

SimSolid Quick Overview Module 6: Thermal Analysis / SimSolid News

Sebastian Karp ● Senior Application Engineer
Arthur Schubert ● Senior Technical Consultant
March 12th 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..**

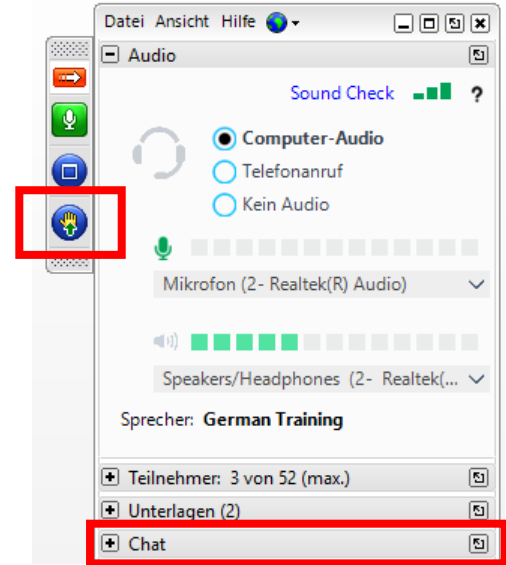
..and provides results in **seconds to minutes**

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



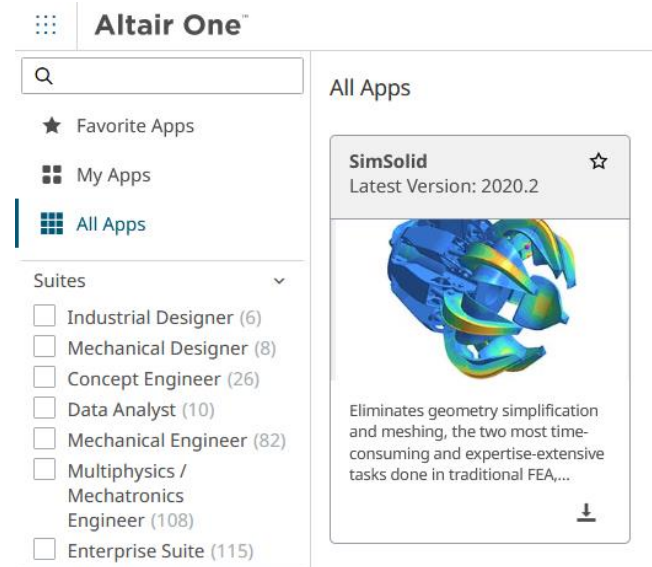
Have a look at...

- Contact us for later communication etc. - trainings@altair.de
- Check out www.altair.com/SIMSOLID



What Customers Are Saying About SimSolid

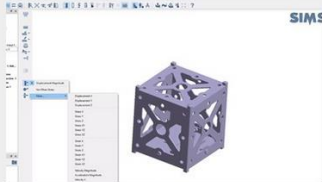
- Download - www.altairone.com



Have a look at...

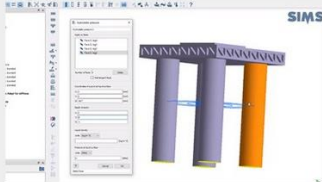
- www.altair.de/resource/altair-simsolid-tutorial-projects

All SimSolid Demo Models, Training Materials, and Tutorials



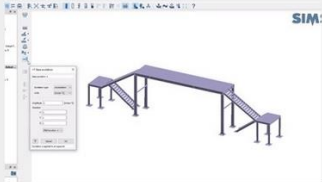
Altair SimSolid - ERP with Constant Loading Tutorial
Learn how to perform a dynamic frequency response...

Tutorials



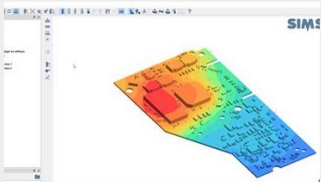
Altair SimSolid - Hydrostatic Pressure Tutorial
Learn how to simulate an hydrostatic pressure in Altair...

Tutorials



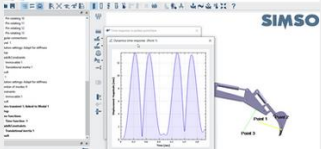
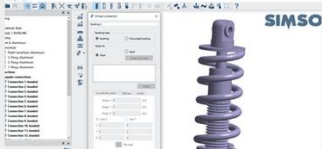
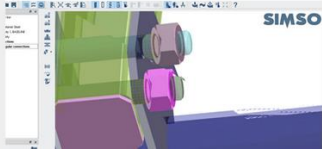
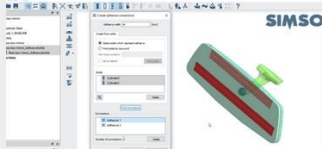
Altair SimSolid - Random Response Analysis Tutorial
Learn how to perform a random response analysis in Altair...

Tutorials



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

Tutorials

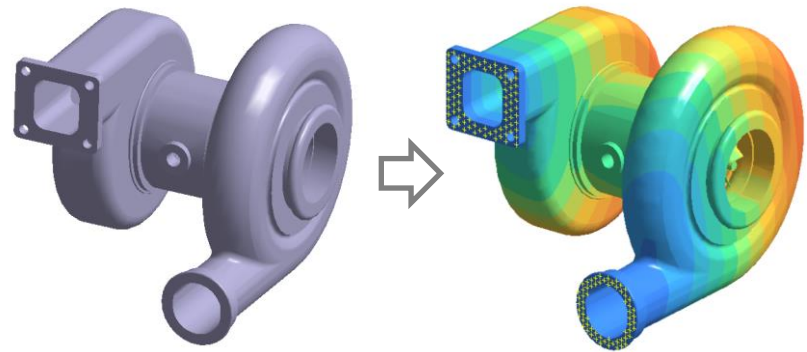
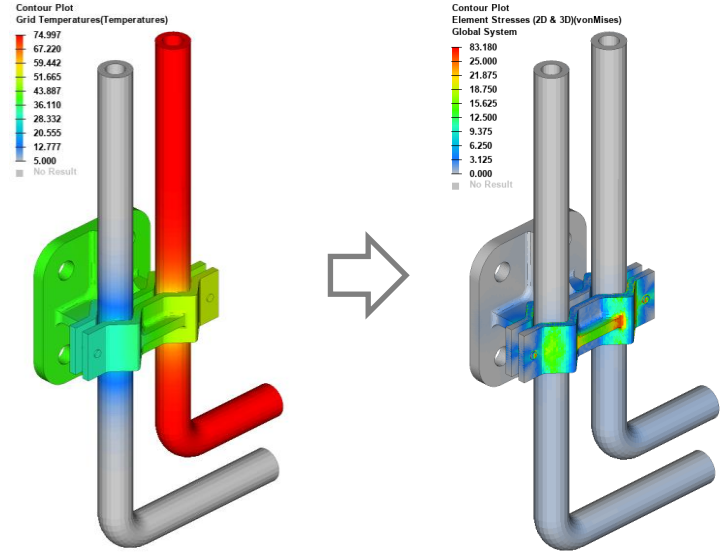


POLL

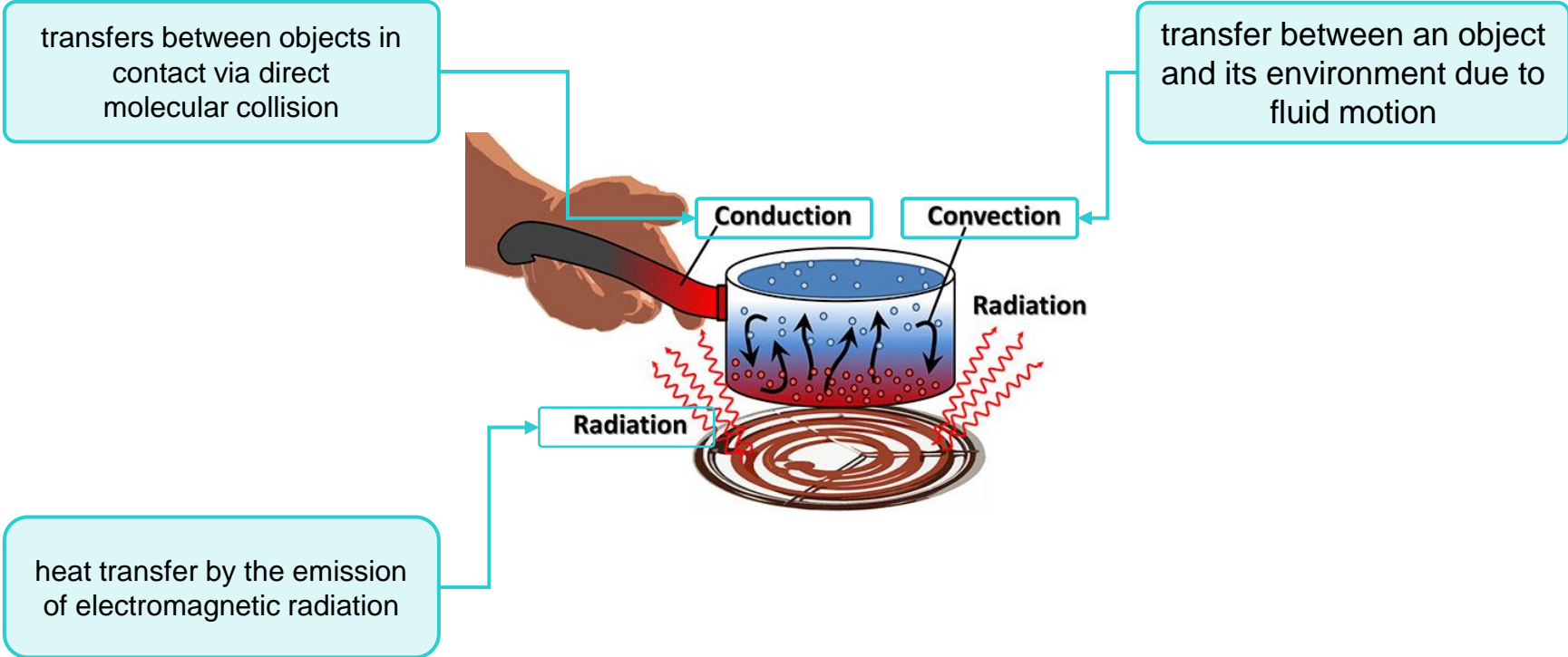
THERMAL ANALYSIS

Motivation

- Temperature distribution
- Thermal induced stresses



Definitions



Supported solutions

- Steady State
 - Temperature and fluxes at the final thermal equilibrium state are of interest
 - Thermal loads time-invariant
- Supported in OptiStruct

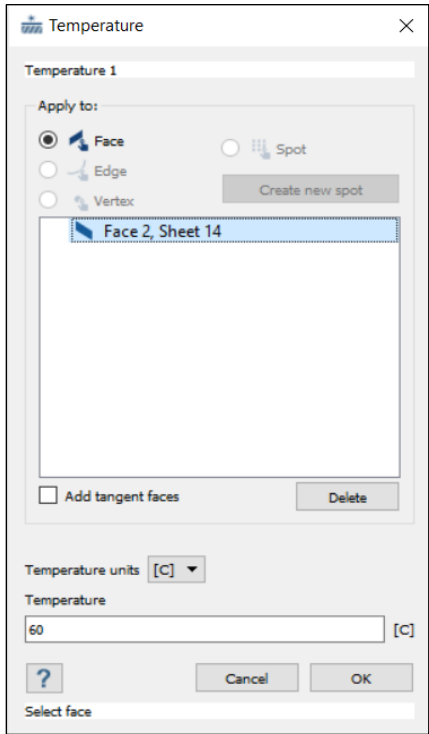
- Linear
 - Material properties such as conductivity, capacity and convection coefficient are temperature independent
- Supported in OptiStruct

- Transient
 - Thermal behavior of a system over a specific period of time
 - Thermal loads time-invariant or -dependent
- Supported in OptiStruct

- Nonlinear
 - Material properties such as conductivity, capacity and convection coefficient are temperature dependent
- Supported in OptiStruct

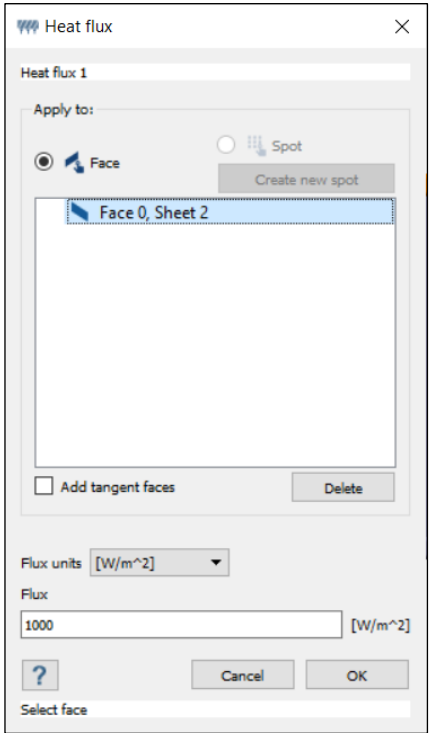
Boundary conditions

- Temperature
 - Definition of fixed temperature
 - Applied to: face, edge, vertex or spot
 - Specified as: °C



Boundary conditions

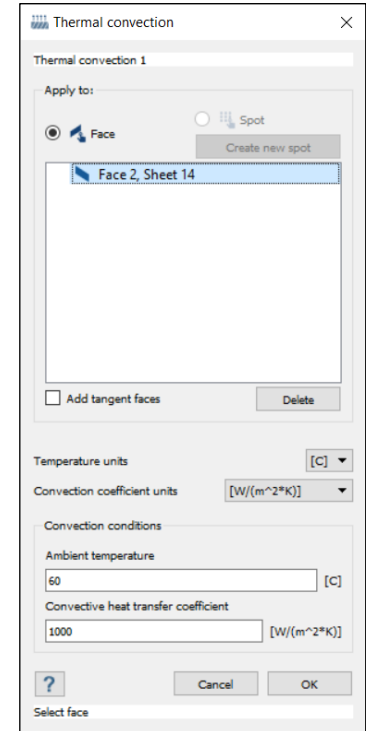
- Heat flux
 - Rate of heat energy transfer through a given surface per unit time
 - Applied to: face or spot
 - Specified as: W/m^2



Boundary conditions

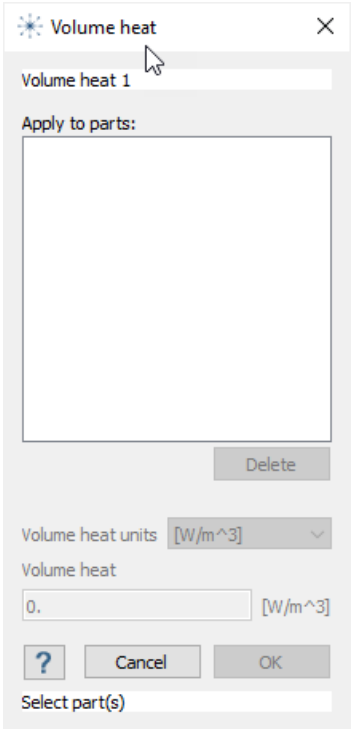
- Convection
 - Rate of heat transfer between the bulk fluid (liquid or gas, buoyant or moving) and a surface of your model
 - Applied to: face or spot
 - Specified as: $W/(m^2 \cdot K)$
 - Ambient temperature : constant temperature of the bulk fluid

Flow type	$W/(m^2 \cdot K)$
Forced convection - low speed flow of air over a surface	10
Forced convection - moderate speed flow of air over a surface	100
Forced convection - moderate speed cross-flow of air over a cylinder	200
Forced convection - moderate flow of water in a pipe	3000
Forced convection - molten metals flowing in tubes	2000 to 45000
Forced convection - oil flowing in tubes	300 to 1700
Forced convection - boiling water in a pipe	50000
Gases in free convection	5 to 37
Water in free convection	100 to 1200
Oil in free convection	50 to 350



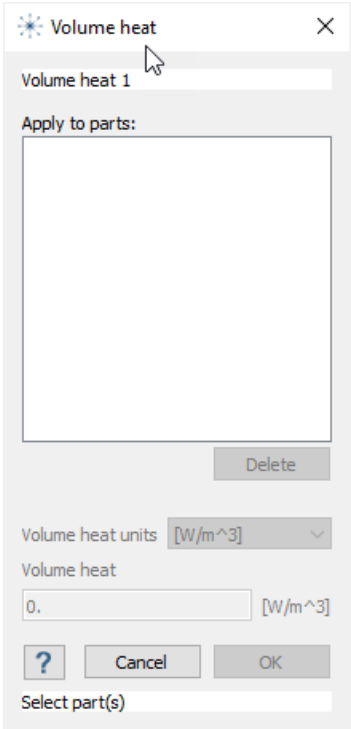
Boundary conditions

- Volumetric heat
 - internal heat generation (heat source) and internal heat absorption (heat sink) in volumes
 - Applied to: part
 - Specified as: $W/(m^3)$
 - A positive volume heat value indicates a heat source and a negative volume heat value indicates a heat sink




Boundary conditions

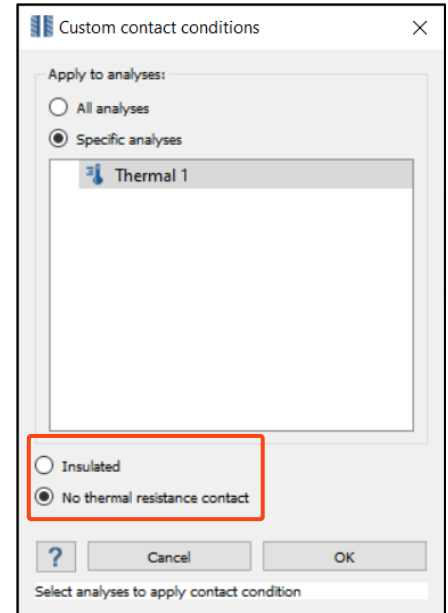
- Volumetric heat
 - internal heat generation (heat source) and internal heat absorption (heat sink) in volumes
 - Applied to: part
 - Specified as: $W/(m^3)$
 - A positive volume heat value indicates a heat source and a negative volume heat value indicates a heat sink



Boundary conditions

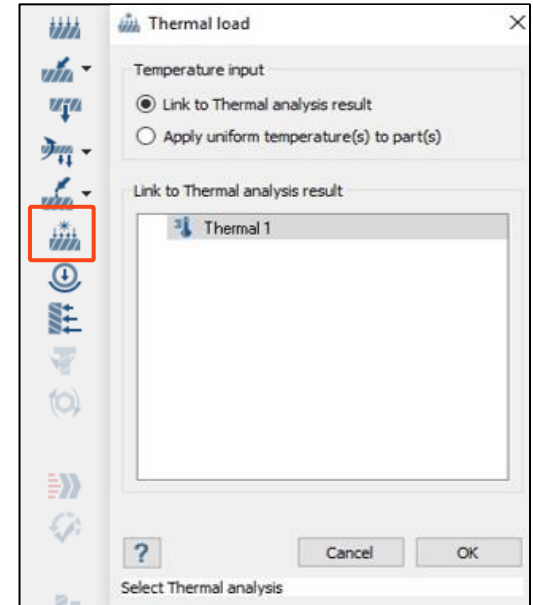
- Thermal contact conditions
 - Thermal specific contact conditions between parts
 - No thermal resistance - default condition. Indicates full heat transfer will occur through the part connection.
 - Insulated - Insulated means the opposite. No heat transfer will occur.

-  on one or more parts > Review part contact conditions > Edit



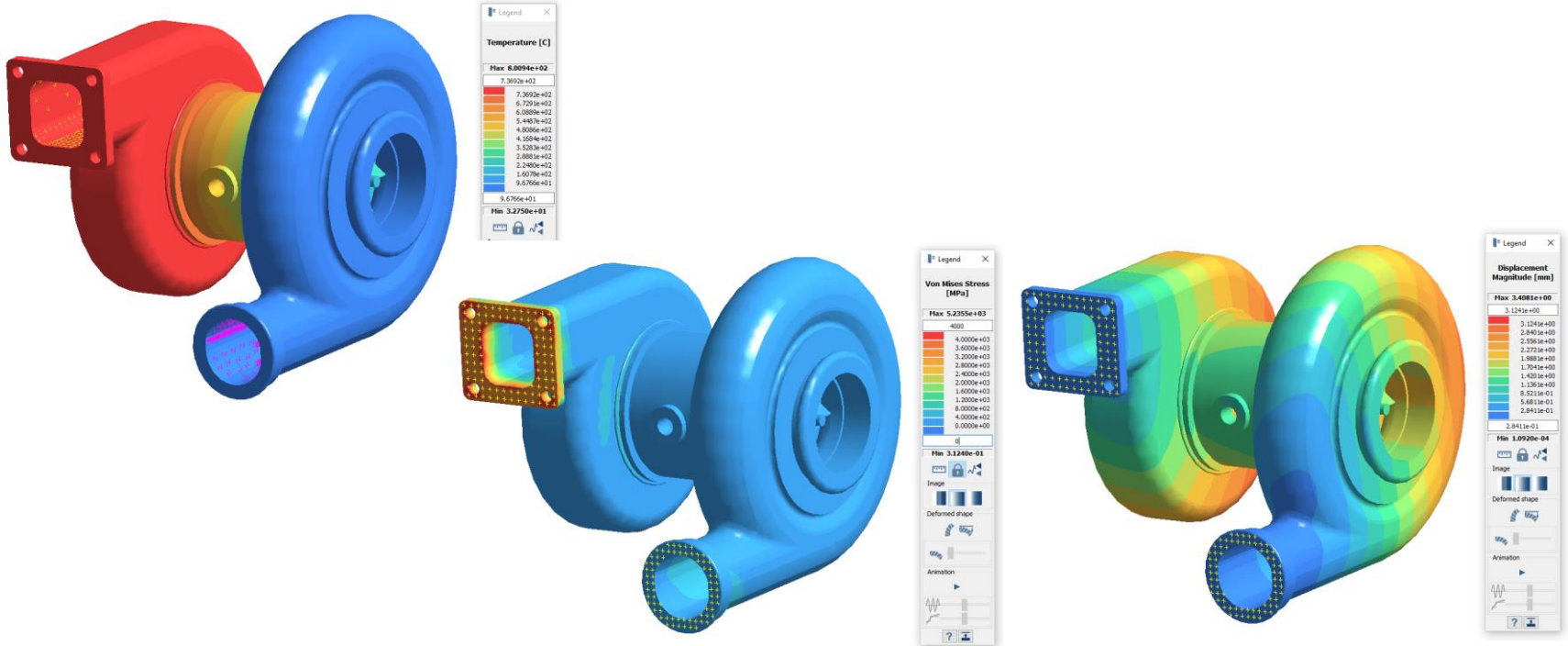
Boundary conditions

- Thermal loads
 - Thermal analysis results can be used in a subsequent structural static analysis as thermal loads
 - Uniform temperature field: a single temperature is assumed to be uniform over the entire model.
 - Part based temperature field: temperatures are applied to individual parts.
 - Thermal analysis result field: the temperatures are imported from an existing Thermal analysis.



Thermal analysis

- Demo



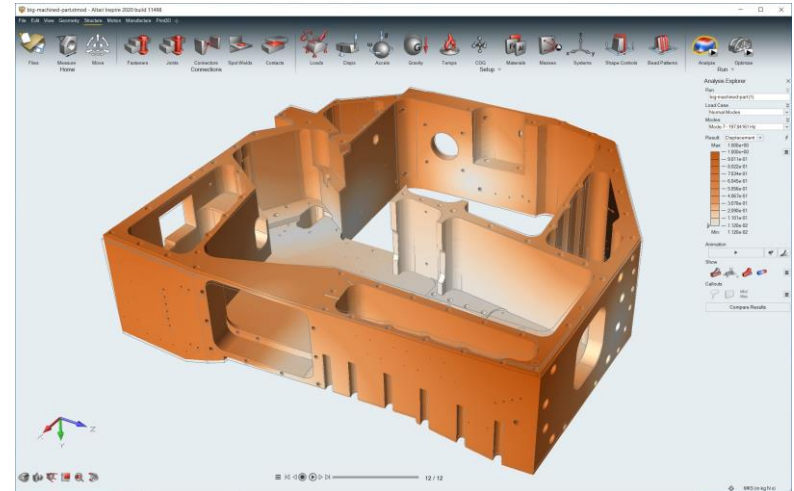
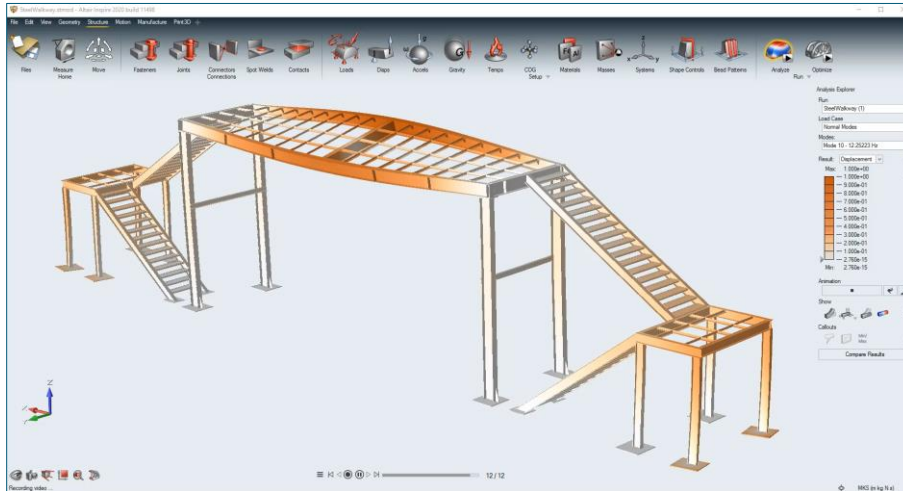
SIMSOLID FEATURES 2020.X AND NEWS

→ Module 7 – next week !!

© Altair Engineering, Inc. Proprietary and Confidential. All rights reserved.

SimSolid Solver in Inspire

Inspire enables easy and quick **geometry modification** for iterative design studies



393 parts

10 modes

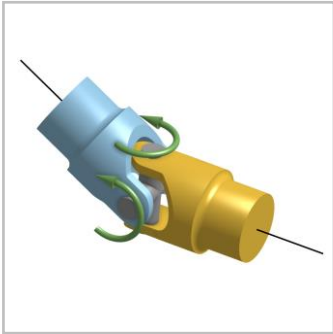
2 min solve

1200 faces

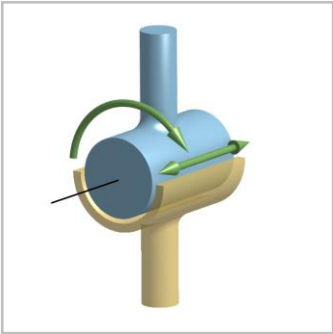
5 modes

<1min solve

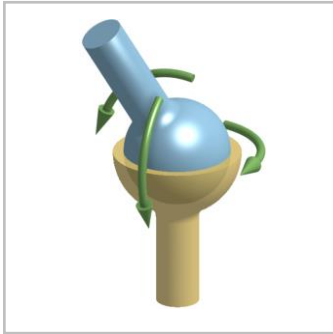
2020: New SimSolid Virtual Joints



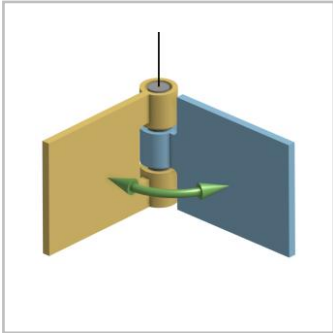
Universal
2 DOF, 2 axes



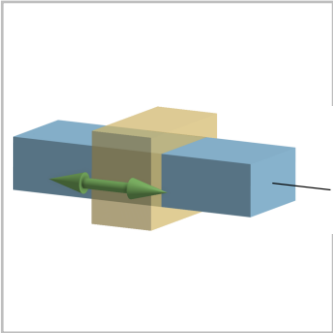
Cylindrical
2 DOF, 1 axis



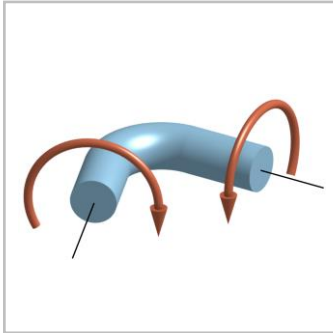
Ball
3 DOF, no axes



Hinge
1 DOF, 1 axis



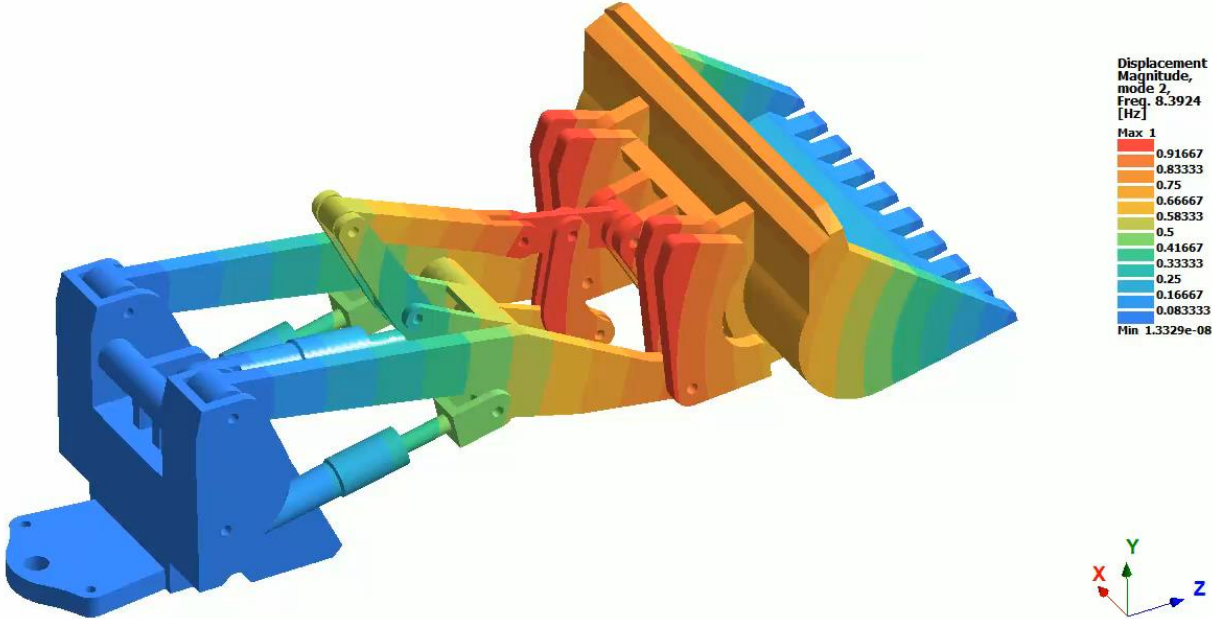
Linear guide
1 DOF, 1 axis



Flexible shaft
5 DOF, 2 axes

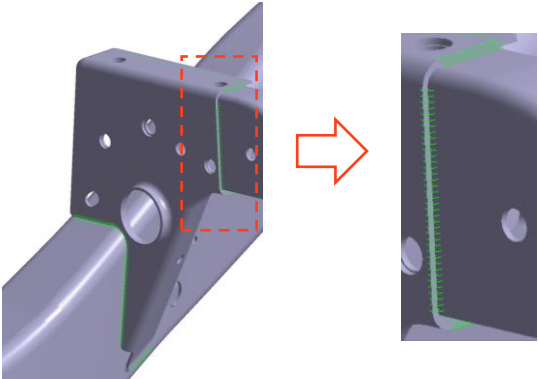
2020: New virtual joints

SIMSOLID

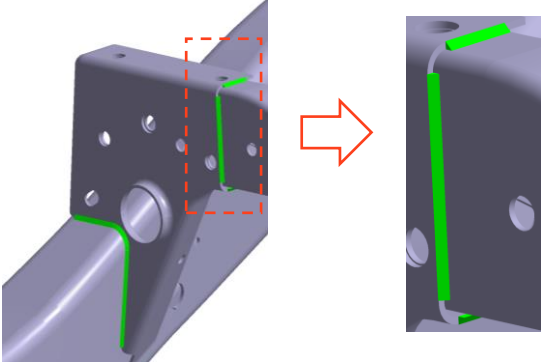


Design study 1 | Modal 1

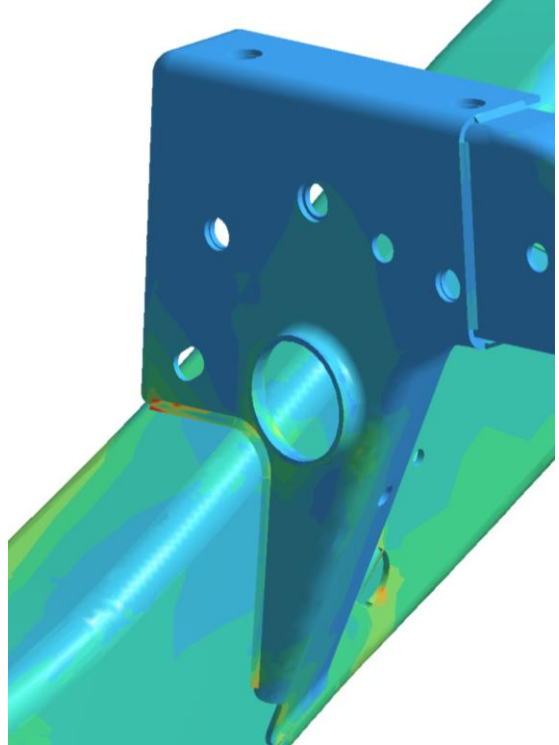
2020: New 3D solid seam welds



Previous – Connection based welds



Release 2020 – 3D solid based welds

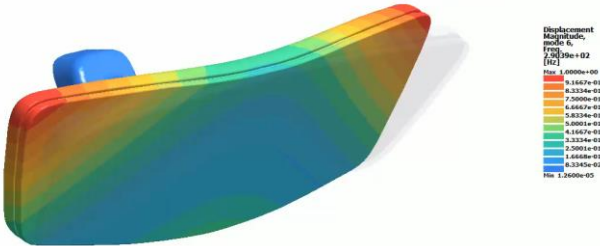


Release 2020 - Weld stresses

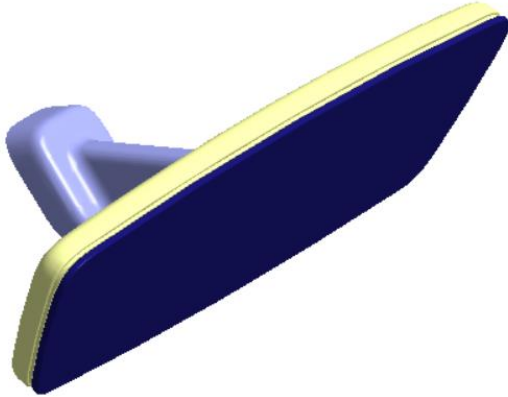
2020: Adhesive Joints

Represents sealants and structural adhesives

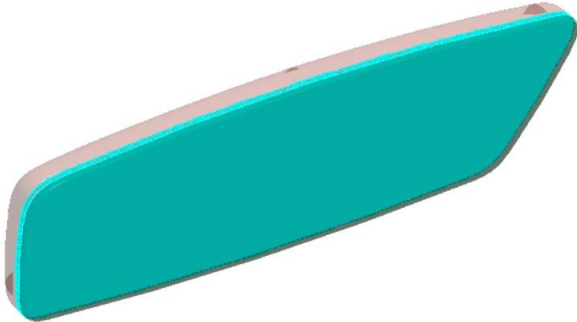
- Adhesives from Connections
- Adhesives from Tubes



Sixth elastic mode



Rearview Mirror

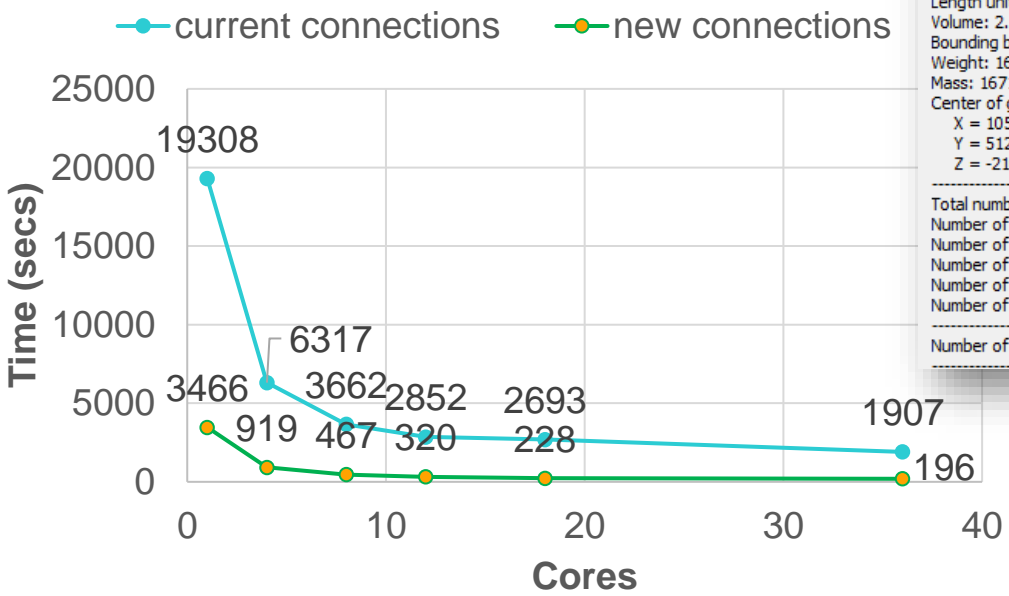


Adhesive connection between the mirror/frame

2020: Performance improvements

Regular Connection Search TIME - 24,000 Connections

Connections Search Time



? Assembly info

Length units: [mm]
Volume: 2.12935e+09 [mm]^3
Bounding box: 21188.6 x 7230.13 x 42250 [mm]
Weight: 163922 [N]
Mass: 16715.4 [kg]
Center of gravity coordinates:
X = 10594.2 [mm]
Y = 5128.03 [mm]
Z = -21095.2 [mm]

Total number of parts: 8427
Number of suppressed parts: 5
Number of bolts: 1594
Number of nuts: 1894
Number of washers: 3835
Number of sheets: 652

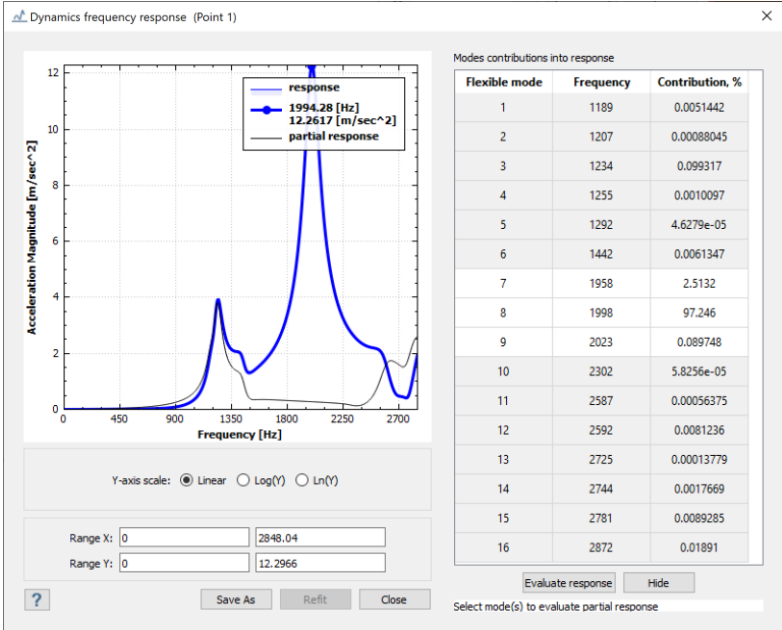
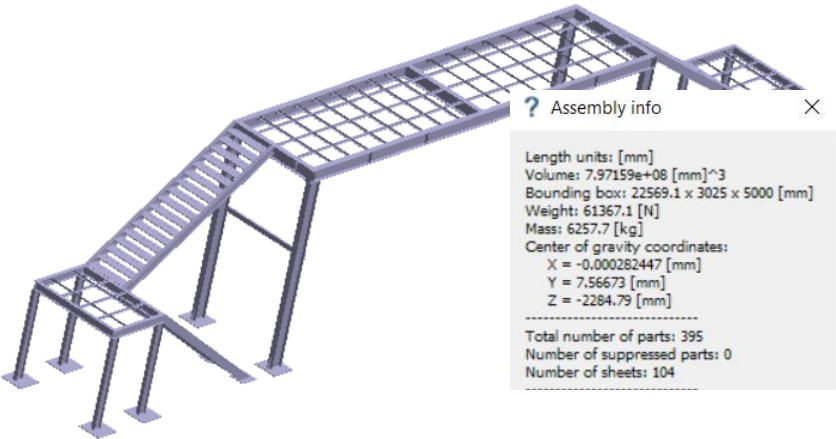
Number of triangles in faceted model: 5566418

Current connection search
7x speedup with 18 cores
10x speedup with 36 cores

New connection search
15x speedup with 18 cores
18x speedup with 36 cores

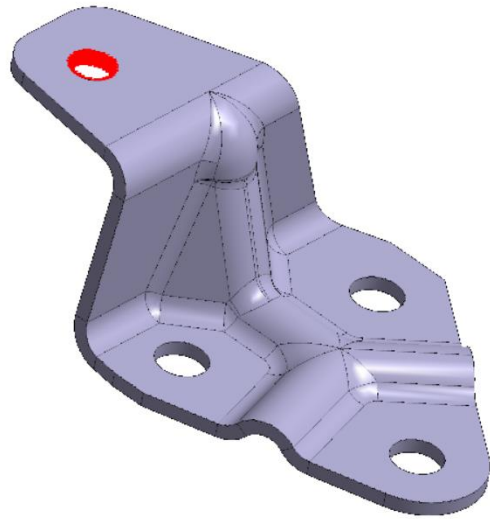
2020: Partial dynamic response

- Modal contributions into response
- Partial dynamics response

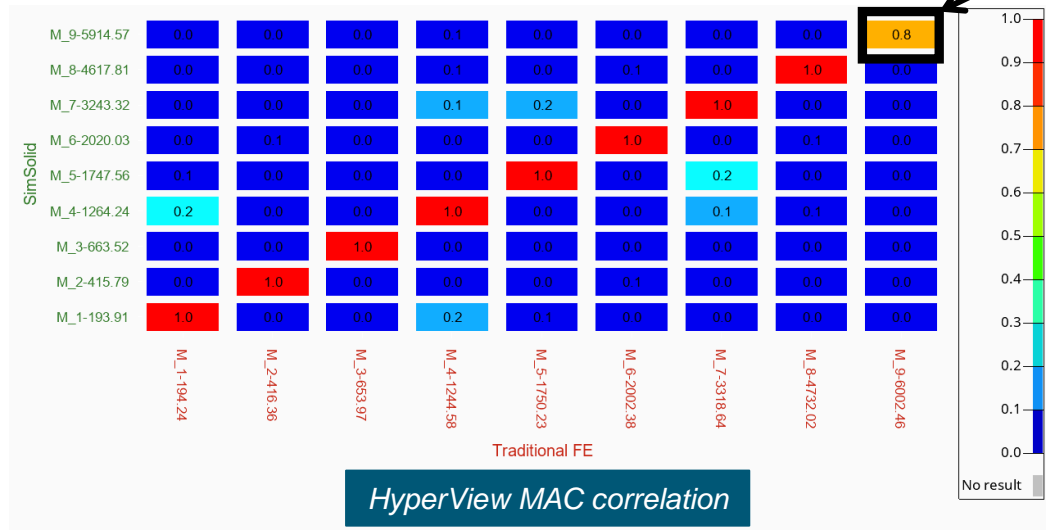


2020: Mode shape export

- **Mode shape export** – mode shapes can now be exported in Universal file (UNV) format at a given set of datum points.

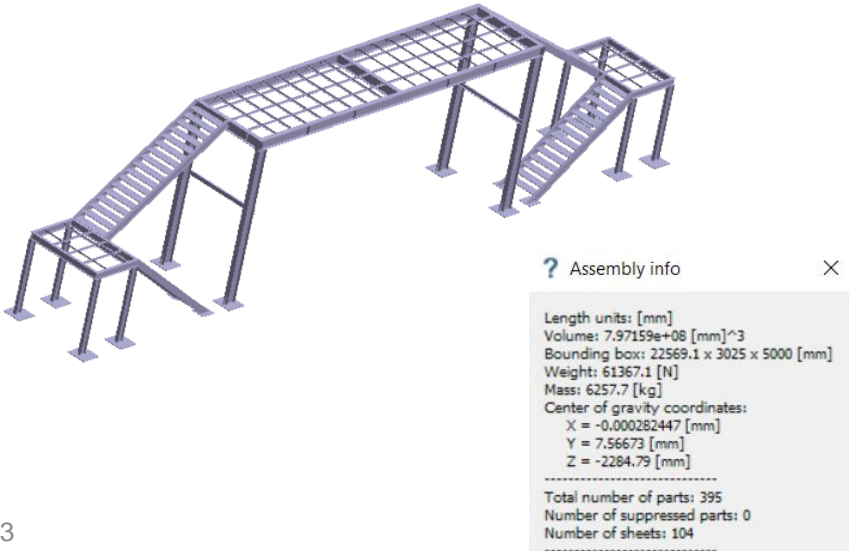


Modal correlation study of a bracket



2020: new Dynamic results

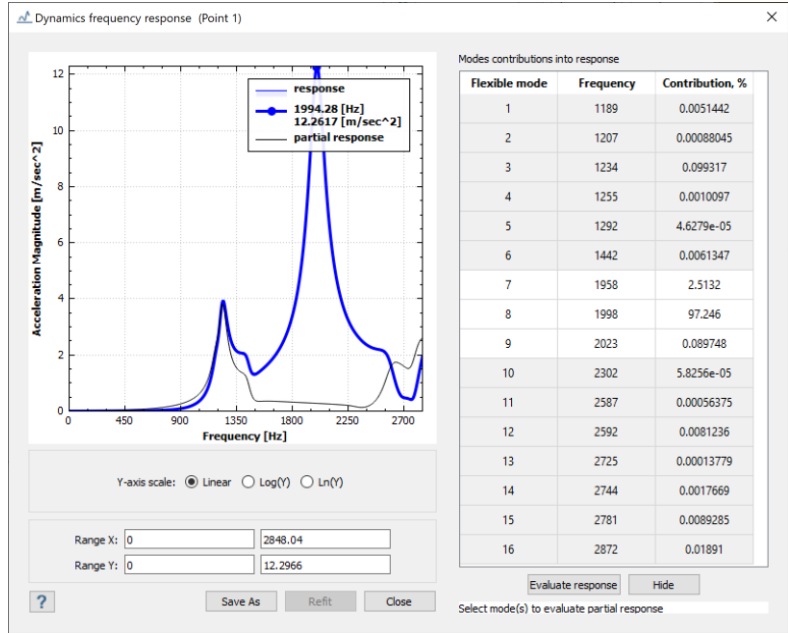
- **New Dynamic results** – New velocity, acceleration and equivalent radiated power (ERP) results available for frequency response analysis.



? Assembly info

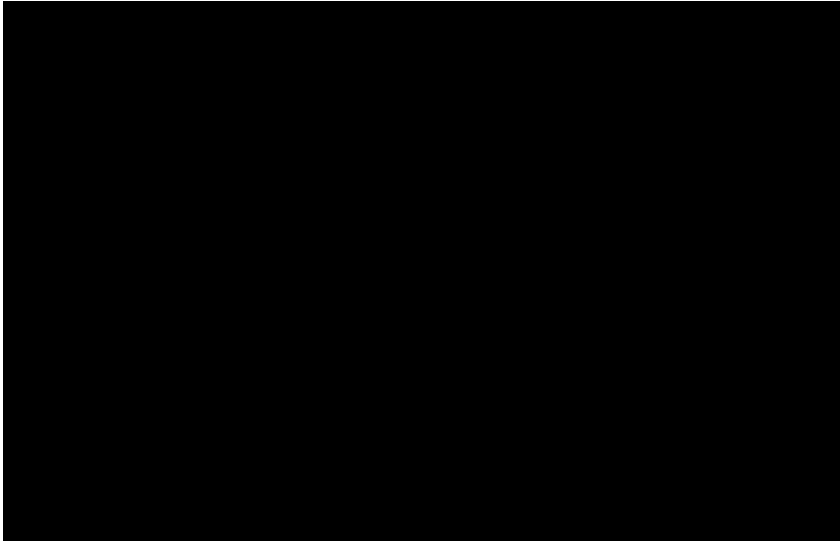
Length units: [mm]
Volume: 7.97159e+08 [mm]^3
Bounding box: 22569.1 x 3025 x 5000 [mm]
Weight: 61367.1 [N]
Mass: 6257.7 [kg]
Center of gravity coordinates:
X = -0.000282447 [mm]
Y = 7.56673 [mm]
Z = -2284.79 [mm]

Total number of parts: 395
Number of suppressed parts: 0
Number of sheets: 104



2020: animation improvements

- **Static animation improvements** – New in frequency response analysis, static animation of results are now available for each frequency



Legend X

Displacement Magnitude [mm]

Max 1.7645e-01

1.6175e-01

1.6175e-01
1.4704e-01
1.3234e-01
1.1763e-01
1.0293e-01
8.8225e-02
7.3521e-02
5.8816e-02
4.4112e-02
2.9408e-02
1.4704e-02

1.4704e-02

Min 2.7325e-11

Image

Deformed shape

Animation

2020: Equivalent Radiated Power (ERP)

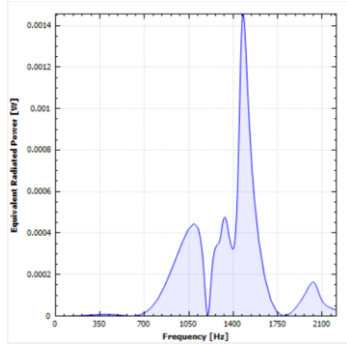
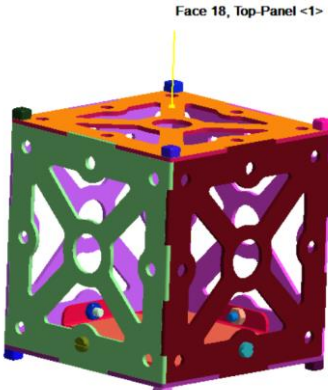
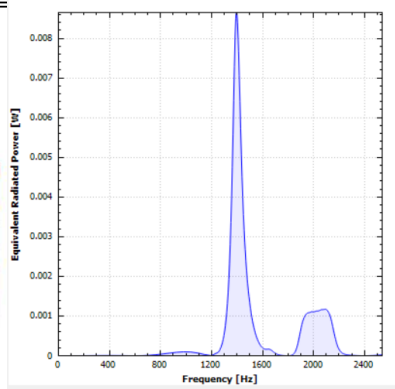
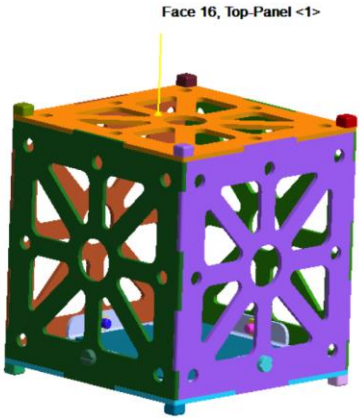
Background: ERP helps in understanding the acoustic contribution of a vibrating structure without running a full-blown acoustic simulation.

$$ERP = ERPRLF * (0.5 * ERPC * ERPRHO) \int V^2 dA$$

V is the normal velocity of the picked point

ERPC (speed of sound in air) = 343 m/s

ERPRHO (density of air) = 1.205 kg/m³

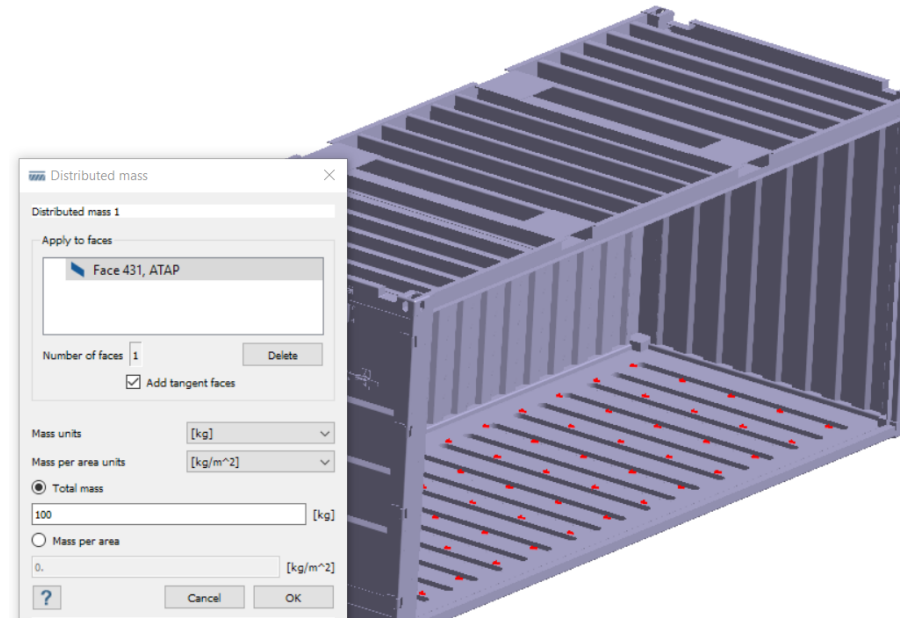


Design 2: Peak Panel ERP = 1.4 e-3 W

Design 1: Peak Panel ERP = 8 e-3 W

2020: Distributed Mass and Liquid Body

- Distributed mass is now available in static and dynamic analysis
- Fluid mass representation is also supported
 - Adds distributed mass to the structure and, therefore, changes its inertia and dynamics.
 - Incompressible fluid can significantly change vibration modes and general deformations at fluid enclosure because the volume of the enclosure is always preserved.



Distributed mass applied to container's floor

2020: Pick Info improvements

- New selection by face option. This provides an average of the result value over the selected face.
- In addition to average values, total face values are provided for:
 - thermal flux
 - equivalent radiated power (ERP)

Pick Info

Response: **Displacement Magnitude**

Output coordinate system
Global coordinate system
Cartesian

Pick point Datum point set
 Pick face

Faces
Face 6, Part 1

Face 6, Part 1

Displacements, [mm] :
Displacement magnitude = 0.00048573
Displacement X = 0.00048116
Displacement Y = 9.8852e-08
Displacement Z = -7.8352e-06

Stresses, [MPa] :
Von Mises stress = 1.1185
Principal stress 1 = 1.1025
Principal stress 2 = 0.059605
Principal stress 3 = -0.062696
Max shear stress = 0.58259
Stress X = 1.0812
Stress Y = 0.015761
Stress Z = 0.0024182
Stress XY = -3.4117e-05
Stress YZ = -9.5228e-05
Stress XZ = -0.0019793

Strains:
Equivalent strain = 4.8093e-06
Principal strain 1 = 5.5169e-06
Principal strain 2 = -1.2097e-06
Principal strain 3 = -1.9985e-06
Strain X = 5.3797e-06
Strain Y = -1.4925e-06
Strain Z = -1.5785e-06
Strain XY = -2.2006e-10
Strain YZ = -6.1422e-10
Strain XZ = -1.2766e-08

Energy density = 3.4957e-06 [MPa]

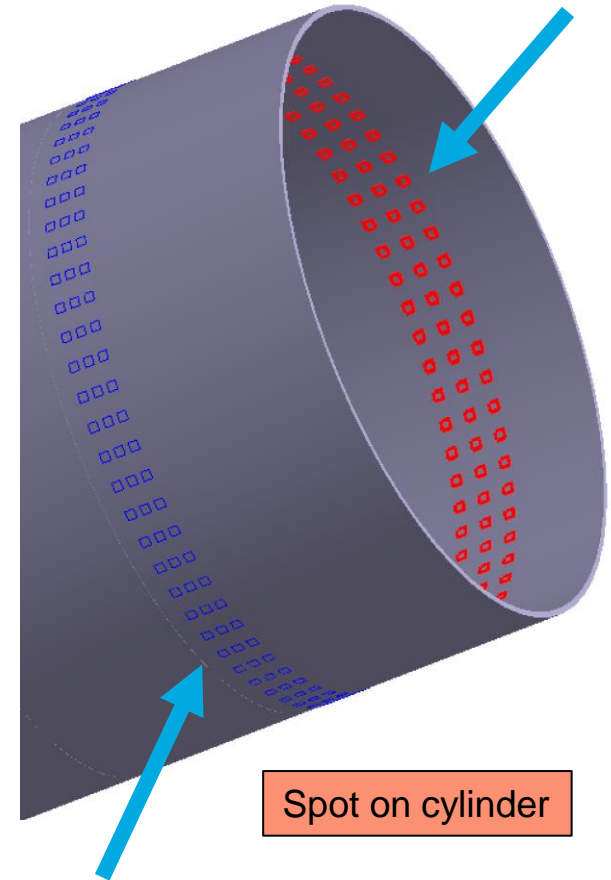
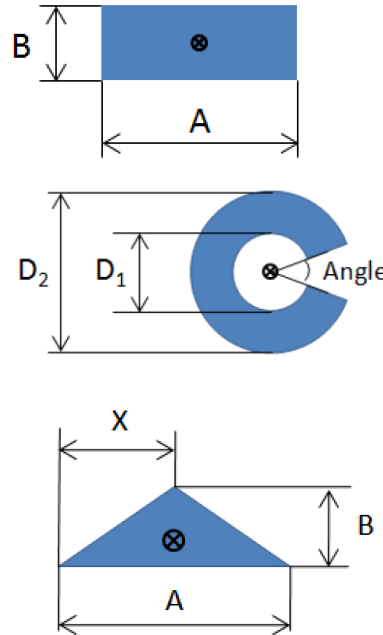
Evaluate average

? Close

Pick a face

2020: Spots' improvements

- Spots define local boundary condition region
- Broad range of existing shapes – rectangle, circle, triangular
- New for 2020 – spot section on cylinders and import from CSV now available
- Spots can be created on internal or external cylindrical faces
- Circular spots can be imported from a CSV file

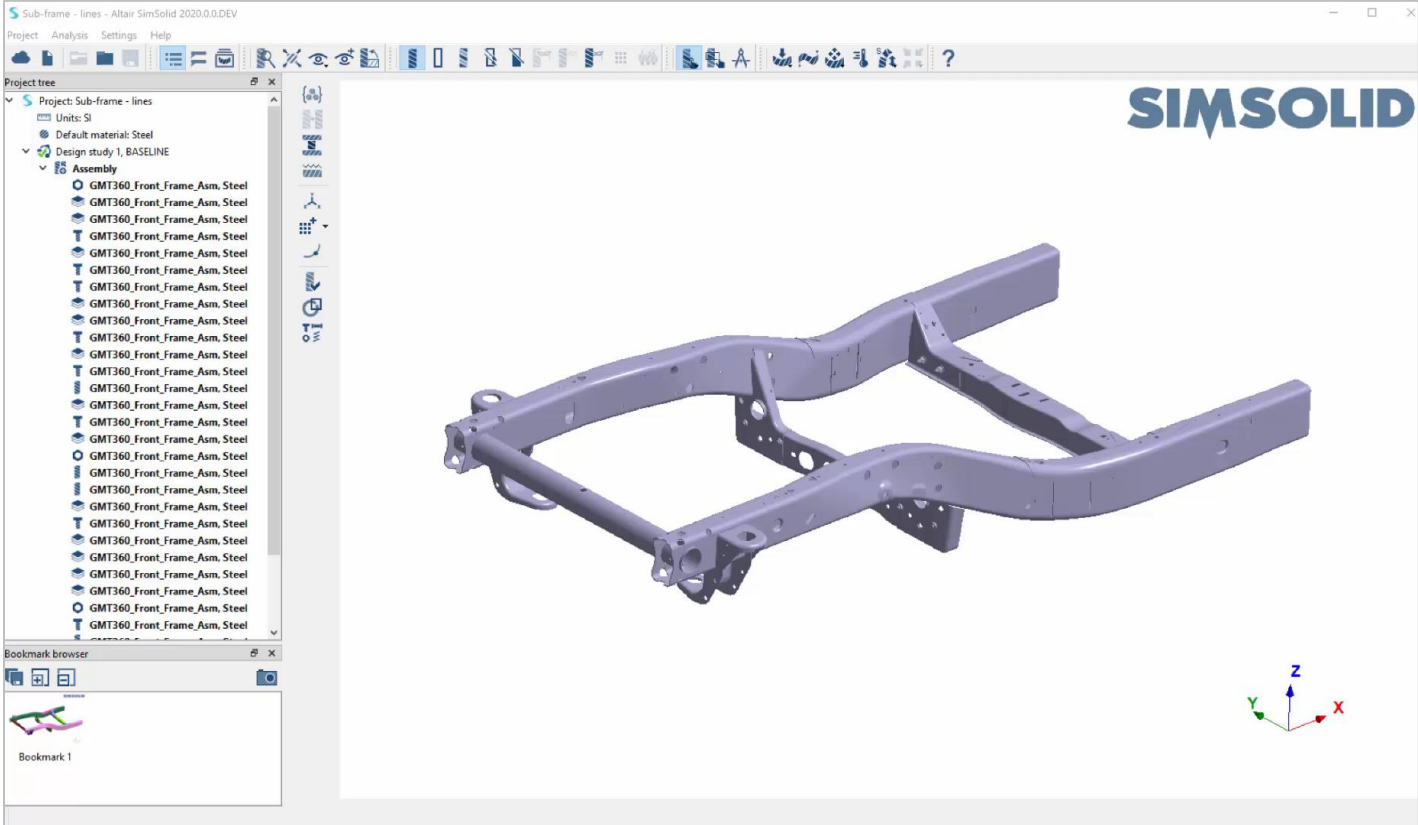


2020.1: Welds from xMCF xml file workflow

The screenshot displays the Altair SimSolid 2020.0.0.DEV interface. On the left, the Project tree shows a sub-frame assembly with multiple 'GMT360_Front_Frame_Asm_Steel' components. The central 3D view shows a complex mechanical assembly with various colored parts (green, pink, blue, purple) and welds. A right-hand panel features the SIMSOLID logo and XML code for defining welds. The XML code includes a header with version and namespace information, followed by a list of welds defined by location IDs and coordinates.

```
<?xml version="1.0" encoding="UTF-8"?>
<xmcf version="1.0" attributeFormDefault="unqualified"
elementFormDefault="qualified"
targetNamespace="http://schemas.altair.com/AltairxMCF"
xmlns="http://schemas.altair.com/AltairxMCF"
xmlns:xsi="http://www.w3.org/2001/XMLSchema-instance"
xmlns:xs="http://www.w3.org/2001/XMLSchema">
  <version>3.0.0</version>
  <date>2020-7-23</date>
  <- connection_group id="1">
    <- connected_to >
    <- connection_list >
      <- connection_1d >
        <- loc_list >
          <loc v="0"> 255.299854 379.070226 654.431309 </loc>
          <loc v="1"> 260.245693 379.071671 654.067411 </loc>
          <loc v="2"> 265.108417 379.071655 653.101572 </loc>
          <loc v="3"> 269.816814 379.071614 651.544412 </loc>
          <loc v="4"> 274.246676 379.071594 649.315215 </loc>
          <loc v="5"> 278.368983 379.071573 646.561026 </loc>
          <loc v="6"> 282.122796 379.071551 643.320259 </loc>
          <loc v="7"> 285.362784 379.071530 639.565989 </loc>
          <loc v="8"> 288.116864 379.071511 635.443582 </loc>
          <loc v="9"> 290.345941 379.071494 631.013530 </loc>
          <loc v="10"> 291.901628 379.071476 626.304779 </loc>
          <loc v="11"> 292.868252 379.071480 621.442163 </loc>
          <loc v="12"> 293.233380 379.071475 616.496352 </loc>
          <loc v="13"> 292.867207 379.071482 611.550709 </loc>
          <loc v="14"> 291.899800 379.071492 606.688249 </loc>
          <loc v="15"> 290.343335 379.071511 601.979654 </loc>
          <loc v="16"> 288.113165 379.071528 597.550211 </loc>
          <loc v="17"> 285.358526 379.071557 593.428195 </loc>
        </loc_list >
      </connection_1d >
    </connection_list >
  </connected_to >
</connection_group >
</xmcf >
```

2020.1: Welds from xMCF xml file workflow



2020.1

- old connection-based seam welds are automatically converted to 3D solid welds wherever possible
- Additional seam weld info on geometry dialogs and other seam weld related info options
- New SSP file format → 3x – 4x smaller filesize
- Enhanced equation solver → Performance improvement of 2x – 5x for each equation solve (once each solution pass)
- New Reverse displayed parts hot key
- Multiple fixes

2020.2

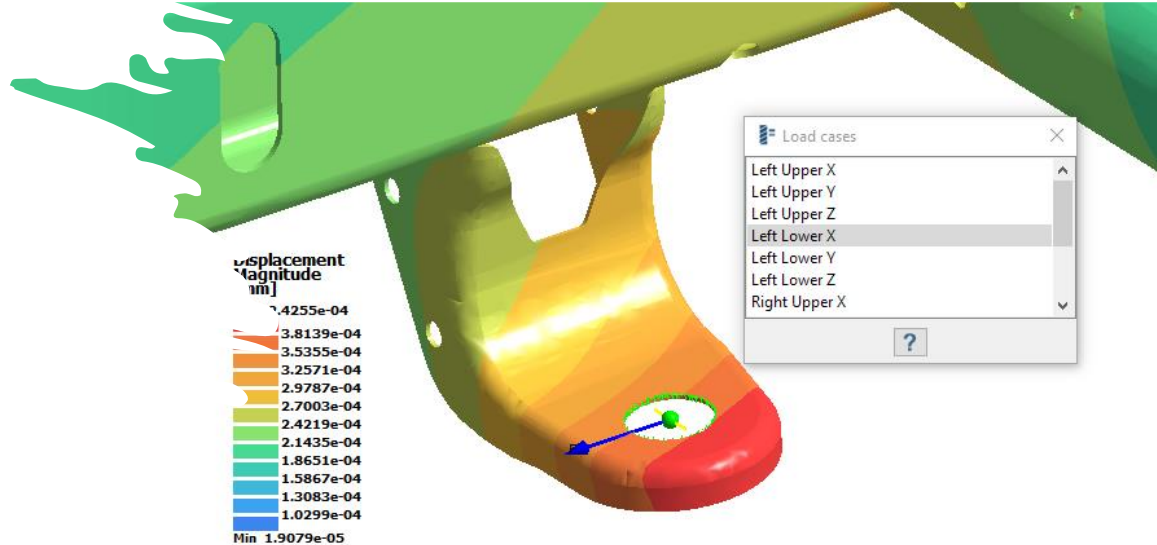
- Fatigue analysis
- Teamcenter integration
- Multi-loadcase workbench
- Weld material assignment
- Seam welds on curvature
- Lighter, faster response mesh
- Faster modal solver

2020.2: Fatigue Analysis

- Stress-life
- Sequential and time-series history
- Mean stress corrections

Multi-loadcases Workbench

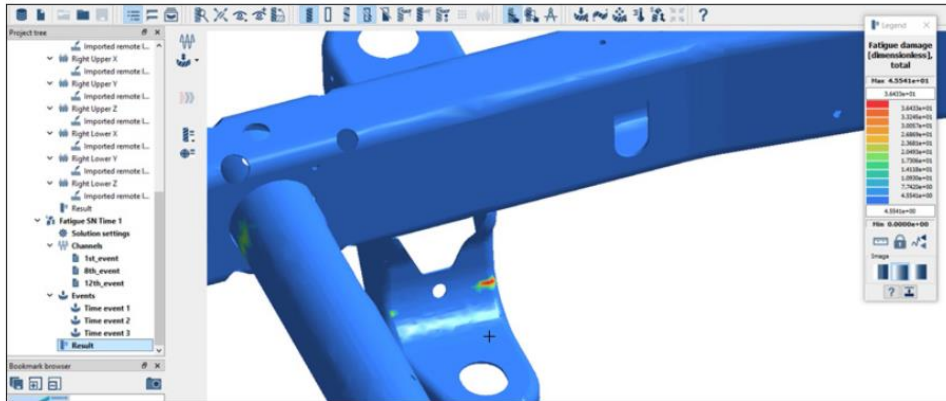
- Multiple static load cases
- Linked to fatigue analysis



2020.2: Fatigue

- Webinar on Demand: “Fatigue Assessment at the Speed of Design”

<https://web.altair.com/2021-fatigue>



This session is ideal for designers and design engineers who want to rapidly explore the fatigue life of their designs during the early CAD stage of development. Join us and see first-hand how new fatigue assessment capabilities within Altair SimSolid can allow you to set up and run a fatigue analysis with multiple load cases in seconds.

Attend & Learn:

- Quickly import and setup large and complex models
- Automatically create loading events and repeats
- Identify damage and product life from loading history

Watch On-Demand

Performance & Robustness

Response Mesh

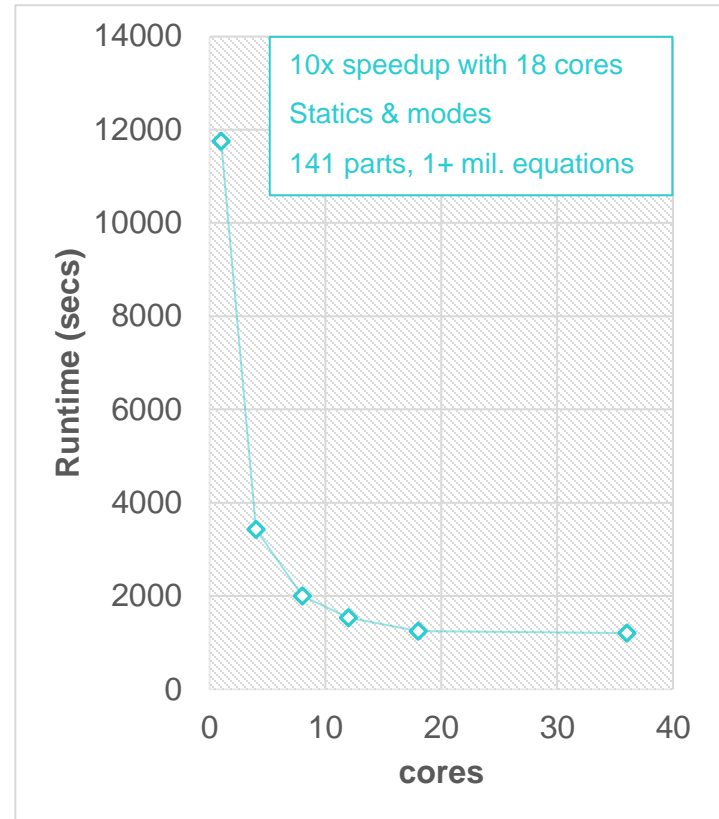
- Up to 16X Faster
- 2X-6X Lighter
- Ability to refine

Equations solver

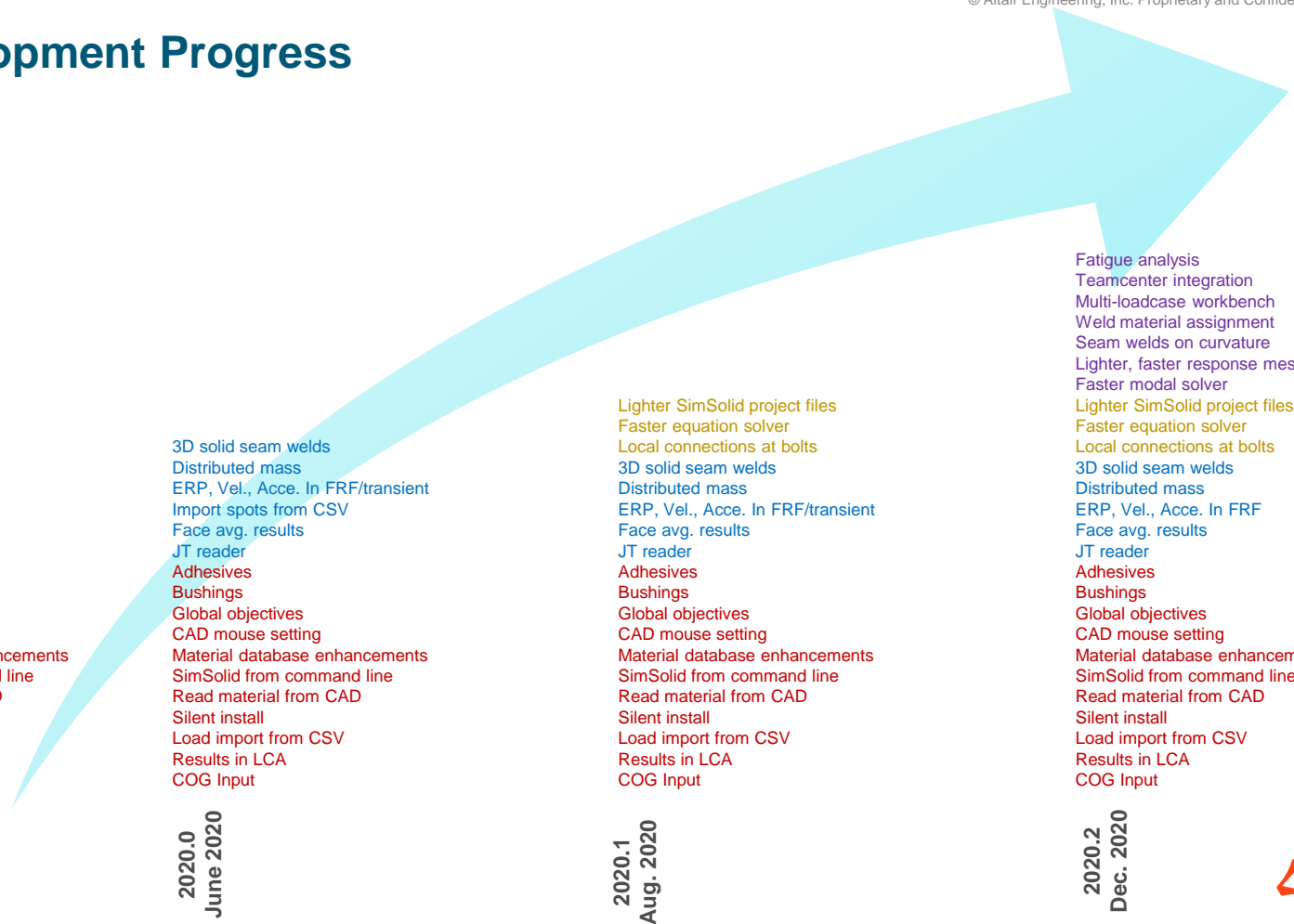
- Up to 2-10X Solution speedup
- Reduced memory requirement
- Modes, linear and nonlinear statics

New SimSolid project file format

- Up to 3-4X lighter
- Faster to write



Development Progress



- Adhesives
- Bushings
- Global objectives
- CAD mouse setting
- Material database enhancements
- SimSolid from command line
- Read material from CAD
- Silent install
- Load import from CSV
- Results in LCA
- COG Input

2020.0
June 2020

- 3D solid seam welds
- Distributed mass
- ERP, Vel., Acce. In FRF/transient
- Import spots from CSV
- Face avg. results
- JT reader
- Adhesives
- Bushings
- Global objectives
- CAD mouse setting
- Material database enhancements
- SimSolid from command line
- Read material from CAD
- Silent install
- Load import from CSV
- Results in LCA
- COG Input

2020.1
Aug. 2020

- Lighter SimSolid project files
- Faster equation solver
- Local connections at bolts
- 3D solid seam welds
- Distributed mass
- ERP, Vel., Acce. In FRF/transient
- Face avg. results
- JT reader
- Adhesives
- Bushings
- Global objectives
- CAD mouse setting
- Material database enhancements
- SimSolid from command line
- Read material from CAD
- Silent install
- Load import from CSV
- Results in LCA
- COG Input

2020.2
Dec. 2020

- Fatigue analysis
- Teamcenter integration
- Multi-loadcase workbench
- Weld material assignment
- Seam welds on curvature
- Lighter, faster response mesh
- Faster modal solver
- Lighter SimSolid project files
- Faster equation solver
- Local connections at bolts
- 3D solid seam welds
- Distributed mass
- ERP, Vel., Acce. In FRF
- Face avg. results
- JT reader
- Adhesives
- Bushings
- Global objectives
- CAD mouse setting
- Material database enhancements
- SimSolid from command line
- Read material from CAD
- Silent install
- Load import from CSV
- Results in LCA
- COG Input

Solution Breadth



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
- Linearized stresses



Materials

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



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
- Total strain energy

2021 New

- Stress linearization
- Strain-life Fatigue (EN uniaxial / multiaxial)
- Material Orthotropy
- Batch mode
- Sheet repair
- Multiple version install

Material name

Material 6

System of units SI Material type orthotropic

Property	Value	Units
▼ Mechanical properties		
Density	0.0000000000e+00	[kg/m ³]
Elasticity modulus X	0.0000000000e+00	[pa]
Elasticity modulus Y	0.0000000000e+00	[pa]
Elasticity modulus Z	0.0000000000e+00	[pa]
Poisson's ratio XY	0.0000000000e+00	[dimensionless]
Poisson's ratio YZ	0.0000000000e+00	[dimensionless]
Poisson's ratio ZX	0.0000000000e+00	[dimensionless]
Shear modulus XY	0.0000000000e+00	[pa]
Shear modulus YZ	0.0000000000e+00	[pa]
Shear modulus ZX	0.0000000000e+00	[pa]

2021 Enhancements

- Material database
 - input of user defined SN / EN curves (Fatigue)
- Total Strain Energy result
- Absolute acceleration output
- Adhesive mass output
 - reviewing mass of the adhesives in model through info dialogue
- Weld export
- Coordinate System
 - rotate local axis at an angle

2021 Enhancements

- Animation of geometrical-non-linear results
- Bearing loads
 - can now be applied to any surface of revolution. Allows input of axial load / torque additional to radial.
- Sheet/face recognition improvement
- International keyboard decimal

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

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 _{VM}	356.5MPa	366.5MPa	2.8%

A summary of results for all benchmarks(NAFEMS)

Check for Other Media Testimonials:



57 https://www.nafems.org/publications/resource-center/bm_jan_20_1/

<https://www.digitalengineering247.com/article/altair-simsolid-walkthrough/simulate>

<https://www.develop3d.com/reviews/review-altair-simsolid-simulation-CAD-design-engineering>

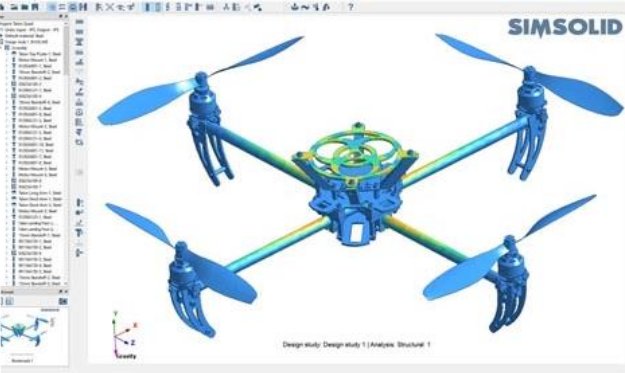
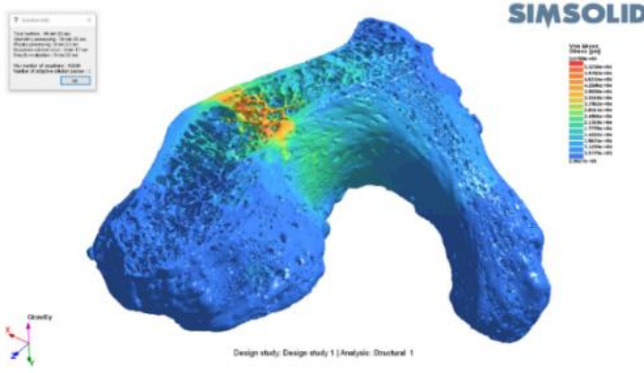
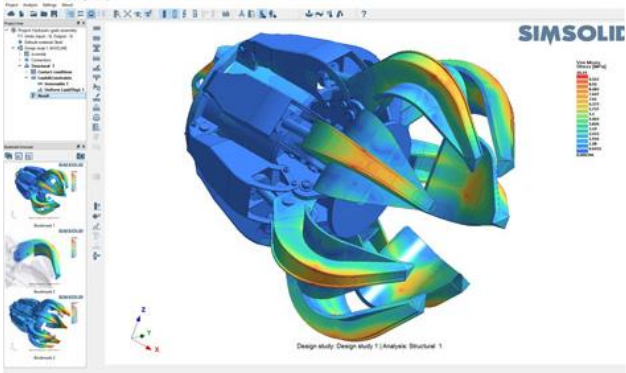


JOIN THE SIMULATION REVOLUTION

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
- or give feedback after meeting finished

QUESTIONS / ANSWERS



THANK YOU

altair.com



#ONLYFORWARD