

# **SIMSOLID QUICK OVERVIEW**

## **MODULE 3 – STRUCTURAL LINEAR ANALYSIS**

Sebastian Karp / Senior Application Engineer / February 19<sup>th</sup> 2021

## SimSolid Vision - A new paradigm for simulation

Altair **SimSolid** is **structural simulation** that operates directly on **original, un-simplified CAD assemblies, does not create a mesh..**

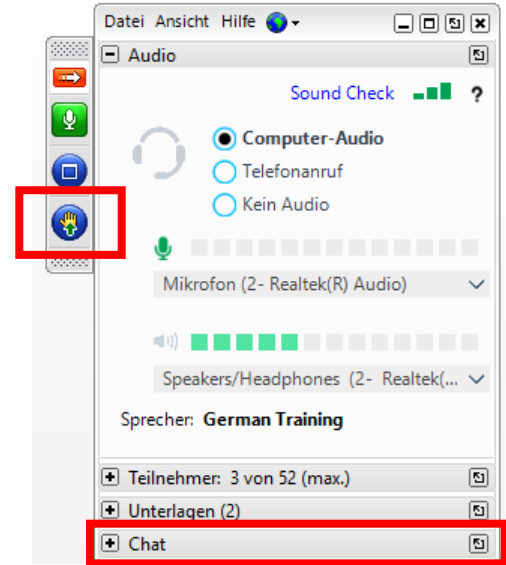
..and provides results in **seconds to minutes**

# Quick Overview Series

- Module 1: Introduction February 5th 2021
  - Module 2: User Interface + Modal Analysis February 12th 2021
  - **Module 3: Linear Analysis** February 19th 2021
  - Module 4: Non-Linear Analysis February 26th 2021
  - Module 5: Dynamic Analysis March 5th 2021
  - Module 6: Thermal and SimSolid news March 12th 2021
  - Module 7: Inspire/SimSolid Solver March 19th 2021
- (all Fridays)

# Organisational

- Session is recorded
- Q/A-block at end of session (not recorded)
- Raise hand and audio will be activated or use chat for questions
- Combined presentation of all modules can be shared



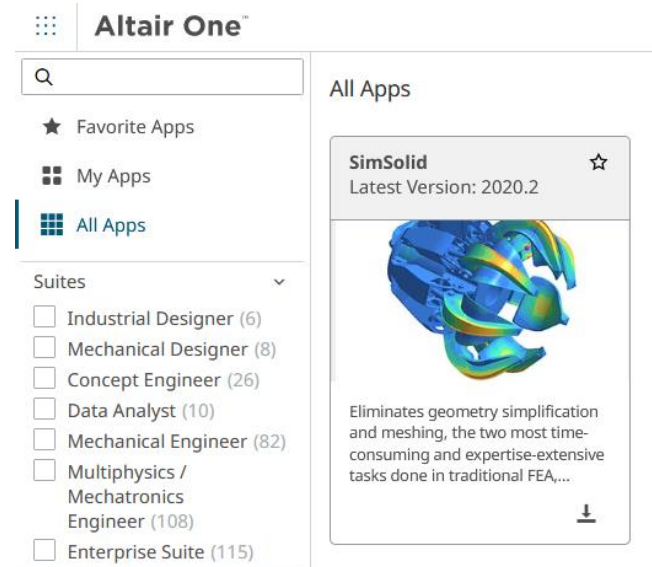
# Have a look at...

- Contact us for later communication etc. - [trainings@altair.de](mailto:trainings@altair.de)
- Check out [www.altair.com/SIMSOLID](http://www.altair.com/SIMSOLID)



**What Customers Are Saying About SimSolid**

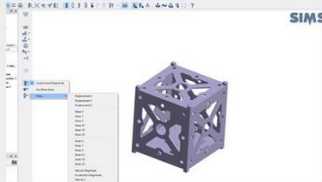
- Download - [www.altairone.com](http://www.altairone.com)



# Have a look at...

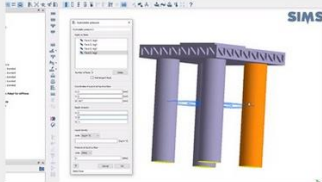
- [www.altair.de/resource/altair-simsolid-tutorial-projects](http://www.altair.de/resource/altair-simsolid-tutorial-projects)

All SimSolid Demo Models, Training Materials, and Tutorials



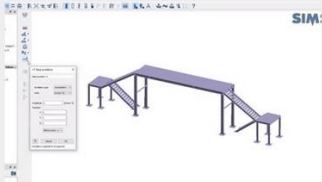
**Altair SimSolid - ERP with Constant Loading Tutorial**  
Learn how to perform a dynamic frequency response...

Tutorials



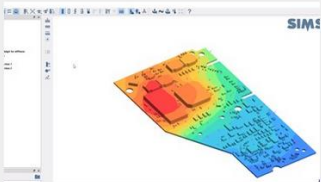
**Altair SimSolid - Hydrostatic Pressure Tutorial**  
Learn how to simulate an hydrostatic pressure in Altair...

Tutorials



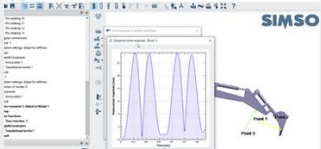
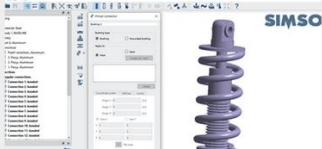
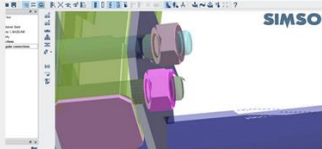
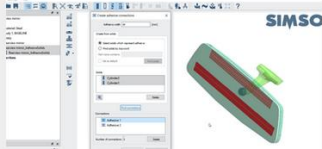
**Altair SimSolid - Random Response Analysis Tutorial**  
Learn how to perform a random response analysis in Altair...

Tutorials



**Altair SimSolid - Thermo-structural Coupling Tutorial**  
Learn how to perform thermo-structural coupling analyses in...

Tutorials



# POLL

# STRUCTURAL LINEAR ANALYSIS

# SimSolid Simulation Capabilities



## Solutions

- Modal
- Linear Statics
- Nonlinear Statics
- Frequency Response
- Linear Transient
- Random Response
- Thermal
- Thermal-Stress
- Inertia Relief
- Bolt Pretension
- Linear Superposition
- **Partial dynamic response**
- **Fatigue Analysis**



## Materials

- Isotropic
- Elastoplastic
- Rigid
- **Fluid bodies**
- User Extensible



## Connections

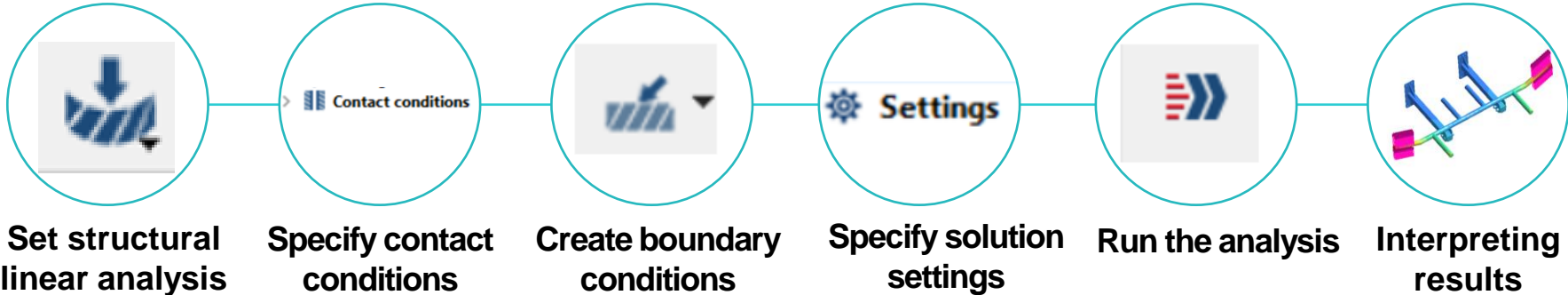
- Auto-connections
- Bonded, Sliding and Separation with Friction
- Bolts
- Spot Welds
- **Solid Seam Welds**
- Bushings
- Rivets
- Virtual Connectors
- **Adhesives**
- **Joints**



## Results

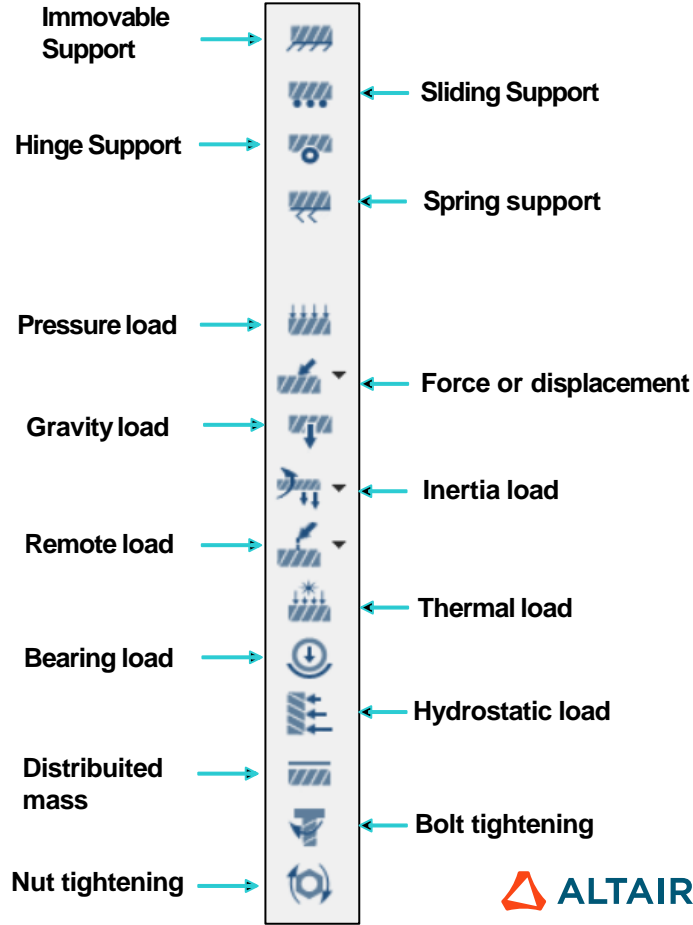
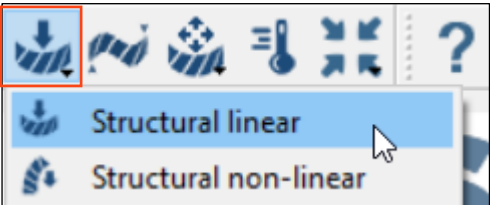
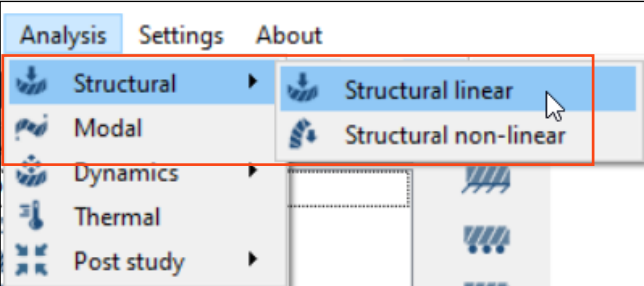
- Contours and Animations
- Displacement, Stress, Strain, **Velocity, Acceleration, ERP**
- Frequency and Mode Shapes
- XY Plots
- Modal Participation Factors
- Forces: Reaction, Contact, Bolts and Welds
- Safety Factors
- Contact Responses
- Strain Energy Density

# Set Structural Linear Analysis



# Set Structural Linear Analysis

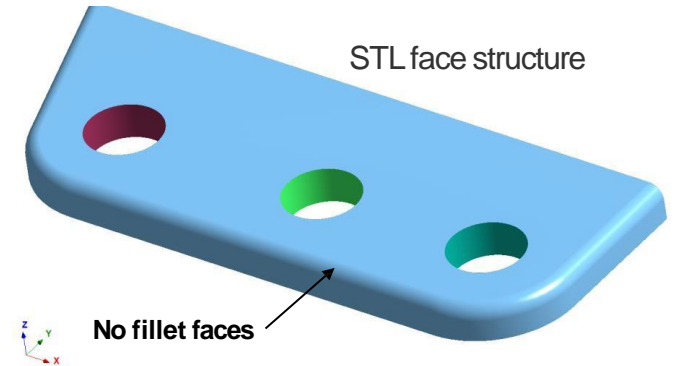
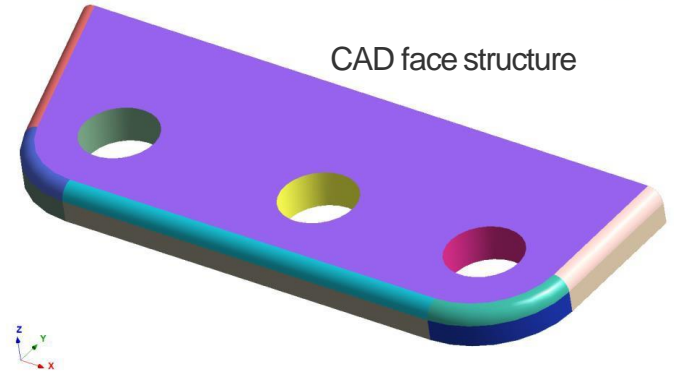
Select from either the **Analysis** menu or **main toolbar**.



# Geometry Considerations

**SimSolid** does not import CAD surface or solid geometry.

- From **CAD** (SolidWorks, Fusion 360, Onshape, ecc)
  - Full CAD hierarchical assembly tree structure used
  - CAD part faces used (preferred)
  - Facets are based on CAD add-in faceting parameters
- From **STL**
  - Multi-body STL used. Flat assembly tree structure only
  - **SimSolid** determined part face structure based on surface curvature (will miss fillet faces, see example on the right)
  - Facets are based on STL file export parameters (must take care as some CAD system export poor quality STL)

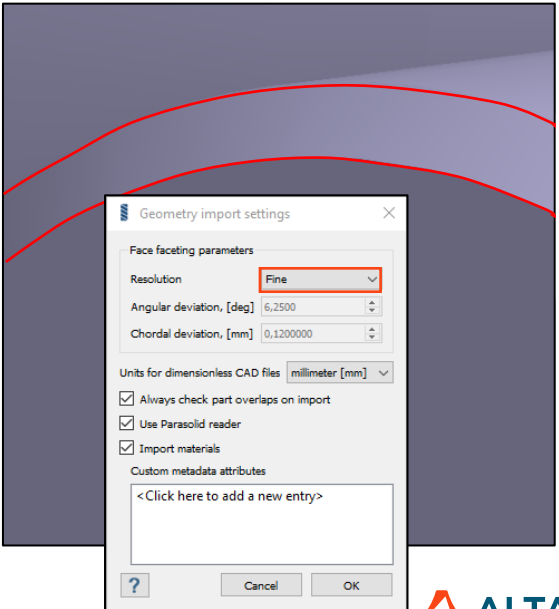
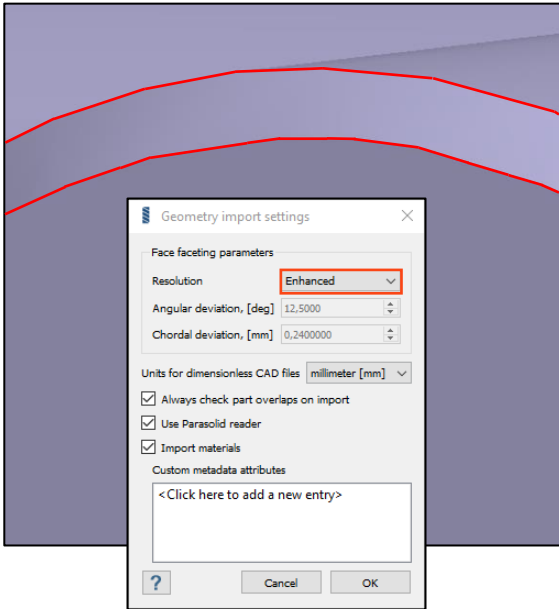
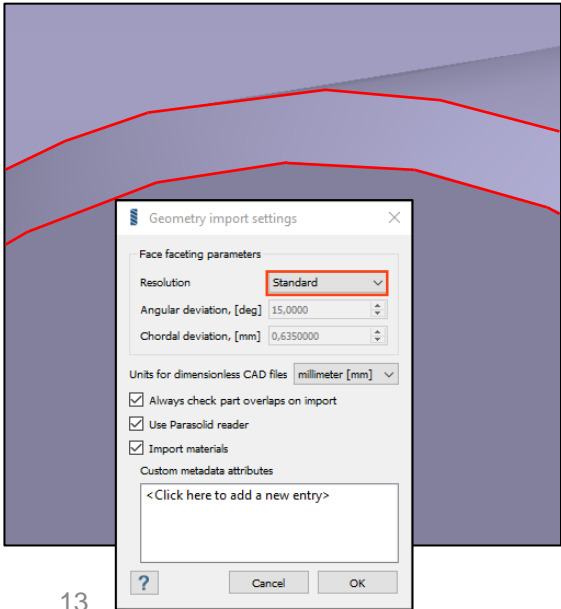


# Geometry Considerations

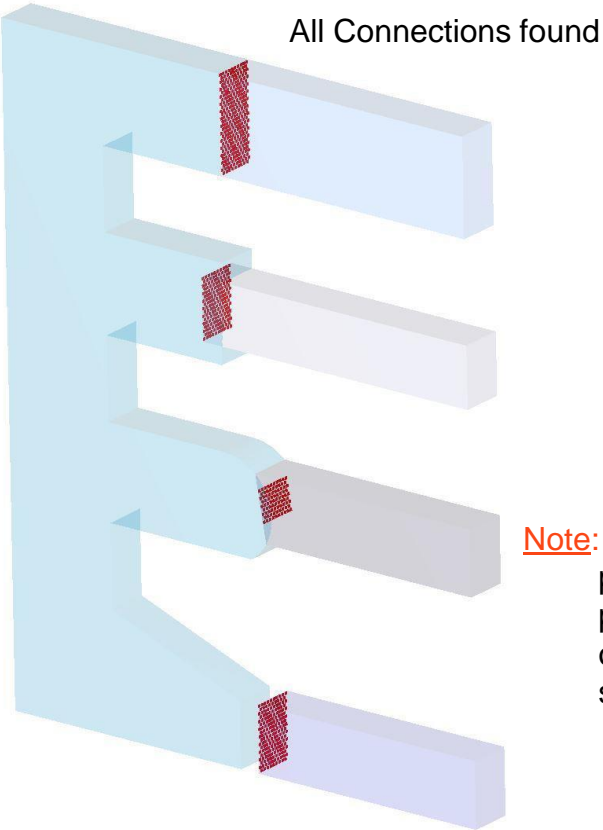
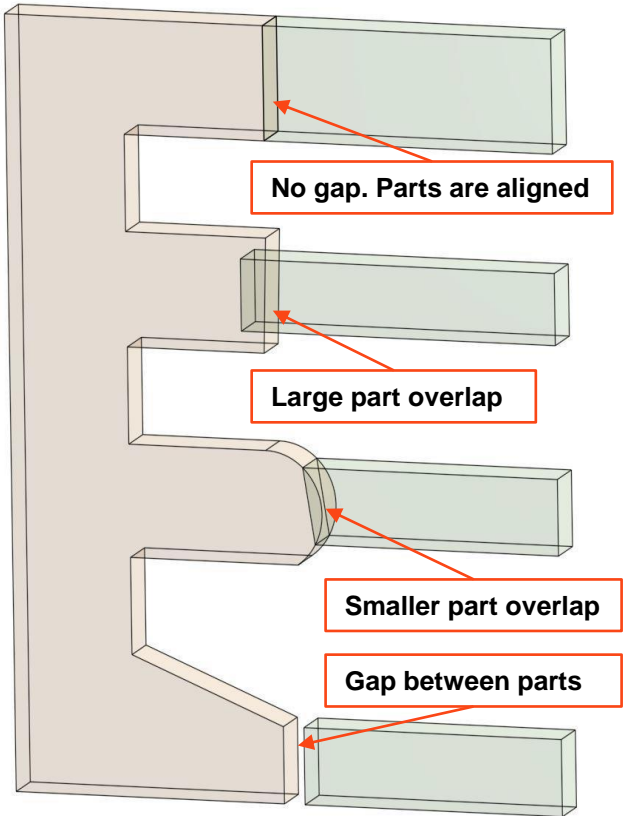
## Facet Settings - Resolution

**4 levels:** standard (default), enhanced, fine, custom.

- Increasing the level of resolution the run time increases, so should be used only when necessary.



# Examples of Permitted Connections



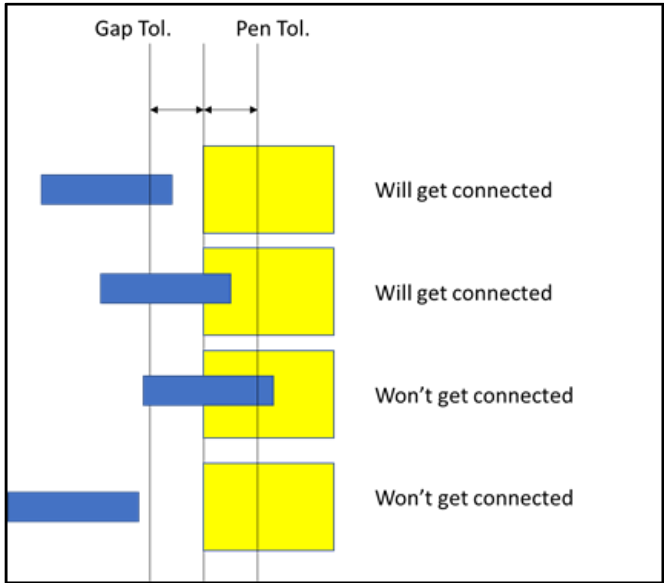
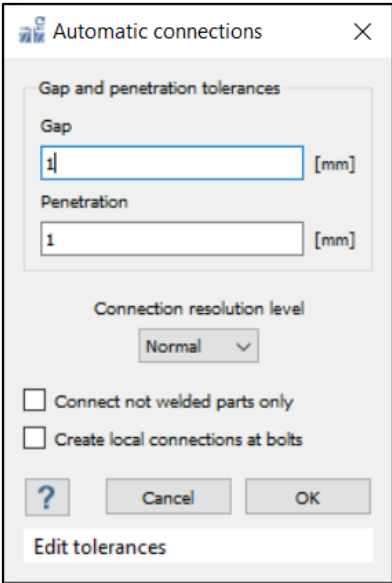
**Note:** While these contact are permitted, it is not a good practice to have excessive overlap in areas where stress detail is desired.

# Automatically Create Connections

**Connections** can be automatically found by SimSolid using the **Automatic connections** button on the Connections toolbar.

## Dialog Settings

- **Gap and penetration tolerances** is the search distances that will be used. Best practice is to set as small a value as possible to still find the connection.
- **Domain Resolution level (Normal, Increased, High)** sets the precision of the connection search algorithm. In most cases **Normal** will be fine. Only needs to be changed for extreme aspect parts - very thin, very long, etc.



# Automatically Create Connections

## Resolution

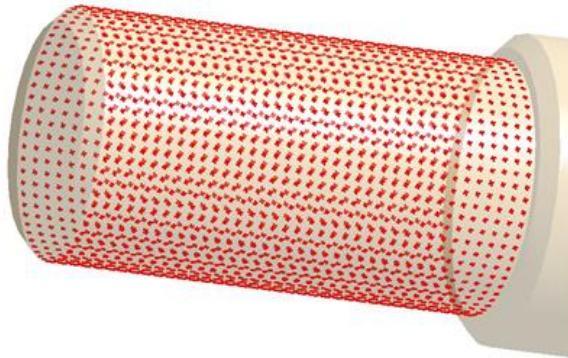
Connections resolution is visualized by icon density.

- Make sure there is adequate coverage in thin or curved regions.
- Connections that are too weak may lead to mechanisms (rigid body motion).

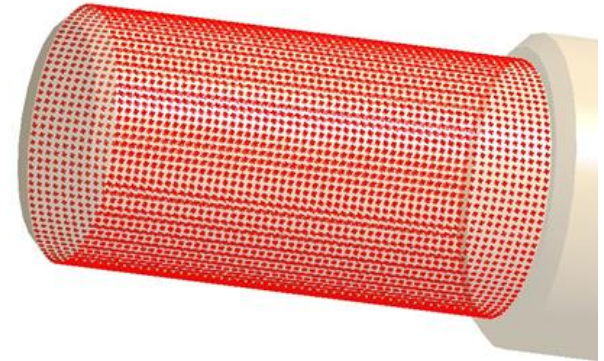
NORMAL



INCREASED



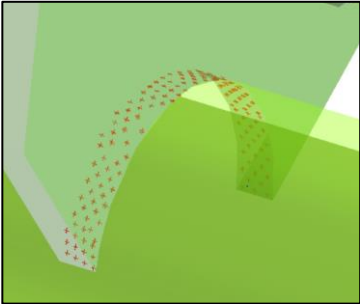
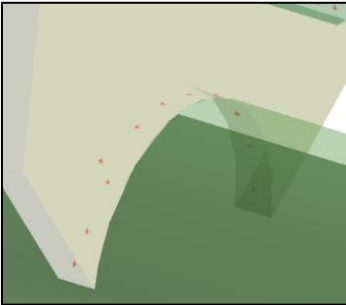
HIGH



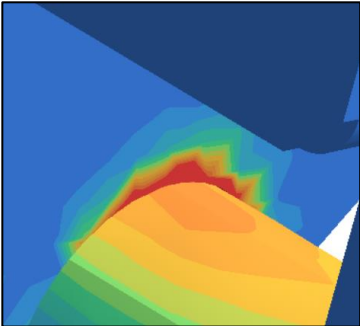
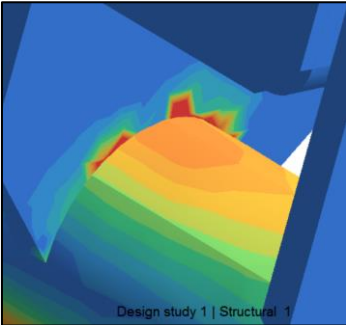
# Automatically Create Connections

## Resolution

Stress evaluation around connections require higher resolution



Normal VS Higher Resolution



**Do Not** always use higher resolution as it really slows the solution without any benefit

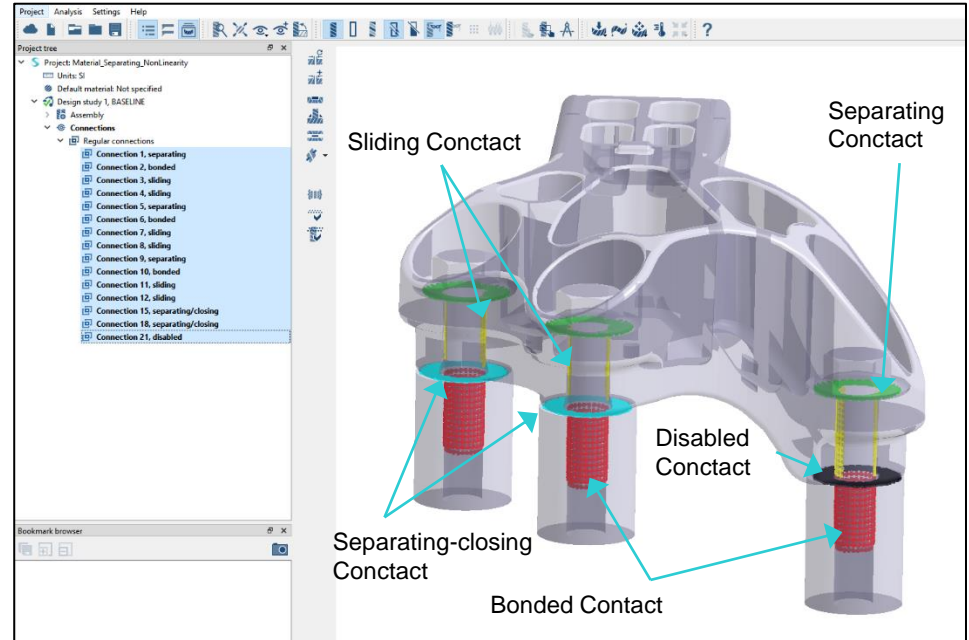
# Review Contact Conditions

Works the same as connections.

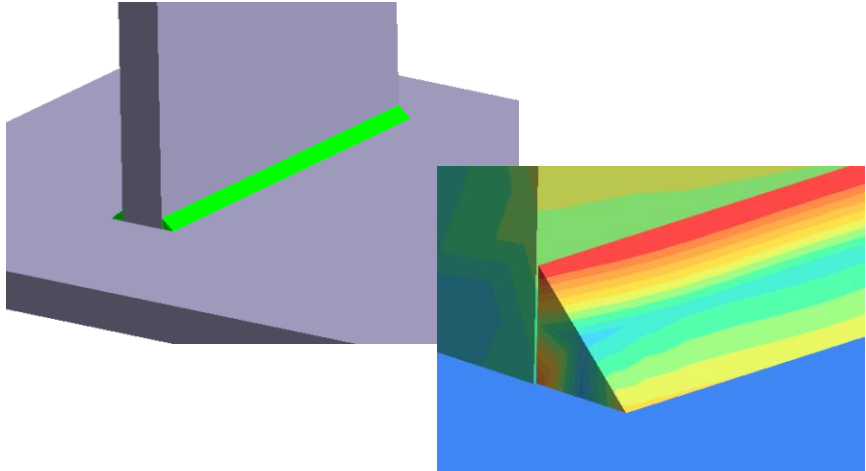
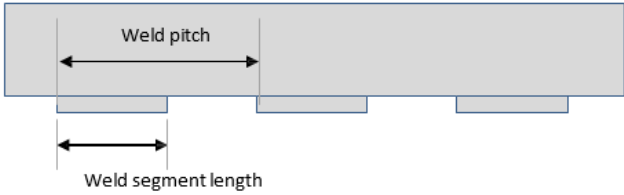
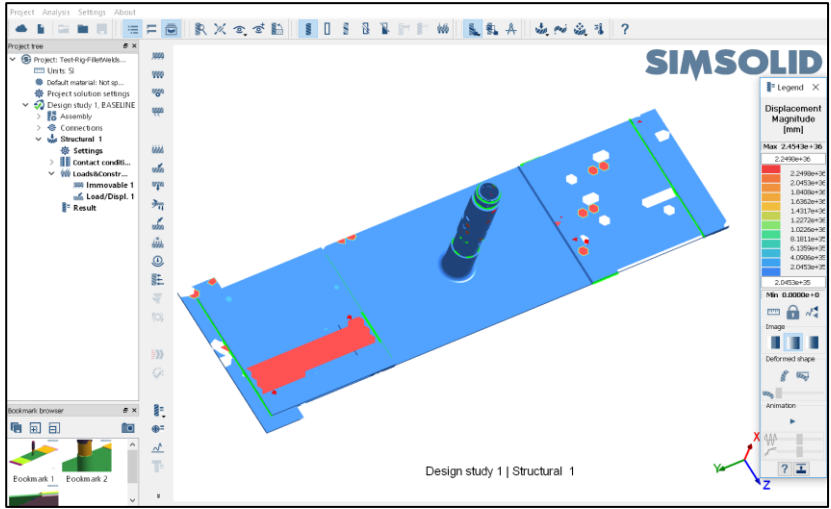
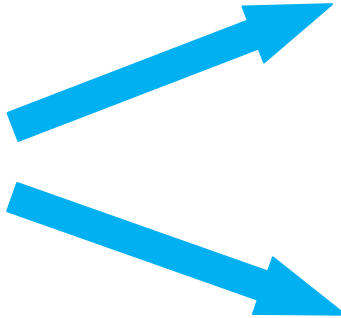
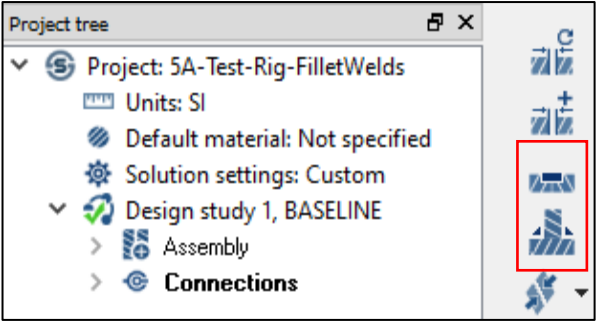
The differences are that contact conditions are color coded:

- Bonded → red
- Sliding → yellow
- Separating → green
- Disabled → black
- Separating-closing → cyan/blue

Also the contact conditions is specific to analysis.




# Spot / Seam Welds

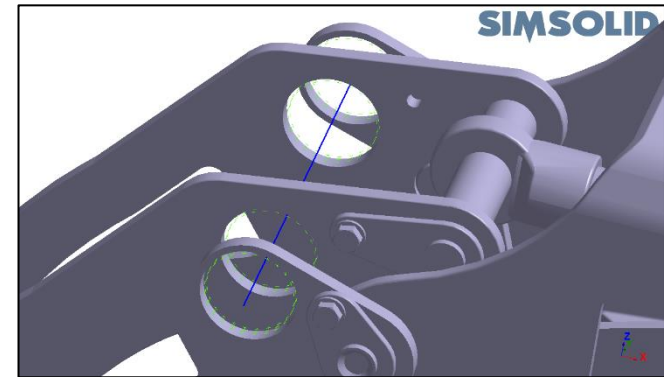
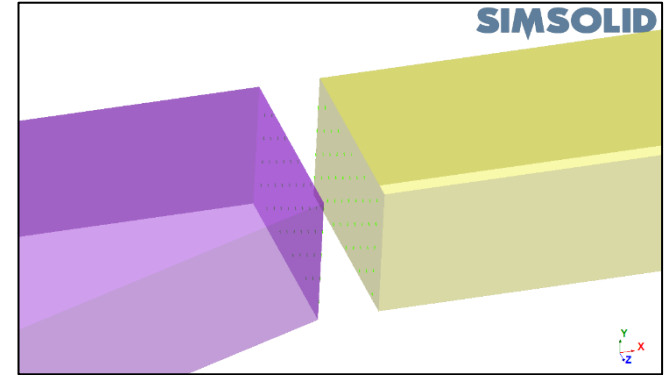
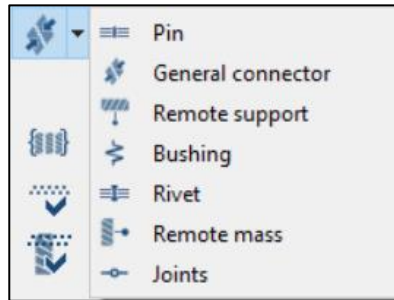


# Virtual Connectors

In SimSolid, **virtual connectors** can be applied in cases where the corresponding connecting parts are missing in the model.

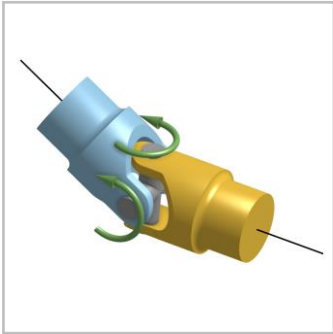
**Virtual connectors** are a rigid imaginary part created between selected faces representing the geometrical support of the connector. The type of contact interaction between a virtual connector and a real part is defined by the type of virtual connector. Read on to find out how virtual connectors are defined and used.

From the **Connection Toolbar**, click on  icon to open the pull-down menu and to visualize the kind of **virtual connectors** supported in SimSolid:

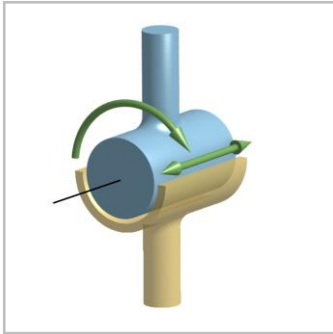


**Note:** Since **virtual connectors** create perfectly rigid boundaries, stresses along the boundary faces can have local concentrations.

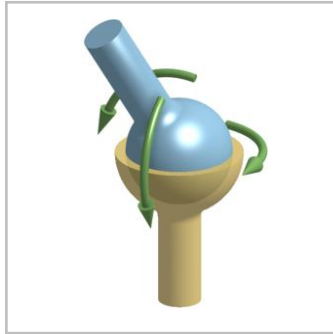
# Virtual Joints



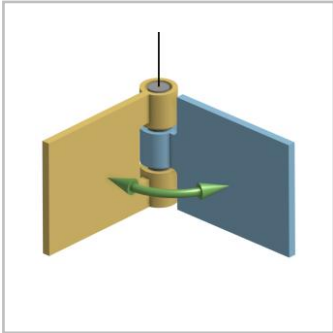
**Universal**  
2 DOF, 2 axes



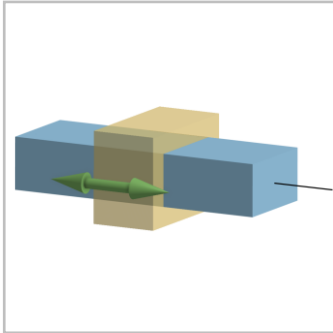
**Cylindrical**  
2 DOF, 1 axis



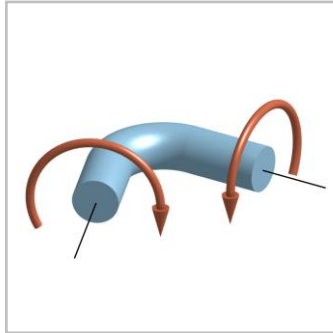
**Ball**  
3 DOF, no axes



**Hinge**  
1 DOF, 1 axis



**Linear guide**  
1 DOF, 1 axis



**Flexible shaft**  
5 DOF, 2 axes

Virtual connector

Ball joint 1

Joint type

- Ball
- Hinge
- Cylinder
- Linear guide
- Universal
- Flexible shaft

Apply to

- Face
- Spot

Create new spot

Face 11, Trim/Extend...

Delete

Joint center

X: 700 [mm]

Y: 542.5 [mm]

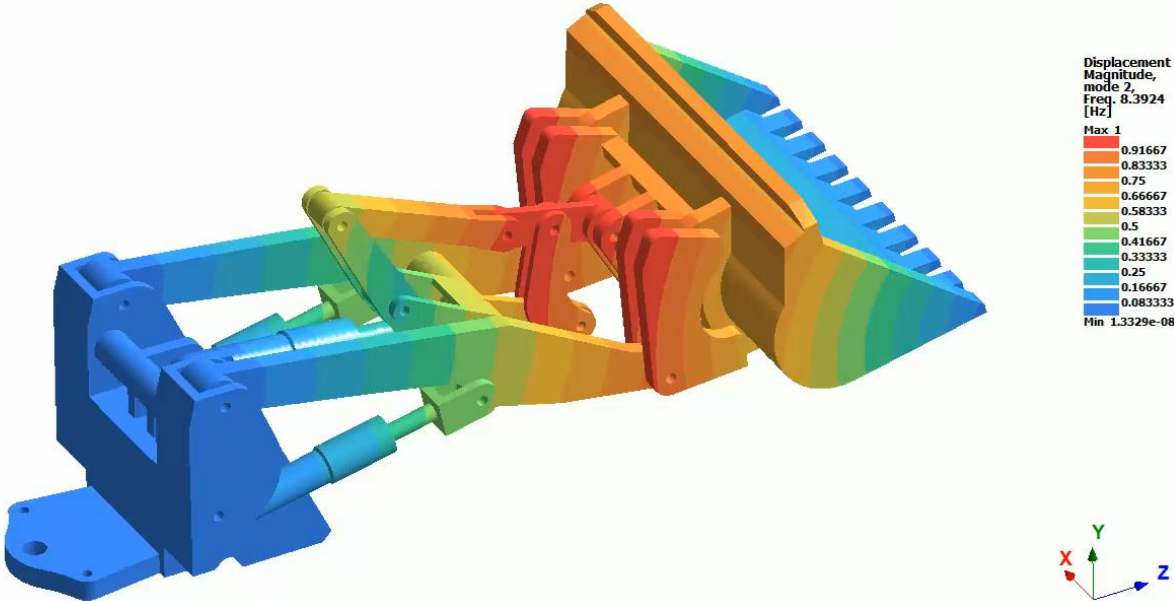
Z: -1450 [mm]

? Close OK

Select faces to connect

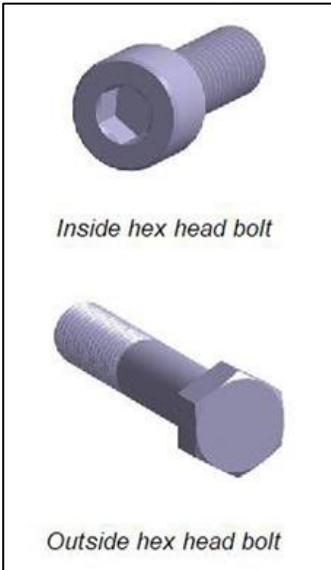
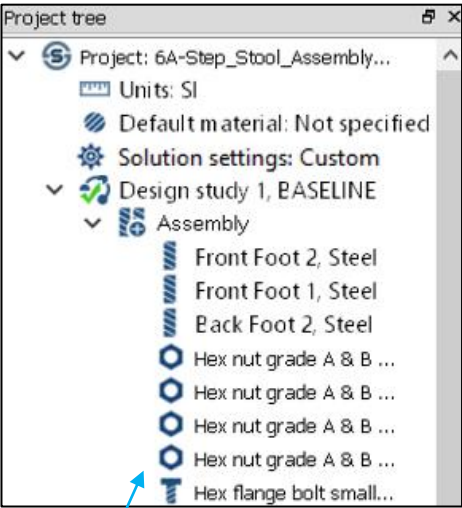
# Virtual Joints

# SIMSOLID

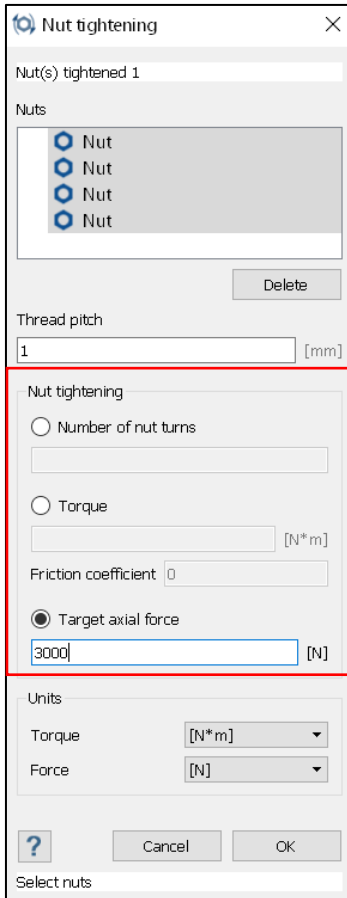
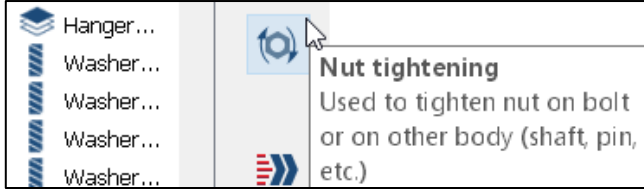
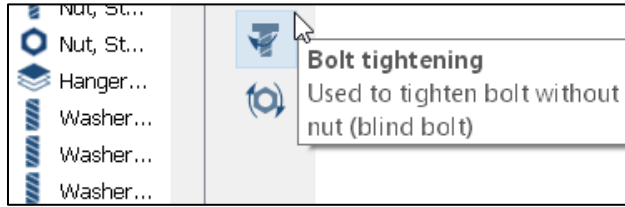


Design study 1 | Modal 1

# Bolted Connectors and Bolt/Nut-Tightening



Bolts, nuts and washers are identified by their icons

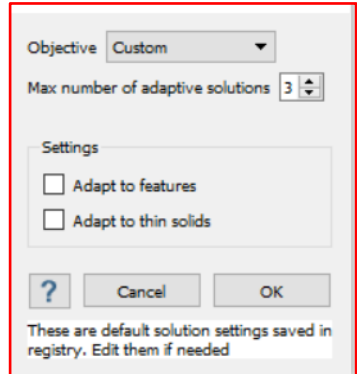
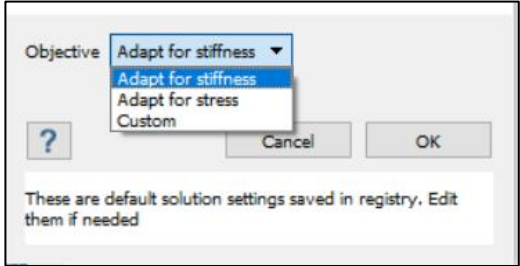


# Solution Settings

→ globally to the entire assembly or locally to individual groups of parts

- **Adapt for Stiffness** - Use this objective for general load path prediction, modes and/or thermal analysis.
- **Adapt for Stress** - Use this objective for general stress calculation.
- **Custom** - Most general objective. All controls can be individually set. Set the **max number of passes** and the setting options.

Objective →	Adapt for stiffness	Adapt for stress	Custom
<b>Primary purpose</b>	General load path prediction, reaction forces, modes and thermal	General stress prediction	Fine grained controls
<b># adaptive passes</b>	3	4	3 - 8
<b>Adapt to features</b>	OFF	ON	User defined
<b>Adapt to thin solid</b>	OFF	ON	User defined
<b>Local part adaption</b>	N/A	N/A	User defined

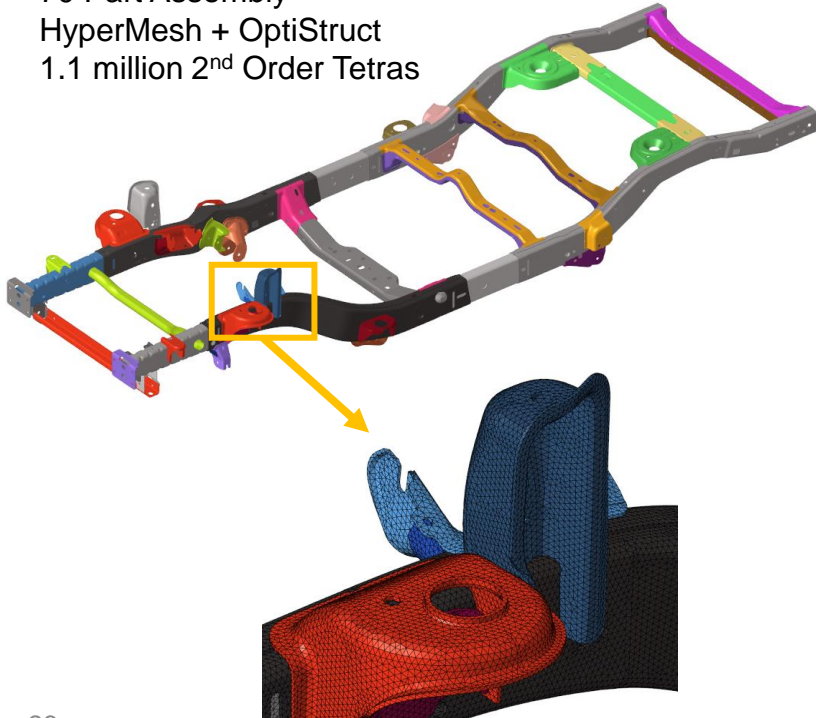


# VERIFICATION VS. FE

# Validation Example – Auto Frame

## Reference Solution

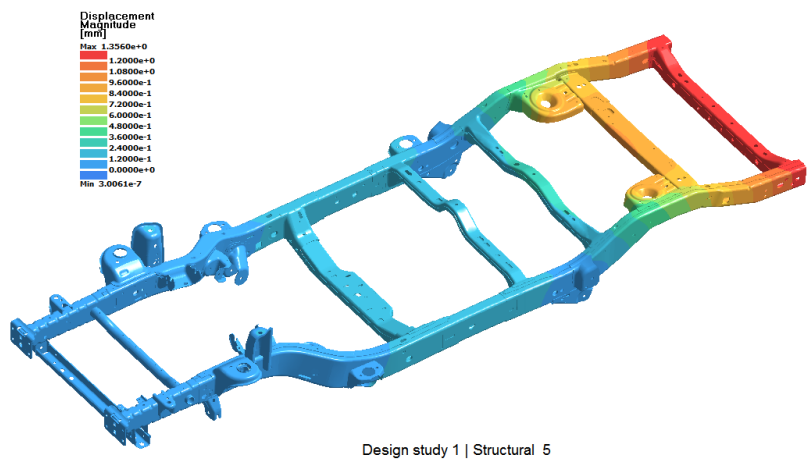
- 70 Part Assembly
- HyperMesh + OptiStruct
- 1.1 million 2<sup>nd</sup> Order Tetras



Mode	SimSolid	OptiStruct
1	33.732 Hz	33.749 Hz
2	41.501 Hz	41.992 Hz
3	59.170 Hz	59.439 Hz
4	64.644 Hz	63.811 Hz
5	79.403 Hz	80.317 Hz
6	80.094 Hz	81.729 Hz
7	94.660 Hz	95.598 Hz
8	100.35 Hz	103.20 Hz
9	114.48 Hz	117.88 Hz

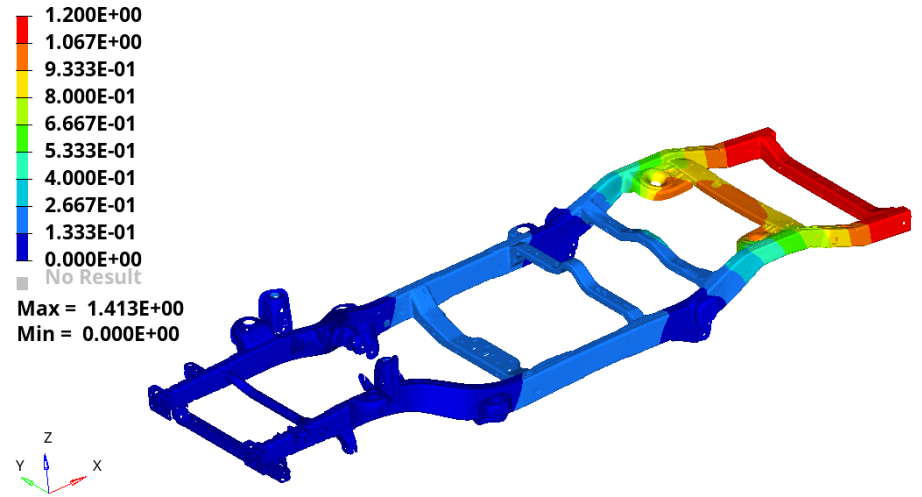
# Frame – linear static analysis

SimSolid Max Disp ~ 1.36 mm



6 solution passes, Adapt to thin solids

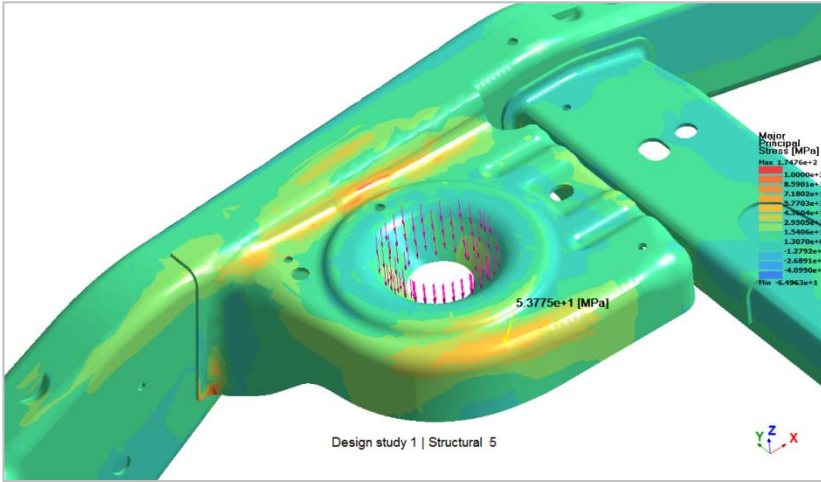
OptiStruct Max Disp ~ 1.41 mm



1.1 million 2<sup>nd</sup> Order Tetras

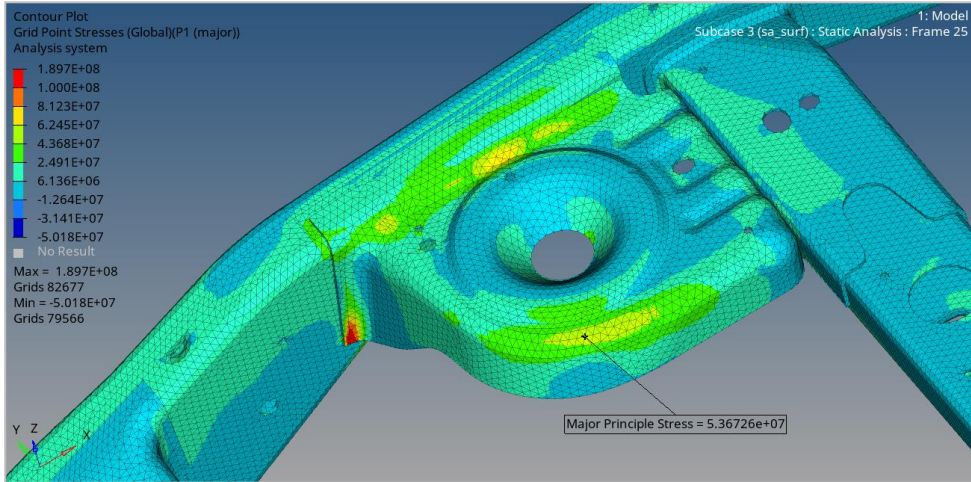
# Frame – linear static analysis

## SimSolid Major Principal Stress



6 solution passes, Adapt to thin solids

## OptiStruct Major Principal Stress

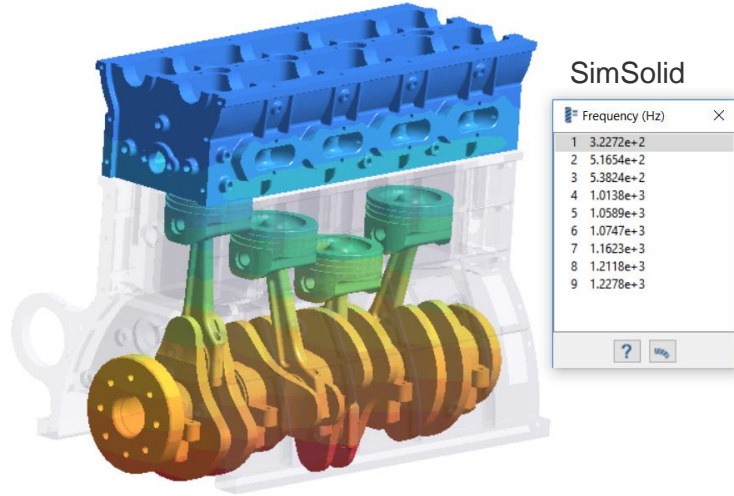
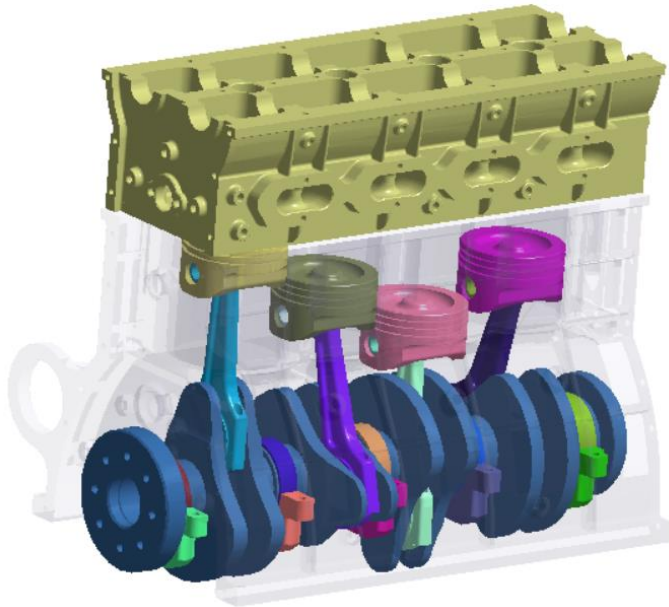


1.1 million 2<sup>nd</sup> Order Tetras

# Validation Example – Engine

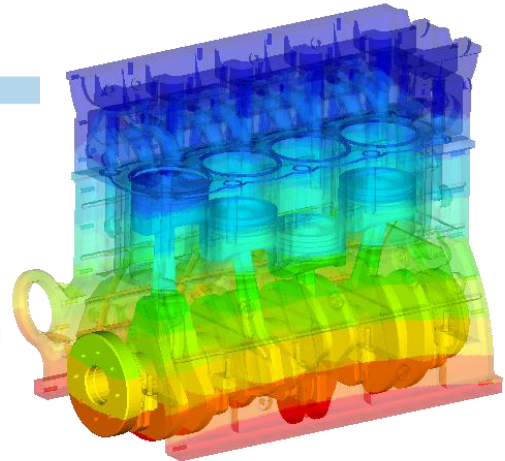
## Reference Solution

- 44 Part Assembly
- HyperMesh + OptiStruct
- 700,000 2<sup>nd</sup> Order Tetras
- Bonded Connections

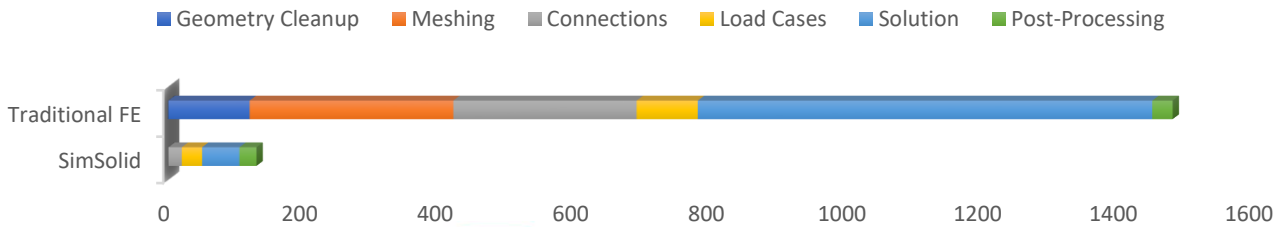


## OptiStruct

- Mode 1 - F = 3.146697E+02
- Mode 2 - F = 5.153883E+02
- Mode 3 - F = 5.370159E+02
- Mode 4 - F = 1.018273E+03
- Mode 5 - F = 1.049172E+03
- Mode 6 - F = 1.061735E+03
- Mode 7 - F = 1.160738E+03
- Mode 8 - F = 1.197509E+03
- Mode 9 - F = 1.255972E+03
- Mode 10 - F = 1.341970E+03



# Validation example - Piston assembly

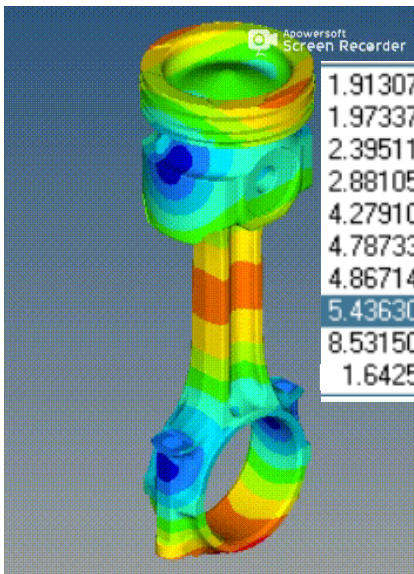


## Piston-Connecting Rod Assembly

- 8 Parts
- 2 Bolts and Nuts

## Modal Analysis

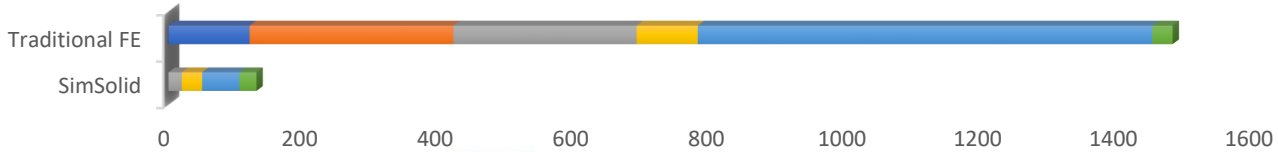
- 10 Modes



Mode	Frequency (Hz)
1	0.0000e+0
2	0.0000e+0
3	0.0000e+0
4	5.3010e-3
5	2.1902e-2
6	3.0078e-2
7	4.7826e+2
8	5.4111e+2
9	8.5070e+2
10	1.6355e+3

# Validation example - Piston assembly

■ Geometry Cleanup ■ Meshing ■ Connections ■ Load Cases ■ Solution ■ Post-Processing

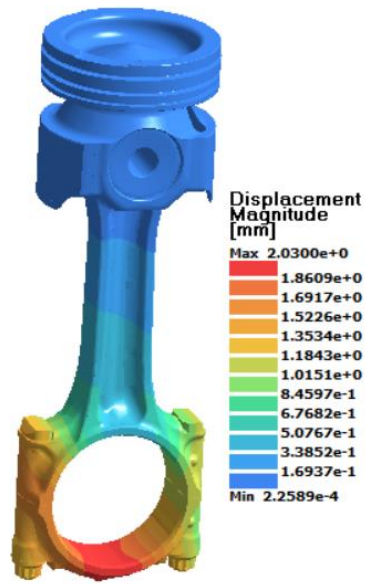
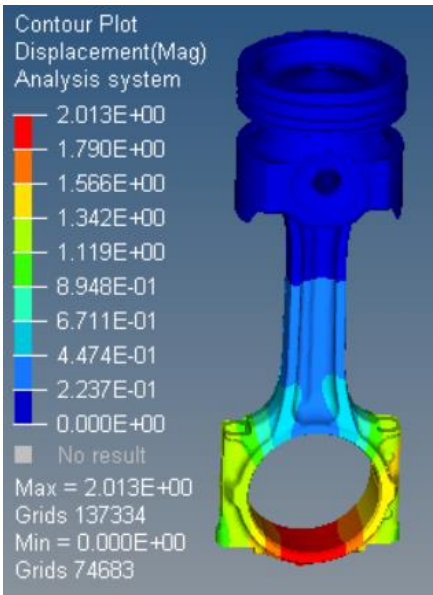


## Piston-Connecting Rod Assembly

- 8 Parts
- 2 Bolts and Nuts

## Linear Static Analysis

- Pressure load on the cap

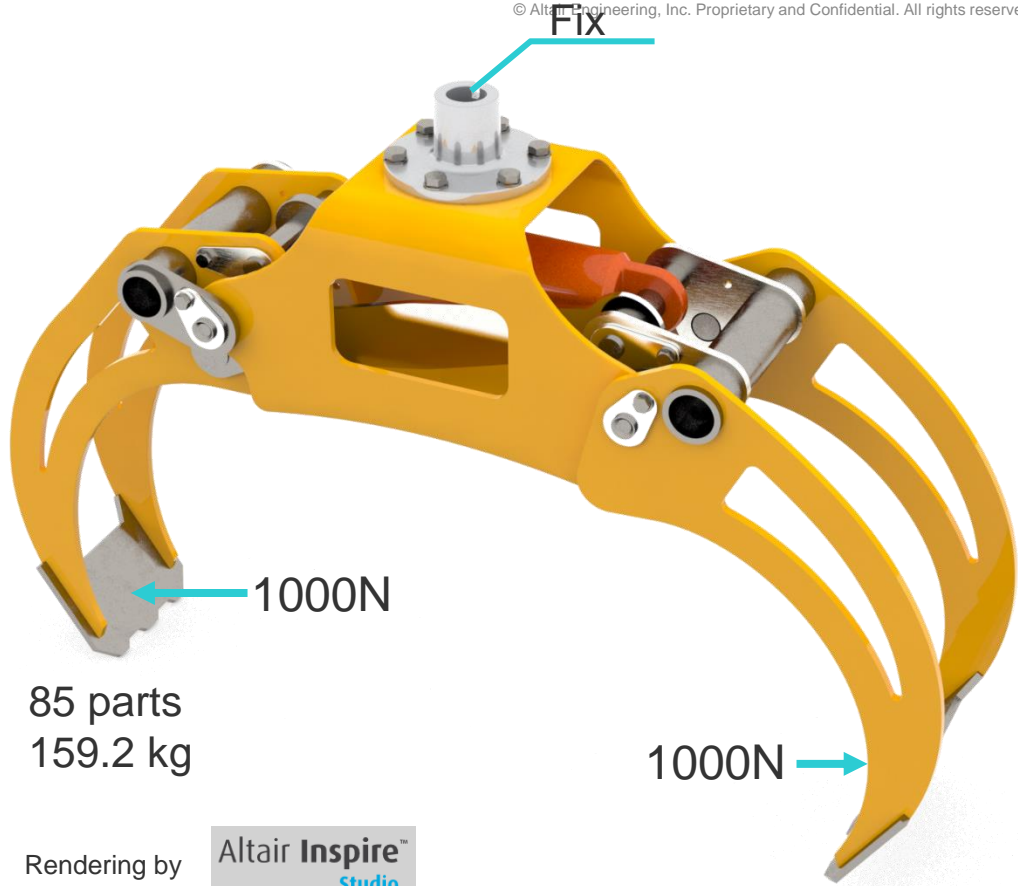


# Grapple

- Material (Steel equivalent)
  - $E=210000 \text{ MPa}$
  - $\text{Nu}=0.3$
  - $\text{Rho}=7.85\text{E-}9 \text{ ton/mm}^3$

## Linear Statics

- Load
  - $F_x = +/-1000\text{N}$
- Constraint
  - Fixed Upper hole
- Contact
  - Bonded all

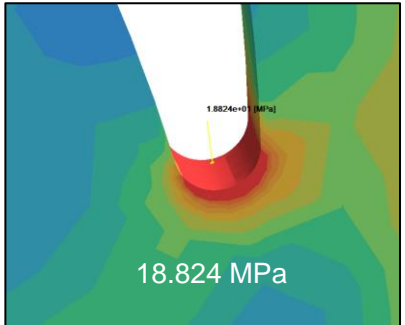
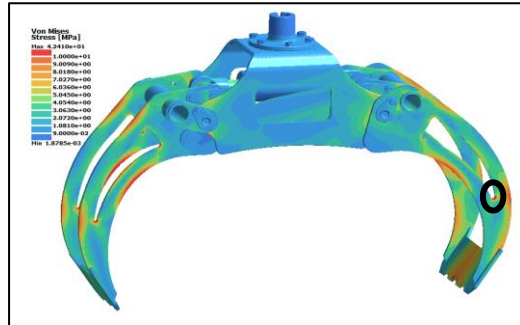
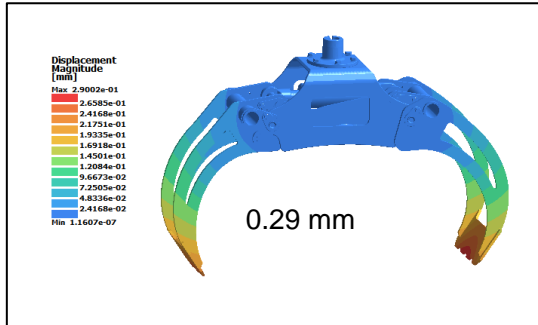


Rendering by Altair Inspire™ Studio

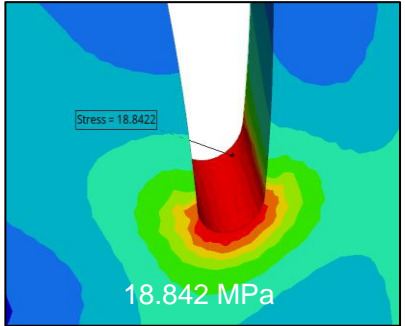
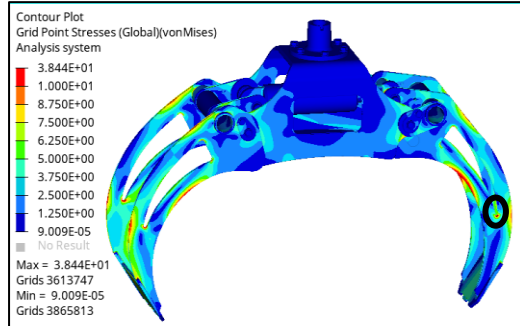
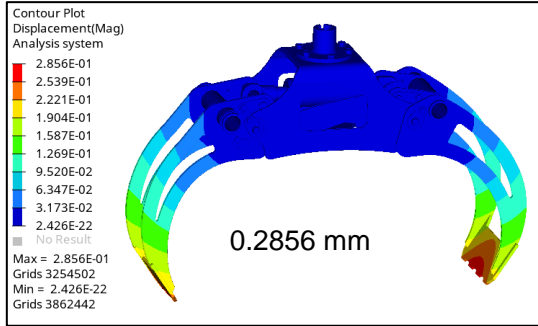
Grapple - V1.step

# Linear static analysis

SimSolid

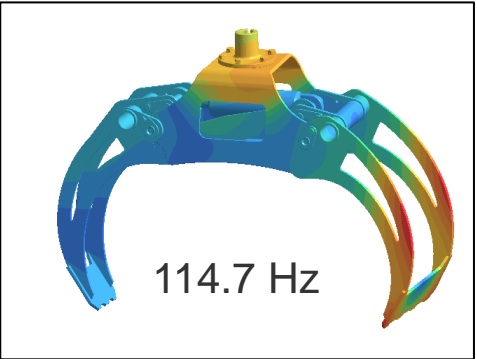
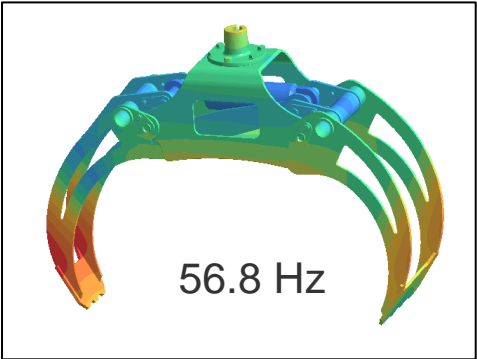
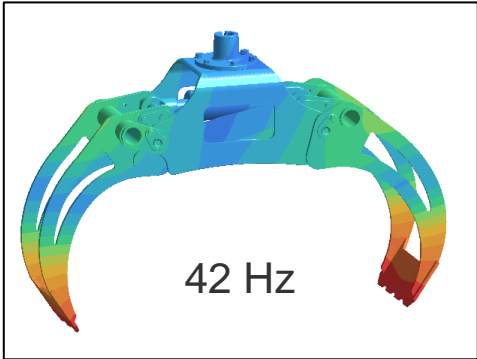


OptiStruct

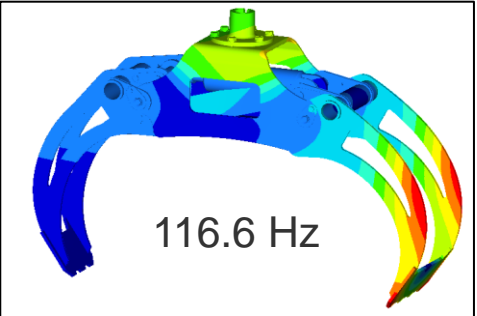
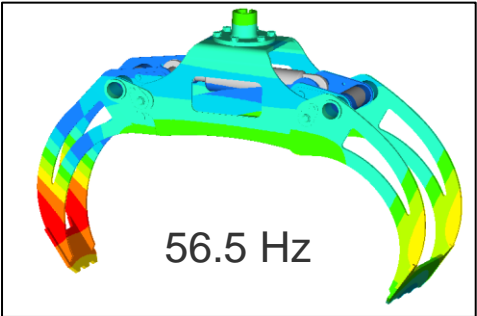
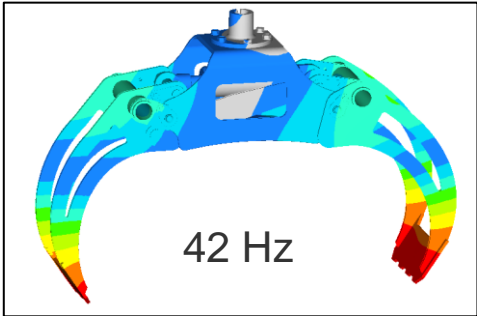


# Eigenvalue Analysis

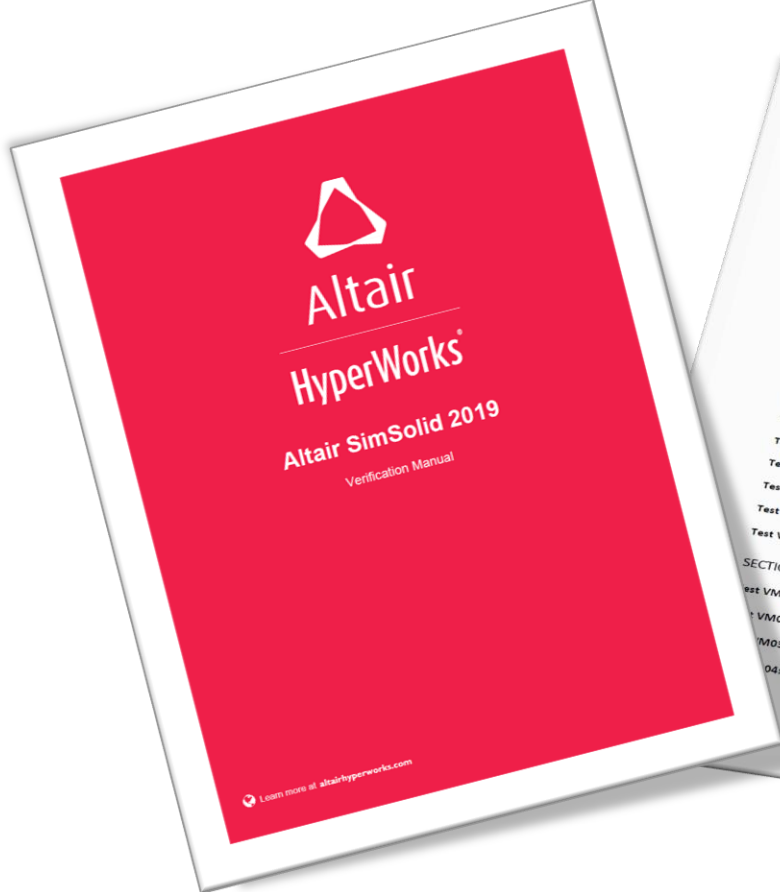
SimSolid



OptiStruct



# Verification Manual



**Table of Contents**

INTRODUCTION

- Document Purpose 7
- Test Documentation Format 7

**SECTION 1: STATIC ANALYSIS TESTS**

- Test VS01: Straight Cantilever Beam 7
- Test VS02: Twisted Cantilever Beam 8
- Test VS03: Curved Beam 8
- Test VS04: Simply Supported Thin Plate 10
- Test VS05: Clamped Thin Plate 12
- Test VS06: Tapered Solid Bar 14
- Test VS07: Cantilever Plate 16
- Test VS08: Plate with a Hole 18
- Test VS09: Scordella-Lo Roof 20
- Test VS10: Curved Thick Elliptical Plate 22
- Test VS11: Raasch Challenge 24
- Test VS12: Notched Plate 26
- Test VS13: Simply supported beam with mid-span load 28
- Test VS14: Skew Plate Under Pressure 29
- Test VS15: Stress Concentration of Filleted Bar 31

**SECTION 2: MODAL ANALYSIS TESTS**

- Test VM01: Free Vibration of a Wedge 32
- Test VM02: In-plane Vibration of a Cantilever Plate 35
- Test VM03: Cylindrical Shell 37
- Test VM04: Free Thin Square Plate 37

- 38
- 39
- 40

# NAFEMS about SimSolid

## SimSolid in the News



Benchmark	Description	Quantity	Target Solution	SimSolid	
				Results	Discrepancy
1	Pressure component	Von Mises stress	534MPa	532MPa	<1%
2	Coil spring	Spring rate	20.8N/mm	20.76N/mm	<1%
3	Skew plate	Maximum principal stress	0.82MPa	0.82MPa	<1%
4	Plate with hole	Maximum principal stress	314MPa	325.7MPa	3.7%
		Minimum principal stress	-114MPa	-117.9MPa	4.2%
5	U-shaped notch	Maximum principal stress	48.2MPa	47.6MPa	1.2%
6	Cantilevered plate	Mode 1	0.42Hz	0.42Hz	<1%
		Mode 2	1.02Hz	1.02Hz	<1%
		Mode 3	2.58Hz	2.56Hz	<1%
		Mode 4	3.29Hz	3.27Hz	<1%
		Mode 5	3.75Hz	3.72Hz	<1%
7	Cantilever under pure bending	Sxx	221MPa	221.7MPa	<1%
		Uz	0.0247m	0.0247m	<1%
8	Cantilever realistic support	S <sub>VM</sub>	356.5MPa	366.5MPa	2.8%

A summary of results for all benchmarks(NAFEMS)

Check for Other Media Testimonials:

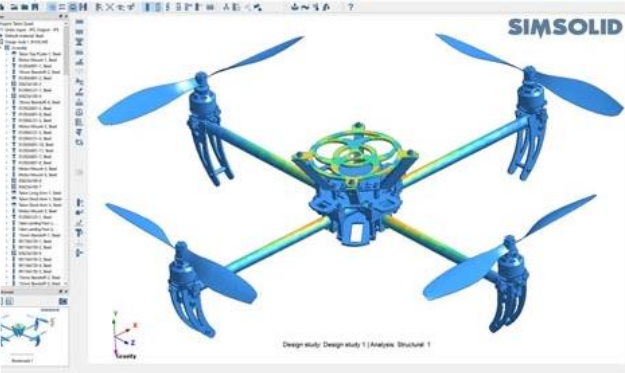
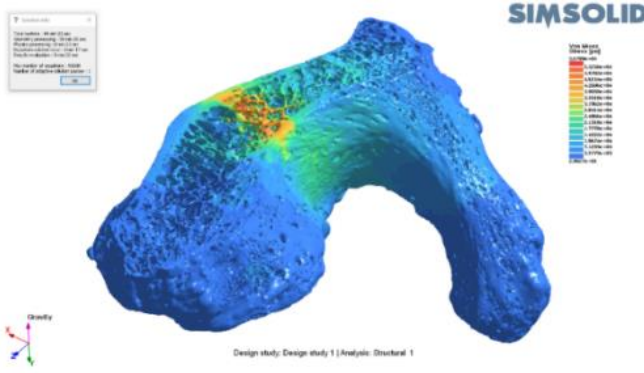
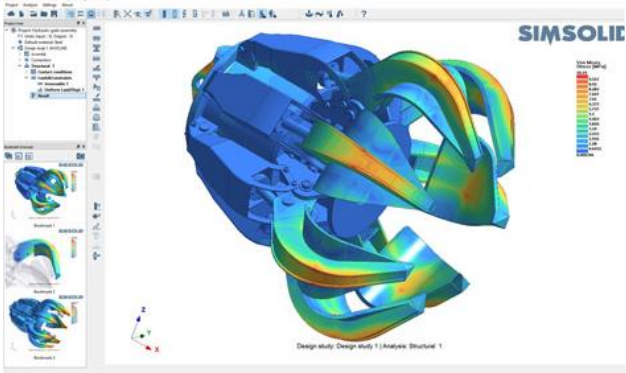


# JOIN THE SIMULATION REVOLUTION

Fast, Easy to Use & Accurate

Expand What is Possible to Solve

Enables Simulation Driven Design



## Quick Overview Series

- Module 1: Introduction February 5th 2021
- Module 2: User Interface + Modal Analysis February 12th 2021
- Module 3: Linear Analysis February 19th 2021
- Module 4: Non-Linear Analysis February 26th 2021
- Module 5: Dynamic Analysis March 5th 2021
- Module 6: Thermal and SimSolid news March 12th 2021
- Module 7: Inspire/SimSolid Solver March 19th 2021

(all Fridays)

- Contact: [trainings@altair.de](mailto:trainings@altair.de)
- or give feedback after meeting finished

# QUESTIONS / ANSWERS



# THANK YOU

[altair.com](https://altair.com)



#ONLYFORWARD