How to convert a 3D model to a 2D drawing in CATIA V5: sprocket gear example

This guide shows you how to convert a 3D model to a 2D drawing in CATIA V5, using a sprocket gear as the example. You open the finished part, switch to the Drafting workbench, and insert the front, projection, and isometric views. You then add dimensions, annotations, and a title block, and export the finished drawing as a PDF.

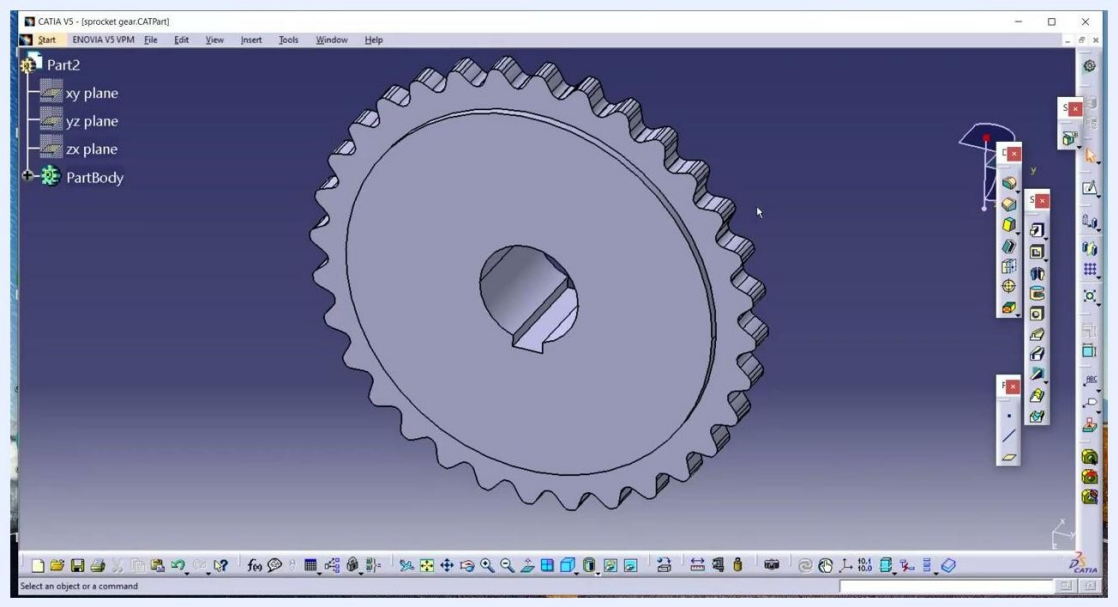
Prerequisites

- CATIA V5 is installed and running.
- Your sprocket gear is fully modeled and ready for drafting.
- The part is open in the workspace, with the part tree (xy plane, yz plane, zx plane, **PartBody**) on the left.

Open the model and create a new drawing

Step 1: Open the final 3D model

Make sure the sprocket gear is fully modeled and open in CATIA V5. The model appears in the workspace, and the part tree is on the left. Confirm the model is complete before you start drafting.



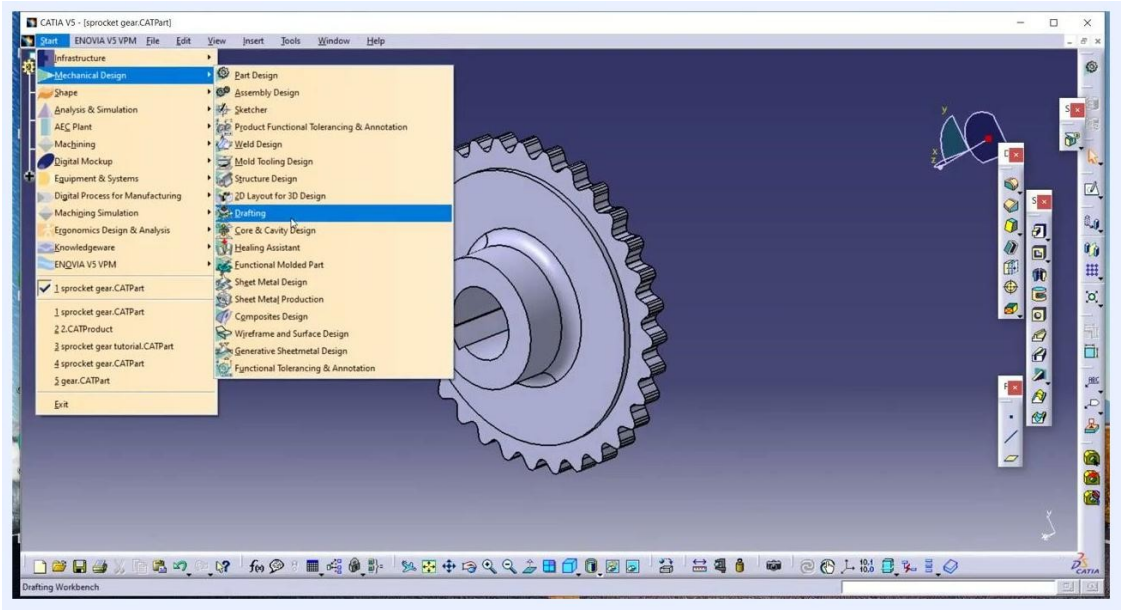
CATIA V5 window showing the completed 3D sprocket gear model, part tree on the left, toolbars on the right and bottom, and the model centered in the workspace

Step 2: Switch to the Drafting workbench

To convert the 3D model to a 2D drawing in CATIA, start in the Drafting workbench.

1. Click the **Start** menu at the top.
2. Hover over **Mechanical Design** in the dropdown.
3. Click **Drafting** in the list of workbenches.

The software may take a moment to switch. Wait until the interface responds and the drafting tools are available.



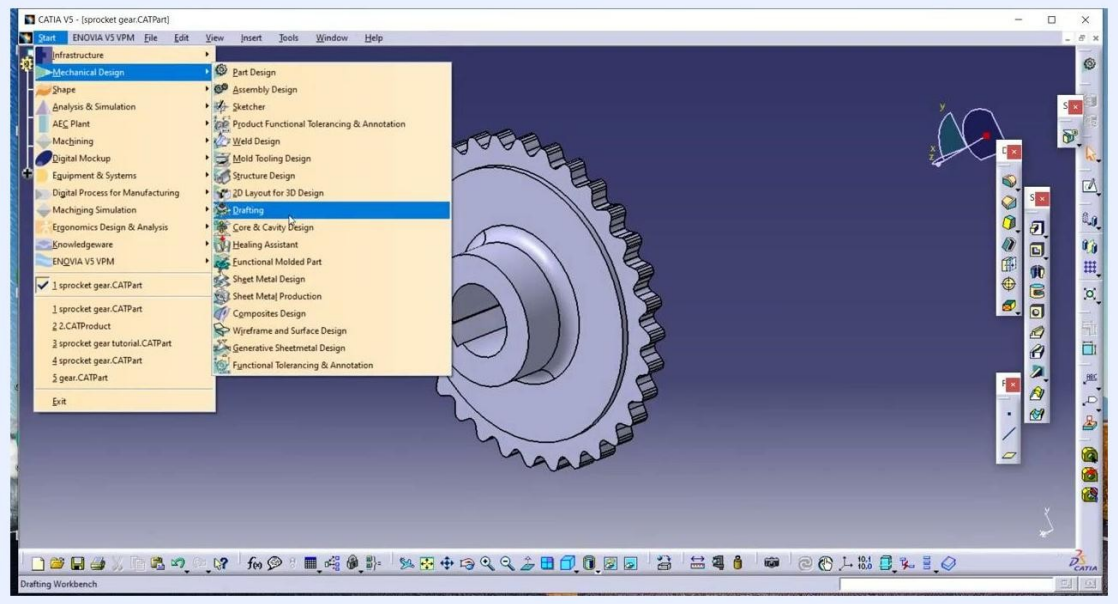
CATIA V5 interface with the Start menu open, Mechanical Design expanded, and Drafting highlighted in the dropdown list

Step 3: Start a new drawing

When prompted, the **New Drawing Creation** dialog appears.

1. Select an automatic layout for your drawing. For example, choose the layout with front, bottom, and right views.
2. Confirm the paper size is 210 x 297 mm and the global scale is 1:1.

3. Click **OK**, or click **Modify** to adjust the settings further.



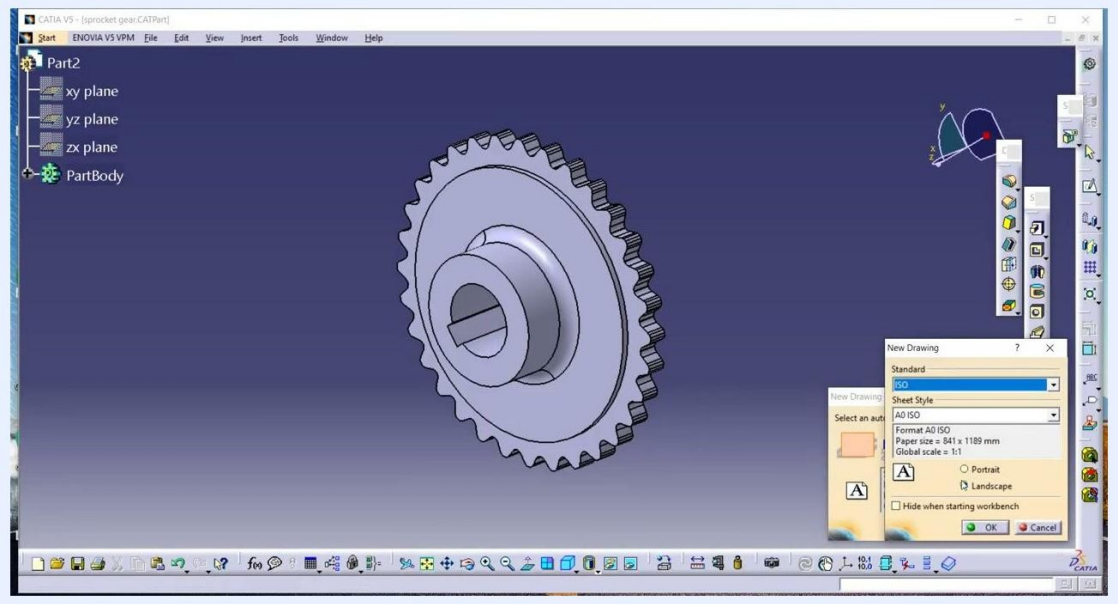
CATIA V5 with the New Drawing Creation dialog open, showing layout options and paper size settings

Step 4: Adjust the paper size and orientation

In the **New Drawing** dialog, set your sheet options.

1. Set the standard to **ISO**.
2. Choose the sheet style you need (for example, A0 ISO).
3. Select **Portrait** or **Landscape**, as required.

4. Click **OK** to confirm.

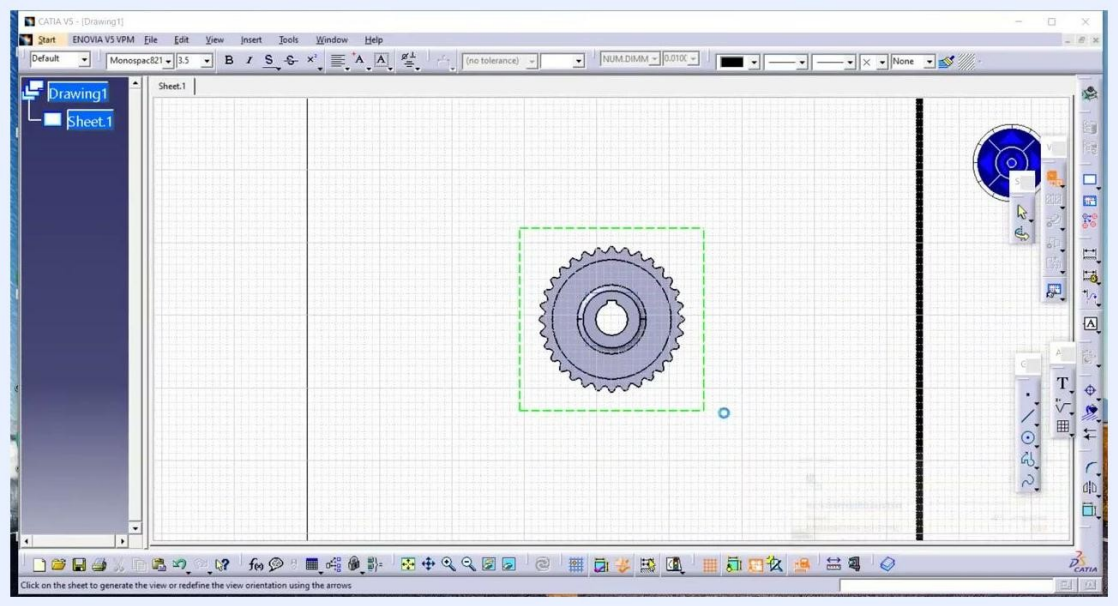


CATIA V5 with the New Drawing dialog open, ISO standard selected, and options for Portrait or Landscape orientation

Insert and scale the front view

Step 5: Start the front view tool

In the Drafting workbench, find the view creation toolbar on the right side of the screen. Select the **Front View** tool (hover text: "Advanced Front View"). Click the **Advanced Front View** icon to begin inserting the front view of your 3D sprocket gear.



CATIA V5 Drafting workbench with the Advanced Front View icon highlighted in the right toolbar, ready to insert a front view

Step 6: Select the reference face in the 3D model

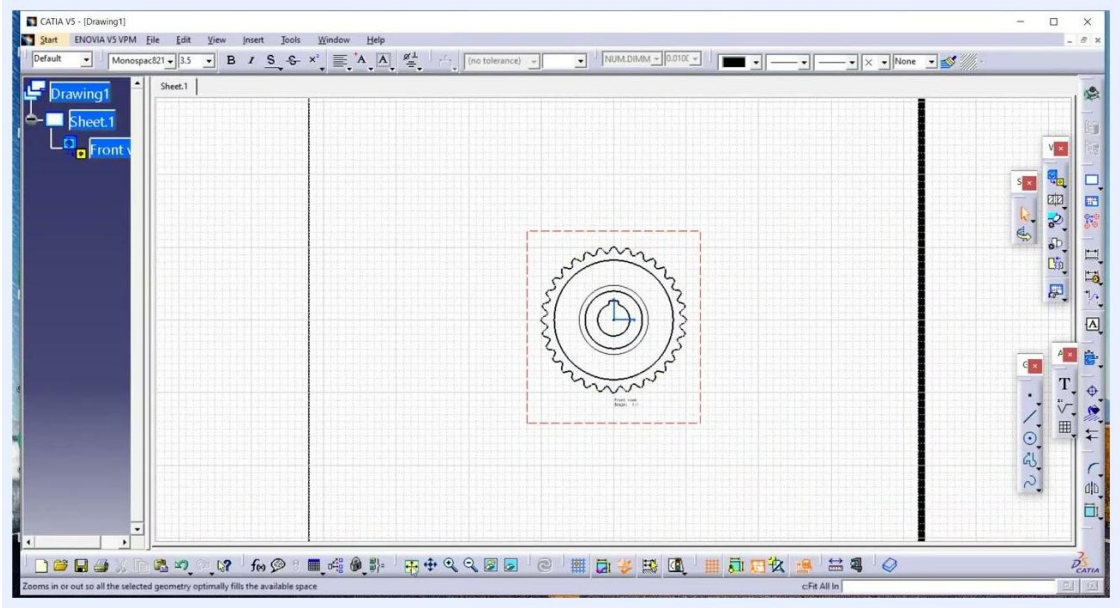
Switch to the 3D model window so you can pick the face for the front view.

1. Click **Window** in the top menu.
2. In the dropdown, select your 3D model file (for example, sprocket gear.CATPart) to bring the model to the foreground.
3. Click the face of the sprocket gear that you want to use as the front view reference.

The selected face is highlighted, and the software generates a preview of the front view on your drawing sheet.

Step 7: Place the front view on the drawing sheet

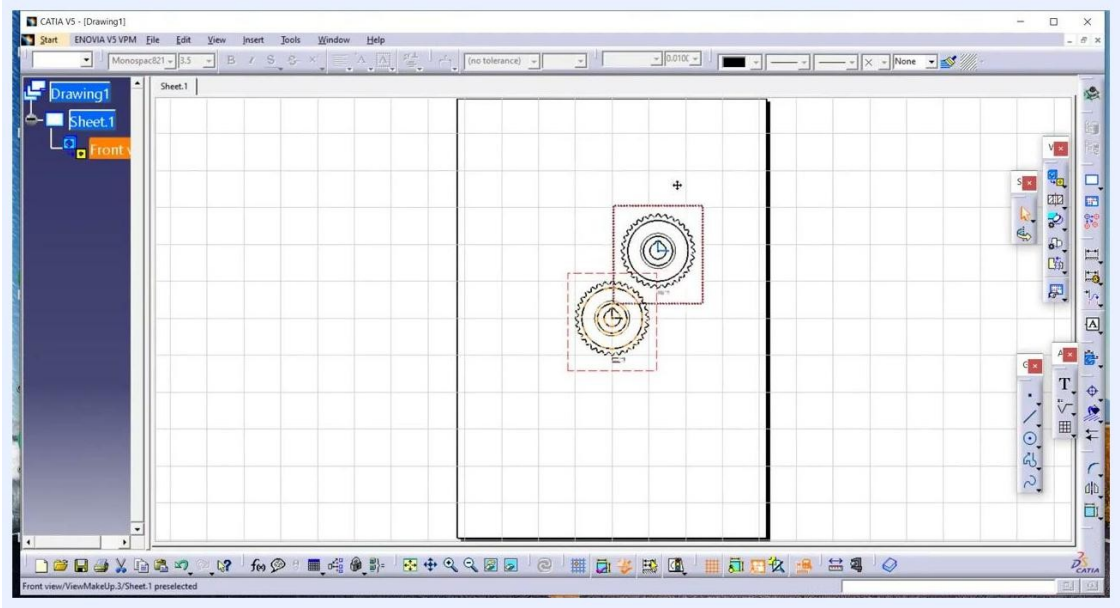
Return to the drawing sheet and click the location where you want the front view. The front view appears, enclosed in a dashed selection box.



CATIA V5 drawing sheet with the front view of the sprocket gear placed, highlighted with a dashed selection box

Step 8: Move the front view

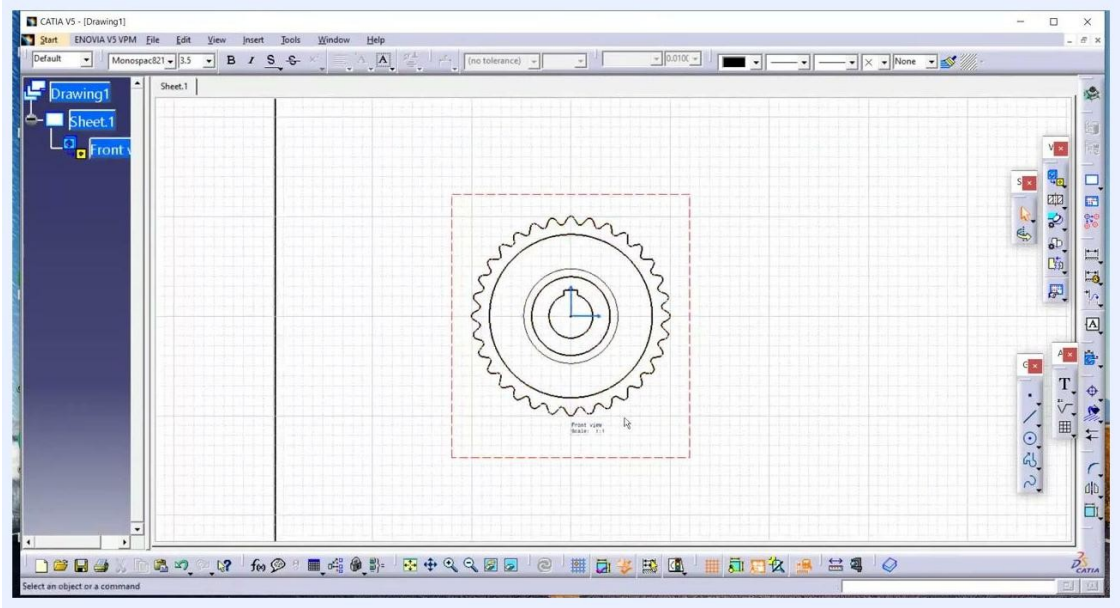
Click and drag the front view to reposition it on the sheet. Arrange it for the best placement and clarity.



CATIA V5 drawing sheet showing two front views of the sprocket gear, one being moved to a new position

Step 9: Change the scale of the front view

First, check the default scale. The label below the front view shows the current scale (for example, "Front view Scale 1:1").

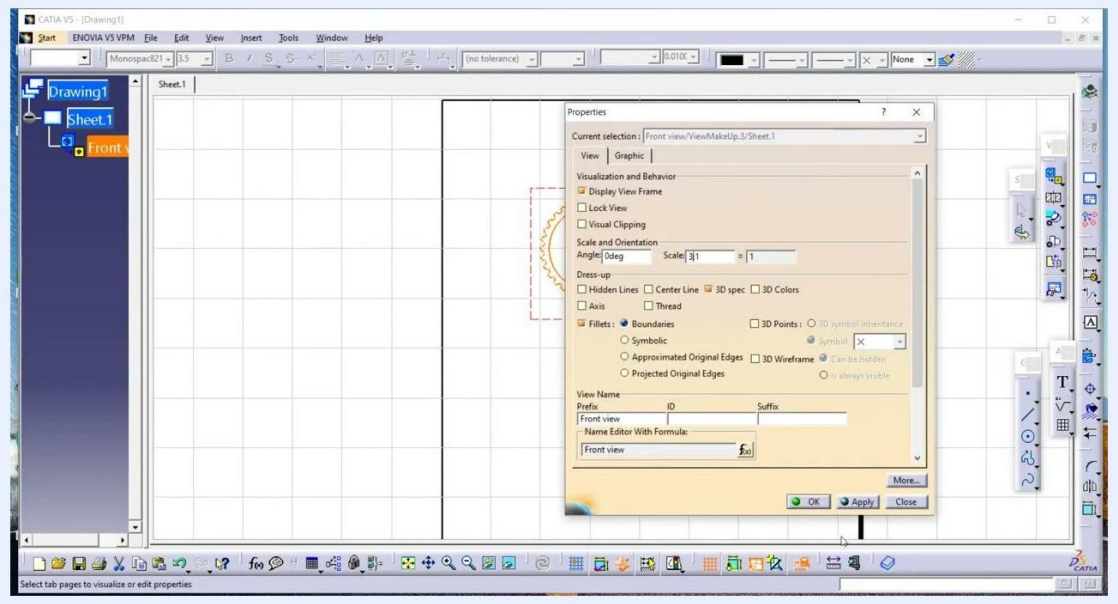


CATIA V5 drawing sheet with the front view of the sprocket gear, scale label visible as "Front view Scale 1:1"

Then change it.

1. Right-click the front view and select **Properties**.
2. In the Properties dialog, open the **View** tab and find the **Scale** field.
3. Enter the scale you want (for example, change 1:1 to 3:1 to enlarge the view).

4. Click **Apply** or **OK**.



CATIA V5 Properties dialog open for the front view, with the Scale field set to 3:1

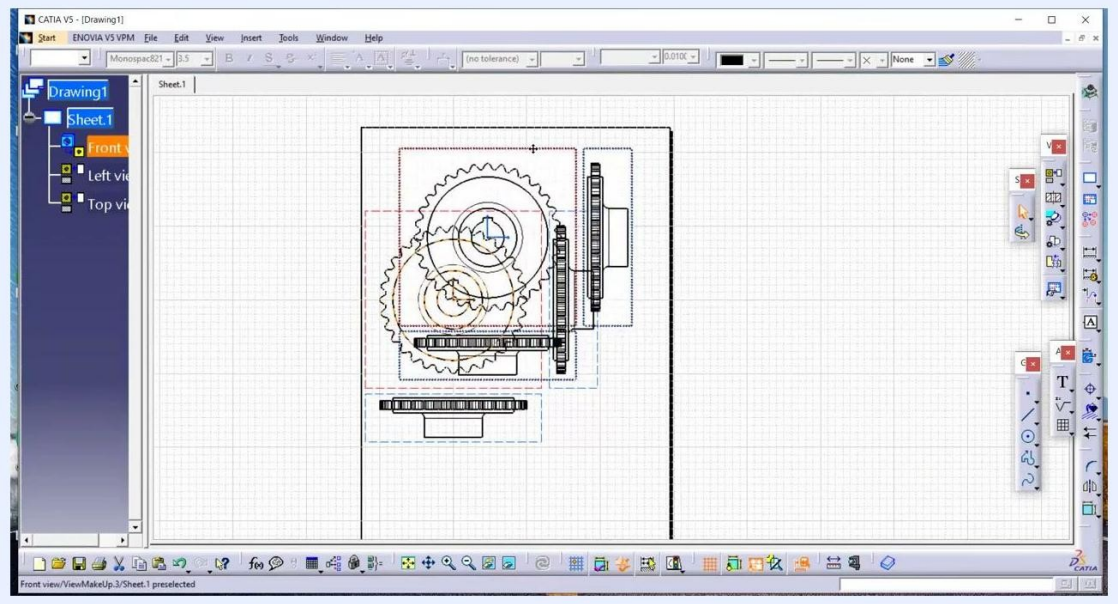
You have now inserted and scaled the front view of the sprocket gear on the 2D drawing sheet. Continue with the projection views below.

Add projection views

Step 10: Set the projection method

1. Right-click the sheet background and select **Properties**.
2. In the Properties dialog, find the **Projection Method** section.
3. Choose **First angle standard** or **Third angle standard**, as your drawing requires.

4. Click **Apply** or **OK**.

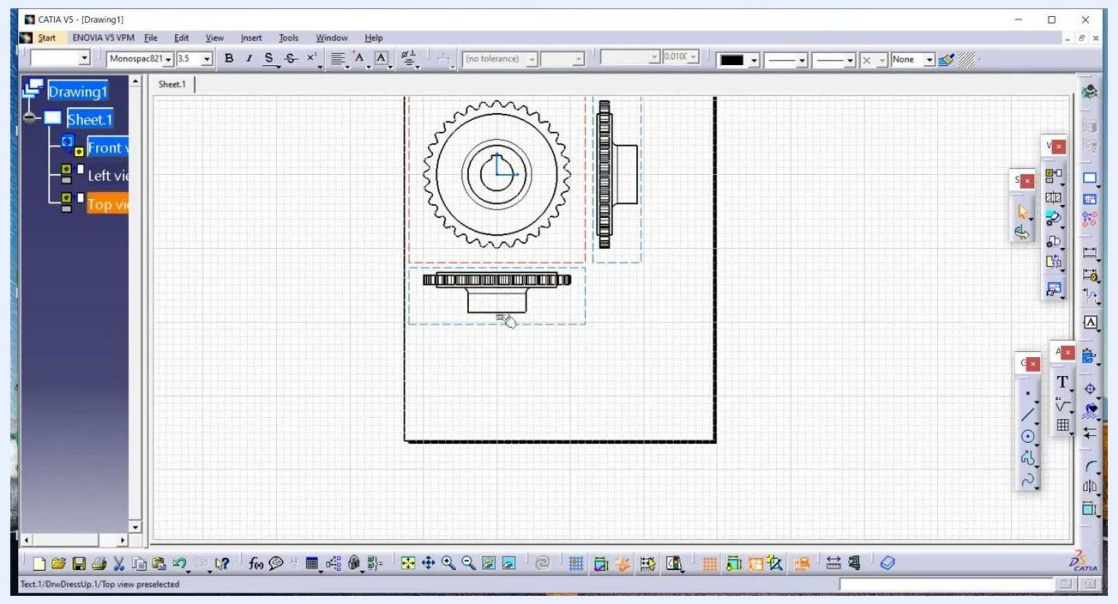


CATIA V5 Properties dialog open for the drawing sheet, showing options for First angle and Third angle projection standards

Step 11: Insert a projection (side) view

1. In the Drafting workbench, select the **Projection View** tool from the right-side toolbar.
2. Click the front view to generate a projection view (for example, a left or right side view).
3. Move the cursor to the location you want, typically to the right of the front view, and click to place the side view.

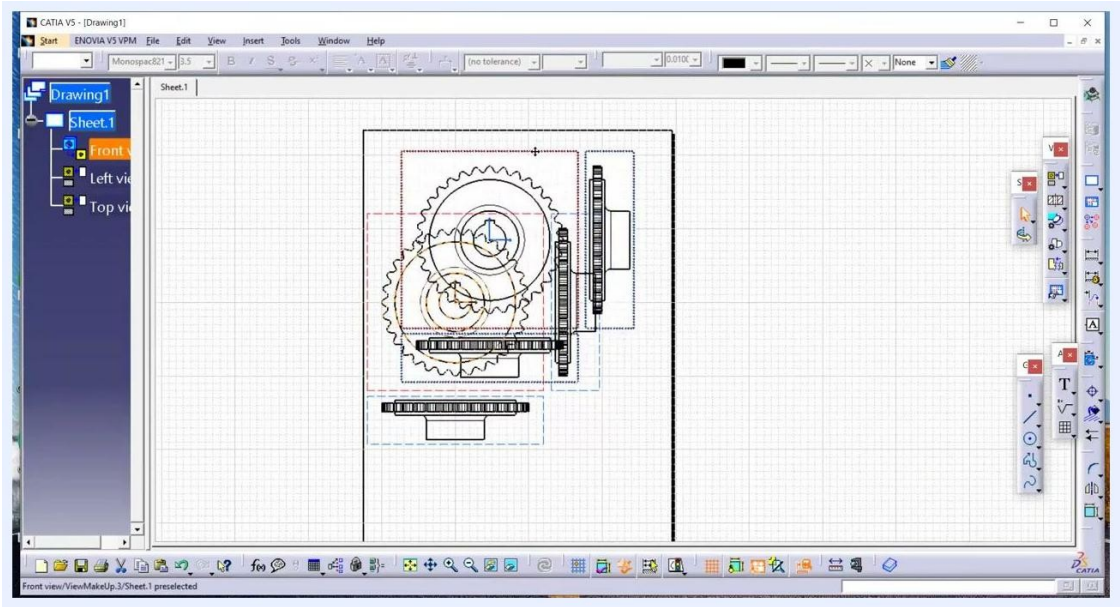
Keep a suitable gap between the front and side views for clarity.



CATIA V5 drawing sheet with both front and left (side) views of the sprocket gear, each in its own selection box

Step 12: Insert additional projection views

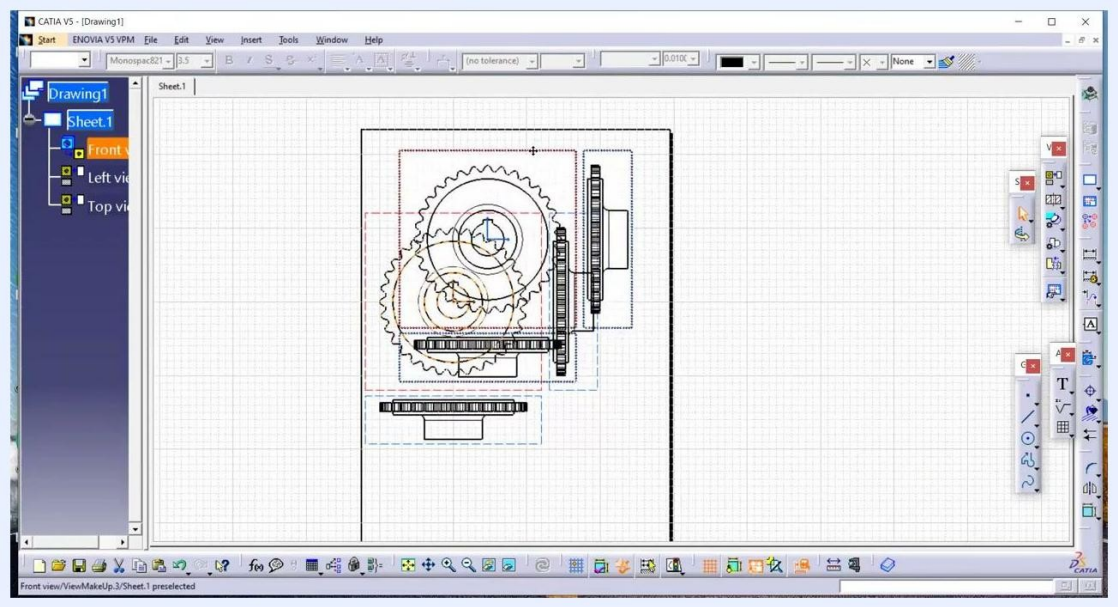
Use the **Projection View** tool again to add more views, such as a top or bottom view. Click below the front view to insert a top or bottom projection view. Arrange the front, side, and top views according to standard engineering drawing conventions.



CATIA V5 drawing sheet with front, left, and top views of the sprocket gear, each in a colored selection box

Step 13: Arrange and align the principal views

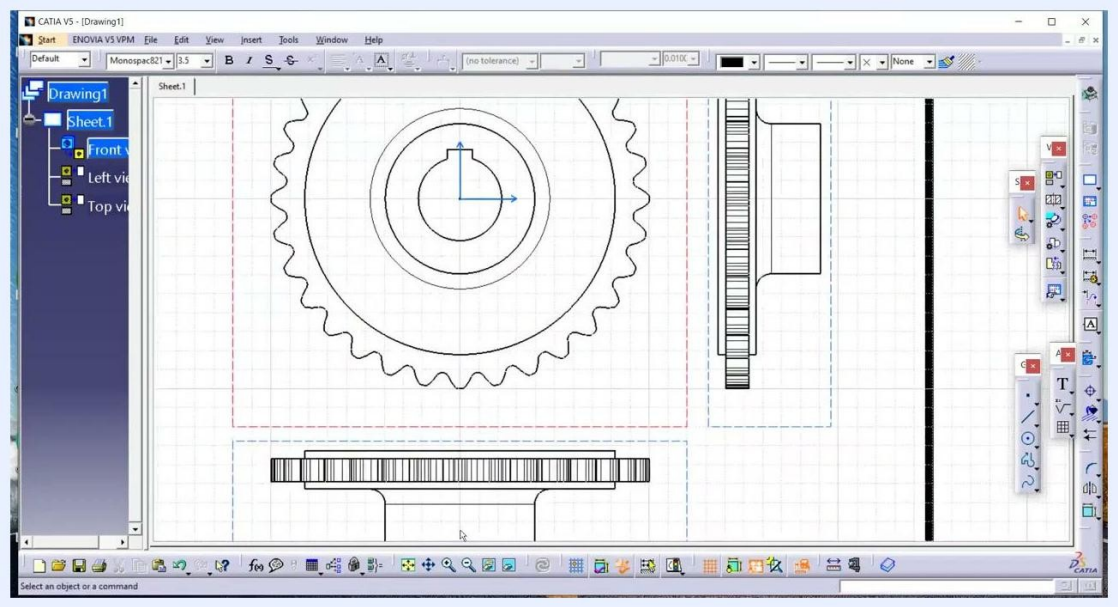
Adjust the positions of the front, side, and top views so they are aligned and evenly spaced. Confirm that all three principal views are clearly displayed on the sheet.



CATIA V5 drawing sheet with the three principal views (front, left, top) of the sprocket gear, neatly arranged

Step 14: Zoom in to verify detail

Zoom in on the drawing sheet to inspect the alignment and detail of each view. Make sure all features are accurately represented and the views are correctly scaled.



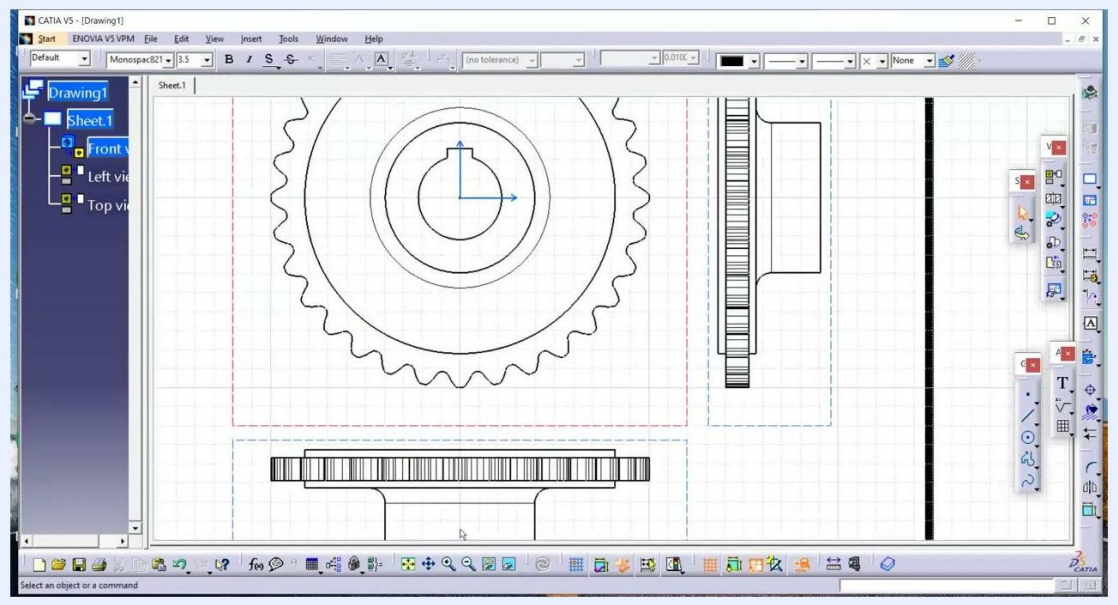
CATIA V5 zoomed-in view of the front, left, and top views of the sprocket gear, showing detailed alignment

Step 15: Customize view properties

You can control hidden lines, axes, center lines, and threads for each view.

1. Select each view (front, side, top) in the tree or on the sheet.
2. Right-click and choose **Properties**.

3. In the Properties dialog, enable or disable display options such as hidden lines, axis lines, center lines, and thread representation, as needed for technical clarity.



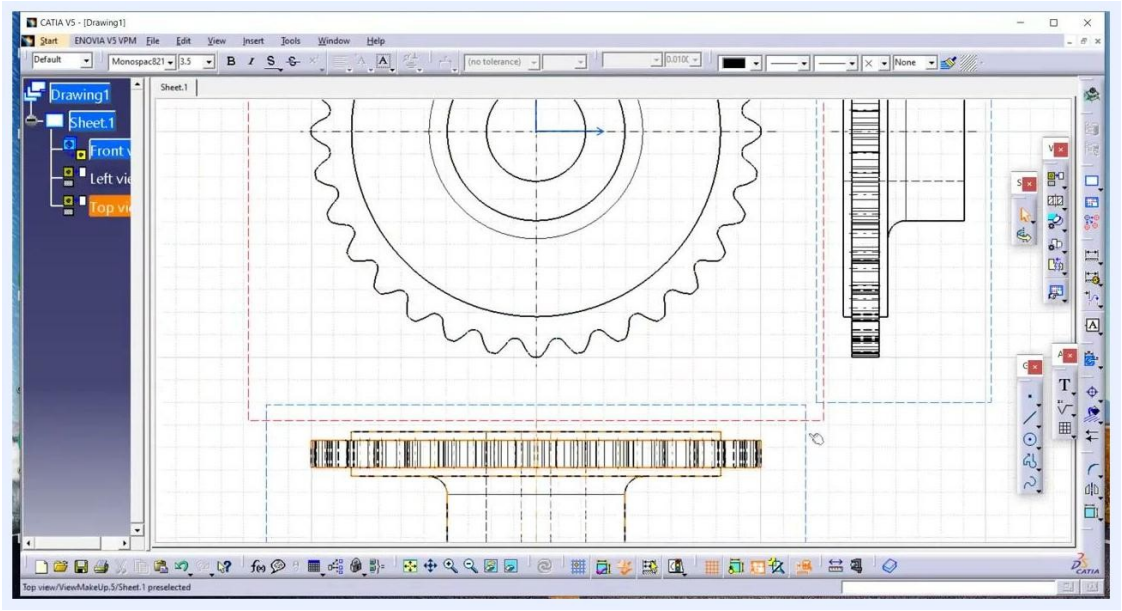
CATIA V5 drawing sheet with front, left, and top views selected, ready for property customization

Step 16: Confirm the view settings

The **Properties** dialog shows options for visualization and behavior, scale and orientation, and dress-up features. Confirm or adjust these settings:

- Angle: 0 deg
- Scale: 2:1
- **Center Line** checked under Dress-up
- Other options, such as **Hidden Lines**, **Axis**, **Thread**, and **Fillets**, as needed

Click **OK** or **Update** to apply the changes. You can follow the update progress in the status window at the bottom right.



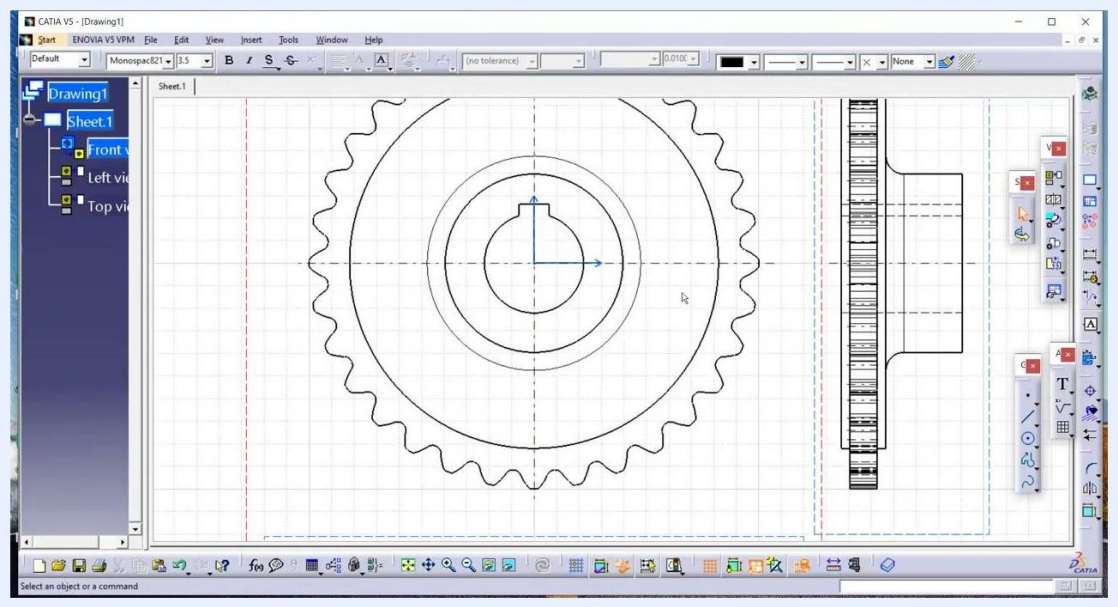
CATIA V5 Properties dialog open, showing settings for view visualization, scale, and dress-up options. Update progress bar visible at bottom right.

The drawing area now shows the gear with visible center lines and axes (XOX axis). Confirm that all properties display as expected.

Add dimensions and annotations

Step 17: Start dimensioning

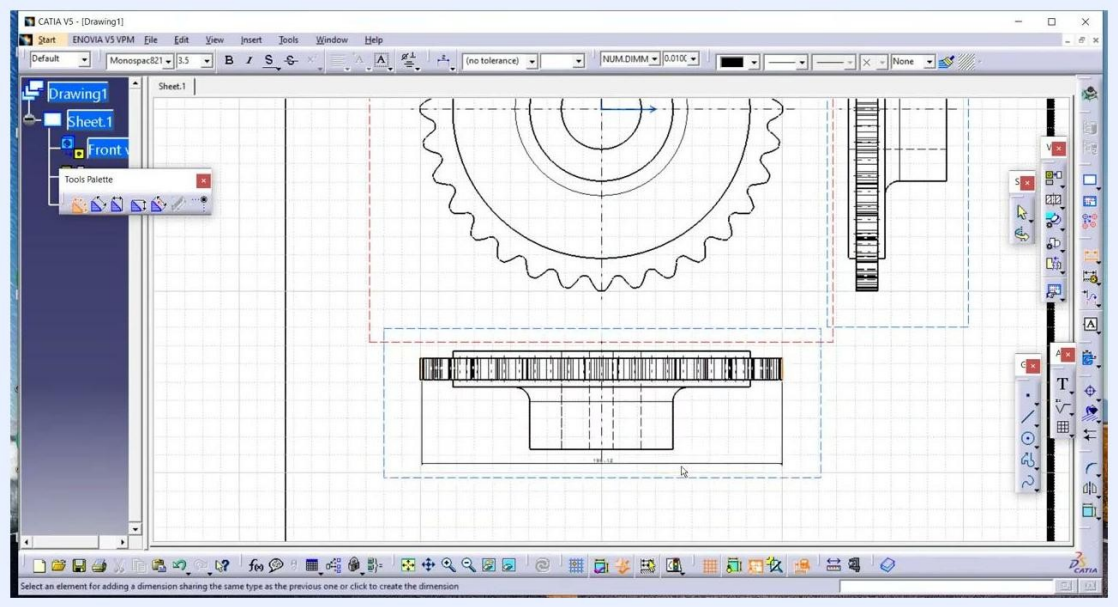
Select the dimensioning tool from the right-side toolbar or the floating dimension palette. The drawing is now ready for dimensioning, and no dimension lines appear yet.



Drawing area with gear and axes visible, ready for dimensioning. No dimension lines yet.

Step 18: Create a linear dimension between two points

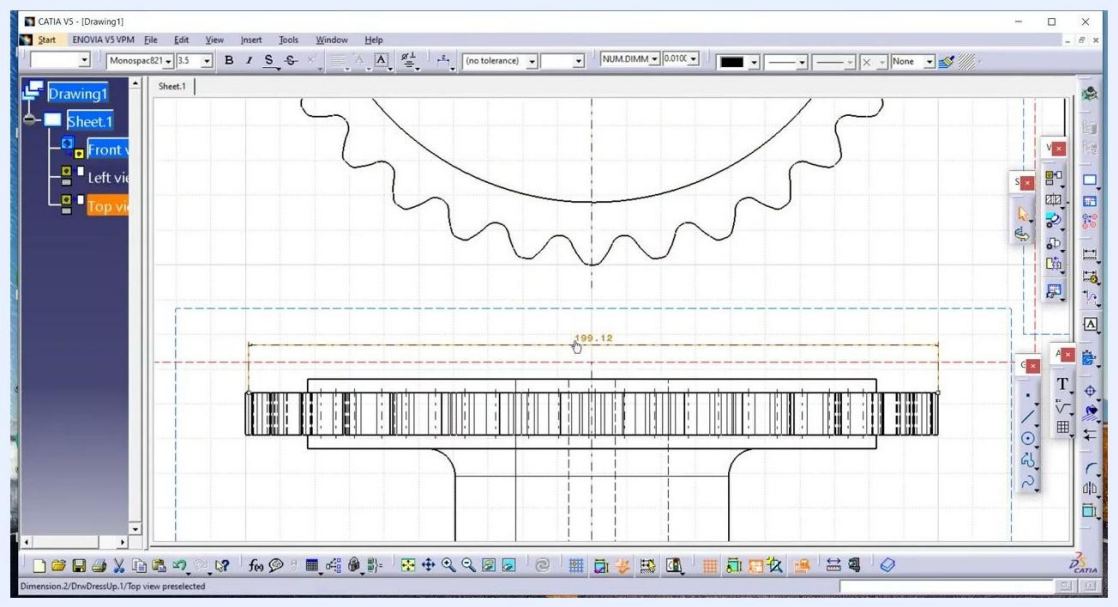
Select the view you want, for example the side view of the gear. Click the first point, then the second point. The dimension line appears between the two points.



Dimension line displayed between two points on the side view of the gear.

Step 19: Add a diametral dimension

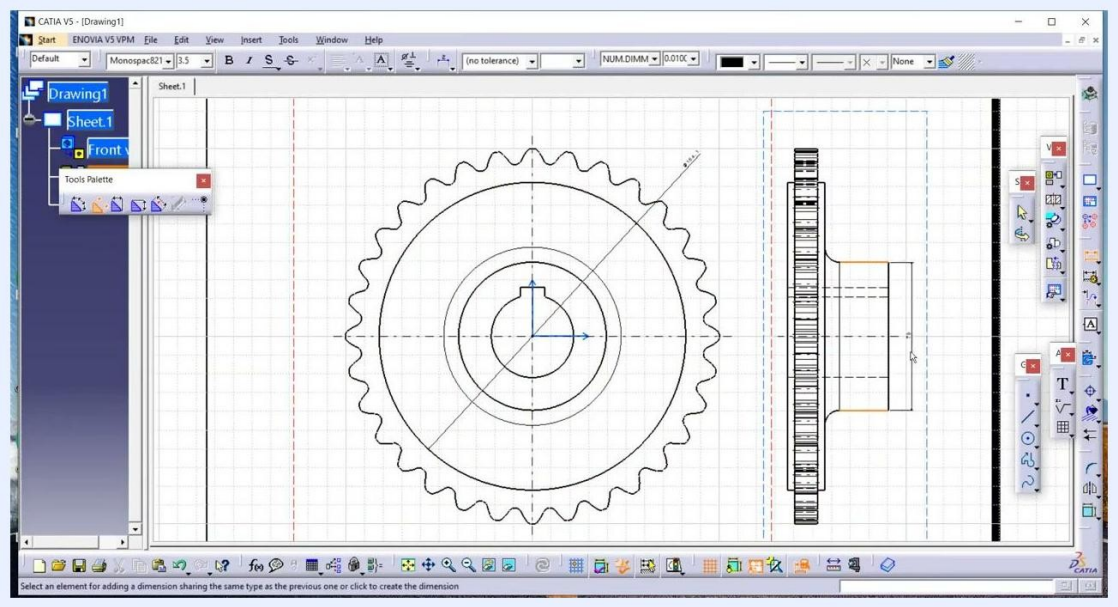
Select the diametral dimension tool from the palette, then click the circular feature you want to dimension. The diametral dimension (for example, 199.12) appears in the drawing area.



Diametral dimension (199.12) displayed above the gear drawing.

Step 20: Add more dimensions or annotations

Use the dimensioning tools to add further dimensions. Use the text tool to add annotations or numbers, such as a label that indicates a specific feature or count.

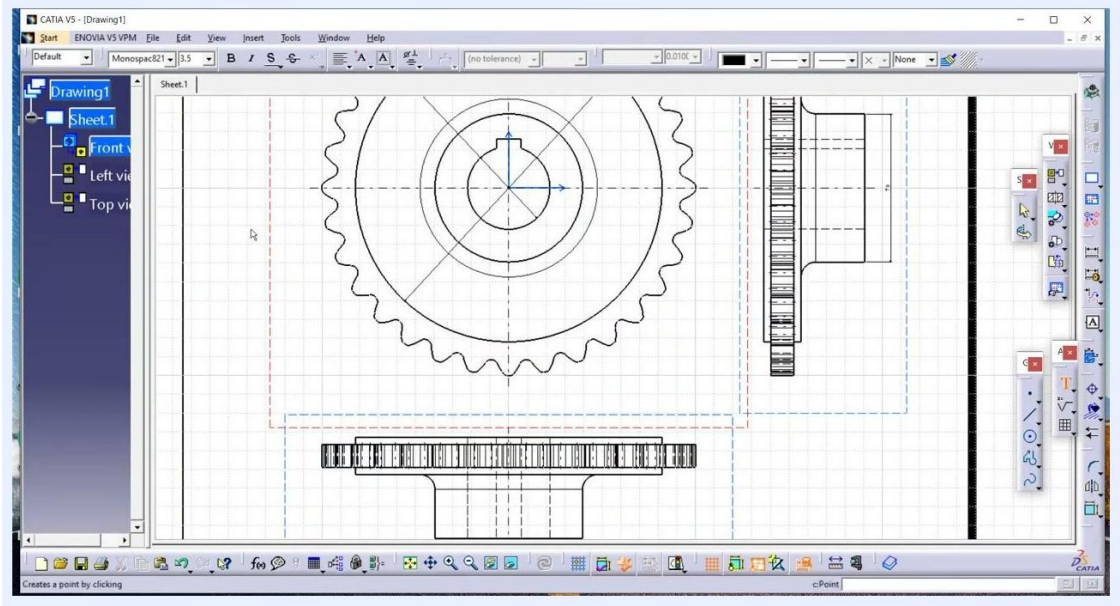


Drawing with additional dimension lines and annotation tools visible.

Step 21: Add a text annotation for the number of teeth

1. Select the text tool from the right-side toolbar.
2. Click in the drawing area above the gear to place the annotation.
3. Enter "Number of tooth" in the **Text Editor** dialog box.

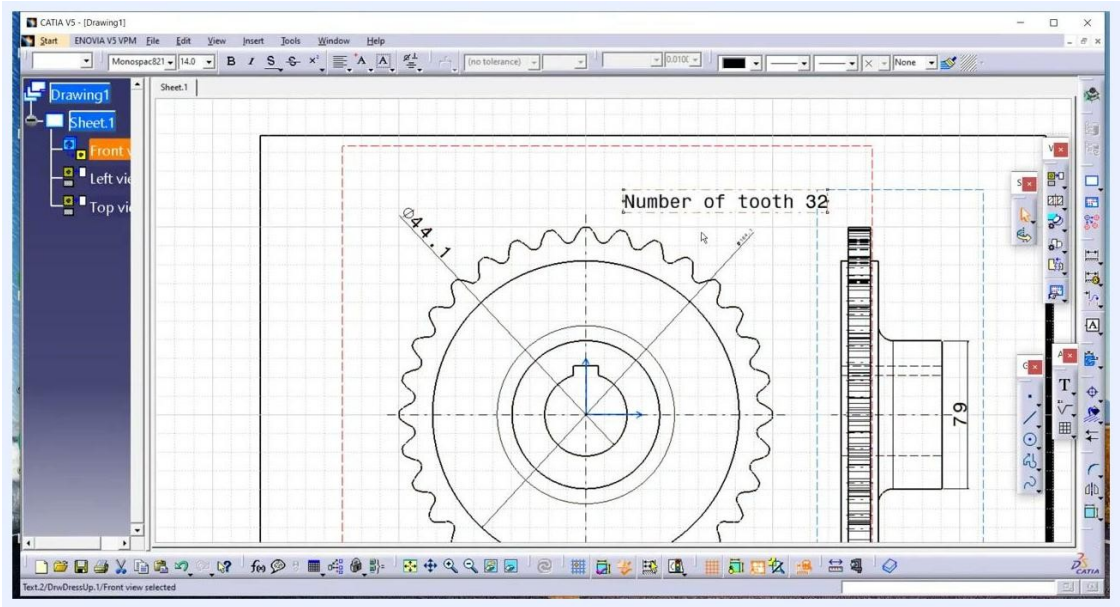
4. Click **OK** to confirm and place the text.



Text Editor dialog open with the annotation text being entered above the gear drawing.

Step 22: Edit and position the annotation

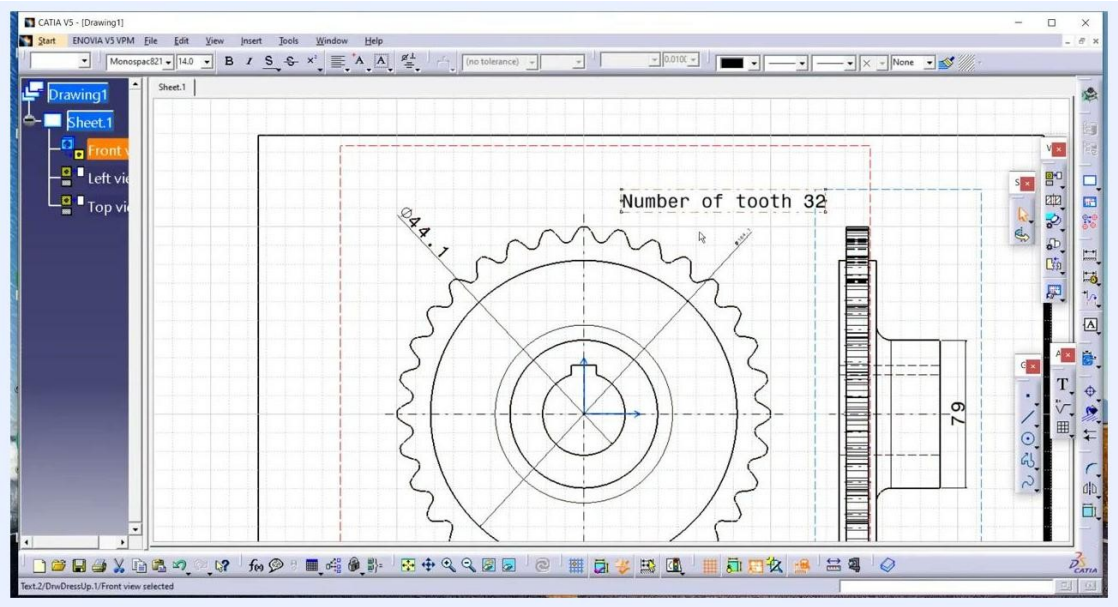
Update the annotation to the correct value if needed (for example, "Number of tooth 32"). Use the toolbar at the top to adjust the font size and style (for example, set the font size to 14.0). Move the annotation to the location you want above the gear drawing.



Annotation "Number of tooth 32" placed above the gear drawing, font size set to 14.0.

Step 23: Review the updated layout

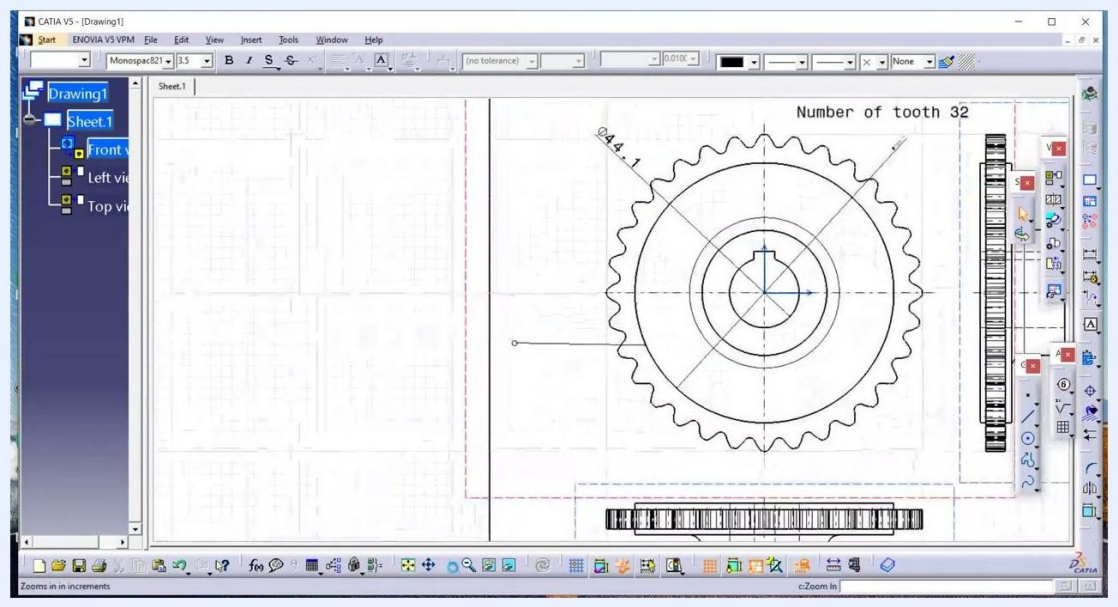
Check that the annotation is clearly visible and correctly positioned. Confirm that all dimension lines (for example, diameter 44.1 and width 79) and annotations are present and legible. The drawing now shows the gear with all required dimensions and the "Number of tooth 32" annotation.



Drawing layout with all views, dimensions, and "Number of tooth 32" annotation visible.

Step 24: Zoom in or out for clarity

Use the zoom tool in the bottom toolbar to focus on specific areas of the drawing. Verify that all details stay clear and that annotations do not overlap other elements.



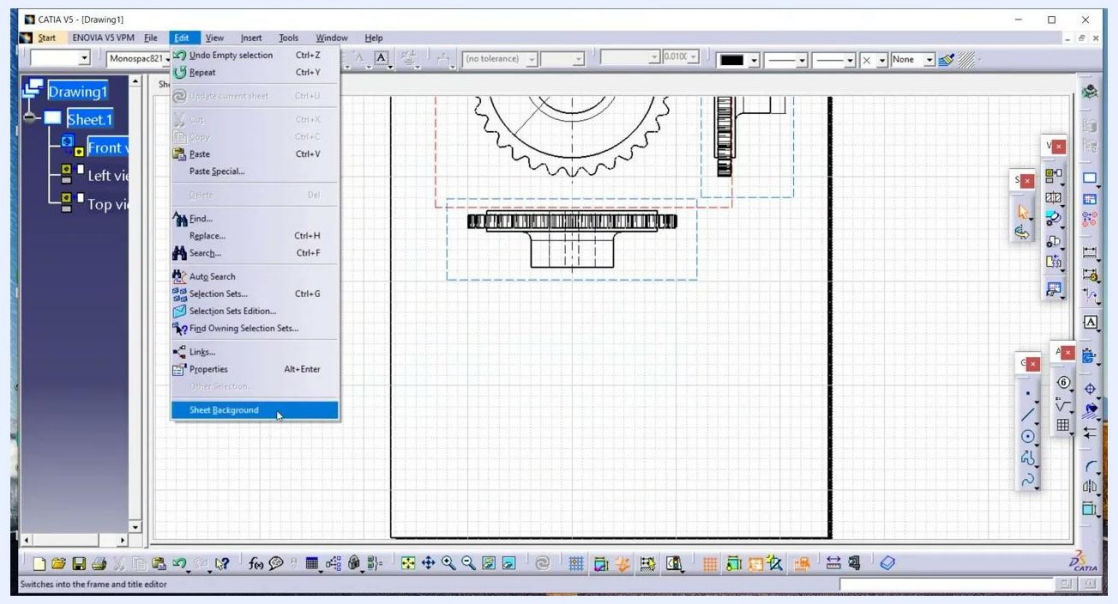
Drawing zoomed in to focus on the gear and annotation "Number of tooth 32".

Add the frame and title block

Step 25: Switch to the sheet background

1. Click **Edit** in the top menu.
2. Select **Sheet Background** at the bottom of the dropdown.

This lets you modify the drawing's frame and title block area.



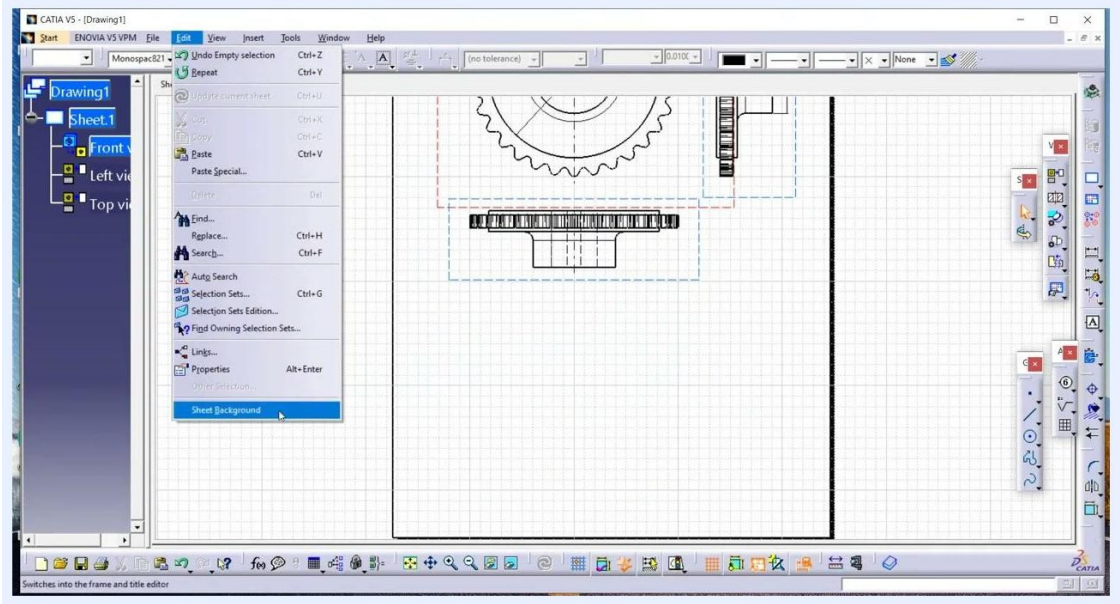
Edit menu open with "Sheet Background" highlighted, drawing area showing gear views and space for title block.

Step 26: Insert a title block

After you switch to the sheet background, the **Manage Frame And Title Block** dialog appears.

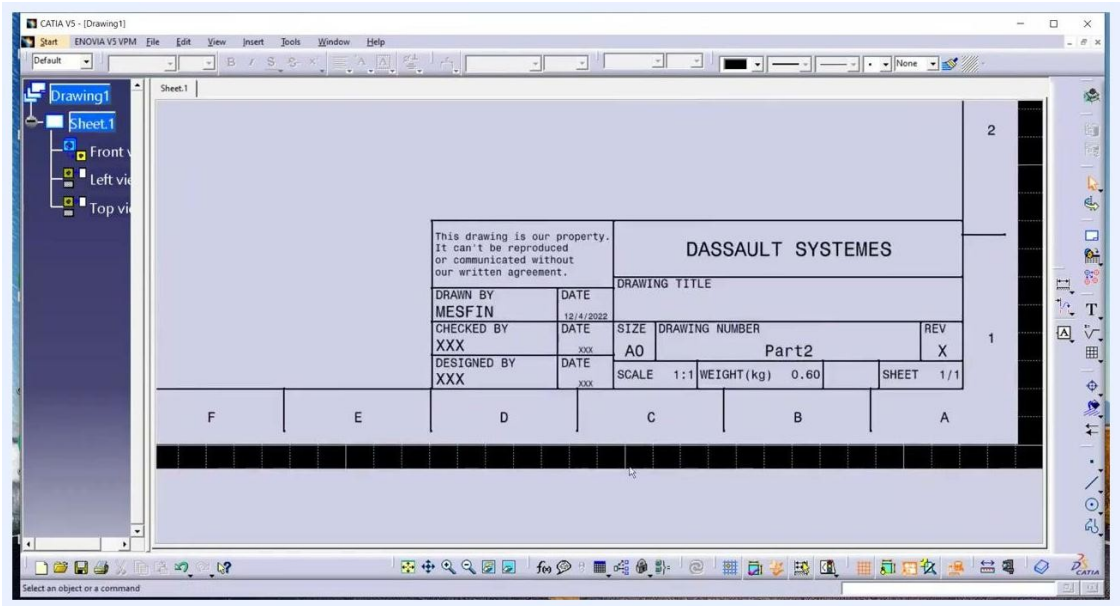
1. Open the **Style of Title Block** dropdown.
2. Select a style, such as **Drawing Titleblock Sample 2**.
3. Under **Action**, make sure **Create** is selected.

4. Check the style in the dialog's preview pane.



Manage Frame And Title Block dialog open, "Drawing Titleblock Sample 2" selected, preview of title block visible.

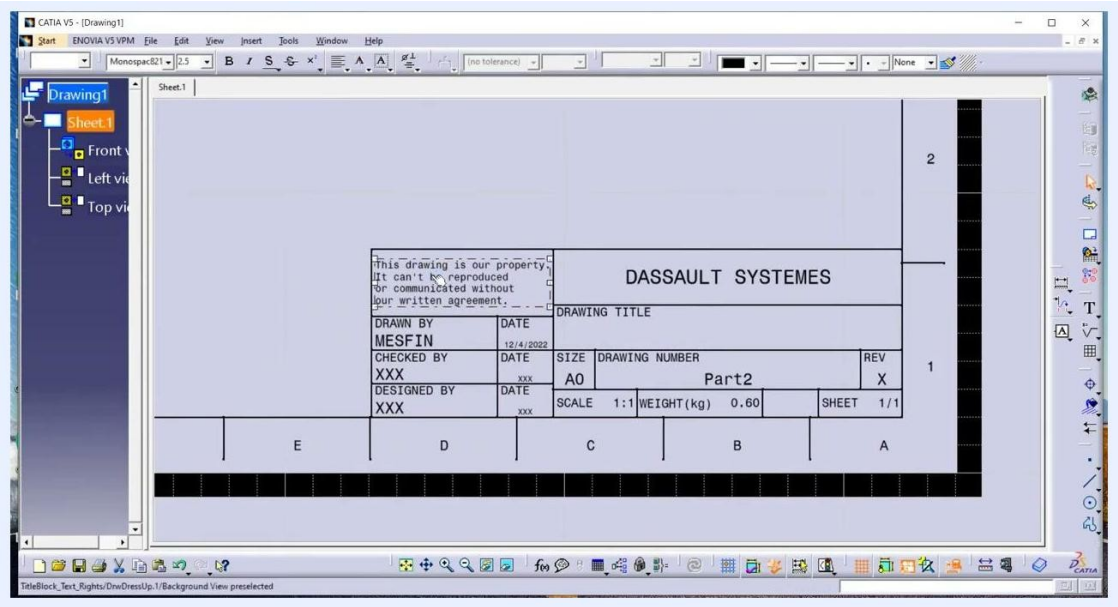
Click **OK** or **Apply** to insert the title block. It appears at the bottom of the sheet with default information: company name, drawing title, author, date, size, drawing number, revision, scale, weight, and sheet number.



Title block inserted at the bottom of the drawing sheet, showing default fields and values.

Step 27: Edit the title block information

Click any text field in the title block to activate it for editing. For example, click the company name field ("DASSAULT SYSTEMES") or the property notice. Use the text editing tools at the top (font size, style, and so on) as needed.

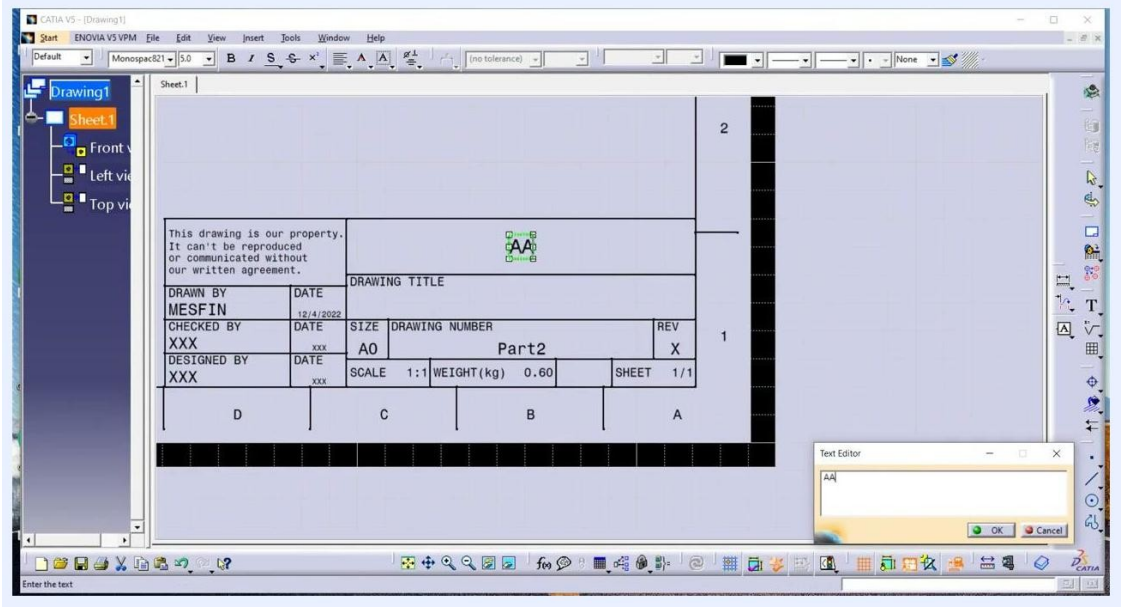


Title block with editable text fields, "DASSAULT SYSTEMES" and property notice selected for editing.

To update the company name or another field:

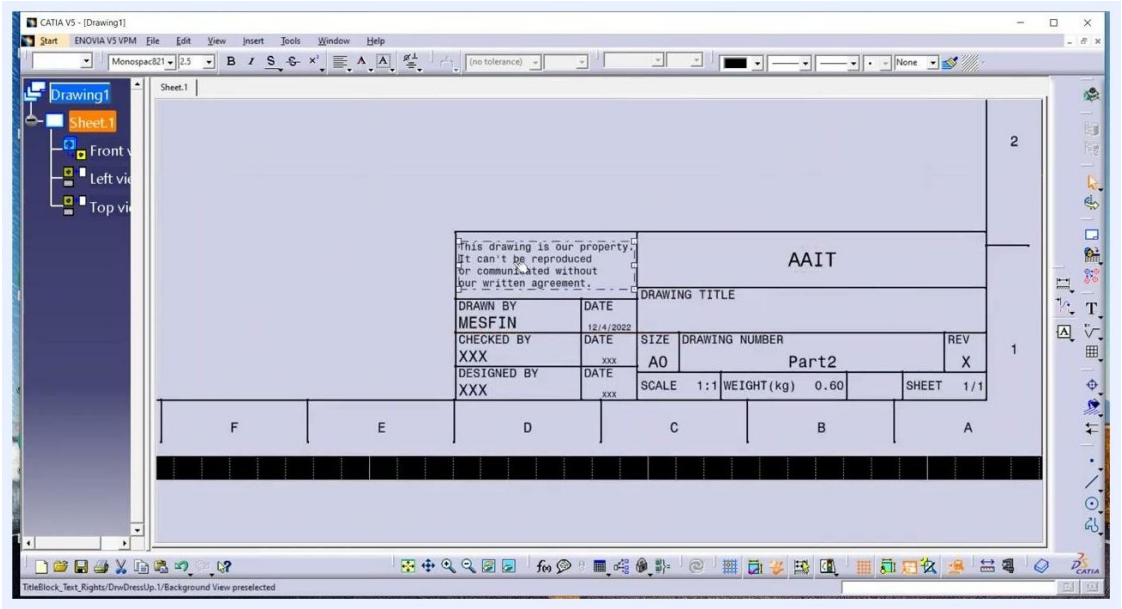
1. Select the field and enter the new value (for example, change "DASSAULT SYSTEMES" to "AAIT").
2. Type the new text in the **Text Editor** dialog that appears.

3. Click **OK** to confirm.



Text Editor dialog open with the new company name being entered, company name field selected in the title block.

Verify that the new value (for example, "AAIT") appears in the title block, and that all other required fields are accurate and current.



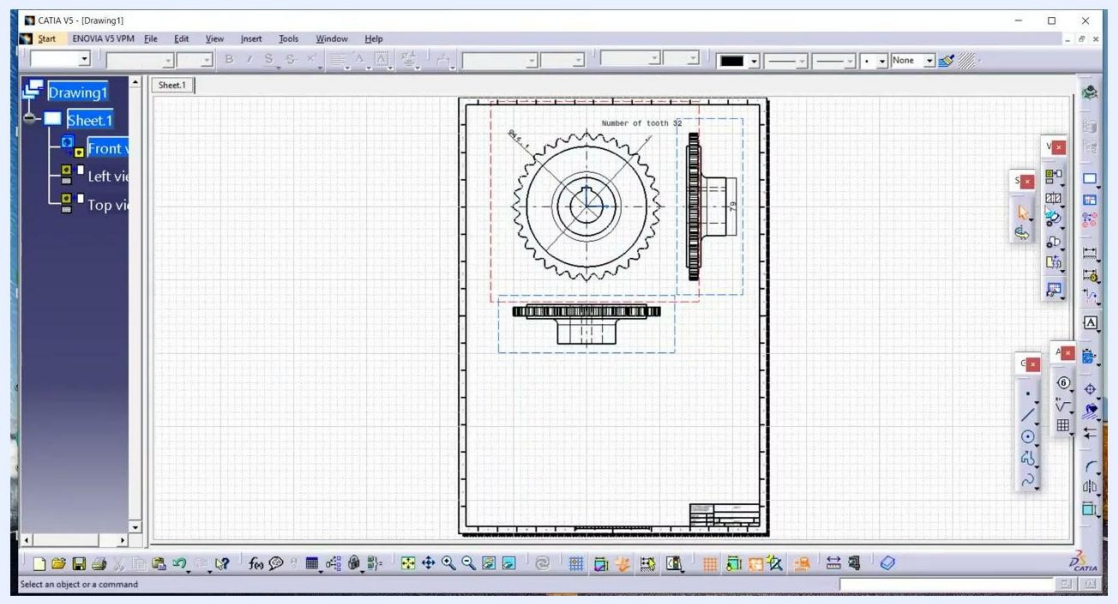
Title block updated with "AAIT" as the company name, all fields visible and editable.

Step 28: Fill out the title block with drawing information

Enter the following details in the title block:

- **Drawn By:** MESFIN
- **Date:** 12/4/2022
- **Checked By:** john
- **Date:** xxx (replace with the actual date if available)
- **Designed By:** XXX (replace with the actual name if available)
- **Date:** xxx (replace with the actual date if available)
- **Drawing Title:** AAIT
- **Size:** A0
- **Drawing Number:** Part2
- **Revision:** X
- **Scale:** 1:1
- **Weight (kg):** 0.60
- **Sheet:** 1/1

Make sure all fields are completed. Confirm that the note "This drawing is our property. It can't be reproduced or communicated without our written agreement." is present. Verify that the sheet grid (A-F, 1-2) is visible for reference.



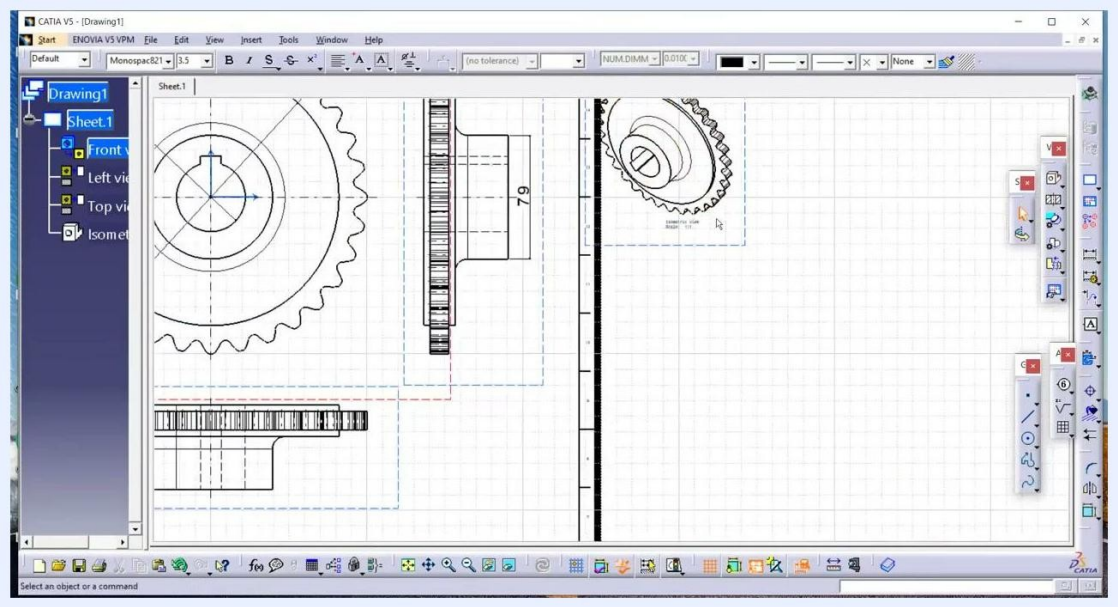
Title block filled with all required information, including names, dates, drawing title, size, drawing number, revision, scale, weight, and sheet number. The CATIA V5 interface is visible with the title block in focus.

Then confirm that all entered information (paper size, weight, sheet number, and so on) displays correctly. Double-check accuracy and completeness before you continue.

Add an isometric view

Step 29: Activate the isometric view tool

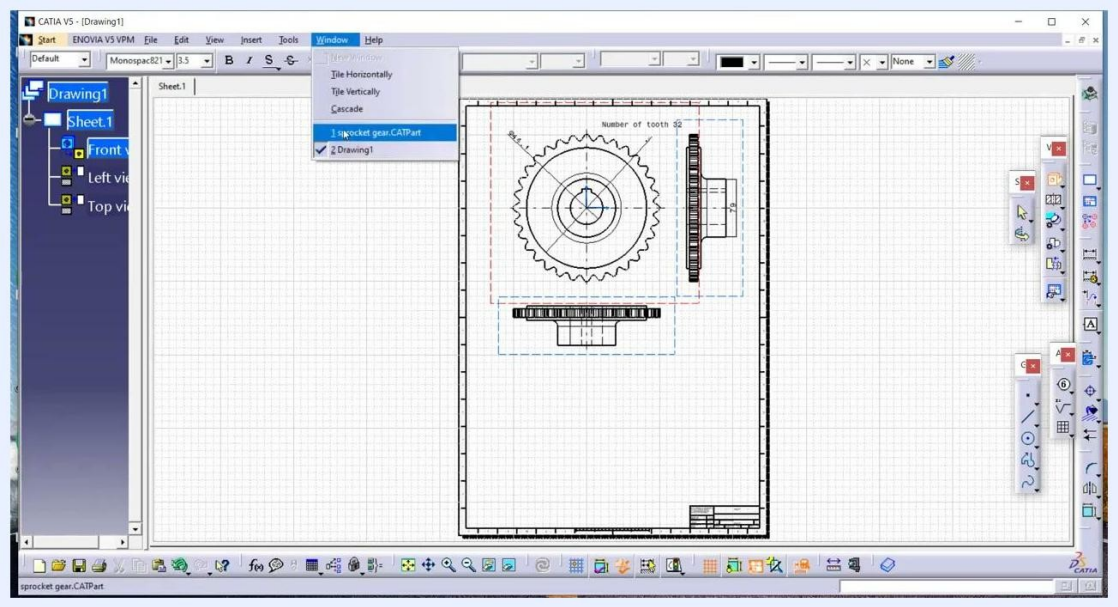
Find the isometric icon in the CATIA V5 toolbar and click it to activate the isometric view function. In the tree on the left, make sure **Isometric** is selected under **Sheet.1**.



Drawing sheet with front, left, and top views of a sprocket gear, with the isometric icon in the toolbar and the CATIA V5 interface visible.

Step 30: Switch to the sprocket gear part window

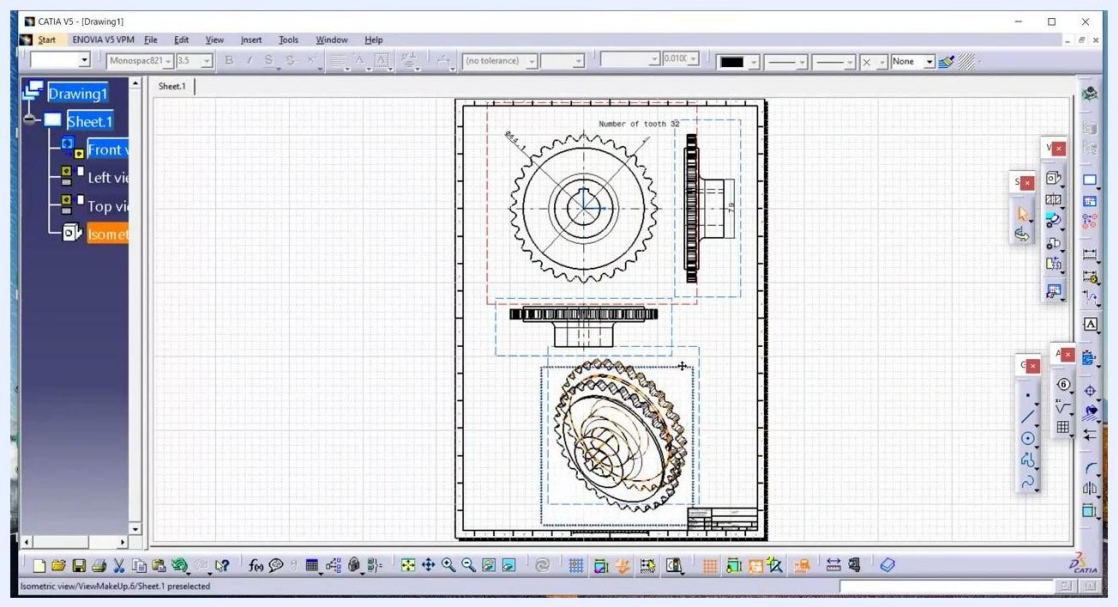
Click **Window** in the top menu and select sprocket gear.CATPart from the dropdown to switch to the part file. This lets you drag the isometric view into your drawing sheet.



CATIA V5 interface with the Window menu open, showing "sprocket gear.CATPart" selected. The drawing sheet with sprocket gear views is visible.

Step 31: Drag the isometric view onto the sheet

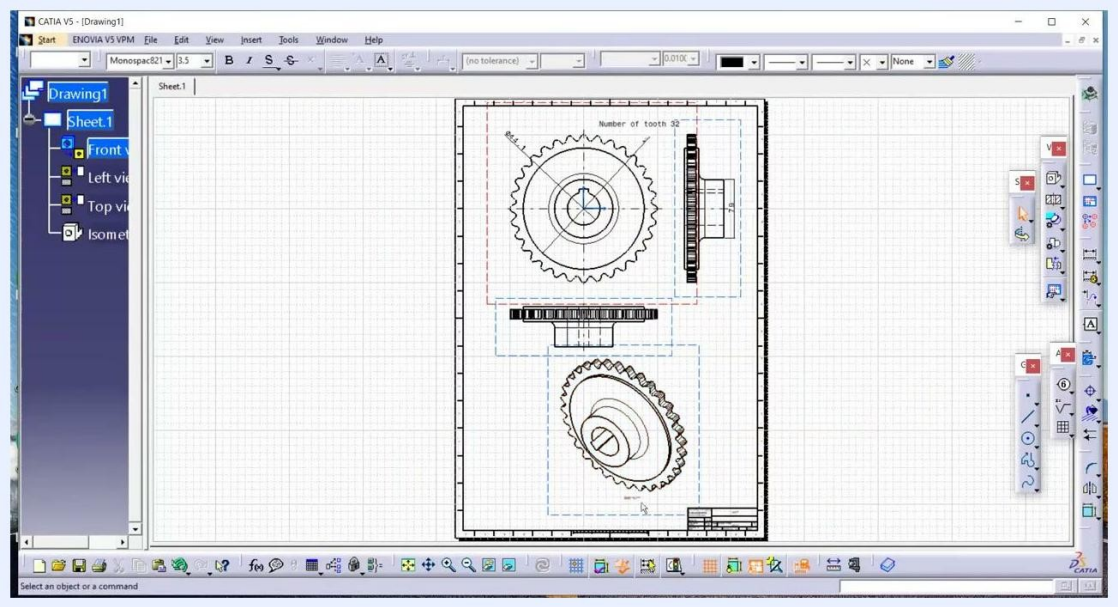
Click and drag the isometric view of the sprocket gear onto the drawing sheet. Position it in the lower section of the sheet so it does not overlap other views. Adjust the placement for clarity and presentation.



Drawing sheet with the isometric view of the sprocket gear being placed in the lower section. The CATIA V5 interface and drawing tree are visible.

Step 32: Adjust and finalize the isometric view

Resize or reposition the isometric view if needed for the best layout. Make sure it is clearly visible and does not obscure other drawing elements. Confirm that all views (front, left, top, isometric) are aligned and labeled.

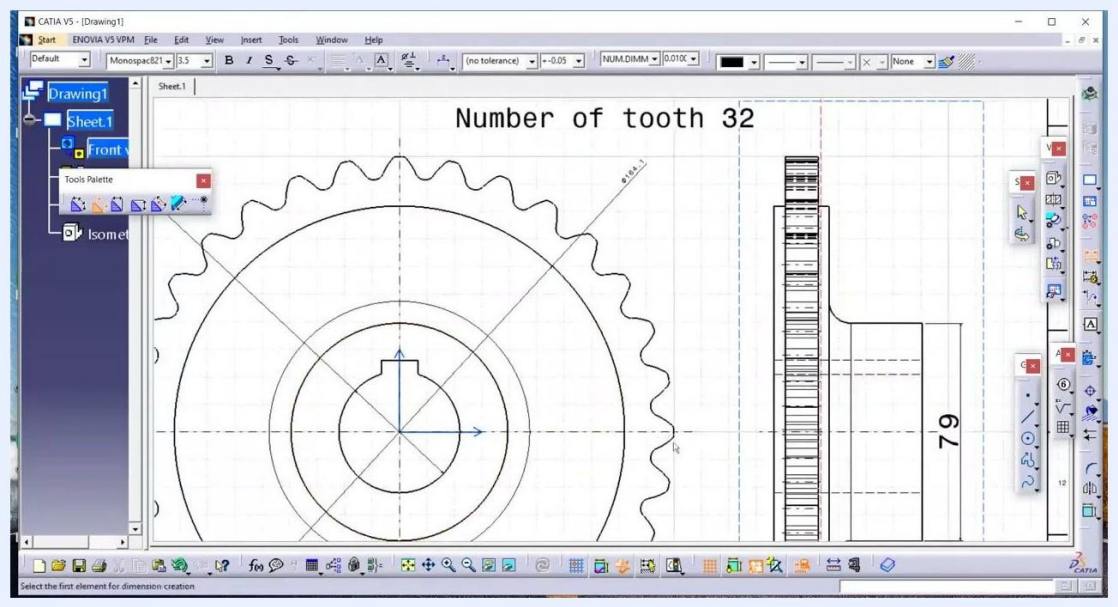


Drawing sheet with the isometric view of the sprocket gear finalized and positioned. All views are clearly visible and well-organized.

Export the drawing as a PDF

Step 33: Review the drawing before export

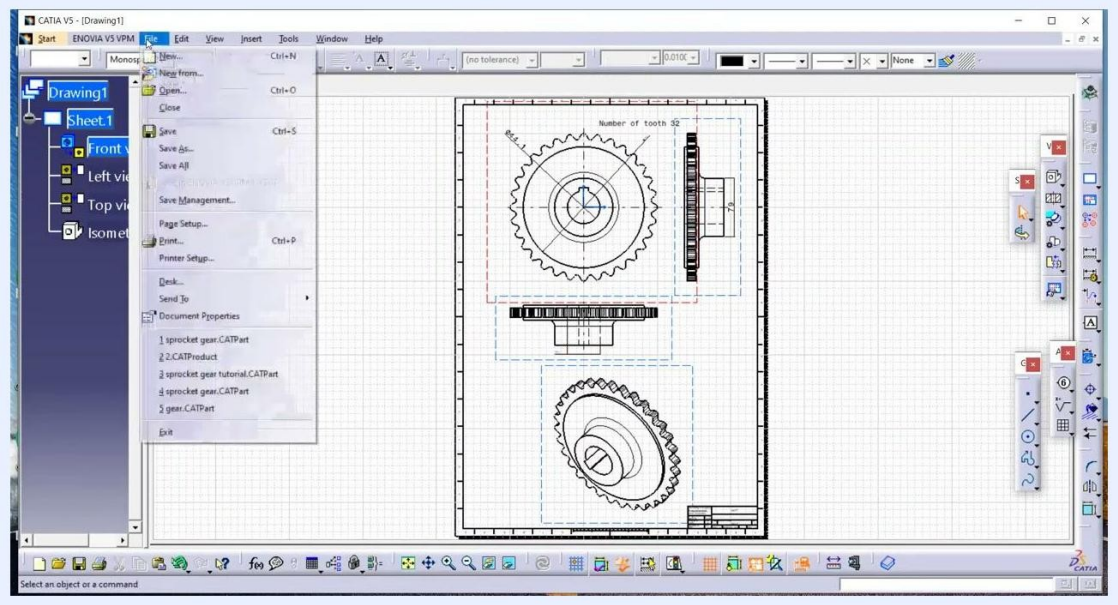
Inspect the drawing sheet in CATIA V5 to make sure all required details are present. Confirm that dimensions (for example, shaft diameter), annotations (for example, "Number of tooth 32"), and all views are clearly visible. The drawing should include the front, side, and isometric views of the sprocket gear.



CATIA V5 drawing sheet showing detailed sprocket gear views, dimension lines, and the annotation "Number of tooth 32". The Tools Palette is open, and the drawing tree is visible on the left.

Step 34: Open Save As

Click **File** in the CATIA V5 menu at the top left. The dropdown shows options such as New, Open, Save, Save As, and Print. Click **Save As...** to open the Save As dialog, where you choose the file format and destination folder.

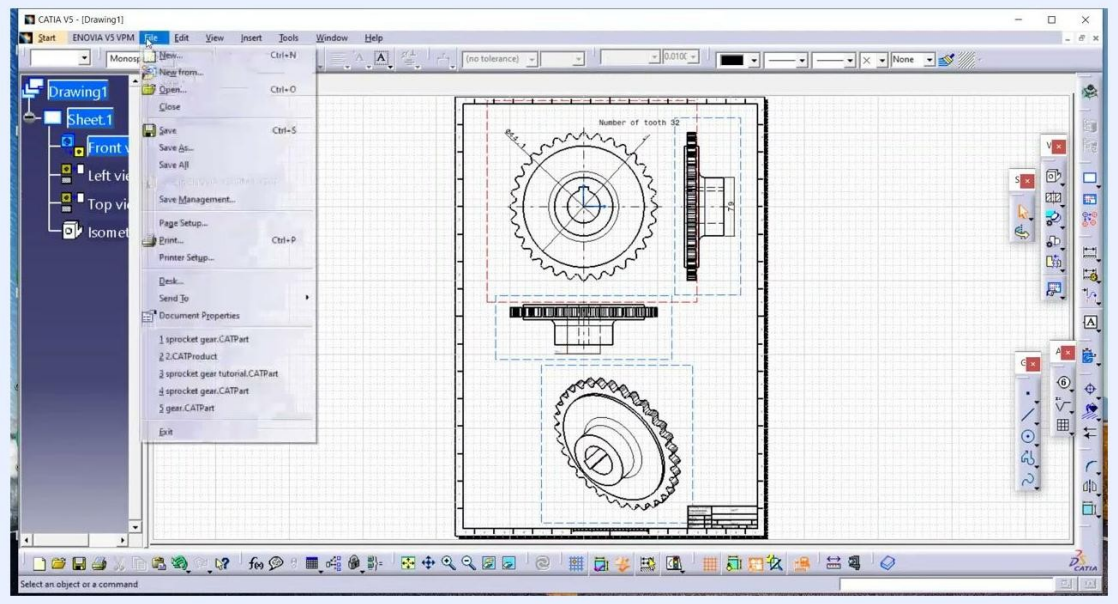


CATIA V5 interface with the File menu open, displaying options including Save, Save As, and Print. The drawing sheet with sprocket gear views is visible.

Step 35: Choose the export location and file type

1. In the Save As dialog, go to the folder you want (for example, "catia tutor").
2. Enter the file name "sprocket gear".

3. In the **Save as type** dropdown, select **pdf**.



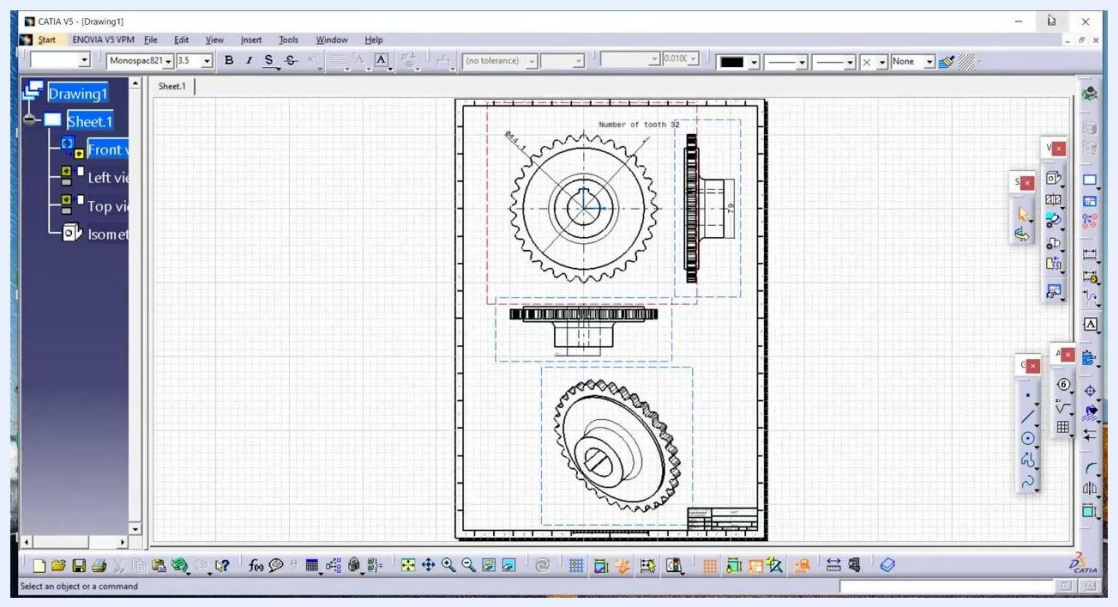
Save As dialog open in CATIA V5, showing the folder "catia tutor", file name "sprocket gear", and file type set to "pdf".

Step 36: Save the PDF file

Click **Save** to export the drawing as a PDF. CATIA creates the file in the selected folder with the name you specified.

Step 37: Verify the exported file

Open the destination folder (for example, "catia tutor") in Windows Explorer. Confirm that sprocket gear.pdf is present alongside your other CATIA files.



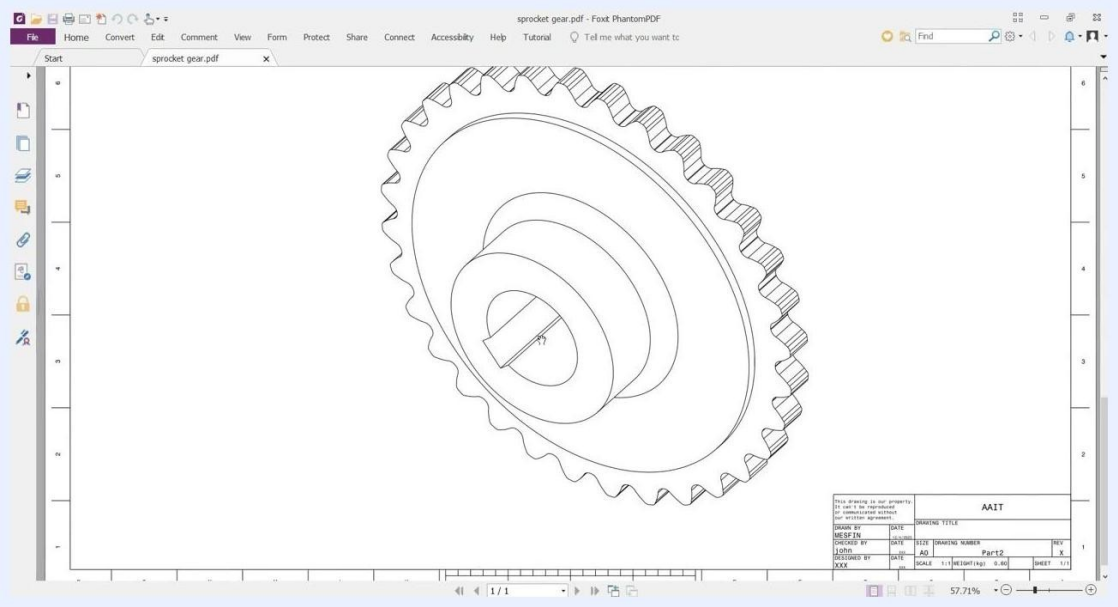
Windows Explorer window open to the "catia tutor" folder, showing the newly created "sprocket gear.pdf" file among other CATIA files. A right-click context menu is visible.

Step 38: Review the exported PDF in detail

Open sprocket gear.pdf in your PDF viewer (for example, Foxit PhantomPDF) and inspect the drawing for clarity and completeness:

- Confirm the isometric view of the sprocket gear is clearly rendered, with all lines and features visible.
- Check the title block in the lower right corner for the drawing title, drawing number, revision, scale (1:1), sheet number (1/1), and the designer and checker names (for example, "DRAWN BY: MESFIN", "CHECKED BY: john", "DESIGNED BY: XXX").
- Check for additional notes such as "This drawing is our property..." and the organization label "AAIT".
- Ensure the drawing is centered and fits within the page margins.
- Verify that dimensions and annotations are present and legible.

Use the PDF viewer's zoom and navigation tools to examine all areas of the drawing. Confirm that the exported PDF matches the original CATIA V5 drawing in layout and detail.



PDF viewer (Foxit PhantomPDF) displaying the exported sprocket gear drawing. The isometric gear view is centered, with a detailed title block in the lower right corner showing fields like DRAWN BY, CHECKED BY, DESIGNED BY, SCALE, and organization label "AAIT".

Summary

You have now converted a 3D sprocket gear model to a 2D drawing in CATIA V5. You:

- Drafted a technical drawing in CATIA V5.
- Exported the drawing as a PDF file.
- Verified the exported PDF for accuracy, completeness, and professional presentation.

What's next

You can now share, print, or archive your technical drawing, knowing it meets documentation standards and preserves the critical design information.