

Working with STL Files in FreeCAD

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12 November 2025



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Learning Objectives

In this tutorial we will learn to,

- ▶ **Export a FreeCAD model to STL format**
- ▶ **Import an STL file into FreeCAD**
- ▶ **Convert an STL mesh into a solid**
- ▶ **Refine the solid and make modifications**



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System Requirements

To record this tutorial, I am using:

- ▶ **Windows 11**
- ▶ **FreeCAD v1.0.0**

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Pre-requisites

To follow this tutorial,

- ▶ **Learners should know the basics of FreeCAD interface and navigation**



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Summary

In this tutorial we learnt to,

- ▶ **Export a FreeCAD model to STL format**
- ▶ **Import an STL file into FreeCAD**
- ▶ **Convert an STL mesh into a solid**
- ▶ **Refine the solid and and modify it in Part Design**



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Assignment

As an Assignment,

- ▶ **Import Example.stl file into FreeCAD**
- ▶ **Convert it to a solid and refine it**
- ▶ **Create a circle of 30 mm in the centre of the hole**
- ▶ **Extrude the sketch to 70 mm**



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