Using LS-DYNA® from ANSYS Workbench Environment

Dr.-Ing. Matthias Hörmann
CADFEM GmbH, Grafing b. München, Germany
mhoermann@cadfem.de

Abstract

Numerical simulations as integral part in the virtual product development process exhibit a huge spectrum. Ranging from simple modal analyses over linear and nonlinear stiffness and strength based problems up to coupled multi-physic analyses, where different physical disciplines interact with each other. Thereby simulation tools in combination with preprocessors must enable users to perform product development tasks faster and therefore more efficient. One essential part is hereby the seamless model file transfer from and to 3D CAD systems. Additionally model and assembly handling in-between the different simulation disciplines in combination with an automatic mesh generation and automatic contact detection is also important to speed up development time. With the Workbench environment, ANSYS took a quantum leap into model analysis and handling different simulation disciplines in one standard user interface in combination with a tight interface from and to almost all common 3D CAD systems. An interface between ANSYS Workbench and LS-DYNA therefore provides the opportunity to use Workbench preprocessing functionalities for LS-DYNA simulations. The German ANSYS and LS-DYNA distributor CADFEM has thus created a unidirectional, interactive graphic interface "Workbench LS-DYNA" for the transfer of data from ANSYS Workbench to LS-DYNA. This not only enables users to transfer the pure structure in form of nodes and elements, but also sections, materials, contact definitions and boundary conditions, including prescribed motions and force loading. LS-DYNA specific control and database options are included from a template file, which can be customized by the user. Moreover any LS-DYNA keyword command can be defined within the Workbench GUI and will be added into the LS-DYNA input file.

Besides using CAD interfaces, automatic mesh generation and contact detection of Workbench, LS-DYNA users will benefit from this interface. With an existing ANSYS Workbench license and LSTC's free of charge LS-PrePost an additional preprocessor causing cost and training effort may no longer be necessary. Even more important, the interface enables easier data exchange with other analysis departments using already ANSYS Workbench. Moreover the CAD interfaces in ANSYS Workbench also allow a closer link to construction departments.

Workflow in Figures

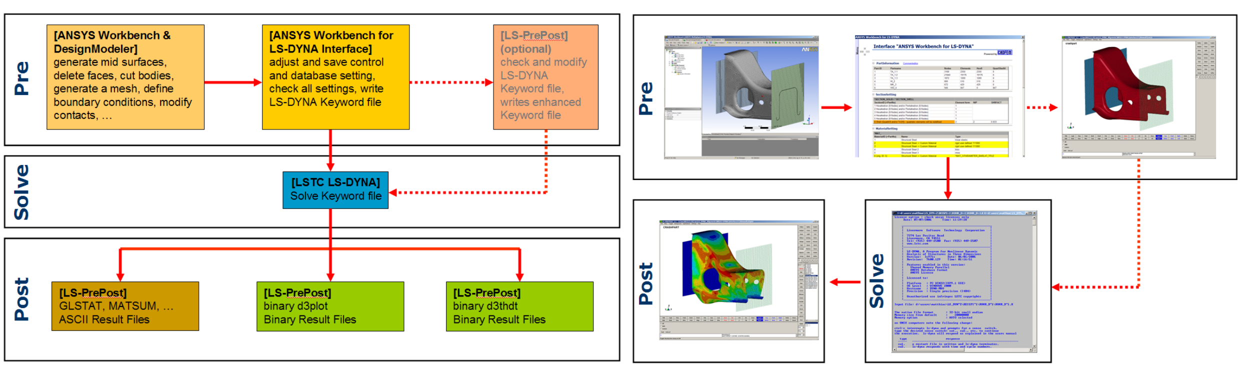


Figure 1: Workflow using ANSYS Workbench, Interface "ANSYS Workbench for LS-DYNA®", LS-DYNA® and LS-PrePost®

Part Definitions

Part-Ids are automatically assigned. Section- and Material-ids are correspondingly used. User has the possibility to define gravity and adaptivity as well as the *PART_INERTIA option of LS-DYNA. Equation of states and hourglass definitions can be defined by the user and will be accounted for.

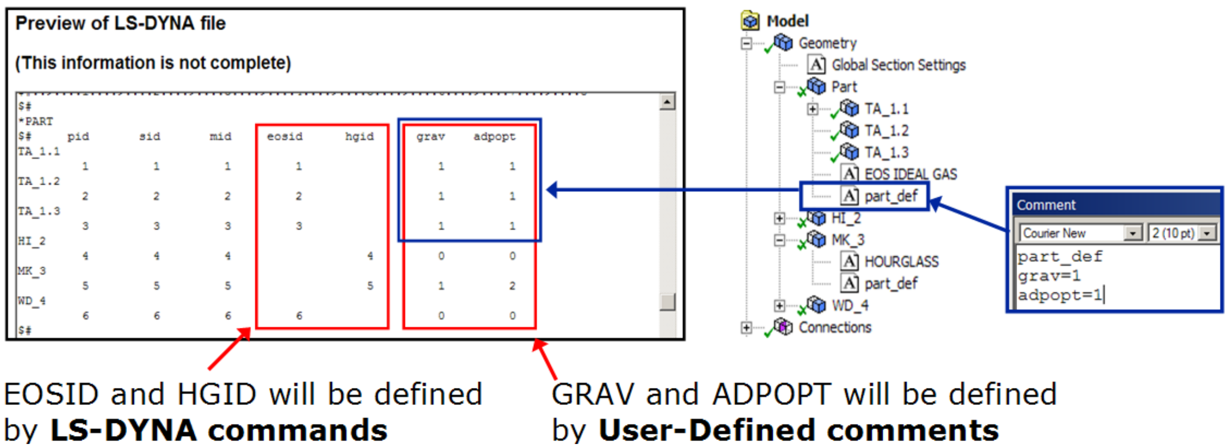


Figure 2: Part definitions

Section Definitions

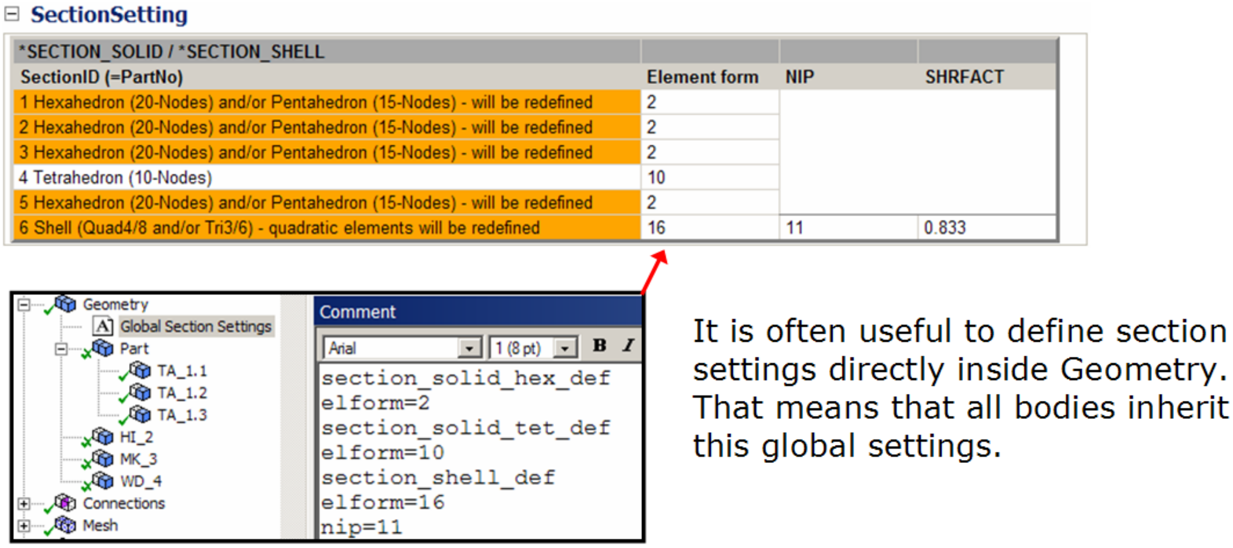
Default element forms are set within the interface. A new default is in terms of tetrahedrons were element form #4 is used. Any element form of LS-DYNA can be accessed via the interface. In case of shell elements one can also defined number of integrations points across thickness and shear correction factor, whereby default is 0.833.

<u>Typically used element forms (ELFORM):</u>

Element	EQ	Description
Hexahedron	1	one point integrated element (**default**)
Hexahedron	2	fully integrated element
Tetrahedron	4	quadratic tetrahedron element with nodal rotations, 5 integration points (**default**)
Tetrahedron	10	linear tetrahedron element w/o nodal rotations
Tetrahedron	16	quadratic tetrahedron element with mid side nodes (10 nodes)
Shell	2	one point integrated shell (**default**)
Shell	16	fully integrated shell

All other element forms available in LS-DYNA can be defined as well

Figure 3: Element form definitions

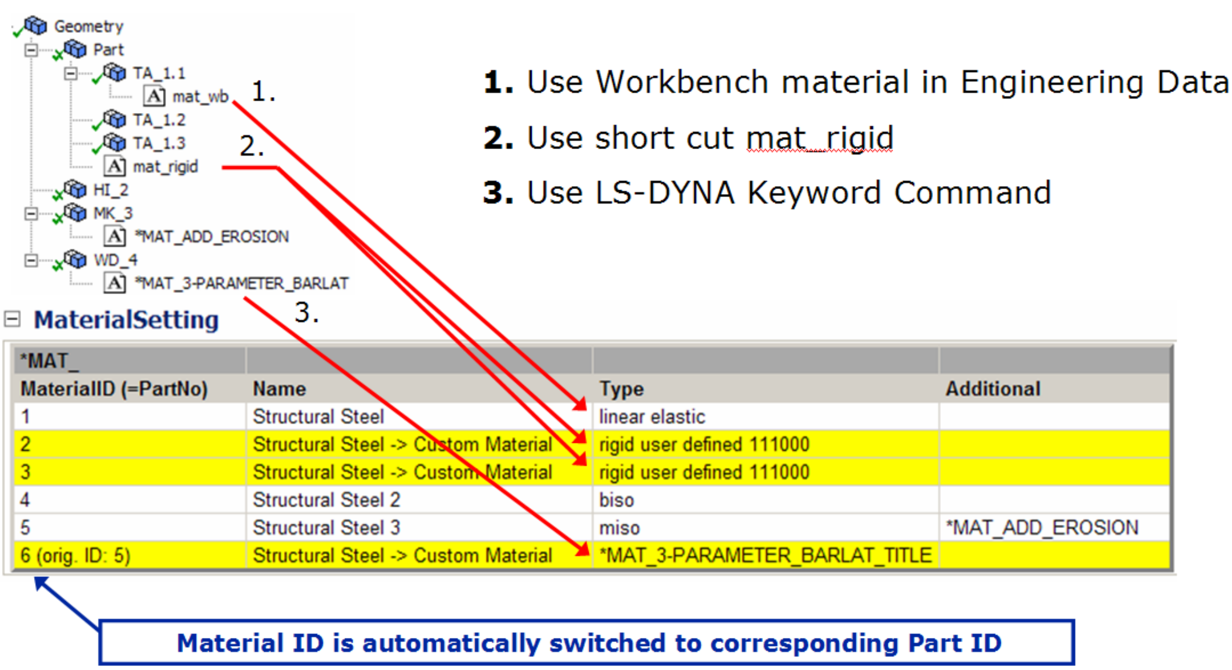


*SECTION_SOLID / *SECTION_SHELL			
SectionID (=PartNo)	Element form	NIP	SHRFACT
1 Hexahedron (20-Nodes) and/or Pentahedron (15-Nodes) - will be redefined	2		
2 Hexahedron (20-Nodes) and/or Pentahedron (15-Nodes) - will be redefined	2		
3 Hexahedron (20-Nodes) and/or Pentahedron (15-Nodes) - will be redefined	2		
4 Tetrahedron (10-Nodes)	10		
5 Hexahedron (20-Nodes) and/or Pentahedron (15-Nodes) - will be redefined	2		
6 Shell (Quad4/8 and/or Tri3/6) - quadratic elements will be redefined	16	11	0.833

Figure 4: Element form definitions and modifications within ANSYS Workbench

Material Definitions

There are three ways to define a material for LS-DYNA within the ANSYS Workbench GUI.
1. Using Workbench material in Engineering Data
2. Use short cut command to switch material to rigid (*MAT_RIGID)
3. Using LS-DYNA keyword command directly in ANSYS Workbench
Consequently all available material models in LS-DYNA can be included into ANSYS Workbench.



*MAT_			
MaterialID (=PartNo)	Name	Type	Additional
1	Structural Steel	linear elastic	
2	Structural Steel -> Custom Material	rigid user defined 111000	
3	Structural Steel -> Custom Material	rigid user defined 111000	
4	Structural Steel 2	biso	
5	Structural Steel 3	miso	*MAT_ADD_EROSION
6 (orig. ID: 5)	Structural Steel -> Custom Material	*MAT_3-PARAMETER_BARLAT_TITLE	

Figure 4: Material definitions

For the rigid material definition it is possible to define the support condition in local or global coordinate system. Additionally the interface checks for rigid bodies which are connected. Consequently a *CONSTRAINED_RIGID_BODIES definition will be automatically written.

The following material models are directly supported from ANSYS Workbench GUI:

Isotropic linear elastic	→ *MAT_ELASTIC (#1)
Orthotropic linear elastic	→ *MAT_ORTHOTROPIC_ELASTIC (#2)
BISO	→ *MAT_PIECEWISE_LINEAR_PLASTICITY (#24)
MISO	→ *MAT_PIECEWISE_LINEAR_PLASTICITY (#24)
BKIN	→ *MAT_PLASTIC_KINEMATIC (#3)

For the rubber material models one can use the curve fitting method in ANSYS Workbench, which takes into account not only uniaxial test data but also biaxial, pure shear, planar shear and compression experimental test.
The fitted rubber constants are then used in LS-DYNA material models *MAT_HYPERELASTIC_RUBBER and *MAT_OGDEN_RUBBER.

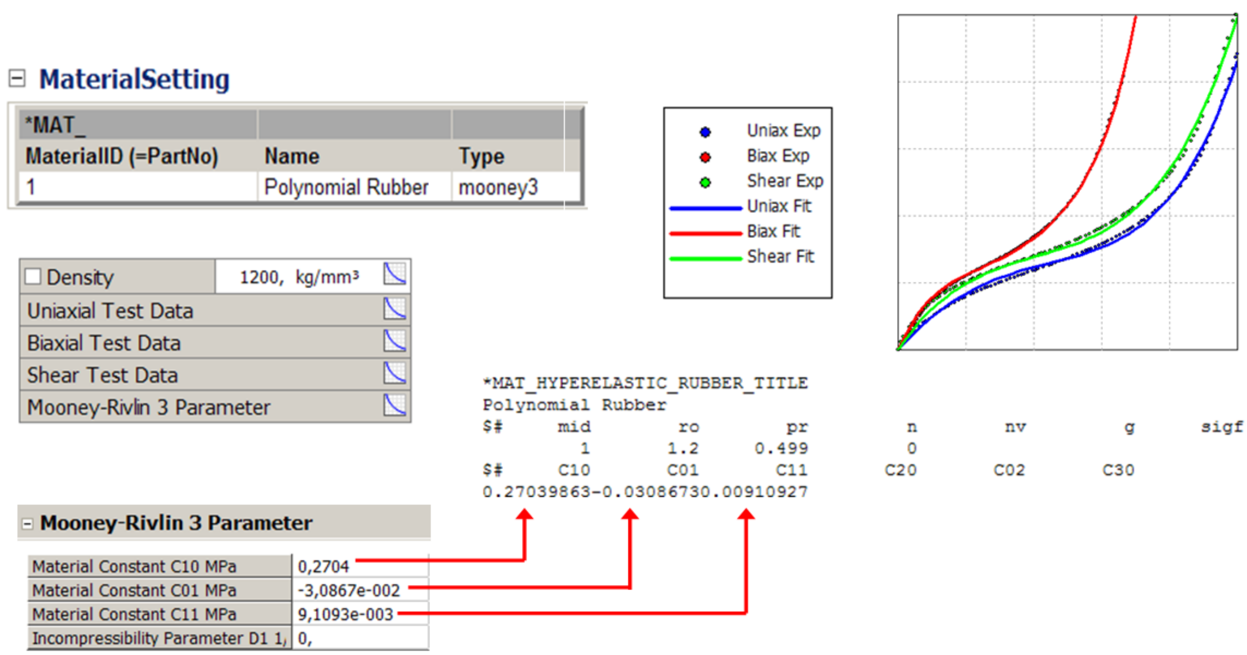


Figure 5: Rubber material data fit and export to LS-DYNA keyword command

*MAT_ADD_EROSION is also possible to be defined.

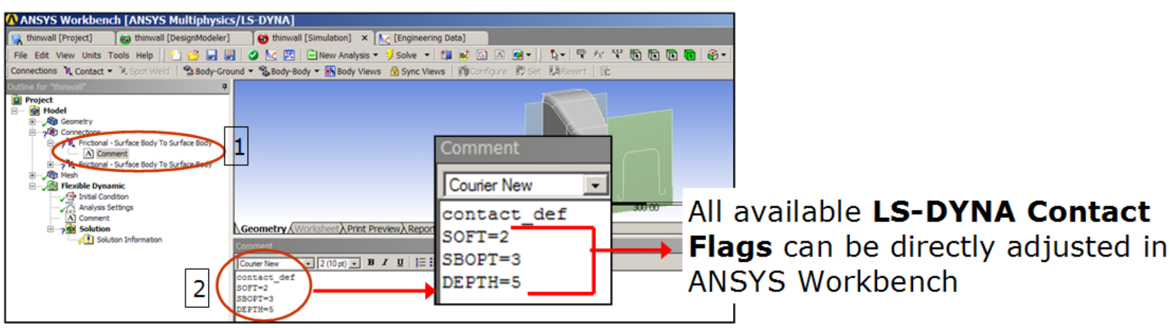
Contact Definitions

The following Fig. 5 shows the available contact definitions, which are accessible through the interface.

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE	*CONTACT_TIED_NODES_TO_SURFACE_OFFSET
*CONTACT_SURFACE_TO_SURFACE	*CONTACT_TIED_SURFACE_TO_SURFACE_OFFSET
*CONTACT_FORMING_SURFACE_TO_SURFACE	
*CONTACT_ERODING_SURFACE_TO_SURFACE	
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE	
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE	
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE	
*CONTACT_AUTOMATIC_SINGLE_SURFACE	
*CONTACT_SINGLE_SURFACE	
*CONTACT_ERODING_SINGLE_SURFACE	
*CONTACT_NODES_TO_SURFACE	
*CONTACT_AUTOMATIC_NODES_TO_SURFACE	
*CONTACT_ERODING_NODES_TO_SURFACE	
*CONTACT_FORMING_NODES_TO_SURFACE	

Figure 6: Available contact definitions via the interface

The following contact flags can be defined and accessed within ANSYS Workbench GUI.



1	SSID	MSID	SSTYP	MSTYP	SBOX	MBOX	SPR	MPR
2	FS	FD	DC	VC	VDC	PENCHK	BT	DT
3	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
A	SOFT	SOFSCL	LCIDAB	MAXPAR	SBOPT	DEPTH	BSORT	FRCFRQ
B	PENMAX	THKOPT	SHLTHK	SNLOG	ISYM	I2D3D	SLDTHK	SLDSTF
C	IGAP	IGNORE	DPRFAC	DTSTIF			FLANGL	

Figure 7: Available contact flags

Boundary Definitions

Beside standard fixed boundary conditions also prescribed motions can be applied and used via ANSYS Workbench.

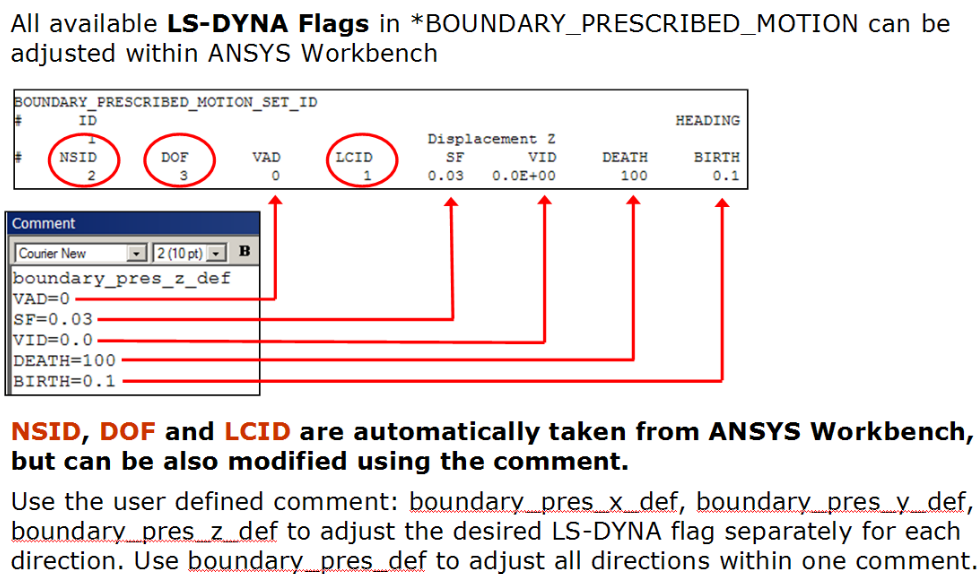


Figure 8: Prescribed Motions
A corresponding listing in the Interface GUI looks like this

BoundarySetting

*BOUNDARY_SPC

ID	Name	DOFX	DOFY	DOFZ	DOFRX	DOFRY	DOFRZ
1	fix_free_free	SPC applied to rigid (PID 8); SPC defined on MAT_RIGID definition -> SPC not written.					
2	fix	1	1	1	1	1	1
3	fix_tabular_tabul	1	0	0	0	0	0
4	free_free_functio	0	0	1	0	0	0
5	fix_free_tabular	1	0	0	0	0	0
6	Displacement	SPC applied to rigid (PID 16); SPC defined on MAT_RIGID definition -> SPC not written.					

*BOUNDARY_PRESCRIBED_MOTION_SET

NSID	Name	Direction	DOF	VAD
12	free_free_tabular	Z	4	0
12	fix_tabular_tabular	Y	2	1
223	fix_tabular_tabular	Z	3	0
Sytem defined	fix_free_tabular	Z	3	2

*BOUNDARY_PRESCRIBED_MOTION_RIGID

PID	Name	Direction	DOF	VAD
16	Displacement 2	X	1	2
16	Displacement 2	Y	2	2
16	Displacement 2	Z	3	2

where checking is done, whether boundary conditions are applied to rigid bodies. A distinction is made for the prescribed motions whether body is rigid or not.

Load and Initial Definitions

Loads, gravity and initial velocities can be defined using the ANSYS Workbench GUI. Corresponding LS-DYNA keyword commands are the written.

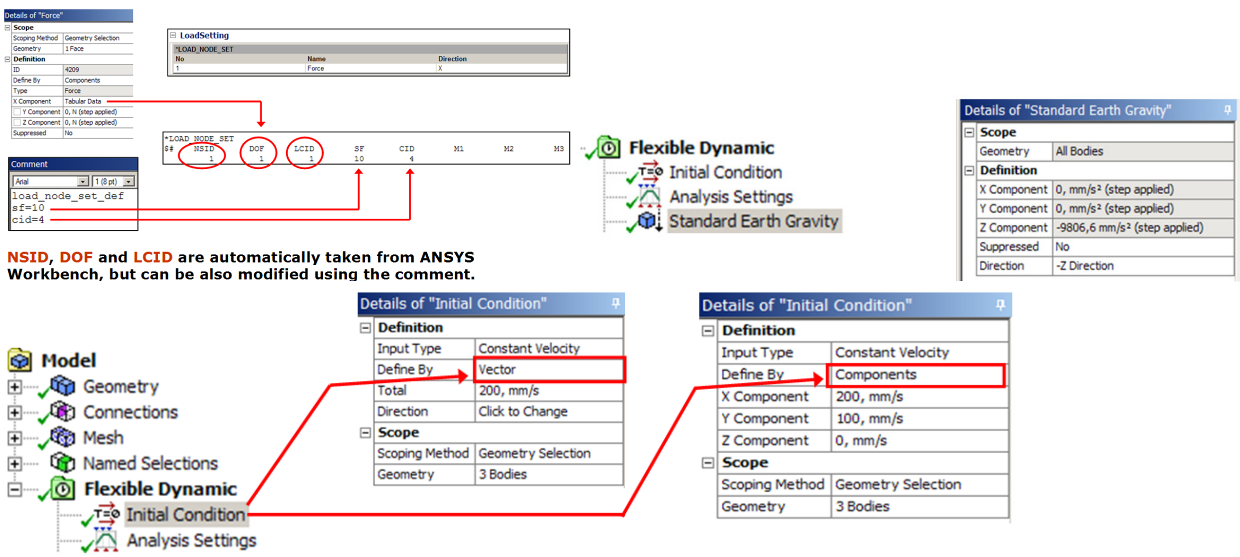


Figure 9: Loads, gravity and initial velocity options

Node Set and Part Set Definitions

Loads, gravity and initial velocities can be defined using the ANSYS Workbench GUI. Corresponding LS-DYNA keyword commands are the written.

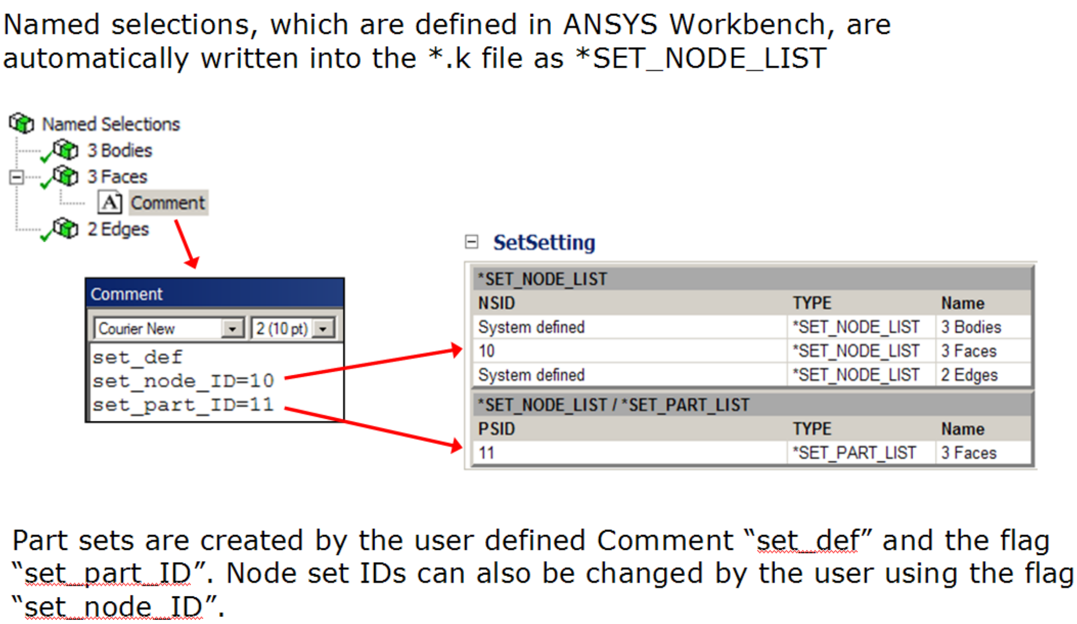


Figure 10: Named selections in form of node and part sets

Control and Database Definitions

Loads, gravity and initial velocities can be defined using the ANSYS Workbench GUI. Corresponding LS-DYNA keyword commands are the written.

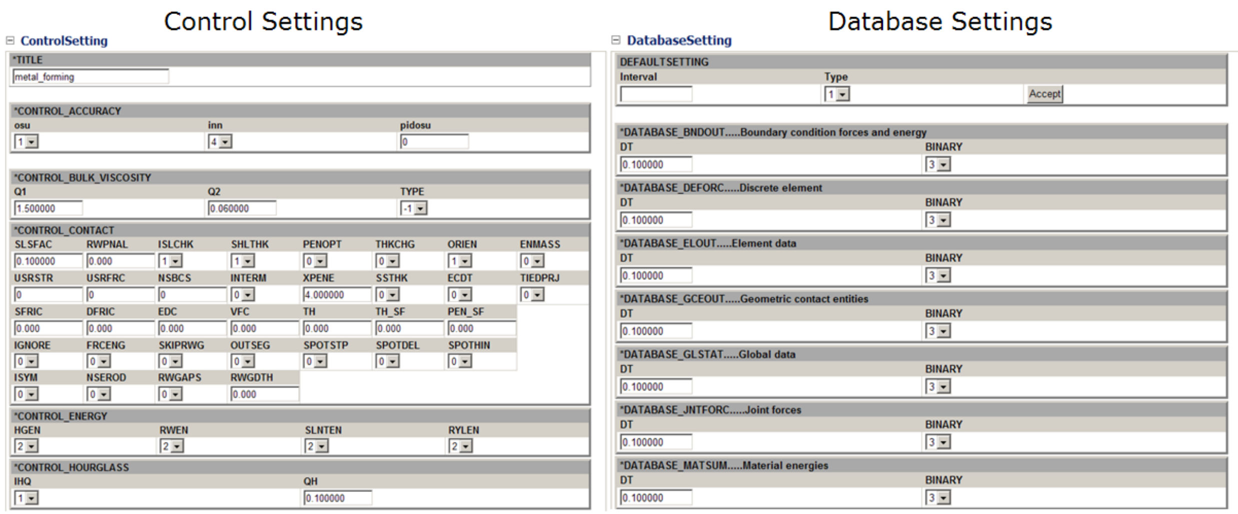


Figure 11: Control and database settings

More Information

Additional information on the interface as well as downloading the interface can be found on http://www.cadfem.de/Workbench-LS-DYNA.3622.0.html

Übersicht | Ablauf | Details | Veranstaltungen | Download | Kontakt

Übersicht "ANSYS Workbench for LS-DYNA"

Workbench Explizit

"ANSYS Workbench for LS-DYNA" ist der Schlüssel für die explizite Berechnung in der Workbench Umgebung, der dem ANSYS Anwender die Tür zu den herausragenden expliziten Möglichkeiten von LS-DYNA öffnet.

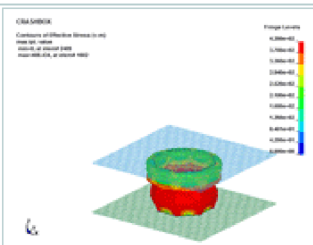
Die von CADFEM entwickelte Schnittstelle erlaubt den Datenaustausch von ANSYS Workbench v11 nach LS-DYNA. Ergebnis: Typische explizite LS-DYNA Aufgabenstellungen - große Deformationen, stark nichtlineares Materialverhalten oder kurzzeitdynamische Vorgänge, z.B. Falltestanalysen - können in der gewohnten komfortablen ANSYS Workbench Umgebung aufbereitet, per Mausklick an LS-DYNA übergeben und dort berechnet werden.

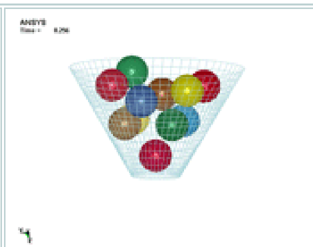
Über die reine Struktur hinaus werden Materialien, Kontaktdefinitionen, Randbedingungen sowie Verschiebungsgrößen- und Kraftvorgaben übertragen.

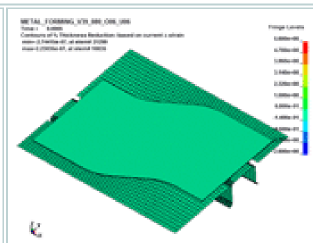
Auf folgenden Seiten finden Sie umfassende Informationen rund um diese Schnittstelle:

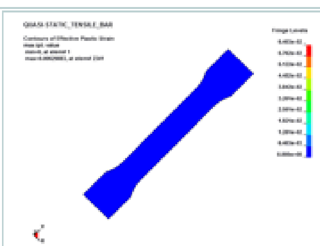
- Beispiel zum Ablauf einer expliziten Berechnung mit LS-DYNA aus ANSYS Workbench
- Details zu Elementtypen, Kontakten, Randbedingungen etc. sowie techn. Präsentation, Handbuch
- Veranstaltungen rund um ANSYS Workbench for LS-DYNA
- Download Schnittstelle "ANSYS Workbench for LS-DYNA"
- Ihre Ansprechpartner

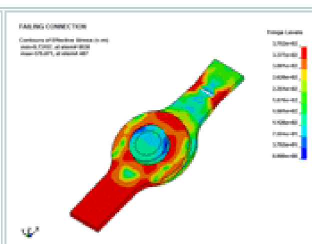
Beispiele

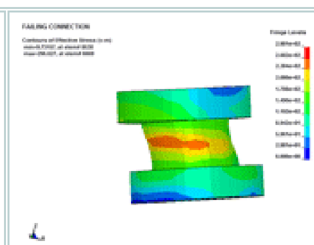

Dynamisches Eindrücken einer Crashbox


Komplexe Kontaktberechnung


Metallumformung


Quasi-Statische Belastung eines Zugstabes


Versagensanalyse Gewindeniet


Versagensanalyse Kunststoffteil