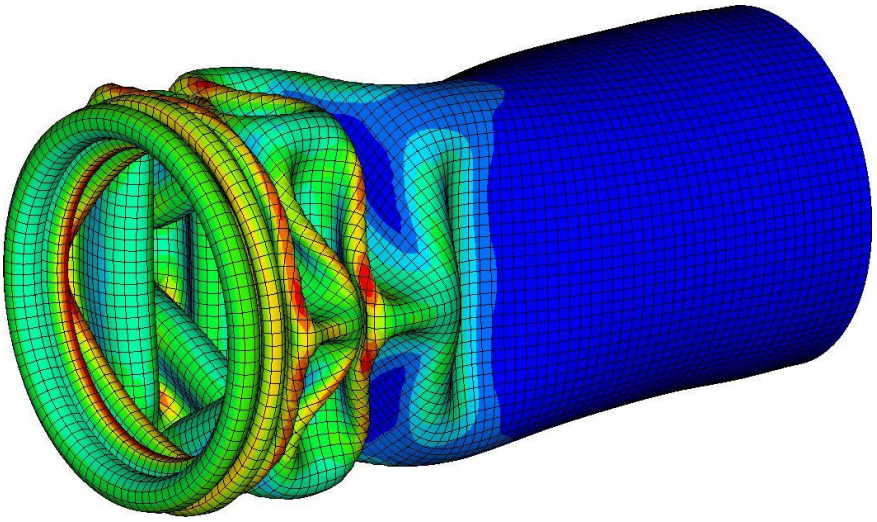
Basic Tutorials

LS-DYNA / LS-PrePost

Ex. 0. Introduction



Contents

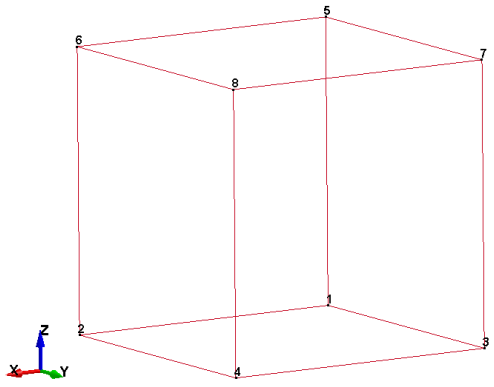
1 LS-DYNA

LS-DYNA from Livermore Software Technology Corporation (LSTC), is a highly advanced general purpose nonlinear finite element program that is capable of simulating complex real world problems. The distributed and shared memory solver provides very short turnaround times on desktop computers and clusters operated using Linux, Windows and UNIX.

LS-DYNA is suitable to investigate phenomena involving large deformations, sophisticated material models and complex contact conditions for structural dynamic problems. LS-DYNA allows switching between explicit and different implicit time stepping schemes. Disparate disciplines, such as coupled thermal analyses, Computational Fluid Dynamics (CFD), fluid-structure interaction, Smooth Particle Hydrodynamics (SPH), Element Free Galerkin (EFG), Corpuscular Method (CPM), Discrete Element Method (DEM) and the Boundary Element Method (BEM) can be combined with structural dynamics. By determining product characteristics before a prototype is built, LS-DYNA is the key to reducing time to market for many products. Carrying out investigations with the aid of LS-DYNA supports the design of robust products with superior performance. For pre- and post-processing, LS-DYNA comes with the LS-PrePost tool. LS-PrePost can be utilized to generate inputs and visualize numerical results

1.1 Keyword format input files

An LS-DYNA input file is a text-file in so called Keyword format usually with a .k, .key or .dyn suffix, e.g. car_crash.k. The keyword format is described in the LS-DYNA Keyword User's Manual, which has three volumes. A short overview of the basic structure of such an input file for a basic 1 element finite element model is given below.



A finite element model in LS-DYNA is built up by different keywords, which is defined for all ingoing definitions and parameters in a model (e.g. ***PART**, ***NODE**). The following description and the map on next page may help to understand the data structure in LS-DYNA.

Consider a cube consisting of one element with eight node points as shown in the figure. We have 1 part, the cube, and use the ***PART** keyword to begin the definition of the finite element model. The keyword

***PART** contains data that points to other attributes of this part, e.g. material properties. Keywords for these other attributes, in turn, point elsewhere to additional attribute definitions. The organization of the keyword input for the cube looks like this:

```
*PART
$#      pid     secid       mid
         1         1         1
*SECTION_SOLID
$#    secid    elform
         1         1
*MAT_ELASTIC
$#      mid        ro         e        pr
         1    7850.0   2.10E11       0.3
*ELEMENT_SOLID
$#    eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       2       4       3       5       6       8       7
*NODE
$#    nid               x               y               z
         1             0.0             0.0             0.0
         2             1.0             0.0             0.0
         3             0.0             1.0             0.0
         4             1.0             1.0             0.0
         5             0.0             0.0             1.0
         6             1.0             0.0             1.0
         7             0.0             1.0             1.0
         8             1.0             1.0             1.0
```

The LS-DYNA Keyword User Manual, volume 1, should be consulted now for a description of the keywords used in the figure. The manual is available in through e.g. LS-RUN, see section 2. A brief description follows:

- ***PART:** We have one part with identification **pid=1**. This part has attributes identified by section identification **secid=1** and material identification **mid=1**.
- ***SECTION_SOLID:** Parts definitions that reference **secid=1** are defined as constant stress 8 node brick elements (**elform=1**).

- ***MAT_ELASTIC:** Parts definitions that reference **mid=1** are defined as an elastic material with density, Young's modulus and Poisson's ratio.
- ***ELEMENT SOLID:** The element with identification **eid=1** are defined by **nid=1** to **nid=8** and belongs to **pid=1**.
- ***NODE:** The node identified by **nid** has coordinates x,y,z.

Boundary conditions and time dependent loads are also set by keywords and are usually applied on nodes, elements, segments or parts. Set definitions are often used to define groups of these entities. Since all loads are time dependent, curves need to be defined that states time vs load unit (force, pressure etc.)

One example of defining boundary conditions and loads on the cube are visualized in the figure below. The LS-DYNA Keyword User Manual should be consulted now for a description of the keywords used in the figure

```
*BOUNDARY_SPC_SET
$#     nsid       cid      dofx      dofy      dofz     dofrx     dofry     dofrz
         1         0         0         1         0         1         0         1
*SET_NODE_LIST
$#      sid
         1
$#     nid1      nid2      nid3      nid4
         1         2         5         6
*LOAD_SEGMENT
$#     lcid        sf        at        n1        n2        n3        n4        n5
         1       1.0       0.0         4         8         7         3         0
*DEFINE_CURVE
$#     lcid
         1
$#               a1                  o1
              0.0                 0.0
              1.0                10.0
```

About the keywords above:

- ***BOUNDARY_SPC_SET:** The node set with identification **nsid=1** are constrained in y-translation and x- and z-rotations.
- ***SET_NODE_LIST:** This keyword defines that node 1, 2, 5 and 6 belongs to node set **sid=1**.
- ***LOAD_SEGMENT:** A pressure load is applied on a segment that are defined by node 4, 8, 7 and 3.
- ***DEFINE_CURVE:** The curve consists of two points that defines the time vs pressure. This curve with identification **lcid=1** are used for the load.

1.2 Consistent units

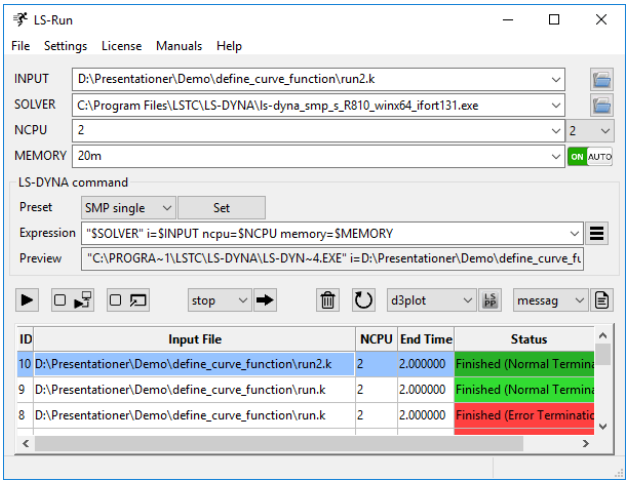
LS-DYNA requires that the set units used is consistent with Newton's second law of motion (f=m*a). Different unit systems are shown in the table below. In these tutorials, we will use the **S2** system.

	S1	S2	S3
length	meter	millimeter	millimeter
time	second	second	millisecond
mass	kilogram	tonne	kilogram
force	Newton	Newton	kiloNewton
Young's modulus of steel	210.0E+09	210.0E+03	210.0
density of steel	7.85E+03	7.85E-09	7.85E-06
gravitation	9.81	9.81E+03	9.81E-03

2 LS-RUN – Running LS-DYNA and access to manuals

On Microsoft Windows, the **LS-RUN** software is the preferred method to start your LS-DYNA simulation. **LS-RUN** also provides:

- Queueing system, to manage you LS-DYNA simulations.
- Managing of running and finished simulations, including starting LS-PrePost et c.
- Access to all LS-DYNA manuals.
- Install and check the LS-DYNA license and license server.



To quickly start an LS-DYNA simulation using LS-RUN: just drag and drop the keyword file on the INPUT-field and then select you preferred solver (SMP single precision version in the above image). Finally press the play button ▶ to start the simulation.

3 LS-PrePost

- LS-PrePost is an advanced pre- and post-processor designed specifically for LS-DYNA
- LS-PrePost is developed for Windows and Linux
- LS-PrePost is ***Free***
- Core Functionality

- Full support of LS-DYNA keyword files
- Full support of LS-DYNA result files
- Robust handling of geometry data (new CAD engine)
- Pre-processing (meshing, model clean-up, entity creation)
- Post-processing (animation, fringe plotting, curve plotting)

3.1 Input and output

- **Input (partial list)**
  - FEM: LS-DYNA Keyword, Nastran, I-DEAS Universal, PAM-CRASH, RADIOSS, ABAQUS
  - CAD: IGES, STEP
  - ASCII: glstat, matsum, etc.
  - Binary: d3plot, binout, etc.
- **Output (partial list)**
  - FEM: LS-DYNA Keyword, Nastran
  - Image: PNG, TIFF, BMP, GIF, JPG, PostScript
  - Movie: AVI, MPEG, Animated GIF, JPEG
  - XY Data: CRV, CSV, XML
  - CAD: IGES, STEP, STL
  - Other: Post.db, Project File

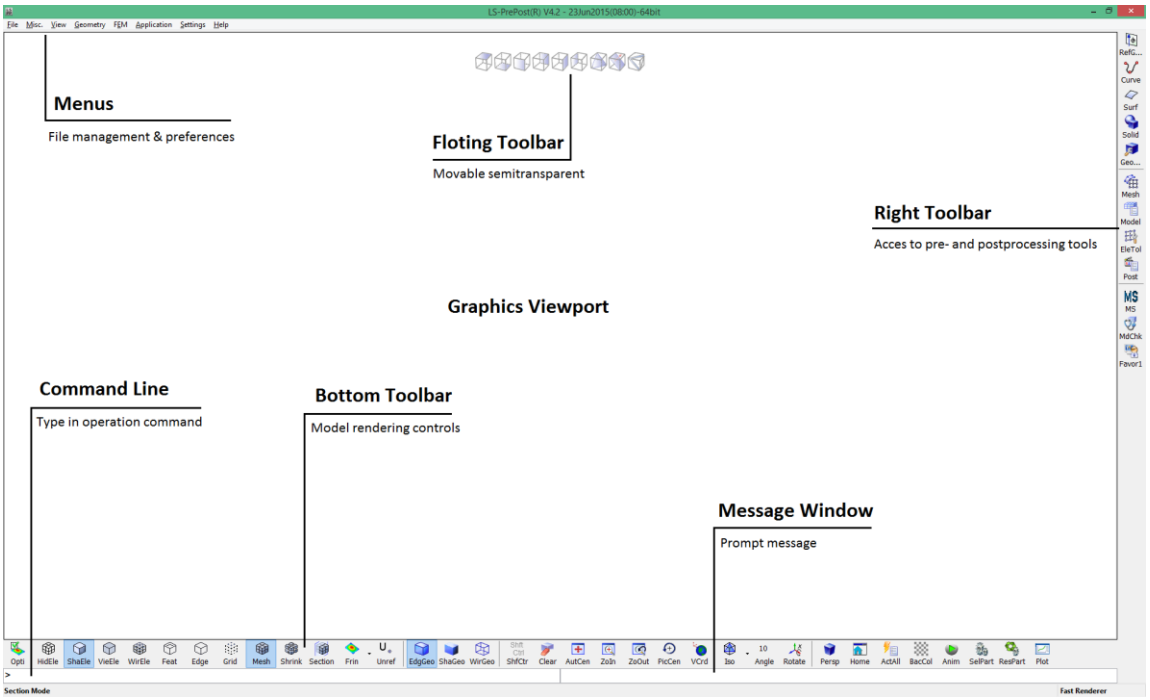
3.2 Mouse and Keyboard

- **Dynamic Model Operation**
  - Rotate: Shift + Left-click
  - Translate: Shift + Middle-click
  - Zoom: Shift + Right-click/Scroll-wheel
  - (Using Ctrl instead of Shift for edge mode)
- **Graphics Selection**
  - Pick (single): Left Click
  - Area (rectangle): Left-click + Drag
  - Poly (polygon): Left-click at corners / Right-click to finish
- **List Selection**
  - Multi-Select: Left-click + Drag / Ctrl + Left-click
- **Mouse over controls for status bar help comments**

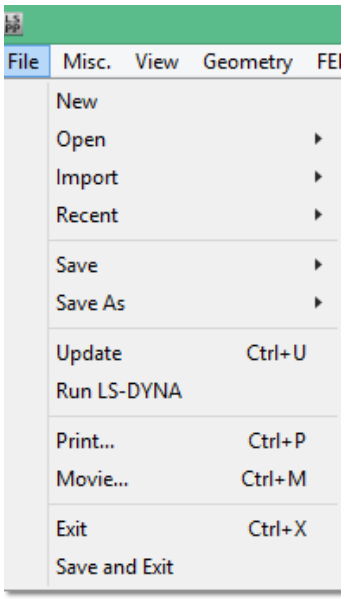
3.3 Graphical user interface

The graphical interface of LS-PrePost is shown in the figure below. On the right hand side, you can see the main toolbar. When clicking on one of these, a sub-toolbar just to the left will be shown. That is the location where you'll find most of the tools needed to create/modify/delete entities in your model.

In the bottom toolbar, you'll find the tools most commonly used to determine how LS-PrePost should render mesh/surfaces, orient the model, etc. There are a couple of the drop-down menus on the top left corner that you will use more or less frequently: File/View/Application/Settings. The Floating Toolbar is used to toggle between different views.



3.4 File Menu



New – Launch a new session of LS-PrePost, all model/data will be closed

Open – Open file (new model created for each file opened)

Import – Import file (adds keyword data to current model)

Recent – Open recent files (stored in /user/.lspp_recent)

Save – Over-write current *Keyword* or *Project* file

Save As – Save any of the following file formats using advanced options: *Keyword, Active Keyword* (visible data), *Project*, *Post.db* (condensed d3plot data), *Geometry, Keyword and Project* (using the same file name).

Update – Load new d3plots for run in progress

Run LS-DYNA – Pop up LS-DYNA job submission dialog, currently only limited to the same local machine LS-PrePost is running

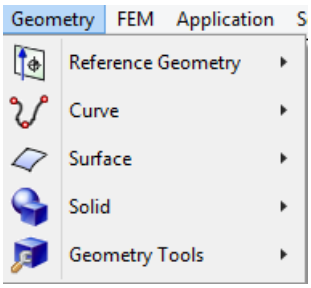
Print... – Launch printing interface (send to printer or image file)

Movie... – Launch movie generation interface

Exit – Exit LS-PrePost

Save and Exit – Save data to current file and exit LS-PrePost

3.5 Geometry Menu



Reference Geometry – Access tools for creating and editing reference geometry (Axis, Plane, Coordinate System, Point, Reference Geometry Edit)

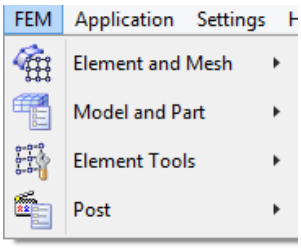
Curve – Access tools for creating and editing curves (Point, Line, Circle, Circular Arc, Ellipse, Elliptical Arc, BSpline Curve, Helix, Composite Curve, Break Curve, Merge Curve, Bridge Edge, Smooth Curve, Middle Curve, Morphing Curve, Fillet Curve, Parabola, Hyperbola, Function, Polygon, Convert, Sketch)

Surface – Access tools for creating and editing surfaces (Plane, Cylinder, Cone, Sphere, Torus, Ellipsoid, Fill Plane, Extrude, Revolve, Sweep, Loft, N-Side Surface, Patch Surface, Bridge Two Faces, Combine Faces, Fit From Points/Mesh, Middle Surface, Surface Morphing, Fit Primary Surface, Break Surface)

Solid – Access tools for creating and editing solids (Box, Cylinder, Cone, Sphere, Torus, Extrude, Revolve, Sweep, Loft, Fillet, Chamfer, Draft, Thicken, Wedge, Boolean, Prism)

Geometry Tools – Access other geometry tools (Delete Face, Blank Entity, Extend Curve, Extend Face, Intersection, Offset, Project, Replace Face, Stitch Faces, Trim, Transform, Reverse Direction, Copy Entity, Management, Heal, Topology Simplify, Measure, Text Object, Array flow)

3.6 FEM Menu

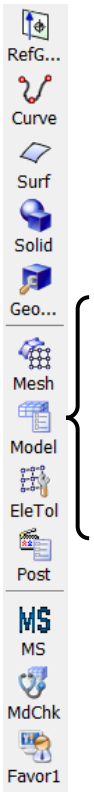


Element and Mesh – Access mesh creation tools (Shape Mesher, Auto Mesher, Solid Mesher, Block Mesher, N-Line Mesher, 2D-Mesher, Tetrahedral Mesher, Blank Mesher, BulkF Mesher, Element Generation, Node Editing, Element Editing, Nurbs, Mass Trimming, Spot Welding, SPH Generation, Disc Sphere Generation, Multiple Solver Mesh, Result Mapping)

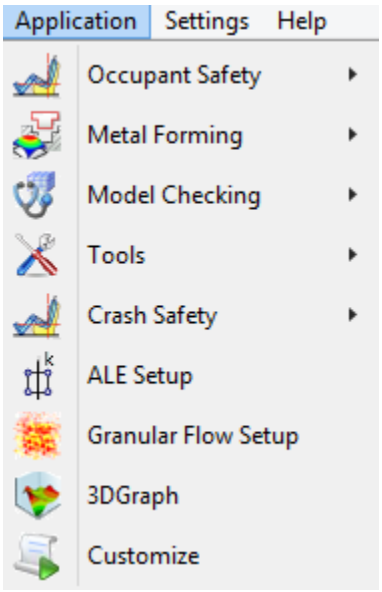
Model and Part – Access model and part tools (Assembly and Select Part, Keyword Manager, Create Entity, Part Data, Display Entity, Reference Check, Renumber, Section Plane, Model Selection, Subsystem Manager, Group, View, Part Color, Appearance, Annotation, Split Window, Explode, Lighting Setup, Reflect Model, Trace Light)

Element Tools – Access element tools (Identify, Find, Blank, Move or Copy, Offset, Transform, Normals, Detach, Duplicate Nodes, Node Editing, Element Editing, Measure, Morph, Smooth, Part Trim, Part Travel, Edge Face, Regionalize)

Post – Access post-processing tools (Fringe Component, Fringe Range, History, XY Plot, ASCII, Binary Output, Follow, Trace, State, Particle, Circle Grid, Chain Model, FLD, Output, Setting, Vector)



3.7 Application Menu



Occupant Safety – Occupant related applications (Airbag Folding, DynaFold, Dummy Positioning, Seat Deformer, Head Impact Positioning, Airbag Impact Setup, Sled Creation, THUMS Positioning, Seatbelt Fitting)

Metal Forming – Setting up metal forming analyses (Easy Setup, Roler Hemming, Die System Module, Scrap Trim Simulation, Best Fit, 3D Drawbead Generation, Blank Size/trim line, Flange Unfolding, New Roller Hemming)

Model Checking – Check contact, element quality, unreferenced entities, etc. (General Checking, ALE Checking)

Tools – Useful tools (Media, DurveGen, jIntegral)

Crash Safety – Crash safety applications (Intrusion Measurements)

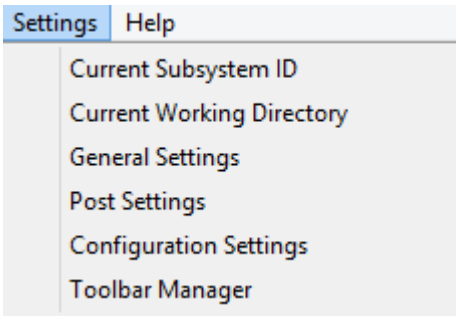
Ale Setup – Setting up Ale (Arbitrary Lagrange-Euler) analyses)

Granular Flow Setup – Setting up granular Flow analyses

3DGraph - Create 3D graphs.

Customize – Read and operate scripting files.

3.8 Settings Menu



Current Subsystem ID – Set current subsystem ID

Current Working Directory – Set current working directory

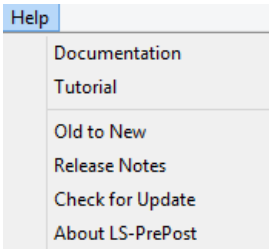
General Settings – Set general parameters

Post Settings – Set post processing parameters

Configuration Settings – Set configuration settings

Toolbar Manager – Customize toolbars

3.9 Extra help



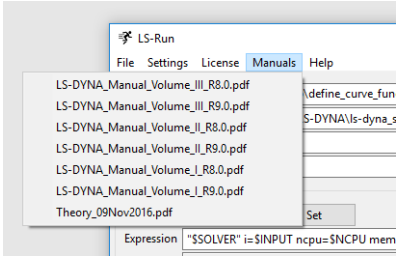
To get more detailed information about LS-PrePost, one can use LS-PrePost's own help system. Located under **Help** in the top menu. Both **Documentation** and **Tutorial** can be used to develop the knowledge in the program.

The first time one want to use this, two files needs to be downloaded. Open **Documentation**, select **Download by LS-PrePost** and click **Download**. This might take some minutes. Do the same thing with **Tutorial**. If this doesn't work, download the files from:

- http://ftp.lstc.com/anonymous/outgoing/lsprepost/ or alternatively
- ftp://ftp.dynamore.se/lsprepost/

If you for example use LS-PrePost version **4.2** on **Windows,** then click **4.2 > doc > win**, then download **Document.chm** and **Tutor.chm**. Move the downloaded files to …LS-PrePost\4.2-x64\resource\HelpDocument (… will be replaced with the folder path where you installed LS-PrePost). You can now go to the **Help** menu in LS-PrePost and open **Documentation** and **Tutorial**.

To get more information about LS-DYNA, the manuals are helpful. Located under **Manuals** in **LS-RUN** (see section 2):



- Vol.1 – Explaining almost every keyword
- Vol.2 – Material Models for LS-DYNA
- Vol.3 – Multi-Physics Solvers
- Theory – information on theory behind LS-DYNA

More workshops for **LS-PrePost** can be found at:

- http://ftp.lstc.com/anonymous/outgoing/lsprepost/Training/Intro/ or alternatively
- ftp://ftp.dynamore.se/lsprepost/Training/Intro/.