

Detail SolidWorks Parts

HOW-SOLIDWORKS-0408 / REV IMPORT-01 / 2026.09.06

This conversation emphasized that an assembly is not required just to create fabrication drawings. A drawing can use Select Bodies or Relative View to isolate one body from the master model, then add projected, section, detail, dimension, and balloon information. A weldment cut list can organize identical items, while Save Bodies is reserved for cases that genuinely need separate released part documents, reuse, configurations, or an assembly deliverable.

STATUS	Recovered / Starting Point
CLASSIFICATION	HOW
MATERIAL	SOLIDWORKS DRAWING / MULTIBODY DETAILING / WELDMENT
DISPOSITION	PUBLIC / LOCAL SUMMARY / WORKFLOW NOTE
CATEGORY	CAD / SolidWorks
RECORD TYPE	drawing-workflow

TECHNICAL NOTES

A recovered method for detailing individual bodies from a SolidWorks multibody or weldment part without converting the entire model first.

GOAL

Produce clear individual fabrication details while keeping the coordinated multibody or weldment model as the source of truth.

EQUIPMENT

- SolidWorks
- Drawing template
- Master multibody or weldment part
- Cut-list properties

METHOD / STEPS

1. Keep intended components as separate solid bodies and update the weldment cut list.
2. Populate consistent cut-list properties such as part number, description, material, thickness, finish, route, length, and quantity.

3. Create an overall drawing with isometric, orthographic, balloons, and a weldment cut list.
4. Use Model View > Select Bodies for a quick individual-body detail.
5. Use Relative View when the body needs a clean manufacturing orientation.
6. Add projected, section, detail, hole callout, dimension, and item-balloon information.
7. Use Save Bodies when independent files, configurations, reuse, or a conventional assembly are genuinely required.

RESULT / WHERE THIS STANDS

A practical drawing strategy that avoids unnecessary file proliferation; the next step is testing it on one representative multibody model.

TROUBLESHOOTING

- Rename the cut-list item, not only the body, when manufacturing information must appear in the drawing.
- Sort the cut list before ballooning because later sorting can change item numbers.
- Imported model dimensions can create clutter; place deliberate manufacturing dimensions when the source is not drawing-ready.
- Check that identical geometry and material are grouped as intended.

SAFETY / LIMITATIONS

- Verify dimensions, material, quantity, and revision information before releasing a fabrication drawing.
- Do not assume a linked drawing is independent after files are moved or references are changed.

SPECIFICATION

FASTEST METHOD	Model View with Select Bodies
ANGLED PARTS	Relative View using two perpendicular faces or planes
DRAWING SET	Overall views, cut list, individual body sheets
PROPERTIES	Part number, description, material, thickness, finish, route
SEPARATE FILES	Use Save Bodies only when a released part is actually needed

DEVELOPMENT HISTORY

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Imported as a separate detailing workflow record from the supplied SolidWorks conversation; confidential system references were omitted.

RELEASED MATERIAL / SOURCE LINKS

HOW-SOLIDWORKS-0408-public-summary.txt

PUBLIC SUMMARY / WORKFLOW NOTE

<https://u225x.com/downloads/record-notes/HOW-SOLIDWORKS-0408-public-summary.txt>

CONNECTED RECORDS

HOW-WELDMENT-0409

<https://u225x.com/record/HOW-WELDMENT-0409>

SYS-SOLIDWORKS-0411

<https://u225x.com/record/SYS-SOLIDWORKS-0411>

RETURN TO THE LIVING RECORD

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This PDF preserves the public notes at this revision. Visit the record for updates and attached images.

<https://u225x.com/record/HOW-SOLIDWORKS-0408>