How to create a 2D drawing in CATIA V5: views, dimensions, and hatching

This guide shows you how to create a 2D drawing in CATIA V5. You start with a blank drawing sheet and insert a front view of your part. Then you add section and auxiliary views, dimension every view, apply hatching, and adjust the isometric view properties.

Prerequisites

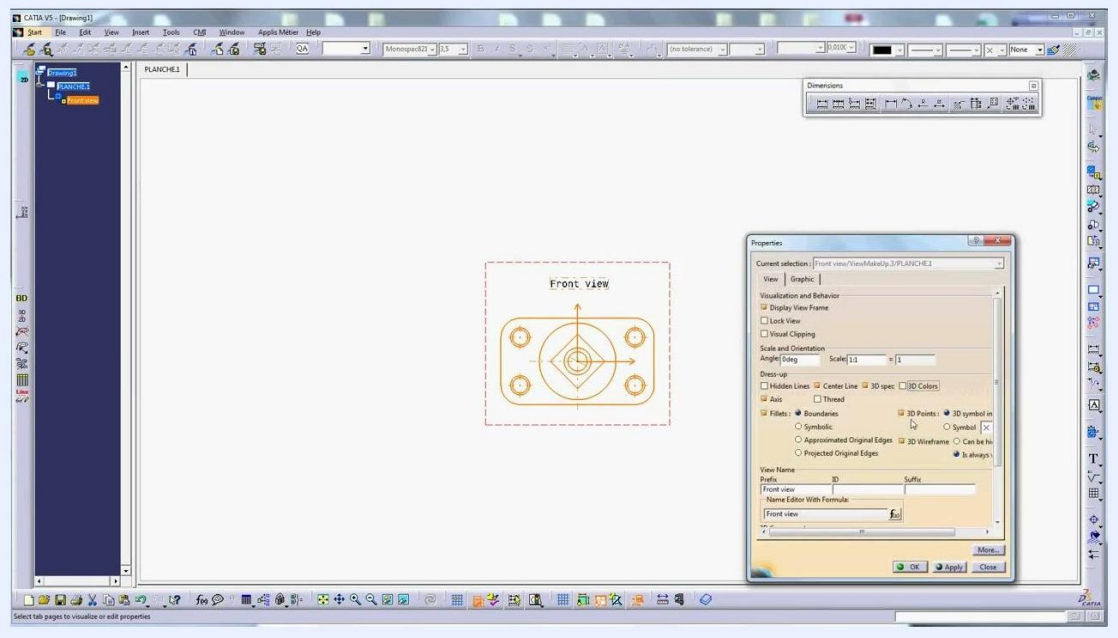
- CATIA V5 with access to the Drawing workbench.
- A part to insert into the drawing sheet as a front view.

Set up the drawing sheet and front view

Step 1: Open a new drawing sheet to start your 2D drawing

Launch CATIA V5 and open a new drawing document. The workspace shows a blank drawing sheet labeled "PLANCHE1" with a thick border.

The left panel shows the drawing structure: Drawing1 > PLANCHE1. Toolbars appear at the top, left, and right of the CATIA V5 Drawing workbench.

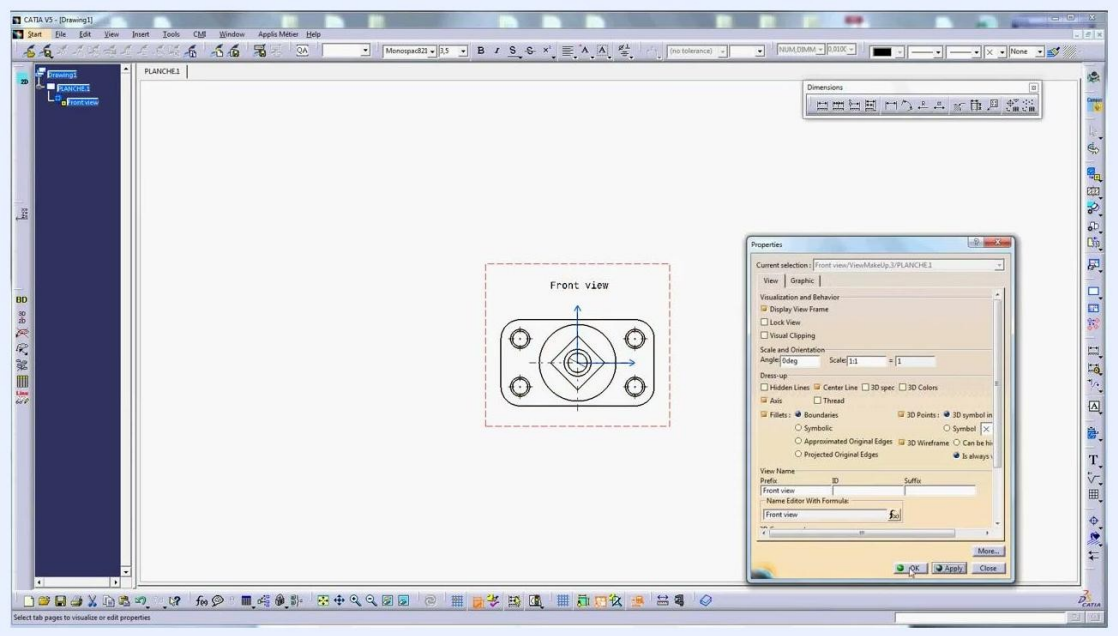


Blank drawing sheet in CATIA V5 with a labeled border and the drawing structure panel visible

Step 2: Insert the front view of the part

Insert a front view of the part into the drawing sheet. The view appears centered on the sheet, outlined in orange, with its axes and the label "Front view" below the drawing.

The left panel now shows Drawing1 > PLANCHE1 > Front view. A progress bar appears at the bottom right while the view is inserted.

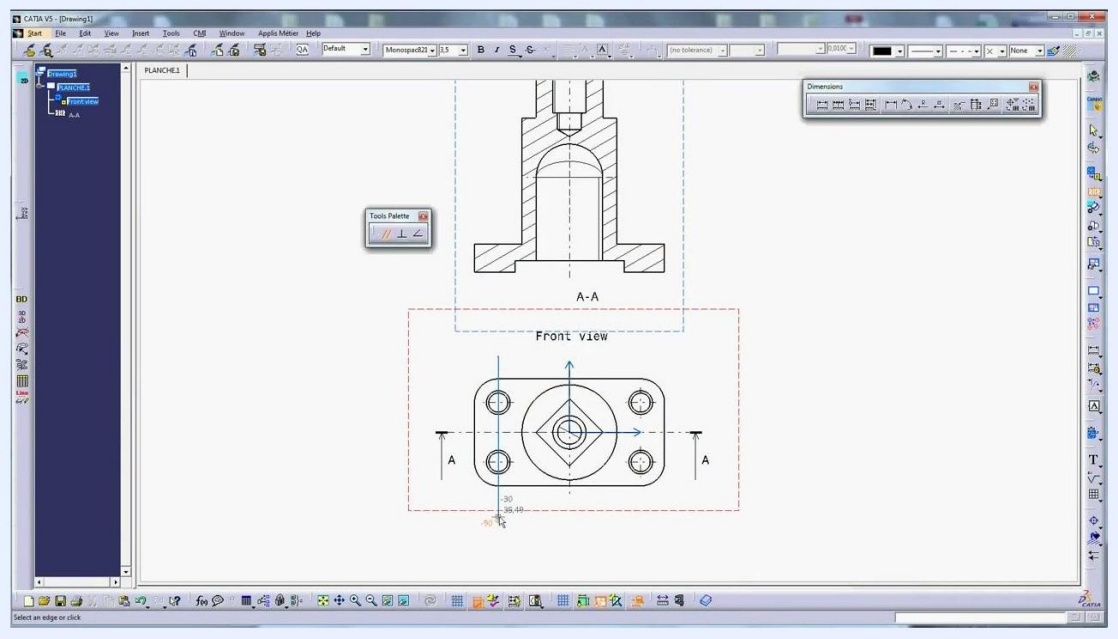


Front view of a mechanical part inserted on the sheet, with axes, a label, and the updated drawing tree

Step 3: Open the view properties and apply your changes

Right-click or double-click the "Front view" to open the **Properties** dialog. The window appears on the right with options for visualization, scale, orientation, and dress-up features.

In this example, the scale is 1:1 and the angle is 0 degrees. Options for hidden lines, axes, threads, fillets, and 3D points are available. The view name is editable and set to "Front view".



Properties dialog open for the front view, showing scale, orientation, and display options

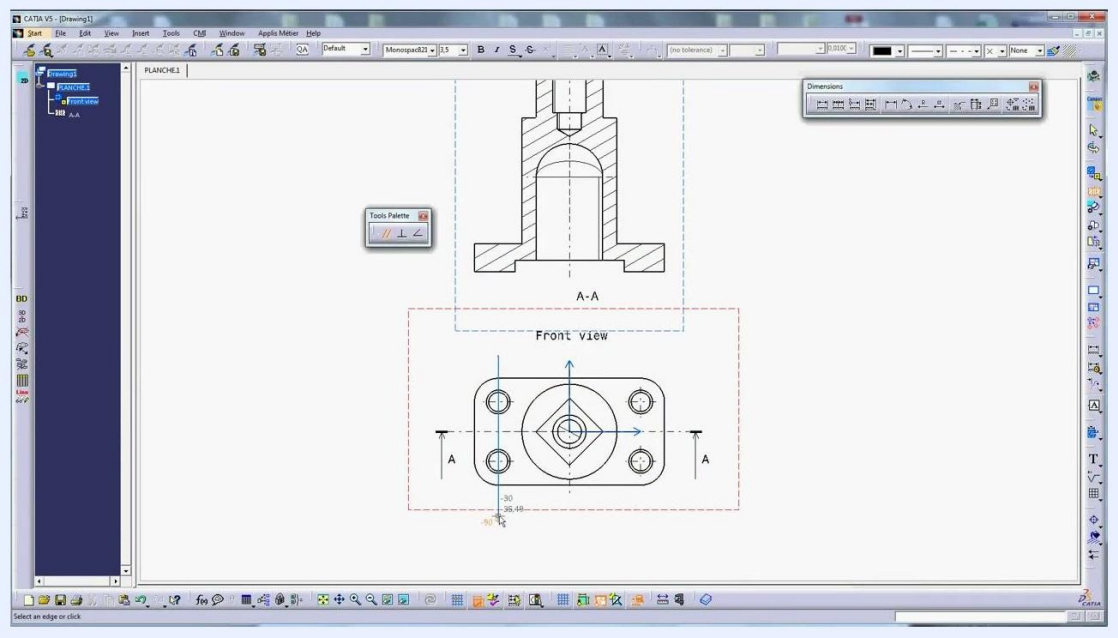
Click **OK** or **Apply** to confirm your changes. The front view updates and is now drawn in black lines. The **Properties** dialog stays open if you need further adjustments.

Add section and auxiliary views

Step 4: Add section view A-A and dimension lines

Insert a section view labeled "A-A" above the front view. It shows an internal cross-section of the part. Add dimension lines and labels such as "A-A" and "B-B" to the front view.

The **Tools Palette** window appears, which shows that the dimensioning or sectioning tools are active. The drawing now contains section view "A-A" at the top and the front view in the center. Dimension lines and section indicators (arrows and labels) are also visible.

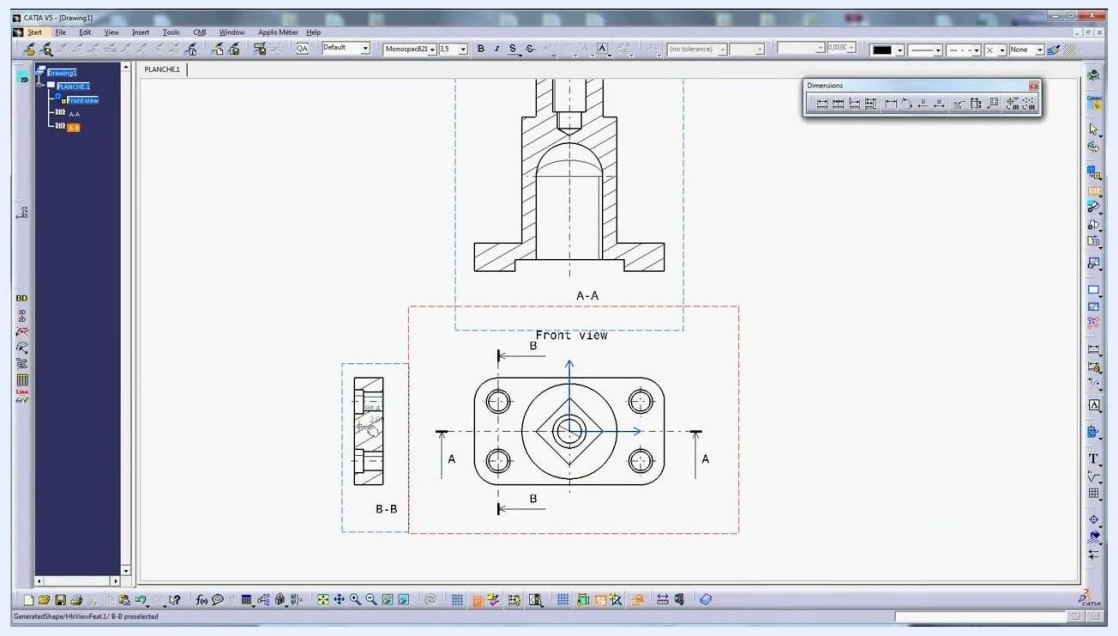


Section view A-A added above the front view, with dimension lines and labels visible and the Tools Palette open

Step 5: Insert section view B-B

Add a second section view labeled "B-B" to the left of the front view. The drawing now shows section view A-A at the top, B-B on the left, and the front view in the center. Dashed boxes with labels enclose all the views.

The left panel updates to show the new section view in the drawing tree.

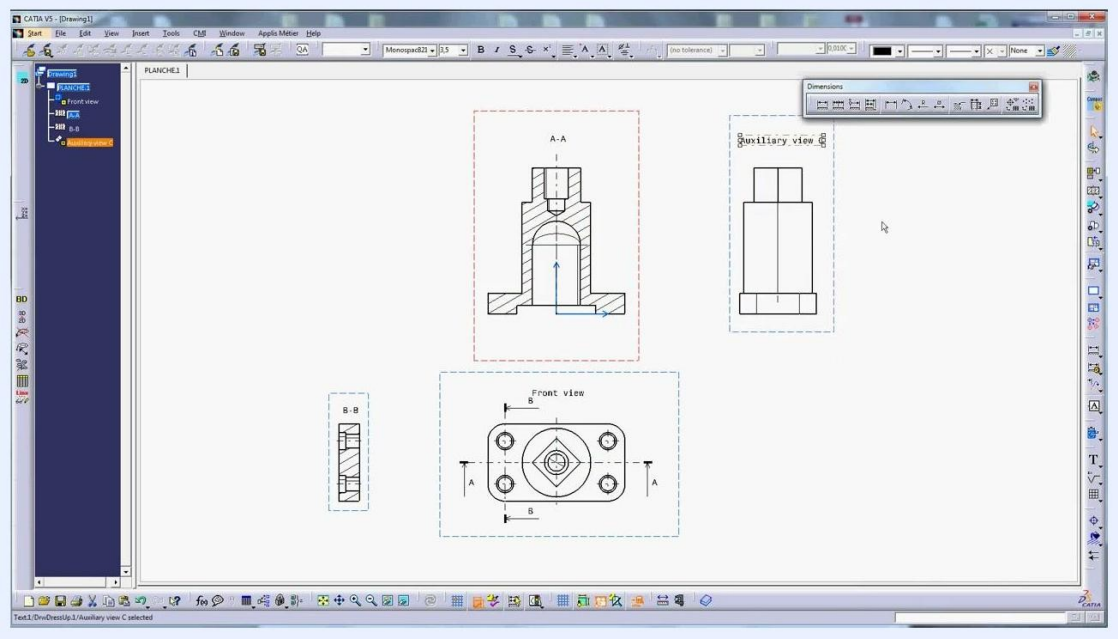


Section view B-B added on the left, with all views labeled and boxed and the drawing tree updated

Step 6: Add an auxiliary view

Insert an auxiliary view labeled "Auxiliary view C" to the right of the front view. It shows a different projection of the part, with a basic outline and dimensions.

The left panel now lists all views: Front view, A-A, B-B, and Auxiliary view C. The sheet contains A-A (top), B-B (left), the front view (center), and Auxiliary view C (right).

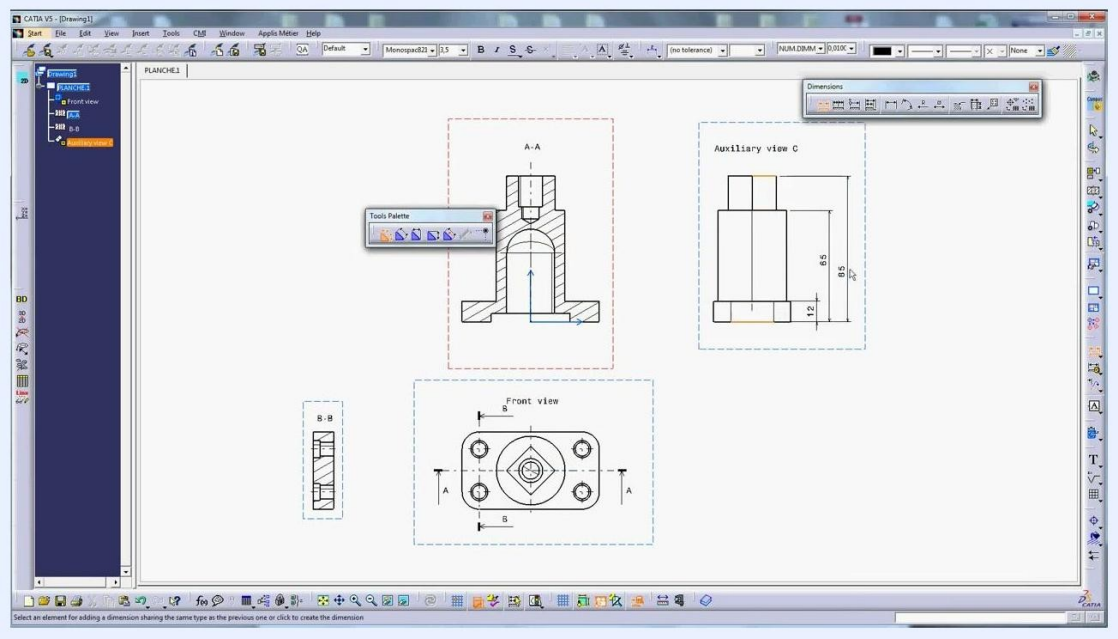


Auxiliary view C added on the right, with all views labeled and boxed and the drawing tree updated

Step 7: Dimension the auxiliary view

Use the **Tools Palette** to add dimensions to the auxiliary view. Dimension lines and values, for example 12, 65, and 85, appear on the auxiliary view.

The **Tools Palette** window is visible near the section view, which shows that dimensioning is active. The drawing now includes dimensioning for all views.



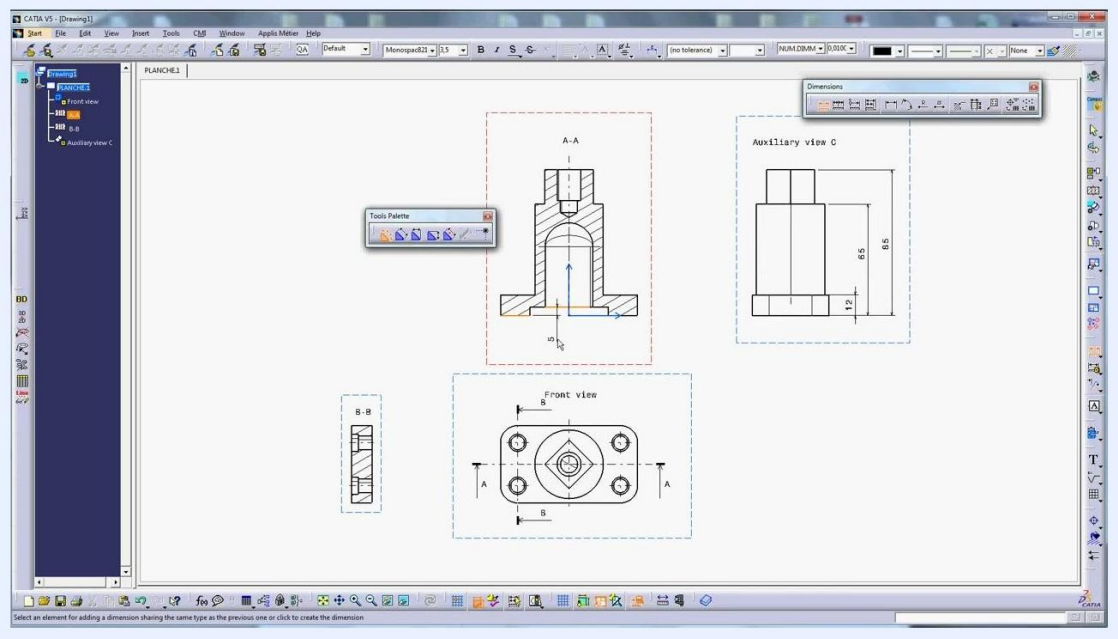
Auxiliary view C dimensioned, with the Tools Palette open and all views labeled and boxed

Dimension section view A-A

Step 8: Start dimensioning section view A-A

Make sure the **Tools Palette** and **Dimensions** toolbars are open and visible. Select section view "A-A", which sits at the top center of the sheet and is outlined in red.

Use the dimensioning tool to add a horizontal dimension at the base of the view. The value "5" appears at the base and indicates the thickness of the lower section.



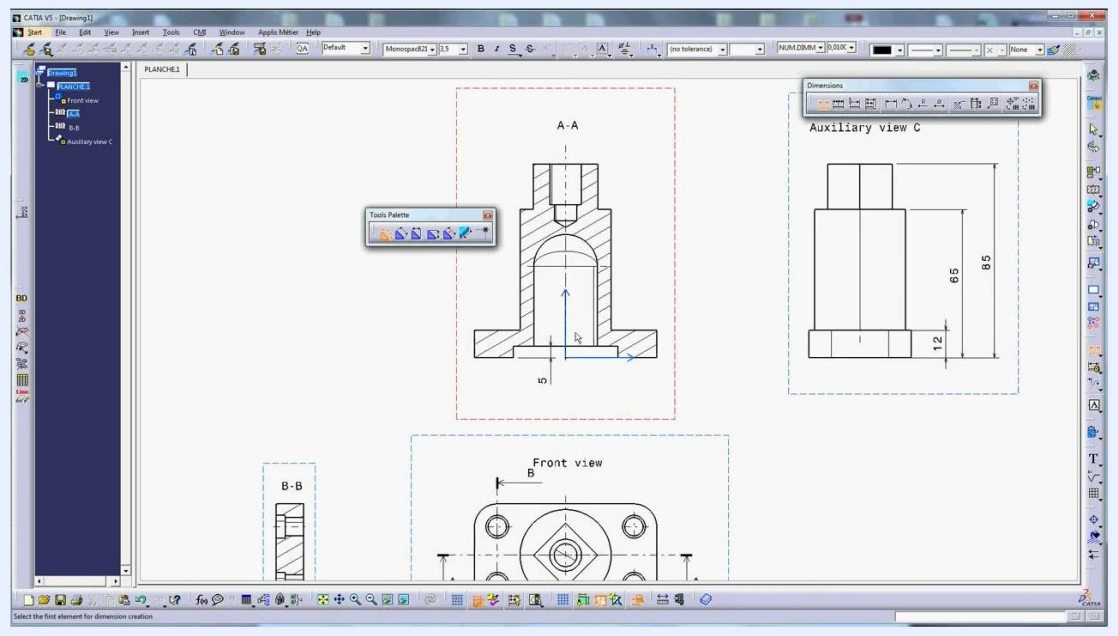
Section view A-A being dimensioned with "5" at the base, and the Tools Palette and Dimensions toolbars visible

Step 9: Zoom in and add vertical dimensions

Zoom in on section view "A-A" for precise dimensioning. Add two vertical dimensions on the right side of the view:

- "18" for the upper segment
- "25" for the total height of the upper section

The **Tools Palette** and **Dimensions** toolbar stay visible for quick access.

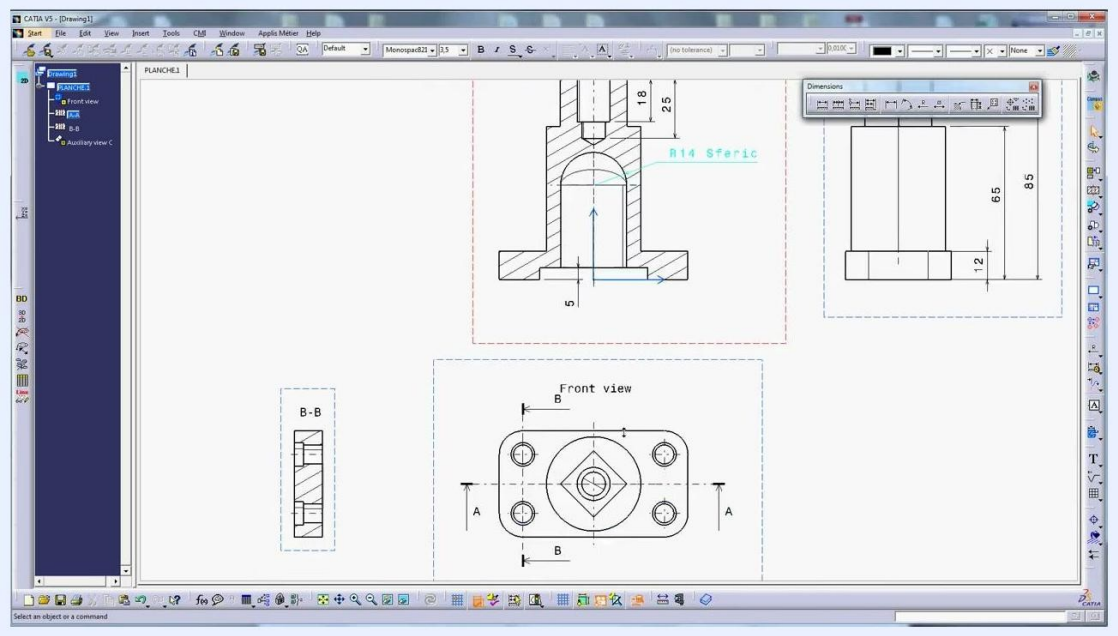


Zoomed-in section view A-A with vertical dimensions "18" and "25" added

Step 10: Annotate the spherical radius

Add a radius annotation to the curved section of the cross-section. The annotation "R14 Sferic" indicates a spherical radius of 14. It appears with an orange leader line pointing to the arc.

The annotation is in edit mode and highlighted in orange, so you can still place or adjust it.



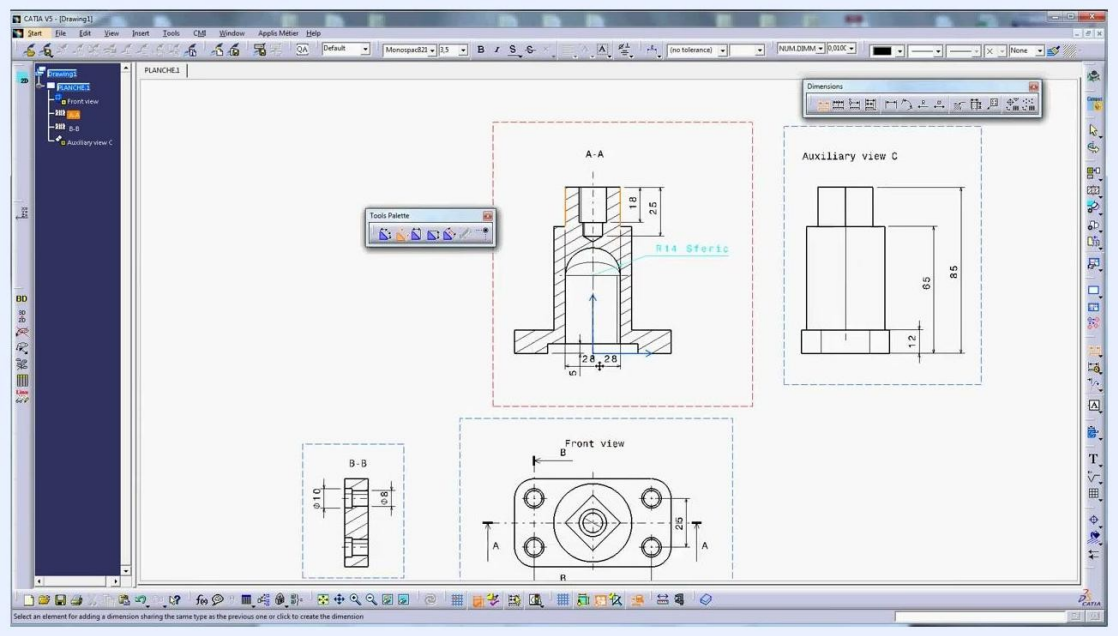
Section view A-A with the "R14 Sferic" annotation in orange, a leader line pointing to the arc, and the annotation in edit mode

Confirm and place the "R14 Sferic" annotation. It turns blue, which indicates it is finalized. The earlier dimensions ("5", "18", and "25") remain visible, and the sheet shows all views with their updated annotations.

Step 11: Add horizontal dimensions to section view A-A

Use the dimensioning tool to add two horizontal dimensions at the base of section view A-A. Both are labeled "28", which indicates symmetrical features.

The **Tools Palette** is active and the section view is outlined in red. All previously placed dimensions and annotations remain visible.

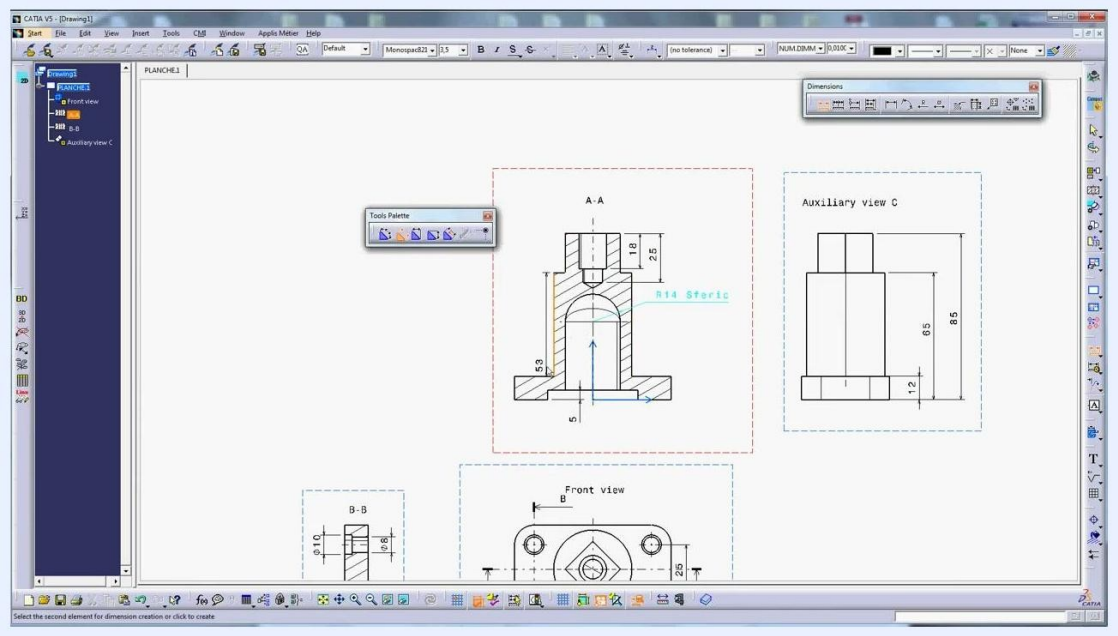


Section view A-A with two new horizontal dimensions "28" added at the base and the Tools Palette active

Step 12: Add a diameter dimension to section view A-A

Place a diameter dimension at the base of section view A-A. The dimension "⌀40" indicates the diameter of the lower section.

The **Tools Palette** remains open for quick tool selection. All other dimensions and the "R14 Sferic" annotation are still visible.



Section view A-A with the diameter dimension "⌀40" added at the base and the Tools Palette open

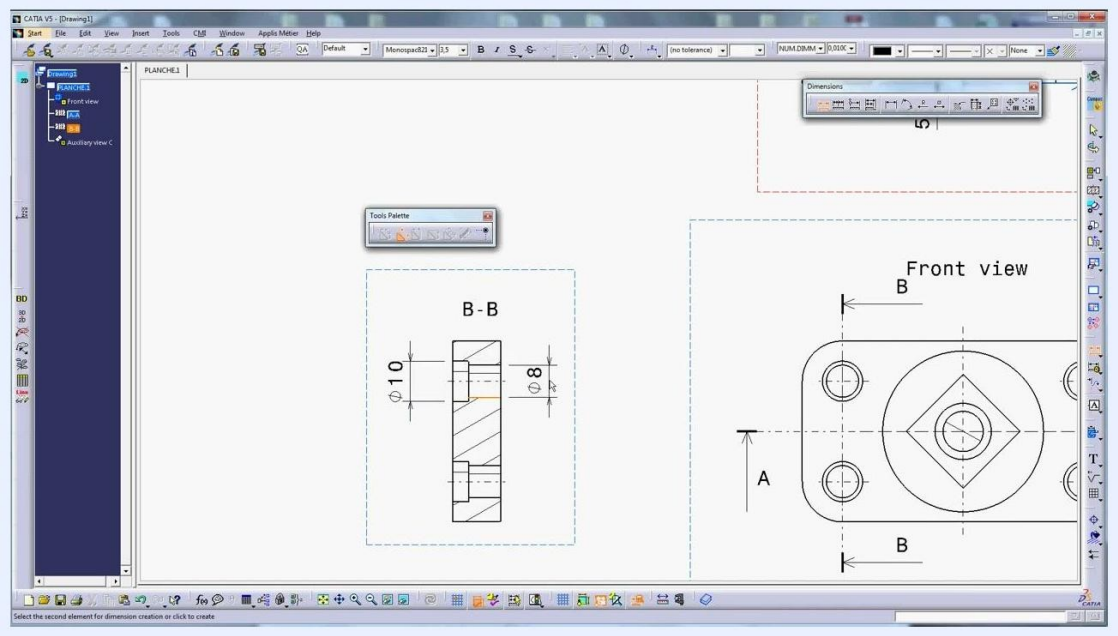
Dimension section view B-B

Step 13: Add diameter dimensions to section view B-B

Move to section view "B-B", which is at the bottom left of the sheet and outlined in blue. Use the dimensioning tool to add two horizontal diameter dimensions:

- "10" on the left side
- "8" on the right side

The **Tools Palette** is active and the cursor is positioned to place or adjust the dimension.



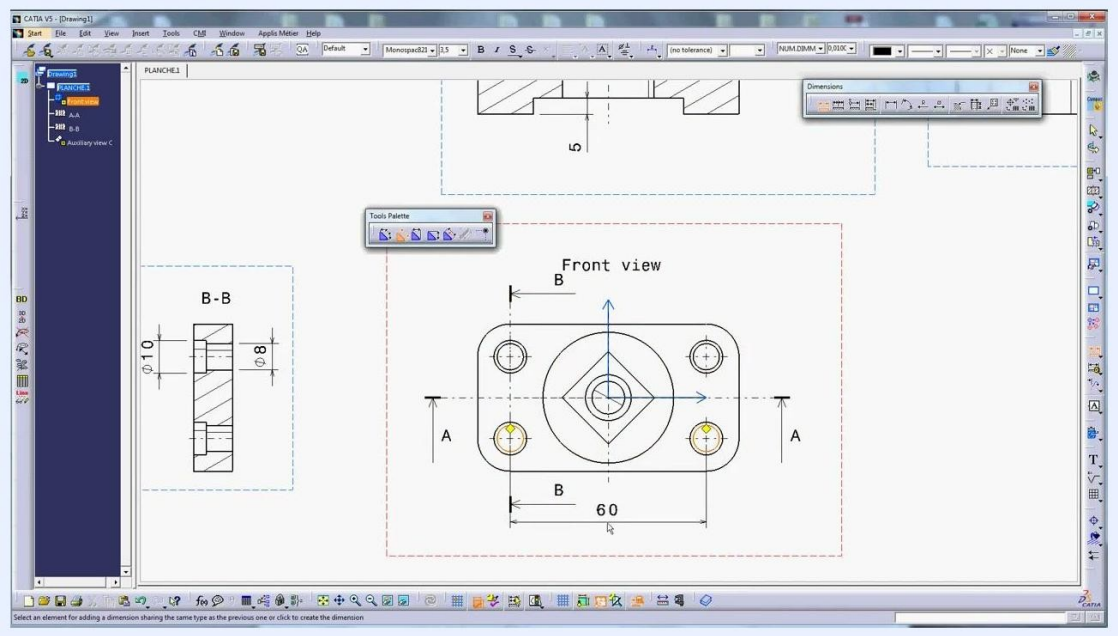
Section view B-B with diameter dimensions "10" and "8" being added and the Tools Palette active

Dimension the front view

Step 14: Add the key dimension to the front view

Use the dimensioning tool to add a key horizontal dimension to the front view. The dimension "60" is placed at the bottom and indicates the overall width between the centers of the outer holes.

The front view is outlined in red, which shows it is the current focus. The **Tools Palette** is active, and the section indicators "A" and "B" remain visible.

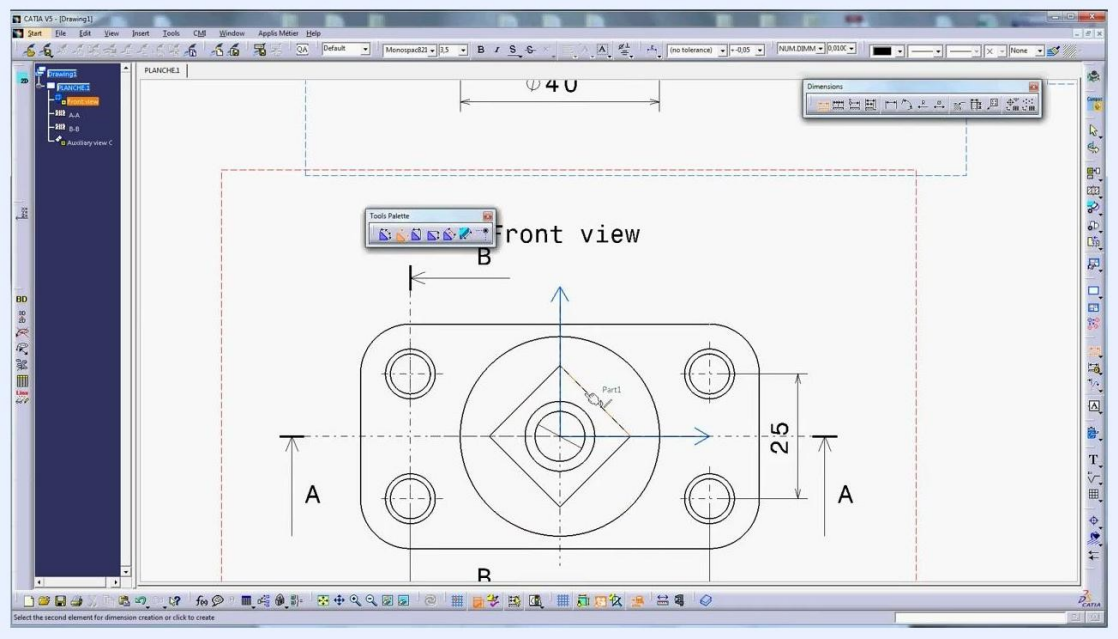


Front view with the dimension "60" added at the bottom, the Tools Palette active, and the section indicators visible

Step 15: Zoom in on the front view

Zoom in on the front view for precise placement of additional dimensions. The **Tools Palette** is positioned near the center of the front view.

The cursor is active and ready to select elements for dimensioning. The "⌀40" dimension from section view A-A is visible at the top of the workspace.

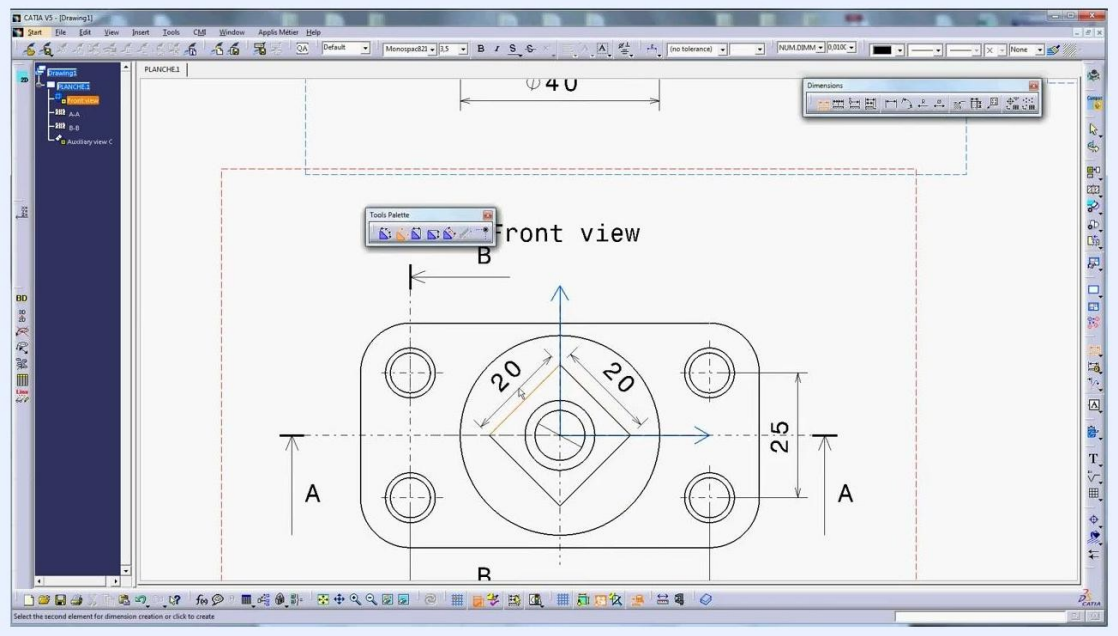


Zoomed-in front view with the Tools Palette near the center, the cursor ready for dimensioning, and "⌀40" visible above

Step 16: Add diameter dimensions to the front view

Use the dimensioning tool to add two diameter dimensions to the front view. Both are labeled "20" and indicate the diameter of the inner and outer circles.

The **Tools Palette** is active and the dimensions are placed symmetrically.

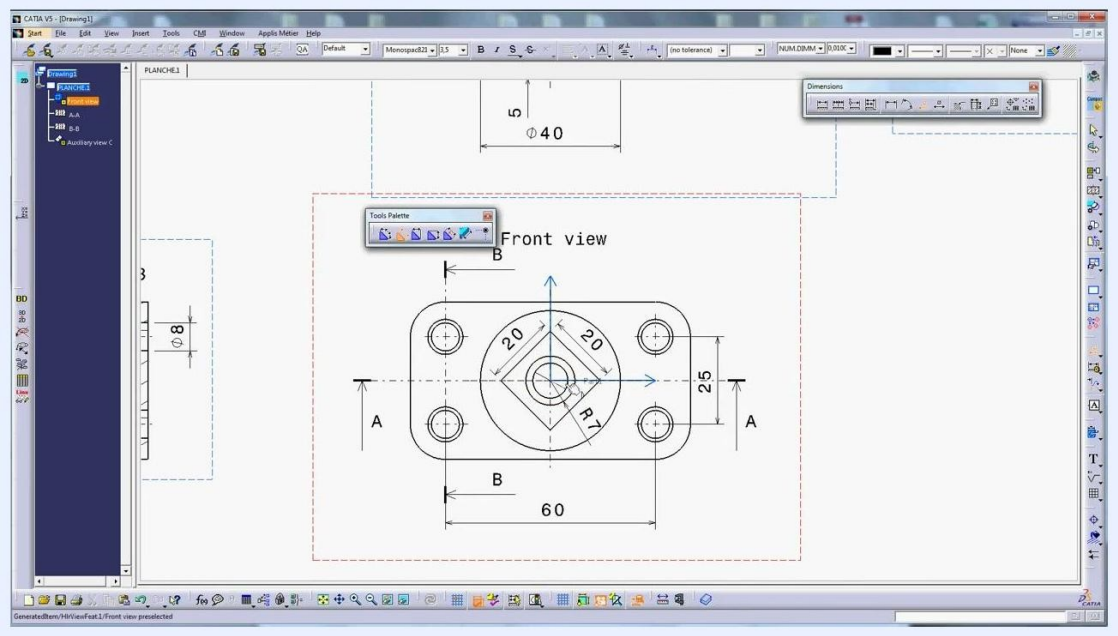


Front view with two diameter dimensions "20" added, the Tools Palette active, and the dimensions placed symmetrically

Step 17: Add a radius dimension to the front view

Place a radius dimension labeled "R7" on the inner square feature of the front view. The dimension sits inside the square and clearly indicates the radius.

The previously placed dimensions ("60", "25", "20", and "20") remain visible.

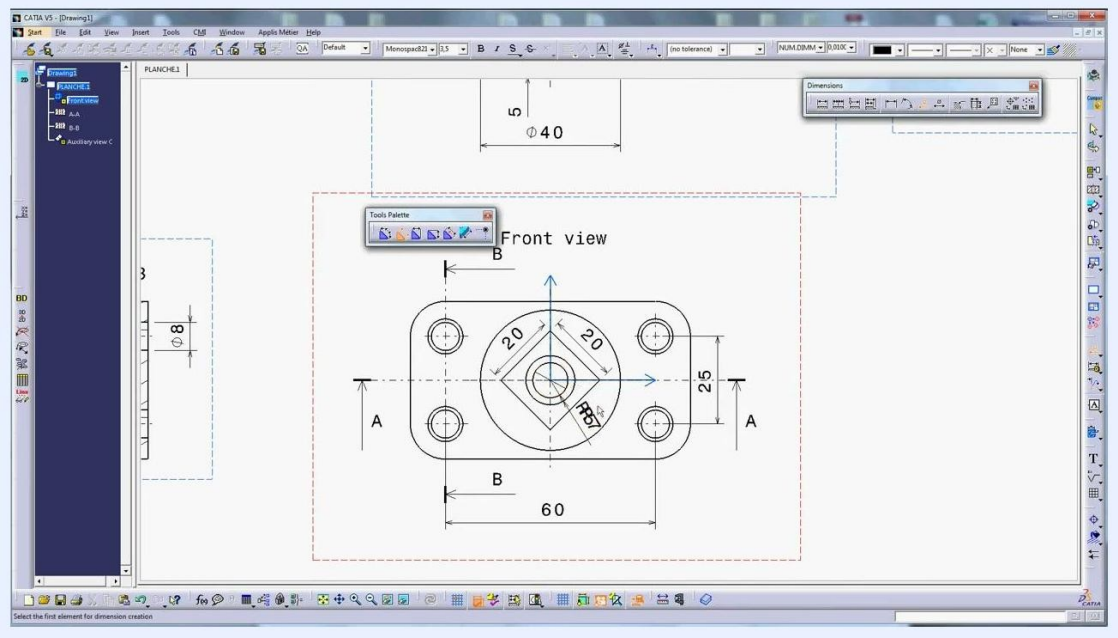


Front view with the radius dimension "R7" added to the inner square and all dimensions visible

Step 18: Adjust the dimension placement

Adjust the placement of the newly added dimensions for clarity and readability. The **Tools Palette** remains open for further adjustments.

The drawing now displays all key dimensions and annotations.

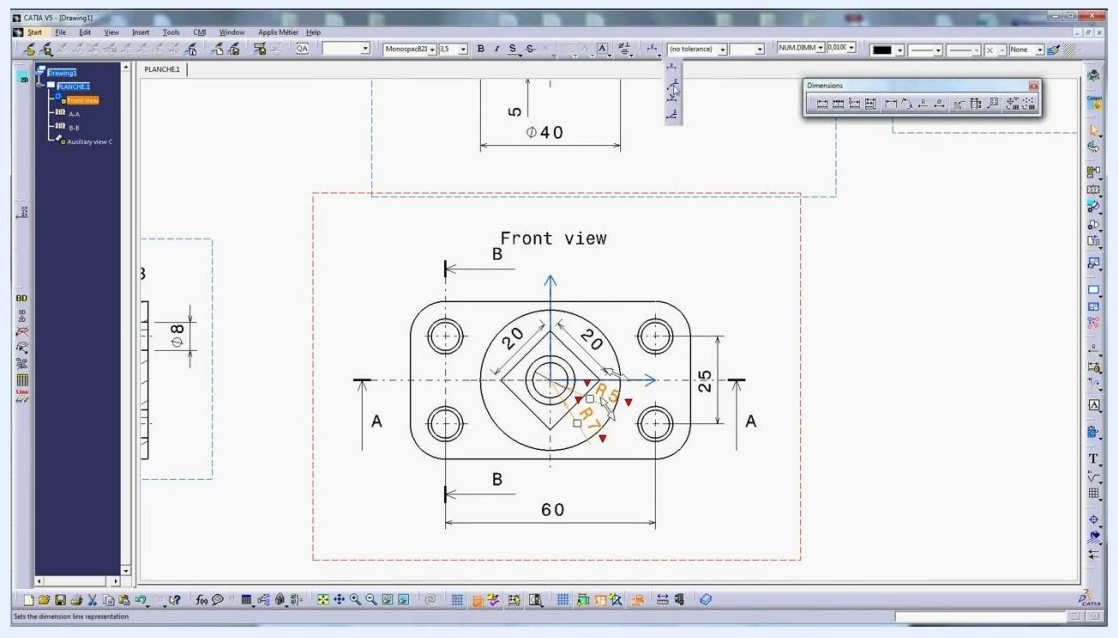


Front view with the dimensions "60", "25", "20", "20", and "R7" clearly placed and the Tools Palette open

Step 19: Open the dimension properties

Select the newly added dimensions to access their properties. Dimension handles and property options appear, so you can customize items such as line representation and tolerance.

All toolbars and palettes stay accessible for a final review.



Front view with dimension handles and property options visible, ready for final review

When you finish, the front view shows all of its dimensions clearly placed, and the drawing is ready for review.

Add dimensions and hatching to section views

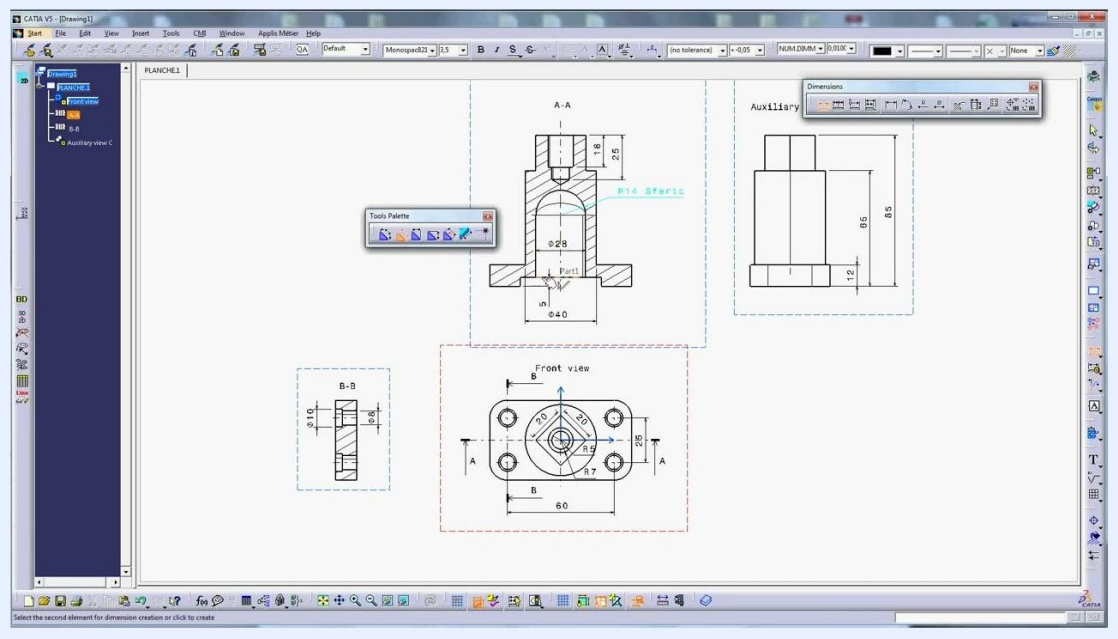
Step 20: Activate the Tools Palette

Open or activate the floating **Tools Palette** window. It contains icons for drawing and editing features such as lines, circles, and hatching. Position the palette where you can reach it easily while you work.

Step 21: Add or edit section view dimensions

Select a section view, for example A-A, by clicking inside its dashed blue boundary. Use the **Dimensions** toolbar to add or modify dimensions. For example, add a diameter dimension (Ø28) or a length dimension (35).

Watch the cursor and the selection highlights as you place or edit the dimension. Confirm that the dimension appears in the correct location on the section view.

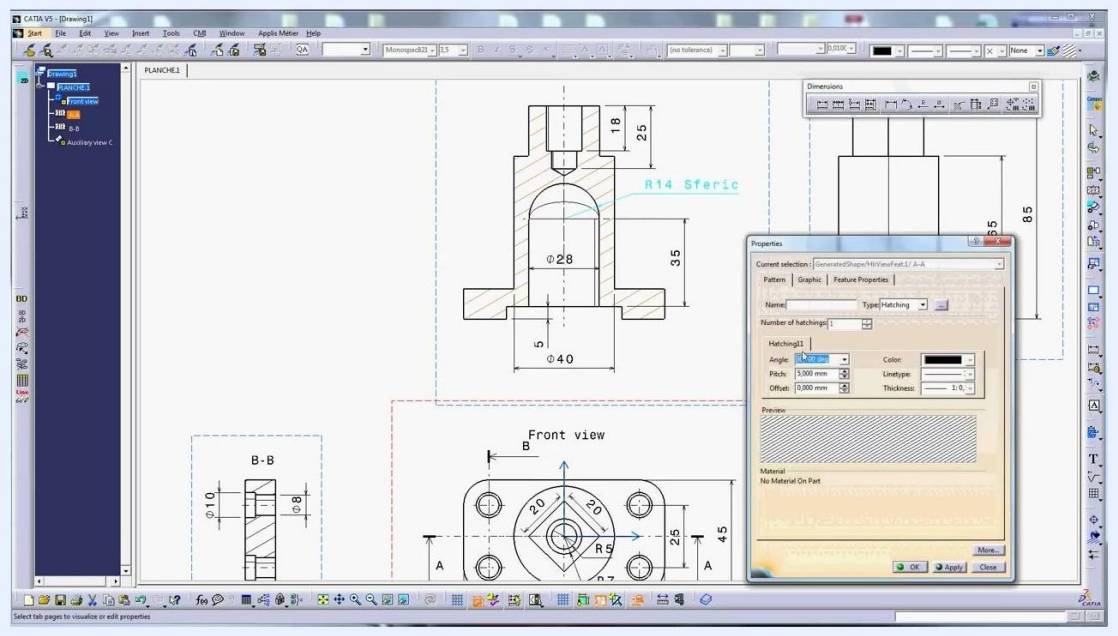


Section view A-A selected with the Tools Palette open and a new Ø28 dimension being added

Step 22: Continue dimensioning the section views

Add more dimensions as needed, such as the vertical height (35) on section A-A. Make sure every critical feature is dimensioned for manufacturing or inspection.

Use the **Dimensions** toolbar to select the appropriate dimension type, such as linear, diameter, or radius.



Section view A-A with a new vertical dimension (35) being added and the Tools Palette and Dimensions toolbar visible

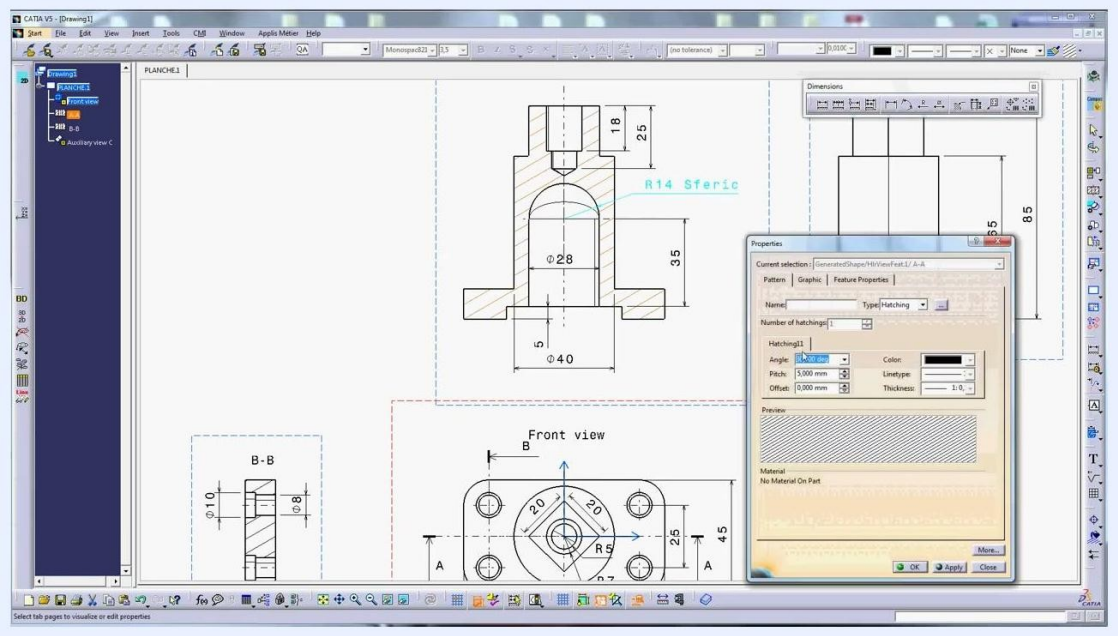
Step 23: Apply hatching to a section view

Double-click or right-click a sectioned area, for example A-A or B-B, to open the **Properties** dialog. In the **Properties** window, go to the **Feature Properties** tab and select **Hatching**.

Set the hatching parameters:

- Angle: 45.000 deg
- Pitch: 5.000 mm (or 4.000 mm, as shown in later steps)
- Offset: 0.000 mm
- Color: Black
- Linetype: Solid
- Thickness: 1.0

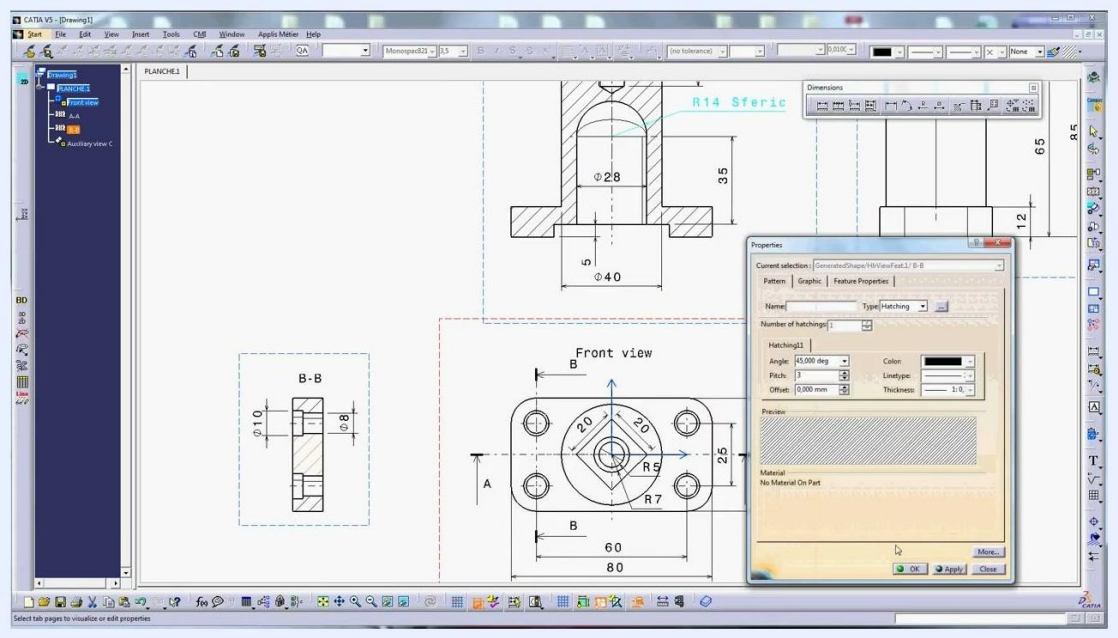
Preview the hatching pattern in the dialog. Then click **Apply** or **OK** to apply the hatching to the selected section.



Properties dialog open for section A-A with the hatching parameters being set: angle 45°, pitch 5 mm, color black, thickness 1.0

Step 24: Repeat the hatching for other sections

Select another section view, for example B-B. Open the **Properties** dialog and set the hatching parameters as you did for the first section. Confirm that the hatching is applied and visible in the section view.



Properties dialog open for section B-B with the hatching parameters being set: angle 45°, pitch 5 mm, color black, thickness 1.0

Set the isometric view properties

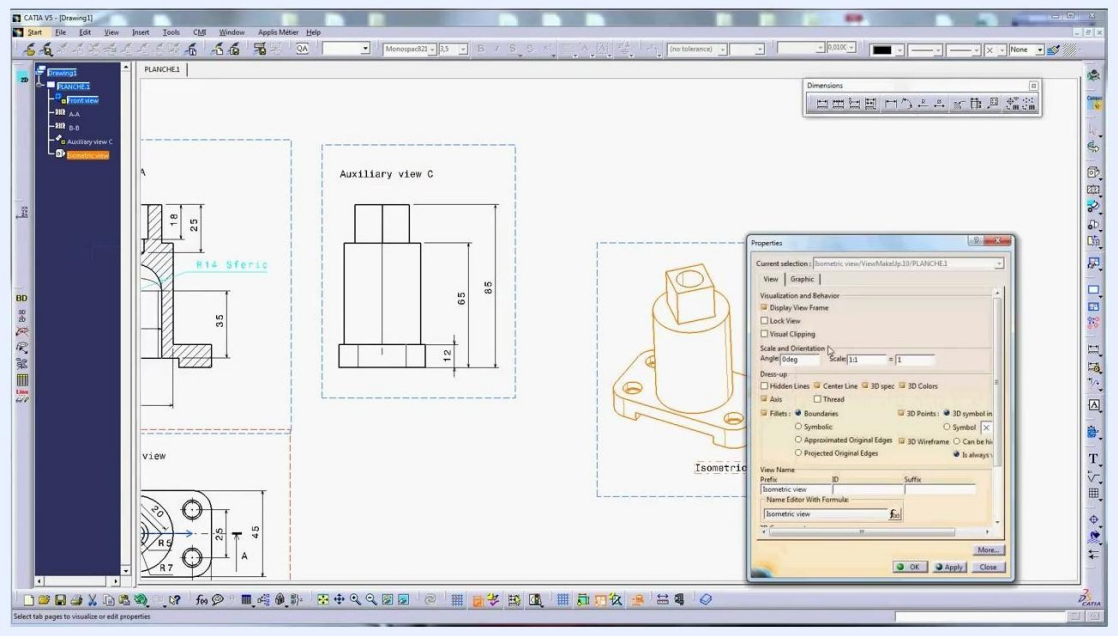
Step 25: Open the properties for the isometric view

Right-click or double-click the isometric view boundary to open the **Properties** dialog. The window is focused on the isometric view, which stays highlighted on the drawing.

The dialog offers these options:

- Visualization and Behavior: Display View Frame, Lock View, Visual Clipping
- Scale and Orientation: Angle 0 deg, Scale 1
- Dress-up options: Hidden Lines, Center Line, 3D spec, 3D Colors, Fillets, Boundaries, 3D Points, Symbol, and more

- View Name: currently "Isometric view"



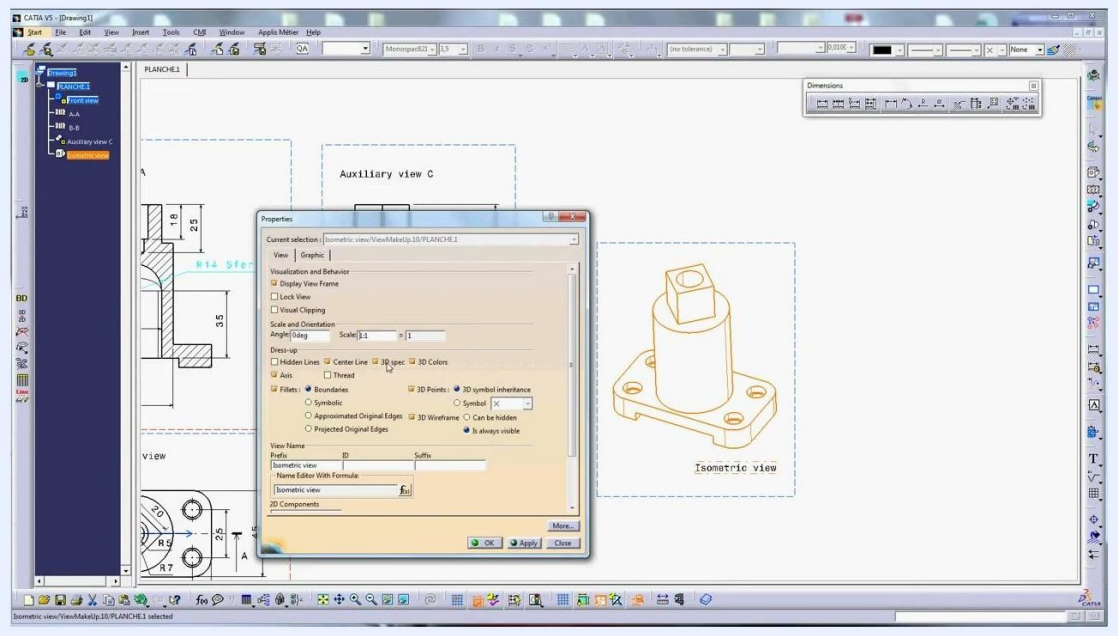
Properties dialog open for the isometric view, showing the visualization, scale, and dress-up options

Step 26: Adjust the isometric view settings

In the **Properties** dialog, review and change these settings as needed:

- **Scale and Orientation:** Set **Angle** to 0 deg and **Scale** to 1 for a true-size isometric view.
- **Dress-up options:** Check or clear options such as Hidden Lines, Center Line, 3D spec (checked), and 3D Colors (checked). You can also set Fillets (Boundaries, Symbolic, Approximated Original Edges, Projected Original Edges), 3D Points, 3D symbol inheritance, and Symbol.
- **3D Wireframe:** Choose Can be hidden or Is always visible.
- **View name:** Confirm or edit the name in the **Name Editor With Formula** field. It currently reads "Isometric view".

The dialog stays open for further adjustments.

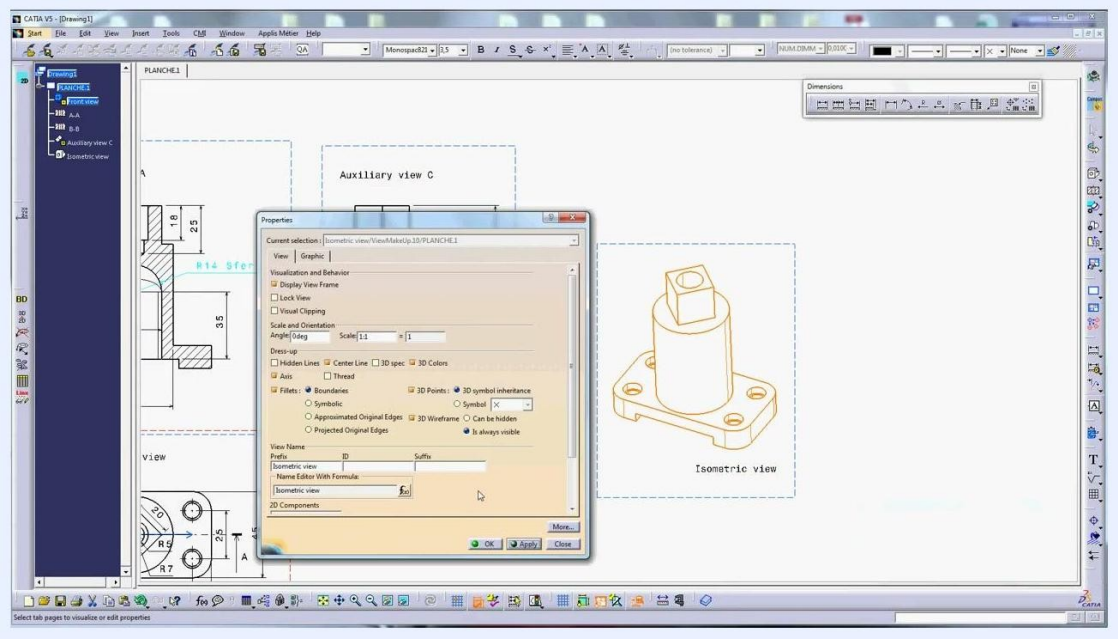


Properties dialog for the isometric view with the 3D spec and 3D Colors options checked

Step 27: Apply and confirm your changes

Click **Apply** to preview the changes in the drawing. Click **OK** to confirm and close the **Properties** dialog.

The isometric view updates according to the selected properties, such as 3D spec and 3D Colors. The drawing now shows the customized isometric view alongside the other views.



Properties dialog for the isometric view, ready to apply or confirm changes

Summary

You now know how to create a 2D drawing in CATIA V5. You opened a drawing sheet and inserted a front view. You added section views A-A and B-B and an auxiliary view C. You dimensioned each view with linear, diameter, and radius dimensions, including the "R14 Sferic" annotation. You also applied hatching to the section views and customized the isometric view properties.

What's next

- Review the drawing for completeness and accuracy.
- Add any necessary notes, tolerances, or finishing details.
- Save and export the finished drawing as required.