Simulation for Fusion 360

Vaclav Prchlik
Autodesk

Learning Objectives

• Understand Simulation for Fusion 360 key concepts
• Learn how to setup a Simulation, Solve and analyze results
• Learn how to analyze design with Linear Static Stress and Modal
• Learn to use Thermal and Thermal Stress analysis

Description

Simulation for Fusion 360 brings powerful simulation tools to every designer and engineer. Now you can make the right design decisions, improve your product functionality, and identify critical areas in your design. In this class you will discover how to setup simulation, create loads and supports, analyze results, and share it on the cloud. You will learn how to analyze your model using Linear Static Stress and Modal analysis.

Your AU Expert

Vaclav Prchlik is a software development manager for Simulation for Fusion 360 3D CAD design app. Vaclav has been working for Autodesk, Inc., for 12 years, leading the team that is implementing easy-to-use structural analysis, drop test, and mechanical engineering generators and calculators. Before joining Autodesk, he was research and development manager in a small software company that focused on knowledge-driven CAD. Vaclav has a master's degree in industrial engineering and management from University of West Bohemia, and he has a PhD in mechanical engineering.
Key Concepts of Simulation for Fusion

Loads

- Loads define forces, moments, etc. applied to your design
- Loads are causing stresses and deformation of your design
- Multiple types of Loads

![Diagram of loads and inputs](image)

Tips and Tricks

You can limit Force area
Constraints

- Constraints (also called Boundary Conditions) limit displacement of your design
- Constraints describe connections of your design to the rest of the world
- Constraints balance Loads
- Multiple types of Constraints

Tips and Tricks

1. Select type of constraint
2. Select Surface
3. Input additional parameters

You can select multiple faces
Contacts

- Contacts (also called Joints) define connection between components
- Contacts describe how components interact with each other
- Automatic Contacts generation
- Multiple types of Contacts

![Image showing different contact types]

**Tips and Tricks**

1. Generate automatic contacts first
2. Select two components and two surfaces
3. Select type of contact

You can specify tolerance of automatic contacts
Results

- Results (also called Post-Processing) allows you to explore calculated stress, strain and deformation and make decision about your design
- In Results you find answers to your question:
  - What is the stress in my design?
  - How my design deforms under Loads?
  - Would it break?
  - Where can I can material to optimize my design?
- Colors are mapped to values using Legend

1. Display results or call Solve
2. Select result type
3. Use Thresholding to look at critical areas inside model
Studies

- Study contains definition of simulation and results
- Studies allows you to do multiple analysis on one design
- Multiple types of Studies:
  - Linear Static Stress
  - Modal Frequencies
  - Thermal (Steady)
  - Thermal Stress

1. Select study type
2. Study is created
Description of Study Types

Linear Static Stress

- Calculate displacements, stresses, strains and reaction forces caused by applied loads
- Assumptions:
  - Small deformations (deflections & rotations)
  - Linear material behavior
- Most commonly used simulation ~80%

Modal (Natural Frequencies)

- Modal Analysis finds natural frequencies for your design
- These are frequencies that should be avoided
Thermal (Steady)

- Analyze temperature distribution in your design caused by heat input and output
- Stabilized heat flux

Thermal Stress

- Two steps analysis
  - Thermal analysis. Heat is causing deformation. If constrained it causes stress.
  - It is combined with Static Stress analysis
**Simple Simulation Step-by-Step**

Following example demonstrates basic workflow for defining simulation, solve and review results.

**Step 1 – Switch to Simulation workspace**

Fusion 360 introduces a new workspace dedicated to Simulations. First step in analyzing your design is to switch to Simulation workspace.

**Step 2 – Create a Study**

Next step is to create a Study. Linear Static Stress is a commonly used Study Type.
**Step 3 – Apply Material**

Each Study can override materials from the original design. You can test multiple variants of material. Material dialog also offers options for Safety Factor for each component.

![Apply Material Diagram]

**Step 4 – Add Constraint**

Fixed Constraint is the easiest way to define how your design is connected with the rest of the technical system that is not part of your Fusion model.

![Add Constraint Diagram]
**Step 5 – Add Load**

Force is typical way to define how your design is loaded.

**Step 6 – Solve and Analyze Results**

Single click to Solve button calculates and shows the Results. Legend maps colors to the numerical values. Safety Factor is a good place to start analyzing feasibility of your design.