

# Getting Started with LS-DYNA Crush Tube Example

The Arup Campus, Blythe Gate, Blythe Valley Park, Solihull, West  
Midlands, B90 8AE

tel: +44 (0) 121 213 3399

email: [dyna.support@arup.com](mailto:dyna.support@arup.com)

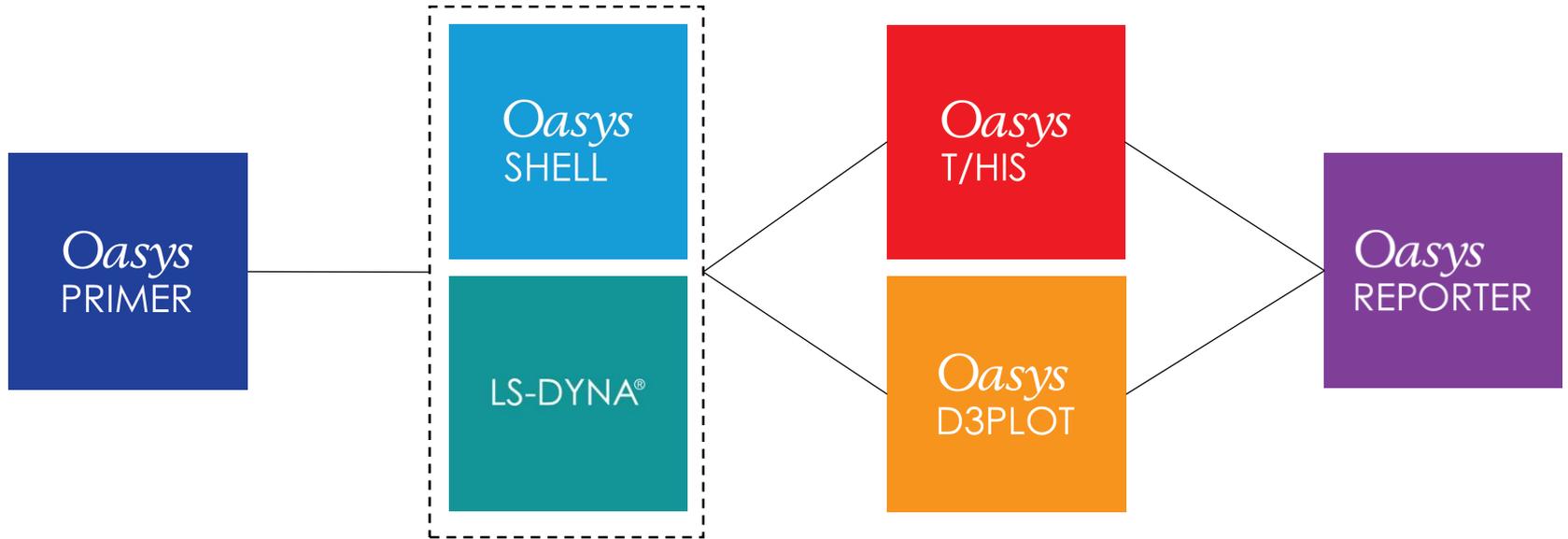
2018

- The purpose of this tutorial is to introduce new users to the basics and modelling workflow of LS-DYNA.
- For demonstration purposes, the Oasys pre- and post-processing software (PRIMER, D3PLOT and T/HIS) are used although any software that supports LS-DYNA can be used (i.e. LS-PrePost).

### Steps covered:

- Pre-processing
  - Import model
  - Section and material cards
  - Boundary and load conditions
  - Contact
  - Control and output (database) cards
  - Write LS-DYNA input deck
- Solver
  - Run Submission
- Post-processing
  - Animate the results
  - Obtain cross section force output
  - Inspect global energy output

An incomplete input deck is provided: **crush\_tube\_start.key**

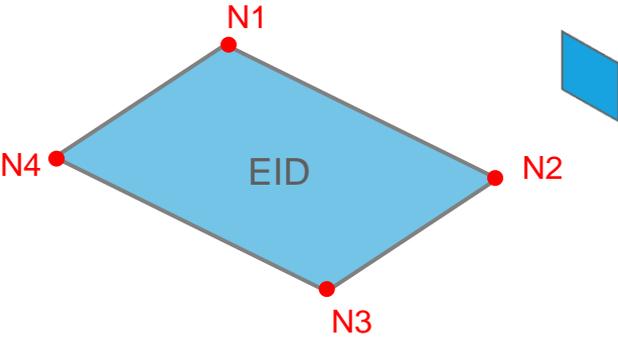


Pre-processing

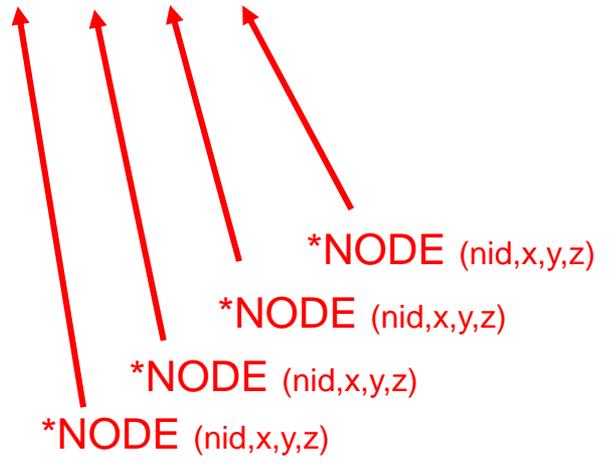
Solver

Post-Processing

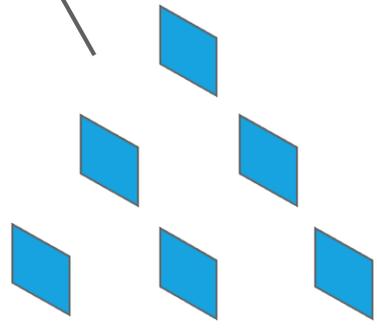
Reporting



**\*ELEMENT\_SHELL** (eid,pid,nid1,nid2,nid3,nid4)



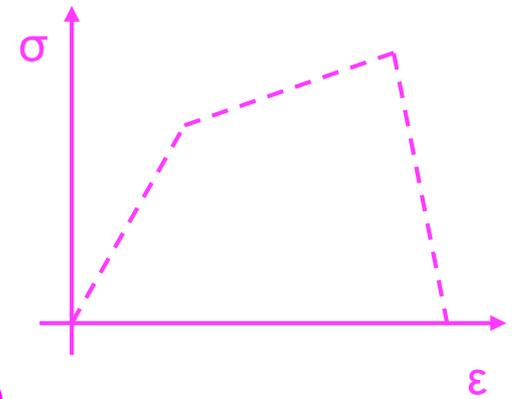
- Parts cannot have multiple element types (e.g. a mix of 2D and 3D elements)
- Parts cannot have elements with different materials or sections
- Parts are a useful way to keep a model organised



**\*PART** (pid,sid,mid)

**\*SECTION\_SHELL**  
(Thickness, formulation)

**\*MAT\_ELASTIC**  
(Material properties)



- LS-DYNA\_manual\_Vol\_I\_....pdf    General
- LS-DYNA\_manual\_Vol\_II\_....pdf    Materials
- (LS-DYNA\_manual\_Vol\_III\_....pdf    Multiphysics)

**Alphabetical list of Keywords**

## \*NODE

**\*NODE**

**\*NODE\_{OPTION}**

Available options include:

<BLANK>

MERGE

Purpose: Define a node and its coordinates in the global coordinate system. Also, boundary conditions in global directions can be specified. Generally, nodes are assigned to elements; however, exceptions are possible, see remark 2 below. The nodal point ID must be unique relative to other nodes defined in the \*NODE section. The MERGE option is usually applied to boundary nodes on disjoint parts and only applies to nodes defined when the merge option is invoked. With this option, nodes with identical coordinates are replaced during the input phase by the first node encountered that shares the coordinate. During the merging process a tolerance is used to determine whether a node should be merged. This tolerance can be defined using the keyword \*NODE\_MERGE\_TOLERANCE keyword, which is recommended over the default value. See the \*NODE\_MERGE\_TOLERANCE input description in the next section.

**Node Cards.** Include as many cards in the following format as desired. This input ends at the next keyword ("\*") card.

Card 1	1	2	3	4	5	6	7	8	9	10

**Options  
Description  
Cards  
Variables  
Remarks**

See variable descriptions below

Character	Hello
Integer (A or A8)	123
Float	1.04e-6

Some cards are additional

Some values are set by default – you don't need to add them

**\*PART** **\*PART**

**Card Sets.** Repeat as many sets data cards as desired (card 1 through 10). This input ends at the next keyword (\*\*\*) card.

Card 1	1	2	3	4	5	6	7	8
Variable	HEADING							
Type	C							
Default	none							
Remarks	1							

Card 2	1	2	3	4	5	6	7	8
Variable	PID	SECID	MID	EOSID	HGID	GRAV	ADDOPT	TMID
Type	I/A	I or A8	I or A8	I or A8	I or A8	I	I	I or A8
Default	none	none	none	0	0	0	0	0

**Inertia Card 1.** Additional Card for the INERTIA option. See remarks 3 and 4.

Card 3	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	TM	IRCS	NODEID		
Type	F	F	F	F	I	I		

**Inertia Card 2.** Additional Card for the INERTIA option.

Card 4	1	2	3	4	5	6	7	8
Variable	IBX	IBY	IBZ	IYX	IYZ	IZZ		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
HEADING	Heading for the part
PID	Part identification. A unique number or label must be specified.
SECID	Section identification defined in the *SECTION section. See Remark 7.
MID	Material identification defined in the *MAT section. See Remark 7.
30-6 (PART)	LS-DYNA R7.1

*PART		*PART	
VARIABLE	DESCRIPTION		
EOSID	Equation of state identification defined in the *EOS section. Non-zero only for solid elements using an equation of state to compute pressure. See Remark 7.		
HGID	Hourglass/bulk viscosity identification defined in the *HOURLASS Section. See Remark 7. EQ.0: default values are used.		
GRAV	Part initialization for gravity loading. This option initializes hydrostatic pressure in the part due to gravity acting on an overburden material. This option applies to brick elements only and must be used with the *LOAD_DENSITY_DEPTH option. EQ.0: all parts initialized, EQ.1: only current material initialized.		

See remarks below

**Remarks:**

- HEADING default is standard material description, e.g. Material Type 1.

30-10 (PART) LS-DYNA R7.1

---

**\*PART** **\*PART**

- The local cartesian coordinate system is defined as described in \*DEFINE\_COORDINATE\_VECTOR. The local z-axis vector is the vector cross product of the x-axis and the in plane vector. The local y-axis vector is finally computed as the vector cross product of the z-axis vector and the x-axis vector. The local coordinate system defined by CID has the advantage that the local system can be defined by nodes in the rigid body which makes repositioning of the rigid body in a preprocessor much easier since the local system moves with the nodal points.
- When specifying mass properties for a rigid body using the inertia option, the mass contributions of deformable bodies to nodes which are shared by the rigid body should be considered as part of the rigid body.

- LS-DYNA input deck are often denoted by the \*.k or \*.key extension
- Open keyword file 'crush\_tube\_start.key' in a text editor of your choice (Notepad++, Vim)
- Input deck starts with \***KEYWORD** and terminates with \***END**
- Comments begin with \$
- Each LS-DYNA keyword begins with an asterisk \*
- Followed by card (lines) of data associated with that keyword:
  - Often more than one card for a keyword
  - Format can be 'fixed' (column specific) or 'free' (comma separated)
  - Each card has a maximum of 80 characters
  - Typically 8 fields of 10 characters, common exceptions are:
    - \***ELEMENT** is 10 fields of 8 characters
    - \***NODE** has 16 characters for coordinates
- Order of keywords in file is arbitrary (few exceptions)

```
*KEYWORD
$
$ This is a comment
$
*PART
Example_Part_Fixed
$ For easy counting:
$234567890$234567890$234567890$234567890$234567890$234567890$234567890$234567890
  1          1          1
*PART
Example_Part_Free
2, 2, 2
*END
```

Fixed - Each card is 8 fields of 10 characters (usually)  
TIP: Add a comment line with count to help you

Free - Each field is separated by a comma

# Oasys PRIMER

*Pre-Processor - .key or .k files*

## Quick-Pick Control

Controls the mouse action when applied within the graphics area.

## Manipulation Tools

Provides access to PRIMER specific functions.

## Keywords

This provides access to the Keywords that are supported and can be edited by PRIMER.

## Open Menu tabs

These control which option is displayed in the current menu panel. Model and Part Tree will always be available in addition to selected options.

## Current menu panel

"Current Menu Panel" Displays the menu for the option currently selected by the menu tabs.

## Graphics Area

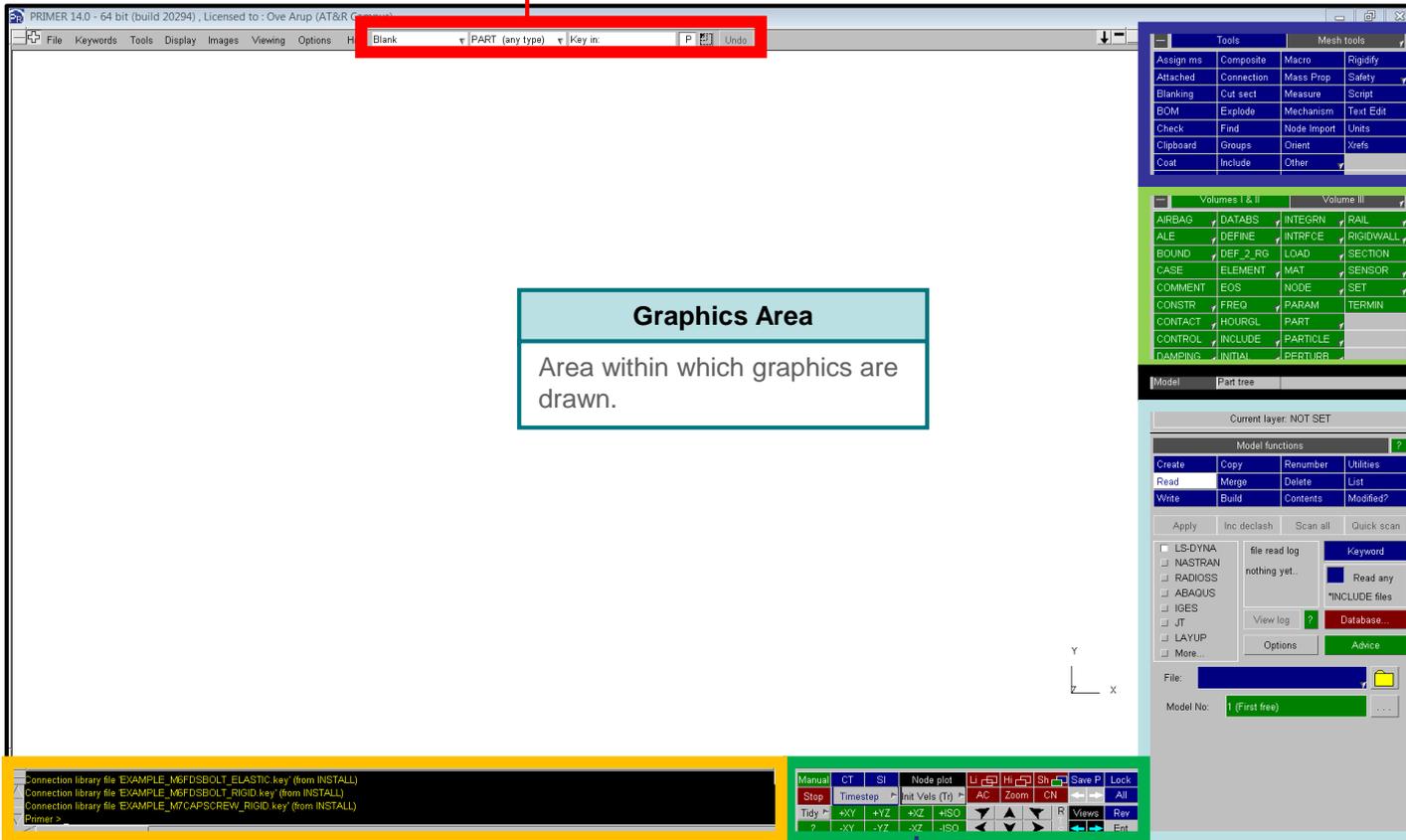
Area within which graphics are drawn.

## Dialogue & List area

Area for command-line input and output, also acts a listing area for messages.

## Viewing & Drawing Commands

Provides all aspects of view control: direction, perspective, scale, etc. Contains the drawing commands and their settings.

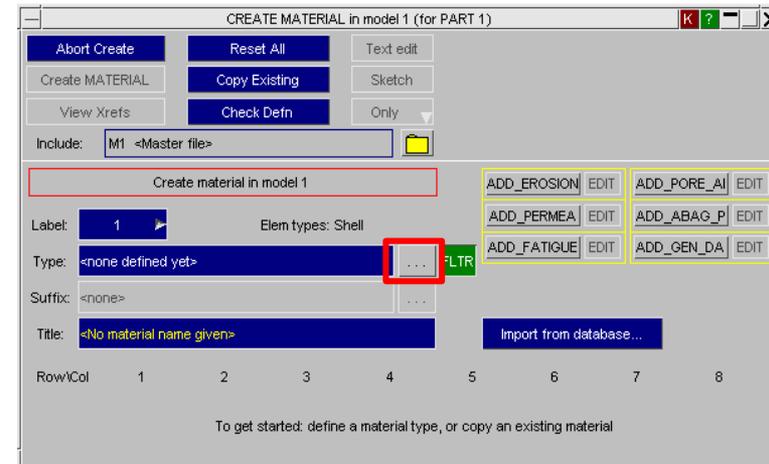
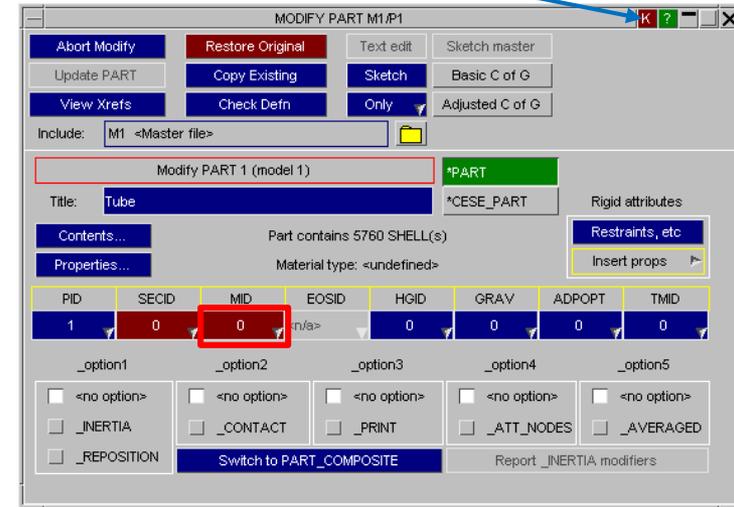


There are over 200 materials types available in LS-DYNA. Refer to the LS-DYNA keyword manual Vol. II for a complete description.

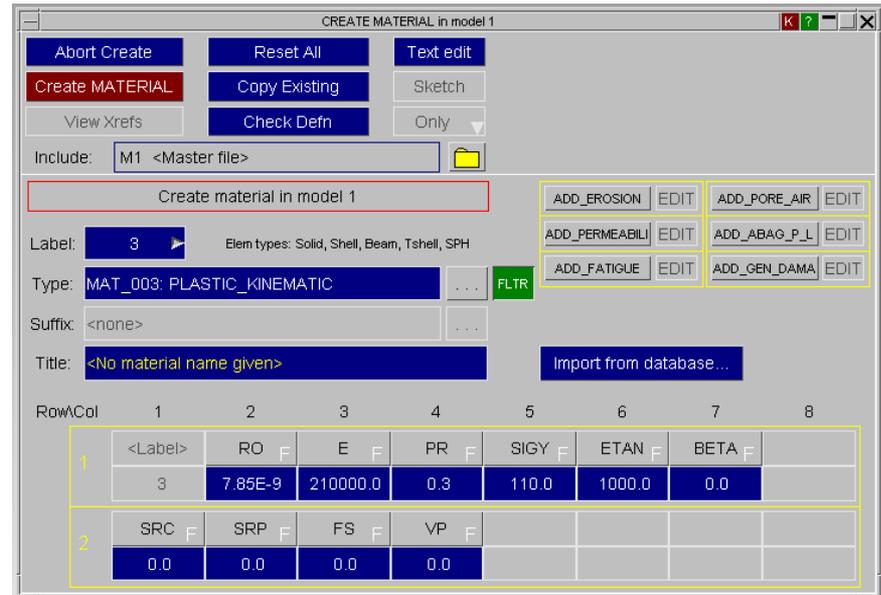
- Read LS-DYNA input deck 'crush\_tube\_start.key' into PRIMER. Press shortcut key 'y' to visualise the elements.
- Right click on the tube part in the graphics window and select 'Edit' from the drop-down menu.
- Now create a new material card by right clicking in the **MID** field (or click the down arrow) and select 'Create'.
- Firstly, specify the material type as **\*MAT\_003: PLASTIC\_KINEMATIC**. Do this by clicking the button to the right of the 'Type' field and selecting the type from the menu.

**MAT\_003** is a cost effective model suited to model isotropic and kinematic hardening plasticity with the option of including rate effects.

[Click here to access keyword manual](#)



- Enter the following values in the material card:
  - RO = 7.85E-9
  - E= 210000
  - PR = 0.3
  - SIGY = 110
  - ETAN = 1000
- Hover the mouse over the parameters to see a description.
- Press **'Create MATERIAL'**.



Note on consistent units: LS-DYNA requires that the set units employed in any model is consistent with Newton's Second Law of Motion ( $F=ma$ ). Different unit systems are shown in the table below. In this exercise, we will use the S2 system of units:

	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

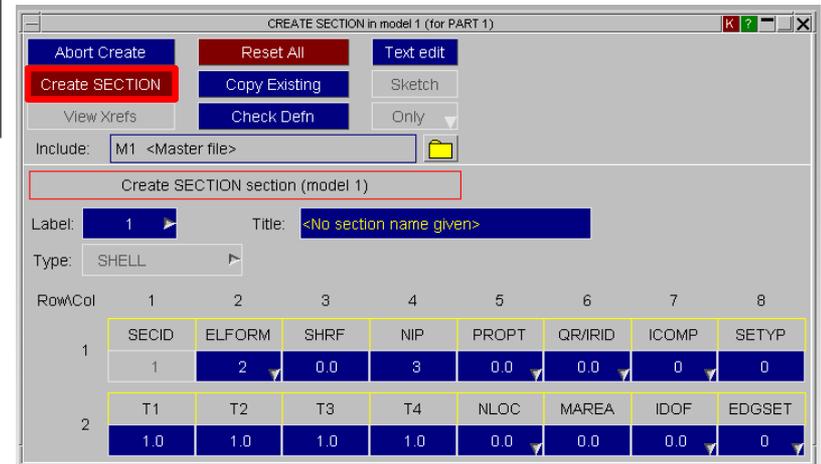
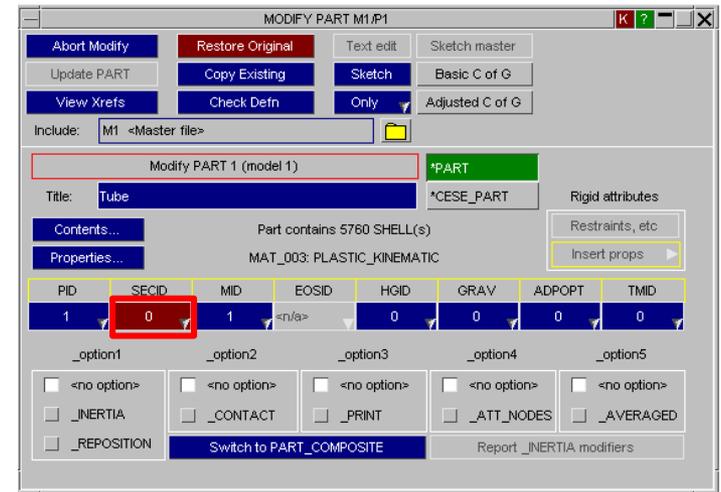
- Create a section card by right clicking in the **SECID** field and select 'Create'.
- Enter the properties of the new section card as follows:
  - **ELFORM** = 2 (default)
  - **NIP** = 3
  - **T1** = 1 (setting T1 = 1 will automatically assign T2, T3 and T4 to the same value)
- Press 'Create SECTION'.

**ELFORM:** Element formulation

**NIP:** Number of through-thickness integration points

**T1:** Shell thickness at node 1

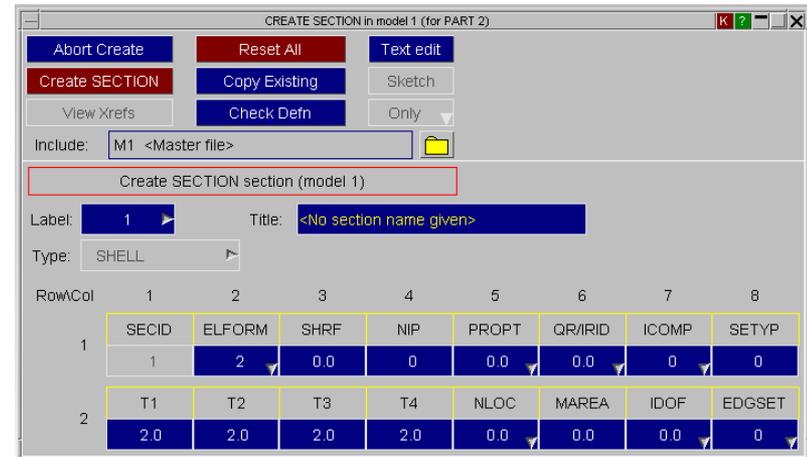
- Click 'Update Part' in 'MODIFY PART' panel.



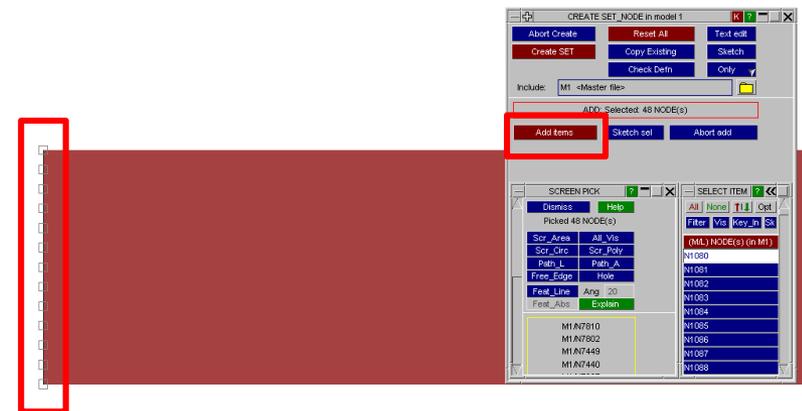
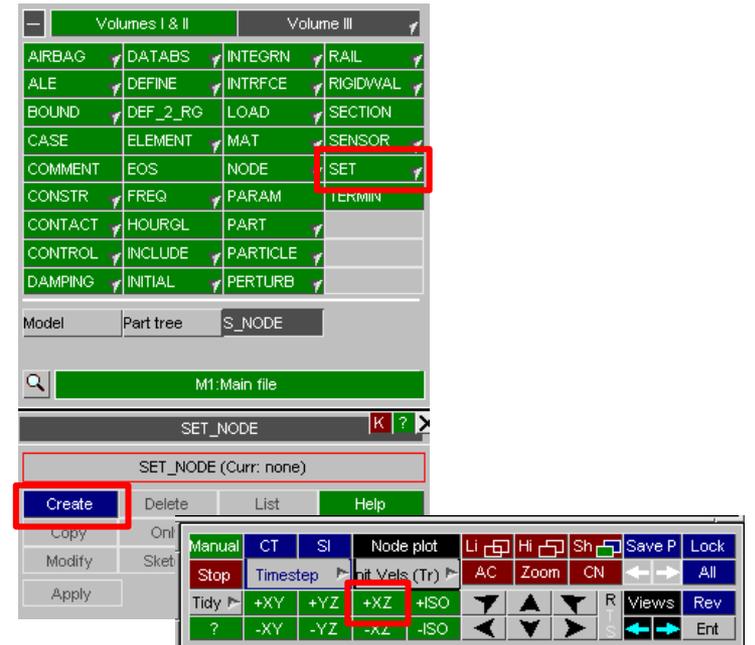
- Create a material and section card for the 'Plate' part in the same way, by right clicking on the plate and clicking 'Edit'.
- **MID:** Enter the properties of the plate's material card as follows:
  - Set type to "MAT\_020: RIGID"
  - $RO = 7.85E-9$
  - $E = 210000$
  - $PR = 0.3$
  - $CMO = 1$
  - $CON1 = 5$
  - $CON2 = 7$

As a rigid body material has been specified, the elastic modulus and Poisson ratio are only used for contact stiffness.

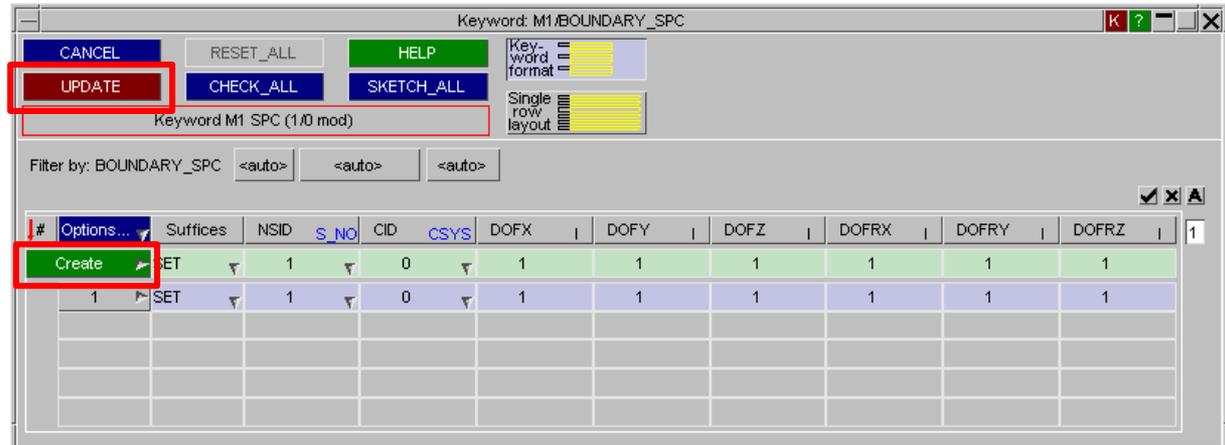
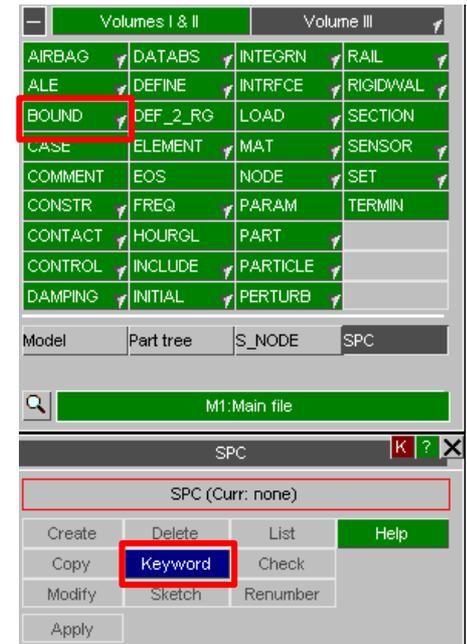
- **SECID:** Enter the properties of the plate's section card as follows:
  - Leave  $ELFORM = 2$  and leave  $NIP = 0$
  - $T1 = 2$



- Next, a single point constraint (SPC) boundary condition should be applied to the far end (negative x) of the tube.
- To make this easier, first create a set of nodes for which the SPC condition should be applied to.
  - Click on **SET** in the keyword panel and select '**NODE**', then click '**Create**'.
  - Click '**Add NODE**' and then select the nodes at the far end of the tube. This is easier when the orientation of the model is set to '**+XZ**', then drag a box over the desired nodes.
  - Then click '**Add items**' followed by '**Create SET**'.



- Now an SPC boundary condition can be applied to the set of nodes to constrain them in all six degrees of freedom: translational and rotational.
- Click '**BOUND**' in the keyword panel and select '**SPC**'.
- Click '**Keyword**' to edit the **\*BOUNDARY\_SPC** card.
- Change the suffice from '**NODE**' to '**SET**' and then '**SELECT**' the previously created node set ID in **NSID**.
- Set all six degrees of freedom (DOF) to 1
- Click '**Create**' and then '**UPDATE**'.

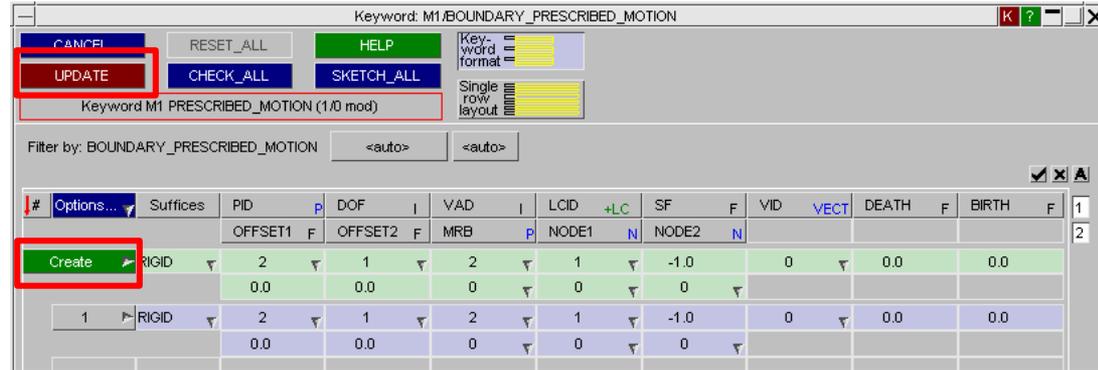
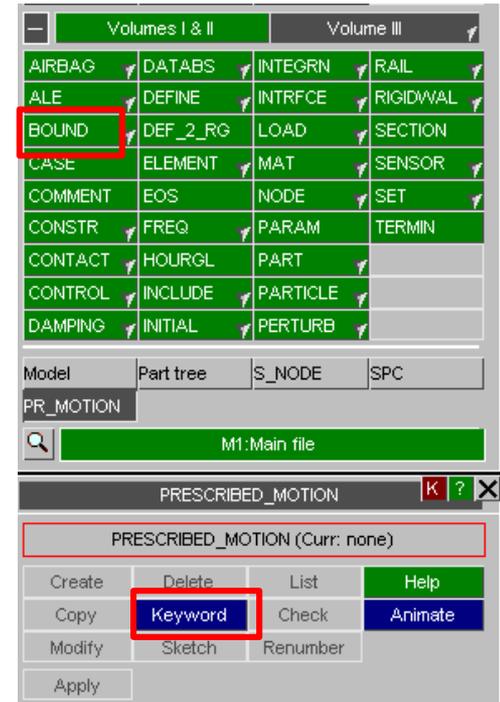


- A prescribed motion boundary condition is applied to the rigid plate part according to a defined load curve **LCID**.
- Click '**BOUND**' in the keyword panel and select '**PRESCRIBED\_MOTION**'.
- Click '**Keyword**' to edit the **\*BOUNDARY\_PRESCRIBED\_MOTION** card.
  - Change the suffice from **NODE** to **RIGID**
  - Enter the part ID (**PID**) for the plate
  - Set the degrees of freedom to the x direction (**DOF = 1**)
  - Set **VAD = 2**
  - Set **LCID = 1**
  - Set **SF = -1**
- Click '**Create**' and then '**UPDATE**'.

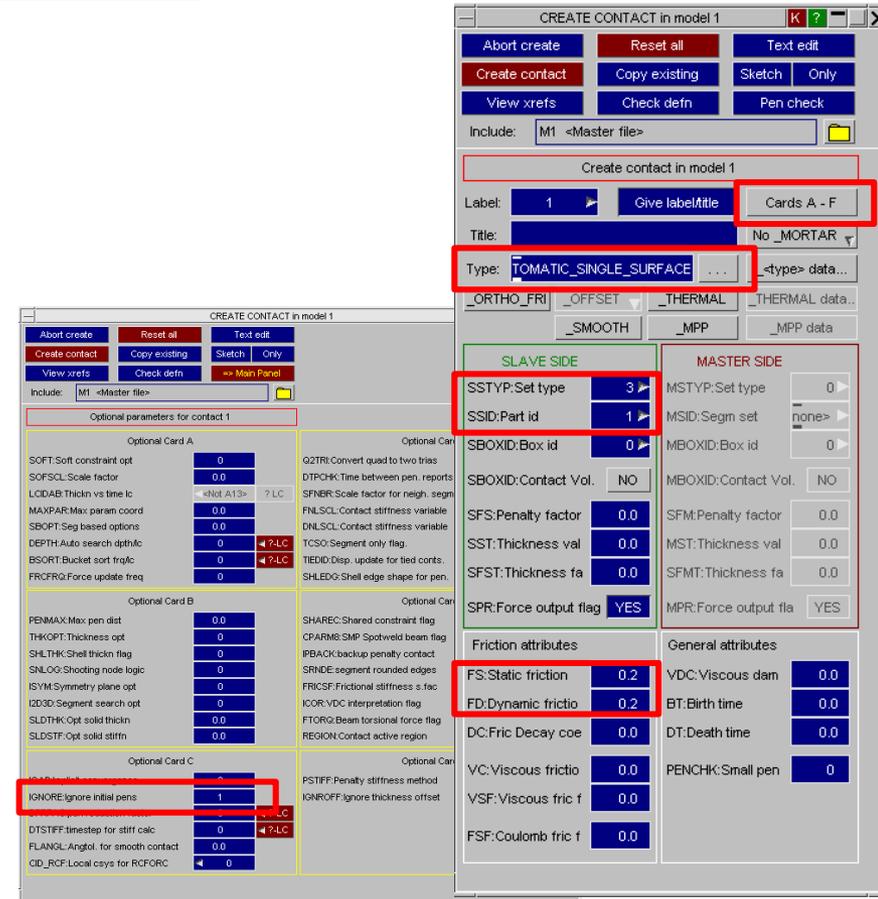
**VAD:**  
Velocity/acceleration/displacement flag.

**LCID:** Load curve ID which is pre-defined in this tutorial.

**SF:** Load curve scale factor.



- Now create a contact definition between the tube and the plate.
- Click '**CONTACT**' in the keyword panel and then '**Create**'.
- In the Create Contact window set the type to '**AUTOMATIC\_SINGLE\_SURFACE**'.
- Set the slave side type (**SSTYP**) to 3 (PART id)
- Select the tube part via the '**PICK**' or '**SELECT**' option in the **SSID** field. There is no master side for this type of contact.
- Change both the static friction and dynamic friction to 0.2.
- Click '**Cards A – F**' to edit optional contact features and set '**Ignore initial pens**' to 1.
- Now click '**Create contact**'.



The **AUTOMATIC\_SINGLE\_SURFACE** is used when the exact position of the contact is unknown. In this case it is used for self contact to prevent the tube from penetrating itself.

## Pre-processing: Contacts

- Next, create a contact surface between the plate and the tube.
- Create a new contact and set the type to **'AUTOMATIC\_SURFACE\_TO\_SURFACE'**.
- Set the slave side type (**SSTYP**) to 3 (Part)
- Select the plate part via the **'PICK'** or **'SELECT'** option in the **SSID** field.
- Do the same for the tube on the master side.
- Change both the static friction and dynamic friction to 0.3.
- Select **'Cards A – F'** to edit optional additional features and set **'Ignore initial pens'** to 1.
- Now click **'Create contact'**.

**'Ignoring initial pens'** helps eliminate any initial stresses that would occur as a result of initial penetrations.

The **AUTOMATIC\_SURFACE\_TO\_SURFACE** contact is used when the direction of contact between two or more components is predictable. Here it is used to define contact between the plate and the tube.

## Getting Started with LS-DYNA

MODIFY CONTACT M1/CONT2

Abort modify Restore original Text edit

Update contact Copy existing Sketch Only

View xrefs Check defn Pen check

Include: M1 <Master file>

Modify contact M1/CONT2

Label: 2 Give label/title Cards A - F

Title: No \_MORTAR

Type: IC\_SURFACE\_TO\_SURFACE \_type> data...

\_ORTHO\_FRI \_OFFSET \_THERMAL \_THERMAL data...

\_SMOOTH \_MPP \_MPP data

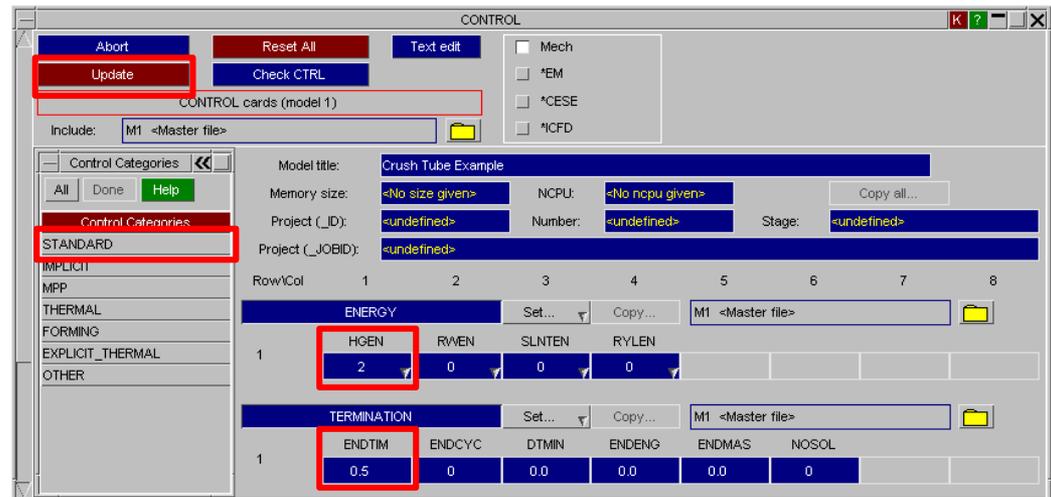
SLAVE SIDE		MASTER SIDE	
SSTYP: Set type	3	MSTYP: Set type	3
SSID: Part id	2	MSID: Part id	1
SBOXID: Box id	0	MBOXID: Box id	0
SBOXID: Contact Vol.	NO	MBOXID: Contact Vol.	NO
SFS: Penalty factor	0.0	SFM: Penalty factor	0.0
SST: Thickness val	0.0	MST: Thickness val	0.0
SFST: Thickness fa	0.0	SFMT: Thickness fa	0.0
SPR: Force output flag	YES	MPR: Force output fla	YES
Friction attributes		General attributes	
FS: Static friction	0.3	VDC: Viscous dam	0.0
FD: Dynamic frictio	0.3	BT: Birth time	0.0
DC: Fric Decay coe	0.0	DT: Death time	0.0
VC: Viscous frictio	0.0	PENCHK: Small pen	0
VSF: Viscous fric f	0.0		
FSF: Coulomb fric f	0.0		

There is a large number of control cards available in LS-DYNA to control aspects of the simulation such as timestep, accuracy, termination and so on. Any cards that are not defined will be given their default values.

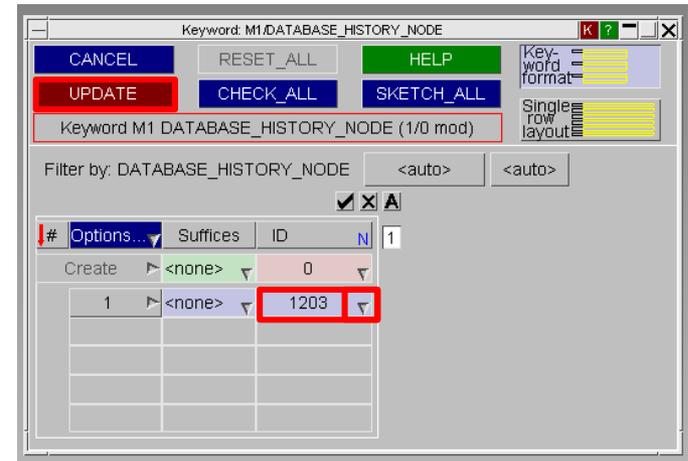
- Click '**CONTROL**' in the keyword panel and then '**Modify**'.
- In the control card window, under 'Control Categories', select '**STANDARD**' then '**ENERGY**'. Now set **HGEN** to 2 to enable hourglass energy calculation.

**It is recommended to set HGEN=2**, otherwise there will be no calculation of hourglass energy. When post processing a run where hourglass energy calculation was off, this may give the false impression that the really is no hourglassing.

- Set up the '**TERMINATION**' control card (also under '**STANDARD** control').
- Set **ENDTIM** to 0.5 so that the simulation ends after 0.5s.
- Click '**Update**' to update the control card.



- Click **'DATABS'** in the keyword panel and then **'HISTORY\_NODE'**.
- To output the nodal history (displacements, velocities) from the free end of the tube, pick or enter the node ID in the ID field. Node 1203 can be used.
- Click **'Create'**.
- The position of the chosen node can be checked by clicking the arrow in the ID field and then **'Sketch'** from the drop down menu.
- Click **'UPDATE'** to dismiss the database card.



- Database cards define what output data is created and at what locations.
- **It is important to remember to set up database cards, they will not affect the calculation but do dictate what data is available in post processing!**
- Here, nodes can be added for which output data is required (displacement or velocity histories for example). N
- Note that the ASCII database cards must also be set up in order for the output data to be available, this is covered in a later slide.

- Click 'DATABS' in the keyword panel and then 'CROSS\_SECTION' and 'Create'.
- In the 'CREATE DATABASE\_CROSS\_SECTION' window, set the local system type to 'Constant X', then select a node that is one element away from the fixed end of the tube.
- Click 'Sketch' to check the position and orientation of the cross section plane, and then 'Create XSECTION'.

**DATABASE\_CROSS\_SECTION** card defines the cross section to obtain the forces and moments in the tube. **DATABASE\_SECFORCE** output card must also be added to instruct the output and frequency of these forces (see next page).



CREATE DATABASE\_CROSS\_SECTION

Buttons: Abort Create, Reset All, Text edit, Undo Create, **Create XSECTION**, Copy Existing, **Sketch**, Check Defn, Only

Include: M1 <Master file>

Pick point P1

Label: 2 Give label \_PLANE \_SET Properties

Title: <No Xsection name given>

PSID	XCT	YCT	ZCT	XCH	YCH	ZCH	RADIUS
0	245.83333	175.0	-75.0	145.83333	175.0	-75.0	0.0
XHEV	YHEV	ZHEV	LENL	LENM	ID	ITYPE	
245.83333	275.0	-75.0	200.0	200.0	0	0	

Drag translate Drag rotate Auto create Move by half eleme

Local system type: **Constant X** Infinite plane Point 1 position: P1 at centre

Offset: 0.0 + - Auto CSYS 1

Auto create PSID

Show only PSID

Reverse direction  Automatically add location to title

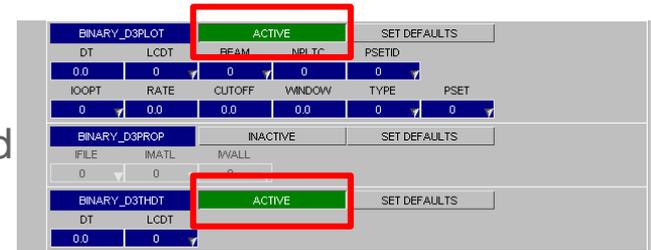
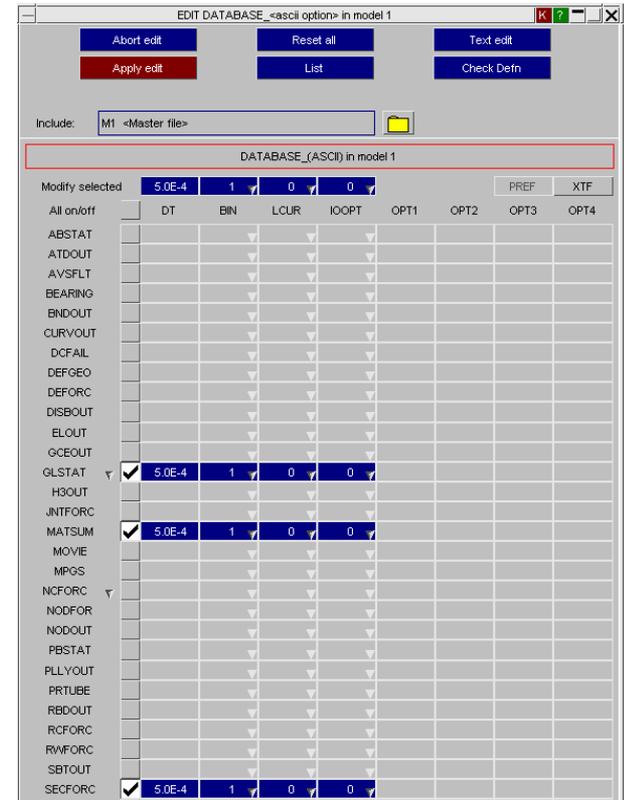
Volume Find 200.0

**ASCII database** options specify output and frequency of data components. Database history cards are also necessary in order to specify the locations (nodes, cross sections etc.) of interest.

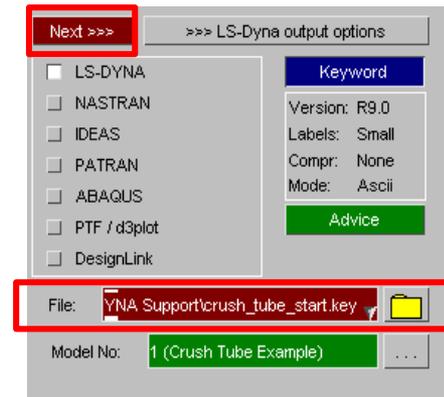
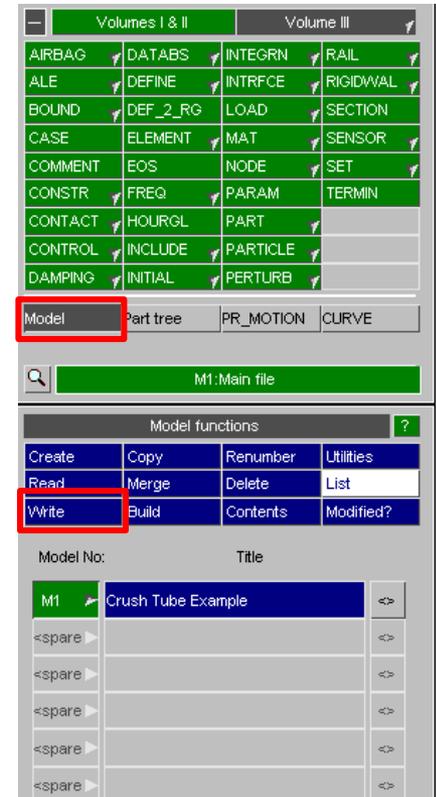
- Click **'DATABS'** in the keyword panel and then **'(ASCII)\_OPTION'**.
- Tick boxes for **'GLSTAT'** (global data), **'MATSUM'** (material energies) and **'SECFORC'** (section forces).
- Click **'Apply edit'** to create this database card.

**Binary database** options specify what output files are generated (eg. the .pft file for post processing using Oasys D3PLOT) and the timestep between output states.

- Click **'DATABS'** in the keyword panel and then **'BINARY'**.
- Click the **'INACTIVE'** button for **'BINARY\_D3PLOT'** and **'BINARY\_D3THDT'**.
- Define a time interval, **DT**, of your choice between output states. This will generate binary output files and associated time histories.
- Click **'Apply edit'** to create this database card.

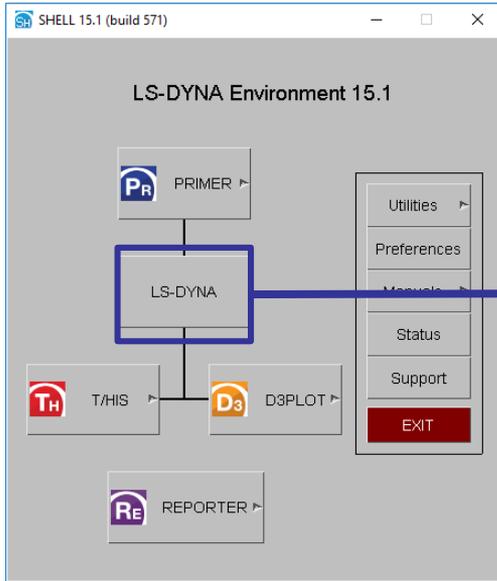


- To save the model click '**Model**' and then '**Write**'.
- Define a file path and name and then click '**Next**' and finally click '**Apply**', to write the file.
- Open the \*.key file in a text editor and inspect the \* LS-DYNA keywords. Use the LS-DYNA Keyword Manual Vol. I and II to assist you.
- The written keyword file can now be submitted to LS-DYNA.



# Oasys SHELL

*Job Submission*



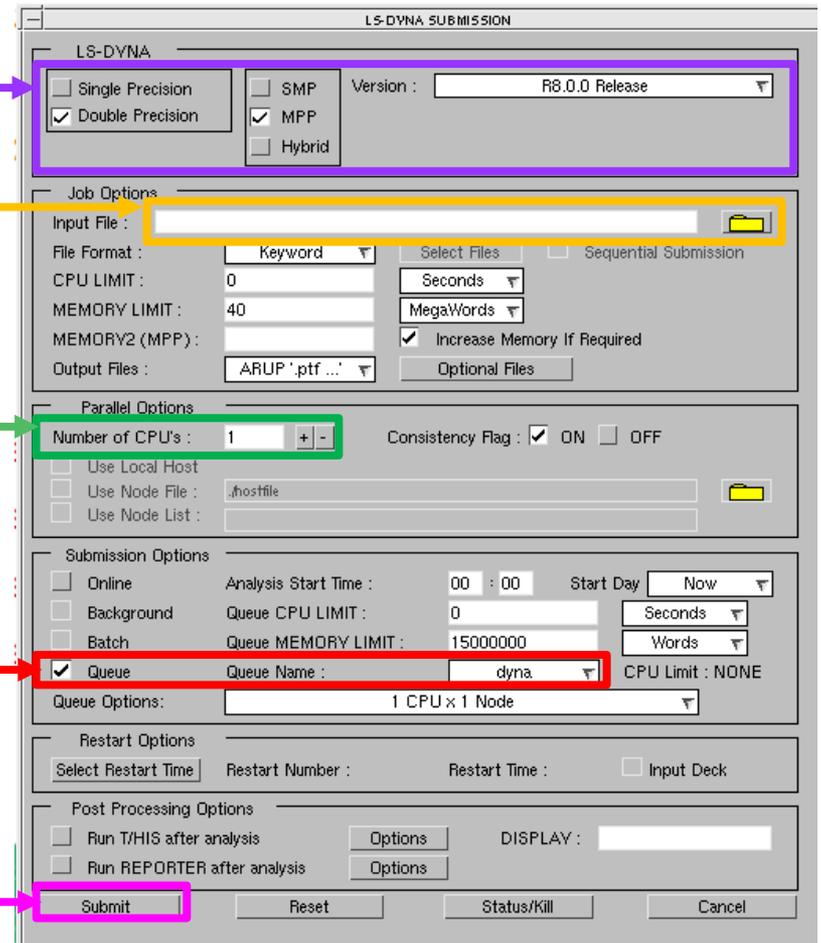
LS-DYNA Version

Input File

Number of CPU (cores)

Queue options

Submit



The list of LS-DYNA versions that can be accessed through the SHELL is controlled via an ASCII file called 'dyna\_versions', see installation manual for further details

Arup file name convention is *<name>.extension* (see the left hand side column), so it is obvious which files belong together.

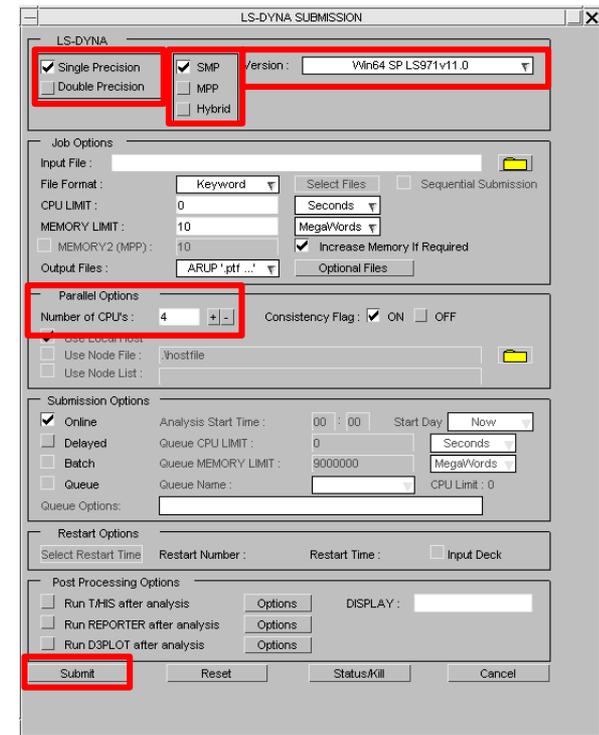
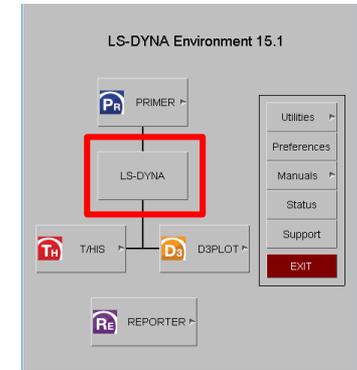
Input Files			
<name>.key		keyword input file	ASCII
<name>.dpf	d3dump	restart file	BINARY
<name>.adf	runrsf	restart file	BINARY

Output Files			
<name>.otf	d3hsp	analysis status	ASCII
<name>.ptf	d3plot	plot state data	BINARY
<name>.thf	d3thdt	time history data	BINARY
binout		analysis results	BINARY
messag		analysis status	ASCII
<name>.log	log	analysis status	ASCII
<name>.dpf	d3dump	restart file	BINARY
<name>.adf	runrsf	restart file	BINARY

Results files are BIG so are often written as a “family” of files (e.g. *ptf*, *.ptf01*, *.ptf02*).

Other specific output files can be requested for specialist data (e.g. *spcforc* for reaction forces). See PRIMER for details of these.

- In Oasys SHELL, select the following settings:
  - **Single Precision:** Single precision (SP) uses 32 bit storage and is used for most LS-DYNA applications. Double precision (DP) would use 64 bit storage and is useful when extremely small deflections or long event times are expected.
  - **SMP:** Shared Memory Processing allows users to distribute the model over multiple processes on the same machine. Scalability is minimal beyond 4-8 processors. Massively Parallel Processing (MPP) allows users to run the LS-DYNA solver over a cluster of machines or use multiple processors on a single machine. Scalable!
  - **Version:** Select the desired SMP LS-DYNA version installed on the client machine.
  - **Number of CPUs:** 4 cores is sufficient here.
  - **Input File:** Navigate to the LS-DYNA input deck
  - Click **'Submit'**.



# Oasys D3PLOT

*Post-Processor - .ptf or d3plot files*

# Post-processing: Oasys D3PLOT

# Getting Started with LS-DYNA

## Top menus

Allows access to basic options, keywords and tools, in a drop-down menu format

## Quick-Pick Control

Controls the mouse action when applied within the graphics area

## Tools

Provides access to D3PLOT specific functions

## T/HIS link

## Menu tabs

These control which option is displayed in the current menu panel. Model and Part Tree will always be available in addition to selected options

## Current menu panel

"Current Menu Panel" Displays the menu for the option currently selected by the menu tabs

## Animation Controls

Controls states and what is displayed during animation

## Window ID

## Graphics Area

Area within which graphics are drawn

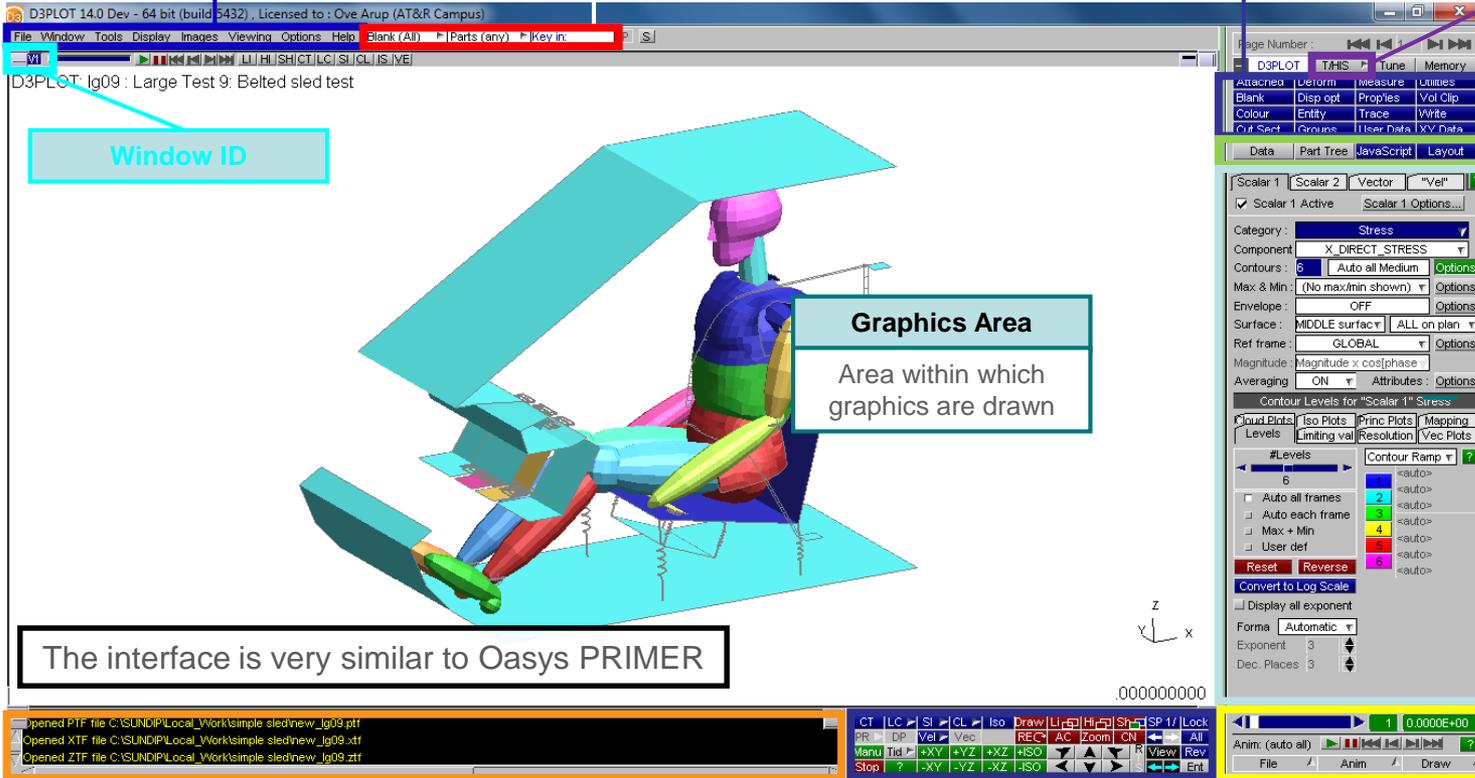
The interface is very similar to Oasys PRIMER

## Dialogue & List area

Area for command-line input and output, also acts a listing area for messages

## Viewing & Drawing Commands

Provides all aspects of view control: direction, perspective, scale, etc. Contains the drawing commands and their settings



Open the .otf file and look for: Error, Warning, total, termination, smallest, added  
 It's useful to look at this file while the model is running. (The *messag* files can be useful too).

The .otf file contains:

- Generic information from reading the model in
- First allocation of memory
- Model initialisation (total masses, smallest timestep, contact surface timesteps (should be less than model timestep), added mass, initial energy, ...)
- Information at each timesteps output files are produced for
- Summary of time taken for each part of the analysis (check it's reasonable).

### 100 smallest timesteps

```

-----
element          timestep
shell    27156      0.35129E-06
shell    12452      0.35129E-06
shell    35534      0.49827E-06
shell    35534      0.49827E-06
shell    35534      0.49827E-06
shell    35534      0.49827E-06
slave surface of interface # 1 type= 13
  surface timestep= 0.761E-06 current minimum= 0.761E-06
slave surface of interface # 2 type= 13
  surface timestep= 0.761E-06 current minimum= 0.761E-06
slave surface of interface # 3 type= 7
  surface timestep= 0.498E-06 current minimum= 0.498E-06
master surface of interface # 3 type= 7
  surface timestep= 0.761E-06 current minimum= 0.498E-06

The LS-DYNA time step size should not exceed 0.498E-06
    
```

```

problem cycle = 1200
time          = 1.2949E-03
added mass    = 4.3696E-04
percentage increase = 2.7420
    
```

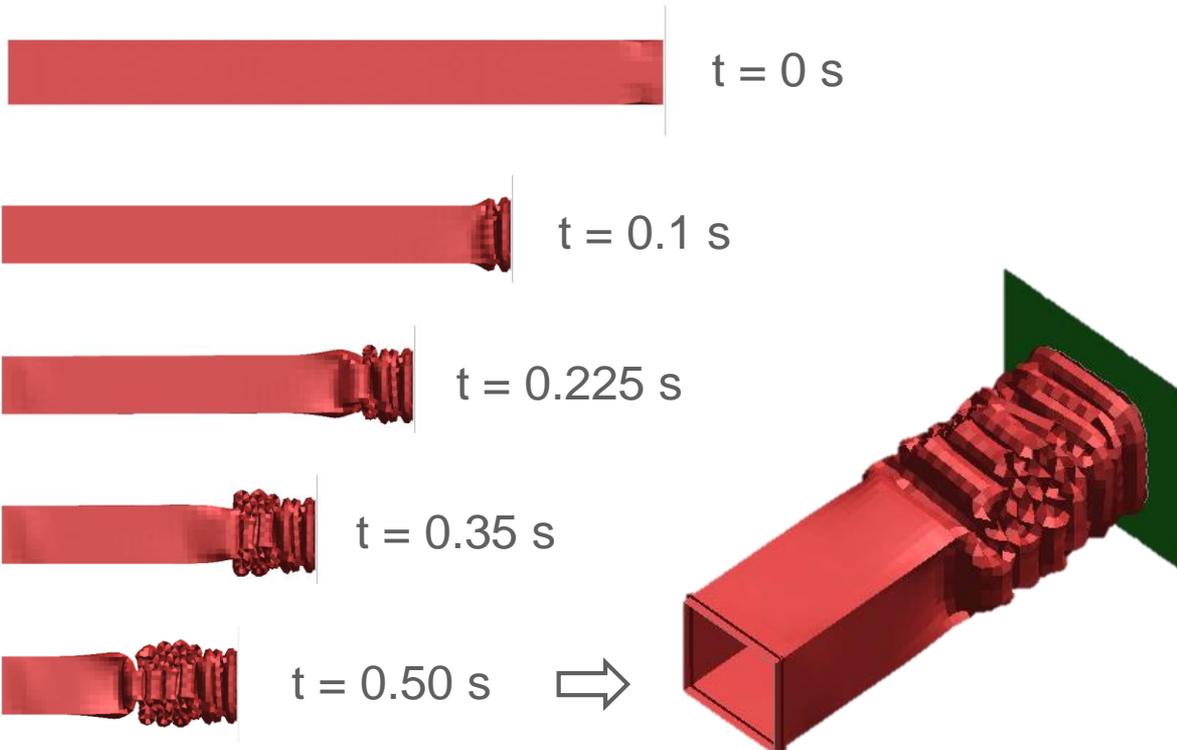
```

Deformable Spotwelds *****
total added spotweld mass = 1.3139E-05
percentage mass increase = 8.2451E-04
    
```

### Timing information

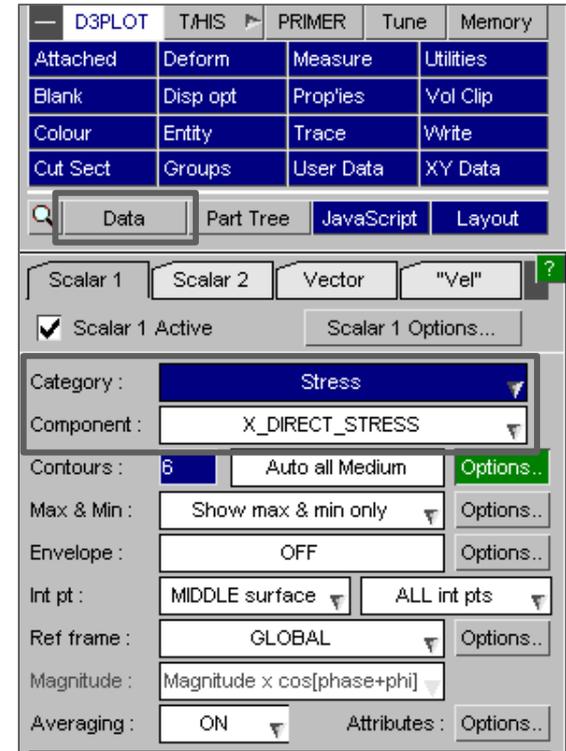
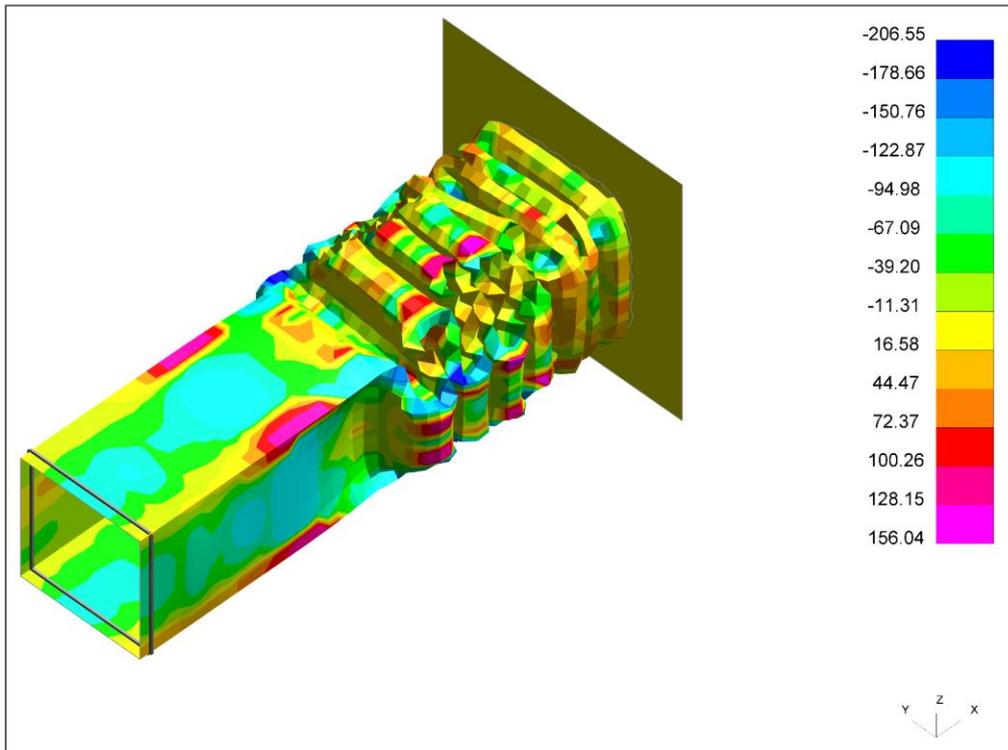
	CPU(seconds)	%CPU	Clock(seconds)	%Clock
Initialization .....	4.7000E-01	0.01	5.0935E-01	0.01
Element processing ...	4.0769E+03	47.79	2.0127E+03	47.58
Binary databases .....	3.3946E+00	0.04	1.7572E+00	0.04
ASCII database .....	2.0024E+00	0.02	1.1212E+00	0.03
Contact algorithm ...	4.3704E+03	51.23	2.1756E+03	51.43
Interface ID 1	3.4023E+03	39.88	1.6947E+03	40.06
Interface ID 2	9.4749E+02	11.11	4.7645E+02	11.26
Contact entities .....	0.0000E+00	0.00	0.0000E+00	0.00
Rigid bodies .....	7.7279E+01	0.91	3.8627E+01	0.91
Implicit Nonlinear ...	0.0000E+00	0.00	0.0000E+00	0.00
Implicit Lin. Alg. ...	0.0000E+00	0.00	0.0000E+00	0.00
<b>Totals</b>	<b>8.5305E+03</b>	<b>100.00</b>	<b>4.2303E+03</b>	<b>100.00</b>
Problem time =	5.0000E-01			
Problem cycle =	1171653			
Total CPU time =	8531 seconds ( 2 hours 22 minutes 11 seconds)			
CPU time per zone cycle =	810 nanoseconds			
Clock time per zone cycle=	402 nanoseconds			
Number of CPU's	2			
NLQ used/max	272/ 272			
Start time	09/13/2010 16:53:41			
End time	09/13/2010 18:04:50			
Elapsed time	4269 seconds( 1 hours 11 min. 9 sec.) for 1171653 cycles			
Normal termination				

- Open the <name>.ptf file from the run directory in Oasys D3PLOT.
- Left Shift + left mouse button to rotate the camera.
- Clicking on items blanks them. Middle mouse button to unblank last item. 'U' to unblank everything.
- Animate the results and check the deformation.



Check that the model behaviour is sensible and as expected. For example, there should not be parts flying away or penetrating each other and contacts should remain intact.

- Select **Category**: Stress
- **Component**: X\_DIRECT\_STRESS via the 'DATA – Scalar 1' panel
- Select **CT** (Continuous Tone) or **SI** (Shaded Image) for contour plotting



By default this will show the **Plastic Strain** component, but other output categories can be selected from the data panel.

# Oasys T/HIS

*Time History - . thf or d3thdt, ...*

## Top menu

Allows access to basic options and tools, in a drop-down menu format

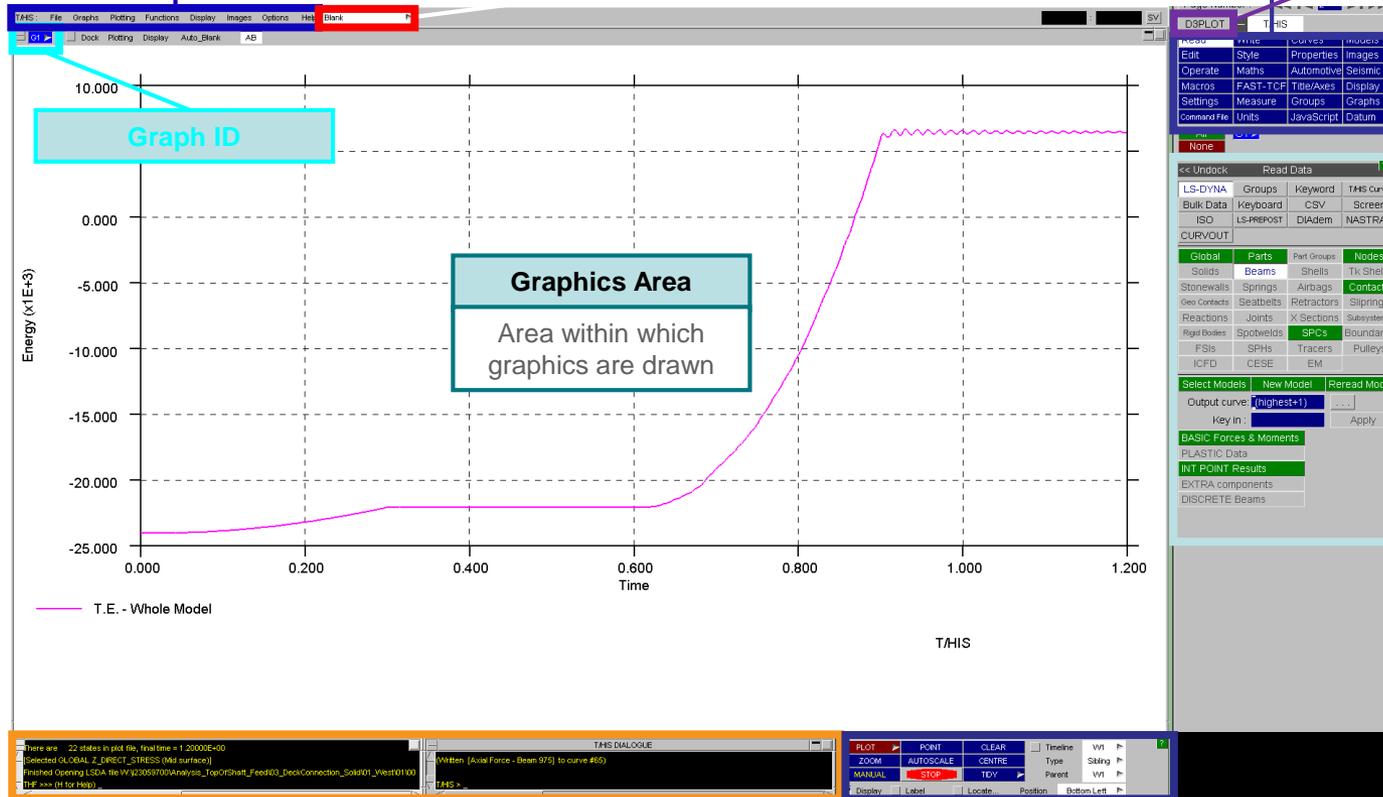
## Quick-Pick Control

Controls the mouse action when applied within the graphics area

## Tools

Provides access to T/HIS specific functions

## D3PLOT Link



## Graph ID

## Graphics Area

Area within which graphics are drawn

## Current menu panel

The entity types for which time-history data is available are shown by the green buttons.

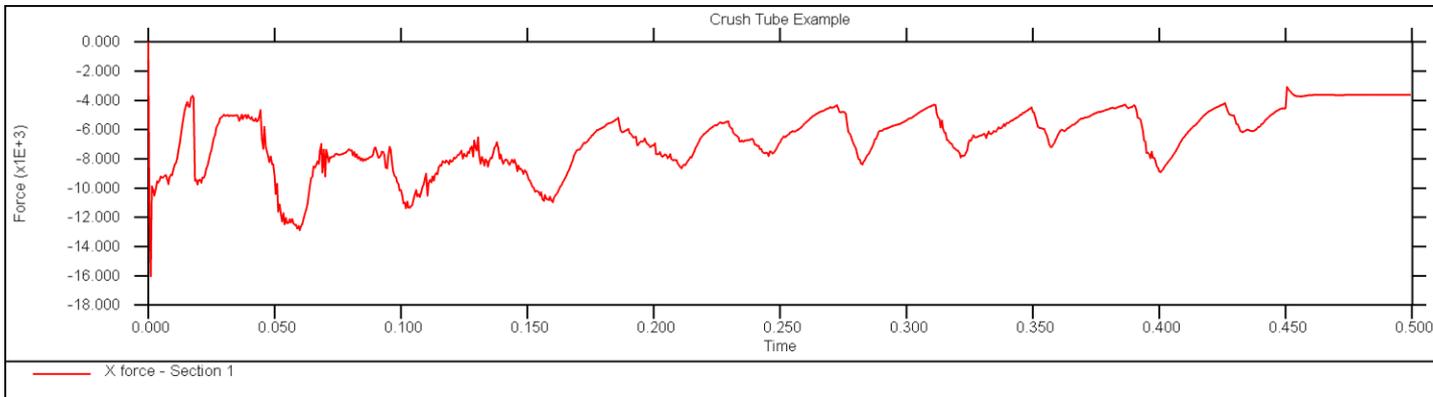
## Dialogue & List area

Area for command-line input and output, also acts a listing area for messages

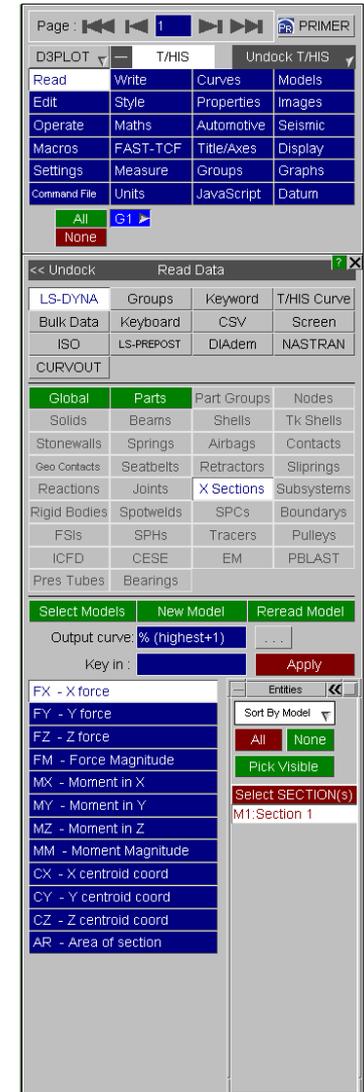
## Viewing Commands

These are different to PRIMER and D3PLOT

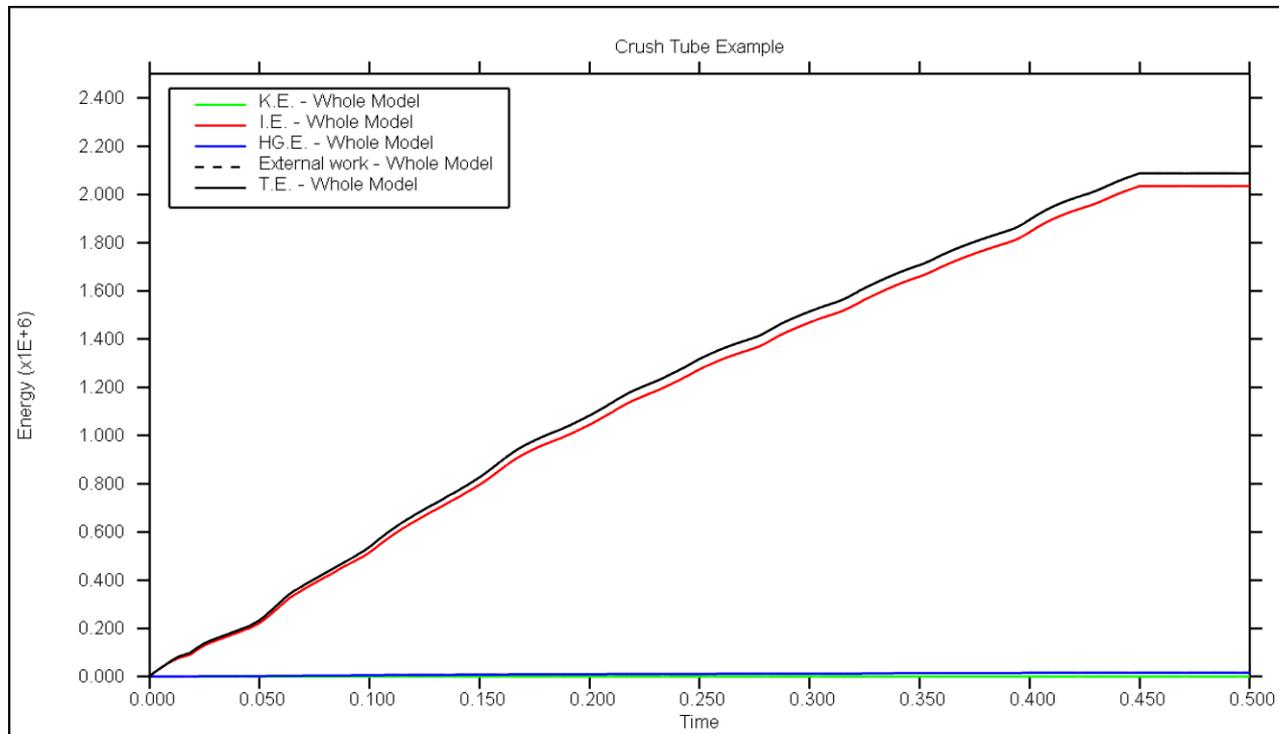
- Click on **T/HIS** button via the Main Menu Option - this will open a T/HIS session within D3PLOT.
- Click “**X Sections**”.
- Select the component of force, i.e. **FX – X force**.
- Select the **SECTION(s)** entities corresponding to the model.
- Click ‘**Apply**’ to plot data graphs.
- Note: Left clicking a line on the graph blanks it. Press middle mouse button to unblank.
- Further graph and curve options can be found under the ‘**T/HIS**’ main menu panel -> ‘**Curves**’.



While D3PLOT allows visual representation of this result and interaction with it, T/HIS allows for plotting and manipulation of time histories. All data for post processing must be requested in the database cards when setting up the run.



- Blank the previous plots and follow equivalent steps as described above.
- Click '**Read**' in the tool panel to access the available time histories. What is available here depends on how the database cards were set up earlier.
- Click '**Global**' and select (holding the 'Ctrl' key) '**KE**', '**IE**', '**HG**', '**EW**' and '**TE**'.
- Check that the energy time histories make sense with what is expected.

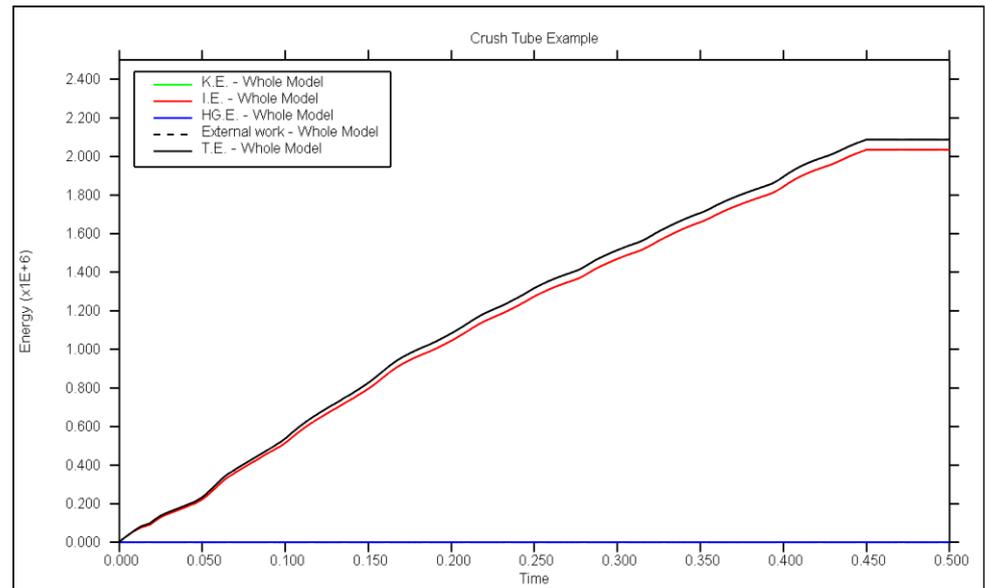


- Check the hourglass energy it should not be more than 5% of internal energy.

If the hourglass energy exceeds 5% of the internal energy, then measures should be taken to reduce the extent of hourglassing. These measures may include:

- Avoid point or line loads and contacts (use pressures or surface contacts instead).
- Avoid bending in single-point-integration solid elements (or use fully integrated elements).
- Use a finer mesh.
- If none of the above work, some benefit can be had from using the **\*HOURGLASS** card.

- The graph on the right shows the energies from the same model having been run without requesting hourglass energy in the **\*CONTROL\_ENERGY** card. Compare this with the graph on the previous slide where hourglass energy (HG.E.) is non-zero.



- Check the work done by the force by performing the integral  $\int F_x dx$ .
- The force plotted is with respect to time  $F_x(t)$ , not displacement  $F_x(x)$ . The work done can be calculated as  $\int F_x(t) \frac{dx}{dt} dt$ .
- Click '**Read**', '**Nodes**', and plot the X Displacement of the defined node.
- Now differentiate this curve by clicking '**Operate**' in the tool panel then '**DIF**' and select the X displacement curve. Press '**Apply**' to generate a curve for  $\frac{dx}{dt}(t)$ .
- Now use the '**MUL (y)**' operation to multiply  $F_x(t)$  with  $\frac{dx}{dt}(t)$ . Then use the '**INT**' operation on the resultant graph to perform the integral  $\int F_x(t) \frac{dx}{dt} dt$ .
- Compare the result against the energies plotted earlier by using the 'Curves' tool.

# ARUP

[www.arup.com/dyna](http://www.arup.com/dyna)

For more information please contact the following:

**UK:**

**Arup**

The Arup Campus  
Blythe Valley Park  
Solihull, West Midlands  
B90 8AE  
UK  
T +44 (0)121 213 3399  
F +44 (0)121 213 3302  
[dyna.support@arup.com](mailto:dyna.support@arup.com)

**China:**

**Arup**

39/F-41/F Huai Hai Plaza  
Huai Hai Road (M)  
Shanghai  
China 200031  
T +86 21 6126 2875  
F +86 21 6126 2882  
[china.support@arup.com](mailto:china.support@arup.com)

**India:**

**nHance Engineering Solutions Pvt. Ltd (Arup)**

Ananth Info Park  
HiTec City  
Madhapur  
Hyderabad - 500081  
India  
T +91 (0) 40 44369797 / 8  
[india.support@arup.com](mailto:india.support@arup.com)