CATIA V5

• 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 familiar with the previous releases, you can upgrade your designing skills with the tremendous improvement in this latest release.

• CATIA, 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 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 available 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.

• Drafting Workbench
• Assembly Design Workbench

• The **Assembly Design** workbench is used to assemble the components using the assembly constraints available in this workbench.

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.

SYSTEM REQUIREMENTS

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

• System unit: An Intel Pentium III or Pentium 4 based workstation running Windows XP, Windows Vista, Windows 7, or Windows 8.

• Memory: 2 GB of RAM is the minimum recommended for all applications. 1 GB of RAM is recommended for DMU applications.

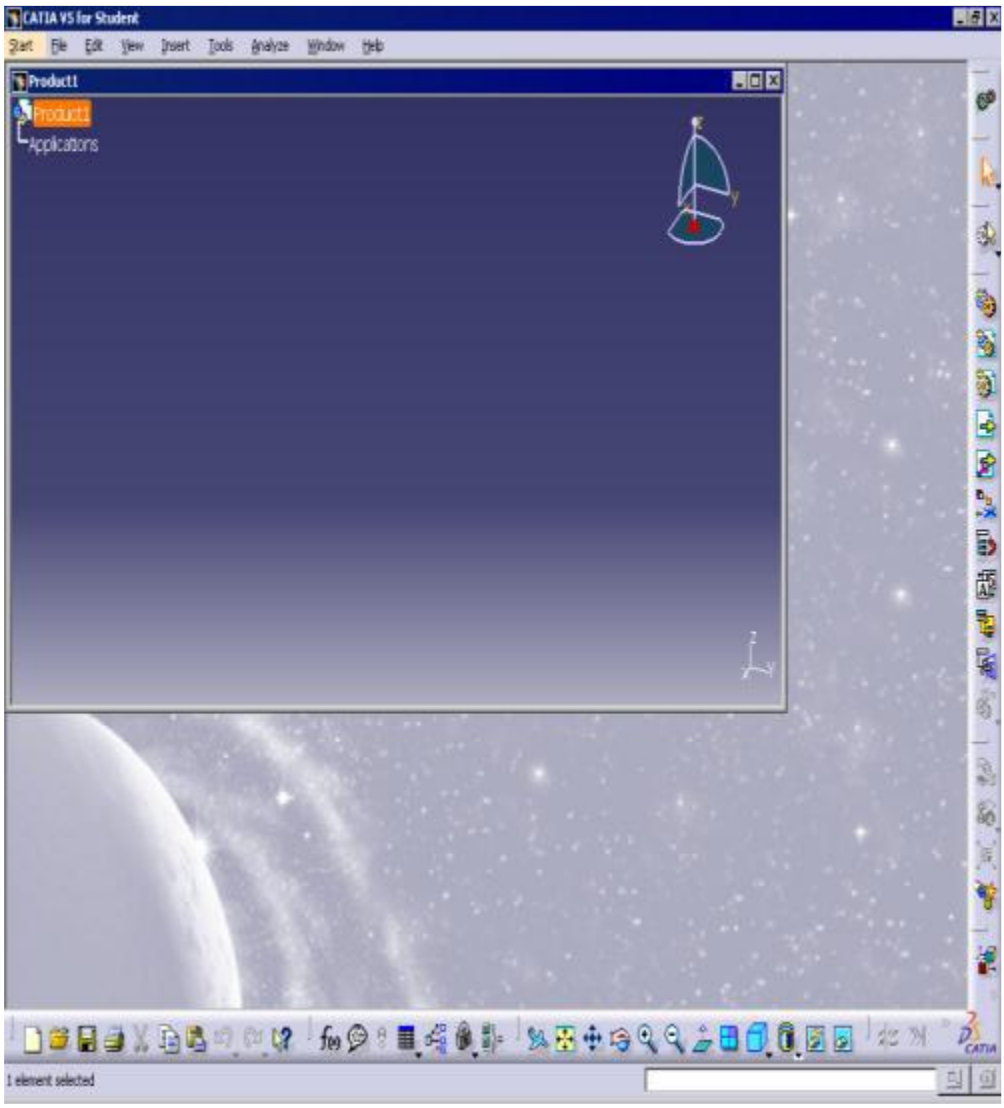
• Disk drive: 4 GB Disk Drive space (Minimum recommended size)

• Internal/External drives: A CD-ROM drive is required for program installation.

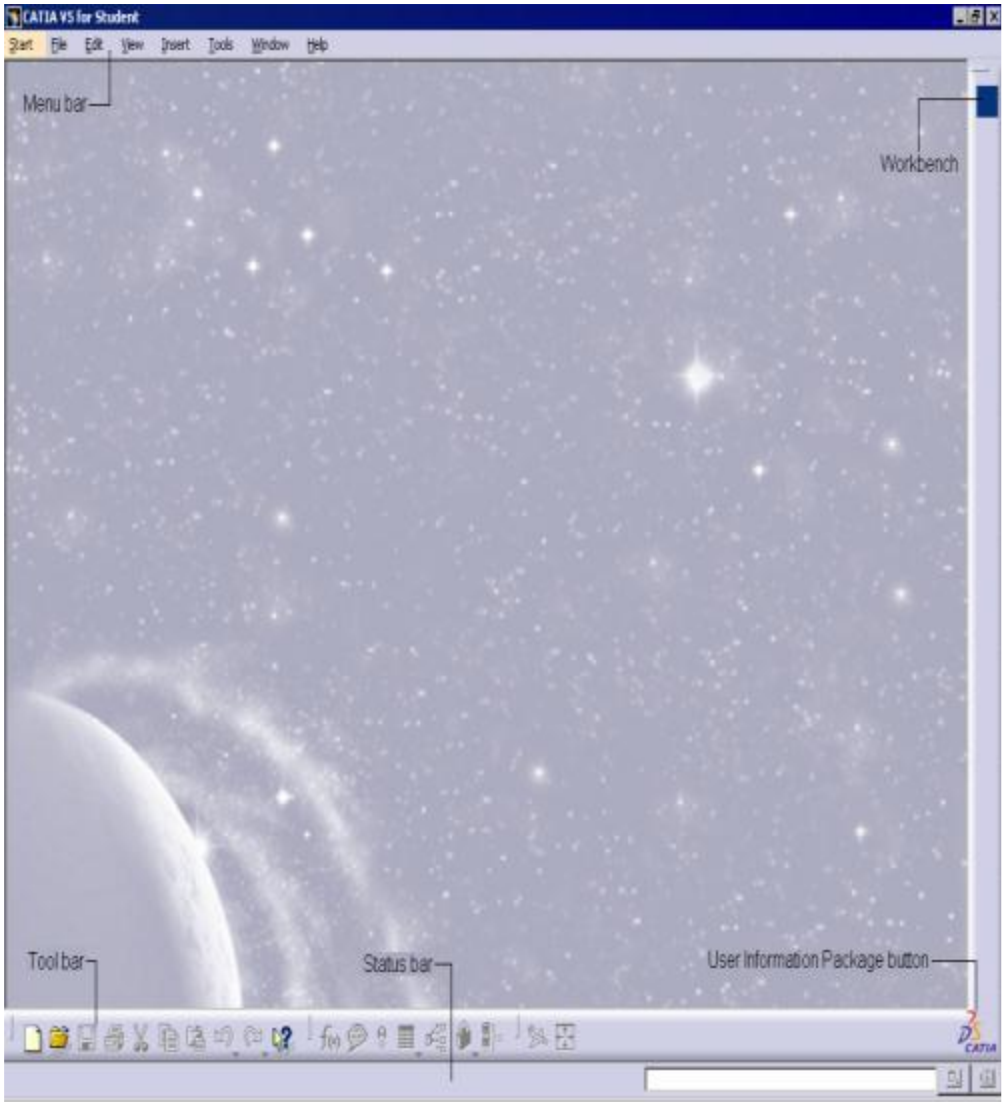
• Display: A graphic color display compatible with the selected platform-specific graphic adapter. The minimum recommended monitor size is 17 inches.

• 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.

Introduction



The initial screen that appears after starting CATIA V5



The screen that appears after closing the initial product file

IMPORTANT TERMS AND DEFINITIONS

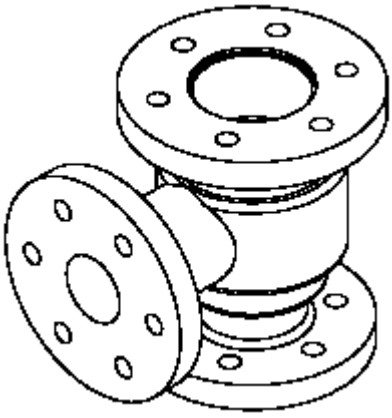
- **Feature-based Modeling**

• Feature is defined as the smallest building block that can be modified individually.

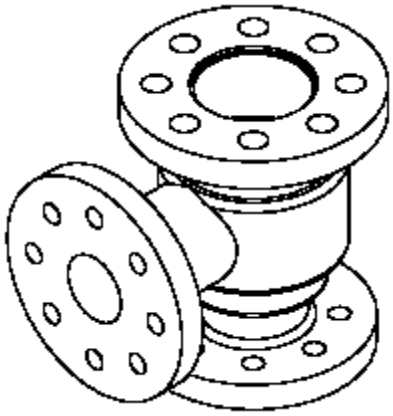
• In CATIA, solid models are created by integrating a number of these building blocks.

• 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

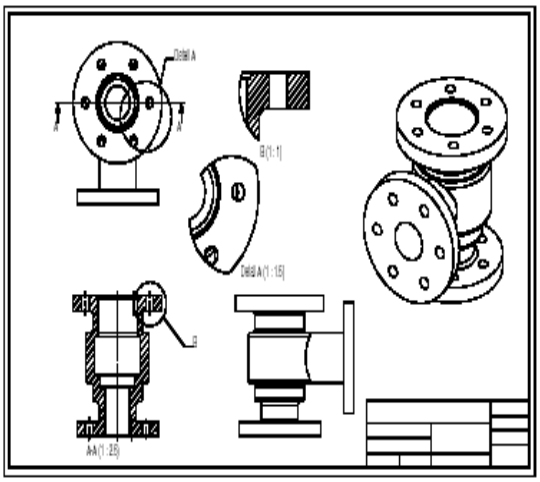


Modified body of pipe housing

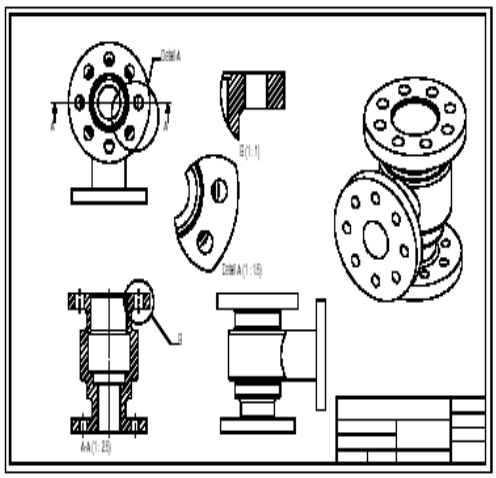
Bidirectional Associativity

• 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



The drawing views of the body part, before making the modifications



The drawing views, after modifications

Introduction

- 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.

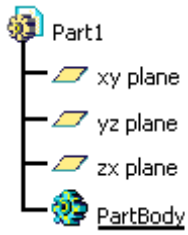
- CATDrawing

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

• Specification Tree

• It is a manager that 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.



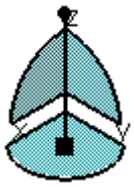
The specification tree that appears on starting a new CATPart file

• 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 using compass.

• By default, it appears on the top right corner of the geometry area.



The Compass

Introduction

• 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.

Introduction

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

Introduction

• 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**

Introduction

• Part Body

• 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

• Wireframe elements are used as a substitute to entities drawn in the **Sketcher** workbench.

• Surface

• A surface is a geometric feature with no thickness.

• Surfaces 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

• In case a feature, body, or a sketch is not 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

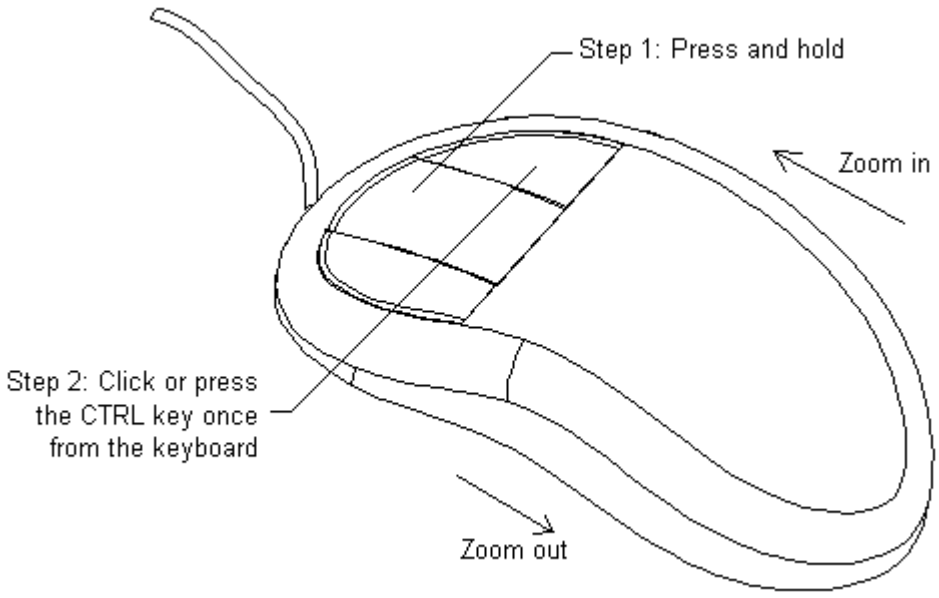
• This option brings the selected feature, body, or sketch in the specification tree to the middle left portion of the geometry area

Introduction

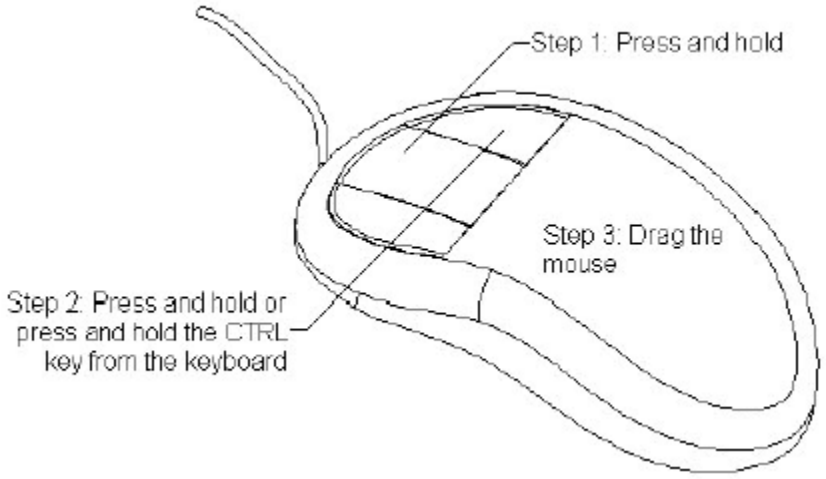
UNDERSTANDING THE FUNCTIONS OF THE MOUSE BUTTONS

• 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



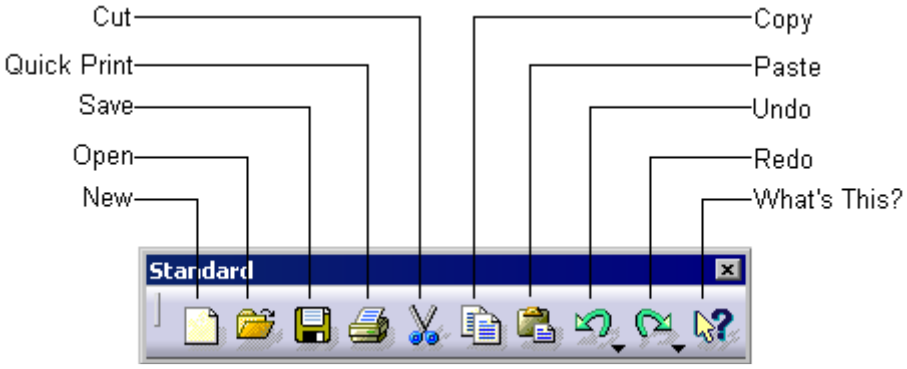
Using the three button mouse to perform the rotate operation

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



• 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.
• Current Information or Dialog Box

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 GETTING STARTED WITH CATIA V5

• Dialog Box Display Button

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

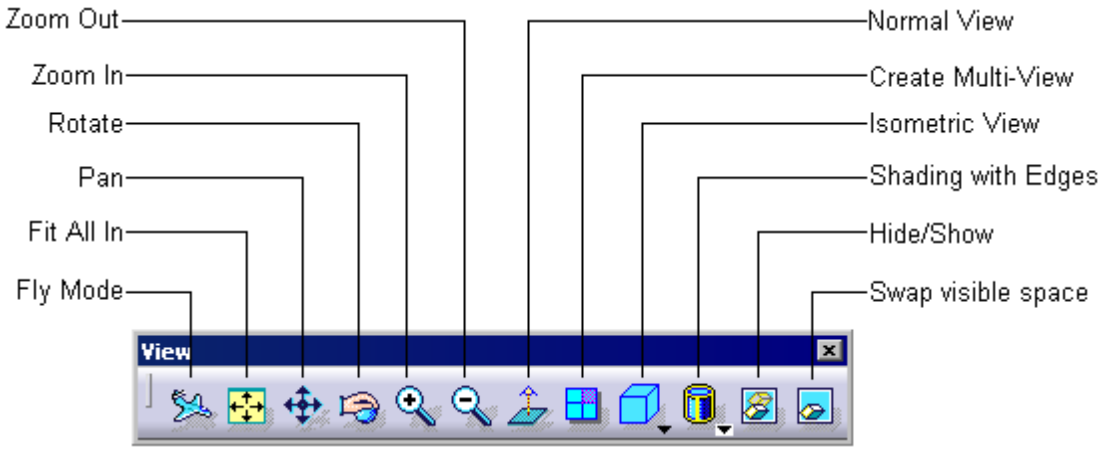
• User Information Package Button

• • Choosing this button will open a window with a default link that is C:\Program Files\Dassault Systemes\B13\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



Select Toolbar



The Select toolbar

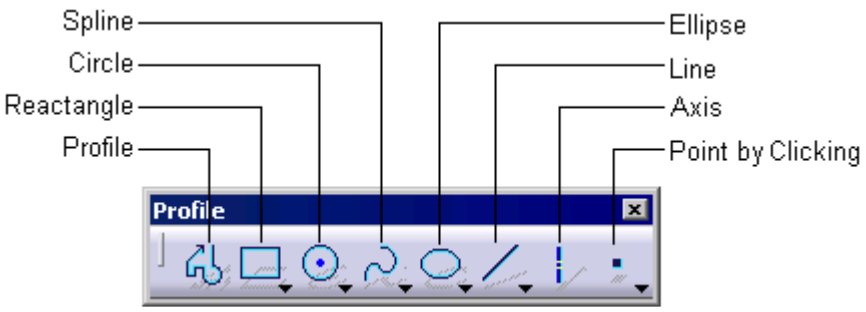
Sketcher Toolbar



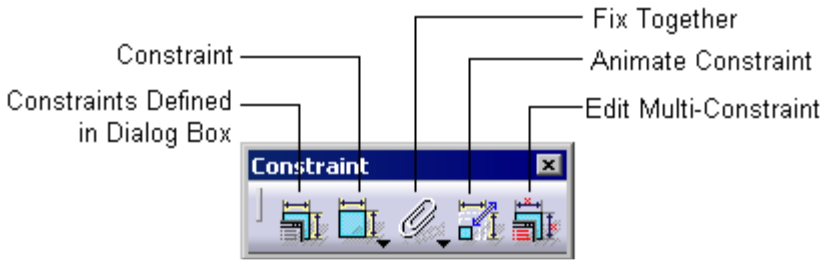
The Sketcher toolbar

Profile Toolbar

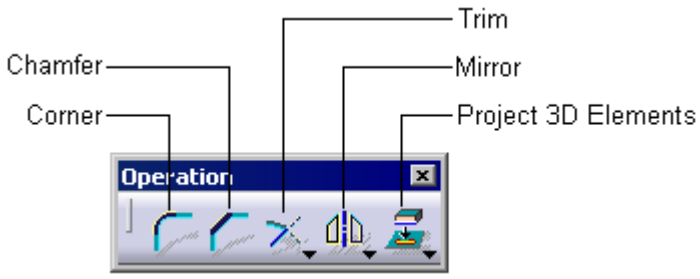
The Profile toolbar



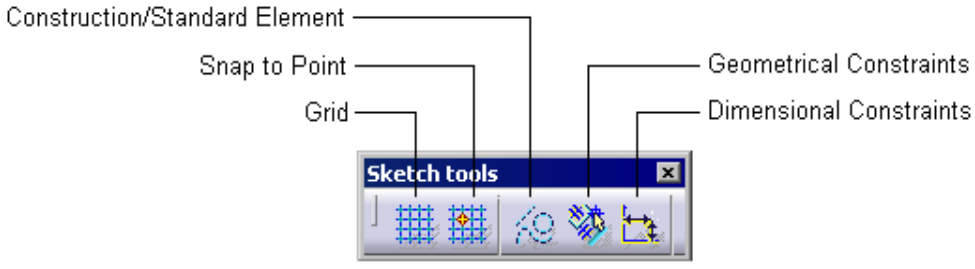
The Constraint toolbar



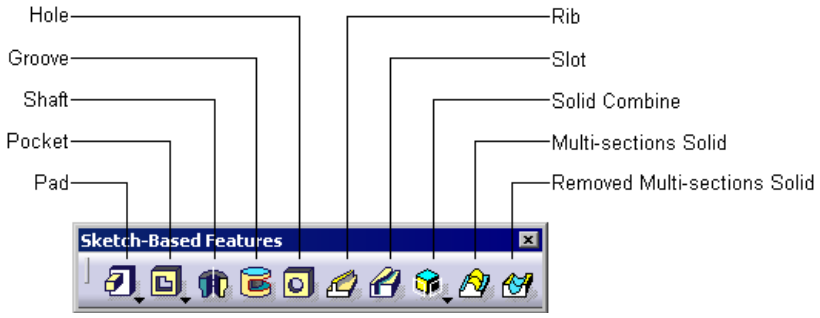
The Operation toolbar



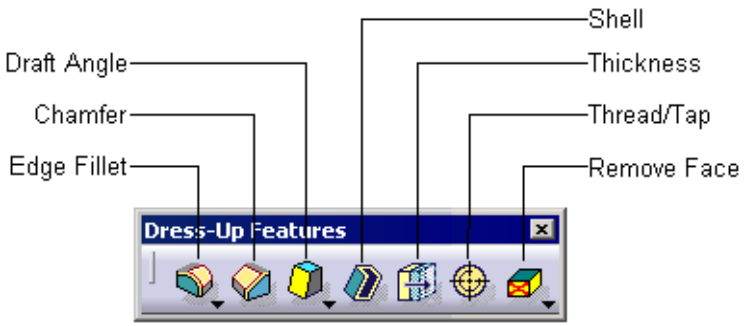
• Sketch tools Toolbar



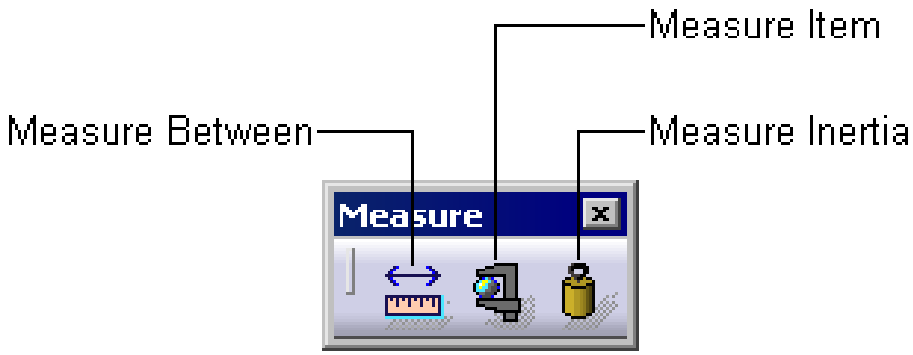
Sketch-Based Features Toolbar



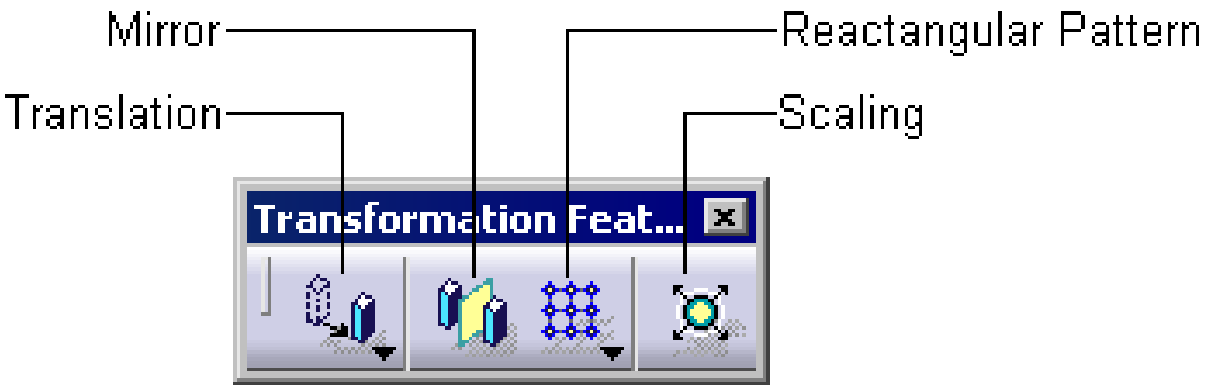
- **Dress-Up Features Toolbar**



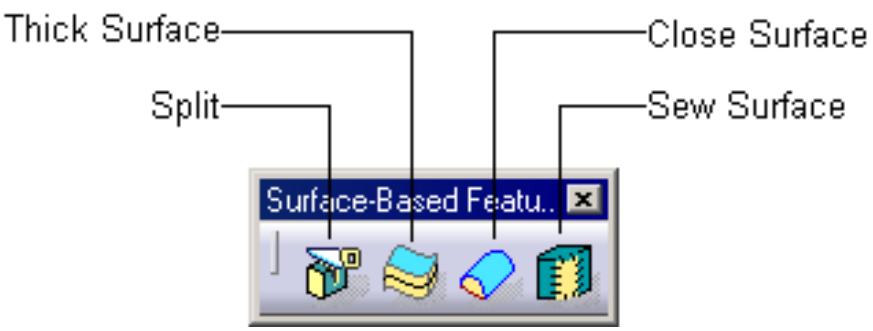
- **Measure Toolbar**



- **Transformation Features Toolbar**



- **Surface-Based Features Toolbar**



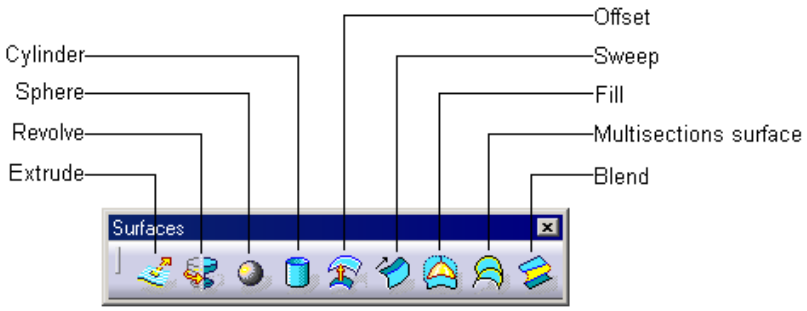
- **Apply Material Toolbar**



- **Wireframe and Surface Design Workbench Toolbars**

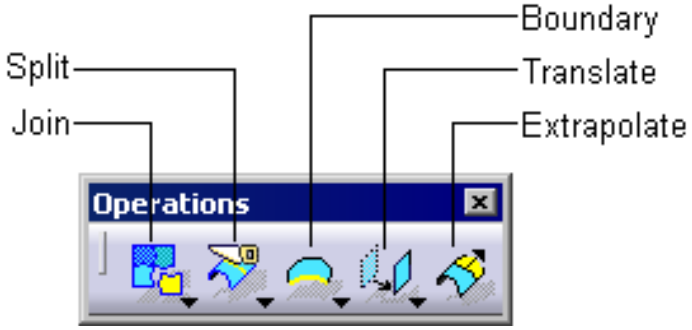
• You can invoke the **Wireframe and Surface Design** workbench from the main menu bar by choosing **Start > Mechanical Design >Wireframe and Surface Design**.

- **Surfaces Toolbar**

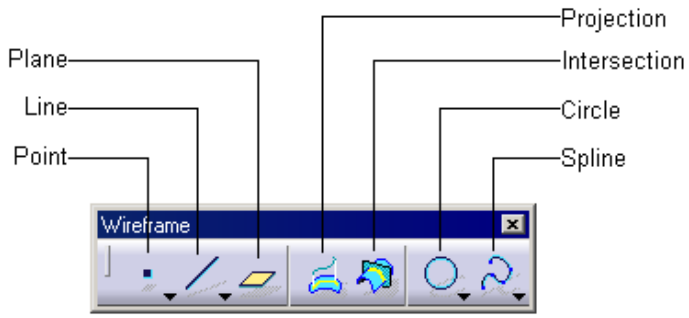


Introduction

- **Operations Toolbar**



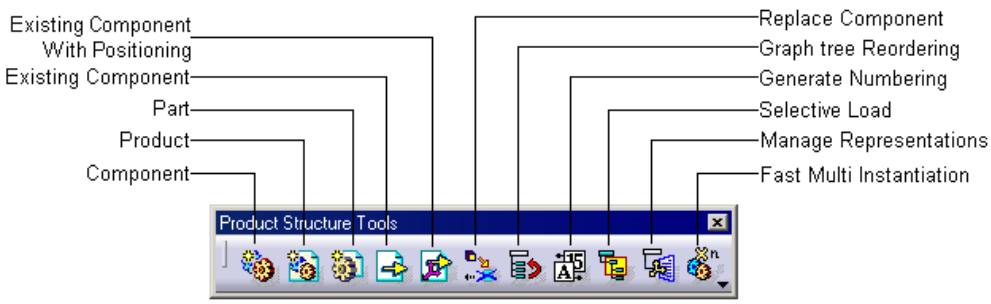
• Wireframe Toolbar



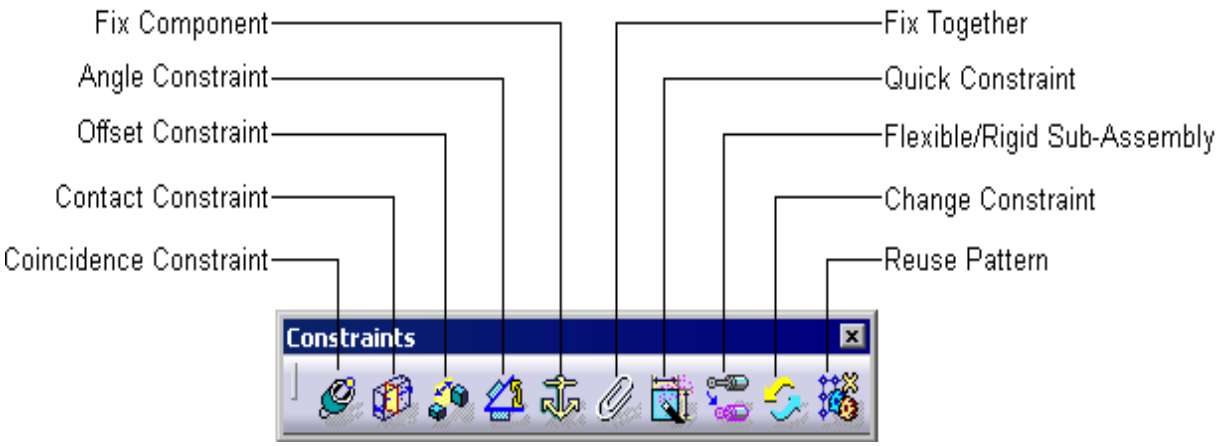
• Assembly Design Workbench Toolbars

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

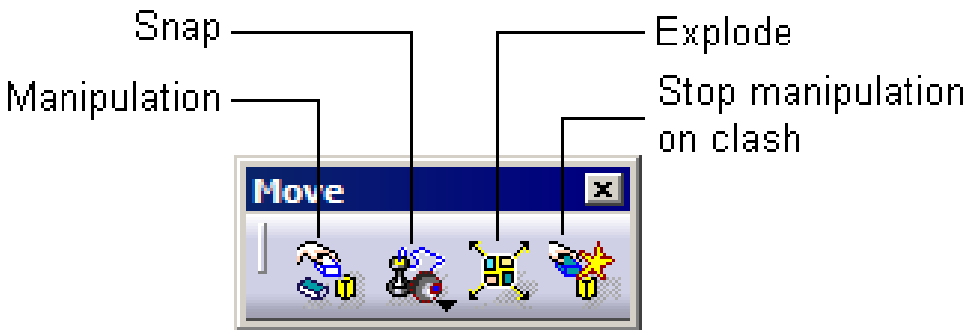
• Product Structure Tools Toolbar



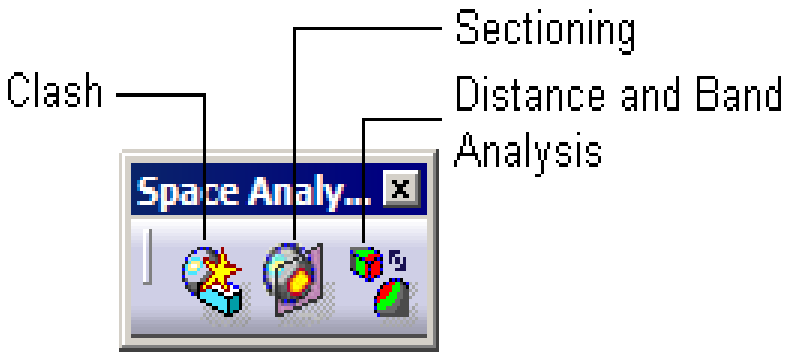
• Constraints Toolbar



- **Move Toolbar**



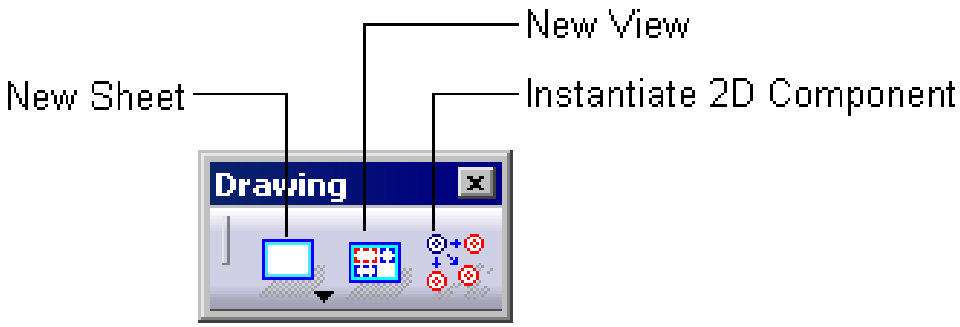
- **Space Analysis Toolbar**



• 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



• Views Toolbar

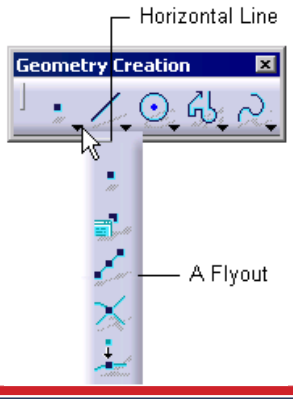
Detail

Offset Section

Front

• Generation Toolbar





The Generation toolbar The flyout that

appears when a

down arrow is chosen

➢ HOT KEYS

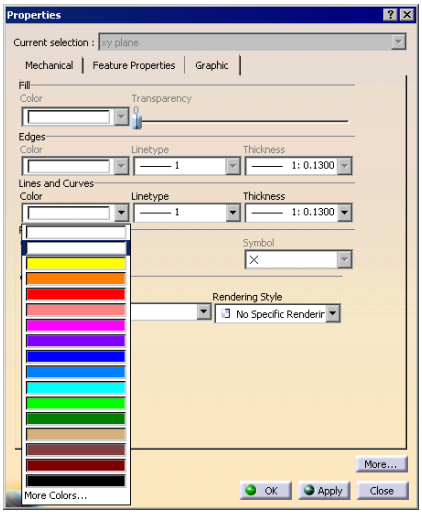
• CATIA V5 is more popularly known for its icon driven structure.

• The hot keys that can be used with CATIA V5, along with their functions, are listed in the table given below.

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 color of the screen and for displaying the entities on the screen.



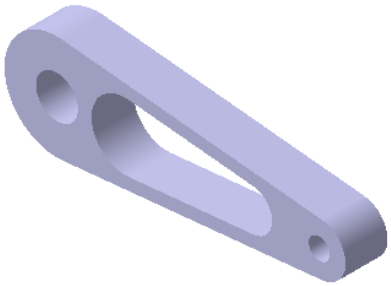
The Properties dialog box

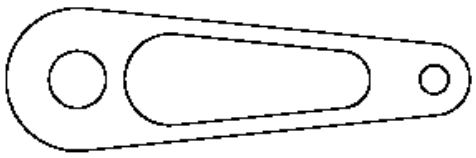
Chapter 1

➢ THE SKETCHER WORKBENCH

• Most of the components designed using CATIA 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.

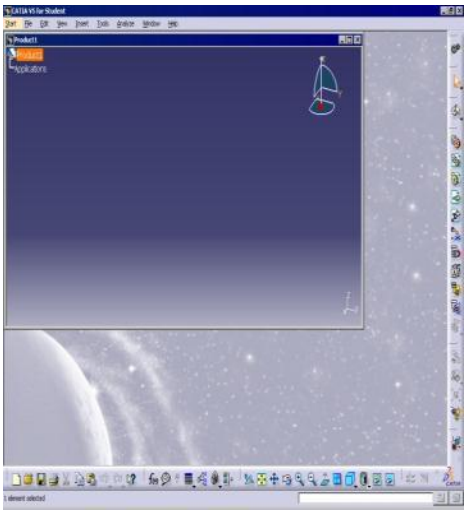




Solid Model of a Link **Base sketch for the solid model.**

➢ STARTING A NEW FILE

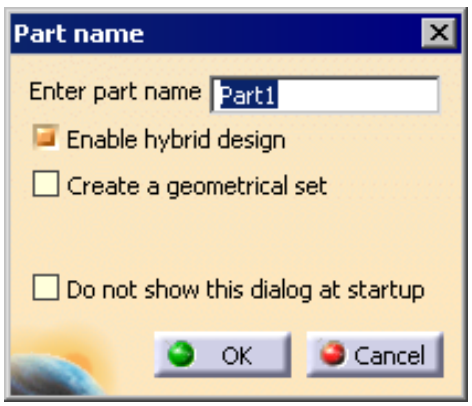
• When you start CATIA V5R14, a new **Product** file with the name **Product1** is displayed on the screen, as shown in the figure.



Initial screen that appears after starting CATIA V5R14

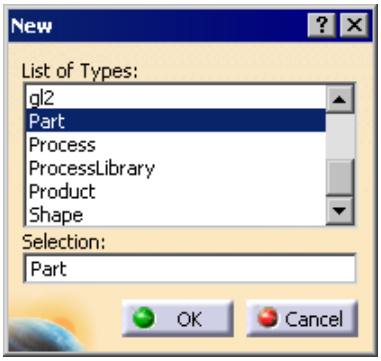
• Choose **Start > Mechanical Design > Part Design** to make sure that you are in the **Part Design** workbench.

• The **Part** name dialog box is displayed, as shown in the figure.



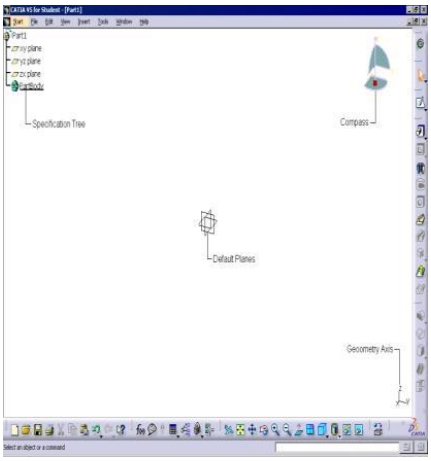
The Part Name dialog box

• Alternatively, you can choose the **File** option from the menu bar; the **New** dialog box is displayed, as shown in the figure.



The New dialog box

• A new file in the **Part Design** workbench is displayed on the screen, as shown in the figure.

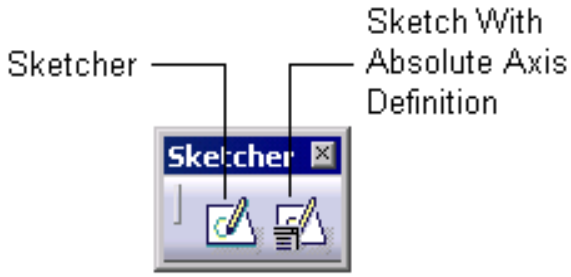


A new Part Design workbench file

➢ INVOKING THE SKETCHER WORKBENCH

• To invoke the **Sketcher** workbench, choose the down arrow on the right of the **Sketcher** button in the **Sketcher** toolbar; a flyout appears.

• The **Sketcher** flyout as an independent toolbar, is as shown in the figure.

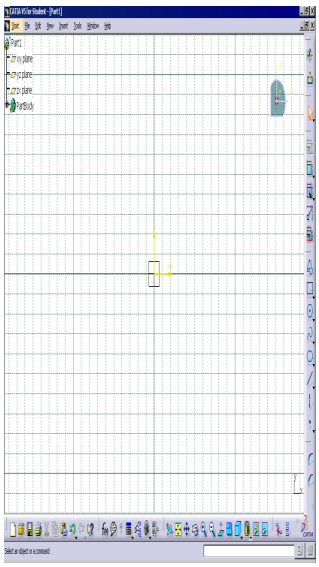


The Sketcher toolbar

• Invoking the Sketcher Workbench using the Sketcher Button

• To invoke the **Sketcher** workbench using the **Sketcher** tool,choose the **Sketcher** 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.

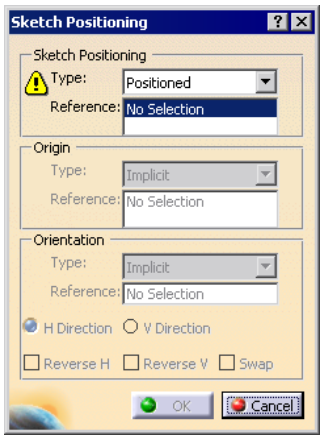


Sketcher workbench invoked using the YZ plane as the sketching plane

- **Invoking the Sketcher Workbench using the Sketch with Absolute Axis Definition Button**

• To invoke the **Sketcher** workbench using this option, choose the **Sketch With Absolute Axis Definition** button from the **Sketcher** toolbar.

• The **Sketch Positioning** dialog box will appear, as shown in the figure.



The Sketch Positioning dialog box

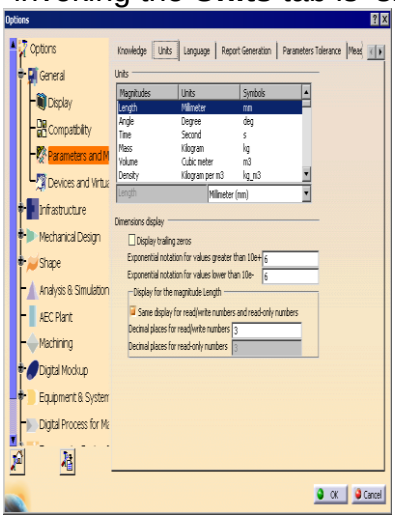
➢ SETTING UP THE SKETCHER WORKBENCH

• 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.



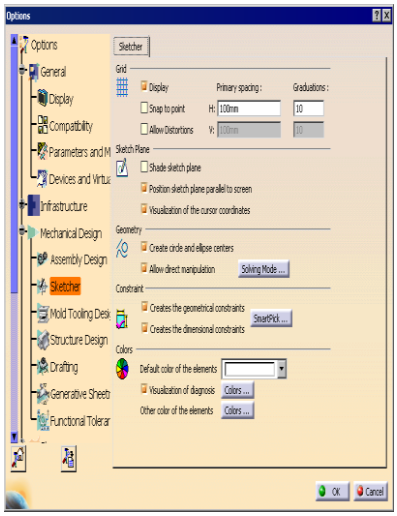
The Options dialog box with the Units tab chosen

• 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.



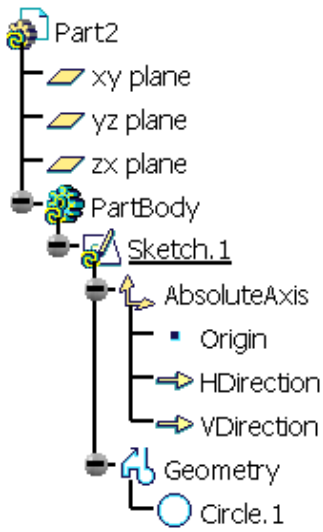
The Options dialog box with the Sketcher option selected

➢ UNDERSTANDING THE SKETCHER TERMS

- **Specification Tree**

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

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

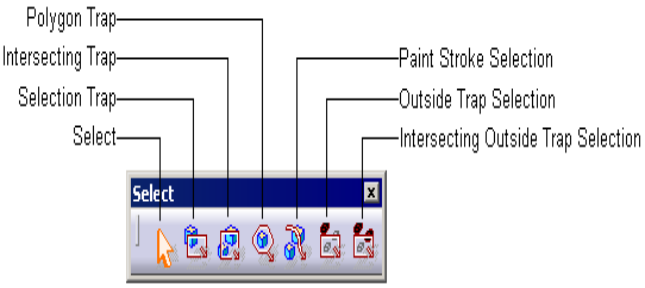


The expanded form of the Specification tree

- **Sketcher Toolbar**



The Sketcher toolbar



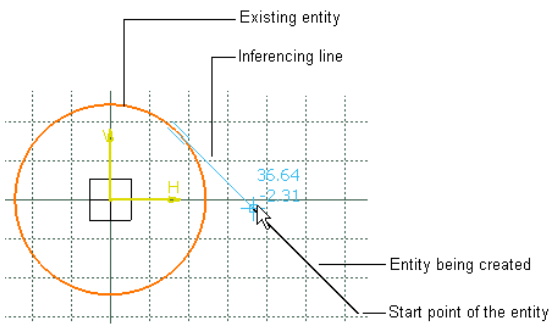
The Select toolbar

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

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

- **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

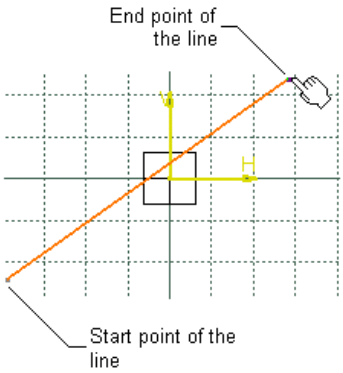
➢ DRAWING SKETCHES USING THE SKETCHER TOOLS

- **Drawing Lines**



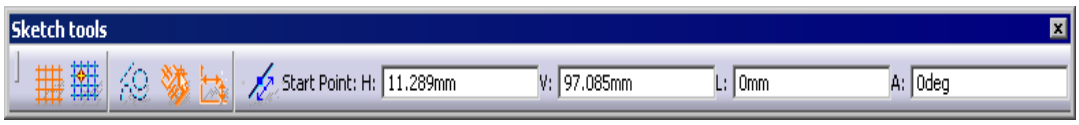
• CATIA being parametric in nature, allows the user to drawa line of any length and at any angle, and then change it to the desired length and angle.

Drawing Lines by Specifying Points in the Geometry Area



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

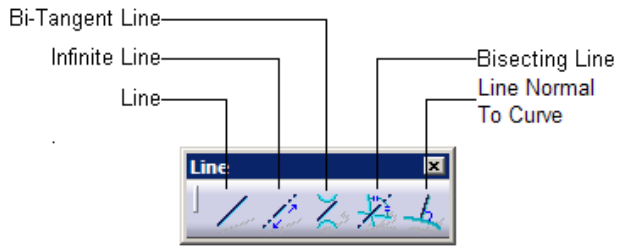
- **Drawing Lines Using the Sketch tools Toolbar**



The expanded form of the

Sketch tools toolbar after invoking the Line tool

• Drawing Lines 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.

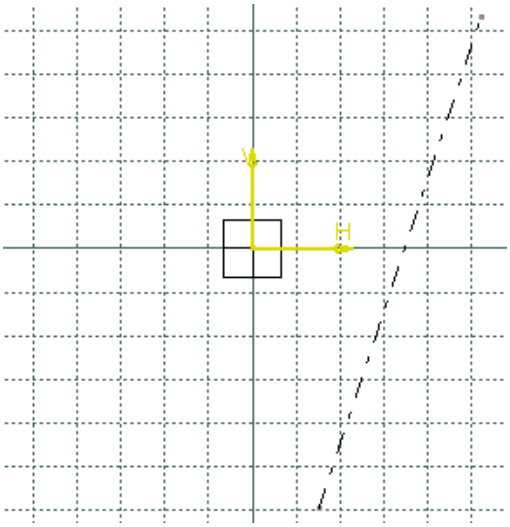
• Drawing Lines Normal To Curve

• 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.

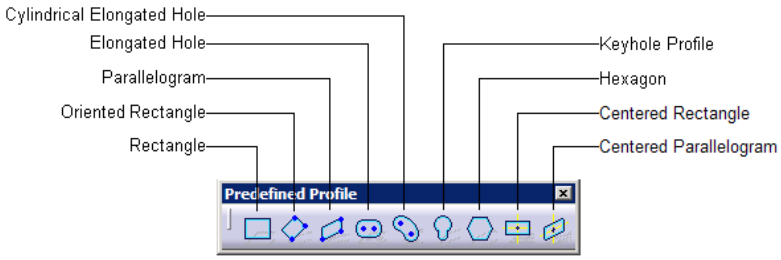


An axis drawn in the geometry area

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

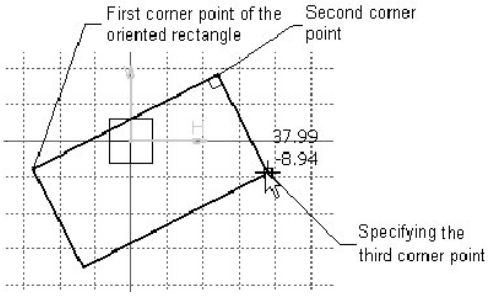
- Rectangle is a basic geometry that comprises of four sides.

- **Drawing Rectangles**



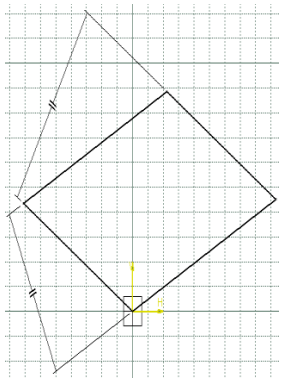
The Predefined Profile Toolbar

- **Drawing Oriented Rectangles**



Selecting the corner points to draw an oriented rectangle

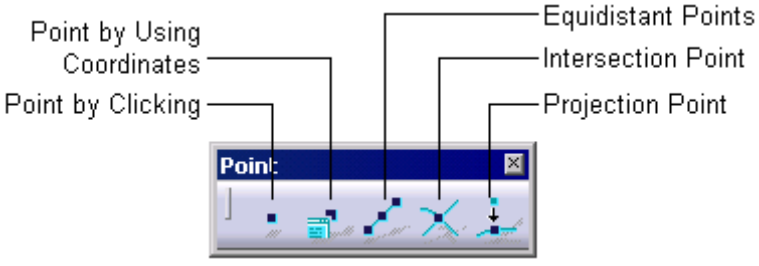
- **Drawing Parallelograms**



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

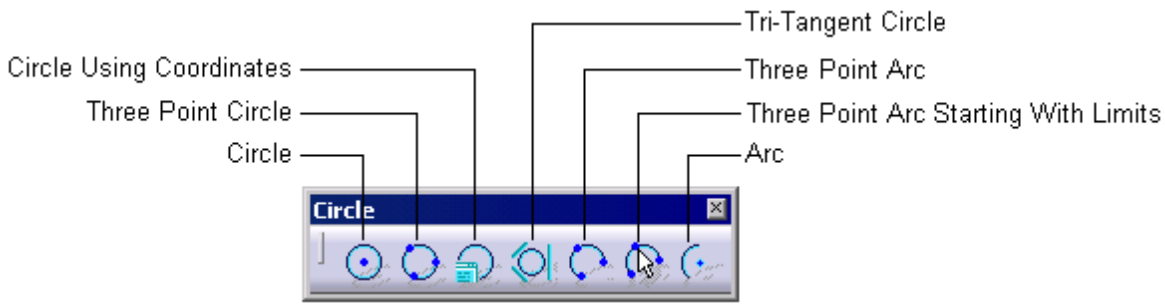
- **Drawing Points by Clicking**

• To draw points by clicking, invoke the **Point by Clicking** tool from the **Point** toolbar; the **Sketch** tools toolbar expands.

- **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

• 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 is displayed.

• You can specify the coordinate values of the center point and radius, using the options in this dialog box.

• Drawing Tri-Tangent Circles

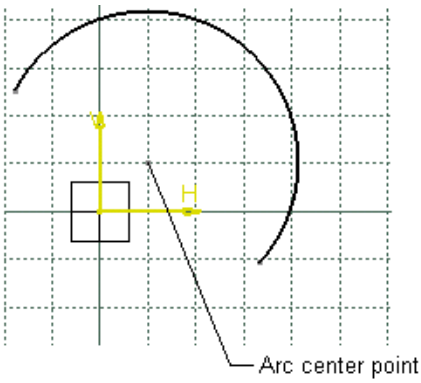
• To draw a tri-tangent circle, invoke the **Tri-Tangent Circle** tool from the **Circle** toolbar.

• Drawing Arcs

• An arc is a geometric element that forms a sector of a circle or ellipse.

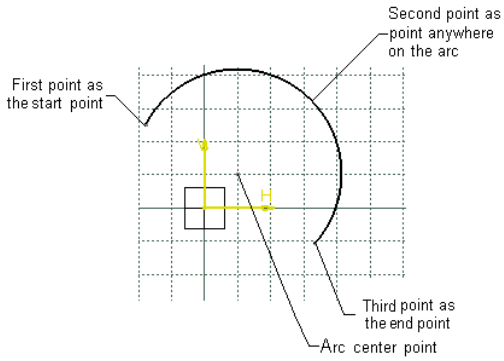
• The tools to draw arcs are available in the **Circle** toolbar.

• Drawing an Arc by Defining the Center Point



An arc

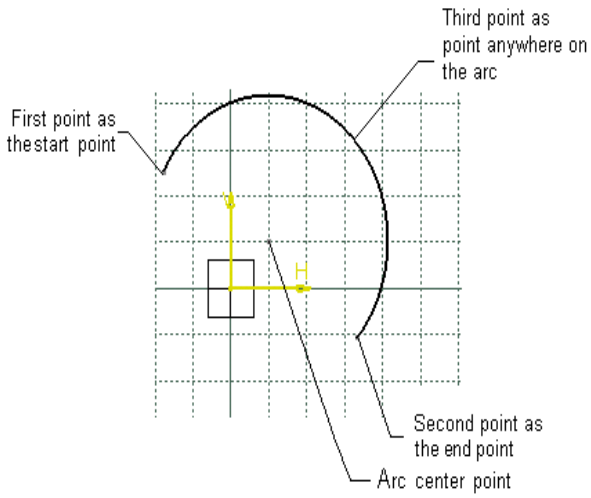
• Drawing Three Point Arc



Selecting points to draw a three point arc

Drawing Three Point Arcs Starting With Limits

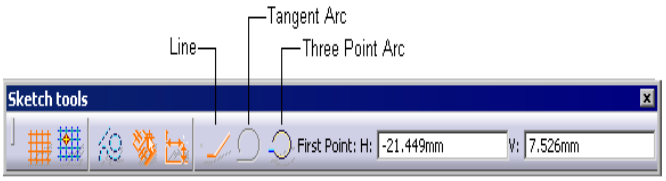
• To draw a three point arc starting with limits type of arc, invoke the **Three Point Arc With Limits** tool from the **Circle** toolbar.



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.

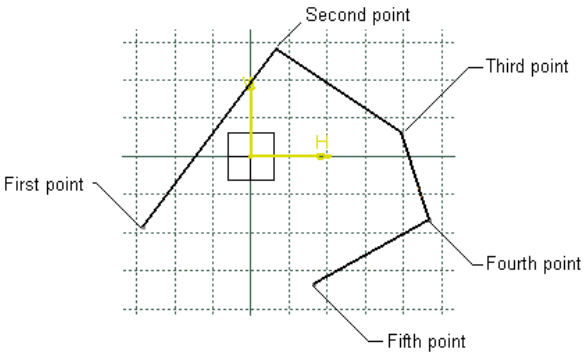


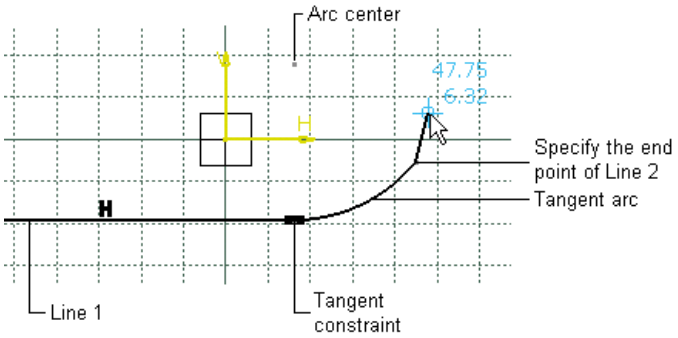
An open profile drawn using the Profile tool

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

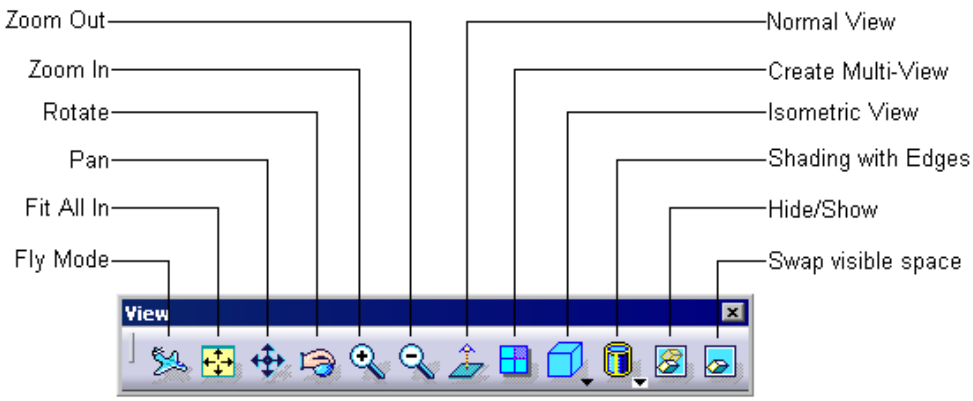
• Drawing Three Point Arcs using the Profile Tool

• 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 allthe sketched elements or geometry are included in the available 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 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 plane in the current Sketcher workbench.

- **Hide/Show**

• To hide a sketcher element, or geometric element, you can invoke the **Hide/Show** tool by choosing the **Hide/Show** button from the **View** toolbar.

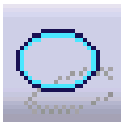
- **Swap Visible Space**

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

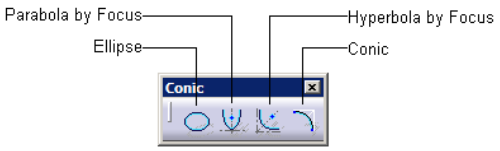
Chapter 2

➢ OTHER SKETCHING TOOLS

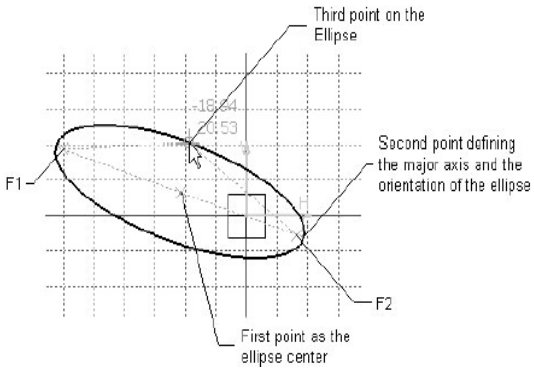
- **Drawing an Ellipse**



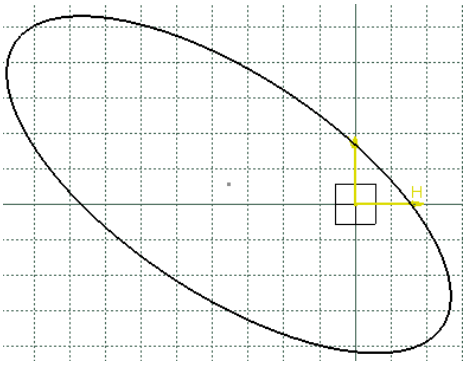
To draw an ellipse, invoke the **Ellipse** tool by choosing the **Ellipse** button from the **Conic** toolbar as shown in the figure.



The Conic toolbar



Specifying the three points to draw an ellipse

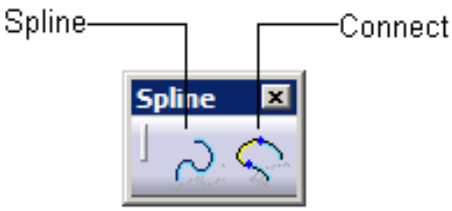


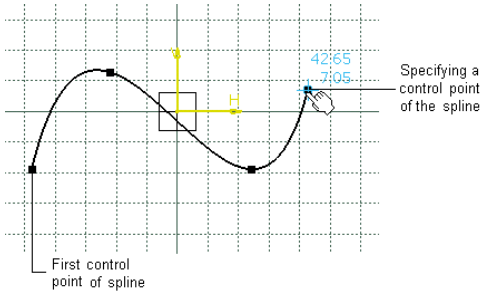
The resulting ellipse

• 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.





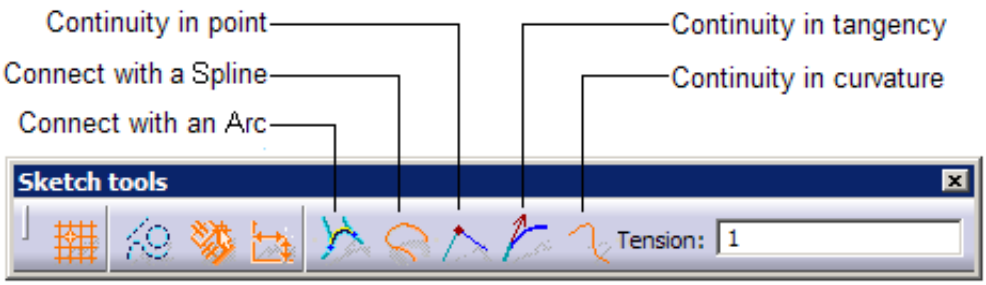
The Spline toolbar

Specifying points for drawing a spline

• Connecting two Elements by an Arc or Spline

• 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.



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.

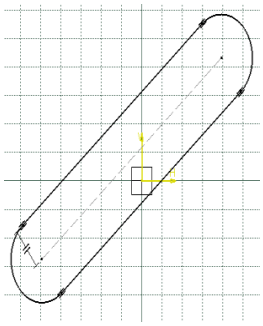
• Connecting Two Elements with an Arc

• To connect two selected elements with an arc, choos **Connect** button from the **Spline** toolbar.

• Choose the **Connects with an Arc** button from the tools toolbar; you are prompted to select the first elconnected.

• Drawing Elongated Holes

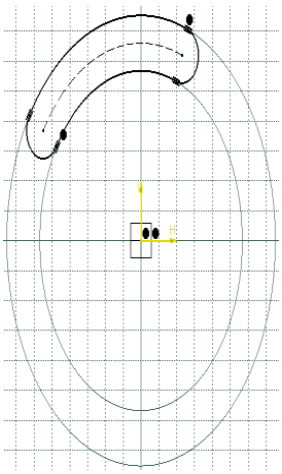
• To draw an elongated hole invoke the **Elongated Hole** tool from the **Predefined Profile** toolbar.



An elongated hole profile

• Drawing Cylindrical Elongated Holes

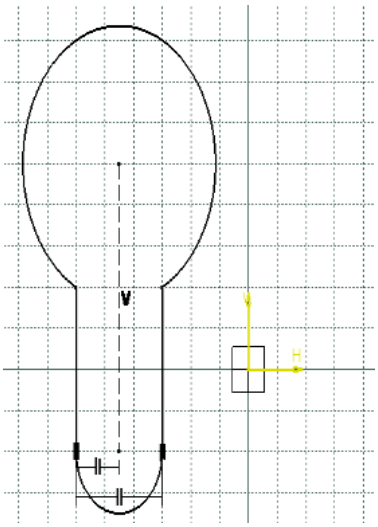
• To draw a cylindrical elongated hole, invoke the **Cylindrical Elongated Hole** tool from the **Predefined Profile** toolbar..



A cylindrical elongated hole

• Drawing Keyhole Profiles

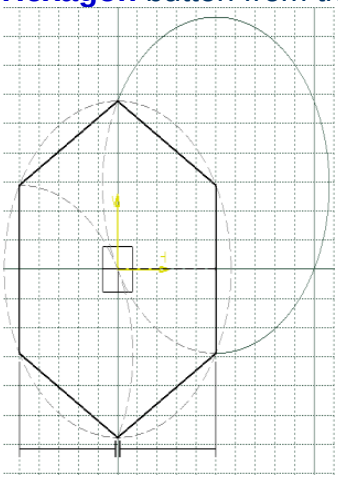
• To draw a keyhole profile, invoke the **Keyhole Profile** tool from the **Predefined Profile** toolbar.



A keyhole profile

- **Drawing Hexagons**

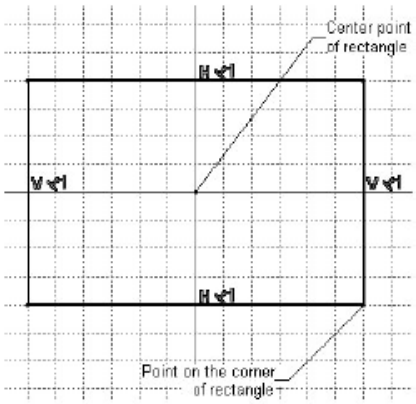
- To draw a hexagon choose the **Hexagon** button from the **Predefined Profile** toolbar.



Hexagon drawn using the **Hexagon** tool

• Drawing Centered Rectangles

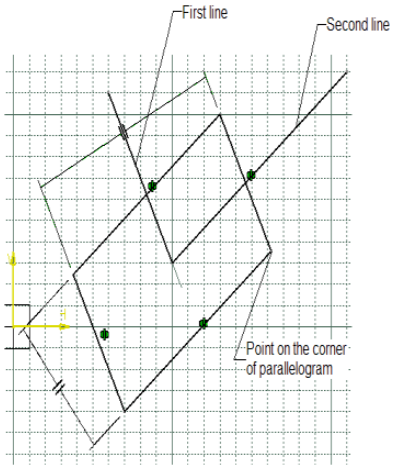
• To draw a centered rectangle, choose the **Centered Rectangle** button from the **Predefined Profile** toolbar.



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.



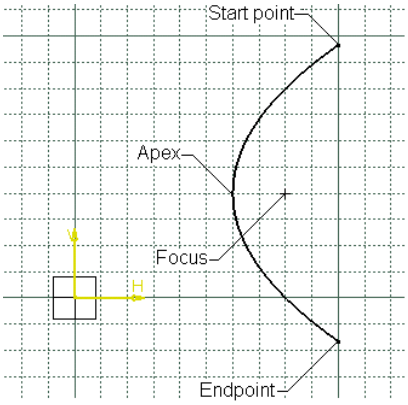
Centered parallelogram with the first line, second line, and the point on parallelogram

• Drawing Conics

• To draw various conics available in CATIA V5R14, choose the down arrow available on the right of the **Ellipse** button in the **Profile** toolbar.

• Drawing a Parabola by Focus

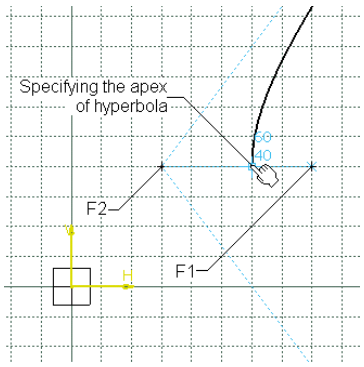
• To draw a parabola by focus, invoke the **Parabola by Focus** tool from the **Conic** toolbar.



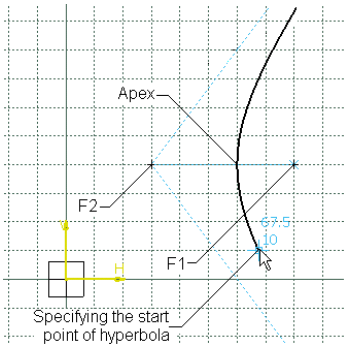
Points used to draw a parabola

• Drawing a Hyperbola by Focus

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



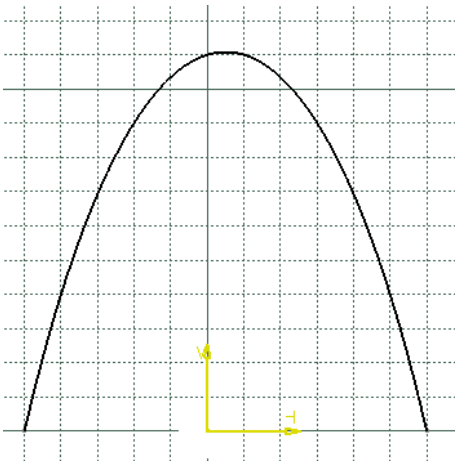
Specifying the apex of the hyperbola



Specifying 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.

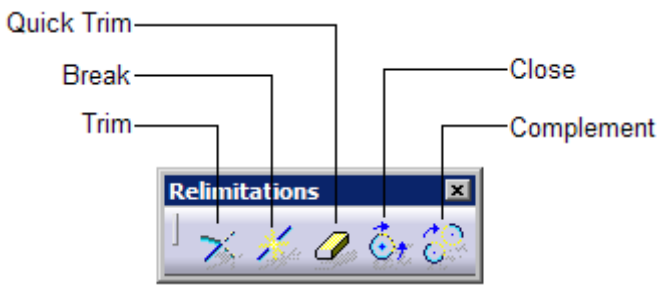
• These include trimming the sketches, using the quick trim, breaking a sketched element, filleting the sketches, adding chamfer to the sketches, and so on.

• Trimming Unwanted Sketched Elements

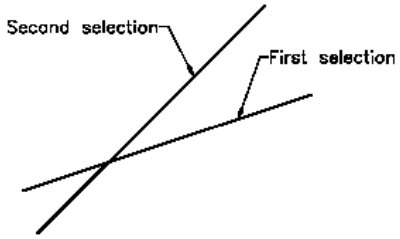


• Invoke the **Relimitations** toolbar by choosing the down arrow provided on the right of the **Trim** button available in the **Operation** toolbar.

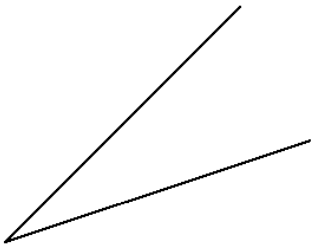
• The **Relimitations** toolbar is shown in the figure.



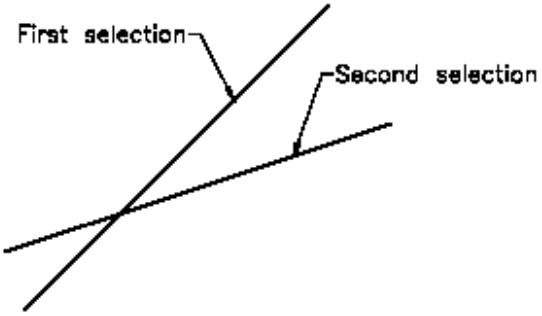
The Relimitations toolbar



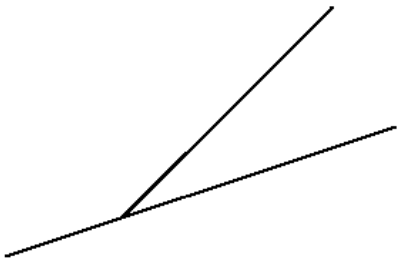
Elements to be selected to trim



The resulting trimmed elements

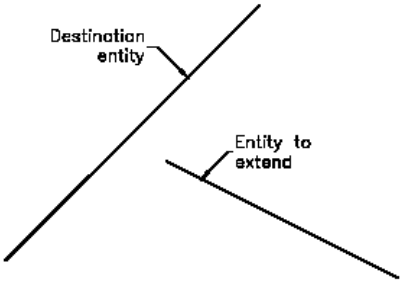


Elements to be selected to trim

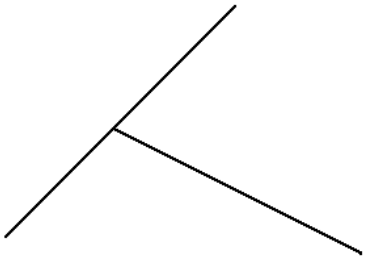


The resulting trimmed elements

Extending Sketched Elements



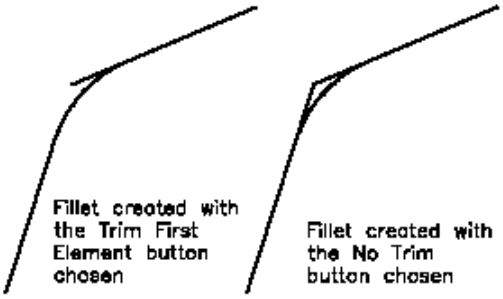
Elements selected to extend



The resulting extended element

• Trimming by Trim Tool

• CATIA V5 also using which you can unwanted sketched

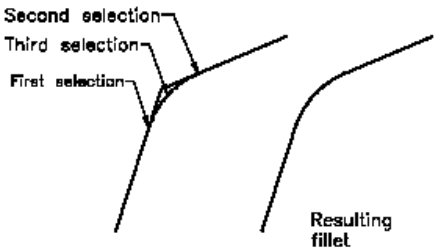


Using the Quick

provides you with a tool quickly trim the elements.

• Filleting Sketched Elements

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

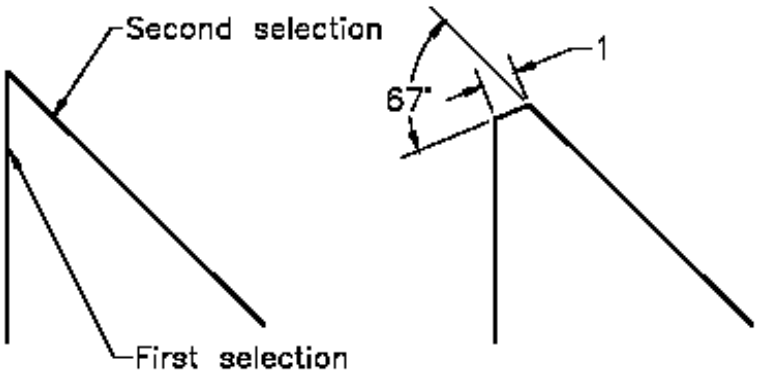


Elements to be selected and the resulting filleted sketched elements

Fillets created using the selected Trim First Element and the NoTrim buttons

• Chamfering Sketched Elements

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

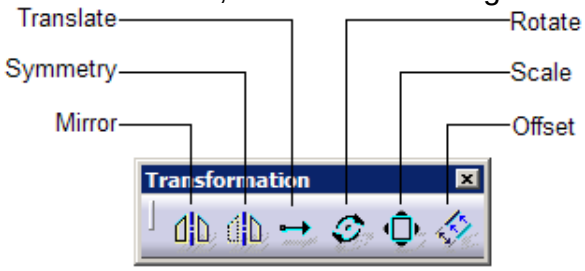


Elements to be selected and the resulting chamfer created

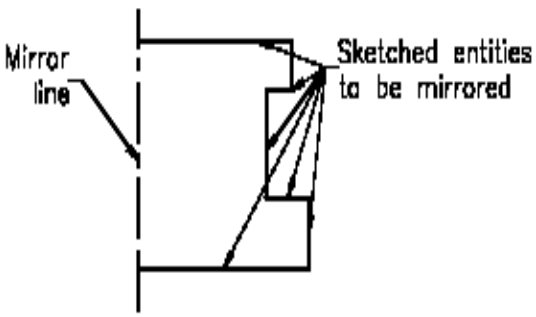
• Mirroring Sketched Elements

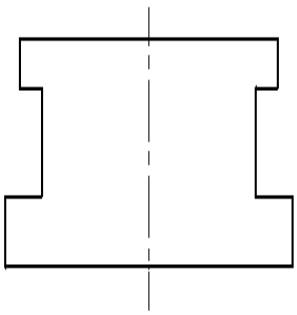
• You can mirror the sketched elements along the mirror line in the **Sketcher** workbench of CATIA V5 using the **Mirror** tool.

• Choose the down arrow on the right of the **Mirror** button provided in the **Operation** toolbar to invoke the **Transform** toolbar, as shown in the figure.



The Transformation toolbar





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

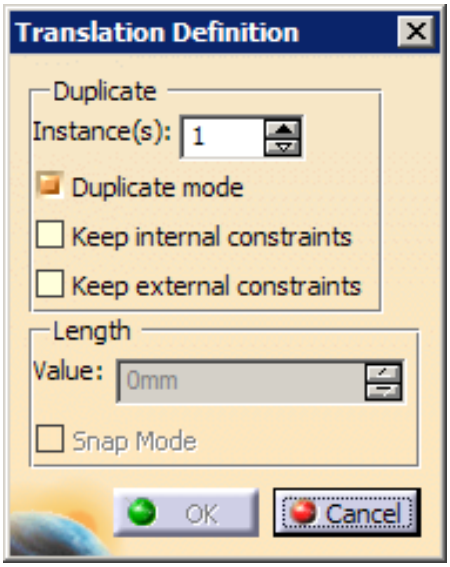
Resulting mirrored sketch

• 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.



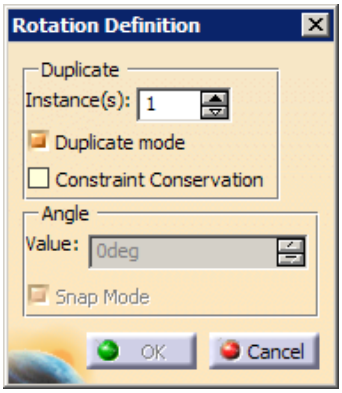
The Translation Definition Dialog box

• Rotating Sketched Elements

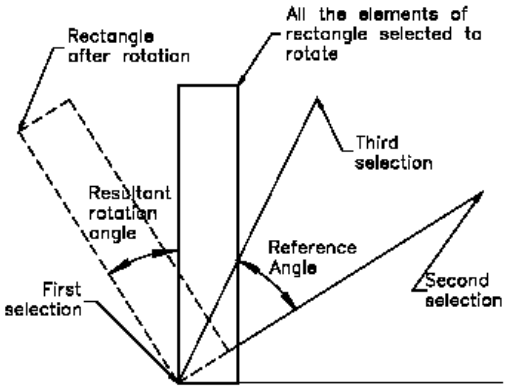
• The **Rotate** tool is used to rotate the sketched elements around a rotation center point.

• 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

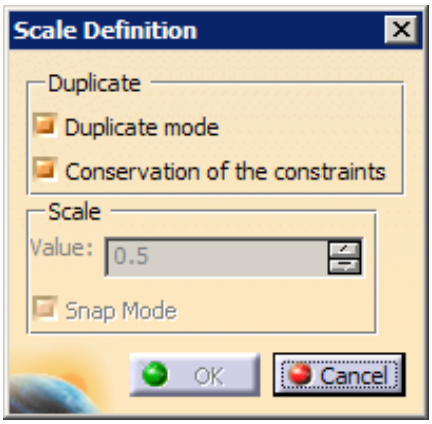


Points to be selected and the rotated elements

• Scaling Sketched Elements

• To scale the sketched elements, select the elements that you need to scale and then choose the **Scale** button from the **Transformation** toolbar.

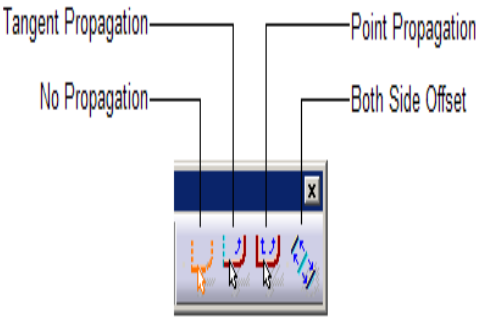
• The **Scale Definition** dialog box is displayed, as shown in the figure.



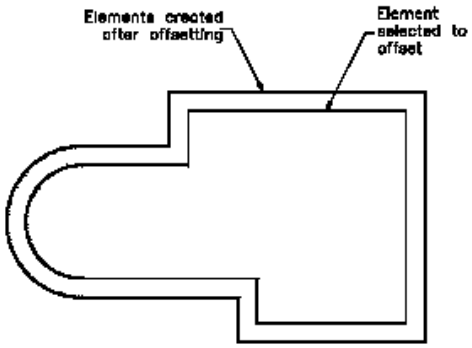
The Scale Definition dialog box

• 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

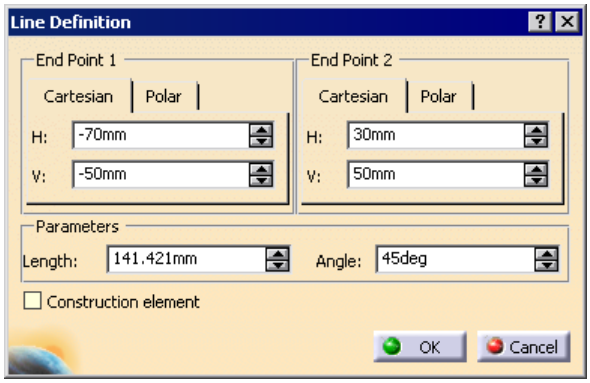
• You can modify the sketched elements, using various tools available in the **Sketcher** environment of CATIA V5.

• 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.



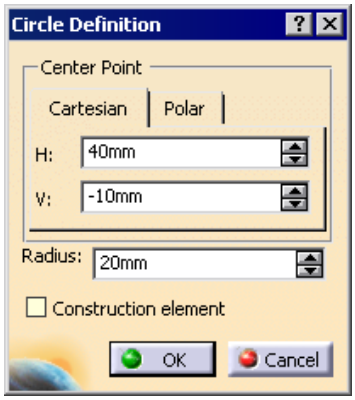
The Line Definition dialog box

• 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

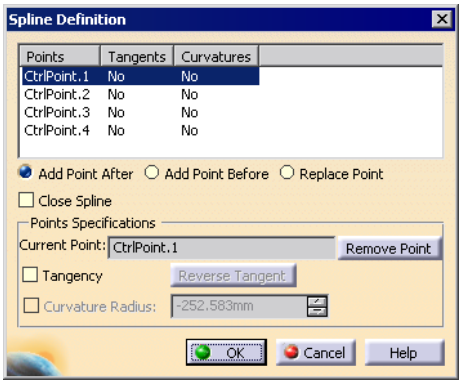
• Modifying the Sketched Arc

• You can modify the sketched arc using the **Circle Definition** dialog box.

• To invoke it, double-click on the arc that you need to modify.

• Modifying the Sketched Spline

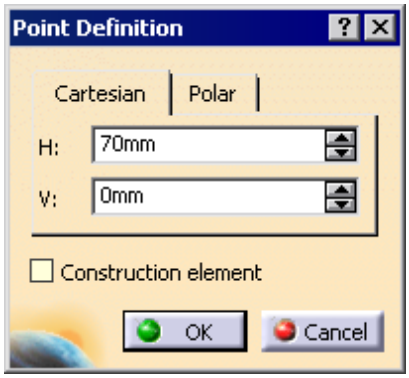
• You can modify a spline using the **Spline Definition** dialog box, which is displayed when you double-click on the spline that you need to modify.



The Spline Definition dialog box

• Modifying the Sketched Point

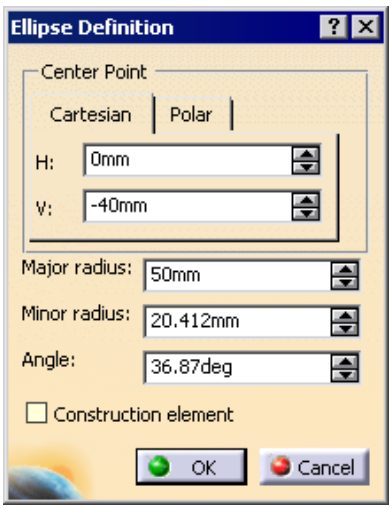
• To modify a sketched point, double click on it; the **Point Definition** dialog box is displayed as shown in the figure.



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.



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 from the keyboard.

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

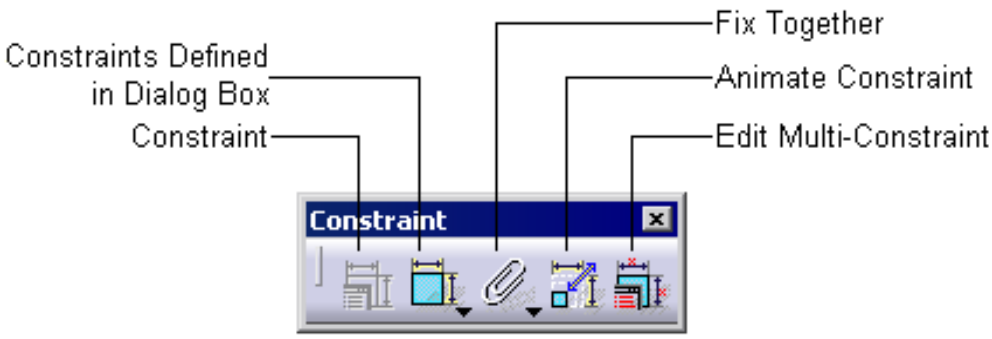
Chapter 3

➢ 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 available 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:

1. 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.

2. Under-Constrained

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

3. Over-Constrained

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

4. 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.

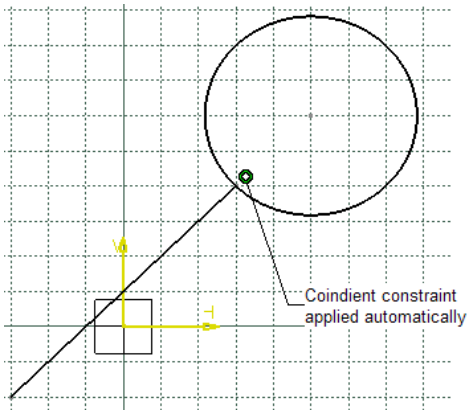
5. Not Changed

• 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

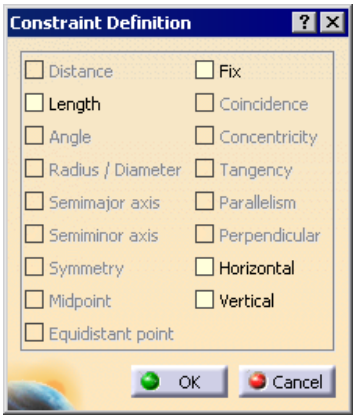
This tool is active by default and therefore,as you draw an element, some constraints are automatically applied to it.



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.



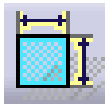
The Constraint Definition dialog box

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**

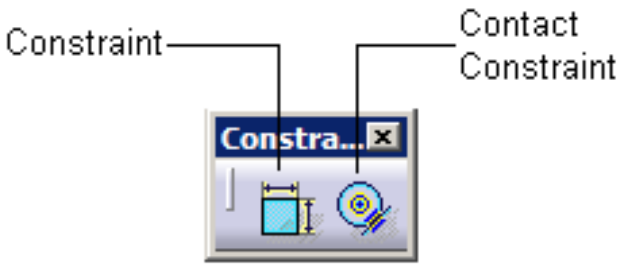
- **Perpendicular**
- **Horizontal**
- **Vertical**

• Applying Dimensional Constraints



• After applying the geometric constraints, you need to apply the dimensional constraints to fully define the sketches, by using the **Constraints Definition** dialog box.

• Choose the **Constraint** button from the **Constraint** toolbar or the **Constraint Creation** toolbar shown in the figure.

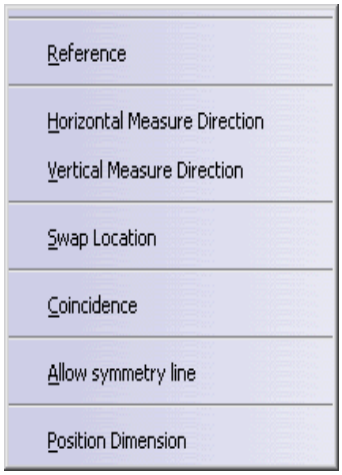


The Constraint Creation toolbar

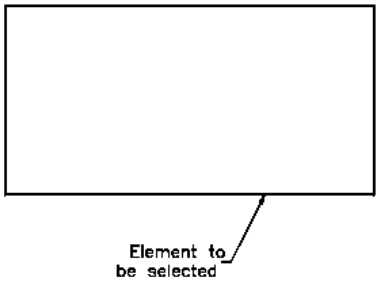
Linear Dimensioning of a Line and Between Two Points

• 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.



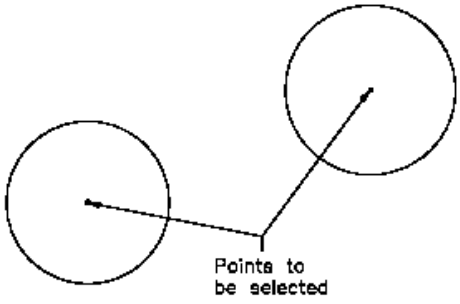
The Contextual menu

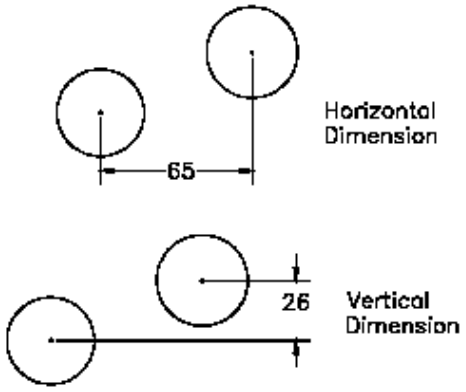


Element selected to be dimensioned



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.

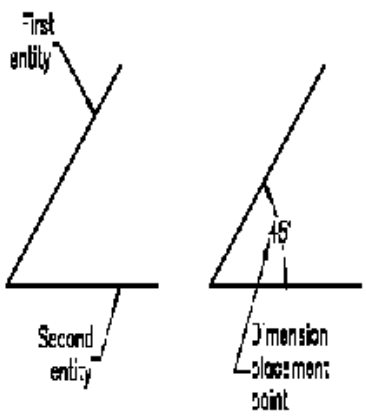
• You can also apply a horizontal or vertical dimension to it.

- **Dimensioning An Arc and A Circle**

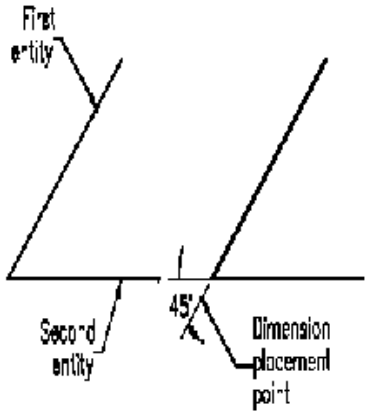
• By default, the diameter dimension is applied to circles, and radius dimension is applied to arcs.

• Invoke the **Constraint** tool and select the arc or the circle that you need to dimension.

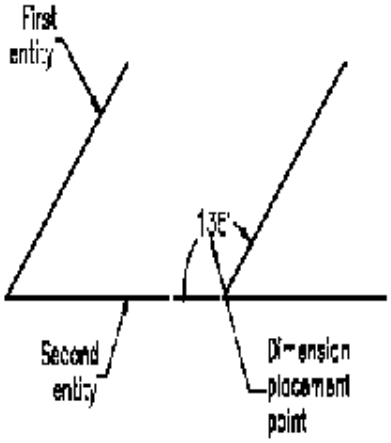
Applying an Angular Dimension



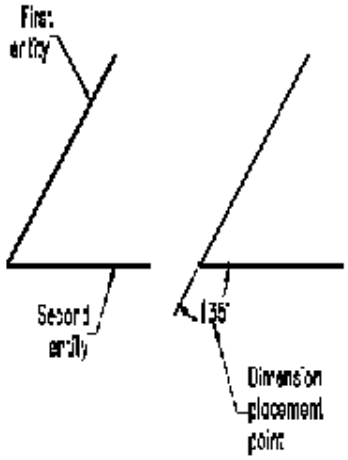
Angular dimension placed according to the placement point



Angular dimension placed according to the placement point

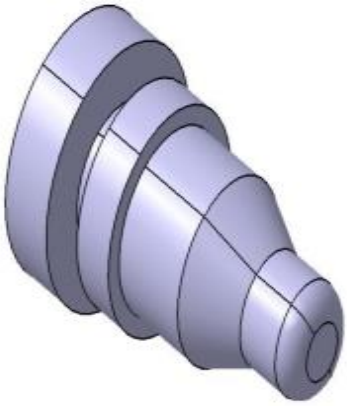


Angular dimension placed according to the placement point

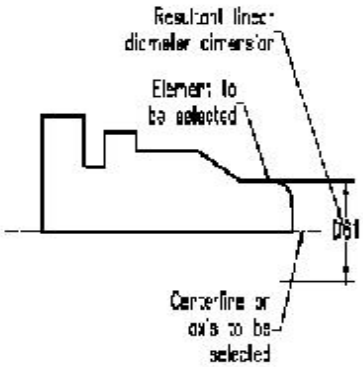


Angular dimension placed according to the placement point

- **Applying Linear Diameter Dimensions**



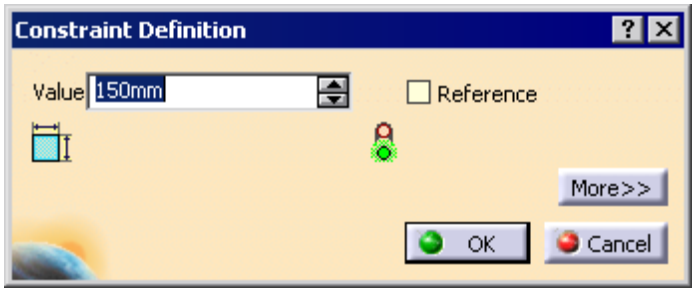
Model created by revolving a sketch around the horizontal center line



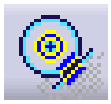
Element and center line selected and the resulting linear diameter dimension

- **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.



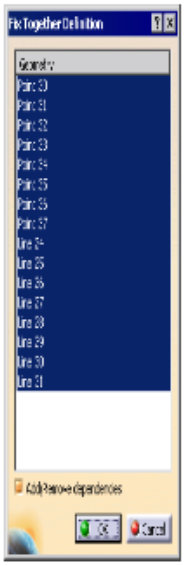
The ConstraintDefinition dialog box invoked after doubleclicking the dimension

• Applying Contact Constraints 

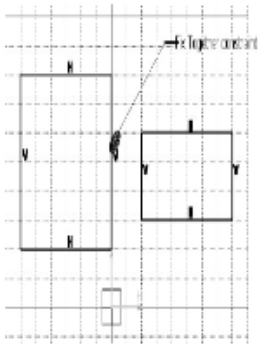
• The **Contact Constraint** tool is used to automatically apply geometrical constraints to the selected elements, depending on their position and geometry.

• 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.

• Applying Fix Together Constraints



The Fix Together Definition dialog box

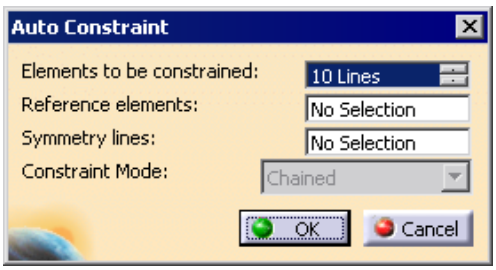


The Fix Together constraint applied to the sketched entities

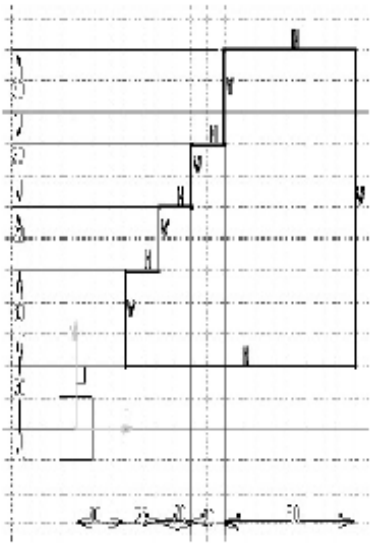
• Applying Automatic Constraints 

• To apply constraints, select the elements to be constrained, and choose the **Auto Constraint** button from the **Constraint** toolbar; the **Auto Constraint** dialog box is displayed.

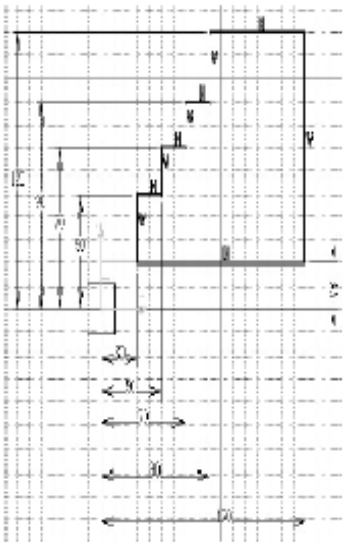
• The number of elements selected is displayed in the **Elements to be constrained** display box, as shown in the figure.



The Auto Constraint dialog box



Automatic dimensions applied using the Chained option

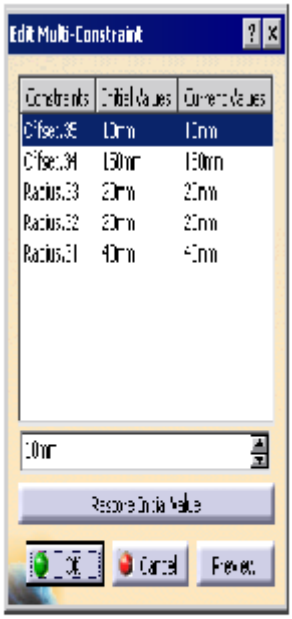


Automatic dimensions applied using the Stacked option

• 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

• If, after applying dimensions to the sketches, the sketch is OverConstrained,you need to delete the over defining constraints,
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 the Figure A.

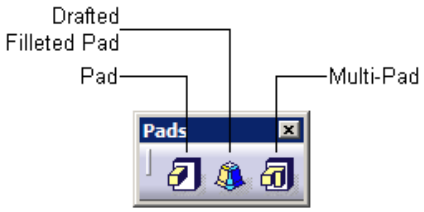


Figure A The Pads toolbar

• Choose the **Pad** button from the **Pads** toolbar; the **Pad Definition** dialog box is displayed, as shown in Figure B.

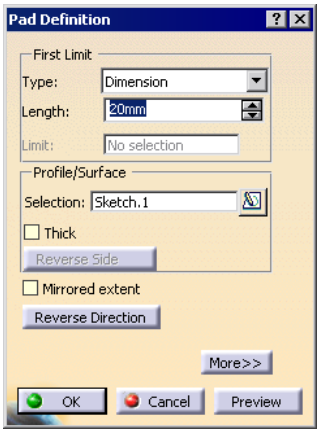
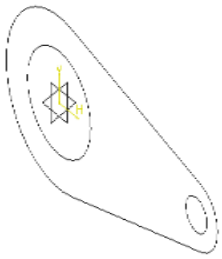
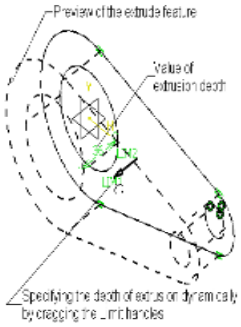


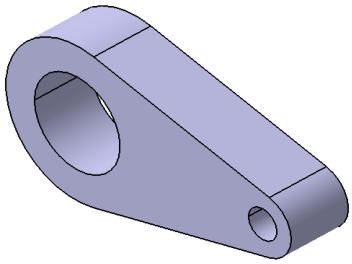
Figure B The Pad Definition dialog box



The sketch, after exiting the Sketcher workbench



Dynamically dragging the Limit handle to specify the depth of extrusion

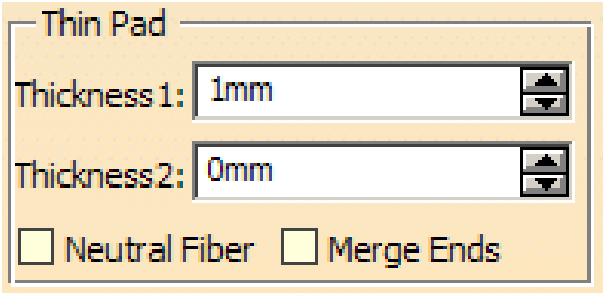


The model, after creating the base Feature

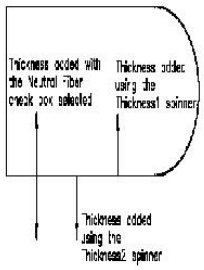
• Creating a Thin Extruded Feature

• To create a thin extruded feature, choose the **Thick** check box from the **Pad Definition** dialog box.

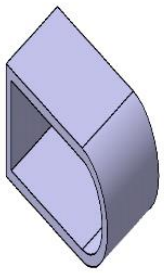
• The dialog box expands and the options in the **Thin Pad** area of the **Pad Definition** dialog box are invoked, as shown in the figure.



The Thin Pad area ofthe Pad Definition dialog box



Thickness added using different options



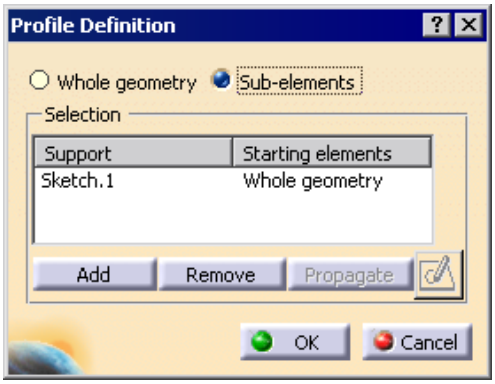
An extruded feature created using the Thin option

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

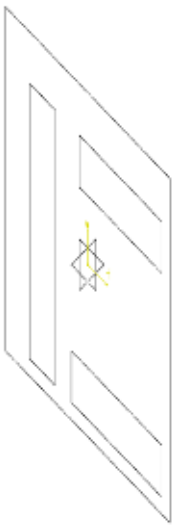
• To create a feature by selecting a particular contour from a multi-contour sketch, draw the sketch and invoke the **Pad Definition** dialog box.

• Right-click on the **Selection** selection area and choose the **Go** to profile definition option.

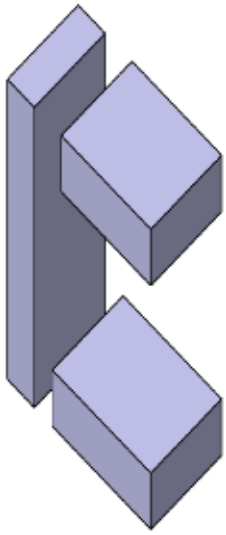
• The **Profile Definition** dialog box is displayed, as shown in the figure.



The Profile Definition dialog box

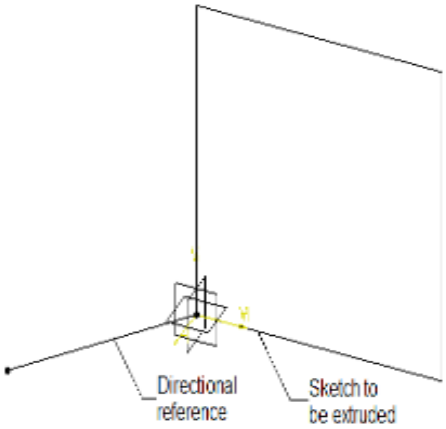


A multi-contoured sketch

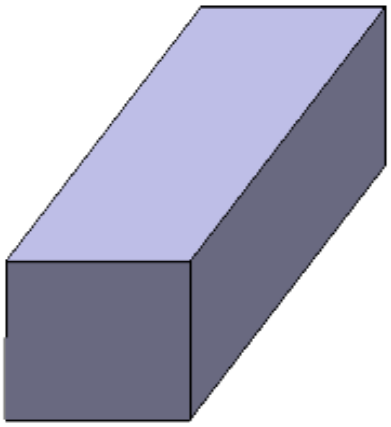


Various contours extruded

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



Sketch to be extruded and the directional reference

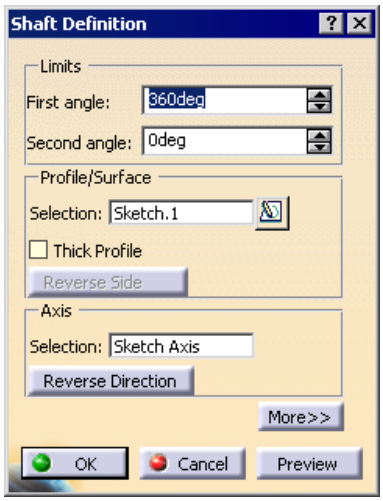


The extrude feature created by extruding a sketch along the directional reference

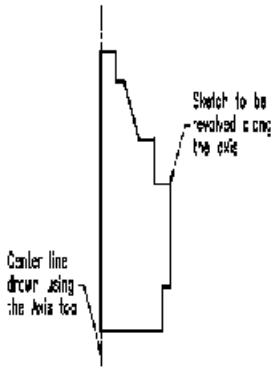
➢ CREATING BASE FEATURES BY REVOLVING SKETCHES

• 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.

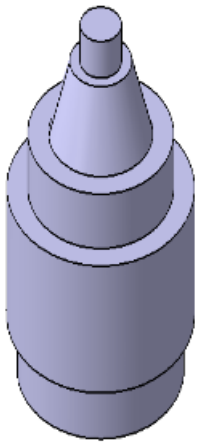
• Next, exit the **Sketcher** workbench and choose the **Shaft** button from the **Sketch-Based Features** toolbar so that the **Shaft Definition** dialog box is displayed.



The **Shaft Definition** dialog box



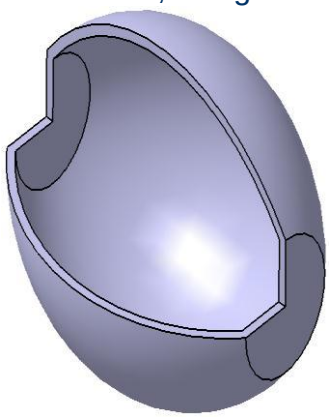
The sketch to be revolved and the axis



The model, after creating the shaft feature

• Creating Thin Shaft Features

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



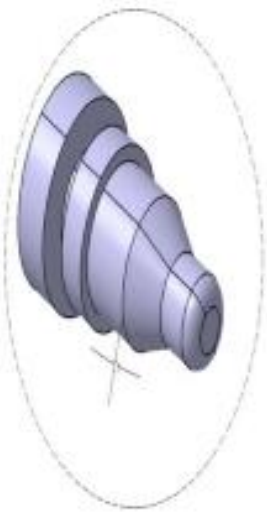
The model, after creating the thin shaft feature using an open profile

➢ 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.

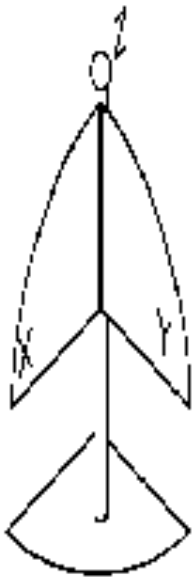
• This tool can be invoked even when you are inside some other tool.

• Rotating the View Using the Rotate Tool



View of the model being rotated using the Rotate tool

• Rotating the View Using the Compass

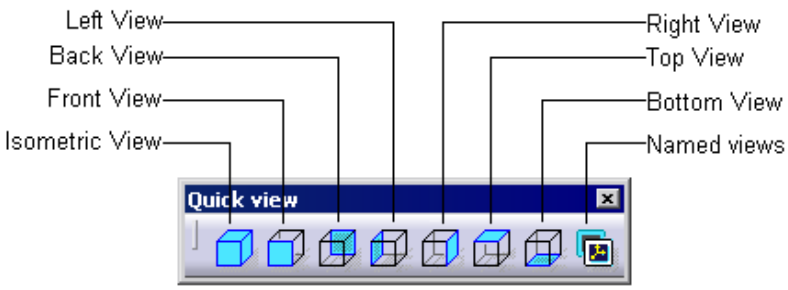


Compass

➢ MODIFYING THE VIEW ORIENTATION

• To invoke the **Quick View** 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.

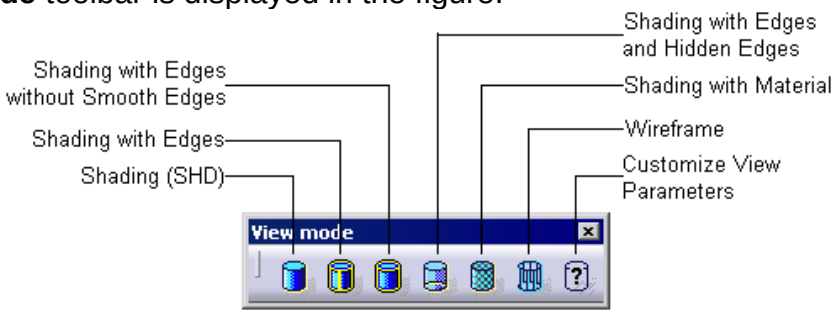


The Named Views dialog box

➢ 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 following are the tools available in the **View mode** toolbar:

• **Shading (SHD)**

• When you choose the **Shading (SHD)** button from the **View** mode toolbar, the model is 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 is displayed in shading with edges but the display of the smooth edges is removed.

• Shading with Edges and Hidden Edges

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

• Shading with Material

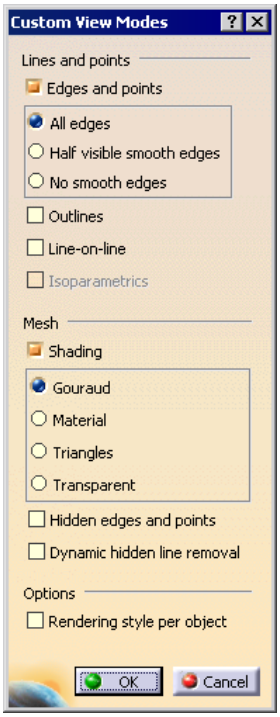
• When you choose the **Shading with Material** button, the model is displayed in the rendered mode.

•Wireframe

• When you choose the **Wireframe** 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 is 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



Model after assigning Aluminium material

Chapter 4

Learning Objectives:

- *Understand the importance of sketching planes.*

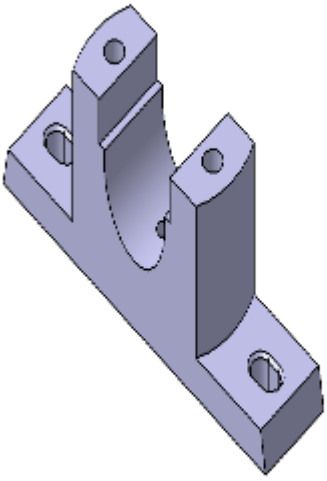
- *Create reference elements.*

- *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*

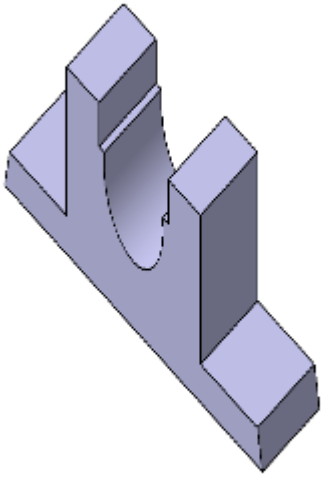
➢ IMPORTANCE OF SKETCHING PLANES

Most mechanical designs consist of multiple features such as sketched features, referenced elements, and placed features that are integrated to complete a model.

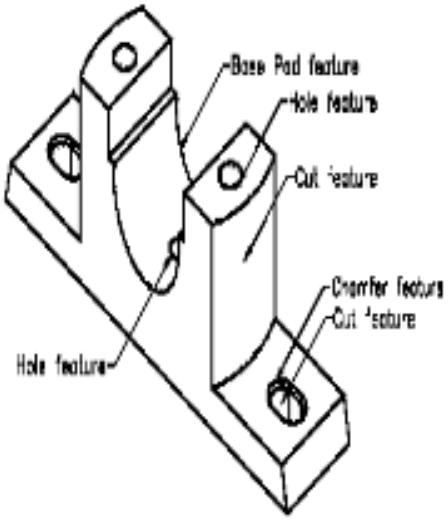
• You can select any one of the default planes as the sketching plane to create the sketch of the base feature.



A multifeatured model



Base feature of the model

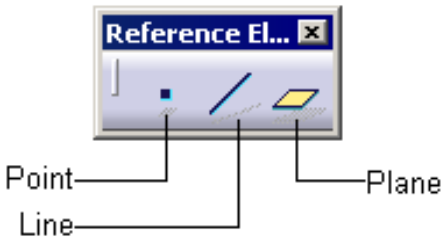


Other features of the final model

➢ 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 that will be used as the sketching plane for other features.

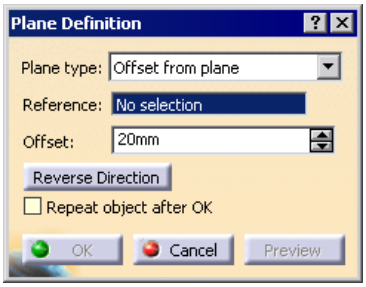
• Default Planes

• 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

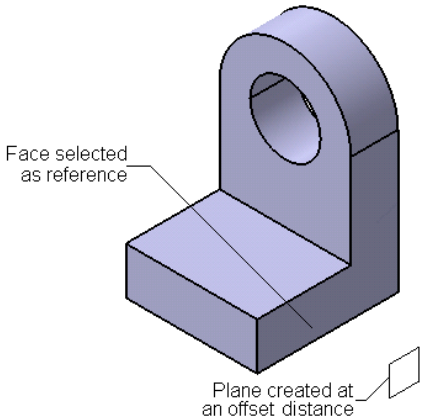
• To create a new plane, choose the **Plane** button from the **Reference Elements (Expanded)** toolbar.

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



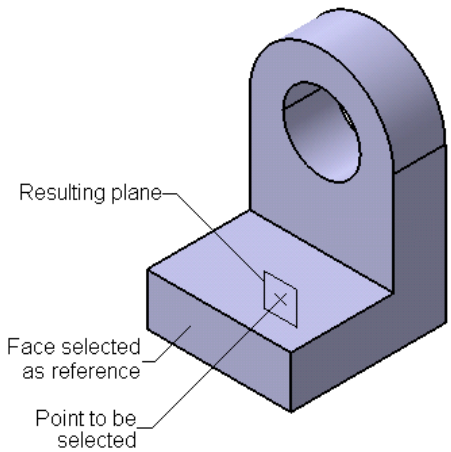
The Plane Definition dialog box

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



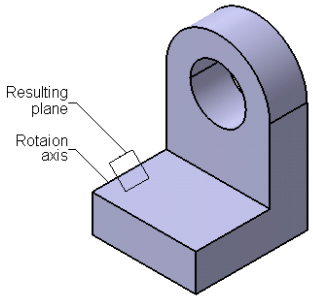
- **Creating a Plane Parallel Through Point**

Reference planar face, sketched point, and the resulting plane Reference planar face and the resulting plane



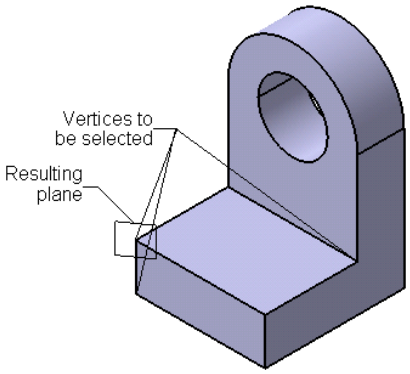
Reference planar face, sketched point, and the resulting plane

- **Creating a Plane at an Angle/Normal to Plane**



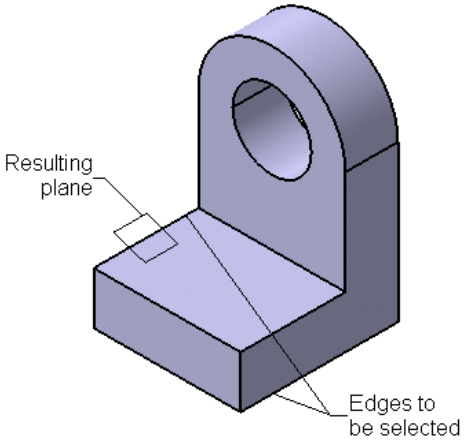
Rotation axis, reference plane, and the resulting plane

- **Creating a Plane Through Three Points**



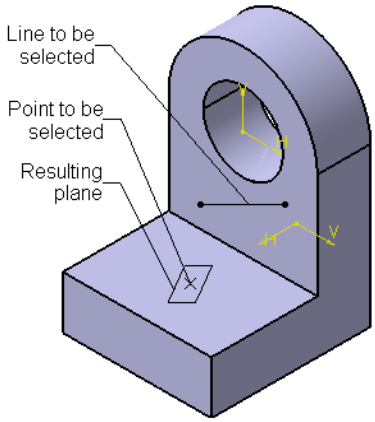
Three vertices to be selected and the resulting plane

Creating a Plane Through two Lines



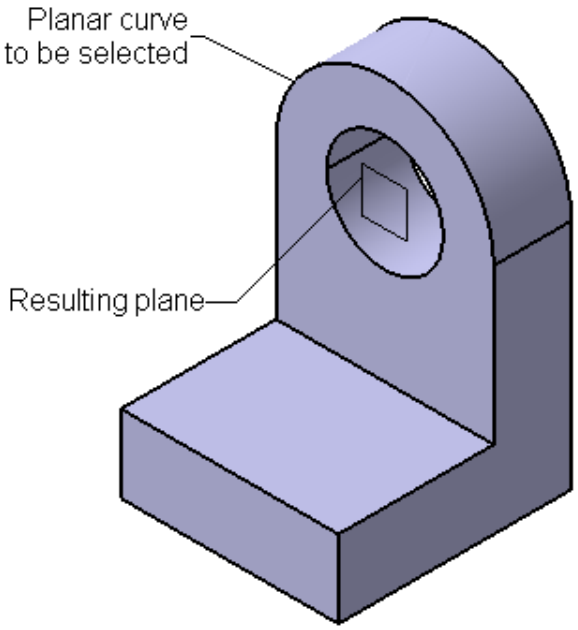
Vertices to be selected and the resulting plane

- **Creating a Plane Through Point and Line**



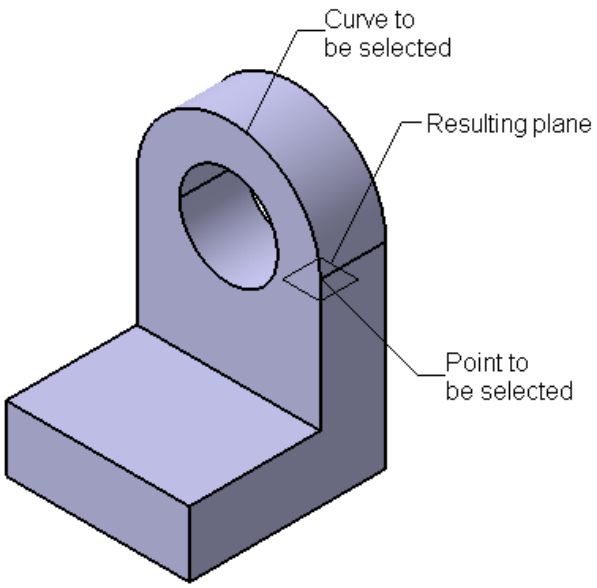
The point and line to be selected, and the resulting plane

- **Creating a Plane Through Planar Curve**



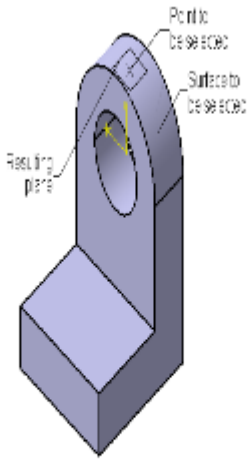
The planar curve to be selected and the resulting plane

- **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.

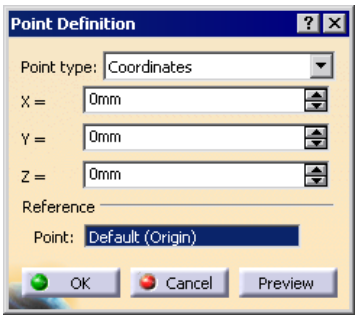
- **Creating a Plane Using a Mean Through Points**

• 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

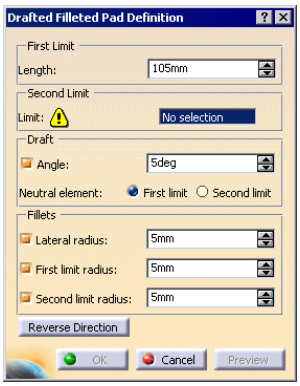
• To create reference lines, choose the **Line** button from the **Reference Elements** toolbar; the **Line Definition** dialog box is displayed.

➢ OTHER SKETCH-BASED FEATURES

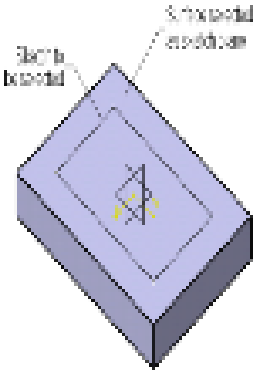
• **Creating Drafted Filleted Pad Features**

• Choose the **Drafted Filleted Pad** button from the **Pads** toolbar.

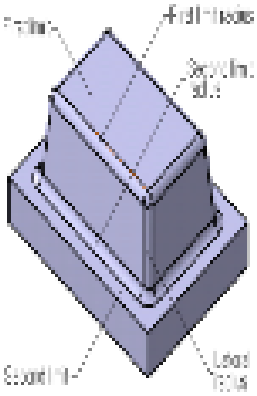
• The **Drafted Filleted Pad Definition** dialog box is displayed, as shown in the figure.



The Drafted Filleted Pad Definition dialog Box



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

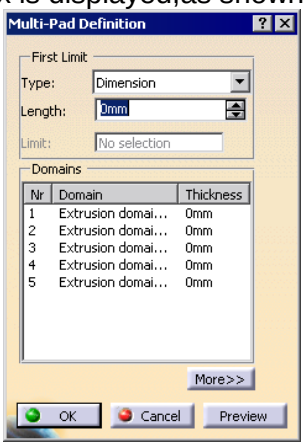


The resulting drafted filleted pad

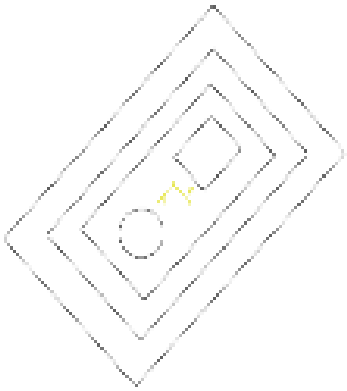
• Creating Multi-Pad Features

• To create a multi-pad feature, choose the **Multi-Pad** button from the **Pads** toolbar and then select the sketch to be extruded.

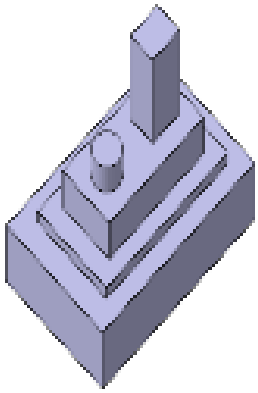
• The **Multi-Pad Definition** dialog box is displayed,as shown in the figure.



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

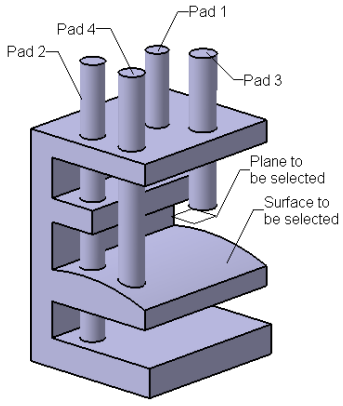
• The **Up to plane** option is used to extrude the sketch from the sketch plane up to the selected plane or the planar face.

• 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.

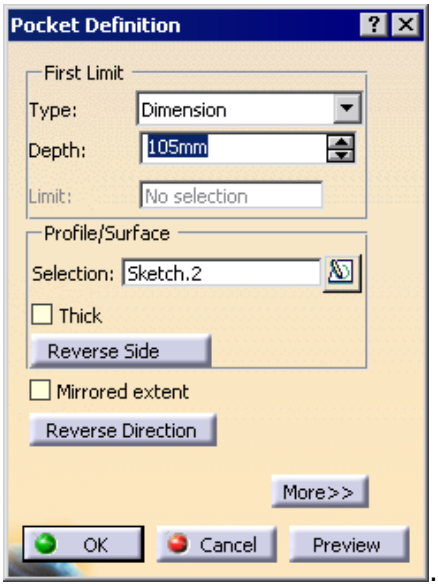


Pad features created using different feature termination options

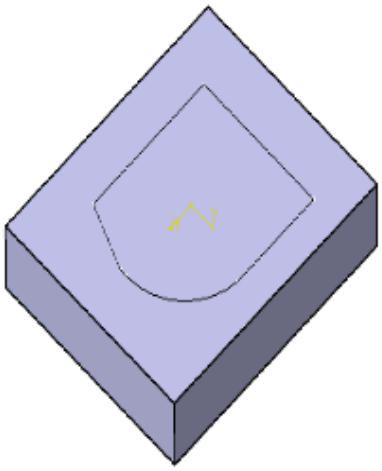
• Creating Pocket Features

• 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.

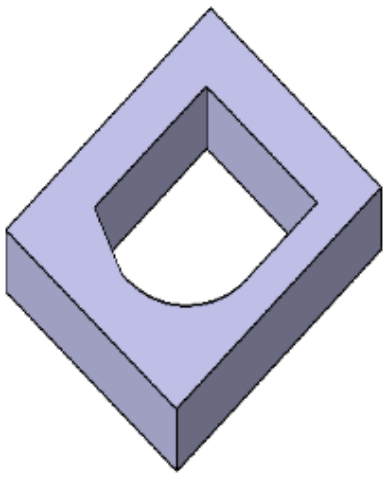
• The **Pocket Definition** dialog box is displayed, as shown in the figure



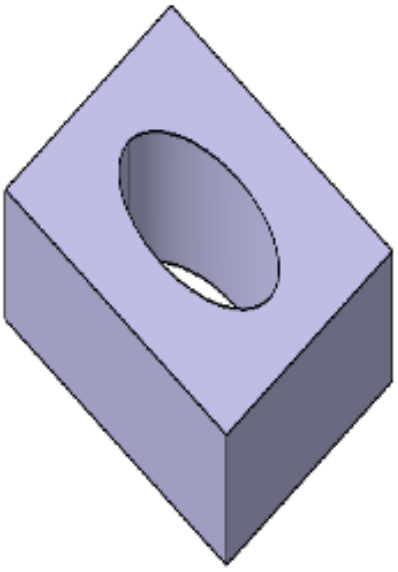
The Pocket Definition dialog box



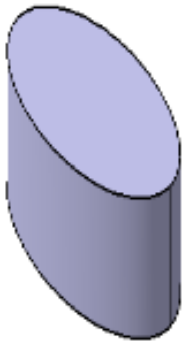
The sketch for the pocket feature



The resulting pocket feature

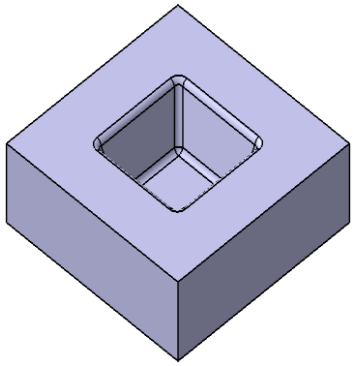


Pocket feature with the default material removal side selected



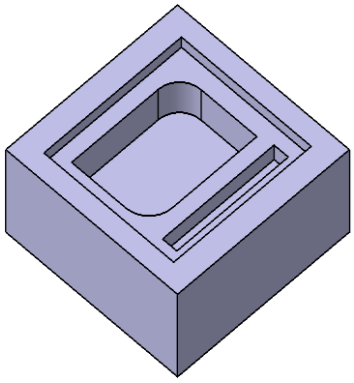
Pocket feature after flipping the material removal direction

- **Creating Drafted Filleted Pocket Features**



The drafted filleted pocket feature

- **Creating Multi-Pocket Features**

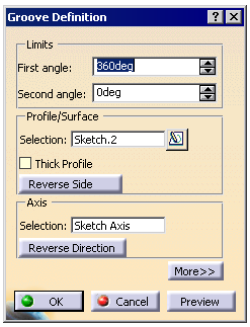


The multi-pocket feature

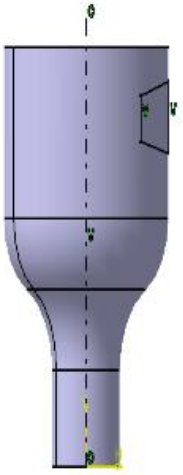
- **Creating Groove Features**

• To create a **Groove** feature, draw the sketch and then choose the **Groove** button from the **Sketch-Based Features** toolbar.

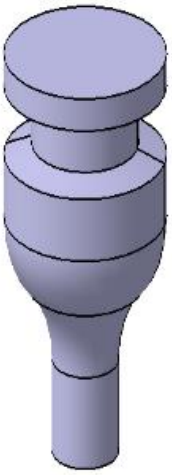
• The **Groove Definition** dialog box is displayed, as shown in the figure.



The Groove Definition dialog box

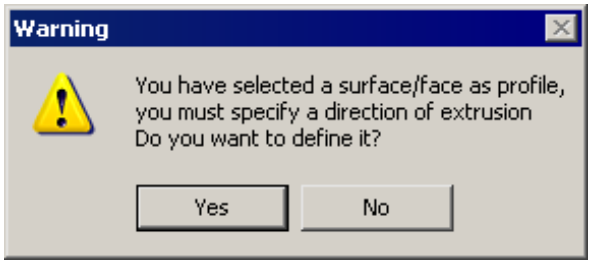


Sketch for creating the groove feature

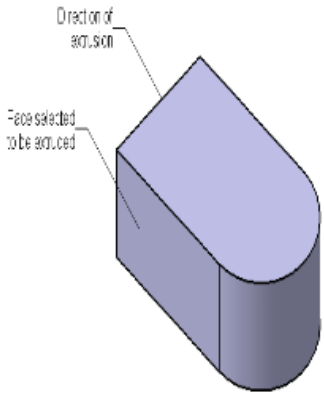


The resulting groove feature

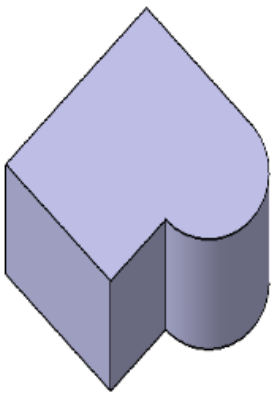
- **Extruding and Revolving Planar and Non planar Faces**



The Warning Message Window



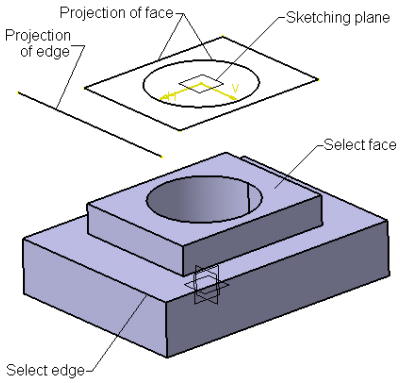
Face selected to extrude and the direction of extrusion



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.

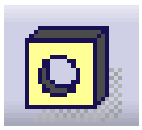


The projected elements

Chapter 5

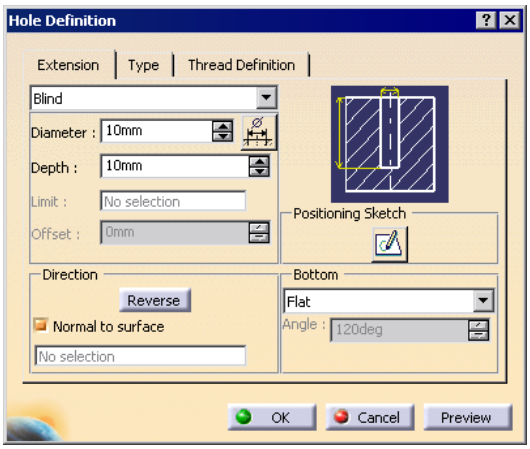
➢ ADVANCED MODELING TOOLS

• Creating Hole Features



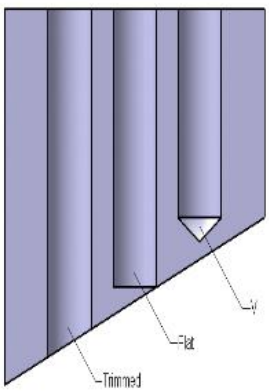
• The preview of the hole feature, with the default values is displayed, along with the **Hole Definition** dialog box.

• The **Hole Definition** dialog box is shown in the figure.

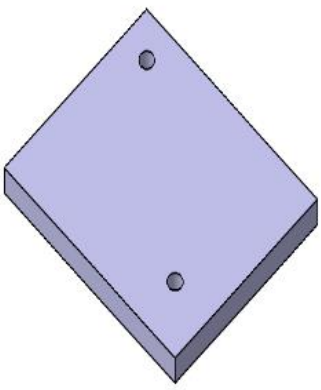


The dialog box

• **Simple Hole**



Types of bottom termination options for a hole feature



Base plate with holes created using the Hole tool

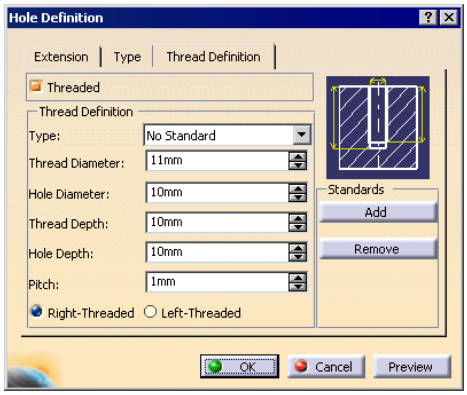
Creating a Hole

• Creating a Threaded Simple 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.

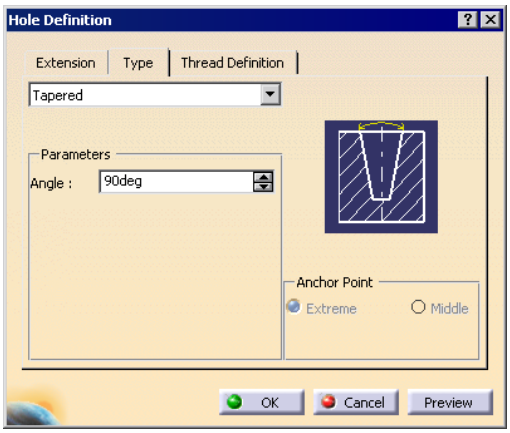
• Select the **Threaded** radio button to invoke the options available in the **Thread Definition** tab.



The Hole Definition dialog box, after selecting the Threaded radio button

• Creating a Tapered Hole

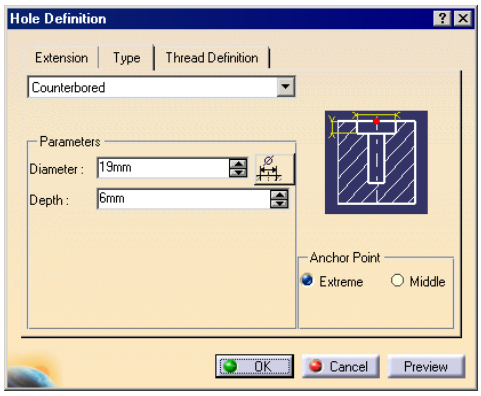
• To create a tapered hole, invoke the **Type** tab of the **Hole Definition** dialog box and select the **Tapered** option from the drop-down list, as shown in the figure.



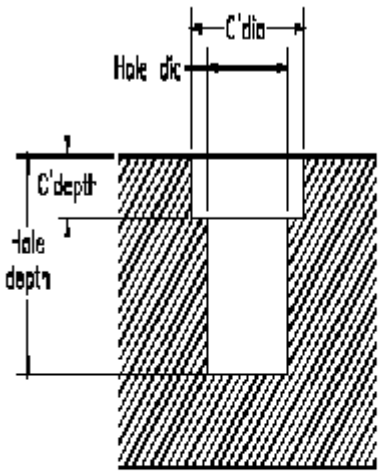
The Hole Definition dialog box after selecting the Tapered option from the drop-down list in the Type tab

• Creating a Counterbored Hole

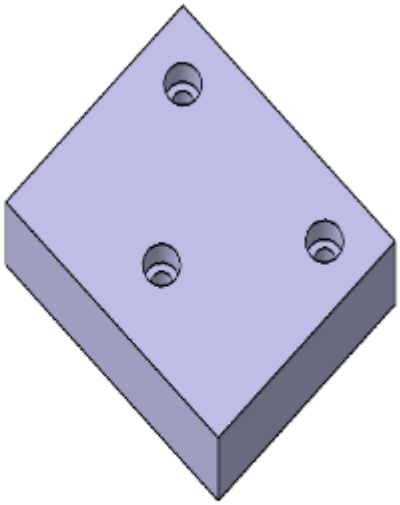
• To create a counterbored hole, select the **Counterbored** option from the drop-down list available in the **Type** tab of the **Hole Definition** dialog box, as shown in the figure.



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



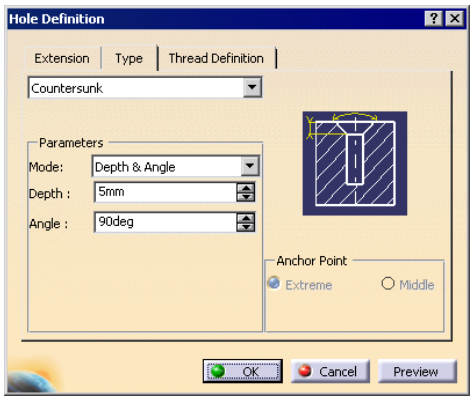
Sectional view of counterbored hole



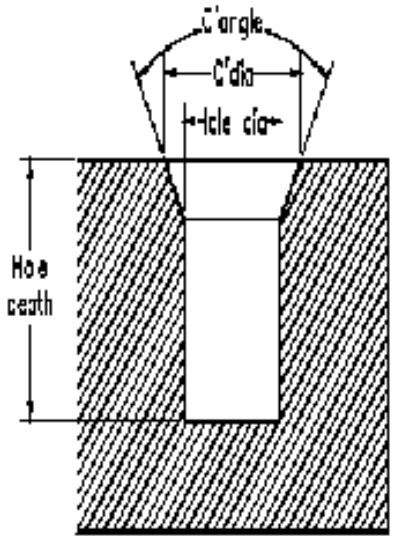
Base plate with counterbored holes

• Creating a Countersunk Hole

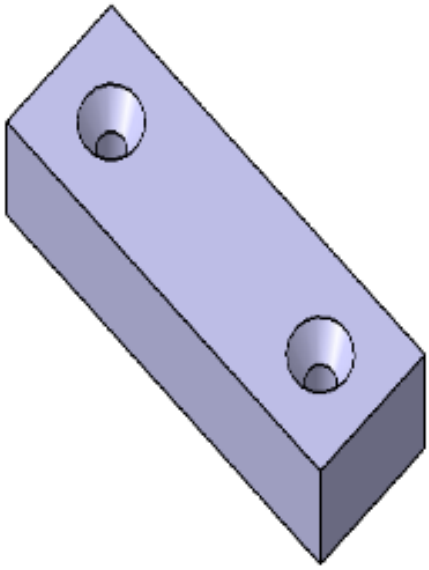
• To create a countersunk hole, select the **Countersunk** option from the drop-down list available in the **Type** tab, as shown in the figure.



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



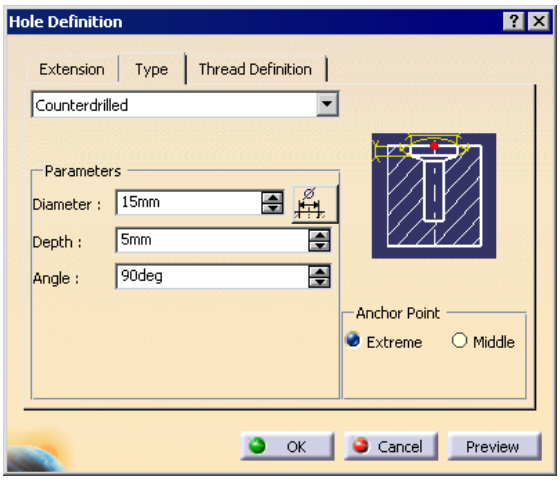
Sectional view of countersunk hole



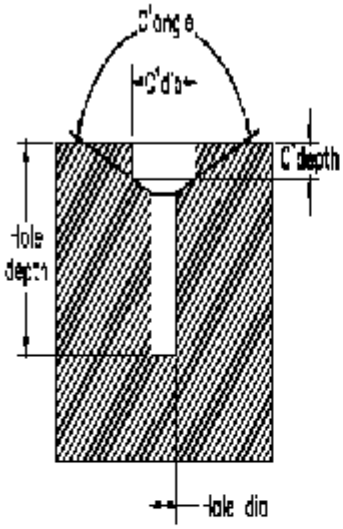
Spacer plate with countersunk holes

• Creating a Counterdrilled Hole

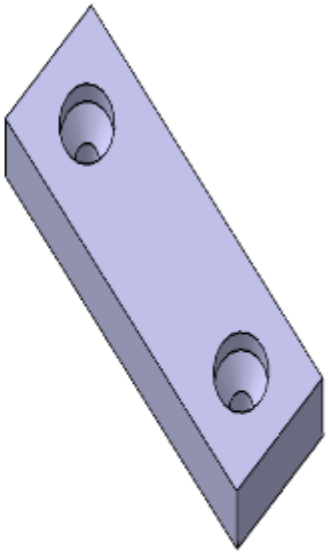
• To create a counterdrilled hole, select the **Counterdrilled** option from the drop-down list available in the **Type** tab; as shown in the figure.



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



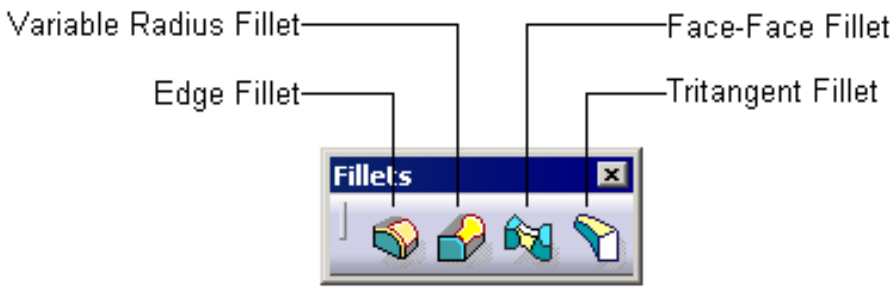
The sectional view of the counterdrilled hole



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.

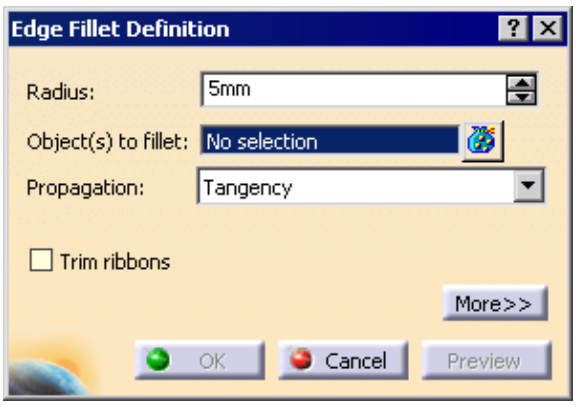


The Fillets toolbar

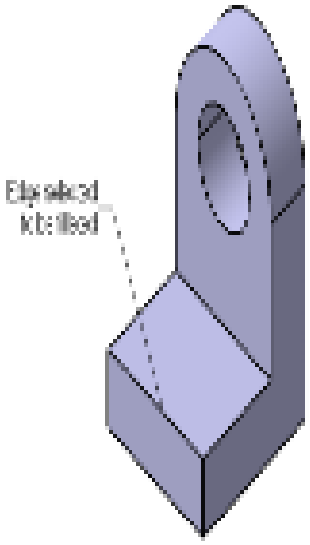
• Creating an Edge Fillet

• To create an edge fillet, choose the **Edge Fillet** button from the **Fillets** toolbar.

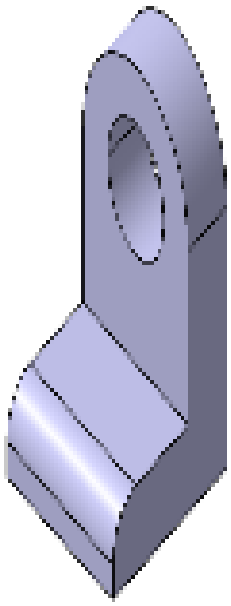
• The **Edge Fillet Definition** dialog box is displayed, as shown in the figure.



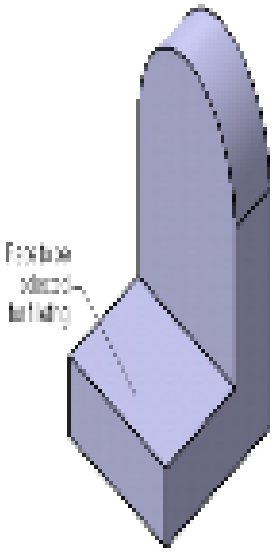
The Edge Fillet Definition dialog box



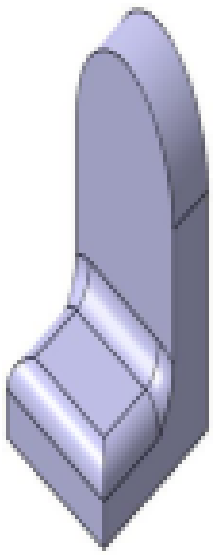
Edge selected to be filleted



Resulting edge fillet



Face to be selected

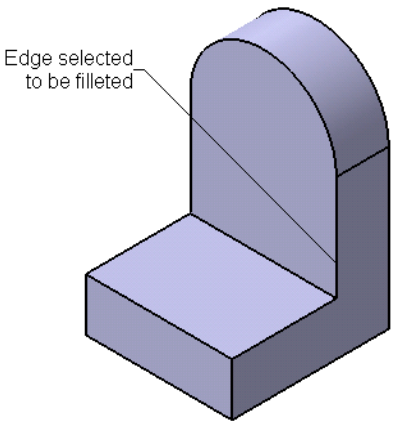


Resulting fillet

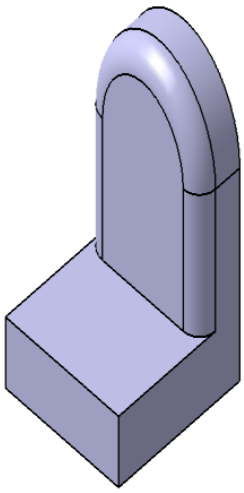
• Managing Selected Entities

• 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

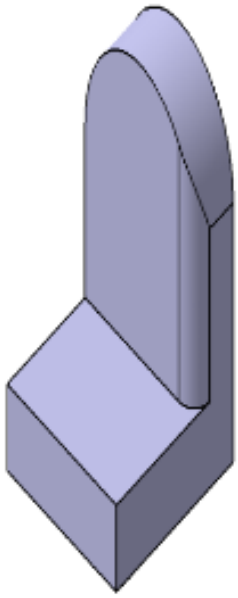


Edge selected to be filleted

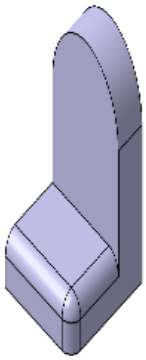


Fillet using the tangent propogation

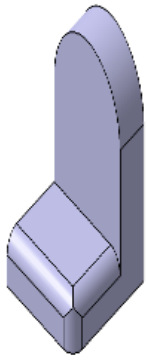
Fillet using the minimal propagation



- Trimming the Overlapping Fillets

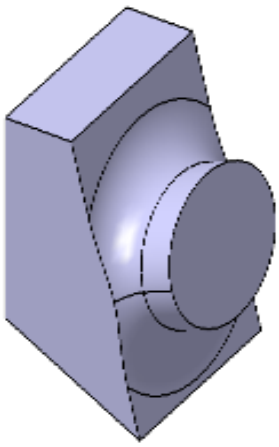


Fillet with the Trim ribbons check box cleared

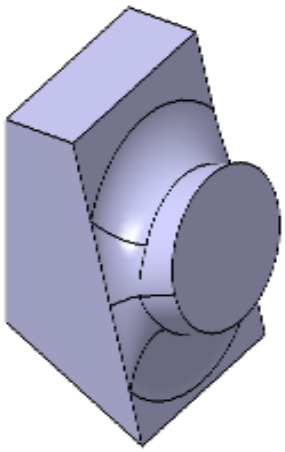


Fillet with the Trim ribbons check box selected

- **Selecting the Edges to Keep**

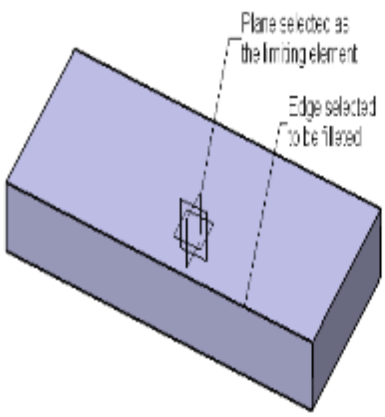


Edges distorted to accommodate the fillet radius

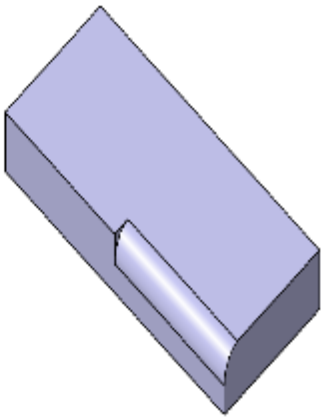


The model, after selecting the edges to keep

- **Setting the Limits of the Fillet**

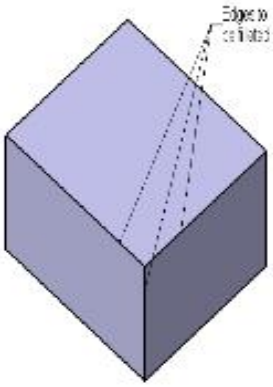


Edge to be filleted and the limiting element

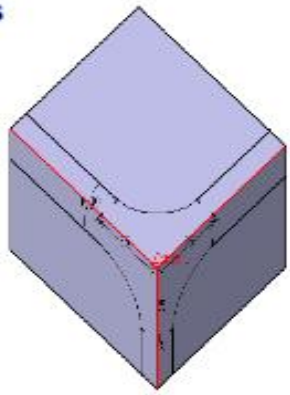


Resulting fillet

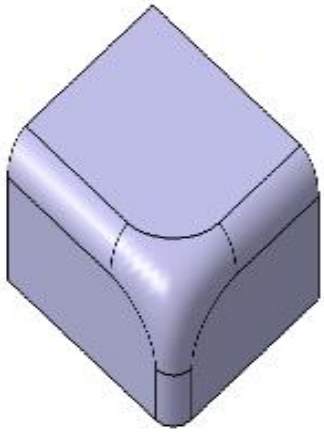
- **Setback Fillet by Blending the Corners**



Edges to be selected

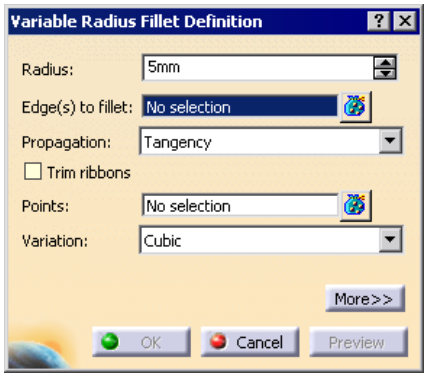


Preview of the setback fillet

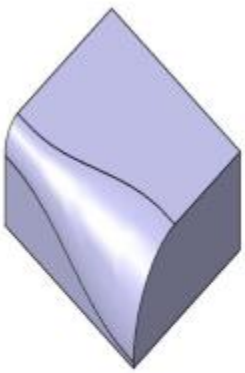


• Creating Variable Radius Fillets

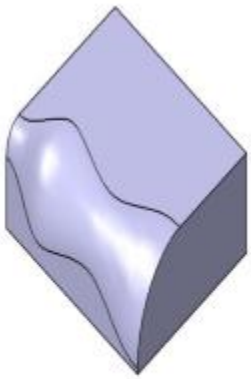
• To create variable radius fillet, choose the **Variable Radius Fillet** button from the **Fillets** toolbar; the **Variable Radius Fillet Definition** dialog box is displayed, as shown in the figure.



The Variable Radius Fillet Definition dialog box



Variable radius fillet created by specifying radii to both the end points of the edge



Variable radius fillet after defining additional control points



Variable radius fillet with the Cubic option selected

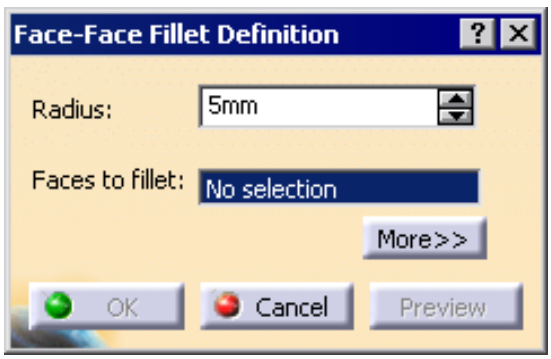


Variable radius fillet with the Linear option selected

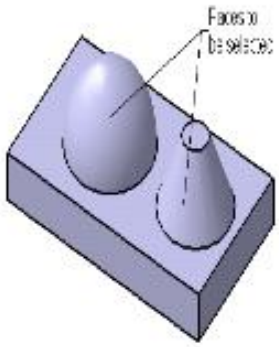
• Creating Face-Face Fillets

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

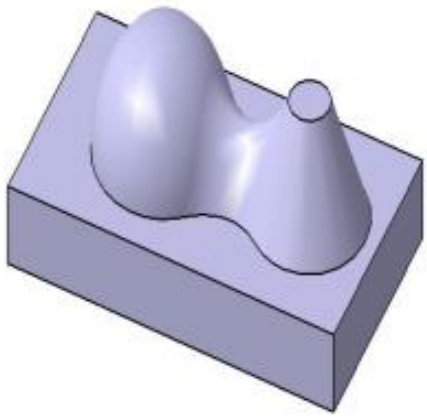
• The **Face-Face Fillet Definition** dialog box is displayed, as shown in the figure.



The Face-Face Fillet Definition dialog box



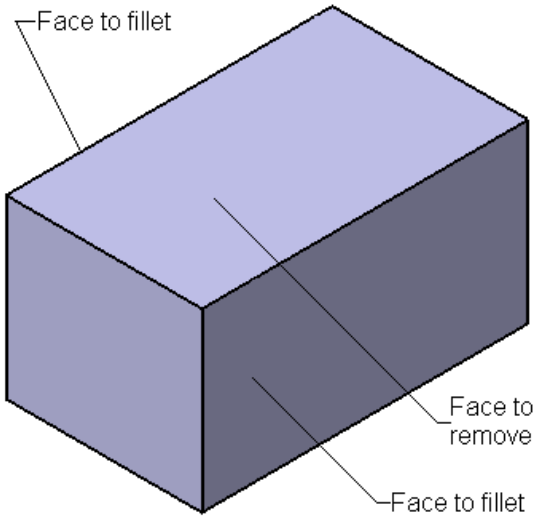
Faces to be selected



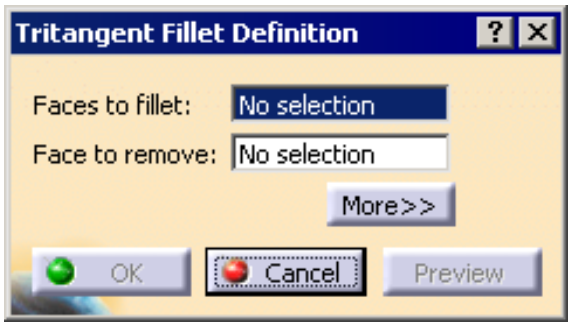
Resulting face-face fillet

• Creating Tritangent Fillets

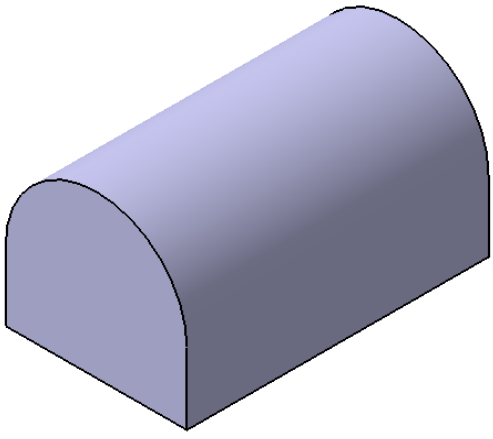
• To create a tritangent fillet, choose the **Tritangent Fillet** button from the **Fillets** toolbar, the **Tritangent Fillet Definition** dialog box is displayed, as shown in the figure.



Faces to be selected



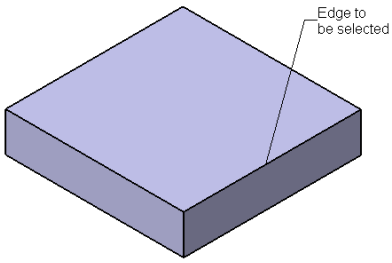
The Tritangent Fillet Definition dialog box



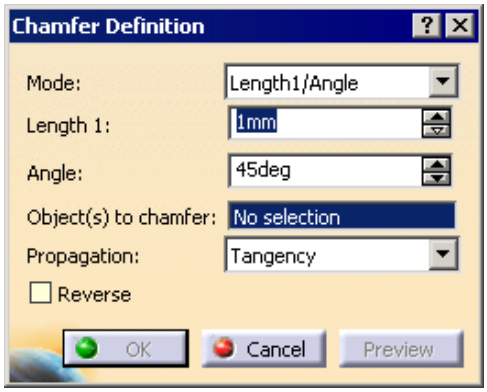
Resulting tritangent fillet

• Creating Chamfers

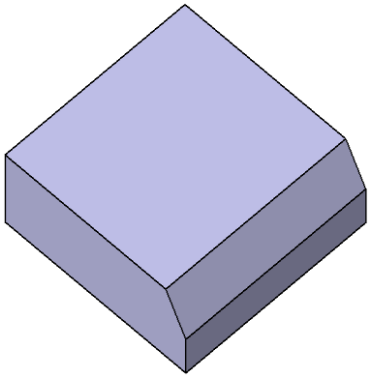
• To chamfer the edges of the model, choose the **Chamfer** button from the **Dress-Up Features** toolbar; the **Chamfer Definition** dialog box is displayed, as shown the figure.



Edge to be selected



The Chamfer Definition dialog box



Resulting chamfer

• Adding a Draft to the Faces of the Model

• 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 casted, molded, or formed.

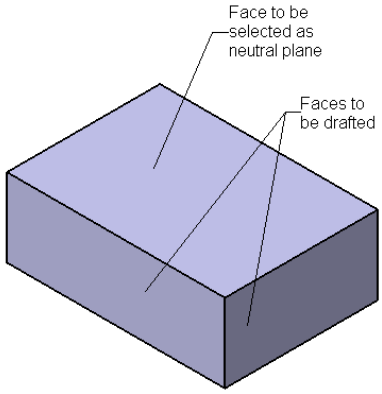
• The **Part** workbench of CATIA V5 provides you with various tools to draft faces of the model.

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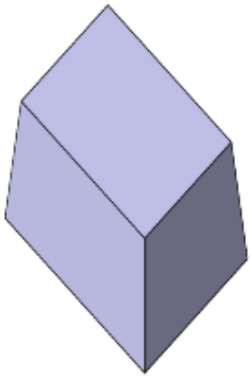
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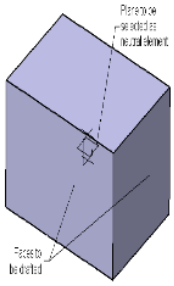
Faces to be selected



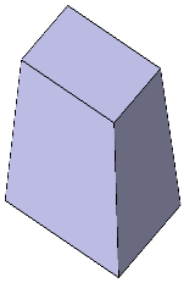
The Draft Definition dialog box



Resulting drafted faces

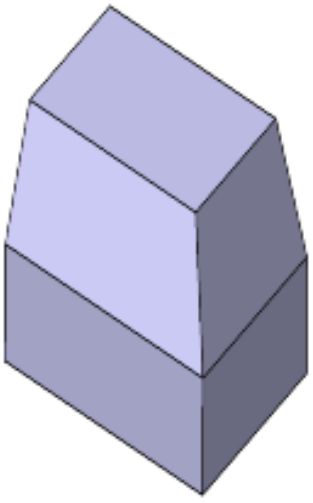


Faces and planes to be selected

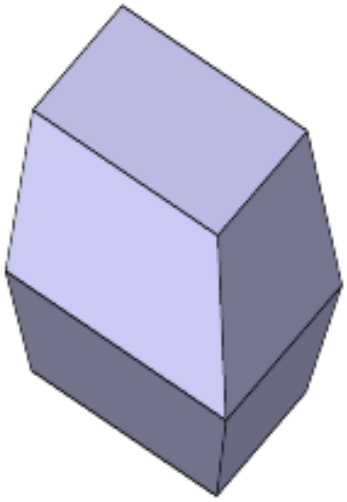


Resulting drafted faces

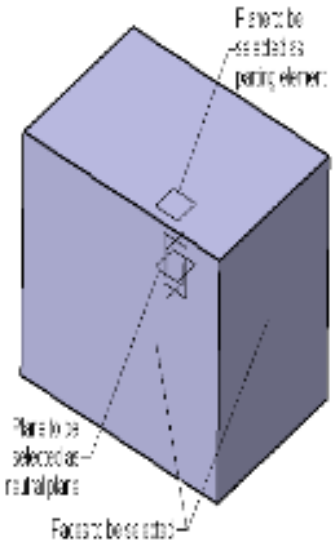
- **Defining the Parting Element While Adding Drafts to the Faces**



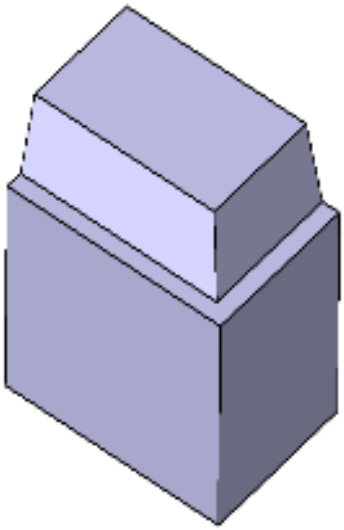
Faces drafted with the **Parting = Neutral** check box selected.



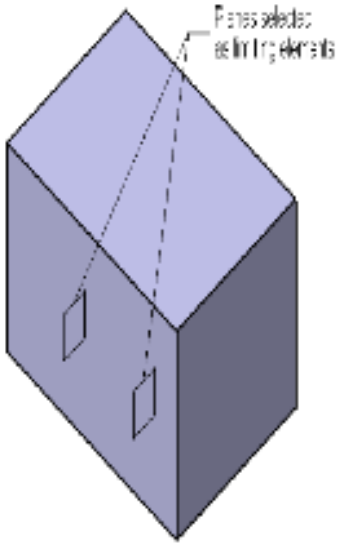
Faces drafted with the **Draft both sides** check box selected



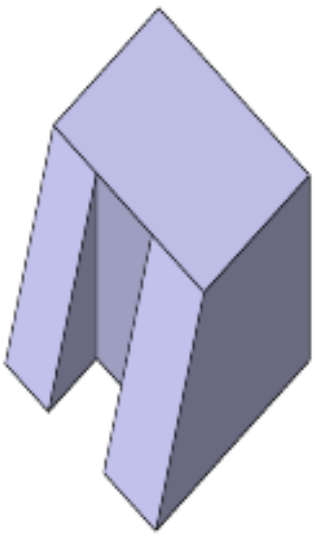
References to be selected



Resulting drafted faces



Limiting faces to be selected

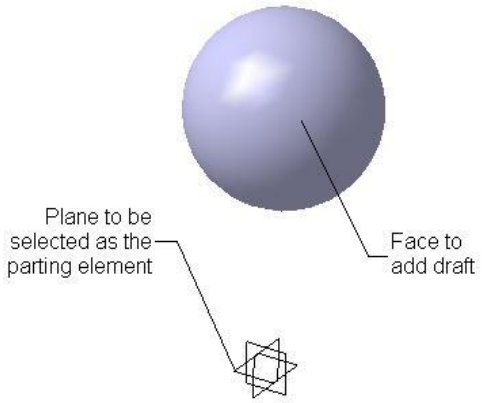


Resulting drafted face

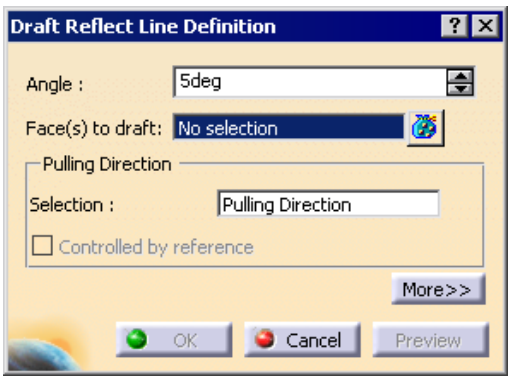
• Adding Drafts using the Reflect Line

• 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.



Face and plane to be selected





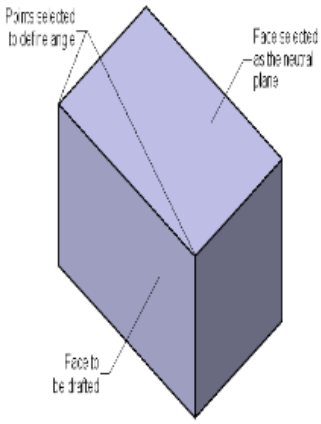
Resulting draft feature

The Draft Reflect Line Definition dialog box

• Adding Variable Angle Draft

• To create a variable angle draft, choose the **Variable Angle Draft** from the **Drafts** toolbar.

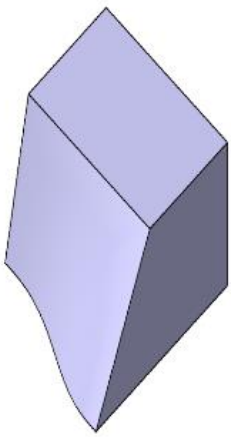
• The **Draft Definition** dialog box is displayed, as shown in the figure.



References to be selected



The Draft Definition dialog box

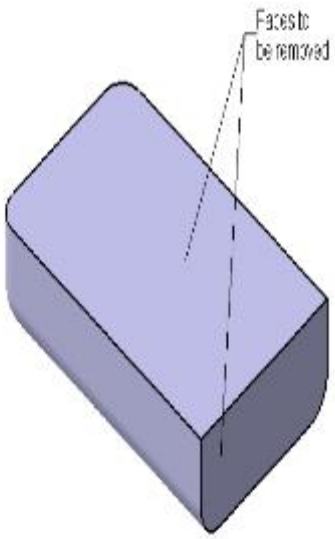


Face after adding draft

• Creating a Shell Feature

• To create a shell feature, choose the **Shell** button from the **DressUp Features** toolbar.

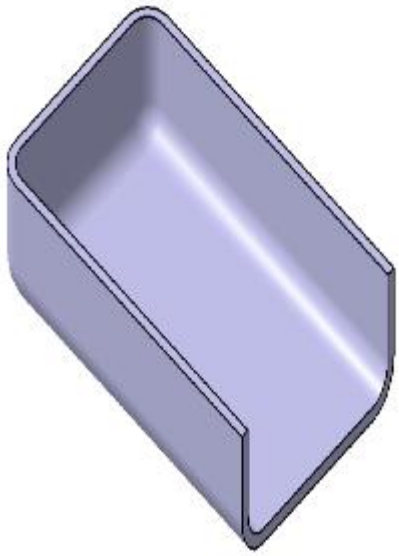
• The **Shell Definition** dialog box is displayed, as shown in the figure.



Faces to be selected for removal

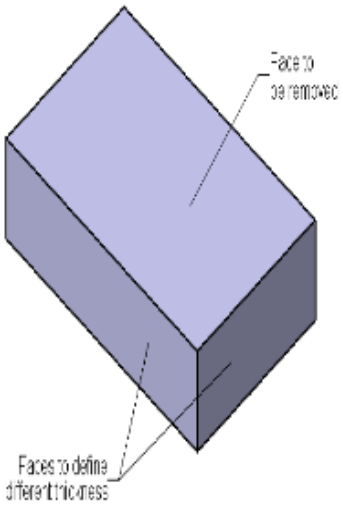


The Shell Definition dialog box

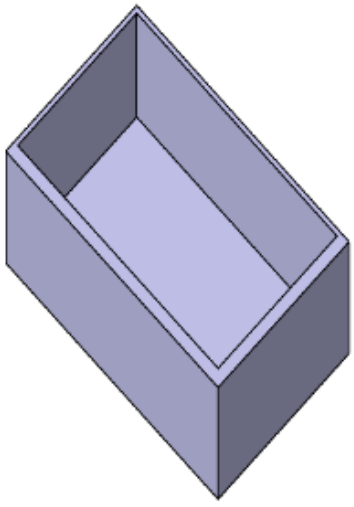


Resulting shelled model

- **Creating a Multithickness Shell**



Faces to be selected



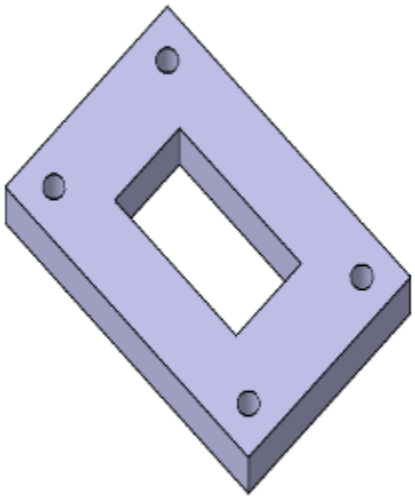
Resulting shelled model

Chapter 6

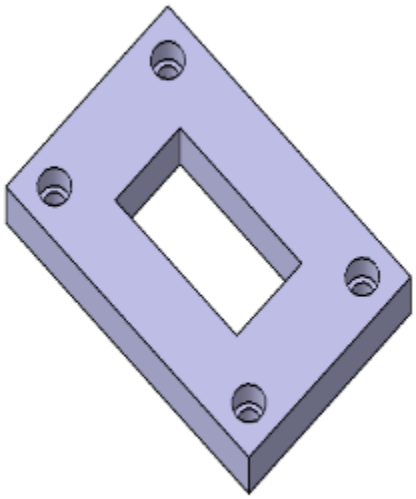
➢ EDITING FEATURES OF A MODEL

• 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 hole



Modified Base Plate

The editing methods available in CATIA are:

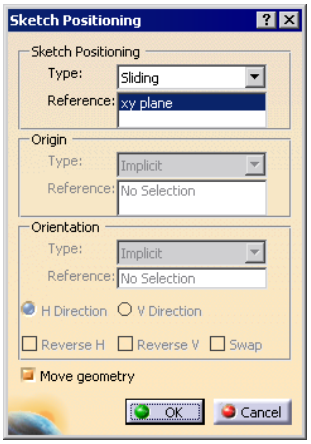
- **Editing Using the Definition Option**
- **Editing by Double-clicking**
- **Editing the Sketch of a Sketch-based Feature**

• Redefining the Sketch Plane of Sketches

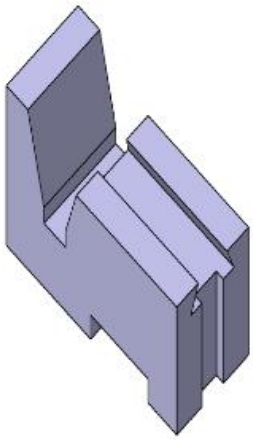
• To edit a sketch, select it from the specification tree and invoke the contextual menu.

• 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.

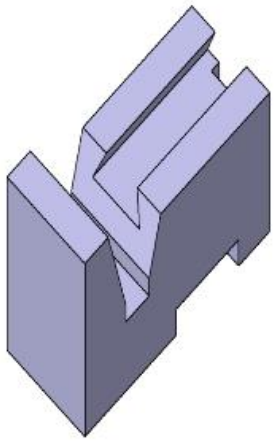
• The **Sketch Positioning** dialog box is displayed, as shown in the figure.



The Sketch Positioning dialog box

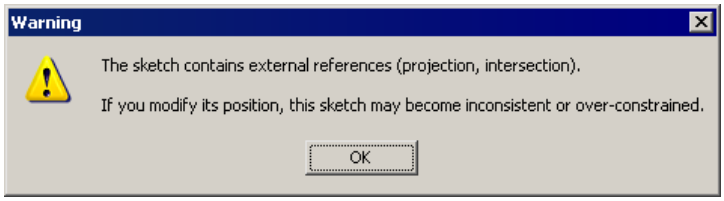


Base feature created on the YZ plane



The model, after redefining the sketch plane of the base feature to ZX plane

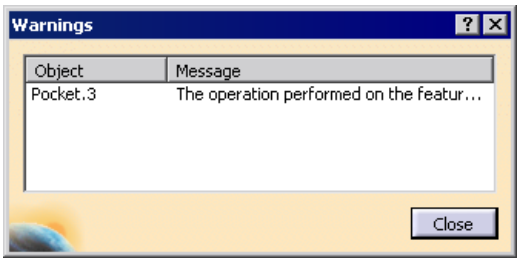
• When you redefine the sketch plane of the sketch-based features, sometimes a **Warning** dialog box is displayed, as shown in the figure.



The Warning dialog box

• Managing the Features and Sketches by Cut,Copy, and Paste Functionality

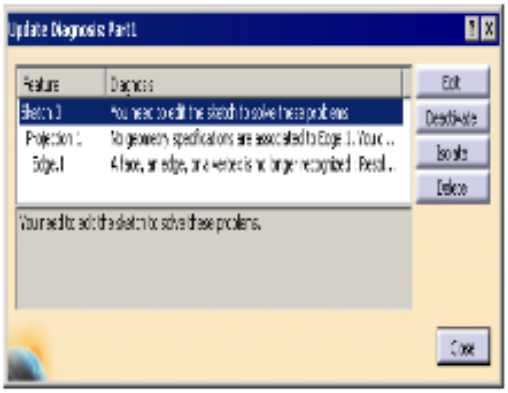
• If the resulting feature merges with the model, the **Warnings** dialog box is displayed, as shown in the figure.



The Warnings dialog box

• Understanding the Concept of Update Diagnose

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

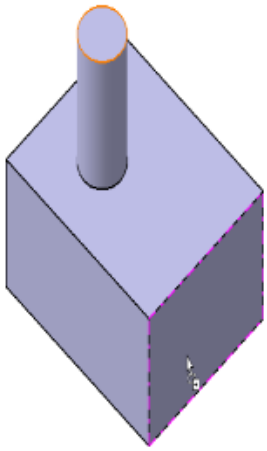


The Update Diagnosis dialog box

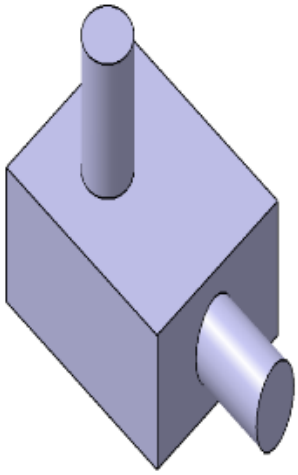
• Cut, Copy, and Paste Features and Sketches from One File to Another

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

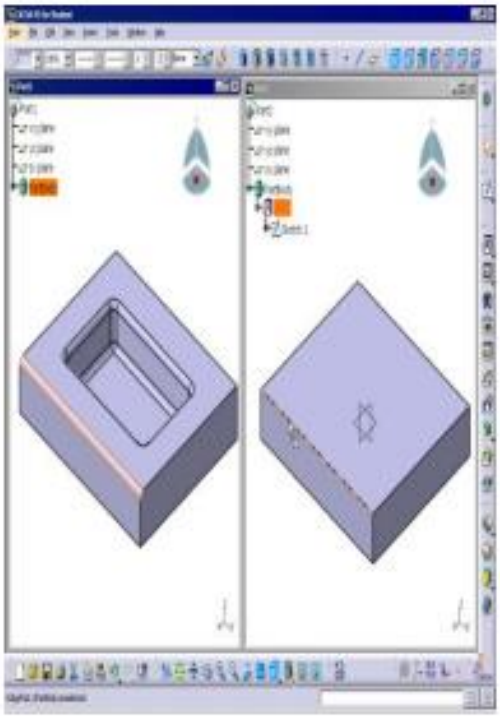
- **Copying Features Using Drag and Drop**



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** and 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.



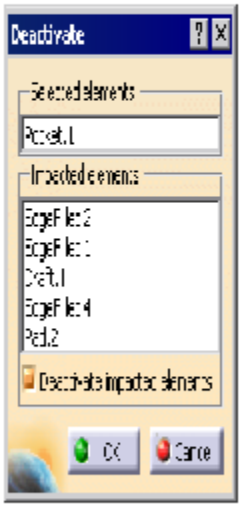
The Paste Special dialog box

- **Deleting Unwanted Features**



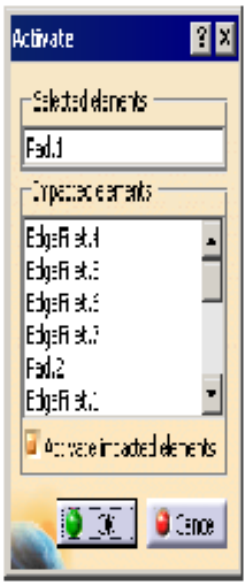
The Delete dialog box

- **Deactivating Features**



The Deactivate dialog box

- **Activating Deactivated Features**



Activate dialog box

• Defining Features in Work Object

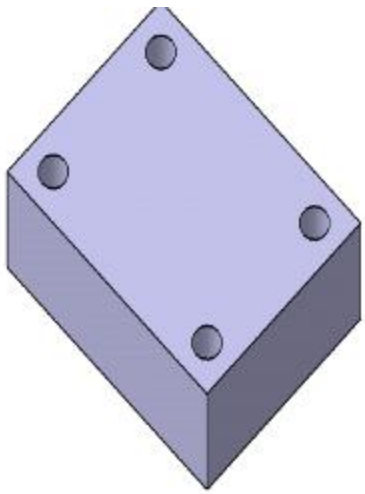
• Defining a feature in the work object is as a process in which you rollback the model to an earlier stage.

• Reordering the Features

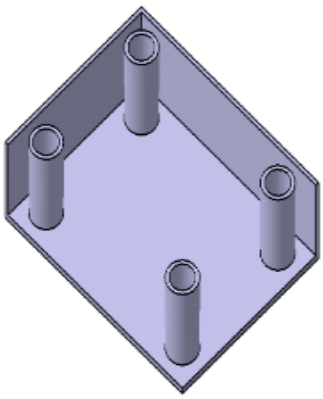
• Reordering is defined as a process of changing the order, in which the features were created in the model.



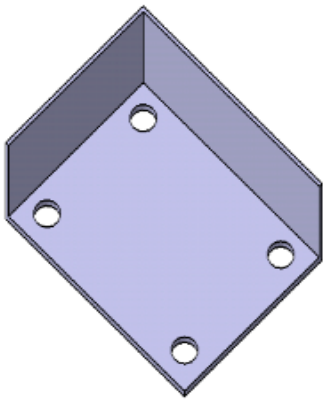
The Feature Reorder dialog box



Model with through holes on the top face



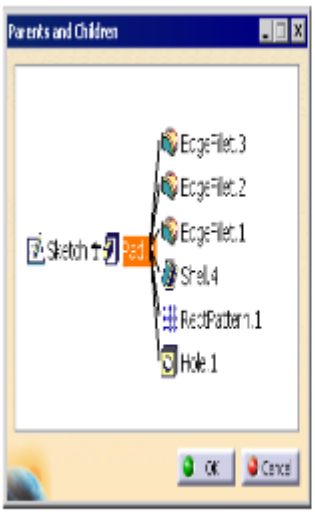
Model after shelling



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.

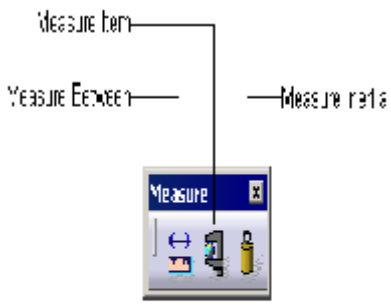


The Parents and Children dialog box

• MEASURING ELEMENTS

• The **Part** workbench of CATIA V5 also provides you with the tools to measure distance, angle, radius, area, and inertia.

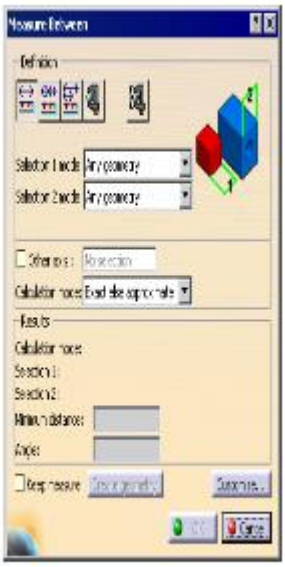
• To measure these elements, you can use the tools available in the **Measure** toolbar, as shown in the figure.



The Measure toolbar

• Measuring Between Elements

• To measure 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.

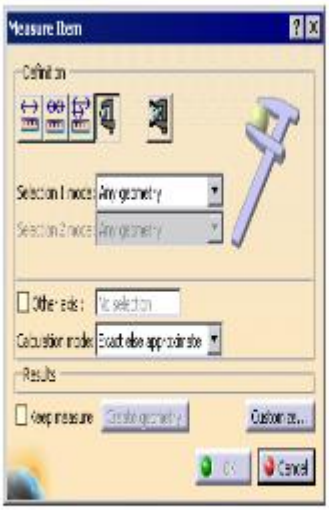


The Measure Distance dialog box

• Measuring Items

• To measure elements using this tool, choose the **Measure Item** button from the **Measure** toolbar.

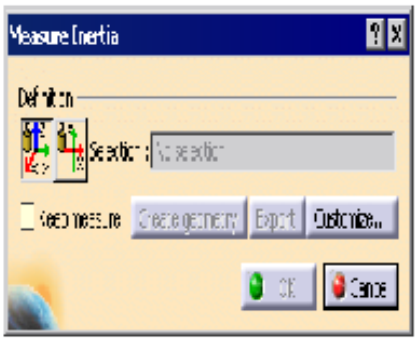
• The **Measure Item** dialog box is displayed, as shown in the figure.



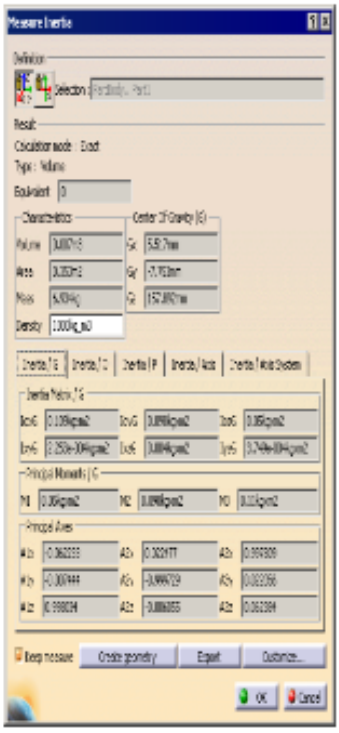
The Measure Item dialog box

• Measuring Inertia

• 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.



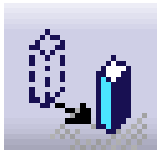
The Measure Inertia dialog box



The expanded Measure Inertia dialog box

Chapter 7

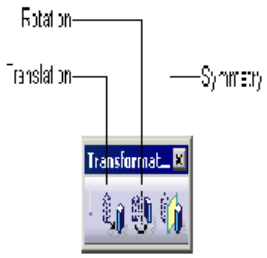
➢ TRANSFORMATION FEATURES



• Translating Bodies

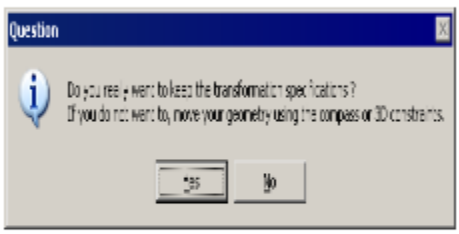
• To translate the current body, invoke the transformation toolbar, by choosing the down arrow on the right of the **Translation** button from the **Transformation Features** toolbar.

• The **Transformations** toolbar is shown in the figure.



The Transformations toolbar

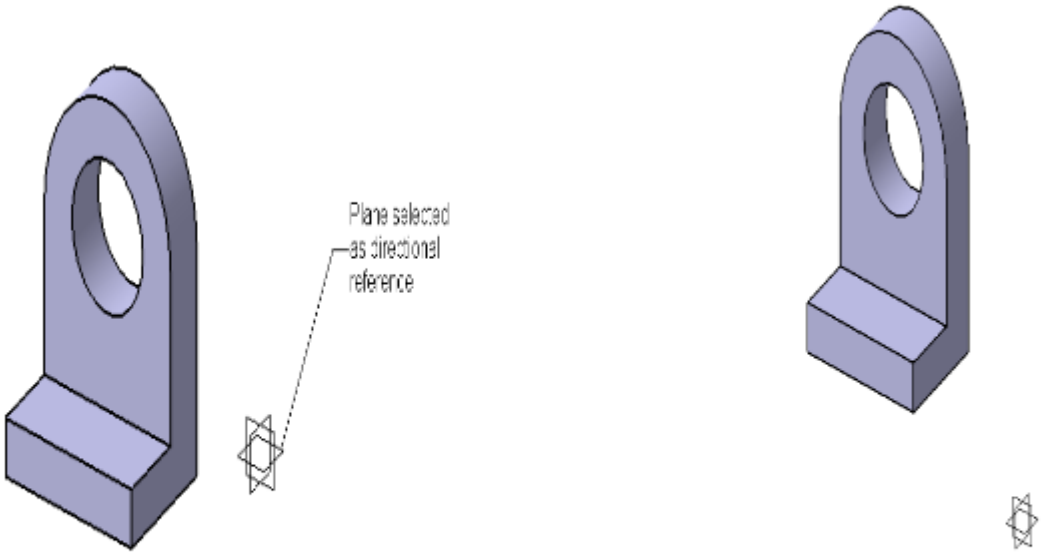
• Choose the **Translation** button from the **Transformations** toolbar. The **Question** dialog box is displayed, as shown in the figure.



The Question dialog box



The Translate Definition dialog box

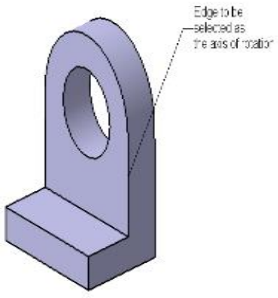


Plane selected as directional reference Resulting translated body

• Rotating Bodies

• Choose the **Rotation** button from the **Transformations** toolbar; the **Question** dialog box is displayed.

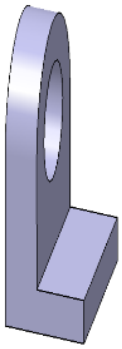
• Choose the **Yes** button from this dialog box; the **Rotate Definition** dialog box is displayed, as shown in the figure.



Edge to be selected as rotation axis



The Rotation Definition dialog box



Resulting rotated body

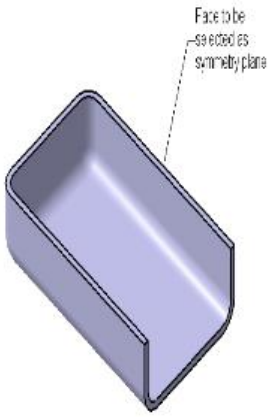
• Creating Symmetry Features

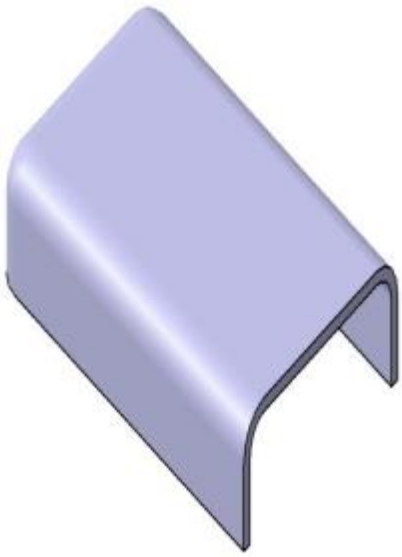
• After choosing the **Symmetry** button from the **Transformations** toolbar, choose the **Yes** button from the **Question** dialog box.

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



The Symmetry Definition dialog box

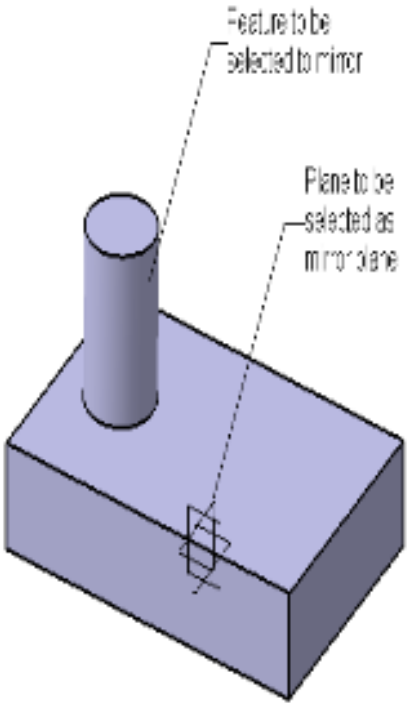




Resulting symmetry feature

• Mirroring Features and Bodies

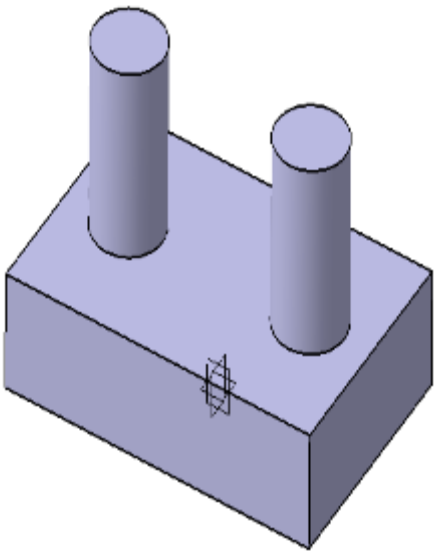
• To mirror the selected feature, select it from geometry area or from the specification tree and choose the **Mirror** button from the **Transformation Features** toolbar; the **Mirror Definition** dialog box is displayed, as shown in the figure.



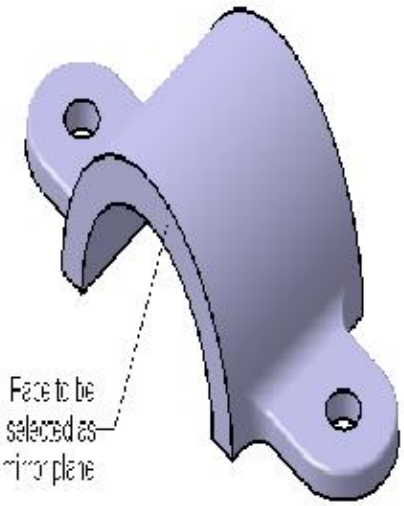
Feature and the mirror plane to be selected



The Mirror Definition dialog box



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**.

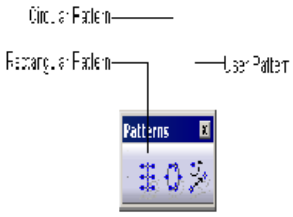


Figure A The Patterns toolbar

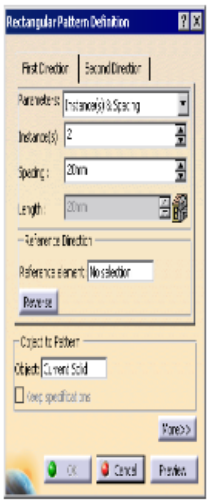
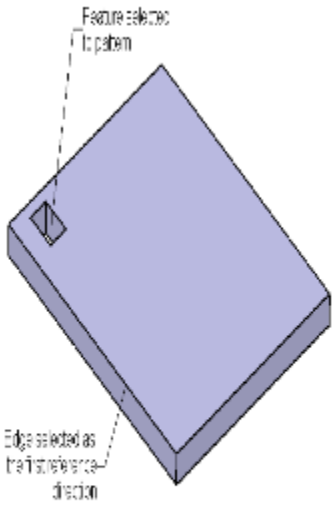
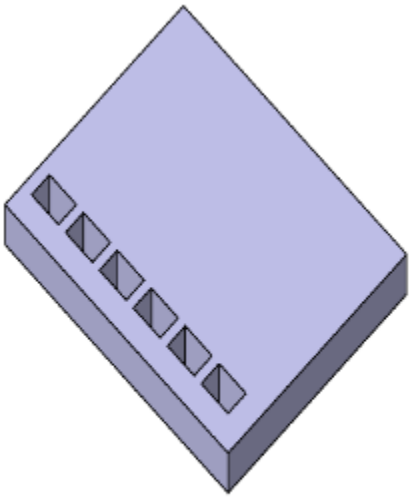


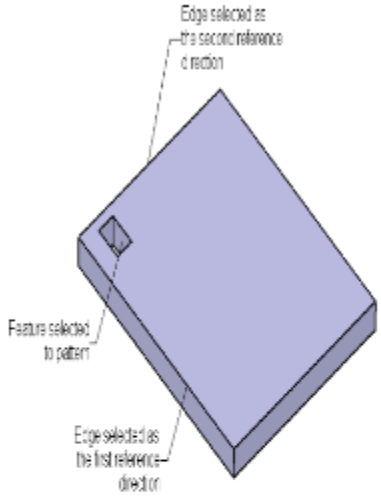
Figure B The Rectangular Pattern Definition dialog box



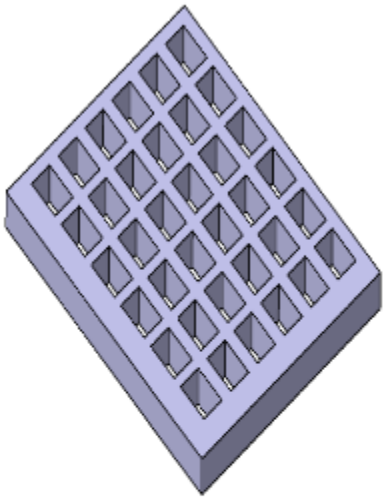
Feature and the edge to be selected



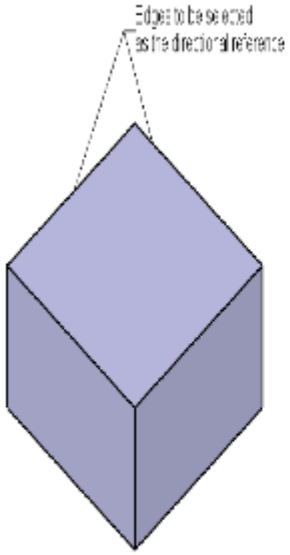
Resulting pattern



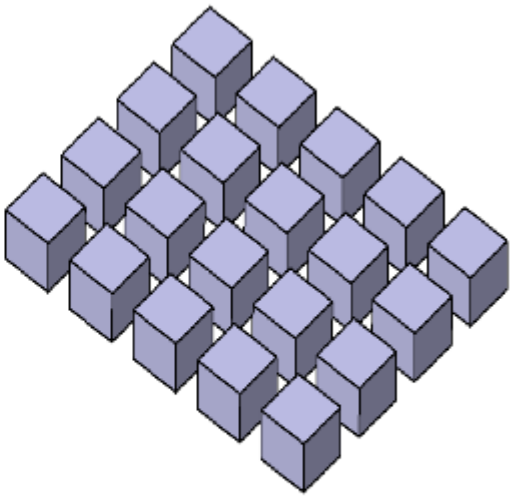
Feature and the directional references to be selected



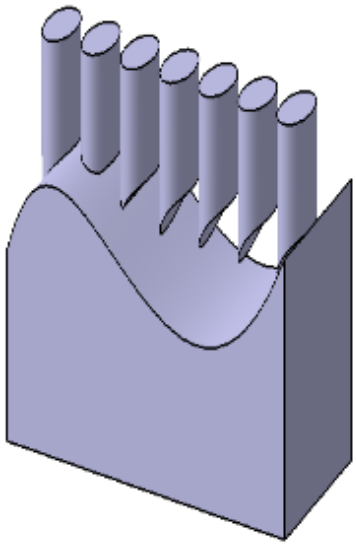
Resulting pattern



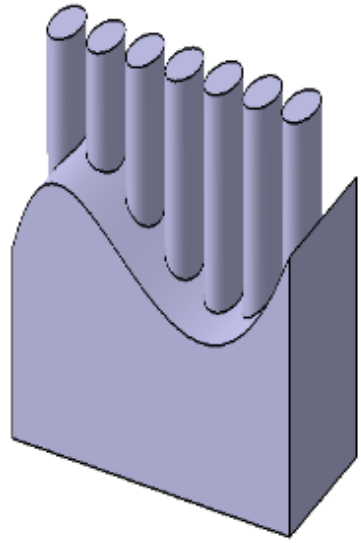
Directional references to be selected



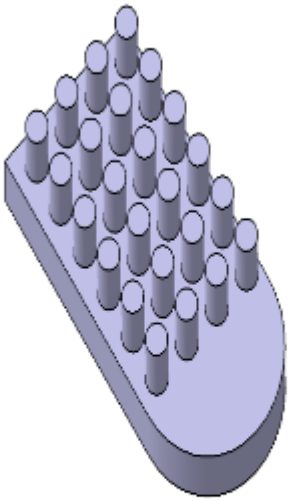
Resulting pattern of the PartBody



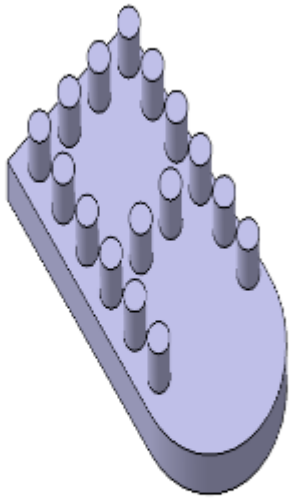
Pattern with the Keep specifications check box cleared



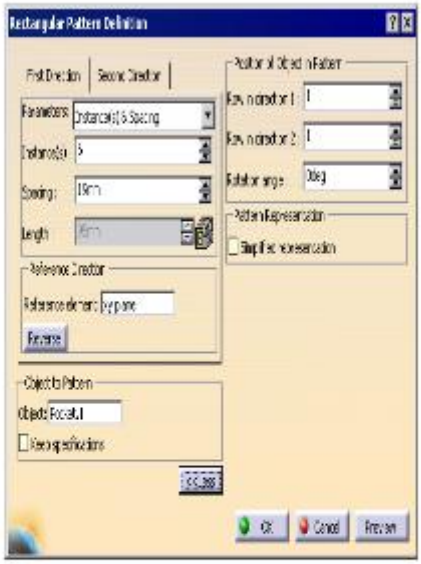
Pattern with the Keep specifications check box selected



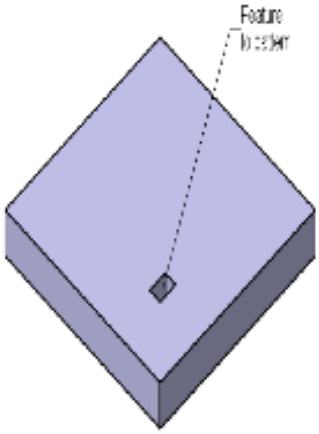
Patterned feature without skipping any instance



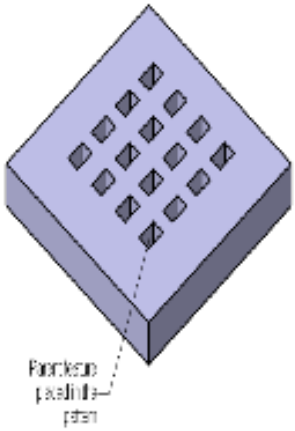
Patterned feature after skipping some of the instances



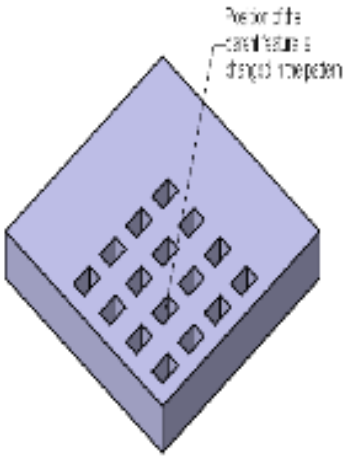
The expanded Rectangular Pattern Definition dialog box



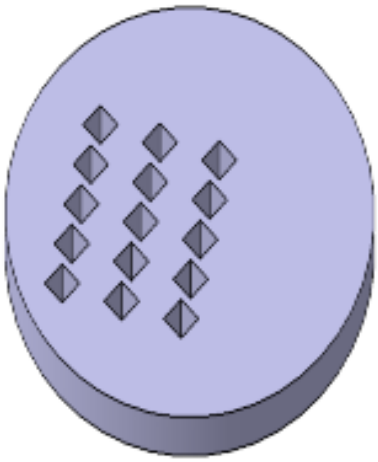
Feature to be pattern



Position of the parent feature in the pattern



Position of the parent feature after modifying the position of the parent feature



Rotated pattern

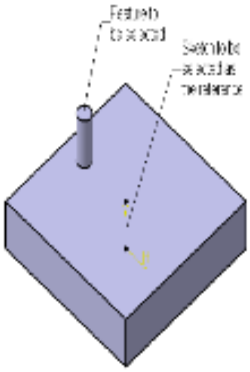
• Creating Circular Patterns

• To arrange features or the current body in a circular manner,choose the **Circular Pattern** button from the **Patterns** toolbar.

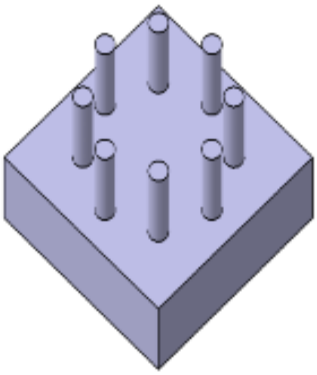
• The **Circular Pattern Definition** dialog box is displayed, as shown in the figure.



The Circular Pattern Definition dialog box



Feature to be patterned and the reference element to be selected



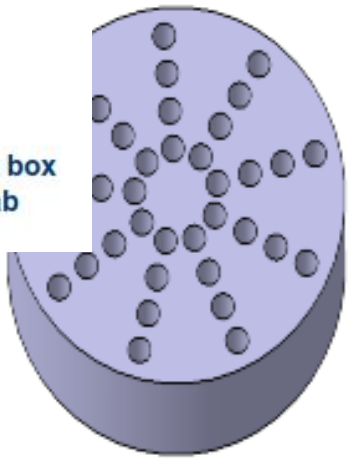
Resulting patterned feature



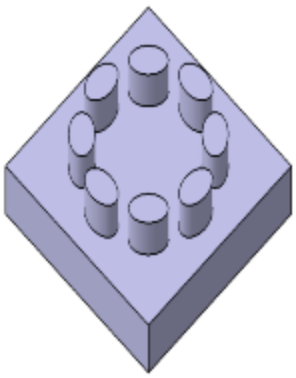
The Circular Pattern Definition dialog box after invoking the Crown Definition tab



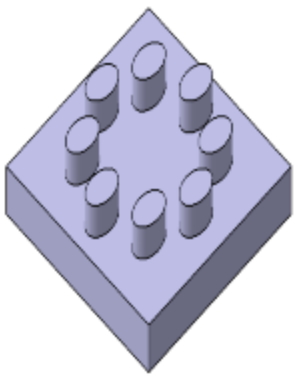
Various parameters used to define the crown in the pattern



Resulting pattern after defining the crown parameters



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

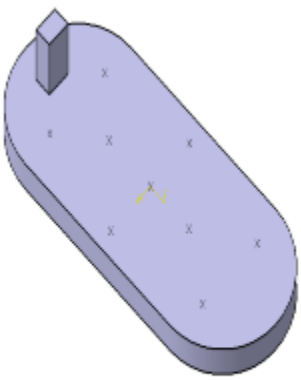


Pattern with the Radial alignment of instance(s) check box cleared

• Creating User Patterns

• To create a user pattern, select the feature to pattern and then choose the **User Pattern** button from the **Patterns** toolbar.

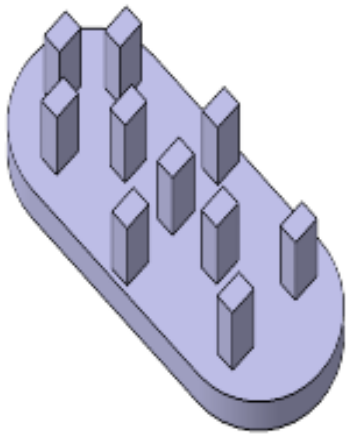
• The **User Pattern** **Definition** dialog box is displayed, as shown in the figure.



Features and the reference sketch



The User Pattern Definition dialog box



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.

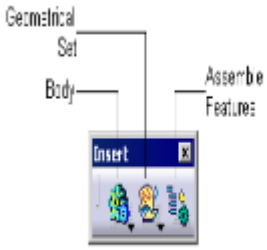


The Scaling Definition dialog box

➢ WORKING WITH ADDITIONAL BODIES

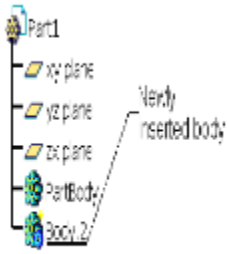
• To invoke the Insert toolbar, move the cursor on any one of the toolbars and invoke the contextual menu.

• Choose **Insert** from it; the **Insert** toolbar will be displayed, as shown in the figure.



The Insert toolbar

• Inserting a New Body



Newly inserted body

• Inserting a New Body Using Dialog Box

• With this release, a new tool is introduced in the **Part Design** workbench.

• To insert a body using this tool, choose the **Body By Dialog Box** button from the **Container Body** toolbar.

• Inserting Features in the Assemble Feature



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.

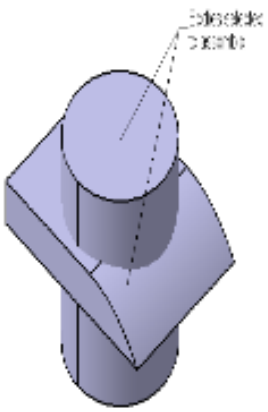
• Assembling Bodies

• To assemble the two bodies together, choose the **Assemble** button from the **Boolean Operations** toolbar.

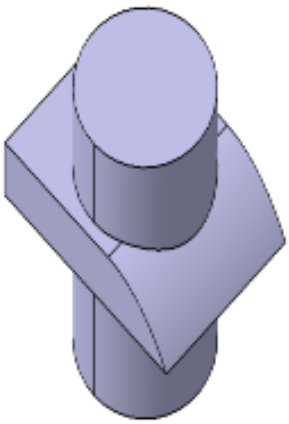
• The **Assemble** dialog box is displayed, as shown in the figure.



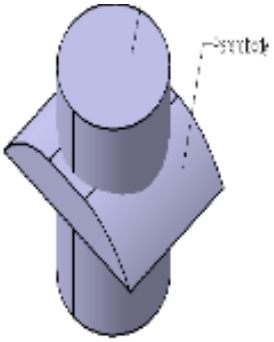
The Assemble dialog box



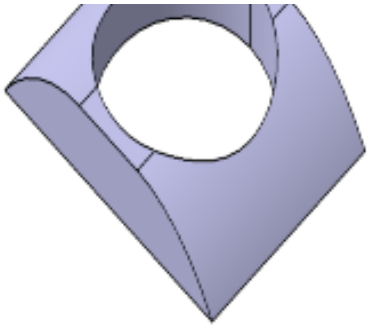
Bodies to be assembled



Resulting assembled body



Pocket body and the parent body

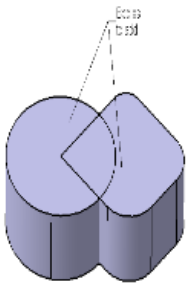


Resulting assembled body

• Adding Bodies

• The **Add** tool is used to add the selected bodies together.

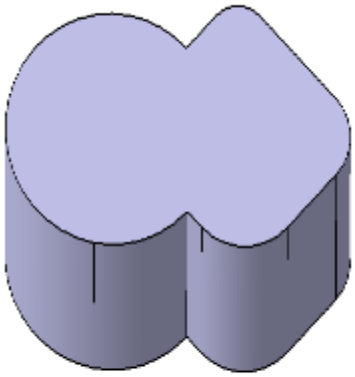
• When you invoke this tool, the **Add** dialog box is displayed, as shown in the figure



Bodies to be added



The Add dialog box



Resulting body

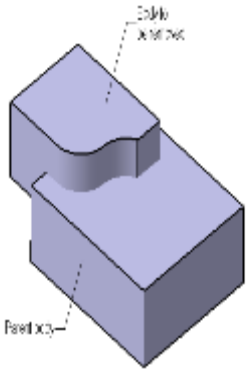
• Subtracting Bodies

• The **Remove** tool is used to subtract the selected body from another body.

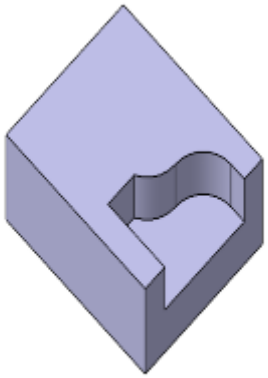
• When you invoke this tool, the **Remove** dialog box is displayed, as shown in the figure.



The Remove dialog box



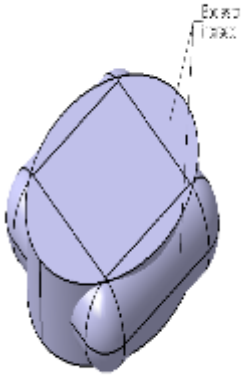
Body to be removed and the parent body



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



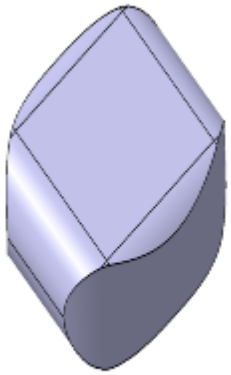
Body to be selected

selected bodies.

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



The Intersect dialog box



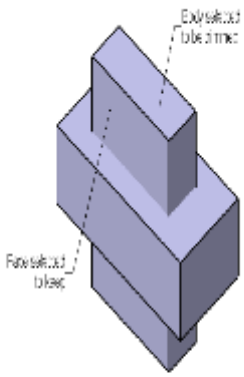
Resulting body

• Trimming Bodies

• When you invoke the **Union Trim** tool, you will be prompted to select the body to trim, hence do so.

• Note that the parent body can not be trimmed.

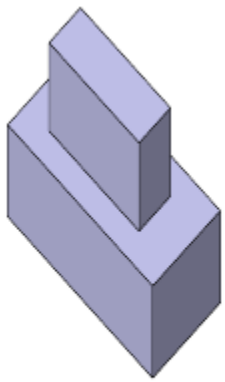
• The **Trim Definition** dialog box is displayed, as shown in the figure.



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



The Trim Definition dialog box

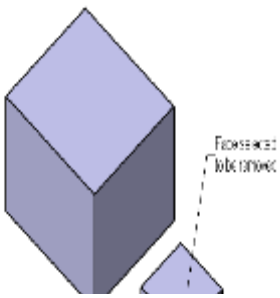


Resulting trimmed body

• Removing Lumps

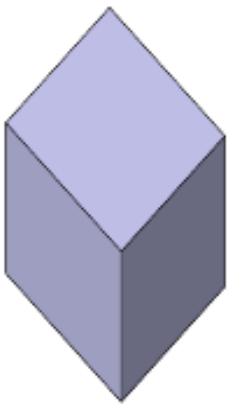
• When you invoke the **Remove Lump** tool, you are prompted to select the body to trim.

• 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



Resulting body after removing the lump

➢ ADDING STIFFENERS TO THE MODEL

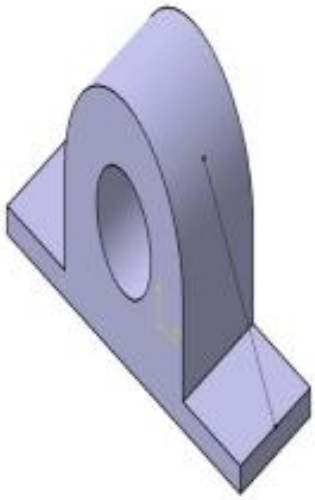
• Stiffeners are generally added to increase the strength of the components under the increased loading conditions.

• To add a stiffener, choose the **Stiffener** button from the **Sketch-Based Features** toolbar.

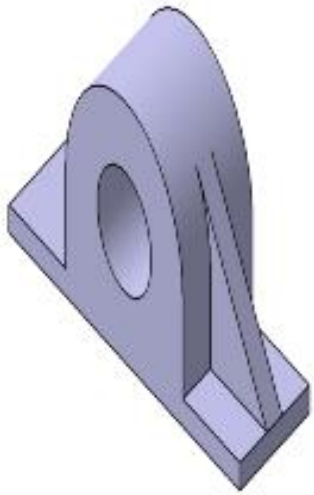
• The **Stiffener Definition** dialog box is displayed, as shown in the figure.



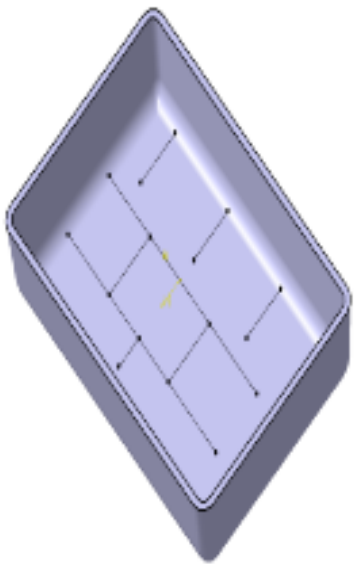
The Stiffener Definition dialog box



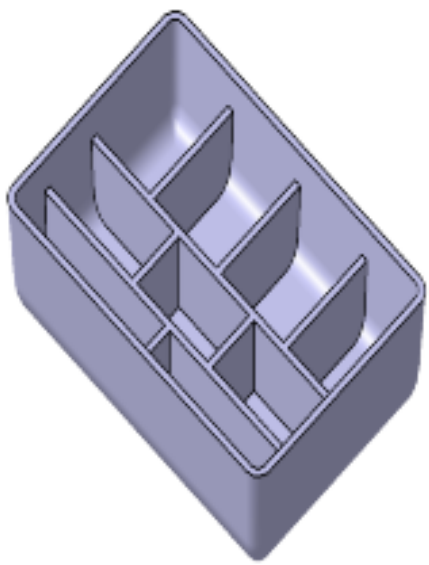
Sketch for the stiffener



Resulting stiffener



Sketch for the stiffener



Resulting stiffener

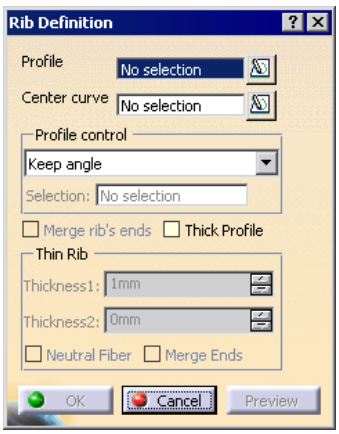
Chapter 8

➢ ADVANCED MODELING TOOLS

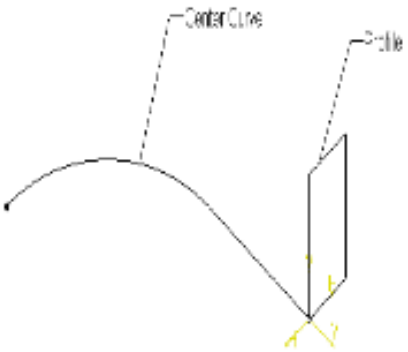
• Creating Rib Features



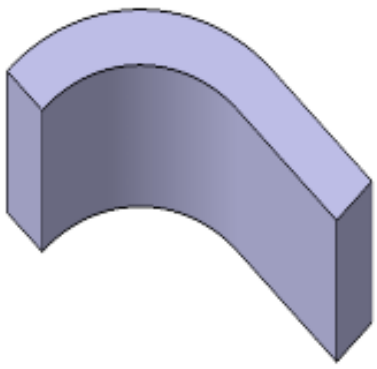
• To create a rib feature, choose the **Rib** button from the **Sketch-Based Features** toolbar; the **Rib Definition** dialog box is displayed, as shown in the figure.



The Rib Definition dialog box



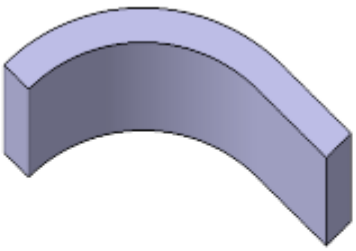
Profile and the center curve to



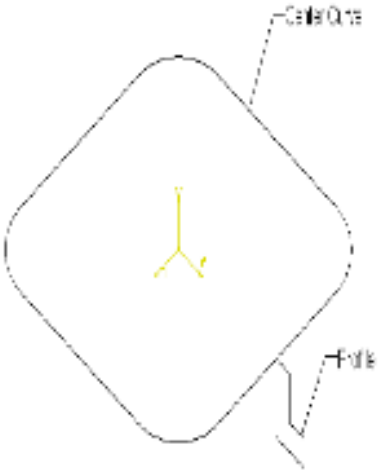
Resulting rib feature



Unattached profile and the center curve



Resulting rib feature

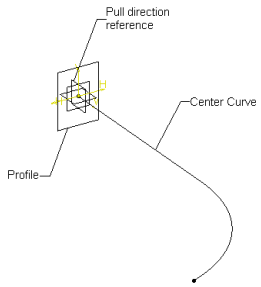


Closed profile and closed center curve

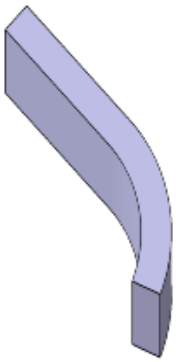


Resulting rib feature

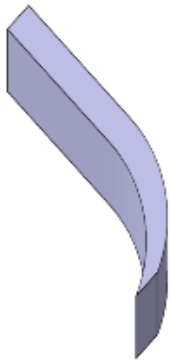
- **Defining the Pulling Direction**



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

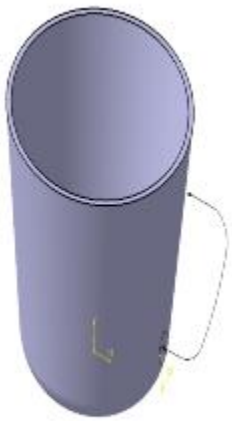


The rib feature without the pulling direction



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.



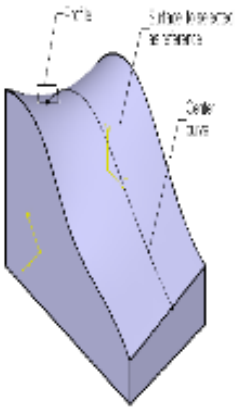
The Thin Rib area of the Rib Definition dialog box



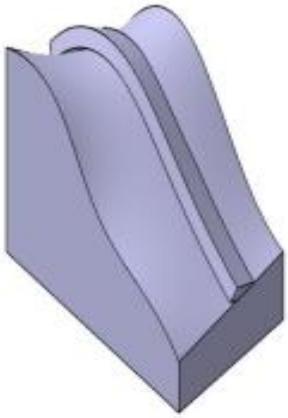
A thin rib feature

- **Defining the Reference Surface**

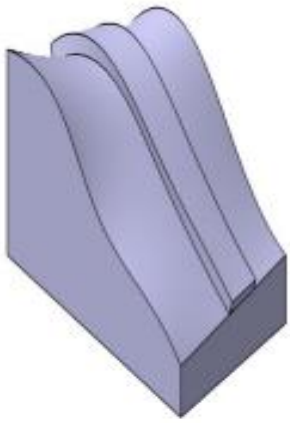
- A reference surface can be defined for creating the rib feature.



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

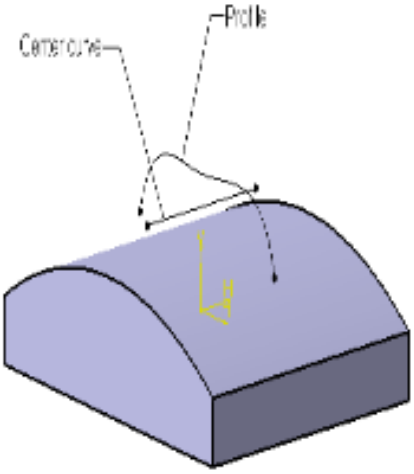


Rib feature with the **Keep Angle** option selected from the **Profile control** area

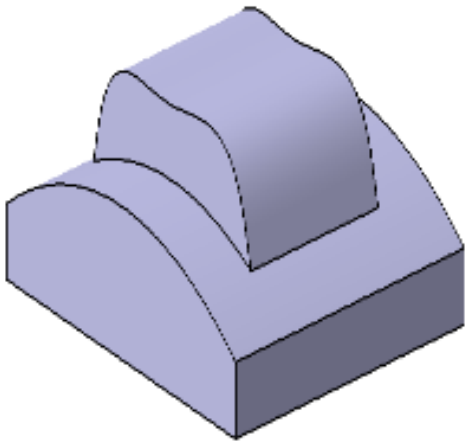


Rib feature after defining the reference surface

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



Profile and the center curve

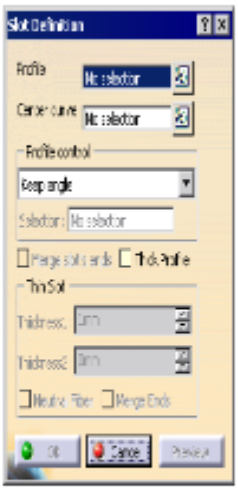


Resulting rib feature

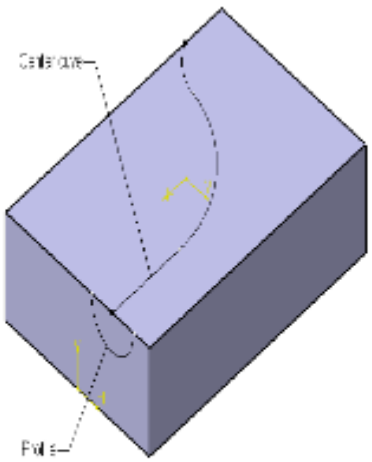
• Creating Slot Features

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

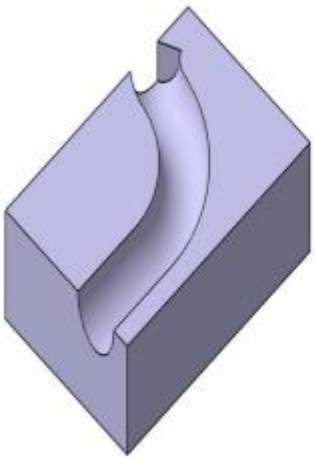
• To create this feature, choose the **Slot** button from the **Sketch Based Features** toolbar; the **Slot Definition** Dialog box is displayed,as showninthe figure.



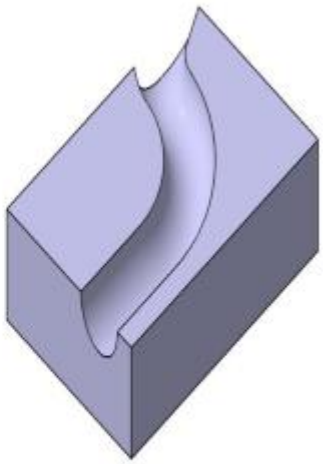
The Slot Definition dialog box



Profile and the center curve



Resulting rib feature



Rib feature after selecting the Merge slot's ends check box

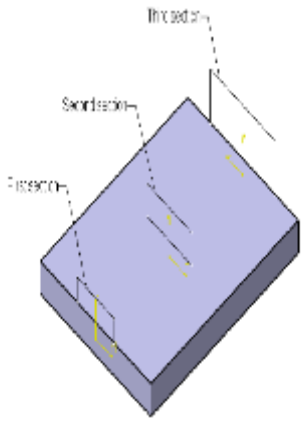
• Creating Lofted Features

• To create a loft feature, choose the **Loft** button from the **Sketch-Based Features** toolbar.

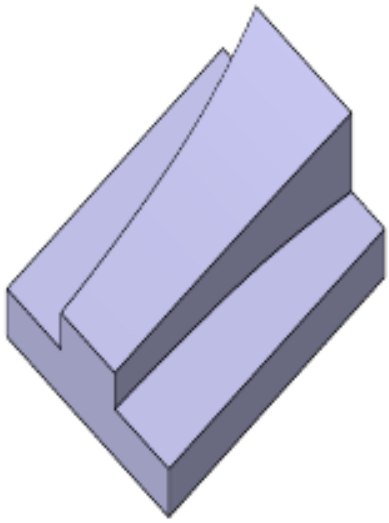
• The **Multi-sections Solid Definition** dialog box is displayed, as shown in the figure.



The Multi-sections Solid Definition dialog box



Sections selected to loft



Resulting lofted feature

• Lofting Sections with Unequal Number of Vertices

• To create this type of feature, invoke the **Multi-sections Solid 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**.

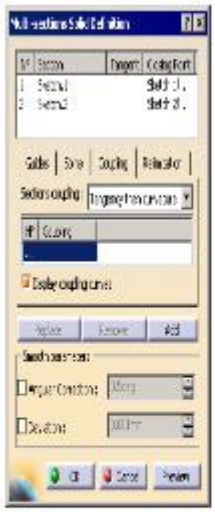
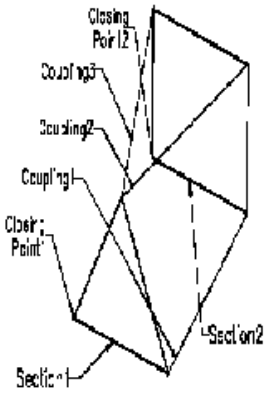


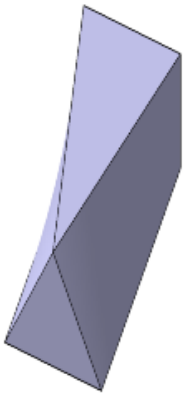
Figure A The Multi-sections Solid Definition dialog box, after invoking the Coupling tab



The Coupling dialog box

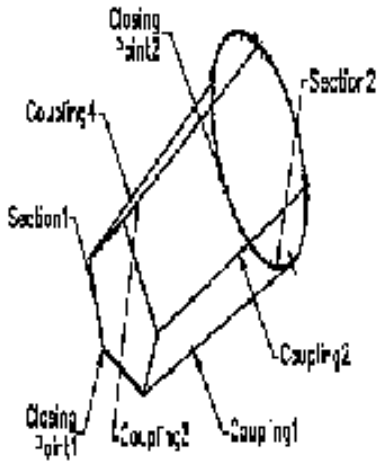


Sections, closing points, and couplings

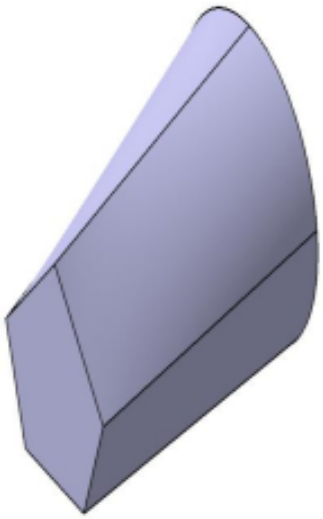


Resulting lofted feature

• Lofting a Circular Section with a Polygonal Section



Sections, closing points, and couplings



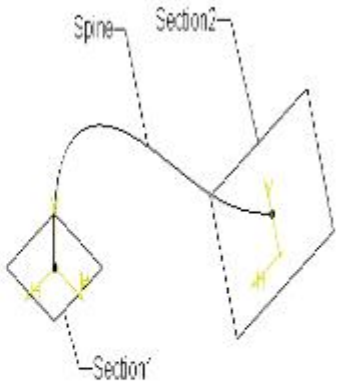
Resulting lofted feature

• Lofting the Sections Along a Spine

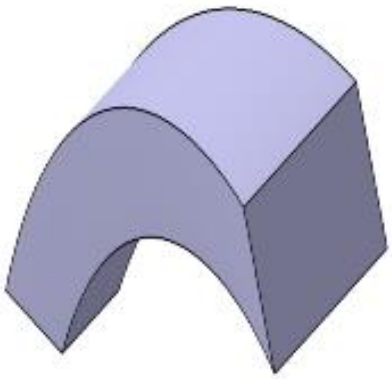
• To create a lofted feature using this option, invoke the **Spine** tab of the **Multi-sections Solid Definition** dialog box, as shown in the figure.



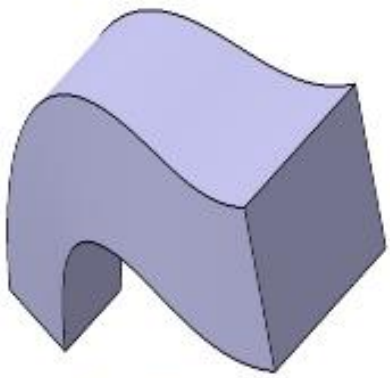
The Multi-sections Solid Definition dialog box after invoking the Spine tab



Sections and the spine to be selected

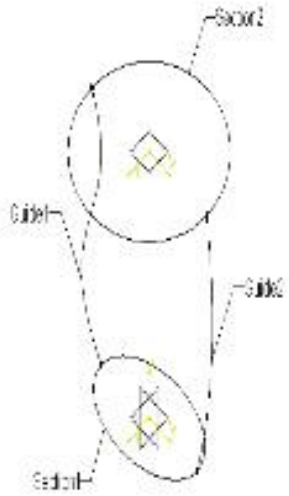


Lofted feature without selecting the spine



Lofted feature after selecting the spine

• Creating Loft Feature with Guides



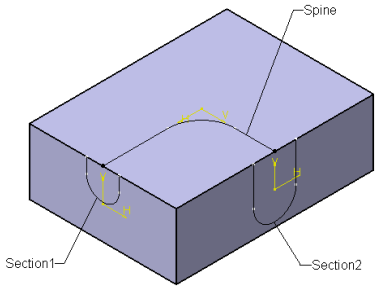
The sections and the guides



Resulting lofted feature

• Creating Removed Loft Features

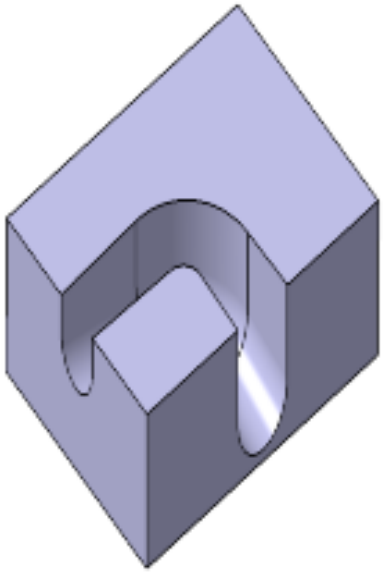
• When you invoke the **Remove Loft** tool, the **Removed Multisections SolidDefinition** dialog box is displayed, as shown in the figure.



Sections and the spine to be selected



The Removed Multi-sections Solid Definition dialog box



Resulting removed loft feature

Chapter 9

➢ SURFACE OPERATIONS

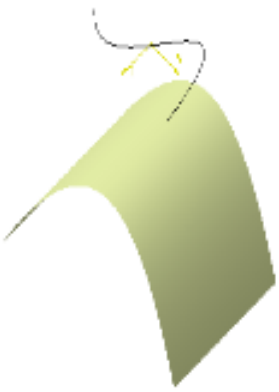
• CATIA V5 also provides you with various tools that are used to modify surfaces, which are known as surface operation tools and are used regularly, while creating surface models.

• 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



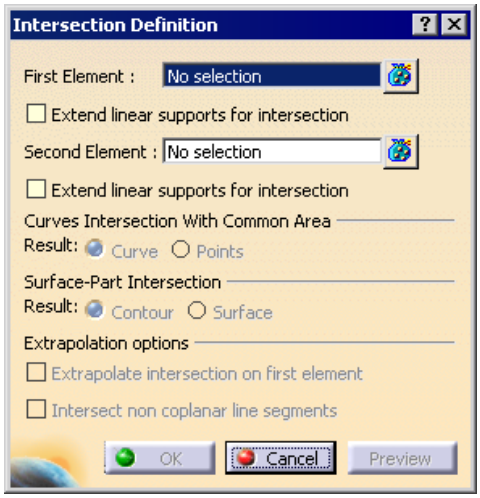
•The curve and the support surface



•The resulting projected element

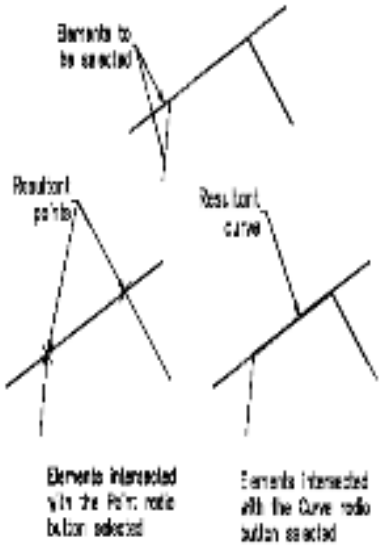
• Creating Intersection Elements

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



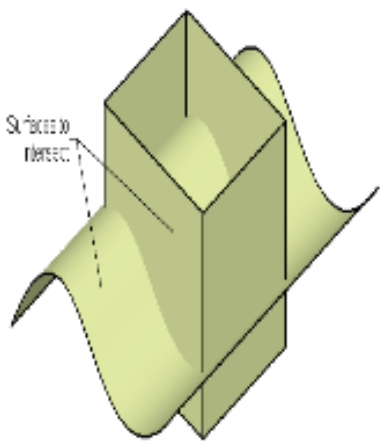
•The Intersection Definition dialog box

- Intersection of Two Curves

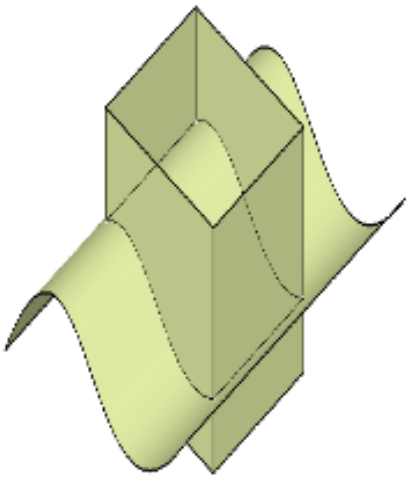


- Intersection of Two Surfaces

•Elements to intersect and the resulting intersections



•Surfaces to be selected



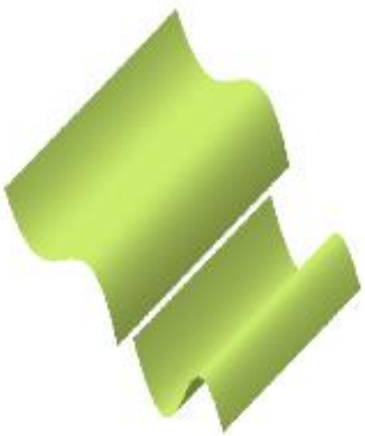
•Resulting intersection curve

• 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 surface to be healed

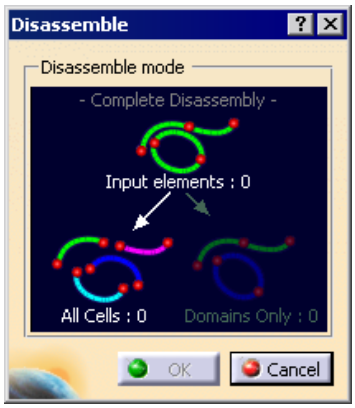


•The resulting healed surface

• 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.



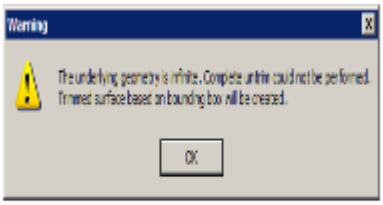
•The Disassemble dialog box

• 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



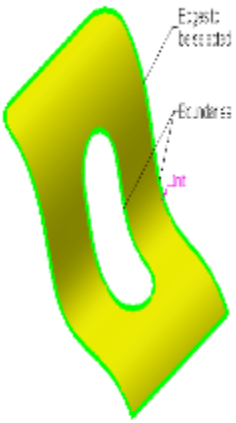
•The trimmed surface



•The untrimmed surface

• Creating Boundary Curves

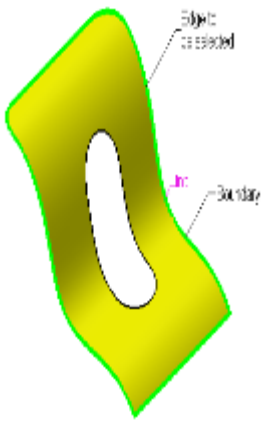
• Choose the Boundary button from the Operation toolbar to invoke the Boundary Definition dialog box, as shown in the figure.



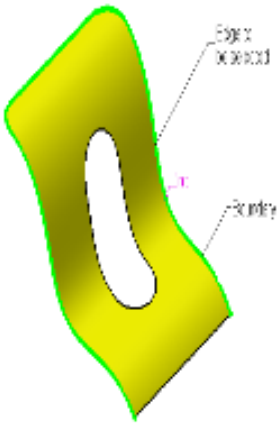
•Boundary with a complete continuity



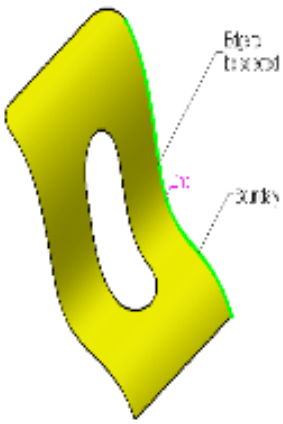
- The Boundary Definition dialog box



- Boundary with a point continuity



- Boundary with a tangent continuity

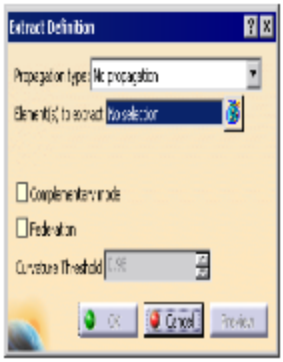


- Boundary with no continuity

• Extracting Geometry

• Choose the Extract button from the Extracts toolbar, which is invoked when you choose the down arrow available on the right of the Boundary button in the Operations toolbar.

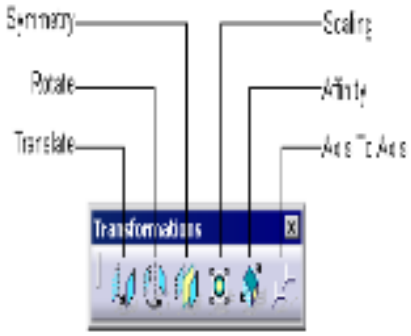
• The Extract Definition dialog box is displayed, as shown in the figure.



•The Extract Definition dialog box

• Transformation Features

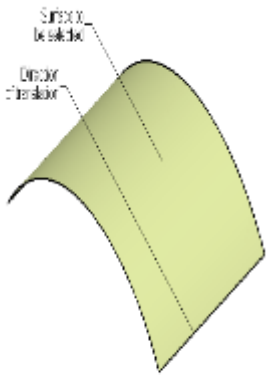
• Transformation features are the features that are used to change the physical position of the geometry, such as translating, rotating, scaling, and so on.



•The Transformations toolbar

• Translating Elements

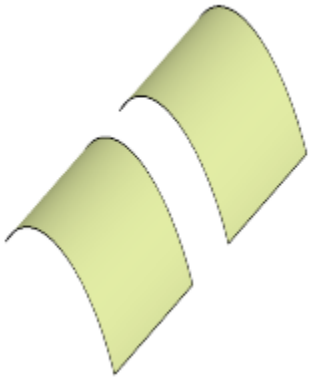
• To translate an element, choose the Translate button from the Transformations toolbar; the Translate Definition dialog box is displayed, as shown in the figure.



•Surface and directional reference



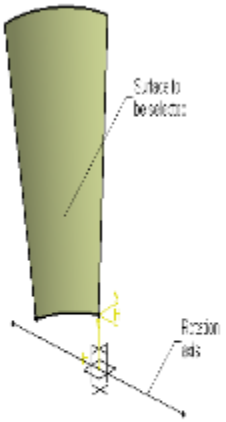
•The Translate Definition dialog box



•Resulting translated surface

• Rotating Elements

• When you choose the Rotate button from the Transformations toolbar, the Rotate Definition dialog box is displayed, as shown in the figure.



•Surface and the axis of rotation



•The Rotate Definition dialog box



•Resulting rotated surface

• Creating Symmetry Elements

• When you invoke the Symmetry tool,

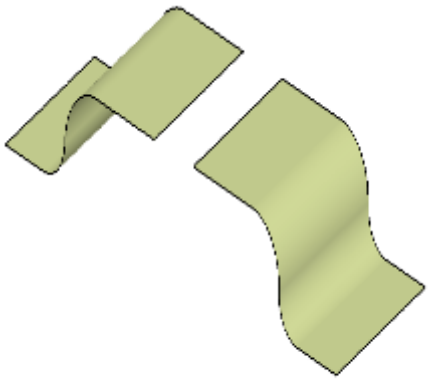


•Surface and the reference element

the Symmetry Definition dialog box is displayed, as shown in the figure.



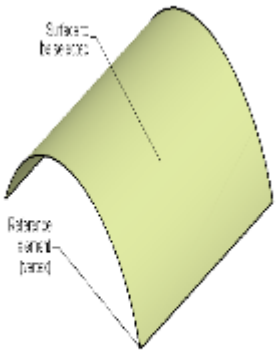
•The Symmetry Definition dialog box



•Resulting mirrored surface

• Scaling Elements

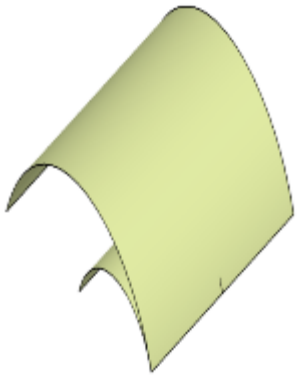
• When you invoke the Scaling tool, the Scaling Definition dialog box is displayed, as shown in the figure.



•Surface and the reference element



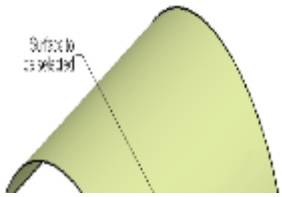
•Scaling Definition dialog box



•Resulting scaled surface

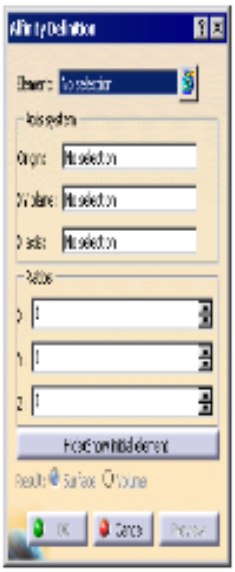
- Non uniform Scaling of Elements

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





•Resulting nonuniform scaling



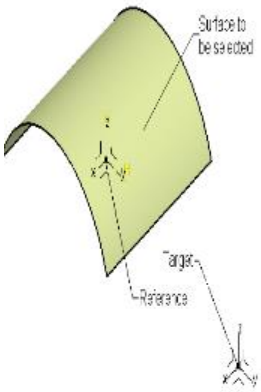
•The Affinity Definition dialog box



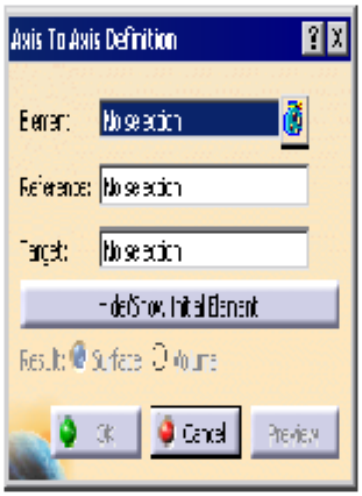
•Resulting nonuniform scaling

• Transforming Element From Axis to Axis

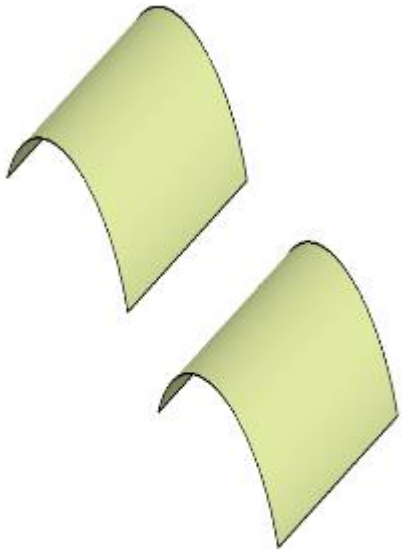
• When you invoke the Axis To Axis tool, the Axis To Axis Definition dialog box is displayed, as shown in the figure.



- Surface and axes to be selected



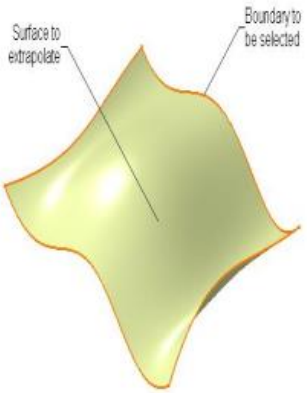
- The Axis To Axis Definition dialog box



- Resulting transformed surface

• Extrapolating Surfaces and Curves

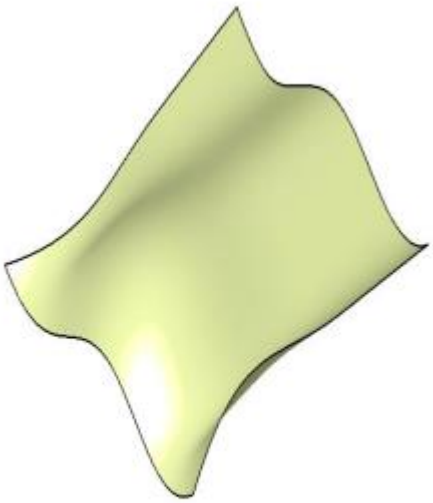
• When you invoke the Extrapolate tool, the Extrapolate Definition dialog box is displayed, as shown in the figure.



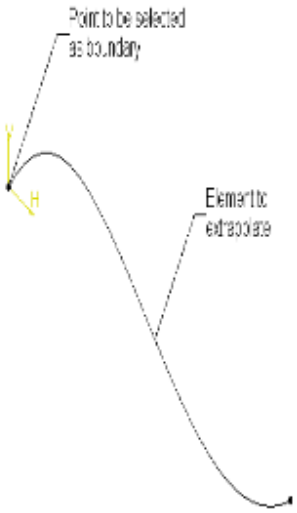
•Boundary and surface to be selected



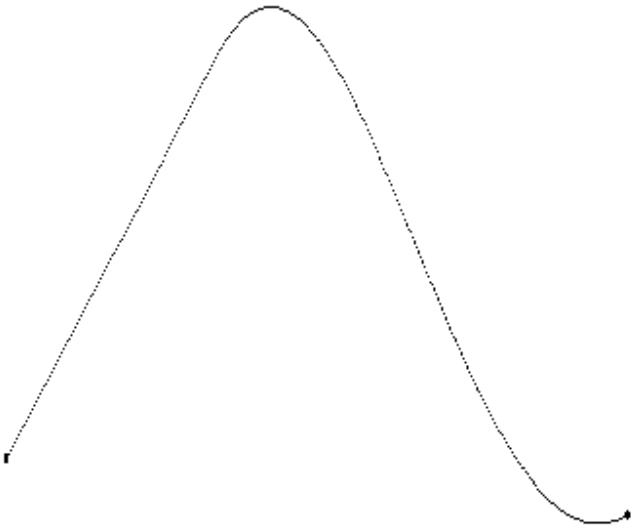
•The Extrapolate Definition dialog box



•Resulting extrapolated surface



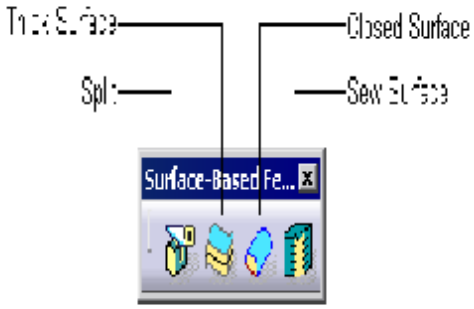
•Boundary and curve to be selected



•Resulting extrapolated curve

- Splitting a Solid Body With a Surface

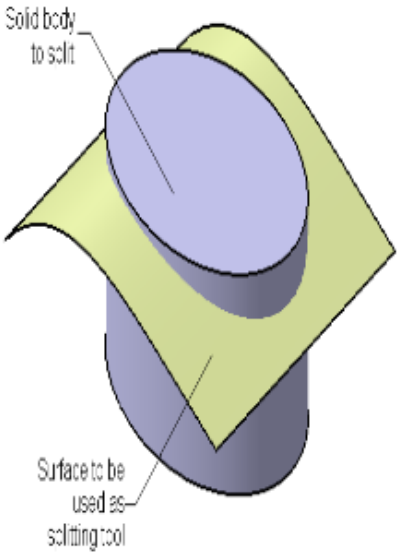
• Choose the Split button from the Surface-Based Features (Extended) toolbar; the Split Definition dialog box is displayed, as shown in the figure.



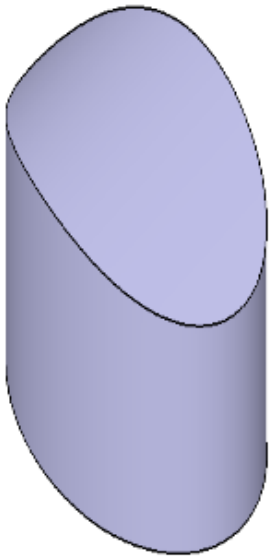
•The Surface-Based Features (Extended) toolbar



•The Split Definition dialog box



•Body to be split and the splitting surface



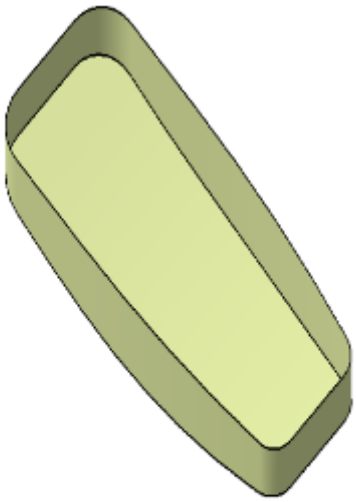
•Resulting solid body after hiding the splitting surface

➢ 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 SurfaceBased Features (Extended) toolbar.

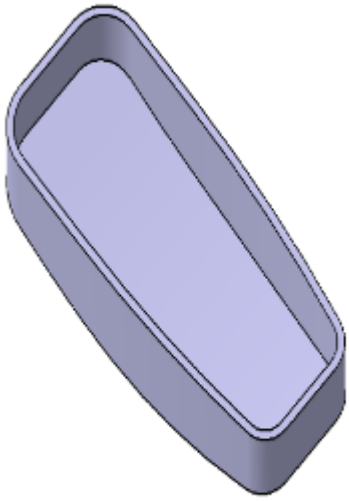
• The ThickSurface Definition dialog box is displayed, as shown in the figure.



•Surface to be thickened



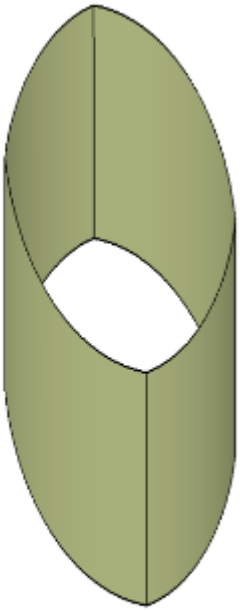
•The ThickSurface Definition dialog box



•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.

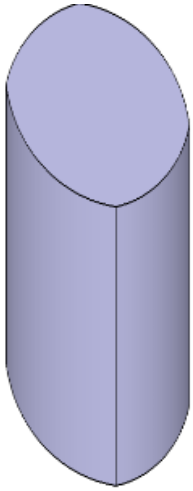


•Surface with open ends



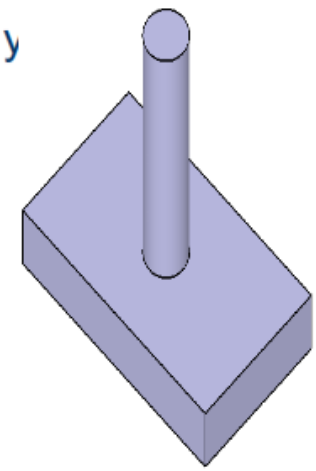
•The CloseSurface Definition dialog box

•Resulting closed body

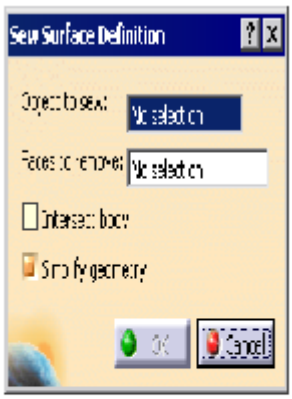


• Sewing a Surface With a Solid Body

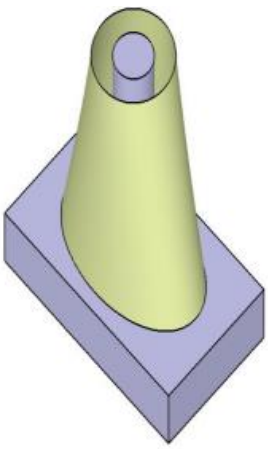
• Choose the Sew Surface button from the Surface-Based Features (Expanded) toolbar; the Sew Surface Definition dialog box is displayed, as shown in the figure.



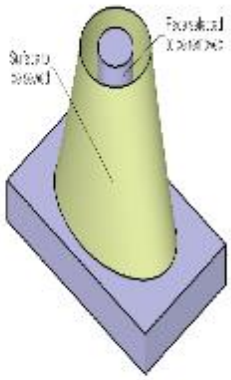
•Solid body



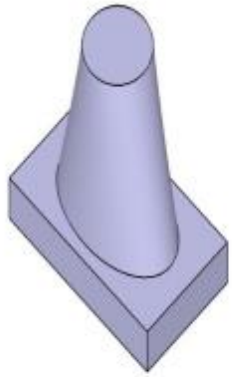
•The Sew Surface Definition dialog box



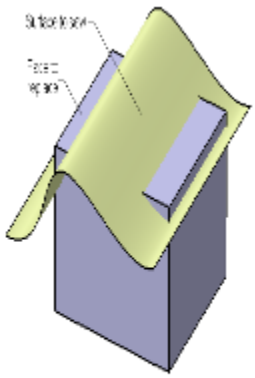
•Surface created on the solid body



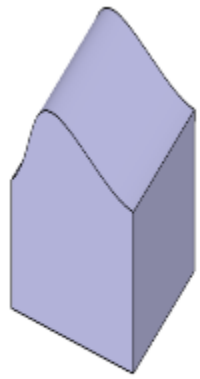
•Surface to be sewed and face to be removed



•Resulting solid body



•Surface to be sewed and face to be removed



•Resulting solid body

Chapter 10

➢ 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 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



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.

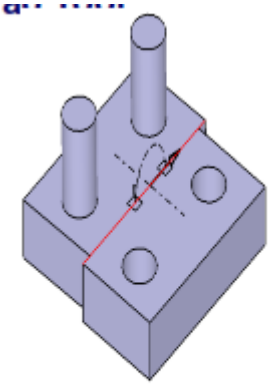
• Moving and Rotating by Using the Manipulation Tool

• 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.

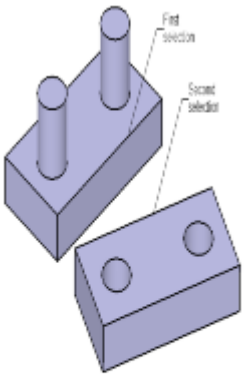


The Manipulation Parameter dialog box

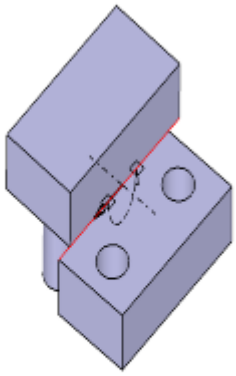
• Moving Components by Using the Snap Tool



Position of components, after snapping



Geometric elements selected to be snapped



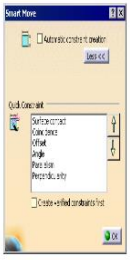
Position of components, after the snapping direction is reversed

• Moving Components by Using the Smart Move Tool

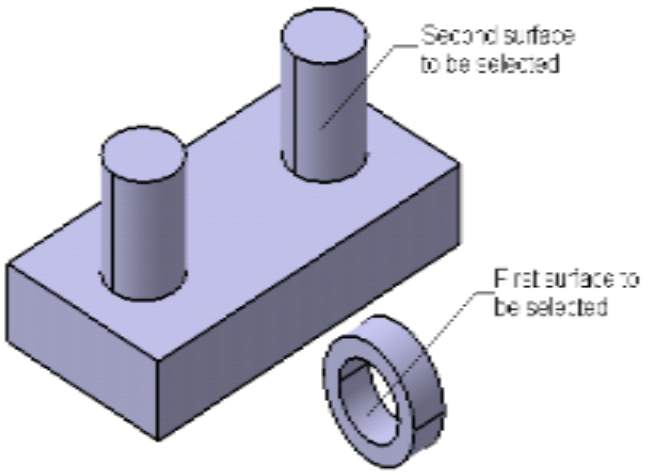
• To invoke this tool, choose the down arrow besides the **Snap** button to invoke the **Snap** toolbar.

• Choose the **Smart Move** button to invoke the **Smart Move** dialog box.

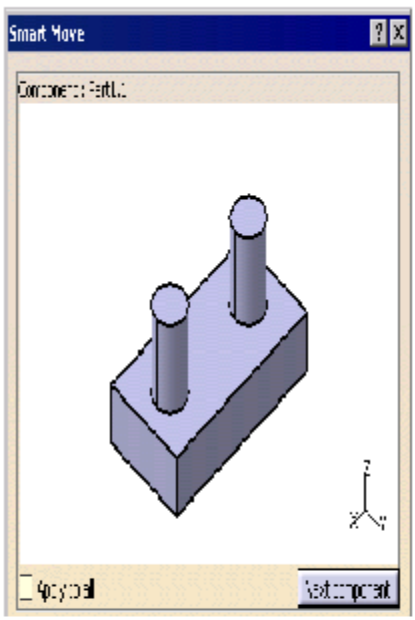
• Choose the **More** button to expand it, as shown in the figure. `



The expanded **Smart Move** dialog box



Surfaces to be selected

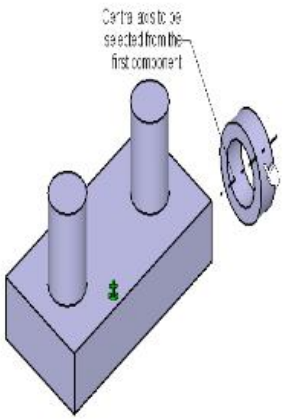


The partial view of the Smart Move dialog box with viewer

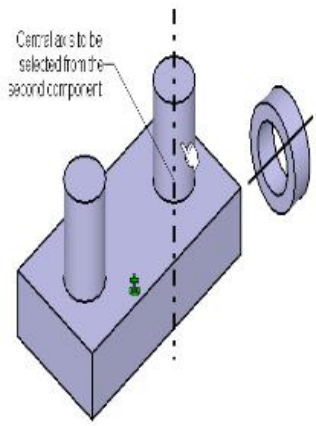
using the Compass

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

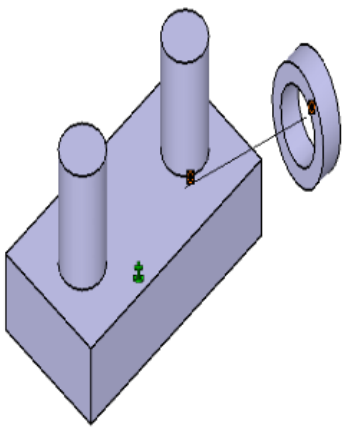
- **Applying Constraints**
- **Fix Component Constraint**
- **Coincidence Constraint**



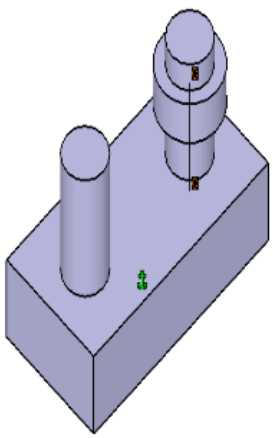
Central axis of the first component to be selected



Central axis of the second component to be selected

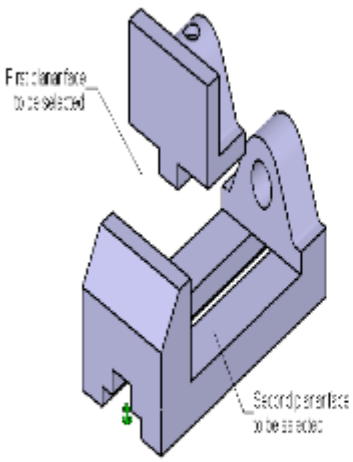


Coincidence constraint applied between two components

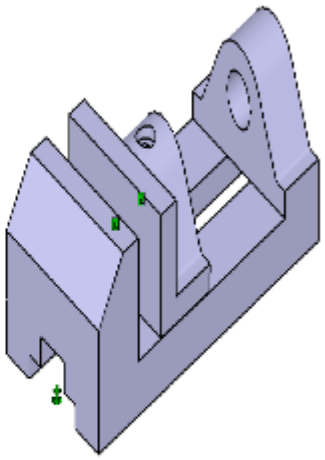


Position of the components after updating

• Contact Constraint



Planar faces to be selected



Position of components, after a constraint is applied and updated

• Offset Constraint

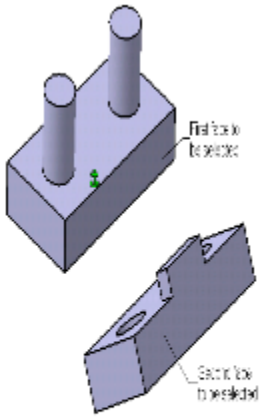
• After invoking the **Offset Constraint** tool, you are prompted to select the first geometric element for the **Offset** constraint.

• Select a planar face, circular face, plane, axis, or a point from the geometry area.

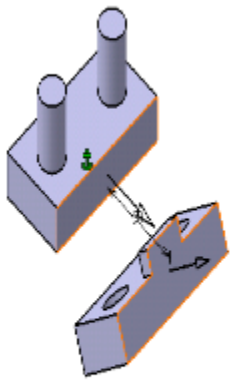
• Select a planar face of another component; the **Constraint Properties** dialog box is displayed, as shown in the figure.



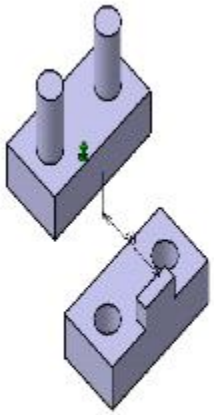
The Constraint Properties dialog box



Faces to be selected



Arrows in the same direction



Components after updating

• Angle Constraint

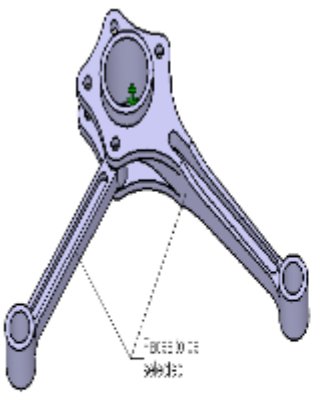
• To invoke this tool, choose the **Angle Constraint** button from the constraint toolbar.

• Now, select the two planar faces from the two different components that you need to place at some angle from each other.

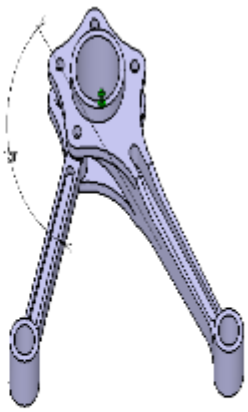
• Once the selection is complete, the **Constraint Properties** dialog box is displayed, as shown in the figure



Constraint Properties dial box for Angle constraint



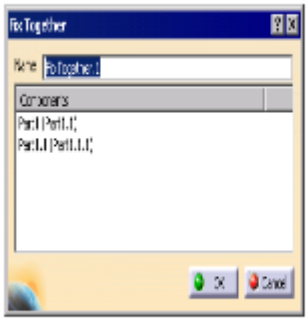
Faces to be selected



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.

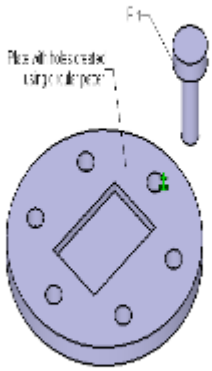


The Fix Together dialog box

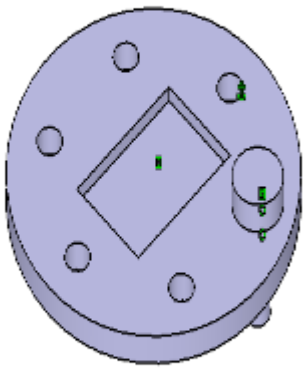
• Quick Constraint

• 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

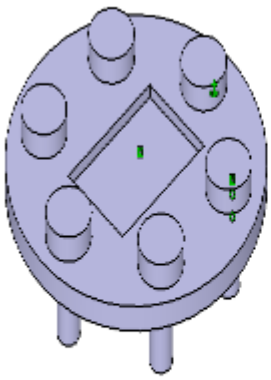
• After selecting the Coincidence constraint, choose the **Reuse Pattern** button from the **Constraints** toolbar.

• 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



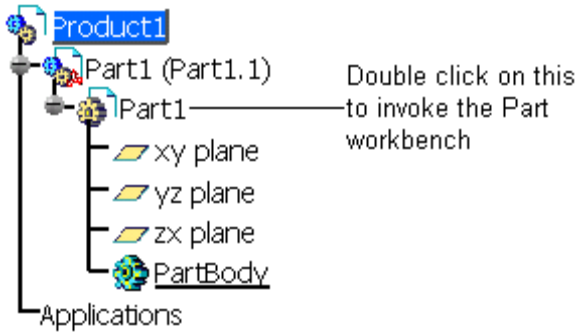
The Specification Tree after creating the component pattern

- **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

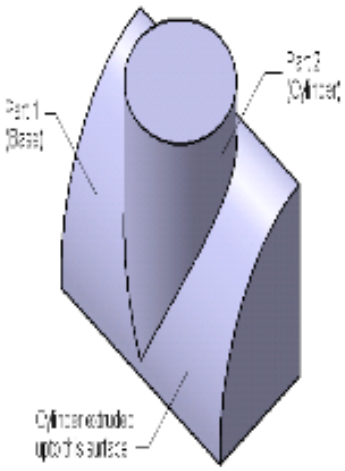
- **Creating Base Part in Top-Down Assembly**



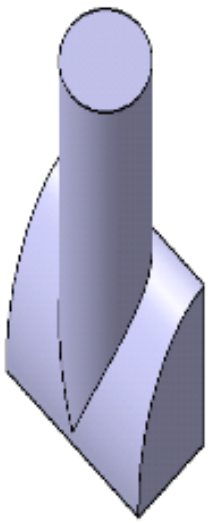
The fully expanded

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**

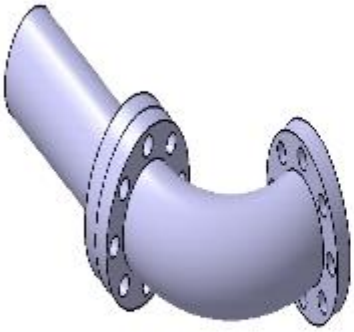
• While working in the **Assembly Design** workbench, you may need to delete some of the constituent parts and subassemblies.



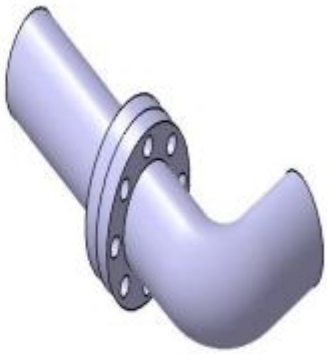
The Delete dialog box

- **Replacing Components**

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



The original component

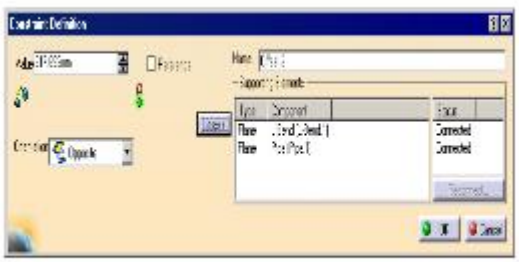


The replaced component

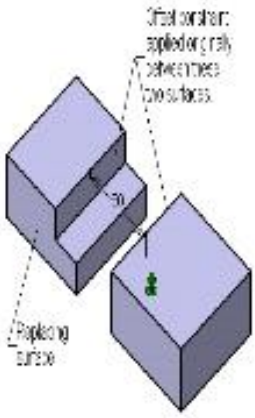
- **Editing Components Inside an Assembly**

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

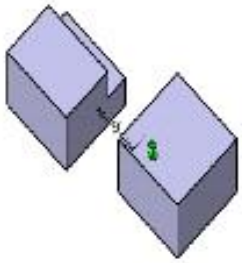
- **Editing Subassemblies Inside an Assembly**
- **Editing the Assembly Constraints**
- **Editing the Constraint Definition**



The Constraint Definition dialog box

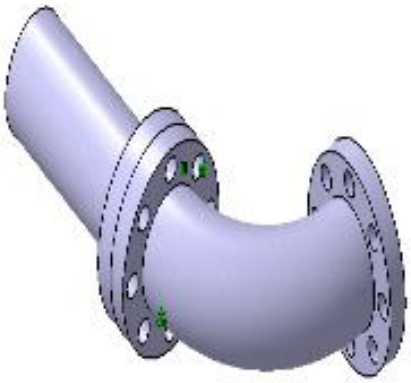


The associated and replacing surface for the **Offset** constraint

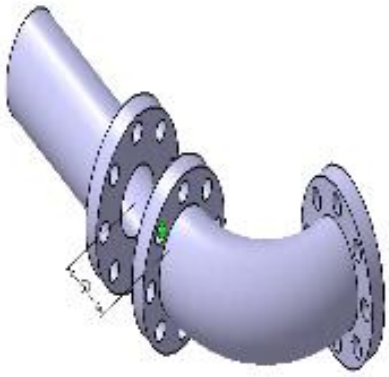


Components after editing the offset constraint and updating it

- **Replacing a Constraint**



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



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.



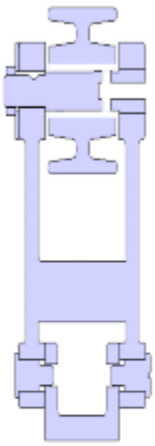
The Check Clash dialog box



The Preview window

• Sectioning an Assembly

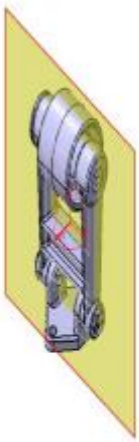
• To section an assembly model, choose the **Sectioning** button from the **Space Analysis** toolbar; the **Sectioning Definition** dialog box is displayed, as shown in the figure.



The 2D section view of the complete assembly



The Sectioning Definition dialog box



The sectioning plane

- **Exploding an Assembly**

• To explode an assembly, from the **Move** toolbar; the displayed, as shown in the



The Belt Tightener in the assembled state



Overlapping components in 3D explosion of the assembly

choose the **Explode** button **Explode** dialog box will be figure.



The Explode dialog b



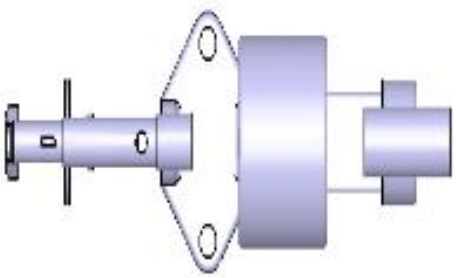
Figure showing the exploded assembly with Constrained selected as the type



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

Chapter-11

➢ 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 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 is displayed; select **Drawing** and choose the **OK** button.

• The **New Drawing** dialog box is displayed, as shown in the figure.



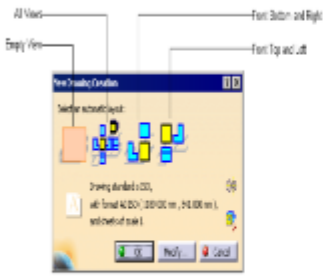
The options available in the **New Drawing** dialog box are:

- **Standard**
- **Format**
- **Orientation**

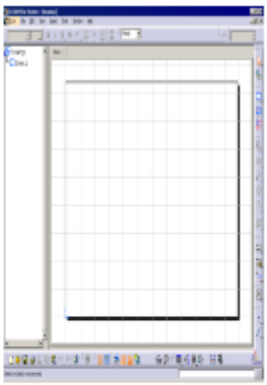
- **Starting a New File in the Drafting Workbench Using the Start Menu**

• 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



Initial screen after starting a new file in the Drafting workbench

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**

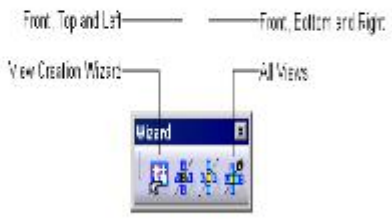
- **Broken View**

- **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.

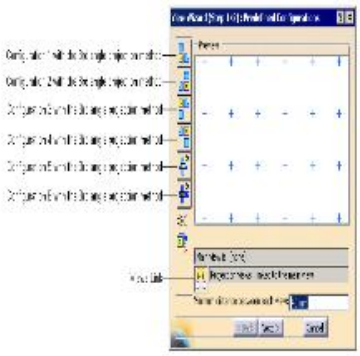
- **Automatically Generating the Views**



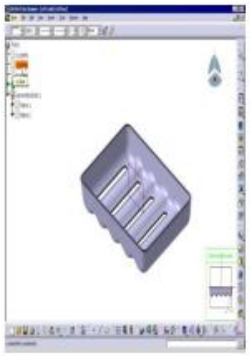
The Wizard toolbar

- **Generating the Views Using the View Creation Wizard**

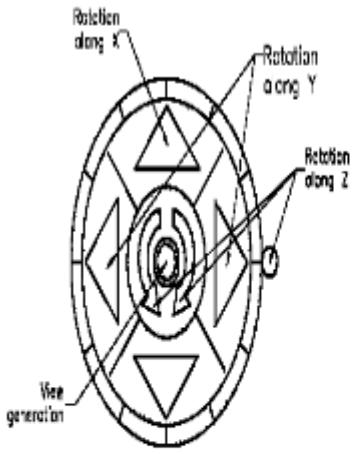
• When you invoke the **View Creation Wizard** tool, the **View Wizard** dialog box is displayed, as shown in the figure.



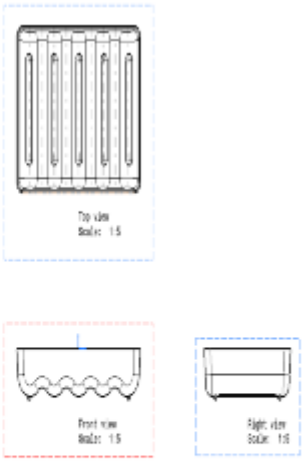
The View Wizard dialog box



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



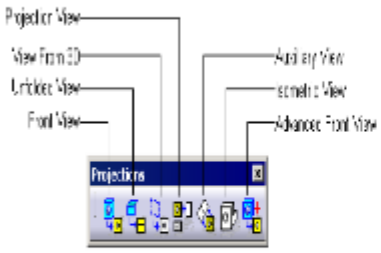
Controls provided on the blue knob



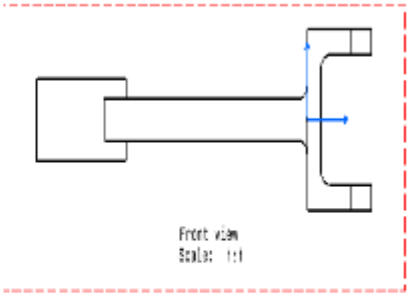
Views generated using the View Creation Wizard dialog box

- 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



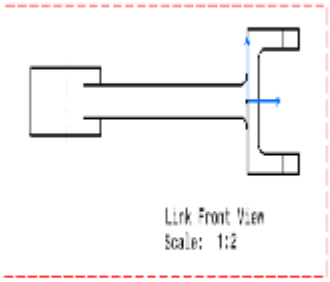
The front view generated using the Front View tool

• 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.

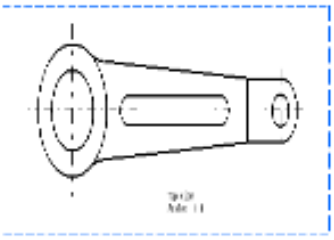


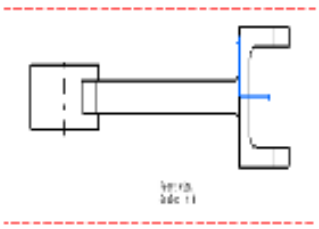
The View Parameters dialog box



The front view generated using the Advanced Front View tool

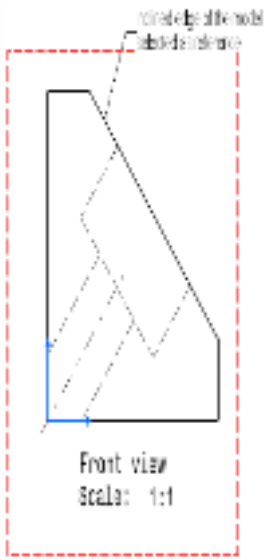
- **Generating Projected Views**



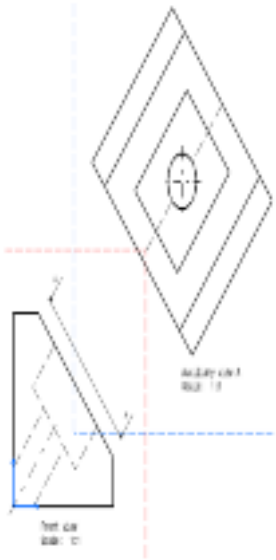


Top view generated from the front view using the Projection View tool

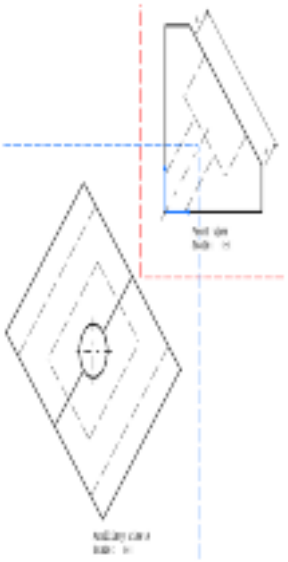
- **Generating Auxiliary Views**



Reference to be selected to generate the auxiliary view

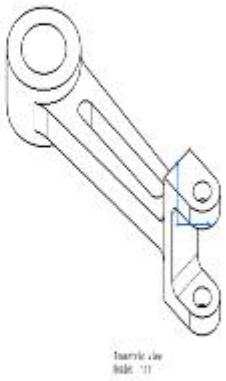


Auxiliary view placed above the parent view



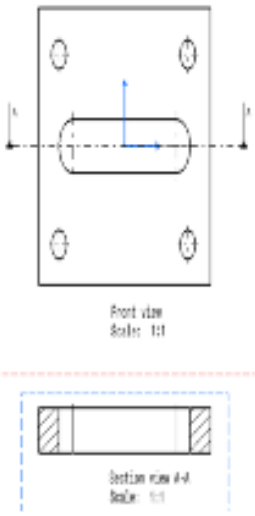
Auxiliary view placed

- **Generating Isometric Views**

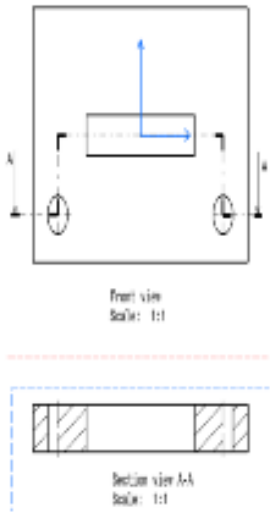


Isometric view generated using the Isometric View tool

- Ger

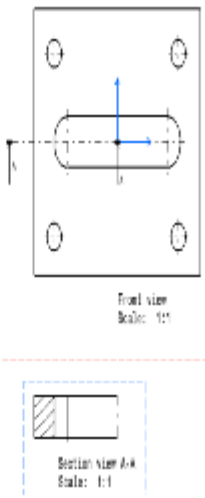


Full section view

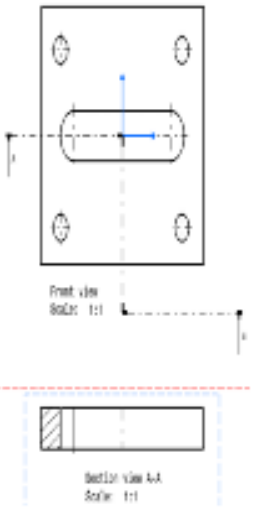


Offset section view

- **Creating the Partial Section View**

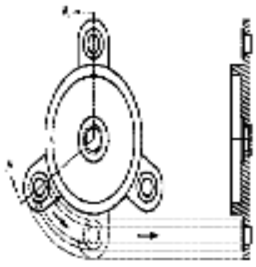


The partial section view

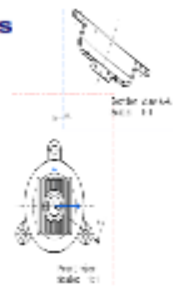


Half section view

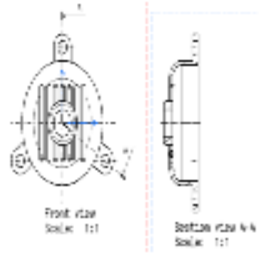
- **Generating the Aligned Section Views**



Aligned section view

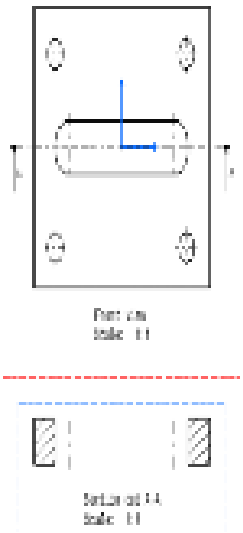


Aligned section view with inclined line drawn first



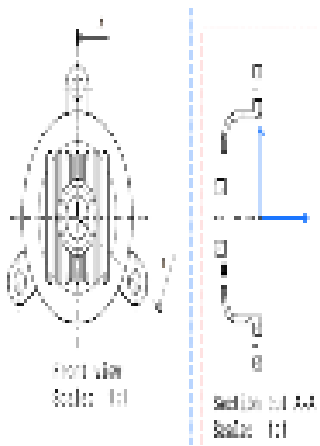
Aligned section view with vertical line drawn first

- **Generating the Offset Section Cut**



Offset section cut

- **Generating the Aligned Section Cut**

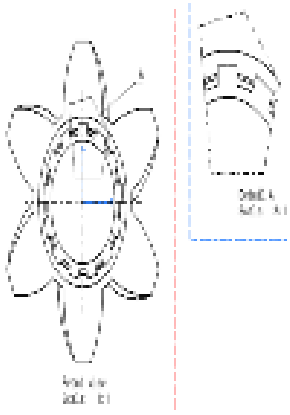


Aligned section cut



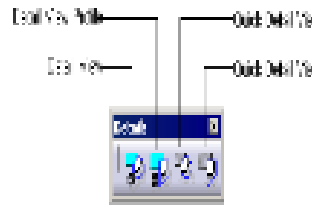
Detail view

- Generating the Detail View Profiles



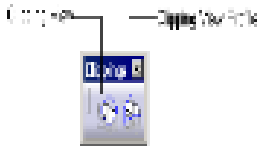
Detail view profile

- Generating Detail Vie

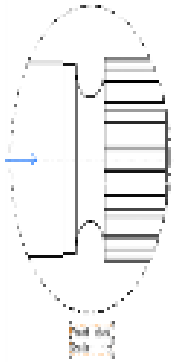


The Details tool

- Generating Clipping Views

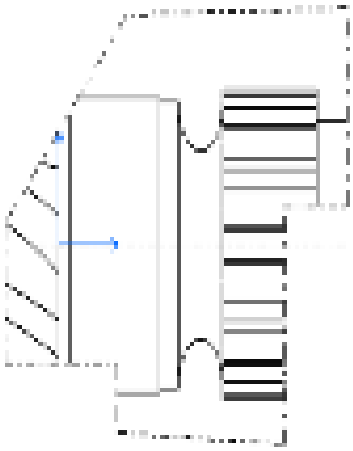


The Clippings toolbar



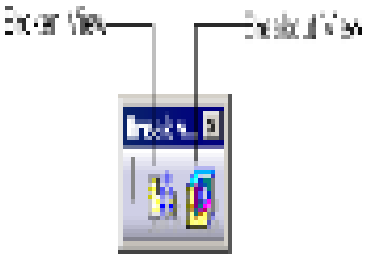
Clipping view

- **Generating the Clipping View Profiles**

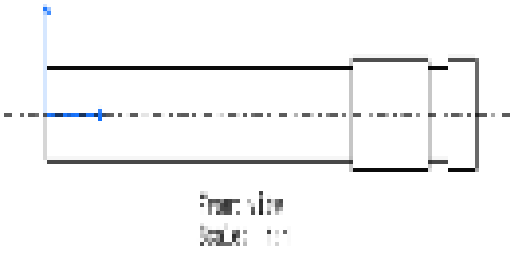


Clipping view profile

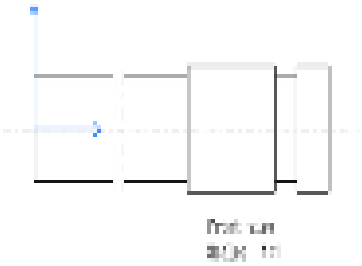
- **Generating the Broken View**



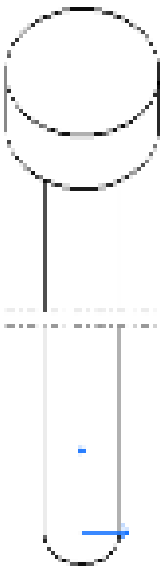
The Break view toolbar



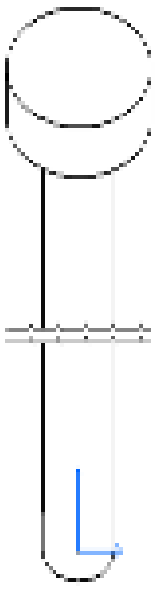
The front view that needs to be broken



Resulting broken view

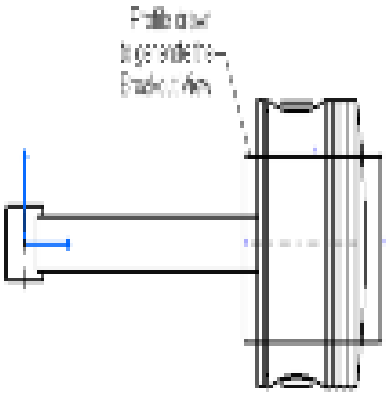


A broken isometric view

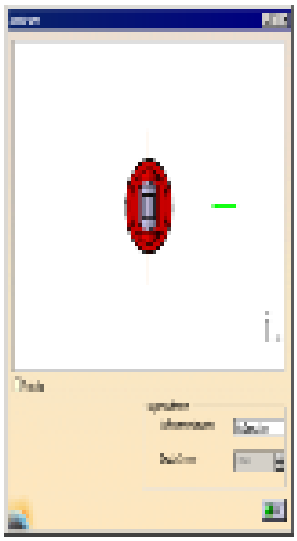


Zigzag broken lines

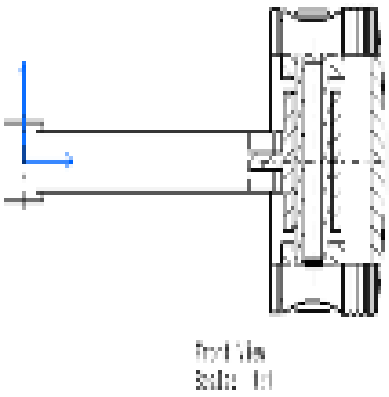
- **Generating the Breakout View**



Zigzag broken lines



The 3D Viewer window



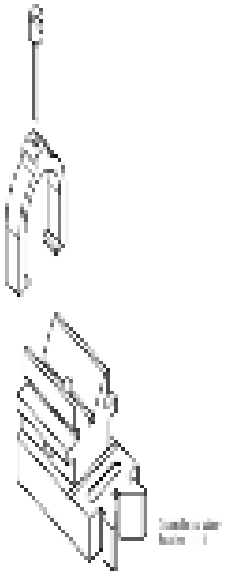
Breakout view

- **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 various types of editing operations and modifications on the drawing views.

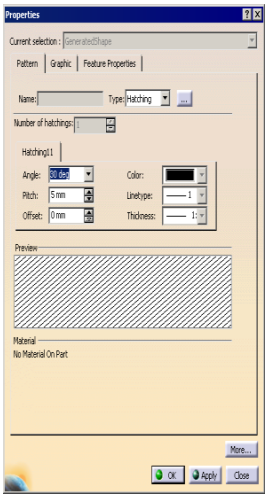
The various options under the **Editing and Modifying**

Drawing Views option are:

- **Changing the Scale of Drawing Views**
- **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



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

Chapter-12

➢ 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.

➢INSERTING THE FRAME AND TITLE BLOCK

• Automatic Insertion of the Frame and Title Block

• Choose the **Frame Creation** button from the **Drawing** toolbar.

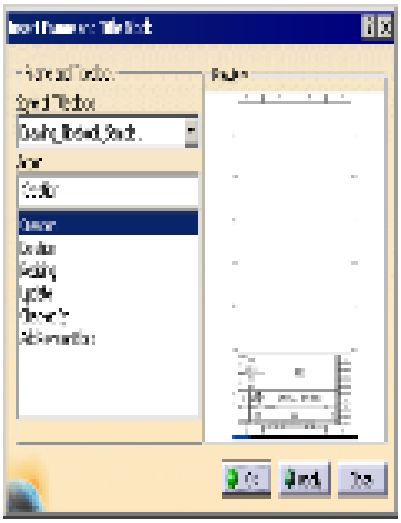
• The **Insert Frame and Title Block** dialog box is displayed, as shown in the figure.

- Style of Titleblock
- Action
- Creation



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

- Deletion
- Resizing
- Update



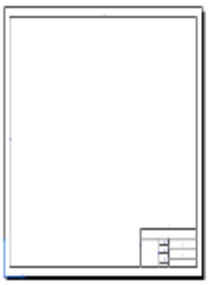
The Insert Frame and Title Block dialog box

- Checked By



The Controller's name dialog box

- Creating the Frame and Title Block Manually



Drawing sheet after drawing the frame and title block



The Text Editor dialog box

- Inserting the Logo



Drawing sheet after completing the frame and title block

• Adding Text in the Title Block

• Select a point on the drawing sheet, where you need to place the text.

• The **Text Editor** dialog box is displayed, as shown in the figure.

➢ ADDING ANNOTATIONS TO THE DRAWING VIEWS

• After generating the drawing views, you need to generate thedimensions 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 thenchoose 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.



Dimension Generation Filter dialog box

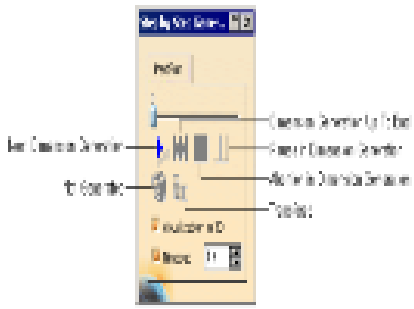
The options available in the **Dimension Generation Filter** dialog box are:

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

- **Generating Dimensions Step by Step**

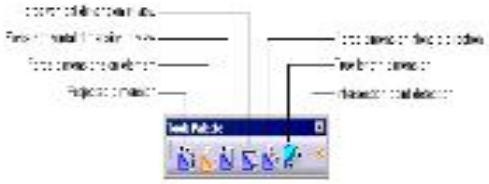
• If the **Dimension Generation Filter** dialog box is displayed,set the option in this dialog box and choose the **OK** button to exit it.

• 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

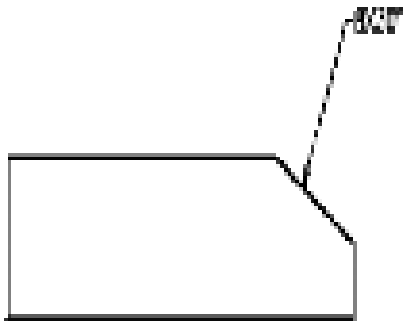
- **Creating the Chamfer Dimensions**



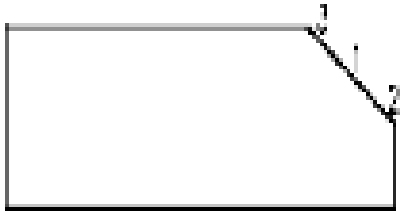
The Tools Palette toolbar



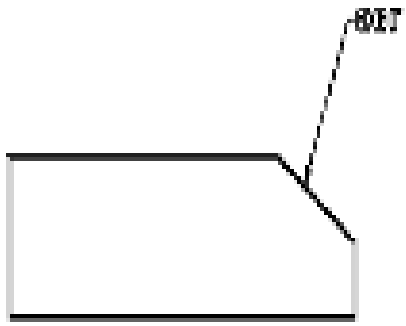
Selection sequence



Resulting chamfer dimension



Selection sequence



Resulting chamfer dimension

Adding Datum Features

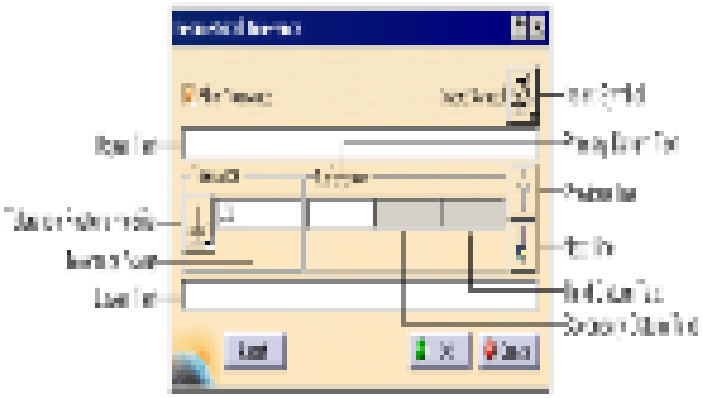
• Move the cursor to the desired location and click a point on the drawing sheet to place the datumfeature; the **Datum Feature Creation** dialog box is displayed, as shown in the figure.



The Datum Feature Creation dialog box

Adding Geometric Tolerance to the Drawing Views

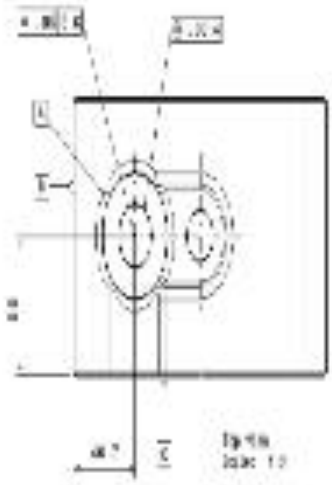
• Move the cursor to an appropriate location and click on the drawing sheet to place the tolerance on that location; the **Geometrical Tolerance** dialog box is displayed, as shown in the figure.



The Geometrical Tolerance dialog box

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

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



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**.

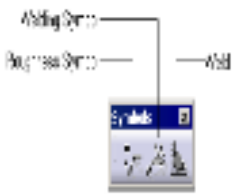


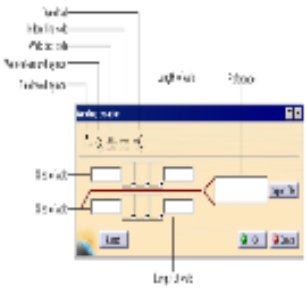
Figure A The Symbols toolbar



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 the preview of the welding symbol; the **Welding creation** dialog box is displayed, as shown in the figure.



The Welding creation dialog box



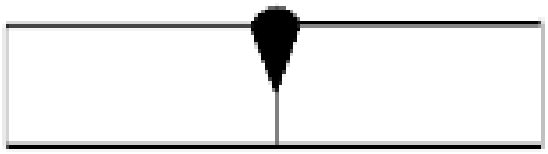
Drawing view after adding the welding symbol

• Applying Weld

• Select the second edge from the drawing sheet; the **Welding Editor** dialog box is displayed, as shown in the figure.



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.

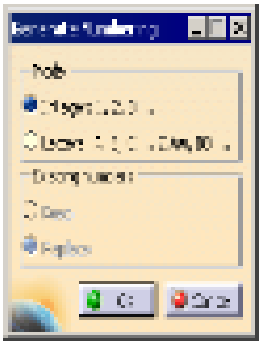
➢ GENERATING THE BI

➢LL OF MATERIAL (BOM)

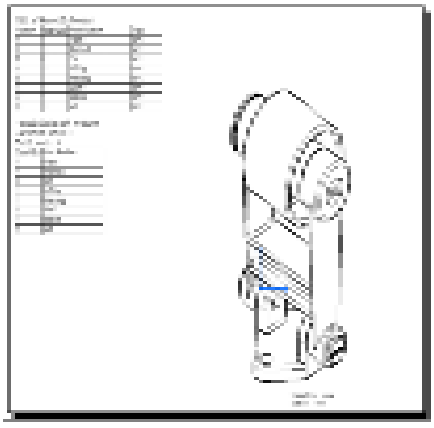
• To number the components, switch to the assembly file from which the drawing views are generated.

• 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.

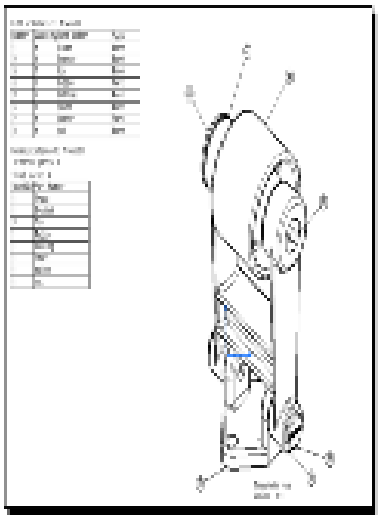


The Generate Numbering dialog box



Drawing sheet after generating the BOM

GENERATING BALLOONS



Drawing sheet after generating balloons