Fusion 360 simulation for dummies

Harv Saund

Technical Specialist – Fusion 360

Wasim Younis

Simulation Sales Manger - Symetri

Join the conversation #AULondon #AU2018

AUTODESK. UNIVERSITY

About the Speakers

Harv Saund

Technical Specialist – Fusion 360, Autodesk UK

harv.saund@autodesk.com | @harv_saund

Joined Autodesk in 2016.

Prior to Autodesk, 5 years as a Solidworks technical consultant, 2 years spent in Race Engine Development & a degree in Motorsport Engineering

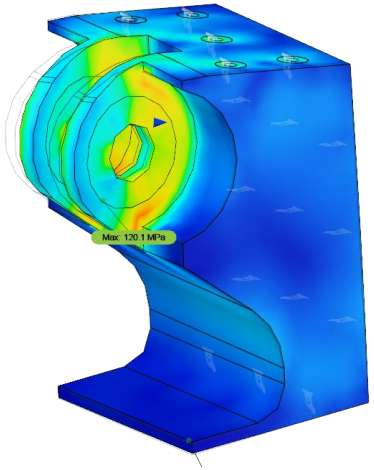
Product focus is the Fusion 360 platform incl. CAM, Eagle, FEA, Generative Design & Library.IO

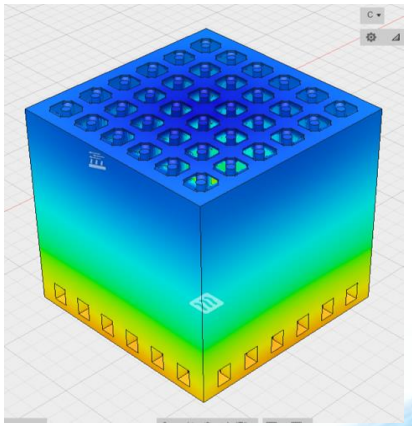
Wasim Younis – Symetri

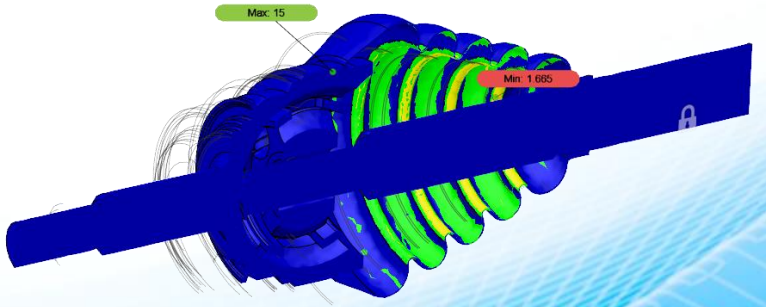
Wasim Younis has more than 20 years of experience in the manufacturing field and has been presenting at Autodesk University for more than 7 years. He has been involved with Simulation software since Autodesk, Inc., first introduced it. He is well known throughout the Simulation community, and has authored the Up and Running with Autodesk Inventor Simulation books, available worldwide.

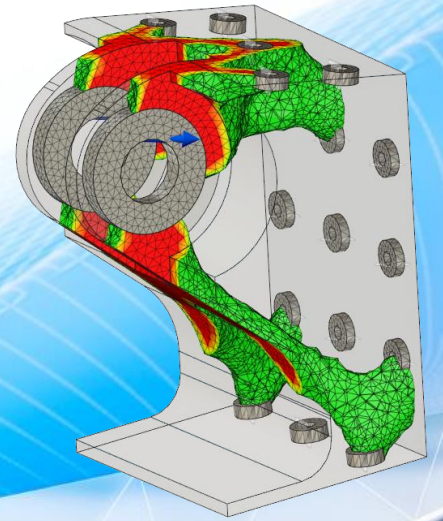
Class Agenda

- What is FEA?
  - Mesh
  - Contacts
  - Constraints
  - Loads
- Static Stress – HANDS ON
- Thermal – HANDS ON
- Non Linear – HANDS ON
- Shape Optimisation – HANDS ON
- Summary





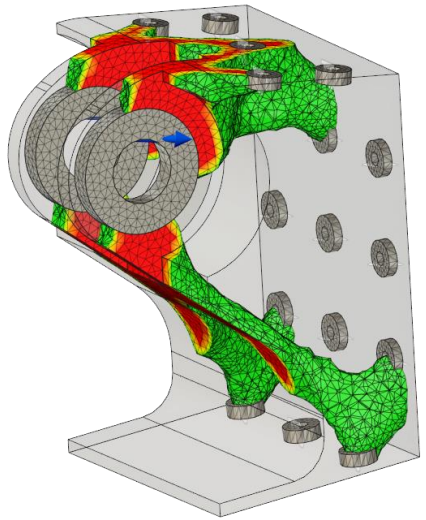


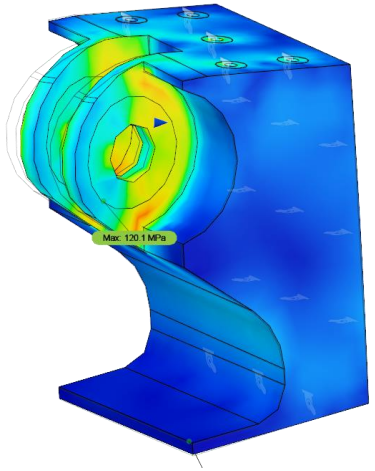


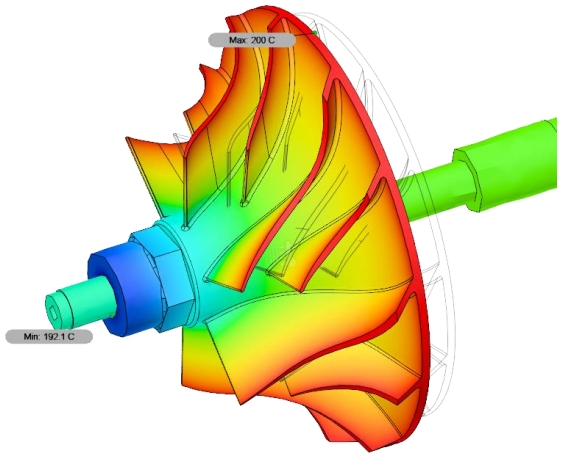
What is FEA?

What is FEA?

Finite Element Analysis (**FEA**) is a computer based method of simulating / analyzing the behavior of engineering structures and components under a variety of conditions. It is an advanced engineering tool that is used in design and to augment, complement or replace experimental testing.

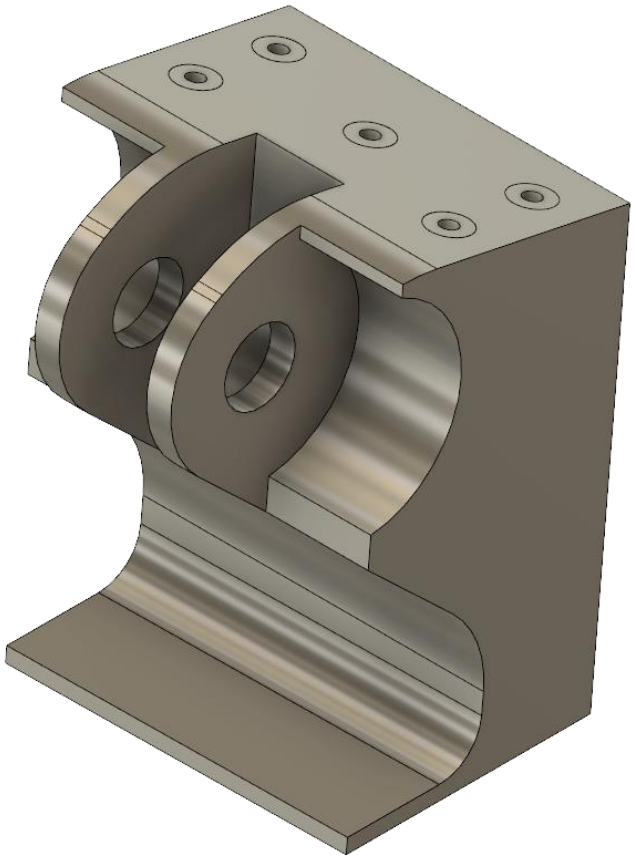


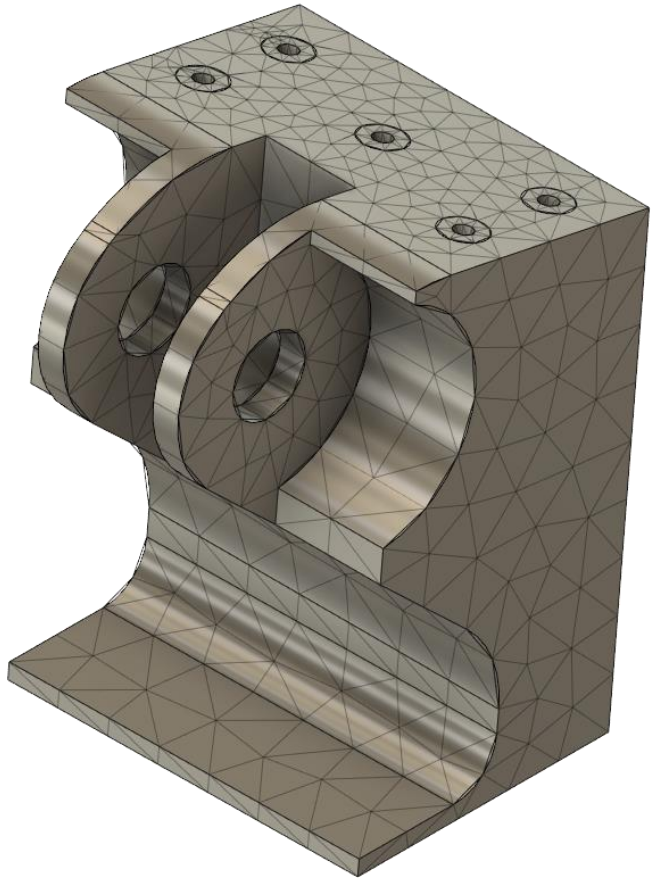




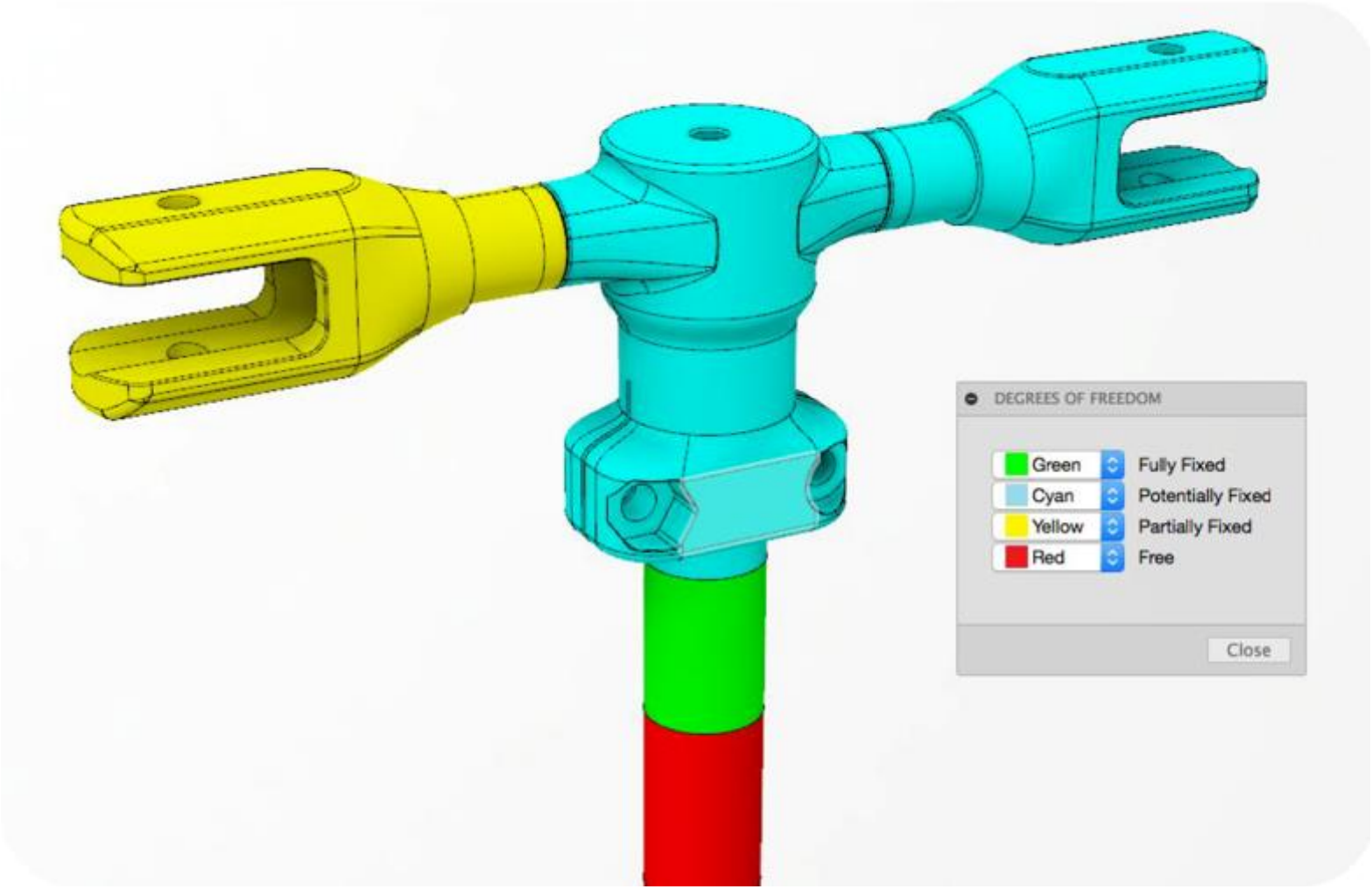
Quote from HERA (www.hera.org.nz)

What is a mesh?

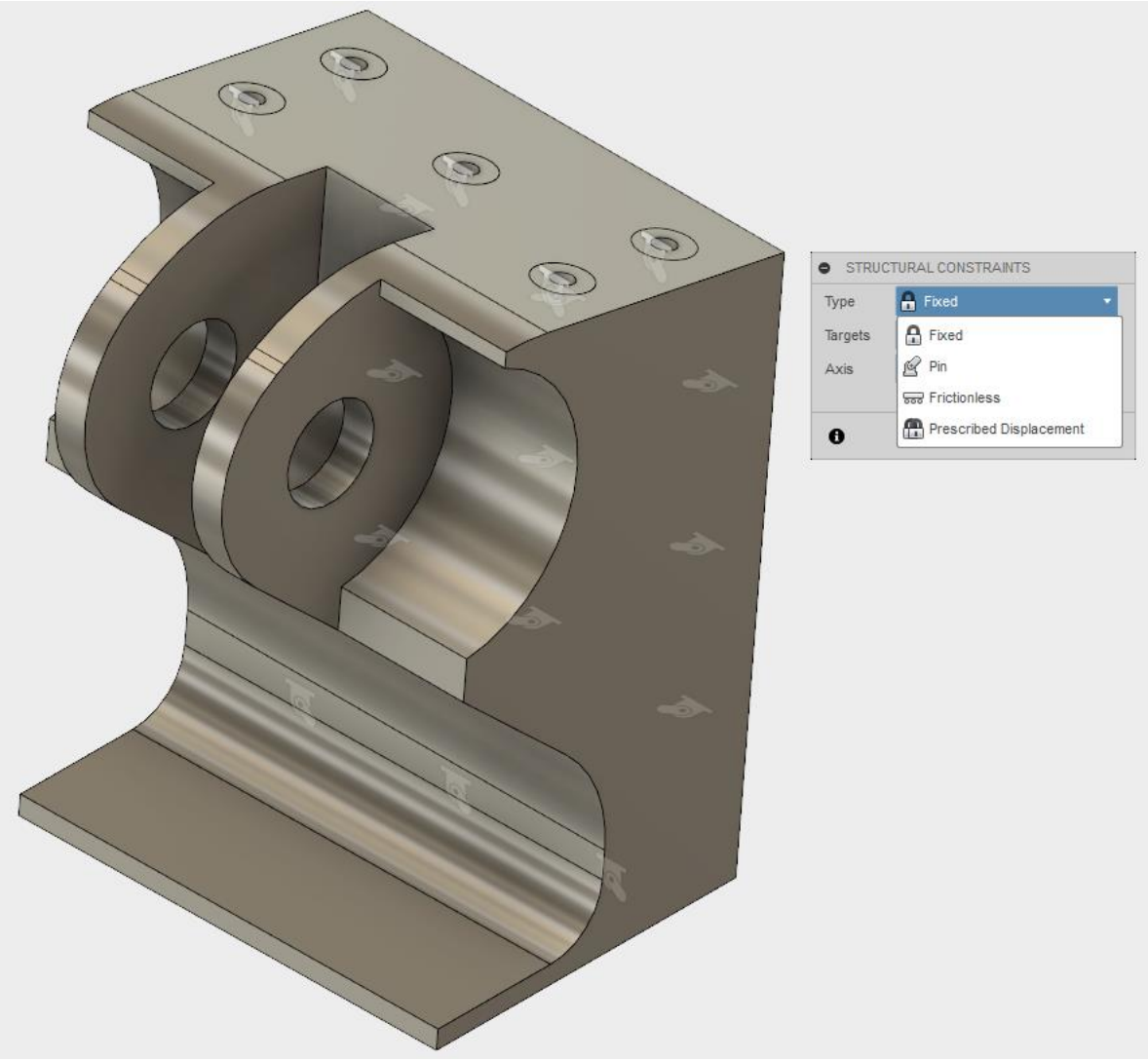




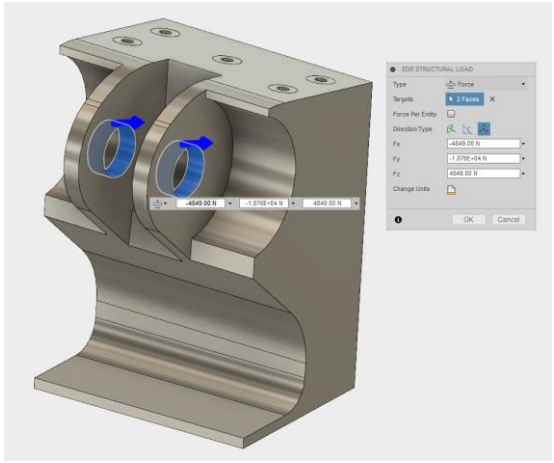
Contacts

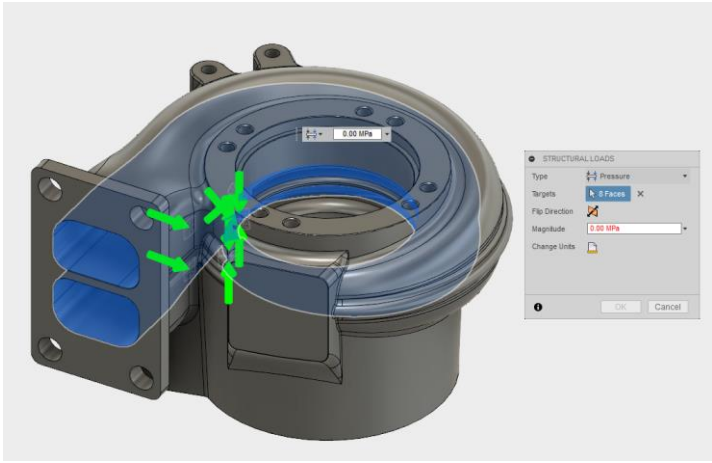


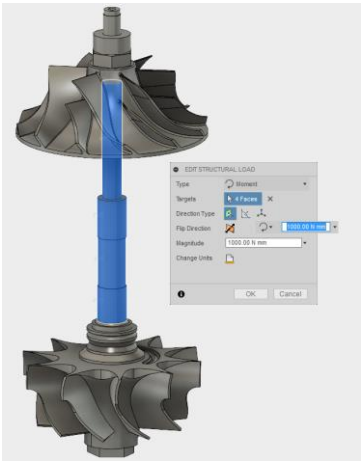
Constraints

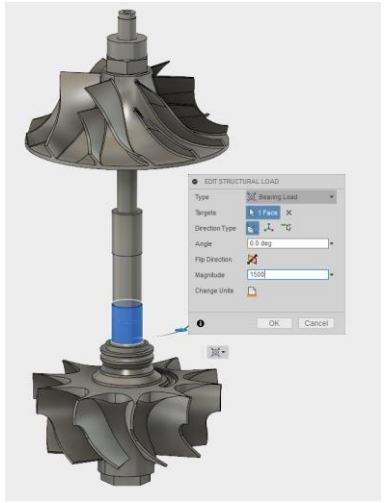


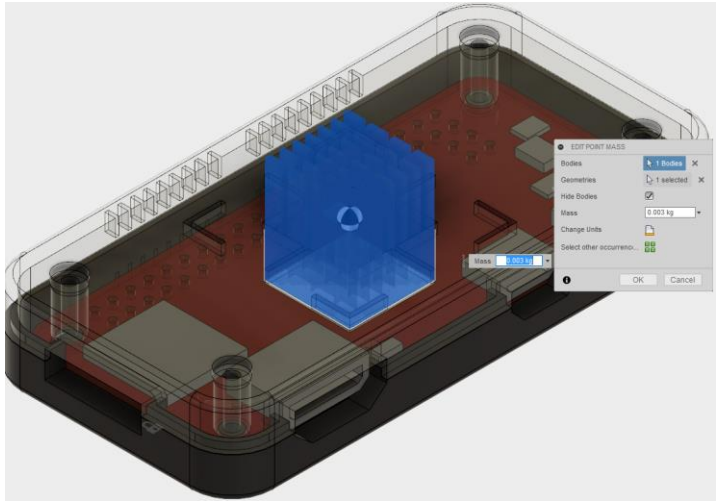
Loads

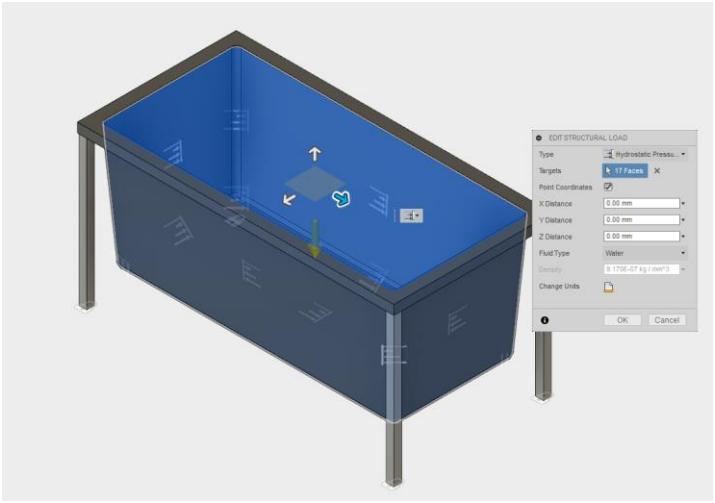






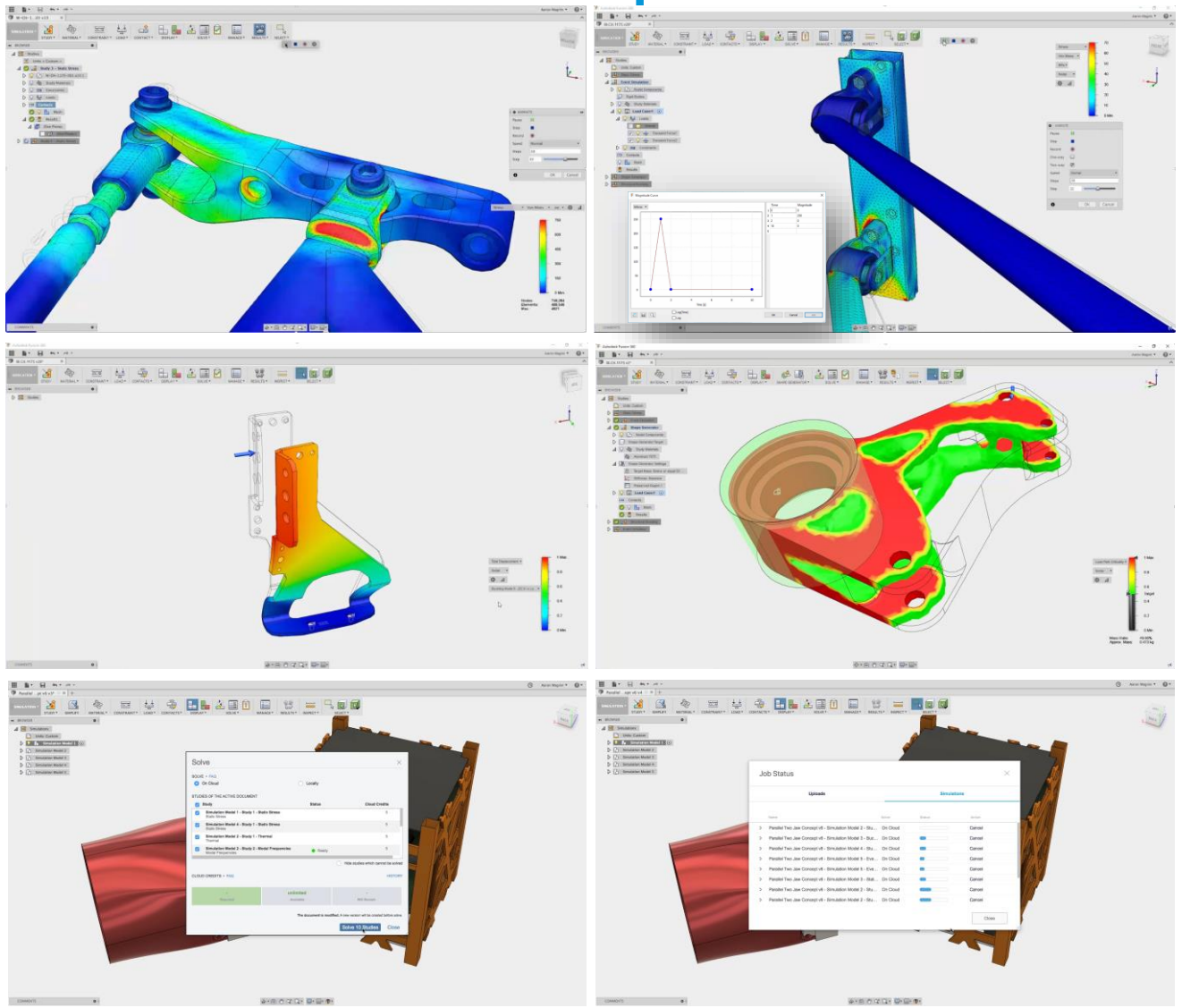




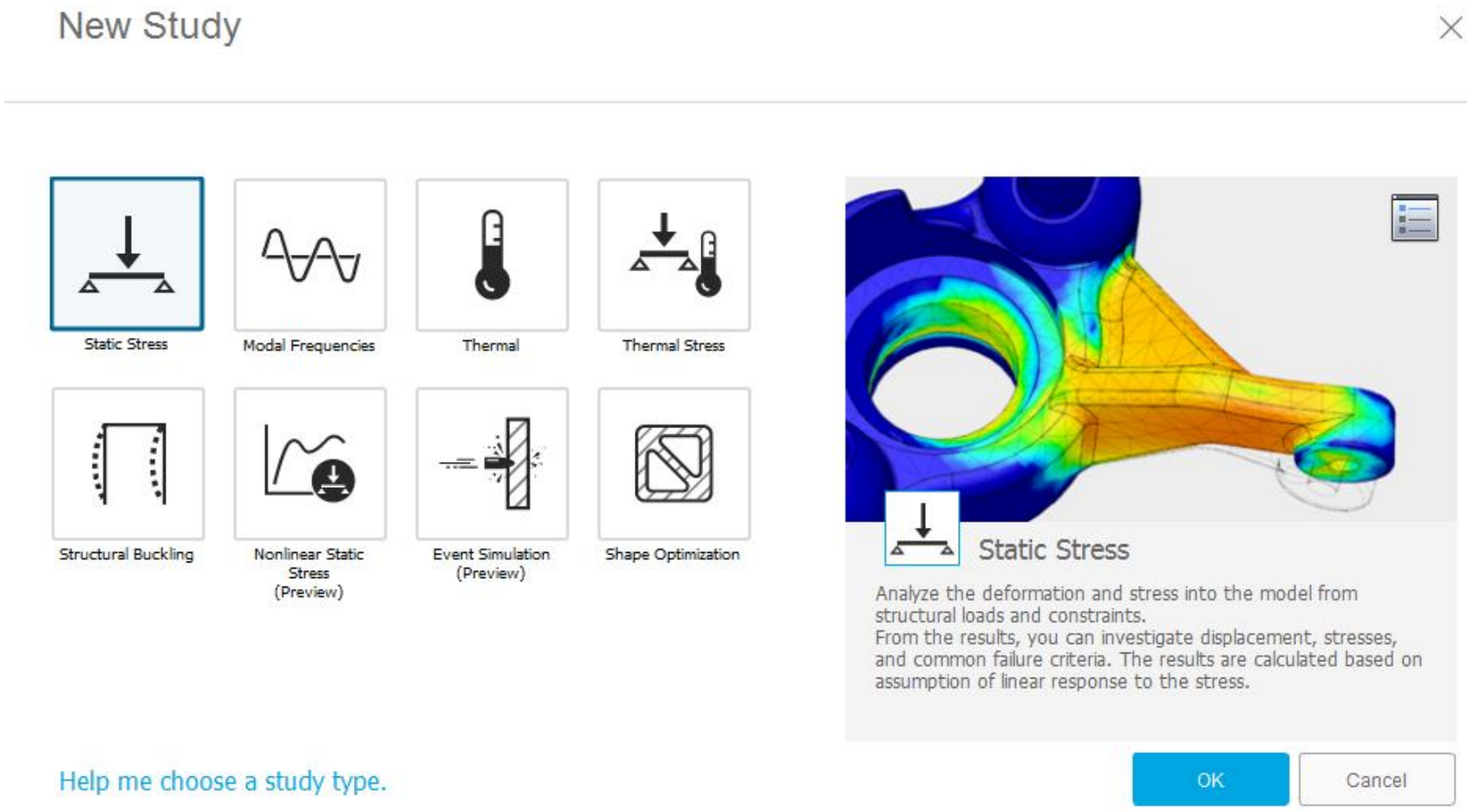


Fusion 360 – Simulation Workspace

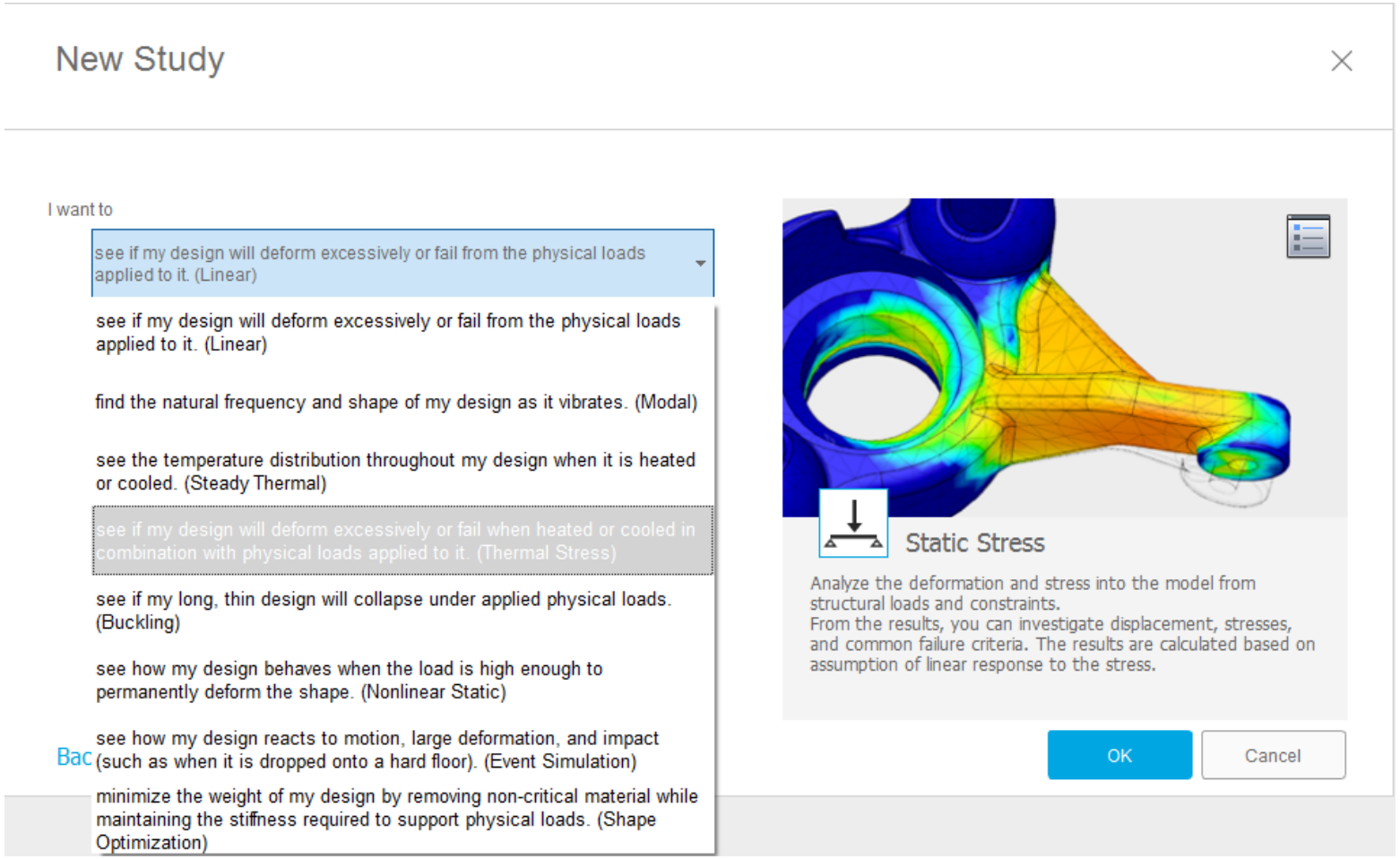
- Powered by Autodesk Nastran Solver
  - Linear Static Stress
  - Modal (Frequency)
  - Thermal
  - Thermal Stress
  - Structural Buckling
  - Shape Optimization
  - Linear Structural Buckling
  - Non-linear Static Stress
  - Event Simulation
- Cloud Solve
- Simplify Workspace
- Compare Workspace



Simulation Types

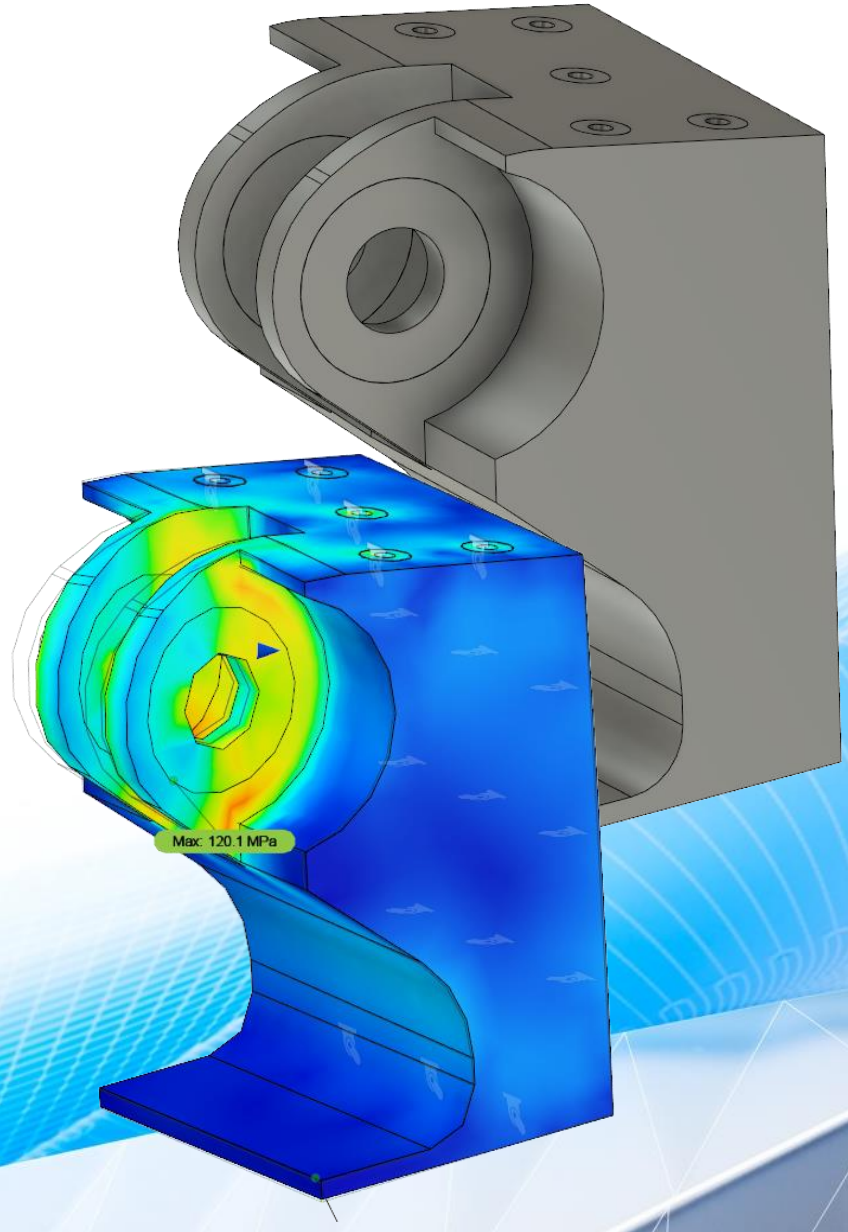


Questions to ask



Hands On

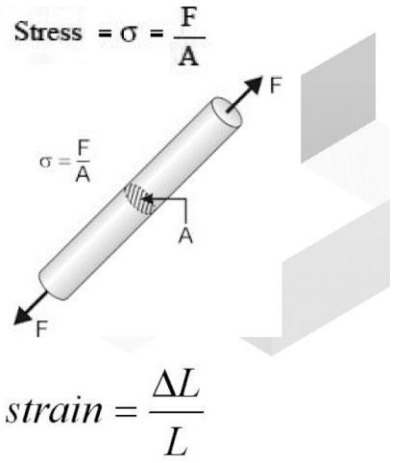
Static Stress

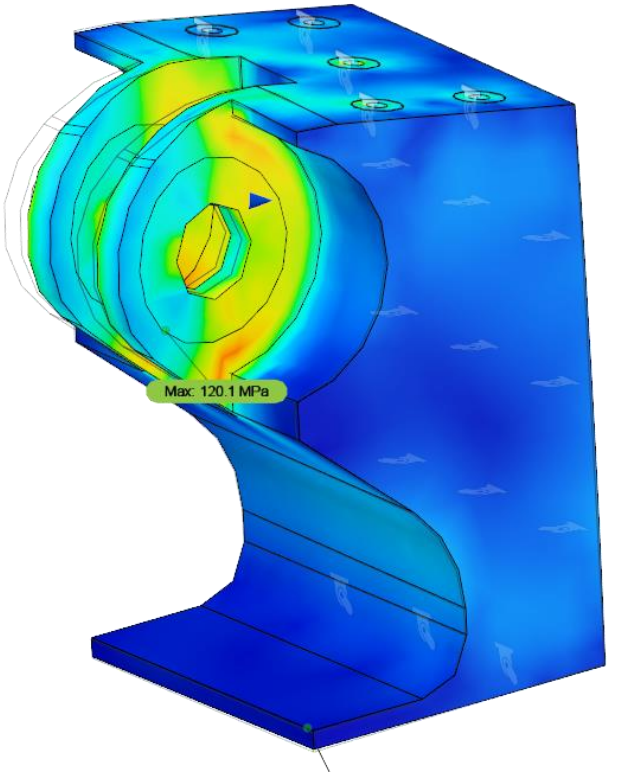


Simulation Types

Static Stress

- Material is “Linear”
- Loads are applied slowly and steadily
- Structural deformations are small



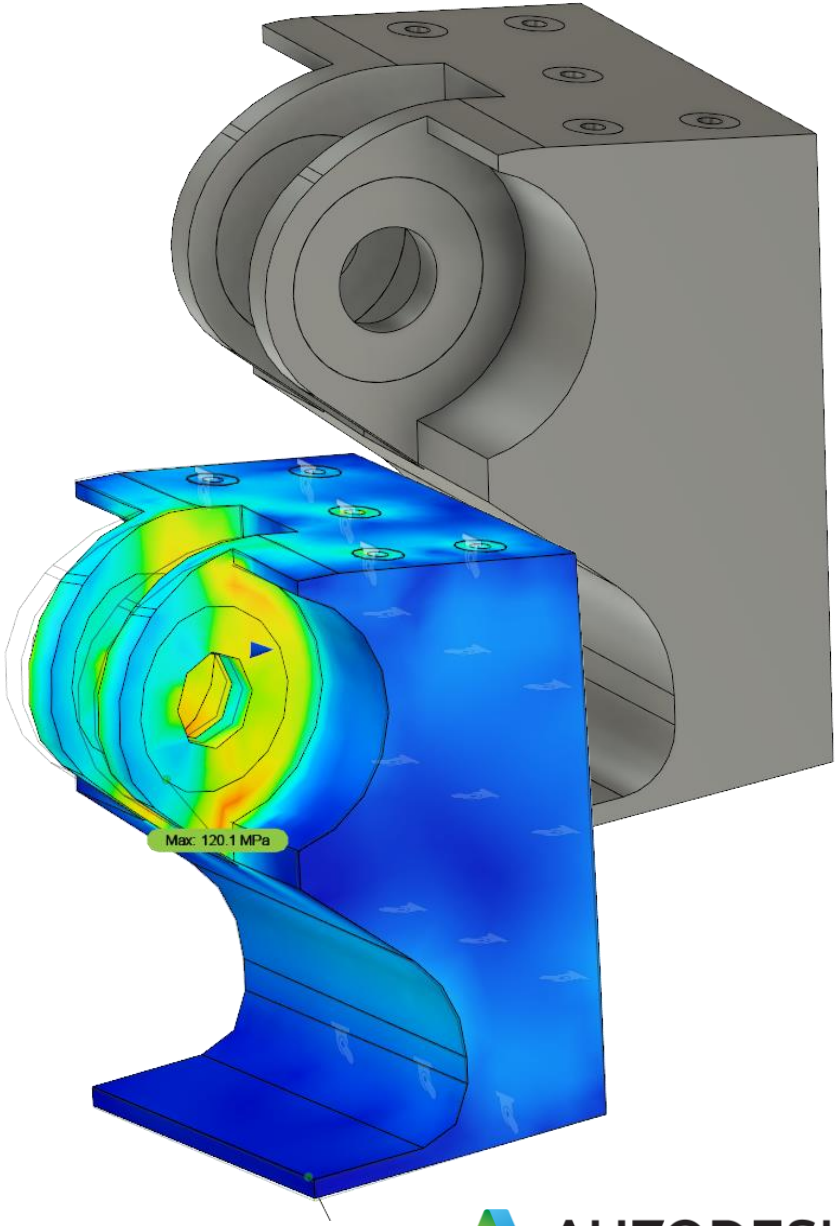


Static Stress

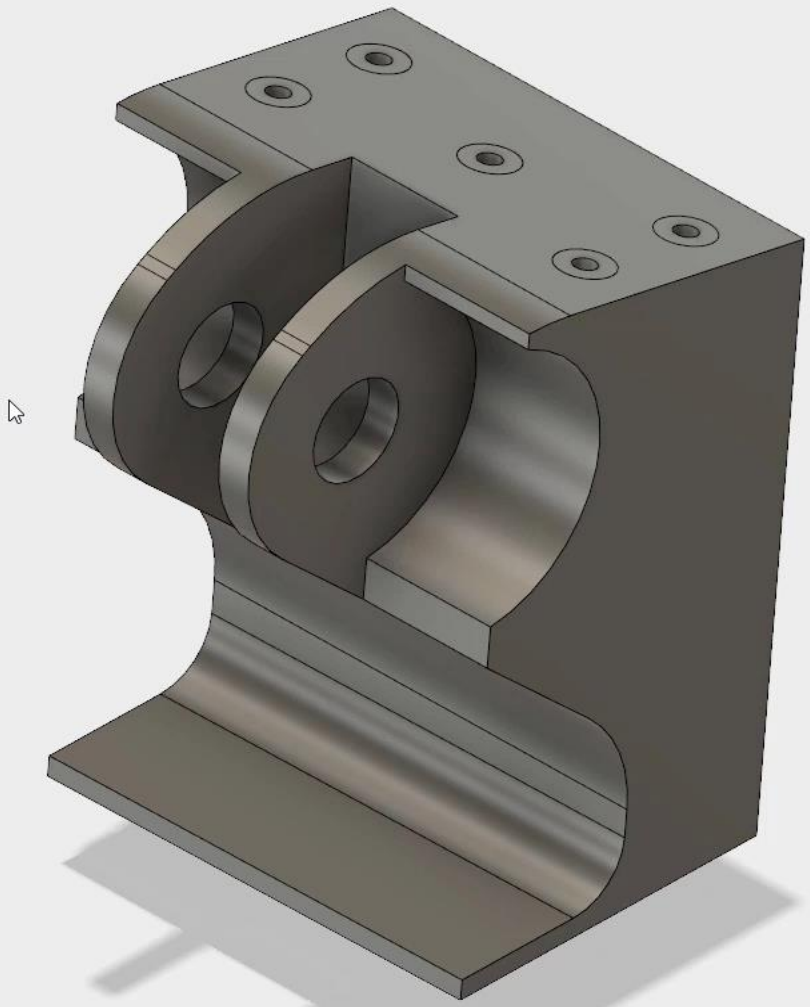
GKN Bracket

Aircraft Elevator Bracket

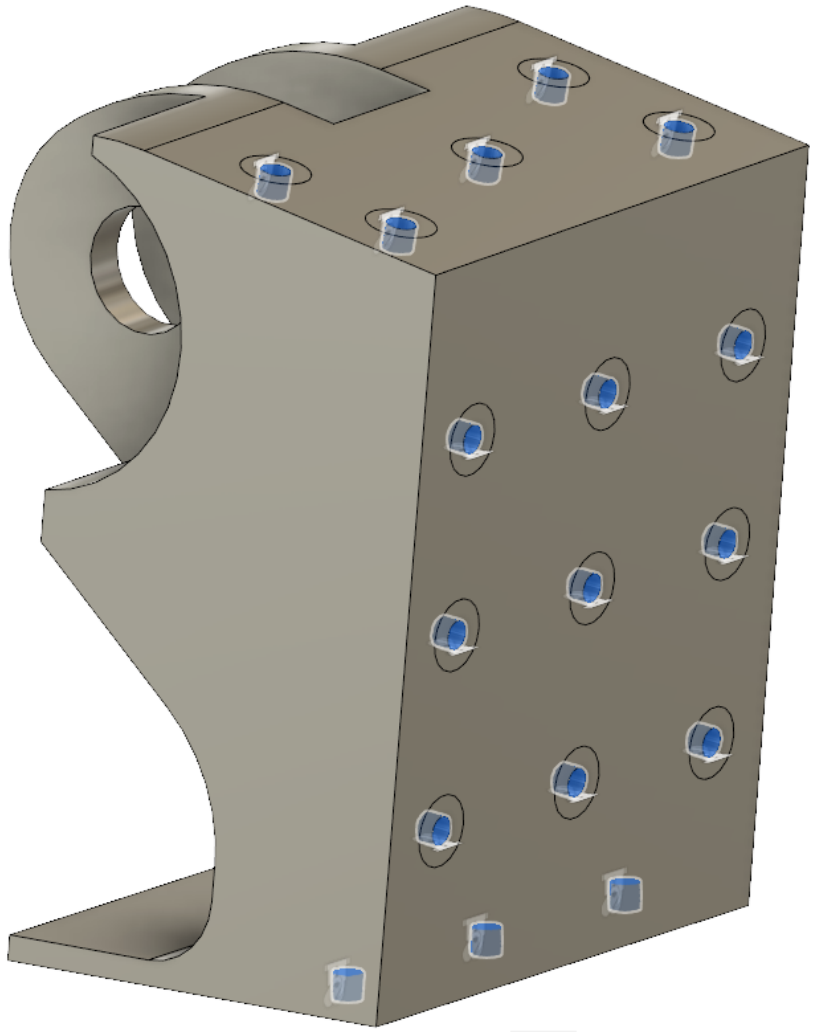
- Material – Ti 6Al 4V
- Minimum FOS – 1.2
- Maximum Displacement – 0.55mm

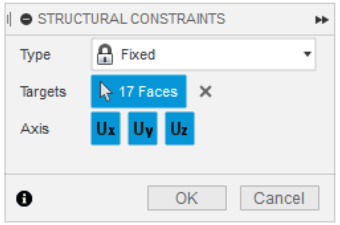


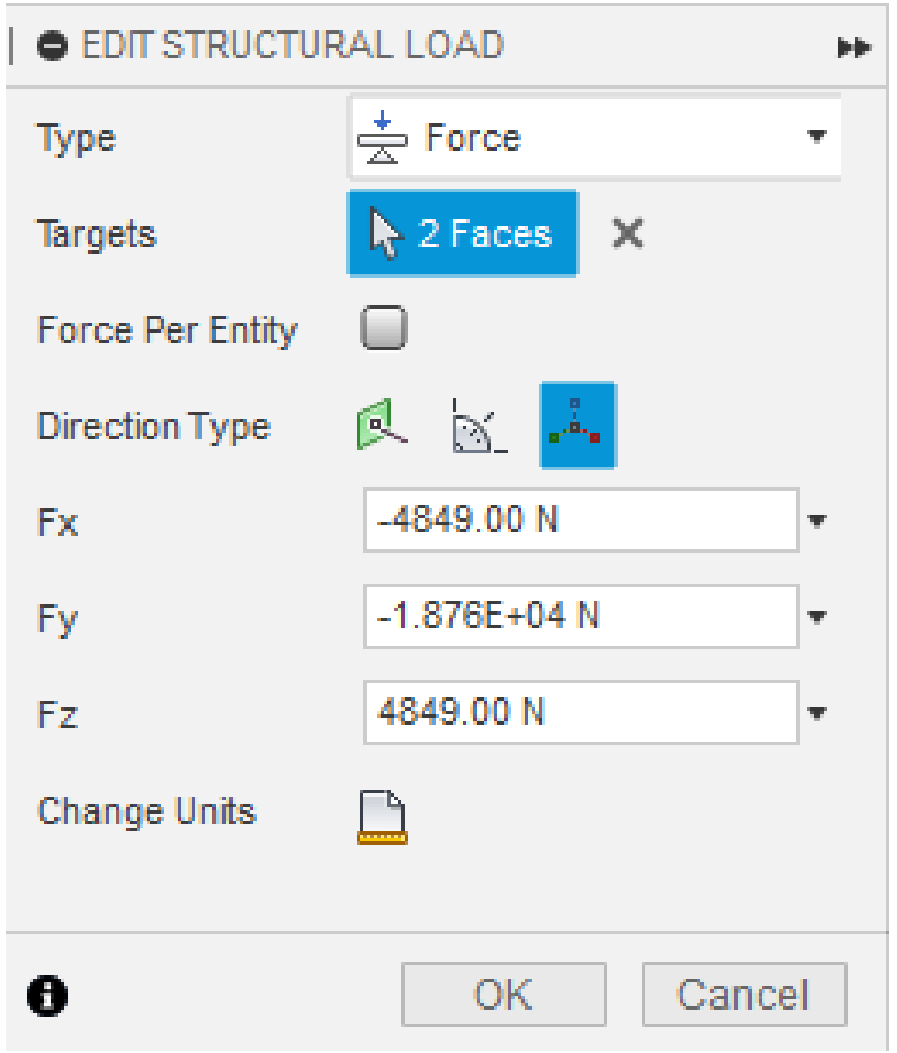
Static Stress



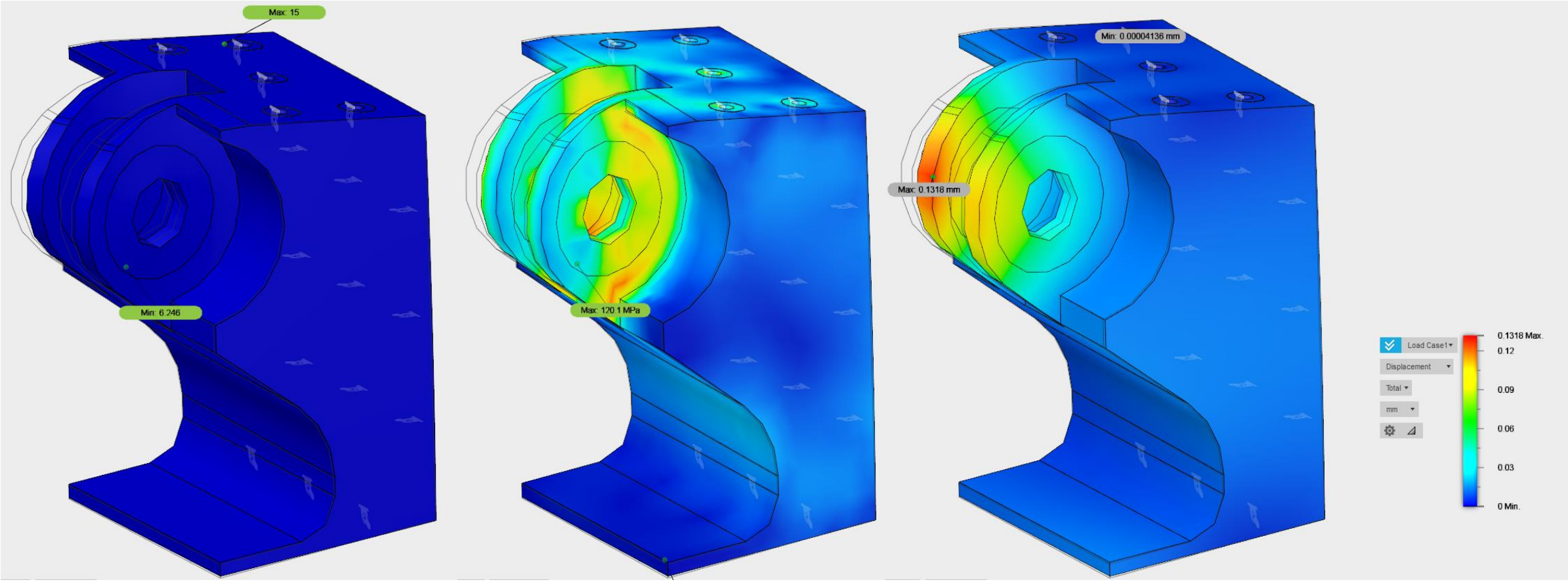
Constraints & Loads



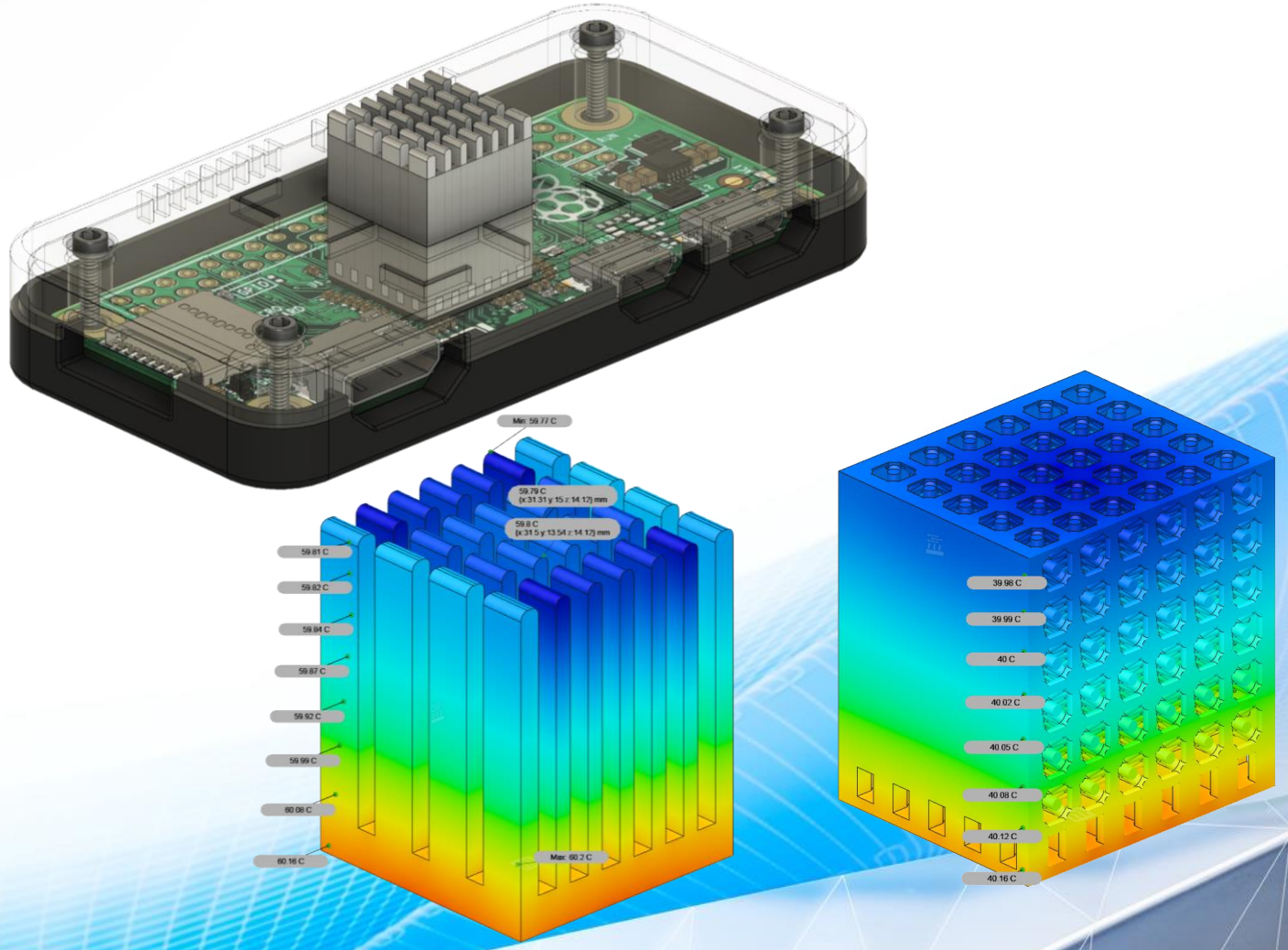




Static Stress



Thermal


Min: 59.77 C
59.79 C
(x 31.31 y 15 z 14.12) mm
59.8 C
(x 31.5 y 13.54 z 14.12) mm
59.81 C
59.82 C
59.84 C
59.87 C
59.92 C
59.99 C
60.06 C
60.16 C
Max: 60.2 C
39.98 C
39.99 C
40 C
40.02 C
40.05 C
40.08 C
40.12 C
40.16 C

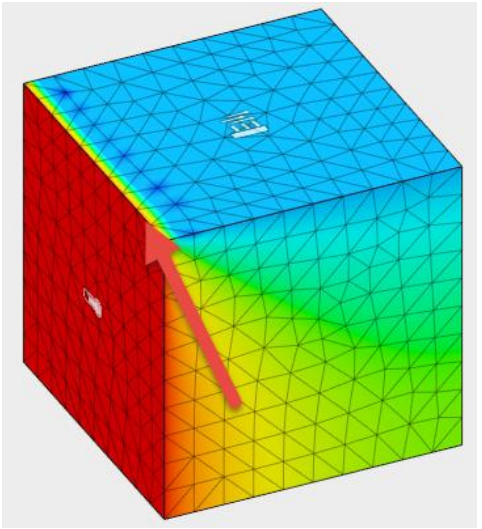
Simulation Types

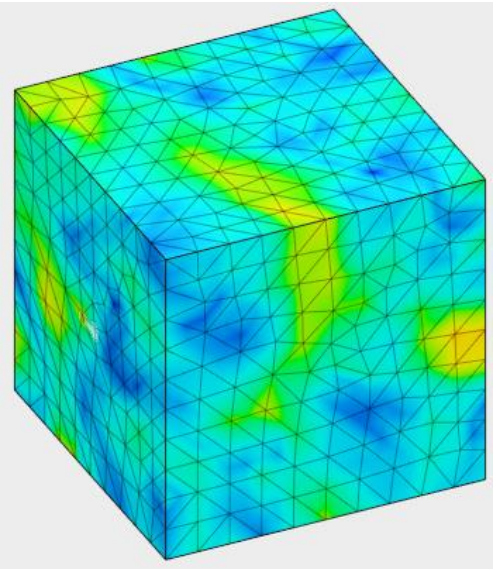
Thermal

Convection of heat

Measure Heat Flux and gradient

Understand how thermal distribution will effect components



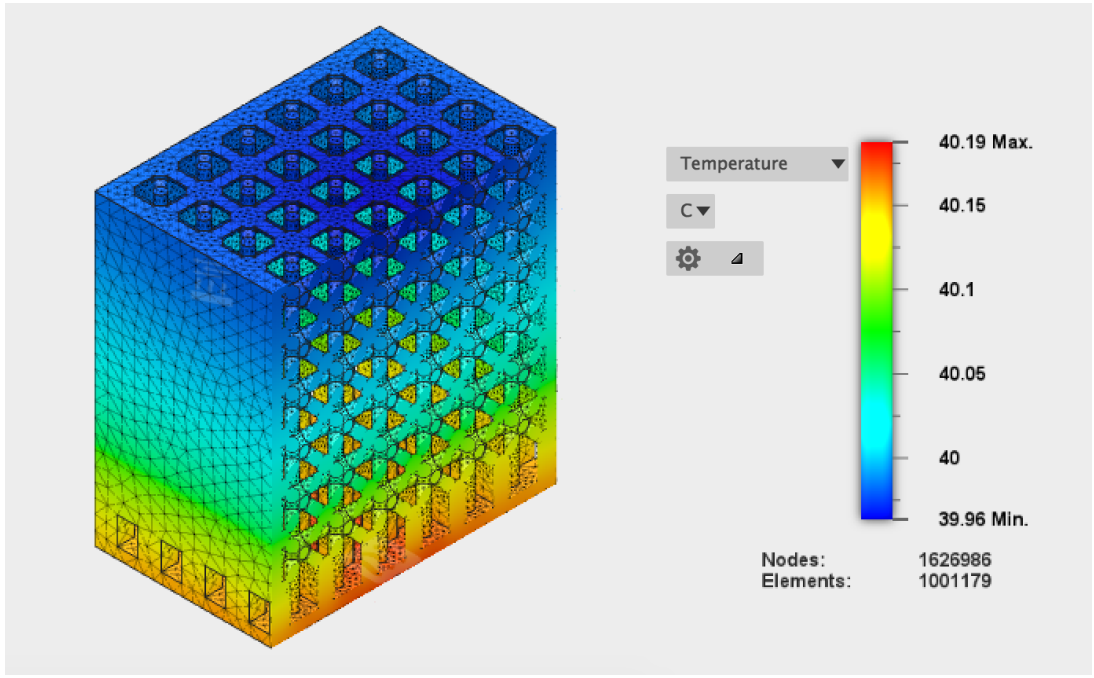


Thermal

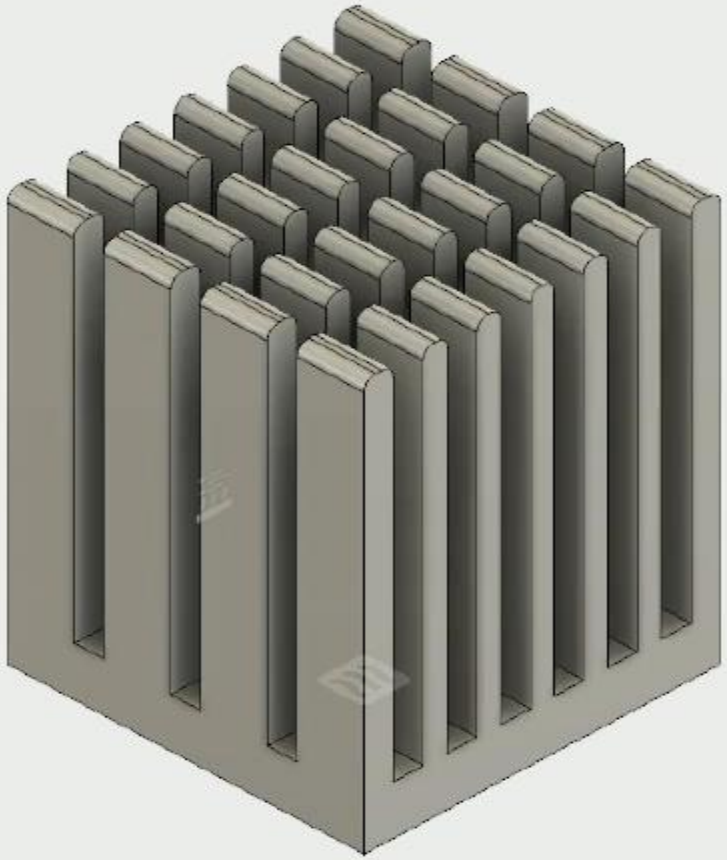
Rasberry Pi PCB

Heat Sink

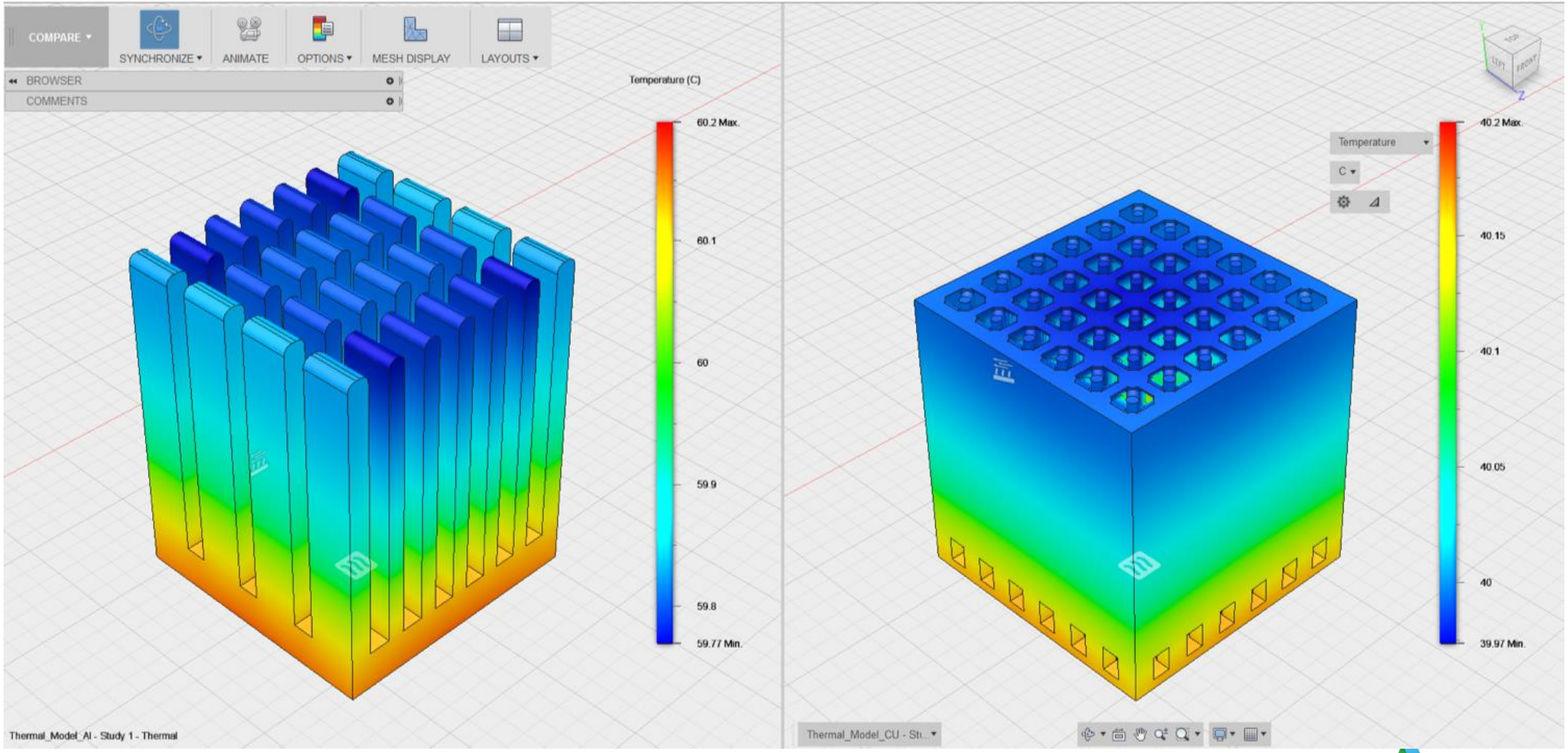
- Material – Aluminum
- Internal Heat – 0.7W
- Convection – 7.00 W / (m^2 K) | 20°C
- Reduce maximum temperature



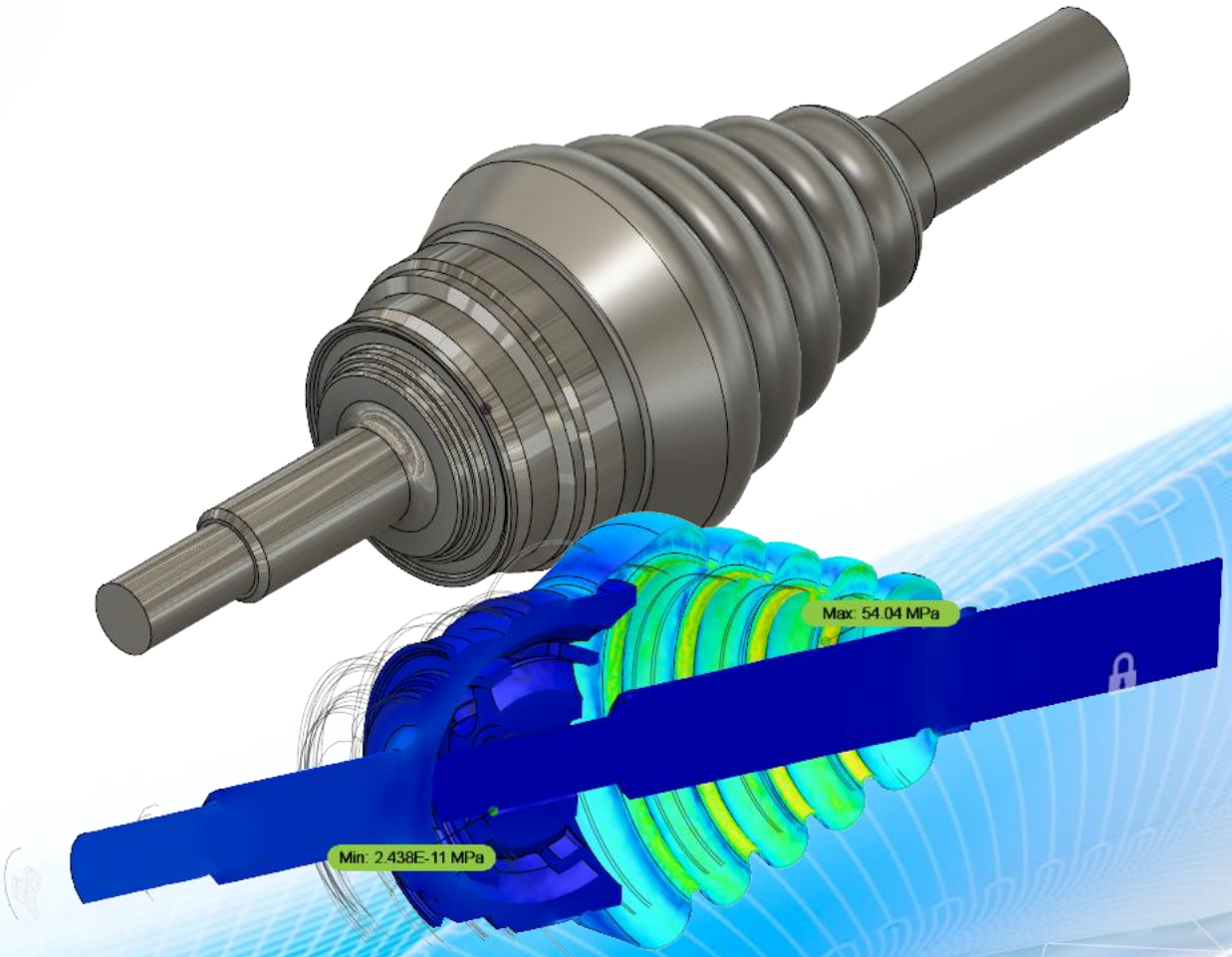
Thermal



Thermal



Non Linear



Simulation Types

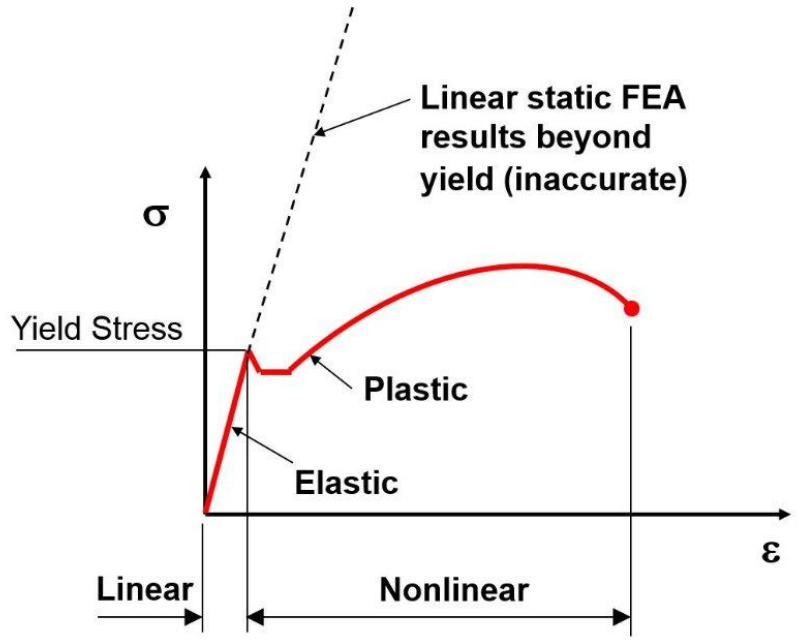
Non Linear

Material is not “Linear”

Direction of load changes as the model deforms

Displacement is large

Stress strain curves deviates from a straight line

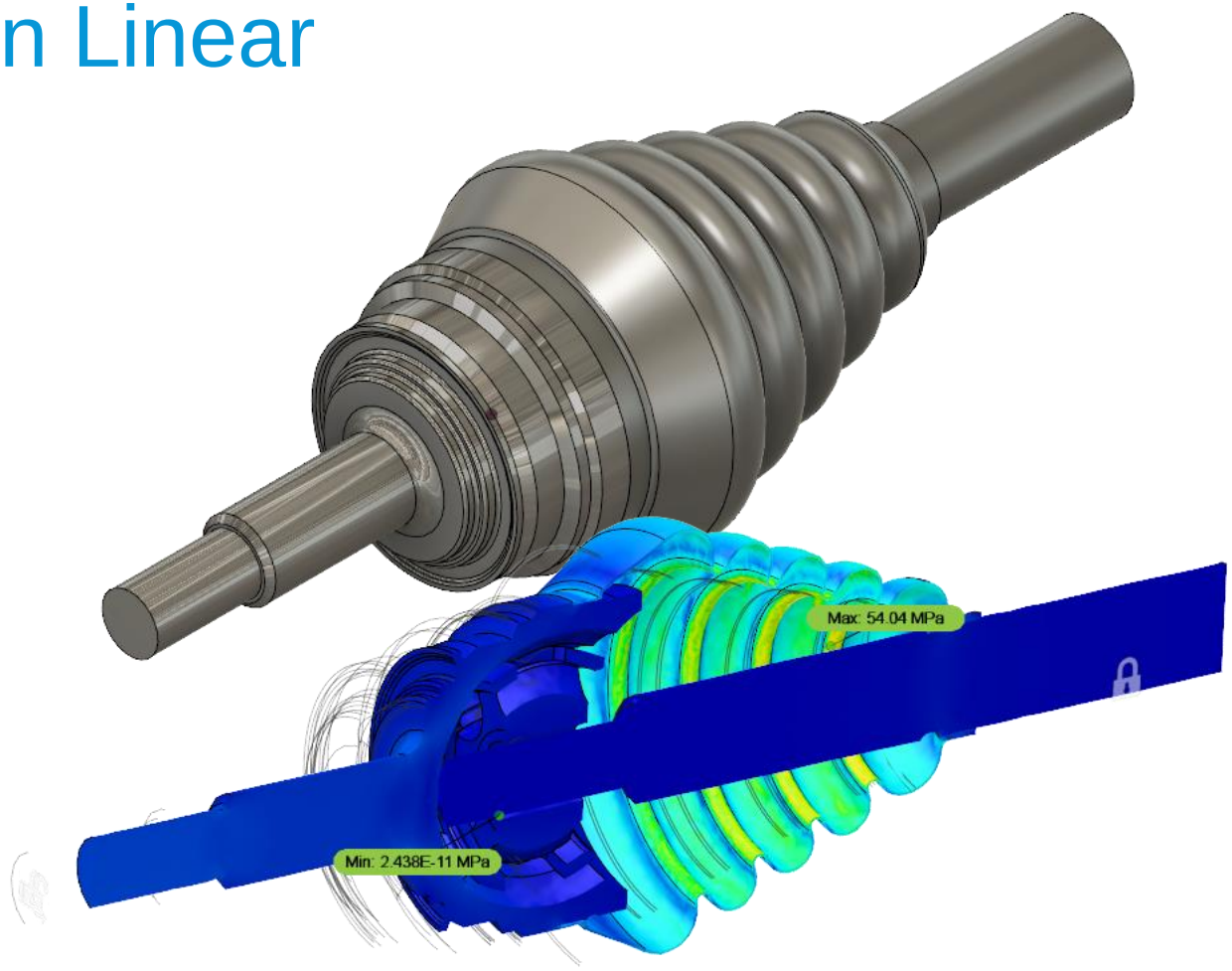


Non Linear

Constant Velocity Joint

CV Joint Gator

- Materials
  - Stainless Steel
  - Rubber (Non Linear)
- FOS > 1.5
- Stress < 75 MPa



Autodesk Fusion 360

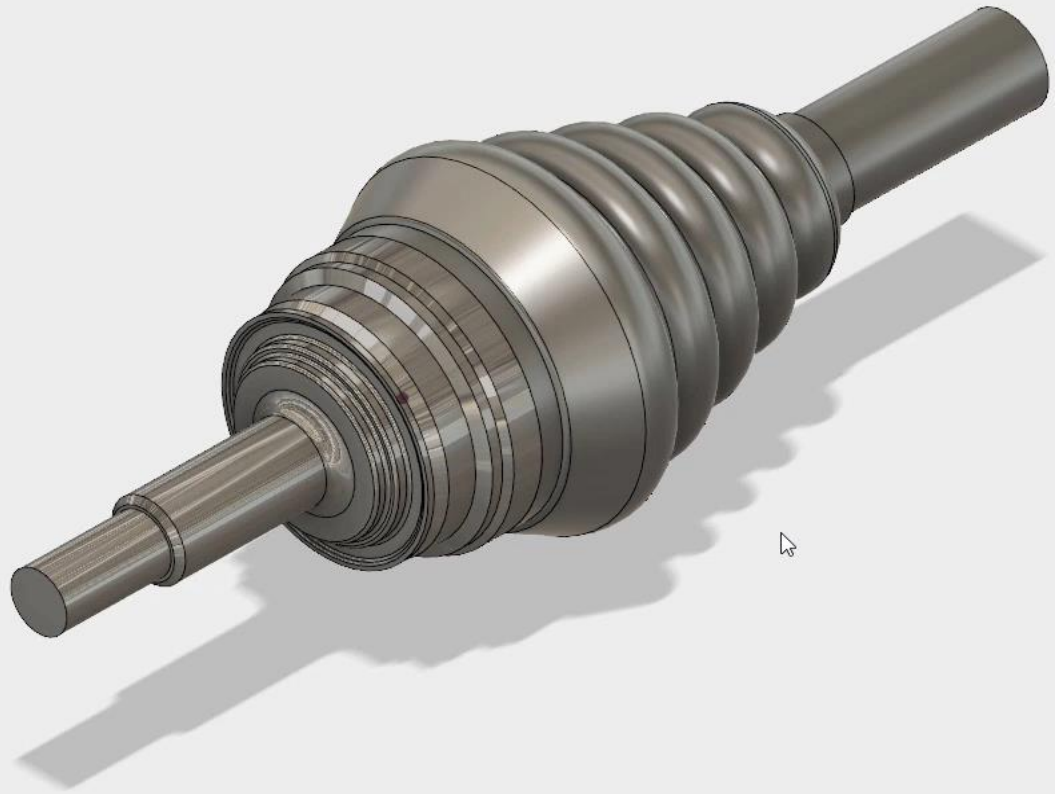
C.V joint START v2

MODEL | SKETCH | CREATE | MODIFY | ASSEMBLE | CONSTRUCT | INSPECT | INSERT | MAKE | ADD-INS | SELECT | SHEETER

Non Linear

BROWSER

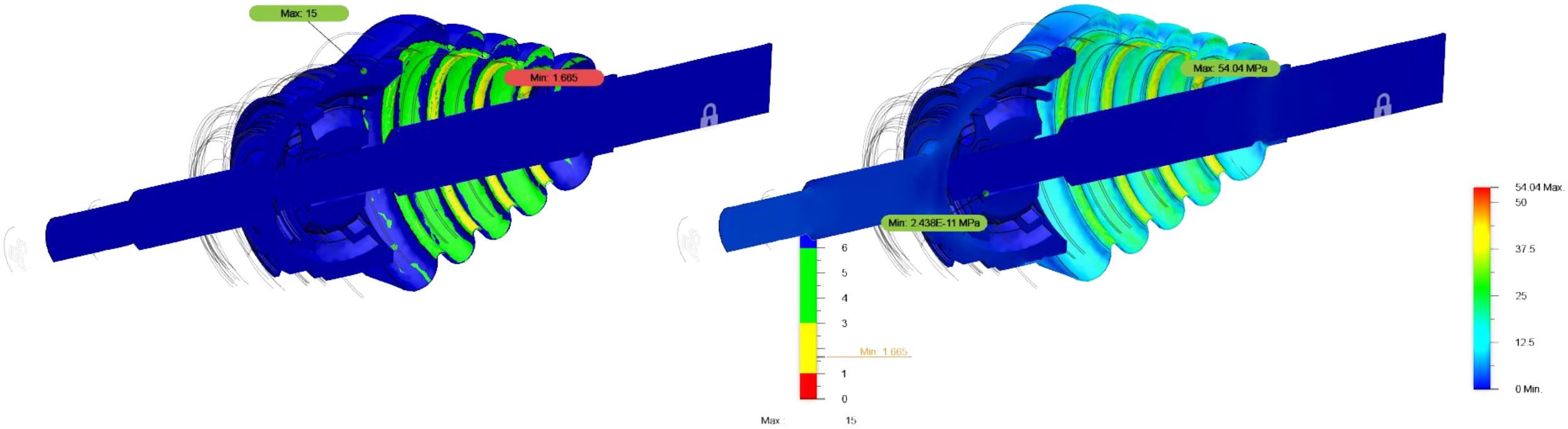
- C.V joint START v2
  - Document Settings
  - Named Views
  - Origin
  - Analysis
  - Joint Origins
  - Joints
  - Sketches
  - Construction
  - 3:1
  - 1:1
  - 2:1
  - Driveshaft:1
  - Gator:1



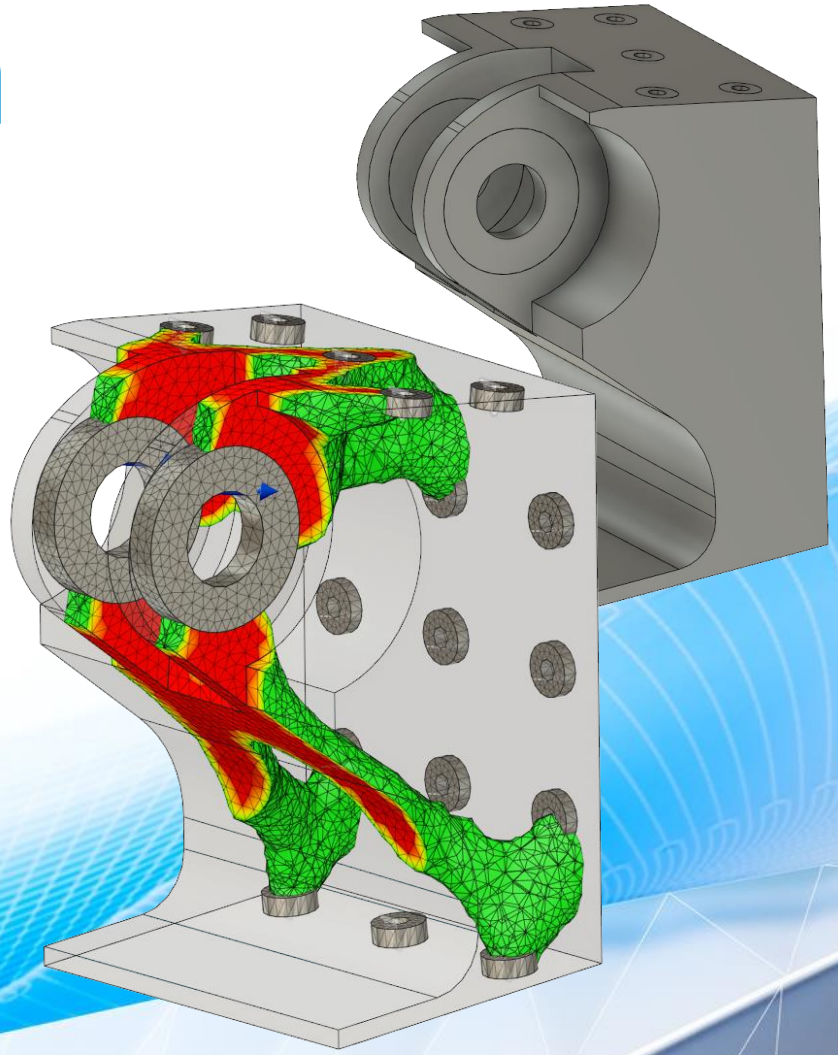
COMMENTS

Non Linear

Results



Shape Optimisation



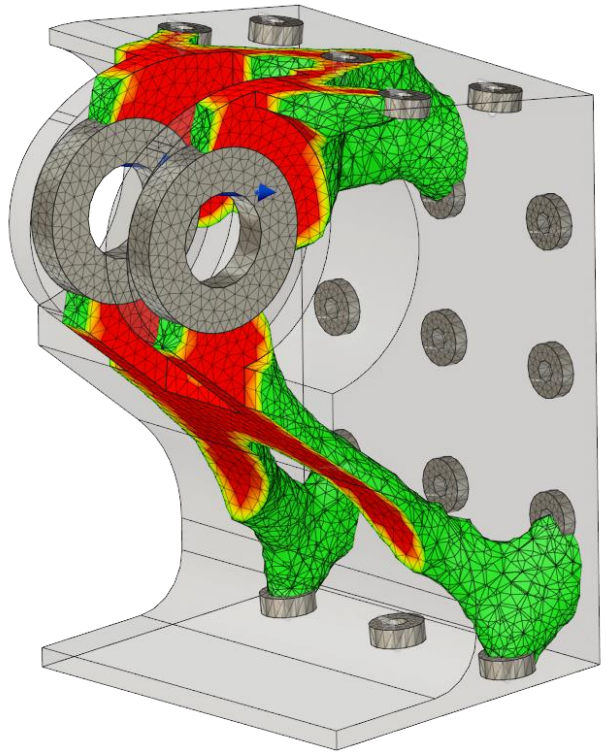
Simulation Types

Shape Optimisation

A Shape Optimization study should be used if you are interested in designing lightweight, structurally efficient parts

Scenarios to use Shape Optimisation

- Design Exploration
- Lightweighting of parts
- Determine where to safely remove material

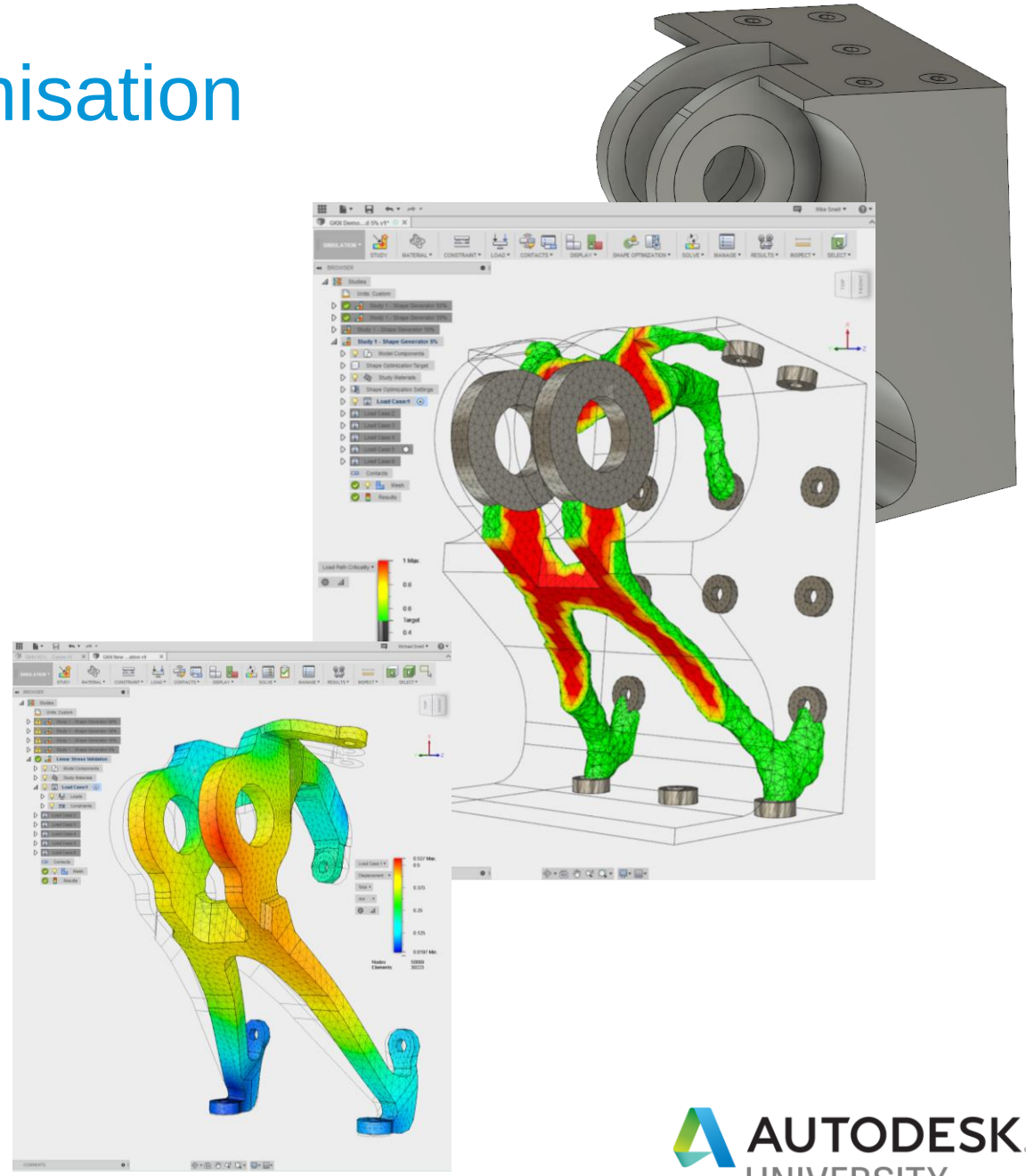


Shape Optimisation

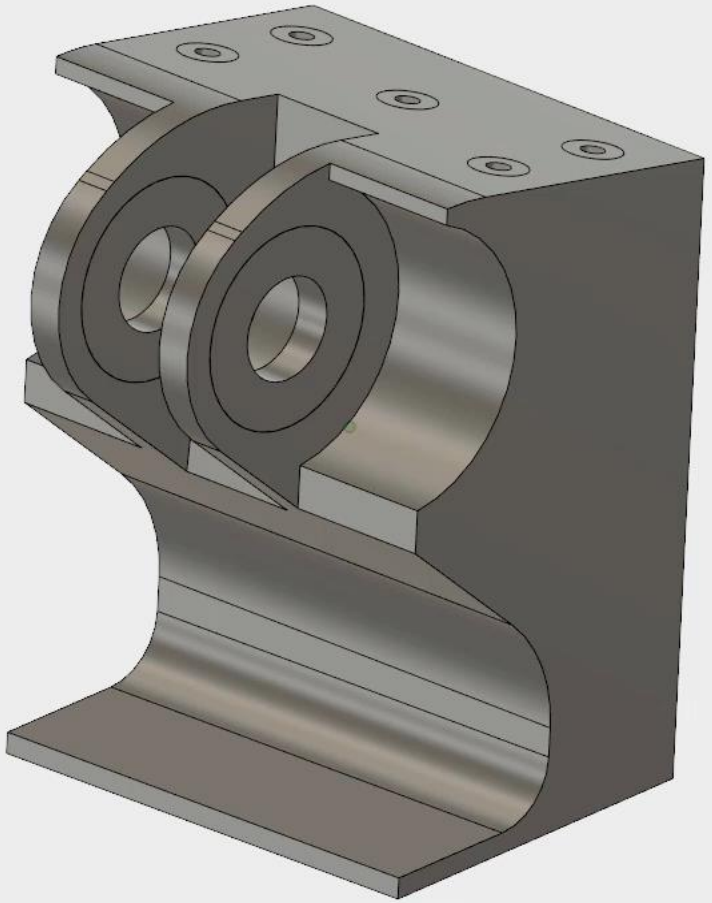
Aircraft Elevator Bracket

GKN Bracket

- Material – Ti 6Al 4V
- Minimum FOS – 1.2
- Maximum Displacement – 0.55mm
- 6 multi-directional load cases @ 1.5x operating loads

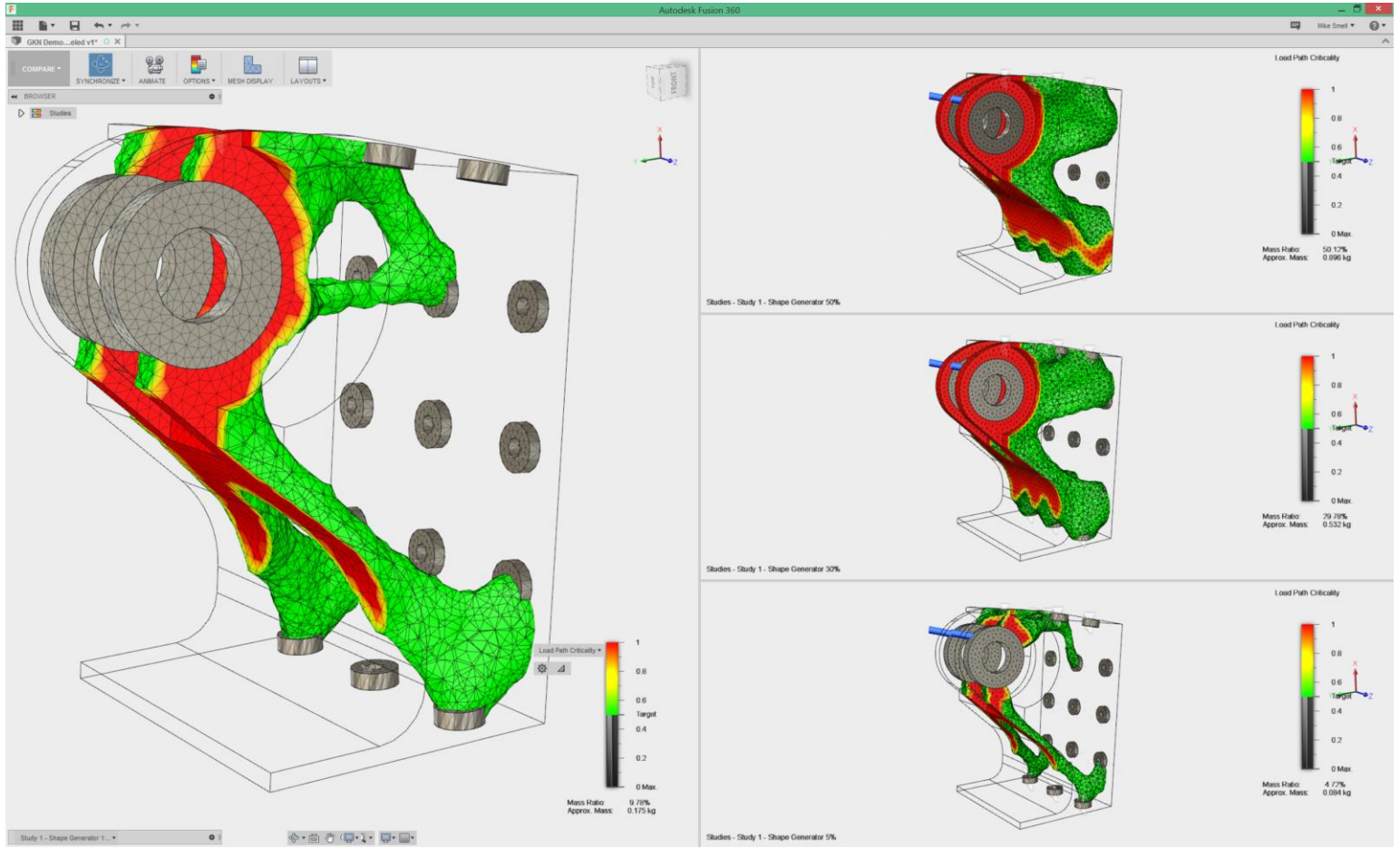


Shape Optimisation



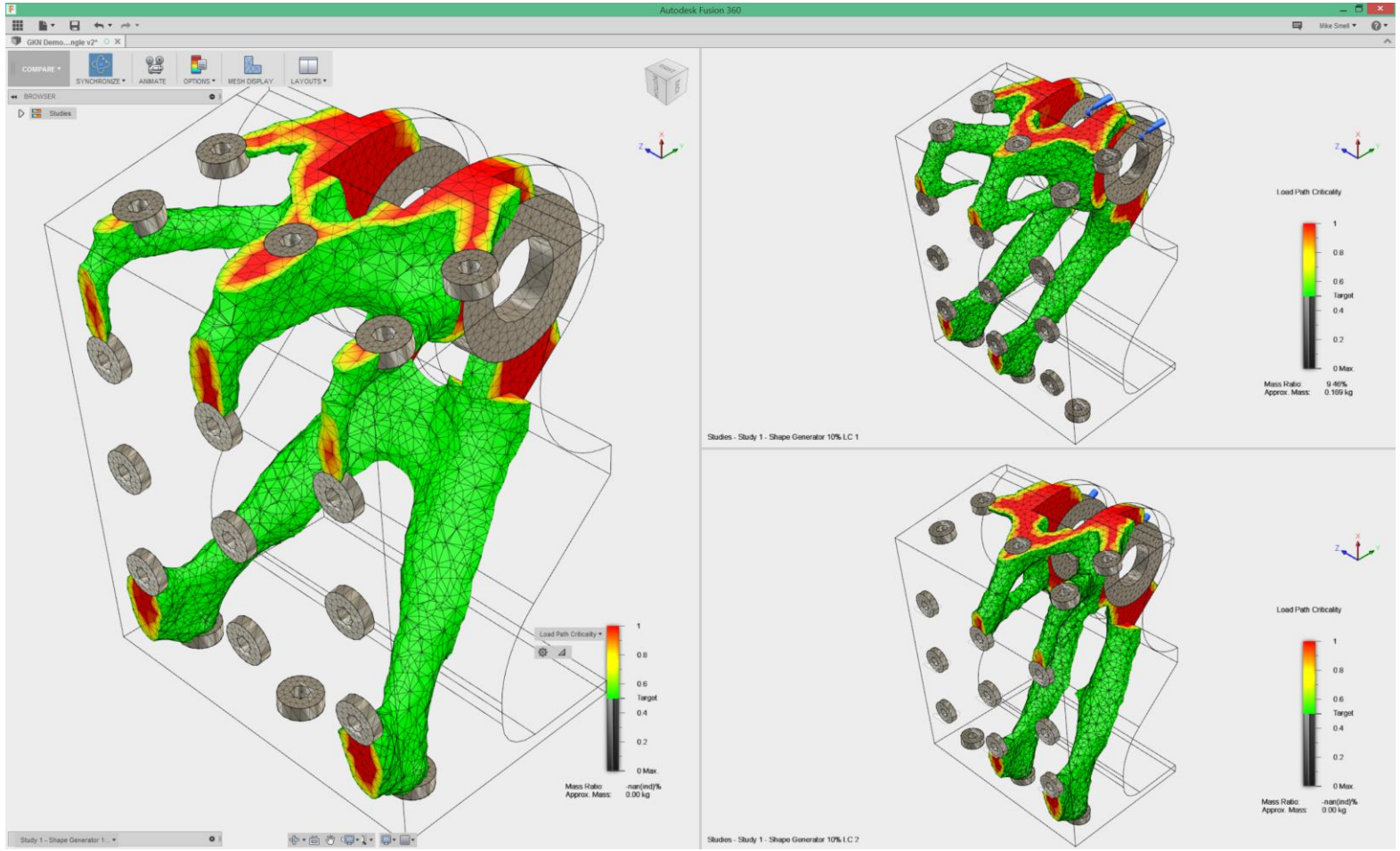
Shape Optimisation

Results – Multiple Mass Targets



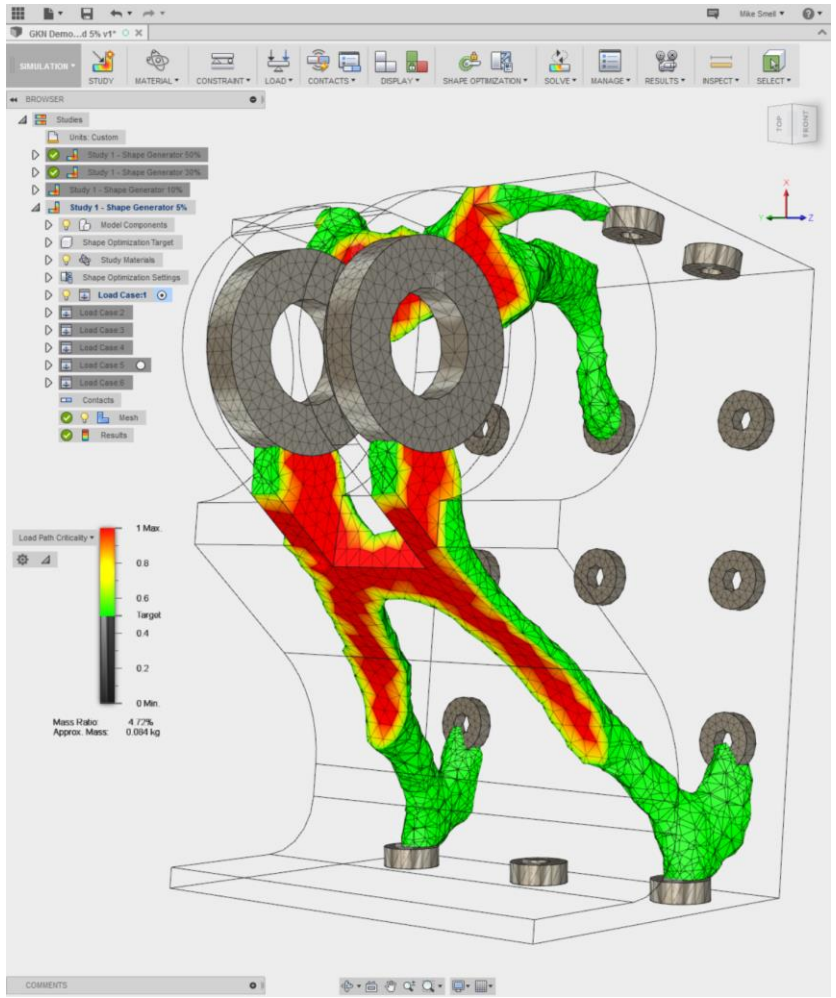
Shape Optimisation

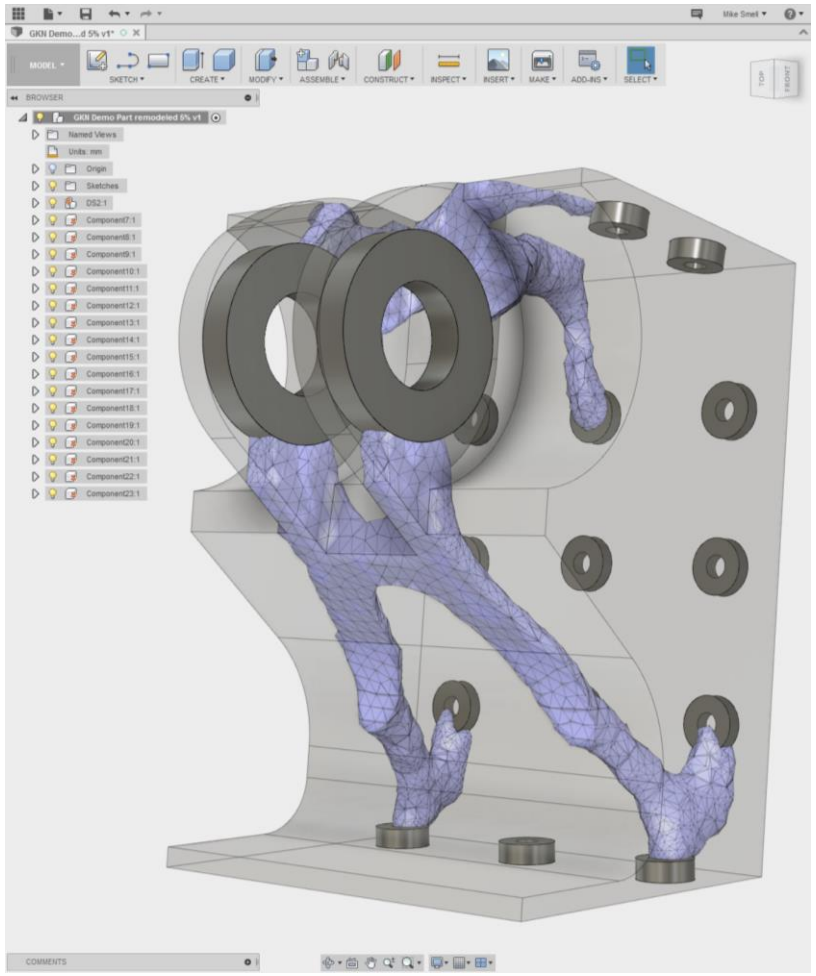
Results – Multiple Load Cases



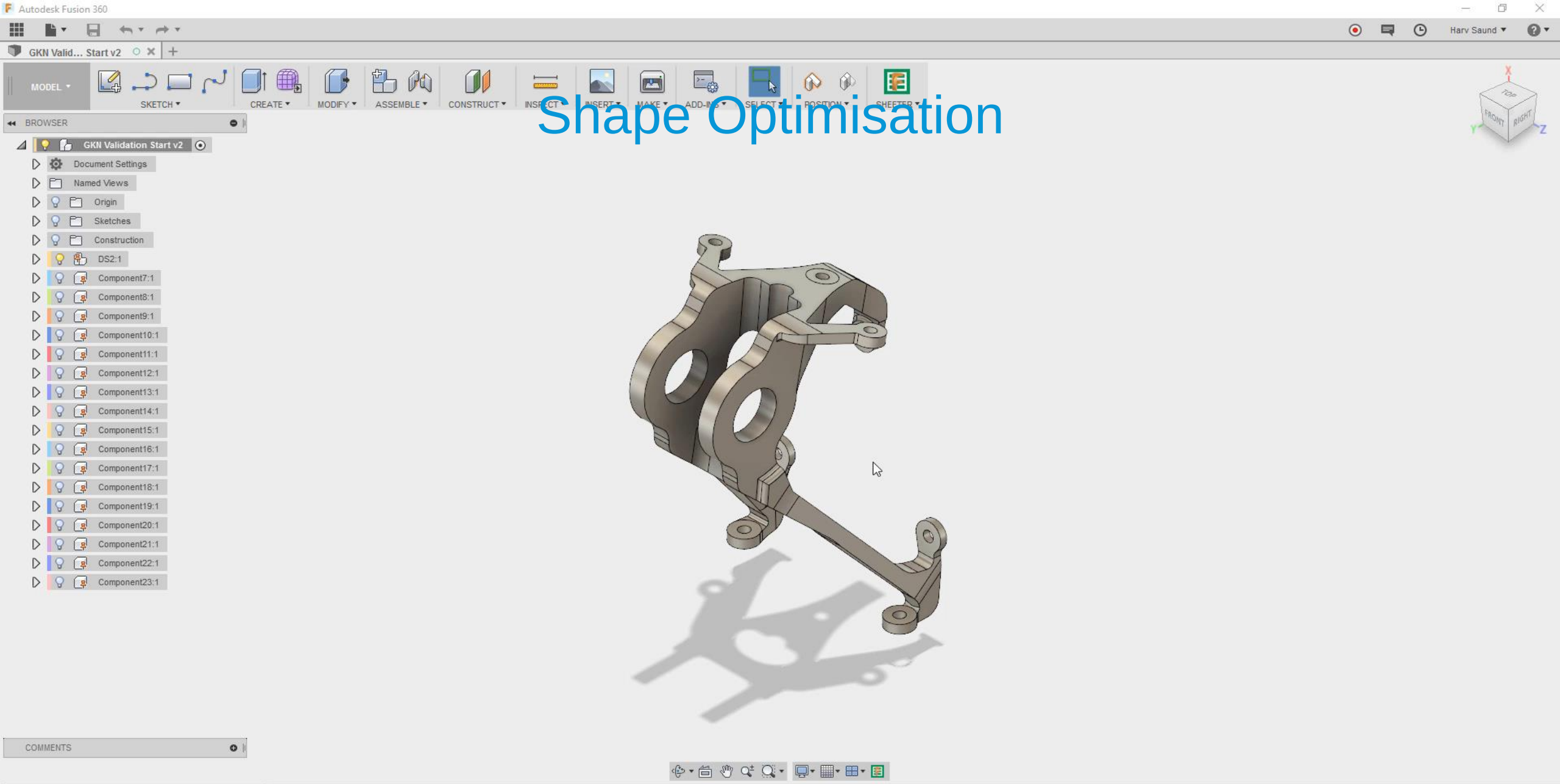
Shape Optimisation

Promote to Model workspace



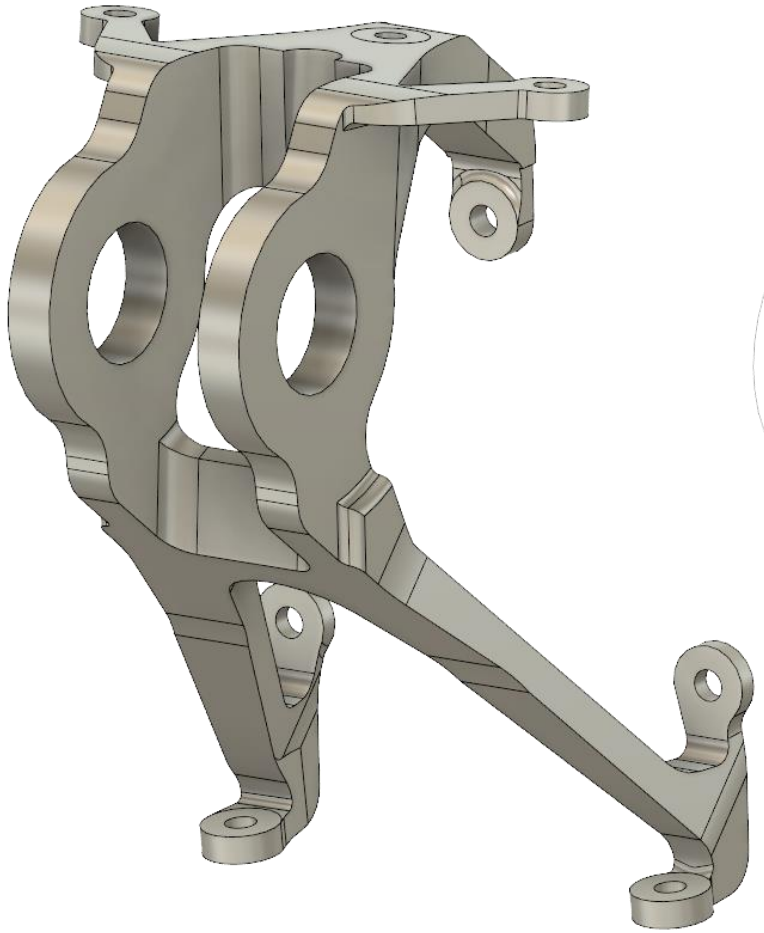


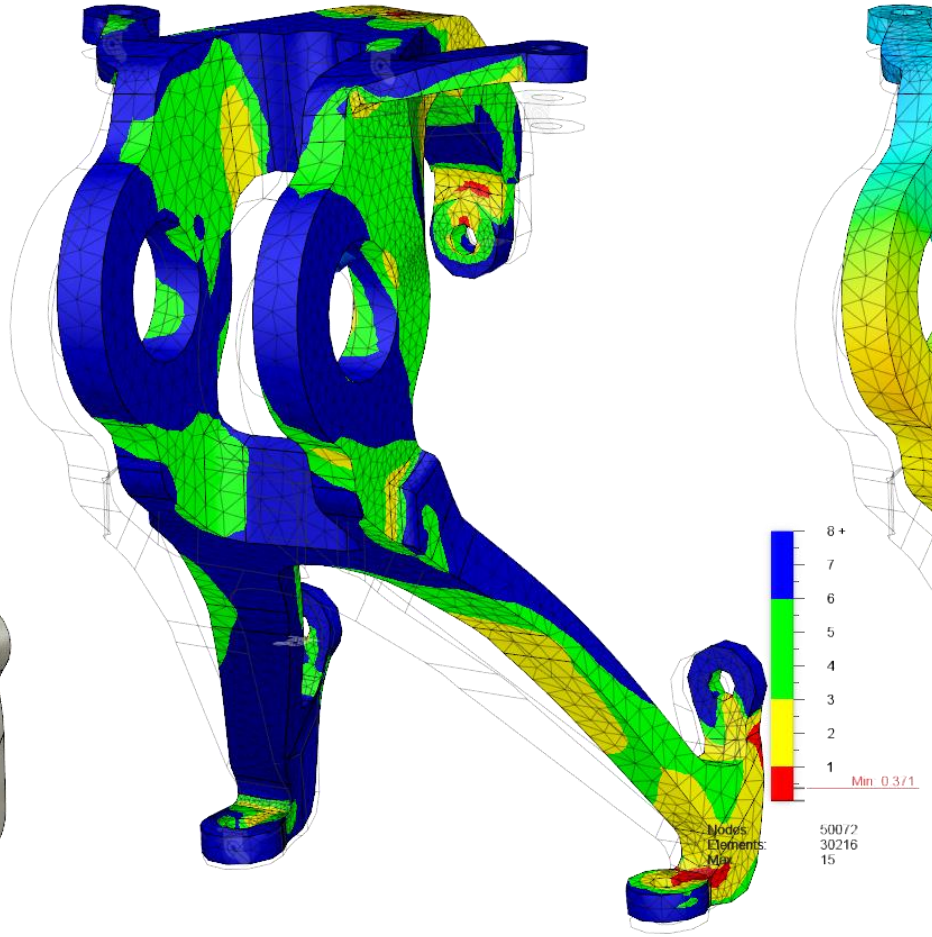
Shape Optimisation

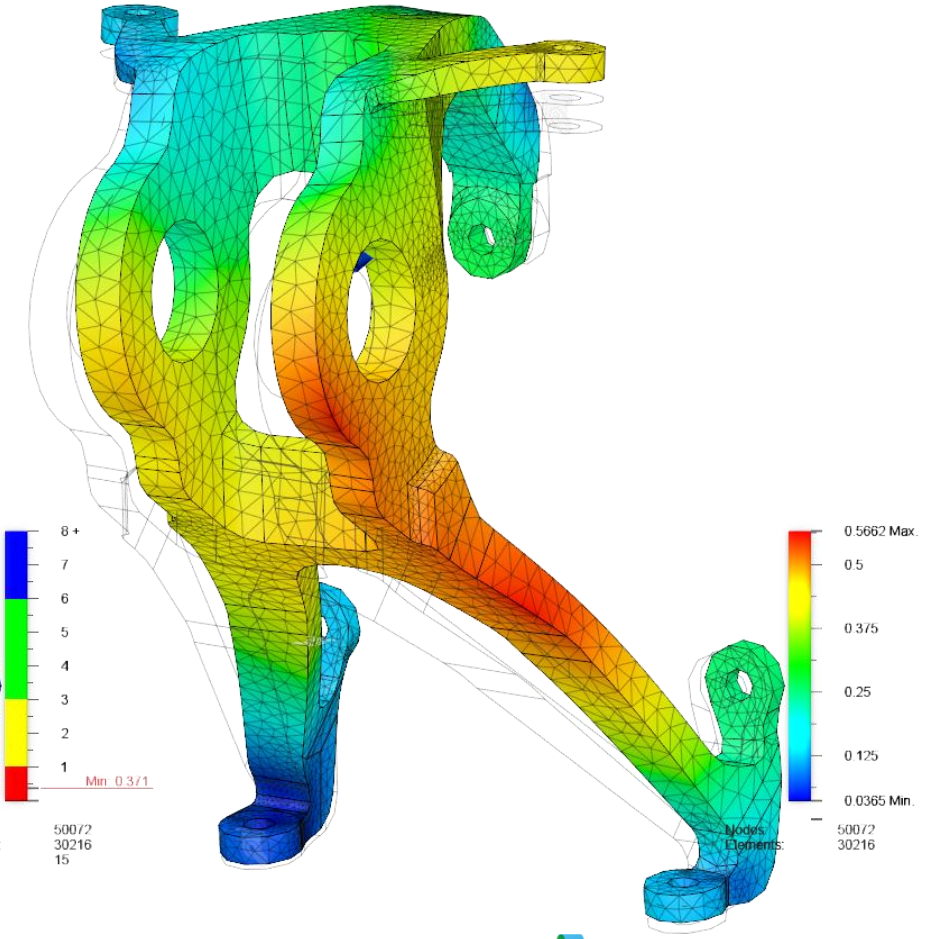


Shape Optimisation

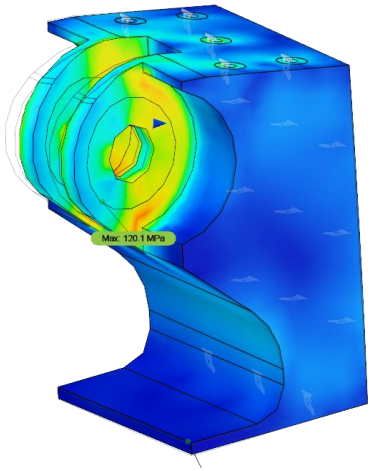
Redesign & Validation

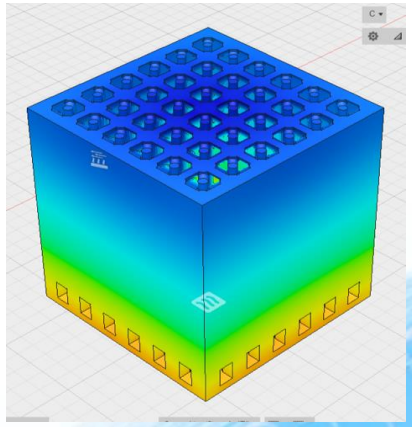


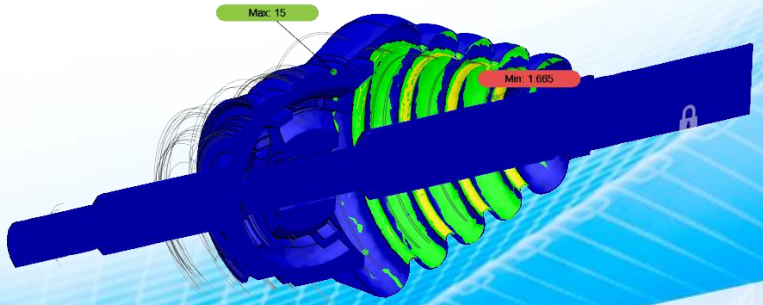


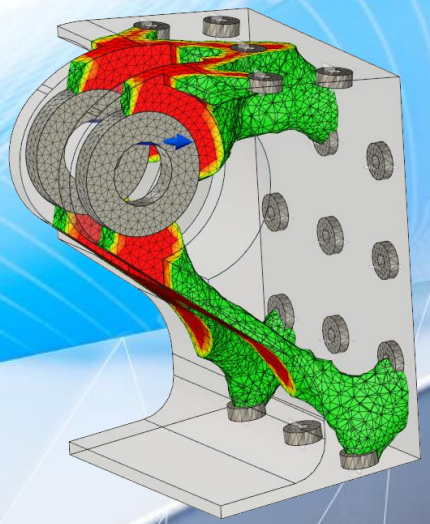


Summary



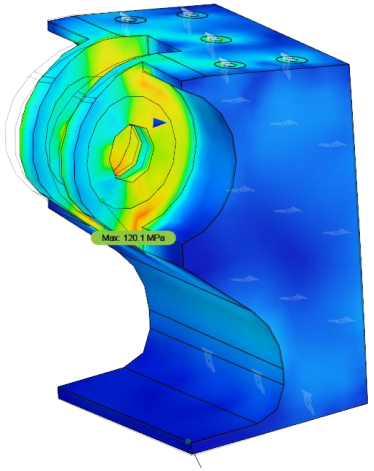


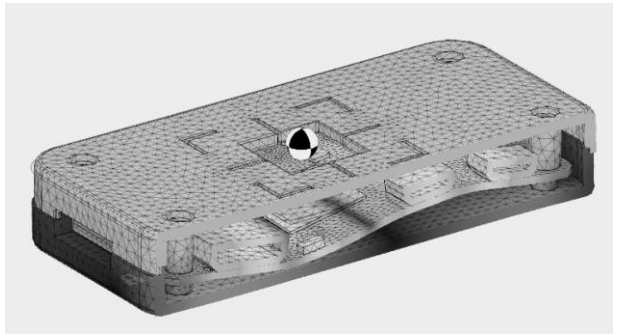


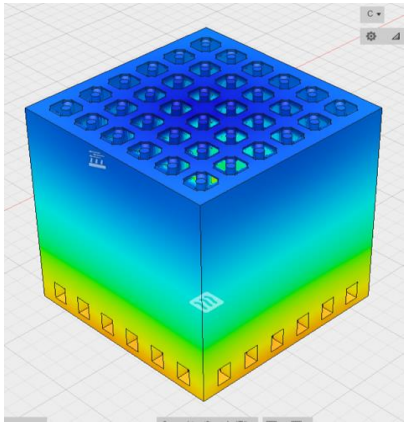


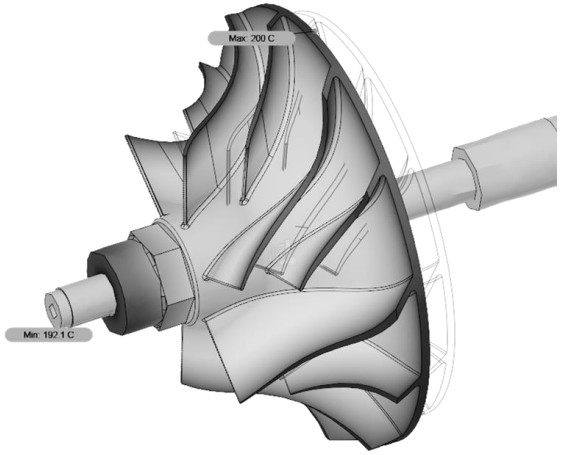
Summary

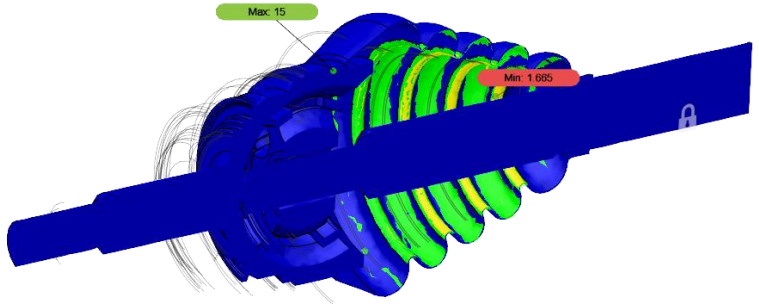
Fusion 360 Simulation Use cases

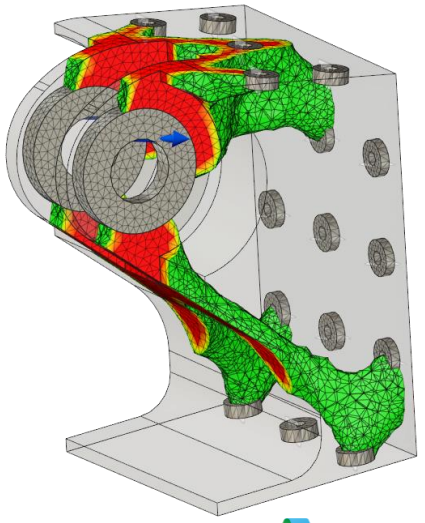












AUTODESK®

Make anything.