

# **SIMSOLID QUICK OVERVIEW**

## **MODULE 4 – STRUCTURAL NON-LINEAR ANALYSIS**

Sebastian Karp / Senior Application Engineer / February 26<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..**

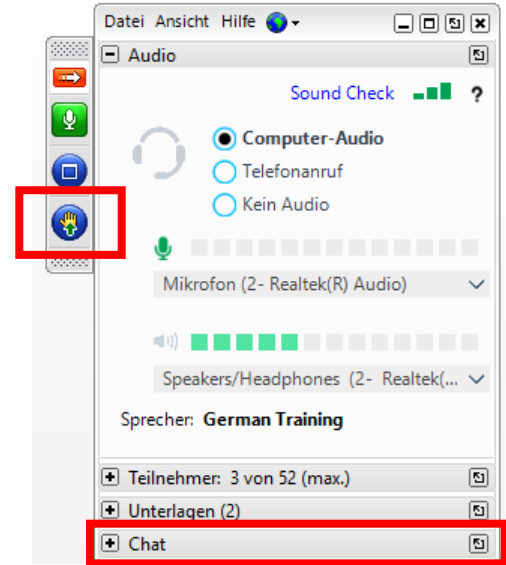
..and provides results in **seconds to minutes**

## 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



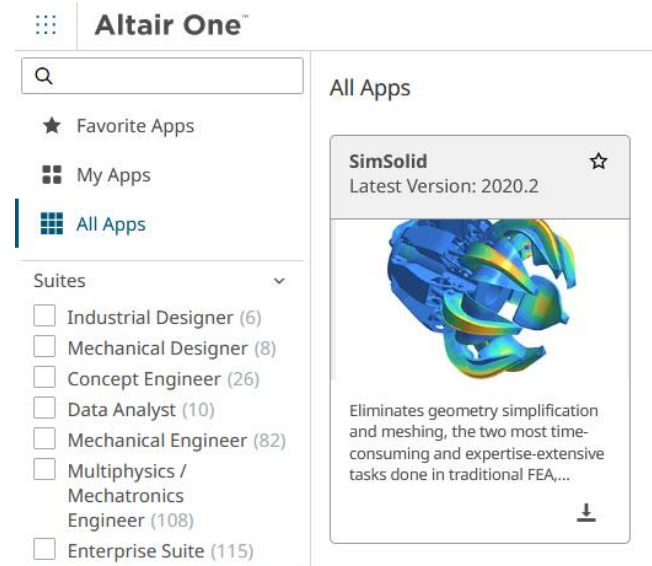
# Have a look at...

- 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**

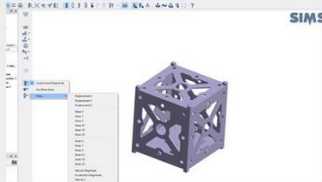
- Download - [www.altairone.com](http://www.altairone.com)



# Have a look at...

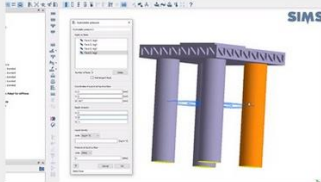
- [www.altair.de/resource/altair-simsolid-tutorial-projects](http://www.altair.de/resource/altair-simsolid-tutorial-projects)

All SimSolid Demo Models, Training Materials, and Tutorials



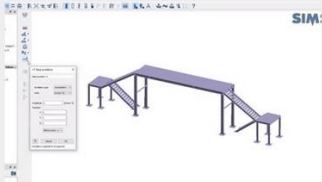
**Altair SimSolid - ERP with Constant Loading Tutorial**  
Learn how to perform a dynamic frequency response...

Tutorials



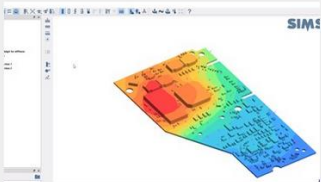
**Altair SimSolid - Hydrostatic Pressure Tutorial**  
Learn how to simulate an hydrostatic pressure in Altair...

Tutorials



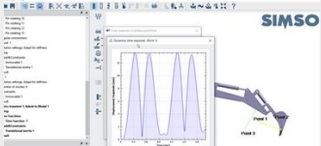
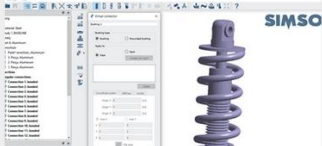
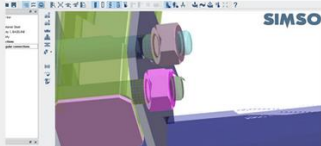
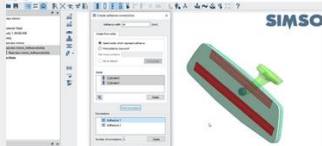
**Altair SimSolid - Random Response Analysis Tutorial**  
Learn how to perform a random response analysis in Altair...

Tutorials



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

Tutorials



# POLL

# STRUCTURAL NON-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**

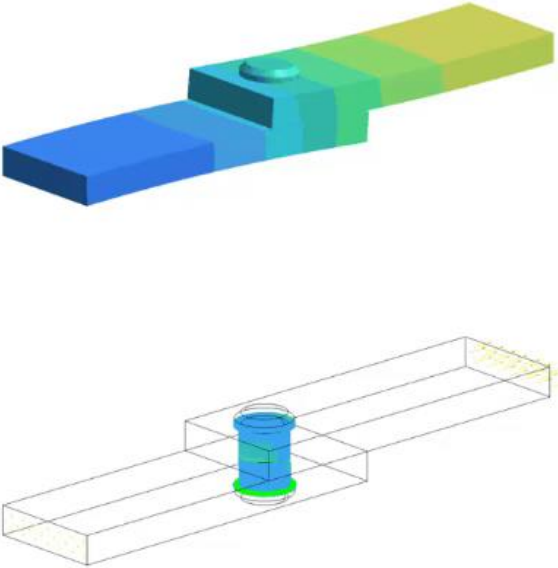


## Results

- 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

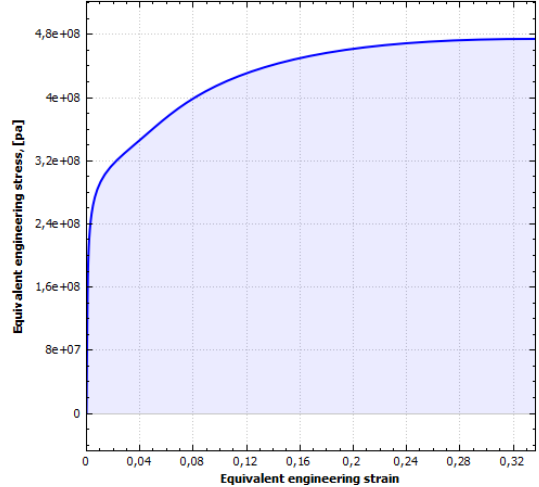
# Non-Linear Analysis on SimSolid

## Contact



Total run time = 19 sec

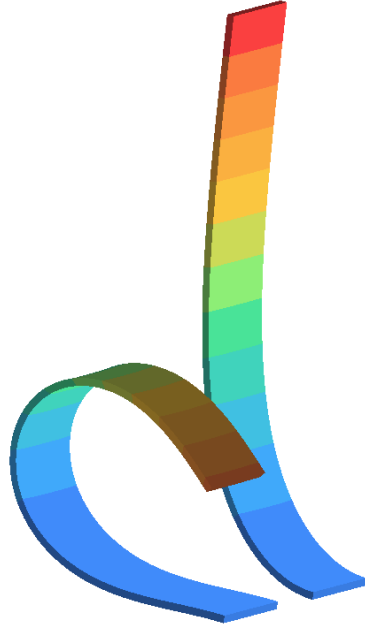
## Material



Total run time = 29 sec



## Geometric



Total run time = 5 sec

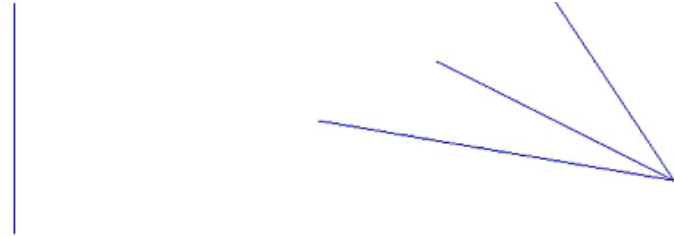
Limitation today: **NOT** possible to combine **Geometric non-linear** with **Separating/closing contact**.



# Introduction to Non-Linearity in Structural Mechanics

**Geometric non-linearity** accounts for the current (unknown) volume of the structure:

- Update of stiffness as the structure deforms.
- Characterized by:
  - Large displacement and large deformation.
  - Large rotation.
  - Buckling, bifurcation and collapse of the structure.



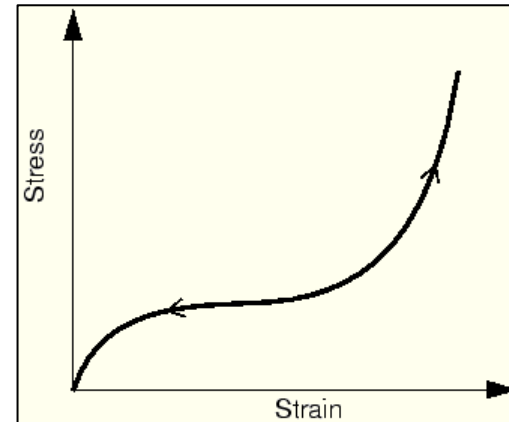
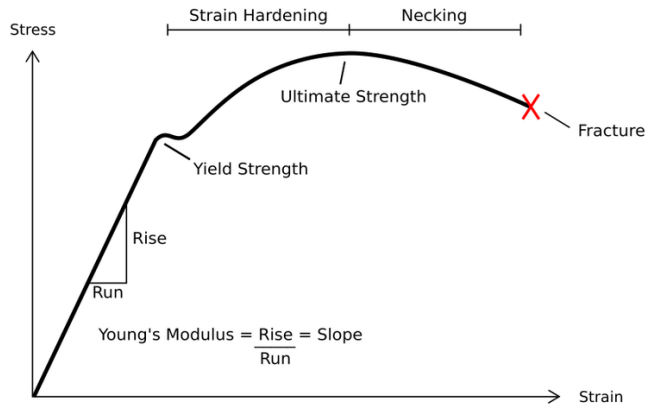
**Boundary non-linearity** stems from changes in a load or constraint in response to deformation or displacement in the structure:

- **Contact problems.**
- **Follower forces** (e.g., pressure load).

# Introduction to Non-Linearity in Structural Mechanics

**Material non-linearity** is used to model dependence of stress state on the existing strain state

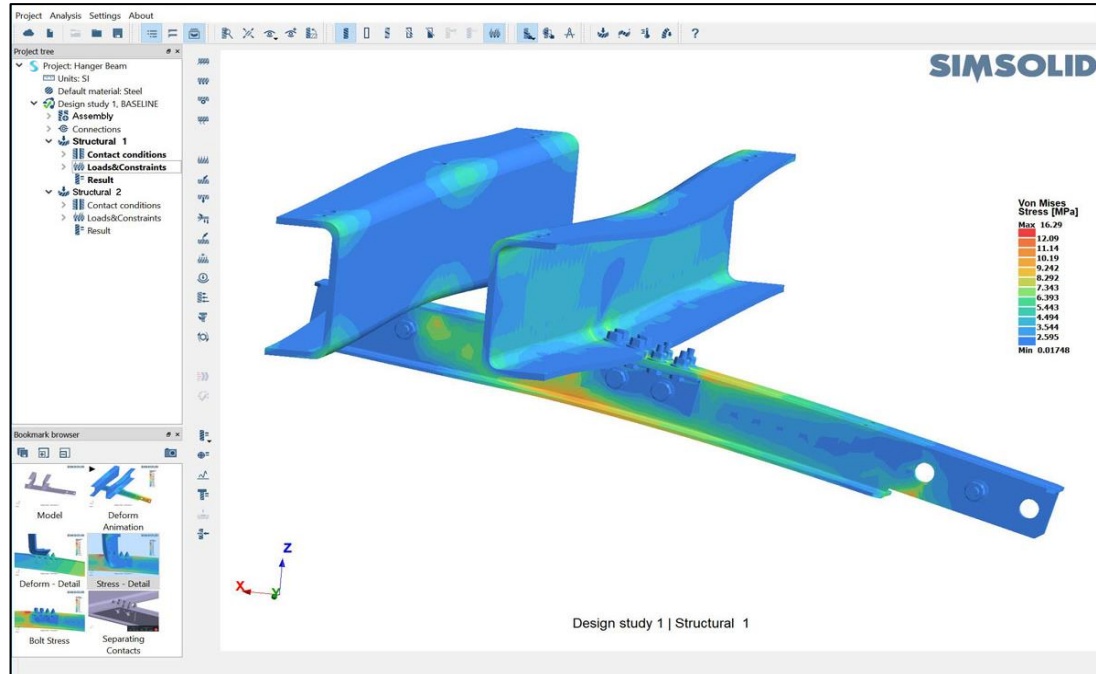
- Different types include:
  - **Plasticity beyond yield**
  - **Nonlinear elasticity** (Hyper-elastic, Viscoelastic etc.) - SimSolid does not simulate this.
  - **Cracking, Necking**, etc. - SimSolid does not simulate this.



# Non-Linear Analysis on SimSolid - Separating Contact

## Separating Contact - Non-Linear contact with friction

SimSolid has a wide variety of methods to connect parts in an assembly. Four contact conditions types are available for each connection - **bonded**, **sliding**, **disabled**, **separating-closing** or **separating contact with friction**.



# Non-Linear Analysis on SimSolid - Material

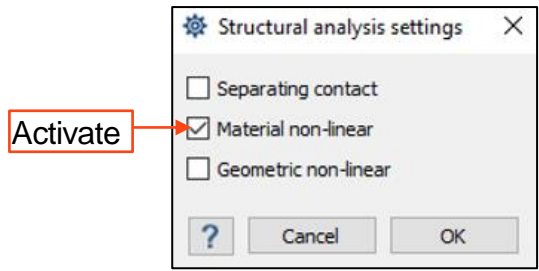
## Material - Material non-linear

Non-linear structural effects are specified in Structural settings dialog.

At least one model material property must contain a non-linear **stress vs strain curve**.

For any analysis that exceed the elastic limit, SimSolid will provide three outputs for each result quantity (more details in next slides):

- **Elasto-plastic**, full load: the results at full load assuming a non-linear stress- strain curve is used.
- **Elastic**, full load: the results at full load assuming a linear (straight) stress-strain curve is used.
- **After unloading**: the residual results after loading has been removed.



Select NL Material

Apply material

Material DB file name: C:\Users\panj\AppData\Local\Temp\SimSolid\SimSolid-Material-DB.ssm

Materials: [A] Generic Materials, [A] Plain Carbon & Low Alloy Steels, [A] Stainless Steels, [A] Cast Irons, [A] Aluminum Alloys, [A] Copper Alloy, [A] Magnesium Alloys, [A] Nickel Alloys, [A] Titanium Alloys, [A] Ceramics, [A] Plastics, [A] Sample non-linear materials, [A] Titanium - High strength - NL, [A] Copper - High strength - NL, [A] External Project Materials

System of units: SI

Material name: Titanium - High strength - NL

Elasticity modulus [Pa]: 1.14000e+11

Stress-strain curve

Review

Stress-strain curve

Point #	Equivalent strain	Equivalent stress, [pa]
1	0	0
2	0.007	7.96666e+08
3	0.0071	8.20113e+08
4	0.00722	8.6056e+08
5	0.00725	8.10006e+08
6	0.00728	8.15451e+08
7	0.00733	8.16895e+08
8	0.00736	8.20339e+08
9	0.00743	8.23781e+08
10	0.00746	8.27223e+08
11	0.0074	8.30665e+08
12	0.00753	8.34101e+08
13	0.00756	8.37538e+08

This strain-stress table was created by power law function: stress = K\*strain^n where K is strength coefficient, n is n-value

Support CSV, Create by In-value, Set elasticity modulus, Add row, Delete row

Apply to all parts, Apply to

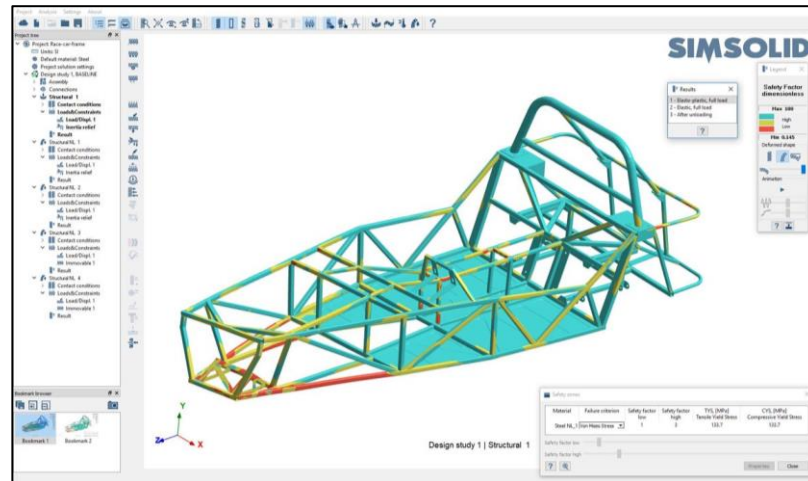
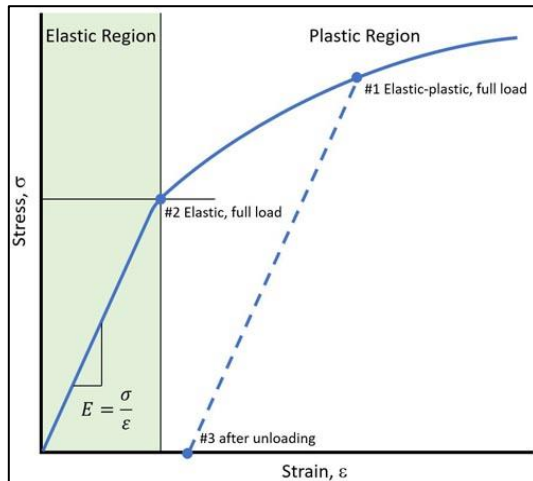
View strain-stress table

# Non-Linear Analysis on SimSolid - Material

## Material - Results evaluation

Once the analysis is complete, you can display result plots the usual way, but with one substantial difference:

- **Elastic-plastic response under full load** - Defines the structural response after all non-linear iterations are complete. See **Point #1** on the stress-strain graph below.
- **Elastic under full load** - Defines the response as if the material was linear at the elastic load limit (**Point #2**).
- **After unloading** - Defines the residual stresses, strains, and deformations once all loads have been removed (**Point #3**).



# Non-Linear Analysis on SimSolid - Geometric

## Geometric non-linear

In **geometric non-linear analysis**, changes in geometry as the structure deforms are taken into account when iteratively solving the strain-displacement and equilibrium equations.

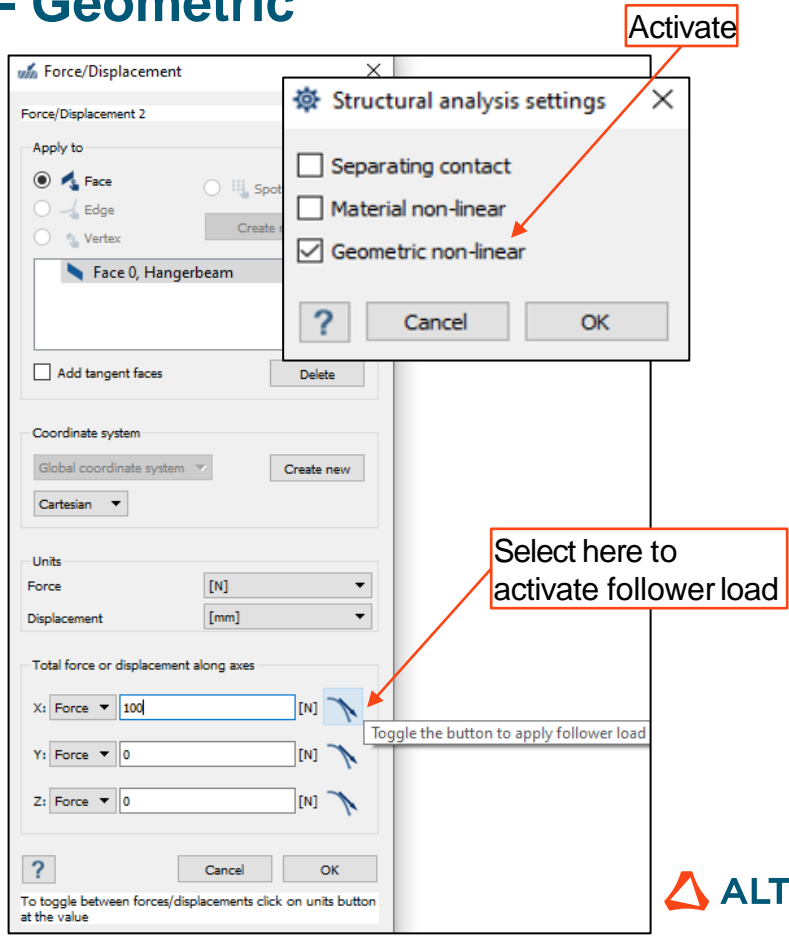
Follower loads can be optionally defined.

Typical applications include slender structures, sheet metal and stability analysis of all types.

Results are processed in the usual manner.

One additional consideration for geometric non-linear analyses is to examine deformations in true scale.

To do this, show a deformed shape plot in the normal manner, then select the **Change max deformation** button.



# Non-linear options summary

\* Today: not with „Separating contact“

Structural analysis settings

- Separating contact
- Material non-linear
- Geometric non-linear

Buttons: ? Cancel OK

Structural analysis settings

- Separating contact
- Material non-linear
- Geometric non-linear

Buttons: ? Cancel OK

Structural analysis settings

- Separating contact
- Material non-linear
- Geometric non-linear

Buttons: ? Cancel OK

Structural 2 - Separating Contact

- Contact conditions
  - Connection 1, separating
  - Connection 2, bonded
  - Connection 3, bonded
  - Connection 4, bonded
  - Connection 5, bonded
  - Connection 6, bonded
  - Connection 7, bonded
  - Connection 8, bonded
  - Connection 9, bonded
  - Connection 10, bonded
  - Connection 11, bonded
  - Connection 12, bonded
  - Connection 13, bonded
  - Connection 14, bonded
  - Connection 15, bonded

Contact conditions

- Disable contact
- Bonded
- Sliding without friction
- Separating

Friction coefficient: 0.1

Choose type of contact

Buttons: ? Cancel OK

Apply material

Material D6 file name: C:\Users\jwcl\AppData\Roaming\Smsold\SDMSOLID-Material-D6.ssm

Materials

- Generic Materials
  - Plain Carbon & Low Alloy Steels
    - Cast Irons
    - Aluminum Alloys
    - Copper Alloys
    - Magnesium Alloys
    - Nickel Alloys
    - Titanium Alloys
    - Ceramics
    - Plastics
    - Sample non-linear materials
      - Steel NL**
      - Custom Materials

System of units: SI

Material name: Steel NL

Elasticity: 2.00000e+11

Poisson's ratio: 0.3

Density: 7.85000e+00

Thermal expansion: 1.20000e-05

Thermal conductivity: 5.60000e+00

Ultimate tensile strength: 0.00000e+00

Tensile yield strength: 1.17316e+08

Compressive yield strength: 1.17316e+08

Default failure criterion: Not Specified

Stress-strain curve

View strain-stress table

Point #	Equivalent strain	Equivalent stress, [pa]
1	0	0
2	0.00098658	1.17316e+8
3	0.000958418	1.27842e+8
4	0.000724973	1.38366e+8
5	0.000799633	1.48888e+8
6	0.000886065	1.59406e+8
7	0.00089201	1.69919e+8
8	0.00111558	1.80426e+8
9	0.0012736	1.90925e+8
10	0.00147511	2.01411e+8
11	0.00173425	2.1188e+8
12	0.00207068	2.22328e+8
13	0.00251042	2.32744e+8
14	0.00308887	2.43118e+8
15	0.00385505	2.53432e+8

Buttons: Refit Cancel OK

Apply to all parts Apply to selected part(s)

Select part(s) to apply material to

Load/displacement

Load/Displ. 1

Apply to:

- Face
- Edge
- Vertex

Create new spot

Face 10, Lowerbeam

Delete

Coordinate system: Global coordinate system

Create new

Units: Force [N], Displacement [mm]

Total load or displacement along axes

X: Force 100 [N]

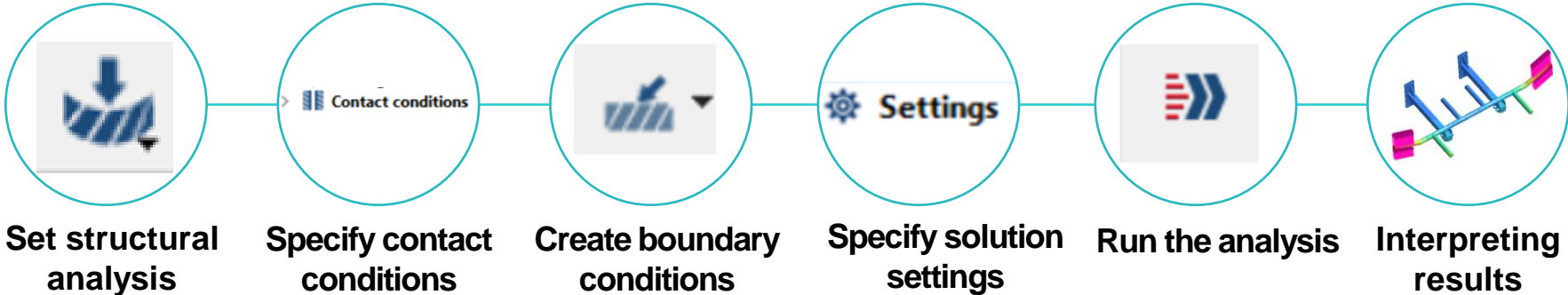
Y: Force 0 [N]

Z: Force 0 [N]

Buttons: ? Cancel OK

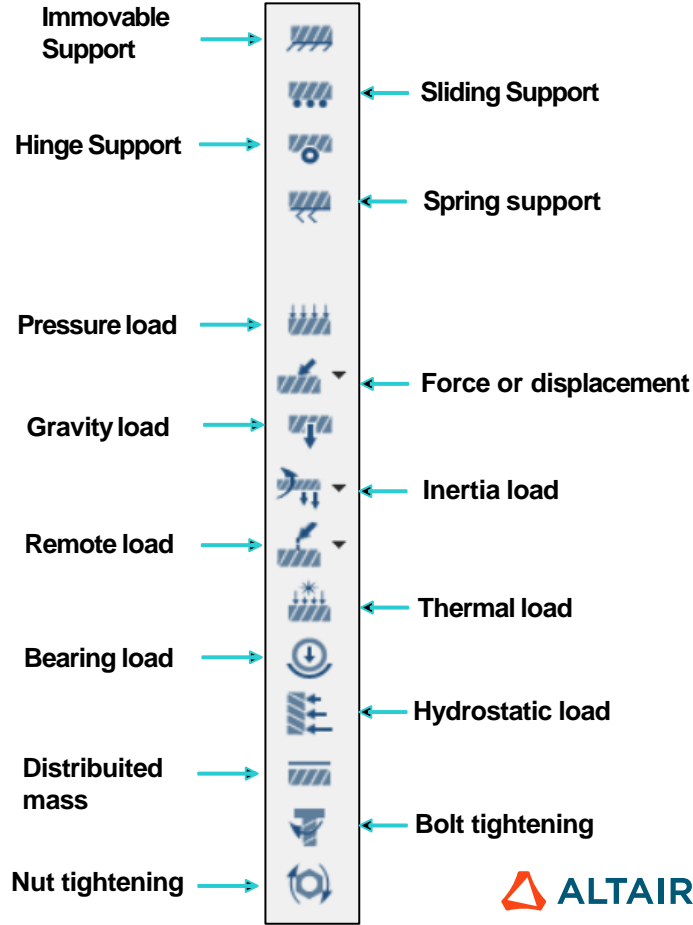
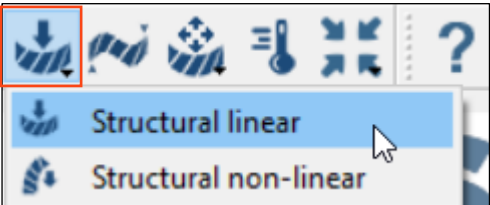
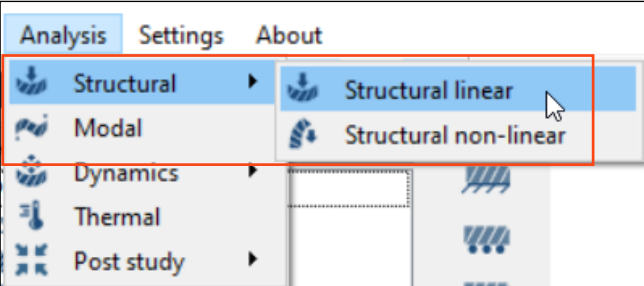
To toggle between loads/displacements click on units button at the value

# Set Structural Analysis



# Set Structural Analysis

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

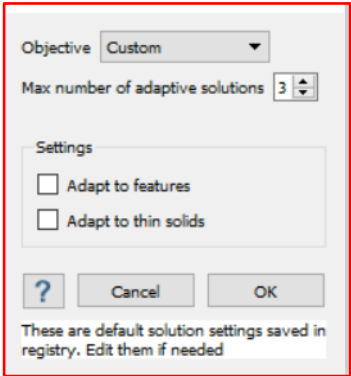
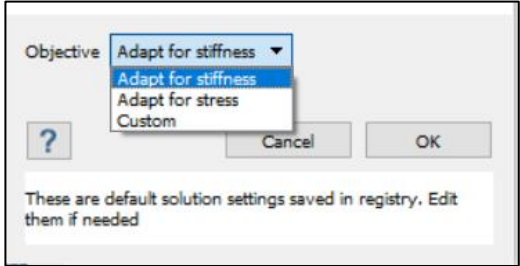


# Solution Settings

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

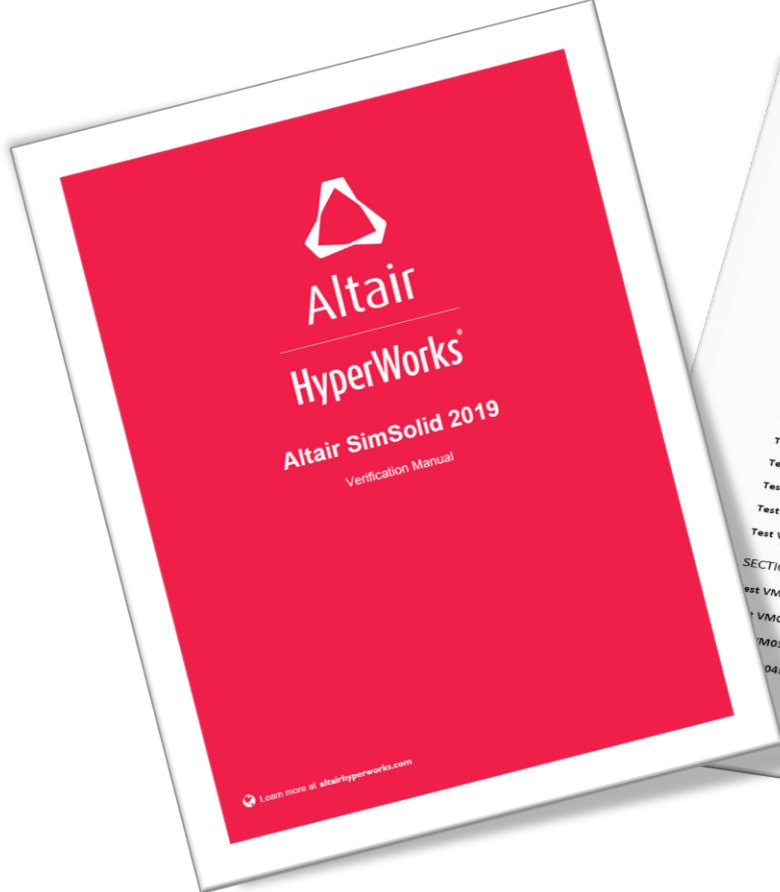
- **Adapt for Stiffness** - Use this objective for general load path prediction, modes and/or thermal analysis.
- **Adapt for Stress** - Use this objective for general stress calculation.
- **Custom** - Most general objective. All controls can be individually set. Set the **max number of passes** and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

# 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

# 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:

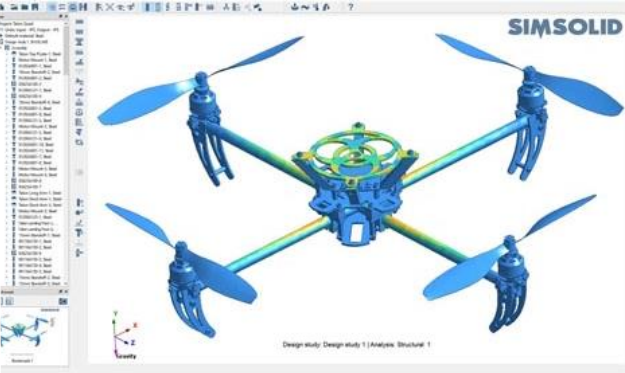
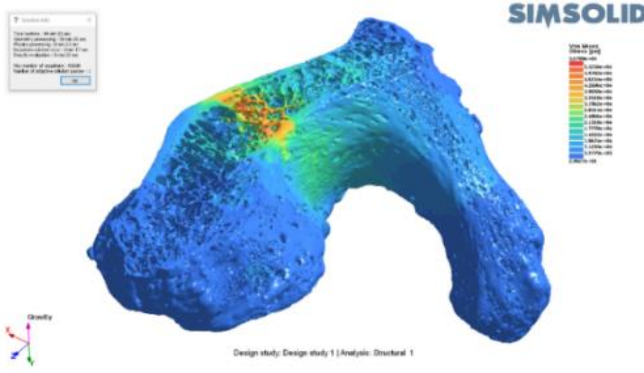
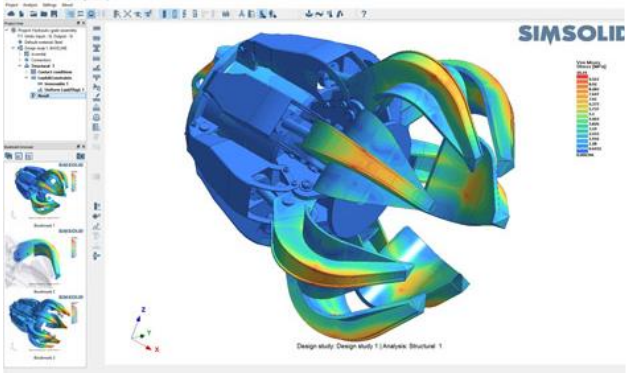


# JOIN THE SIMULATION REVOLUTION

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