



# How to open stp file in solidworks?

## Description

After several searches on the internet on a question like How to open stp file in solidworks?, I could see the lack of information on CAD software and especially of answers on how to use for example Solidworks. Our site CAD-Elearning.com was created to satisfy your curiosity and give good answers thanks to its various Solidworks tutorials and offered free.

Engineers in the fields of technical drawing use Solidworks software to create a coherent design. All engineers must be able to meet changing design requirements with the suite of tools.

This CAD software is constantly modifying its solutions to include new features and enhancements for better performance, more efficient processes.

And here is the answer to your How to open stp file in solidworks? question, read on.

## Introduction

1. STEP AP203/214/242 (\*.step; \*.**stp**).
2. IGES (\*.igs; \*.iges)
3. ACIS (\*.sat)

Additionally, how do I open a **STP file**?

1. Find the STEP file on your computer or device.
2. Double click or right-click and select **File > Open**.
3. Your **file** will **open** with the associated STEP reading software already installed, like Adobe Acrobat.

Furthermore, how do I convert a STEP file to SOLIDWORKS?

1. Click Open (Standard toolbar) or File > Open.
2. In the dialog box, in Files of type, select the desired file format: STEP AP203/214/242 (\*).
3. In the dialog box, browse to the desired file.
4. Click Options.

5. In the System Options dialog box, set options including:
6. Click OK.

As many you asked, how do I edit a STP file in SOLIDWORKS?

1. In the FeatureManager design tree, right-click the feature created from the imported document, and select Edit Feature.
2. In the Files of type list, select the desired format.
3. Browse to select the desired file to import.

Also the question is, what is a **SOLIDWORKS** STP file? Doing so is a very simple process. Open up the **file** using Onshape, and once inside, select the object, right-click anywhere on the viewport and select "Export". Once there, simply choose the file format as STL, and now you can convert STEP to STL files, offline, for free!

## How can I view STP files for free?

FreeCAD. FreeCAD is a free and open-source CAD designing software that allows you to view STP files. It can be used to view various other 2D/ 3D model files too, such as DWG, DXF, OBJ, DAE, PLY, IGS, IFC, STL, WRL, VRML, etc.

## How do I open a STEP file in SOLIDWORKS 2020?

1. Click Open (Standard toolbar) or File > Open.
2. In the dialog box, in Files of type, select the desired file format: STEP AP203/214/242 (\*.
3. In the dialog box, browse to the desired file.
4. Click Options.
5. In the System Options dialog box, set options including:
6. Click OK.

## Are STP and STEP files the same?

The answer is that there is no difference between a \*. step file and a \*. stp file. Both are related, interchangeable and meet ISO standards.

## Can SOLIDWORKS export STP?

You can set the export options when you export SOLIDWORKS part or assembly documents as STEP files. To set the STEP export options: Click File > Save As. Select a STEP file type for Save as type, then click Options.

## How do you import bodies into SOLIDWORKS?

1. With a part document open, click Insert Part (Features toolbar) or Insert > Part.

2. Browse to a part document and click Open.
3. Under Transfer, select any combination of items from the inserted part to be included in the derived part.

## **How do you extract parts from an assembly in SOLIDWORKS?**

1. Click Tools > Options > System Options > Export.
2. For File Format, select SLDPRT from assembly.
3. Specify the following options, and click OK.

## **How do I enable FeatureWorks in SOLIDWORKS?**

To set FeatureWorks to activate whenever you open SOLIDWORKS, click Tools > Add-Ins and select FeatureWorks under the Active Add-ins and Start-up columns.

## **How do I convert STEP files to Sldasm?**

Open solidworks, on the top part there is a button: "Open". Then select your file on there and start waiting. After that click to another button: "Save As". Finally select your file extension: ".sldasm".

## **How do I convert STP to PDF?**

Open your. Stp file with a reader which can open the file. Print the file on the virtual PDF24 PDF printer. The PDF24 assistant opens, where you can save as a PDF, email, fax, or edit the new file.

## **Can I open STP file in AutoCAD?**

Though AutoCAD opens STEP (STP) files, it does not natively support the STEP file format. It does not read or write STP files natively; instead, converting files using a built-in translation tool.

## **Is STP the same as STL?**

The STP format uses a data algorithm that presents the data in more detail than the STL format. This is due to the transformation of the surface into a kind of mesh of triangles, simplifying the representation of shape in STL format.

## **Can you 3D print a STP file?**

STEP files are a useful 3D file type due to their precision and customizability. Read on to learn more about the STEP file format! There are hundreds if not thousands of different file types out there, and dozens of them just in the realm of 3D printing and computer-aided design (CAD).

## Can Cura use STEP files?

Supported CAD File Formats With Cura 4.10, the following CAD file formats are currently supported, with more to be added in the future: STEP. IGES. DXF/DWG.

## What software can open a STEP file?

You can open a STEP file in a variety of CAD programs, including Autodesk Fusion 360 (cross-platform), IMSI TurboCAD (Windows, Mac), FreeCAD (cross-platform), and Dassault Systemes CATIA (Windows, Linux).

## Can Adobe Open STP file?

Adobe RoboHelp opens STP files that are for stop lists. You can use Microsoft's SharePoint for opening STP files that are SharePoint Template files.

## Bottom line:

Everything you needed to know about How to open stp file in solidworks? should now be clear, in my opinion. Please take the time to browse our CAD-Elearning.com site if you have any additional questions about Solidworks software. Several Solidworks tutorials questions can be found there. Please let me know in the comments section below or via the contact page if anything else.

The article clarifies the following points:

- How can I view STP files for free?
- How do I open a STEP file in SOLIDWORKS 2020?
- How do you extract parts from an assembly in SOLIDWORKS?
- How do I enable FeatureWorks in SOLIDWORKS?
- How do I convert STEP files to Sldasm?
- How do I convert STP to PDF?
- Can I open STP file in AutoCAD?
- Is STP the same as STL?
- Can Cura use STEP files?
- What software can open a STEP file?