IM123395

Using SIMSOLID with Autodesk Fusion 360

Ken Welch
SIMSOLID Corporation

Learning Objectives

- Learn a new complementary software application for Fusion 360
- Learn how to do structural analysis without meshing
- Learn how to do large assembly design studies with evolving design geometry
- Learn how to do structural simulation of lightweight generative designed parts within the context of a large assembly

Description

SIMSOLID™ is next generation, high capacity, structural FEA. It uses new computational methods which allow the solution of assemblies with hundreds to thousands of parts directly on a standard desktop computer. SIMSOLID completely eliminates geometry simplification and meshing, the two most time consuming, expertise extensive and error prone tasks done in traditional FEA. SIMSOLID is the perfect complement to existing CAD embedded simulation. It extends the analysis range and provides feedback in seconds to minutes.

This class will provide an introduction to SIMSOLID working with Autodesk Fusion 360. It will give an in-depth overview of SIMSOLID's unique meshless approach and will provide numerous industry examples including large assemblies and lattice based designs. The attendee will learn new techniques for performing preliminary design analysis on rapidly evolving designs.

Speaker

Ken is cofounder and CEO at SIMSOLID Corporation. Ken has over 30 years experience in the CAE industry in a variety of technical, sales and executive roles at companies including PDA Engineering, Rasna, Moldflow and MSC Software. Ken holds a B.S. and M.S. in civil engineering from the University of California, Davis. Ken is located in Newport Beach, California USA.

Introducing SIMSOLID – innovative structural analysis partner application for Fusion 360

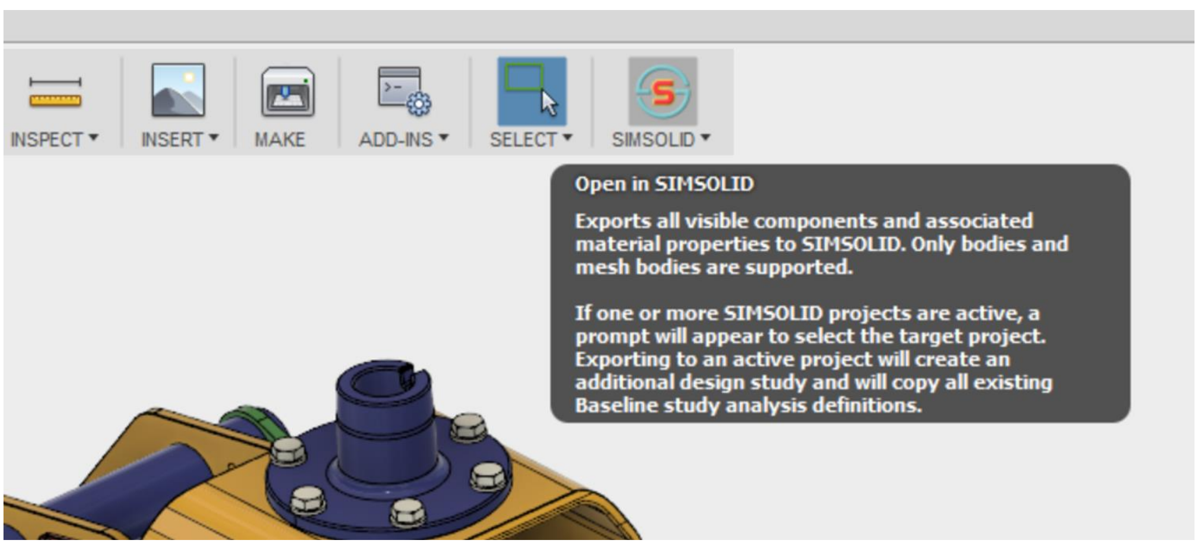
SIMSOLID™ is next generation, high capacity, structural FEA. It uses new computational methods which allow the solution of assemblies with hundreds to thousands of parts directly on a standard desktop computer. SIMSOLID eliminates geometry simplification and meshing, the two most time consuming, expertise extensive and error prone tasks done in traditional FEA.

SIMSOLID is the perfect complement to existing Fusion 360 embedded simulation. It extends the analysis range to more complex shapes and provides feedback in seconds to minutes. SIMSOLID operates with Fusion 360 as a tightly integrated attached simulation environment. It has a shared memory pipeline for Fusion 360 geometry and materials and has a unique design study process that allows associative update of geometry without needing to recreate analysis definitions.

How the Fusion 360 integration works

SIMSOLID's integration with Autodesk Fusion 360 works with both solids and mesh based geometry. This is especially useful for complex 3D printed parts containing complex lattice type structures.

A FUSION 360 Add-in is used to connect the two applications. There is complete integration between both the assembly tree and the related material properties. To invoke SIMSOLID from the Fusion 360 Model menu, simply pick the SIMSOLID icon and all visible geometry will be loaded into SIMSOLID.



Here is a video that demonstrates how the integration works: https://youtu.be/Ny-zkBIPFww

Huh? Structural simulation without meshing, how is that possible?

SIMSOLID is the world's first structural simulation system that has been developed from the ground up as a large assembly solver. SIMSOLID is a generalization of classical FEA. While it uses the same variational principals, the difference comes in how we build approximation functions and associate them with geometry. SIMSOLID doesn't break the system into finite elements, instead it discretizes the original geometry in a more abstract mathematical way.

SIMSOLID uses high order functions built on the fly during the solution phase. Geometry is decoupled from these functions, which allows the freedom to use the most accurate functions possible. For example, when performing thermal analysis SIMSOLID uses harmonic polynomials which precisely meet the thermal equation. Multiple solution passes are performed and with each pass errors are quantified and the functions are adapted. In contrast to other commercial FEA products, adaptive analysis is not an option. In SIMSOLID, it is always active, even for large assemblies. In this respect, SIMSOLID is unique in the industry.

Notable features of SIMSOLID include linear and nonlinear statics, modal, thermal and thermal-stress solution methods; expansive connector types including bolted, bonded, sliding, separating with friction and welded (spot, laser, fillet); and in-depth reaction forces including spot weld forces, connection forces and part resultant forces.

SIMSOLID compared to traditional FEA

Here is a quick summary of how SIMSOLID and traditional FEA compare.

NUMERICAL METHODS	
Traditional FEA	**SIMSOLID**
Simple regions – TET, etc.	Arbitrary regions – whole part can be a region
DOF is associated with a node – it is point-wise	DOF is not point-wise. It can be associated with volumes, surfaces, lines and/or point clouds
DOF are nodal Ux, Uy, Uz displacements	DOF are integrals over corresponding geometrical objects, not nodal
3 DOF per node	Many DOF per single associated geometry object are possible, depends on solution adaptation
Shape functions are simple low degree interpolation polynomials	Shape functions can be of arbitrary class • complete standard polynomials • divergence-free polynomials • harmonic polynomials • non-polynomials

Here is a summary that compares the accuracy workflow of SIMSOLID with traditional FEA. Note that there are far fewer operations and settings required in SIMSOLID.

ACCURACY WORKFLOW	
Traditional FEA	**SIMSOLID**
Geometry level of detail decision by user	Full geometry detail – modeling errors minimized
Types of elements decision by user	No elements
Mesh density and distribution based controls decision by user	No meshing
Correct interpretation of analysis settings by user • Solver & solution methods • Tolerances and options	No settings in dynamics and non-linear analyses including separating contact
Solution adaptation is mostly based on local energy density change, it is relative • Rarely used for assemblies	Solution adaptation is based on local energy density change and absolute errors on boundary • Always active • Easy to set both global (whole assembly) and local (part based) solution adaption • Reaction forces at support and connections are very accurate

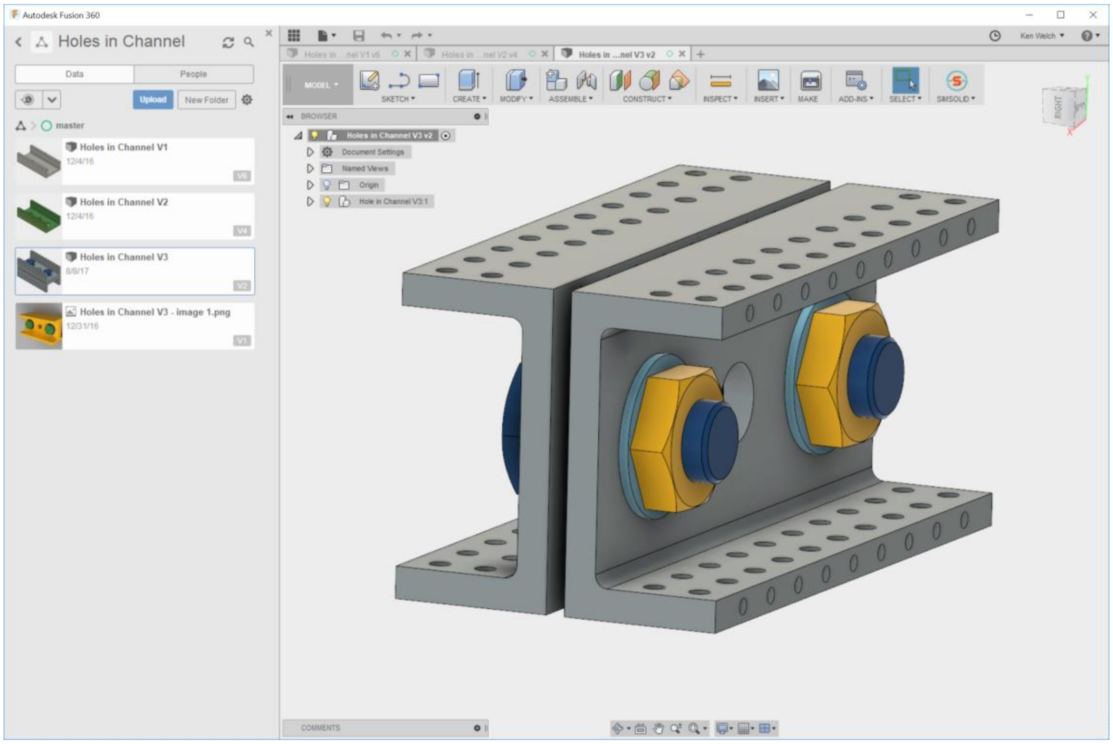
SIMSOLID applications

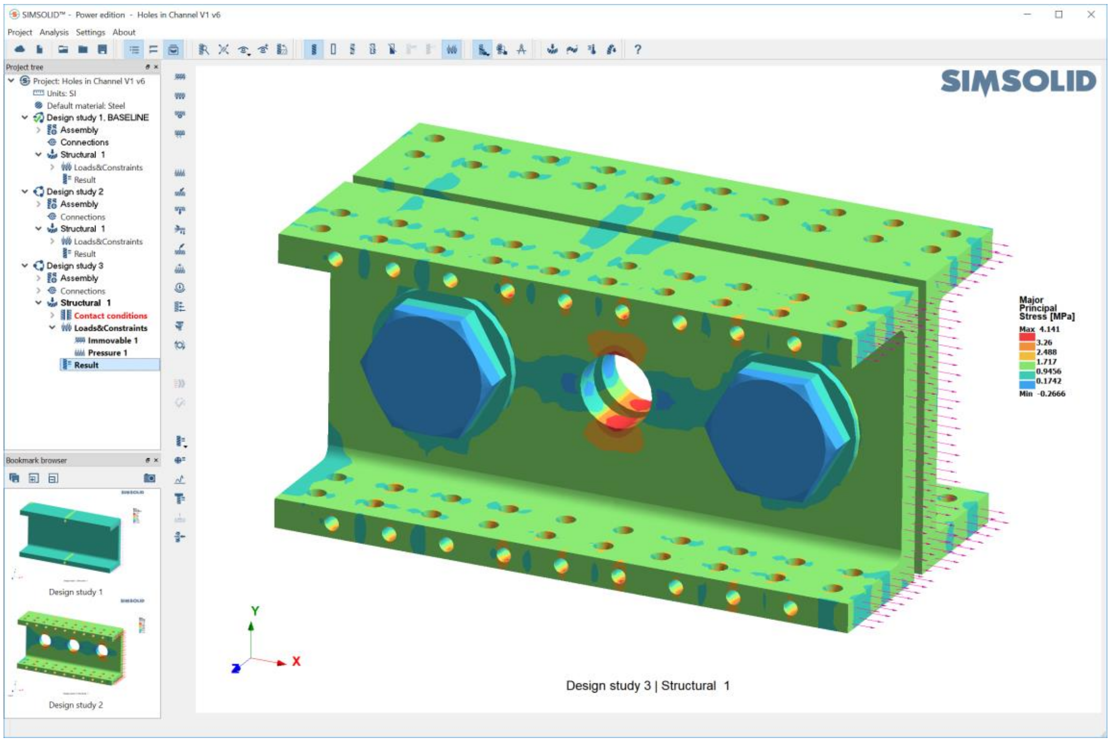
For the remained of the session, various applications of SIMSOLID working with Fusion 360 will be presented.

Demonstration 1 – Fusion 360 integration, smart functions, design studies

The goal of this demonstration is to show the tight integration between Fusion 360 and SIMSOLID. All parts and material properties are linked. Further, design modifications are automatically processed as SIMSOLID design studies. All associated analysis definitions are automatically synced between the studies.

Solution accuracy is controlled by the SIMSOLID adaptive solution process. By default, 3 adaptive solution passes are performed and error analysis is done with each pass. The solution is aided by SIMSOLID smart functions which automatically recognize through hole geometry features and apply specific stress functions to increase accuracy and speed convergence.





Demonstration 1 – Fusion 360 design model (top) and SIMSOLID design studies (bottom) of all configurations

Demonstration 2 – Nonlinear separating contact, bolt tightening

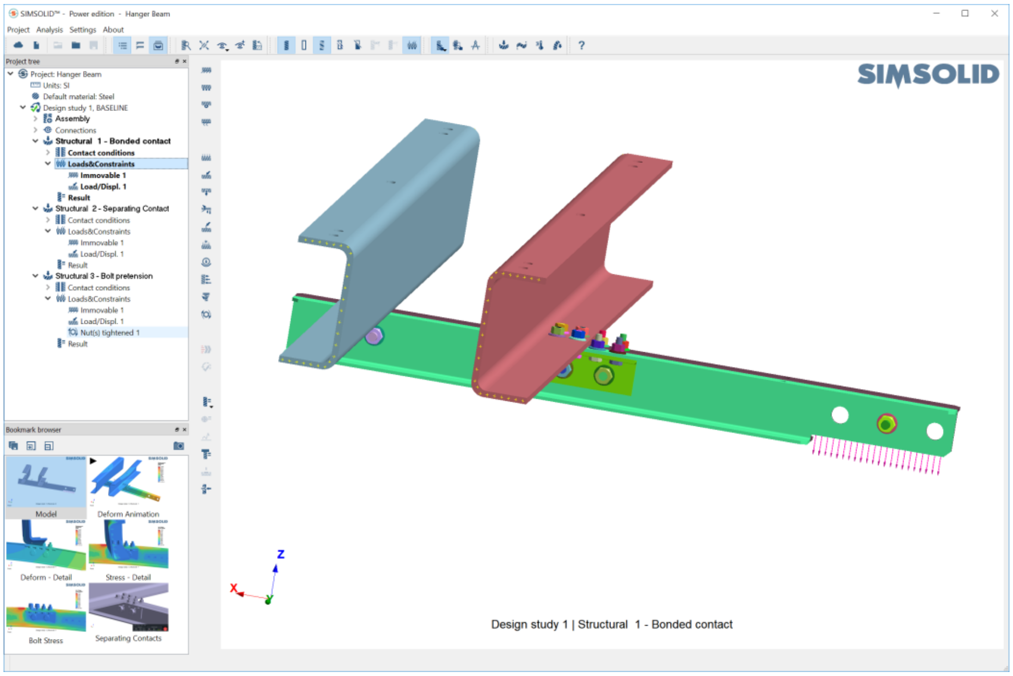
The goal of this demonstration is to show how quickly SIMSOLID can model and analyze various assembly connection configurations.

The model of a generic hanger beam consists of 54 parts, including bolts, nuts and washers. Three configurations of the connection between the cross member beams is considered.

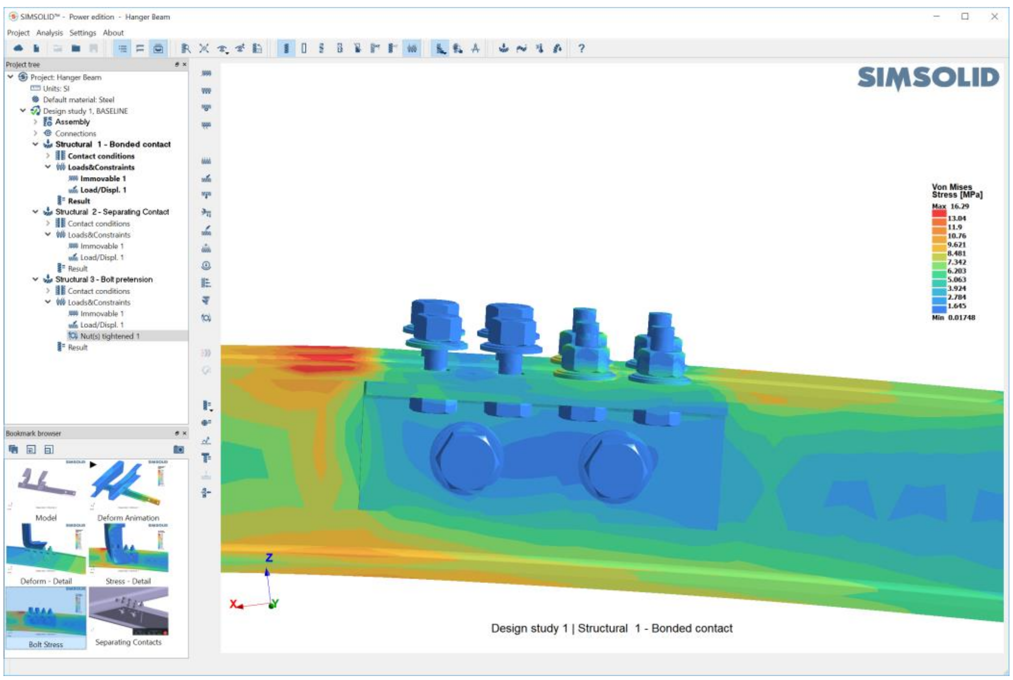
In the first analysis, the joint is assumed completely bonded. This is the most common modeling technique and it is a fast way to understand the overall load path of the assembly. The disadvantage of this method is that the load is transferred between the beams and not the connecting bolts. Solution time for this analysis with 4 adaptive solution passes is 11 seconds.

To better understand the bolt loads, separating contact with friction is applied to all connections between beams. Now the bolts will carry the load and a better representation of the bolt stress and reaction forces is obtained. Note that this makes the analysis nonlinear. Additional iterations are required. Solution time for this analysis with 4 adaptive solution passes is now 30 seconds which is still reasonable. In SIMSOLID, advanced connection modeling that includes separating contact with friction is very practical, even on larger assemblies.

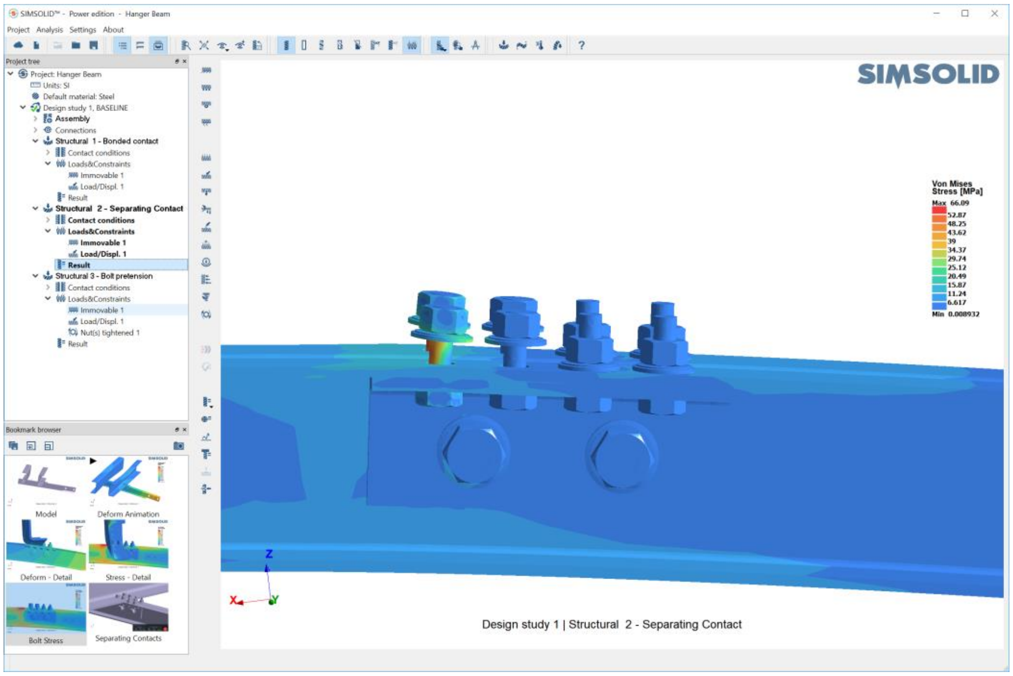
One final analysis refinement is done by applying both separating contact and bolt tightening loads to the bolts in the joint. In SIMSOLID, the tightening load can be prescribed by turn-of-nut, torque plus friction or target axial load. In our case, a target axial load was prescribed. This matches the design spec for the joint and is the most detailed representation of the stress state.



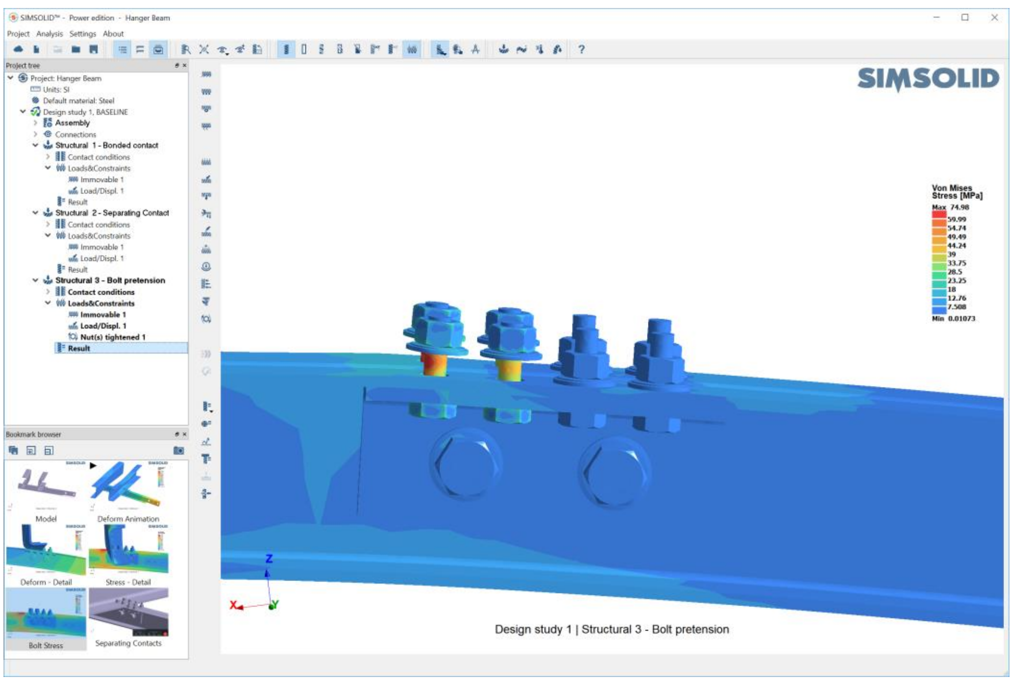
SIMSOLID analysis model of hanger beam showing constraints and loads.



Stress detail, bonded case. Top beam hidden for better viewing clarity.



Stress detail, separating contact case. Top beam hidden for better viewing clarity. The back bolts take the majority of the load.



Stress detail, separating contact + bolt tightening case. Top beam hidden for better viewing clarity. Since all 4 bolts are tightened, the stress is better distributed among them.

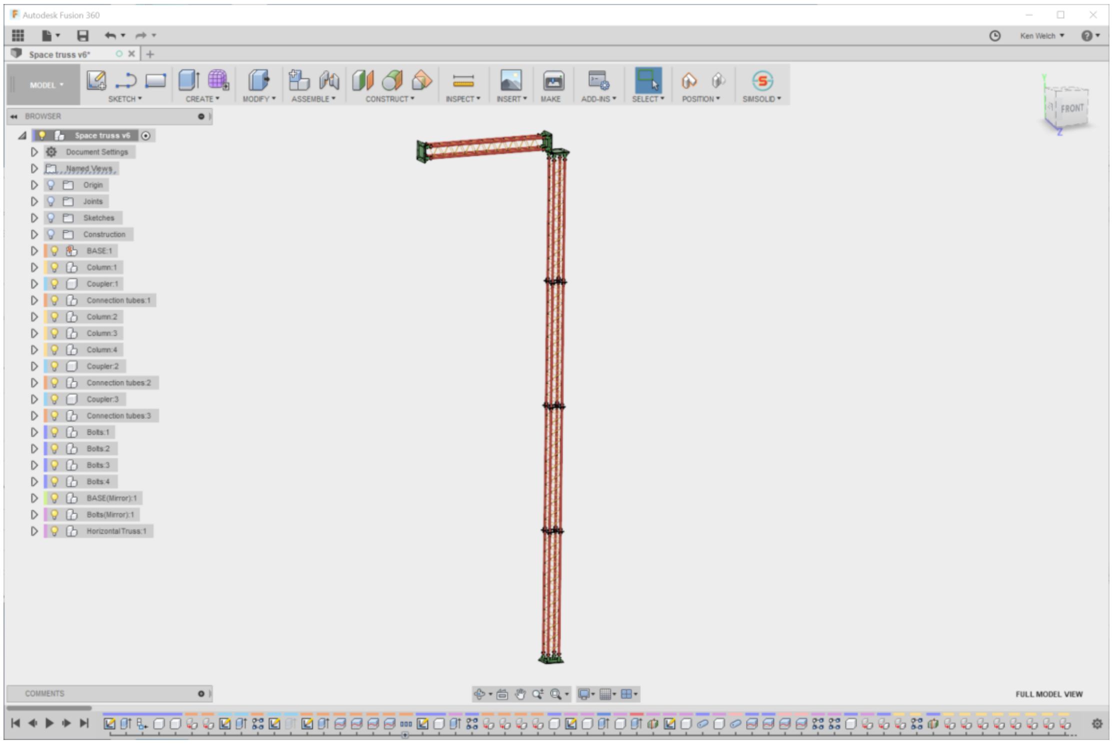
Demonstration 3 – Modal analysis, virtual connectors

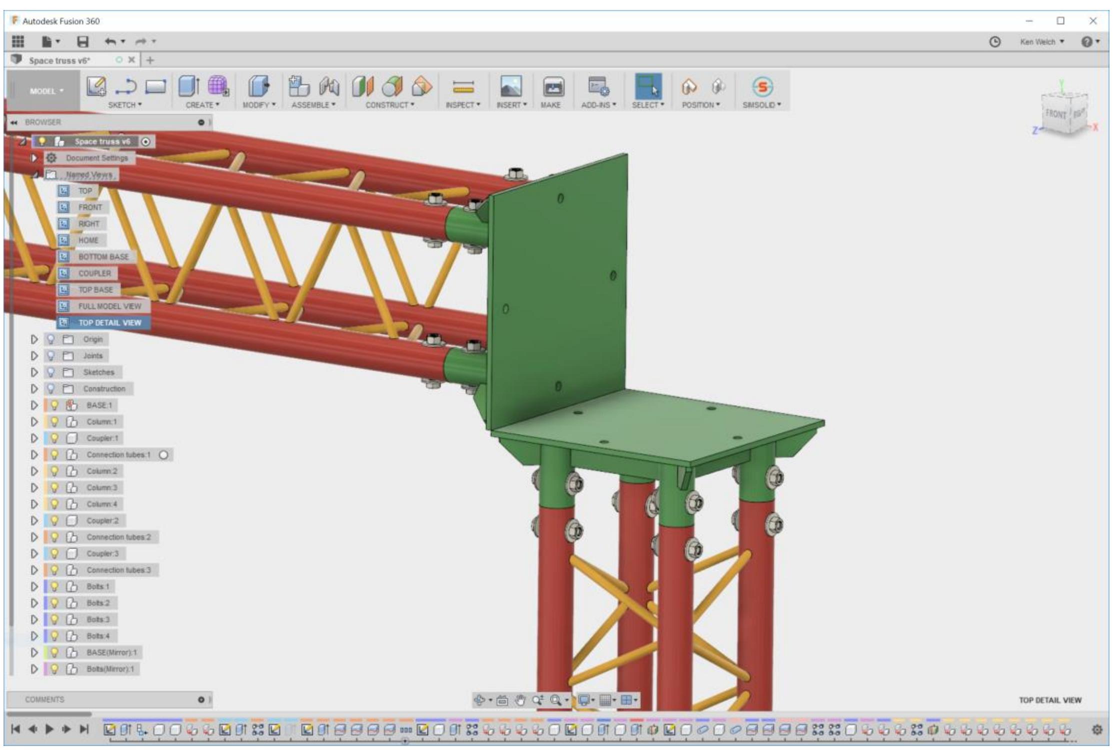
SIMSOLID is used to quickly evaluate the modal properties of a mast structure.

The goal of the simulation is to calculate the primary dynamic modes of vibration for a fixed base mast, then to examine the effect on these modes when the additional horizontal section is added. Since the connection block between the mast sections has not yet been designed, SIMSOLID virtual connectors are used to quickly model the joint.

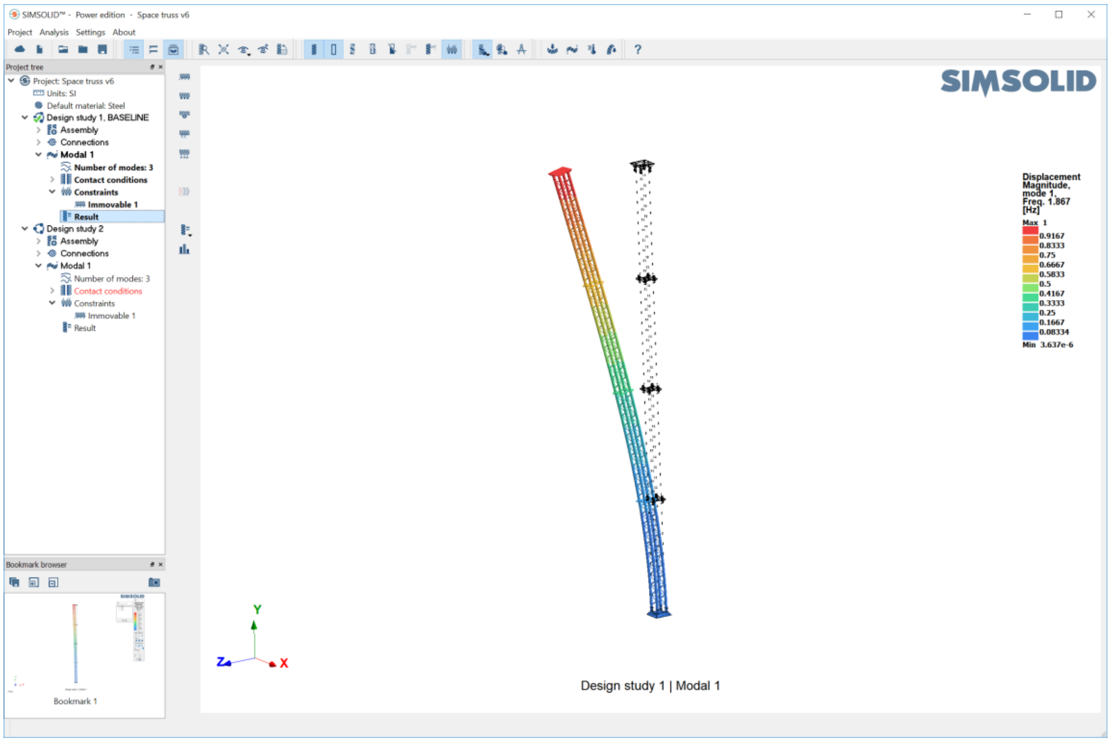
The initial steel mast structure contained 545 parts consisting of solid bars, hollow tubes, full fidelity bolts, nuts and washers. The additional horizontal section added another 182 parts bring the entire assembly to 727 parts. No simplification of the geometry as required. Even the threads on the bolts remain.

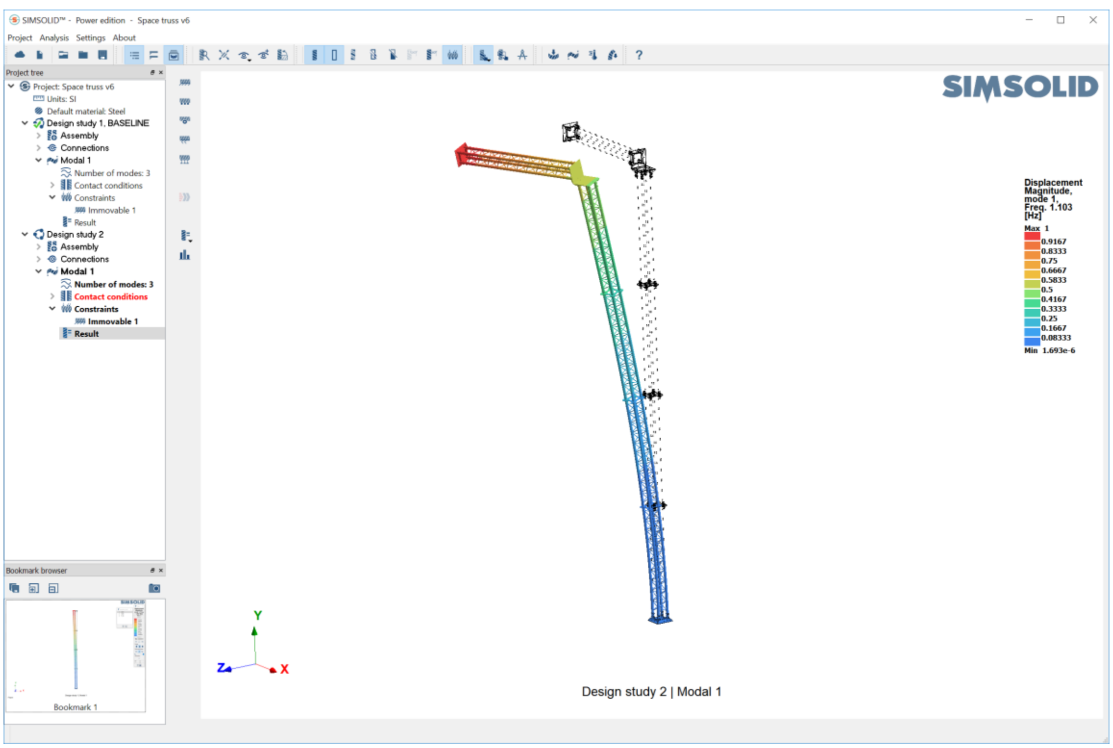
Note, one more quick refinement of the analysis is possible. The mass of the connection block can easily be included using a SIMSOLID rigid part instead of the virtual connector. In this case, the envelope of the connection block is created in Fusion 360 and the appropriate mass of the block is added in SIMSOLID's rigid part definition.





Demonstration 3 – Two views of the Fusion 360 design model





Demonstration 3 – SIMSOLID analysis model with two design studies – single mast structure (top) and multi-section mast (bottom)

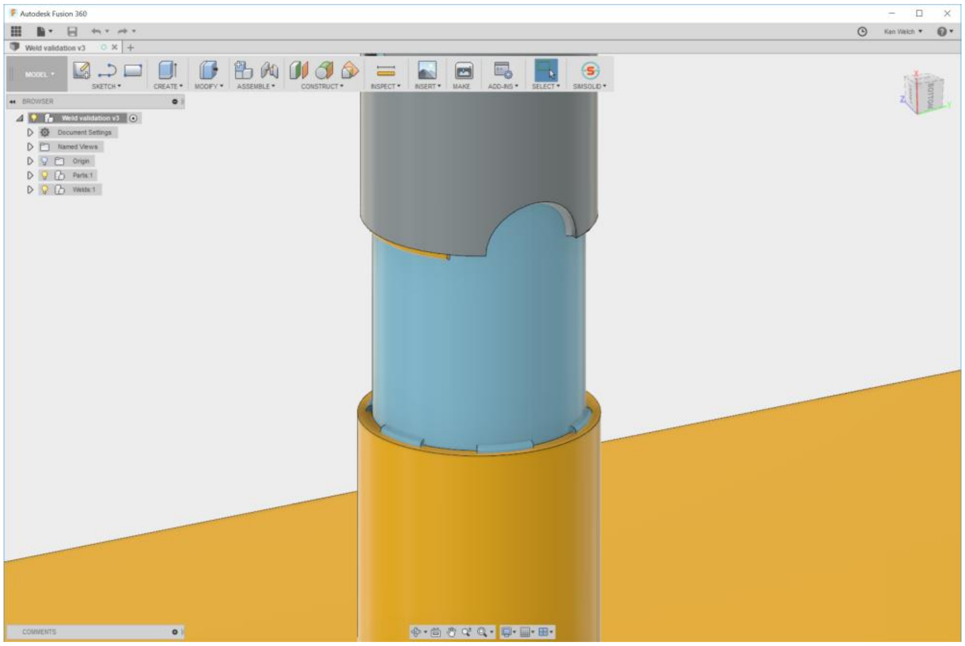
Demonstration 4 – Fusion 360 working with advanced welded connection

SIMSOLID has specific definitions for modeling spot, laser or fillet (seam) welded joints. This demonstration show how advanced fillet welds can be defined in Fusion 360 and then automatically transferred to SIMSOLID for analysis.

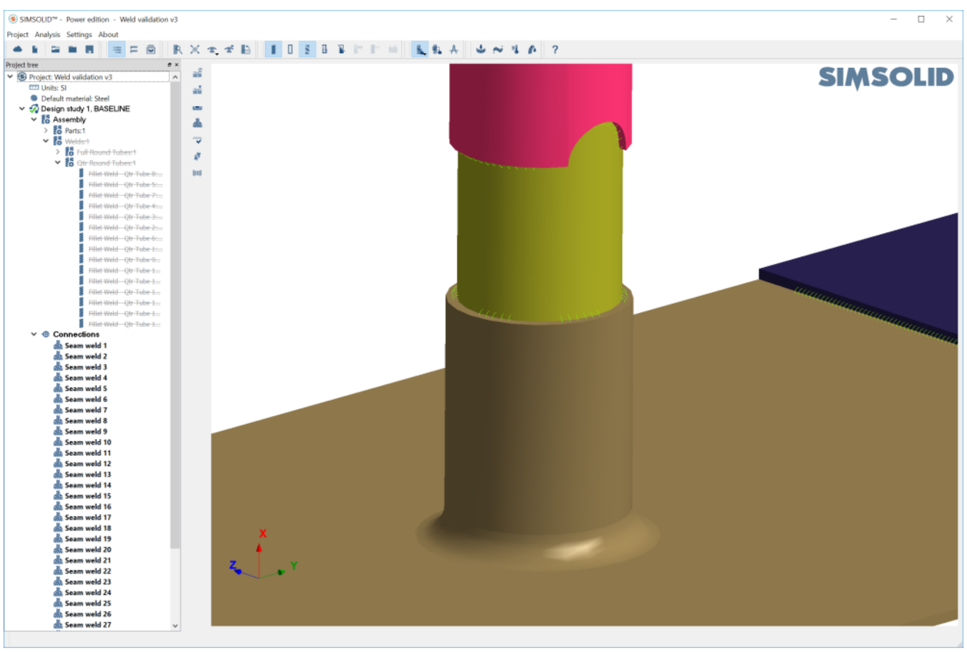
Fillet (seam) welds in Fusion 360 are modeled at solid tubes or wedges. These entities can be straight or curved and are located at the part boundaries where the weld is to be applied. Note that they can overlap the real part geometry. They are used for weld definition only and will be removed prior to doing the analysis. The diameter of the solid tubes is not used in the definition. It only defines the part edge where the weld will be defined along with the start and end location. Some of you may note that this technique is a common method used by many in the auto industry.

To create these welds in SIMSOLID do the following. First transfer the model including the weld solids to SIMSOLID. Next open the weld definition dialog and select the welds from solids tab. Specify the desire weld diameter. In our case we will use 5 mm. Now select the tubes and SIMSOLID will convert them to welds. Don't want to pick the welds individually? SIMSOLID has a short cut. Select all parts and SIMSOLID will automatically remove the parts that are not weld related. Select OK to complete the fillet weld definition. The tubes will be suppressed so that they are not included in the analysis.

Using this technique welds can be quickly defined and the model is analysis ready in seconds.



Seam weld solid tubes in Fusion 360.



Seam weld definitions automatically created in SIMSOLID. Note, a single SIMSOLID operation creates the welds and suppresses the weld definition solids.

Demonstration 5 – combining solids and mesh based models

SIMSOLID does not create a mesh. This makes it easier to integrate design models containing both solids and mesh based geometry. Analysis of large assemblies with hybrid (solid/mesh) part models is practical. This is especially useful for complex 3D printed parts containing complex lattice type structures.

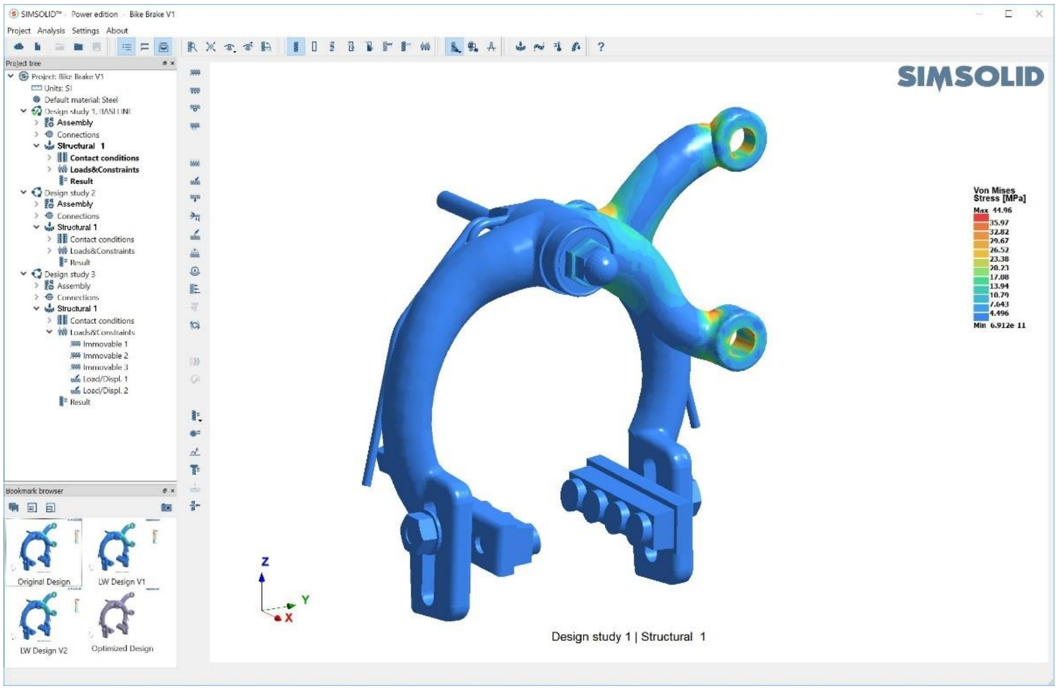
In this demonstration, we highlight one possible workflow for 3D lattice based design.

To begin, the original design is analyzed in SIMSOLID. From this, an initial stress state is calculated and reviewed. Next the stresses are exported and used as input on a lattice creation system. In this example, nTopology uses this stress state to create lightweight lattice models for both of the main brake arms. The stress fields guides the lattice such that more material is located in the high stress areas, more lattice in the low stress areas, etc.

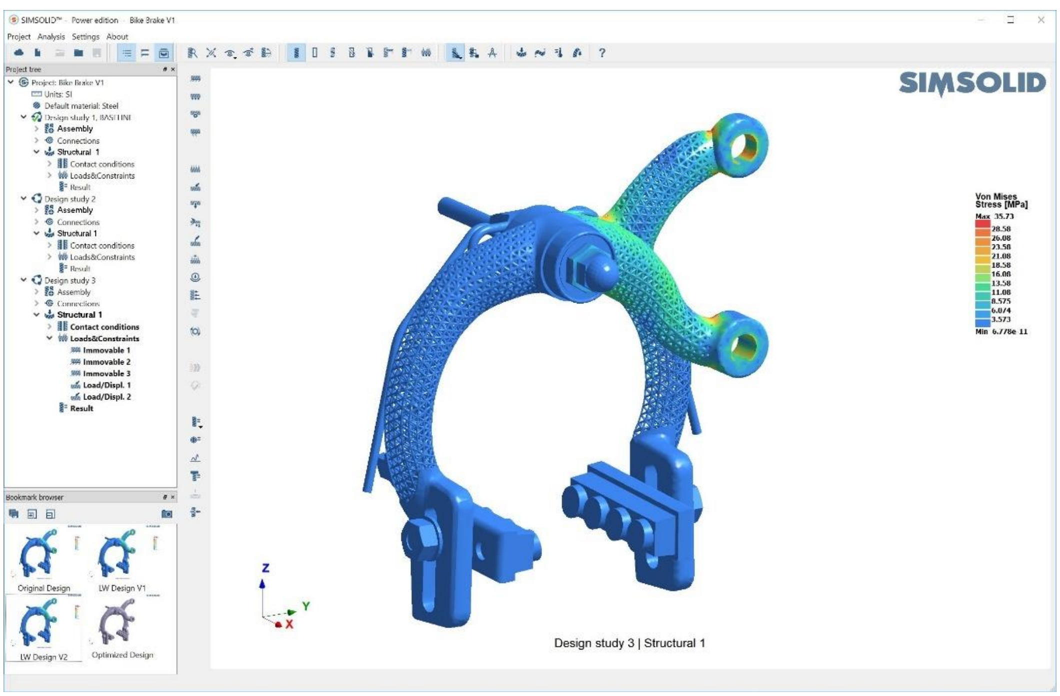
Next this lightweight model is imported back into SIMSOLID and placed in a second design study. The analysis constraint and load environment from the original design study is automatically reapplied to the second study. Finally, the second study is analyzed and the results compared with the original all solid based design.

The analysis time requirements are more substantial for the complex lattice based model. The original model contains 21 parts and the total solution time for 3-adaptive solution passes is only 19 seconds. The fully assembly including the lightweight lattice arms runs in about 5-1/2 minutes. Still this is a very reasonable time considering this was done on generic 4-core desktop systems. With larger systems, faster time are possible. SIMSOLID solver is very parallel and takes full advantage of all cores that are available.

Other workflows are possible. SIMSOLID works great with Fusion 360 mesh and solids based hybrid models.



Bike brake assembly. Stress analysis on original solid parts



Bike brake assembly. Stress analysis on lightweight lattice based design

Summary and sources for more information

This class introduces you to a new Fusion 360 partner product SIMSOLID. It explains what SIMSOLID is and how it works. It provides details on how to pass design-analysis models seamlessly between both applications. Finally, it provides several examples including doing large assembly design studies with evolving design geometry and simulation of lightweight generative designed parts within the context of a large assembly.

For more information please refer to the following sources of information:

WEB:	www.simsolid.com
FACEBOOK:	www.facebook.com/simsolid.corporation
TWITTER:	https://twitter.com/simsolid
LINKEDIN:	http://www.linkedin.com/company/simsolid-corporation