

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

© Oasys 1985 – 2017

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

Contents

GSA to LS-DYNA	5
LS-DYNA model in GSA	5
GSA to LS-DYNA workflow	5
Data transferred to LS-DYNA	5
Creating an LS-DYNA model	6
Materials	6
Composite Shells	7
Analysis stages	8
LS-DYNA preference	8
LS-DYNA analysis task	8
Supported keywords	9
General	9
Model	10
Materials	10
Properties	11
Constraints	11
Loading	11
Miscellaneous	12

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 in a GSA analysis. A number of features 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

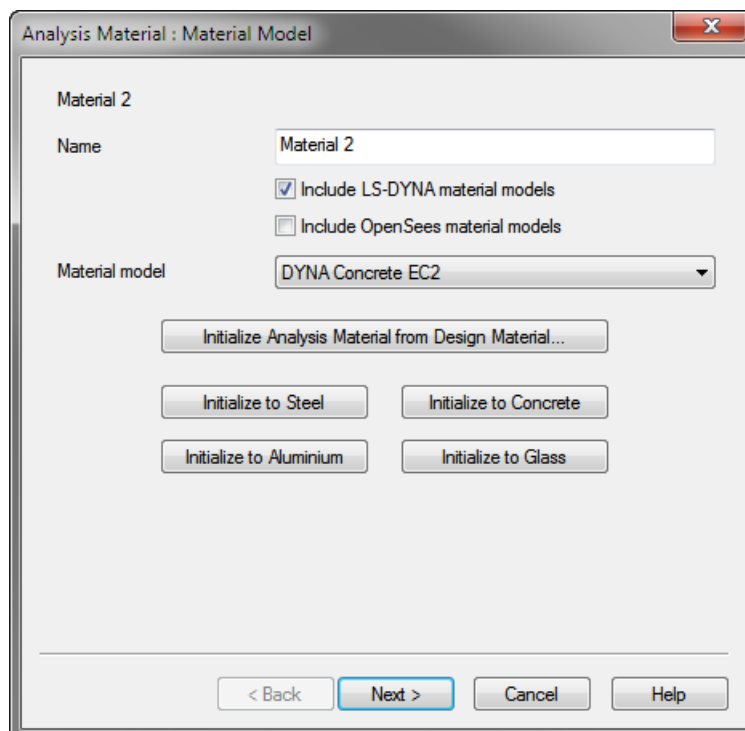
- axis sets
- properties - sections, 2D properties (including composite), 3D properties, springs, dampers
- rigid constraints (diaphragms)
- joints
- loads – nodes, beams, shells
- load curves
- sets – nodes, beams, shells

Creating an LS-DYNA model

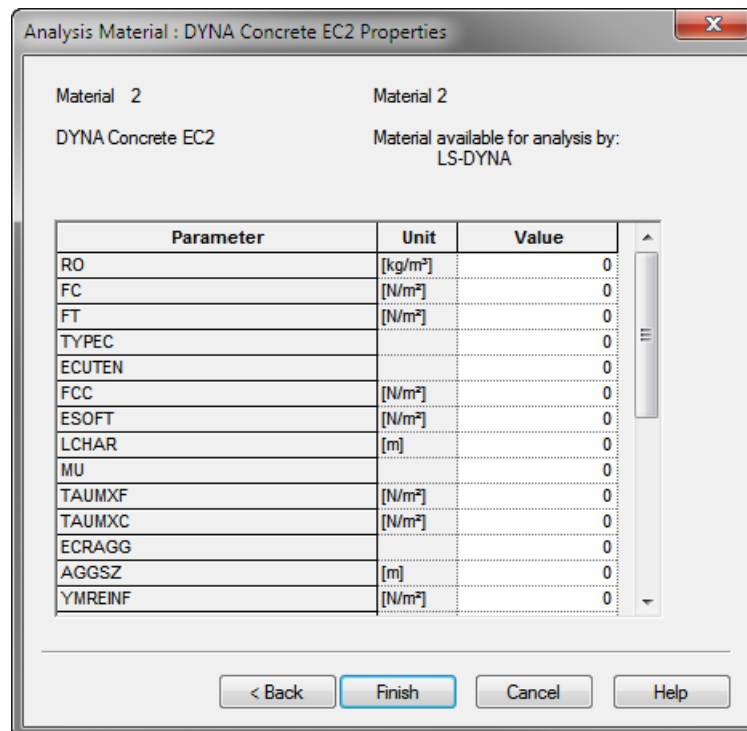
There are a number of features that make it easier to define the model for an LS-DYNA analysis.

Materials

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

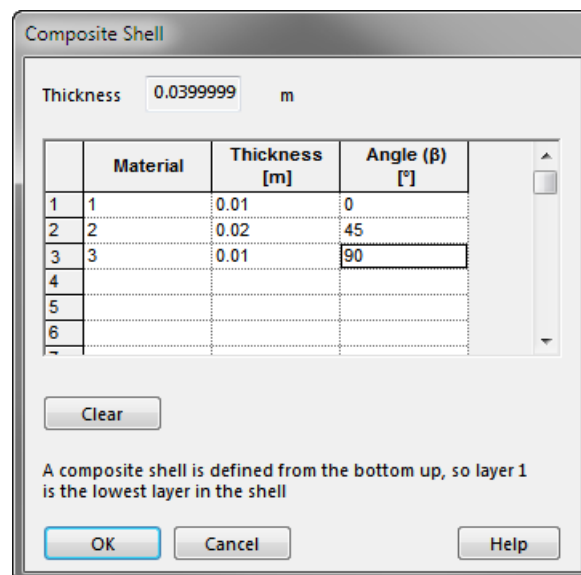


This leads to a page where the data to define the model can be entered. All parameters are defined as in the LS-DYNA Keyword Manual.



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



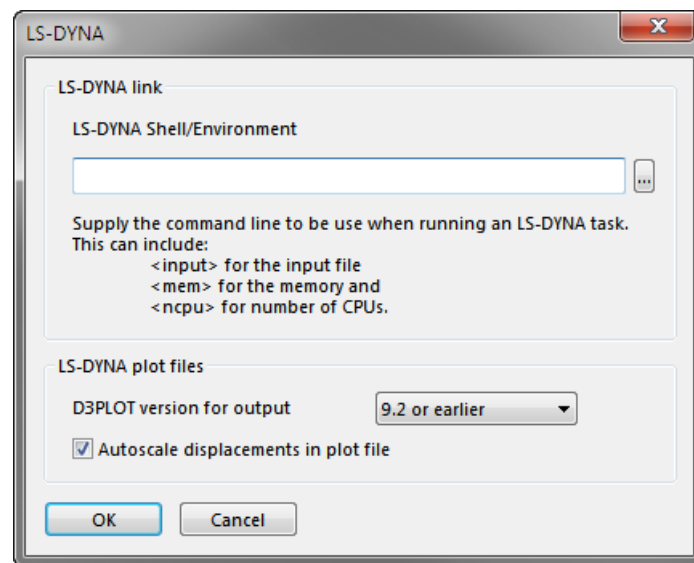
the way we have implemented the link is to let the engineer set up a DYNA analysis task and the “run” option will create the keyword file.

Analysis stages

The most convenient way to map between the GSA and LS-DYNA models is to use 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.

Analysis Wizard : LS-DYNA Analysis Parameters

Task 2: LS-DYNA Analysis - Stage: Whole model

Analysis termination time s Max. no. cycles

Excitation

Force excitation

Load description

Load curve

Base excitation

Base nodes

Excitation	X	<input type="text" value="1"/>	<undefined>
	Y	<input type="text" value="1"/>	<undefined>
	Z	<input type="text" value="none"/>	<undefined>

Output options

Save results at interval of s

Interval for time-history results s

Nodes

Elements

Assemblies for cross sections

< Back Next > Cancel Help

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 prompts 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 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_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