



## LS-DYNA® Analysis for Structural Mechanics

An overview of the core analysis features used by LS-DYNA® to simulate highly nonlinear static and dynamic behavior in engineered structures and systems.

### **Please Note: PARTIAL CLASS NOTES FOR PROSPECTIVE ATTENDEES**

This training book has been edited from its original format. It has been shortened to 73 pages from its original 195 pages. For more on our training options, please visit <http://predictiveengineering.com/support-and-training/ls-dyna-training>



## Acknowledgements

These notes were constructed from numerous sources but special thanks should be given to the following people:

Technical Support Team at Livermore Software and Technology Corporation (LSTC)

With special mention to:

Satish Pathy, LSTC

Jim Day, LSTC

Philip Ho, LSTC

*And the invaluable team at:*

DYNAmore, GmbH, Germany

### **Trademarks:**

LS-DYNA® and LS-PrePost® are registered and protected trademarks of LSTC.

### **Disclaimer:**

The material presented in this text is intended for illustrative and educational purposes only. It is not intended to be exhaustive or to apply to any particular engineering design or problem. Predictive Engineering nor the organizations mentioned above and their employees assumes no liability or responsibility whatsoever to any person or company for any direct or indirect damages resulting from the use of any information contained herein.

## COURSE OUTLINE

1.	INTRODUCTION .....	10
1.1	WHAT THE STUDENT CAN EXPECT .....	10
1.2	WHAT WE COVER .....	10
1.3	HOW WE DO IT .....	10
1.4	HOW TO BE SUCCESSFUL IN LEARNING LS-DYNA .....	10
1.5	GENERAL APPLICATIONS .....	11
1.6	SPECIFIC APPLICATIONS (COURTESY OF PREDICTIVE ENGINEERING) .....	12
2.	WHAT IS LS-DYNA? .....	20
2.1	HOW WE VISUALIZE THE LS-DYNA ANALYSIS PROCESS .....	20
3.	IMPLICIT VERSUS EXPLICIT ANALYSIS .....	21
3.1	WHAT WE ARE SOLVING .....	21
3.2	EXPLICIT (DYNAMIC) – ONE MUST HAVE “MASS” TO MAKE IT GO .....	22
3.3	IMPLICIT (DYNAMIC OR STATIC) .....	22
3.4	PROS AND CONS OF EXPLICIT V IMPLICIT .....	22
4.	LS-DYNA GETTING STARTED WITH THE FUNDAMENTALS .....	23
4.1	LS-DYNA KEYWORD MANUAL .....	23
4.2	KEYWORD SYNTAX .....	23
4.3	UNITS .....	24
4.4	REFERENCE MATERIALS AND PROGRAM DOWNLOAD .....	25
4.4.1	Internal LSTC FAQ - <a href="ftp://ftp.lstc.com/outgoing/support/FAQ">ftp://ftp.lstc.com/outgoing/support/FAQ</a> .....	25
4.5	SUBMITTING LS-DYNA ANALYSIS JOBS AND SENSE SWITCHES .....	26
4.5.1	A Better Way to Run LS-DYNA Jobs: LS-RUN .....	26
4.6	LS-DYNA OUTPUT FILES (RESULTS AND MESSAGE FILES) AND DATABASE REQUESTS AND MANAGEMENT .....	27
4.7	WORKSHOP: 1 - LS-DYNA GETTING STARTED .....	28
5.	FUNDAMENTAL MECHANICS OF EXPLICIT ANALYSIS .....	29
5.1	TIME STEP SIGNIFICANCE .....	29
5.1.1	Explicit Time Integration .....	30
5.2	TIME STEP SIGNIFICANCE (COURANT-FRIEDRICHS-LEWY (CFL) CHARACTERISTIC LENGTH) .....	31
5.3	MASS SCALING: (EVERYBODY DOES IT BUT NOBODY REALLY LIKES IT) .....	32
5.3.1	Workshop: 2 - LS-DYNA Mass Scaling Basics .....	33
5.3.2	Instructor Led Workshop: 2 - Mass Scaling Advanced .....	35

5.4	IMPLICIT MESH VERSUS EXPLICIT MESH CHARACTERISTICS.....	36
5.4.1	Instructor Led Workshop: 3 - Implicit versus Explicit Mesh Differences .....	36
6.	EXPLICIT ELEMENT TECHNOLOGY .....	37
6.1	ELEMENT TYPES IN LS-DYNA.....	37
6.2	ONE GAUSSIAN POINT ISOPARAMETRIC SHELL ELEMENTS AND HOURGLASSING.....	38
6.2.1	Instructor Led Workshop: 4 - Explicit Element Technology   A: Side Bending .....	38
6.2.2	Instructor Led Workshop: 4 - Explicit Element Technology   B: Out-of-Plane Bending with Plasticity.....	39
6.2.3	Workshop: 3 - Building the Better Beam .....	40
6.2.4	Workshop: 4 - Hourglass Control/Hourglass.....	41
6.3	WORKSHOP: 5 – SOLID ELEMENT TECHNOLOGY – HEX AND TET FORMULATIONS .....	42
6.3.1	Workshop 5 – Solid Element Technology – Hourglass Control .....	43
6.4	SCALAR ELEMENTS (E.G., NASTRAN CBUSH) OR LS-DYNA “DISCRETE BEAM”.....	44
6.4.1	Workshop: 6 - Discrete Beam (Spring Away) .....	47
7.	LS-PREPOST.....	48
7.1	WORKSHOP: 7 - LS-PREPOST   WORKSHOP 6 (SKIP EXERCISES ON PAGES 67-69) .....	48
8.	MATERIAL MODELING .....	49
8.1	PART I: METALS .....	49
8.1.1	Engineering Stress-Strain vs True Stress-Strain.....	49
8.1.2	Basic Review of Material Models Available in LS-DYNA .....	50
8.1.3	Material Failure and Experimental Correlation.....	51
8.2	WORKSHOP: 8 - ELASTIC-PLASTIC MATERIAL MODELING (*MAT_024) .....	52
8.3	MATERIAL MODELING OF STAINLESS STEEL - *MAT_024 (CURVE) OR *MAT_098 (EQUATION) .....	54
8.4	STRAIN RATE SENSITIVITY OF METALS .....	55
8.5	PART II: PLASTICS, ELASTOMERS AND FOAMS .....	56
8.5.1	Modeling Plastics, Elastomers vs Foams (Viscoelasticity).....	56
8.6	MATERIAL MODELS FOR MODELING FOAMS.....	57
8.7	MODELING TECHNIQUES FOR ELASTOMERS AND FOAMS .....	58
8.7.1	Workshop: 9 - Modeling an Elastomer (*MAT_181) Ball with Hex and Tet Elements .....	59
8.8	PART III: COMPOSITE OR LAMINATE MATERIAL MODELING .....	60
8.8.1	Workshop: 10 - Composite Materials - Basic Understanding Using *MAT_054.....	61
8.9	PART IV: EQUATION OF STATE (EOS) MATERIAL MODELING .....	63
8.9.1	Modeling Water with *EOS_GRUNEISEN and *MAT_NULL.....	65
8.10	MATERIAL FAILURE SIMULATION .....	66



8.10.1	Basic Methods of Modeling Failure: Material versus Bond Failure.....	66
8.11	WORKSHOP: 11 - MODELING GENERAL MATERIAL FAILURE.....	67
8.12	MODELING RIGID BODIES.....	68
8.12.1	Rigid Materials (*MAT_020 or *MAT_RIGID) .....	68
8.12.2	Workshop: 12 - Using Rigid Bodies.....	69
8.12.1	Instructor Led Workshop: 5 – Connections From RBE2 /CNRB and RBE3/ CI.....	70
9.	CONTACT.....	71
9.1	DEFINITION OF CONTACT TYPES.....	71
9.2	GENERAL CONTACT TYPES.....	72
9.2.1	Additional Options: SOFT=2 “The Default” .....	72
9.2.2	Contact when things ERODE .....	73
9.2.3	MORTAR Contact.....	74
9.3	WORKSHOP: 13 - UNDERSTANDING BASIC CONTACT MECHANICS.....	76
9.3.1.1	Student Notes for Workshop – Understanding Basic Contact Mechanics.....	79
9.3.1.2	Contact Energy .....	80
9.3.1.3	Addendum to Workshop: Contouring Contact Pressures .....	81
9.4	INSTRUCTOR LED WORKSHOP: 6 – MANAGING CONTACT INTERFERENCE IN THE REAL WORLD .....	82
9.5	WORKSHOP: 14 - EDGE-TO-EDGE CONTACT.....	84
9.6	MISCELLANEOUS COMMENTS ON CONTACT .....	85
9.6.1	Contact Numerical Efficiency or Why Not All _MORTAR All the Time?.....	85
9.6.2	Why Paying Attention to the Contact time Step is Important .....	86
9.7	CONTACT BEST PRACTICES.....	88
9.8	MESH TRANSITIONS: TIED CONTACT FOR WELDING AND GLUING.....	89
9.8.1	Tied Contact or Gluing .....	89
9.8.2	Summary Table for Tied Contact .....	90
9.8.2.1	Summary Table for New Approach to _TIED Contact .....	91
9.8.3	Workshop: 15 - Tied Contact for Hex-to-Tet Mesh Transitions (TIED_SURFACE_TO_SURFACE).....	92
9.8.4	Workshop: 16 - Tied Contact for Gluing Things Together (BEAM_OFFSET) .....	93
9.8.5	Mesh Transitions: Mastering Tied Contact (Student Extra Credit).....	94
9.8.6	Instructor Led Workshop: 7 - _TIED Bad Energy (or why we use _BEAM_OFFSET) .....	95
10.	CONNECTIONS VIA JOINTS .....	96
10.1	JOINTS OR *CONSTRAINED_JOINT_ .....	96
10.2	HOW JOINTS WORK .....	97

10.3	WORKSHOP: 17A – SPHERICAL JOINT BETWEEN A SHELL AND SOLID .....	98
10.4	WORKSHOP: 17B - CYLINDRICAL JOINT BETWEEN TWO NESTED CYLINDERS .....	99
10.4.1	Who Uses Joints?.....	100
11.	DAMPING.....	101
11.1	GENERAL, MASS AND STIFFNESS DAMPING .....	101
11.1.1	*DAMPING_option .....	101
11.1.2	*DAMPING_FREQUENCY_RANGE_DEFORM.....	101
11.1.3	Material Damping (e.g., elastomers and foams).....	102
11.1.4	General Example on Material Damping.....	102
11.2	INSTRUCTOR LED WORKSHOP: 8 – DAMPING OF TRANSIENT VIBRATING STRUCTURES.....	103
12.	LOADS, CONSTRAINTS AND RIGID WALLS .....	104
12.1	LOADS.....	104
12.1.1	Initialization Loads (*INITIAL_).....	104
12.1.2	Point and Pressure Loads (*LOAD_POINT & _SEGMENT).....	104
12.1.3	Body Loads (*LOAD_BODY_).....	104
12.1.4	Rigid Walls (e.g., *RIGIDWALL_MOTION) .....	104
12.1.5	Boundary (e.g., *BOUNDARY_PRESCRIBED_) .....	104
12.2	WORKSHOP: 18 - DROP TEST OF PRESSURE VESSEL .....	105
13.	DATA MANAGEMENT AND STRESS AVERAGING .....	108
13.1.1	Stress Reporting and Stress Averaging in LS-DYNA/LSPF .....	109
	Instructor Led Workshop: 9 - Stress Reporting and Stress Averaging   Shells .....	109
14.	LOAD INITIALIZATION BY DYNAMIC RELAXATION AND IMPLICIT ANALYSIS .....	110
14.1	INITIALIZATION OF GRAVITY, BOLT PRELOAD AND OTHER INITIAL STATE CONDITIONS .....	110
14.2	WORKSHOP: 19 - DYNAMIC RELAXATION - BOLT PRELOAD PRIOR TO TRANSIENT .....	111
15.	IMPLICIT-EXPLICIT SWITCHING FOR BURST CONTAINMENT .....	112
15.1	HIGH-SPEED ROTATING EQUIPMENT – *CONTROL_ACCURACY.....	112
15.1.1	Workshop: 20 - Implicit-Explicit Switching for Turbine Spin Up.....	113
16.	SMOOTHED PARTICLE HYDRODYNAMICS (SPH) .....	114
16.1	INTRODUCTION .....	114
16.1.1	A Little Bit of Theory (skip this if you don't like math...).....	114
16.1.2	Lagrangian vs Eulerian .....	116
16.1.3	Types of Simulations with SPH .....	117
16.1.4	Common Keywords for SPH .....	117

16.2	WORKSHOP: 21A - SPH GETTING STARTED – BALL HITTING SURFACE .....	118
16.3	WORKSHOP: 21B - SPH GETTING STARTED - FLUID MODELING.....	119
16.4	WORKSHOP: 21C – SPH GETTING STARTED – BIRD STRIKE .....	120
16.4.1	Bird Strike Models .....	121
16.5	REFERENCES .....	122
17.	EXPLICIT EXAMINATION .....	123
18.	EXPLICIT MODEL CHECK-OUT AND RECOMMENDATIONS .....	125
18.1	UNITS .....	125
18.2	MESH .....	125
18.2.1	Using Surface Elements to Improve Stress Reporting Accuracy .....	125
18.3	D3HSP FILE (LS-DYNA EQUIVALENT TO THE NASTRAN F06 FILE).....	125
18.4	HISTORY PLOTS .....	126
18.5	MATERIAL MODELING ERRORS .....	126
18.6	CONTACT OPTIONS WITH RECOMMENDATIONS AND *CONTROL_CONTACT OPTIONS .....	127
18.6.1	*CONTROL_TIED Global Recommendation .....	128
18.7	CONTROL CARDS WITH RECOMMENDATIONS .....	129
18.8	DATABASE CARDS WITH RECOMMENDATIONS.....	130
18.9	EXPLICIT ELEMENT RECOMMENDATIONS.....	130
18.10	ETC.....	130
19.	IMPLICIT ANALYSIS .....	131
19.1	INTRODUCTION .....	131
19.1.1	Why Implicit? .....	131
19.1.2	What we cover .....	131
19.1.3	What Sort of Problems Can We Solve in Implicit? .....	132
19.2	IMPLICIT VERSUS EXPLICIT ANALYSIS.....	135
19.2.1	What We Are Solving.....	135
19.2.2	Review of Mathematical Foundation of Nonlinear Dynamic Implicit Analysis .....	136
19.3	LINEAR ELASTIC IMPLICIT ANALYSIS.....	137
19.3.1	Keywords Used in this Section for Shell Elements .....	137
19.4	SHELL ELEMENT TECHNOLOGY FOR LINEAR ELASTIC IMPLICIT ANALYSIS.....	138
19.4.1	In-Plane Gaussian Integration .....	138
19.4.1.1	Workshop: 22A - Linear Elastic Analysis – Shells - Stress Concentrations .....	139
19.4.2	Out-of-Plane Gaussian Integration.....	140

19.4.2.2	<b>Workshop: 21B – Linear Elastic Analysis – Shells – Out-of-Plane Integration .....</b>	<b>145</b>
19.5	SOLID ELEMENT TECHNOLOGY FOR LINEAR ELASTIC STRESS ANALYSIS.....	146
19.5.1.1	<b>Keywords Used in this Section for Solid Elements .....</b>	<b>147</b>
19.5.1.2	<b>Workshop: 23 – Linear Elastic Analysis – Solids - Hex &amp; Tets.....</b>	<b>148</b>
19.6	BEAM ELEMENT TECHNOLOGY FOR LINEAR ELASTIC STRESS ANALYSIS .....	151
19.6.1	Beam Integration (QR) Setting: Rectangular .....	151
19.6.2	Beam Integration (Cylindrical Solid and Tube).....	152
19.6.3	Workshop: 24 - Linear Elastic Analysis – Beam Analysis .....	153
19.7	CHECKLIST FOR IMPLICIT LINEAR ELASTIC ANALYSIS IN LS-DYNA.....	154
19.8	GEOMETRIC AND MATERIAL NONLINEARITY.....	155
19.8.1	New Keyword Commands Used in this Section for Nonlinear Implicit Analysis .....	156
19.8.2	Workshop: 25 – Nonlinear Buckling of Beer Can .....	157
19.8.3	Checklist for Implicit Nonlinear Analysis in LS-DYNA.....	158
19.9	CONTACT .....	159
19.9.1	General Comment and Focus on Mortar Contact .....	159
19.9.2	General Mortar Contact Types.....	160
19.9.2.1	<b>Workshop: 26A – Implicit Contact – Basic Mechanics with Bolt Preload .....</b>	<b>161</b>
19.9.2.2	<b>Workshop: 26B - Contact - Shrink Fit Analysis .....</b>	<b>162</b>
19.9.3	Tied Contact for Mesh Transitions, Welding and Gluing .....	163
19.9.4	Checklist for Implicit Nonlinear Contact Analysis in LS-DYNA.....	164
19.9.5	But My Boss Says That Using “Dynamics” is Wrong for a Static Solution? .....	165
19.10	RIGID BODY USAGE.....	166
19.10.1	RBE2 to CNRB .....	166
19.11	LOAD INITIALIZATION .....	167
19.11.1	Workshop: 27 – Load Initialization – Implicit-Explicit Switching.....	167
19.12	LINEAR DYNAMICS: IT IS ALL ABOUT THE NORMAL MODES .....	169
19.12.1	Normal Modes Analysis .....	170
19.12.1.1	<b>Workshop: 28 - Normal Modes Analysis.....</b>	<b>171</b>
19.12.2	Response Spectrum Analysis or Shock Response Analysis.....	173
19.12.2.1	<b>Workshop: 29 - Response Spectrum Analysis of Bracket.....</b>	<b>174</b>
19.12.3	PSD Analysis .....	175
19.12.3.1	<b>Workshop: 30 - PSD Analysis and Zero-Crossing Frequencies.....</b>	<b>176</b>
20.	IMPLICIT MULTI-PHYSICS: COUPLED THERMAL-STRESS ANALYSIS .....	178

20.1.1.1	Getting Started with Coupled Thermal-Stress Analysis .....	178
20.1.2	Workshop 31: Couple Thermal-Stress Analysis.....	179
20.2	IMPLICIT CHECK-OUT AND RECOMMENDATIONS.....	180
20.2.1	Model Construction Recommendations .....	181
20.2.2	Implicit Keyword Cards and Recommendations .....	182
20.2.3	For post processing of results .....	184
20.2.4	Convergence Troubleshooting and Solution Speed Optimization .....	185
20.2.5	General Troubleshooting .....	185
20.2.6	Convergence .....	185
20.2.7	Lessons from the Field .....	186
20.2.7.1	What Careful Postprocessing Can Tell You .....	186
20.2.7.2	Workshop On Convergence Verification .....	186
20.2.7.3	Spotweld Elements are Rods (3 DOF) .....	187
20.2.8	Comments on LS-DYNA Output Messages and Their Significance .....	188
21.	DISCRETE ELEMENT METHOD .....	189
22.	FLUID STRUCTURE INTERACTION AND MULTI-PHYSICS IN LS-DYNA .....	190
23.	LSPP SUMMARY LIST OF OPERATIONS AND CORRESPONDING WORKSHOPS .....	191
24.	APPENDIX.....	192
24.1	LICENSE NOTES .....	192
24.1.1	Licensing Notes: Making It Work With LSTC_FILE .....	193

## 1. INTRODUCTION

### 1.1 WHAT THE STUDENT CAN EXPECT

This class is directed toward the engineering professional simulating highly nonlinear, static and dynamic problems involving large deformations and contact between multiple bodies. What this means in more layman terms is that we will provide a realistic foundation toward the practical usage of LS-DYNA.

### 1.2 WHAT WE COVER

- Nonlinear Explicit and Implicit FEA Mechanics
- The technology of creating accurate nonlinear, transient FEA models
- How to do your own research to create more advanced simulations
- Our condensed experience and that of our colleague's to help you *not* repeat our mistakes

### 1.3 HOW WE DO IT

- The class covers the basics in a hands-on manner as taught by an engineer that has had to live by what they have validated.
- Each day will have six to eight Workshops. Each Workshop is part theory, part demonstration and part hands-on practice. Videos are provided for each Workshop allowing the student to relax and follow along at their own pace. These videos cover the basics and also provide insight into the many tips and tricks that make LS-DYNA the world's most complete and accurate simulation code.
- Breaks are provided every two hours where students can pause, relax and ask the instructor more detailed questions.
- Students are encouraged to turn off their email, text messaging and other forms of digital/social media during class time (8:00 am to 5:00 pm).

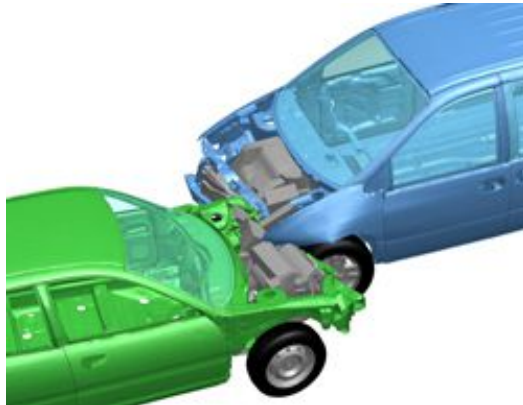
### 1.4 HOW TO BE SUCCESSFUL IN LEARNING LS-DYNA

- You are already way ahead of the pack by simply reading this far. You have started on the journey of how to be successful with LS-DYNA. It is this simple. To be successful as far as we know, it requires:
  - Reading (very traditional but necessary to read the manual (RTM), read again and most likely for us normal people, read again.
  - Attend courses since it breaks up the learning process and opens doors to new avenues of learning and knowledge
  - Be open to new ideas and then once again, read, read and read some more.
  - After all this reading, one has to do some organic learning. That means building small models to explore options and mechanics and to suffer a bit prior to calling your colleagues for help.
  - Lastly, don't be hesitant to reach out for help once you have read, built small models to explore options, read some more and then are posed to asked questions that will lead you quickly toward the right solution for your project. Without this background, your questions will often be wild, untamed and often just a waste of time to you and your colleague.

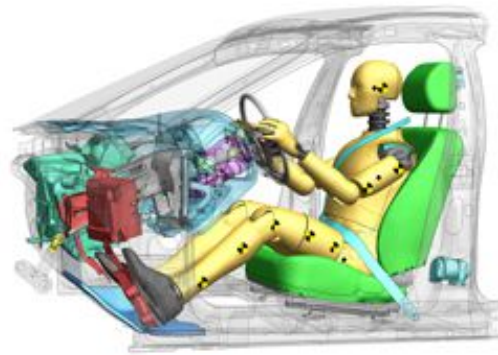


## 1.5 GENERAL APPLICATIONS

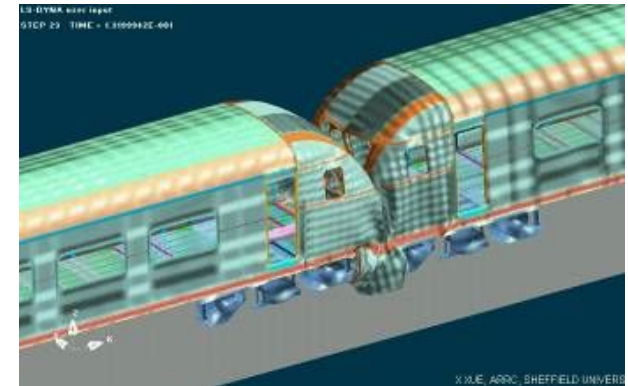
Crashworthiness



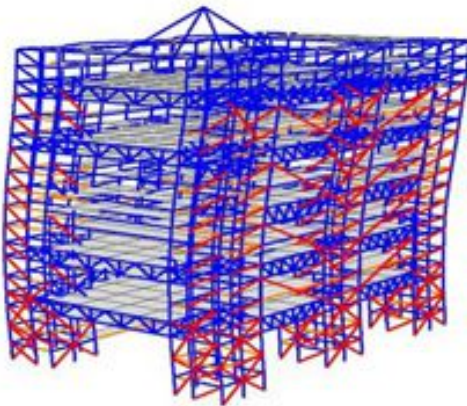
Driver Impact



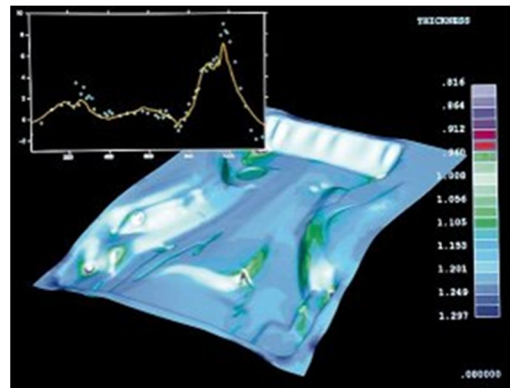
Train Collisions



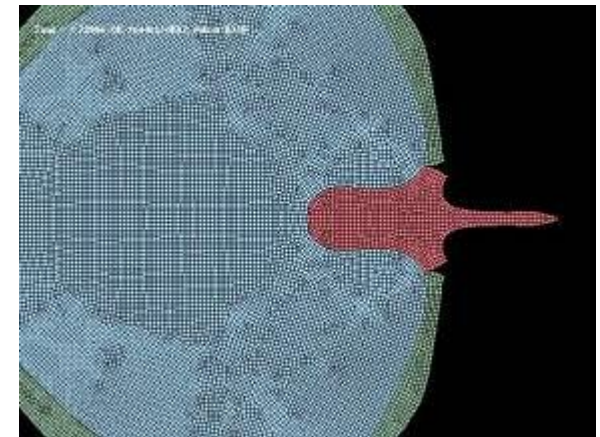
Earthquake Engineering



Metal Forming

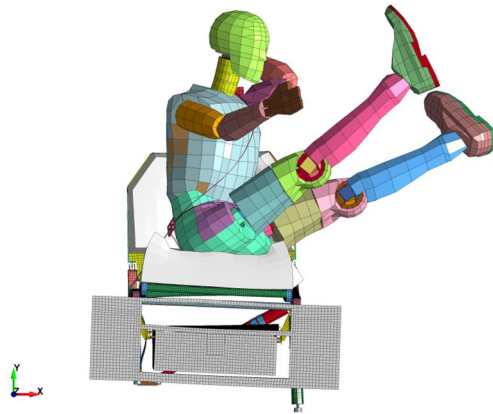


Military

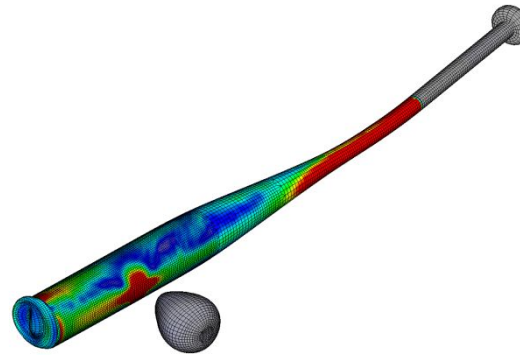


## 1.6 SPECIFIC APPLICATIONS (COURTESY OF PREDICTIVE ENGINEERING)

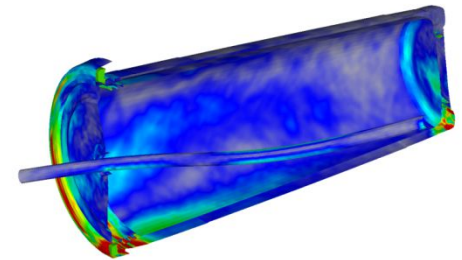
CS-FAR 25 16g Sled Analysis



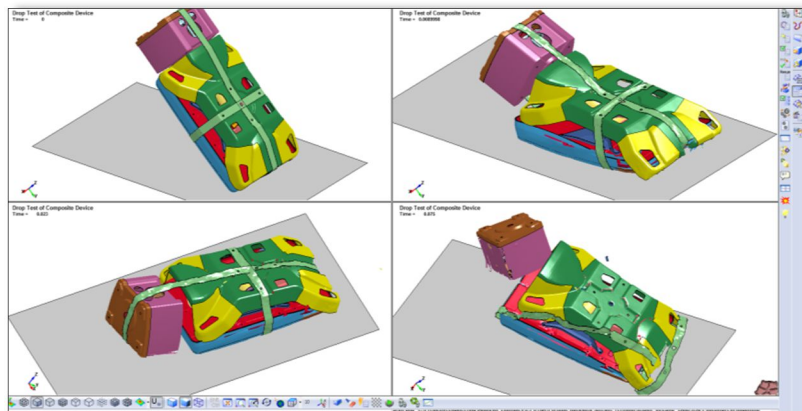
Sporting Goods Equipment



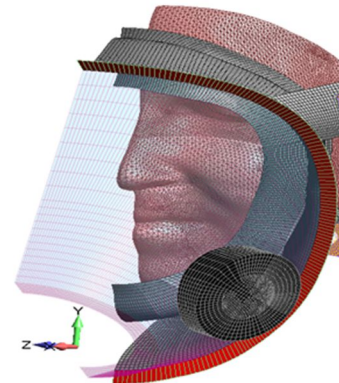
Drop Test Consumer Products



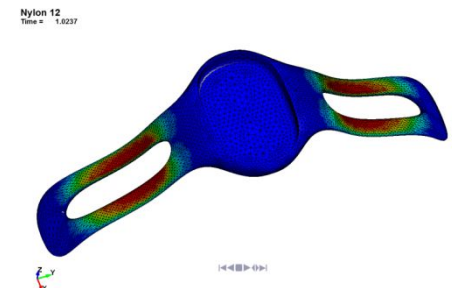
Drop Test of Composites / Electronics



Human Biometrics



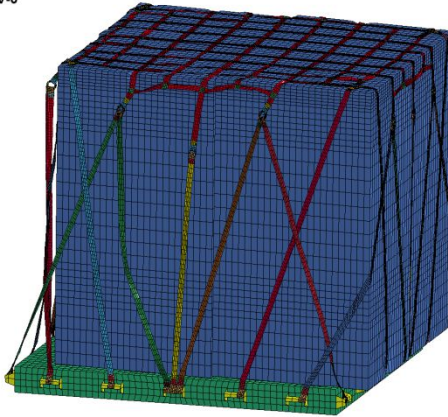
Large Deformation of Plastics



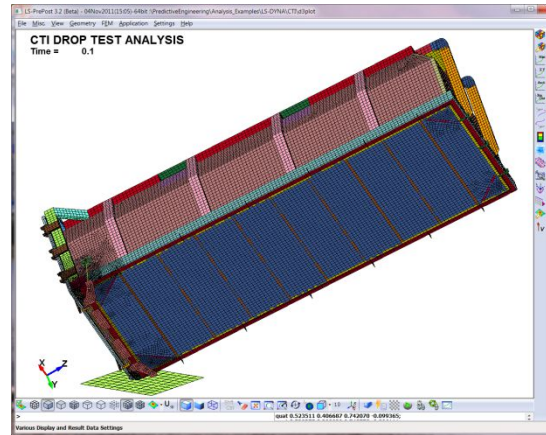


Crash Analysis of Cargo Net

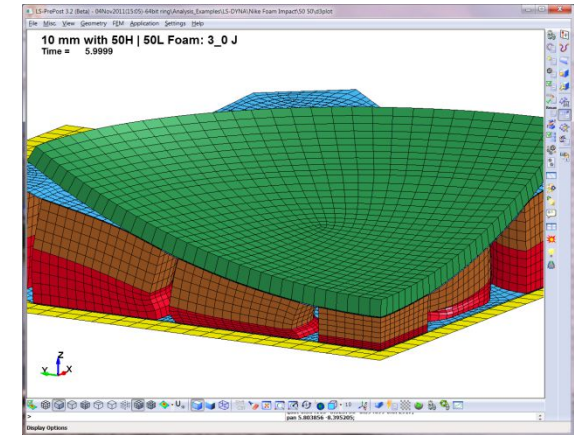
REV-0



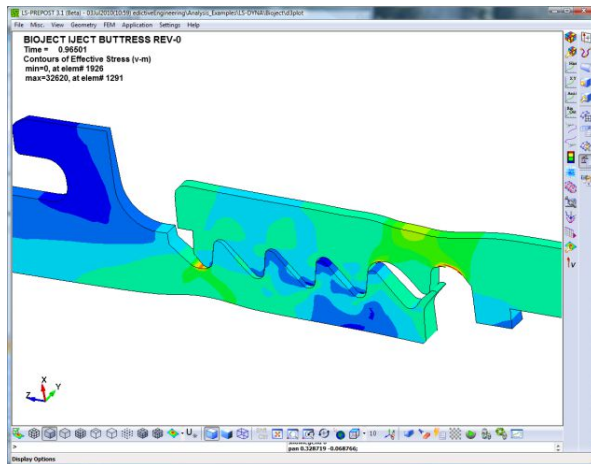
Drop Test of Nuclear Waste Container



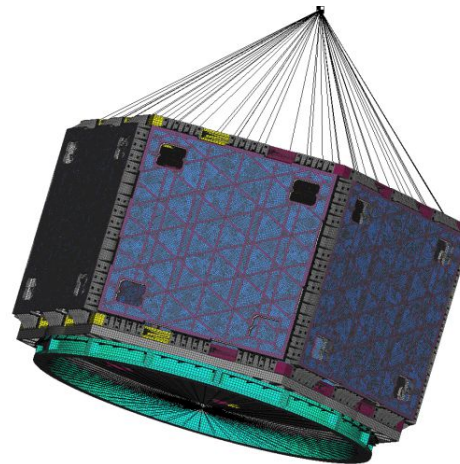
Impact Analysis of Foams



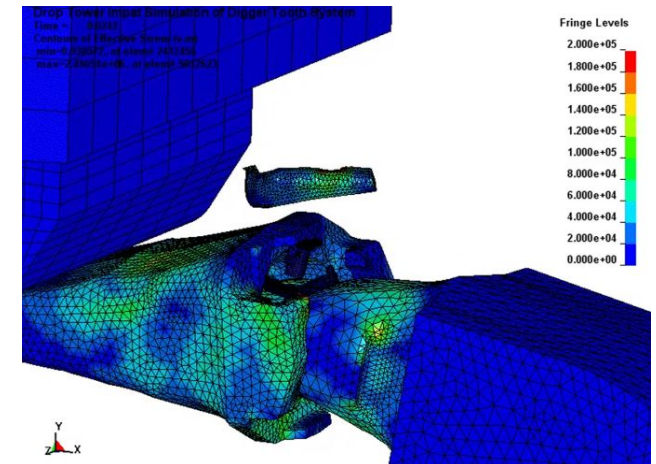
Plastic Thread Design



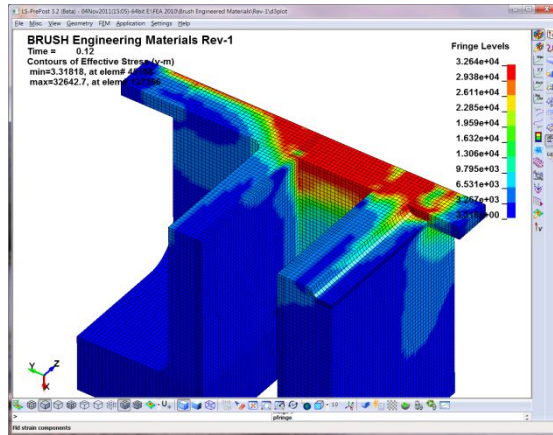
PSD / Modal Analysis



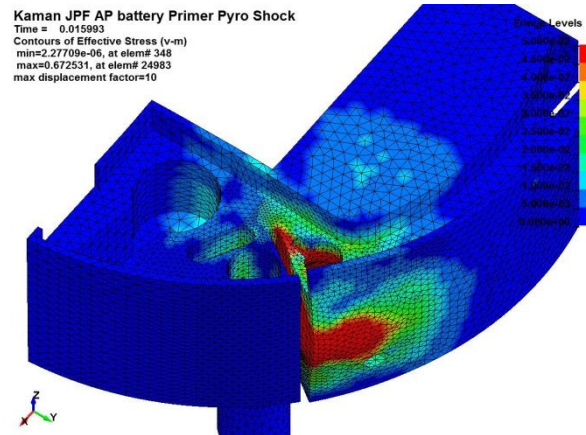
Digger Tooth Failure



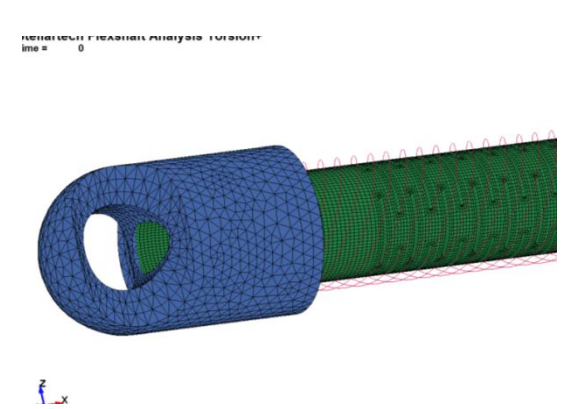
## Electron Beam Welding



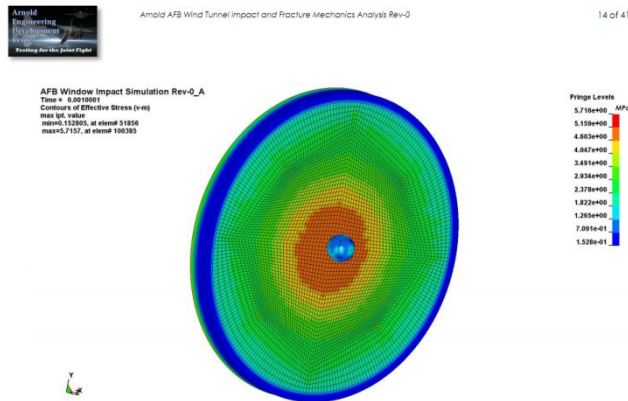
## Pyro-Shock Analysis



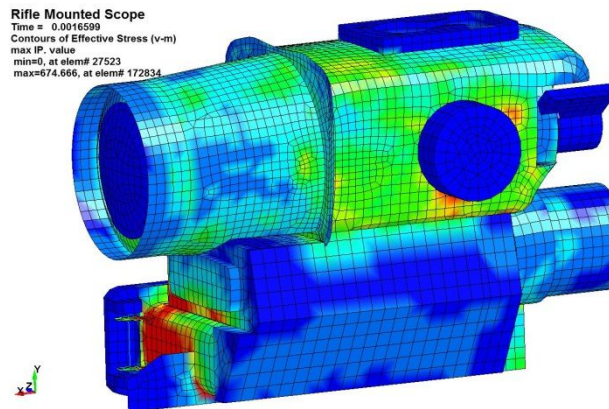
## Medical Equipment



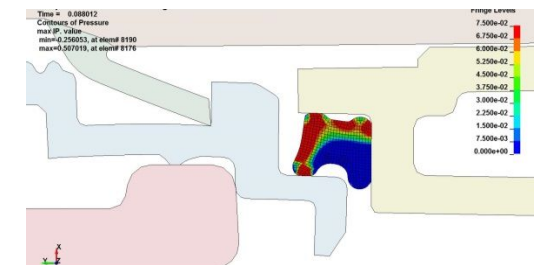
## Fracture Mechanics of Glass



## Ballistic Shock Loading of Optical Equipment

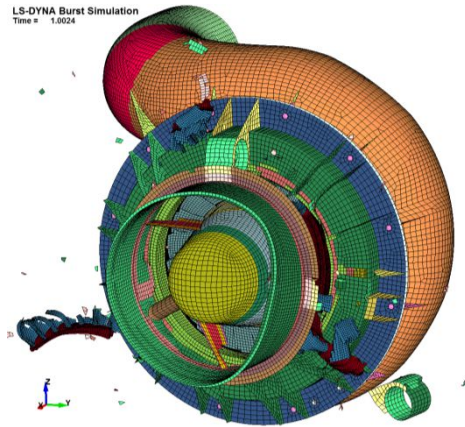


## Hyperelastic Medical Seal Analysis

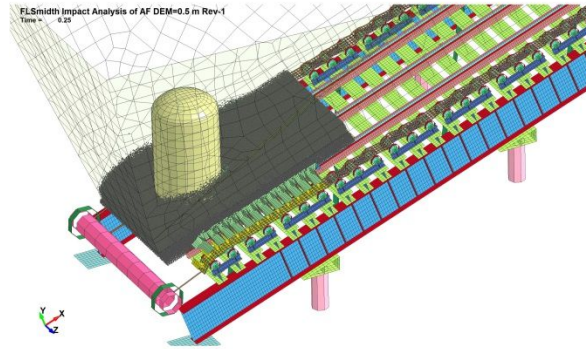




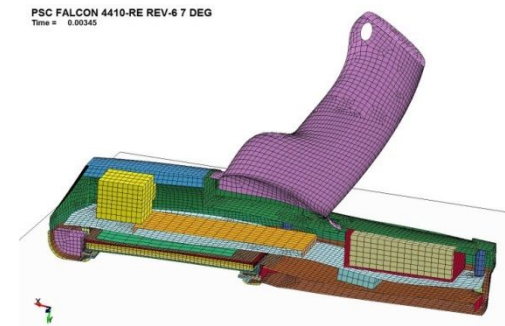
Blade-Out Analysis



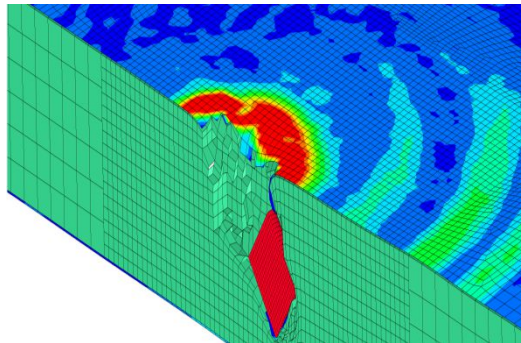
Discrete Element Method for the Mining Industry



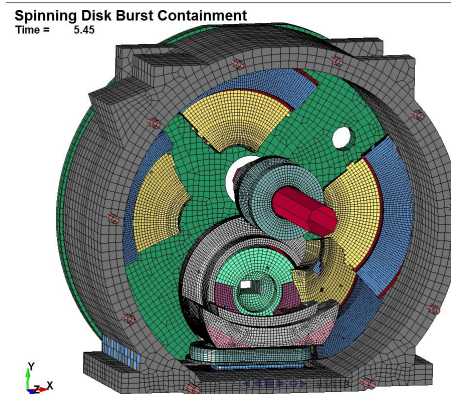
Drop-Test of Hand Held Electronics



Ballistic Penetration of Al/Foam Panel

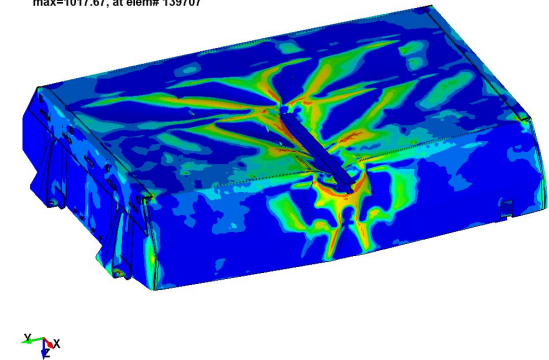


High-Speed Spinning Disk Containment

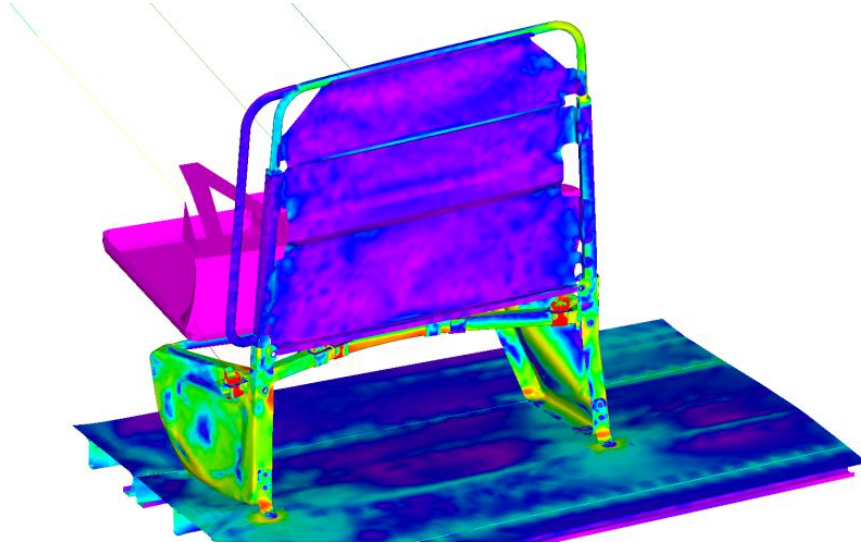


Locomotive Fuel Tank

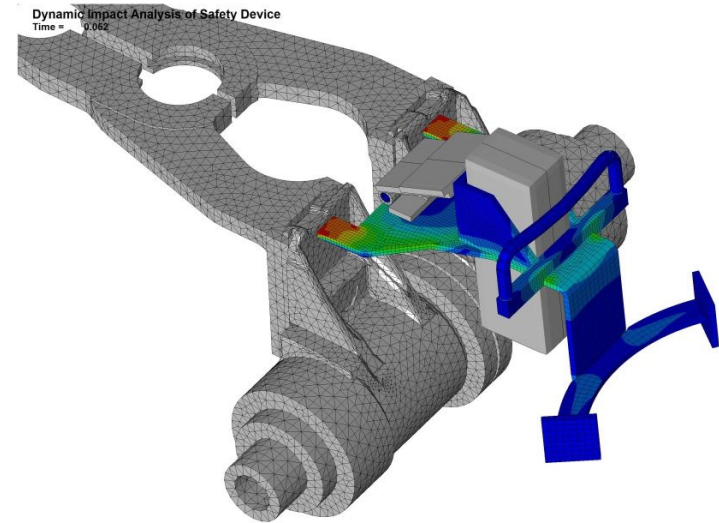
Locomotive Fuel Tank Crushing Analysis  
Time = 0.050001  
Contours of Maximum Principal Stress  
Ipt #2 and Ipt #3  
min=-0.0048221, at elem# 276069  
max=1017.67, at elem# 139707



FMVSS Virtual Testing of Bus Seats

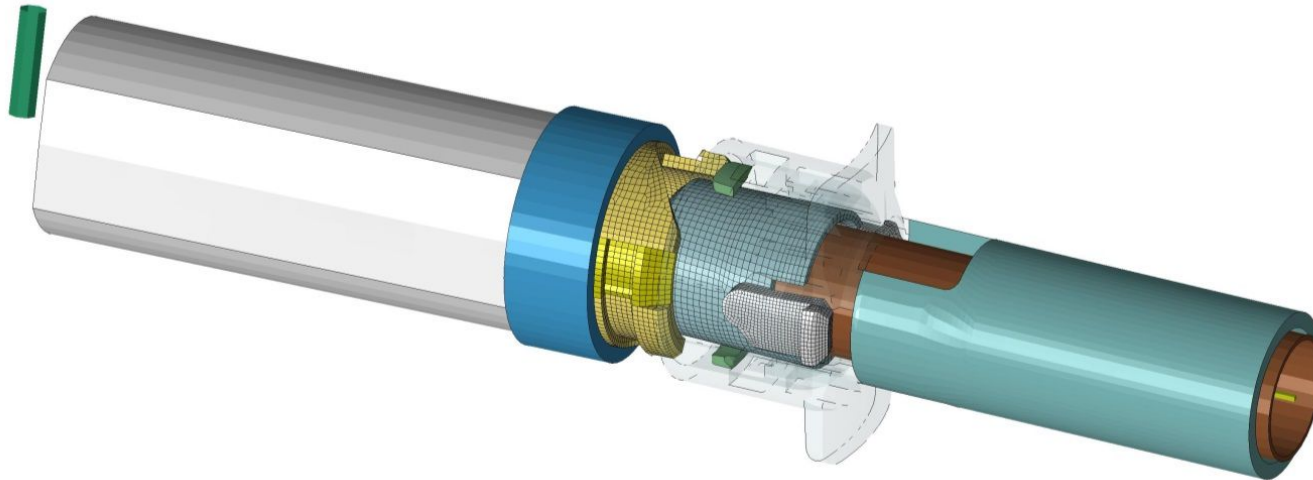


Impact Analysis of Safety Block Device



Snap-Fit Analysis – All Plastic Medical Device

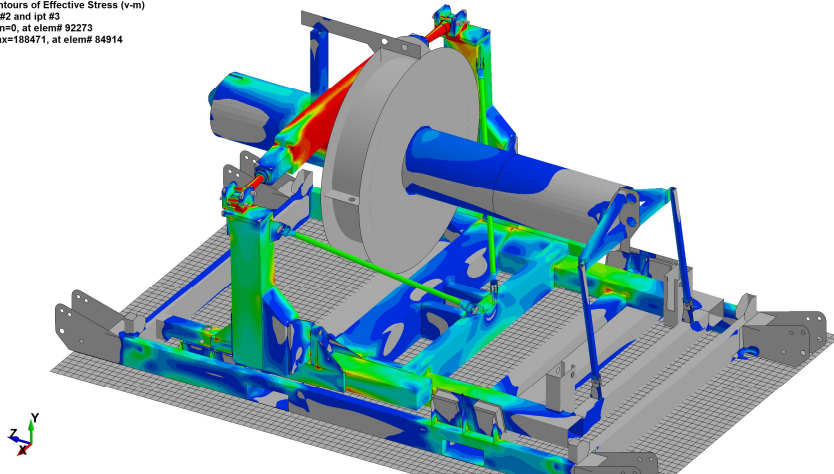
Plastic Assembly Snap-Fit  
Time = 12



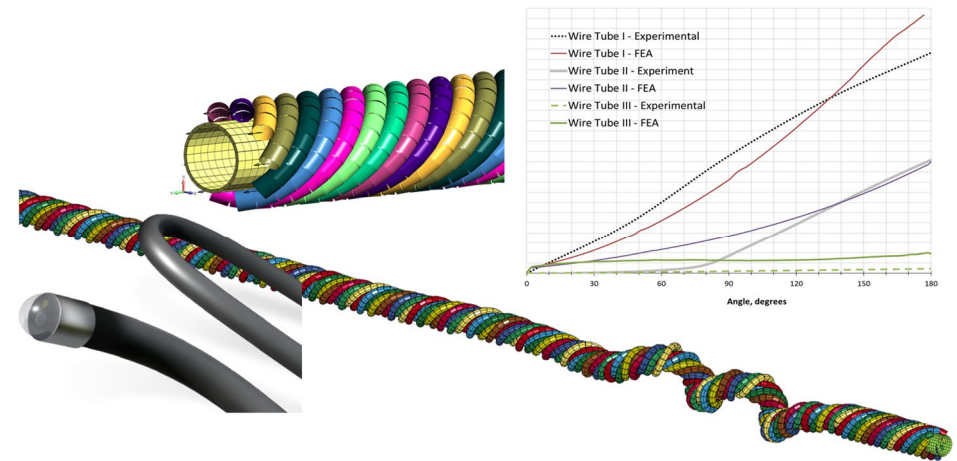


## 9g Crash Analysis of Jet Engine Stand

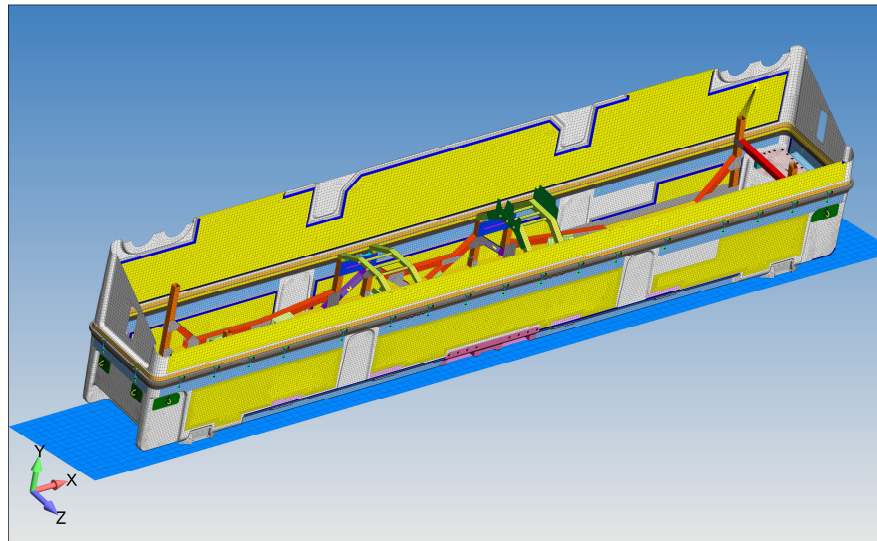
Time = 1.0031  
Contours of Effective Stress (v-m)  
lpt #2 and lpt #3  
min=0, at elem# 92273  
max=188471, at elem# 84914



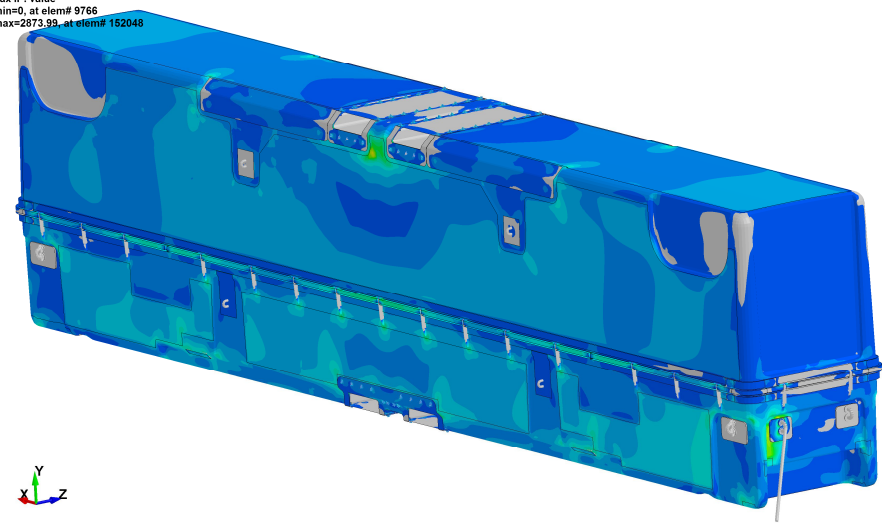
## Torque Analysis of Endoscopic Medical Device



## Drop, Rail Impact and PSD Analysis of Composite Container

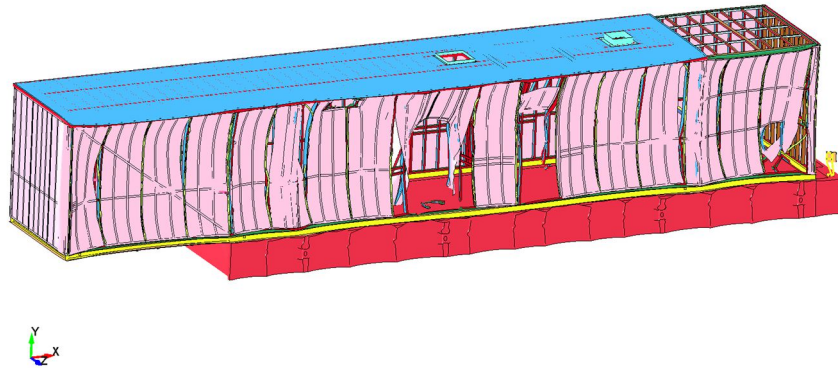


Contours of Effective Stress (v-m)  
max IP value  
min=0, at elem# 9766  
max=2073.95, at elem# 152048



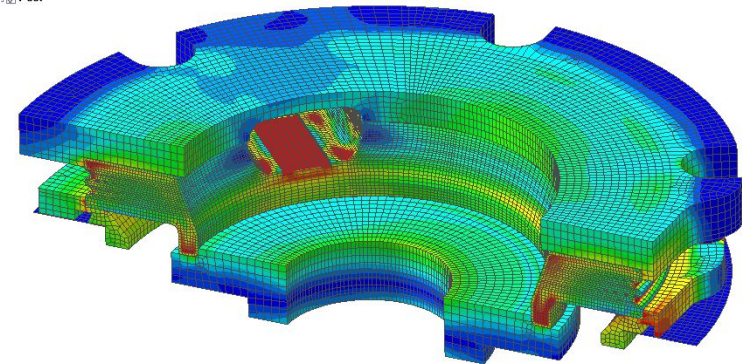
## ConWep Air Pressure Blast Analysis of Generator Housing

LS-DYNA Air Pressure ConWep Blast Analysis Rev-1  
Time = 0.0225



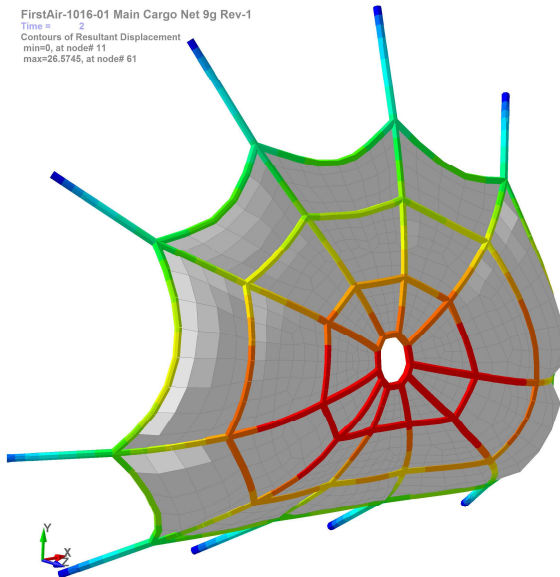
## Alumina-Stainless Steel Braze Process Simulation

FEI-1116-01 Insert F Braze Process Simulation Ambient to Solidus Rev-0  
Time = 0.68133  
Contours of Effective Stress (v-m)  
outer shell surface  
min=0, at elem# 171025  
max=80.7882, at node# 243293  
Post

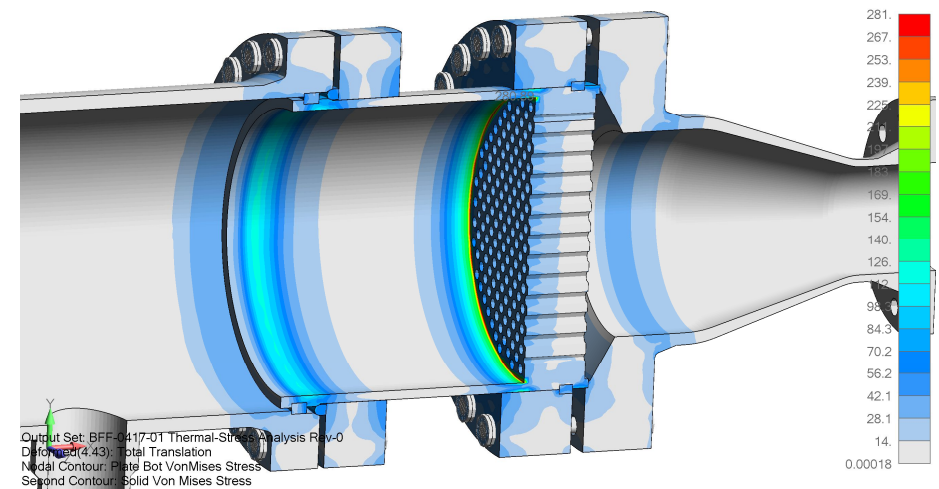


## Air Freighter 9g Cargo Net Analysis

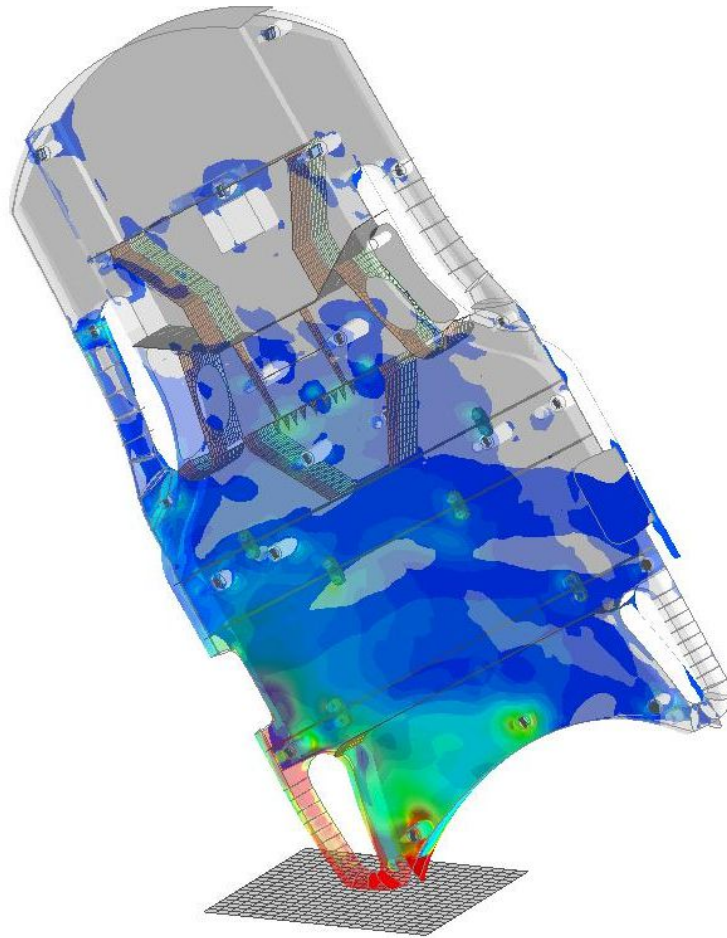
FirstAir-1016-01 Main Cargo Net 9g Rev-1  
Time = 2  
Contours of Resultant Displacement  
min=0, at node# 11  
max=26.5745, at node# 61



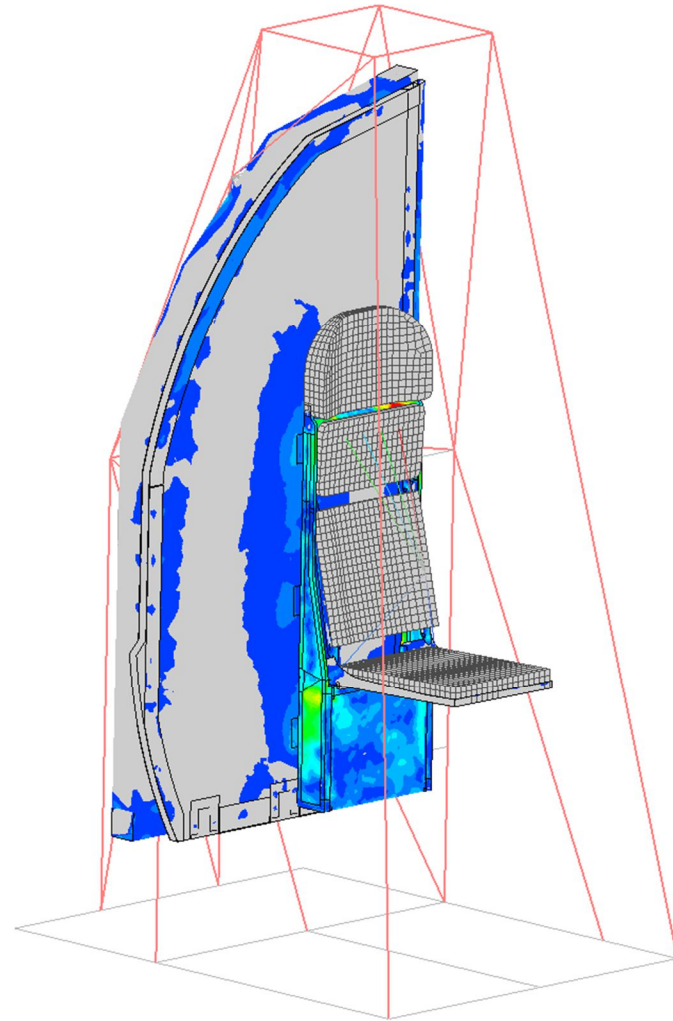
## Thermal-Stress Fatigue Analysis of ASME Evaporator Vessel



Drop-Test of First-Responder Medical Equipment



Composite Lavatory Wall with Attendant Seat 16g Sled Test





## 2. WHAT IS LS-DYNA?

LS-DYNA is a finite element analysis (FEA) solver. It is the motor that generates results based on what the user provides as input. In other words, it is not a mesh generation tool or one that can create stress contour plots but the world's most sophisticated and complex FEA solver. The workflow is to provide LS-DYNA an ascii text based deck (with a suffix as \*.k or \*.dyn) with nodes, elements, loads, constraints, material laws, etc. and then LS-DYNA solves this input and generates another file (\*.f06) with the requested results.

One can read an LS-DYNA analysis deck with any text editor. A lot of useful information about the LS-DYNA code and its structure can be found in the LS-DYNA Keyword Manual Vol. 1. For every new user, it is time well spend to read the Introduction and Getting Started sections. It provides some very nice background on the LS-DYNA code.

### 2.1 HOW WE VISUALIZE THE LS-DYNA ANALYSIS PROCESS

No matter where you build your deck, LSTC's LS-PrePost (henceforth LSPP) is often an invaluable tool along the path to a validated FEA model. This course is focused on setting up a simulation model that will solve using LS-DYNA and generate results that are verifiable and will lead to a validated solution. We do not focus on how the nodes and elements may be generated but we do focus on their quality.

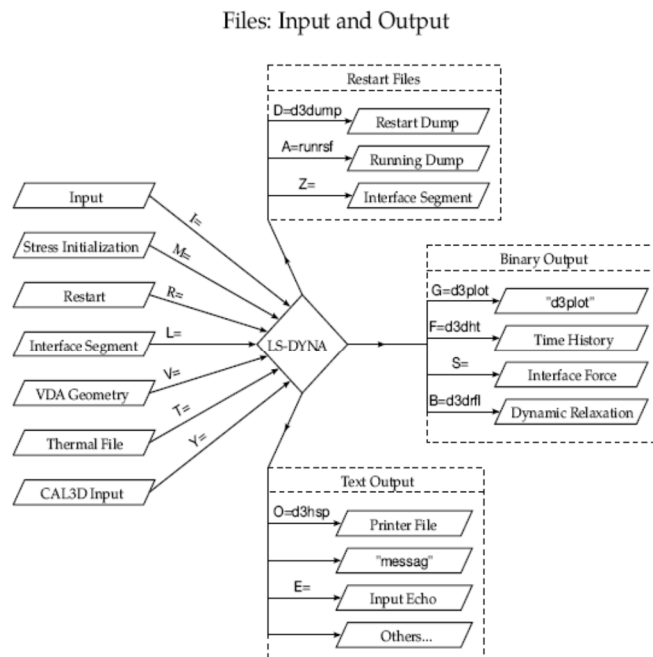


Figure 2-2. Files Input and Output.

Pre-Processor  
(Nodes, Elements, Etc.)

Commercial Software  
{Ansys, Hypermesh,  
FEMAP, etc.}

LSPP

*Pre-Processing to Create Keyword Deck*

**LS-DYNA**

*Post-Processing FEA Results*

Commercial Software  
{Ansys, Hypermesh, FEMAP,  
etc.}

LSPP

Proprietary Software



### 3. IMPLICIT VERSUS EXPLICIT ANALYSIS

LS-DYNA is a non-linear transient dynamic finite element code with both explicit and implicit solvers.

#### 3.1 WHAT WE ARE SOLVING

Explicit only works when there is acceleration of **mass** (dynamic) whereas an implicit approach can solve the dynamic and the static problem (**no mass**). For dynamic problems, we are solving the following equation:

$$ma^n + cv^n + kd^n = f^n$$

where n=time step. A common terminology is to call the  $kd^n$  part the internal force in the structure. The basic problem is to determine the displacement at some future time or  $d^{n+1}$ , at time  $t^{n+1}$ .

In conceptual terms, the difference between Explicit and Implicit dynamic solutions can be written as:

$$\textit{Explicit: } d^{n+1} = f(d^n, v^n, a^n, d^{n-1}, v^{n-1}, \dots)$$

All these terms are known at time state “n” and thus can be solved directly. This means that the solution marches forward regardless of the element deformation or contact behavior or whatever nonlinearities (*importantly, no residual – see below*). However, it doesn’t mean that it might not blow up if elements get too distorted and it doesn’t mean that contact will always contact.

For *Implicit*, the solution depends on nodal velocities and accelerations at state n+1, quantities which are unknown:

$$\textit{Implicit: } d^{n+1} = f(v^{n+1}, a^{n+1}, d^n, v^n, \dots)$$

Given these unknowns, an iterative solution is required to calculate the displacement at this future time. If the nonlinearity is mild, the implicit approach allows one to use a comparably large time step as that compared to the explicit analysis and the run time can be advantageous. This is because an implicit solution must perform an iterative solution to reduce the residual within each time step:

$$ma^n + cv^n + kd^n - f^n = \textit{Residual}$$

If the nonlinearity is severe, the implicit solution may require a very small time step and a large number of iterations within each step to reduce the residual to something reasonable (i.e., a converged solution). In contrast, an explicit solution has no residual and just solves but requires a small time step (more will be said about this later). Thus, when faced with large nonlinearities, an explicit solution is more robust whereas, if the nonlinearity is mild, an implicit solution is often more practical to get the job done quickly.

### 3.2 EXPLICIT (DYNAMIC) – ONE MUST HAVE “MASS” TO MAKE IT GO

Internal and external forces are summed at each node point, and a nodal acceleration is computed by dividing by nodal mass. The solution is advanced by integrating this acceleration in time. The maximum time step size is limited by the Courant condition, producing an algorithm which typically requires many relatively inexpensive time steps. Using this criterion, the solution is unconditionally stable. Since the solution is solving for displacements at nodal points, the time step must allow the calculation to progress across the element without “skipping” nodes, that is, the time step must ensure that the stress wave stays within the element. Hence, the explicit solution is limited in time step by the element size and the speed sound in that element under study. Much more will be said about element size and the speed of sound in materials since execution speed for an explicit analysis is often of great importance given that careful meshing can mean the difference between a run time of days or hours. Just to plant the seed, but an explicit analysis is all about mass since everything has a time step (e.g., contact, 1D spring elements, CNRB’s, etc.).

Advantages:

### 3.3 IMPLICIT (DYNAMIC OR STATIC)

A global stiffness matrix is computed, decomposed and applied to the nodal out-of-balance force to obtain a displacement increment. Equilibrium iterations are then required to arrive at an acceptable “force balance”. The advantage of this approach is that time step size may be selected by the user. The disadvantage is the large numerical effort required to form, store, and factorize the stiffness matrix. Implicit simulations therefore typically involve a relatively small number of expensive time steps. The key point of this discussion is that the stiffness matrix (i.e., internal forces) has to be decomposed or inverted each time step whereas in the explicit method, it is a running analysis where the stiffness terms are re-computed each time step but no inversion is required. Since this numerical technique is independent of a time step approach, element size is not of direct concern only the size of the model (nodes/elements) directly affects the run time.

### 3.4 PROS AND CONS OF EXPLICIT V IMPLICIT

Explicit		Implicit	
Pros	Cons	Pros	Cons
It solves directly since the solution marches forward.	Solution time step controlled by wave speed and element mechanics.	Large time steps can be used since the solution is iterative.	Requires iterative process to converge.
Dynamic solution	Long run times for simulations that require long event times.	Static and Dynamic solutions	Requires iterative process to converge which can lead to long run times.
Extreme nonlinearity is easily handled.	Of course, solution can blow up due to twisted elements or contact problems.	Linear and Nonlinear solutions	Implicit struggles with extreme nonlinearity
Pretty much all physics can be solved.	No Cons since you are solving the impossible.	Provides the missing link in LS-DYNA to solve standard linear static and dynamic problems.	Focused on solid mechanics so don’t expect to see meshfree methods anytime soon.

## 4. LS-DYNA GETTING STARTED WITH THE FUNDAMENTALS

### 4.1 LS-DYNA KEYWORD MANUAL

LS-DYNA has perhaps one of the most basic learning methods. It is organic. One simply has to dig in and learn the basics and there is no substitute for doing it yourself. The Keyword Manual also provides recommended usage guidelines and examples on how to use the commands. It is your first and best resource. Given the frequency of program updates, the Keyword manuals are likewise being constantly updated. Fairly recent versions of the four Keyword manuals can be found in the *Class Reference Notes / Keyword Manuals*.

*Analyst's Note: Please keep in mind that LS-DYNA is an analysis engine that runs off of an ascii deck (a text file) and that oftentimes the fastest path to an optimum solution is to edit the deck.*

### 4.2 KEYWORD SYNTAX

- Commands are strings of words separated by an underscore, e.g., \*BOUNDARY\_PRESCRIBED\_MOTION\_RIGID.
- Text can be uppercase or lowercase
- Commands are arranged alphabetically in User's Manual
- Order of commands in input deck is **mostly** unimportant (except \*KEYWORD, \*DEFINE\_TABLE, \*INCLUDE\_TRANSFORM, ?)
- Keyword command must be left justified, starting with an asterisk
- A "\$" in the first column indicates a comment
- Input values can be in **fixed fields or comma-delimited**
- A blank or zero parameter indicates that the default value of parameter will be used (or taken from \*CONTROL\_option)

## LS-DYNA® KEYWORD USER'S MANUAL

### VOLUME I

03/03/17 (r:8240)  
LS-DYNA Dev

LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)

### Required Commands:

\*KEYWORD  
 \*CONTROL\_TERMINATION  
 \*NODE  
 \*ELEMENT  
 \*SECTION  
 \*MAT  
 \*PART  
 \*DATABASE\_BINARY\_D3PLOT  
 \*END

### 4.3 UNITS

Many a fine analysis model has been brought down by bad units. Although one may wonder why in this modern age one still has to twiddle with units and not have it addressed by the interface is philosophical-like engineering debate between the ability to hand-edit the “deck” or be hand-cuffed to a gui (pronounced “gooey”) interface. Moving past this discussion, to use LS-DYNA effectively, one should have a rock-solid and un-shakable conviction in your chosen system of units.

Since the majority of LS-DYNA work is dynamic, the analyst will often be looking at the energies of the system or velocities, in addition to displacements and stresses. Hence, a consistent set of units that are easy to follow can provide significant relief in the debugging of an errant analysis. A general guide to units can be viewed within the Class Reference Notes / Units (see [Consistent units — LS-DYNA Support.pdf](#)). Saying all that, here are the four unit systems that I have standardized on for analysis work. It doesn’t mean they are the best but at least they are generally accepted.

#### Consistent Unit Sets for LS-DYNA Analysis

Mass	Length	Time	Force	Stress	Energy	Density Steel	Young’s	Gravity
kg	m	s	N	Pa	J	7,800	2.07e+9	9.806
g	mm	ms	N	MPa	N-mm	7.83e-03	2.07e+05	9.806e-03
kg	mm	ms	kN	GPa	kN-mm	7.83e-06	2.07e+02	9.806e-03
Ton (1,000 kg)	mm	s	N	MPa	N-mm	7.83e-09	2.07e+05	9.806e+03
Lbf-s <sup>2</sup> /in (snail)	in	s	lbf	psi	lbf-in	7.33e-04	3.00e+07	386

## 4.4 REFERENCE MATERIALS AND PROGRAM DOWNLOAD

The first site to visit: [www.lsdynasupport.com](http://www.lsdynasupport.com)

Another great site: [www.dynasupport.com](http://www.dynasupport.com)

LS-DYNA Examples: [www.DYNAExamples.com](http://www.DYNAExamples.com)

LS-DYNA Conference Papers: [www.dynalook.com](http://www.dynalook.com)

Newsletter: [www.FEAInformation.com](http://www.FEAInformation.com)

Newsletter and Seminars: [www.DYNAmore.com](http://www.DYNAmore.com)

Yahoo Discussion Group: [LS-DYNA@yahoogroups.com](mailto:LS-DYNA@yahoogroups.com)

Aerospace Working Group: [awg.lstc.com](http://awg.lstc.com)

Varmit AI's Material Database (google'it)

## LSTC Program Download Site

<ftp://user:computer@ftp.lstc.com>

SMP Version: ls-dyna

MPP Version: mpp-dyna

SMP/Windows: pc-dyna

For Development Versions:

<ftp://beta.keyboard@ftp.lstc.com>

### 4.4.1 INTERNAL LSTC FAQ - [FTP://FTP.LSTC.COM/OUTGOING/SUPPORT/FAQ](ftp://ftp.lstc.com/outgoing/support/FAQ)

This is the most outstanding information and is a recommended "desktop-reference" for those inquisitive simulation engineers.

#### Some specific popular FAQs include:

consistent units

[ftp://ftp.lstc.com/outgoing/support/FAQ/consistent\\_units](ftp://ftp.lstc.com/outgoing/support/FAQ/consistent_units)

An overview of Contact

<ftp://ftp.lstc.com/outgoing/support/FAQ/contact.overview>

Soft Contact

<ftp://ftp.lstc.com/outgoing/support/FAQ/contact.soft1>

General guidelines for Crash Analysis

<ftp://ftp.lstc.com/outgoing/support/FAQ/guidelines.pdf>

Hourglass Control

[ftp://ftp.lstc.com/outgoing/support/FAQ/hourglass\\_condensed](ftp://ftp.lstc.com/outgoing/support/FAQ/hourglass_condensed)

Dealing with Instabilities

<ftp://ftp.lstc.com/outgoing/support/FAQ/instability.tips>

Dealing with long run times

[ftp://ftp.lstc.com/outgoing/support/FAQ/long\\_run\\_times](ftp://ftp.lstc.com/outgoing/support/FAQ/long_run_times)

Mass Scaling

[ftp://ftp.lstc.com/outgoing/support/FAQ/mass\\_scaling](ftp://ftp.lstc.com/outgoing/support/FAQ/mass_scaling)

Negative Volume in Brick Elements

[ftp://ftp.lstc.com/outgoing/support/FAQ/negative\\_volume\\_in\\_brick\\_element.tips](ftp://ftp.lstc.com/outgoing/support/FAQ/negative_volume_in_brick_element.tips)

Quasi-static simulations

<ftp://ftp.lstc.com/outgoing/support/FAQ/quasistatic>

Restarting Analyses

<ftp://ftp.lstc.com/outgoing/support/FAQ/restart>

Modeling spinning bodies

<ftp://ftp.lstc.com/outgoing/support/FAQ/spin>

Spring Back

<ftp://ftp.lstc.com/outgoing/support/FAQ/springback>

Stress vs Strain for plasticity models

[ftp://ftp.lstc.com/outgoing/support/FAQ/stress\\_vs\\_strain\\_for\\_plasticity\\_models](ftp://ftp.lstc.com/outgoing/support/FAQ/stress_vs_strain_for_plasticity_models)

User-defined materials

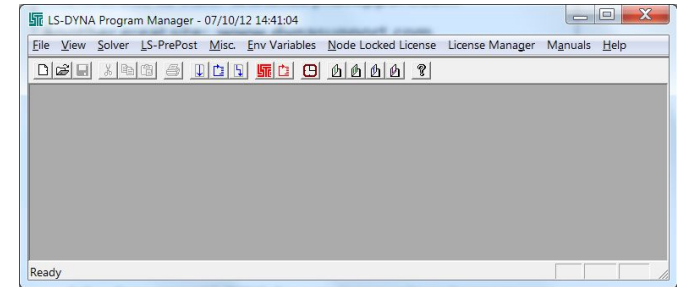
[ftp://ftp.lstc.com/outgoing/support/FAQ/user\\_defined\\_materials.faqFAQs](ftp://ftp.lstc.com/outgoing/support/FAQ/user_defined_materials.faqFAQs)

## 4.5 SUBMITTING LS-DYNA ANALYSIS JOBS AND SENSE SWITCHES

Analysis jobs can be submitted directly with command line syntax or using the Windows manger (shown on the right); however, it is only for the SMP solver and cannot run jobs using the MPP solver.

While LS-DYNA is running, the user can interrupt the analysis and request mid-analysis information. This interrupt is initiated by typing **ctrl-c** on keyboard and then a "sense switch" can be activated by typing the following:

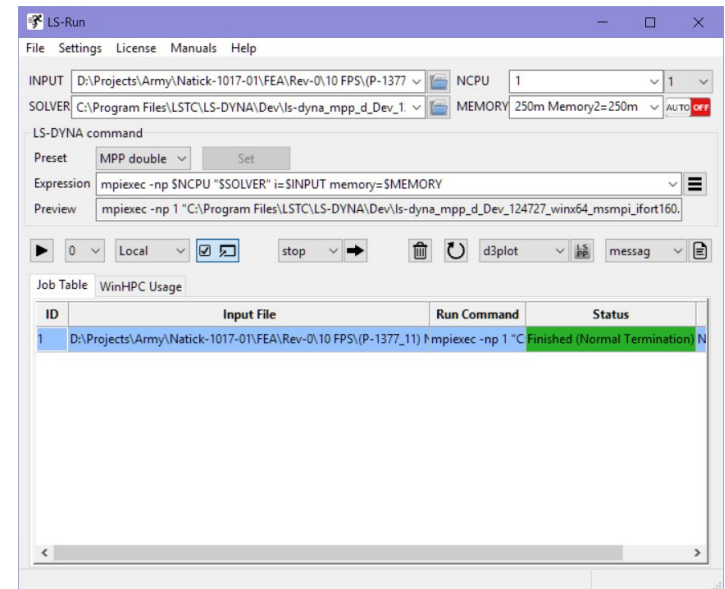
- **sw1** A restart file is written and LS-DYNA terminates
- **sw2** LS-DYNA responds with current job statistics
- **sw3** A restart file is written and LS-DYNA continues
- **sw4** A plot state is written and LS-DYNA continues
- **swa** Dump contents of ASCII output buffers
- **stop** Write a plot state and terminate



### 4.5.1 A BETTER WAY TO RUN LS-DYNA JOBS: LS-RUN

LS-Run is a LS-DYNA job tool that allows you to run jobs using multiple solvers and to queue jobs up for multiple runs. It can be downloaded from DYNAmore site but I would just do a search using the term LS-RUN. New versions come out from time to time.

*Analysts' Note: We will be using the MPP (massively parallel processing) version of LS-DYNA since it provides the greatest scalability above four CPU-Cores. We also standardize on the double-precision version since it provides the highest precision and is required for an implicit analysis. As one can guess, once you have a solution, one can seek efficiency by using the single precision version (~15% faster) or using the shared memory processing (SMP) version with four or fewer CPU-Cores (~20%).*



## 4.6 LS-DYNA OUTPUT FILES (RESULTS AND MESSAGE FILES) AND DATABASE REQUESTS AND MANAGEMENT

**Introduction:** LS-DYNA is built for speed and its file formats and file request are likewise designed for speed and the ability to efficiently handle gigantic file sizes (e.g., hundreds of GBytes). This section provides a brief overview and is not comprehensive or a replacement for the wealth of information provided in the Keyword Manual. A fundamental recommendation is to build small models and explore options with the Keyword Manual close at hand.

### Results and Message Files

File Name	Type	Description
d3plot, d3plot0#	binary	Database for entire model (stress, displacements, strain and energy information (kinetic energy, internal energy, energy ratio). When LSPP reads in the d3plot file, it automatically reads in its daughter files (d3plot01, d3plot02, .....). This file can be augmented with additional results information by *DATABASE_EXTENT_BINARY
d3hsp	ascii	This file contains an echo of the submitted Keyword Deck and provides detailed analysis statistics from contact penetration, mass values of each part, warning messages and more. This file can be read into LSPP (Misc. / D3hsp View) and a summary overview is provided. Very useful for model verification (e.g., mass of model).
mes00#	ascii	Text file of on-screen messages during analysis. This file is often requested by technical support since it provides documentation on the LS-DYNA solver used, warning messages and solution statistics.
binout00#	binary	Non-contour'able results database (e.g., energies, spcforc (reaction forces at constraints), etc.). A key advantage is the ability is to request high-frequency output of specific items and not suffer from data overload within the d3plot file.
glstat, spcforc, bndout, etc.	ascii	If one is using the LS-DYNA SMP solver (in the class we default to the MPP solver), then one can also output non-contour'able results in ascii format. This will be apparent with multiple files with characteristic names.

### Database Requests – The Minimum For Most Explicit Analyses (But Still One Must Read The Manual (RTM))

Keyword Command	Description
*DATABASE_BINARY_D3PLOT	One of the nine required Keywords for an LS-DYNA FEA model and of course, necessary if one wants to visualize the FEA results. This command creates the d3plot files.
*DATABASE_EXTENT_BINARY	The essential daughter Keyword command to _D3PLOT it controls what gets dumped into the binary file. Although optional it seems that most analyses require a few of the options contained within this Keyword to supplement the results dumped into the d3plot files. Too many to discuss and thus RTM.
*DATABASE_GLSTAT, _SPCFORC, & _MATSUM	No rule-of-thumb is provided but usually one wants to see the energies of the model (_GLSTAT), reaction forces (_SPCFORC) and individual energy, hourglass, mass scaling for each PART (_MATSUM).



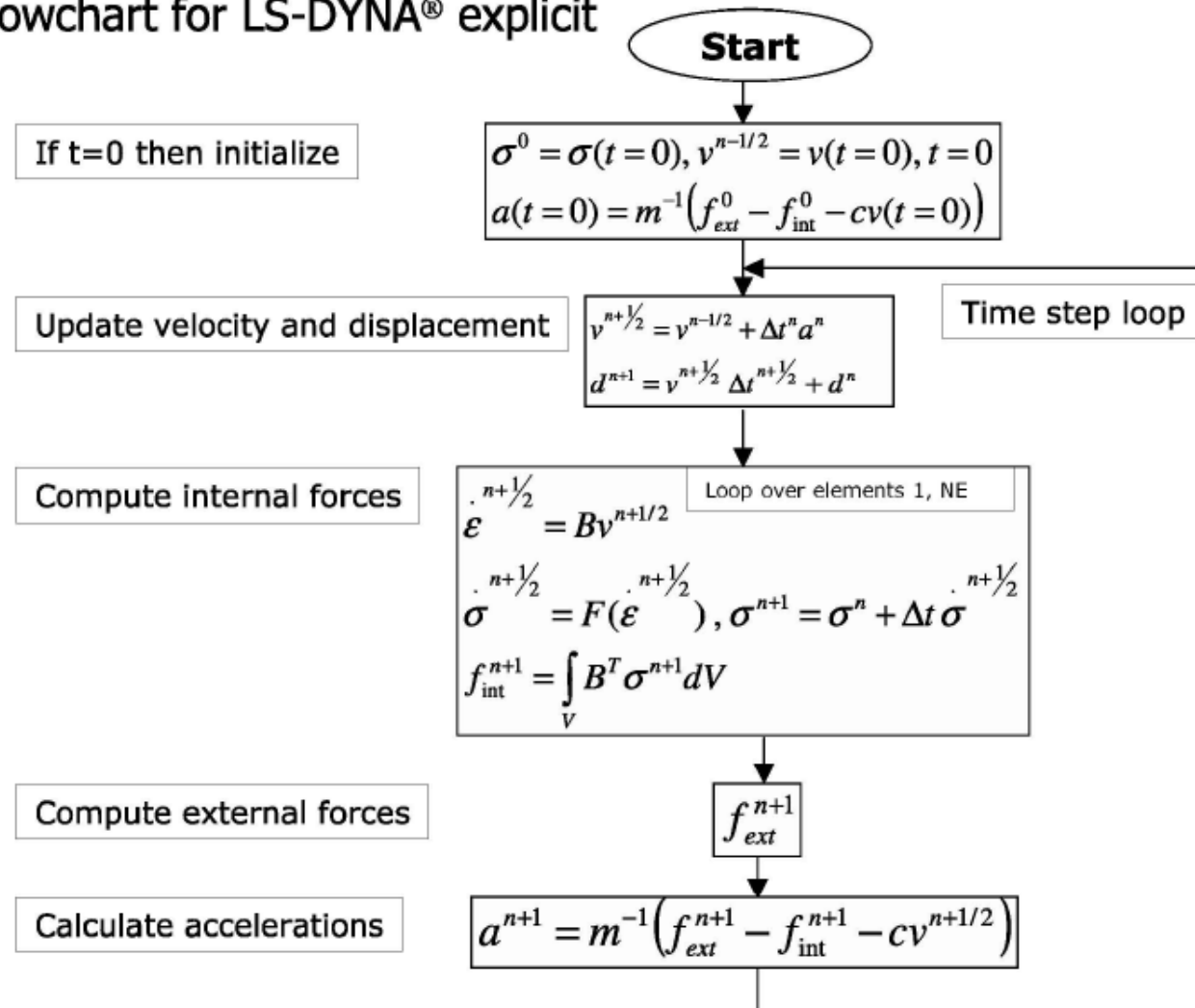




## 5. FUNDAMENTAL MECHANICS OF EXPLICIT ANALYSIS

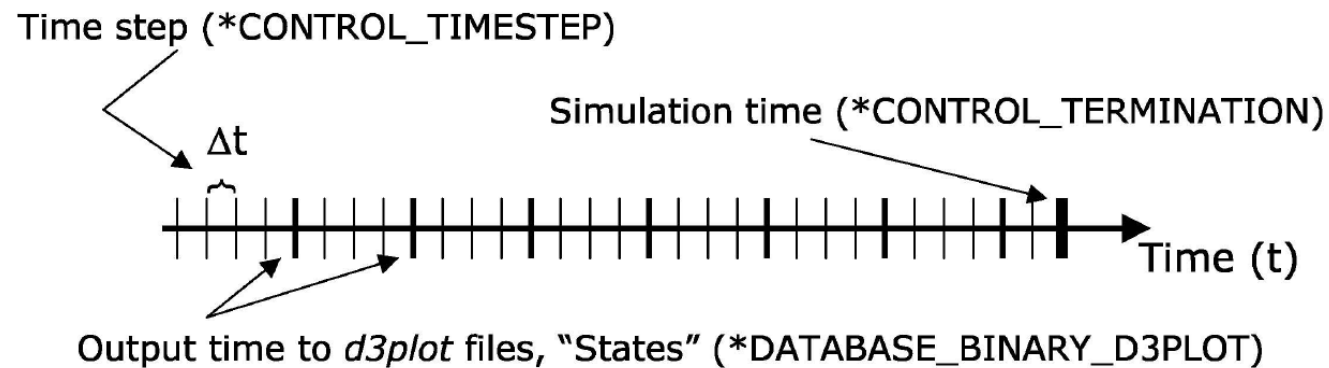
### 5.1 TIME STEP SIGNIFICANCE

#### Flowchart for LS-DYNA® explicit



### 5.1.1 EXPLICIT TIME INTEGRATION

- Very efficient for large nonlinear problems (CPU time increases only linearly with DOF)
- No need to assemble stiffness matrix or solve system of equations
- Cost per time step is very low
- Stable time step size is limited by Courant condition (i.e., time for stress wave to traverse an element)
- Problem duration typically ranges from microseconds to tenths of seconds
- Particularly well-suited to nonlinear, high-rate dynamic problems
- Nonlinear contact/impact
- Nonlinear materials
- Finite strains/large deformations



**Figure 1: How Solution Time and Result Outputs Are Defined in Explicit**

## 5.2 TIME STEP SIGNIFICANCE (COURANT-FRIEDRICHS-LEWY (CFL) CHARACTERISTIC LENGTH)

- In the simplest case (small, deformation theory), the timestep is controlled by the acoustic wave propagation through the material.
- In the explicit integration, the numerical stress wave must always propagate less than one element width per timestep.
- The timestep of an explicit analysis is determined as the minimum stable timestep in any **one (1) deformable finite element in the mesh**. (Note: As the mesh deforms, the timestep can similarly change)
- The above relationship is called the Courant-Friedrichs-Lewy (CFL) condition and determines the stable timestep in an element. The CFL condition requires that the explicit timestep be smaller than the time needed by the physical wave to cross the element. Hence, the numerical timestep is a fraction (0.9 or lower) of the actual theoretical timestep. Note: the CFL stability proof is only possible for linear problems.
- In LS-DYNA, one can control the time step scale factor (TSSFAC). The default setting is 0.9. It is typically only necessary to change this factor for shock loading or for increased contact stability with soft materials.

$$C_{AcousticWaveSpeed} = \sqrt{\frac{E_{Material}}{\rho_{Material}}}$$

$$\Delta ExplicitTimestep = \frac{Length_{Element}}{C_{Wavespeed}}$$

$$\Delta Timestep_{CFL} = (0.9)\Delta ExplicitTimestep$$

*Analyst's Note: Based on this conditions, the time step can be increased to provide faster solution times by artificially increasing the density of the material (e.g., mass scaling, lowering the modulus or by increasing the element size of the mesh).*



### 5.3.1 WORKSHOP: 2 - LS-DYNA MASS SCALING BASICS

#### What You Will Learn:

Simple class exercise to reinforce the concept of mass scaling basics and how to view the explicit time step within LSPP.

The model is just a simple plate that is hit with a short force pulse along its bottom edge. This force pulse then propagates a stress wave through the bar. The physics are classic and your job is to manage the explicit time step.

#### Tasks:

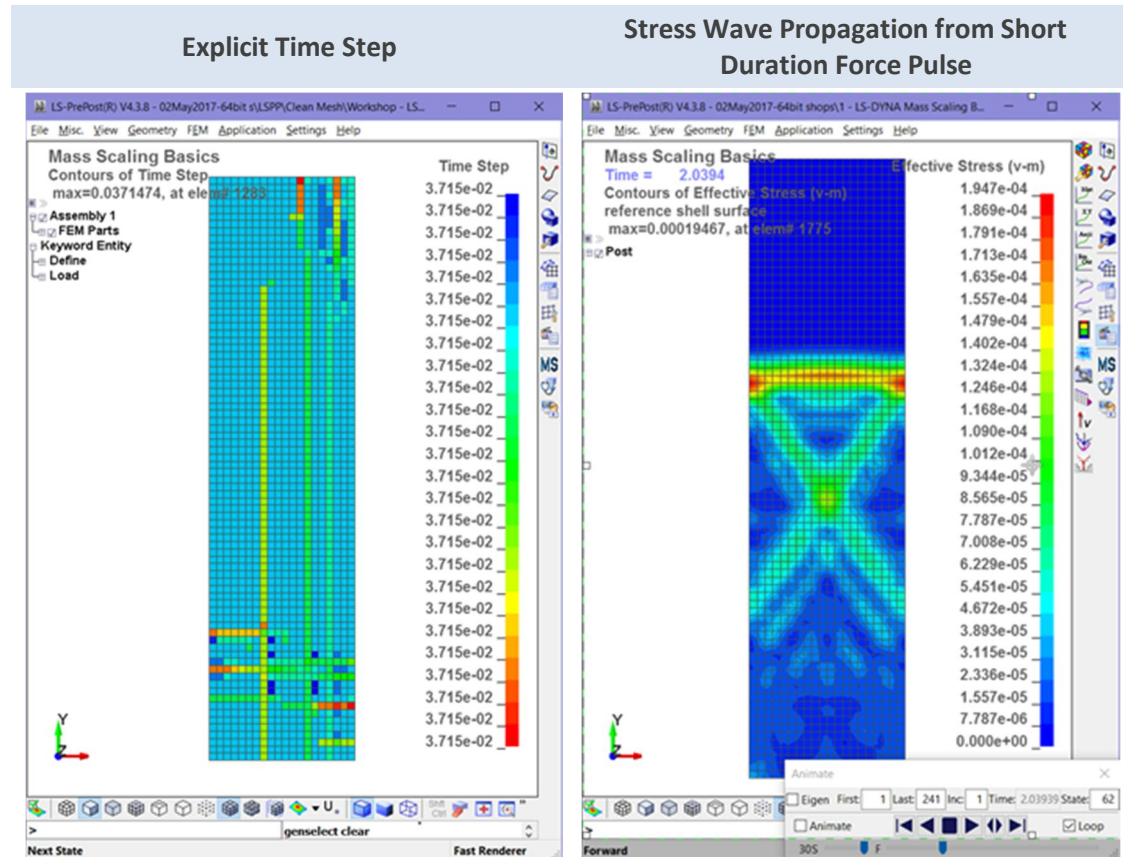
- Open Keyword deck: /LS-DYNA Mass Scaling Basics / LSPP / Clean Mesh / LS-DYNA Mass Scaling Basics - Clean.dyn in LSPP. Verify elastic isotropic material (\*MAT\_ELASTIC) properties and then shell property (\*SECTION\_SHELL) with elform=2 and thickness = 1.0.
- Check explicit time step using LSPP's command Application / Model Checking / General Checking / Element Quality / Shell check item / *check* Time step.
- Change elastic modulus from 70 to 35 and re-contour time step.
- Submit the model for analysis using LS-Run with the solver set to MPP Double-Precision

*Analysts' Note: Although this model would run faster using SMP single-precision, we are aiming to keep it simple and use the most robust solver platform. As always, once a solution is in hand, one can seek efficiencies using SMP when your number of CPU-cores is limited to less than four or to use single precision if the run is not using the implicit solver.*

**Units:** kN-mm-ms-kg (stresses in GPa)

**Linear, Elastic Material:**

E	$\nu$	$\rho$
70	0.33	2.71e-6



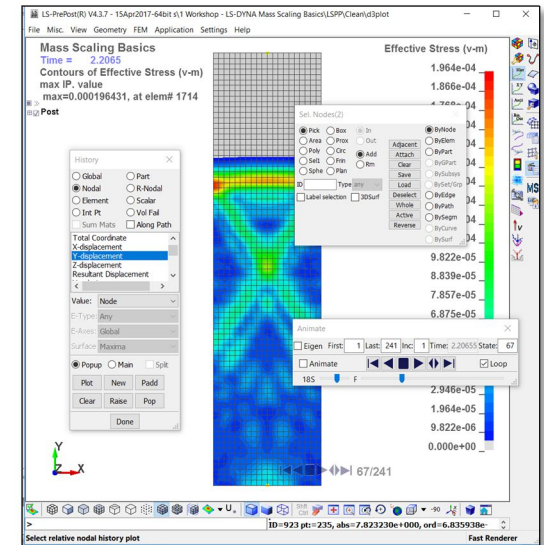
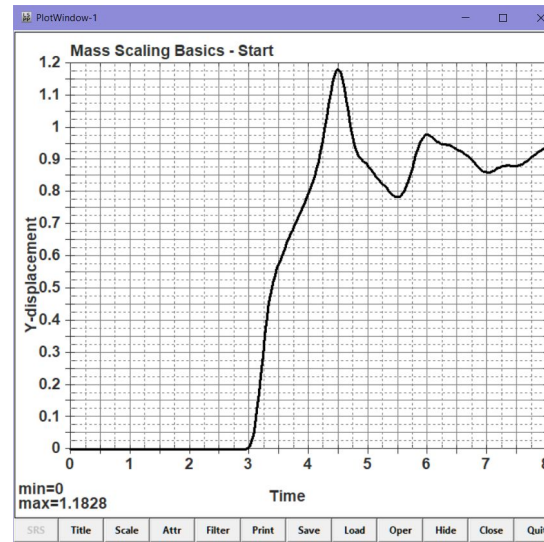


## WORKSHOP: 2 – LS-DYNA MASS SCALING BASICS (CONTINUED)

With the model working, let's harvest some data. We are going to make several runs of this model to investigate the relationship between mesh, explicit time step and mass scaling. As part of this process, you'll get comfortable working with LSPP and LS-Run. Our metric is going to be the maximum displacement from a node at the end of the bar (Node #1).

### Tasks:

- Within existing LSPP model, open History, select Node, Y-Displacement and then pick the node at the very top of the bar (node #1) and hit Plot. (Note: The node is attached is a rigid link or in 'DYNA'ese a constrained nodal rigid body).
- Note that the maximum displacement at the top of the bar is 1.18 mm.



### Documenting the Learning Objective:

Open the Keyword Deck LS-DYNA Mass Scaling Basics - Skewed Mesh - Start.dyn in your favorite text editor and apply conventional mass scaling (CMS) to the \*CONTROL\_TIMESTEP keyword card via the dt2ms option. The idea is to match the original time step in the clean mesh example and understand that mass scaling is invaluable but alas has drawbacks (i.e., one should carefully check your results).

*(Note: Remember that the tssfac=0.9 and thus to get an explicit step of 0.0334, one must use a value of dt2ms=-0.0371.)*

Model	Time Step	% Mass Added by Mass Scaling	Max. Displacement
Starting Point	0.0334 ms	0.00%	1.18 mm
Skewed Mesh (-4x)	0.0184 ms	0.00%	mm
Skewed Mesh with Mass Scaling I	0.0334 ms	%	mm
Skewed Mesh with Mass Scaling II	0.0290 ms	%	mm

### 5.3.2 INSTRUCTOR LED WORKSHOP: 2 - MASS SCALING ADVANCED

#### Explicit Time Step Mass Scaling (\*CONTROL\_TIMESTEP)\*

- Mass scaling is no free lunch. For dynamic systems, added mass can affect the response of the system (i.e., like additional *un-wanted* KE).
- It is just something to monitor and make an engineering judgment about its effectiveness; time savings versus potential detrimental effects. Mass scaling is my universal modeling condiment and the aim is typically no more than 5% additional mass.

*Analyst Question: Would mass scaling make your dynamic ( $F=ma$ ) analysis more conservative or less?*

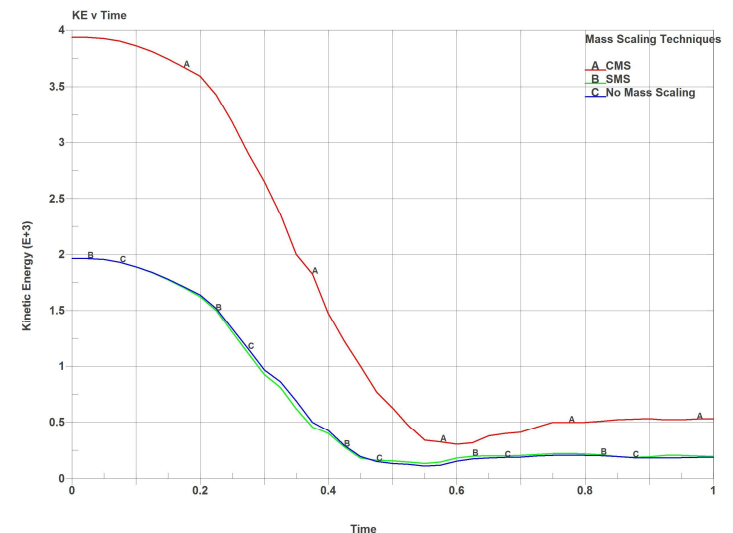
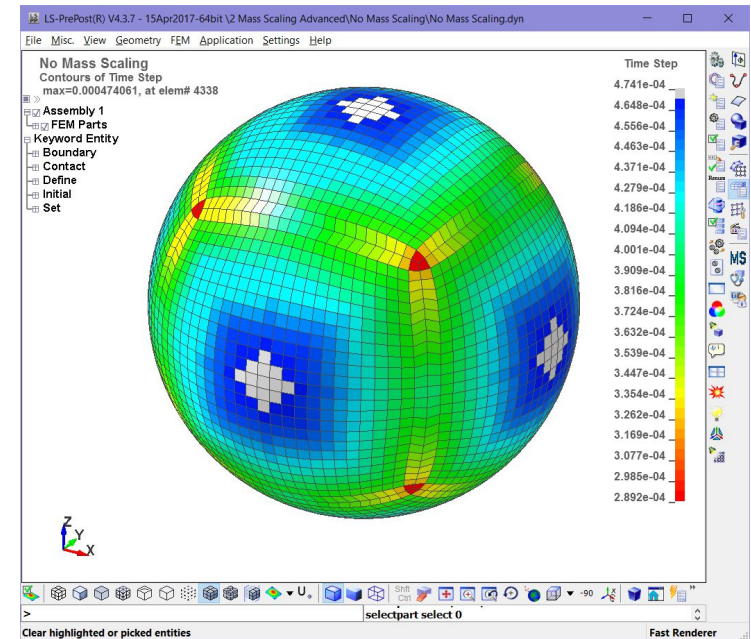
- Conventional mass scaling (CMS) has morphed to using the negative (-)DT2MS option as the recommended default.
- Selective mass scaling (SMS): Using selective mass scaling, only the high frequencies are affected, whereas the low frequencies (rigid body bodies) are not influenced; thereby, a lot of artificial mass can be added to the system without adulterating the global solution.
- This method is very effective, if it is applied to limited regions with very small critical timesteps. SMS is invoked with the IMSCL command over a single part or multiple parts.

*Analyst Note: Please understand that CMS is used **on all other parts** not called out within the IMSCL command (see Keyword Manual)*

Solution time is 28 seconds for no mass scaling and 15 and 13 seconds for SMS and CMS respectively. SMS is more computationally expensive but has large benefits for some models.

*Example Courtesy of [www.DynaSupport.com](http://www.DynaSupport.com)*

Time step ranges from 2.89 to 4.74e-4



## 5.4 IMPLICIT MESH VERSUS EXPLICIT MESH CHARACTERISTICS

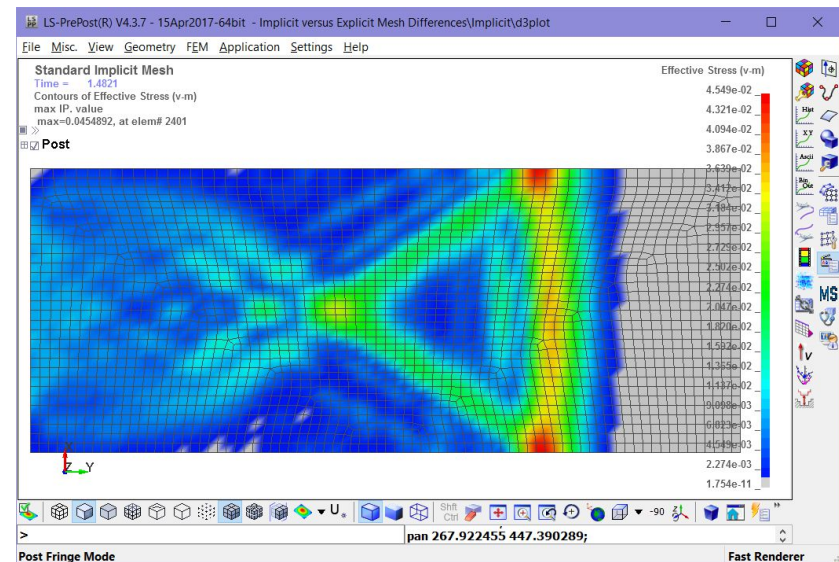
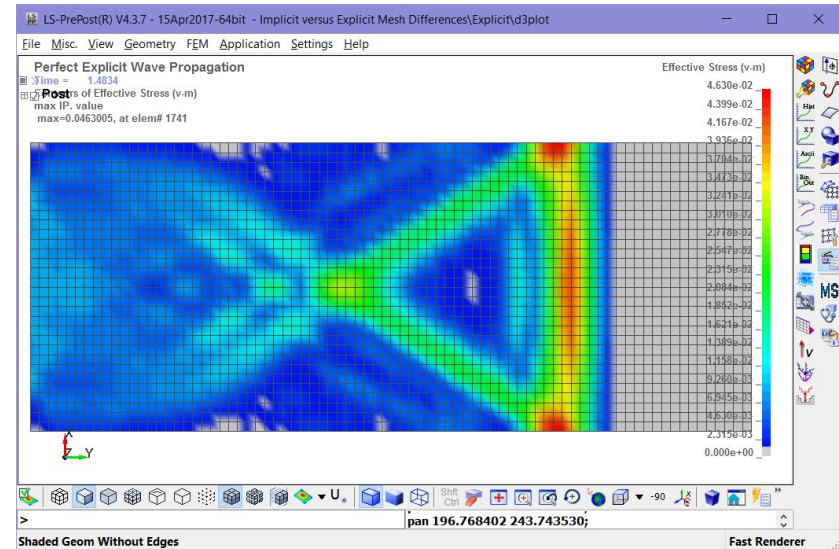
### 5.4.1 INSTRUCTOR LED WORKSHOP: 3 - IMPLICIT VERSUS EXPLICIT MESH DIFFERENCES

#### Meshing for Accuracy

- Solution time (number of nodes + time step) is often one of the most important considerations in setting up an explicit analysis; care should be exercised in setting up the mesh density.
- A good implicit mesh **does not** typically work well for an explicit analysis.
- In an explicit analysis, linear, elastic stresses are not often the most important analysis result. Typically, plastic strain, energy, crushing behavior, etc. are more important. These results are not as mesh sensitive as linear, elastic stresses and permit a much larger element size to be used.

Since the time step is controlled by wave propagation, the mesh should be graded gradually to likewise allow a smooth wave propagation through the structure whenever possible.

*Analyst's Note: Mass scaling is great but it needs to be combined with a reasonable mesh gradient.*





## 6. EXPLICIT ELEMENT TECHNOLOGY

### 6.1 ELEMENT TYPES IN LS-DYNA

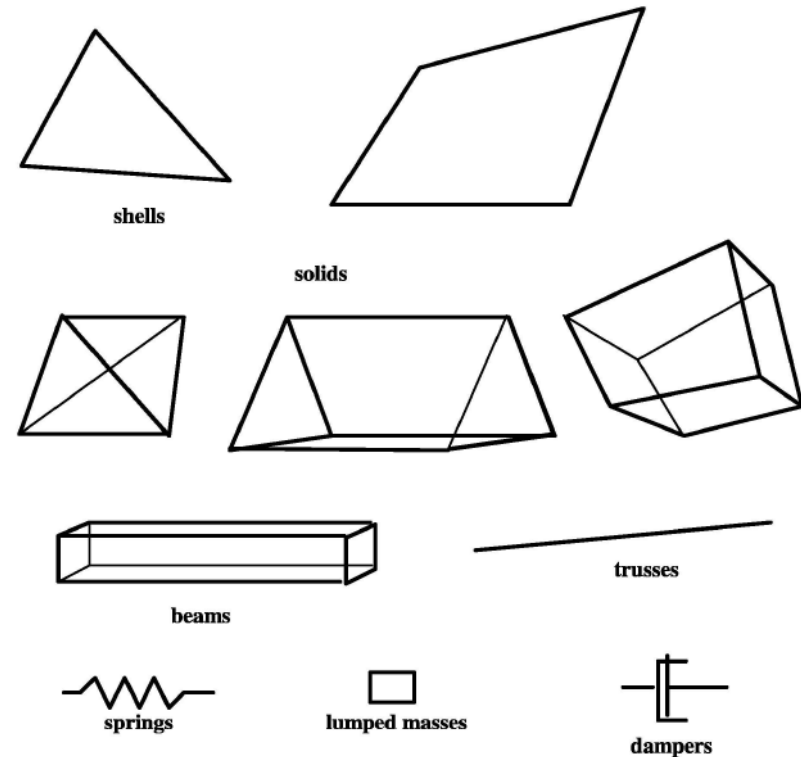
#### Element Toolbox:

If it numerically exists, then LS-DYNA most likely has it:

- Point elements (mass, inertia)
- Discrete elements (springs, dampers)
- Beams, cables, discrete-beams, etc.
- Solids (2D and 3D, Lagrangian, Eulerian, ALE)
- Shells
- Thick Shells (8 noded)
- Cohesive elements
- Seatbelts (and related components)
- EFG and SPH (meshless)

#### Extremely Brief Recommendations:

- Hughes-Liu Integrated Beam, ELFORM=1, is default. Stresses are calculated at the mid-span of the beam. Special requirements for stress output.
- For solid elements, the default is ELFORM=1 and uses one-point Gaussian Integration (constant) stress. This element is excellent for very large deformations. It is the standard recommend for explicit simulations.
- Shell elements are covered in detail.



#### Detailed Element Recommendations (see Student's Class Reference Notes)

Review of Solid Element Formulations Erhart.pdf

Aerospace Working Group - Aerospace\_MGD\_v16-2.pdf

## 9.7 CONTACT BEST PRACTICES

Recommendation	Why
<code>_AUTOMATIC_SINGLE_SURFACE_MORTAR</code>	It is the most robust and <code>_SINGLE_SURFACE</code> as a default will track penetrations and slowly remove them. Once your explicit simulation is stable, one can always remove <code>_MORTAR</code> later on for a speed improvement.
<code>_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR</code>	If high negative sliding interface energy is noted, it is useful to break up your contacts into part-to-part or {part set}-to-{part set} via <code>SURFACE_TO_SURFACE</code> to pinpoint the contact region that is causing problems.
<code>_AUTOMATIC_SINGLE</code> or <code>SURFACE_TO_SURFACE</code> w/ <code>soft=2</code>	This is our recommended backup if greater numerical efficiency is required. The <code>soft=2</code> option on the optional A card switches the contact formulation to a segment base and adjusts the contact stiffness for better contact. Likewise, <code>IGNORE=1</code> (However, it is set to <code>=1</code> as default when <code>SOFT=2</code> ).
Check your model for penetrations (LSPP – Application / Model Checking / General Checking / Contact Check). Keep in mind that <code>_MORTAR</code> , as a default, automatically accommodates small penetrations in explicit ( <code>IGNORE=1</code> ).	Be aware of their magnitude since they contribute to negative sliding interface energy, poor contact behavior and spurious contact forces.
For contact between deformable bodies, aim for a uniform mesh pattern.	Contact works by segments and transfers contact forces to the adjacent nodes. If the mesh is course, the contact forces will be high and dynamic. The smoother the patter the better the contact.
Verification	Why
<code>*CONTROL_ENERGY, SLNTEN=2</code>	Automatically calculated when contact is present although its request is provided directed within <code>*CONTROL_ENERGY</code> . Provides one small verification that contact is working well. Global (all contacts) are provided in GLSTAT while individual contact sliding energies are provided in a separate ASCII file under SLEOUT.

## 9.8 MESH TRANSITIONS: TIED CONTACT FOR WELDING AND GLUING

### 9.8.1 TIED CONTACT OR GLUING

Given the idealization difficulty of system modeling, the ability to tie together different mesh densities (e.g., hex-to-hex or tet-to-hex), snap together parts along a weld-line or just glue sections together (e.g., plate edge to a solid mesh) is an amazingly useful ability and LS-DYNA provides a very complete Tied Contact tool box to work with.

The emphasis of this course to provide an overview of the basics to get started efficiently with LS-DYNA, a short list of recommended \*KEYWORDS for Tied Contact are presented that work for both implicit and explicit solution sequences.

#### When the Mesh is Co-Planar (Translational DOF Tied)

- \*CONTACT\_TIED\_SURFACE\_TO\_SURFACE
- \*CONTACT\_TIED\_NODES\_TO\_SURFACE

#### When the Mesh is Co-Planar (All Six DOF Tied)

- \*CONTACT\_TIED\_SHELL\_EDGE\_TO\_SURFACE

#### When the Mesh is Offset (All Six DOF Tied)

- \*CONTACT\_TIED\_SHELL\_EDGE\_TO\_SURFACE\_BEAM\_OFFSET

The utility of using a very brief subset is that one can build up experience and confidence without the expense of trying out a rather daunting list of tied contact Options.

#### Theory Into Practice

Tied contacts are not really “contacts” but a constraint or penalty relationship that uses the \*CONTACT card entry format. For reference, the slave side is tied to the master side; that is, the slave nodes follow the master nodes and thus the slave side should be the finer mesh since it is theoretically possible for a master side node to penetrate the slave side surface. For an explicit analysis, the default tied option ties the slave to the master accelerations (see **Theory Manual**) while for implicit, the displacements are tied. This explains why the default constraint method can’t be used to tie rigid bodies or have SPCs attached to any \_TIED slave node.

With the \_OFFSET formulation, the penalty method is used. If the \_SHELL\_EDGE option is also employed all six DOF’s are tied together by essentially using little springs between the nodes. Since the tied contact is enforced by springs, the nodes can be offset, rigid bodies can be tied together and even SPCs can be applied to the tied interface nodes. The downside to the penalty formulation is that it may create spurious negative sliding interface energy. To correct this pathology, it is recommended to use a modern formulation with \_BEAM\_OFFSET.

## 9.8.2 SUMMARY TABLE FOR TIED CONTACT

{Before reading this page and wondering how to ensure the correct application, one should jump forward to the next page.}

{PS: Kept this page and the prior discussion to emphasize how fast LS-DYNA is changing and how important it is to stay informed and on-top of new advances.}

*CONTACT_TIED_	Recommended Usage	Type	Pros/Cons
_SURFACE_TO_SURFACE	Gluing solid mesh transitions together where the two meshes are co-planar/adjacent.	Constraint	<b>Pros:</b> Provides smooth displacement and stress interpolation across dissimilar meshes between hex-to-hex or tet-to-hex. <b>Cons:</b> If the meshes are co-planar, there are no cons.
_NODES_TO_SURFACE	Useful for creating weld edge lines between two solid elements parts that are co-planar; simulates a “fillet weld”. However, is identical to _SURFACE_TO_SURFACE except one uses nodes on the slave surface.	Constraint	<b>Pros:</b> Allows the logical modeling of edge contact between solid parts. If the meshes are co-planar, there are no cons. <b>Cons:</b> Not to be used with plate or beam elements since rotational DOF are not correctly handled.
_SHELL_EDGE_TO_SURFACE	Welding plate or beam nodes together when the mesh is co-planar and captures all six DOF.	Constraint	<b>Pros:</b> Handles all six DOF’s using a constrain method. <b>Cons:</b> Not designed for solid elements.
_SURFACE_TO_SURFACE_OFFSET	Gluing solid mesh transitions when the meshes are not co-planar.	Penalty	<b>Pros:</b> Allows one to glue together dissimilar meshes that are offset. Also one may glue deformable bodies onto rigid bodies. <b>Cons:</b> Not suitable for plate and beam connections. Old formulation.
_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET	Ideal for welding together plates/beams or for plates/beams onto solids whether deformable and/or rigid with offsets.	Penalty	<b>Pros:</b> The grand slam of tied contact. Handles all six DOF and offsets. Typically very numerically stable with little negative sliding interface energy. Also may glue deformable bodies onto rigid bodies. <b>Cons:</b> None except that it is penalty based.

### 9.8.2.1 Summary Table for New Approach to \_TIED Contact

Now, after years of wondering why it was necessary to have so many different \_TIED contact formulations, it was reasoned that at the end-of-the-day, one could handle the full range of \_TIED possibilities with two \_TIED contact formulations. This is the new go-forward approach for both explicit and implicit analyses. Although the two Workshops are using the older formulations, the student can swap out these formulations after a successful completion and demonstration to themselves that the same results can be obtained.

Keyword	Card Option	Comment
_TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET	IPBACK=1	General purpose _TIED contact for tying solids (3 DOF nodes) to shells and solids. If the tie interface is coplanar or offset, the _OFFSET feature handles both situations. Plus being _CONSTRAINED it eliminates any problems with a spring formulation. The IPBACK=1 option is useful if one needs to tie to rigid bodies. If a rigid body is present in the tie definition, then the formulation is automatically switched to a penalty based algorithm.
_TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET	IPBACK=1	With SHELL_EDGE option, one ties all 6 DOF of the slave side nodes and if one needs to tie to rigid bodies, the IPBACK=1 option is available.

*Analyst's Note: This recommendation comes after many years of \_TIED introspection and troubleshooting. It is a work in progress but given the maturity of the LS-DYNA program, the go-forward quest is to only use only two \_TIED formulation to handle all tied contacts. The advantage of the \_CONSTRAINED formulation is that springs are not use and the constraint relationships are created between the adjacent surfaces. This eliminates the possibility of forming negative sliding interface energy and assures a clean tie that will work well for explicit and implicit analyses.*

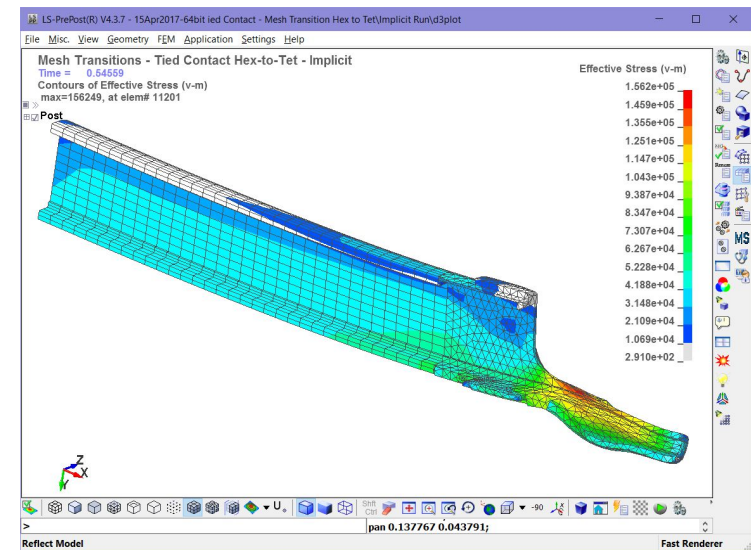
### 9.8.3 WORKSHOP: 15 - TIED CONTACT FOR HEX-TO-TET MESH TRANSITIONS (TIED\_SURFACE\_TO\_SURFACE)

#### Background for TIED Contact Analysis:

The advantages of using a hex mesh for explicit work centers on better shape control during large deformation and the ability to maintain a larger time step. The last item is often pivotal in keeping your solution running fast without having the program add excessive mass if automatic mass scaling is invoked (\*CONTROL\_TIMESTEP (DTMS = negative timestep value)). In the implicit world, the use of hex elements are desired for the ability to provide equivalent stress mapping using far less nodes (eight node brick versus the use of five 10-node tetrahedrals to fill the same space or 8/26 nodes) and often times, cleaner stress contours. This workshop shows how to setup the mesh transition and run the analysis using the implicit solution.

#### Tasks:

- Start with opening up the Keyword deck: Tied Contact - Mesh Transition Hex to Tet - Start.dyn and inspect the Keywords. It is setup for an explicit run. Input the settings required to enable `_TIED_SURFACE_TO_SURFACE`. Our recommendation is by \*PART id.
- Run model and wait for it to finish. Better idea, apply CMS. What value? Open model in LSPP and assess the time step Application / Model Checking / etc. Apply reasonable value of -2d2ms that will give you something under <10% mass scaling. Run model.
- Inspect model, check applied force (plot SPCFORC) and look at ratio of kinetic/internal energy. Is the model at equilibrium?
- Now, let's verify that the `_TIED` contact is working as intended. Open up model in LSPP then go to Application / Model Checking / General Checking / Contact Check and pick Tied and highlight the `_TIED` contact and click on the Check button. One should see that the `_TIED` interface is not as one might suspect with more nodes `_TIED` then desired. This can be fixed in two ways, reverse the interface and make the Hex elements the slave or better yet, tell the `_TIED` contact not to search so deep during the tying process (MST=-1e-5).
- Time to clean up the model. Inspect mess0000 file and notice warnings. To fix this one has to create a node or segment set to restrict the slave side.



*Analyst's Note: The prior model was run as implicit (see folder Implicit) and if one finishes the prior workshop quickly, one can inspect the Keyword deck and learn a bit more about implicit.*

Done Early? Reset the `_TIED` interface to the recommended usage and save the file in another folder (as to not overwrite the existing d3plot files and compare the results).

### 9.8.4 WORKSHOP: 16 - TIED CONTACT FOR GLUING THINGS TOGETHER (BEAM\_OFFSET)

**Objective:** To understand how tied contact can easily glue structures together that are not coincident. There is much to learn and there are limitations. But if one understands the theory, it leads to confidence in how the tying or gluing is done.

#### Tasks:

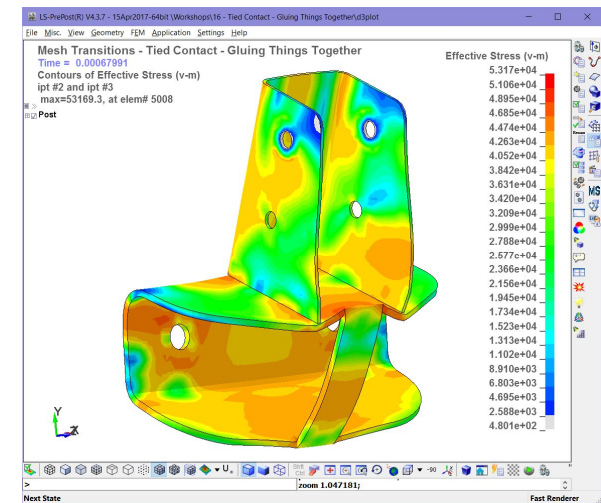
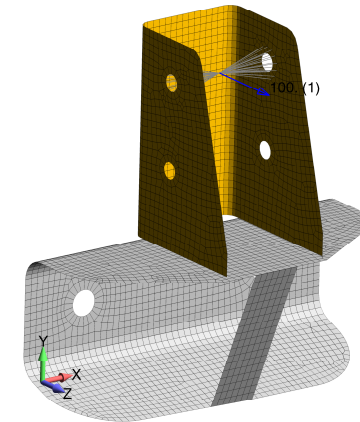
- Open up the start file: Tied Contact - Gluing Things Together - Start.dyn. It is set up to run minus the `_TIED` contact definition. That is your job. There is a predefined slave node set and one can use the Part id for the master side definition. Since it is shells with six dof and the surface is offset, these hints should lead you to choose the correct `_TIED` setup. The real question is whether one uses the `_OFFSET` method or the plain vanilla non-offset method?
- For Curiosity, use a non-offset method and see what happens to the slave side. Then reset the `_TIED` contact method and use an `_OFFSET` method. The – Finish `_OFFSET` and – Finish non-offset are both setup to provide guidance if your intuition and research fails you.

Non-offset: slave nodes are moved to master surface (if within reach, if not one can set `sst` and `mst` to a negative number that reaches)

`_OFFSET`: does not move slave nodes but they have to be in reach.

- With Any luck, you'll see something like this when you are done.

Done Early? Rework the `_TIED` formulation to that of the recommended formulation and verify that it works as intended.



*Analyst's Note: The Tied option considers a node "tied" if it is within 5% of the element's thickness. This applies to all `_TIED` formulations. As mentioned, the constraint option moves the slave node to be adjacent to the master surface while the offset option accounts for the gap; but whether or not it is tied, depends upon the separation of the nodes. To override the default setting, one can set the `SST` to a negative number that reflects the absolute distance to search for a tie relationship between the slave and master nodes.*

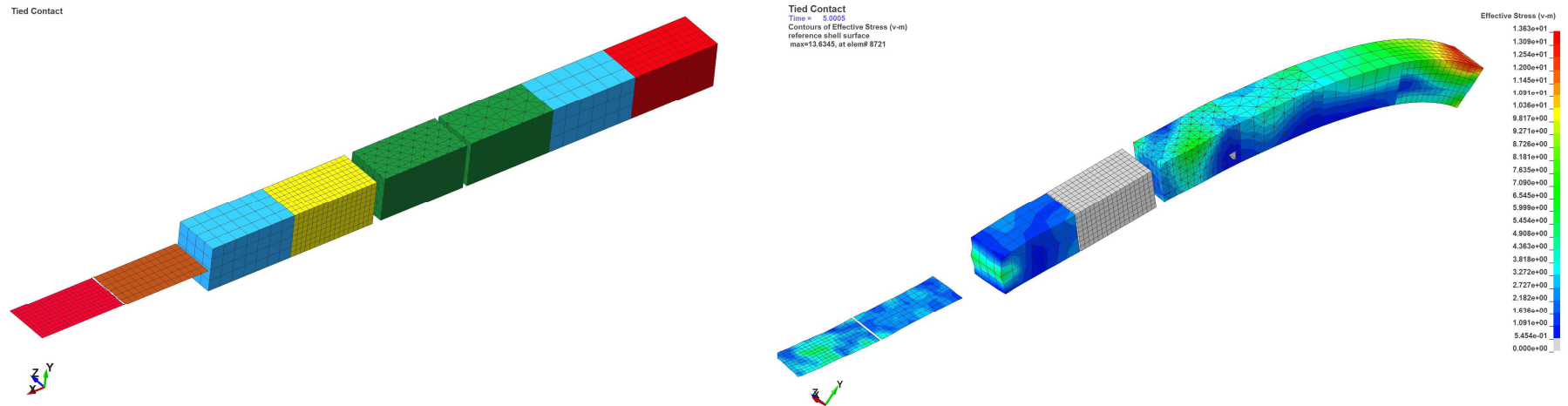


### 9.8.5 MESH TRANSITIONS: MASTERING TIED CONTACT (STUDENT EXTRA CREDIT)

#### Master Class \_TIED Example:

Tied contact is so useful and so powerful that it merits a bit of exploration on its many feature. Tied contact can be used to idealized a glue bond (\_TIEBREAK) or to glue a deformable body onto a rigid body (using a tied penalty formulation, i.e., \_OFFSET) or to cleanly attach a shell edge (6 DOF) onto a face of some solid elements (3 DOF).

The model provided below in a start (no \*CONTACT\_TIED\_... defined) and then in a Finish format covers a broad range of \_TIED applications.

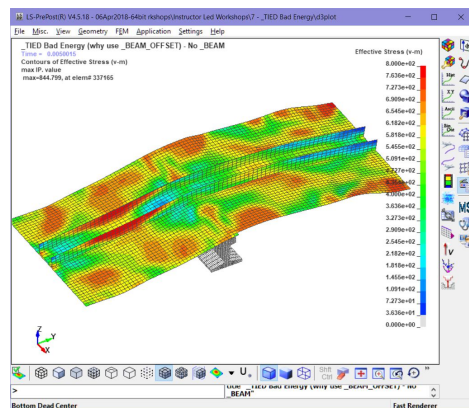
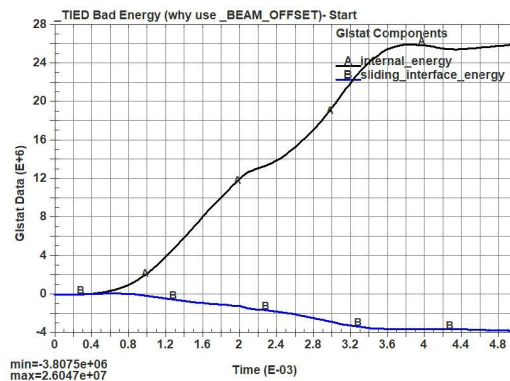


## 9.8.6 INSTRUCTOR LED WORKSHOP: 7 - \_TIED BAD ENERGY (OR WHY WE USE \_BEAM\_OFFSET)

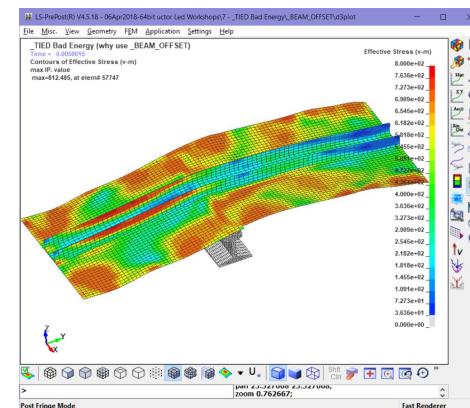
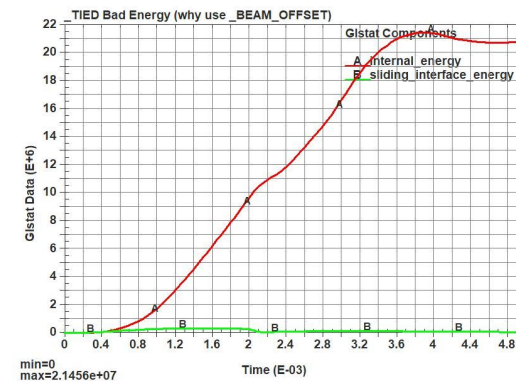
If anything this little dialog is to remind myself to be careful with Tied Contact's with "OFFSET". As mentioned, the Offset option indicates that the algorithm is using the penalty method to enforce the locked motion between parts. When there is "penalty" one has the opportunity to create negative sliding interface energy since springs are used to enforce the locked positions. This behavior killed a rather simple analysis. It was a bit amazing how it completely changed the behavior of the structure. The fix is just to change the contact to \_BEAM\_OFFSET.

The two models are provided. Which model is right? {on the full model it was a complete FUBAR}.

**\*CONTACT\_TIED\_SHELL\_EDGE\_TO\_SURFACE\_OFFSET\_TITLE**



**\*CONTACT\_TIED\_SHELL\_EDGE\_TO\_SURFACE\_BEAM\_OFFSET\_TITLE**



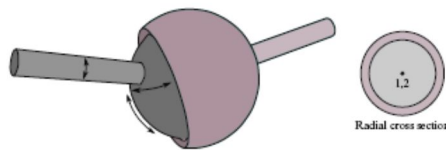
*Analyst's Note: Why din't we just use \_BEAM\_OFFSET as the default? It was an inherited model and I wasn't quite aware that \_BEAM\_OFFSET is the recommended go-forward formulation by LSTC and hey, how could a \_TIED contact kill your model anyway?. It pays to understand your craft and stay abreast of new developments since this problem cost me a couple of days.*

## 10. CONNECTIONS VIA JOINTS

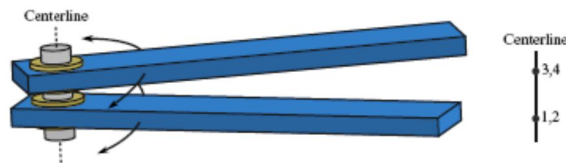
### 10.1 JOINTS OR \*CONSTRAINED\_JOINT\_

To model the motion and likewise, large movements in engineering systems, one needs joints. LS-DYNA has a very sophisticated set of commands that will allow one to model many types of common joints (e.g., hinges, spherical bearings, etc.). They are not that hard to setup if one just goes slow and perhaps build small pilot models of each joint that they are trying to simulate since debugging a large model can be laborious.

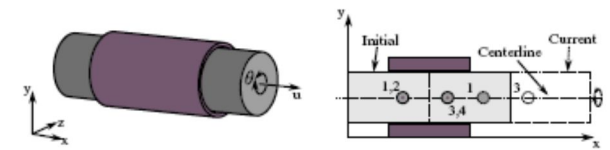
Spherical Joint



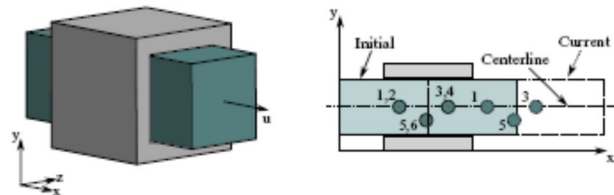
Revolute Joint



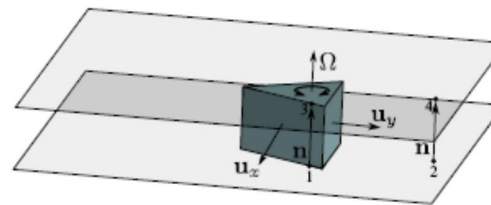
Cylindrical Joint



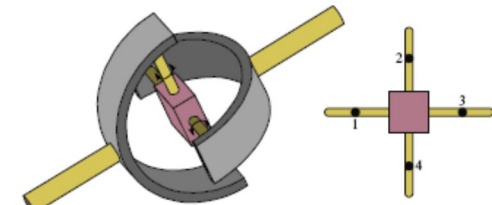
Translational Joint



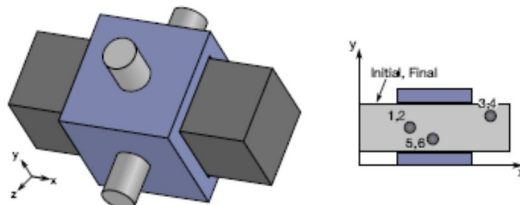
Planar Joint



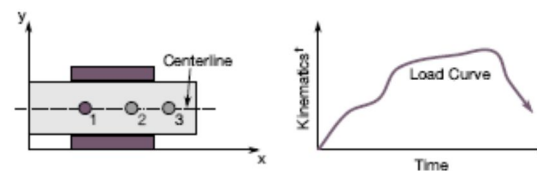
Universal Joint



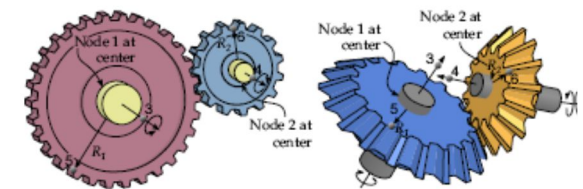
Locking Joint



Translation Motor Joint



Gear Joint, etc.



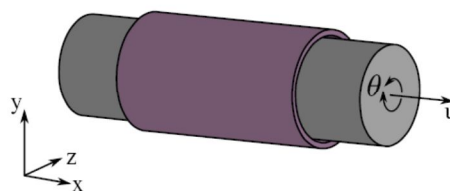
## 10.2 HOW JOINTS WORK

The foundation of joints lie in the use of \*CONSTRAINED\_NODAL\_RIGID\_BODIES (CNRB) to provide the framework for the action of the joints. The joint mechanical behavior is implemented using the penalty method (i.e., like contact). As such, a joint has a stiffness and energy. The stiffness of the joint is calculated based on more math that I want to describe in this brief note ([Please see LS-DYNA Theory Manual 2014 for the complete description](#)) what is directly of importance is that the joint stiffness is based on the mass of the joint's CNRB and their geometry, and that the reaction forces to enforce the joint's behavior, are applied at the center-of-mass of the CNRB.

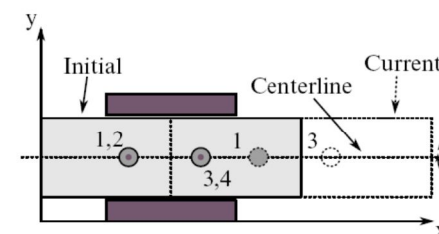
*Analyst's Note: Given that joints use the penalty method, one can have joint failure (i.e., unexpectedly flying apart) by using CNRB's with too little mass and having the opposing CNRB's center-of-mass to close to resist displacements/moments. One may want to read the Keyword \*CONTROL\_RIGID for some interesting notes on CNRB w.r.t. joints and mass scaling.*

The simplest joint is the spherical joint which connects two coincident nodes between two CNRB's. A more interesting joint is the \*CONSTRAINED\_JOINT\_CYLINDRICAL which requires two colinear nodes within the opposing CNRB's. Please note that the node sets are separated by some reasonable distance to ensure that the joint doesn't "blow-up". What is reasonable? It depends on the mass of the CNRB's and the magnitude of the force that is applied to the joint. If this joint is expected to receive a high bending load (MZ), then it would be good to ensure a large separation between the node sets. There are no hard guidelines but the reality is that most joints function just fine.

**\*CONSTRAINED\_JOINT**



**\*CONSTRAINED**



**Figure 10-13. Cylindrical Joint.** This joint is derived from the rotational joint by relaxing the constraints along the centerline. This joint admits relative rotation and translation along the centerline.

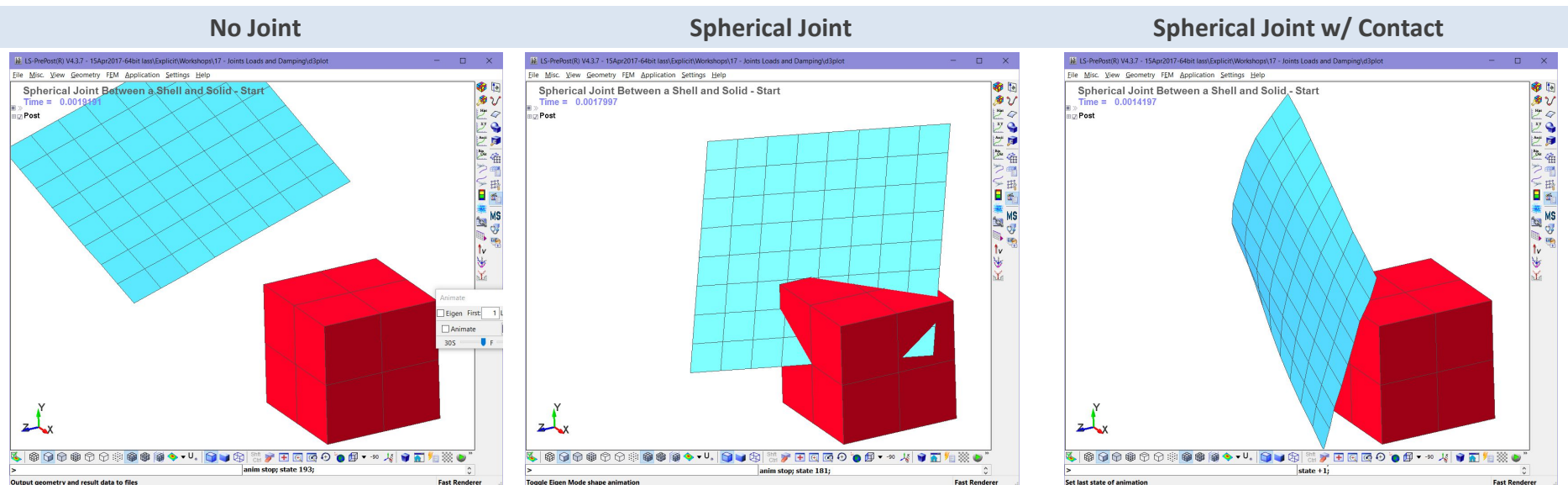
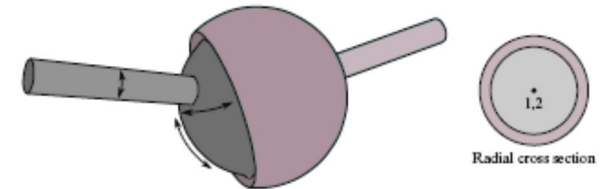
### 10.3 WORKSHOP: 17A – SPHERICAL JOINT BETWEEN A SHELL AND SOLID

**Objective:** Understand how a spherical joint is setup and how it functions.

**Background:** The spherical joint is the easiest to setup of all the joints. All you need is two rigid bodies (Constrained Nodal Rigid Body (CNRB) having one node from each CNRB coincident with each other. As with all joints, try to ensure that the CNRB is well distributed (it has some spatial reach and mass).

#### Tasks

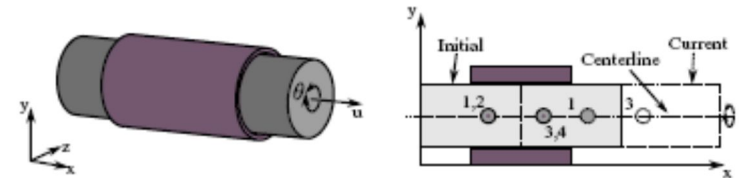
1. Open up Spherical Joint Between a Shell and Solid - Start.dyn and inspect the deck.
2. Read the Keyword Manual for \*CONSTRAINED\_JOINT\_SPHERICAL and then setup your joint.
3. Run the model and see if it makes sense.
4. Add contact to the model and rerun.



## 10.4 WORKSHOP: 17B - CYLINDRICAL JOINT BETWEEN TWO NESTED CYLINDERS

**Objective:** Understand how a spherical joint is setup and how it functions.

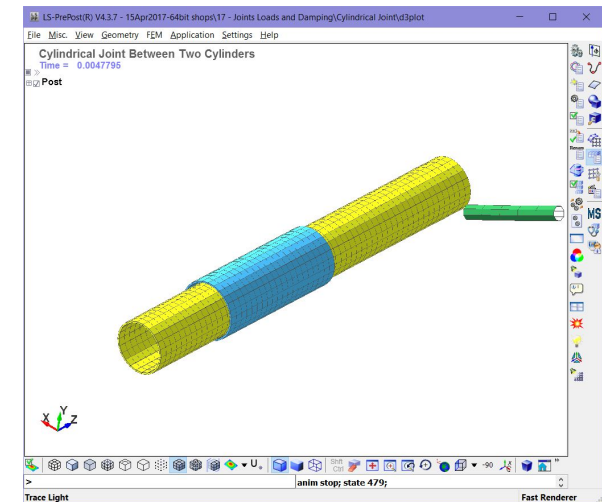
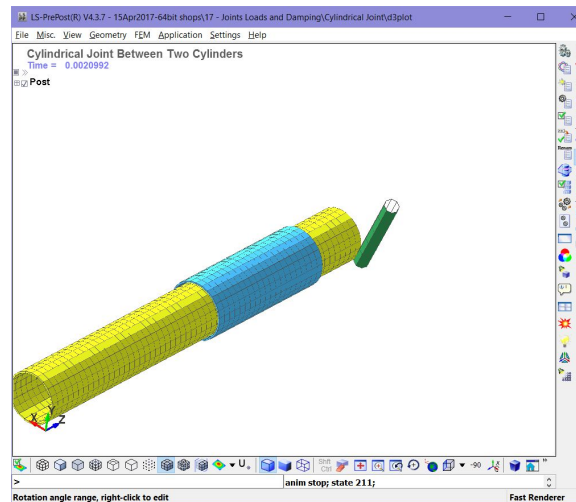
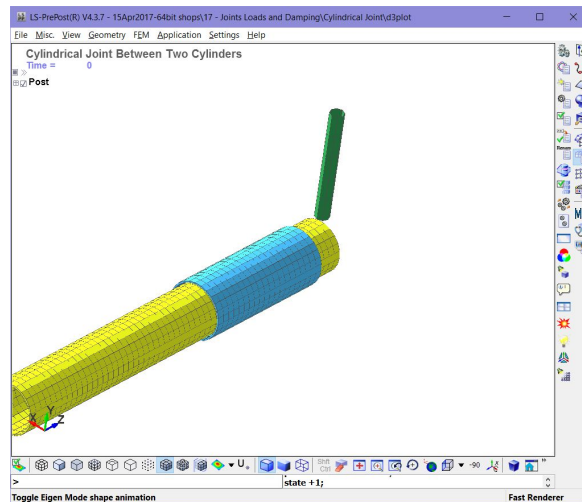
**Background:** The spherical joint is the easiest to setup of all the joints. All you need is two rigid bodies (Constrained Nodal Rigid Body (CNRB) having one node from each CNRB coincident with each other. As with all joints, try to ensure that the CNRB is well distributed (it has some spatial reach and mass).



### Tasks

5. Open up Cylindrical Joint Between Two Cylinders - Start.dyn and inspect the deck.
6. Read the Keyword Manual for \*CONSTRAINED\_JOINT\_CYLINDRICAL and then setup your joint.
7. Run the model and see if it makes sense.
8. Add contact to the model and rerun. Notice how \*CONTACT is setup, change sfs=1 (default) and rerun.

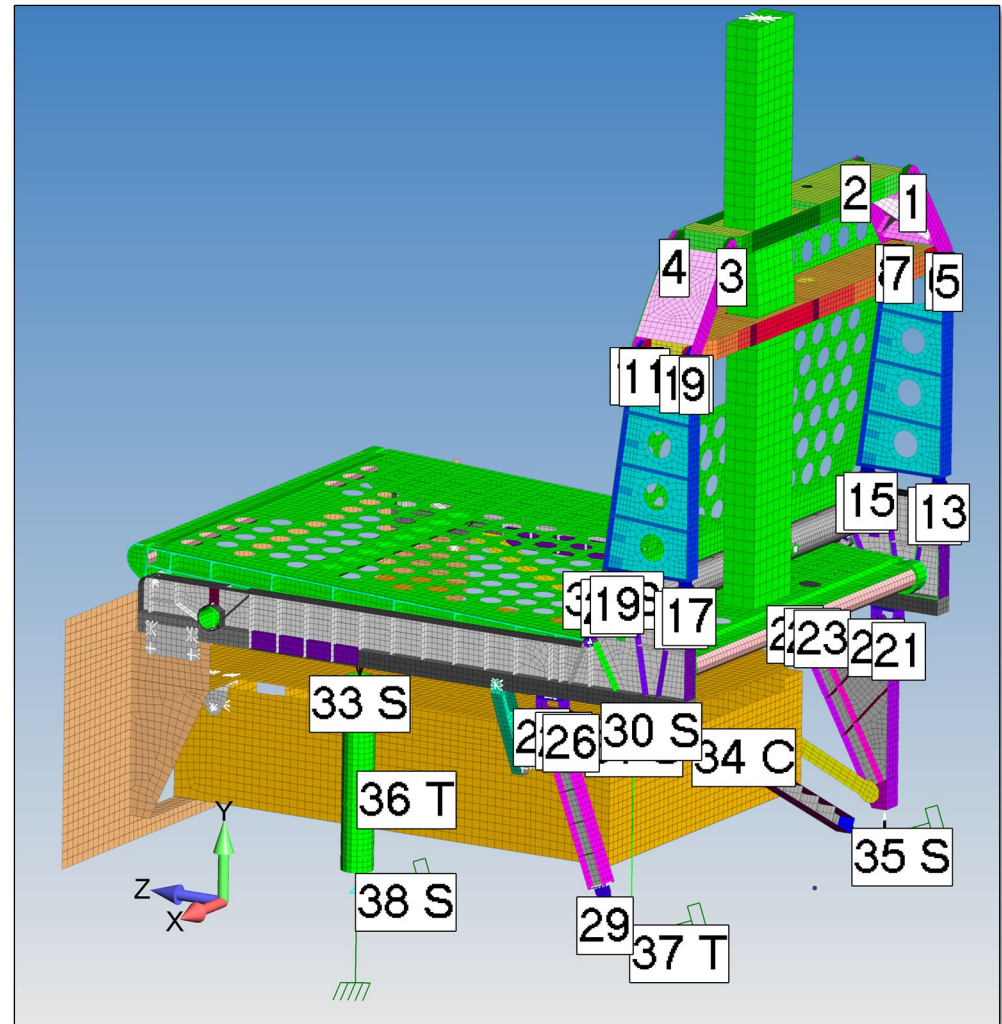
### Cylindrical Joint w/Whipping Beam Contact





### 10.4.1 WHO USES JOINTS?

**Background:** Joints are very prevalent in seat analysis and automotive simulation. One could spend days working on joint setup and our only comment is that they do work if one is careful with the setup.



### 19.5.1.1 Keywords Used in this Section for Solid Elements

The practice going forward is only to mention Keywords that are unique to the implicit analysis procedure and it is assumed that the reader is familiar with the standard Keywords to run an explicit analysis. For example, the Keyword \*CONTROL\_TERMINATION is not discussed although it is required to run the analysis.

Keyword	Card Variable	Description
*CONTROL_ACCURACY	IACC=1	The IACC=1 is new and is something specific for implicit to account for the solution potentially taking large steps through the solution. This card setup is a recommended standard for all implicit analysis work. While OSU=1 is standard for rotating equipment, it is not necessary for standard implicit work unless one has large rotations.
*CONTROL_IMPLICIT_GENERAL	IMFLAG=1 & DTO=1.0	This is where we start by telling LS-DYNA that an implicit analysis is being requested.
*CONTROL_IMPLICIT_SOLUTION	NSOLV=1	The <b>linear elastic hex element</b> requires this trigger.
*CONTROL_OUTPUT	SOLSIG=1	For stress extrapolation of fully-integrated solid elements.
*DATABASE_EXTENT_BINARY	NINTSLD=8	Writes out all integration points for fully-integrated solid elements.
*SECTION_SOLID	ELFORM=18 (Hexs) ELFORM=16 (Tets)	18: 8 point enhanced strain solid element for <b>linear</b> statics. 16: 10-node tetrahedral.

### 19.5.1.2 Workshop: 23 – Linear Elastic Analysis – Solids - Hex & Tets

**Objective:** Verify that Bricks and Tets can provide high-quality linear stress results.

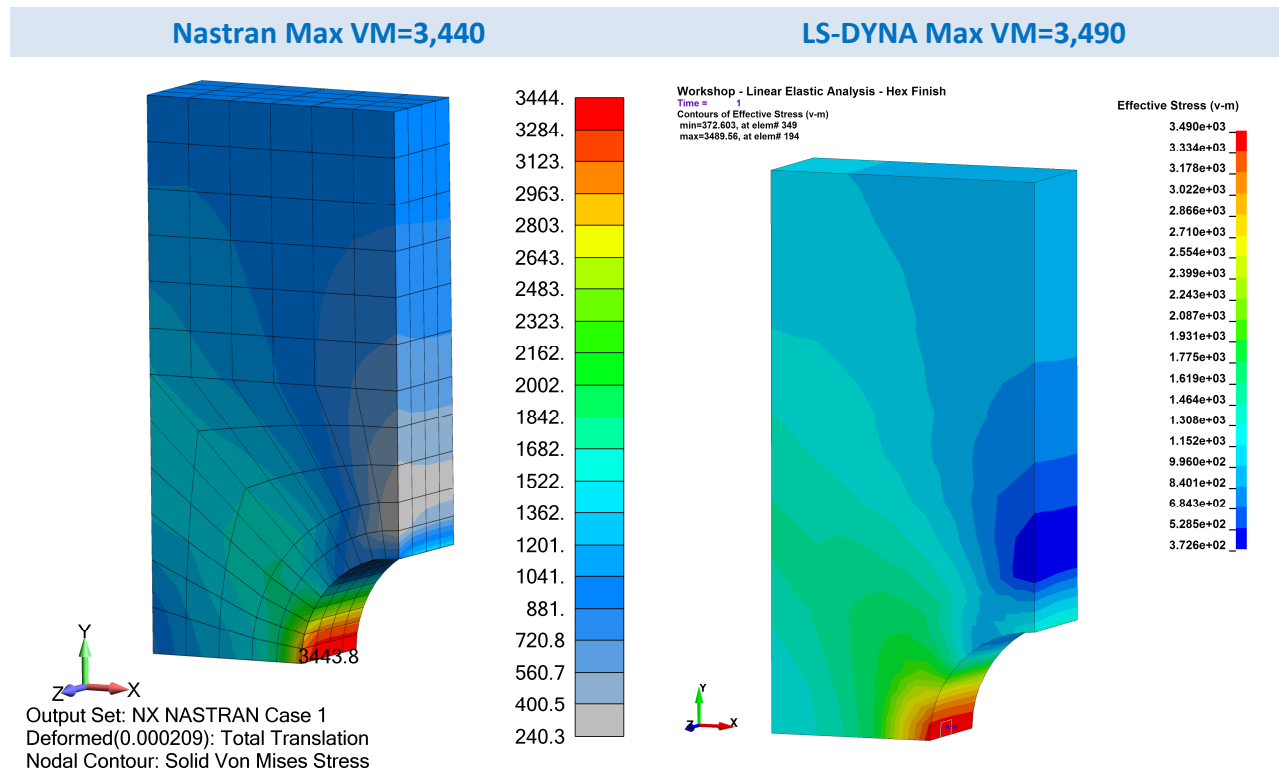
**Problem Description:** A quarter-symmetry plate of unit thickness is given a uniform pressure load of 1,000 psi. Nastran results are presented for the baseline comparison. (Note: Stress interpolation is done by by \*CONTROL\_OUTPUT, solsig=1 so make sure that extrapolate is turned off. ). A movie file is not provided since this Workshop follows prior practice of filling out the – Start keyword deck and if the student runs into trouble, there is a – Finish deck available.

#### Tasks:

##### Hexahedral Analysis

- Load Keyword file: / Linear Elastic Analysis – Solids / Hex / Linear Elastic Analysis - Hex - Start.dyn into a text editor and start filling in Keywords.
- Run and post-process.
- Change elform=-1 and note any differences.

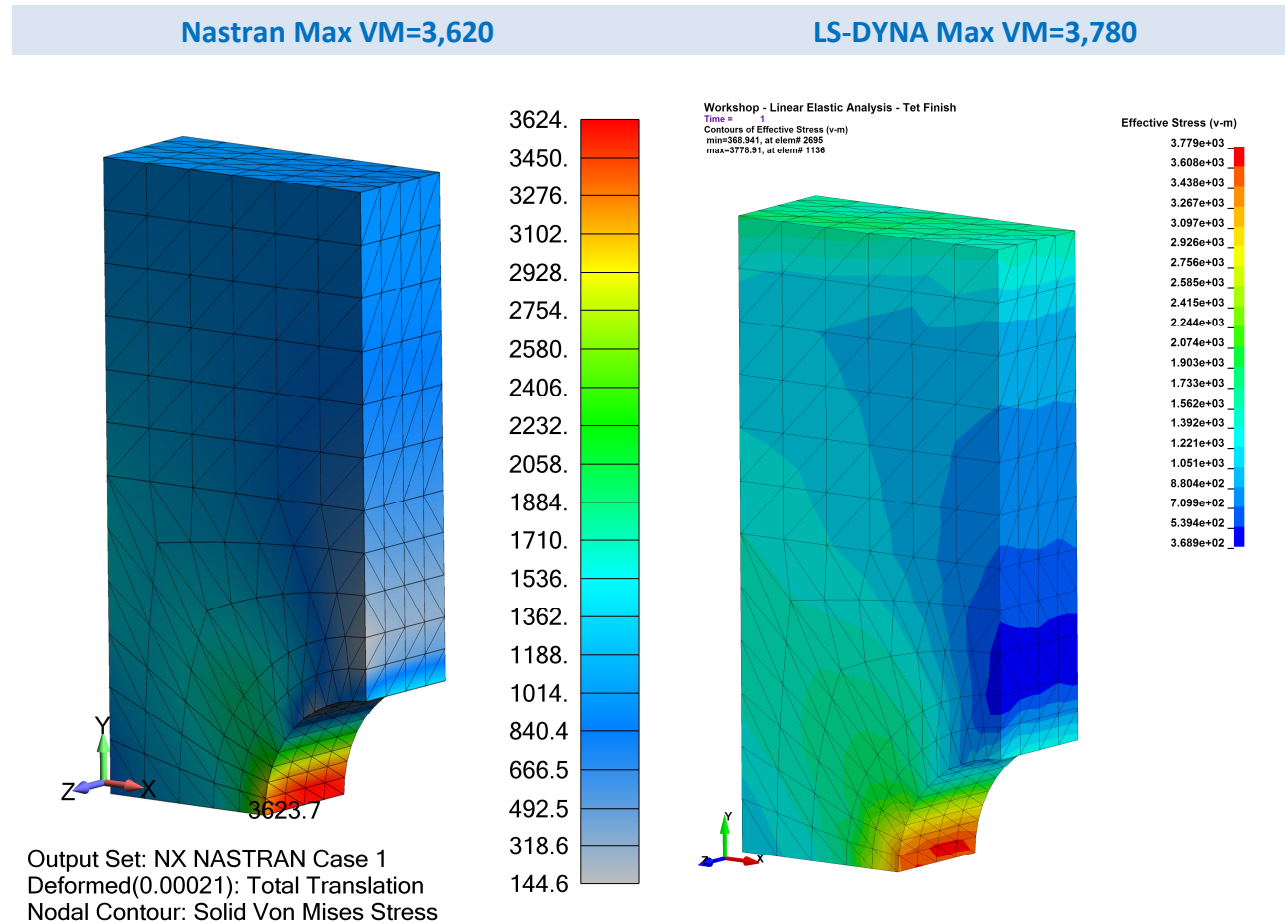
Elform	Max Stress
18	
-1	



## Tasks:

### Tetrahedral Analysis

- Not much to discuss. The model is only presented for completeness.



### 19.5.1.2.1 SOLID ELEMENTS: STRESS POST-PROCESSING IN LSPP

#### How Does LSPP Display Stresses in LSPP for Solid Elements?

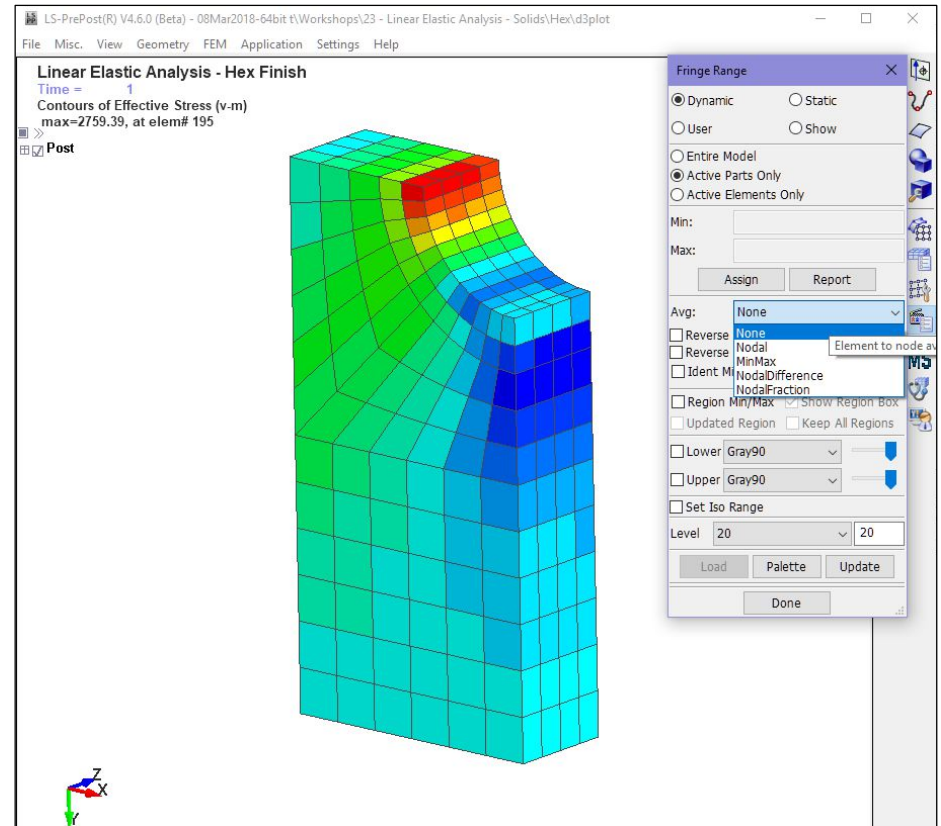
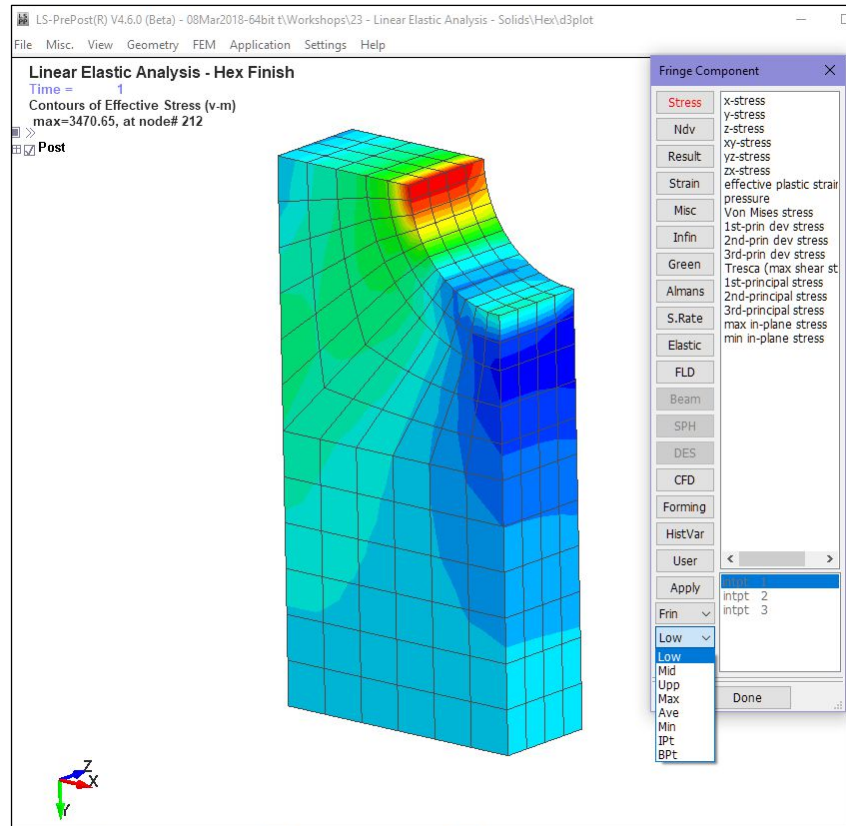
##### Inside the Element

In Fringe Component, the Low, Mid and Upp all do the same thing for solid elements, i.e., nothing. Each node has a unique stress as calculated by Solsig or un-extrapolated (assuming fully-integrated element formulation).

Then we have Max, Ave and Min which take the Max, Ave or Min of the eight integration points and uses that as the “single-value” stress item for that element.

##### Outside the Element

Under Fringe Range is where one determines how to average (Avg:) the stresses between connected elements. The options allow no averaging (none) and then Nodal and MinMax (averaging the Min and then the Max value at the connected nodes). The last two options can be used to assess the mesh convergence (NodalDifference and NodalFraction).



## 19.6 BEAM ELEMENT TECHNOLOGY FOR LINEAR ELASTIC STRESS ANALYSIS

LS-DYNA beam element formulations are quite diverse and provide broad modeling flexibility. When the term “resultant” is used for the ELFORM description, it signifies that only displacements and forces are calculated for the element. For explicit work, ELFORM=1 is the default while for implicit work ELFORM=4 is recommended. Although ELFORM=13 matches exactly with standard Nastran beam element formulation, it is a resultant formulation and stresses are not calculated and it is only applicable to linear behavior.

For all the standard LS-DYNA beam element formulations where stress is calculated, one needs to think about the type of quadrature rule (QR) is desired to recover stresses. It is identical to that for shell and solid elements but with a twist. For example, LS-DYNA calculates stress at the mid-point of the beam element while Nastran codes perform the calculate at both ends of the beam. Also, the stress recovery is different (see below). Nevertheless, the same identical results can be obtained for linear elastic analysis as that generated by hand-calculations or standard implicit codes. And, if nonlinear, elastic-plastic, large strain behavior is required, LS-DYNA provides quite a bit of flexibility to obtain high-accuracy results.

### 19.6.1 BEAM INTEGRATION (QR) SETTING: RECTANGULAR

Beam integration is defined by the QR setting and can be one point or 2x2 Guass, 3x3 Guass, 3x3 Lobatto or 4x4 Guass as shown in the figure. Additionally, if necessary, one can also define specialized integration rules via \*INTEGRATION\_BEAM.

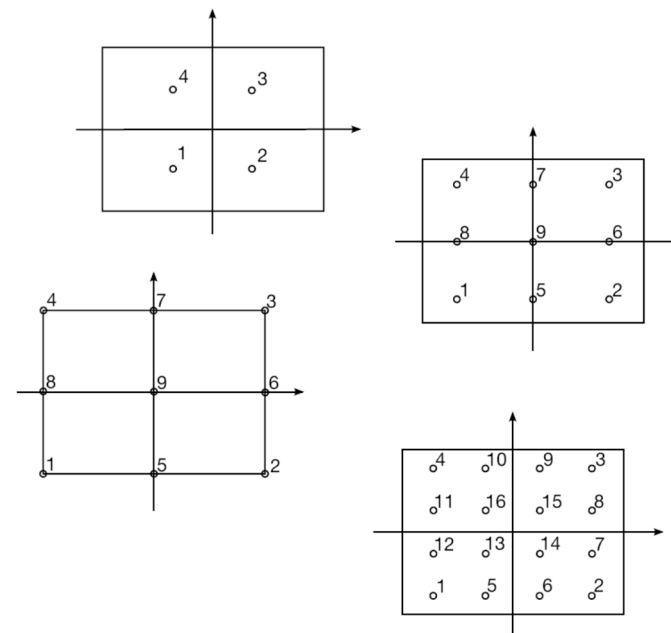
For beam elements, the integration is performed at one-point along the axis (mid-point of the beam) and then multiple points in-plane per QR=2, 3, 4 or 5.

The integration rule can be considered similar to that for shell elements based on the NIP setting to capture through thickness plasticity where for accuracy reasons NIP=5 is preferred.

For linear analysis, QR=4 is recommended and for plasticity analysis, QR=5 is a better choice.

LS-DYNA Theory Manual

Hughes-Liu Beam



**Figure 6.3.** Integration possibilities for rectangular cross sections in the Hughes-Liu beam element.



### 19.6.2 BEAM INTEGRATION (CYLINDRICAL SOLID AND TUBE)

For circular solid (cylinders) or tubes, the default beam integration formulation is based on a circumference of integration points at a radius defined by:

$$Integration_{Radius} = \sqrt{(R_{Inner}^2 + R_{Outer}^2)/2}$$

Unfortunately, there is not a Lobatto type integration scheme for circular cross-sections. Please note that the standard integration schemes only allow 4 (QR=2) or more points around one fixed circumference (e.g., the figure on the right shows nine integration points (QR=3 or 4).

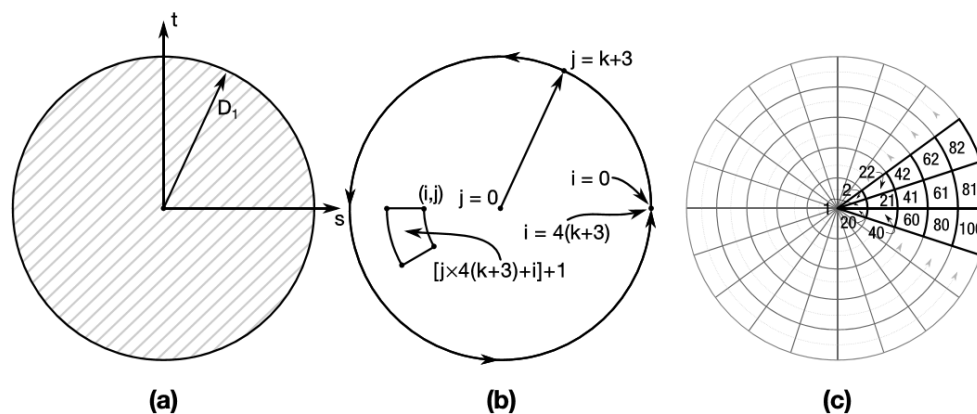
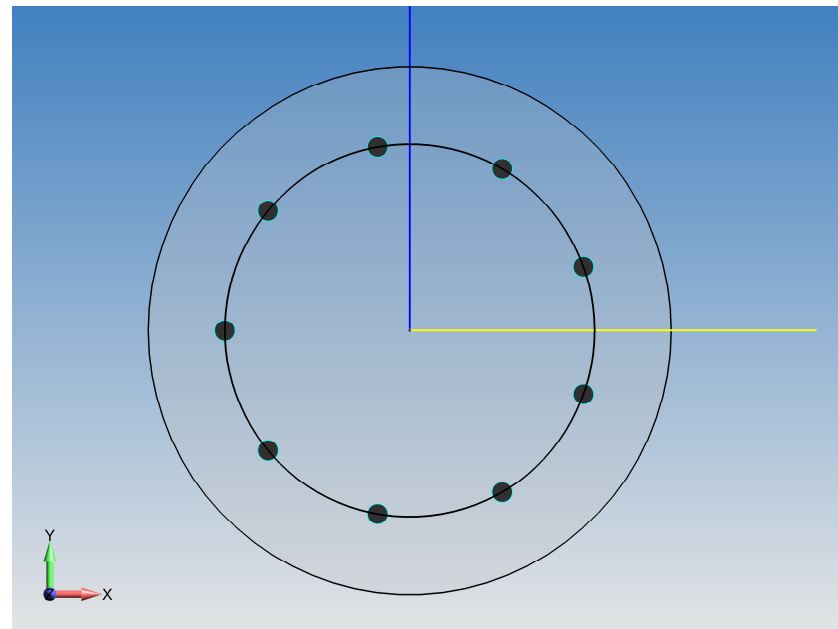
This scheme has advantages and disadvantages. For explicit analysis with large strain plasticity, it works fairly well but for linear stress analysis, where stresses on the surface of the beam are of more importance, it presents some limitations.

#### \*INTEGRATION\_BEAM

Although it may appear computationally costly to request 100 beam integration points (K=2), most FE models don't have hundreds of thousands of beam elements. The only input requirements for a solid cylinder is to specify  $D_1$  and then K.

To output this data into the binary file, the \*DATABASE\_EXTENT\_BINARY, beamip=100 must be set. An example is presented on the next page.

Overall, the simulation engineer should know that the results are exact to theory with respect to the stress results being extracted at the mid-point of the beam (midway between the nodes) and at the specified integration point.



**Figure 24-10.** Type 8: Circular. (a) Cross section geometry. (b) Integration point numbering. (c) Example for  $k = 2$ .

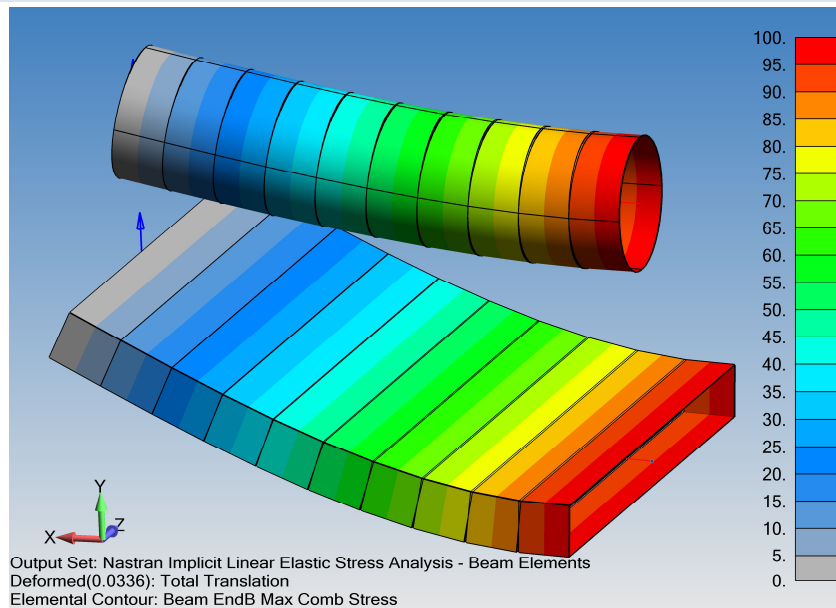
### 19.6.3 WORKSHOP: 24 - LINEAR ELASTIC ANALYSIS – BEAM ANALYSIS

**Objective:** Setup a beam model to generate linear elastic stress analysis results that match a Nastran run.

**Problem Description:** Cantilevered beam with a load of 16.67 units at its end. The student needs to setup the file to run and to be able to contour the beam results in LSPP.

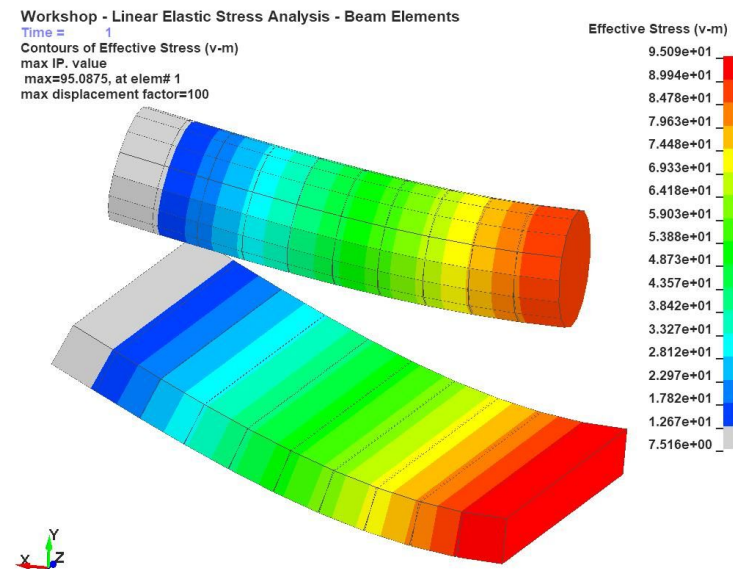
**Tasks:** Using the – Start file, the student will need to set the standard linear analysis commands (e.g., NSOLVR=? ) and then some new options. The first is to setup the circular section to use \*INTEGRATION\_BEAM. Review the manual and then it should make sense to set icst=8, k=2 and d1=1.28505. The irid=10 and thus under \*SECTION\_BEAM, qr/irid=-10. Then, we will need to request beam stress information to be dumped into the D3Plot files. This is done within the \*DATABASE\_EXTENT\_BINARY, BEAMIP={student input – see manual for k=2}. For post-processing, if all is setup correctly one can contour beam results under FriComp / Beam / Von Mises stress. To get the displacement scaled up, see Settings (up on the main bar) / Post Settings / Displacement Scale Factor. The image shown below is using a Displacement Scale Factor = 100.

#### Nastran Results



*Analysis Notes: Nastran beam stress results are provided at the end of the beams. Each beam has a length of 1 unit. Total length of the cantilevered beam is 10.*

#### LS-DYNA Results



*Analysis Notes: LS-DYNA calculates beam stress results at the middle of the beam. To compare with Nastran, multiply by (10/9.5)  
Importantly, the stress results for the rectangle beam analysis are identical once adjusted for stress recovery location. The rod still lacks – why?*

Extra Credit: Increase the \*INTEGRATION\_BEAM variable k from 2 to 3 (remember to change the BEAMIP to 144) and reanalyze.

## 19.7 CHECKLIST FOR IMPLICIT LINEAR ELASTIC ANALYSIS IN LS-DYNA

- ☑ \*CONTROL\_ACCURACY with IACC=1
- ☑ \*CONTROL\_IMPLICIT\_GENERAL with IMFLAG=1 & DTO=1.0
- ☑ Check element formulation: (i) Shells ELFORM=21; (ii) Bricks ELFORM=18; (iii) 10-node Tets ELFORM=16 and Beams ELFORM=4 with QR=3 (Lobatto). If your beams are circular, then one might want to use \*INTEGRATION\_BEAM.
- ☑ \*CONTROL\_SHELL with INTGRD=1 for Lobatto integration (only for linear analysis using shells)
- ☑ To ensure linear elastic solution (no geometric nonlinearity), set NSOLVR=1 (\*CONTROL\_IMPLICIT\_SOLUTION)
- ☑ Use \*MAT\_1 since it is a linear elastic analysis
- ☑ For consistency set load curve to end at Time=1.0 and Termination=1.0
- ☑ Be aware of your integration scheme (Gaussian or Lobatto) and your setting on \*DATABASE\_EXTENT\_BINARY to recover all required integration point data for shells (i.e., MAXINT=-3), solids (i.e., NINTSLD=8) and beams (i.e., BEAMIP=9). For pure linear, think Lobatto to the surface and extrapolate to the nodes. However, if material plasticity occurs, than Gaussian integration is a better choice.
- ☑ Think about extrapolation, whether for shells (LSPP, extrapolate 1) or for solids via SOLSIG=2 (*and remember to turn off Extrapolation when contour stresses over solid elements*)
- ☑ Within LSPP, be careful with how you pick stresses. In general, your selection choices will depend upon your element type.

## 19.8 GEOMETRIC AND MATERIAL NONLINEARITY

In this section we show how LS-DYNA implicit handles geometry and material nonlinearity. For the explicit solver, the solution of high nonlinearities is typically not an issue unless we are talking about massive element distortions or compaction of foams. We will treat this section in the classic sense and cover geometric nonlinearity as buckling and material nonlinearity as plasticity.

A nice engineering review of nonlinearity and how LS-DYNA can solve these problems using the implicit technique is found in the Class Reference Notes / Implicit Analysis / DYNAmore Implicit Users Guide /LS-DYNA Implicit Users Guide.pdf. If you have not read this article, one should review it while the Workshop exercise on buckling is solving (since it'll take a few minutes).

To summarize what it means to do nonlinear analysis, the DYNAmore article provides a succinct list:

- non-linear material models (plasticity);
- contact;
- large deformations;
- non-linear constraints (such as joints);
- non-linear loading (such as follower forces, where the force direction is defined relative to the deformed geometry);
- stress stiffening (guitar string effect).

### 19.8.1 NEW KEYWORD COMMANDS USED IN THIS SECTION FOR NONLINEAR IMPLICIT ANALYSIS

Keywords for Nonlinear Implicit Analysis		
Keyword	Card Variable	Description
*CONTROL_ACCURACY	IACC=1	The IACC=1 is new and is something specific for implicit to account for the solution potentially taking large steps through the solution. This card setup is a recommended standard for all implicit analysis work. While OSU=1 is standard for rotating equipment, it is not necessary for standard implicit work unless one has large rotations.
*CONTROL_IMPLICIT_AUTO	IAUTO=1 & DTMAX=?	When in doubt, leave everything in default. However, if you know how the solution might behave, one should control it. By setting DTMAX to a specific value we force the implicit solution to only advance at steps no greater than this value. However, when in doubt just leave it alone at the default. For a buckling analysis this encourages the solution to not overshoot and then have to fight its way back down. If one looks at the DYNAmore example, they control the maximum timestep with a curve.
*CONTROL_IMPLICIT_DYNAMICS	IMASS=1, GAMMA=0.6 & BETA=0.38	Adds dynamics with numerical damping to stabilize the implicit solution. If this technique is used, the kinetic energy (KE) should be checked at the end of the solution to verify that it is very low as compared to the internal energy. Damping only removes energy from the solution when there is motion or kinetic energy. If the KE is near zero than the dissipative energy (fictional energy removed by damping) will also be zero. <i>Note: make sure your material cards have mass! Also please note that for a true dynamics analysis, reset Gamma and Beta to their defaults. These settings (0.6 / 0.38) are to simulate a damped response and their only purpose is to provide convergence stability to a nonlinear solution.</i>
*CONTROL_IMPLICIT_SOLUTION	NLPRINT=2	NLPRINT is for extra information



## 19.8.2 WORKSHOP: 25 – NONLINEAR BUCKLING OF BEER CAN

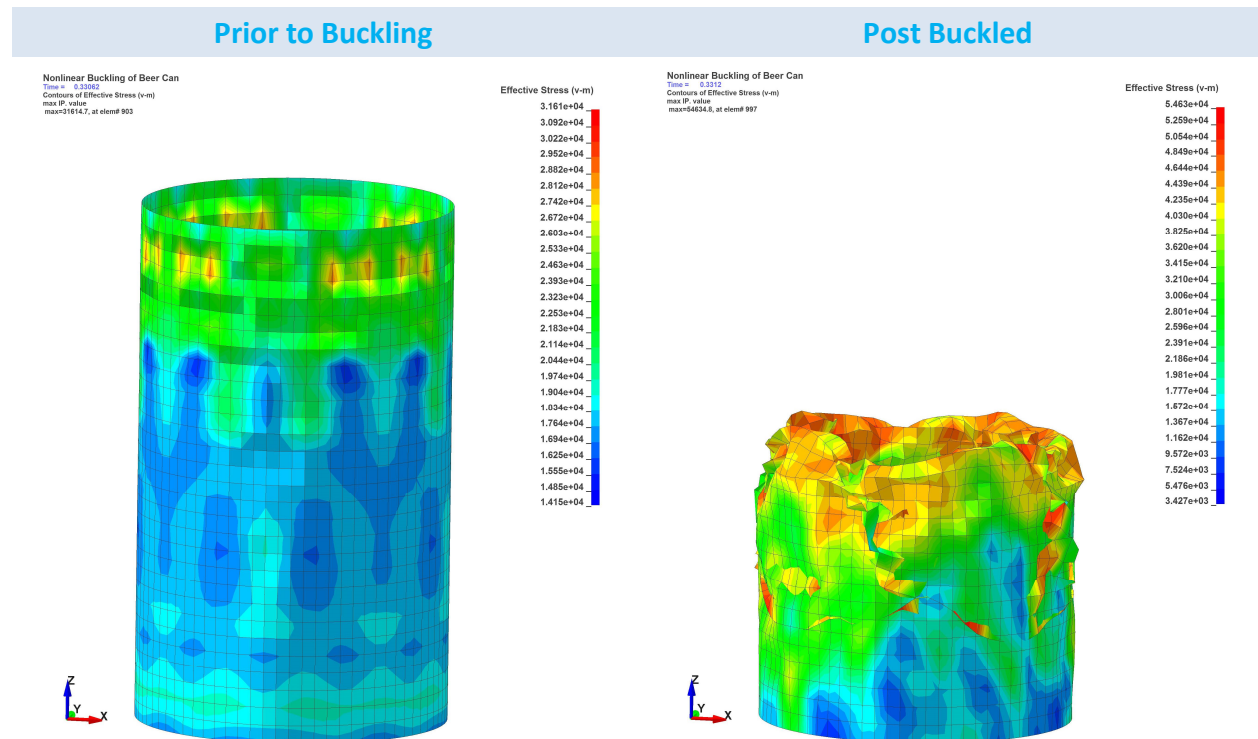
**Introduction:** The classic example is a slender column buckling and one can solve this problem with a simple hand calculation using Euler's formula or solve it with a standard FEA technique. A review of basic FEA buckling using FEMAP and NX Nastran can be found in the Reading Assignments folder as: Predictive Engineering Buckling White Paper Rev-1. To keep things interesting, we'll do a beer can and make the material out of aluminum using \*MAT\_098.

**Objective:** Setup model to solve one of the trickiest nonlinear geometric buckling problems in mechanics

**Problem Description:** The buckling of thin-walled cylinder is a very complex problem. We are going to make it even more difficult by using a force to drive the buckling and not a slowly increasing displacement load (although a companion model is given with a moving translational load). The cylinder is constrained on both ends and the force load is applied at the top over a load interval where buckling occurs prior to full load application.

### Tasks:

- Open in a text editor Nonlinear Buckling of Beer Can – Start.dyn and inspect the Keywords.
- Enter data in the Keywords discussed within this section and then run the model.
- Review results and create XY cross-plot in LSPP showing Force (Y-axis) and Displacement (X-axis). The accompanying Movie file will show how to do a cross plot and also add a Timeline plot to the graphic.



*Extra Credit: If you have time, rerun the model as a buckling analysis within LS-DYNA. This can be done by adding the \*CONTROL\_BUCKLE card and then changing the endtime to a time prior to buckling (we just want to load the can and not crush it). The buckling value (Freq = ) is the load scale factor that would cause buckling.*

### 19.8.3 CHECKLIST FOR IMPLICIT NONLINEAR ANALYSIS IN LS-DYNA

- ☑ \*CONTROL\_IMPLICIT\_ACCURACY with IACC=1
- ☑ \*CONTROL\_IMPLICIT\_AUTO with IAUTO=1 and DTMAX=0.1 or use a “curve” to define solution keypoints
- ☑ \*CONTROL\_IMPLICIT\_DYNAMICS with IMASS=1, GAMMA=0.6 and BETA=0.38 for damped dynamics to assist in solution stability, e.g., when buckling occurs. *Note: If a true transient dynamic analysis is required, then the values of GAMMA and BETA should be set to their defaults.*
- ☑ \*CONTROL\_IMPLICIT\_GENERAL with IMFLAG=1 & DTO at 0.1 or some thoughtful initial timestep value
- ☑ Check element formulation: (i) Shells ELFORM= -16 with NIP=5; (ii) Bricks ELFORM=-1; (iii) 10-node Tets ELFORM=16 and Beams ELFORM=4 with QR=4
- ☑ \*CONTROL\_IMPLICIT\_SOLUTION (all defaults) and NLPRINT=3 to print out convergence information to the screen
- ☑ \*CONTROL\_OUTPUT with SOLSIG=2 for solid element extrapolation of integration point stresses to the nodes
- ☑ If the analysis is not dynamics based, set load curve to end at Time=1.0 and Termination=1.0 for consistency
- ☑ \*DATABASE\_EXTENT\_BINARY to recover integration point data for shells, solids and beams via MAXINT=-3, BEAMIP=16 and NINTSLD=8.
- ☑ Stress averaging for shells or solids can be done as per the notes within the linear section.

## 19.9 CONTACT

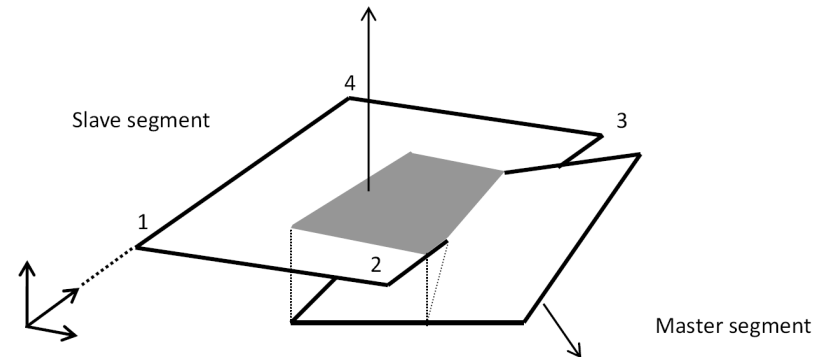
### 19.9.1 GENERAL COMMENT AND FOCUS ON MORTAR CONTACT

LS-DYNA was developed specifically to solve contact problems (see Class Reference Notes / History of LS-DYNA). We are not going to talk at length about contact theory since sources are available via the LSTC Theory Manual and many fine application notes (see Reading Assignments / Mortar Contact for Implicit Analysis). Contact can be effortlessly implemented or it can be bewitching in complexity. A reasonable treatment of contact is a multi-day course in itself. For this reduced treatment for LS-DYNA implicit, we will focus only on Mortar contact and just a bit on `_TIED` applications.

#### Efficient Contact Modeling

Whenever possible, interferences between parts should be avoided. It is standard contact practice that any initial interference is removed (nodes are shifted) and as such, sharp stress spikes can occur where parts/plates overlap.

Setting up contact surfaces appropriately that account for plate thickness can be time consuming. If necessary, contact thickness can be overridden within the `*CONTACT` Keyword card or tracked via `IGNORE=1`. This is especially important with implicit since the solution stability depends upon clean contact.



“Mortar contact is a penalty based segment-to-segment contact with finite element consistent coupling between the non-matching discretization of the two sliding surfaces”

Thomas Borrvall, DYNAmore Nordic AB

*In other words, no `Soft=2` setting is needed and pretty-much all other varied and sundry contact settings can be left un-touched.*

## 19.9.2 GENERAL MORTAR CONTACT TYPES

**For general contact, the list is short and simple:**

4. \*CONTACT\_AUTOMATIC\_SINGLE\_SURFACE\_MORTAR
5. \*CONTACT\_AUTOMATIC\_SURFACE\_TO\_SURFACE\_MORTAR

How Mortar contact treats penetrations is a subject of some discussion since it is one known mechanism that can destroy convergence. From the Keyword Manual – Implicit Section (and now would be a good time to do some reading):

### *Initial Penetrations*

As mentioned above, initial penetrations are always reported in the message file(s), including the maximum penetration and how initial penetrations are to be handled. The IGNORE flag governs the latter and the options are

IGNORE < 0	Same functionality as the corresponding absolute value, but contact between segments belonging to the same part is ignored completely
IGNORE = 0	Initial penetrations will give rise to initial contact stresses, i.e., the slave contact surface is not modified
IGNORE = 1	Initial penetrations will be tracked, i.e., the slave contact surface is translated to the level of the initial penetrations and subsequently follow the master contact surface on separation until the unmodified level is reached
IGNORE = 2	Initial penetrations will be ignored, i.e., the slave contact surface is translated to the level of the initial penetrations, optionally with an initial contact stress governed by MPAR1
IGNORE = 3	Initial penetrations will be removed over time, i.e., the slave contact surface is translated to the level of the initial penetrations and pushed back to its unmodified level over a time determined by MPAR1
IGNORE = 4	Same as IGNORE = 3 but it allows for large penetrations by also setting MPAR2 to at least the maximum initial penetration

Most of these options are self-explanatory but IGNORE=4 provides the ability to handle initial interpenetrations. A dedicated workshop will cover this behavior since it is quite useful for interference fits.

### 19.9.2.1 Workshop: 26A – Implicit Contact – Basic Mechanics with Bolt Preload

**Objective:** Get familiar with contact and being able to check your model by investigating the contact forces generated between the two surfaces pulled together by a bolt preload and then by the “lever” load (Y-axis).

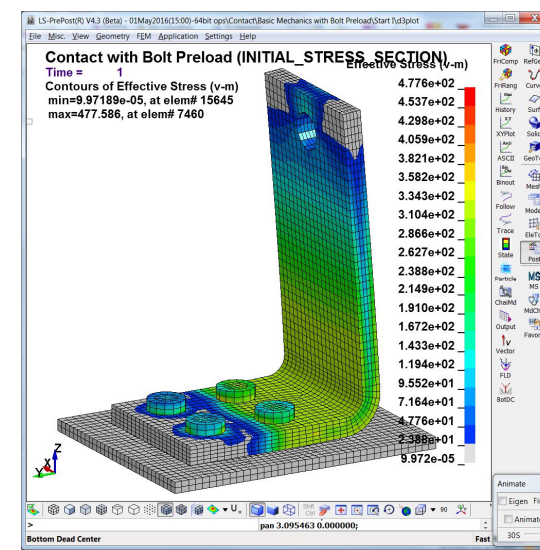
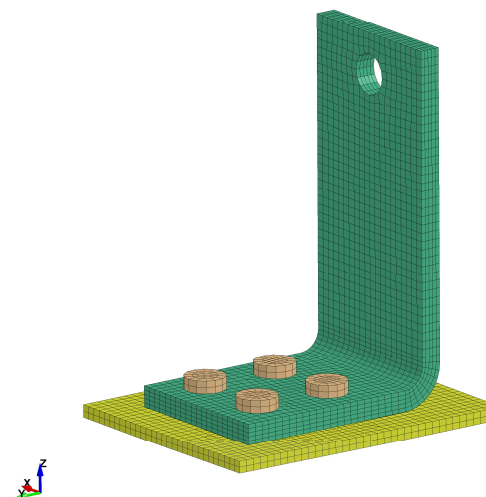
**Model Description:** A simple plate structure is bolted together with the bolts given a hefty bolt preload. Once this is done, a horizontal (Y-axis) load is applied that tilts the vertical plate over.

**Acknowledgements:** This example is courtesy from DYNAmore with some modifications.

#### Workshop Tasks

- Load model Basic Mechanics with Bolt Preload – Start and setup mortar contact between the three parts using SURFACE\_TO\_SURFACE between parts 1 & 3 and then 1 & 2. Investigate how bolt preload is done via a section through part 3 (the bolts). Run note lack of convergence. Add IGNORE=1 to contact card.
- Check for contact interference within LSPP (Application / Model Checking / General Checking / Contact Check. Check interference, make mental note and proceed to analyze model. Note excessive iterations (100+) on first effort.
- Kill analysis and add IGNORE=1 or =2 via \*CONTROL\_CONTACT (Why do you think in this case they provide equivalent results?). Re-analyze.
- Plot the resultant contact force between the L-Bracket (part 1) and the bottom plate (part 2). Note that the preload increases the contact force linearly up to the application of the horizontal “lever” load. (\*DATABASE\_RCFORC)
- Create a Part Set with all three parts and then switch the contact to SINGLE\_SURFACE and create a \*CONTACT\_FORCE\_TRANSDUCER for the base part. Re-analyze and plot the force transducer result.

Contact with Bolt Preload (INITIAL\_STRESS\_SECTION)





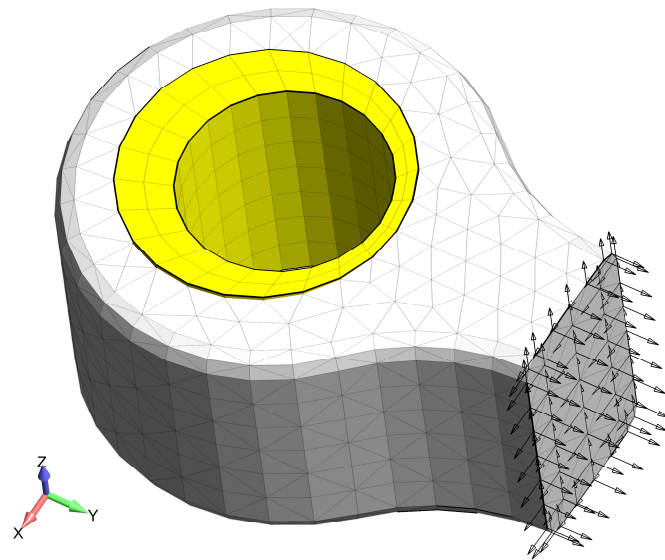
### 19.9.2.2 Workshop: 26B - Contact - Shrink Fit Analysis

**Objective:** We explore how IGNORE=4 works for Mortar contact to handle interference fits

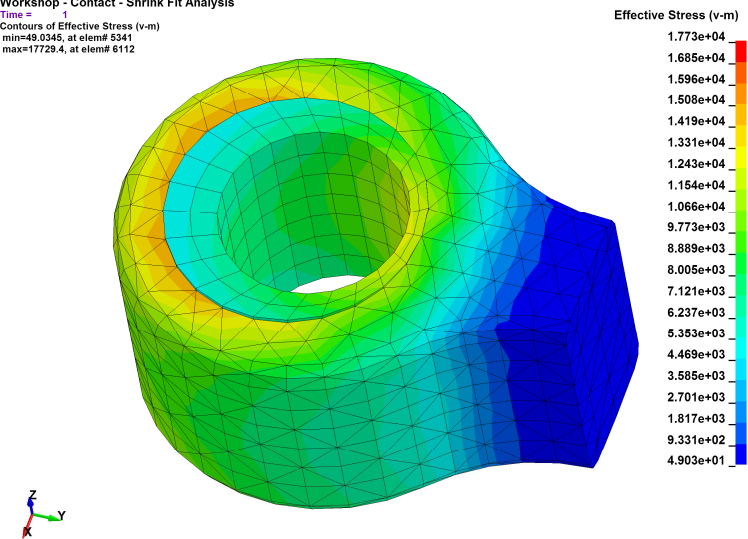
**Problem Description:** A sub-section of a larger model is used to illustrate how contact can be used to develop an interference fit. No load will be used and the analysis will be run with only the geometric interference fit of 0.0035" is used to drive the analysis.

**Tasks:** Setup Mortar Contact with the correct parameters to enforce the interference action

- Review settings and set the appropriate value for \*CONTROL\_OUTPUT (SOLSIG), \*CONTROL\_IMPLICIT\_SOLUTION (NLPRINT), \*DATABASE\_EXTENT\_BINARY (NINTSLD) and more complex within the \*CONTACT card the IGNORE, MPAR1 & MPAR2 values. Don't forget friction or your bushing might slide away.
- Run the model and see if you get lucky.



Workshop - Contact - Shrink Fit Analysis  
 Time = 1  
 Contours of Effective Stress (v-m)  
 min=49.0345, at elem# 5341  
 max=17729.4, at elem# 6112



### 19.9.3 TIED CONTACT FOR MESH TRANSITIONS, WELDING AND GLUING

Given the idealization difficulty of system modeling, the ability to tie together different mesh densities (e.g., hex-to-hex or tet-to-hex), snap together parts along a weld-line or just glue sections together (e.g., plate edge to a solid mesh) is an amazingly useful ability and implicit LS-DYNA provides a very complete Tied Contact tool box to work with.

The emphasis of this course to provide an overview of the basics to get started efficiently with LS-DYNA, a short list of recommended \*KEYWORDS for implicit Tied Contact are presented.

It is also suggested that this would be good time to review the implicit section of the Keyword Manual under “tied” (Appendix P). Keep in mind that it is better to use fewer options but understand more completely what each command does. For example, the \_SURFACE option tells LS-DYNA to only look at the surface of the MST that is provided within the Keyword Card. For example, if a Part is specified, it only looks at the surface of this Part as eligible for Tied Contact and nothing in its interior.

#### When a Solid Mesh is Used (Translational DOF Tied)

- \*CONTACT\_TIED\_NODES\_TO\_SURFACE (planer connection)
- \*CONTACT\_TIED\_NODES\_TO\_SURFACE\_CONSTRAINED\_OFFSET (When the mesh is offset)

#### When a Shell Mesh is Used or attaching beams to a shell mesh (All Six DOF Tied)

- \*CONTACT\_TIED\_SHELL\_EDGE\_TO\_SURFACE
- \*CONTACT\_TIED\_SHELL\_EDGE\_TO\_SURFACE\_CONSTRAINED\_OFFSET
- \*CONTACT\_TIED\_SHELL\_EDGE\_TO\_SURFACE\_BEAM\_OFFSET

#### General Comments

- \_TIED (no \_OFFSET) implies that the slave surface is co-planer or if it isn't, it'll move it to be adjacent to the Master surface. And, BTW, it'll only move it if is “close” to the Master surface (Read The Manual (RTM)). One can override the default search distance by specifying a negative search distance on the SST MST fields. Please note that one has to specify both SST and MST with a negative number for the search distance. When in doubt – RTM.
- \_OFFSET will not move your slave surface but you may need to expand your search distance to get things to tie together. This is where setting negative values on SST and MST come in handy. Be aware that on the \_NODES side that if the distance is too large you can pick up interior nodes if you specify the SST using a PART.

#### **New School: Two (2) Tied Contact to do it all**

*\*CONTACT\_TIED\_NODES\_TO\_SURFACE\_CONSTRAINED\_OFFSET with CARD E option IPBACK=1 (to tie to rigid bodies) – 3 DOF (solid) tie*

*\*CONTACT\_TIED\_SHELL\_EDGE\_TO\_SURFACE\_CONSTRAINED\_OFFSET with CARD E option IPBACK=1 (to tie to rigid bodies) – 6 DOF (shells) tie*

## 20.2.8 COMMENTS ON LS-DYNA OUTPUT MESSAGES AND THEIR SIGNIFICANCE

**\*|du|/|u|** - If this value stays pegged at 1.0 then there is a good chance that you have something moving in your structure. Even with \_IMPLICIT\_DYNAMICS turned on, if something is moving quickly in your model, this value will often stay at 1.00 or maybe drop to 0.9 or 0.8 but quickly bounce back to 1.00. The solution is to kill the analysis and start hunting for something that is un-restrained or not connected or has a missing contact.

**\*Ei/E0** – If this value stays high (e.g., 1e-4) then you have something that is deforming massively (energy = force\*displacement) or poorly formed elements or something that is not behaving in a physical manner (e.g., wrong choice of material law). When this value stays high, the debugging can be difficult.

Here's one suggested short list:

- Check element quality (explicit time step & Jacobian)
- Check material formulation and curves that are used to define these materials
- Check CNRB's (implicit only supports full 6-DOF formulation as of this writing)
- Check loads

```

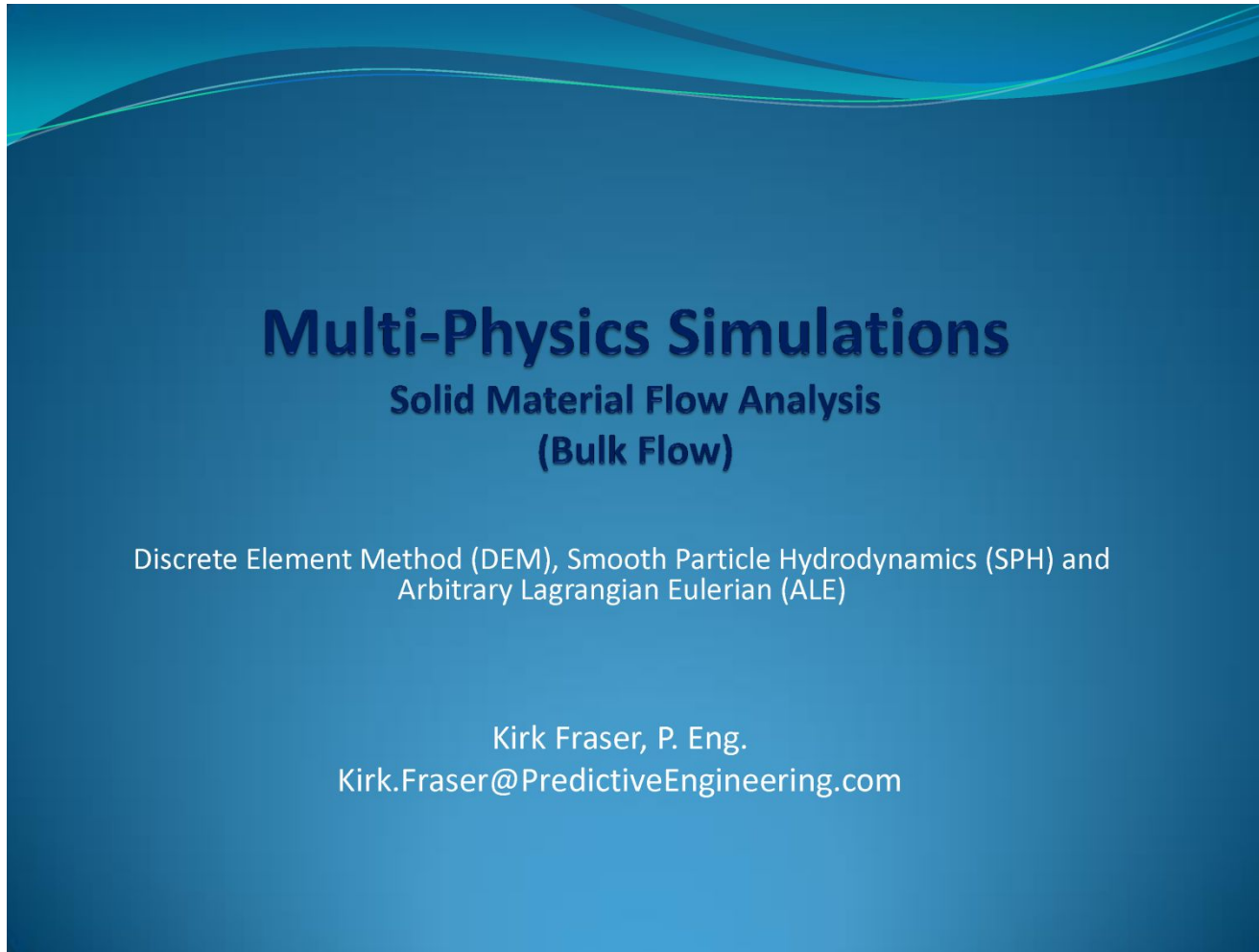
EGIN implicit dynamics step 13 t= 6.2447E-02 07/28/16 03:09:37
=====
time = 6.24468E-02
current step size = 2.49408E-03
Iteration: 1 *|du|/|u| = 3.4944549E-01 *Ei/E0 = 1.4412338E-06
Iteration: 2 *|du|/|u| = 6.2556077E-01 *Ei/E0 = 2.3195510E-06
Iteration: 3 *|du|/|u| = 1.6109298E-02 *Ei/E0 = 3.1429765E-08
Iteration: 4 *|du|/|u| = 4.2269288E-03 *Ei/E0 = 4.2768351E-09
Iteration: 5 *|du|/|u| = 1.0832160E-03 *Ei/E0 = 9.7970873E-10
Iteration: 6 *|du|/|u| = 1.4506872E-03 *Ei/E0 = 2.2795808E-09
Iteration: 7 *|du|/|u| = 1.4156571E-03 *Ei/E0 = 1.2233175E-09
Iteration: 8 *|du|/|u| = 3.5458310E-03 *Ei/E0 = 2.3347172E-09
Iteration: 9 *|du|/|u| = 3.4936389E-03 *Ei/E0 = 3.1520275E-09
Iteration: 10 *|du|/|u| = 1.0644974E-03 *Ei/E0 = 8.7925570E-10
Iteration: 11 *|du|/|u| = 1.7903880E-03 *Ei/E0 = 1.6156436E-09
Iteration: 12 *|du|/|u| = 7.5351489E-04 *Ei/E0 = 4.5485773E-10
Equilibrium established after 12 iterations 07/28/16 03:10:18

EGIN implicit dynamics step 14 t= 6.4941E-02 07/28/16 03:10:22
=====
time = 6.49408E-02
current step size = 2.49408E-03
Iteration: 1 *|du|/|u| = 3.5824293E-01 *Ei/E0 = 2.1956909E-06
Iteration: 2 *|du|/|u| = 1.0000000E+00 *Ei/E0 = 5.6207147E-04

ONE SEARCH stepsize = 0,
Automatically REFORMING stiffness matrix...

```

## 21. DISCRETE ELEMENT METHOD

A presentation slide with a blue gradient background and wavy lines at the top. The title 'Multi-Physics Simulations' is in large white font, followed by 'Solid Material Flow Analysis (Bulk Flow)' in smaller white font. Below that, the text 'Discrete Element Method (DEM), Smooth Particle Hydrodynamics (SPH) and Arbitrary Lagrangian Eulerian (ALE)' is in white. At the bottom, the presenter's name 'Kirk Fraser, P. Eng.' and email 'Kirk.Fraser@PredictiveEngineering.com' are listed in white.

**Multi-Physics Simulations**  
**Solid Material Flow Analysis  
(Bulk Flow)**

Discrete Element Method (DEM), Smooth Particle Hydrodynamics (SPH) and  
Arbitrary Lagrangian Eulerian (ALE)

Kirk Fraser, P. Eng.  
Kirk.Fraser@PredictiveEngineering.com

See Class Reference Notes / DEM / Predictive Engineering Discussion of LS-DYNA Meshfree Methods.pptx

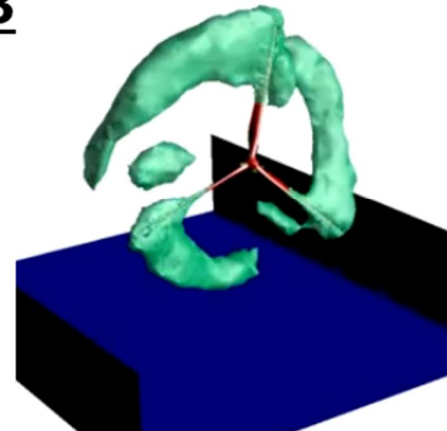
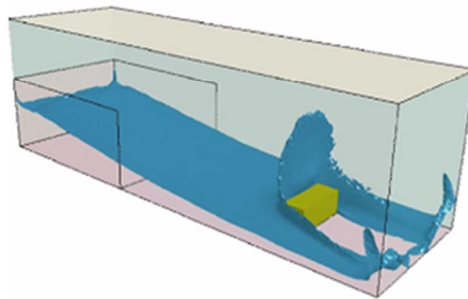
## 22. FLUID STRUCTURE INTERACTION AND MULTI-PHYSICS IN LS-DYNA

*{A bit dusty but still functional for the curious}*

# Fluid Structure Interaction with LS-DYNA Multiphysics



August 2013



See Class Reference Notes / Multi-Physics / LS-DYNA Multi-Physics.ppsx



## 23. LSPP SUMMARY LIST OF OPERATIONS AND CORRESPONDING WORKSHOPS

This sheet of LSPP Operations can be filled out by the student during the class.

LSPP Operation	Where the Operation is Discussed Within Movie File
Contouring Explicit Time Step, setting contour mode (none or node), stress legend - reverse	Workshop – LS-DYNA Mass Scaling Basics
Measuring nodal data from the glstat history files	Workshop – LS-DYNA Mass Scaling Basics
Changing Part Colors in LSPP	Workshop – Building the Better Beam
Displacement Scaling of D3PLOT data (results) {Settings / Post}	Instructor Led Workshop - Damping
Fast Fourier Transform of Transient Data	Instructor Led Workshop - Damping
Post-Processing Eigenvalue Results	Instructor Led Workshop - Damping
Section Cuts Through Model to Display Results	Workshop – Drop Test of Pressure Vessel
Visualization of Beams with View / Beam Prism	Workshop – Dynamic Relaxation
Plot Beam Element Forces	Workshop – Dynamic Relaxation
Post-Processing Stress via Fringe Range: User (defined stress range) and Stress Avg: {option}	Workshop – Dynamic Relaxation

# End

LS-DYNA Analysis for  
Structural Mechanics Class  
Notes

Partial Class Notes for  
Potential Attendees

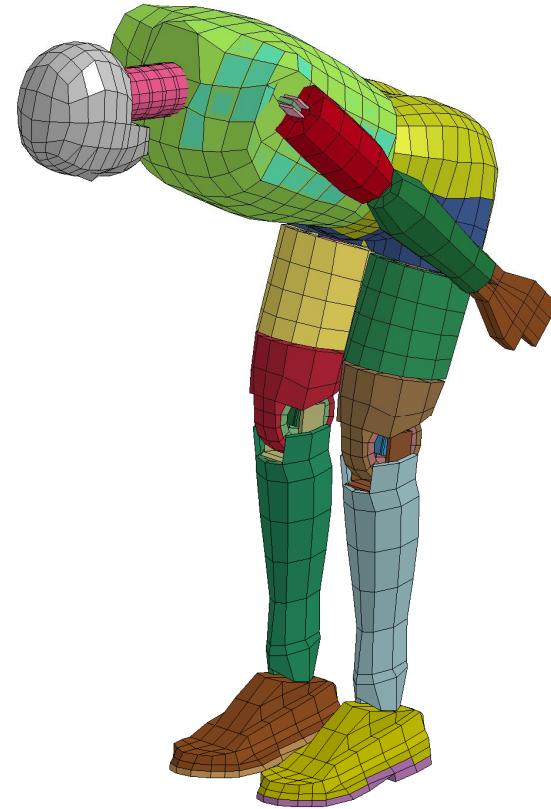
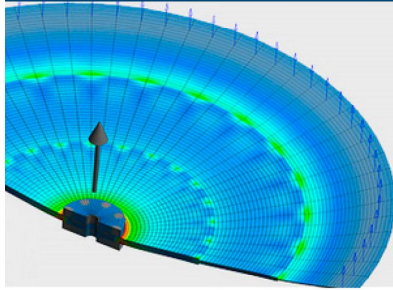
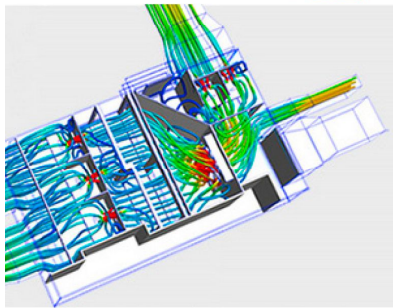


Image courtesy of DYNAmore

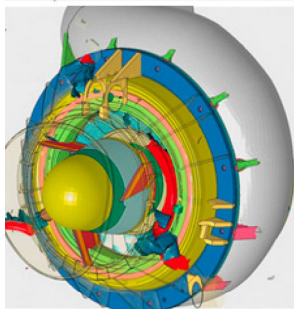
## OUR SERVICES



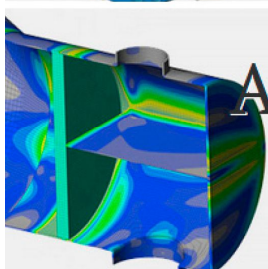
FEA



CFD



LS-DYNA



ASME BPVC

## Our History

Over the years, Predictive Engineering has successfully completed more than 800 projects, and has set itself apart on its strong FEA, CFD and LS-DYNA consulting services. Since 1995, Predictive Engineering has continually expanded its client base. Our clients include many large organizations and industry leaders such as SpaceX, Nike, General Electric, Navistar, FLIR Systems, Sierra Nevada Corp, Georgia-Pacific, Intel, Messier-Dowty and more.

## Purchasing LS-DYNA Software

At Predictive Engineering, we are longtime experts in LS-DYNA software and can help you buy LS-DYNA and then guide you through the acquisition, licensing, installation, support and in-depth LS-DYNA training. We like to say that we don't sell LS-DYNA but we advocate LS-DYNA to clients where it will make them money.

If you would like to learn more about purchasing and learning LS-DYNA, please contact us for our in-depth PDF: LS-DYNA The advanced simulation tool for nonlinear, linear, dynamic, static analysis and multi-physics (DEM, FSI, CFD, Electro-magnetics, SPH, Galerkin EFM & More)



2505 SE 11th, Suite 310, Portland, Oregon 97202

503-206-5571

[info@PredictiveEngineering.com](mailto:info@PredictiveEngineering.com)

[www.PredictiveEngineering.com](http://www.PredictiveEngineering.com)