Become a Fusion Simulation expert in 60 minutes

Dr. Shekar Sub

Sr.Principal Engr/Architect
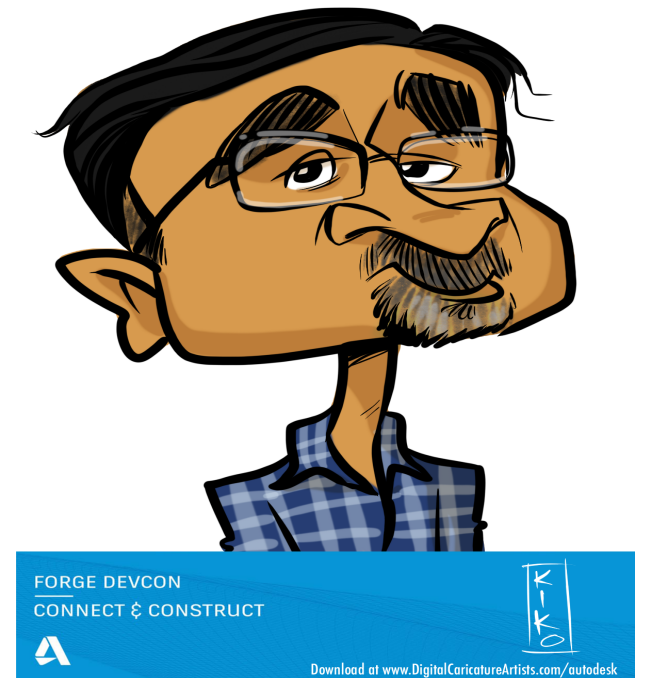
Contents

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

About the speaker

- Currently working on ANSYS collaboration

- 21 years @ Autodesk

- Lead for Inventor and Fusion Sim

- Many times @ AU presenting class

- Inventor and Fusion community forums

- Volunteer for FIRST robotics

- Co-author of "Mastering Inventor...."

Why Simulation?

- Samsung Note 7 phone fire!!!

- Recalled 1 million of 2.5 million phones manufactured

- Every single phone recalled

- Lost ~$5 billion

Why Simulation?

Fusion Simulation

- Easy to use

- Local & Cloud solve

- Meshing (Tetrahedral)

- Industry acclaimed Nastran, Explicit Solvers

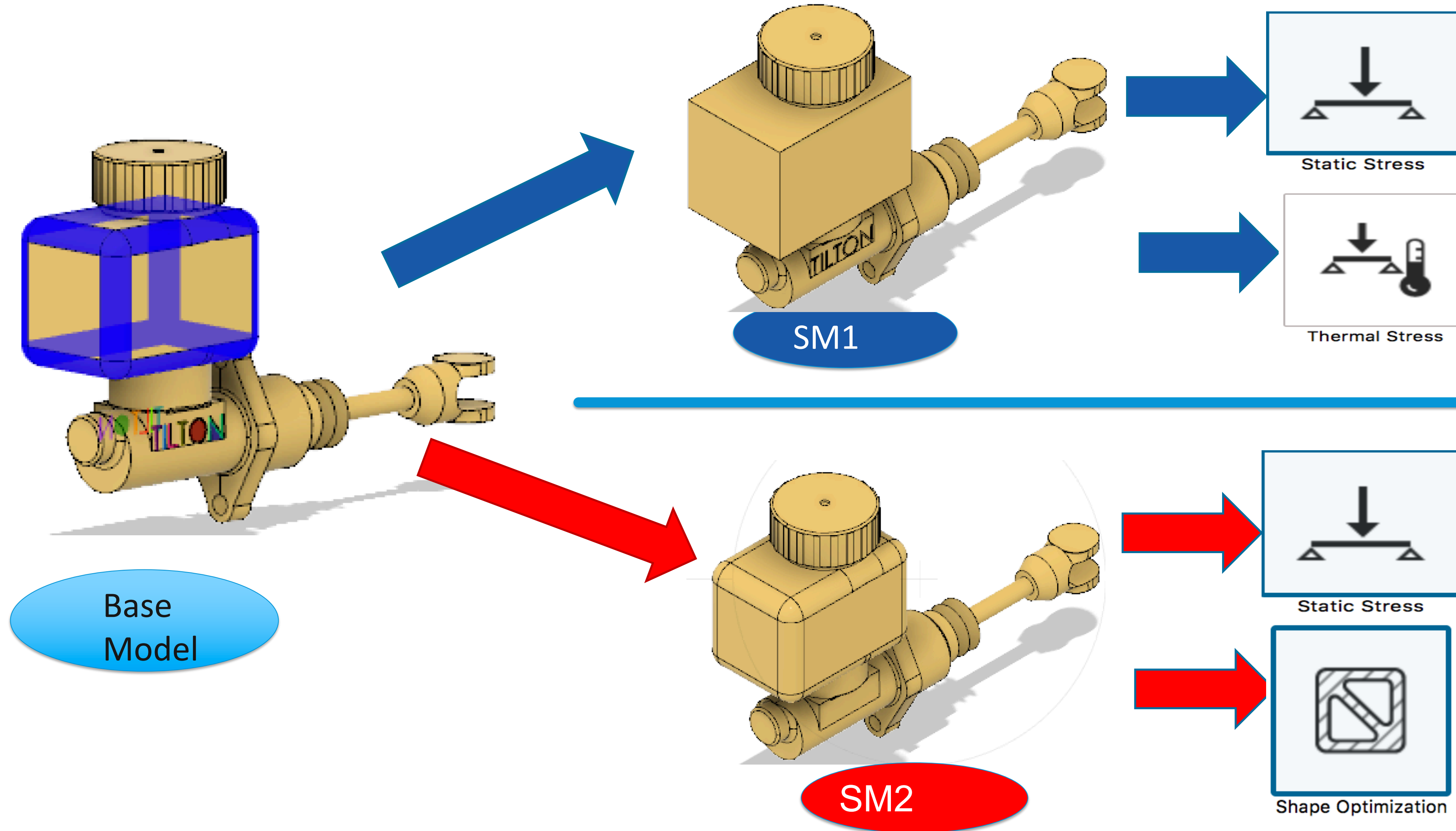- Multi-threaded

- Multi-platform

Simulation Steps

Simplify → Create Studies → Edit materials → Apply L&C → Contacts → Mesh → Solve → Results

Simplification
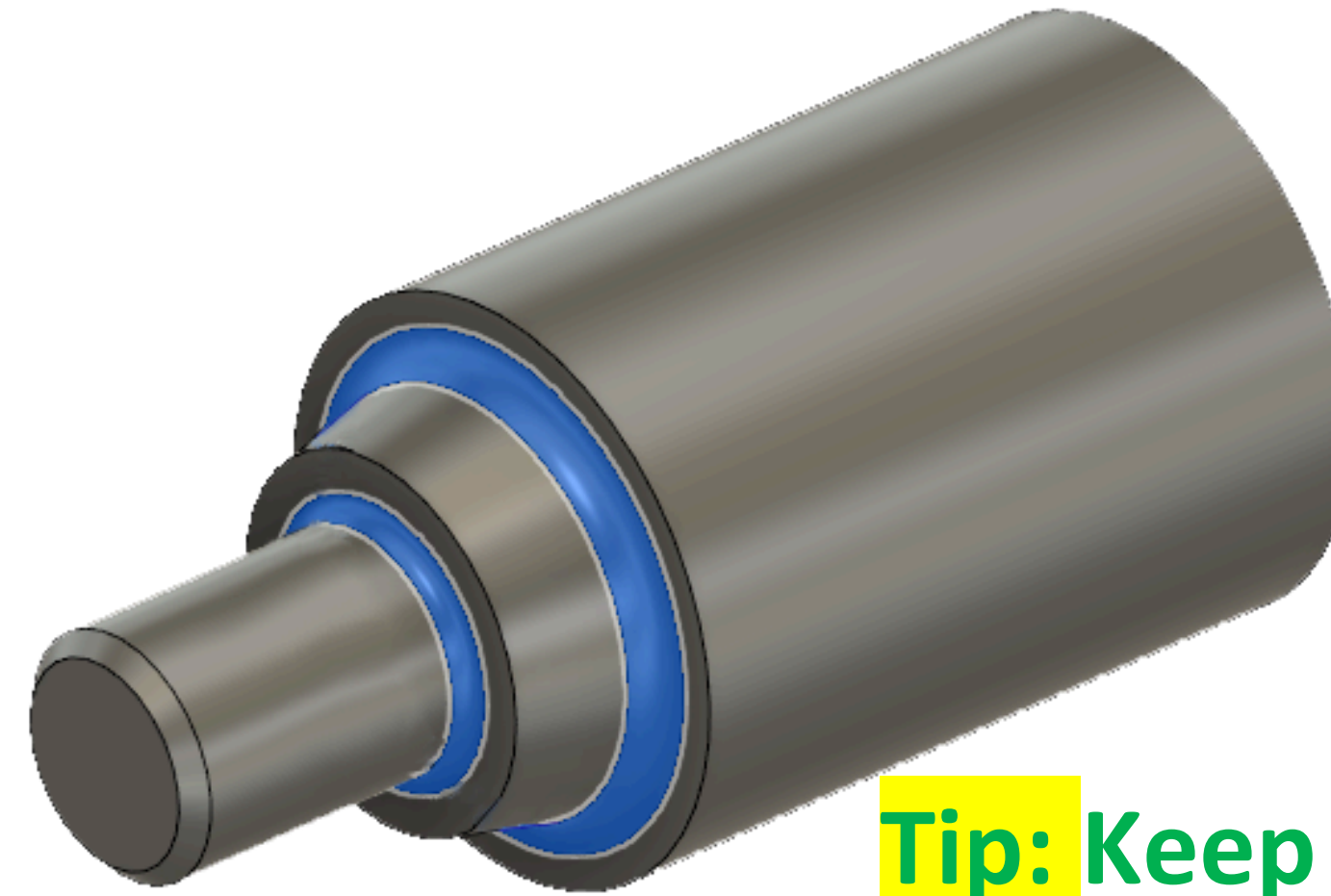
Simplify workspace

- **"What-if"** workspace (Simplify)

- Remove unneeded geometry

  o Features

  o Bodies/Components

- Multiple variants of the base production model

Simulation Models

Base Model

SM1

Static Stress

Thermal Stress

SM2

Static Stress

Shape Optimization

# What to Remove? What to Use?

- Small features

  o Removed from critical stress regions

  o No impact on overall stiffness

  o Does not alter mass for frequency analysis

- Screws and bolts. Use connectors

- Lifting eyes or handles

- Name plates, panel switches or indicator lights. Use point masses.

Tip: Keep these internal fillets

Symmetry

- Split body + Remove

- Model is statically stable
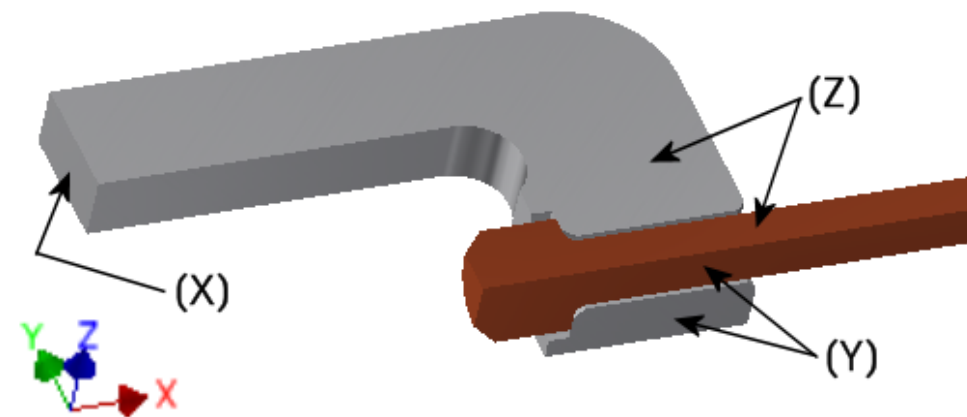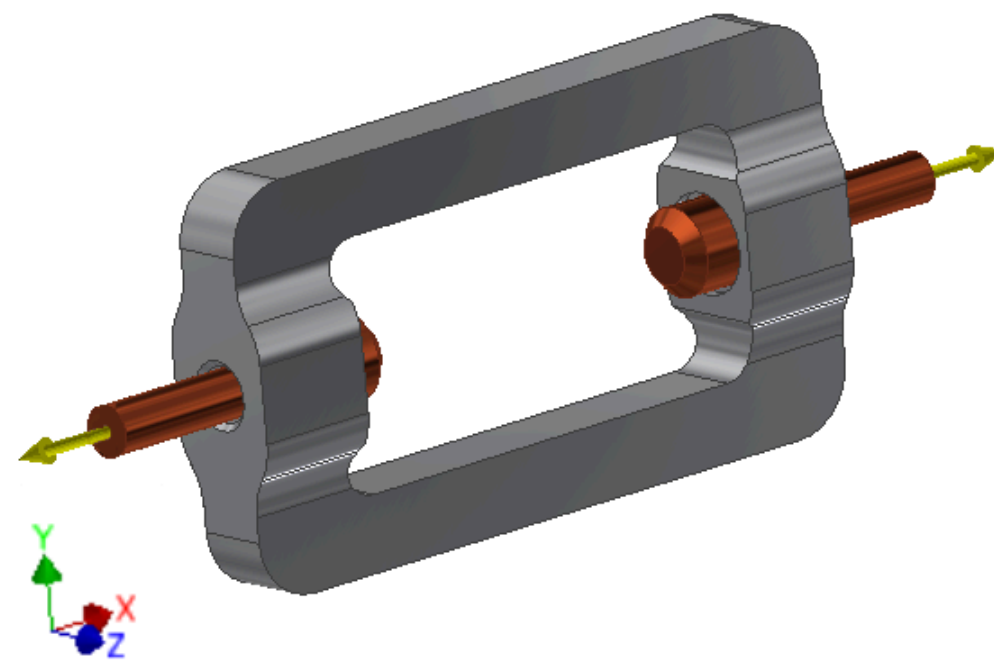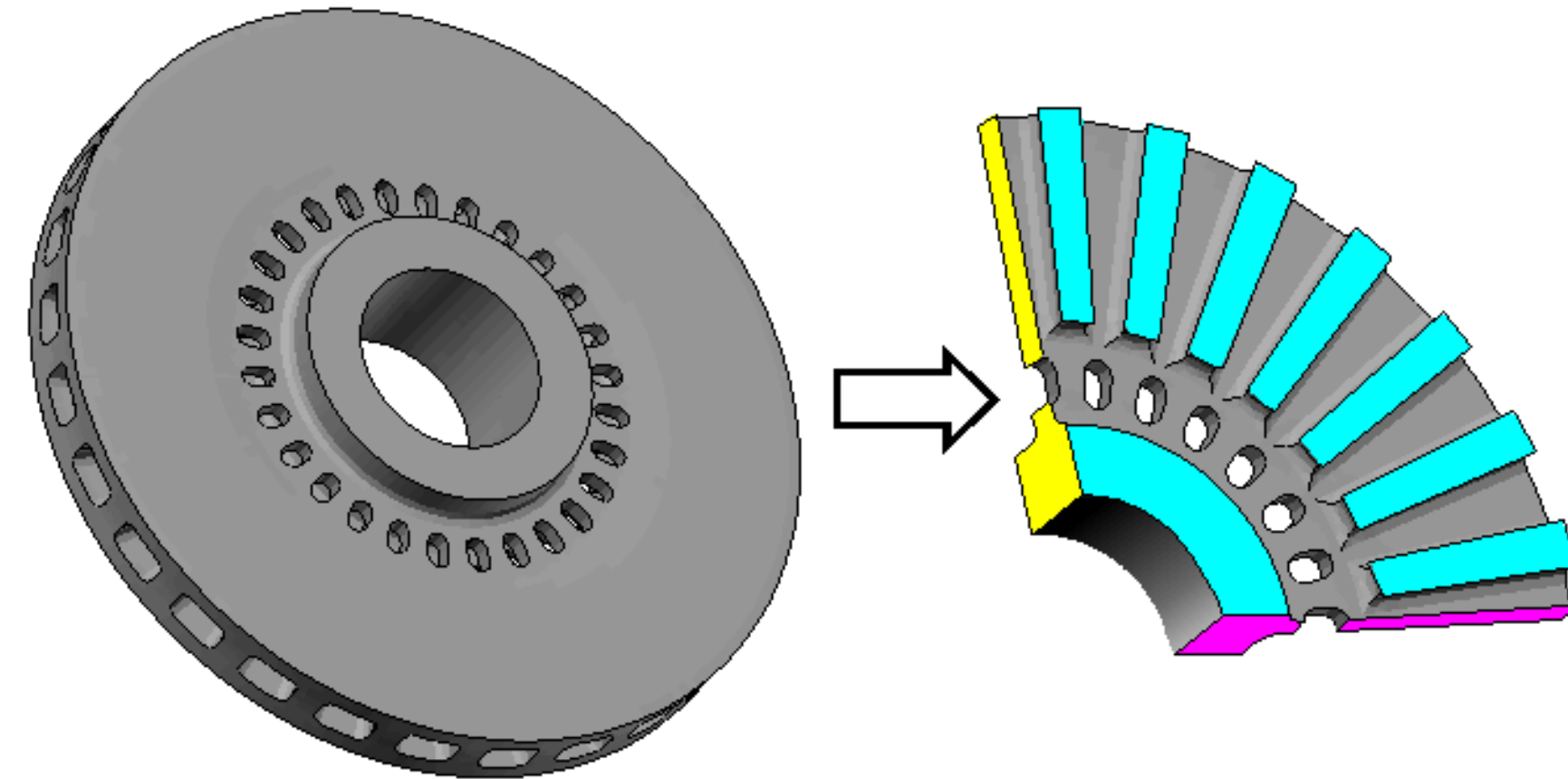
- Tip: Do symmetry changes in Simplify workspace. Results have slight variations.

- 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.
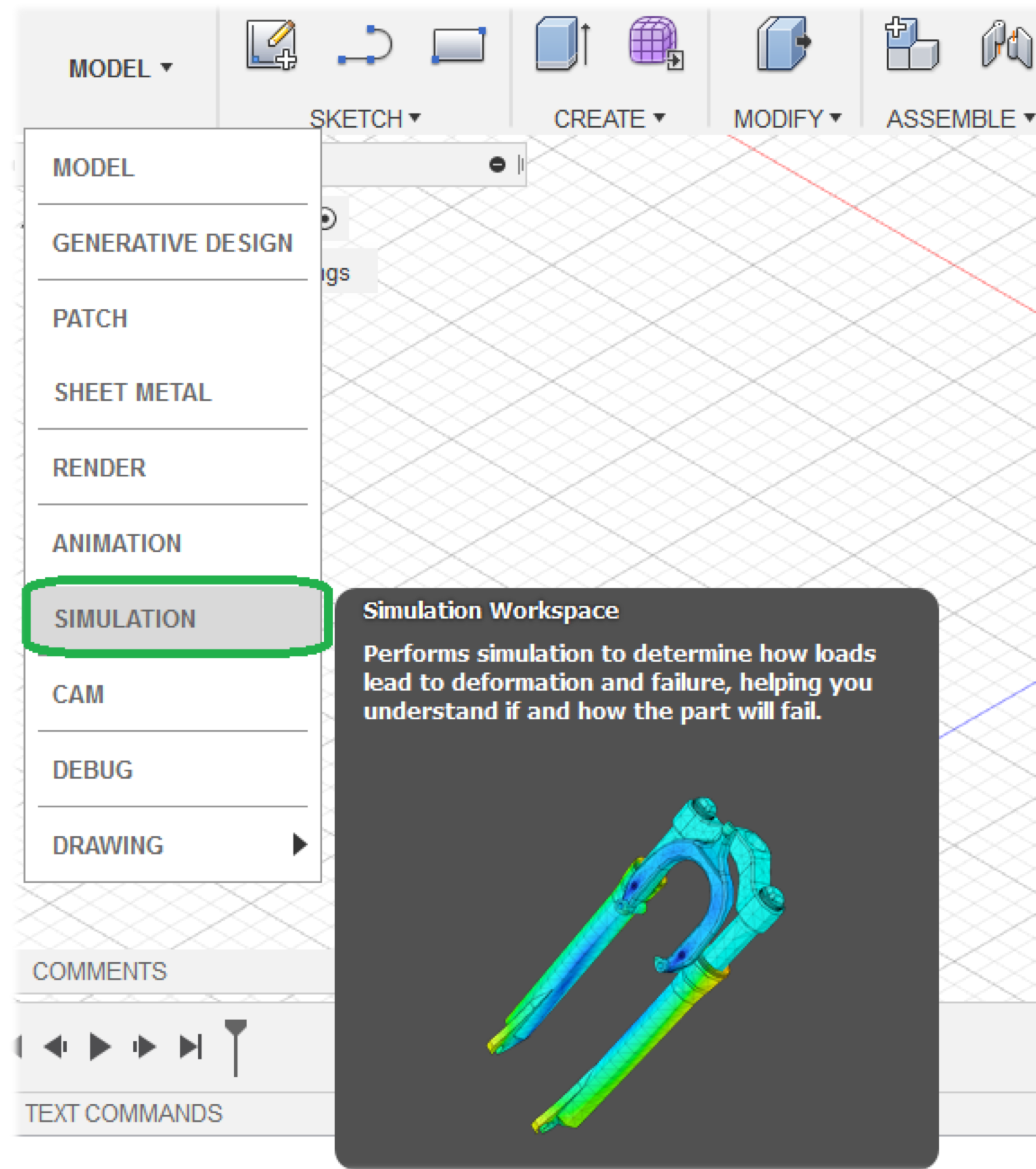
Demo

- <mark>Tip:</mark> In some fillet over fillet/complex fillet cases it is difficult/impossible to remove the fillet. Add Spheres at the intersections and then use Remove faces.

- [Simplify tools](#)

Studies

Sim Workspace

- Integrated

- Tip: No geometry creation

- Setup->Mesh->Solve->Results

- Fully associative

- Has Compare workspace

- Sibling of Generative Design workspace

MODEL ▼

SKETCH ▼   CREATE ▼   MODIFY ▼   ASSEMBLE ▼

MODEL

GENERATIVE DESIGN

PATCH

SHEET METAL

RENDER

ANIMATION

SIMULATION

CAM

DEBUG

DRAWING ▶

Simulation Workspace

Performs simulation to determine how loads lead to deformation and failure, helping you understand if and how the part will fail.

COMMENTS

TEXT COMMANDS

Fusion Simulation UI

1. QAT

2. Toolbar

SIMULATION ▾ | STUDY ▾ | SIMPLIFY ▾ | MATERIALS ▾ | CONSTRAINTS ▾ | LOADS ▾ | CONTACTS ▾ | DISPLAY ▾ | SOLVE ▾ | MANAGE ▾ | RESULTS ▾ | INSPECT ▾ | SELECT ▾

Simulations
Units: Custom
Simulation Model 1
  Named Views
  Origin
  Model Components
  Study 1 – Static Stress
    Study Materials
    Load Case1
    Contacts
    Mesh
    Results
  Study 2 – Shape Optimization
  Study 3 – Modal Frequencies

3. Browser

4. Marking menu

Repeat Results Details

Study Materials | Structural Loads
Undo | Redo
Manual Contacts | Structural Constraints

DOF View

Set Orbit Center
Reset Orbit Center
Workspace ▸
Study ▸
Simplify
Materials ▸
Constraints ▸
Loads ▸
Contacts ▸
Display ▸
Solve ▸
Manage ▸
Results ▸
Inspect ▸
Select ▸

5. Overflow menu

Study types

- 8
- Help with "Choose a study type" available
- Tip: Create & then Edit. Studies are interchangeable
- Clone, Delete, Properties available
- Tech Preview: Event Simulation
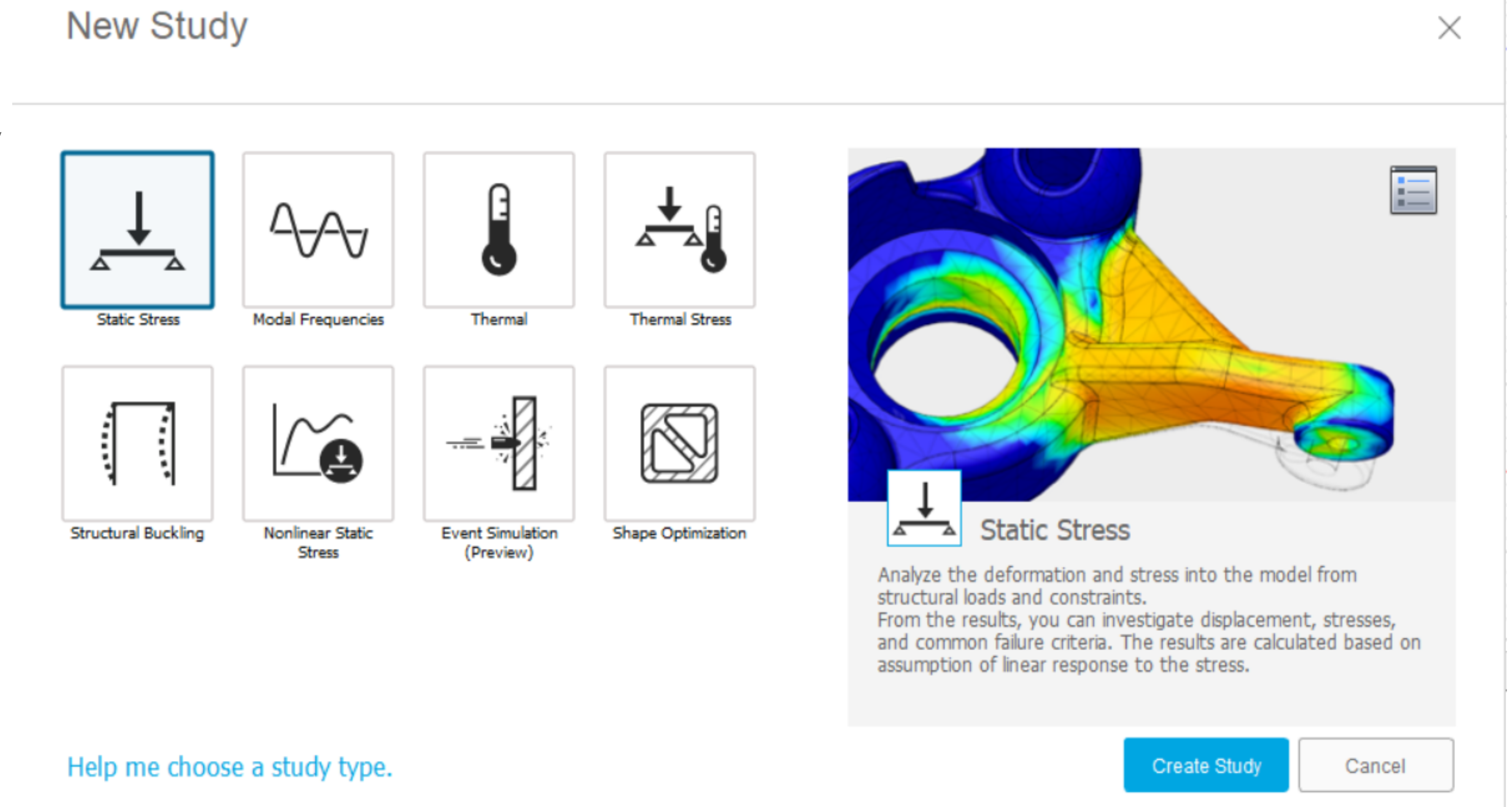
Main Study types

Static stress	Non linear	Thermal	Shape

Static stress

Model response to L&C

Small displacement,

Linear response

Local/Cloud

Non linear

Large deformation, motion.

Non-linear material

Steps

Cloud only

Thermal

Temp distribution

Heat flow

<mark>Tip: 1</mark> thermal load is must

Local/Cloud

Shape

Lightweighting

Stress, displacement objectives

Cloud only

<mark>Tip: Use fine mesh size</mark>

Materials

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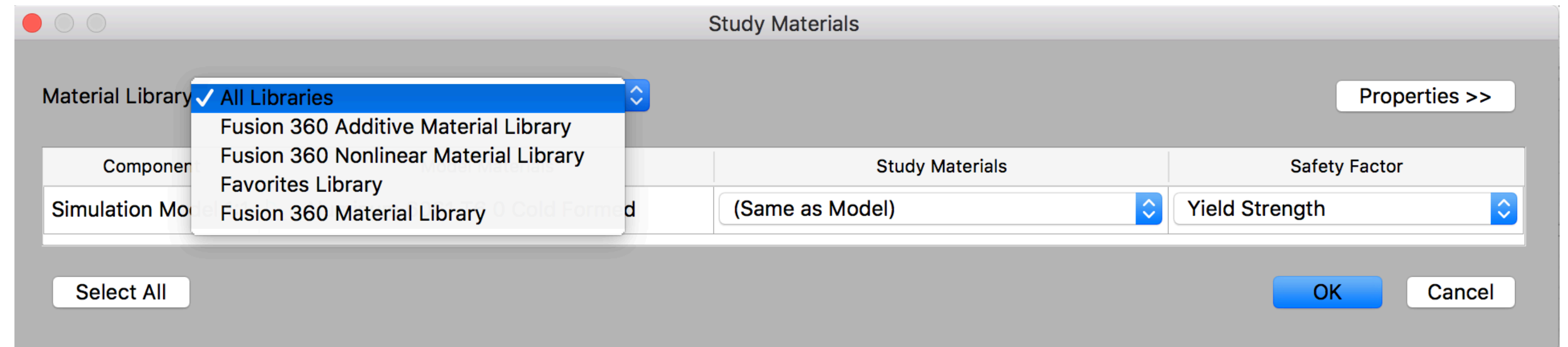
- Material cannot be used for solve
- Value missing or not allowable
- Non-linear material for Linear solve

Materials

- You can [define a stress-strain](#) curve for a non-linear material. Bring in from MatWeb website.
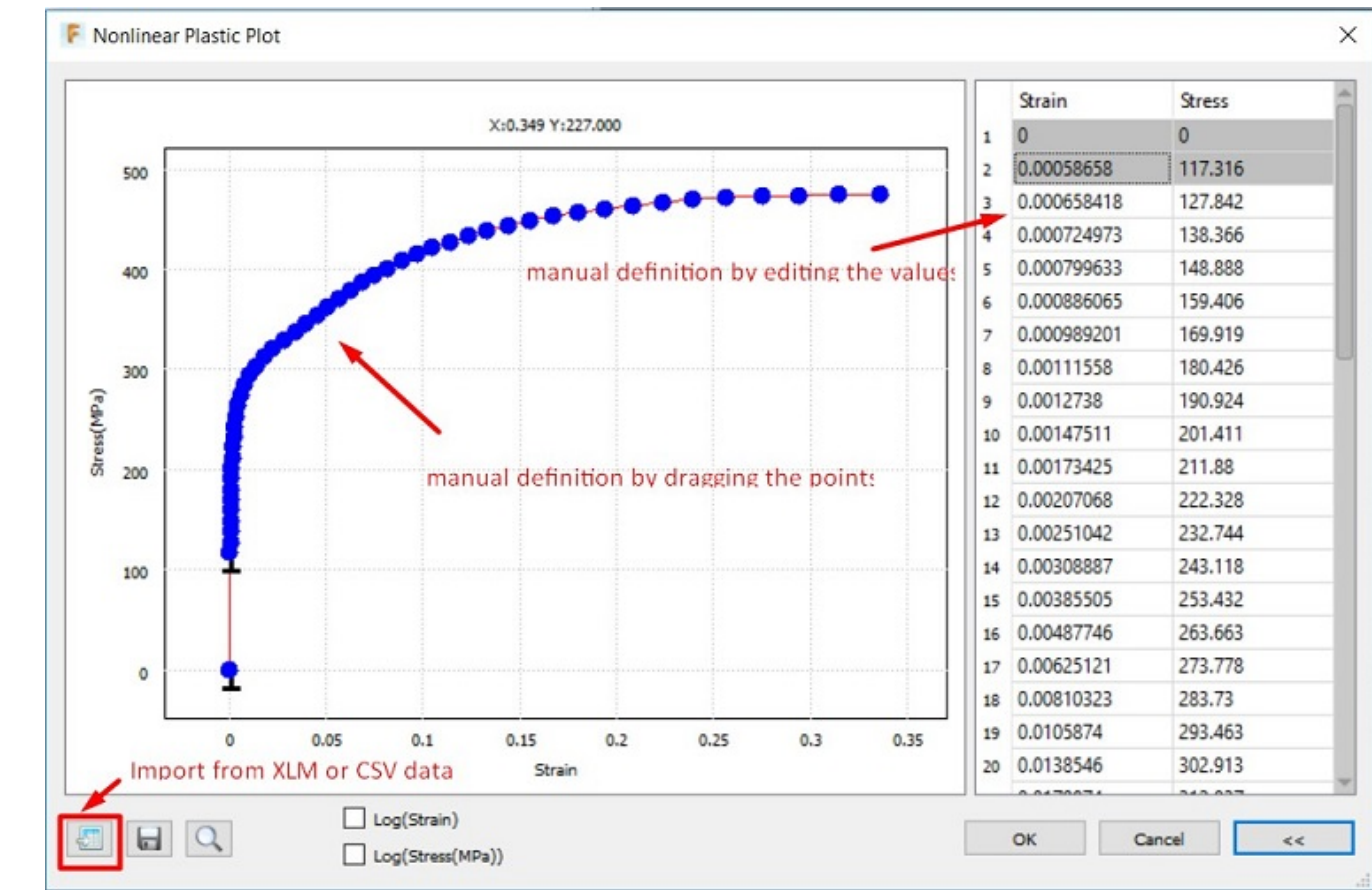
- <mark>Tip:</mark> Ctrl to add rows in Study Materials dialog. Shift to select a bunch of rows

- <mark>Tip:</mark> RMB on a material in the browser to access the **Study Materials** command, all components that use the same material are automatically preselected

Constraints

Constraints

- Goal: Limit translational, rotational motion

- Need at least a few

- Entities: Faces, Edges, Vertices

STRUCTURAL CONSTRAINTS

Type: Fixed

Targets: Select

Axis: Ux Uy Uz

OK   Cancel

Fixed
Fixed
Pin
Frictionless
Prescribed Displacement
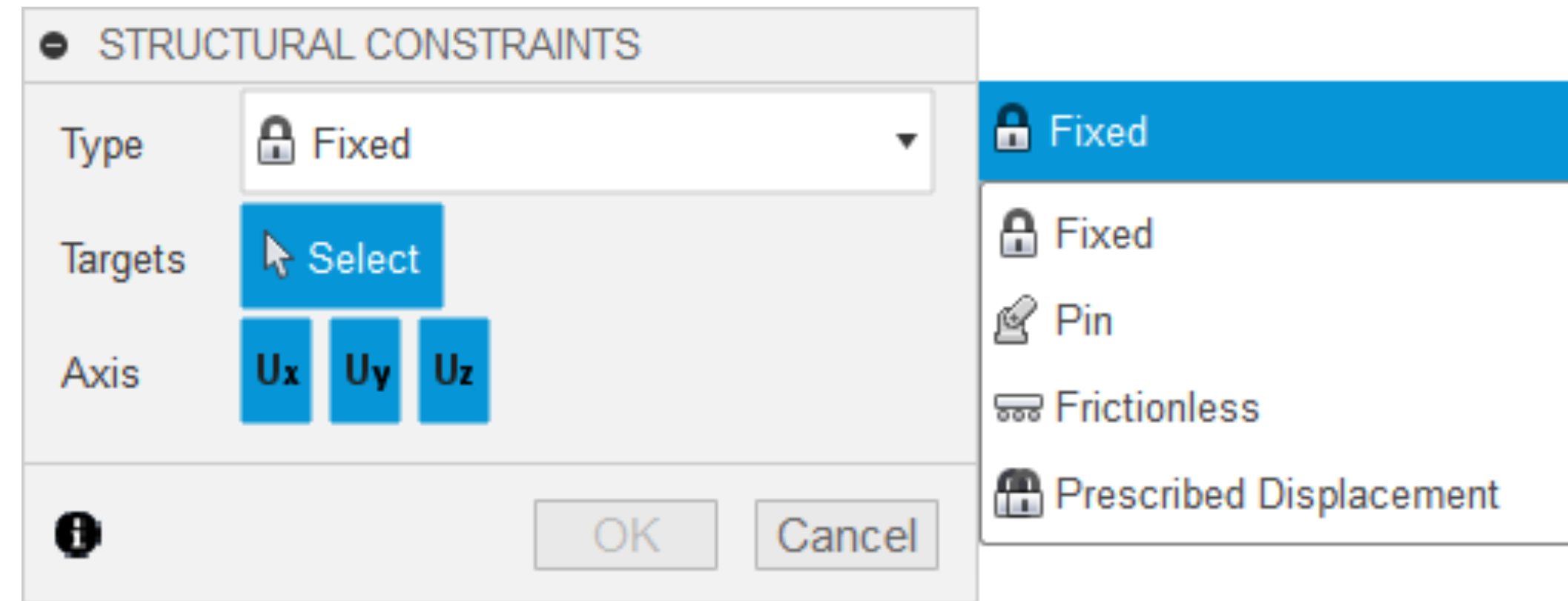
- Tip: In some situations partially constrain the model and use the **Remove rigid body** modes option. Solver will apply an acceleration load to keep model statically stable.
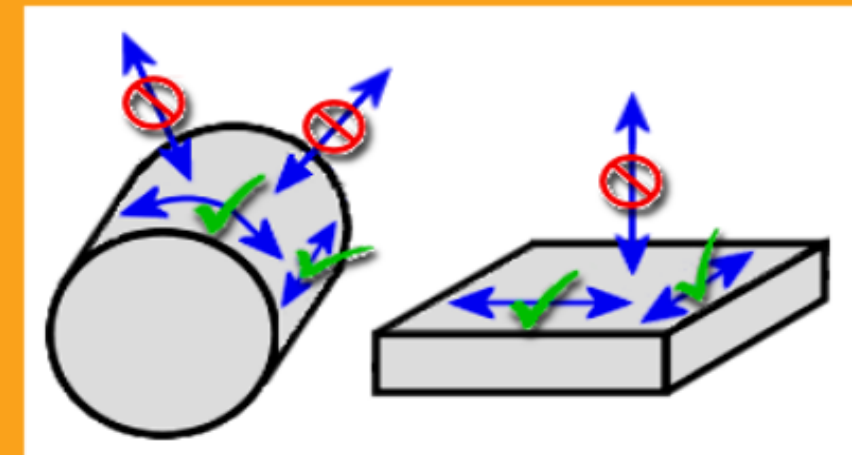
Constraint types
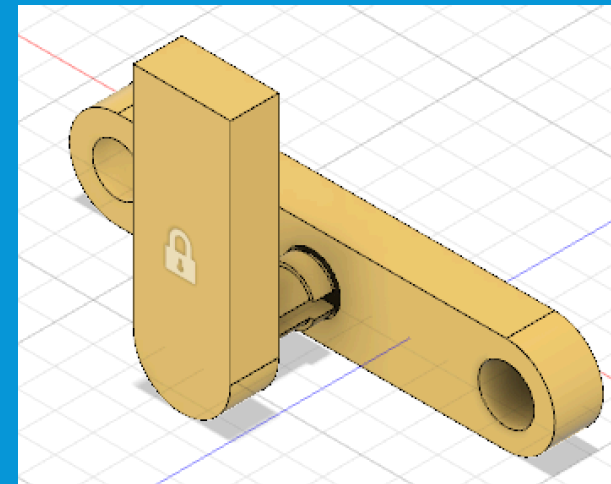
Fixed

DOF Ux Uy Uz

Unselect to unfix
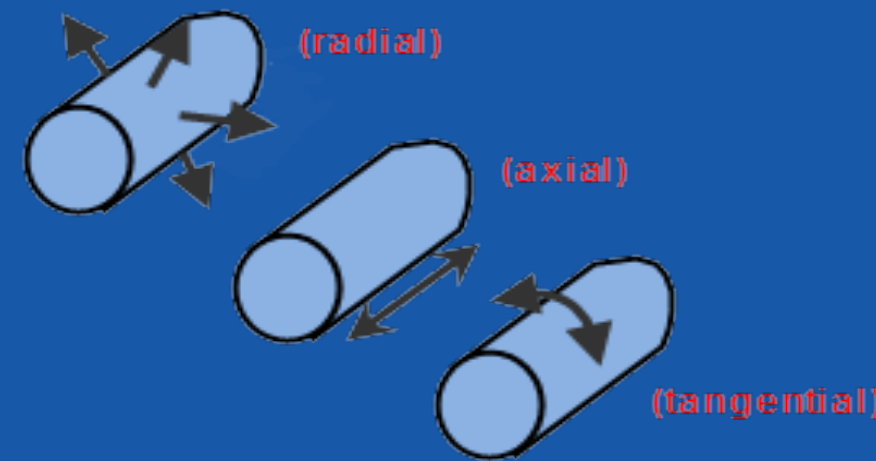
Pin

Uniformly distributed

Always Normal

Radial Axial Tangential

Multiple entities

Frictionless

No movement normal
to surface
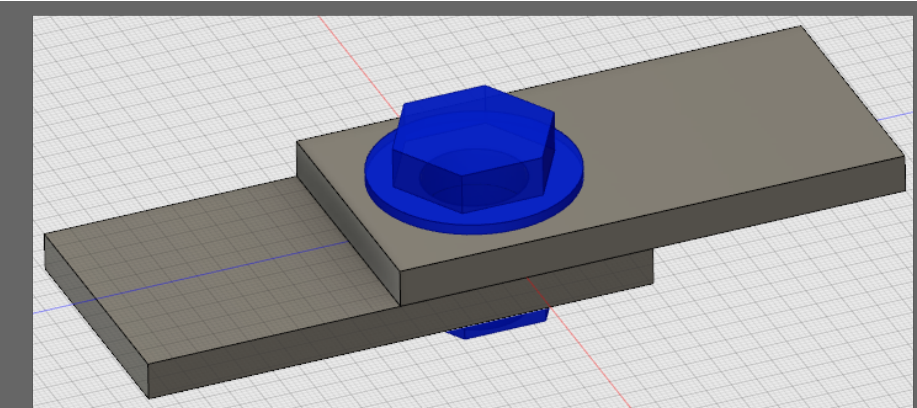
Connector

Bolt

Rigid

No geometry is created

Loads

Loads

- Goal: Specify load magnitude and type accurately

- causes displacements

- Force

  - Limit Target

  - Per entity
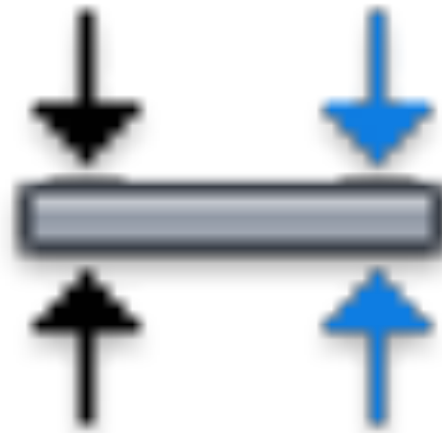
  - Normal, Angle, Vectors
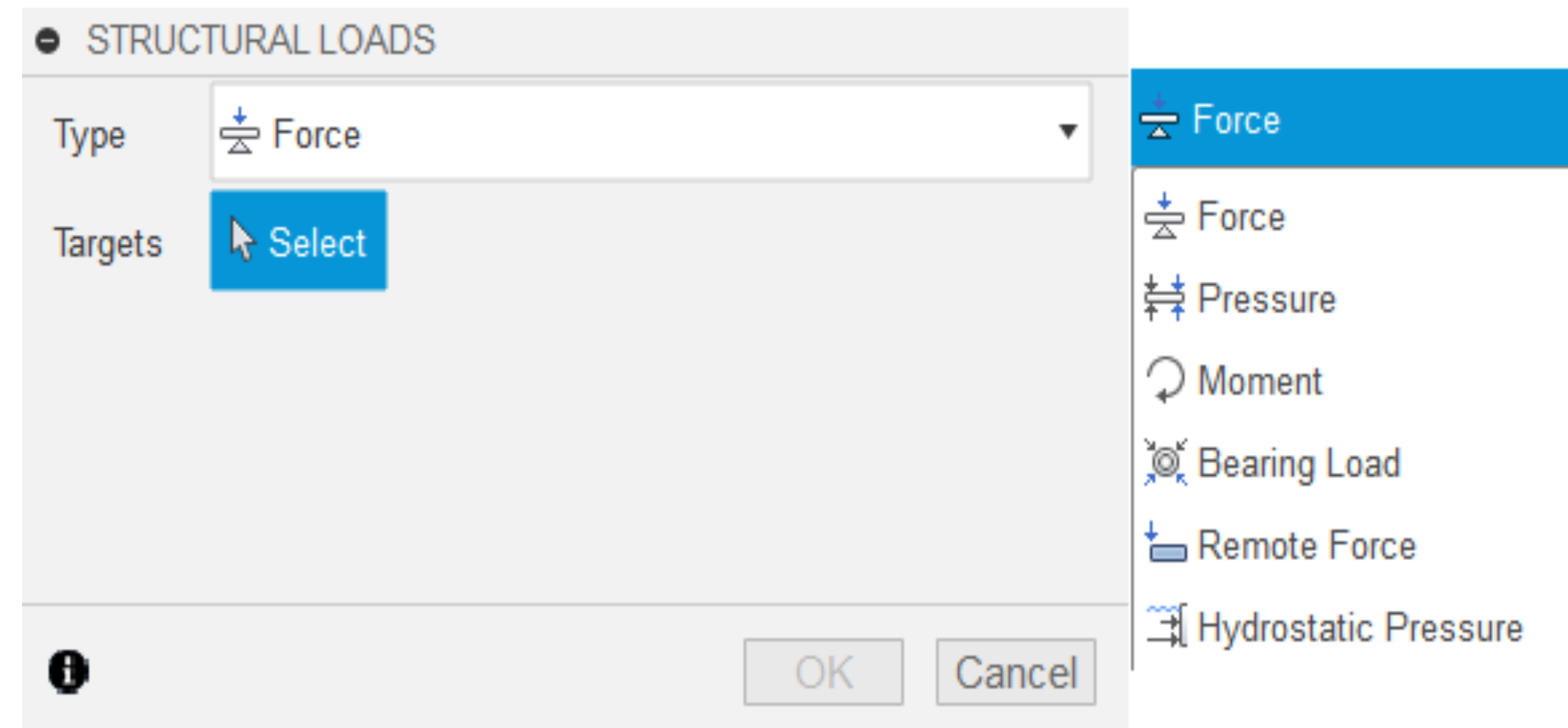
Main Load types

Force

Pressure

Moment

Bearing

Normal/Any direction

Uniformly distributed

Centroid of faces

Limit target

Always Normal

Axis passes centroid

Force per entity

Multiple entities

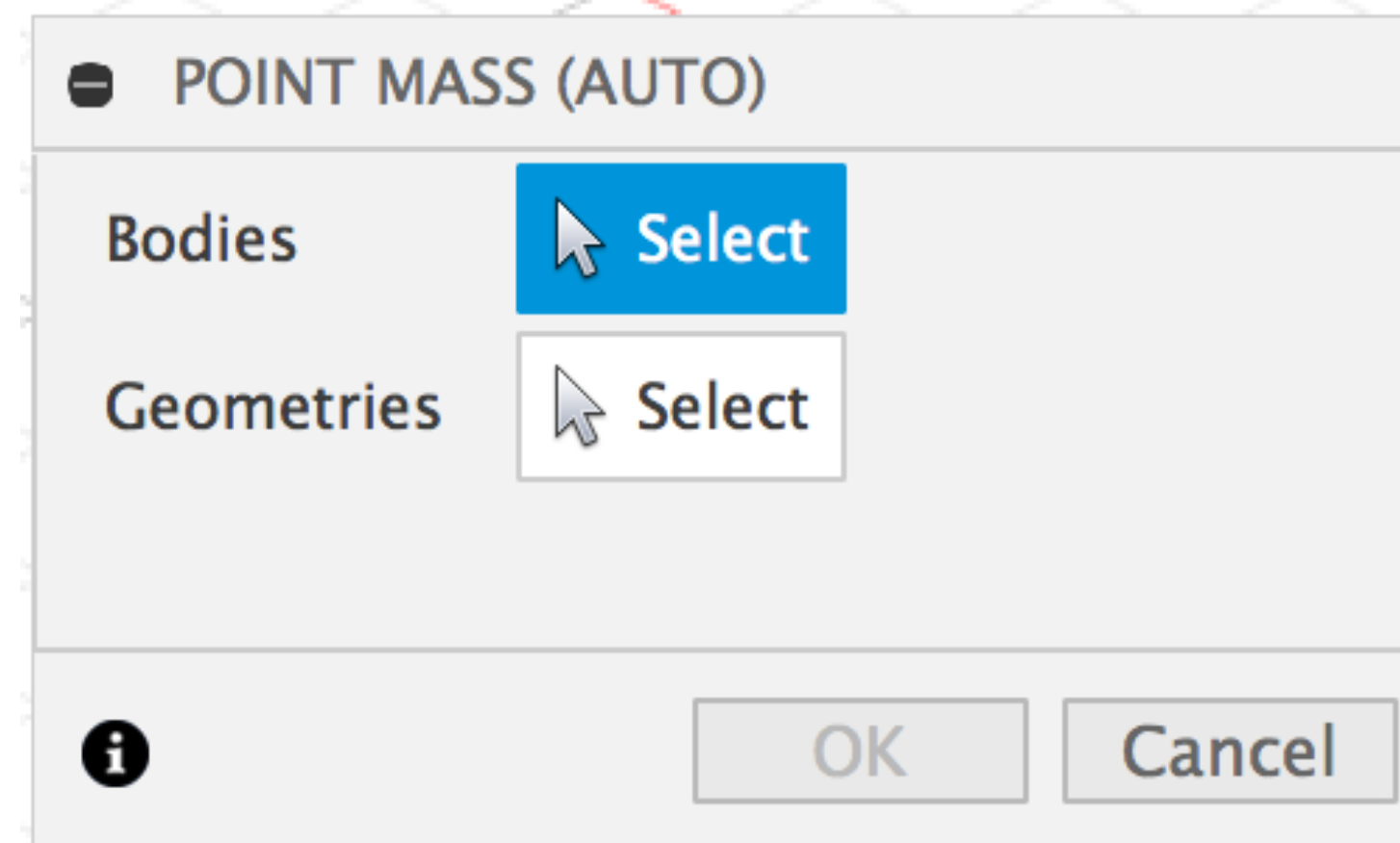Multiple entities

Multiple entities
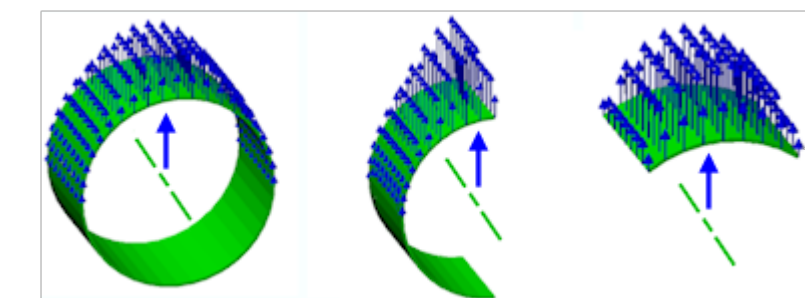
Load Cases

- Load Case 1: Effects of gravity
- Load Case 2: Effects of L&C
- Not unique to a load case
  - Suppressed components
  - Materials
  - Contacts
  - Mesh settings
  - Local mesh control
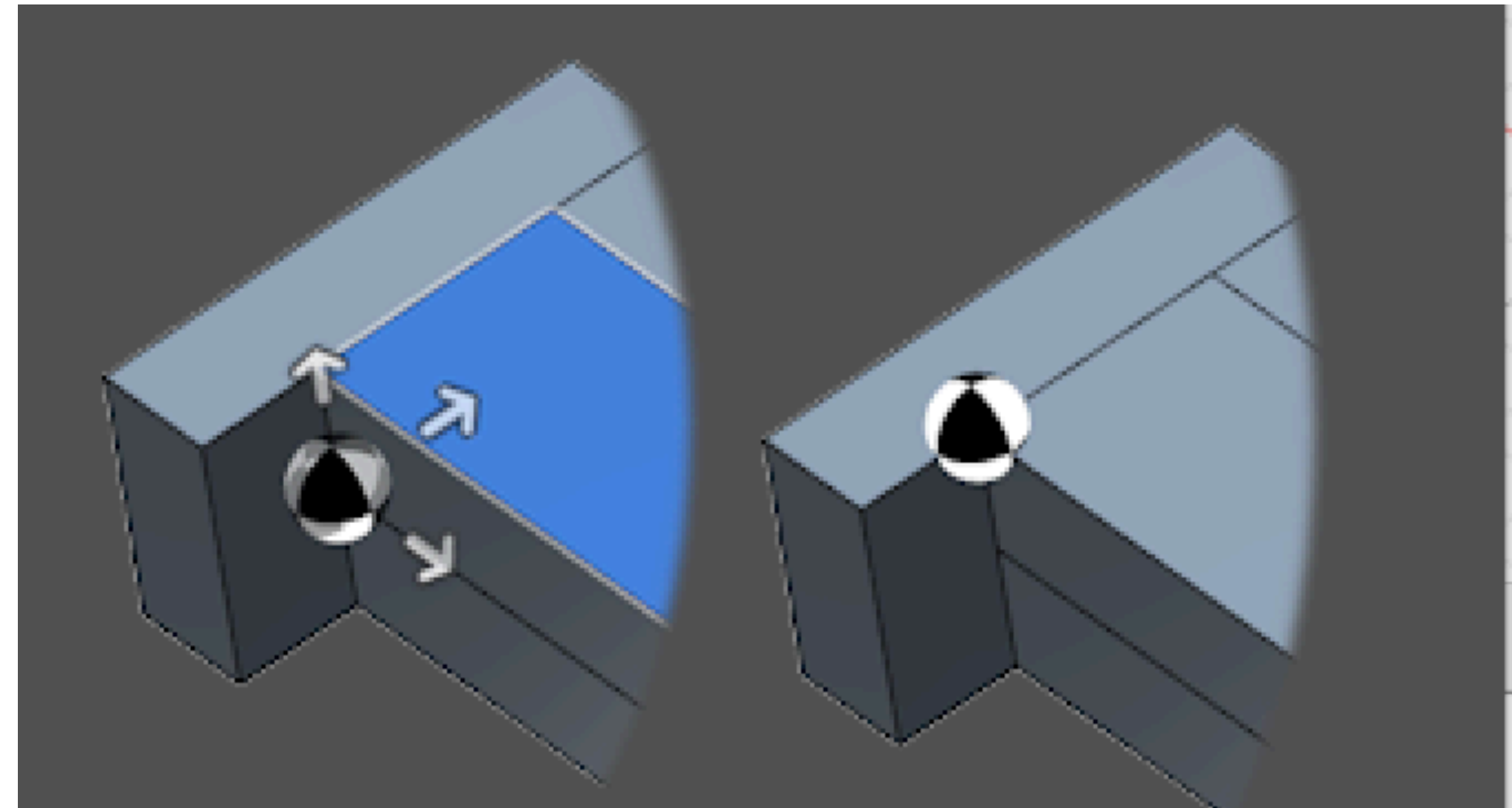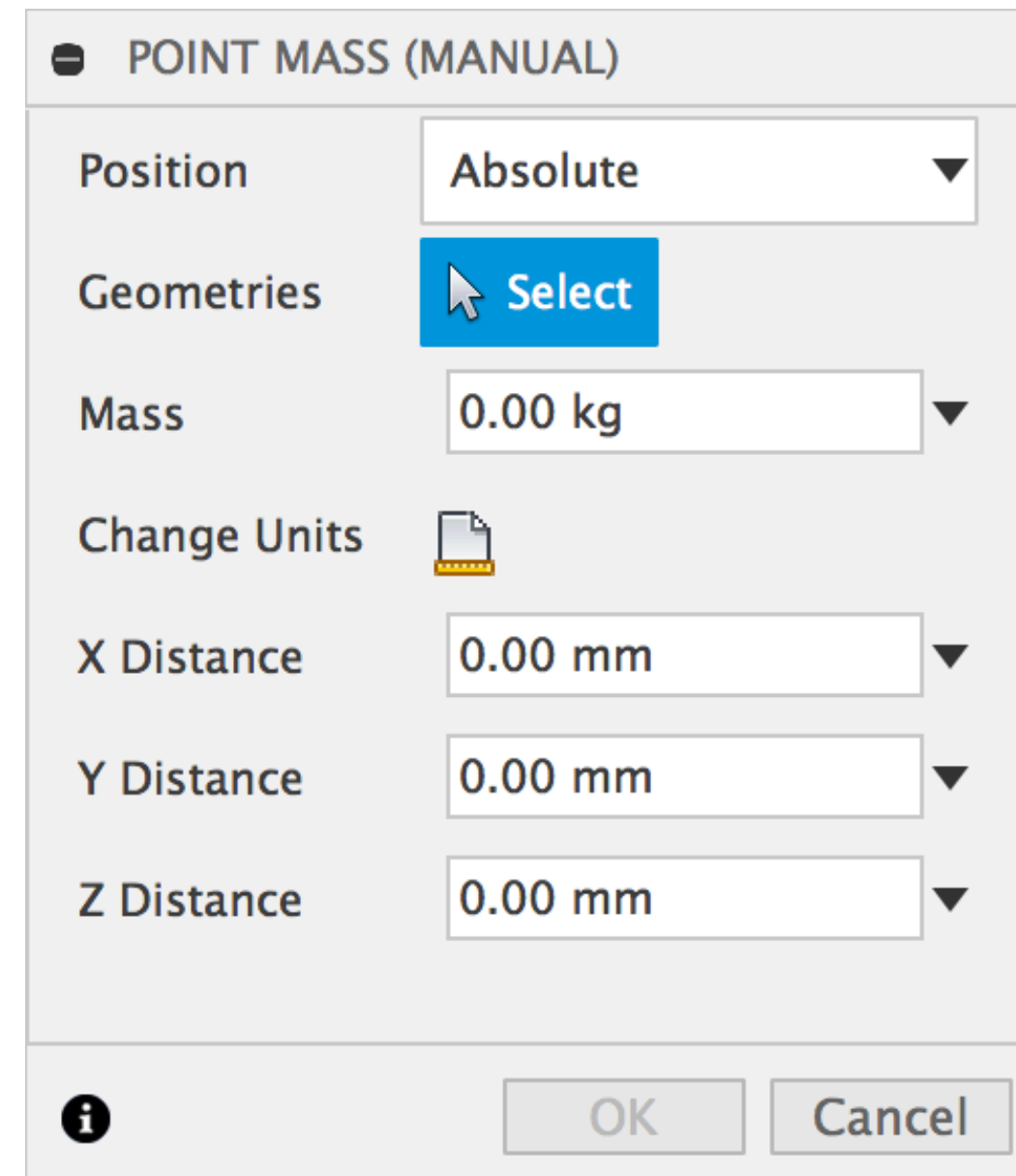 - Tip: Double-click activates a load case. Cannot have 0 LCs
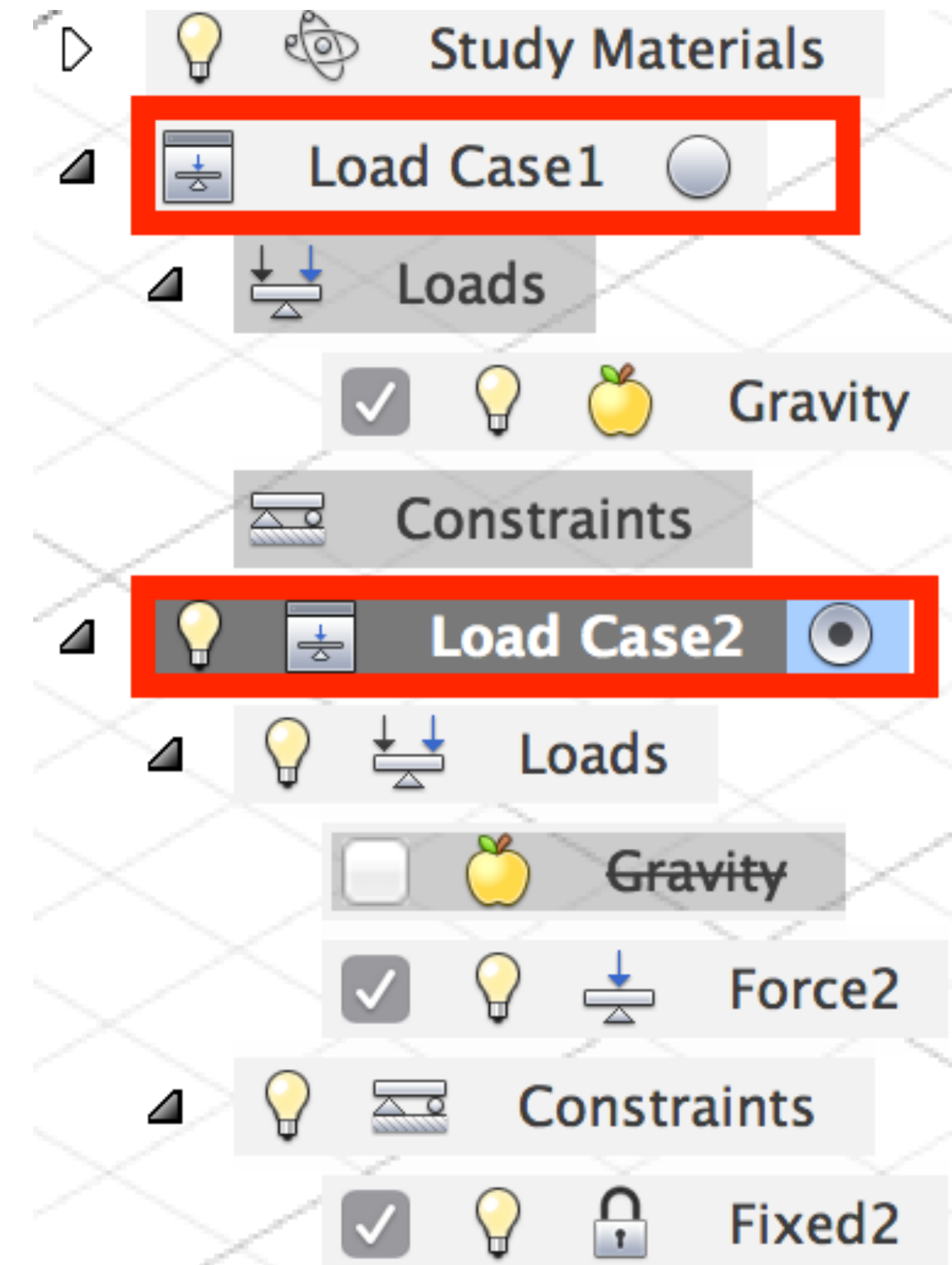
Point Masses - Auto
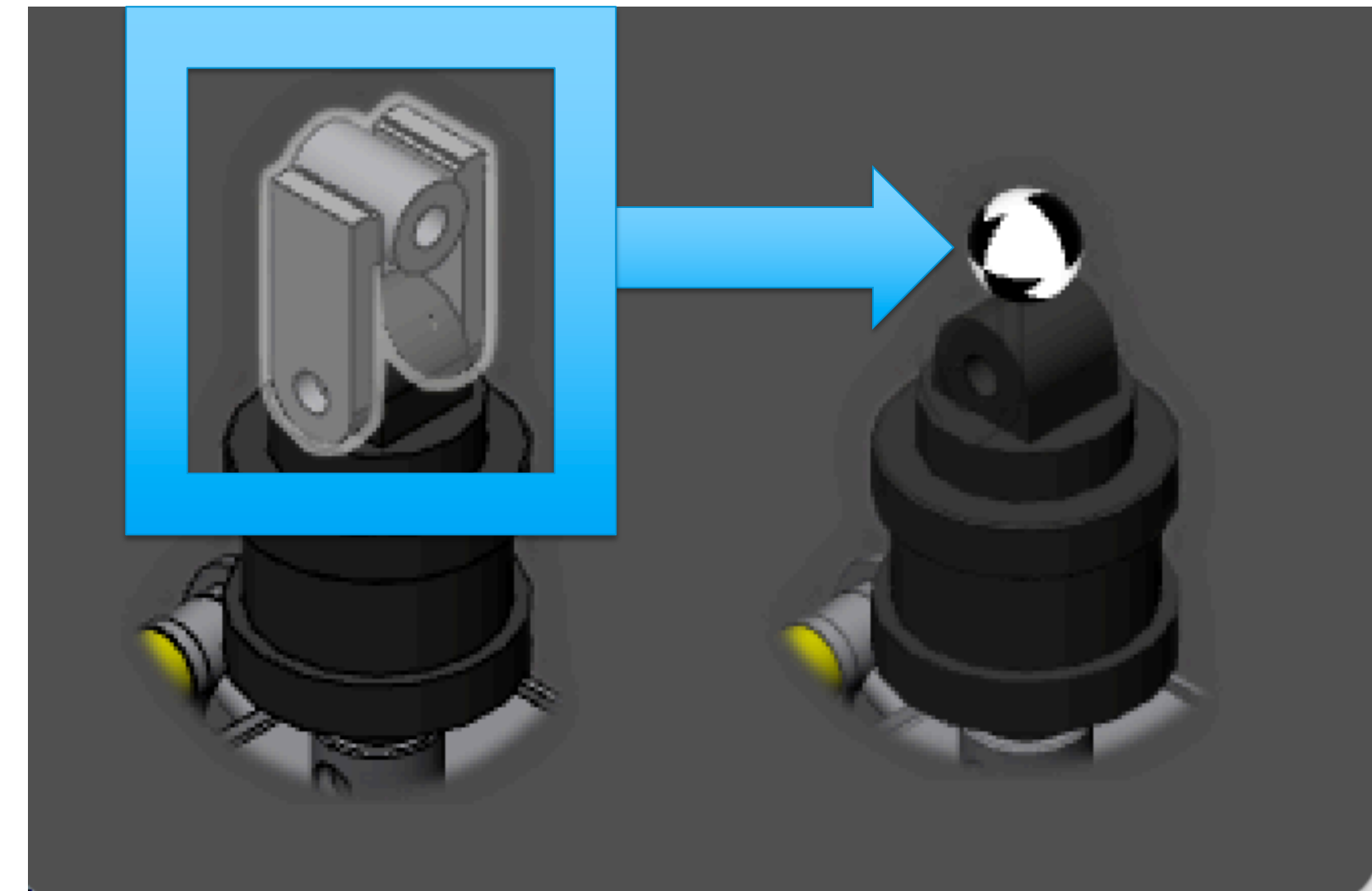
- Effects of components not present in the model

- Reduces file size, element count and processing time

- Existing solid bodies will be hidden

Point Masses - Manual

- Not based on existing geometry. 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

  o Face: Normal

  o Edge: Average vector of normal vectors @ edge

  o Vertex: Average vector of all faces @ vertex

  o <mark>Tip:</mark> 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.
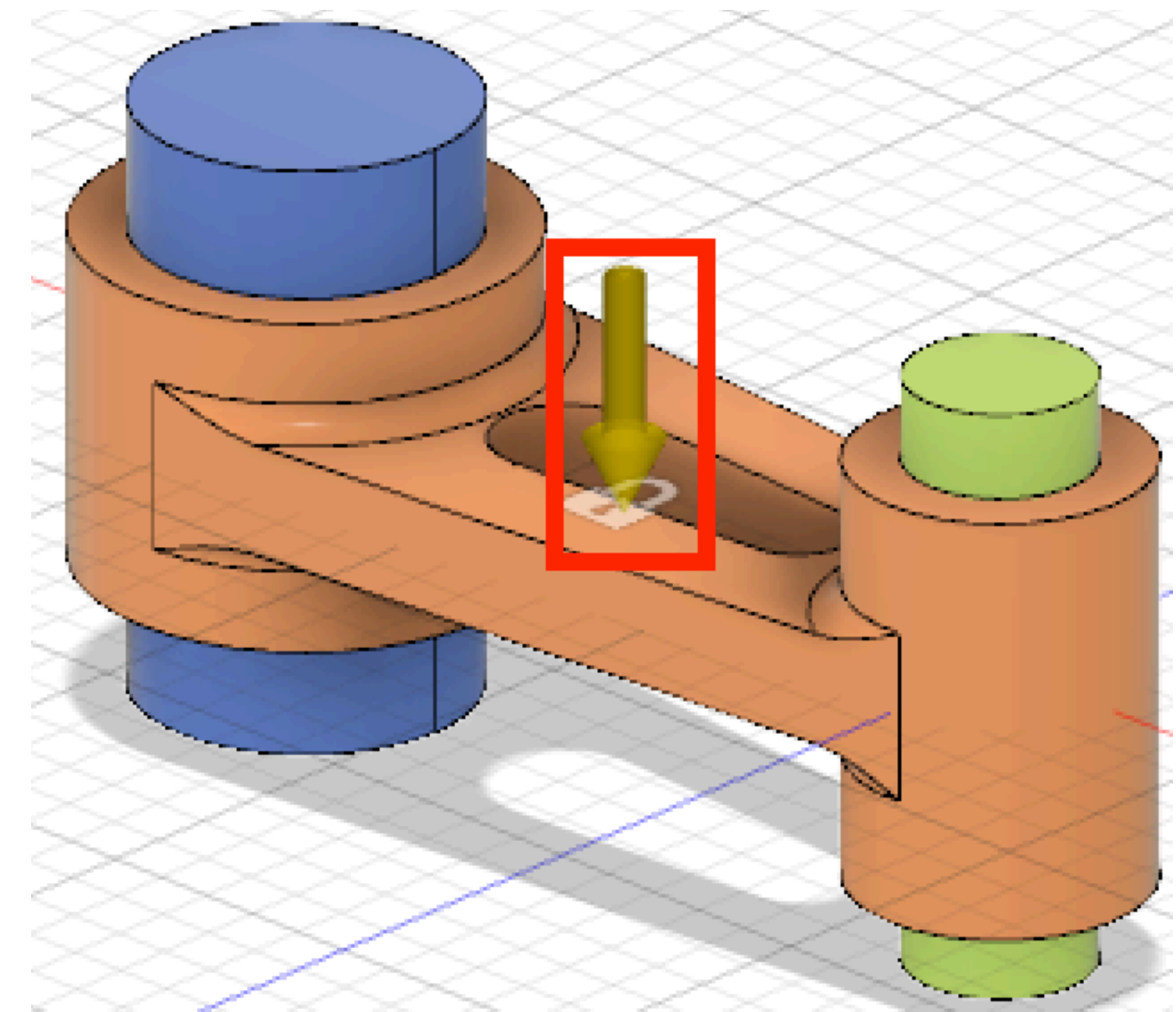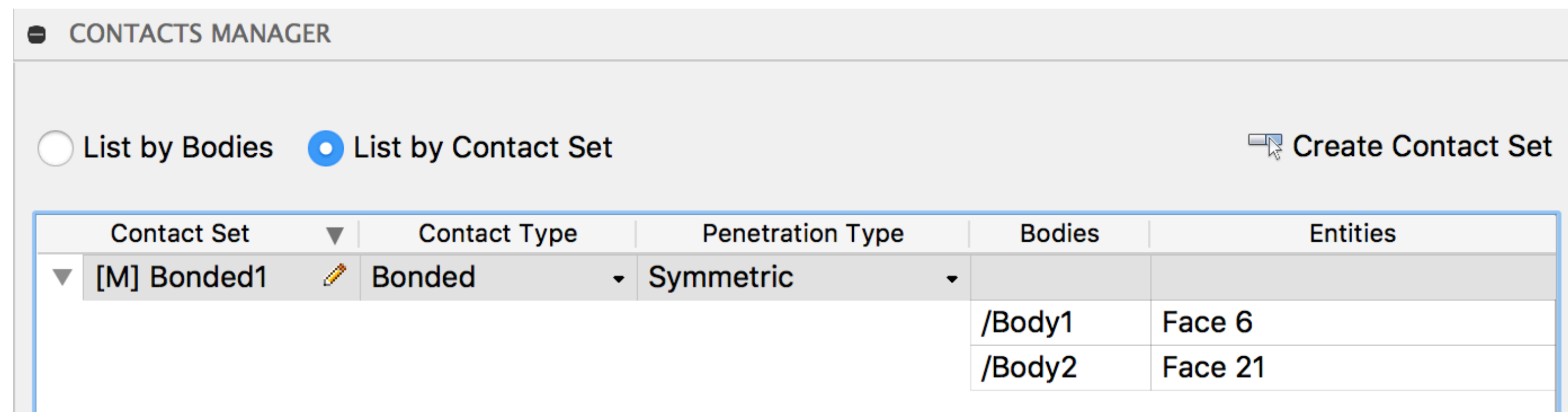
Contacts

Contacts

- Specify how 2 bodies are connected

- Has no relation to joints in assembly

- Use Contacts, Manage Contacts to edit contacts

Contact types

Bonded	Separation	Sliding	Rough

Bonded
Welded
Offset allowed

Separation
No penetration
Partial or full separation
Slide freely

Sliding
No penetration
No separation
Sliding allowed

Rough
No penetration
Partial or full separation
No sliding

Contacts

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

Meshing

Mesh quality

Average Element Size

Model-based Size   1%   10%
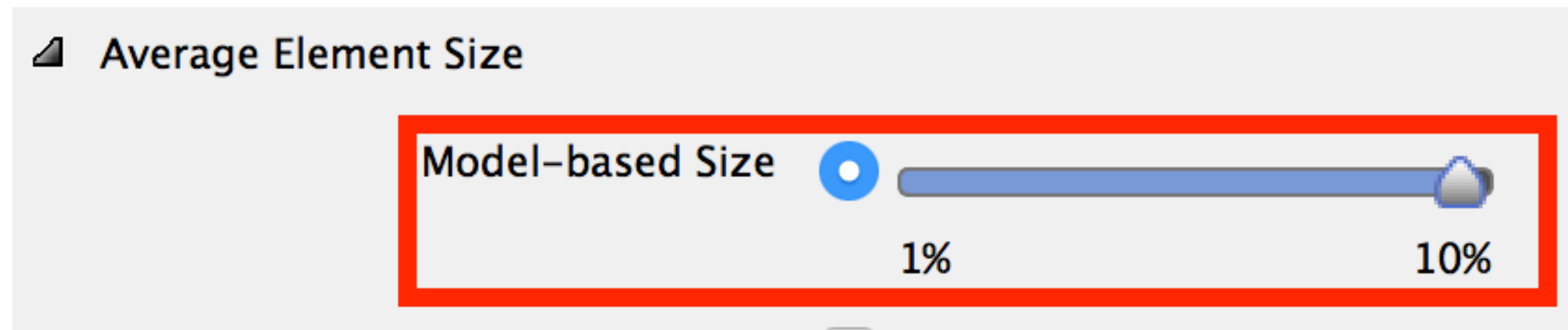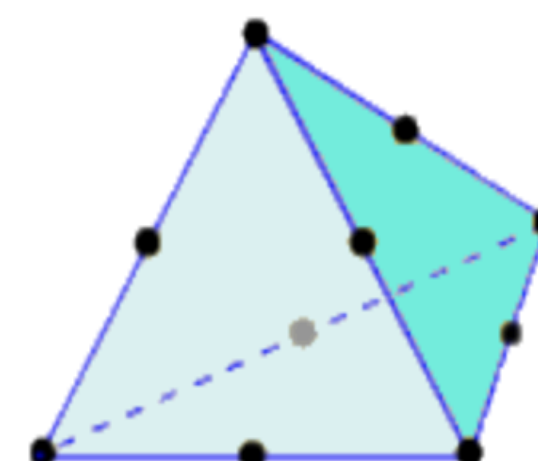
- How large is your mesh element? <mark>Tip:</mark> For Shape Optimization use a small mesh size to get reasonable results.

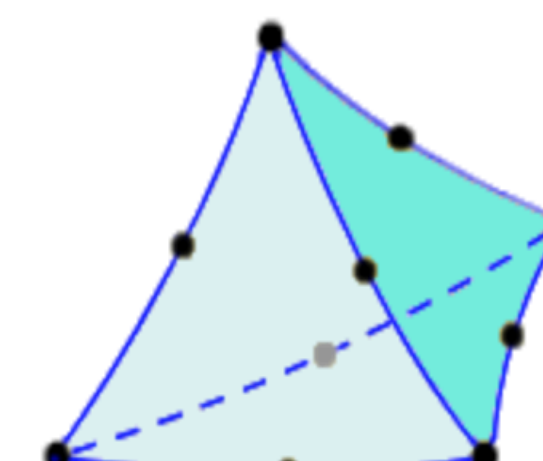**Element Order** ✓ **Parabolic**
                      Linear
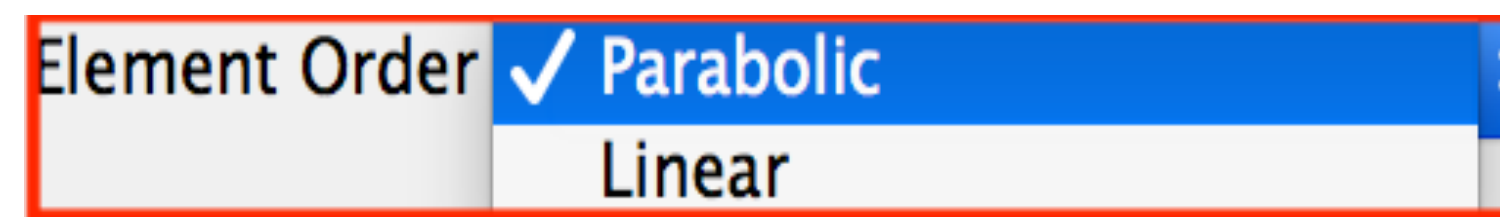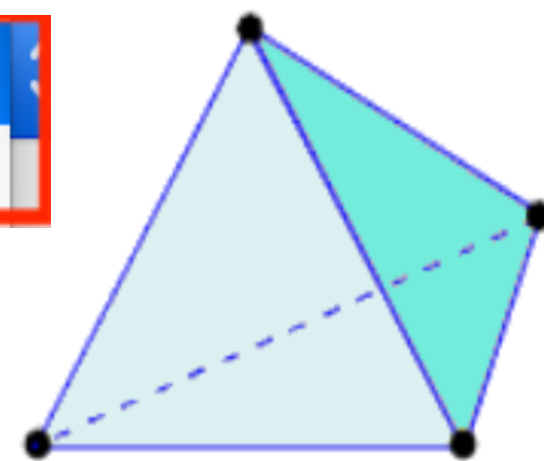
- Linear:
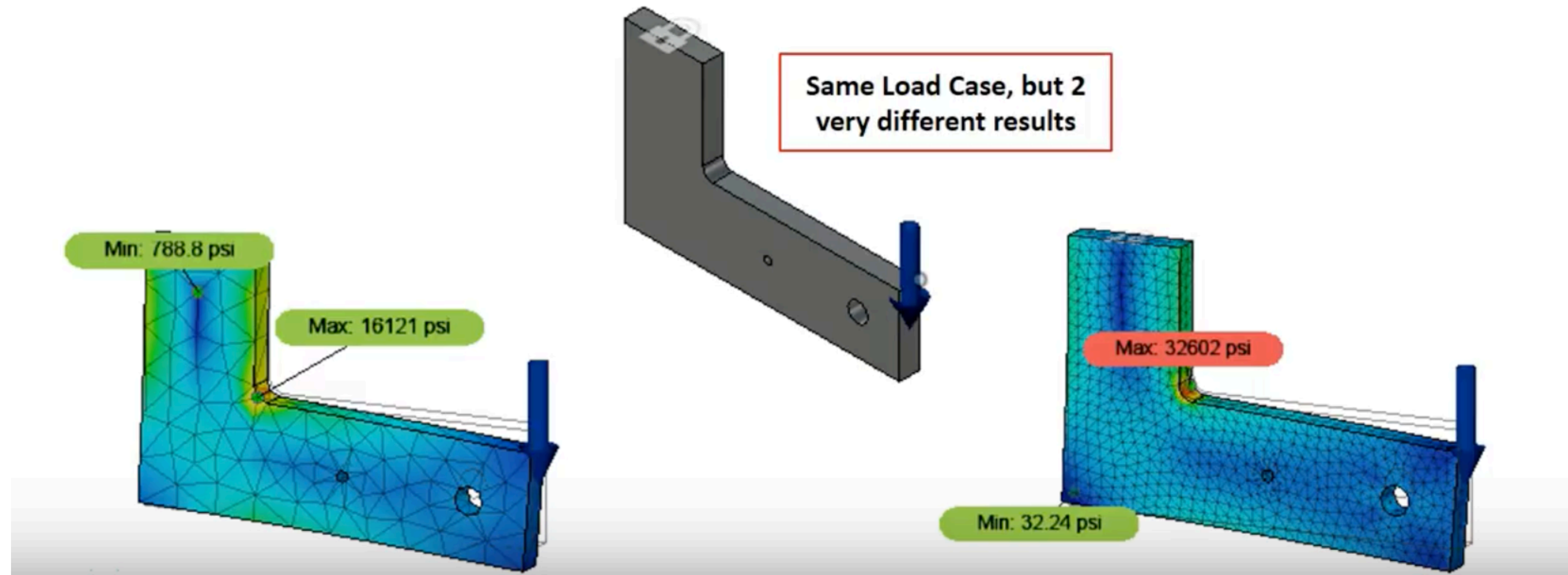
- Parabolic:

Linear Tetrahedron
(4 Nodes)

Parabolic Tetrahedron
(10 Nodes)
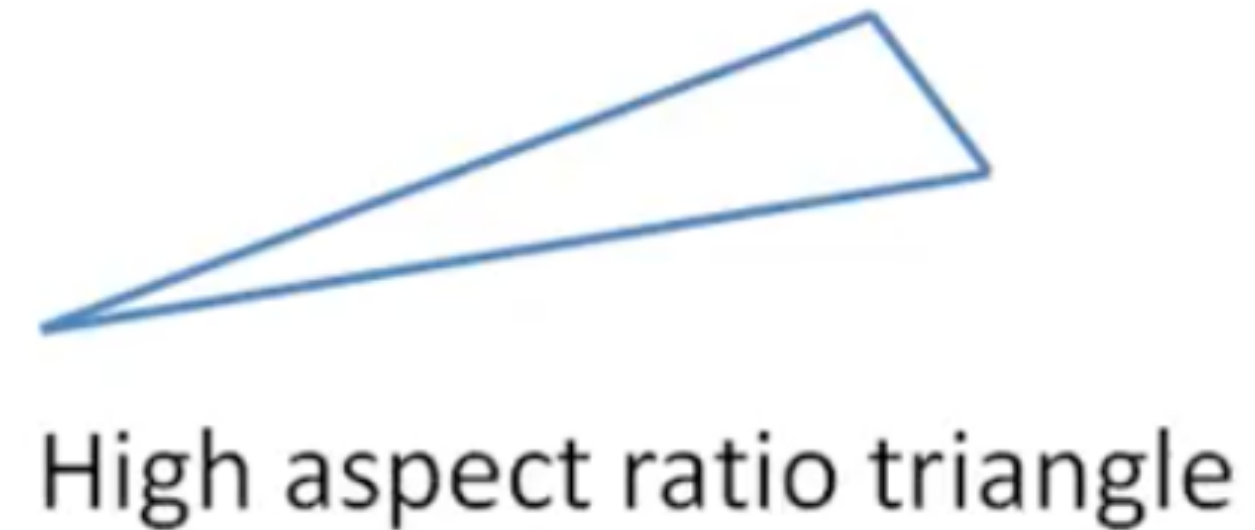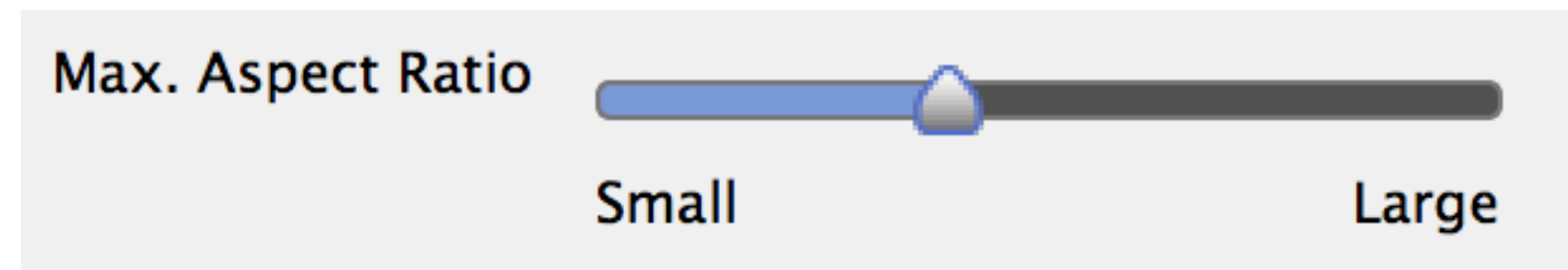
**Parabolic Tetrahedron with Curved Edges**
(10 Nodes)

Mesh Settings

Same Load Case, but 2 very different results

Min: 788.8 psi

Max: 16121 psi

Max: 32602 psi

Min: 32.24 psi

- <mark>Tip:</mark> Good mesh extremely important for good results. [Video Link](#)

Mesh Settings

- Aspect ratio

Max. Aspect Ratio | Small | Large
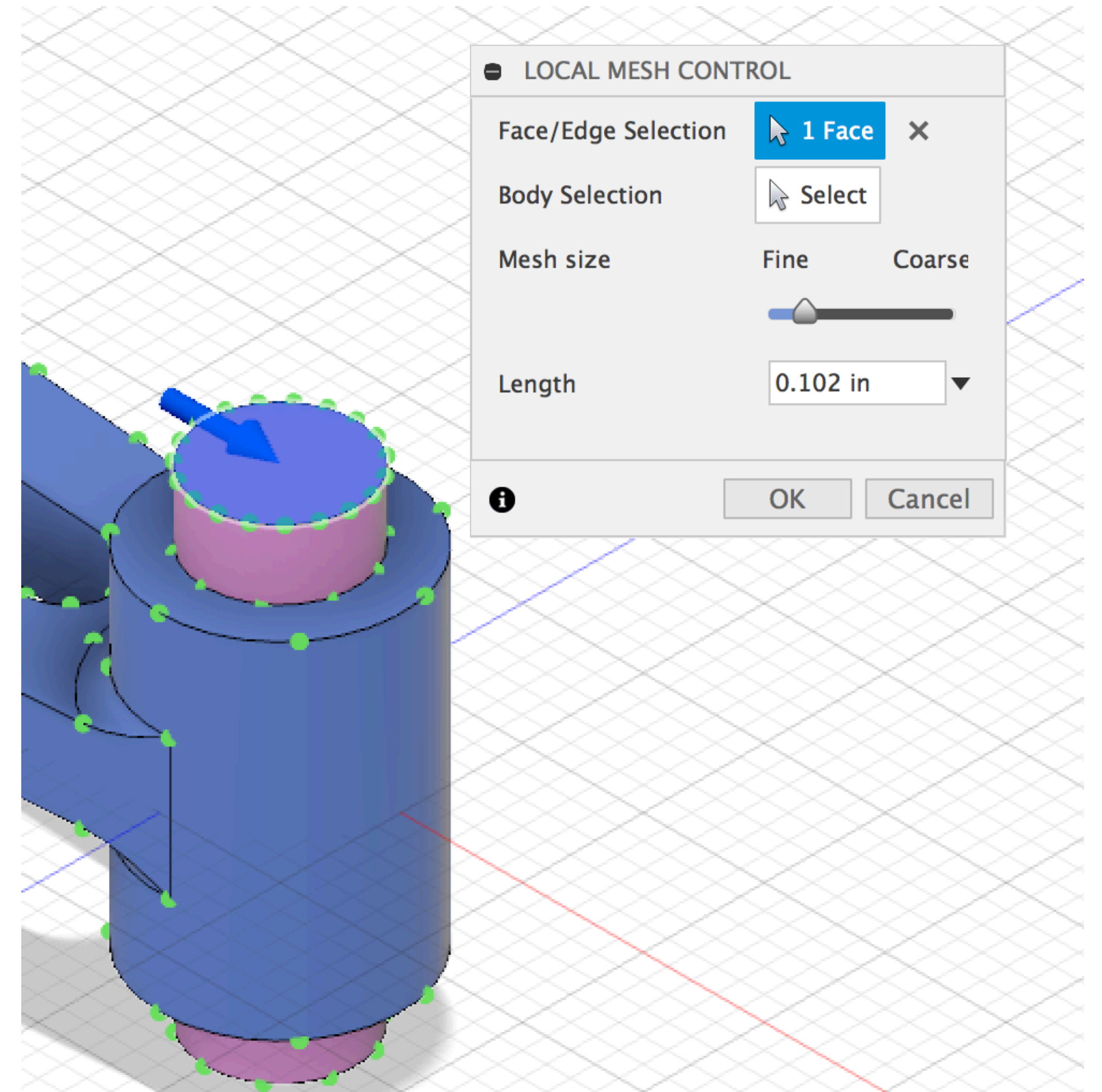
Aspect ratio = 1   High aspect ratio triangle

- Maximum turn angle

- Tip: Lower the turn

angle smoother the circle
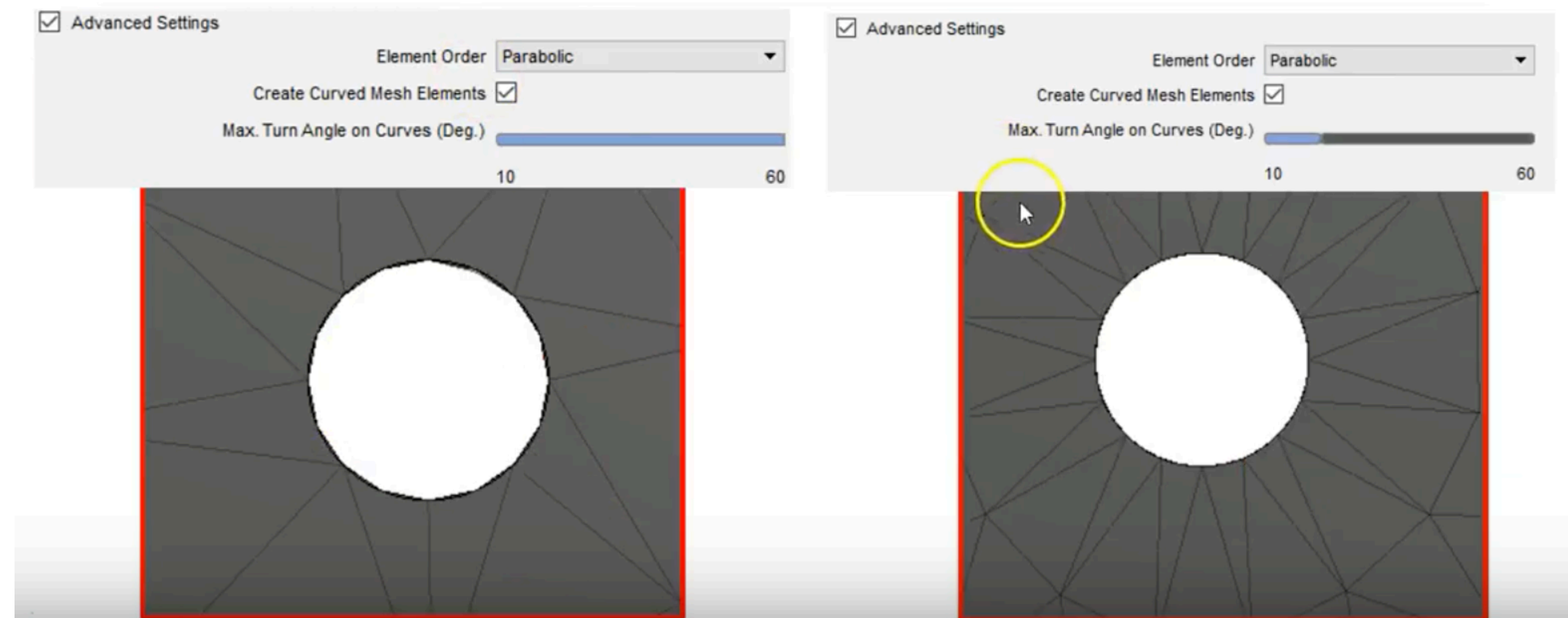
Local Mesh control

- Mesh needs to be fine in localized regions

- Faces, edges or bodies

- Adaptive mesh refinement
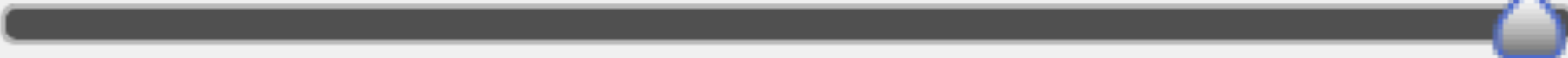
Adaptive Mesh refinement

- Static, Modal, Thermal and Thermal Stress

Adaptive Mesh refinement

- **Maximum # of mesh refine**ments ==Tip:== <8 gives good results

- **Results Convergence Tolerance**:% change between 2 iterations <= tolerance it stops

- **Portion of elements to refine(%)**

  o X%: Top X% w.r.t critical result are refined

- **Frequency Mode**: Modal frequencies only. Basis for refinement

Adaptive mesh refinement example
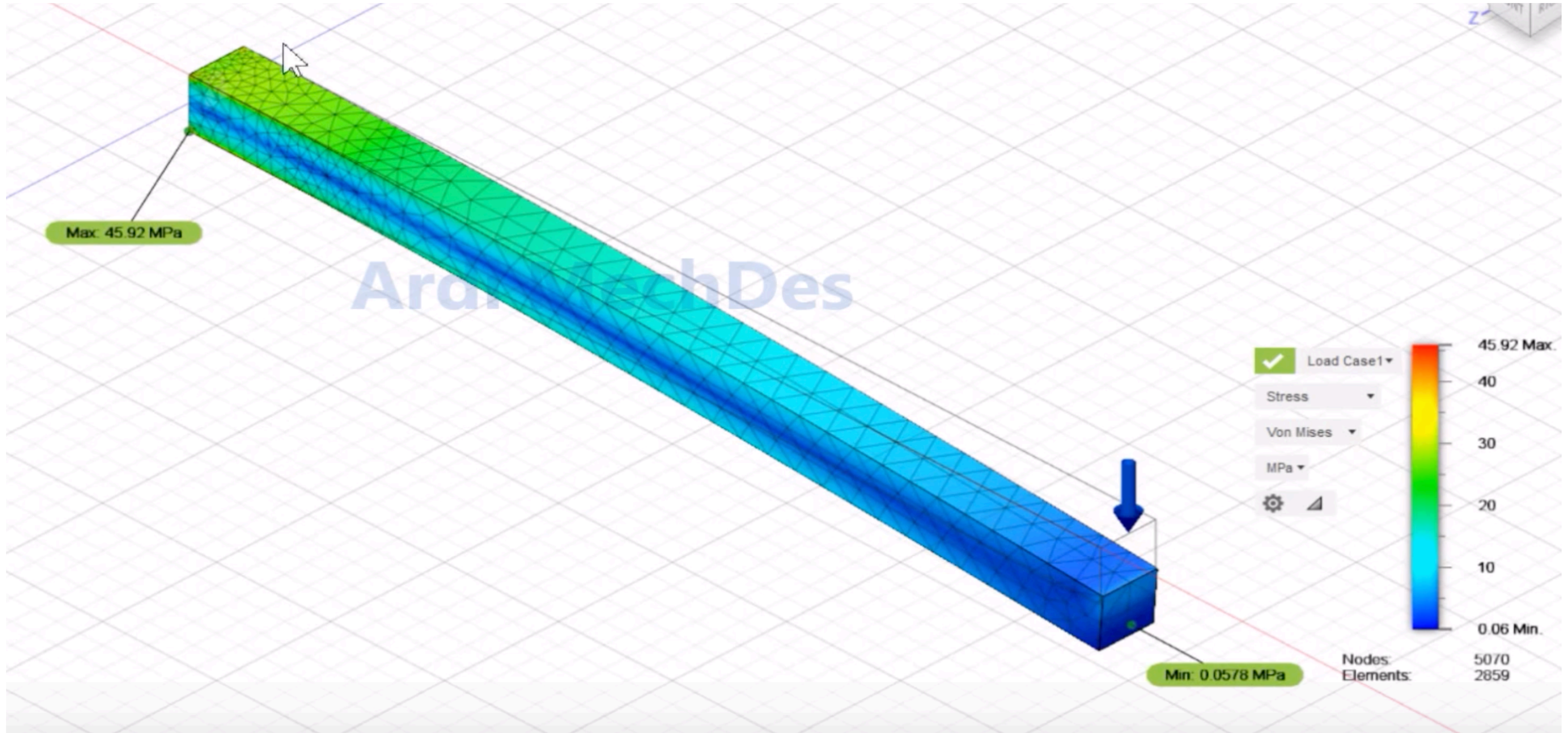
Pre-Check, Solve

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 non-linear material for linear analysis

Solve dialog

- Local only 1 study. Synchronous

- Cloud: Multiple. Asynchronous

- Studies that cannot be solved can be hidden

- **Tip:** To resolve a solved study, uncheck and check the checkbox next to a load or constraint

- **Tip:** Even though you solve locally, your results are automatically uploaded to cloud after the solve.

Cloud Credits

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

Results

Results for baseline accuracy

- **Static Stress**

  o Von Mises

  o 1st principal

  o 3rd principal

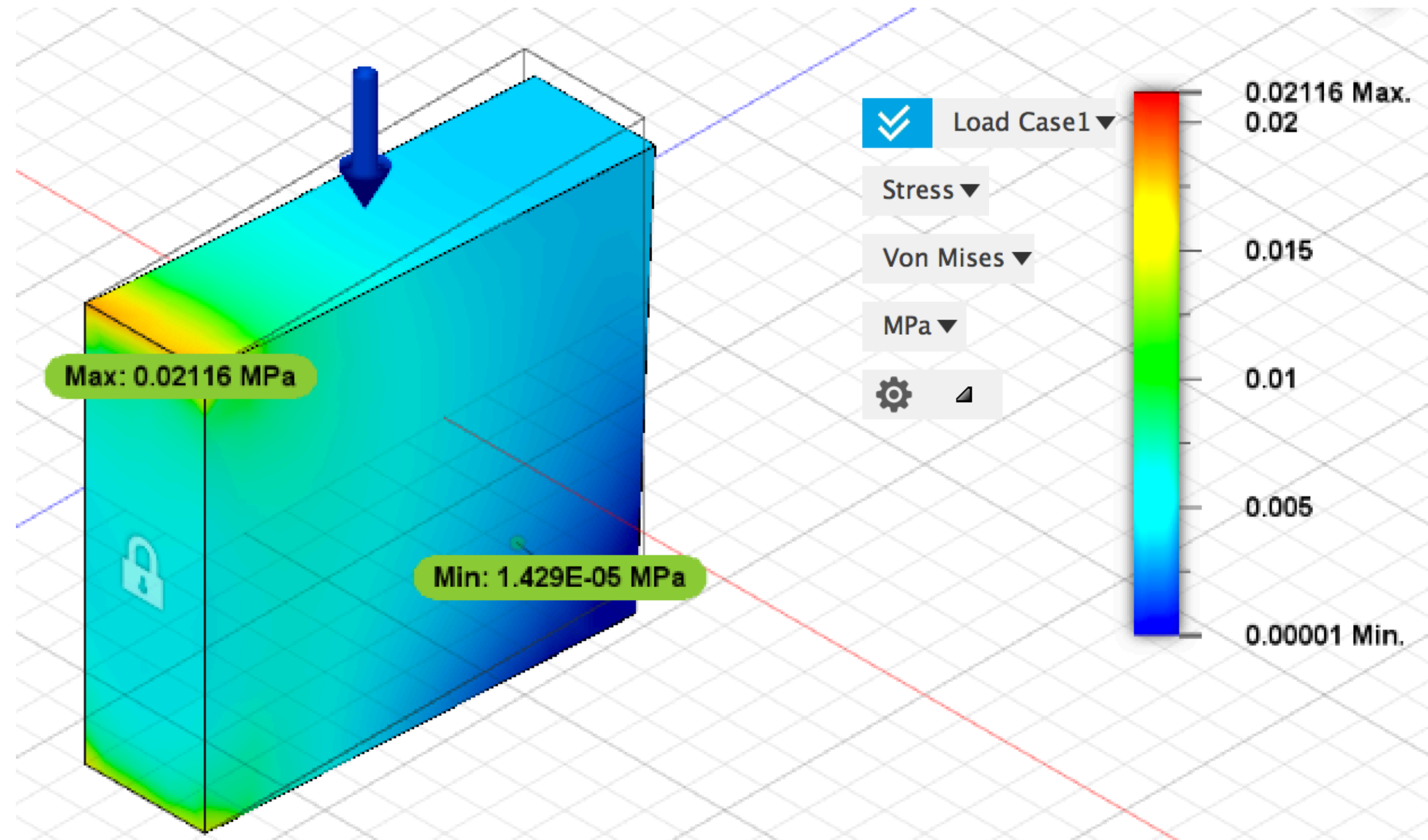 o Displacement, total

- **Thermal**

  o 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
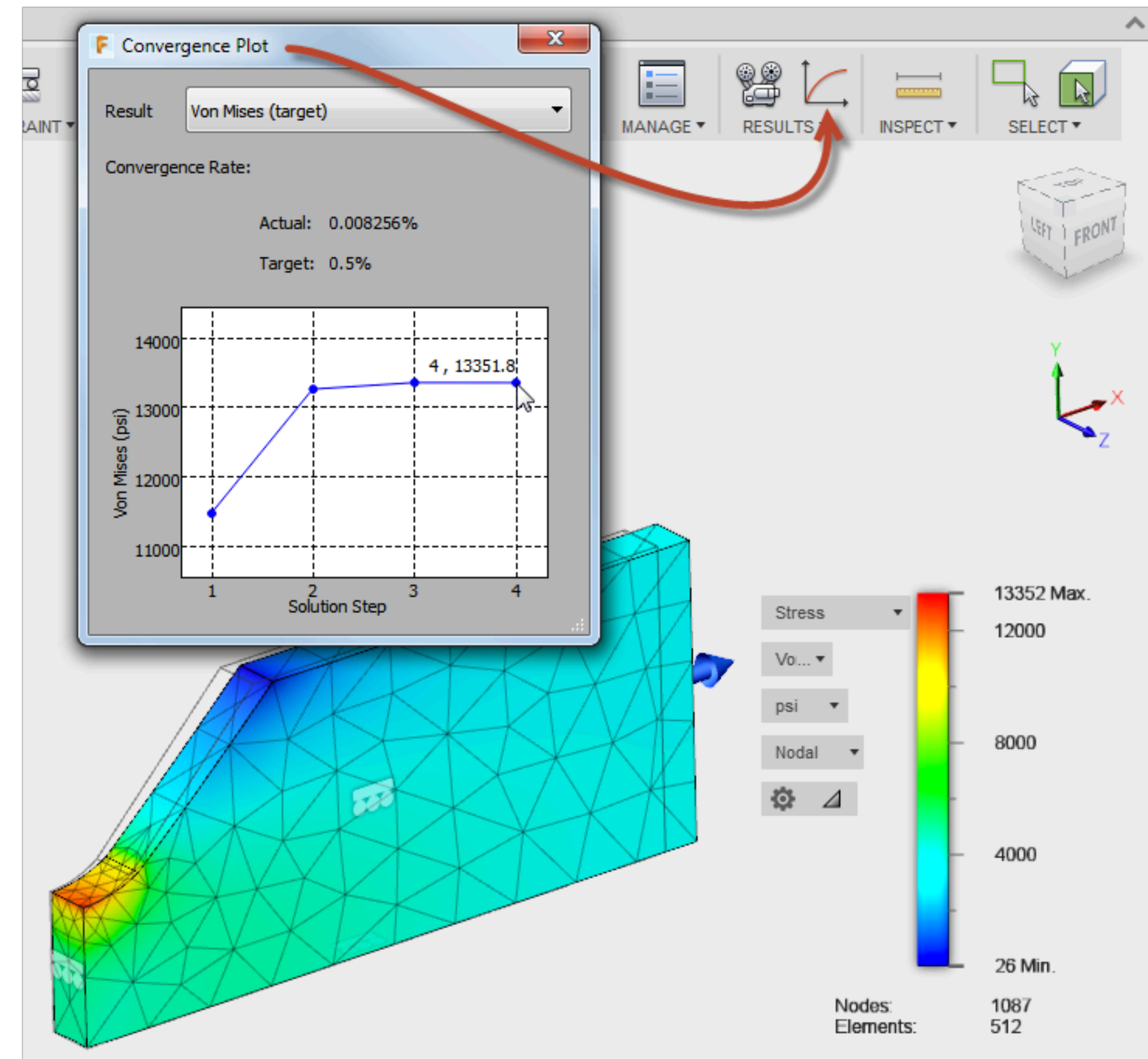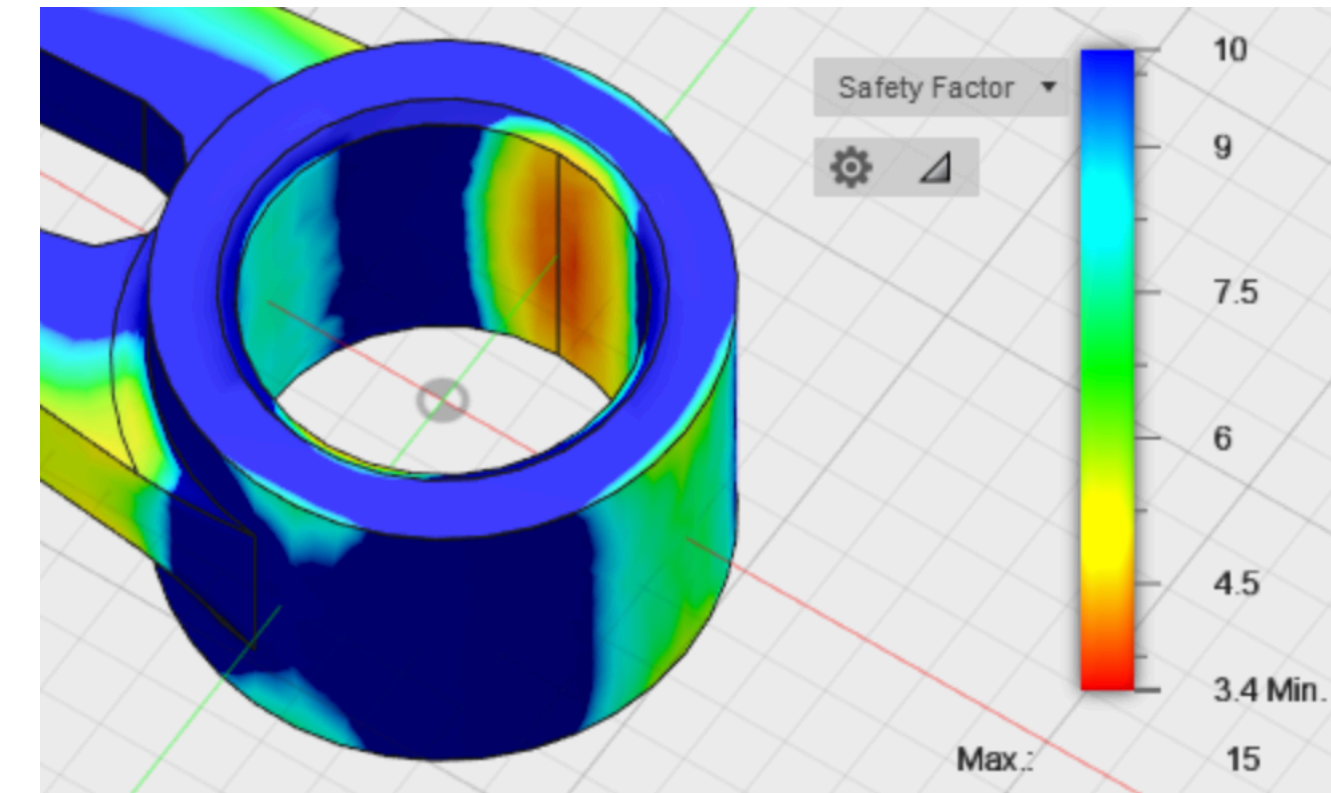
Results

- 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
Shape Optimization	Load path criticality
Structural buckling	Total Displacement, Displacement X, Displacement Y, Displacement Z

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
![red warning]	Insufficient	Bends/breaks.	Material > YS Reinforce weaker areas NLSS for bending
![orange warning]	Marginal	Transitional area	Investigate SF Mesh convergence
![green check]	Sufficient	Good	Run other studies Slender->buckling
![blue check]	Excessive	Over-engineered	Material < YS Reduce weight, SO

RESULTS DETAILS

Actual Minimum Safety Factor **15.00**

The design appears to be over-engineered for the current analysis criteria. Ensure the Safety Factor Targets meet the standards of your company, application and industry.

▼ **Safety Factor Targets**

Default Values ☑

Upper Target | 6.00

Lower Target | 3.00

▼ **Recommendations**

1. Use Shape Optimization to remove unnecessary material.

2. Try testing weaker, less expensive materials to reduce cost.

Show strongest areas of design

Deformation Scale | Adjusted
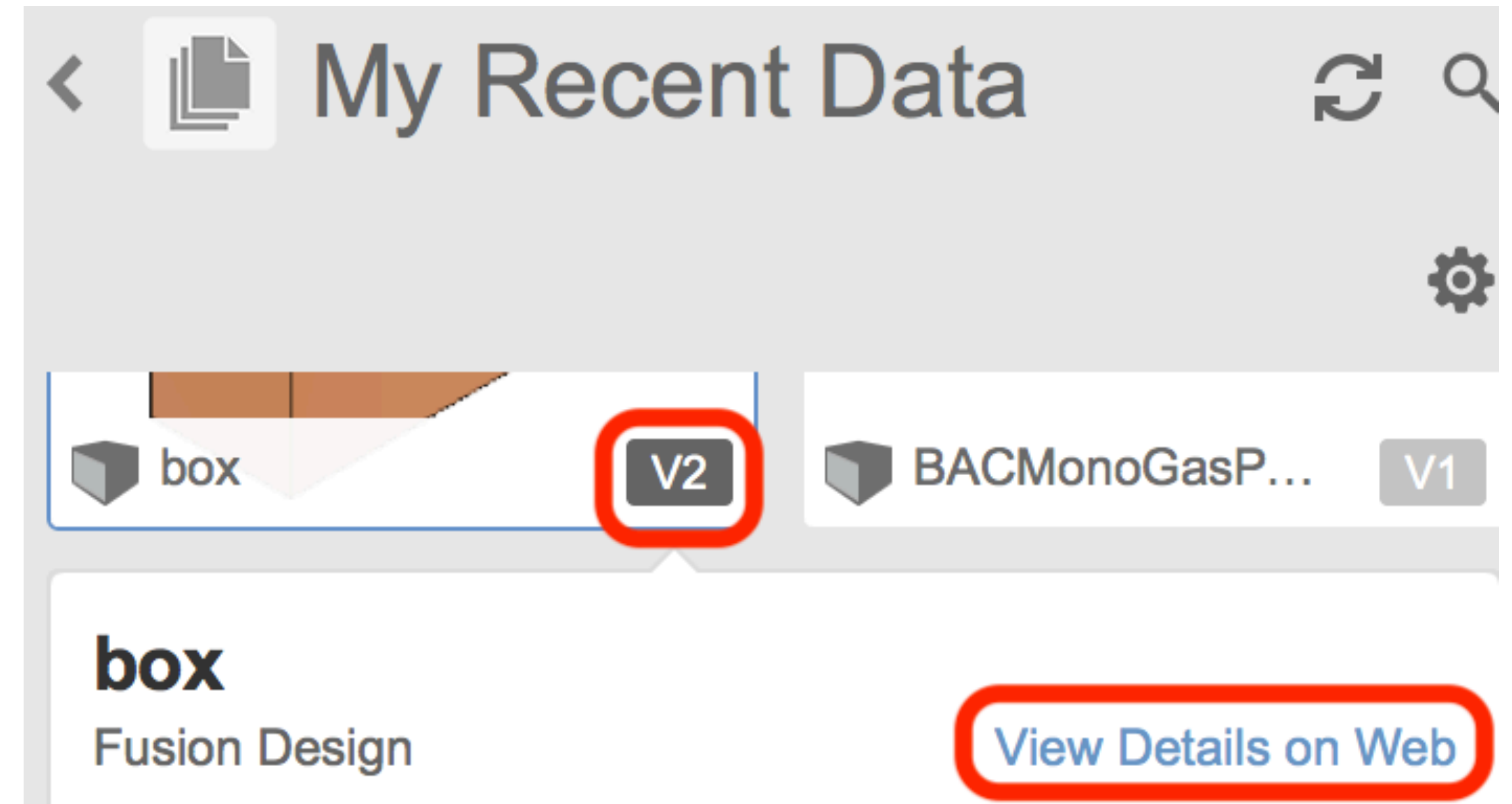
Don't show this automatically ☐

Close

Results sharing - Reports

- Report generates reports with images

- Customizable

- NLSS result charts can be included for steps

- <mark>Tip:</mark> Use Dynamic Content (Javascript) option which provides collapsible sections

Results sharing – A360

- Generate results and save the file

- Data Panel, click on versions v2
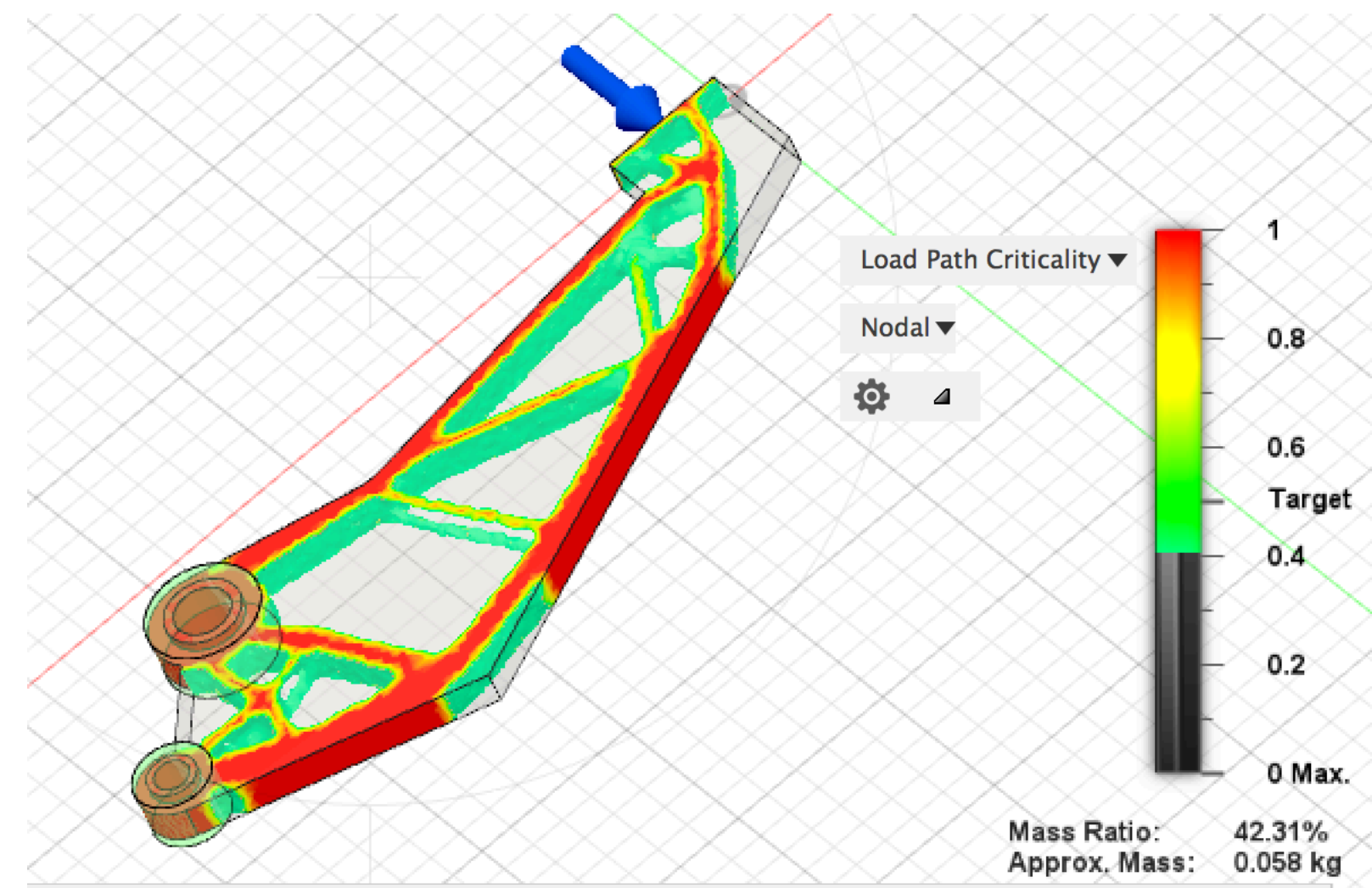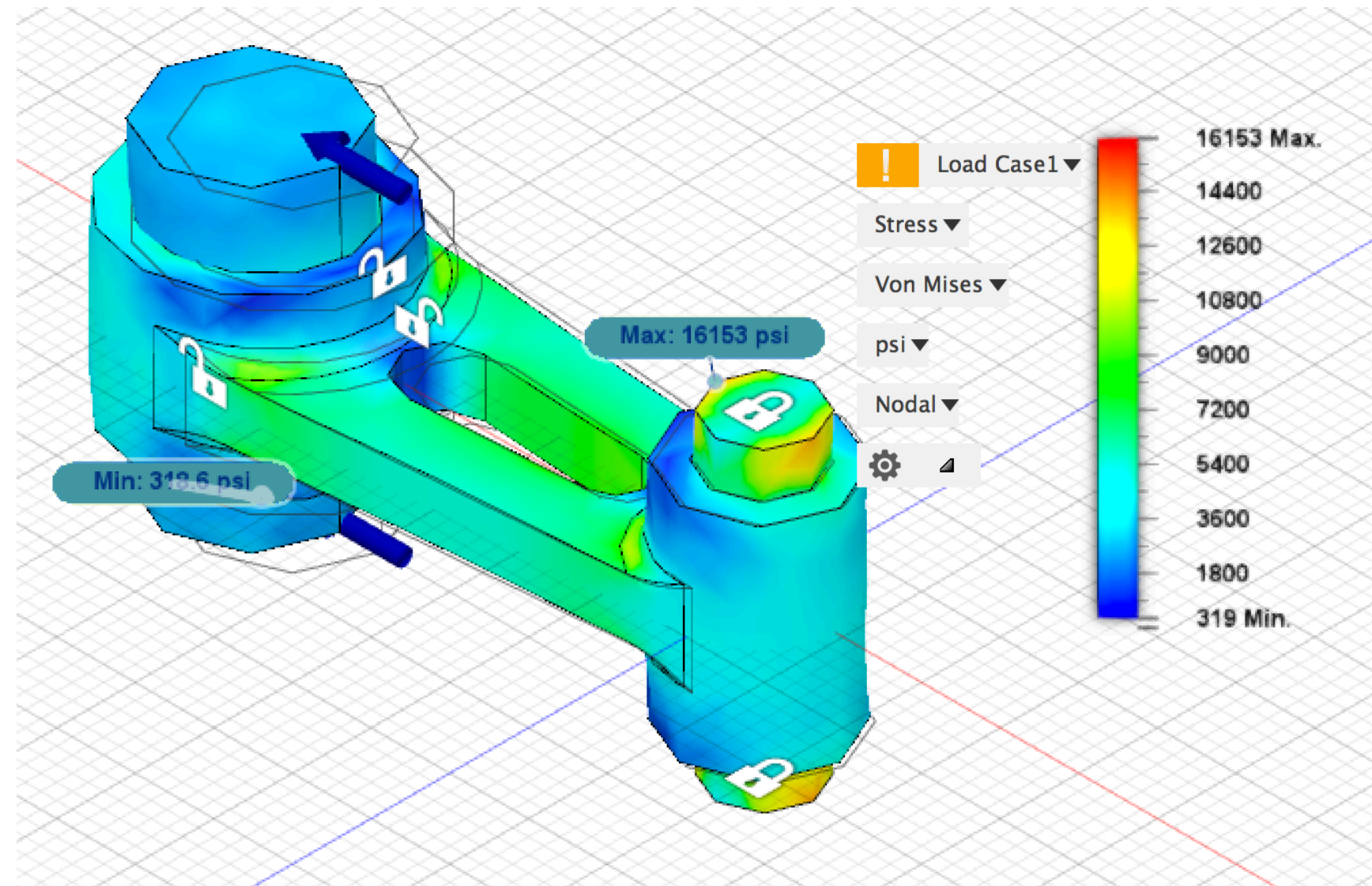
- Click on "View Details on Web"

- In A360 click on Share

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
- Video

Demo

Links

- [About Fusion 360 Simulation Learning](): Main learning link

- [How to](): Specific and general steps for managing studies

- [Hands-on exercises](): Tutorials

AUTODESK
Make anything™