Numerical Simulation of a 9m Free Drop Test in a 1:2 Scale Model Cask for Spent Fuel Elements of Nuclear Research Reactors

Carlos Alexandre de J. Miranda
Miguel Mattar Neto
Gerson Fainer
IPEN/CNEN-SP
Nuclear Engineering Center

Project Purpose & Partners

Main purpose - to develop modeling and results evaluation methods in order to apply in future prototypes design, as part of a Latin American multinational project sponsored by IAEA

The project partners are the Institutes IPEN & CDTN (from Brazil) and South American countries with research reactors, to qualify a shipping cask for their burned fuel elements

Licensing Background

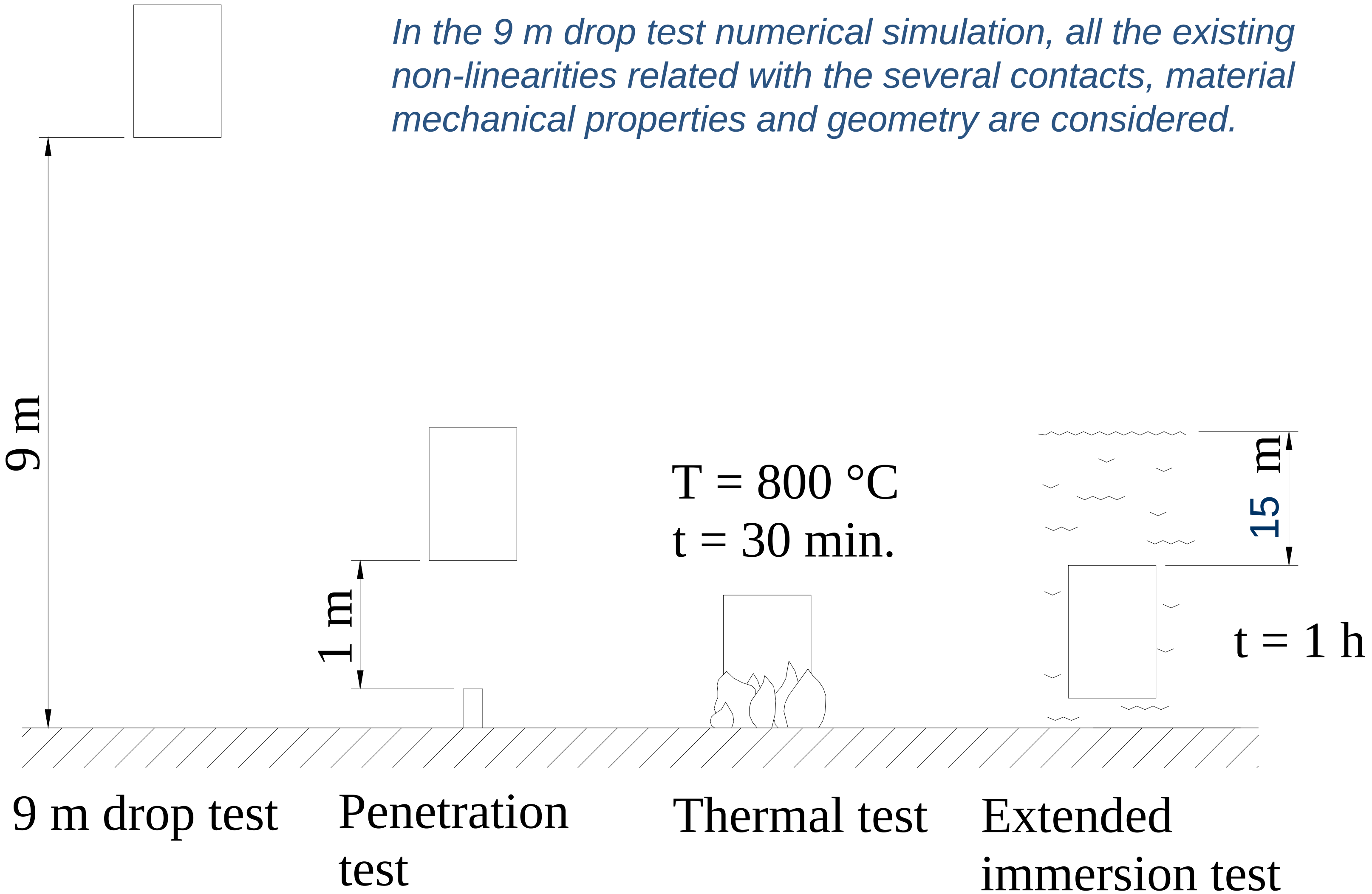
- Safety Analysis Report for the Packaging - SARP (according to USNRC Regulatory Guide 7.9, r1)
  - General Information (packaging description, drawings, QA)
  - Shielding Evaluation (gamma and neutron sources)
  - Criticality Evaluation (criticality models for MTR and TRIGA fuels)
  - Operating Procedures (loading and unloading, dry/wet (un)loading)

Licensing Background

- Safety Analysis Report for the Packaging - SARP
  (according to USNRC Regulatory Guide 7.9, rev 1)
  - Structural Evaluation (materials, lifting and tiedown devices, normal and accident conditions)
  - Thermal Evaluation (thermal properties of materials, normal and accident conditions)
  - Containment (containment boundary, normal and accident conditions)
  - Acceptance Tests and Maintenance Program (visual inspection; structural, thermal and leak tests; shielding integrity verification)
  - Shielding Evaluation (gamma and neutron sources)
  - Criticality Evaluation (criticality models for MTR and TRIGA fuels)
  - etc.

Prescribed tests

In the 9 m drop test numerical simulation, all the existing non-linearities related with the several contacts, material mechanical properties and geometry are considered.



Tests x Simulations

Physical testing X Numerical simulation

- Model (1:2 Scale)
  - 9 m drop test + penetration test + thermal test
  - Damaged model
  - Extended immersion test (200 m) numerically simulated using damaged model
- Prototype: All tests numerically simulated

Cask design

- Design criteria: 21 MTR or 78 TRIGA, max. weight 10 t, Type B fissile package

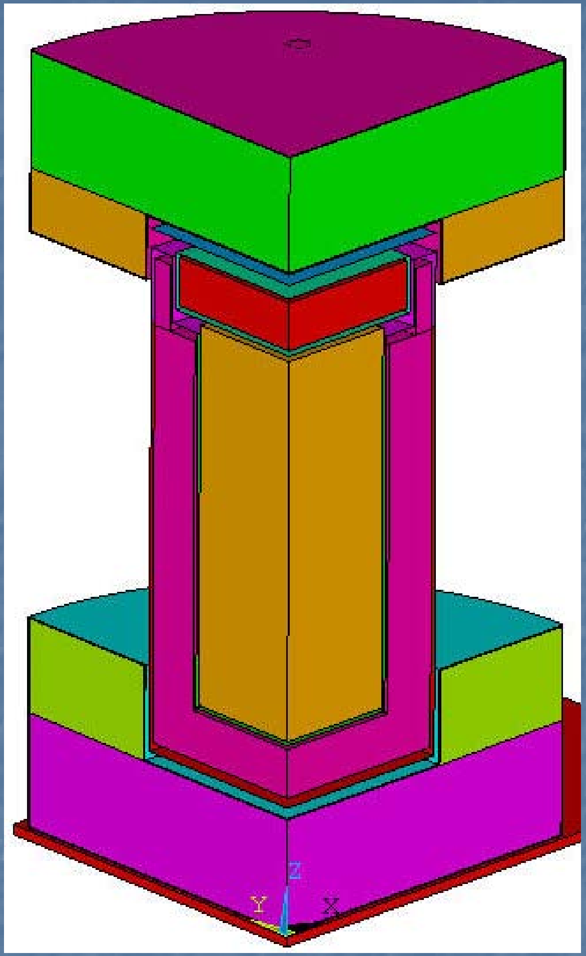
- Design goal:
  . 125 g in the internal basket

- Main parts:
  . Main body;
  . Basket;
  . Lids (In & External);
  . Lead;
  . Heads;
  . Dampers;
  (Impact limiters)

Cask model description

The cask - stainless steel cylinder with flat heads (the bottom one is welded and the upper one has flanges with threaded connections – not modeled) and internal basket (for the fuel elements)

- It is surrounded by a biological shield of lead
- it has also upper and lower wood dampers contained in stainless steel shells



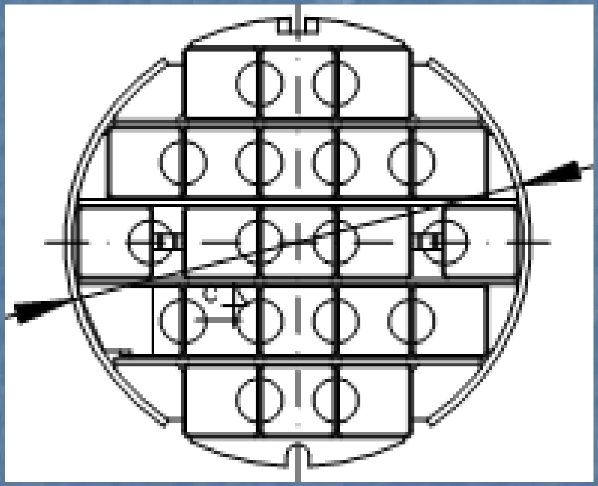
90° FE model

Main body

External wall: resist puncture and external pressure

Internal wall: resist lead contraction

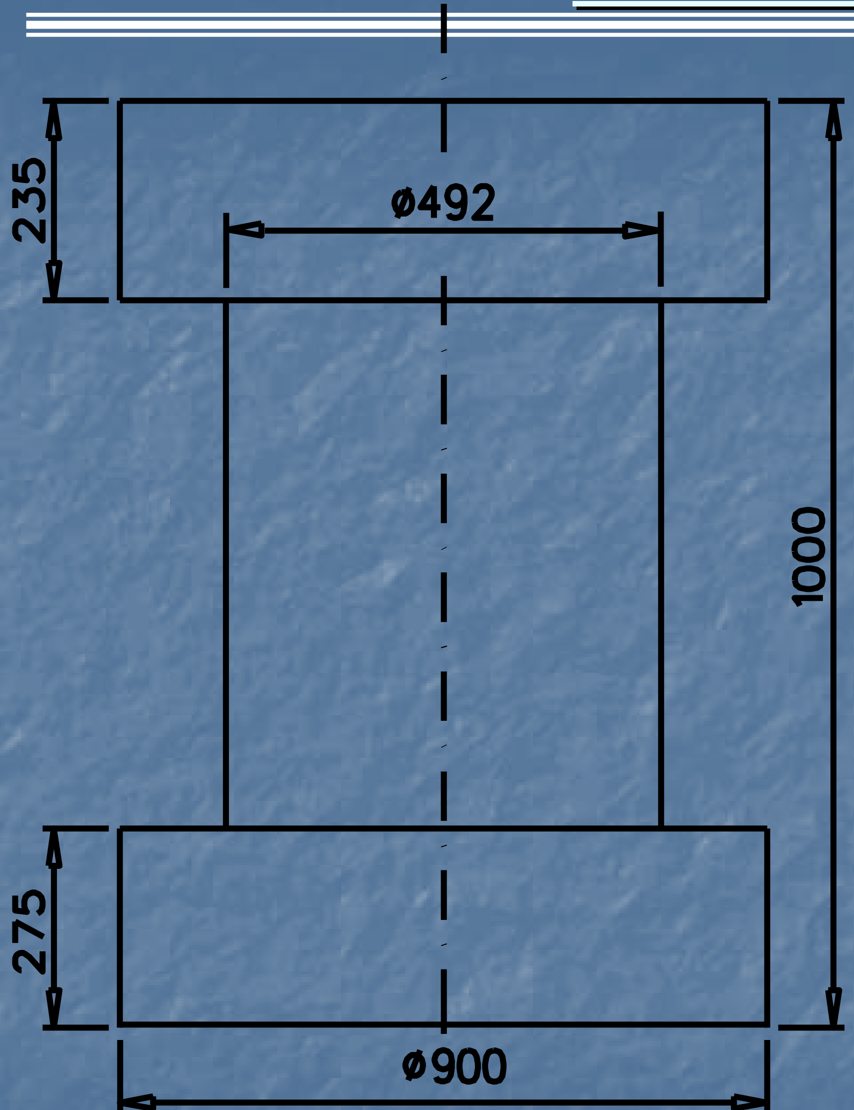
Internal basket (dummy mass in the model)



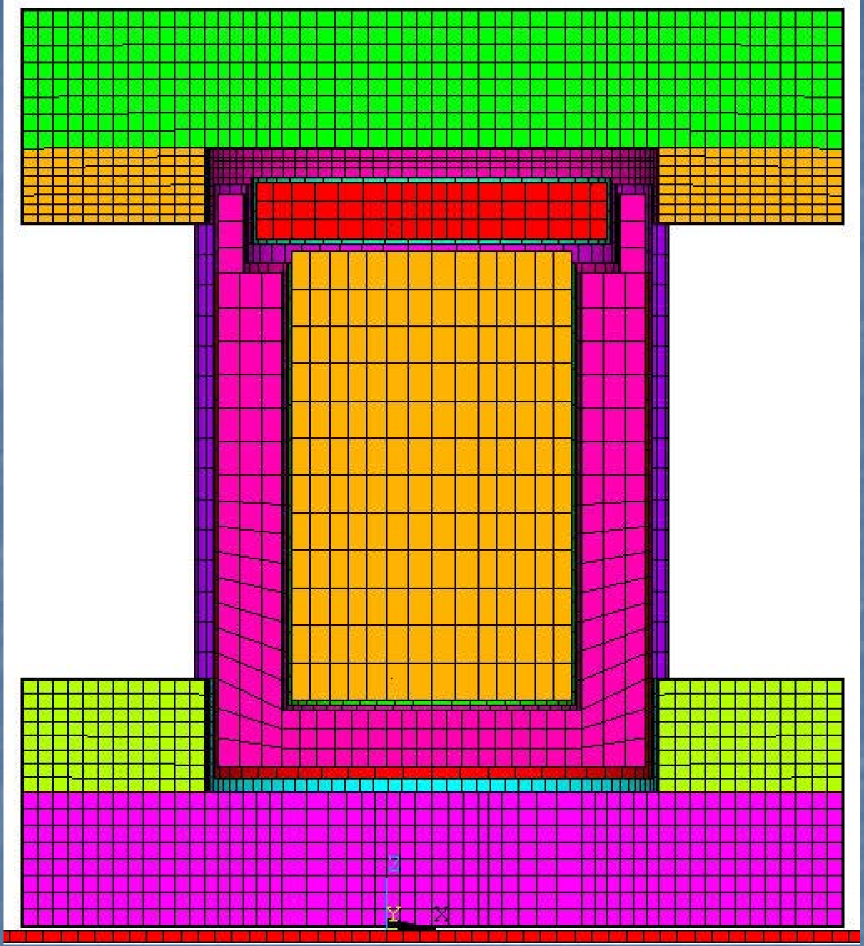
1st approach:

- Only for MTR fuel elements
- 20 positions for fuel elements, one position for accelerometer

Half scale model



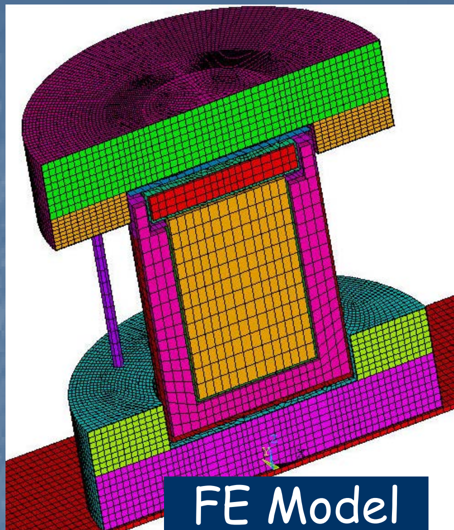
Overall dimensions

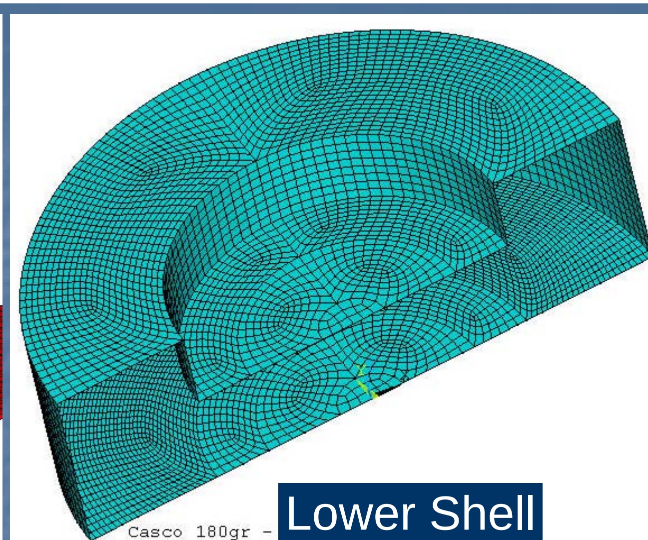


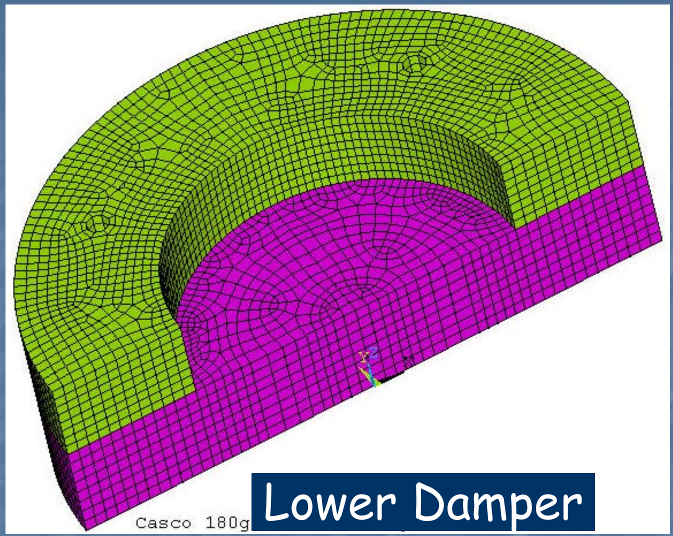
180° FE model

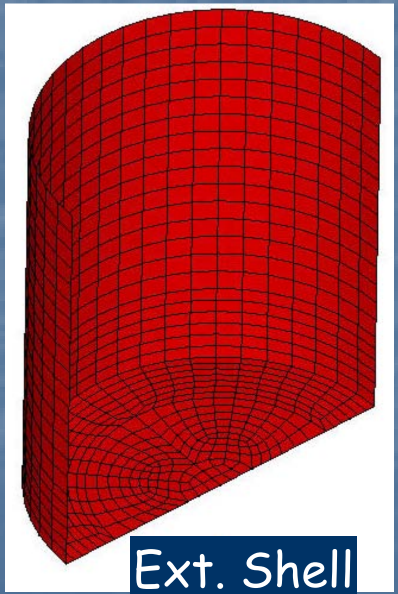
CENM

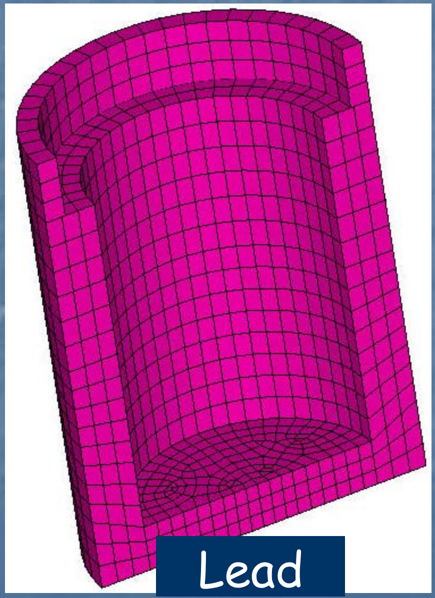
FE model details

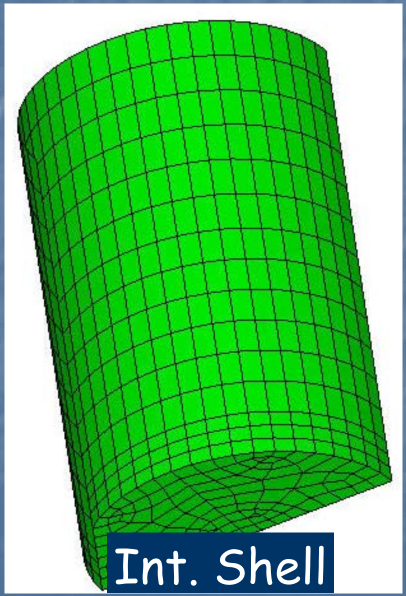


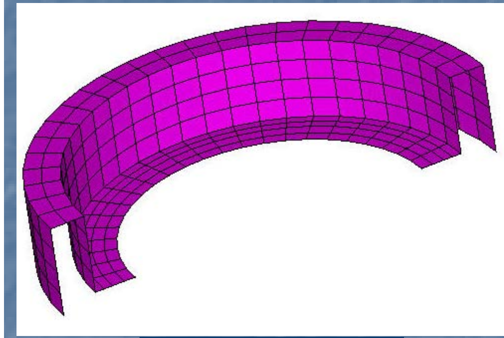








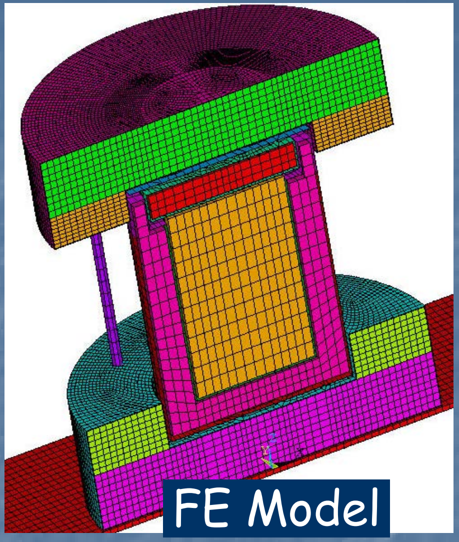


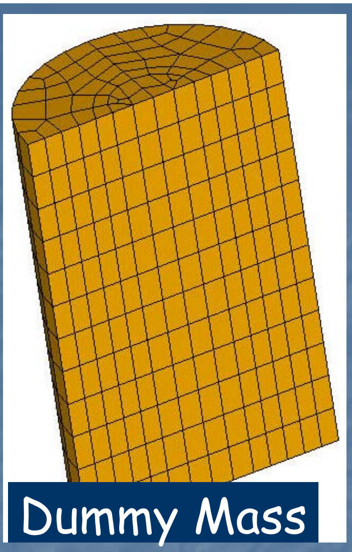


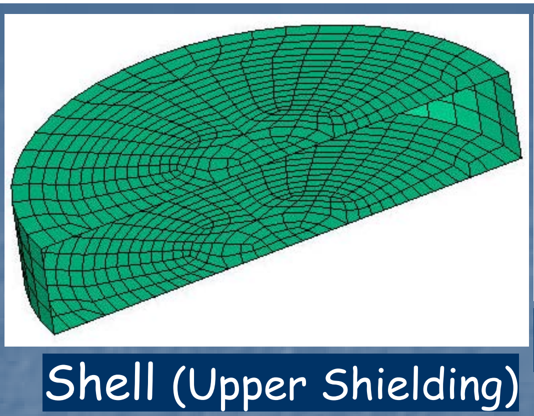
Conection

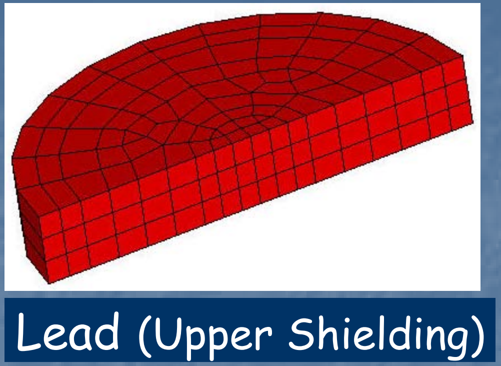
CENM

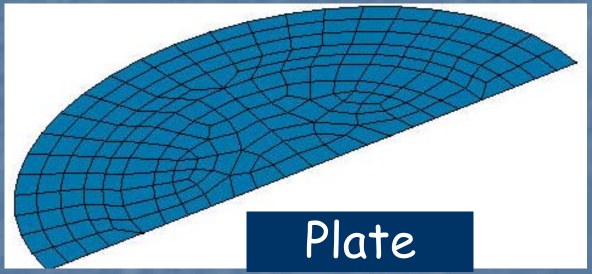
FE model details

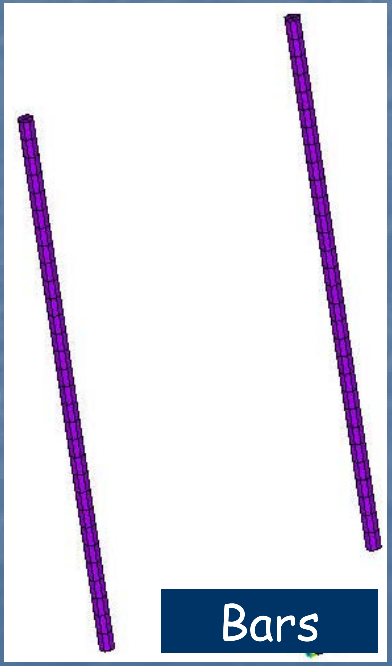


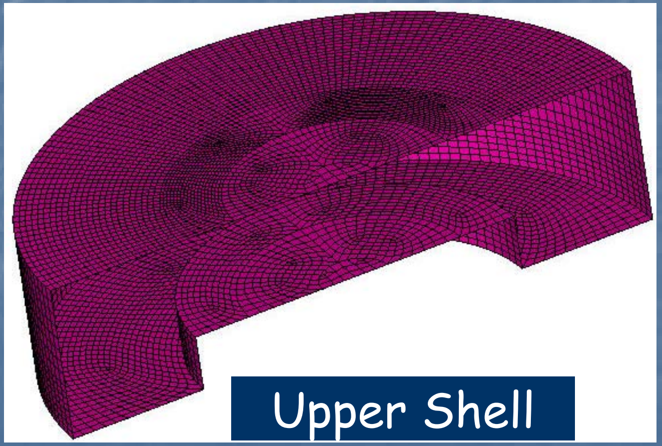


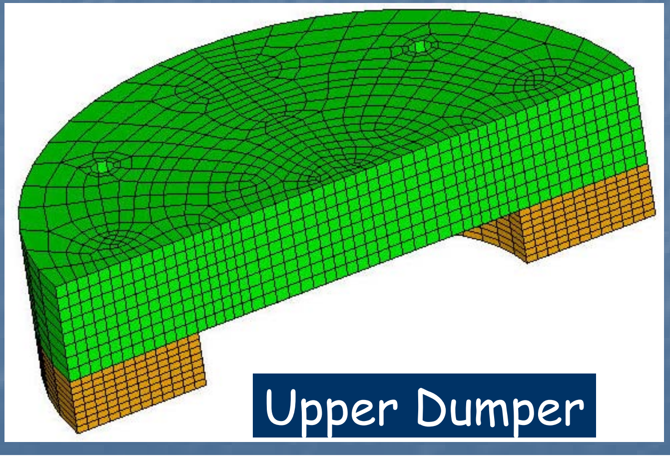




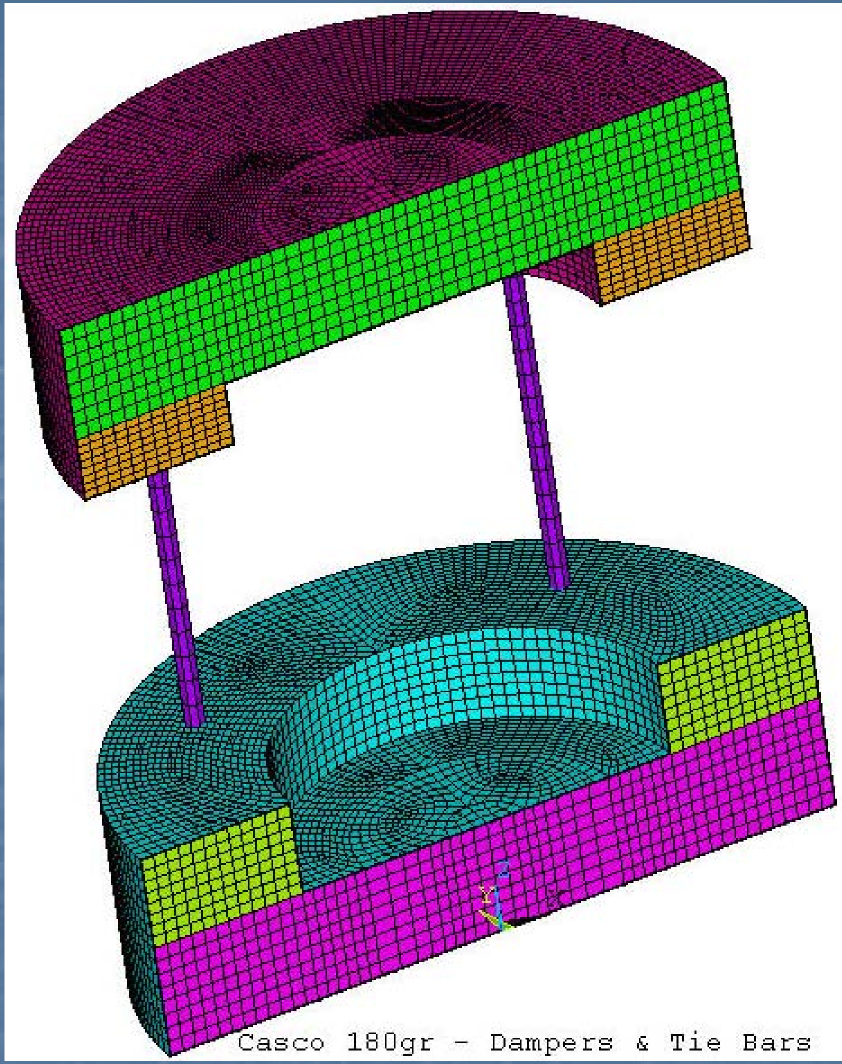




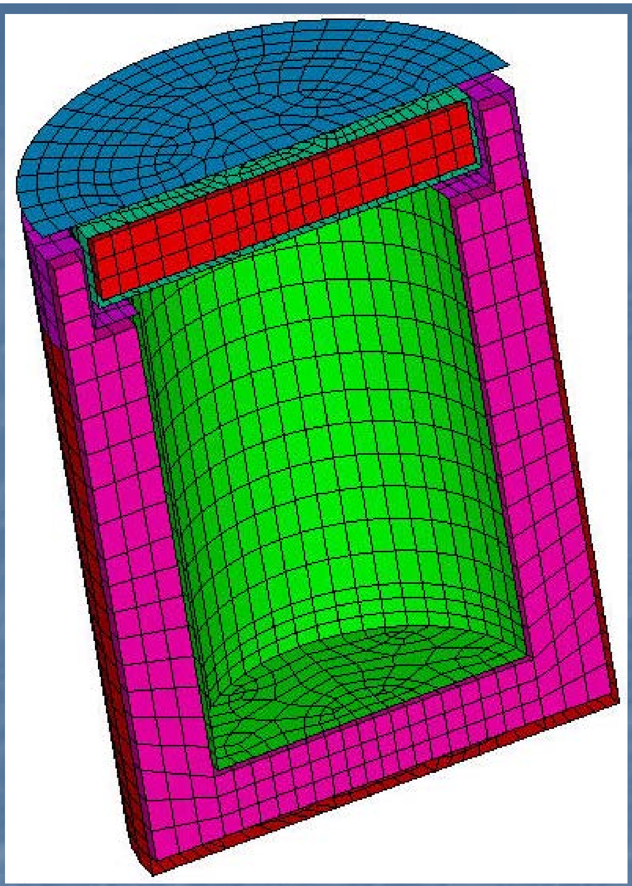




FE model details



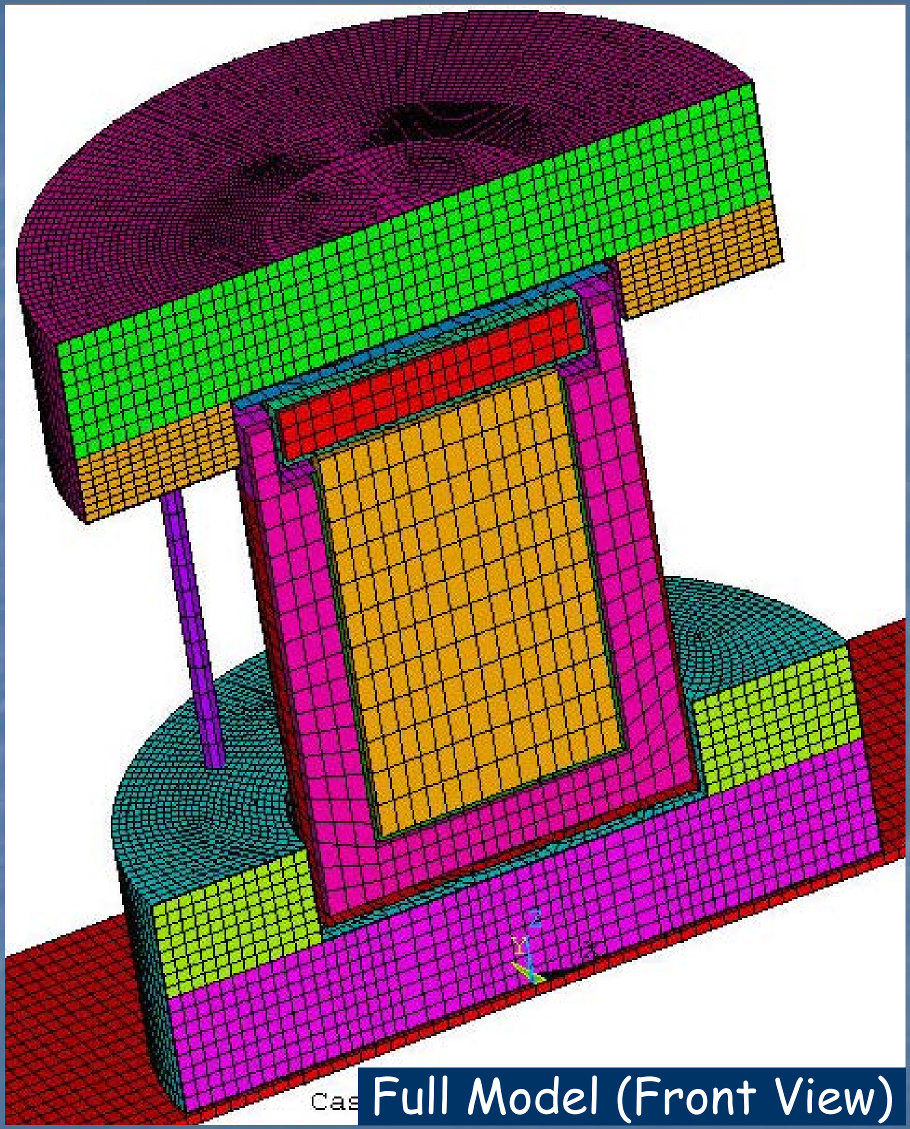
Dampers and Tie Bars



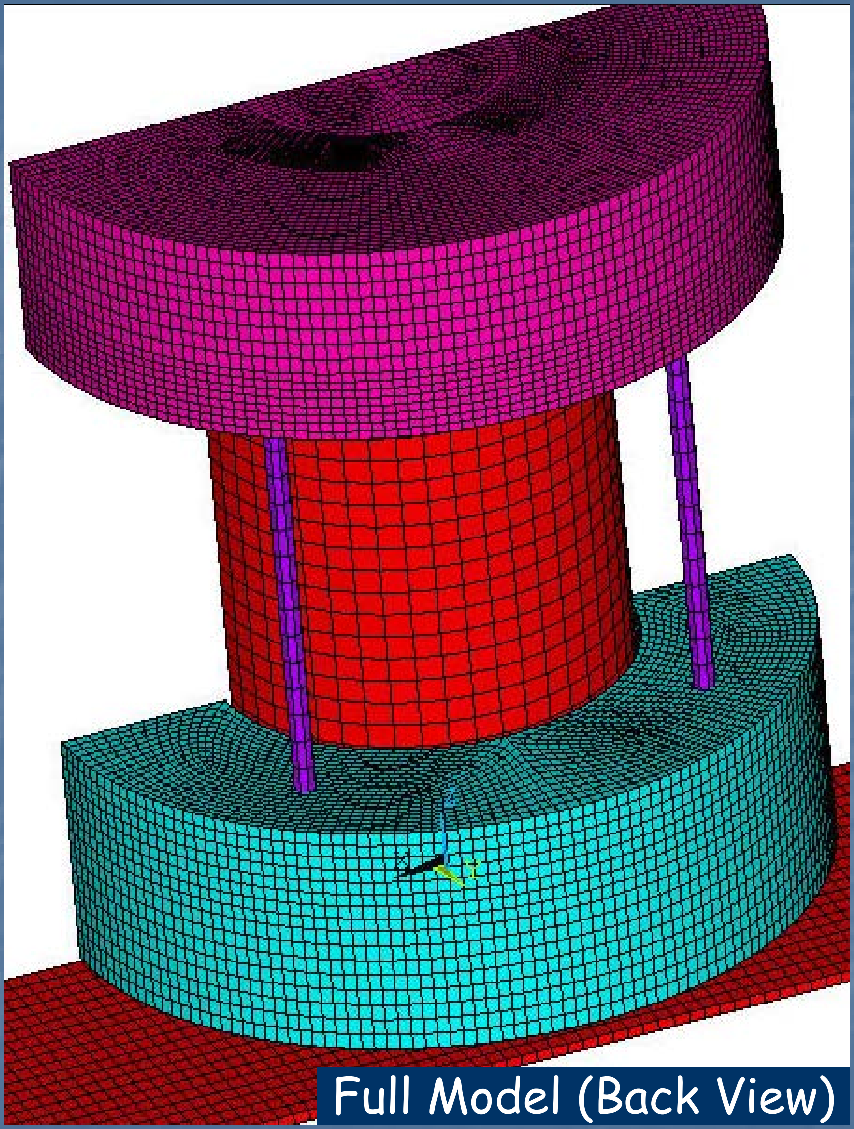
Cylinders (Ext&Int), Lead, Connection, Upper Shielding (Shell&Lead), Plate

CENM

FE model details



Full Model (Front View)



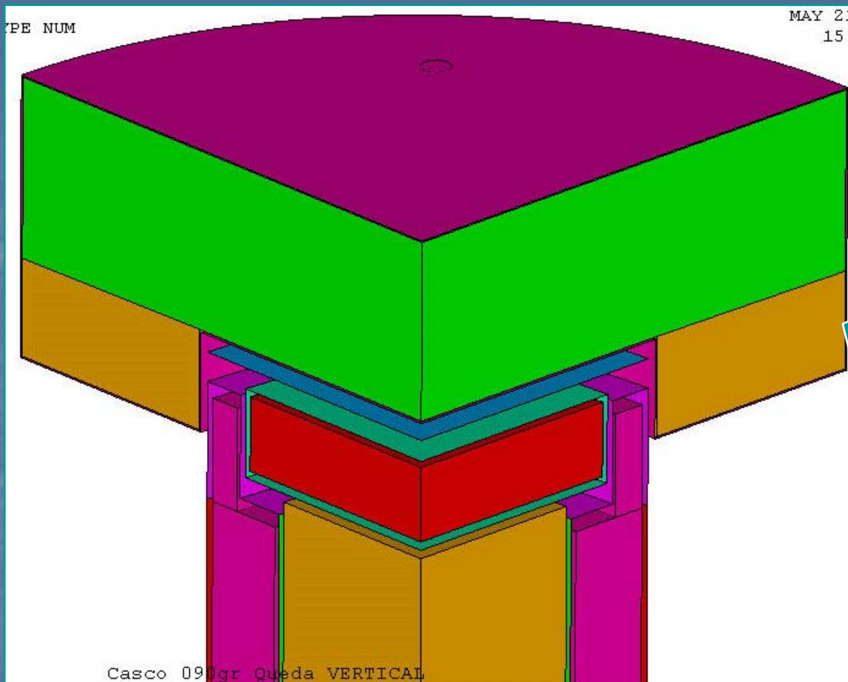
Full Model (Back View)

Half scale model - general data

Cask Part	Material	Dimensions
Lower shell	stainless steel	dia = 900 mm
Lower Damper	wood (**OSB**)	dia = 894 mm
Inner Shell	stainless steel	dia = 328 mm
Lead	Lead	
Outer Shell	stainless steel	dia = 492 mm
Upper Damper	wood (**OSB**)	dia = 894 mm
Upper shell	stainless steel	dia = 900 mm
Tie bars	stainless steel	dia = 30 mm

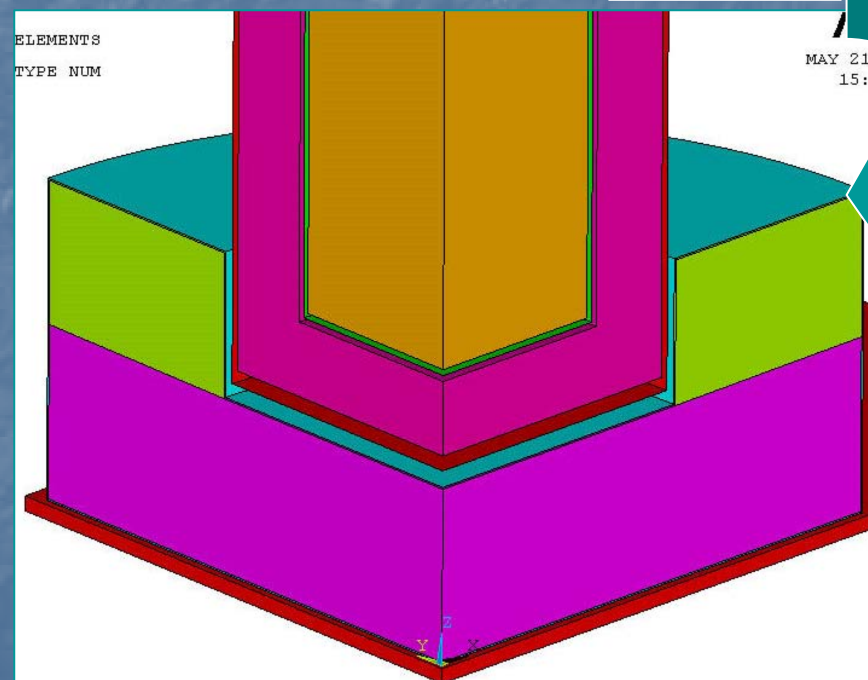
CENM

Impact limiters



upper damper

lower damper



- Filling material: wood (OSB)
  (Oriented Strand Board)

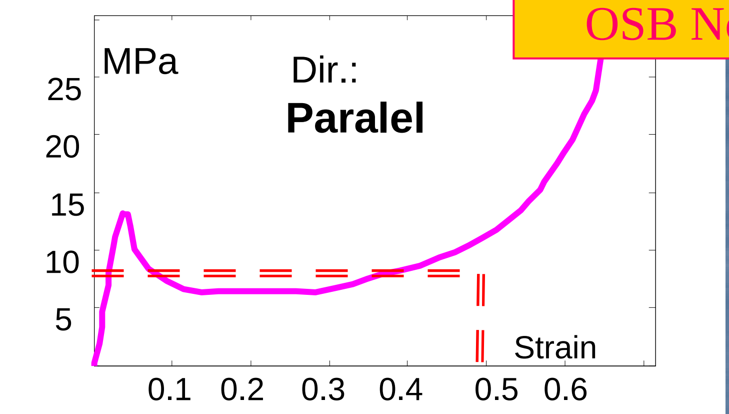
Impact limiters – Possible materials

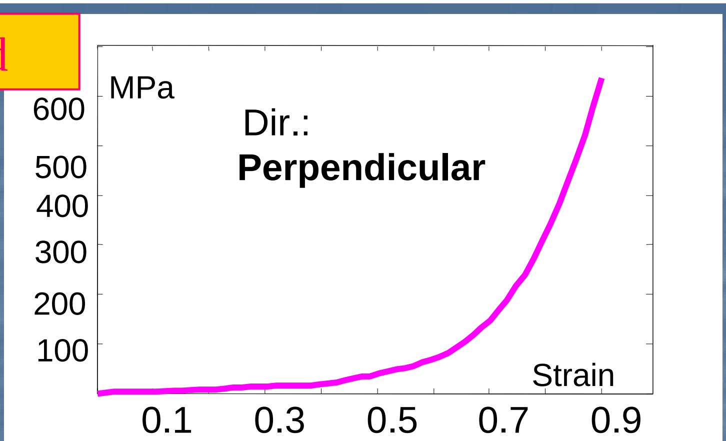
Filling material	Advantages	Disadvantages
Solid wood (Eucaliptus, Pinus)	▪High energy absorption capacity (tenacity)	▪Flammable ▪Hard to model ▪Anisotropic, not homogeneous
Reconstituted wood (OSB)	▪Homogeneous	▪Flammable ▪Hard to model ▪Anisotropic
Polyurethane foam	▪Only slightly anisotropic ▪Easily modeled	▪Challenging manufacture ▪Organic, flammable
Light mortar	▪Homogeneous, isotropic ▪Good resistance to fire ▪Inorganic ▪Easily modeled	▪Needs baking

CENM

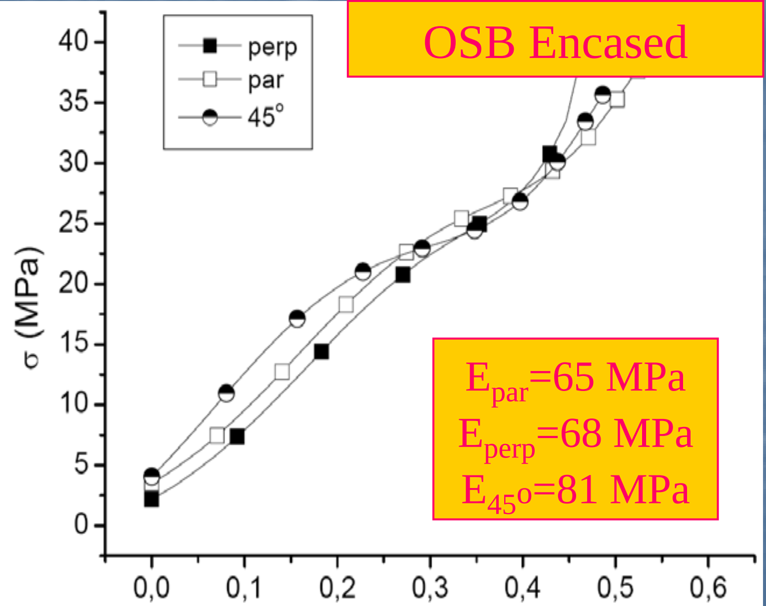
Impact limiters - Material Properties

OSB Not encased





OSB Encased

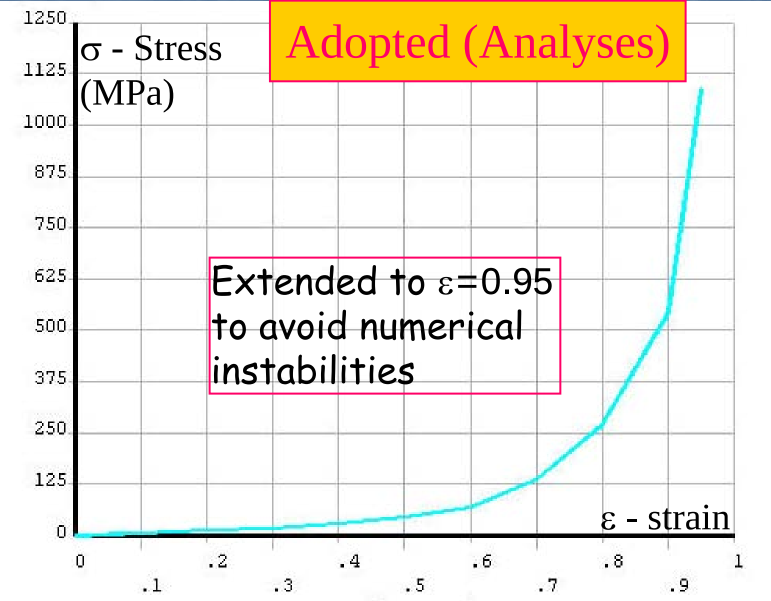


E_{par}=65 MPa

E_{perp}=68 MPa

E_{45^o}=81 MPa

Adopted (Analyses)

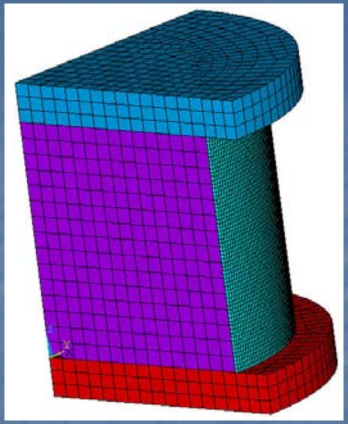


Extended to ε=0.95 to avoid numerical instabilities

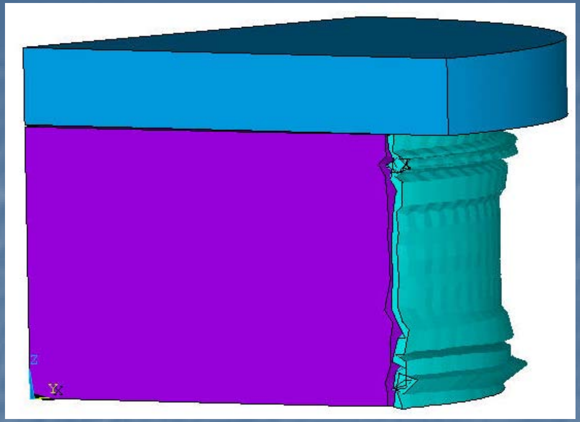
CENM

Impact limiters - Test Simulation

OSB Encased Test



OSB Encased Model



OSB Encased Result

Contacts, Materials Models & Loading

1. Contacts: . defined as ASTS in the ANSYS LS-DYNA

   (Automatic Surface-To-Surface Contact)

   . defined as TDSS in the ANSYS LS-DYNA

   (Tied Surface-To-Surface Contact)

2. All materials, but the OSB and the rigid surface, were modeled as Bilinear Isotropic Material (BISO)

   The rigid surface was modeled with the RIGID option and same properties as the steel

3. Loading - **initial velocity** (corresponds to 9 m drop) plus the **gravity acceleration**

Other Material Properties

	Material / Component					
	Steel	Lead	Fictitious Mass	Bar	Rigid Surface	units
Young's modulus	200 e9	14 e9	2 e9	200 e9	200 e9	N/m^2
Poisson´s Coefficient	0.30	0.42	0.0	0.30	0.30	---
Density	7500	11500	600	7500	7500	Kg/m^3
Yielding stress	310 e6	14 e6	0.0	310 e6	-----	N/m^2
Tangent modulus	7.6 e8	1.0 e7	0.0	7.6 e8	-----	N/m^2

Scale Correction

Scale correction:

Some values measured or obtained during the 1:2 scale model testing or the 1:2 scale model analyses should not be directly associated with the full cask (the prototype).

In particular, for the acceleration:

$$Acc_{model} = \frac{Acc_{protot}}{S_f} \Rightarrow \text{Half scale} \rightarrow S_f = \tfrac{1}{2}$$

So, the obtained acceleration in the 1:2 scale model should be divided by 2 to have the correspondent value in the prototype.

Scale Correction

Scale correction – Other factors

Subscripts: ‘m’ - scale model
‘p’ - prototype

Parameter	Relationship		Parameter	Relationship
1. Weight:	$W_m = S_f^3.W_p$		4. Velocity:	$V_m = V_p$
2. Force:	$F_m = S_f^2.F_p$		5. Acceleration:	$a_m = a_p / S_f$
3. deformation:	$\varepsilon_m = S_f.\varepsilon_p$		6. Duration:	$t_m = S_f.t_p$

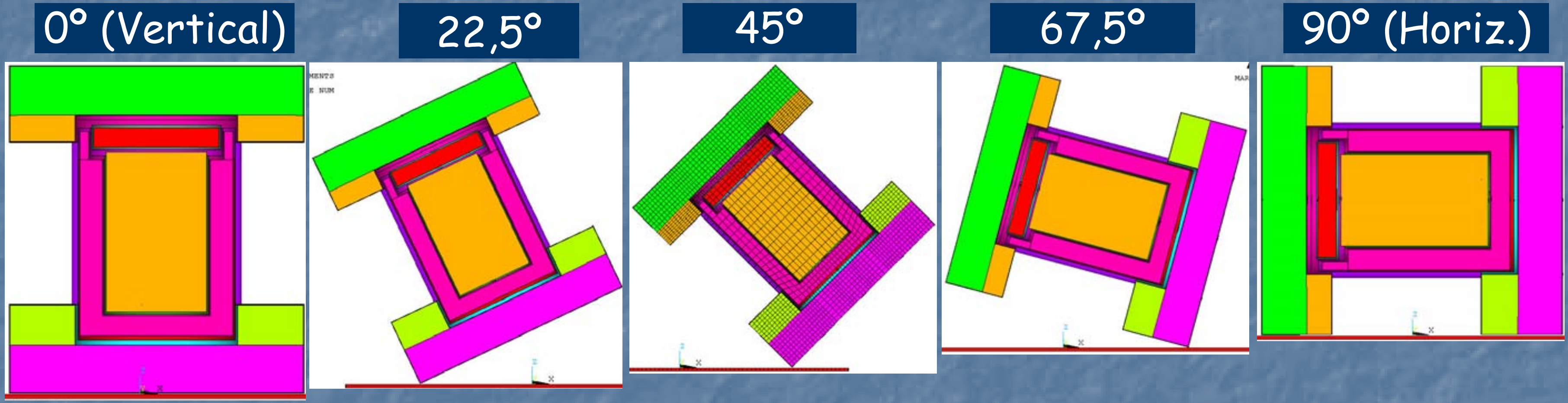
half-scale (1:2) model → $S_f = ½$

Analyses done

. The most damaging position should be investigated

. The first one analyzed was the upright one, 0º (Vertical)

. Some skewed positions were also analyzed

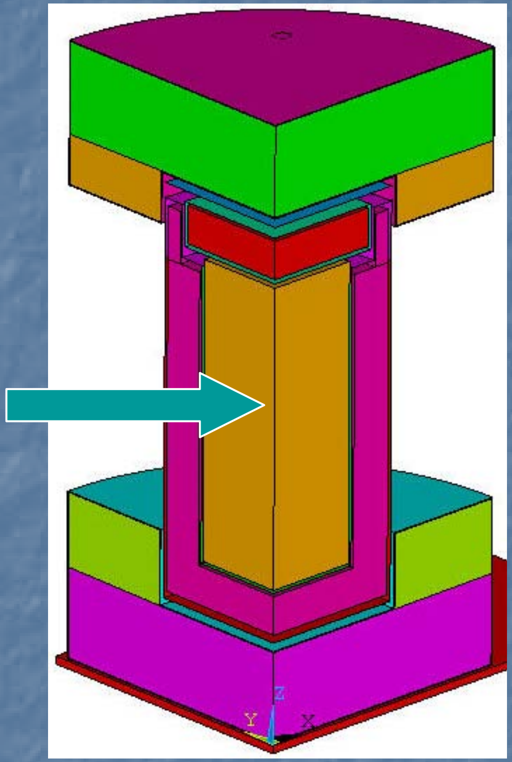


Some of the Analyzed Positions

FE model Results

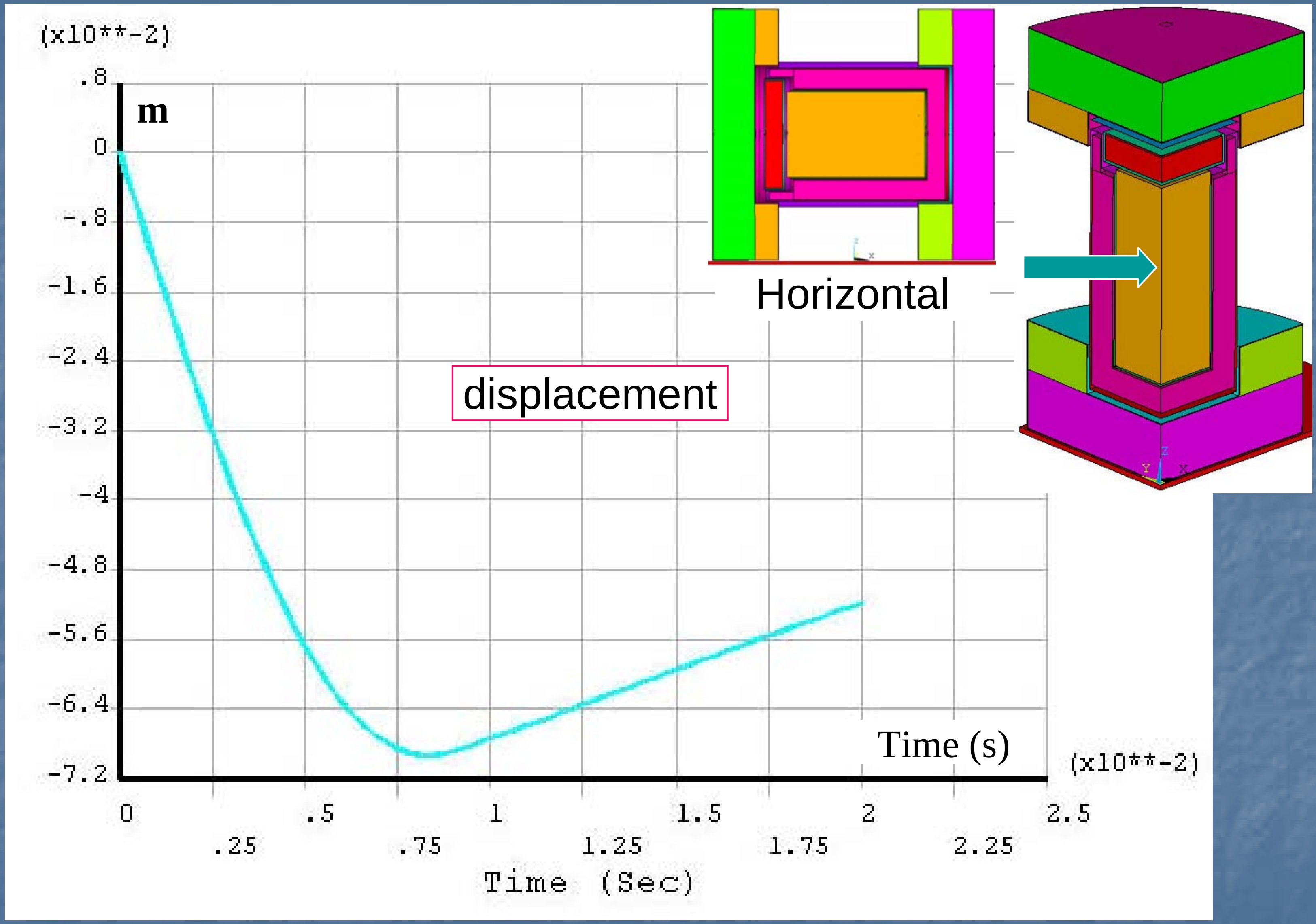
Results:

1. For the indicated node (in the middle of the internal (dummy) mass):
    - the vertical displacement
    - the vertical velocity
    - the vertical acceleration
2. The deformed structure at the instant of max. deformation
3. The deformation of the structure during the analysis first instants (≈until 12ms)

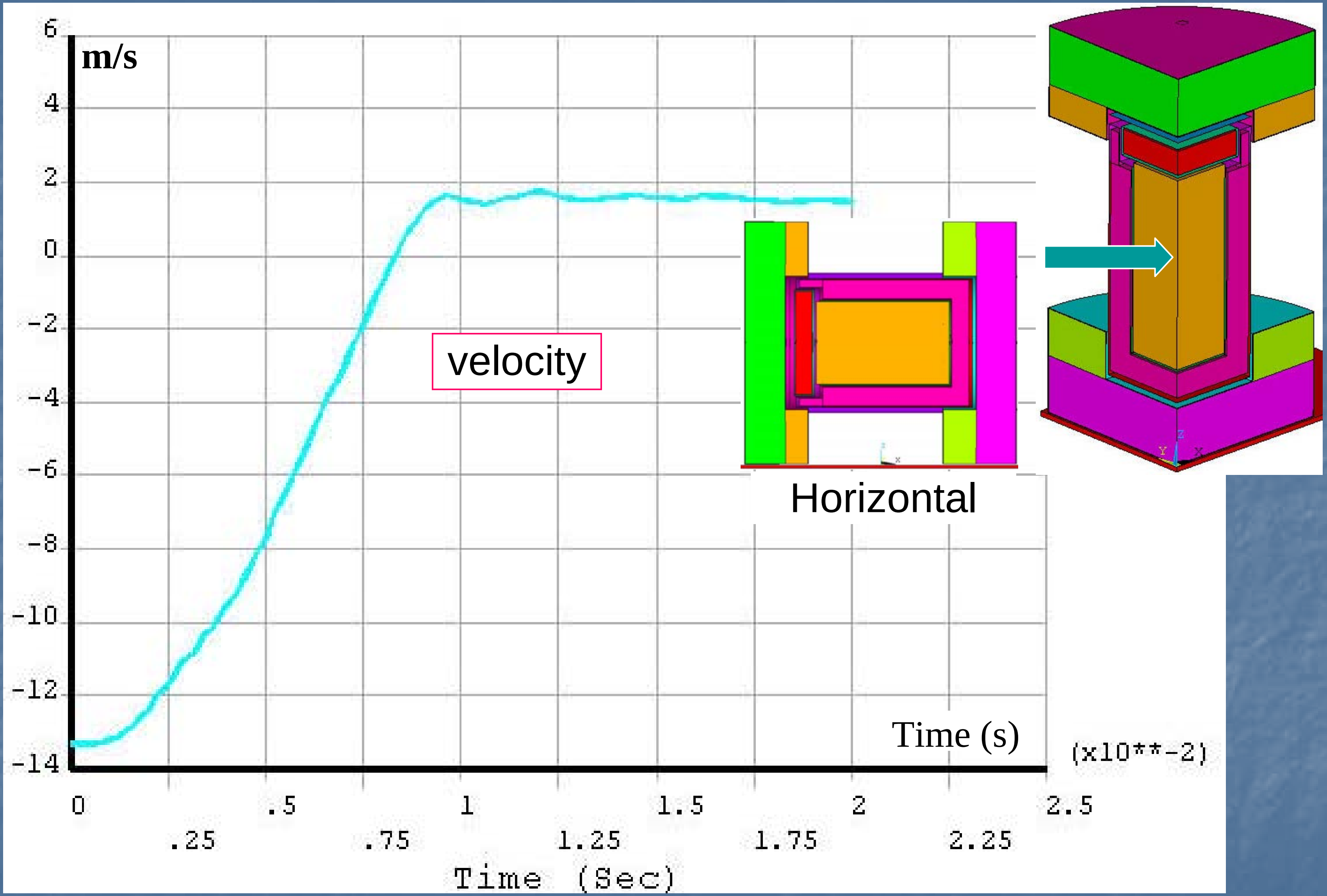


Mostly of the analyses were performed until 20 ms

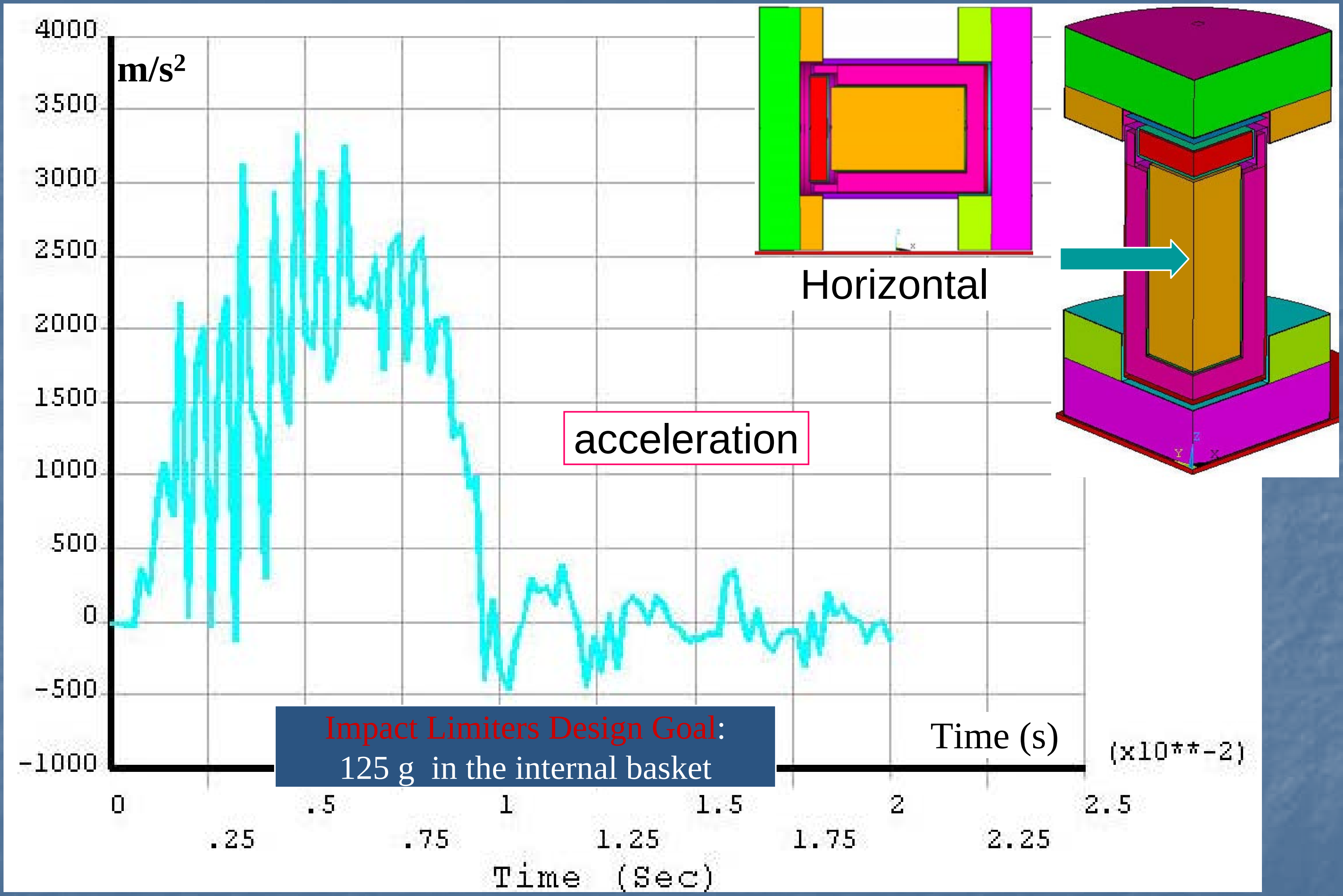
FE model Results



FE model Results



FE model Results



Impact Limiters Design Goal:
125 g in the internal basket

FE model Results

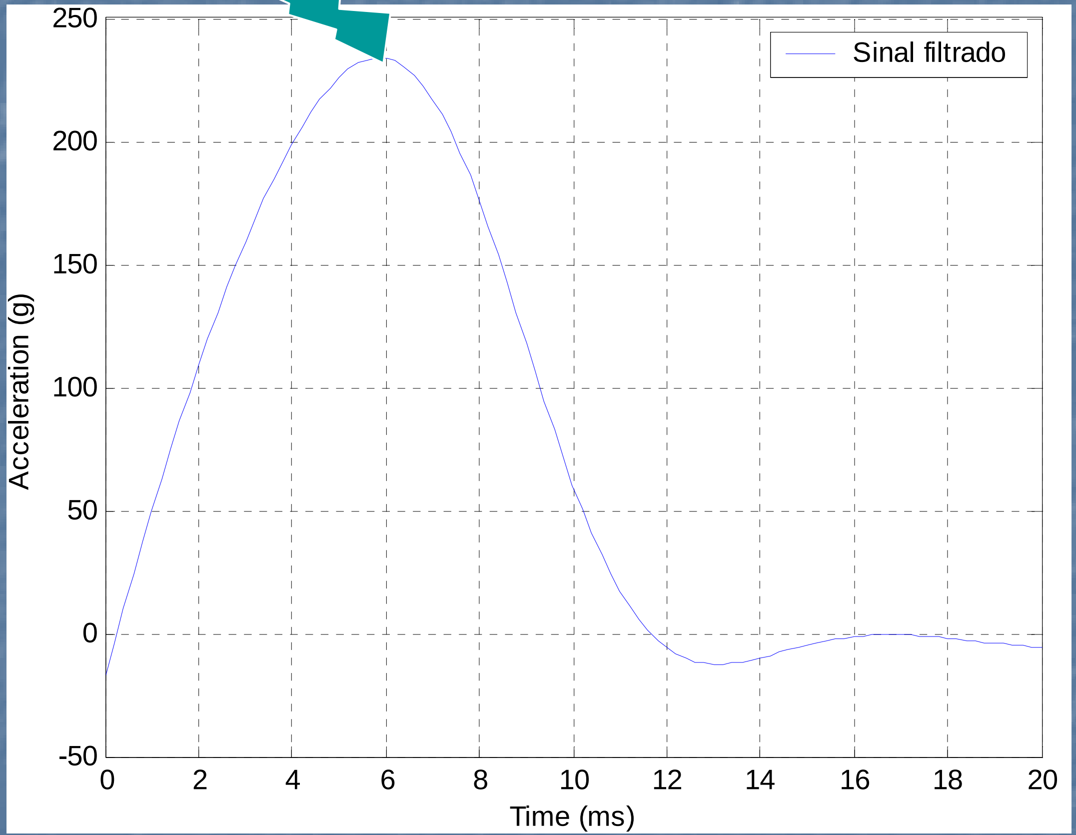
1:2 Model

Max. Acceler. ≈ 240 g

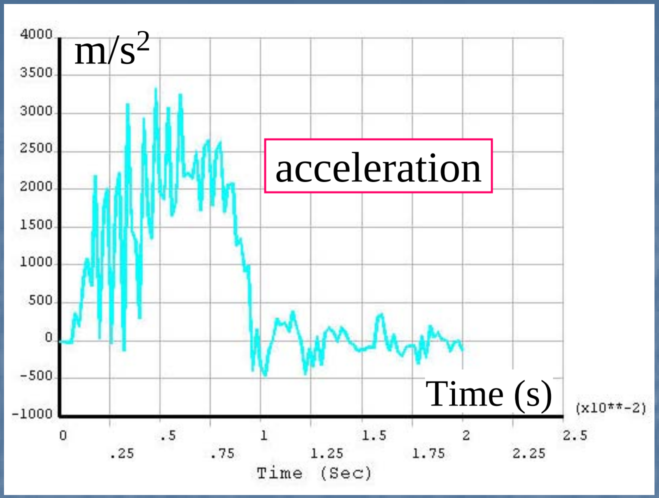
Scale correction

Prototype

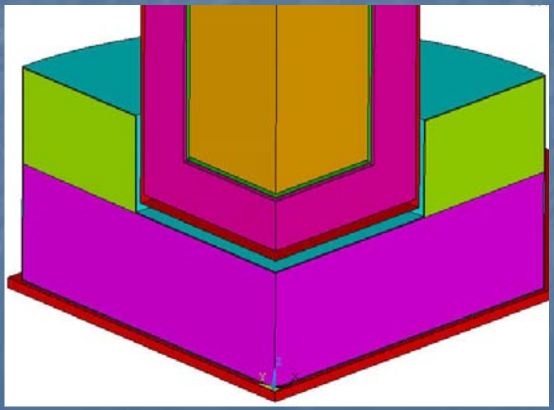
Expected Max. Acceler. ≈ 120 g



Acceleration - After Signal Filtering (Butterworth)

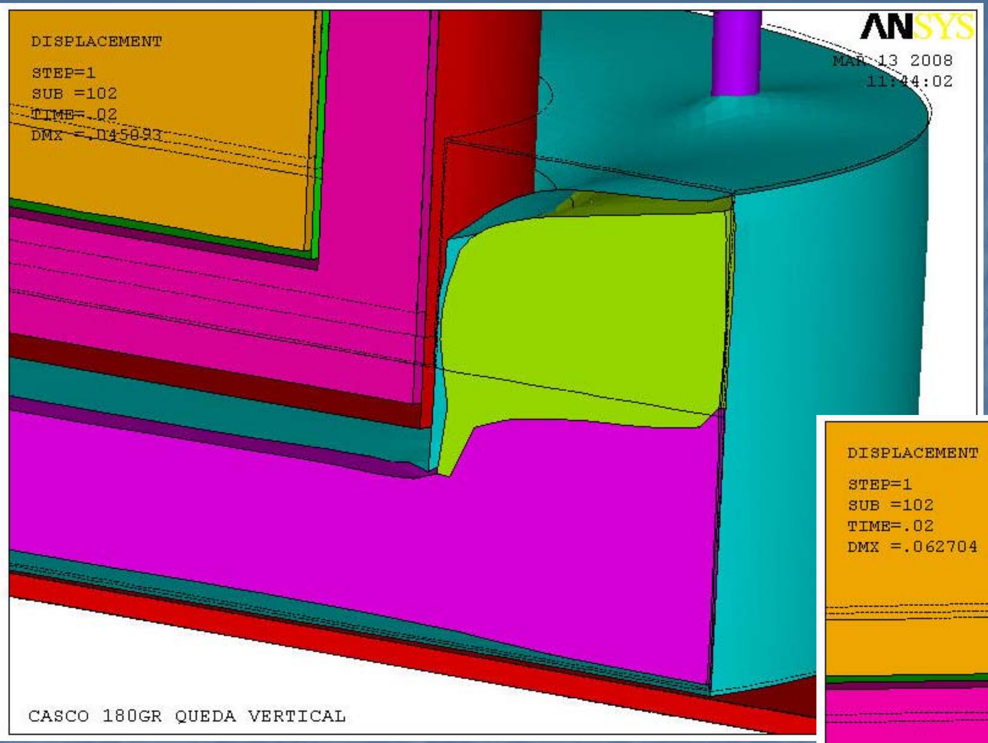


Impact limiters Design Goal:
125 g in the internal basket

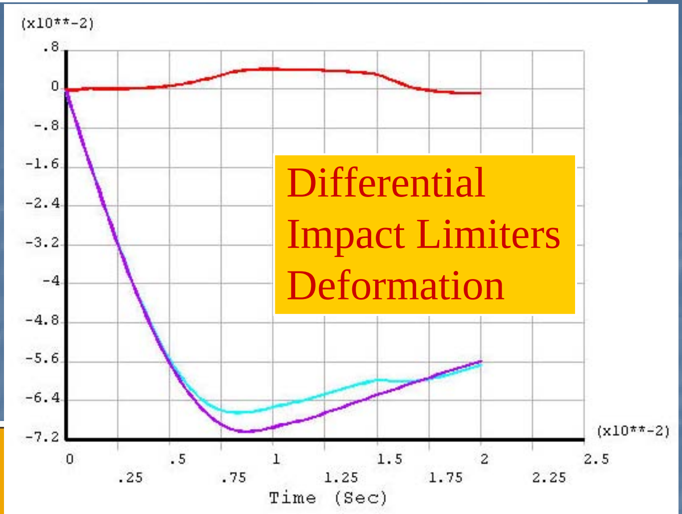


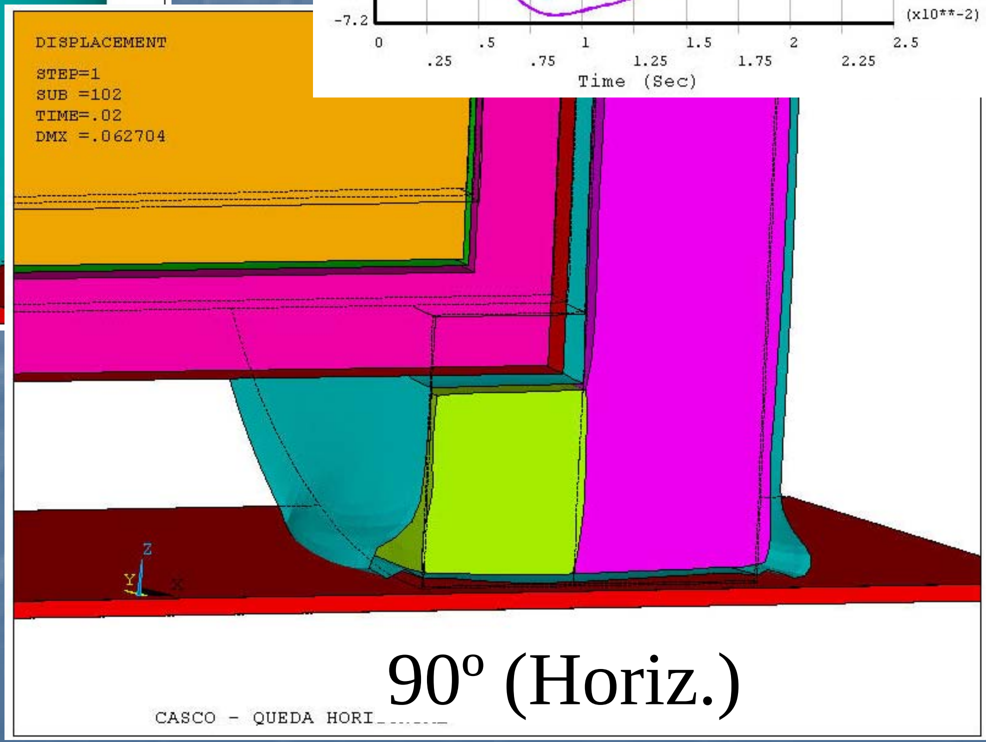
CENM

FE model Results



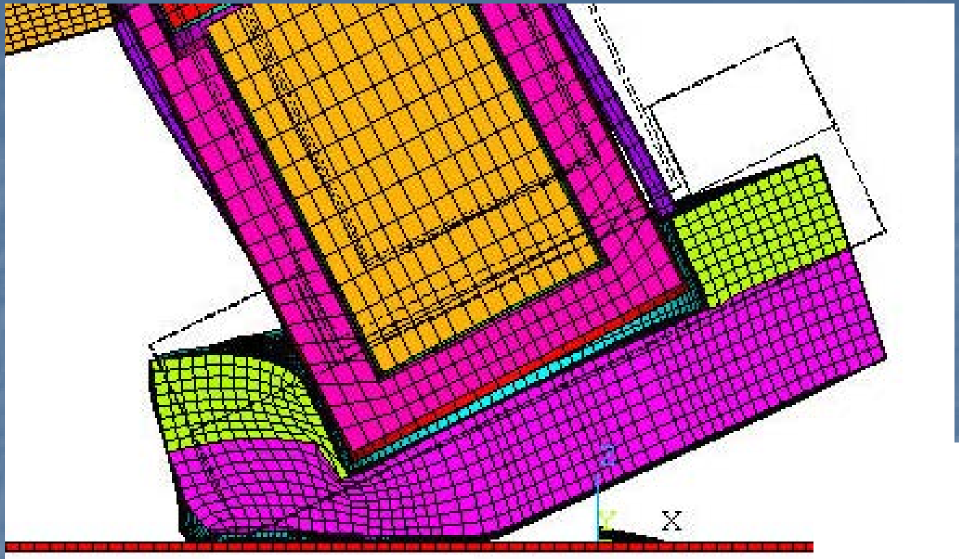
0º (Vertical)





90º (Horiz.)

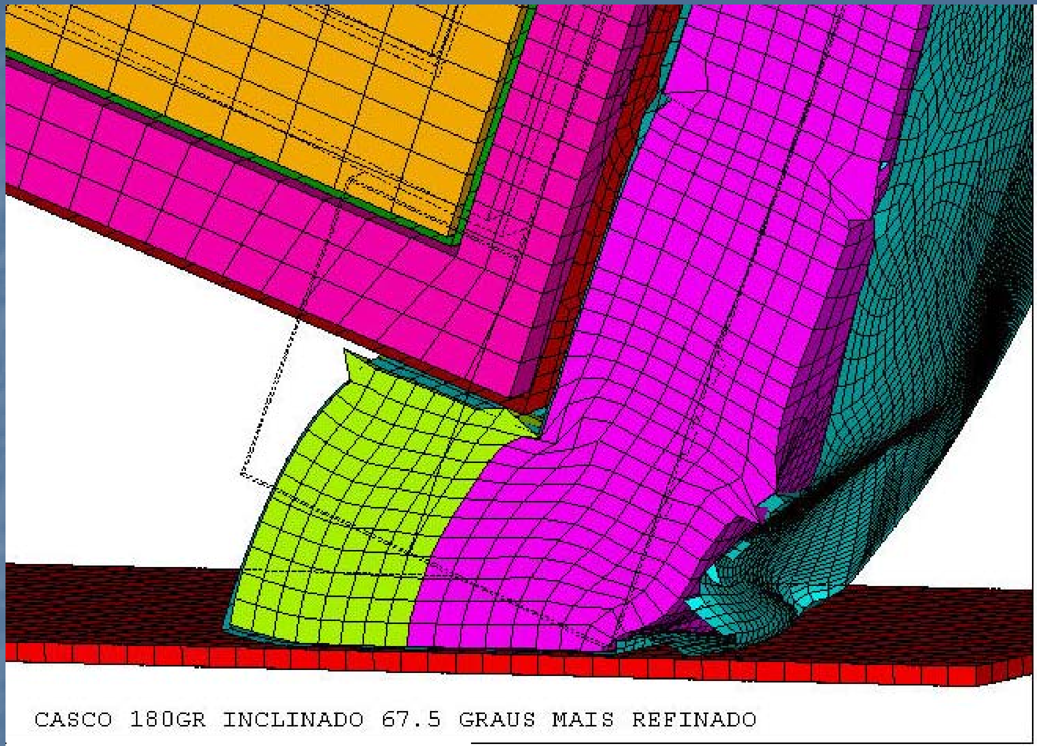
FE model Results



22,5º

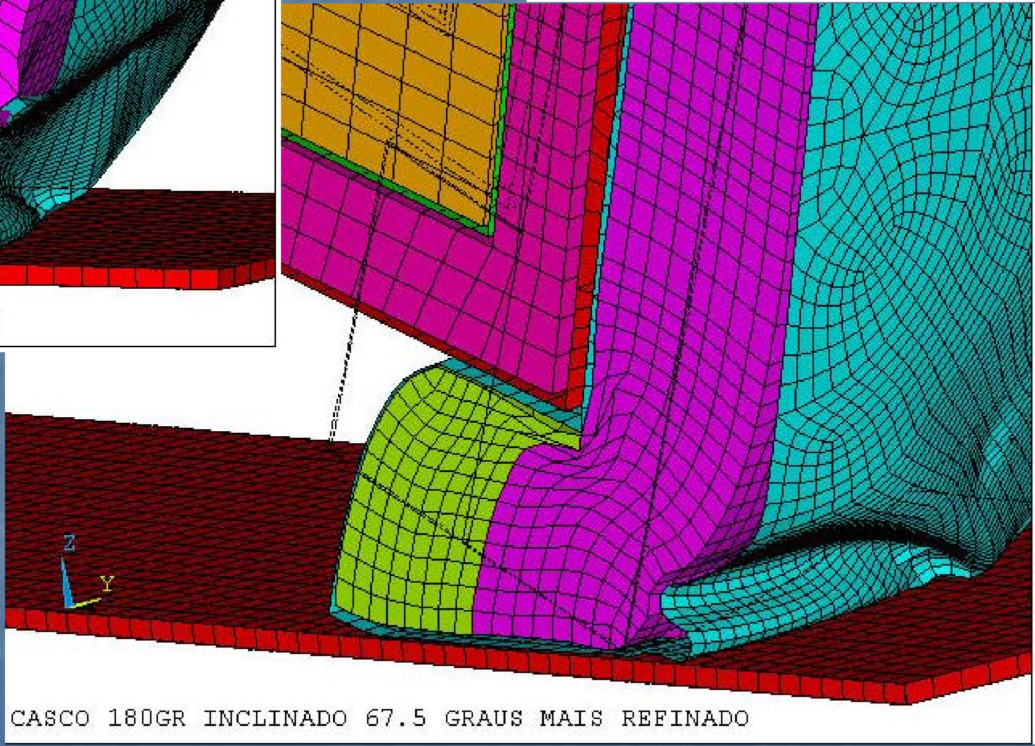
45,5º

FE model Results



67,5°
(10ms ANSYS)

67,5°
(LS-DYNA Solver)

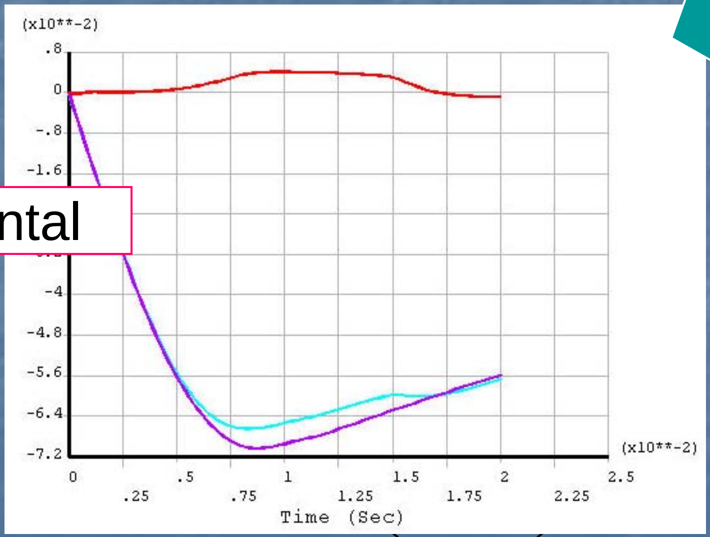


FE model Results

Some Specific Results

	Deceleration (g)	Deformation (mm)
Vertical	≈ 430	45.5
Horizontal (90º)	≈ 250	66.1 & 69.1 (avg=67.6)
Corner (45°)	≈ 130	153.6

Horizontal



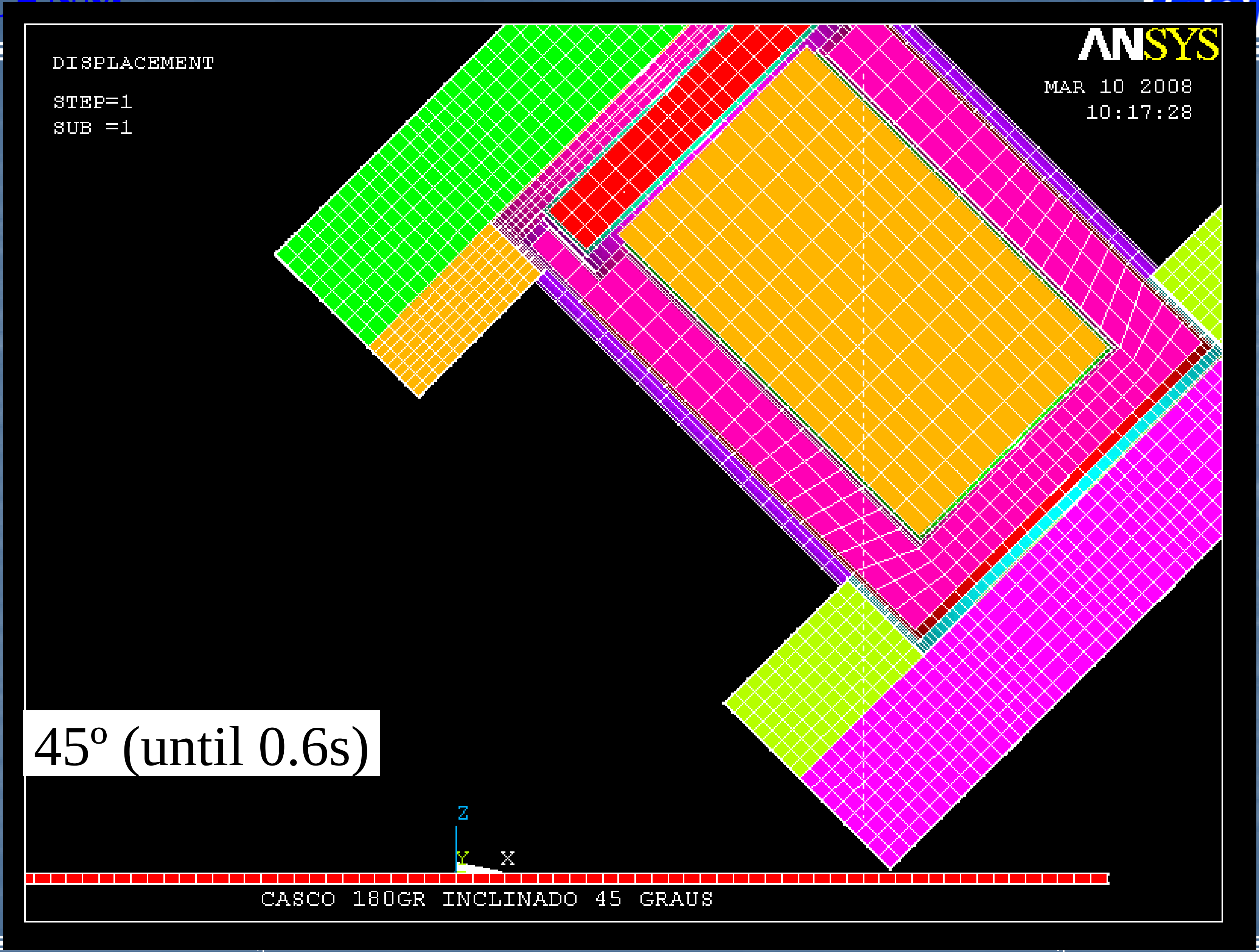

DISPLACEMENT
STEP=1
SUB =1
ANSYS
MAR 6 2008
15:18:42
Z
Y
X
CASCO 188GR QUEDA VERTICAL

0º (Vertical)

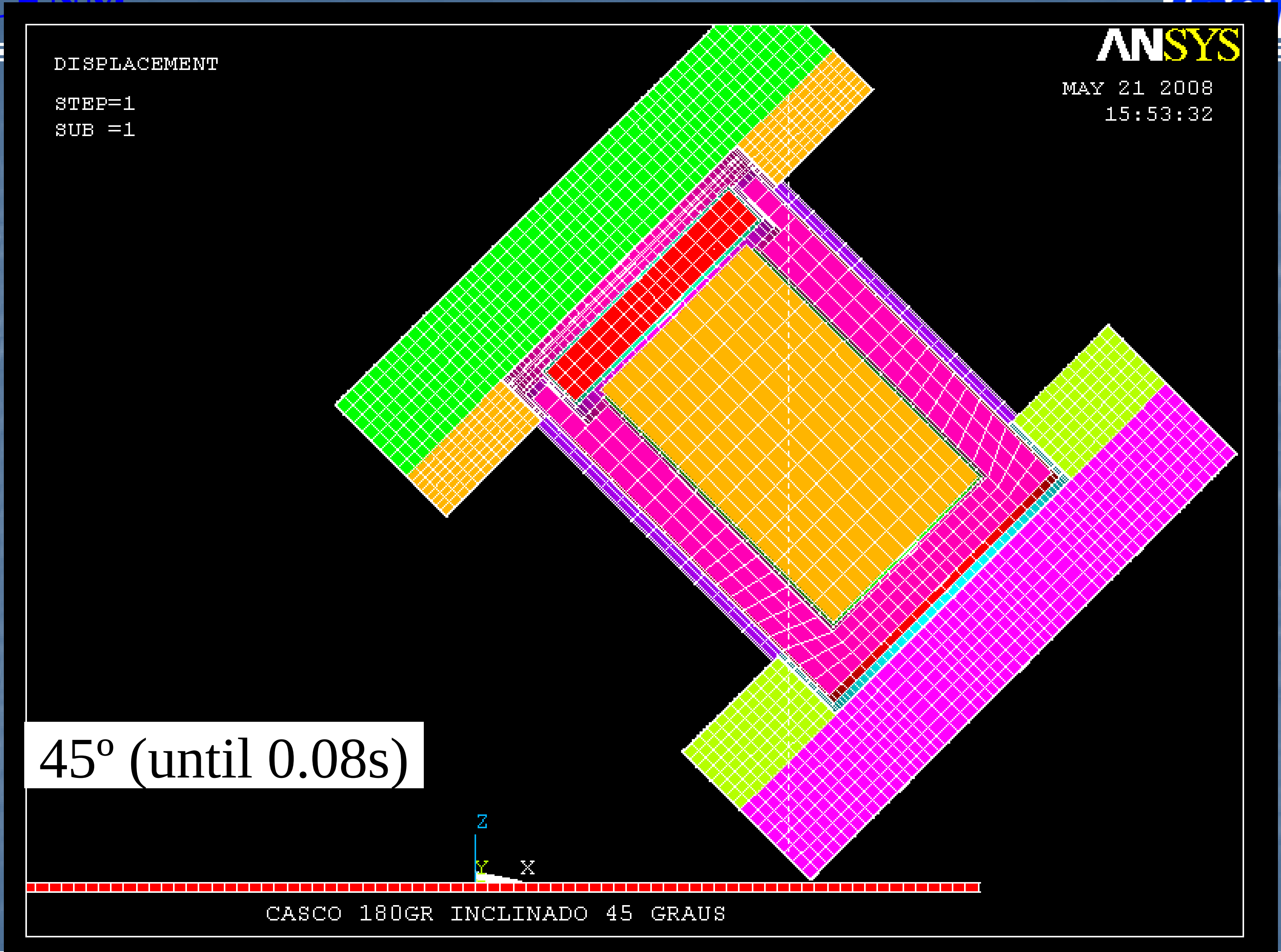
CENM


1
DISPLACEMENT
STEP=1
SUB =1
ANSYS
MAY 20 2008
09:48:06
Z
Y
X
CASCO 180GR INCLINADO 25.5 GRAUS MAIS REFINADO

25,5º

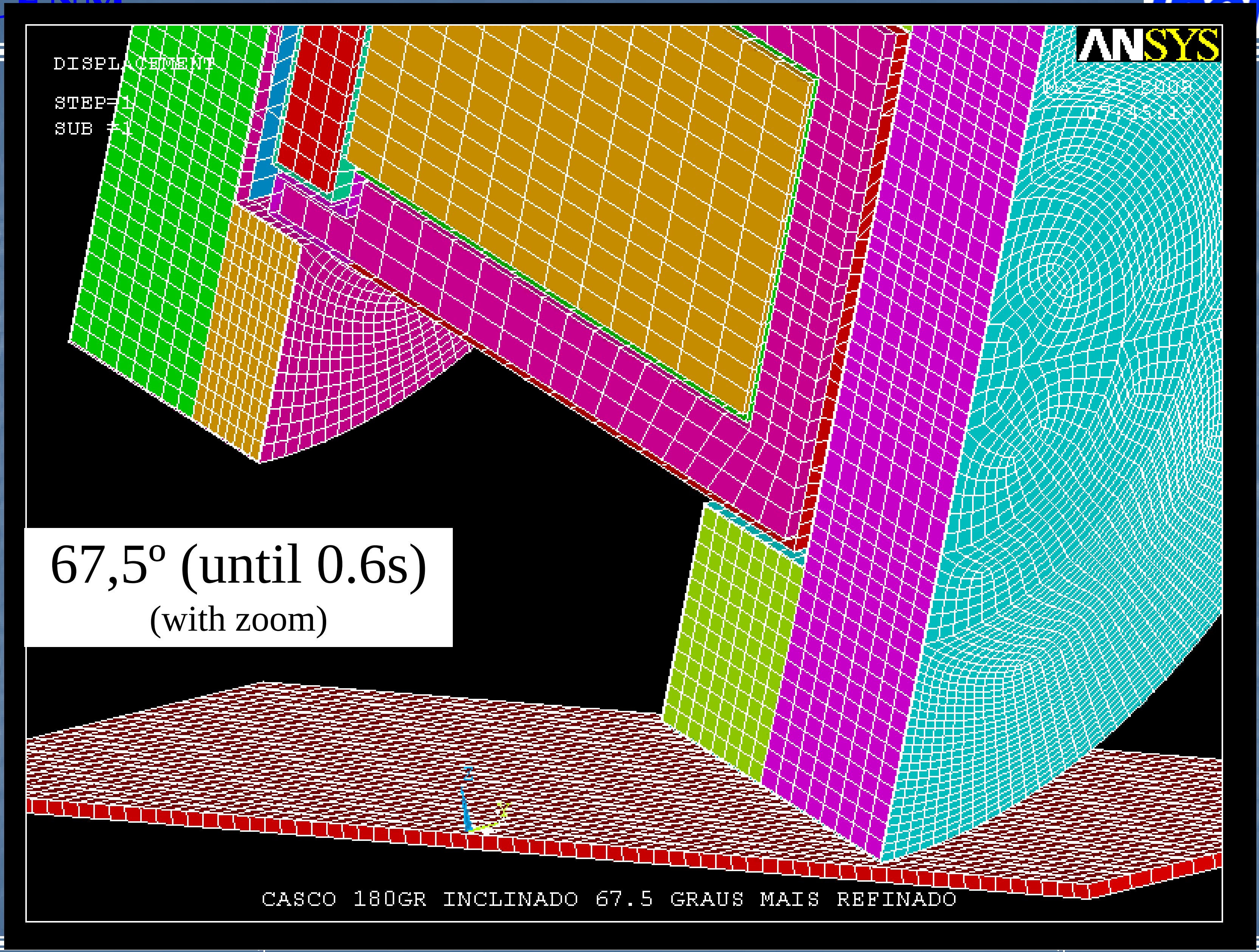


45º (until 0.6s)

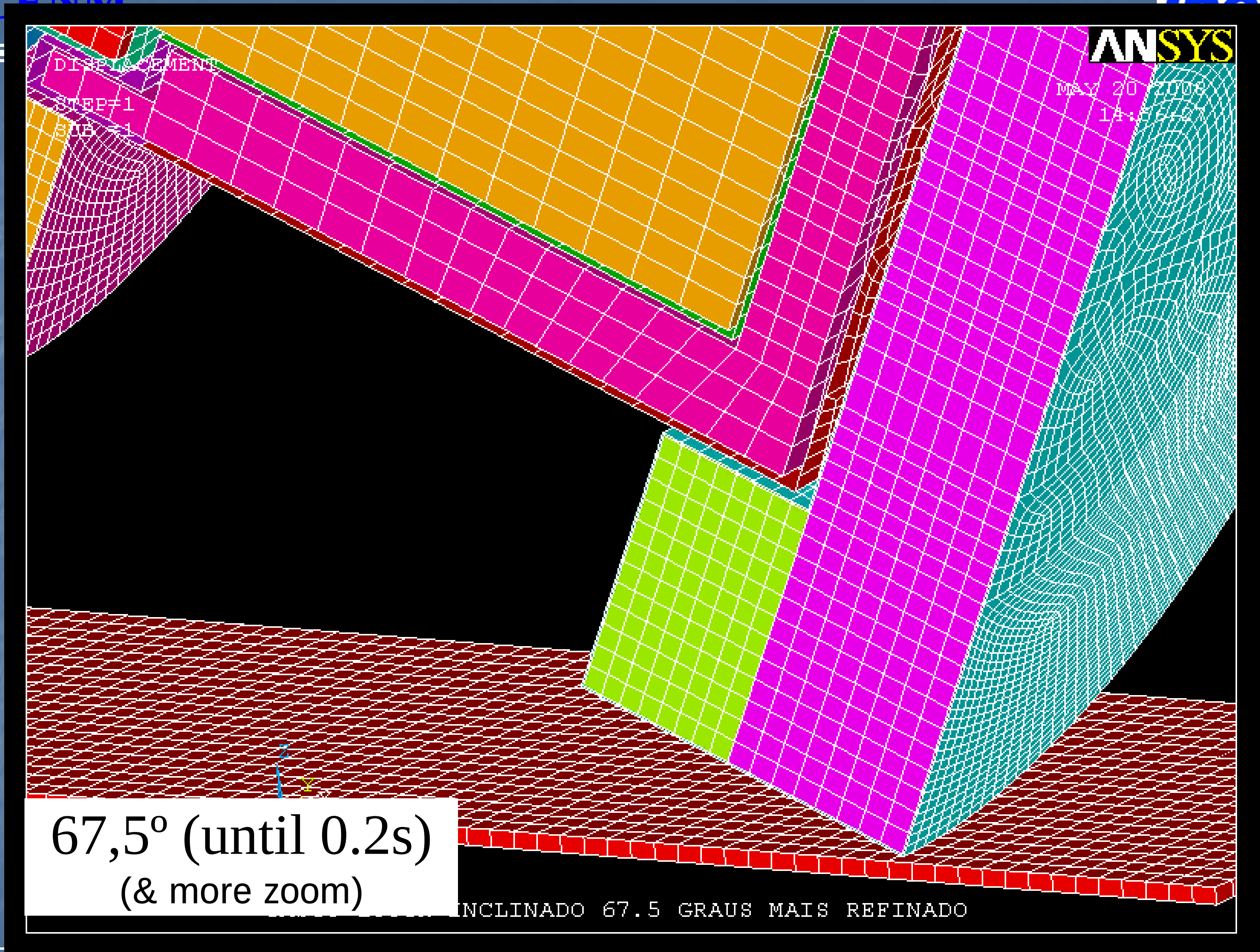


45º (until 0.08s)

CENM

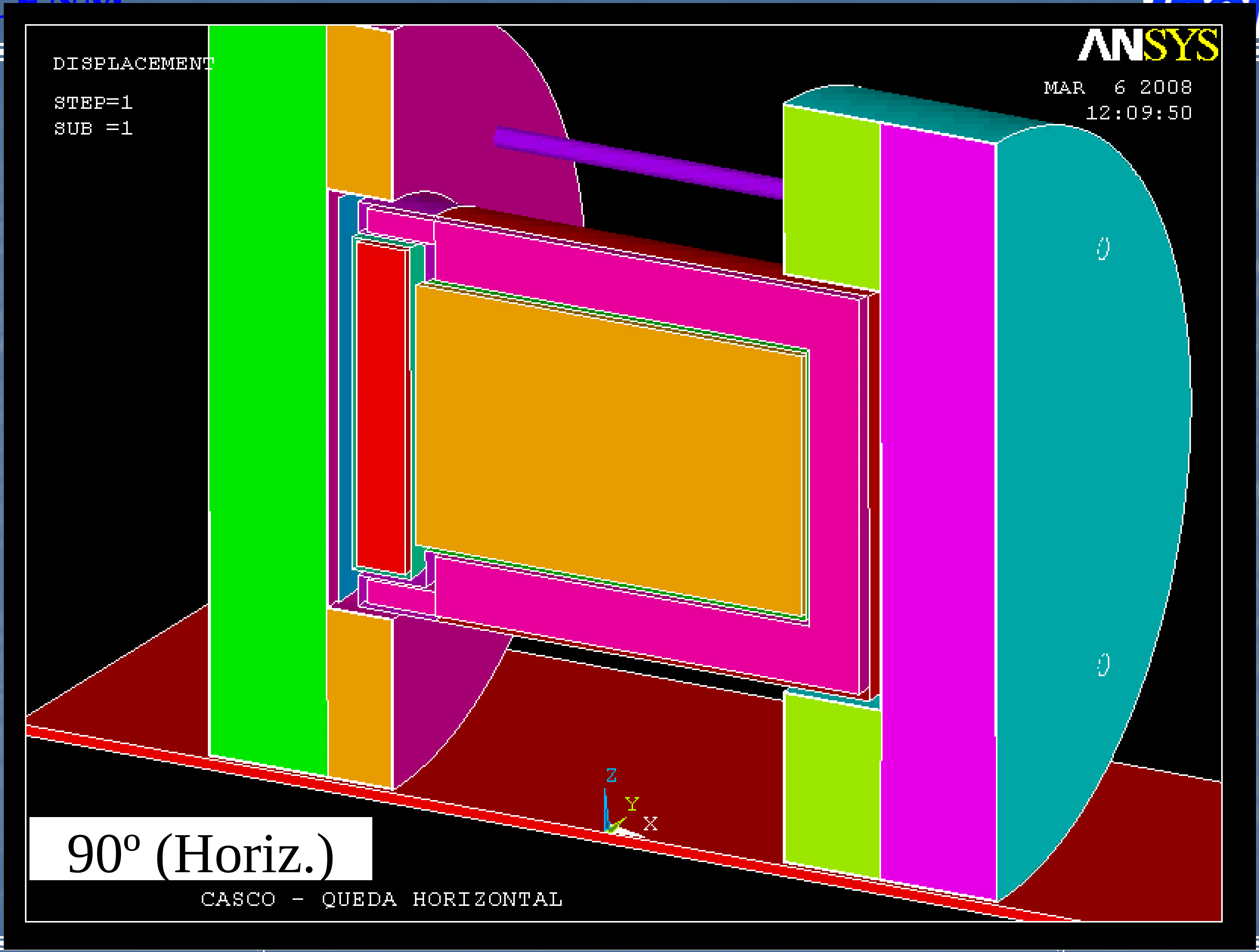


67,5º (until 0.6s)
(with zoom)



67,5º (until 0.2s)
(& more zoom)

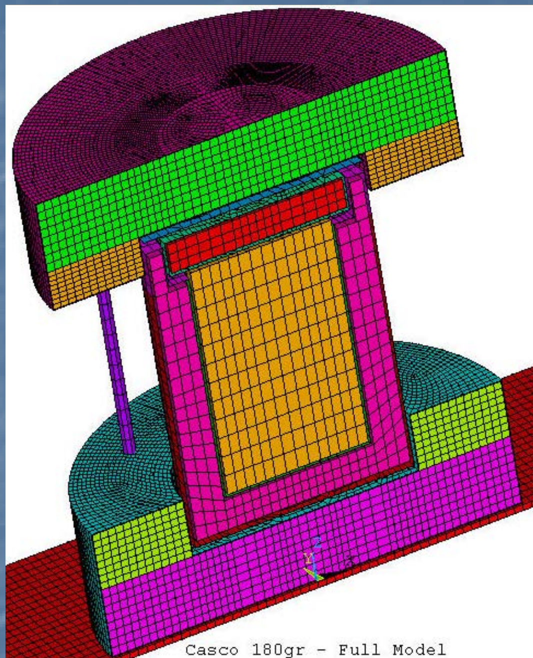
CENM

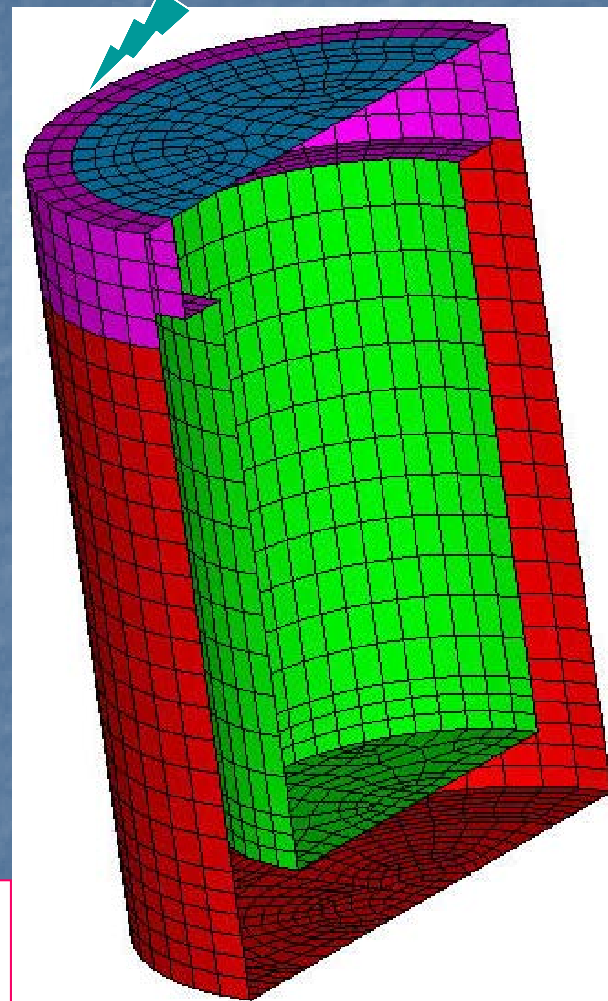

DISPLACEMENT
STEP=1
SUB =1
ANSYS
MAR 6 2008
12:09:50
Z
Y
X
90º (Horiz.)
CASCO - QUEDA HORIZONTAL

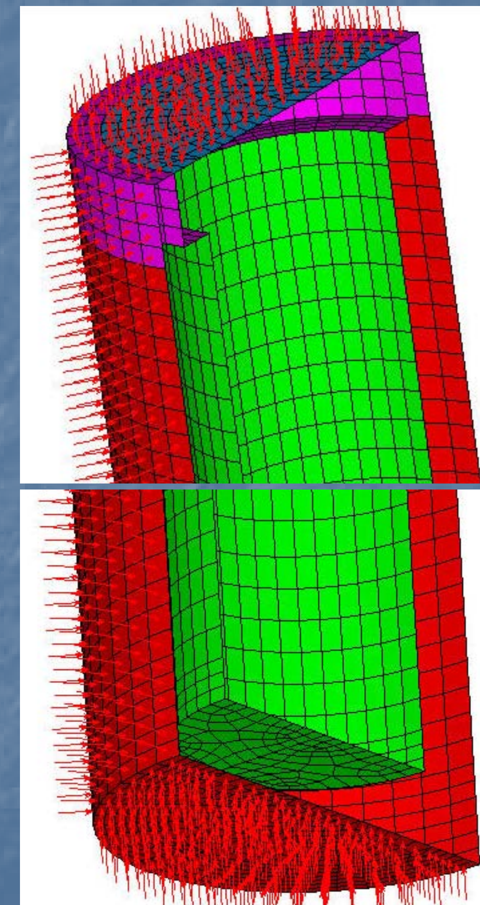
FE model - Ext. Pressure

Merged nodes
(Connection & Lid)

1.5 bar → 0.15 MPa







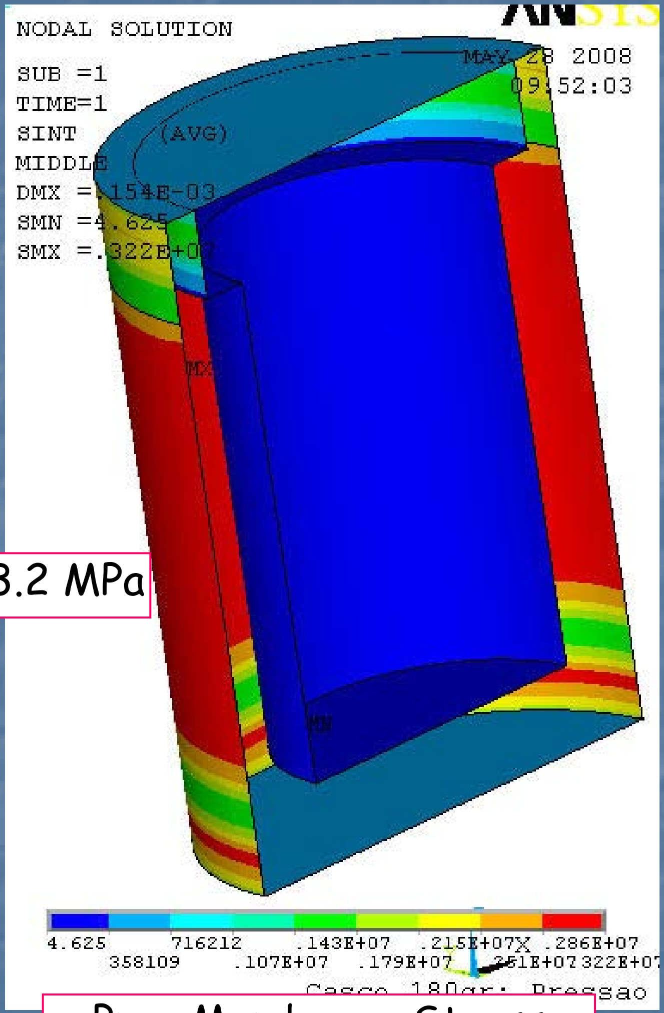
Boundary Conditions:
Symmetry on XZ plane

CENM

FE model - Ext. Pressure

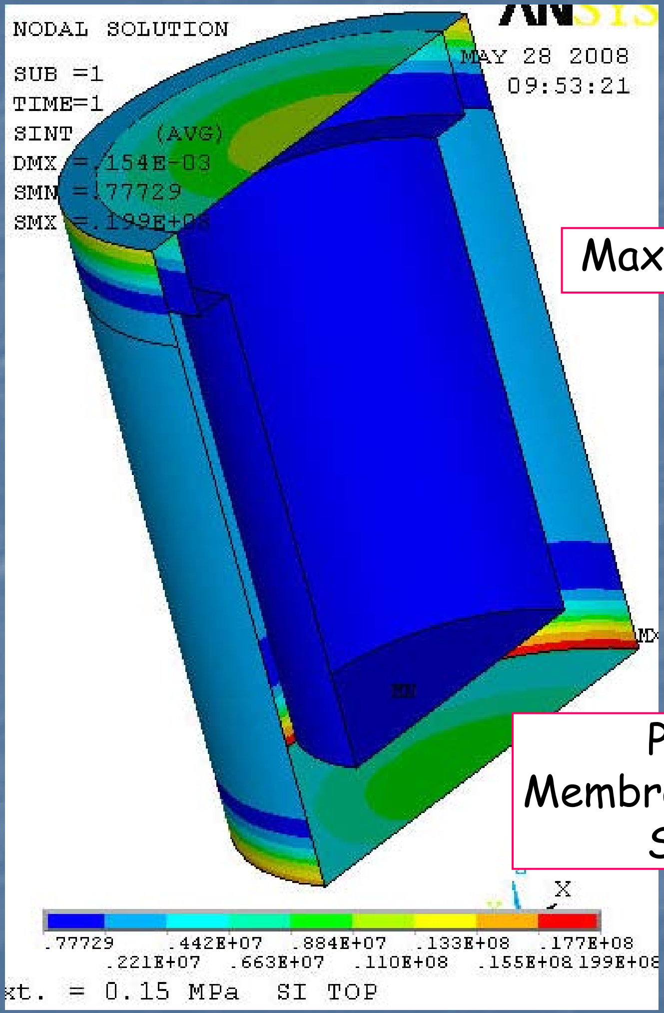
p = 0.15 MPa

Results: Pm & Pl+Pb Stresses



Max = 3.2 MPa

Pm - Membrane Stress



Max = 19.9 MPa

Pl+Pb – Membrane+Bending Stress

Discussion and Conclusions

1. These analyses were performed with preliminary data (mostly for the materials and, in some aspects, for modeling)

2. Scale correction: for the acceleration & scale model 1:2:

$$S_f = ½ \Rightarrow Acc_{protot} = Acc_{model} / 2$$

3. The obtained max. accel. (≈250 g) in the ½ scale model during the HORIZONTAL drop predicts ≈125 g in the internal mass prototype which is compatible with the project design hypothesis (≈125 g) – The same should be done for the other analyses.

4. For the analyses in skewed positions the 180º FE model has symmetries defined accordingly

Discussion and Conclusions

6. The model for the final simulations will be calibrated with experimental data

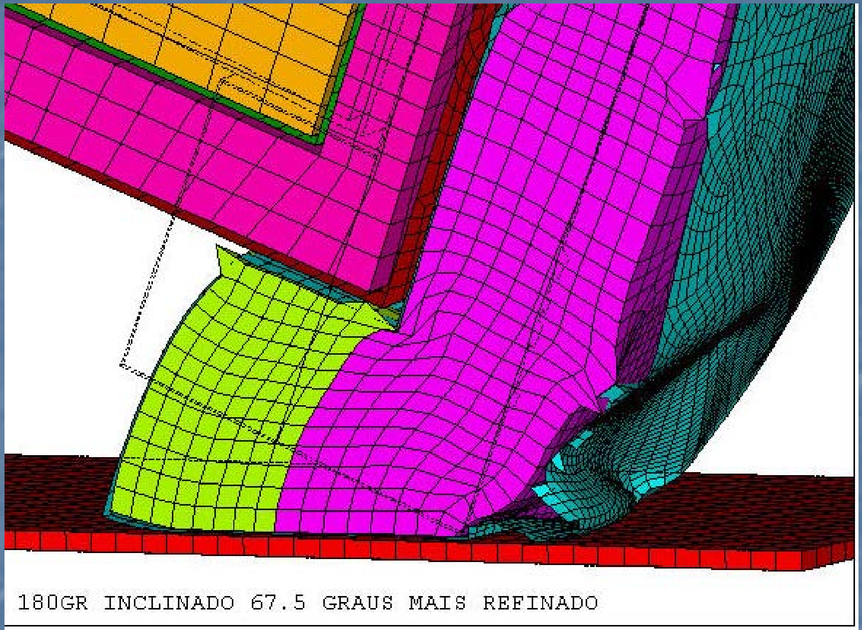
Once the numerical model is calibrated, and all open issues clarified, the prototype structural qualification will be done numerically

7. External Pressure – As expected, the stresses in the cylinder due to the external pressure of 1.5 bar (0.15 MPa) are low (Pm≈3.2 MPa and Pl+Pb ≈19.9 MPa)

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

FE model Results

ANSYS LS-DYNA _X_ LS-DYNA Solver

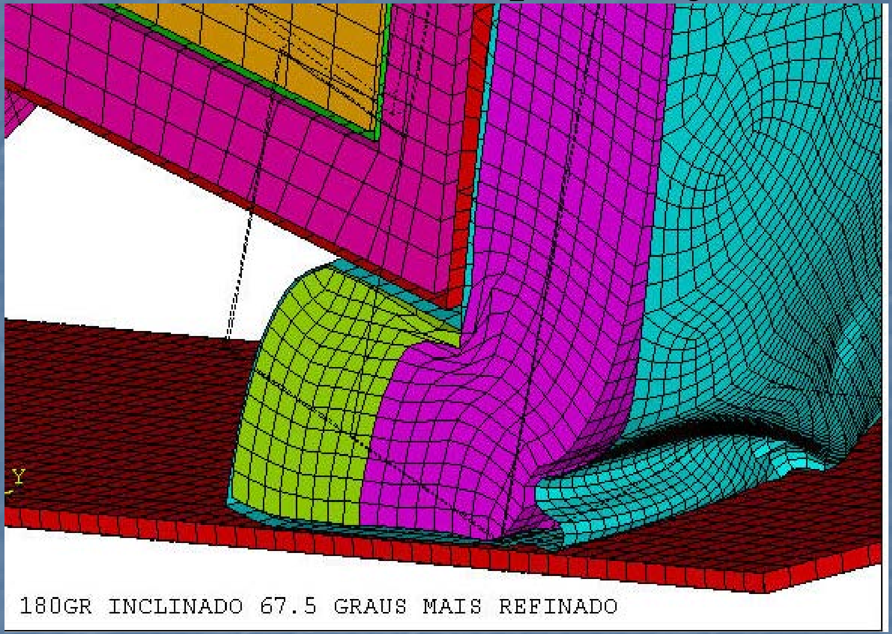


ANSYS

Three steps analysis

LS-DYNA Solver

Five steps analysis



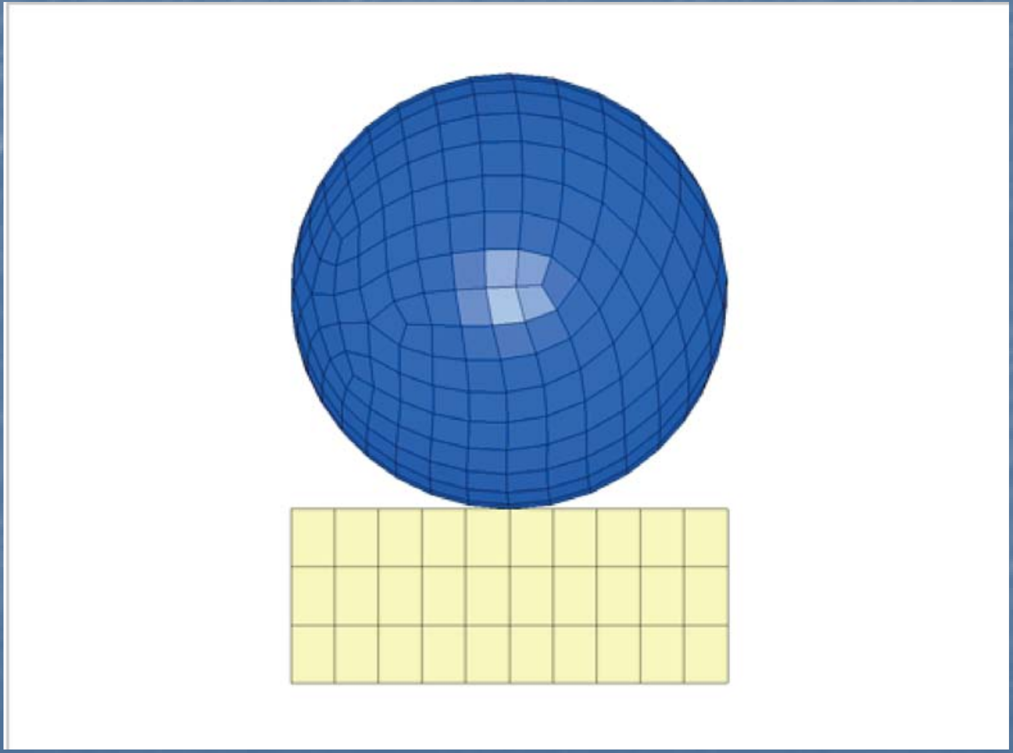
The ANSYS LS-DYNA version does not have the same Contact Controls the (full) LS-DYNA has

Discussion and Conclusions

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

Best Practices for Modeling Contact/Impact with Low Stiffness Materials

Example: Rigid steel ball (solid elements) impacting a block of foam (solid elements)



Discussion and Conclusions

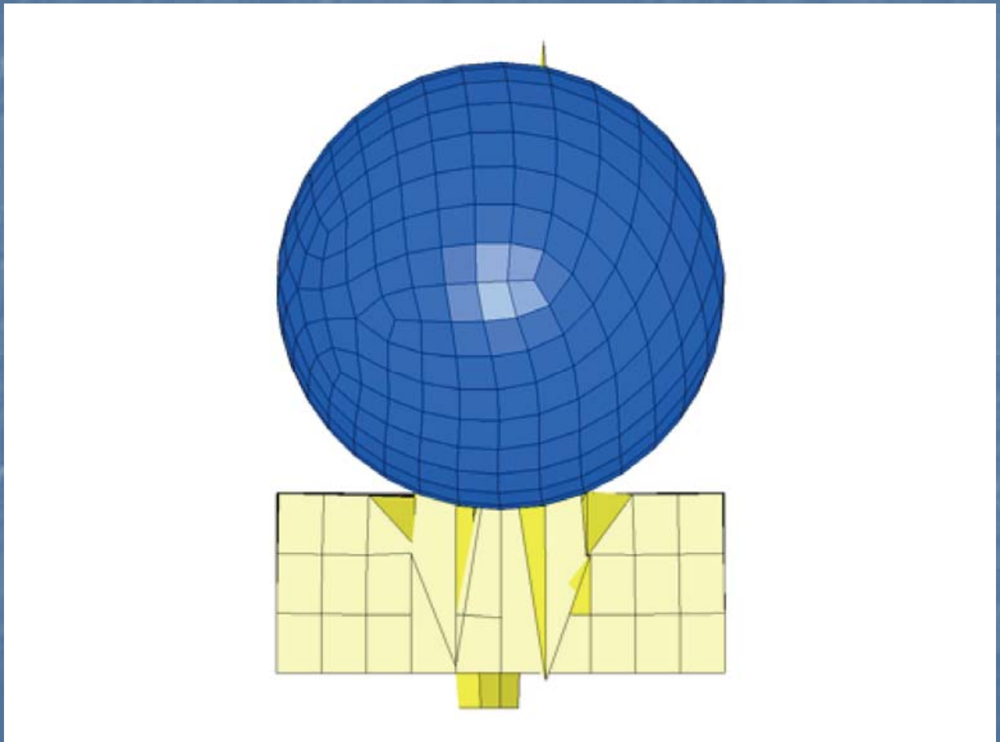
8. ANSYS LS-DYNA _X_ LS-DYNA Solver

To model the interaction between the rigid block and the foam, a two-way contact such as *AUTOMATIC_SURFACE_TO_SURFACE is included.
The recoverable low density foam is modeled using *MAT_LOW_DENSITY material model whose inputs include density, elastic-modulus, and a load curve to define its engineering stress-strain behavior for compression.
The tensile behavior for this model is elastic (uses the compression Young's modulus, E) and optionally has a cut-off stress value after which the stress remains constant in tension.
For simplicity, any hysteresis is ignored and it is assumed that the unloading curve follows the loading curve.
Starting with the simulation of the impact using all default parameters and then incrementally updating the modeling parameters by monitoring the simulations results.

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

(1) Default Parameters

This simulation consists of all default parameters and with this definition, the job terminated abruptly with messages of ‘Negative Volume’ and ‘Complex sound speed’.



8. ANSYS LS-DYNA _X_ LS-DYNA Solver

Discussion — Timestep and Contact Stiffness for Foam Materials

By default, LS-DYNA computes the timestep and the contact stiffness based on the maximum value of [Young's Modulus (E, specified in the material card), the maximum slope from the stress-strain curve (Ecurve)]. This default approach is conservative to ensure that the computed timestep is stable for all compressive strains. The default value of the modulus from this approach could either be too small (if E is greater than Ecurve) or too large (if the Ecurve is greater than E). LS-DYNA allows us to override this default logic by using a non-zero value of KCON.

When KCON is non-zero, the default comparison of E and Ecurve is skipped and LS-DYNA uses KCON for the timestep and the contact stiffness calculations. In the above default simulations, the Ecurve is 10 and E is 50. Consequently, the E value of 50 was used in the timestep and contact stiffness calculations which is roughly 0.025% of the modulus of steel (200000). This huge disparity in stiffness values between the impacting bodies is naturally going to cause instabilities in contact.

Discussion and Conclusions

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

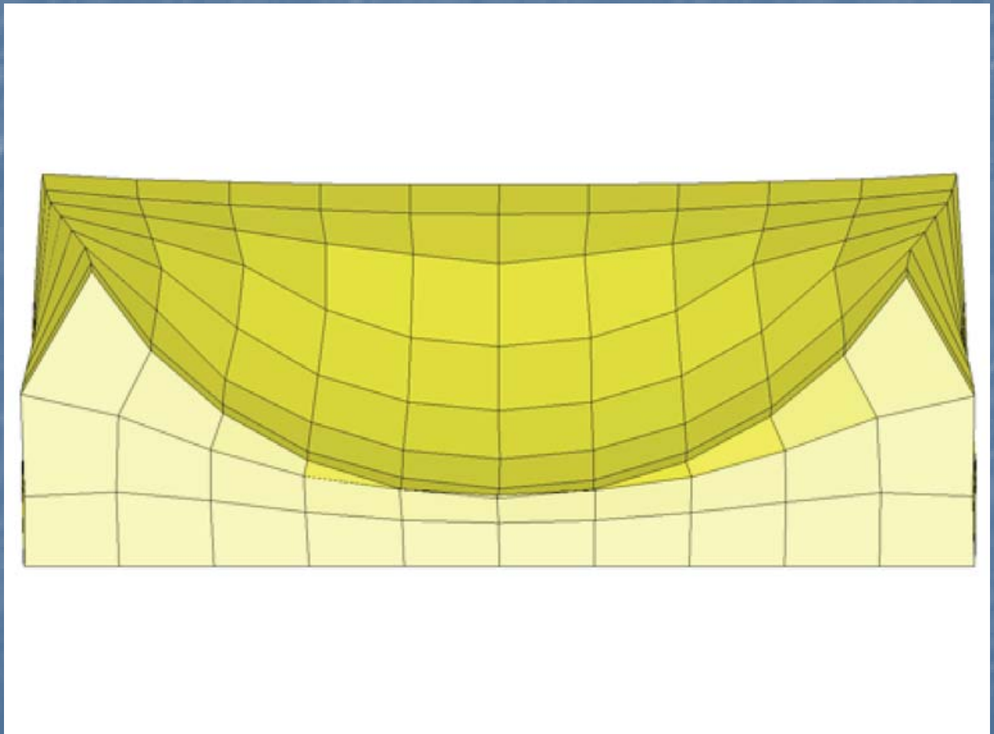
Discussion — Timestep and Contact Stiffness for Foam Materials

The alternative approach is to eliminate dependencies on elastic contacts for contact is to use a penalty contact using soft-constraint algorithm (SOFT=1 in *CONTACT). However, LS-DYNA always uses a penalty based approach based on material stiffness for contact between a rigidbody and deformable bodies. Since our ball is modeled as rigid, switching to a soft-constraint formulation will make no difference in the simulation. Therefore, using KCON is recommended to alter the modulus and the recommended value is at least 1% of the modulus of the impacting material which in our case is steel. So for our next simulation, we set the KCON to 2000 which is 1% of the steel ball modulus of 200,000.

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

(2) KCON = 2000 (1% of Steel Ball Modulus of 200000)

This simulation goes much beyond our run with default parameters but terminates due to similar messages. The final deformed shape is as shown below.



Discussion and Conclusions

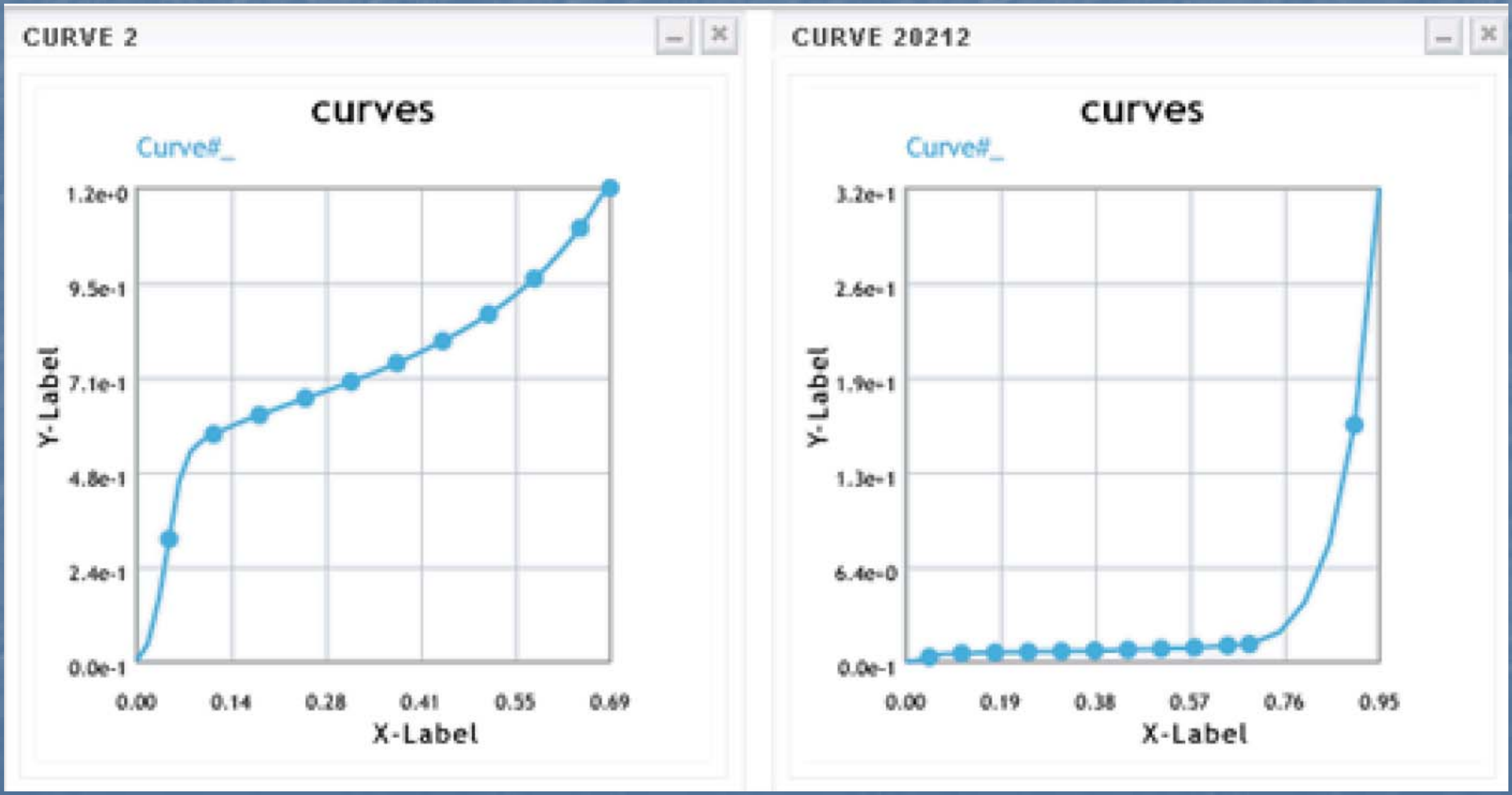
8. ANSYS LS-DYNA _X_ LS-DYNA Solver

Discussion

As one can see from the cross-section image, the first row of foam elements has collapsed which would explain the negative volume issues. The reason for this is the stress determined by LS-DYNA at large compressive strains. The complete stress strain curve is as shown below (left). As it can be seen, the last strain point is limited to around 70% but the simulation results shows the strain has reached nearly 100%. For strain magnitudes larger than the last input point in the curve, LS-DYNA extrapolates using the last slope. This may yield small stress values and fails to model the bottoming out effect that occurs at large compressive strains. The fix to this is to manually provide an exponentially increasing curve to cover compressive strains to a minimum of 95-99%. It must be noted that the manual curve must be smooth. The modified curve is as shown below (right).

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

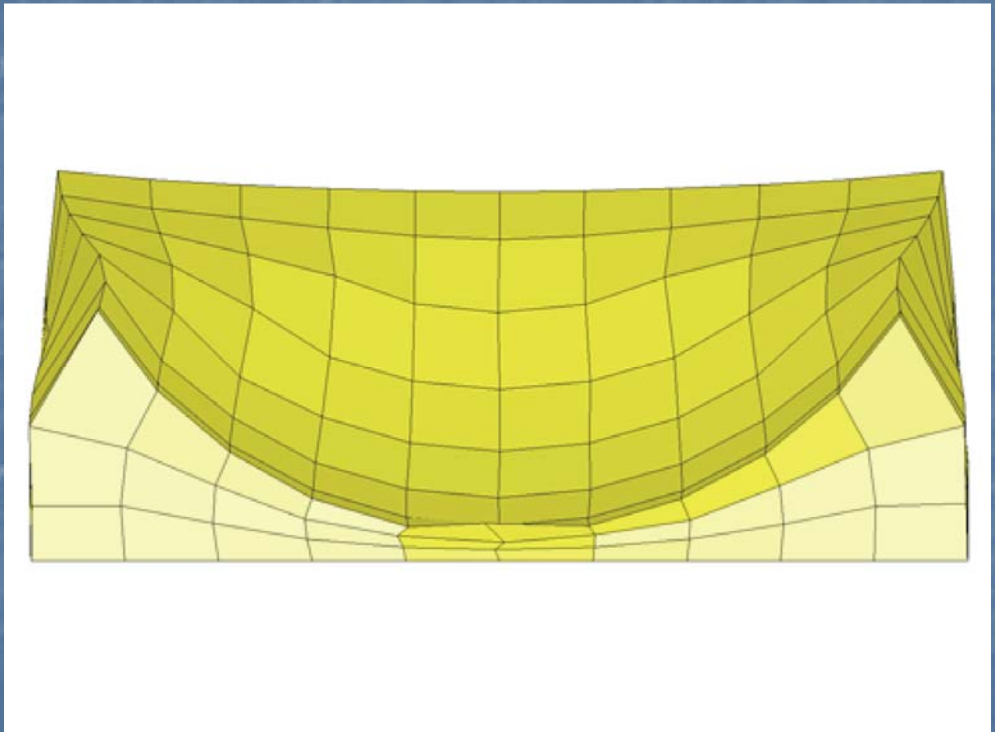
Discussion



8. ANSYS LS-DYNA _X_ LS-DYNA Solver

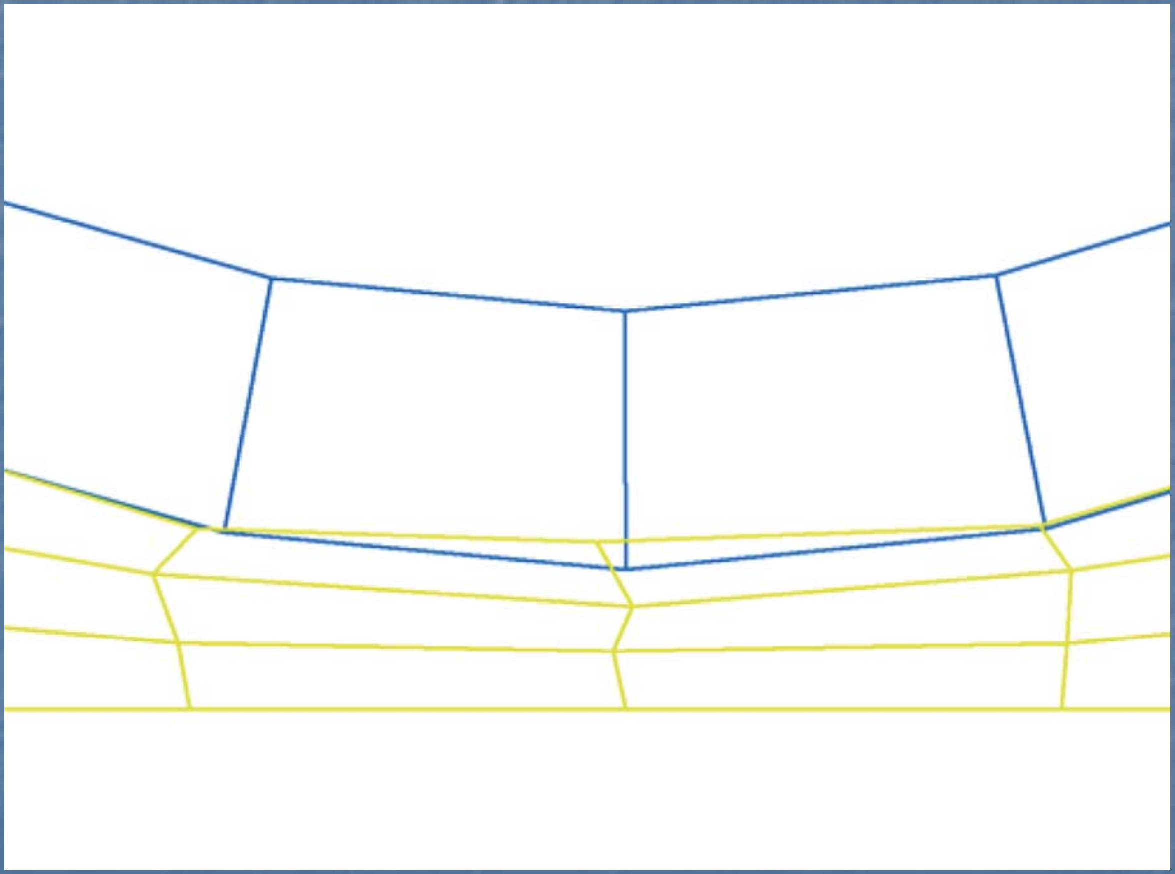
(3) Extended Stress-Strain Curve

This simulation worked as expected with no instability or error message and terminated normally. The final deformed shape and the transient movie is as shown below.



8. ANSYS LS-DYNA _X_ LS-DYNA Solver

Zooming



Discussion and Conclusions

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

Discussion

Upon close examination of results from the run, one can find that there is some penetration between the ball and the foam block during the maximum compressive strain as shown in following figure.

This is attributed to the way the segment thickness is computed for solid elements only for nodal release criteria when using AUTOMATIC type contacts.

Much like the shell elements, in which the mid-surface is offset in both directions of the segment normal, the solid segments maximum allowable thickness is computed.

The amount of maximum allowable thickness is based on a small percentage (5%) of the solid element diagonal which based on the element geometry could be very small.

Small offset values make it vulnerable to nodal release logic used by all AUTOMATIC contacts when there is excessive penetration.

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

Discussion

For solid elements, except for type 4 and type 26 contacts, the tolerance used for determining the nodal release is 5% of the computed solid segment offset distance.

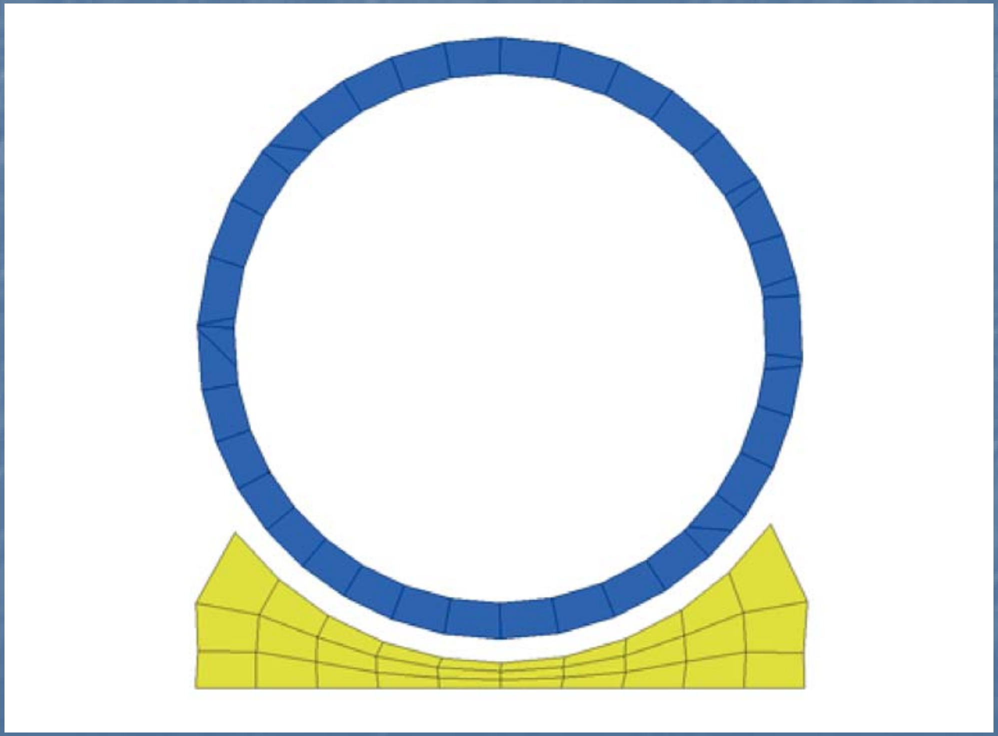
The previous images are a classic case of nodes being released and the fix is to increase the offset thickness.

One can do this by setting SLDTHK in Optional Card ‘B’ to a value greater than 1mm to ensure no nodes are released from contact.

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

(4) Increased Solid Element Thickness in Contact

This simulation works as expected and shows no contact penetration even for large compressive strains and meets all stability and accuracy criteria. The final deformed shape and the transient results are shown below.



8. ANSYS LS-DYNA _X_ LS-DYNA Solver

Ten Recommendations to Improve Contact Stability

(1) Avoid duplicate contact definitions.

The general thumbrule is no pair of node/segment or segment/segment (in soft=2) should be treated for penetration by more than one contact defintion

(2) It is extremely important that the interacting segments are similar is size and stiffness.

Rigid segments , in particular, interacting with deformable segments always use penalty (soft=0) treatment.

(3) Always use a thickness offset (SLDTHK) and reasonable stiffness (SLDSTF) for segments that belong to solid elements.

This is important when using non-structural materials such as foams, rubbers, honeycomb, etc.

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

Ten Recommendations to Improve Contact Stability

(4) IGNORE is the recommended option for models with penetrations. Always set this option to 1

(5) Shell segments in single surface contact do not use the actual shell thickness. It is recommended to use SSTHK=1 for uniform and true shell thickness.

(6) With regards to 5th point, ensure to use realistic shell thickness values.

(7) Mesh interacting objects such that the force distribution is uniform and not concentrated.

8. Use SOFT=1 when interacting pair of segments have mis-match (order of magnitude) material properties.

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

Ten Recommendations to Improve Contact Stability

9. Avoid thin segments (for contact). Segments less than 0.6mm-0.8mm is considered thin.
By doing this we are ensuring that no node travels more than 40% of the interactive segment thickness in one single timestep. The other alternative is reducing the timestep (TSSFAC). But usually the problem lies in unrealistic (very thin) shell thickness for contact.

10. Use default values unless experts recommend a non-default value.
Lastly, always perform 100 cycles of **shakedown** with no loads and boundary conditions to ensure zero energies in all elements. This must be frequently depending on how much the design evolves. If you are swapping parts or making a number of updates over a short duration, then a weekly shakedown is probably a good idea to remove instabilities.

Discussion and Conclusions

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

ANSYS LS-DYNA environment (called ANSYS CLASSIC) does not allow entering all commands available in LS-DYNA.

It is necessary to establish an alternative procedure.

Discussion and Conclusions

8. ANSYS LS-DYNA _X_ LS-DYNA Solver

. Pre-processing

. Solve (indirectly calls the LS-DYNA Solver)

. Pos-processing

. ANSYS Pre-processing

. ANSYS Solve → file ‘.k’ (kill solver)

. File ‘.k’ Editing (LS-Dyna Contact parameters)

. LS-DYNA Solver

. ANSYS Pos-processing

Open Issues

1. Lead Slump and Contraction
2. Impact factor for the quasi-static analyses
3. Degree of modeling detail:
   . Contacts & Shell x 3D elements ?
   . Overall meshing ?
   . Dampers geometric details & Bolted connections ?
   . etc.
4. Impact limiters material properties → dynamic ?
   static ?
   encased ?
5. Impact limiters design → detailed design

Discussion

1.
2.
3.
4.
5.