FINITE ELEMENT ANALYSIS
Predictive Engineering

LS-DYNA® Analysis for Structural Mechanics

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

LSTC
Livermore Software Technology Corp.

Acknowledgements

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

Technical Support Team at Livermore Software Technology Corporation (LSTC)
With special mention to:
Jim Day, LSTC
Philip Ho, LSTC

External to LSTC, we have:
Paul Du Bois, Hermes Engineering, NV

And

DYNAmore, Gmbh, Germany

COURSE OUTLINE

1. INTRODUCTION

1.1 What The Student Can Expect

This class is directed toward the engineering professional simulating highly nonlinear, transient 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

- Explicit 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 toward inoculating you against repeating our mistakes

1.3 How we do it

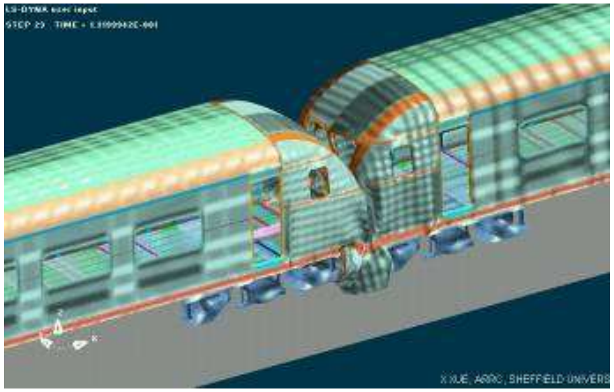
- The class covers the basics in a hands-on manner as taught by an engineer that has had to live by what they have killed.
- Each day will have six to eight Workshops. Each Workshop is part theory, part demonstration and part hands-on practice. Videos are provided for each Workshop allowing the student 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 General Applications

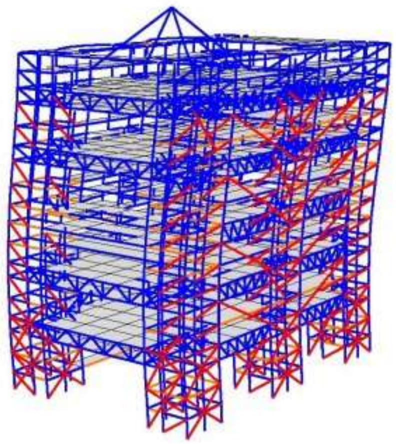
Crashworthiness

Driver Impact

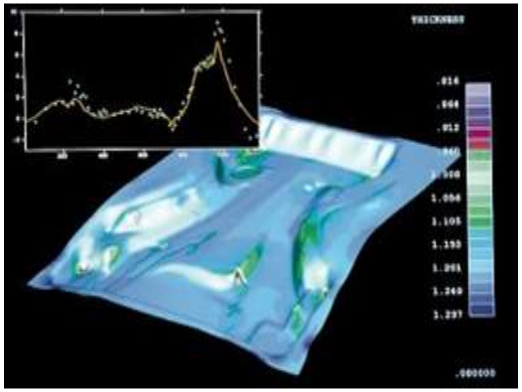
Train Collisions



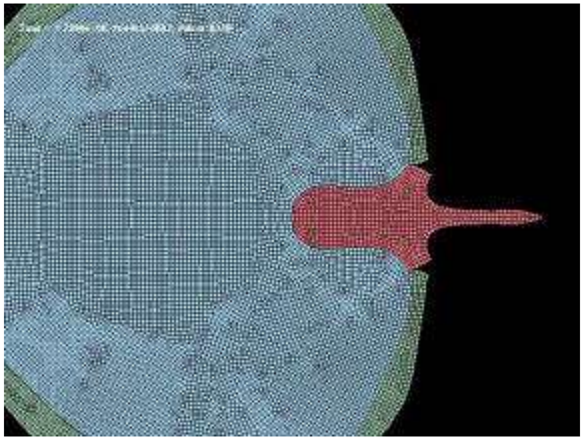
Earthquake Engineering



Metal Forming

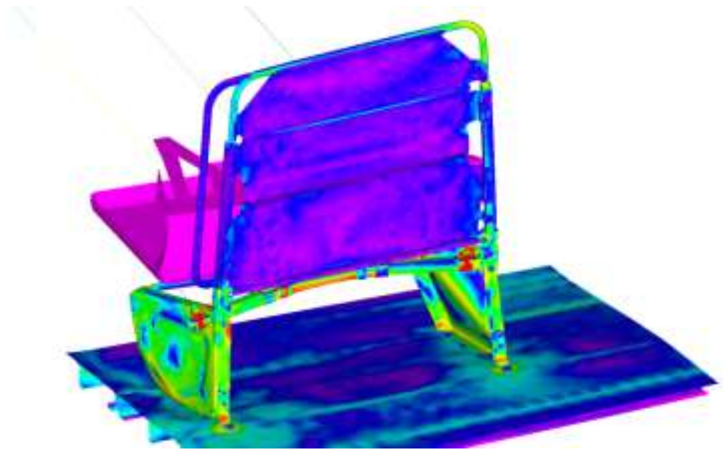


Military

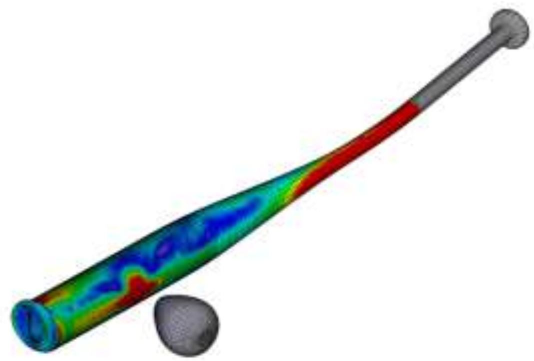


1.5 Specific Applications (Courtesy of Predictive Engineering)

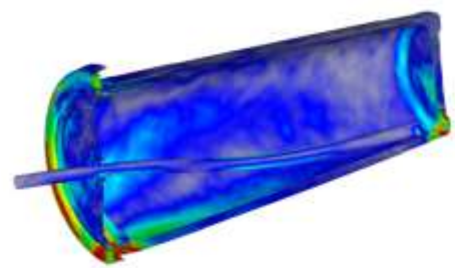
Crash Analysis of Seats



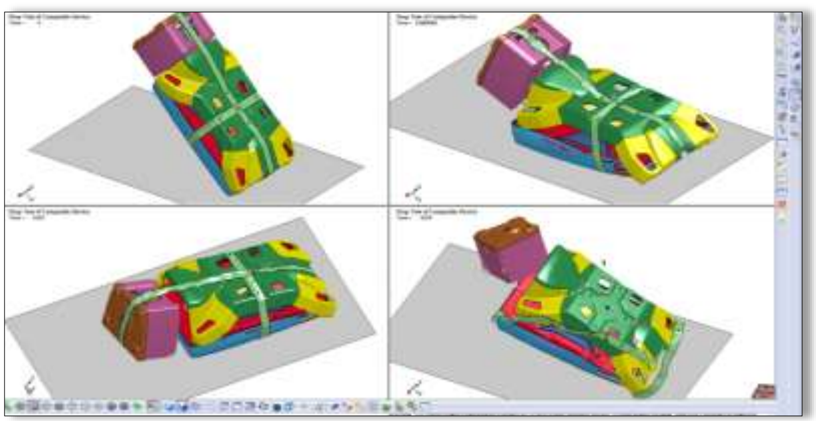
Sporting Goods Equipment



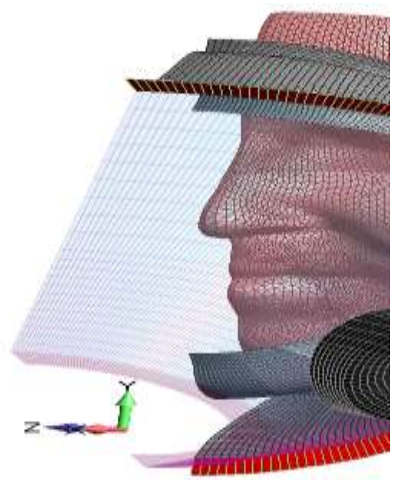
Drop Test Consumer Products



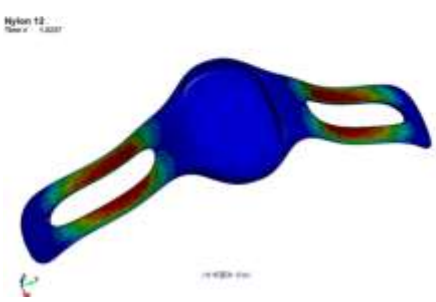
Drop Test of Composites / Electronics



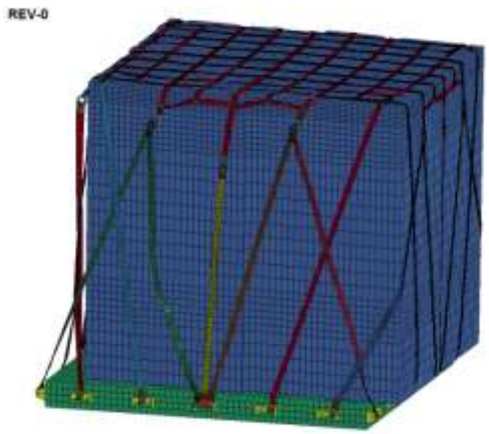
Human Biometrics



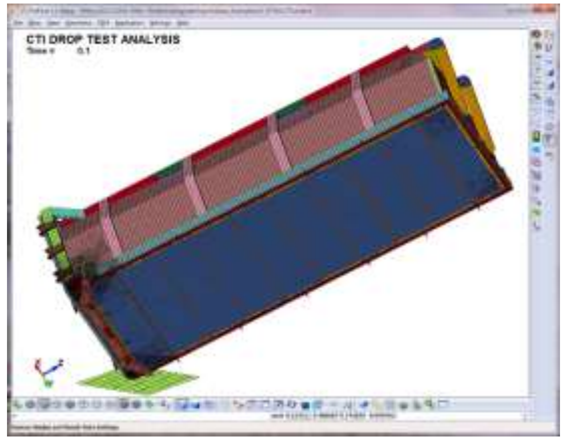
Large Deformation of Plastics



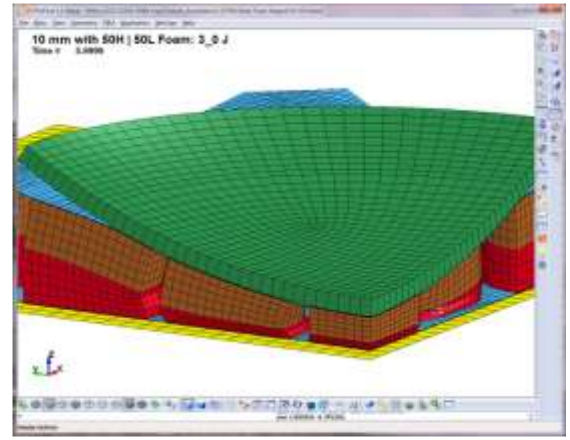
Crash Analysis of Cargo Net



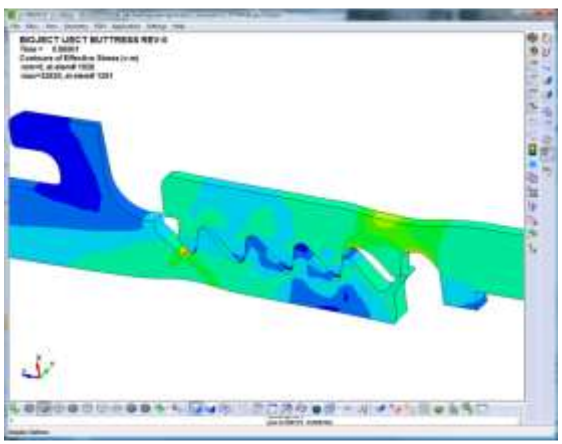
Drop Test of Nuclear Waste Container



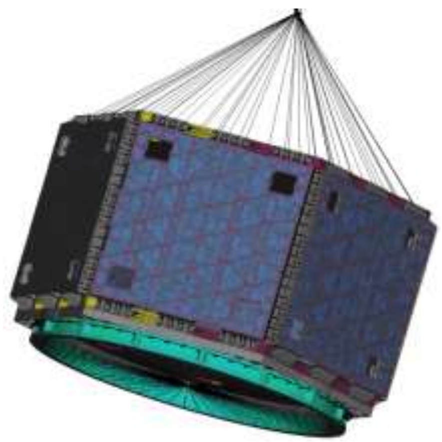
Impact Analysis of Foams



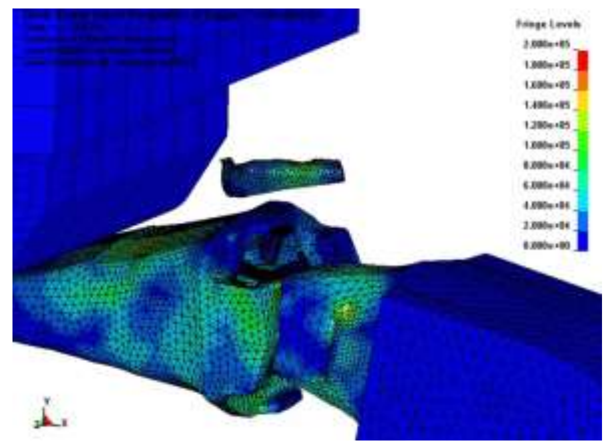
Plastic Thread Design



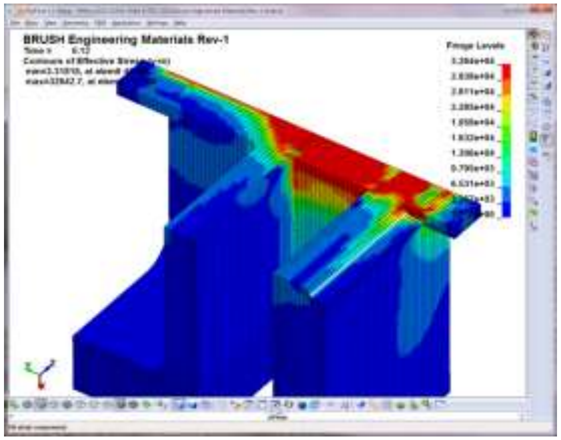
PSD / Modal Analysis



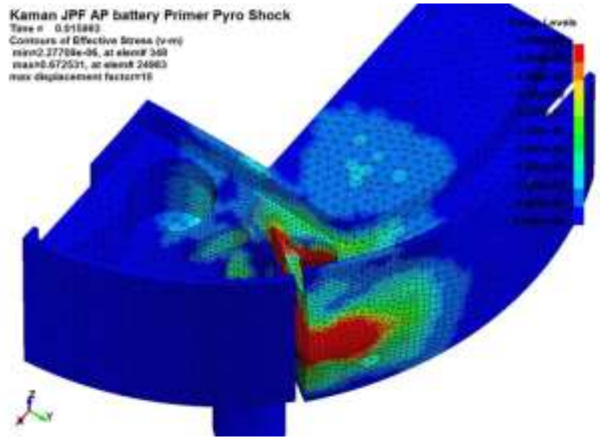
Digger Tooth Failure



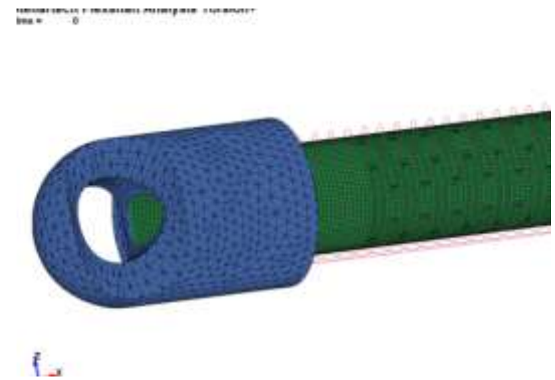
Electron Beam Welding



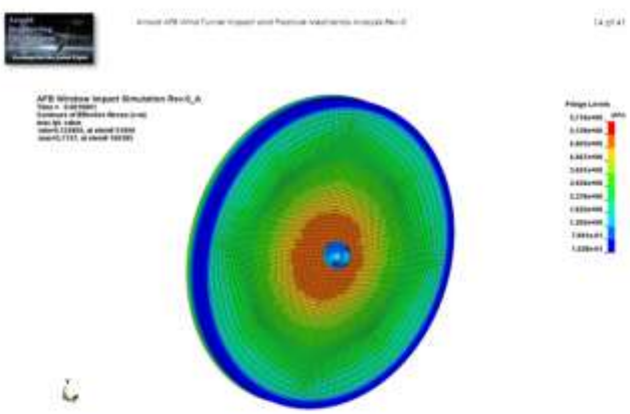
Pyro-Shock Analysis



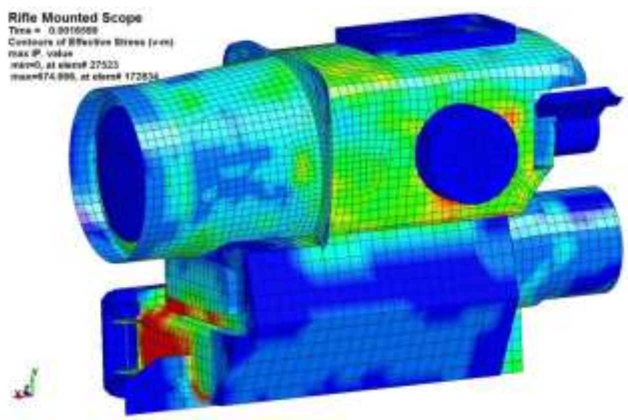
Medical Equipment



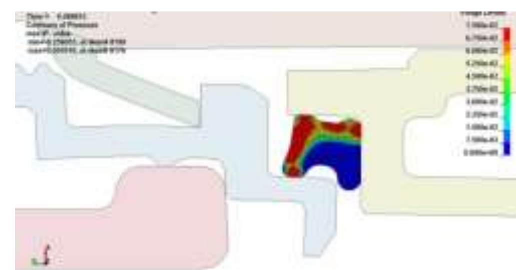
Fracture Mechanics of Glass



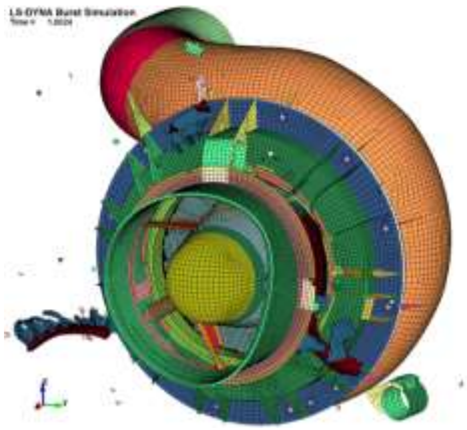
Ballistic Shock Loading of Optical Equipment



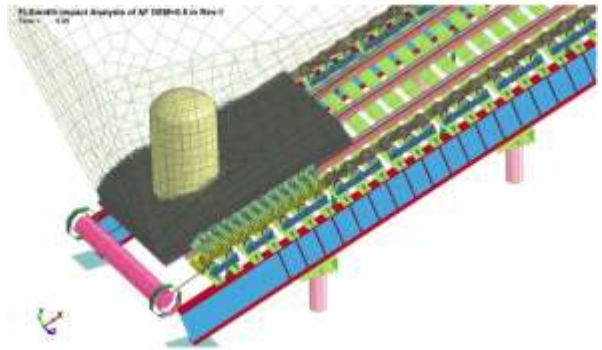
Hyperelastic Medical Seal Analysis



Blade-Out Analysis

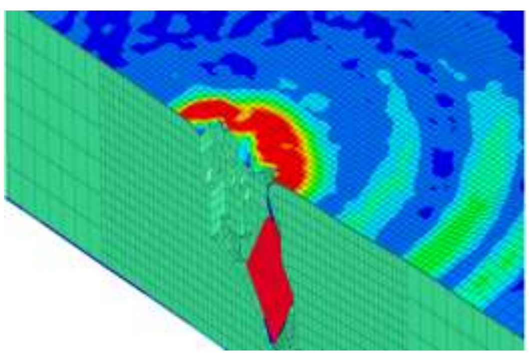


Discrete Element Method for the Mining Industry

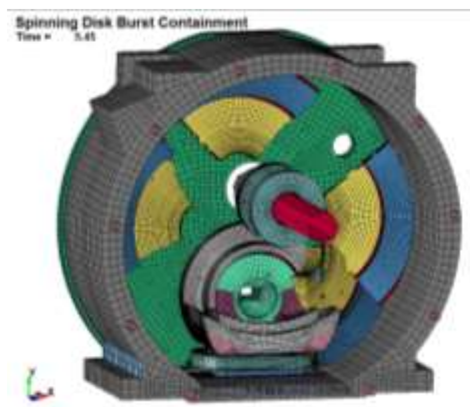


Drop-Test of Hand Held Electronics

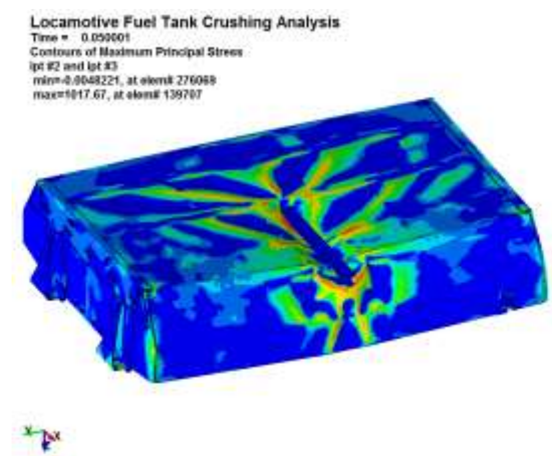
Ballistic Penetration of Al/Foam Panel



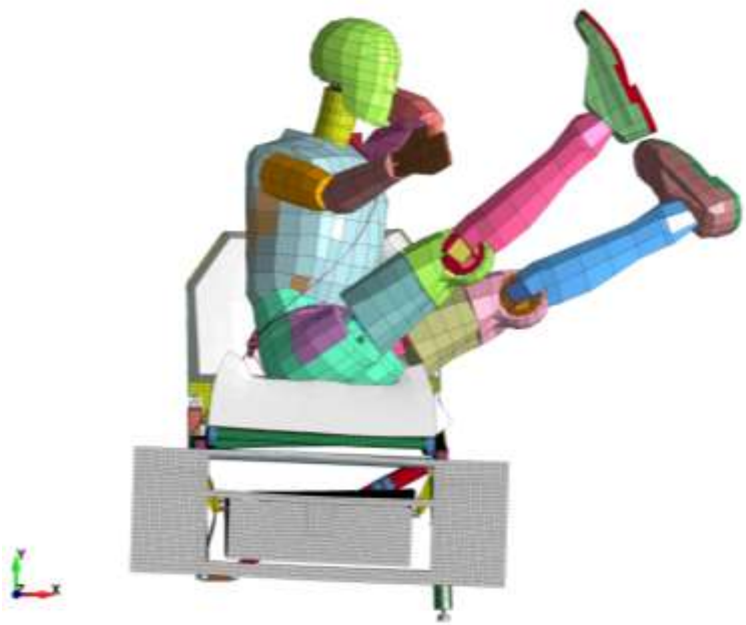
High-Speed Spinning Disk Containment



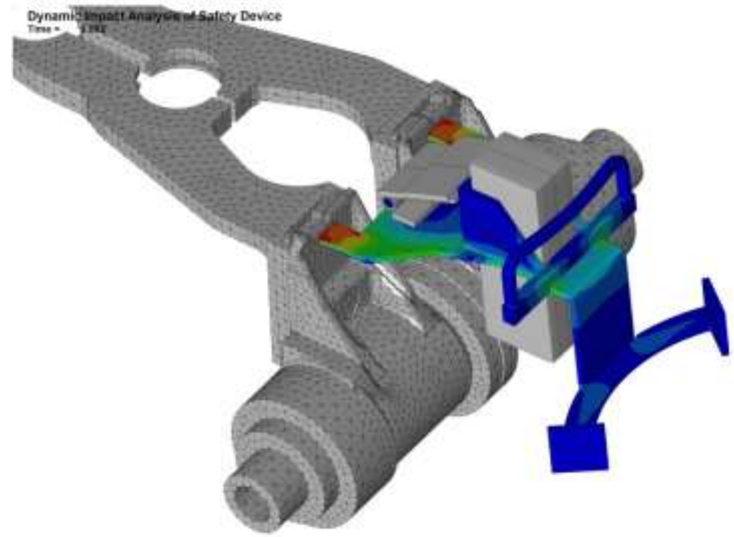
Locomotive Fuel Tank



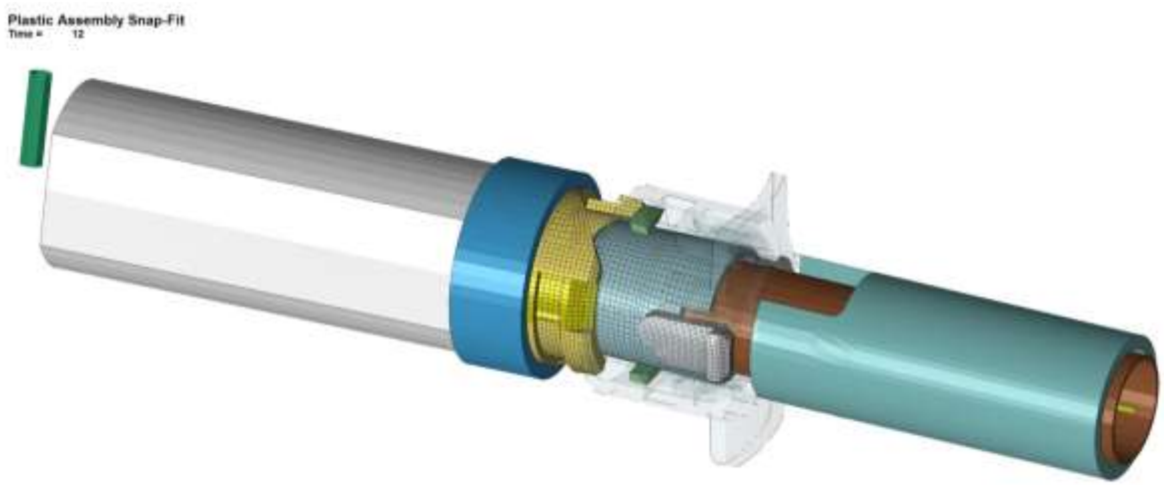
Airplane 16g Crash Analysis



Impact Analysis of Safety Block Device



Snap-Fit Analysis



2. IMPLICIT VERSUS EXPLICIT ANALYSIS

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

2.1 What We Are Solving

Explicit only works when there is acceleration (dynamic) whereas an implicit approach can solve the dynamic and the static problem. For dynamic problems, this means that 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:

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

All these terms are known at time state "n" and thus can be solved directly.

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

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

Given these unknowns, an iterative solution at each time step is required.

2.2 EXPLICIT (DYNAMIC)

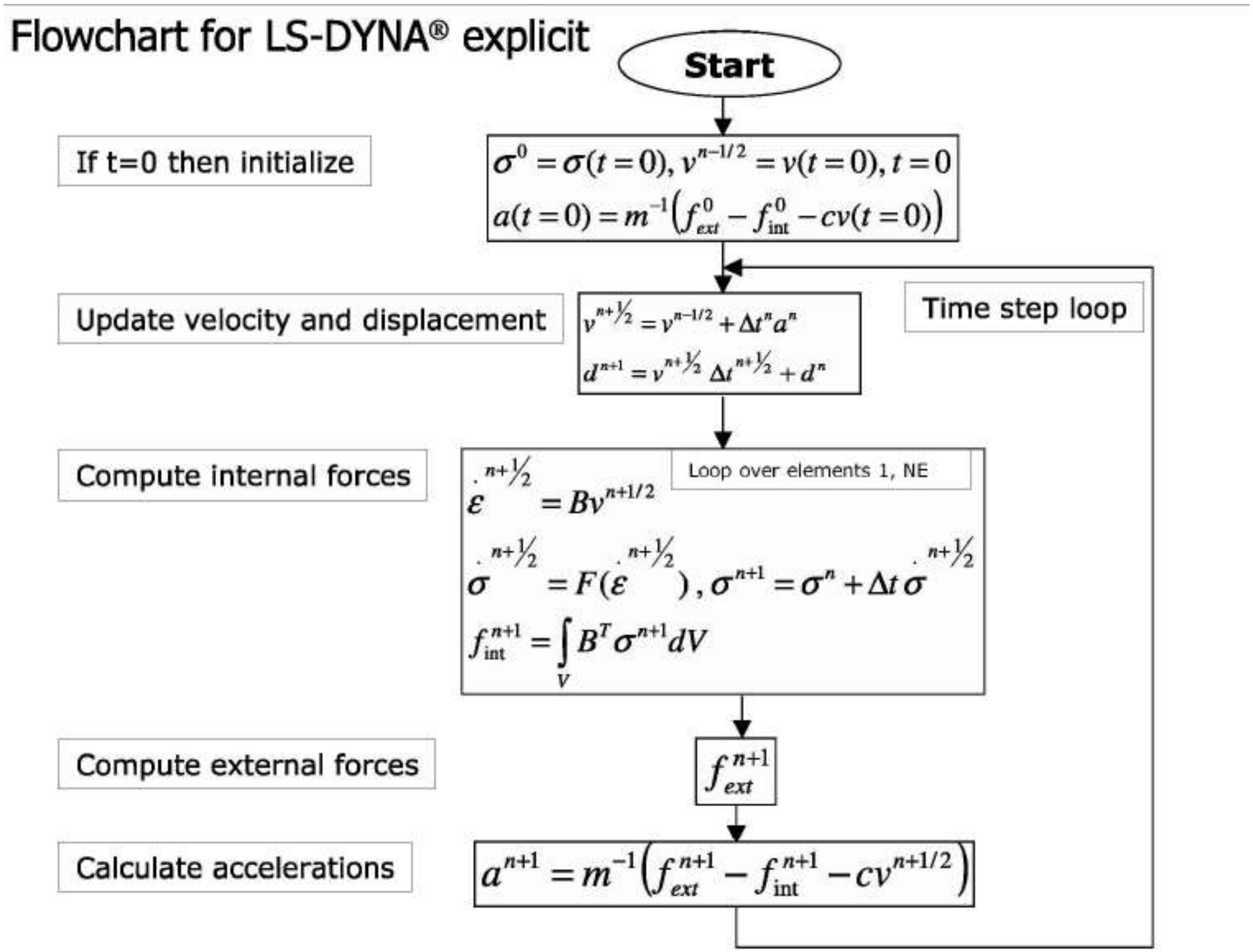
Internal and external forces are summed at each node point, and a nodal acceleration is computed by dividing by nodal mass. The solution is advanced by integrating this acceleration in time. The maximum time step size is limited by the *Courant condition*, producing an algorithm which typically requires 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.** Hence, the explicit solution is limited in time step by the element size and the speed sound in the material under study. Even worse, the smallest element in the mesh can dictate the time step for the whole solution and likewise combined with the stiffest material (fastest speed of sound).

2.3 IMPLICIT (DYNAMIC)

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.

3. FUNDAMENTAL MECHANICS OF EXPLICIT ANALYSIS

3.1 Time Step Significance



3.1.1 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
- 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

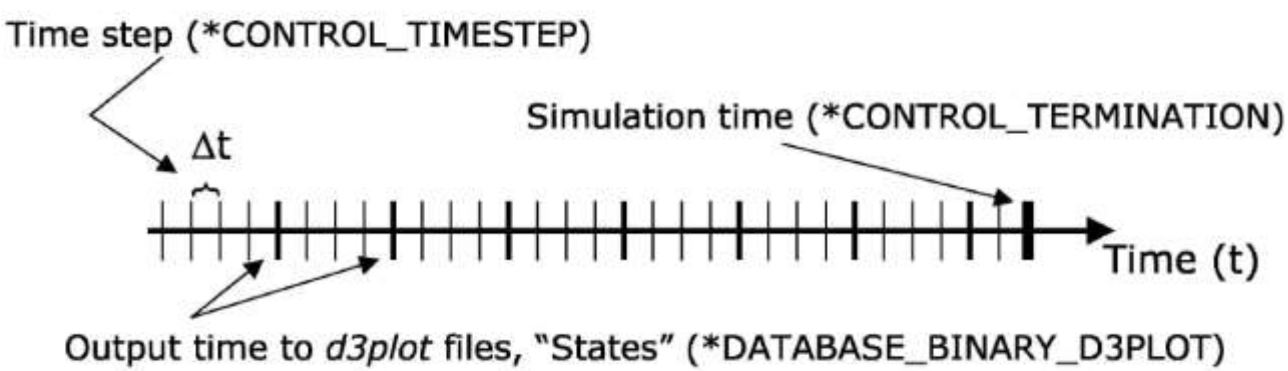


Figure 1: How Solution Time and Result Outputs Are Defined in Explicit

3.2 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.

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

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

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

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.

3.3 Mass Scaling: (Everybody Does it But Nobody Really Likes It)

Explicit Time Step Mass Scaling (*Control_Timestep)*

- Mass scaling is very useful and directly increases the timestep. The concept is simple, **Larger Timestep = Lower Solution Time**
- One can also just simply increase the global density of the material for non-dynamic simulations (i.e., where inertia effects can be considered small).
- *CONTROL_TIMESTEP: Conventional mass scaling (CMS) (negative value of DT2MS): The mass of small or stiff elements is increased to prevent a very small timestep. Thus, artificial inertia forces are added which influence all eigenfrequencies including rigid body modes. This means, this additional mass must be used very carefully so that the resulting non-physical inertia effects do not dominate the global solution. This is the standard default method that is widely used.
- With CMS, a recommended target is not to exceed 5% of the mass of the system or 10% of the mass of any one part. Added mass can be tracked with *DATABASE options of GLSTAT for entire model and MATSUM for individual parts. (*Note: All general recommendations and tips are given in Explicit Model Check-Out and Recommendations.)*

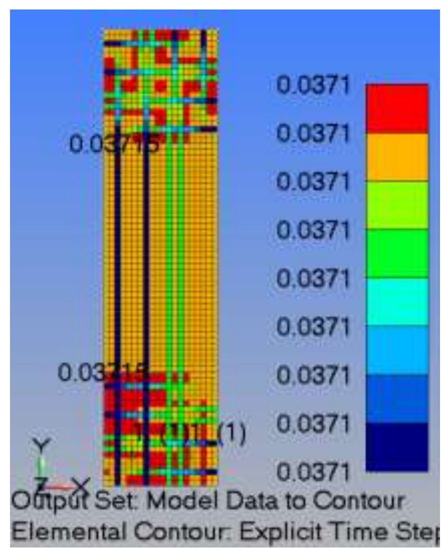
$$\Delta Timstep_{CFL} = TSSFAC \frac{Length_{Element}}{\sqrt{\frac{E}{\rho * Mass\ Scaling}}}$$

$$C_{Aluminum} = \sqrt{\frac{\frac{70}{(1-\upsilon^2)}}{2.71x10^{-6}}} = 5{,}384 \text{ mm/ms}$$

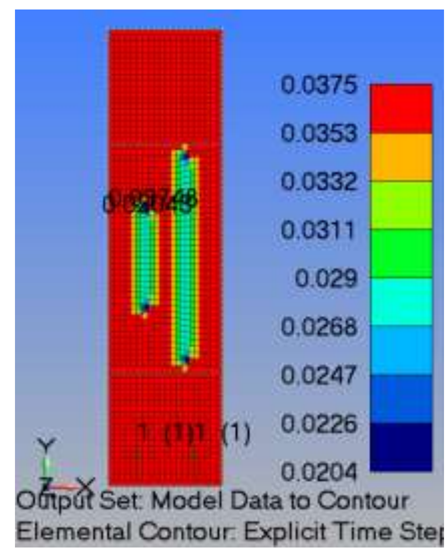
$$\Delta Timestep_{Al} = 0.9 \cdot \frac{200}{5{,}384} = 0.9 \cdot 0.0371 = 0.0334\, ms$$

- LS-DYNA time step is different between Femap and LS-DYNA due to TSSFAC=0.9 (default)
- Mesh quality affects Time Step – just tweak it

Clean Mesh



Mesh Toolbox Tweaked



3.3.1 Workshop: Femap to LS-DYNA with Mass Scaling Basics

What You Will Learn

How to build your first LS-DYNA model from the ground up and how to improve the run times using mass scaling.

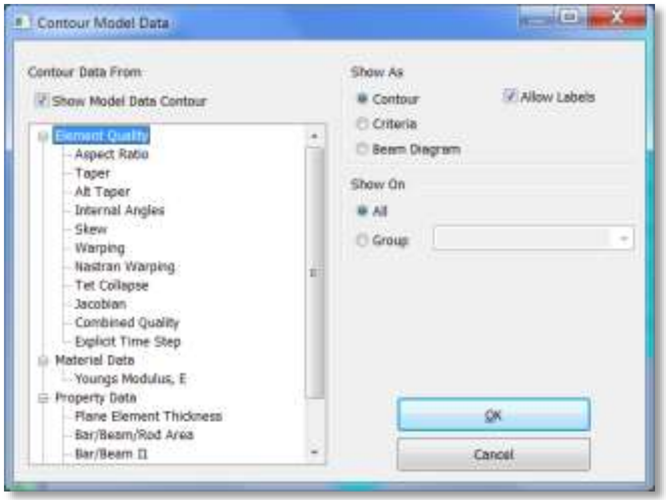
Tasks

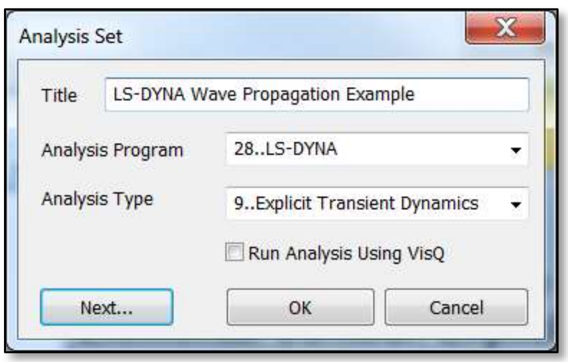
- Open Femap model (Femap to LS-DYNA with Mass Scaling Basics Start.modfem) and create LS-DYNA Isotropic Material and then plate property with thickness = 1.0 (investigate Formulation for plates and just note that there is a lot going on with LS-DYNA).
- Mesh surface and check Explicit Time Step (see Model Data Contour / Show Model Data Contour). *Note that the time step is in milliseconds given the unit system and that the element time step is defined by its shape.*
- Change elastic modulus from 70 to 35 and re-contour time step. Note change and then un-do (CTRL-Z) within Femap.
- Setup Analysis Manager for LS-DYNA Analysis. The analysis will run for 8 milliseconds with results sets at intervals of 0.01. Make sure to select a Load for the analysis. Export to the same directory as the Femap model file. While doing this verify the load application and the constraint set (there ain't any….).

Units: kN-mm-ms-kg

Linear, elastic material model of aluminum:

E	v	ρ
70	0.33	2.71e-6

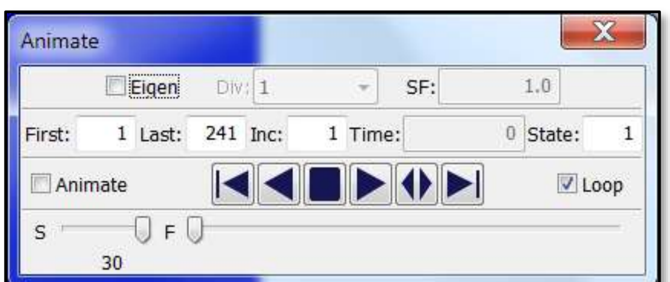


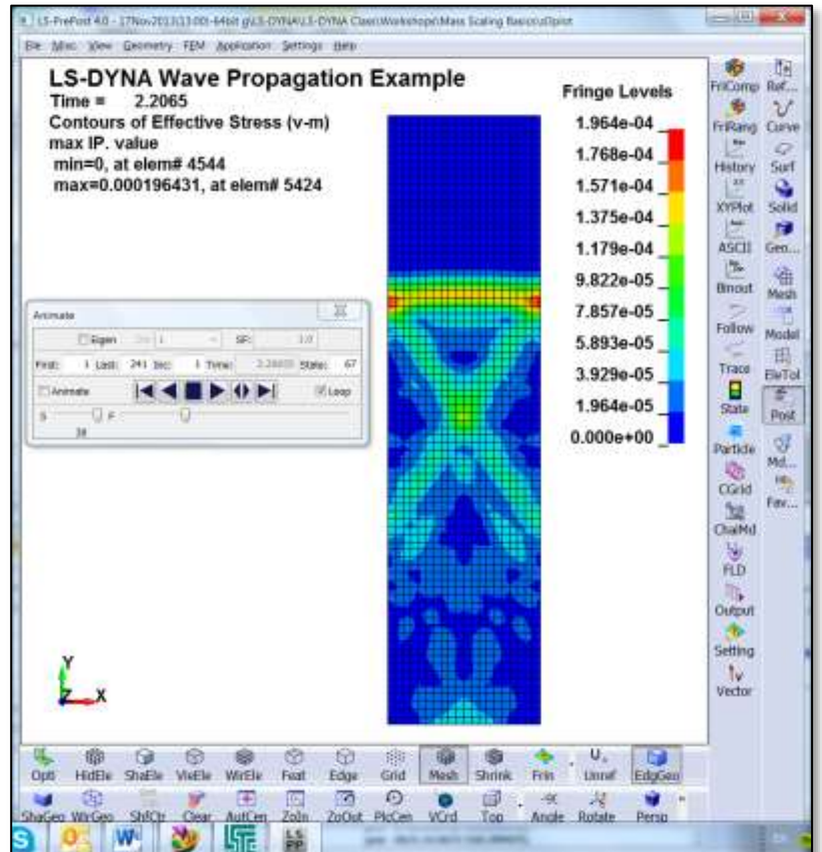


Femap to LS-DYNA with Mass Scaling Basics (continued)

Tasks

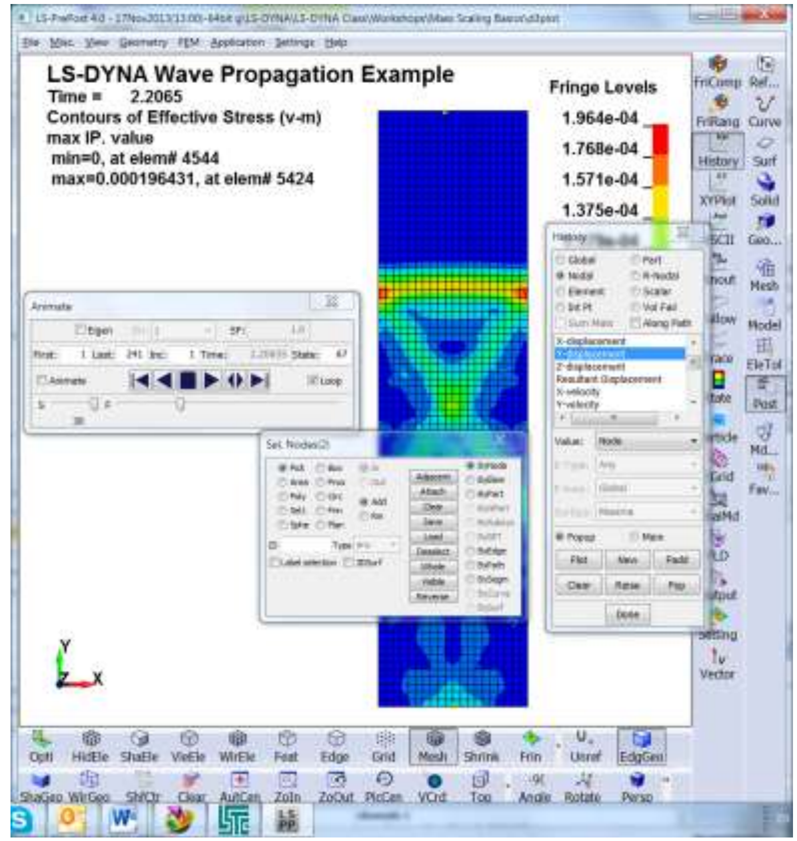
- Run LS-DYNA model file using LS-DYNA.
- Take a look at the Analysis Output and find the time step listing. One will note that it is reduced by 0.9 from what Femap calculates. The reason for this is to ensure that the Courant condition is met.
- Open up LS-PrePost (LSPP) and load the d3plot file (File / Open / LS-DYNA Binary Plot). This “mother file” will then load its subsequent children files, i.e., d3plot01, d3plot02, etc. automatically. To make it easier the first time around, go to View / Toolbar and set the toolbars to Text and Icon.
- Then we will contour the von Mises Stress (Toolbar **Post**, FrinComp, von Mises Stress. Then hit the play button on the Animate dialog box. Sit back and enjoy the stress wave colors.





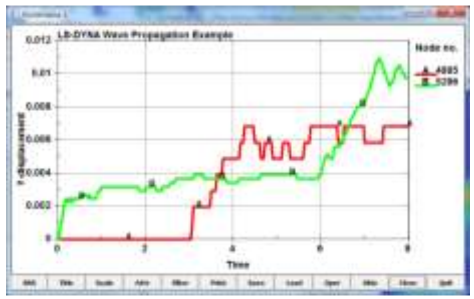
Femap to LS-DYNA with Mass Scaling Basics (continued)

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 Femap, LS-DYNA Program Manager and LSPP. Our test metric is going to be the maximum displacement from a node at the end of the bar.



Tasks

- Within existing LSPP model, open History, select Node, Y-Displacement and then pick a node at the very top of the bar near the center (any'ol node near the center) and then likewise at the bottom, near the center. When done you should have two nodes selected and then hit Plot within the History dialog box. When finished something like this should appear as shown on the right.

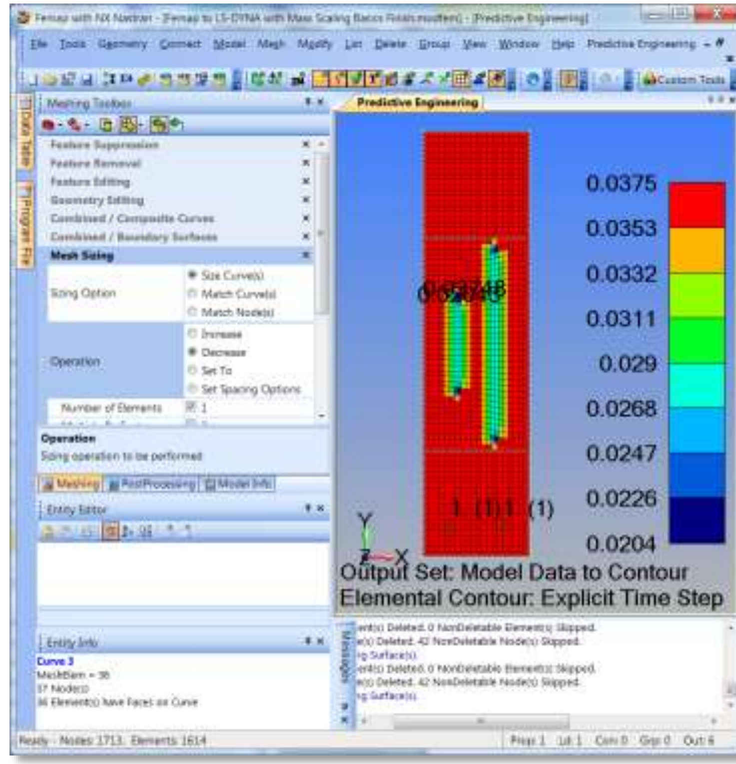


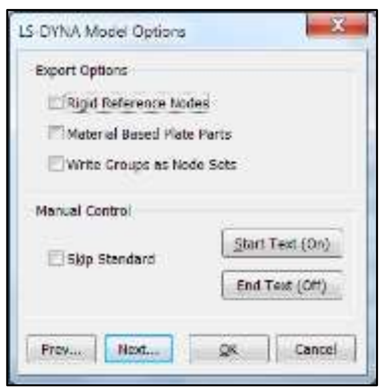
- Note that the maximum displacement at the top is 0.00781 mm with an explicit timestep = 0.0334 ms.

Femap to LS-DYNA with Mass Scaling Basics (continued)

Tasks

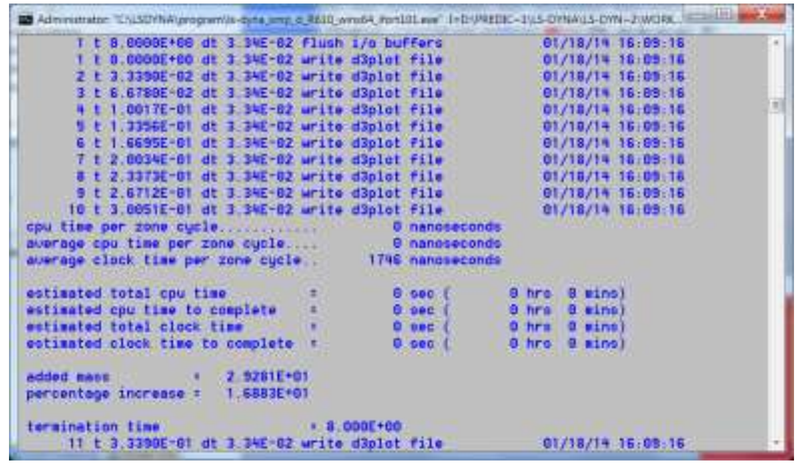
- Go back to the Femap model file and open up the Meshing Toolbox Pane. Decrease the mesh sizing on the right-hand side curve in the middle by four increments. Although we are making the mesh size larger (i.e., the timestep should increase), one will note that the explicit time step will decrease due to non-uniform shaped elements used to transition the mesh. This is more reflective of FEA reality where the mesh is rarely uniform.
- Export Model to LS-DYNA, analyze and create a plot as in the steps above.
- Apply mass scaling to arrive at an explicit time step of 0.0334 ms. See LS-DYNA Keyword *CONTROL_TIMESTEP and insert this keyword into the Femap Analysis Set Manager (see graphic) and rerun the model. *Note: keywords can be entered in a simple comma separated format as shown in the graphic on the right and when the deck is exported, they are inserted into the LS-DYNA analysis file.*





Femap to LS-DYNA with Mass Scaling Basics (continued)

The following task is to get comfortable reading the LS-DYNA output. Although this information can be found in the d3hsp and messag files, it is a bit easier to grab it directly off of the screen. The information we are looking for is the "percentage increase" of mass that the –DTMS option within the *CONTROL_TIMESTEP has added to our model. In this example, we have added mass of 16.9%.



Objective

Apply mass scaling to obtain similar Max. Displacements. Keep in mind that the higher the time step, the faster your analysis will run and in practice, it can take a 120 minute run and allow it to run in 80 minutes with no adverse effects. It is a huge productivity boost.

Model	Time Step	% Mass Added by Mass Scaling	Max. Displacement
Starting Point	0.0334 ms	0.00%	0.00781 mm
Skewed Mesh (-4x)	0.0184 ms	0.00%	0.00781 mm
Skewed Mesh with Mass Scaling	0.0334 ms	16.8%	0.00684 mm

Class Bonus: What happens when the CFL Criterion is not met? (e.g., set TSSFAC=2.0)

3.3.2 Instructor Led Workshop I: 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.
- 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.

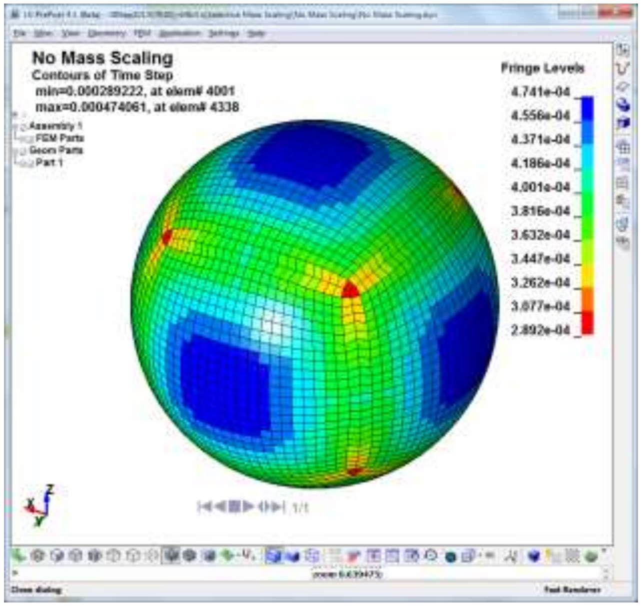
Analyst Note: Would this make your dynamic (F=ma) analysis more conservative or less?

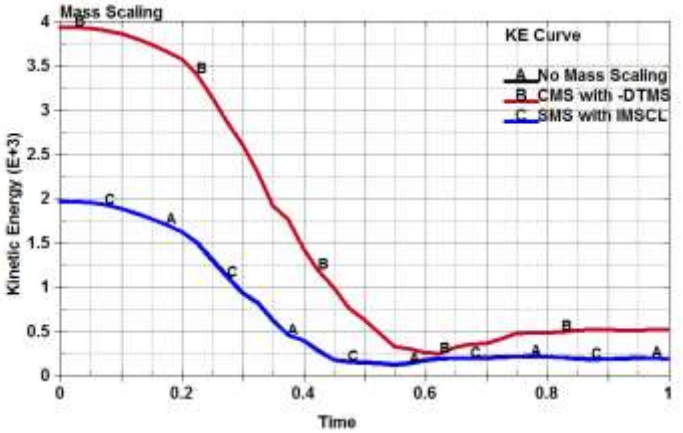
- Conventional mass scaling (CMS) has morphed to using the negative (-)DT2MS option as the recommended default.
- Selective mass scaling (SMS): Using selective mass scaling, only the high frequencies are affected, whereas the low frequencies (rigid body bodies) are not influenced; thereby, a lot of artificial 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.

Solution time is 10 seconds for no mass scaling and 5 and 6 seconds for SMS and CMS respectively. SMS is more computationally expensive but has large benefits for some models.

Example Courtesy of www.DynaSupport.com

Time step ranges from 2.89 to 4.74*e*-4





3.4 Implicit Mesh versus Explicit Mesh Characteristics

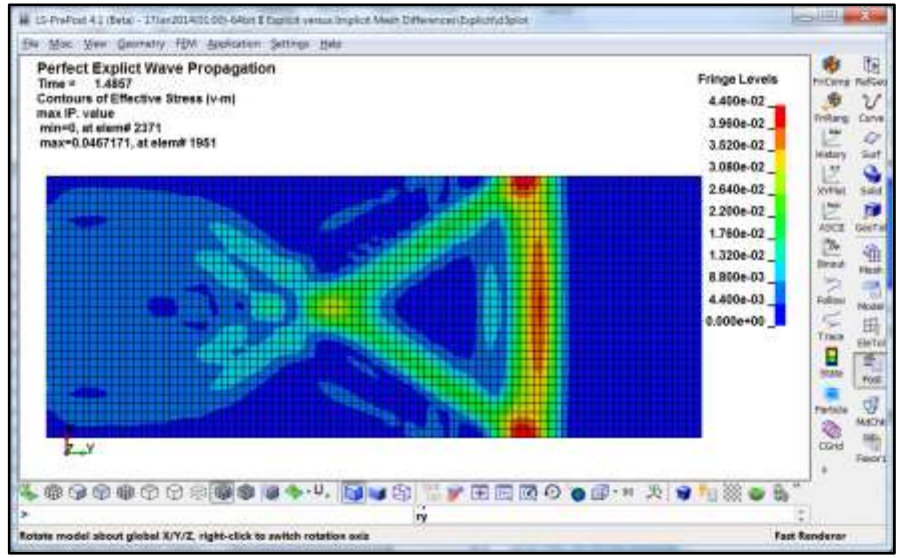
3.4.1 Instructor Led Workshop II: Implicit versus Explicit Mesh Differences

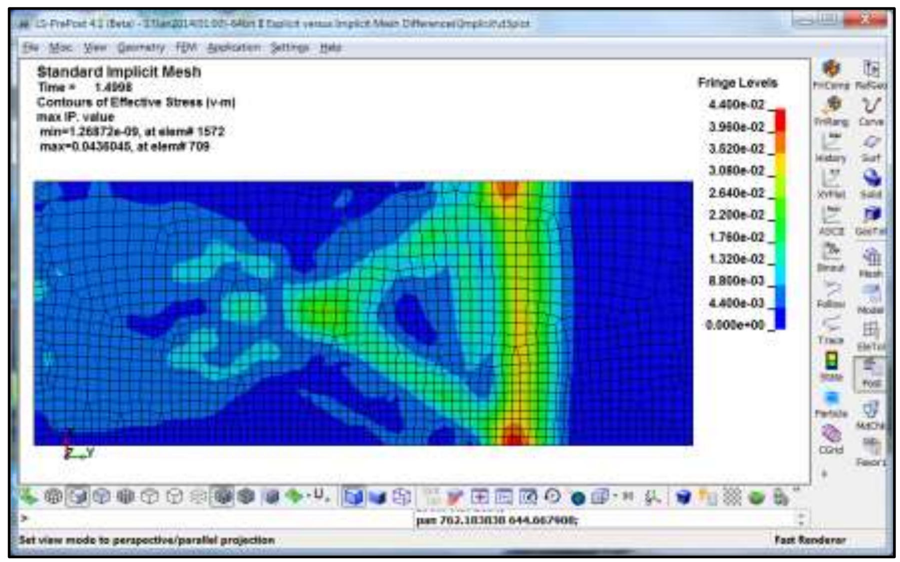
Meshing for Accuracy

- 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 depth, etc. are more important. These parameters 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.

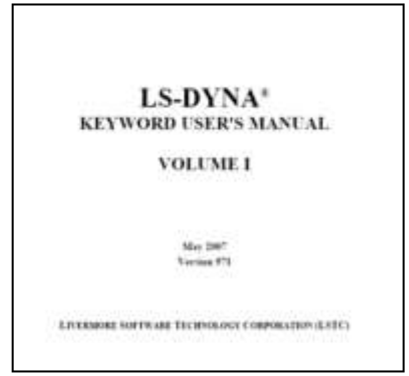




4. FIRST LS-DYNA MODEL: GETTING STARTED

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.


LS-DYNA®
KEYWORD USER'S MANUAL
VOLUME I

LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)

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 **unimportant** (except *KEYWORD and *DEFINE_TABLE)
- 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*)

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
Ton (1,000 kg)	mm	s	N	MPa	N-mm	7.83e-09	2.07e+05	9.806e+03
Lbf-s^2/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**

Another great site: **www.dynasupport.com**

LS-DYNA Examples: **www.DYNAExamples.com**

LS-DYNA Conference Papers: **www.dynalook.com**

Newsletter: **www.FEAInformation.com**

Yahoo Discussion Group: **LS-DYNA@yahoogroups.com**

Aerospace Working Group: **awg.lstc.com**

LSTC Program Download Site

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

SMP Version: ls-dyna

MPP Version: mpp-dyna

SMP/Windows: pc-dyna

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).

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:

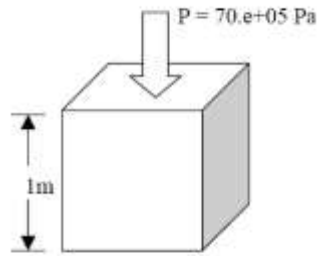
- **sw1** A restart file is written and LS-DYNA terminates
- **sw2** LS-DYNA responds with current job statistics
- **sw3** A restart file is written and LS-DYNA continues
- **sw4** A plot state is written and LS-DYNA continues
- **swa** Dump contents of ASCII output buffers
- **stop** Write a plot state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

4.6 WORKSHOP I: LS-DYNA GETTING STARTED EXAMPLE

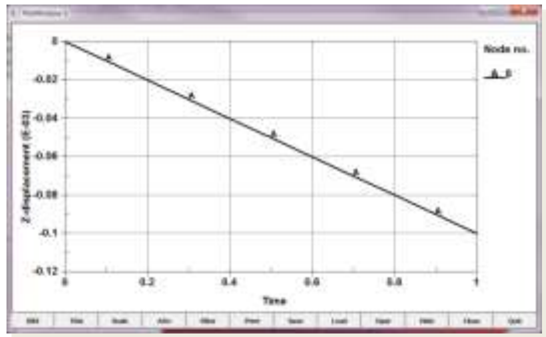
This workshop uses the LSTC Getting Started Example material and a LS-DYNA model has been prepared. This material can be found in the Students' "Class Reference Notes" folder.

Goals

- Open Windows Notepad and build LS-DYNA Keyword deck by hand-entry. Use comma separated format.
- For each command, consult the Keyword Manual.
- Analyze your model.
- Post process the results within LS-PrePost
- If time exists proceed to other examples.



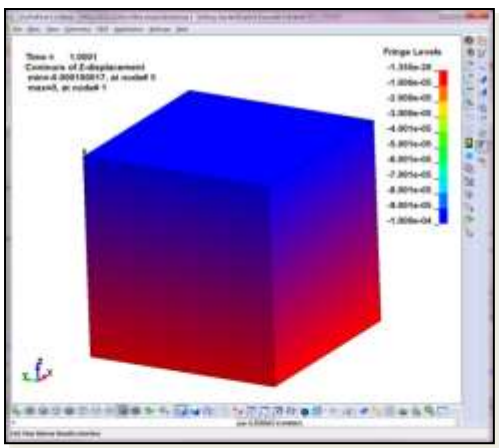
Aluminum 1100-O	
density	2700 kg/m^3
modulus of elasticity	70.e+09 Pa
Poisson Ratio	0.3
coefficient of expansion	23.6e-06 m/m K
heat capacity	900 J/kg K
thermal conductivity	220 W/m K



The vertical displacement due to a 70.0e+05 Pa pressure load can be calculated by

$$\Delta l = \frac{Pl}{E} = \frac{(70e+05)(1)}{(70e+09)} =$$

1.0e-04 m

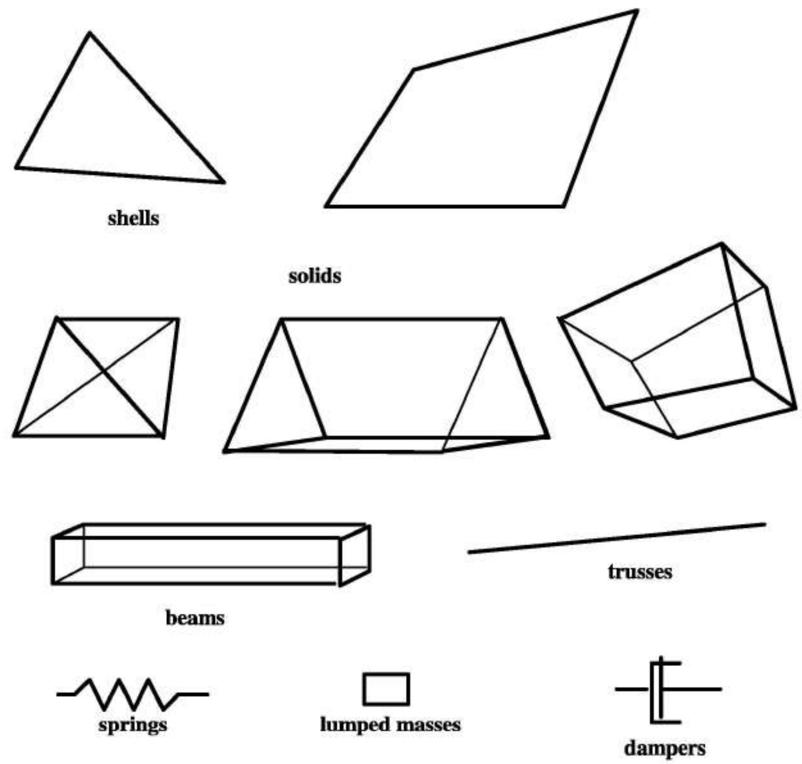


5. EXPLICIT ELEMENT TECHNOLOGY

5.1 ELEMENT TYPES IN LS-DYNA

There are many different element types in LS-DYNA:

- Point elements (mass, inertia)
- Discrete elements (springs, dampers)
- Beams
- Solids (20 and 3D, Lagrangian, Eulerian, ALE)
- Shells
- Thick Shells (8 noded)
- 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 Guassian 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_v12-1.pdf

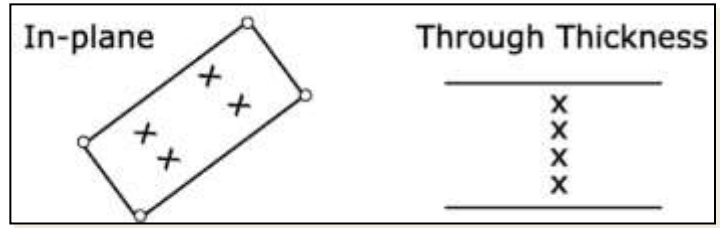
5.2 One Guassian Point Isoparametric Shell Elements and Hourglassing

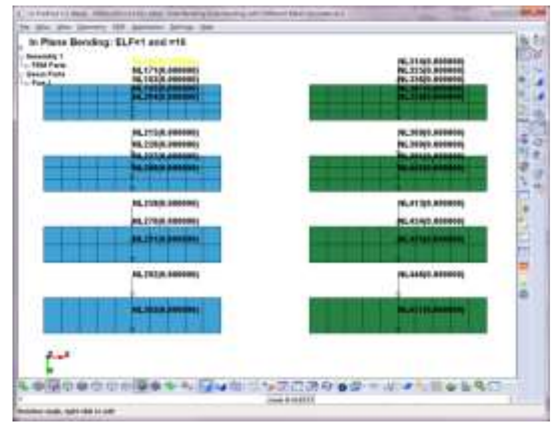
5.2.1 Instructor Led Workshop III: Explicit Element Technology | A: Side Bending

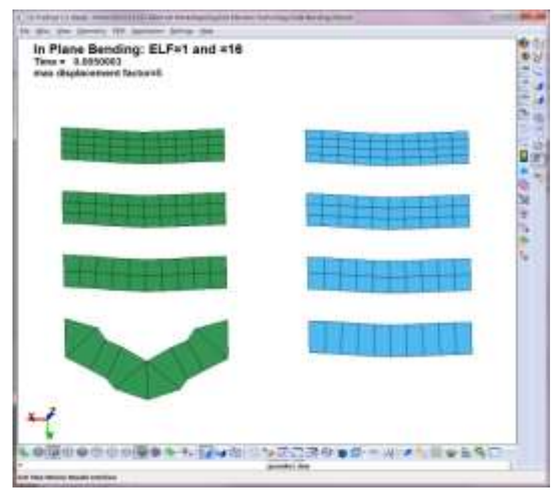
Isoparametric Shell Elements

Default element is **one** Guass point in-plane (ELFORM=2)

- This default formulation is efficient and generally the most robust formulation for large deformations.
- The example shows that under-integrated elements have severe problems in bending. The recommended number of through thickness elements is three (3). However, fully integrated (ELFORM=16) does an adequate job with one or two. Computationally 3x more expensive than the default formulation (ELFORM=2).
- Importantly, it is not always possible to use only ELFORM=16 due to computational expense and care must be taken with using the default formulation in situations where only one element through thickness is possible.
- Increasing the number of elements can be problematic due the CFL timestep condition since three elements over a narrow width of strip will always cause a severe reduction in timestep.
- Recommended size is 5 mm for steel and aluminum and thus yields a time step of approximately 1 µs.







This section courtesy of LSTC and Paul Du Bois, Hermes Engineering NV

5.2.2 Instructor Led Workshop III: Explicit Element Technology | B: Out-of-Plane Bending with Plasticity

Isoparametric Shell Elements*

- Only two element formulations are recommended: ELFORM=2 and ELFORM=16 (keep it simple).
- Number of through-thickness integration points (NIP) controlled by user:

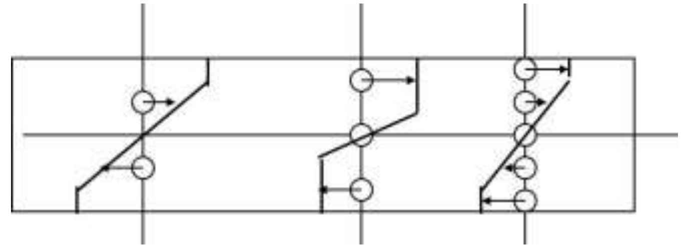
NIP 1: Membrane Behavior

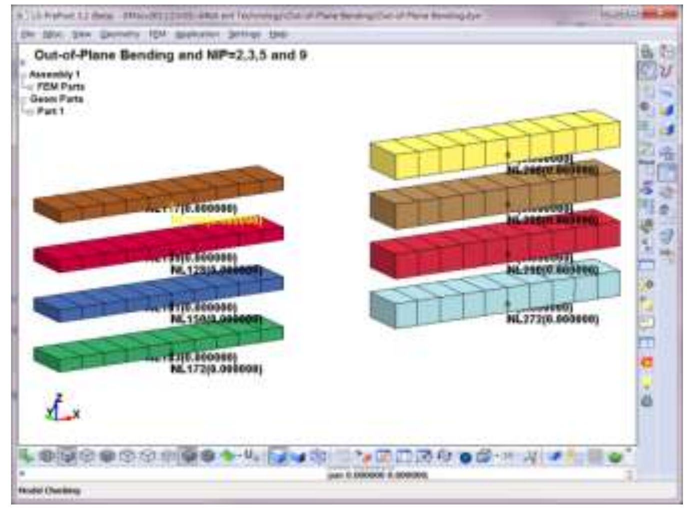
NIP 2: Linearly Elastic Behavior (default)

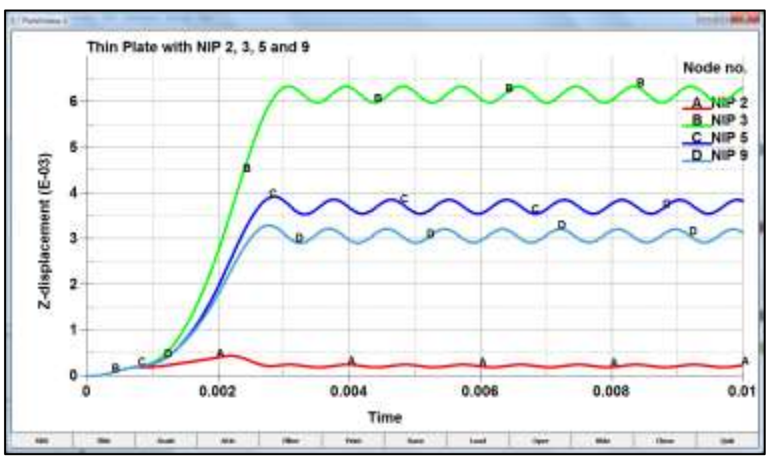
NIP 3+: Recommended for Nonlinear Materials

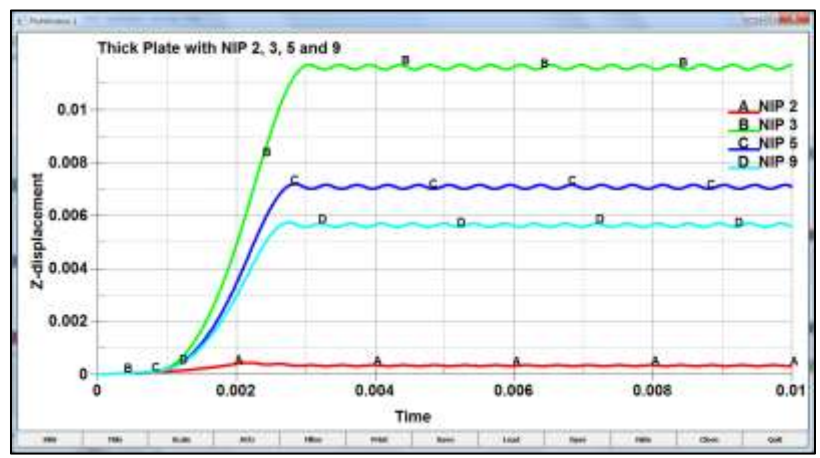
Optimum NIP for Nonlinear Plasticity = 5

- ELFORM=2 not well-suited to warped geometries unless BWC warping stiffness with full projection is invoked or use ELFORM=16. Warping is something that is troublesome whether the analysis is implicit or explicit.





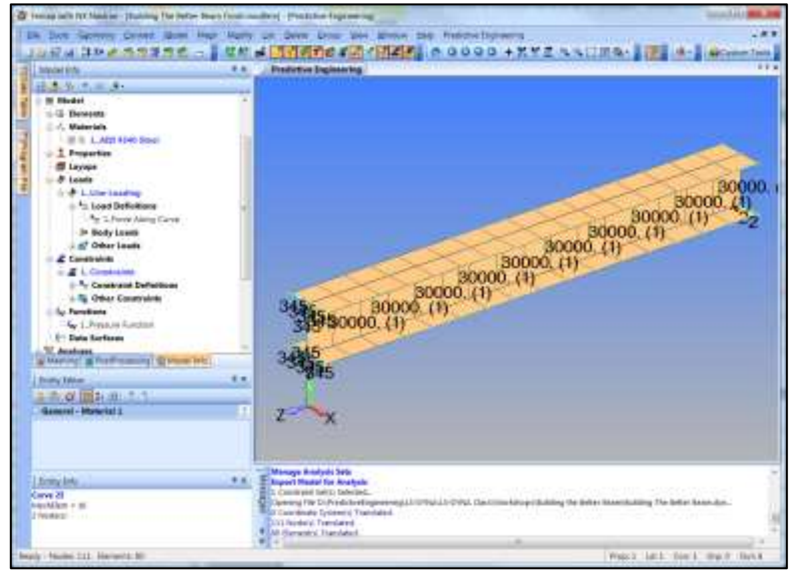




5.2.3 WORKSHOP: BUILDING THE BETTER BEAM

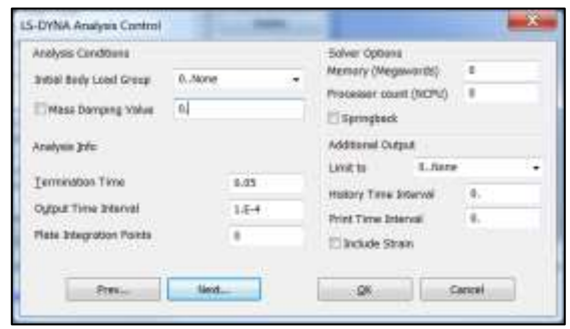
Objective

The importance of mesh density and plate out-of-plane integration will be demonstrated through the use of this simply-support I-beam model (half-symmetry). The material model is steel with a yield stress of 100,000 psi and a tangent modulus of 200,000 psi. The workshop will start with a Femap model shown on the right.



Tasks

- Setup LS-DYNA Analysis (Femap Analysis Set Manager) with a termination time = 0.05 and Output Time Interval =0.0001.
- Measure maximum displacement at the end of the beam and record this value on the table below.
- Then, increase mesh density in web, record maximum displacement. Lastly increase Plate Integration Points = 5 and record maximum displacement.



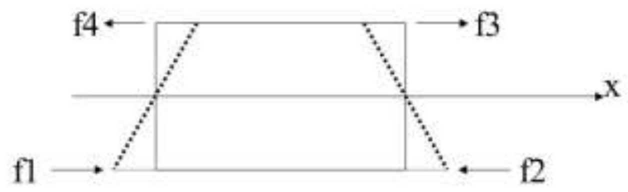
Model	Mesh Density	Integration	Max. Disp.
Start	1	0	60
Refine	4	0	3
		5	

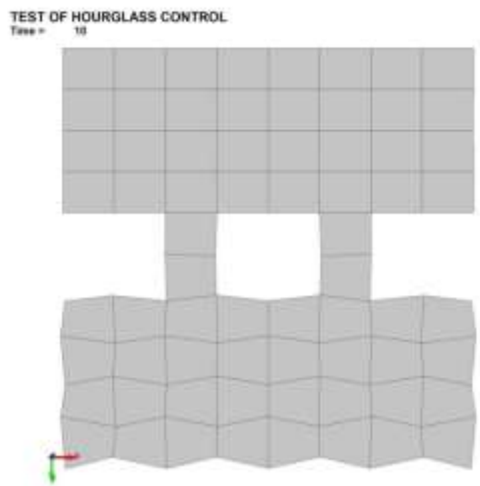
*Extra Task: Open up LS-DYNA deck in LSPP (File / Open / LS-DYNA Keyword File) and look at the Keyword *SECTION (Model / Keywrd / *SECTION and note where the plate integration points are defined, read the manual if necessary for clarification. Go back to one element thru the web and change the element type to full-integration, fast.*

5.2.4 Workshop II: Hourglass Control/Hourglass

Isoparametric Shell/Solid Elements and Hourglassing*

- All under-integrated isoparametric elements (one Guass point) have hourglassing present. It is a non-physical "zero-energy" mode of deformation.
- Fully Integrated formulations do not hourglass. Additionally, tetrahedron and triangular elements do not hourglass but are overly stiff in many applications.
- *Control_Hourglass or *Hourglass to set hourglass control.
- Use default unless additional documentation is consulted (e.g., see Review of Solid Element Formulations Erhart.pdf (Class Reference Notes / Solid Elements).
- Hourglass energy should be less than 10% of the internal energy at any stage of the analysis (use *CONTROL_ENERGY (HGEN=2) to calculate hourglass energy).
- In LSPP, check glstat for total hourglass energy and then matsum for individual part energy.





How to Limit Hourglassing

- Apply pressures instead of point loads.
- Refine mesh
- Selectively use ELFORM=16 (3x computational cost)

Workshop II: Hourglass Control

- Evaluate current model for hourglassing. Plot internal energy and hourglass energy.
- Read Hourglass Material.
- Attempt fix with different hourglass type.
- Switch ELFORM to 16.

5.2.5 Workshop III: Meshing for Explicit Success | Meshing for Explicit Success Start.modfem

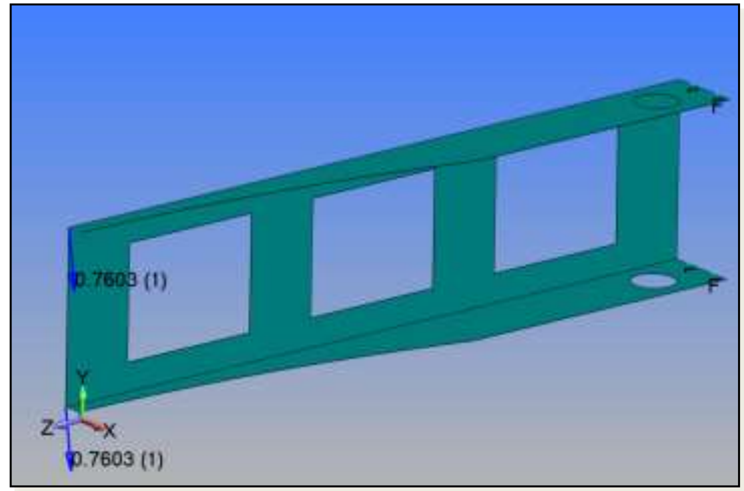
Element Quality for Explicit Analysis

Orthogonal Meshes

In explicit analysis, regular meshes are important to represent the wave propagation problem with minimal dispersion. When the mesh is irregular, the wave front will become dispersed and degrade the quality of the results.

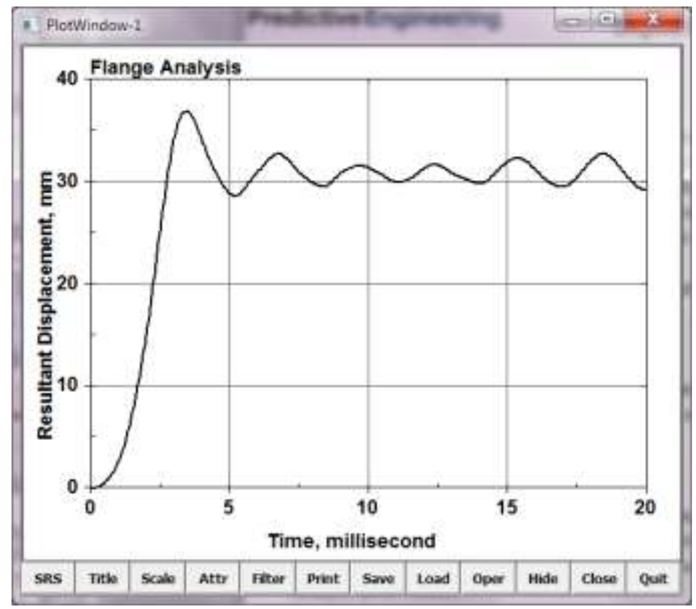
Ideally, mesh lines should be orthogonal or parallel to the stress wave front (and thus orthogonal).

Note: The stress wave front is that induced by the dominant load.



Workshop III: Mesh for Explicit Success

The student meshes the geometry to obtain the best quality results possible with the minimum element count and CPU cost.



Tasks

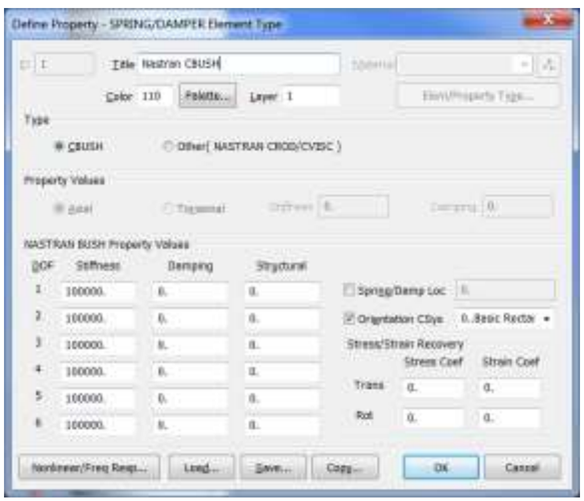
- Mesh geometry using a mesh sizing of 5 mm.
- Setup Analysis Manager with a termination time of 10 milliseconds with data output at 50 μs (0.050 ms) intervals.
- Review results and plot resultant displacement at the top corner node where the load was applied (+Y).
- Investigate the following by recording the effects of: i) Plate NIP, ii) Mesh Density, iii) ELFORM & iv) Mass Scaling on the displacement at the end of the beam.

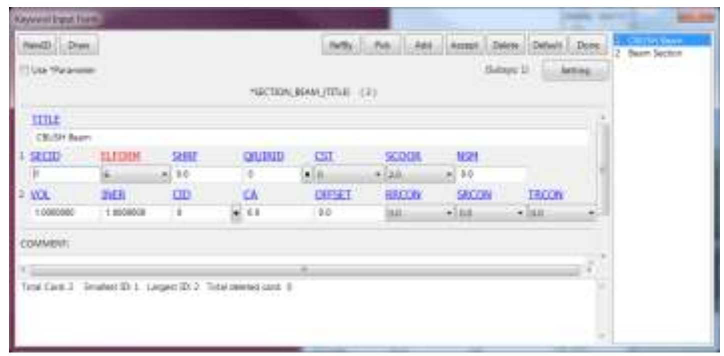
5.3 Scalar Elements (Nastran CBUSH Equivalents)

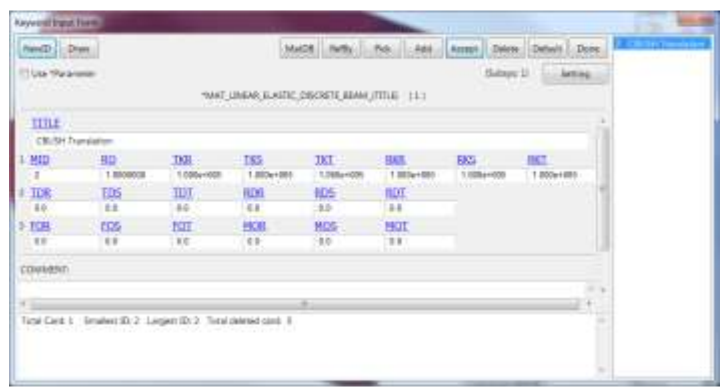
The CBUSH element is of such general utility that it merits its own special section on how to obtain equivalent behavior within LS-DYNA. One of the concepts that merit attention is that an explicit analysis always requires mass whereas a static implicit analysis only requires stiffness. Hence, in a static analysis, one can have elements with zero length (i.e., zero mass) whereas in an explicit analysis mass must be present and likewise a finite length.

Although LS-DYNA has several methodologies to arrive at simulation the behavior of Nastran CBUSH element (e.g., *ELEMENT_DISCRETE), we will present the most basic method and the one recommended by LSTC. In Nastran, the CBUSH property card specifies orientation and stiffness. In LS-DYNA, these capabilities are handled by two cards: (i) *ELEMENT_SECTION, ELFORM=6 (orientation) and (ii) *MAT_LINEAR_ELASTIC_BEAM (stiffness values).

Nastran CBUSH **LS-DYNA Equivalent *SECTION_BEAM & *MAT_66**

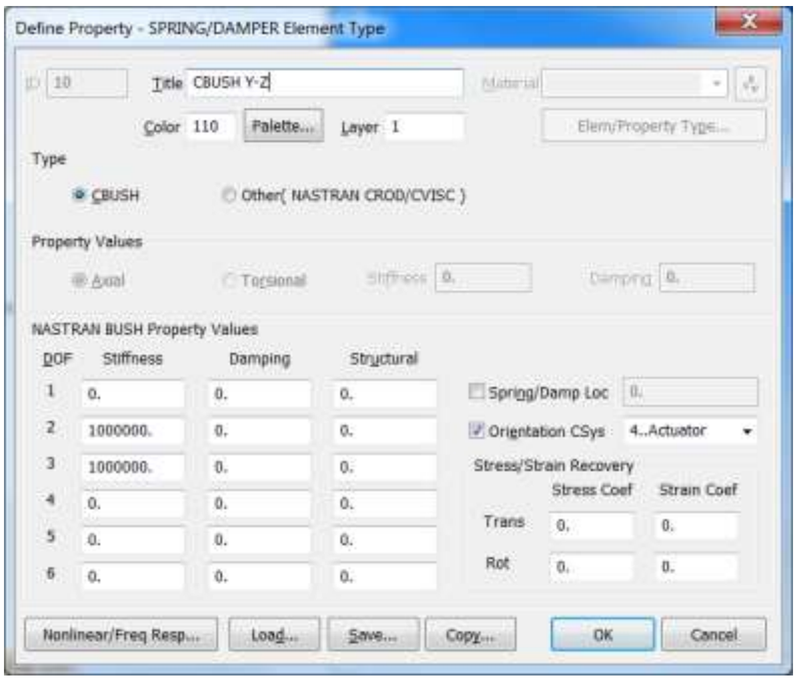






If the connection has zero length, then the *SECTION_BEAM (ELFORM=6) field SCOOR should be set to a value corresponding to the requirements of the analysis. In other words, take a look at the manual and www.dynasupport.com under discrete beams since the choice of SCOOR is not obvious. Our general default is SCOOR=3. Importantly, a CID must be defined or if CID=0, then the beam follows the global coordinate system (just like Nastran).

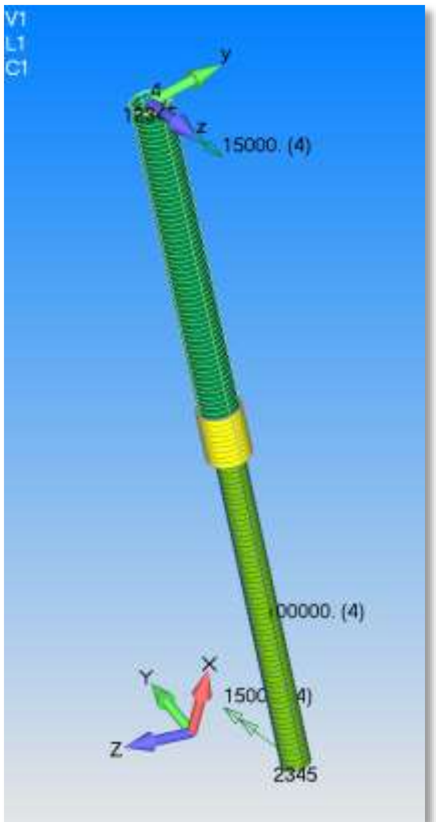
The mass of the element is calculated the field VOL and INER (if torque is present) and the mass density of the material. A basic element is given below for a zero length "CBUSH" element in a unique coordinate system:

Nastran CBUSH Property Card	Equivalent LS-DYNA Keywords
	***SECTION_BEAM_TITLE** CBUSH Equivalent Element $# secid elform shrf qr/irid cst scoor nsm 1 6 1.000000 2 2 3.000000 0.000 $# vol iner cid ca offset rrcon srcon trcon 1.000000 1.000000 1 0.000 0.000 0.000 0.000 0.000 ***ELEMENT_BEAM** $# eid pid n1 n2 n3 rt1 rr1 rt2 rr2 local 1 1 400 401 0 0 0 0 0 2 ***DEFINE_COORDINATE_SYSTEM** 4, 488.999, -28.99994, 414.4192, -105.6686, -180.7164, 227.1846 670.4222, 17.28623, -199.297 ***MAT_LINEAR_ELASTIC_DISCRETE_BEAM_TITLE** Spring Stiffnesses $# mid ro tkr tks tkt rkr rks rkt 1 0.001000 0.000 1.0000E+6 1.0000E+6 0.000 0.000 0.000 $# tdr tds tdt rdr rds rdt 0.000 0.000 0.000 0.000 0.000 0.000 $# for fos fot mor mos mot 0.000 0.000 0.000 0.000 0.000 0.000

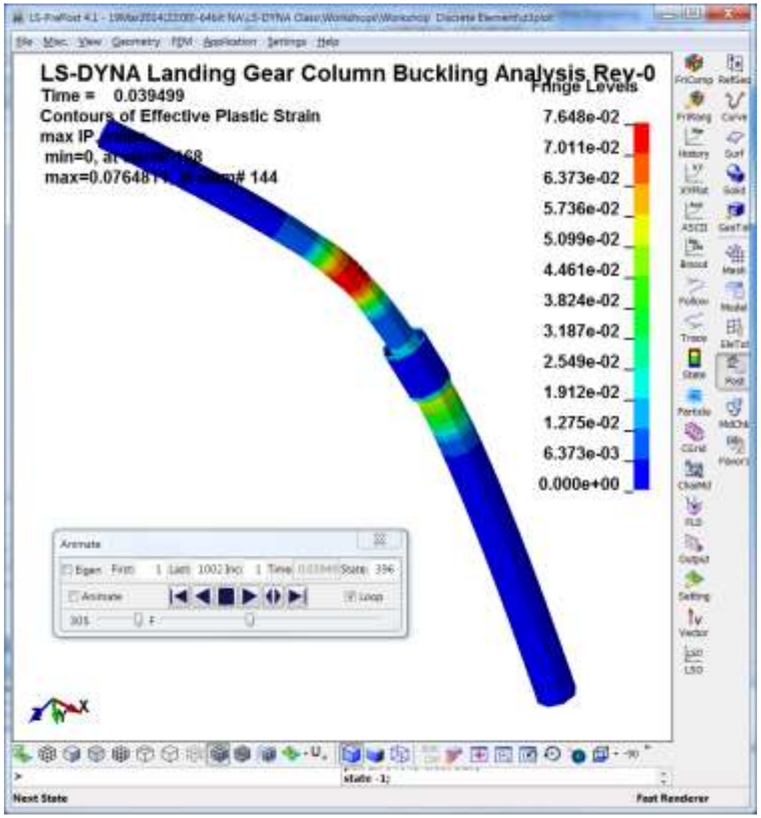
Some things to note is that a third node is not defined within the *ELEMENT_BEAM card since the orientation of the element is handled by the CID definition. One will note that VOL and INER are both given values of 1 for simplicity since the mass of the element is then controlled by just the mass density on the material card. And, don't forget that a *PART card is also required to tie together the *SECTION and *MAT cards. In this example the coordinate system is fixed and does not rotate with the system.

5.3.1 Workshop Discrete Elements (Non-Linear Buckling Analysis)

Femap Model



LS-DYNA Non-Linear Buckling Results



6. LS-PREPOST

6.1 Workshop IV: LS-PrePost | Workshop 7 & 8

Introduction to LS-PrePost

Reference and tutorial materials are provided at the www.LSTC.com site.

Model manipulation is by Shift Key and the left, right and middle mouse button.

Class Referenced Note's Section

The detailed usage of LS-PrePost in its own rights is a two day class (see Class Reference Notes / LS-PrePost Introduction Manual / **LS-PrePost Introduction.pdf**).

Workshop Goal: To Get Comfortable with LSPP Post-Processing

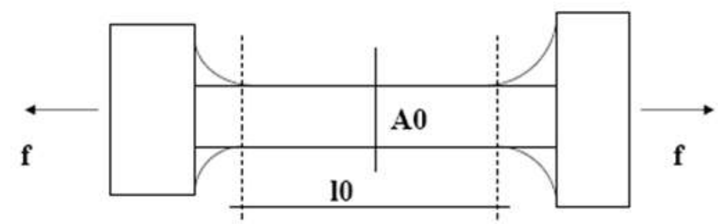
- Open PDF file and review contents. Take 10 minutes.
- Walk through PDF until arriving at Workshop 7.
- Perform exercises as listed as Bookmarks.

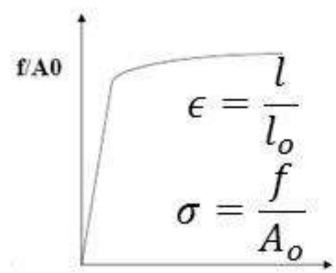
7. MATERIAL MODELING

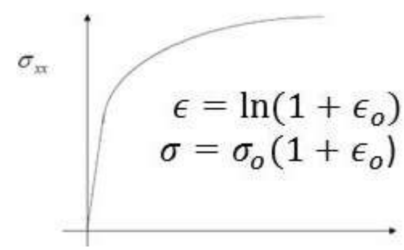
7.1 Part 1. Metals

7.1.1 Engineering Stress-Strain vs True Stress-Strain

The constitutive large-strain modeling of all materials is based on the true stress-strain response of the material. Starting with a simple tensile test, the engineering stress strain is just the force over the original cross-sectional area of the coupon. The true stress-strain response accounts for the necking of the cross-section and can be elastically stated as shown on the graph on the right.

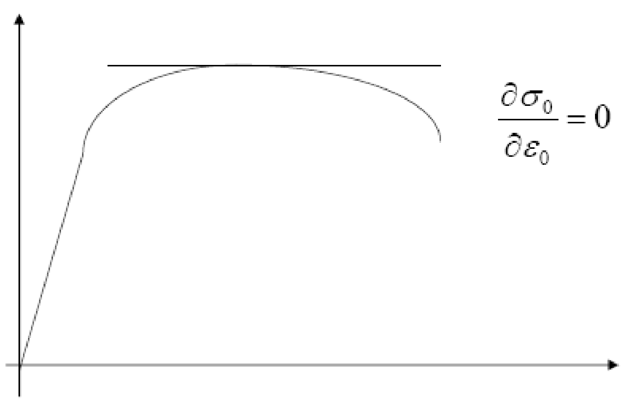


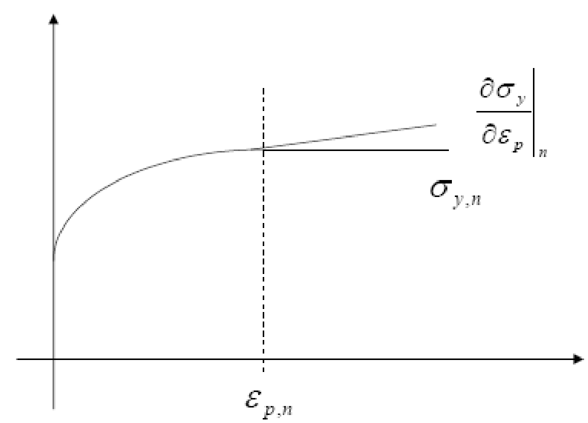




This method has its limitations and once the coupon starts to neck locally, this approach is no longer valid and an iterative approach must be used to calculate the true stress-strain response.

In many simulations, it is not exactly critical to have the necking response accurately characterized since once the material starts to neck we only have the end point data available and hence we draw a straight line from the initiation of necking to failure.





7.1.2 Review of Material Models Available in LS-DYNA

The broad array of material models in LS-DYNA can be over-whelming. In the material manual they are listed in numerical order based on their insertion into the code. Hence, the elastic material model (*MAT_001) was the first material model developed. These earlier models are well validated since they have been used extensively over the years. Later material models, e.g., *MAT_181 Simplified Rubber, was developed in early 2000, and although a somewhat recent development, it has seen wide-spread usage due to its advanced formulation and robustness. For elastomers, it is my default choice.

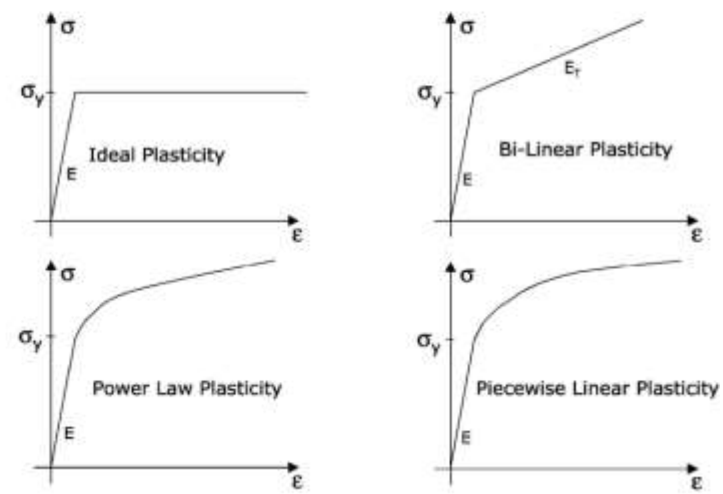
Although many material models exist for metals, one of the most robust models is *MAT_024 or *MAT_PIECEWISE_LINEAR_PLASTICITY. This material model is the standard workhorse and is the recommended starting point for elastic-plastic simulation of metals and general plastics since it can also handle viscoelastic behavior (i.e., strain-rate dependency).

Note: It is recommended practice to build a pilot model using the material law of choice and replicate the material's experimental data.

Major Categories

(LS-DYNA_manual_Vol_II_R7.0.pdf)

- Elastic
- Elastic-Plastic
- Rigid
- Orthotropic/Anisotropic
- Hyperelastic (Rubber)
- Foams
- Composites
- Viscoelastic
- Heart/Lung/Tissue
- Acoustic material
- Fabric
- Concrete/Soil
- High Explosives
- Laminated Glass
- User-defined



7.1.3 Material Failure and Experimental Correlation

Material Failure

Simulation of material failure is a broad research avenue. Prediction of brittle material failure, as shown by the glass shatter patterns on the right, can be extremely mesh sensitive. For ductile materials, the failure prediction is more robust since ferrous and non-ferrous materials often have high-energy absorption characteristics prior to failure. That is to say, metallic materials generally tend to tear while brittle materials tend to snap.

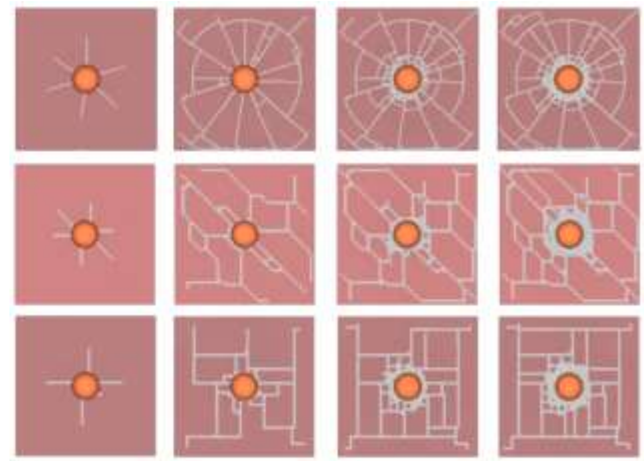
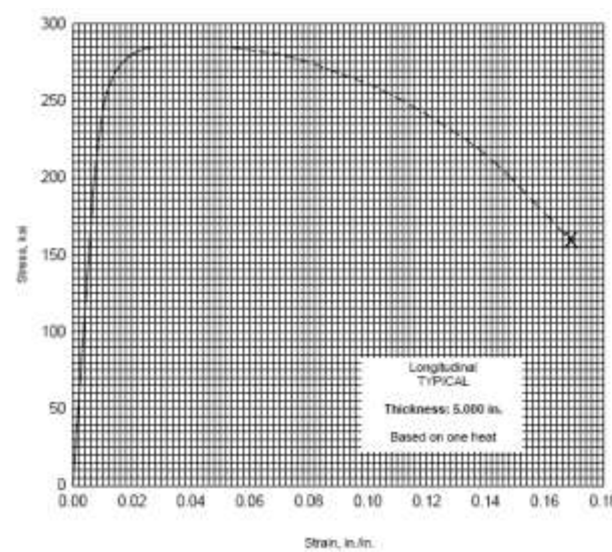


Image from article: A finite element model for impact simulation with laminated glass, Timmel, Kolling, Osterrieder, DuBois, IJIE, 2006.

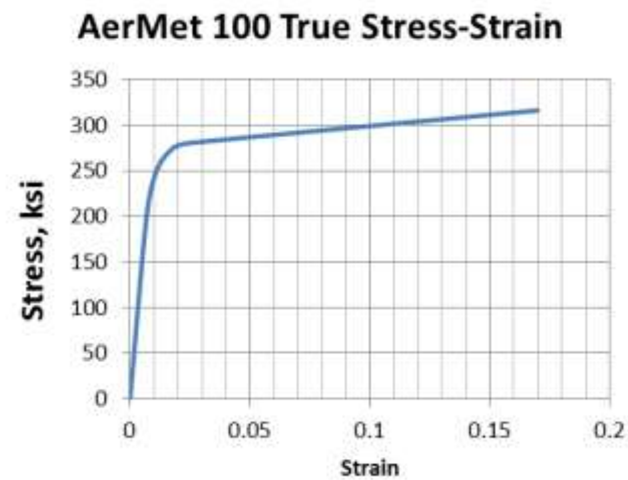
Experimental Correlation

The development of a material model often starts with stress-strain data from a standard mechanical test. This data can then be converted into true stress-strain and a very simplified approach used to extend this curve to failure.

In the high-strength steel shown on the right (see Class Reference Notes; MMPDS, AerMet 100 Steel Bar), the stress-strain curve drops as necking occurs. The presented data is all that is needed to generate a useful true stress-strain curve for the material.

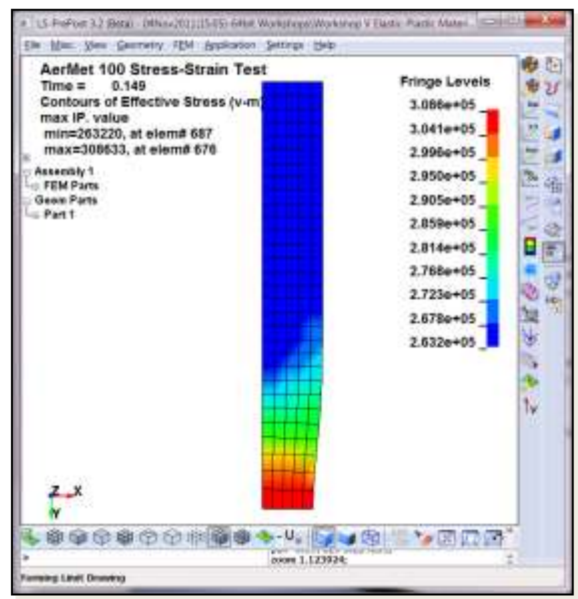


7.2 Workshop V: Elastic-Plastic Material Failure



Objective: To approximately model the material deformation and failure of the AerMet 100 steel.

A simplified approach is presented using the piecewise linear plasticity approach available in *MAT_024. Data is estimated from the presented curve and corrected to true stress-strain up to the point of necking. After necking, a straight line is drawn to the ultimate failure point. The final processed, true stress-strain curve is shown on the right. The data spreadsheet can be found within the Workshop file folder.



Goal: Get Comfortable With Constitutive Modeling

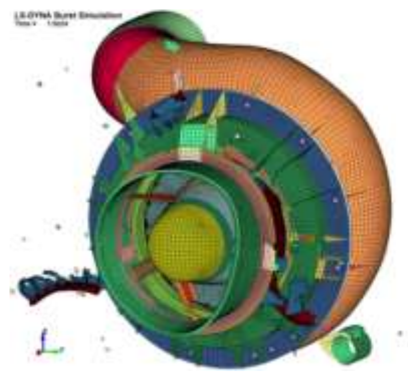
Run initial model (Elastic-Plastic Failure Start.dyn) and investigate the current material law. Plot von Mises and Plastic Strain (History Plots).

Change material law to Piecewise Linear Plasticity using Excel Spreadsheet or PDF file in Workshop Directory and rerun model. *Note correct format for entering plastic strain.*

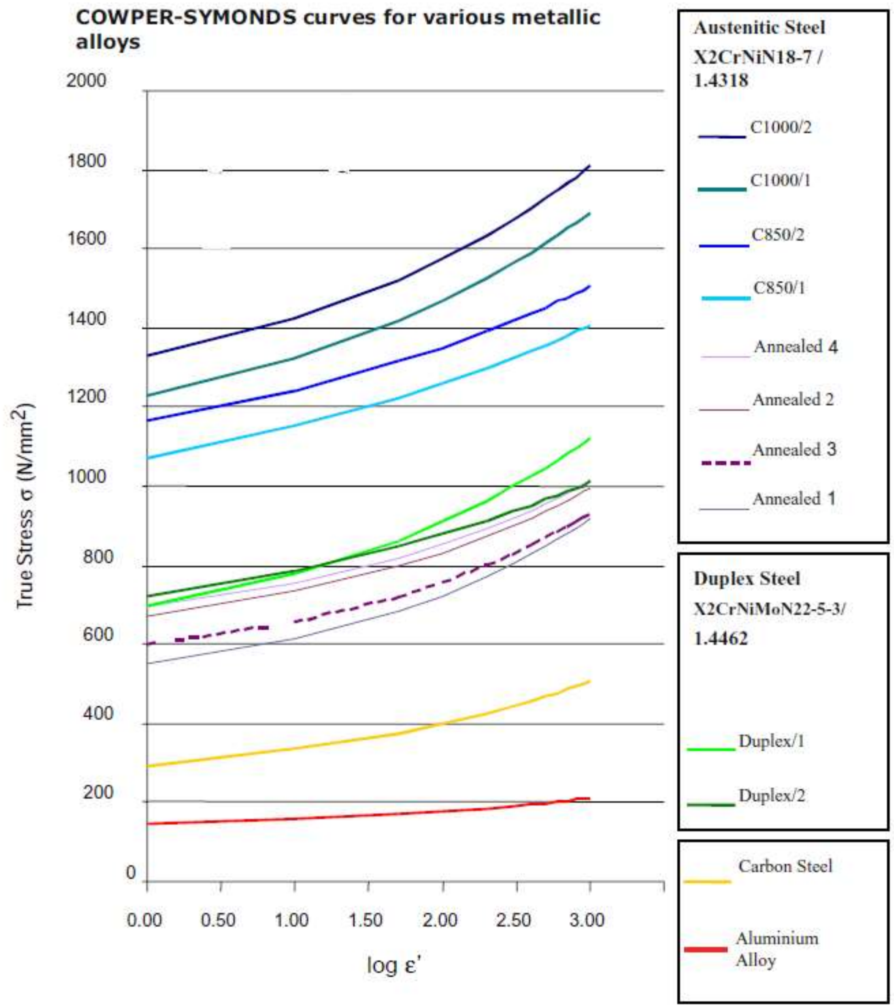
Compare results, check hourglass energy balance and validate material law model.

7.3 Strain Rate Sensitivity of Metals

- This mechanical metallurgical behavior is due to the movement of dislocations within the crystalline structure. Dislocations move up to the speed of sound within the metal.
- The graph on the right provides a rough-order-of-magnitude idea of how strain rate affects the true stress in steels and aluminum. The strain rate is in seconds.
- An example of strain rate effects might be that for rotor burst. The rotor spins at 55,000 rpm. The tip velocity of the turbine blade at burst is 575 m/s. As the blade impacts the containment ring, one could expect to see significant strain rate effects.



- For example, given a 10 mm tall, carbon steel cylinder that is compressed 10% at 575 m/s. This would give a strain rate of 575 s^{-1} or log ε' = 2.8. From the chart on the right, one could expect an increase in the yield stress of >50%.
- Strain-rate effects can be evaluated by numerical testing (i.e., exploratory work looking at maximums).



7.4 Part 2: Elastomers and Foams

7.4.1 Modeling Elastomers vs Foams (Viscoplasticity)

The Material section of the LS-DYNA keyword manual provides a wealth of practical information on the modeling of elastomers (soft plastics and rubbers) and foams. Plastic materials have very unusual engineering stress-strain curve due to necking. However, once corrected for true stress-strain, the curve looks very reasonable.

The curve on the right is for a uniaxial test performed on rubber. Experimental uniaxial data can be directly entered into the material card. For the modeling of rubbers, this is a very common approach.

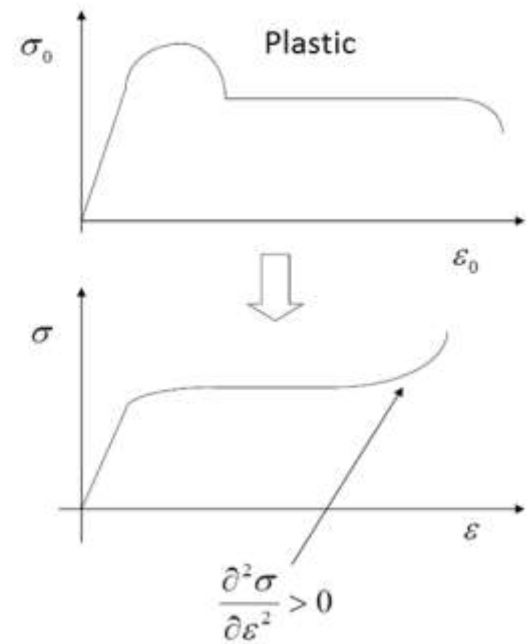
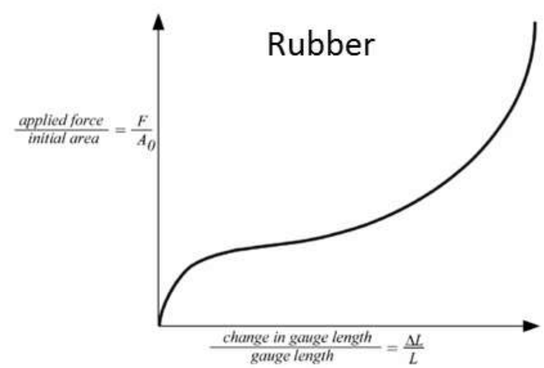
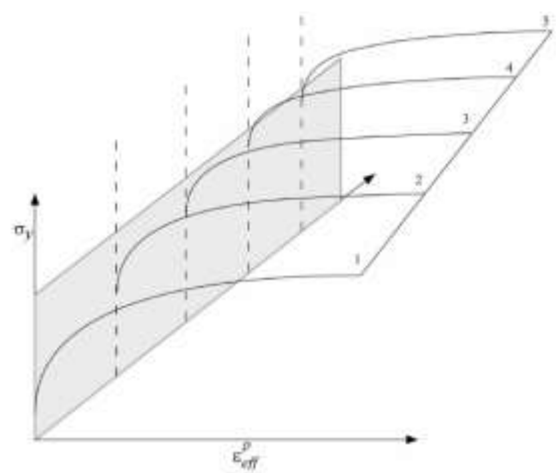


Figure 27.1. Uniaxial specimen for experimental data.



Viscoplasticity

This concept is fundamental in the modeling of plastic, rubber or foam materials. These materials deform via the stretching of their long-chain hydrocarbon network. As such, they are very sensitive to strain rate effects. As the strain rate increases, their complete stress/strain curve will shift upward.



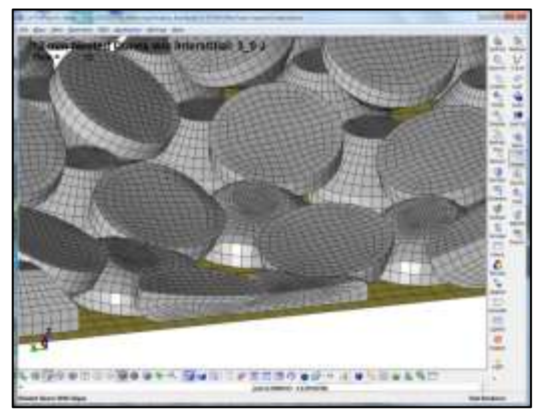
Analyst's Note: Whenever a new material model is simulated, a virtual test coupon analysis should be done and the results compared to the mechanical test data.

7.5 Material Models for Modeling Foams

The following table presents the currently available foam material models in use within LS-DYNA. One recommended foam model is that of *MAT_083 or *MAT_FU_CHANG_FOAM. Foams are perhaps the most challenging material to model due to their extreme nonlinear behavior upon loading and unloading plus their tendency to become crushed or damaged during loading, and then upon unloading, present a completely different stress/strain response.

Foam Material Models Available in LS-DYNA			
5	* MAT_SOIL_AND_FOAM	**142**	*MAT_TRANSVERSELY_ANISOTROPIC_CRUSHABLE_FOAM
14	* MAT_SOIL_AND_FOAM_FAILURE	**144**	*MAT_PITZER_CRUSHABLE_FOAM
26	*MAT_HONEYCOMB	**154**	*MAT_DESHPANDE_FLECK_FOAM
126	*MAT_MODIFIED_HONEYCOMB	**163**	*MAT_MODIFIED_CRUSHABLE_FOAM
53	*MAT_CLOSED_CELL_FOAM	**177**	*MAT_HILL_FOAM
57	*MAT_LOW_DENSITY_FOAM	**178**	*MAT_VISCOELASTIC_HILL_FOAM
62	* MAT_VISCOUS_FOAM	**179**	*MAT_LOW_DENSIlY_SYNTHETIC_FOAM
63	*MAT_CRUSHABLE_FOAM	**180**	*MAT_LOW_DENSIlY_SYNTHETIC_FOAM_ORTHO
73	*MAT_LOW_DENSIlY_VISCOUS_FOAM		
75	*MAT_BILKHU/DUBOIS_FOAM		
83	*MAT_FU_CHANG_FOAM		

Analyst's Note: Since foams are modeled using solid elements, it is not uncommon to have numerical problems as the foam becomes highly compressed and crushed since the elements used to idealized this behavior, likewise become highly distorted or crushed. Typically, the workarounds are to use highly structured meshes with large element sizes. Another technique that is gaining utility is to use SPH (Smooth Particle Hydrodynamics) to model the foam material. This "mesh-free" technique will be covered at the end of these course notes.

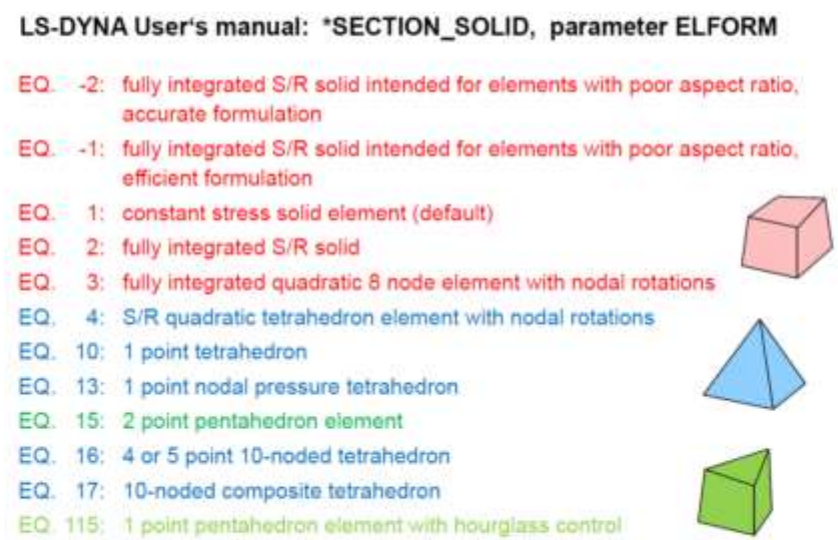


7.6 *Workshop VI:* Modeling an Elastomer Ball with Hex and Tet Elements

Solid Element Meshing for Soft Materials: Hexahedral versus Tetrahedral

Whenever possible, a hex mesh should be used for the modeling of soft materials. The recommended element formulation is ELFORM=1 (hex) or =13 (tet). Both are one-point Guassian integration formulations and can handle large-deformations without element aspect failure problems (not negative volume). Another recommendation is to use ELFORM=-1 for non-perfect hex's.

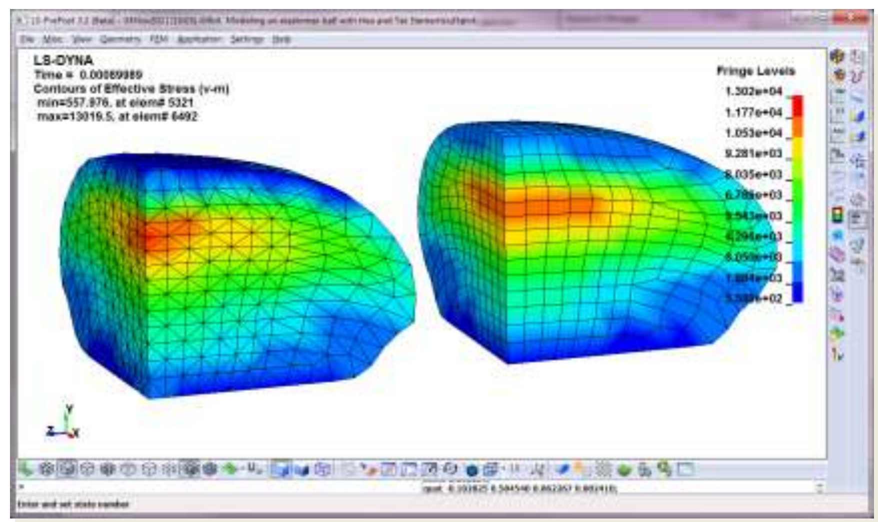
Please take a read within: Review of Solid Element Formulations Erhart.pdf (Class Reference Notes / Solid Elements) and it has an excellent section on Hourglass Control.



***CONTROL_SOLID |ESORT=1 (?)**

Element Quality and Negative Volumes

Large-deformation behavior in soft materials is highly sensitive to mesh characteristics. In an ideal situation, brick elements are always preferred due to their regularity of formation (shape and distribution). Even with a high-quality mesh, under high compression loading, brick elements can generate negative volumes. The best solution is to refine the load application and/or improve the mesh quality. Prior to embracing any one path, it is recommended to consult references (i.e., www.DYNASupport.com and Keyword Manual).



7.7 Equation of State (EOS) Material Modeling

An equation of state is required for materials that undergo significant deformation (can be very large plastic deformation or a compressible fluid). The Cauchy stress tensor can be separated into a hydrostatic stress tensor ($p\delta_{ij}$) and a deviatoric stress tensor ($\sigma`_{ij}$):

$$\sigma_{ij} = \sigma`_{ij} + p\delta_{ij}$$

The deviatoric stress is calculated by the material model constitutive law. The pressure term, p, must then come from an equation of state (EOS). The EOS provides a relationship between pressure and the volume (can also be a relation of temperature and/or energy). Depending on the compressibility of the material, different types of EOS's are possible. A very popular EOS is the Gruneisen equation of state. The full version of this EOS (compression) is:

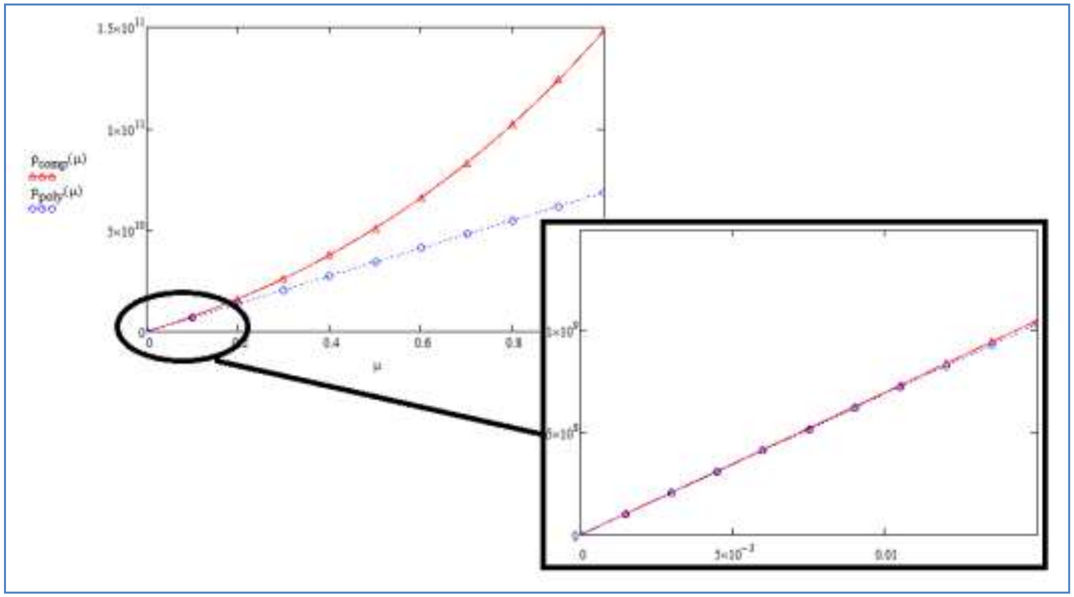
$$p = \frac{\rho_0\, C^2\, \mu \left[1 + \left(1 - \frac{\gamma_0}{2}\right)\mu - \frac{a}{2}\mu^2\right]}{\left[1 - (S_1 - 1)\mu - S_2\left(\frac{\mu^2}{1+\mu}\right) - S_3\left(\frac{\mu^3}{(1+\mu)^2}\right)\right]^2} + (\gamma_0 + a\,\mu)E$$

The equation appears rather complicated at first glance, if we focus on a weakly compressible fluid (many engineering fluids can be considered this way), the equation of state can be reduced to:

$$p = \rho_0\, C^2\, \mu$$

Where ρ_0 is the initial reference density for the material, C is the speed of sound in the material and $\mu = \frac{\rho}{\rho_0} - 1$. All the other parameters are curve found by curve fitting to a set of compression experiments. These parameters are typically only needed when the pressure loading is very high as in shock waves. In the figure shown on the next page, we can see a comparison between an EOS specified with only the speed of sound (linear dependency of pressure on volume) and an EOS to give a cubic dependency of pressure on volume.

In the subsequent workshops, the units are $p\ [Pa]$, $\rho_0\ [kg/m^3]$ (specified as RO on *MAT_NULL) ,and $C\ [m/s]$ (specified as C0 on *EOS_GRUNEISEN) and μ is a dimensionless parameter.



Comparison between weakly compressible and compressible

Fluids in LS-DYNA Explicit need to be described by a constitutive material law (such as *MAT_NULL for example) and an appropriate EOS. The reason is that solving the set of Euler equations (or full Navier-Stokes in the presence of viscocity) with a strictly explicit time integration scheme requires an equation of state to directly determine the pressure at each node point. A truly incompressible algorithm requires solving a Poisson equation (elliptical partial differential equation) to ensure that the flow is divergence free. The Poisson equation can only be solved iteratively or using Implicit time integration.

All this truly means is that in LS-DYNA SPH Explicit, a fluid that is commonly considered incompressible can be treated as weakly compressible with a simple EOS by defining only two parameters; initial density and the speed of sound in the material. Throughout the workshops, we use *MAT_NULL, but other material models such as *MAT_JOHNSON_COOK, *Elastic_Plastic_Hydrodynamic, etc. can be used with an EOS to describe various engineering materials.

7.7.1 Modeling Water with *EOS_GRUNEISEN and *MAT_NULL

As basic as this may sound, it is not that obvious. The technique for modeling water is well described in Class Reference Notes / Aerospace Working Group / Aerospace_MGD_v12-1.pdf at page 66, Section 3.9.2 Water. The entry is as simple as the graphic shown on the right.

The only real difficulty is ensure that one gets the units handled correctly.

*Analyst's Note: Make sure you remember to set the *PART card to use the *EOS law.*

3.9.2 Water

In LS-DYNA models, *MAT_NULL and *EOS_GRUNEISEN are commonly used to represent water and other liquids. Some commonly used input constants for water at 20 deg C follow.

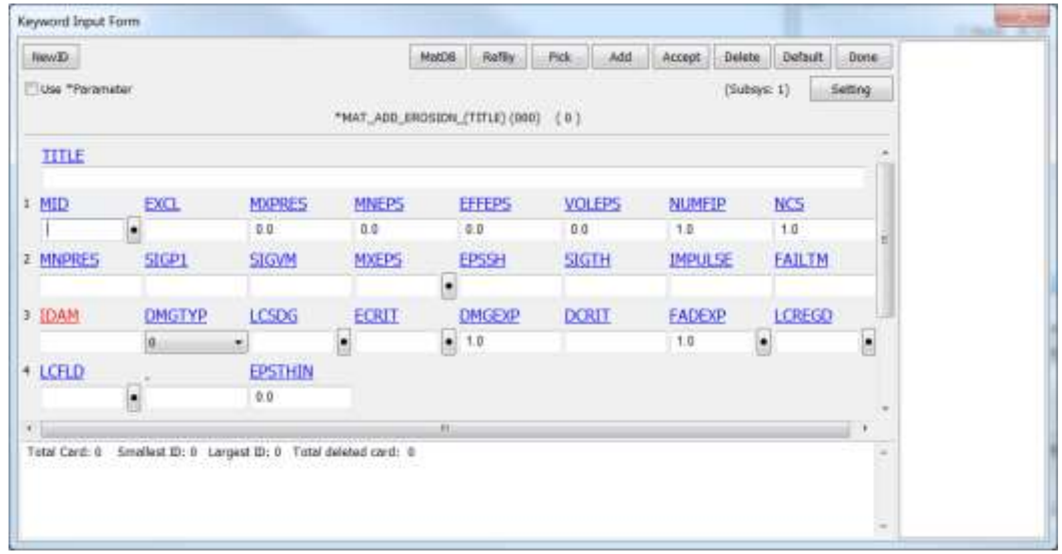
*MAT_NULL:
Mass density, RO = 1.e-6 kg/mm^3
1.0 g/cm^3
1000. kg/m^3
1.e-9 tonnes/mm^3
9.37e-5 lbf-s^2/in^4
Dynamic viscosity, MU = 1.0e-3 N-s/m^2 (often taken as 0.0)
Pressure cutoff, PC = -100 Pa (often taken as zero)
All other parameters in *MAT_NULL should be set to zero or left blank.

*EOS_GRUNEISEN:
Nominal sound speed C = 1500 mm/ms
0.15 cm/microsec
1500 m/s
1500e3 mm/s
59055 in/s
E0 = 0
V0 = 1.0 (unitless)

7.8 Material Failure Simulation

7.8.1 Basic Methods of Modeling Failure: Material versus Bond Failure

Standard failure in the modeling of materials is by specifying some sort of material based failure criterion. My favorite approach is to use the *MAT_ADD_EROSION card to specify the exact failure criteria that is needed. For metals, one approach is to set the EFFEPS (maximum effective strain at failure) to 3x MXEPS (maximum principal strain at failure). This ensures that the material does not prematurely fails under compressive plastic deformation but still remains true to the mechanical test data.



Exercise: Open LS-DYNA_manual_Vol_II_R7.0.pdf (see Class Reference Notes / Keyword Manuals) and read the *MAT_ADD_EROSION card section.

Another way of modeling failure is by *CONSTRAINED_TIED_NODES_FAILURE. With this formulation, bond failure can be modeled in a direct and simple manner by setting the plastic strain required to pull apart the nodes. Of course, this plastic strain is taken from that elements integration point. The setup for this failure mechanism is to take a clean mesh and let LSPP create the tied connections. This is done by breaking apart the elements and then tying together the adjacent nodes. Upon failure, the elements fly apart but are not deleted. An example of this concept is can be found at www.dynaexamples.com / Intro by J. Reid / Sphere Plate. Given all that, I prefer the simplicity of *MAT_ADD_EROSION.

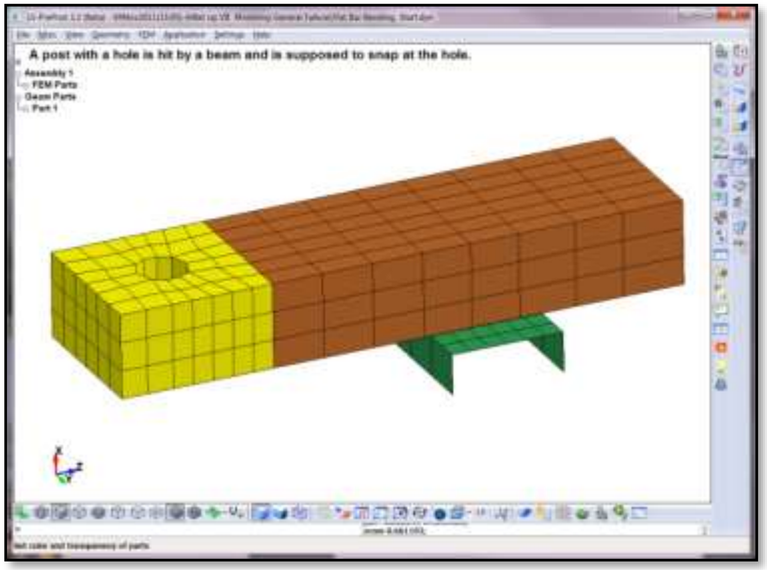
7.9 Workshop VII: Modeling General Failure

Objective: Modeling of material failure is not as complex as one might think if a reasonable expectation is taken from the outset. It should be noted that: "All models are wrong, but some models are useful" and hence when trying to replicate failure in a structure one should strive for upmost simplicity prior to adding complexity. This concept segues into another saying: "elegant simplicity is deceptively difficult to achieve". In the following workshop, a basic failure mechanism within the *MAT_24 card is improved upon by using the *MAT_ADD_EROSION approach.

Model Introduction: A flat bar is held at its end (near the hole) and the rail is given an initial velocity to impact against the bar and move it upward.9

Workshop Tasks

- Open Flat Bar Bending Start.dyn and inspect material law used for the default simulation.
- Understand unit system since it is not completely uncommon in the world of LS-DYNA.
- Run Model and inspect failure mode. Note: Rigid material modeling technique.
- Delete failure criterion from material card.
- Open LS-DYNA Material Manual and read-up on the *MAT_ADD_EROSION card. Then, within LSPP add a tensile strain failure criterion (MXEPS) = 0.25 to the existing material law.
- Rerun model

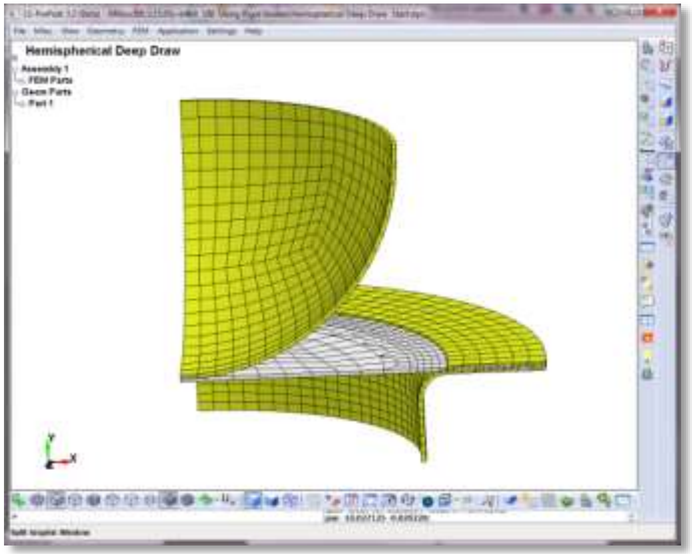


Student Bonus: Open up LSPP with your final model file and increase the initial velocity of the bar and note the results.

7.10 Modeling Rigid Bodies

7.10.1 Rigid Materials (*MAT_020 or *MAT_RIGID)

This is one of the most powerful modeling techniques within LS-DYNA. By setting bodies (i.e., parts) to use *MAT_RIGID, where deformation and stresses are not of interest, significant CPU savings can be realized. In the background, LS-DYNA retains the surface mesh of the part for contact behavior and calculates an inertia matrix to simulate the dynamic behavior of the body. What is useful with this approach is that the body still retains its inertia and physical characteristics as it interacts with other bodies within the simulation but at a fraction of the numerical cost of dragging around a fully deformable body. For example, the model on the right is of a deep drawing operation and only the plate is deformable.



*Note: Rigid bodies cannot have constraints applied to them. To constrain a rigid body, the CMO flag is set within the *MAT_RIGID card.*

For a very nicely done reference, please see Class Reference Notes / Rigid Bodies / LS-DYNA Intro Class Chapter 9 Rigid Bodies.pdf.

Other Common Usage: Rigid Links and Joints

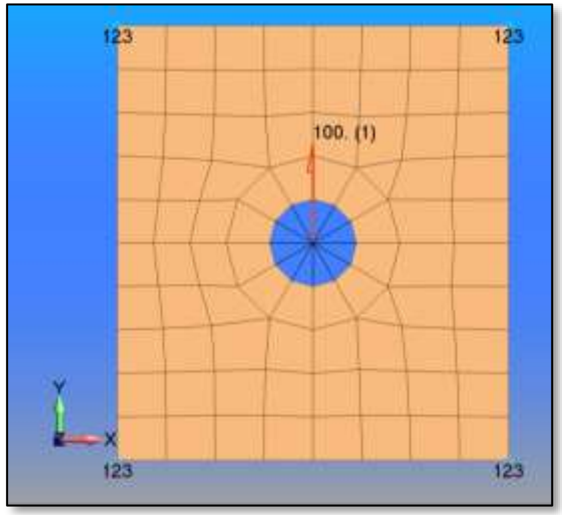
Nastran multi-point-constraints (MPC) equations, of which their two most common flavors are the RBE2 and RBE3 elements, are translated into two different LS-DYNA formulations. Our discussion will just focus on the RBE2 translation. The RBE2 is translated into a rigid body where the nodes are placed into a group and then constrained rigidly per the number of dependent DOF's specified in the RBE2 element. The inertia or mass properties for this nodal rigid body are obtained from the elements attached to the nodes of the rigid body. Although this may sound a bit odd coming from the implicit world, in explicit mechanics everything needs a bit of mass to enable its calculation. Hence, for the rigid link to behave correctly, it borrows mass from its attached elements and two node CNRB's should be avoided.

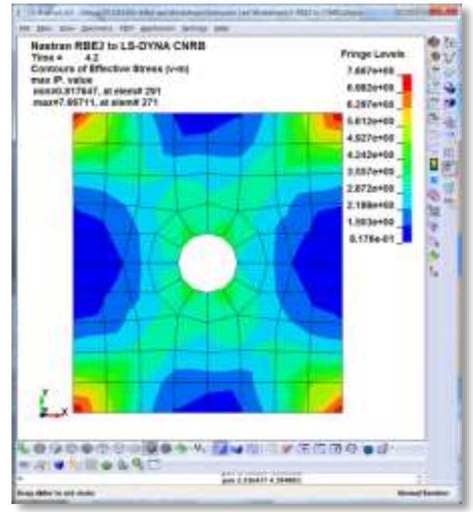
The LS-DYNA card that is used is *CONSTRAINED_NODAL_RIGID_BODY (CNRB) with the CMO card specifying what DOF's are to be released.

7.10.2 Instructor Led Workshop V: RBE2 to CNRB

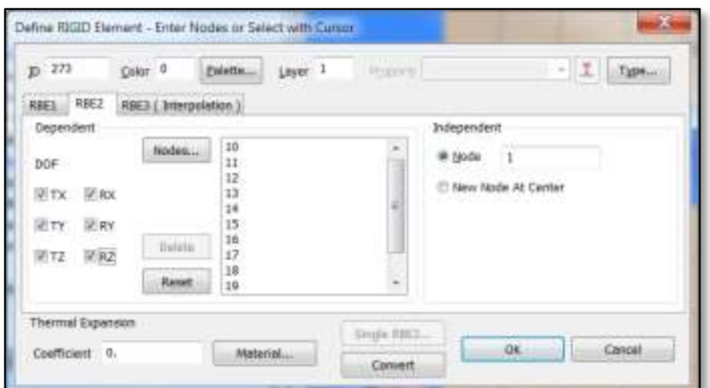
A Y-direction load is applied to the independent node of a RBE2 element around a hole in a plate model. The corners of the plate model are pinned. This example will be used to demonstrate how a RBE2 element is translated into LS-DYNA and its behavior. The process is self-discovery and no walk-through video is provided. The instructor will walk through the steps and then the student is left to explore.

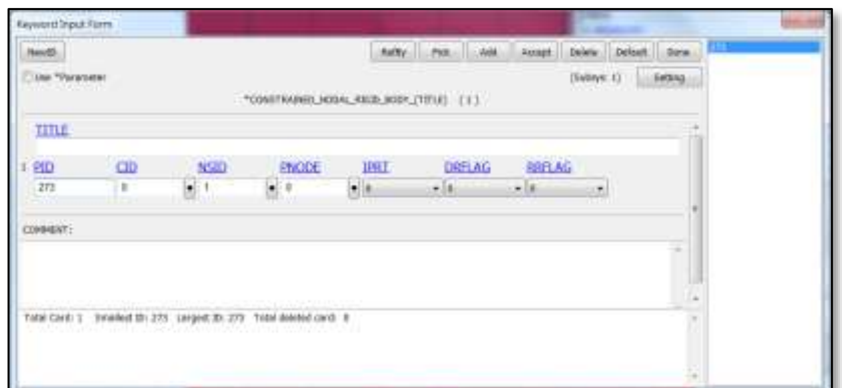
Goals: Open Femap model and inspect Rigid Link. Export to LS-DYNA model and review card within LSPP. Run model. Then release CNRB in Y-Direction and re-run model.





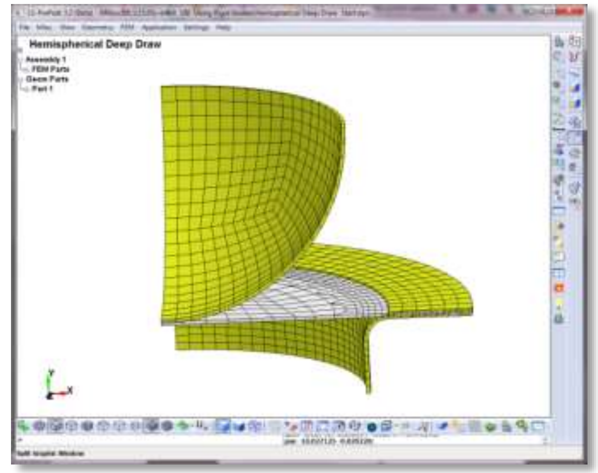
Stress results from the default run. Given the model setup, what stress units are we seeing?





Check out the DRFLAG and RRFLAGS.

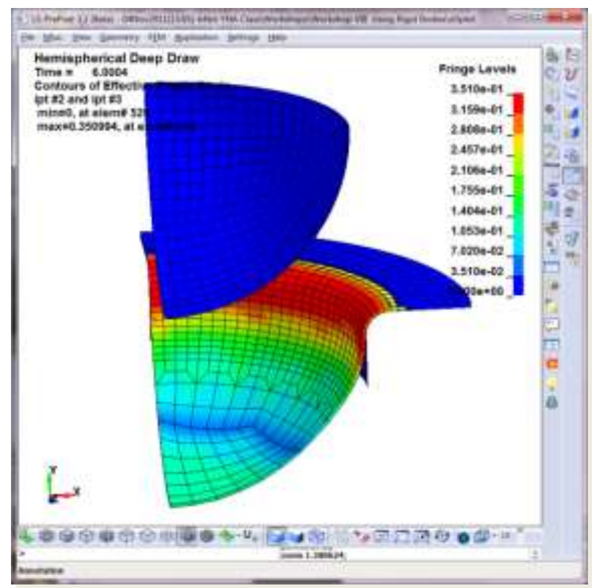
7.10.3 Workshop VIII: Using Rigid Bodies



The implementation of rigid bodies open doors to making your models run faster and allows you to focus on what is important and avoid getting distracted. For complex system level models, the use of rigid bodies is invaluable and if desired can always be switched later on to deformable.

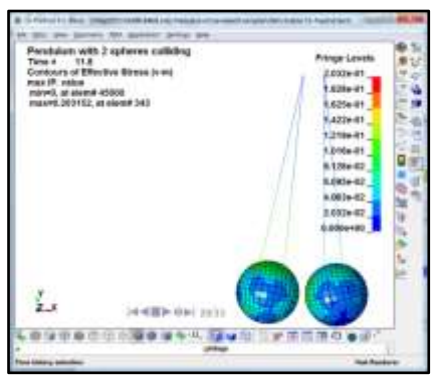
What You Will Learn

- How Rigid Bodies work within a simulation and how easy it is to switch to deformable to gather deformation and stress information.
- Defining a fixed movement (i.e., punch movement) as a load case.
- Getting more familiar with the LSPP Interface for making model updates.



Student Led Workshop I: Switching Rigid Bodies to Deformable Bodies

The pendulum model starts out rigid and then is switched to deformable at the last moment prior to impact. See Class Reference Notes / LSTC General Examples Manual and Examples / EXAMPLES / DEFORMABLE_TO_RIGID.

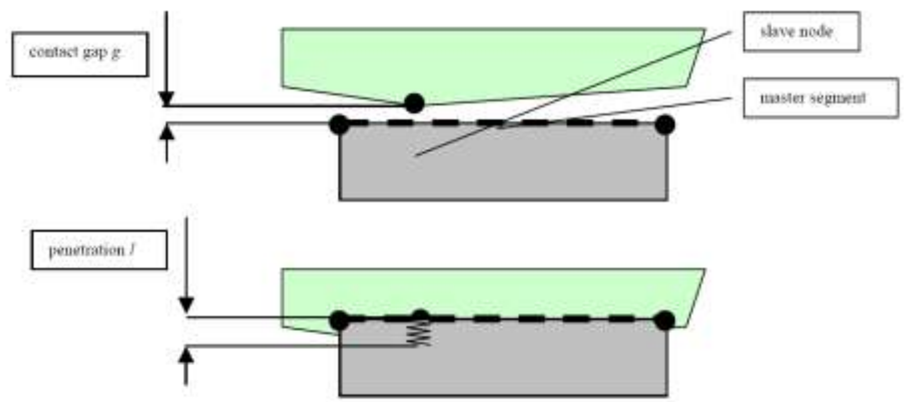


This example provides insight in how to do Restarts (See Class Reference Notes / Keyword Manuals / LS-DYNA_manual_Vol_I_R7.0.pdf / Section RESTART INPUT DATA). Two files are provided that covers the complete run in the directory noted above. Run the regular model and then perform a restart. A small movie file is provided to show how the process works (see Workshops / Student Led Workshops.

8. CONTACT

8.1 Definition of Contact Types

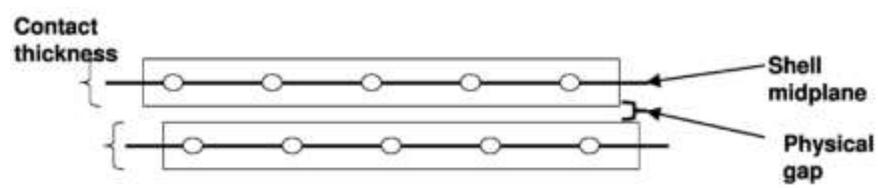
LS-DYNA was developed specifically to solve contact problems (see Class Reference Notes / History of LS-DYNA). Contact behavior is enforced by three methods: (i) Penalty-Based using finite springs (see graphic); (ii) Constraint-Based where no penetration is allowed and cannot be used with rigid bodies; and (iii) Tied Contact (discussed within its own section). The most commonly used approach is that of the penalty method.



Contact can be effortlessly implemented or it can be bewitching in complexity. A reasonable treatment of contact is a multi-day course in itself. To start the learning process see Class Reference Notes / Contact User's Guide / Contact User's Guide.pdf.

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

8.2 General Contact Types

Although there are numerous contact types given in the Keyword Manual, these workhorse formulations are recommended:

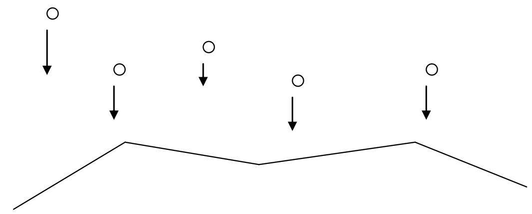
1. *CONTACT_AUTOMATIC_GENERAL
2. *CONTACT_AUTOMATIC_SINGLE_SURFACE
3. *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE.

The first formulation is the "kitchen sink" and is computationally expensive but is very robust and will enforce contact between beam elements and other components. It is a *single-surface formulation* and only the slave side is defined (it assumes that everything contacts everything else).

The second formulation is a less CPU expensive form of GENERAL (e.g., edge-to-edge contact is not checked). While the last formulation is general purpose and numerically efficient.

Classical Contact

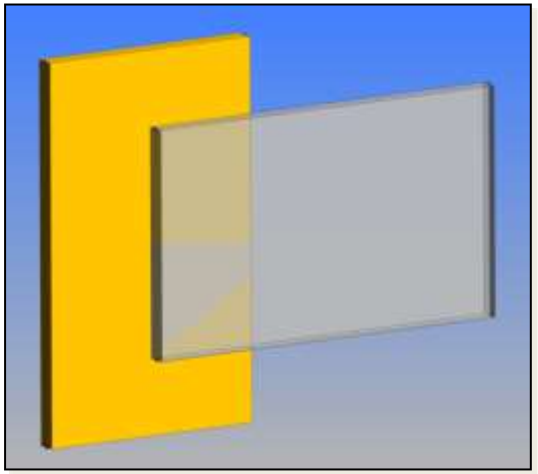
"Keeping Nodes on the Right Side"



8.2.1 Additional Options: SOFT=2 "The Default"

The standard contact search routine is based on nodes looking for faces (i.e., segments). Sometimes, contact might be missed for a few steps, and when finally engaged a large restoring force is required to separate the interfaces or contact might just not occur. Additionally, the standard penalty approach calculates the spring stiffness based on global material stiffnesses. This can lead to contact instability in soft materials.

To counter these problems, the contact option SOFT=2 is recommended. This switches the contact search routine to segment-to-segment and locally calculates the stiffness for the penalty approach. It should be considered the "default".

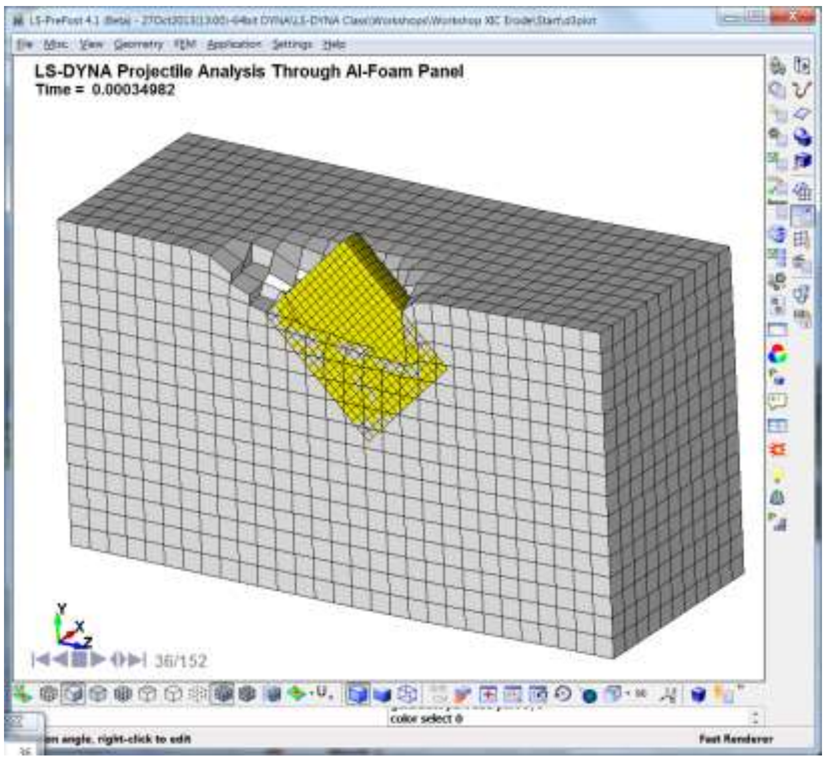


8.2.2 Contact when things ERODE

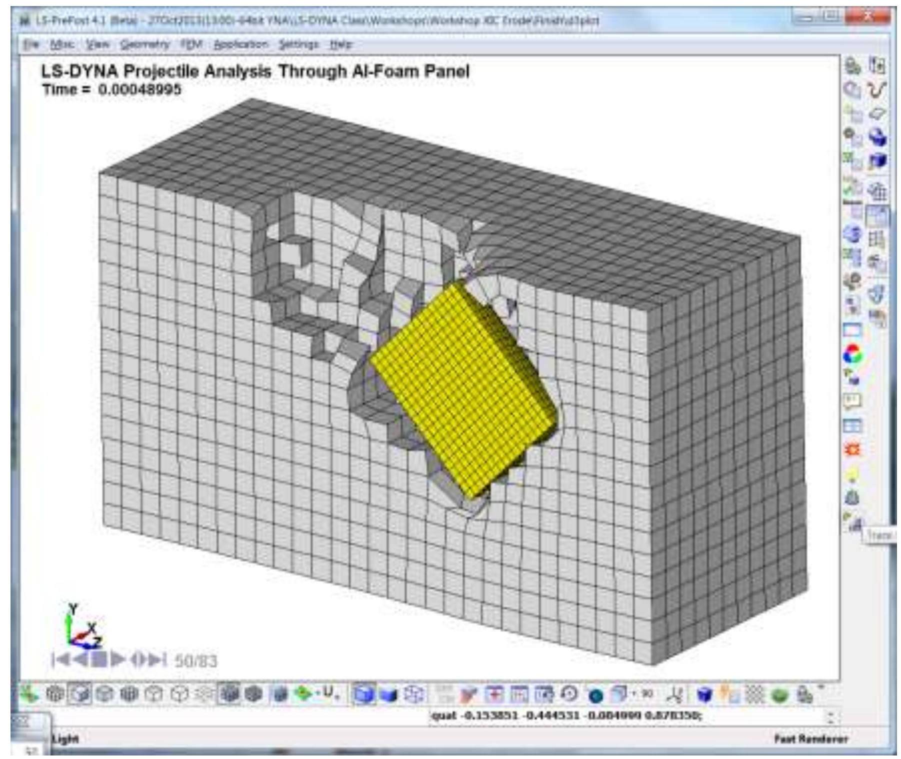
For numerical efficiency, the contact algorithm only looks at free edges and faces. If element erosion occurs (i.e., element failure), the standard contact algorithm is not prepared to look for contact on these newly generated faces. If one knows ahead of time, then the contact can be switched to _ERODING with EROSOP=1. The EROSOP option is required to allocate memory storage for the newly created element surfaces.

This option should ***not*** be evoked for most element erosion situations since it is not very numerically efficient. For example, plate models are typically just fine without using element erosion, but as shown below, it is critical.

_AUTOMATIC_SINGLE_SURFACE



_ERODING_SINGLE_SURFACE



8.3 Workshop IX: Understanding Basic Contact Mechanics

Often times, the challenge to modeling contact is not setting up the contact model but checking the results. In this workshop, the goal is to verify the contact behavior and plot the contact force for each tube. The contact behavior should make good engineering sense.

Workshop Tasks

- Run model (Pipe on Pipe Contact Start.dyn) and look at contact behavior.
- Investigate logical contact behavior and change to Automatic and try using SOFT=2 (FINISH I)
- Measure contact forces between the parts (*CONTACT_FORCE_TRANSDUCER) and set ASCII file parameters RCFORCE & RWFORCE = 1e-6 (FINISH II)
- Plot rigid wall impact force (RWFORCE) via PADD on XY Plot Dialog
- Lastly, if one has time, setup model to show contact pressure (see Student Bonus Option).

Keywords Discussed

*DATABASE

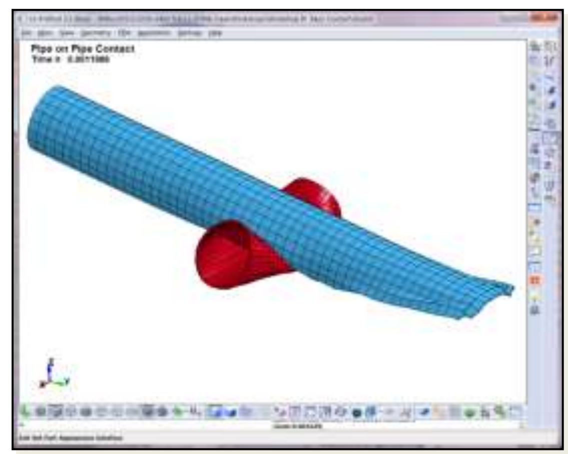
_ASCII (GLSTAT, RCFORCE, SLEOUT, RWFORC)

_BINARY_INTFOR

*RIGIDWALL_GEOMETRIC_FLAT

8.3.1 Student Bonus Option: Contouring Interface Pressures

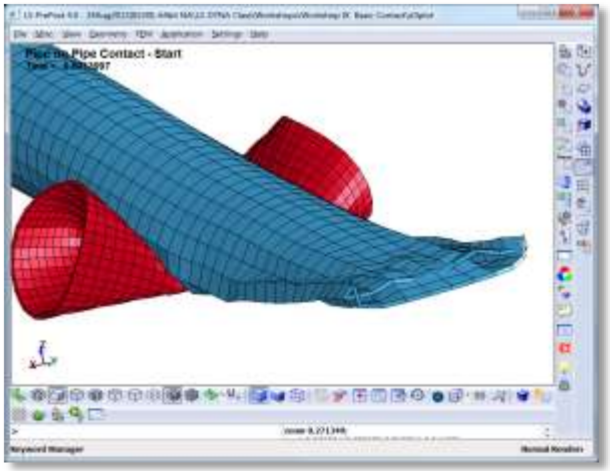
LS-DYNA doesn't automatically generate interface pressures developed during contact. To obtain this information, three items are required: (i) *DATABASE_BINARY_INTFOR DT={time interval} must be set; (ii) Print flag(s) on card 1 of *CONTACT_ must be set to SPR=1 and/or MPR=1; and (iii) and upon analysis (LS-DYNA Program Manager), one must use the Advanced tab and include the Interface Force s=(provide your own filename) option. This creates a separate binary file can then be read by LS-PrePost as a separate post-processed item (just load the filename specified above).



8.3.2 Workshop IX: Comparison of Results

START

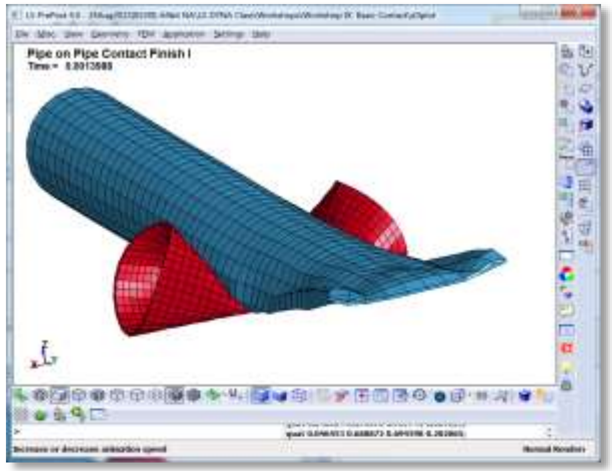
*CONTACT_SINGLE_SURFACE



Interpenetration between surfaces is noted which should never happen.

FINISH I

*CONTACT_AUTOMATIC_
SINGLE_SURFACE (SOFT=2)



With _AUTOMATIC and SOFT=2, the correct contact behavior is noted.

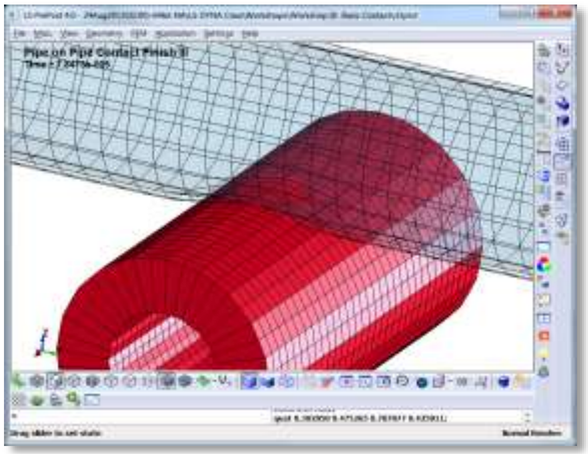
FINISH II

*CONTACT_FORCE_
TRANSDUCER w/ ASCII outputs
RCFORC & RWFORC 1E-6

Although _SINGLE_SURFACE is used, one can still obtain surface-to-surface contact forces. The rigid wall forces are also reported with the ASCII option RWFORC. Note that the rigid wall force is much greater than the tube-to-tube force since it represents the combined force of both tubes hitting the "ground floor".

FINISH III

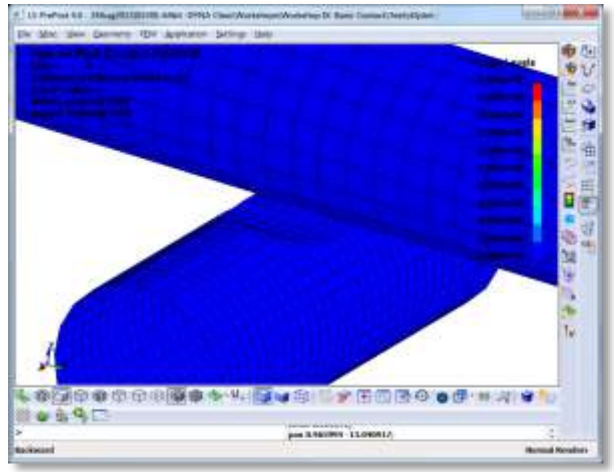
*SECTION_SHELL (Thickness=50)



With _SINGLE_SURFACE contact, initial interpenetrations are not instantly removed but automatically tracked. This is equivalent of setting the IGNORE=1 option.

FINISH IV

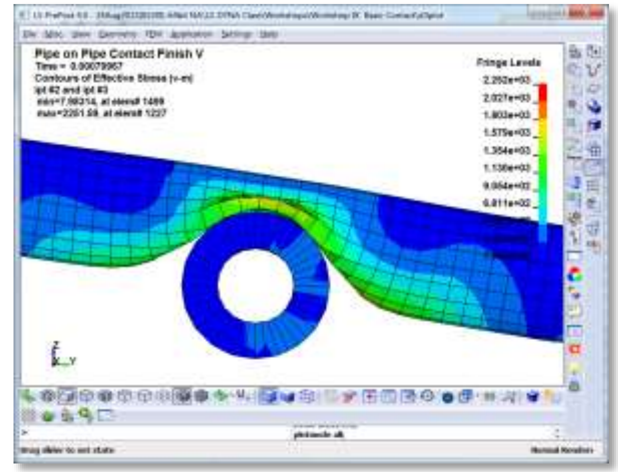
*CONTACT_AUTOMATIC_ SURFACE_TO_SURFACE



With _SURFACE_TO_SURFACE contact, Interpenetration is enforced and the slave surface nodes are immediately moved to accommodate the profile of the master surface. *Note: One should add back in SINGLE_SURFACE contact for the top tube with SOFT=2.*

FINISH V

*CONTACT_AUTOMATIC_ SURFACE_TO_SURFACE (SOFT=2)



With the Soft=2 option interpenetrations are gracefully handled (i.e., tracked). It is what makes the Soft=2 option quite useful in many contact applications.

Analyst Note: The reason that so many contact options exist is partly due to legacy requirements and also for numerical efficiency. When working with large assemblies, the contact algorithm can often represent the most numerically costly part of the simulation (e.g., >50%). To pare down run time, it can be very handy to remove the SOFT option and to explore the many other more economical contact algorithms; such as NODES_TO_SURFACE or to remove the _AUTOMATIC option.

8.4 Instructor Led Workshop VI Contact: Negative Sliding Interface Energy

Contact sliding energy has three contributions: (i) interfacial forces; (ii) energy dissipated due to friction (FS ≠ 0.0); and (iii) energy dissipated by contact damping (on the Contact Card VDC ≠ 0.0). In a well-defined model, the sliding interface energy (SLEOUT) should be positive and perhaps no more than 10% of the total internal energy of the model. However, a common numerical pathology for contact is to generate negative sliding energy and is a "red-flag" for that particular contact. Numerically, negative sliding energy is typically generated by: (i) parts sliding past each other (not friction) and the penalty method has difficulty maintaining the surfaces apart (interpenetrations) and (ii) rough mesh, large time step, etc. where the contact behavior has trouble pushing the surfaces apart cleanly and can get numerically lost in which direction to push the surfaces apart.

Analysts Note: It is common practice to report Sliding Energy in comparison to the analysis internal energy and also to provide a plot of individual Sliding Energy for each contact (SLEOUT).

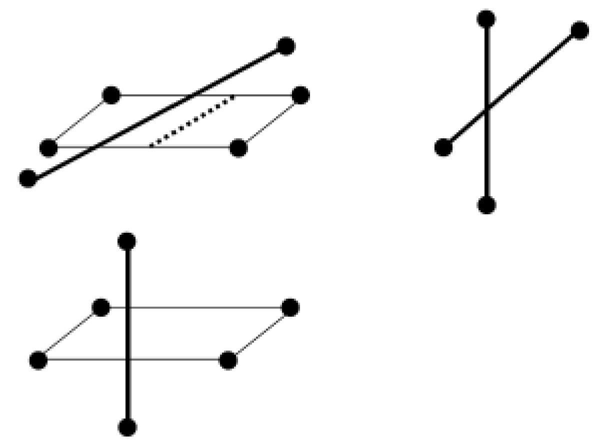
How to Fix (when it is more than a couple of percent of internal energy but even then it depends)

Eliminate initial penetrations, check for redundant contacts, reduce the time step scale factor (*CONTROL_TIMESTEP TSSFAC <0.9), refine the mesh, set contact card options back to default except SOFT=1 and IGNORE=1, explore the use of the DEPTH option.

8.5 Intermediate Contact Analysis

8.5.1 Beam and Edge-to-Edge Contact Modeling

Where Regular _AUTOMATIC Contact Fails



***CONTACT_AUTOMATIC_GENERAL**

Beam and direct edge-to-edge contact creates special challenges since contact is based on a line between two nodes. For beam elements, the contact surface is enforced as a cylinder regardless of the beam's cross-section. The algorithm checks contact along the length of the beam and at its end. With this capability, all of the more complex interactions shown on the right are found and prevented. These are other features to this contact such as

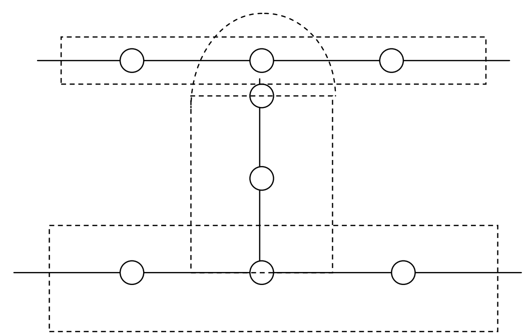
_AUTOMATIC_GENERAL is a "single-surface" contact and the user only defines the slave set for contact.

8.5.2 Special Contact Options

***CONTACT_AUTOMATIC_**

To handle contact at the end of plates, a semi-circular projection is made as shown on the graphic on the right for all _AUTOMATIC contacts. The radius of the projection is ½ the plate thickness and the option SHLEDG lets the user chose between round or square edge.

End Contact Occurs for all "Automatic" Contacts via Projection



This end contact option is the "gotcha" when constructing the "perfect mesh"

ISYM

This option controls how the contact algorithm handles symmetry edges. If your model has a symmetry plane enforced by SPC constraints, you'll want to set ISYM=1.

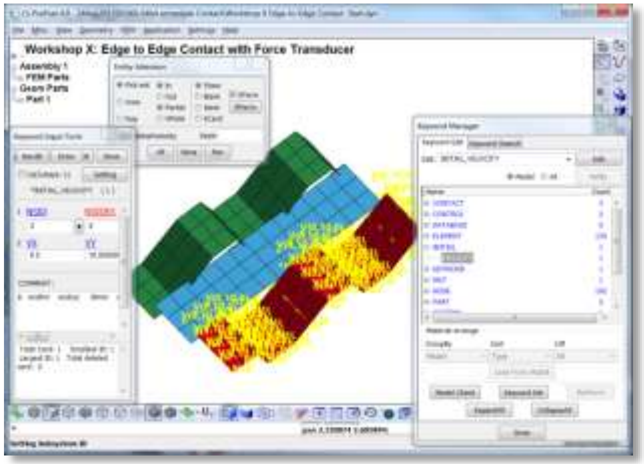
***CONTROL_CONTACT**

Allows "one-stop-shopping" to set your contact options.

8.6 Workshop X: Edge-to-Edge Contact

Problem Statement

A thin (0.002 m) corrugated plate is slammed into two other plates using a recommended default contact algorithm of _AUTOMATIC_SINGLE_SURFACE. Upon inspection, it doesn't work. How would you fix it?

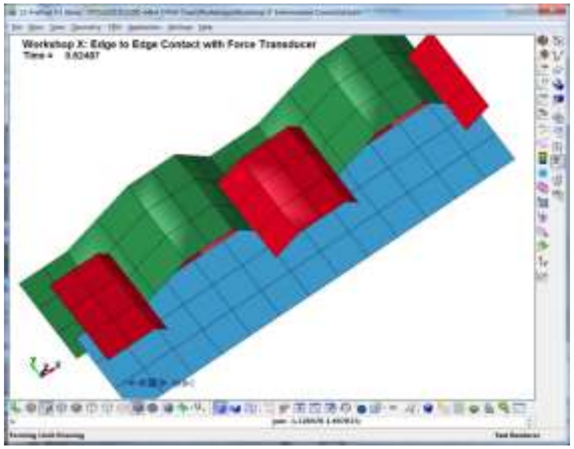


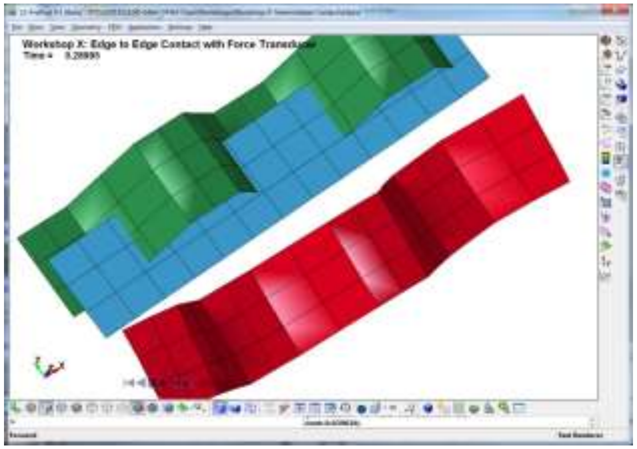
Script

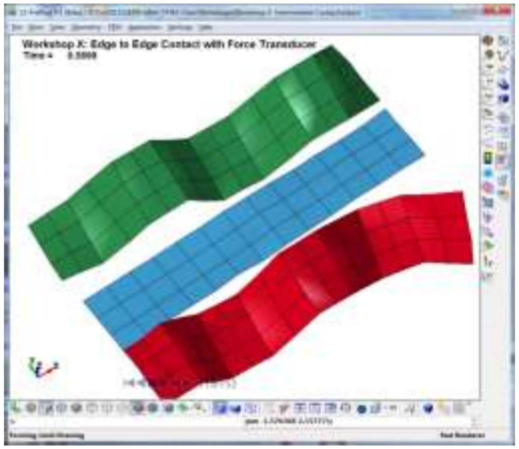
- Run model and look at contact behavior.
- Change contact to _AUTOMATIC_GENERAL
- Re-analyze model and check contact

Analyst's Note: One may note that LS-DYNA is warning you that the time step should not exceed 1.4e-5 while the model's running time step is 2.1e-4.

- Reduce timestep via TSSFAC to 0.5 (add *CONTROL_TIMESTEP card)
- Re-analyze and then lastly remove the interpenetration by "translation"







8.7 Tied Contact for Mesh Transitions, Welding and Gluing

8.7.1 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_SURFACE_TO_SURFACE_BEAM_OFFSET
- *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.

*Analyst's Note: Tied contacts are not really "contacts" but a constraint relationship that uses the *CONTACT card entry format. For an explicit analysis, the constraint option ties the adjacent nodal velocities together while for implicit, the displacements are tied. This explains why the nodes must be on the same plane or adjacent and also why this formulation can't be used with rigid bodies or have SPCs attached to any node that is tied. Additionally, it is only generally applicable for just translational DOF (TX, TY & TZ).*

With the OFFSET formulation, the penalty method is used. If the BEAM option is employed all six DOF's are tied together by essentially using little springs between the nodes. This is the most computationally expensive tied contact but the most robust. 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 nodes.

The reason that we like constraint Tied Contact is that it is stable whereas penalty Tied Contact _Offset is penalty based and has all the standard pathologies of regular contact such as the possibility of creating negative sliding interface energy.

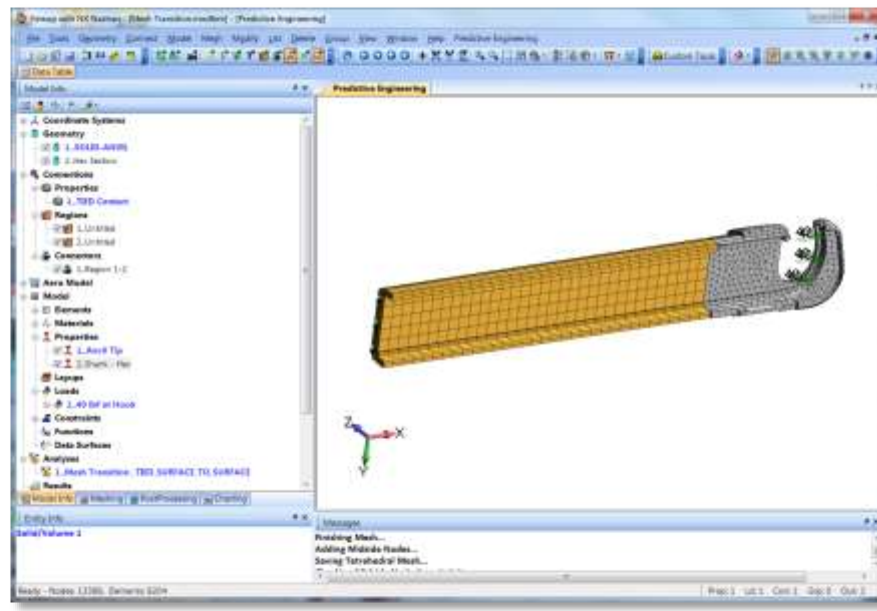
8.8 Summary Table for Tied Contact

*CONTACT_TIED_	Recommended Usage	Type	Pros/Cons
_SURFACE_TO_SURFACE	Gluing solid mesh transitions together where the two meshes are co-planar/adjacent.	Constraint	***Pros:*** Provides smooth displacement and stress interpolation across dissimilar meshes between hex-to-hex or tet-to-hex. **Cons:** If the meshes are co-planar, there are no cons.
_NODES_T0_SURFACE	Useful for creating weld edge lines between two solid elements parts that are co-planar; simulates a "fillet weld".	Constraint	***Pros:*** Allows the logical modeling of edge contact between solid parts. If the meshes are co-planar, there are no cons. ***Cons:*** 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	***Pros:*** Handles all six DOF's using a constrain method. ***Cons:*** Not designed for solid elements.
_SURFACE_TO_SURFACE_ BEAM_OFFSET	Gluing solid mesh transitions when the meshes are not co-planar.	Penalty	***Pros:*** Allows one to glue together dissimilar meshes that are offset. Also one may glue deformable bodies onto rigid bodies. ***Cons:*** Not suitable for plate and beam connections.
_SHELL_EDGE_SURFACE_ BEAM_OFFSET	Ideal for welding together plates/beams or for plates/beams onto solids whether deformable and/or rigid with offsets.	Penalty	***Pros:*** 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. ***Cons:*** None except that it is penalty based.

8.9 Workshop XI-A: 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 items 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.

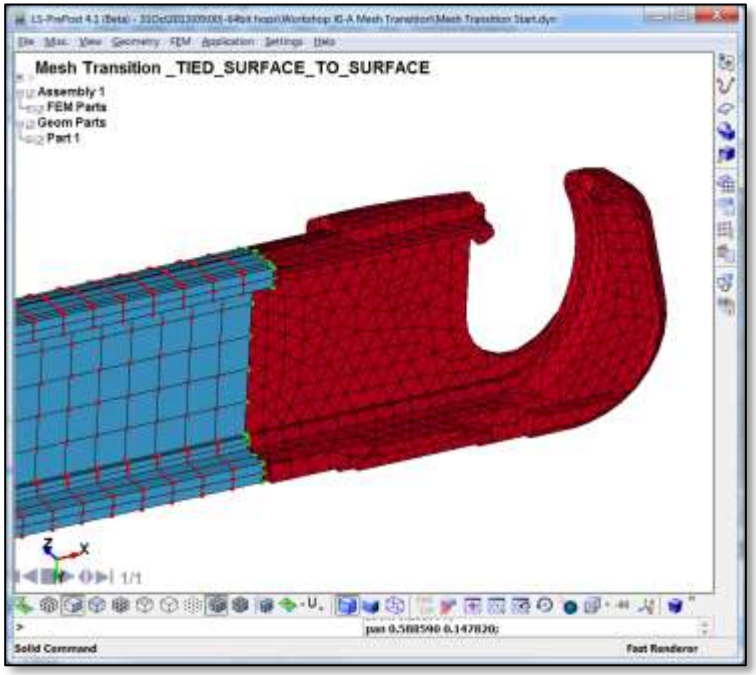


Workshop Script

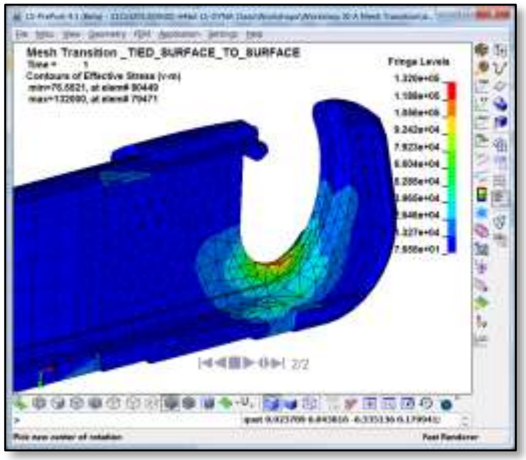
A small endoscopic, surgical stapling anvil is given a 40 lbf QC proof load. The yield stress of the powder metallurgy SS is 110 ksi. Is this device safe for surgical use?

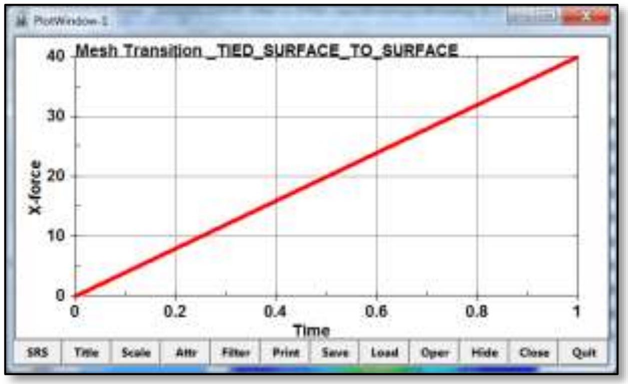
- One starts with the provided Femap model. Inspect how contact was defined. Export model using the LS-DYNA translator.
- Open model in LSPP, inspect how contact was translated and verify *CONTROL cards for analysis.
- Run Analysis and contour stresses
- Use the XY plot capability to verify reaction forces
- Would you recommend this design?

Export to LSPP and Verify Loads, Constraints and Contact



Run Implicit Analysis and Verify Load





This image created by Application / Model Checking / General Checking / Contact Check. Highlight the “Contact” and hit the Check button.

After contour stresses, make an XY plot SPC force plot by summing the SPC’s (see ASCII button).

Keyword Commands Used

*CONTROL_IMPLICIT_GENERAL (IMFLAG=1), *CONTACT_TIED_SURFACE_TO_SURFACE, *DATABASE_ASCII (SPCFORC=1)

8.9.1 Student Bonus

If this is working for you, change the DTO field to 0.1 within the *CONTROL_IMPLICIT_GENERAL card and watch what happens. Then, change the TIED contact to that of *CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET. If you are wondering what is what, read the manual.

Another avenue is to investigate how plasticity affects the analysis. Implement material plasticity using a yield stress of 100,000 psi with a tangent modulus of 100,000 psi. If this is too boring, change the yield stress to 30,000 psi with a tangent modulus of 10,000 psi.

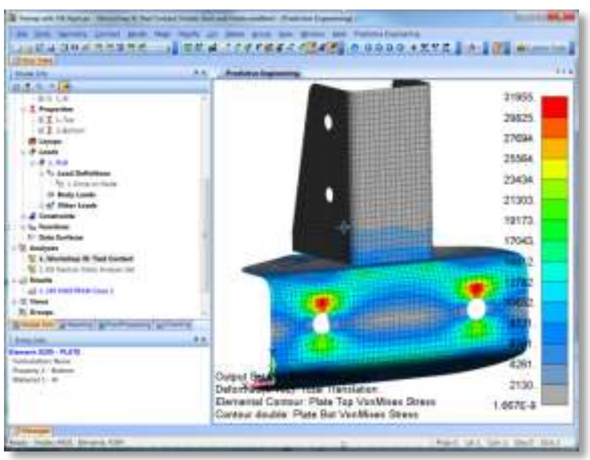
8.10 Workshop XI-B: Tied Contact for Gluing Things Together (BEAM_OFFSET)

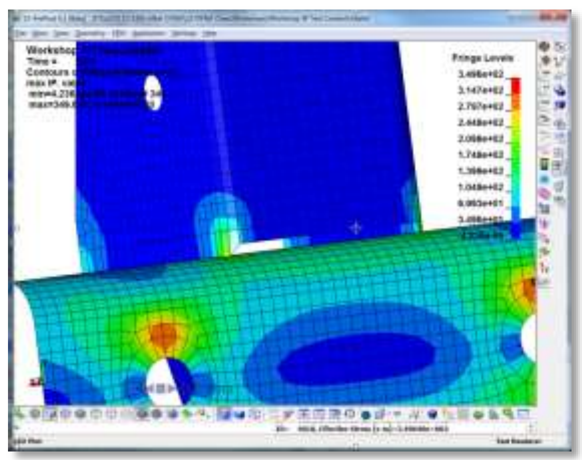
This workshop takes a typical example of where to use Tied Contact and walks through the setup and debugging process. A comparison is also provided between the Nastran and LS-DYNA model.

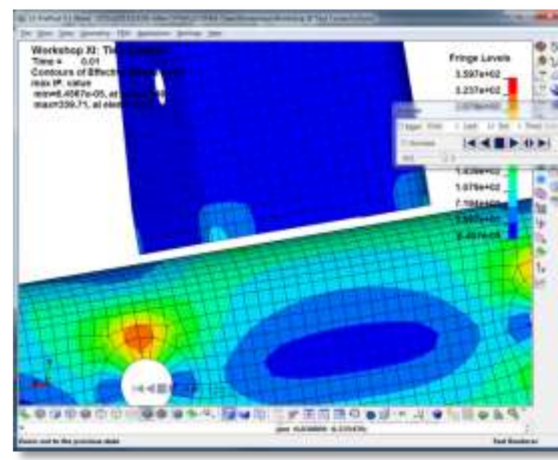
Script: Review contact setup within Femap; Run NX Nastran Analysis; Set Analysis Manager to LS-DYNA model and export; Run LS-DYNA model and interrogate results; Change contact to _BEAM_OFFSET; Verify Tied Status and correct accordingly; Re-analyze model and interrogate results.

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 and MST to a negative number that reflects the absolute distance to search for a tie relationship between the slave and master nodes.

Required Reading: Class Reference Notes / Keyword Manuals / LS-DYNA_manual_Vol_I_R7.0.pdf – General Remarks on *CONTACT on page 549 to 555.





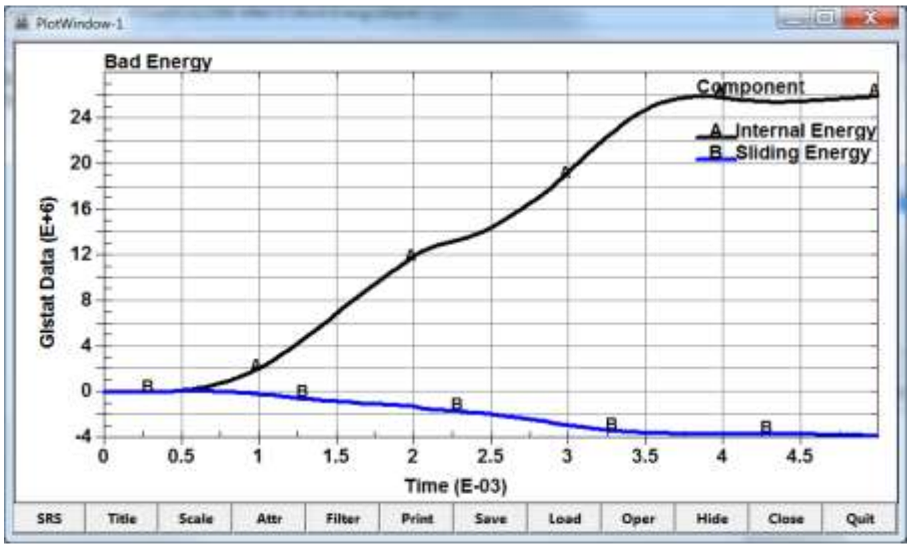


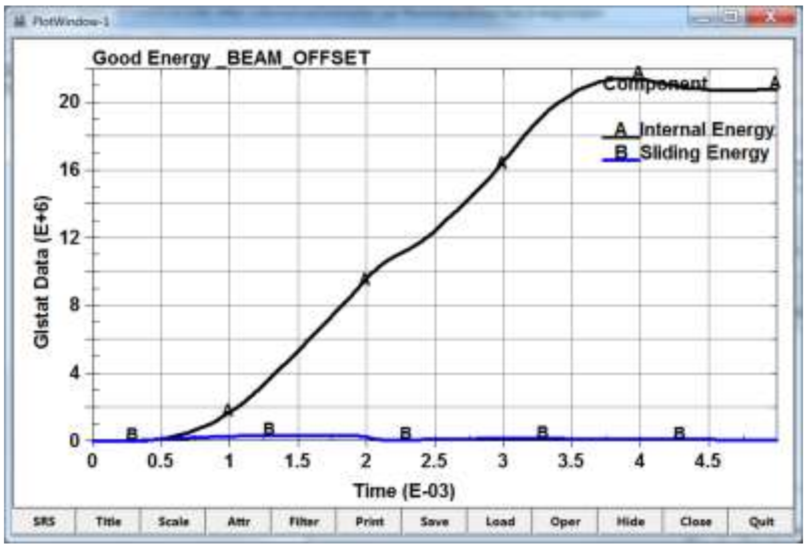
8.10.1 INSTRUCTOR LED BONUS WORKSHOP BAD ENERGY WITH TIED CONTACT

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. 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.





9. DAMPING

9.1 General, Mass and Stiffness Damping

In dynamics, there often can be some oscillations that the analyst would prefer to have damped out or to account for viscous behavior of some materials (see Material Damping). By default, LS-DYNA is undamped.

9.1.1 *DAMPING_*OPTION*

Introduces Rayleigh proportional damping based on: $[c] = \alpha[m] + \beta[k]$

- The mass damping constant α is specified by *DAMPING_GLOBAL, *DAMPING_PART_MASS and *DAMPING_RELATIVE.
- The stiffness damping constant β is activated by *DAMPING_PART_STIFFNESS.

Mass damping is for low frequency response (rigid body modes), while stiffness damping is more effective at higher frequencies. Since they are dissipative, their energy loss should be tracked. This can be done with the *CONTROL_ENERGY option of RYLEN=2. Energy loss is then reported in the glstat and matsum files.

9.1.2 *DAMPING_FREQUENCY_RANGE

This is a more elegant approach to damping and allows the user to specify the critical damping coefficient and the frequency range to damp. It is effective when used with low amounts of damping (e.g., 1 or 2%) and when the frequency range is no more than a factor of 30x (e.g., 100 to 3,000).

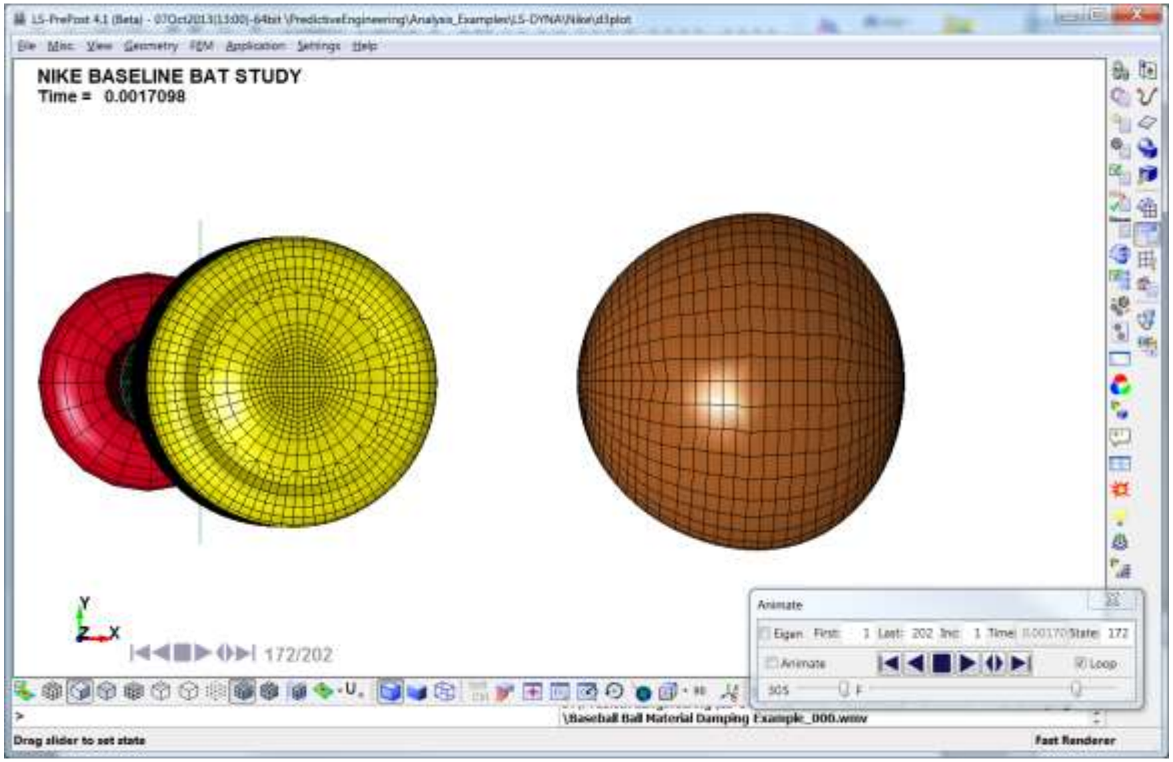
Analyst's Notes: I know of no shortcut to producing good agreement with the observed loss in a test I can only suggest good judgment and a trial-and-error approach in order to tune the numerical damping.

9.1.3 Material Damping (e.g., elastomers and foams)

Elastomers (e.g., *MAT_181 Simplified Rubber) and foams (e.g., *MAT_053 Fu Chang foam) have the ability to add damping directly within the material card. Recommended values are between 0.05 and 0.5. However, there is no true recommended value since each material application is a bit different and requires some observation by the analyst to determine the appropriate value.

9.1.4 Student Example on Material Damping

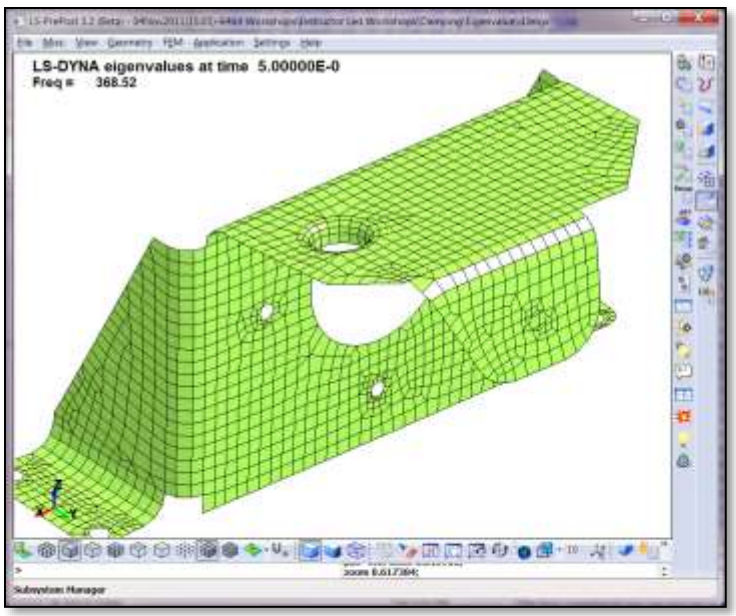
An example of material damping is provided in the Class Reference Notes / Damping titled: Baseball Ball Material Damping Example.dyn along with a movie file of what one can expect to witness.

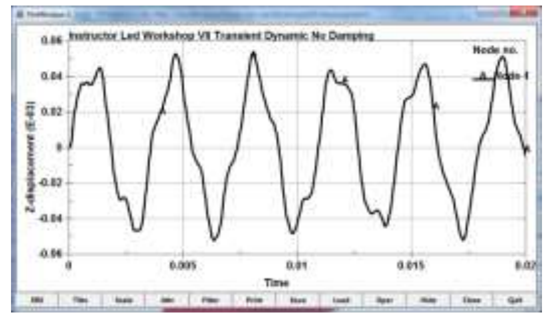


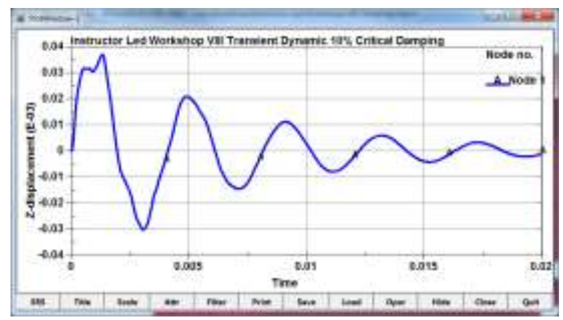
9.2 Instructor / Student Led Workshop VII: Damping

An example is given of a vibration problem run by NX Nastran normal modes analysis and then by undamped and damped transient analysis. If the student desires, the model is easy to switch to LS-DYNA implicit for an Eigenvalue run.

Script: The model is inspected within Femap and and Nastran normal modes analysis is run. The first natural frequency is noted. The model is then exported out to LS-DYNA and directly analyzed. A History plot is made of node 1 in the Z-direction. Using LSPP X-Y plot tools, under Oper, a fft is applied and note the frequency at the first big spike on the FFT graph. The model will then be given 10% critical damping between 250 and 350 Hz. Save and plot results.







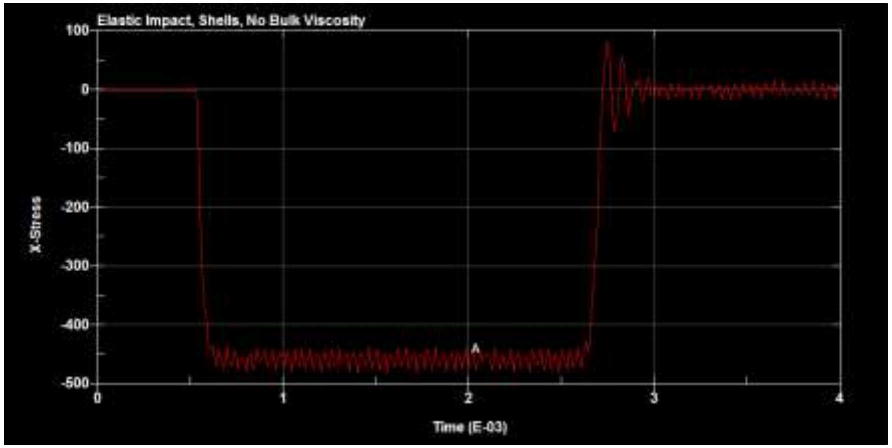
Student Bonus: One can switch it to an Eigenvalue analysis by adding these two *CONTROL cards: _IMPLICIT_EIGENVALUE (NEIG=5) and IMPLICIT_GENERAL (IMFLG=1). Once analyzed you'll want to view the d3eigv.

9.3 BULK VISCOSITY

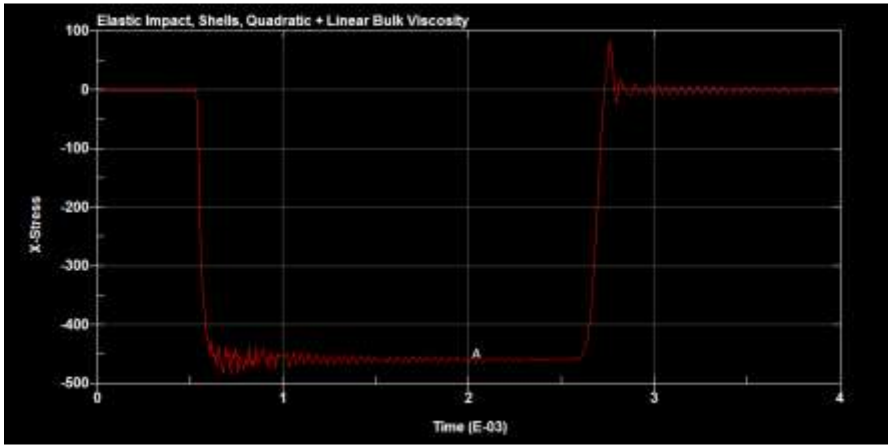
Although the use of bulk viscosity as a form of damping is not exactly classical since it was developed for the treatment of shock waves through the FEA mesh, it has grown into a recommended procedure for general dynamic structural FEA work even when shock waves are not present. The *CONTROL_BULK_VISCOSITY card has several options and the default are recommended. If shell elements are present in the analysis, one should set TYPE=-2 to activate this calculation for shells. The reason it is not default is that shock waves are more of a solid element behavior. In the case of shells, we are more interested in the linear bulk viscosity mechanism (see LS-DYNA Theory Manual for more details).

Example of the effects of bulk viscosity on an impact analysis is shown below courtesy of Suri Bali and his excellent website Blog2.d3view.com.

Standard Response



Slightly Smoothed w/ Bulk Viscosity



Images courtesy of Suri Bali via his Blog2.d3view.com – full article can be found in Class Reference Notes / Bulk Viscosity

*Analyst Note: Although not necessary, it is recommended to add *CONTROL_BULK_VISCOSITY to any transient structural FEA analysis since it tends to slightly damper out crazy oscillations at little numerical cost and with little energy cost.*

10. LOADS, CONSTRAINTS AND RIGID WALLS

10.1 Loads

10.1.1 Initialization Loads

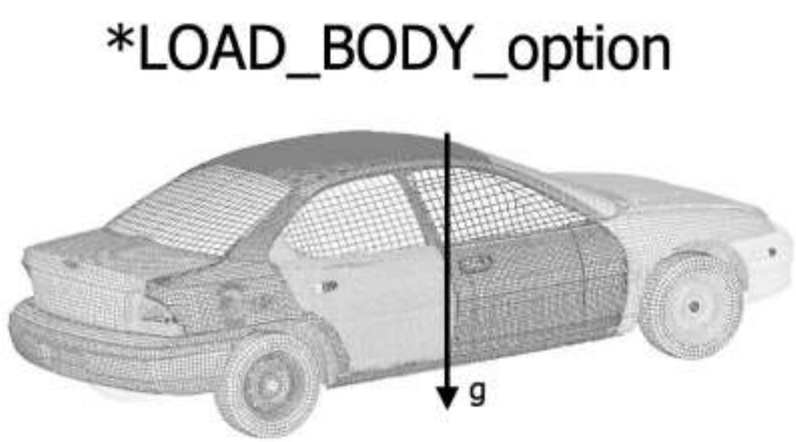
The most common initialization load is *INITIAL_VELOCITY_*option*. For example, for any type of drop test, the structure is given a uniform initial velocity and then allowed to instantaneously hit the target. The other common initialization command is *INITIAL_TEMPERATURE for thermal analysis work.

10.1.2 Point and Pressure Loads

There is nothing complex to these loads. Point loads are those loads applied at nodes while pressure loads are applied over element faces. In LS-PrePost, pressure loads are applied onto segments (i.e., faces).

10.1.3 Body Loads

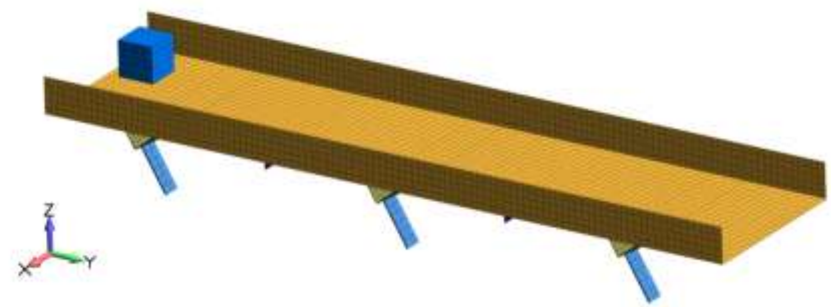
Body loads are most commonly defined as constant acceleration to capture the effect of gravity. Keep in mind that LS-DYNA treats body acceleration loads differently and that to obtain the same direction of gravity in Nastran one must switch the sign of the acceleration load.



10.2 Workshop: Applying Moving Boundary Conditions

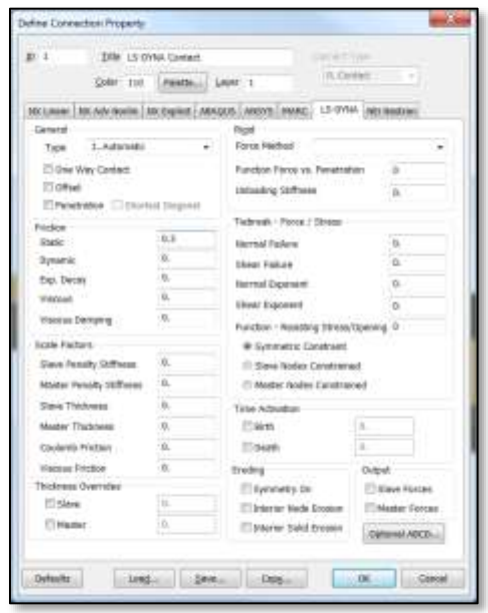
This workshop is quite simple and just demonstrates how one can create a variety of boundary conditions using some of the LS-DYNA commands.

Objective: Interrogate the Femap model and notice how the load condition is defined with a 5 Hz function with gravity. We are going to bounce the cube down this shaker but the model needs some work.



Tasks

- Setup contact between the cube and the shaker pan within Femap with a static friction of 0.3
- Define the two Regions using Properties and then setup the Connector.
- Following this procedure, define glued contact (TIED) between the shaker table springs (the lats) and the bed supports. Keep it simple and use OFFSET. To bridge the gap between the components, set the slave/master thickness to 2. For ease, create Regions using Properties.
- Set up Analysis Manager for a run time of 5.0 seconds with a results frequency of 0.05. Run model. Please note estimated run time and time step.
- Assume we don't need to have the shaker table bed, supports as "deformable", switch the material model to "rigid" and note the speed-up.

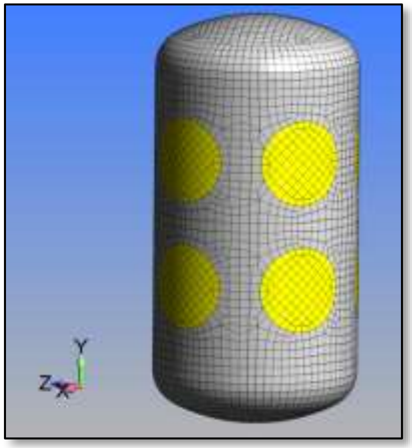


10.3 Workshop XII: Drop Test of Pressure Vessel

This exercise is geared toward increasing your confidence in using LS-DYNA. We are going to do a standard drop test where the body is given an initial velocity and a rigid wall is placed directly beneath the vessel. At the end of this exercise we'll fill the vessel with a fluid. The objective is to get familiar with manipulating a model within LS-PrePost (LSPP). The workshop starts by exporting the model from Femap where one has defined the materials (aluminum and rubber) and the mesh with a few predefined LS-DYNA options within the Femap Analysis Set Manager.

Workshop Script

- A thin-walled (0.05") aluminum vessel that is 24" in diameter and 36" tall is impacted at a speed of 100 MPH (1,760 in/s) against a rigid wall. The first task is to set the initial velocity for the vessel followed by creating the rigid wall.



- Use *INITIAL_VELOCITY_GENERATION to get the vessel going. One will need to create a part set to define what you want to initialize (see *SET_PART). Set initial velocity to Vy=-1,760 and all other options default.
- Define rigid wall with *RIGIDWALL_PLANAR. We are placing this wall perpendicular to the bottom of the vessel. The bottom of the vessel is at 0, 0, 0. To avoid initial contact with the rigid wall, it is placed below the outer skin of the vessel at 0, -0.025, 0 and the head of the vector at 0, 1, 0 to define its position.
- Run model and notice that the pressure vessel skin folds in upon itself.
- Add *CONTACT_AUTOMATIC_SINGLE_SURFACE (use prior part set for your "single surface") and set SOFT=2 for improved contact behavior. Run model and it should look better. This model has **Finish I** appended.
- The next step is to apply a pressure load. Pressure loads are typically applied to "segments" which is LS-DYNA terminology for element faces. This operation could be done in Femap but we'll proceed with LSPP. One will need to

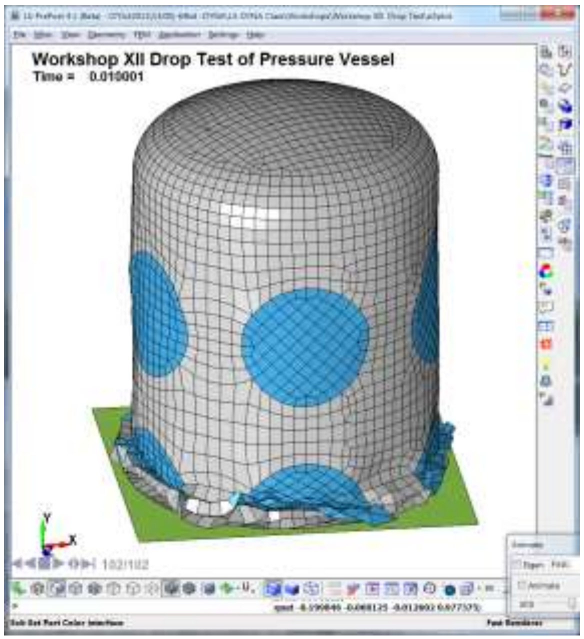
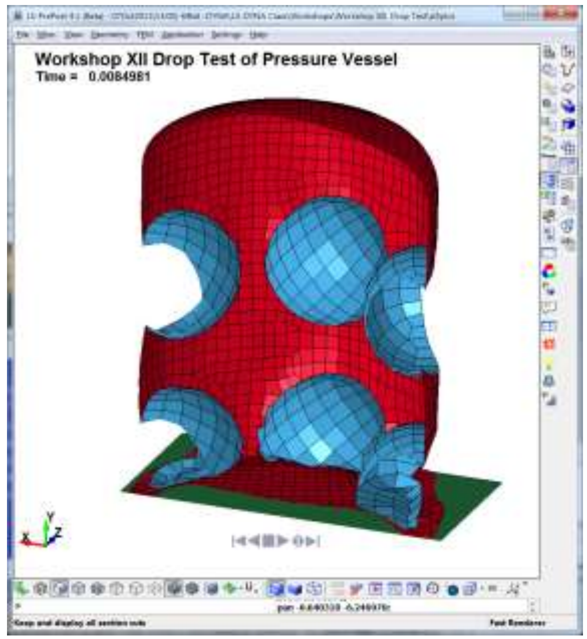
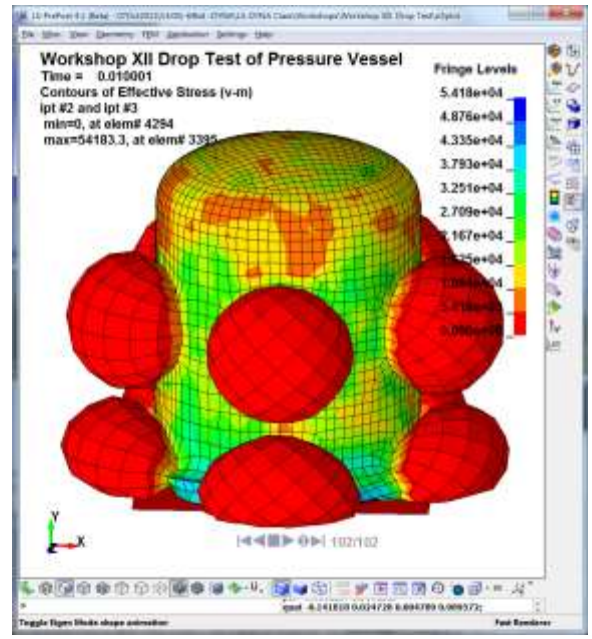
create a segment set, define a load curve and then create the pressure load.

- To create your segment set, use the Create Entity option within LSPP. This option is located within the Model and Part toolbar below the Keyword Manager button. Within the Create Entity screen, expand the Set Data option and select *SET_SEGM option. Another pane will appear. Within this pane you'll want to Create (Cre) the new segment set. I would recommend picking the segments using the ByPart option. After picking, hit Apply and you're done.
- Create your pressure load curve using the *DEFINE_CURVE command. Set the curve to unity over time, e.g., 0,1 and 1,1. When done, this should be curve number 2.
- The pressure load is finally created using the *LOAD_SEGMENT_SET command. We are setting the pressure to 5 psi by setting SF=5. Run the model and interrogate. This model has **Finish II** appended to its name.

- To simulate the incompressible fluid behavior within the vessel, add *AIRBAG_LINEAR_FLUID to the model with BULK=5e5 and RO=1e-4 with all other options default. The segment set created for the pressure load is re-used to define the enclosed fluid volume. The completed card is shown on the right.
- Rerun model. You should see the rubber portals bulge. This model is noted as **FINISH III**.

10.3.1 Student Bonus

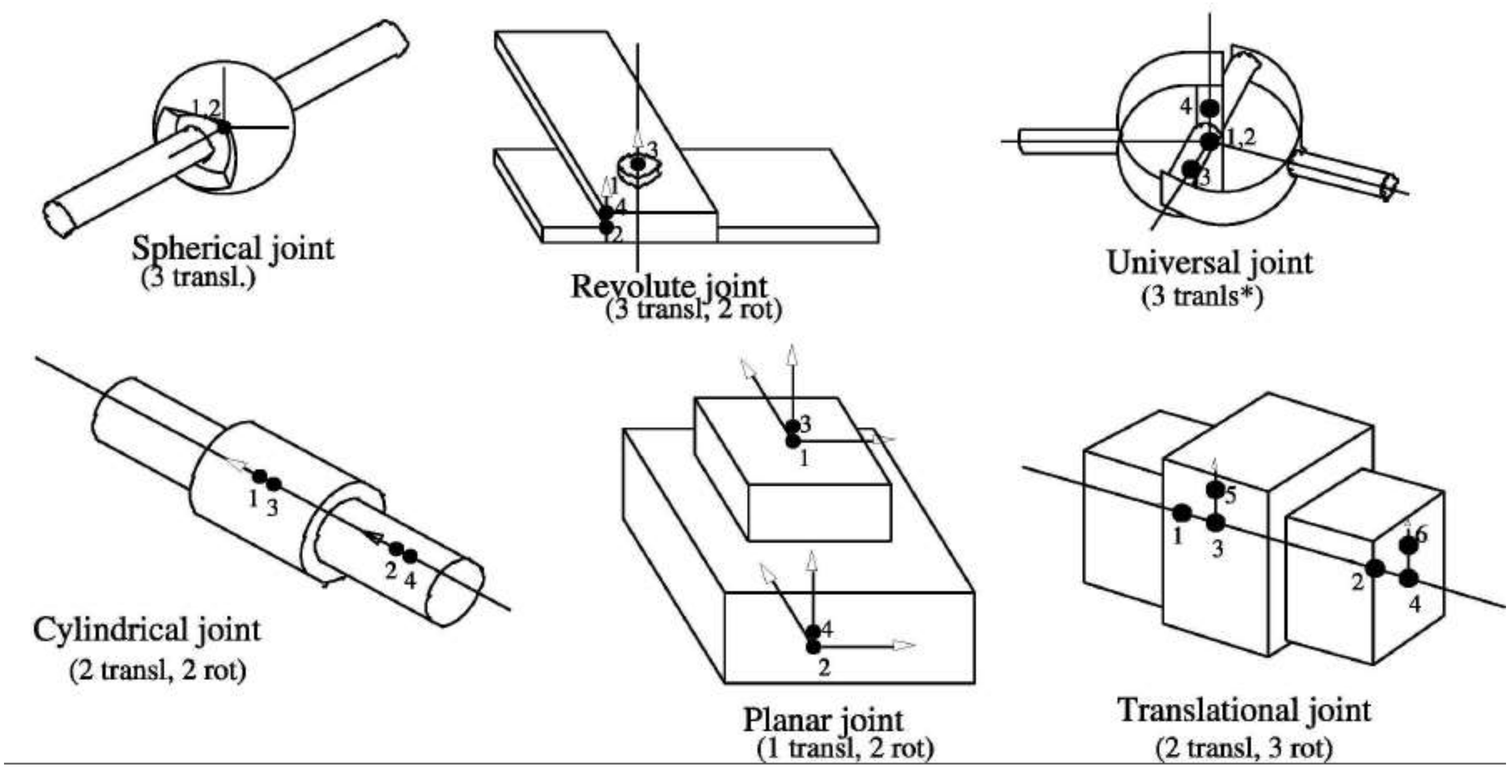
Take the prior model and reorient the *RIGIDWALL_PLANAR to make it vertical or in the XY plane. Then change the *INTIAL_VELOCITY_GENERATION to have the vessel hit normal to the XY plane or in the Z-direction. We are simply re-orienting the impact from smashing down on the ZX plane to hitting “sideways” against a XY plane (so-to-speak).

FINISH I (Change Colors)	FINISH II (Create Section Cut)	FINISH III (Contour Stresses)
		
Clean and neat drop test against rigid wall.	The pressure is inward (function of the plate element normals or sign of the pressure load.	As the vessel is collasped, the fluid volume pushes the rubber portals outward.

11. CONNECTIONS VIA JOINTS, BOLTS AND SPOTWELDS

11.1 Joints or *CONSTRAINED_JOINT_

To model the motion and likewise, large deformation 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. If time permits, we'll do a simple model.



12. LOAD INITIALIZATION BY DYNAMIC RELAXATION AND IMPLICIT ANALYSIS

12.1 Initialization of Gravity, Bolt Preload and Other Initial State Conditions

Stress Initialization

Given that explicit analysis work often involves timesteps in the range of microseconds, one can image the challenges of using an explicit approach to obtain quasi-static or static stress states in structures subjected to uniform constant loading. There are many applications where the ***start*** of the explicit analysis requires the initialization of steady-state loads within the structure. Here's a short list: rotating equipment (e.g., fans, turbine blades or rotating flywheels), pressurized vessels or tires, bolt preloads, shrink-fit parts, or spring mounted structures under constant gravity.

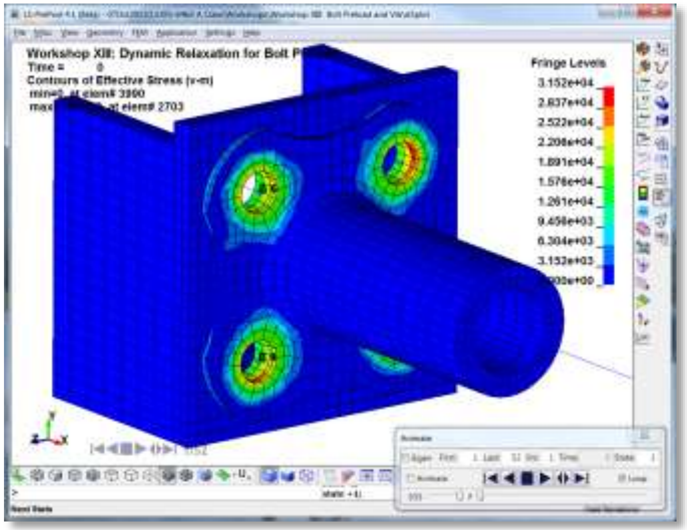
These static stress-state conditions can be simulated in LS-DYNA using two techniques: Explicit Dynamic Relaxation or Implicit Static Analysis.

Dynamic Relaxation (DR) *CONTROL_DYNAMIC_RELAXATION

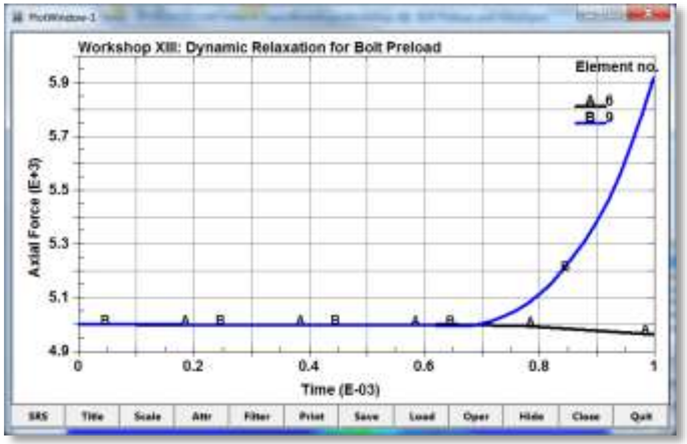
DR is a heavily damped explicit analysis that is initiated prior to the main transient analysis. It has all the characteristics of a standard explicit run but it is assumed that stresses are relatively elastic and that displacements are small. The solution is heavily damped and unexpected results may be generated. Nevertheless, with some models, it does a great job with bolt preload, tire inflation or application of a shrink-fit. In the DR process, the load is applied (e.g., bolt preload) as a transient load with a sharp ramp up and then a steady-state application. The model dynamically responds to this load application with all the characteristics of a standard explicit transient analysis. As the model is solved, the nodal velocities are reduced at each timestep by the dynamic relaxation factor (default = 0.995). The kinetic energy (KE) is calculated at prescribed steps and when this energy has decreased sufficiently against the initial KE, the solution is considered converged and the DR process shuts down and the solution is handed over to the regular, normal explicit transient analysis sequence. There are lots of options to manage this process and the student is referred to the Class Reference Notes for more information.

Analyst's Note: I have often struggled to get DR to work correctly. It seems "finicky" since the process is uses a heavily damped response and it just ain't natural. As such, whenever possible I strive to use the implicit approach.

12.2 Workshop XIII: Dynamic Relaxation



As modal frequency response analysis is known as "the poor man's transient dynamics", Dynamic Relaxation might also be known as "the poor man's implicit". Nevertheless, in many cases it gets the job done a lot faster than straight-up implicit and is easier to setup.



Script

- Using the "Start" model, create bolt loading curve (*DEFINE_CURVE) with the following three points: 0, 0 | 1e-4, 5000 | 1e-2, 5000. Set the SIDR=1 (please see Manual about details on this option).
- Apply bolt preload load to the four beam elements. This is done by *INITIAL_AXIAL_FORCE_BEAM (please see Manual about details on this Keyword). However, one will first need to create a beam set. This has already been done for you – thus proceed gently forward using this predefined beam set. Please note unique Material Law required for the use of this Keyword.
- Apply load of 1,000 units at the far end of the structure's tip. This done by creating another curve with the following four points: 0, 0 | 5e-4, 0 | 6e-4, 1000 | 1e-2, 1000 and SIDR=0. Then, under *LOAD_NODE_POINT, pick the node at the far end of the beam (it'll be 6352) with DOF=2 and LCID=2 (with any luck) and hit Accept.
- Run analysis.

Student Extra: Take the prior model and add mass scaling to model. Remember to set the DT2MS value to a negative number. Forgot why? Please read the manual since "structured organic learning sticks the best."

Analyst's Note: After completing Workshop XIV, come back to this example and switch to implicit.

13. IMPLICIT-EXPLICIT SWITCHING FOR BURST CONTAINMENT

An application for implicit startup is the initialization of the steady-state stress field for rotating equipment. For systems with high-speed rotating components, the model can be initialized to its steady-state spinning condition using an implicit analysis. To model any downstream event that may be highly nonlinear and dynamic, the model can then be switched over to an explicit analysis. The trick is performing this type of analysis is to remember that LS-DYNA lets you apply boundary conditions and manage contact behavior in a "birth / death" manner. For example, one can apply constraints to all secondary structures that are not relevant, or in the case of a turbine analysis, not spinning to ensure that the implicit analysis runs smoothly and then remove them (i.e., death) once the explicit analysis starts up. Likewise, this can be done with contact behavior within the model to avoid numerical difficulties during the implicit solve. Both of these tricks can dramatically speed-up the implicit solution without affecting the accuracy of the simulation.

Analyst Note: The implicit-to-explicit switching can be a bit tricky. What is recommended is that one switches the simulation to explicit prior to desired end-point of the implicit simulation. For example, if the implicit end point is 1.0 and the implicit time step is 0.25, then the curve used to control the switch from implicit-to-explicit could be set to any time point after 0.75 and before 1.0(e.g., 0.9) and the explicit "on" switch set to the end time of the implicit simulation (e.g., 1.000001).

13.1 HIGH-SPEED ROTATING EQUIPMENT – *CONTROL_ACCURACY

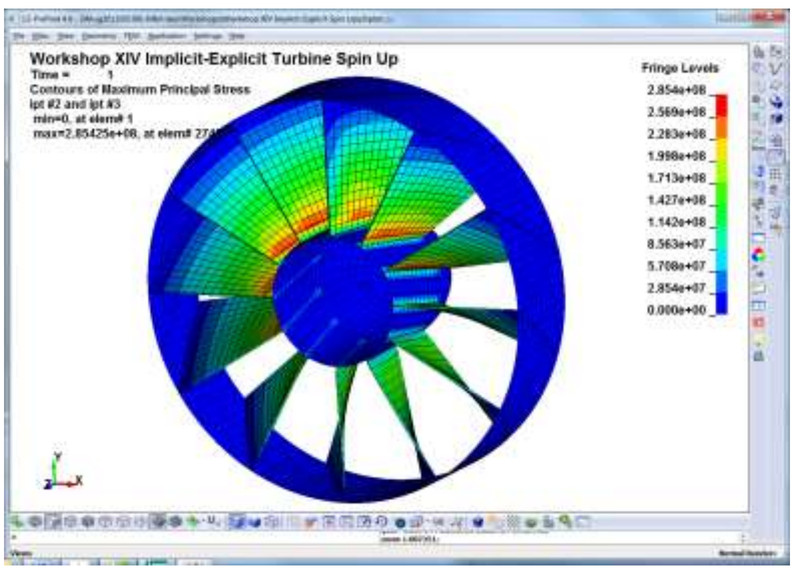
For structures that rotate, it is recommended that the *CONTROL_ACCURACY option OSU = 1 and INN = 2 be set. The OSU option adds additional terms to the stress update and improves the accuracy of the simulation while the INN option sets invariant node numbering to ensure the accurate calculation of element forces within elements that are become highly twisted and/or rotate through space. Interesting enough, the INN = 2 option is default for implicit calculations. Both of these options will slow the simulation down by as much as 10%.

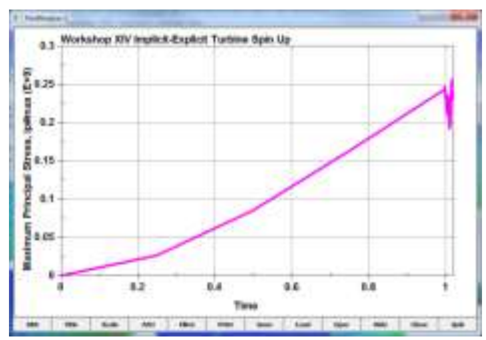
13.1.1 WORKSHOP XIV: IMPLICIT-EXPLICIT TURBINE SPIN UP

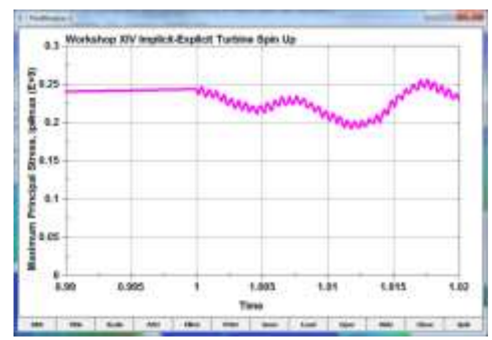
This workshop will show the student how to setup a turbine simulation from implicit ramp-up to steady-state rotating movement. The model has been mostly prepared and the student is required to finish the setup using the following Keywords:

*CONTROL_ACCURACY (OSU=1, INN=2), *CONTROL_IMPLICIT_GENERAL (IMFLAG=-1, DTO= 0.25), *CONTROL_IMPLICIT_SOLVER (LSOLVR=6), *INITIAL_VELOCITY_GENERATION (PHASE=1), *INITIAL_VELOCITY_GENERATION_START_TIME (STIME=1.00001), *INITIAL_VELOCITY_RIGID_BODY (VZR=225), *LOAD_BODY_PARTS (PSID=1), *LOAD_BODY_RZ (LCID=2, SF=225), *BOUNDARY_SPC_SET_BIRTH_DEATH (NSID=1, DOFX=1, DEATH=1.0).

For any implicit analysis, it is recommended to use the double-precision solver. This is activated within the LS-D YNA Program Manager under Solver, Select LSDYNA Solver. When completed, one should see the graphic on the right a small "d" within the solver name.







14. SMOOTHED PARTICLE HYDRODYNAMICS (SPH)

14.1 Introduction

A short introduction video for this section can be found in Workshops / SPH / Introduction to SPH Modeling

14.1.1 A Little Bit of Theory (skip this if you don't like math…)

Kirk Fraser, Sr. Staff Engineer, Predictive Engineering

Smoothed particle hydrodynamics (SPH) was developed in the 1970's by Monaghan, Gingold and Lucy for astrophysics problems. Monaghan has published an enormous number of papers on the SPH method. Libersky et al. [1] were the first to apply the method to solid mechanics problems. Lacome [2] was one of the first to implement SPH in LS-DYNA.

Mesh-based methods do a great job for all kinds of engineering calculations. When the deformation gets really large, mesh-based methods start to fail due to negative element volume, excessive mesh distortion and/or mesh tangling within contact region which then causes problems with the explicit time step and so on and so forth.

SPH is a Lagrangian based mesh-free method that can handle unlimited plastic deformation. The rate of change of the field variables for a given particle "i", with ***N*** "j" neighbors in the support domain is given by Lui and Lui [3]: $W_{i,j}$ is the smoothing function (interpolation kernel) and can take on many different forms depending on the type of problem being studied (e.g., the cubic spline function is popular), $\pi_{i,j}$ is an

Continuity Equation

$$\frac{D\rho_i}{Dt} = \sum_{j=1}^{N} m_j ({v_i}^{\beta} - {v_j}^{\beta}) \frac{\partial W_{i,j}}{\partial x_i}$$

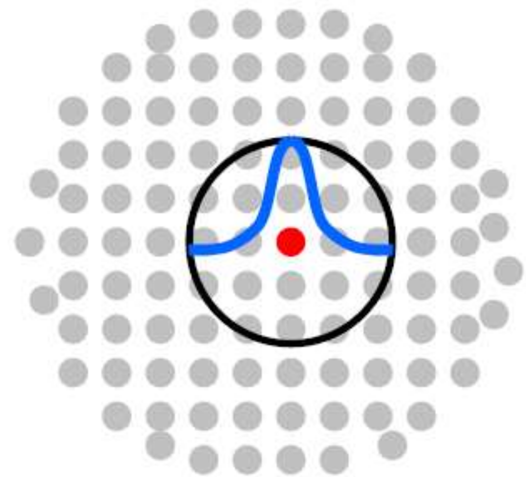
Conservation of Momentum

$$\frac{D{v_i}^{\alpha}}{Dt} = -\sum_{j=1}^{N} m_j \left(\frac{{\sigma_i}^{\alpha\beta}}{{\rho_i}^2} + \frac{{\sigma_j}^{\alpha\beta}}{{\rho_j}^2} + \pi_{i,j}\right) \frac{\partial W_{i,j}}{\partial x_i}$$

Conservation of Energy

$$\frac{De_i}{Dt} = \frac{1}{2}\sum_{j=1}^{N} m_j \left(\frac{p_i}{{\rho_i}^2} + \frac{p_j}{{\rho_j}^2} + \pi_{i,j}\right) \left({v_i}^{\beta} - {v_j}^{\beta}\right) \frac{\partial W_{i,j}}{\partial x_i} + \frac{1}{\rho_i} {\tau_i}^{\alpha\beta} {\varepsilon_i}^{\alpha\beta} + H_i$$

artificial viscosity term and H_i is an artificial heating term. The general idea is to use a finite number of neighbors within a radius of influence (also known as the smoothing length) on the central element. The graphic on the right depicts how the smoothing length can be visualized for a SPH mesh.



The method converts a set of partial differential equations (PDE) into a set of ordinary differential equations (ODE). The ODE's can be integrated in time with many different schemes; in LS-DYNA a multi-step (fractional step) explicit method is used. This means that there is a stability condition on the time-step size (CFL):

$$\delta t_{min_i} = \xi \left(\frac{h_i}{c_i + v_i}\right) m_j$$

ξ is a constant and is typically 0.2 to 0.4. Implicit time integration a possibility in the future in LS-DYNA, but for now, only explicit is available. For the truly die-hard SPH gear-heads, I recommend the book by Damien Violeau [4] (you gotta really love math to enjoy this book) or for a less math intensive approach, see the book by William G. Hoover [5].

Lastly, for modeling constitutive relationships in SPH, one can use many of the same material cards (i.e., laws) as a regular Lagrangian analysis.

14.1.2 Lagrangian vs Eulerian

The two most common frames of reference for numerical simulations is Lagrangian or Eulerian. You can think of Lagrangian reference like you are sitting on a plane and Eulerian like you are on the ground (not moving) and watching the plane go by. Lagrangian makes following the history of each material point very easy compared with the Eulerian description. CFD codes (finite difference, finite volume and finite element) are written from an Eulerian formulation.

The Lagrangian nature of SPH makes it a very powerful numerical method, this opens many doors that Eulerian method closes (or makes it difficult to open the door).

14.1.3 Types of Simulations with SPH

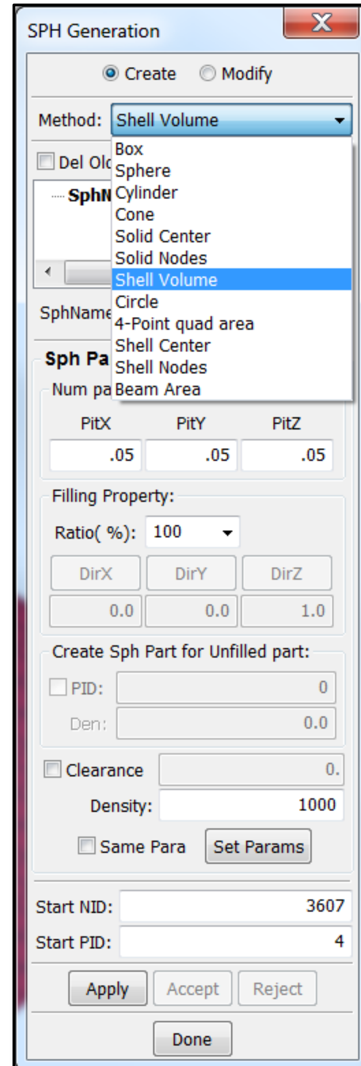
- Impact and ballistics (e.g., bird strike)
- Fracture and fragmentation
- Fluid structure interaction (e.g., sloshing)
- Linear and non-linear vibrations
- Microstructure evolution
- Heat transfer
- And many more…

14.1.4 Common Keywords for SPH

- *CONTROL_SPH
- *SECTION_SPH
- *CONTACT_AUTOMATIC_NODES_TO_SURFACE with SOFT=1 is recommended
- All *EOS_ and most *MAT_ (see Keywords manual for details, [6] and [7]) cards can be directly used
- Most of the standard keywords work for SPH
- Node sets need to be used to define contact, boundary conditions, etc.

14.2 SPH Workshop I: Getting Started

This example (courtesy of Kirk Fraser) is geared toward demystify the process of creating and running a SPH model. The concept is that any box, sphere, cylinder, etc. can be used to create a SPH mesh. Once the mesh is created, the model is setup and analyzed.

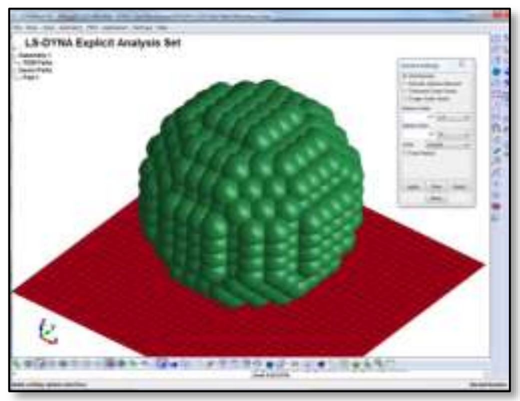


Start-Up

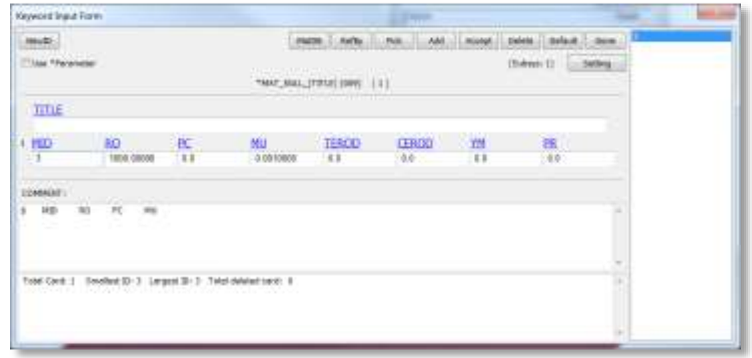
- Open the Femap model SPH / SPH Workshop 1 / SPH 1 Ball and Plate Example (Setup for SPH).modfem and then fire up the movie file.

Goals

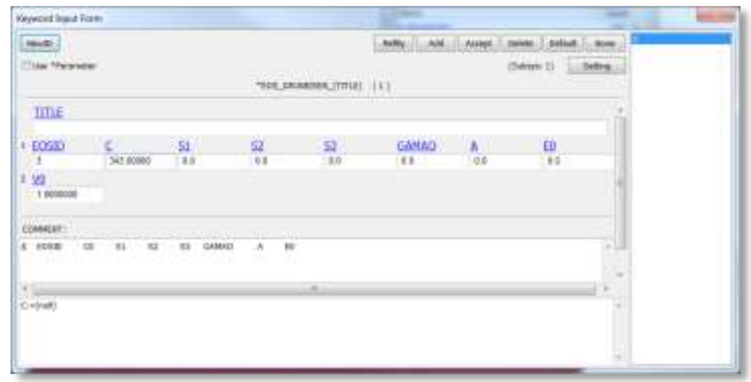
- Get familiar with the concept of creating a SPH mesh.
- The procedure is pretty simple and offers flexibility
- Visualize your SPH spheres
- Convert model to full SPH analysis
  - *CONTROL_SPH IDIM=3
  - *SECTION_SPH
  - *PART
  - *CONTACT_AUTOMATIC_NODES….
  - *TERMINATION 0.025 s
- View results
- At the end of this workshop, see Class Reference Notes / SPH / LSTC SPH Short Course Notes.pdf



14.3 SPH Workshop II: Fluid Modeling



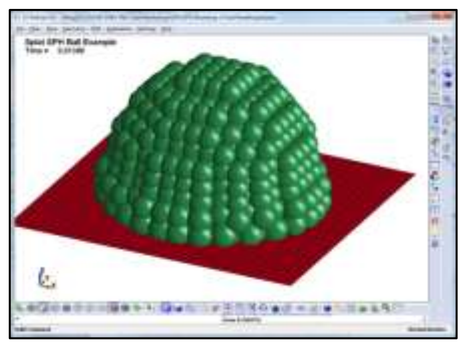
In this example (courtesy of Kirk Fraser), the plate is made rigid and the ball is turned into water by using two cards: *MAT_NULL and *EOS_GRUNEISEN.

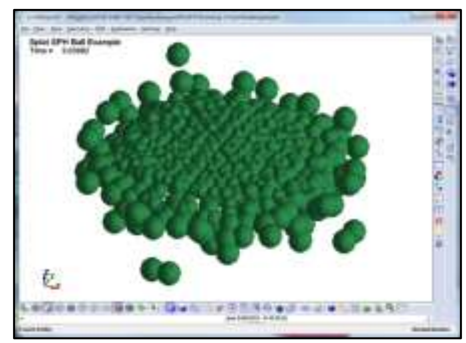


Learning Objective

- Material modeling of fluids is not that difficult. It is equivalent to modeling an isotropic solid material with E and υ. For a fluid at its simplest form, there is speed of sound "*c*", density and viscosity.
- The c_{water} is 342 m/s and has a density of 1,000 kg/m^3. The viscosity of water is 0.001 Pa*s.
- Run, explore and then change mu=1,000.
- SPH can also be used for a sloshing analysis.

*Analyst's Note: SPH doesn't need hourglass control but if a default value is applied (see *CONTROL_HOURGLASS) it will get used! Since the Null material has no shear stiffness, the default hourglass coefficient can lead to significant energy losses. Given this feature, avoid the use of *CONTROL_HOURGLASS for SPH simulations and apply hourglass control directly to the part of interest.*



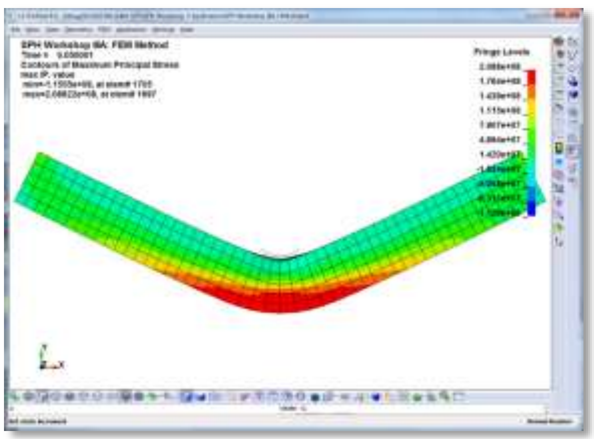


14.4 SPH Workshop III: Verification

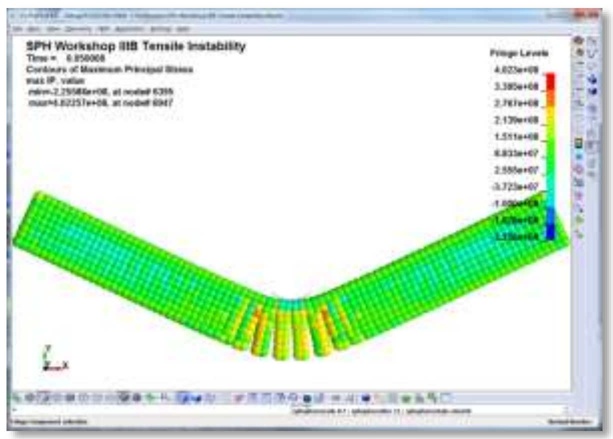
This workshop (courtesy of Kirk Fraser) compares the SPH method against the standard FEM method and discusses a common numerical difficulty with SPH method known as "Tensile Instability". This introduction is not meant to be anywhere complete but just to get the student started in how to debug their SPH models and what they might want to look out for while interrogating the model. If this subject is of greater interest, LSTC provides a two-day course on the SPH method and the course notes can be viewed within Class Reference Notes / SPH / LSTC SPH Short Course Notes.pdf.

Three models are provided: IIIA FEM Method, IIIB Tensile Instability and IIIC Recommended for the student to compare and explore the SPH method. The workshop movie file provides all the steps and a bit of discussion. Along with learning about tensile instability, the student will learn how to post-processes SPH results and create comparison graphs between principal stress and contact force.

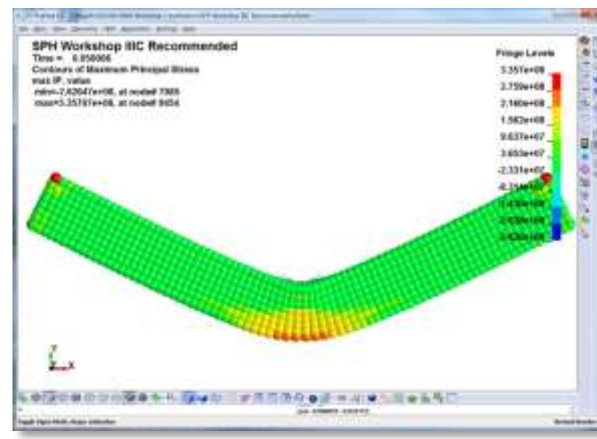
FEA Mesh Benchmark



SPH Default (FORM=0)

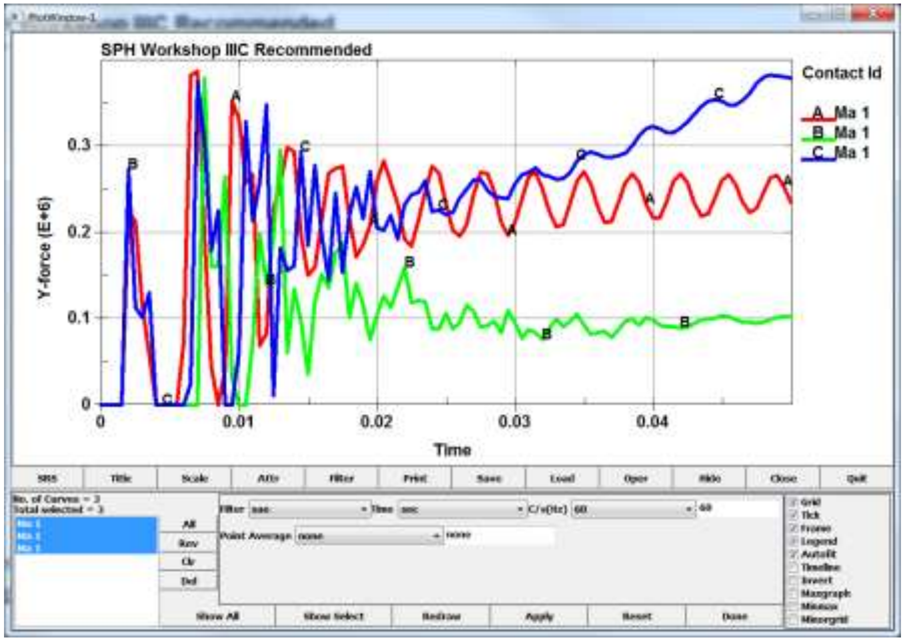


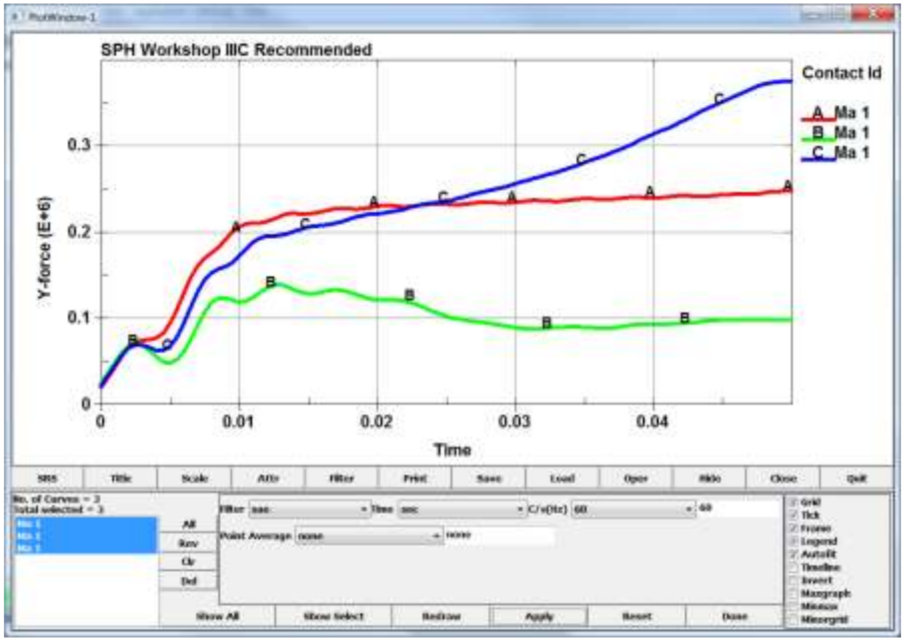
SPH (FORM=7)



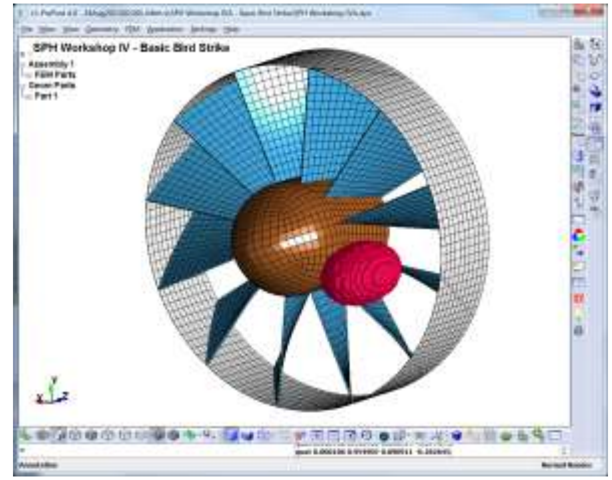
14.4.1 SPH Verification III: Workshop Bonus

At the end of the prior workshop, the student should have made graphs of the maximum principal stresses and contact forces between the FEM, SPH Tensile Instability and SPH Recommended. In particular, the contact force graph will have quite a bit of numerical noise. LSPP provides a Filter option to remove such noise. The student should explore this option since it is invaluable in the interpretation of dynamic results.





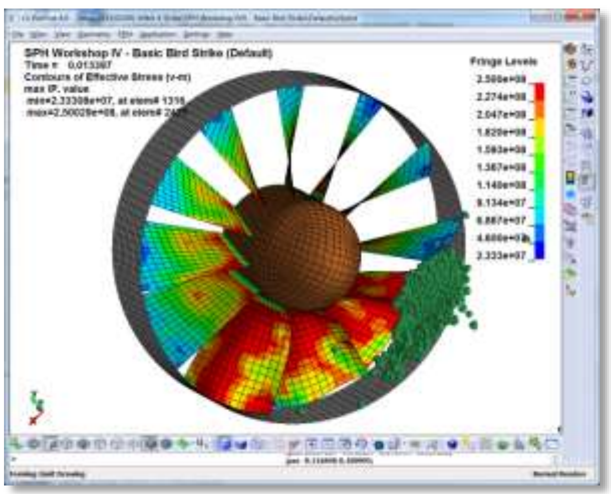
14.5 SPH Workshop IV: Bird Strike



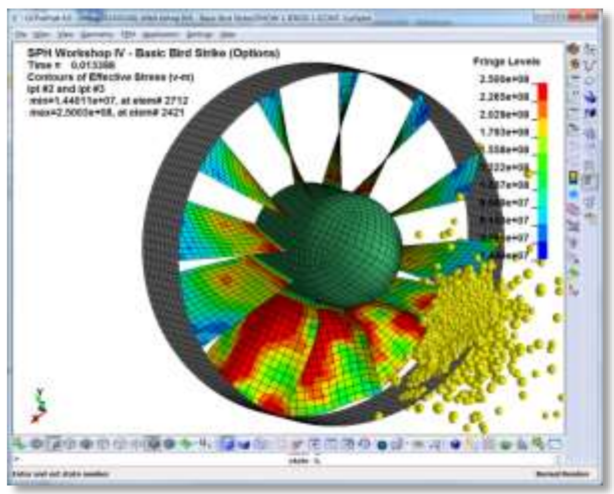
The Bird Strike workshop essentially leverages all the prior workshops and explores in a bit greater detail the *CONTROL_SPH control card options ISHOW, IEROD and ICONT to make the simulation more efficient. Lastly, the student is encouraged to set the fan blade failure criterion using *MAT_ADD_EROSION (MXEPS=0.05) while enforcing contact between the parts.

Hands-On Script: The procedure is to open the starting file (Workshops / SPH / SPH Workshop IV – Bird Strike / Default / SPH Workshiop IV – Default Start.dyn, inspect the model and then use this baseline model to create the other two simulations. This is an interactive instructor/student Workshop and questions should be asked if any of the operations don't feel right. Please note that file folders exist with the completed, final *.dyn model within the folder. Lastly, if one is bored or has time, change the failure criterion for the bird.

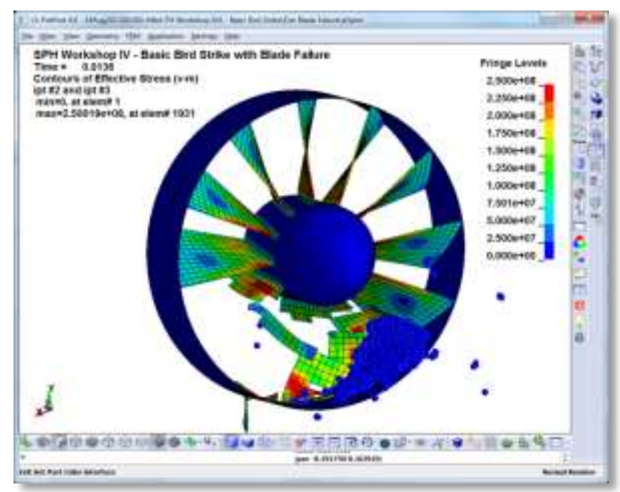
Defaults



ISHOW=1, IEROD=1 & ICONT=1



BLADE FAILURE



14.5.1 Bird Strike Models

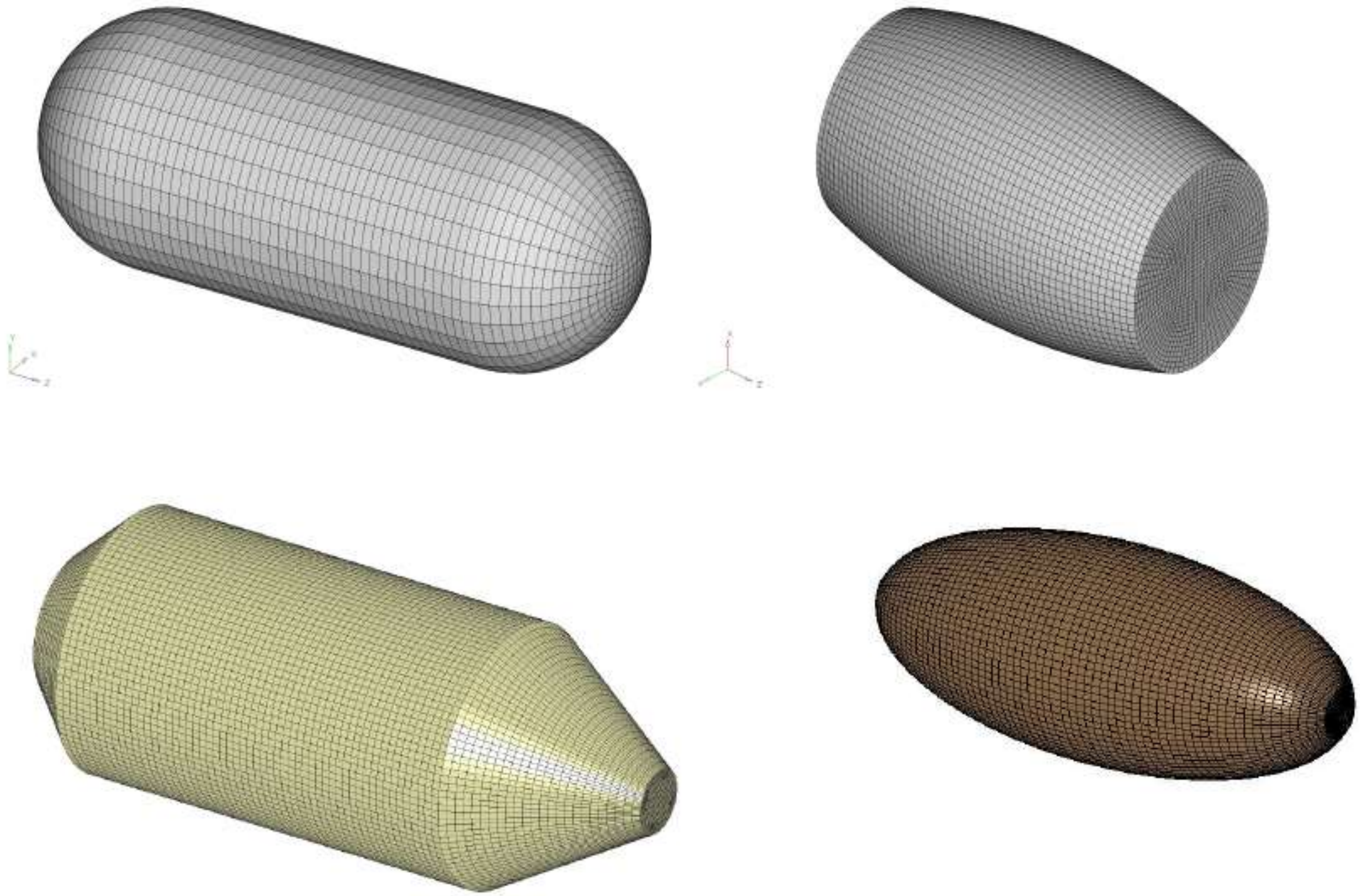


Figure 5.4 Various bird models used in rotating bird strike analyses.

If one ever wonders what a spherical chicken looks like. Image courtesy of Aerospace_MGD_v12-1.pdf, page 101 (see Class Reference Notes / Aerospace Working Group)

14.6 References

[1] L. D. Libersky, A. G. Petschek, T. C. Carney *et al.*, "High Strain Lagrangian Hydrodynamics: A Three-Dimensional SPH Code for Dynamic Material Response," *Journal of Computational Physics,* vol. 109, no. 1, pp. 67-75, 11//, 1993.

[2] J. L. Lacome, "Smooth Particle Hydrodynamics (SPH): A New Feature in LS-DYNA."

[3] G.-R. Liu, and M. B. Liu, *Smoothed particle hydrodynamics : a meshfree particle method*, Hackensack, New Jersey: World Scientific, 2003.

[4] D. Voileau, *Fluid Mechanics and the SPH Method: Theory and Applications*, Oxford, UK: Oxford University Press, 2012.

[5] W. G. Hoover, *Smooth Particle Applied Mechanics: The State of the Art (Advanced Series in Nonlinear Dynamics)*, Singapore: World Scientific Publishing, 2006.

[6] LSTC, "LS-DYNA Keywords User Manual Volume 1," no. Version 971, July 12, 2012, 2012.

[7] LSTC, "LS-DYNA Materials User Manual Volume 2," no. Version 971, July 12, 2012, 2012.

[8] S. Marrone, A. Colagrossi, D. Le Touzé *et al.*, "Fast free-surface detection and level-set function definition in SPH solvers," *Journal of Computational Physics,* vol. 229, no. 10, pp. 3652-3663, 2010.

15. EXPLICIT MODEL CHECK-OUT AND RECOMMENDATIONS

If you think you might have a simulation that is working, here's a short list of things to check for and review.

Here's an order of checking: Units | Mesh | d3hsp File | History Plots | Material Modeling | Contact Behavior | Etc.

15.1 Units

It is recommended to settle on one unit system for as much of your LS-DYNA work as possible to avoid unit problems when one is unfamiliar with a specific system. A commonly recommend system for dynamic events is the kN, mm, ms, kg system. Stresses are then in GPa. We have covered this before but it is hard to overstate the importance of getting your units straight. In a dynamic analysis, the mass of the system should always be checked.

15.2 Mesh

When looking at your mesh, it should look good and if it looks good, it will generate a smooth stress contour. This is never more so important than for an explicit analysis. If this sounds odd, please see Class Reference Notes / Stress Visualization / Desktop Engineering Stress Visualization Article March 2011.pdf.

Besides this Zen of meshing statement, here are some bulleted items to consider:

- Is the mesh density sufficient to capture the mechanical response? (Remember one-point Guass Integration)
- If Hourglassing is significant, remesh and likewise, if contact is poor (high SLEOUT), remesh, etc.
- Lastly, check the explicit time step. Seriously, a couple bad elements can completely explode the analysis (personal experience that cost me a weekend). This can be done easily via LSPP and viewing the D3hsp file under 100 smallest timesteps.

15.3 D3hsp File (LS-DYNA Equivalent to the Nastran F06 File)

- The *d3hsp* file summarizes the input in descriptive terms and can be viewed via a text editor or from within LSPP via Misc. and then D3hsp View.

    Within the d3hsp, one should review:

    - Verify mass of system (find "summary'')

- 100 elements listed in ascending order of time step (find "smallest")
- Review rigid bodies for any that might be deleted or with super-small mass
- Warning and error messages (find "Warning" or "Error")

15.4 History Plots

The GLSTAT file is your first stop for checking the analysis. A fundamental check is that your Energy Ratio should be 1.0 (+/- 0.01). This can be quickly checked within the ASCII, GLSTT file under Energy Ratio. An example of a more complex energy plot is shown on the right. The high Sliding Energy is because the model is simulating a burst containment of a fragmenting X-ray target with friction. For more information on Energy Data, see www.dynasupport.com under LS-DYNA User's Guides and then Energy Data also see Class Reference Notes / Energy Balance / Total energy LS DYNA Support.pdf

If the GLSTAT checks out, it is time to go over the MATSUM file via individual entries. The internal energy should be plotted against the hourglass energy and margins of 5% or less of the peak internal energy should be observed.

Sliding energy (GLSTAT - global) and sliding interface energy (ASCII – local via) say a lot about the numerical validity of your contacts. If friction is zero, than the GLSTAT value should be 1% or less of the internal energy. Interrogate local sliding energy by plotting all of the SLEOUT values. If they are negative greater than 5% of the peak internal energy, be worried and start digging.

15.5 Material Modeling Errors

Although much-ado is made about strain rate sensitivity, for most engineering applications, the only really strain rate sensitive materials are carbon based materials (rubbers, elastomers, plastics and foams) and to obtain such data is not that difficult from any good material testing laboratory. What is more common is just plain screwing up the material model. Thus, it is almost mandatory to demonstrate with a virtual coupon test model that one can match test data with the LS-DYNA model. Such correlation should be within every engineering report. Never underestimate the power of "KISS" and always attend to the basics before making your life more difficult.

15.6 Contact Options with Recommendations

SOFT=2	Soft=2 is really a quite good standard contact option
IGNORE=1	For handling small penetrations that are not worth the time to fix, this option is very useful. Please don't expect it to do wonders or correctly fundamental contact modeling problems
VDC=20	Contact is often noisy and adding 20% damping can be a nice option to smooth things out. It is one of those tweaks that is often times worth investigation once you have the model working.
ISYM=1	If you have symmetry faces via SPC's, please be aware of this option.

15.7 Control Cards with Recommendations

Commonly used options are shown in the table below

*CONTROL_BULK_VISCOSITY	TYPE=-2	For dynamic structural-only analyses, this option provides a mild smoothing of oscillations with little performance or energy cost.
*CONTROL_CONTACT	IGNORE=1	If you have small penetrations in your model, this option makes good sense but don't expect to help you with large modeling contact errors.
	ISYM=1	Symmetry Control
*CONTROL_ENERGY	HGEN=2 & SLNTEN=2	Calculates hourglass energy and sliding interface energy. Although it adds computational expense it should be added and checked and then one can delete it later.
*CONTROL_HOURGLASS	IHQ=4 & QH=0.05	Improved formulations and the default value of 0.1 can often be lowered if needed.
*CONTROL_SHELL	ESORT=1	Ensures highly twisted elements don't cause harm to your simulation
	NFAIL1=1 & NFAIL=1	Deletes highly distorted elements prior to them causing harm to your simulation
*CONTROL_SOLID	ESORT=1	Automatic sorting of tetrahedron and pentahedron elements to treat degenerate tetrahedron and pentahedron elements as tetrahedron (formulation 10). However, most LS-DYNA models don't have tetrahedrals but it is included for completeness.
*CONTROL_TIMESTEP	DT2MS (negative)	Just admit it, you'll going to use some mass scaling. If so, don't forget to contour the added mass to verify it didn't get to crazy (see below for recommendation on contouring added mass).
	ERODE=1	Just good practice when using solid elements. If the element

		becomes highly distorted to the point of a negative volume, it'll be deleted without killing the simulation.
*CONTROL_RIGID	PLOTEL=1 or 2	If one is using RB2 or 3 elements from a Nastran model or directly using CNRB's, this command will add plot only elements to the d3plot file and allow visualization of the elements.

15.8 Database Cards with Recommendations

*DATABASE_ASCII	GLSTAT, MATSUM, SLEOUT & SPCFORC	This is the minimum recommended set. Please note if you have rigidwalls in your simulation, you should also have RWFORC enabled.
*DATABASE_EXTENT_BINARY	BEAMIP (4) & STSSZ=3	Beam elements are isoparametric elements and have integration points, setting BEAMIP=4 covers the default beam formulation. **STSSZ=3** is to dump out added mass information and then also check MSSCL=1 or =2 to indicate incremental or percentage increase of added mass (my choice is MSSCL=2). This is very useful to check your model for mass scaling effects. To contour the added mass, see LSPP, Fcomp / Misc / Time Step Size (whereas in this case, it is the added mass). *Please note, it is really mandatory if you are going to be aggressive with your mass scaling to contour this item and be aware of where you are adding mass to your structure.*

15.9 Etc

I'm a big fan of building stupid, simple, itty-bitty test models to evaluate a proposed behavior. A standard downfall of many simulations is an attempt to model all the physics out-of-the-gate without prior evaluation of the effects of individual items, in brief, the more complex the model, the more heinous is the debugging.

16. IMPLICIT CHECK-OUT AND RECOMMENDATIONS

Implicit analyses are generally much more difficult to obtain convergence and that is often the downfall of its usage. To truly get spun up to speed on implicit, the student should start by reading (see Class Reference Notes / Implicit Analysis / LSTC Implicit Class Notes by Morten Jensen 2012 Draft.pdf).

For implicit analyses used for the initialization of explicit runs, it is recommended to use the *CONTACT_ birth option (BT) to turn off all non-essential contacts until the start of the explicit run. Likewise use *BOUNDARY_SPC_NODE (or SET)_BIRTH_DEATH to lock down any parts of the structure that are not relevant to the implicit initialization. These two steps will greatly facility a fast and efficient implicit kick-off.

For troubleshooting LS-DYNA implicit analyses don't be shy about locking down (SPC'ing) large parts of your structure and ripping out contacts and nonlinear material laws. Once you have something running, it is a lot easier to add in complexity step-by-step-by-step than struggle with behemoth that is taking 30 minutes to finally error out.

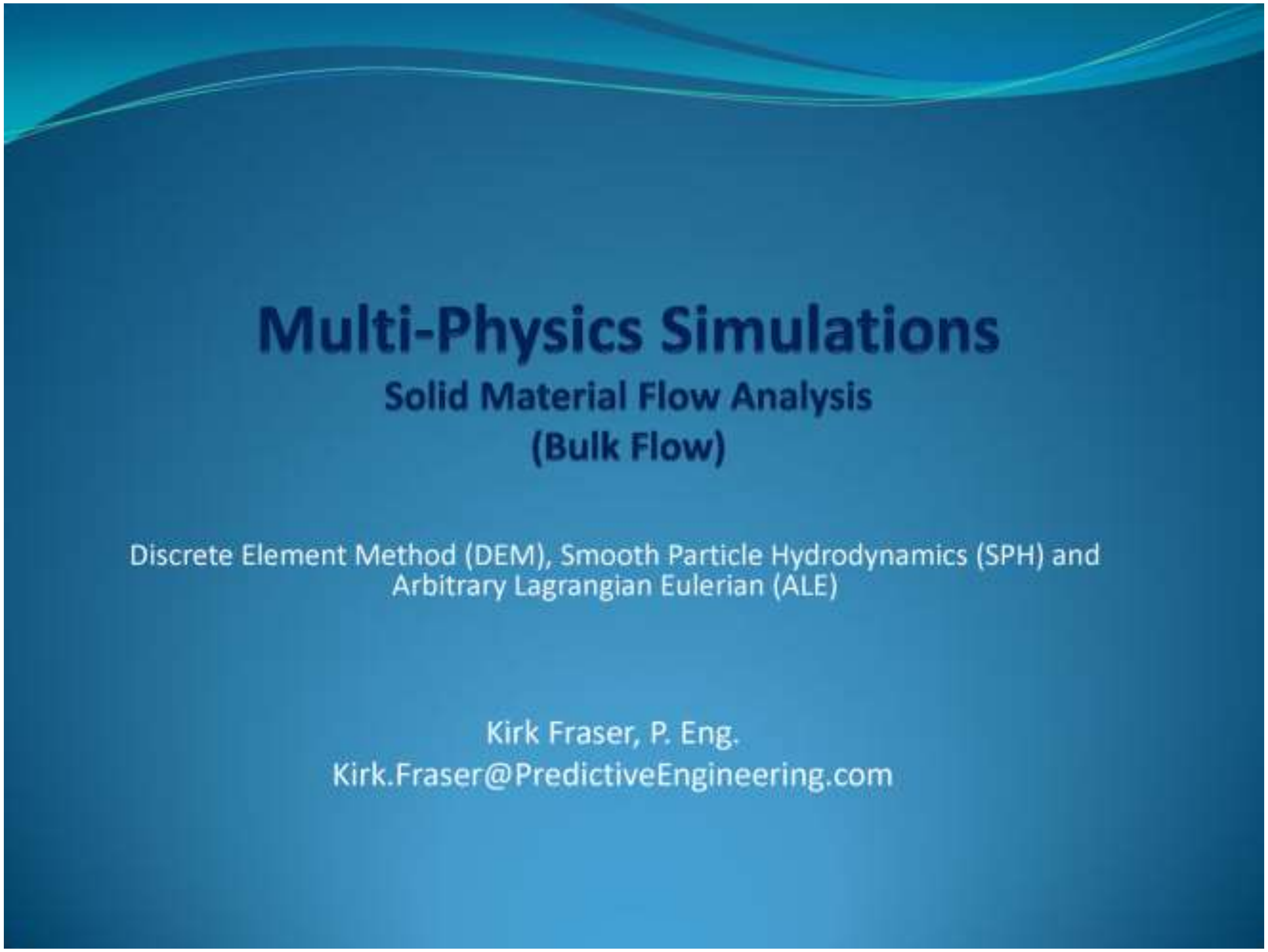
LS-DYNA also can automatically switch to explicit if the implicit solver is struggling to find convergence (see Class Reference Notes / Implicit Analysis / LSTC Implicit Class Notes by Morten Jensen 2012 Draft.pdf, page 42). It may sound attractive but don't really expect it to perform miracles since it most likely has to return to the implicit solution to finalize the run and most likely it would just fail to find convergence again (please note that most typical LS-DYNA runs are so highly nonlinear that often times it is a miracle that the implicit run even works).

A little thing to note is that LS-DYNA has near identical element formulations to that of standard implicit codes but as of this writing (Fall 2013), stresses are only provided at the centroid and integration points. Hence, one will note significant differences in stresses due to lack of Jacobian extrapolation from the integration points out to the nodal points. Of course, if your mesh is fine enough the differences will be small. A trick to overcome this limitation in solid meshes is to "skin" the solid with very thin layer of plate elements and then take your stresses from this layer. This technique is well known and leveraged within the implicit community for obtaining high-quality stresses for strain gauge and fatigue work.

16.1 Implicit Keyword Cards and Recommendations

*CONTACT_ -- _MORTAR	This algorithm was essentially developed for implicit analyses. It is super expensive for explicit and not recommended. However, keep in mind that with the BT and DT contact card option one can always turn it off and switch to another contact formation for the explicit run.	
*CONTROL_IMPLICIT_SOLVER	LSOLVR=6	BCSLIB-EXT (Boeing) solver that is recommended for implicit solutions for your standard plate, beam, CNRB, hex solid model.
	LSOLVR=10	For large tetrahedral models, the iterative solver performs well (just like Nastran).
*CONTROL_IMPLICIT_AUTO	IAUTO=1	Not a big fan of auto stepping. It will sometimes help but often is slower than just specifying a small DTO within the *CONTROL_IMPLICIT_SOLUTION card.
Recommended Only for Pure Implicit Analyses where Stress Recovery in the Elastic Range is Important Since for Explicit Analyses it Generates 4 to 8x larger D3Plot Files due to Integration Point Stresses		
*DATABASE_EXTENT_BINARY	MAXINT=-3	A negative number dumps out all integration point stresses.
	NINTSLD=8	This dumps out all the integration point stress data for solid elements. Very useful for implicit work if you would like to something that approaches a normal linear stress result.
LSPP (LS-PrePost) v4.1	Extrapolate -1	This command will correctly extrapolate the integration point stresses for ELFORM=20 (shells) ELFORM=18 (solids) and aligns well with Nastran-type element results (**with MAXINT=-3**). Note: For shells one can use -1 or 1, only for solids is -1 required.

17. DISCRETE ELEMENT METHOD

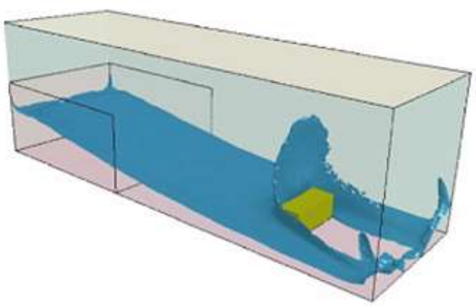


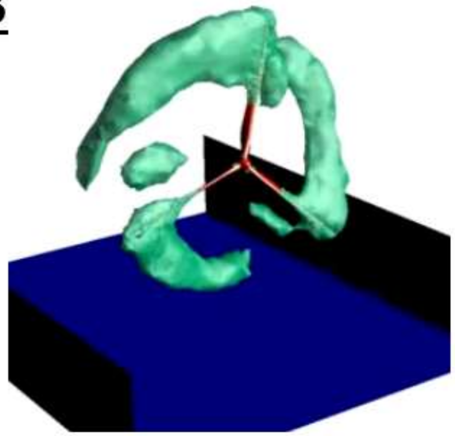
See Class Reference Notes / DEM / Predictive Engineering Discussion of LS-DYNA Meshfree Methods.pptx

18. FLUID STRUCTURE INTERACTION AND MULTI-PHYSICS IN LS-DYNA

Fluid Structure Interaction with LS-DYNA Multiphysics

August 2013





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

19. LS-DYNA RECENT DEVELOPMENTS

See Class Reference Notes / LS-DYNA Recent Developments / LSTC_2012_Conference_Recent Developments.pdf

END

End of LS-DYNA Analysis for Structural Mechanics Class Notes

Your comments would be welcomed

On a scale of 1 to 5, where “1” means not satisfactory and a “5” indicates that it was very satisfactory.

How were the class notes and the workshops?	1 2 3 4 5
Did the instructor do a good job in presenting the material?	1 2 3 4 5
Was the pace of the class adequate to learn the material?	1 2 3 4 5
Quality of the experience?	1 2 3 4 5

If you could do one or two things to make it better, what would they be?

General Comments?

When done just tear out this sheet and leave it at your desk. Thank you.