

Convert Weldment to Assembly

HOW-WELDMENT-0409 / REV IMPORT-01 / 2026.09.06

The cleanest general path in this recovered conversation is Save Bodies: prepare and rebuild the master part, update the cut list, confirm separate solid bodies, name the outputs, and create the assembly from the same workflow. The resulting parts normally retain the original coordinate relationship, which is useful for rigid fabricated frames. Moving components still need real assembly mates, and every generated file should be checked for transferred properties, naming, references, and BOM behavior.

STATUS	Recovered / Starting Point
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CLASSIFICATION	HOW
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MATERIAL	SOLIDWORKS WORKFLOW / WELDMENT / MULTIBODY PART
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DISPOSITION	PUBLIC / LOCAL SUMMARY / WORKFLOW NOTE
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CATEGORY	CAD / SolidWorks
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RECORD TYPE	software-workflow
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TECHNICAL NOTES

A practical workflow for turning a SolidWorks weldment or multibody part into derived parts and an assembly with Save Bodies.

GOAL

Create usable component parts and an assembly from a coordinated weldment or multibody master model.

EQUIPMENT

- SolidWorks
- Master weldment or multibody part
- Assembly and drawing templates
- Project folder with controlled file names

METHOD / STEPS

1. Rebuild the master model with Ctrl+B and Ctrl+Q.
2. Update the Solid Bodies or Cut List folder and remove unwanted construction bodies.
3. Confirm which bodies should remain separate and give them meaningful names.
4. Use Insert > Features > Save Bodies and provide controlled part filenames.
5. Choose Create Assembly when an assembly deliverable is needed.
6. Rebuild the derived parts and assembly, then verify positions, references, transferred properties, drawings, and BOM output.
7. Keep references during active design and lock them only as a deliberate release step.

RESULT / WHERE THIS STANDS

A repeatable starting workflow for a rigid weldment; the exact property and BOM behavior still needs testing on a representative model.

TROUBLESHOOTING

- Identical members may need repeated instances instead of unique filenames.
- Cut-list properties and document custom properties are not always the same field.
- A generated part can retain a link to the master, so moving or renaming files can break references.
- Moving components must be floated and mated intentionally rather than relying on a rigid weldment layout.

SAFETY / LIMITATIONS

- Work from copies and a controlled project folder before changing references.
- Verify the resulting BOM and manufacturing properties before release.

SPECIFICATION

PRIMARY COMMAND	Insert > Features > Save Bodies
BEFORE EXPORT	Rebuild master part and update the cut list
OUTPUTS	Derived part files plus optional assembly
RIGID WELDMENT	Keep common coordinates; avoid unnecessary mates
VALIDATION	Check properties, references, BOM, and filenames

DEVELOPMENT HISTORY

2026.09.06 / IMPORT-01

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

RELEASED MATERIAL / SOURCE LINKS

HOW-WELDMENT-0409-public-summary.txt

PUBLIC SUMMARY / WORKFLOW NOTE

<https://u225x.com/downloads/record-notes/HOW-WELDMENT-0409-public-summary.txt>

CONNECTED RECORDS

HOW-SOLIDWORKS-0408

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

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-WELDMENT-0409>