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 (implicit) and dynamic (implicit/explicit) 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:**
ANSYS®, LS-DYNA® and LS-PrePost® are registered and protected trademarks of ANSYS.

**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

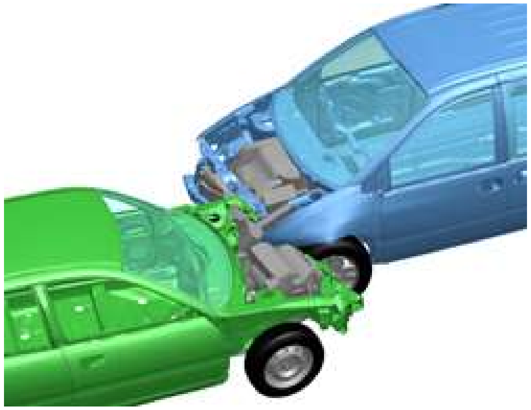
- The class covers the basics in a hands-on manner as taught by engineers that has had to live by what they have validated.
- Each day (four hour session) will have three to four Workshops. Each Workshop is part theory, part demonstration and part hands-on practice. Videos are provided for most Workshops 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)

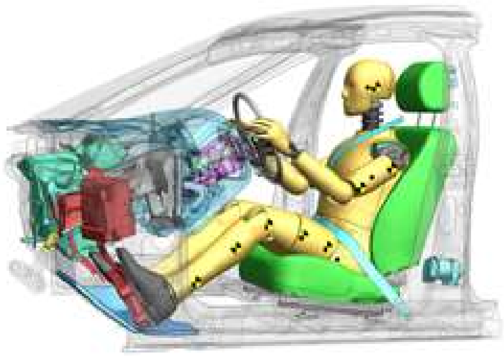
- You are already way ahead of the pack by simply attending this class. You have started on the journey of how to be more successful with LS-DYNA. It is this simple. To be successful, as far as we know, it requires:
  - Reading (very traditional but with LS-DYNA it is necessary to read the manual (RTM), 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

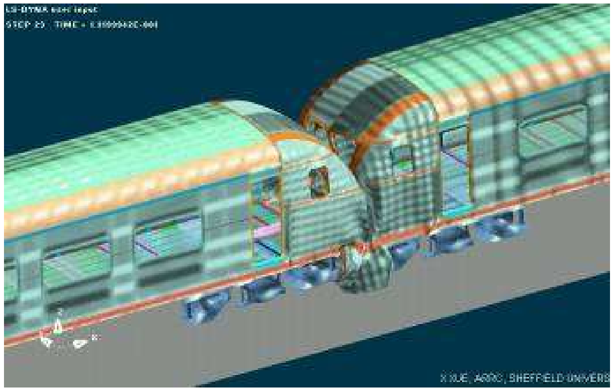
Crashworthiness



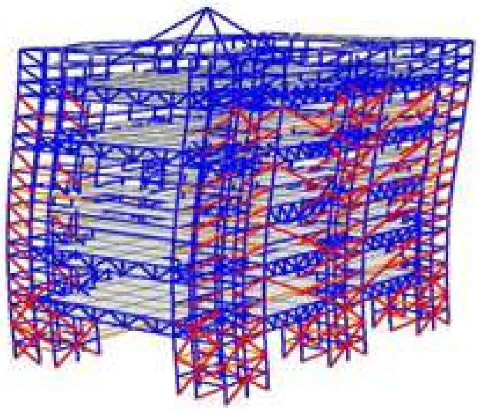
Driver Impact



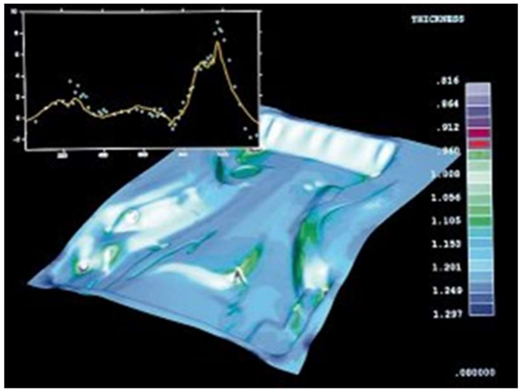
Train Collisions



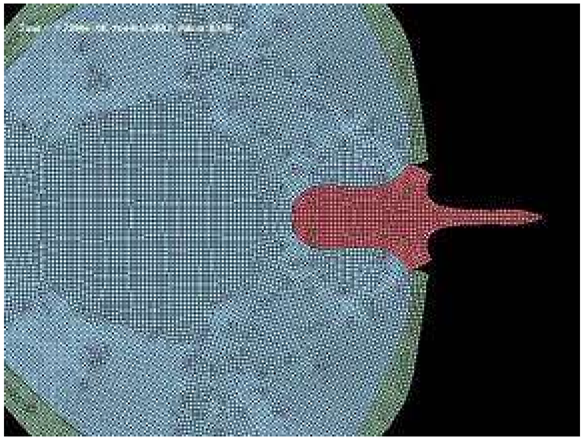
Earthquake Engineering



Metal Forming

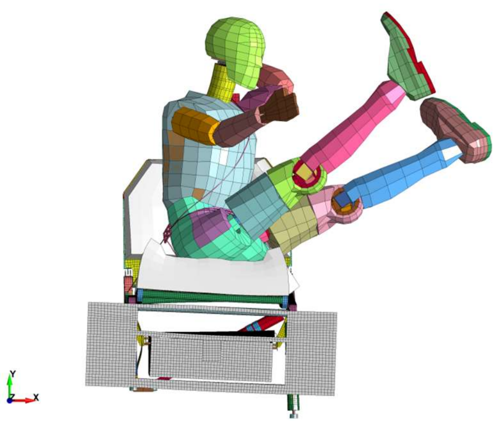


Military

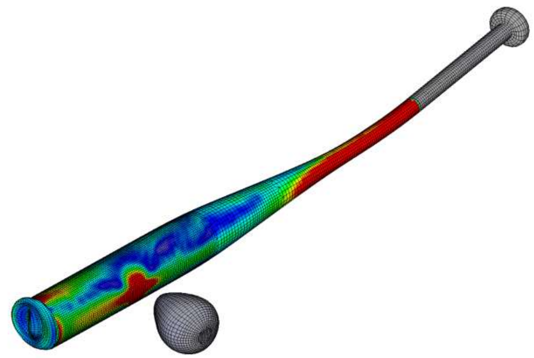


## 1.6 Specific Applications (Courtesy of Predictive Engineering)

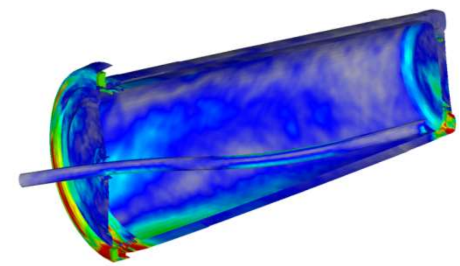
CS-FAR 25 16g Sled Analysis



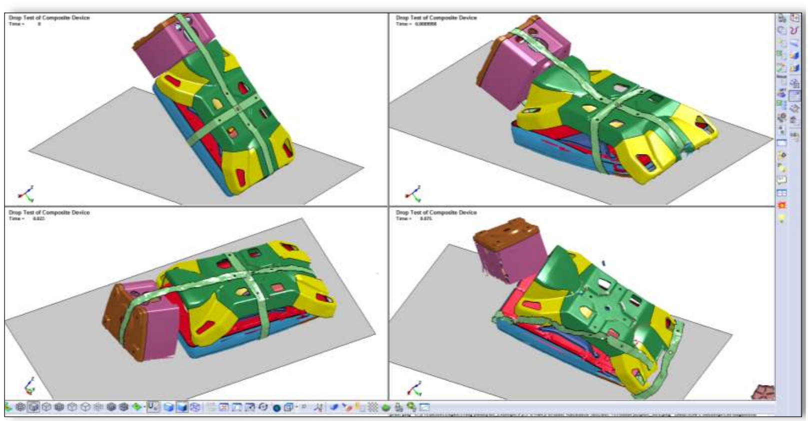
Sporting Goods Equipment



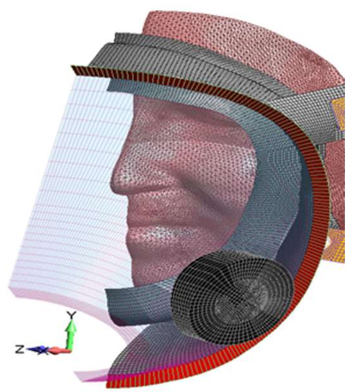
Drop Test Consumer Products



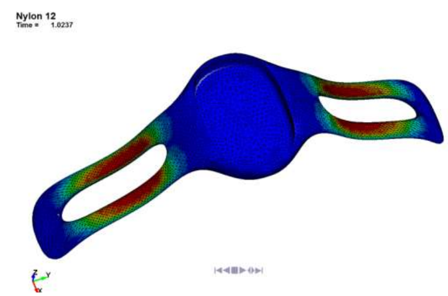
Drop Test of Composites / Electronics



Human Biometrics



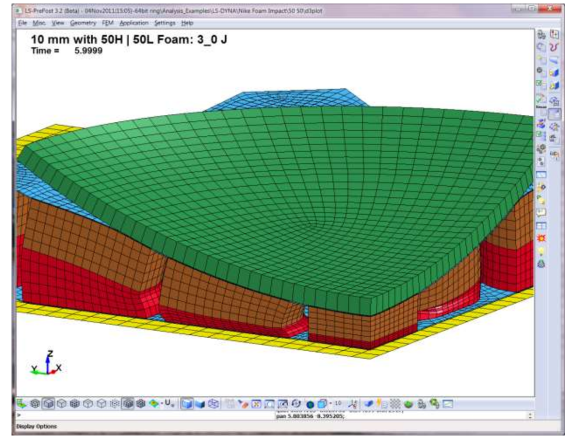
Large Deformation of Plastics



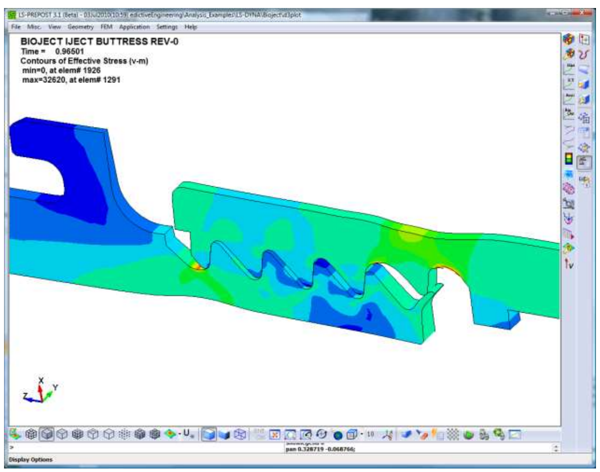
## Crash Analysis of Cargo Net


## Drop Test of Nuclear Waste Container

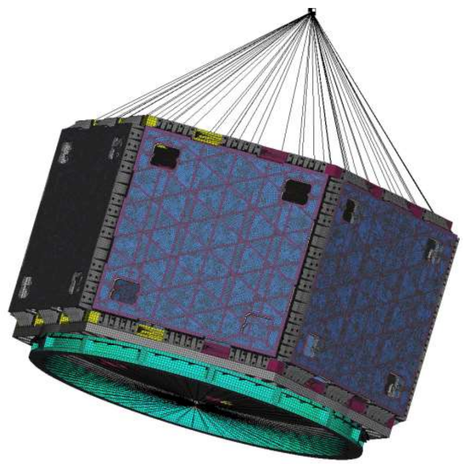

## Impact Analysis of Foams



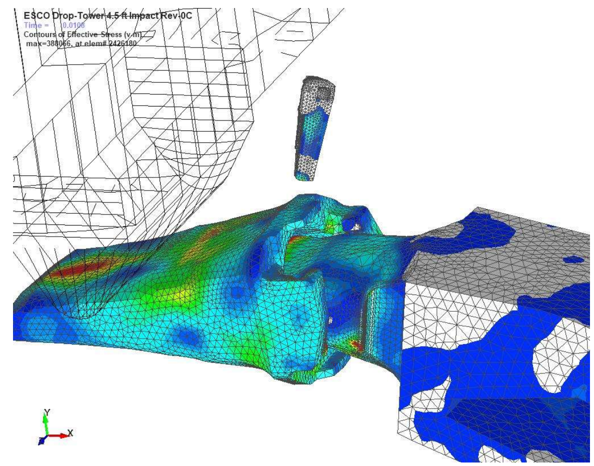

## Plastic Thread Design



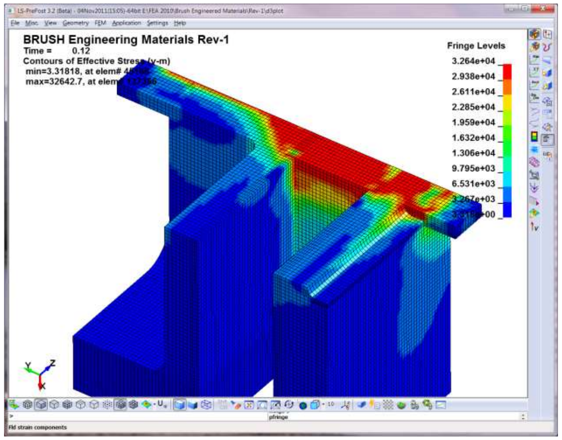

## PSD / Modal Analysis



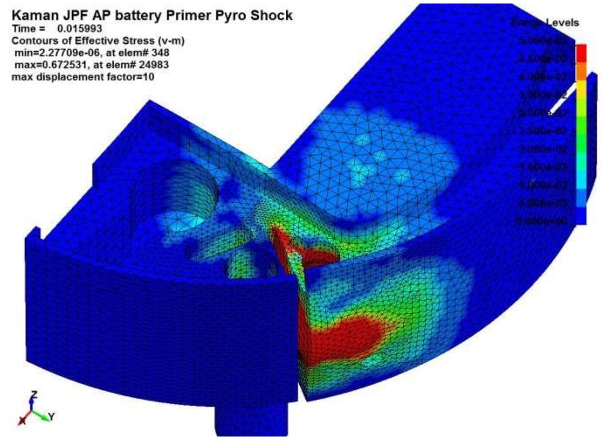
## Digger Tooth Failure



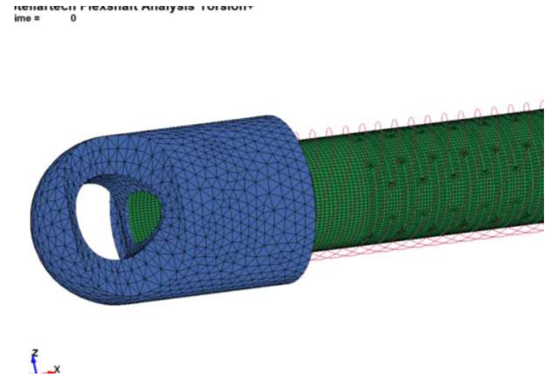
Electron Beam Welding



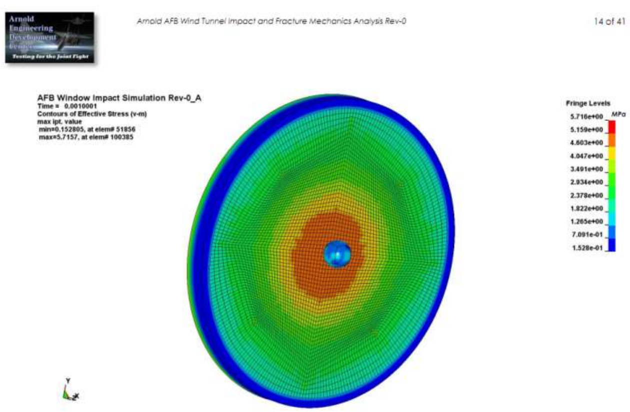

Pyro-Shock Analysis



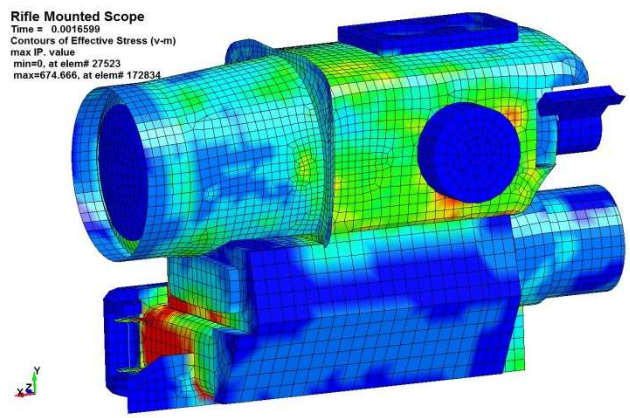

Medical Equipment



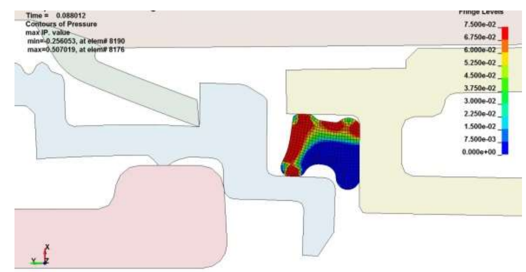
Fracture Mechanics of Glass



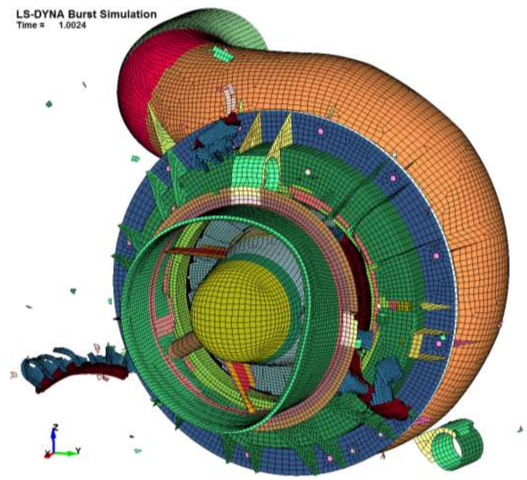

Ballistic Shock Loading of Optical Equipment



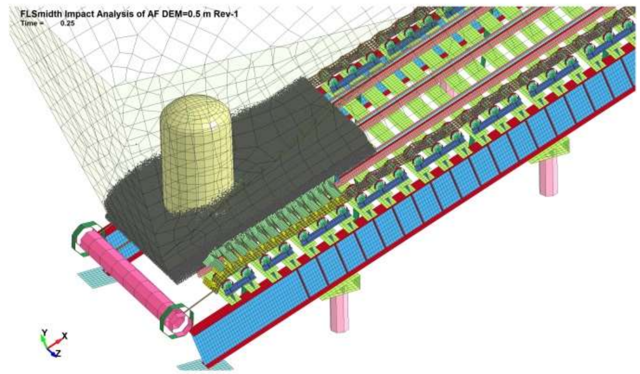

Hyperelastic Medical Seal Analysis



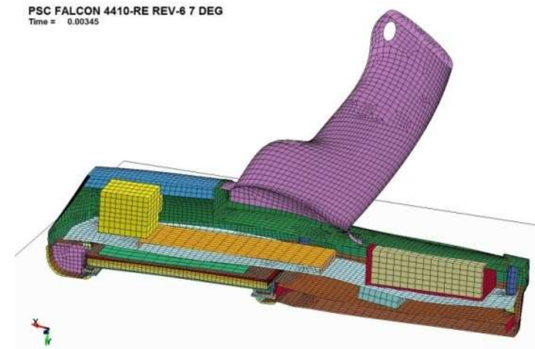
## Blade-Out Analysis



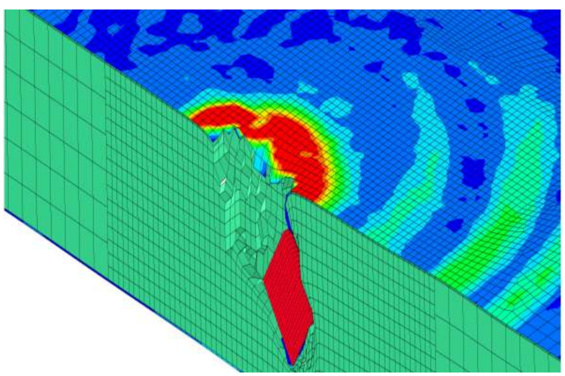

## Discrete Element Method for the Mining Industry



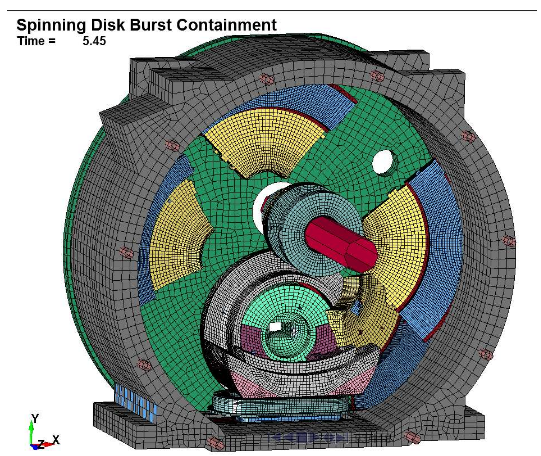

## Drop-Test of Handheld Electronics



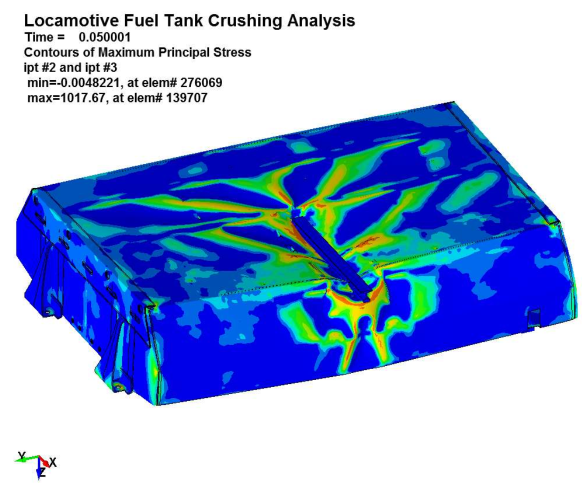

## Ballistic Penetration



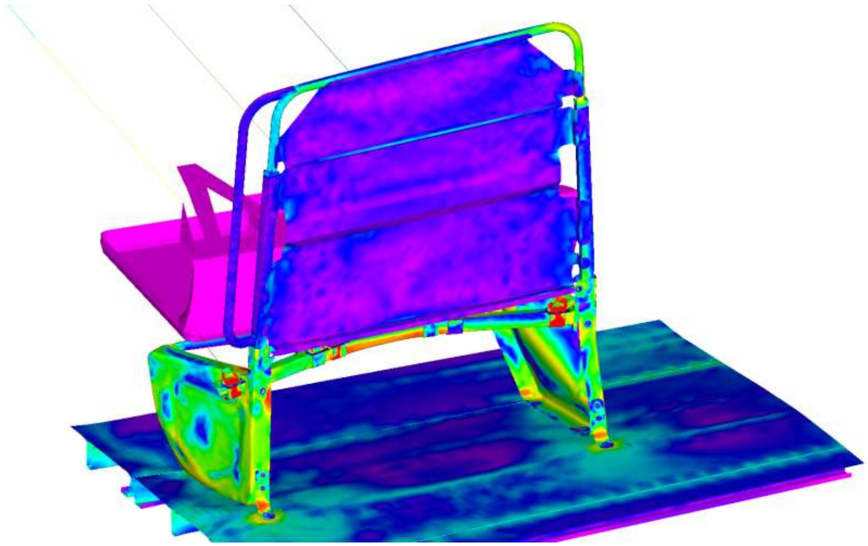
## High-Speed Spinning Disk Containment



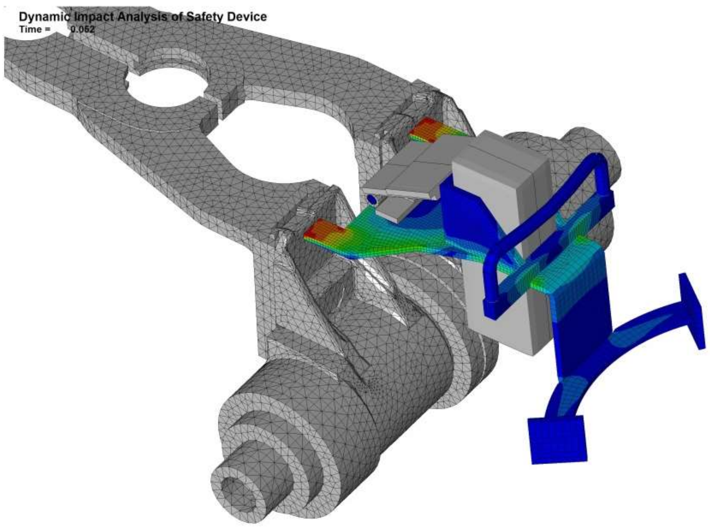

## Locomotive Fuel Tank



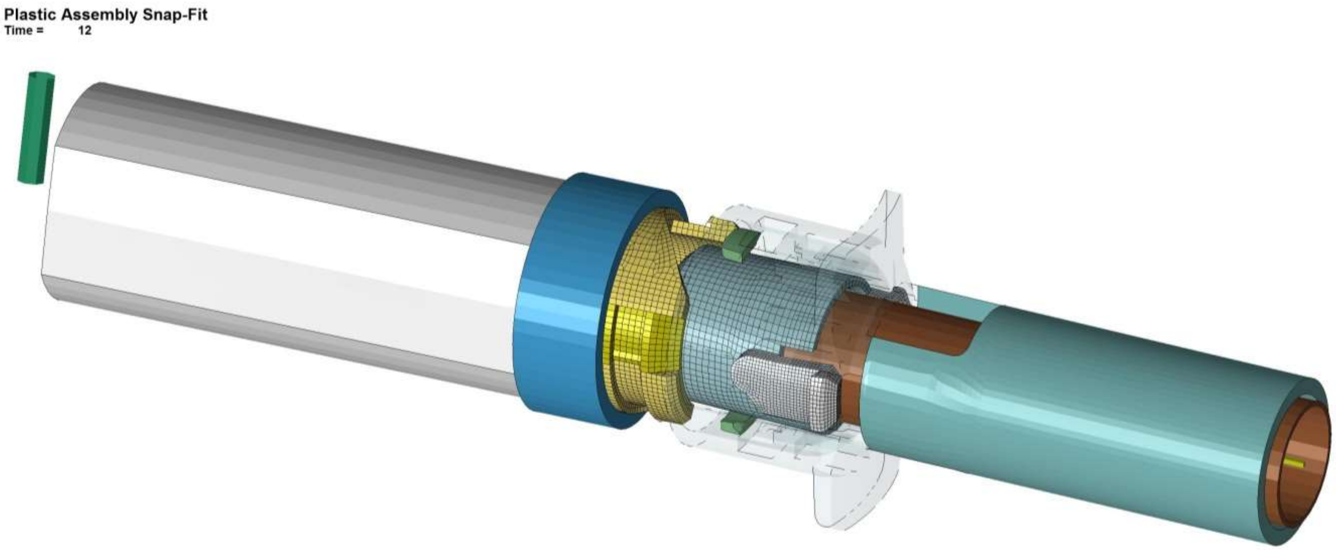
FMVSS Virtual Testing of Bus Seats



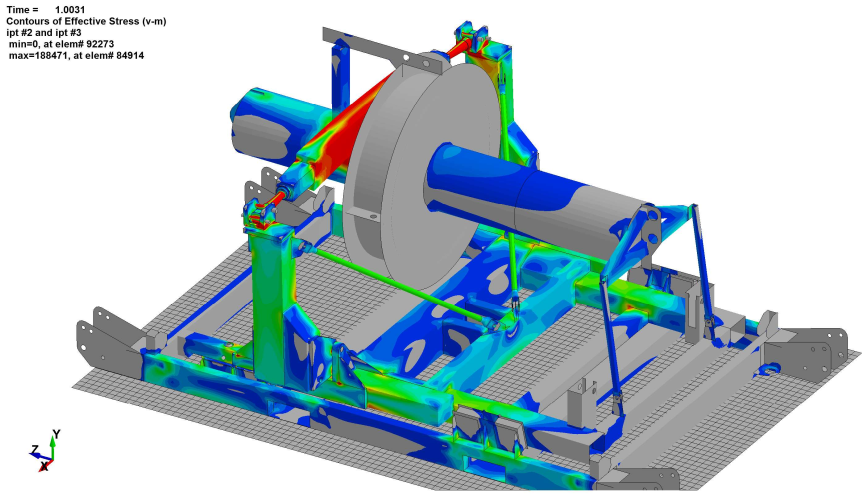
Impact Analysis of Safety Block Device



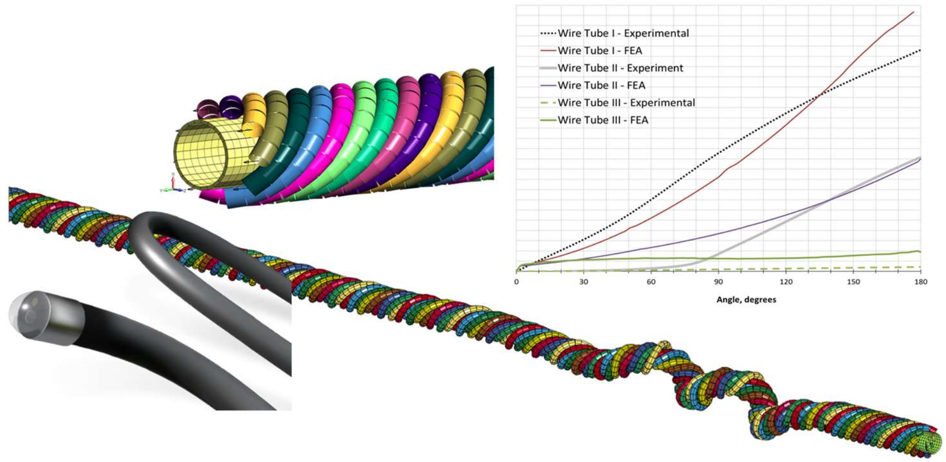

Snap-Fit Analysis – All Plastic Medical Device



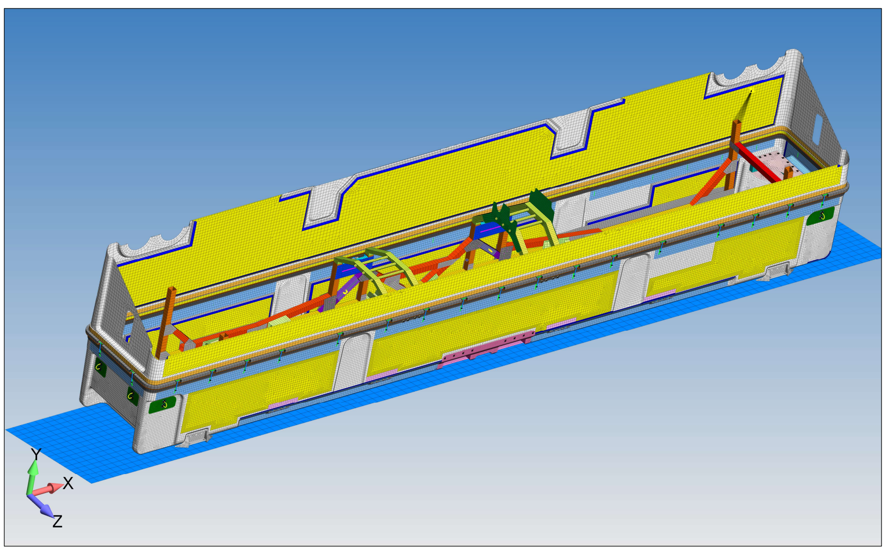
## 9g Crash Analysis of Jet Engine Stand

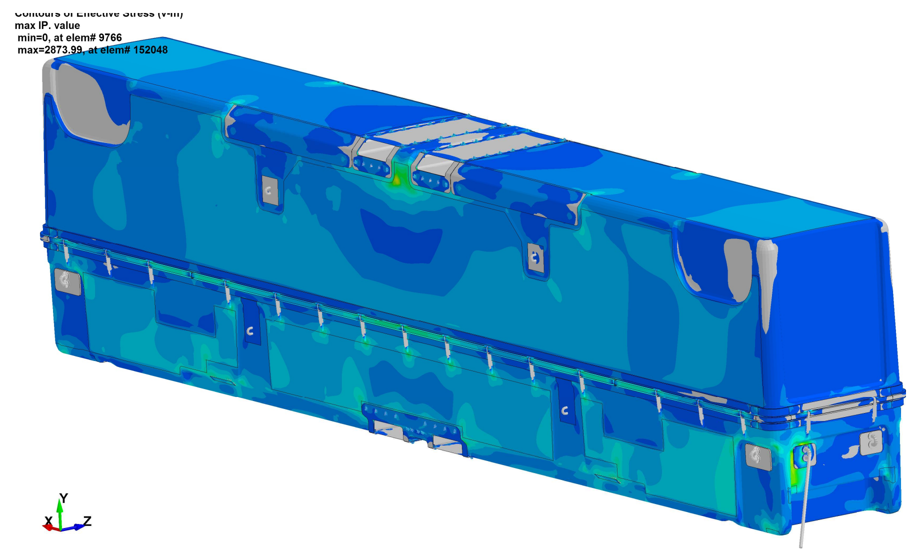



## Torque Analysis of Endoscopic Medical Device

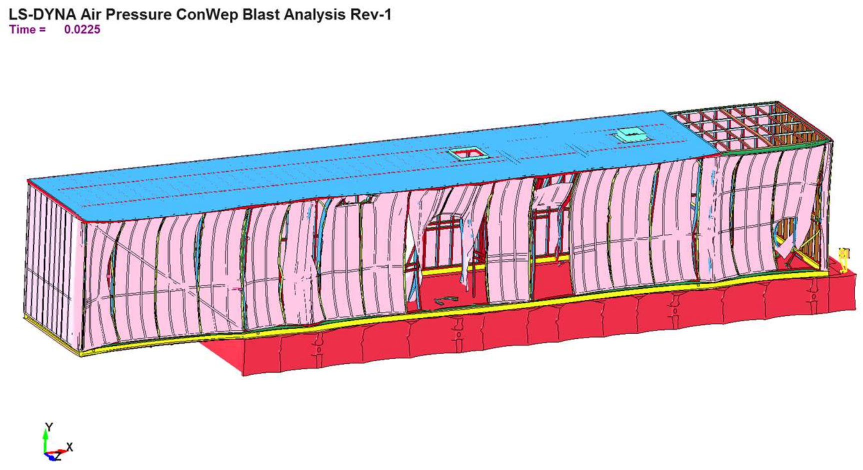



## Drop, Rail Impact and PSD Analysis of Composite Container

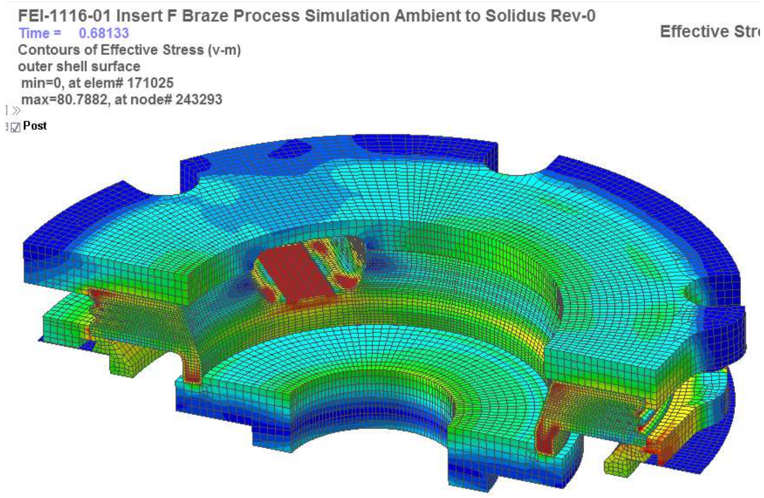


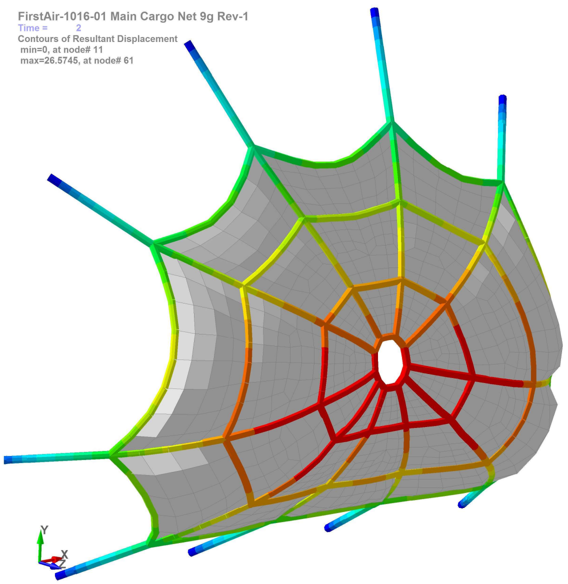
## ConWep Air Pressure Blast Analysis of Generator Housing



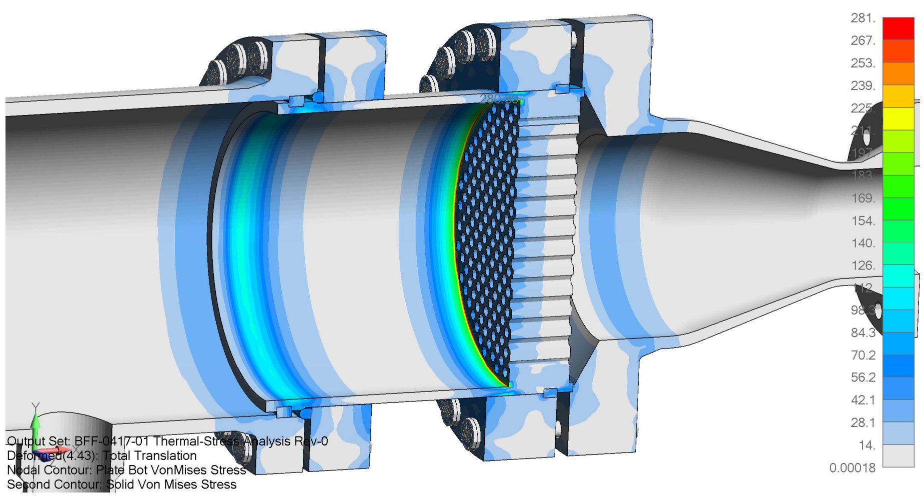

## Alumina-Stainless Steel Braze Process Simulation



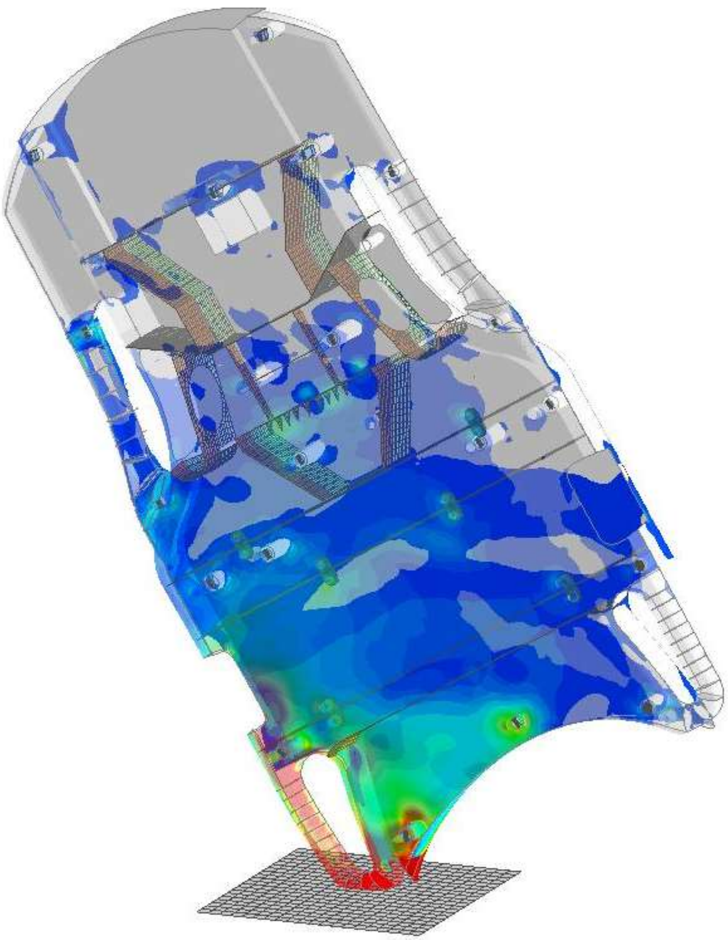

## Air Freighter 9g Cargo Net Analysis



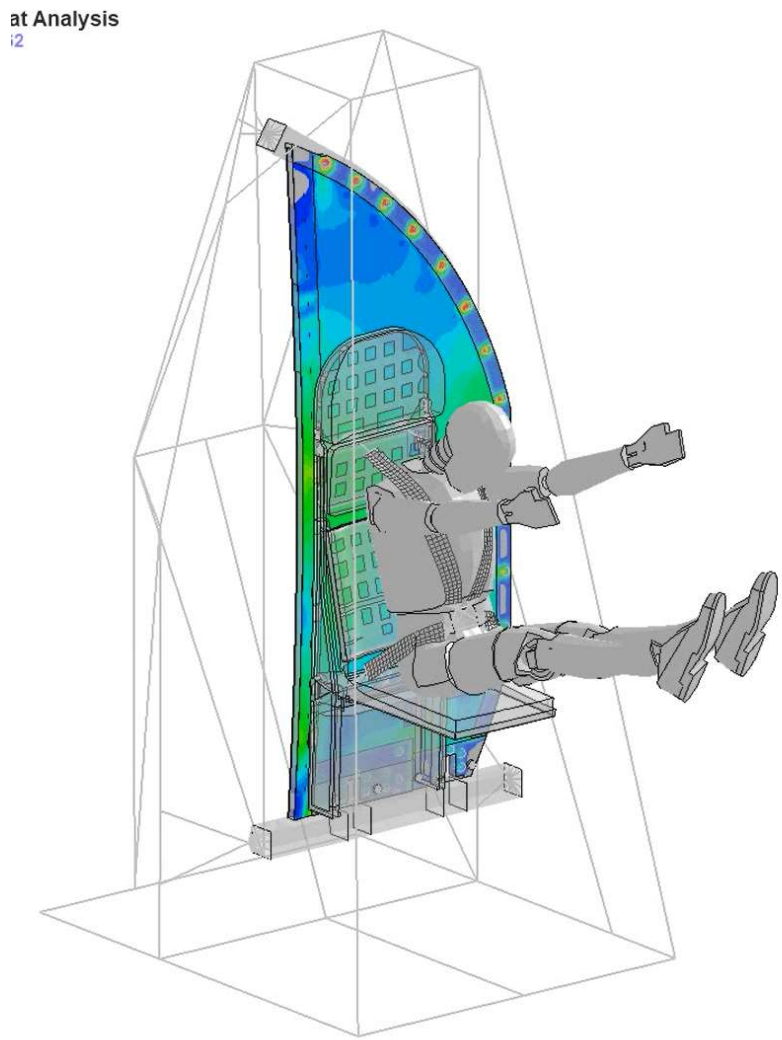

## Thermal-Stress Fatigue Analysis of ASME Evaporator Vessel



Drop-Test of First-Responder Medical Equipment



Composite Lavatory Wall with Attendant Seat 16g Sled Test



# 2. WHAT IS LS-DYNA?

LS-DYNA is a finite element analysis (FEA) solver. It is the motor that generates results based on what the user provides as input. In other words, it is not a 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.

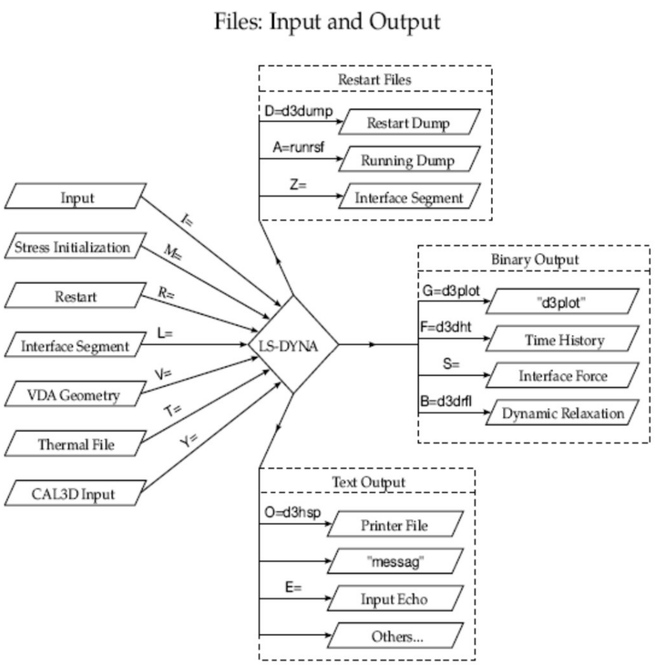



**Figure 2-2.** Files Input and Output.

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:

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

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:

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

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.*

*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.*

LS-DYNA®
KEYWORD USER'S MANUAL

VOLUME I

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

Livermore Software Technology Corporation (LSTC)

## 4.2 Keyword Syntax

- Commands are strings of words separated by an underscore, e.g., *BOUNDARY_PRESCRIBED_MOTION_RIGID.
- Text can be uppercase or lowercase
- Commands are arranged alphabetically in User's Manual
- Order of commands in input deck is ***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.*