

# Implicit Setup Tool



# What is the Implicit Setup Tool?

- The Implicit Setup Tool can be used to help setup the **\*CONTROL** cards required for various implicit analysis types.
- This tool can be accessed via the **Tools → Implicit** button.
- This tool should ideally be used when no **\*CONTROL\_IMPLICIT\_...** keywords already exist in the model, however the tool has settings to control how it behaves when they do.

**NOTE:** this tool will not setup any loads/boundary constraints/etc. that may be needed for the analysis – these must be setup outside of this tool.



# Implicit vs. Explicit



# Implicit vs. Explicit

Applications	
<b>Implicit</b> <ul style="list-style-type: none"><li>• Low rate dynamic analyses</li><li>• Linear and non-linear Static analysis</li><li>• Modal and vibration analysis</li><li>• Strength and buckling</li><li>• Springback</li><li>• Gravity loading &amp; Pre-loading</li></ul>	<b>Explicit</b> <ul style="list-style-type: none"><li>• High rate dynamic analyses</li><li>• Car crash</li><li>• Impact / Penetration problems</li><li>• Explosives</li></ul>
Advantages / Disadvantages	
<b>Implicit</b> <ul style="list-style-type: none"><li>• Unconditionally stable (no timestep limit)</li><li>• Can be used for static analysis</li><li>• Relatively inexpensive for long duration analyses</li><li>• Often requires a large amount of memory</li><li>• Can have problems with strongly non-linear models</li></ul>	<b>Explicit</b> <ul style="list-style-type: none"><li>• Computationally fast</li><li>• Robust even for strongly non-linear models</li><li>• Low core / RAM intensive</li><li>• Conditionally stable (timestep limit)</li><li>• Expensive to conduct long duration analyses</li></ul>

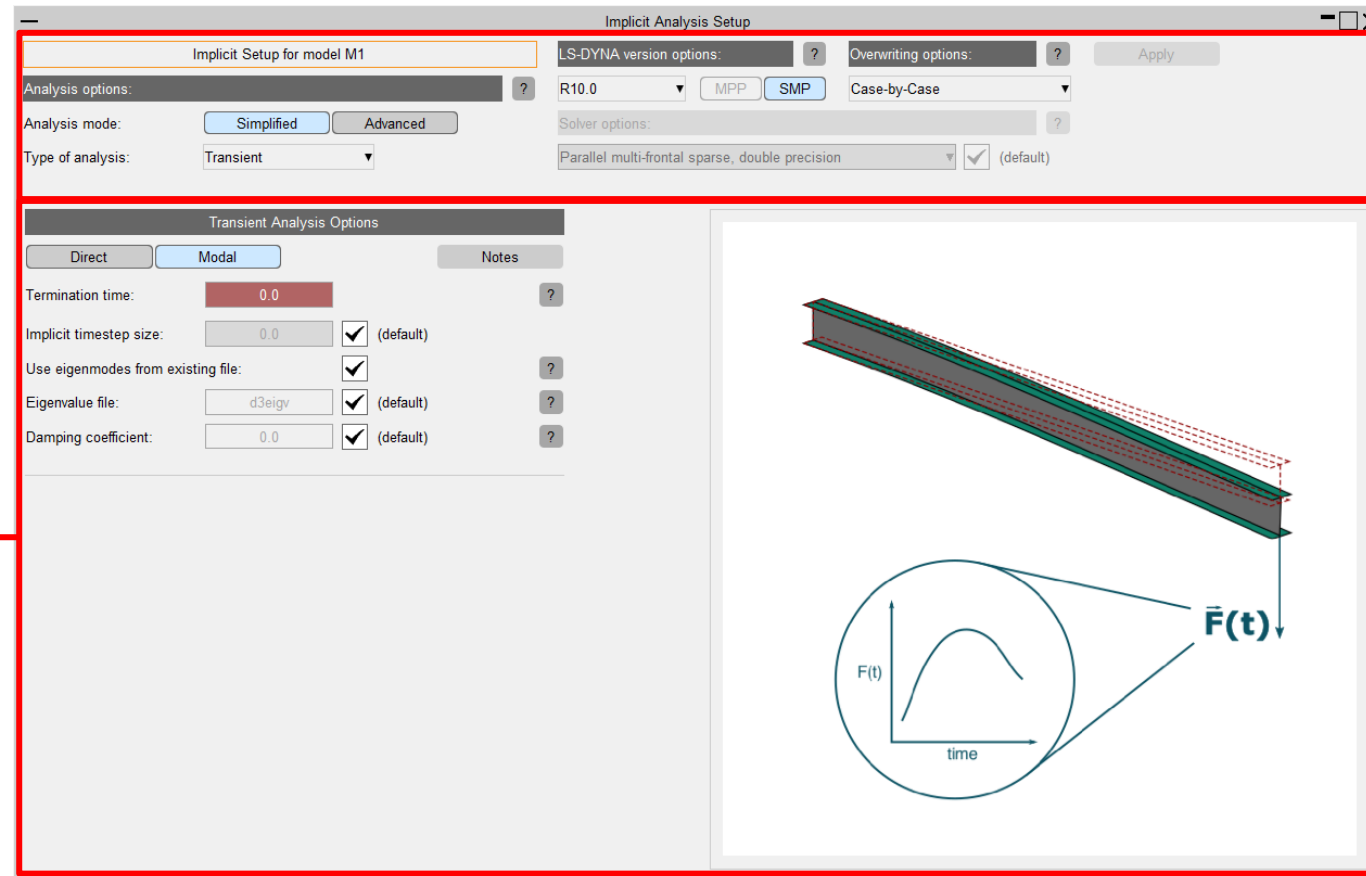
# General window overview



# Implicit Setup Tool – Top section options

The Implicit Setup Tool can be split into two sections:

- **Top Section** – settings that apply to all analysis types.
- **Analysis options** – settings that apply specifically to the chosen analysis type.

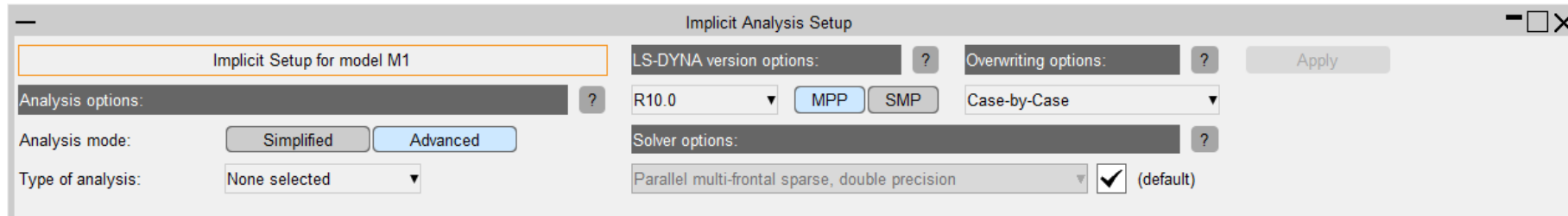


# Top section settings



# Implicit Setup Tool – Top section options

The settings at the top of the window apply to all analyses:



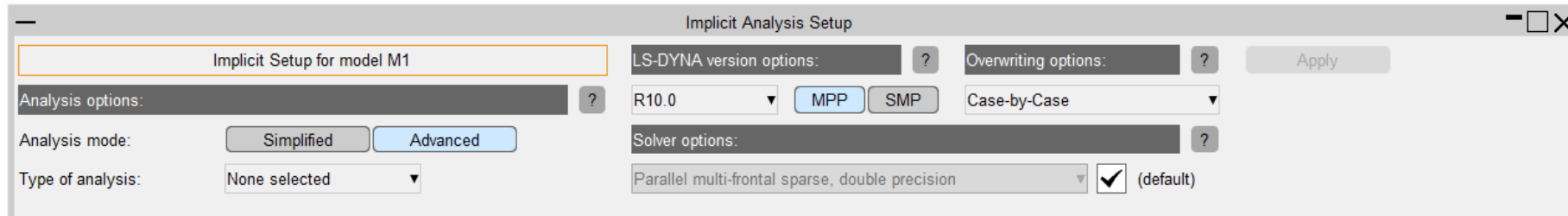
- **Simplified/Advanced** – Switch between **Simplified** mode: default/best guess settings are used to reduce the amount of input required; and **Advanced** mode: more options are available for a more customised/complex analysis.
- **Type of analysis** – Select from: Static, Transient (Direct or Modal), Buckling, Eigenvalue, or Frequency Domain (FRF).
- **LS-DYNA version options** – Set the output LS-DYNA version (this must be honoured when saving the model as it can affect what options are available), and choose between MPP and SMP (this must match the option used when submitting the analysis job).





# Implicit Setup Tool – Top section options

The settings at the top of the window apply to all analyses:



- **Solver options** – Select which linear solver method will be used to solve the inverse of the stiffness matrix (see \*CONTROL\_IMPLICIT\_SOLVER LSOLVR for details). *NOTE: This option can only be changed when using Advanced mode.*
- **Overwriting options** – Select how the tool will handle writing to keywords that already exist in the model:
  - **Overwrite** will overwrite all keywords that clash with the options from the tool.
  - **Ignore** will not overwrite any keywords that clash.
  - **Case-by-Case** will present the keywords that clash and offer the choice whether to ignore/overwrite them.

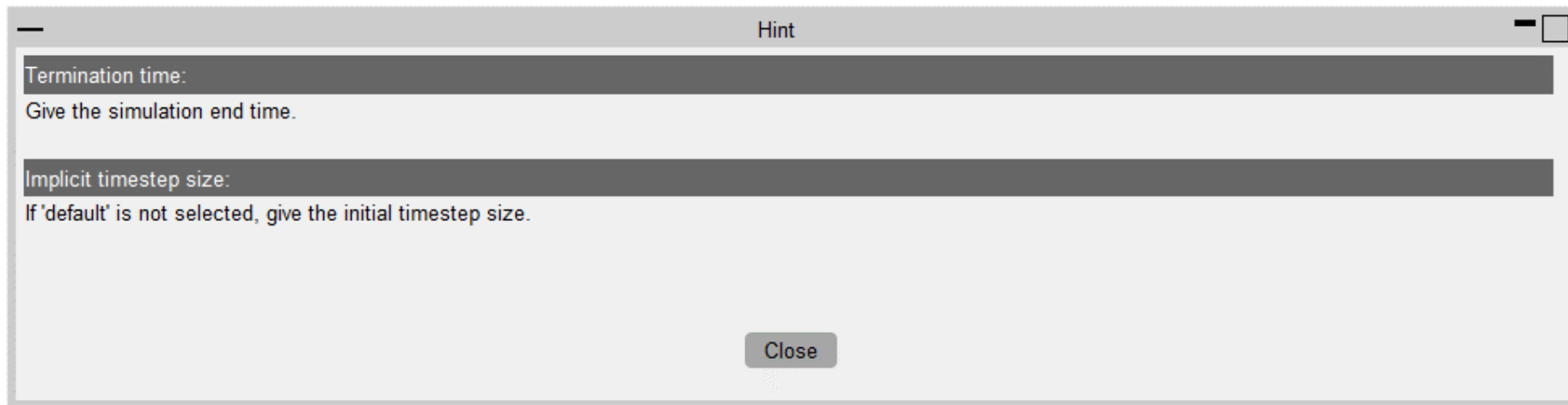


# Guide to input



# Implicit Setup Tool – Top section options

- The **Apply** button will only become available when all required input has been provided. This can be checked by looking for textboxes with a red background – these have invalid/missing input. When using the **Advanced** mode, sections may need expanding to reveal which input is still required.
- Clicking the **?** buttons will present a hint window describing the expected input for the prompts in the same vicinity as the **?** button.
- The **Advanced** mode will offer more options than the **Simplified** mode for each analysis type. However, some options will be initially selected such that similar input is required for the SAME analysis in the simplified mode.



# Implicit Setup Tool – Simplified Mode

Implicit Analysis Setup

Implicit Setup for model M1

Analysis options: ?

Analysis mode: **Simplified** Advanced

Type of analysis: Transient

LS-DYNA version options: R10.0 MPP **SMP**

Overwriting options: Case-by-Case Apply

Solver options: Parallel multi-frontal sparse, double precision ☒ (default)

Transient Analysis Options

Direct **Modal** Notes ?

Termination time: 0.0 ?

Implicit timestep size: 0.0 ☒ (default)

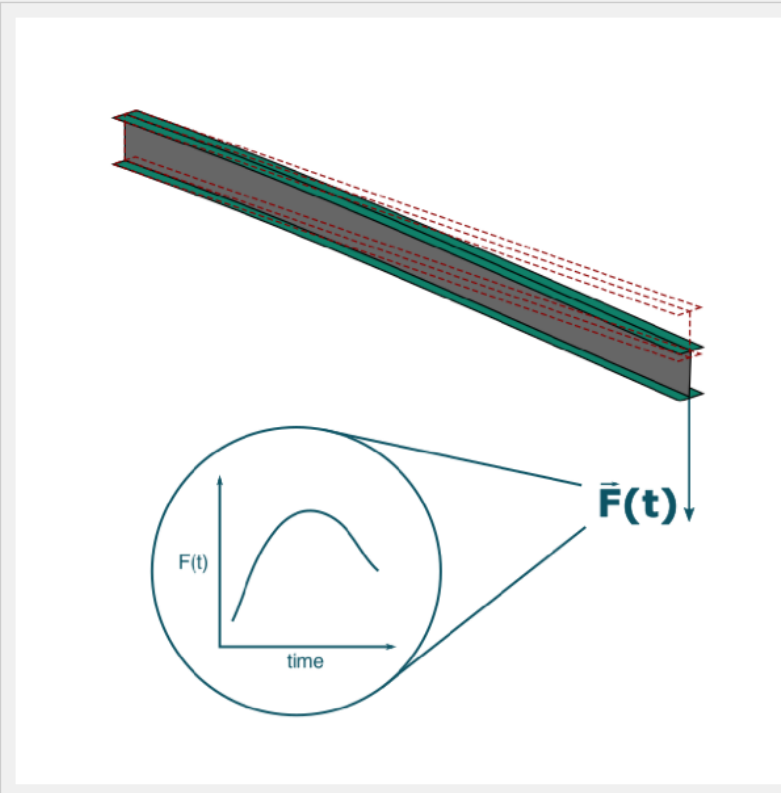
Use eigenmodes from existing file: ☒ ?

Eigenvalue file: d3eigv ☒ (default) ?

Damping coefficient: 0.0 ☒ (default) ?

Currently the only *required* input before **Apply** becomes available.

Clicking on these will present hint windows describing the analysis/input.



The diagram shows a 3D model of a beam structure. A force  $\bar{F}(t)$  is applied at the end of the beam. Below the beam, a circular inset shows a graph of  $F(t)$  versus time, illustrating the time-varying nature of the force.

# Implicit Setup Tool – Advanced Mode

Implicit Analysis Setup

Implicit Setup for model M1

Analysis options: ?

Analysis mode: Simplified Advanced

Type of analysis: Transient

LS-DYNA version options: R10.0 MPP SMP

Overwriting options: Case-by-Case

Solver options: Parallel multi-frontal sparse, double precision (default)

Transient Analysis Options

Direct Modal

Timestep control

Stiffness matrix options

Use eigenfrequencies from file: d3eigv

Eigenvalue Setup

Apply loads to:

Node Set

Entire model

Node Set: 0

Damping options

In this case, *required* input can be found by expanding this section.

3D model of a beam with a force vector  $\bar{F}(t)$  and a graph of  $F(t)$  vs time.

# Writing out options



# Implicit Setup Tool – Output

Once all required options have been set (these will be highlighted in red when current input is invalid/required) the **Apply** button will become available. Clicking this button will present a window reviewing the affected keywords.

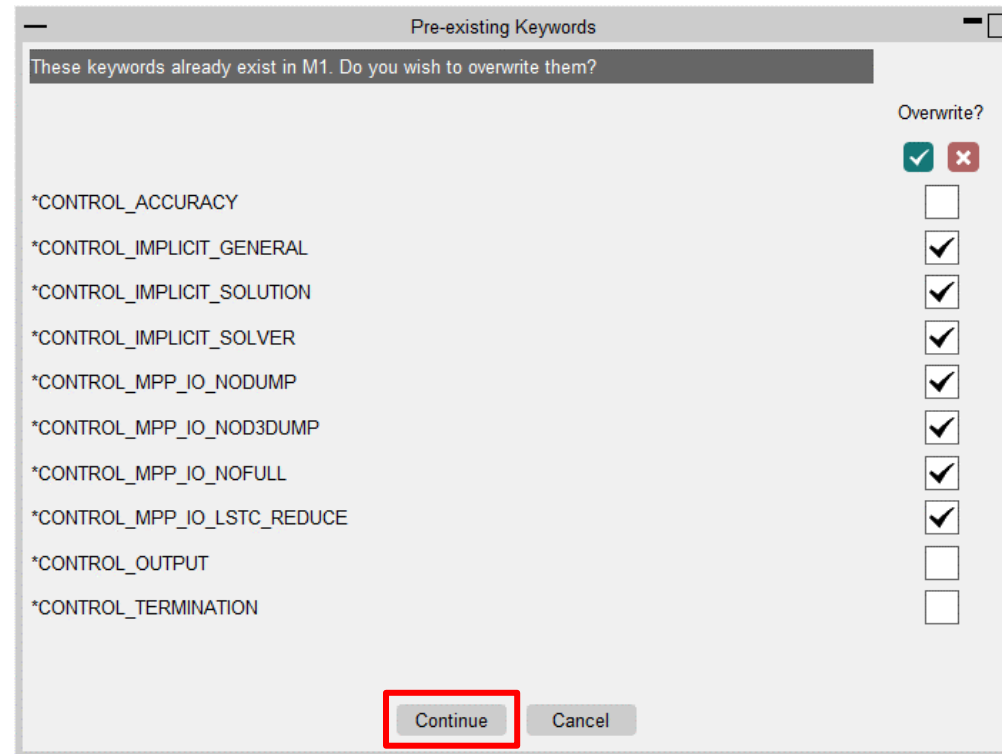
The screenshot shows the 'Implicit Analysis Setup' dialog box. The title bar reads 'Implicit Analysis Setup'. The main area is divided into several sections:

- Implicit Setup for model M1**: A text field at the top left.
- Analysis options:** A section with a help icon (?). It includes:
  - Analysis mode:** Two buttons, 'Simplified' (selected) and 'Advanced'.
  - Type of analysis:** A dropdown menu set to 'Static'.
- LS-DYNA version options:** A section with a help icon (?). It includes:
  - A dropdown menu set to 'R10.0'.
  - Two buttons, 'MPP' (selected) and 'SMP'.
- Overwriting options:** A section with a help icon (?). It includes:
  - A dropdown menu set to 'Case-by-Case'.
- Solver options:** A section with a help icon (?). It includes:
  - A dropdown menu set to 'Parallel multi-frontal sparse, double precision'.
  - A checked checkbox and the text '(default)'.
- Static Analysis Options**: A section with a title bar. It includes:
  - Two buttons, 'Linear' (selected) and 'Non-linear'.
  - A 'Notes' button.
  - Termination time:** A text field set to '1.0' with a help icon (?).
  - Implicit timestep size:** A text field set to '1.0' with a checked checkbox and the text '(default)'.

The 'Apply' button, located in the top right corner of the dialog, is highlighted with a red rectangle.

# Implicit Setup Tool – Output

If the **Overwriting** option was set to **Case-by-Case**, any clashing keywords can be completely overwritten (corresponding box ticked) or ignored after clicking **Apply**. Clicking **Continue** will proceed to the final summary window.





# Implicit Setup Tool – Output

Finally, the **Output Overview** window will summarise which keywords will be written to the model, and which will be ignored (these keywords already exist in the model). Clicking **Confirm** will apply the settings and write the keywords to the model.



# Example: Frequency Domain FRF



## Example: Frequency Domain FRF

This Example will cover setting up the **\*CONTROL** cards for a Frequency Domain Frequency Response Function (**FRF**) analysis. The analysis will be set to:

- Apply an excitation **Force** to **Node Set 100** in the -Z direction.
- The analysis will adopt modes of frequencies up to **2000.0Hz**.
- A **Constant Damping** coefficient of **0.01** will be applied across all modes.
- The frequency response will be taken from **Node Set 500**, in the form of **Acceleration**, in the translational Z direction.
- The response for **400 frequencies** in the range of **1.0 – 400.0Hz** will be output.

**NOTE:** This exact setup can only be achieved in the Advanced mode, as the Simplified mode will use all modes found in the initial eigenvalue analysis, not just those < 2000.0Hz, and will apply the input excitation in the +Z direction.



# FRF Example: Simplified mode



# Example: Frequency Domain FRF - Simplified

100 modes have been requested for the initial eigenvalue analysis to ensure frequencies of up to 2000.0Hz are covered. Now click **Apply**.

Implicit Analysis Setup

Implicit Setup for model M1

Analysis options: ?

Analysis mode: **Simplified** Advanced

Type of analysis: Frequency Domain

LS-DYNA version options: ? R9.0 MPP SMP

Overwriting options: ? Case-by-Case

Solver options: ? Parallel multi-frontal sparse, double precision (default)

Frequency Domain - Frequency Response Function (FRF) Analysis Options

Notes

Number of eigenvalues: 100 ?

Input excitation type: Nodal force ?

Excitation direction: Z-direction ?

Apply input to: Node Set 100 ?

Damping coefficient: 0.01 (default) ?

Output response type: Acceleration ?

Response direction: Trans' Z-direction ?

Get response from: Node Set 500 ?

No. output frequencies: 400 ?

Frequency range: 1.0 to: 400.0

Input Excitation

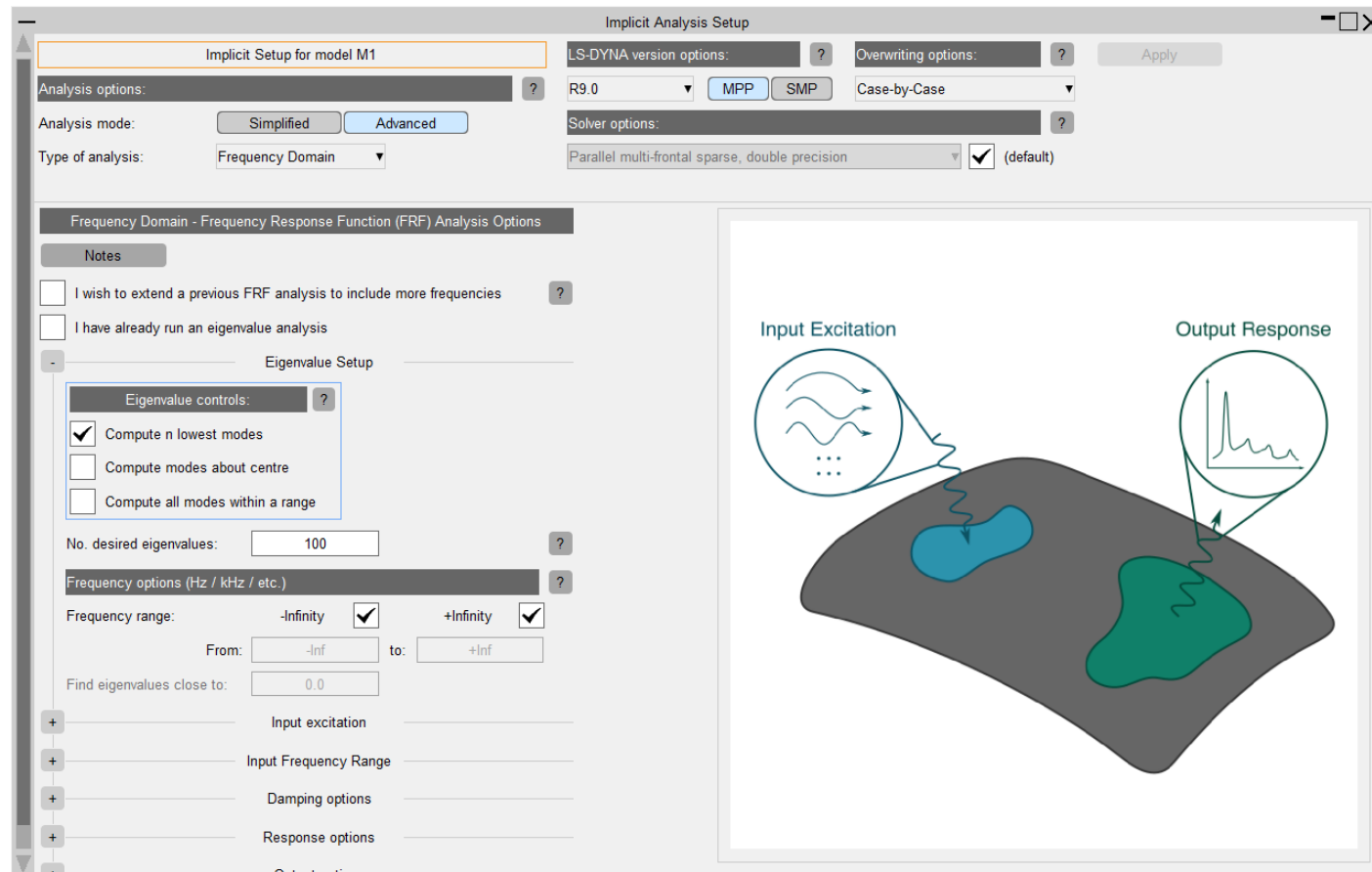
Output Response

# FRF Example: Advanced mode



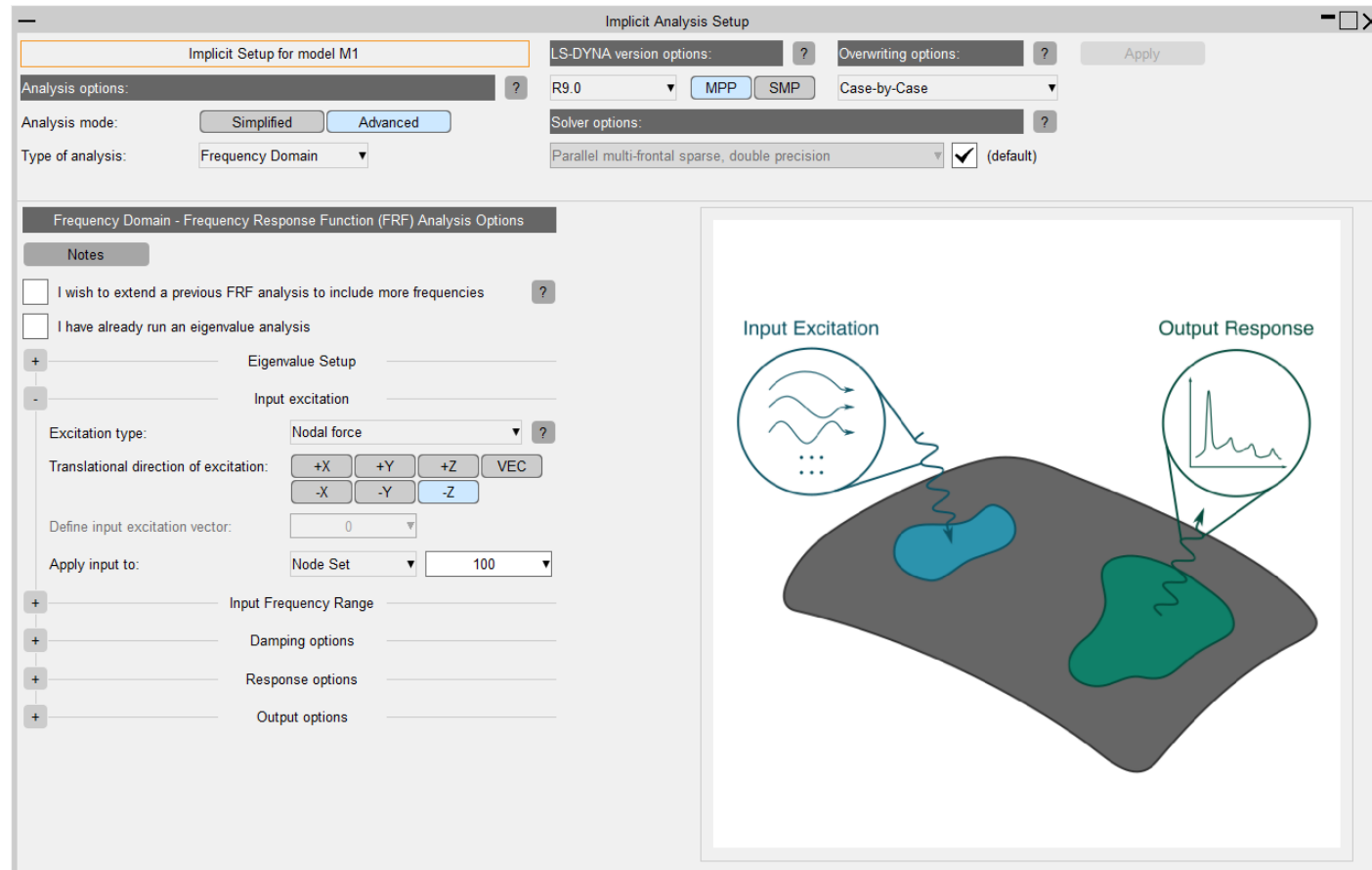
# Example: Frequency Domain FRF - Advanced

Begin by setting the number of eigenvalues to find in the initial eigenvalue analysis. 100 has been chosen here to ensure enough frequencies are covered (up to 2000.0Hz).



# Example: Frequency Domain FRF - Advanced

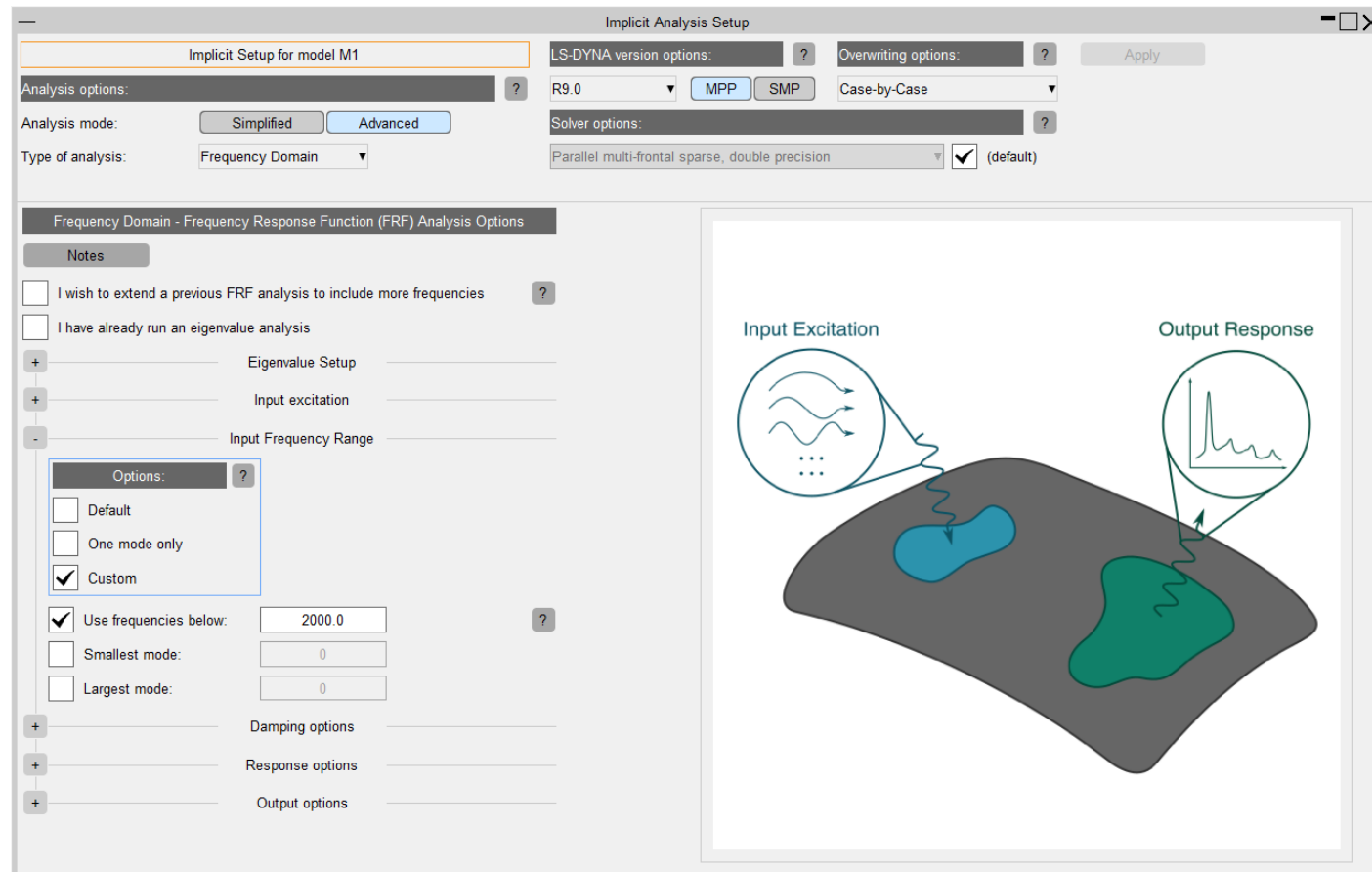
Input excitation in the form of **Nodal Force**, in the -Z direction, applied to Node Set 100.





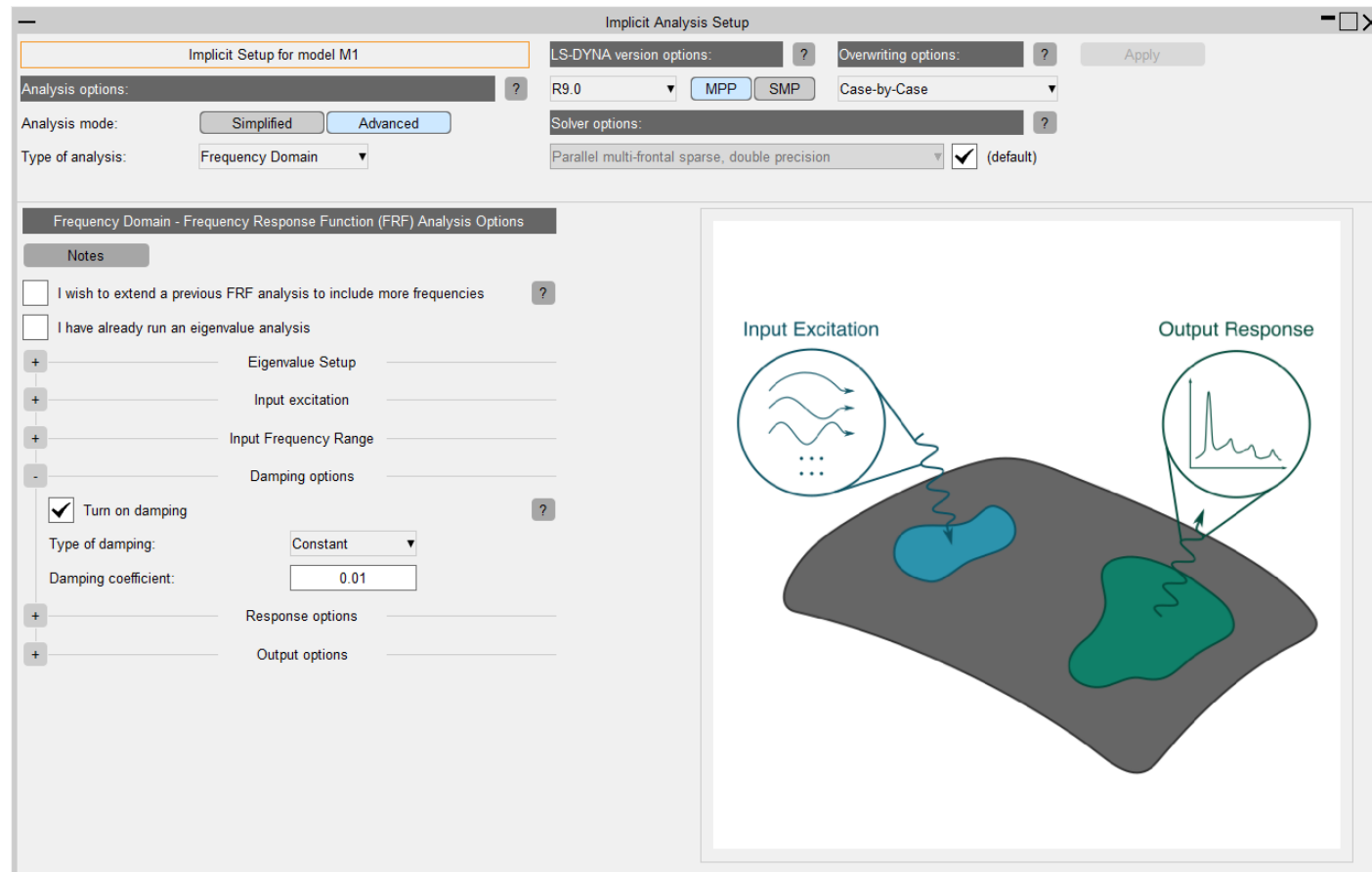
# Example: Frequency Domain FRF

Adopt modes of frequencies less than 2000.0Hz.



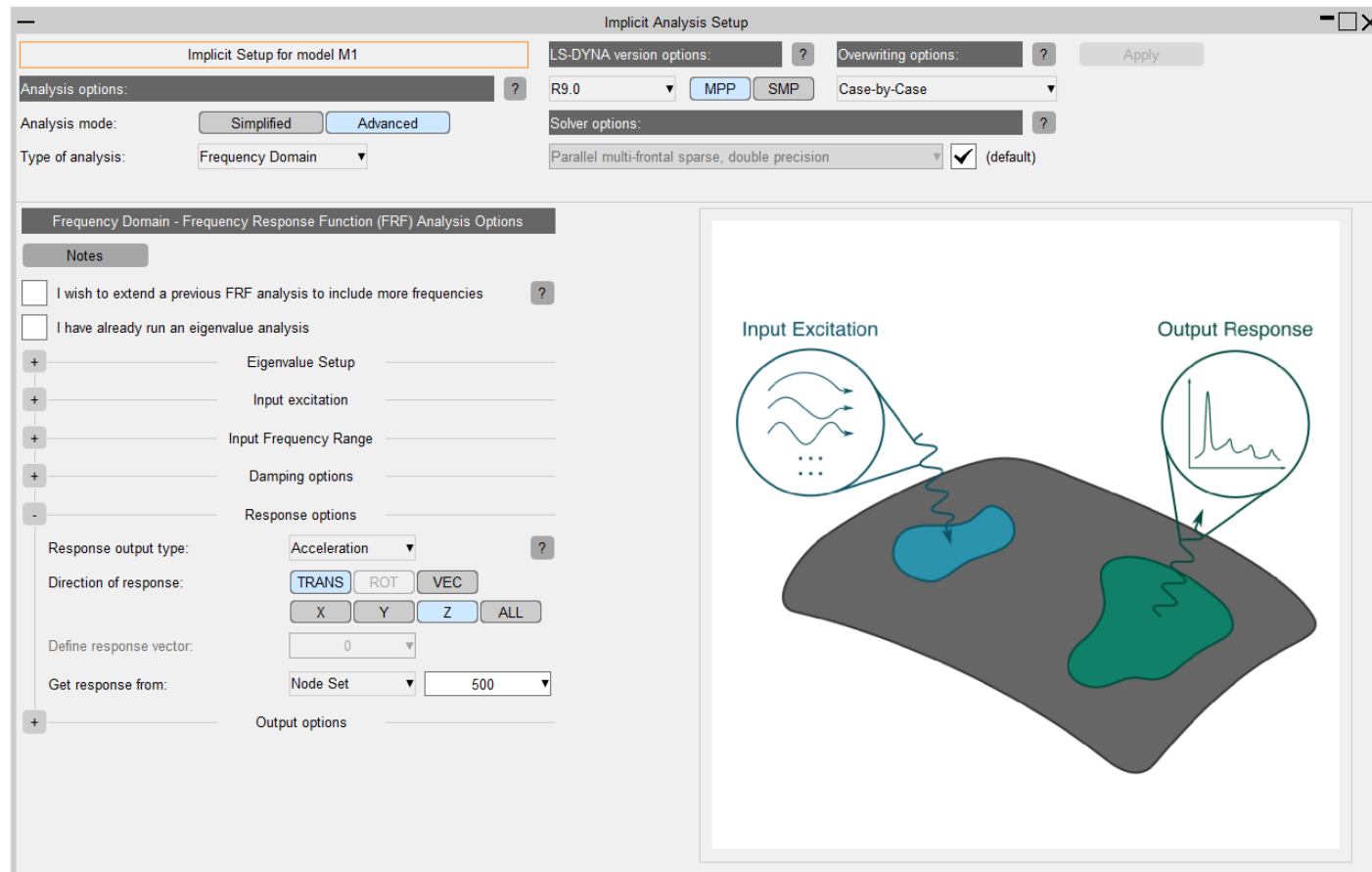
# Example: Frequency Domain FRF - Advanced

Apply a **Constant Damping** coefficient of 0.01 across all modes.



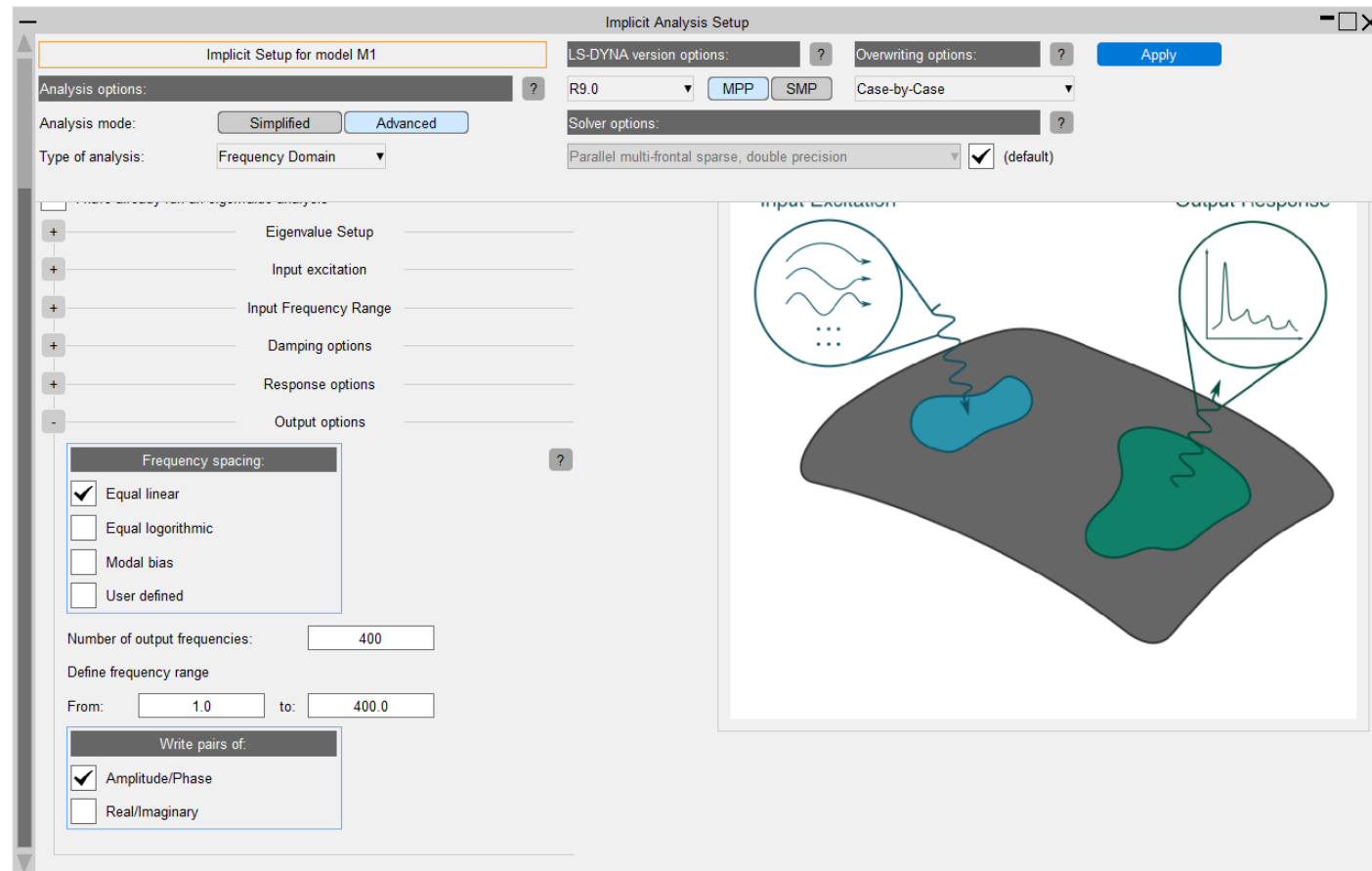
# Example: Frequency Domain FRF - Advanced

Take the output **Response** from Node Set 500, in the form of **Acceleration** in the translational Z direction.



# Example: Frequency Domain FRF - Advanced

**Output** the response for 400 frequencies in the range of 1.0 – 400.0Hz. Now click **Apply**.



# Contact us

## Global / UK

T: +44 121 213 3399

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

## India

T: +91 40 69019723 / 98

E: [india.support@arup.com](mailto:india.support@arup.com)

## China

T: +86 21 3118 8875

E: [china.support@arup.com](mailto:china.support@arup.com)

## USA

T: +1 415 940 0959

E: [us.support@arup.com](mailto:us.support@arup.com)

Subscribe to  
our newsletter:



Follow us on:



@Oasys LS-DYNA  
Environment



@Oasys LS-DYNA  
Environment



@Oasys



@Oasys

<https://www.oasys-software.com/dyna/>