National Aeronautics and
Space Administration

Marc Simulations at NASA Johnson Space Center

Part 1: Inflatables and Softgoods Simulations

MSC Software Nonlinear Users Meeting
April 17, 2017
Ann Arbor, Michigan, USA

Satish Reddy
Chief Engineer, Structural Analysis

Jacobs Technology

View in Slide Show Mode

OVERVIEW

Major mass reduction opportunities exist when spacecraft and habitation modules are built as inflatables instead of conventional composite or metallic hard structure. These inflatables as well as parachutes used for spacecraft descent and landing use soft good straps and tether cords extensively.

Sophisticated analytical finite element analysis techniques exist to model metallic and composite structure but those that model soft goods and straps are very rudimentary. Safety margins of metallic links and structure that are tethered to are dependent on the loads transferred by such soft goods straps. Their elasticity and ability to adequately transfer loading to the metallic parts become critical to avoid the use of excessive safety factors that result in overdesign.

The methodology to simulate soft goods in finite element analysis is developed in this work. It addresses structural analysis modeling and simulations of the Orion parachute system and extends it to inflatable spacecraft and habitats.

INNOVATION

Exploited MSC Marc Finite Element Analysis Software’s rebar & elastomer modeling elements to simulate soft good fabric weave, strength and stiffness in conjunction with its contact simulation abilities to model soft goods and their interactions with hard structural components.

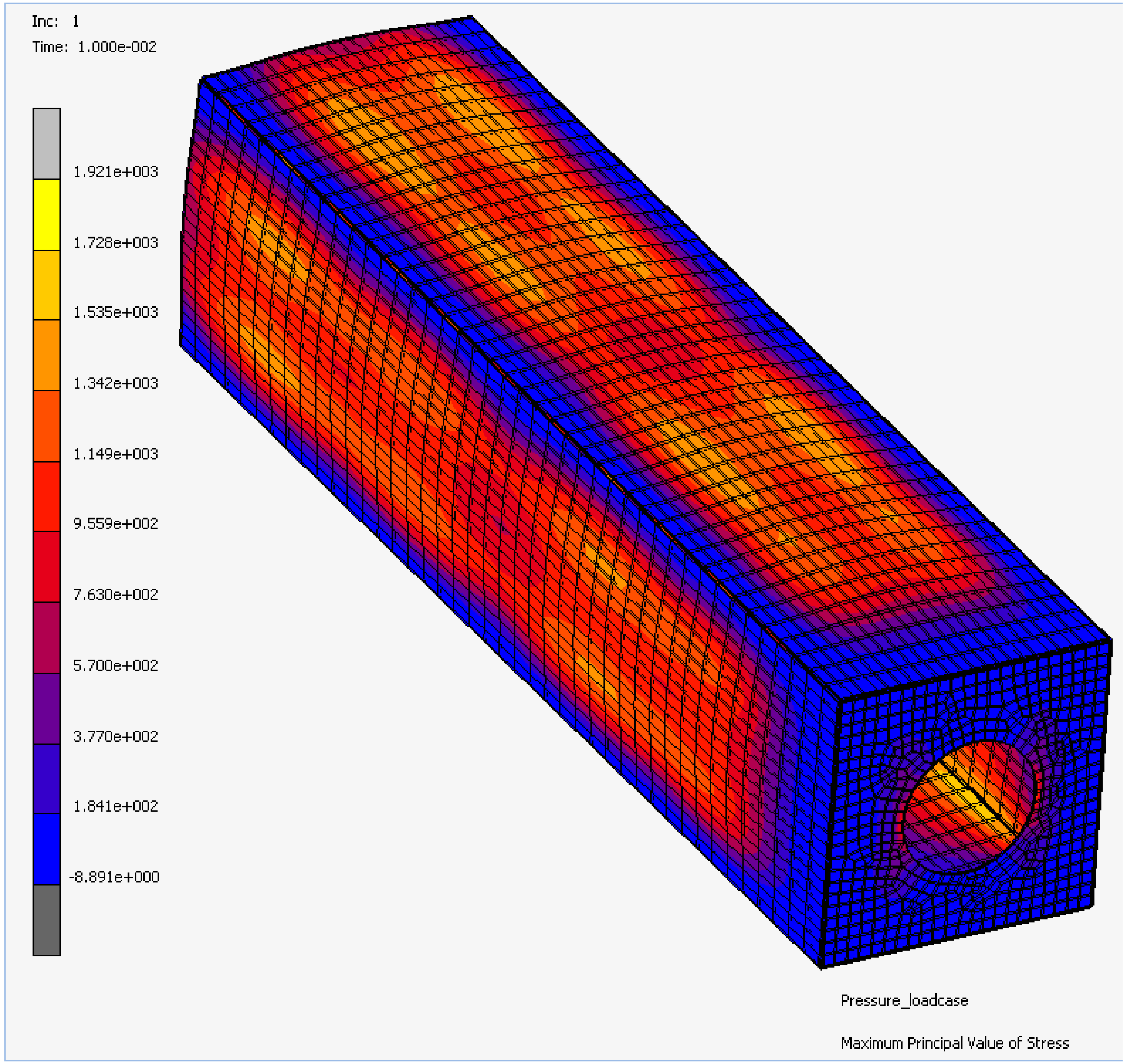
OUTCOME

Demonstrated successful simulation of:

- Parachute tethers under static and dynamic loads.
- Inflated habitats with multilayer fabric spread on collapsible rigid member frames.
- Inflated spacecraft modules (‘module in a bag’) with reinforced tether straps interfaced with rigid docking ports. These may be revolved to create 1g during interplanetary space travel or in planetary orbit.
- Inflatable seals for misalignment tolerant spacecraft module joints.

Mitigates technical risk in the development of such safety critical structures - designs may be optimized for strength and stiffness, loads and structural safety margins better predicted and weight saving opportunities realized.

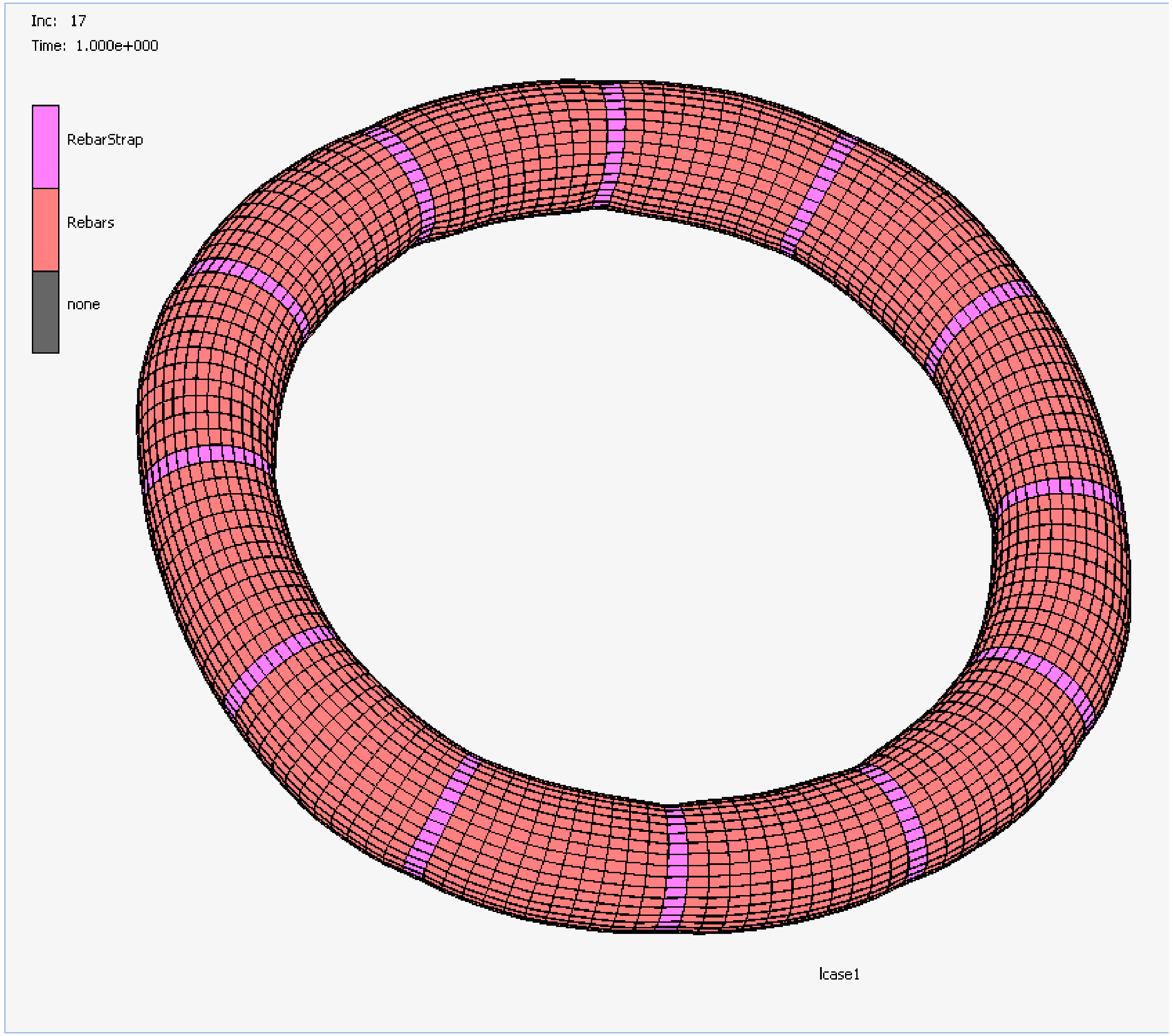
EXAMPLE 1 : RECTANGULAR BOX HABITAT



Modelled a collapsible frame with a rigid circular docking port on one side

Fabric stress and deflection pattern look realistic

EXAMPLE 2 : TOROIDAL INFLATED SOFTGOODS SPACECRAFT, SPUN ABOUT AXIS TO CREATE ARTIFICIAL GRAVITY

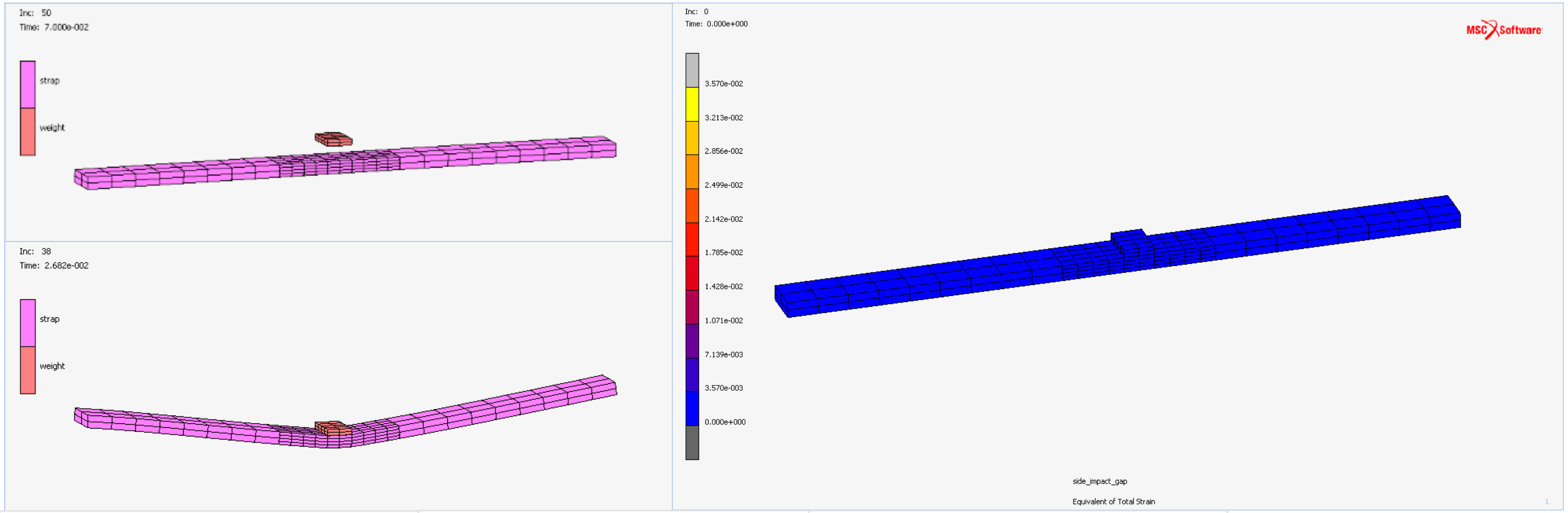


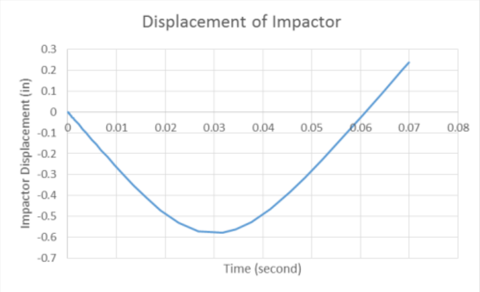
2001 Space Odyssey movie: https://youtu.be/1wJQ5UrAsIY

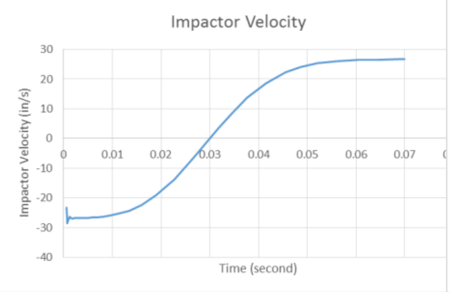
- No rigid frame. Geometry obtained thru inflation.
- Stiff strap loop used in periodic intervals
- Inflated to 1 atm (14.4 psi)

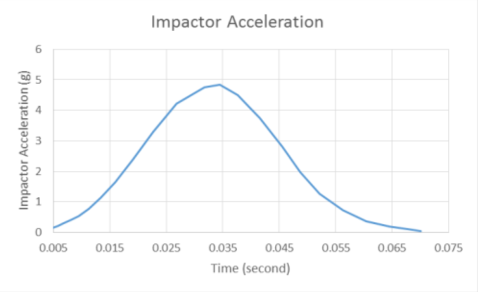
Radius of section, inner	$r_i := 60\,in = 5\cdot ft$
Radius of section, outer	$r_o := 61\,in$
Radius of toroid, inner	$R_i := 360\,in = 30\cdot ft$
Radius of toroid, median	$R_m := R_i + r_i = 420\,in$
Radius of toroid, outer	$R_o := R_i + 2\cdot r_i = 480\,in$
Height of crew member	$h := 70\,in = 5.833\,ft$
Radius of his CG	$R_{crew} := R_o - \frac{h}{2} = 445\,in$
To create 1g at crew member CG,	$\omega := \sqrt{\frac{1g}{R_{crew}}} = 0.931\cdot\frac{rad}{s}$
	$\omega = 8.895\,\frac{rev}{min}$

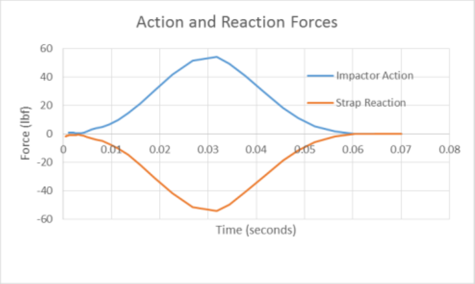
EXAMPLE 3 :   STRAP STRUCK WITH A MOVING MASS IMPACTOR

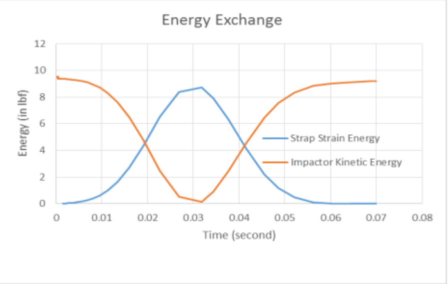










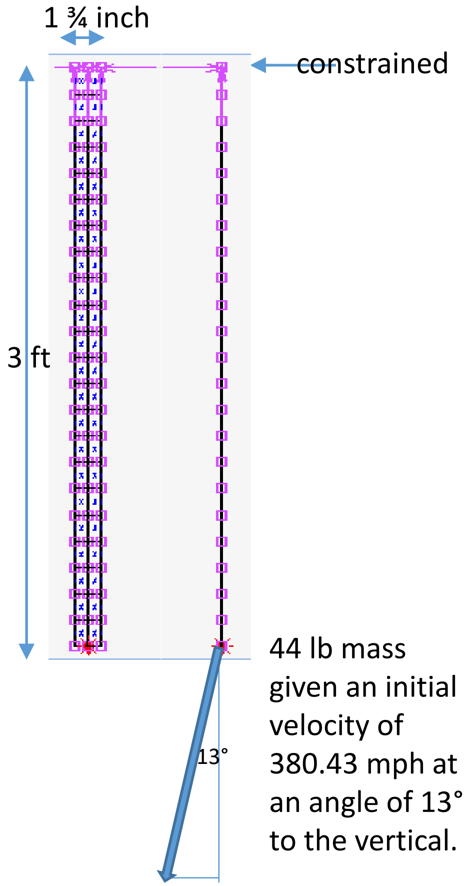


A 10 lb impactor travelling at 12.23 mph strikes a fabric strap.

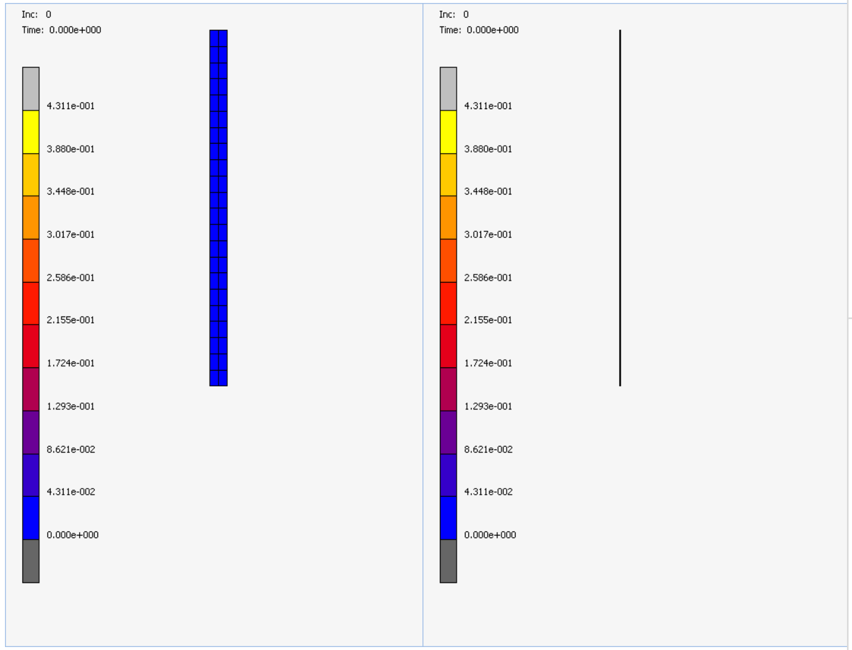
It deflects the fabric strap, transferring its kinetic energy into strain energy in the strap.

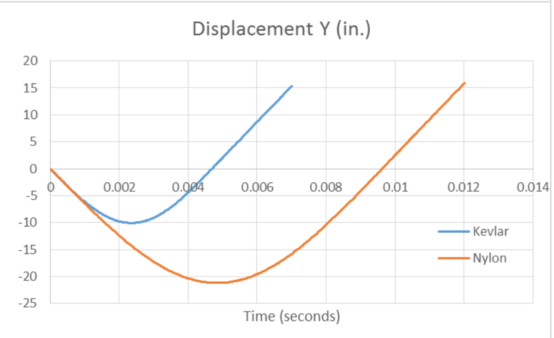
The strap being elastic, then returns the energy back to the impactor.

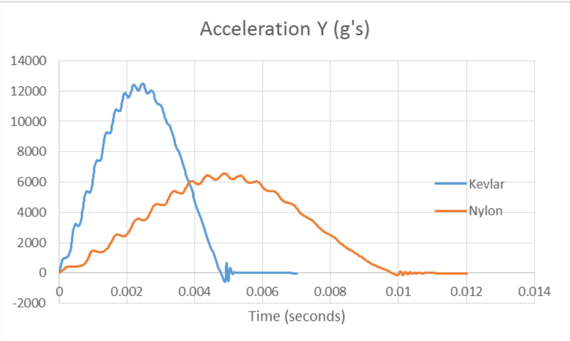
EXAMPLE 4 : KEVLAR VS NYLON STRAP USED TO STOP A MOVING MASS ATTACHED TO ONE END

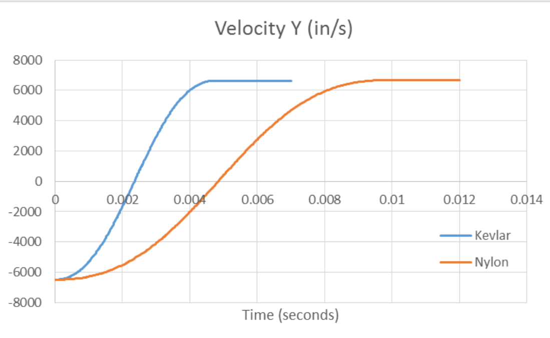


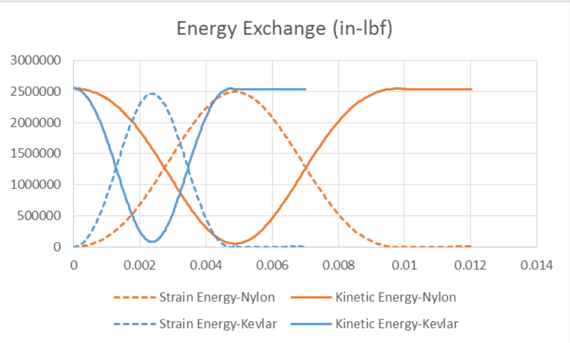
44 lb mass given an initial velocity of 380.43 mph at an angle of 13° to the vertical.





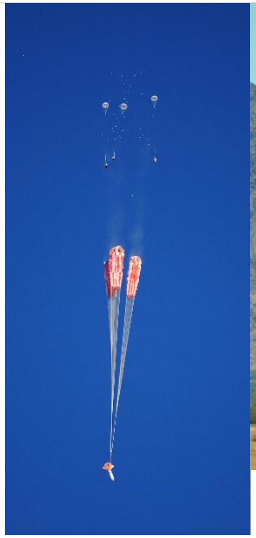






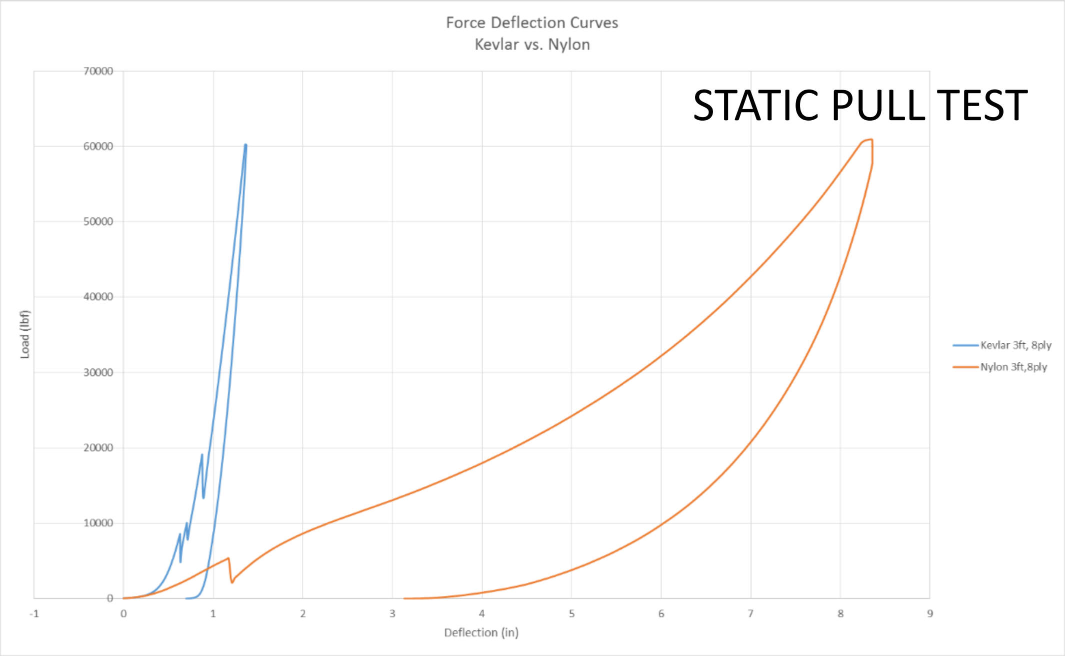
- This attempts to simulate a parachute drop test with a 3 foot Kevlar deployment strap vs. a 3 foot Nylon deployment strap jerking a 44 lb fitting off a shelf at >300 mph making an angle of 13° to the horizontal.

- Kevlar and Nylon fiber simply modeled as linear elastic for proof of concept purposes. Fiber young’s modulus, poisons ratio, fiber modeled as 0°, 90° fabric in a thin rubber matrix.
- Transient Dynamic Analysis successfully converges to a solution and shows expected differences between Nylon and Kevlar straps.
- In reality, some of the kinetic energy will be converted into heat – damping not modeled here, material is not linear, all will lead to lower accelerations.

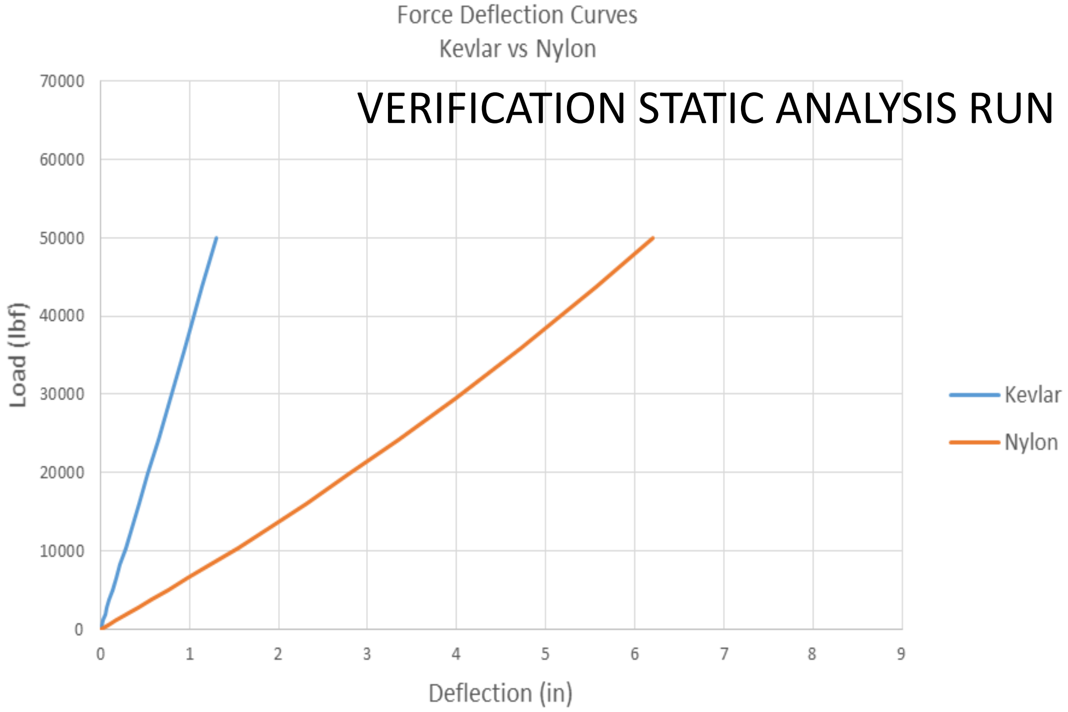


EXAMPLE 4 : KEVLAR VS NYLON STRAP USED TO STOP A MOVING MASS ATTACHED TO ONE END

The stiffness of the strap used in the dynamic analysis was obtained from a simple 2 point approximation from test data. A static analysis run was made before the dynamic analysis run to ensure that the strap stiffness modeled matched test. Though the straps consisted of 8 plys. The model was built using only 2 plys that were assigned the same stiffness as 8 plys – for modeling convenience.

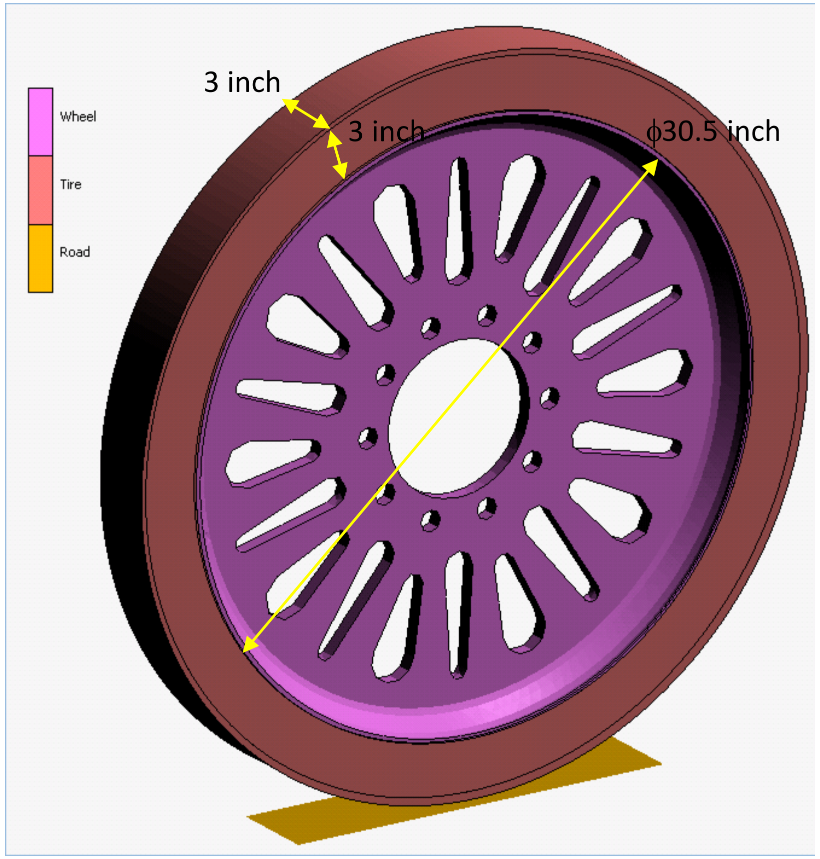

Force Deflection Curves
Kevlar vs. Nylon

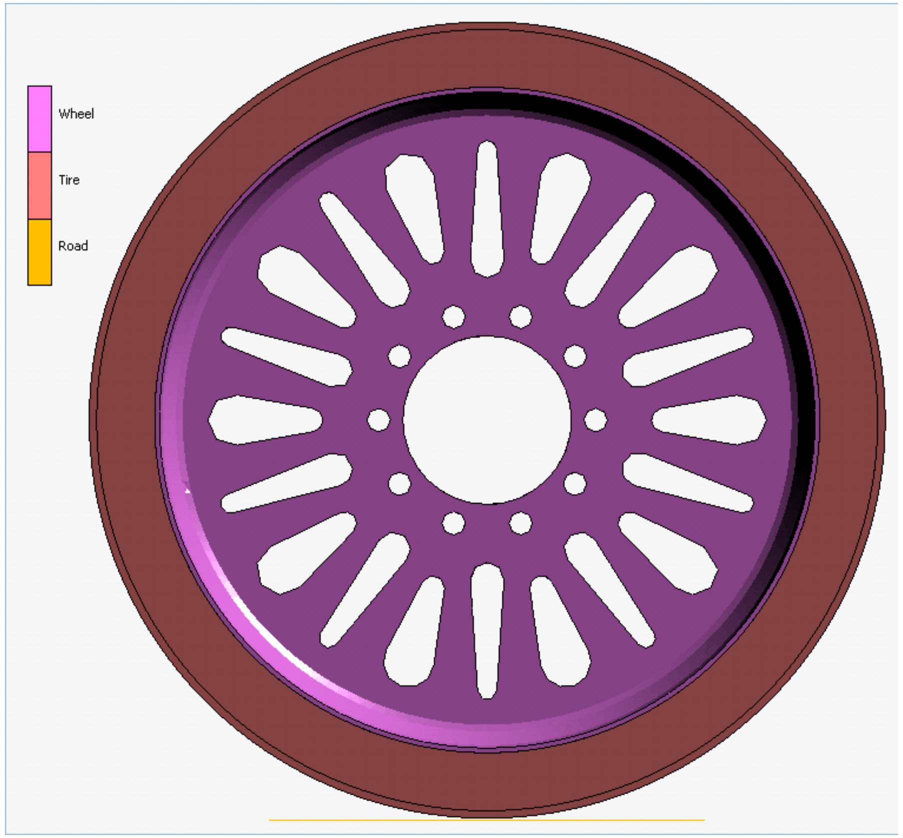
STATIC PULL TEST


Force Deflection Curves
Kevlar vs Nylon

VERIFICATION STATIC ANALYSIS RUN

EXAMPLE 5 : INFLATED EMERGENCY TRUCK WHEEL UNDER DYNAMIC LOADS



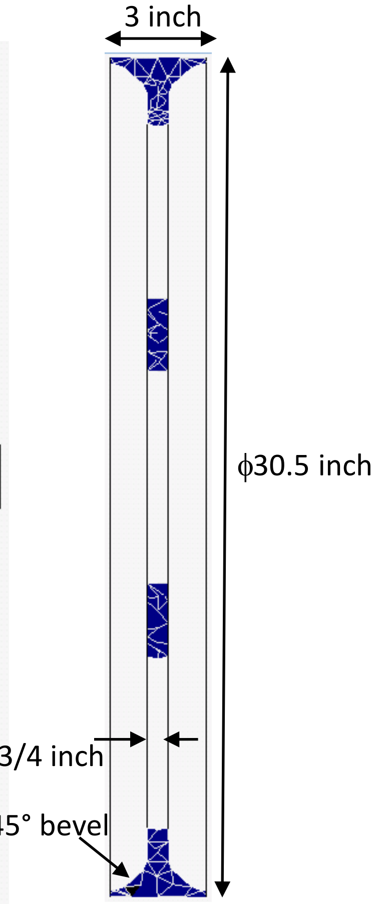


Inflatable temporary spare cord reinforced pneumatic spare tire inflated to pressure of 120 psi, and rated to handle a static vehicle weight of 9000 lbf per wheel. Tire and wheel dimensions as shown. Roughly translates to a tire spec of ST 77/100B30.5. Nylon cord reinforced sidewalls and steel belt reinforced tread modeled.

EXAMPLE 5 : INFLATED EMERGENCY TRUCK WHEEL UNDER DYNAMIC LOADS

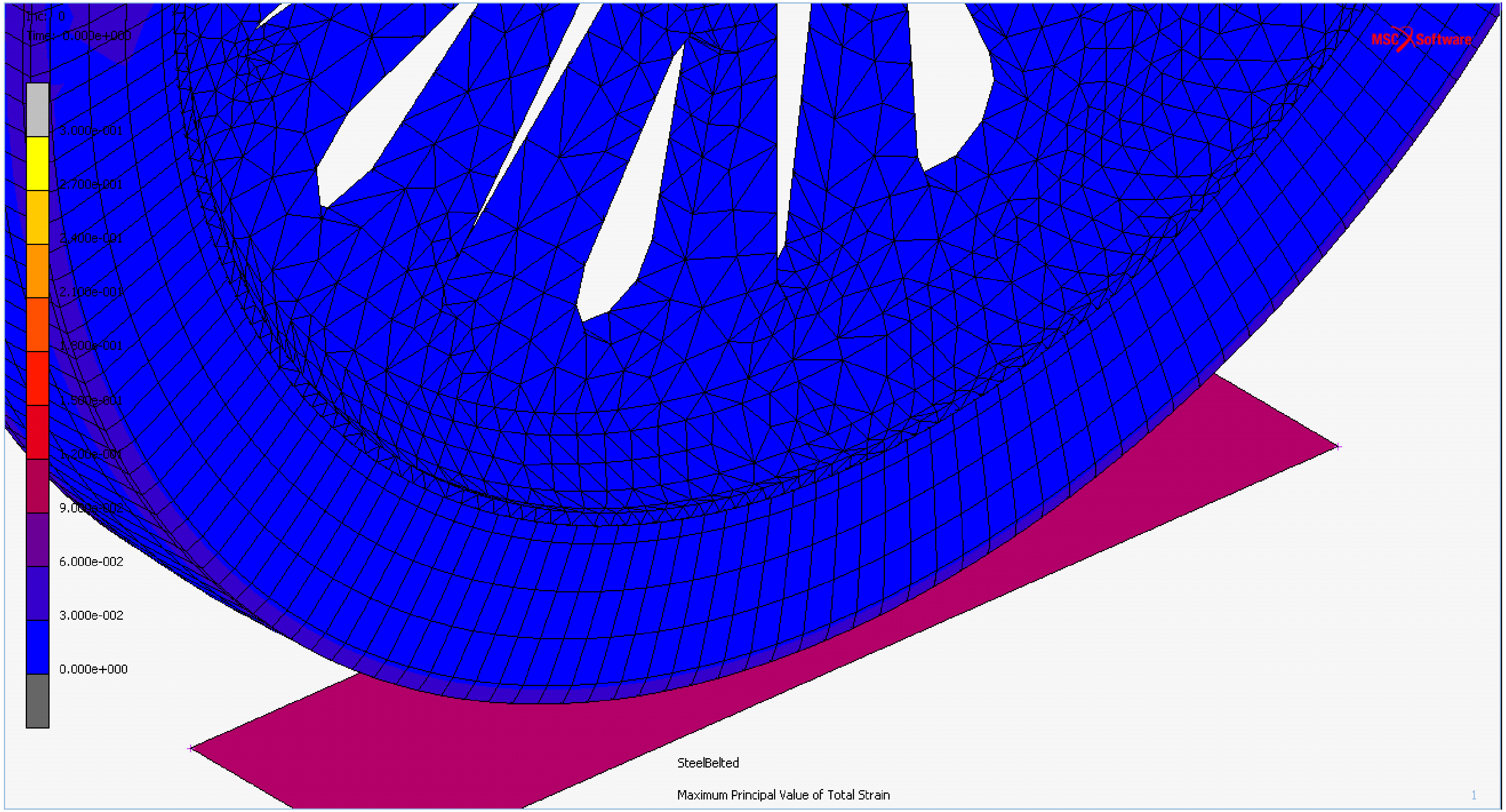
Estimated weights:
Wheel = 51.6 lb
Tire = 10.1 lb

Mounting studs: x10
M22x1.5 4 29/64"L
- M-3202, Grade 10.9



The A356 cast alloy wheel was optimized to have lower stresses and provide sufficient rigidity. The lightening hole pattern is important – a bad pattern can cause large local stresses. The 45° bevel is important, as this allows even distribution of stresses across the rim.

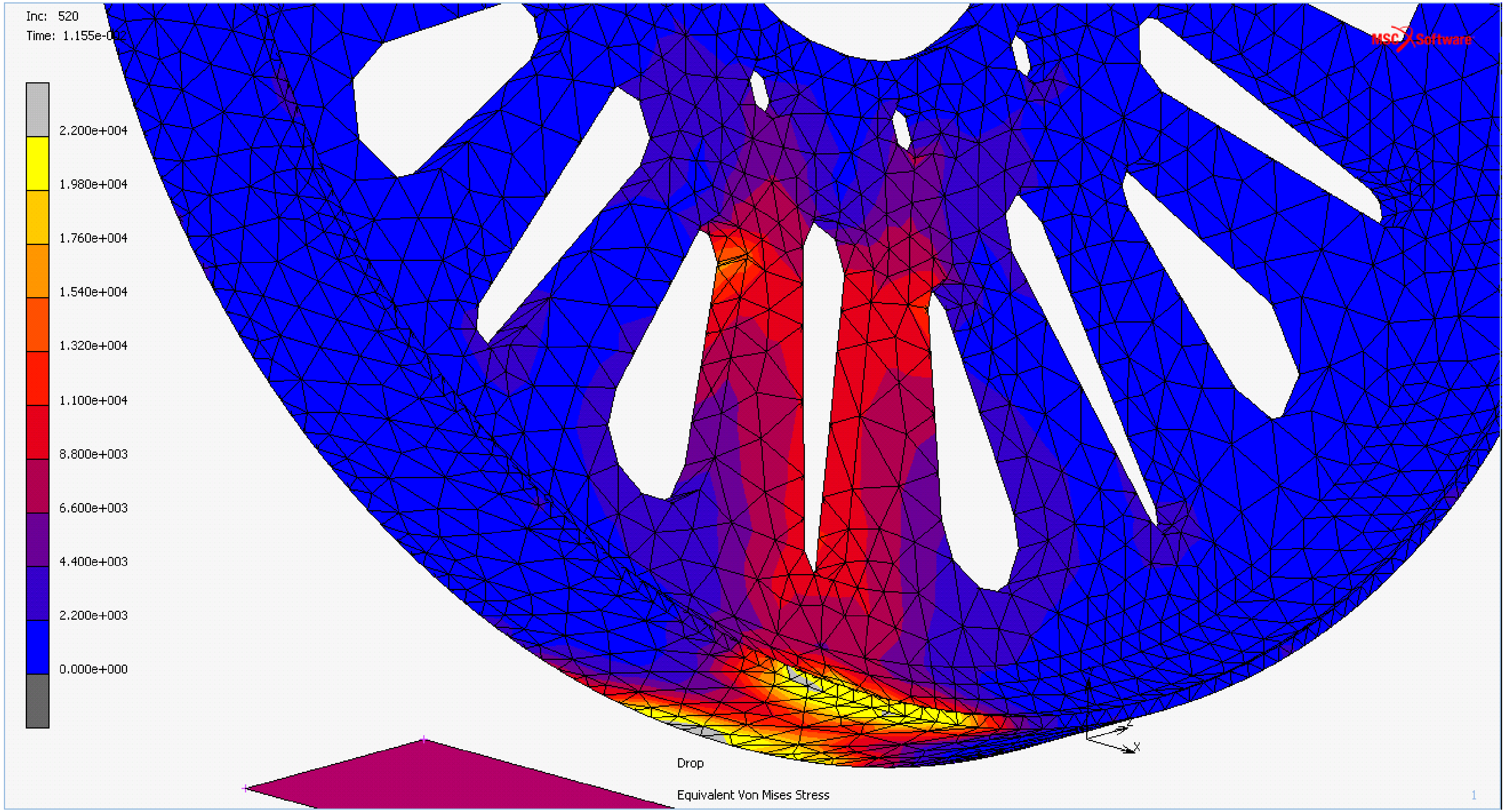
EXAMPLE 5 : INFLATED EMERGENCY TRUCK WHEEL UNDER DYNAMIC LOADS (SATOP PROJECT)



Load Case: 1 inch drop during engagement of emergency wheel with road

- Maximum principal strain in tire rubber is less than 30%. So, rubber is not excessively deformed

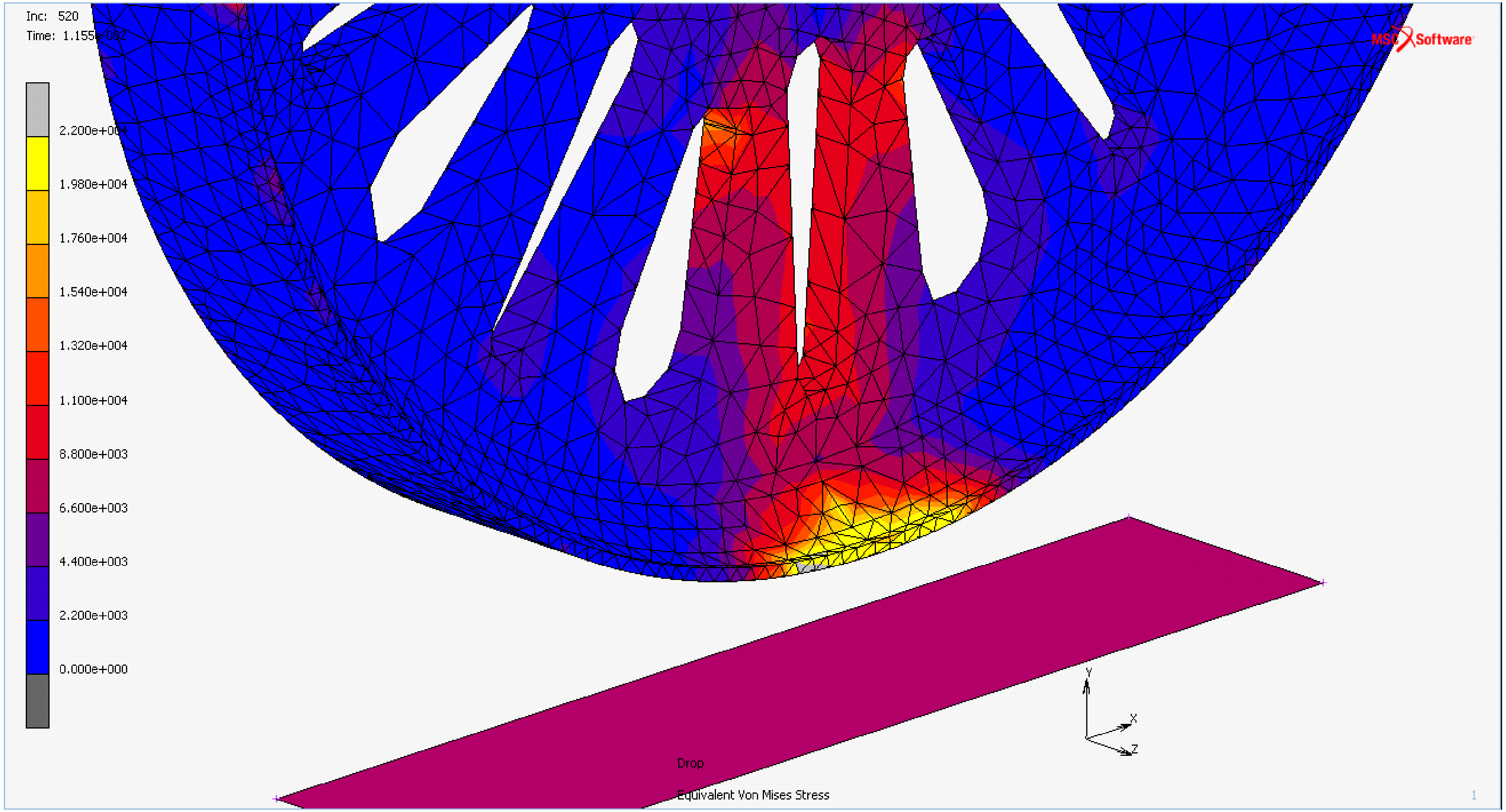
EXAMPLE 5 : INFLATED EMERGENCY TRUCK WHEEL UNDER DYNAMIC LOADS (SATOP PROJECT)



Load Case: 1 inch drop during engagement of emergency wheel with road

- The 45° fillet at wheel web-rim interface was optimized so that peak vonMises stress is just under yield strength.

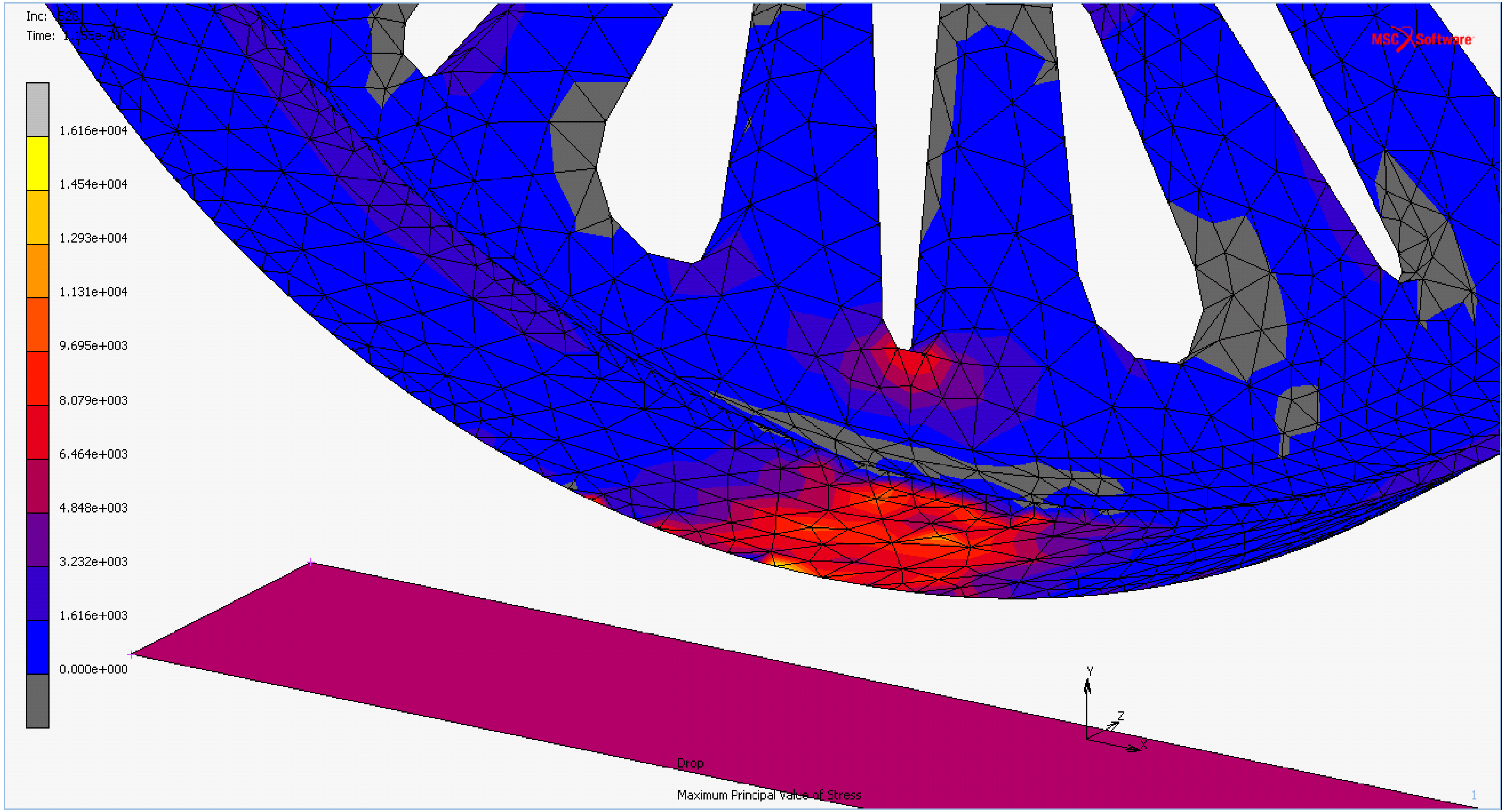
EXAMPLE 5 : INFLATED EMERGENCY TRUCK WHEEL UNDER DYNAMIC LOADS (SATOP PROJECT)



Load Case: 1 inch drop during engagement of emergency wheel with road

- The 45° fillet at wheel web-rim interface was optimized so that peak vonMises stress is just under yield strength.

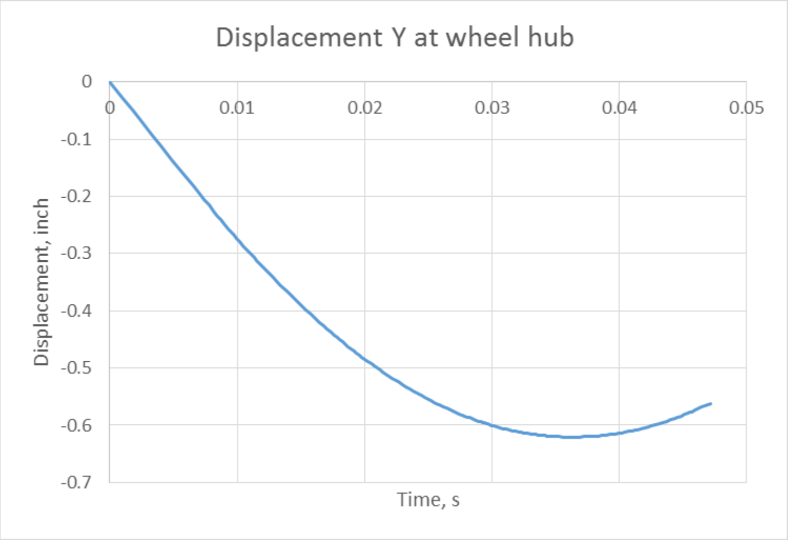
EXAMPLE 5 : INFLATED EMERGENCY TRUCK WHEEL UNDER DYNAMIC LOADS (SATOP PROJECT)

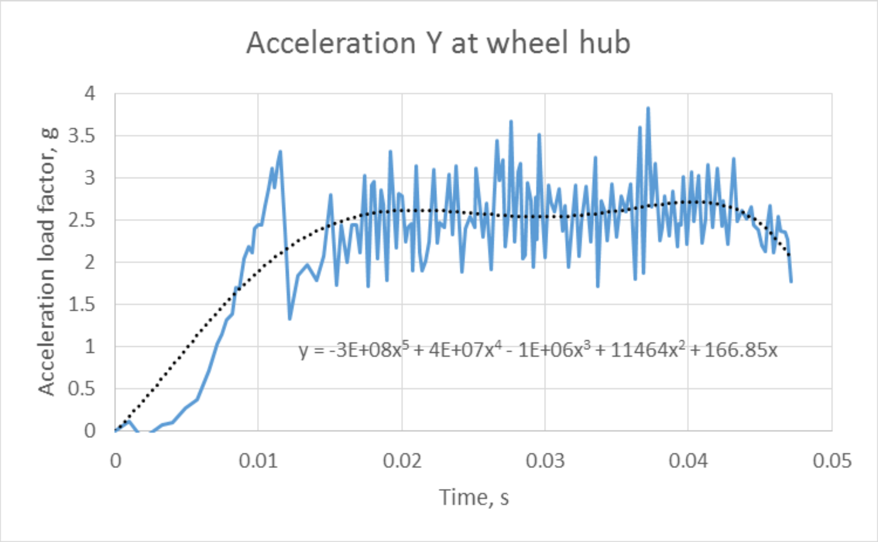


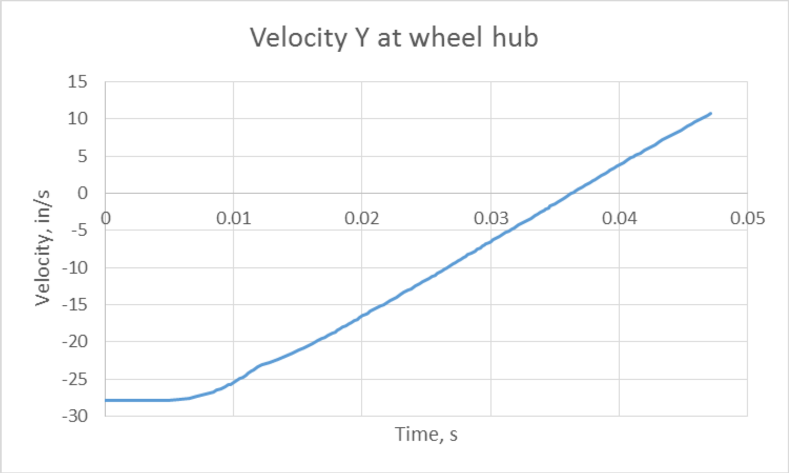
Load Case: 1 inch drop during engagement of emergency wheel with road

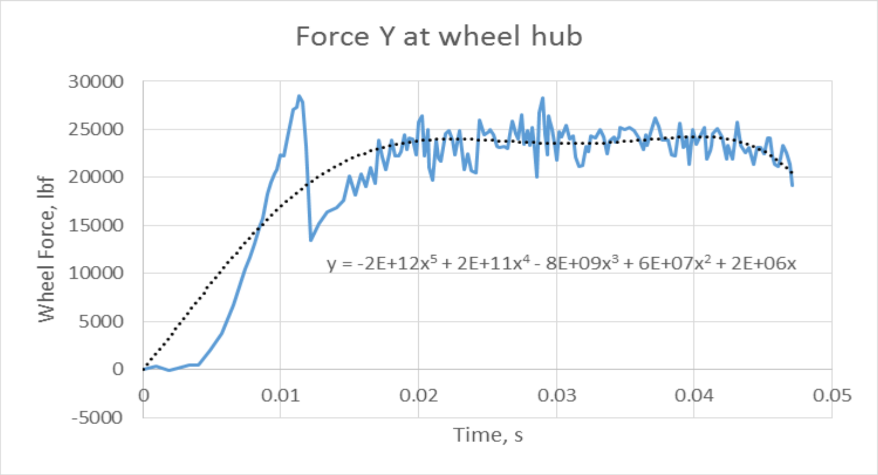
- Max principal stress is about half of the ultimate tensile strength of the material. This gives a safety factor of 2 and good fatigue life.

EXAMPLE 5 : INFLATED EMERGENCY TRUCK WHEEL UNDER DYNAMIC LOADS (SATOP PROJECT)







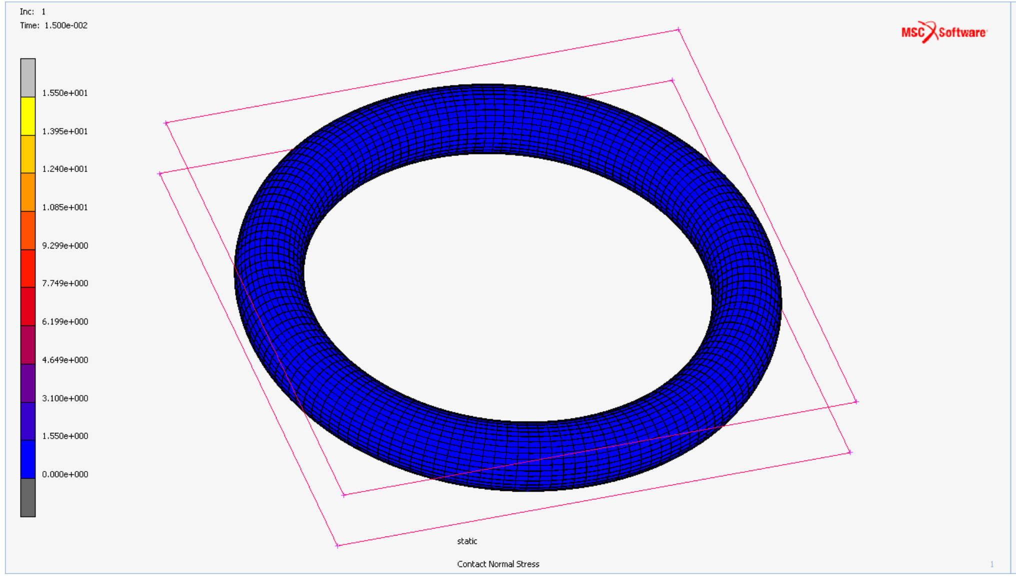


Peaks wheel hub force is about 28600 lbf, Drop impact load factor is about 3 g.
These are substantially less than the solid rubber tire models where (7 – 9 g were obtained)

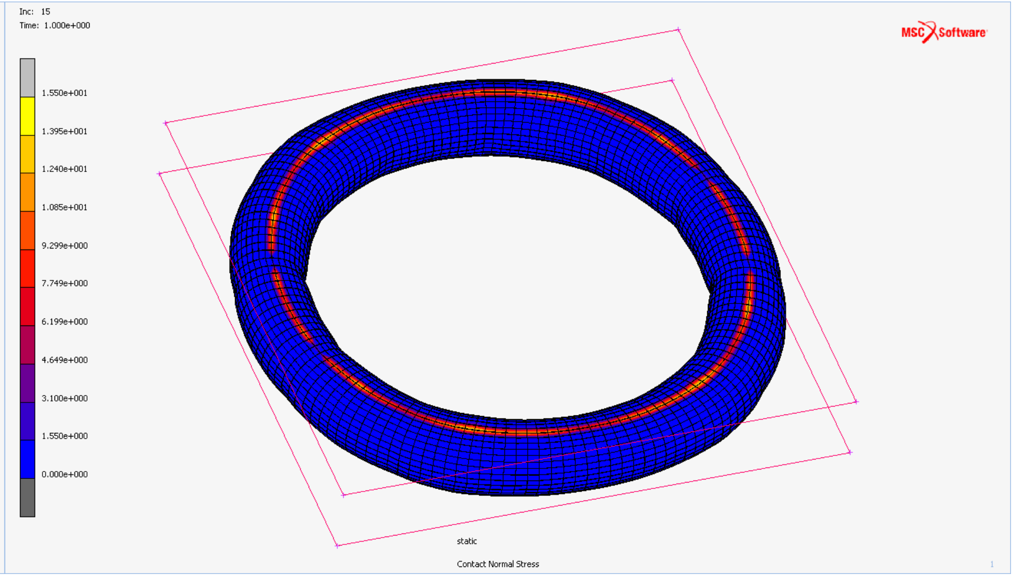
EXAMPLE 6 : MISALIGNMENT TOLERANT INFLATABLE SEAL

Commercial Inflatable door seal: https://youtu.be/8emG1LYboHo, https://youtu.be/jjFZfDb1wBs

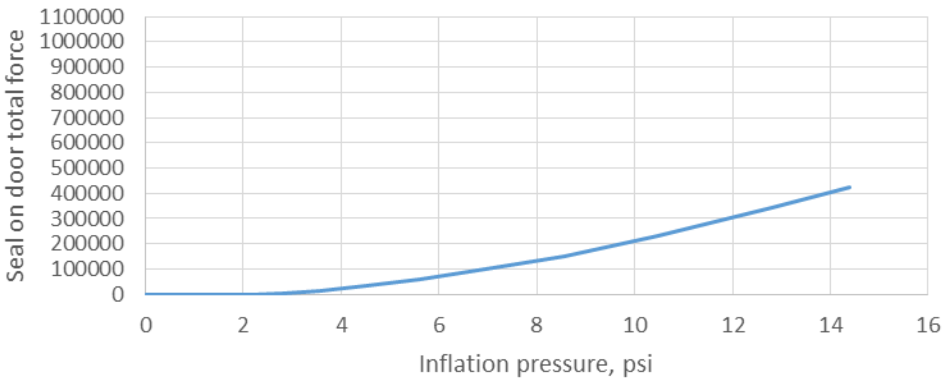
Step 1: Inflation of the seal between two surfaces



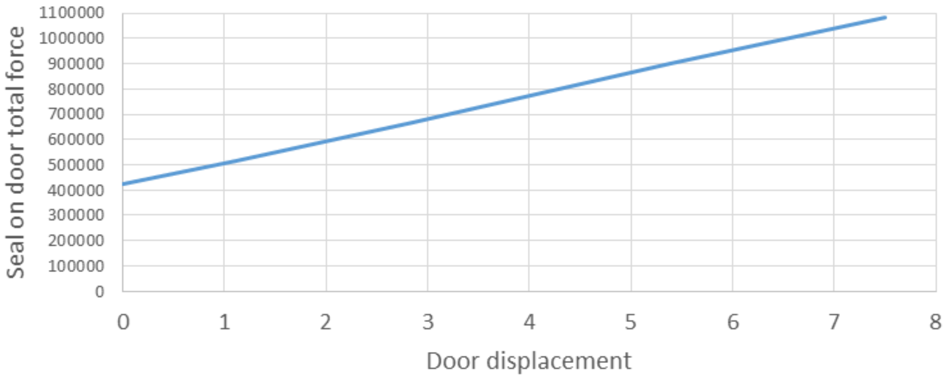
Step 2: The two surfaces then come closer together to compete the tight fit



Seal on door force vs. inflation pressure



Seal on door force vs. door displacement



These two steps may be interchanged to get similar result

Successfully demonstrated use of MARC to model such problems.

Seals may be made of rubber or composite fabric.

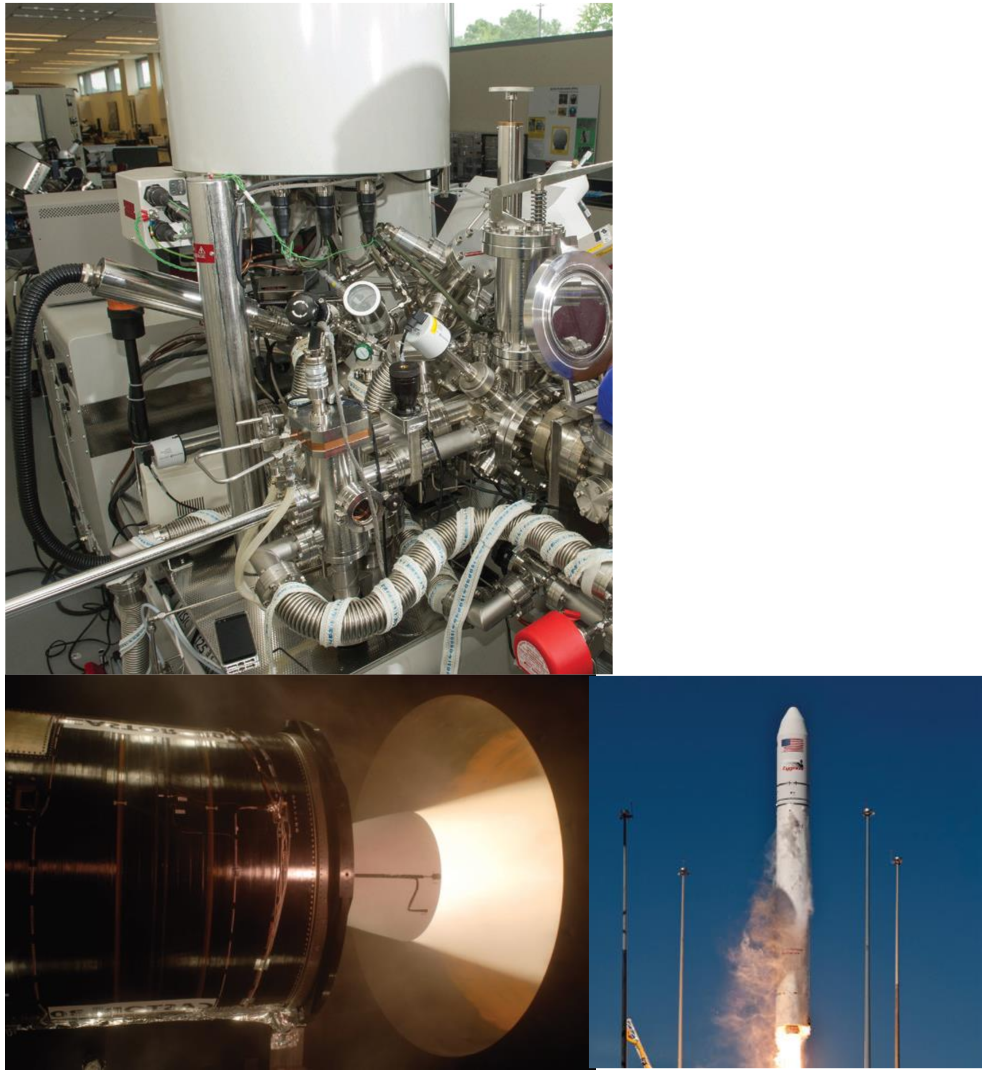
May be used to develop seals between spacecraft modules, Lunar/Martian habitat modules, rover door seals, etc.

SUMMARY OF FINDINGS:

1. The REBAR feature in MSC Marc may be exploited to successfully model fabric weave.
2. The REBAR elements will need to be constructed superimposed on matrix elements.
3. The model so built has been shown to work well for static and transient dynamic problems.
4. Test and analysis correlation for static pull loads have been obtained
5. The restart functionality in Marc may be used to set up a multi step problem (example: step 1 - inflate, step 2 - preload, step 3 - impact load).
6. Softgood – metallic structure integrated models have been shown to work well (tire-wheel).
7. Marc handles geometric nonlinearities, material nonlinearities and contacts between bodies very well

CONCLUSION:

1. The technology developed demonstrated proof of concept and applied it successfully to NASA and non NASA problems.
2. The technology may be applied to paying projects, with care being taken to ensure that preliminary simulation results do match equivalent tests.
3. The results are as good as the fidelity of the math model.



Marc Simulations at NASA Johnson Space Center

Part 2: Fracture Simulations

MSC Software Nonlinear Users Meeting
April 17, 2017
Ann Arbor, Michigan, USA

Satish Reddy
Chief Engineer, Structural Analysis

Jacobs Technology

View in Slide Show Mode

OVERVIEW

Currently, the NASGRO suite of programs is used to analyze fatigue crack growth and fracture, perform structural life assessments and analyze fatigue crack initiation. It also has a very strong experimentally determined materials database consisting of crack growth rate and fracture toughness data that is necessary for such analyses.

NASGRO was originally developed at NASA Johnson Space Center to perform fracture control analysis on NASA space systems. Today, Southwest Research Institute (SwRI) under a space act agreement manages an industry consortium (Airbus, Boeing, Bombarier, Embraer, Lockheed Martin, ULA, SpaceX, Sikorsky, Mitsubishi Aircraft, Israel Aerospace, NASA, ESA,…) of users. The core team of developers of this software include several of the authors of this proposal. NASGRO is considered the global industry standard used for fracture control.

The goal of this work was to evaluate the state of the art in fracture prediction using MSC Software Corporation’s Marc and benchmark its advertised abilities in fracture prediction against the robust NASGRO database and experience at JSC. Marc provides a highly visual pre and post processor combined with a very powerful analysis solver that includes advanced contact, remeshing, ductile damage, composite failure and delamination, and crack propagation features.

BENEFITS

Currently NASGRO doesn’t automatically tell the analyst where a crack will initiate. Marc appears to do this. And if it does it correctly, that would be a tremendous value addition to projects and programs.

Designers and analysts may be able to engineer components such that a potential crack would propagate in a benign direction or be effectively arrested thus avoiding catastrophic failures.

The ability to reliably visualize crack initiation and propagation in complex geometries is the next frontier for stress and fracture analysis. We will not be just communicating stress plots with our customers, but will be showing them how and where cracks will initiate and propagate.

Increase the reliability of structures that will be developed for the human exploration of Mars, where it will not be possible to send spare parts if they were to fail.

OUTCOME

- Marc Analysis Simulation results show correlation with the NASGRO standard indicating feasibility for the cases investigated.

- Simulations clearly show expected crack propagation direction for a variety of cases.

SOME NASGRO CRACK CASES

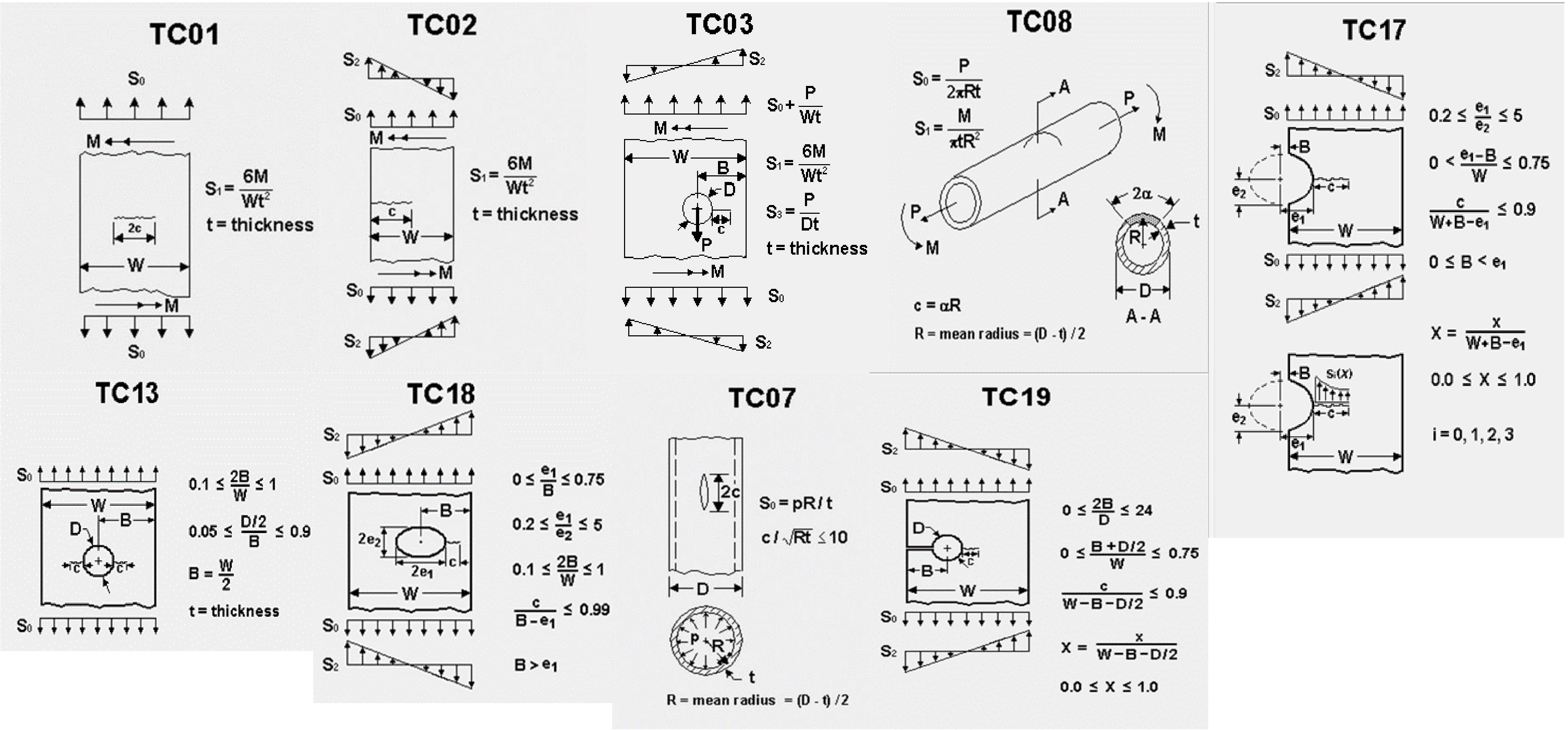
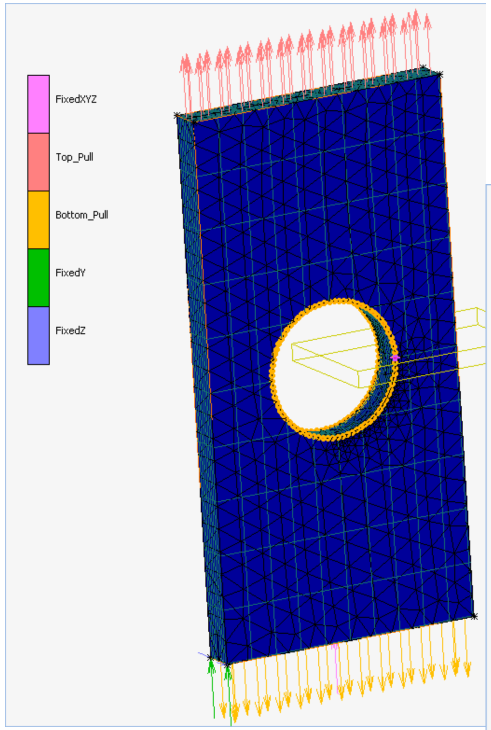
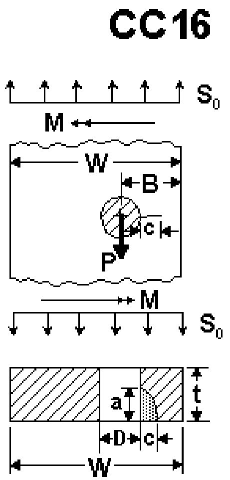


PLATE WITH A HOLE UNDER TENSILE CYCLIC LOADING



MARC analysis model



NASGRO model

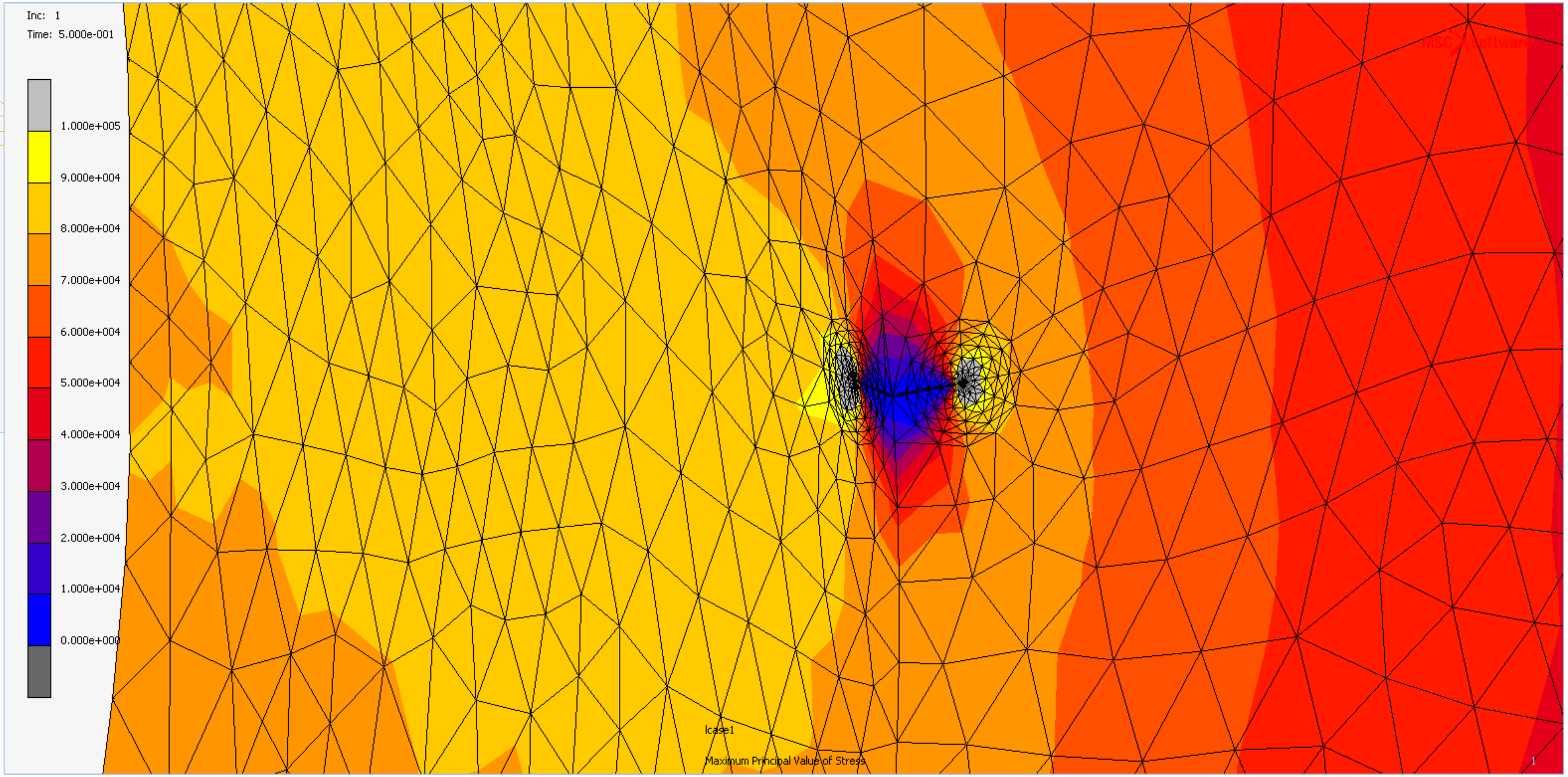
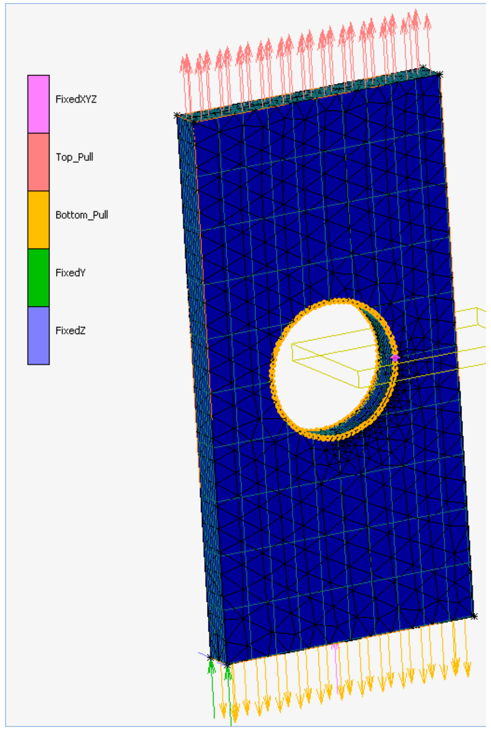
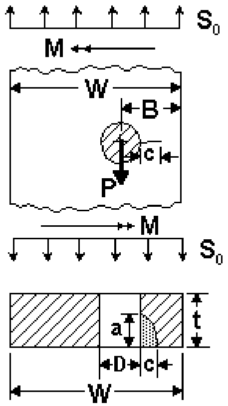


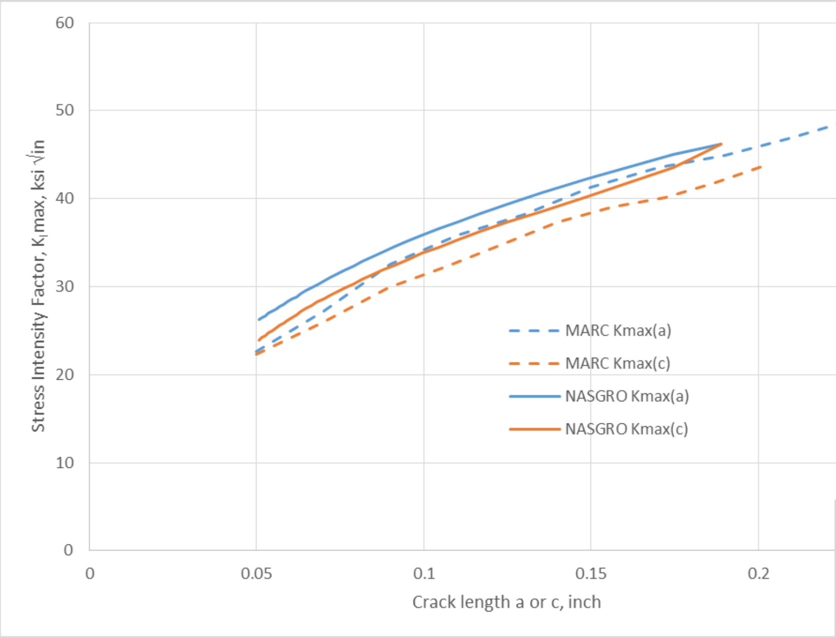
PLATE WITH A HOLE UNDER TENSILE CYCLIC LOADING



MARC analysis model

CC16





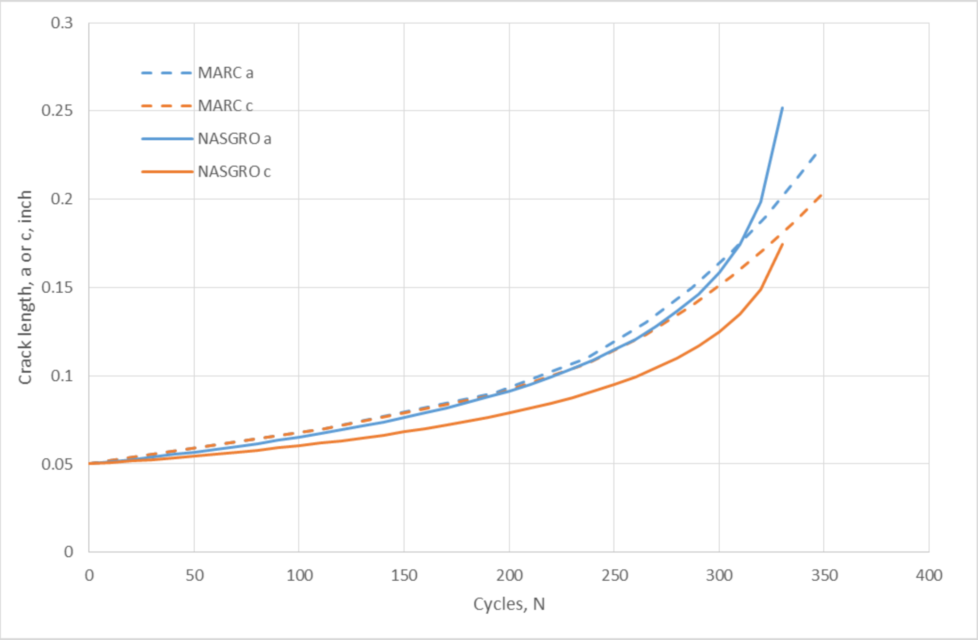
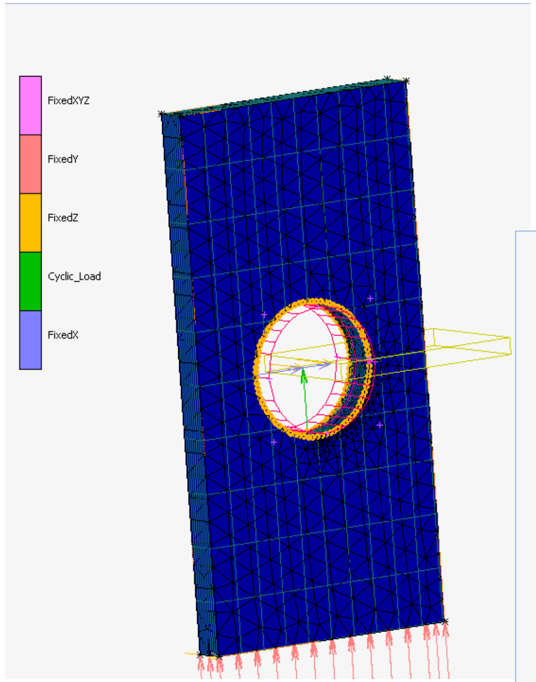
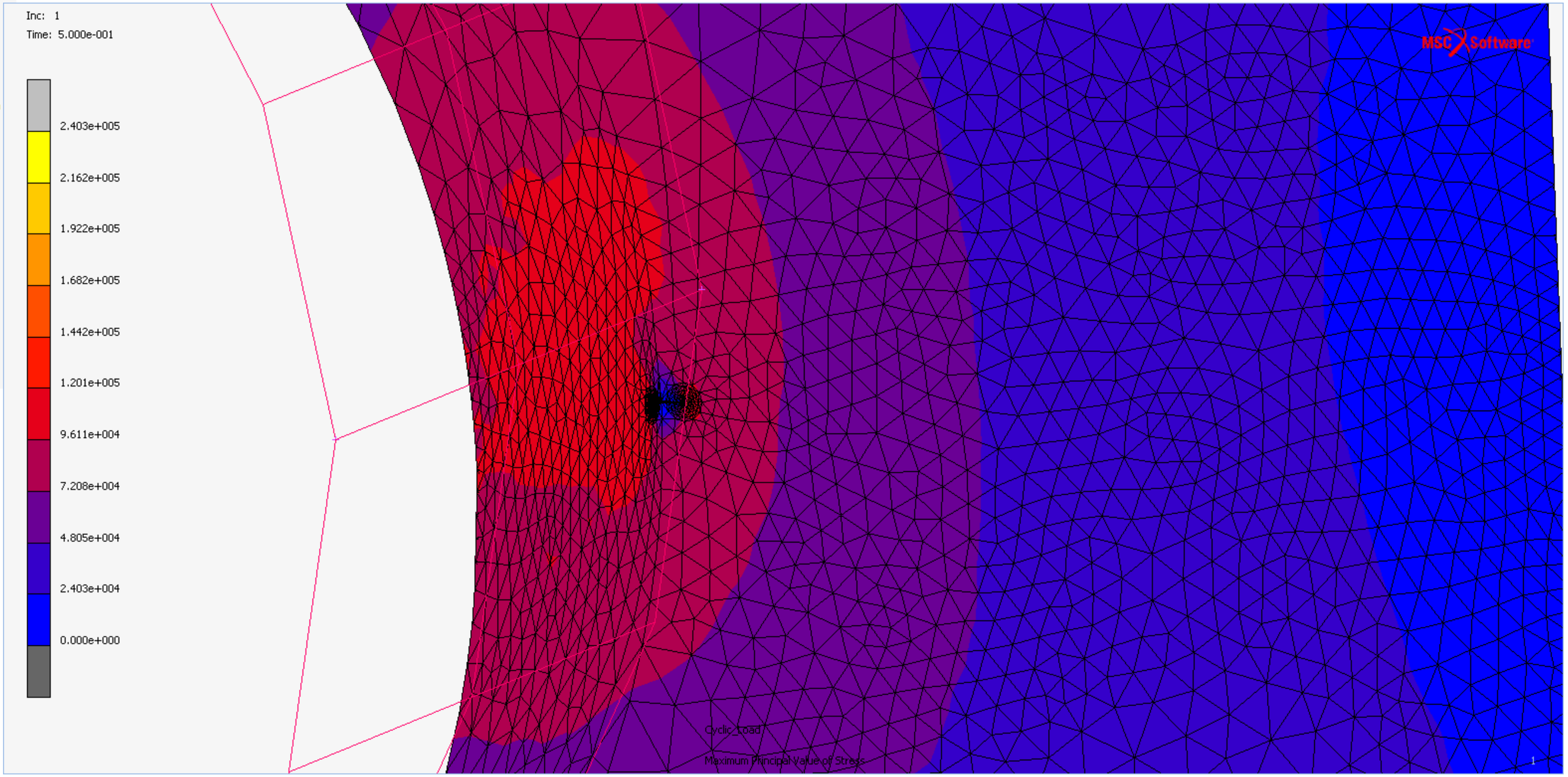
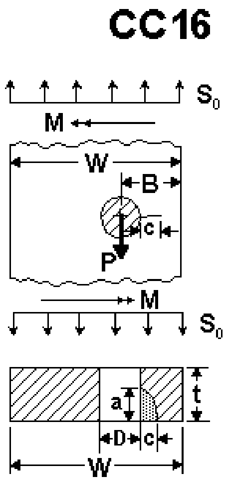


PLATE WITH A HOLE UNDER CYCLIC LOADING FROM A PIN



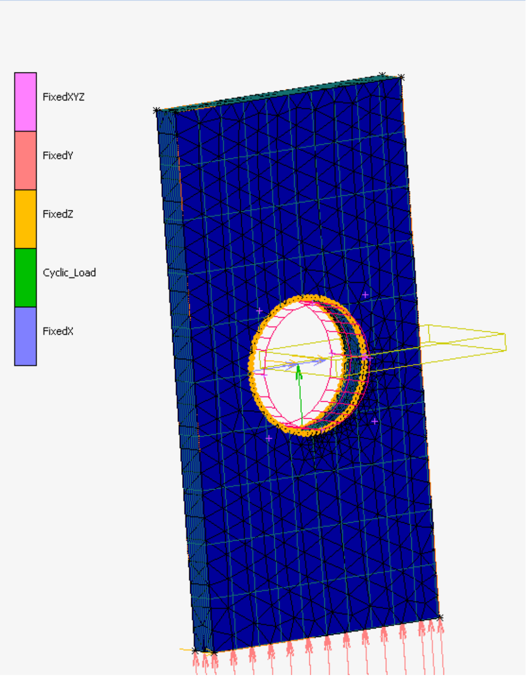
MARC analysis model





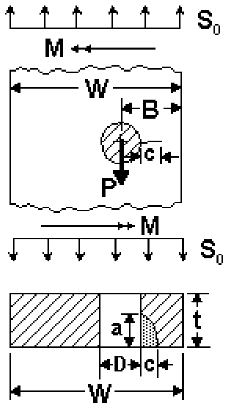
NASGRO model

PLATE WITH A HOLE UNDER CYCLIC LOADING FROM A PIN

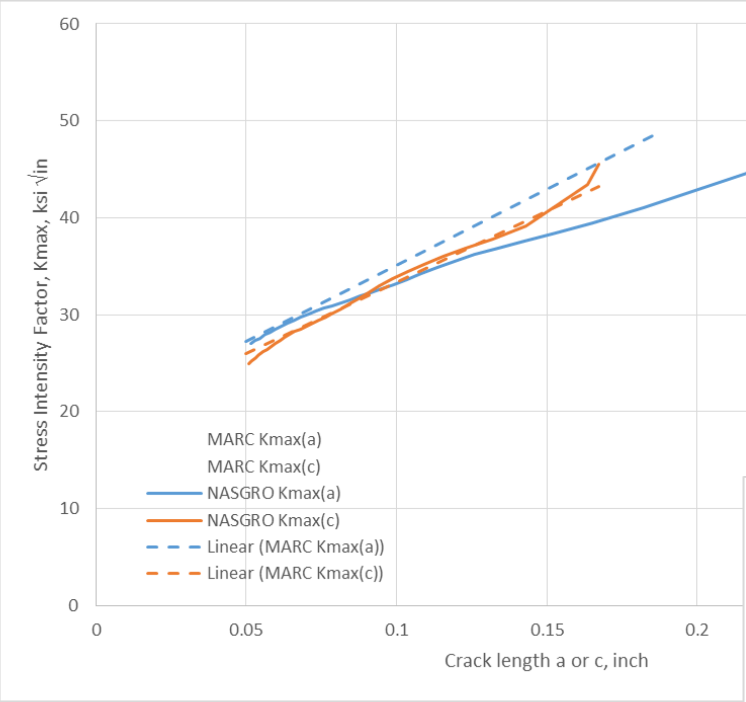


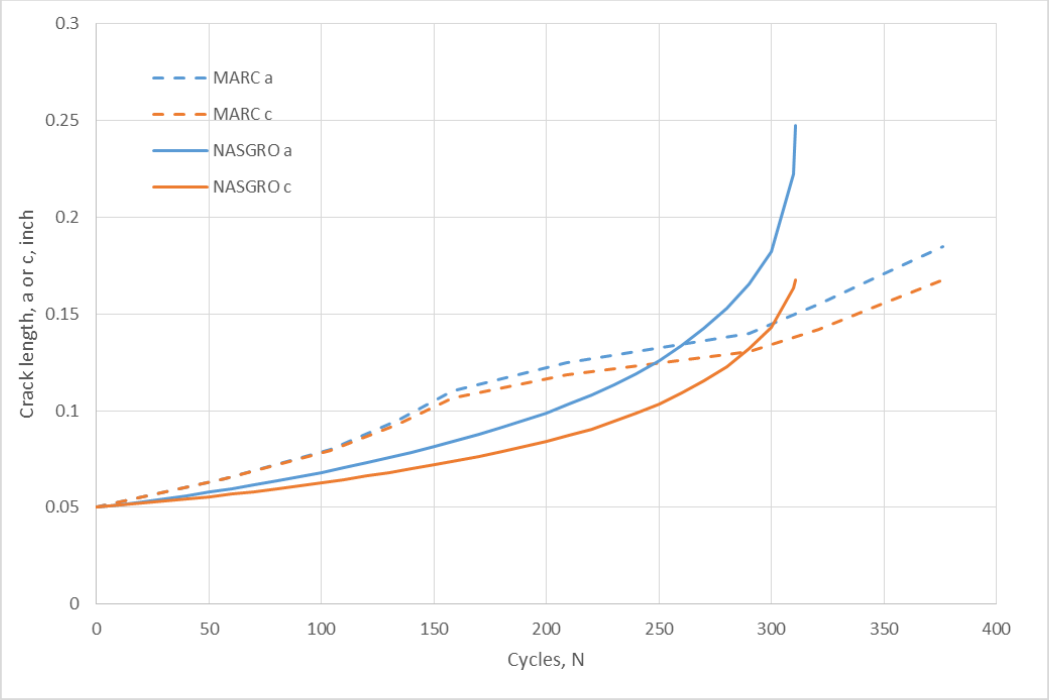
MARC analysis model

CC16

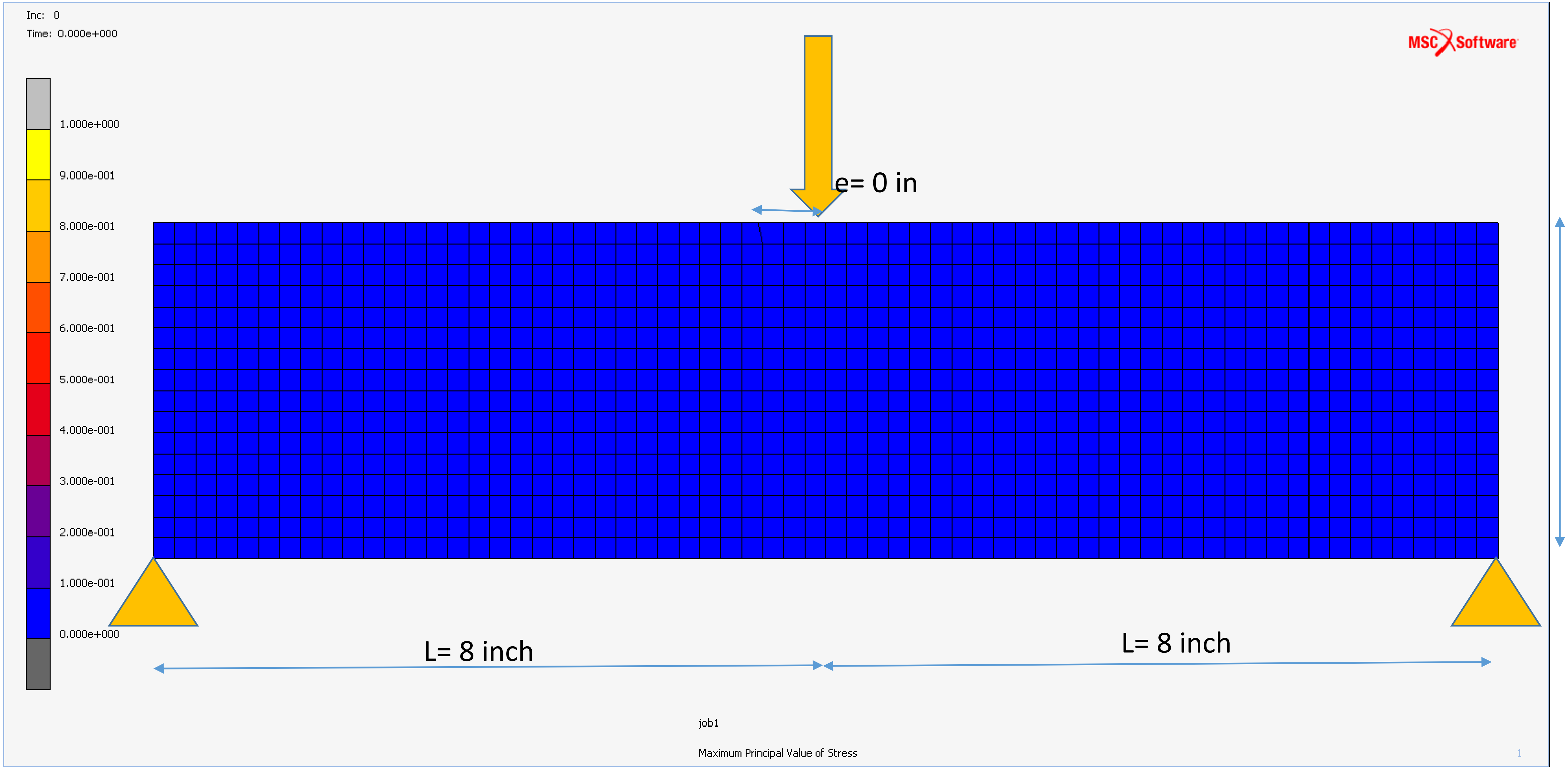


NASGRO model





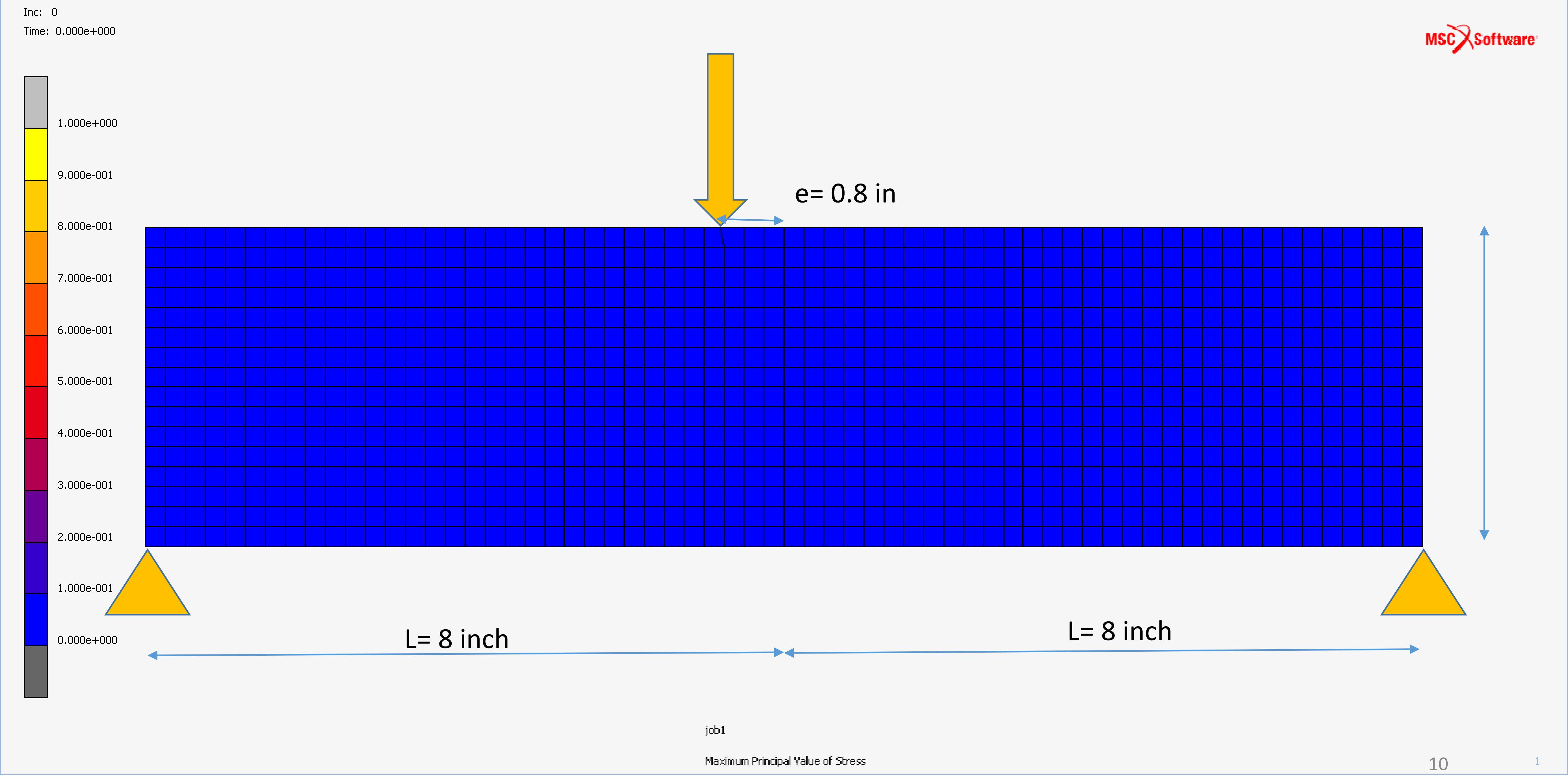
3 Point Bend Specimen, e/L = 0



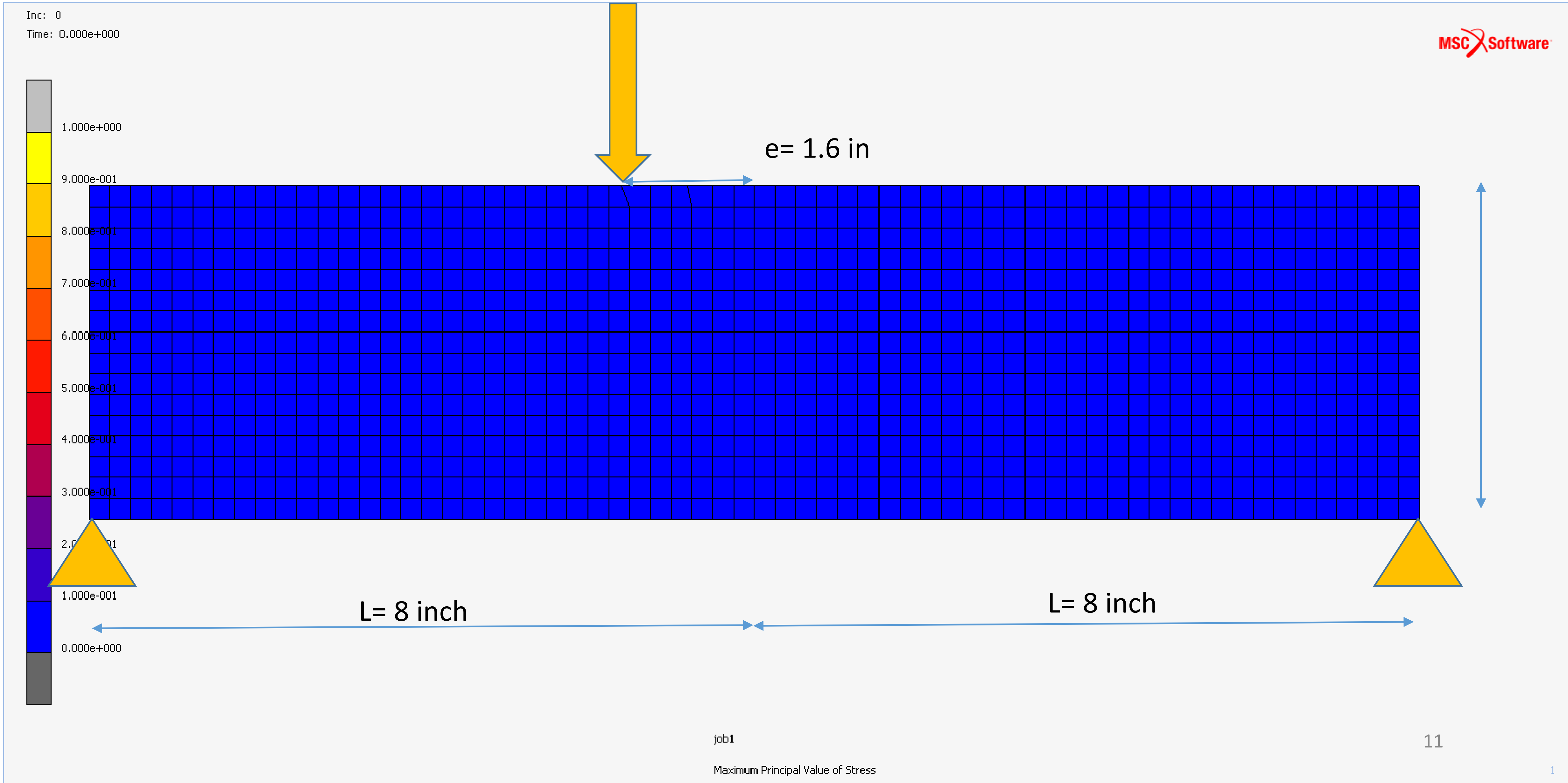
Inc: 0
Time: 0.000e+000

Maximum Principal Value of Stress

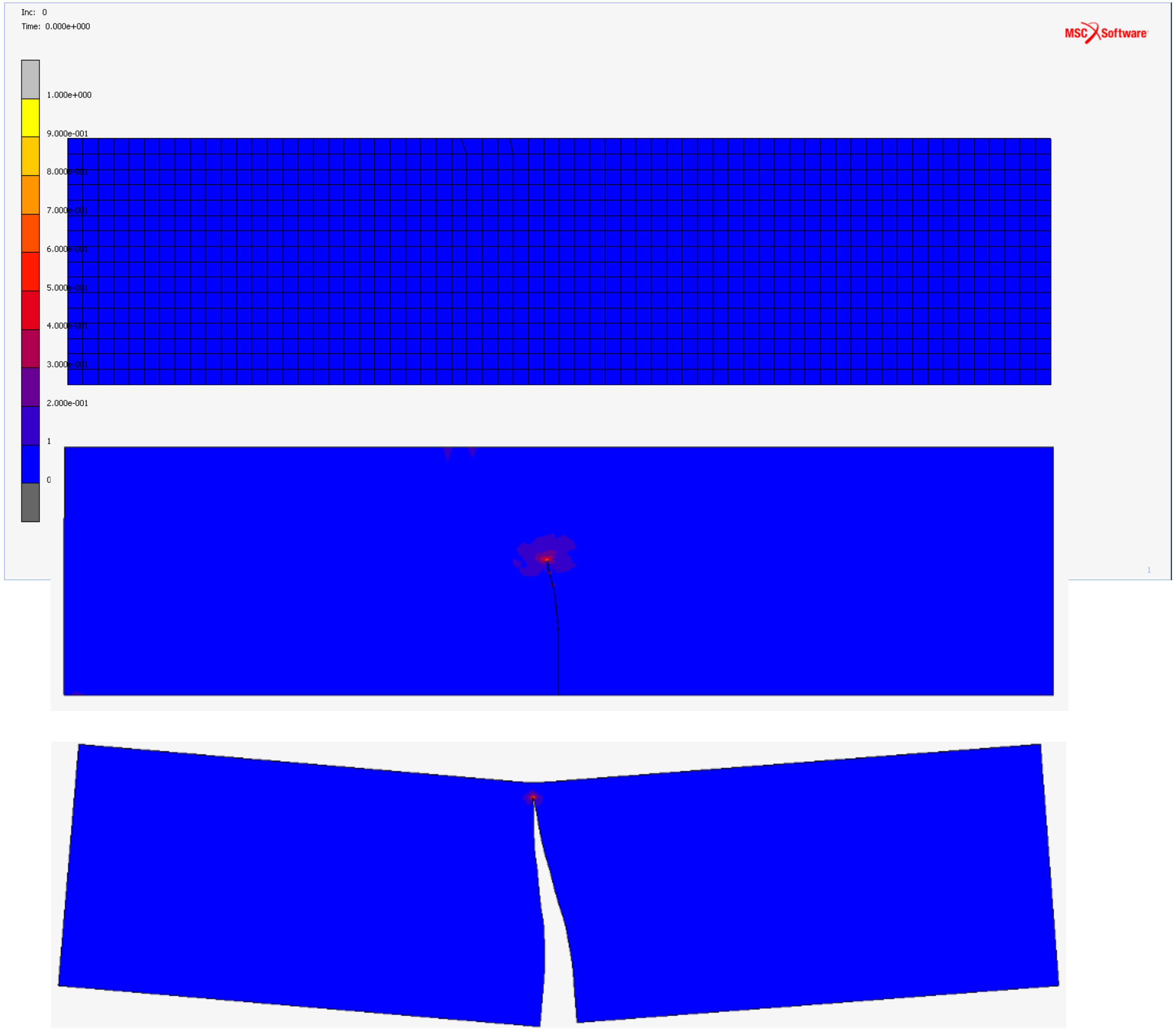
3 Point Bend Specimen, e/L = 0.1



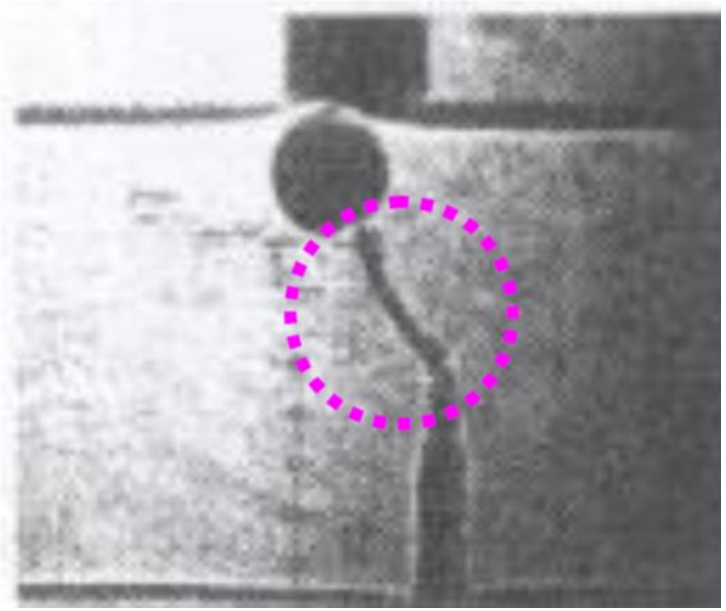
3 Point Bend Specimen, e/L = 0.2



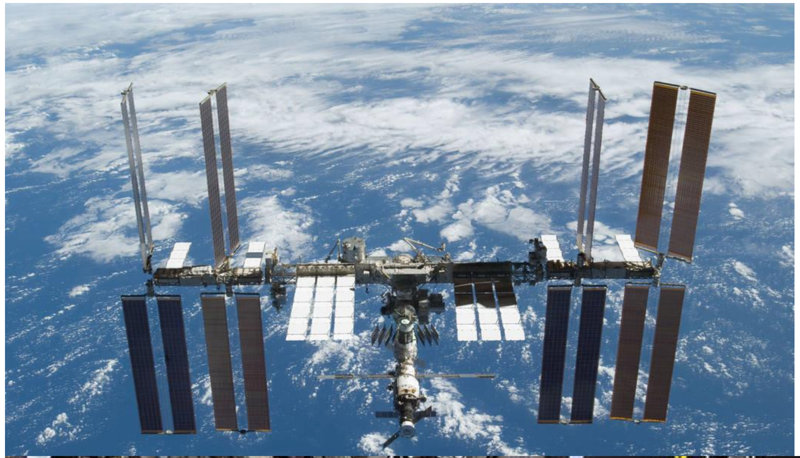
3 POINT BEND FRACTURE



Fracture of a flawed beam in 3 point bending. Crack growth path and eventual failure successfully predicted



National Aeronautics and
Space Administration



Marc Simulations at NASA Johnson Space Center

Part 3: Gasket Sealing

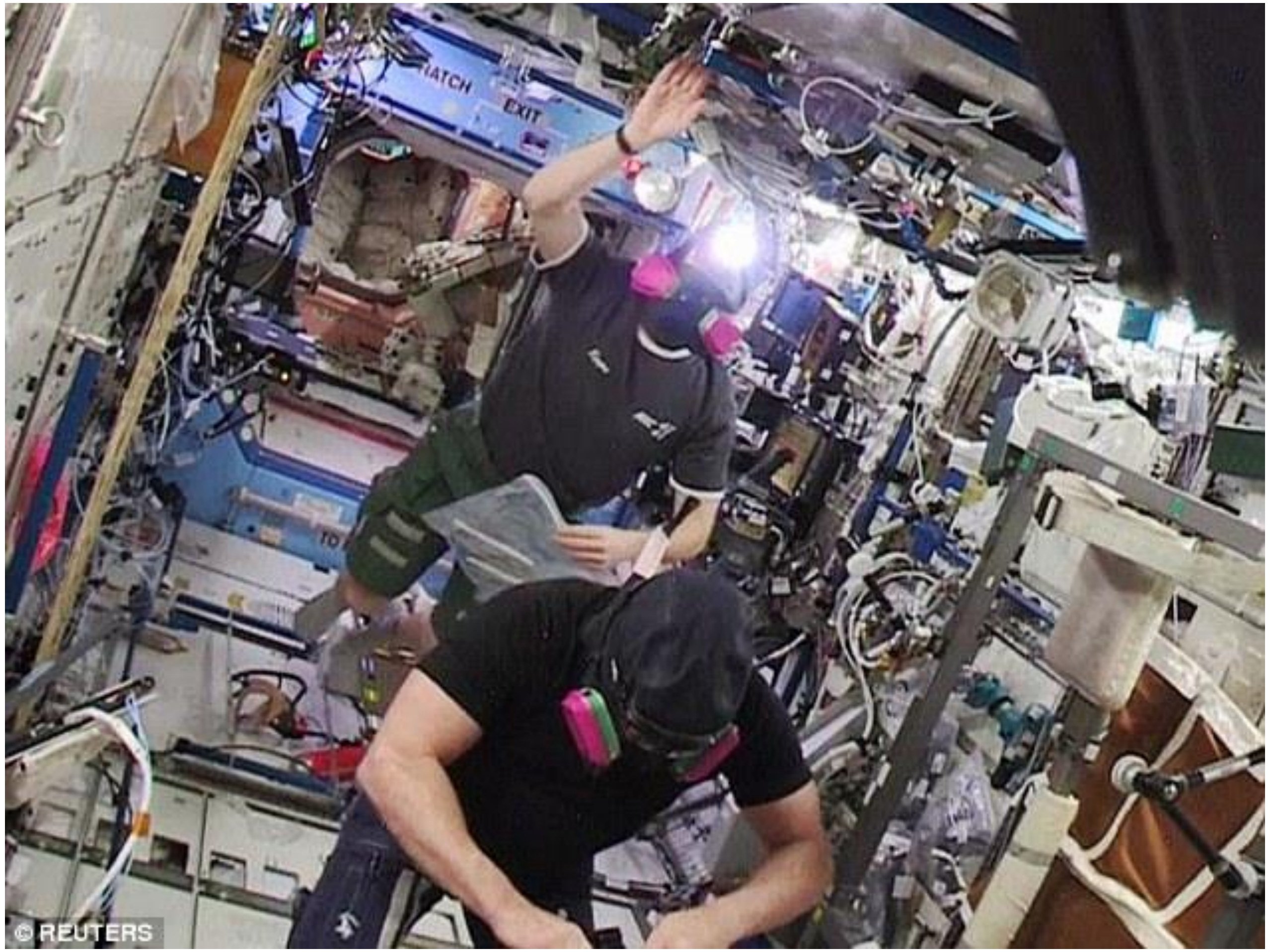
MSC Software Nonlinear Users Meeting
April 17, 2017
Ann Arbor, Michigan, USA

Satish Reddy
Chief Engineer, Structural Analysis

Jacobs Technology

View in Slide Show Mode

EXAMPLE: INTERNATIONAL SPACE STATION EMERGENCY MASK

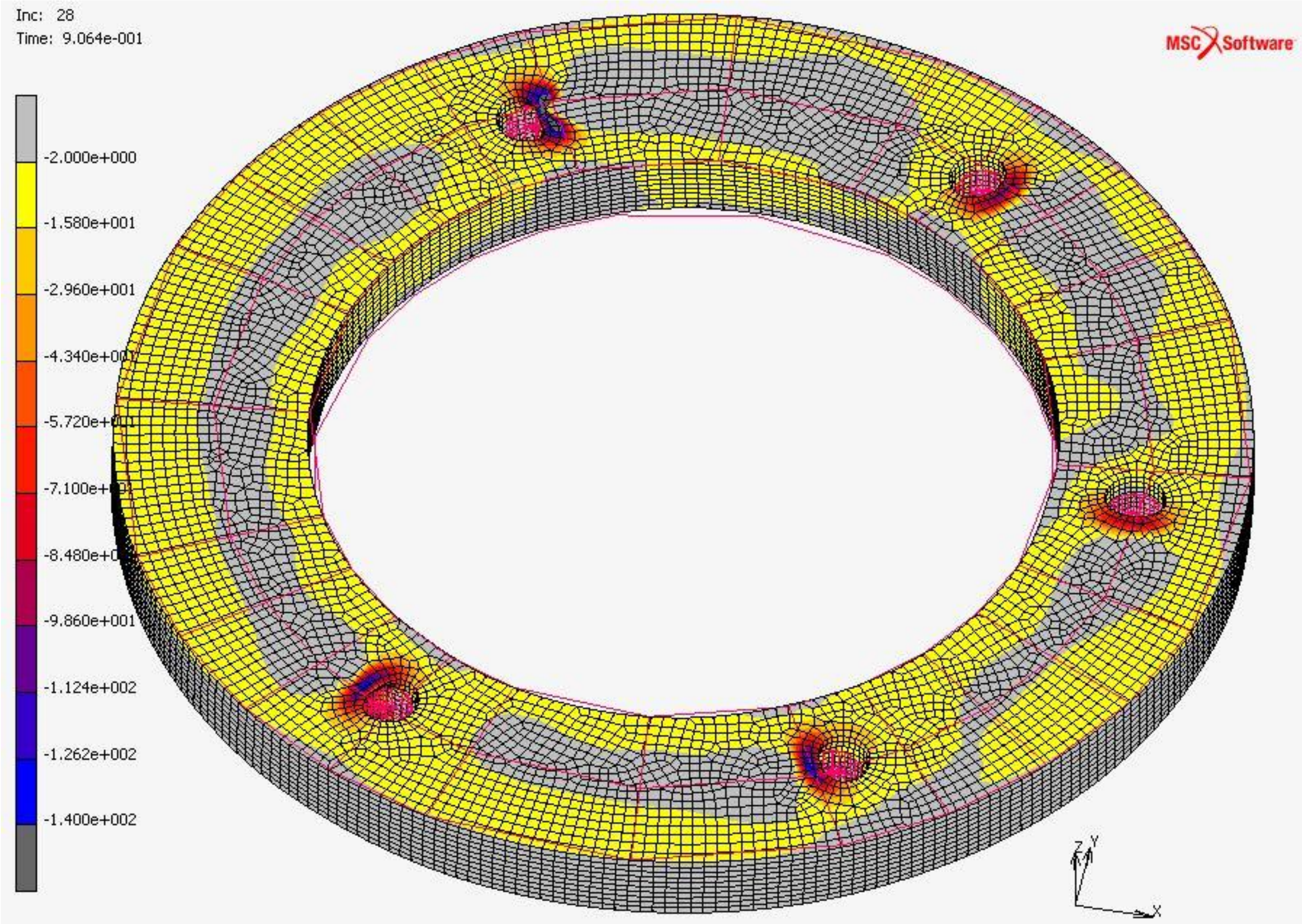


Members of expedition 42 (pictured) sample the air in the US segment after initially evacuating to the Russian side of the ISS, due to warnings of a possible ammonia leak

Emergency Mask with Fire Cartridges

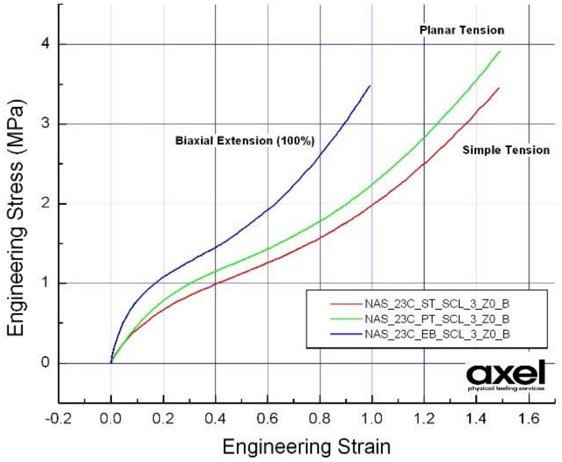
Emergency Mask without Cartridges

EPDM SEAL DESIGNED TO ENSURE EFFECTIVE SEALING

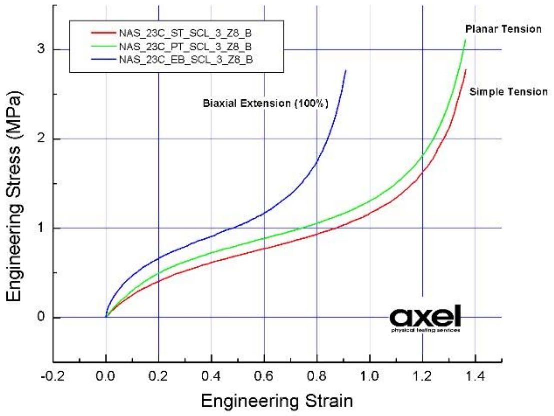


The EPDM Hyperelastic properties were characterized by testing done at Axel Products, Inc, Ann Arbor, Michigan.

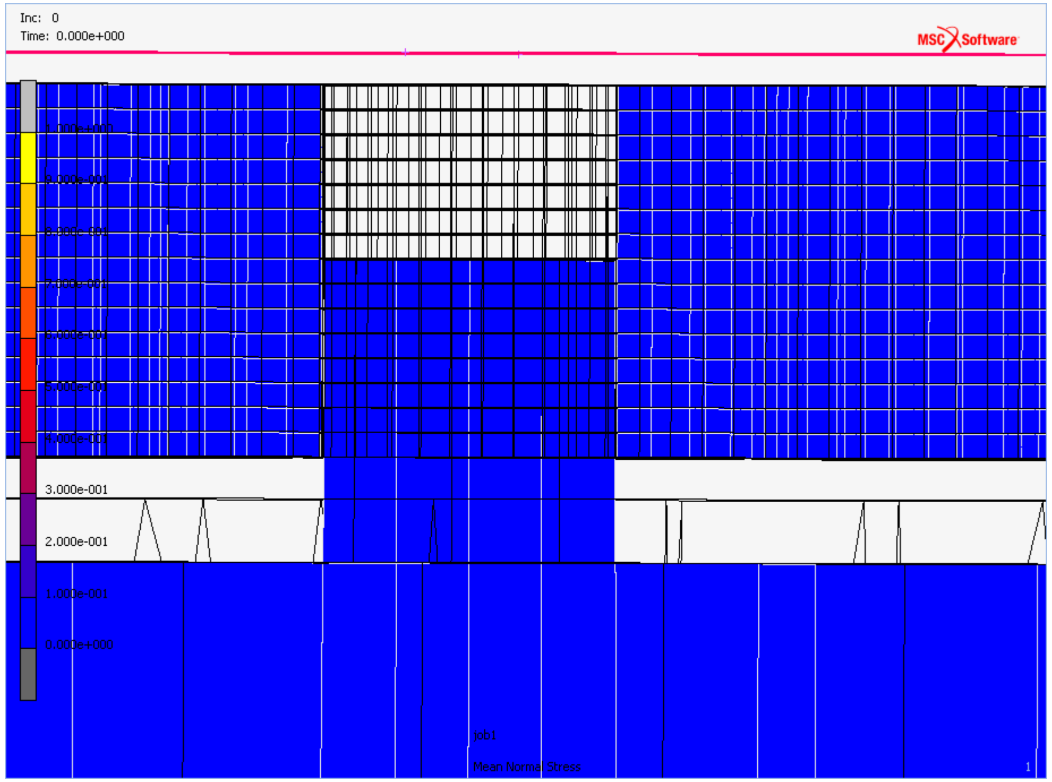
NAS 23°C 150% First Loading

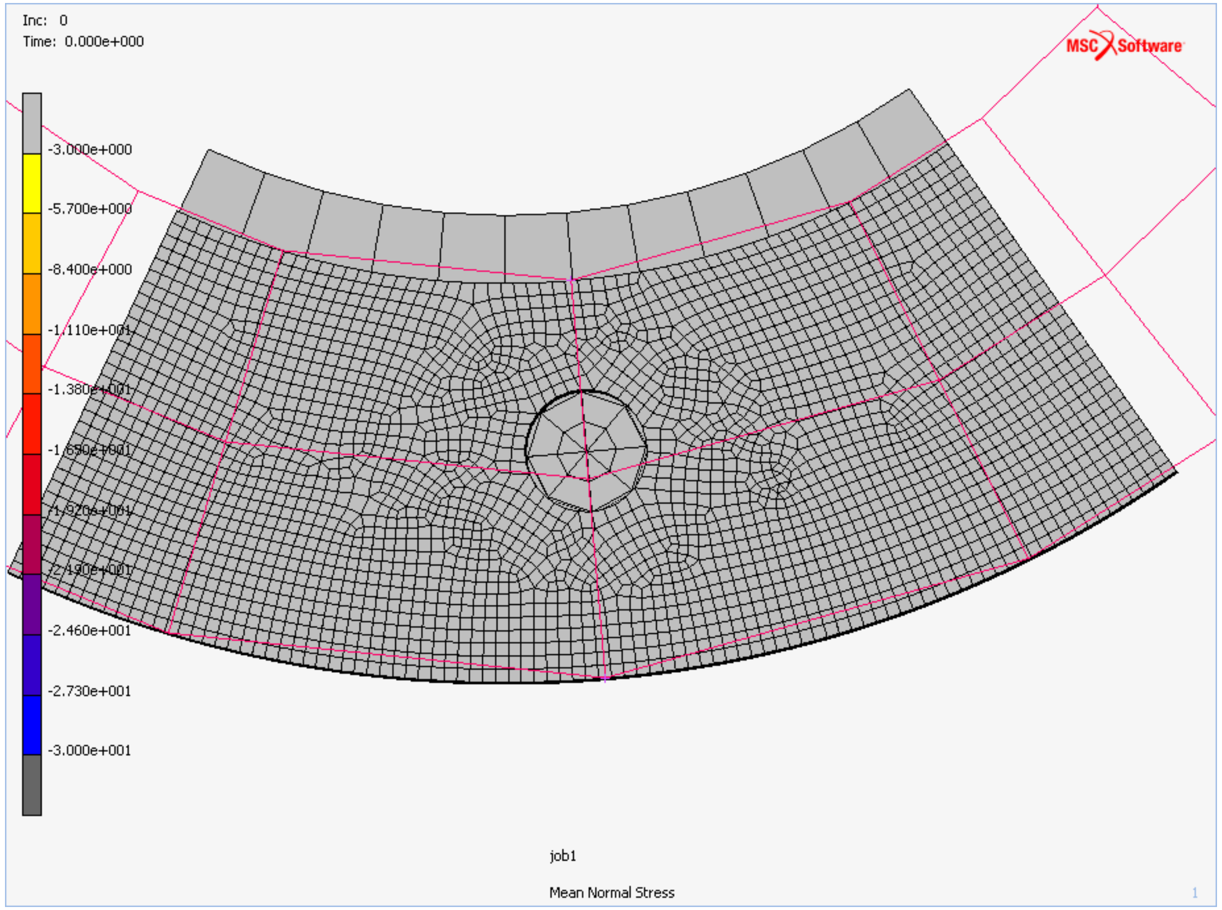


NAS 23°C 150% Strain Summary

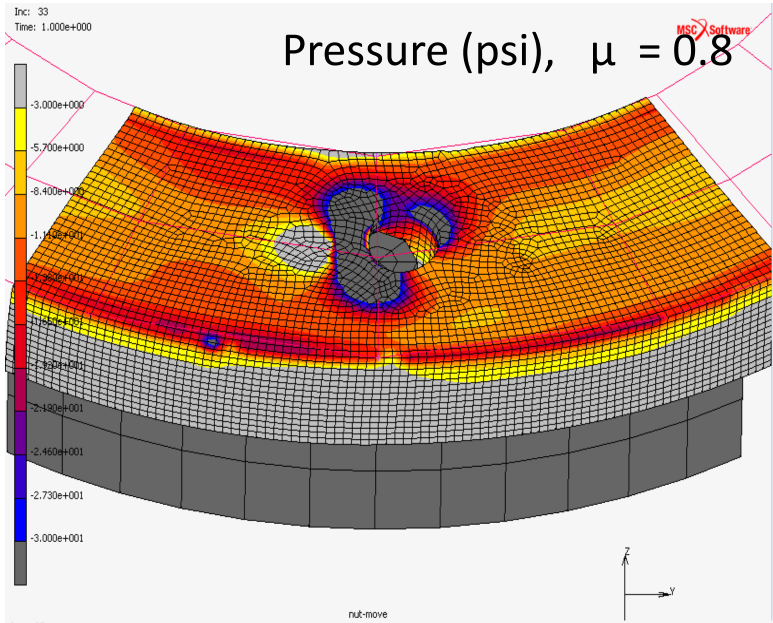


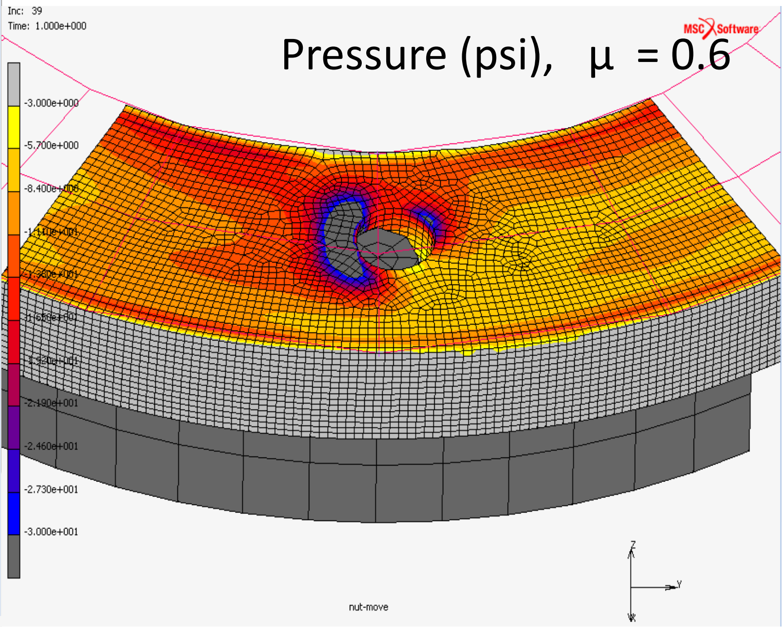
SEAL INTERFACE EVALUATED FOR VARIOUS FRICTIONAL COEFFICIENTS

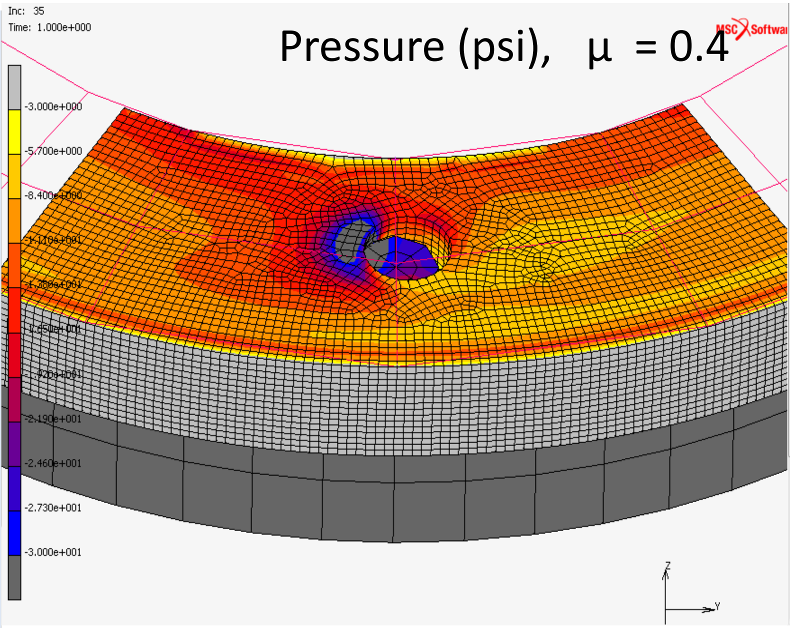


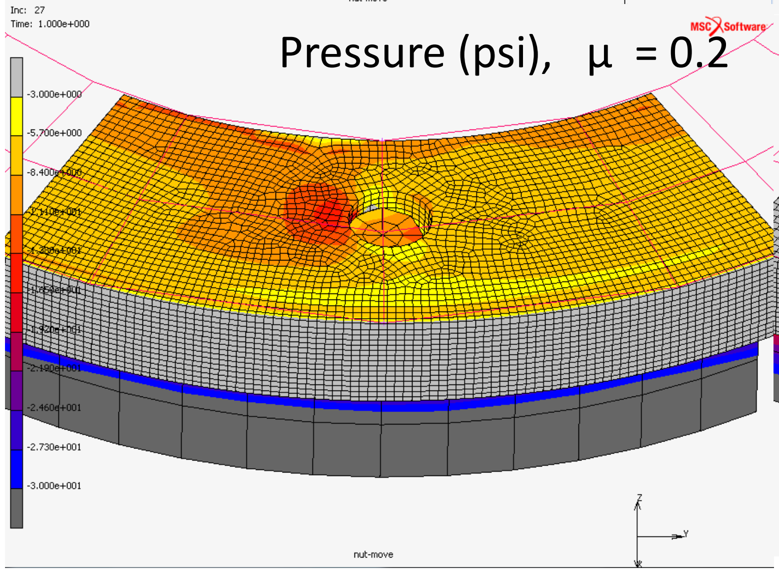


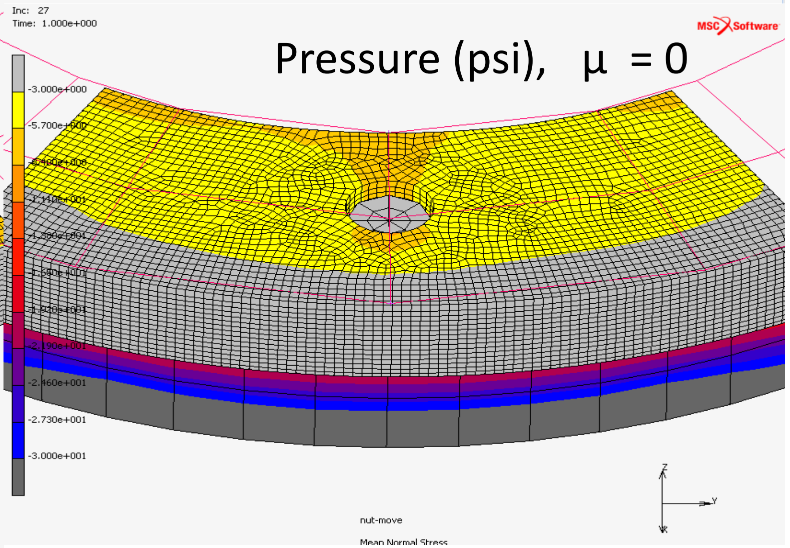
SEAL INTERFACE EVALUATED FOR VARIOUS FRICTIONAL COEFFICIENTS







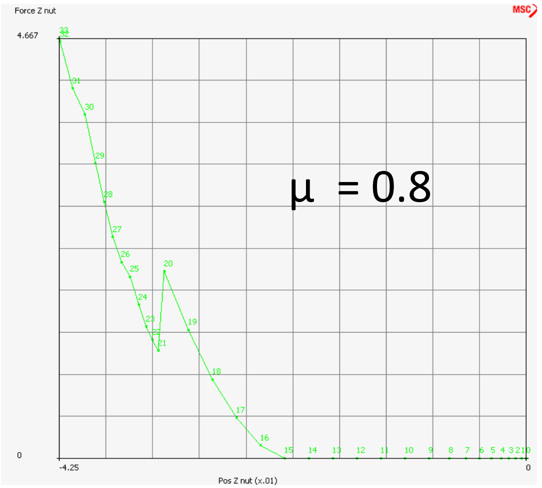




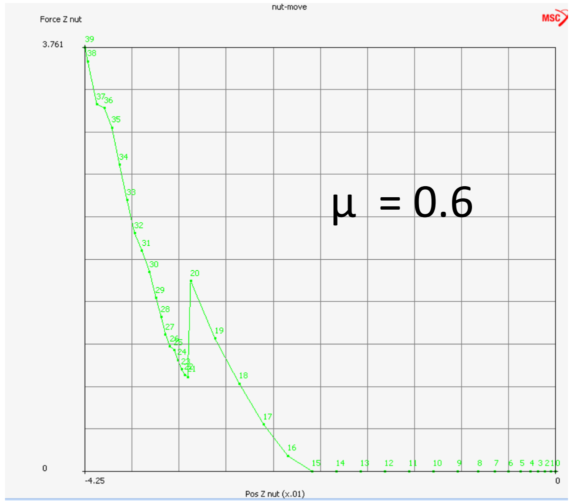
Pinching of the seal material over the positioning boss lead to reduction in effectiveness of the seal.

Thus reducing friction was important

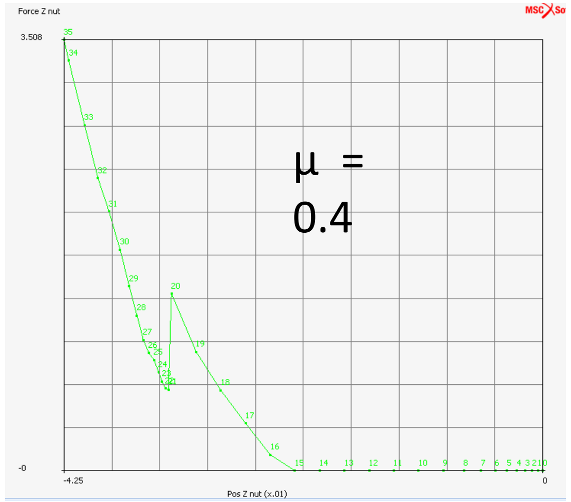
SEAL INTERFACE EVALUATED FOR VARIOUS FRICTIONAL COEFFICIENTS



μ = 0.8

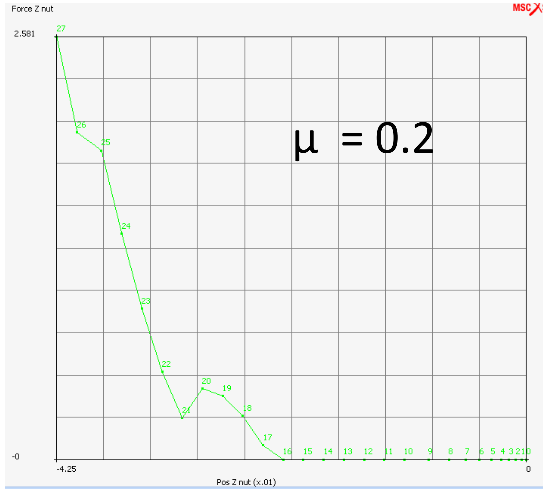


μ = 0.6

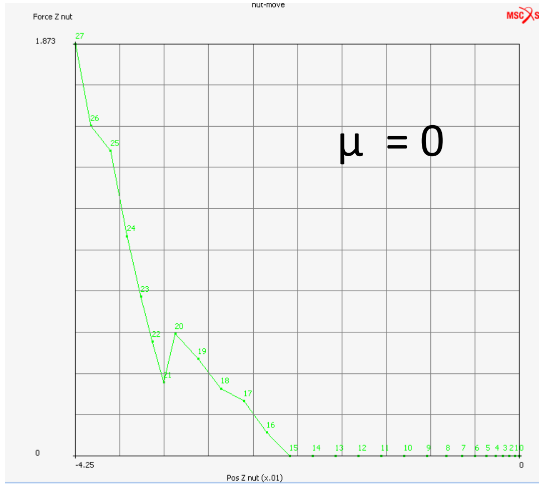


μ = 0.4

This indicates that much of the axial force is just wasted on squeezing the seal material trapped between the pin and the nut.



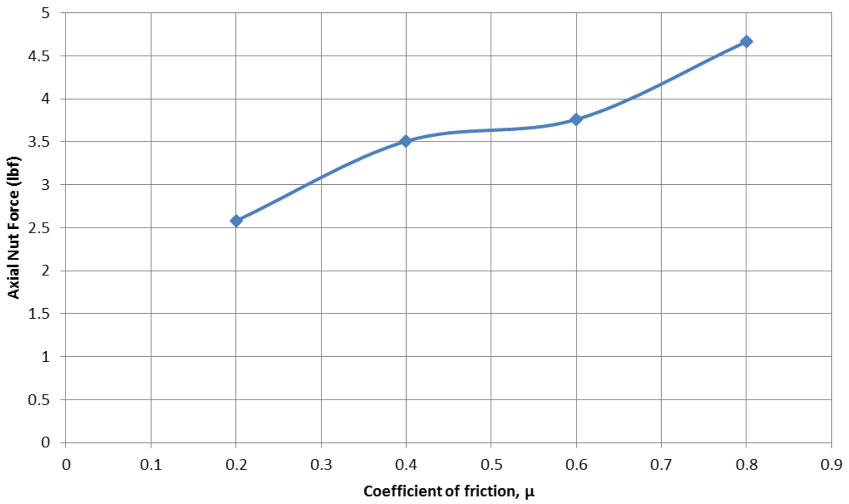
μ = 0.2



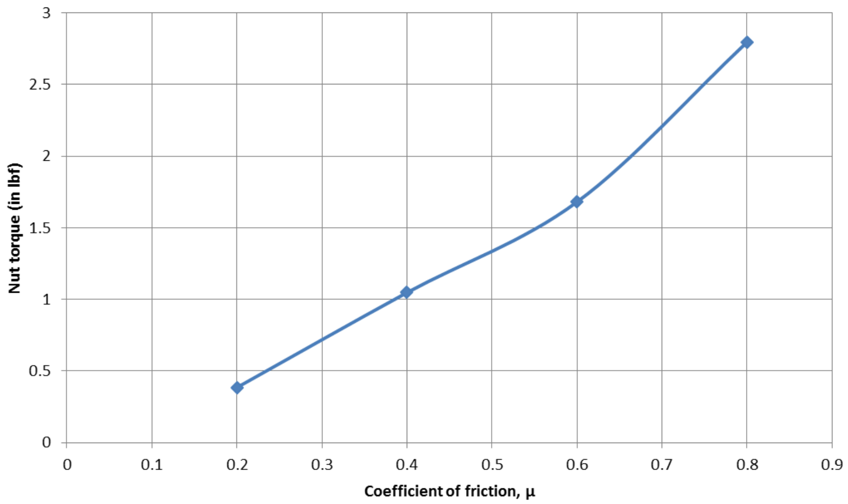
μ = 0

SEAL INTERFACE EVALUATED FOR VARIOUS FRICTIONAL COEFFICIENTS

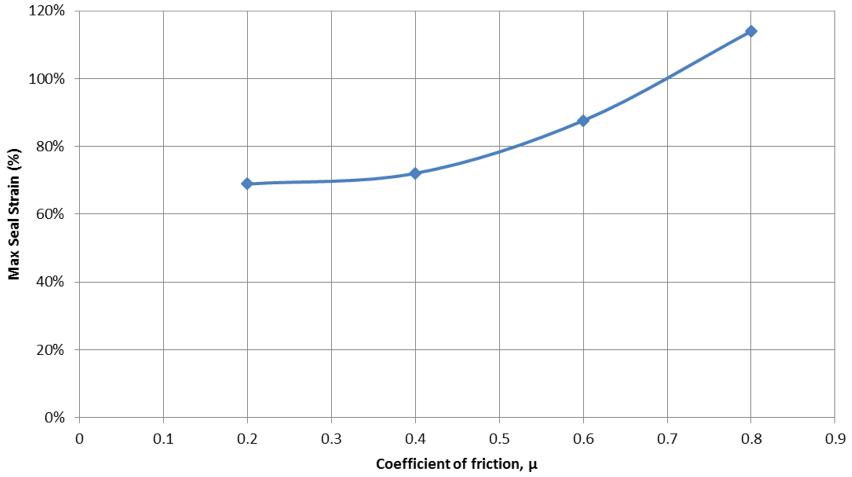
Effect of Friction on Axial Nut Force



Effect of Friction on Nut Torque



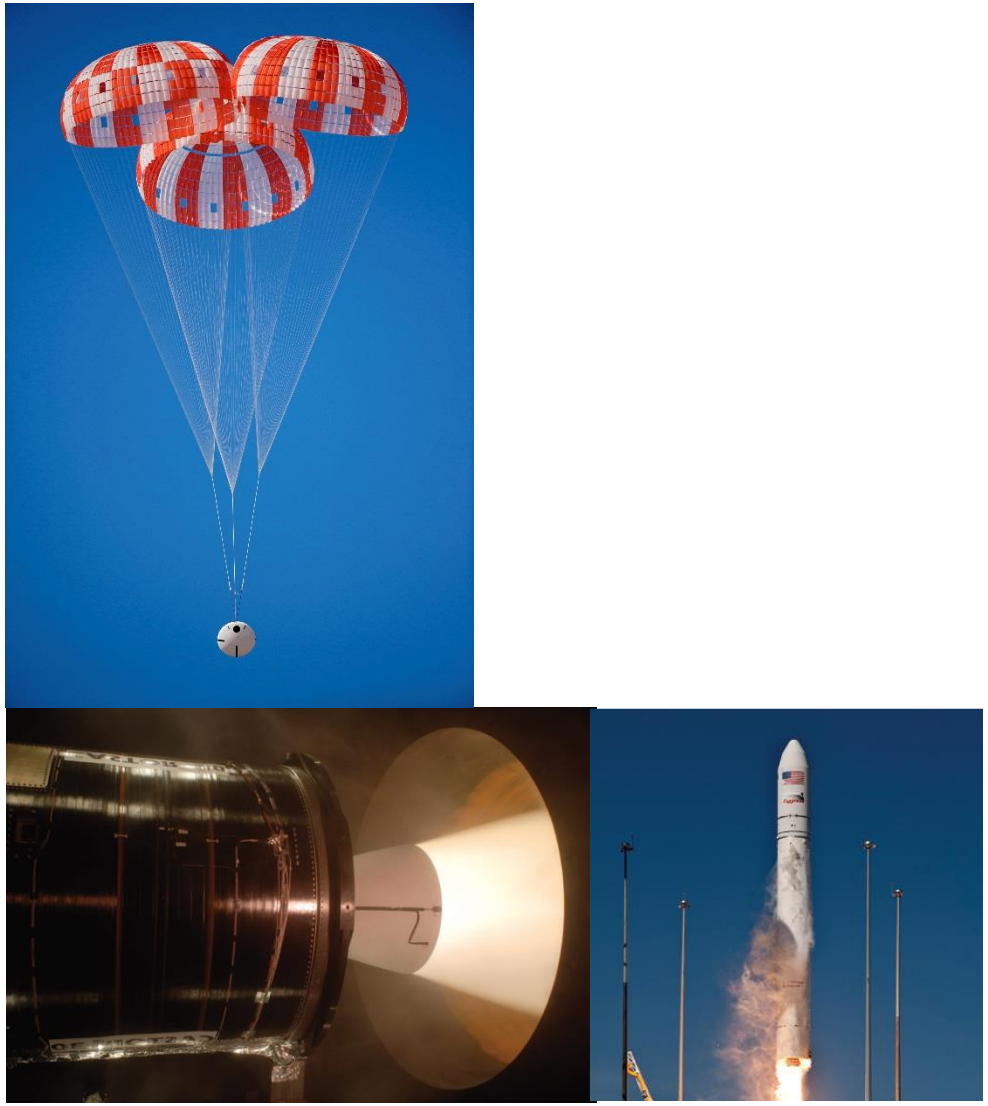
Effect of Friction on Max Seal Strain



OUTCOME

Marc’s hyper-elastic modeling of rubber like materials with appropriate material testing was able to effectively solve the Emergency Mask sealing issues.

National Aeronautics and
Space Administration



Marc Simulations at NASA Johnson Space Center

Part 4: Impact Simulation

MSC Software Nonlinear Users Meeting
April 17, 2017
Ann Arbor, Michigan, USA

Satish Reddy
Chief Engineer, Structural Analysis

Jacobs Technology

View in Slide Show Mode

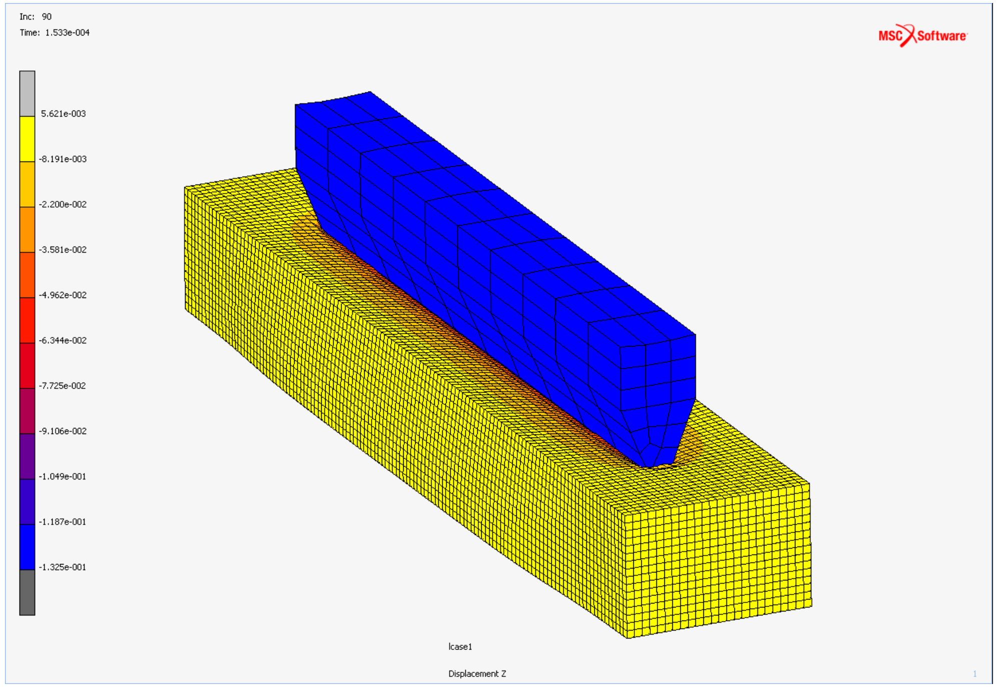
EXAMPLE: BLADE IMPACT SIMULATION

The purpose of this simulation is to identify the max forces that the cutter housing will see during this very high speed, high impact event.

The cutter blade, made of a very hard alloy strikes a relatively soft steel (CRES300 series) anvil at over 200 mph.

The impact of the blade on the anvil drives the blade into the anvil. After a test, it is typically found embedded in the anvil about 0.2 inches deep.

Violent impacts such as this have lead to housing distortion and failure. A better understanding of this impact event will help lead to improved design.



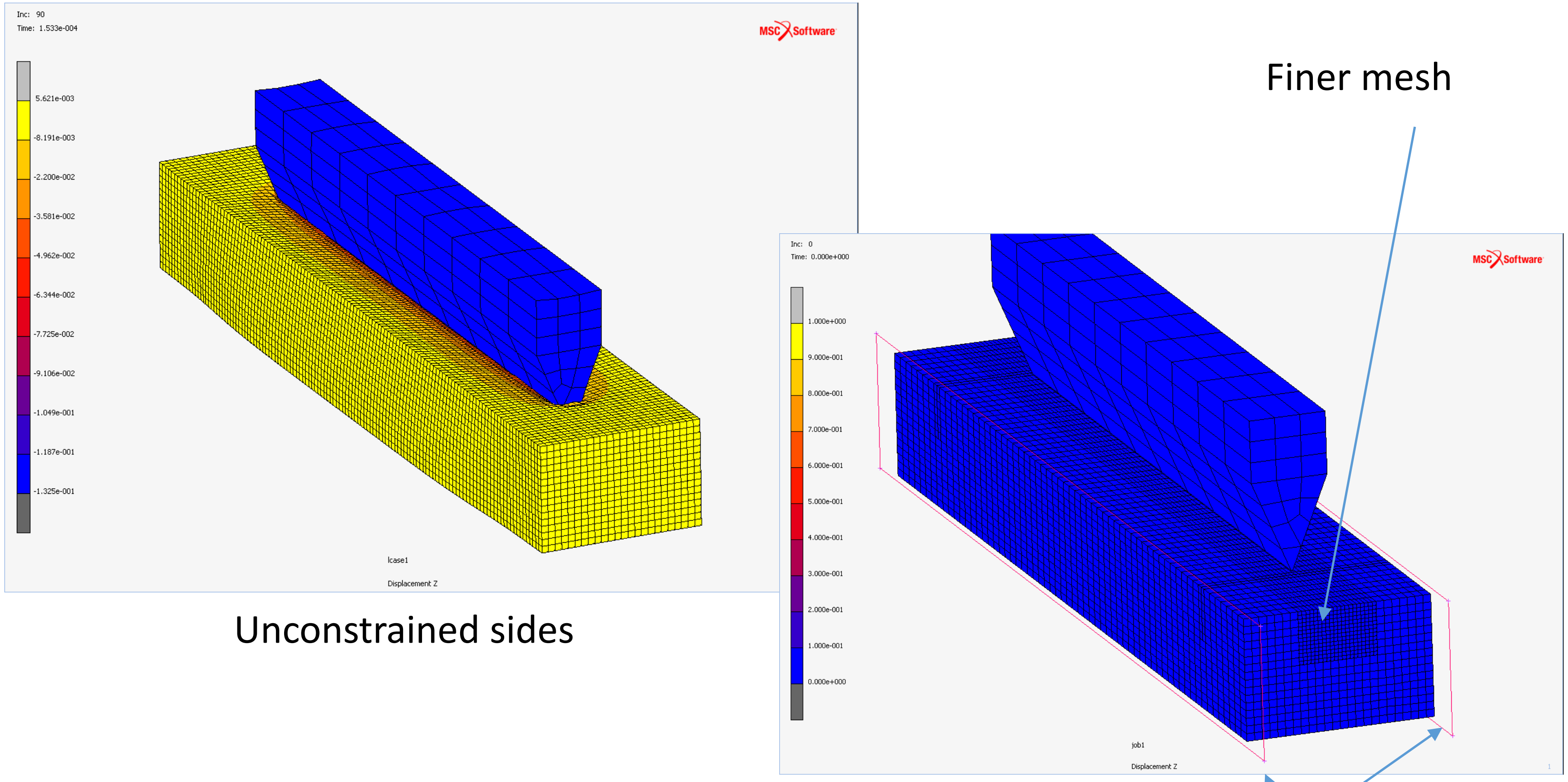
Approach

In MSC Marc non linear dynamic analysis:

1. Build a 3d FE model of the blade and anvil. Assign piston-cutter blade impeller mass as density to the blade. Apply appropriate constraints in the bottom and side of the anvil.
2. Model the blade and anvil contact as a ‘glued’ contact. This simulates the fact that post impact the blade remains stuck in the anvil. It doesn’t bounce back.
3. Provide the impact velocity as the initial velocity of the blade.
4. If the analysis doesn’t exit normally, observe the animation of the last increment. Usually the problem is that the anvil mesh is not fine enough.

Cases

Two cases were investigated. One where the anvil had no side constraint and the other where it was side constrained. Both cases had the bottom fully supported.



Finer mesh

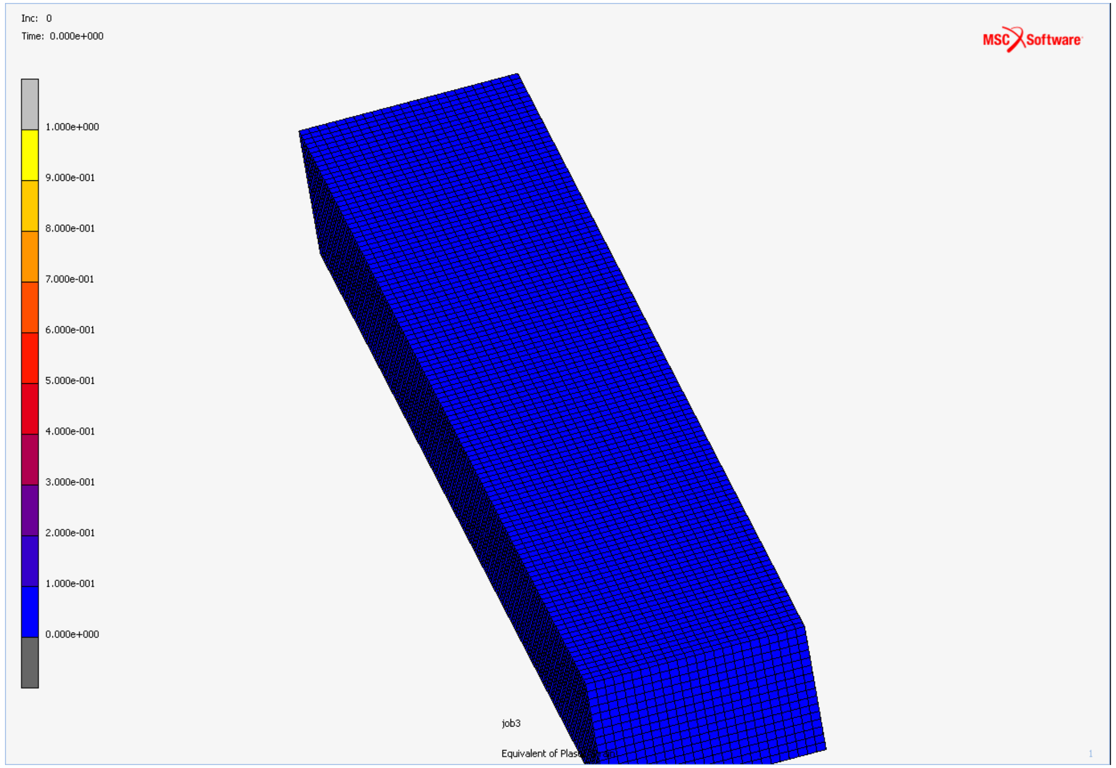
Unconstrained sides

Vertical displacement

Side constrained

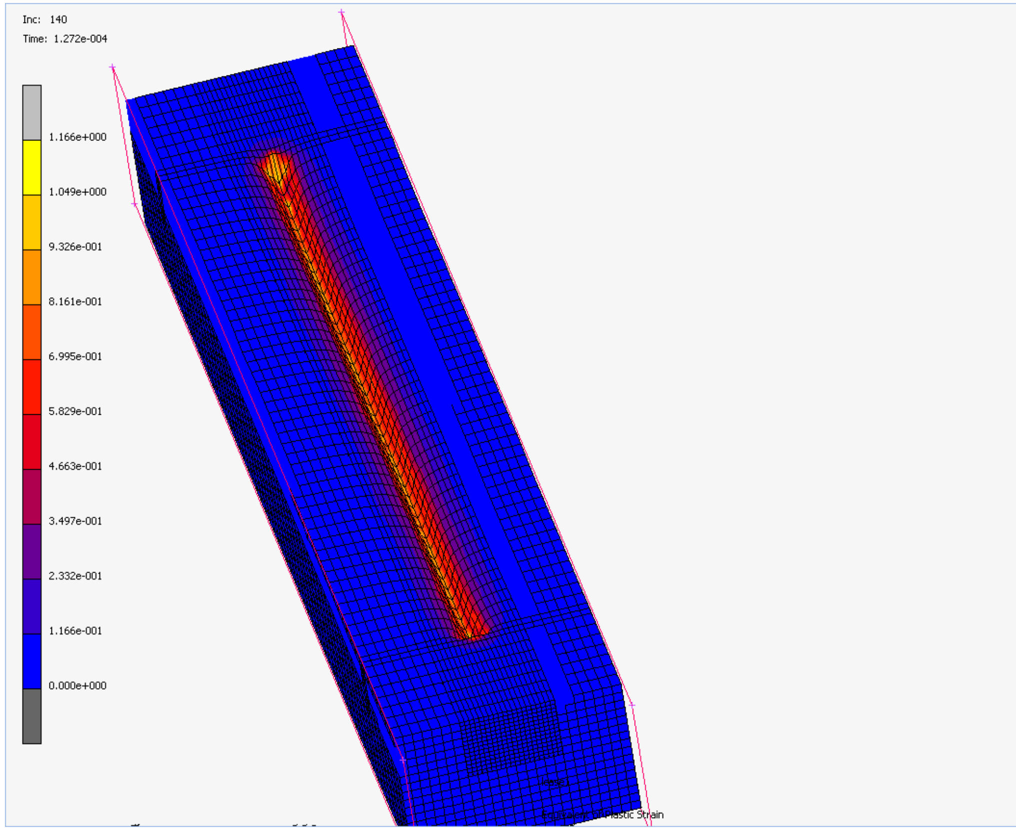
Results: Plastic Strain in anvil

Two cases were investigated. One where the anvil had no side constraint and the other where it was side constrained. Both cases had the bottom fully supported.



Unconstrained sides

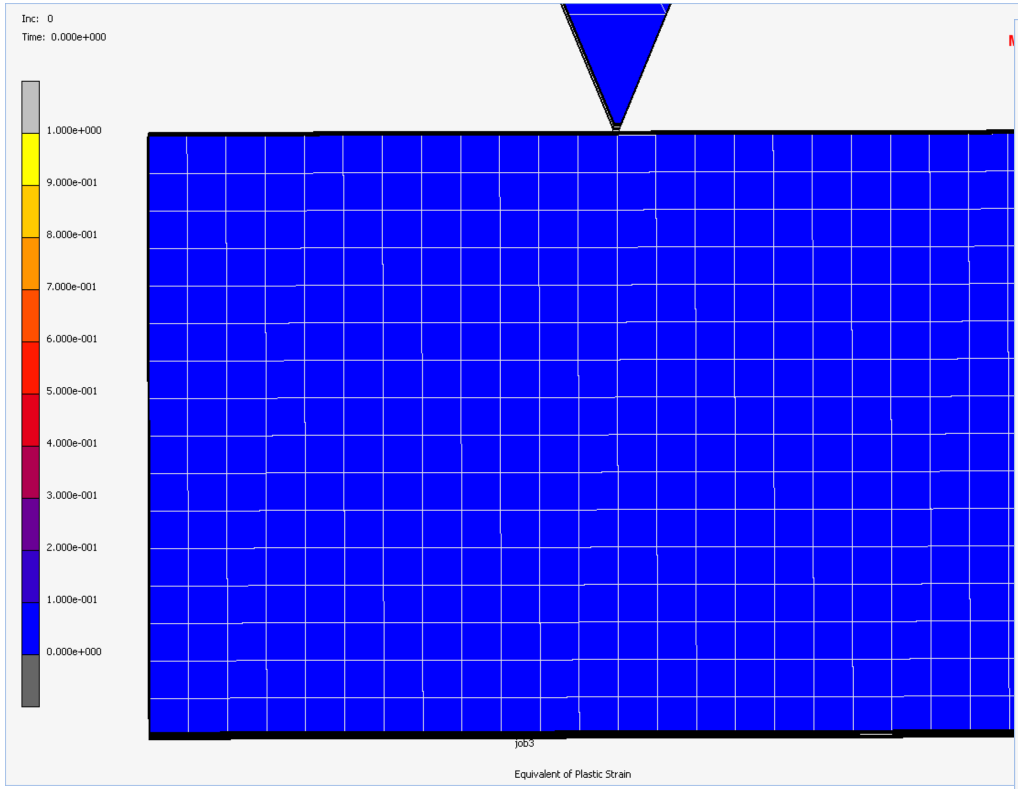
Equivalent of plastic strain



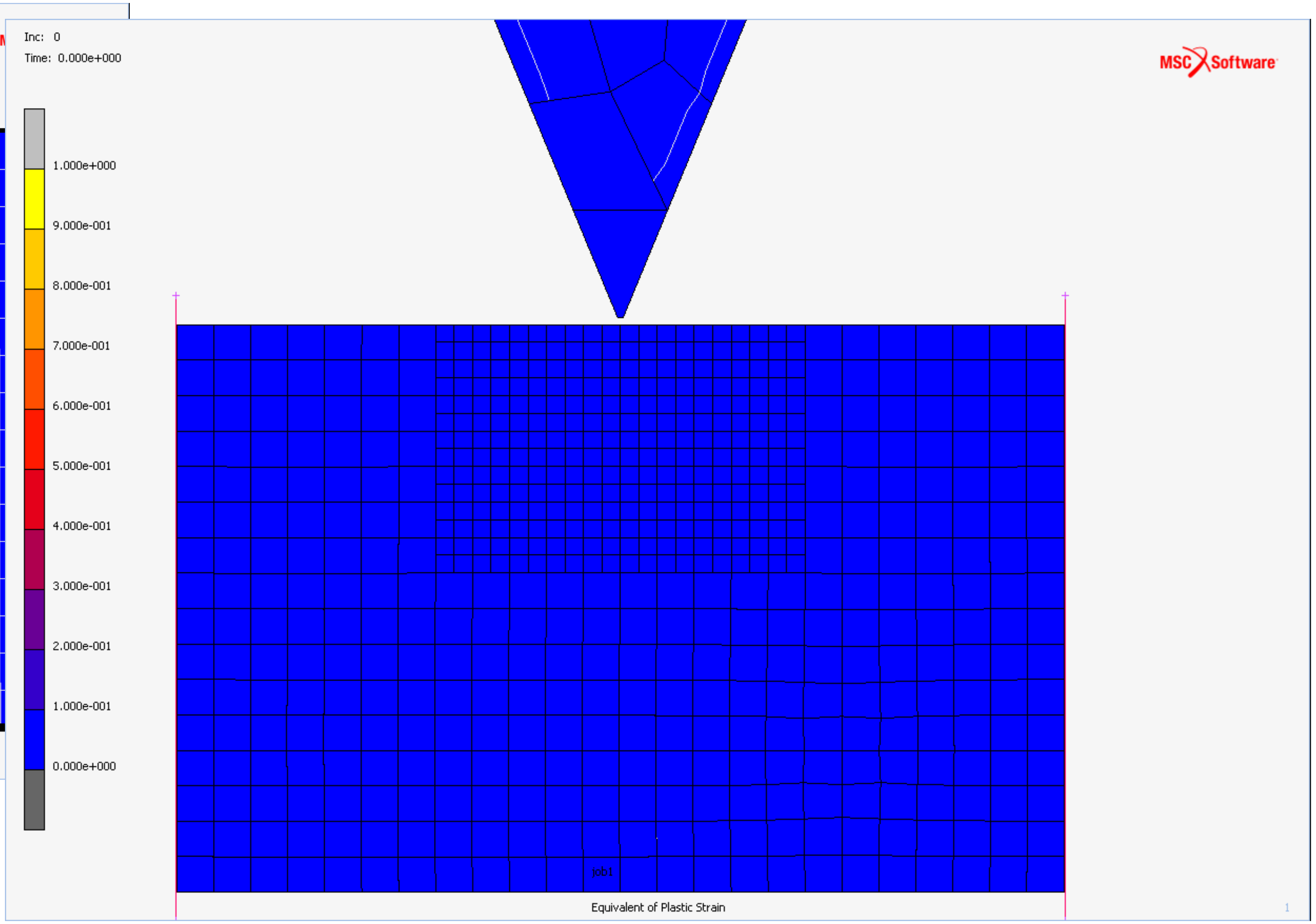
Side constrained

Results: Plastic Strain – section view

Two cases were investigated. One where the anvil had no side constraint and the other where it was side constrained. Both cases had the bottom fully supported.



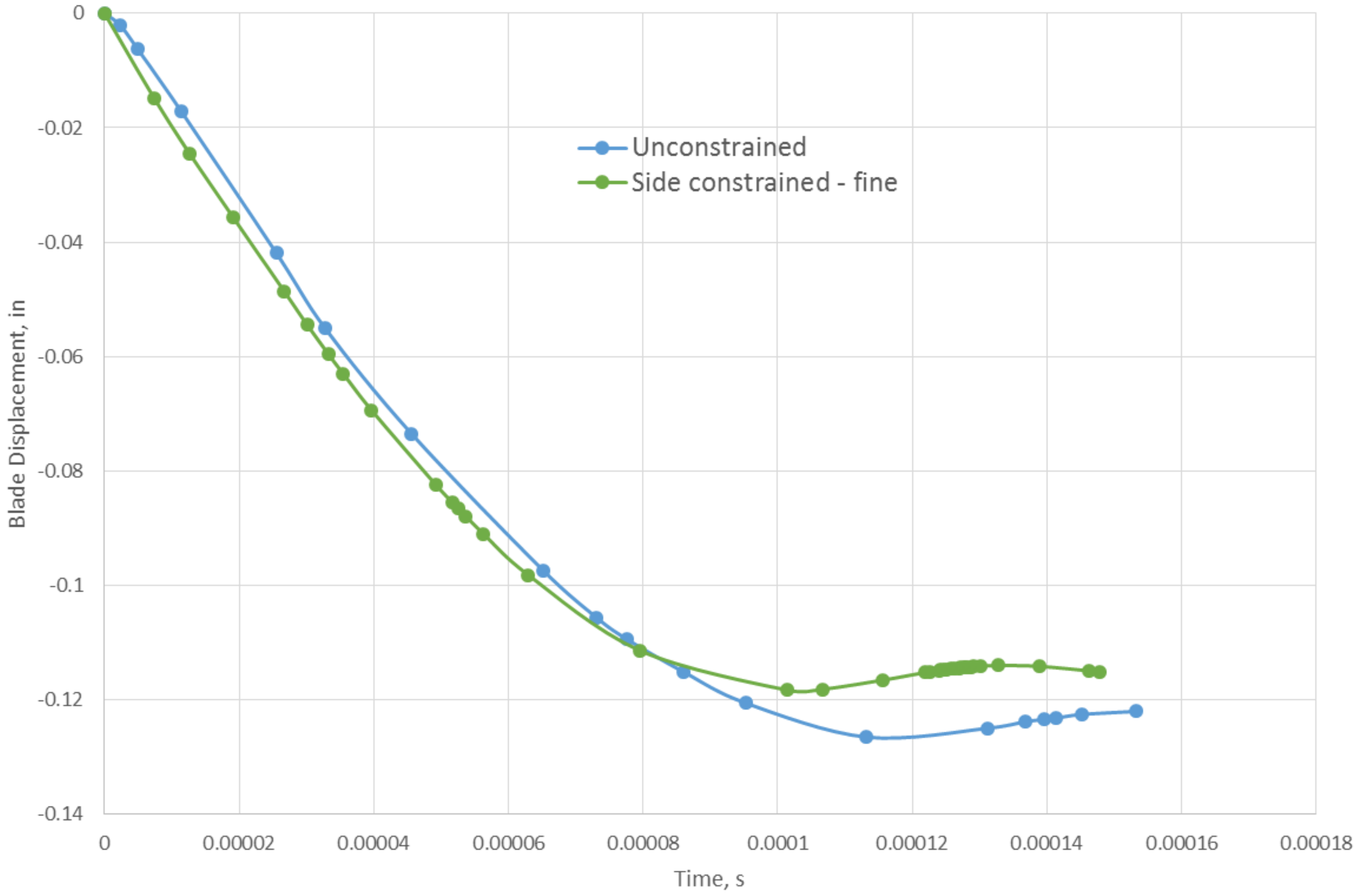
Unconstrained sides



Equivalent of plastic strain

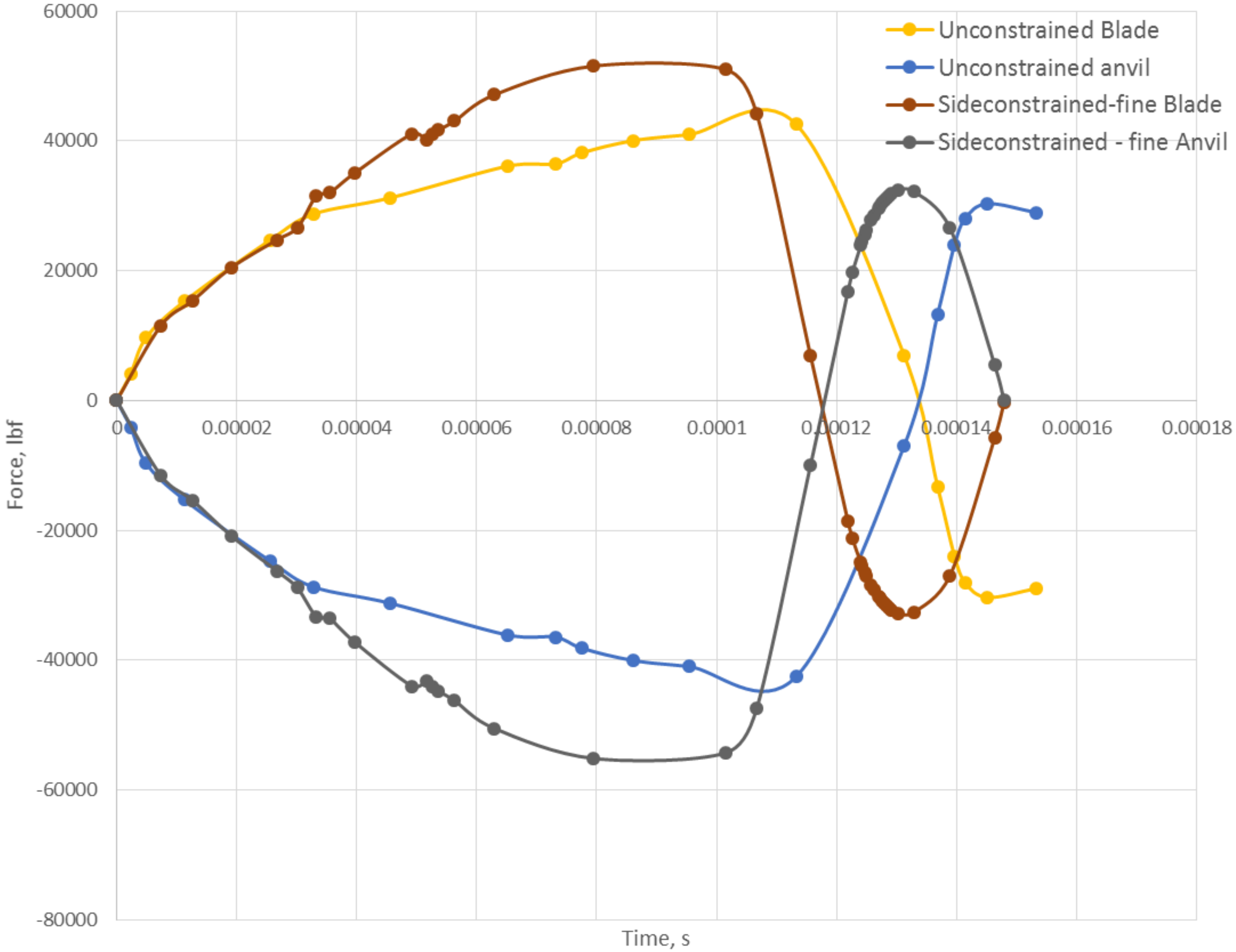
Side constrained

Results: Blade Displacement vs. Time



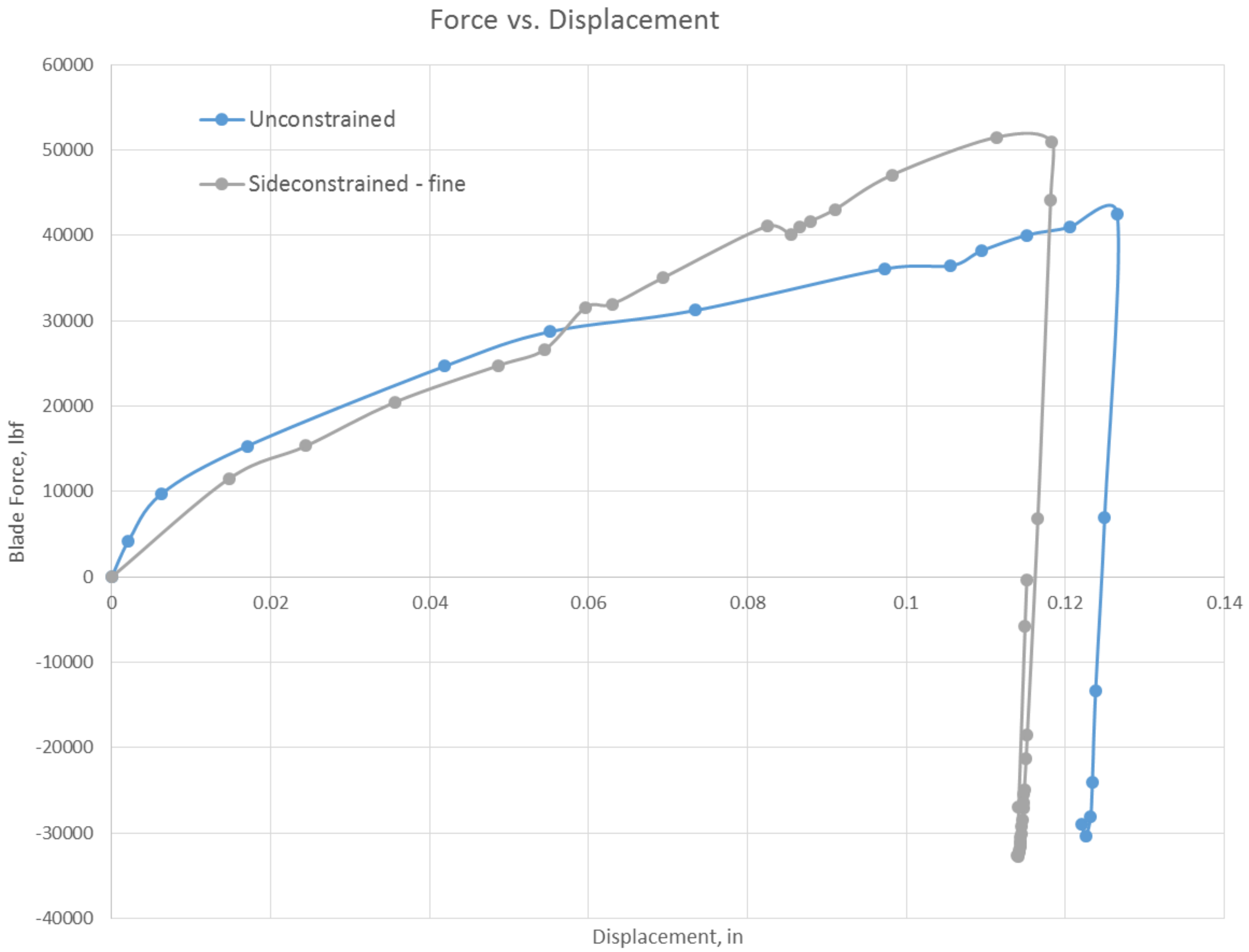
Side constrained case shows smaller vertical displacement as expected.

Results: Blade and Anvil Forces vs. Time



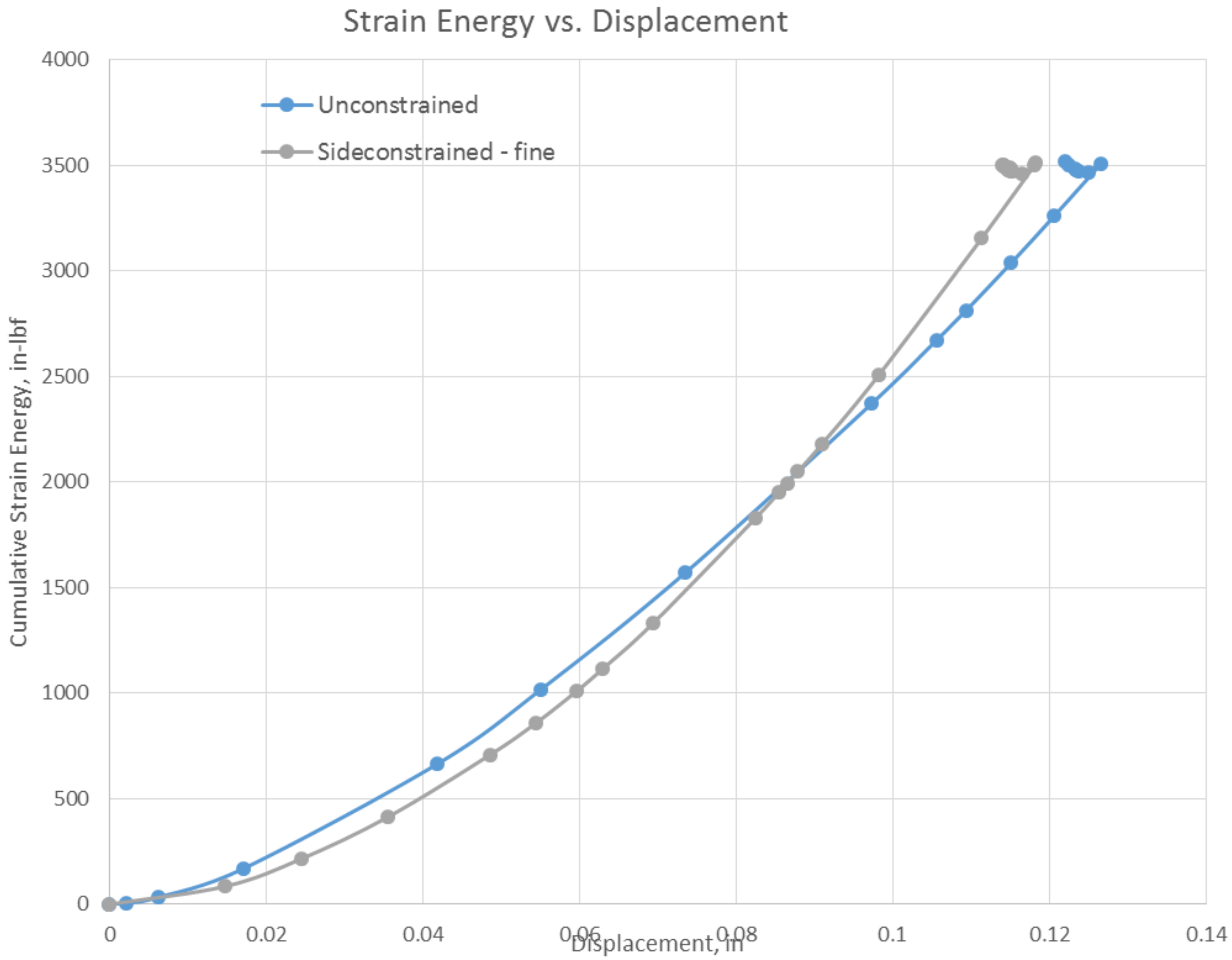
Side constrained case shows larger vertical forces as expected. Anvil forces are equal and opposite to blade forces as expected.

Results: Force vs. Displacement



Side constrained case shows a stiffer result as expected – higher peak force, lower max displacement.

Results: Strain Energy vs. Displacement



Both cases show the same total strain energy, this is because the initial kinetic energy of the blades for both cases were the same.

Analysis Conclusions

1. Peak impact forces are
    - Unconstrained: 42,572 lbf
    - Side constrained: 51,526 lbf

2. The greater the constraint provided to the anvil, greater is the peak impact force. Thus the critical element in the simulation is the geometry and constraints provided to the anvil.

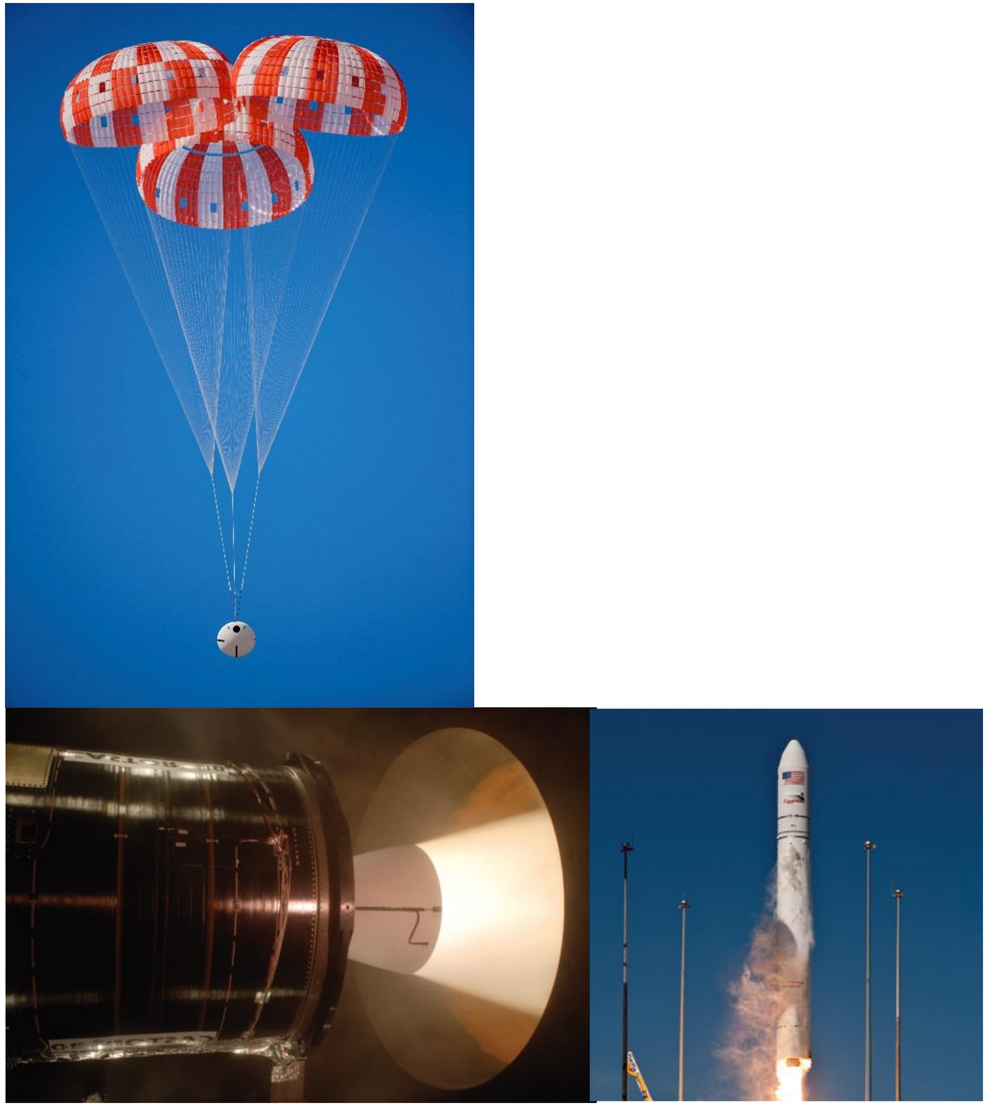
3. These forces are high enough to explain the damage seen in tests

4. These forces when imparted to an FE model of the anvil retainer should help locate potential damage areas and assist in reinforcing them.

5. Once a good stable simulation is accomplished, the anvil retainer may be incorporated into the FE model to run an integrated solution. Set a glued constraint between anvil retainer and anvil to simulate the blade getting stuck in the anvil.

OUTCOME

Marc’s modeling of a impact simulation with gross flow of material mimicked testing validating the use of this kind of simulation.



Marc Simulations at NASA Johnson Space Center

Part 5: High Plastic Deformation Simulation

MSC Software Nonlinear Users Meeting
April 17, 2017
Ann Arbor, Michigan, USA

Satish Reddy
Chief Engineer, Structural Analysis

Jacobs Technology

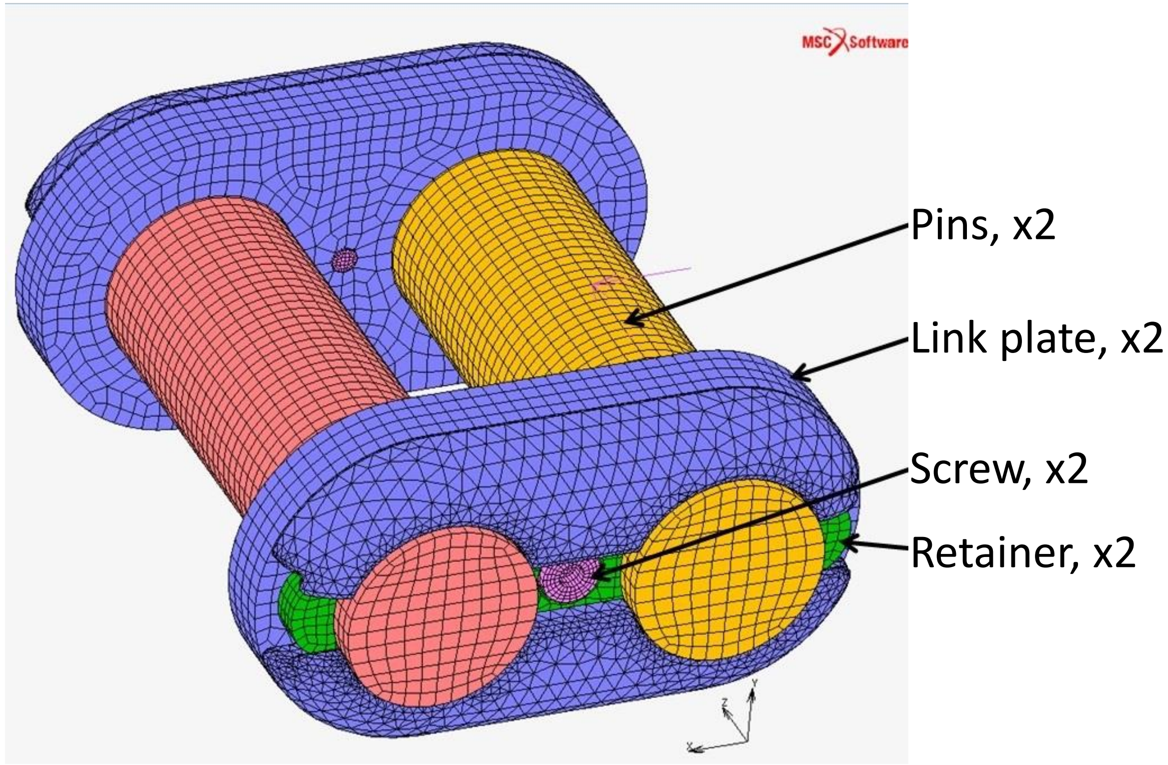
View in Slide Show Mode

EXAMPLE: PULL TO FAILURE OF A LINK

The primary purpose of this simulation is to see how well Marc simulates a pull to failure.

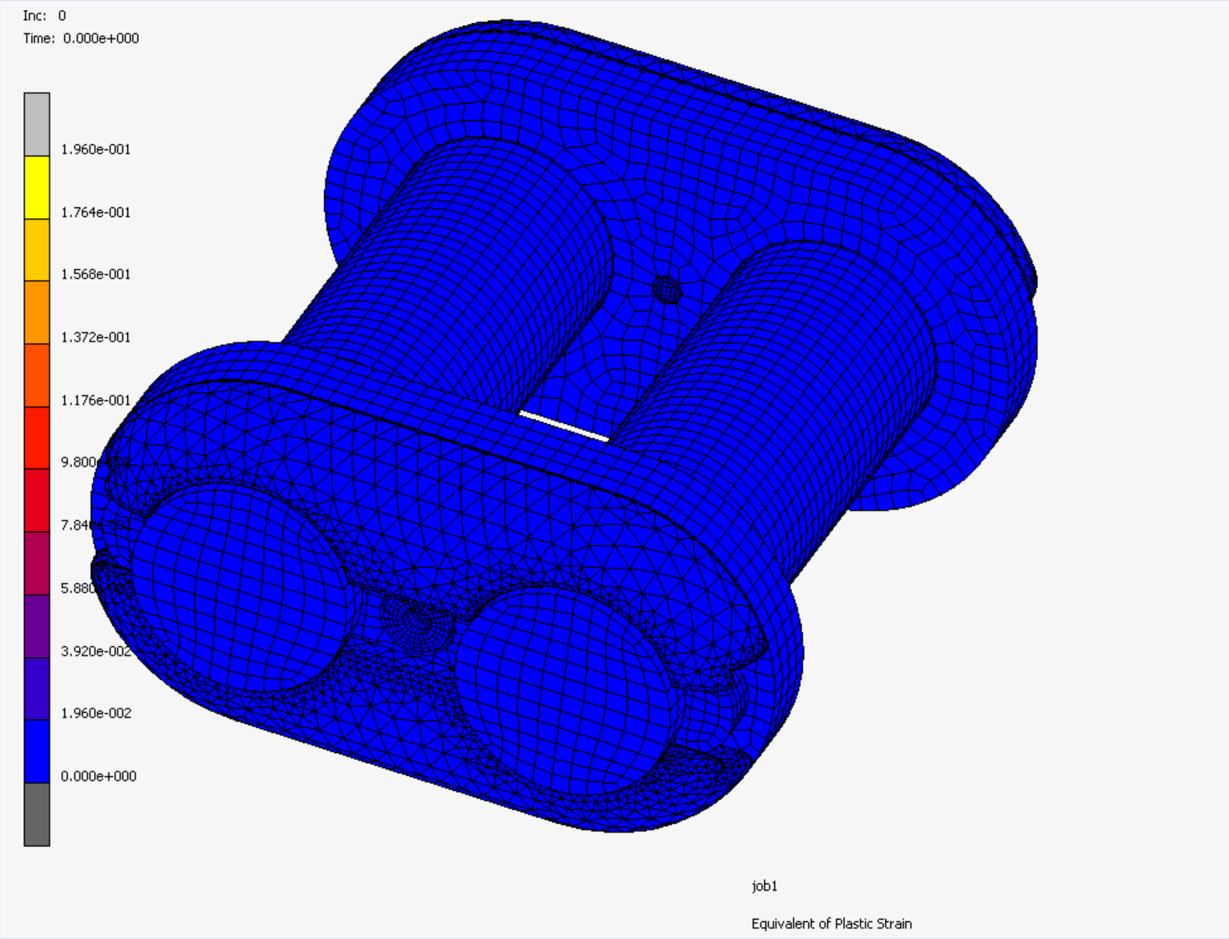
Other objectives include:

- Marc’s contact simulation – glued, touching
- Nonlinear material property simulation
- Mesh density required for adequate simulation
- Interaction of different types of elements
- Geometric nonlinearity – high deformation problems



The Test

The Simulation Prediction



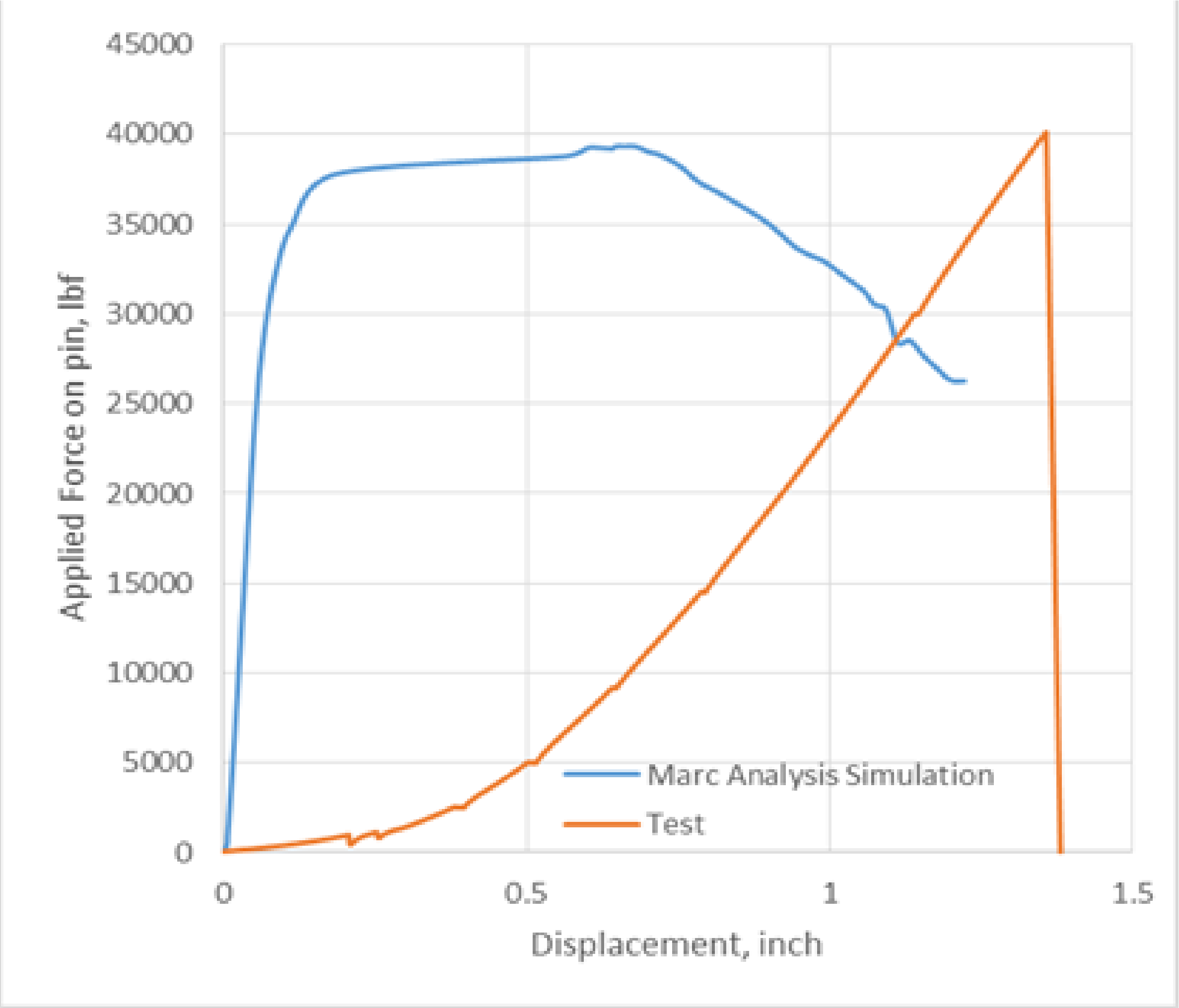
The Test Result

Comparison

Ultimate Load

- Marc simulation prediction: 39352 lbf
- Test Result: 40127 lbf
- 1.9% difference

The test displacement is dominated by the stretch of the fabric strap used to hold the link explaining the difference in shape of the two curves.



OUTCOME

Marc’s modeling of a ultimate load evaluation pull test showed:

- Excellent ultimate load prediction
- Effectively simulated the failure mode

EXAMPLE: EXCESSIVE SET SCREW FORCE

The primary purpose of this simulation is to investigate the failure of a clutch on the Advanced Resistive Exercise Device (ARED).

This is a weight lifting machine that works in the International Space Station microgravity environment to help astronauts maintain bone and muscle strength.

A root cause analysis indicated that excessive set screw force may have distorted a clutch housing to the point that it interfered with a underlying shaft.

https://www.nasa.gov/pdf/553871main_AP_ST_Phys_ARED.pdf

https://www.youtube.com/watch?v=YxImeOomkUk

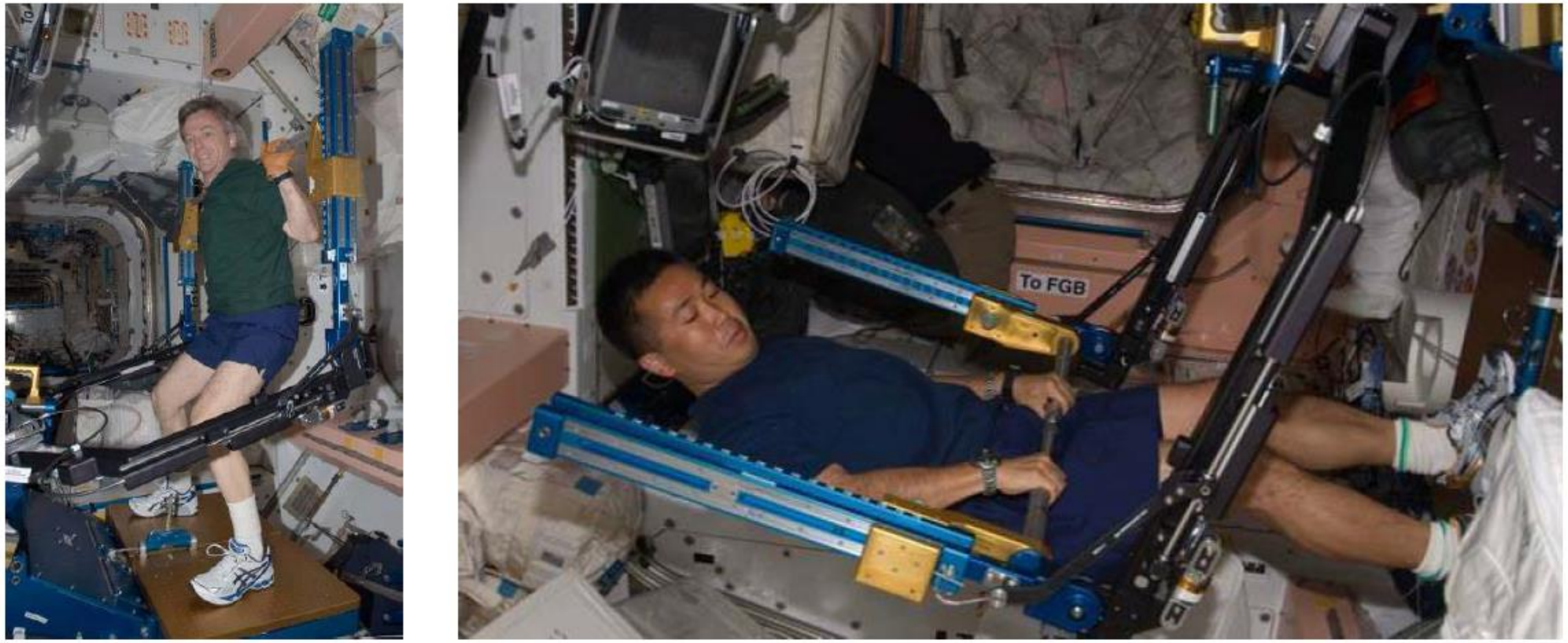
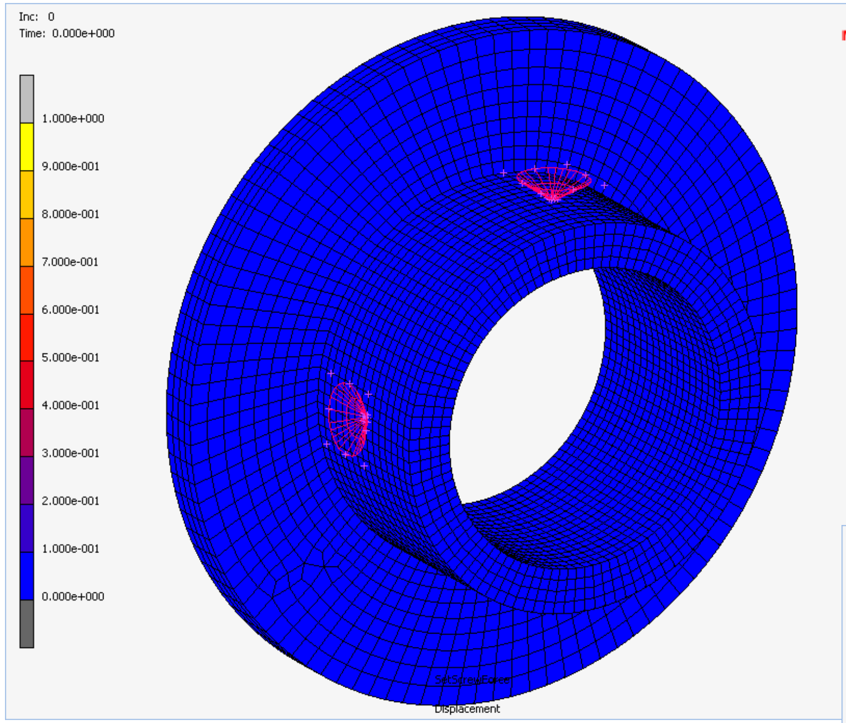


Figure 1: Canadian Astronaut Robert Thirsk (left) and Japanese Astronaut Koichi Wakata (right) use ARED onboard the ISS.

The problem:



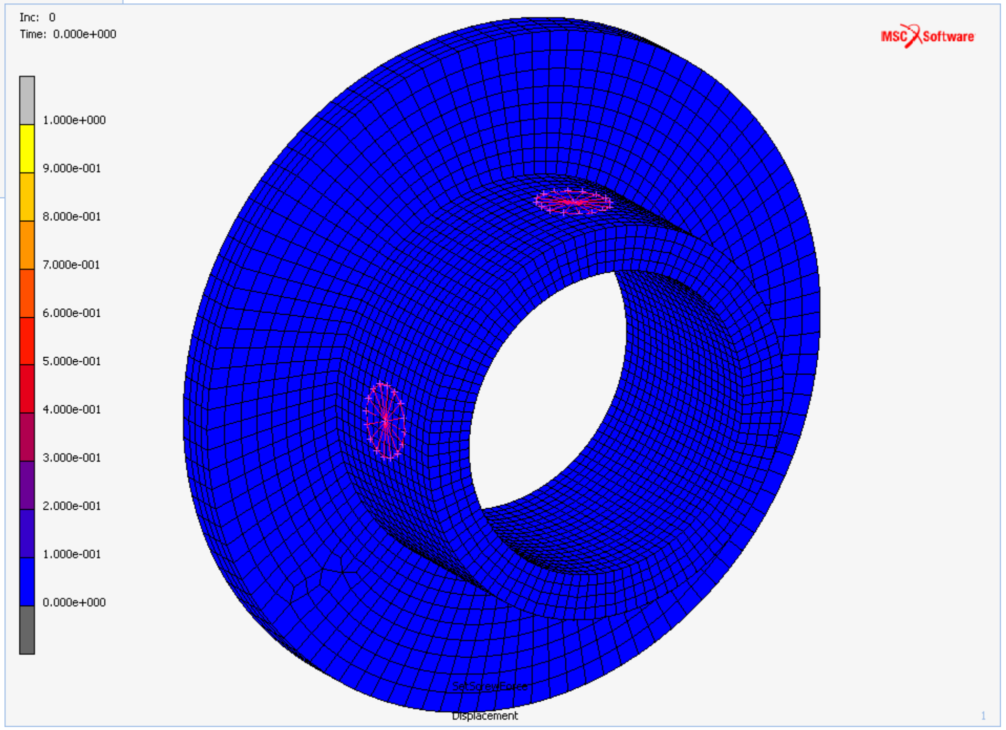
Pointed tip set screw

Max set screw torque: 85 in lbf (-303 design)

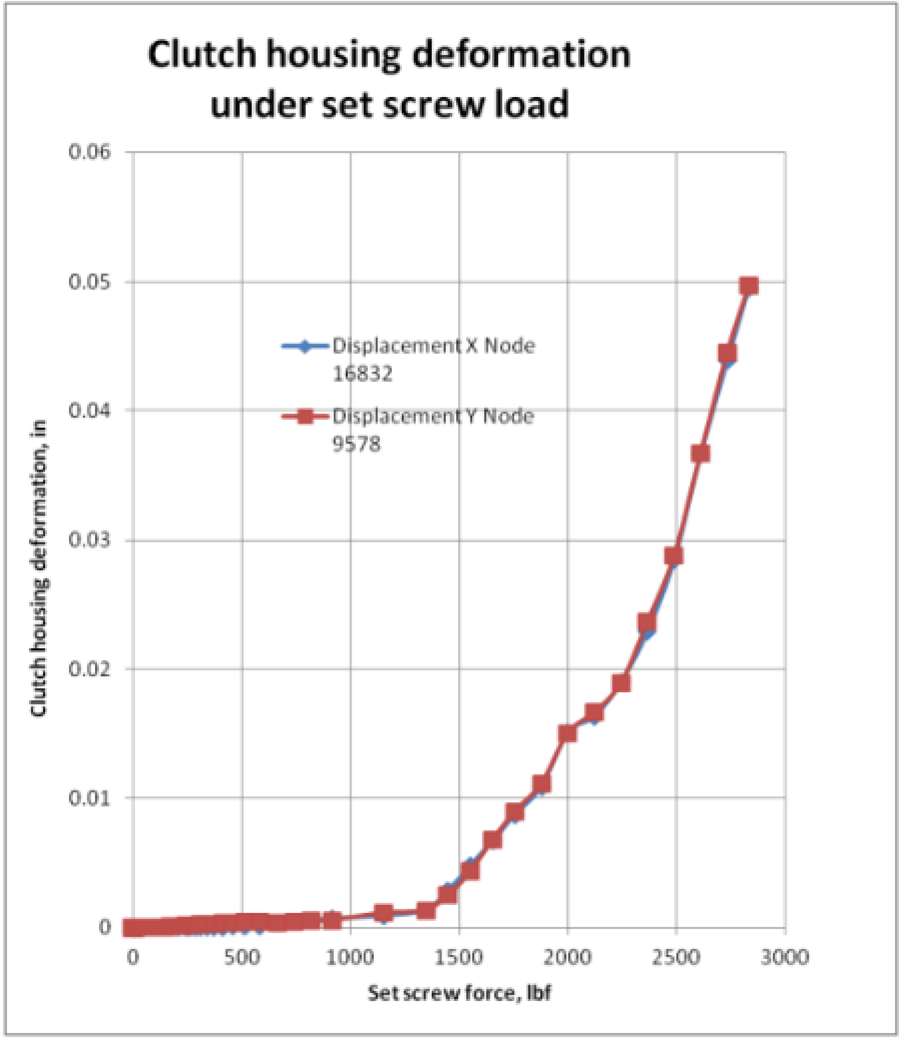
The solution:

Flat tip set screw

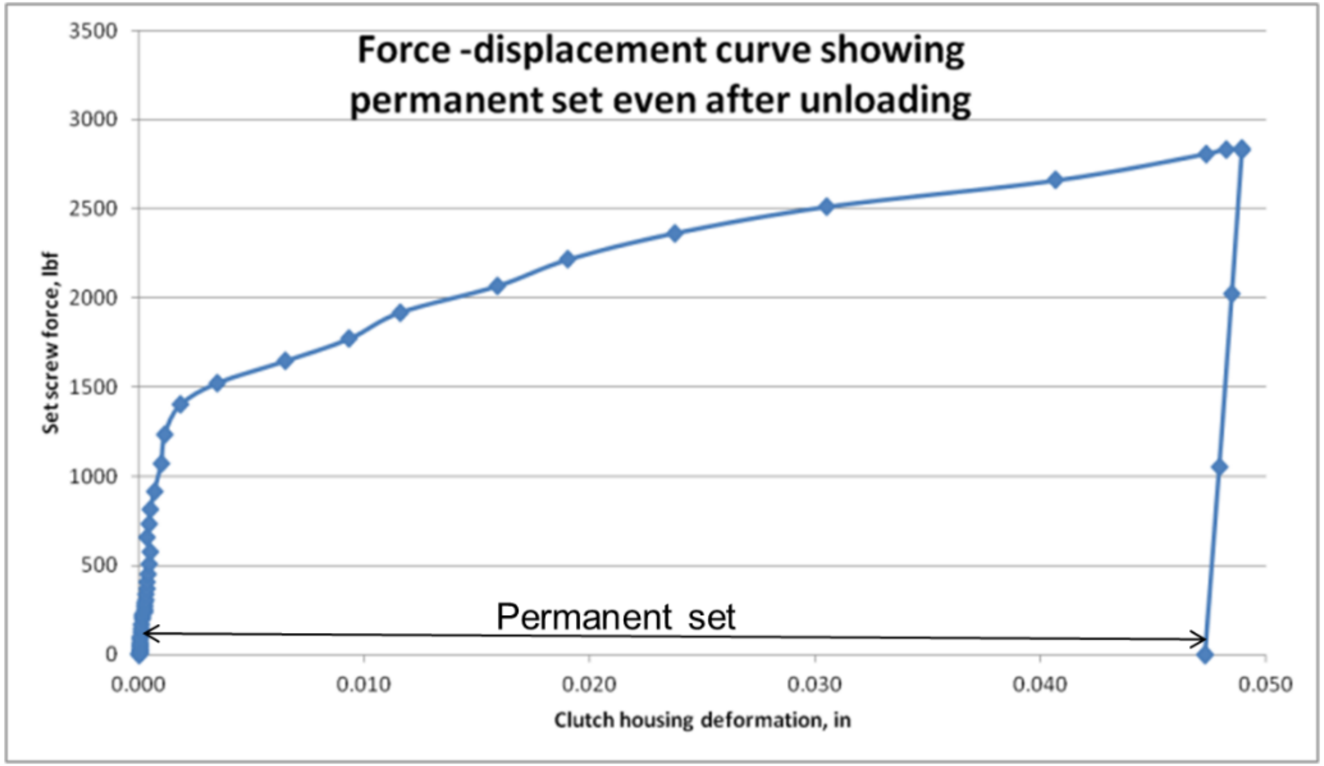
Max set screw torque: 85 in lbf (-303 design)



Results

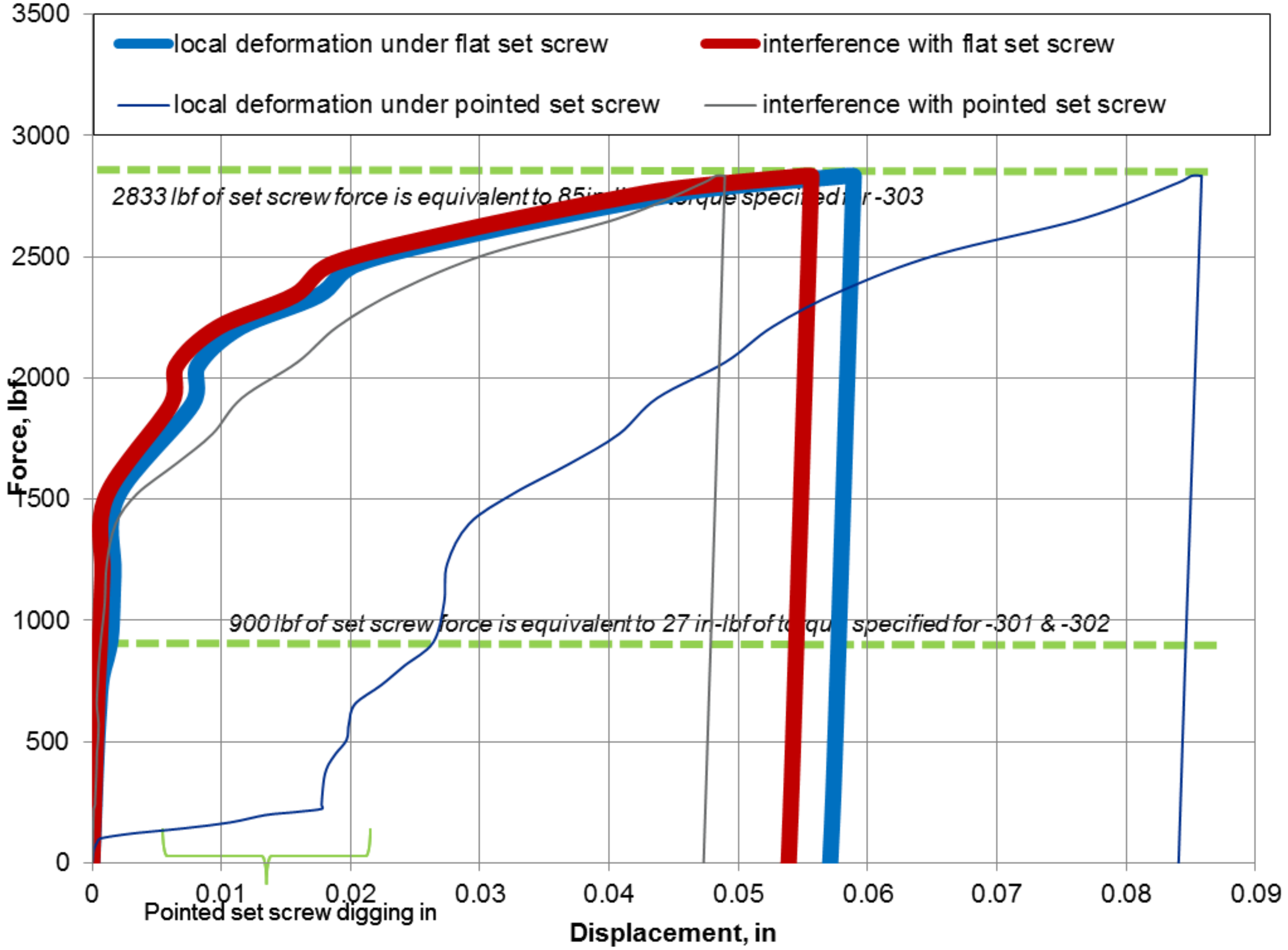


- Small deformations until about 1400 lbf set screw load.
- Large deformations occur beyond 1400 lbf set screw load
- Peak deformation of 0.050 in under a set screw force of 2833 lbf.
- This is sufficient to cause binding at the clutch bearing.



Comparison of pointed set screw vs flat set screw:

- The pointed set screw was replaced by a flat tipped set screw and the non linear analysis re-run.



1. The flat tipped set screws don’t dig in as much as previously seen with the pointed ones. Negligible damage to housing at -301/-302 torques of 27 in-lbf (900 lbf force).
2. The pointed set screws begin digging into the soft housing at just about a 100 lbf of set screw loading. To prevent this from happening, the torque will need to be backed off to finger tight. This will not retain the screw during vibration due to operation

OUTCOME

Marc’s modeling of the clutch with set screw resulted in:

- Confirming that the set screw was causing the interference and jamming of the mechanism
- Helped identify a solution – replacing a pointed set screw with a flat set screw
- Figure out the set screw torque that will ensure jam free working of the mechanism