

FreeCAD

Import and Edit STEP File

Import

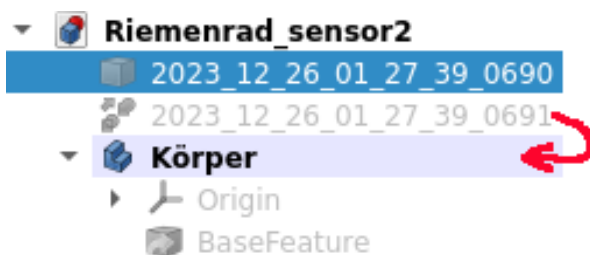
1. Launch FreeCAD
2. Create a new project (File - New)
3. Import STEP (stp) file: (File - Import)

Convert Imported Drawing

1. Workbench: Select Part
2. Refine shape (Part - Create copy - Refine shape)
3. Check the result. Select the newly created „Part“ and then: Part - Check geometry
4. Run the check, no errors should be displayed. Close.

Move the Part into an Editable Body

1. Workbench: Select Part Design
2. Create a new body
3. Drag the part with the mouse onto the new body, this creates the „BaseFeature“:



The BaseFeature is the imported form ready for editing. Here, all operations available in FreeCAD can now be performed. Afterwards, save, export, etc., as usual.

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Last update: **2023/12/26 01:13**

