Document information

Doc. no.: N/a

Revision: 1

Prepared for: Public webinar

Project no.: N/a

Approved by: MG

Release date: 2022-06-16

Distribution: Approved for public release

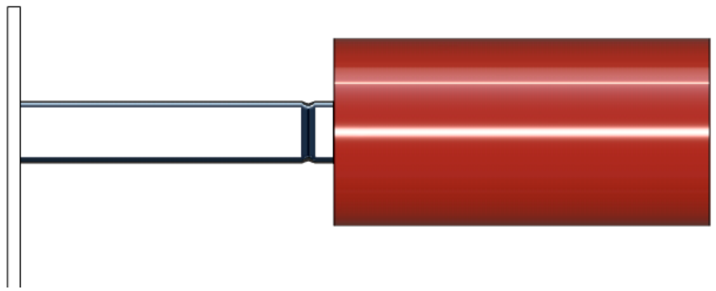
DYNAmore Nordic AB
Brigadgatan 5
SE-587 58 LINKÖPING
Sweden

Org. no. 556819-8997
EC VAT:
SE556819899701

Phone: +46 (0)13 236680
Fax: +46 (0)13 214104
E-mail: info@dynamore.se
Web: www.dynamore.se

Webinar 2022-06-16

Explicit Model Breakdown



Marcus Gustavsson, DYNAmore Nordic

DYNAmore Nordic

Support, training, guidelines, and more

- Support
  - Support manager: Dr. Jimmy Forsberg
  - E-mail: support@dynamore.se
    - Target to answer in 4 hours, carried out
  - Call: +46 13 236680
- Training & seminars including on-line & on-site: www.dynamore.se
- Secure file server: files.dynamore.se
  - Software and license download, client area with guidelines and more
- www.dynamore.se - information on LS-DYNA, Seminars, Conferences
- www.dynalook.com – Papers from international LS-DYNA conferenses
- www.dynasupport.com – General support for LS-DYNA
- www.dynaexamples.com – LS-DYNA example models from crash to DEM.

Agenda

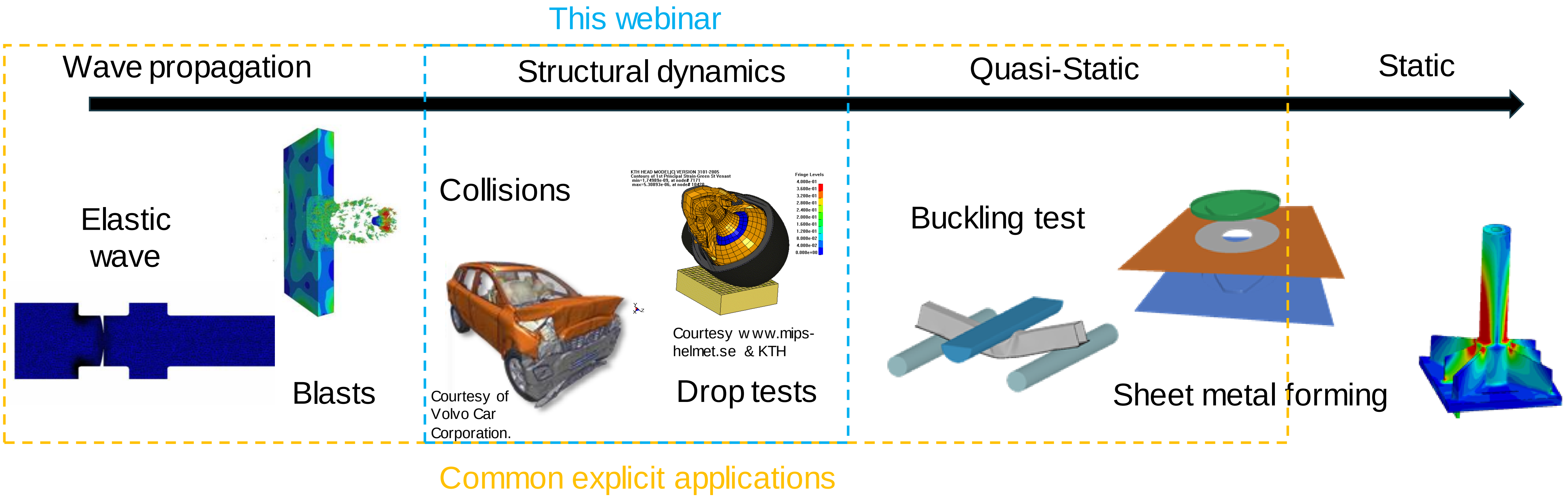
Webinar Content

- About explicit simulation
- LS-DYNA’s keyword format & model structure
  - FE-geometry
  - Material
  - Boundary conditions
  - Control & output

LS-DYNA Explicit

Applications

- Dynamic problems
  - The mass inertia affects the results
  - Time is real
- Quasi-Static and Static problems
  - The mass inertia should not affect the results
  - The time should not affect the results



LS-DYNA Explicit

Time Integration

- Explicit schemes gives the configuration at time t_{n+1} as an explicit function of earlier configurations, i.e.,

$$\mathbf{x}_{n+1} = \mathbf{f}(\mathbf{x}_n, \mathbf{x}_{n-1}, \ldots, t_{n+1}, t_n, \ldots)$$

- Explicit conditionally stable, a sufficiently small time step needed
  - Critical time step size is dependent on the speed of sound

$$\Delta t_{\text{critical}} = \frac{l_e}{c_e} \qquad c_e \sim \sqrt{\frac{E_e}{\rho_e}}$$

  - Smaller mesh ➡ smaller time step ➡ longer simulation time (wall clock time)
- No information can propagate across more than one element per time step

LS-DYNA Keyword Format

The Basics

- LS-DYNA's input files are called keyword files (.k, .key)
- Input is structured in "keywords" that start with "*"
- A comment card starts with "$"

Keyword

Cards

```
*PART
$#                                                                 title
Tube 1
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1         0         1         0         0         0
```

Field

- 0 in a field most often result in a default value
- The number of cards and field formatting differ between keywords.
- The keyword file always start with `*KEYWORD` and end with `*END`
- Read more about formatting specifics in the latest LS-DYNA manual www.dynasupport.com/manuals

```
*KEYWORD
*TITLE
Test example
$ Define nodes and elements
*NODE
*ELEMENT_SHELL
$ Define section and material
*PART
$ Define element types and integration
*SECTION_SHELL
$ Define material properties
*MAT_ELASTIC
$ Define loads and BC
*LOAD_NODE
*BOUNDARY_PRESCRIBED_MOTION_RIGID
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE
$ Define output of results
*DATABASE_BINARY_D3PLOT
*DATABASE_GLSTAT
$ Control cards govern entire simulation
*CONTROL_TERMINATION
*CONTROL_TIMESTEP
*END
```

LS-DYNA Keyword Format

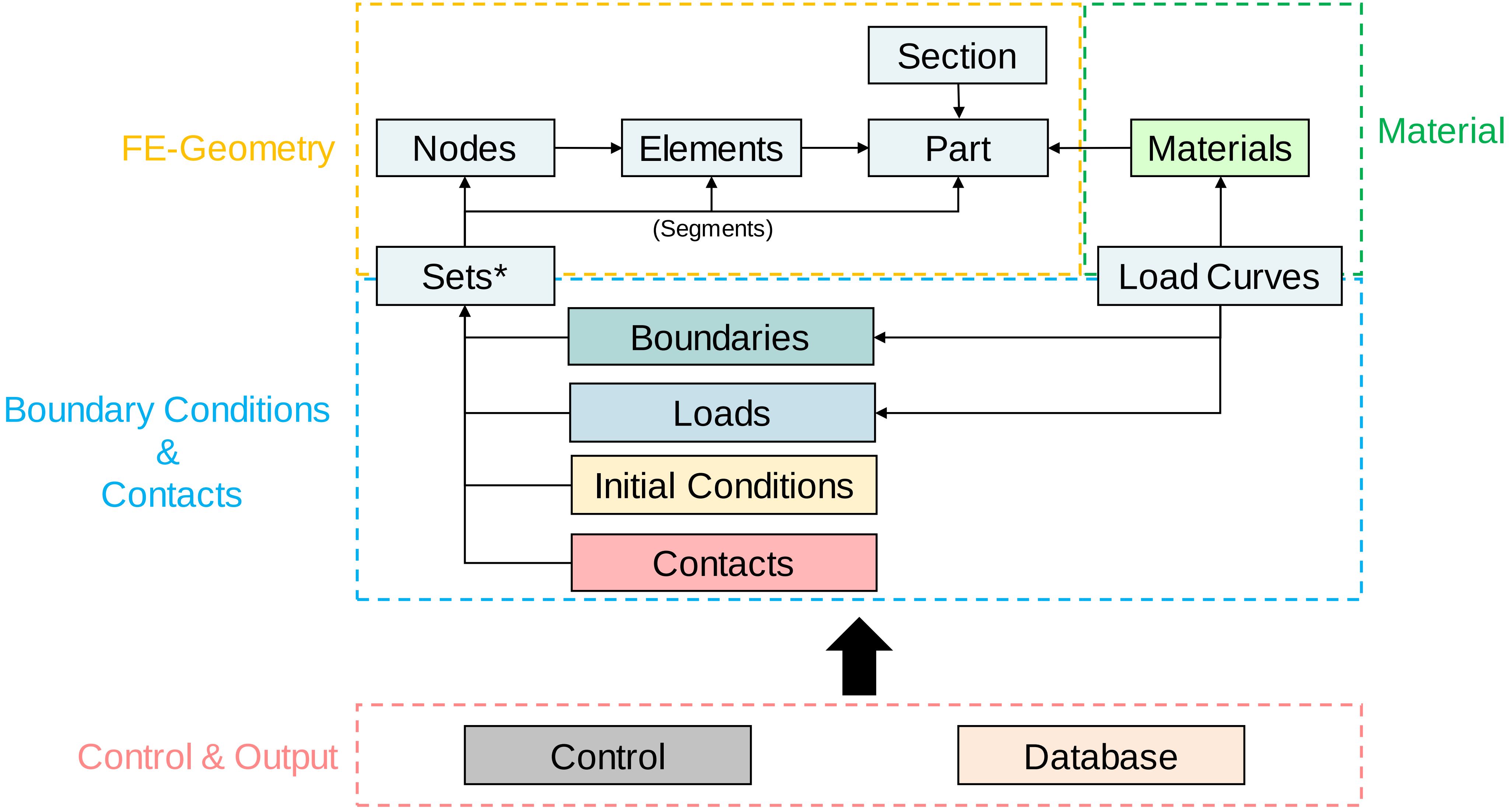
Consistent Units

- LS-DYNA has no fixed unit system
  - It simply requires units that are consistent with Newton’s second law, F=ma
  - Unit system is implicitly defined by the units of material properties and model length that are input

MASS	LENGTH	TIME	FORCE	STRESS	ENERGY	DENSITY	YOUNG's	GRAVITY
kg	m	s	N	Pa	J	7.83e+03	2.07e+11	9.806
kg	cm	s	1.0e-02 N	Pa	J	7.83e-03	2.07e+09	9.806e+02
kg	mm	ms	kN	GPa	kN-mm	7.83e-06	2.07e+02	9.806e-03
g	cm	s	dyne	dyne/cm²	erg	7.83e+00	2.07e+12	9.806e+02
g	mm	s	1.0e-06 N	Pa	1.0e-09 J	7.83e-03	2.07e+11	9.806e+03
g	mm	ms	N	MPa	N-mm	7.83e-03	2.07e+05	9.806e-03
ton	mm	s	N	MPa	N-mm	7.83e-09	2.07e+05	9.806e+03
lbf-s²/in	in	s	lbf	psi	lbf-in	7.33e-04	3.00e+07	386
slug	ft	s	lbf	psf	lbf-ft	1.52e+01	4.32e+09	32.17

LS-DYNA Model Structure

In Simple Terms



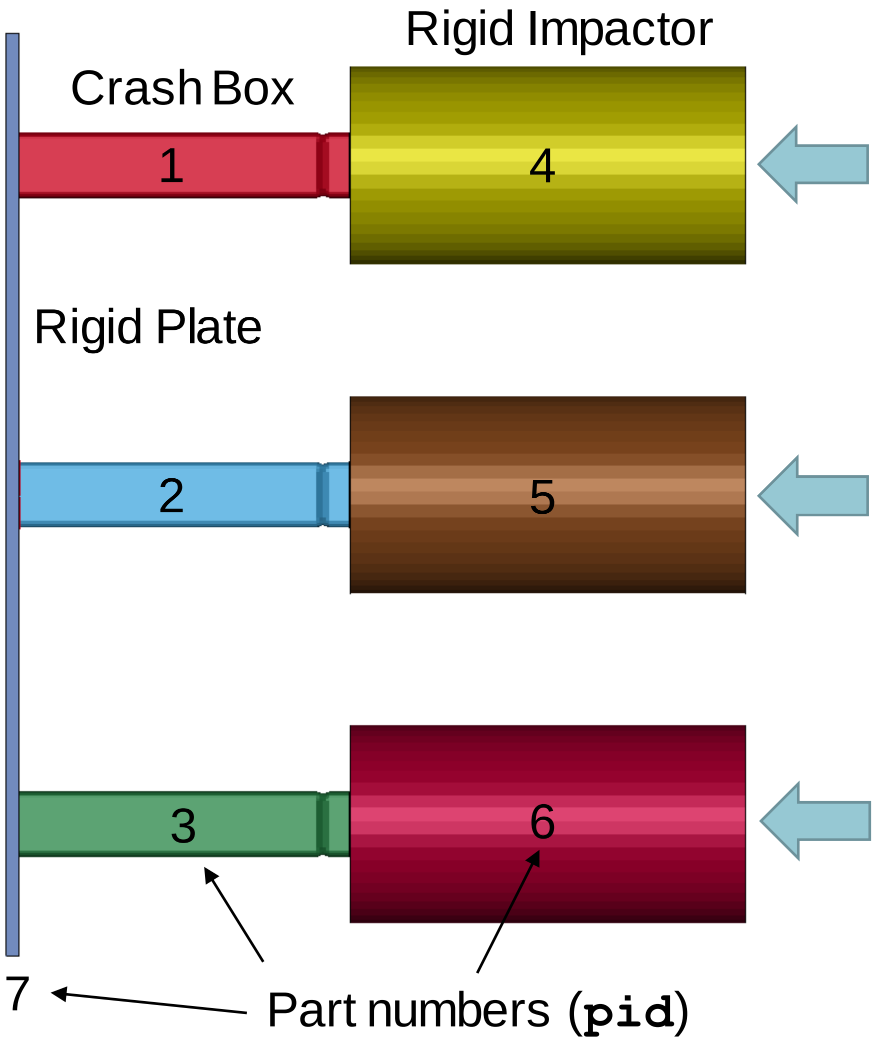
* BC:s can also be directly applied to parts, elements, nodes, etc.

Explicit Model Breakdown

Crash Box Model

- Our model is a rather simple crash box based on the one from our explicit guideline
  - 100x100x500 mm, thickness 2 mm
- The system of units used in the model is mm, ms, kg, kN, GPa.
- The three boxes and impactors are identical but have different definitions for
  - Boundary conditions
    - Impactor movement
    - Crash box fixation
  - Contact definition

Client Area > 1_LS-DYNA_Guidelines > 1_LS-DYNA_Explicit_Guidelines

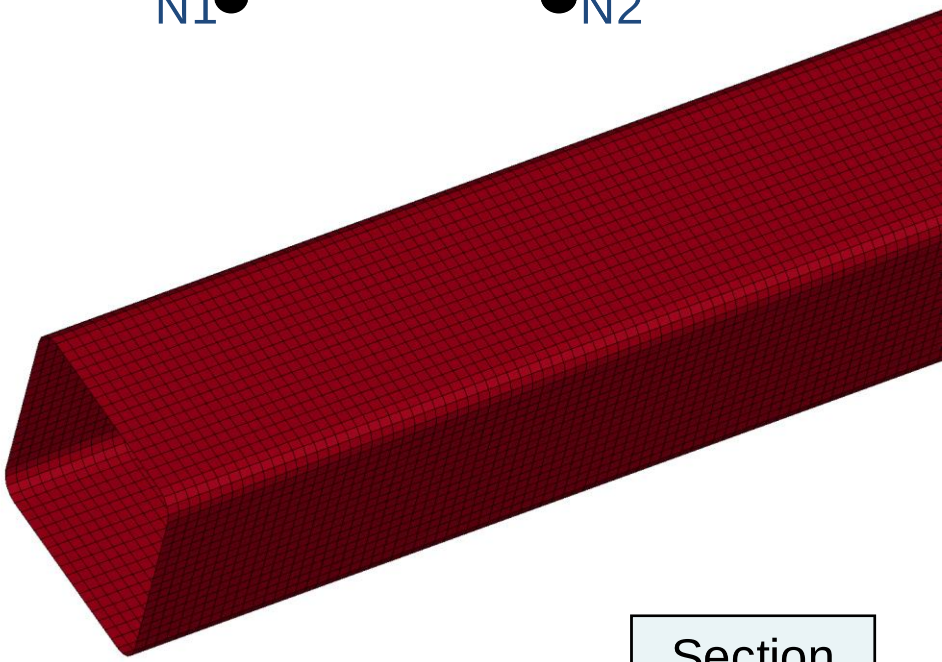


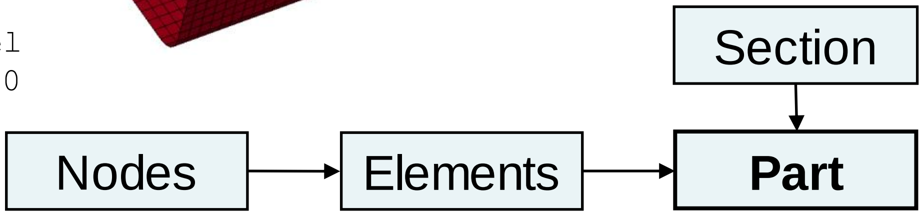
FE-Geometry

*PART – Always at the center

```
*NODE
$#   nid               x               y               z      tc      rc
       1           -20.0          -450.0          -200.0       0       0
...
*ELEMENT_SHELL
$#   eid     pid      n1      n2      n3      n4      n5      n6      n7      n8
       1       1       1       2       5       4
...
*PART
$#                                                                         title
Tube 1
$#     pid     secid       mid     eosid      hgid      grav    adpopt      tmid
         1         1         1         0         1         0         0         0
$#
$#
*SECTION_SHELL
$#   secid    elform      shrf       nip
         1       -16                   5
$#      t1        t2        t3        t4
       2.0       2.0       2.0       2.0
$#
$#
*MAT_PIECEWISE_LINEAR_PLASTICITY
$#     mid        ro         e        pr      sigy      etan      fail      tdel
         17.85000E-6     210.0       0.3       0.0      0.01.00000E21       0.0
$#       c         p      lcss      lcsr        vp
       0.0       0.0       420         0       1.0
```

Hourglass id

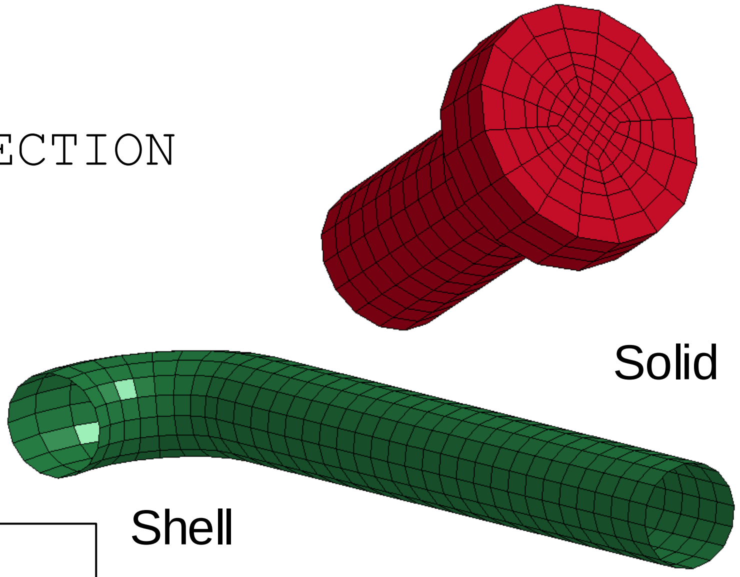




FE-Geometry

`*SECTION` – Chose element type

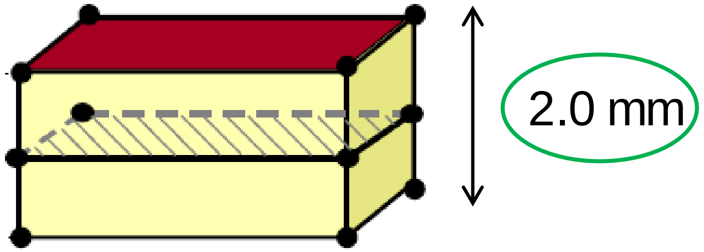
- The element formulation of a `*part` is controlled by `elform` on `*SECTION`
- Multiple `*part` can use the same `*SECTION` id (`secid`)
- <u>Crash box solids use elform 1 and shells elform -16</u>




```
*SECTION_SOLID
$#    secid    elform       aet    unused    unused    unused    cohoff    unused
          1         1
```

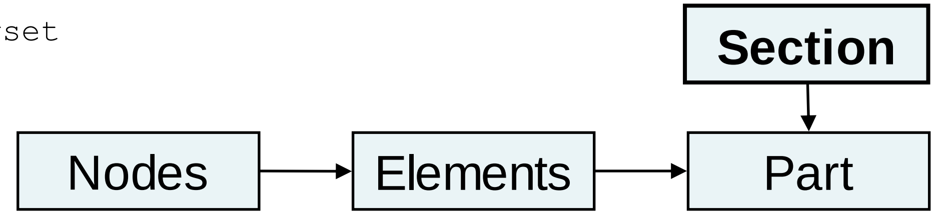
elfrom (ex for solids):
EQ.1: Constant stress solid element (default)
EQ.2: 8 point hexahedron
EQ.10 : 1 point tetrahedron
EQ.13 : 1 point nodal pressure tetrahedron
EQ.16 : 4 or 5 point 10 - noded tetrahedron

elfrom (ex for shells):
EQ.1: Hughes-Liu,
EQ.2: Belytschko-Tsay (default)
EQ.16: Fully integrated shell element
EQ.-16: Fully integrated shell element modified for higher accuracy




```
*SECTION_SHELL
$#    secid    elform      shrf       nip     propt   qr/irid     icomp     setyp
          1       -16                   5
$#       t1        t2        t3        t4      nloc     marea      idof    edgset
        2.0       2.0       2.0       2.0
```

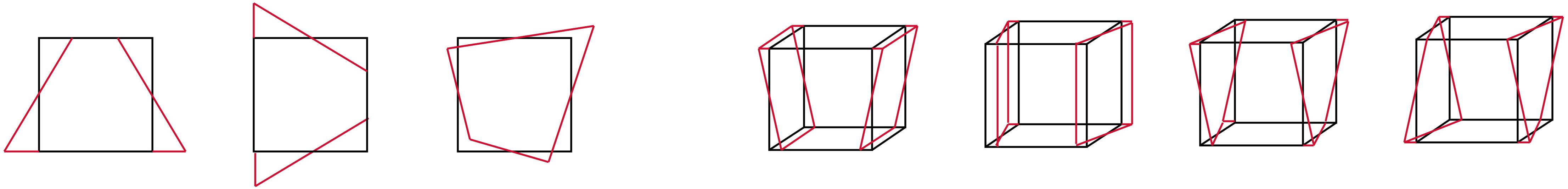
Geometrical attributes, such as thickness, are defined on *SECTION for shells (and beams)



FE-Geometry

*HOURGLASS – Control Element Behaviour

- It is not uncommon to use under integrated elements in explicit simulation.
  - Inherit to under integrated elements are zero energy/stiffness modes, so called hourglass modes. In order to mitigate this an artificial stiffness is added via *HOURGLASS or *CONTROL_HOURGLASS



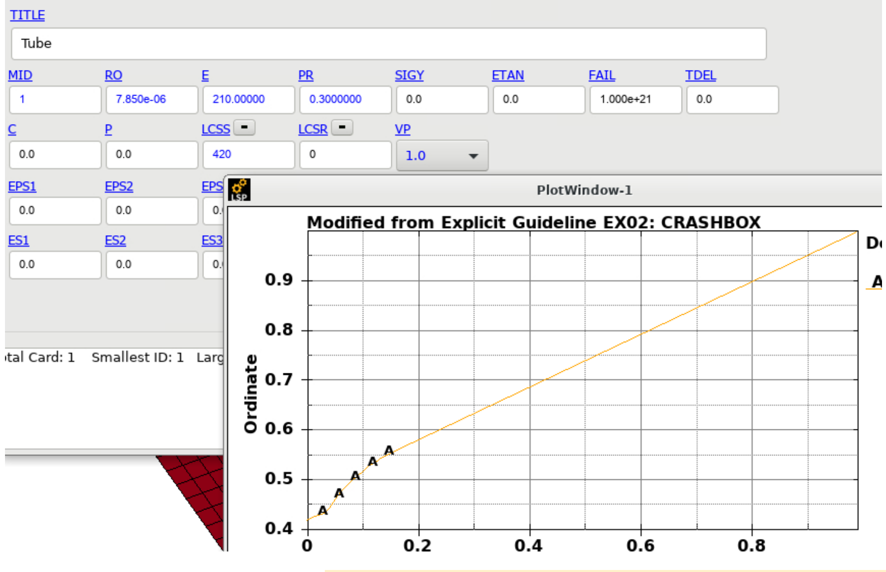
- For the Crash Box we use fully integrated shell type 16, however, we still use hourglass IHQ=8. This activates full projection warping stiffness for shell formulations 16 and -16, NOT "hourglass" per say.

```
*HOURGLASS
$#    hgid       ihq        qm       ibq        q1        q2    qb/vdc        qw
         1         8      0.03         0       1.5      0.06       0.1       0.1
```

Material

Overview

- 200+ material models in LS-DYNA that model different behaviors, plasticity, strain rate, failure, etc.
- Most materials are developed for modelling a specific material type, e.g., metals, composites, plastics; however, they are not bound to any specific physical material
- The complexity varies between material models, from the simplest isotropic elastic models to advanced elasto-plastic, rate-dependent, anisotropic models with failure. It is important to note that the more complex material model the more is required in terms of material testing and calibration

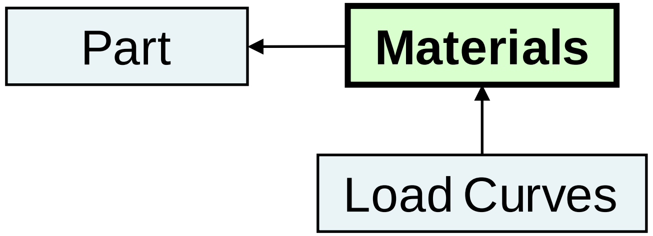


- In this webinar we consider two of the most used materials models
  - *MAT_020 – A rigid material “model” that cannot deform.
  - *MAT_024 – An elasto-viscoplastic material model

Short video - [Material hardening — Dynamore Nordic AB]

Webinar - Client Area > 2_webinars > 3_Material_and_Failure

Webinars on numerous material types and phenomena; failure, plastics, rubber, composites, and more.



Material

Crash Box Model

- Crash boxes (1,2,3) are assigned *MAT_024
  - Elasto-viscoplastic behavior is captured
- Impactors and the plate (4,5,6,7) are rigid and use *MAT_020
  - Undeformable geometry

*MAT_RIGID_(TITLE) (020) (2)

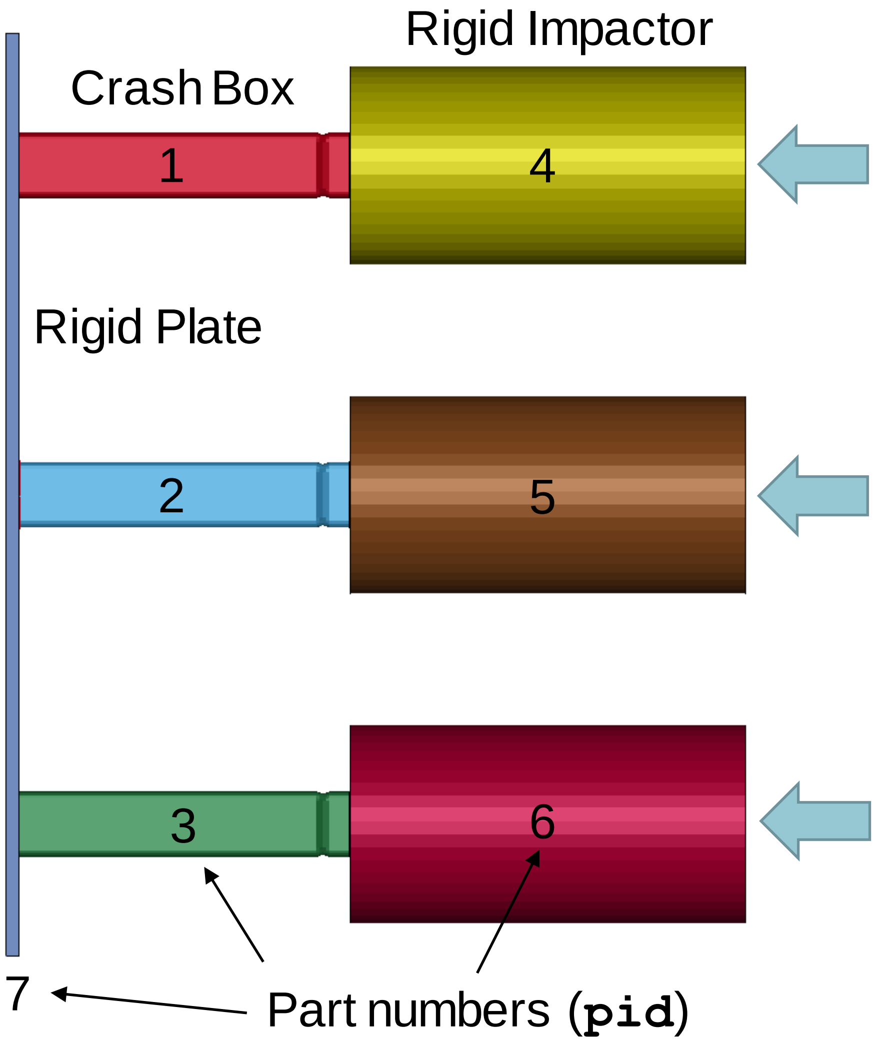
TITLE: Impactor free in x translation

1	MID	RO	E	PR	N	COUPLE	M	ALIAS
	4	1.655e-05	210.00000	0.3000000	0.0	0	0.0	
2	CMO	CON1	CON2					
	1.0	5	7					
3	LCO OR A1	A2	A3	V1	V2	V3		
	0.0	0.0	0.0	0.0	0.0	0.0		

*MAT_PIECEWISE_LINEAR_PLASTICITY_(TITLE) (024) (1)

TITLE: Tube

1	MID	RO	E	PR	SIGY	ETAN	FAIL
	1	7.850e-06	210.00000	0.3000000	0.0	0.0	1.000e+21
2	C	P	LCSS	LCSR	VP		
	0.0	0.0	420	0	1.0		

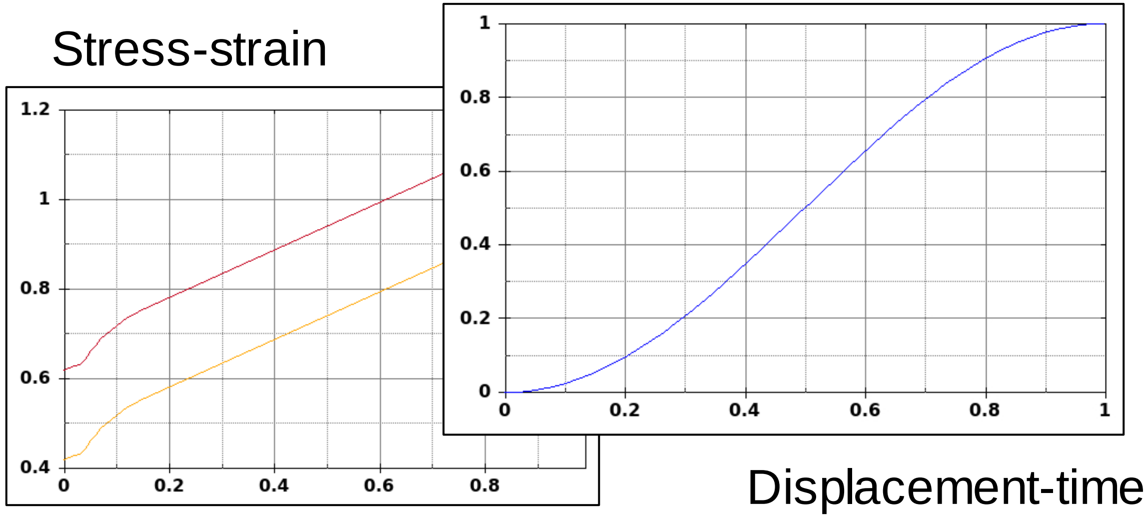


Curve Input

*DEFINE_CURVE

- Before we go on, we need to introduce `*DEFINE_CURVE`
- Input of curve data for most things in LS-DYNA
  - Stress-strain for material (hardening)
  - Displacement-time for prescribed displacement
- Extent time-dependent curves beyond termination to ensure correct value at last time step!

Stress-strain



Displacement-time

```
*DEFINE_CURVE_TITLE
strain rate 0 /s
$#      lcid      sidr       sfa       sfo...
         420         0       0.0     0.001
$#                  a1                  o1
                   0.0              419.44
              0.010075              424.68
              0.021059               428.0
              0.031039              432.16
              0.041086              443.33
              0.050897              462.12
              0.060735               475.1
              0.070969              488.87
              0.080324              498.38
              0.090179              507.77
              0.100131              517.96
              0.110688              527.06
              0.120154              534.18
              0.130916              542.01
              0.141085              548.53
              0.150458              554.24
              0.154148              556.43
                  0.99              1000.0
```

Load curve id

"x" scale factor

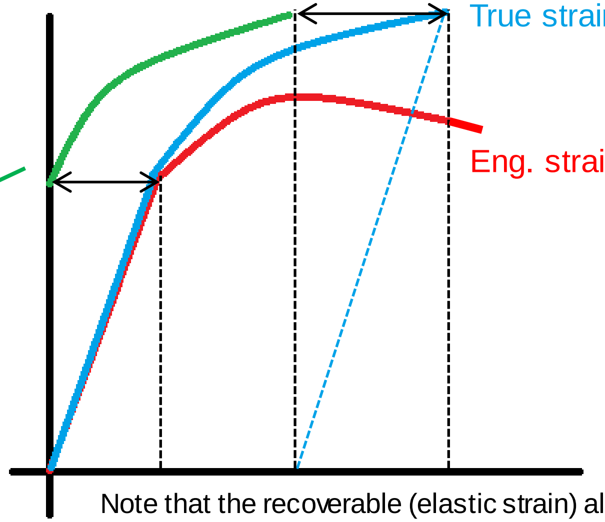
"y" scale factor

Material

*MAT_024/*MAT_PIECEWISE_LINEAR_PLASTICITY

- “The workhorse of LS-DYNA”
  - Captures the behavior of many materials adequately in many situations
    - Most metals in most situations and often adequate for plastics.

```
*MAT_PIECEWISE_LINEAR_PLASTICITY_TITLE
Tube
$#     mid        ro         e        pr      sigy      etan      fail      tdel
         17.85000E-6     210.0       0.3       0.0       0.01.00000E21       0.0
$#       c         p      lcss      lcsr        vp
       0.0       0.0       420         0       1.0
...
```



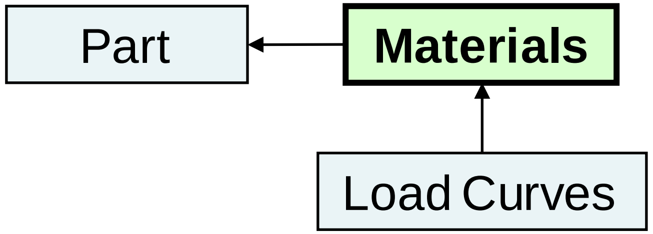
  - Isotropic linear elastic
  - Von Mises yield surface
  - Plastic hardening
    - `sigy` & `etan` or
    - `*DEFINE_CURVE` -> `LCSS` (true stress vs. true plastic strain)
    - Extrapolates linearly if $\varepsilon_{pl.} > \max(\varepsilon_{pl.curve})$, make sure final slope is appropriate
    - Only valid until necking
  - Viscoplastic

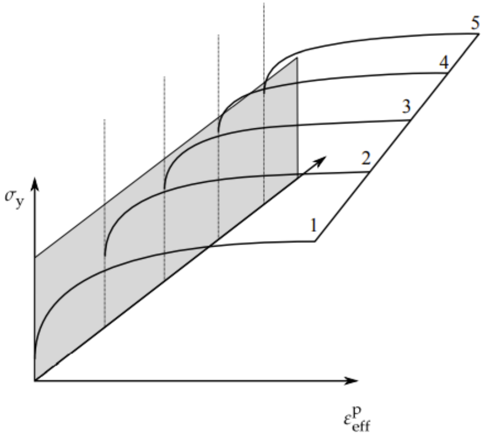
Short video - Material hardening — Dynamore Nordic AB

Material

Crash Box Model – ***MAT_024**

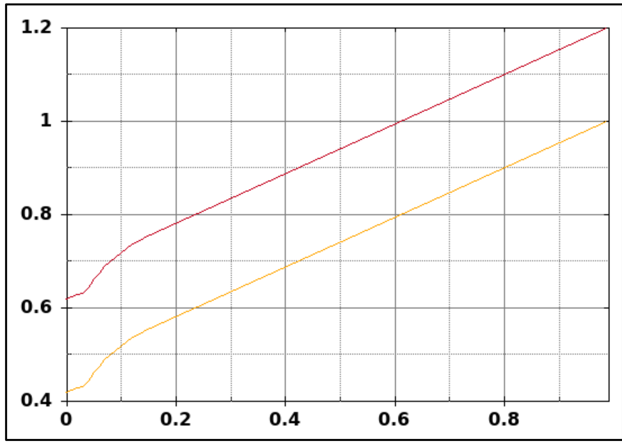
- The crash box use a viscoplastic input for `*MAT_024`
- Viscoplasticity is activaded by
  - Setting VP=1
  - Input two or more *DEFINE_CURVE on LCSS via *DEFINE_TABLE
    - Intermediate strain rate values in are interpolated between the curves
- In the case of the crash box the same curve is used wit an offset





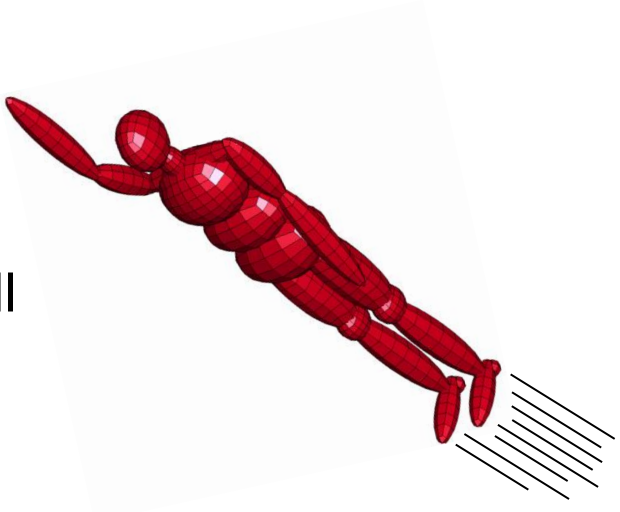
lcss
420

```
*DEFINE_TABLE
$#     tbid          sfa        offa
        420
$#                 value        lcid
                     0.0         421
                     1.0         422
*DEFINE_CURVE_TITLE
strain rate 0 /s
$#     lcid         sidr         sfa         sfo        offa        offo ...
        421            0         0.0       0.001         0.0         0.0
$#                    a1                      o1
                     0.0                  419.44
                0.010075                  424.68
...
*DEFINE_CURVE_TITLE
strain rate 1000 /s
$#     lcid         sidr         sfa         sfo        offa        offo ...
        422            0         0.0       0.001         0.0       200.0
$#                    a1                      o1
                     0.0                  419.44
...
```



Material

*MAT_020/*MAT_RIGID



- Assigned to a part just like any other mechanical material
  - The part becomes a rigid body and is bypassed in the element routine
- The rigid body formulation itself is a (real) constraint, penalty works well
  - Cannot be combined or connected to other constraints
- There are special boundary conditions for rigid bodies
- Constraints can also be defined directly on the material card

Elastic material properties
(Used in contact definition)

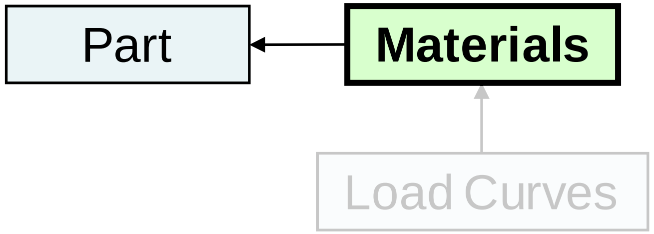
```
*MAT_RIGID_TITLE
Impactor free in x translation
$#     mid        ro         e        pr         n    couple ...
         41.65540E-5     210.0       0.3       0.0       0.0
$#     cmo      con1      con2
       1.0         5         7
$#lco or a1        a2        a3        v1        v2        v3
       0.0       0.0       0.0       0.0       0.0       0.0
```

Constraint options

- Learn more from our webinars or short videos
  - Client Area > 2_webinars > 1_Explicit_and_General > LS-DYNA_Rigid_Bodies
  - Introduction to Rigid Body Materials in LS-DYNA — Dynamore Nordic AB
  - Introduction to Rigid Body Joints in LS-DYNA — Dynamore Nordic AB

Material

Crash Box Model – ***MAT_RIGID**



- Impactors and the plate (4,5,6,7) are rigid
- Constraints on the material cards are utilized
- Parts 4,5,6 use `mid` 4
- Part 7 use `mid` 7

```
*MAT_RIGID_TITLE
Impactor free in x translation
$#     mid         ro          e         pr ...
         41.65540E-5       210.0        0.3
$#     cmo       con1        con2
       1.0          5           7
...
```

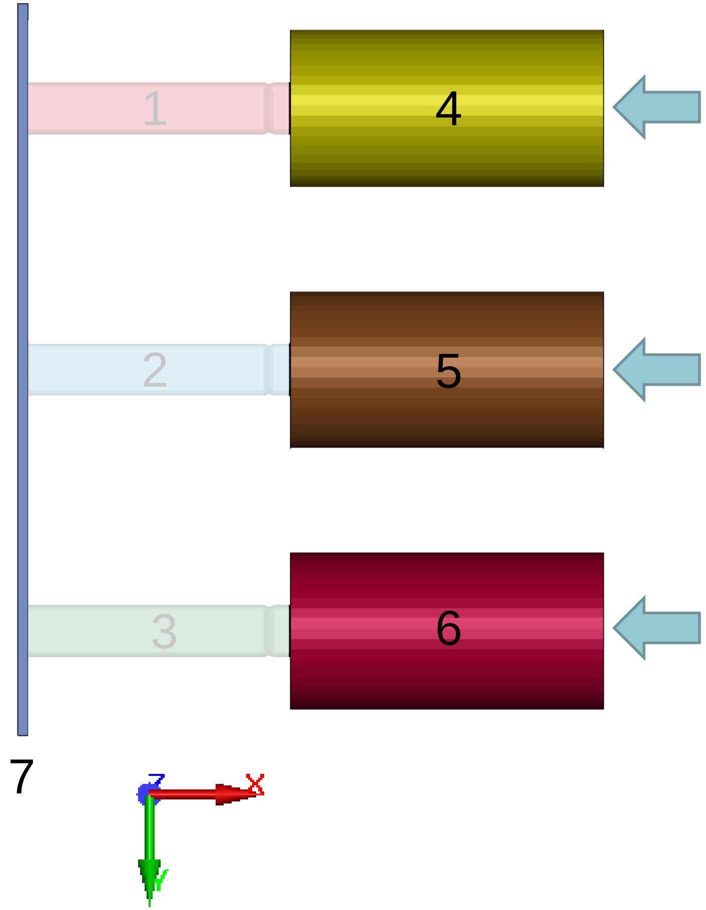
Only free in global x translation

```
*MAT_RIGID_TITLE
Plate
$#     mid         ro          e         pr ...
         77.85000E-6       210.0        0.3
$#     cmo       con1        con2
       1.0          7           7
...
```

Fixed in al DOF:s

CON1:=Global translational constraint:
EQ.0: no constraints,
EQ.1: constrained x displacement,
EQ.2: constrained y displacement,
EQ.3: constrained z displacement,
EQ.4: constrained x and y displacements,
EQ.5: constrained y and z displacements,
EQ.6: constrained z and x displacements,
EQ.7: constrained x, y, and z displacements.

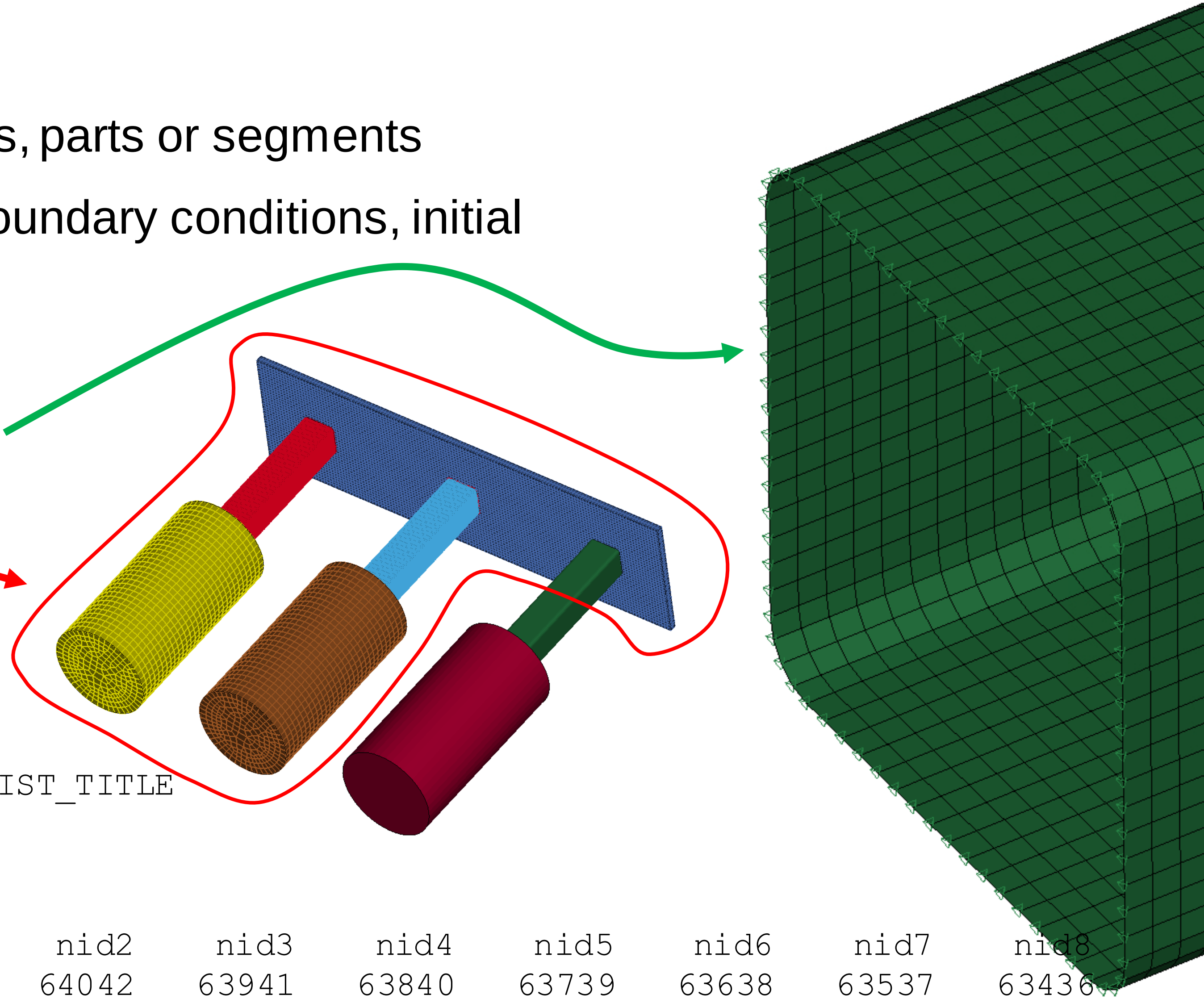
CON2:=Global rotational constraint:
EQ.0: no constraints,
EQ.1: constrained x rotation,
EQ.2: constrained y rotation,
EQ.3: constrained z rotation,
EQ.4: constrained x and y rotations,
EQ.5: constrained y and z rotations,
EQ.6: constrained z and x rotations,
EQ.7: constrained x, y, and z rotations.



Define Sets

`*SET` – Collect FE-entities

- `*SET` lets you define groups of nodes, elements, parts or segments
- These sets can be used for defining contacts, boundary conditions, initial conditions, and more
- The crash box model use
  - `*SET_NODE_LIST` (for boundary conditions)
  - `*SET_PART_LIST` (for contacts)
- There are many more options
  - `*SET_BEAM`
  - `*SET_SHELL`
  - `*SET_SEGMENT`
  - `*SET_..._ADD` (combine sets)
  - ...

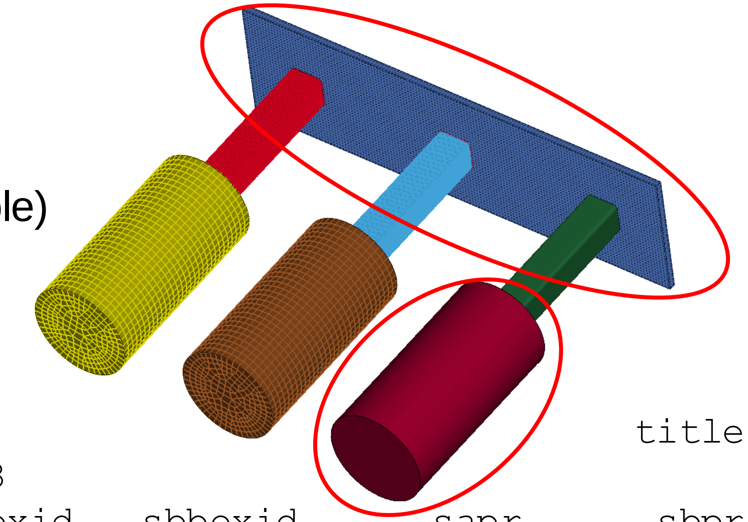


```
*SET_NODE_LIST_TITLE
Tube 1 edge
$#      sid
          1
$#     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
      64143     64042     63941     63840     63739     63638     63537     63436
...
```

Contacts - Sliding

Surface to Surface

- There are many different contacts in LS-DYNA, well go through the basics of some common types
  - Huge number of inputs on *CONTACT, however, in most cases most of the fields are left to their defaults.
  - There are 3 mandatory cards, see below
- Contacts for explicit are generally on penalty formulation, meaning springs are introduced to mitigate penetration.
- For the crash box we have defined a surface-to-surface contact using a part set of the plate (7) and part id of impactor (6)
- Other than that, the most common settings are friction, `fs` & `fd`, and damping, `vdc`
  - Common starting values: `fs` = `fd` = 0.2, and damping, `vdc` = 20 (as used in the example)



ID type of SURFB:

EQ.0: Segment set ID

EQ.1: Shell element set ID

EQ.2: Part set ID

EQ.3: Part ID

EQ.5: Include all (SURFB field is ignored).

EQ.6: Part set ID for exempted parts. All are included in the contact.

EQ.7: Branch ID; see *SET_PART_TREE.

Contact interface definition

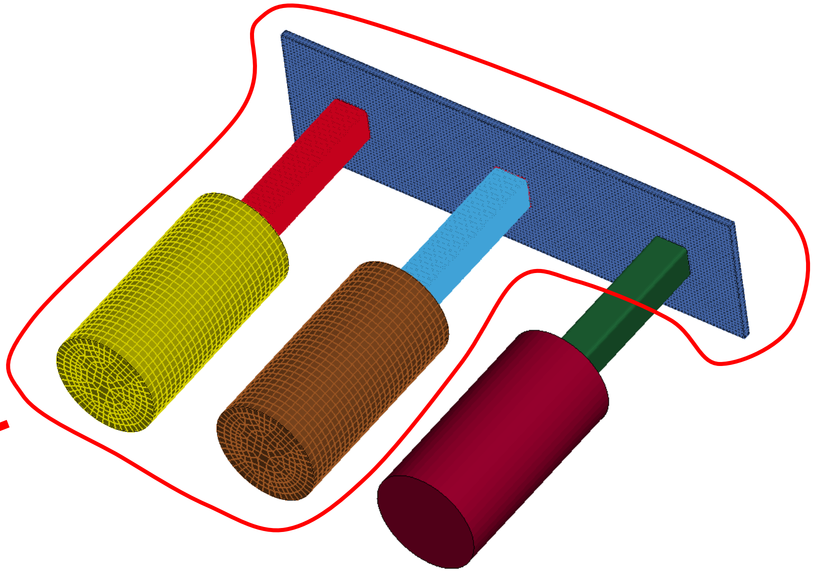
```
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_ID
$#     cid                                                                 title
         5Surface to surface contact for Tube 3
$#   surfa     surfb  surfatyp  surfbtyp   saboxid   sbboxid      sapr      sbpr
         3         4         3         2         0         0         0         0
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
       0.2       0.2       0.0       0.0      20.0         0       0.01.00000E20
$#    sfsa      sfsb      sast      sbst     sfsat     sfsbt       fsf       vsf
       1.0       1.0       0.0       0.0       1.0       1.0       1.0       1.0
```

Friction & damping

Contacts - Sliding

Single Surface

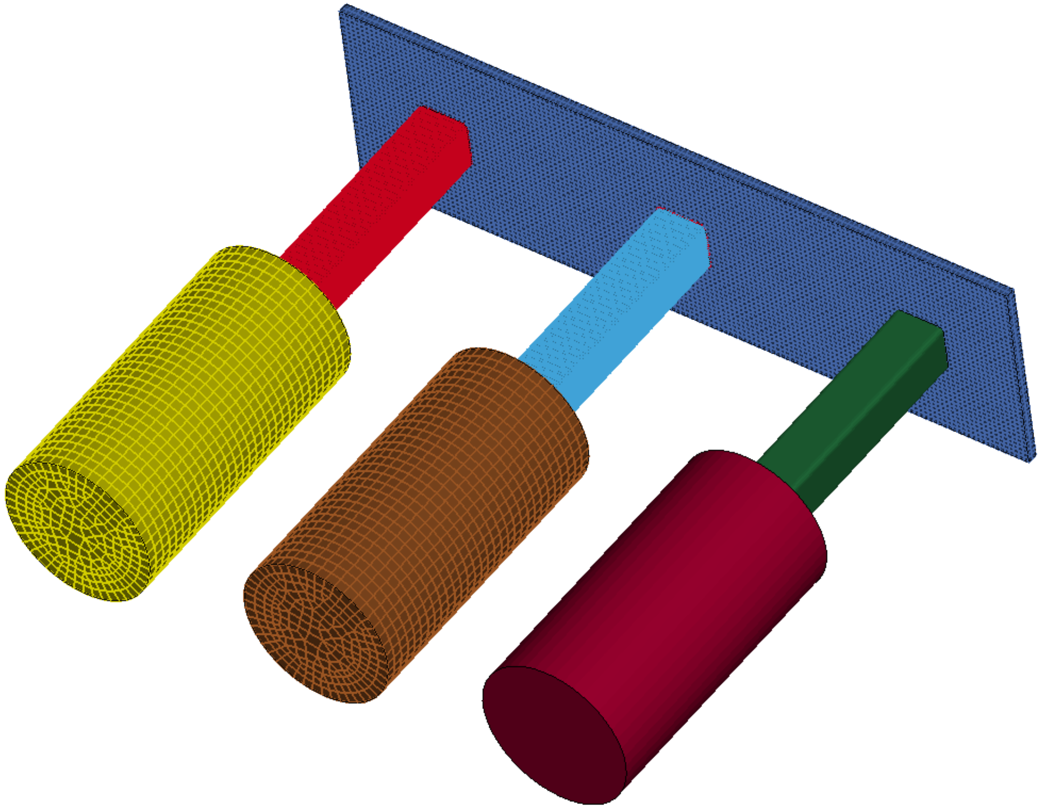
- Then there are single surface contacts
  - Offers self contact, e.g., a part contacting itself
  - Input is very similar to surface to surface
  - One only have to define one side of the contact
    - This makes it possible to gather a large number of contact entities in one contact
  - By adding *FORCE_TRANSDUCER, one for each impactor, the force between each impactor and its corresponding crash box can be measured.



```
*CONTACT_AUTOMATIC_SINGLE_SURFACE_ID
$#     cid                                                                 title
         1Single Surface Contact for Tube 1 and 2
$#   surfa     surfb  surfatyp  surfbtyp   saboxid   sbboxid      sapr      sbpr
         1         0         2         0         0         0         0         0
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
       0.2       0.2       0.0       0.0      20.0         0       0.01.00000E20
$#    sfsa      sfsb      sast      sbst     sfsat     sfsbt       fsf       vsf
       1.0       1.0       0.0       0.0       1.0       1.0       1.0       1.0
*CONTACT_FORCE_TRANSDUCER_ID
$#     cid                                                                 title
         2Impactor 4
$#   surfa     surfb  surfatyp ...
         4         0         3
...
```

Contacts

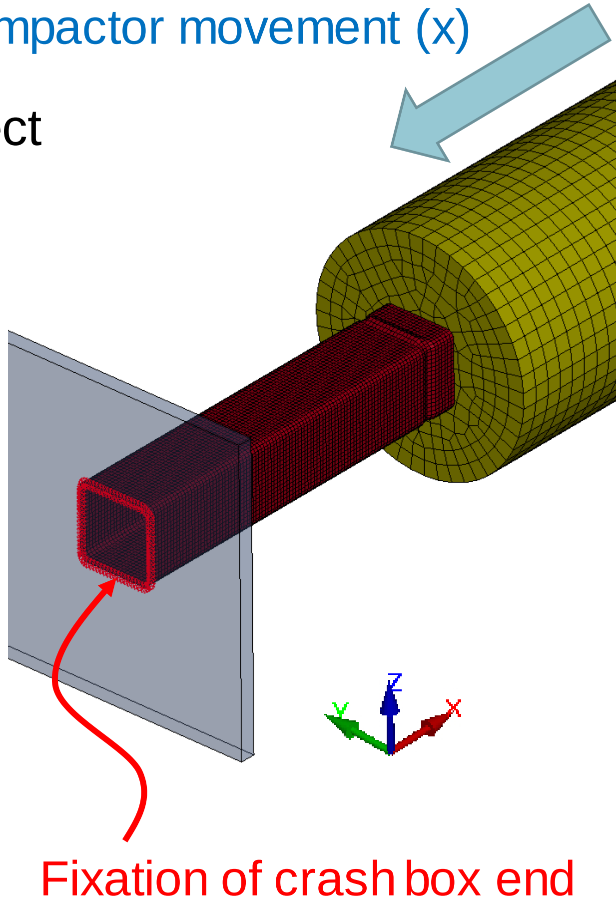
- Feels like something is missing in crash box 3, but I can’t figure out what…

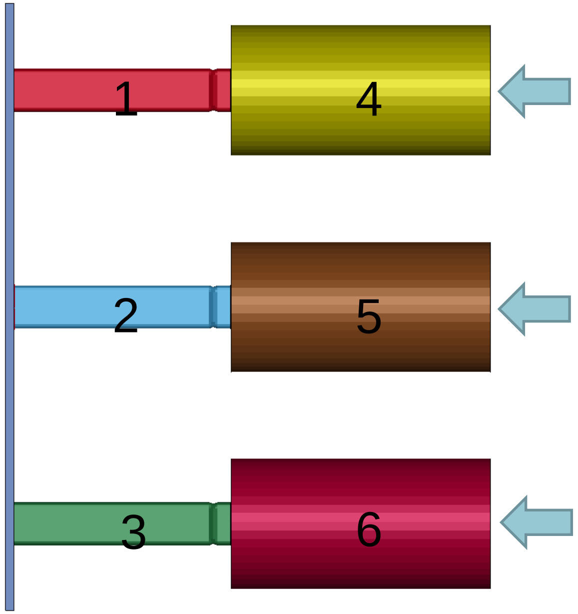


Boundary Conditions

Crash Box Model – Overview

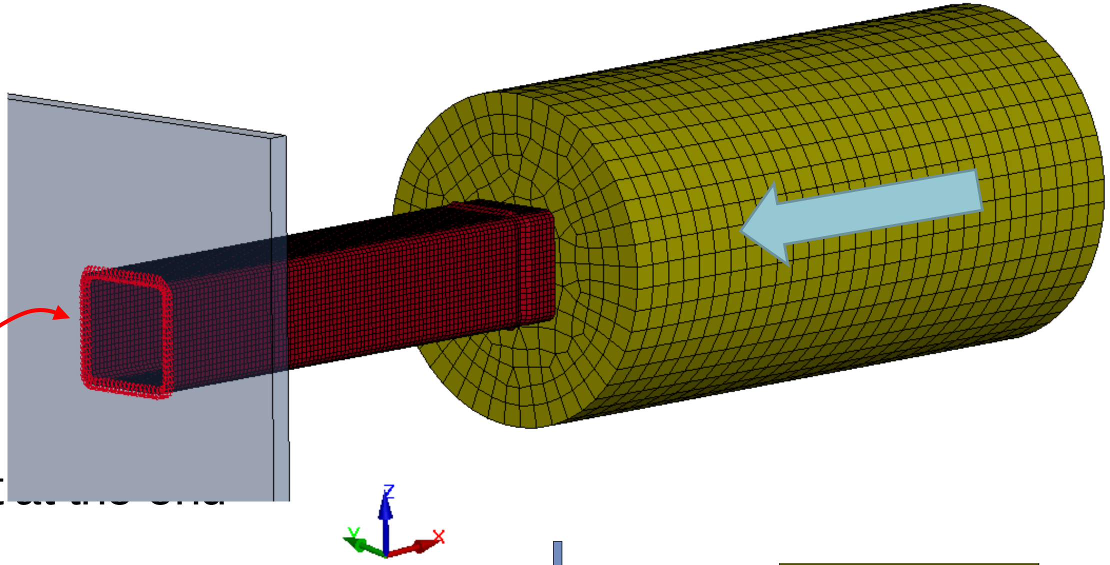
- The crash boxes and impactors are identical but subject to different boundary conditions
  - Impactor movement
  - Crash box fixation
- Recall, the rigid plate (7) is fixed and impactors (4,5,6) fixed except x-trans. through `*MAT_RIGID`
- 1 – 4
  - `*BOUNDARY_SPC`
  - `*INITIAL_VELOCITY(_RIGID)`
- 2 – 5
  - `*CONSTRAINED_NODAL_RIGID_BODY`
  - `*LOAD(_RIGID)`
- 3 – 6
  - `*CONTACT_TIED_NODES_TO_SURFACE(_OFFSET)`
  - `*BOUNDARY_PRESCRIBED_MOTION(_RIGID)`





Boundary Conditions

Crash Box Model – Crash Box 1 to Impactor 4

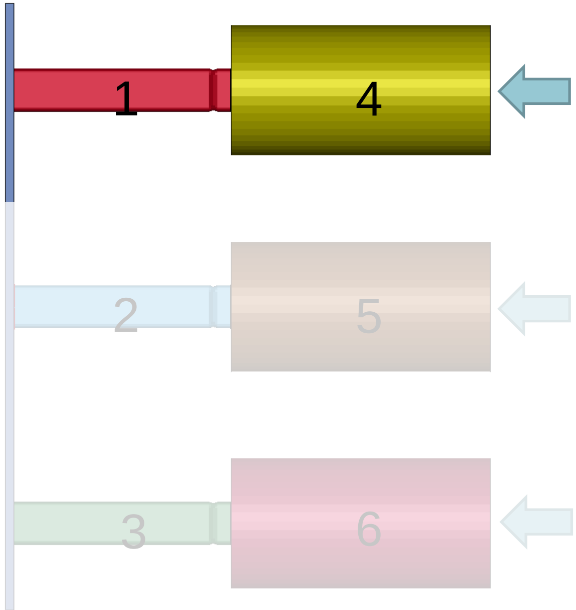


- *BOUNDARY_SPC(_SET)

```
*BOUNDARY_SPC_SET
$ NSID    CID   DOFX    DOFY    DOFZ   DOFRX    DOFRY    DOFRZ
     1             1       1       1       1        1        1
```

Translational constraints (DOFX, DOFY, DOFZ) — Rotational constraints (DOFRX, DOFRY, DOFRZ)

  - The SPC (Single Point Constraint) is applied to a node set at the end of the crash box
  - It is fixed in all DOF:s (0=free, 1=fixed)



- *INITIAL_VELOCITY(_RIGID)

```
*INITIAL_VELOCITY_RIGID_BODY
$#     pid        vx        vy        vz       vxr       vyr       vzr      icid
         4   -8.3333       0.0       0.0       0.0       0.0       0.0         0
```

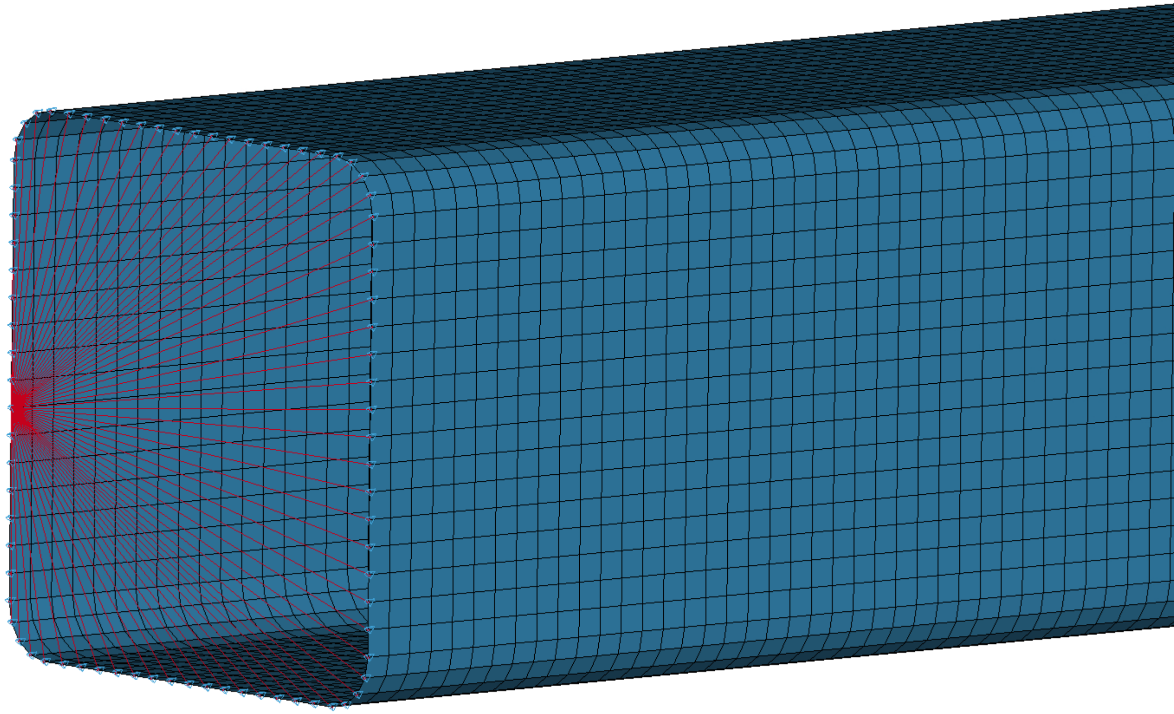
  - The impactor is given an initial velocity in negative x-direction at t = 0
  - Assigned via the pid

Boundary Conditions

Crash Box Model – Crash Box 2 to Impactor 5

- *CONSTRAINED_NODAL_RIGID_BODY

```
*CONSTRAINED_NODAL_RIGID_BODY_SPC
$#      pid         cid         nsid        pnode ...
         10           0            1            0
$#      cmo        con1         con2
        1.0           7            7
```

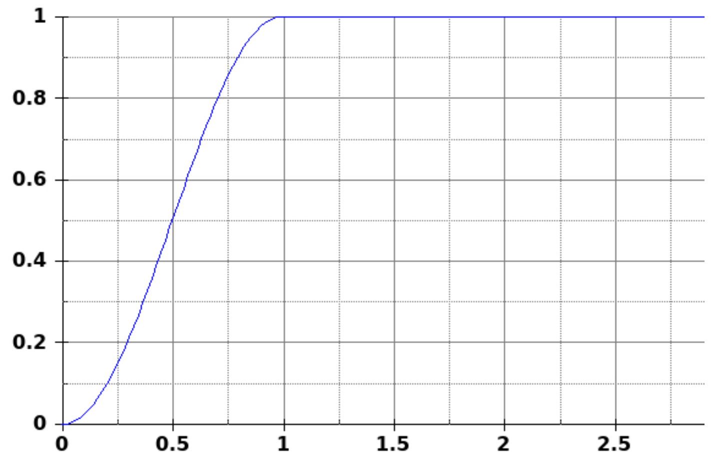


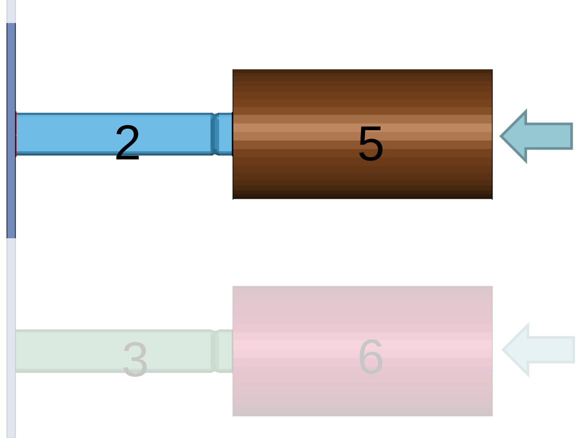
  - Makes a set of nodes a rigid body via node set
    - Can be assigned to free nodes or nodes that belong to a deformable body
    - Constraints like *MAT_RIGID
    - Assign unique pid

- *LOAD(_RIGID)

Direction

```
*LOAD_RIGID_BODY
$#      pid         dof         lcid          sf ...
          5           1            5      -150.0
```



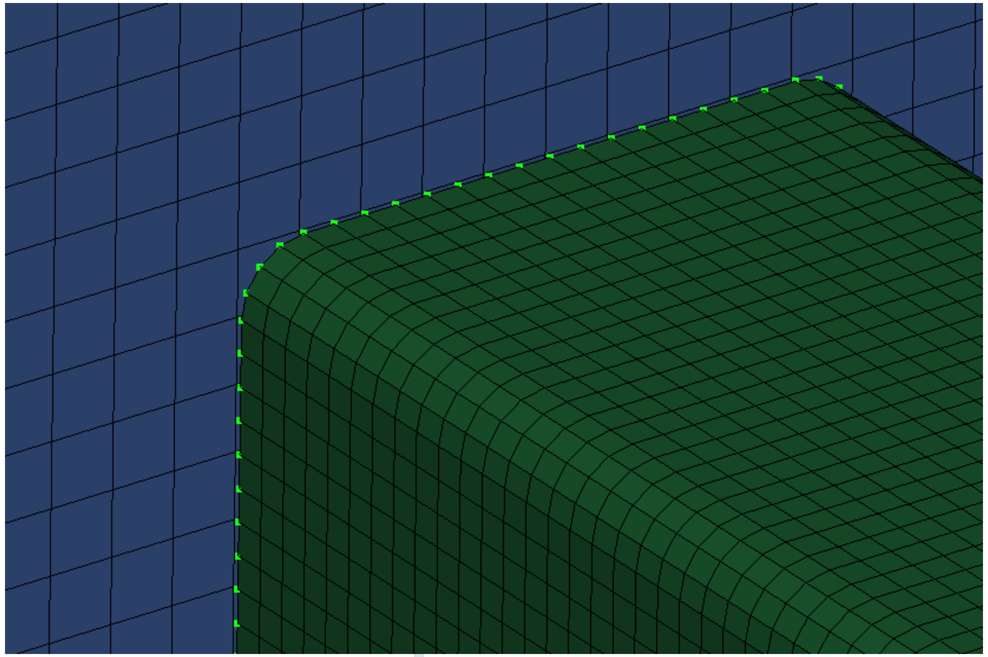


  - Try to use smooth curves
  - Applied using pid
  - Scale factor can be used (150 kN)

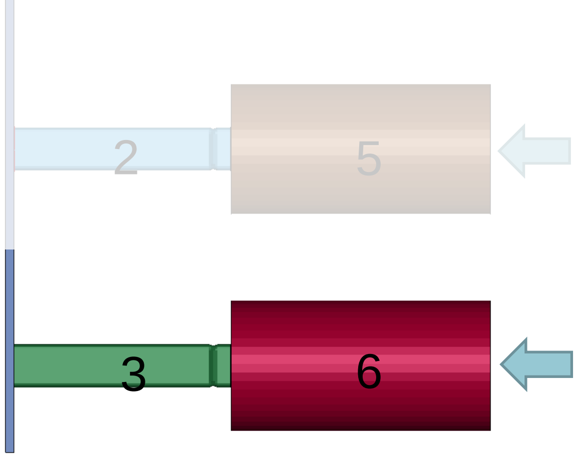
Boundary Conditions

Crash Box Model – Crash Box 3 to Impactor 6

- *CONTACT_TIED_NODES_TO_SURFACE(_OFFSET)
  - A tied contact is used to fixate the crash box end
    - Assigned via node (box edge nodes) set to part (plate 7)
  - Tied contacts are generally constraints
    - As the plate is a rigid body, we must add the _OFFSET option to invoke penalty formulation
  - There are many different tied contacts for different purposes
    - For example, some transfer rotation and moment while others don't; therefore, it is important consider what contact situation you have.



```
*CONTACT_TIED_NODES_TO_SURFACE_OFFSET_ID
$#     cid
         6end of Tube 3 tied to rigid plate
$#   surfa     surfb  surfatyp  surfbtyp   saboxid   sbboxid      sapr      sbpr
         3         7         4         3         0         0         0         0
$#      fs        fd        dc        vc       vdc    penchk        bt        dt
       0.0       0.0       0.0       0.0       0.0         0       0.01.00000E20
$#    sfsa      sfsb      sast      sbst     sfsat     sfsbt       fsf       vsf
       1.0       1.0       0.0       0.0       1.0       1.0       1.0       1.0
```

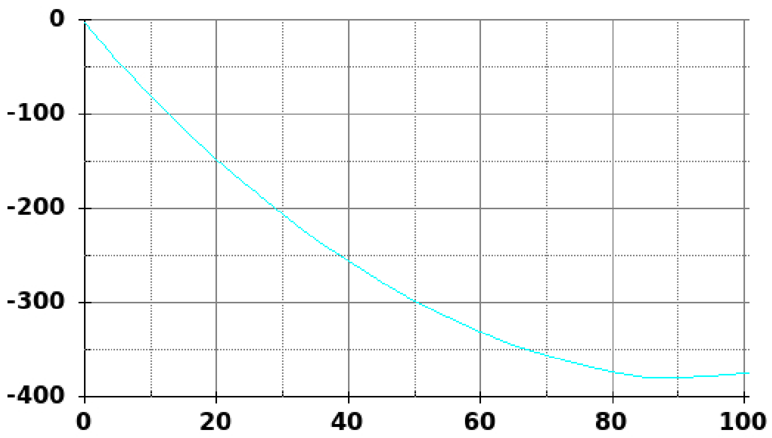


- *BOUNDARY_PRESCRIBED_MOTION(_RIGID)
  - Impactor movement is displacement controlled

Learn more! : Client Area > 2_webinars > 1_Explicit_and_General > Tied_and_Tiebreak_Contacts

Boundary Conditions

Crash Box Model – Crash Box 3 to Impactor 6



- *CONTACT_TIED_NODES_TO_SURFACE(_OFFSET)
  - A tied contact is used to fixate the crash box end
- *BOUNDARY_PRESCRIBED_MOTION(_RIGID)
  - Impactor movement is displacement controlled

Degree of freedom, 1-8 available for rigid bodies

Vel/acc/disp flag

Curve/function ID

Scale factor

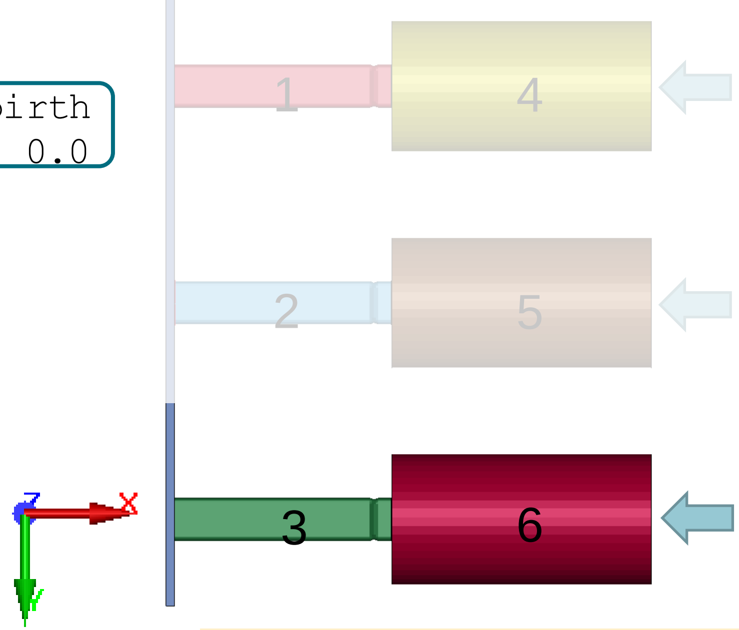
Birth and death of motion

```
*BOUNDARY_PRESCRIBED_MOTION_RIGID
$#   pid     dof     vad    lcid      sf     vid      death      birth
       6       1       2       6     1.0       01.00000E28        0.0
```

Part ID

1 = x-direction	0 = velocity
2 = y-direction	1 = acceleration
3 = z-direction	2 = displacement

Vector for arbitrary orientation (dof 4 & 8)

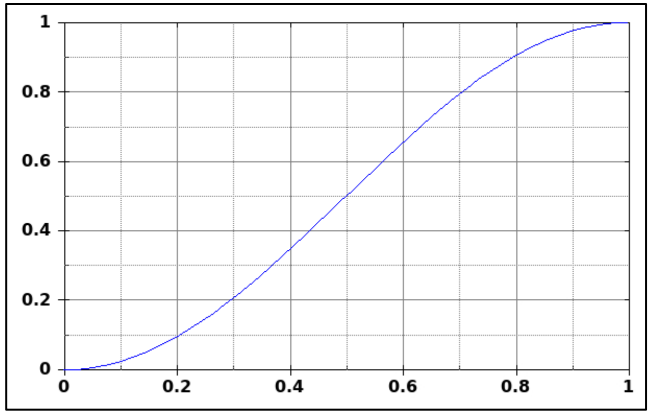


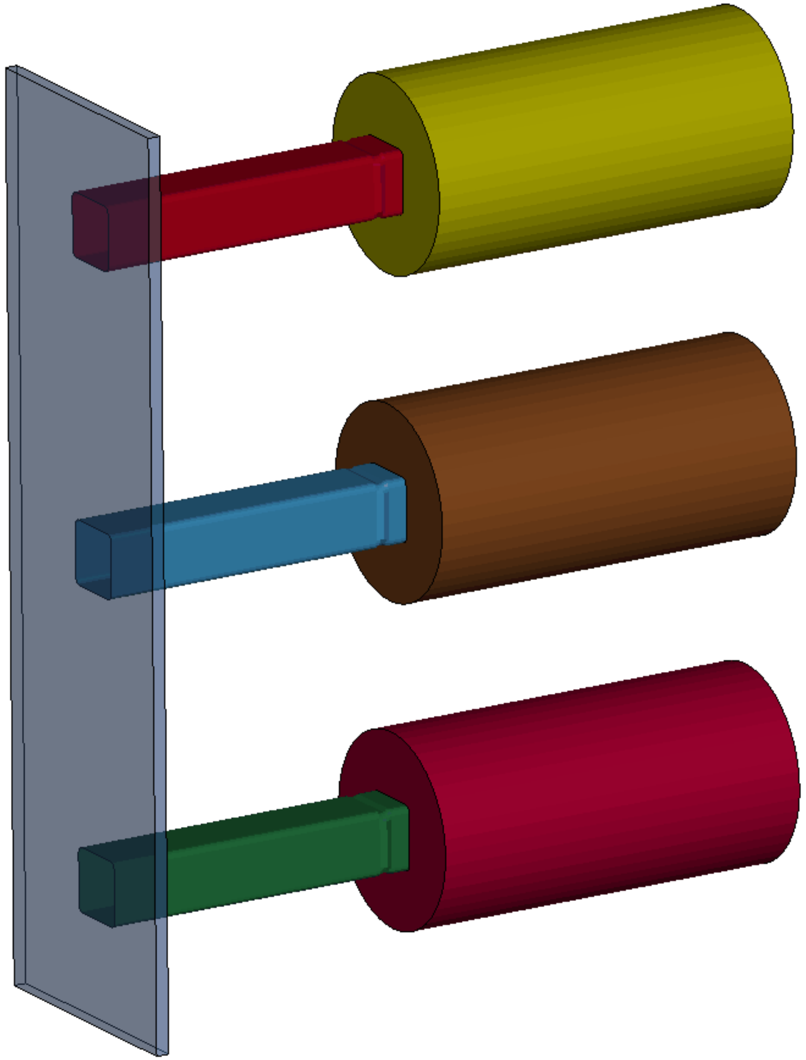
- Learn more from our webinars or short videos
  - Client Area > 2_webinars > 1_Explicit_and_General > LS-DYNA_Rigid_Bodies
  - Introduction to Rigid Body Materials in LS-DYNA — Dynamore Nordic AB
  - Introduction to Rigid Body Joints in LS-DYNA — Dynamore Nordic AB

Boundary Conditions

Results Sneak peak

- General comments
  - Use prescribed motions instead of forces when possible
  - Use a smooth load curve
  - Consider inertial effects (loading rate)
  - Can’t combine rigid bodies with (real) constraints

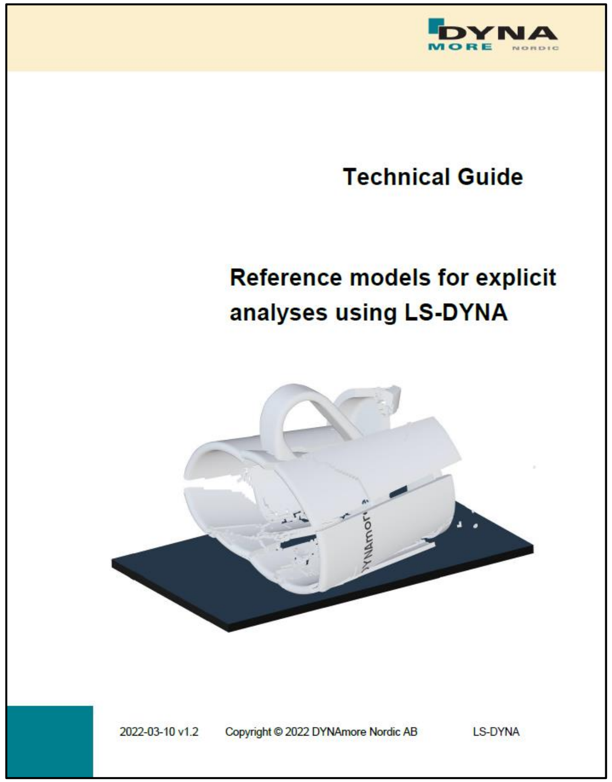




Control & Output

*CONTROL Cards

- Read and start from our recommended setting
  - Client Area > 1_LS-DYNA_Guidelines > 1_LS-DYNA_Explicit_Guidelines




```
*CONTROL_ACCURACY
$#     osu       inn    pidosu      iacc
         1         4         0         0
$ Note: PIDOSU can be used to restrict OSU to a part set.
$ Read the manual for details.
*CONTROL_BULK_VISCOSITY
$#      q1        q2      type     btype    tstype
       1.5      0.06        -2         0         0
*CONTROL_CONTACT
$#  slsfac    rwpnal    islchk    shlthk    penopt    thkchg     orien    enmass
       0.1       1.0         2         0         1         1         1         0
$#  usrstr    usrfrc     nsbcs    interm     xpene     ssthk      ecdt   tiedprj
         0         0         0         0       4.0         1         0         0
$#   sfric     dfric       edc       vfc        th     th_sf    pen_sf
       0.0       0.0       0.0       0.0       0.0       0.0       0.0
$#  ignore    frceng   skiprwg    outseg   spotstp   spotdel   spothin
         0         1         0         0         0         1       0.0
$#    isym    nserod    rwgaps    rwgdth     rwksf      icov    swradf    ithoff
         0         0         1       0.0       1.0         0       0.0         0
$#  shledg    pstiff    ithcnt    tdcnof     ftall    unused    shltrw    igactc
         0         0         0         0         1                 0.0         0
*CONTROL_ENERGY
$#    hgen      rwen    slnten     rylen     irgen     maten
         2         2         2         2         2         1
*CONTROL_MPP_DECOMPOSITION_AUTOMATIC
*CONTROL_MPP_IO_LSTC_REDUCE
*CONTROL_MPP_IO_NODUMP
*CONTROL_OUTPUT
$#   npopt    neecho    nrefup    iaccop     opifs    ipnint    ikedit    iflush
         1         3         1         0       0.0         0       100      5000
$#   iprtf    ierode   tet10s8    msgmax    ipcurv      gmdt   ip1dblt      eocs
         0         1         2        50         0       0.0         0         0
$#   tolev    newleg    frfreq     minfo    solsig    msgflg    cdetol
         2         0         1         0         0         0      10.0
$# phschng    demden   icrfile   spc2bnd    penout    shlsig   hisnout    engout
         0         0         0         0         0         0         1         0
$#    insf    isolsf      ibsf      issf    mlkbag
         0         0         0         0         0
*CONTROL_SHELL
$#  wrpang     esort     irnxx    istupd    theory       bwc     miter      proj
      30.0         1        -1         4         1         2         1         1
$# rotascl    intgrd    lamsht    cstyp6    thshel
       1.0         0         0         1         0
$# psstupd   sidt4tu     cntco    itsflg    irquad    w-mode   stretch      icrq
         0         0         1         0         2       0.0       0.0         0
$#  nfail1    nfail4   psnfail    keepcs     delfr   drcpsid    drcprm   intperr
         0         1         0         0         0         0       1.0         0
*CONTROL_SOLID
$#   esort   fmatrix   niptets    swlocl    psfail   t10jtol      icoh    tet13k
         1         0         4         1         0       0.0         0         0
$#   pm1     pm2     pm3     pm4     pm5     pm6     pm7     pm8     pm9    pm10
       0       0       0       0       0       0       0       0       0       0
$#  tet13v
```

- Common *CONTROL cards
  - *CONTROL_ACCURACY
    - INN = 4 Invariant node numbering
  - *CONTROL_BULK_VISCOCITY
  - *CONTROL_CONTACT
  - *CONTROL_ENERGY
  - *CONTROL_OUTPUT
  - *CONTROL_SHELL/SOLID
    - ESORT = 1 (mixed meshes, quad & tria / hex & penta)
  - *CONTROL_SOLUTION
- In main.k
  - *CONTROL_TERMINATION
    - Define simulation termination time
  - *CONTROL_TIMESTEP

Control & Output

Explicit Time Control

- LS-Dyna will automatically identify the minimum time step and use this for the simulation
- The 100 smallest timesteps with corresponding element can be found in the ASCII file d3hsp
- A safety factor is used for the critical time step. The safety factor can be changed in `*CONTROL_TIMESTEP (TSSFAC)`

$$\Delta t = 0.9\Delta t_{\text{critical}}$$ Default safety factor

$$\Delta t = 0.67\Delta t_{\text{critical}}$$ Recommended for high explosives

The simulation time step changes due to deformed elements

```
    1 t 0.0000E+00 dt 5.52E-07
    1 t 0.0000E+00 dt 5.52E-07
 1876 t 9.9962E-04 dt 5.04E-07
 3920 t 1.9999E-03 dt 4.84E-07
 5000 t 2.5189E-03 dt 4.80E-07
 6046 t 2.9996E-03 dt 4.35E-07
 8630 t 3.9997E-03 dt 3.52E-07
10000 t 4.4709E-03 dt 3.39E-07
11540 t 4.9999E-03 dt 3.52E-07
14428 t 5.9998E-03 dt 3.38E-07
15000 t 6.1934E-03 dt 3.39E-07
17317 t 6.9998E-03 dt 3.55E-07
20000 t 7.9439E-03 dt 3.45E-07
```

Control & Output

Explicit Time Control – Time & Mass Scaling

- Termination time is set on `*CONTROL_TERMINATION` (`endtime`)
- Mass scaling can be used to increase the time step

$$\Delta t_{\text{critical}} = \frac{l_e}{c_e} \qquad c_e \sim \sqrt{\frac{E_e}{\rho_e}} \qquad \rho_e \uparrow \;\Longrightarrow\; \Delta t_{\text{critical}} \uparrow$$

- The mass scaling is activated in `*CONTROL_TIMESTEP`
- Set the time step in `DT2MS`. Use a negative value to only scale the elements that needs mass scaling

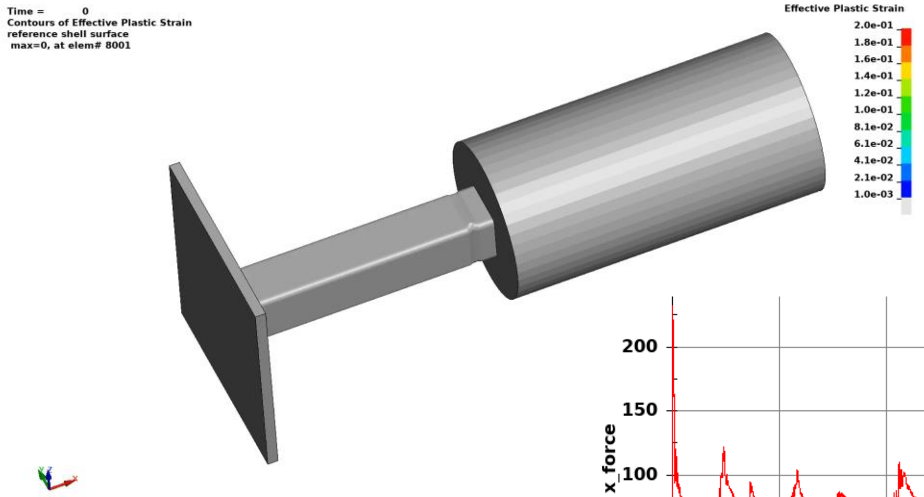
```
*CONTROL_TERMINATION
$#  endtim    endcyc     dtmin    endeng    endmas     nosol
     100.0         0       0.0       0.01.000000E8         0
*CONTROL_TIMESTEP
$#  dtinit    tssfac      isdo    tslimt     dt2ms      lctm     erode     ms1st
       0.0       0.9         0       0.0   -1.0E-3         0         0         0
$#  dt2msf   dt2mslc     imscl    unused    unused     rmscl    unused      ihdo
       0.0         0         0                           0.0                   1
```

Webinar - Client Area > 2_webinars > 1_Explicit_and_General > Masscaling_and_Subcycling

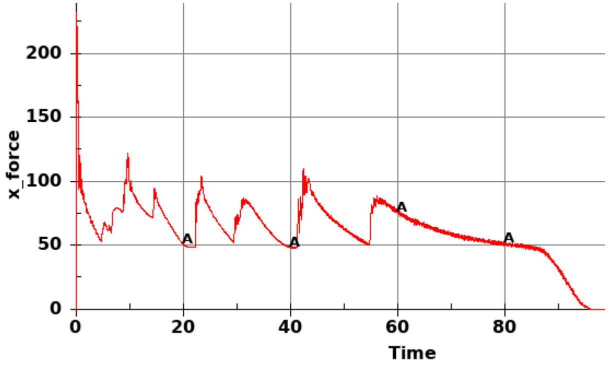
Control & Output

D3PLOT & BINOUT

- D3PLOT – 3D data
  - View displacements, strains, stress, etc.



binout0000
binout0001



- BINOUT – Scalar data
  - GLSTAT (global data)
  - MATSUM (material energies)
  - RCFORC (resultant interface/contact forces)
  - SLEOUT (sliding interface energy)
  - NODOUT (nodal point data)
  - ELOUT (element data)
  - SECFORC (cross section forces)
  - SPCFORC (SPC reaction forces)
  - BNDOUT (boundary condition force/ energy)
  - RBDOUT (rigid body data)
  - and many more…

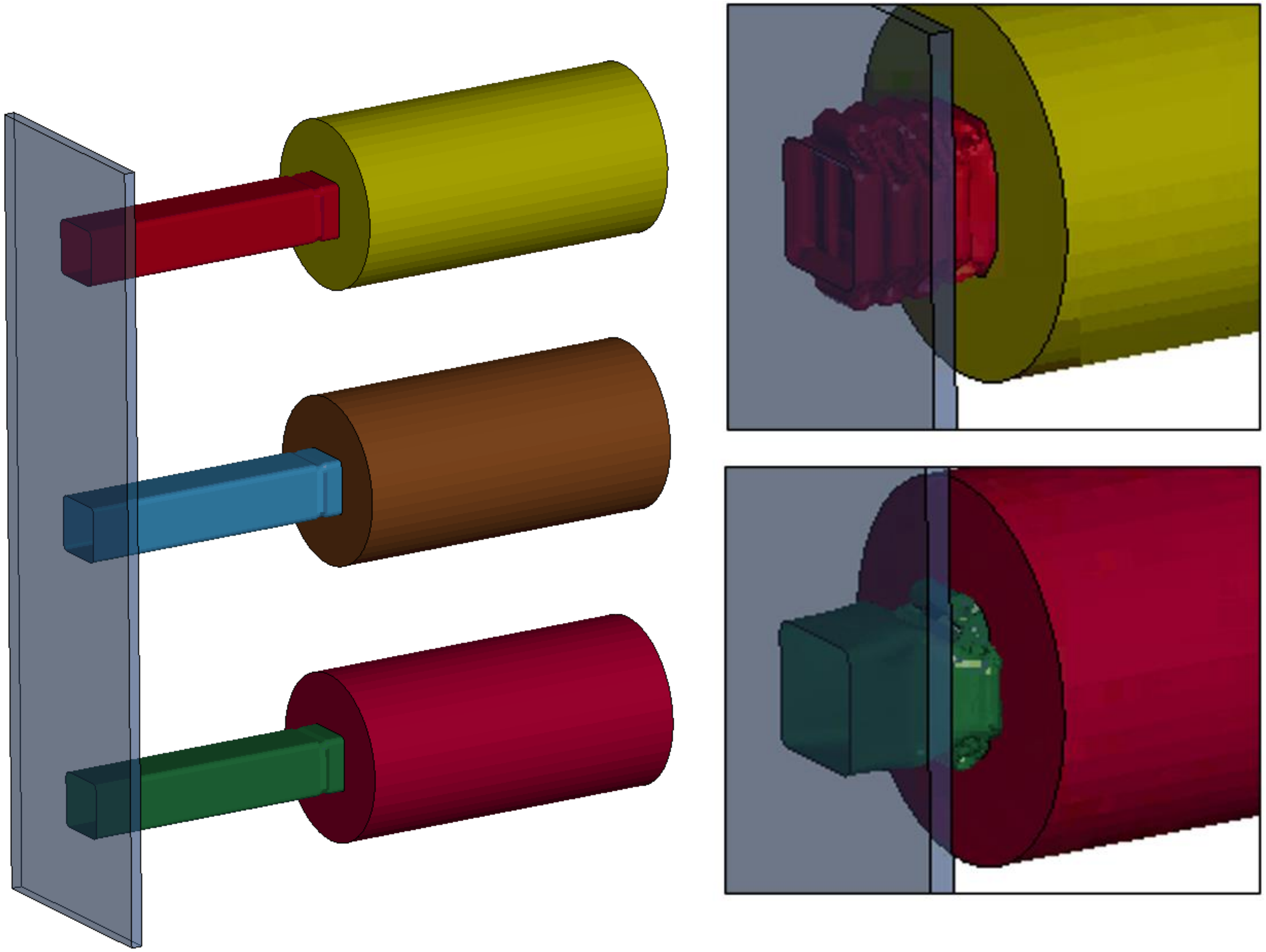
```
*DATABASE_BINARY_D3PLOT
$#        dt      lcdt      beam     npltc    psetid
         4.0         0         0         0         0
$#     ioopt
           0
```

```
*DATABASE_GLSTAT
$#        dt    binary      lcur     ioopt
        0.01         0         0         1
…
```

Frequency of output
(output time step)

Results

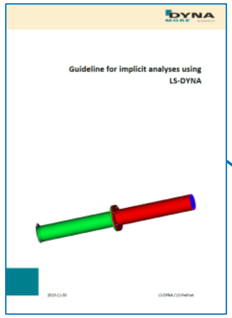
Open Model

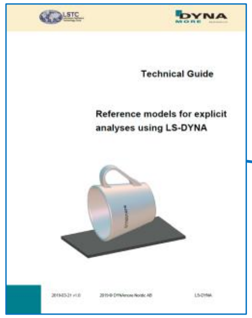


Limited creasing in crash box 3 (green) due to missing self contact

Final notes - Suggested Modelling Structure

*INCLUDE





Main.k

```
*INCLUDE  *CONTACT
*LOAD     *BOUNDARY
*INITIAL  *SET_…_ADD
…
```

Control_cards.k

```
*CONTROL_ACCURACY
*CONTROL_BULK_VISCOSITY
*CONTROL_CONTACT
*CONTROL_ENERGY
*CONTROL_HOURGLASS
*CONTROL_OUTPUT
*CONTROL_SHELL
*CONTROL_SOLID
*CONTROL_SOLUTION
…
```

Database_cards.k

```
*DATABASE_…
*DATABASE_BINARY_D3PLOT
*DATABASE_BINARY_INTFOR
*DATABASE_EXTENT_BINARY
*DATABASE_HISTORY…
…
```

Often Combined

Materials.k

```
*MAT_020/RIGID
*MAT_024/PIECEWISE…
…
```

Geometry.k

```
*NODE
*ELEMENT…
*PART
*SECTION…
*SET_NODE
*SET_ELEMENT
*SET_PART
*DATABASE_HISTORY_NODE
```

Rarely changed ⟹

- Few errors are introduced.
- All normal results are always available.
- Prepare settings for different types of analyses.

- Make use of `*INCLUDE` !
- `*INCLUDE_TRANSFORM`
  - Use multiple instances of the same part, e.g., bolts
  - Unit conversion, e.g., for including material database…

Final Notes

- Explicit LS-DYNA is usually run using single precision
  - For long time events double precision may be required to mitigate accuilative errors
  - Rigid body rotations with small increments may be require double precision

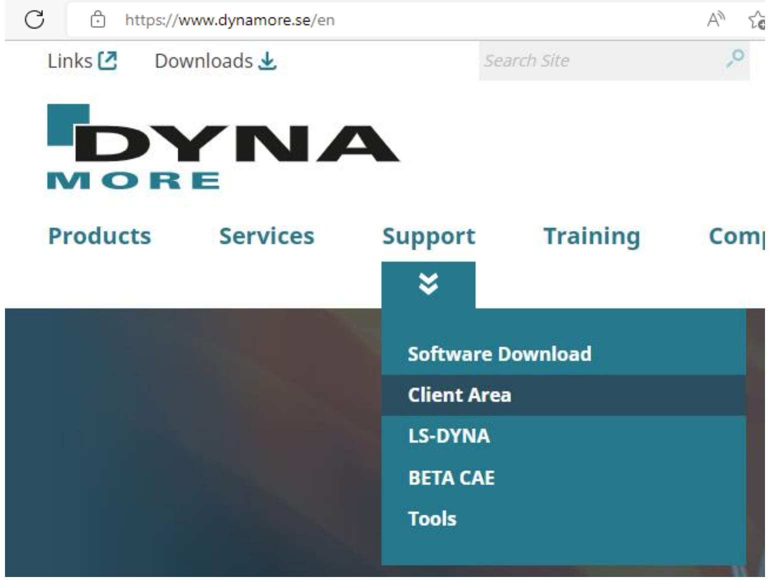
- Some things that didn't make it into the webinar but can be good to know about
  - Damping – Remove unwanted oscillations or introduce energy dissipation
    - Short video - Damping in LS-DYNA — Dynamore Nordic AB
    - Webinar - Client Area > 2_webinars > 1_Explicit_and_General > Damping_in_LS-DYNA
  - Preload/pre-tensioning
    - Webinar - Client Area > 2_webinars > 1_Explicit_and_General > Bolt_Pretension
    - Webinar - Client Area > 2_webinars > 1_Explicit_and_General > Pretension_and_Dynamic_Relaxation
    - Short video - Initializing models with Dynamic Relaxation — Dynamore Nordic AB
  - Time scaling for quasi-static simulation

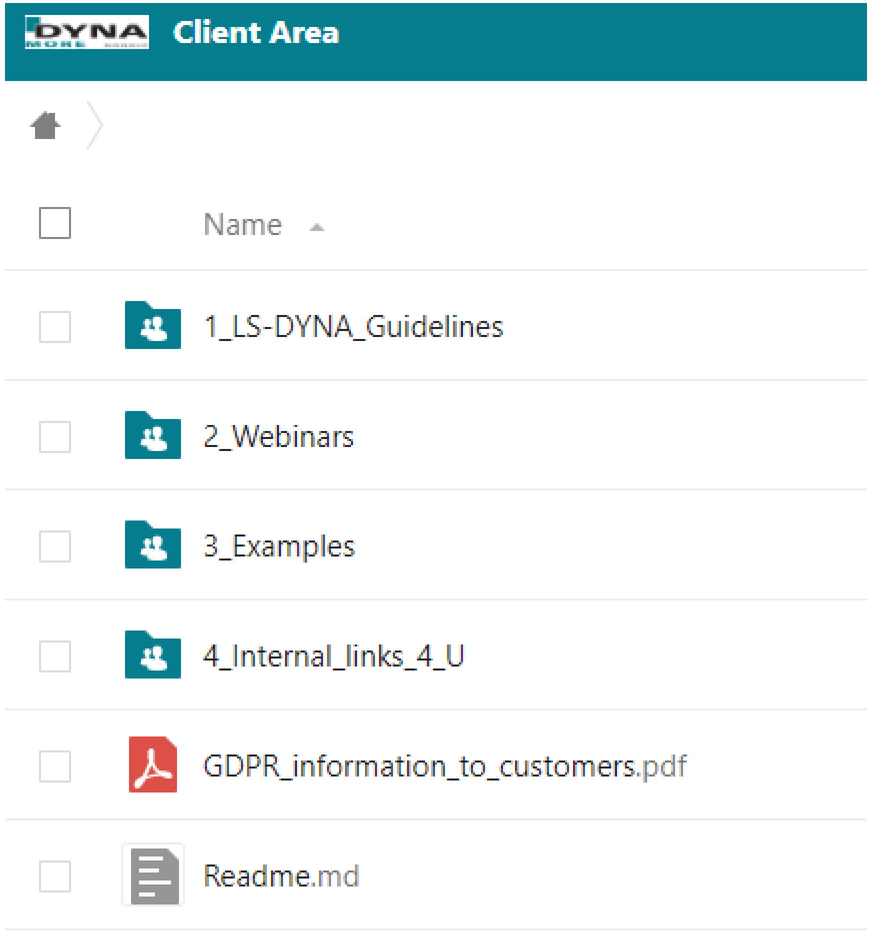
- For more in-depth recommendations about specific settings for certain applications in explicit LS-DYNA we urge you to read our Technical guide for Explicit, "Reference models for explicit analyses using LS-DYNA", that you find on our client area!

Final Notes – Training Resources

Client Area

- Comprehensive training and guideline material for LS-DYNA
- <u>All our customers have access</u>
- Password: *Contact support@dynamore.se*
  - NOTE! Password is changed each 6 months. Contact support again for the new password.
- Use link files.dynamore.se/shares/client_area, or find it via our webpage

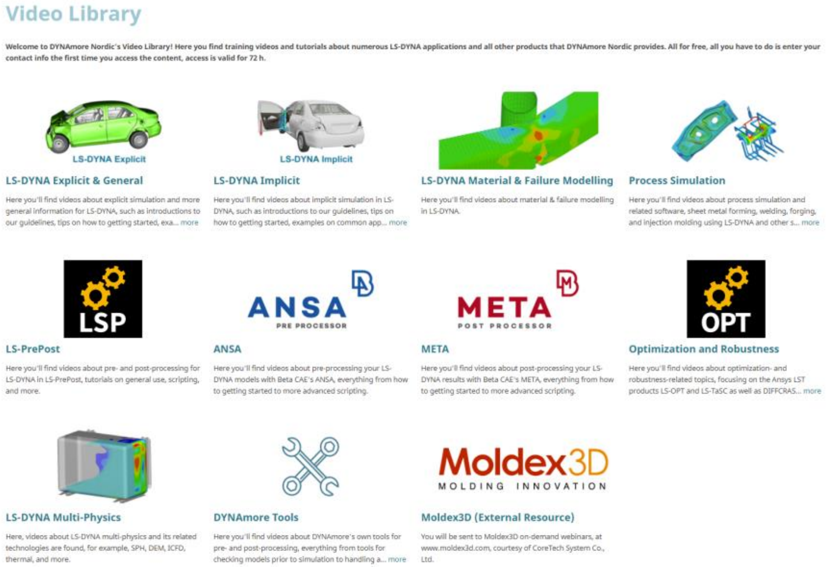


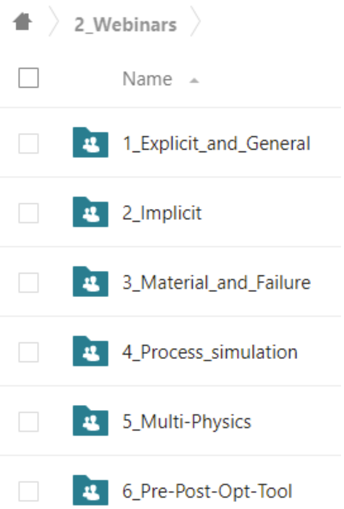


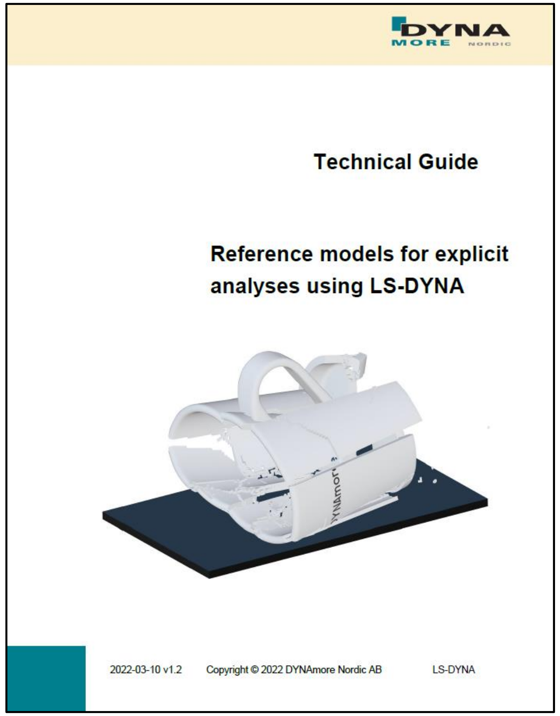
Final Notes – Training Resources

Summary

- **Courses** at Seminars — Dynamore Nordic AB
- **Short videos** on www.dynamore.se/en/training/video-library for inspiration and overview
- **Client Area**, files.dynamore.se/shares/client_area, guidelines, webinars, and examples.
  - Explicit Guideline: Client Area > 1_LS-DYNA_Guidelines > 1_LS-DYNA_Explicit_Guidelines
- www.dynasupport.com, a site where you will find answers to many questions related to LS-DYNA.
- Explicit Time Control
  - Client Area > 2_webinars > 1_Explicit_and_General > Masscaling_and_Subcycling
- General
  - Client Area > 2_webinars > 1_Explicit_and_General > Model_Checking
- Damping
  - Short video - Damping in LS-DYNA — Dynamore Nordic AB
  - Client Area > 2_webinars > 1_Explicit_and_General > Damping_in_LS-DYNA
- Rigid bodies
  - Short video - Introduction to Rigid Body Materials in LS-DYNA — Dynamore Nordic AB
  - Short video - Introduction to Rigid Body Joints in LS-DYNA — Dynamore Nordic AB
  - Client Area > 2_webinars > 1_Explicit_and_General > LS-DYNA_Rigid_Bodies
- Preloading
  - Client Area > 2_webinars > 1_Explicit_and_General > Bolt_Pretension
  - Client Area > 2_webinars > 1_Explicit_and_General > Pretension_and_Dynamic_Relaxation
  - Short video - Initializing models with Dynamic Relaxation — Dynamore Nordic AB
- Material
  - Short video - Material hardening — Dynamore Nordic AB
  - Client Area > 2_webinars > 3_Material_and_Failure
    - Webinars on numerous material types and phenomena; failure, plastics, rubber, composites, and more.
- Contacts
  - Client Area > 2_webinars > 1_Explicit_and_General > Tied_and_Tiebreak_Contacts
  - Client Area > 2_webinars > 1_Explicit_and_General > Mortar_In_Explicit_Simulation

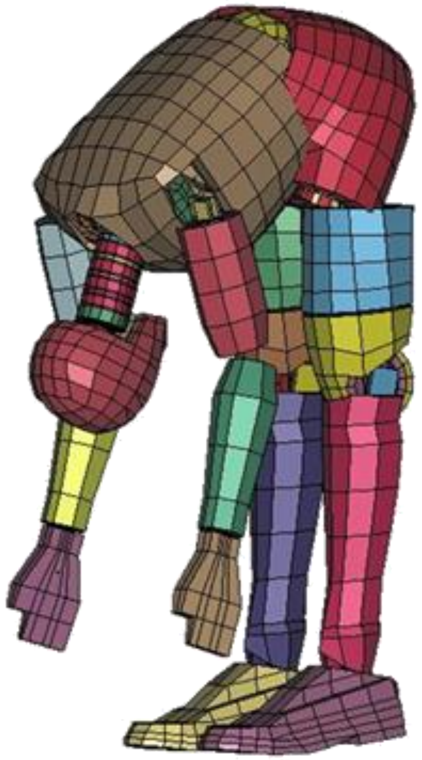







Register Today!
DYNA MORE NORDIC
Nordic LS-DYNA Users' Conference
2022 October 18-19 Gothenburg

Thank You



DYNAMore Nordic AB
Brigadgatan 5
587 58 Linköping, Sweden

Tel.: +46 - (0)13 23 66 80
info@dynamore.se

www.dynamore.se
www.dynaexamples.com
www.dynasupport.com
www.dynalook.com

Find us on

DYNAmore worldwide
Germany ▪ France ▪ Italy ▪ Sweden ▪ Switzerland ▪ USA