Getting Started with LS-DYNA Implicit

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

tel: +44 (0) 121 213 3399

email: dyna.support@arup.com

2018

Introduction

- The purpose of this tutorial is to introduce new users to the basics of LS-DYNA Implicit and to convert an existing explicit model to an implicit one.
- 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:

- Review LS-DYNA explicit model and results of cantilever beam, which is fixed at one end and subject to point load at free end.
- Submit eigenvalue analysis to check and eliminate rigid body modes.
- Submit linear static analysis.
- Verify results against hand calculations.

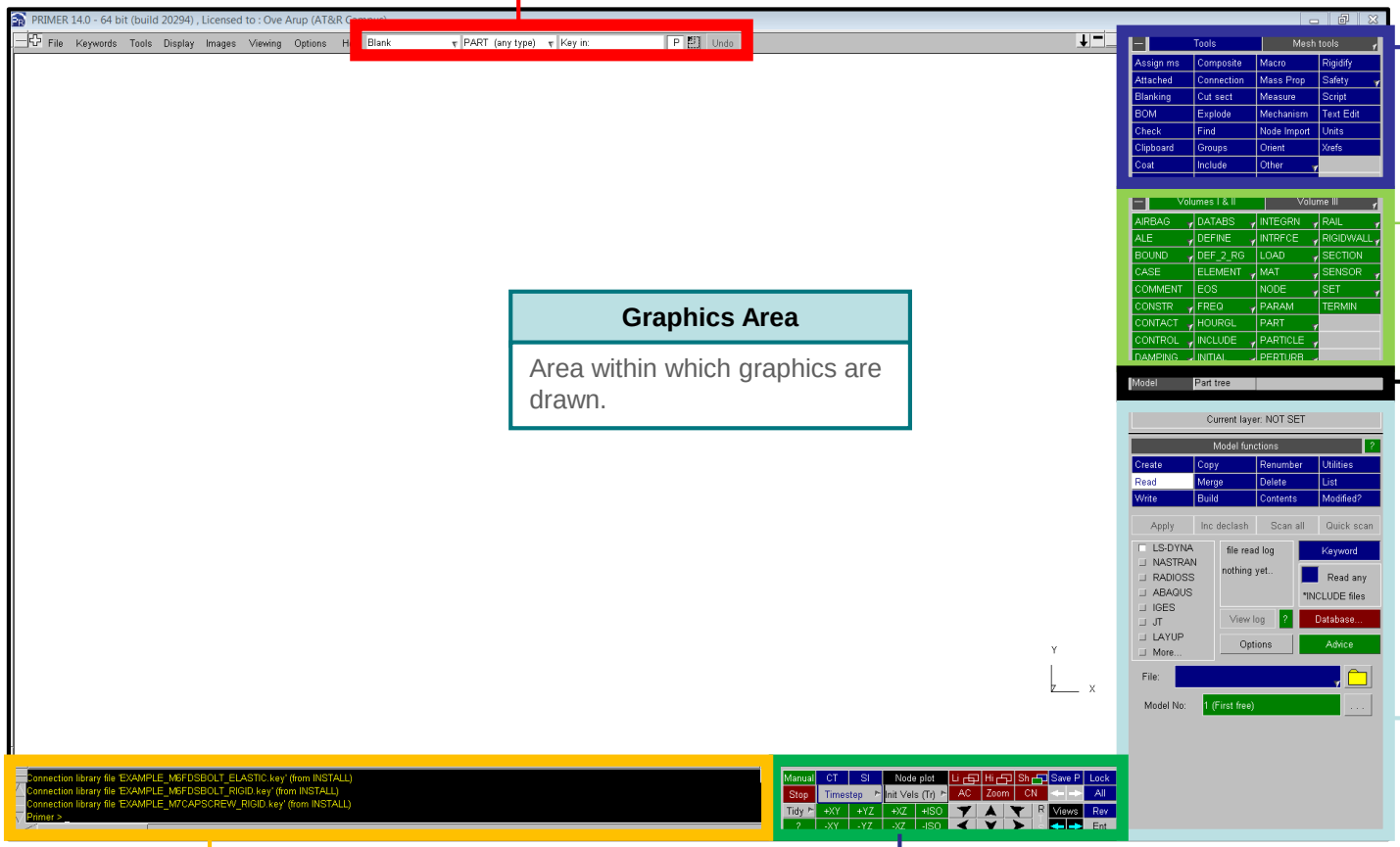
An incomplete input deck is provided: **linear_static_explicit.key**

Pre-Processing: PRIMER Environment

LS-DYNA Implicit

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.

Graphics Area

Area within which graphics are drawn.

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.

Dialogue & List area

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

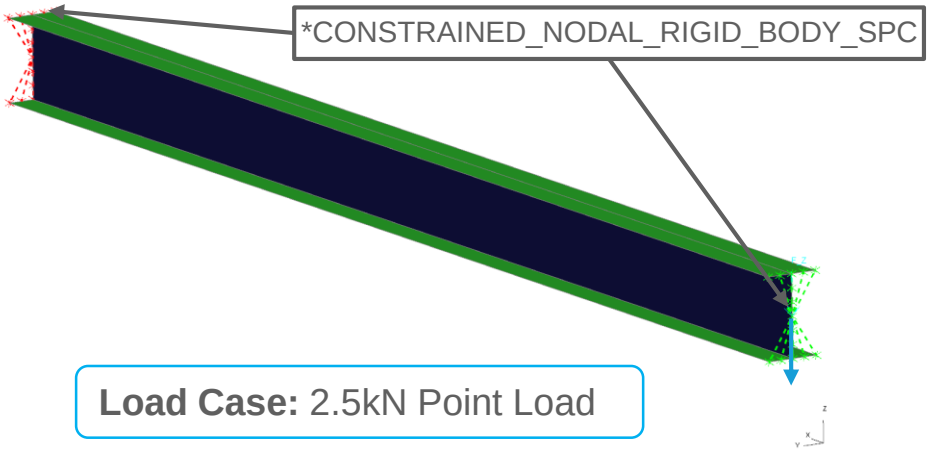
Viewing & Drawing Commands

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

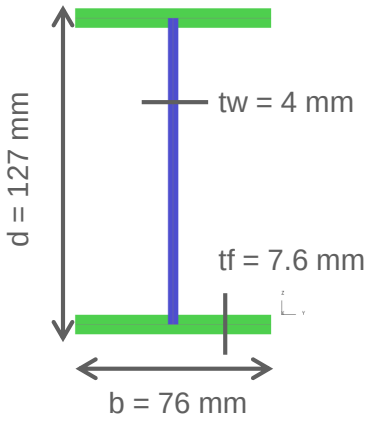
Model Description: Cantilever Beam Analysis

LS-DYNA Implicit

Cantilever beam (I section) modelled in shell elements, fixed at one end



Load Case: 2.5kN Point Load



E = 205000 N/mm^2
L = 2000 mm
Density = 7.85e-9 T/mm^3
Pr = 0.3
I_y=46902000 mm^4

Hand calculation: Tip Deflection

$$d = \frac{PL^3}{3EI}\left(1 + \frac{3f_sEI}{GAL^2}\right) \text{ where } f_s = \frac{A}{A_{web}}$$

Hand calculation: Max Bending Moment

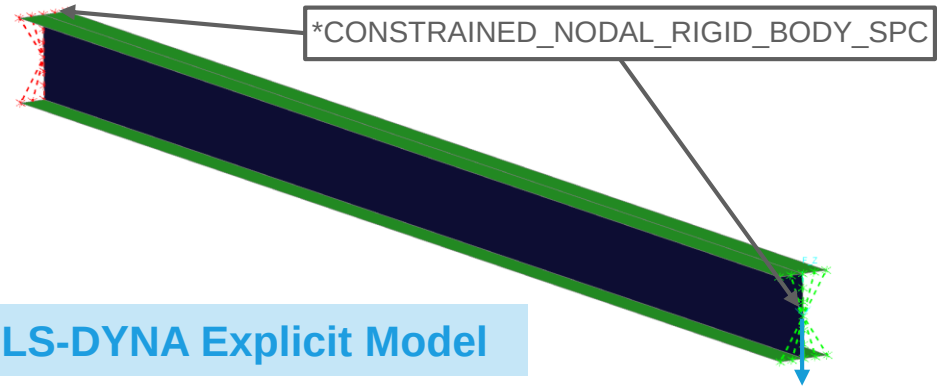
$$M_y = P \cdot L = 2500 \cdot 2000 = 5e6\ Nmm$$

$$d_z = \frac{PL^3}{3EI_y}\left(1 + \frac{3f_sEI_y}{GAL^2}\right) = \frac{2.5 \cdot 2000^3}{3 \cdot 20500 \cdot 4690200}\left(1 + \frac{3 \cdot (1632.8/477.6) \cdot 20500 \cdot 4690200}{7.88e^4 \cdot 1632.8 \cdot 2000^2}\right) = 13.9 \cdot (1 + 0.019) = 7.07\ mm$$

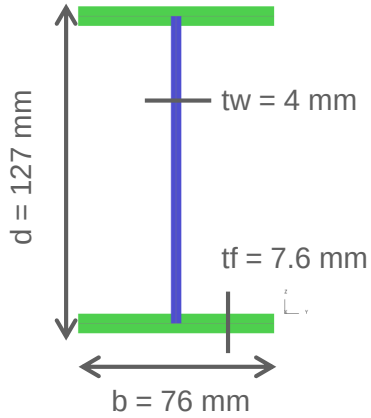
$$G = \frac{E}{2(1+v)}$$

Model Description: Cantilever Beam Analysis

LS-DYNA Implicit

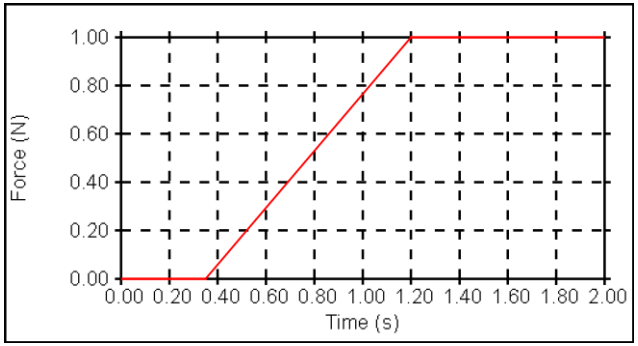


LS-DYNA Explicit Model



E = 205000 N/mm^2
L = 2000 mm
Density = 7.85e-9 T/mm^3
Pr = 0.3
P = 2500 N
I_y=46902000 mm^4

```
*MAT_ELASTIC_TITLE
Steel
$:      mid        ro         e        pr        da        db         k
          1   7.85E-9  205000.0       0.3       0.0       0.0       0.0
*SECTION_SHELL
$:    label    elform      shrf       nip     propt        qr     icomp     setyp
          1        16       0.0         5       1.0       0.0         0         0
$:       t1        t2        t3        t4      nloc     marea      idof    edgset
        7.6       7.6       7.6       7.6       0.0       0.0       0.0         0
$
$:    label    elform      shrf       nip     propt        qr     icomp     setyp
          2        16       0.0         5       1.0       0.0         0         0
$:       t1        t2        t3        t4      nloc     marea      idof    edgset
        4.0       4.0       4.0       4.0       0.0       0.0       0.0
*PART
Flange
$:      pid     secid       mid     eosid      hgid      grav    adpopt      tmid
          2         1         1         0         0         0         0         0
$
*PART
Web
$:      pid     secid       mid     eosid      hgid      grav    adpopt      tmid
          3         2         1         0         0         0         0         0
```

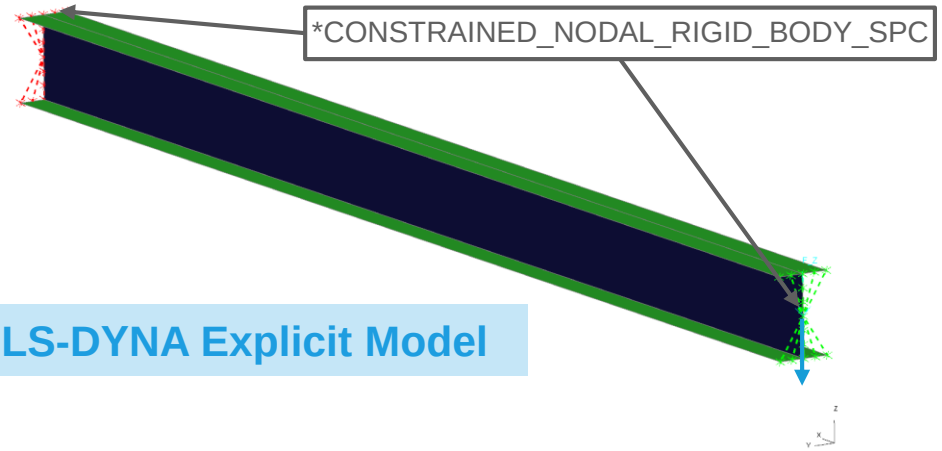


LCID

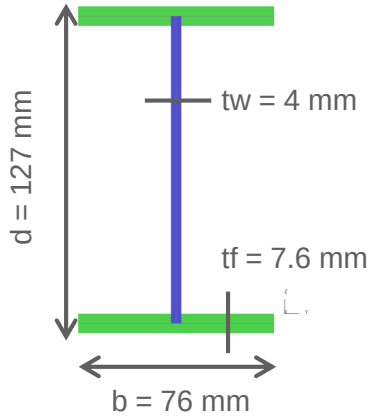
```
*LOAD_NODE_POINT
$:nid/nsid       dof       lcid         sf
      1277         3          1    -2500.0
```

Model Description: Cantilever Beam Analysis

LS-DYNA Implicit



LS-DYNA Explicit Model



$E = 205000\ N/mm^2$

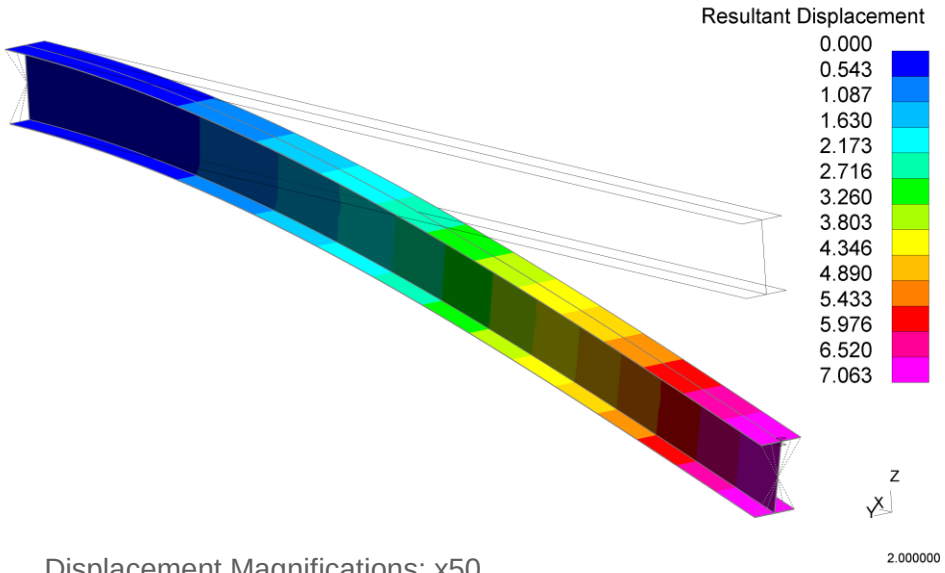
$L = 2000\ mm$

$Density = 7.85e\text{-}9\ T/mm^3$

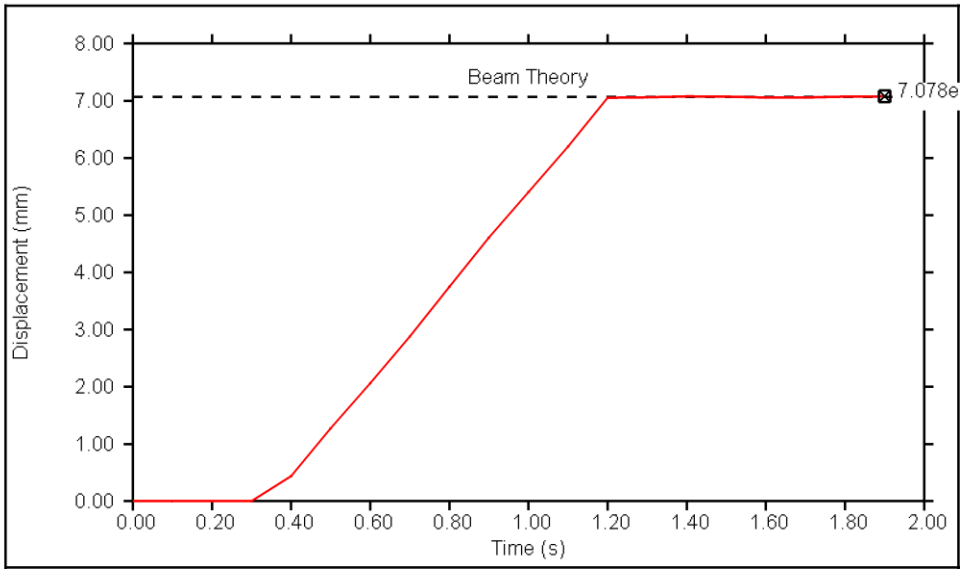
$Pr = 0.3$

$P = 2500\ N$

$I_y = 46902000\ mm^4$

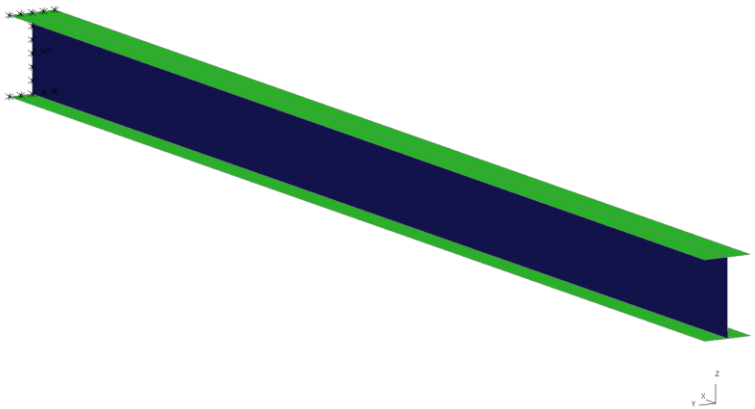


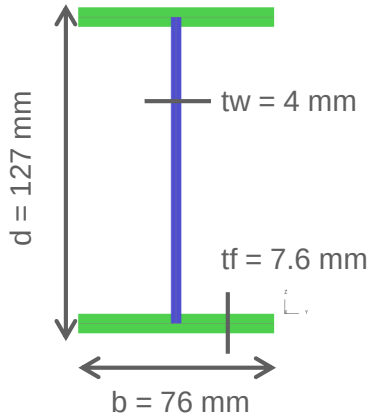
Displacement Magnifications: x50



Explicit-implicit model conversion

Recommended steps





E = 205000 N/mm^2
L = 2000 mm
Density = 7.85e-9 T/mm^3
Pr = 0.3
P = 2500 N
I_y=46902000 mm^4

Submit eigenvalue analysis to check and eliminate rigid body modes

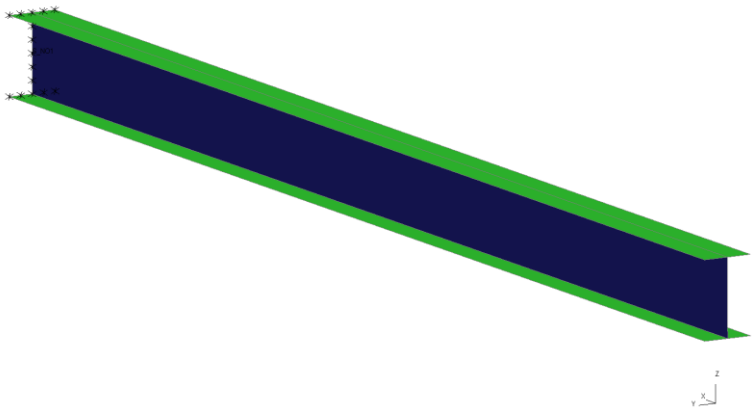
- Set up recommended *CONTROL_, *DATABASE_, and *SECTION cards for implicit analysis
- Set-up specific eigenvalue analysis *CONTROL_ cards
- Submit the analysis
- Check the results and compare against analytical solution

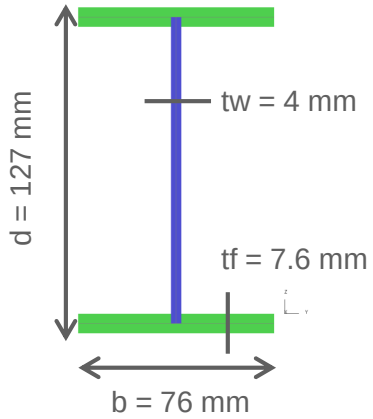
Submit linear static analysis

- Setup implicit load case
- Submit the analysis
- Check the results and compare against beam theory

Explicit-implicit model conversion

Recommended steps





E = 205000 N/mm^2
L = 2000 mm
Density = 7.85e-9 T/mm^3
Pr = 0.3
P = 2500 N
I_y=46902000 mm^4

Submit eigenvalue analysis to check and eliminate rigid body modes

- Set up recommended *CONTROL_, *DATABASE_, and *SECTION cards for implicit analysis
- Set-up specific eigenvalue analysis *CONTROL_ cards
- Submit the analysis
- Check the results and compare against analytical solution

Submit linear static analysis

- Setup implicit load case
- Submit the analysis
- Check the results and compare against beam theory

Eigenvalue Analysis

For an implicit static analysis to solve there must be no rigid body modes in the model. If rigid body modes are present these must be eliminated by adding suitable restraints to the model. For this reason the first step in running a static analysis should be an eigenvalue analysis. Capturing the first 20 modes is more than sufficient.

Disable all *CONTROL cards in explicit input deck and activate the following:

```
$
*CONTROL_IMPLICIT_GENERAL
$:    imflag        dt0    imform      nsbs       igs     cnstn      form    zero_v
          1.0        0.0         0         0         0         0         0         0
$
$*CONTROL_IMPLICIT_EIGENVALUE
$:      neig    center     lflag    lftend     rflag    rhtend    eigmth     shfscl
         20.0       0.0         0       0.0         0       0.0         0       0.0
$:   isolid     ibeam    ishell   itshell    mstres    evdump   mstrscl
          0         0         0         0         0         0       0.0
*DATABASE_FORMAT
$:    iform   ibinary
          0         1
$
*DATABASE_BINARY_D3PLOT
$:       dt      lcdt    nobeam     npltc    psetid
        0.1         0         0         0         0
```

Turn on implicit solver

Set number of eigenvalues to solve for

32 bit **output files**

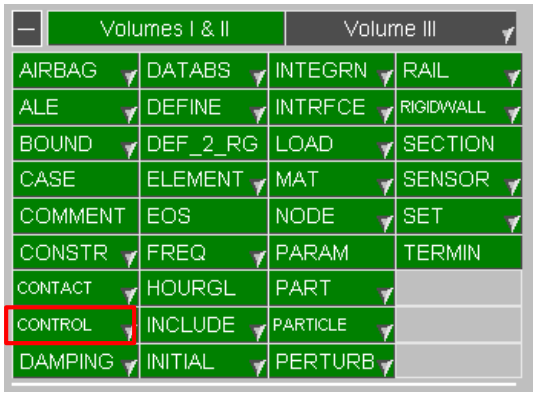
Set a higher **D3PLOT output** than you would for an explicit analysis. Make sure you capture any key points in the analysis.

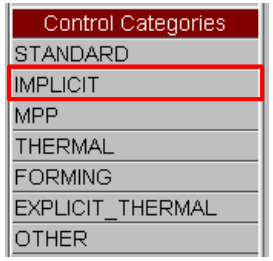
Eigenvalue Analysis

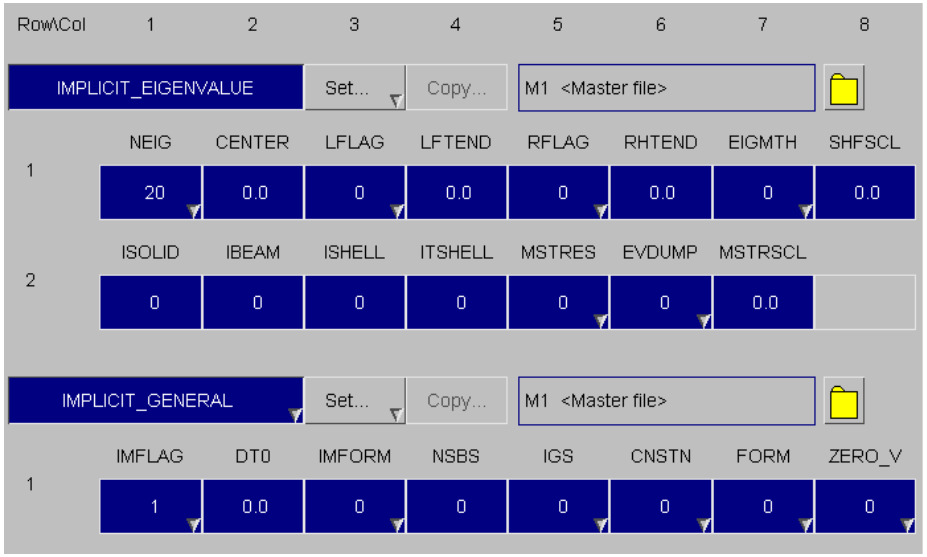
For an implicit static analysis to solve there must be no rigid body modes in the model. If rigid body modes are present these must be eliminated by adding suitable restraints to the model. For this reason the first step in running a static analysis should be an eigenvalue analysis. Capturing the first 20 modes is more than sufficient.

Disable all *CONTROL cards in explicit input deck and activate the following implicit cards:

- Click '**CONTROL**' in the Oasys PRIMER Keyword panel and then '**Modify**'
- Under '**Control Categories**', select '**IMPLICIT**'





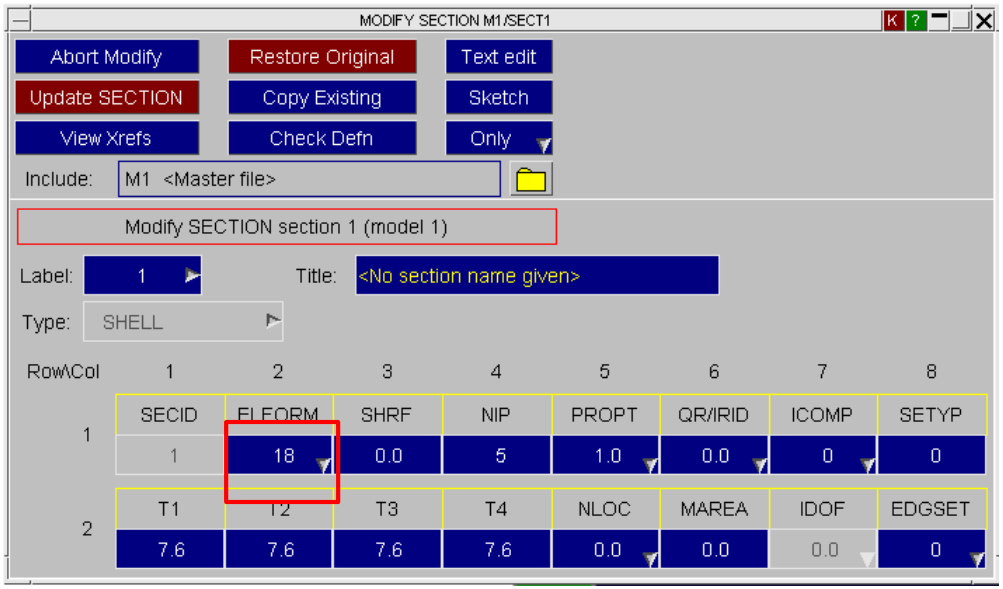


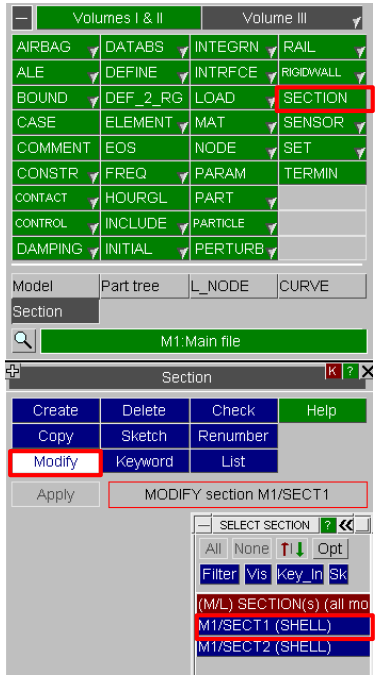
Element Formulations: Implicit

Particular element formulations (**ELFORM** on the *SECTION cards) are recommended for implicit analyses to give the most accurate results.

Linear Analyses (including eigenvalue)

- Solid Elements – Element formulation type 18 (20, 21, 99)
- **Shell Elements – Element formulation type 18 (20, 21, 99)**
- Beam Elements – Element formulation type 13





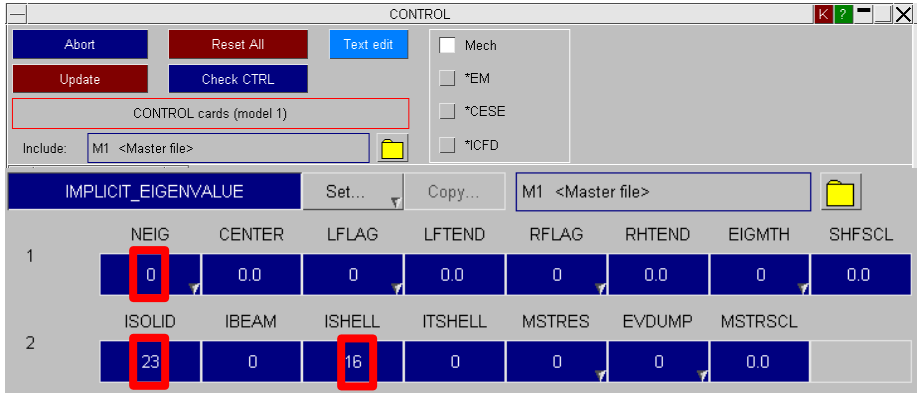
Element Formulations: Implicit

Rather than modifying an existing explicit model the **ISOLID, IBEAM, ISHELL, ITSHELL** options in the *CONTROL_IMPLICIT_EIGENVALUE card can be used to automatically reset the element type.

Note: As long as **NEIG**=0 on the *CONTROL_IMPLICIT_EIGENVALUE card LS-DYNA will not run an eigenvalue analysis.

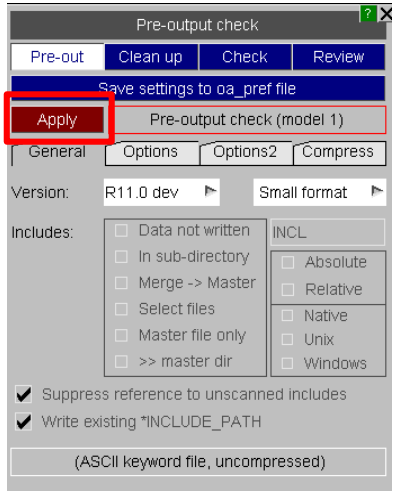
Example

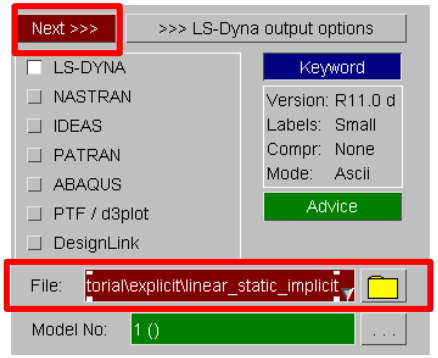
*CONTROL_IMPLICIT_EIGENVALUE settings for switching all solid elements to formulation 23 and all shell elements to formulation 16:

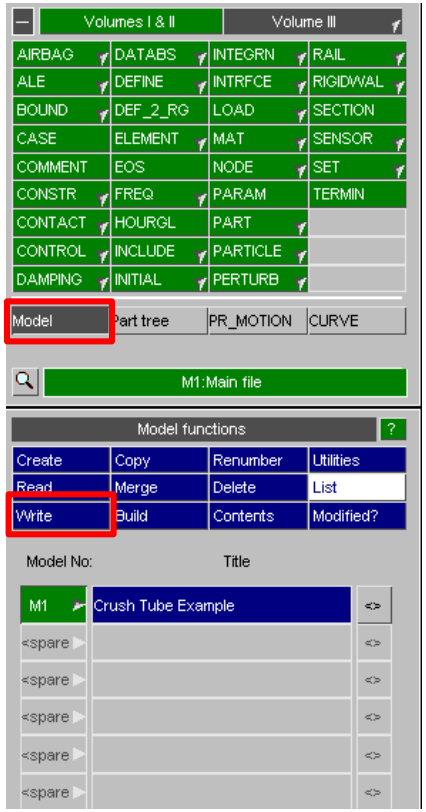


Pre-processing: Saving the model

- To save the model click '**Model**' and then '**Write**'.
- Define a file path and name (**linear_static_eigv.key**) and then click '**Next**' and finally click '**Apply**', to write the file.
- The written keyword file can now be submitted to LS-DYNA.



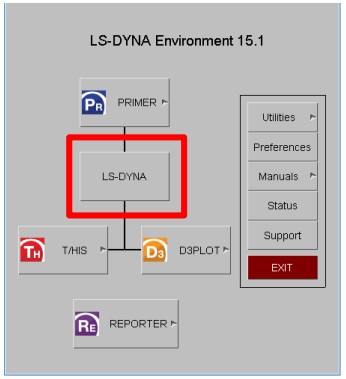


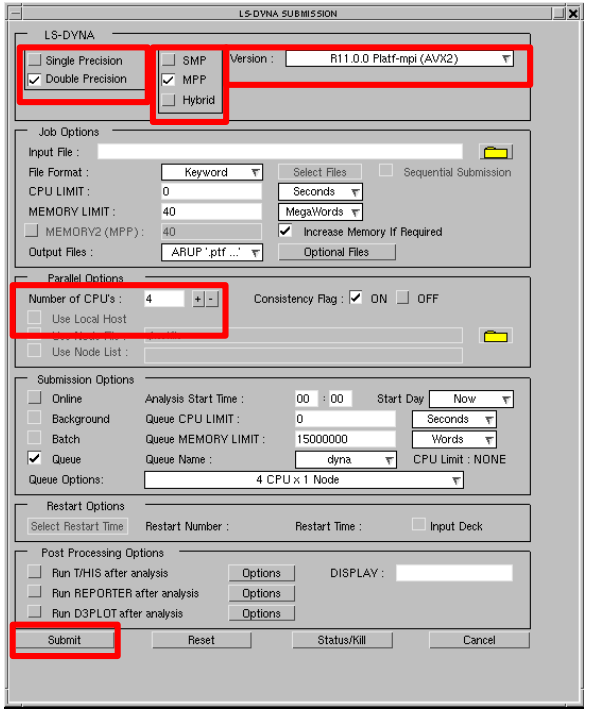


Solver: Submit LS-DYNA job

- In Oasys SHELL, select the following settings:
  - **Double Precision:** Double precision (DP) use 64 bit storage and is useful when extremely small deflections or long event times are expected. Recommended for implicit analyses.
  - **MMP:** 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. Recommended for implicit analyses.
  - **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**'.

For best implicit performance, it is important to provide enough memory to allow the stiffness matrix factorization to run in-core





Eigenvalue Analysis - “eigout” file

Run the analysis in LS-DYNA and check for rigid body (zero frequency) modes in the “eigout” file, see below.

Check for frequencies at or near zero

problem time = 1.00000E+00

(all frequencies de-shifted)

		frequency		
MODE	EIGENVALUE	RADIANS	CYCLES	PERIOD
1	6.915707E+03	8.316073E+01	1.323544E+01	7.555472E-02
2	5.298313E+04	2.301807E+02	3.663439E+01	2.729676E-02
3	5.618522E+04	2.370342E+02	3.772517E+01	2.650750E-02
4	2.678063E+05	5.175000E+02	8.236269E+01	1.214142E-02
5	8.222794E+05	9.067963E+02	1.443211E+02	6.928993E-03
6	1.842911E+06	1.357539E+03	2.160590E+02	4.628366E-03
7	2.029133E+06	1.424476E+03	2.267125E+02	4.410873E-03
8	3.998877E+06	1.999719E+03	3.182652E+02	3.142034E-03
9	6.946053E+06	2.635537E+03	4.194587E+02	2.384025E-03
10	7.207740E+06	2.684723E+03	4.272870E+02	2.340347E-03

λ $\omega = \sqrt{\lambda}$ $f = \omega / 2\pi$ $T = 1/ f$

The modes are output to the “d3eigv” file which can be animated in Oasys D3PLOT.

Post-processing: Oasys D3PLOT

LS-DYNA Implicit

Top menus

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

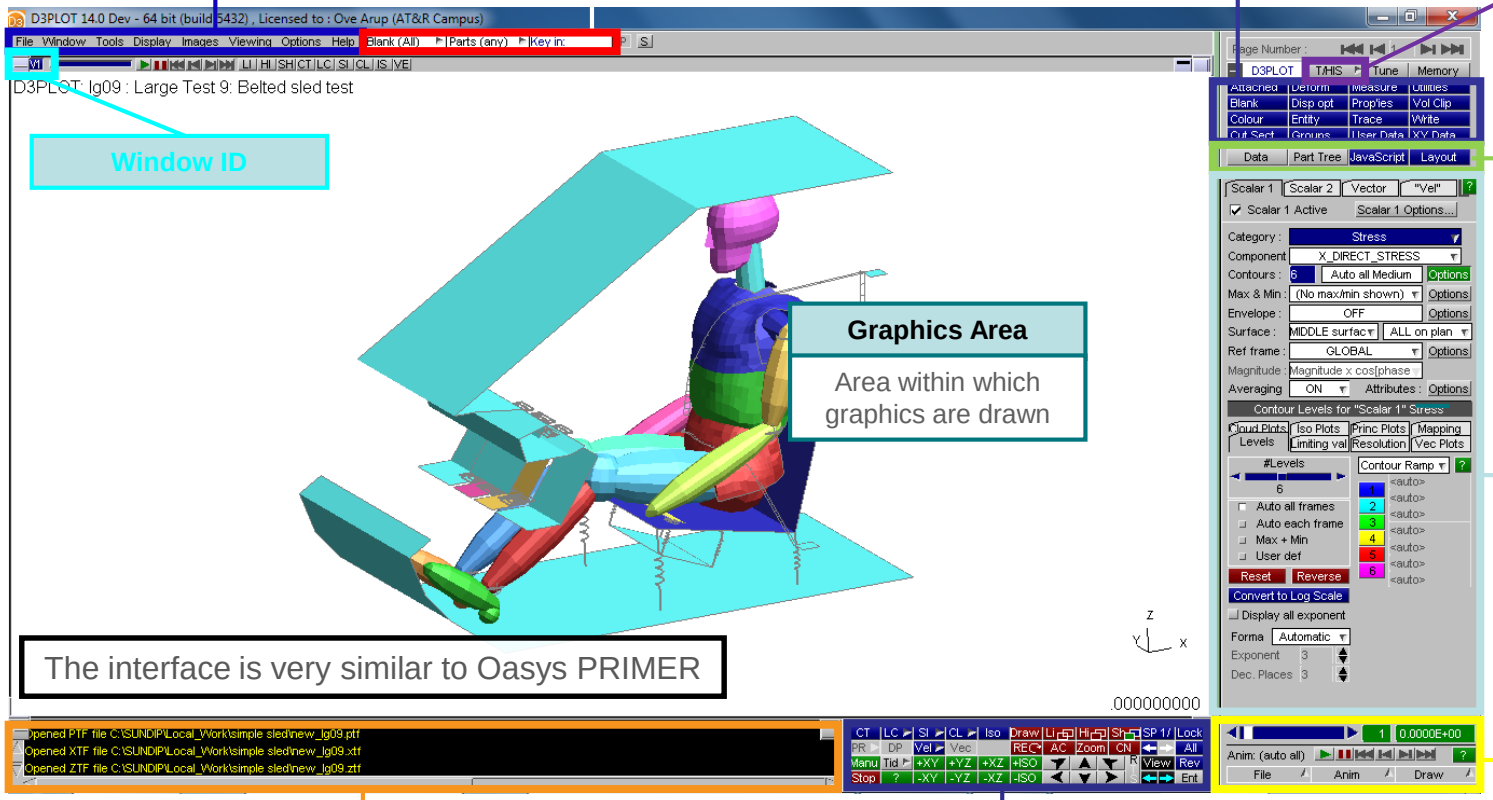
Quick-Pick Control

Controls the mouse action when applied within the graphics area

Tools

Provides access to D3PLOT specific functions

T/HIS link



Window ID

Graphics Area

Area within which graphics are drawn

The interface is very similar to Oasys PRIMER

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

Dialogue & List area

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

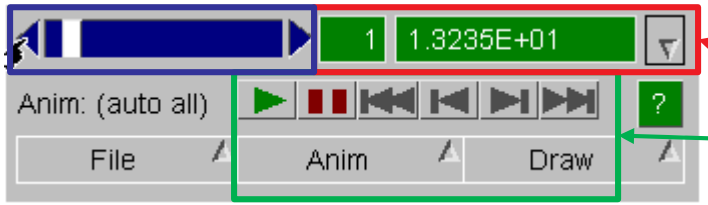
Viewing & Drawing Commands

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

Post-processing: Read in d3eigv file

- Open the d3eigv file from the run directory in Oasys D3PLOT.
- Note: Eigenvalue analysis is a special and the usual d3plot/ptf file will not display any results
- Animate the results and check the mode shapes of the beam.

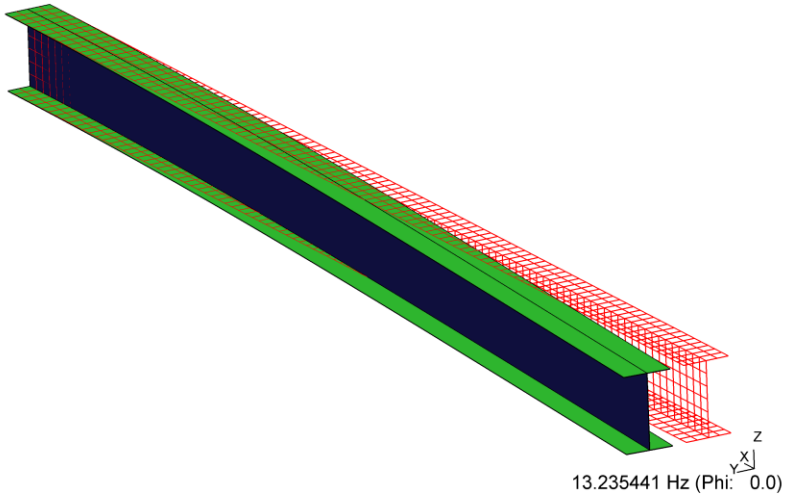
Position the slide on the state number

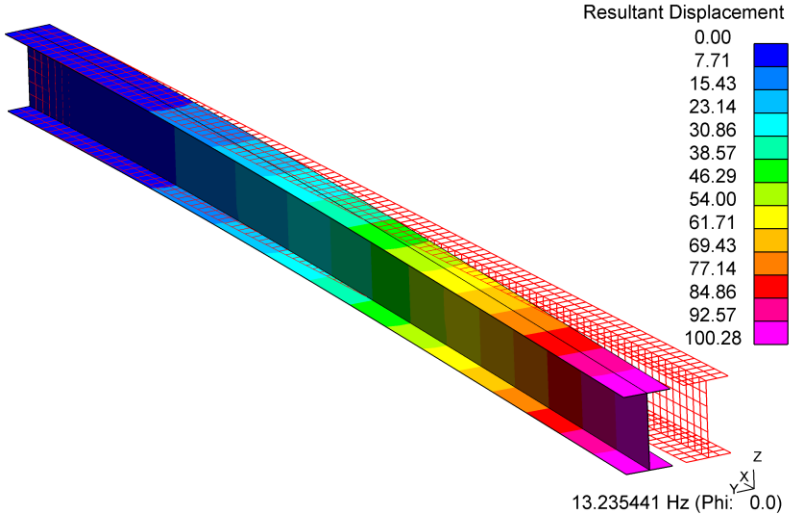


Natural frequency

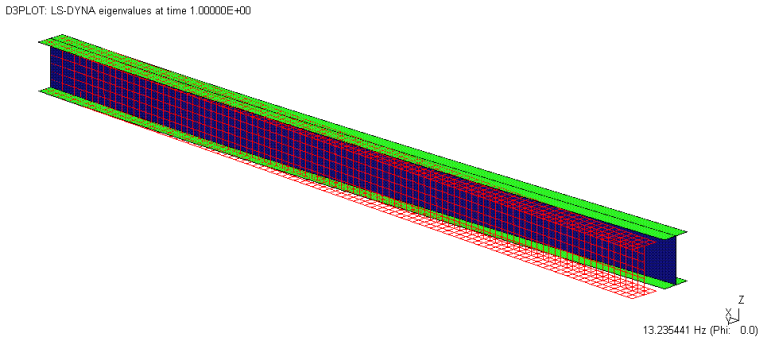
Animate results

Contour plots

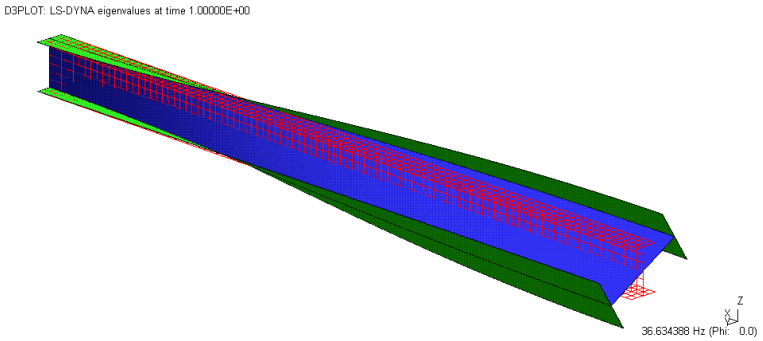




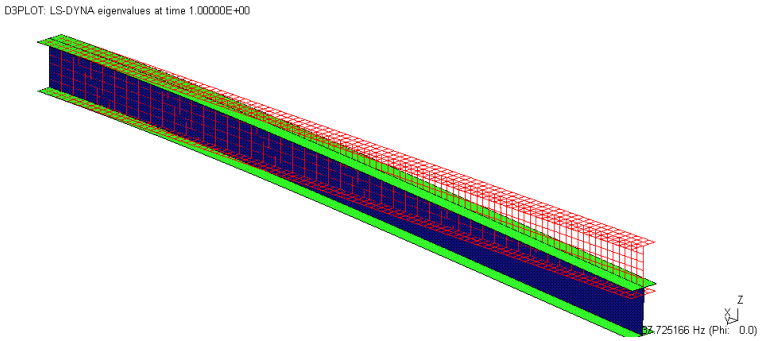
Eigenvalue Analysis - “d3eigv” file



Mode 1 – 13Hz



Mode 2 – 37Hz

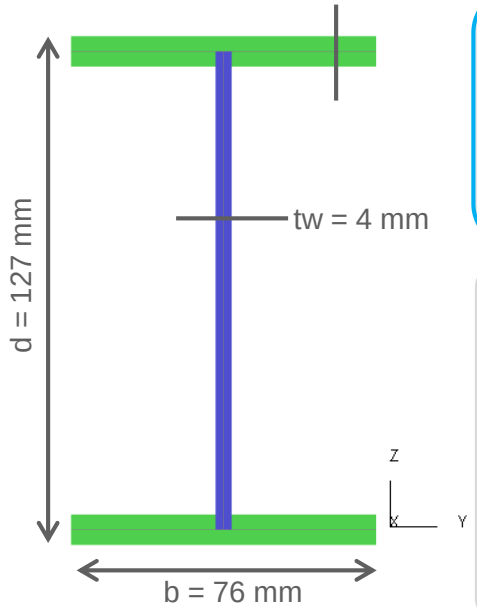


Mode 3 – 38Hz

No Rigid Body Modes

This is because the beam has been suitably restrained. (Try removing the SPC: rigid modes will appear)

Eigenvalue Analysis – Hand calculation



Hand calculation: 1st Natural Frequency

$$f_1 = \frac{1.875^2 \sqrt{\frac{EI}{\rho A l^4}}}{2\pi}$$

E = 205000 N/mm2
L = 2000 mm
Density = 7.85e-9 T/mm3

$$I_y = \sum \left(I_{yi} + A d_z{}^2\right) = \frac{1}{12} 4 \cdot 119.4^3 + 2 \cdot \left(\frac{1}{12} 76 \cdot 7.6^3 + (7.6 \cdot 76)\left(\frac{119.4}{2}\right)^2\right) = 4690200 mm^4$$

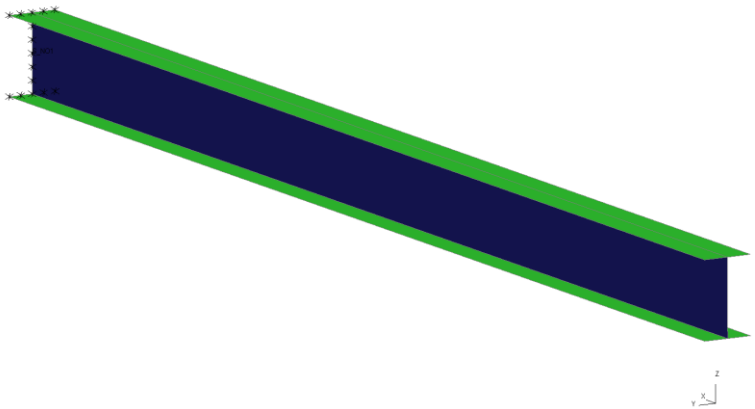
$$f_1 = \frac{1.875^2 \sqrt{\frac{EI_y}{\rho A l^4}}}{2\pi} = \frac{1.875^2 \sqrt{\frac{205000 \cdot 4690200}{7.85e^{-9} \cdot 1632.8 \cdot 2000^4}}}{2\pi} = \mathbf{38.3Hz}$$ ✓

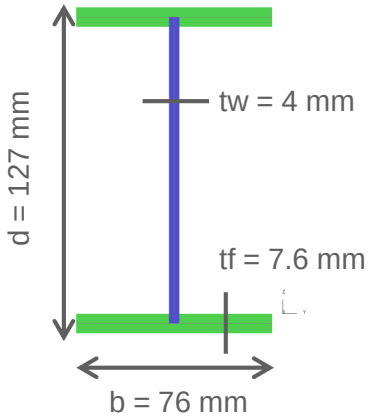
$$I_z = \sum \left(I_{zi} + A d_y{}^2\right) = \frac{1}{12} 119.4 \cdot 4^3 + 2 \cdot \left(\frac{1}{12} 7.6 \cdot 76^3\right) = 556673.07 mm^4$$

$$f_1 = \frac{1.875^2 \sqrt{\frac{EI_z}{\rho A l^4}}}{2\pi} = \frac{1.875^2 \sqrt{\frac{205000 \cdot 556673.07}{7.85e^{-9} \cdot 1632.8 \cdot 2000^4}}}{2\pi} = \mathbf{13.2Hz}$$ ✓

Explicit-implicit model conversion

Recommended steps





E = 205000 N/mm^2
L = 2000 mm
Density = 7.85e-9 T/mm^3
Pr = 0.3
P = 2500 N
I_y=46902000 mm^4

Submit eigenvalue analysis to check and eliminate rigid body modes

- Set up recommended *CONTROL_, *DATABASE_, and *SECTION cards for implicit analysis
- Set-up specific eigenvalue analysis *CONTROL_ cards
- Submit the analysis
- Check the results and compare against analytical solution

Submit linear static analysis

- Setup implicit load case and boundary conditions
- Submit the analysis
- Check the results and compare against beam theory

Linear Static Analysis: Output forces

When running a **static analysis** either linear or non-linear it is often useful to request output to the “binout” file so that the boundary forces can be checked against the applied load. If there is an imbalance then the results are likely to be in error.

Restrained nodes

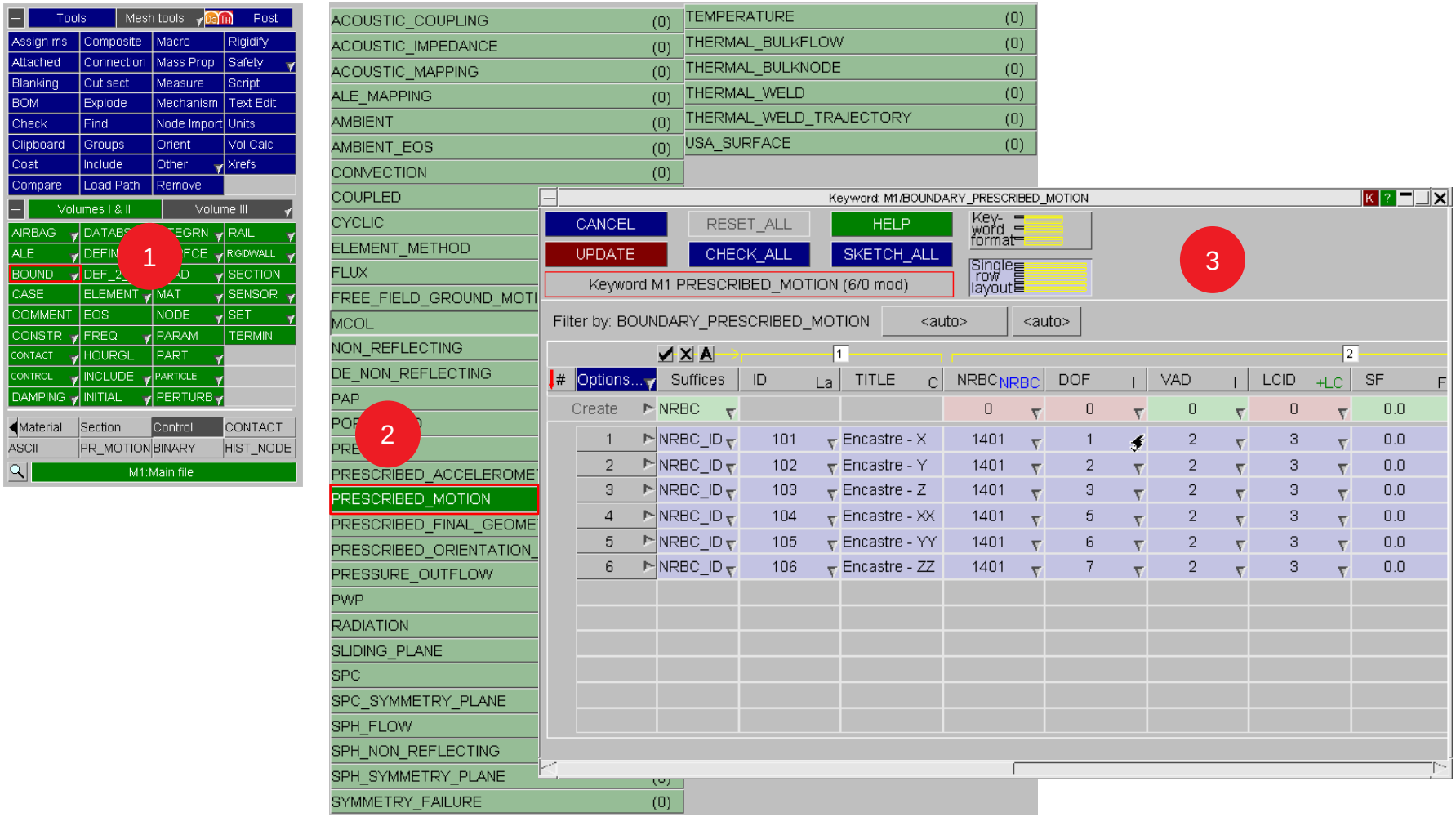
- Output to “spcforc” file or
- Output sets of nodes to “nodfor” file

Rigid Parts and Nodal Rigid Bodies

- If these are restrained then at present it is not possible to get the restraining forces output.
- To get these forces the restraints must be replaced by:
  - *BOUNDARY_PRESCRIBED_MOTION_NRBC and
  - Setting the “scale factor” to zero
  - Turn on output to the “bndout” file

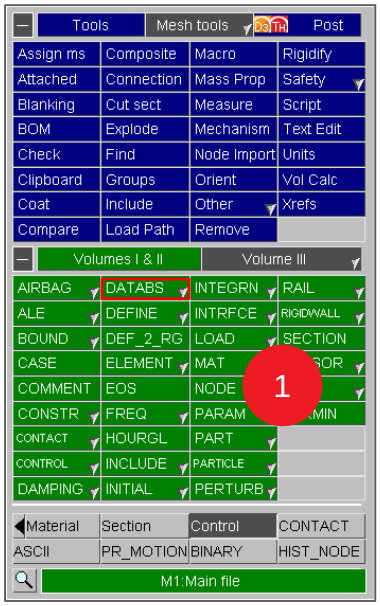
Linear Static Analysis: Output forces

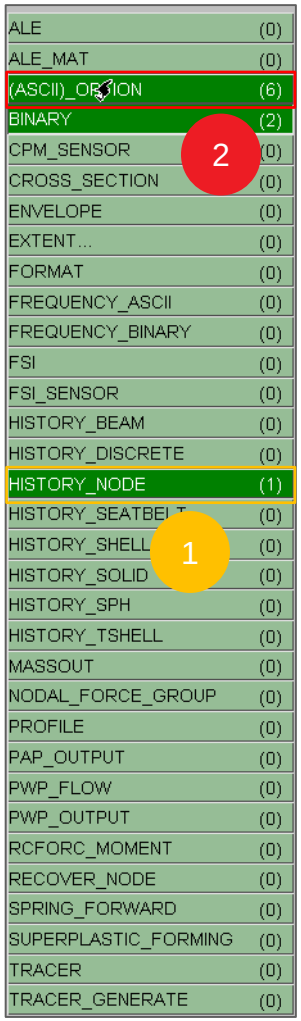
*BOUNDARY_PRESCRIBED_MOTION_NRBC

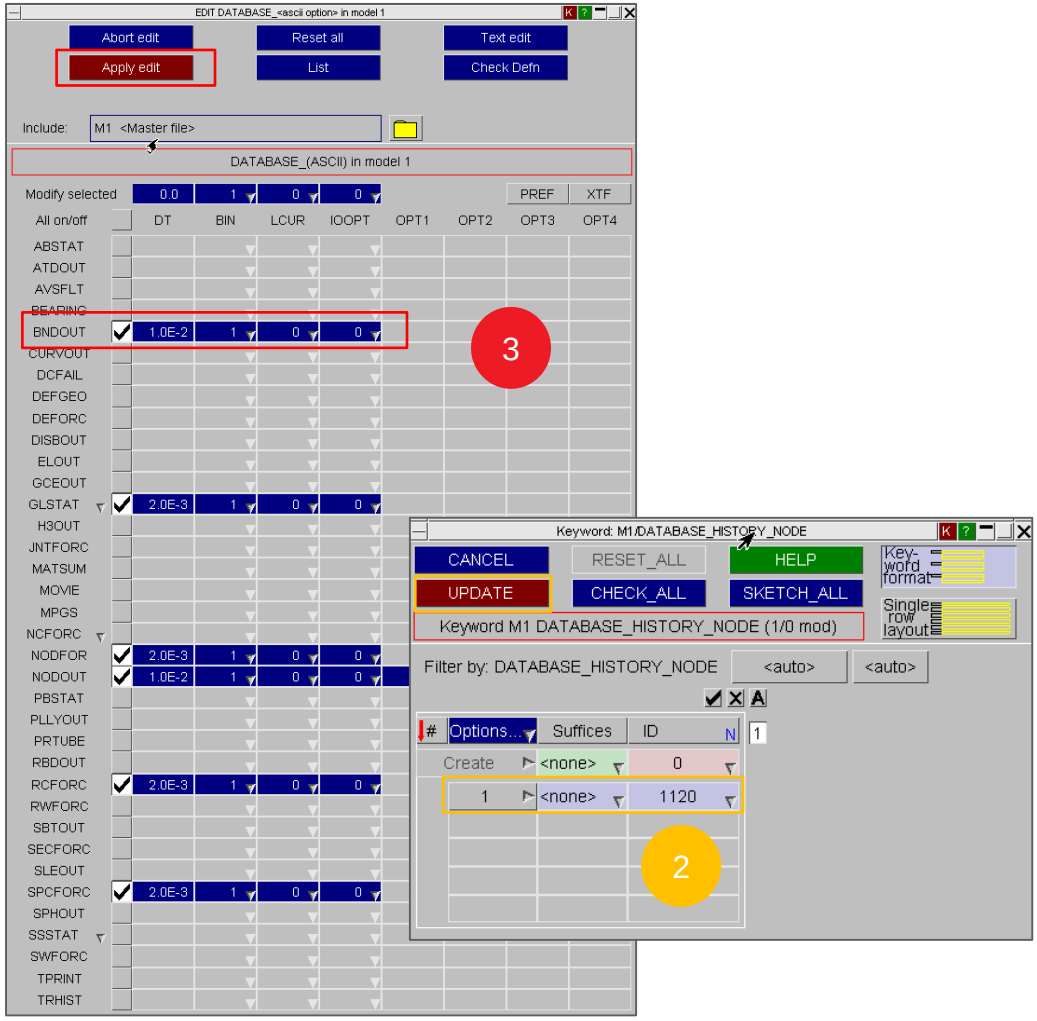


Linear Static Analysis: Output forces

*DATABASE (ASCII)_OPTION/*DATABSE_HISTORY_NODE







Linear Static Analysis: Control Cards


```
*CONTROL_ACCURACY
$       OSU       INN    PIDOSU      IACC
          1         2                    1
*CONTROL_MPP_IO_NODUMP
*CONTROL_MPP_IO_NOD3DUMP
*CONTROL_MPP_IO_NOFULL
*CONTROL_MPP_IO_LSTC_REDUCE
*CONTROL_ENERGY
$      HGEN      RWEN    SLNTEN     RYLEN
          2         2         2         2
*CONTROL_HOURGLASS
$       IHQ        QH
          6      0.10
*CONTROL_OUTPUT
$:    npopt    neecho    nrefup    iaccop     opifs    ipnint    ikedit    iflush
          1         3         0         0       0.0         0         0         0
$:    iprtf    ierode     tet10    msgmax    ipcurv      gmdt   ip1dblt      eocs
          0         0         0         0         0       0.0         0         0
$:    tolev    newleg    frfreq     minfo    solsig    msgflg    cdetol
          0         0         0         1         0         0       0.0
$===========================================================================
*CONTROL_IMPLICIT_GENERAL
$:   imflag       dt0    imform      nsbs       igs     cnstn      form    zero_v
          1       1.0         0         0         0         0         0         0
$
*CONTROL_IMPLICIT_SOLUTION
$:   nsolvr    ilimit    maxref      dctol     ectol     rctol     lstol     abstol
          1         0         0       0.0       0.0       0.0       0.0       0.0
$:    dnorm    diverg     istif   nlprint    nlnorm   d3itctl     cpchk
          0         0     99999         0         0         0         0
*CONTROL_IMPLICIT_SOLVER
$:   lsolvr    lprint     negev     order      drcm    drcprm   autospc   autotol
          5         2         0         0         0       0.0         2       0.0
$
*CONTROL_TERMINATION
$:   endtim    endcyc      dtmin     endeng    endmas      nosol
         3.0         0       0.0       0.0       0.0          0
```

Turn on **objective stress update**, **invariant node numbering** and **implicit accuracy**.

Remove excessive dump files

Turn on **implicit solver**

Linear solver on: Don't update stiffness matrix

Use solver 5 – Only solver in MPP, Get statistics

Termination Time (Load step) set to 3.0

Solver: .otf file (Analysis status file)

Open the .otf file and look for: `Error, Warning, t o t a l, 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 (`t o t a l` 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      38524
shell      3
shell
shell
```

```
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
```

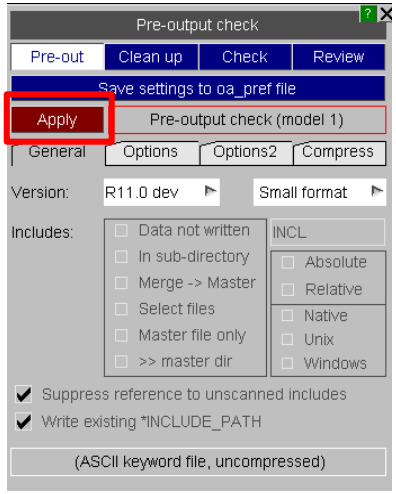
```
problem cycle        =         1200
time                 =   1.2949E-03
added mass           =   4.3696E-04
percentage increase  =   2.7420
```

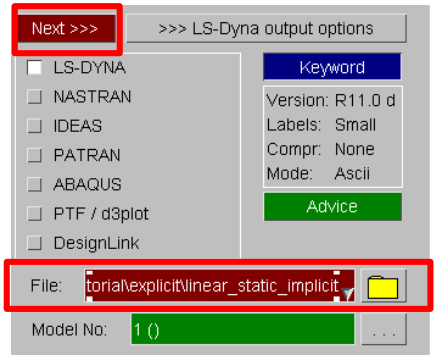
```
Deformable Spotwelds *****
total added spotweld mass =    1.3139E-05
percentage mass increase  =    8.2451E-04
```

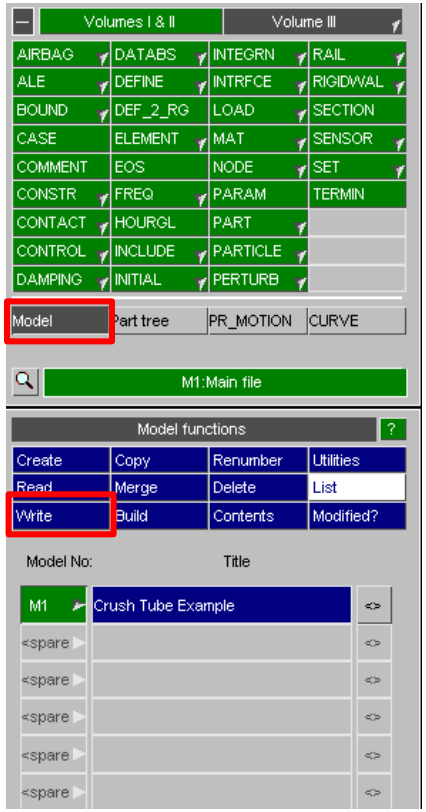
```
 T i m i n g   i n f o r m a t i o n
                      CPU(seconds)    %CPU  Clock(seconds) %Clock

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
-------------------------------------------------------------------
 T o t a l s          8.5305E+03  100.00     4.2303E+03  100.00

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

N o r m a l   t e r m i n a t i o n
```

Pre-processing: Saving the model

- To save the model click '**Model**' and then '**Write**'.
- Define a file path and name (**linear_static_implicit.key**) and then click '**Next**' and finally click '**Apply**', to write the file.
- The written keyword file can now be submitted to LS-DYNA.



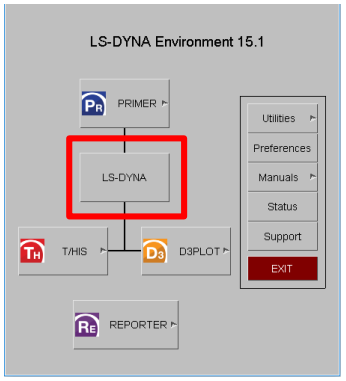


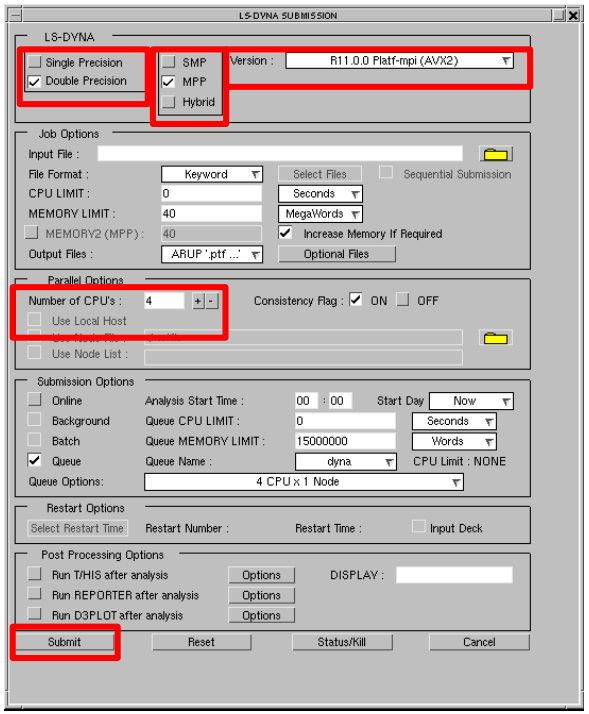


Solver: Submit LS-DYNA job

- In Oasys SHELL, select the following settings:
  - **Double Precision:** Double precision (DP) use 64 bit storage and is useful when extremely small deflections or long event times are expected. Recommended for implicit analyses.
  - **MMP:** 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. Recommended for implicit analyses.
  - **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**'.

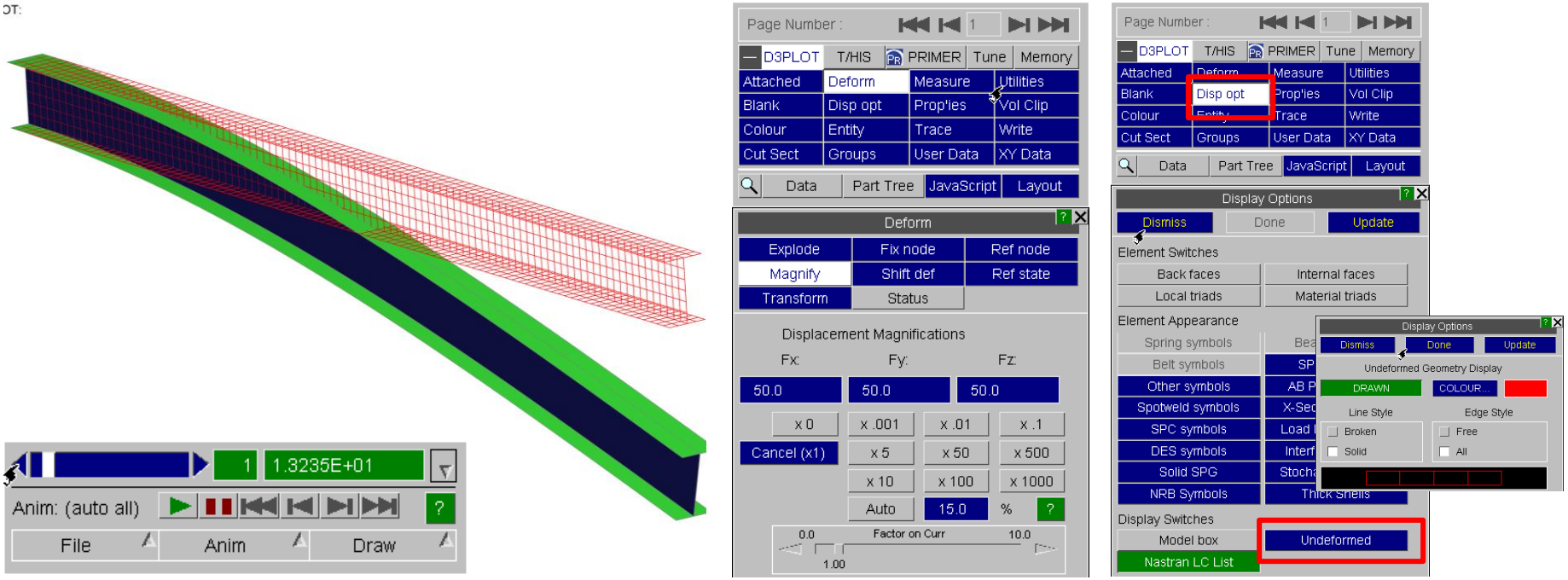
For best implicit performance, it is important to provide enough memory to allow the stiffness matrix factorization to run in-core





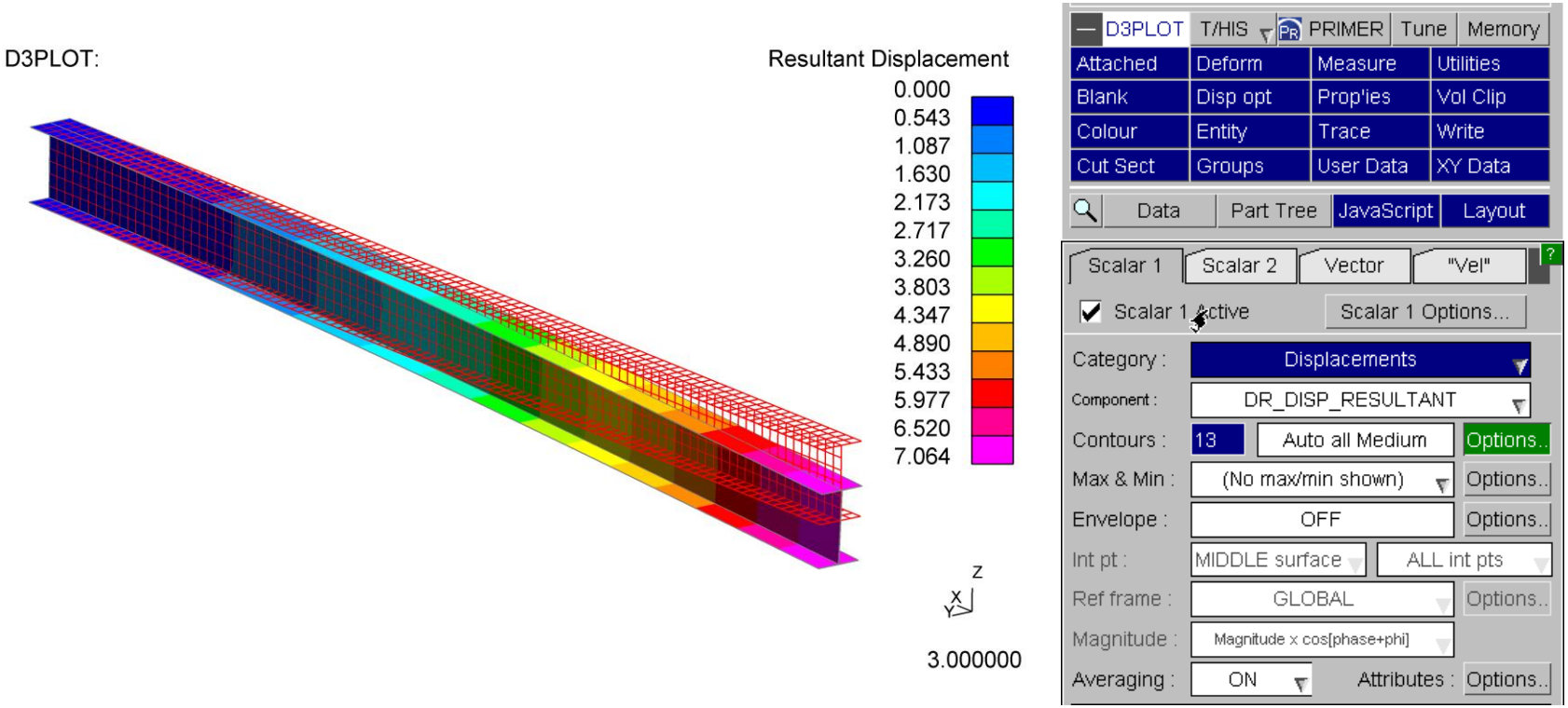
Post-processing: Animate the results

- 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.
- Magnify the displacements and display the undeformed geometry.,



Post-processing: Contour deflection

- Select **Category**: Displacements
- **Component**: DR_DISP_RESULTANT 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.

Post-processing: Oasys T/HIS

LS-DYNA Implicit

Top menus

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

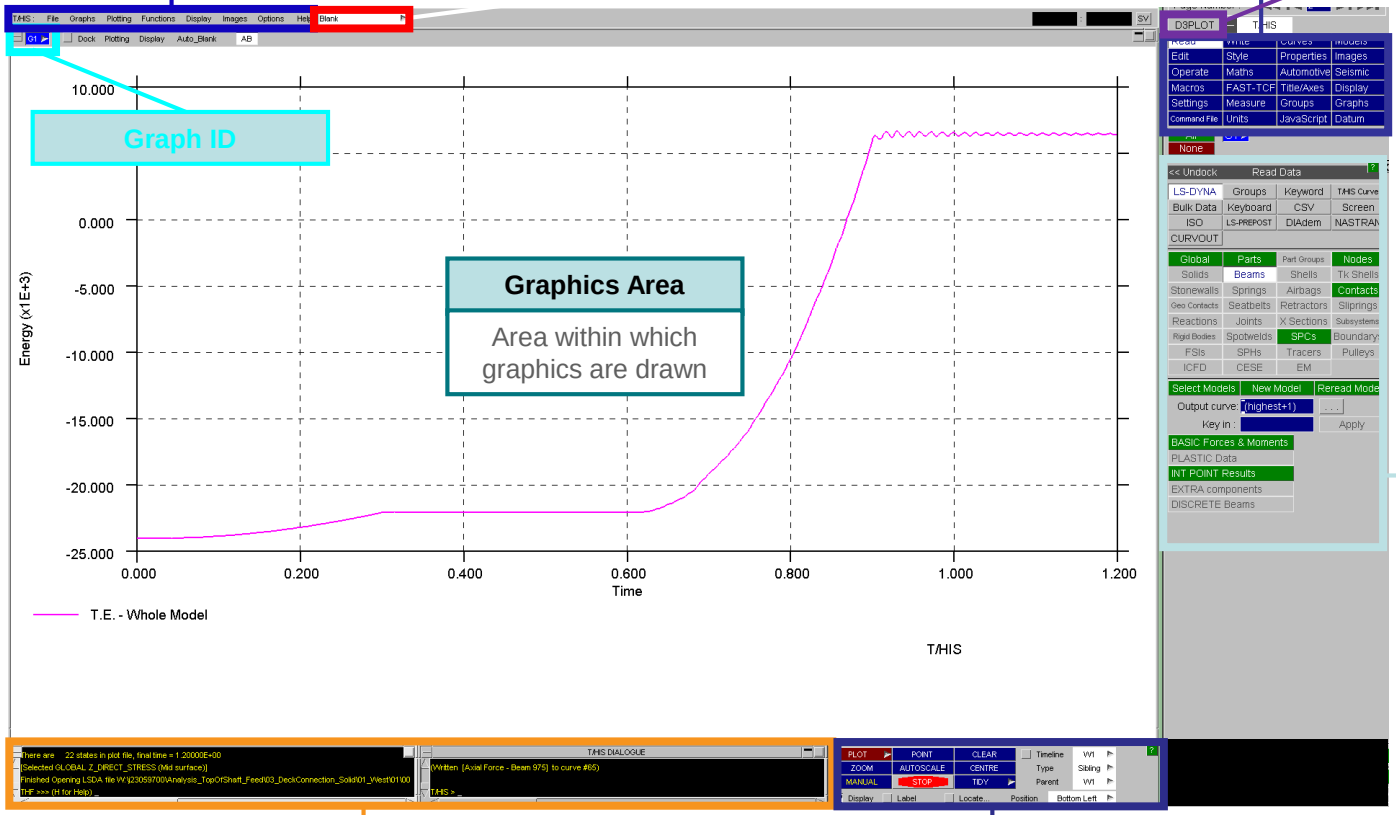
Quick-Pick Control

Controls the mouse action when applied within the graphics area

Tools

Provides access to T/HIS specific functions

D3PLOT Link



Dialogue & List area

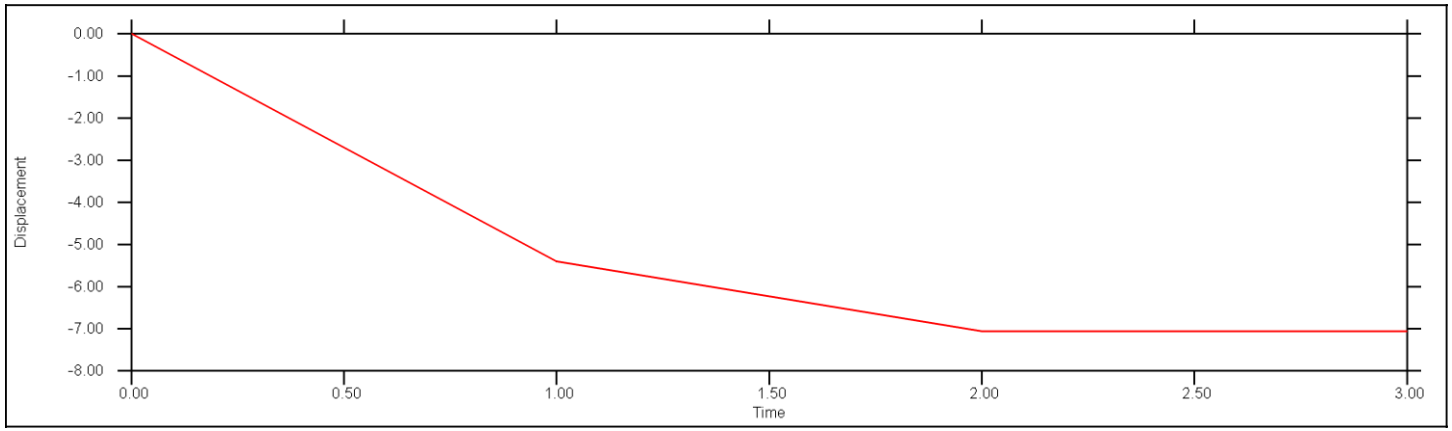
Area for command-line input and output, also acts a listing area for messages

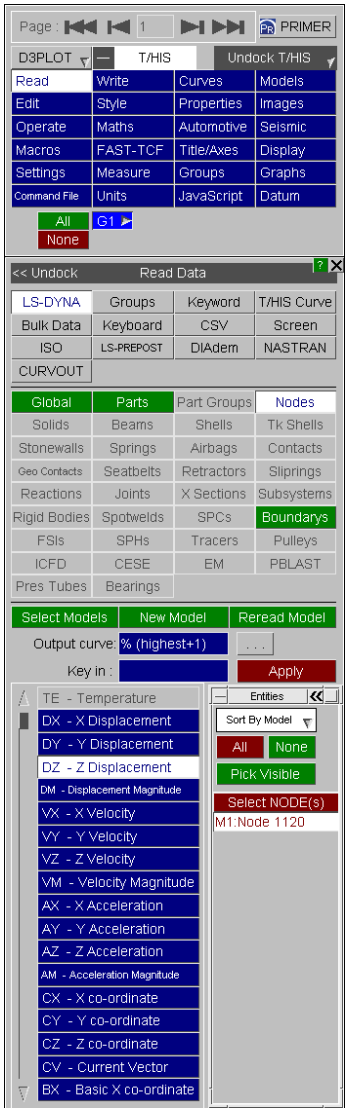
Viewing Commands

These are different to PRIMER and D3PLOT

Post-processing: Beam deflection

- Click on **T/HIS** button via the Main Menu Option - this will open a T/HIS session within D3PLOT.
- Click "**Nodes**".
- Select '**DZ – Z Displacement**'**.**
- Click '**Apply**' to plot data graph.
- 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'.

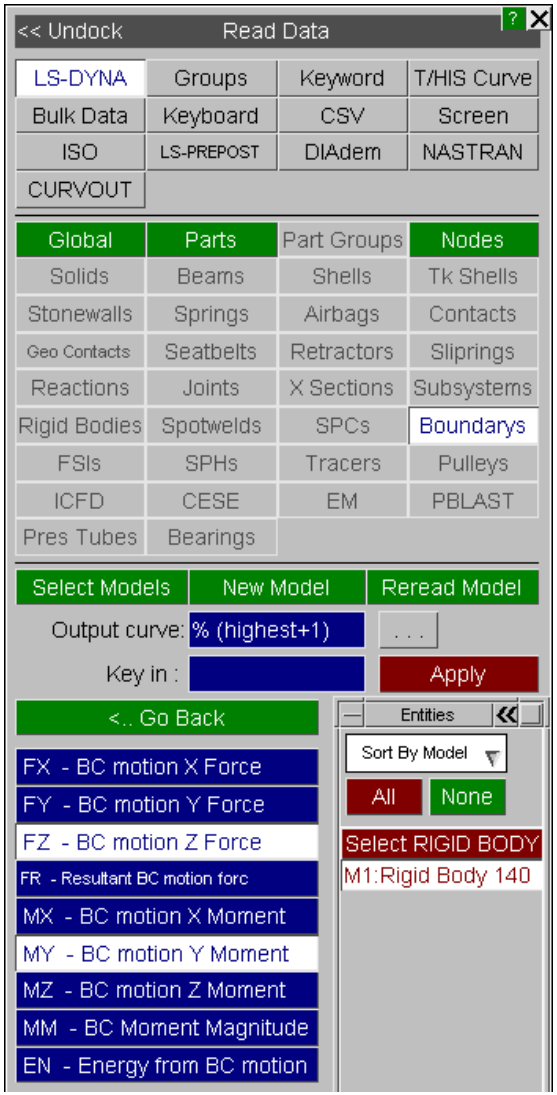


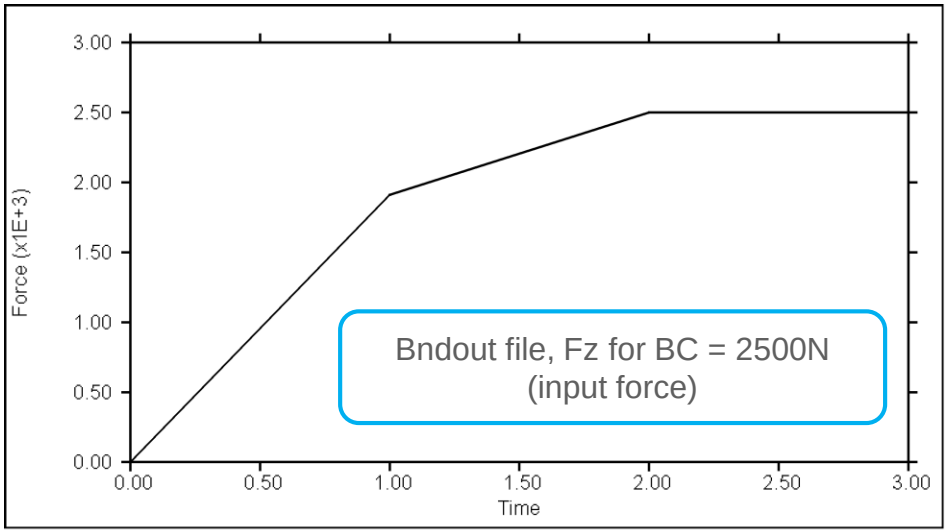


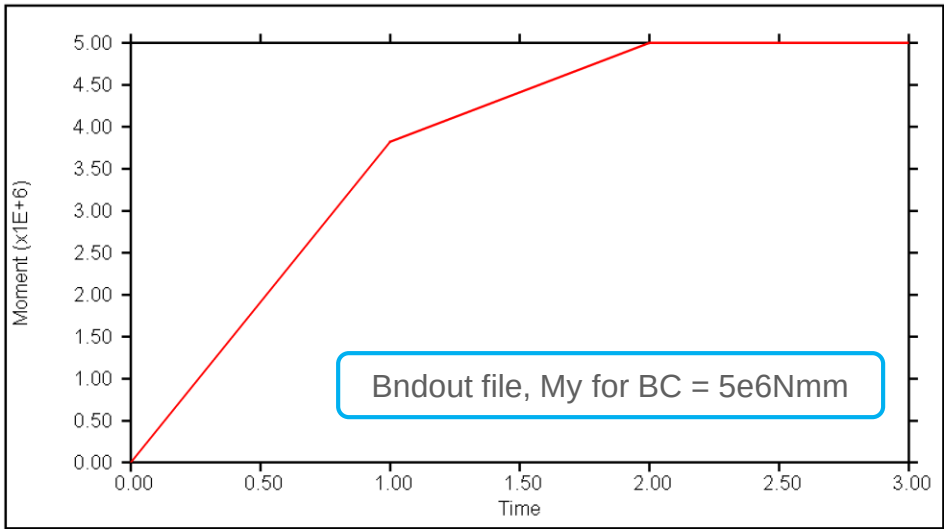
While D3PLOT allows visual representation of this result and interaction with it, T/HIS allows for plotting and manipulation of time histories. All data for post processing must be requested in the database cards when setting up the run.

Post-processing: Reaction forces and moments

LS-DYNA Implicit







Check reaction forces and moments

Linear Static Analysis: Results

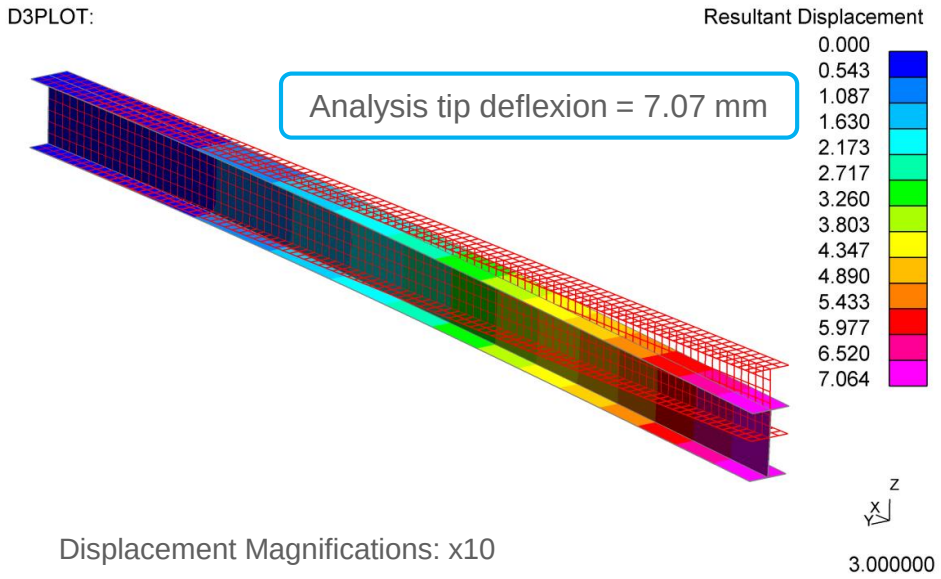
Hand calculation: Tip Deflection

$$\mathrm{d} = \frac{PL^3}{3EI}\left(1 + \frac{3f_sEI}{GAL^2}\right) \text{ where } f_s = \frac{A}{A_{web}}$$

E = 205000 N/mm2
L = 2000 mm
Density = 7.85e-9 T/mm3
P=5000N
pr=0.3
Iy=46902000mm4

$$d_z = \frac{PL^3}{3EI_y}\left(1 + \frac{3f_sEI_y}{GAL^2}\right) = \frac{2.5 \cdot 2000^3}{3 \cdot 20500 \cdot 4690200}\left(1 + \frac{3 \cdot \left({}^{1632.8}/_{477.6}\right) \cdot 20500 \cdot 4690200}{7.88e^4 \cdot 1632.8 \cdot 2000^2}\right) = 13.9 \cdot (1 + 0.019) = 7.07\ mm$$

$$G = \frac{E}{2(1+\upsilon)}$$



Hand calculation: Max Bending Moment

$$M_y = P \cdot L$$

$$M_y = P \cdot L = 2500 \cdot 2000 = 5e6Nmm$$

Bndout file, My for BC = 5e6Nmm
Bndout file, Fz for BC = 2500N (input force)

ARUP

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

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

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