IM463460

Become a Simulation expert in 60 minutes

Shekar Sub, Hugh Henderson
Autodesk Inc

Learning Objectives

- 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

Description

Fusion Simulation software offers a rich set of analysis types to simulate real-world problems. Whether it's simple static stress, optimizing a shape to reduce weight, or simulating a bird hitting an airplane, it's all there. One of the biggest challenges is to set up the simulation properly so the results are reasonable. Interpretation of results to selecting the best alternative for manufacturing is another challenge. While demystifying simulation with tips and tricks from community forums, we will also highlight the pitfalls one needs to avoid. Collaboration and knowledge sharing is key to mastering simulation tools.

Speaker(s)

Shekar Sub

- Dev lead for Fusion-Ansys collaboration
- Working on Fusion Sim & Generative
- 23 years @ Autodesk
- Many times @ AU
- Bachelors Masters Doctorate in Mech Engg
- Co-author of "Mastering Inventor…."
- Volunteer for FIRST robotics
- Walking, Yoga and Tennis

Hugh Henderson

- Quality Assurance Engineer
- 18 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

Introduction

A vast amount of knowledge exists on the internet on Fusion Simulation. Youtube has videos about Fusion Simulation that are very helpful. Instead of rewriting a whole new handout we are providing links about Fusion Simulation.

Figure 1: Simulation Steps

About Fusion360 Simulation: Learn as to why you need to do simulation and the value behind it.

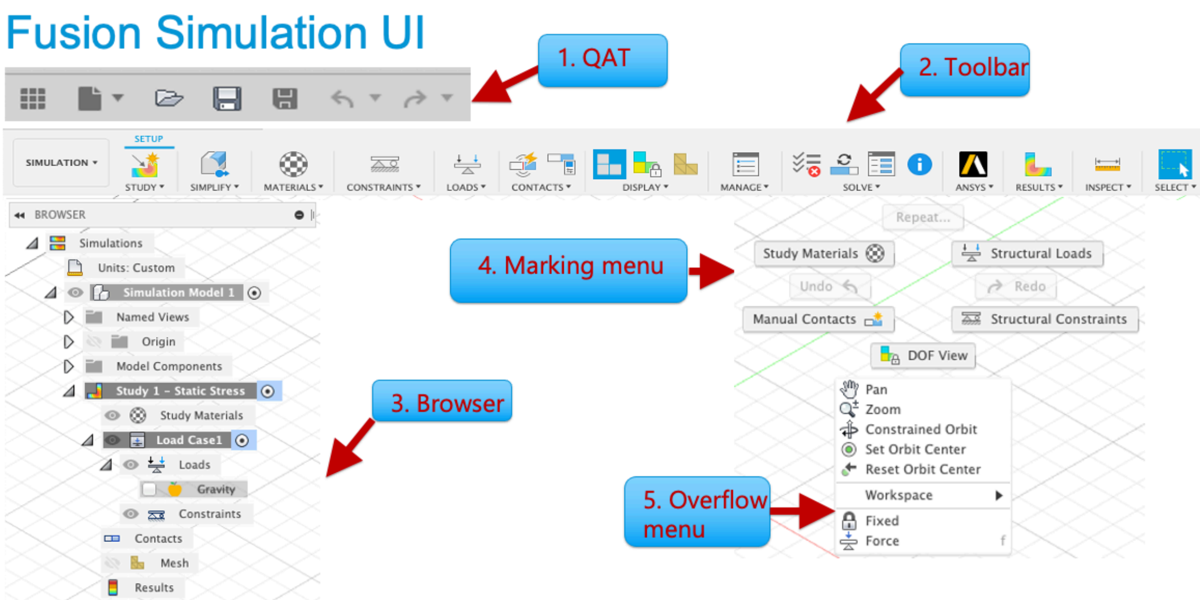

Figure 2: Fusion Simulation User Interface

The Simulation toolbar is a good starting place to familiarize yourself with all the commands needed for Simulation analysis.

1. Simplification

Remove any unneeded geometry for your simulation. During this phase, strategize and plan to figure out what geometry needs to remain in the model for simulation.

- Unneeded Fillets
- Embossed Text
- Actual threads
- Leverage symmetry

- Consider body/components that could be approximated by point masses

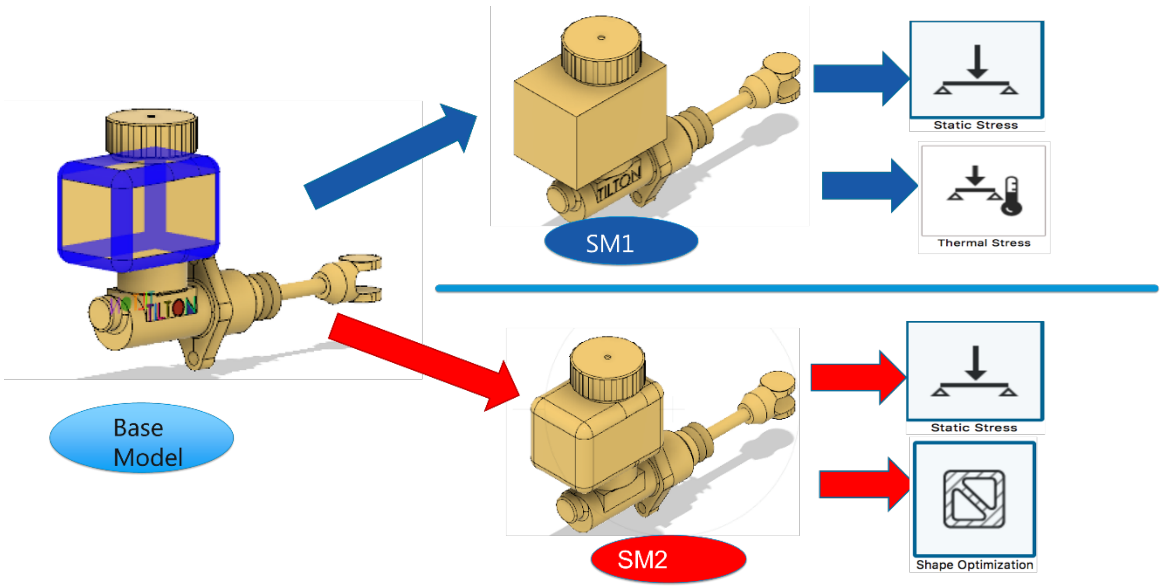

Figure 3: Base Model, Simulation models and studies

[Demo Video]

2. Studies

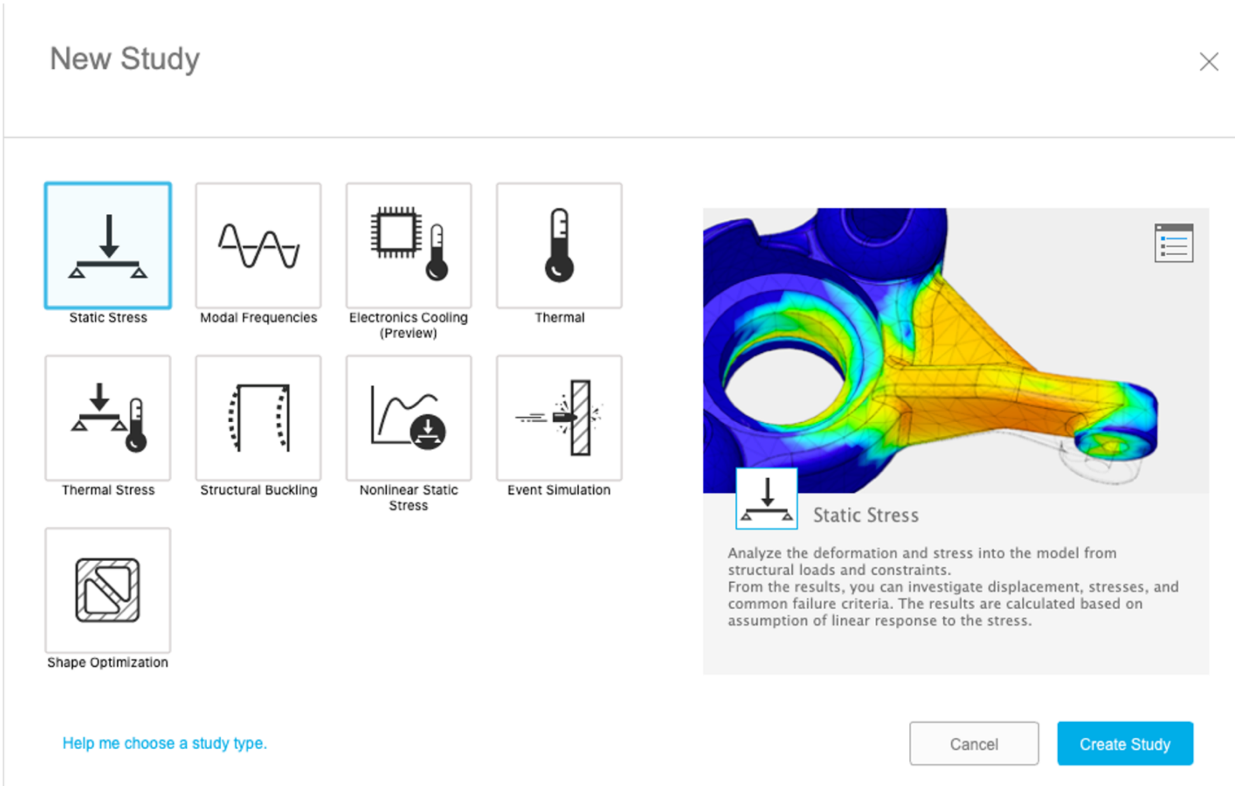

Figure 4: Simulation studies

There are nine different studies that you can select from to do your analysis. This provides step-by-step procedure to setup an LSS analysis. Tip: Create & then Edit

- 2.1 How to create a study?
- 2.2 [Static Stress]
- 2.3 [Modal frequencies]
- 2.4 [Structural Buckling]
- 2.5 [Thermal]:
- 2.6 [Thermal Stress]
- 2.7 [Shape Optimization]
- 2.8 [Non Linear Static Stress]
- 2.9 [Event Simulation]

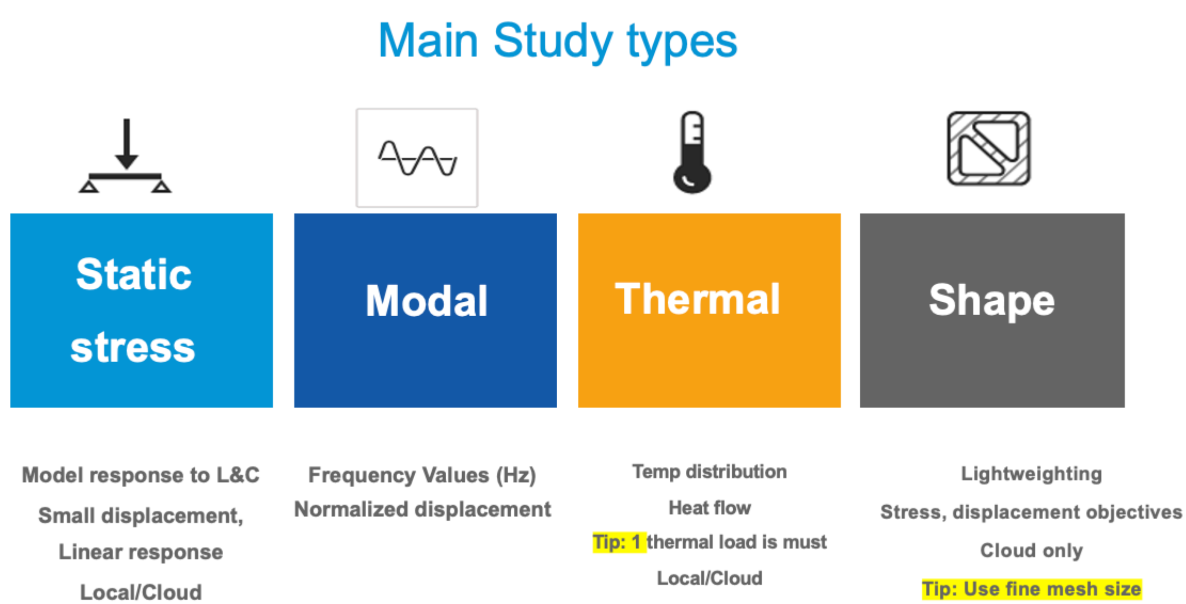

Figure 5: Main study types

3. Materials

Simulation Materials may be different than Model materials. You can create your own custom material.
Tip: Ctrl to add rows in Study Materials dialog. Shift to select a bunch of rows. RMB on a material in the browser to access the **Study Materials** command, all components that use the same material are automatically preselected

4. Constraints

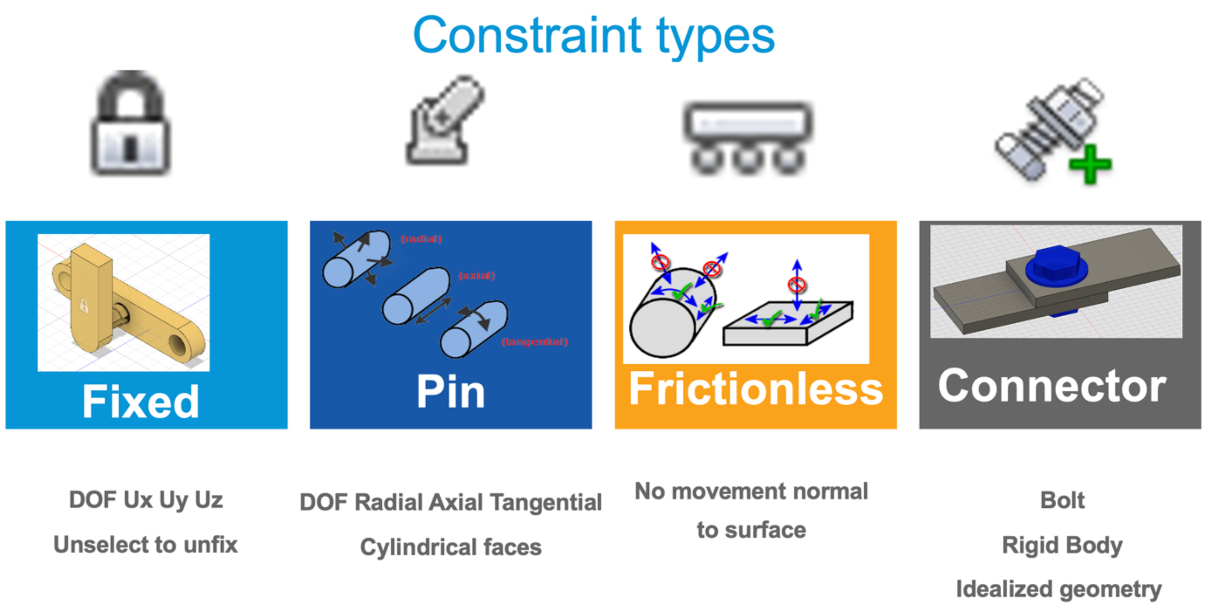

Figure 6: Constraint types

To restrict the model to a particular location apply Constraints. Apply any of the four different constraints. Tip: In some situations, partially constrain the model and use the **Remove rigid body** modes option. Solver will apply an acceleration load to keep model statically stable.

5. Loads

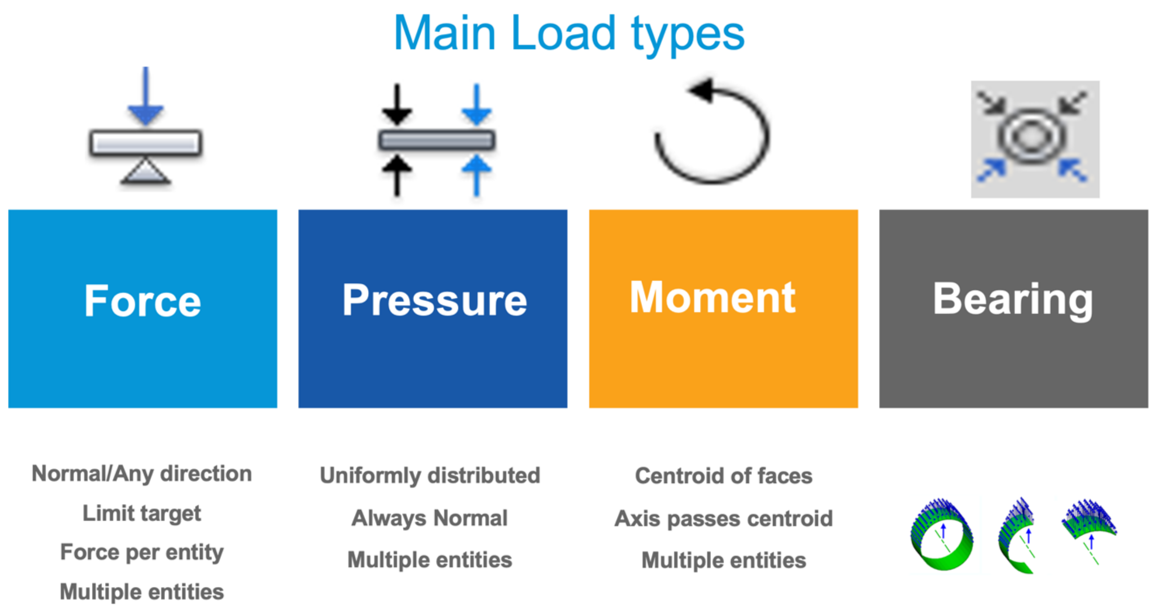

Figure 7: Main Load types

How much load does the model need to resist? Apply any of the six different Structural Loads needed for your simulation.

- 5.1 **Load Cases** Study different load cases and evaluate how the model performs. Tip: Double-click activates a load case. Cannot have 0 LCs
- 5.2 **How to assign a point mass**: Substitute geometry with a point-mass or create a point mass to idealize non-created geometry. 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.
- 5.3 **How to assign global loads** **(Linear, Angular):** Learn how to apply linear acceleration, angular velocity, and angular acceleration loads.

6. Contacts

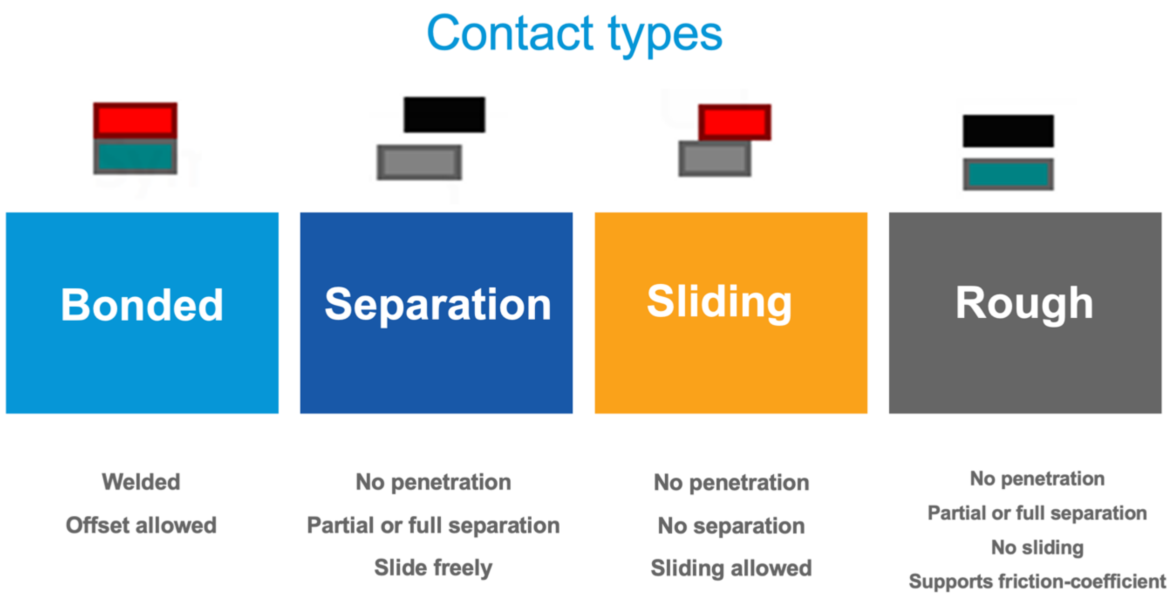

Figure 8: Contact types and their properties

Loads need to transfer across bodies so run Automatic contacts and create manual [contacts] where necessary.

Contacts

Type	What	Penetration	DOF of 2 entities	Separation	Frictionless (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	Yes	Yes, in tangential direction	
Rough	Similar to separation but no sliding	No	Separate	No gaps or separations	Yes/Defined	No	

Figure 9: A comparison of different types of contacts

7. Meshing

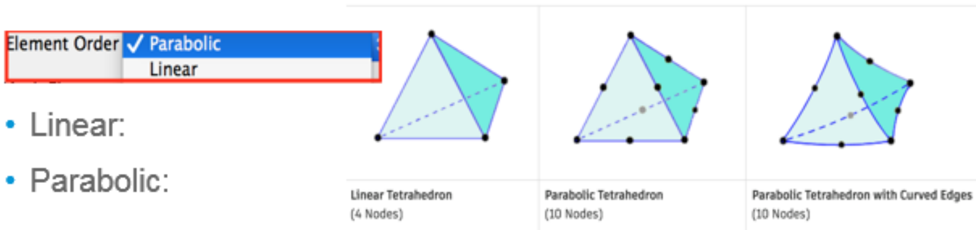

Figure 10: Element types

Good quality [Meshing]() is key to produce good results. Understanding [node and element]() types. The aspect ratio need to be adjusted if you get stress concentrations.

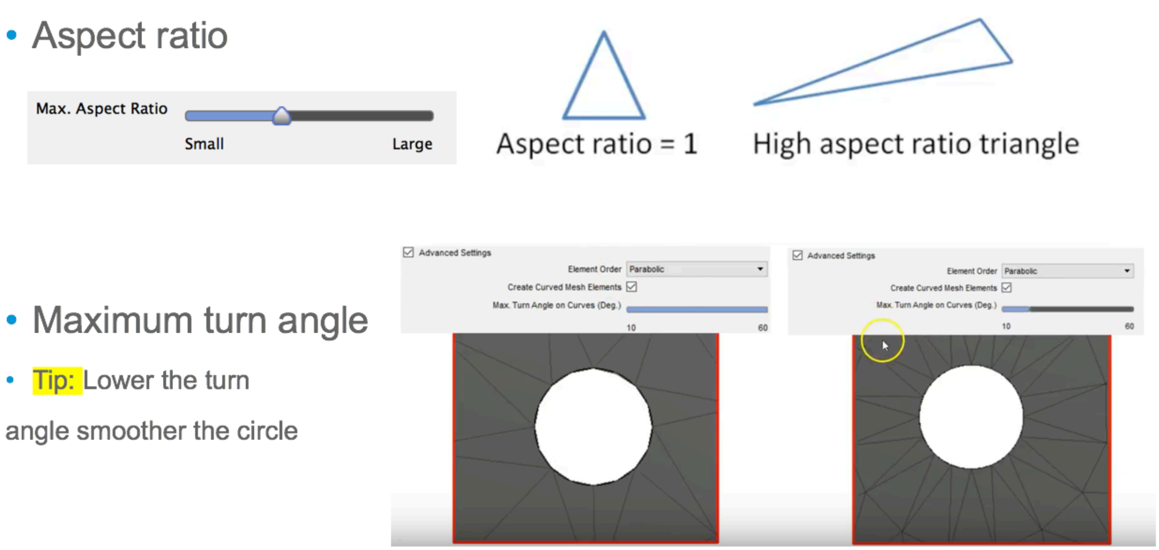

Figure 11: Aspect ratio and maximum turn angle

8. Pre-Check

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

Figure 12: Pre-check warnings and their meanings

Pre-check your studies before Solving and saves time. Once you come into Simulation workspace you can keep pressing Pre-check and satisfy the inputs needed for solve.

9. Solve

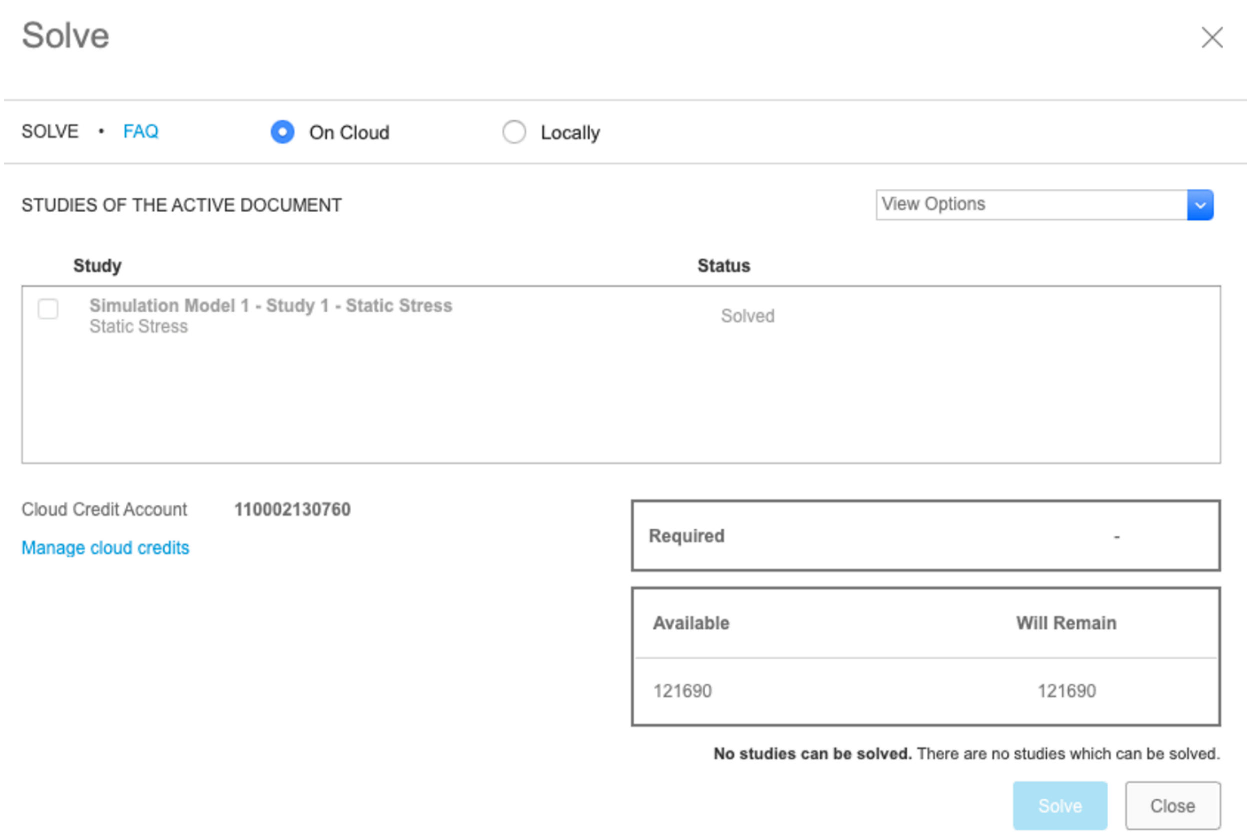

Figure 13: Solve dialog

- 9.1: **Solve dialog**: One stop dialog to do local or cloud solves. Also manage cloud credits. Tip: To resolve a solved study, uncheck and check the checkbox next to a load or constraint. No CC charged for cancelled solves. You can only cancel 1 job at a time
- 9.2: **Solve Status**: Display status of simulation jobs
- 9.3: **Solve Details**: Details of mesh for troubleshooting

10. Results

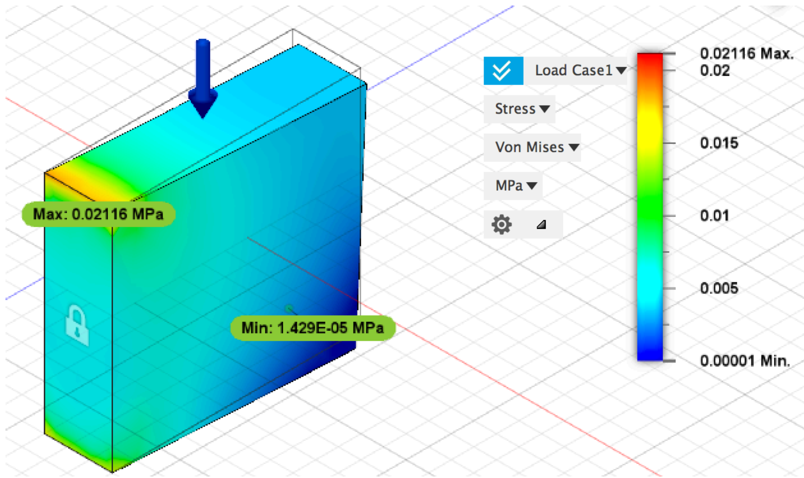

Figure 14: Results legend

Visualize results 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

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
Structural buckling	Total Displacement, Displacement X, Displacement Y, Displacement Z (Normalized), Critical Load Factor

Figure 15: Result types for various types of studies

10.1 : Display

10.2 : Animate

10.3 : Display Minimum and Maximum value labels

10.4 : Create Slice Plane

10.5 : Surface Probes

10.6 : Point Probes

10.7 : Legend

10.8 : Reactions

10.9 : Deformation Scale

10.10 : Comparing Simulation Results

10.11 : Results Details

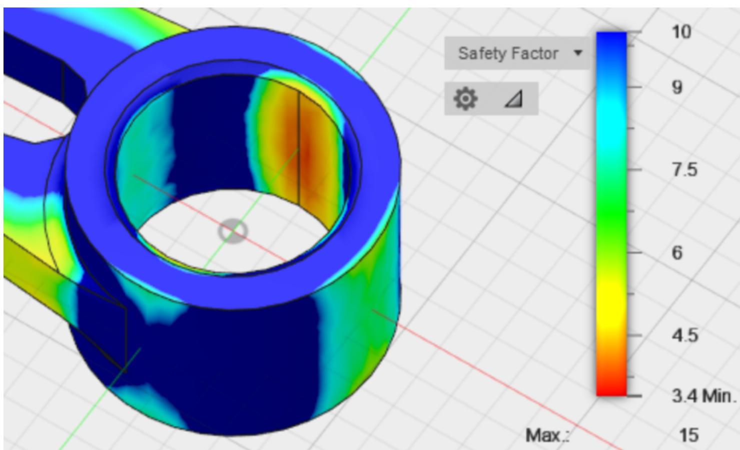

Figure 16: Safety factor result

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.

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

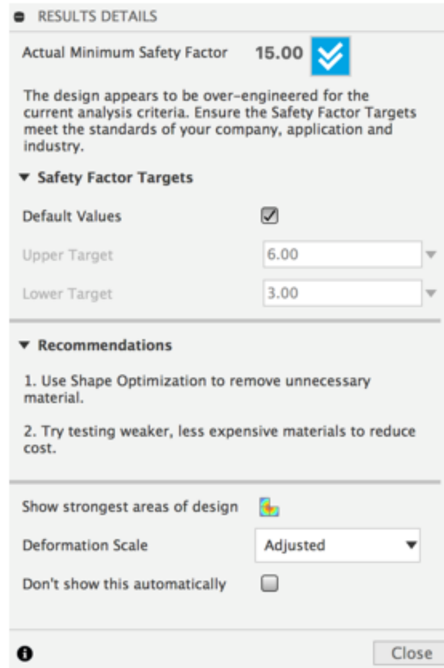

Figure 17: Result details

Tip: Use Dynamic Content (Javascript) option which provides collapsible sections
Tip: Compare workspace available after results generation
Compare workspace [video]

11. Conclusion

Here is a link for [tutorials] [hands-on exercises].

Demo

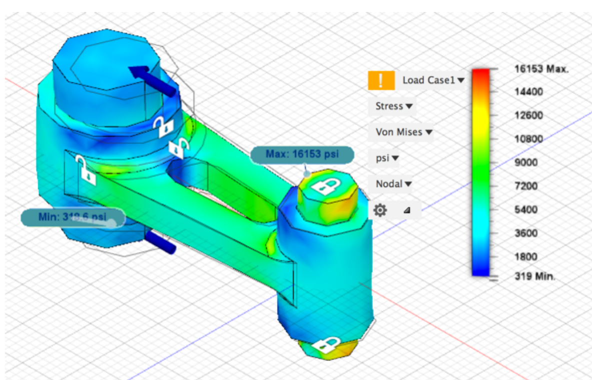

Figure 18: LSS and SO studies demo

- How create a Static Stress Analysis and Solve?
- How to create a Shape Optimization study and Solve?