

# **Drafting**

## **Drafting Workbench**

# What is Drafting

- **Generative Drafting workbench automatically generates the views of geometry from the 3D as well as dimensions**
- **A drawing can be generated at any time. If the part or product is changed, the drawing will be automatically updated because of the associative link.**

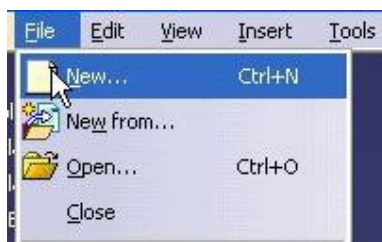
# How to enter Drafting workbench.

**1. Select Start -> Mechanical Design -> Drafting from the menu bar.**



**Or**

**2. Select File -> New and select Drawing in List of Types.**

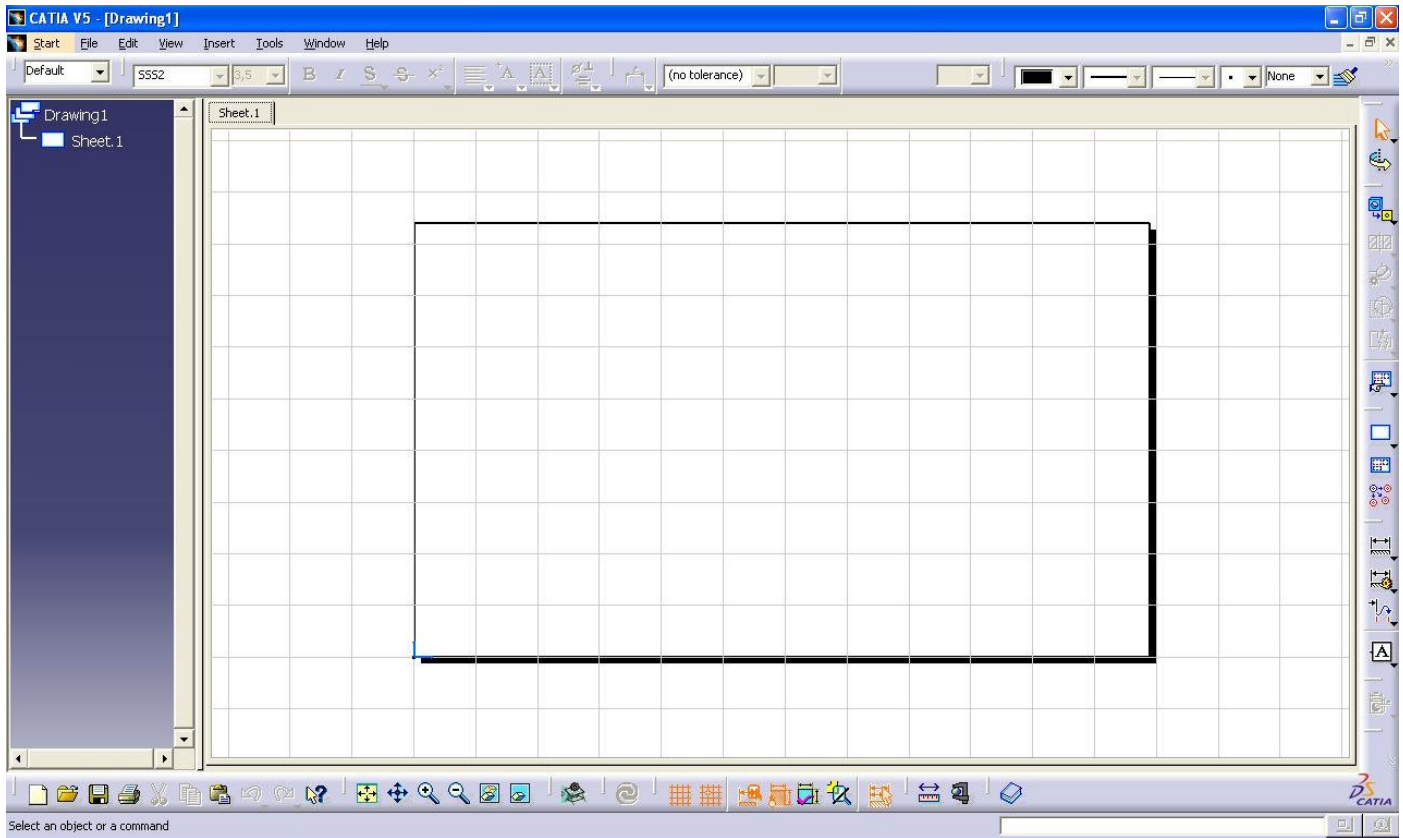


**The New Drawing dialog box appears. Then choose Standard and Sheet Style.**



# Drawing interface

The Drawing workbench looks as follows (Fig. 3)

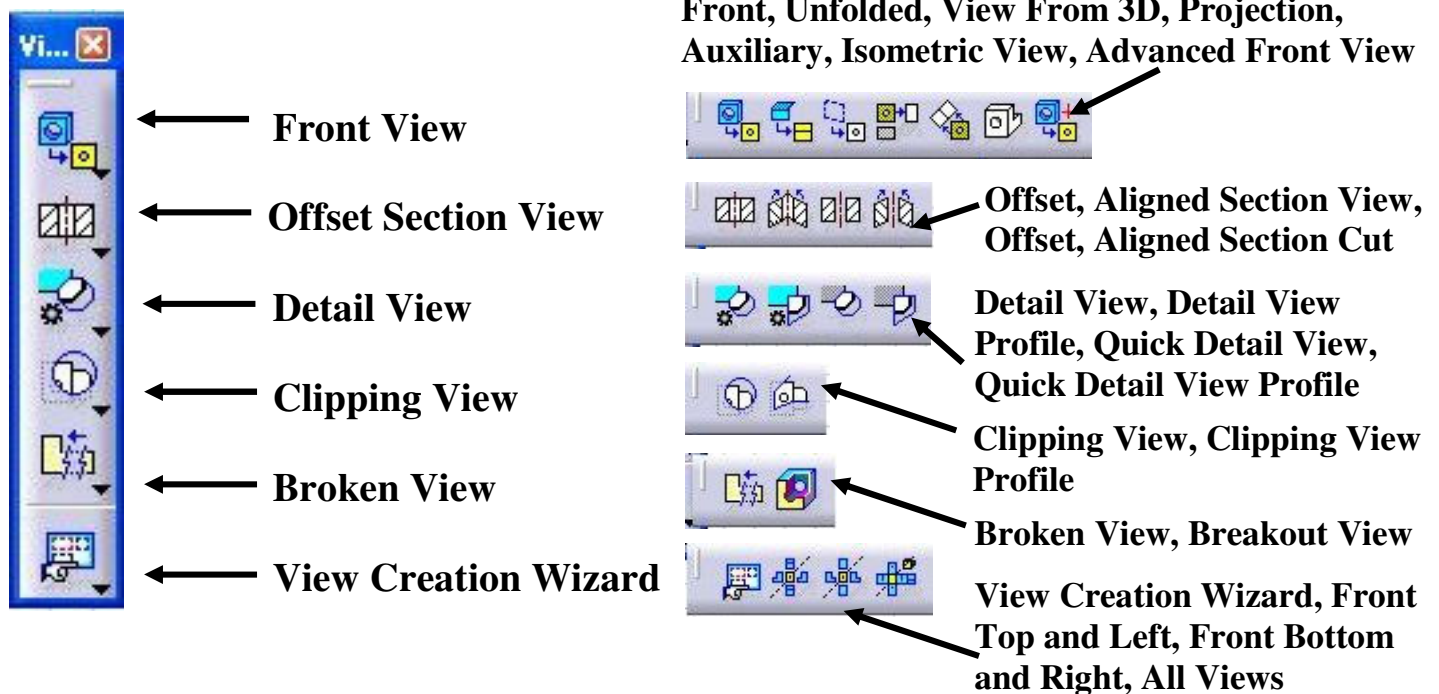


**Fig. 3. The Drawing workbench.**

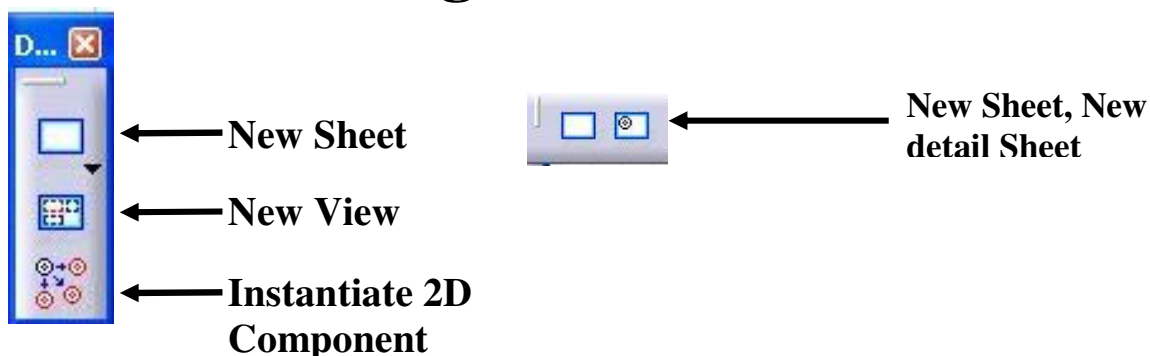
# Drafting Tool Bars

There are four standard tool bars found in Drafting Workbench. The four tool bars are shown below. Some tools have an arrow located at the bottom right of the tool icon. To display the other tool options you must select and hold the left mouse button on the arrow.

## 1. The Views Tool Bar



## 2. The Drawing Tool Bar



### 3. The Annotations Tool Bar



Text

Roughness Symbol

Table

Text, Text with Leader, Text Replicate, Balloon, Datum Target, Text Template Placement



Roughness Symbol, Welding Symbol, Weld



Table, Table from CSV



### 4. The Dimensioning Tool Bar

Dimensions, Chained, Cumulated, Stacked, Length/Distance, Angle, Radius, Diameter, Chamfer, Thread and Coordinate Dimensions, Hole Dimension Table, Coordinate Dimension Table



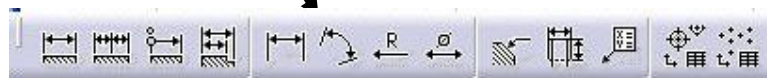
Dimensions

Technological Feature Dimensions

Re-route Dimension

Datum Feature

Datum Feature, Geometrical Tolerance



Technological Feature Dimensions, Multiple Intra, Chained, Length, angle, Radius and Diameter Technological Feature Dimensions

Re-route Dimension, Create and Remove Interruption(s)

# Generative Drafting

## Creating Views

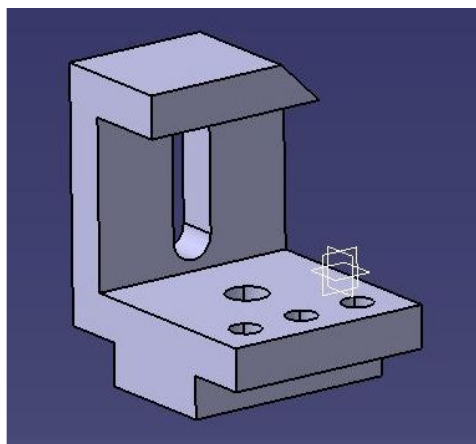
**Views can be created in various ways:**

- **Individually one by one**
- **View Wizard**

**Example No. 1. Individually one by one**

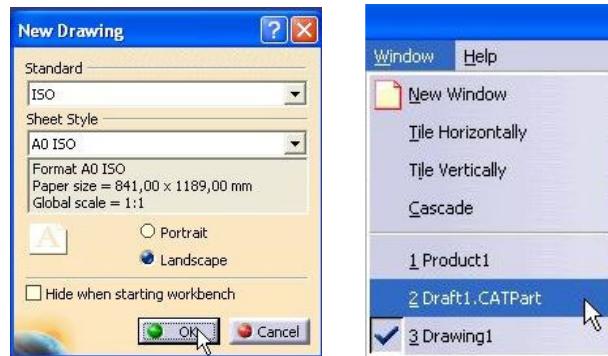
**Many types of views can be created one by one in an approach.**

**1. Open Draft1 file.**

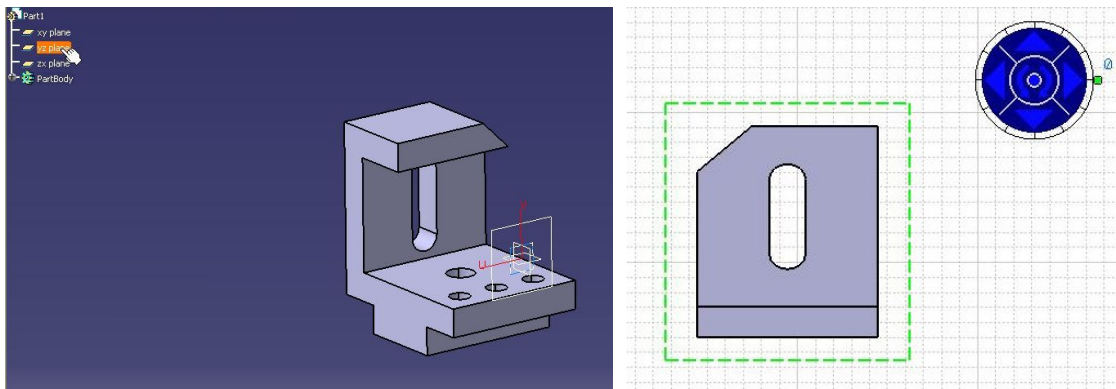


**2. Then open Drafting Workbench and in Standard options select ISO standard and in Sheet Style choose A0 ISO format. Then select**

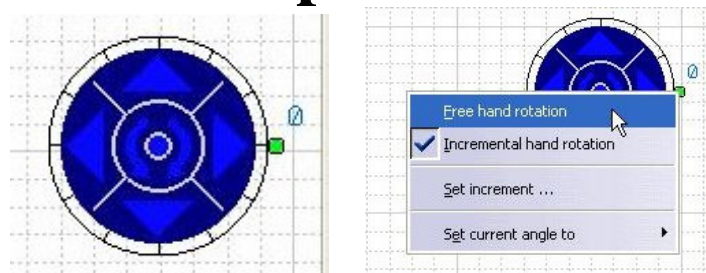
**The Front Views icon . Then activate the CATPart (Draft1) document.**



**3. Select your front view plane (select yz plane). A preview will be shown on the drawing sheet.**



**4. Before accepting the view you can reorient it with the view manipulator.**

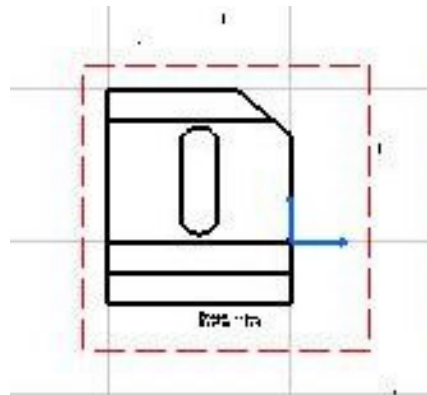



**Up, down right and left arrows flip background plane view about 90 deg.**

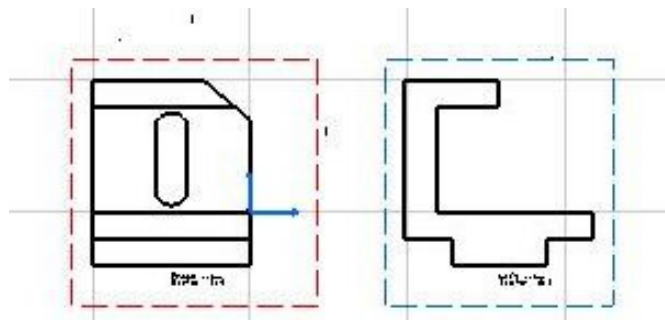


**Center left and right arrows rotate view 30 deg about the same plane. The 30 deg increment can be changed with the contextual menu on the dial.**

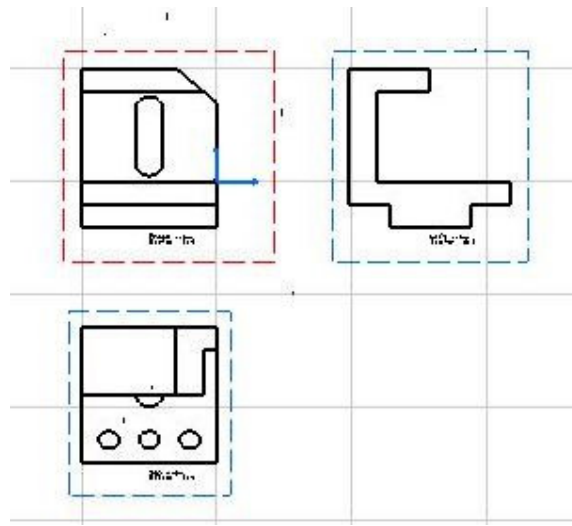
**5. Click two times on right arrow and click on dial center or on sheet to generate the front view.**



**6. Then add a new view to this drawing. Activate the front view and select the Projection View icon . Then move the drawing where you want it to be projected. Select desired location to place the view.**




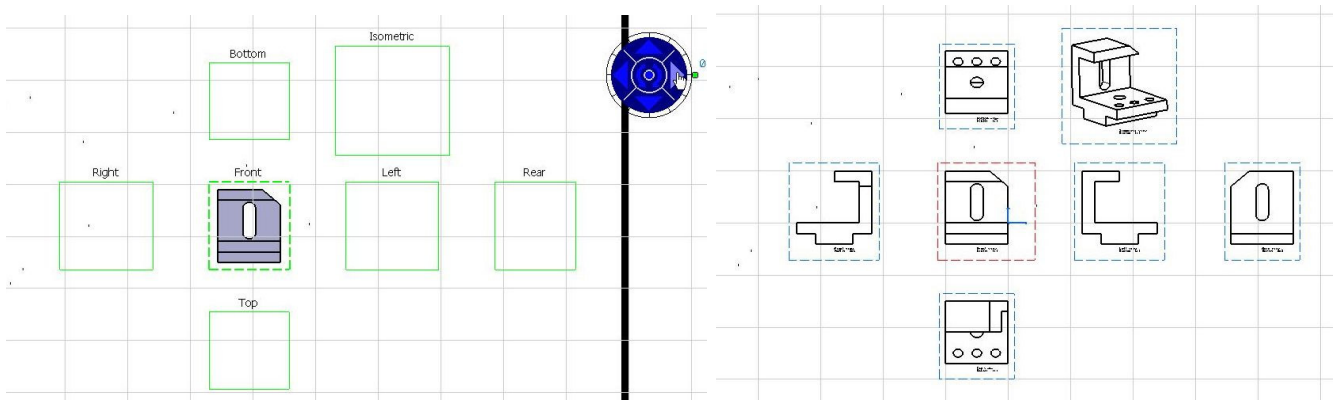
**Generate another view (repeat the procedure).**



## **Example No. 2. The View Wizard**

**The View Wizard provides with the ability to quickly create a variety of standard view configurations or build a specific view configuration.**

**1. Select the All Views icon . Then activate the CATPart (Draft1) document and select “yz plane”. Rotate front view about 180deg on left and click on dial center or on sheet to generate the views**

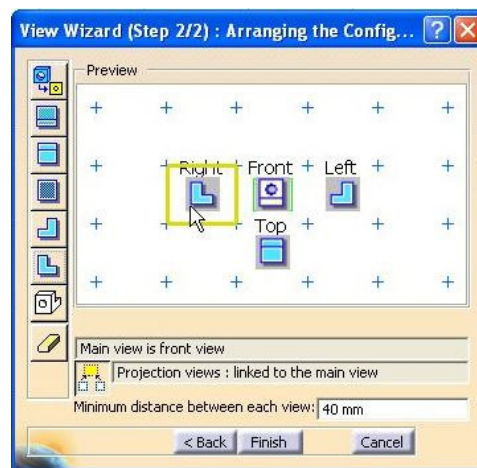
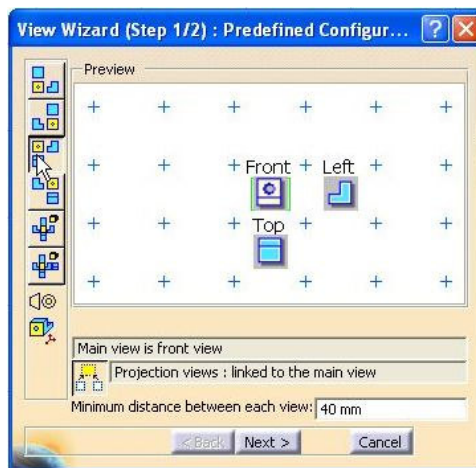


**You can choose different configuration views.**

### **Example No. 3. View Creation Wizard.**

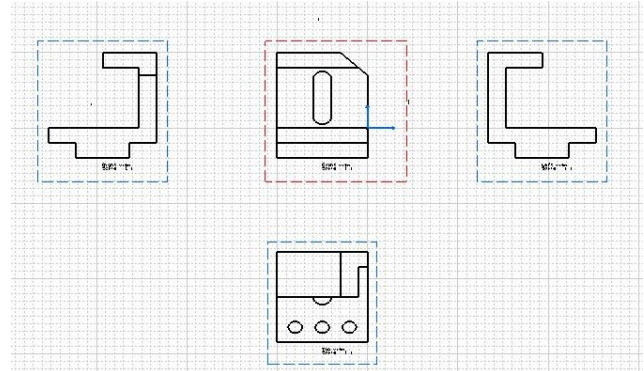
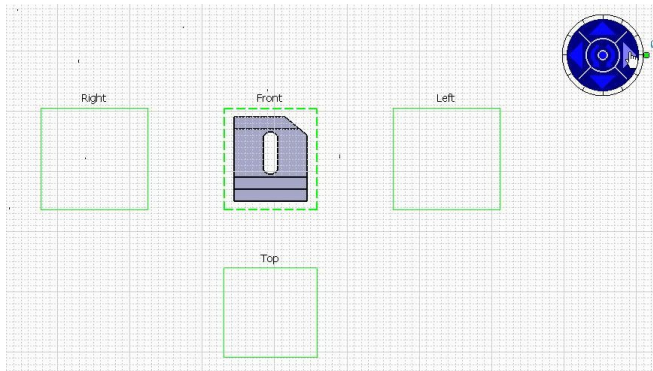
**1. Select the View Creation Wizard icon** .

**The View Wizard: Predefined Configurations dialog box appears.**



**2. Select one of the view configurations and the Next button for additional views. The View Wizard: Arranging Configurations dialog box appears. Then, select and place any additional views (Right view) to the existing**


**view configuration and confirm by clicking Finish button. Then activate the CATPart (Draft1) document and select “yz plane”. Rotate front view about 180deg on the left and click on dial center or on sheet to generate the views.**



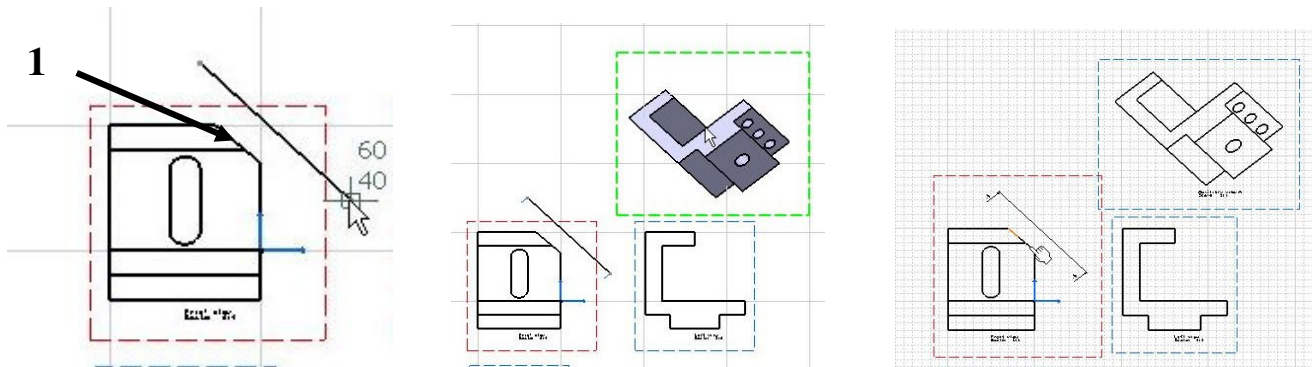
**Active view is in red frame (for active view you can generate additional views). Double-click on the blue frame to activate it or use contextual menu.**

## **Additional View Generation**


### **Example No. 4. Auxiliary View.**

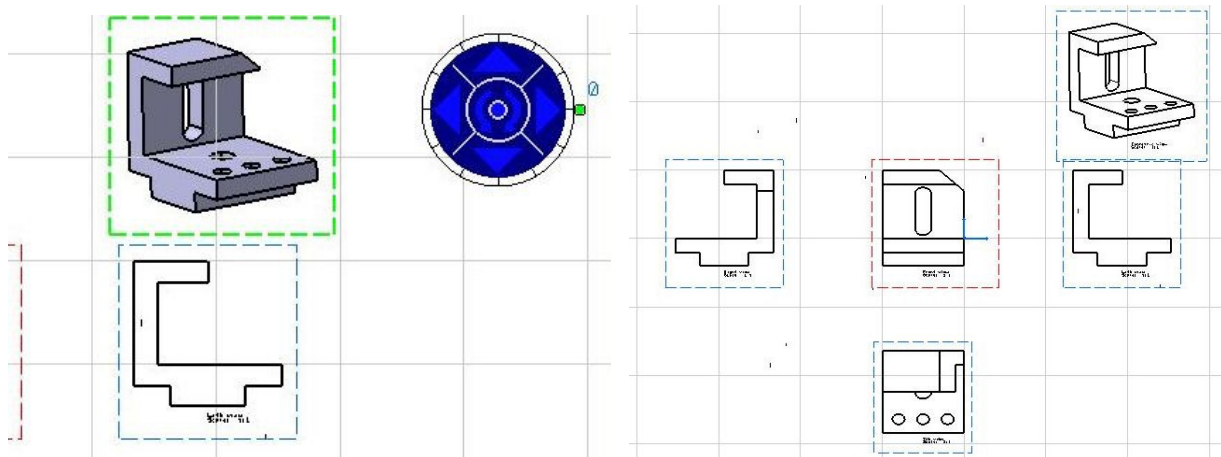
**1. Activate the front view and select the Auxiliary View icon . Then sketch the representation of the plane or select an edge (1) on the drawing and drag the mouse to see**

**the preview of auxiliary view. Then select a place on the drawing to create the auxiliary view.**



## **Example No. 5. Isometric View.**

**1. Select the Isometric View icon . Then, activate the CATPart (Draft1) document and select a face of the part. A preview of the isometric view displays. You can rotate it (use the View Manipulator). Then select a place on the drawing to create the isometric view.**





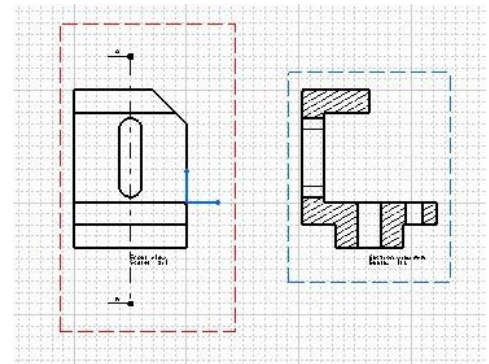
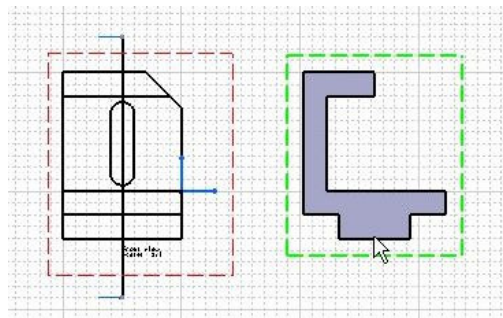
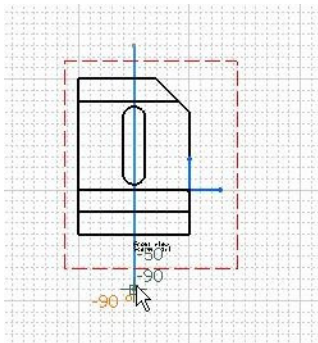
# Sections Views

There are three main types of sectional views: full section, offset or aligned depending on how the cutting plane is defined.

## Example No. 6. Offset Section View.

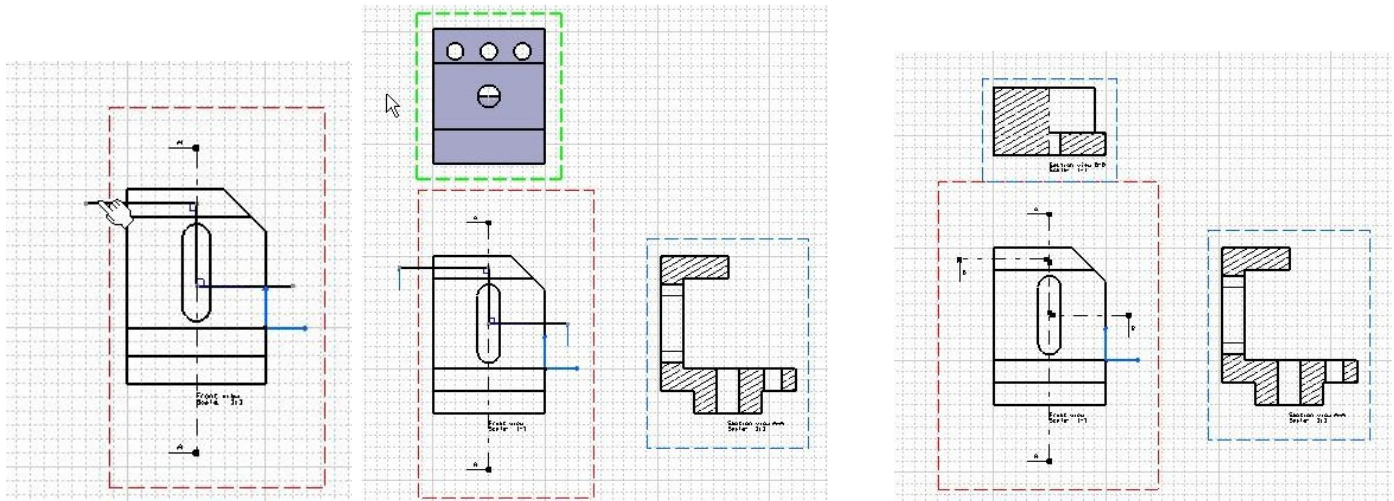
a) Simple offset section – a full section with a simple single plane fully passing through the part.

1. Select the Offset Section View icon . Then sketch cutting line and select a place on the drawing to create the offset section view.



b) Complex offset section – a full section in which the cutting plane is offset to pass through important features

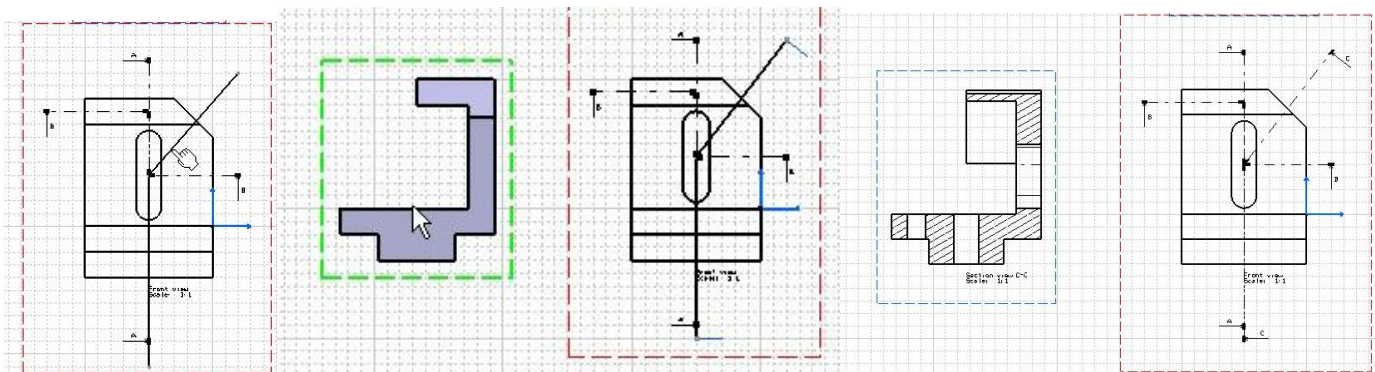
**2. Select the Offset Section View icon . Then sketch cutting line and select a place on the drawing to create the offset section view.**



## **Example No. 7. Aligned Section View.**


**Aligned section has a cutting plane revolving around an axis**

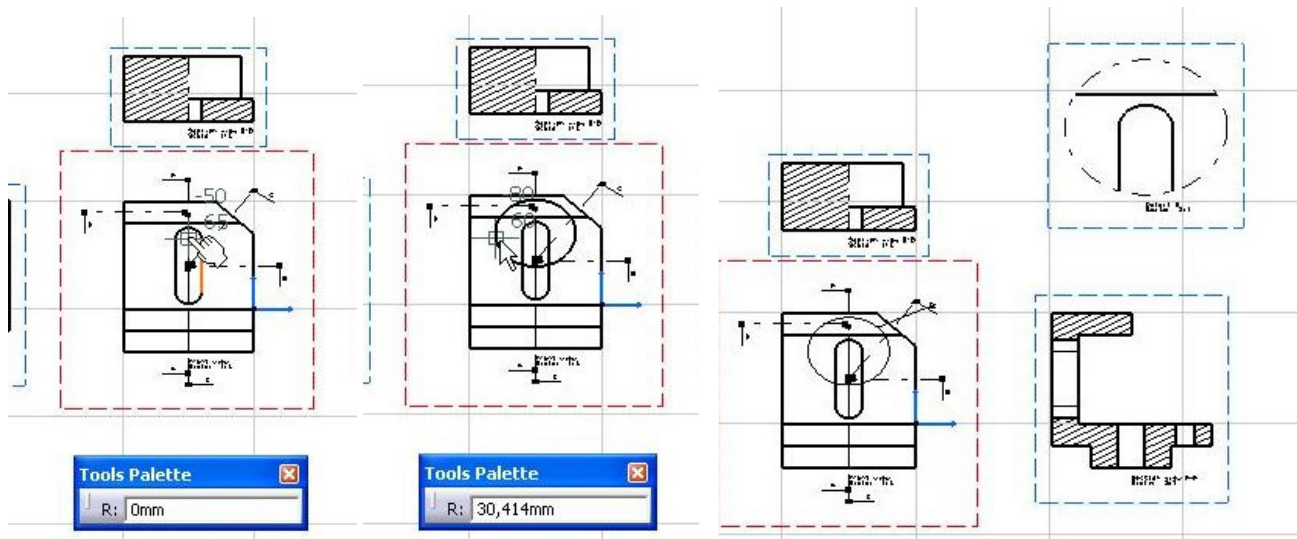
**1. Select the Aligned Section View icon . Then sketch cutting line and select a place on the drawing to create the offset section view.**



## **Example No. 8. Detail View.**

**Detail views can be fully detailed views that are calculated from 3D part or quick detailed views that are calculated from the defining view.**

**1. Select Quick Detail View icon . Then define the center of the circle by clicking on the front view and click to define the circle radius, then, move mouse to place the detail view.**



**Detail can be defined by circular perimeter or profile perimeter (Quick Detail View Profile ).**

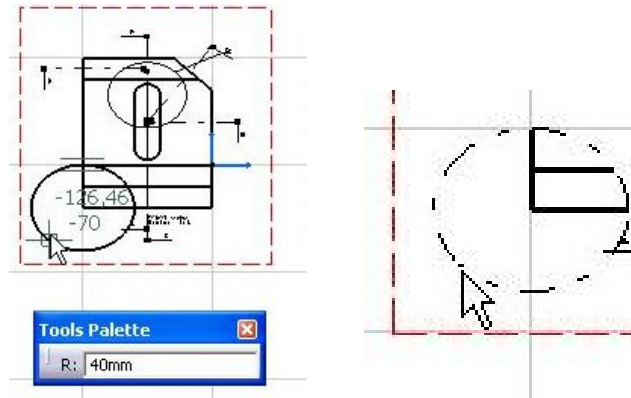
**Example No. 9. Clipping View.**

**Clipping view can be defined by circular perimeter or profile perimeter. Clipping will**



**remove all the existing view's geometry that is not in the “callout.”**

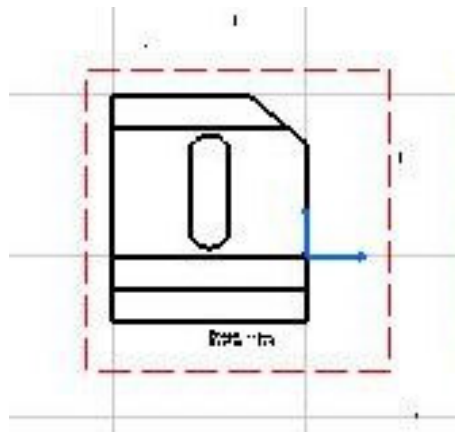
**1. Select Clipping View icon . Then define the center of the circle by clicking on the front view and click to define the circle radius. The result is displayed.**



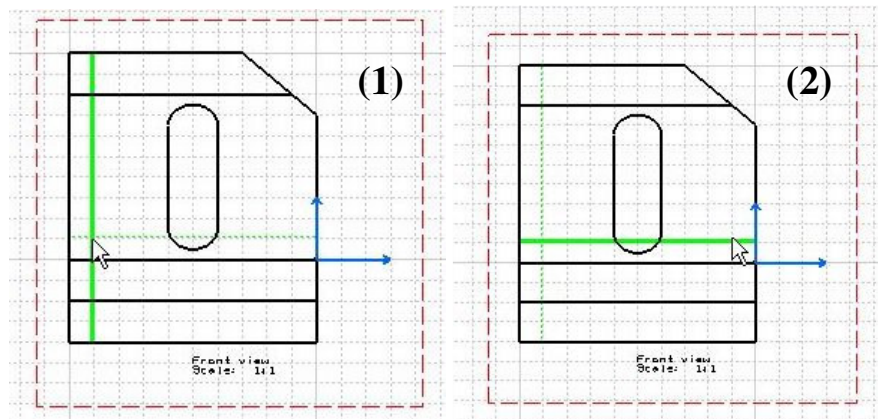
## **Example No. 10. Broken View.**

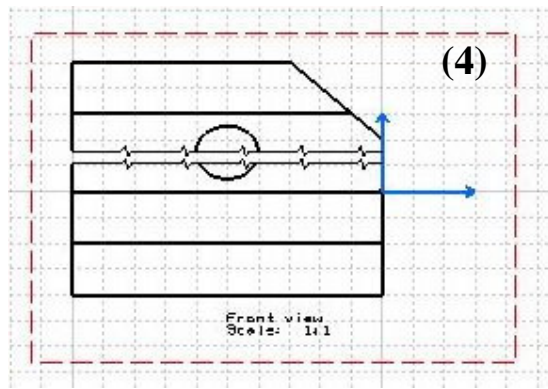
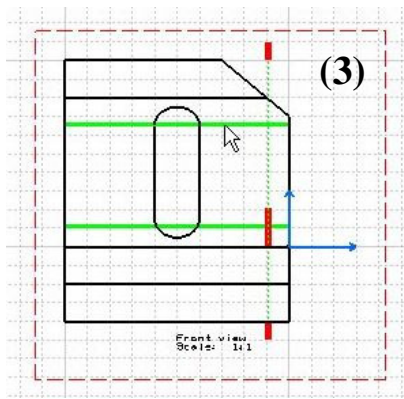
**Broken view allows for creation of local cut (by a plane) in order to see the inside of part without cutting it totally.**

**1. Select New Sheet icon  and generate new front view.**



**2. The select Broken View icon . Then, define the break out area by clicking the location for the first break limit line (select point inside the view) (1), click on the desired area to get the vertical break out (2). Then, select a point inside the view to indicate the position of the second breakout line (3) and click on the sheet to generate the broken view (4).**







## Modifying Section and Detail Profiles

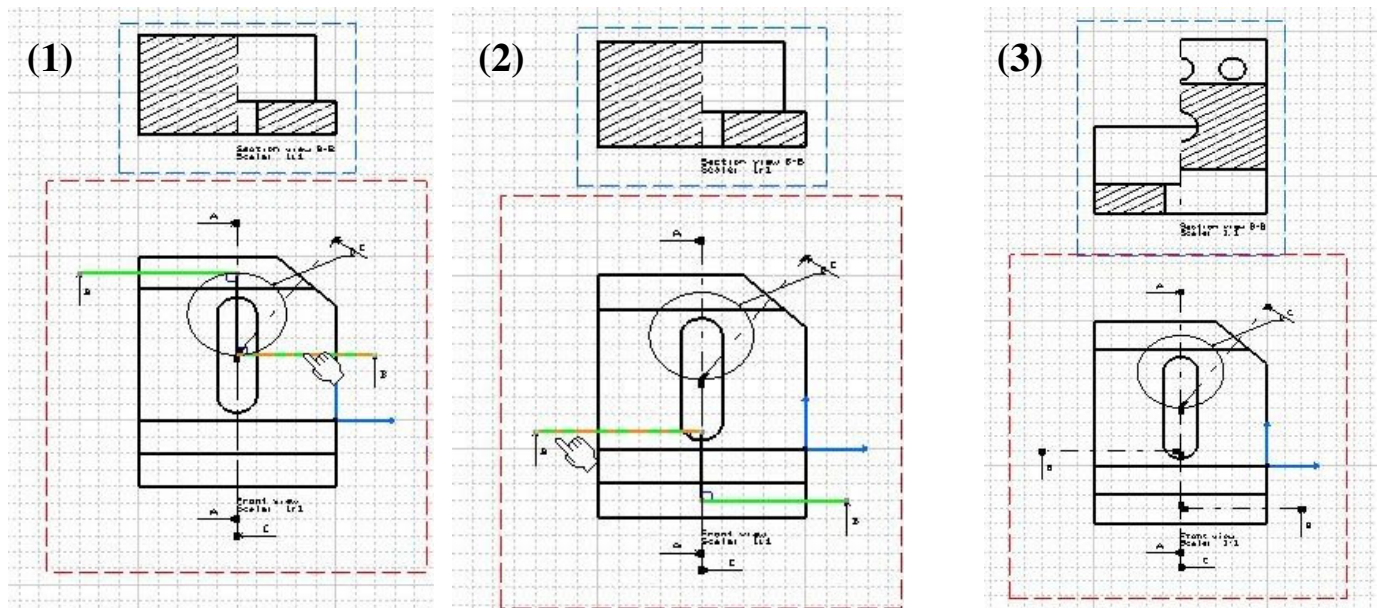
You can modify section and detail profiles at every time.




### Example No. 11. Modifying Section View Profile.

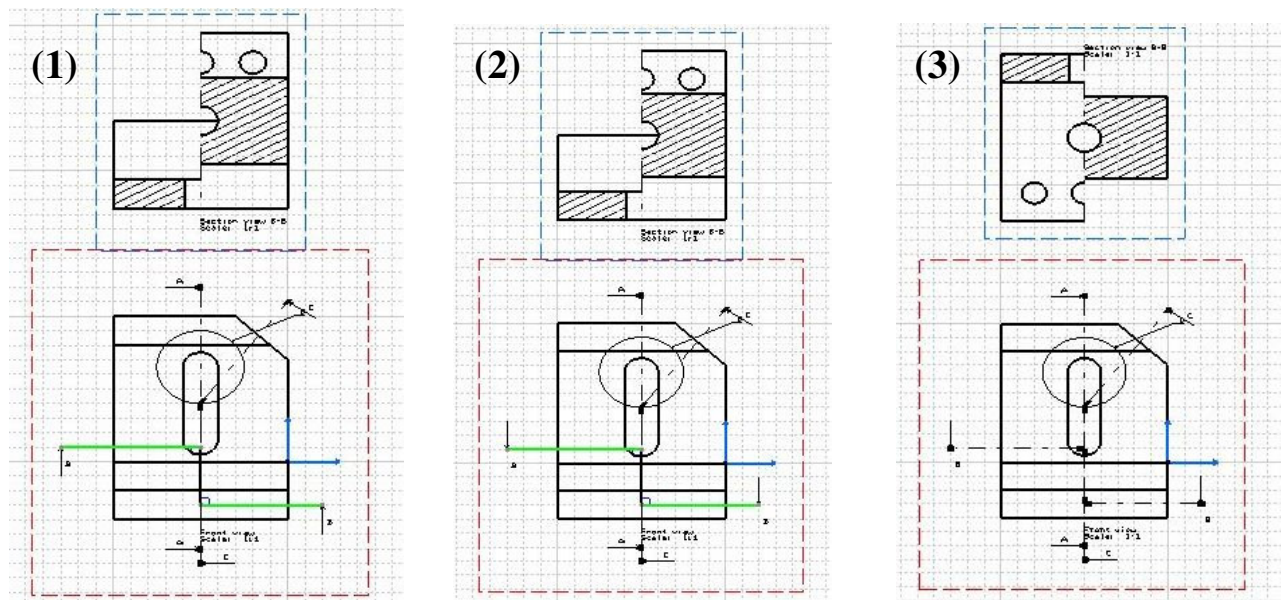
1. Double-click on the Section view profile

B-B (1) to open Edit/Replace toolbar .

Select the Replace Profile icon  and create new callout profile (2) and select the End Profile Edition icon  to apply the modification (3).

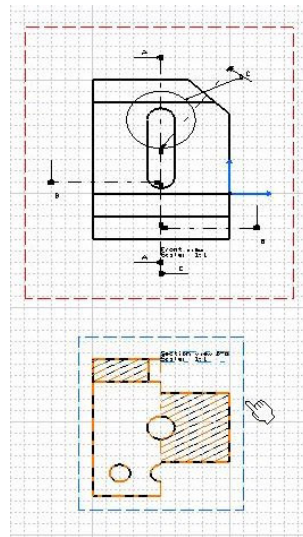


**2. Double-click on the Section view profile B-B (1) to open Edit/Replace toolbar** . **Select the Invert Profile Direction icon**  **(2) and select the End Profile Edition icon**  **to apply the modification (3).**




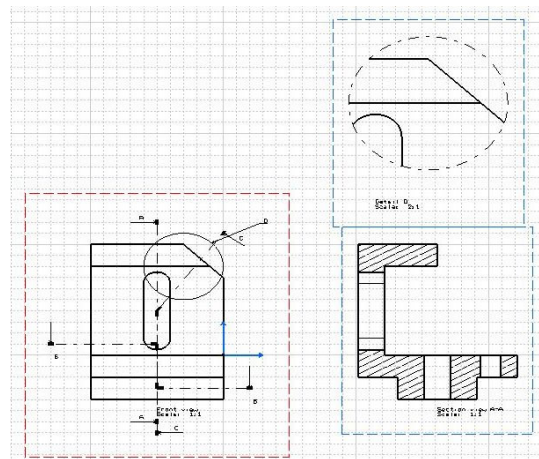
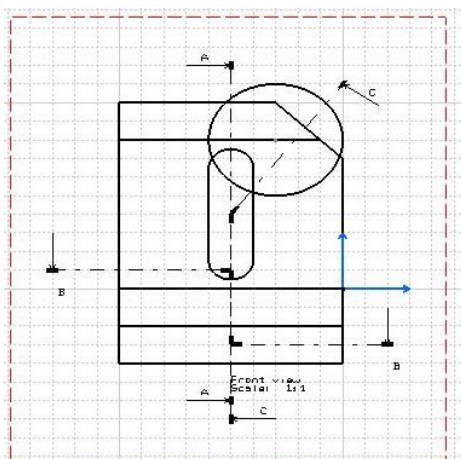
**3. Then, click on the section view B-B frame and move it to the bottom front view.**





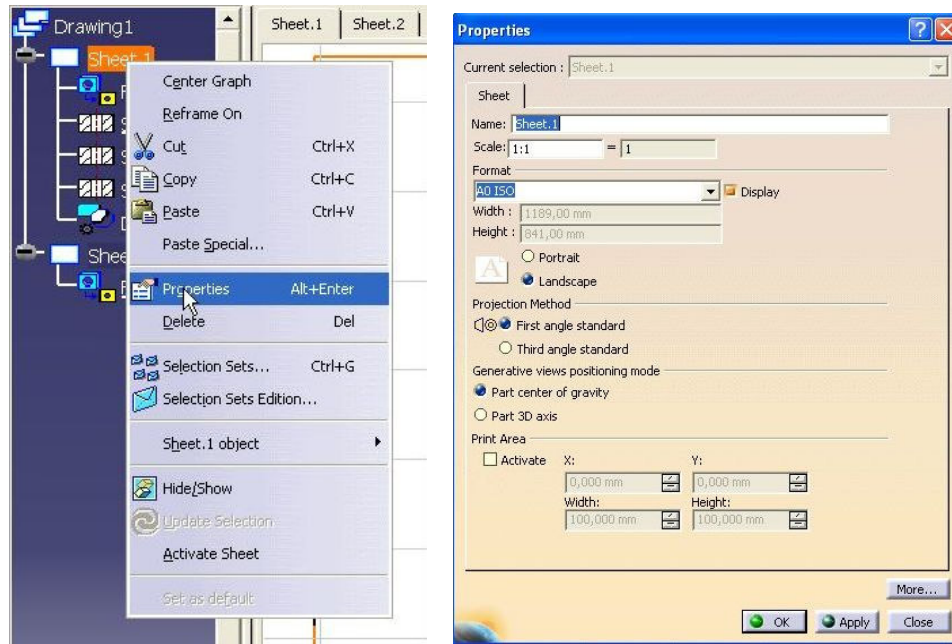
## Example No. 12. Modifying Detail View Profile.

1. Double-click on the Detail view profile to open Edit/Replace toolbar . Select the Replace Profile icon and create new callout profile (1) (new center point and radius of circle) and select the End Profile Edition icon to apply the modification (2).

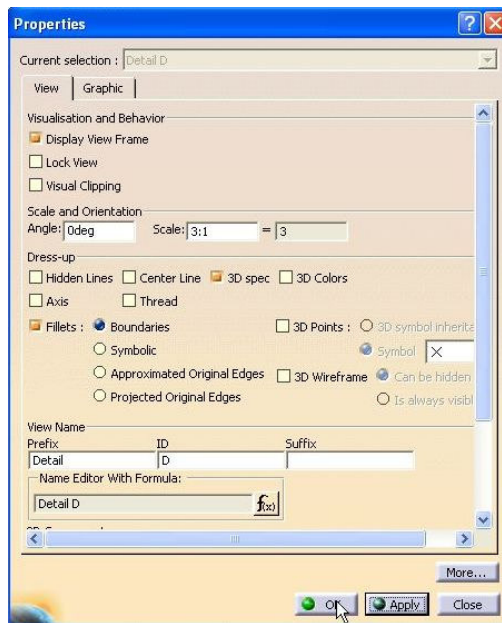


# Sheet properties

Select the Sheet.1 to modify the properties in the tree and Properties with the contextual menu.



The Properties dialog box appears. You can change a name, scale or projection method. You can change view properties. Select the Detail view to modify the properties in the tree or detail view frame and Properties with the contextual menu. The Properties dialog box appears. You can change a name, scale (write in 3:1) or orientation.




# Dimension Generation

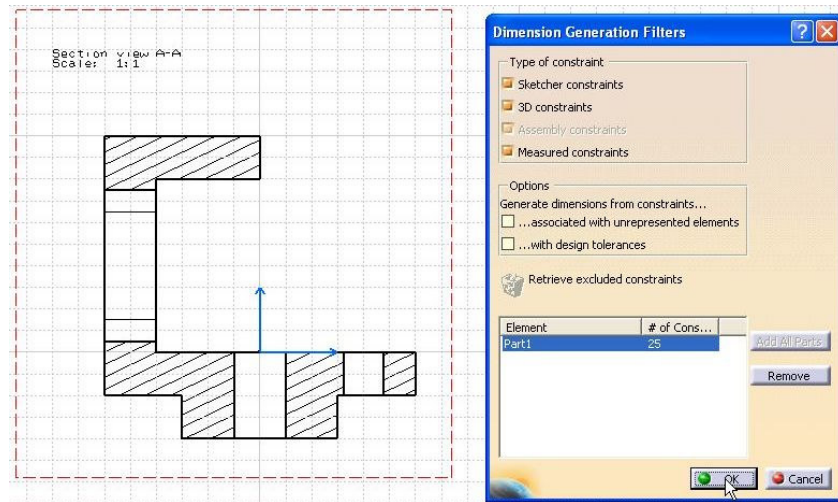
## Generate Dimension (Automatic).

**Generate Dimension** is a option that generates dimensions from existing 3D Part constrains (dimension constraints). Dimensions can be generated with two methods:

- **In One Step**
- **Step by Step**

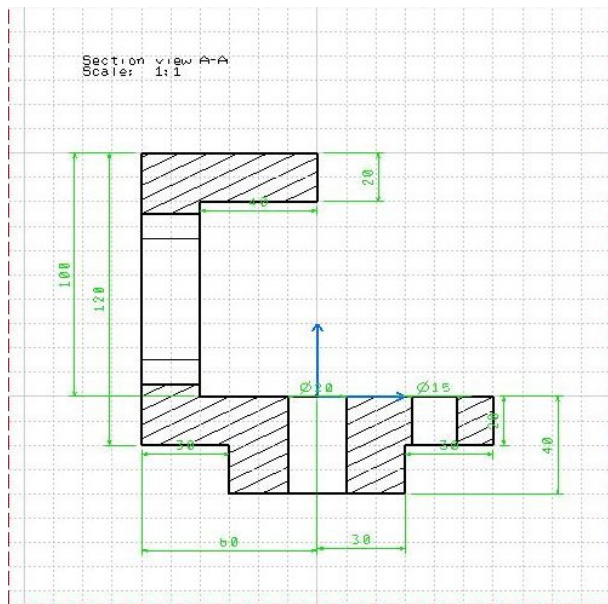
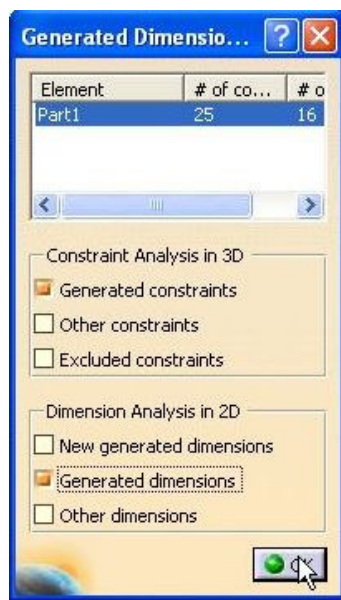
**Example No. 13. Generate Dimension (one step)**

**1. Activate the right Section view and select Generate Dimension icon . The Dimension Generation Filters dialog box appears. In Type of Constraints options mark Sketcher constraints, 3D constraints and Measured constraints and click OK.**




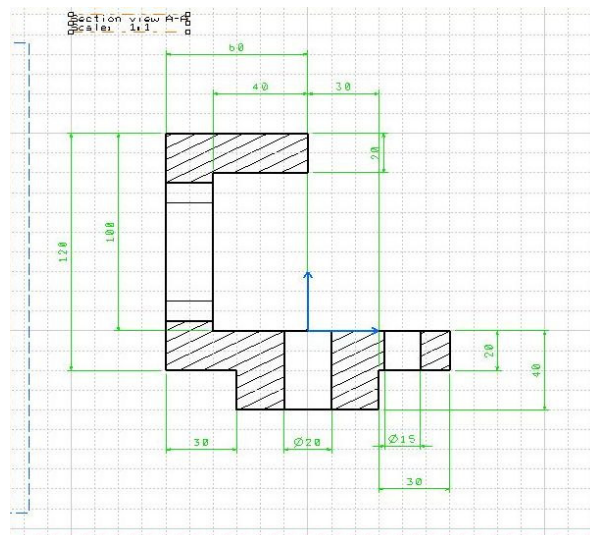
**2. The Generated Dimension Analysis dialog box appears. In Constraint Analysis 3D options mark Generated constraints and in Dimension Analysis 2D options mark Generation dimensions and click OK.**






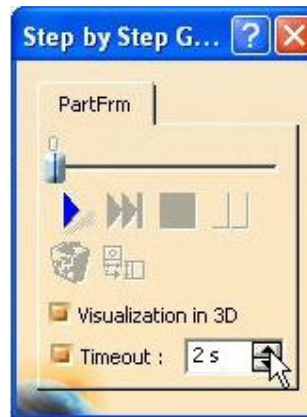
**3. Then you can use Automatic Positioning options to reposition dimensions or reposition dimensions manually.**

**Select the Section view and select Dimension Positioning icon . Dimensions have been repositioned automatically.**



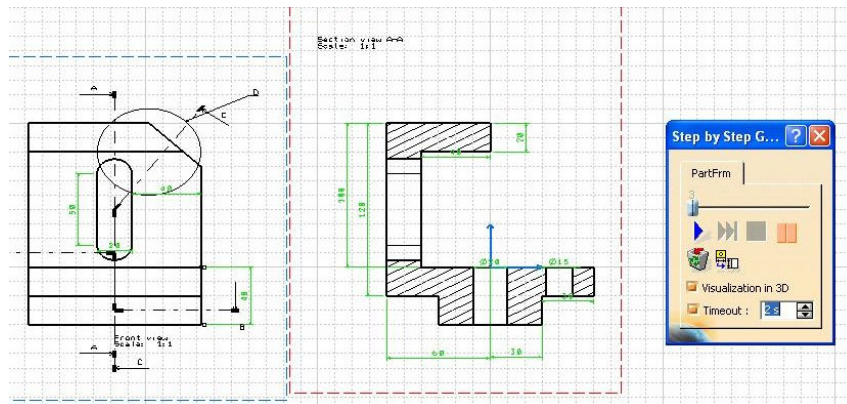
**Example No. 14. Generate Dimension (step by step)**

**1. Remove dimensions generated previously. Then select section view and select Generate Dimension Step by Step icon . The Dimension Generation Filters dialog box appears. In Type of Constraints options mark Sketcher constraints, 3D constraints and Measured constraints and click OK.**



**2. The Step by Step Generation Dimension dialog box appears. In Timeout options write 2 seconds (this is a delay between each dimension generated) and select the Next Dimension Generation icon. Dimension will be generated. Select Pause icon – now you can use Transfer option to relocate a dimension to another view (select the frame of the view to which the dimension is to be transferred) or Not Generated option to discard any unwanted dimensions. Then continue**



**generation process by selecting Next icon again.**



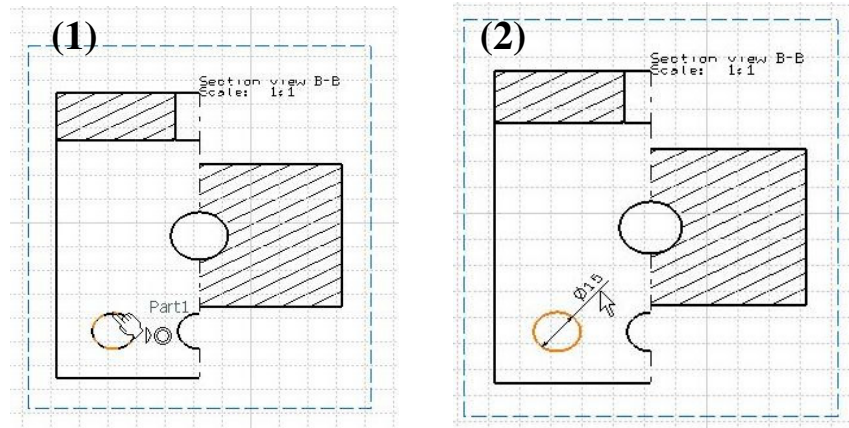
## **Adding Dimensions (manually)**

**You can create additional dimensions manually. Many options to manual dimensioning are in Dimensioning toolbar (those options are similar to options in AutoCAD application)**

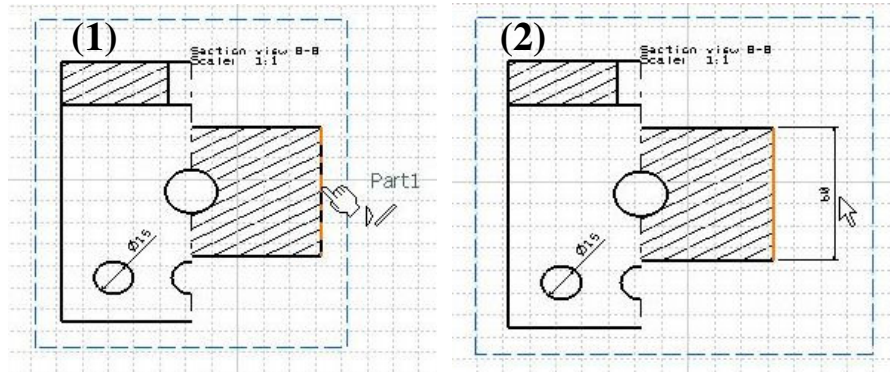
### **Example No. 15**

**1. Activate bottom Section view. Then select Diameter Dimension icon  and select a hole on the view (1). The dimension type is automatically defined according to the selected elements (or in the Tools Palette ). The command options in the**

**Tools Palette allows you to position the dimension using one of the modes. Then choose location dimension on the drawing (2).**



**2. Create a linear dimension. Select Dimensions icon  and select the vertical line (1) and choose location on the drawing (2).**




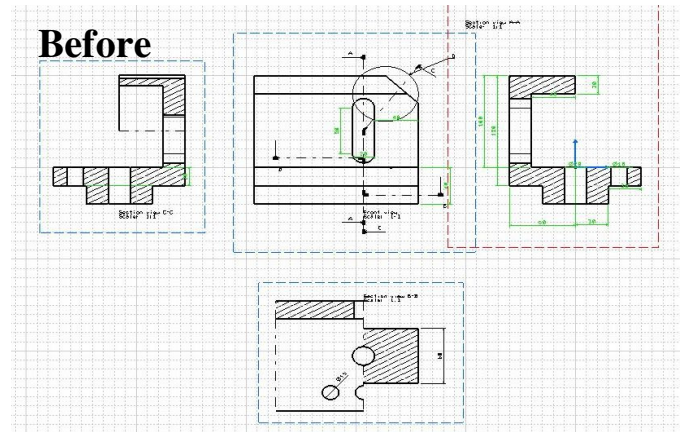
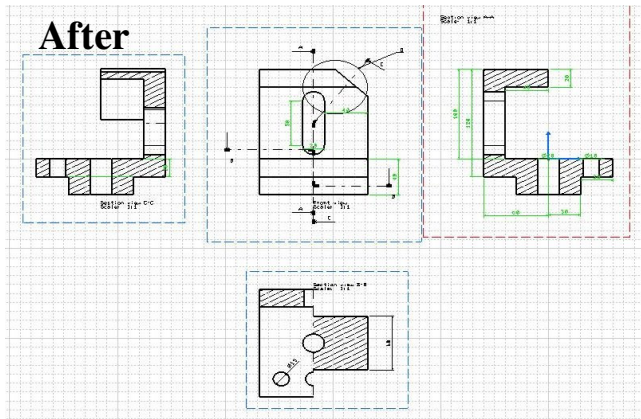
## **Modifications part**

**If you modify your part in Part Design, you can update your sheet.**

**Example No. 16**



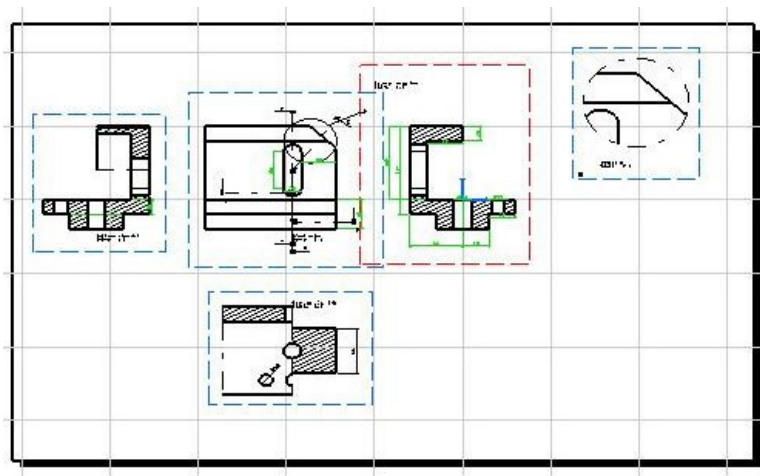
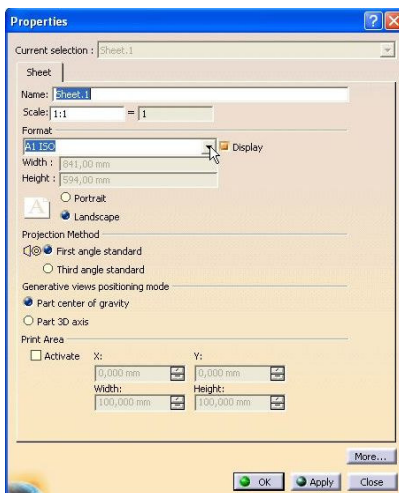
**1. Activate the CATPart (Draft1) document and change Pad length value (on 150). Then activate the CATDrawing document and select Update current sheet icon .**



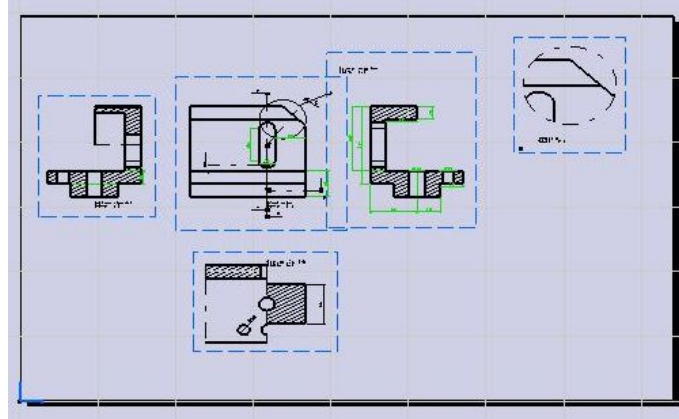
## Adding Title Block

### Example No. 17

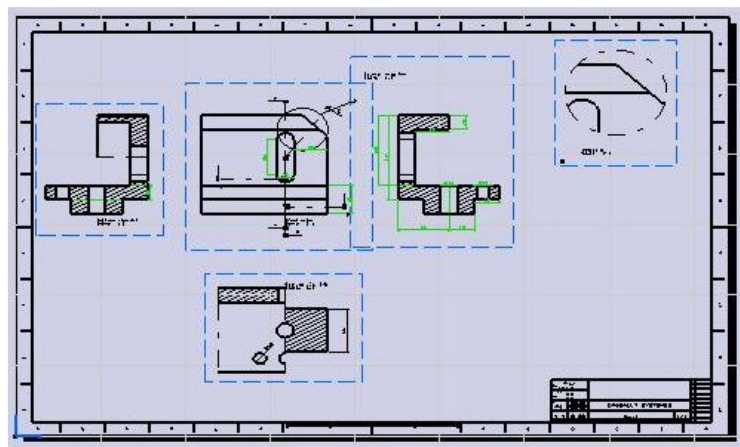
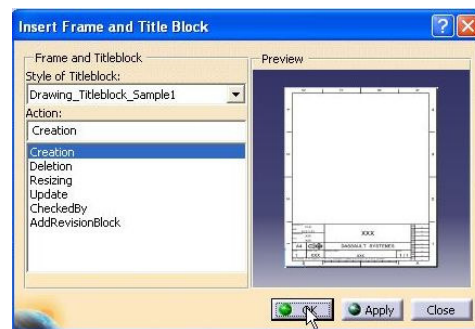
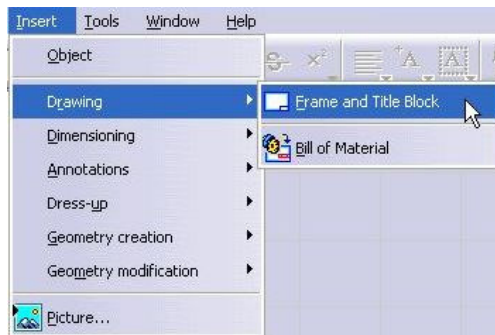
**1. Change format sheet in Sheet Properties options on A1 ISO.**



**2. Then change the sheet Background with Edit → Background menu.**



**3. Then select Insert → Drawing → Frame and Title Block menu. The Insert Frame and Title Block dialog box appears. In Style of Titleblock options you can chose different styles of title block. Confirm by clicking OK button.**



**4. Use the geometric tools to draw the title block and use text tools to fill in all necessary information. When finished, go back to Working View from the Edit menu.**

### **Task No. 1**

**Create drawing as the following one (open the Draft2 file)**

