EDU[**CAD**]TION **STUDY GUIDE**

How to Export STEP Files

Chapter 4, Lesson 3

SendCutSend

STEP File = 3D Solid

- A **STEP** (.stp or .step) file is the universal 3D file type for sharing solid CAD models between software platforms.

- STEP files preserve accurate geometry, but unlike native CAD formats, they do not retain design history or editable features.

- In contrast, **MESH files** (STL, OBJ) represent only the outer surface and are better suited for 3D printing, not CNC machining.

Where to Export STEP Files

- **STEP** files are exported from the component level, not individual bodies.

- This is a key distinction:
    - **Components** contain one or more bodies and represent complete parts.
    - **Bodies** are individual 3D volumes without the metadata that defines assembly context.

- When exporting from the body level, Fusion will only show **mesh** options (like STL). To access the **STEP** format, right-click the component instead of the body (see Figure 1.1).

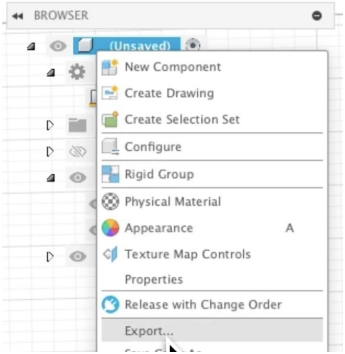

Figure 1.1

Exporting a STEP File

- **Steps**:
  1. In the browser, right-click the main component that contains your 3D body.
  2. Select **Export** (see Figure 1.2).
  3. In the export dialog box, the default format will appear as `.f3d` (Fusion's native file).
  4. Use the drop-down menu to choose STEP (.stp) as the file type (see Figure 1.3).
  5. Name the file and click **Export**.

Result: The exported STEP file contains your full 3D geometry, ready for upload to SendCutSend or most manufacturing services.

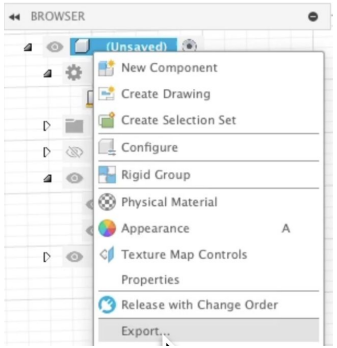

Figure 1.2

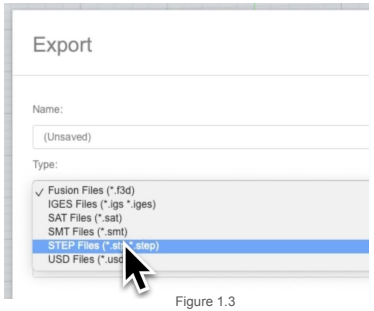

Figure 1.3

Working with Multi-Body Components

• If your design contains multiple bodies under one component, exporting the entire component will combine all bodies into a single STEP file.

• This creates a merged solid that cannot be separated during manufacturing.

- To export each body separately:
  1. Hide all but one body in the Browser by clicking the **eye icon** next to each body you want to hide (see Figure 1.4). Once only the desired body is visible, complete the steps from the previous page.
  2. Right-click the main component and select **Export**.
  3. Choose STEP (.stp, name the file (e.g., `Part_1.step`), and save.
  4. Repeat for each visible body. Hidden bodies will not be included in the export, ensuring each part is saved independently.

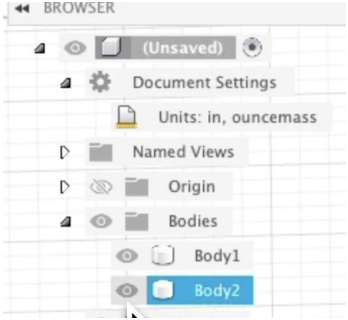

Figure 1.4

Best Practice: Separate Components

- In earlier lessons, we discussed organizing your CAD models using components rather than just bodies.

- This structure becomes critical when exporting STEP files. By maintaining one component per part, each file can be exported cleanly and shared with manufacturers without unwanted geometry overlap.

Alternate Export Location

Another method to export a STEP file:

- Go to File → Export in the top menu (see Figure 1.5).
- Choose your desired file type (DXF, STEP, etc.). You will see all the file types here (see Figure 1.6).

Note: *attempting to export a DXF from a 3D body will trigger an error, DXFs are for 2D geometry only.*

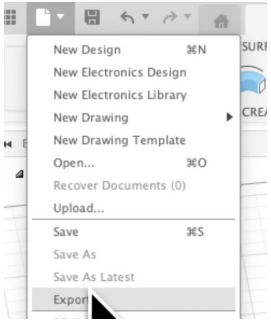

Figure 1.5

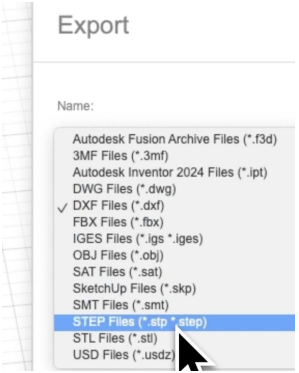

Figure 1.6

STEP File Limitations

- **No Design History**: STEP files are static solids. They do not include sketches, extrusions, or timelines. Once exported, the parametric design data is lost.

- **Limited Editability**: Manufacturers cannot modify STEP files easily. If a change is needed, you must return to your original CAD file (e.g., Fusion 360 or SolidWorks), make the adjustment, and re-export.

- **The Manufacturing Handshake**: Sending a STEP file to a manufacturer signifies approval, it's your agreement that the part is ready for production. Once shared, edits are your responsibility, not the shop's.

Troubleshooting

- **Issue**: The STEP option doesn't appear in the export menu.
  Fix: Ensure you're right-clicking the component, not a body.

- **Issue**: Multiple bodies merge into one STEP file.
  Fix: Hide all unwanted bodies before exporting.

- **Issue**: Manufacturer can't modify your file.
  Fix: Make changes in your CAD software and re-export the STEP file.

Summary

STEP files are the bridge between your CAD environment and real-world manufacturing. They provide accurate geometry for machining, fabrication, and inspection. Exporting correctly from Fusion 360 ensures clarity, precision, and compatibility with services like SendCutSend.

Learn more at https://sendcutsend.com/education/