How to do drafting in CATIA V5: create 2D drawings from 3D models

This guide shows you how to do drafting in CATIA V5. You start with a 3D part, create a 2D drawing sheet, and place standard views. You then format annotations, show hidden lines and centerlines, and adjust view scale. Finally, you add dimensions, notes, an isometric view, and a title block.

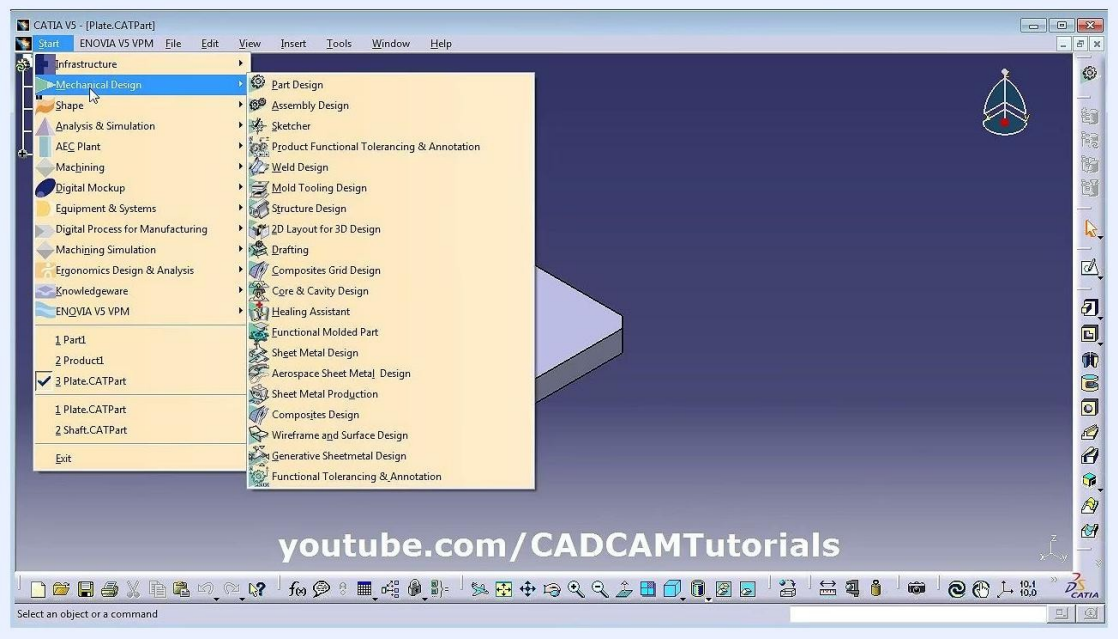
Prerequisites

- CATIA V5 installed on your computer.
- A 3D part file to draft, such as Pin, Flange, Plate, or Vertical Plate. The examples store these in Documents\CATIA.

Prepare the model for drafting

Step 1. Launch CATIA V5

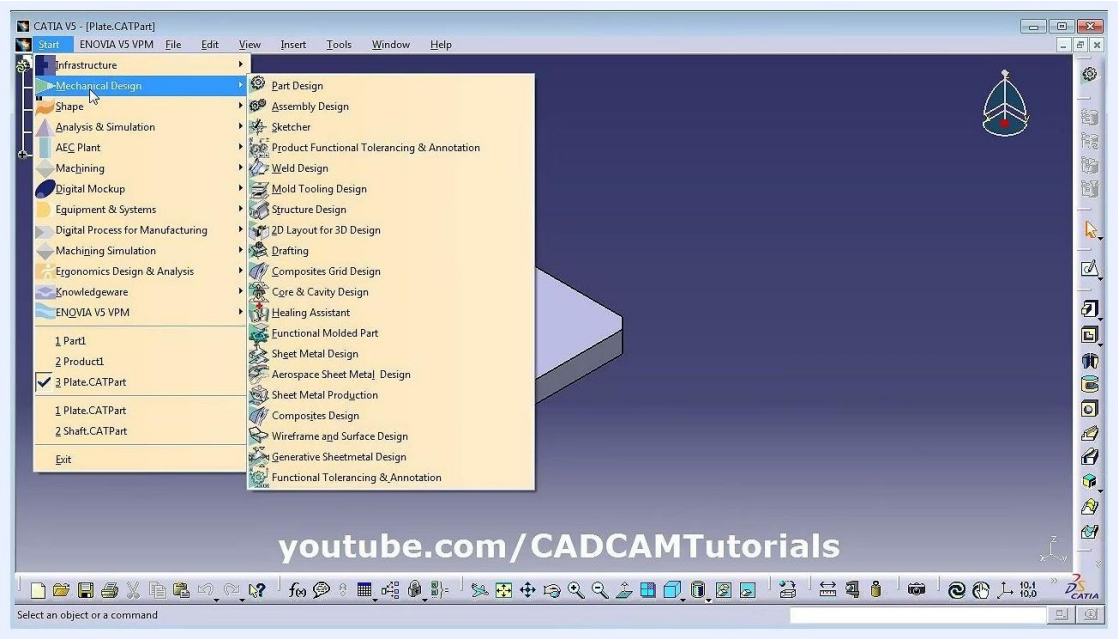
Launch CATIA V5 and confirm you are on the main screen. Make sure the workspace is clear and ready for a new operation.



CATIA V5 main screen with navigation menus visible

Step 2. Open the 3D part

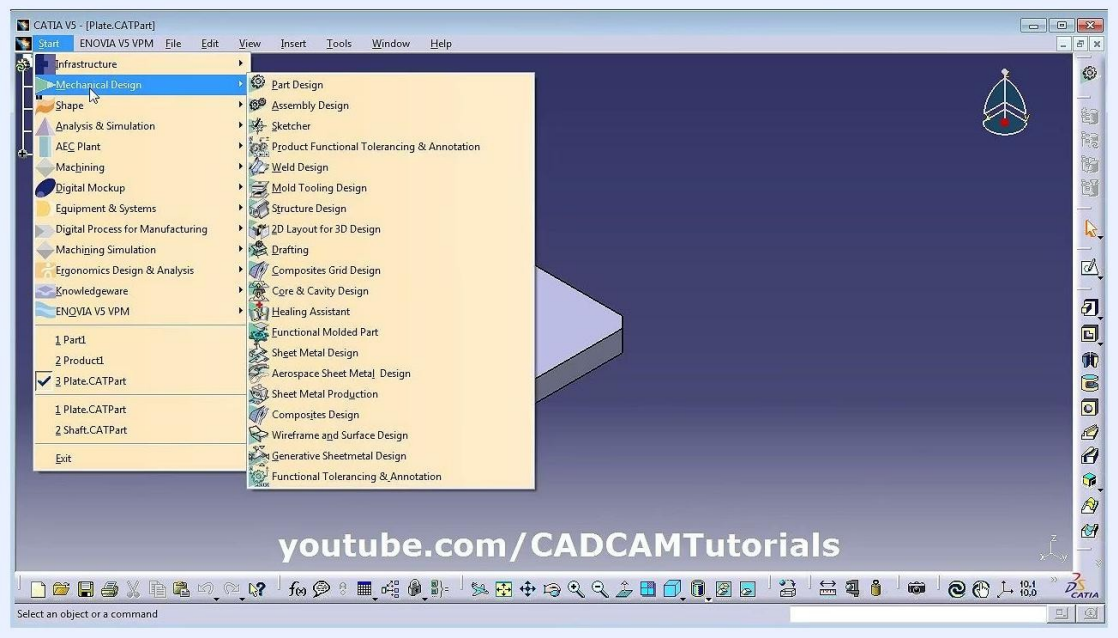
Click the **Open** icon, or select **File > Open**. In the file selection dialog, go to the folder that contains your 3D part files, for example Documents\CATIA. Select the part you want to draft (for example, Pin, Flange, or Plate), then click **Open**.



File selection dialog listing CATIA part files such as Flange, Pin, Plate, Product1, Product2, Shaft, and Vertical Plate

Step 3. Switch to isometric view and start the Drafting workbench

Set the model to isometric view for better visualization. Then click the **Start** menu in the top left and hover over **Mechanical Design** to expand its submenu. Select **Drafting** from the list of modules.

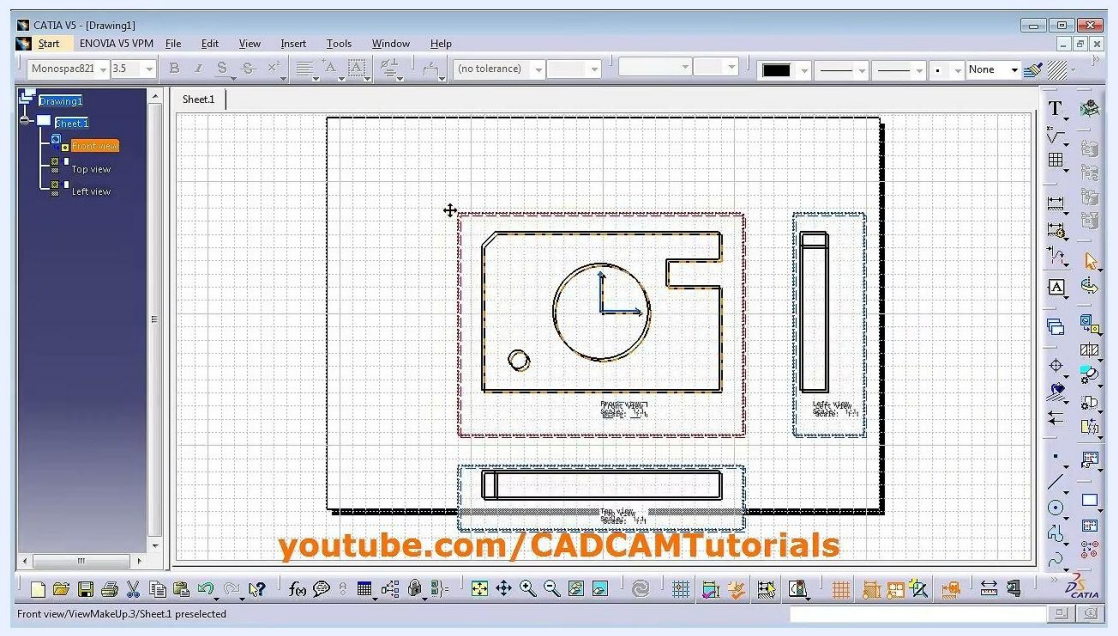


Start menu expanded with Mechanical Design highlighted and the Drafting option visible

Create the drawing sheet

Step 4. Choose a drawing sheet configuration

In the **New Drawing Creation** dialog, choose your layout. You can start with an empty sheet and add views manually. You can also choose a preset such as **All Views**, **Front, Bottom, and Right**, or **Front, Top, and Left**. This tutorial uses **Front, Top, and Left**.



New Drawing Creation dialog with the Front, Top and Left layout selected, showing the preview and sheet options

Step 5. Set the sheet standard and size

In the same dialog, click **Modify** to adjust the sheet settings:

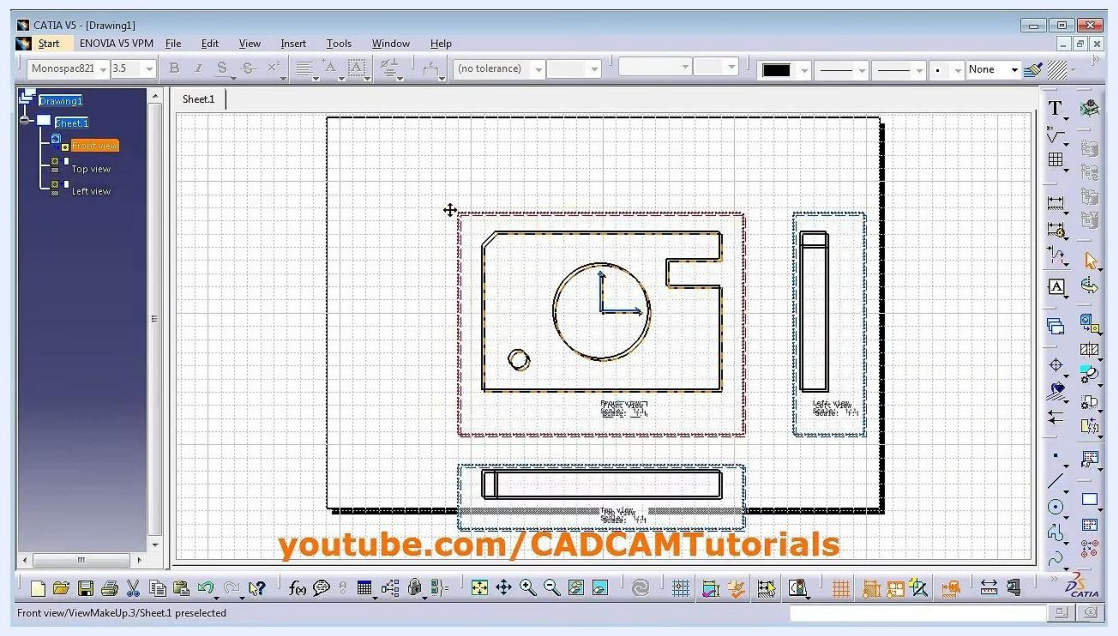
- Set **Standard** to **ISO**.
- Set **Sheet Size** to **A3** (297 x 420 mm).
- Set the orientation to **Landscape**.

Click **OK** to confirm.

Step 6. Position the views on the sheet

The sheet displays the selected Front, Top, and Left views. Click and drag the views to position them as you like.

The views are interlinked. Moving the main view moves the associated views with it, and you can move side views only horizontally.

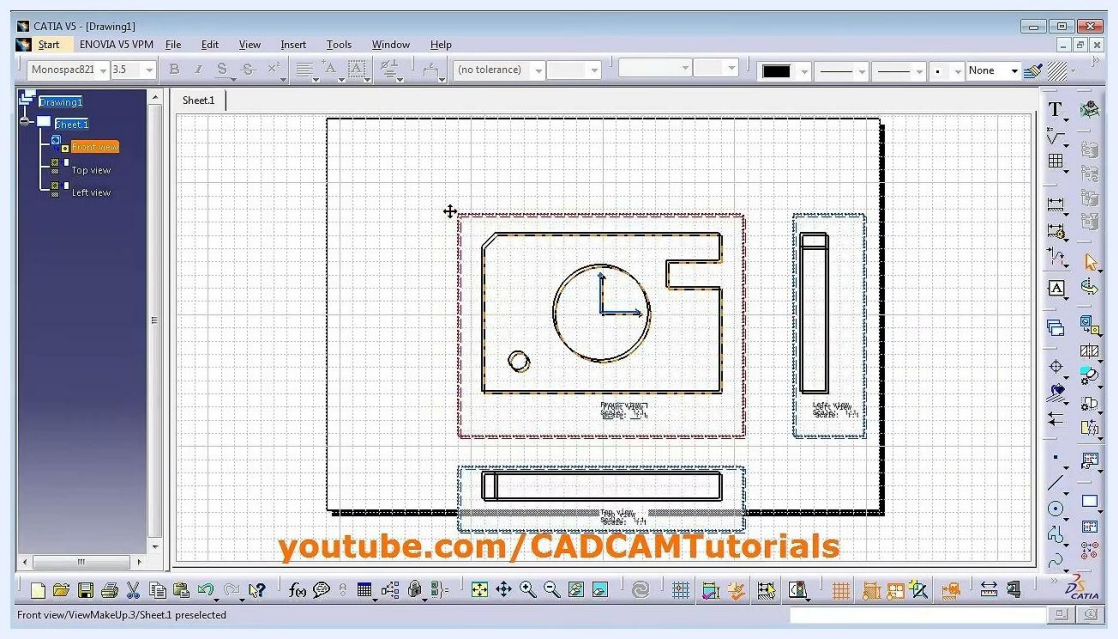


Drawing sheet with front, top, and left views of the part that you can reposition by dragging

Format labels and annotations

Step 7. Adjust and format view labels

Select a label, such as **Front View**, under the view you want to change. In the text editor, change the font, increase the font size, or apply bold or italic formatting. Click **OK** to confirm.



Text editor open for the Front View label with the font dropdown visible and drawing views on the sheet

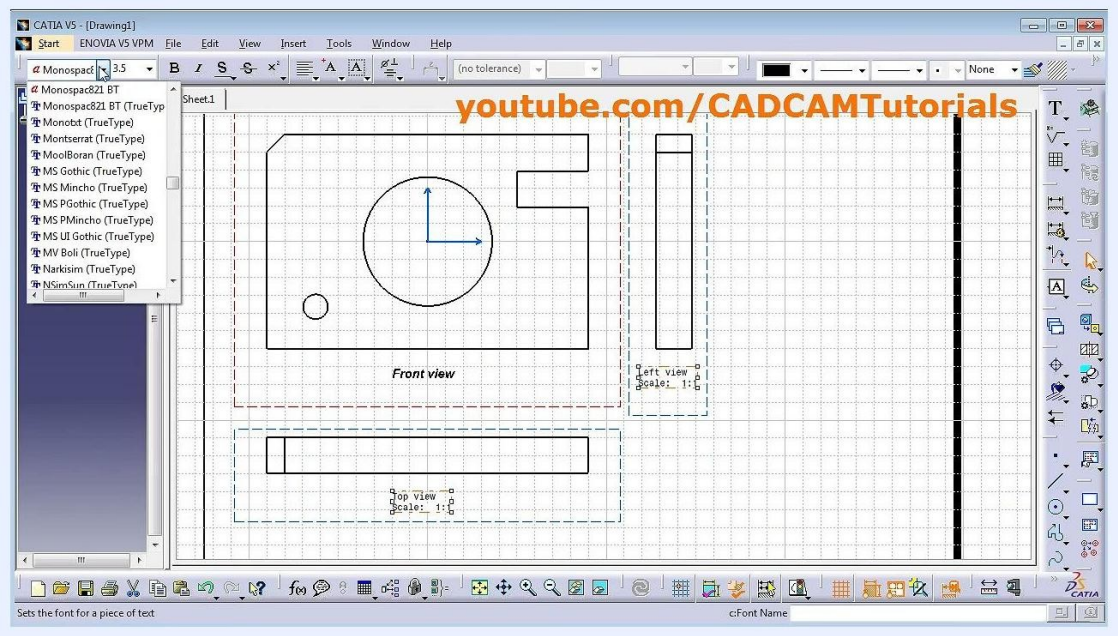
Step 8. Move and format annotations

Click and hold the left mouse button on an annotation, such as a view label or scale text, and drag it to a new location.

To remove the scale annotation, double-click the scale text (for example, "Scale: 1:1") under the view. Delete its content in the text editor.

To change an annotation's formatting, double-click it to open the text editor and select the text. Choose a different font from the font dropdown and adjust the size, bold, italic, and other options. Click **OK** to apply the changes.

To format several annotations at once, hold Ctrl and click each annotation. Then apply the font, size, bold, or italic formatting to all of them together.

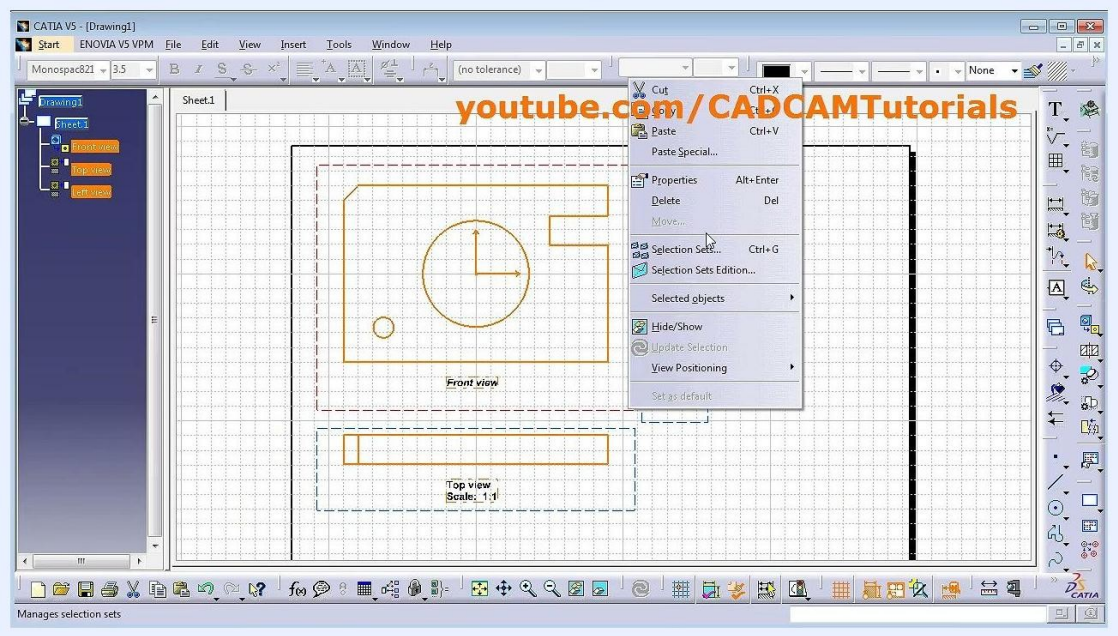


Multiple annotations selected with the font dropdown expanded for batch formatting

Control view display and scale

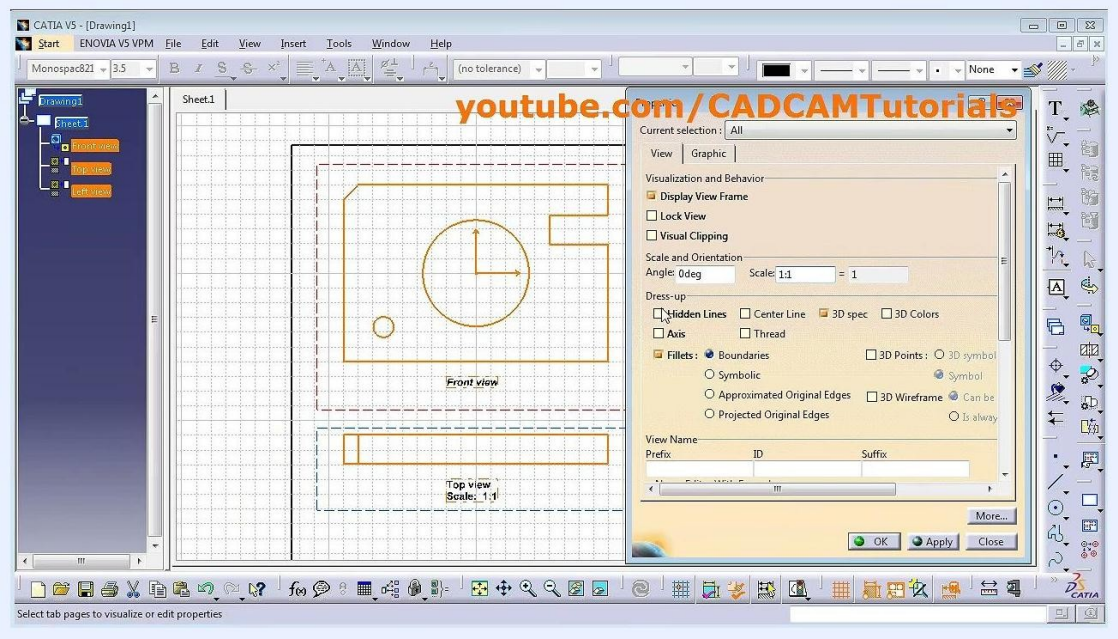
Step 9. Display hidden lines and centerlines

First, hold Ctrl and select all the views you want to modify. The selected views are highlighted in orange in the tree and on the sheet.



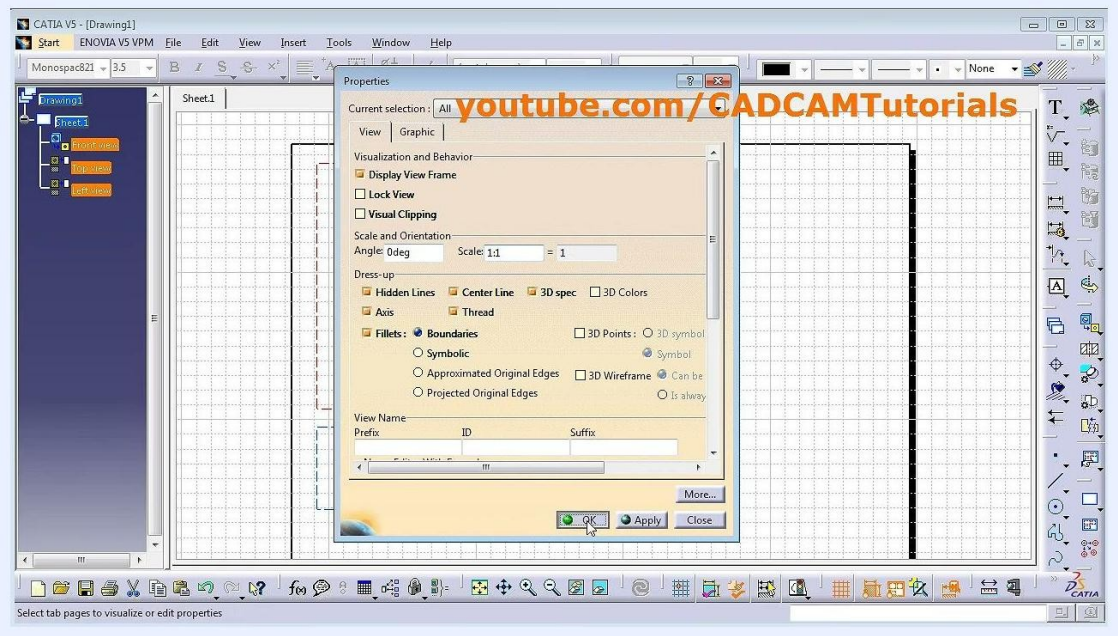
All drawing views selected in the tree and on the sheet, ready for property changes

Right-click any selected view and choose **Properties**. In the Properties dialog, open the **View** tab and find the **Dress-up** section. Check **Hidden Lines**, **Center Line**, **Axis**, and **Thread** as needed.



Properties dialog with the Hidden Lines, Center Line, Axis, and Thread options in the Dress-up section

Click **Apply** or **OK** to confirm and close the dialog.



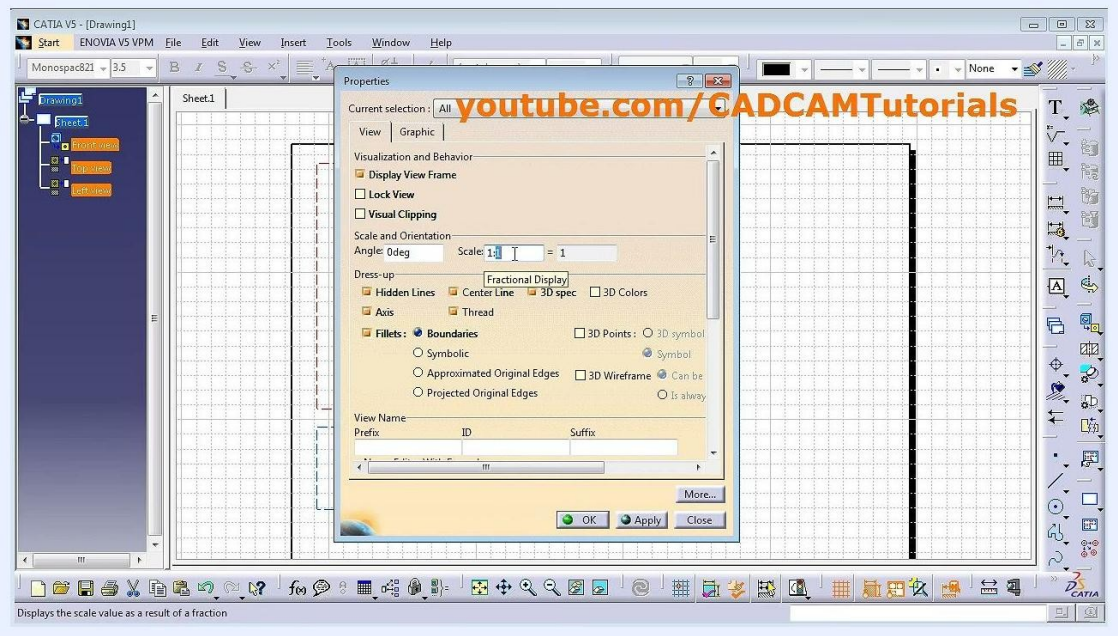
Properties dialog with hidden lines and center lines enabled and the Apply and OK buttons visible

Step 10. Adjust the scale of drawing views

Open the Properties dialog as in the previous step and find the **Scale** field under **Scale and Orientation**. A higher ratio enlarges the view, and a lower ratio reduces it:

- Setting the scale to 2:1 doubles the view size.
- Setting the scale to 1:2 makes the view half its original size.

Click **Apply** or **OK** to update the view scale.

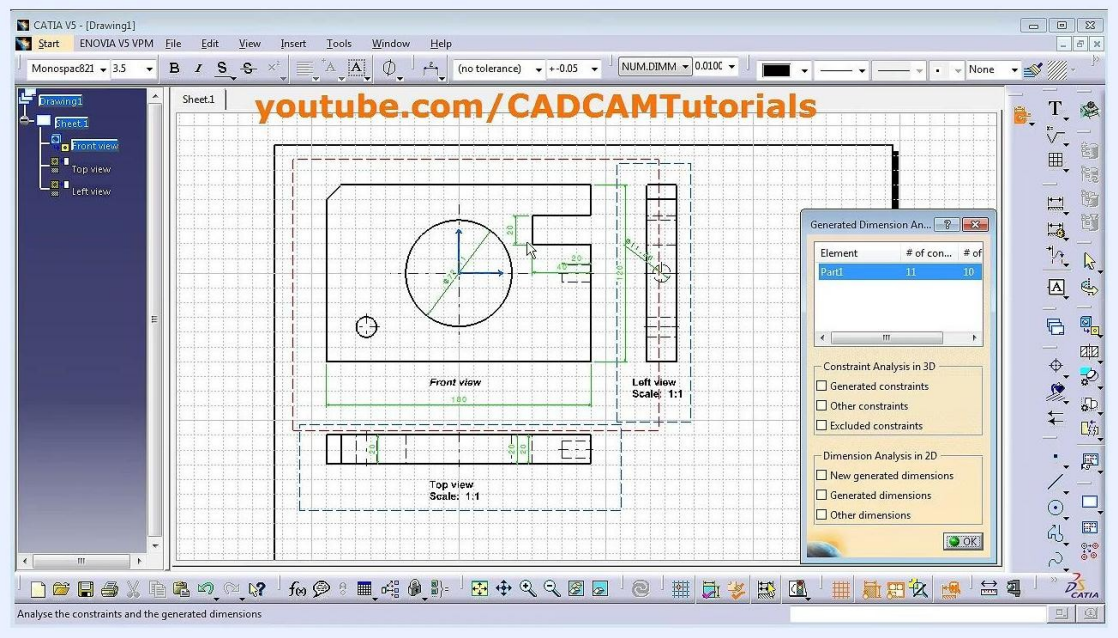


Properties dialog with the Scale field highlighted for adjusting the view scale ratio

Generate dimensions automatically

Step 11. Display the generated dimensions

Find the **Generate Dimensions** tool on the right toolbar (the icon with a dimension symbol) and click it. The **Generated Dimension Analysis** dialog appears and lists the part and the number of constraints and dimensions. Click **OK** to display all the generated dimensions on your drawing views.



Drawing sheet with generated dimensions on all views and the Generated Dimension Analysis dialog open

Step 12. Arrange and clean up dimensions

To reposition a dimension, press and hold the left mouse button on it and drag it to a new location on the sheet.

To align diameter values horizontally:

1. Hold Ctrl and select the diameter dimensions you want to align.
2. Right-click one of them and choose **Properties**.
3. Click the **Dimension Line** tab.
4. Under **Representation**, select the two-part horizontal option.
5. Click **OK**.
6. Click outside the dialog, then drag the diameter dimensions to your preferred positions.

To remove duplicate or unnecessary dimensions, select them and press Delete. For example, you can delete a "20" dimension that appears three times.

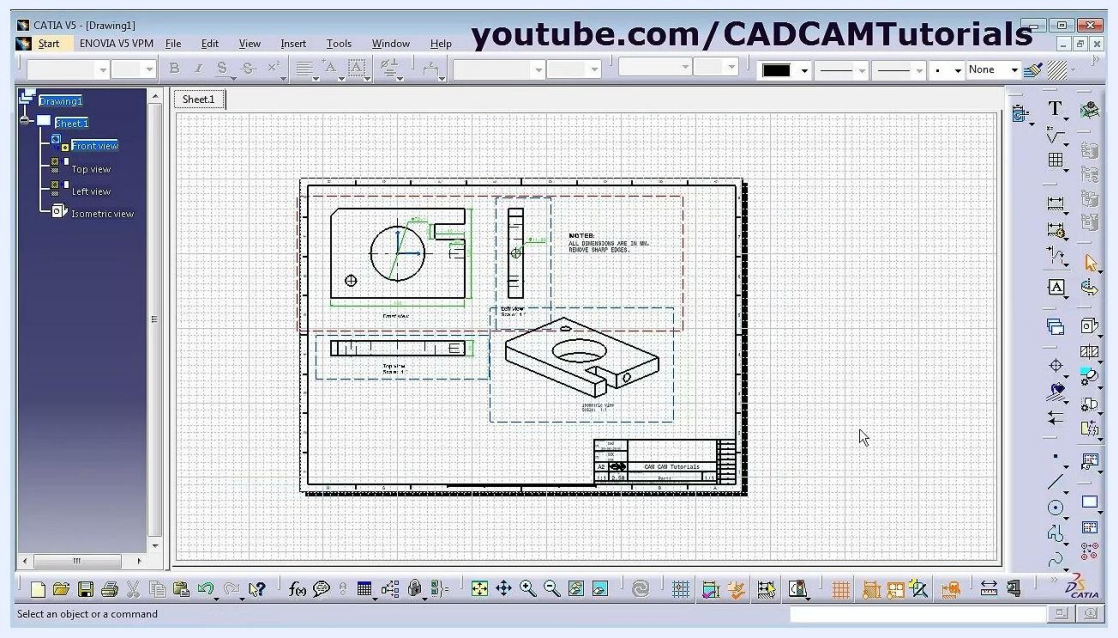
Step 13. Add an isometric view

Click the **Isometric View** tool, typically in the view creation toolbar, and select **Isometric View** in the prompt that appears. Click the drawing sheet where you want to place the view. In the window that opens, choose your part (for example, "Part1" or "Plate") and select the orientation you want.

Add a frame and title block

Step 14. Create the title block

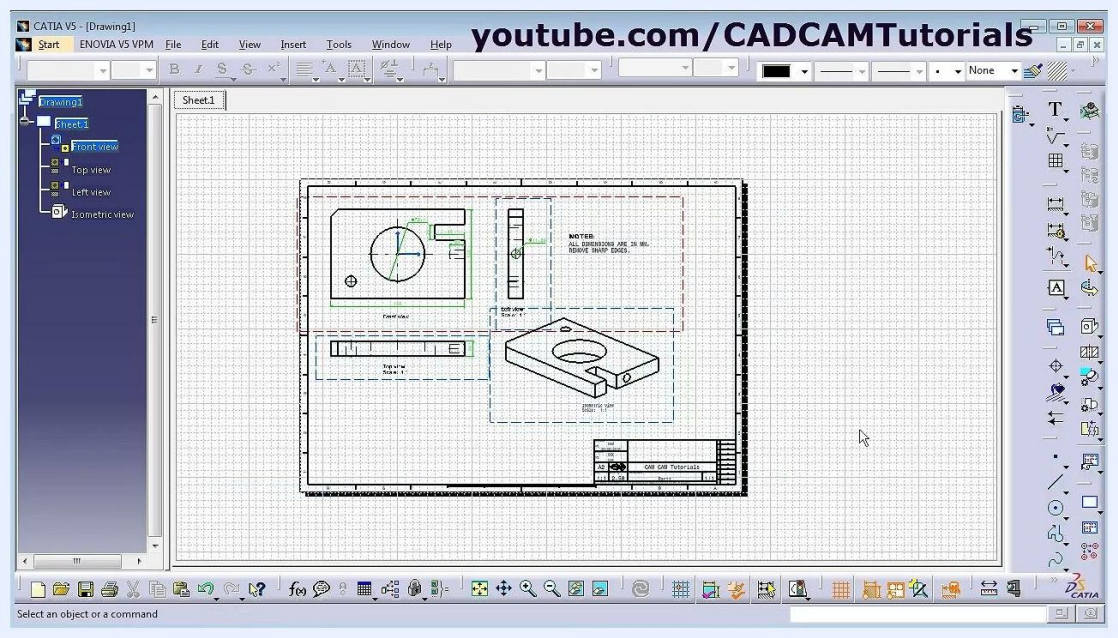
Open the **Manage Frame And Title Block** dialog. Set **Style of Title Block** to **Drawing Titleblock Sample 1** and set **Action** to **Create**. Click **Apply**, then **OK**. The sheet now displays the title block and border.



Manage Frame And Title Block dialog with the Create action selected, and the drawing sheet showing multiple views and the title block

Step 15. Edit the title block text

Double-click the title block area to open the **Text Editor**. Enter the text you want, for example "CAM CAM Tutorials", and click **OK** to save.

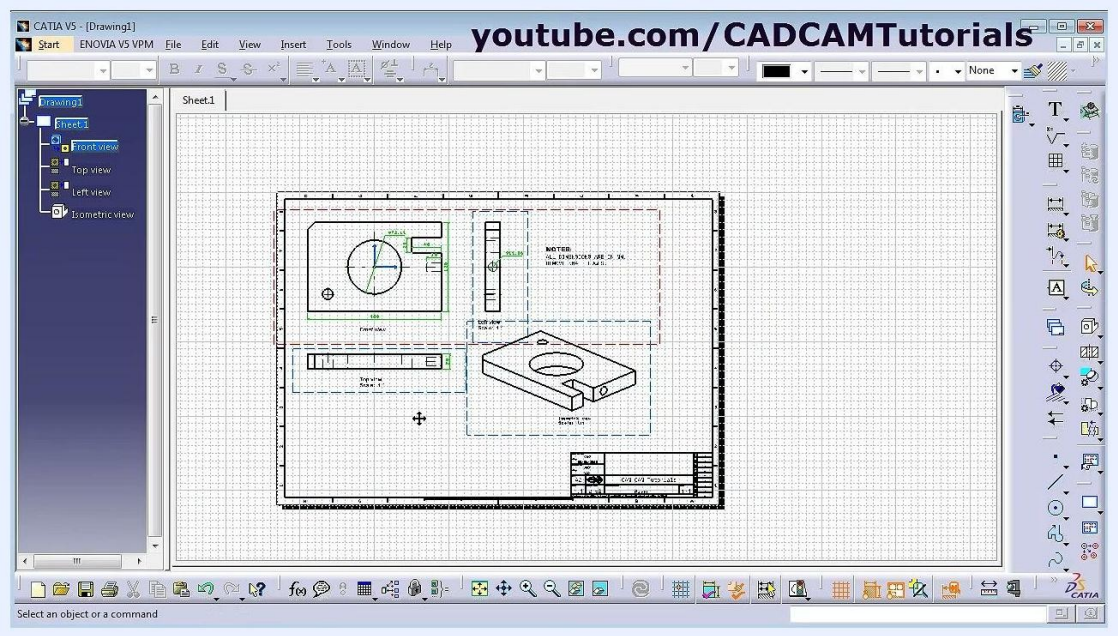


Text Editor with "CAM CAM Tutorials" entered and the title block highlighted for editing

Add and set up a second sheet

Step 16. Add a new sheet

In the tree on the left, right-click to add a new sheet (Sheet.2). The new sheet appears as a blank drawing area with a grid background.



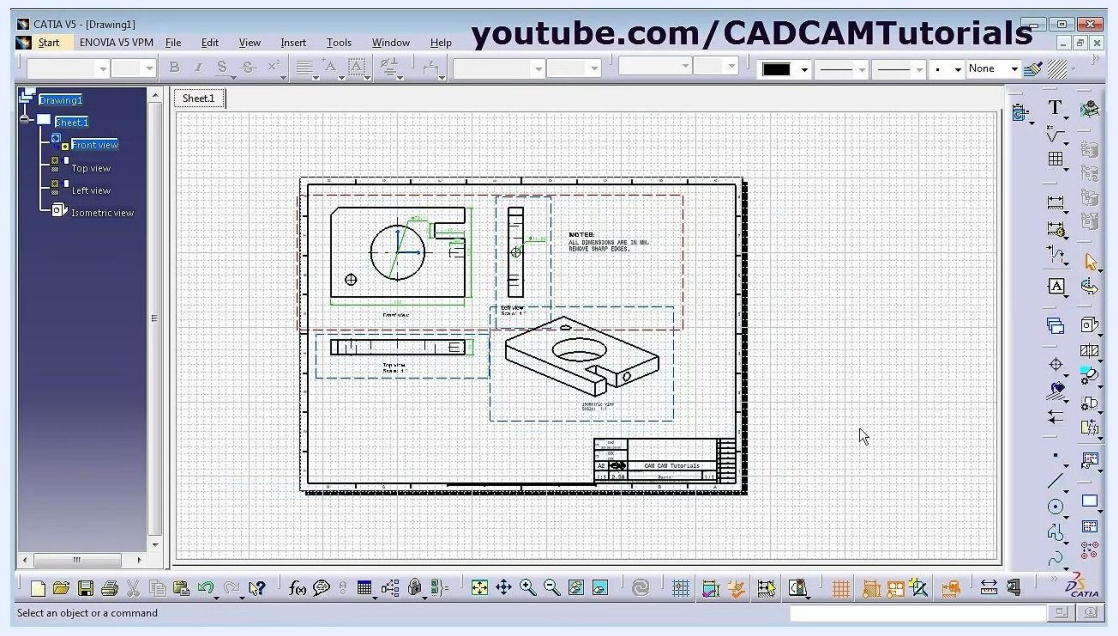
Drawing tree with two sheets, Sheet.2 selected and blank

Step 17. Set the sheet properties

In the **Properties** dialog for Sheet.2, set the following:

- **Name:** Sheet.2
- **Scale:** 1:1
- **Format:** A3 ISO (Width: 420.00 mm, Height: 297.00 mm)
- **Orientation:** Portrait or Landscape, as needed
- **Projection Method: First angle standard** or **Third angle standard**, as required
- **Generative views positioning mode: Part bounding box center**

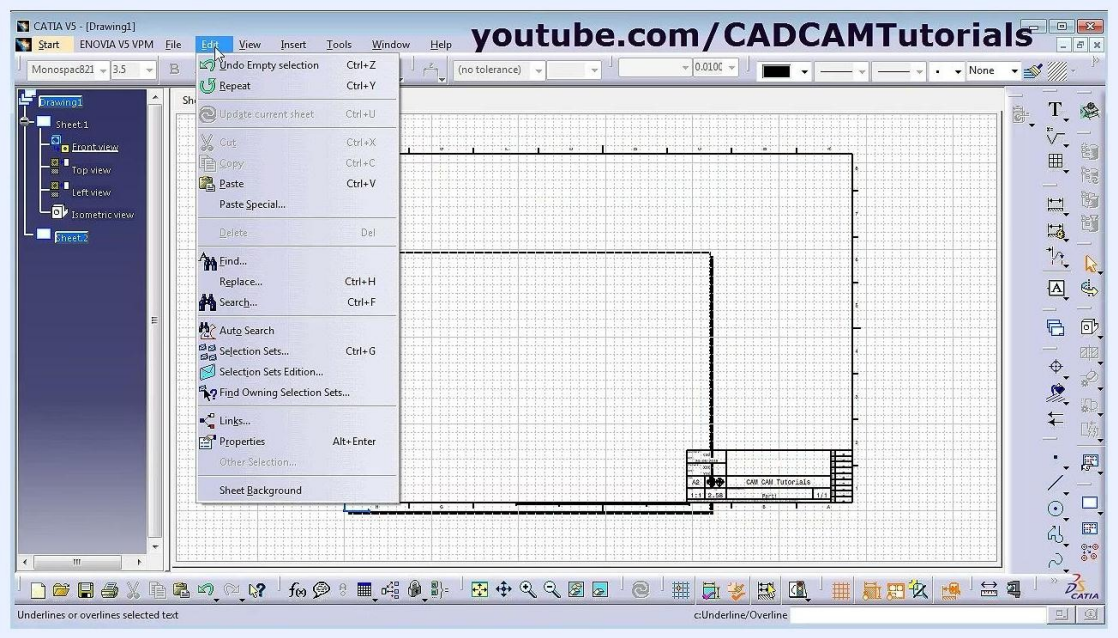
Click **OK** to apply the settings.



Properties dialog for Sheet.2 with A3 ISO format, 1:1 scale, and first angle standard selected

Step 18. Open the sheet background

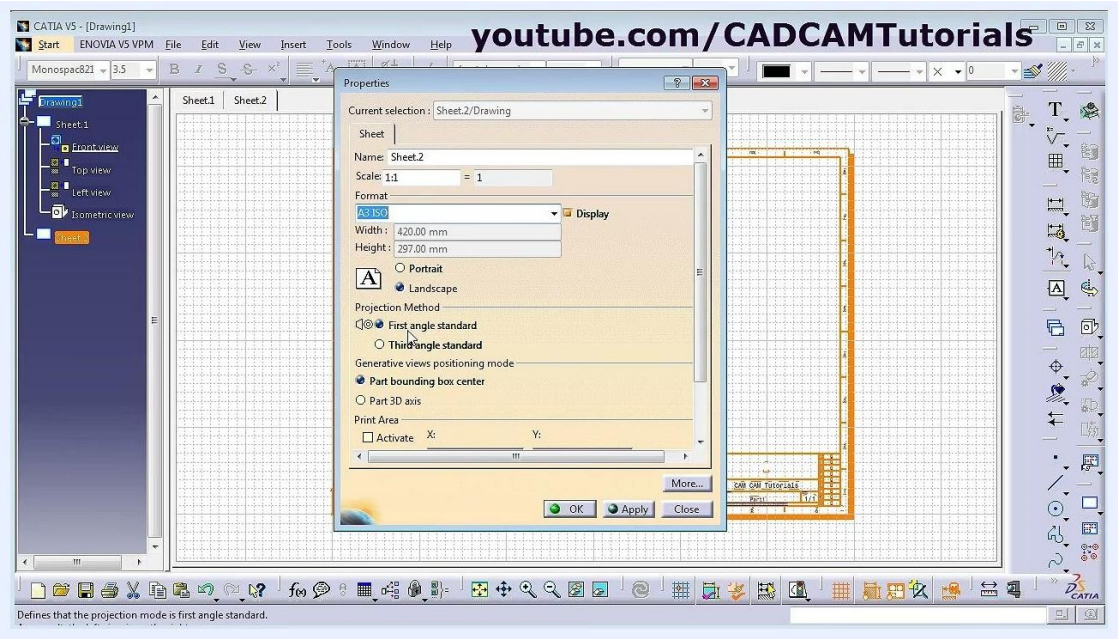
To add a frame and title block to the new sheet, go to the **Edit** menu and select **Sheet Background**. This lets you modify the background elements of the sheet.



Edit menu open with Sheet Background highlighted and a blank sheet background visible

Step 19. Add the frame and title block to the new sheet

Open the **Manage Frame And Title Block** dialog for the new sheet. Set **Style of Title Block** to **Drawing Titleblock Sample 1**. Under **Action**, select **Create** or **Resize**, as needed. Click **Apply**, then **OK** to update the frame and title block.

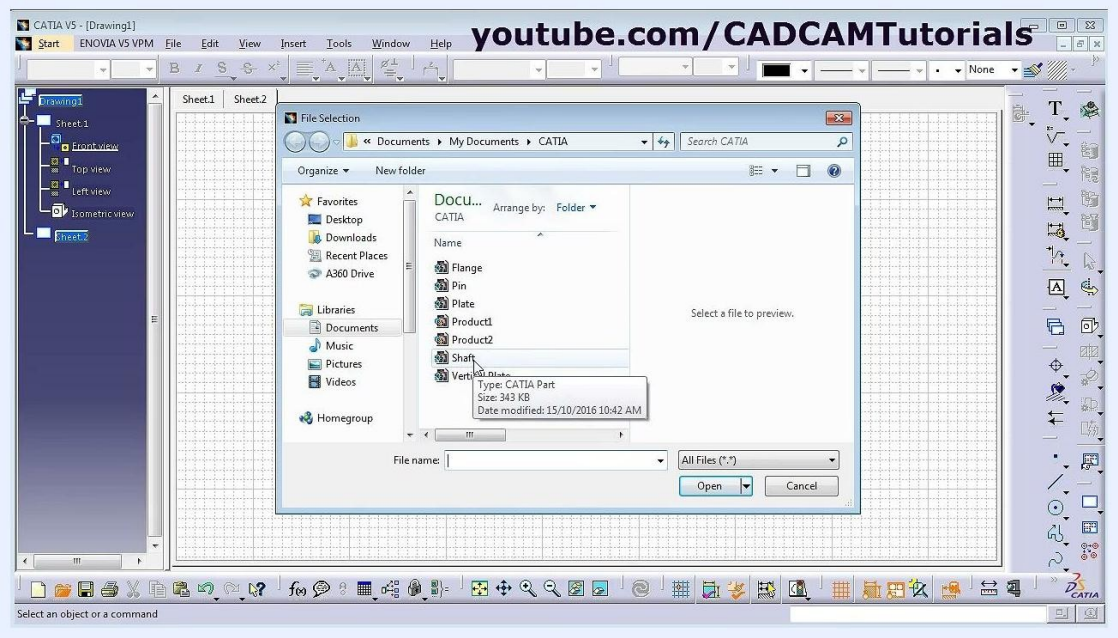


Manage Frame And Title Block dialog open on Sheet.2 with the Create action selected

Draft another part manually

Step 20. Open another part

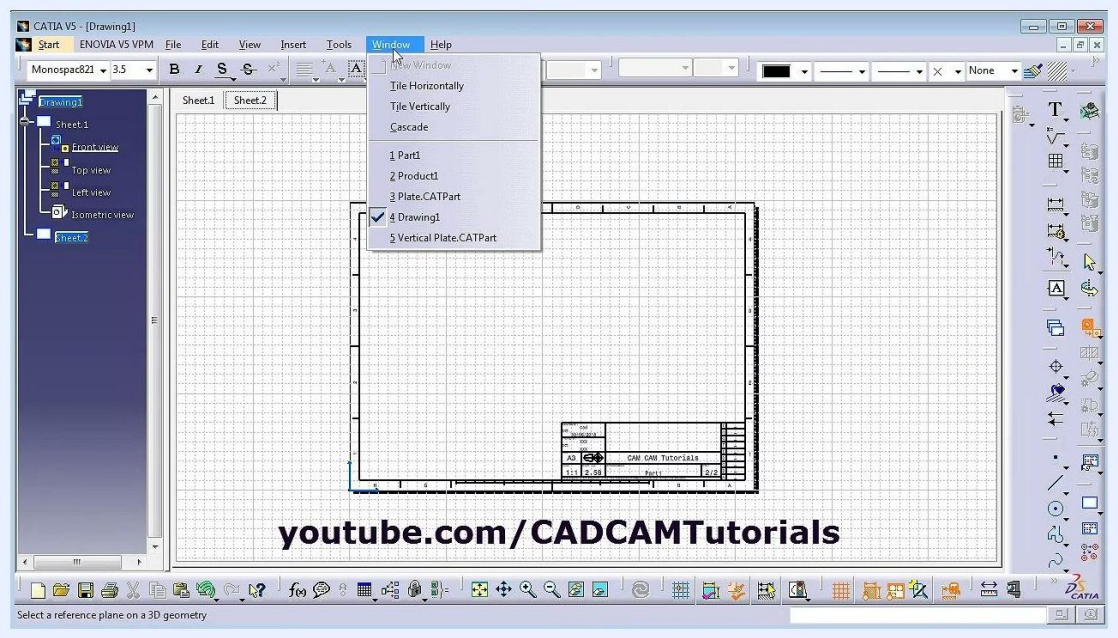
To create a drawing for another part, select **File > Open** or click the open icon. In the file selection dialog, go to the folder with your CATIA parts and select the part file, for example "Vertical Plate.CATPart". Click **Open** to load it.



File selection dialog with Vertical Plate.CATPart highlighted for opening

Step 21. Return to the drawing sheet

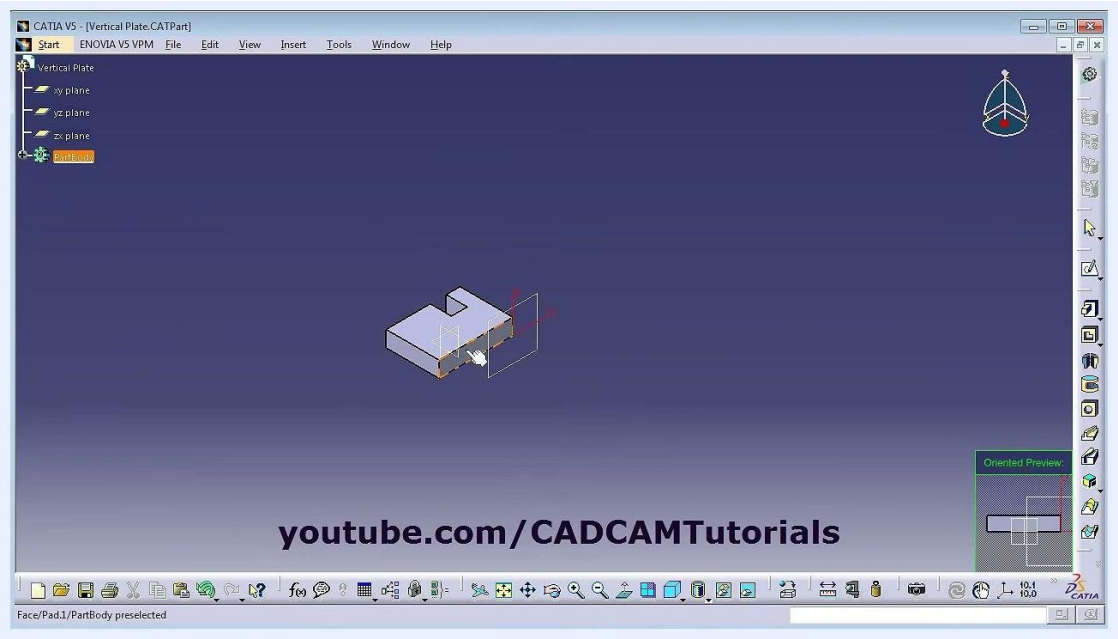
Click the **Window** menu and select your drawing document (for example, "Drawing1") to return to the drawing environment. Make sure the correct sheet, such as Sheet.2, is active and visible.



Drawing sheet with title block and the Window menu open, listing Drawing1 and Vertical Plate.CATPart

Step 22. Select the front view face

In the part window (for example, "Vertical Plate.CATPart"), locate the 3D model. Click the face you want as the front view. For example, select the large flat face if you want that side in front. The selected face is highlighted, and a reference plane may appear to show your selection.

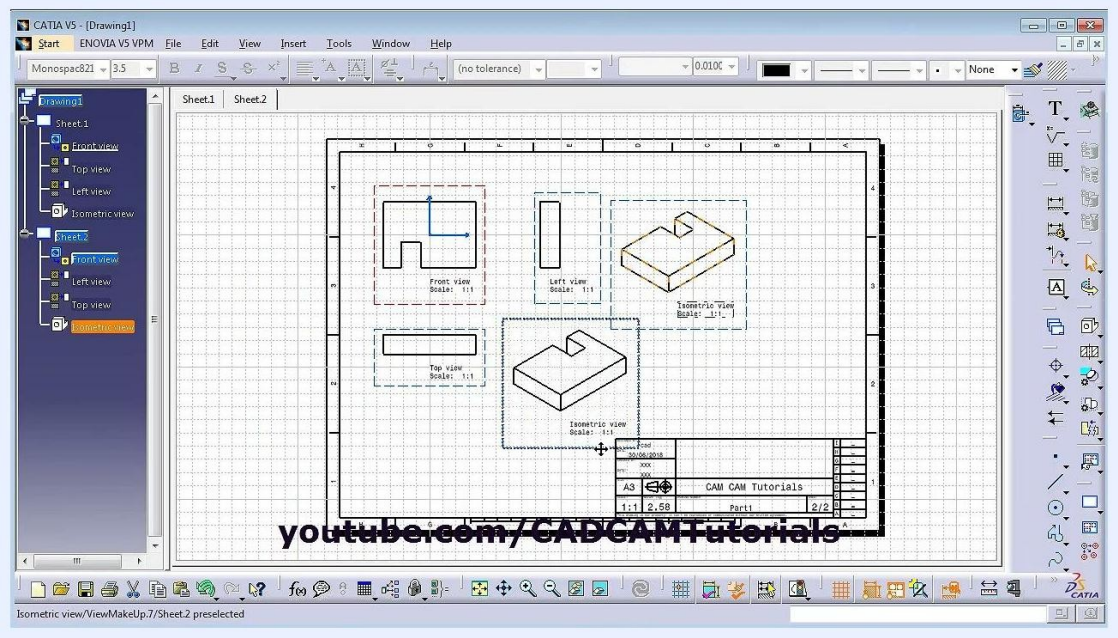


Part window with the Vertical Plate model, the front view face selected, and the reference plane visible

Insert views and add details manually

Step 23. Add a projection view

Click the **Projection View** tool on the right-side toolbar. Move the cursor next to the front view and click to place the left view. The new left view appears aligned with the front view, and the tree shows "Left view" under Sheet.2.

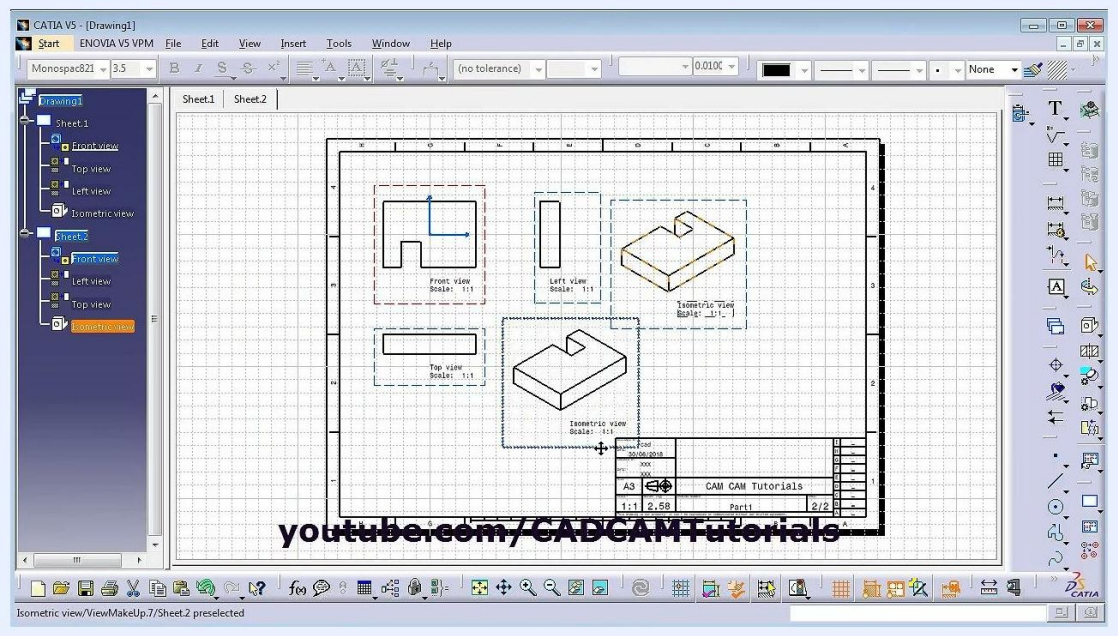


Front and left views placed on the drawing sheet, with both views listed in the tree

Step 24. Add more projection views and an isometric view

Click **Projection View** again and place a top view below the front view. The tree now lists "Front view", "Left view", and "Top view" under Sheet.2. Each view is labeled with its name and scale (for example, "Scale: 1:1").

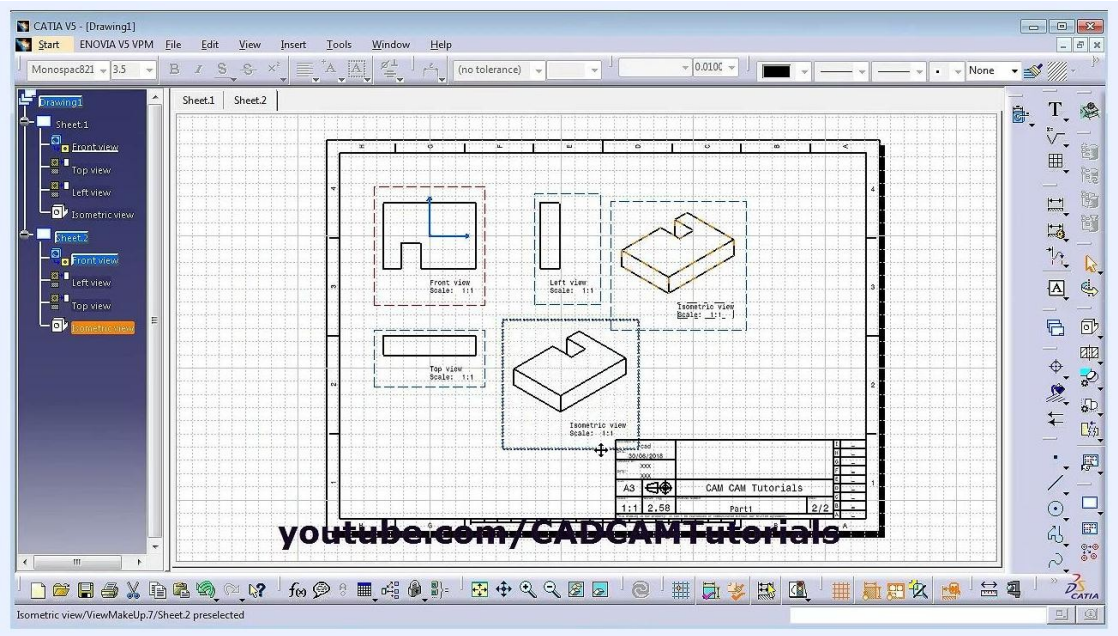
To add an isometric view, click the **Isometric View** tool on the right-side toolbar. Move the cursor to the location you want.



Front, left, and top views placed with the Isometric View tool highlighted, ready to place the isometric view

Step 25. Place and arrange the isometric view

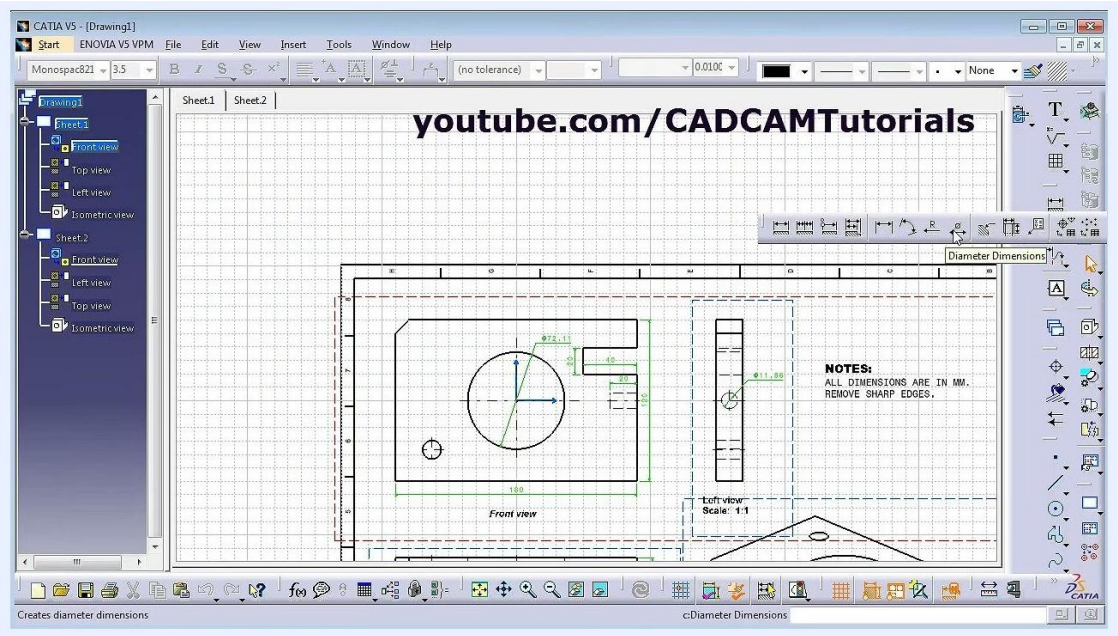
Click to place the isometric view on the sheet, then move and arrange the views for clarity. Select any view to access options for hiding elements or adding centerlines. The drawing now shows the front, left, top, and isometric views.



Front, left, top, and isometric views arranged on the drawing sheet

Step 26. Add dimensions manually

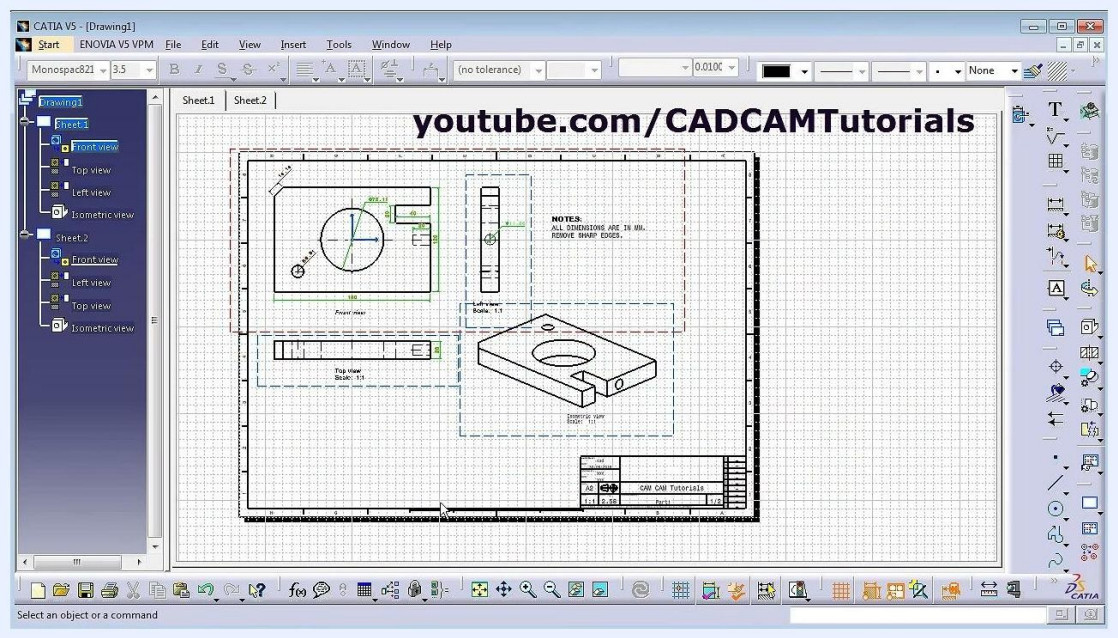
Use the dimensioning tools on the right-side toolbar. For a diameter, select the appropriate tool (for example, **Diameter Dimensions**) and click the circle or arc you want to dimension. The dimension appears in green with its value, such as "Ø72.11". Zoom in for precise placement.



Dimensioning tools active with a diameter dimension being added to a circle in the front view

Step 27. Add radius or chamfer dimensions

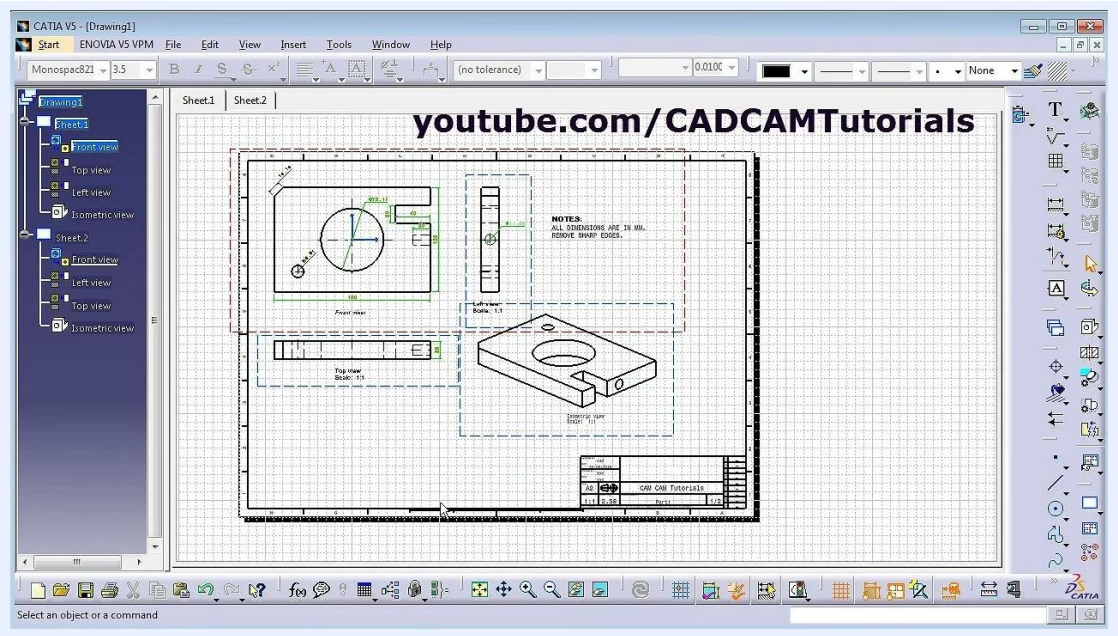
For a radius, select the **Radius Dimension** tool (often marked "R") and click the arc or circle. For a chamfer, select the **Chamfer Dimension** tool and click the chamfered edge. Adjust the position of each dimension for clarity.



Radius and chamfer dimension tools ready to apply to selected geometry

Step 28. Add notes

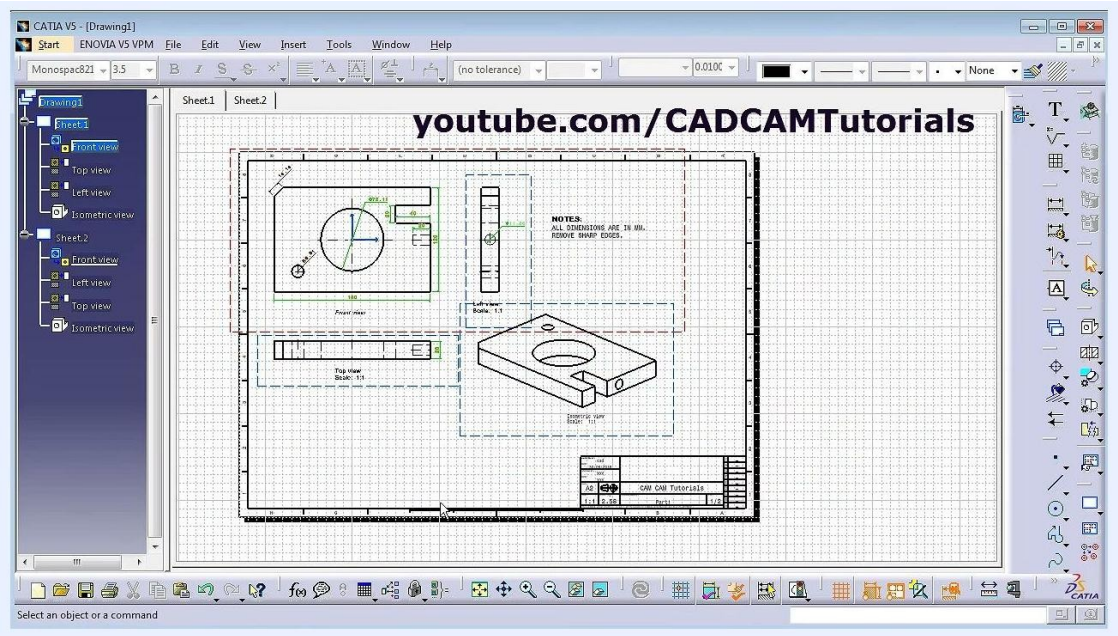
Use the text tool to add notes to the drawing, for example "ALL DIMENSIONS ARE IN MM. REMOVE SHARP EDGES." Place notes in a clear area, typically near the views or in a designated notes section. Include all the details and instructions needed for manufacturing or inspection.



Drawing sheet with notes, views, and dimensions clearly shown

Step 29. Detail multiple parts on one sheet

To detail another part on the same sheet, repeat the process. Open the part file and use the **Front view**, **Projection View**, and **Isometric View** tools to place its views. Add dimensions and notes as required. Organize the views and notes clearly so several parts fit on one sheet.



Drawing sheet with multiple detailed views, notes, and the title block visible

Review the completed drawing

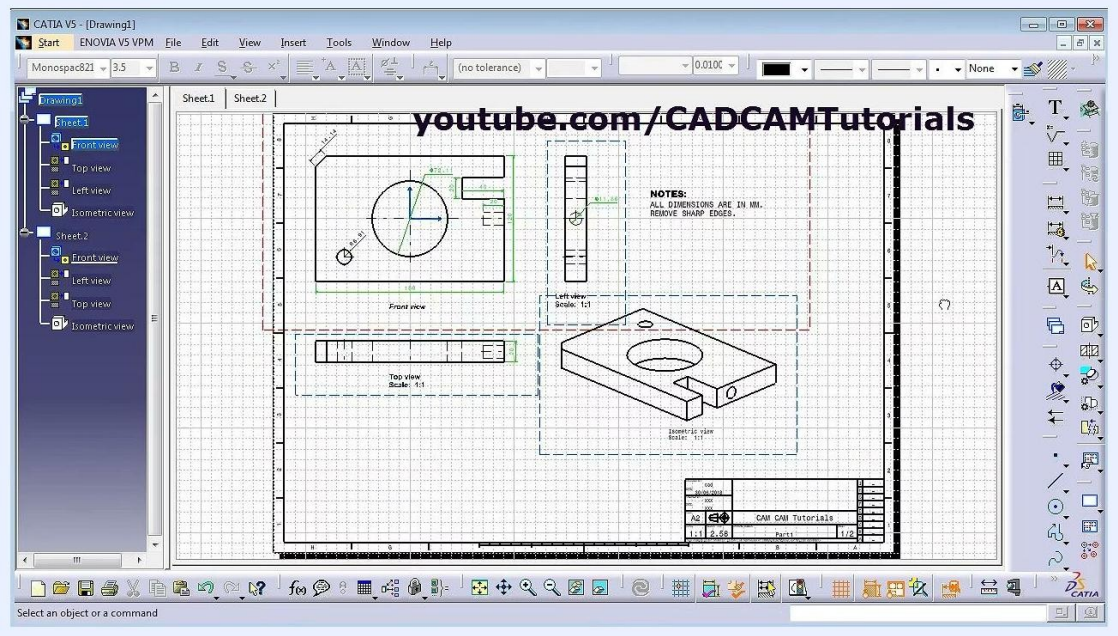
Step 30. Check the sheet and save

Examine the sheet to confirm all required views and details are present:

- The front view (top left), left view (upper center), top view (bottom center), and isometric view (bottom right) are placed and labeled with their scales, such as "Scale: 1:1".
- Each view is enclosed in a dashed colored box.
- Dimensions appear in green, such as the diameter and length values in the front and left views.
- The note reads: "NOTES: ALL DIMENSIONS ARE IN MM. REMOVE SHARP EDGES."
- The title block at the bottom right shows details such as sheet size (A2 in the example), scale (1:1), part name, and drawing number.

- The tree on the left shows Drawing1 with Sheet.1 (its own set of views) and Sheet.2 (active, with Front, Left, Top, and Isometric views).

Confirm that all annotations, notes, and dimensions are legible and correctly placed. Make sure no views or text overlap and that the title block contains the information needed for manufacturing or inspection. Save the drawing file to preserve your work.



Completed CATIA V5 drawing sheet with front, left, top, and isometric views, dimensions, notes, title block, and the drawing tree on the left

Summary

You now know how to do drafting in CATIA V5. You created a drawing from a 3D model, configured the sheet, and formatted labels and annotations. You also displayed hidden lines and centerlines, adjusted view scale, and generated and arranged dimensions. Finally, you added title blocks, extra sheets, manual views, and notes. The result is a complete drawing sheet ready for production or review.

What's next

- Repeat the manual view workflow to draft additional parts on new sheets.
- Try the other preset configurations, such as **All Views** or **Front, Bottom, and Right**, when you create a drawing.