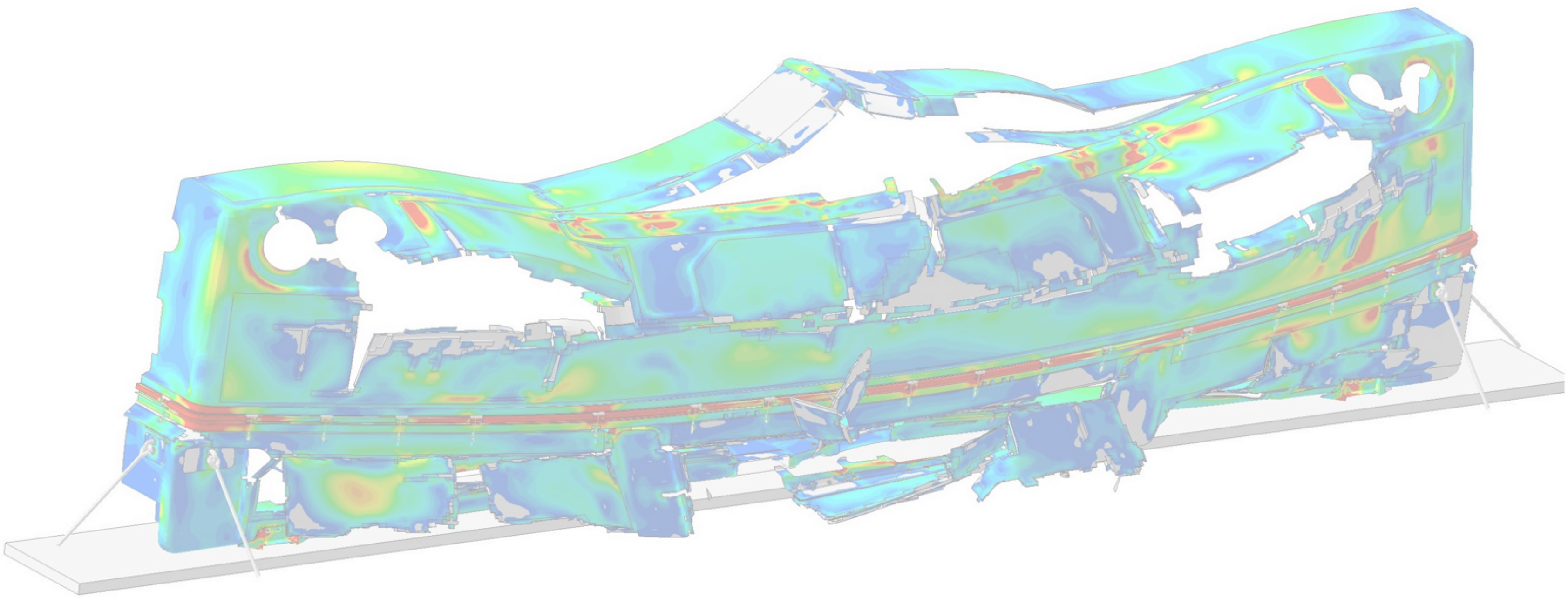


Linear and Nonlinear Analysis with Femap and LS-DYNA

An introduction to the world of LS-DYNA using Femap for pre and post processing

Femap Symposium – Cypress, CA – May 15th, 2018

Adrian Jensen, PE | Senior Staff Mechanical Engineer

FINITE ELEMENT ANALYSIS
Predictive Engineering

LS-DYNA Sales, Support & Consulting

www.PredictiveEngineering.com

AppliedCAx

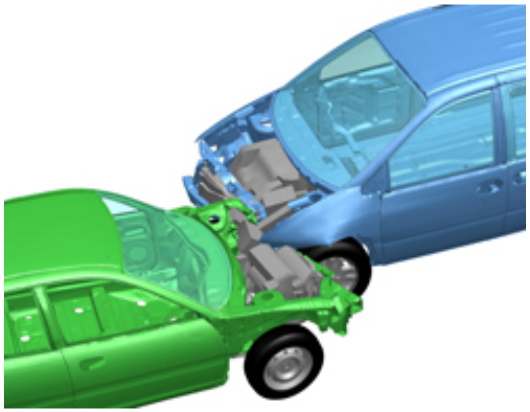
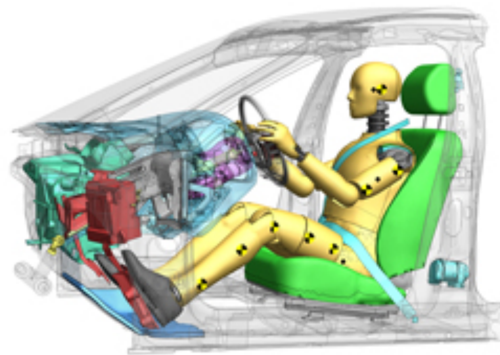
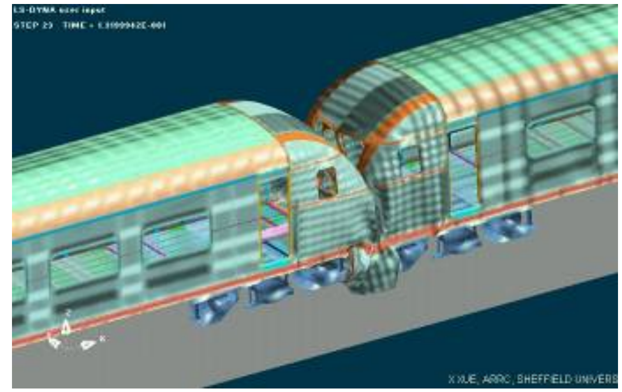
Siemens PLM Software Sales & Support
CAD I CAM I CAE I Teamcenter

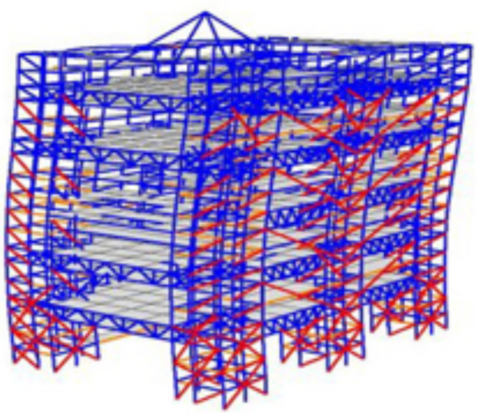
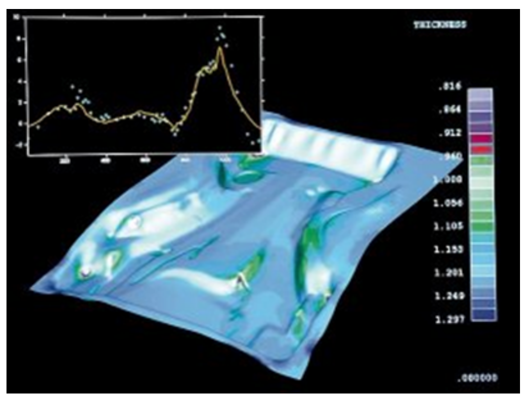
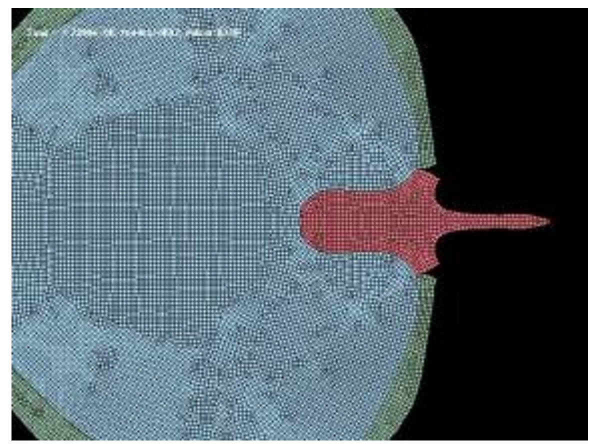
www.AppliedCAx.com

TABLE OF CONTENTS

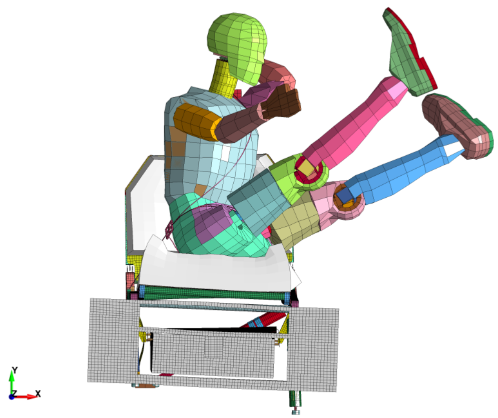
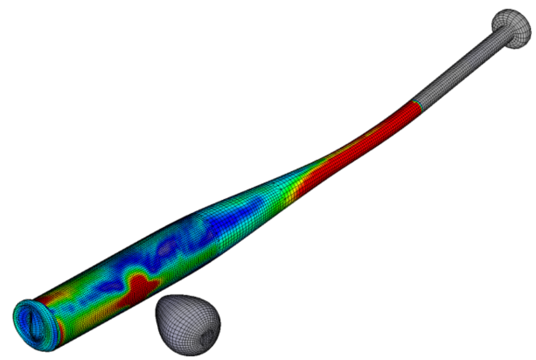
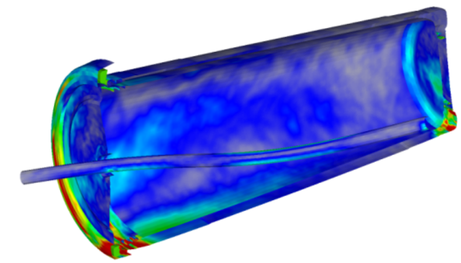
1. WHAT IS LS-DYNA?

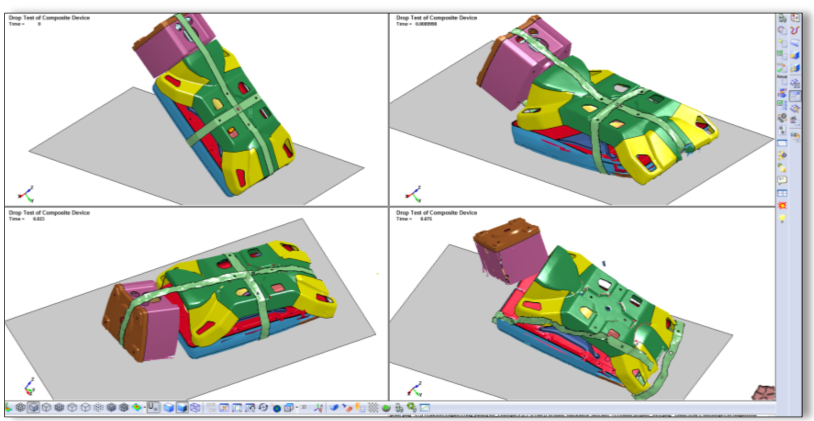
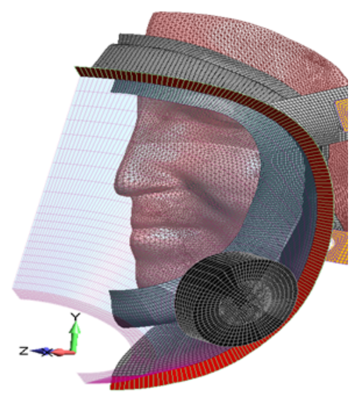
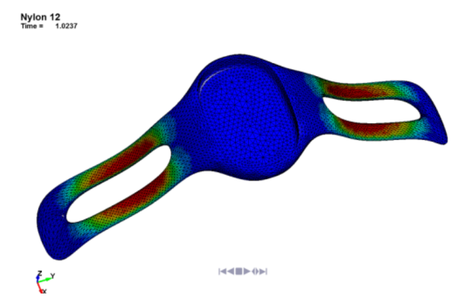
1.1 General Applications

Crashworthiness	Driver Impact	Train Collisions
		

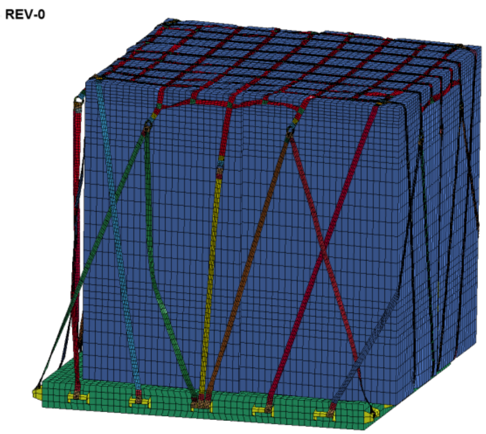
Earthquake Engineering	Metal Forming	Military
		

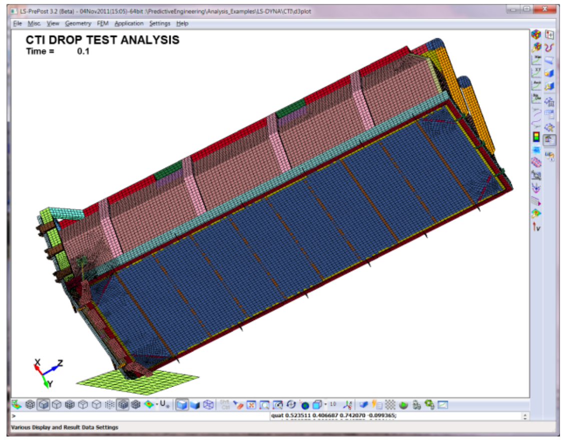
1.2 Specific Applications (Courtesy of Predictive Engineering)

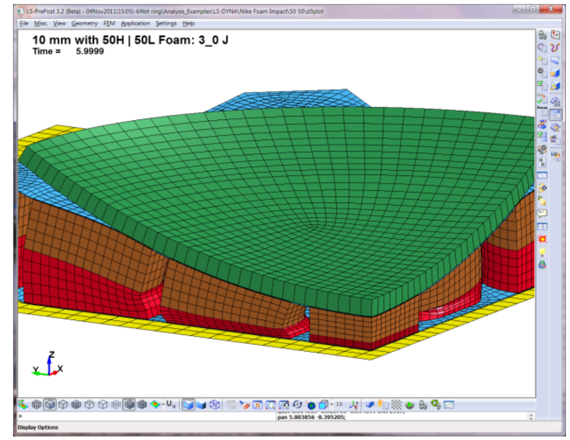
Airplane 16g Crash Analysis	Sporting Goods Equipment	Drop Test Consumer Products
		

Drop Test of Composites / Electronics	Human Biometrics	Large Deformation of Plastics
		

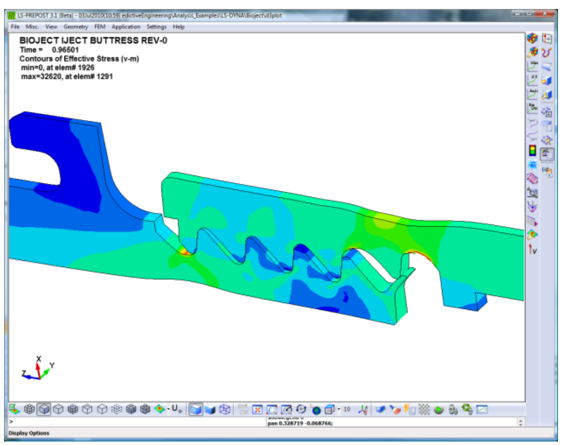
Crash Analysis of Cargo Net	Drop Test of Nuclear Waste Container	Impact Analysis of Foams

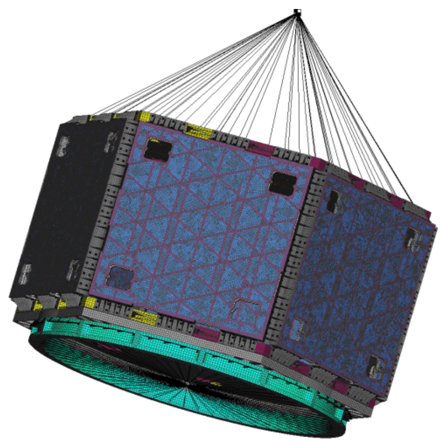


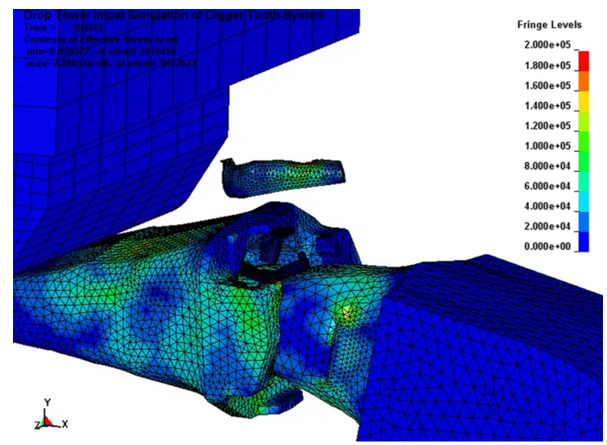




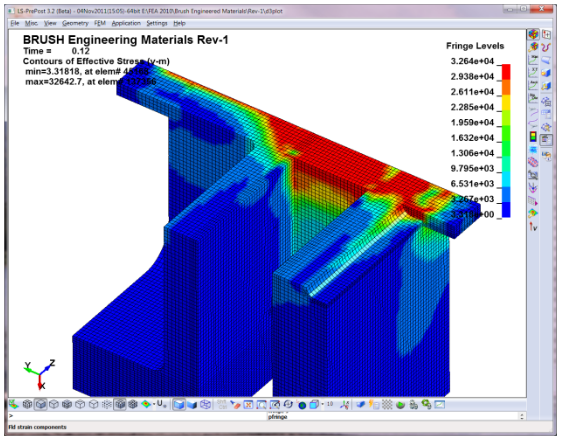
Plastic Thread Design	PSD / Modal Analysis	Digger Tooth Failure



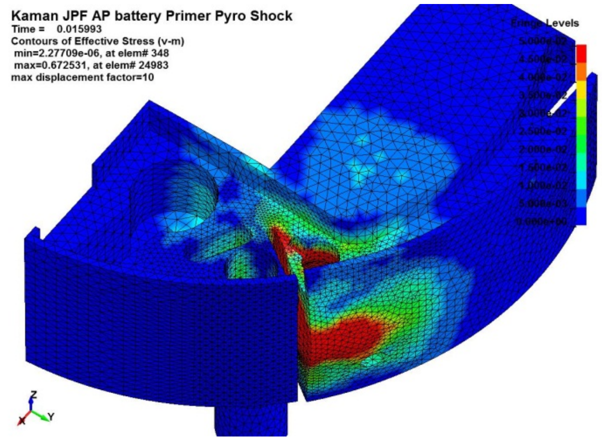




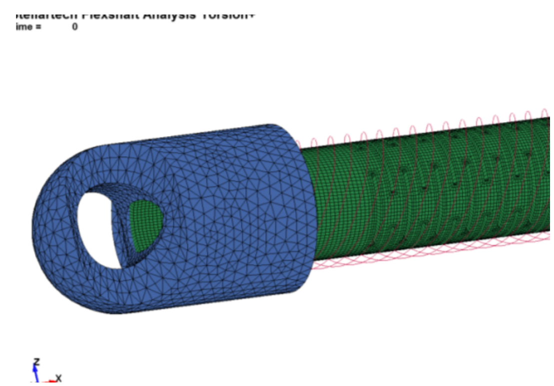
Electron Beam Welding



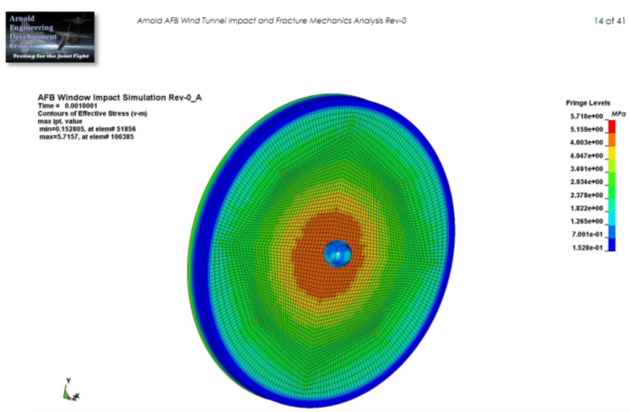
Pyro-Shock Analysis



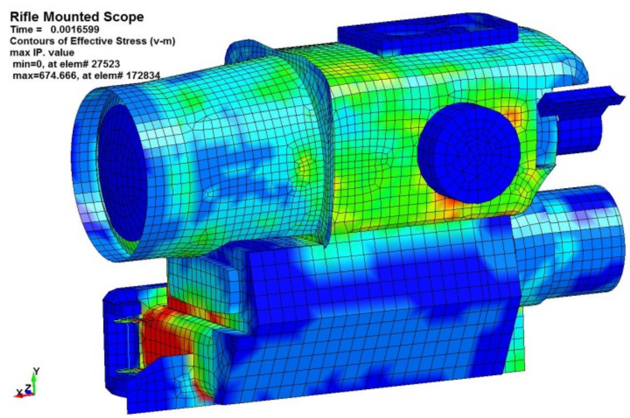
Medical Equipment



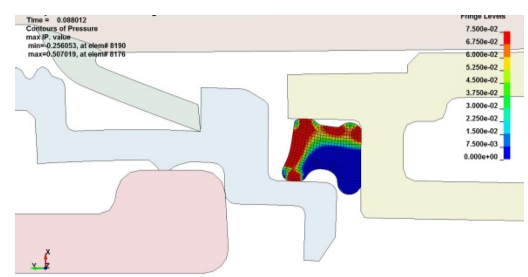
Fracture Mechanics of Glass



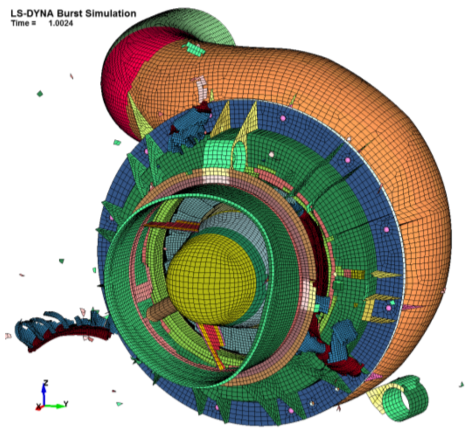
Ballistic Shock Loading of Optical Equipment

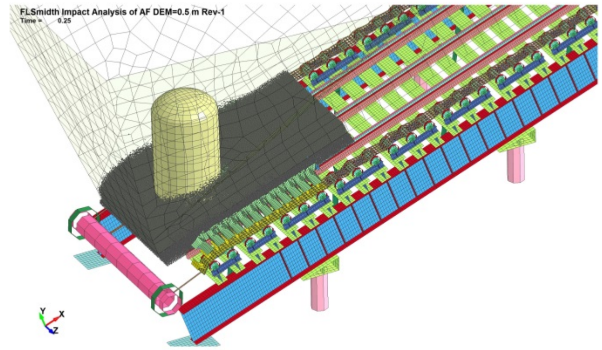


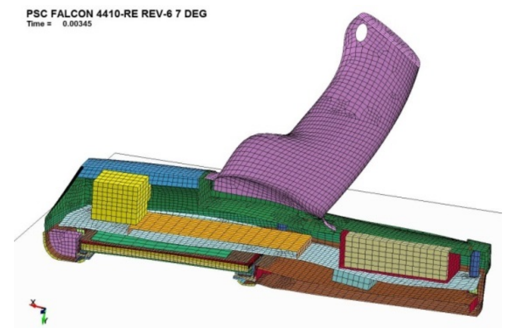
Hyperelastic Medical Seal Analysis



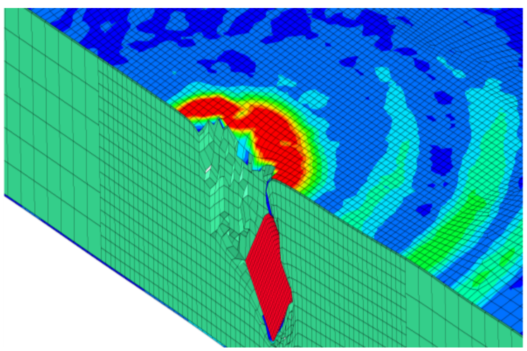
Blade-Out Analysis	Discrete Element Method (Mining Industry)	Drop-Test of Hand Held Electronics

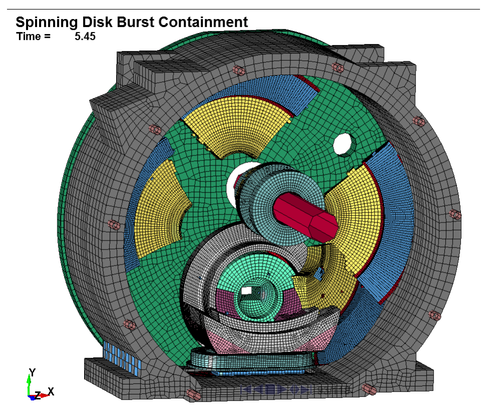


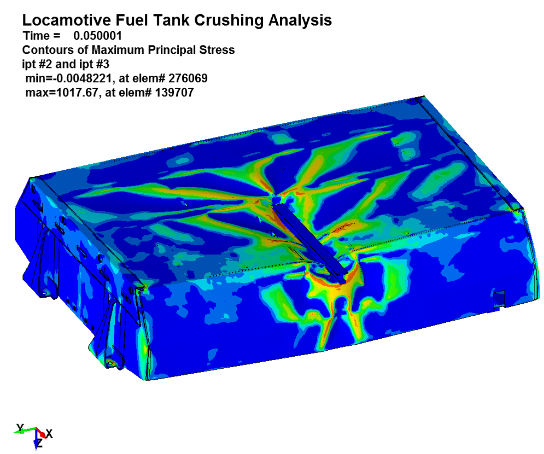


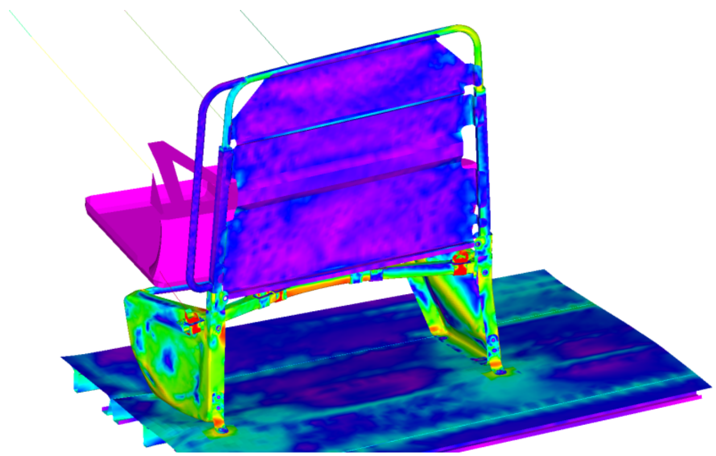
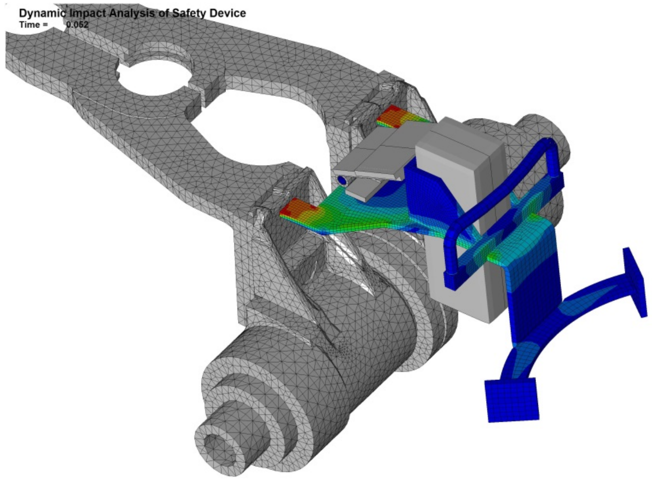


Ballistic Penetration of Al/Foam Panel	High-Speed Spinning Disk Containment	Locomotive Fuel Tank

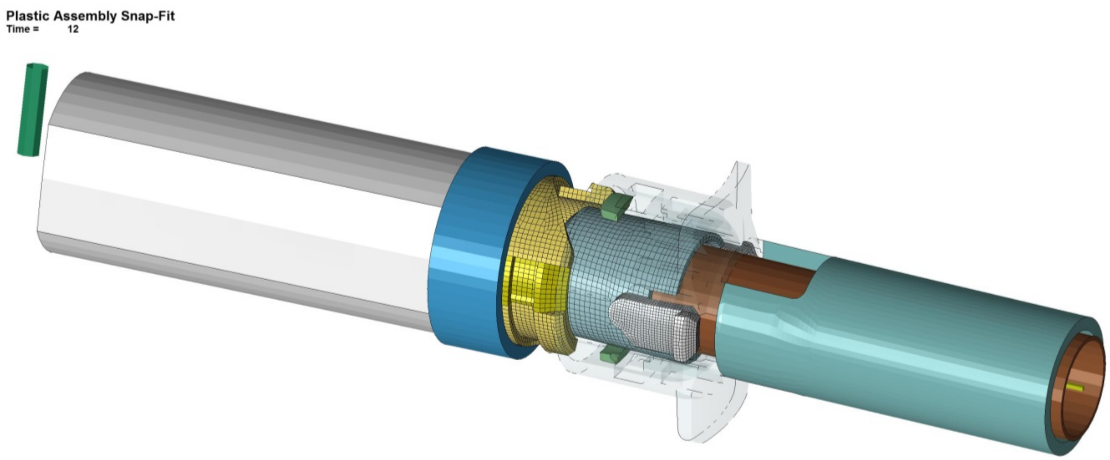




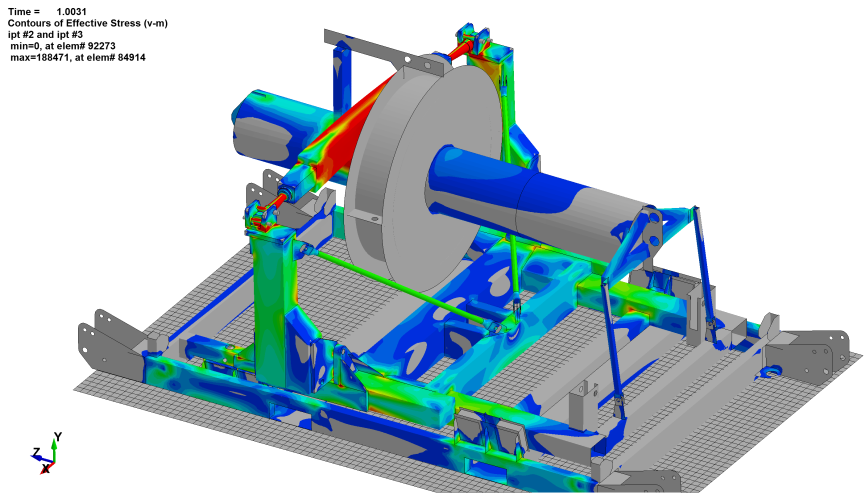


Crash Analysis of Bus Seats	Impact Analysis of Safety Block Device
	

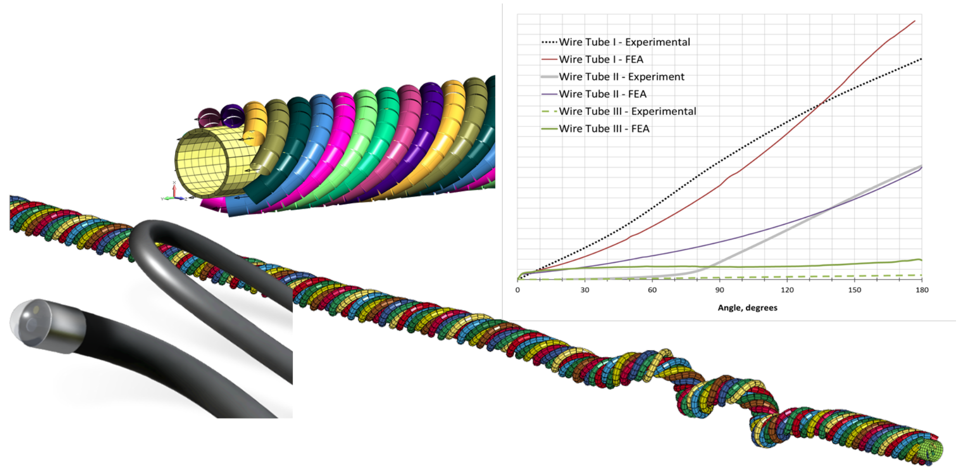
Snap-Fit Analysis – All Plastic Medical Device



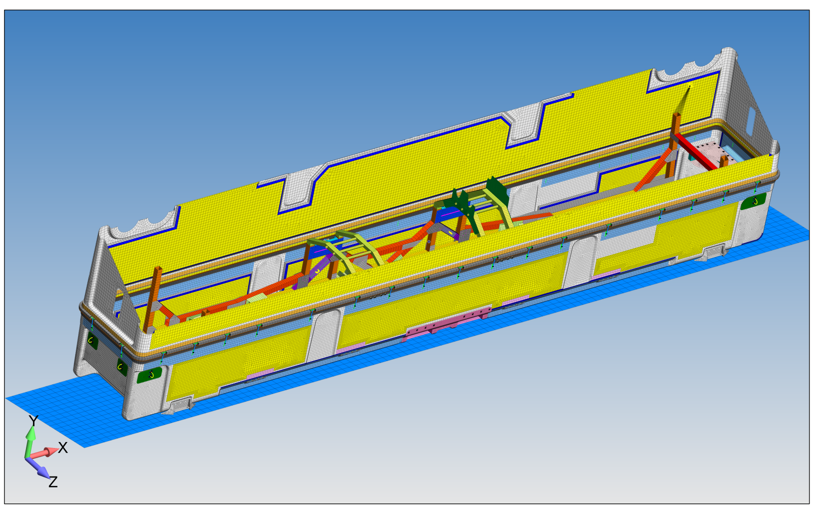
9g Crash Analysis of Jet Engine Stand

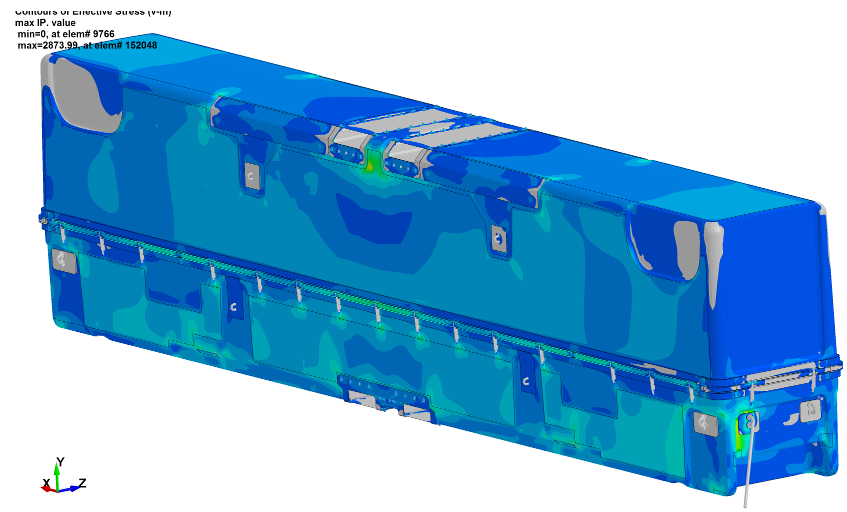


Torque Analysis of Endoscopic Medical Device



Drop, Rail Impact and PSD Analysis of Composite Container

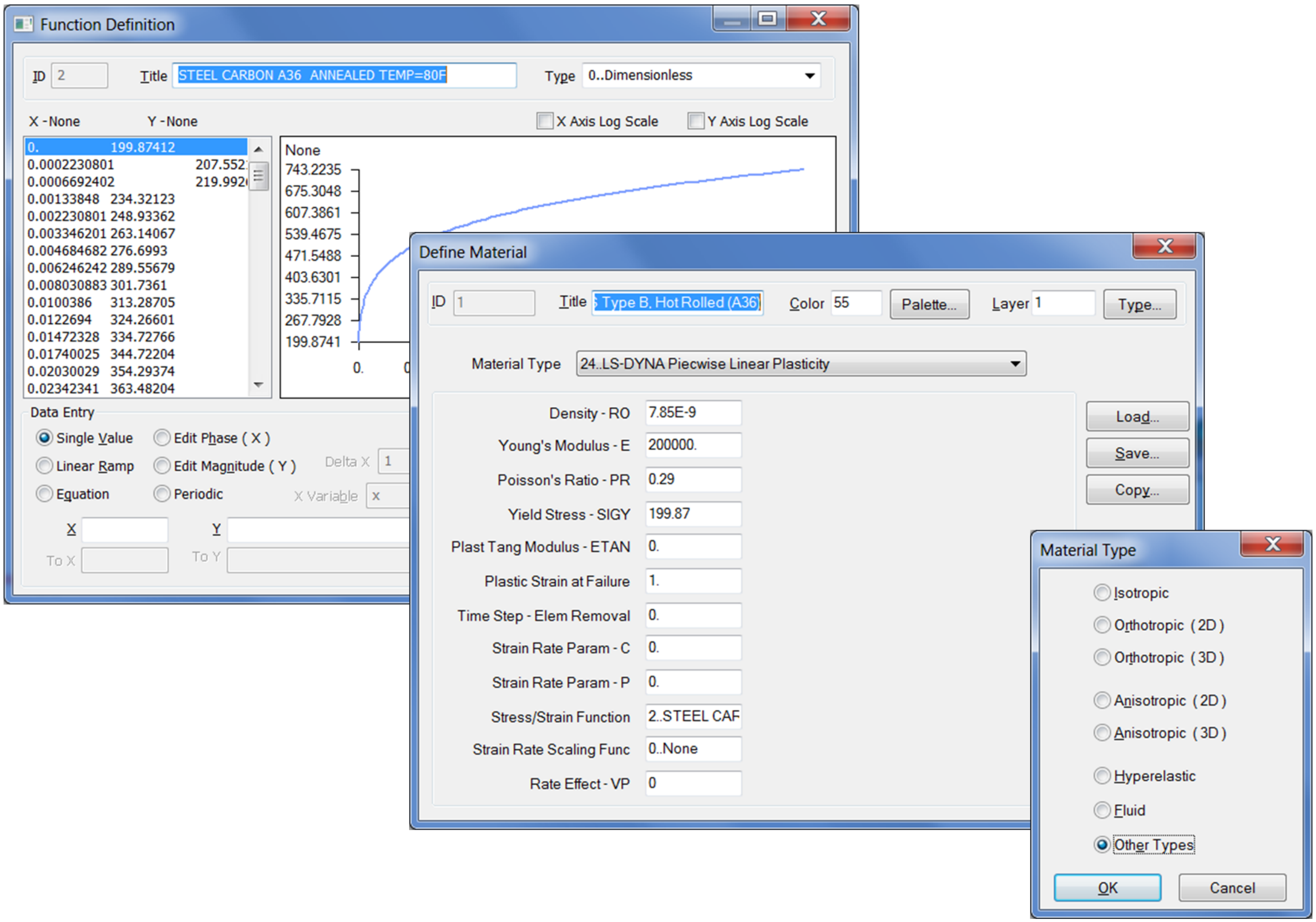




2. FEMAP – AN INTERFACE THAT OPENS ANALYSIS POSSIBILITIES ACROSS MULTIPLE PLATFORMS

Let's build a simple model. There are a few key steps that are necessary to get your model up-and-running in LS-DYNA.

2.1 MATERIALS



2.2 MESHING

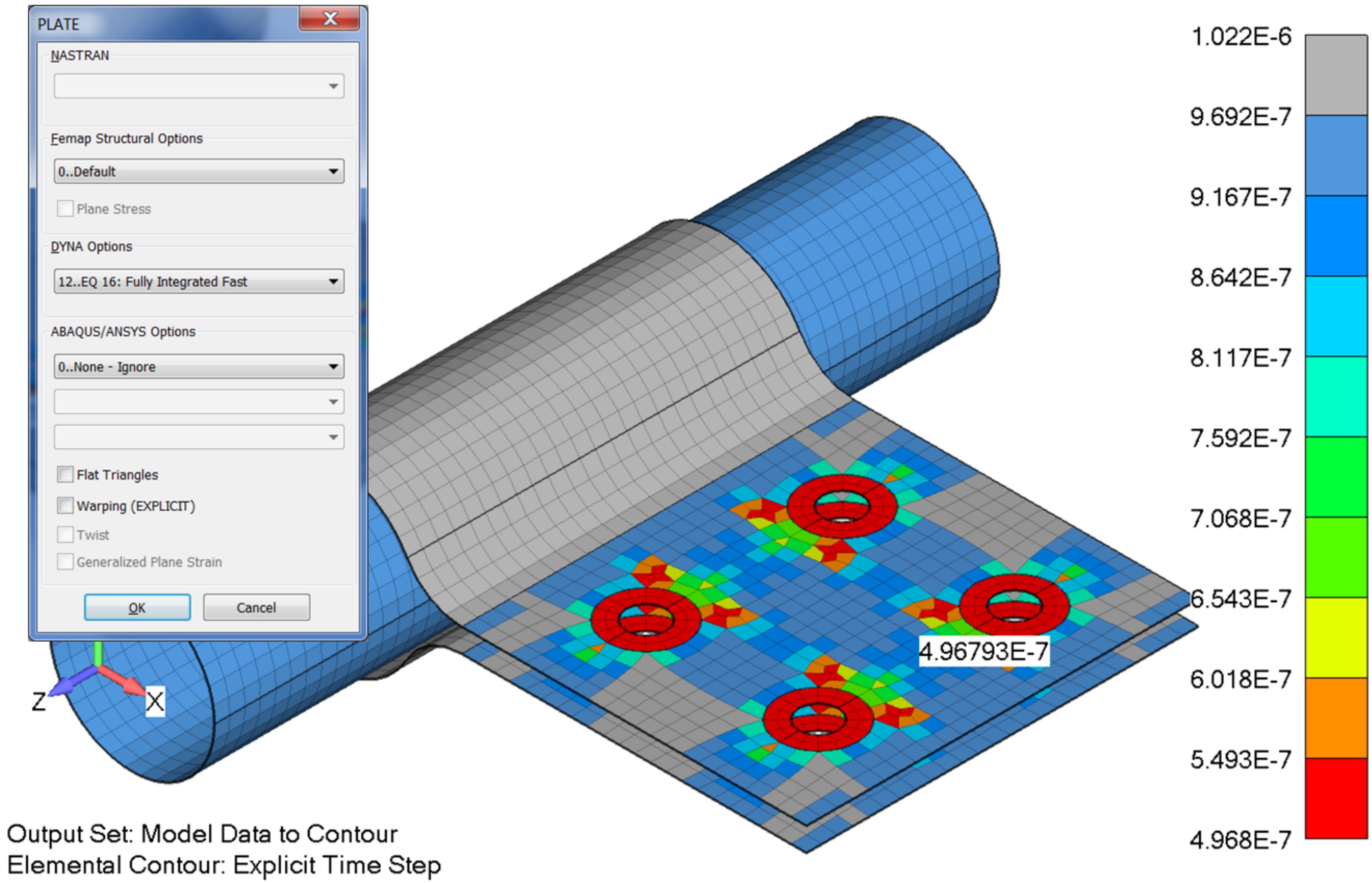
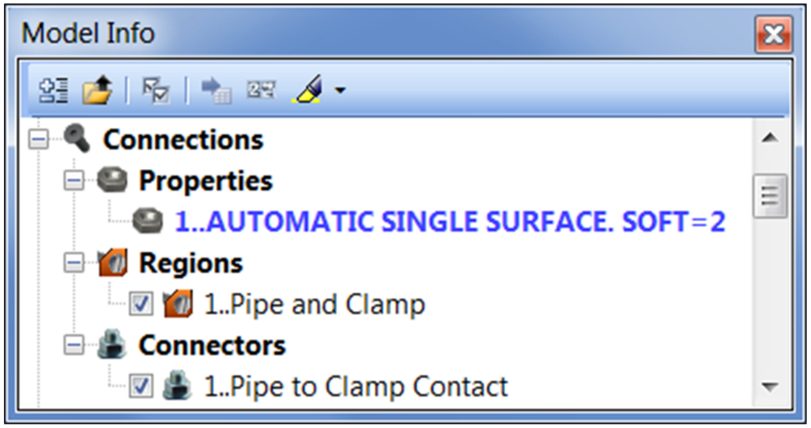
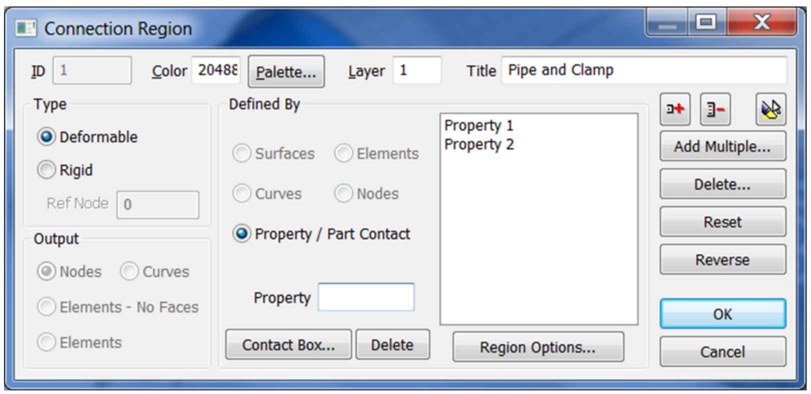
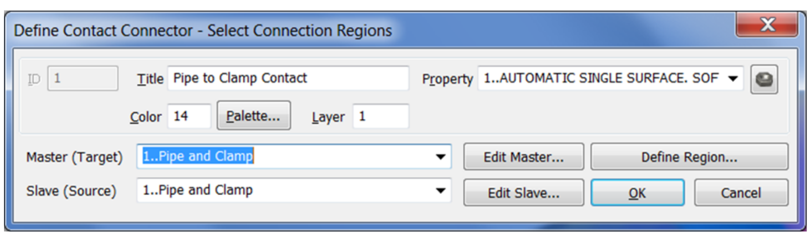


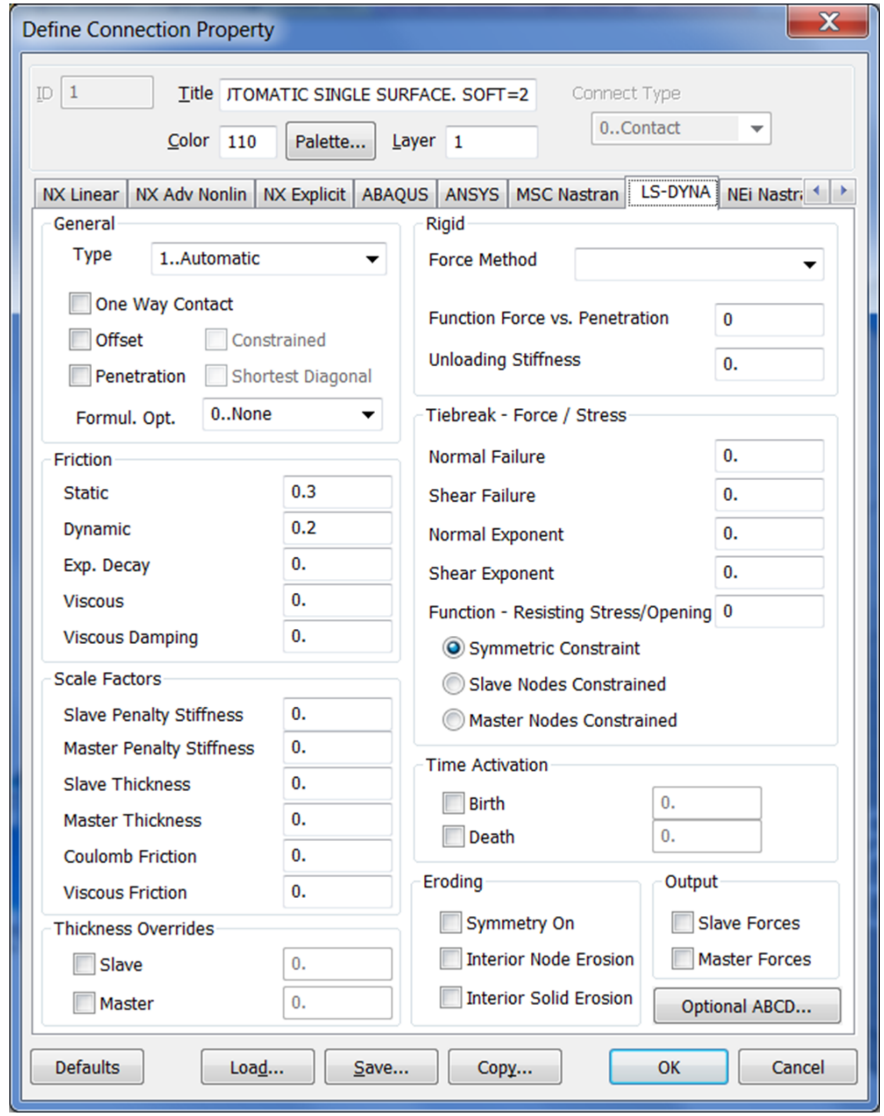
PLATE
NASTRAN
Femap Structural Options
0..Default
Plane Stress
DYNA Options
12..EQ 16: Fully Integrated Fast
ABAQUS/ANSYS Options
0..None - Ignore
Flat Triangles
Warping (EXPLICIT)
Twist
Generalized Plane Strain
OK
Cancel
4.96793E-7
1.022E-6
9.692E-7
9.167E-7
8.642E-7
8.117E-7
7.592E-7
7.068E-7
6.543E-7
6.018E-7
5.493E-7
4.968E-7
Z
X
Output Set: Model Data to Contour
Elemental Contour: Explicit Time Step

2.3 CONTACT

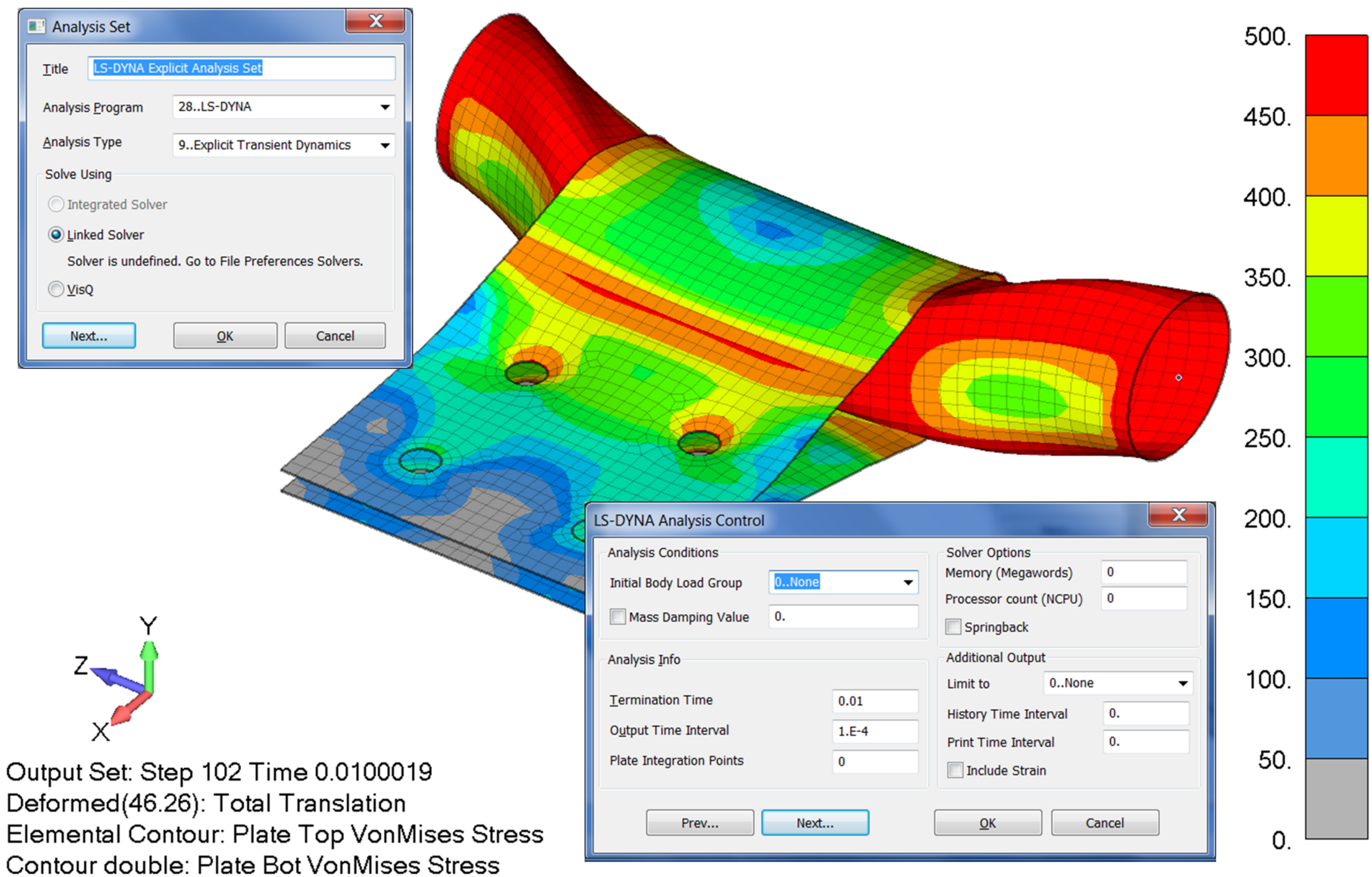

Model Info
Connections
Properties
1..AUTOMATIC SINGLE SURFACE. SOFT=2
Regions
1..Pipe and Clamp
Connectors
1..Pipe to Clamp Contact


Connection Region
ID 1
Color 20488
Palette...
Layer 1
Title Pipe and Clamp
Type
Deformable
Rigid
Ref Node 0
Defined By
Surfaces
Elements
Curves
Nodes
Property / Part Contact
Property
Property 1
Property 2
Add Multiple...
Delete...
Reset
Reverse
OK
Cancel
Output
Nodes
Curves
Elements - No Faces
Elements
Contact Box...
Delete
Region Options...


Define Contact Connector - Select Connection Regions
ID 1
Title Pipe to Clamp Contact
Property 1..AUTOMATIC SINGLE SURFACE. SOF
Color 14
Palette...
Layer 1
Master (Target) 1..Pipe and Clamp
Edit Master...
Define Region...
Slave (Source) 1..Pipe and Clamp
Edit Slave...
OK
Cancel


Define Connection Property
ID 1
Title JTOMATIC SINGLE SURFACE. SOFT=2
Connect Type
0..Contact
Color 110
Palette...
Layer 1
NX Linear
NX Adv Nonlin
NX Explicit
ABAQUS
ANSYS
MSC Nastran
LS-DYNA
NEi Nastra
General
Type 1..Automatic
One Way Contact
Offset
Constrained
Penetration
Shortest Diagonal
Formul. Opt. 0..None
Friction
Static 0.3
Dynamic 0.2
Exp. Decay 0.
Viscous 0.
Viscous Damping 0.
Scale Factors
Slave Penalty Stiffness 0.
Master Penalty Stiffness 0.
Slave Thickness 0.
Master Thickness 0.
Coulomb Friction 0.
Viscous Friction 0.
Thickness Overrides
Slave 0.
Master 0.
Rigid
Force Method
Function Force vs. Penetration 0
Unloading Stiffness 0.
Tiebreak - Force / Stress
Normal Failure 0.
Shear Failure 0.
Normal Exponent 0.
Shear Exponent 0.
Function - Resisting Stress/Opening 0
Symmetric Constraint
Slave Nodes Constrained
Master Nodes Constrained
Time Activation
Birth 0.
Death 0.
Eroding
Symmetry On
Interior Node Erosion
Interior Solid Erosion
Output
Slave Forces
Master Forces
Optional ABCD...
Defaults
Load...
Save...
Copy...
OK
Cancel

2.4 ANALYSIS MANAGER

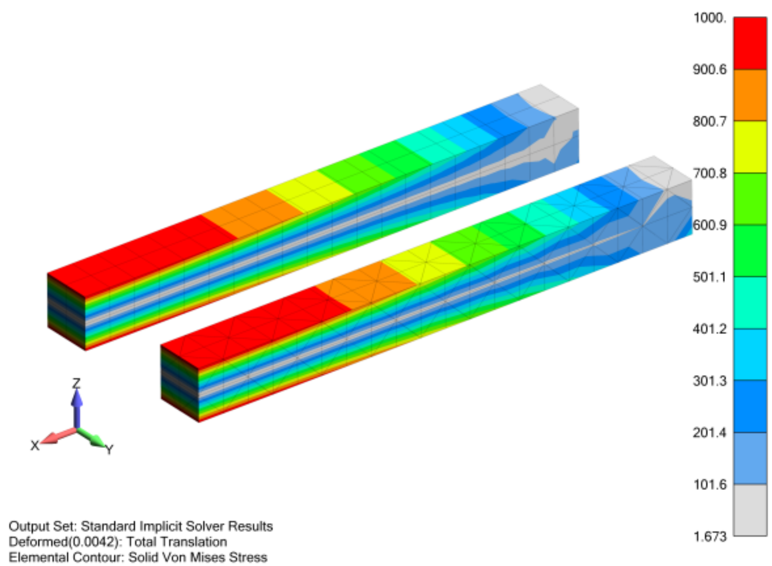


3. FOUNDATIONS OF FINITE ELEMENT ANALYSIS

3.1 Simply-Supported Beam

A simply-supported half-symmetric beam is analyzed using beam, shell and solid (8-node brick and 10-node tetrahedral) elements. This basic example is used to demonstrate that LS-DYNA can solve the most basic of linear elastic problems.

Standard FEA Implicit Code (Nastran)	LS-DYNA Implicit Analysis



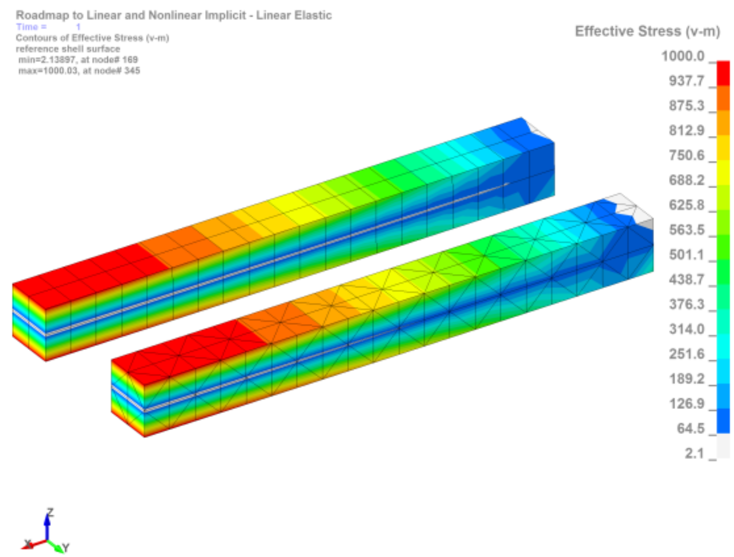


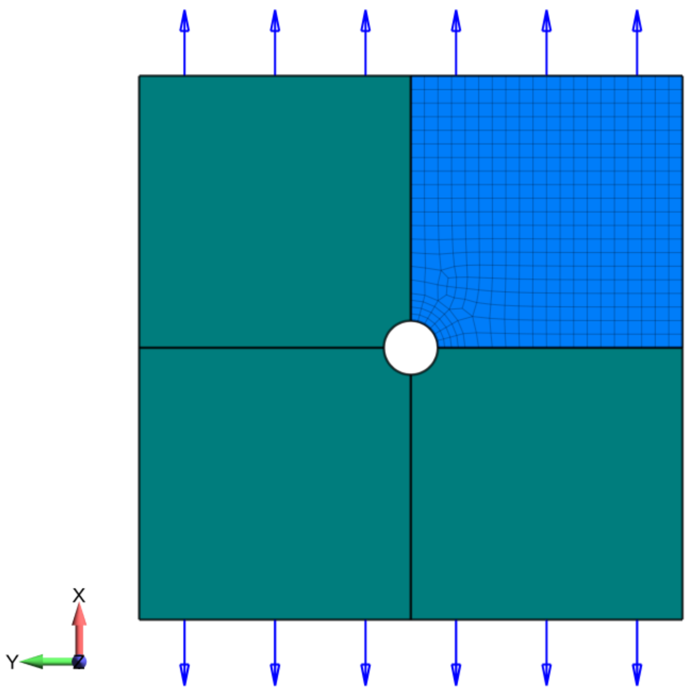
Table 1: Summary of linear elastic implicit verification results

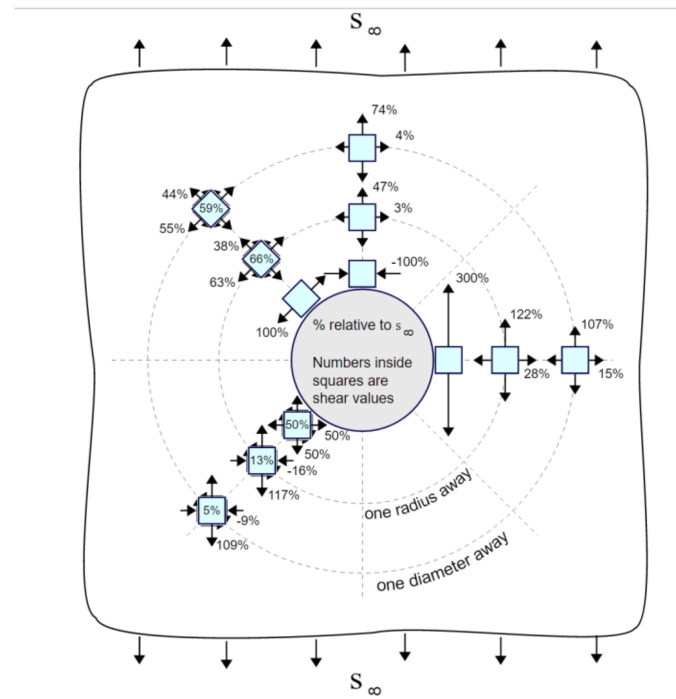
Model	Hex		10-node Tet		Shell		Beam	
	Stress	Disp	Stress	Disp	Stress	Disp	Stress	Disp
Standard	999.0	4.185e-3	1000.	4.194e-3	999.1	4.192e-3	1000.	4.190e-3
LS-DYNA	999.3	4.184e-3	1000.	4.192e-3	999.0	4.192e-3	1000.	4.192e-3
% Difference	0.03	0.02	0.00	0.05	0.01	0.00	0.00	0.05

3.2 PLATE WITH A HOLE

For a uniformly loaded, infinite plate with a hole, the maximum stress concentration is 3x the far field stress.

Plate with Hole Under Uniform Tension	Courtesy of www.fracturemechanics.org





The following graphics show stress results for the solid and shell element formulations. As in the prior example, the same meshes were used between the two programs. In contouring solid element stresses within LS-PrePost, several options are available for averaging nodal stresses: mid, ave and max. The mid option takes a simple average between connected nodes and was used in the hex and tet models. For the shell model, extrapolate 1 was used within LS-PrePost to extrapolate the stresses from the integration points and then averaged using the default setting (mid).

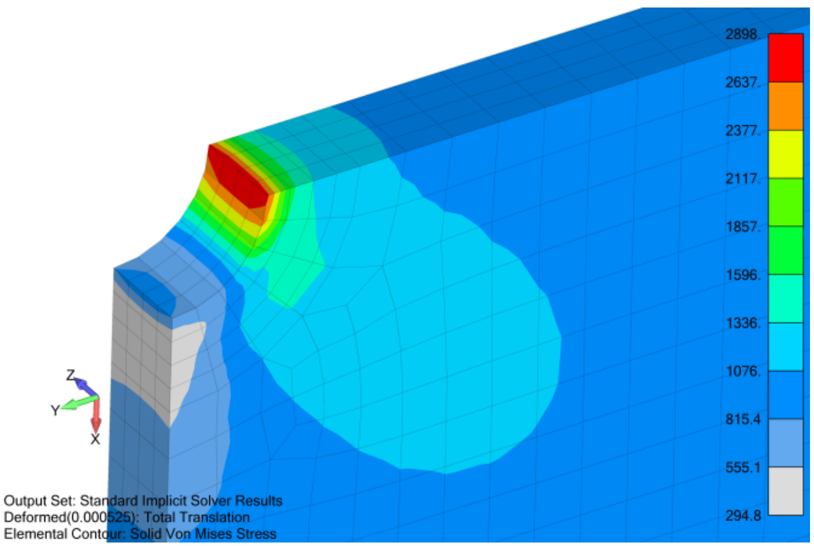
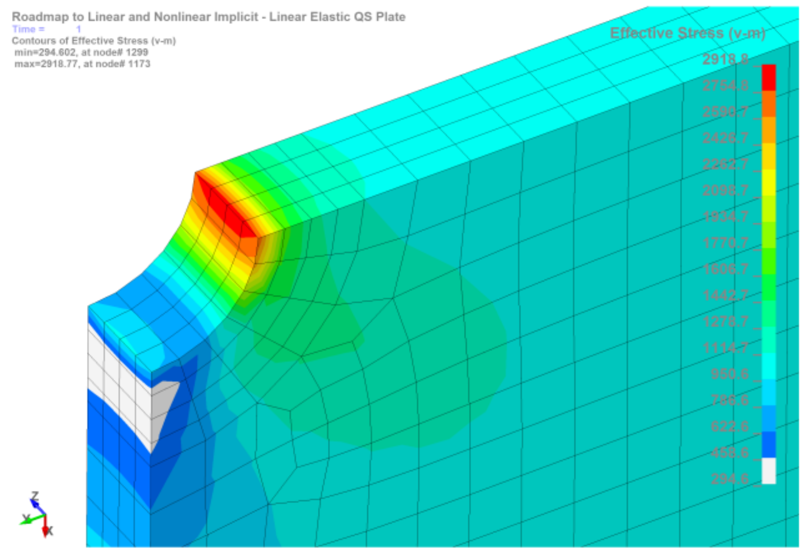
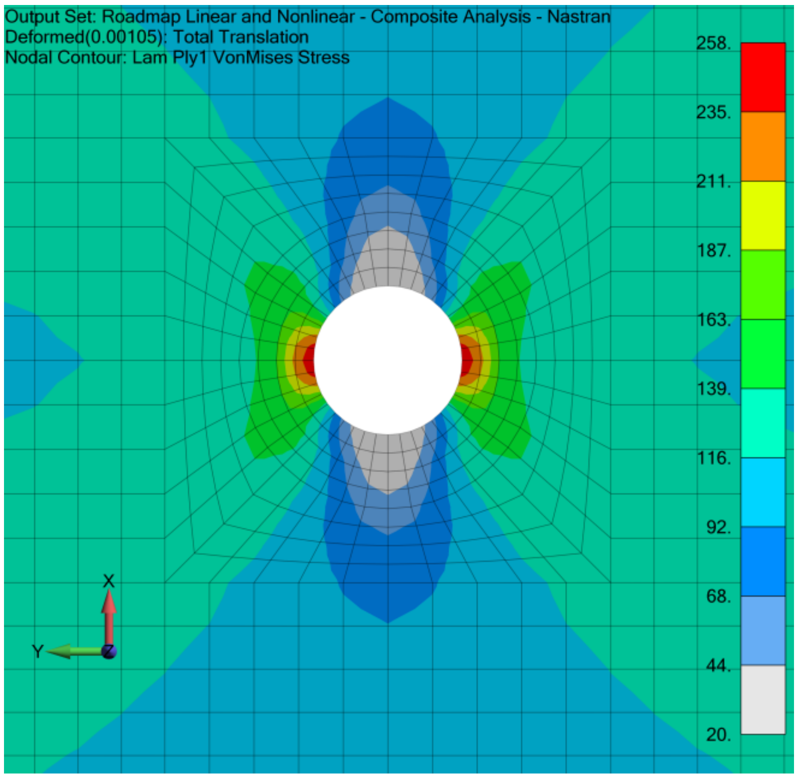
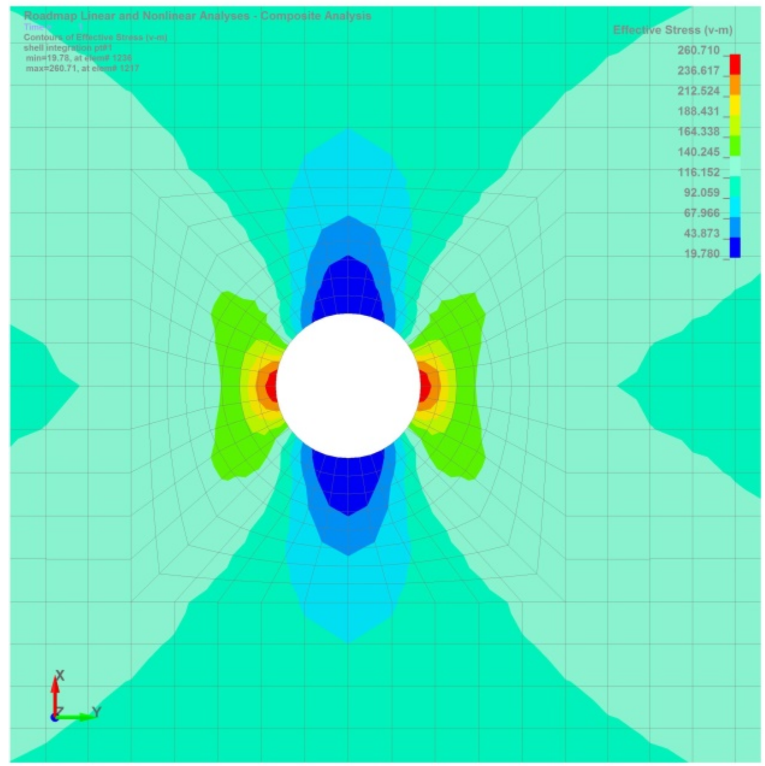
Standard FEA Implicit Code (Nastran) - Hex	LS-DYNA Implicit Analysis - Hex
	

Table 2: Summary of linear elastic results for QS plate with hole

	Hex	Tet	Shell
Standard	2898	3063	2865
LS-DYNA	2919	3063	2865
% Difference	0.72	0.00	0.00

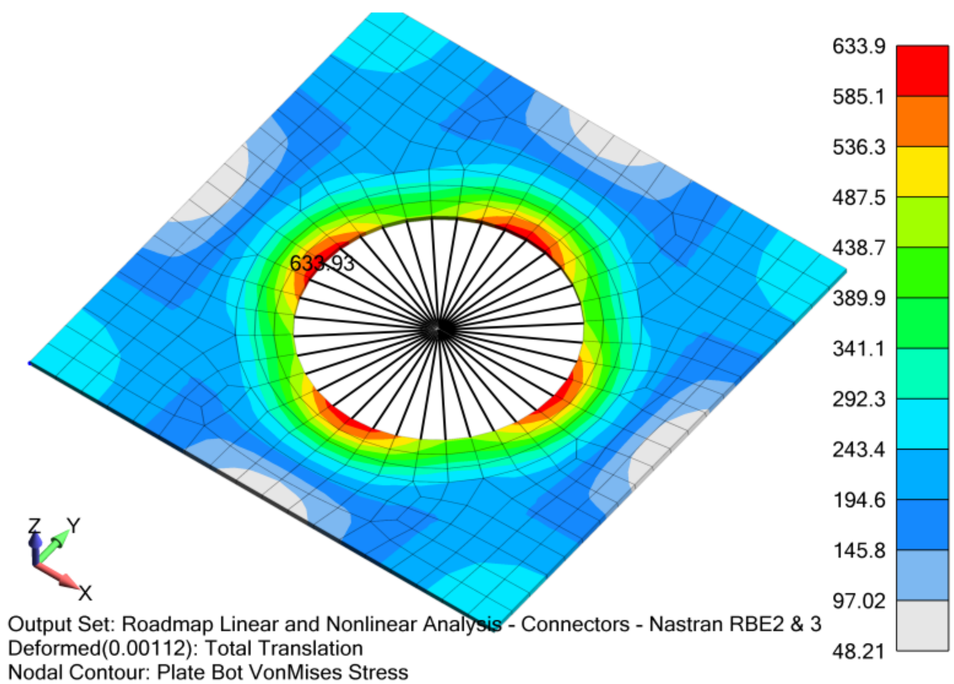
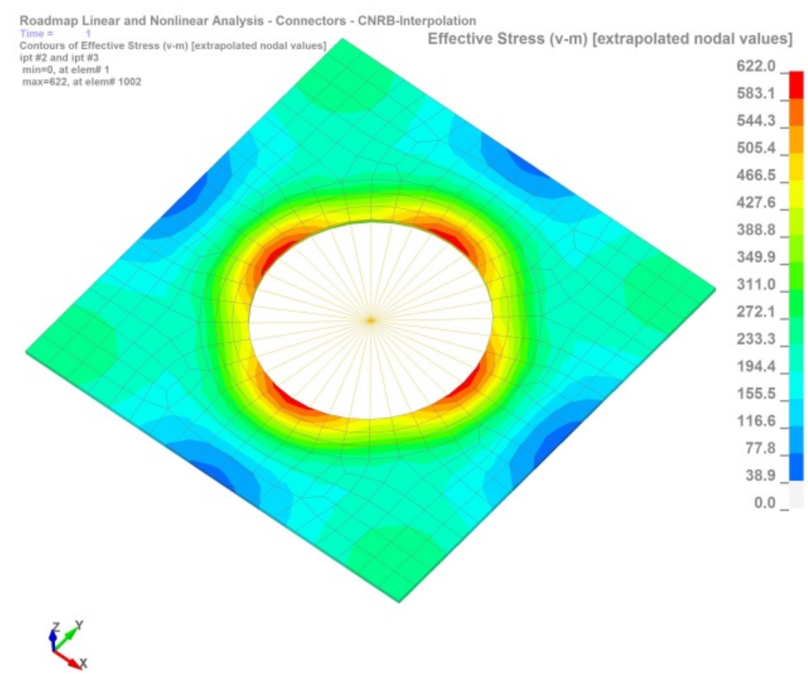
3.3 COMPOSITE ANALYSIS

The images below provide a comparison between the first plies of an eight ply laminate composite plate with a hole. The analysis is linear elastic. For the LS-DYNA model, the shell formulation is ELFORM=-16 (minus sign 16). To request ply information, use the *DATABASE_EXTENT_BINARY setting maxint=8 to write out integration point data for each ply. The reason for not requesting all integration points on each layer (i.e., ply) using -8 is that Nastran only reports the centroid value as a default and we don't wish to make this comparison more difficult than necessary.

Nastran – Ply 1	LS-DYNA Ply 1
	

3.4 Linear Connectors (Equivalent Nastran Multi-Point Constraint Elements)

In Nastran implicit analysis, it is quite common to use connectors that are based on constraint relationships between stiffness terms within the stiffness matrix. In Nastran they are termed multi-point constraint elements (MPC's) and depending upon their formulation are also known as rigid elements (e.g., RBE1 and RBE2) or force interpolation elements (e.g., RBE3).

MPC – RBE2 (Rigid 6 DOF's)	LS-DYNA CNRB (Rigid 6 DOF)
	

4. WHAT ABOUT NOT-SO-SIMPLE MODELS?

4.1 COMPLEX CONSTRUCTION FOR COMPLEX LOADING

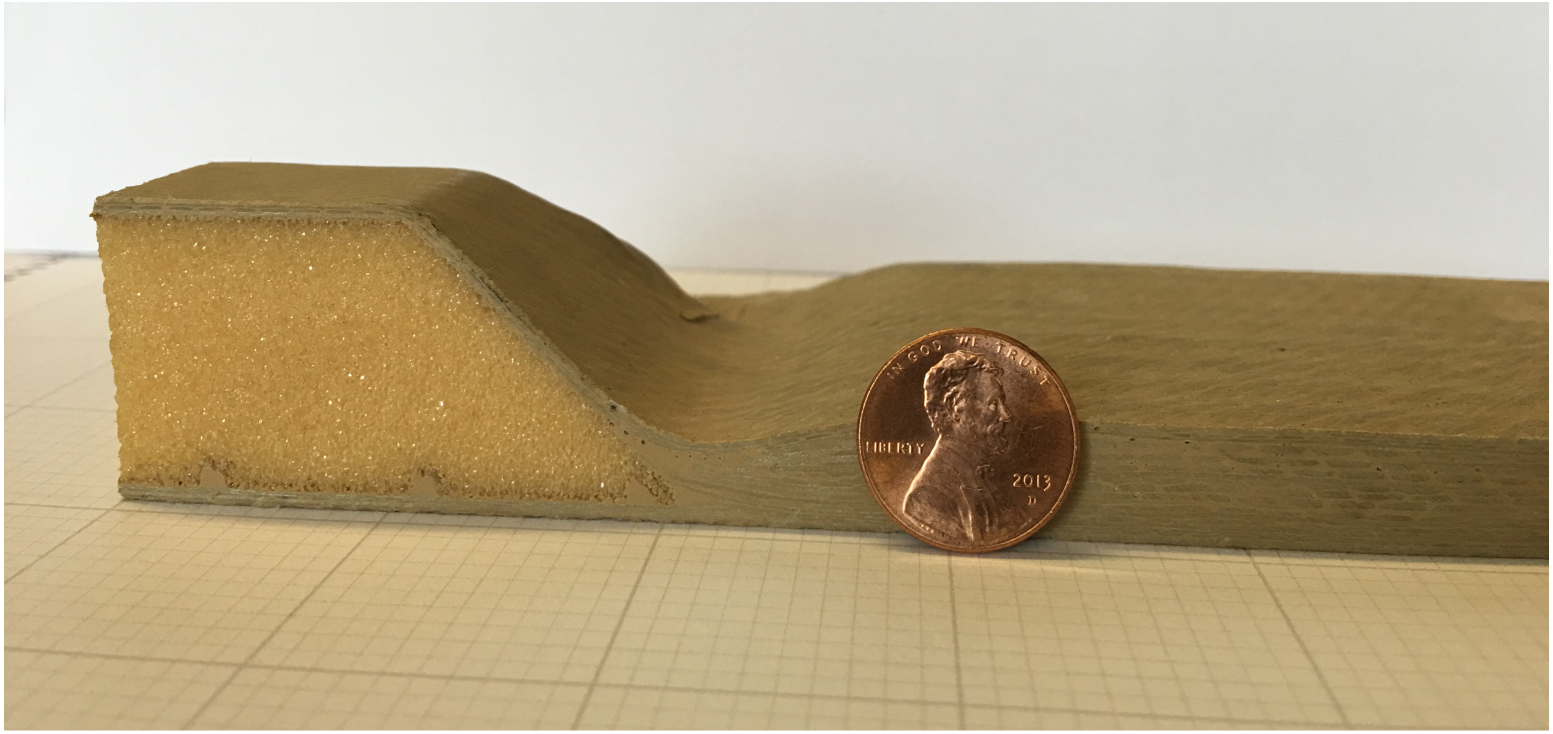
A large composite shipping container was analyzed for drop, impact, PSD random vibration and general stress analysis. The main shell of the container was a glass-fiber vacuum infused composite with closures made of aluminum. Lifting rings and other major structural load points were attached to the composite container using thick aluminum plates with preloaded bolts to distribute point loads into the shell.

Example of Large Composite Transportation Container

4.2 COMPOSITE-LAMINATE MATERIAL MODELING

The uniqueness of this work was that one base model could address progressive composite failure whether under static conditions (implicit) or during drop test analysis (explicit) along with bolt preload and extensive nonlinear contact behavior at closures, skid plates and load rings. Analysis recommendations are provided for general implicit analysis for: (i) PSD random vibration with bolt preload; (ii) progressive failure of composites with *MAT_54; (iii) contact modeling and (iv) optimization of run times using MPP LS-DYNA. The explicit analysis of the container was rather simplistic but some comments will be made about the analysis setup and runtimes.

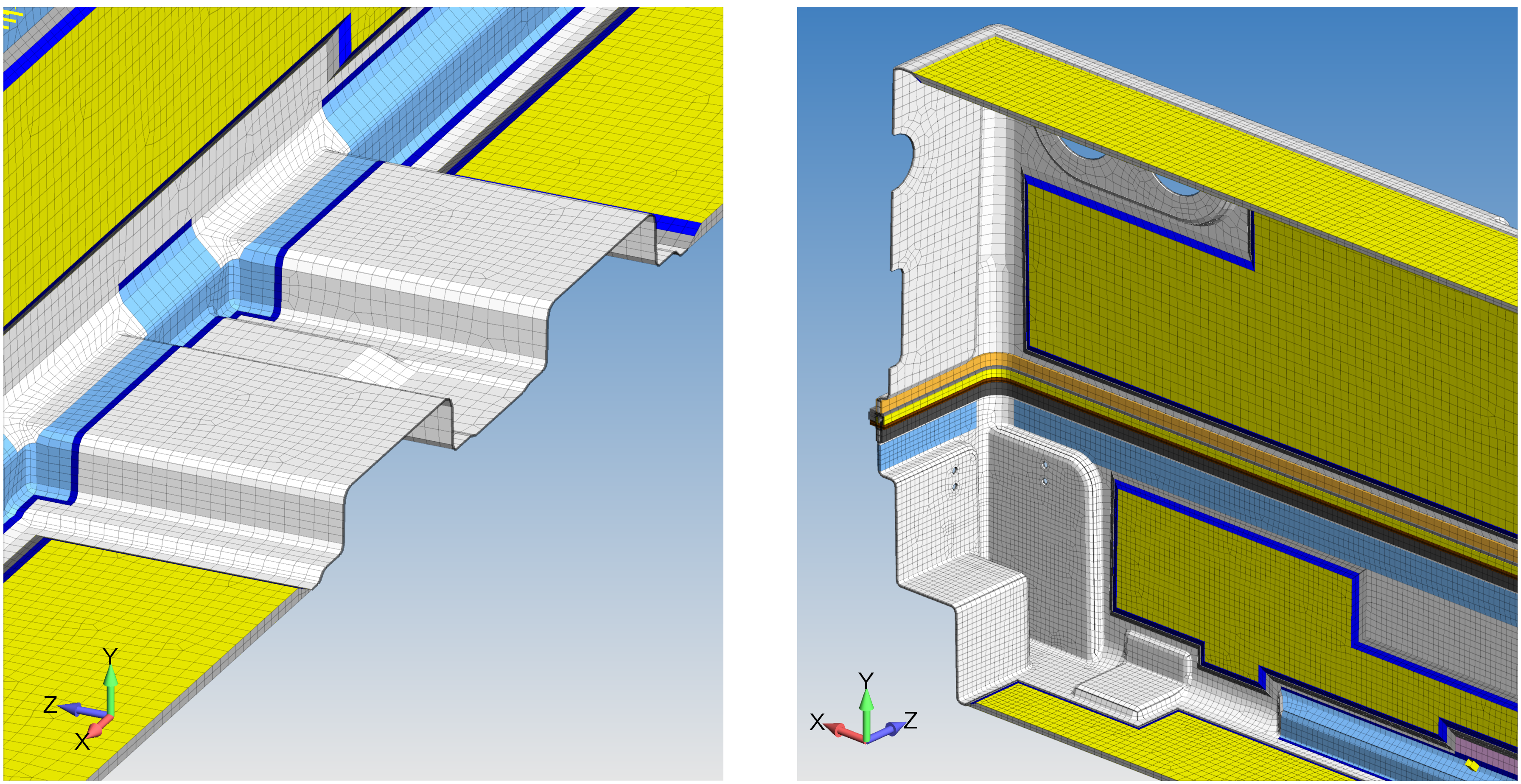
Sandwich and Solid Laminate Composites with Transition Region



4.3 Sandwich Panel - Surface and Solid Meshing

Given that the model was going to be run in explicit, the aimed time step was 1 µs. With a nominal composite wave speed of 3500 m/s, element sizes could be as small as 3.5 mm if needed, but in general, the mesh sizing was set to 25 mm. The model contains 150k elements and 125k nodes.

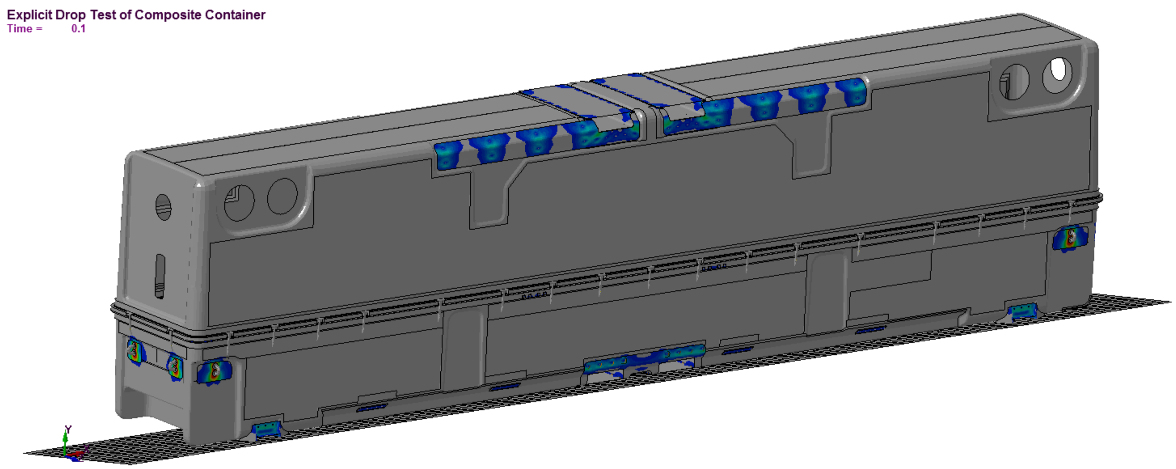
Sandwich and Solid Laminate Composites with Transition Region

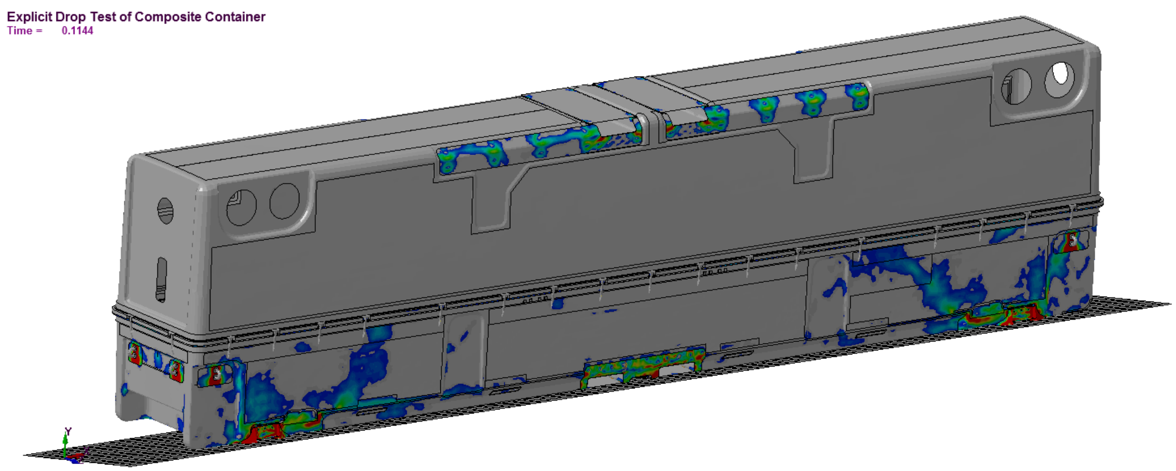


4.4 Multi-Step Analysis

The top image shows the preload state prior to initial velocity initialization while the second image shows the impact stresses. Scaling is as with prior work.

Bolt Preload Following by Impact (Implicit/Explicit)



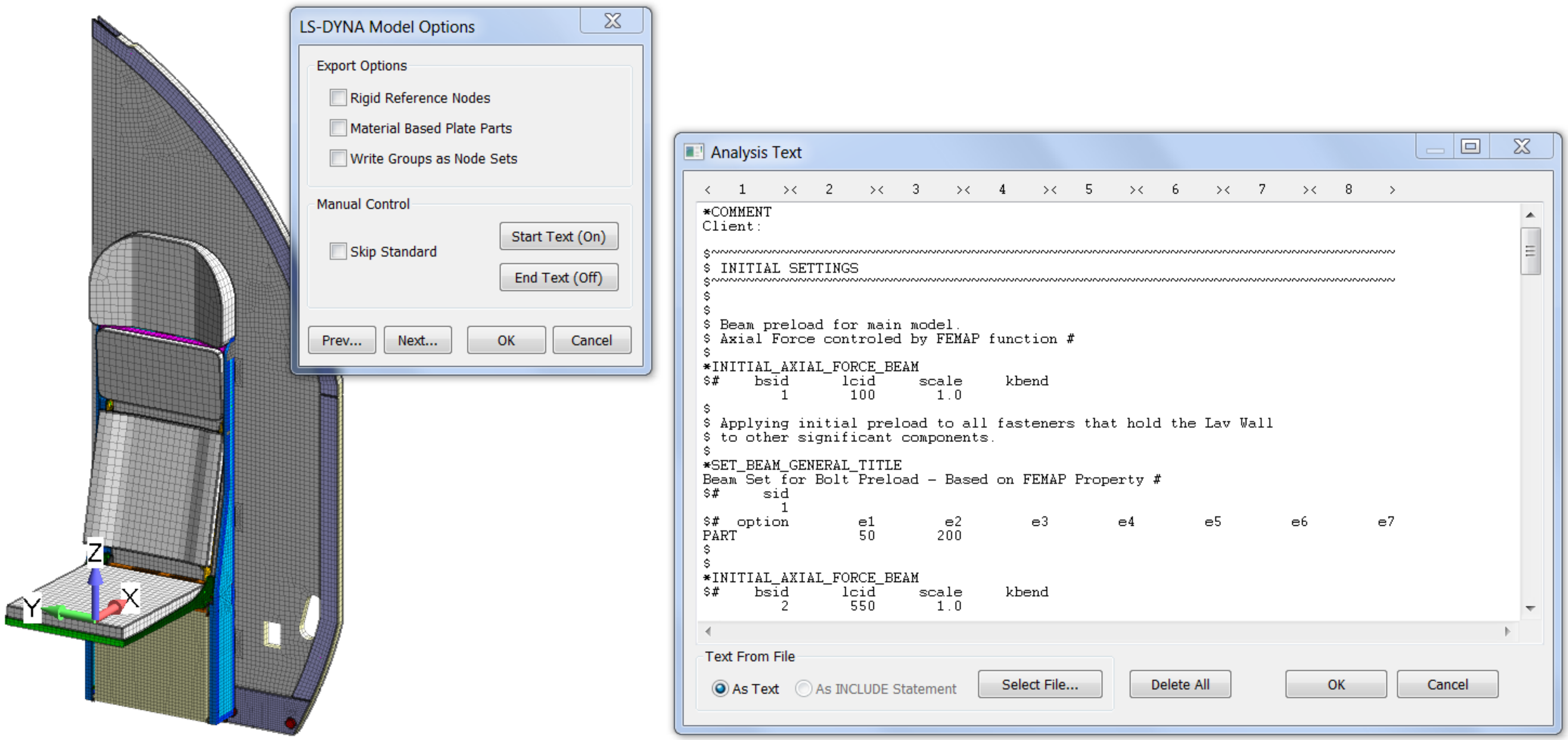


5. FEMAP AEROSPACE FROM INTERIORS TO SLED TESTING

5.1 EVEN WHEN IT'S NOT AVAILABLE, IT'S STILL AVAILABLE IN FEMAP

It's a tall order to ask that a pre and post processor keeps 100% up-to-date with all the developments for a portfolio of solvers. For features not yet incorporated into the Femap GUI, the ability to add analysis text allows one to access even more LS-DYNA features.

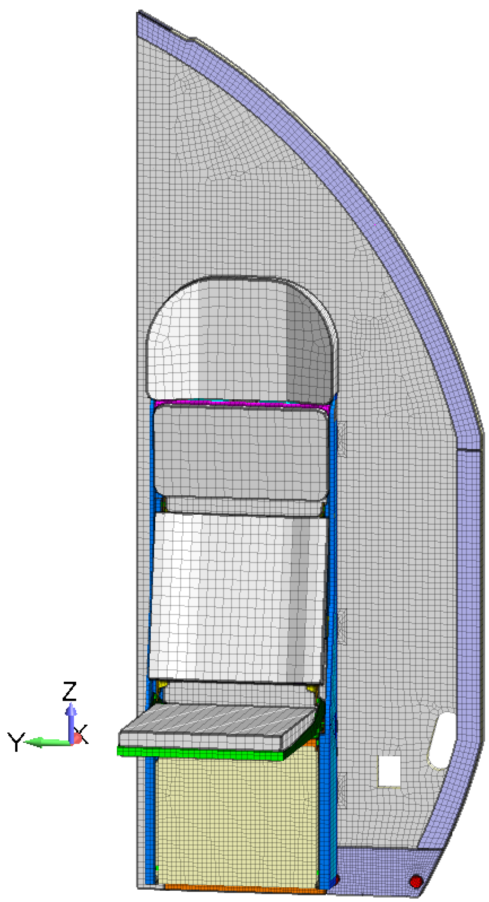
Analysis Text to Incorporate Additional Analysis Control

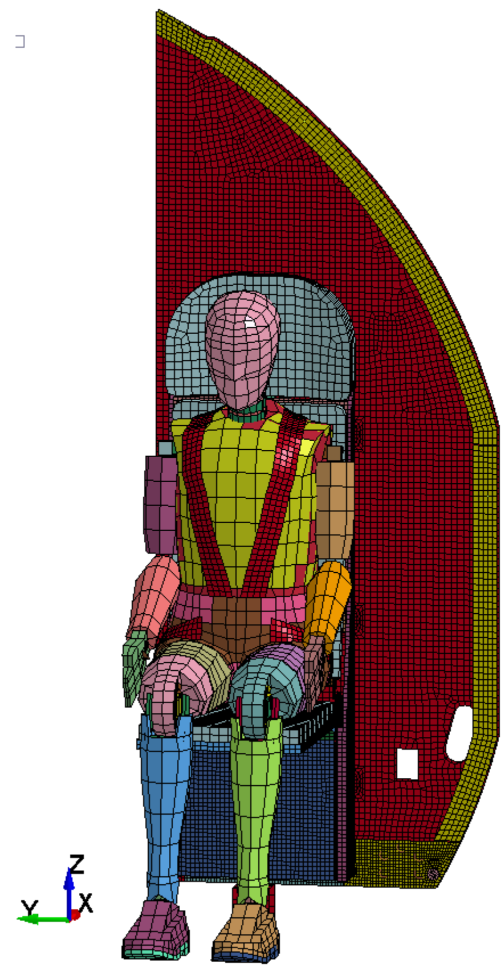


5.2 EXPORT AND READY TO RUN

For even more flexibility in analysis file management, the analyst can use the *INCLUDE keyword to point the analysis manager to external files. This is especially well when working with complicated 3rd party models, like an ATD.

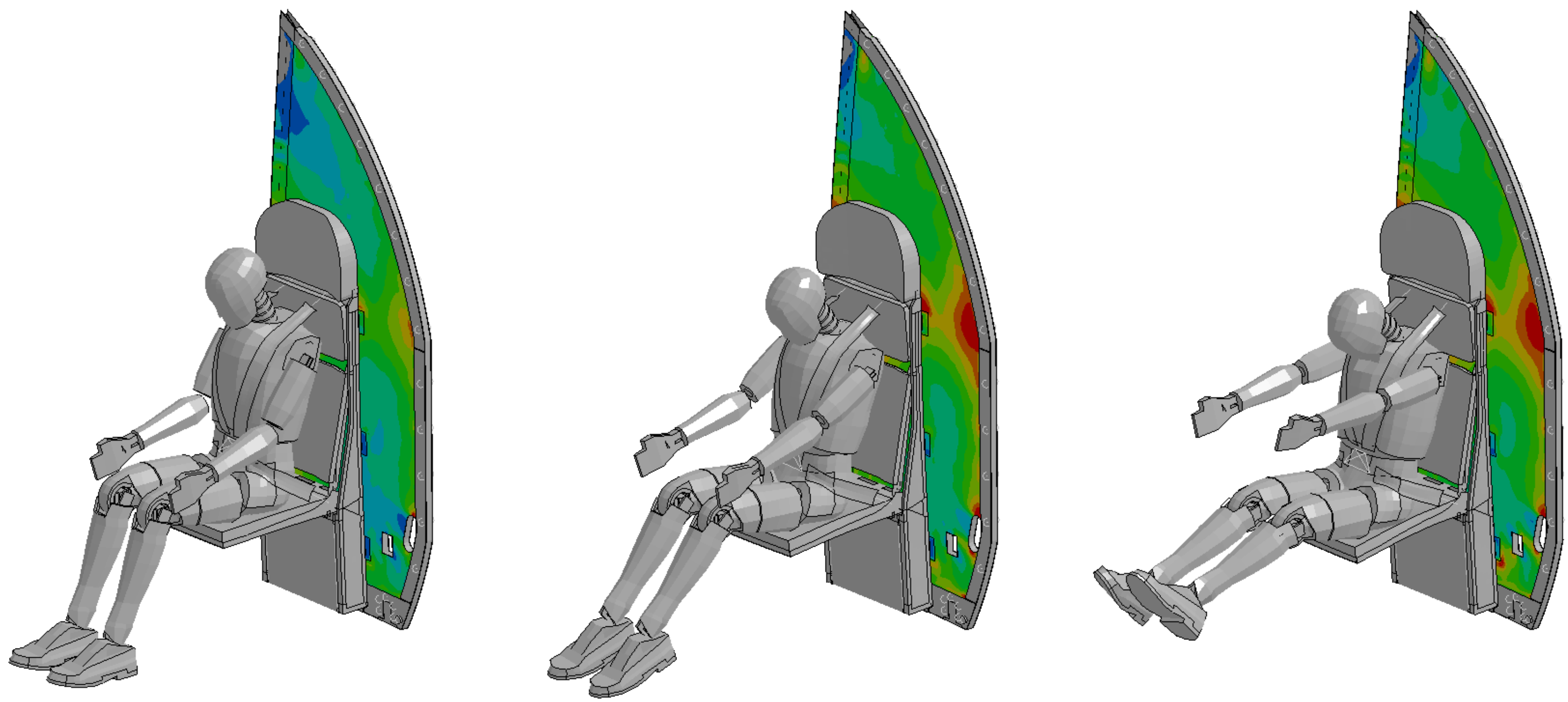
Using *INLCUDE to Integrate External Analysis Files





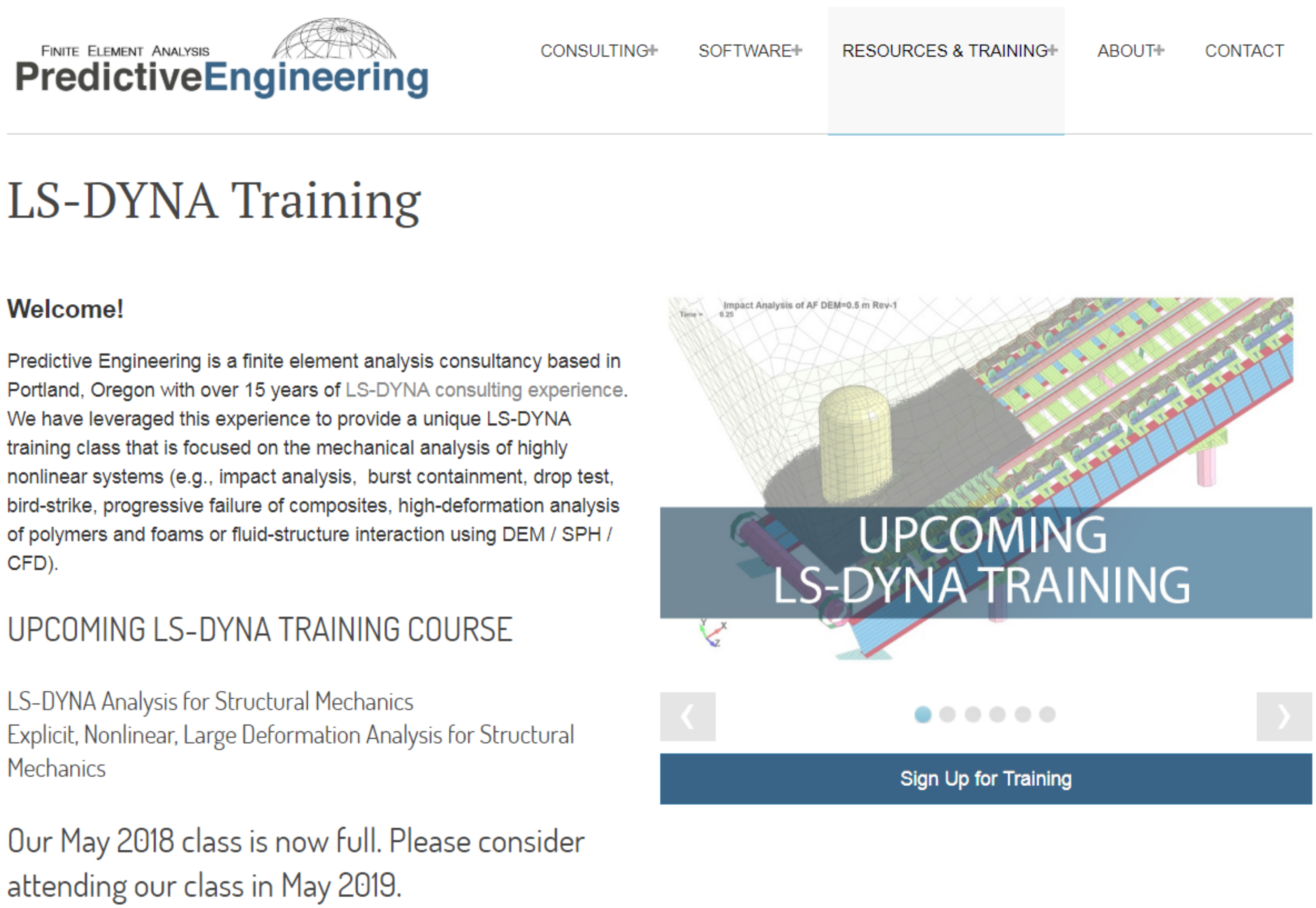
Considering the geometry preparation tools and meshing toolbox; the ability to create LS-DYNA material models and set element formulations, and the added flexibility of analysis text and include files, - Femap allows you to work from CAD geometry and generate a run-ready LS-DYNA analysis deck.

Full Nonlinear Dynamic Tranisent Analysis – Straight from Femap



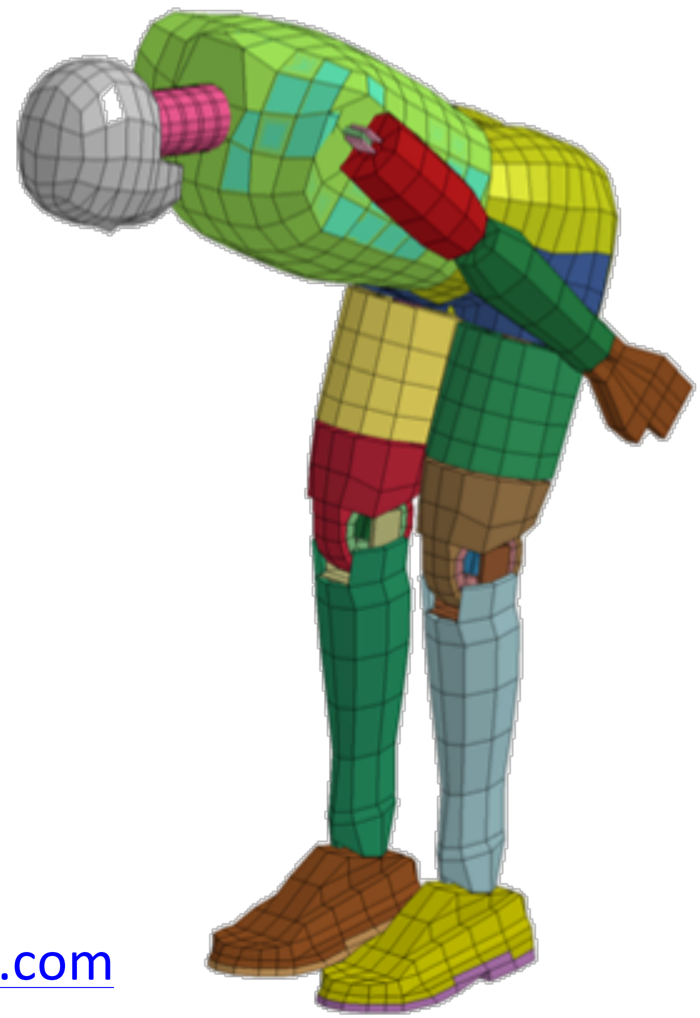
6. GETTING BY WITH A LITTLE HELP FROM YOUR FRIENDS

It seems that outside the core LSTC staff (which includes the DYNAmore team), true mastery of LS-DYNA is quite elusive. One could spend years studying the manuals and examples and only scratch the surface of capabilities. However, it doesn't mean that one can't get meaningful results in a reasonable timeframe. With the right training and resources, a good simulation engineer can use Femap and LS-DYNA without too much blood, sweat and tears.



Thank you for your Time!

Questions?



Adrian Jensen
Sr. Staff Engineer
Predictive Engineering, Inc.

Phone: (503) 206-5571
E-mail: adrian.jensen@predictiveengineering.com