AUTODESK UNIVERSITY

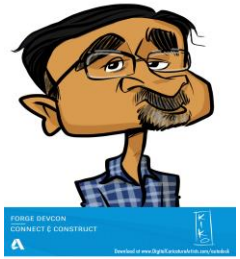
IM500020:Become a Fusion Simulation expert in 60 minutes

Shekar Sub, Hugh Henderson

© 2021 Autodesk, Inc.

Introduction

Shekar Sub



- Working on Fusion Sim & Generative
- 24 years @ Autodesk
- AU speaker
- Bachelors Masters Doctorate in Mech Engg
- Co-author of “Mastering Autodesk Inventor….”
- Volunteer for FIRST robotics
- Walking, Yoga and Tennis

Hugh Henderson

- Quality Assurance Engineer
- 19 years @ Autodesk
- Fusion Simulation (past Inventor Sim)
- Fixture Design Engineer – Industry Exp.
- BSME, Univ. of Illinois at Urbana-Champaign ('95-'98) Thermo, FEA, Simulation focus

Key Learnings

- Learn how to set up Fusion Simulation analysis with loads and constraints
- Learn how to interpret simulation results
- Gain tips and tricks
- Learn how to avoid pitfalls when using Fusion Simulation

Contents

- Introduction
- Simplification
- Studies
- Materials
- Constraints
- Loads
- Contacts
- Meshing
- Pre-check/Solve
- Results

Why Simulation?

- Samsung Note 7 phone fire!!!
- Recalled 1 million of 2.5 million phones manufactured
- Lost ~$5 billion

Why Simulation?

Fusion Simulation

- Integrated with CAD, CAM
- Easy to use
- Local & Cloud solve
- Meshing (Tetrahedral)
- Industry acclaimed Nastran, Explicit Solvers, Moldflow, CFD solvers. Send to Ansys
- Multi-threaded
- Multi-platform (Windows, MAC)

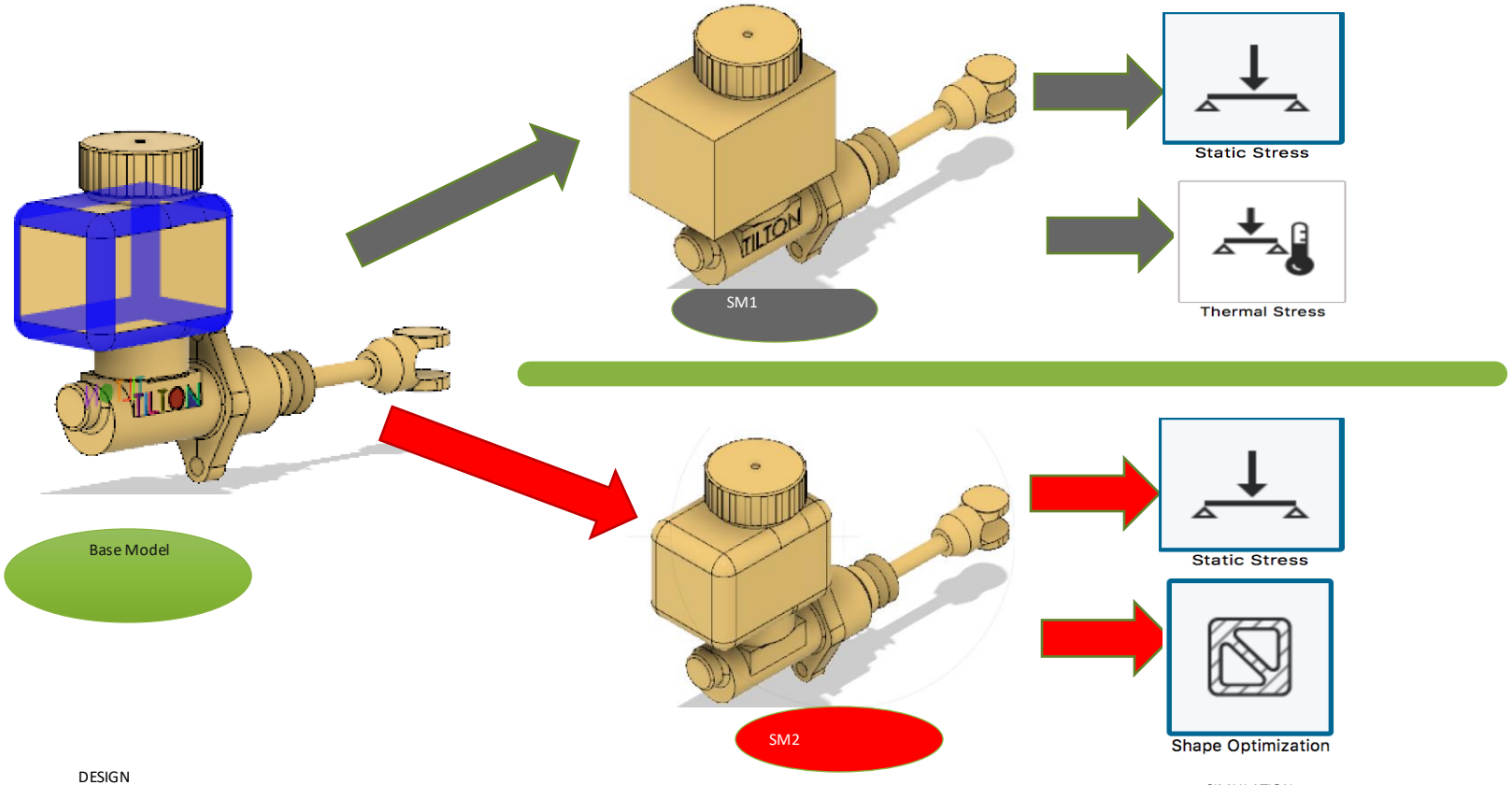
Simulation Steps

Simplification

Simplify workspace

- Create multiple variants of the design workspace model ("What-if" scenarios)
- Simplify the simulation models
- Remove unneeded geometry
- Advantages
  - Reduced element count
  - Faster simulation solves (~6X)

Simulation Models



What to Remove? What to Use?

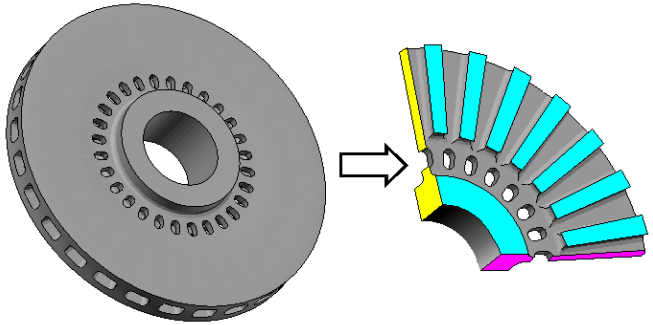
- Small features
  - Fillet/Chamfer faces with no load/constraint
  - No impact on overall stiffness
  - Shouldn't significantly alter mass for frequency analysis

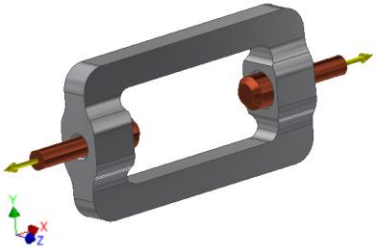
Tip: Keep these internal fillets

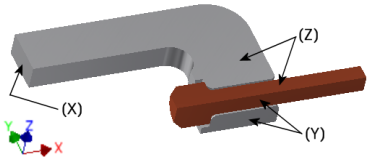
- Screws and bolts. Use connectors
- Lifting eyes or handles, Name plates: Decorative
- Panel switches or indicator lights. Use point masses

Symmetry

- Split body + Remove
- Facilitates making Model statically stable





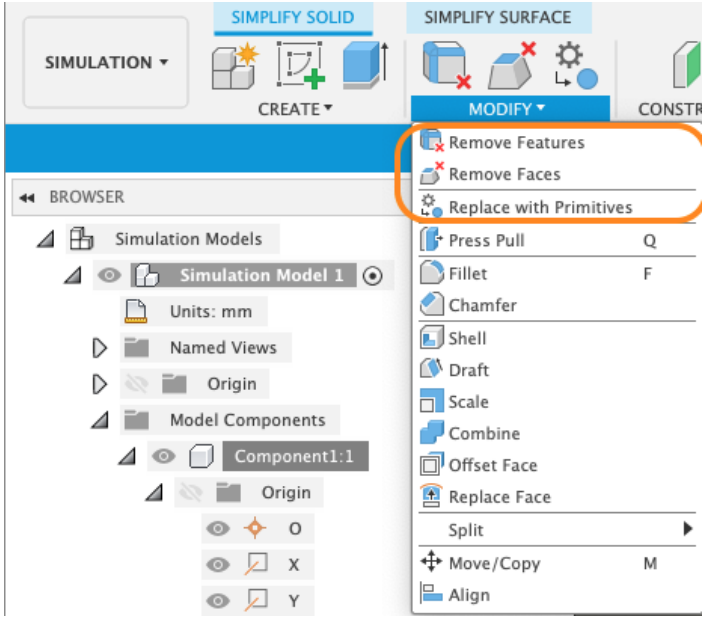


- Tip: Do symmetry changes in Simplify workspace.
- Tip: Avoid modeling with symmetry when performing Modal Frequencies or Structural Buckling simulations. Even symmetrical structures have asymmetrical vibration modes, such as when the structure is twisting.

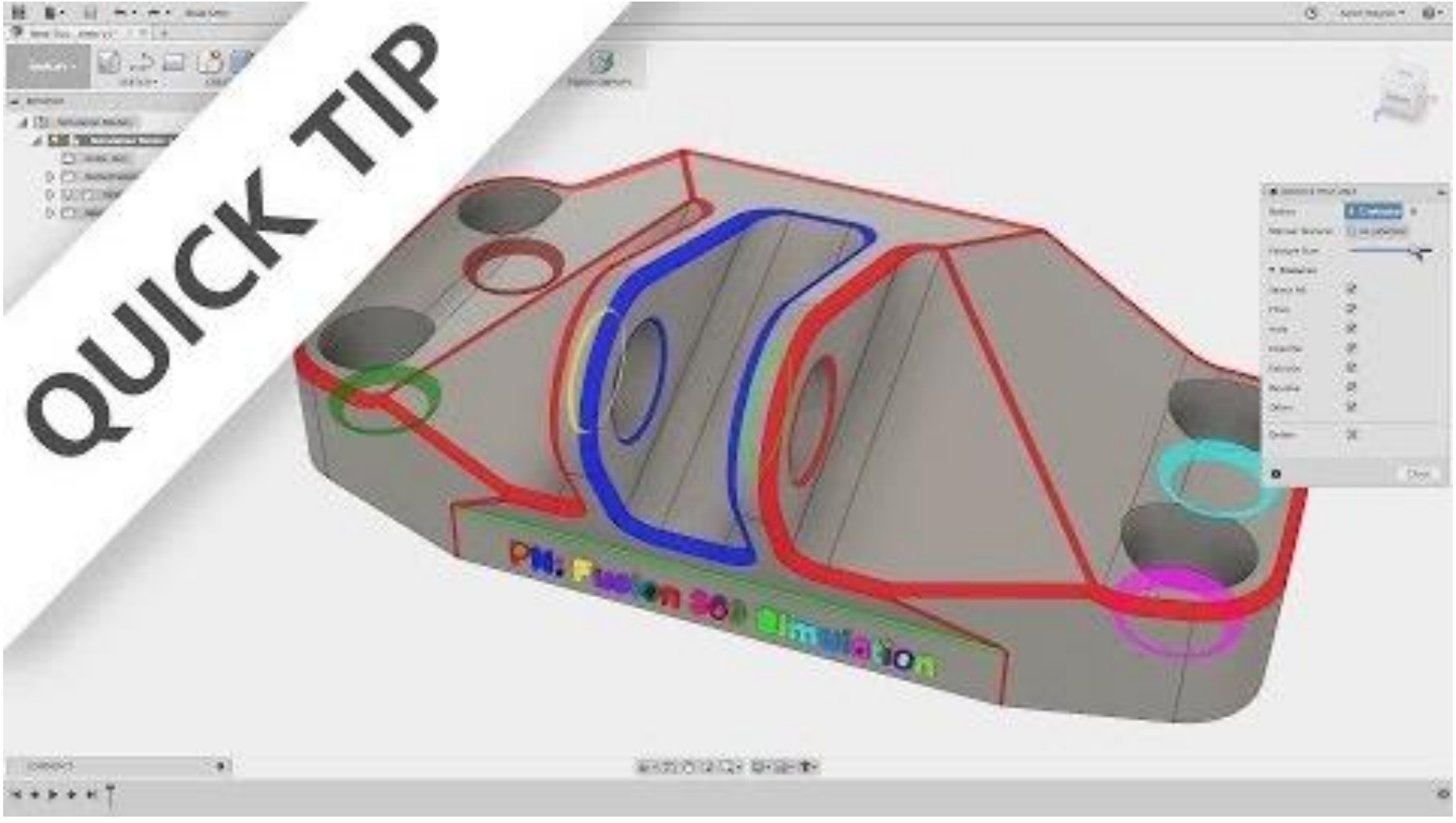
Simplify commands

- What
  - Features: **Remove Features**
  - Faces**: Remove Faces (for fillets)**
  - Bodies/Components:
    - **Remove**
    - **Remove All except Selected**
- Replace with Primitives

  (Box, Sphere, Cylinder)



Tip: "Regularization failed" error message indicates need for simplification.


QUICK TIP

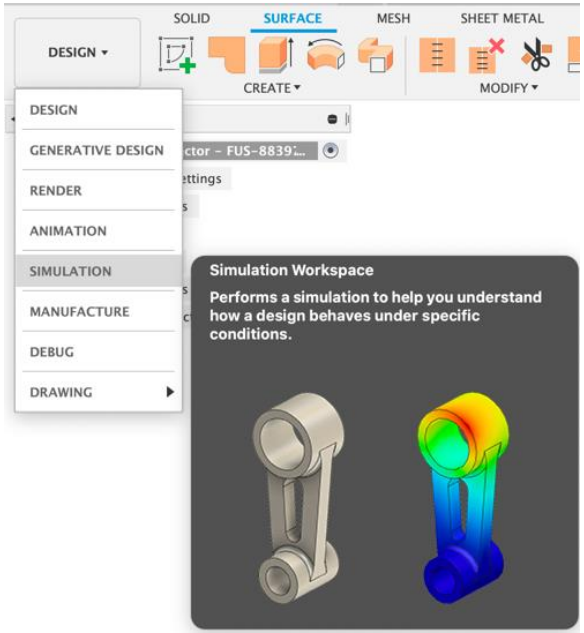
Demo

- Tip: In some fillet over fillet/complex fillet cases it is difficult/impossible to remove the fillet. Add Spheres at the intersections of fillet faces and then use Remove faces command.
- Demo: Simplify tools

Studies

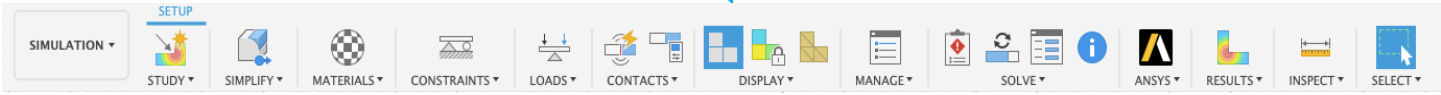
Sim Workspace

- Integrated
- Fully associative copy of Design and Simulation Model
  - Tip: No geometry creation
- Has Compare workspace
- Similar to Generative Design workspace

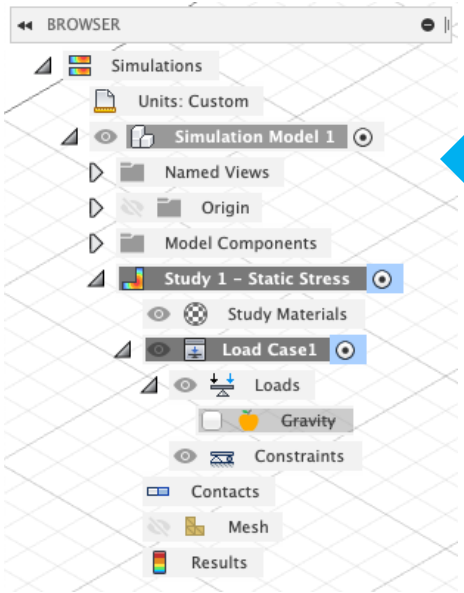


Fusion Simulation UI

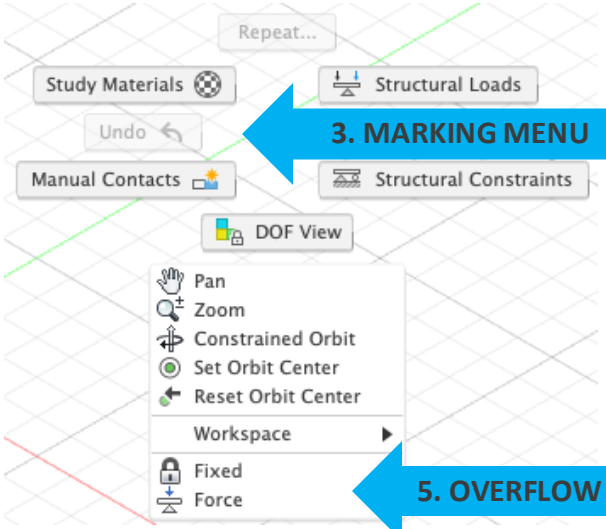
1. QAT



2. TOOLBAR



4. BROWSER

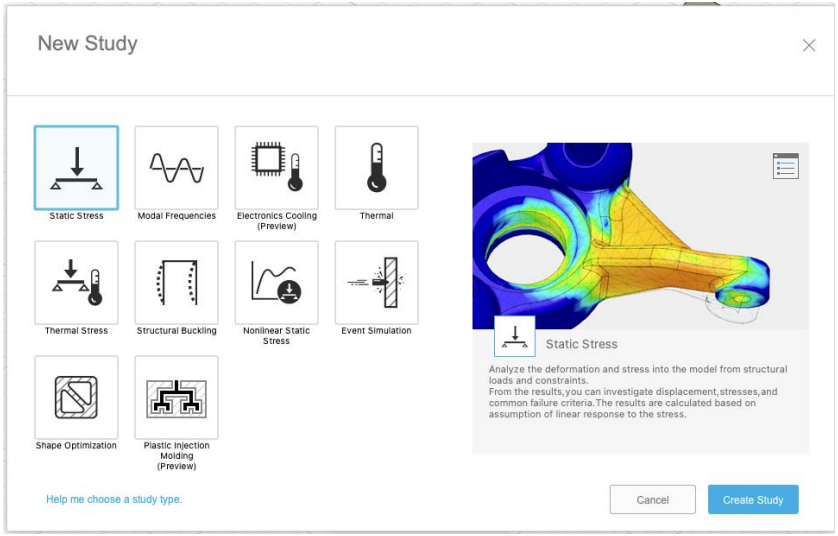


3. MARKING MENU

5. OVERFLOW MENU

Study types

- Ten study types
- Help with “Choose a study type” available
- Tip: Clone & then Edit type. Most studies are interchangeable
- Clone, Delete, Properties available
- Public Preview: Injection Molding & Ecooling



Main Study types

Static stress

Model response to L&C

Small displacement, Linear response

Local/Cloud

Modal

Frequency Values (Hz)

Normalized displacement

Thermal

Temp distribution

Heat flow

Tip: 1 thermal load is must

Local/Cloud

Shape

Lightweighting

Stress, displacement objectives

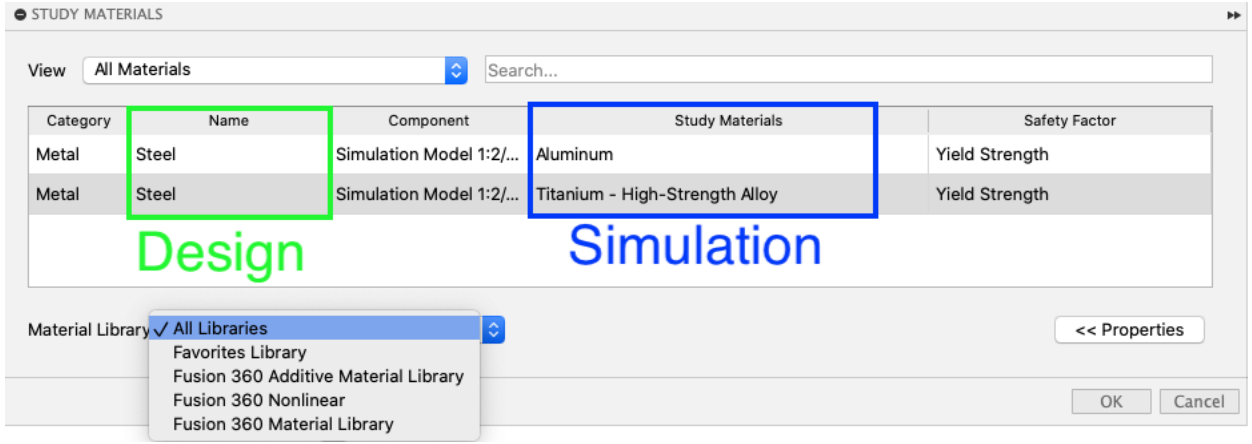
Cloud only

Tip: Use fine mesh size

Materials

Physical Materials

- Assign to bodies in Simulation workspace

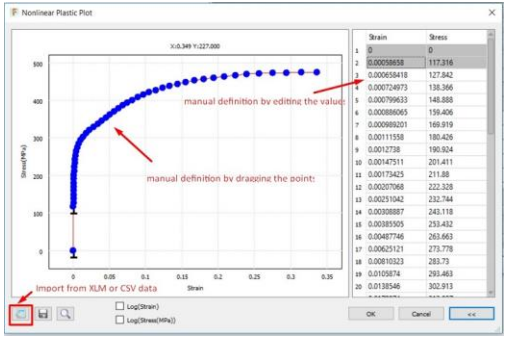


- Material cannot be used for solve
- Value missing or not allowable
- Linear material for Non-Linear solve

Physical Materials

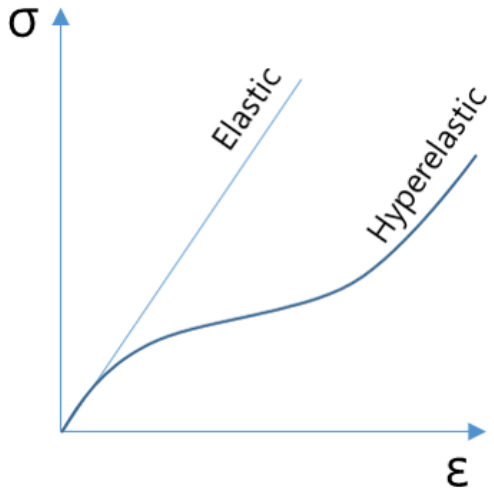
- Isotropic material properties
- Custom materials (Example:Steel material A516M Grade 65**)**
- Orthotropic (Wood)
- Visco-Elastic (Foam, Shoe insoles)
- Tip: RMB on a material in the browser to access the Study Materials command, all components that use the same material are automatically preselected

Advanced Materials



- You can define a stress-strain curve for a non-linear material.
- Import XML or a CSV file using Import from Curve command

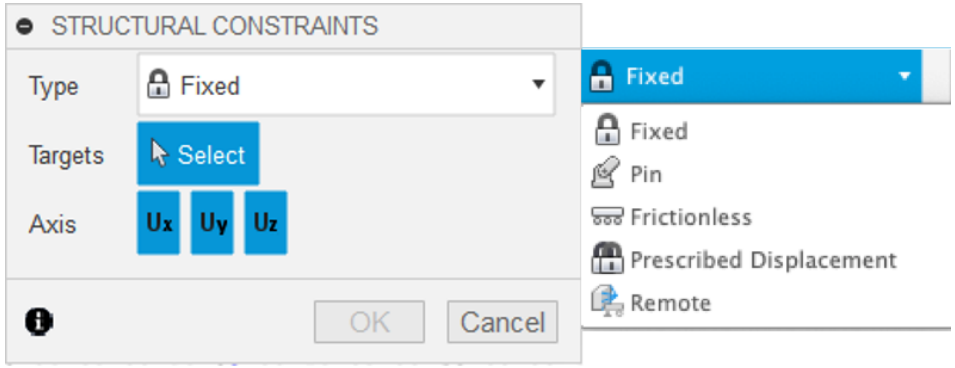
- Tip: Ctrl to add rows in Transient Properties plot dialog. Shift to select a bunch of rows
- Non-Linear
  - Plastic
  - Elastic
  - Elasto-Plastic
- Hyperelastic: High flexibility under high loads . Car-tires and fluid seals
- Temperature-dependent
- Additive



Constraints

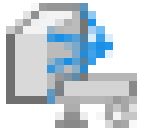
Constraints

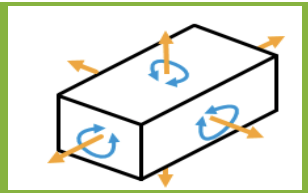
- Goal: Limit translational, rotational motion
- Need at least a few
- Entities: Faces, Edges, Vertices



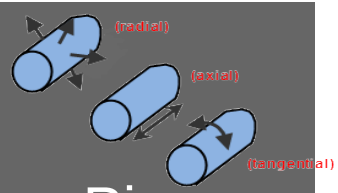
- Tip: In some situations partially constrain the model and use the **Remove rigid body** modes option(RMB, Study Settings). Solver will apply an acceleration load to keep model statically stable.

Constraint types

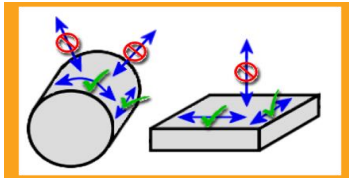




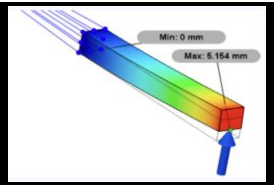
Fixed



Pin



Frictionlesss



Remote

DOF Ux Uy Uz

Unselect to unfix

DOF Radial Axial Tangential

Cylindrical faces

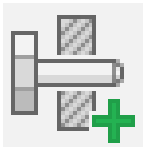
No movement normal to surface

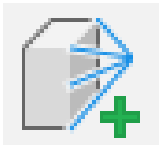
Remote location

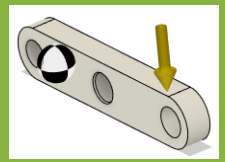
Ux Uy Uz

Rx Ry Rz

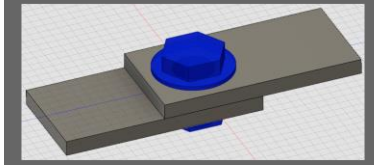
Idealization types



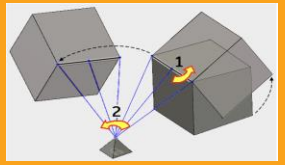




Point Mass



Bolt



Rigid Body

Not available

Auto or Manual

Threaded

With Nut

No movement normal to surface

Spring Connector

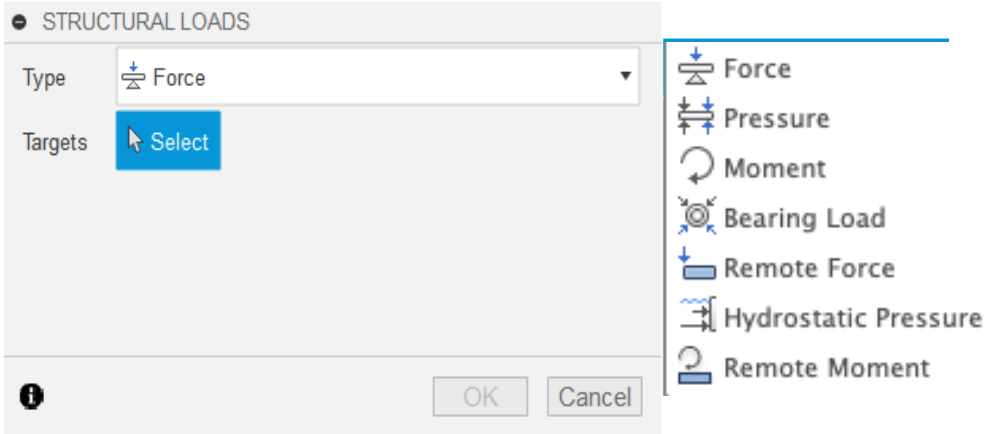
Shell

Beam

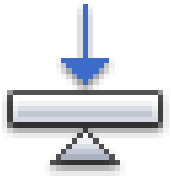
Loads

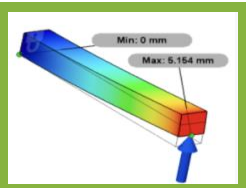
Loads

- Forces applied during operation
- Causes stresses & displacements
- Goal: Specify load magnitude and type accurately



Main Load types





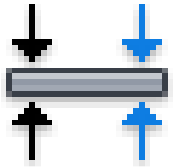
Force

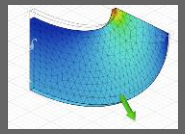
Normal/Any direction

Limit target

Force per entity

Multiple entities



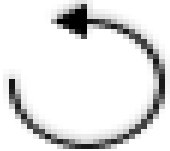


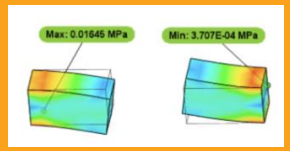
Pressure

Uniformly distributed

Always Normal

Multiple entities



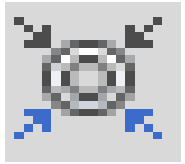


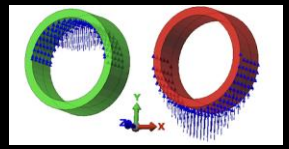
Moment

Centroid of faces

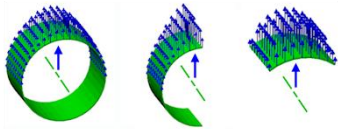
Axis passes centroid

Multiple entities



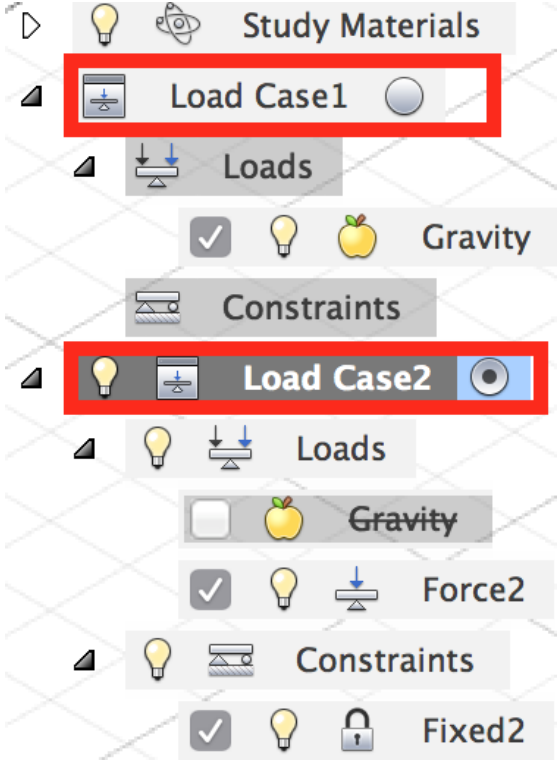


Bearing

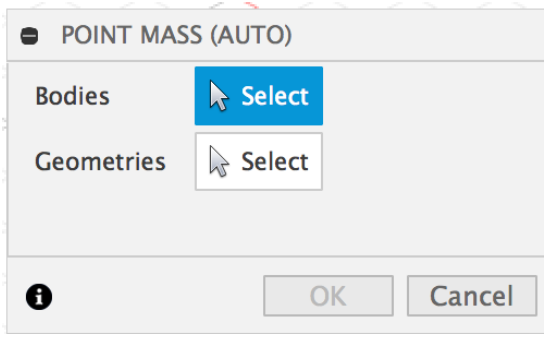


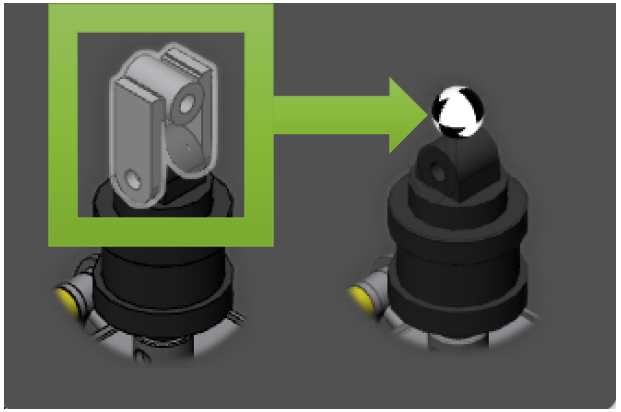
Load Cases

- Load Case 1: Weight of rider
- Load Case 2: Accelerating/Braking
- Load Case 3: Cornering forces
- Not unique to a load case
  - Removed components
  - Materials
  - Contacts
  - Mesh settings
  - Local mesh control
  - Tip: Double-click activates a load case. Available in LSS & Shape-Opt. View Results by LC.



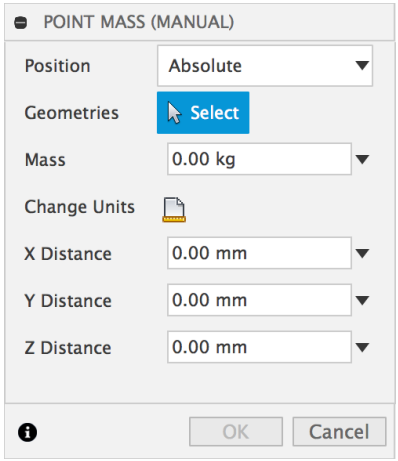
Point Masses - Auto

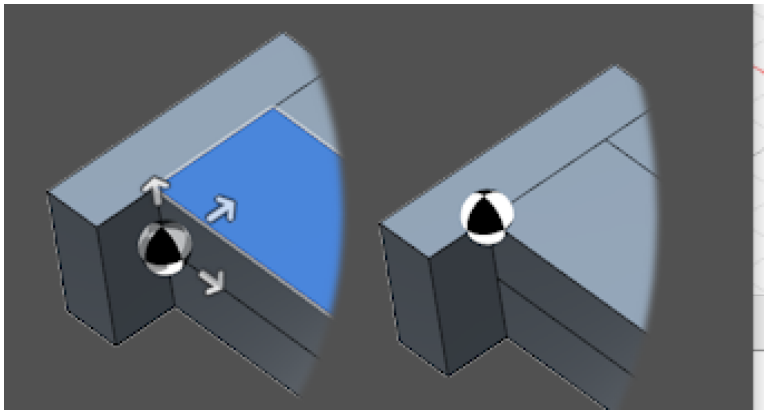




- Idealize geometry
- Reduces file size, element count and processing time
- Selected solid bodies will be suppressed

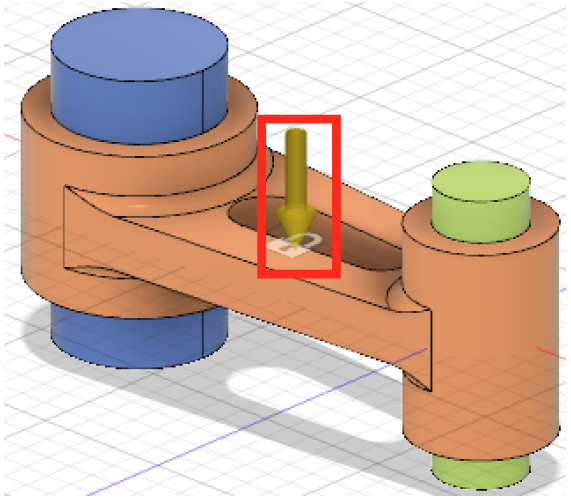
Point Masses - Manual





- No geometry needed. Specify point for centroid.
- **Tip:** Which input field corresponds to which offset direction? Drag a manipulator arrow. Then, notice which Distance field has a changing value while you are dragging the arrow.

Gravity



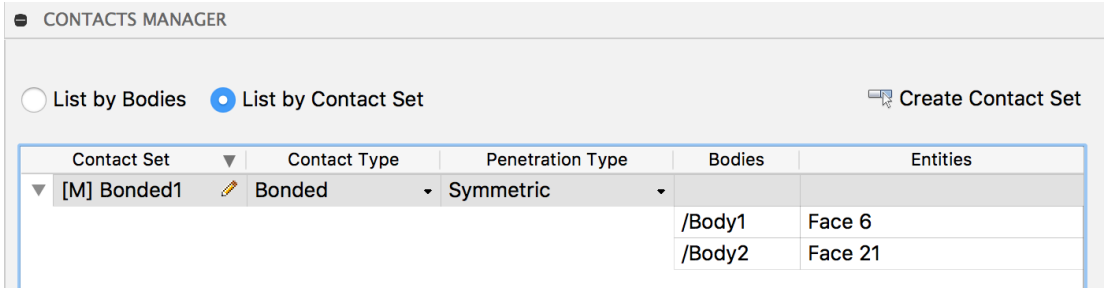
- Global load, affects point masses
- ON/OFF
- Gravity direction
  - Face: Normal
  - Edge: Average vector of normal vectors @ edge
- Different than Forces

  - Tip: When you apply a Hydrostatic Pressure load to any face of the model, the program automatically activates gravity. The direction of gravity controls the direction of increasing pressure for this type of load.

Contacts

Contacts

- Specify how 2 bodies are connected
- Has no relation to joints in assembly



- Use Contacts, Manage Contacts to edit contacts

Contact types

Bonded

Welded

Offset allowed

Separation

No penetration

Partial or full separation

Slide freely

Support friction-coefficient

Sliding

No penetration

No separation

Sliding allowed

Rough

No penetration

Partial or full separation

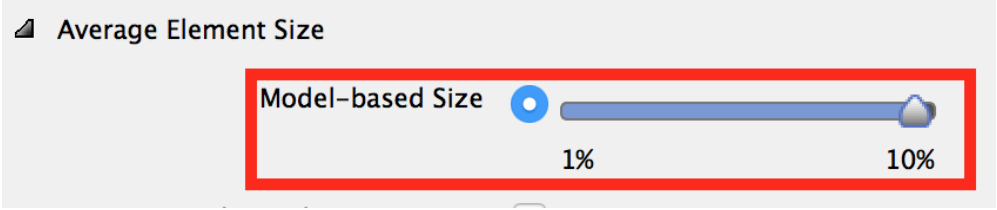
No sliding

Contacts

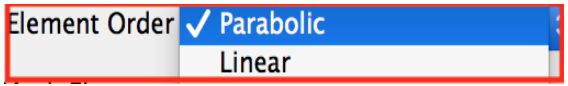
Type	What	Penetration	DOF of 2 entities	Separation	Friction (Mu)	Sliding	Other
Bonded	"welded together".	**No**	**Same**	**No**	**No**	**No**	Treated as single body. Same equal deformation for adjacent nodes
Separation	**Separates and slides**	**No**	**Separate**	**In normal direction**	**Yes**	**Yes, in tangential direction**	**Tip:** Further constraints may be required to modify the DOF's for each body.
Sliding	**No separation between parts**	**No**	**Separate**	**No**	**No**	**Yes, in tangential direction**	
Rough	**Similar to separation but no sliding**	**No**	**Separate**	**No gaps or separations**	**No**	**No**	

Meshing

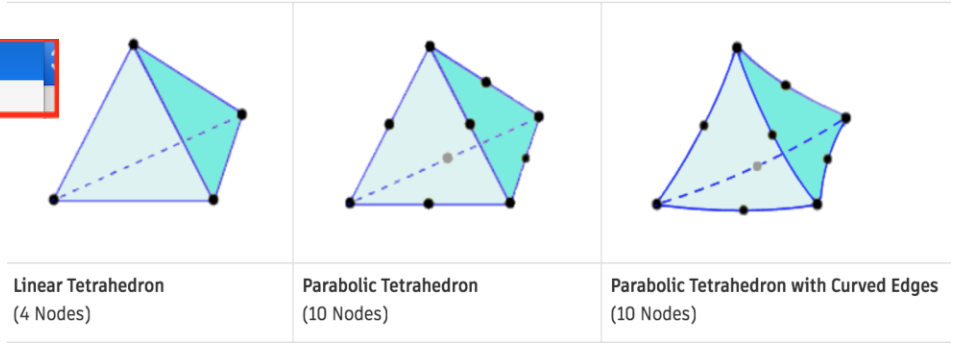
Mesh Settings



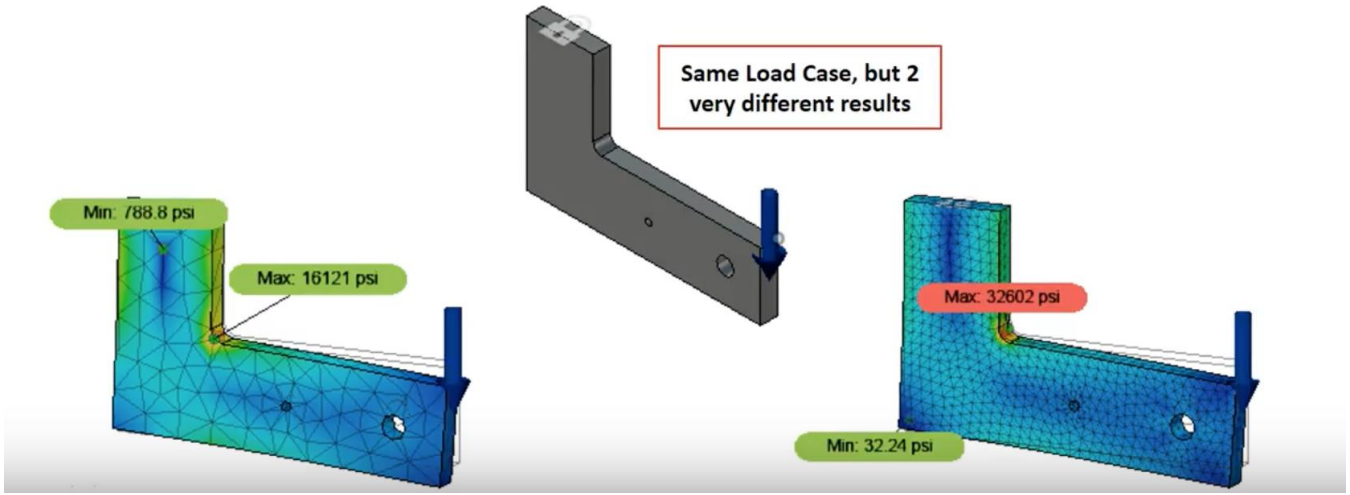
- How large is your mesh element? Tip: For Shape Optimization use a small mesh size to get reasonable results.



- Linear:
- Parabolic:



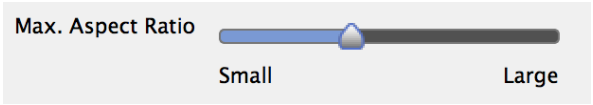
Mesh Settings



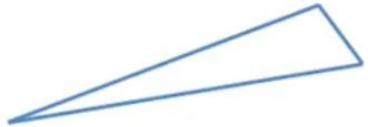
- Tip: Good mesh extremely important for good results.

Mesh Settings

- Aspect ratio



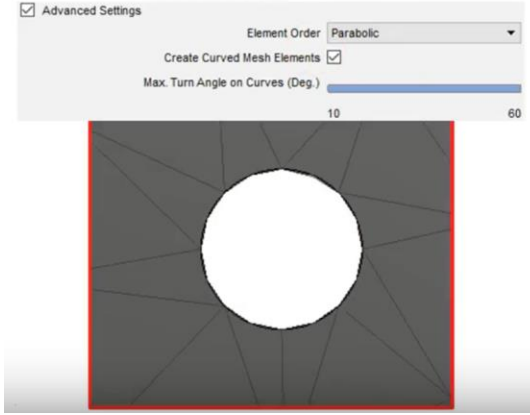
Aspect ratio = 1

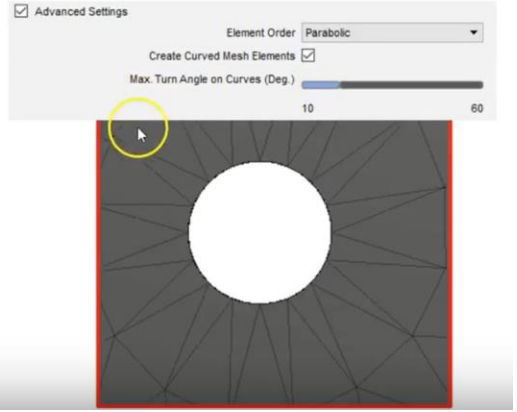


High aspect ratio triangle

- Maximum turn angle
- Tip: Lower the turn

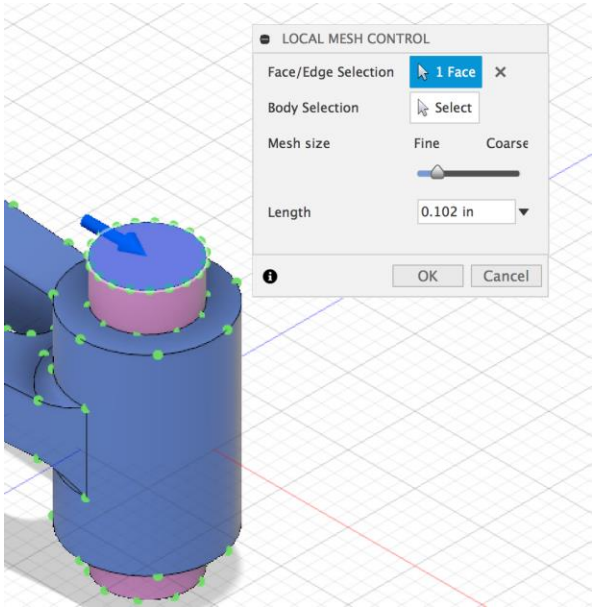
angle smoother the circle





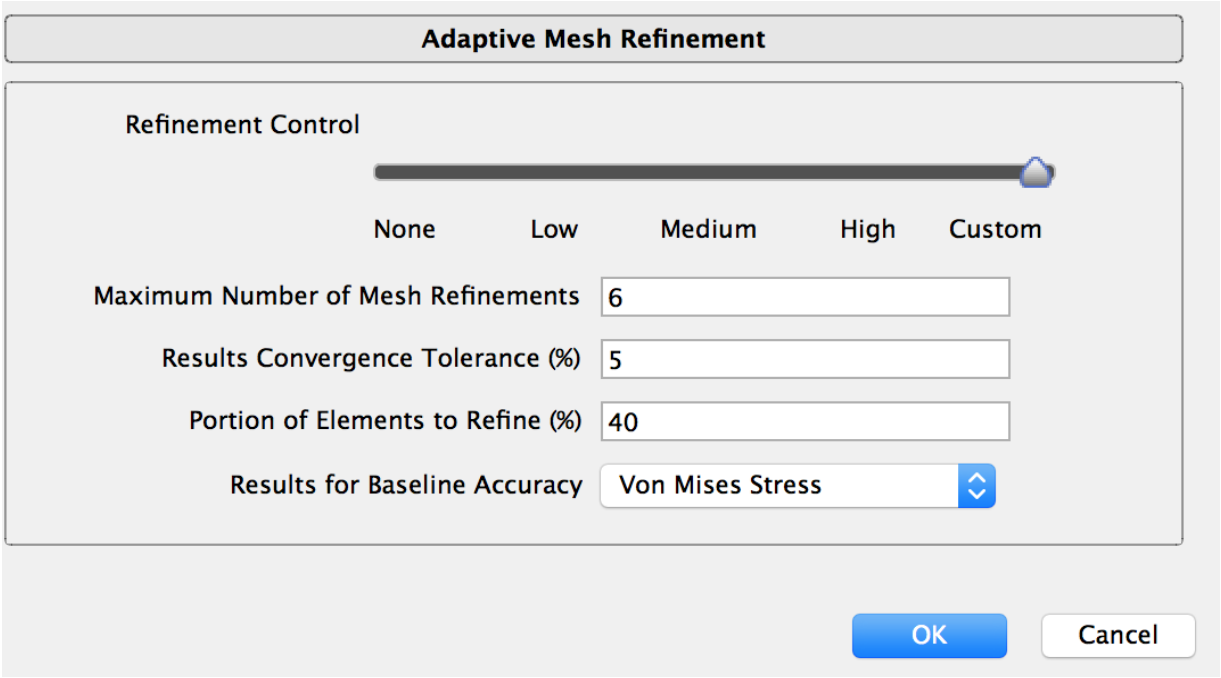
Local Mesh control

- Manual mesh refinement in localized regions
- Faces, edges or bodies
- Finer than overall mesh



Adaptive Mesh refinement

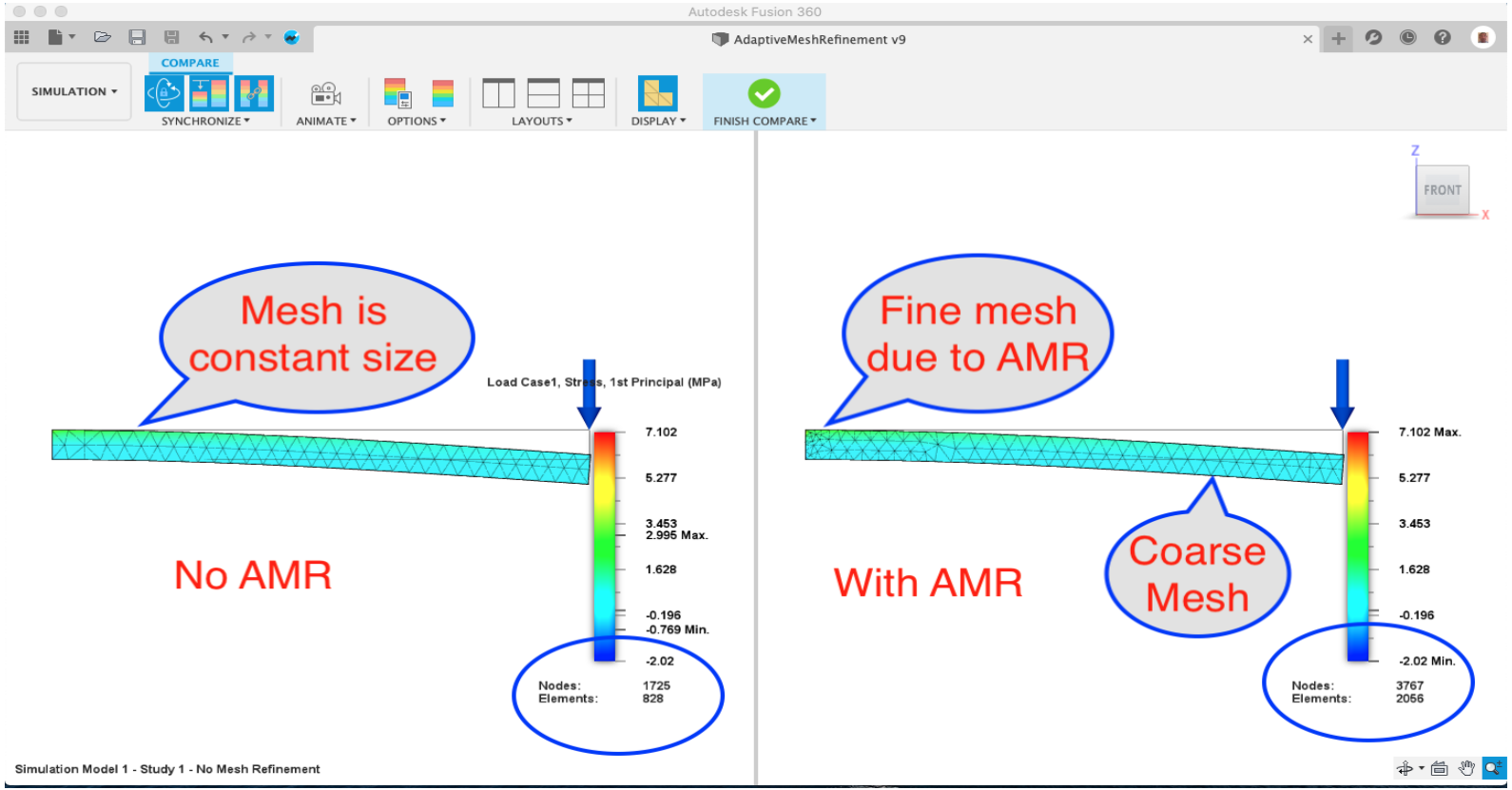
- Static, Modal, Thermal and Thermal Stress

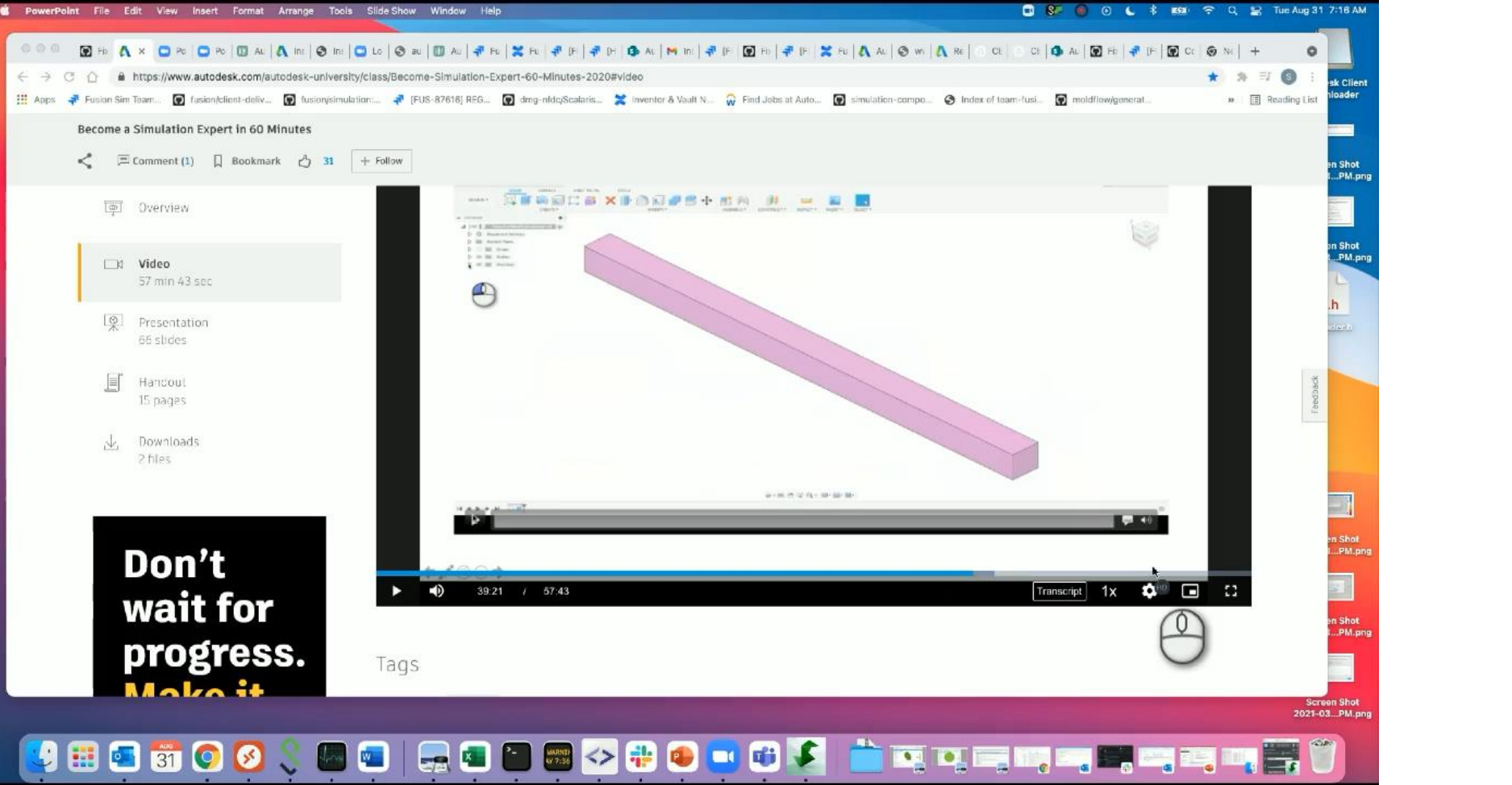


Adaptive Mesh refinement

- Maximum # of mesh refinements Tip: <3 gives good results
- Results Convergence Tolerance:% change between 2 iterations <= tolerance it stops (3%)
- Portion of elements to refine(%)
  - X%: Top X% w.r.t critical result are refined
- Frequency Mode: Modal frequencies only. Basis for refinement

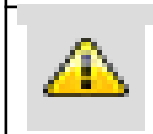
Adaptive mesh refinement demo




PowerPoint File Edit View Insert Format Arrange Tools Slide Show Window Help
Tue Aug 31 7:16 AM
https://www.autodesk.com/autodesk-university/class/Become-Simulation-Expert-60-Minutes-2020#video
Become a Simulation Expert in 60 Minutes
Comment (1)
Bookmark
31
Follow
Overview
Video
57 min 43 sec
Presentation
66 slides
Handout
15 pages
Downloads
2 files
39:21 / 57:43
Transcript
1x
Don't wait for progress.
Tags
Feedback

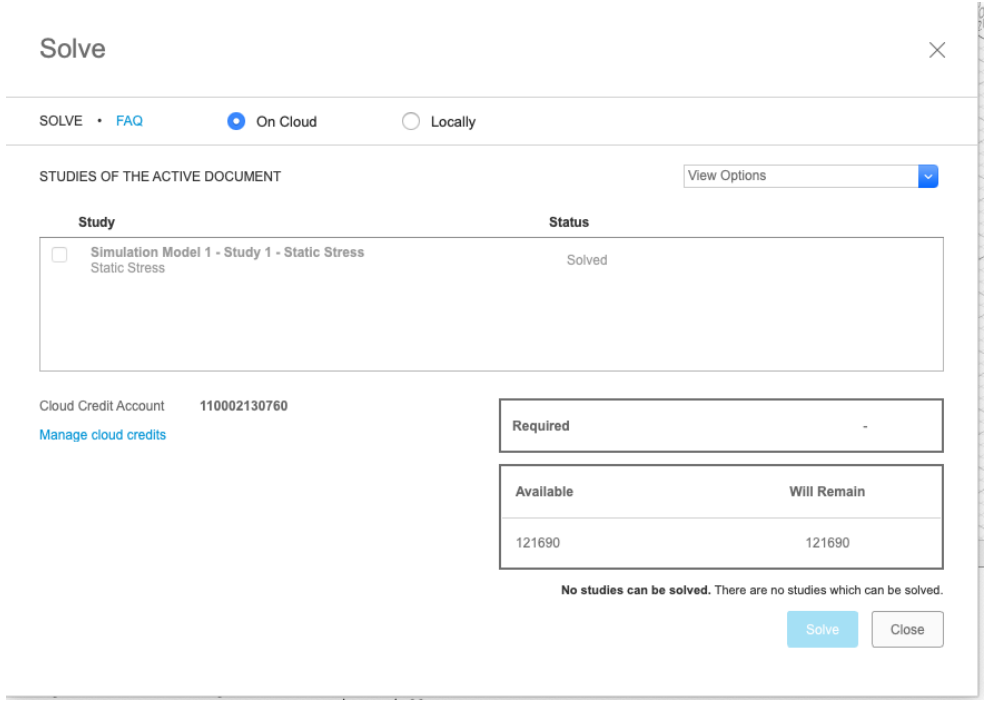
Pre-Check, Solve

Pre-Check

Icon	What it means	Study can be solved?	Examples
	Serious issues, missing inputs.	No	Missing loads, constraints, materials
	Potential issues. Solve may issue warnings	Yes	Unconstrained fully
	All inputs are supplied	Yes	Tip: Desired state

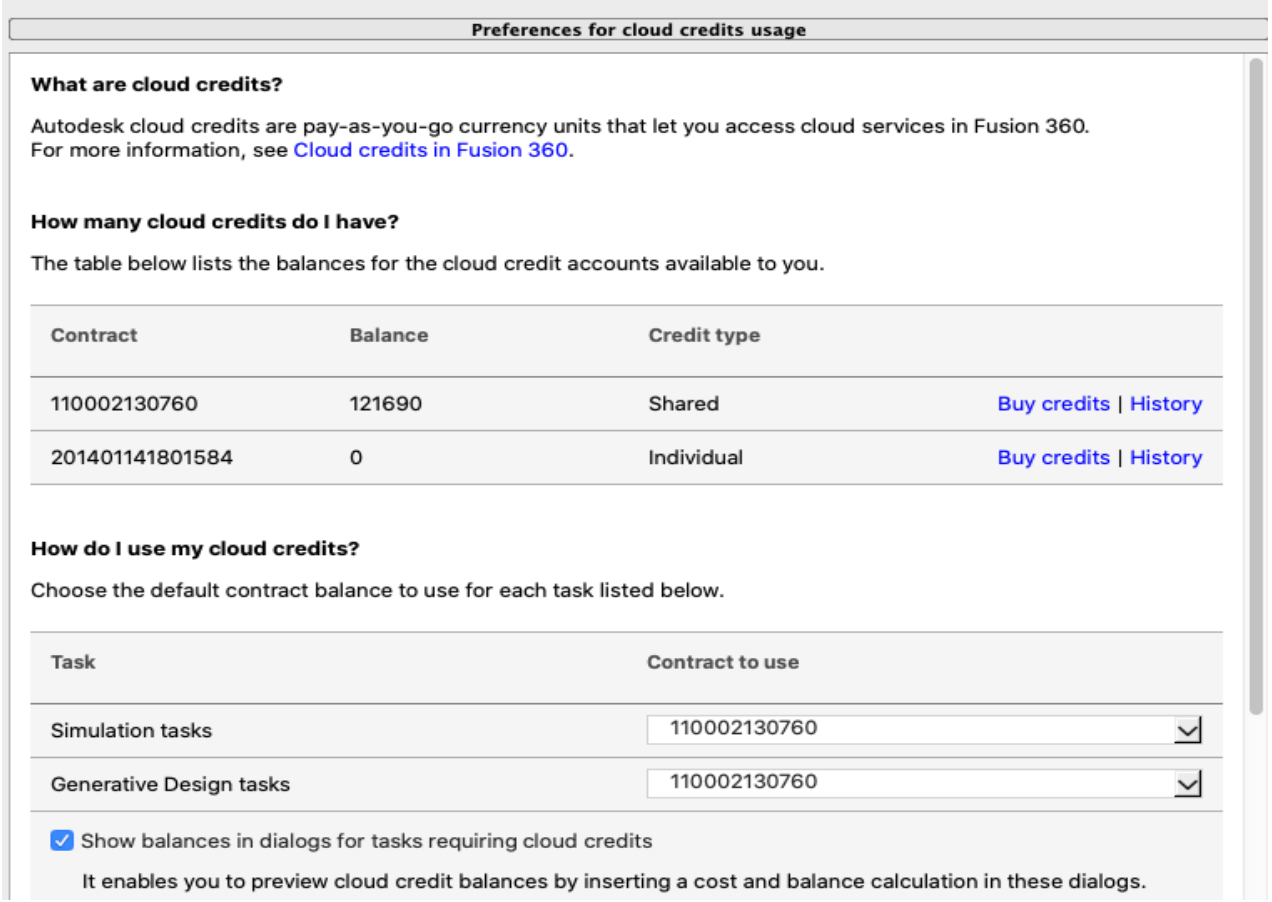
Tip: Error v/s Warning: Missing loads v/s using linear material for non-linear analysis

Solve dialog



- Local only 1 study. Synchronous
- Cloud: Multiple. Asynchronous
- Studies that cannot be solved can be hidden
- Tip: Even though you solve locally, your results are automatically uploaded to cloud after the solve.
- Tip: You can share results with anyone

Cloud Credits

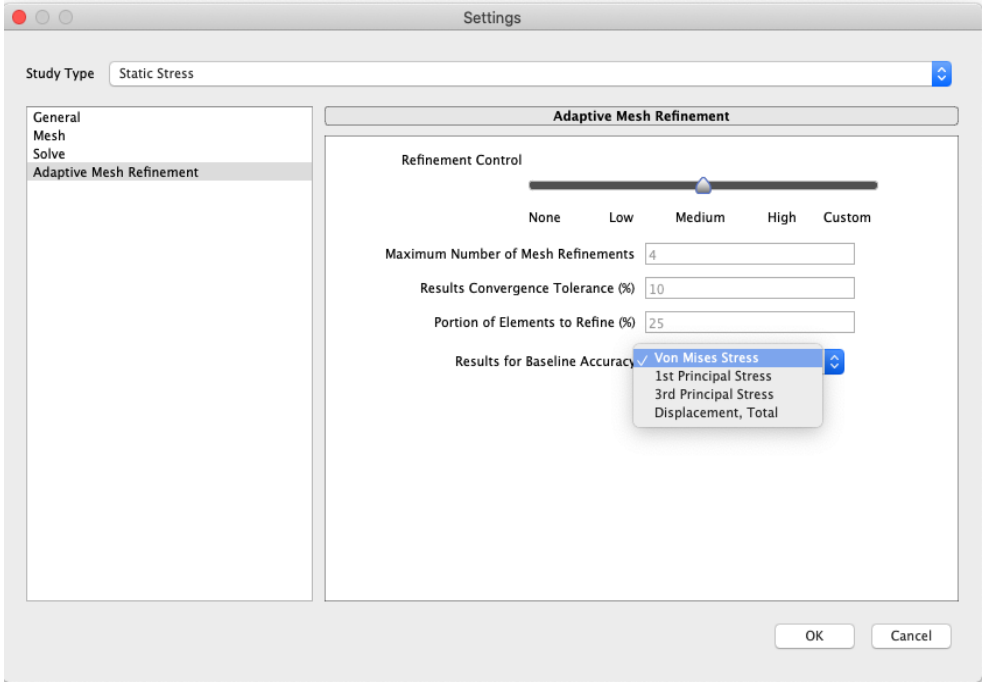


Tip: No CC charged for canceled/failed solves. You can only cancel 1 job at a time.

Results

Results for baseline accuracy

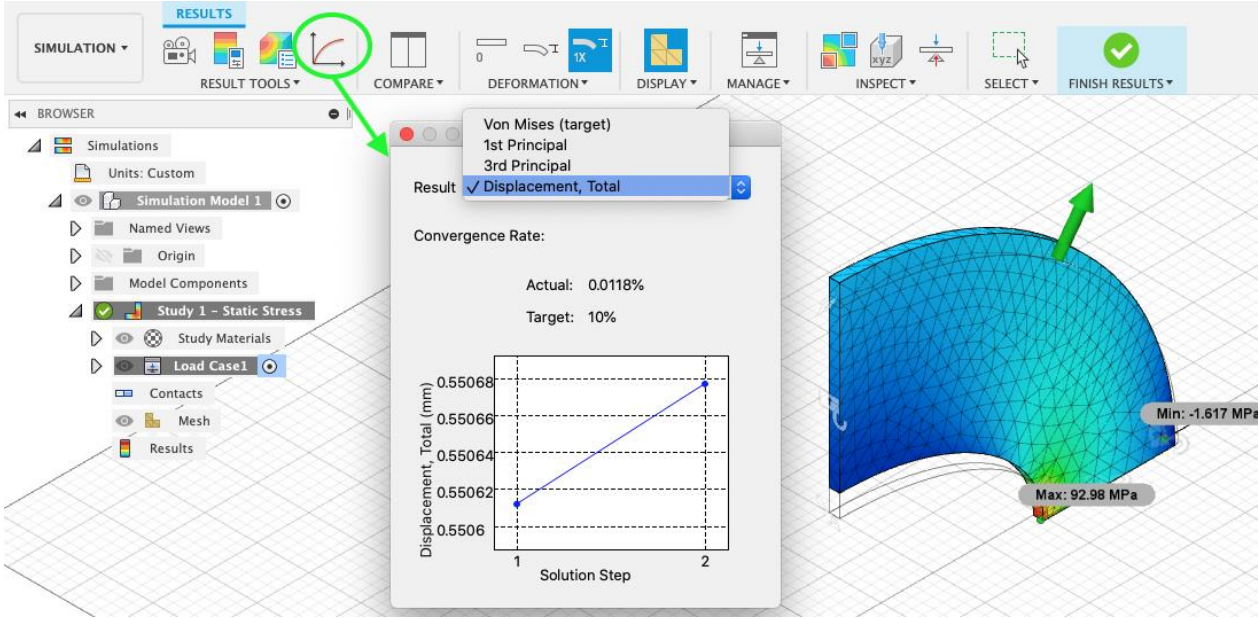
- **Static Stress**
  - Von Mises
  - 1st principal
  - 3rd principal
  - Displacement, total
- **Thermal**
  - Heat flux, temperature
- **Thermal stress**: All previously listed



- Tip: You can specify the desired result on which to base the convergence test regardless of whether you are using a refinement preset or custom settings. Use displacement for faster analysis

Results

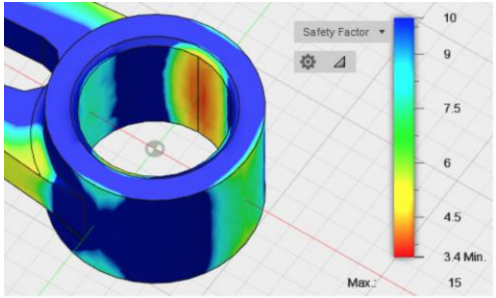
- 3D graphical results
- Legend
- Result type
- Units
- Convergence plot



Result types

Study type	Result type
Linear Static Stress, Non-Linear, Thermal Stress, Explicit	Safety Factor, Stress, Displacement, Reaction Force Reaction Moment, Strain
Thermal, Thermal Stress	Temperature, Heat Flux, Thermal Gradient, Applied Heat Flow
Modal Frequencies	Total Modal Displacement, Modal Displacement X, Modal Displacement Y, Modal Displacement Z (Normalized)
Shape Optimization	Load path criticality, (Promote Mesh)
Structural buckling	Total Displacement, Displacement X, Displacement Y, Displacement Z (Normalized), Critical Load Factor

Results: Safety factor



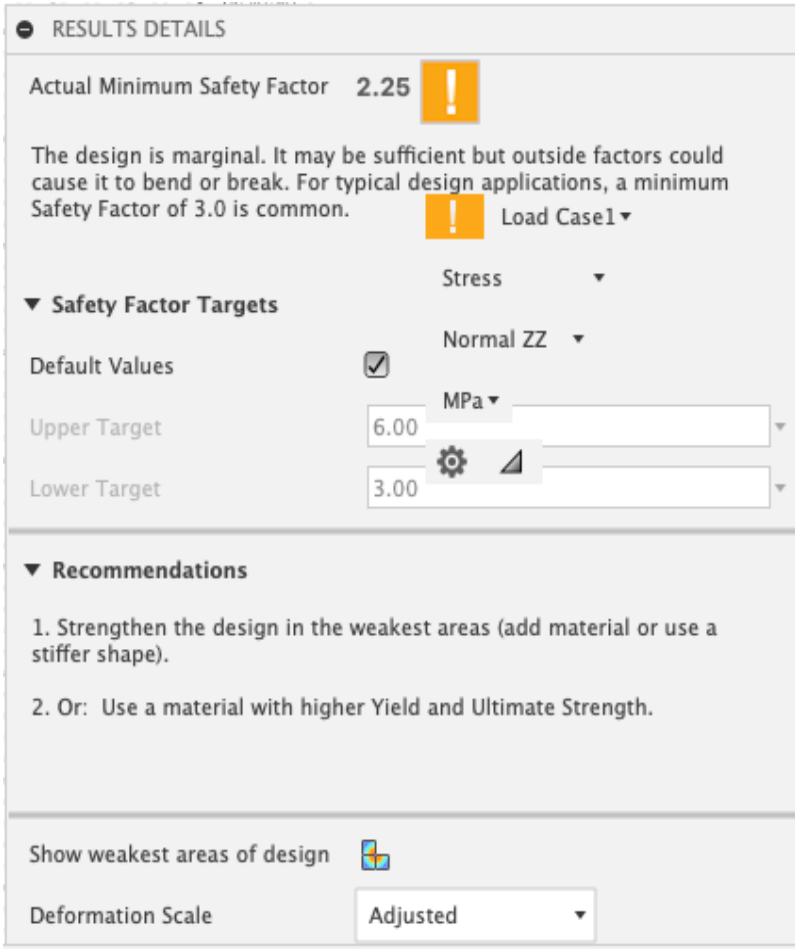
- Safety factor is the default result
- For example NASA® (aerospace industry) and the pressure vessel and crane industries have strict guidelines for design safety factors.
- Tip: A safety factor of <=1.0 means it will fail and not good. For example, an elevator should be designed using higher safety factors than a bracket used to mount a camera.
- Safety Factor = Material Strength / Actual Stress

Results: Contact Pressure

- Tip: Contact Pressure results are generated only where Separation contact is defined between two adjacent parts of a model. Contact pressure results are not computed for any other contact type (such as Bonded, Rough, or Sliding).

Result details

Icon	Indicator	Issue?	Action
	Insufficient	Bends/breaks.	• Material > YS • Reinforce weaker areas • NLSS for bending
	Marginal	Transitional area	• Investigate SF • Mesh convergence
	Sufficient	Good	• Run other studies • Slender->buckling
	Excessive	Over-engineered	• Material < YS • Reduce weight, SO • Switch to GD

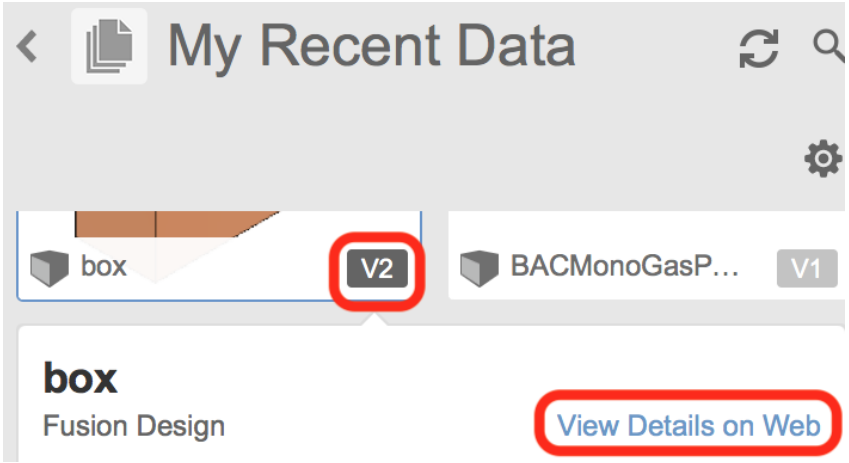


Results sharing - Reports

- Report generates reports with images
- Customizable
- NLSS and Event Sim result charts can be included for steps
- Tip: Use Dynamic Content (Javascript) option which provides collapsible sections

Results sharing – A360

- Generate results and save the file
- Data Panel, click on versions v2
- Click on ”View Details on Web”

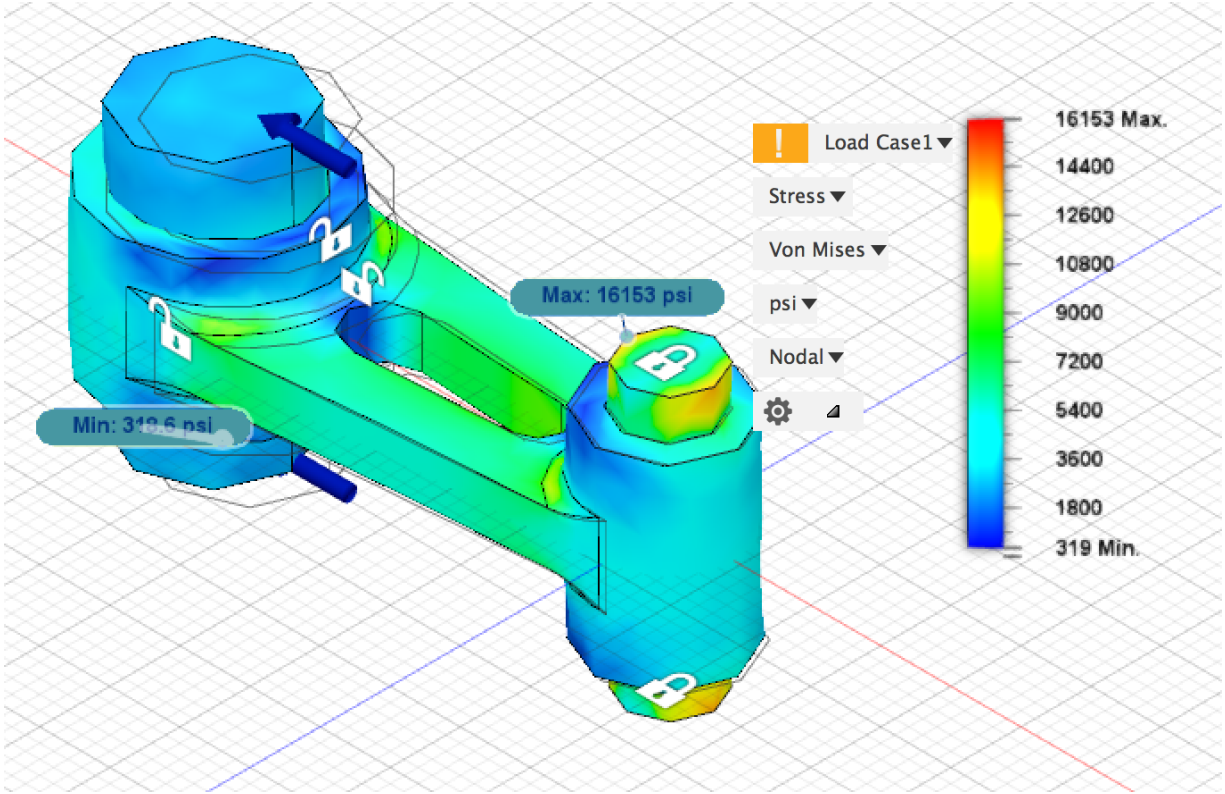


- In Fusion-Team click on the Share icon

Compare workspace

- When
  - Mesh sensitivity results
  - Different load cases
  - Check performance after edit geometry
- What
  - Same study different result type
  - Different studies
- Tip: Available after results generation
- Demo

Demo


Load Case1
Stress
Von Mises
psi
Nodal
Max: 16153 psi
Min: 319.6 psi
16153 Max.
14400
12600
10800
9000
7200
5400
3600
1800
319 Min.

PowerPoint File Edit View Insert Format Arrange Tools Slide Show Window Help

Tue Aug 31 7:25 AM

https://www.autodesk.com/autodesk-university/class/Become-Simulation-Expert-60-Minutes-2020#video

Apps Fusion Sim Team... fusion/client-deliv... fusionsimulation:... [FUS-87618] REG... dmg-nfdcjScalaris... Inventor & Vault N... Find Jobs at Auto... simulation-compo... Index of team-fusi... moldflow/generat... Reading List

Become a Simulation Expert in 60 Minutes

Comment (1) Bookmark 31 + Follow

Search this class

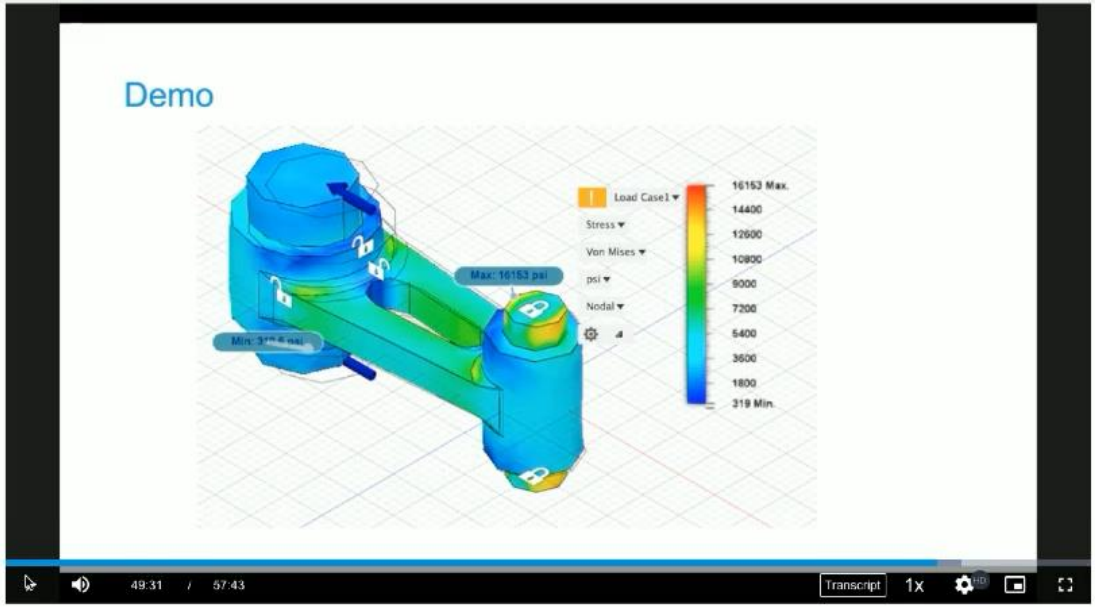
Overview

Video
57 min 43 sec

Presentation
66 slides

Handout
15 pages

Downloads
2 files



Don't wait for

Feedback

Screen Shot 2021-03...PM.png

Links

- About Fusion 360 Simulation Learning: Main learning link
- How to: Specific and general steps for managing studies
- Hands-on exercises: Tutorials

AUTODESK UNIVERSITY

Autodesk and the Autodesk logo are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and/or other countries. All other brand names, product names, or trademarks belong to their respective holders. Autodesk reserves the right to alter product and services offerings, and specifications and pricing at any time without notice, and is not responsible for typographical or graphical errors that may appear in this document.

© 2021 Autodesk. All rights reserved.