FINITE ELEMENT ANALYSIS
Predictive Engineering

LS-DYNA® Handbook

Analysis Theory and Techniques for Structural Mechanics

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

ANSYS LST

ANSYS CERTIFIED CHANNEL PARTNER

Acknowledgements

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

Technical Support Team at the ANSYS Livermore Software and Technology (LST)
With special mention to:
Satish Pathy, LST
Jim Day, LST
Todd Slavic, LST
Ushnish Basu, LST
Philip Ho, LST

And the invaluable team at:
DYNAmore, Gmbh, Germany
And in particular, Dr. T Borrvall, DYNAmore Nordic AB, Linköping, Sweden

Trademarks:

Disclaimer:

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

Special Acknowledgement:

These class notes stand on the shoulders of the outstanding team at the ANSYS LST and their colleagues who have provided suggestions and support through years of collaboration. I am sincerely indebted to them for their help and of course, any errors or omissions are due to me.

COURSE OUTLINE

1. INTRODUCTION

1.1 What The Student Can Expect

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

1.2 What we cover

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

1.3 How we do it

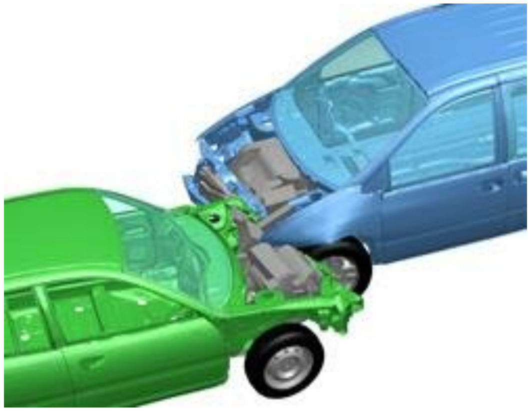
- The class covers the basics in a hands-on manner as taught by engineers that has had to live by what they have validated.
- Each day (four hour session) will have three to four Workshops. Each Workshop is part theory, part demonstration and part hands-on practice. Videos are provided for most Workshops thereby 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.
- A breaks is provided mid-way where students can pause, stretch and perhaps ask the instructor more detailed questions that might not be appropriate to involve the full class.
- Students are encouraged to turn off their email, text messaging and other forms of digital/social media during class time.

1.4 How To Be Successful with as a LS-DYNA Simulation Engineer (top-of-the-pack)

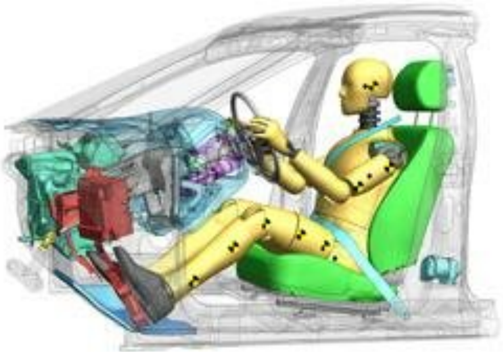
- You are already way ahead of the pack by simply attending this class. You have started on the journey of how to be more successful with LS-DYNA. It is this simple. To be successful, as far as we know, it requires:
    - Reading (very traditional but with LS-DYNA it is necessary to read the manual (RTM),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

1.5 General Applications

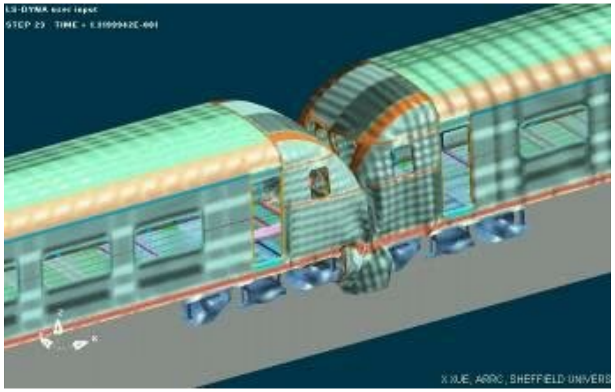
Crashworthiness



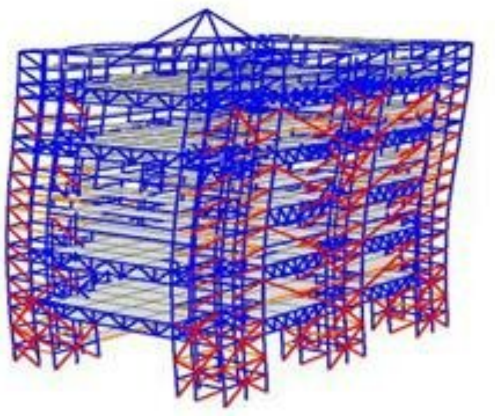
Driver Impact



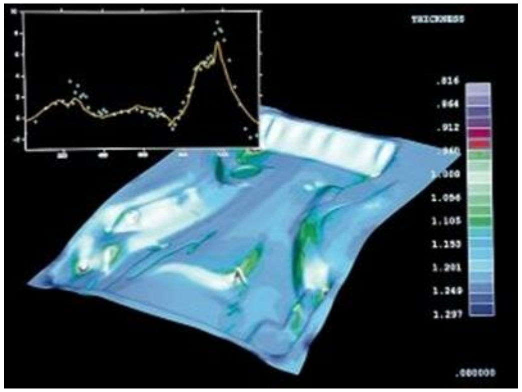
Train Collisions



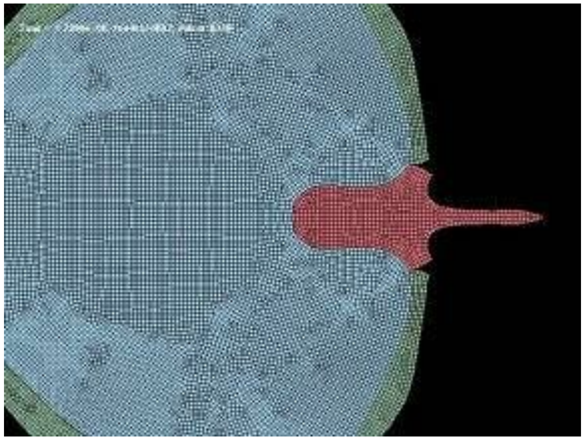
Earthquake Engineering



Metal Forming

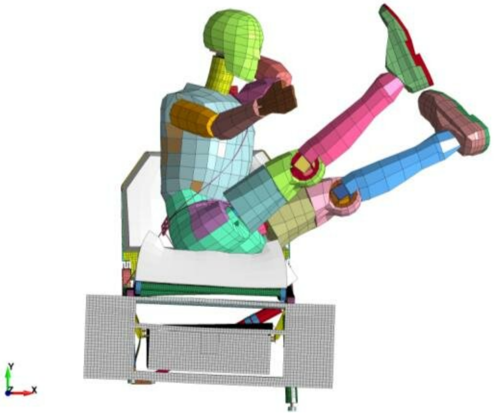


Military

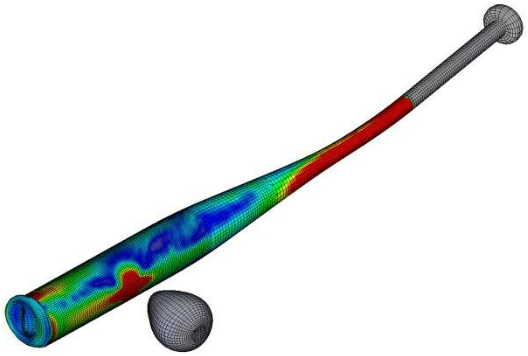


1.6 Specific Applications (Courtesy of Predictive Engineering)

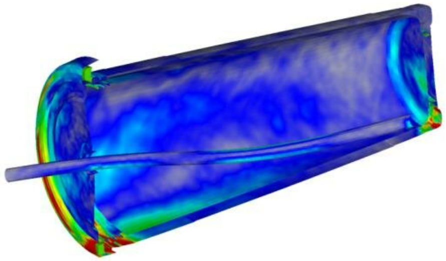
CS-FAR 25 16g Sled Analysis



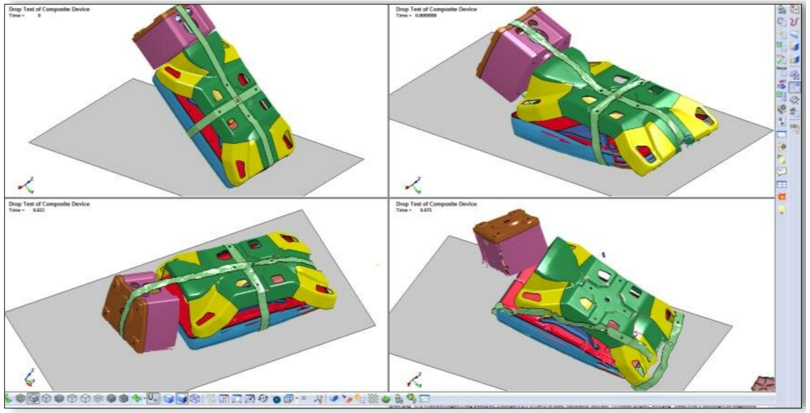
Sporting Goods Equipment



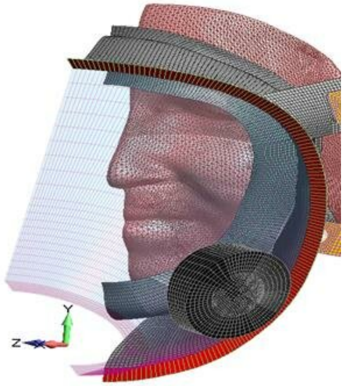
Drop Test Consumer Products



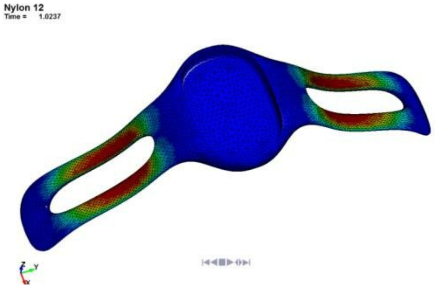
Drop Test of Composites / Electronics


Drop Test of Composite Device
Time = 0
Drop Test of Composite Device
Time = 0.0089998
Drop Test of Composite Device
Time = 0.021
Drop Test of Composite Device
Time = 0.075

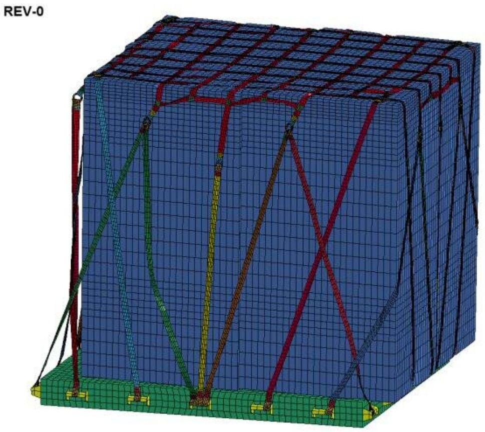
Human Biometrics



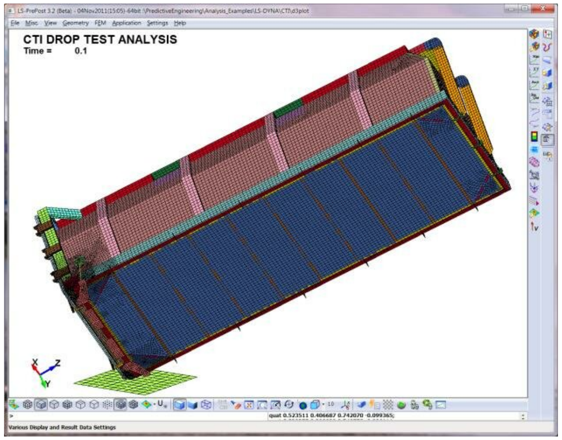
Large Deformation of Plastics



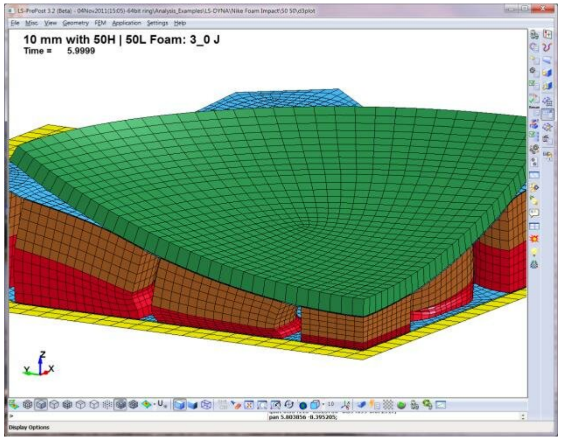
Crash Analysis of Cargo Net



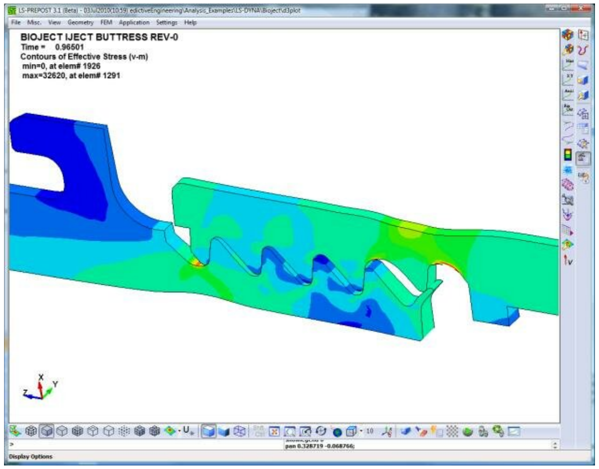
Drop Test of Nuclear Waste Container



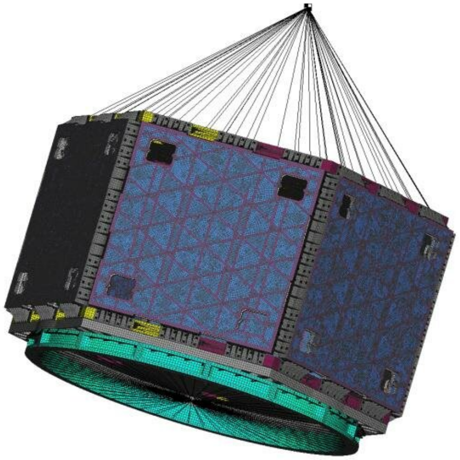
Impact Analysis of Foams



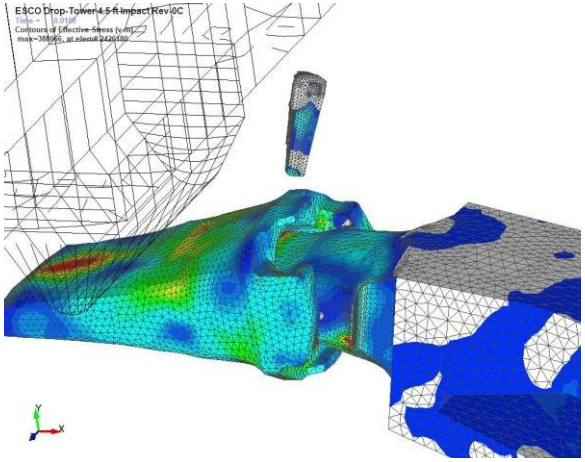
Plastic Thread Design


BIOJECT IJECT BUTTRESS REV-0
Time = 0.96501
Contours of Effective Stress (v-m)
min=0, at elem# 1926
max=32620, at elem# 1291

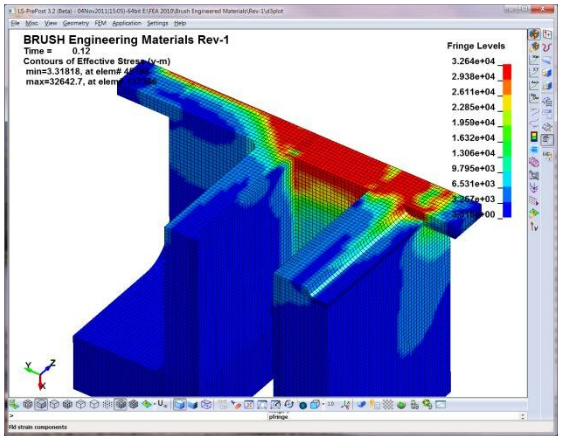
PSD / Modal Analysis



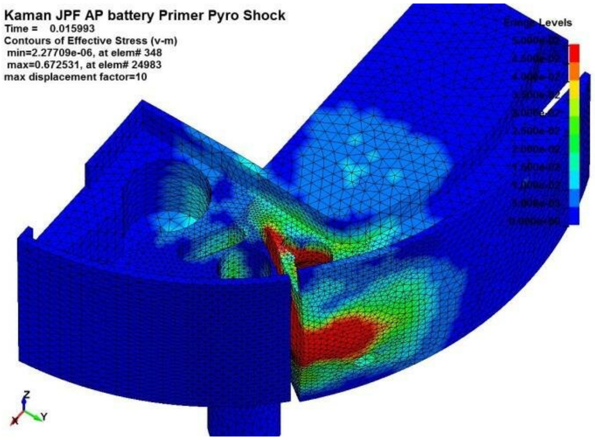
Digger Tooth Failure


ESCO Drop-Tower 4.5 ft Impact Rev-0C
Time = 0.0198
Contours of Effective Stress (v-m)
max=388066, at elem# 2426180

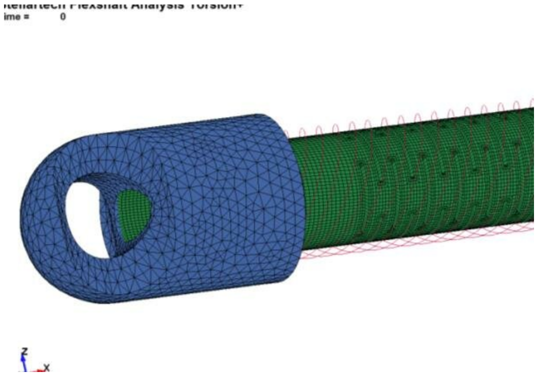
Electron Beam Welding


LS-PrePost 3.2 (Beta) - 04Nov2011(15:05)-64bit E:\FEA 2010\Brush Engineered Materials\Rev-1\d3plot
File Misc View Geometry FEM Application Settings Help
BRUSH Engineering Materials Rev-1
Time = 0.12
Contours of Effective Stress (v-m)
min=3.31818, at elem# 4
max=32642.7, at elem
Fringe Levels
3.264e+04
2.938e+04
2.611e+04
2.285e+04
1.959e+04
1.632e+04
1.306e+04
9.795e+03
6.531e+03
3.267e+03
fringe
Fld strain components

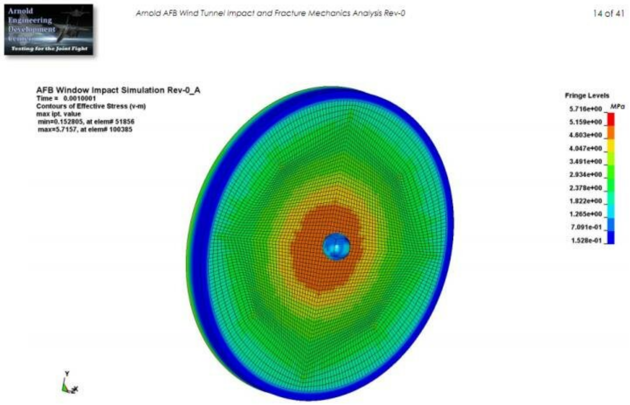
Pyro-Shock Analysis


Kaman JPF AP battery Primer Pyro Shock
Time = 0.015993
Contours of Effective Stress (v-m)
min=2.27709e-06, at elem# 348
max=0.672531, at elem# 24983
max displacement factor=10
Fringe Levels

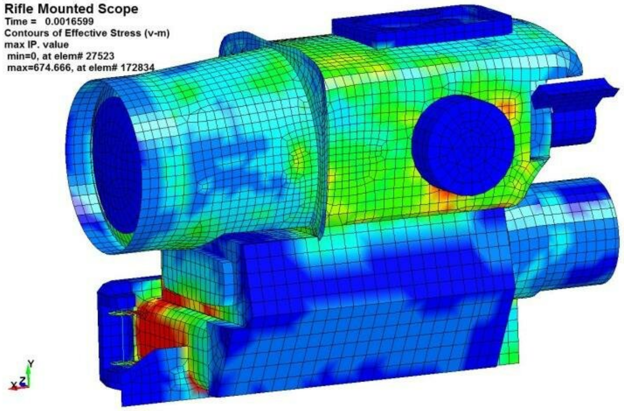
Medical Equipment



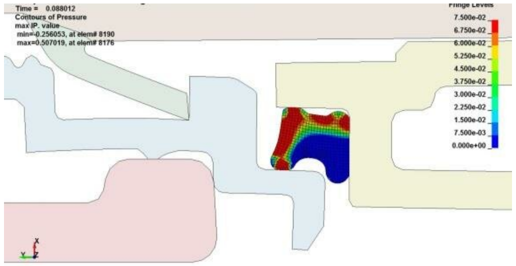
Fracture Mechanics of Glass


Arnold Engineering Development Center
Testing for the Joint Fight
Arnold AFB Wind Tunnel Impact and Fracture Mechanics Analysis Rev-0
14 of 41
AFB Window Impact Simulation Rev-0_A
Time = 0.0010001
Contours of Effective Stress (v-m)
max ipt. value
min=0.152805, at elem# 51856
max=5.7157, at elem# 100385
Fringe Levels
MPa
5.716e+00
5.159e+00
4.603e+00
4.047e+00
3.491e+00
2.934e+00
2.378e+00
1.822e+00
1.265e+00
7.091e-01
1.528e-01

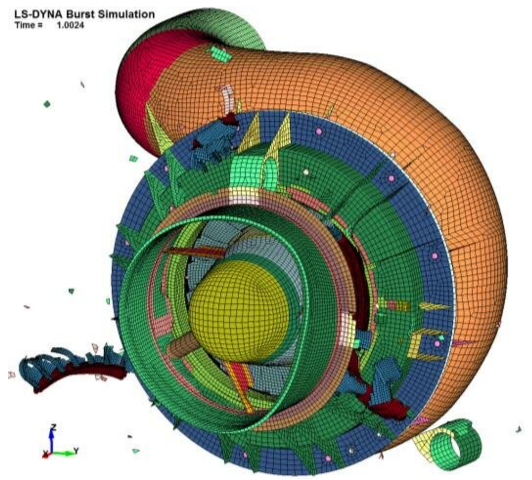
Ballistic Shock Loading of Optical Equipment



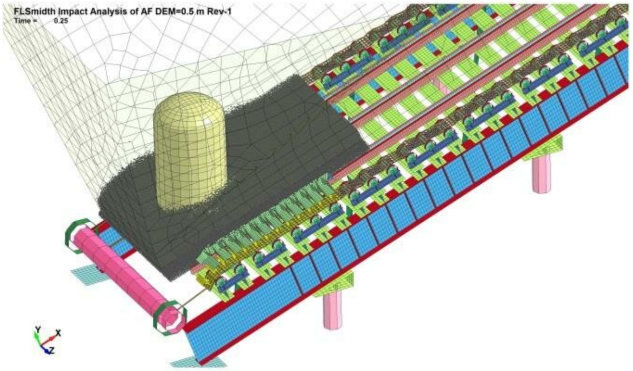
Hyperelastic Medical Seal Analysis



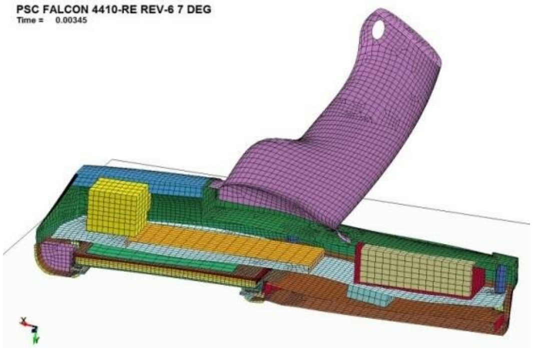
Blade-Out Analysis



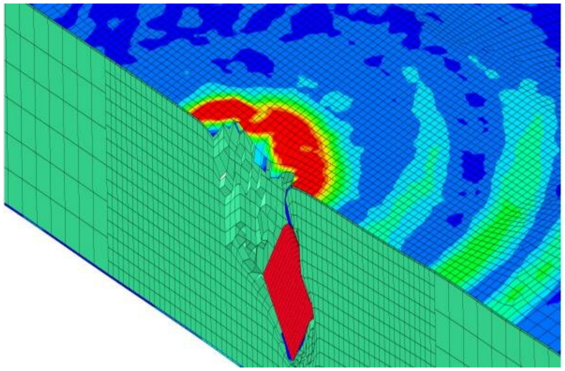
Discrete Element Method for the Mining Industry



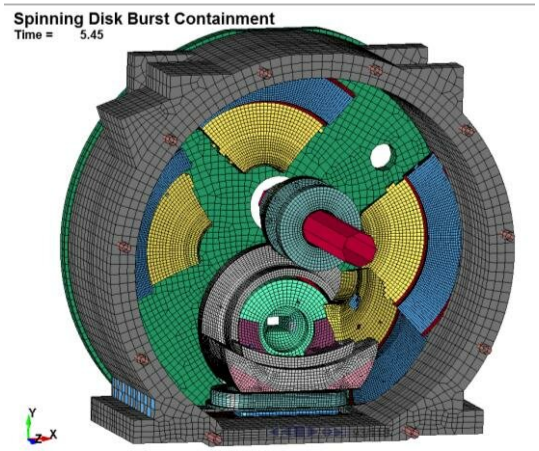
Drop-Test of Handheld Electronics



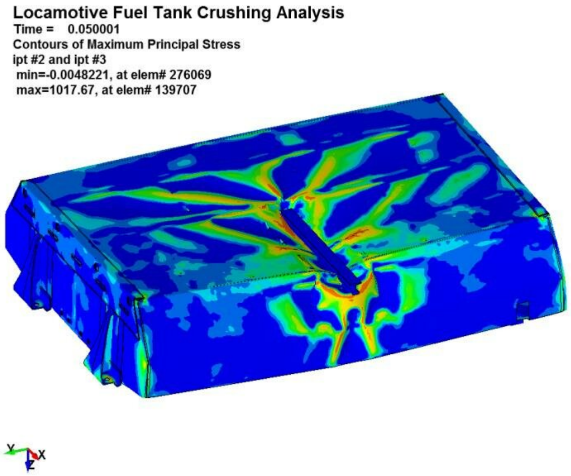
Ballistic Penetration



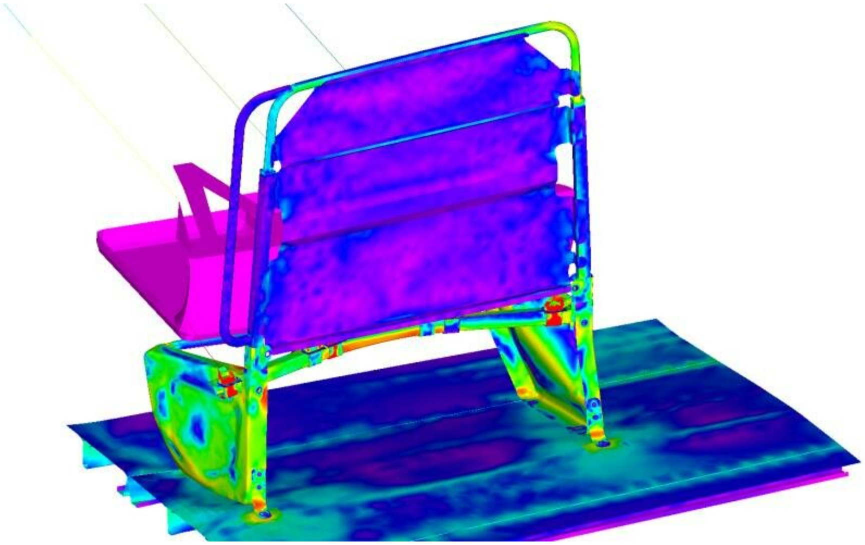
High-Speed Spinning Disk Containment



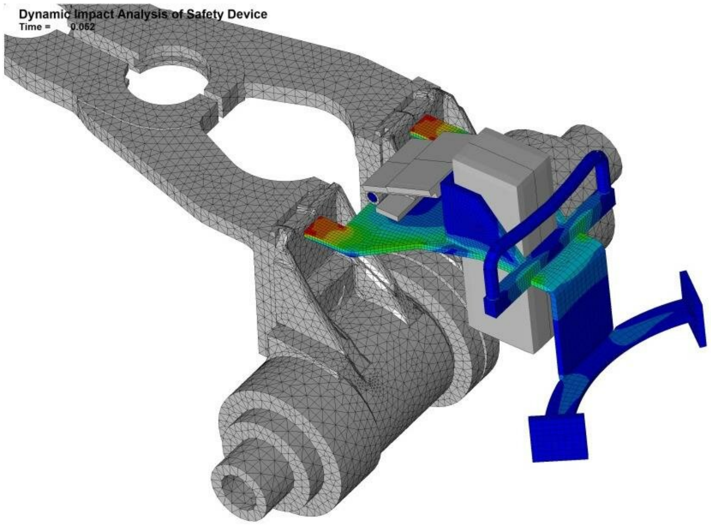
Locomotive Fuel Tank


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

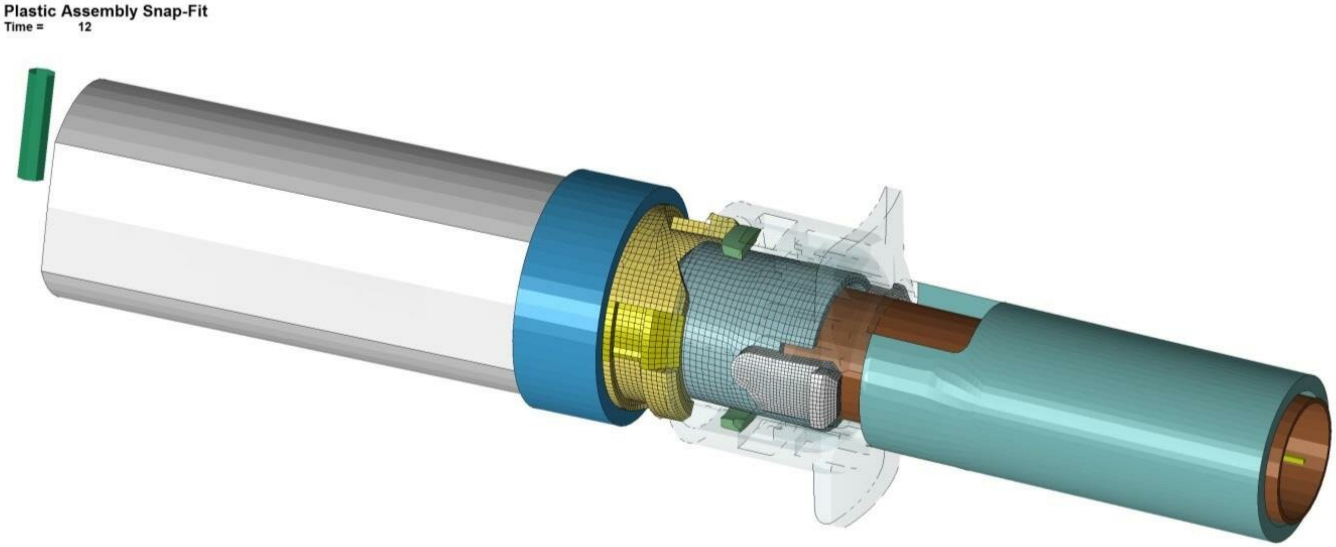
FMVSS Virtual Testing of Bus Seats



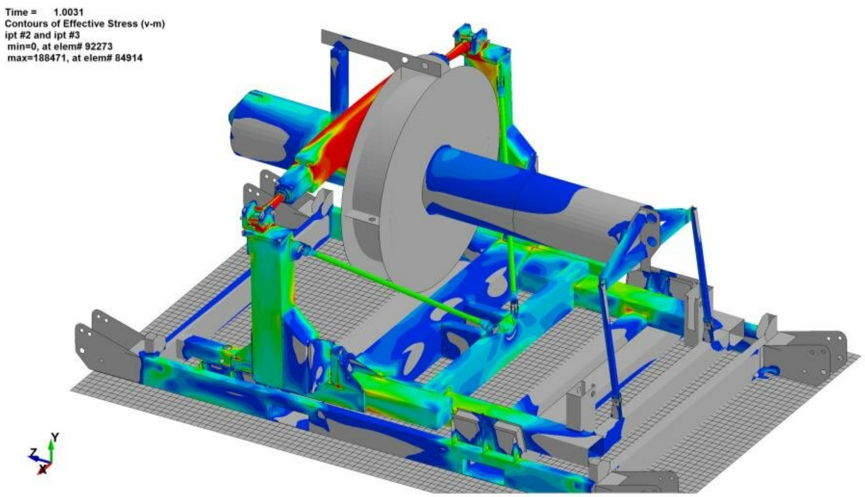
Impact Analysis of Safety Block Device



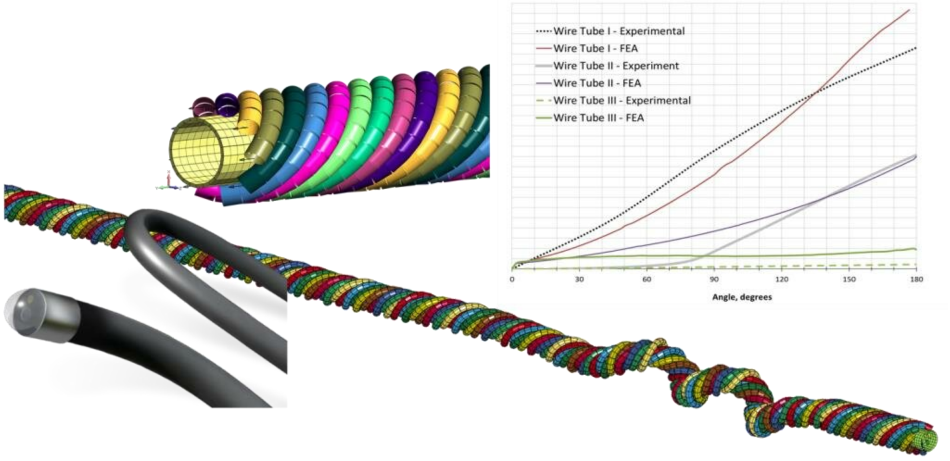
Snap-Fit Analysis – All Plastic Medical Device



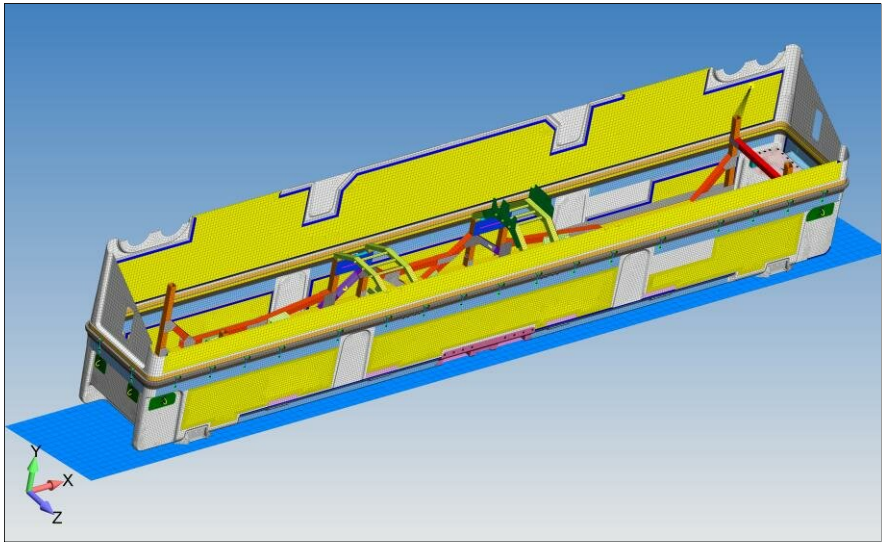
9g Crash Analysis of Jet Engine Stand

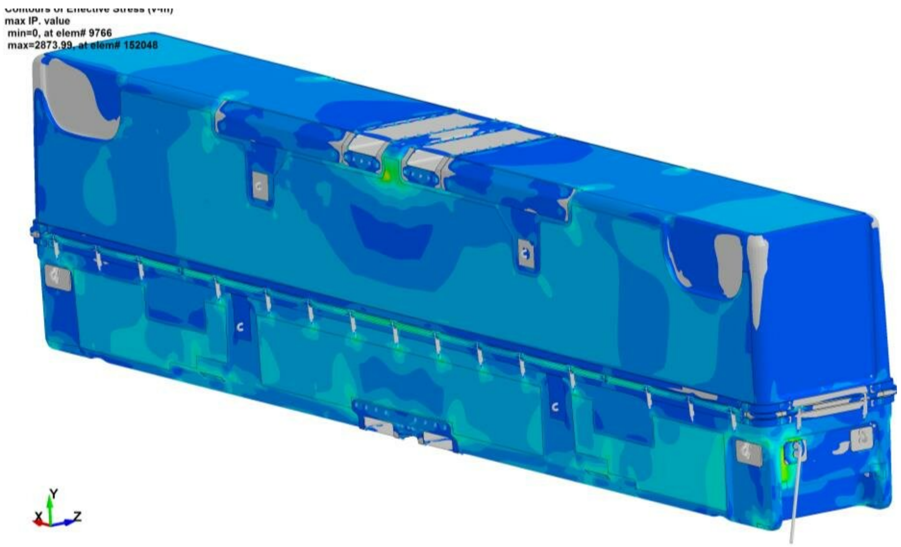

Time = 1.0031
Contours of Effective Stress (v-m)
ipt #2 and ipt #3
min=0, at elem# 92273
max=188471, at elem# 84914
Y
Z
X

Torque Analysis of Endoscopic Medical Device

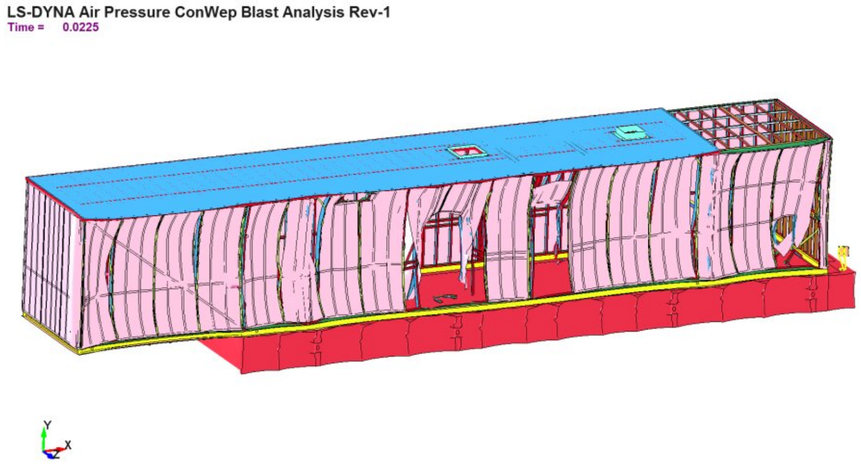


Drop, Rail Impact and PSD Analysis of Composite Container

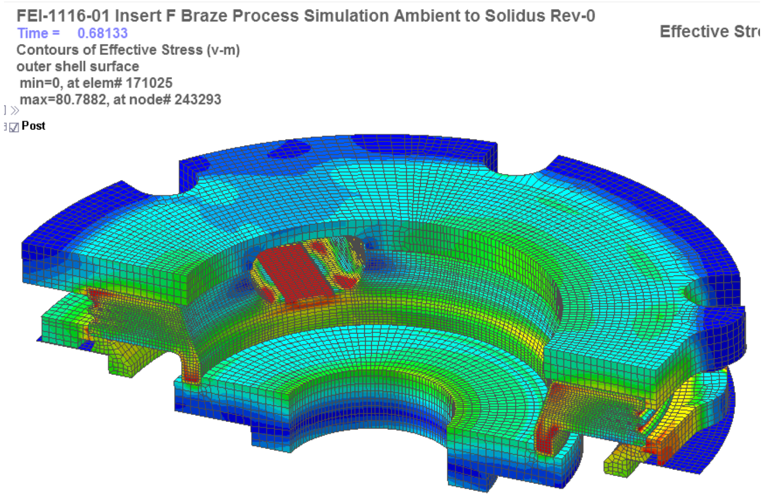



Contours of Effective Stress (v-m)
max IP. value
min=0, at elem# 9766
max=2873.99, at elem# 152048
Y
X
Z

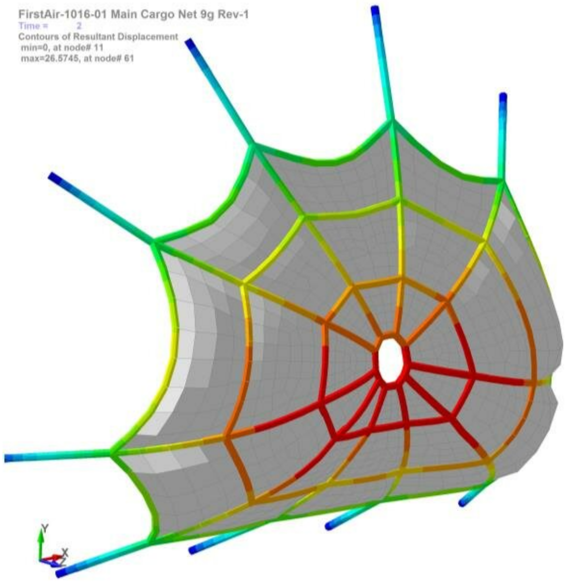
ConWep Air Pressure Blast Analysis of Generator Housing



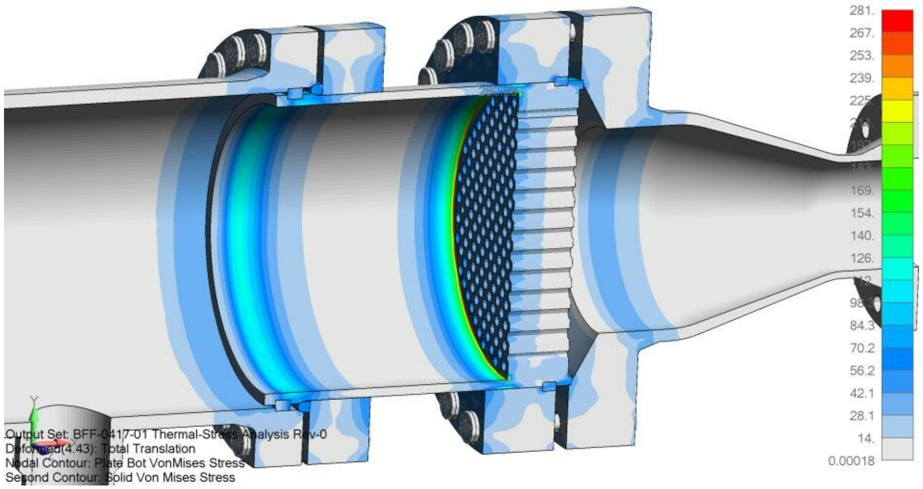
Alumina-Stainless Steel Braze Process Simulation


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

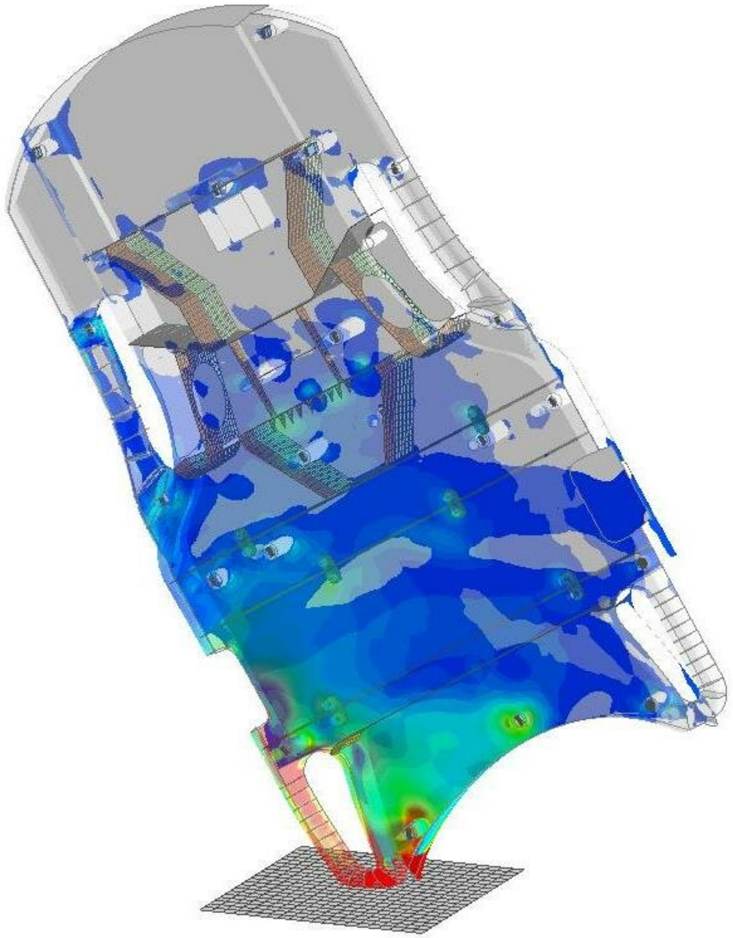
Air Freighter 9g Cargo Net Analysis


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

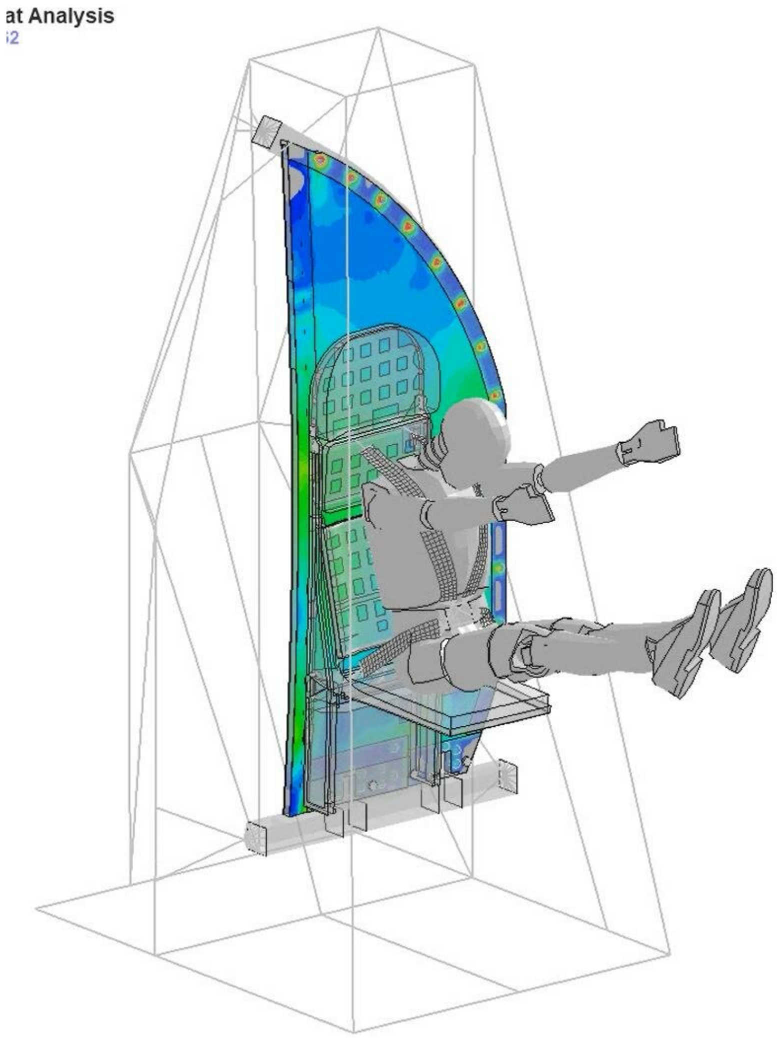
Thermal-Stress Fatigue Analysis of ASME Evaporator Vessel


281.
267.
253.
239.
225
169.
154.
140.
126.
98
84.3
70.2
56.2
42.1
28.1
14.
0.00018
Output Set: BFF-0417-01 Thermal-Stress Analysis Rev-0
Deformed(4.43): Total Translation
Nodal Contour: Plate Bot VonMises Stress
Second Contour: Solid Von Mises Stress

Drop-Test of First-Responder Medical Equipment



Composite Lavatory Wall with Attendant Seat 16g Sled Test



2. WHAT IS LS-DYNA?

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

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

2.1 How We Visualize the LS-DYNA Analysis Process

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

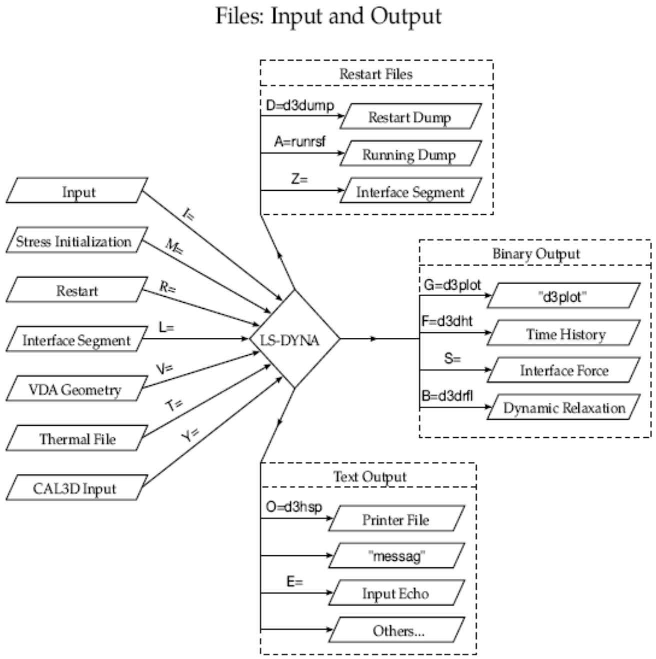


Figure 2-2. Files Input and Output.

LSPP

Pre-Processor
(Nodes, Elements, Etc.)

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

Pre-Processing to Create Keyword Deck

LS-DYNA

Post-Processing FEA Results

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

LSPP

Proprietary Software

3. IMPLICIT VERSUS EXPLICIT ANALYSIS

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

3.1 WHAT WE ARE SOLVING

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

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

where n=time step. A common terminology is to call the kd^n part the internal force in the structure. The basic problem is to determine the displacement at some future time or d^{n+1}, at time t^{n+1}. However, this is where it gets interesting, explicit is based on acceleration whereas implicit is displacement.

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

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

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

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

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

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

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

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

3.2 Explicit (dynamic) – One Must have "Mass" to make it go

Internal and external forces are summed at each node point, and a nodal acceleration is computed by dividing by nodal mass. The solution is advanced by integrating this acceleration in time. The maximum time step size is limited by the Courant-Friedrichs-Lewy (CFL) criterion (to be discussed). For now let's say that the solution marches forward in time using a fixed time step that is calculated based on the element size and the speed of sound in the material (i.e., CFL). Much more will be said about element size and the speed of sound in materials since execution speed for an explicit analysis is often of great importance given that careful meshing can mean the difference between a run time of days or hours. Just to keep this theme in the forefront of our discussion: an explicit analysis is all about mass since everything has a time step (e.g., contact, 1D spring elements, CNRB's, etc.).

3.3 Implicit (dynamic or Static)

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

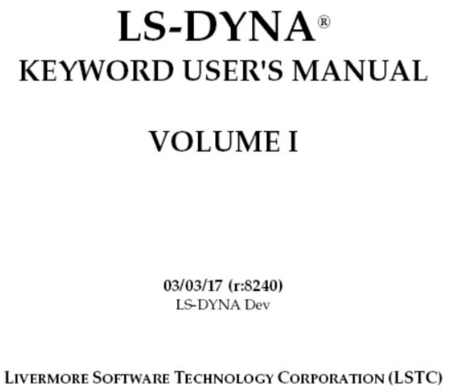
3.3.1 PROS AND CONS OF EXPLICIT V IMPLICIT

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

4. LS-DYNA GETTING STARTED WITH THE FUNDAMENTALS

4.1 LS-DYNA KEYWORD MANUAL

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



LS-DYNA®

KEYWORD USER'S MANUAL

VOLUME I

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

LIVERMORE SOFTWARE TECHNOLOGY CORPORATION (LSTC)

Analyst's Note: Please keep in mind that LS-DYNA is an analysis engine that runs off of an ascii deck (a text file) and that oftentimes the fastest path to an optimum solution is to edit the deck. It took me years to embrace the "deck" and I'm better for it.

4.2 KEYWORD SYNTAX

- Commands are strings of words separated by an underscore, e.g., *BOUNDARY_PRESCRIBED_MOTION_RIGID.
- Text can be uppercase or lowercase
- Commands are arranged alphabetically in User's Manual
- Order of commands in input deck is ***mostly*** unimportant (except *KEYWORD, *DEFINE_TABLE (but then one can use *DEFINE_TABLE_2D if this is a problem), *INCLUDE_TRANSFORM, ?)
- Keyword command must be left justified, starting with an asterisk
- A "**$**" in the first column indicates a comment
- If one would like to screen print out comments, use *COMMENT
- Input values (card data) can be *anywhere within* **fixed fields or/and comma-delimited** (Although one will notice that I like to right-justify values within fixed fields but it is not necessary.)
- A blank parameter indicates that the default value of the parameter will be used (or taken from *CONTROL_*option*)
- Please keep in mind that *every* Keyword starts with "*" and that each line below the Keyword is a "card" per the LST-ANSYS Keyword Manual.

Required Commands:

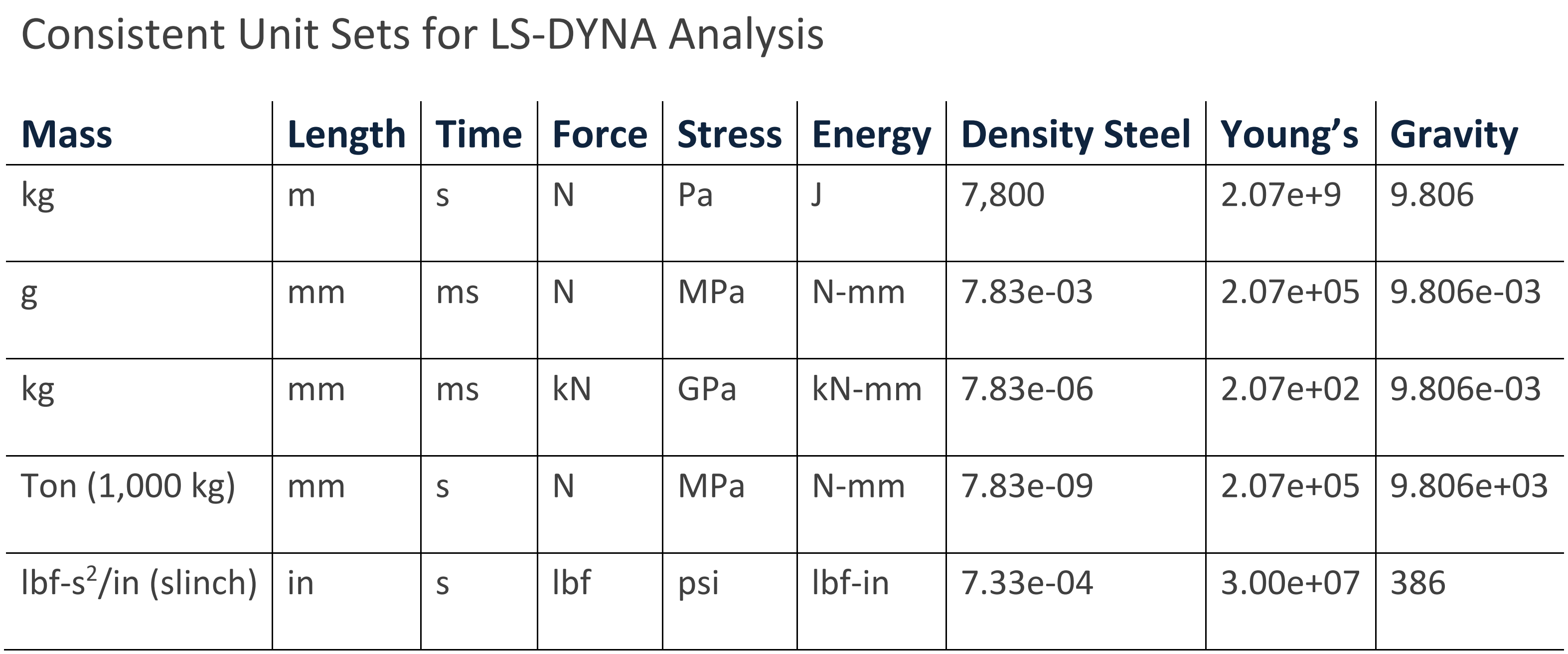
*KEYWORD
*CONTROL_TERMINATION
*NODE
*ELEMENT
*SECTION
*MAT
*PART
*DATABASE_BINARY_D3PLOT
*END

*Analyst's Note: Want more Keyword information – read Appendix V: How to Read Card Summaries. This Appendix explains the philosphy behind the *KEYWORD structure and its syntex. It should be required reading for any 'DYNA addict.*

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 five unit systems that I have standardized on for analysis work. It doesn't mean they are the best but at least they are generally accepted.



Consistent Unit Sets for LS-DYNA Analysis

Mass	Length	Time	Force	Stress	Energy	Density Steel	Young's	Gravity
kg	m	s	N	Pa	J	7,800	2.07e+9	9.806
g	mm	ms	N	MPa	N-mm	7.83e-03	2.07e+05	9.806e-03
kg	mm	ms	kN	GPa	kN-mm	7.83e-06	2.07e+02	9.806e-03
Ton (1,000 kg)	mm	s	N	MPa	N-mm	7.83e-09	2.07e+05	9.806e+03
lbf-s^2/in (slinch)	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**
Newsletter and Seminars: **www.DYNAmore.com**
Yahoo Discussion Group: **LS-DYNA@yahoogroups.com**
Aerospace Working Group: **awg.lstc.com**
Varmit Al's Material Database (google'it)
Ed Wilson's Blog: http://www.edwilson.org/History/1Library.htm

LST-ANSYS Program Download Site

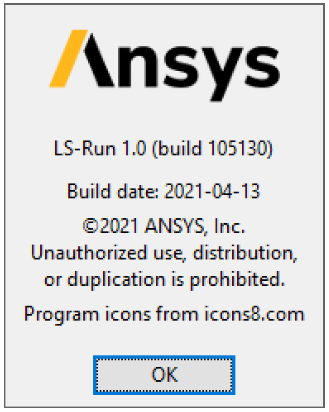
https://ftp.lstc.com/user Username: user Password: computer
SMP Version: ls-dyna
MPP Version: mpp-dyna
SMP/Windows: pc-dyna

4.5 Submitting LS-DYNA Analysis Jobs with LS-Run

LS-Run is a LS-DYNA job tool that allows you to run jobs using multiple solvers and to queue jobs up for multiple runs. It can be downloaded the LST (see above) site.

Analysts' Notes: MPP (Massively Parallel-Processing or SMP (Symmetric Multi-Processing) depends on the number of CPU-Cores. SMP is faster using eight or fewer cores while MPP's sweet spot is eight and above CPU-Cores. In general, we recommend using the Double-Precision version of the code for initial analysis work although it can be anywhere from 10 to 20% slower; however, double-precision is required for the implicit solver.

See Class Reference Notes / MPP versus SMP / for additional discussion and scaling information.

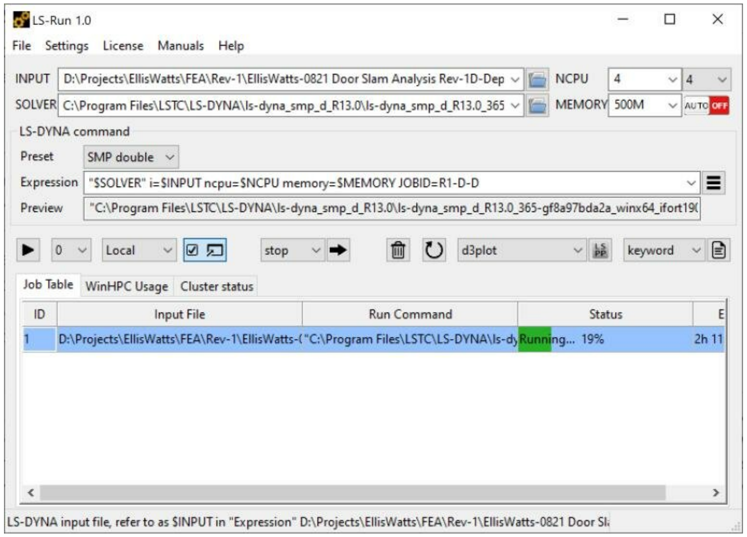

Ansys

LS-Run 1.0 (build 105130)

Build date: 2021-04-13

©2021 ANSYS, Inc.
Unauthorized use, distribution, or duplication is prohibited.

Program icons from icons8.com



INPUT D:\Projects\EllisWatts\FEA\Rev-1\EllisWatts-0821 Door Slam Analysis Rev-1D-Dep
NCPU 4 4
SOLVER C:\Program Files\LSTC\LS-DYNA\ls-dyna_smp_d_R13.0\ls-dyna_smp_d_R13.0_365
MEMORY 500M
Preset SMP double
Expression "$SOLVER" i=$INPUT ncpu=$NCPU memory=$MEMORY JOBID=R1-D-D
Preview "C:\Program Files\LSTC\LS-DYNA\ls-dyna_smp_d_R13.0\ls-dyna_smp_d_R13.0_365-gf8a97bda2a_winx64_ifort19(

▶ 0 Local ☑ stop ➡ d3plot keyword

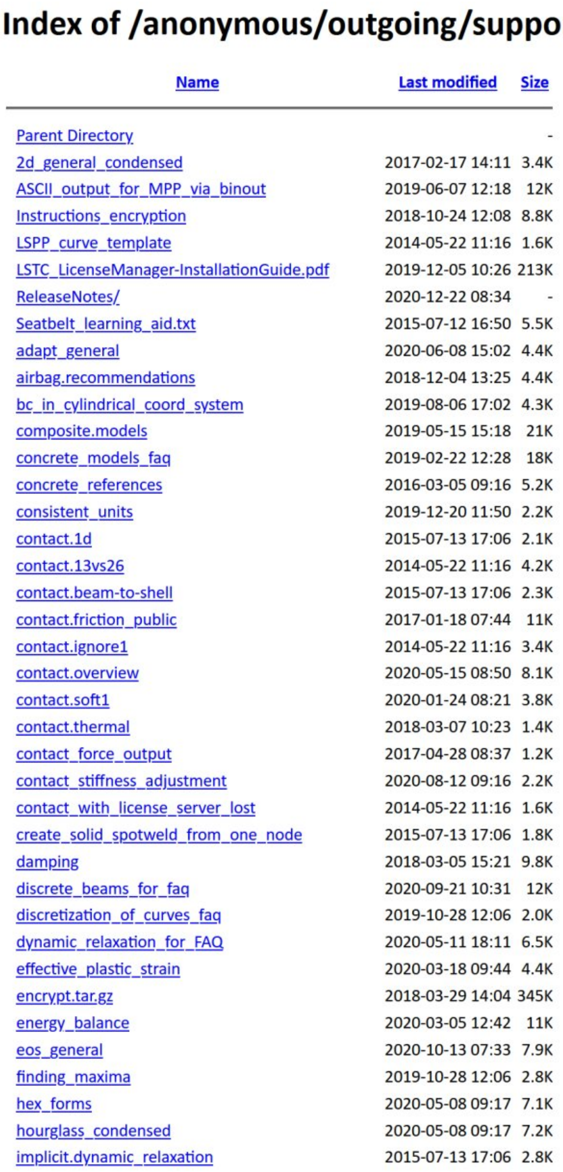
ID	Input File	Run Command	Status	E
1	D:\Projects\EllisWatts\FEA\Rev-1\EllisWatts-(	"C:\Program Files\LSTC\LS-DYNA\ls-dy	Running... 19%	2h 11

LS-DYNA input file, refer to as $INPUT in "Expression" D:\Projects\EllisWatts\FEA\Rev-1\EllisWatts-0821 Door Sl

Attention Windows Users: *LS-DYNA does not handle spaces in file folder names or regular keyword deck names. A simple workaround is to enable 8dot3name on your systems harddisks. This is done by opening up a CMD window (run as Administrator) and typing fsutil 8dot3name set 0 to enable the creation of 8dot3name'ing convention on all drives. With this setting, long file folder names and file names are truncated to 8 characters with a 3 character extension. Please note that after getting Windows setup for 8dot3name, one may have to copy file folders onto another drive and then copy them back to allow Windows to assign the 8dot3name "mask".*

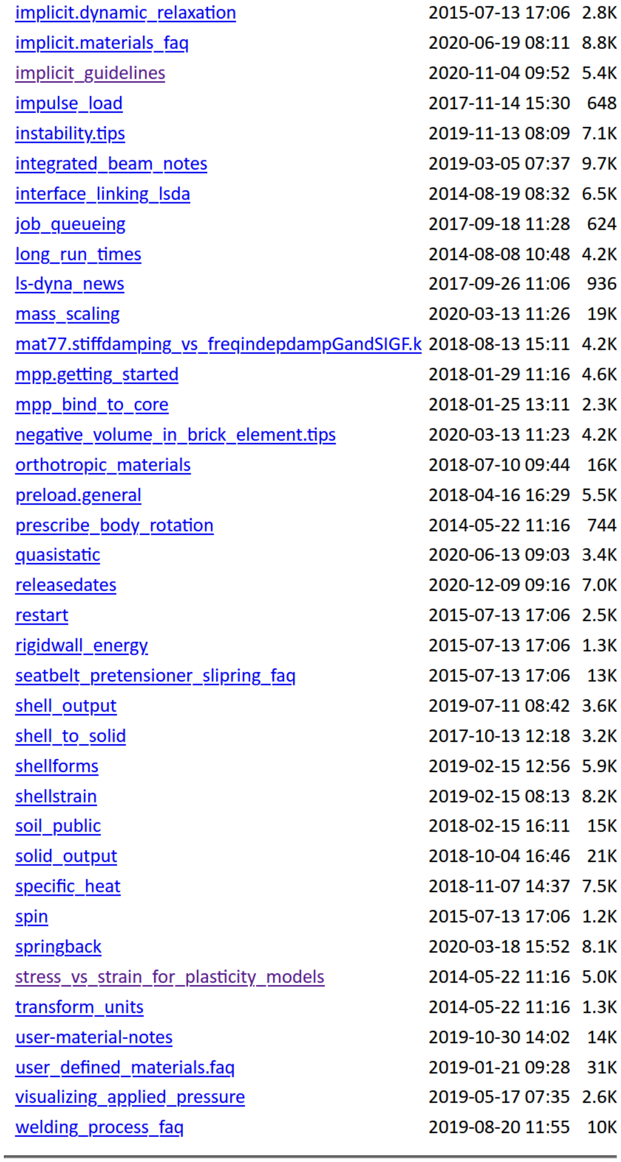
4.5.1 INTERNAL LST FAQ - HTTPS://FTP.LSTC.COM/ANONYMOUS/OUTGOING/SUPPORT/FAQ/

This is the most outstanding information and is a recommended "desktop-reference" for those inquisitive simulation engineers. Please note that this link was valid as of 02/10/2021 and may be removed at the discretion of ANSYS LST – no guarantee is provided by Predictive Engineering, Inc.



Index of /anonymous/outgoing/support/FAQ

Name	Last modified	Size
Parent Directory		-
2d_general_condensed	2017-02-17 14:11	3.4K
ASCII_output_for_MPP_via_binout	2019-06-07 12:18	12K
Instructions_encryption	2018-10-24 12:08	8.8K
LSPP_curve_template	2014-05-22 11:16	1.6K
LSTC_LicenseManager-InstallationGuide.pdf	2019-12-05 10:26	213K
ReleaseNotes/	2020-12-22 08:34	-
Seatbelt_learning_aid.txt	2015-07-12 16:50	5.5K
adapt_general	2020-06-08 15:02	4.4K
airbag.recommendations	2018-12-04 13:25	4.4K
bc_in_cylindrical_coord_system	2019-08-06 17:02	4.3K
composite.models	2019-05-15 15:18	21K
concrete_models_faq	2019-02-22 12:28	18K
concrete_references	2016-03-05 09:16	5.2K
consistent_units	2019-12-20 11:50	2.2K
contact.1d	2015-07-13 17:06	2.1K
contact.13vs26	2014-05-22 11:16	4.2K
contact.beam-to-shell	2015-07-13 17:06	2.3K
contact.friction_public	2017-01-18 07:44	11K
contact.ignore1	2014-05-22 11:16	3.4K
contact.overview	2020-05-15 08:50	8.1K
contact.soft1	2020-01-24 08:21	3.8K
contact.thermal	2018-03-07 10:23	1.4K
contact_force_output	2017-04-28 08:37	1.2K
contact_stiffness_adjustment	2020-08-12 09:16	2.2K
contact_with_license_server_lost	2014-05-22 11:16	1.6K
create_solid_spotweld_from_one_node	2015-07-13 17:06	1.8K
damping	2018-03-05 15:21	9.8K
discrete_beams_for_faq	2020-09-21 10:31	12K
discretization_of_curves_faq	2019-10-28 12:06	2.0K
dynamic_relaxation_for_FAQ	2020-05-11 18:11	6.5K
effective_plastic_strain	2020-03-18 09:44	4.4K
encrypt.tar.gz	2018-03-29 14:04	345K
energy_balance	2020-03-05 12:42	11K
eos_general	2020-10-13 07:33	7.9K
finding_maxima	2019-10-28 12:06	2.8K
hex_forms	2020-05-08 09:17	7.1K
hourglass_condensed	2020-05-08 09:17	7.2K
implicit.dynamic_relaxation	2015-07-13 17:06	2.8K



implicit.dynamic_relaxation	2015-07-13 17:06	2.8K
implicit.materials_faq	2020-06-19 08:11	8.8K
implicit_guidelines	2020-11-04 09:52	5.4K
impulse_load	2017-11-14 15:30	648
instability.tips	2019-11-13 08:09	7.1K
integrated_beam_notes	2019-03-05 07:37	9.7K
interface_linking_lsda	2014-08-19 08:32	6.5K
job_queueing	2017-09-18 11:28	624
long_run_times	2014-08-08 10:48	4.2K
ls-dyna_news	2017-09-26 11:06	936
mass_scaling	2020-03-13 11:26	19K
mat77.stiffdamping_vs_freqindepdampGandSIGF.k	2018-08-13 15:11	4.2K
mpp.getting_started	2018-01-29 11:16	4.6K
mpp_bind_to_core	2018-01-25 13:11	2.3K
negative_volume_in_brick_element.tips	2020-03-13 11:23	4.2K
orthotropic_materials	2018-07-10 09:44	16K
preload.general	2018-04-16 16:29	5.5K
prescribe_body_rotation	2014-05-22 11:16	744
quasistatic	2020-06-13 09:03	3.4K
releasedates	2020-12-09 09:16	7.0K
restart	2015-07-13 17:06	2.5K
rigidwall_energy	2015-07-13 17:06	1.3K
seatbelt_pretensioner_slipring_faq	2015-07-13 17:06	13K
shell_output	2019-07-11 08:42	3.6K
shell_to_solid	2017-10-13 12:18	3.2K
shellforms	2019-02-15 12:56	5.9K
shellstrain	2019-02-15 08:13	8.2K
soil_public	2018-02-15 16:11	15K
solid_output	2018-10-04 16:46	21K
specific_heat	2018-11-07 14:37	7.5K
spin	2015-07-13 17:06	1.2K
springback	2020-03-18 15:52	8.1K
stress_vs_strain_for_plasticity_models	2014-05-22 11:16	5.0K
transform_units	2014-05-22 11:16	1.3K
user-material-notes	2019-10-30 14:02	14K
user_defined_materials.faq	2019-01-21 09:28	31K
visualizing_applied_pressure	2019-05-17 07:35	2.6K
welding_process_faq	2019-08-20 11:55	10K

4.6 LS-DNA Output Files (Results and Message Files) and Database Requests and Management

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

Results and Message Files

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

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

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

4.7 Workshop: 1A - LS-DYNA Getting Started – Common Keyword Deck Format Errors

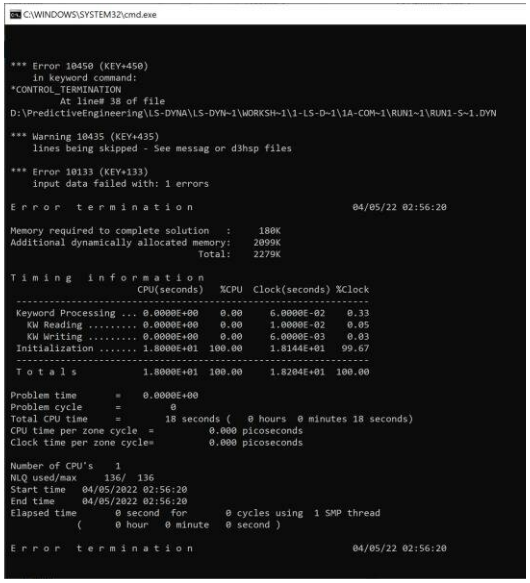
Objective: We will be working directly with the LS-DYNA Keyword deck and if the format is not "as required" – it'll bark at you with cryptic messages.

Introduction to LS-Run: These little workshops aims to get you a bit relaxed about working with Keyword Decks and using LS-Run. The runs listed on the right are prepared Decks where one runs'em, read the error messages and then correct the decks to run correctly to "Normal Termination".

What To Be Aware of About LS-DYNA Keyword Deck Formating

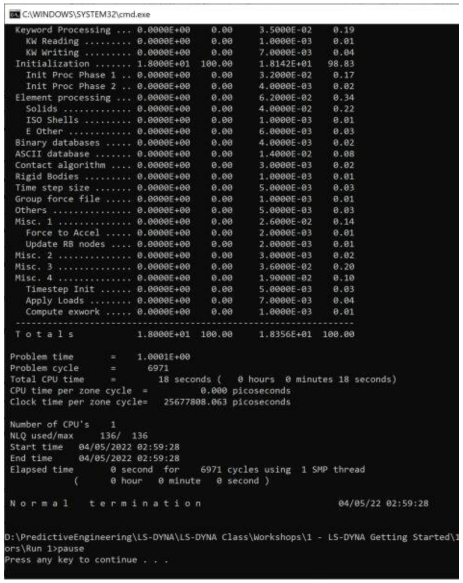
- Run 1 - Start: Extra space before "*"{Keyword}
- Run 2 – Start: Data formatting error – Improperly Formatted Data
- Run 3 – Start: Data entry error – Two values within one Keyword field

Run 1 - Start



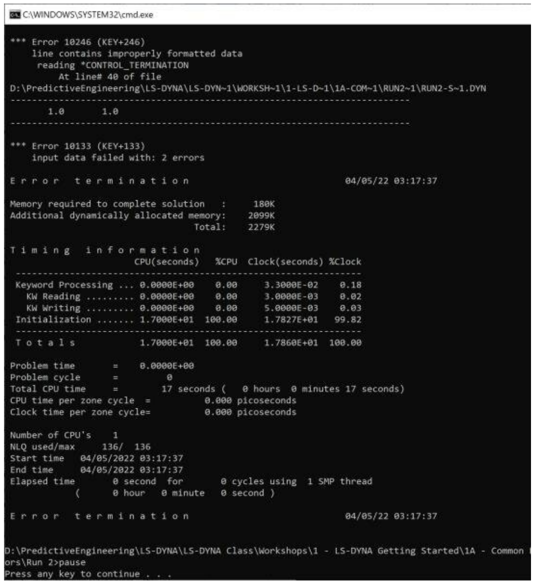
```
*** Error 10450 (KEY+450)
    in keyword command:
*CONTROL_TERMINATION
         At line# 38 of file
D:\PredictiveEngineering\LS-DYNA\LS-DYN~1\WORKSH~1\1-LS-D~1\1A-COM~1\RUN1~1\RUN1-S~1.DYN

*** Warning 10435 (KEY+435)
    lines being skipped - See messag or d3hsp files

*** Error 10133 (KEY+133)
    input data failed with: 1 errors

E r r o r   t e r m i n a t i o n                            04/05/22 02:56:20

Memory required to complete solution   :     180K
Additional dynamically allocated memory:    2099K
                                  Total:    2279K

T i m i n g   i n f o r m a t i o n
                       CPU(seconds)   %CPU  Clock(seconds) %Clock
 ----------------------------------------------------------------
 Keyword Processing ... 0.0000E+00    0.00     6.0000E-02    0.33
   KW Reading ......... 0.0000E+00    0.00     1.0000E-02    0.05
   KW Writing ......... 0.0000E+00    0.00     6.0000E-03    0.03
 Initialization ....... 1.8000E+01  100.00     1.8144E+01   99.67
 ----------------------------------------------------------------
 T o t a l s            1.8000E+01  100.00     1.8204E+01  100.00

Problem time       =    0.0000E+00
Problem cycle      =         0
Total CPU time     =        18 seconds (   0 hours  0 minutes 18 seconds)
CPU time per zone cycle  =          0.000 picoseconds
Clock time per zone cycle=          0.000 picoseconds

Number of CPU's    1
NLQ used/max     136/  136
Start time   04/05/2022 02:56:20
End time     04/05/2022 02:56:20
Elapsed time       0 second  for       0 cycles using  1 SMP thread
            (      0 hour   0 minute   0 second )

E r r o r   t e r m i n a t i o n                            04/05/22 02:56:20
```

Run 1 - Finished



```
 Keyword Processing ... 0.0000E+00    0.00     3.5000E-02    0.19
   KW Reading ......... 0.0000E+00    0.00     1.0000E-03    0.01
   KW Writing ......... 0.0000E+00    0.00     7.0000E-03    0.04
 Initialization ....... 1.8000E+01  100.00     1.8142E+01   98.83
   Init Proc Phase 1 .. 0.0000E+00    0.00     3.2000E-02    0.17
   Init Proc Phase 2 .. 0.0000E+00    0.00     4.0000E-03    0.02
 Element processing ... 0.0000E+00    0.00     6.2000E-02    0.34
   Solids ............. 0.0000E+00    0.00     4.0000E-02    0.22
   ISO Shells ......... 0.0000E+00    0.00     1.0000E-03    0.01
   E Other ............ 0.0000E+00    0.00     6.0000E-03    0.03
 Binary databases ..... 0.0000E+00    0.00     4.0000E-03    0.02
 ASCII database ....... 0.0000E+00    0.00     1.4000E-02    0.08
 Contact algorithm .... 0.0000E+00    0.00     3.0000E-03    0.02
 Rigid Bodies ......... 0.0000E+00    0.00     1.0000E-03    0.01
 Time step size ....... 0.0000E+00    0.00     5.0000E-03    0.03
 Group force file ..... 0.0000E+00    0.00     1.0000E-03    0.01
 Others ............... 0.0000E+00    0.00     5.0000E-03    0.03
 Misc. 1 .............. 0.0000E+00    0.00     2.6000E-02    0.14
   Force to Accel ..... 0.0000E+00    0.00     2.0000E-03    0.01
   Update RB nodes .... 0.0000E+00    0.00     2.0000E-03    0.01
 Misc. 2 .............. 0.0000E+00    0.00     3.0000E-03    0.02
 Misc. 3 .............. 0.0000E+00    0.00     3.6000E-02    0.20
 Misc. 4 .............. 0.0000E+00    0.00     1.9000E-02    0.10
   Timestep Init ...... 0.0000E+00    0.00     5.0000E-03    0.03
   Apply Loads ........ 0.0000E+00    0.00     7.0000E-03    0.04
   Compute exwork ..... 0.0000E+00    0.00     1.0000E-03    0.01
 ----------------------------------------------------------------
 T o t a l s            1.8000E+01  100.00     1.8356E+01  100.00

Problem time       =    1.0001E+00
Problem cycle      =      6971
Total CPU time     =        18 seconds (   0 hours  0 minutes 18 seconds)
CPU time per zone cycle  =          0.000 picoseconds
Clock time per zone cycle=   25677808.063 picoseconds

Number of CPU's    1
NLQ used/max     136/  136
Start time   04/05/2022 02:59:28
End time     04/05/2022 02:59:28
Elapsed time       0 second  for    6971 cycles using  1 SMP thread
            (      0 hour   0 minute   0 second )

N o r m a l    t e r m i n a t i o n                         04/05/22 02:59:28


D:\PredictiveEngineering\LS-DYNA\LS-DYNA Class\Workshops\1 - LS-DYNA Getting Started\1
ors\Run 1>pause
Press any key to continue . . .
```

Run 2 - Start



```
*** Error 10246 (KEY+246)
    line contains improperly formatted data
     reading *CONTROL_TERMINATION
         At line# 40 of file
D:\PredictiveEngineering\LS-DYNA\LS-DYN~1\WORKSH~1\1-LS-D~1\1A-COM~1\RUN2~1\RUN2-S~1.DYN
--------------------------------------------------------------------------
       1.0       1.0
--------------------------------------------------------------------------

*** Error 10133 (KEY+133)
    input data failed with: 2 errors

E r r o r   t e r m i n a t i o n                            04/05/22 03:17:37

Memory required to complete solution   :     180K
Additional dynamically allocated memory:    2099K
                                  Total:    2279K

T i m i n g   i n f o r m a t i o n
                       CPU(seconds)   %CPU  Clock(seconds) %Clock
 ----------------------------------------------------------------
 Keyword Processing ... 0.0000E+00    0.00     3.3000E-02    0.18
   KW Reading ......... 0.0000E+00    0.00     3.0000E-03    0.02
   KW Writing ......... 0.0000E+00    0.00     5.0000E-03    0.03
 Initialization ....... 1.7000E+01  100.00     1.7827E+01   99.82
 ----------------------------------------------------------------
 T o t a l s            1.7000E+01  100.00     1.7860E+01  100.00

Problem time       =    0.0000E+00
Problem cycle      =         0
Total CPU time     =        17 seconds (   0 hours  0 minutes 17 seconds)
CPU time per zone cycle  =          0.000 picoseconds
Clock time per zone cycle=          0.000 picoseconds

Number of CPU's    1
NLQ used/max     136/  136
Start time   04/05/2022 03:17:37
End time     04/05/2022 03:17:37
Elapsed time       0 second  for       0 cycles using  1 SMP thread
            (      0 hour   0 minute   0 second )

E r r o r   t e r m i n a t i o n                            04/05/22 03:17:37


D:\PredictiveEngineering\LS-DYNA\LS-DYNA Class\Workshops\1 - LS-DYNA Getting Started\1A - Common
ors\Run 2>pause
Press any key to continue . . .
```

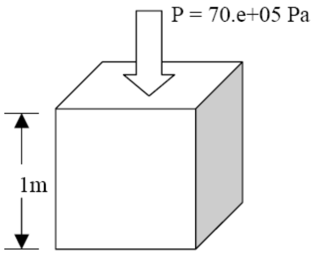
***Analyst's Note:** "Know what you Know" – With nonlinear analysis codes there are so many options, that one should only change those defaults that one knows (which often means reading the manual and maybe creating a pilot model to understand its effect).*

4.8 WORKSHOP 1B – LS-DYNA GETTING STARTED

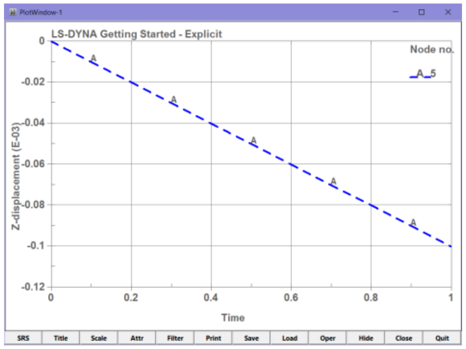
Objective: This workshop uses the LSTC Getting Started Example material and a LS-DYNA model has been prepared. This material can also be found in the Students' "Class Reference Notes" folder. A Workshop video is provided to walk you through the post-processing of the data but your job is to create the one element model that has an applied pressure load.

Tasks:

- Open your favorite text editor and build LS-DYNA Keyword deck using the existing deck: /Explicit Example 1 / ex01 – Start.dyn. The node positions and their constraints have been pre-entered to save you some of the more mundane work. The rest of the Keywords you'll have to figure out (***Workshop - GettingStarted.pdf and Keyword Manual Vol I. Please note that Material Keywords are located in Manual Vol II***).
- Analyze your model using LS-Run and post process the results within LSPP
- 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

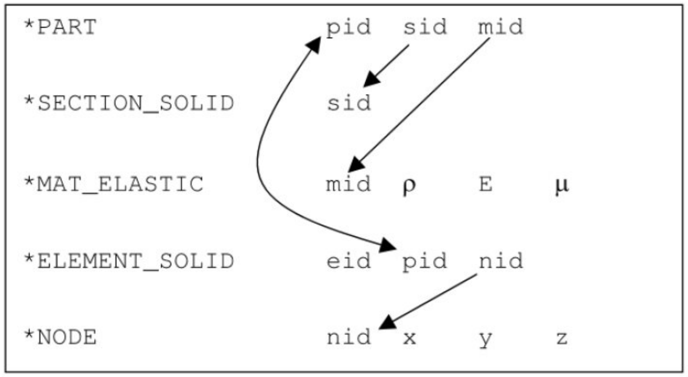


The vertical displacement due to a 70.0e+05 Pa pressure load can be calculated by

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

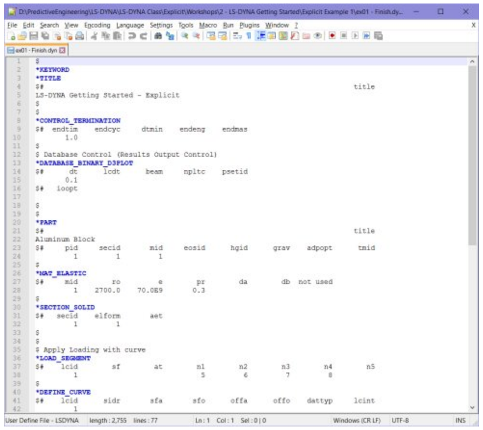
1.0e-04 m

Pivotal Understanding: The "*PART"



```
*PART              pid  sid  mid

*SECTION_SOLID     sid

*MAT_ELASTIC       mid  ρ    E    μ

*ELEMENT_SOLID     eid  pid  nid

*NODE              nid  x    y    z
```

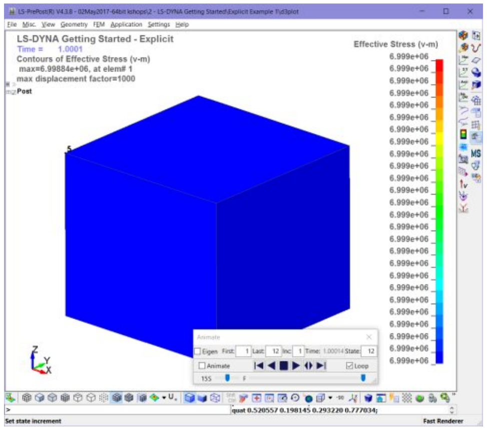
Advantages of Using Notepad++


D:\PredictiveEngineering\LS-DYNA\LS-DYNA Class\Explicit\Workshops\2 - LS-DYNA Getting Started\Explicit Example 1\ex01 - Finish.dy...
File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?
ex01 - Finish.dyn

```
$
*KEYWORD
*TITLE
$#                                                                         title
LS-DYNA Getting Started - Explicit
$
$
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas
       1.0
$
$ Database Control (Results Output Control)
*DATABASE_BINARY_D3PLOT
$#      dt      lcdt      beam     npltc    psetid
       0.1
$#   ioopt

$
$
*PART
$#                                                                         title
Aluminum Block
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1
$
*MAT_ELASTIC
$#     mid        ro         e        pr        da        db  not used
         1    2700.0    70.0E9       0.3
$
*SECTION_SOLID
$#   secid    elform       aet
         1         1
$
$
$ Apply Loading with curve
*LOAD_SEGMENT
$#    lcid        sf        at        n1        n2        n3        n4        n5
         1                             5         6         7         8
$
*DEFINE_CURVE
$#    lcid      sidr       sfa       sfo      offa      offo    dattyp     lcint
         1
```

User Define File - LSDYNA   length : 2,755   lines : 77   Ln : 1   Col : 1   Sel : 0 | 0   Windows (CR LF)   UTF-8   INS

You Should See This

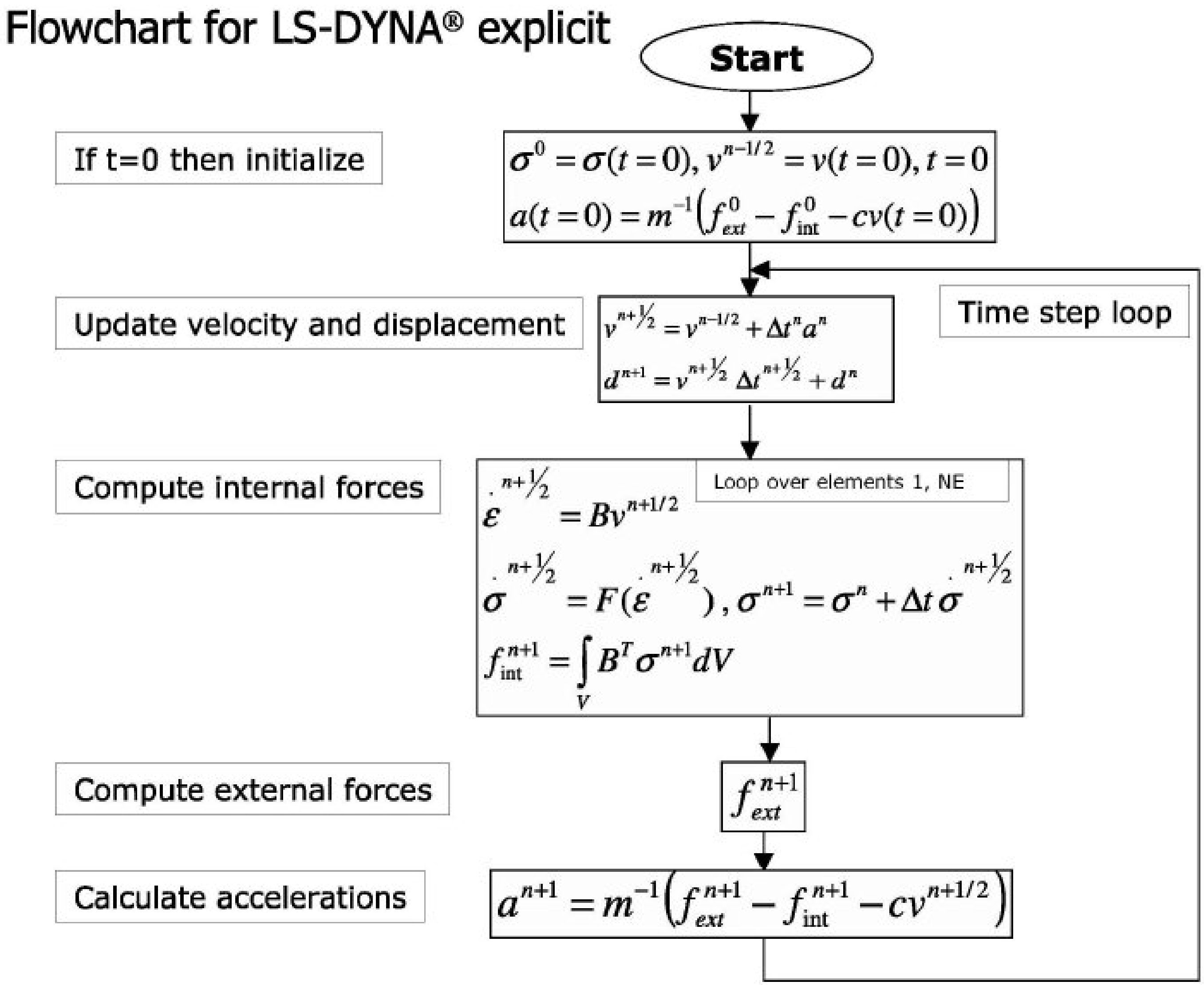

LS-PrePost(R) V4.3.8 - 02May2017-64bit kshops\2 - LS-DYNA Getting Started\Explicit Example 1\d3plot
File Misc. View Geometry FEM Application Settings Help
Effective Stress (v-m)
6.999e+06
5
Set state increment
Fast Renderer

LS-DYNA Getting Started - Explicit
Time = 1.0001
Contours of Effective Stress (v-m)
max=6.99884e+06, at elem# 1
max displacement factor=1000
Post

Take Away (Importance of Workshop): What is involved in building a LS-DYNA FEA model; that simple and direct.

5. FUNDAMENTAL MECHANICS OF EXPLICIT ANALYSIS

5.1 EXPLICIT NUMERICAL FLOWCHART



Flowchart for LS-DYNA® explicit

Start

If t=0 then initialize

$$\sigma^0 = \sigma(t=0),\ v^{n-1/2} = v(t=0),\ t=0$$
$$a(t=0) = m^{-1}\left(f^0_{ext} - f^0_{\text{int}} - cv(t=0)\right)$$

Update velocity and displacement

$$v^{n+\frac{1}{2}} = v^{n-1/2} + \Delta t^n a^n$$
$$d^{n+1} = v^{n+\frac{1}{2}}\,\Delta t^{n+\frac{1}{2}} + d^n$$

Time step loop

Compute internal forces

Loop over elements 1, NE

$$\dot{\varepsilon}^{n+\frac{1}{2}} = Bv^{n+1/2}$$
$$\dot{\sigma}^{n+\frac{1}{2}} = F(\dot{\varepsilon}^{n+\frac{1}{2}}),\ \sigma^{n+1} = \sigma^n + \Delta t\,\dot{\sigma}^{n+\frac{1}{2}}$$
$$f^{n+1}_{\text{int}} = \int_V B^T \sigma^{n+1} dV$$

Compute external forces

$$f^{n+1}_{ext}$$

Calculate accelerations

$$a^{n+1} = m^{-1}\left(f^{n+1}_{ext} - f^{n+1}_{\text{int}} - cv^{n+1/2}\right)$$

5.2 Time Step Significance (Courant-Friedrichs-Lewy (CFL) Characteristic Length)

- In the simplest case (small, deformation theory), the timestep is controlled by the acoustic wave propagation through the material.
- In the explicit integration, the numerical stress wave must always propagate less than one element width per timestep.
- The timestep of an explicit analysis is determined as the minimum stable timestep in any ***one (1) deformable finite element in the mesh***. (Note: As the mesh deforms, the timestep can similarly change)
- The above relationship is called the Courant-Friedrichs-Lewy (CFL) condition and determines the stable timestep in an element. The CFL condition requires that the explicit timestep be smaller than the time needed by the physical wave to cross the element. Hence, the numerical timestep is a fraction (0.9 or lower) of the actual theoretical timestep. Note: the CFL stability proof is only possible for linear problems.
- In LS-DYNA, one can control the time step scale factor (*tssfac*). The default setting is 0.9. It is typically only necessary to change this factor for shock loading or for increased contact stability with soft materials.
- As a note, the *tssfac* doesn't change the "wave speed" only the time step.

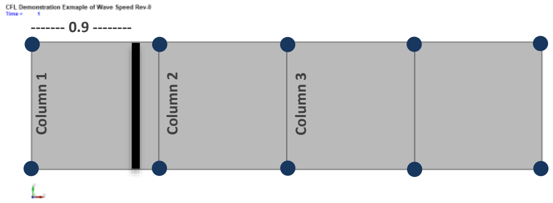
$$C_{AccousticWaveSpeed} = \sqrt{\frac{E_{Material}}{\rho_{Material}}}$$

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

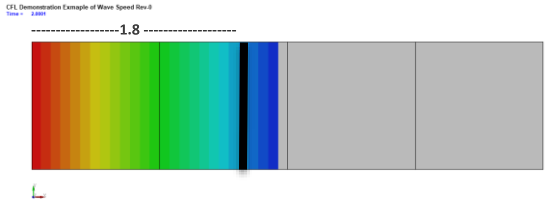
$$\Delta Timstep_{CFL} = (0.9)\Delta ExplicitTimestep$$

A bar is given a whack on its end. At time = 0.0, the state of the system is unknown except where loads and constraints are applied. At the first explicit time step, the stress wave (black bar) has advanced 90% or 0.9 into the element. The explicit calculation then can calculate the acceleration at the first column of nodes but everything else is still unknown. Hence, we still have zero stress in the bar. At the second explicit time step, we finally have all the information that we need to calculate stress in the first element. The stress state can now be contoured. Be aware that the stress fringe shown is extrapolated from the data set. We still do not know what the stresses are in the second element until the third time step has completed.

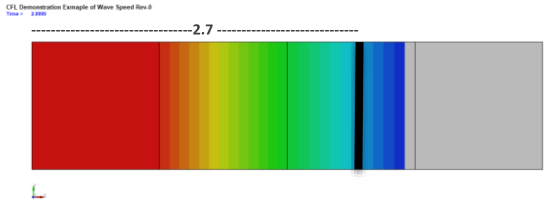
First Explicit Time Step



Second Explicit Time Step



Third Explicit Time Step

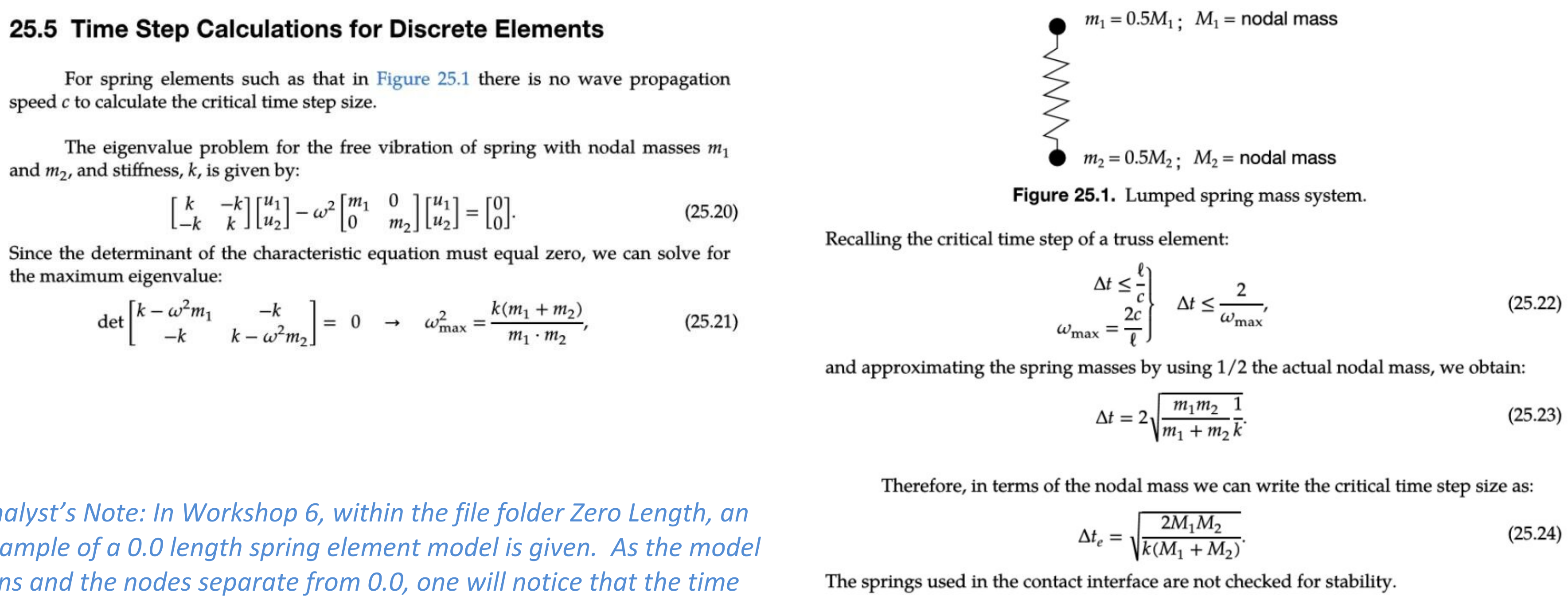


Analyst's Note: Based on these conditions, the time step can be increased to provide faster solution times by: (i) increasing the density of the material (e.g., mass scaling; (ii) lowering the modulus or; (iii) by increasing the element size of the mesh.

5.2.1 Is the CFL based on Elements or Nodes?

The CFL is based on neither but the product of stiffness and mass, i.e., in a purely mathematical sense. It is just common to describe it using wave speed and a characteristic element length whether for a rod (1D), shell (2D) or solid (3D) elements. But then this leaves out springs, which can be defined with coincident nodes, which can be denoted as 0D elements. Within LST's LS-DYNA Theory Manual is a section on time step calculations that covers the CFL explicit time step calculation for springs (0D). As shown, all one needs is stiffness and mass to calculate an explicit time step. At the end of this section, a little note is given: "*The springs used in the contact interface are not checked for stability.*" That is, when LS-DYNA starts an analysis, it will sweep through all elements from 0D to 3D and calculate the explicit time step but does not check contact interface explicit time step for ***stability*** for global control of the model's time step (although it does provide an estimate of the interface's time step). In LS-DYNA, the penalty formulation (virtual springs) *CONTACT has an explicit time step based on the stiffness and mass of the opposing surfaces. Although the explicit time step of the contact interface is not normally a problem, it is something that one should be aware of and if a warning message is displayed, it should be paid heed.

LS-DYNA Theory Manual



25.5 Time Step Calculations for Discrete Elements

For spring elements such as that in Figure 25.1 there is no wave propagation speed c to calculate the critical time step size.

The eigenvalue problem for the free vibration of spring with nodal masses m_1 and m_2, and stiffness, k, is given by:

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \end{bmatrix} - \omega^2 \begin{bmatrix} m_1 & 0 \\ 0 & m_2 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix}. \qquad (25.20)$$

Since the determinant of the characteristic equation must equal zero, we can solve for the maximum eigenvalue:

$$\det\begin{bmatrix} k-\omega^2 m_1 & -k \\ -k & k-\omega^2 m_2 \end{bmatrix} = 0 \quad \rightarrow \quad \omega_{\max}^2 = \frac{k(m_1+m_2)}{m_1 \cdot m_2}, \qquad (25.21)$$

$m_1 = 0.5M_1$; M_1 = nodal mass

$m_2 = 0.5M_2$; M_2 = nodal mass

Figure 25.1. Lumped spring mass system.

Recalling the critical time step of a truss element:

$$\left.\begin{array}{l} \Delta t \le \frac{\ell}{c} \\ \omega_{\max} = \frac{2c}{\ell} \end{array}\right\} \quad \Delta t \le \frac{2}{\omega_{\max}}, \qquad (25.22)$$

and approximating the spring masses by using 1/2 the actual nodal mass, we obtain:

$$\Delta t = 2\sqrt{\frac{m_1 m_2}{m_1+m_2}\frac{1}{k}}. \qquad (25.23)$$

Therefore, in terms of the nodal mass we can write the critical time step size as:

$$\Delta t_e = \sqrt{\frac{2M_1M_2}{k(M_1+M_2)}}. \qquad (25.24)$$

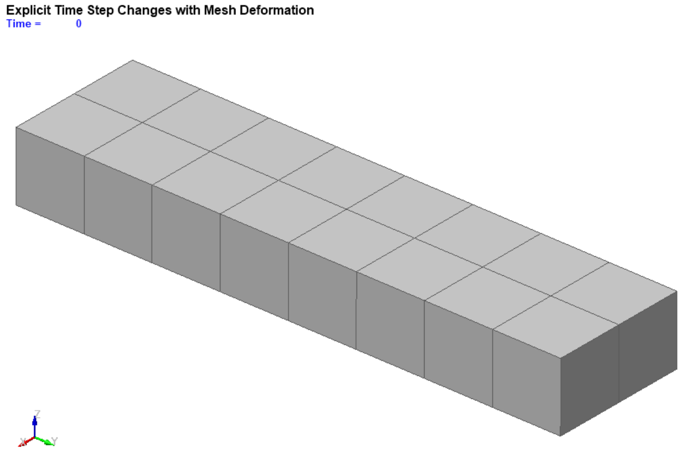
The springs used in the contact interface are not checked for stability.

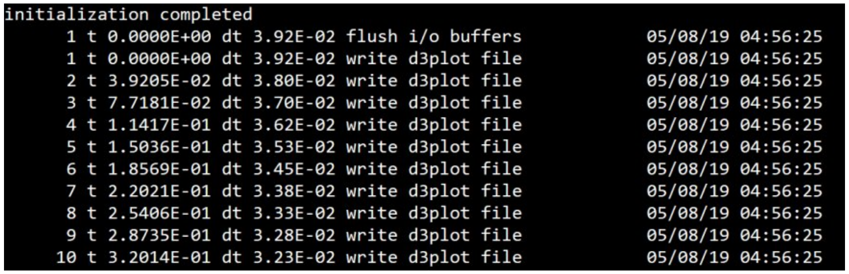
Analyst's Note: In Workshop 6, within the file folder Zero Length, an example of a 0.0 length spring element model is given. As the model runs and the nodes separate from 0.0, one will notice that the time step does not change.

5.2.2 As the Mesh Size Changes, So Does the Explicit Time Step

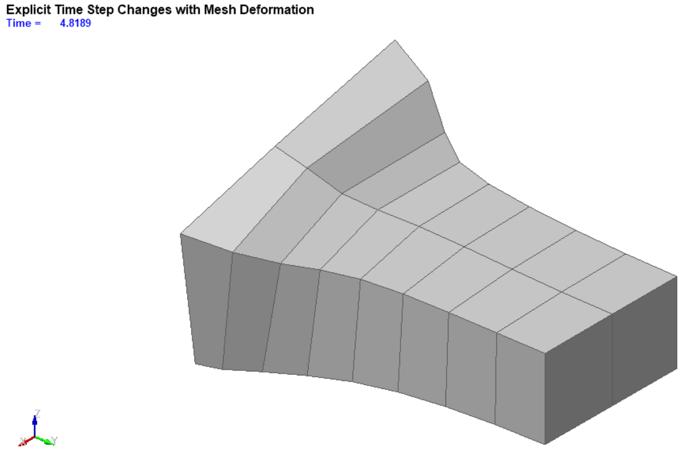
Given the theoretical background of how the explicit time step is calculated, one must also consider that as the analysis progresses and the mesh deforms, the explicit time step will likewise change.

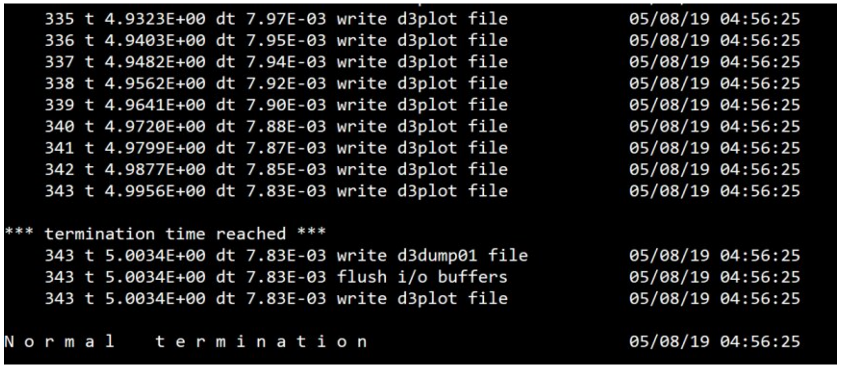
FEA Mesh | **Explicit Analysis: Start dt = 3.92e-2 / Finish dt = 7.83e-3**





```
initialization completed
      1 t 0.0000E+00 dt 3.92E-02 flush i/o buffers            05/08/19 04:56:25
      1 t 0.0000E+00 dt 3.92E-02 write d3plot file            05/08/19 04:56:25
      2 t 3.9205E-02 dt 3.80E-02 write d3plot file            05/08/19 04:56:25
      3 t 7.7181E-02 dt 3.70E-02 write d3plot file            05/08/19 04:56:25
      4 t 1.1417E-01 dt 3.62E-02 write d3plot file            05/08/19 04:56:25
      5 t 1.5036E-01 dt 3.53E-02 write d3plot file            05/08/19 04:56:25
      6 t 1.8569E-01 dt 3.45E-02 write d3plot file            05/08/19 04:56:25
      7 t 2.2021E-01 dt 3.38E-02 write d3plot file            05/08/19 04:56:25
      8 t 2.5406E-01 dt 3.33E-02 write d3plot file            05/08/19 04:56:25
      9 t 2.8735E-01 dt 3.28E-02 write d3plot file            05/08/19 04:56:25
     10 t 3.2014E-01 dt 3.23E-02 write d3plot file            05/08/19 04:56:25
```





```
    335 t 4.9323E+00 dt 7.97E-03 write d3plot file            05/08/19 04:56:25
    336 t 4.9403E+00 dt 7.95E-03 write d3plot file            05/08/19 04:56:25
    337 t 4.9482E+00 dt 7.94E-03 write d3plot file            05/08/19 04:56:25
    338 t 4.9562E+00 dt 7.92E-03 write d3plot file            05/08/19 04:56:25
    339 t 4.9641E+00 dt 7.90E-03 write d3plot file            05/08/19 04:56:25
    340 t 4.9720E+00 dt 7.88E-03 write d3plot file            05/08/19 04:56:25
    341 t 4.9799E+00 dt 7.87E-03 write d3plot file            05/08/19 04:56:25
    342 t 4.9877E+00 dt 7.85E-03 write d3plot file            05/08/19 04:56:25
    343 t 4.9956E+00 dt 7.83E-03 write d3plot file            05/08/19 04:56:25

*** termination time reached ***
    343 t 5.0034E+00 dt 7.83E-03 write d3dump01 file          05/08/19 04:56:25
    343 t 5.0034E+00 dt 7.83E-03 flush i/o buffers            05/08/19 04:56:25
    343 t 5.0034E+00 dt 7.83E-03 write d3plot file            05/08/19 04:56:25

N o r m a l    t e r m i n a t i o n                          05/08/19 04:56:25
```

*Analysts' Note: As one can imagine, with severe deformation, the explicit time step could approach zero. LS-DYNA allows one to stop the analysis or delete highly deformed elements based on their timestep (see *CONTROL_TERMINATION, dtmin and then *CONTROL_TIMESTEP, erode)*

5.3 Mass Scaling: (Everybody Does it But Nobody Really Likes It) – Changing the Wave Speed

Explicit Time Step Mass Scaling (*Control_Timestep)*

- Mass scaling is very useful and directly lowers the wavespeed and therefore increases the timestep given that the element sizing doesn't change. 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* *(Note in Keyword Manual "LT" means a negative value or "-" in front of the number))*: 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. But I prefer to visualize it within LSPP using *DATABASE_EXTENT_BINARY with *stssz* and/or *msscl* settings (RTM).

Analysts' Note: General recommendations and tips are given in Explicit Model Check-Out and Recommendations.

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

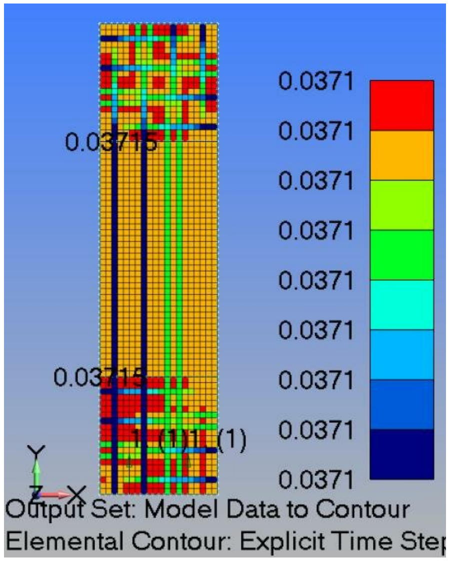
$$C_{Aluminum} = \sqrt{\frac{\frac{70}{(1-\upsilon^2)}}{2.71x10^{-6}}} = 5{,}384 \text{ mm/ms}$$

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

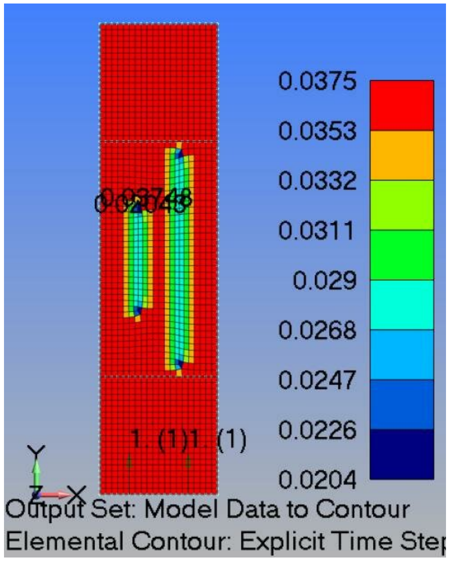
- LS-DYNA time step is different between LSPP and LS-DYNA due to *tssfac*=0.9 (default)
- Mesh quality affects Time Step – just tweak it

5.3.1 Instructor Led Workshop: 1 – Mass Scaling

Clean Mesh



Tweaked



5.3.2 Workshop: 2 - LS-DYNA Mass Scaling Basics

What You Will Learn:

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

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

Tasks:

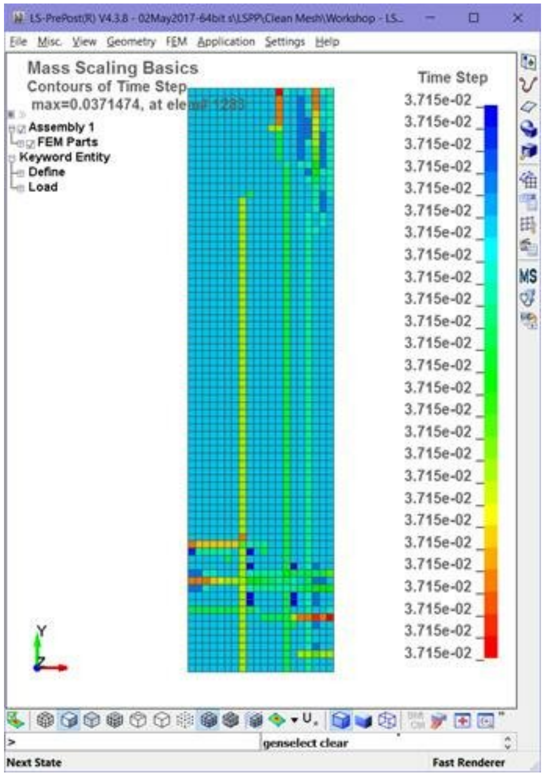
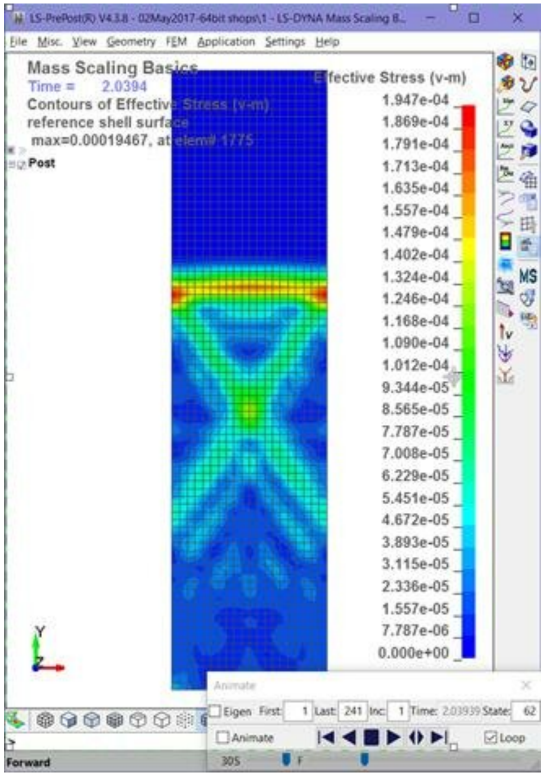
- Open Keyword deck: /LS-DYNA Mass Scaling Basics / LSPP / Clean Mesh / LS-DYNA Mass Scaling Basics - Clean.dyn in LSPP. Verify elastic isotropic material (*MAT_ELASTIC) properties and then shell property (*SECTION_SHELL) with *elform*=2 and thickness = 1.0.
- Check explicit time step using LSPP's command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

Analysts' Note: Although this model would run faster using SMP single-precision (~10 to 20%), we are aiming to keep it simple and use the most robust solver platform. As always, once a solution is in hand, one can seek efficiencies try running it with single-precision.

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

Linear, Elastic Material:

E	v	ρ
70	0.33	2.71e-6

Explicit Time Step	Stress Wave Propagation from Short Duration Force Pulse
	

Mass Scaling Basics
Contours of Time Step
max=0.0371474, at elem# 1283

Mass Scaling Basics
Time = 2.0394
Contours of Effective Stress (v-m)
reference shell surface
max=0.00019467, at elem# 1775

Eigen First: 1 Last: 241 Inc: 1 Time: 2.03939 State: 62
□ Animate ☑ Loop

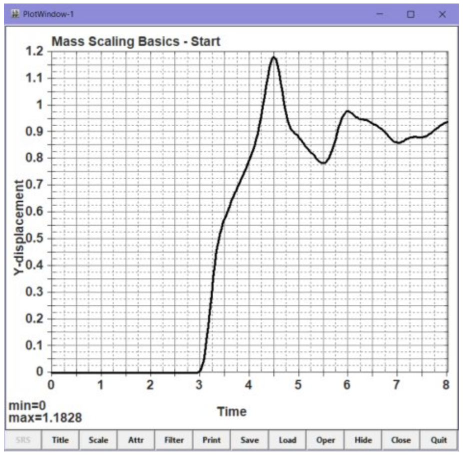
Workshop: 2 – LS-DYNA Mass Scaling Basics (continued)

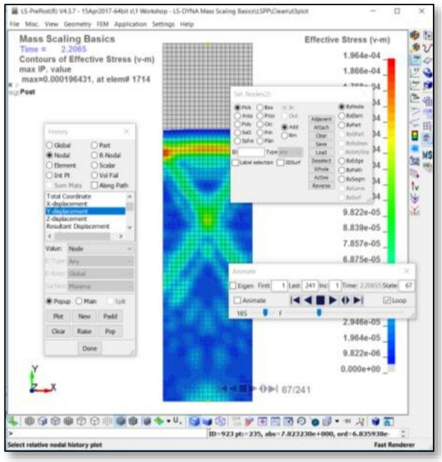
With the model working, let's harvest some data. We are going to make several runs of this model to investigate the relationship between mesh, explicit time step and mass scaling. As part of this process, you'll get comfortable working with LSPP and LS-Run. Our metric is going to be the maximum displacement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

Tasks:

- Within existing LSPP model, open History, select Node, Y-Displacement and then pick the node at the very top of the bar (node #1) and hit Plot. (Note: The node is attached to a constrained nodal rigid body).
- Note that the maximum displacement at the top of the bar is 1.18 mm.
- Start filling out the Table at the bottom of this page.

{A filled-out Table is provided for you to check your work within a nested file folder labeled "Table".}





Documenting the Learning Objective:

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

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

Model	Time Step	% Mass Added by Mass Scaling	Max. Displacement
Starting Point	0.0334 ms	0.0%	1.18 mm
Skewed Mesh (-4x)	0.0184 ms	0.0%	mm
Skewed Mesh with Mass Scaling I	0.0334 ms	%	mm
Skewed Mesh with Mass Scaling II	0.0290 ms	%	mm

Got Extra Time? Open up: Abandon All Hope {Workshop - LS-DYNA Mass Scaling Basics - Skewed Mesh - Violation of CFL – FINISH.dyn} and see what happens when one forces LS-DYNA to ignore the CFL criterion. It'll bark at you but it'll run. For LS-DYNA non-newbies, take a look at the EXTRA file folder and contour the mass percentage added via CMS and create an XY plot of the added mass to the three PART's of the model.

5.3.3 Instructor Led Workshop: 2 - Mass Scaling Advanced

Explicit Time Step Mass Scaling (*CONTROL_TIMESTEP):*

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

Example:

We are impacting a ball against a plate. The mesh is not uniform and mass scaling is used to speed up the analysis. The ball is first analyzed with no mass scaling, then with CMS and finally with SMS over the whole ball. To verify our analysis, we plot the kinetic energies from all three runs.

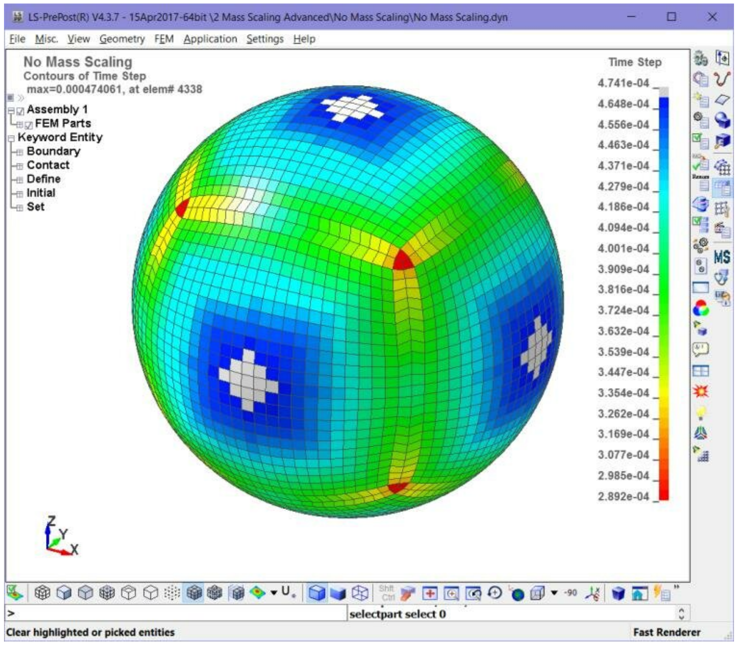
*Analysts' Note: Please understand that CMS is used **on all other parts** not called out within the IMSCL command (see Keyword Manual*

Some Finer Points:

- Solution time is 28 seconds for no mass scaling and 15 and 13 seconds for SMS and CMS respectively. SMS is more computationally expensive but has large benefits for some models.
- Question: Would mass scaling make your dynamic (F=ma) analysis more conservative or less?

Example Courtesy of www.DynaSupport.com

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

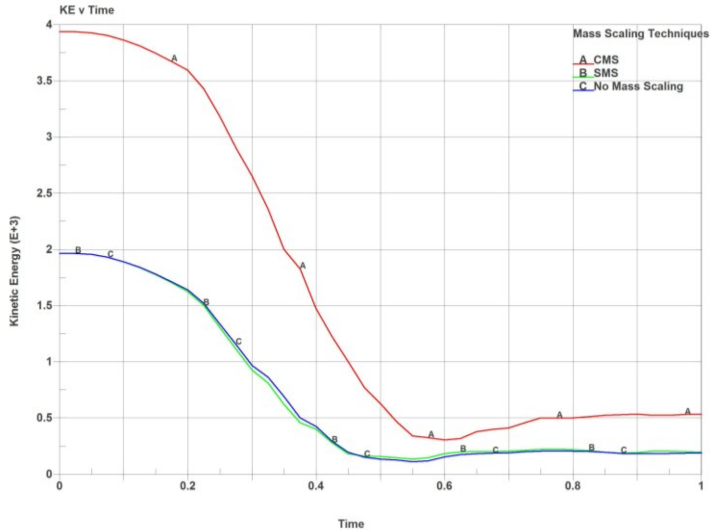


LS-PrePost(R) V4.3.7 - 15Apr2017-64bit \2 Mass Scaling Advanced\No Mass Scaling\No Mass Scaling.dyn

No Mass Scaling
Contours of Time Step
max=0.000474061, at elem# 4338

> selectpart select 0

Clear highlighted or picked entities



5.4 Implicit Mesh versus Explicit Mesh Characteristics

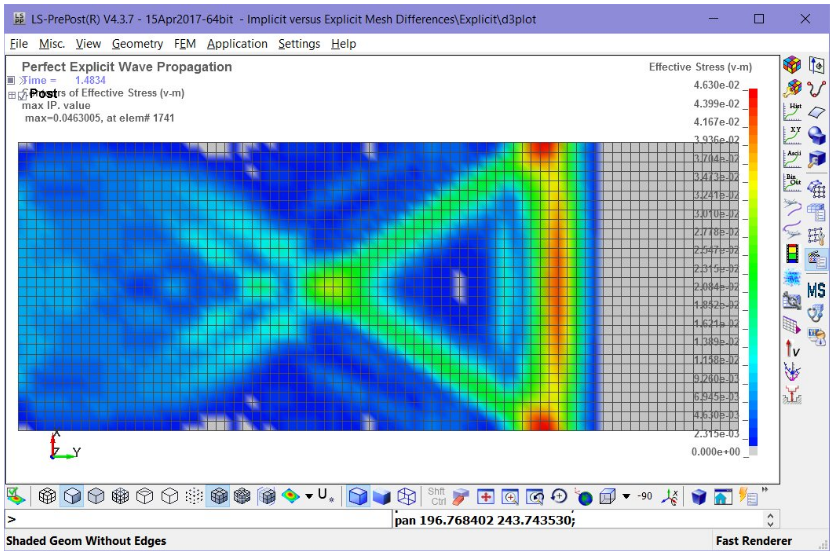
5.4.1 Instructor Led Workshop: 3 - Implicit versus Explicit Mesh Differences

Meshing for Accuracy

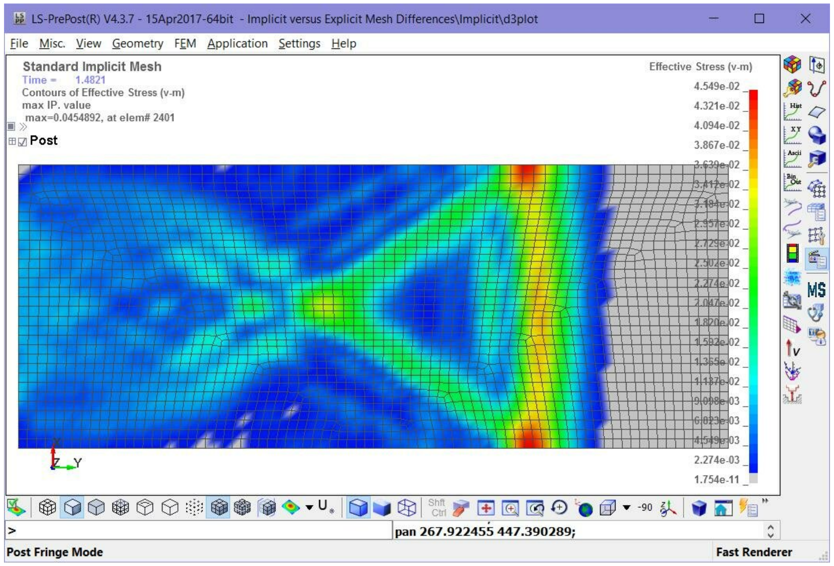
- Solution time (number of nodes + time step) is often one of the most important considerations in setting up an explicit analysis; care should be exercised in setting up the mesh density.
- A good implicit mesh ***does not*** typically work well for an explicit analysis.
- In an explicit analysis, linear,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

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

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


LS-PrePost(R) V4.3.7 - 15Apr2017-64bit - Implicit versus Explicit Mesh Differences\Explicit\d3plot
File Misc. View Geometry FEM Application Settings Help
Perfect Explicit Wave Propagation
Time = 1.4834
Contours of Effective Stress (v-m)
Post
max IP. value
max=0.0463005, at elem# 1741
Effective Stress (v-m)
4.630e-02
4.399e-02
4.167e-02
3.936e-02
3.704e-02
3.473e-02
3.241e-02
3.010e-02
2.778e-02
2.547e-02
2.315e-02
2.084e-02
1.852e-02
1.621e-02
1.389e-02
1.158e-02
9.260e-03
6.945e-03
4.630e-03
2.315e-03
0.000e+00
Z
Y
Shaded Geom Without Edges
Fast Renderer

pan 196.768402 243.743530;



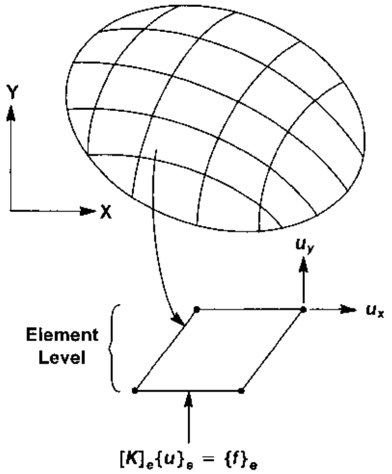
pan 267.922455 447.390289;

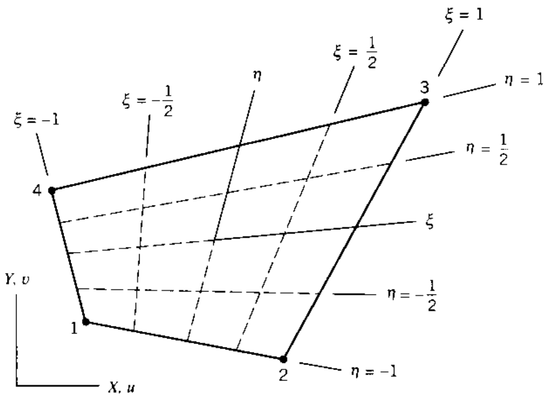
5.4.2 A Short Discussion on Element Quality (aka Jacobian)

Although this section covers material that might be understood by most students, it provides an introduction to the importance of element quality in performing explicit and implicit analyses. If one is not sure what are isoparametric elements, take a quick read from a chapter of Ed Wilson's book located in the Class Reference Notes / Elements / Isoparametric Element Theory (www_EdWilson_org - Book-Wilson - 05-iso.pdf).

Isoparametric (having the same parameters under different coordinate systems) are the bedrock of modern FEA. Simple functions are used to discretize oddly shaped surfaces or volumes. The basis of this method is given in the subsequent slides. Although the theory is given in 2-D it can be directly leveraged into the third dimension.

One starts with a random region that is normalized into a -1 to +1 coordinate system and two formulas that use a simple linear shape function to define interior coordinates and interior displacements:





$$x_{xp} = \sum_{i=1}^{4} N_i(\xi,\eta)x_{xi}$$

$$u_{xp} = \sum_{i=1}^{4} N_i(\xi,\eta)u_{xi}$$

N_i is known as the shape function, which does double duty as the interpolation function for both coordinates (x) and displacements (u). This is the "iso" in the isoparametric. With these formulas we can map displacements in the interior of our element and also map any coordinates. An example of a linear shape function for a four-node quadrilateral element (see FEA textbooks for quadratic shape functions use in parabolic eight-node quadrilateral elements):

5.4.2.1 An Example of the Assembly of Equations for Static Stress Analysis

We start with basic mechanics and apply the isoparametric method to these equations.

Step 1: Satisfy static equilibrium

$$\sum F = 0$$

Step 2: Relate strain to displacements (simple 2D example)

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} \partial/\partial_x & 0 \\ 0 & \partial/\partial_y \\ \partial/\partial_y & \partial/\partial_x \end{bmatrix} \begin{Bmatrix} u \\ v \end{Bmatrix}; \qquad \varepsilon = \partial u$$

Step 3: Incorporate the shape function

This is where it gets a little complicated. To get our generalized displacements (u, v), the shape functions discussed on the prior slide are used to take corner point displacements (nodes) u_i and v_i and generate displacements anywhere within the element.

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & \cdots \\ 0 & N_1 & 0 & N_2 & \cdots \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ \vdots \end{Bmatrix}; \qquad u = Nd$$

Step 4: Relate strain to displacements (using the B matrix)

Matrix “B” is called the strain-displacement matrix and is common FEA matrix jargon. The concept is that you are using the shape function to determine the “strain characteristics” within the quadrilateral element.

$$\varepsilon = \partial N d; \qquad \varepsilon = Bd; \qquad B = \partial N$$

Step 5: Relate stress to strain

$$\sigma = E\varepsilon; \qquad \sigma = EBd$$

Step 6: Relate force to stress

$$F = E\varepsilon A; \qquad F = EBdA$$

Step 7: Relate force to displacement

$$F = Kd; \qquad K = EBA; \qquad u = d$$

The pivotal part is that “EBA” is the stiffness term of the element. It is this component that is calculated to form the stiffness matrix [K]. It doesn’t seem that hard but just calculating the area of a quadrilateral by brute force (double integration) is a numerically very intensive task.

$$\{F\} = [K]\{u\}$$

The prior expression that formulated K=EBA leaves out a few numerical details. To actually calculate individual stiffness terms for the element, the formula EBA must be numerically integrated over the area or volume of the element. This is done by the following standard equation:

$$[K] = \iint [B]^T [E][B] dxdy$$

However, if a standard generalized numerical integration would be used, this operation would be very slow and model sizes would be limited to a few thousands of elements and not hundreds of thousands as in the norm today. To accelerate the numerical area or volume calculation, a process called Gaussian Integration is used. For this process to work, the first step is to transform the generalized X and Y coordinates into normalized -1 to +1 space. This is a linear transformation or a mapping process. The transformation matrix is called the Jacobian. Every element will have a unique transformation from its generalized coordinates into a normalized system.

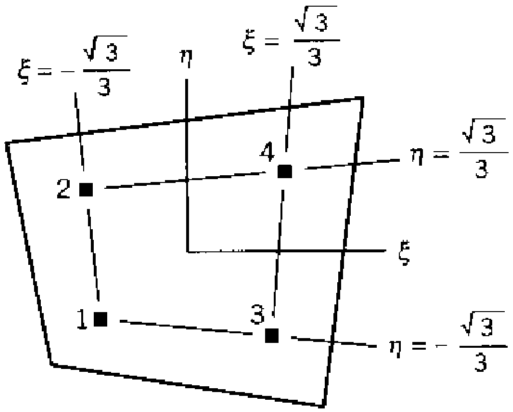
$$[K] = \iint_{-1}^{1} [B]^T [E][B][J]\, d\xi d\eta$$

The Jacobian is also a popular measure of the element’s quality. If the element is distorted, one might say that the Jacobian has a lot of work to do in normalizing the element into a -1, +1 space. If the element is a square or a clean rectangle, the Jacobian practically does nothing. A value of 0.0 indicates a perfectly shaped element and a value of near 1.0 indicates something that might not be solvable.

5.4.2.2 Gaussian Integration for Isoparametric Elements

To numerically integrate the isoparametric element a technique known as Gauss Quadrature is employed. This technique is based on the element having a normalized coordinate system of -1, +1. Essentially, the inner terms of the stiffness equation given below are only solved at discrete points within the element and weighting functions based on Gauss Quadrature are then applied. The discrete points where this numerical integration is carried out are called Gaussian Integration Points (Gauss points). An example of the location of Gauss points in a quadrilateral element is given below.

Gaussian integration is at its best (i.e., most accurate) when the element is as near as possible to a perfect square. During the integration process, tabulated weighting values are used (terms Wi and Wj) to arrive at the final integrated value (I) for the elements area or volume:


$\xi = -\frac{\sqrt{3}}{3}$
η
$\xi = \frac{\sqrt{3}}{3}$
$\eta = \frac{\sqrt{3}}{3}$
ξ
$\eta = -\frac{\sqrt{3}}{3}$
1
2
3
4

$$\boldsymbol{I} = \sum_{i=1}^{n} \sum_{j=1}^{m} W_i W_j \phi(\xi_i \eta_j)$$

The location of these Gauss points is also used for strain recovery and with strain we have stress. That is, in isoparametric elements, stresses are calculated at the Gauss points and extrapolated out to the nodal points for contouring. Hence, a high-quality element (low Jacobian) will provide double benefits with a more accurate [K] and cleaner stress calculation.

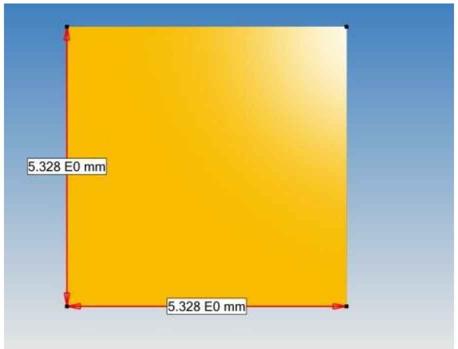
For additional reading on the subject, see the following references:

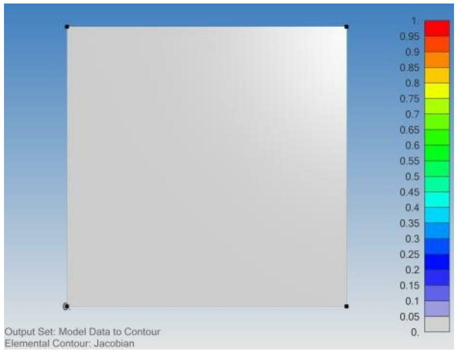
R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt, "Concepts and Applications of Finite Element Analysis," 4th Edition, 2001.

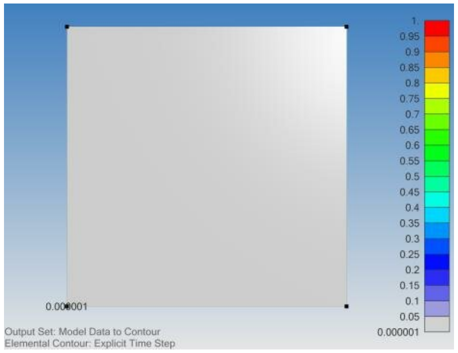
K. J. Bathe, "Finite Element Procedures," 2007.

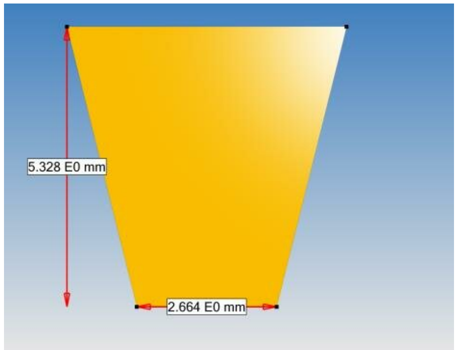
5.4.2.3 How Can One Leverage Element Quality to Create Higher Quality Analyses?

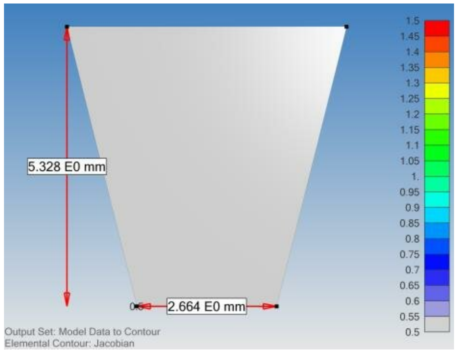
An efficient measure of a element's quality is it's Jacobian since it mathematically describes the transformation of the element's global coordinate system into a set of normalized coordinates whether in 2D or 3D. Contouring of the Jacobian can be readily done in most FEA pre-processors. Please note that the Jacobian quality check can't help you if the element is warped. Thus, a warping check should be included when working with shells elements that are meshed on curved surfaces. Below are some simple examples; however, there is not a 1 to 1 connection between a Jacobian value and the explicit time step (see LS-DYNA Theory Manual for explanation). At the end of the day, it is Jacobian (first) AND Explicit Time Step (second) AND Warping (last).

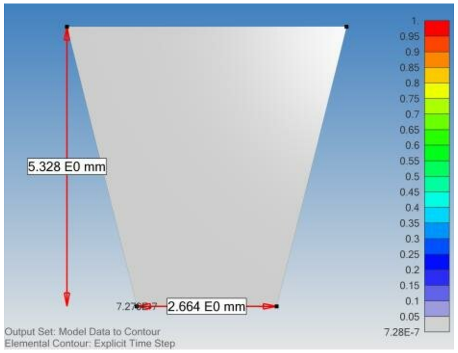


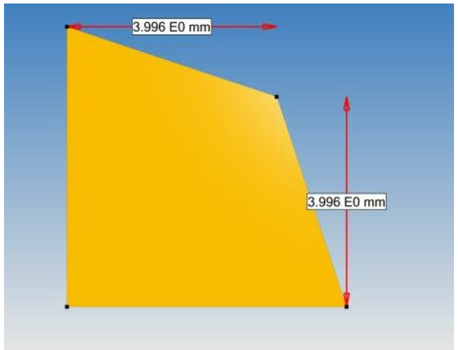


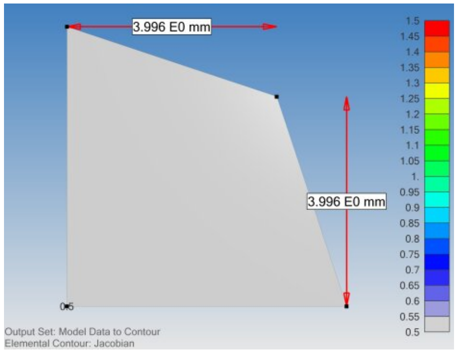


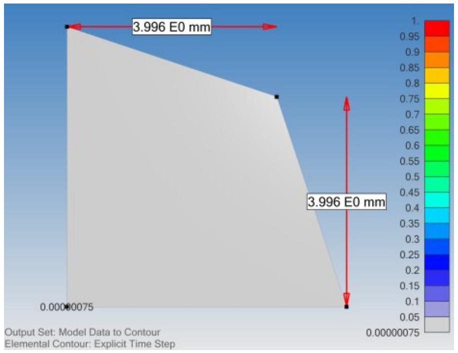












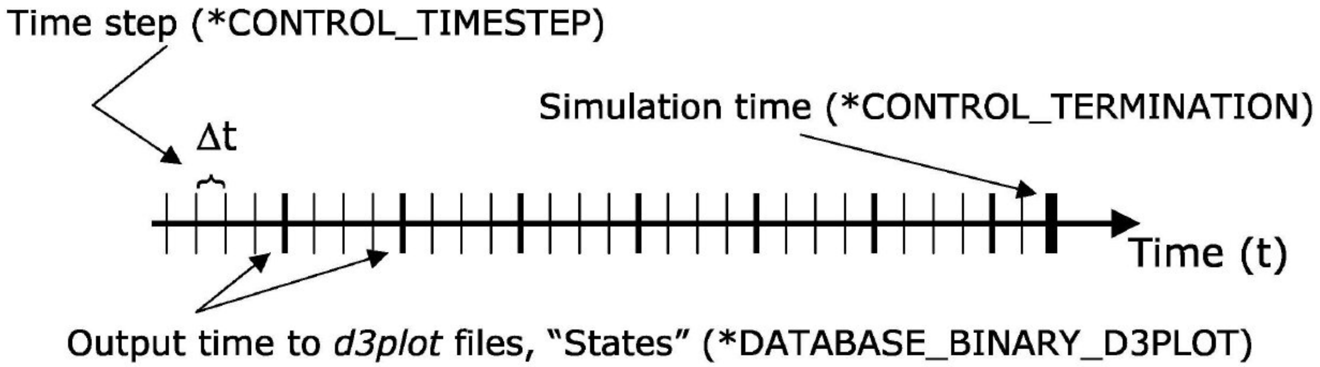
5.5 Summary of Explicit Time Integration

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

Some LS-DYNA explicit terminology that helps explain the relationship between these Keywords:

Time step (*CONTROL_TIMESTEP)

Simulation time (*CONTROL_TERMINATION)



Output time to *d3plot* files, "States" (*DATABASE_BINARY_D3PLOT)

6. EXPLICIT ELEMENT TECHNOLOGY

6.1 Element Types in LS-DYNA

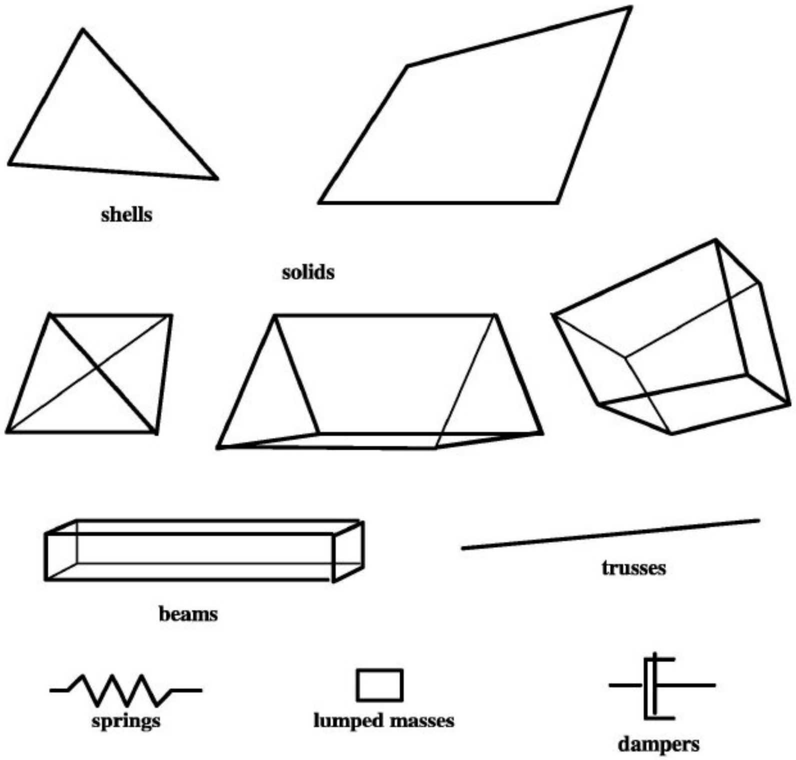
Element Toolbox:

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

- Point elements (mass, inertia)
- Discrete elements (springs, dampers)
- Beams, cables, discrete-beams, etc.
- Solids (20 and 3D, Lagrangian, Eulerian, ALE)
- Shells
- Thick Shells (8 node)
- Cohesive elements
- Seatbelts (and related components)
- EFG and SPH (meshless methods)

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. Only your imagination limits the type of cross-sections available for beams using the *INTEGRATION_BEAM option. Since LS-DYNA is designed for nonlinear mechanics, beams require integration and care should be taken. More on beam element theory and modeling is provided in the Implicit Section of these notes.
- For solid elements, the default is *elform*=1 and uses one-point Gaussian Integration (constant) stress. This element is excellent for very large deformations. It is the standard recommend for explicit simulations.
- Shell elements are covered in detail.



Detailed Element Recommendations (see Class Reference Notes)

{Elements / Solid Elements} Review of Solid Element Formulations Erhart.pdf

{Aerospace Working Group} - AWG LS-DYNA Modeling Guidelines MGD_v19-2 (December 2019).pdf

Analyst's Note: Beam elements represent the highest form of idealization and offer the most opportunities for optimization due to their ease of shape modification (i.e., cross-section modification). LS-DYNA is extremely powerful and can model all standard beams through the linear and nonlinear regime. However care must be taken to understand how stresses are calculated at integration points and that contact is with a cylindrical representation.

6.2 ONE GAUSSIAN POINT ISOPARAMETRIC SHELL ELEMENTS AND HOURGLASSING

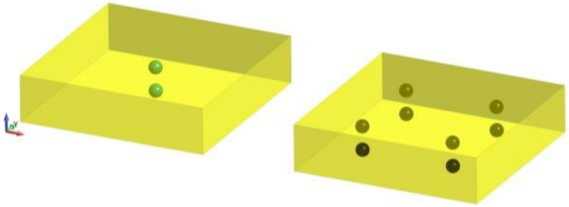
6.2.1 INSTRUCTOR LED WORKSHOP: 4 - EXPLICIT ELEMENT TECHNOLOGY | A: SIDE BENDING

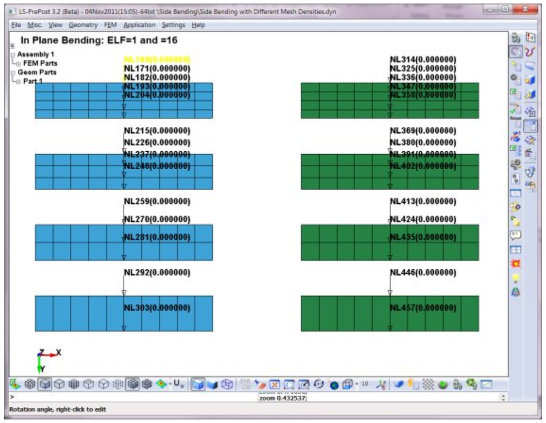
Isoparametric Shell Elements

Default element is **one** Gauss 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.

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

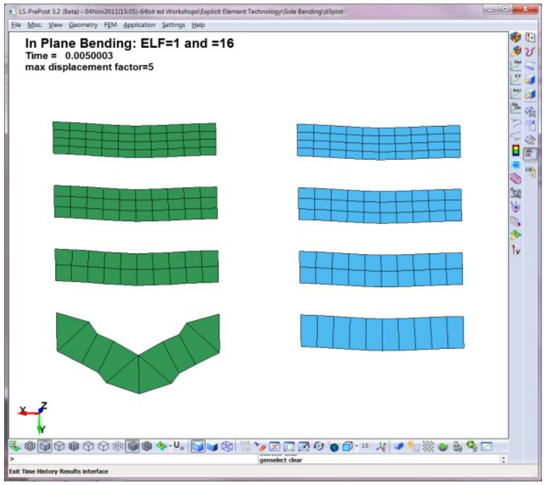




LS-PrePost 3.2 (Beta) - 04Nov2011(15:05)-64bit \Side Bending\Side Bending with Different Mesh Densities.dyn

In Plane Bending: ELF=1 and =16

```
>
zoom 0.432537;
Rotation angle, right-click to edit
```



LS-PrePost 3.2 (Beta) - 04Nov2011(15:05)-64bit ed Workshops\Explicit Element Technology\Side Bending\d3plot

In Plane Bending: ELF=1 and =16
Time = 0.0050003
max displacement factor=5

```
genselect clear
Exit Time History Results interface
```

A Mechanics Observation on In-Plane and Out-of-Plane (Through Thickness Integration

For shell elements, the stiffness of the element is calculated based on the in-plane integration points. If the element is perfect (i.e., 1 x 1), then the stiffness calculation is exact for 1-point and for 4-point Gauss integration. Given that perfectly shaped shell elements are rare, the reality is that 4-point integration does a much better job given randomly shaped elements. The out-of-plane integration points or the through-thickness planes of Gauss integration only serve for the calculation of plastic strain. An example of how the through-thickness integration planes effect the calculation of a plate under bending given material plasticity is shown on the next page. The key observation is that in-plane gives you the in-plane stiffness calculation (K) while through-thickness captures the plastic strain behavior or one might say, the out-of-plane stiffness behavior if and only if plastic strain occurs.

6.2.2 Instructor Led Workshop: 4 - Explicit Element Technology | B: Out-of-Plane Bending with Plasticity

Isoparametric Shell Elements*

- Only one formulation is recommended: *elform*=-16.
- Number of through-thickness integration points (*nip*) controlled by user:

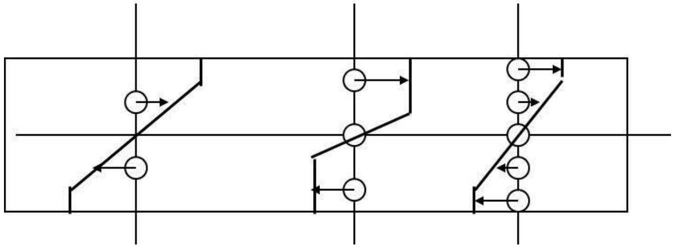
nip 1: Membrane Behavior

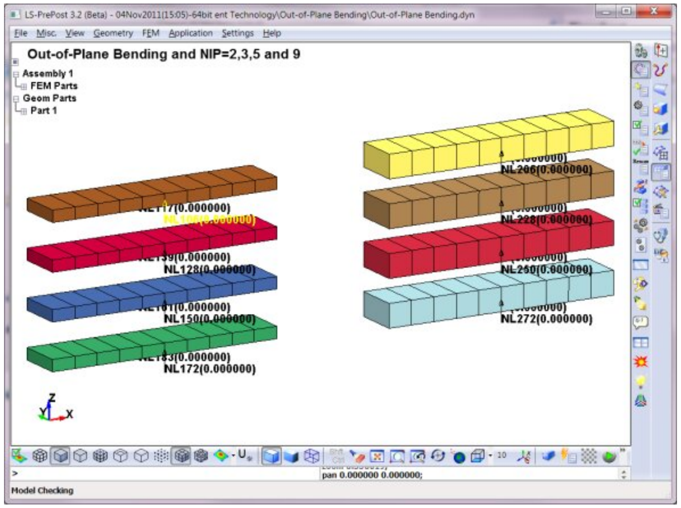
nip 2: *Barely Adequate* (default)

nip 5: Recommended for Nonlinear Materials

Recommend *elform* and *nip* for Nonlinear Plasticity = 5

- *elform*=-16 with *nip*=5

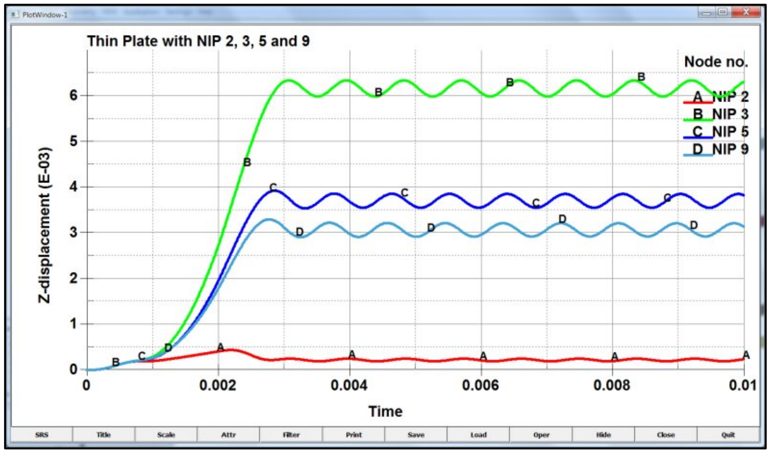


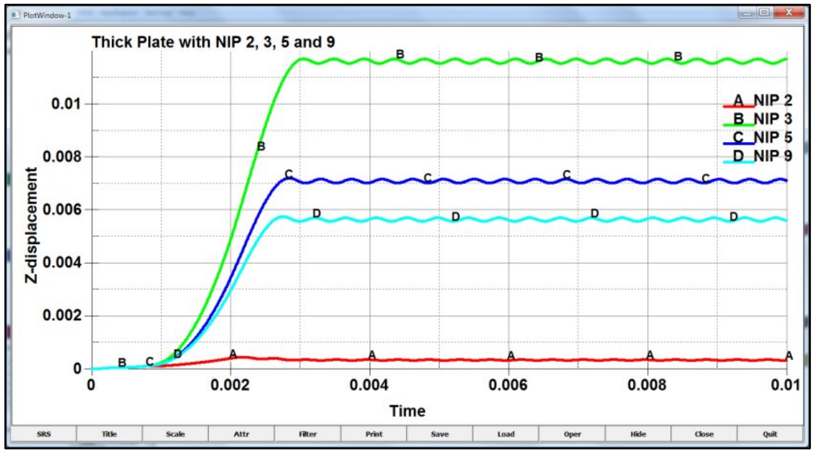


LS-PrePost 3.2 (Beta) - 04Nov2011(15:05)-64bit ent Technology\Out-of-Plane Bending\Out-of-Plane Bending.dyn

Out-of-Plane Bending and NIP=2,3,5 and 9

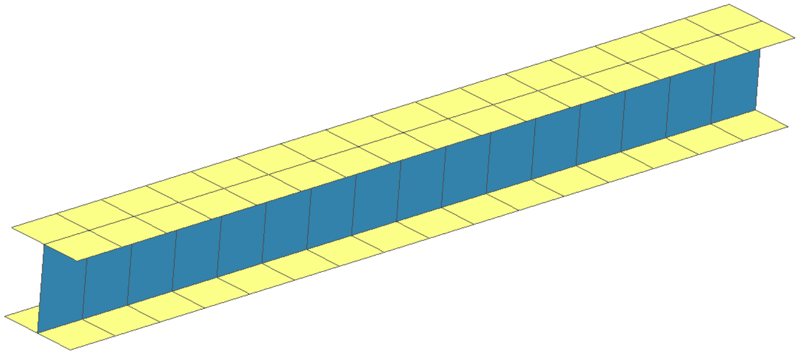
pan 0.000000 0.000000;





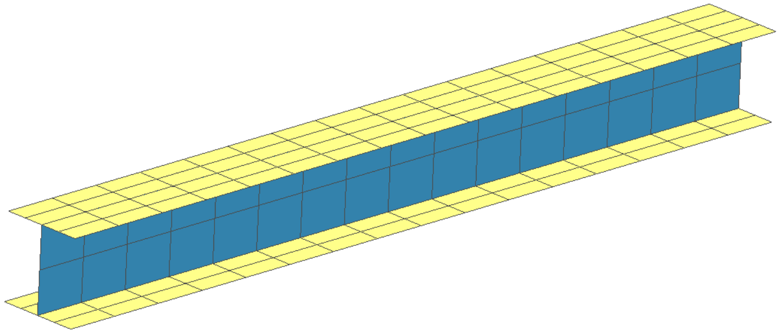
6.2.3 Workshop: 3 - Building the Better Beam

Objective: The importance of mesh density and plate out-of-plane integration will be demonstrated through the use of this simply-supported 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 course model and then modify the mesh density and then finally change the element formulation (*elform*) from under-integrated (*elform* = 2) to fully-integrated (*elform* = -16). The results are surprising, and one has to think about the element shape function (linear) and how element integration calculates the stiffness of the element.

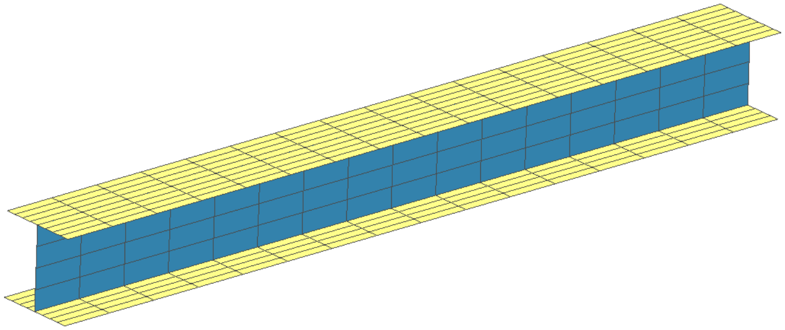


Tasks:

- Open / LSPP / Start / Building the better Beam – Start.dyn in LSPP. Inspect the model and note that it is using shell elements with *elform*=2 (see *SECTION_SHELL keyword command). Run the model and measure the maximum displacement at the end of the beam and record this value on the table below.
- Then, repeat this exercise for the three other models within the folder. Note that each time you run the model in the same folder it will overwrite the existing d3plot files. Record the end displacements for the 2x and 4x mesh refinement models.
- Now, edit the Keyword deck – Start and change its *elform*=-16. Rerun the model and note the end displacement. Do the same for the other two models (2x and 4x).



		Maximum Displacement	
Model	**Mesh Density**	***elform* = 2**	***elform* = -16**
Start	1x	-57.5	-2.7
Refine 2x	2x	-3.1	-2.9
Refine 4x	4x	-3.0	-2.8

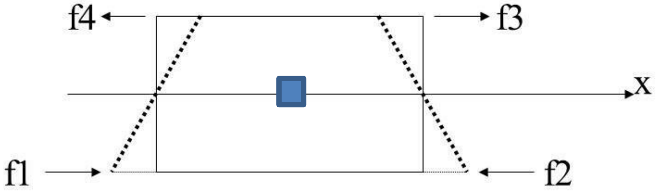


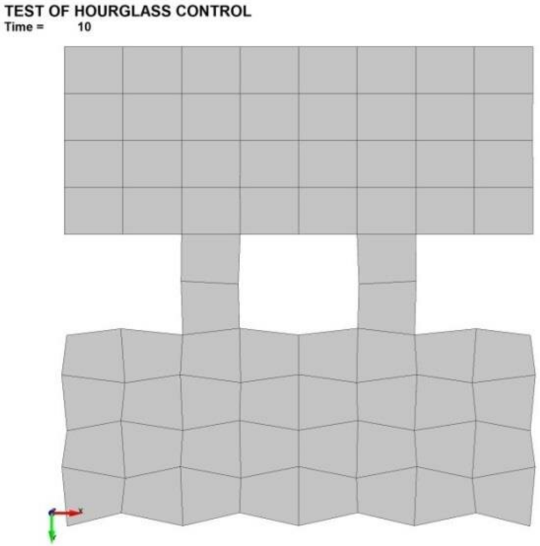
Analyst's Note: It can be surprising to see how the results move around but the mechanics are what they are given the mesh density and the problem physics.

6.2.4 Workshop: 4 - Hourglass Control/Hourglass

Isoparametric[1] Shell/Solid Elements and Hourglassing

- All under-integrated isoparametric elements (one Gauss 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 5% 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.
- For most applications, *hgen*=4 with *qh*=0.03 (see Class Reference Notes / Hourglass).





How to Limit Hourglassing

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

Workshop Tasks:

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

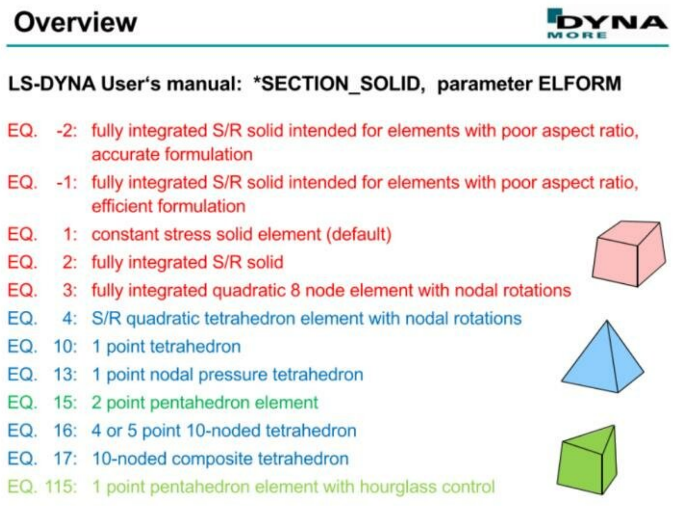
[1] *History Note: According to Ed Wilson (see www.edwilson.org) "The introduction of the isoparametric element formulation by Bruce Irons in 1968 was the single most significant contribution to the field of finite element analysis during the past 40 years."*

6.3 Workshop: 5 – Solid Element Technology – Hex and Tet Formulations

Objective: Be knowledgeable in your selection of solid element formulation (*elform*) whether brick or tetrahedron and its integration scheme.

Introduction: A simple way to become confident in your *elform* selection is to read and then build simple models of the behavior you would like to explore. The model simulates a simple supported beam under uniform loading. We have shell, hex and tet elements. Explore what can be done by simply changing the *elform*. The analytical solution for this model is:

$$\delta_{max} = \frac{5 \cdot \omega \cdot L^4}{384 \cdot E \cdot I} = \frac{5 \cdot 10 \cdot 10^4}{384 \cdot 1e7 \cdot 0.0013021} = 0.10\ inch$$



Overview

LS-DYNA User's manual: *SECTION_SOLID, parameter ELFORM

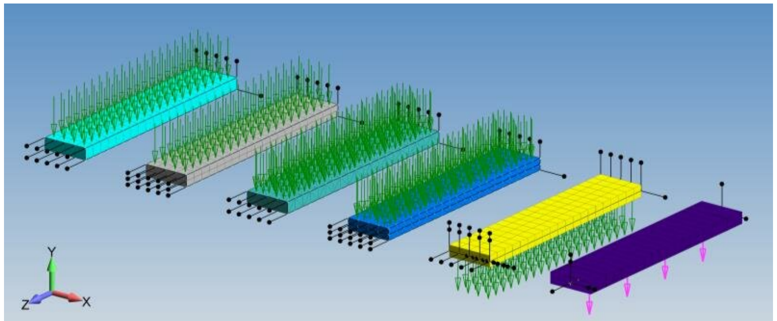
EQ. -2: fully integrated S/R solid intended for elements with poor aspect ratio, accurate formulation
EQ. -1: fully integrated S/R solid intended for elements with poor aspect ratio, efficient formulation
EQ. 1: constant stress solid element (default)
EQ. 2: fully integrated S/R solid
EQ. 3: fully integrated quadratic 8 node element with nodal rotations
EQ. 4: S/R quadratic tetrahedron element with nodal rotations
EQ. 10: 1 point tetrahedron
EQ. 13: 1 point nodal pressure tetrahedron
EQ. 15: 2 point pentahedron element
EQ. 16: 4 or 5 point 10-noded tetrahedron
EQ. 17: 10-noded composite tetrahedron
EQ. 115: 1 point pentahedron element with hourglass control

This paper is in Class Reference Notes / Elements / Solid Elements / Review of Solid Element Formulations Erhart.pdf

Tasks:

- Inspect the starting model in LSPP (Workshop - Solid Element Technology – Start.dyn). We have hex, tet, shell and beam elements);
- Look at the *elform* for each element type and look at the Manual under *SECTION to confirm your understanding of their behavior;
- Then record the end displacements for the models using:
- Run 1 – Hex: *elform* = 1; Tet: *elform* = 10
- Run 2 - Hex: *elform* = -18; Tet: *elform* = 13
- Run 3 – Shell – *nip* = 1

Please note that one should pick the lower, right-hand corner node of the solid and shell elements and, since the last one is a beam element, just the end node. If picked per the video, one will notice that the node numbers on the plot will be 1 to 6 going from left to right as picked on the model.



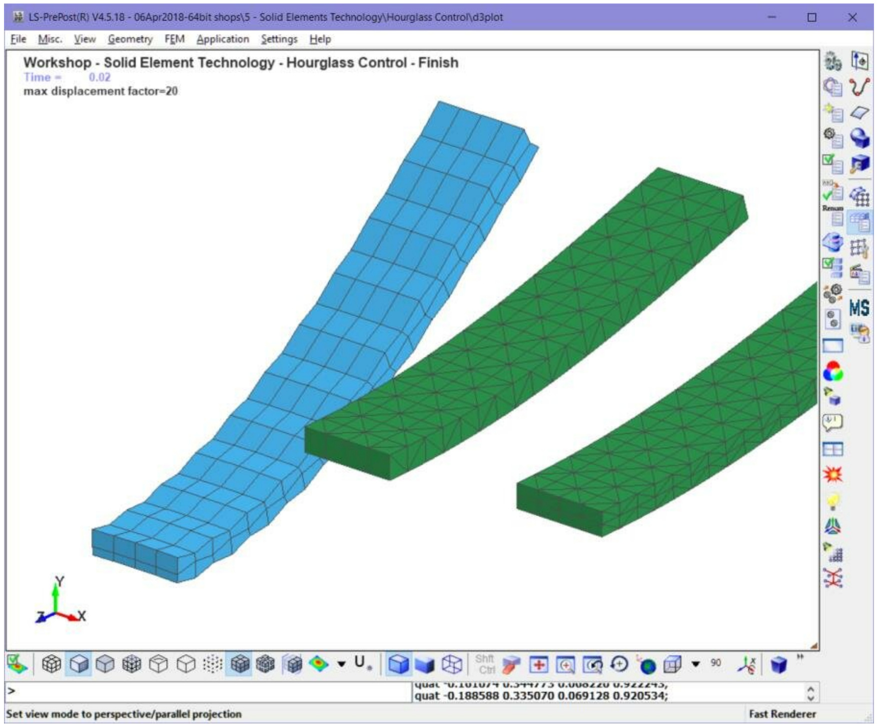
Run	Description	Hex – One	Hex - Two	Tet - One	Tet - Two	Shell	Beam
1	Hex: *elform* = 1; Tet: *elform* = 10						
2	Hex – *elform* =-18 Tet – *elform* = 13						
3	Shell – *nip* = 1						

6.3.1 Workshop 5 – Solid Element Technology – Hourglass Control

Objective: Leveraging the prior workshop on Hourglass Control, we gain a better understanding on how hourglassing works on standard explicit models

Introduction: When using under-integrated elements, hourglassing is real. A little side note is that hourglassing doesn't exist for tetrahedral elements (RTM). We didn't cover this at the beginning of the Workshop to keep the information flow manageable. The image on the left shows Run 1 with the displacement scaled by 20x (Please note that the one-layer hex model has flown off the screen at 20x!).

Your job is to add Hourglass Control and Re-Analyze.


LS-PrePost(R) V4.5.18 - 06Apr2018-64bit shops\5 - Solid Elements Technology\Hourglass Control\d3plot

Workshop - Solid Element Technology - Hourglass Control - Finish
Time = 0.02
max displacement factor=20

```
>
quat -0.181074 0.344773 0.008220 0.922243;
quat -0.188588 0.335070 0.069128 0.920534;
```

Set view mode to perspective/parallel projection
Fast Renderer

Tasks:

- Run Start model in file folder Hourglass Control. Scale model by 20x.
- Edit deck and add Hourglass Control of *lhq* =4 and *qh* = 0.1 and also enable the calculation of hourglass energy (*hgen* = 2). One will also note that _GLSTAT has been set to write out results.
- Run model and note results. Check Internal vs Hourglass Energy
- Re-run model with *qh* = 0.01

Run	Description	Hex – One	Hex - Two	Hourglass Energy Acceptable?
1	*Ihq* = 4 / *qh* = 0.1			
2	*Ihq* = 4 / *qh* = 0.01			

Analyst's Note: If you have time take a look at Pathology of elform 10 and also Implicit file folders. There are notes within each file folder as to what is what. If it doesn't make sense, ask me a few questions. If you have even more time, take a look at the 10-Node tetrahedral folder.