Webinar: LS-DYNA Introduction to contacts

Emily Owen

7th January 2020

Oasys

Outline

- Contact Familiarisation

  - *Terminology*

  - *Contact Types*

  - *Theory*

- Defining a contact

- Contact Checking

- Practical Considerations

Oasys

LS-DYNA Contacts Overview

- Contacts are surfaces defined from existing nodes and elements to prevent parts of the model from passing through (or separating from) each other.

- LS-DYNA offers a large number of contact types and a number of parameters that can be used to improve the contact performance for a specific case.

.000000000

Oasys
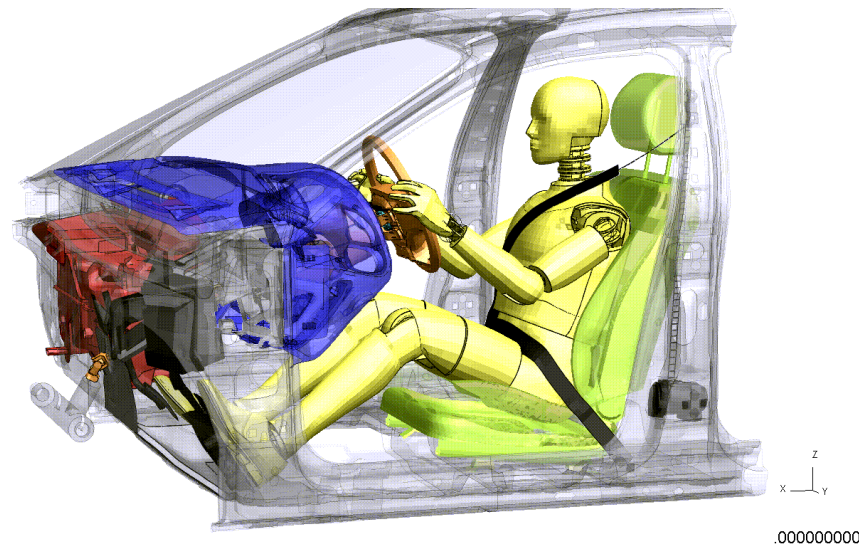
Contact Terminology

- ***Slave*/*master*** - names for two opposing sides of a contact surface.

- ***Segment*** - face of an element included in the contact
    - e.g. 3 or 4-noded shell or one face of a solid.
    - You can define a contact in several different ways (e.g. Part Set), but whichever method you use, LS-DYNA converts the faces of surfaces into segments during the analysis

.000000000

Oasys
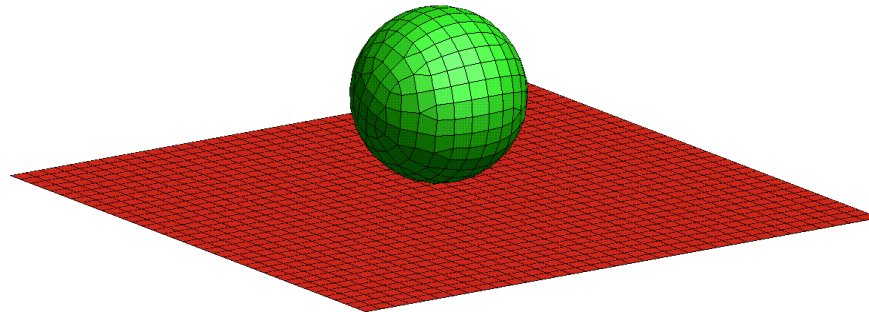
Contact Familiarisation

- Contact is defined through the keyword;

> ***CONTACT_OPTION1_{OPTION2}_...**

- **OPTION1_** determines contact type and underlying contact algorithm

    - One way vs. two way

    - Automatic vs. Non-automatic

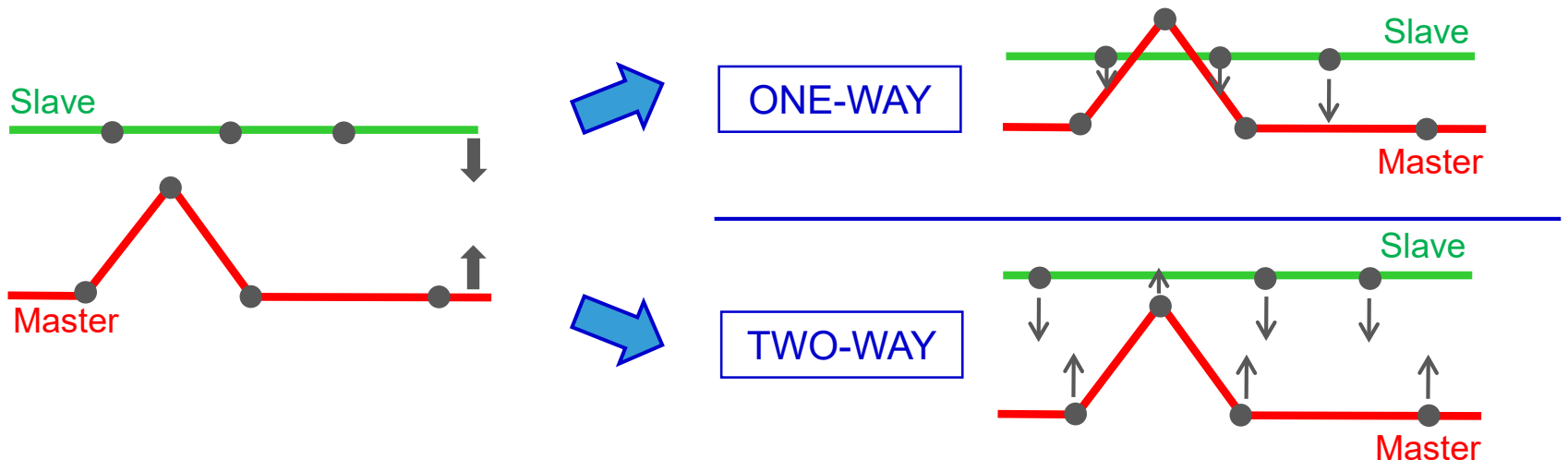   - Contact algorithm; penalty or constraint etc.

e.g.

    *CONTACT_SURFACE_TO_SURFACE

    *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE

    *CONTACT_ONE_WAY_AUTOMATIC_SURFACE_TO_SURFACE

Contact Types

- Contact can use **one-way** or **two-way** definition.

- **One-way contacts** – only the slave nodes are checked for penetration.

- **Two-way contacts** - masters and slaves are reversed and contact checked both ways.

- Typically, if the name of the contact does not specify it, the contact is two-way; single surface contacts are two way.

ONE-WAY

TWO-WAY

Slave

Master
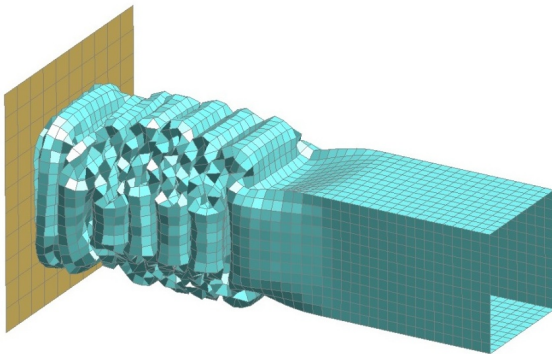
Slave

Master

Slave

Master

Oasys

Contact Types

- Contacts which have '**AUTOMATIC**' in the name are an improved algorithm invoked to compute contact.

- *Automatic contacts are:*

  - Non-oriented, i.e. can detect penetration coming from either side of the shell element

    - Always take into account shell thickness,

    - More robust

    - Recommended

Oasys

Contact Algorithms
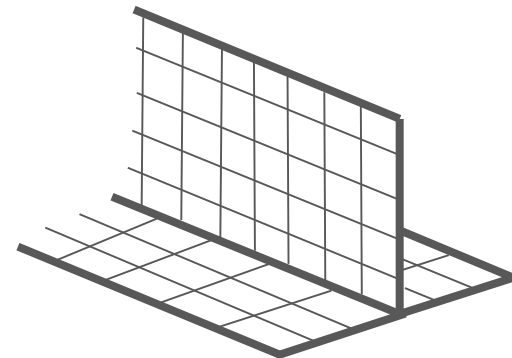
- **Penalty based** – use numerical springs to:

  - Prevent penetration of nodes of one surface into segments of the other

  - Transfers loads between two parts.

  - E.g. *CONTACT_..._SINGLE_SURFACE, *CONTACT_..._SURFACE_TO_SURFACE

- **Constraint based** – create a kinematic constraint between nodes of one part and segments of the other.

  - E.g. *CONTACT_TIED_...

| Penalty Based	Constraint Based

Oasys

Penalty Based Theory

- A slave node penetrates the master segment. This penetration is detected & the penetration depth D_P is calculated.

- A force is applied to the slave node. This "penalty" force is applied on the slave node;

  Force = Stiffness * D_P

- The effect is to project the node back up to the surface of the master segment.

- A reaction force is applied to the master segment nodes, such that the total force on the master nodes equals the applied slave node force.

- Both a <u>normal</u> (F_N) and a <u>friction</u> ($F_f = \mu * F_N$) penalty force are applied to the slave node.

$$F_N = k * D_P$$

Slave node

Oasys
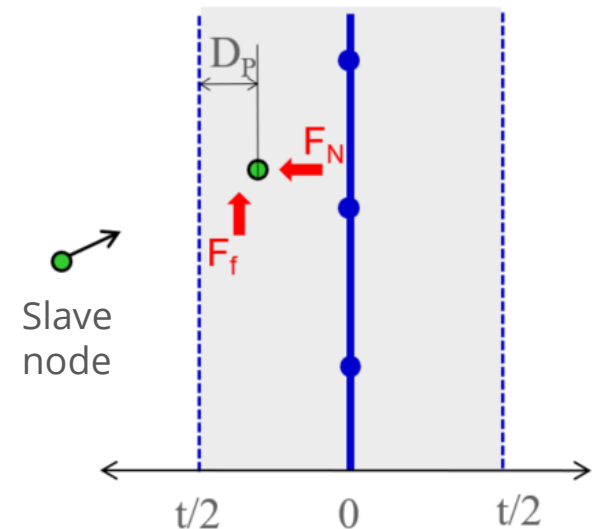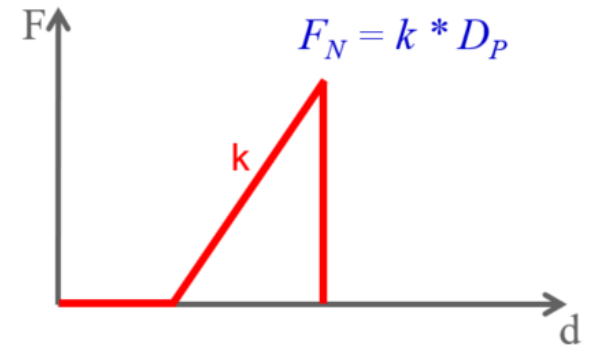
Contact Theory - Contact Thickness

- Two types of thicknesses in LS-DYNA;

- **Shell Thickness** – defined in *SECTION_SHELL or *ELEMENT_SHELL_THICKNESS

- **Contact Thickness** – determines thickness offsets in contact.

  - Default contact thickness = Shell thickness

  - The default contact thickness for single surface contact is:

    - min(shell thickness, 40% of element edge length)

    - Unless **SSTHK =1** on *CONTROL_CONTACT or **OPTT** on *PART_CONTACT is used.

Oasys
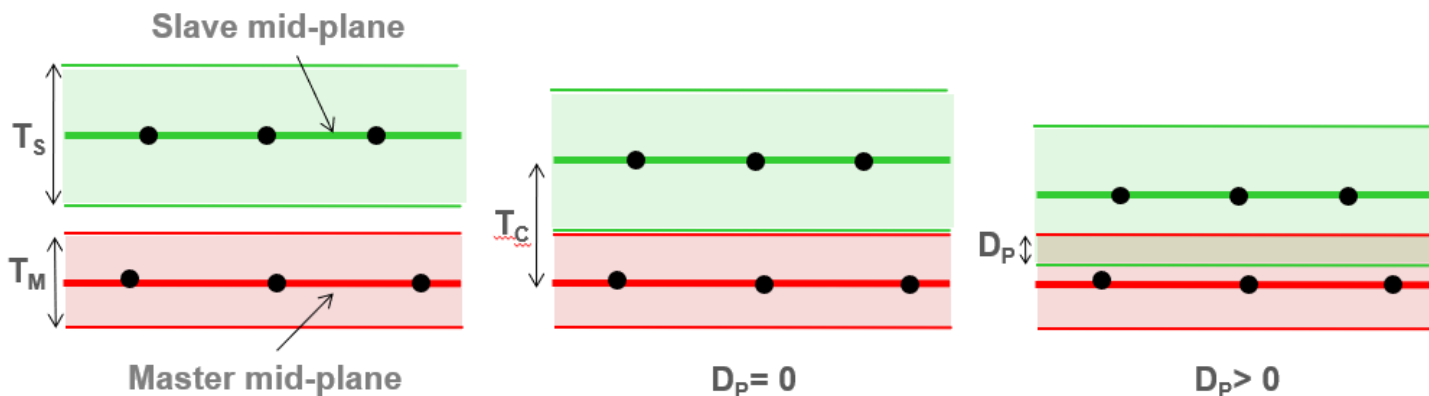
Contact Theory – Contact Stiffness

The _default_ calculation of the stiffness of a contact segment is as follows:

$$k = \frac{fs \cdot Area^2 \cdot K}{Volume}$$

for segments on _solid_ elements

$$k = \frac{fs \cdot Area \cdot K}{Maximum\_diagonal}$$

for segments on _shell_ elements

where:
- **_fs_** – the penalty factor
- **_Area_** – the area of the contact segment,
- **_K_** – the bulk modulus of the contacted element

$$K = \frac{E}{3(1-2\nu)}$$

- A rigid material still requires **_E_** and **_ν_** values, to calculate the bulk modulus, _K_, for the contact stiffness.
- For contact between parts with different stiffness, LS-DYNA choses the lowest of the stiffnesses.
- For shells on solids, the shells determine the contact stiffness.

Oasys

Contact Theory – Contact Stiffness

- **SOFT** parameter is on optional Card A of *CONTACT and controls the contact formulation for each surface.

SOFT	
0	Default penalty formulation. Suited for contacts between stiff materials
1	Activates Soft Constraint Penalty. Used for contact between parts with dissimilar material properties, or dissimilar mesh densities
2	Activates Segment-Based Penalty formulation. Switches the node-to-segment formulation in to segment-segment formulation.

SOFT = 2

SOFT=2 Activates Segment-Based Penalty formulation. Switches the node-to-segment formulation in to segment-segment formulation.

If no slave nodes are detected when they are projected on to the master surface, as in the examples below, penetrations will occur:

Master

Slave

Edge contact
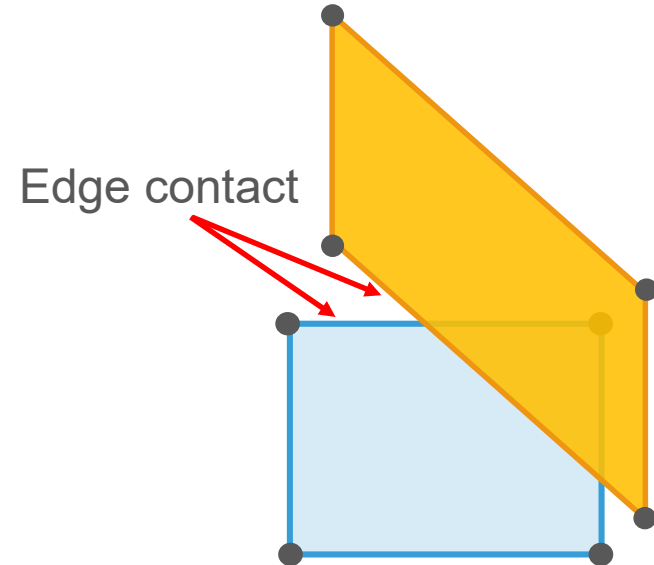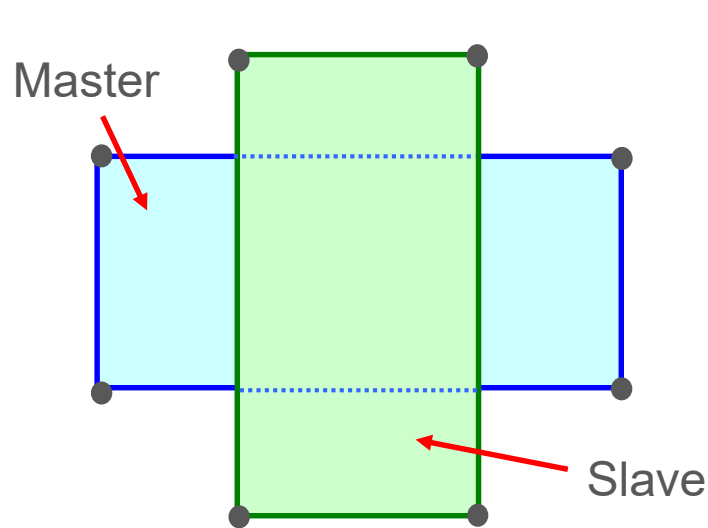
Only works with shell and solid elements and with
*CONTACT_AUTOMATIC_SURFACE_ TO_SURFACE and
*CONTACT_AUTOMATIC_SINGLE_ SURFACE.

SBOPT = 3
DEPTH =5

Oasys

Contact Parameters

- Parameters are set in a hierarchy:

    1. *CONTROL_CONTACT sets overall defaults,

    2. *CONTACT_... sets parameters for that contact – *overrides 1 above*,

    3. *PART_CONTACT sets parameters for that part in the contact – *overrides 1 & 2 above*.

- Important parameters are:

    - Contact thickness and scaling factors

    - Penalty stiffness scale factors or **SOFT** options

    - Friction values

    - Treatment of initial penetrations with **IGNORE** option

Oasys

Defining a Contact

Defining a contact

- In general, a contact analysis requires the following steps:

  1. Definition of the contact surfaces

     1. Nodes/Elements to be included.

     2. Master/slave surfaces

  2. Choice of a suitable contact type for the interacting surfaces

     1. One-way vs. Two-way

     2. Automatic/non-automatic

     3. Contact Formulation

  3. Post-processing of the contact information to ensure that a physically realistic contact has been defined.

DEMO

Creating a Contact

Mandatory cards: _Card 1_

Card 1	1	2	3	4	5	6	7	8
Variable	SSID	MSID	SSTYP	MSTYP	SBOXID	MBOXID	SPR	MPR
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none			0	0
Remarks	1	2			optional	optional	0=off	0=off

- Define Slaves (**SSTYP**) and Masters using (**MSTYP**):

    - Set of Segments (0), set of Shells (1), set of Parts (2), Part (3), Node Set (4) - slave nodes for node to surface contact.

    - All (5) - for single surface definitions (no master defined).

    - Set of exempted parts (6) - slave side only.

Oasys

Creating a Contact

Mandatory cards: _Card 2_

Card 2	1	2	3	4	5	6	7	8
Variable	FS	FD	DC	VC	VDC	PENCHK	BT	DT
Type	F	F	F	F	F	I	F	F
Default	0.	0.	0.	0.	0.	0	0.	1.0E20
Remarks								

- Define friction **FS** (static) and **FD** (dynamic):

  - Fstatic = Fdynamic = 0.2 is typical

- Contact birth time (**BT**) and death time (**DT**).

Oasys

Creating a Contact

Mandatory cards: _Card 3_

Card 3	1	2	3	4	5	6	7	8
Variable	SFS	SFM	SST	MST	SFST	SFMT	FSF	VSF
Type	F	F	F	F	F	F	F	F
Default	1.	1.	element thickness	element thickness	1.	1.	1.	1.

- Penalty stiffness scale factors for slave (**SFS**) and master (**SFM**) – increase this if contact not stiff enough, but must take care as model can become unstable.

- Optional thickness for slave (**SST**) and master (**MST**) – overrides the true shell thickness for contact calculations only.

- Scale factor for slave (**SFST**) and master (**SFMT**) surface thickness – scales the true shell thickness for contact calculation only.

Oasys

Creating a Contact

Optional cards: *Card A*

Optional Card A	1	2	3	4	5	6	7	8
Variable	SOFT	SOFSCL	LCIDAB	MAXPAR	SBOPT	DEPTH	BSORT	FRCFRQ
Type	I	F	I	F	F	I	I	I
Default	0	.1	0	1.025.	0.	2	10-100	1
Remarks			type a13					

SOFT – previously discussed, used to switch contact formulation.

SBOPT – segment-based contact options – default (2), assumes planar segments

DEPTH – Search depth in automatic contact - default 2.

BSORT – bucket sort is performed every BSORT cycles during a calculation

Oasys

Creating a Contact – Additional Parameters

Optional Card B:

**PENMAX -** defines maximum penetration allowed – different use for different contacts, default recommended:

- for example for *CONTACT_AUTOMATIC_SINGLE_SURFACE the default is 0.4, which means 40% of segment thickness.
- PENMAX can be adjusted to allow more/less penetration before the nodes are released.

Optional Card C:

IGNORE - Ignore initial penetrations – more about this later

Oasys

Extent of Contact Surface

- In theory, ALL parts can be included into one single surface contact.

  o e.g. *AUTOMATIC_SINGLE_SURFACE, *AUTOMATIC_GENERAL.

- A *DEFINE_BOX can be used to reduce contact size and save CPU:
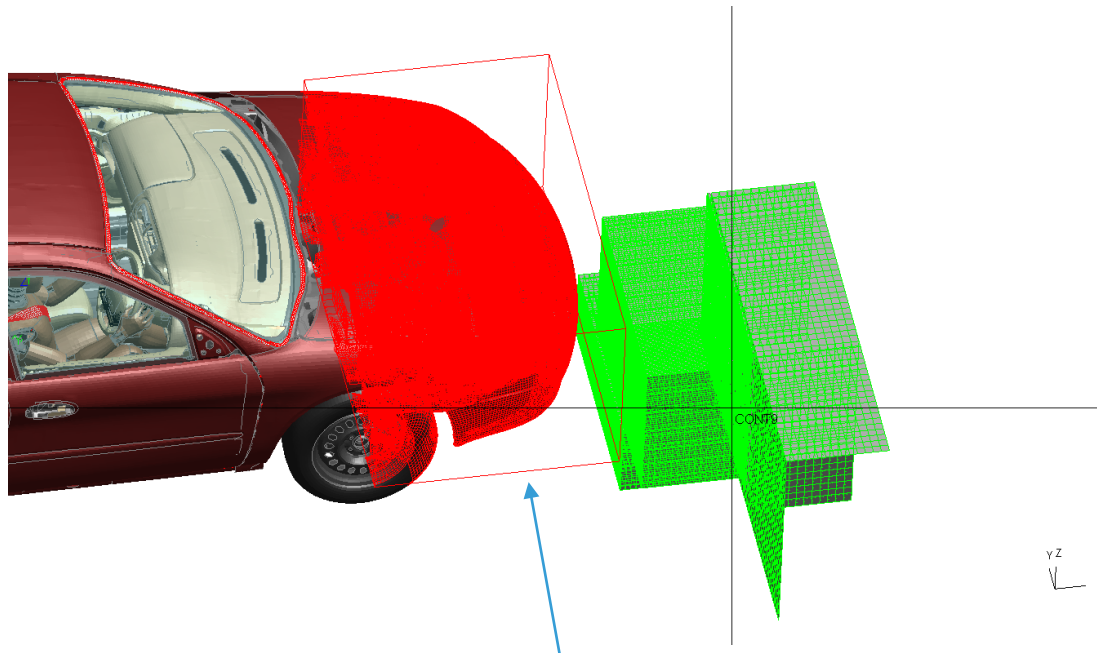
Define box is a volume that you define in Cartesian co-ordinates under SBOXID and MBOXID in *CONTACT_xx

Extent of Contact Surface

- In practise, the model should be split into several contacts. The following items need to be excluded from the "whole model" contact:
  - Spotweld beams, bushes, springs, etc should be excluded.
  - Any parts made of <u>solid</u> elements that can fail. These need to use *CONTACT_ERODING... as this will recalculate the contact surface after element failure.
  - Parts which penetrate other parts.
  - Contacts of special interest should be put into a separate contact

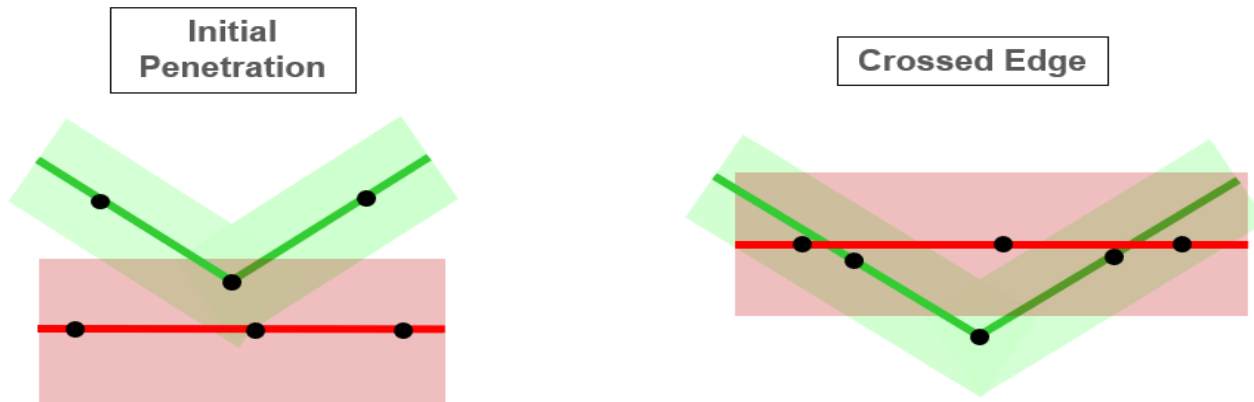- Contacts must not be duplicated – this will lead to numerical instabilities

Contact Checking

Contact Checking

• In Pre-processor:

 • Try to check for penetrations and crossed elements while creating the mesh, e.g.:


• After initialising in LS-DYNA:

  • Check .otf file for messages for initial penetrations.

  • Magnify displacements x100 or x1000, plot model after 5 or 10 cycles, look for deformation caused by initial penetrations.

  • Check contact timestep (not needed for SOFT=1 & 2): timestep reported should be greater than model timestep or instability could occur during the run.

Contact Checking – Initial Penetrations

- If the model contains initial penetrations, these will affect the contacts.

- By default LS-DYNA moves the nodes in order to remove the initial penetrations.

- An alternative is to set parameter IGNORE=1 (available for Automatic Contact types).

- This can be set on *CONTROL_CONTACT (card 4) for all contacts in the model or selectively on each contact (Optional Card C).

Default Contact
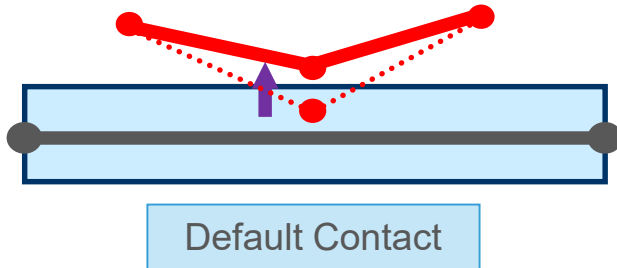
The penetrating node is projected back to outer edge of the contact surface


IGNORE=1
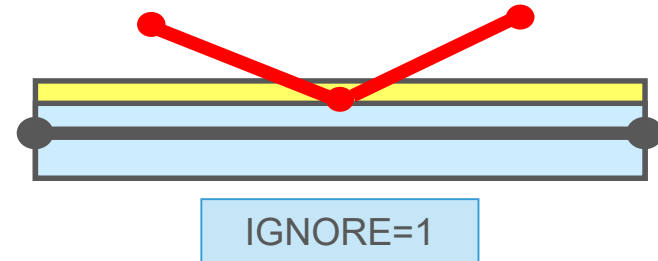
The node is not moved, the contact thickness is reduced for the penetrating node, such that is doesn't penetrate.
(other nodes not initially penetrating still see the full thickness)

- If later on the node moves away from the surface, the full contact depth is reinstated.

- IGNORE=2 gives the same result as 1 but outputs more info – see manual

Contact Checking – Contact not stiff enough

When a sliding contact is not stiff enough and parts start to penetrate through each other, there are several approaches to fixing the problem:

1. _Set SOFT=1_ – _if applicable_

2. _Set SOFT=2_ – _if applicable_

3. _Contact Thickness change_:
   - New thickness which overrides true thickness **SST**/**MST** (F2)
   - Scale true thickness **SFST**/**SFMT** (F2)

4. _Penalty Stiffness change_:
   - Increase penalty stiffness factors **SFS**/**SFM** (F1)
   - Or, if dealing with a rigid or null material, increase Young's modulus (F1)

$$F = k * D_P$$
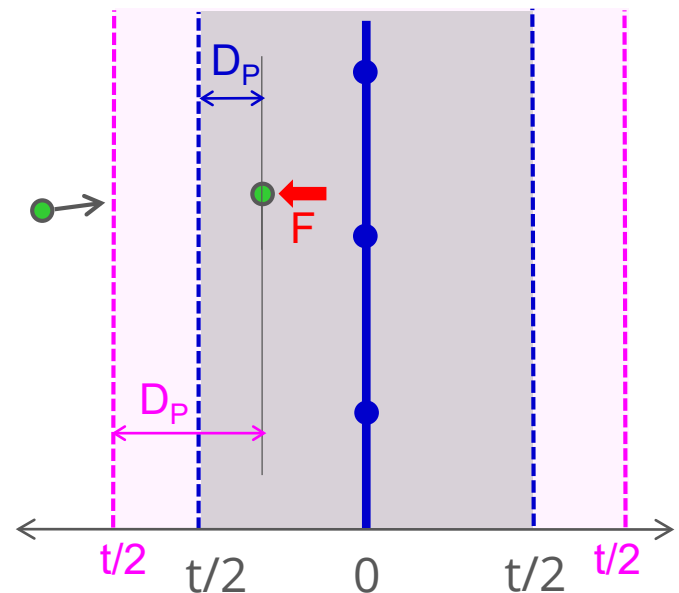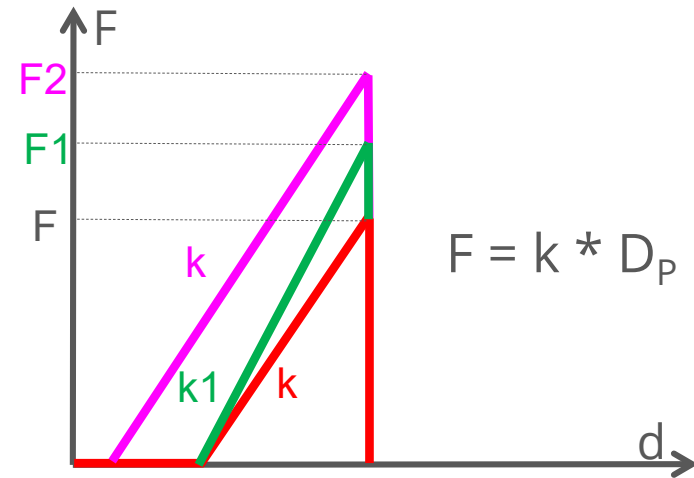
Contact Checking – Contact Timestep

- Penalty based contacts have a timestep, dependant on the contact stiffness.

- The contact timestep is not the same as the element timestep because the contact stiffness is calculated in a different way.

- The contact timestep is given in the otf file

  o search for smallest – the contact timestep summary is above this

```
****************************************************************

    slave surface of interface    =         1
    type                          =       a 3
    surface timestep              = 6.610E-07
    controlling slave node ID     =      1888
    part ID of slave node         =         2
    current minimum time step     = 6.610E-07

    master surface of interface   =         1
    type                          =       a 3
    surface timestep              = 1.830E-05
    controlling master node ID    =      1421
    part ID of master node        =         1
    current minimum time step     = 6.610E-07

The LS-DYNA time step size should not exceed    6.610E-07
to avoid contact instabilities.  If the step size is
bigger then scale the penalty of the offending surface.

S t o r a g e    a l l o c a t i o n

Memory required to begin solution      :      348912
Additional dynamically allocated memory:      183285
                         Total:      532197

100 smallest timesteps
--------------------------------------------------
```

Note that the contact interface number in the otf file will not necessarily match the Contact ID in the keyword file; single surface contacts are not included in this list.

type refers to the contact type, e.g. a3 is automatic surface to surface.

The timestep for slave and master sides are given for two-way contacts. Note master and slave timesteps are different.

Oasys

Other Keyword Cards

ASCII output files: *DATABASE_option

| |
|---|
| **GLSTAT: Global Statistics** |
| Total Sliding Energy: value representing energy dissipated in the contact. With no friction or damping present ideally should always be null |
| **RCFORC: Resultant Contact Forces** |
| Resultant interface forces is the sum of force over all nodes in the interface due to contact |
| **SLEOUT: Contact Energy** |
| Sliding Interface Energies, for master and slave side of the interface. With no friction/damping these should be equal in magnitude but opposite in sign |
| **NCFORC: Contact forces at each node** |
| Nodal interface forces, force acting on each node on the interface |

- Contact forces & energies are calculated incrementally every timestep.

- It is advisable to always switch **FRCENG**=1 on *CONTROL_CONTACT. This will help in understanding the magnitude of frictional contact energy & contact energy due to contact forces.

Oasys

Contact Checking – Monitoring Forces

- The contact force time histories for *Surface-to-surface* and *nodes-to-surface* contacts can be output. This is requested under *DATABASE_ASCII (RCFORC).

- The forces <u>on particular parts</u> in *single surface* contacts can be measured using the *CONTACT_FORCE_TRANSDUCER_PENALTY card
    - Despite the name, this is not a contact surface and it does not generate any contact forces – it only measures the contact force from any contacts that have already been defined.

- A *DEFINE_BOX can be used to identify which contact event is being measured.
- The example shown will measure forces on selected segments are green
- For (axi)symmetric problems, if all segments are included, the total force would sum to zero; therefore, use force transducers with care.


$$F_{front} \qquad F_{back}$$

Oasys
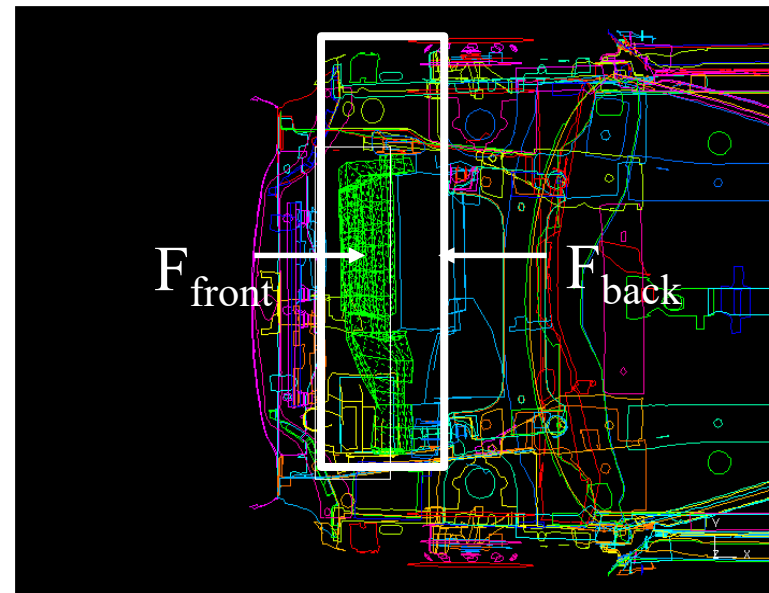
Contact Problems

- The contact energy time history should be smooth and a positive number (energy cannot be negative).
  - This is energy stored in the contact "springs" and should be small compared with deformation energy.
  - Contact energy in a contact spring = ½*(contact force)*(penetration)

    = ½*(contact stiffness)*(penetration)^2

2 usual problems:

- Sudden steps in contact energy indicate numerical instabilities in the contact surface, e.g. nodes catching and getting stuck, instead of sliding smoothly.

- Negative energies indicate large penetrations in the contact (if the energy is initially negative, it means there are large initial penetrations).
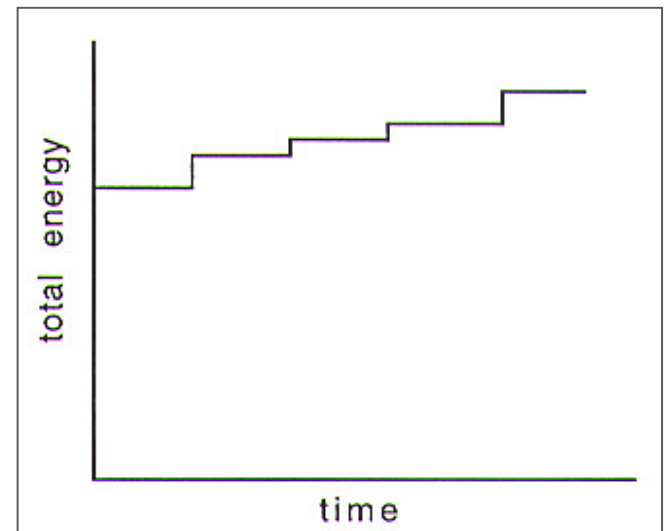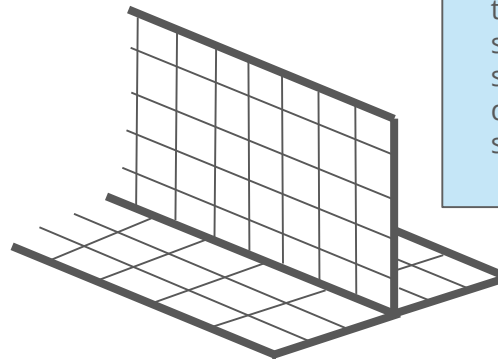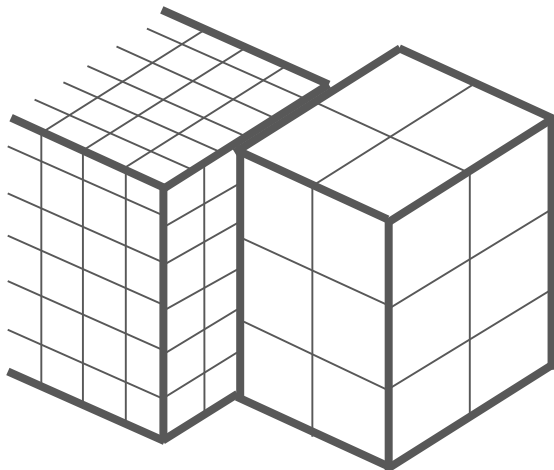
Tied Contacts

Tied Contacts

- ***Tied contact*** – ties the slave side of the contact to the master side.

- Tied contacts can use either a _constraint_ or a _penalty_ formulation.

- Useful for solid mesh transitions, modelling welds etc.

- Slave nodes must be "close" to be TIED – close defined as:

$$D < 0.6 * (master\ thickness + slave\ thickness)$$

- If the node is too far away it will not be tied and a warning will be printed in the d3hsp file (.otf file) – example below.

```
*** Warning Slave node is not constrained
          since it is too far from segment.
   tied interface #         =      1
   slave node #             =     220
   segment #                =     126
   distance from segment    =  0.2009796E+01
   segment nodes:
      118     165     164     113
```

Oasys

Tied Contacts

- Several types of TIED contact available:
  - *CONTACT_TIED_NODES_TO_SURFACE – constrains translational DOF only.
  - *CONTACT_TIED_SURFACE_TO_SURFACE – constrains translational DOF only.
  - *CONTACT_TIED_SHELL_EDGE_TO_SURFACE – (TSE2S) constrains translational AND rotational DOF.
  - *CONTACT_AUTOMATIC_..._TIEBREAK - allows defined failure forces/stresses.
  - *CONTACT_SPOTWELD (alternative name for TSE2S).

- Important considerations when using the default definition:
  - Basic tied contacts use a _constraint_ formulation and cannot be used with rigid parts or other constraints.
  - Slave nodes are projected onto the master segment at initialisation altering geometry if D < 0.6 * (master thickness + slave thickness):

$$D < 0.6*(T_S + T_M)$$

Slave

Master
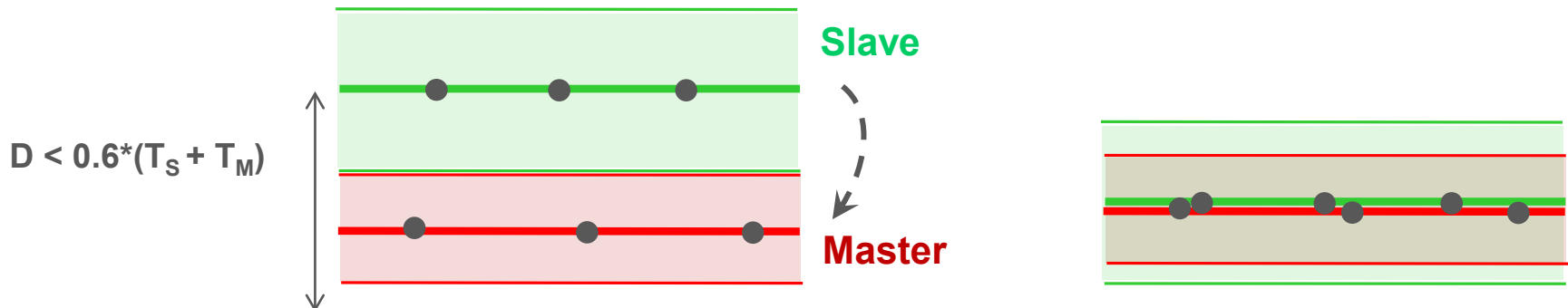
Tied Contacts

Additional options are available for TIED contact types, which maintain the offset of slave nodes (i.e. do not alter geometry):

- **_OFFSET** – maintains offset and switches to _penalty_ formulation.

- **_BEAM_OFFSET** – maintains offset and switches to _penalty_ formulation (TSE2S only); the name 'BEAM' is due to the fact that it uses 'artificial beams' in the contact calculation, which allows for moment transfer, i.e. it is more accurate for the offset option (and so it is not limited to beams).

- **_CONSTRAINED_OFFSET** – _constraint_ formulation (use with TSE2S only), moments are computed.

Oasys

Practical Considerations

Recommend Sliding Contact Types

Application	Contact type	Special Parameters
General model	*CONTACT_AUTOMATIC_SINGLE_SURFACE	
Model with many initial penetrations	*CONTACT_AUTOMATIC_SINGLE_SURFACE	IGNORE=1
Edge contact problems	*CONTACT_AUTOMATIC_SINGLE_SURFACE or *CONTACT_AUTOMATIC_GENERAL (no SOFT option required)	SOFT=2, SBOPT=3, DEPTH=5 Apply selectively in problem areas
Deformable barrier to model	*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE	As above
Beams or other non-continuous mesh	*CONTACT_AUTOMATIC_NODES_TO_SURFACE or *CONTACT_AUTOMATIC_GENERAL (considers cross sectional area of beams along its length, but expensive)	
Folded airbag	*CONTACT_AIRBAG_SINGLE_SURFACE	SOFT=2, DEPTH=3 BSORT = 4 or 1 in extreme cases

Contact types and usage

- The following table presents a summary of commonly used sliding contacts and their application for various contact situations.

  - Beam to Beam

  - Beam to Shell Edge

  - Beam to Shell Surface

  - Shell Edge to Shell Edge (External edges)

  - Shell Segment to Segment

  - Shell Edge to Shell Edge (Internal edges)

- The examples were run with SOFT values of 0, 1 and 2

- The following results are reported:

  - PASS   - Contact worked

  - FAIL   - Contact failed

  - Reverted to SOFT=1 - Contact switched to SOFT=1 internally by LS-DYNA

- The examples were run in R6.0, revision 70232, MPP, single precision
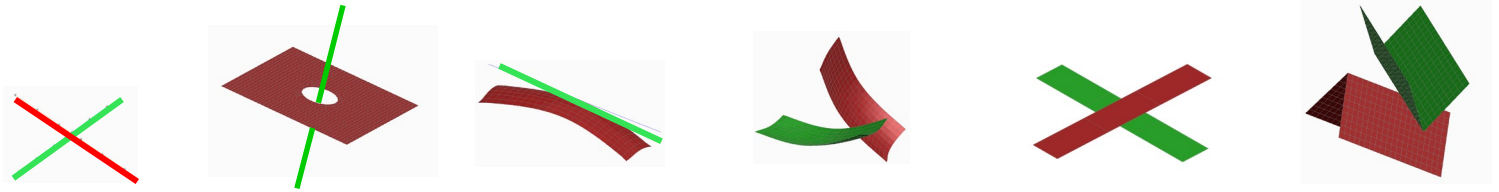
Oasys

Contact types and usage

Contact	SOFT	Beam to beam	Beam to shell	Beam to surf	Edge to edge 1	Segm to segm	Edge to edge 2
AUTOMATIC_ GENERAL	0	PASS	PASS	PASS	PASS	PASS	FAIL
	1	PASS	PASS	PASS	PASS	PASS	FAIL
	2	Reverted to SOFT=1	Reverted to SOFT=1	Reverted to SOFT=1	Reverted to SOFT=1	Reverted to SOFT=1	FAIL
AUTOMATIC_ GENERAL_ INTERIOR	0	PASS	PASS	PASS	PASS	PASS	FAIL
	1	PASS	PASS	PASS	PASS	PASS	FAIL
	2	Reverted to SOFT=1	Reverted to SOFT=1	Reverted to SOFT=1	Reverted to SOFT=1	Reverted to SOFT=1	FAIL
AUTOMATIC_ SURFACE_TO_ SURFACE	0	FAIL	FAIL	FAIL	FAIL	FAIL	FAIL
	1	FAIL	FAIL	FAIL	FAIL	FAIL	FAIL
	2	FAIL	FAIL	FAIL	PASS	PASS	PASS
AUTOMATIC_ SINGLE_SURFACE	0	FAIL	FAIL	PASS	FAIL	FAIL	FAIL
	1	FAIL	FAIL	PASS	FAIL	FAIL	FAIL
	2	FAIL	FAIL	FAIL	PASS	PASS	PASS

Contact types and usage
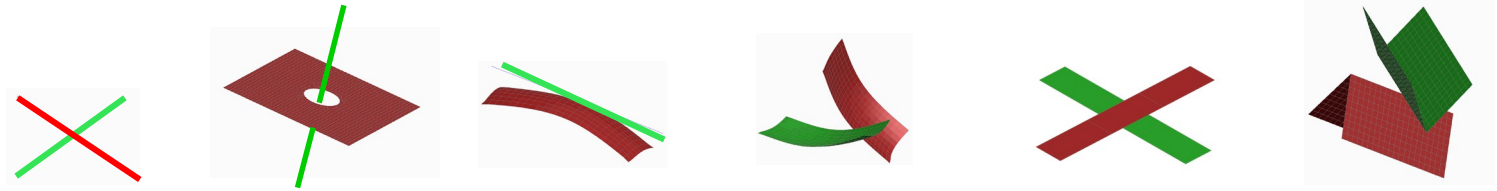
Contact	SOFT	Beam to beam	Beam to shell	Beam to surf	Edge to edge 1	Segm to segm	Edge to edge 2
AUTOMATIC_ BEAMS_TO_ SURFACE	0	FAIL	PASS	PASS	FAIL	FAIL	FAIL
	1	FAIL	PASS	PASS	FAIL	FAIL	FAIL
	2	FAIL	PASS	PASS	FAIL	FAIL	FAIL
SINGLE_EDGE	0	FAIL	FAIL	FAIL	PASS	FAIL	FAIL
	1	FAIL	FAIL	FAIL	PASS	FAIL	FAIL
	2	FAIL	FAIL	FAIL	Reverted to SOFT=1	FAIL	FAIL
AIRBAG_SINGLE_ SURFACE	0	FAIL	FAIL	PASS	FAIL	FAIL	FAIL
	1	FAIL	FAIL	PASS	FAIL	FAIL	FAIL
	2	FAIL	FAIL	FAIL	PASS	PASS	PASS

QUESTIONS

Next webinar

- When: 4th February 2020

- Topic: ICFD an introduction

- Register on our website here:

https://www.oasys-software.com/dyna/training/webinars/ls-dyna-intro-to-icfd/

Oasys

Next training courses

Oasys LS-DYNA team is offering complimentary spaces on Arup-delivered training courses. Upcoming courses in 2020:

- 21 - 23 January   Introduction to LS-DYNA

- 5 February   Introduction to Oasys PRIMER

- 12 February   Oasys PRIMER: seatbelt fitting and dummy positioning

- 18 February   NEW: ICFD: an introduction

Location: Arup office in Solihull, UK

Newsletter sign up

To keep up with our latest news please sign up to our monthly newsletter via: www.oasys-software.com/dyna/signup

Thank you for your time!

Contact Information

ARUP

www.arup.com/dyna

For more information please contact us:

UK	**China**	**India**	**USA West**
The Arup Campus	Arup China	Arup India	Arup Americas
Blythe Valley Park	39/F-41/F Huaihai Plaza	Ananth Info Park, HiTec City	c/o 560 Mission Street Suite 700
Solihull	1045 Huaihai Road (M)	Madhapur Phase-II	San Francisco
B90 8AE	Xuhui District, Shanghai	Hyderabad	CA 94105
United Kingdom	200031	500081, Telangana	United States
	China	India	
T: +44 121 213 3399	T: +86 21 3118 8875	T: +91 40 44369797 / 98	T: +1 415 940 0959
dyna.support@arup.com	china.support@arup.com	india.support@arup.com	us.support@arup.com

or your local Oasys distributor

Oasys