LS-DYNA UPDATE

Oasys LS-DYNA Update Meeting

Arup Campus

Solihull

16th January 2008

- Versions of LS-DYNA
- LS-971_R3 Updates
- LS-971_R4 Updates
- LS-980 Updates

VERSIONS OF LS-DYNA

- **LS-971**
  - Release 2 – 7600.1224 Current release version.
  - Release 3 – To be released Q2-2008.
  - Release 4 – To be released Q4-2008 / 2009
    - A new build is being developed of the single precision version in which the nodal coordinates, displacements and rotations are stored and calculated in double precision.

- **LS-980**
  - Beta releases available now
  - Full release 2009 / 2010

LS-971_R3

- Version is compatible with new LS-971 printed manual.
- Code was frozen to new features in October 2007, and currently undergoing debugging and testing.
- All new features are being added to LS971_R4

*CONTROL_CONTACT

- Parameters **SPOTSTP** and **SPOTDEL** now apply to both beam and solid element spot welds.
  - **SPOTSTP** has a new option, 2. Action to take if a spotweld node or face cannot be found on the master surface of the tied contact.
    - 0 Continue
    - 1 Terminate
    - **2 Delete the weld and continue the calculation**
  - **SPOTDEL.** Solid elements are now deleted when elements constraining the nodes on either the upper or lower contact surface are deleted

*CONTROL_SHELL

- Automatic sorting of degenerate quadrilateral shells to treat them as triangular shells is extended. The **ESORT** flag now permits 3 values:
  - EQ.0: no sorting required
  - EQ.1: full sorting to C0 triangular shells
  - EQ.2: full sorting to DKT triangular shells
- **PSNFAIL**, Optional shell part set ID specifying which part ID's are checked for negative Jacobians. Works with NFAIL1 and NFAIL4 parameters.
  - If zero, all shell part ID's are included.
- **|PSSTUPD|** is the optional shell part set ID specifying which part ID's have or do not have their thickness updated.
  - LT.0: shells in part set are excluded from updates
  - EQ.0: all shells have their thickness updated
  - GT.0: shell in part set are included in updates

*DATABASE_MATSUM

- Output eroded internal and kinetic energy into the MATSUM file by part ID.
- Output the kinetic energy from the added mass under part ID 0, which includes mass defined under *ELEMENT_MASS
- **In the R3 release nonstructural mass distributions defined in *SECTION_SHELL and *ELEMENT_ MASS_PART are now added to their respective part ID's.**

*DATABASE_HISTORY_{OPTION}

New options to limit the output to the DEFORC and SBTOUT files.

To limit output for the DEFORC file:

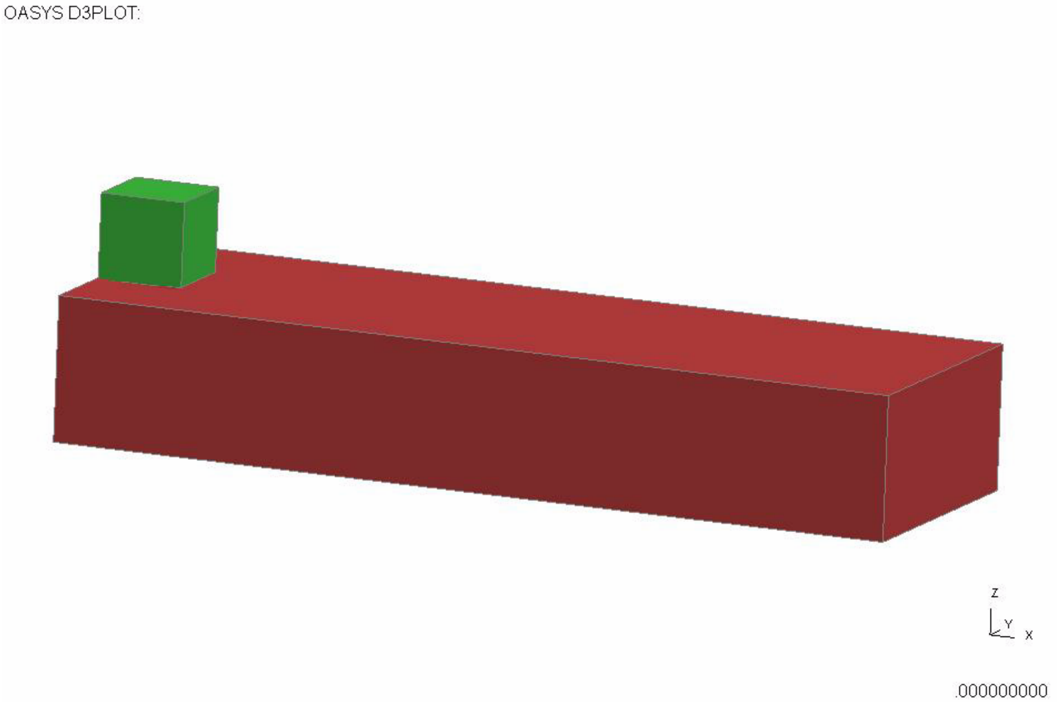
- DISCRETE
- DISCRETE_ID
- DISCRETE_SET

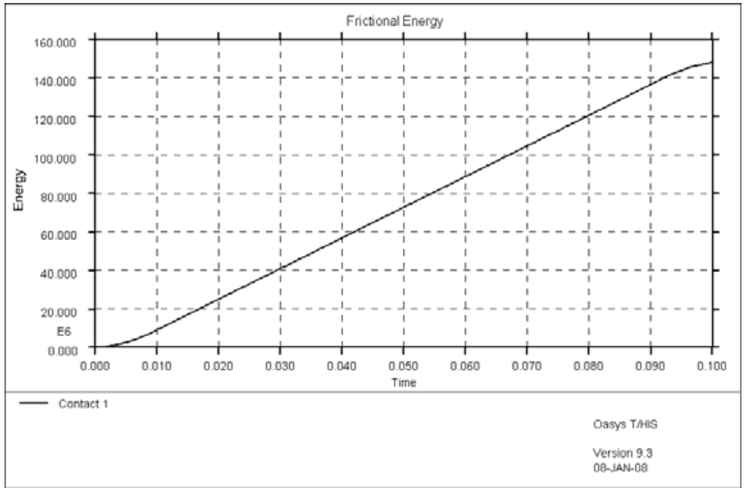
To limit output for the SBTOUT file:

- SEATBELT
- SEATBELT_ID

*DATABASE_SLEOUT

Frictional energy dissipation is now computed and output for each contact interface.





*INTEGRATION_BEAM

- Material definition can now change from integration point to integration point. Note the material model must be that same at all integration points defined in the section.

*MAT_ADD_THERMAL_EXPANSION

- Allows thermal expansion to be added to any part defined in the keyword file.
  - Thermal expansion can be defined as a function of temperature

*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK

- New options for this contact
  - Type 7 – Dycoss Discrete crack model
  - Type 8 – Offset option of type 6
  - Type 9 – Extension of option 7, Discrete crack model with power law and B-K damage.
  - Type 10 – Offset option of type 7 (Not in manual)
  - Type 11 – Offset option of type 9 (Not in manual)

Dycoss Discrete Crack Model

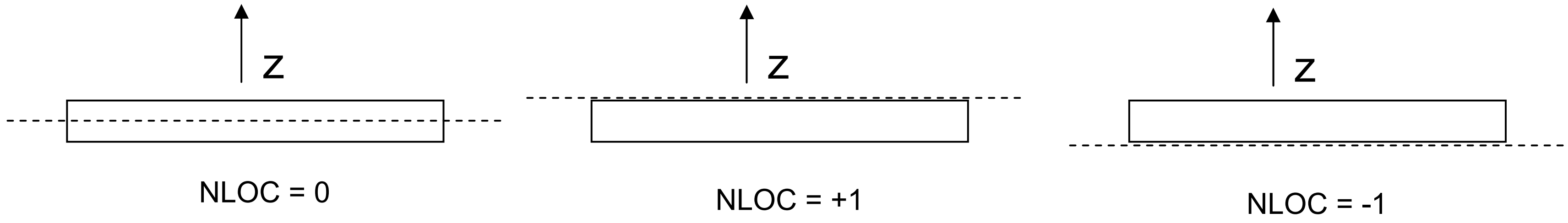
- Developed jointly by US Navy and Royal Netherlands Navy
- Initially bonded interface
- Post-failure: contact with friction and gap opening
- Failure Model
  - Elastic - brittle damage (one damage parameter)
  - Normal and shear failure stresses
  - Softening given by critical energy release rates
  - Friction angle adds strength for compressive normal stress

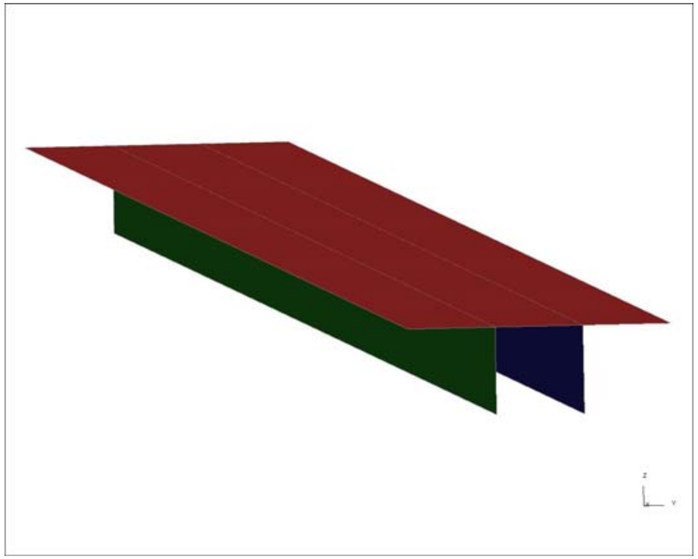
*ELEMENT_SHELL_OFFSET

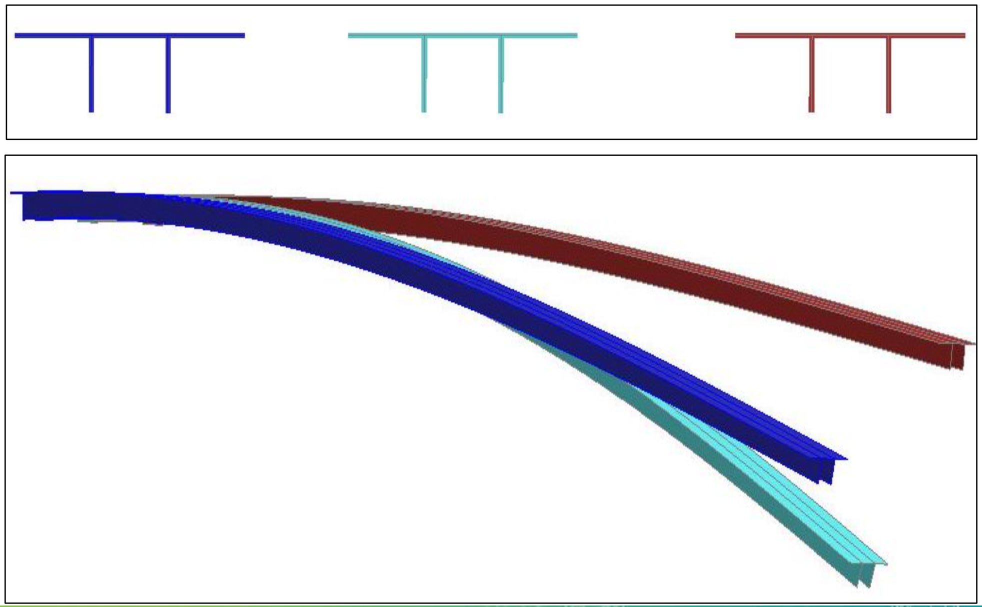
- **All 971 Versions**
  - "Offset" option has been added for all shell elements in version 971.
  - The offset is included when defining the connectivity of the shell element
  - The mid-surface is projected along its normal vector
  - Offsets greater than the shell thickness are permitted
  - Overrides the offset specified in the *SECTION_SHELL input
  - Nodal inertia is modified to account of the offset and provide a stable time step of explicit computations
- **R3 Enhancements**
  - **In the R3 release, shell thickness offsets are accounted for in the single-surface and surface-to-surface contact options.**
  - **Shells can be generated on CAD surfaces and then offset**
  - **Contact now accounts for the offsets during the analysis**

*ELEMENT_SHELL_OFFSET

Cantilevered Stiffener – Top flange with NLOC = 0, +1, -1

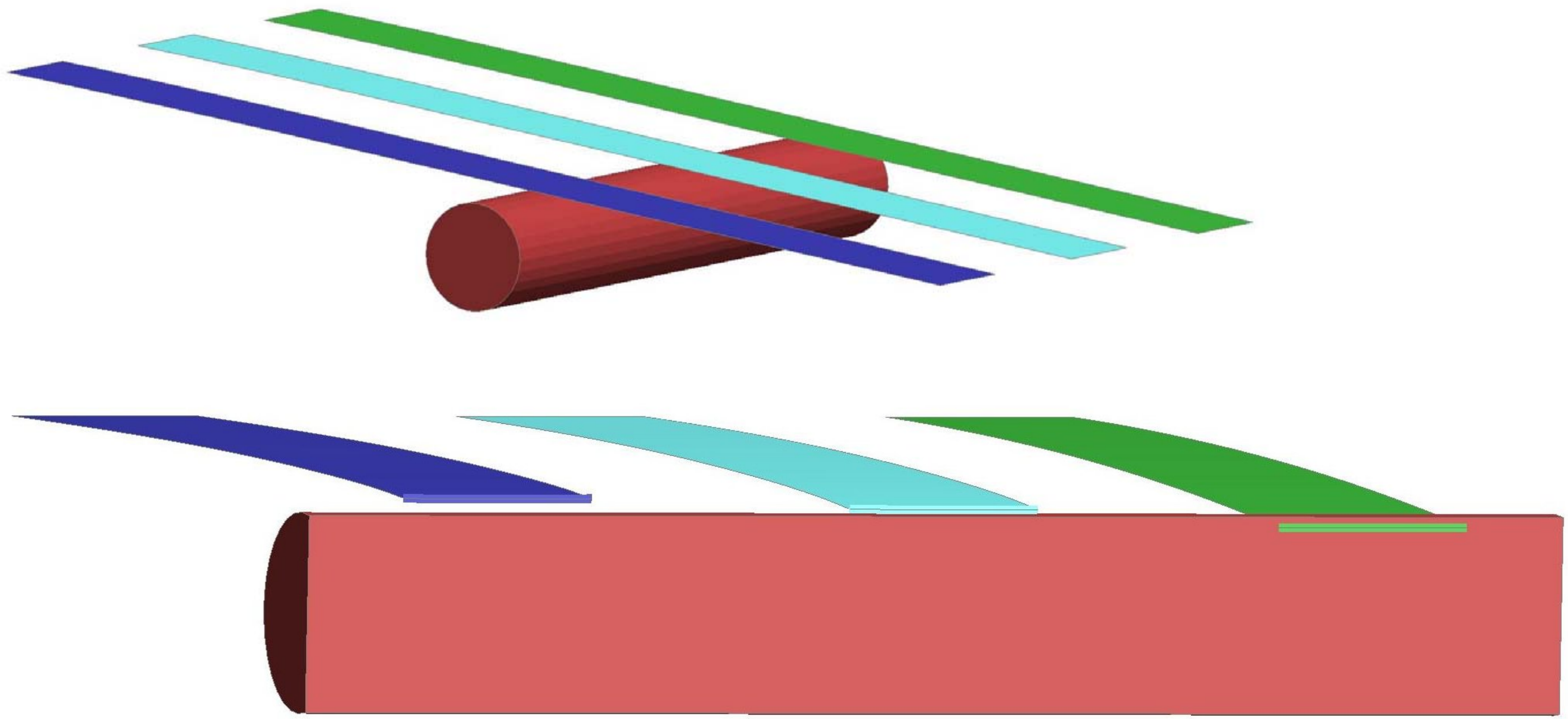






***ELEMENT_SHELL_OFFSET**

Simple Cantilever Beams – OFFSET = -2, 0, +2



PART_DUPLICATE** – New keyword that *does **not appear in 971 Keyword manual.

- Provides a method of duplicating parts or part sets without the need to use the *INCLUDE_TRANSFORM option

- Keyword Card Format

Card1	PTYPE	TYPEID	IDPOFF	IDEOFF	IDNOFF	TRANID		
Type	A	I	I	I	I	I		
Default	none	none	0	0	0	0		

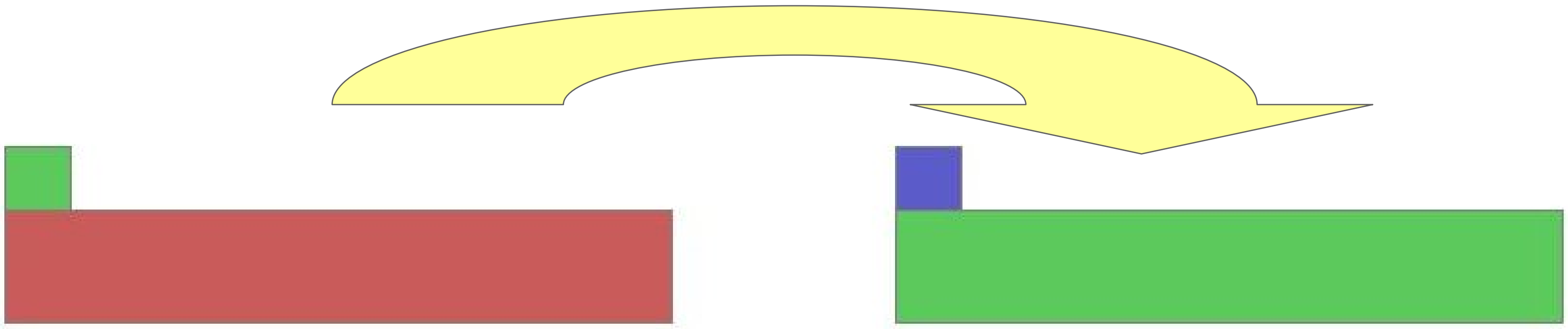
*PART_DUPLICATE

- PTYPE = "PART" to duplicate a single part or "PSET" for a part set.
- TYPEID = ID of part or part set to be duplicated.
- IDPOFF = ID offset of newly created parts.
- IDEOFF = ID offset of newly created elements.
- IDNOFF = ID offset of newly created nodes.
- TRANID = ID of *DEFINE_TRANSFORMATION to transform nodes.

```
$
*PART_DUPLICATE
      PART         1   1000000   1000000   1000000        1
      PART         2   1000000   1000000   1000000        1
*DEFINE_TRANSFORM
1
TRANSL,400,0,0
$
```

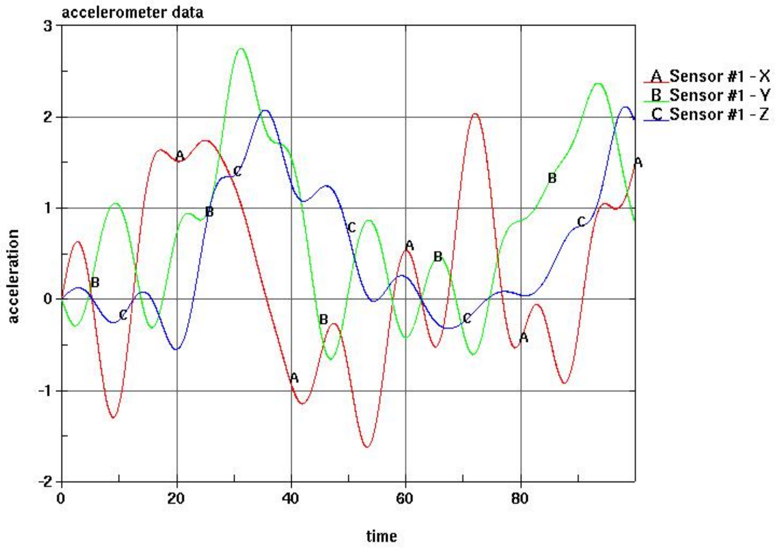
*PART_DUPLICATE

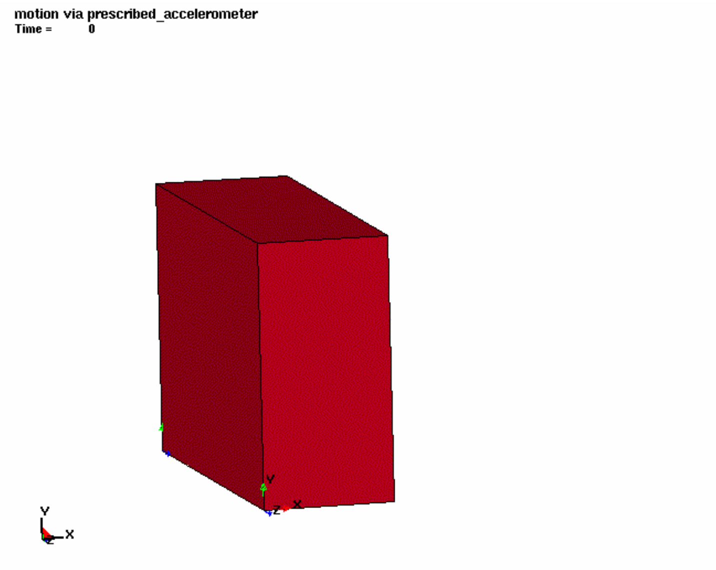
```
$
*PART_DUPLICATE
      PART         1   1000000   1000000   1000000        1
      PART         2   1000000   1000000   1000000        1
*DEFINE_TRANSFORM
1
TRANSL,400,0,0
$
```



*BOUNDARY_PRESCRIBED_ACCELEROMETER_RIGID

- Prescribes rigid body motion based on experimentally obtained accelerometer data
- Required input includes x, y and z acceleration traces and local coordinate system of each sensor
    - Accepts data from any number of sensors (a minimum of three is required)
    - Redundancy from the plurality of data is addressed automatically – information from only the most well conditioned signals is used
    - Filtered or unfiltered data can be handled

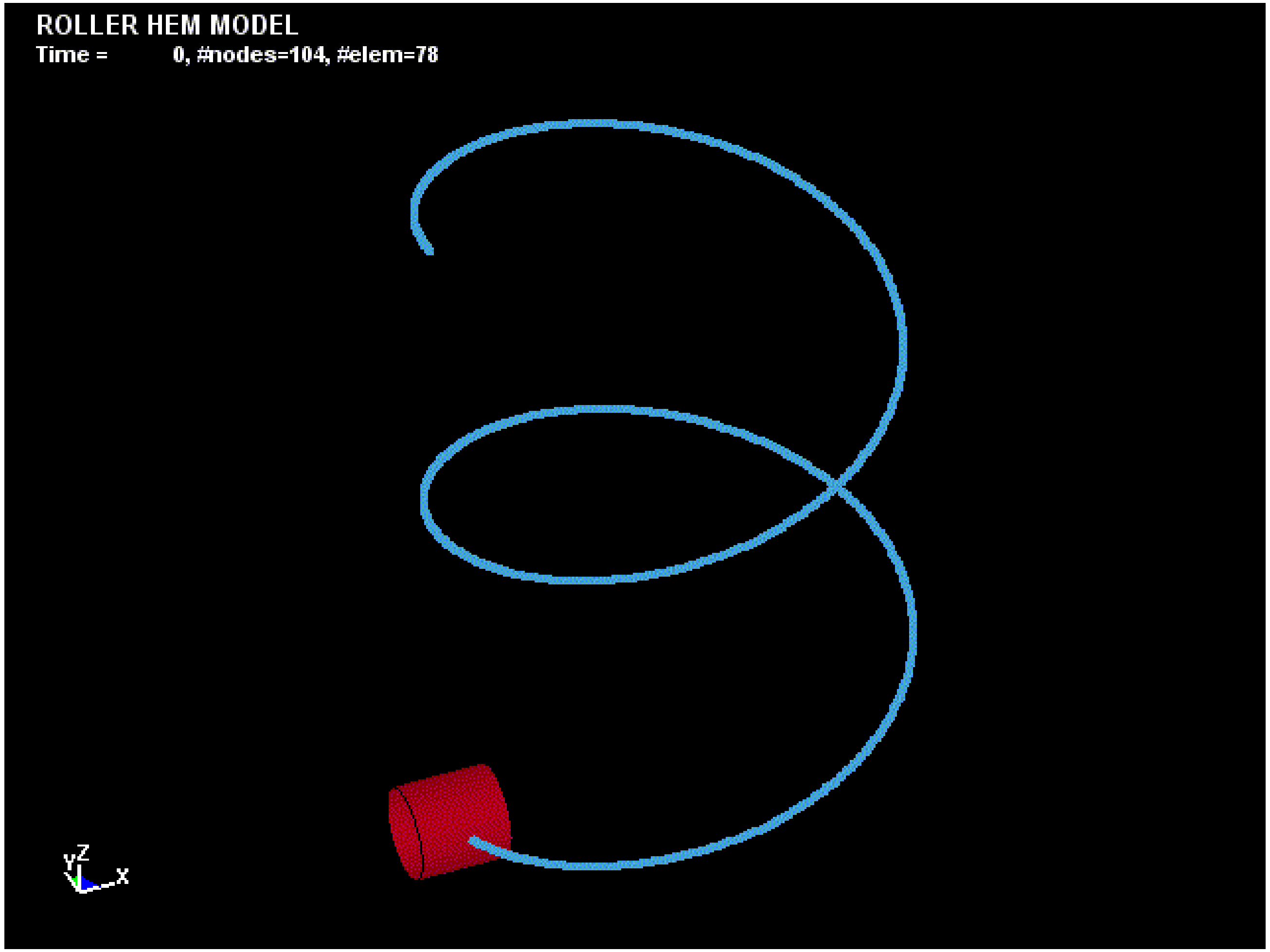




BOUNDARY_PRESCRIBED_ORIENTATION_*RIGID_{OPTION}

- Allows the orientation of a rigid body to be prescribed as a function of time.
- Uses a total formulation which is more precise than the incrementally based.
- Options:
  - **_ANGLES**
    - Specify a sequence of rotations about either body or space fixed axes and the associated orientation angles qi(t) (i=1,2,3) as time histories using *DEFINE_CURVE.
  - **_DIRCOS**
    - Nine elements of the direction cosine matrix are input as functions of time, Cij(t) (i,j=1,2,3)
  - **_EULERP**
    - Provide as functions of time four Euler parameters, εi(t) (i=1,..,4)

BOUNDARY_PRESCRIBED_ORIENTATION_*RIGID_{OPTION}



ENCRYPTED INPUT

- Encrypted input decks can now be read into LS-DYNA.
- Uses open PGP standard format.
  - Can use widely available tools such as “gpg” to encrypt data.
  - Public key encryption with 1024 bit DSA key and 128 bit AES.
- Any part of the input file may be encrypted, except for the initial *KEYWORD and any *INCLUDE statements.
- Multiple sections of the deck may be encrypted.
- Material properties defined in encrypted blocks are not echoed to d3hsp (*.otf) or other output files.

*DEFINE_COORDINATE_NODE

- New option to allow nodes N1 – N2 to define either the local x (default), y or z axis.

*LOAD_BODY_GENERALIZED

- Current input allows specification of body forces on a range of nodes N1 to N2 inclusive
- An arbitrary number of body load definitions are possible
- Two new keyword options are available to define the active node set
  - *LOAD_BODY_GENERALIZED_SET_NODE
  - *LOAD_BODY_GENERALIZED_SET_PART
- Body forces can now be applied in a local coordinate system

ELEMENT_MASS_PART_*{SET}

- Defines additional non-structural mass to be distributed by an area weighted distribution to all nodes of a given part or part set ID
  - The total added mass can be specified
  - **The final mass of the part or part set can be specified and the added mass computed automatically**
- Applies only to part ID's defined by shell elements.
- Provides an alternative method to giving the non-structural mass per unit area in the section definition

*LOAD_SEGMENT_NONUNIFORM

- Provides a method to load segments with a distributed load
  - Loading acts in direction defined by a vector
  - vector is defined in a local coordinate system
- Birth and death time for loading
- Scale factors are defined for each node of the segment
- Linear and quadratic segments considered

*SET_NODE_ADD_ADVANCED

- Define a node set by combining nodes from sets of
  - Nodes
  - Shells
  - Beams
  - Solids
  - Segments
  - Discretes
  - Thick shells

	1	2	3	4	5	6	7	8
Variable	SID1	TYPE1	SID2	TYPE2	SID3	TYPE3	SID4	TYPE4
Type	I	I	I	I	I	I	I	I

LOAD CURVE DISCRETISATION

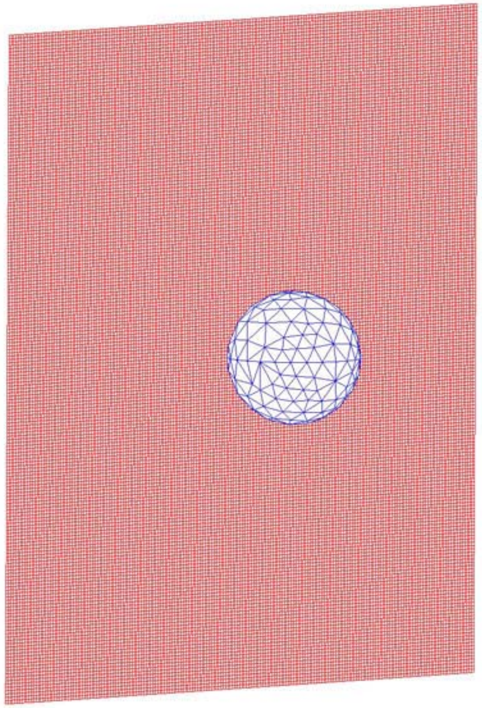
- LS-DYNA load curves used in material models are internally discretised.
  - Reduces the run time of the problem
  - Has been fixed to 101 equally spaced points (100 divisions) in previous versions.
  - 100 divisions considered too coarse for some applications
- Number of divisions for discretisation can now be specified on *CONTROL_SOLUTION.
  - Default is 100 divisions
  - All load curves use the same number of divisions.

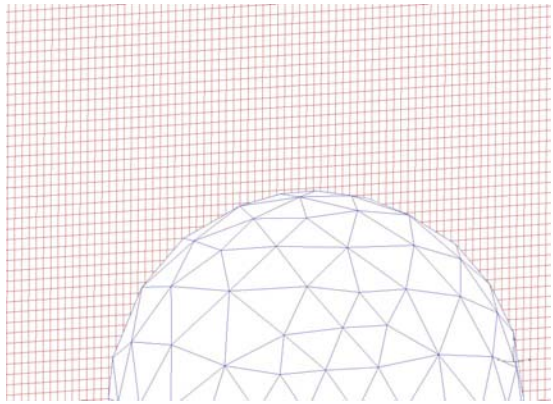
SELECTIVE MASS SCALING

- **Conventional mass scaling**
  - Lumped nodal masses are increased to give a stable timestep.
  - Can introduce non-physical inertia effects.
  - Models may have to be run with a small timestep to avoid or reduce non-physical results.
- **Selective mass scaling**
  - Mass is increased under the constraint that the rigid body translational behaviour is preserved.
  - Lowers high frequencies, which allows for a larger timestep, leaves the low frequency domain relatively unaffected.
  - Can be performed on entire model or a subsets of the parts.
    - As this option is both memory and CPU intensive it should only be applied to small finely meshed parts.
  - Can be run together with conventional mass scaling.
  - Set on *CONTROL_TIMESTEP card.

SELECTIVE MASS SCALING - Example 1

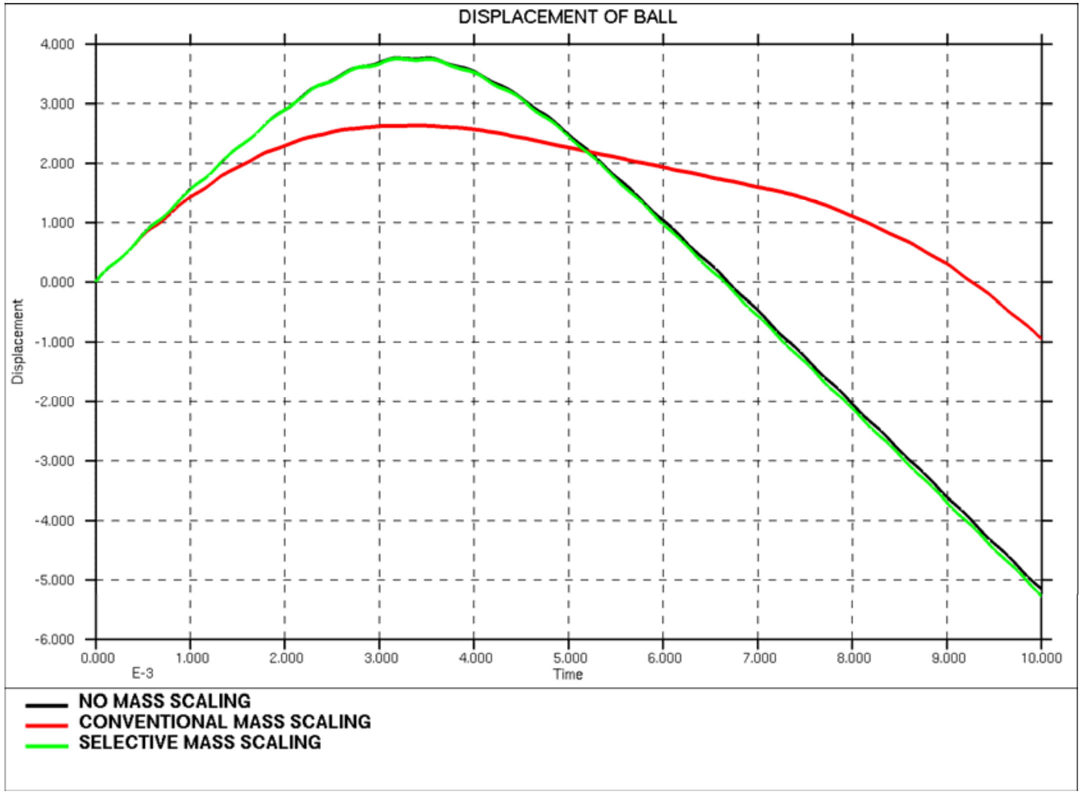
- Ball hitting a finely meshed plate
  - Run 1 – No mass scaling
  - Run 2 – Conventional mass scaling
  - Run 3 – Selective mass scaling





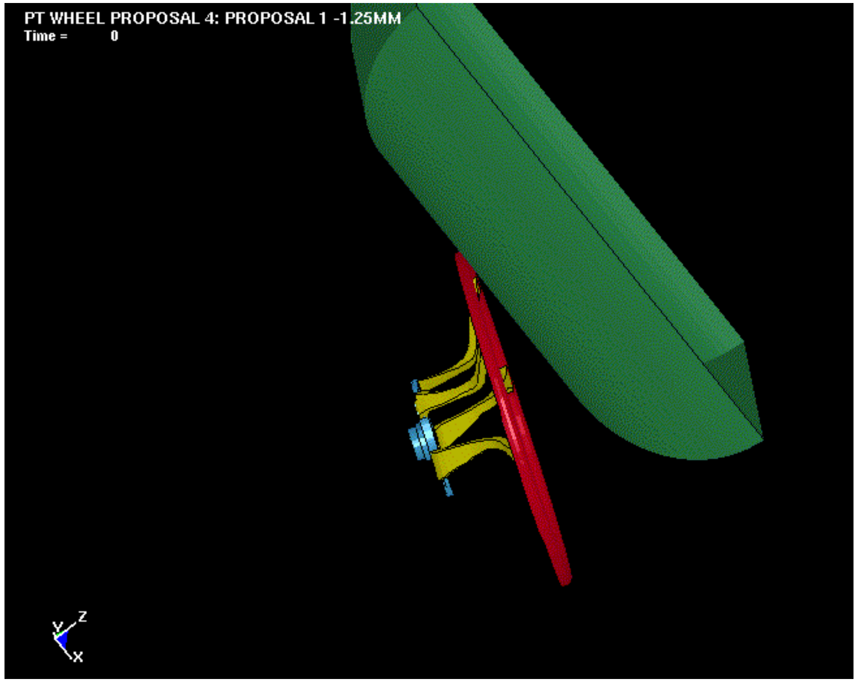
SELECTIVE MASS SCALING - Example 1

• Run 1 – No mass scaling	dt = 0.3 e-6
• Run 2 – Conventional mass scaling	dt = 1.0 e-6
• Run 3 – Selective mass scaling	dt = 1.0 e-6



SELECTIVE MASS SCALING – Example 2

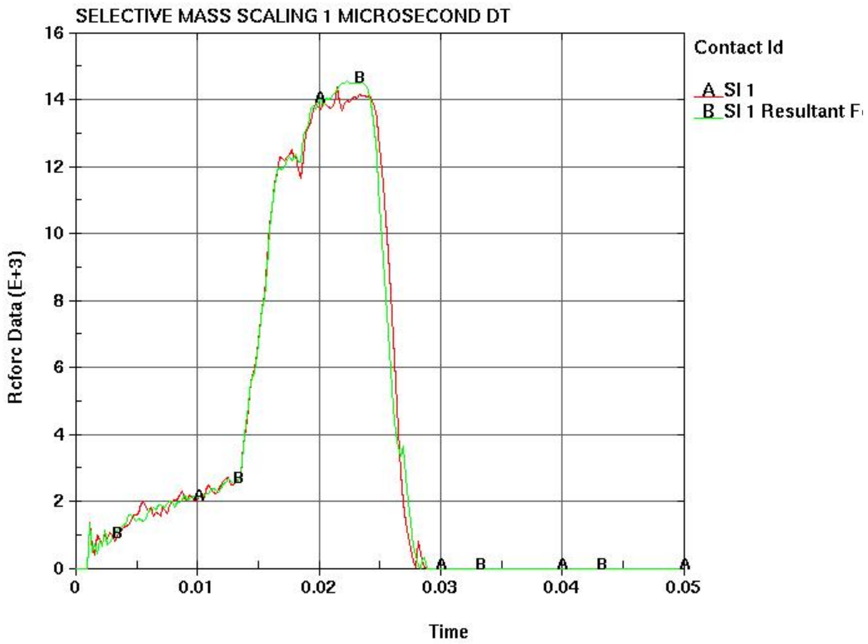
- Steering wheel impacted by body block
- Solid elements dt < 0.1e-6
- Simulation time to 50 ms
  - 5.5 hours with conventional mass scaling, dt=1.0e-7
  - 0.7 hours with selective mass scaling, dt=1e-6

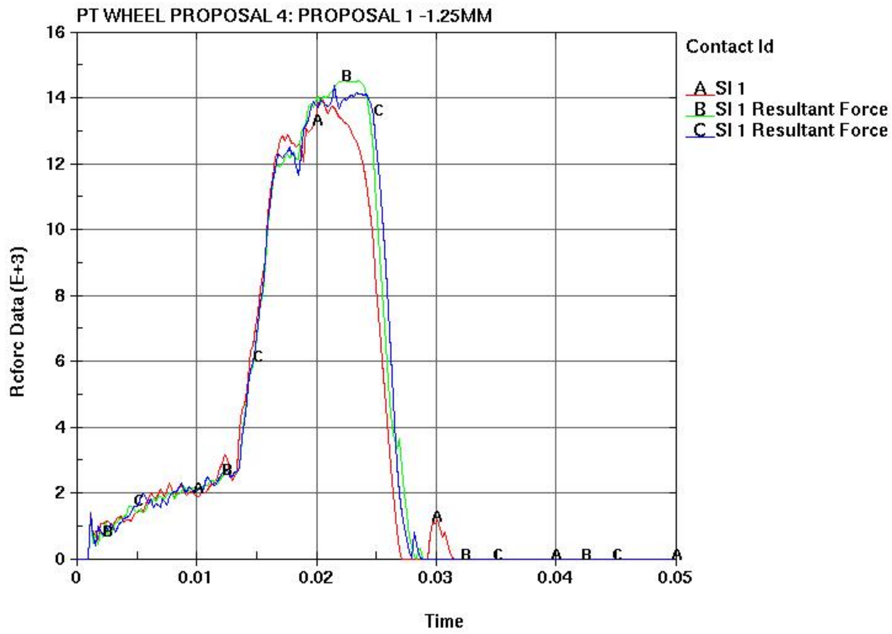


SELECTIVE MASS SCALING – Example 2

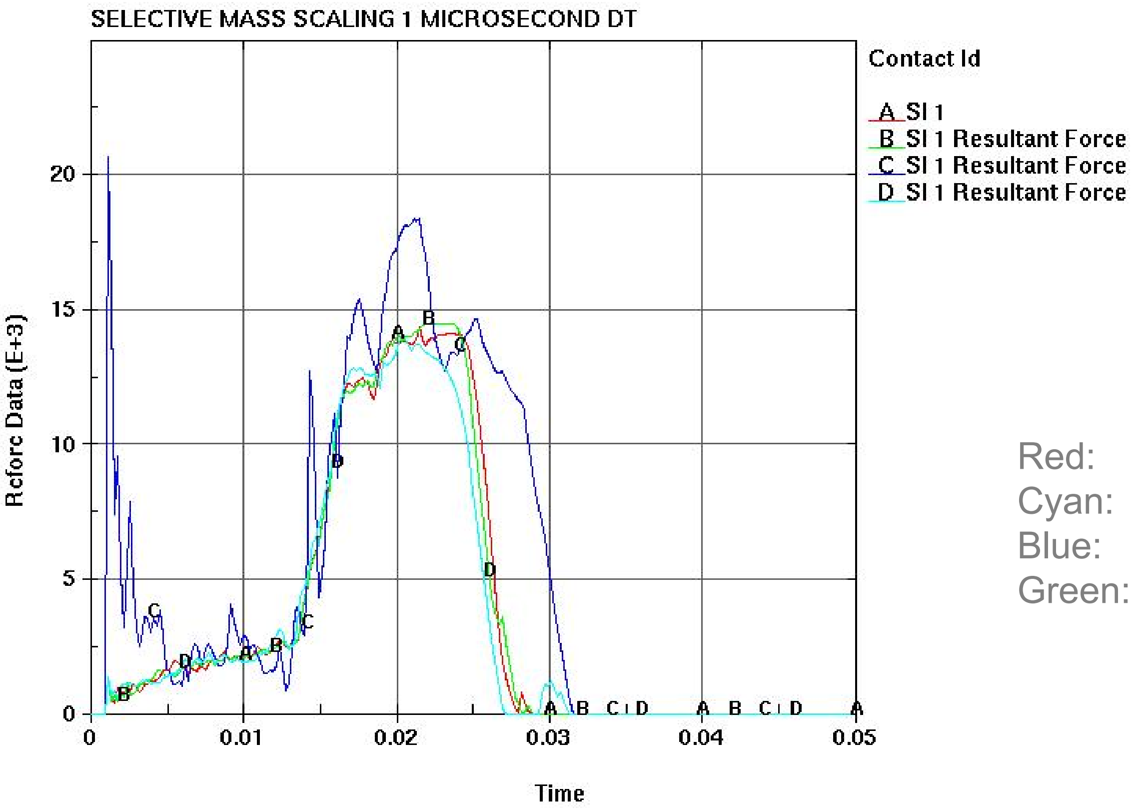
Red: dt=1.e-06 SMS
Green: dt=1.e-07 CMS

Red: dt=2.e-06 SMS
Blue: dt=1.e-06 SMS
Green: dt=1.e-07 CMS





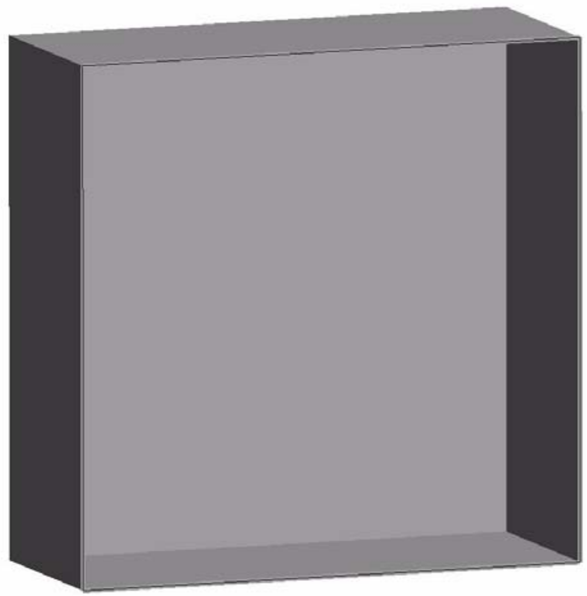
SELECTIVE MASS SCALING – Example 2



Red:	dt=1.e-06	SMS
Cyan:	dt=2.e-06	SMS
Blue:	dt=2.e-06	CMS
Green:	dt=1.e-07	CMS

PARTICLE METHOD

- A new method for airbag analysis
  - Added to LS971_R3
  - Further developments in LS971_R4
- Sometimes referred to as the Corpuscular Method (CP)
- Method based on Kinetic Molecular Theory
  - Ideal gas law
  - Bernoulli and Maxwell
- Simple Input format – Similar to Control Volume
- Venting and Porosity supported
- Internal baffles defined through sid2
  - Flow through internal baffles can be monitored
  - External as well as internal vents can now be specified in vent section.
- Multiple inflators
- Switching from Particle method to CV
- Each node (particle) represents in the order of $1*10^{18}$ molecules



PARTICLE METHOD – Keyword Format

***AIRBAG_PARTICLE**

SID1	STYP1	SID2	STYP2	BLOCK	HCONV		
NP	UNIT	VIS	PATM	TATM	NVENT	TEND	TSW
IAIR	NGAS	NORIF	NID1	NID2	NID3		
SID3	STYPE3	C23	LCTC23	LCPC23	*EVO*		

vent hole data

R4 - Option

Extra card if IAIR=1:

XM_{air}	A_{air}	B_{air}	C_{air}	P_{air}	T_{air}

initial air data

NGAS cards:

XM_1	A_1	B_1	C_1	LCM_1	LCT_1	$INFG_1$
XM_2	A_2	B_2	C_2	LCM_2	LCT_2	$INFG_2$.
.						
XM_n	A_n	B_n	C_n	LCM_n	LCT_n	$INFG_n$

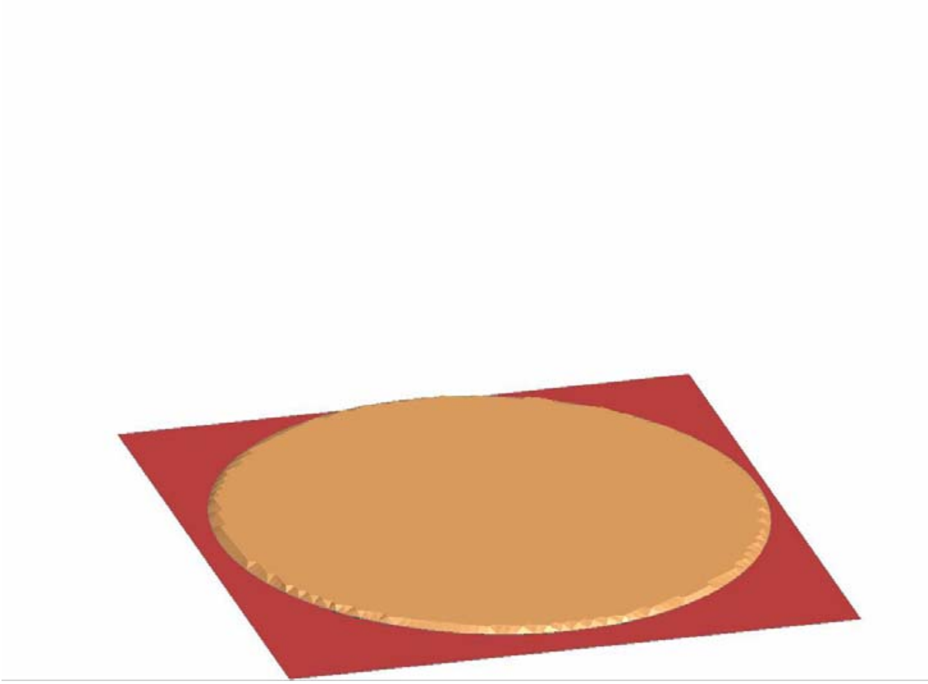
gas component data

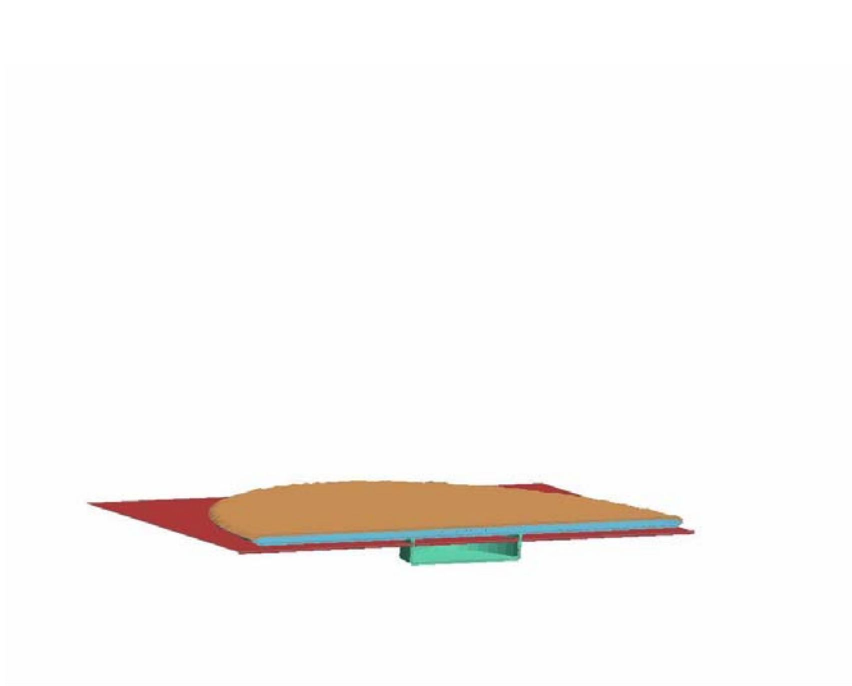
NORIF cards:

NID_1	AN_1	VX_1	VY_1	VZ_1	$INFO_1$
NID_2	AN_1	VX_2	VY_2	VZ_2	$INFO_2$
.					
NID_n	AN_n	VX_n	VY_n	VZ_n	$INFO_n$

inlet orifice data

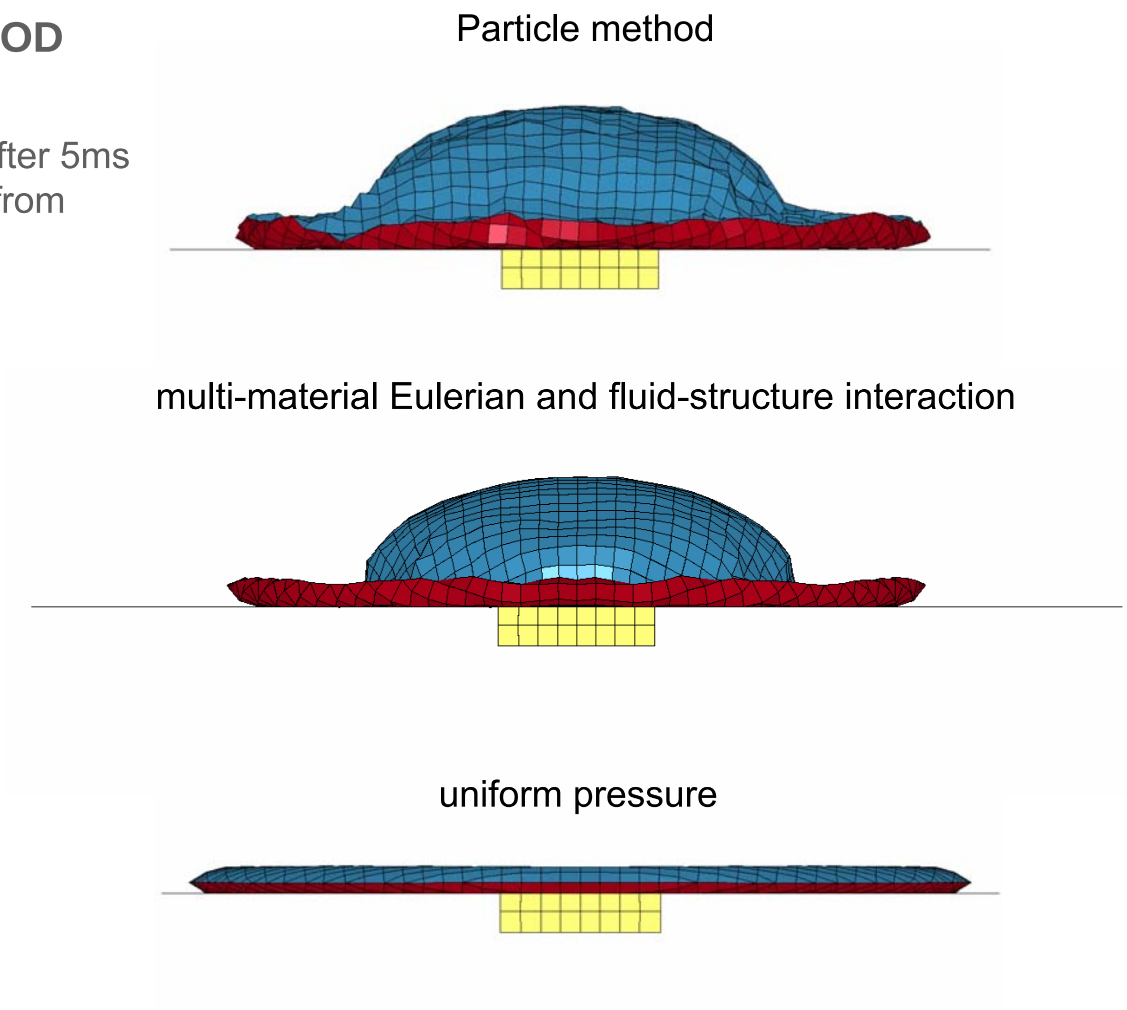
PARTICLE METHOD – Simple Airbag Example



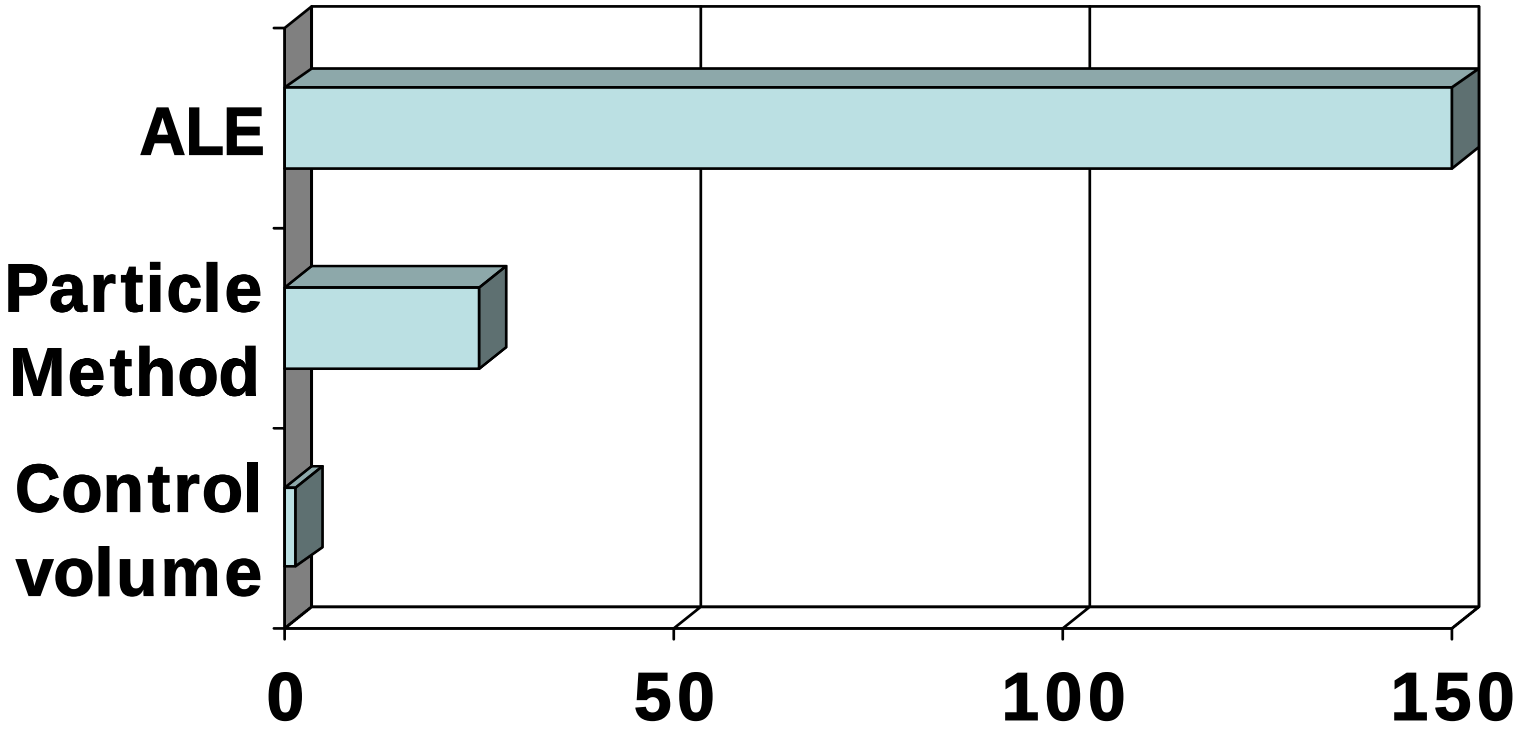


PARTICLE METHOD

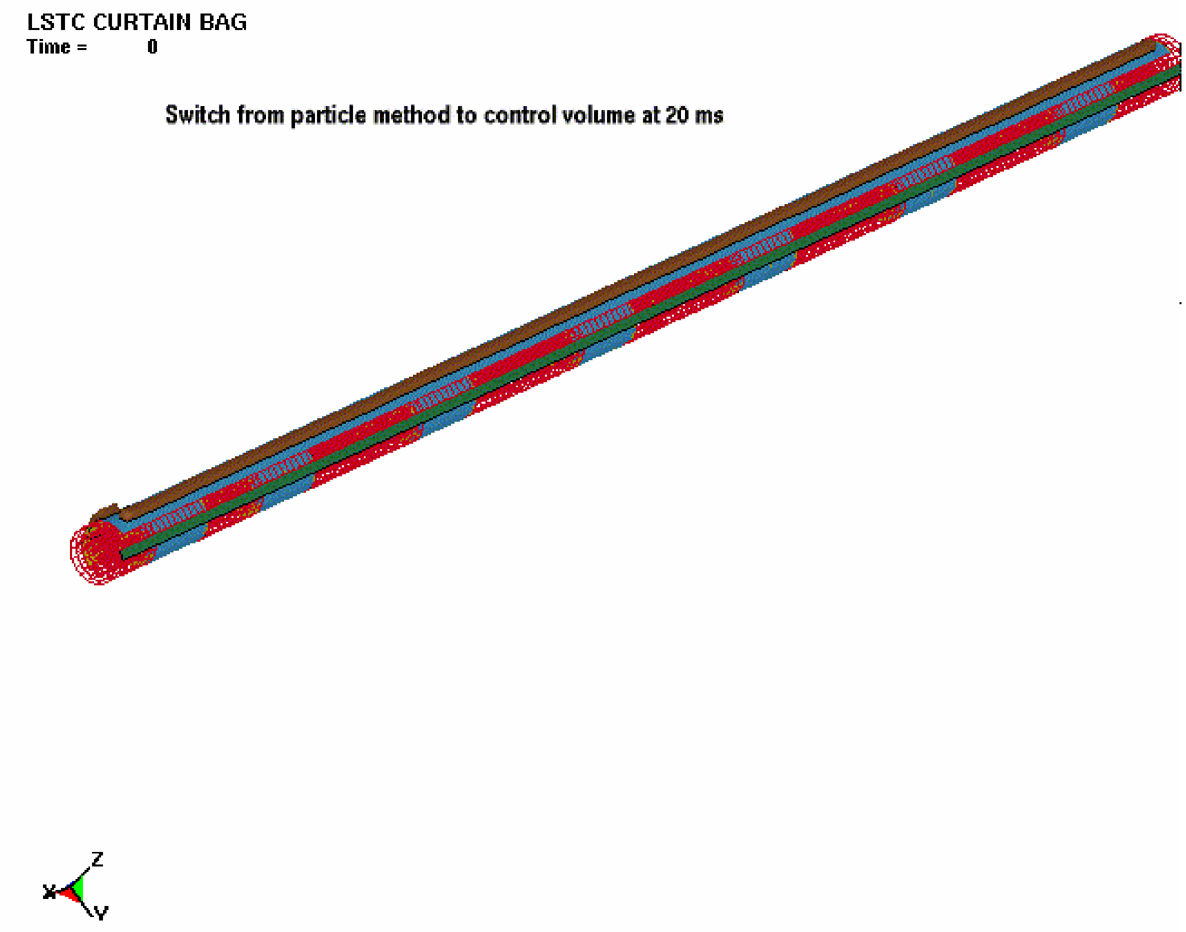
• Example: flat bag after 5ms compared to results from Eulerian and uniform pressure models



PARTICLE METHOD – Typical Run times.



PARTICLE METHOD – Curtain Airbag – LSTC Example

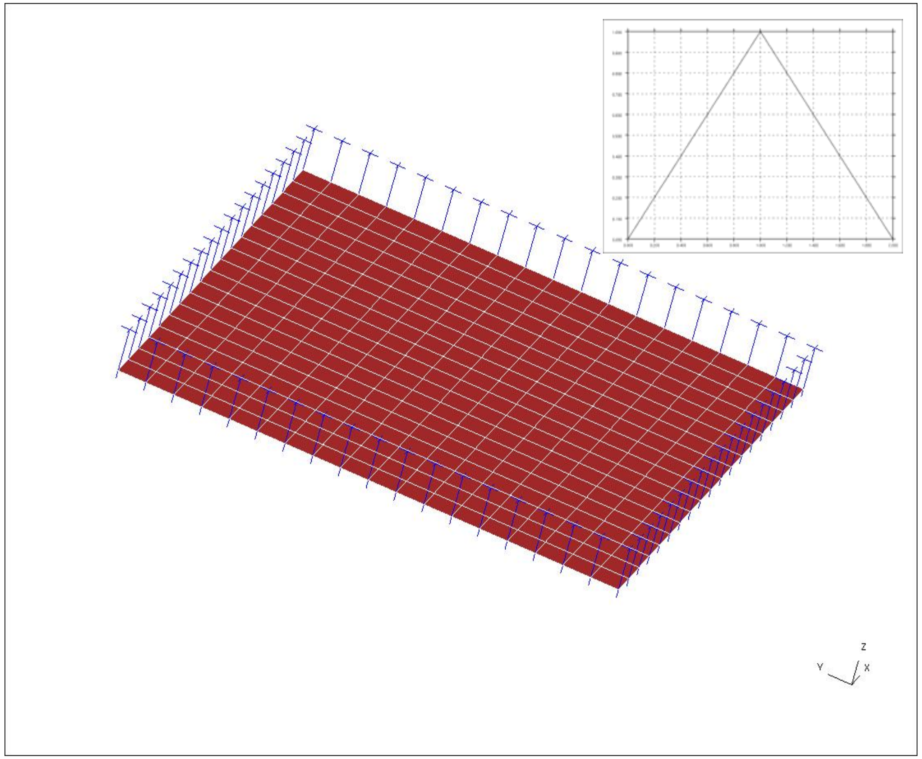


*MAT_CONCRETE_EC2

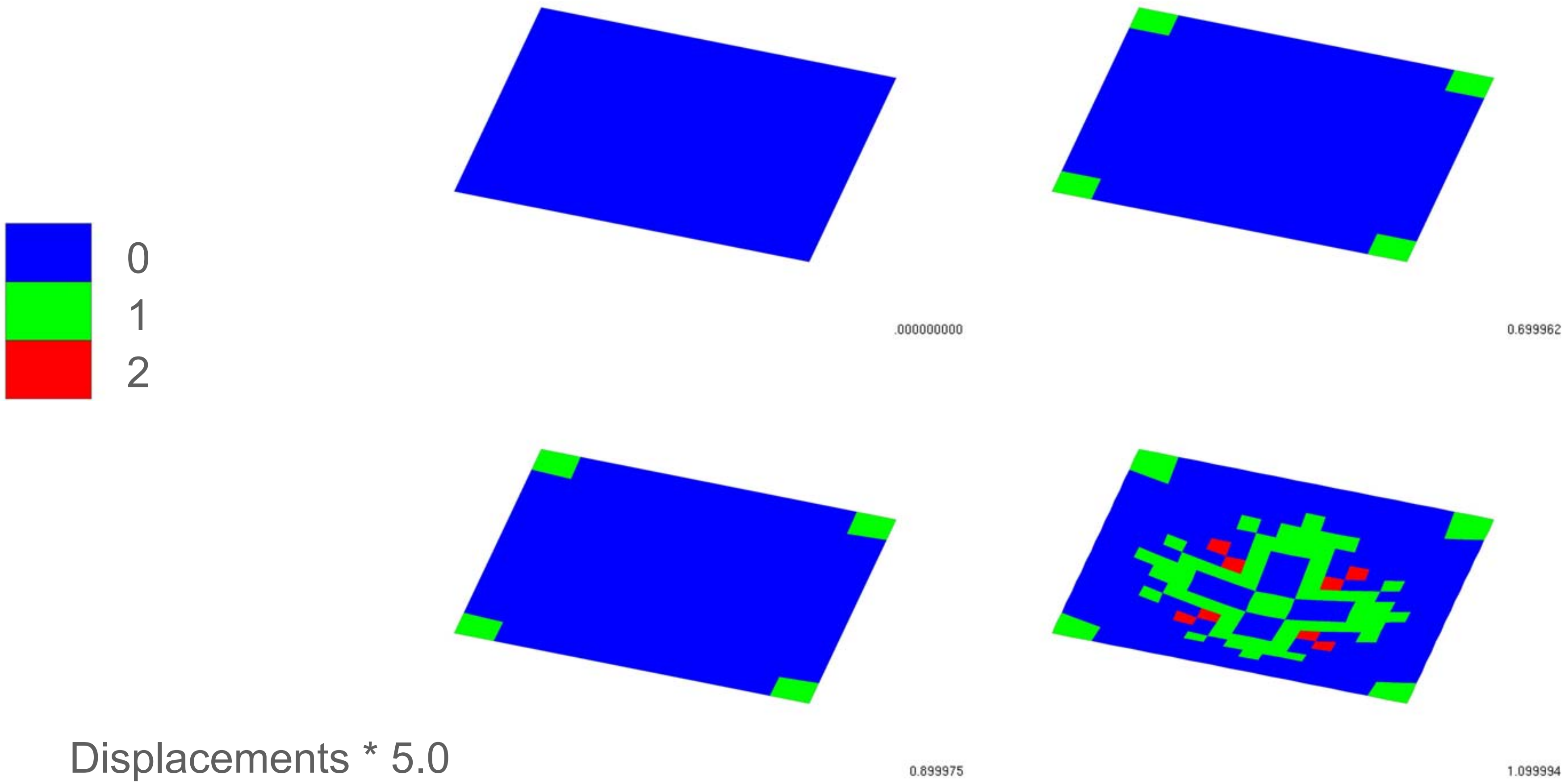
- Material model based on Eurocode 2 Part 1.2 (General rules – Structural fire design)
- For shell and Hughes-Liu beams
- Material model can represent
  - Plain concrete
  - Reinforcing steel only
  - Smeared combination of concrete and reinforcement
- Model includes
  - Concrete cracking in tension
  - Concrete crushing in compression
  - Reinforcement yield, hardening and failure
- Properties are thermally sensitive for fire analysis
- Extra variable output allows the plotting of:
  - Current crack opening strain
  - Number of cracks
  - Thermal strain
  - Current crack opening strain 90 deg to 1st
  - Max crack opening strain 90 deg to 1st
  - Equivalent uniaxial strain for concrete compression
  - Temperature
  - Current crack opening strain (1st crack)
  - Max crack opening strain (1st crack)

*MAT_CONCRETE_EC2 – Example

- Simply supported square plate
- Pressure loading over entire plate
- *INTEGRATION_SHELL with 8 layers
  - concrete
  - steel (reinforcement)
  - concrete
  - concrete
  - concrete
  - concrete
  - steel (reinforcement)
  - concrete
- *MAT_CONCRETE_EC2
  - Two definitions
    - Plain concrete
    - 100% reinforcement

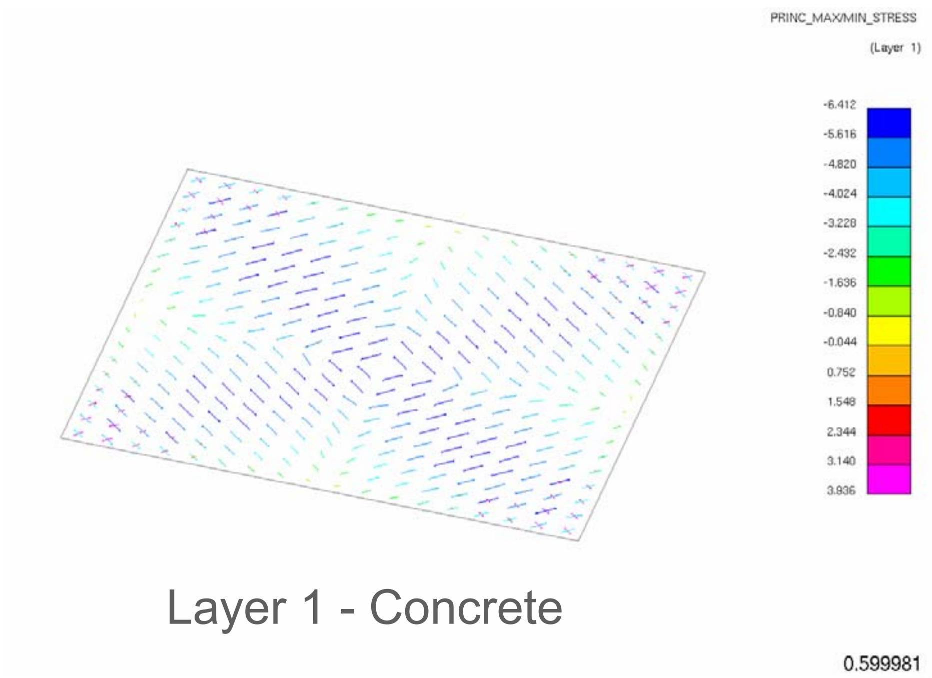


*MAT_CONCRETE_EC2 – Example – Top Layer number of cracks - concrete



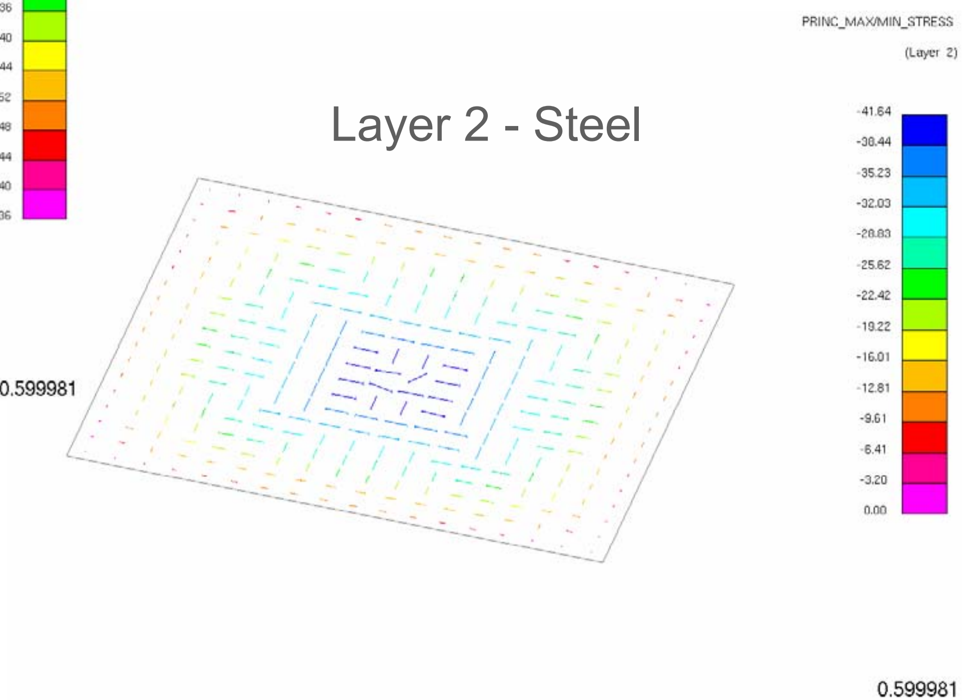
Displacements * 5.0

*MAT_CONCRETE_EC2 – Example – Max/Min Principal Stress



Layer 1 - Concrete

Layer 2 - Steel

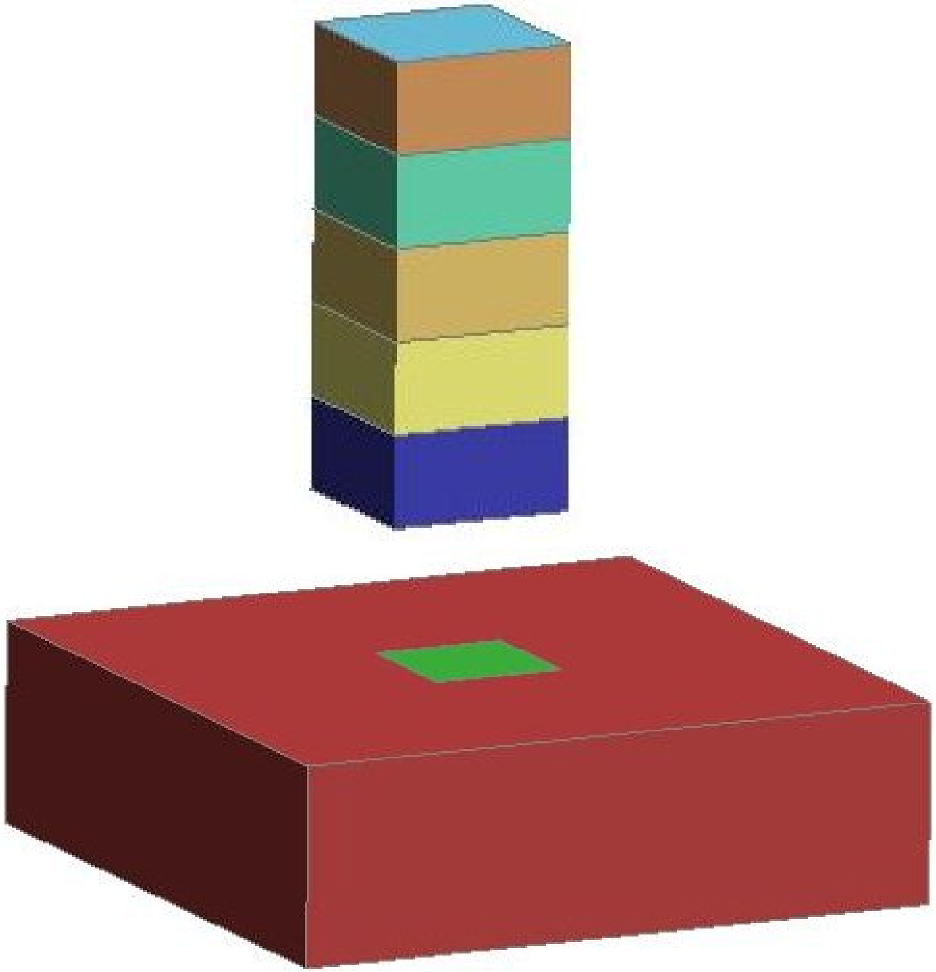


Displacements * 5.0

STAGE_CONSTRUCTION

- Initially developed for civil engineering applications - Its use can be extended to other areas of application where it is desirable to add and delete parts in an automated fashion.
- Controlled by three main Keywords
  - *CONTROL_STAGED_CONSTRUCTION
  - *DEFINE_CONSTRUCTION_STAGES
  - *DEFINE_STAGE_CONSTRUCTION_PART
- A "dynain" file is written at the end of each stage to allow individual stages to be re-run without re-running the whole analysis.
- During a stage a part can
  - Become active – Its stiffness and weight are gradually added during the initial phase of the stage, its is then fully included for the rest of the stage.
  - Be deleted – Its stiffness and weight are gradually reduced during the initial phase of the stage, it is then deleted for the rest of the stage.

STAGE_CONSTRUCTION - Example



Parts 3 – 13 Structure

Parts 1 – 2 Soil

STAGE_CONSTRUCTION - Example

```
*CONTROL_STAGED_CONSTRUCTION
       0.0         1         9      10.0    1.0E-9         0
*DEFINE_CONSTRUCTION_STAGES
         1       0.0       1.0       0.5       0.0       0.0
         2       1.0       2.0       0.5       0.0       0.0
         3       2.0       3.0       0.5       0.0       0.0
         4       3.0       4.0       0.5       0.0       0.0
         5       4.0       5.0       0.5       0.0       0.0
         6       5.0       6.0       0.5       0.0       0.0
         7       6.0       7.0       0.5       0.0       0.0
         8       7.0       8.0       0.5       0.0       0.0
         9       8.0       9.0       0.5       0.0       0.0
*DEFINE_STAGED_CONSTRUCTION_PART
         1         0         0
         2         0         1
         3         3         0
         4         2         0
         5         4         0
         6         5         0
         7         5         0
         8         6         0
         9         6         0
        10         7         0
        11         7         0
        12         8         0
        13         8         0
```

Define 9 stages

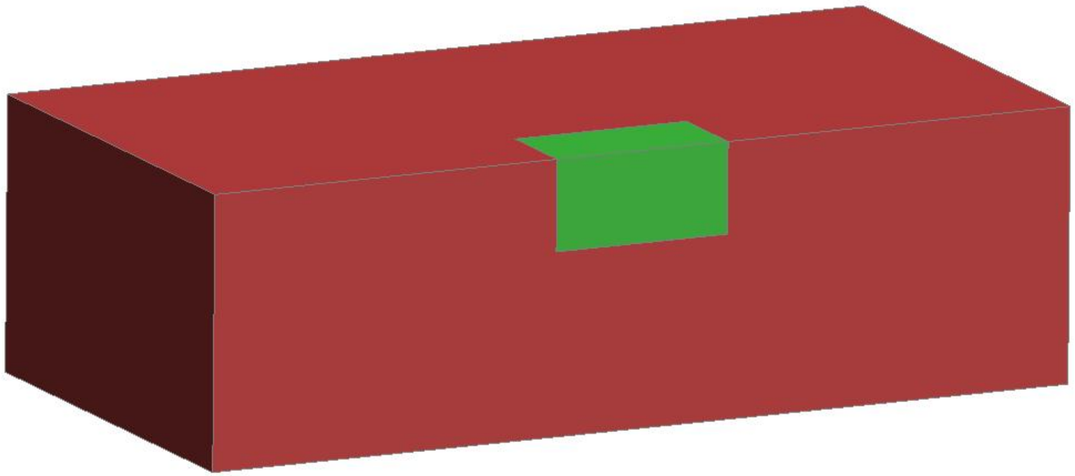
- Stage id
- Start Time
- End Time
- Ramp duration

For each part define active stage

- Part id
- Stage in which part is added
- Stage in which part is removed

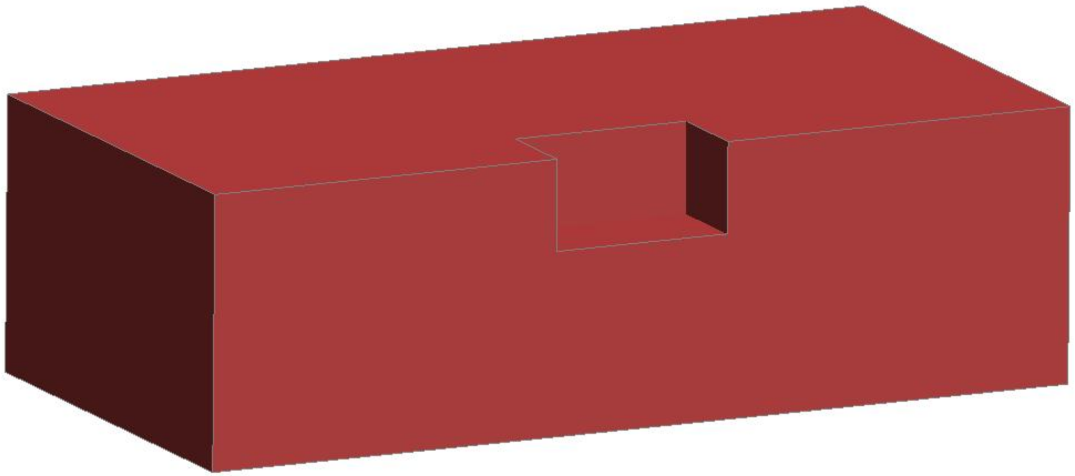
STAGE_CONSTRUCTION - Example

Stage 0



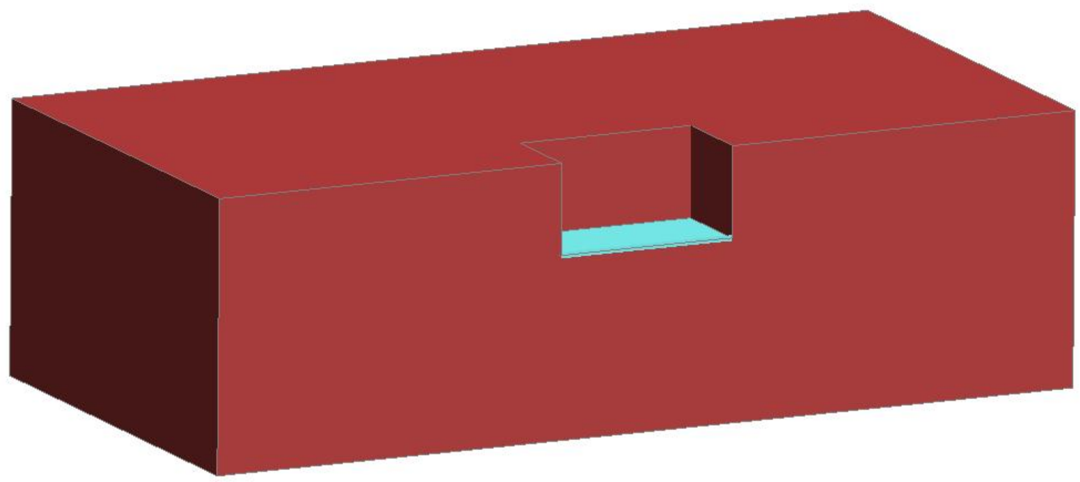
STAGE_CONSTRUCTION - Example

Stage 1



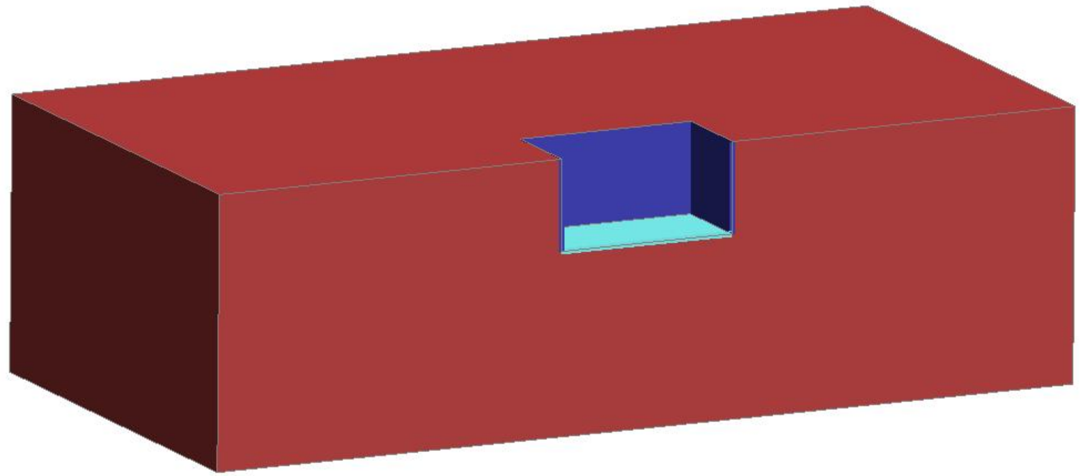
STAGE_CONSTRUCTION - Example

Stage 2



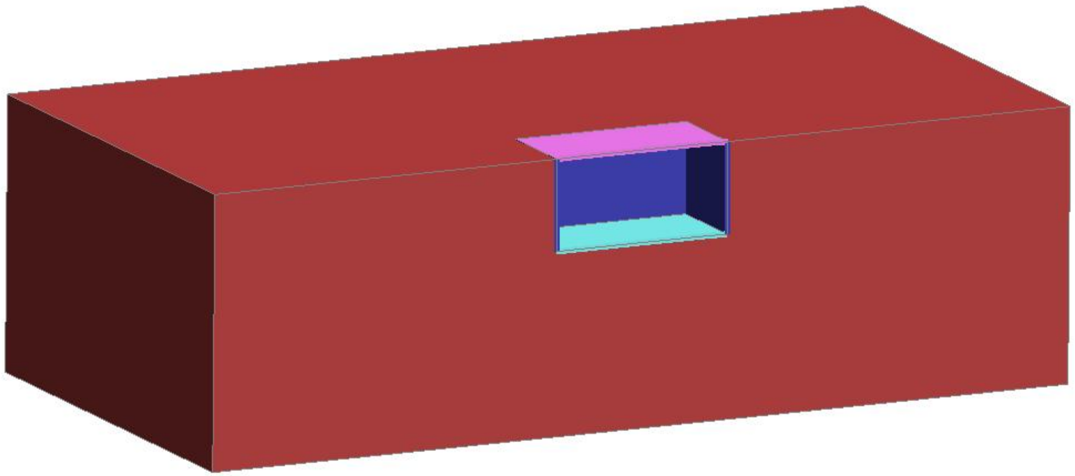
STAGE_CONSTRUCTION - Example

Stage 3



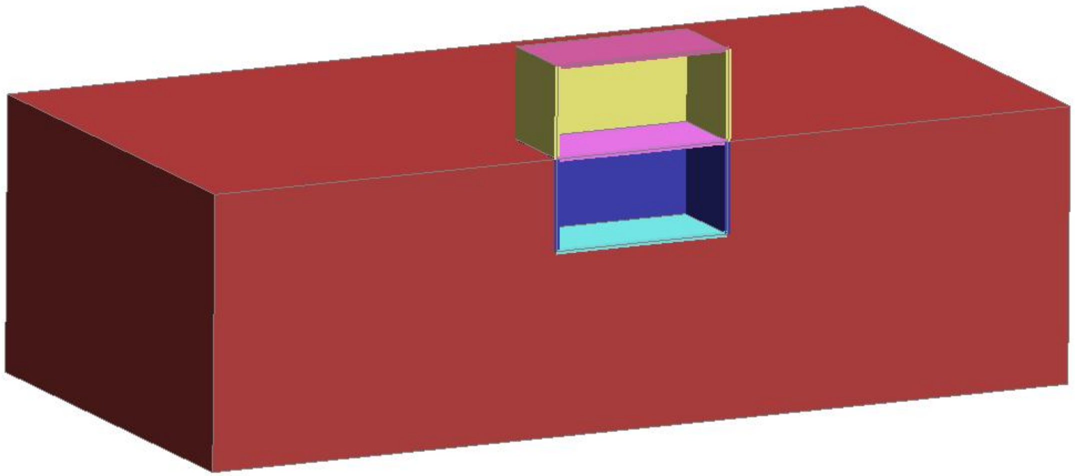
STAGE_CONSTRUCTION - Example

Stage 4



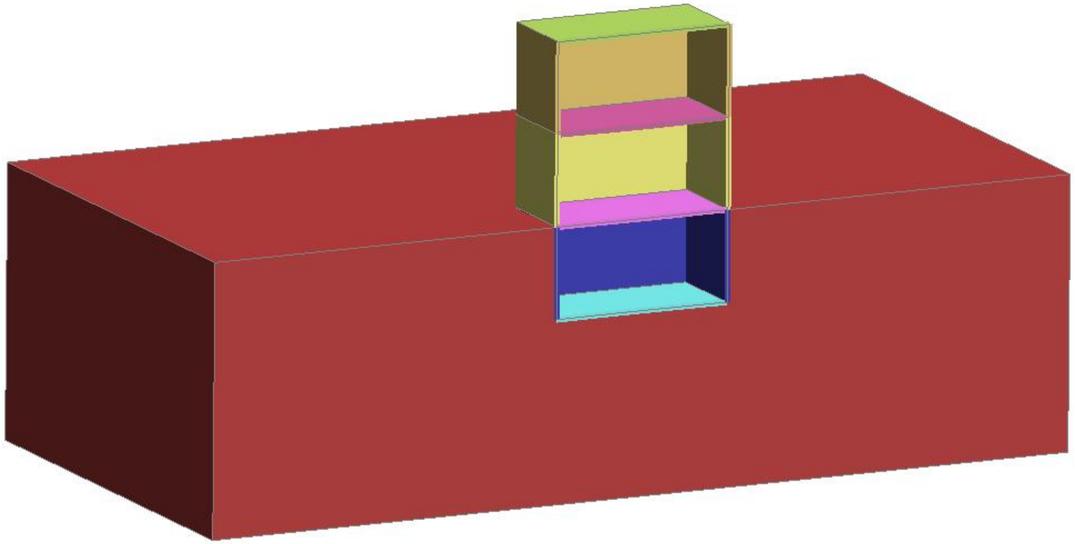
STAGE_CONSTRUCTION - Example

Stage 5



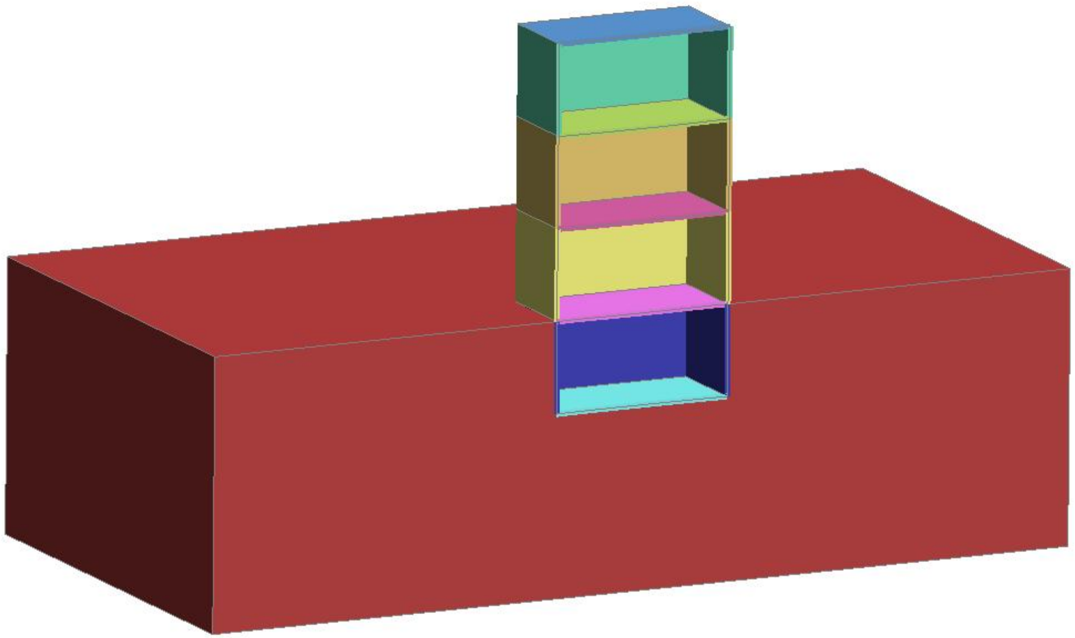
STAGE_CONSTRUCTION - Example

Stage 6



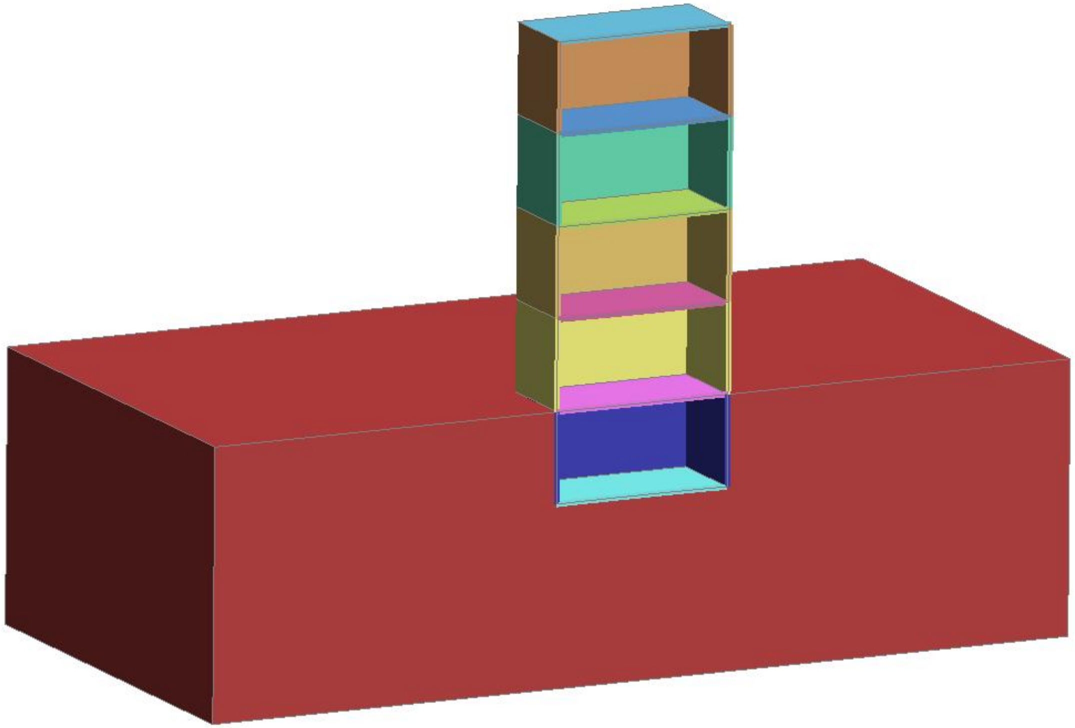
STAGE_CONSTRUCTION - Example

Stage 7

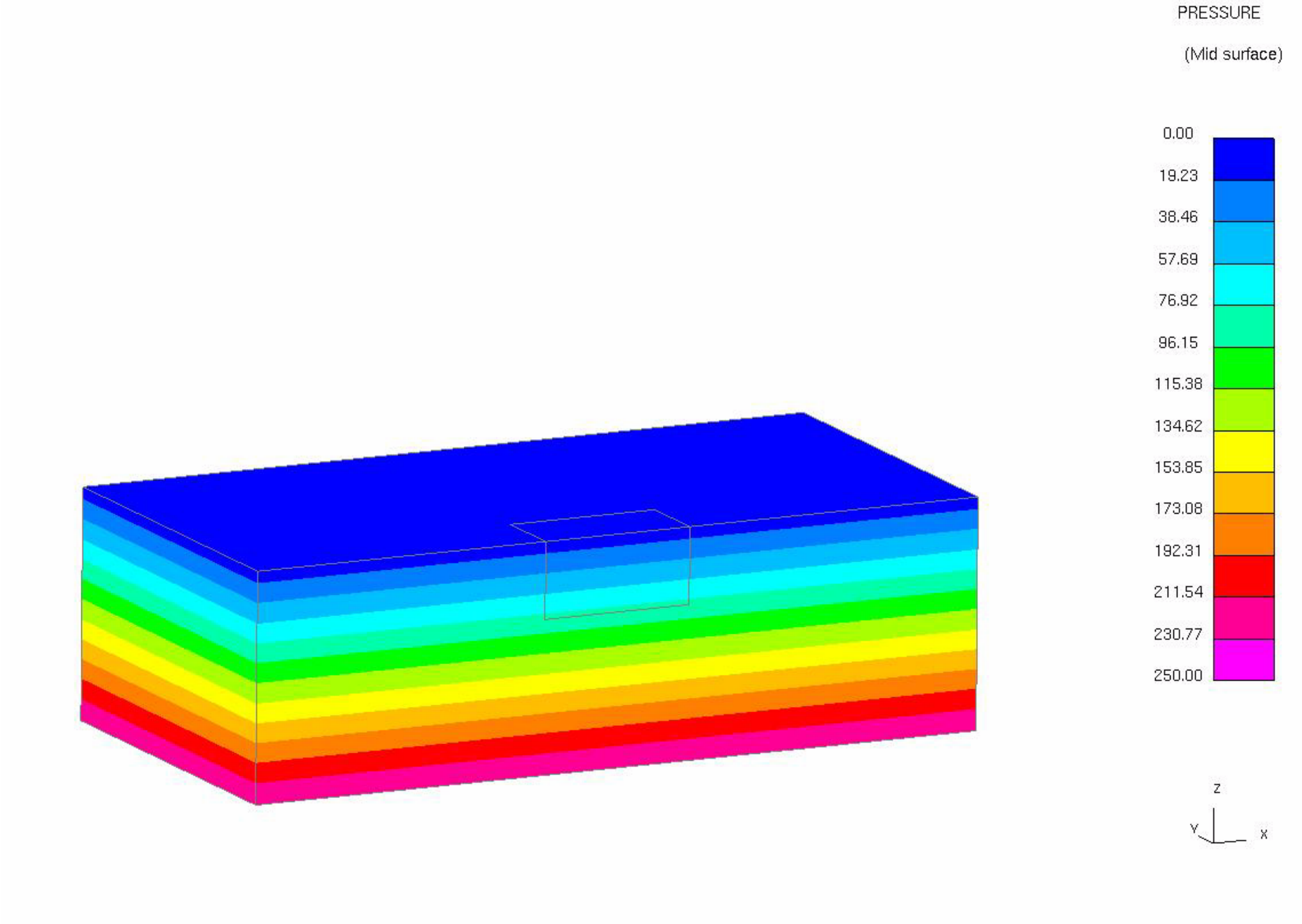


STAGE_CONSTRUCTION - Example

Stage 8

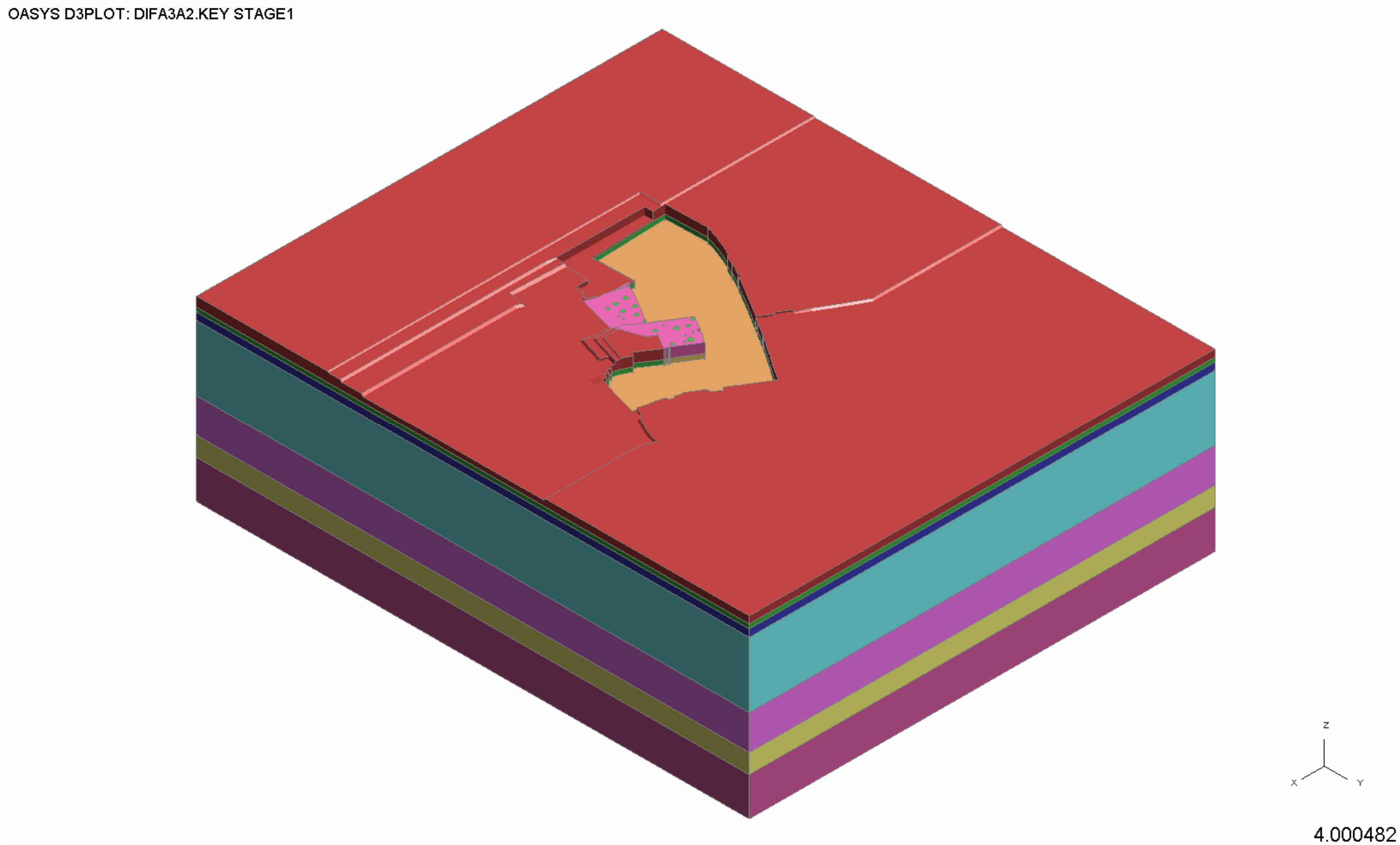


STAGE_CONSTRUCTION - Example



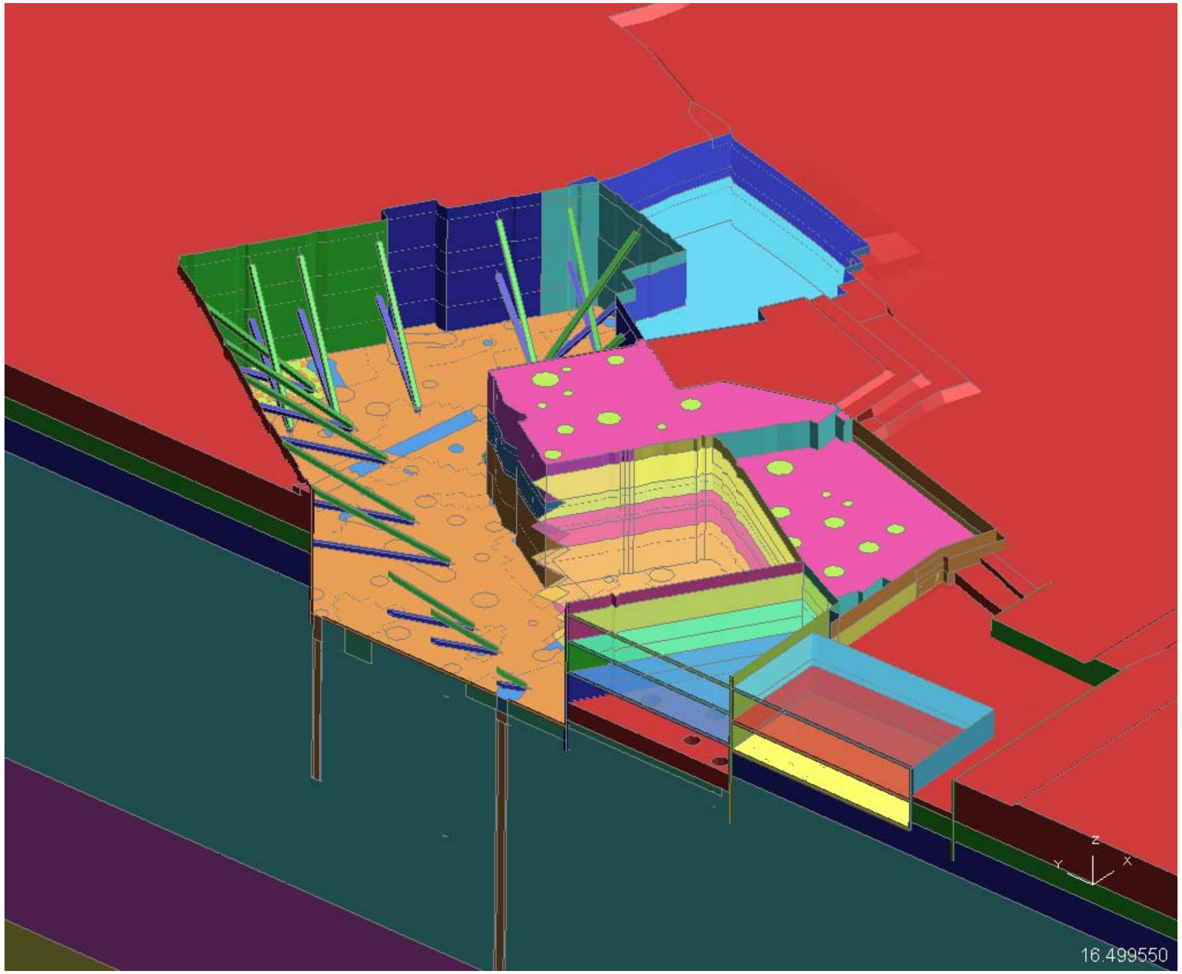
STAGED CONSTRACTION – LARGE EXAMPLE

- Remove existing building



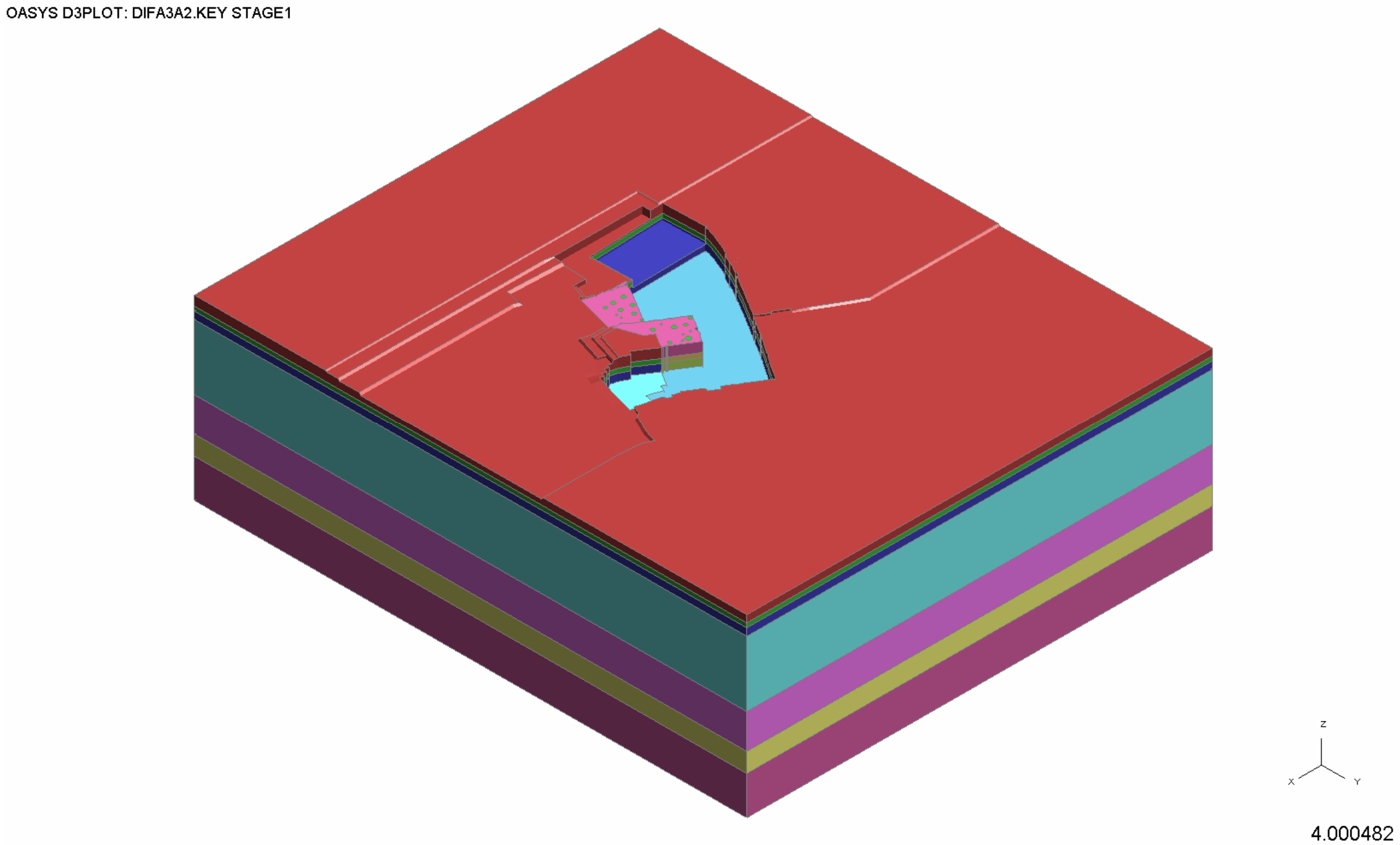
STAGED CONSTRACTION – LARGE EXAMPLE

- Temporary supports during demolition



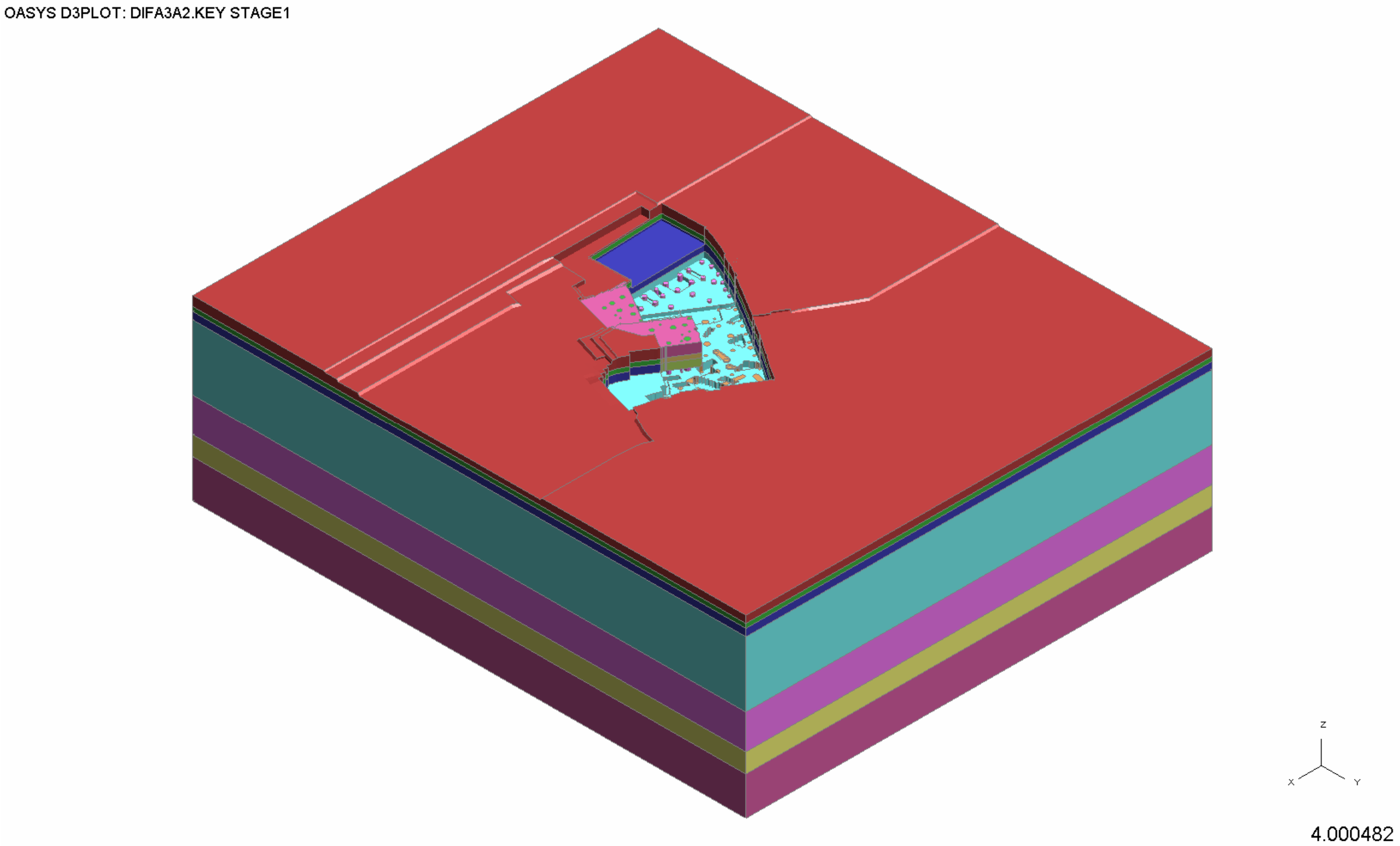
STAGED CONSTRACTION – LARGE EXAMPLE

- Install New Retaining Wall and dig to new basement level



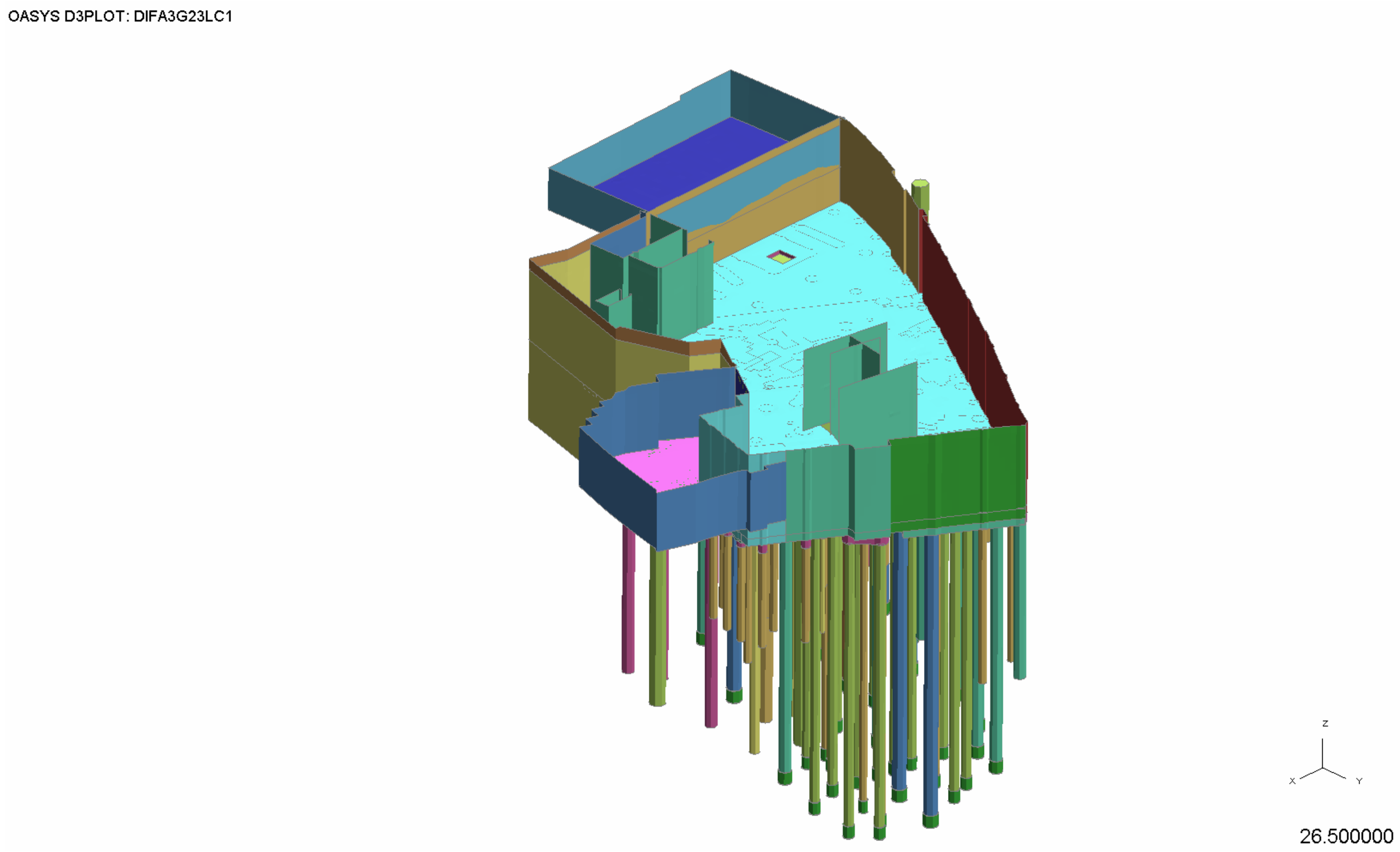
STAGED CONSTRACTION – LARGE EXAMPLE

- Install New Piles and construct Pile cap



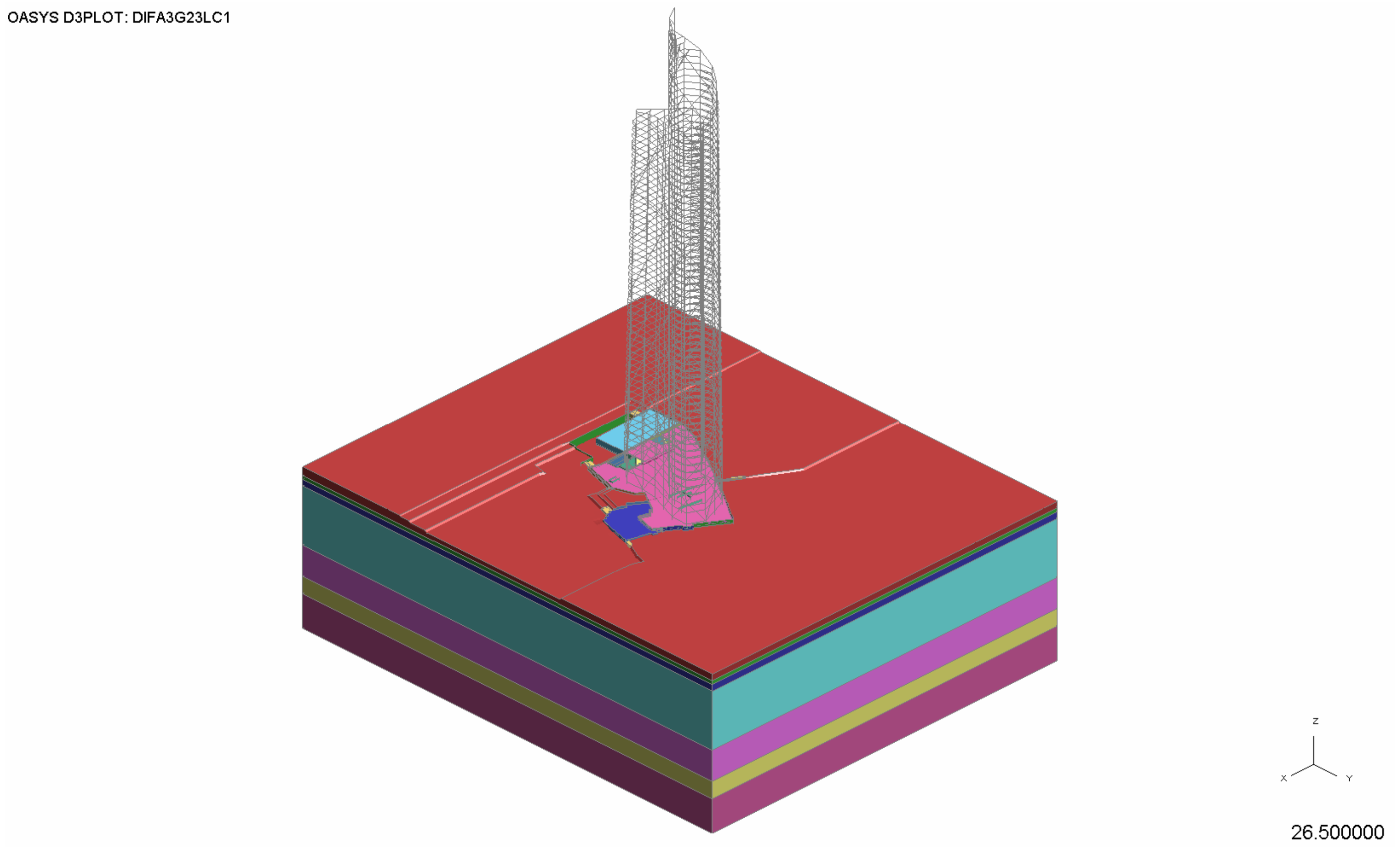
STAGED CONSTRACTION – LARGE EXAMPLE

- Install base slab and internal core walls



STAGED CONSTRACTION – LARGE EXAMPLE

- Install Internal slabs and the super-structure

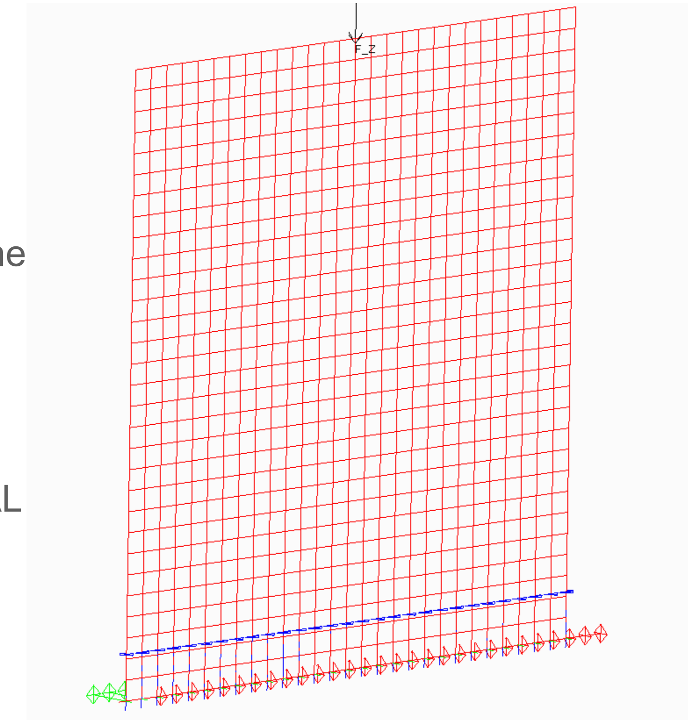


LOAD_THERMAL_VARIABLE_SHELL_*{OPTION}

- Allows through thickness temperatures to be defined for shell elements.
- Temperature at each through thickness point is defined as a function of time.
- Through thickness point is defined in the range -1 (bottom) to +1 (top) surface.

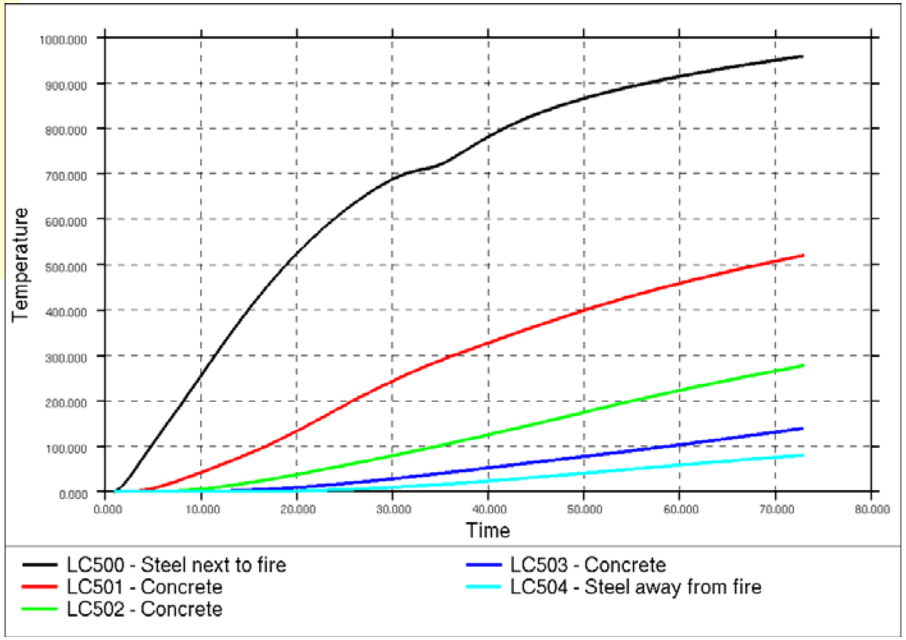
Example – Steel – Concrete – Steel Composite wall

- Temperatures defined via *LOAD_THERMAL_VARIABLE_SHELL
- Temperatures obtained from 3rd party thermal analysis of fire on one side of wall
- Point load to top of wall
- Gravity loading
- Steel modelled using *MAT_ELASTIC_VISCOPLASTIC_THERMAL
- Concrete modelled using *MAT_CONCRETE_EC2

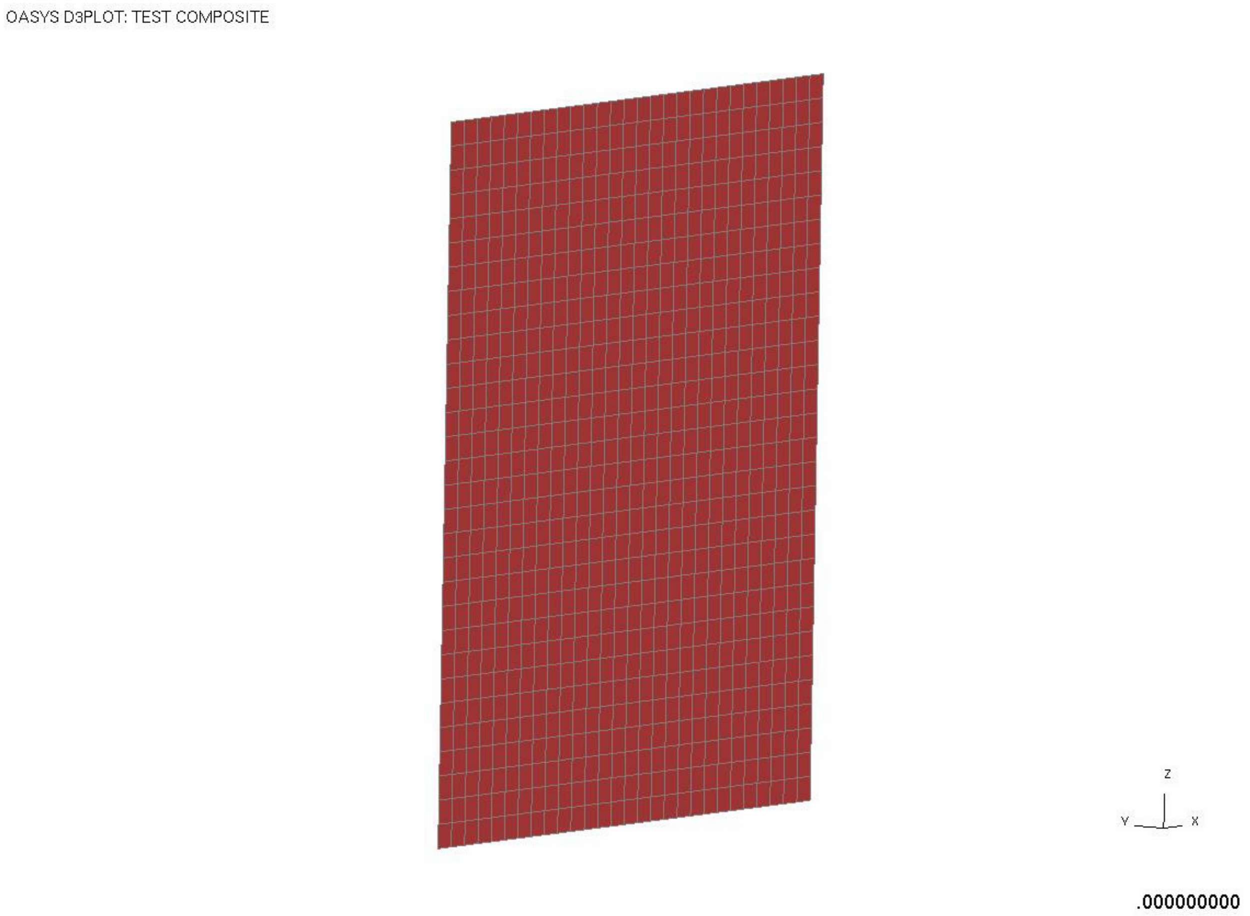


LOAD_THERMAL_VARIABLE_SHELL - Example

```
*LOAD_BODY_Z
         2       9.8         0
*LOAD_NODE_POINT
       442         3         2-2011800.0         0
*LOAD_THERMAL_VARIABLE_SHELL_SET
         1         1
      20.0       1.0       500         0   -0.9388
      20.0       1.0       500         0    -0.921
      20.0       1.0       500         0   -0.8777
      20.0       1.0       500         0    -0.763
      20.0       1.0       501         0    -0.453
      20.0       1.0       502         0       0.0
      20.0       1.0       503         0     0.453
      20.0       1.0       504         0     0.763
      20.0       1.0       504         0    0.8777
      20.0       1.0       504         0     0.921
      20.0       1.0       504         0    0.9388
```



LOAD_THERMAL_VARIABLE_SHELL – Example



PORE FLUID

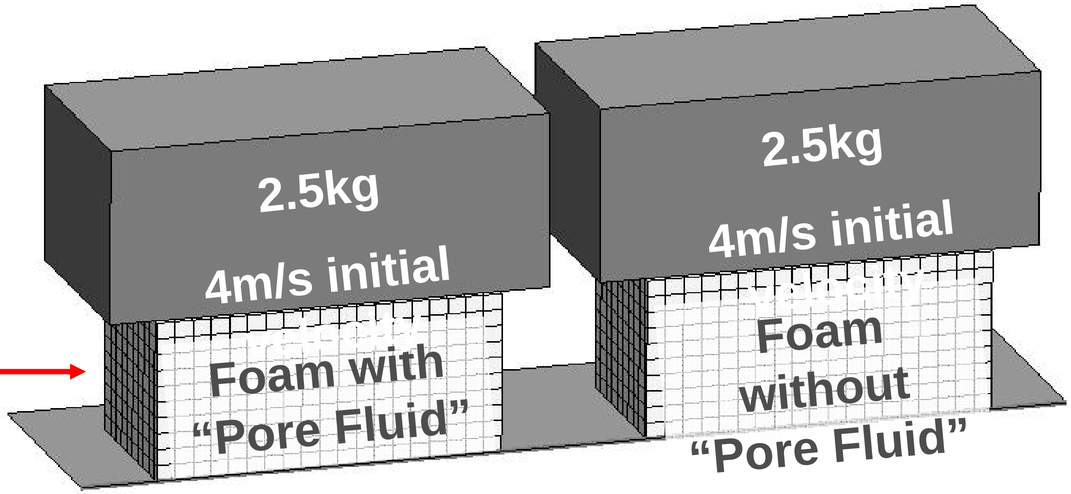
- Allows the modelling of pore water pressure dissipation within soils
- Defined using the following keyword cards – Note: Not in printed LS971 manual.
  - *CONTROL_PORE_FLUID
  - *BOUNDARY_PWP_*{OPTION}*
  - *DATABASE_PWP_FLOW
  - *DATABASE_PWP_OUTPUT
  - *MAT_ADD_PERMEABILITY
  - *INITIAL_PWP_DEPTH
- Originally developed for consolidation in soils.

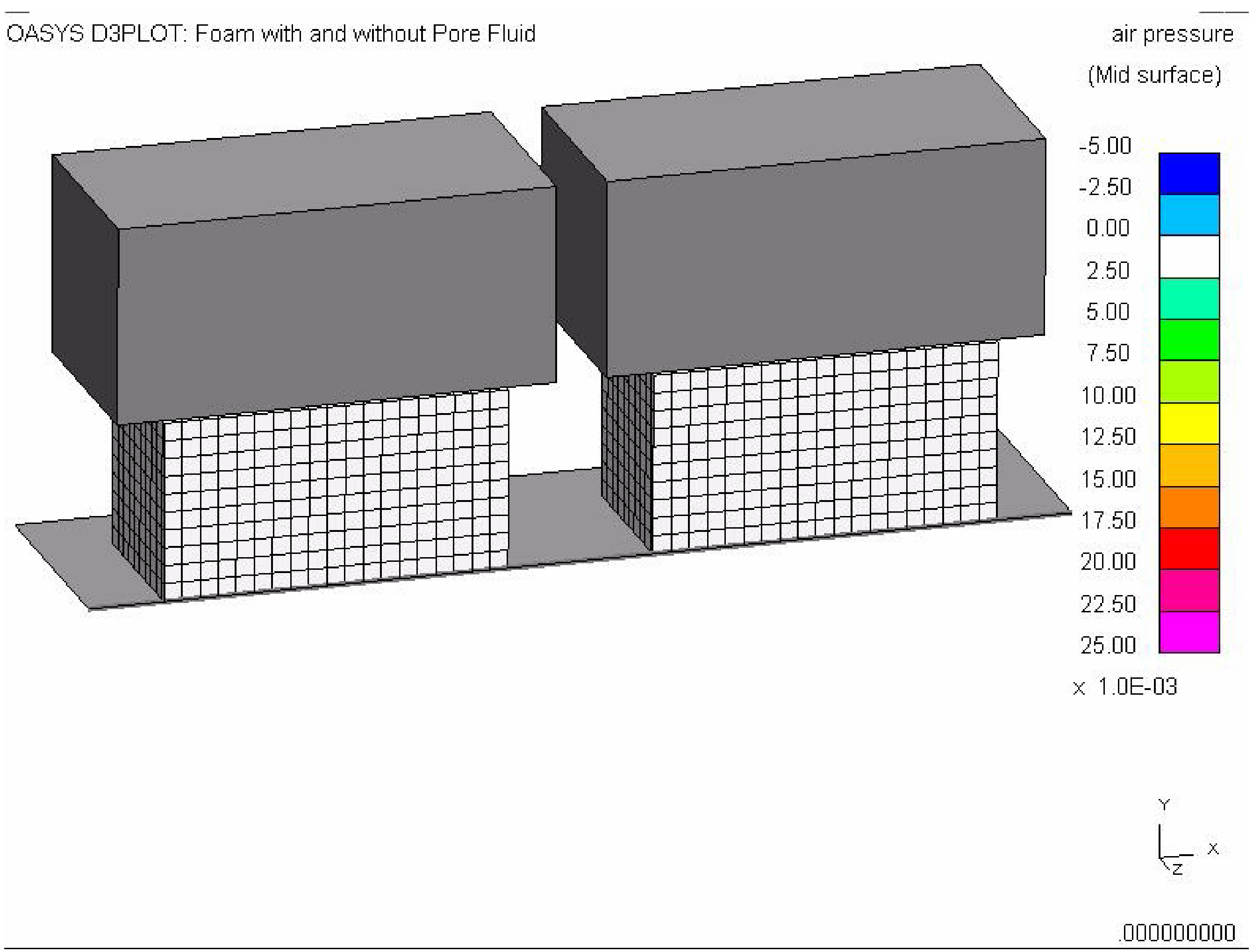
- Example showing use in different application.
- Pore fluid treatment can be added to any LS-DYNA material. The fluid pressure is calculated independently of the material model and added to the total stress. In this case we will input air properties for the pore fluid.

PORE FLUID EXAMPLE

- Motion of fluid through the mesh is governed by Darcy's law: v = -k(∇ pf)
  - » v = fluid velocity vector
  - » k = permeability
  - » pf = fluid pressure
- Two blocks of foam, identical except that "Pore Fluid" has been enabled in the left-hand block to represent air within the foam. Fluid properties are defined, together with the permeability K.

Sides of block have boundary condition: air pressure = 0

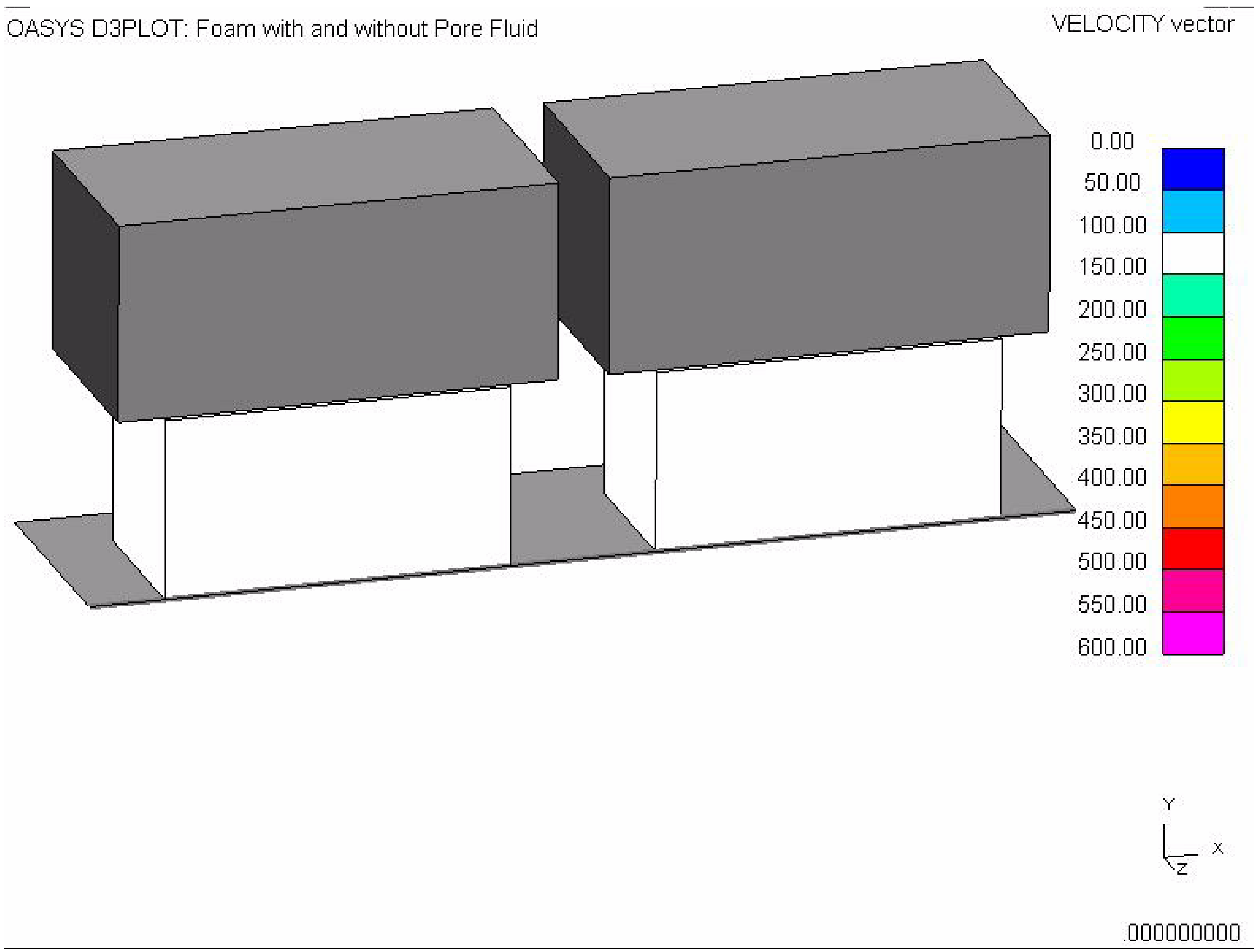




With Pore Fluid **Without Pore Fluid**

Air pressure rises as the block is compressed. The air escapes through the sides of the block, so the pressure reduces towards the sides. During rebound, suction develops (blue contours).

The air pressure in the left-hand block adds to the force on the drop-weight and hence reduces total compression of the foam.



Air flows out of the sides of the block during the compression phase. During rebound, air is sucked back in.

With Pore Fluid **Without Pore Fluid**

LS-971_R4

EFG – MESH FREE SHELLS

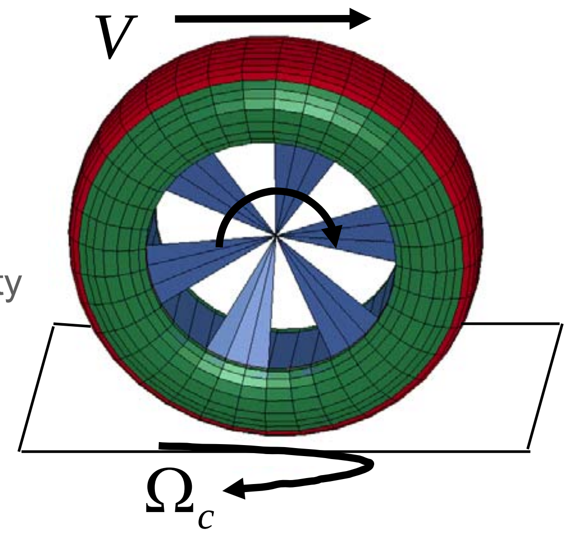
- Under development for the past 4 years
  - Mesh free shells improves accuracy with reasonable efficiency
  - Higher-order approximations yields more accurate solution and smoother stress/strain distribution
  - Adaptive mesh-free shell surface fits better to real surface
  - Implicit implementation for springback
- CPU cost on example forming problem are higher
  - Type 2 - 13.25 minutes
  - Type 16 - 21.41 minutes
  - Type 26 - 31.05 minutes
  - Type 42 - 41.26 minutes (Mesh Free)

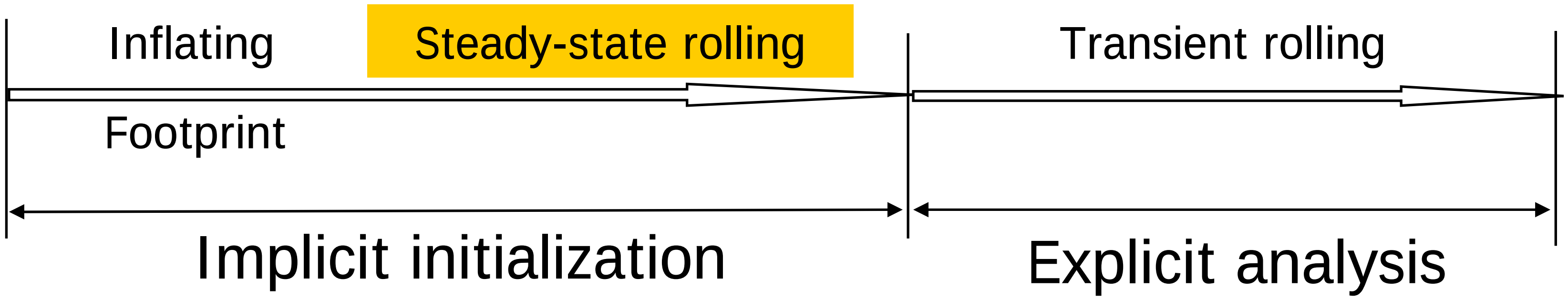
ISOGEOMETRIC SHELLS

- Finite elements using NURBS for basis functions.
  - Allow exact match to dies.
  - Treat as solid to eliminate zero normal stress requirement.
  - Explore through-thickness kinematics and locking.
  - Adaptivity within patches.
  - Explore implicit and implicit solution strategies.
  - Weights as generalized coordinates.
- Dies described by NURBS patches.
  - Smooth contact.
  - Exact geometry.
- CPU costs may be an issue

STEADY STATE ROLLING

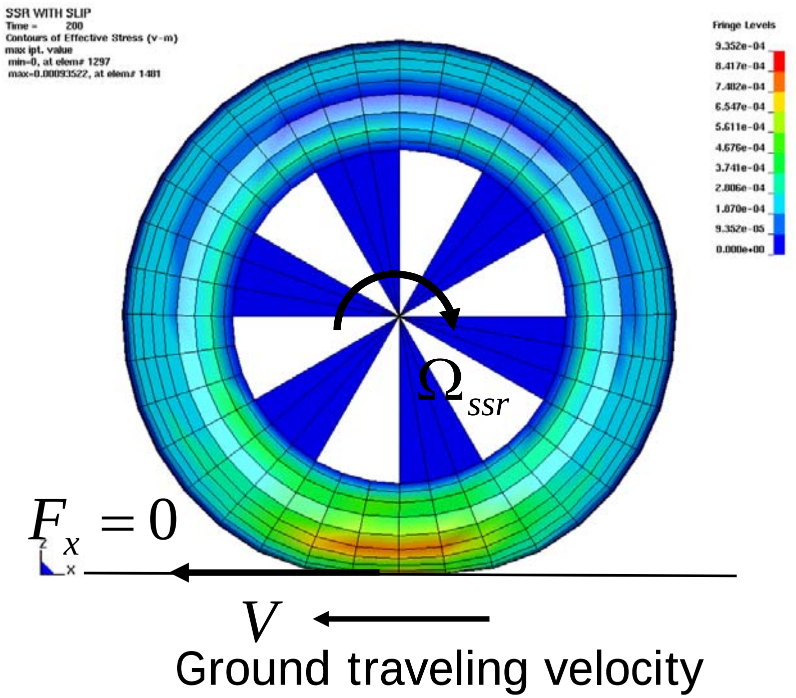
- Quick method to set up tyre rolling problem.
- New keywords
  - *LOAD_STEADY_STATE_ROLLING
    - Define tyre rotation, corning angle and translational velocity
  - *CONTROL_STEADY_STATE_ROLLING
    - Define a curve of the scale factors for friction coefficients

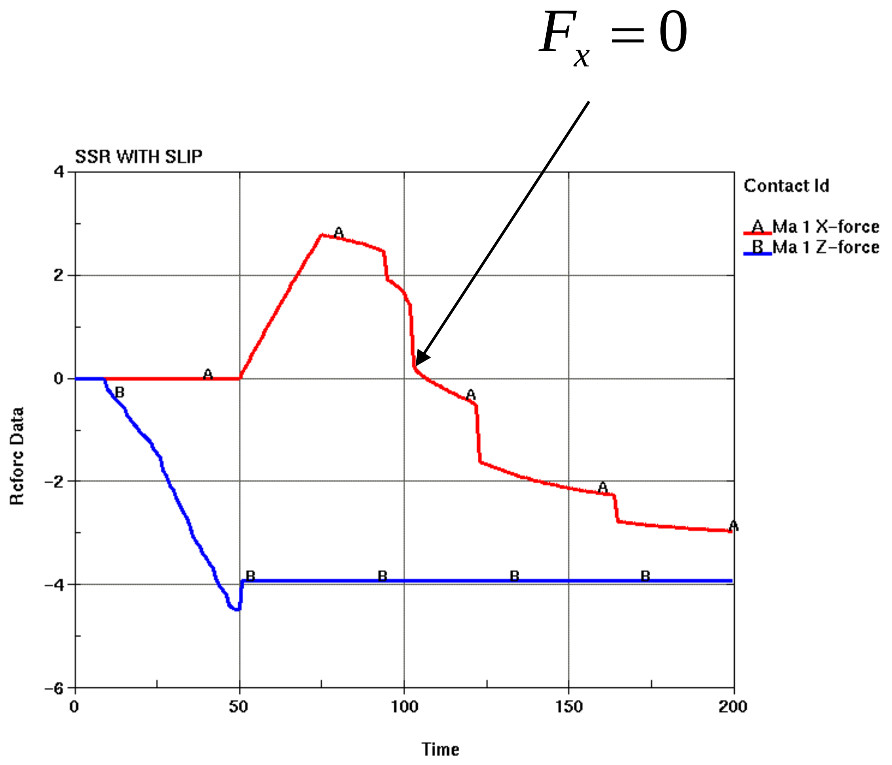




STEADY STATE ROLLING

• Obtain the angular velocity at steady state rolling Ω_{ssr} by checking the contact force at tyre running direction $F_x = 0$ in one run with an input of a curve of angular velocity change.





*DEFINE_FUNCTION

- Arithmetic expression involving a combination of independent variables and other functions, i.e.,
  - f(a,b,c)=a*2+b*c+sqrt(a*c)
  - where a, b, and c are the independent variables.
- The function name, f(a,b,c), must be unique since other functions can then use and reference this function.
  - g(a,b,c,d)=f(a,b,c)**2+d.
- Implemented for two keywords
  - *MAT_024 define a function for the yield stress as a function of plastic strain
    - Timing studies shows only small slowdown over built in functions
  - *BOUNDARY_PRESCRIBED_MOTION
- No change in input is required. If a curve ID is not found, then the function ID's are checked

*INITIAL_AXIAL_FORCE_BEAM

- Beam axial forces can now be initialized to allow a simplified method to model initial tensile force in bolts.
- Works with *MAT_SPOTWELD and beam type 9. Failure of bolts are based on the force resultants as in the spotwelds
- Load curves are used to initialize forces
- Required input requires beam set ID and a corresponding load curve ID

EXTENDED FORMATS

- Customer requests for extended formats are increasing since 8 character IDs are too restrictive
  - All keyword formats are being optionally extended
  - 2 cards will be read for each existing card
  - I10 > I20, I8 > I16, E10.0 > E20.0
  - Mixed default and extended formats will be read
  - Structured input will uniformly go to I20 and E20.0
- Current formats are being kept and all changes will be backwards compatible.
  - All old input files, both structured and keyword, will be read.
- New formats are being added to 971_R4 for early release since preprocessors need to be updated. A 2-5 year lead time is expected.

EXPLICIT CONSTRAINTS

- Historically, LS-DYNA Explicit has applied constraints each time step by constraint type.
  - Multi-point constraints
  - Tied constrained contacts
  - Joint constraint
  - Rigid bodies
- In explicit calculations constraints involving the same node but different constraint types may not be properly applied.
  - A node cannot be both a dependent node in one constraint and an independent node in another
- LS-DYNA Implicit uses a global view of constraints so multiple constraints are consistently applied.
  - Constraints can form closed chains
- LSTC has started a project entitled **Consistent Constraint Explicit, CCE**, where the Implicit technology will be applied to explicit problems.

EXPLICIT CONSTRAINTS

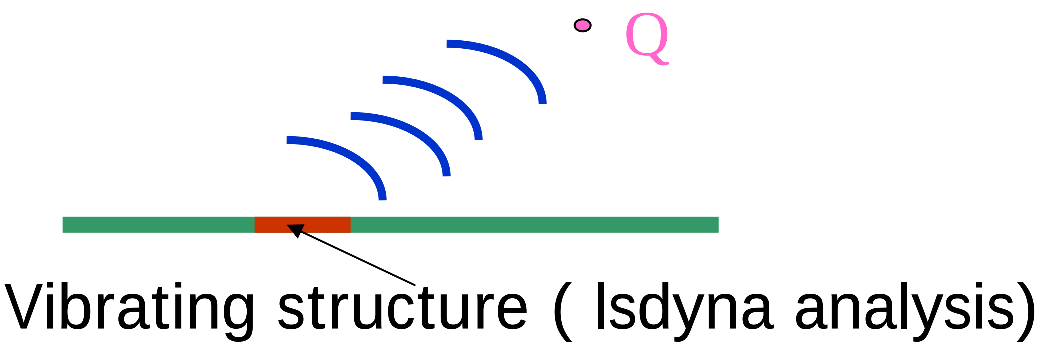
- Build a global constraint matrix C and associated right hand side g.

$$\begin{aligned} &Solve && Ma = f \\ &Subject\ to && Ca = g \end{aligned}$$

- LSTC are looking at using an iterative solver on the null space of the constraints.
- Results will be a consistent application of constraints for explicit models.
- It is expected that CPU time and computer memory requirements will be increased for explicit simulations.
    - Approach will be optional
    - No input changes required

BOUNDARY ELEMENT METHOD FOR ACCOUSTICS

The fluid mesh is not needed, only the structural mesh is required



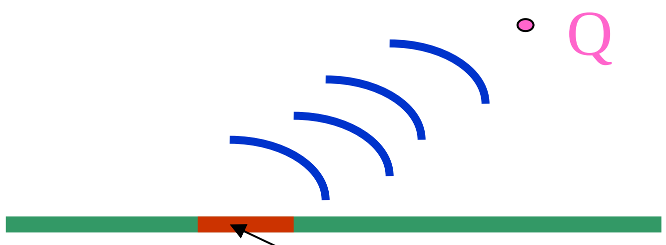
BOUNDARY ELEMENT METHOD FOR ACCOUSTICS

The fluid mesh is not needed, only the structural mesh is required

V : velocity

p : pressure on structure boundary computed in lsdyna (BEM Method)

G : Green's function



Vibrating structure (lsdyna analysis)

$$P(Q) = \int_{Boundary} \left(i\rho\omega v(\omega) G + p(\omega) \frac{\partial G}{\partial n} \right) ds$$

BOUNDARY ELEMENT METHOD FOR ACCOUSTICS

- LS971 i=input_file.key bem="filename"

LSDYNA - Structure analysis

Velocities or Acceleration stored in a binary file as specified by "filename" on command line.

FFT - Compute Velocities in frequency domain

Compute Acoustic pressure at any location in the frequency domain

ASCII Curve files

window_fft, Press_db, Press_Pa

BOUNDARY ELEMENT METHOD FOR ACCOUSTICS

***BOUNDARY_ELEMENT_METHOD_ACOUSTIC**

• Keyword format has changed and is not as given in Dyna manual.

Card 1

Variable	Dens	Sound Speed	Min Freq	Max Freq	NFreq	Dt Output	Start Time	Ref Pres
Type	F	F	F	F	I	I	F	F
Default	none	none	none	none	0	0	0	0

Card 2

Variable	Ext Pres	Type Ext	Int Pres	Type Int	FFT Window			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

BOUNDARY ELEMENT METHOD FOR ACCOUSTICS

Card 3

Variable	**BEM Method**	Max Iter	Residual	NDD				
Type	I	I	F	I				
Default	0	100	1e-6	1				

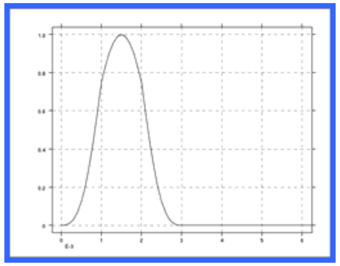
Card 4

Variable	SSID	SSTYPE	Norm	BEM Type	Restart			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

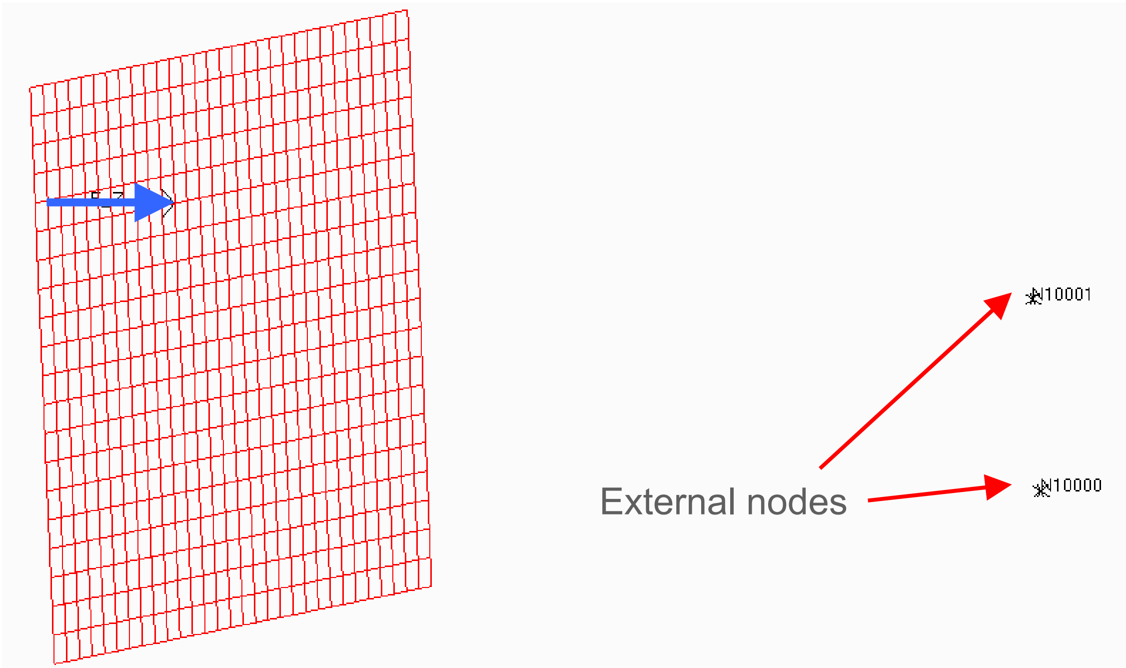
BOUNDARY ELEMENT METHOD FOR ACCOUSTICS

- **BEM Method**
  - 0 Rayleigh method (Not BEM)
  - 1 Kirchhoff method coupled to FEM (*MAT_ACOUSTIC) (Not BEM)
  - 2 **Boundary Element Method**

•Example.
  - Vibrating Plate
  - Excited by point load
  - Two external nodes

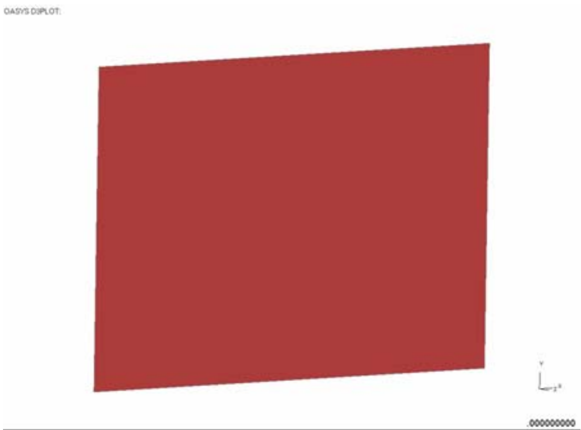


Point Load

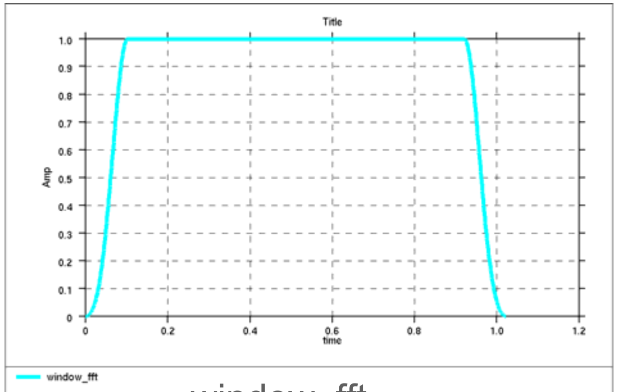


BOUNDARY ELEMENT METHOD FOR ACCOUSTICS

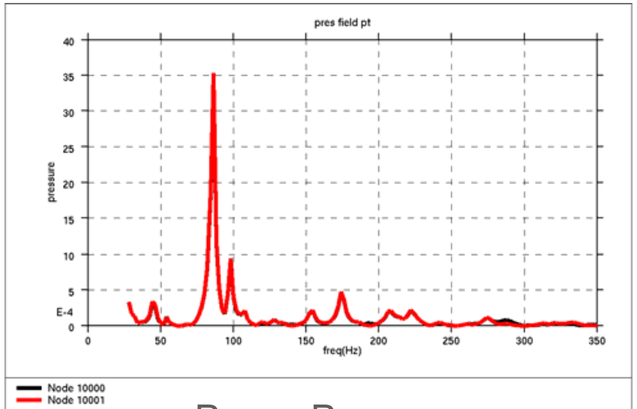
```
*BOUNDARY_ELEMENT_METHOD_ACOUSTIC
$      dens   sp-sound   min-freq   max-freq      nfreq         dt    t_start   ref_pres
       1.21       340.        28.       350.        162      1.e-3        0.0    20.e-6
$ Ext_Pres   Type_Ext   Int_Pres   Type_Int  FFT_window
          1          1          0          0          4
$ IBEM_Met   Max_iter        Res        NDD
          2
$      SSID     SSTYPE       Norm   BEM_Type    Restart
          1          1          0          0          0
$
*SET_NODE
          1
10001,10000
*NODE
   10000 0.300000012E+00 0.150000006E+00 1.000000000E+00
   10001 0.300000012E+00 0.350000006E+00 1.000000000E+00
```



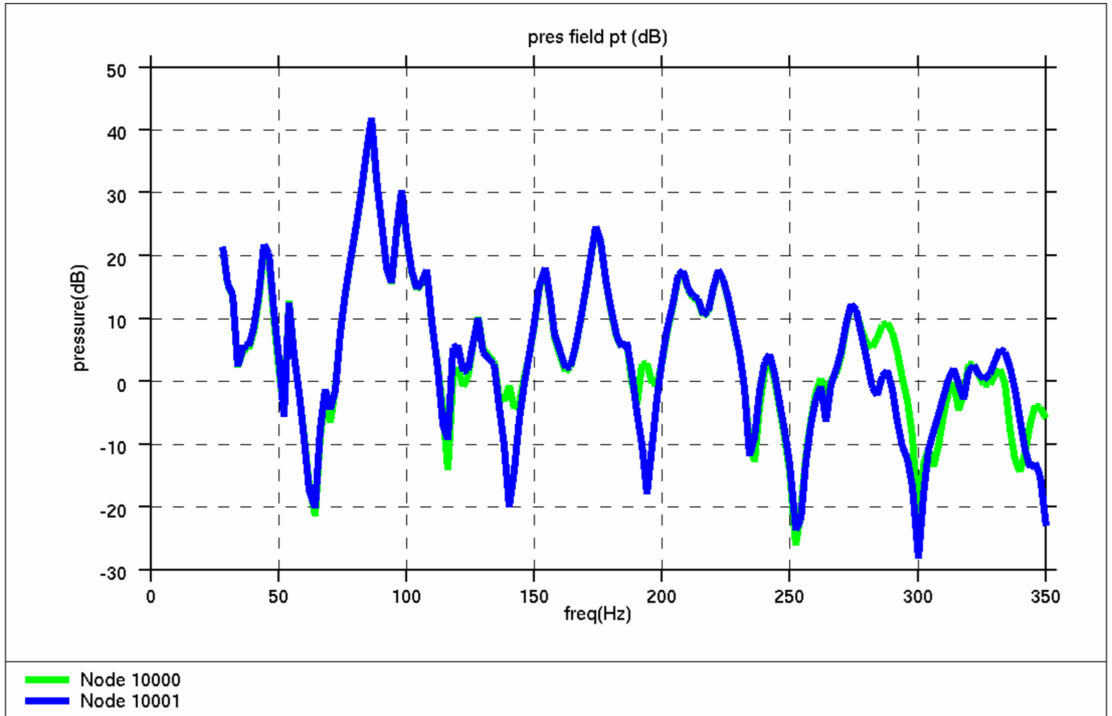
BOUNDARY ELEMENT METHOD FOR ACCOUSTICS



window_fft



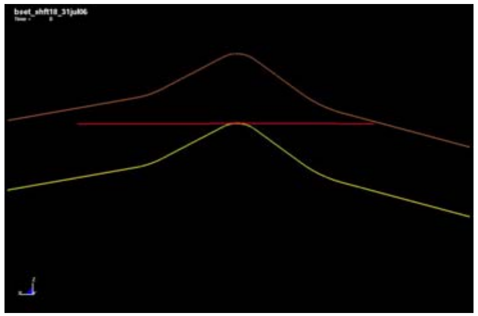
Press_Pa



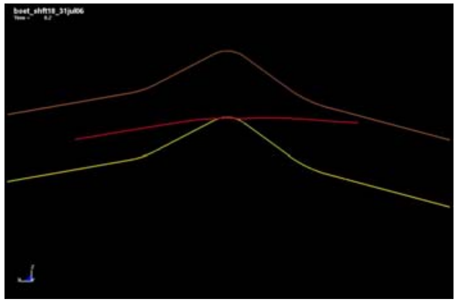
press_db

METALFORMING ENHANCEMENTS

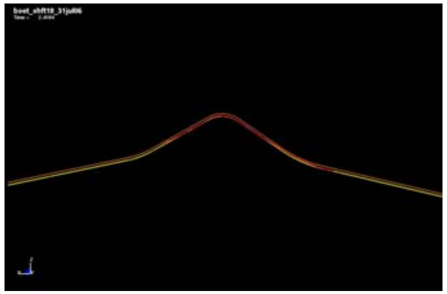
- **STATIC IMPLICIT BINDER WRAPPING**
  - A new algorithm has been developed
    - Automatically determine time step
    - More robust calculations
    - Reduce dynamic effect
    - Reduce CPU costs
    - Can be combined with gravity loading simulation



Initial Position



Gravity Loading



Final Position

METALFORMING ENHANCEMENTS

- **MODELLING OF ELASTIC DIE STRUCTURES IN THE STAMPING PROCESS**
  - Durability of die
  - Deflection effects on stamping
  - Static condensation to create die superelements
  - All surface nodes are kept and internal degrees-of-freedom are eliminated
  - Keyword: *CONTROL_IMPLICIT_STATIC_CONDENSATION
  - Equations for die motion are integrated explicitly
  - Stress recovery for durability analysis
  - Two options
    - Create superelements prior to analysis
    - Define an elastic part set in LS-Dyna input

LS-980

LS-980

- **New compressible fluid solver**
  - CESE - Conservation Element & Solution Element
  - Flux conservations in space and time (locally & globally)
  - Both strong shocks and small disturbances can be handled very well simultaneously
  - Boundary conditions can be implemented easily & accurately
  - Some applications:
    - All speed compressible flows
    - Low speed, High speed flows (subsonic, hypersonic)
    - Especially for high speed flows with complex shock patterns
    - Acoustics (noise)

- **Incompressible fluid solver**
  - Error Control and remeshing simplifies pre-processing.
  - MPP implementation.
  - Allows weak and strong coupling depending upon the problem.
  - Separate meshes for fluid and structure.
  - Coupling to explicit and implicit mechanics.

LS-980

- **Electromagnetism**
  - Introduction of electrical currents in solid conductors.
  - These currents generate magnetic fields, electric fields, as well as induced currents.
  - The magnetic fields coupled with the currents generate Lorentz forces on the conductors.
  - The forces induce motion and deformation of the conductors.
  - This motion has an effect on the fields and the currents.
  - The currents generate Joule heating in the conductors, changing the temperature, and thus some mechanical as well as electromagnetic properties (conductivity for example)

Contact Information

www.arup.com/dyna

For more information please contact the following:

UK:

Arup
The Arup Campus
Blythe Valley Park
Solihull, West Midlands
B90 8AE, UK
T +44 (0)121 213 3399
F +44 (0)121 213 3302
dyna.support@arup.com

USA:

Arup
1625 West Big Beaver
Suite C
Troy
Michigan 48084, USA
T +1 248 822 5050
F +1 248 822 4072
us.support@arup.com

China:

Arup
39/F-41/F Huai Hai Plaza
Huai Hai Road (M)
Shanghai
China 200031
T +86 21 6126 2875
F +86 21 6126 2882
china.support@arup.com

India:

nHance Engineering Solutions Pvt. Ltd (Arup)
101 Cyber Heights
Plot 13, Road No. 2
Banjara Hills, Hyderabad 500033
India
T +91 40 2354 4420
india.support@arup.com