

COMPUTATIONAL SIMULATION OF BLAST LOADING IN AN OPEN CYLINDER MODEL USING ANSYS AND LS-DYNA

**Ramprashad Prabhakaran
Graduate Student
Department of Mechanical Engineering
University of Nevada, Las Vegas (UNLV)**

**MEG 795 PROJECT REPORT - Fall 2003
(Type 2 Cylinder: Lagrange ConWep)**

December 5, 2003

INTRODUCTION

- Open cylinder models can be used for testing the materials, which are used in nuclear reactors and containers that are used to store and/or test explosive materials.
- Reduction in the integrity of the cylinder at any stage of transportation or storage can be fatal to the people in the vicinity as well as to the environment.
- Care should be taken to ensure that these cylinders could resist blast-loading during accidents.
- FEA techniques succeed in reducing the time and effort spent in building and testing prototypes.

OBJECTIVE

- FEA is used to simulate the blast-loading in cylinders made of basalt plastic and steel 20 and to validate with the results obtained experimentally.
- The displacement of the cylinder is monitored throughout the simulation and compared to the actual displacement of the cylinder.
- To study the behavior of the candidate structural materials, namely basalt plastic and steel 20 that can used in nuclear reactors and containers for storing and/or test explosive materials
- To develop the computational modeling capability to efficiently design cylinder, which can be used for storing and testing explosives
- To benchmark the ability of ANSYS and LS-DYNA to simulate the blast-loading in the cylinder

OBJECTIVE (Cont'd.)

- The goal of this project is focused on the determination of the following:
 1. Peak tensile strain in the circumferential direction: ε_1
 2. The time to reach the peak tensile strain in the circumferential direction: t_1
 3. Radial oscillation fundamental tone period: T
 4. Peak compressive strain in the circumferential direction: ε_2
 5. Peak tensile and compressive strain in the longitudinal direction: ε_3
 6. Peak circumferential strain rate: $d\varepsilon_1/dt_1$

BLAST LOADING TEST

- The blast-loading test in cylinders is performed to investigate the dynamic performance of various candidate structural materials.
- The primary objective is to conduct detailed theoretical and computational studies of the crucial elements of the cylinder.
- The design of the cylinder must ensure complete localization of the detonation products.
- The length of the cylinder is designed to completely contain the blast loading due the explosive that is placed at the center of its' longitudinal axis.
- This test helps in performing a detailed analysis of stress-strain state of the container and also to determine the explosion resistance of the basalt plastic and steel 20.

STEPS IN FINITE ELEMENT MODELING

The blast loading experiment is simulated computationally using ANSYS and LS-DYNA to analyze the large deformation dynamic response of the cylinder.

Any finite element simulation has three main stages:

- Pre-processing
- Solution
- Post-processing

PRE-PROCESSING : Material Properties

- **Basalt plastic** (EDT - 10 Binder, RB9 - 1200 complex basalt thread roving): Thread diameter = 9 mm ; Winding angles: 90° and $\pm 35^\circ$
- **Steel 20**: Density = 7850 kg/m^3 ; Yield Strength = 340 MPa ; $E = 200 \text{ GPa}$ $n = 0.33$
- **Overall composite properties in the X-Y coordinate system:**
- The blast loading test in cylinders is performed to investigate the dynamic performance of various candidate structural materials.

Fibers are interchanged between $q = \pm 35^\circ$ and $q = 90^\circ$

$E_x = 15.44 \text{ GPa}$; $E_y = 48.54 \text{ GPa}$; E_z (thickness direction) = 14 GPa

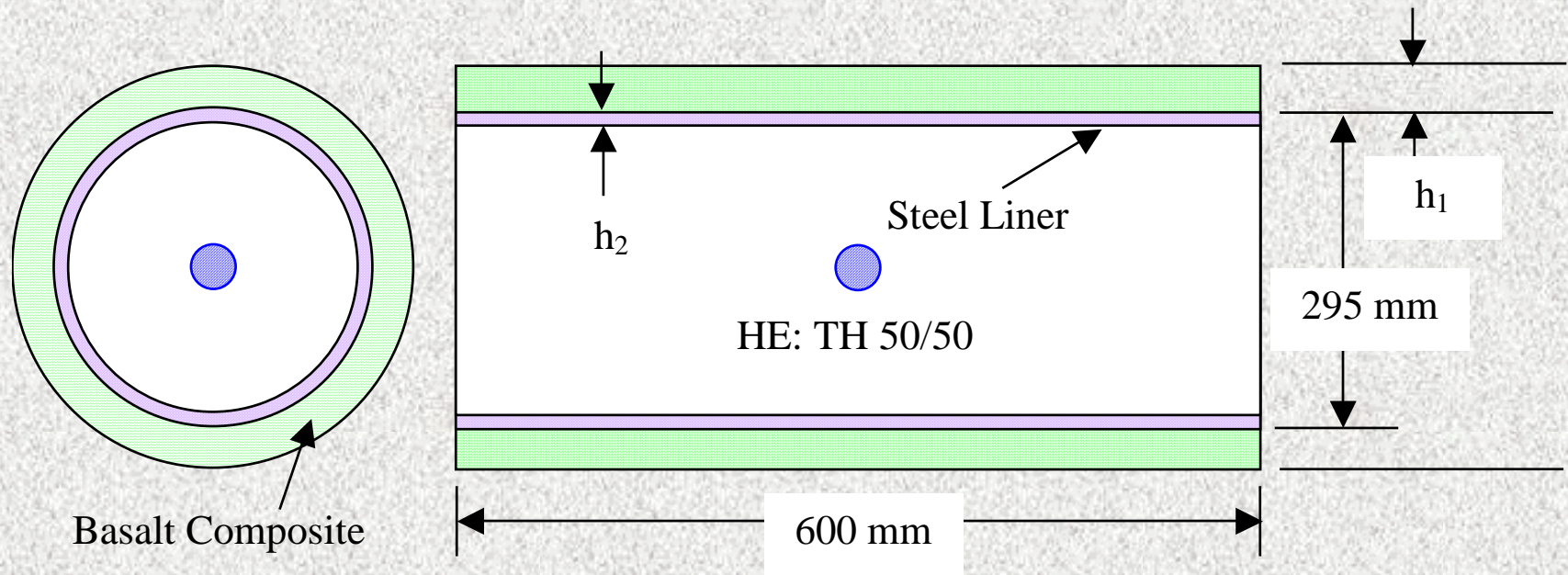
$G_{xy} = 6.645 \text{ GPa}$; $G_{yz} = 3 \text{ GPa}$; $G_{xz} = 3 \text{ GPa}$

$n_{xy} = 0.098$; $n_{xz} = n_{yz} = 0.3$; Density = 2.06 g/cm^3

PRE-PROCESSING : Material Geometry

- ANSYS version 7.1 was used to create the geometry of the modeled cylinder.
- Here the modeled cylinder is made of composite material (basalt plastic) having a length of 600 mm and an outer and inner diameter of 321.14 mm and 295 mm, respectively.
- The model also has a liner made of steel 20 of thickness 2.0 mm.

LINE DIAGRAM OF HOLLOW CYLINDER



Comp. Thick. , h_1 [mm]	13.07
Outer Radius , R_{out} [mm]	160.62
Steel Thick. , h_2 [mm]	2.0
$M_1 + M_2$ [g]	23,700
M_{HE} [g]	407.3
Scale factor	1.13

PRE-PROCESSING : Material Model

Here, two cases were considered.

- **Case I:** Basalt plastic was considered to be solid while steel was considered as shell
- **Case II:** Both basalt plastic and steel was considered as solid
- **Case I & II:**

The element types were defined. In case of basalt plastic, the material model chosen was elastic orthotropic whereas for the steel liner, elastic isotropic material model was chosen.

```
*MAT_ORTHOTROPIC_ELASTIC
      1 0.206E+03 0.154E+11 0.485E+11 0.140E+11 0.098000 0.300000 0.300000
0.664E+10 0.300E+10 0.300E+10          0.0
      0.0      0.0      0.0      0.0      0.0      0.0
      0.0      0.0      0.0      0.0      0.0      0.0
*MAT_ELASTIC
      2 0.785E+04 0.200E+12 0.330000      0.0      0.0      0.0
```

- **Case 1**

The real constant set number for thin shell was defined. Then the cylinder was modeled by using cylinder option in create command in modeling module.

- **Case 2**

Since shell was not considered, real constant set number option was not used here and the cylinder was modeled by using cylinder option in create command in modeling module.

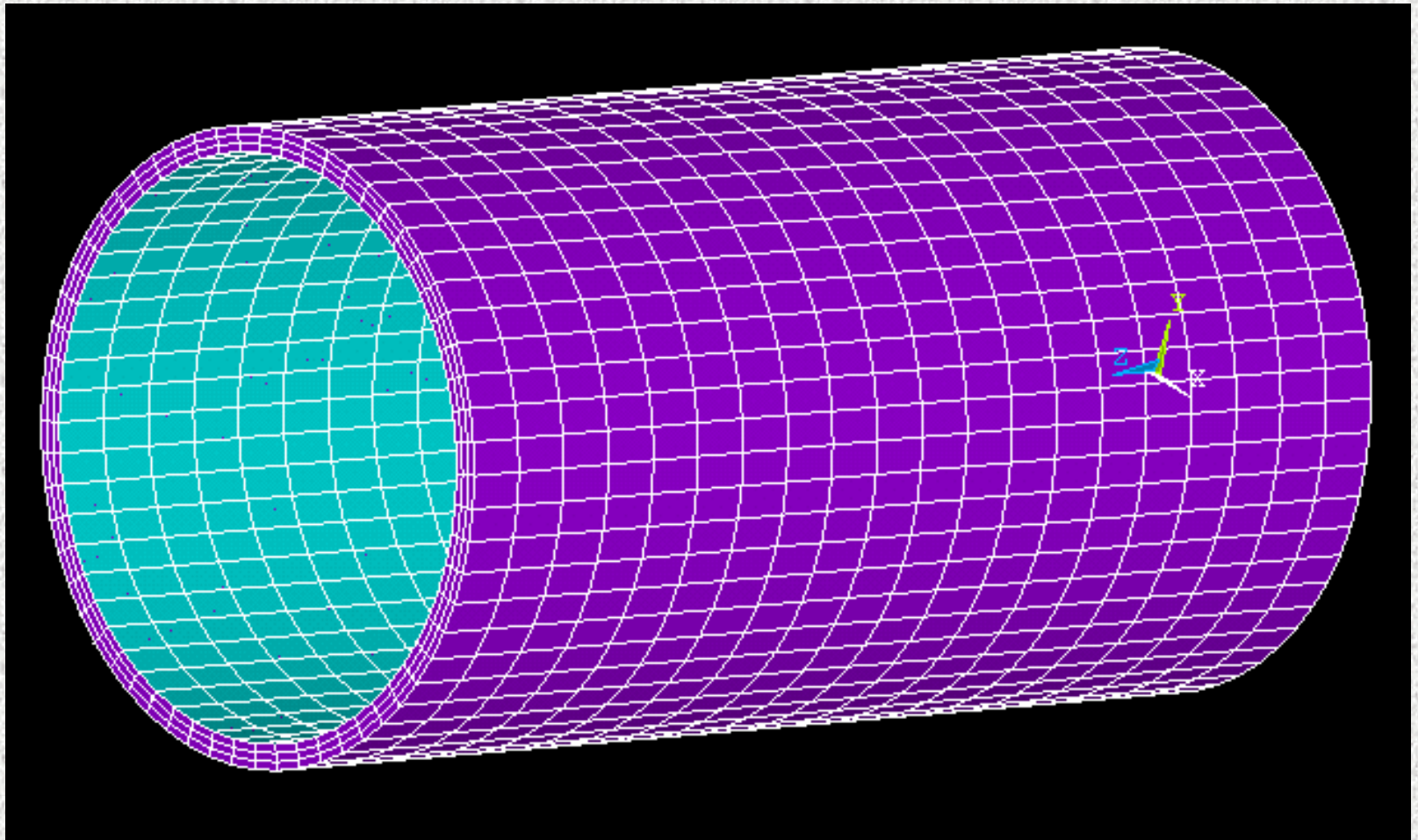
PRE-PROCESSING : Meshing

Case 1:

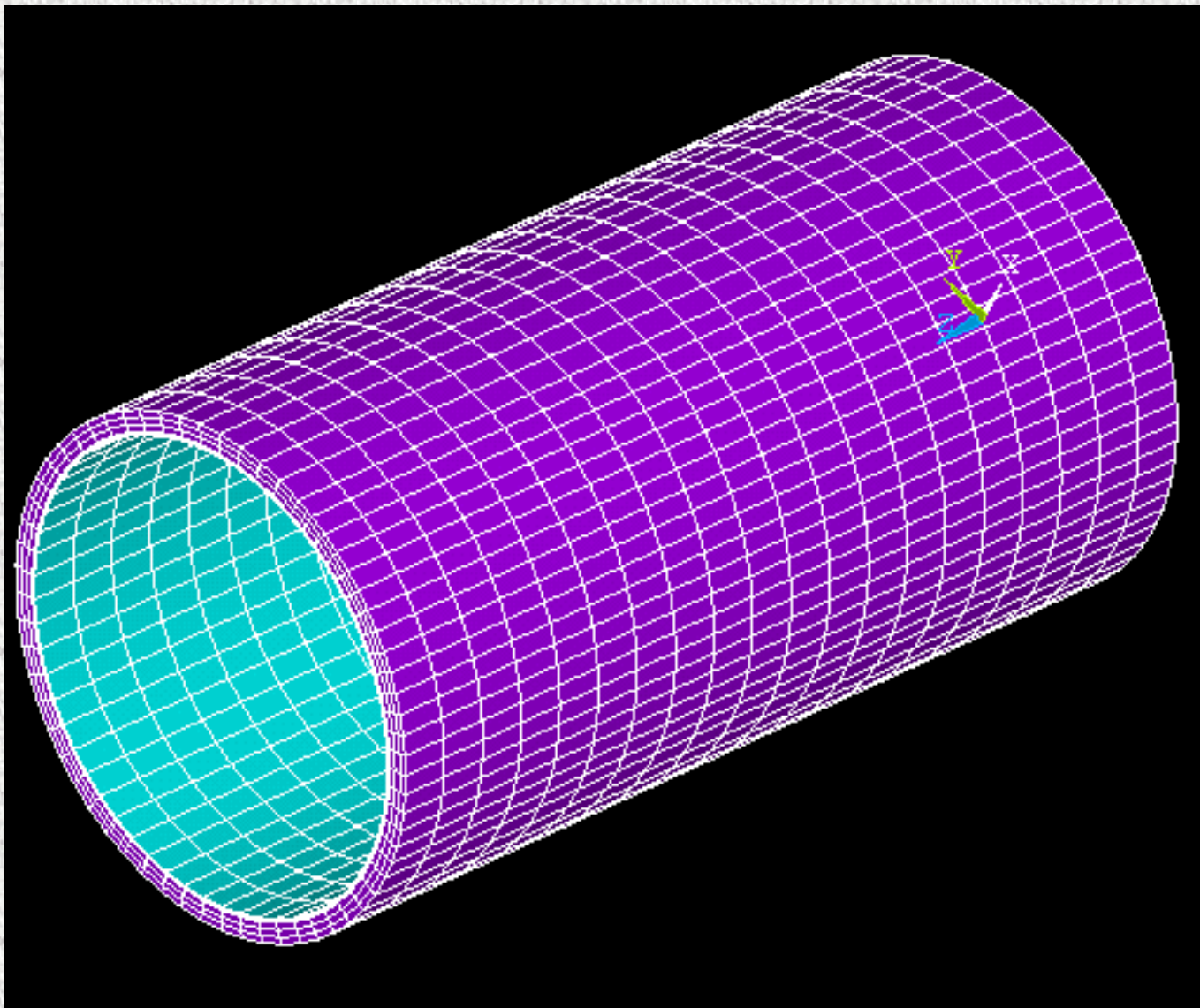
Here the cylinder was meshed in the form of both shell and solid elements. The steel liner was meshed as the shell elements while the basalt plastic was meshed as solid elements. The model had a total of 6000 elements, out of which 1200 are shell elements and 4800 are solid elements.

Case 2:

Here the cylinder was meshed in the form of solid elements only. Both the steel liner and basalt plastic was meshed as solid elements. The model had 6500 solid elements.



Meshed Model of a Cylinder
(Case I: Composite: Solid and Steel: Shell)



Meshed Model of a Cylinder
(Case II: Composite and Steel: Solid)

In order to specify the type and frequency of generating the output files, the following keywords were used:

***DATABASE_BINARY_DEPLOT:** This creates a database file, named, “d3plot” that contains the plotting information of the plot data over the three dimensional geometry of the model. Under this option, the user can specify the frequency of the outputs. In other words, one can choose the frequency with which the plot data can be generated. Careful selection of this frequency, depending on the total time of simulation, reduces simulation time.

***DATABASE_BINARY_D3THDT:** This generates a database file, named, “d3thdt” that contains time history data of element subsets or global information. Under this option too, the frequency of output can be specified.

CONWEP FUNCTION

The following were the parameters that were specified before running the solution:

- **Case I:** To call the ConWep function, the input deck must contain *LOAD_SHELL_SET and *LOAD_BLAST cards.
- The purpose of *LOAD_BLAST card to is define an airblast function for the application of pressure loads due to explosives in conventional weapons. This option determines the pressure values when used in conjunction with the *LOAD_SHELL SET.

```
*LOAD_SHELL_SET
$(Defines which shell to apply *LOAD_BLAST)
$---+---1---+---2---+---3---+---4---+---5---+---6---+---7---+---8
$      SID      LCID      SF      AT
      222      -2       1       0

*LOAD_BLAST
$ WGT_XBO YBO ZBO TBO IUNIT ISURF
$---+---1---+---2---+---3---+---4---+---5---+---6---+---7---+---8
      0.4602      0       0      0.3      0       2       2
```

- In addition, the following cards were also included in the .k file as follows

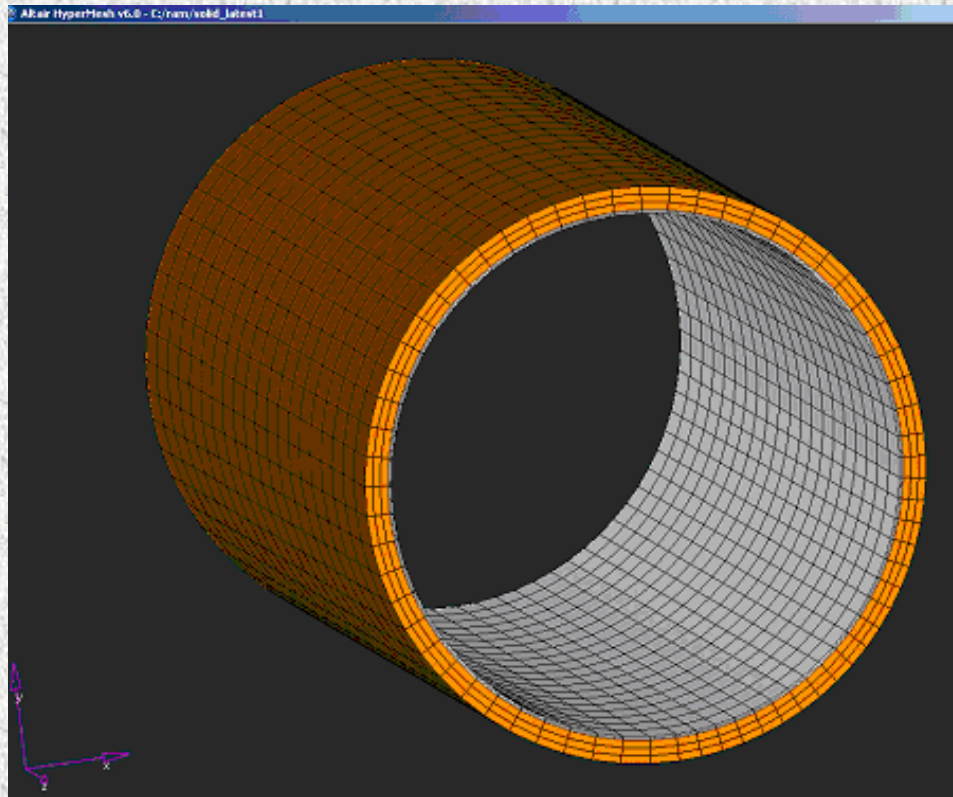
```

*SET_SHELL_LIST_GENERATE
$---+---1---+---2---+---3---+---4---+---5---+---6---+---7---+---8
$      SID      DA1      DA2      DA3      DA4
      222
      1      360
*DEFINE_CURVE
$---+---1---+---2---+---3---+---4---+---5---+---6---+---7---+---8
      4      0      1.000      1.000      0.000      0.000
0.000000000000E+00 0.000000000000E+00
1.000000000000E+00 0.000000000000E+00
$
$
*DEFINE_CURVE
$---+---1---+---2---+---3---+---4---+---5---+---6---+---7---+---8
      25      0      1.000      1.000      0.000      0.000
0.000000000000E+00 0.000000000000E+00
1.000000000000E+00 0.000000000000E+00
$
$

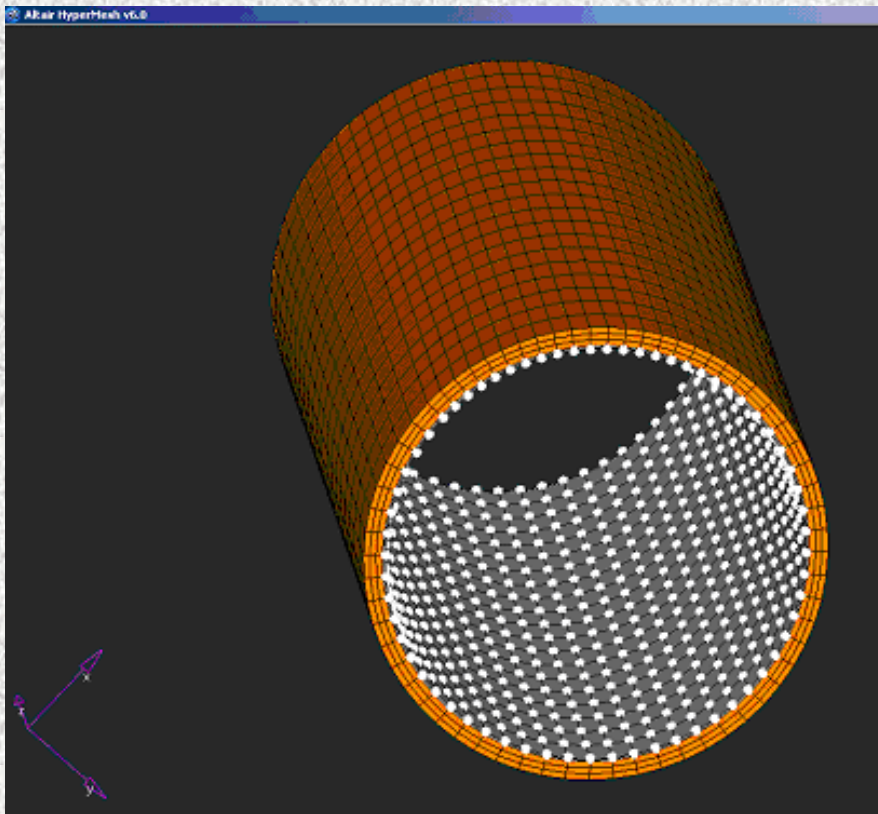
```

- This input file thus generated was fed to LS-DYNA solver.

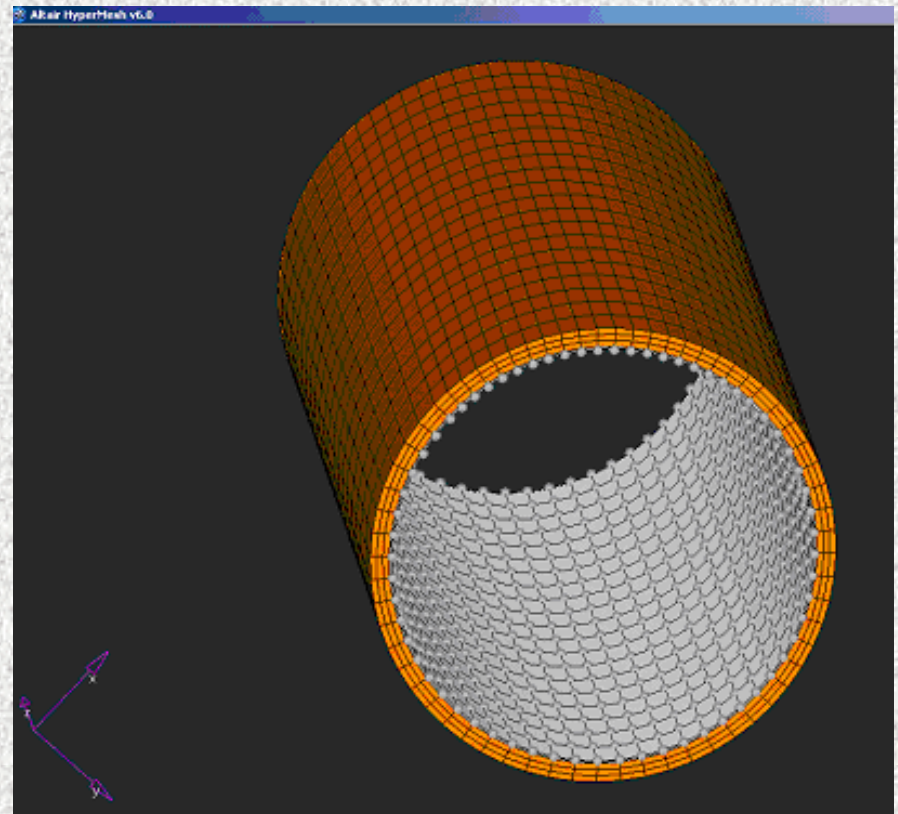
- **Case II:** Since, here both materials are considered as solid, it is necessary to identify nodes and elements on the innermost circumference of the solid. This is done by importing the .k file generated from ANSYS to HyperMesh and by using entity sets option.



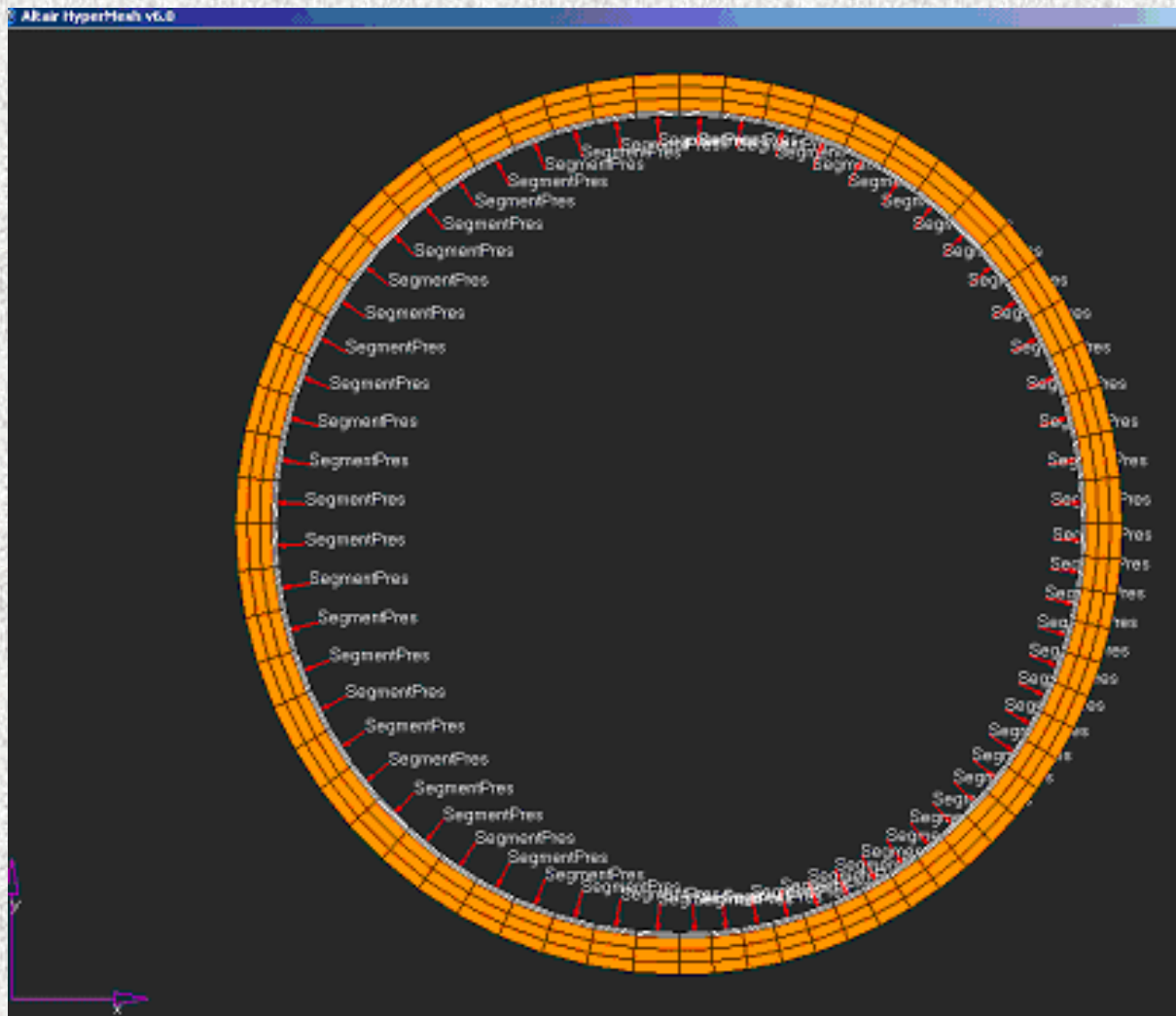
Picked Elements on the Innermost Surface



Picked Nodes on the Innermost Surface



Picked Elements and Nodes on the Innermost Surface



Applying Pressure on the Innermost Surface

- **Case II:** To call the ConWep function, the input deck must contain *LOAD_SEGMENT *DEFINE_CURVE and *LOAD_BLAST cards.
- The purpose of *LOAD_SEGMENT is to apply pressure load over one segment defined by four nodes. Here input for LCID is -2, in order to call ConWep function.

```

*LOAD_SEGMENT
$---+---1---+---2---+---3---+---4---+---5---+---6---+---7---+---8
$      LCID      SF      AT      N1      N2      N3      N4
      -2      -1.0      0      1160      1179      1180      1161
      ....
      -2      -1.0      0      2466      2485      2486      2467
      -2      -1.0      0      2467      2486      2505      2504

*LOAD_BLAST
$ WGT XBO YBO ZBO TBO IUNIT ISURF
$---+---1---+---2---+---3---+---4---+---5---+---6---+---7---+---8
      0.460      0      0      0.3      0      2      2

*DEFINE_CURVE
$---+---1---+---2---+---3---+---4---+---5---+---6---+---7---+---8
      4      0      1.000      1.000      0.000      0.000
0.000000000000E+00 0.000000000000E+00
1.000000000000E+00 0.000000000000E+00
*DEFINE_CURVE
$---+---1---+---2---+---3---+---4---+---5---+---6---+---7---+---8
      25      0      1.000      1.000      0.000      0.000
0.000000000000E+00 0.000000000000E+00
1.000000000000E+00 0.000000000000E+00

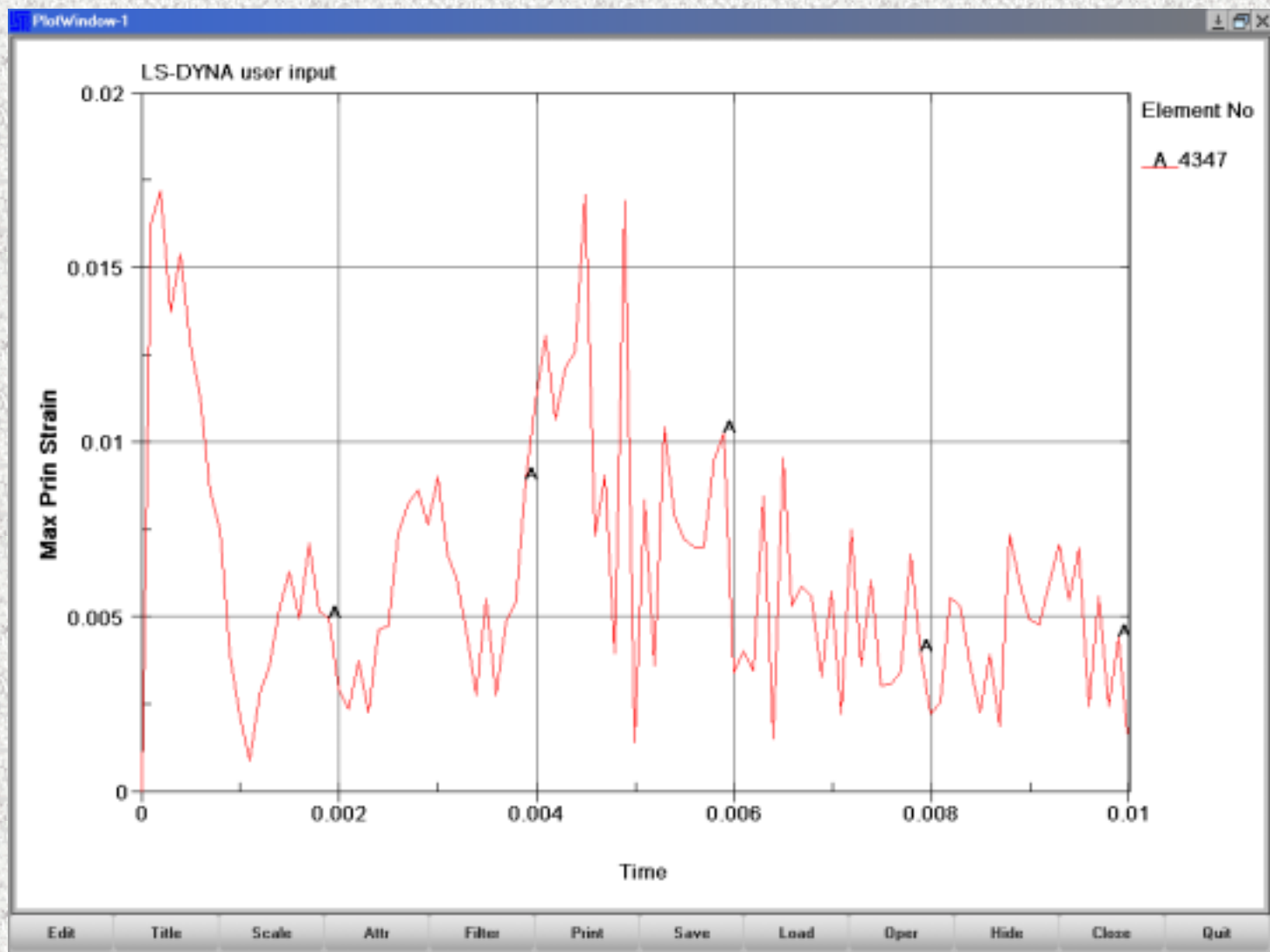
```

SOLUTION

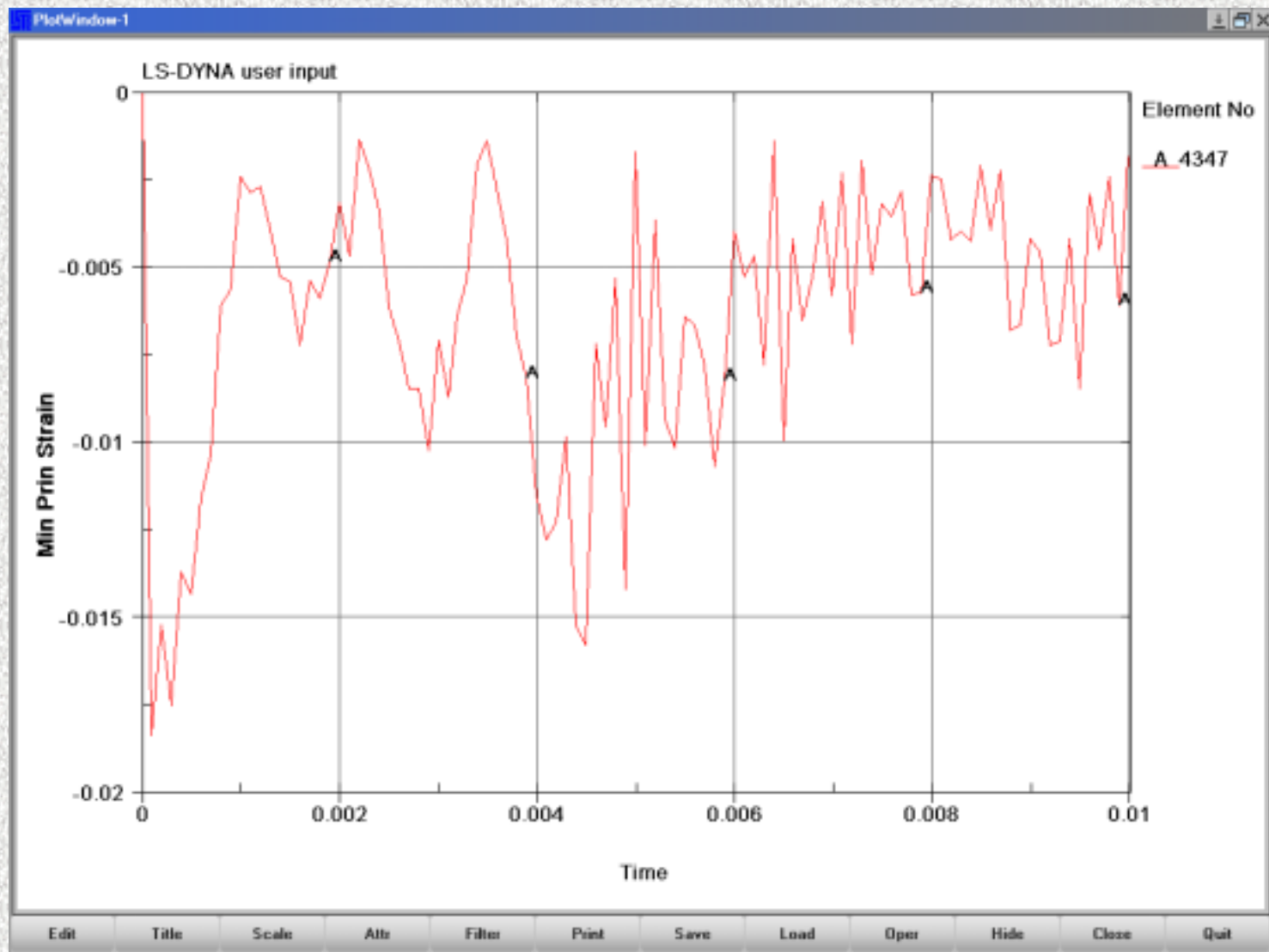
The final version of .k file is then run by using LS-DYNA solver. Once the problem is successfully solved using LS-DYNA solver, the results can be viewed using either Hyper View, LS-POST, etc. Here, LS-POST was chosen.

POST-PROCESSING

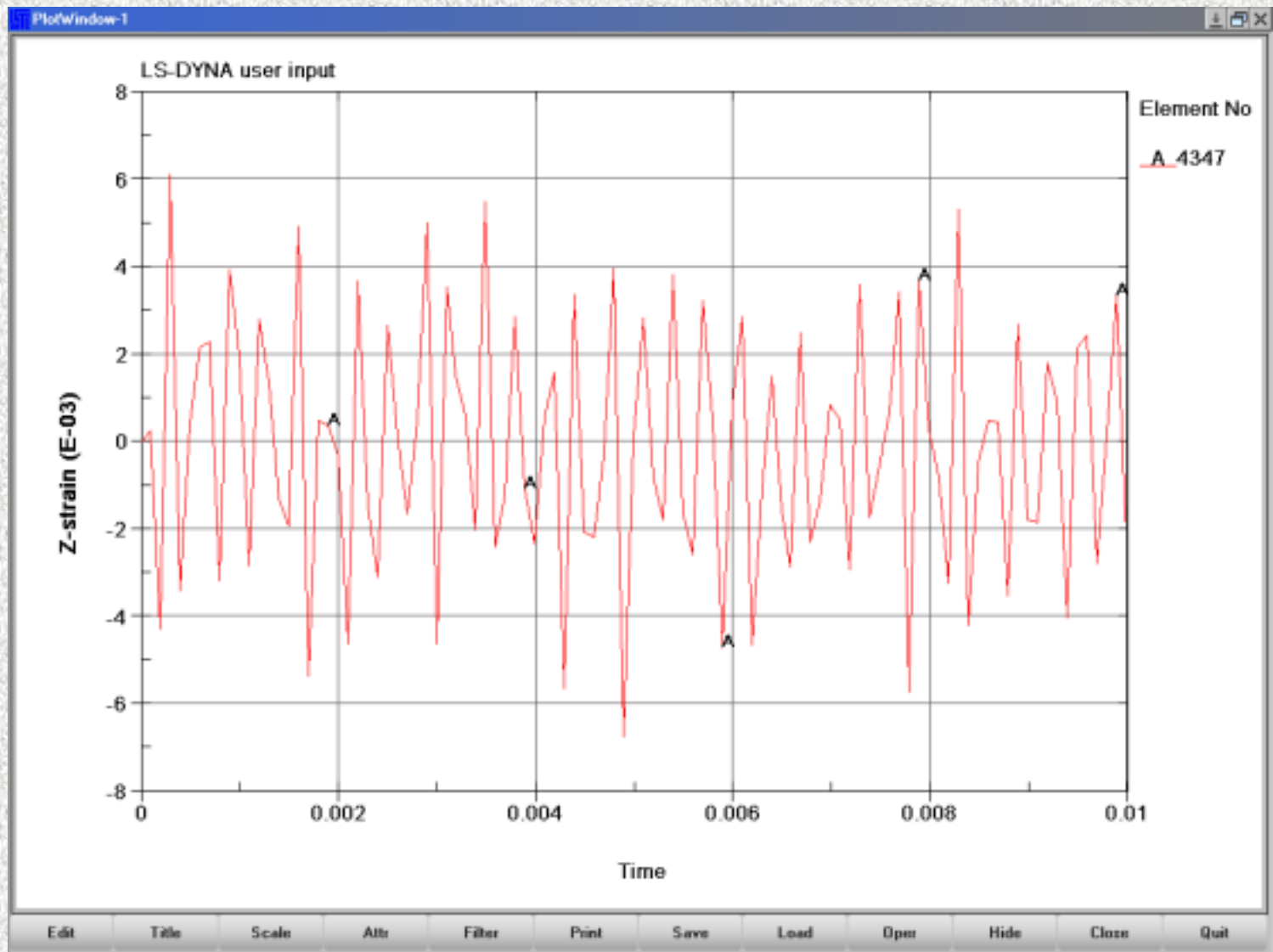
- LS-DYNA is a general-purpose finite element code for analyzing the large deformation dynamic response of structures.
- LS-DYNA currently contains approximately one hundred constitutive models and ten equations-of-state to cover wide range of material behavior.
- Once the problem is successfully solved in LS-DYNA solver, the results can be viewed in many software programs like Hyper View, ANSYS, POST, etc. LS-LS-POST was chosen here.
- LS-DYNA solver outputs a d3plot output file, which is compatible with LS-POST
- Once this file is loaded, LS-POST allows viewing the nodal and elemental results.



Determination of ε_1 , t_1 and T (Case I)



Determination of ε_2 (Case I)

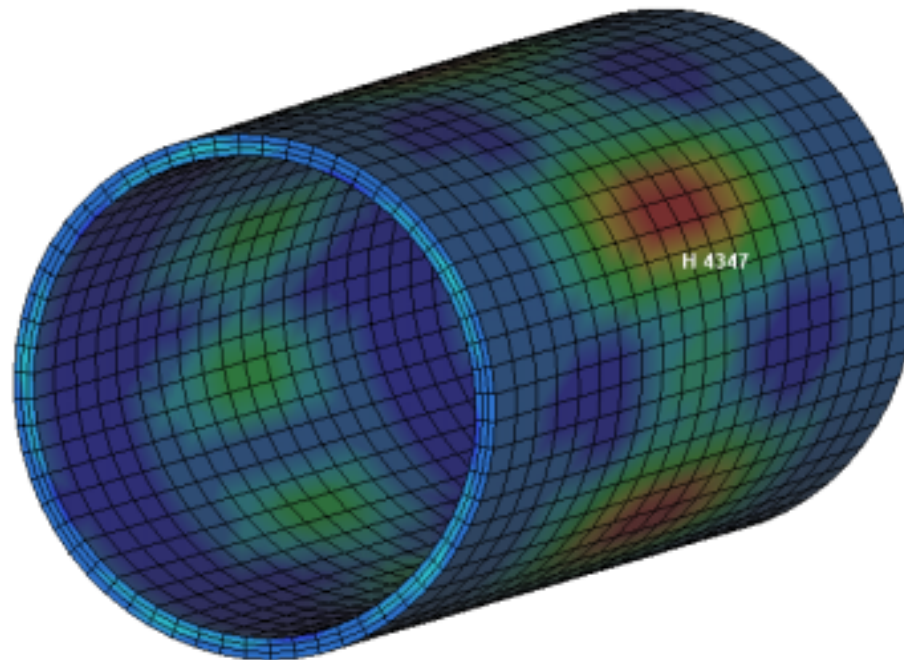


Determination of ε_3 (Case I)

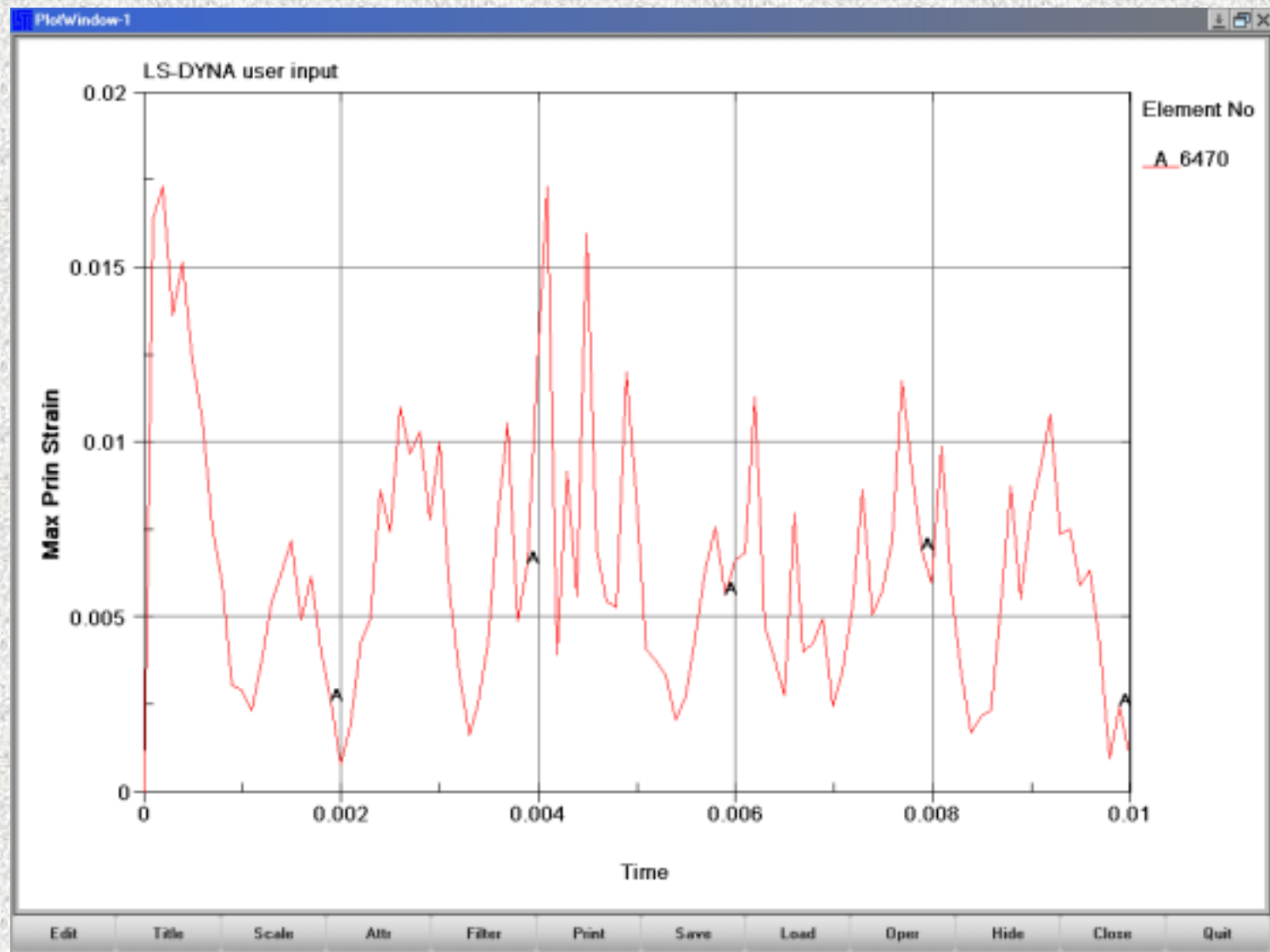
LS-DYNA user input
Time = 0.0047995
Contours of Max principal Strain-Strainrate
min=47.6706, at node# 1729
max=565.596, at node# 42371

Fringe Levels

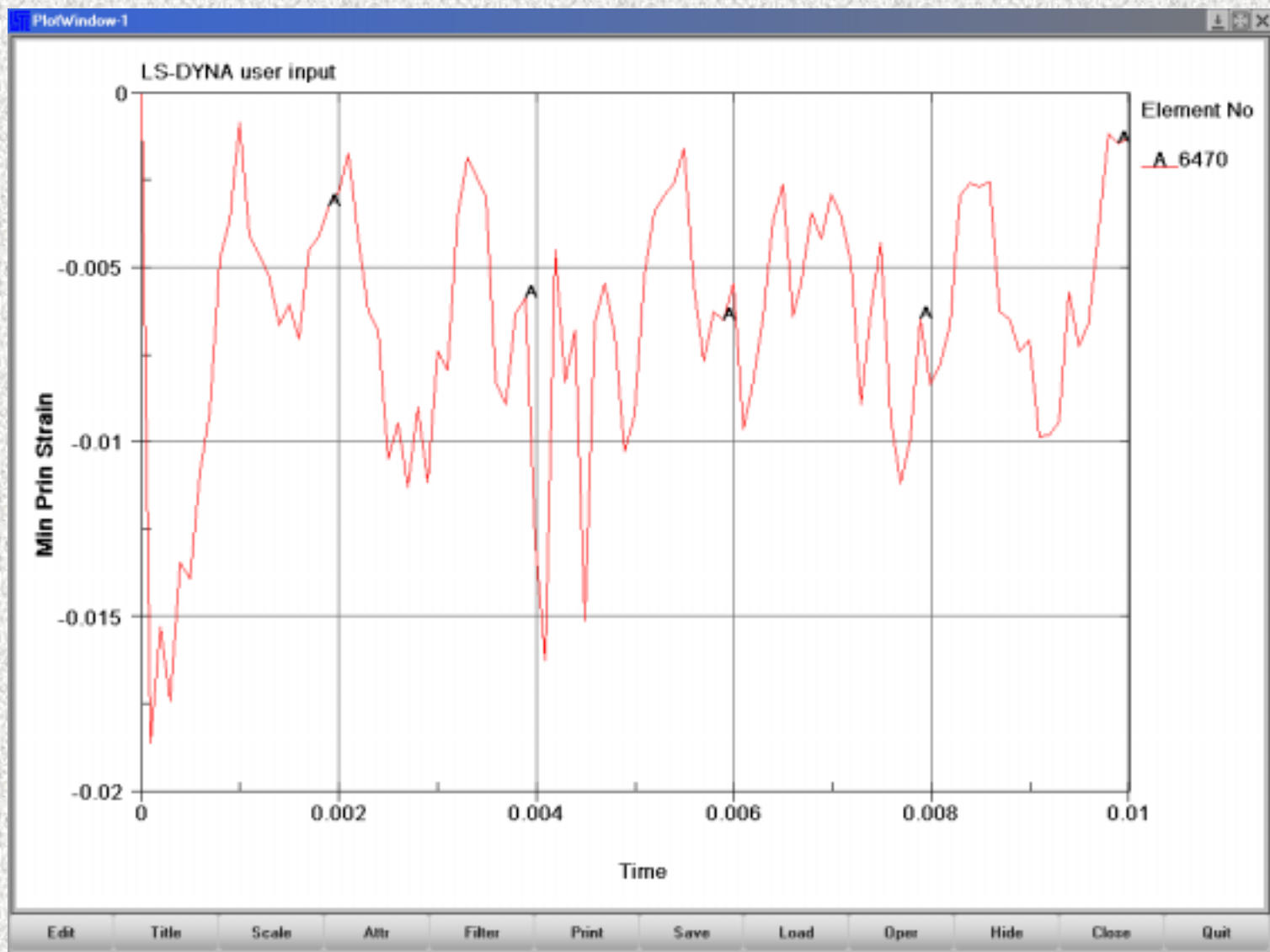
5.656e+002
5.138e+002
4.620e+002
4.102e+002
3.584e+002
3.066e+002
2.548e+002
2.030e+002
1.513e+002
9.946e+001
4.767e+001



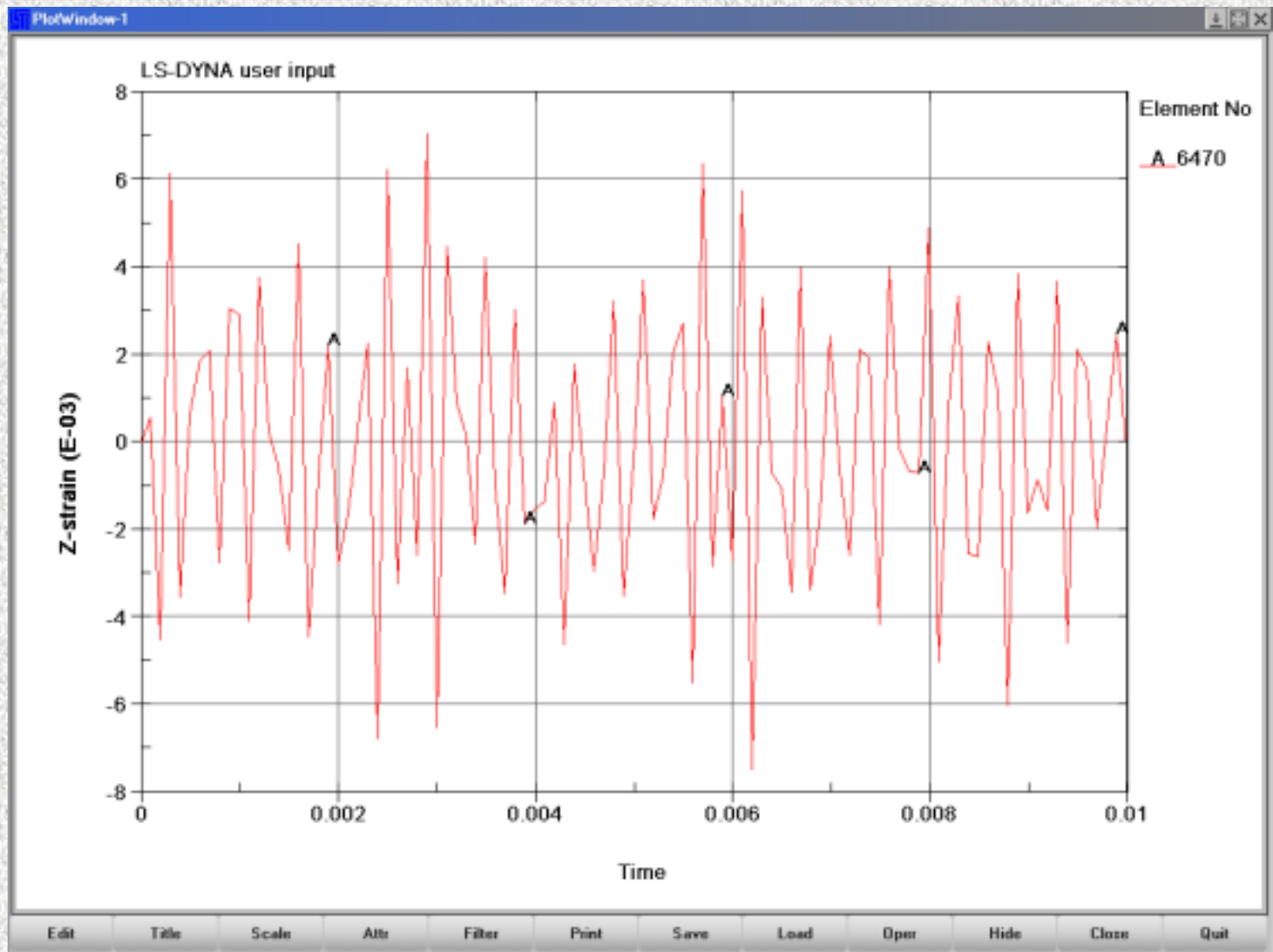
Determination of $d\epsilon_1/dt$ (Case I)



Determination of ϵ_1 , t_1 and T (Case II)



Determination of ϵ_2 (Case II)

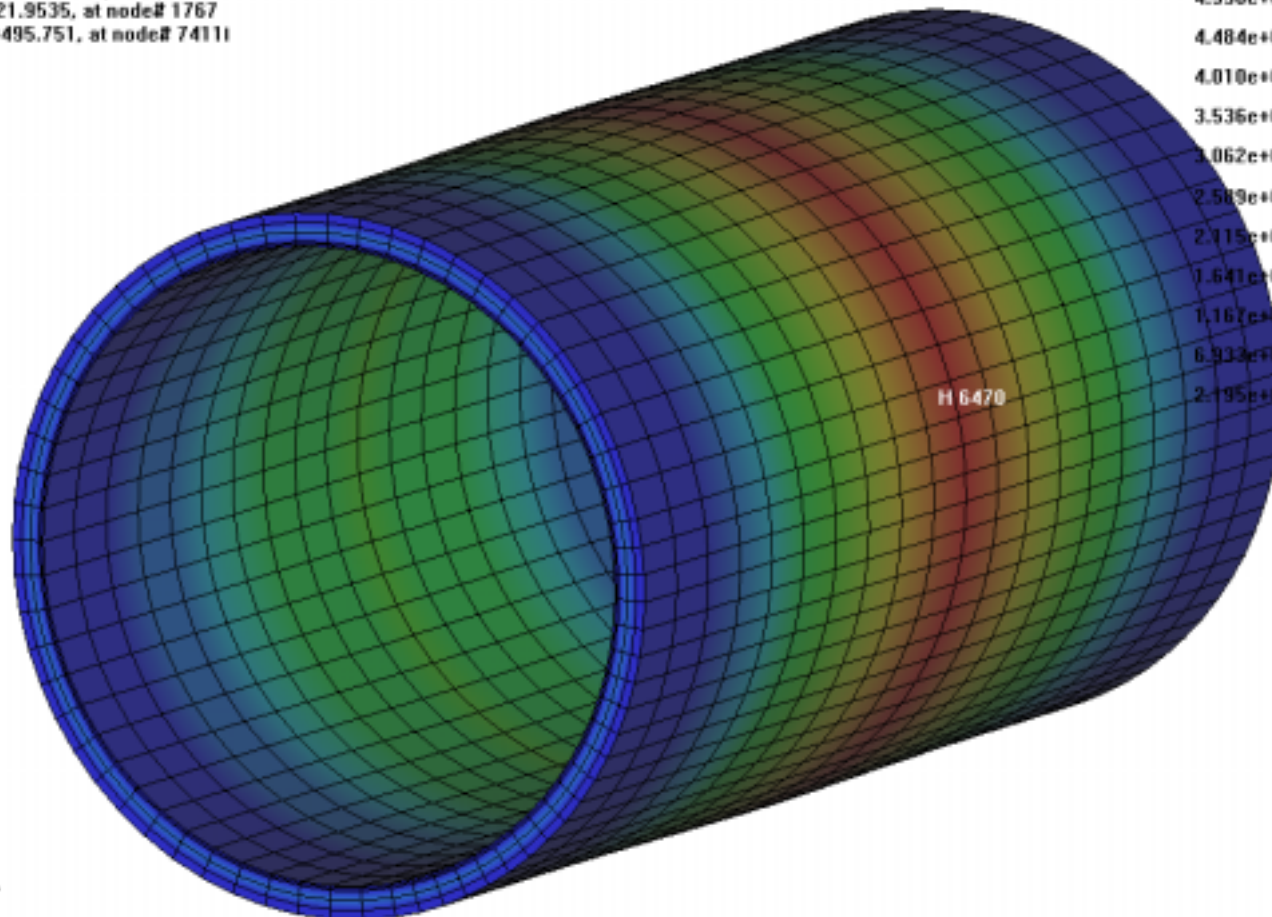


Determination of ϵ_3 (Case II)

LS-DYNA user input
Time = 0.0006999
Contours of Max principal Strain-Strainrate
min=21.9535, at node# 1767
max=495.751, at node# 74111

Fringe Levels

4.958e+002
4.484e+002
4.010e+002
3.536e+002
3.062e+002
2.589e+002
2.115e+002
1.641e+002
1.167e+002
6.933e+001
2.195e+001



Determination of $d\varepsilon_1/d\tau$ (Case II)

COMPARISON OF FEA AND EXPERIMENTAL RESULTS

	Experimental Values	FEA Values (Case I)	% Error (Case I)	FEA Values (Case II)	% Error (Case II)
Comp. Thick. , $h_1^{(1)}$ [mm]	13.07	13.07	-	13.07	-
Outer Radius, $R_{out}^{(1)}$ [mm]	160.62	160.62	-	160.62	-
Steel Thick. , $h_2^{(1)}$ [mm]	2.0	2.0	-	2.0	-
(M1 + M2) $^{(1)}$ [g]	23,700	23,700	-	23,700	-
$M_{HE}^{(1)}$ [g]	407.3	407.3	-	407.3	-
Scale factor $^{(1)}$	1.13	1.13	-	1.13	-
Peak Tensile Hoop Strain, ϵ_1 [%]	3.2	1.7246	46.10	1.737	45.72
Time to reach ϵ_1, τ_1 [μs]	97	199	105.15	199	105.15
Peak Comp. Hoop Strain, ϵ_2 [%]	-0.65	-1.83	181.53	-1.86	186.15
Peak Long. Strain, ϵ_3 [%]	No Data	0.61 (-0.67)	-	0.71 (-0.75)	-
Period, T [μs]	No Data	200	-	199.33	-
Max. Hoop Strain Rate, $d\epsilon_1/d\tau$ [1/s]	524	565.596	7.93	495.751	5.39
Comments	Lot 2	Lot 2	-	Lot 2	-

Lot 2: Interchanging 13 double ring layers and 13 double spiral layers. Binder average mass fraction = 20.0 %.

CONCLUSIONS AND RECOMMENDATIONS

- Open cylinder models can be used for testing the materials, which are used in nuclear reactors and containers that are used to store and/or test explosive materials.
- It has to maintain its integrity in case of accidents, else it can be fatal to the people in the vicinity as well as to the environment and hence care should be taken to ensure that these cylinders could resist blast loading.
- To validate the results obtained experimentally, finite element analysis using ANSYS and LS-DYNA was performed.
- The values of ε_1 , ε_2 , t_1 , ε_3 , T and $d\varepsilon_1/dt$ obtained from LS-POST were compared with the experimental results. The difference between the experimental and FEA results was found to be good in most cases.
- However, in some cases the percentage of error was quite large and this can be attributed to the discrepancies in ConWep function and/or material properties of the composite cylinder. In addition to this, a much more finer mesh would have given better results.

REFERENCES

1. Brendan J. O'Toole, Notes obtained through email.
2. LS-DYNA Keywords User's Manual, Version 960, Livermore Software Technology Corporation, March 2001.
3. http://www.me.unlv.edu/~mbt/Impact_Characteristics/Interfacing%20ANSYS%20and%20LS-DYNA%20.htm
4. Pepper, D.W. and J.C. Heinrich (1992): The Finite Element Method: Basic Concepts and Applications, Hemisphere Publishing Co.
5. Reddy, J. N, An Introduction to the Finite Element Method, McGraw-Hill Book Co., New York, 1984, 1993
6. <http://www.cams.unlv.edu/>
7. www.matweb.com