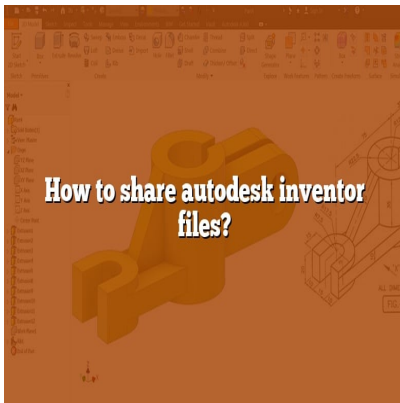




# How to share autodesk inventor files?

## Description

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

Engineers in the fields of technical drawing use Autodesk inventor 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 share autodesk inventor files? question, read on.

## Introduction

1. Open the file you want to **share** a view of.
2. Access one of the following: Select File **Share** Share View.
3. In the Create a Shared View dialog box, specify options.
4. Select Share to upload a visual representation of the file in **Autodesk** Viewer.

Best answer for this question, how do I send an Inventor file?

1. Open the part or assembly file you want to share.
2. The 3D viewable will be created from the present view of the open part or assembly file.
3. If not already open, click the Design **Share** button in the bottom right corner of the graphics window .

You asked, how do I export from Autodesk Inventor? Right-click a sketch in the model browser, and select Export Sketch As DWG (AutoCAD/LT or DXF (AutoCAD R12/LT2). Saves a file with a specified name and file type. Exports **Inventor** data to other file formats. Saved **files** remain open.

Subsequently, can you collaborate on Inventor? In detail, **Inventor** Connected Design allows users to: Collaborate on 3D designs on virtually any device. Frequently share designs-in-progress without committing changes or otherwise impacting data management. Conduct reviews much more quickly than with traditional methods such as 2D screenshots and/or email.

People ask also, how do I send Inventor assembly? If you have Solidworks and Inventor installed on the same computer then you can do the following: Open Solidworks. Import a **Inventor** part / assembly file. Solidworks will open Inventor in the background and convert the part / assembly file.

## How do I save an Autodesk Inventor file as an STL?

1. Navigate to File>Save As>Save Copy As.
2. Select Save as type and select STL.
3. Select Options and input your required scale settings.

## How do you save an Inventor as a DWG?

1. From the File menu, click Export Export to DWG.
2. In the DWG Model File Export Options dialog box, set the following options, and then click Next. Specify the object types to export.
3. Click OK to create the drawing.

## How do you save an Inventor as an STL?

1. Select IPro > Print > 3D Print Preview.
2. Select Options and choose desired resolution and click OK.
3. Within the preview window, select Save Copy As or Send to 3D Print Service.
4. Save As type to STL File (\*.stl)

## Is fusion better than Inventor?

Inventor will provide you with a wide range of tools for very complex objects and Fusion 360 with professional options for mechanical projects. Although similar, Fusion 360 has a bit more of a fresh look and Inventor is more stable and traditional. Both though will give you stable performance and necessary tools.

## Can we collaborate in MIT App Inventor?

The collaborative programming environment within MIT App Inventor provides users a new approach to teach and learn. For example, it enables “teacher-student” or “mentor-mentee” roles inside MIT App Inventor. Teachers can share the projects with students in read-only mode to demonstrate ideas and demos.

## Can you collaborate on MIT App Inventor?

I present a collaborative programming environment to MIT App Inventor, a web-based visual platform for building Android applications with blocks, which enables many programmers to collaborate in real time on the same MIT App Inventor project.

## How do I save an assembly file in Inventor?

1. Save the files, and then close Inventor.
2. Then, using Windows Explorer, drag and drop the files from your local drive to your connected drive.

## Can Fusion 360 Open Inventor files?

Fusion 360 offers the ability to open Inventor parts (IPT) and assemblies (IAM) directly. After selecting the file the upload process begins, which you can dismiss and allow to complete in the background.

## What is the file extension for Autodesk Inventor part files?

Unlike AutoCAD and Mechanical Desktop, which maintain one file with multiple types of data, Autodesk Inventor maintains data in multiple associated files. 3D part model data is contained in a file with the extension . ipt.

## Which is better SOLIDWORKS or Inventor?

Features. Compared to Autodesk Inventor, SOLIDWORKS has much more powerful solid feature options, allowing for much more complex geometries to be created faster and easier.

## How do I convert Autodesk to SOLIDWORKS?

1. EXPORTTOAUTOCAD. A new dwg will be created which converts all specific Plant-components like pipe, equipment, to 3D Solids. This converted file can be imported to Solidwords. â&#x2013;
2. Using Export-function and selecting iges-format. Afterwards import the iges-file into SolidWorks.

## How do I open an IPT File in SOLIDWORKS?

Click Open (Standard toolbar) or File > Open. In the Open dialog box, set Files of type to Inventor Part (\*. ipt) or Inventor Assembly (\*. iam) and click Options.

## How do I convert an IPT file to STL?

## What does STL file stand for?

The name STL is an acronym that stands for stereolithography – a popular 3D printing technology. You might also hear it referred to as Standard Triangle Language or Standard Tessellation Language. Each file is made up of a series of linked triangles that describe the surface geometry of a 3D model or object.

## How do I create an STL file?

1. Choose Your Preferred CAD Program and Open It.
2. Create 3D Models Using Your Software's Tools.
3. Save and Export Your Design to the Computer.
4. Choosing a Slicer Software.
5. Slicing Your File Into a G-code File.

## Wrap Up:

I believe you now know everything there is to know about How to share autodesk inventor files?. Please take the time to examine our CAD-Elearning.com site if you have any additional queries about Autodesk inventor software. You will find a number of Autodesk inventor tutorials. If not, please let me know in the comments section below or via the contact page.

The article makes the following points clear:

- How do you save an Inventor as an STL?
- Is fusion better than Inventor?
- Can we collaborate in MIT App Inventor?
- Can you collaborate on MIT App Inventor?
- What is the file extension for Autodesk Inventor part files?
- Which is better SOLIDWORKS or Inventor?
- How do I open an IPT File in SOLIDWORKS?
- How do I convert an IPT file to STL?
- What does STL file stand for?
- How do I create an STL file?