AUTODESK UNIVERSITY

IM501506

Turbocharge How You Design and Manage Large Assemblies in Autodesk Fusion 360

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Learning Objectives

- Learn new methods on how to effectively manage large assemblies in Autodesk Fusion 360.
- Learn the difference between top-down and bottom-up modeling.
- Learn how to simplify models to improve assembly performance.
- Learn how to use the manage extension to manage several parts in a multidisciplinary team.

Description

Modern consumer products and industrial machines are made up of hundreds or thousands of components to form large assemblies. These components could be machined, 3D printed, molded, electrical, electronic, or off-the-shelf items using various manufacturing and design processes. Effectively managing these assemblies can be a real challenge for companies with multidisciplinary teams that simultaneously contribute to the project from all Autodesk Fusion 360 workspaces. This technical instruction will provide attendees with techniques and methods to improve their development processes when creating large assemblies. Such methods will include: simplifying into subassemblies, using top-down or bottom-up modelling where appropriate, using the manage extension for data traceability, and simplifying geometry with configurations to reduce the load on the PC. You will learn new and improved ways to optimize your product development and collaborative processes.

Speaker(s)

I am a Technical Consultant in the Machine Data and Process Insights team with Autodesk. I have a passion for design and innovation with a background in mechanical engineering design.

Introduction

This class will teach you several modelling and collaboration techniques in Fusion 360 to improve the way you design and manage large assemblies.

These techniques were developed and implemented during “Project Vesuvius” a project where a large-scale 3D printer was designed and created with over three thousand components using several Fusion 360 workspaces and several collaborators.

What Is a “Large Assembly?”

The definition of a large assembly can differ on several factors including:

- Number of Components
- Complexity of Components
- Industry Sector
- Processing Power
- Physical Size

The definition in terms of Fusion 360 can be defined as: “Any assembly where performance is noticeably decreased”

Modelling Methods

The main two modelling methods are known as “Bottom Up and Top Down”

Bottom-up

This method is where individual components are modelled in their own Fusion 360 files and then combined in a separate Assembly file and joints applied to position them.

Strength:

Multiple people can work on each file at the same time speeding up the process of designing.

Weakness:

There are no references to other parts of geometry that may prove useful. E.g. hole positions that go through multiple components.

Top-down

In this method, multiple components are modelled in the same Fusion 360 file. Joins and as build joints are then added to position and fix them.

Strength:

Because all components are created in the same file, we can use geometry from other components through projections to reference position or size of features.

Weakness:

Because all components are in the same file only one person at a time to make changes without overwriting versions

Skeleton modelling

With this method, we can combine the advantages of both bottom-up and top-down modelling. Initially a “skeleton model” is created. This model is comprised of basic sketches, planes, axes and other reference features. These features will map out the key size, position and shape of all the components in the assembly.

This skeleton model is then imported via deriving into separate files for each individual component. It can then be referenced to create the full 3D geometry with details added. If the skeleton model is changed the changes will then proliferate through into each individual component.

Simplifying geometry

When components with high face counts such as gears, bolts, generative design components or other miscellaneous components are loaded and rendered by Fusion 360 a lot of computer power. To remedy this for large assemblies’ simpler versions of these high face count models can be created.

Again, deriving can be used to import the relevant geometry needed to create a simplified version. Derived parameters can also be imported which can include things like extrude distances, sweep distances etc.

Collaboration

A potential problem when working on large assemblies is not technical issues arising from modelling techniques but working efficiently with a large group of collaborators with different skill sets and different locations.

On technique for clearly defining at what stage in the design process a particular part is we can use the Fusion manage extension. This will clearly define to the manufacturer if the design iteration is in a draft stage or final. To do this we need to do the following steps:

- Open the manage extension tab in fusion and assign a unique item number.
- Release with a change order.
- Select a lifecycle on each item.
- You should now change to the web interface for the manage extension.
- You should now fill out all relevant team members who will be working on, reviewing, and finally approving designs.
- From the approval workflow menu, you can move the part from each stage. When it is moved the relevant people will be notified to do their tasks i.e., start work on design changes, reviewing and approve.