



Autodesk® Fusion 360™ for the Autodesk® Inventor® User

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MA1514

This class applies an Autodesk® Inventor® software user's perspective to sketching and modeling techniques in Autodesk Fusion 360 software as a service (SaaS). We also present some use cases from that perspective. Have you ever wanted to learn a new program but the instructor didn't understand your background or know where you were coming from? Then this is the class for you! We cover how to sketch and model in the Fusion 360 model environment and explore how it differs from Inventor by using history-free modeling with push/pull concepts. We explore the sculpt environment and talk about how the understanding of topology differs from modeling in Inventor. Finally, we show how T-spline modeling can solve the problem of complex surface modeling in Inventor.

Learning Objectives

At the end of this class, you will be able to:

- Identify the Autodesk Fusion 360 environments and explain how they differ
- Sketch and model in the Fusion 360 model environment
- Explain how the Fusion 360 sculpt environment and T-splines can be used to solve complex surfacing challenges in Inventor
- Use Fusion 360 with Inventor data

About the Speaker

Based in Novi, Michigan, Timothy works as a technical consultant for i GET IT Online Training For Engineers from Tata Technologies. Focusing on online e-training for engineers using Autodesk software, Timothy has created numerous online training courses for i GET IT in both text and video format. Courses he has created include the programs Autodesk® Inventor®, AutoCAD®, Autodesk® Revit®, Autodesk® Inventor® Fusion, & Autodesk® Fusion 360. Prior to working for Tata Technologies, Timothy worked in manufacturing as a designer and engineer in the tooling and special machine field for 15 years. Most of that time was in Automotive manufacturing and assembly using Autodesk® Inventor®.

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Feature Comparison

Features	Autodesk Fusion 360	Autodesk Inventor 2014
Parametric Modeling (History Based)		X
History Free Modeling	X	X (Edit Base Solid)
2D Sketching	X	X
3D Sketching		X
3D Solid Modeling	X	X
3D Surface Modeling	X	X
T-Spline Modeling	X	
Multibody Parts	X	X
Assembly Files		X
Assembly Joints	X (Multibody Components)	X
Assembly Constraints		X
Cloud File Storage	X	X
Local File Storage		X
Autodesk Vault support		X
Local License Required		X
Access From Any Machine With Internet	X	
Rendering	X	X
Drawings		X
Sheet Metal Parts		X
Plastic Part Feature Tools		X
Weldments		X

This is list of major features*, not all features shown.

* Features as of 12/01/14

Autodesk Fusion 360 Workspaces

Sculpt



The Sculpt workspace is used to create organic shapes for industrial design.

Using T-Splines you can create complex surface bodies much easier than in other 3D design software programs.

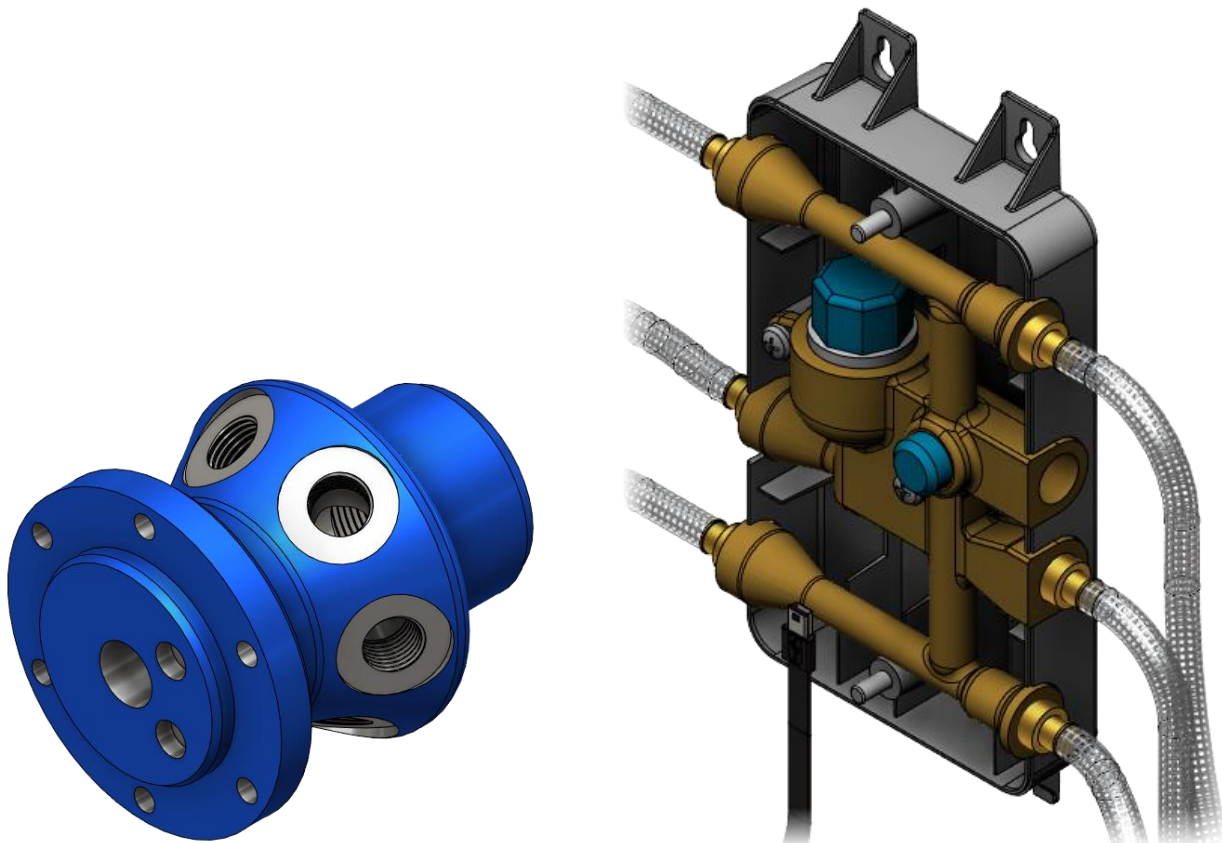


Model



The Model workspace is used to create prismatic shapes for mechanical design or backside geometry.

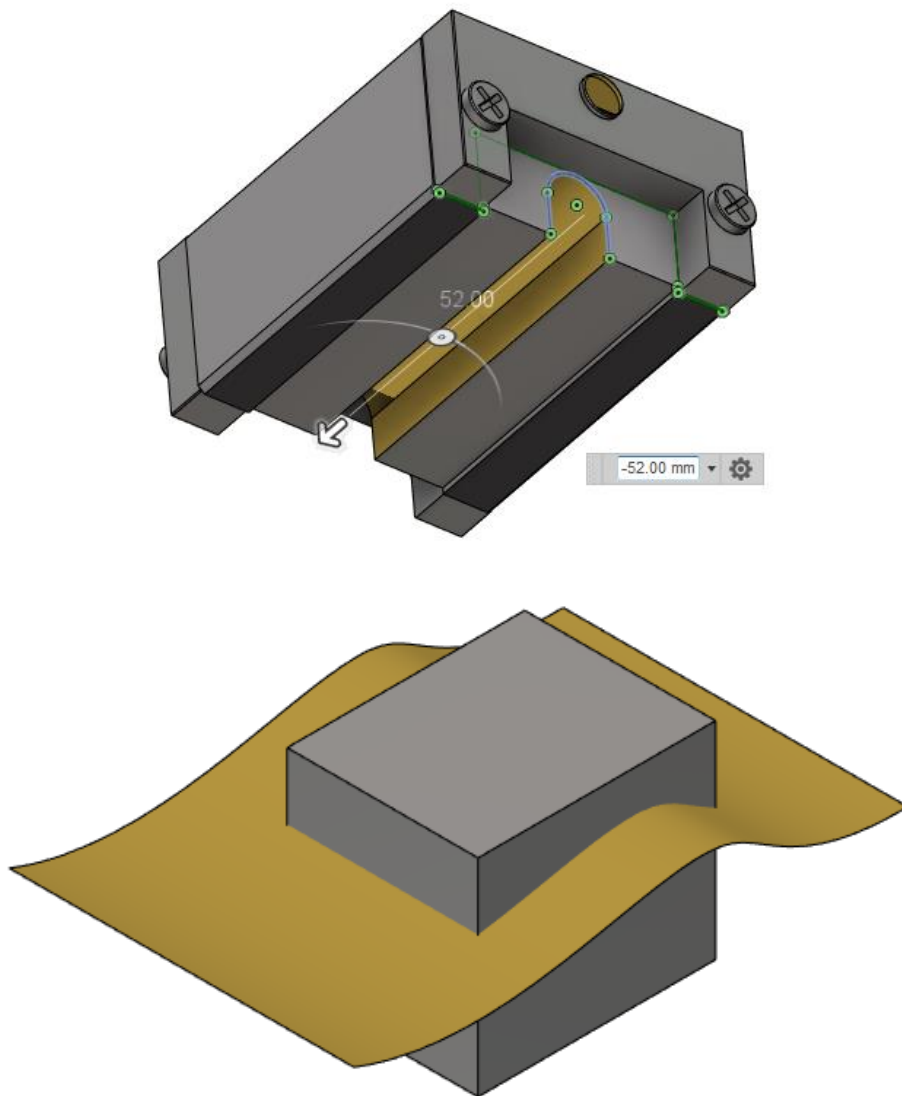
Using more traditional modeling methods in a History Free environment, direct manipulation is used to edit and modify geometry.



Patch

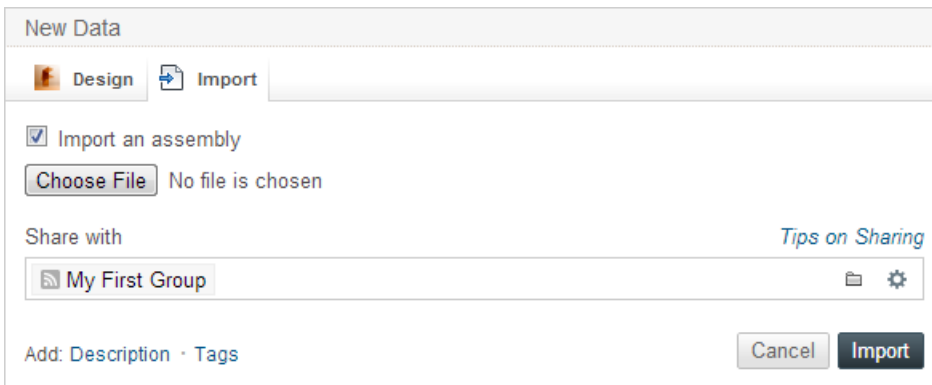


The Patch workspace is used to create construction surfaces and repair imported surfaces.

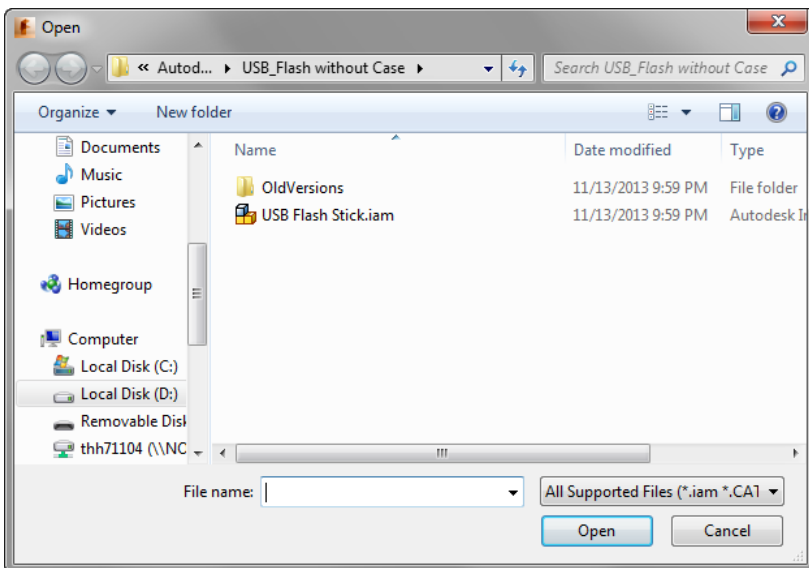


Importing Inventor Data

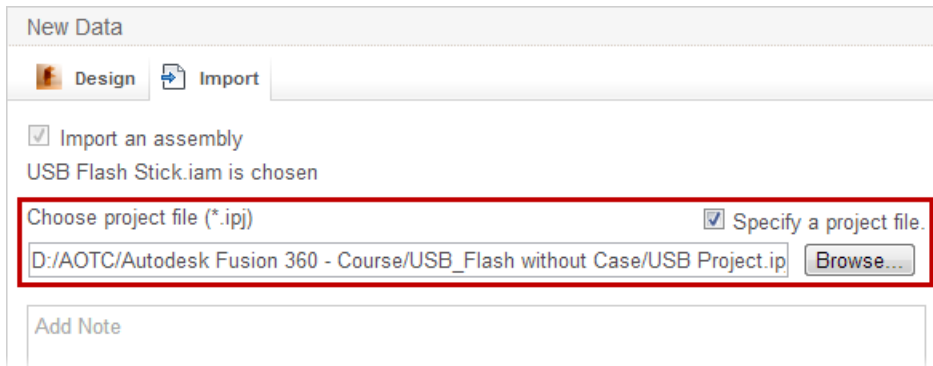
- Import New Data from the **User Hub**
NOTE: Select Import an Assembly to import an Autodesk Assembly



- Choose file(s) to import

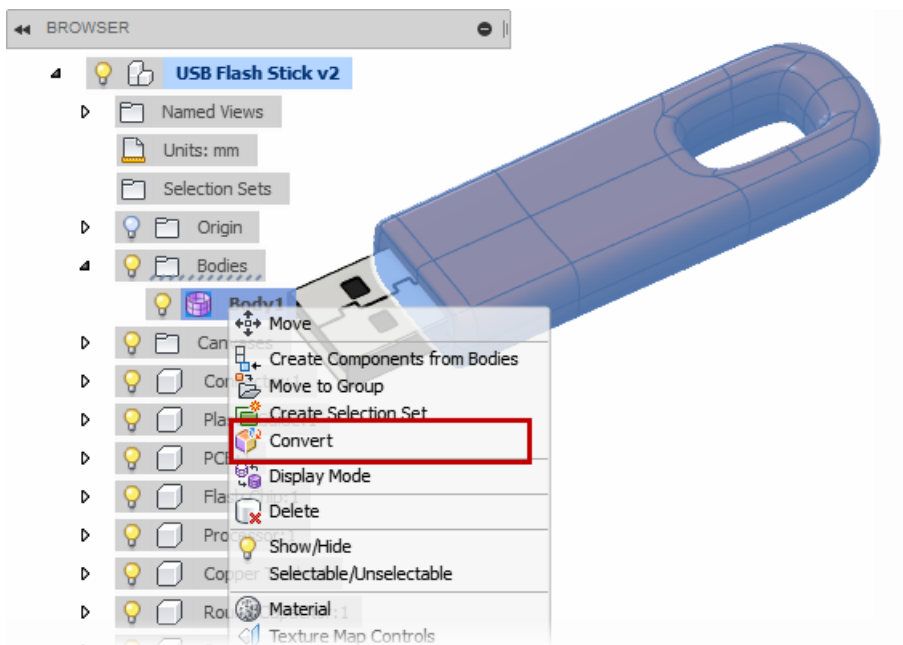


- If importing an assembly file, select the appropriate project file.

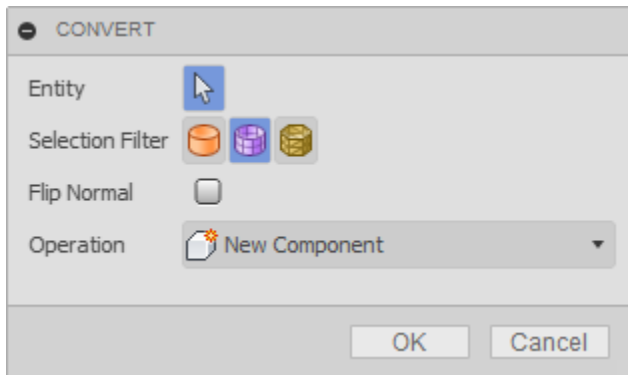


Exporting Design For Use In Autodesk Inventor

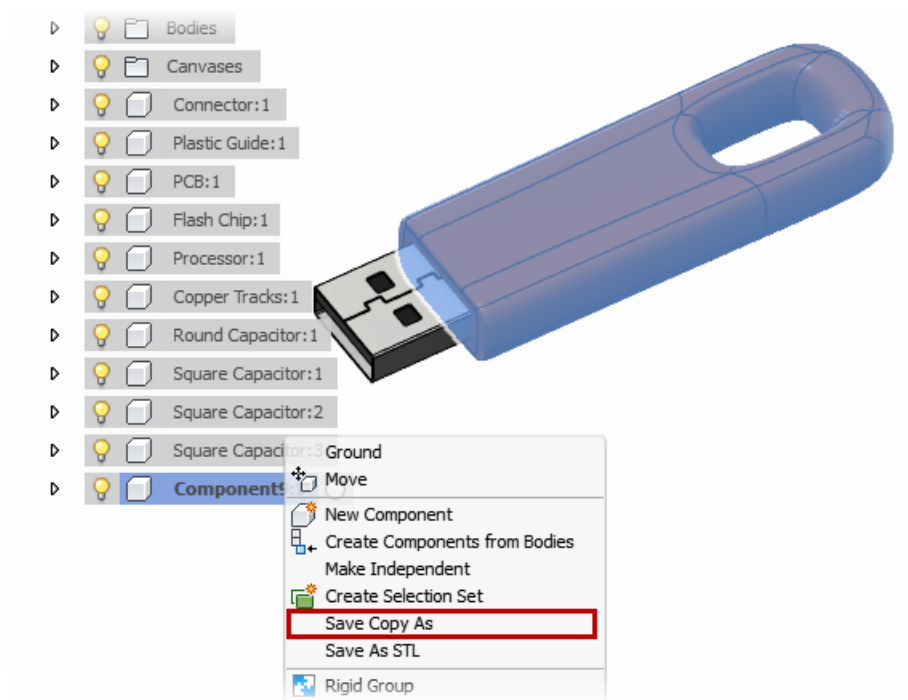
- Convert T-Spline bodies to BRep solid model



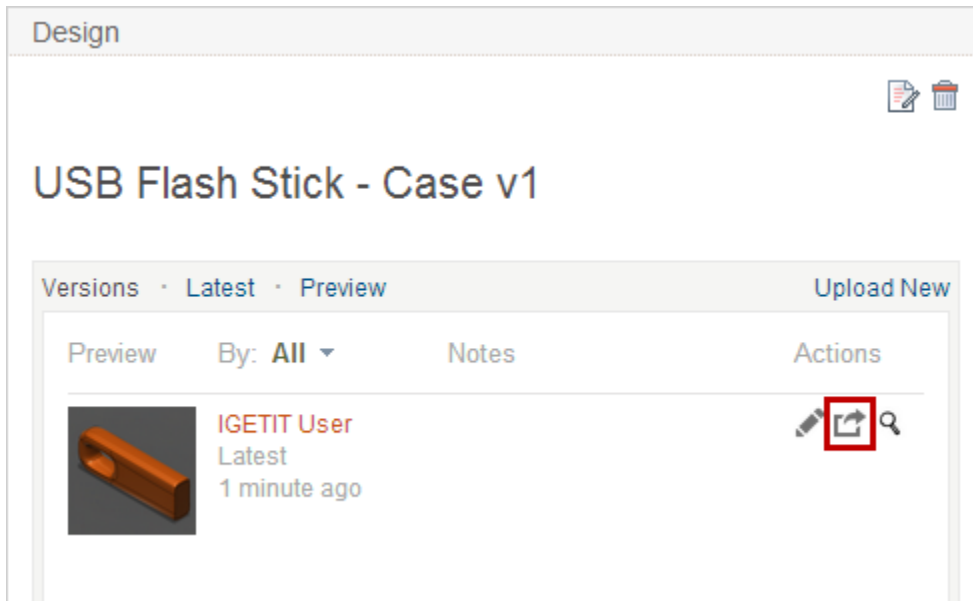
- Create as a **New Component**



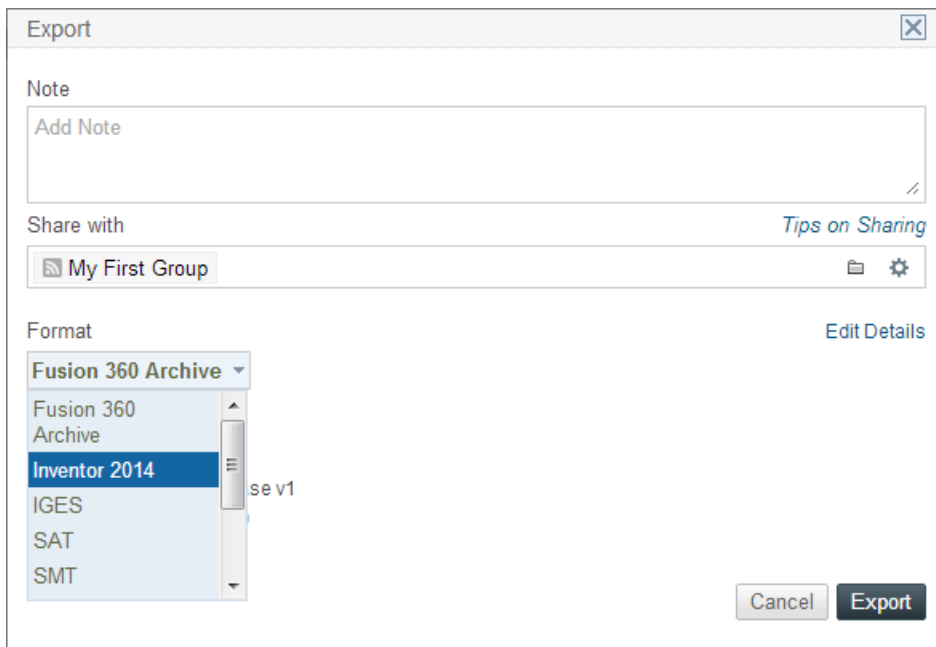
- Right click the new component in the Browser
- Select **Save Copy As**



- In the **User Hub**, Select the newly saved design and **Export** it from the **Versions**.



- Export as **Inventor 2014**



- When export operation is complete, Download the file.

File

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Additional Information



90 Day Free Trial

<http://fusion360.autodesk.com>



Online Training for Engineers

Sign up for a free account.

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Free course available - 8 hour long course - Videos and Projects.

Autodesk Fusion 360 Basics