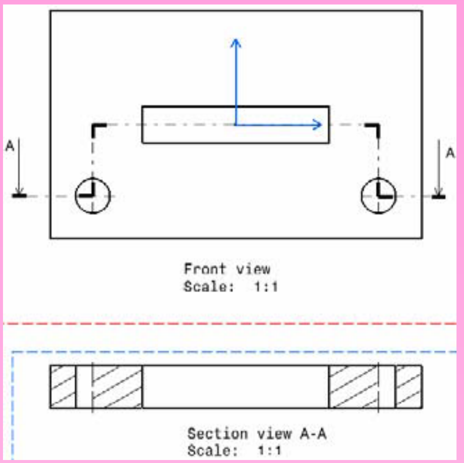
- Generating Section Views



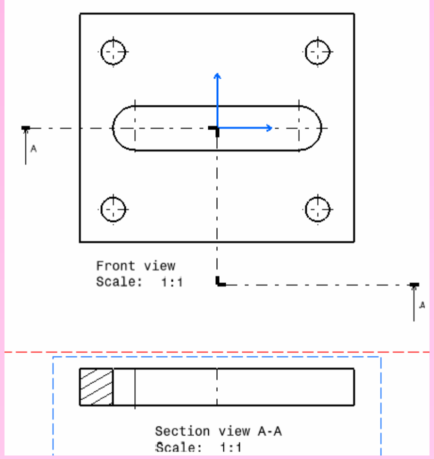
The **Sections** toolbar



Full section view

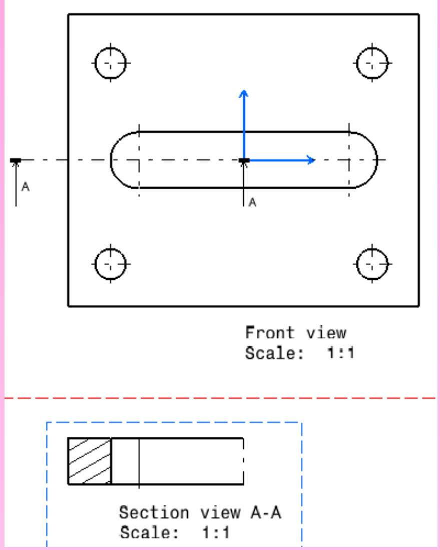


Offset section view



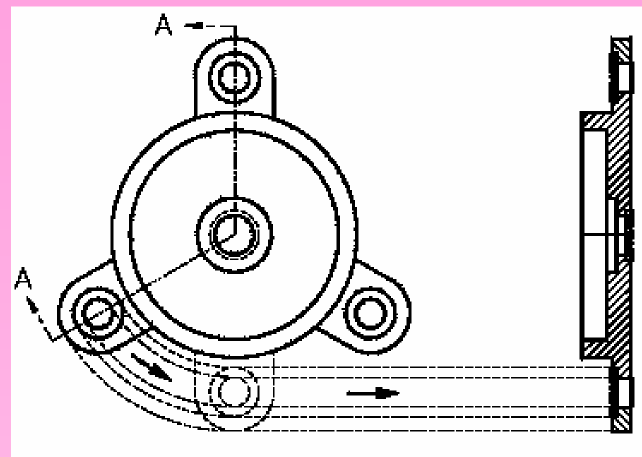
Half section view

- Creating the Partial Section View

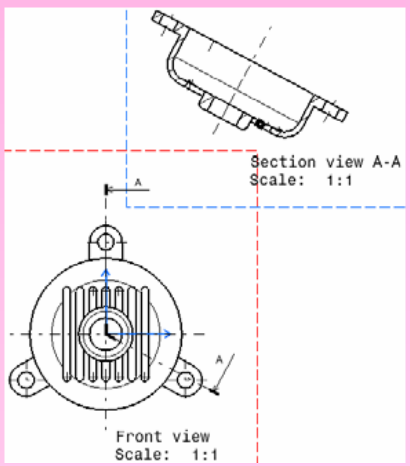


The partial section view

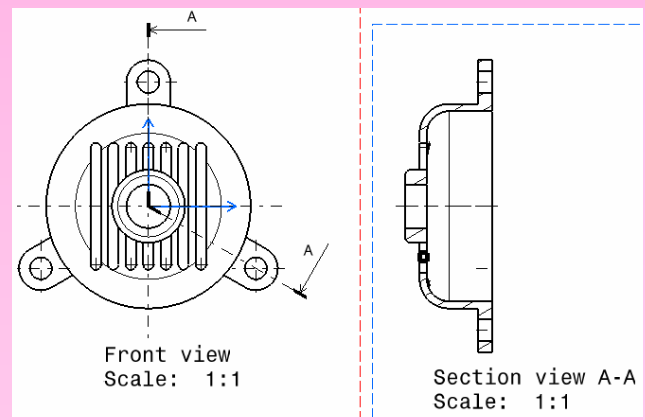
• Generating the Aligned Section Views



Aligned section view

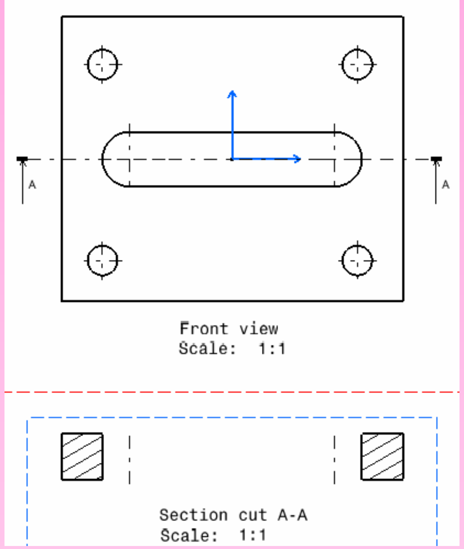


Aligned section view with inclined line drawn first



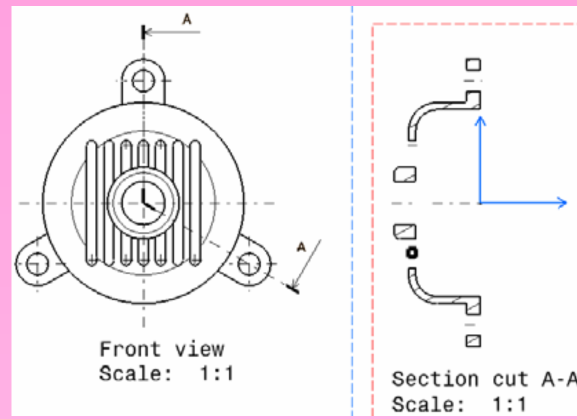
Aligned section view with vertical line drawn first

- Generating the Offset Section Cut



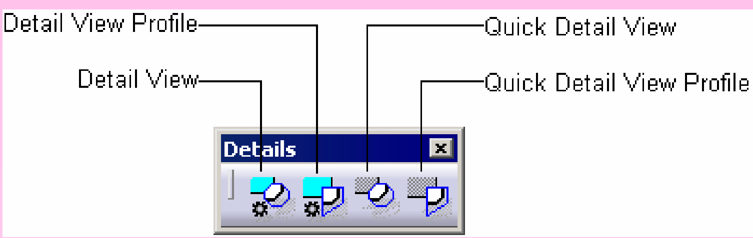
Offset section cut

- **Generating the Aligned Section Cut**

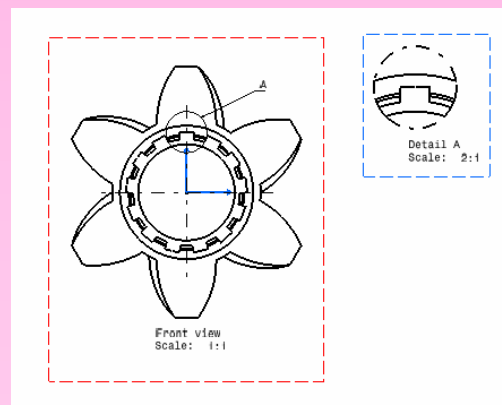


Aligned section cut

- **Generating Detail Views**

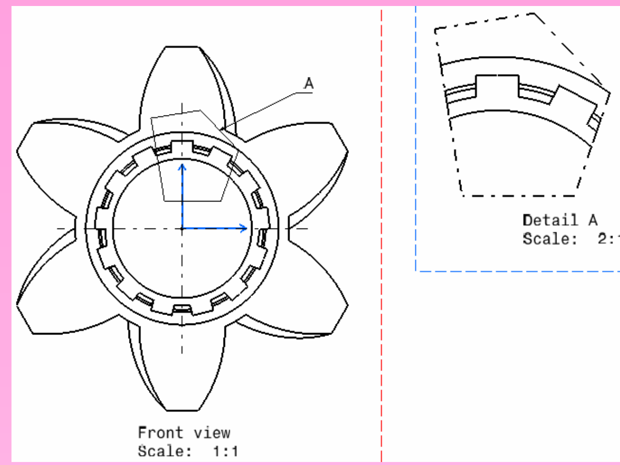


The Details toolbar



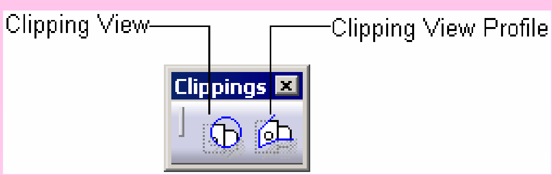
Detail view

- **Generating the Detail View Profiles**

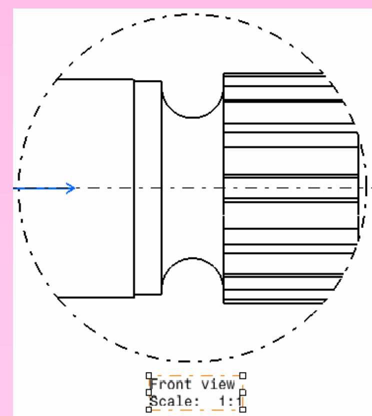


Detail view profile

- **Generating Clipping Views**

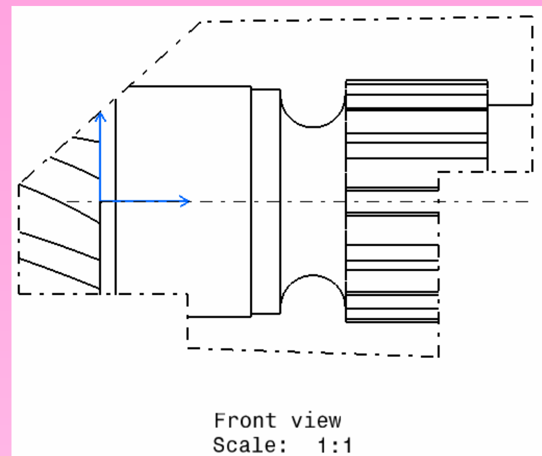


The Clippings toolbar



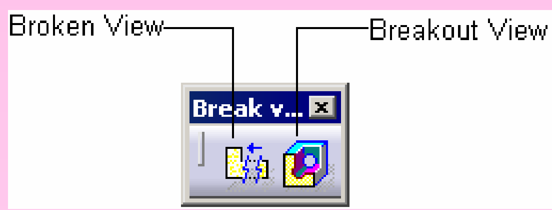
Clipping view

- **Generating the Clipping View Profiles**

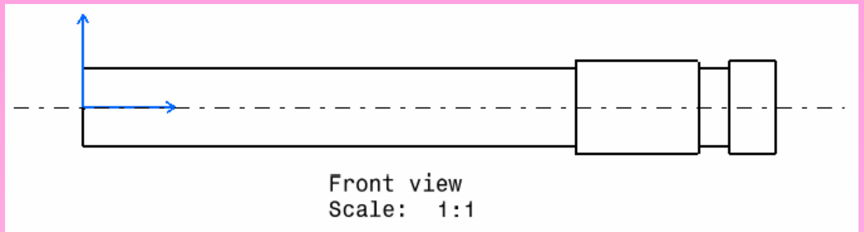


Clipping view profile

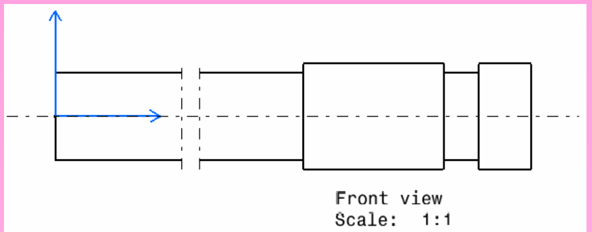
- **Generating the Broken View**



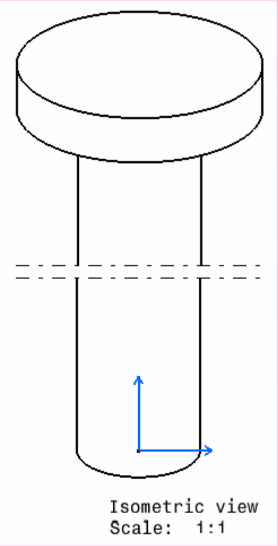
The Break view toolbar



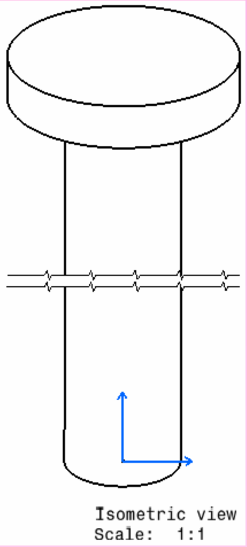
The front view that needs to be broken



Resulting broken view

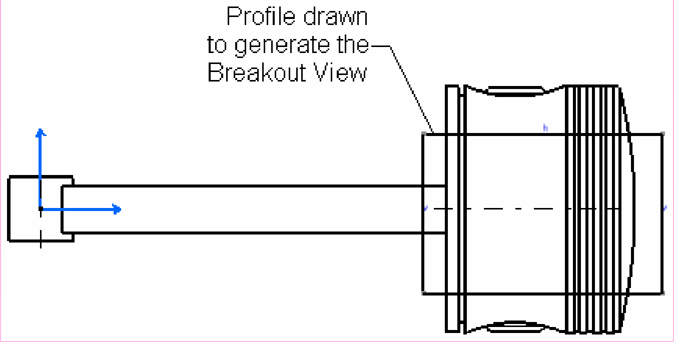


A broken isometric view

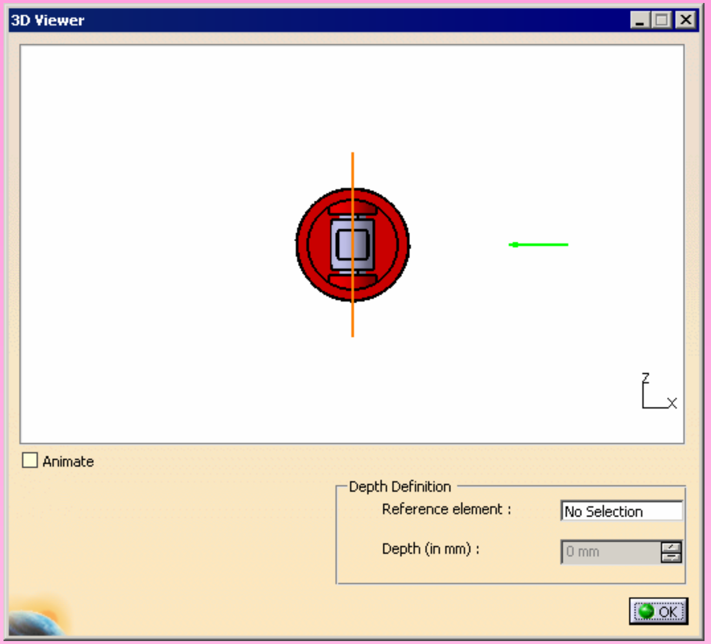


Zigzag broken lines

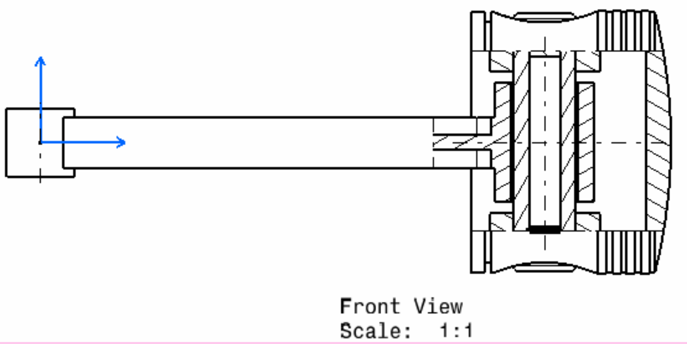
• Generating the Breakout View



Zigzag broken lines



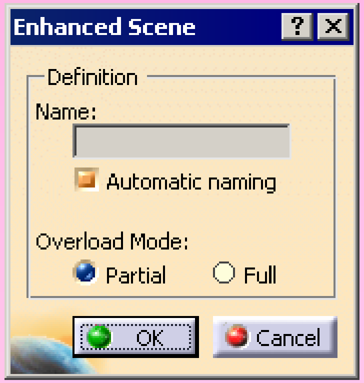
The 3D Viewer window



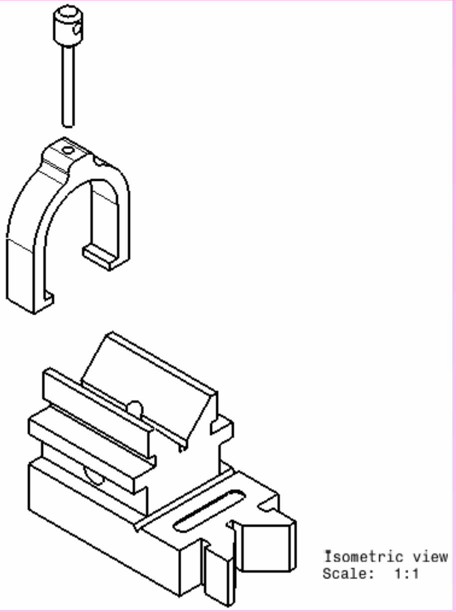
Breakout view

- **Generating the Exploded View**

Choose the **Enhanced Scene** button from the **DMU Review Creation** toolbar to display the **Enhanced Scene** dialog box, as shown in the figure.



The Enhanced Scene dialog box



Drawing view of the exploded assembly

WORKING WITH INTERACTIVE DRAFTING IN CATIA V5

- You can also sketch the 2D drawing views in the **Drafting** workbench of CATIA V5.
- In technical terms, sketching 2D drawings is known as interactive drafting.

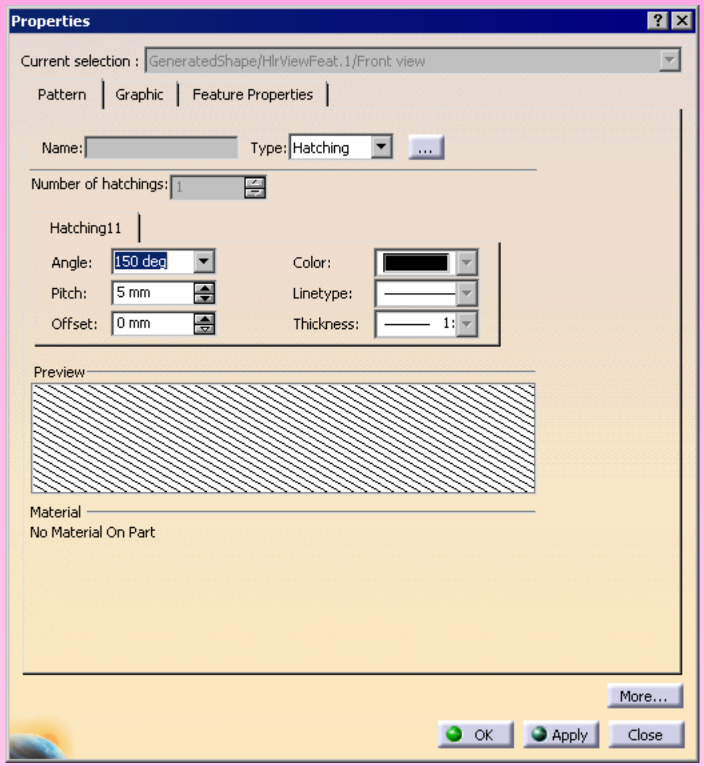
EDITING AND MODIFYING THE DRAWING VIEWS

- In CATIA V5, you can perform editing operations and modifications on the drawing views.
- The various options under the **Editing and Modifying Drawing Views** option are:

- **Changing the Scale of Drawing Views**
- **Modifying the Project Plane of the Parent View**
- **Deleting Drawing Views**
- **Rotating Drawing Views**
- **Hiding Drawing Views**

MODIFYING THE HATCH PATTERN OF SECTION VIEWS

When you generate a section view of an assembly or a component, a hatch pattern based on the material assigned to the components in the part document is applied to the component or components.



The Properties dialog box to modify the hatch pattern

Tutorial 1

In this tutorial, you will generate the drawing views of the model of Motor Cover created in Tutorial 2 of Chapter 7. You will generate the front view, top view, aligned section views from the top view, detail view, and the isometric view. You will use the A4 standard sheet size and the third angle projection. The drawing sheet after generating all the views is shown in **Figure A**. **(Expected time: 30 min)**

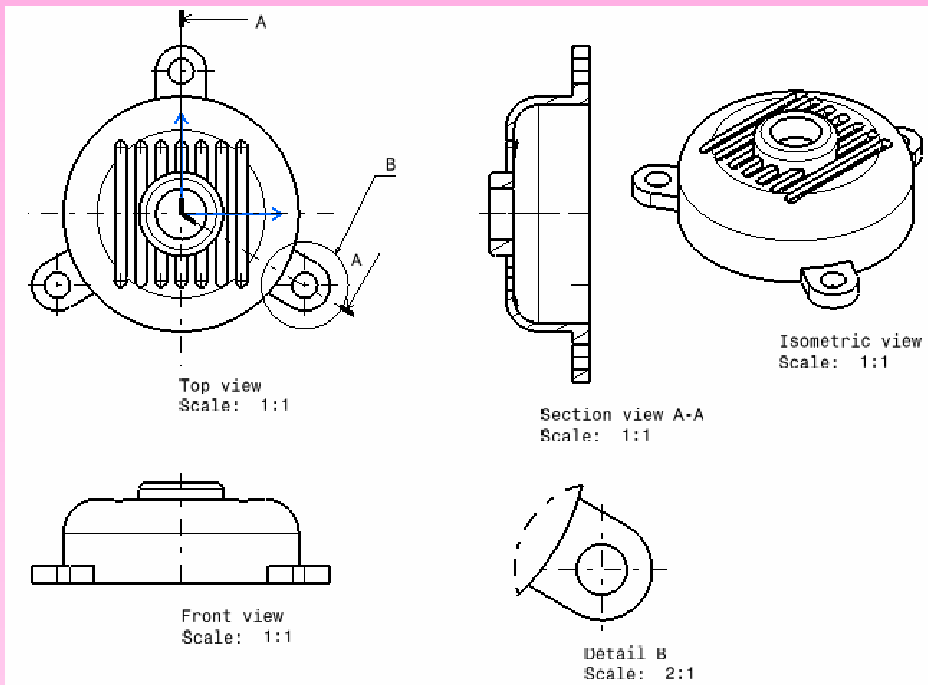


Figure A Drawing views to be generated

1. Copy the part document of Tutorial 2 from Chapter 7 to the folder of the current chapter.
2. Start a new file in the **Drafting** workbench with the standard A4 sheet size.
3. Set the projection standard to third angle.
4. Generate the front view, which will be used as the main view, as shown in **Figure B**.

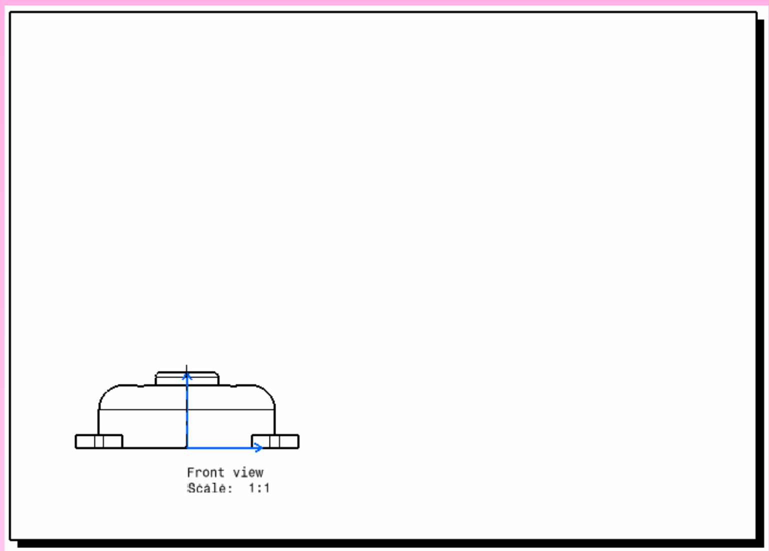


Figure B The drawing sheet after generating the front view

5. Generate the top view using the **Projection View** tool, as shown in **Figure C**.

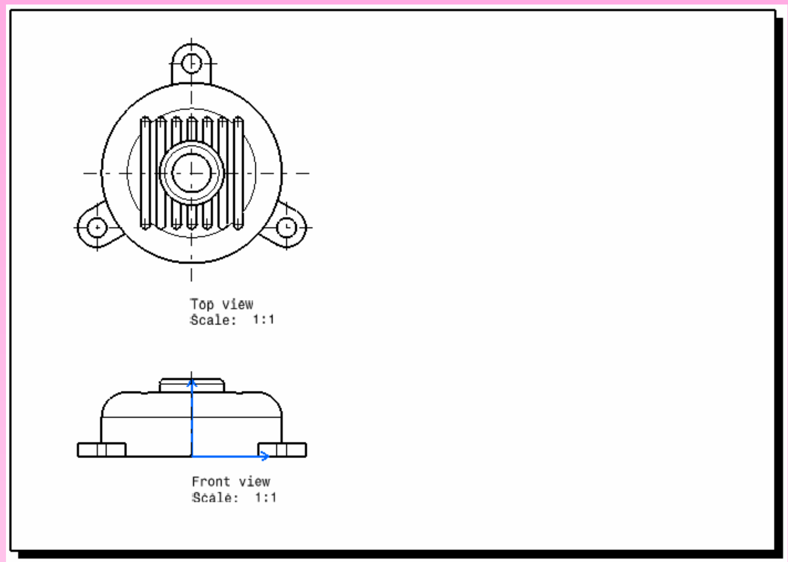


Figure C The drawing sheet after generating the top view

6. Generate the aligned section view from the top view after activating it, as shown in **Figure D** and **Figure E**.

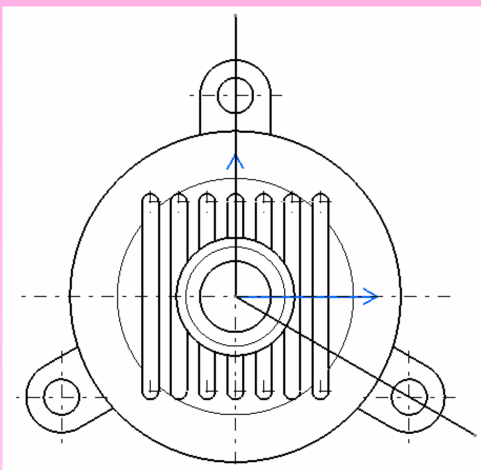


Figure D Sketch for the section view

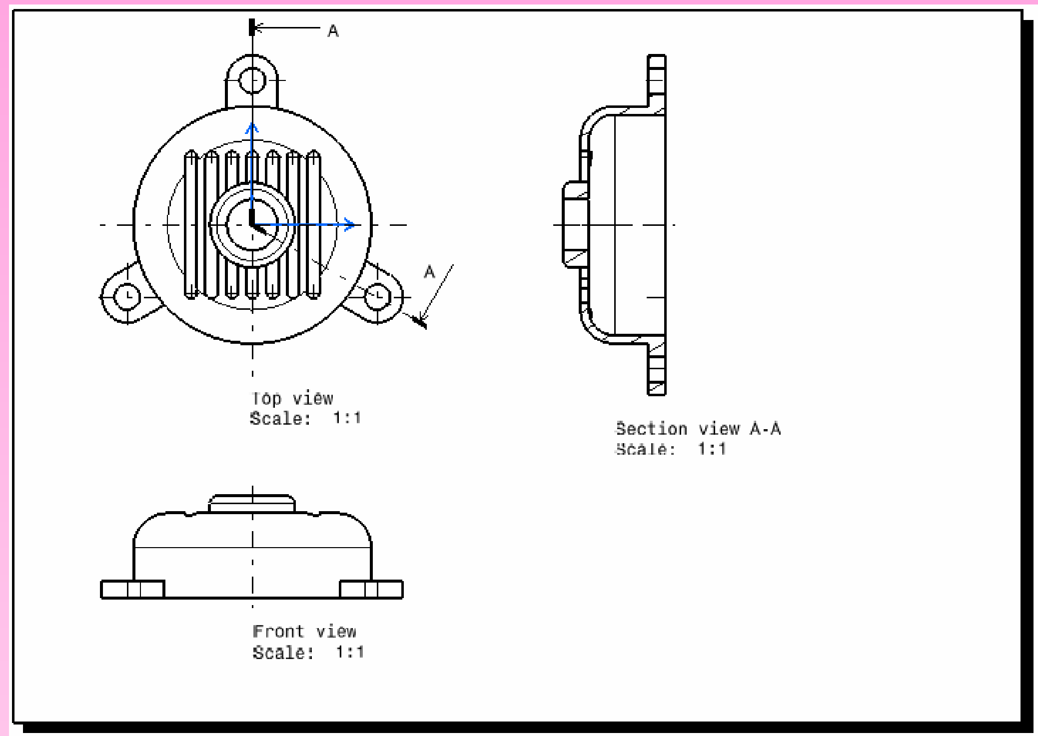


Figure E The drawing sheet after generating the aligned section view

7. Generate the detail view, as shown in **Figure F** and **Figure G**.

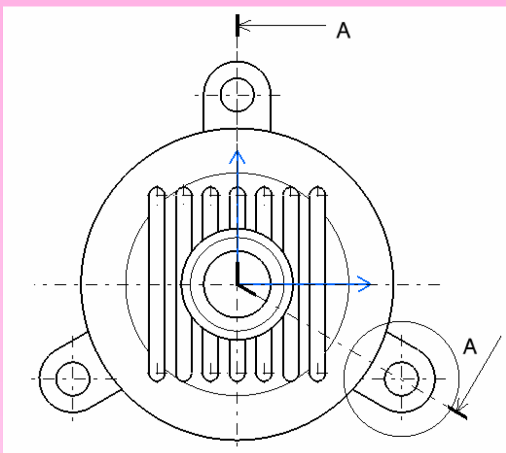


Figure F Sketch for the detail view

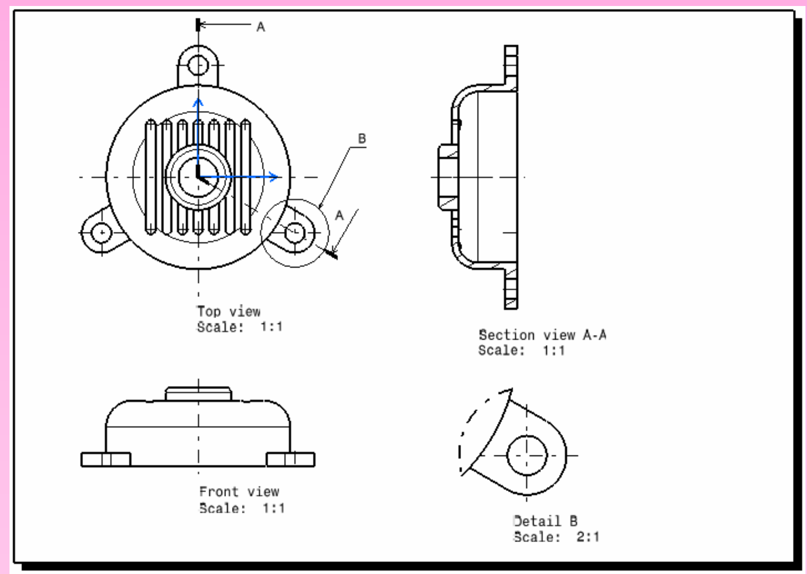


Figure G The drawing sheet after generating the detail view

8. Generate the isometric view, as shown in **Figure H**.

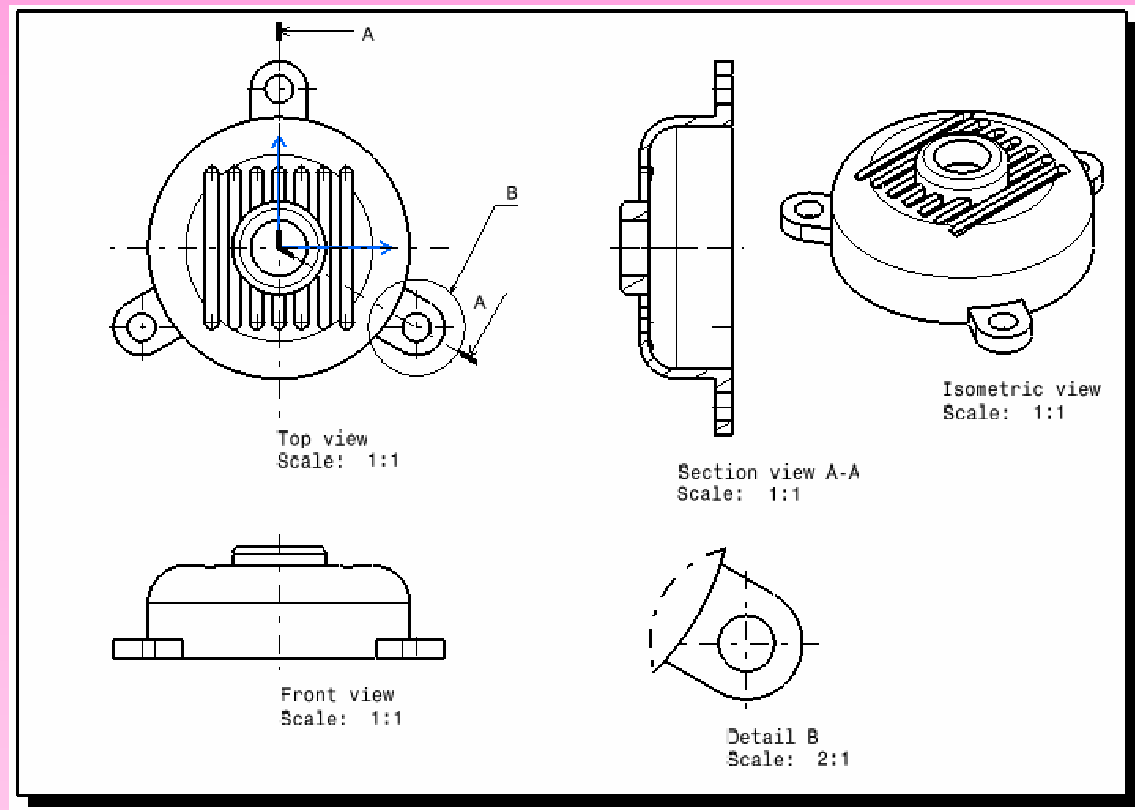


Figure H The final drawing sheet with all views

9. Save the file in **\My Documents\CATIA\c12** folder and then close it.

❑ Tutorial 2

In this tutorial, you will create the Bench Vice assembly and then generate the front, top, right, and isometric views of the same, as shown in **Figure A**. **Figure B**, **Figure C**, **Figure D**, and **Figure E** shows the views and dimensions of the components of the Bench Vice assembly. **(Expected time: 2 hrs)**

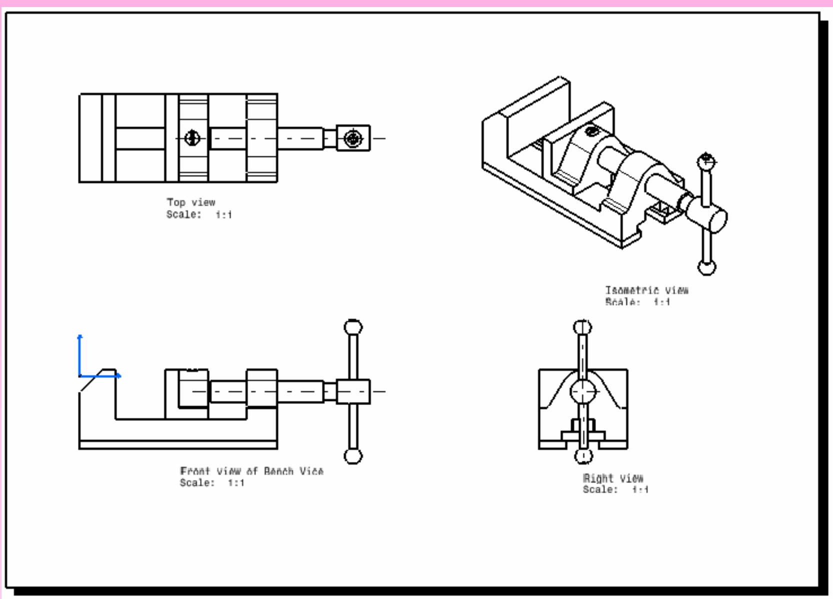


Figure A Drawing view of the Bench Vice assembly

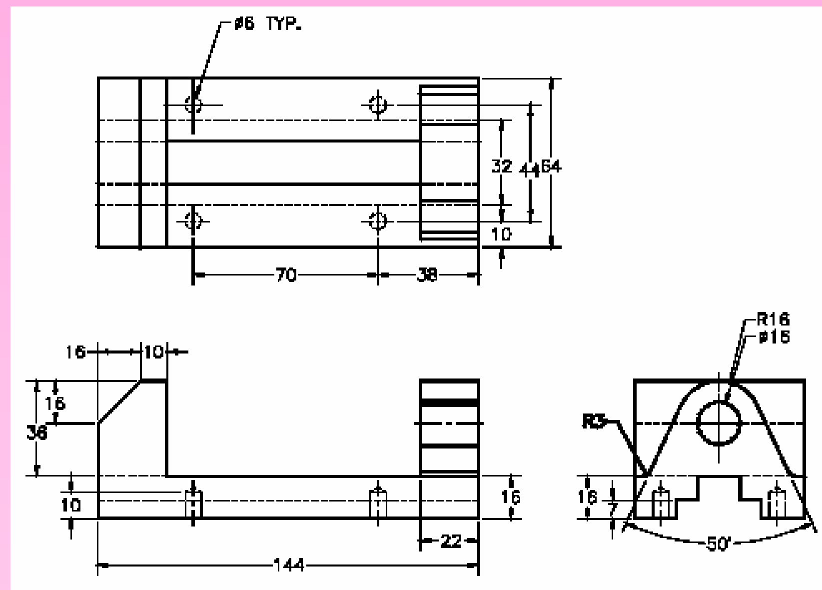


Figure B Views and dimensions of the Vice Jaw

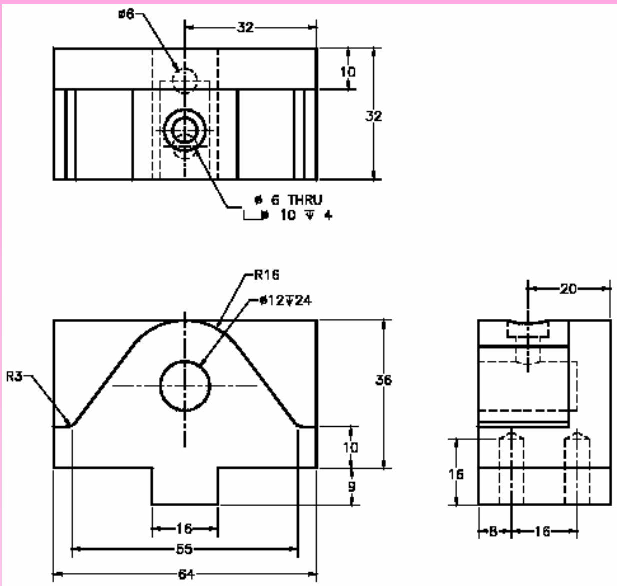


Figure C Views and dimensions of the Vice Jaw

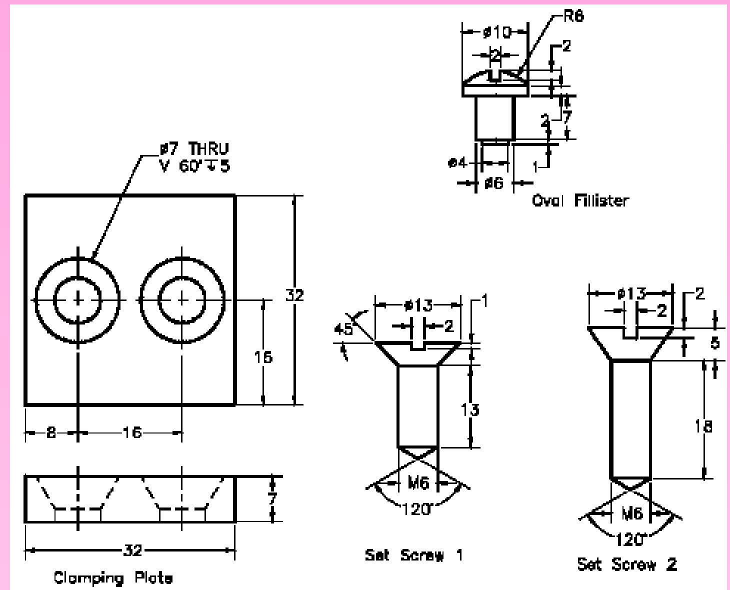


Figure D Views and dimensions of various components of the Bench Vice

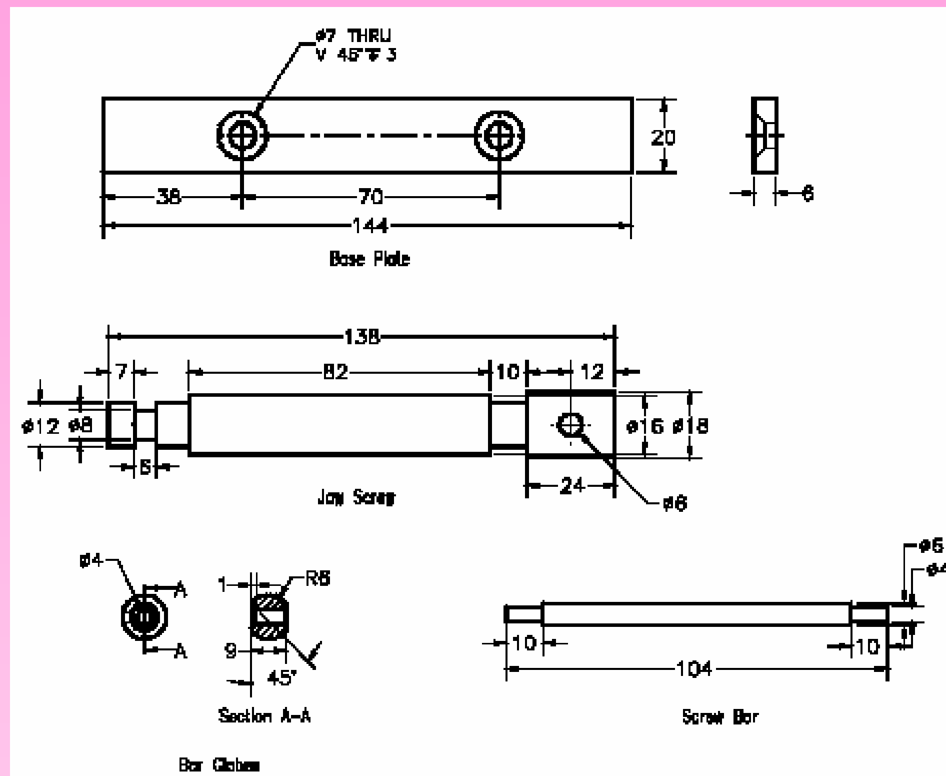


Figure E Views and dimensions of various components of the Bench Vice

1. Create all components of the Bench Vice assembly and assemble them.
2. Start a new file in the **Drafting** workbench using the A2 size sheet.
3. Set the standard of the projection to third angle.
4. Generate the front view, as shown in **Figure F**.

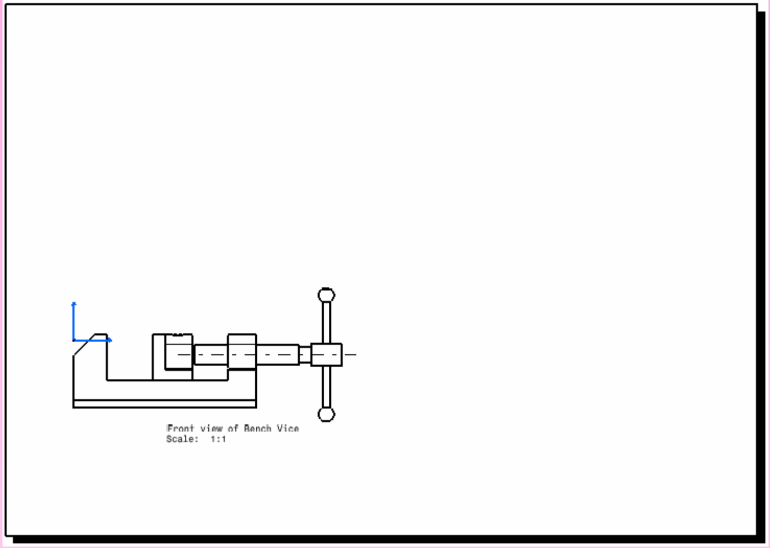


Figure F The Drawing sheet after generating the front view

5. Generate the top view, as shown in **Figure G**.

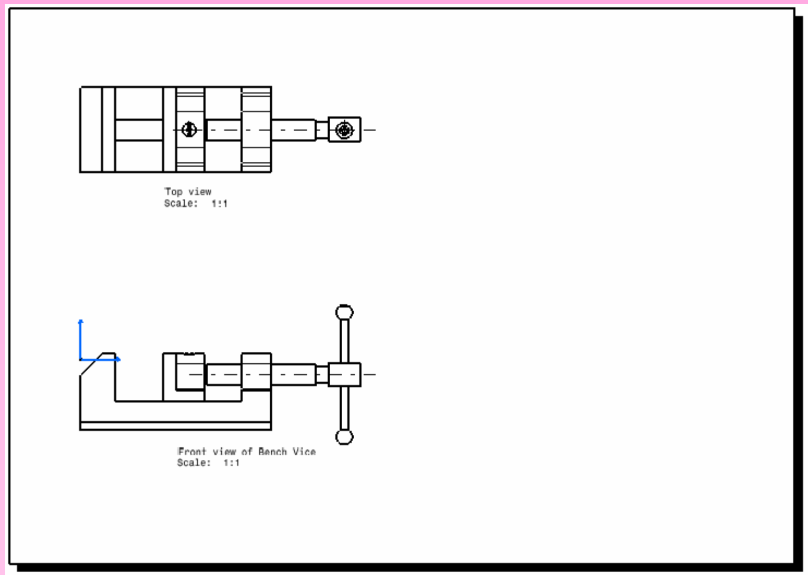


Figure G The drawing sheet after generating the top view

6. Generate the right view, as shown in **Figure H**.

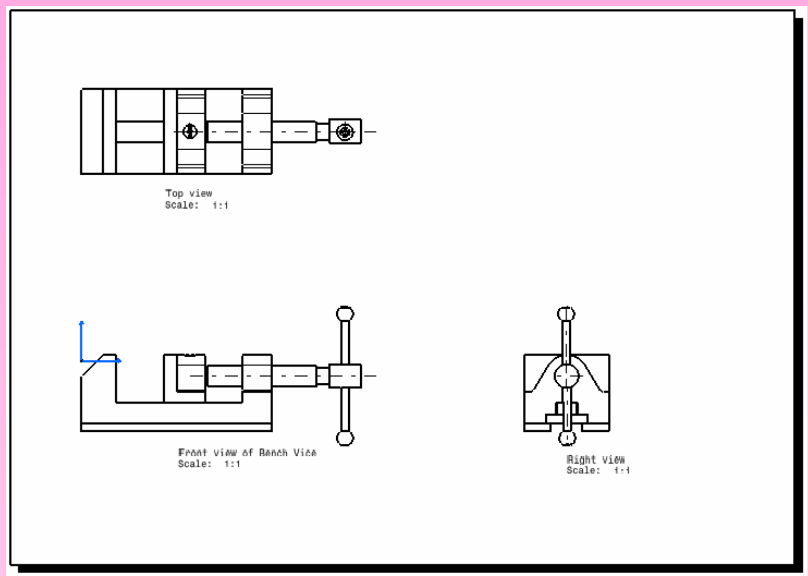


Figure H **The drawing sheet after generating the right view**

7. Generate the isometric view, as shown in **Figure I**.

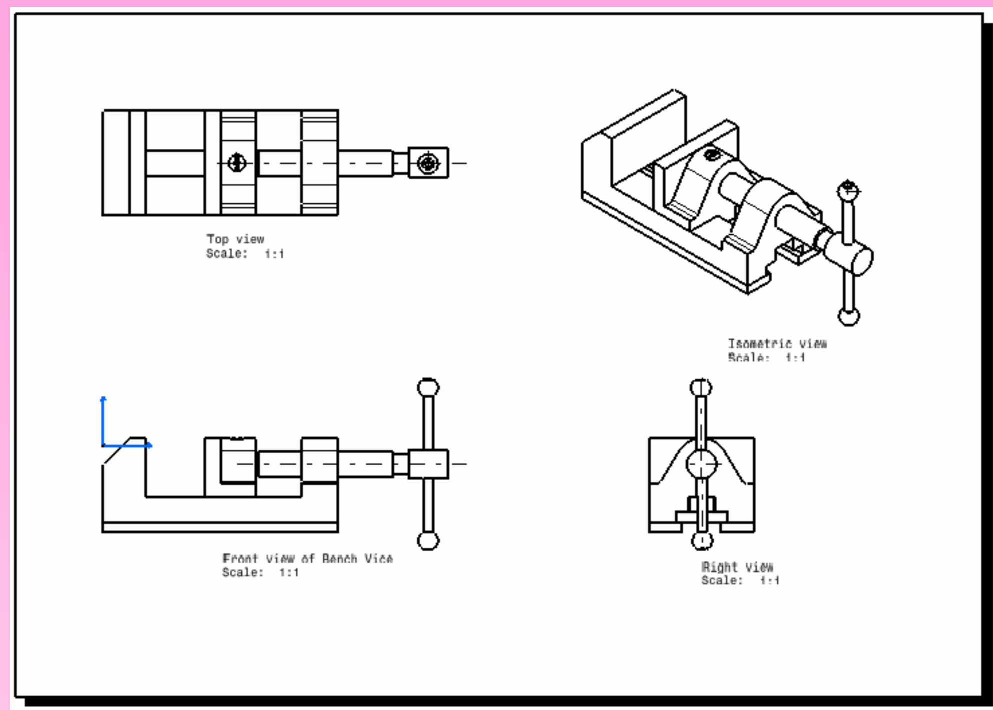


Figure I Final drawing sheet after generating all views

8. Save the file in **\My Documents\CATIA\c12** folder and then close it.

Exercise 1

Create the components of the V-Block and generate the front, top, right, detail, and isometric views, as shown in **Figure A**. The views and dimensions of the components of the V-Block are shown in **Figure B** and **Figure C**. **(Expected time: 1 hr)**

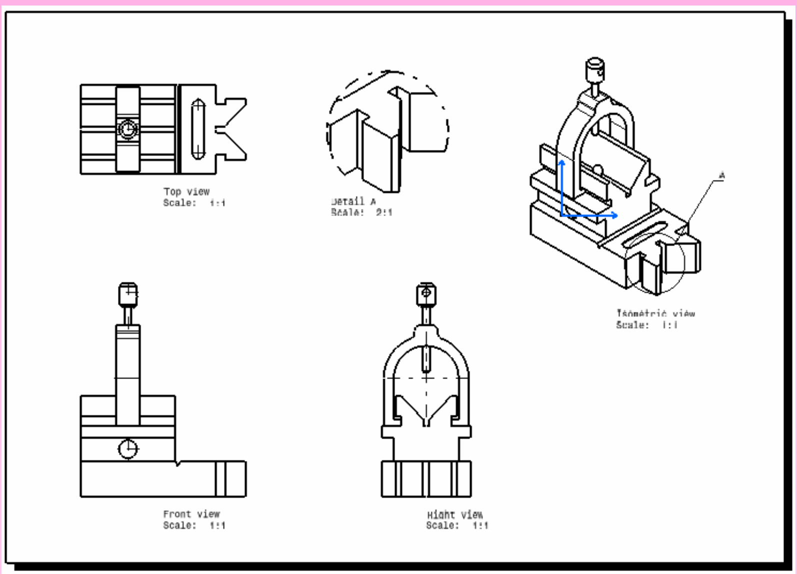


Figure A Drawing views of the V-Block assembly

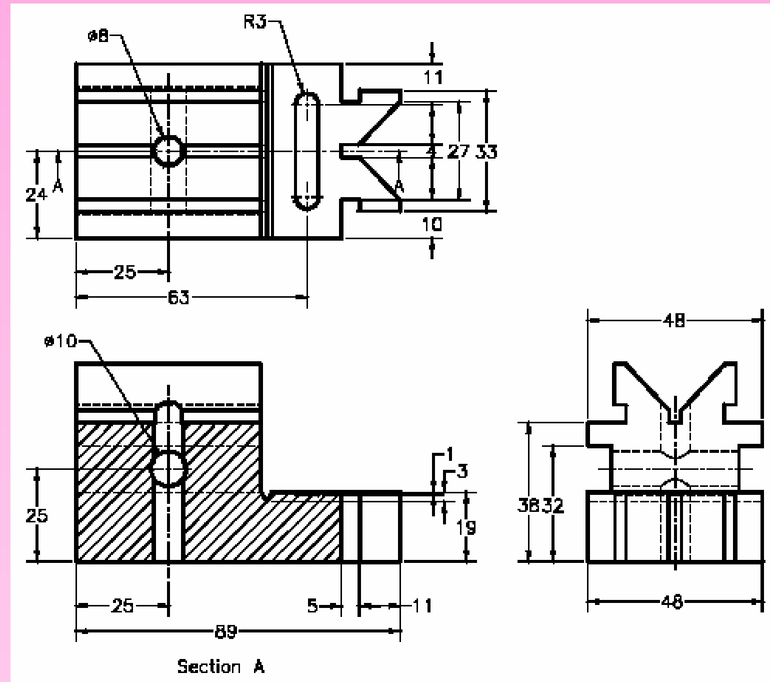


Figure B Views and dimensions of the V-Block body

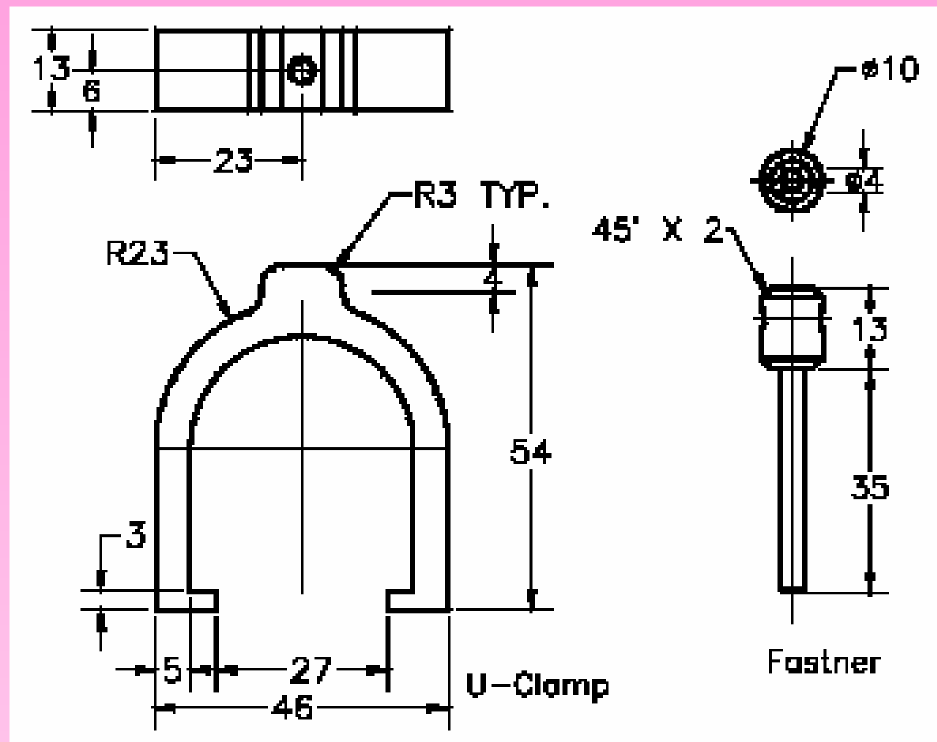


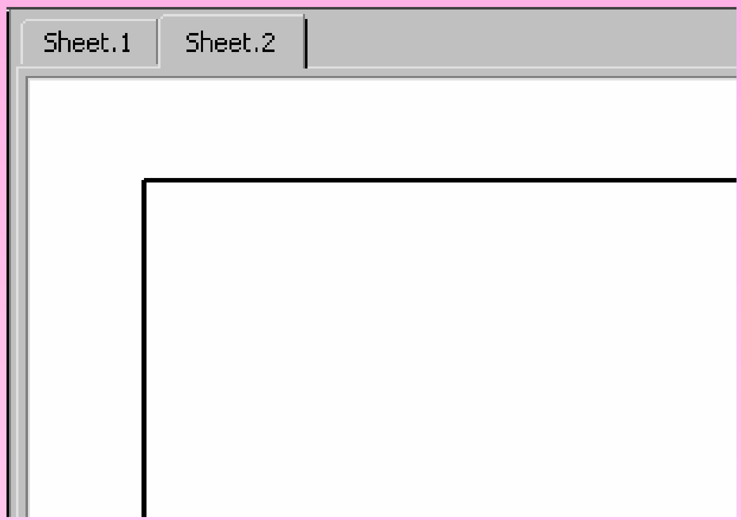
Figure C Views and dimensions of the U-Clamp and Fastener

Learning Objectives:

- *Insert additional sheets in the current drawing file.*
- *Insert frames and title blocks.*
- *Add annotations to the drawing views.*
- *Edit the annotations.*
- *Generate the Bill of Material (BOM).*
- *Generate balloons.*

INSERTING SHEETS IN THE CURRENT FILE

- You can insert additional sheets to the current drafting file.
- This is good practice when you need to generate the drawing views of all the components of an assembly and also its other views such as the isometric view or a view with Bill of Material (BOM), and balloons in a single drawing file.

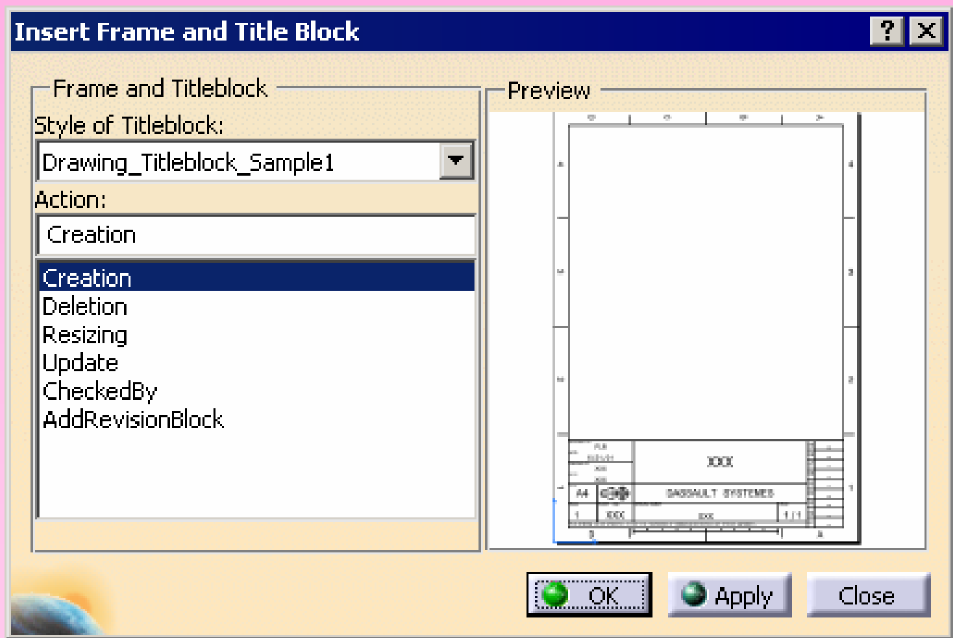


Partial view of the drawing with a new sheet added to the current file

INSERTING THE FRAME AND TITLE BLOCK

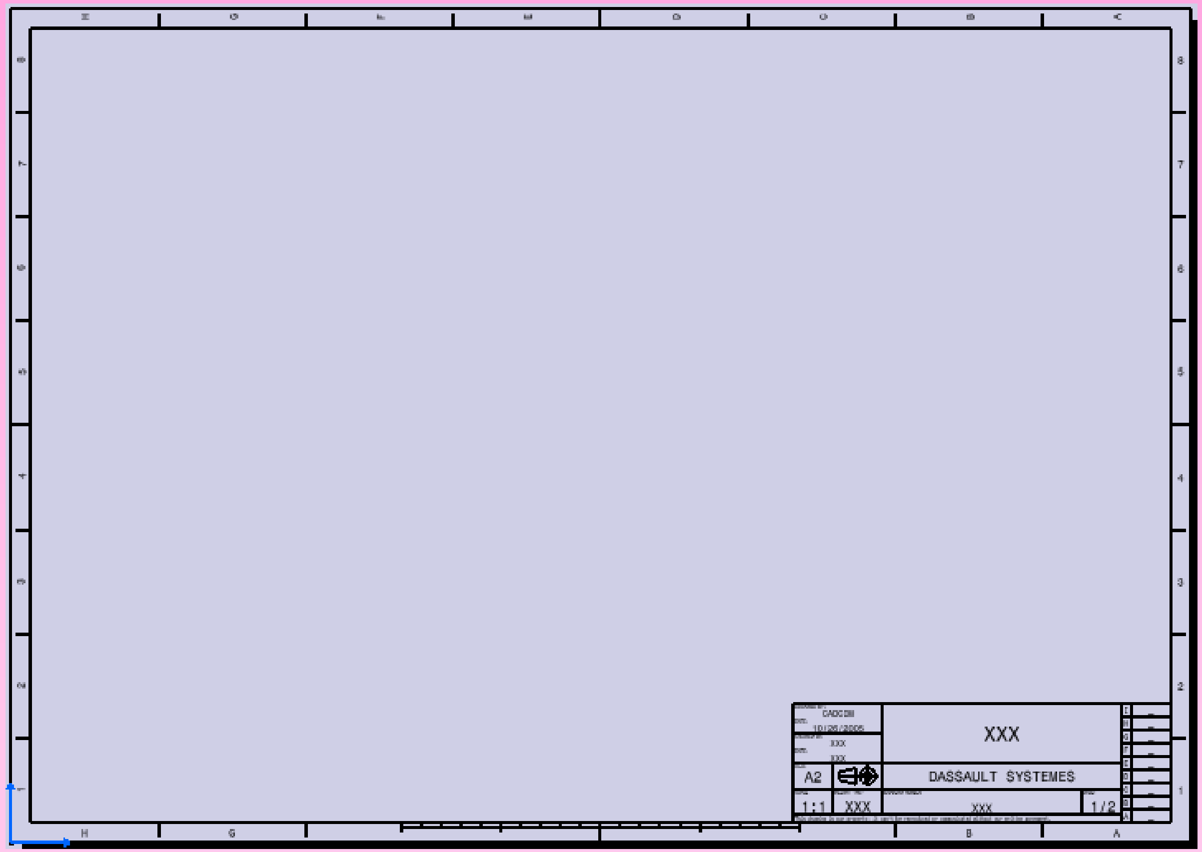
Automatic Insertion of the Frame and Title Block

- Choose the **Frame Creation** button from the **Drawing** toolbar.
- The **Insert Frame and Title Block** dialog box is displayed, as shown in the figure.

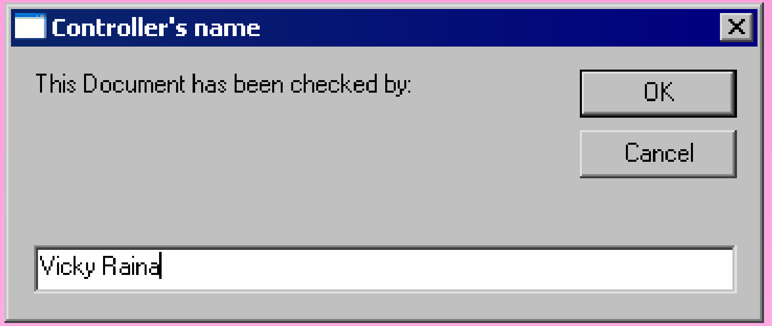


The Insert Frame and Title Block dialog box

- Style of Titleblock
- Action
  - Creation
  - Deletion
  - Resizing
  - Update
  - Checked By
  - AddRevisionBlock

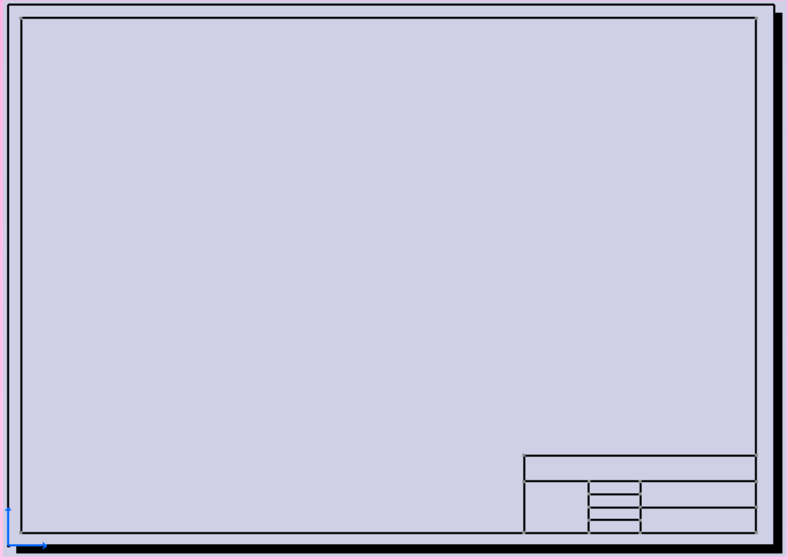


The drawing sheet after adding the frame and title block to the drawing sheet



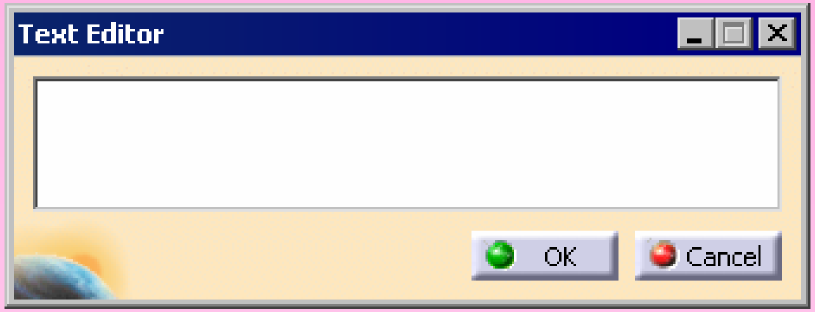
The Controller's name dialog box

- **Creating the Frame and Title Block Manually**



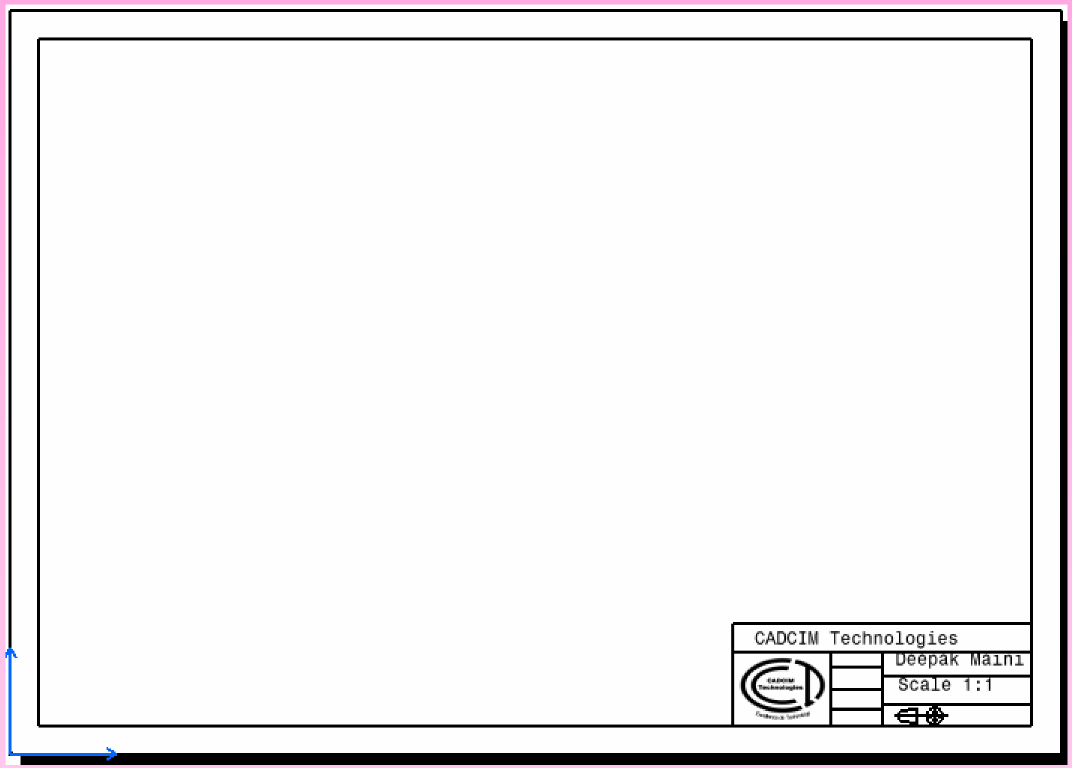
The drawing sheet after drawing the frame and title block

- **Adding Text in the Title Block**
  - Select a point on the drawing sheet to place the text.
  - The **Text Editor** dialog box is displayed, as shown in the figure.



The Text Editor dialog box

- **Inserting the Logo**



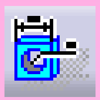
The drawing sheet after completing the frame and title block

ADDING ANNOTATIONS TO THE DRAWING VIEWS

After generating the drawing views, you need to generate the dimensions in the drawing views and add the other annotations, such as notes, surface finish symbols, geometric tolerance, and so on.

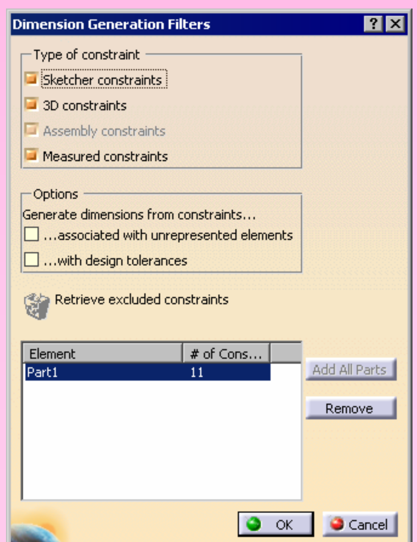
• Generating the Dimensions

• Generating all Dimensions Together



- Exit the **Options** dialog box and then choose the **Generate Dimensions** button from the **Dimension Generation** toolbar, after invoking it from the **Generation** toolbar.

- The **Dimension Generation Filters** dialog box is displayed, as shown in the figure.



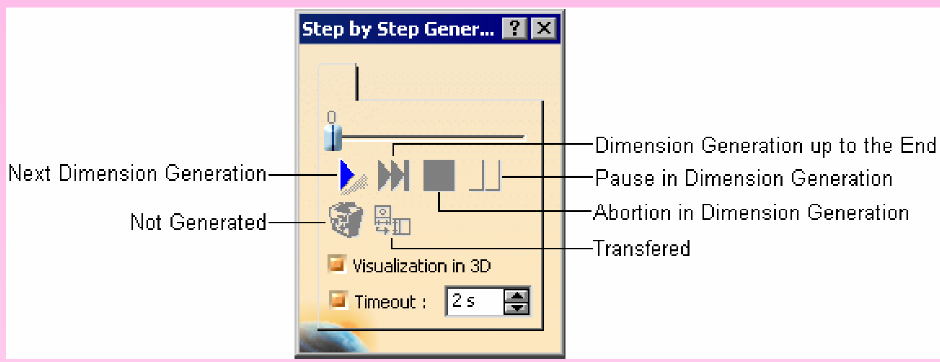
The Dimension Generation Filter dialog box

The options available in the **Dimension Generation Filter** dialog box are:

- **Type of constraint Area**
- **Options Area**
- **Retrieve excluded constraints**
- **Add All Parts**

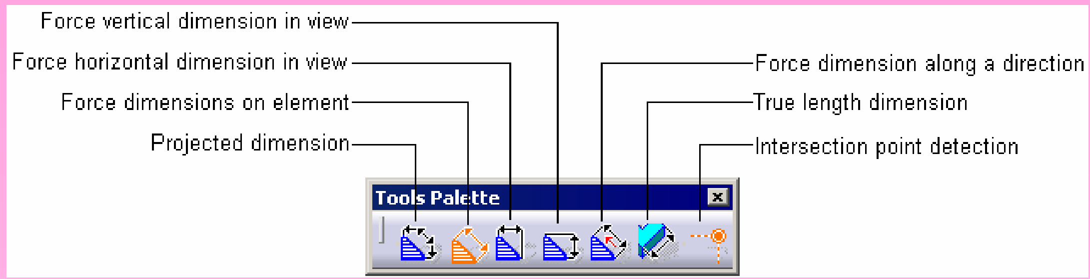
- **Generating Dimensions Step by Step**

- If the **Dimension Generation Filter** dialog box is displayed, set the option in this dialog box and choose the **OK** button to exit it.
- The **Step By Step Generation** dialog box is displayed, as shown in the figure.



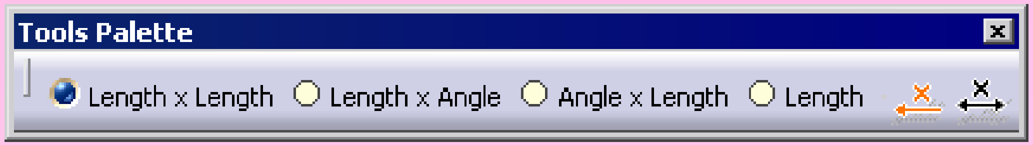
The Step by Step Generation dialog box

- Creating Reference Dimensions

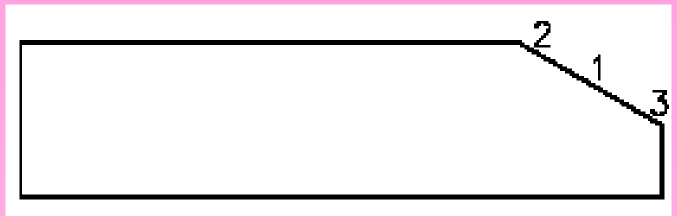


The Tools Palette toolbar

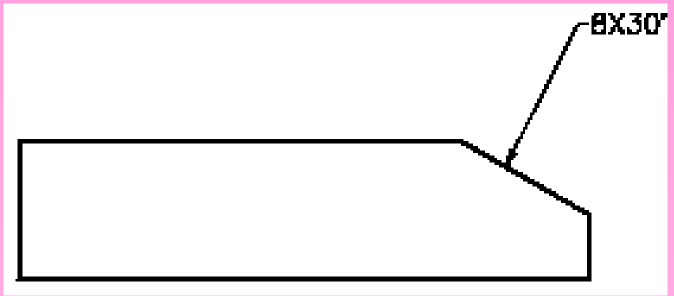
- Creating the Chamfer Dimensions



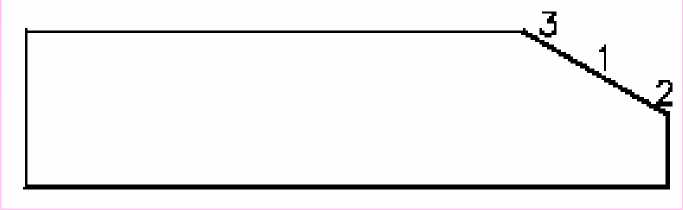
The Tools Palette toolbar



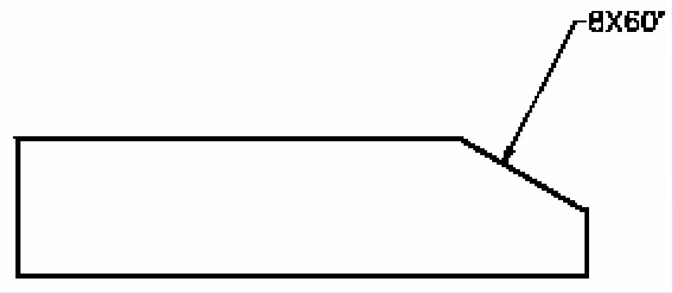
Selection sequence



Resulting chamfer dimension



Selection sequence



Resulting chamfer dimension

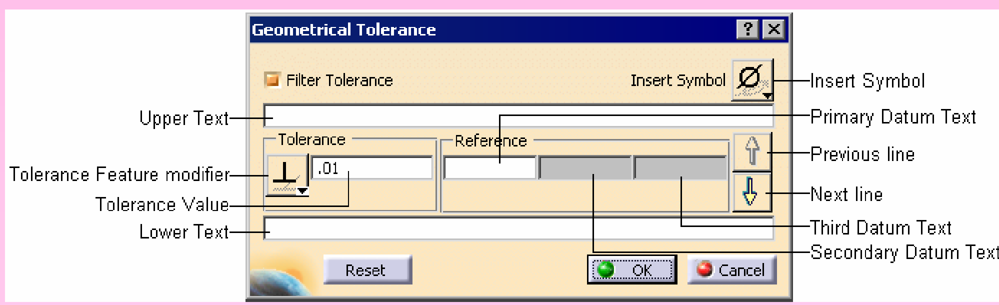
• Adding Datum Features

Move the cursor to the desired location and click a point on the drawing sheet to place the datum feature; the **Datum Feature Creation** dialog box is displayed, as shown in the figure.

The Datum Feature Creation dialog box

• Adding Geometric Tolerance to the Drawing Views

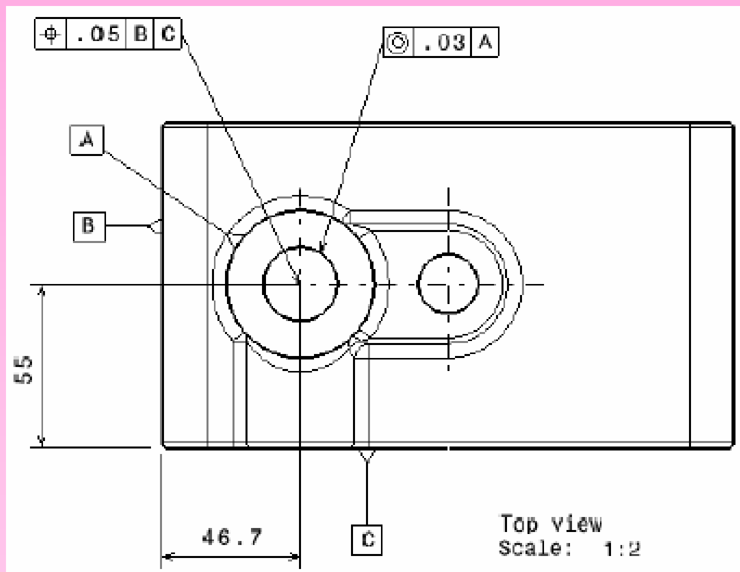
Move the cursor to an appropriate location and click on the drawing sheet to place the tolerance on that location; the **Geometrical Tolerance** dialog box is displayed, as shown in the figure.



The Geometrical Tolerance dialog box

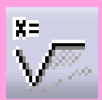
The options available in the **Geometrical Tolerance** dialog box are:

- **Tolerance**
- **Reference**



The drawing view after adding datum feature and tolerance

• Adding Surface Finish Symbols



- Choose the **Roughness Symbol** button from the **Symbols** toolbar, after invoking it from the **Annotations** toolbar.
- The **Symbols** toolbar is shown in **Figure A**.
- Click on the surface from where you need to place the roughness symbol.
- Its preview is displayed attached to the selected point and the **Roughness Symbol** dialog box is displayed, as shown in **Figure B**.

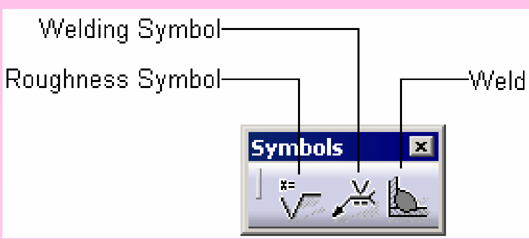


Figure A The Symbols toolbar

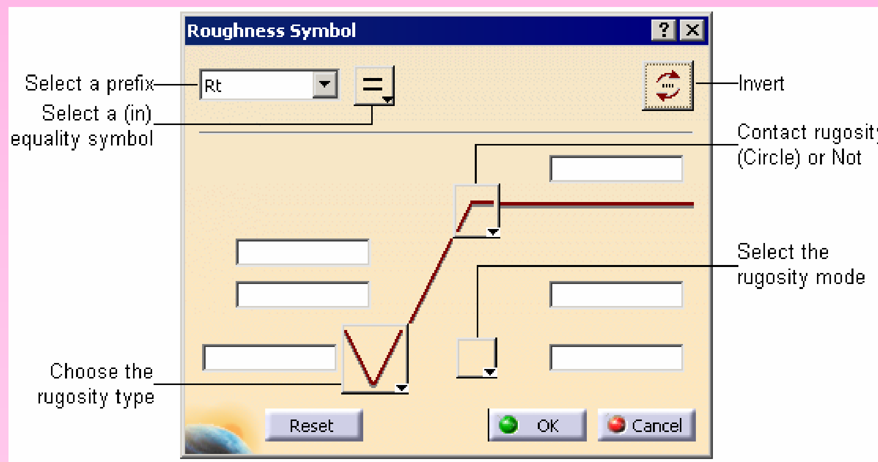
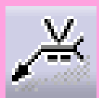
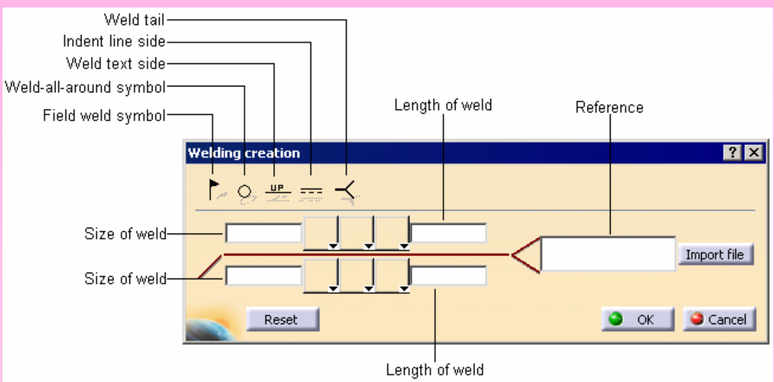


Figure B The Roughness Symbol dialog box

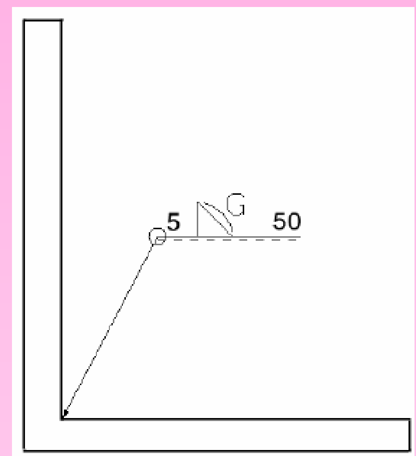
• Adding Welding Symbols



Move the cursor and click on an appropriate location on the drawing sheet to place it; the **Welding creation** dialog box is displayed, as shown in the figure.

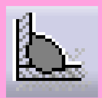


The **Welding creation** dialog box



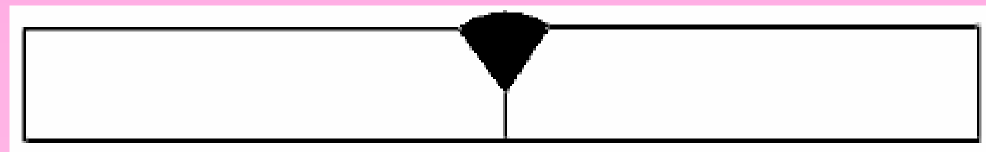
The drawing view after adding the welding symbol

- **Applying Weld**



To apply a weld, choose the **Weld** button from the **Symbols** toolbar.

The **Welding Editor** dialog box



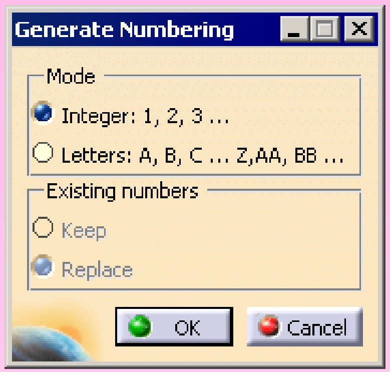
Drawing view after adding the welding symbol

➢ EDITING ANNOTATIONS

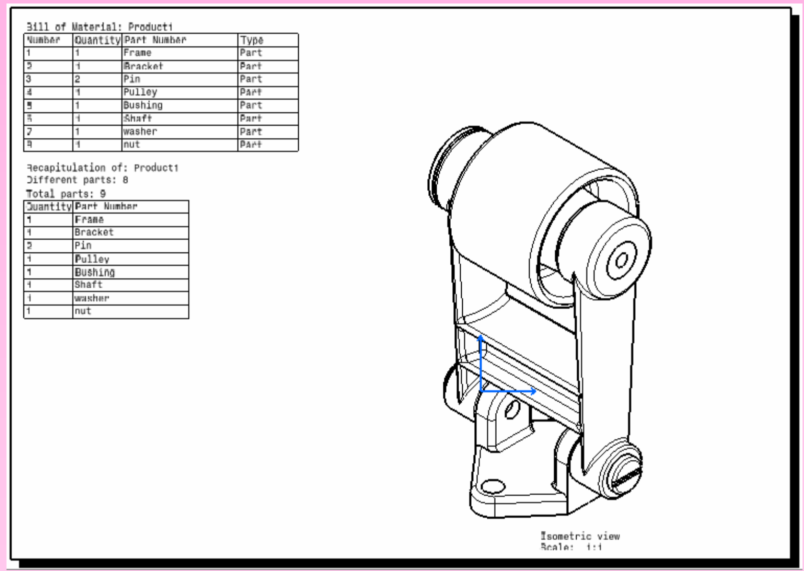
You can edit the annotations added to the drawing views by double-clicking on them to display their respective dialog boxes.

➤ GENERATING THE BILL OF MATERIAL (BOM)

- To number the components, switch to the assembly file from which the drawing views are generated.
- Choose the **Generate Numbering** button from the **Product Structure Tools** toolbar.
- Now, select **Product1** from the **Specification Tree**; the **Generate Numbering** dialog box is displayed, as shown in the figure.



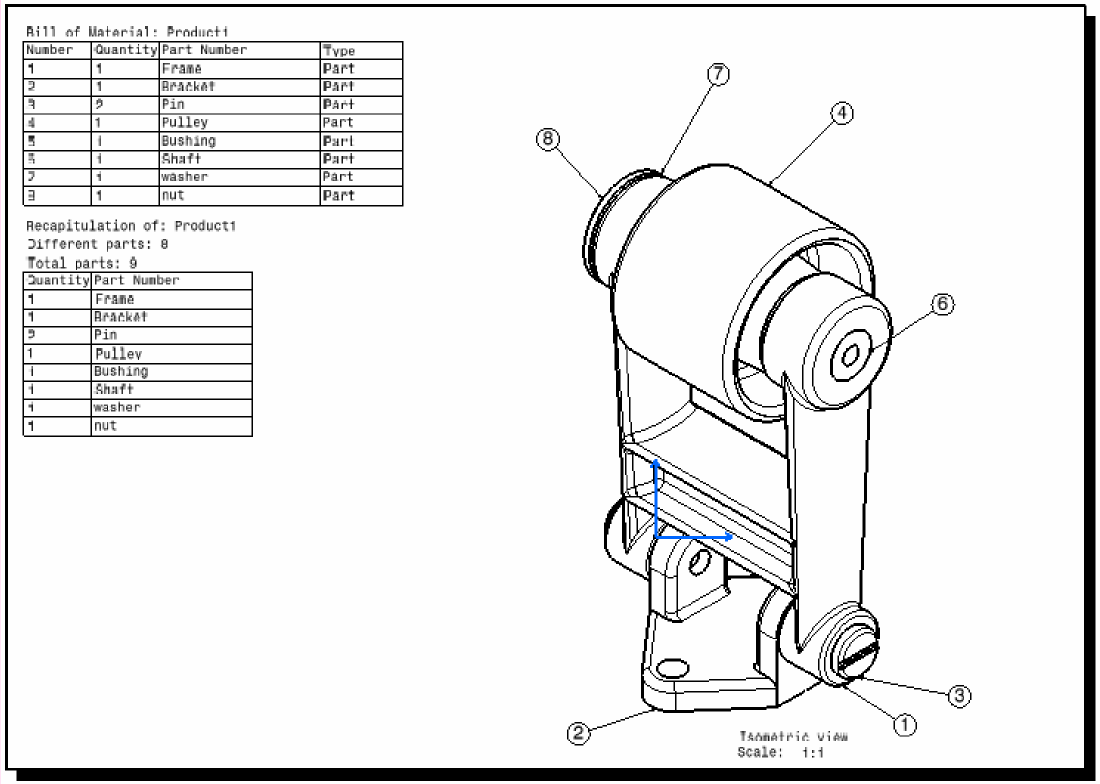
The Generate Numbering dialog box



The drawing sheet after generating the BOM

GENERATING BALLOONS

The **Generate Balloons** tool is used to generate balloons that are attached to the drawing view of an assembly.



Bill of Material: Product1

Number	Quantity	Part Number	Type
1	1	Frame	Part
2	1	Bracket	Part
3	2	Pin	Part
4	1	Pulley	Part
5	1	Bushing	Part
6	1	Shaft	Part
7	1	washer	Part
8	1	nut	Part

Recapitulation of: Product1
Different parts: 8
Total parts: 9

Quantity	Part Number
1	Frame
1	Bracket
2	Pin
1	Pulley
1	Bushing
1	Shaft
1	washer
1	nut

The drawing sheet after generating balloons

Tutorial 1

In this tutorial, you will create the model shown in **Figure A**. After creating it you need to generate the front, top, and the isometric views in the **Drafting** workbench. You also need to generate the dimensions in the drawing views. The views and dimensions are shown in **Figure B**. **(Expected time: 45 min)**

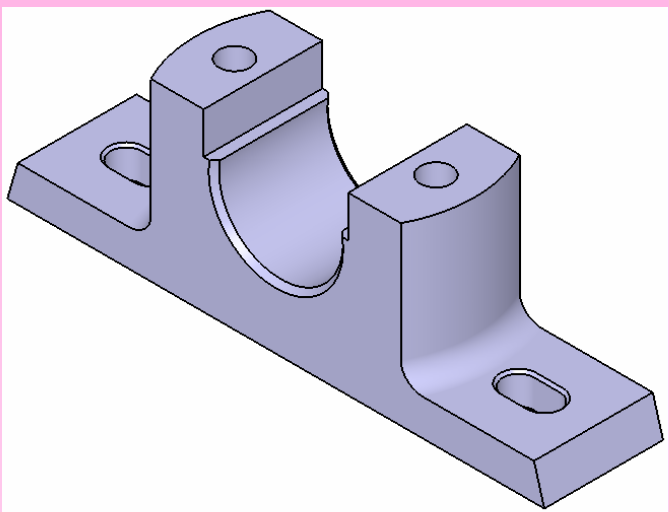


Figure A Model for Tutorial 1

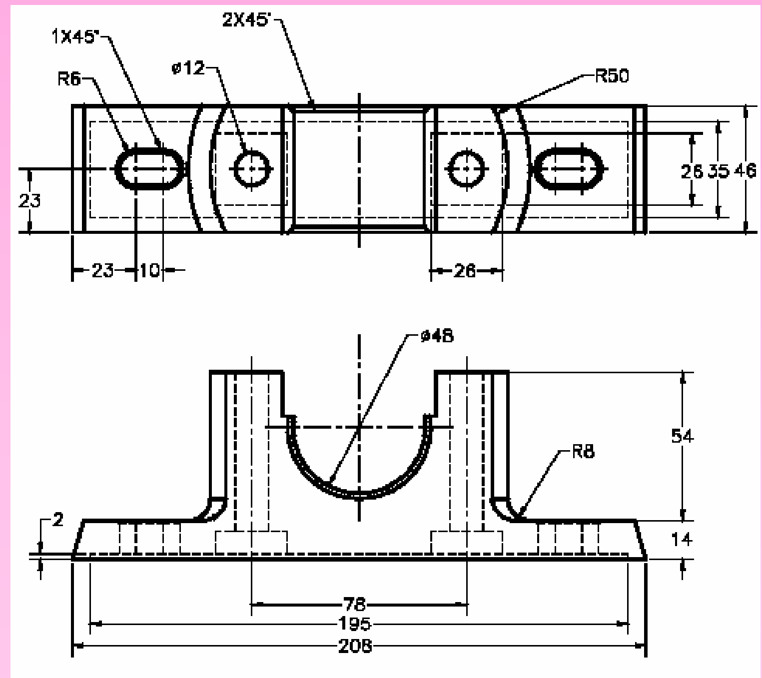


Figure B Views and dimensions for Tutorial 1

1. Create the model in the **Part** workbench and save the file in the c13 folder.
2. Start a new file in the **Drafting** workbench with the standard sheet of A2 size.
3. Set the projection standard to third angle.
4. Create a standard title block and frame in the background editing mode, as shown in **Figure C**.

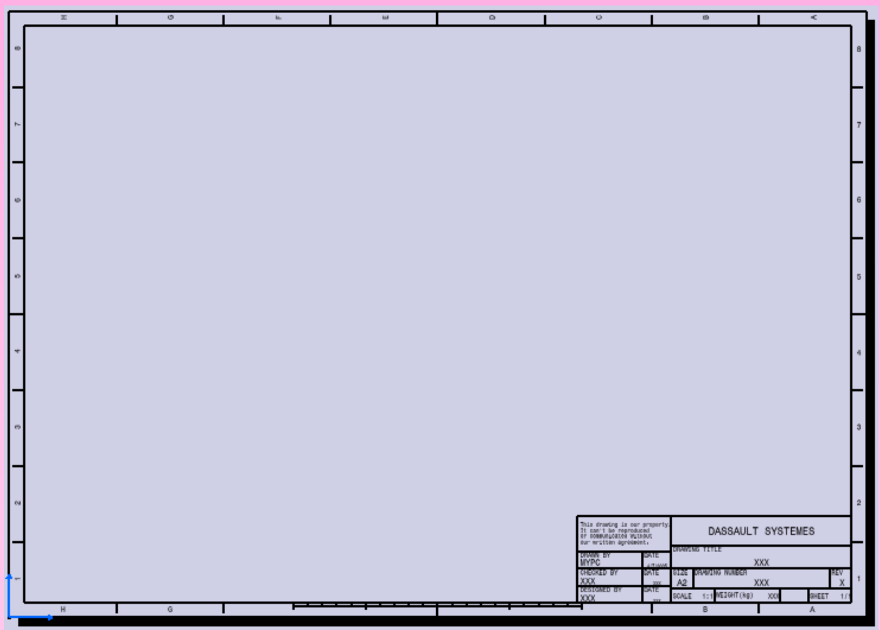


Figure C The drawing sheet after creating the title block and frame

5. Generate the front view, as shown in **Figure D**.

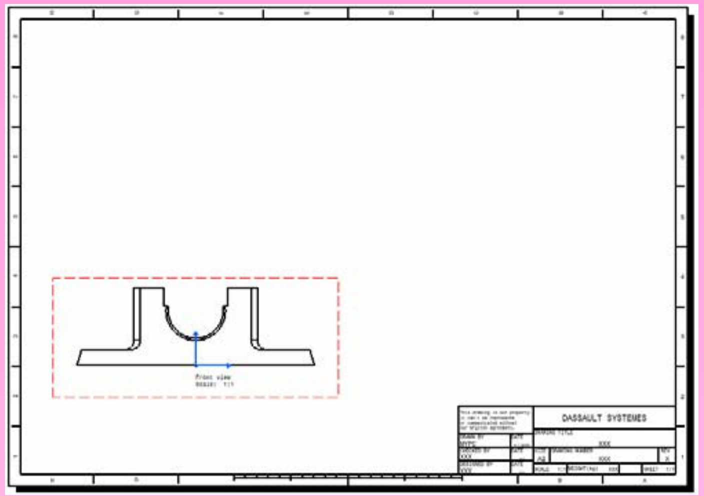


Figure D The drawing sheet after generating the front view

6. Generate the top view, as shown in **Figure E**.

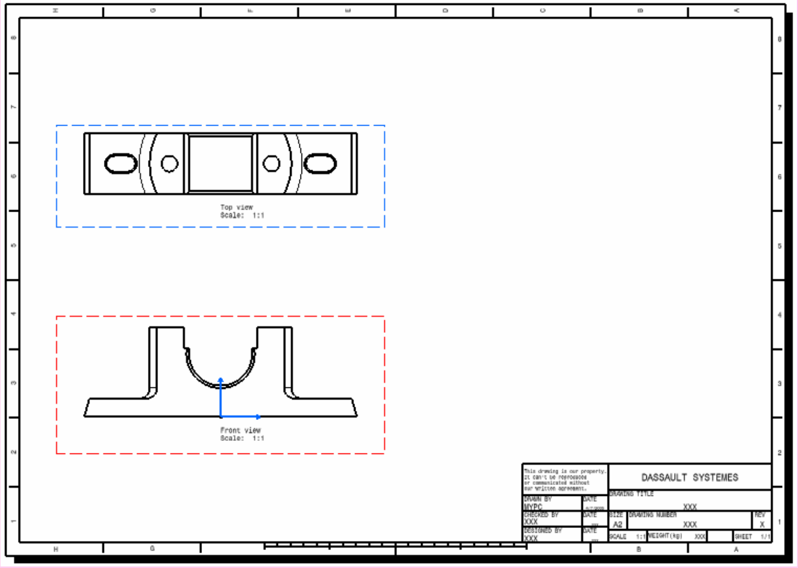


Figure E The drawing sheet after generating the top view

6. Generate the isometric view, as shown in **Figure F** and **Figure G**.

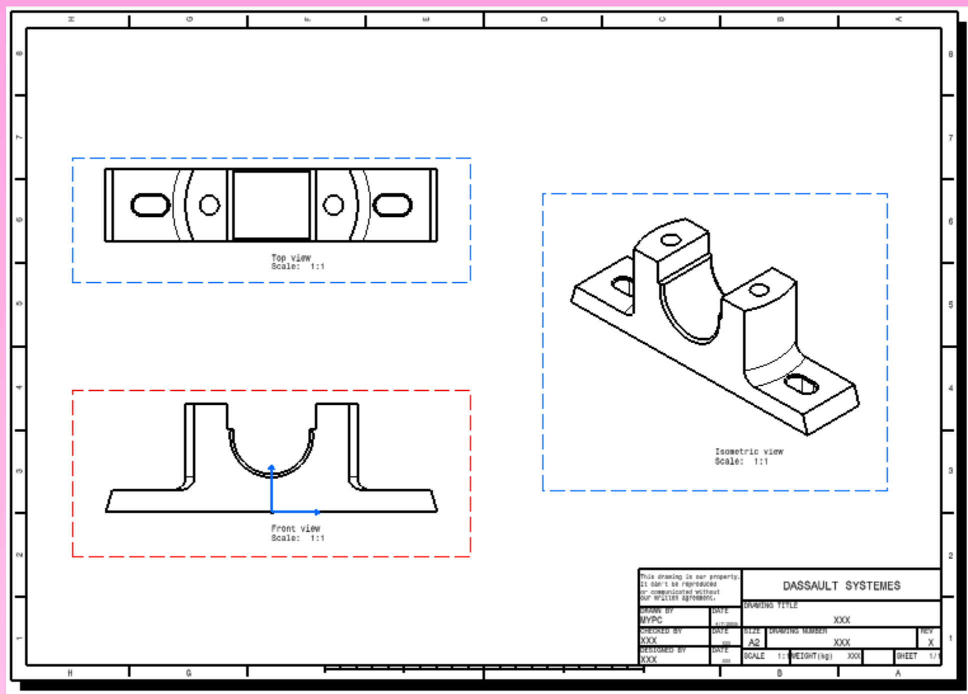


Figure F Drawing sheet after generating the isometric view

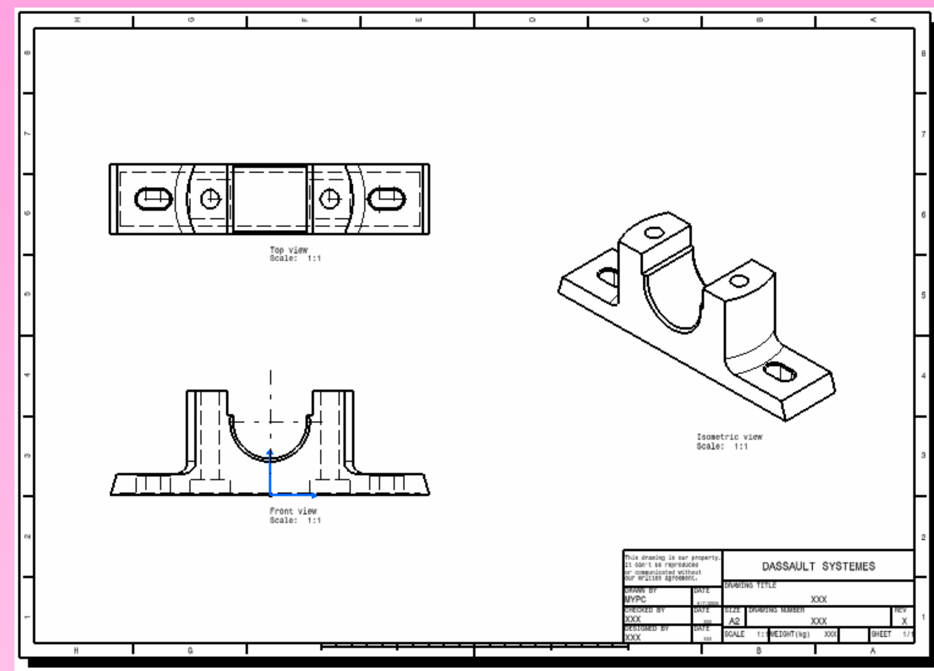


Figure G Drawing sheet after modifying the display

7. Generate the dimensions, as shown in **Figure H**.

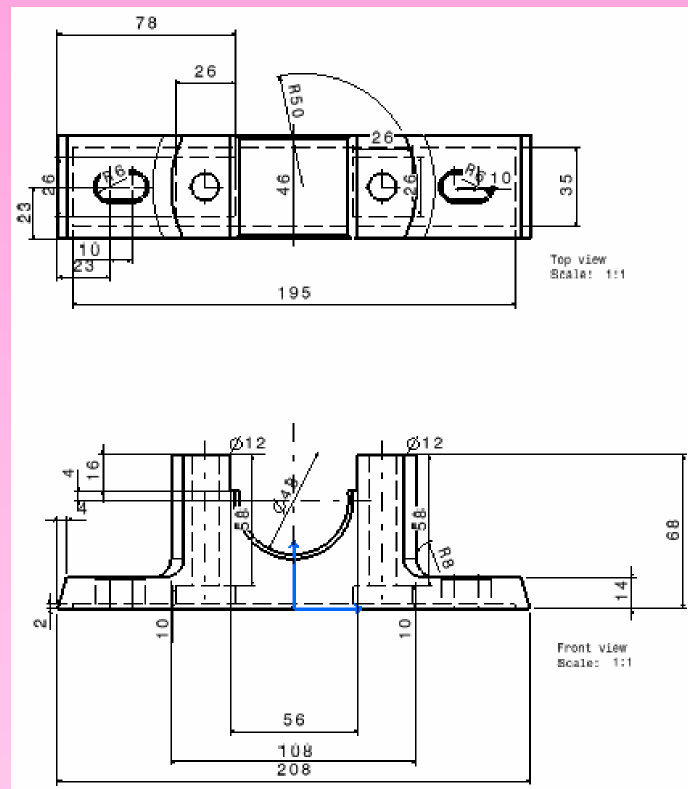


Figure H The drawing sheet after generating dimensions

8. Arrange the dimensions and delete the unwanted dimensions,as shown in **Figure I** and **Figure J**.

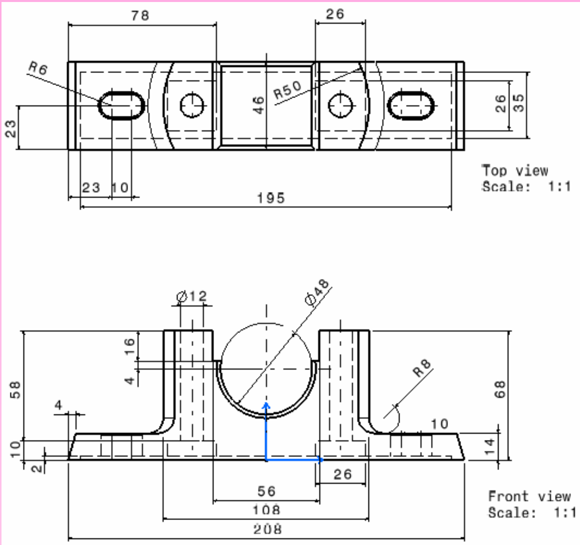


Figure I Drawing sheet after arranging dimensions

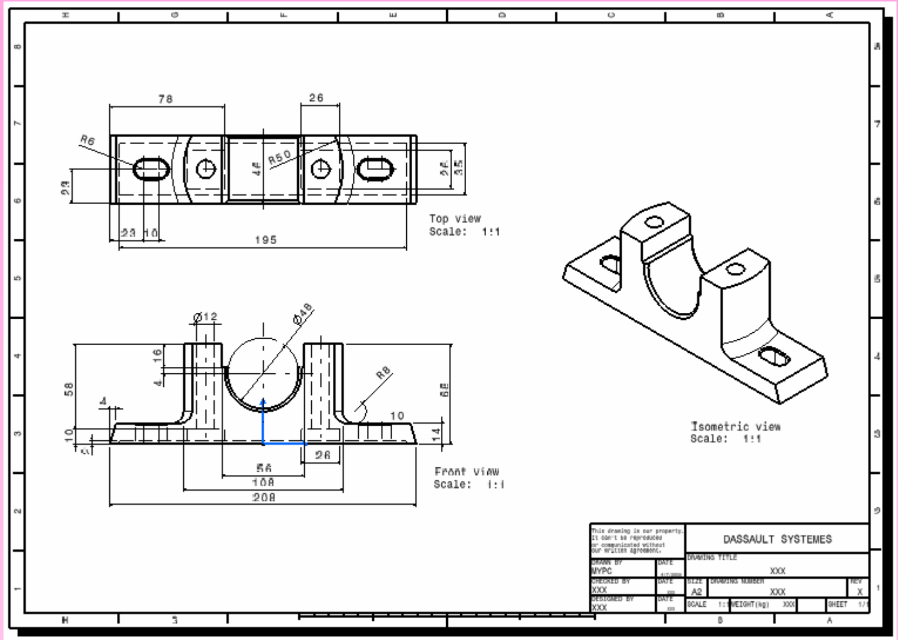


Figure J Final drawing sheet

9. Save the file in **\My Documents\CATIA\c13** folder and then close it.

Tutorial 2

In this tutorial, you will generate the front, top, right, and isometric views of the V-Block assembly created in the Exercise 1 of Chapter 12. You also need to generate the BOM and balloons. **(Expected time: 30 min)**

1. Copy the V-Block folder to the c13 folder.
2. Open the assembly file.
3. Start a new file in the **Drafting** workbench.
4. Generate the front view.
5. Generate the top view.
6. Generate the right view.
7. Generate the isometric view.

8. The drawing sheet after generating all the drawing views is shown in **Figure A**.

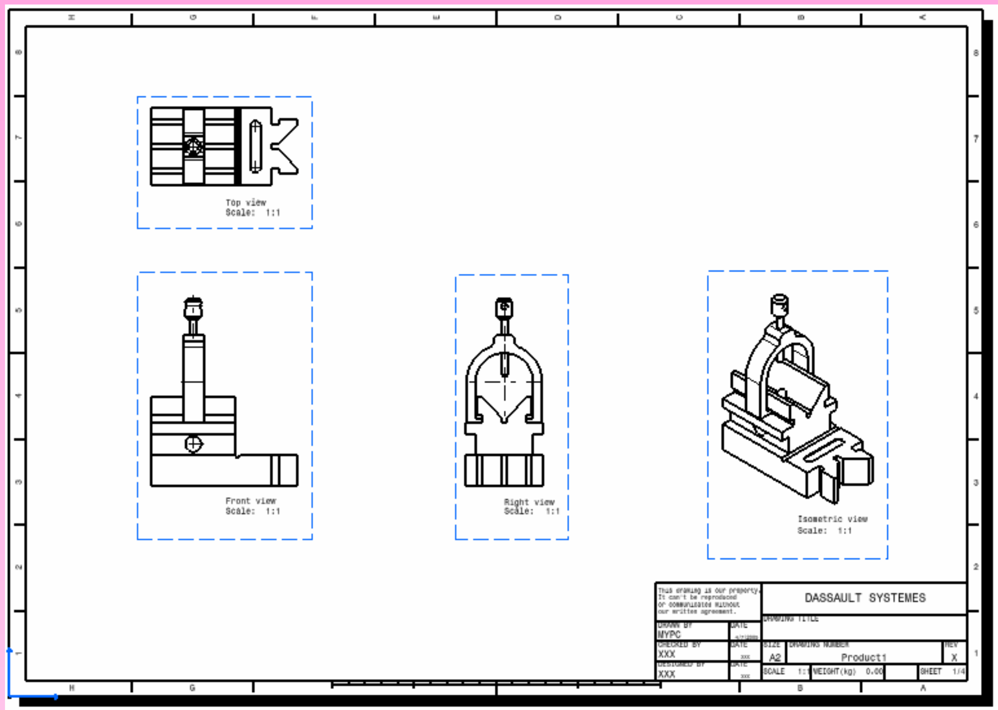


Figure A The drawing sheet after generating the front, top, right, and isometric views

9. Generate the BOM.
10. Generate the balloons.
11. The final drawing sheet after generating the BOM and balloons is shown in **Figure B**.

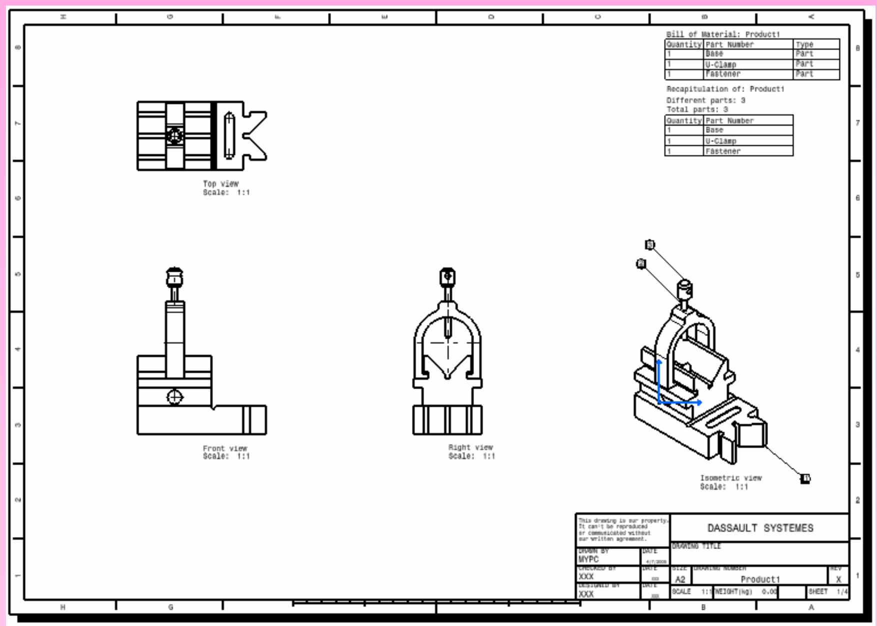


Figure B The final drawing sheet

12. Save the file in **\My Documents\CATIA\c13** folder and then close it.

Exercise 1

Generate the front, top, right, and isometric views of the Blower assembly created in Tutorial 1 of Chapter 11. The drawing view will be generated with the scale factor of 1:5. After generating the drawing views, you need to generate the BOM and balloons, as shown in **Figure A**. **(Expected time: 30 min)**

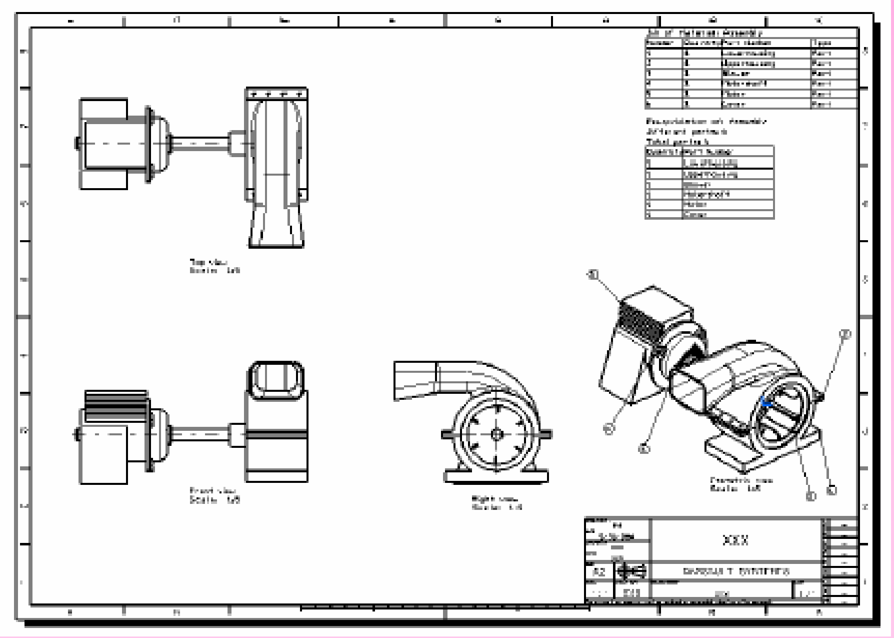


Figure A Drawing views for Exercise 1