AUTODESK. UNIVERSITY

IM500020

Become a Fusion 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

- Sofware design and development
- Working on Fusion Sim & Generative
- 24 years @ Autodesk
- Many times @ AU
- Bachelors Masters Doctorate in Mech Engg
- Co-author of "Mastering Inventor" book

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

Introduction

A vast amount of knowledge exists on the internet on Fusion Simulation. Youtube has videos about Fusion Simulation that are very helpful. You will find key links and tips to navigate through Fusion Simulation.

kil-lic3

Figure 1: Simulation Steps

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

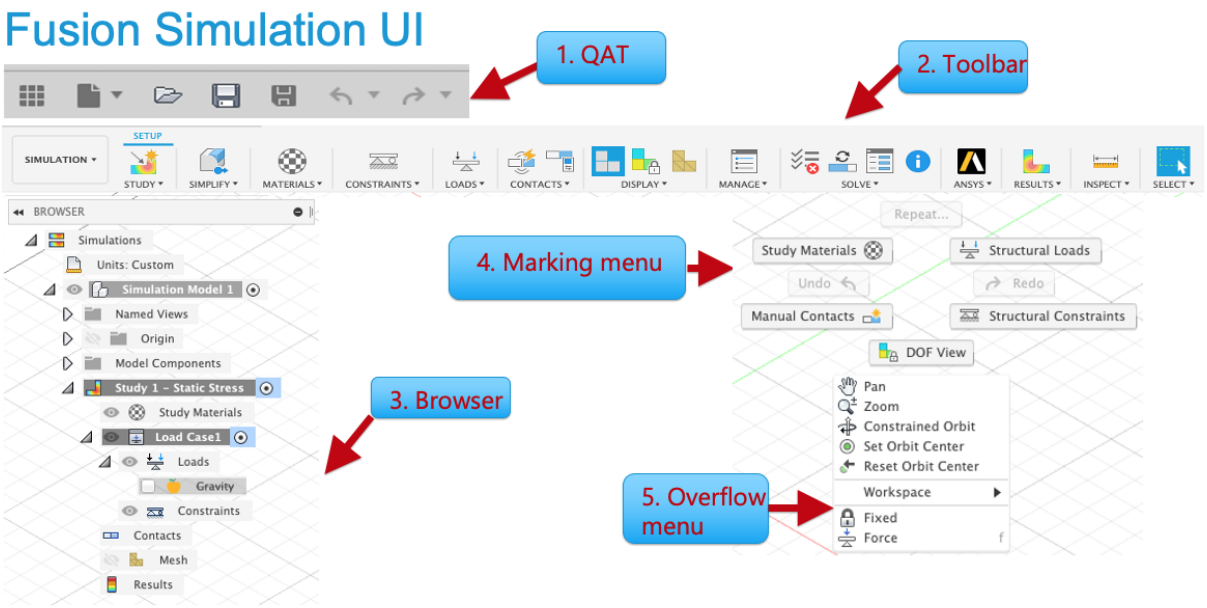

Figure 2: Fusion Simulation User Interface

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. Consider simplifying the model by removing

- Unneeded Fillets
- Embossed Text

- Actual threads
- Leverage symmetry
- Body/components that could be approximated by point masses. The Point Mass command automatically suppresses the selected body/components .

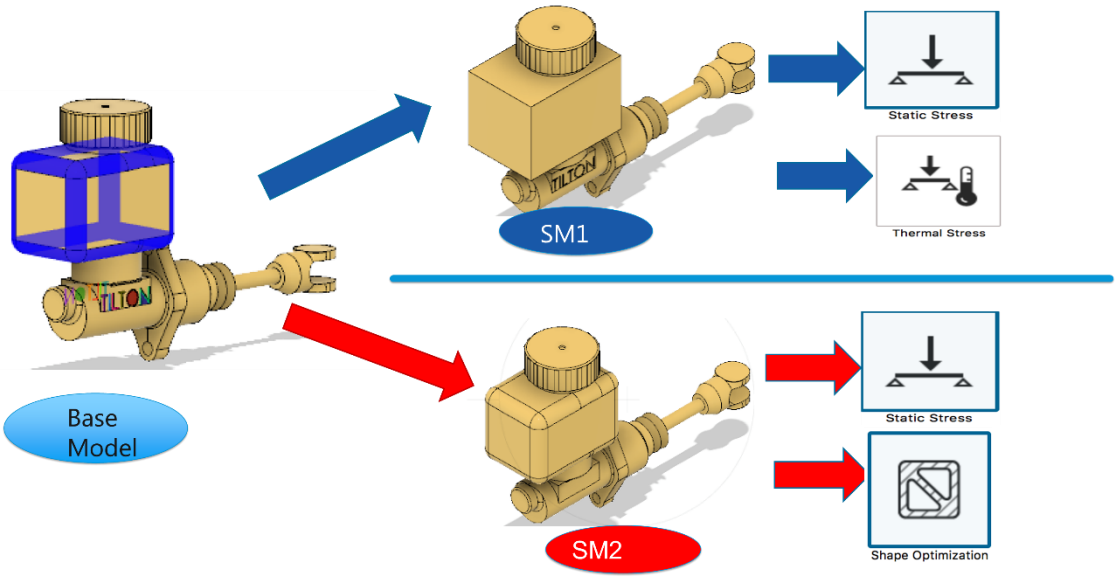

Figure 3: Base Model, Simulation models and studies

Demo Video: This video shows an example of how you can reduce the complexity of the model to get a 6X speedup and still have meaningful results of your simulation.

2. Studies

Figure 4: Simulation studies

There are ten 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

How to create a study?

- 2.1 Static Stress
- 2.2 Modal frequencies
- 2.3 Structural Buckling
- 2.4 Thermal
- 2.5 Thermal Stress
- 2.6 Shape Optimization
- 2.6 Non Linear Static Stress
- 2.7 Event Simulation
- 2.8 Electronics Cooling
- 2.9 Event Simulation
- 2.10 Plastic Injection Molding
- 2.11 Fusion to Ansys: Fusion study setups can be sent to Ansys and in Ansys various study types including but not limited to Fatigue, Vibration, advanced non-linear analysis studies can be created.

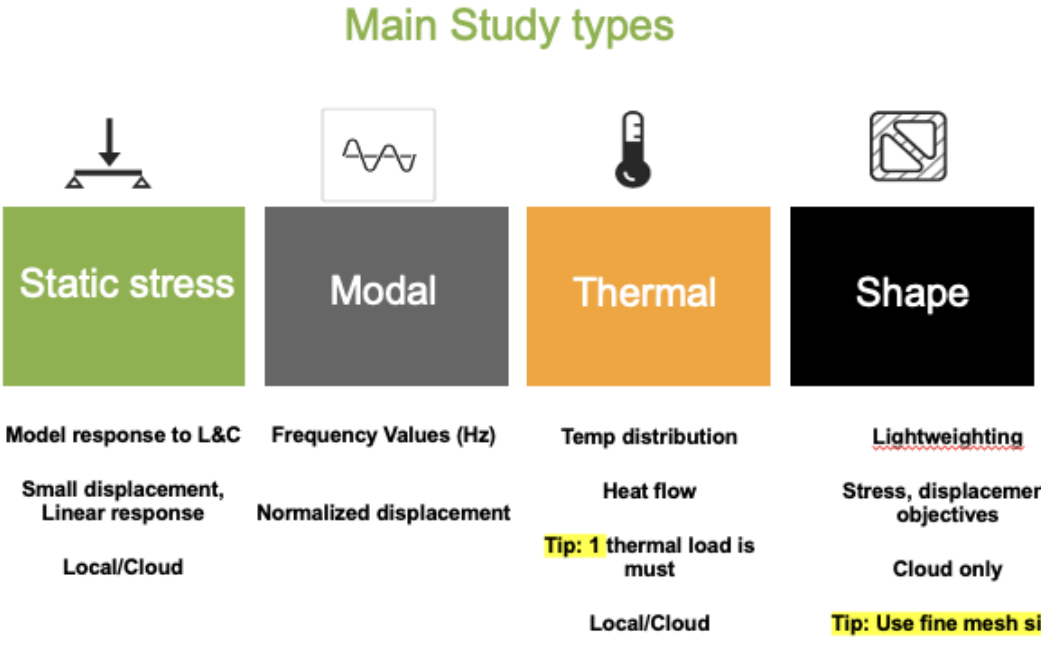

Figure 5: Main study types

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. Right Mouse Button(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

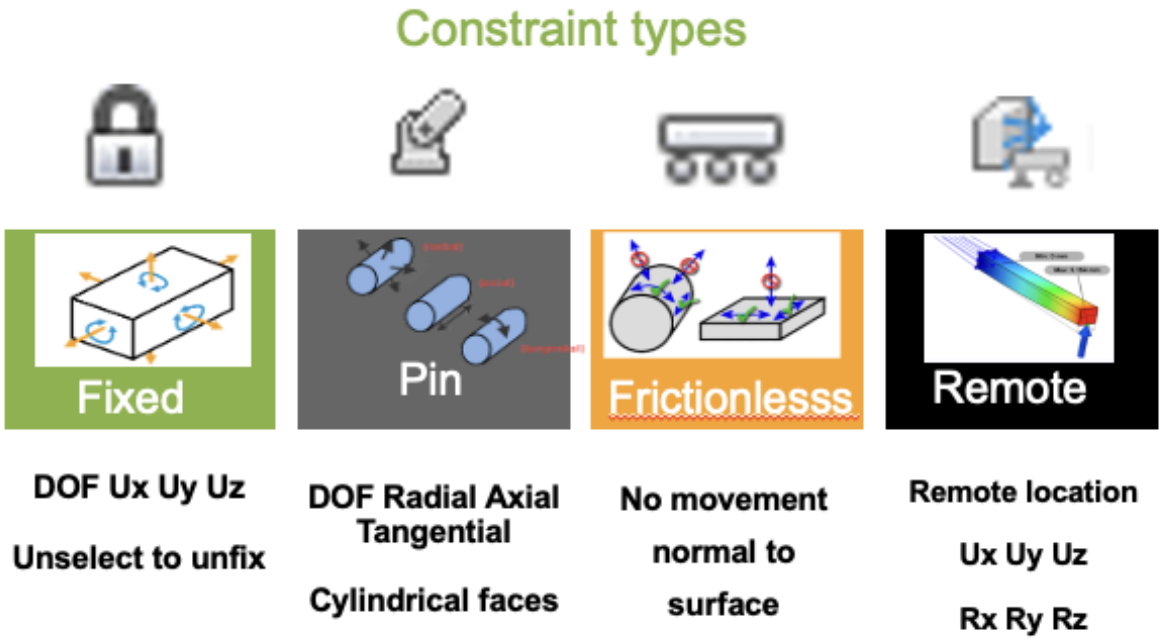

Figure 6: Constraint types

To restrict the model to a particular location apply Constraints. Apply any of the 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.

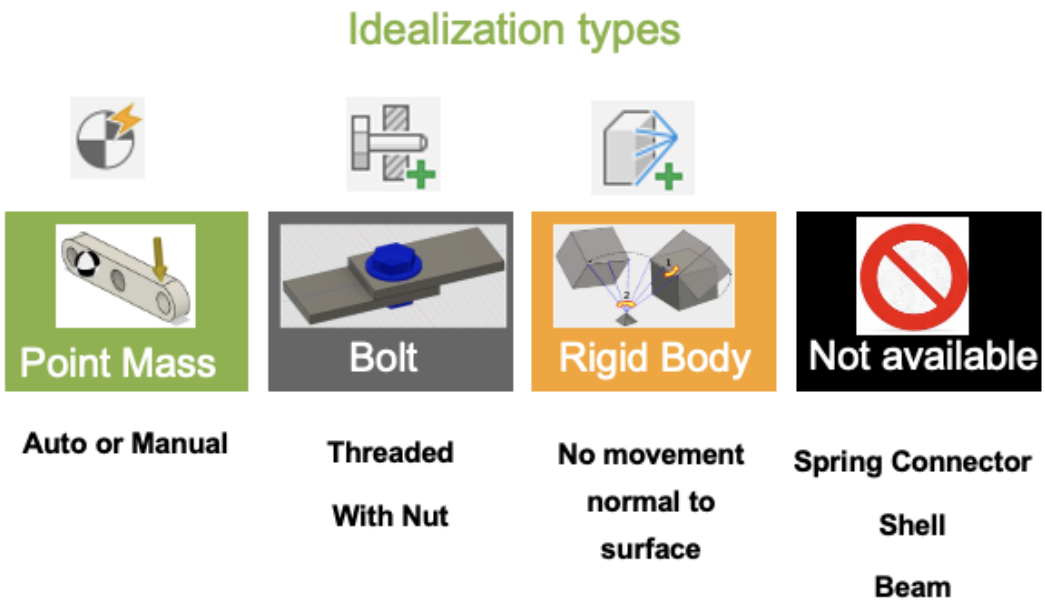

Figure 7: Idealization types

Geometry which is difficult to model such as sand on the back of a truck OR vehicle on a bridge can be approximated to point-masses. 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. Bolt connectors are helpful to create without having to model the actual bolt geometry. All loads and constraints, except pin constraints and frictionless constraints, can be applied to rigid bodies in an Event Simulation study.

5. Loads

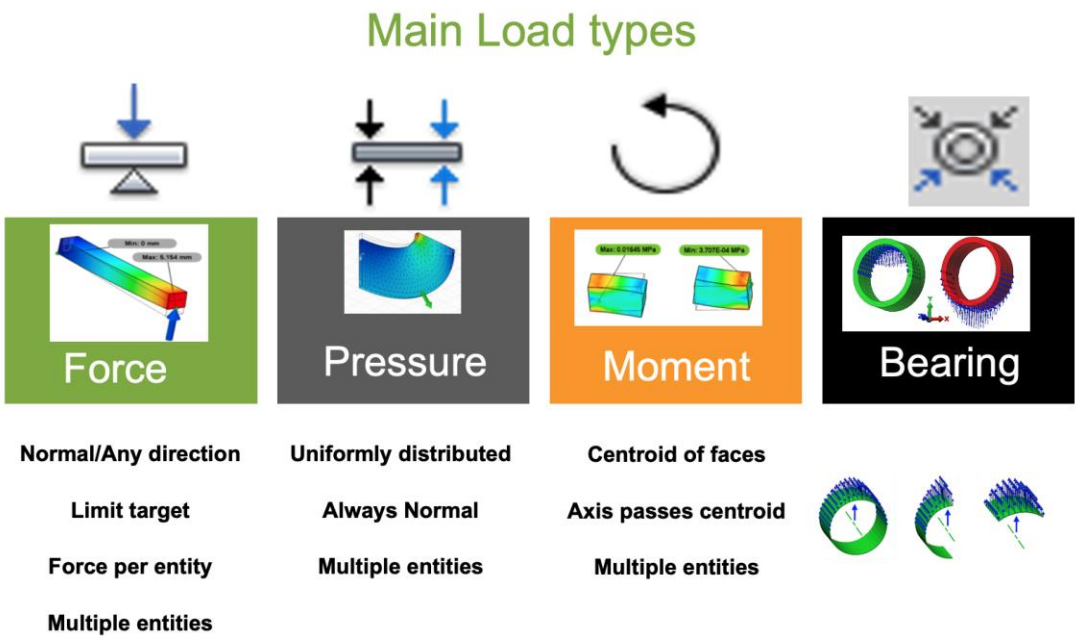

Figure 8: Main Load types

How much load does the model need to resist? Apply any of the 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
- 5.3 **How to assign global loads** **(Linear, Angular):** Learn how to apply linear acceleration, angular velocity, and angular acceleration loads**.**

6. Contacts

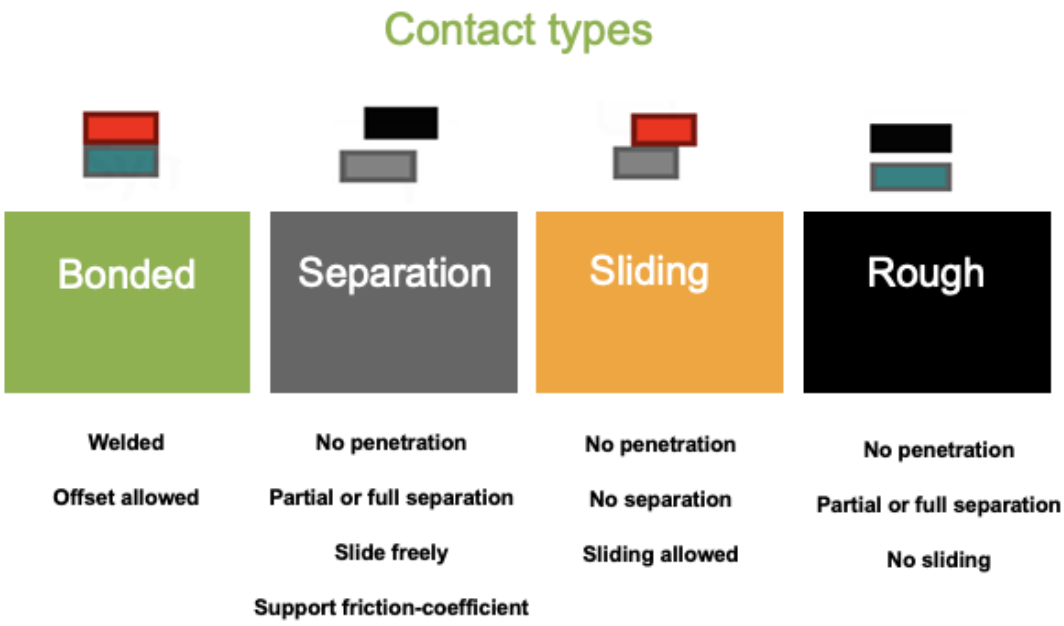

Figure 9 Contact types and their properties

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	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	

Figure 10: A comparison of different types of contacts

7. Meshing

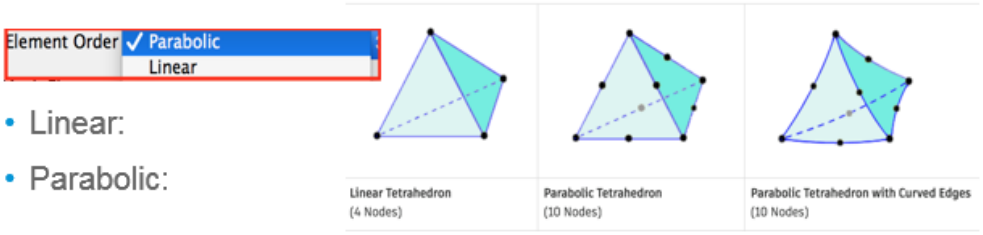

Figure 11: Element types

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.

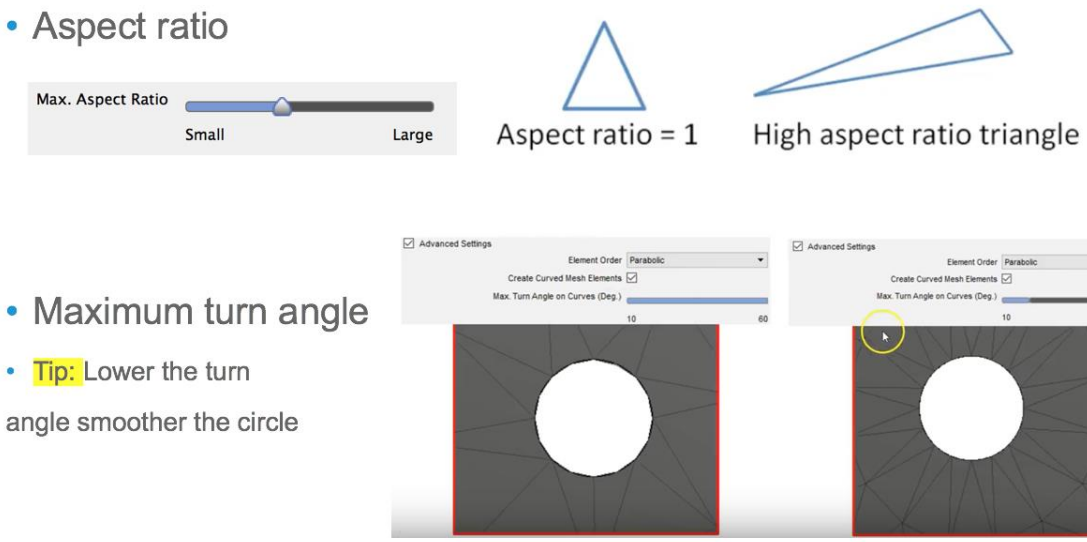

Figure 12: Aspect ratio and maximum turn angle

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 13: Pre-check warnings and their meanings

Pre-check your studies before Solving to know the errors and warnings. Once you come into Simulation workspace you can keep pressing Pre-check and satisfy the inputs needed for solve. This is a good way to satisfy all the inputs needed tor a succesful solve.

9. Solve

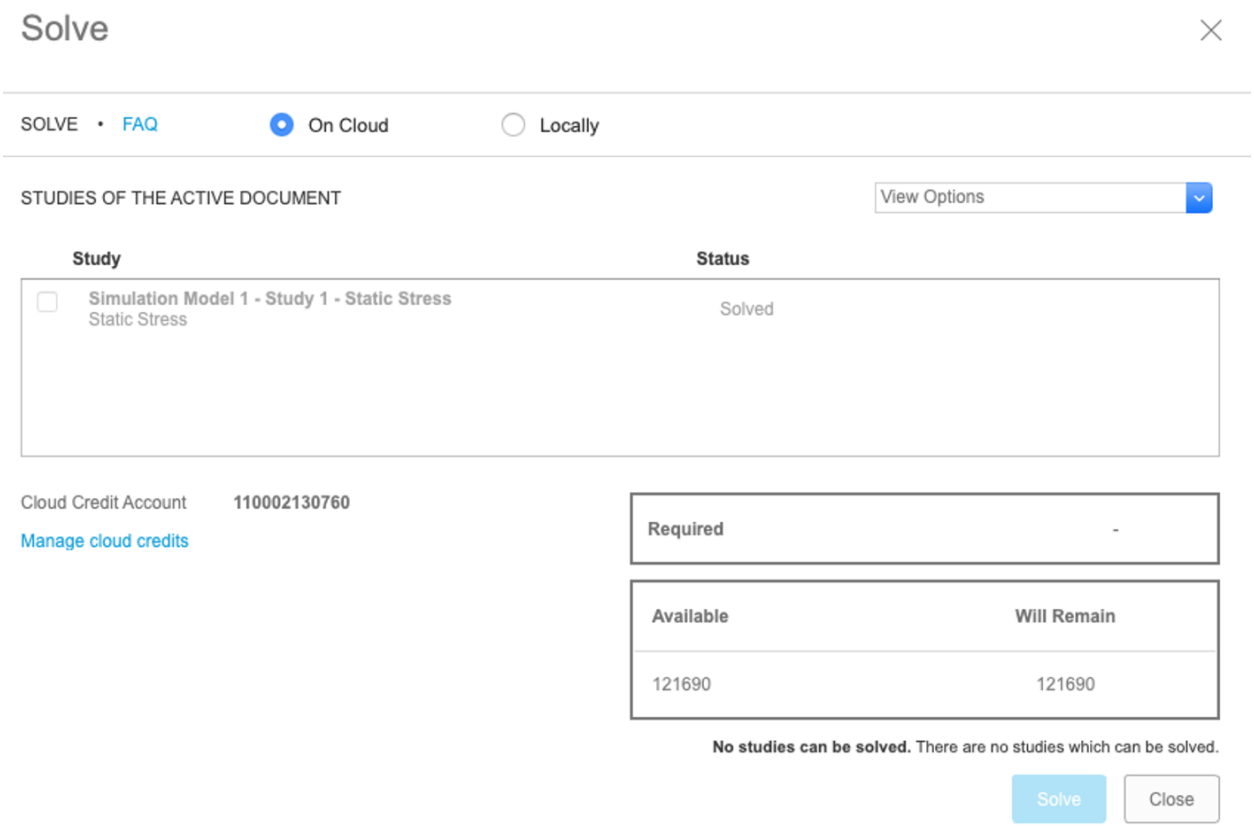

Figure 14: Solve dialog

- 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

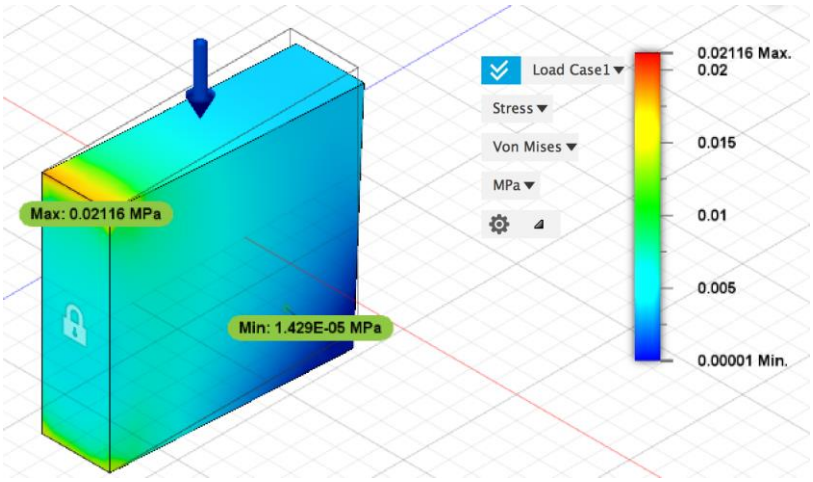

Figure 15: Results legend

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

Figure 16: Result types for various types of studies

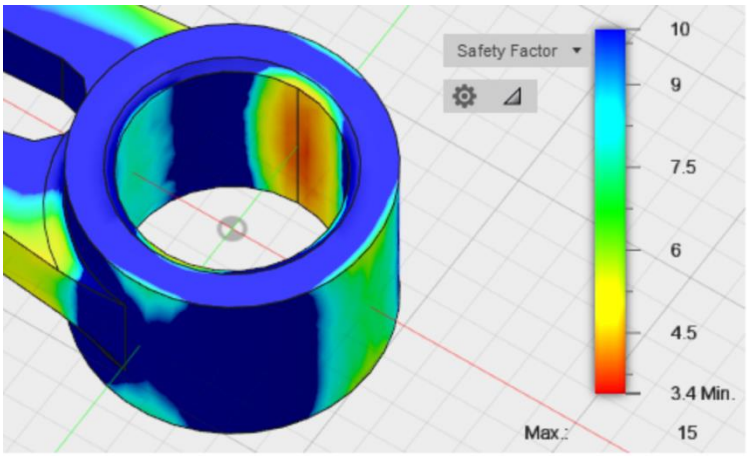

Figure 17: Safety factor result

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

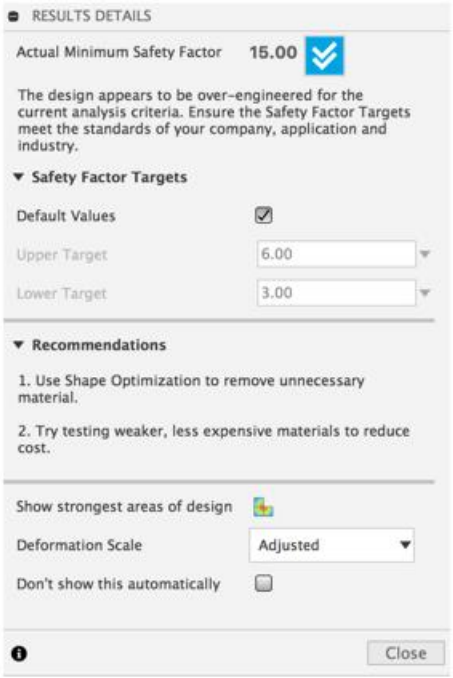

Figure 18: Result details

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

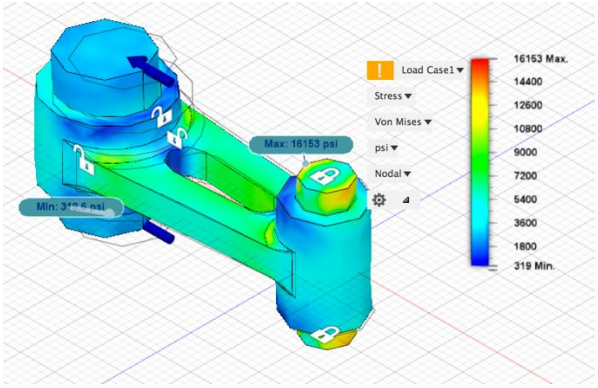

Figure 19: LSS and SO studies demo

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