LS-DYNA in Ansys Workbench
- Case study of ROPS and FOPS simulation

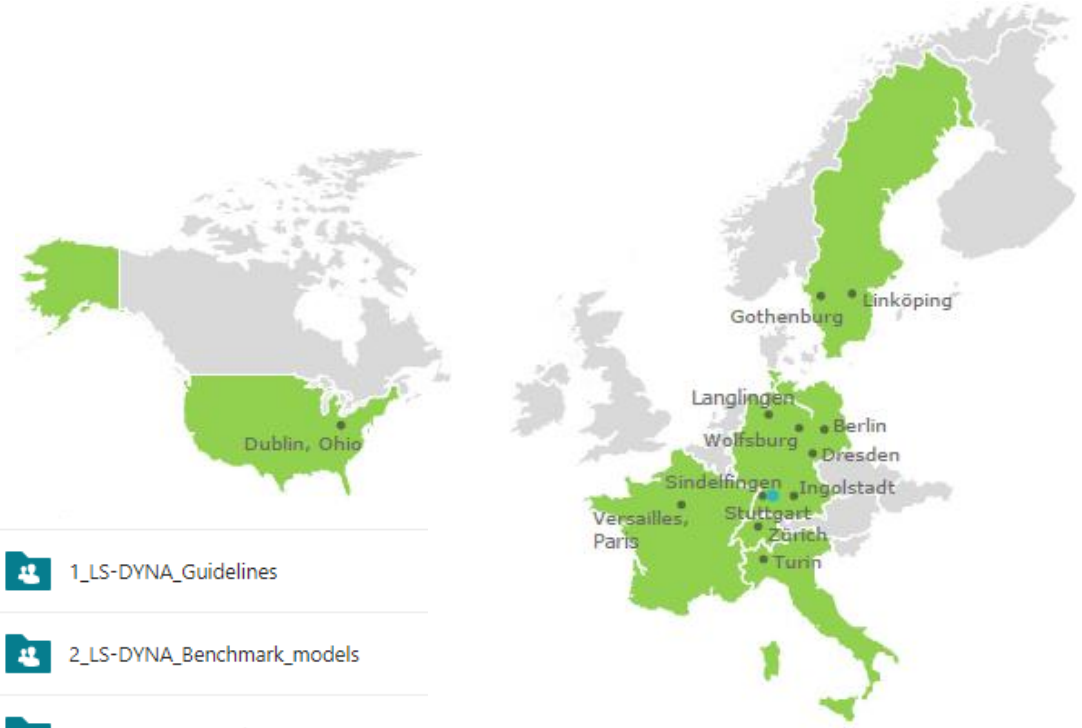
Anders Bernhardsson, DYNAmore Nordic AB
anders.bernhardsson@dynamore.se

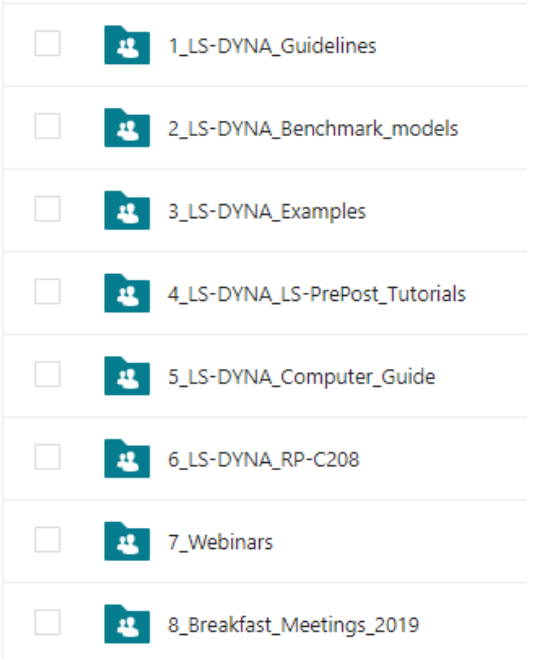
Content

- DYNAmore Nordic
- Background
- What is LS-DYNA
- What is Ansys Workbench
- LS-DYNA Workbench integration
- Case study: ROPS and FOPS
- Additional input to LS-DYNA model
- Conclusion and Future

DYNAmore Group

- Totally ~100 employees
- Developers of ANSYS/LST: 14
- HQ: Stuttgart, Germany
- Nordic offices: Linköping & Göteborg
- Webinars: 1 per week
- Guidelines for DYNAmore customers
- Information
  - www.dynamore.se
  - www.dynasupport.com
  - www.dynalook.com





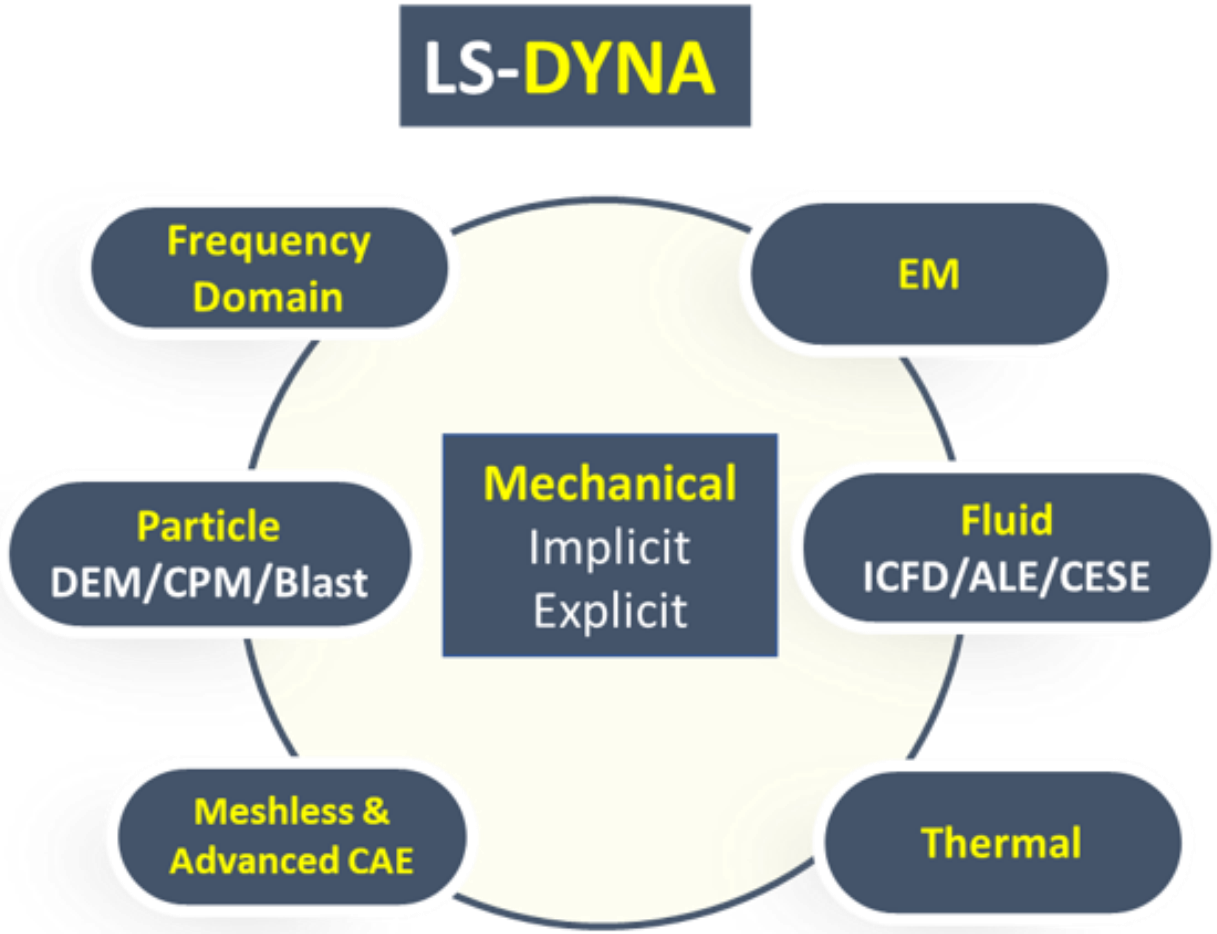
Cooperation between DYNAmore Nordic, EDRMEDESO and ANSYS

- ANSYS/LS-DYNA
  - EDRMEDESO customers can purchase ANSYS/LS-DYNA licenses from EDRMEDESO and get support from DYNAmore Nordic if this is agreed before the purchase
  - DYNAmore Nordic customers can purchase ANSYS/LS-DYNA or LSTC/LS-DYNA licenses from DYNAmore Nordic

Background

- Even before Ansys acquisition of LSTC it was possible to use the LS-DYNA solver through Ansys products
- The LS-DYNA integration into Ansys Mechanical have existed many years by the Workbench LS-DYNA module
- The module allows the user to set up an LS-DYNA keyword model and postprocess the results in Ansys Mechanical

What is LS-DYNA – One Code for multi-physics solutions



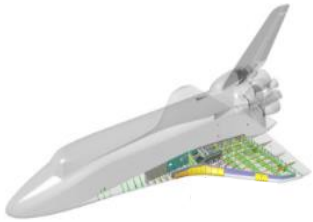
Note: All solvers are included in the license

What is LS-DYNA - Applications

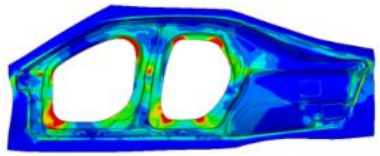
- Crash and impact
- Manufacturing processes
- Overload
- Misuse
- Drop tests
- Advanced material models, failure etc.
- Large models
- Contacts
- Fatigue incremental or dynamic
- FSI
- Explosions

Automotive

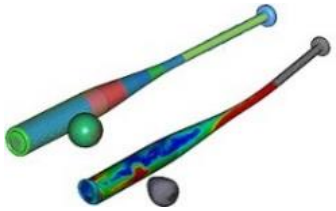
Aerospace



Manufacturing



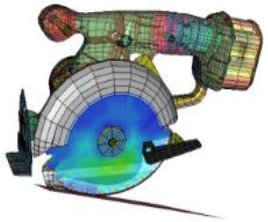
Consumer Products



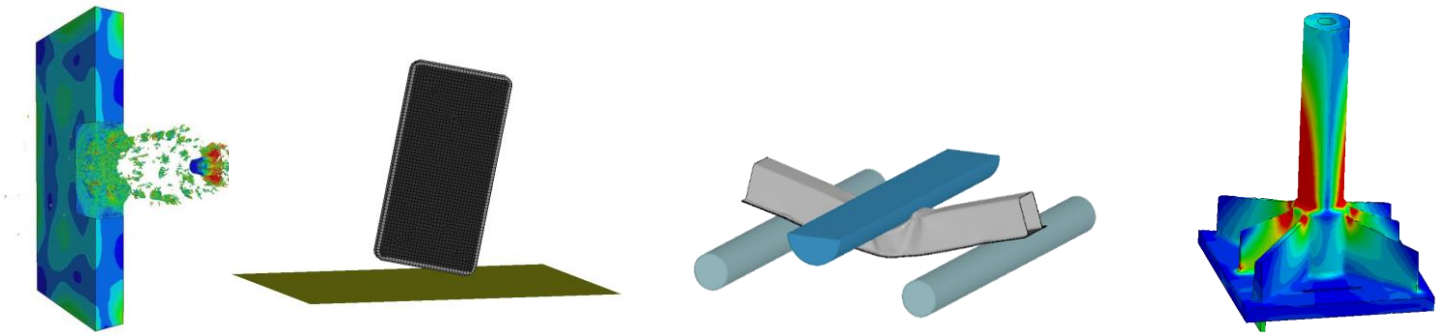
Biomechanics

Defense

Electronics



What is LS-DYNA – Explicit vs. Implicit solver



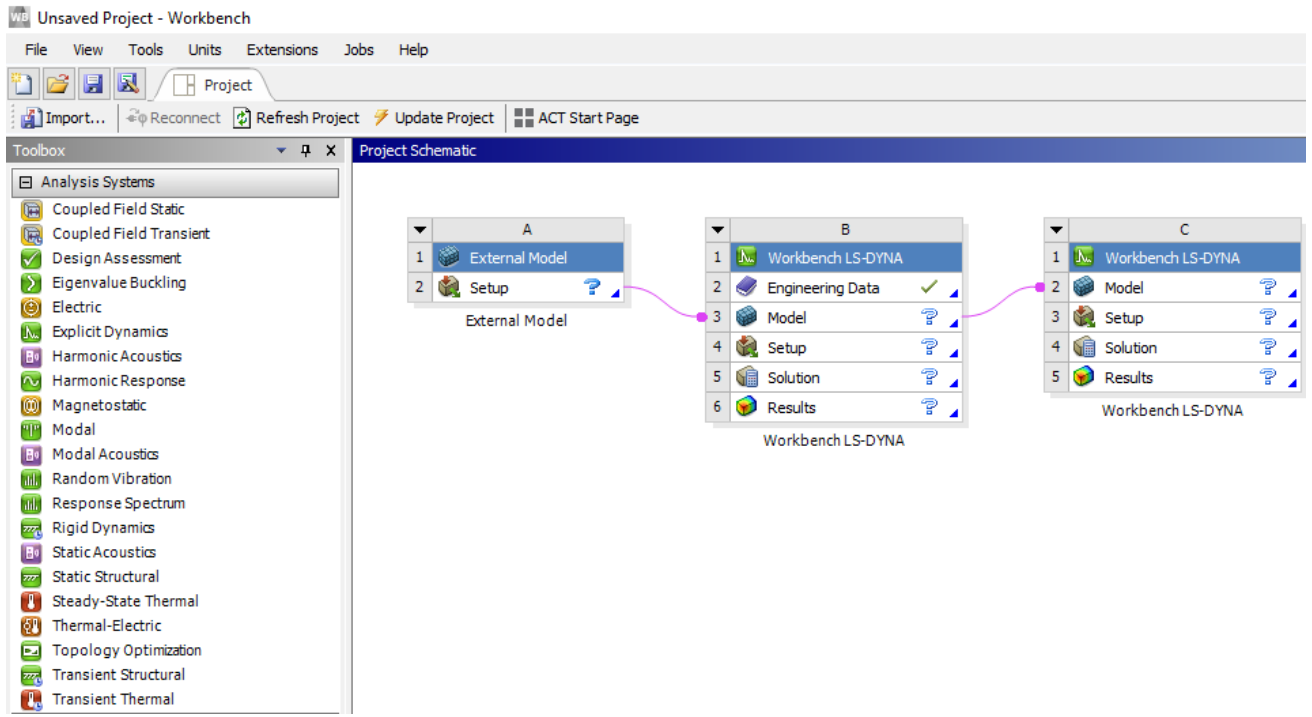
Wave Propagation	Structural Dynamics	Quasi-static	Static

Explicit Solver

Implicit Solver

What is Ansys Workbench

- The Ansys Workbench interface gives the user an easy overview of the project
- By using different Analysis Systems in the project it is possible to transfer models and results between the systems



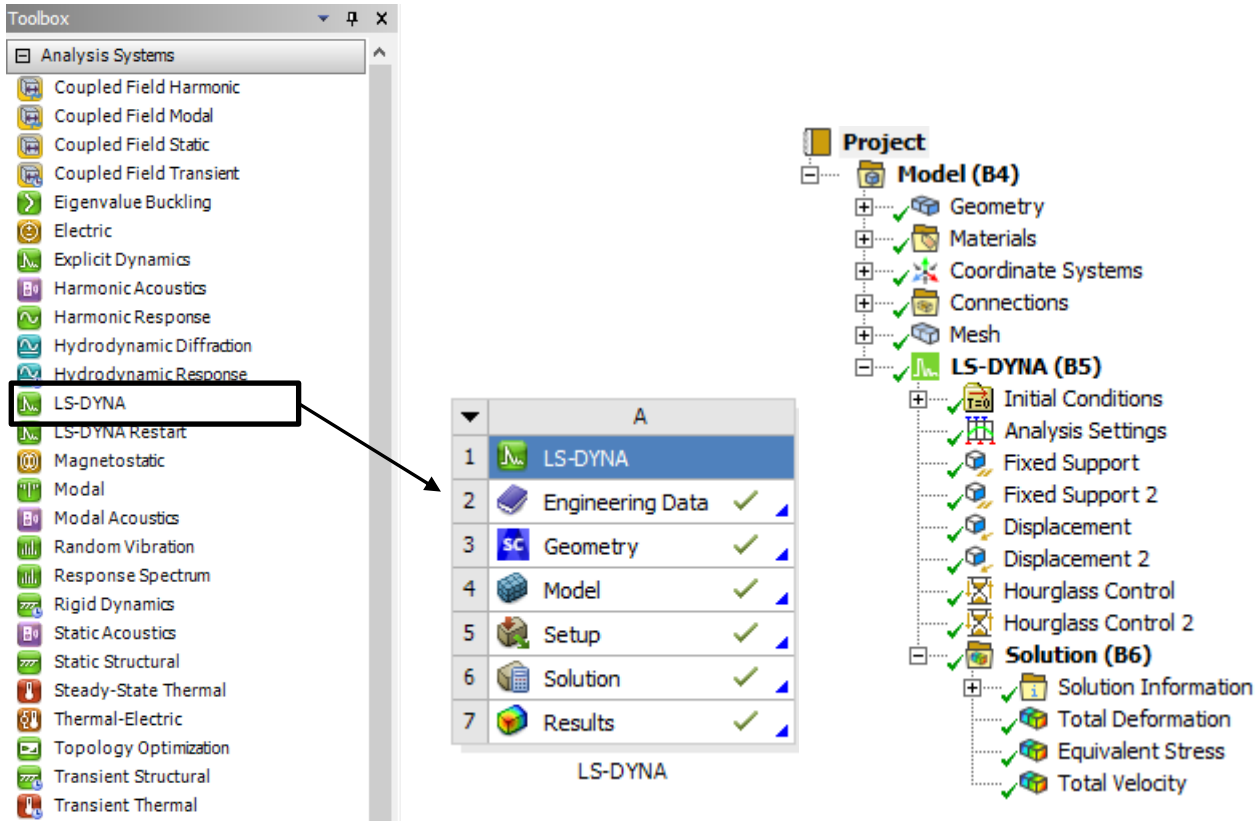
- The Analysis Systems are made up from different components called Cells. These Cells can either be opened inside Workbench interface or launch a separate application.

LS-DYNA Workbench integration

- Previously the LS-DYNA system was included as an extension or ACT (Ansys Customization Toolkit), but from Ansys 2020 R2 it is included as a regular Analysis System

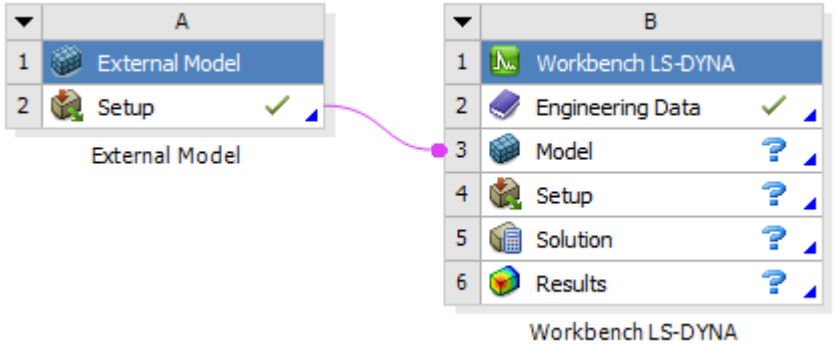
- The system consists of the following cells:
  - Engineering Data: Material database for the analysis
  - Geometry: Create or modify CAD model in DesignModeler or SpaceClaim
  - Model:
  - Setup:
  - Solution:
  - Results:

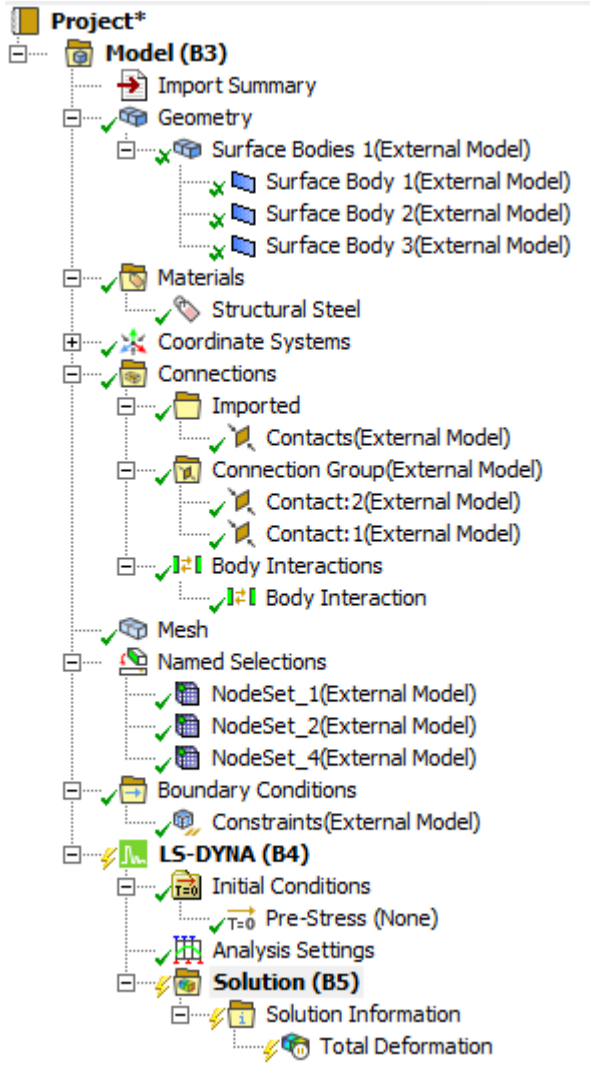
  FE setup; meshing, analysis settings, post-processing. All these steps are performed in Ansys Mechanical



LS-DYNA Workbench integration

- Import a keyword file into Workbench
- Supported keywords to import
  - Nodes and elements
  - Parts
  - Node sets
  - SPCs
  - Contact regions
  - Initial thickness and stresses (from dynain)

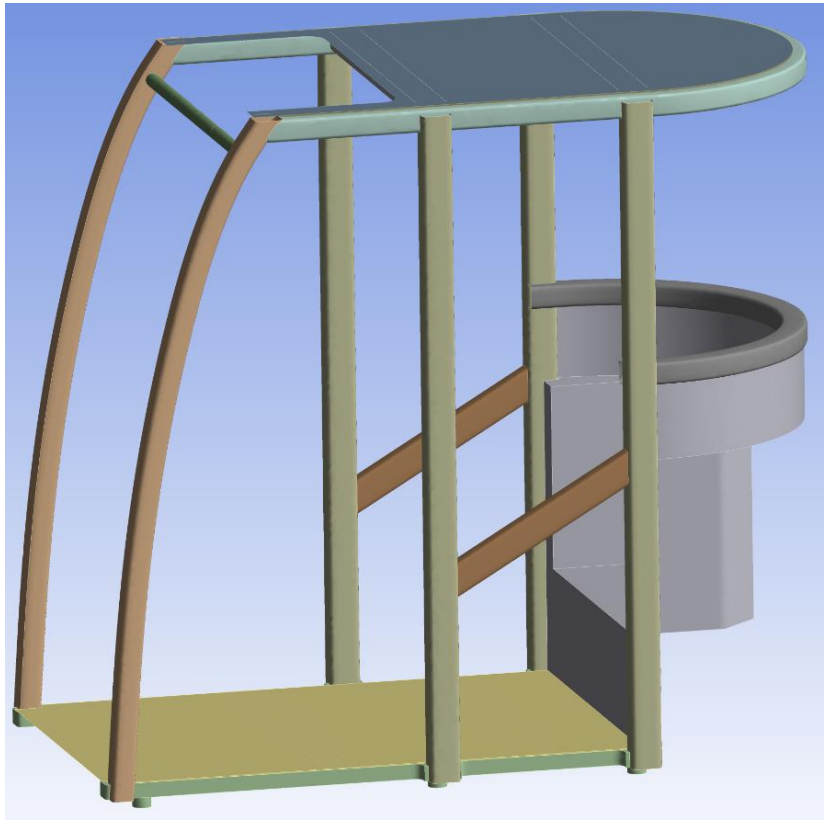




Case study: ROPS and FOPS

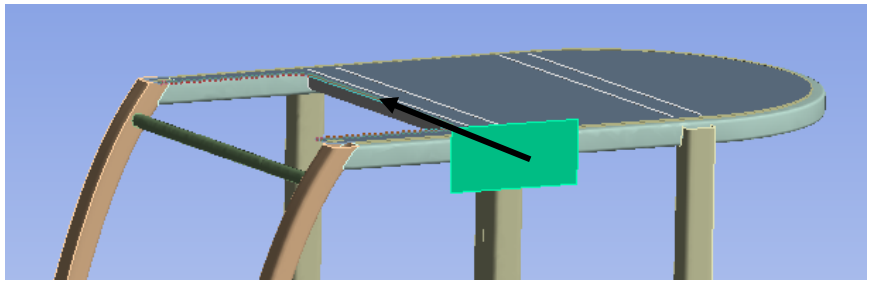
- ROPS : Roll Over Protective Structure
- FOPS : Falling Object Protective Structure

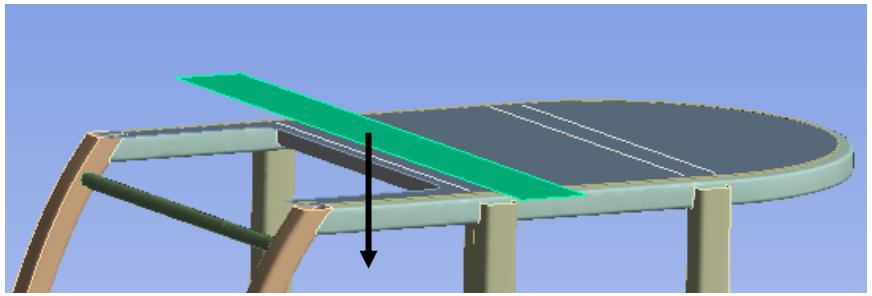
- Used in agricultural, forestry, earth-moving and mining machinery
- The testing procedures for these are harmonized in different standard documents

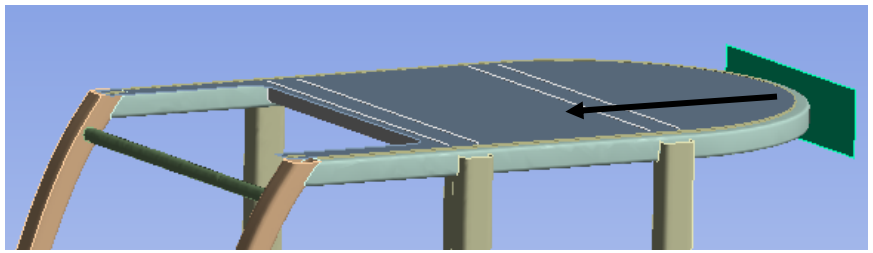


ROPS

- ROPS test procedure
  - Load cases are to be performed in a sequence
  - In this example:
    - Lateral
    - Vertical
    - Longitudinal
  - The loading requirement depends on the machine type and weight
  - For some load cases, both a force and energy requirement must be fulfilled
  - The ROPS structure must not be deformed so it enters the area occupied by the driver

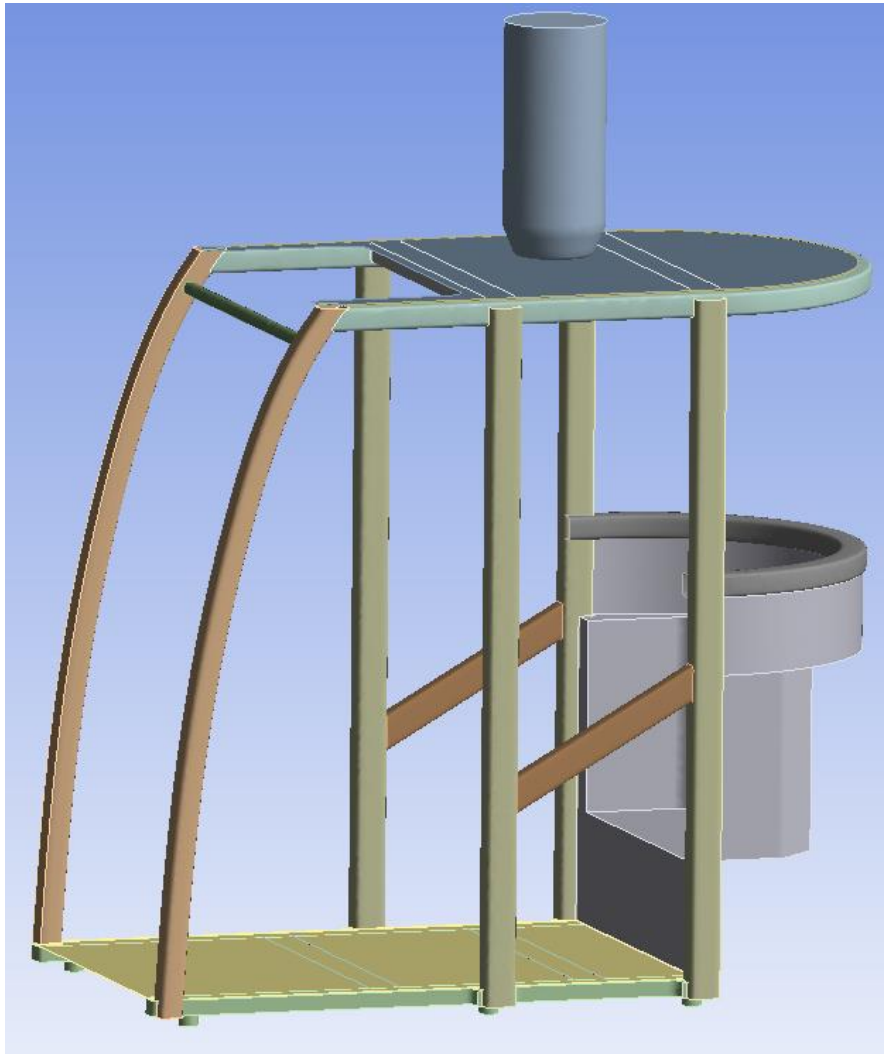






FOPS

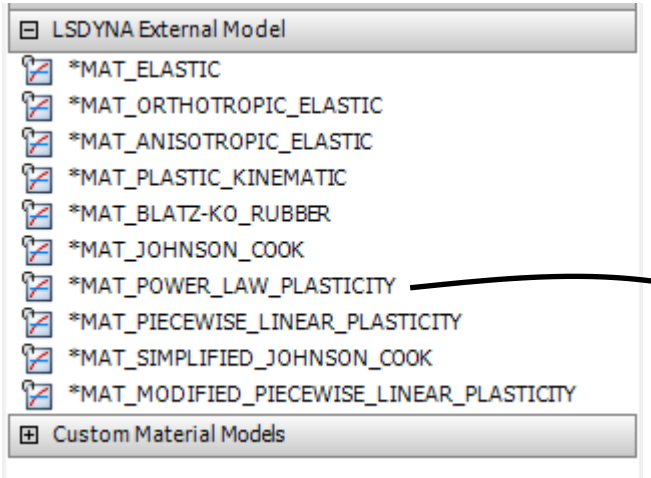
- FOPS test procedure
  - A test object is impacting the top of the FOPS
  - The mass and initial velocity of the test object determines the impact energy
  - The FOPS structure must not be deformed so it enters the area occupied by the driver



Material database

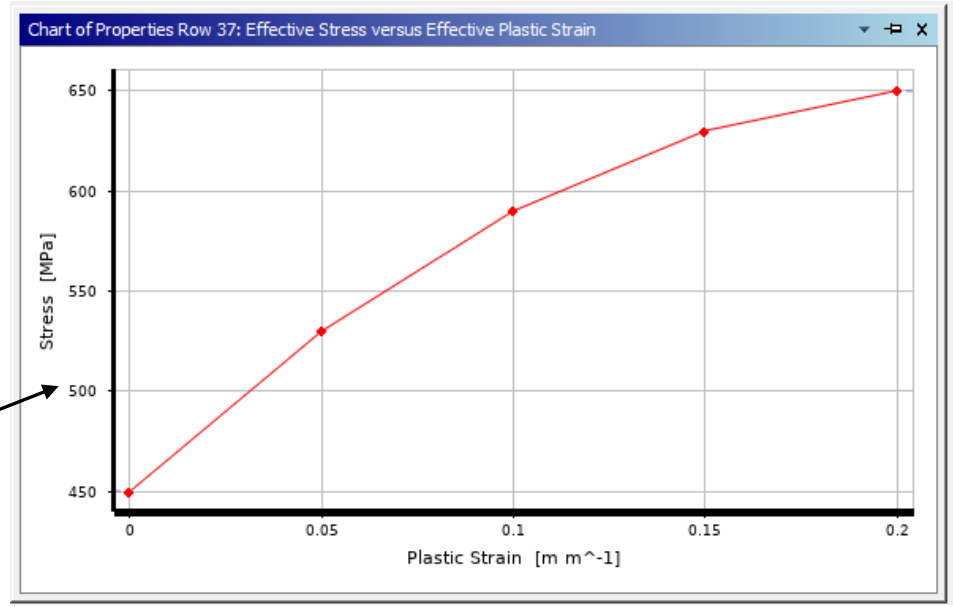
- From version 2021 R1 it´s now possible to select LS-DYNA material models directly in Engineering Data
- Other LS-DYNA material models are available from the database, but is not visible directly in Engineering Data

	Plastic Strain (m m^-1)	Stress (MPa)
2	0	450
3	0.05	530
4	0.1	590
5	0.15	630
6	0.2	650



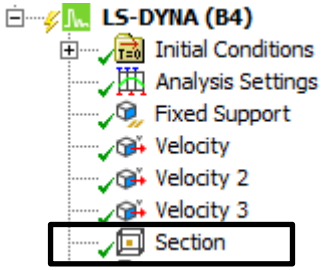
Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	7850	kg m^-3		
4	Isotropic Elasticity				
5	Derive from	Young's ...			
6	Young's Modulus	2E+11	Pa		
7	Poisson's Ratio	0.3			
8	Bulk Modulus	1.6667E+11	Pa		
9	Shear Modulus	7.6923E+10	Pa		
10	Isotropic Thermal Conductivity	60.5	W m^-1...		
11	Specific Heat Constant Pressure, C_p	434	J kg^-1 ...		
12	*MAT_PIECEWISE_LINEAR_PLASTICITY				
13	Definition				
37	Effective Stress versus Effective Plastic Strain	Tabular			
38	Scale	1			
39	Offset	0	MPa		



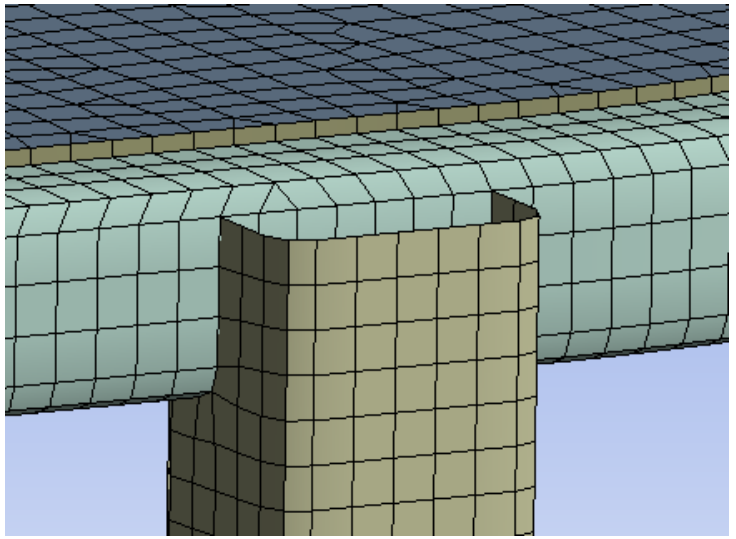
Elements

- As the explicit solver will be used for the simulations, aim for a uniform mesh in the model
- By default, under integrated elements are used for both shell and solid elements
- Possible to change the element formulation and the number of integration points through thickness
- Hourglass controls can be modified in Analysis Settings



Details of "Section"

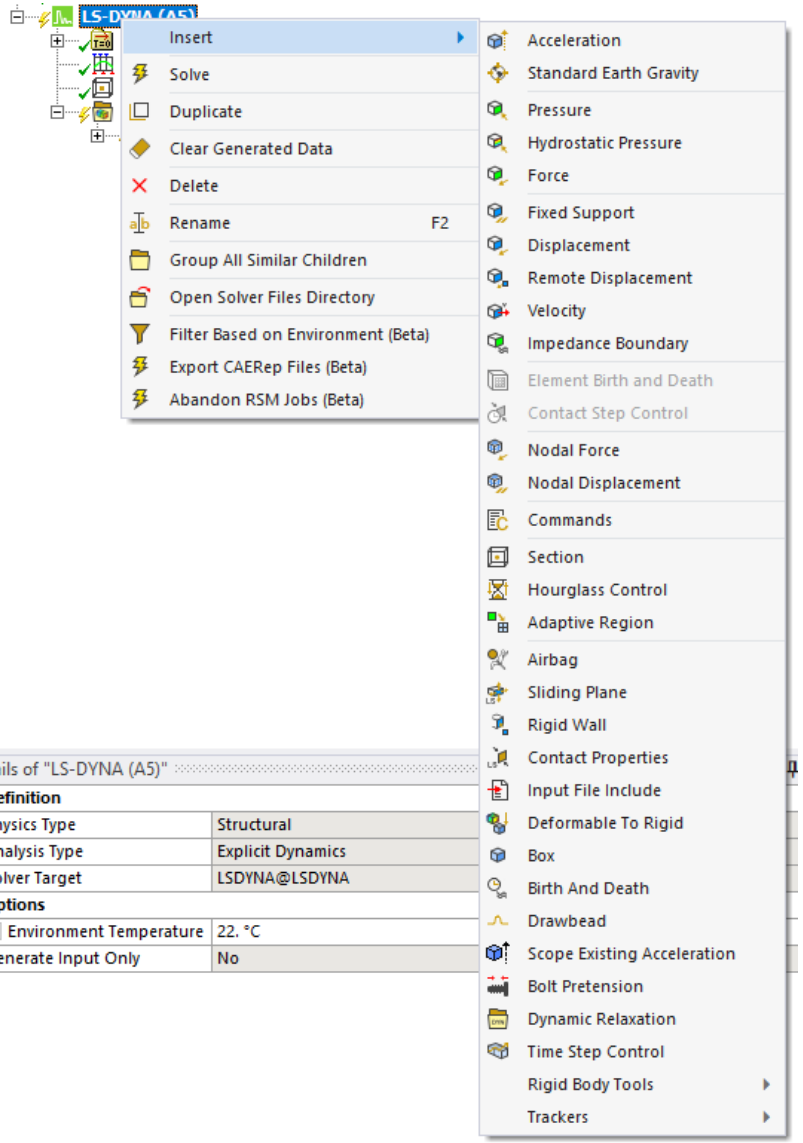
Geometry	
Scoping Method	Geometry Selection
Geometry	1 Body
General	
ALE	No
Formulation	Belytschko-Tsay
LS-DYNA ID	2
Type	Section Shell
Through Thickness Integration Points	5 Point

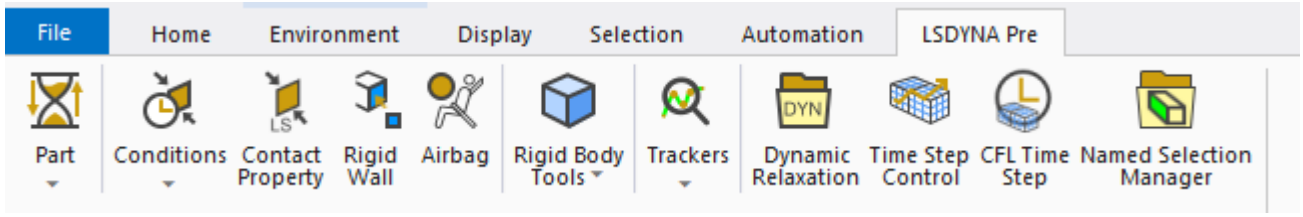


Hourglass Controls	
Hourglass Type	Flanagan-Belytschko Stiffness Form
LS-DYNA ID	4
Default Hourglass Coefficient	0.1

Boundary Conditions

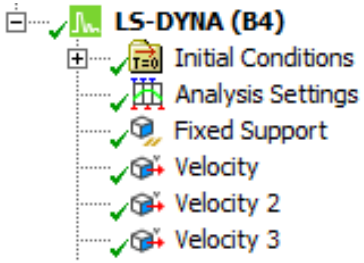
- The typical boundary and initial conditions used in a structural analysis can be selected
- The specific keywords are not seen when choosing the boundary conditions, but the naming of the inputs shows quite easily which LS-DYNA keyword that will be used

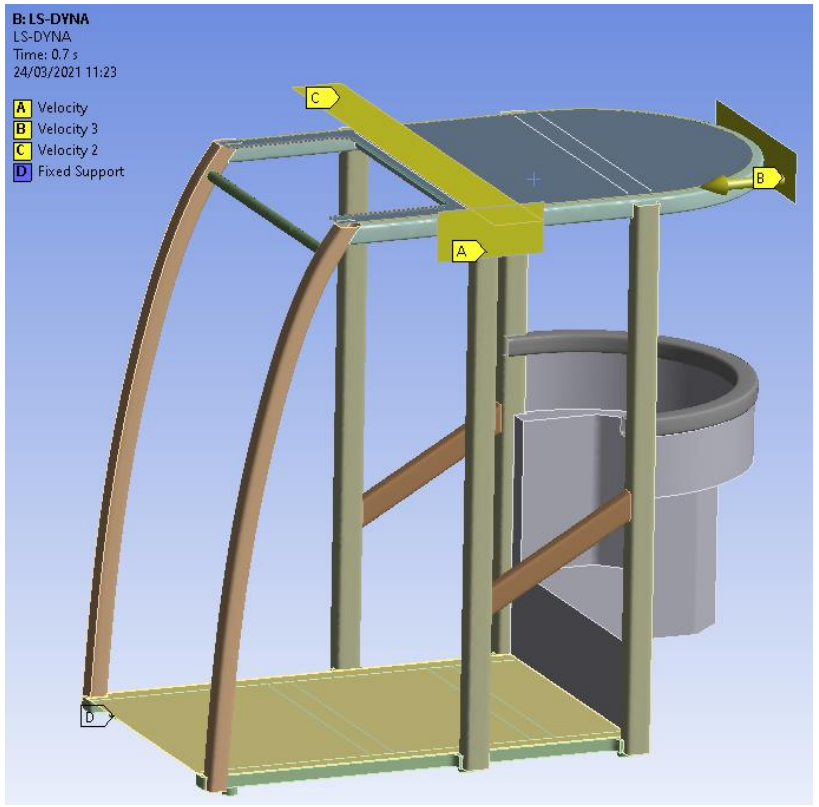


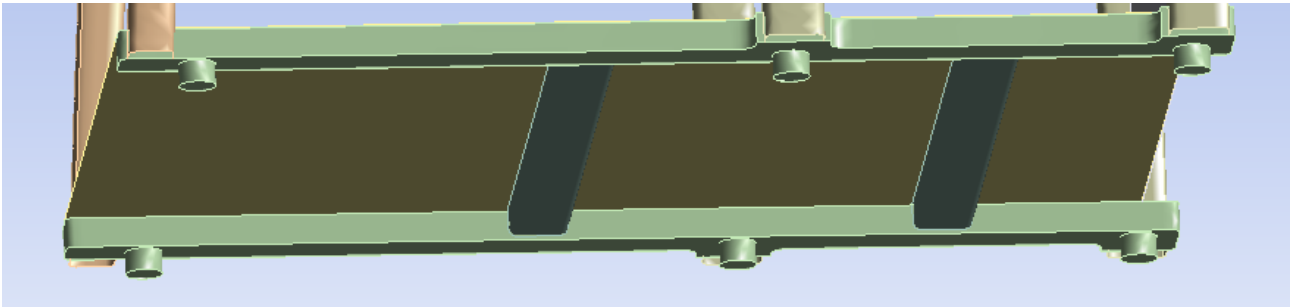


Boundary Conditions

- ROPS
  - Fixed supports on bottom plate
  - Prescribed velocities on impactors

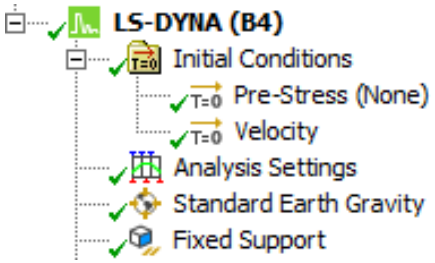


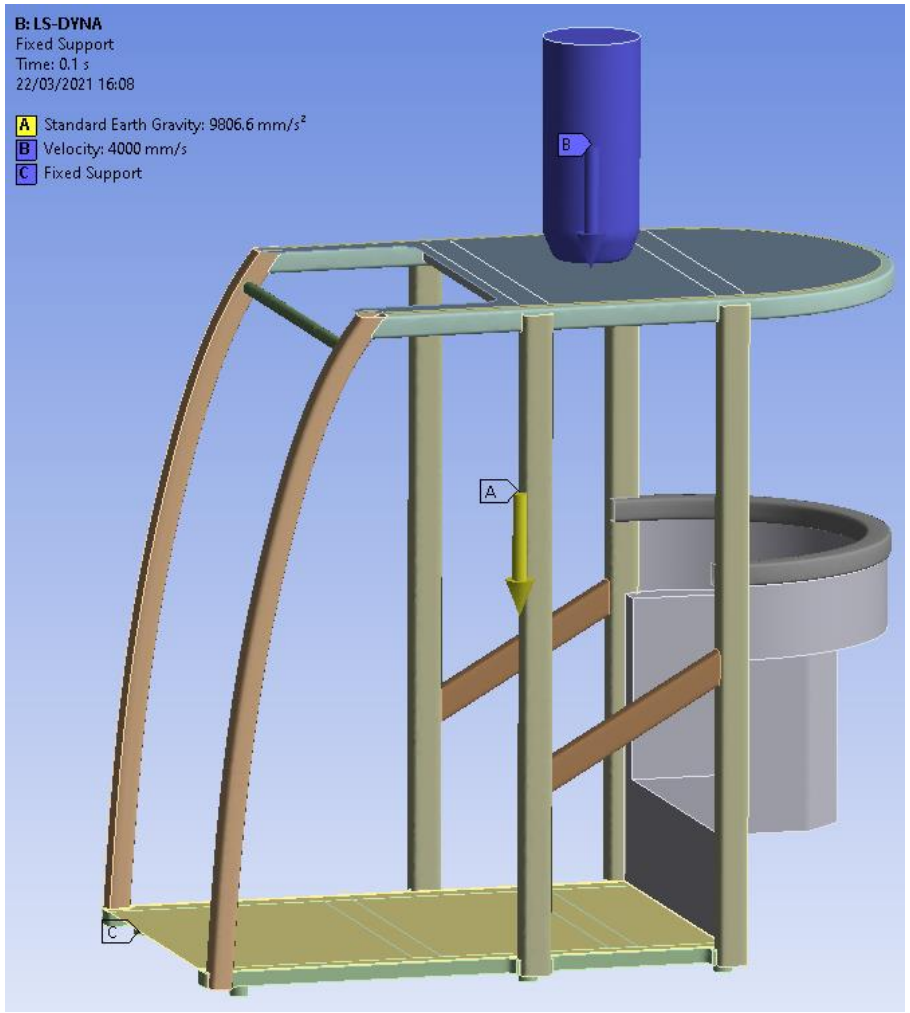




Boundary Conditions

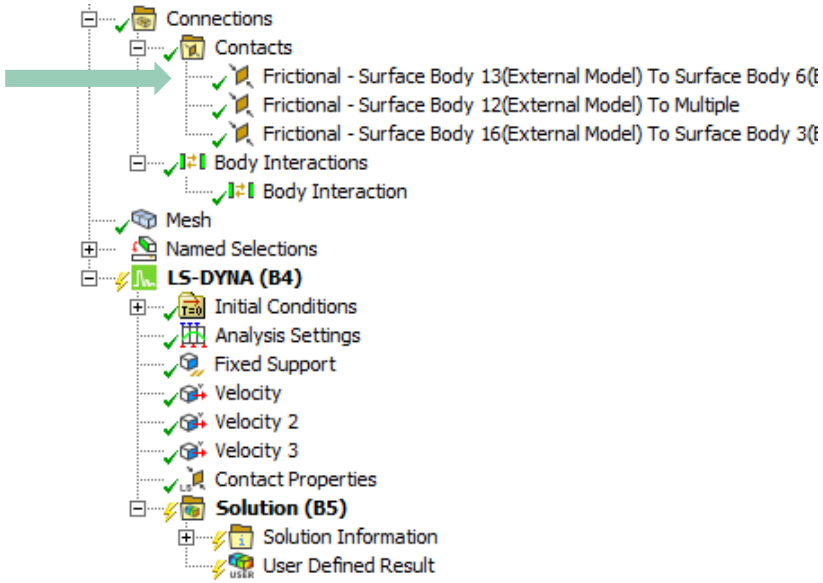
- FOPS
  - Fixed supports on bottom plate
  - Initial velocity on impactor
  - Gravity included





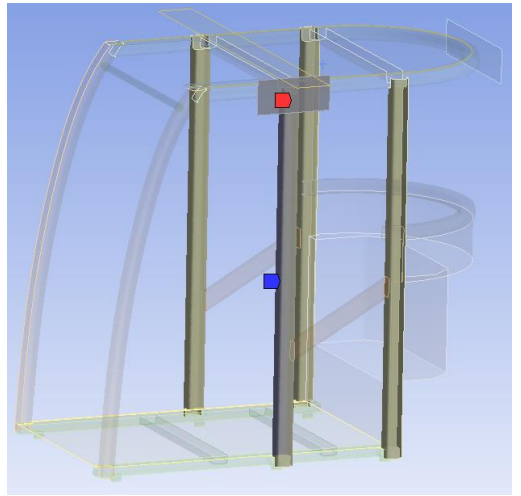
Contacts

- In the Contact Regions the user select the part/face/edge for the Contact and Target side (slave and master)
- Some general settings for the contact are also made
  - Type: Bonded/frictional
  - Friction coefficients
  - Behavior: Symmetric/Asymmetric



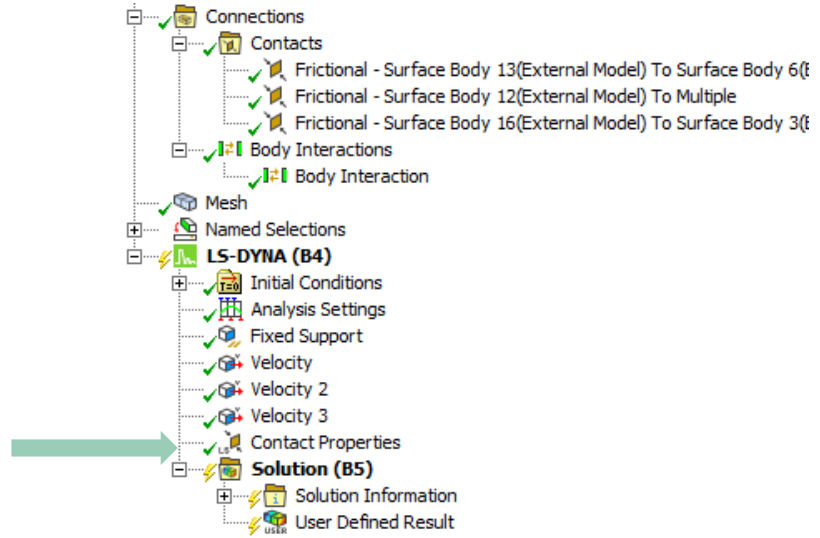
Details of "Frictional - Surface Body 13(External Model) To Surface Body

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	Surface Body 13(External Model)
Target Bodies	Surface Body 6(External Model)
Contact Shell Face	Top
Target Shell Face	Top
Protected	No
Definition	
Type	Frictional
Friction Coefficient	0.3
Dynamic Coefficient	0.
Decay Constant	0.



Contacts

- In the Contact Properties section the user define the LS-DYNA contact settings for each contact region
- Depending on the type and behavior in the contact region (bonded/frictional and symmetric/asymmetric) and the Type selection we can set the specific formulation used in LS-DYNA
- Some other familiar options for the contact keyword are also available in these section



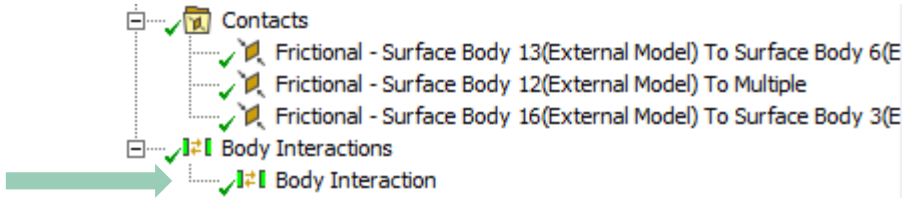
- Note that this is not mandatory. If not added the default settings will be used and is sufficient for most cases

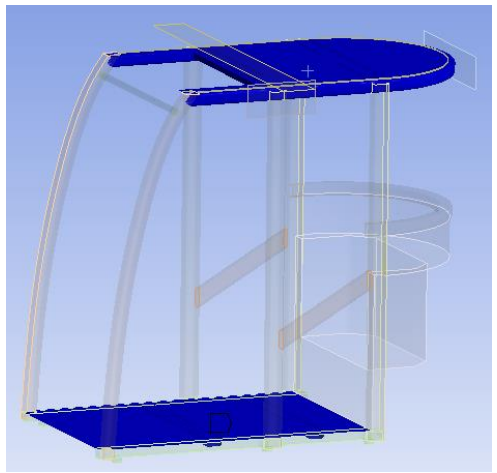
Details of "Contact Properties"

General	
Contact	Frictional - Surface Body 13(External M...
Type	Program Controlled
Formulation	AUTOMATIC_SURFACE_TO_SURFACE
Common Controls	
Birth Time	0 s
Death Time	0 s
Viscous damping coefficient	10
Slave Penalty Scale Factor	0
Master Penalty Scale Factor	0
Advanced Controls	
Optional Thickness for Slave Surface	0 mm
Optional Thickness for Master Surface	0 mm
Optional Solid Element Thickness	0 mm
Soft Constraint Formulation	Program Controlled
Soft Constraint Scale Factor	0.1
Depth	5

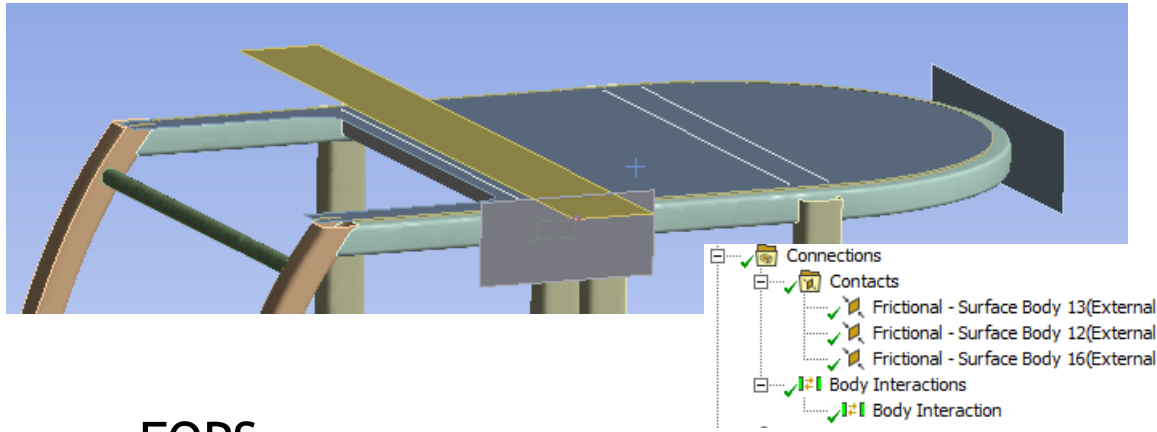
Contacts

- Body interaction to handle the internal contact between specific parts
  - *CONTACT_AUTOMATIC_SINGLE_SURFACE
- Use frictional contact for the rest of the contacts

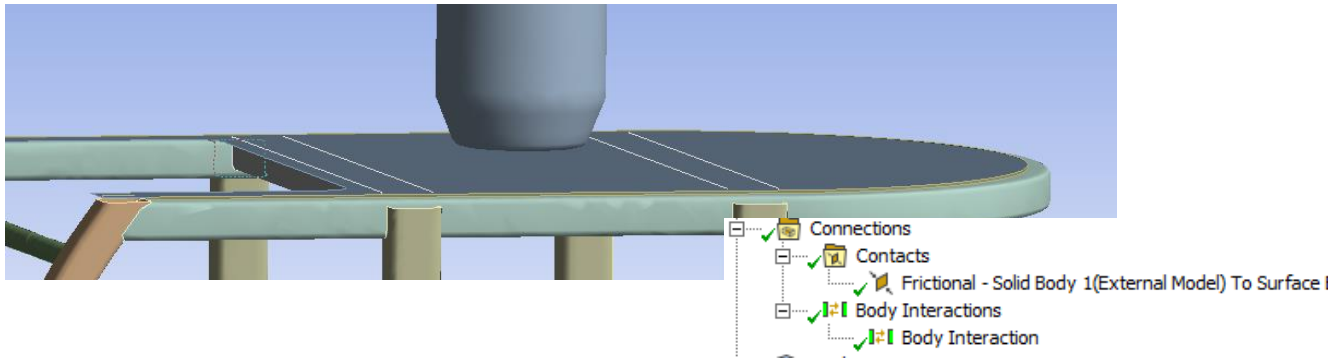




- ROPS
  - Contact between impactors and frame



- FOPS
  - Contact between test object and roof



Analysis settings

- The explicit solver will be used in both simulations
- The only needed input is the simulation end time
- ROPS
  - The ROPS test is a quasi-static test. In the simulation it will be a balance between not introducing dynamic effects in the loading and reducing the computational time.
  - Mass-scaling can be used to increase the explicit time step
- FOPS
  - The simulation time is set so it covers the impact
  - Mass-scaling can be used here as well but always check location and amount of added mass in the model

Step Controls	
End Time	0.7 s
Time Step Safety Factor	0.9
Maximum Number Of Cycles	10000000
Automatic Mass Scaling	Yes
Time Step Size	1.8E-06 s

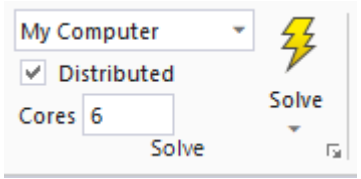
From message file

```
calculation with mass scaling for minimum dt
   added mass    =    2.6363E-02
   physical mass=    4.2625E-01
   ratio         =    6.1850E-02
```

Run LS-DYNA

- In the Analysis Settings there are some general simulation settings together with the solver settings
  - SMP/MPP
  - NCPU
  - Memory
  - Single/double precision
  - Solver version

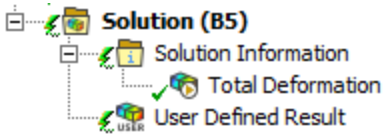
CPU and Memory Management	
Memory Allocation	Program Controlled
Number Of CPUS	6
Processing Type	SMP
Solver Controls	
Solver Type	Program Controlled
Solver Precision	single
Unit System	nmm
Invariant Node Numbering	Off
Second Order Stress Update	No
Solver Version	11.2




```
 Livermore Software Technology, LLC

 7374 Las Positas Road
 Livermore, CA 94551
 Tel: (925) 449-2500  Fax: (925) 449-2507
 www.lstc.com

 LS-DYNA, A Program for Nonlinear Dynamic
 Analysis of Structures in Three Dimensions
 Date     : 10/15/2020    Time: 07:56:29
 Version : smp s R11.2.0
 Revision: R11.2-66-g4cdf4c9c50

 Features enabled in this version:
   Shared Memory Parallel
   FFTW (multi-dimensional FFTW Library)
   Interactive Graphics
   ANSYS License (ANSYS211)
   MADYMO indirect coupling (7.2 or higher)

 Licensed to:
 Issued by   : ANSYS

 Platform    : WINDOWS X64 (SSE2)
 OS Level    : Windows 8/10 & Srv 2012/2016 R2 ua
 Compiler    : Intel Fortran XE 2017 MSVC++ 2017
 Hostname    : canary
 Precision   : Single precision (I4R4)

 Unauthorized use infringes LSTC copyrights
```

Run LS-DYNA

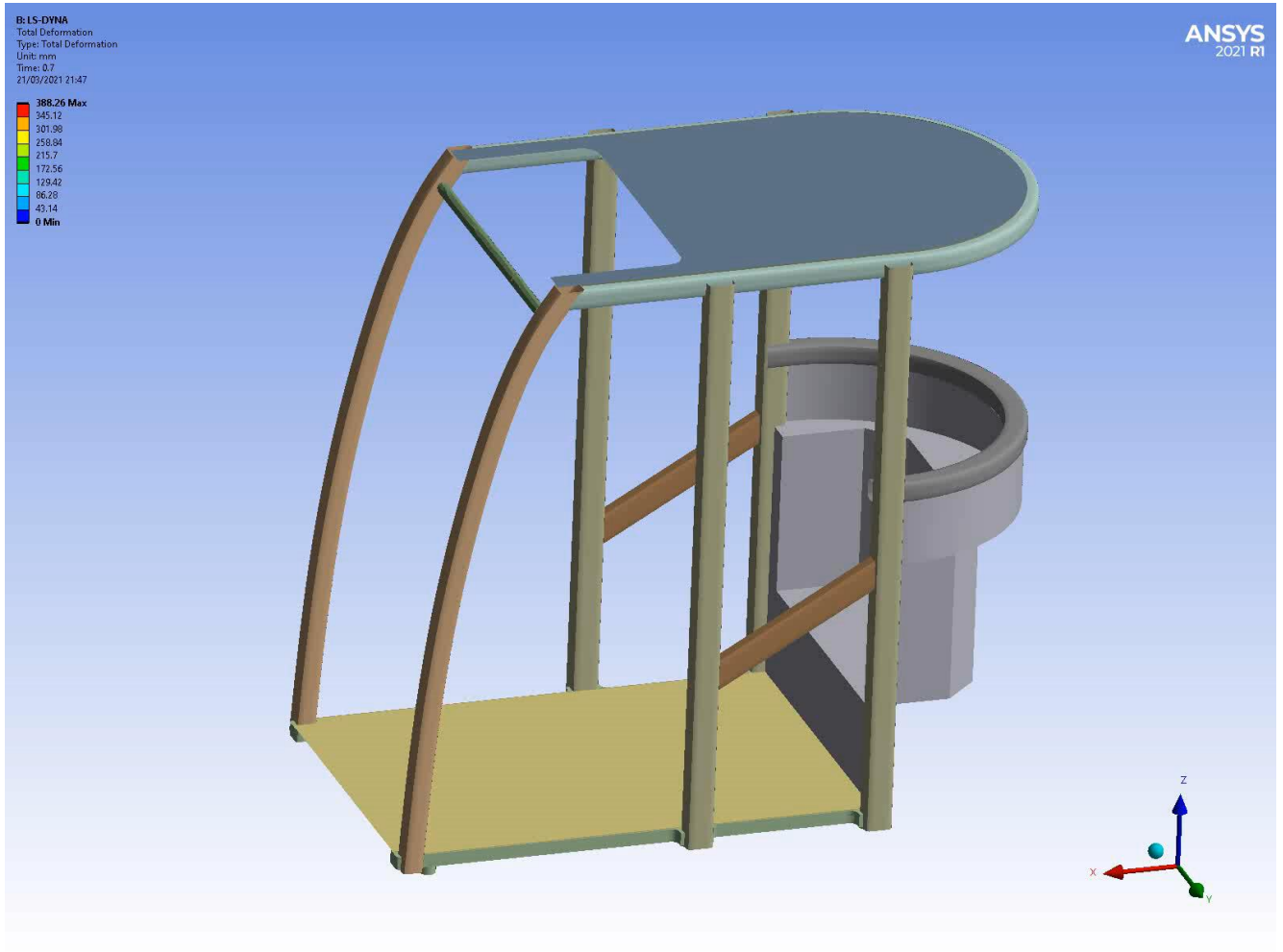
- The LS-DYNA message file can be viewed during the simulation in Solution Information
- The results and animation can be viewed “live” as the result files are created




```
calculation with mass scaling for minimum dt
   added mass    =   8.7199E-02
   physical mass=   1.6174E+00
   ratio         =   5.3914E-02
      1 t 0.0000E+00 dt 1.62E-06 flush i/o buffers            03/21/21 21:12:03
      1 t 0.0000E+00 dt 1.62E-06 write d3plot file            03/21/21 21:12:03
      1 t 0.0000E+00 dt 1.62E-06 write intfor file            03/21/21 21:12:03
cpu time per zone cycle.............          0 nanoseconds
average cpu time per zone cycle....       1133 nanoseconds
average clock time per zone cycle..        504 nanoseconds
estimated total cpu time          =       5618 sec (       1 hrs 33 mins)
estimated cpu time to complete    =       5610 sec (       1 hrs 33 mins)
estimated total clock time        =       2504 sec (       0 hrs 41 mins)
estimated clock time to complete  =       2496 sec (       0 hrs 41 mins)
added mass          =   8.7198E-02
percentage increase =   5.3913E+00
termination time                  = 1.000E-01
termination cycle                 =  10000000
    618 t 9.9954E-04 dt 1.62E-06 write d3plot file            03/21/21 21:12:12
    619 t 1.0012E-03 dt 1.62E-06 write intfor file            03/21/21 21:12:12
   1235 t 1.9991E-03 dt 1.62E-06 write d3plot file            03/21/21 21:12:19
   1236 t 2.0007E-03 dt 1.62E-06 write intfor file            03/21/21 21:12:19
   1852 t 2.9986E-03 dt 1.62E-06 write d3plot file            03/21/21 21:12:25
   1853 t 3.0002E-03 dt 1.62E-06 write intfor file            03/21/21 21:12:26
   2470 t 3.9998E-03 dt 1.62E-06 write d3plot file            03/21/21 21:12:33
   2471 t 4.0014E-03 dt 1.62E-06 write intfor file            03/21/21 21:12:33
   3087 t 4.9993E-03 dt 1.62E-06 write d3plot file            03/21/21 21:12:39
   3088 t 5.0009E-03 dt 1.62E-06 write intfor file            03/21/21 21:12:39
   3704 t 5.9989E-03 dt 1.62E-06 write d3plot file            03/21/21 21:12:46
   3705 t 6.0005E-03 dt 1.62E-06 write intfor file            03/21/21 21:12:46
   4321 t 6.9984E-03 dt 1.62E-06 write d3plot file            03/21/21 21:12:54
   4322 t 7.0000E-03 dt 1.62E-06 write intfor file            03/21/21 21:12:54
   4939 t 7.9996E-03 dt 1.62E-06 write d3plot file            03/21/21 21:13:01
   4940 t 8.0012E-03 dt 1.62E-06 write intfor file            03/21/21 21:13:01
   5000 t 8.0984E-03 dt 1.62E-06 flush i/o buffers            03/21/21 21:13:02
   5556 t 8.9991E-03 dt 1.62E-06 write d3plot file            03/21/21 21:13:09
   5557 t 9.0007E-03 dt 1.62E-06 write intfor file            03/21/21 21:13:09
   6173 t 9.9986E-03 dt 1.62E-06 write d3plot file            03/21/21 21:13:16
   6174 t 1.0000E-02 dt 1.62E-06 write intfor file            03/21/21 21:13:16
```

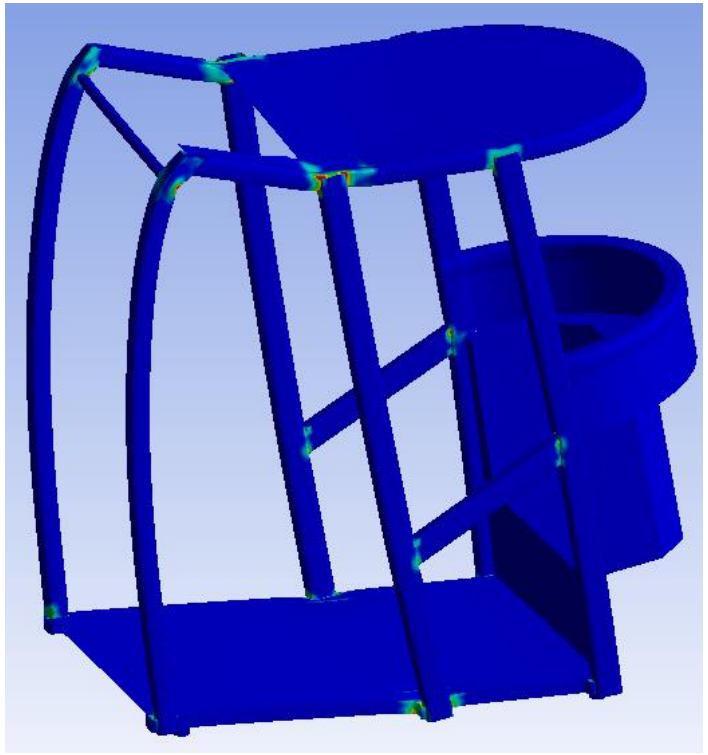
Postprocessing

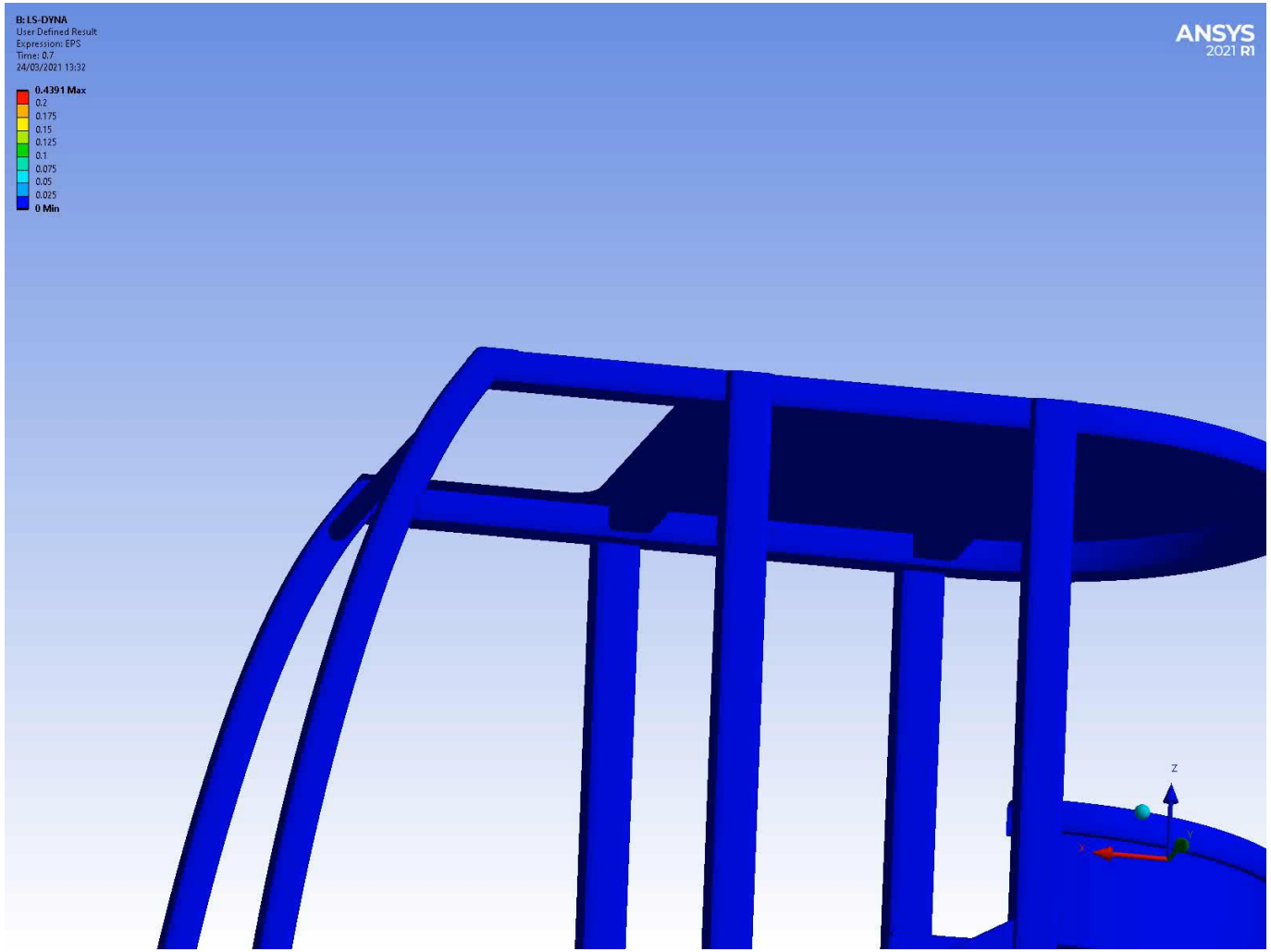
- ROPS
  - Simulation time: 0.7 s
  - Computational time: 1.28 h (6 cpus on laptop)



Postprocessing

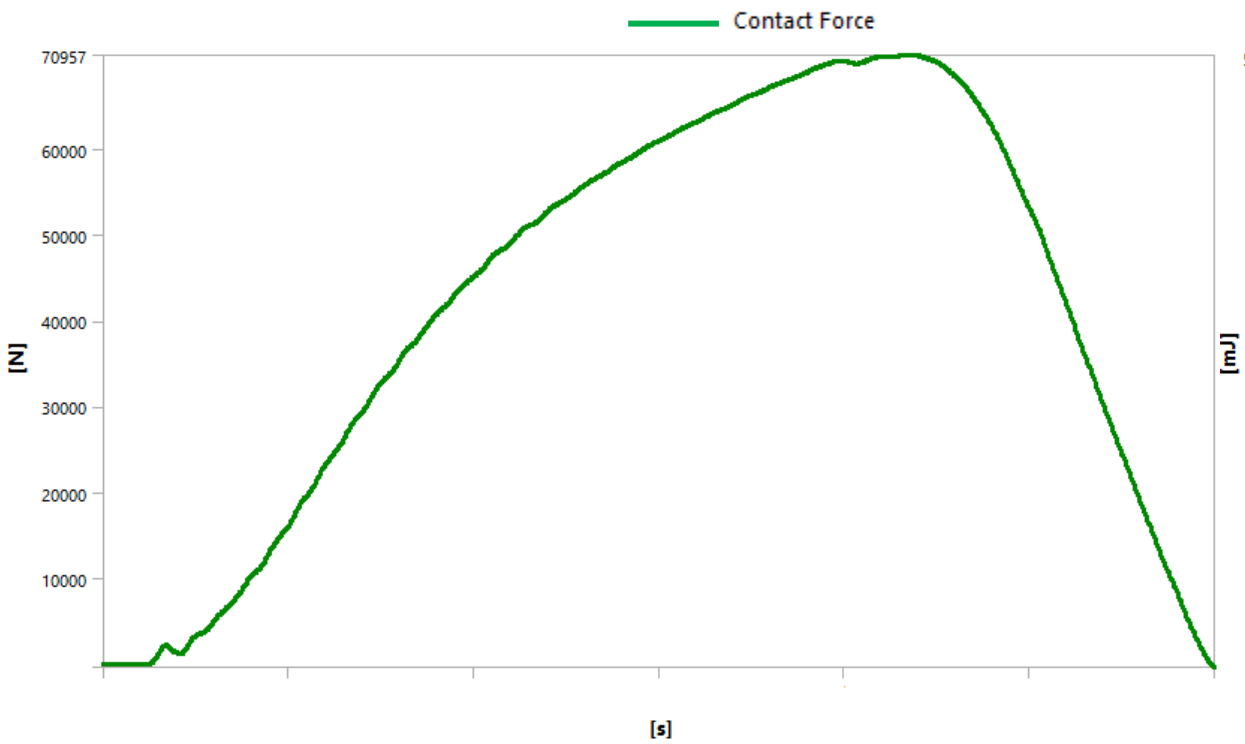
- ROPS
  - Plastic strain

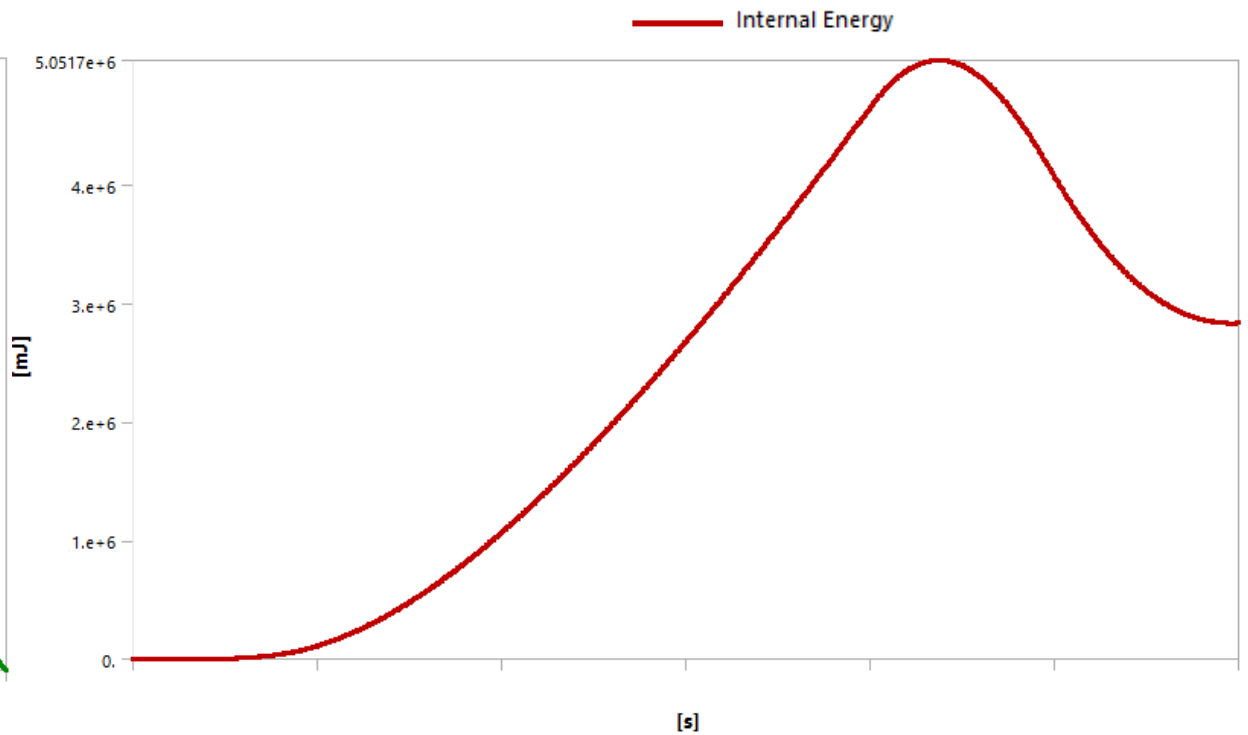




Postprocessing

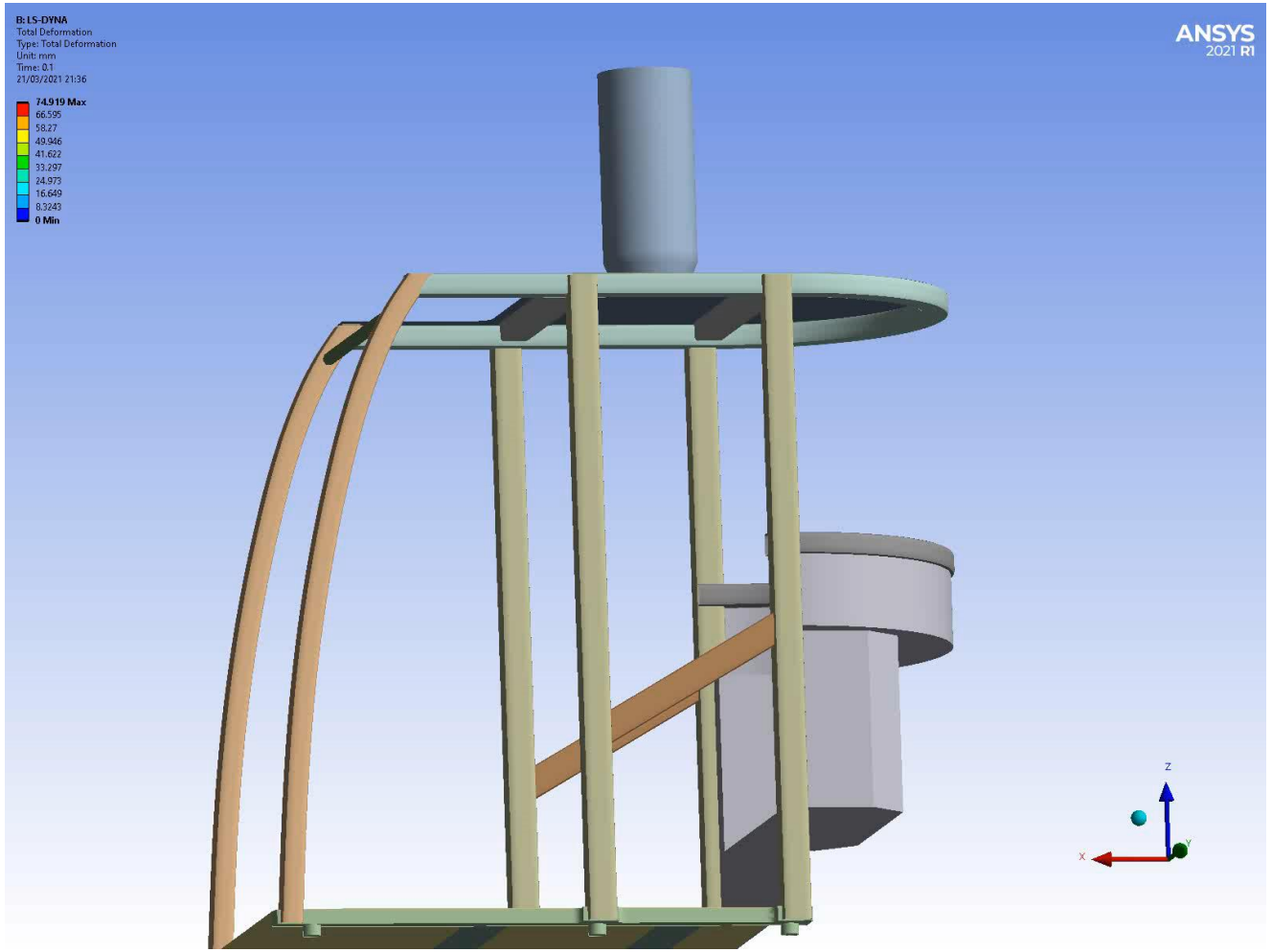
- ROPS
  - Contact force and energy for lateral load case





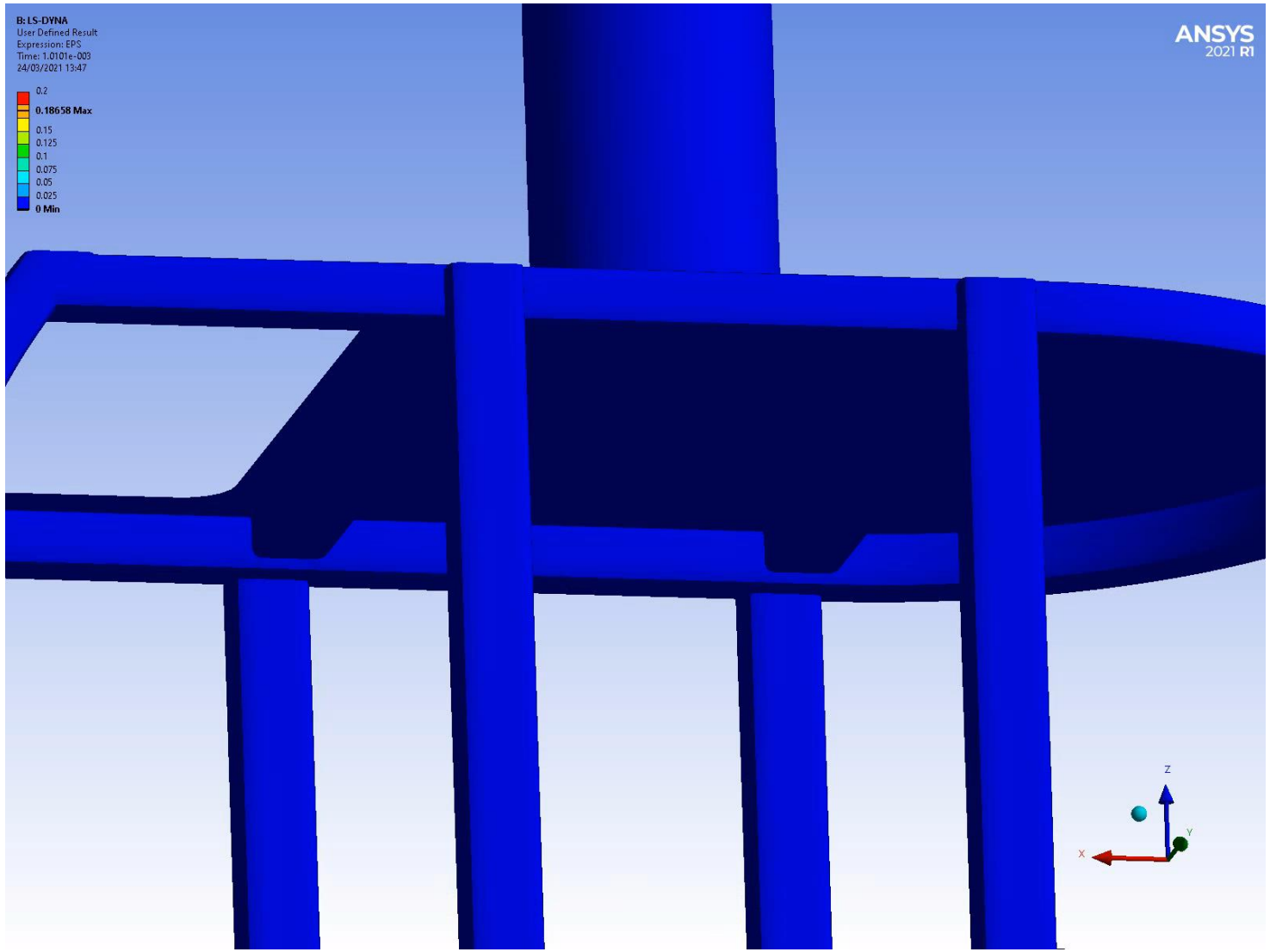
Postprocessing

- FOPS
  - Simulation time: 0.1 s
  - Computational time: 13 min (6 cpus on laptop)



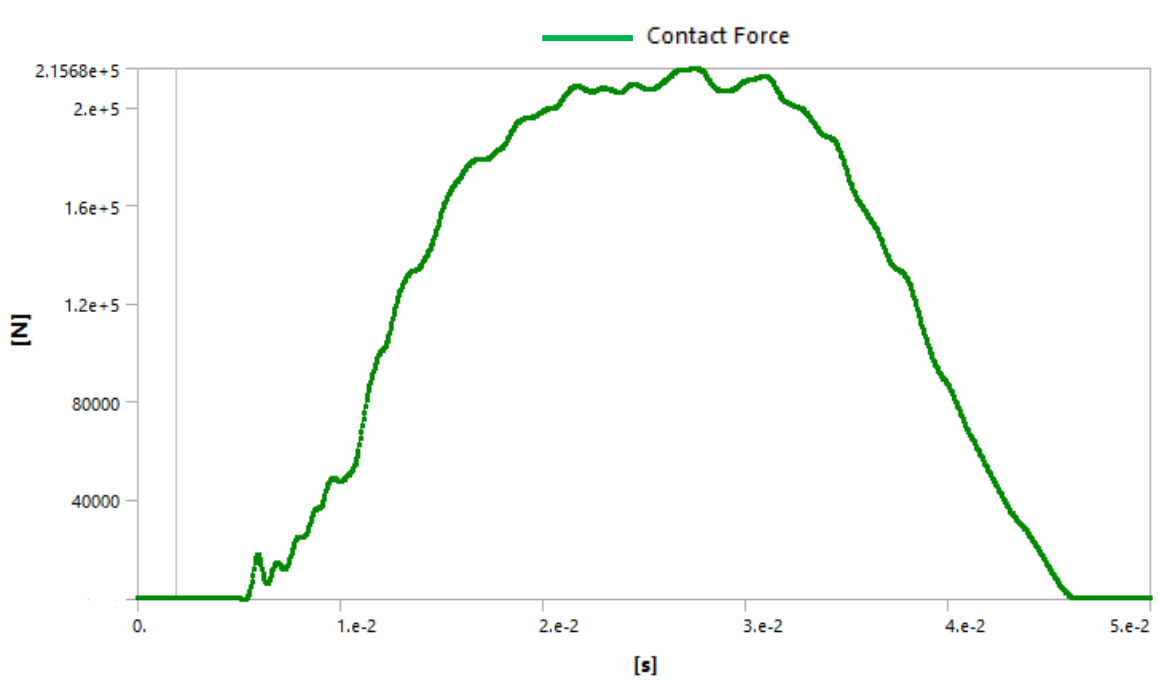
Postprocessing

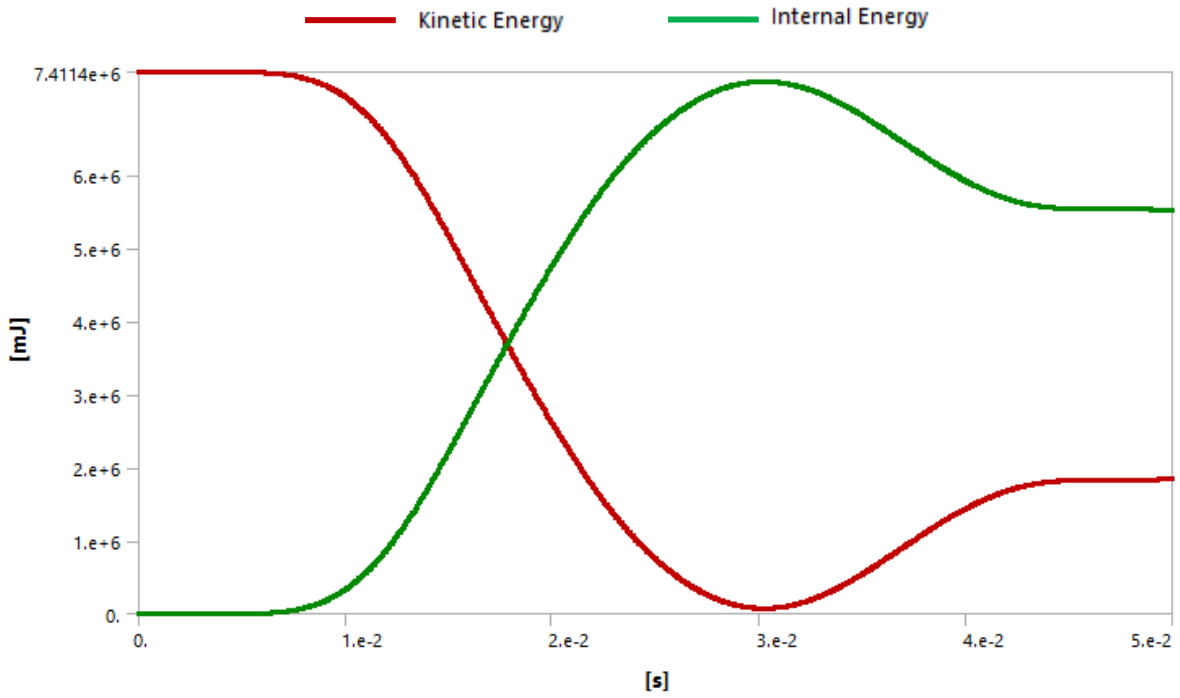
- FOPS
  - Plastic strain



Postprocessing

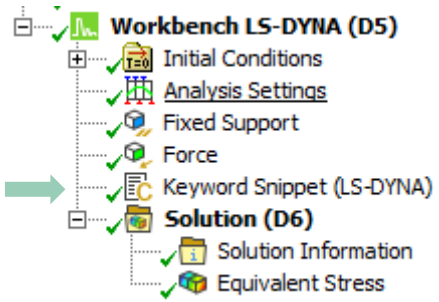
- FOPS
  - Contact force and energies





Additional input to LS-DYNA model

- Not all existing LS-DYNA keywords are accessible through Workbench LS-DYNA
- The Keyword Snippet function gives the user to input the wanted keywords

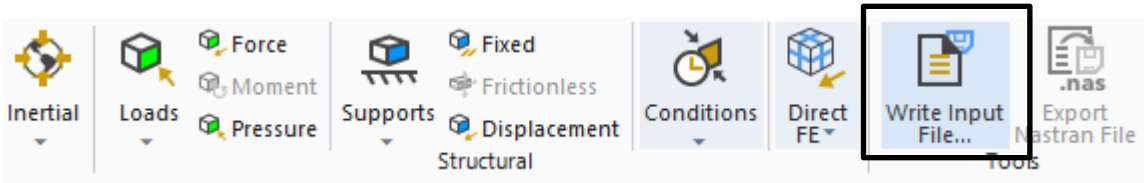


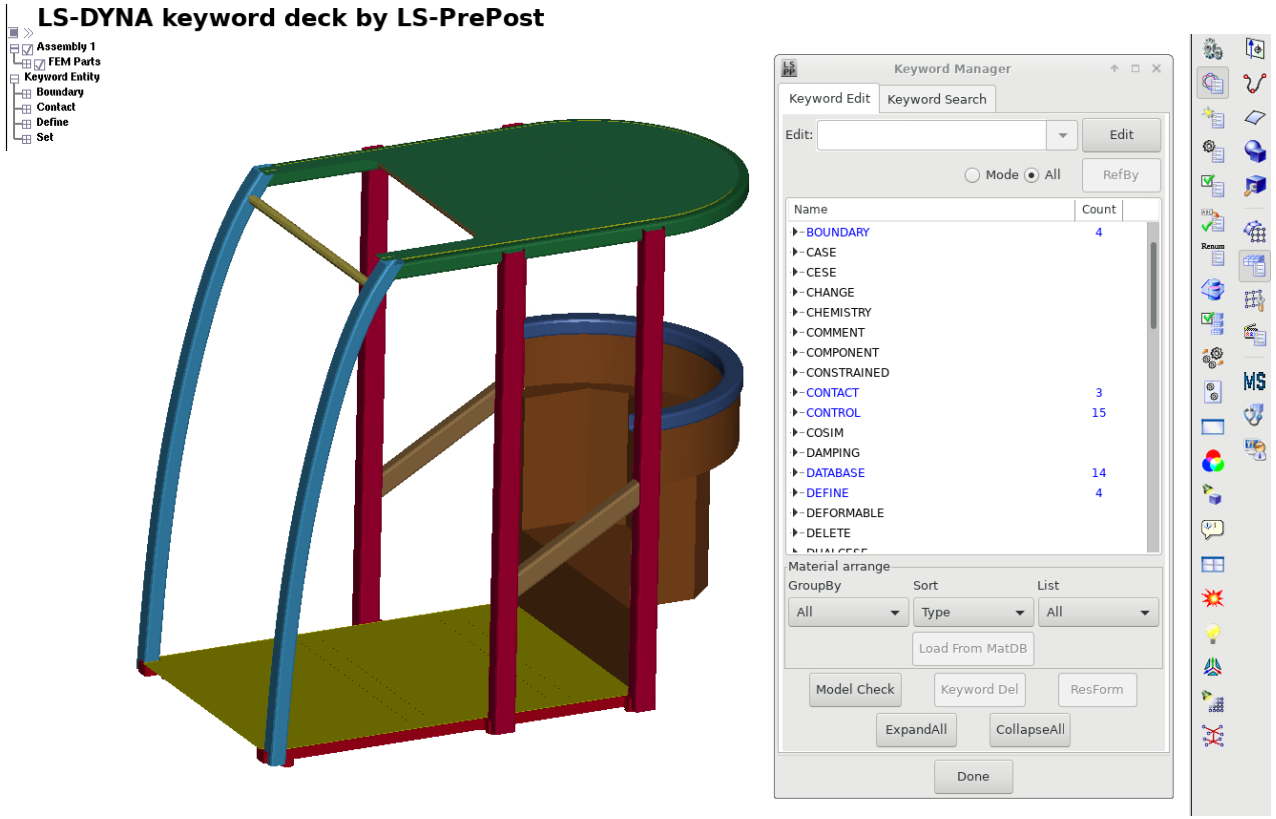
Commands

```
1  $   Keyword Snippet
2  $   Do not leave any unintentional empty lines in this editor
3  *CONTROL_IMPLICIT_GENERAL
4  $#  imflag       dt0    imform      nsbs       igs     cnstn      form    zero_v
5           1       0.1         2         1         2         0         0         0
6  *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_ID
7  $#      cid                                                              title
8           10
9  $#     ssid      msid     sstyp     mstyp    sboxid    mboxid       spr       mpr
10          3         1         3         3         0         0         0         0
11 $#       fs        fd        dc        vc       vdc    penchk        bt        dt
12         0.2       0.0       0.0       0.0       0.0         0 0.01.00000E20
13 $#      sfs       sfm       sst       mst      sfst      sfmt       fsf       vsf
14         1.0       1.0       0.0       0.0       1.0       1.0       1.0       1.0
15
```

Additional input to LS-DYNA model

- Export the keyword file from Mechanical
- LS-PrePost can then be used to add extra keywords that doesn´t exists in Workbench LS-DYNA
- LS-PrePost is a free software and supports all existing keywords in LS-DYNA





Additional input to LS-DYNA model

- Alternatives for ROPS model

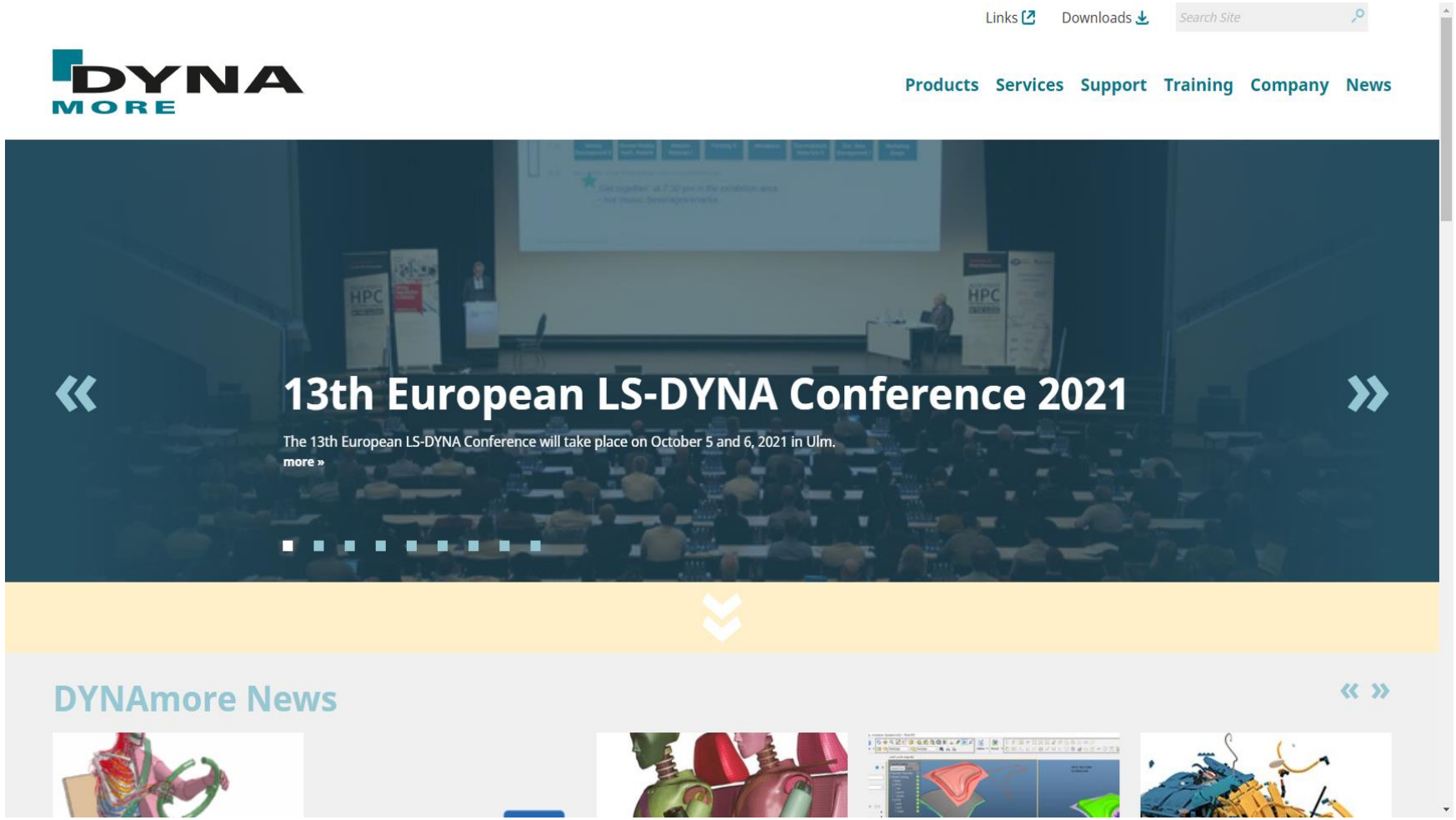
- Implicit solver
  - Add the necessary implicit keywords with Keyword snippets
  - Change element formulation in Mechanical
  - Use double precision for solver
- CASE option
  - Start one simulation that automatically runs individual simulations for different load cases
  - Individual keywords and parts can be added for each individual load case
  - Use LS-PrePost for setup
- Sensors
  - The sensors can monitor responses (e.g. contact force and energy)
  - When sensor criterias are met, boundary conditions can start or stop
  - Useful for the prescribed motion of the impactors
  - Use LS-PrePost for setup

Conclusion and Future

- Workbench LS-DYNA is a user-friendly application to set up and run LS-DYNA models
  - In many cases the default settings are sufficient to run a structural explicit simulation
  - By using Keyword Snippets or LS-PrePost the functionality can be increased
- More functions and keywords will be available in the user interface

- DYNAmore Nordic will continue to work with Workbench LS-DYNA and further investigate the functionality in upcoming releases
- An introduction course is available on Ansys Learning Hub

Conclusion and Future



Thank you!

DYNA MORE

Your LS-DYNA distributor and more