Oasys

Oasys GSA

LS-DYNA Reference

Oasys
YOUR IDEAS BROUGHT TO LIFE

13 Fitzroy Street
London
W1T 4BQ
Telephone: +44 (0) 20 7755 3302
Facsimile: +44 (0) 20 7755 3720

Central Square
Forth Street
Newcastle Upon Tyne
NE1 3PL
Telephone: +44 (0) 191 238 7559
Facsimile: +44 (0) 191 238 7555

e-mail: oasys@arup.com
Website: oasys-software.com

Oasys GSA

Contents

GSA to LS-DYNA

While GSA provides a powerful tool for structural modelling there are times when more powerful analysis is required. In such cases it can be useful to run the analysis on LS-DYNA. LS-DYNA is widely used for problem in mechanical engineering but has powerful features that are suitable for advanced structural analysis.

In general creating a new LS-DYNA model from the GSA model gives scope for data to be missed and for the GSA and LS-DYNA models to get out of sync. The idea behind the GSA to LS-DYNA link is to allow the engineer to carry out his modelling in GSA and keep the reference model in GSA and to create a lightweight LS-DYNA model (keyword file) for analysis. If this model is 'ready to run' in LS-DYNA then when changes are made to the GSA model it is straightforward to discard the LS-DYNA model and simply create a new model.

LS-DYNA model in GSA

The GSA model is seen as the master model however this can lead to problems where an LS-DYNA analysis required features or materials that are not used I a GSA analysis. A number of feature in GSA allow the engineer to define this extra data. The three key features are:

- LS-DYNA analysis task
- LS-DYNA materials
- Composite 2D elements

GSA to LS-DYNA workflow

The way the link is intended to work is to:

- set up a model in GSA
- add extra modules such as DYNA materials
- create an analysis stage to map the GSA sections and materials to DYNA sections and material
- create an LS-DYNA analysis task specifying the relevant DYNA control parameters
- create a 'ready to run' DYNA keyword file.

Data transferred to LS-DYNA

The link should be able to transfer:

- control information
- nodes
- elements – beams, shells, 3D, springs, dampers, masses

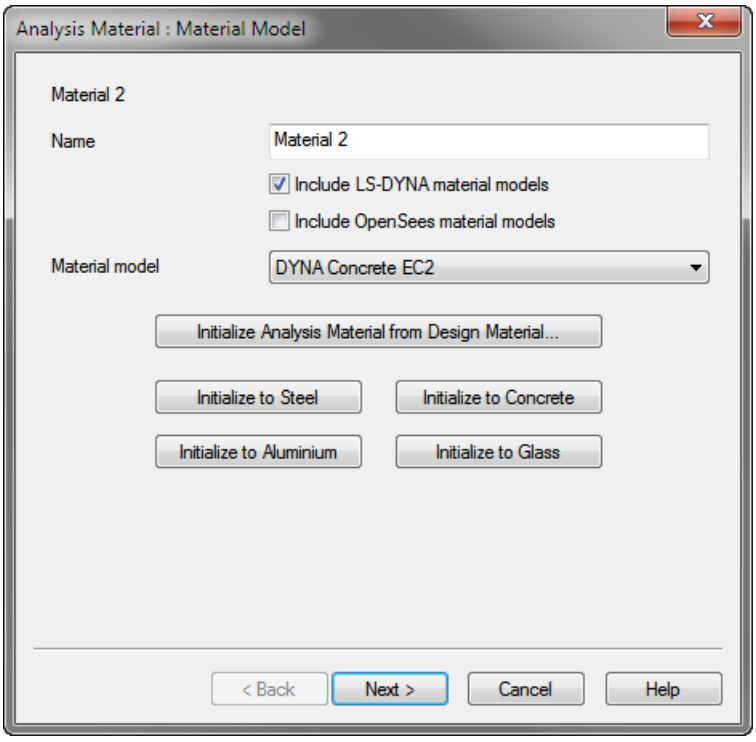
- axis sets
- properties - sections, 2D properties (including composite), 3D properties, springs, dampers
- rigid constraints (diaphragms)
- joints
- loads – nodes, beams, shells
- load curves
- sets – nodes, beams, shells

Creating an LS-DYNA model

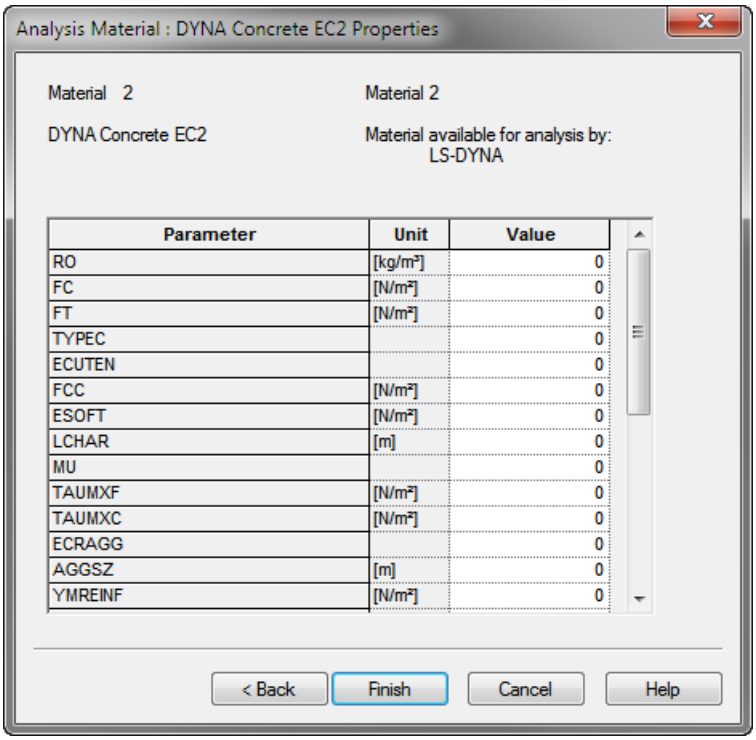
There are a number of features that make it easier to define the model for an LS-DYNA analysis.

Materials

In order to create LS-DYNA specific materials use the analysis material wizard and select the include LS-DYNA material models option

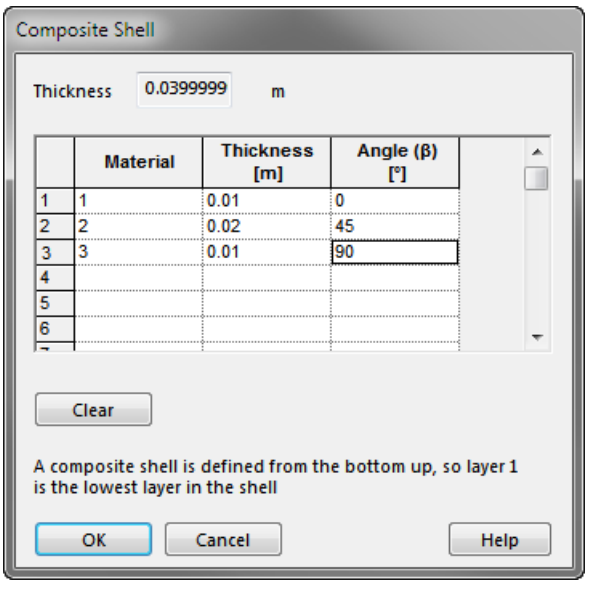


This leads to a page where the data to define the model can be entered. All parameters are defined as in the LS-DYNA Keyword Manual.



Composite Shells

To define a composite shell use the 2D property wizard. And select the LS-DYNA Composite button to define the lay-up of the composite



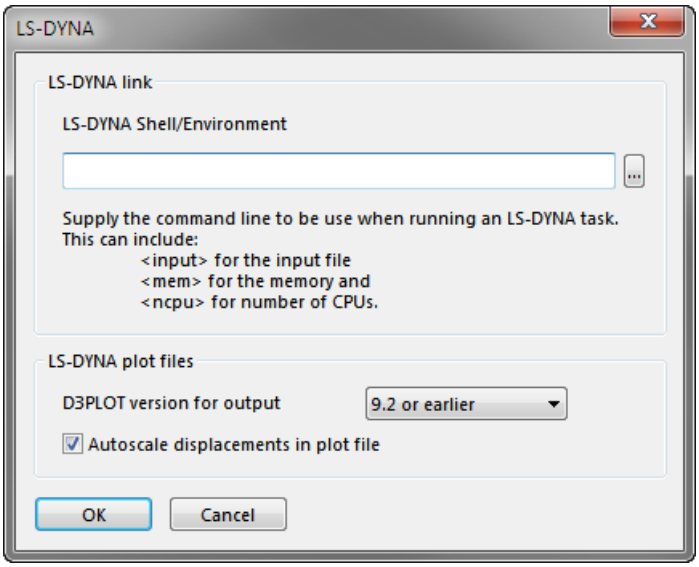
he way we have implemented the link is to let the engineer set up a DYNA analysis task and the "run" option will create the keyword file.

Analysis stages

The most convenient way to map between the GSA and LS-DYNA models is to used analysis stages and in-particular analysis stage properties or analysis stage materials. This allows the LS-DYNA materials to replace the normal GSA materials for an LS-DYNA specific stage.

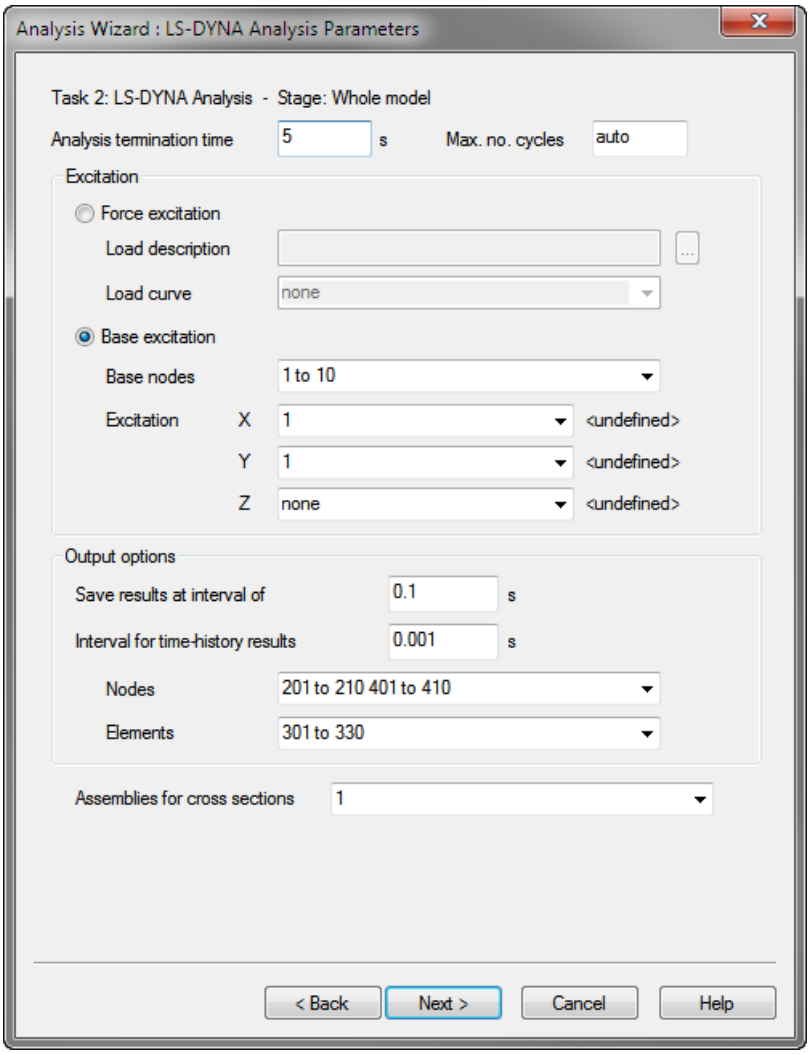
LS-DYNA preference

The LS-DYNA preferences available from Preferences | Miscellaneous allow a default LS-DYNA environment to be specified. This can give the command line for running a LS-DYNA preprocessor such as LS-PrePost or Oasys PRIMER.



LS-DYNA analysis task

The final stage is to create an LS-DYNA analysis task. From the analysis wizard select the LS-DYNA task option and the appropriate analysis stage. This allows the definition of the LS-DYNA analysis, selecting the excitation type and output options for both plot and time history files.



The subsequent page defines how the mass is to be assigned to the model. In GSA loads can be considered as additional mass. If this is required for the LS-DYNA analysis the export of the keyword file will include nodal masses corresponding to those that GSA would generate.

Damping can include either damper elements, Rayleigh damping or both.

The final page prompt for the file name for the keyword file and the command line to run on finish. If no command line is supplied the keyword will be still be written but GSA will not try to do anything with the file.

Supported keywords

General

*TITLE

*CONTROL_TERMINATION

*DATABASE_BINARY_D3PLOT

*DATABASE_BINARY_D3THDT

*DATABASE_BINARY_INTFOR

*DATABASE_BINARY_D3DUMP

*DATABASE_BINARY_RUNRSF

*DATABASE_HISTORY_NODE

*DATABASE_HISTORY_BEAM

*DATABASE_HISTORY_DISCRETE

*DATABASE_HISTORY_SHELL

*DATABASE_HISTORY_SOLID

*DAMPING_GLOBAL

Model

*DEFINE_COORDINATE_SYSTEM

*NODE

*DEFINE_COORDINATE_NODES

*BOUNDARY_SPC_NODE

*ELEMENT_BEAM

*ELEMENT_BEAM_OFFSET

*ELEMENT_DISCRETE

*ELEMENT_MASS

*ELEMENT_INERTIA

*ELEMENT_SHELL

*ELEMENT_SOLID

Materials

*MAT_ELASTIC

*MAT_PLASTIC_KINEMATIC

*MAT_ORTHOTROPIC_ELASTIC

*MAT_FABRIC

*MAT_CONCRETE_EC2

*MAT_STEEL_EC3

*MAT_HYSTERETIC_BEAM

*MAT_HYSTERETIC_REINF

*MAT_PLASTIC_KINEMATIC

*MAT_PARK_ANG_BEAM

*MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM

*MAT_SEISMIC_ISOLATOR

*MAT_RIGID

*MAT_LINEAR_ELASTIC_DISCRETE_BEAM

*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM

*MAT_SPRING_ELASTIC

*MAT_SPRING_INELASTIC

*MAT_DAMPER_VISCOUS

*MAT_CABLE_DISCRETE_BEAM

Properties

*PART_COMPOSITE

*SECTION_BEAM

*SECTION_BEAM_TITLE

*SECTION_DISCRETE

*SECTION_SHELL

*SECTION_SOLID_TITLE

*INTEGRATION_BEAM

*DAMPING_PART_STIFFNESS

Constraints

*CONSTRAINED_RIGID_BODIES

*CONSTRAINED_LINEAR_LOCAL

*CONSTRAINED_LINEAR_LOCAL

*CONSTRAINED_NODAL_RIGID_BODY

*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_ID

Loading

*BOUNDARY_PRESCRIBED_MOTION_NODE

*LOAD_NODE_POINT

*LOAD_BEAM_ELEMENT

*LOAD_SHELL_ELEMENT

*LOAD_BODY_X

*LOAD_BODY_Y

*LOAD_BODY_Z

Miscellaneous

*DATABASE_CROSS_SECTION_PLANE_ID

*DEFINE_CURVE

*PART

*SET_PART_LIST

*SET_NODE

*SET_NODE_LIST

*SET_BEAM

*SET_SHELL

*SET_SHELL_LIST

*END