How to make a BOM in CATIA V5 with the Assembly Design workbench

If you need to know how to make a BOM in CATIA V5, you can generate a Bill of Materials directly from your assembly. This guide walks you through opening the assembly, creating the BOM from the **Analyze** menu, customizing its format, saving it to a file, and checking the exported result.

Prerequisites

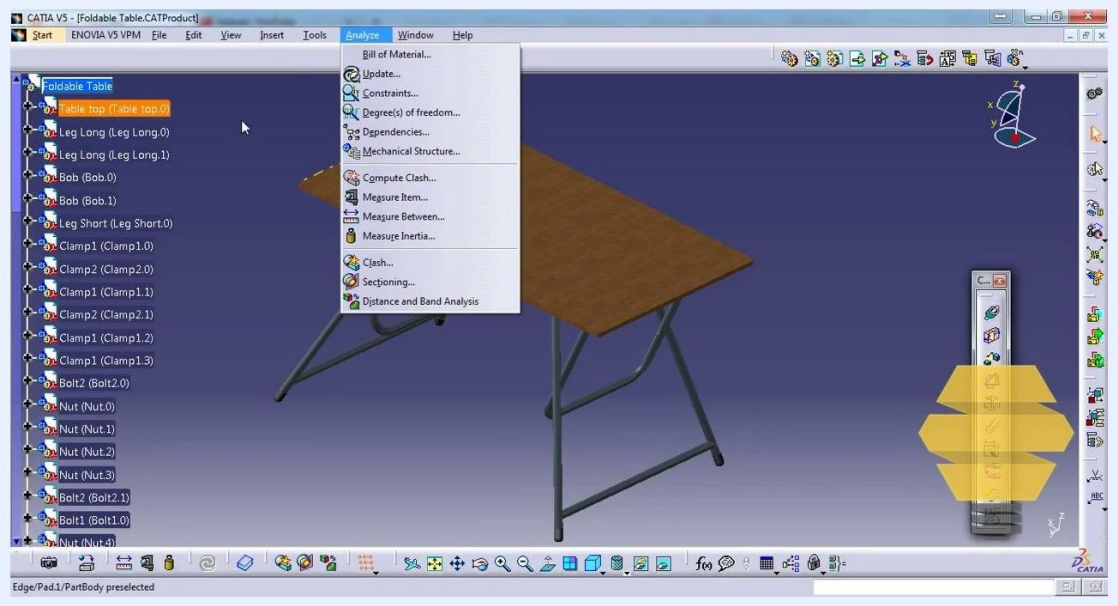
Before you begin, make sure you have the following:

- CATIA V5 installed and running.
- An assembly file, such as Foldable Table.CATProduct.
- Access to the **Assembly Design Workbench**, or to **Product Structure** within the Infrastructure Workbench.
- A text editor, such as Notepad, to open the exported file.

Step 1: Open your assembly to make a BOM in CATIA

Launch CATIA V5 and load your assembly file, for example Foldable Table.CATProduct. Make sure you are in the **Assembly Design Workbench** or in **Product Structure** within the Infrastructure Workbench.

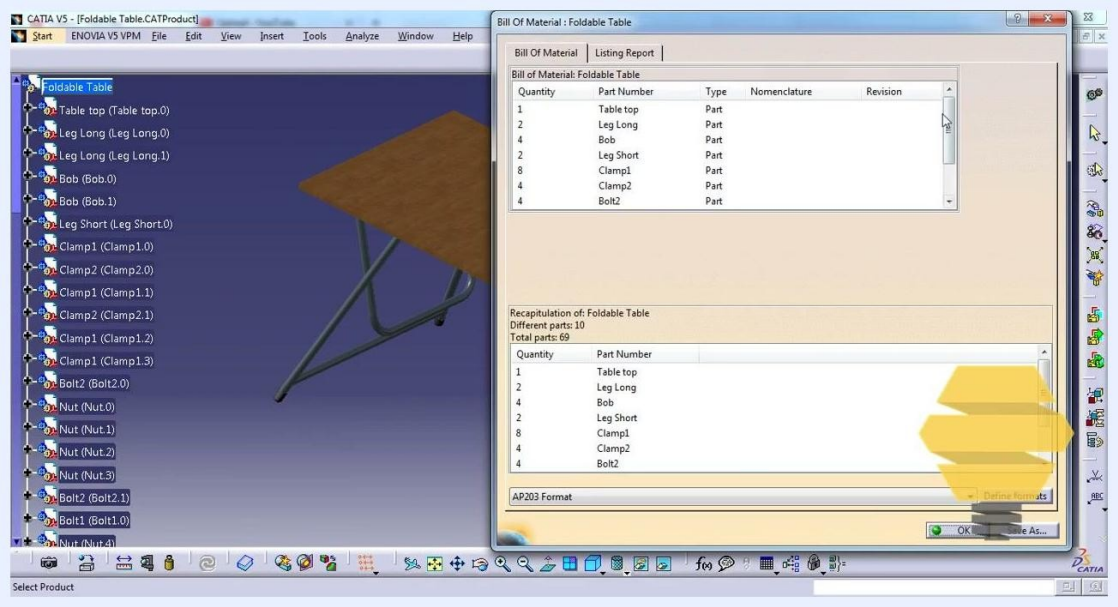
The assembly tree appears on the left and the 3D model appears in the center.



CATIA V5 interface showing the foldable table assembly loaded, with the assembly tree on the left and the 3D model in the center

Step 2: Open the Bill of Material dialog from the Analyze menu

In the top menu bar, click **Analyze**. In the dropdown, select **Bill of Material...** to open the BOM dialog.



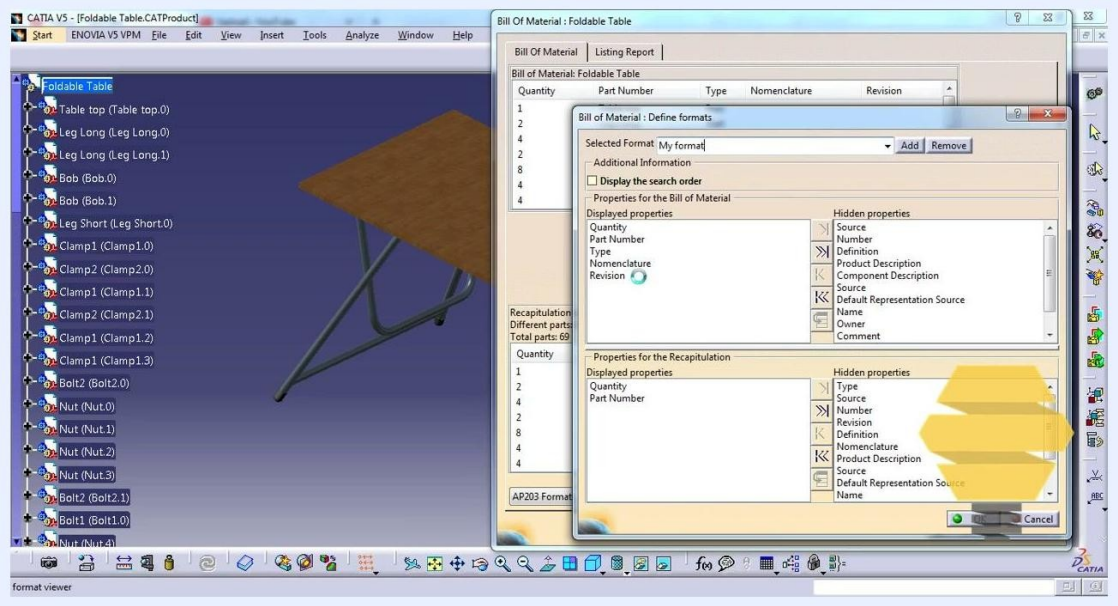
Analyze menu expanded with Bill of Material... highlighted, showing the assembly tree and table model

Step 3: Review the Bill of Materials

The **Bill of Material: Foldable Table** window appears. Review the details listed for each component:

- Quantity
- Part Number
- Type
- Nomenclature
- Revision

The lower section of the window shows a recap. In this example, it reads "Different parts: 10, Total parts: 69".



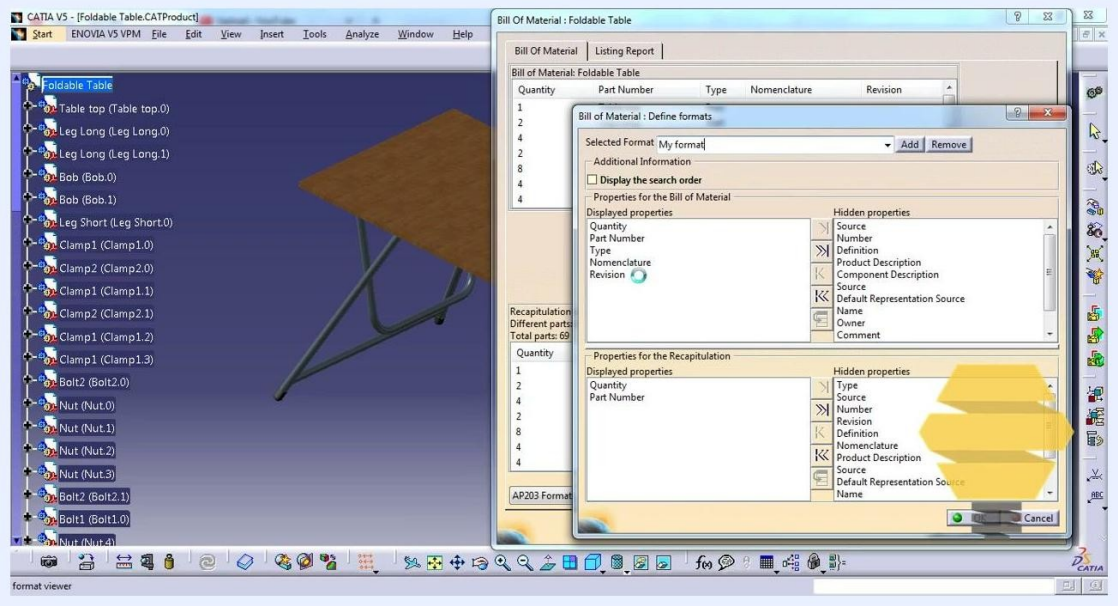
Bill of Material dialog open, displaying part quantities, numbers, types, and a summary recap

Step 4: Customize the BOM format (optional)

To change the displayed columns or the format, click **Define Formats**. Then follow these steps:

1. In the **Define formats** dialog, enter a name for your format, for example My format, and click **Add**.
2. Select the properties you want to display, such as Quantity, Part Number, Type, Nomenclature, and Revision.
3. Move any unwanted properties to the hidden list.

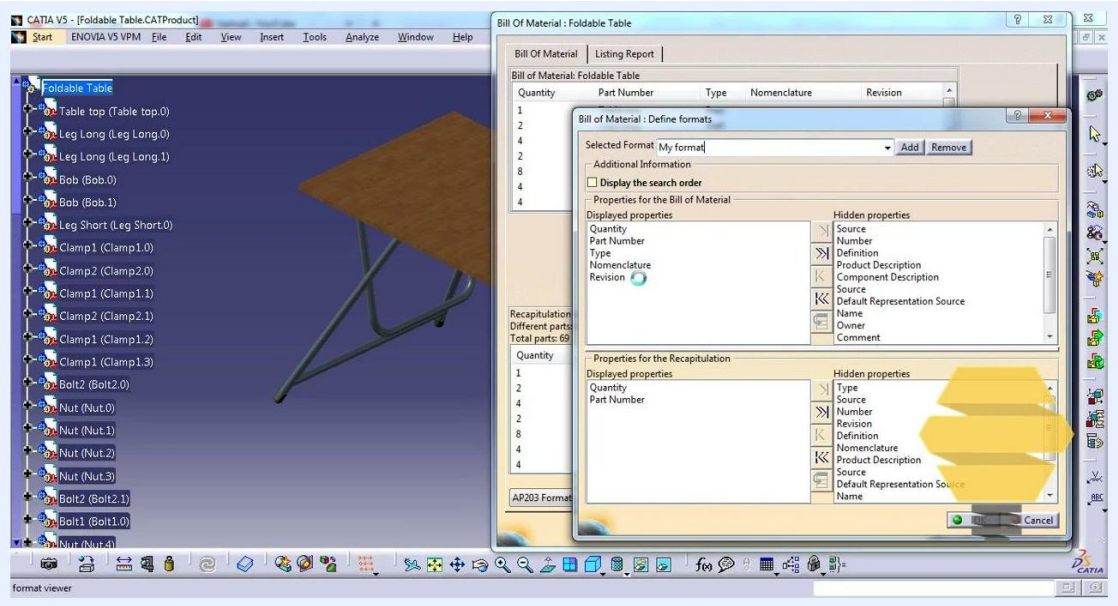
4. Click **OK** to apply your custom format.



Define formats dialog open, with My format entered and property selection options visible

Step 5: Save the BOM to a file

With your desired format selected, click **Save As** in the BOM dialog. Choose the file type, for example .txt, and enter a filename such as bill of material.txt. Click **Save** to export the BOM.



Bill of Material dialog with custom format applied, ready for export

Step 6: Check the structure of the exported BOM file

Open the exported BOM file with a text editor such as Notepad. Review the following parts of the file.

At the top, find the **Compute Date**, for example 30 Januari 2017 14:06:11. CATIA generates this timestamp automatically. It shows when the BOM was created, which is useful for version control if you have multiple BOMs.

The main BOM table lists these columns:

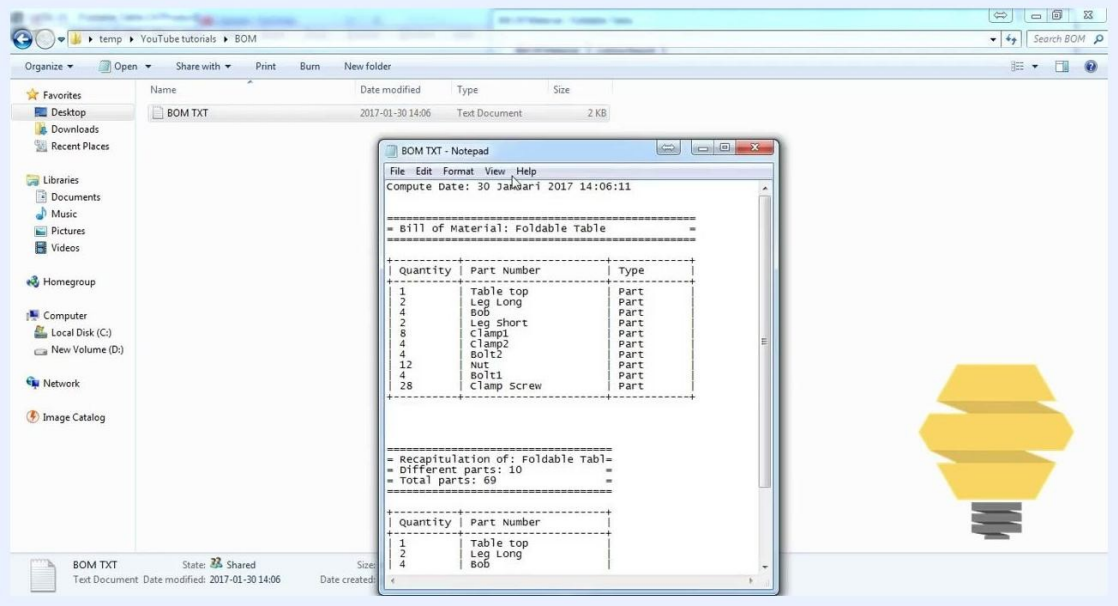
- **Quantity**: the number of each part required.
- **Part Number**: the name or identifier of each part, such as Table top, Leg Long, BoB, Leg Short, Clamp1, Clamp2, Bolt2, Nut, Bolt1, and Clamp Screw.
- **Type**: whether the item is a "Part". If the item were an assembly, it would show "Assembly".

Confirm that all parts and their quantities match your assembly structure.

At the bottom, the **Recapitulation** section summarizes the BOM:

- **Different parts**: the total number of unique parts, for example 10.
- **Total parts**: the sum of all quantities, for example 69.

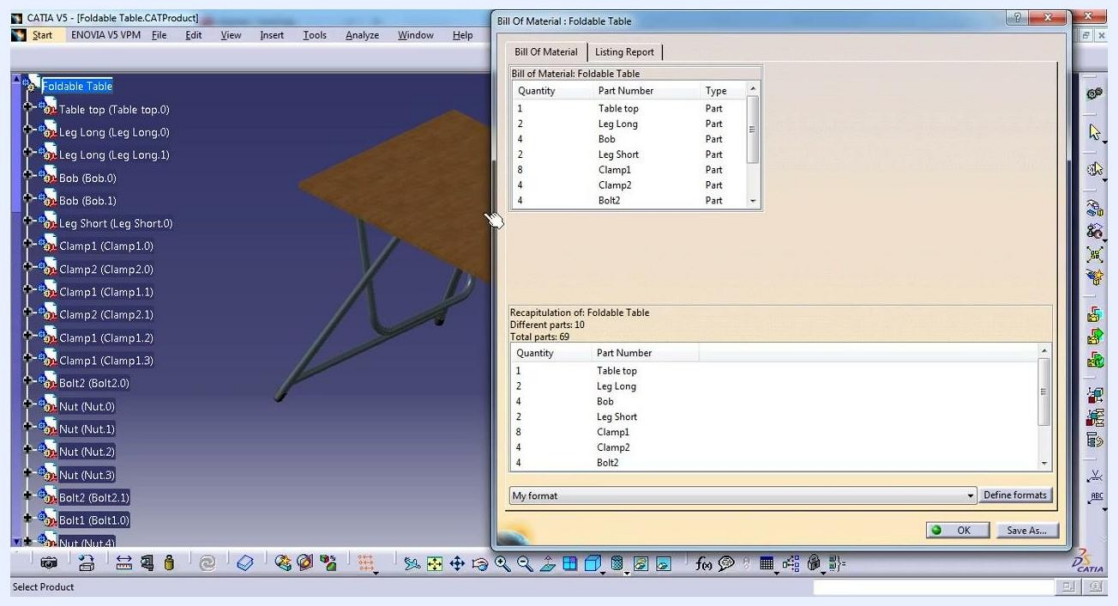
Use this summary to quickly verify that your BOM is complete and accurate.



Notepad window displaying the exported BOM TXT file, showing compute date, part list with quantities and types, and recapitulation summary

Step 7: Cross-check the recapitulation totals

In the **Recapitulation** section, verify the number of different parts (for example, 10) and the total number of parts (for example, 69). Then add up the quantities in the main BOM table to confirm that the total matches the recapitulation value.



Notepad window with the BOM TXT file, highlighting the recapitulation section with different parts and total parts

Summary

You now know how to make a BOM in CATIA V5. You opened your assembly, generated the Bill of Material from the **Analyze** menu, optionally customized the format, saved the BOM as a text file, and verified its contents and totals against the recapitulation.

What's next

- Create additional BOMs with different formats using **Define Formats**.
- Use the **Compute Date** to track versions when you keep multiple BOM files.