International Journal of Scientific Engineering and Research (IJSER)
ISSN (Online): 2347-3878
Index Copernicus Value (2015): 62.86 | Impact Factor (2015): 3.791

Shape Optimization of Thin-Walled Tubes Under_Axial_Velocity

M Vinay Babu[1], K Veeraswamy[2]

Dept Mechanical Engineering MLRIT, Dundigal, Hyderabad, Andhra Pradesh, India

Abstract: ***In this study, the objective is to maximize the crash worthiness of thin-walled tubes under axial impact loads by shape optimization. As design variables, parameters defining the cross-sectional profile of the tube as well as parameters defining the longitudinal profile like the depths and lengths of the circumferential ribs and the taper angle are used. The methodology is applied to the design optimization of a Crush Cans supporting the bumper beam of a vehicle for the loading conditions in standard crash regulations. The crash event is simulated using explicit finite element method. Three Models are considered for the thin walled tubes which are modeled with feasible element quality criteria, and the Barrier is given rigid property. All the necessary loads and boundary conditions were applied as per the real time considerations and explicit analysis is done. The results indicate significant improvement in the crashworthiness over the benchmarks.***

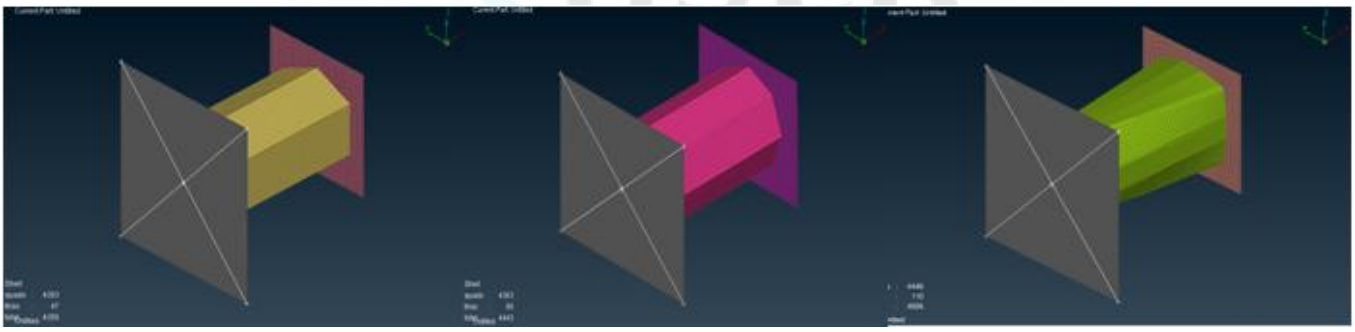
Keywords: Thin walled Tubes, Crush Cans, Crash box, Shape Optimization

1. Modeling of the Thin walled Tubes

Two standard models are considered with uniform cross-section of Hexagonal and Octagonal Cross-sections. And the third model is the unique design which is optimized for better energy absorption and results. The Three Models are compared with same load case and boundary condition for three different thicknesses which is 2.5mm, 3mm and 3.5mm. The Total energy absorption capacity and all other energies absorbed are compared between the three models for three different thicknesses.

The geometric modeling of the Crushcans is done in CATIA V5 and the mesh, boundary conditions, material properties and section properties are defined in the preprocessor. The generated models are meshed using ANSA software tool, version 13.3. The whole structure is modeled using the Mixed type, which would include Quad and Tria elements. Only reduced integration type of elements are used in order to get effective results in the crash simulation, ignoring unnecessary stiffness and right integration. The connection between the crush can and the bumper beam or the crush can and the front member plates is a seam weld. So the FE is modeled by the node to node paste. The rear end plate is constrained in all the directions both in translational and rotational. The Elements to constrain are selected and created and ELSET. And Constraint is created for that ELSET. Composite S3RS/S4RS element formation is used, because it gives better results with bending stress. And the Rigid Barrier is modeled by creating using element of type *R3D4 with for nodes. The rigid properties are assigned to the rigid element. The Shell elements are also assigned with respective properties *SHELL SECTION, in which thickness and NIP [number of Integration points] are given. The NIP value is given 5, which is feasible and best in general case.

The following are the FE models of the three designs, as visible in ANSA tool with element count seen in the left bottom of the screen.



2. Boundary Conditions

The Rear end plate in the model is constrained by creating an element set ELSET for the elements representing the rear plate and creating the boundary condition. Creating a constraint for that element set, so arrest the displacement in all the six degrees of freedom.

The initial and boundary conditions defined in the finite element model should reflect the conditions of the crash tests. Otherwise, the response of the crash-box cannot be correctly predicted. In the present finite element model, The Rigid Barrier is given the Velocity, and the rear end of the crush-can is constrained.

The Rigid element is applied with load on its independent node, which present in its C.O.G. The rigid element is given a boundary condition i.e., *initial velocity, which defines that the rigid element is initially, before starting the simulation run, is having an initial velocity. The velocity given is 16.64kmph or 4633mm/sec.

3. Analysis Method

In the FE simulations, the steel column was fixed at one end and crushed by a rigid wall at the other end. For this, the rear end of the Thin Walled tubes are constrained in all DOF's and the Rigid Barrier is given Velocity. When the simulation run begins, the rigid barrier which is having initial velocity, advances towards the Crush cans and the contacts occurred between the rigid wall and the column. Since the rear end of the part is constrained, it starts to be crushed by the rigid barrier. During the crushing process, it absorbs the kinetic energy, which is obtained from the reaction forces.

4. Results and Discussion

Here, all the three Models are analyzed with the same loading and boundary conditions with three varying thicknesses, to check the consistency. The results for all the three models with three different thickness conditions, which in total nine simulations are run and results are obtained. The results are plotted between the three models for each thickness.

4.1 Total Energy Absorption

Table 5.1: Total Kinetic Energy Absorbed (in KN)

Thickness	*2.5 mm*	*3.0 mm*	*3.5 mm*
Model-1	1.398E+04	1.677E+04	1.394E+04
Model-2	1.152E+04	1.705E+04	1.842E+04
Model-3	1.282E+04	1.776E+04	1.862E+04

From the above table, we can see that the total kinetic energy absorption of the optimized shape is considerably more than the other two subjects even in all varying thicknesses. The optimized design is able to absorb more energy than the standard designs, which is also proved with varying thicknesses.

5.2 External Work done

The External work done is the work done by the externally applied loads. The work done in the model, at each time frame is recorded in the output results file and is plotted with time in seconds on X-axis and Energy in Newton on Y-Axis. The following are the graphs plotted for work done curves of three designs for three thicknesses.

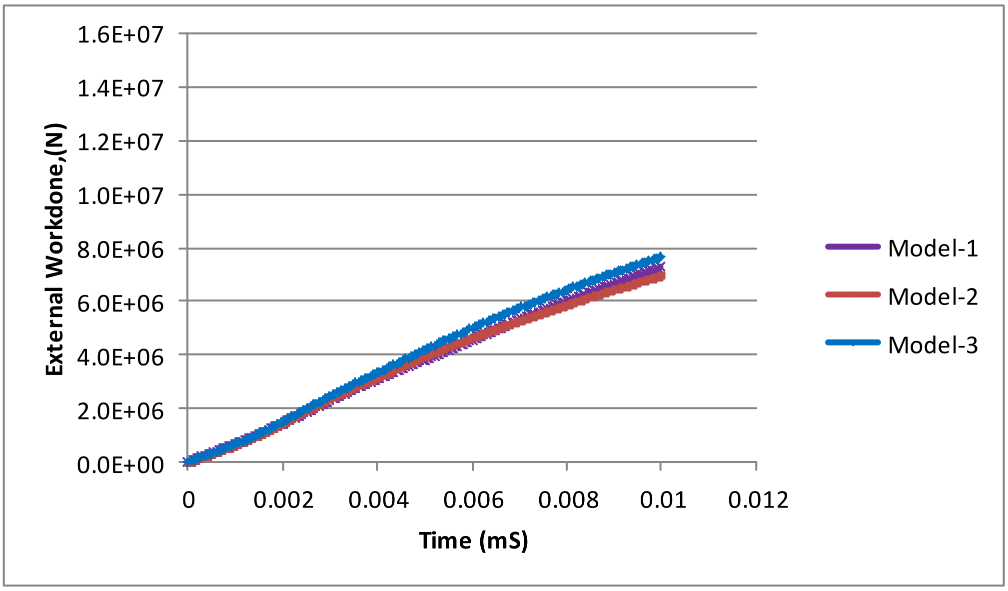


Figure 5.2: Work done comparison for thickness of 2.5 mm

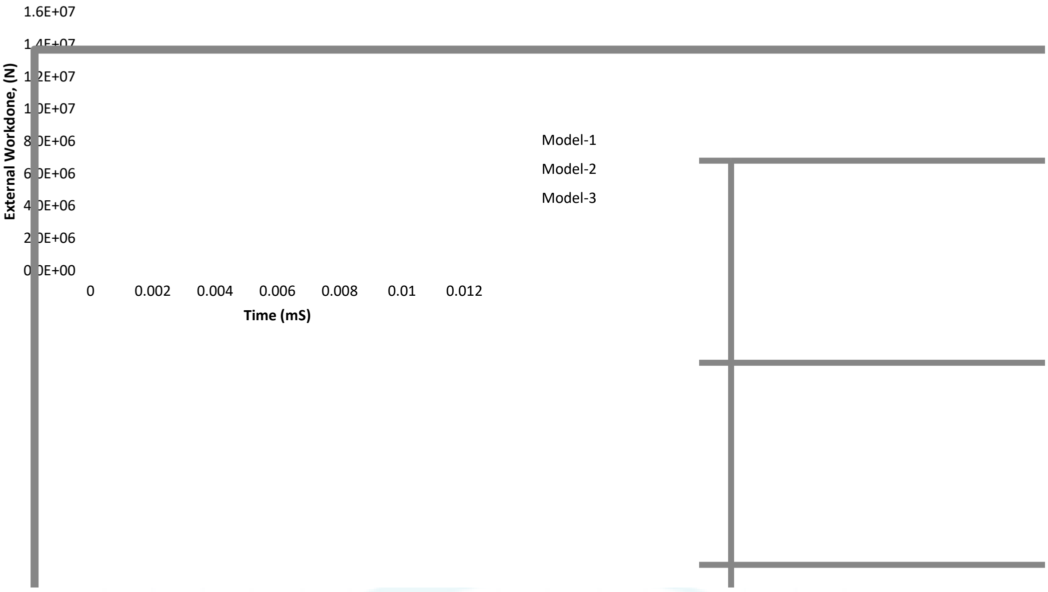


Figure 5.3: Work done comparison for thickness of 3.0 mm

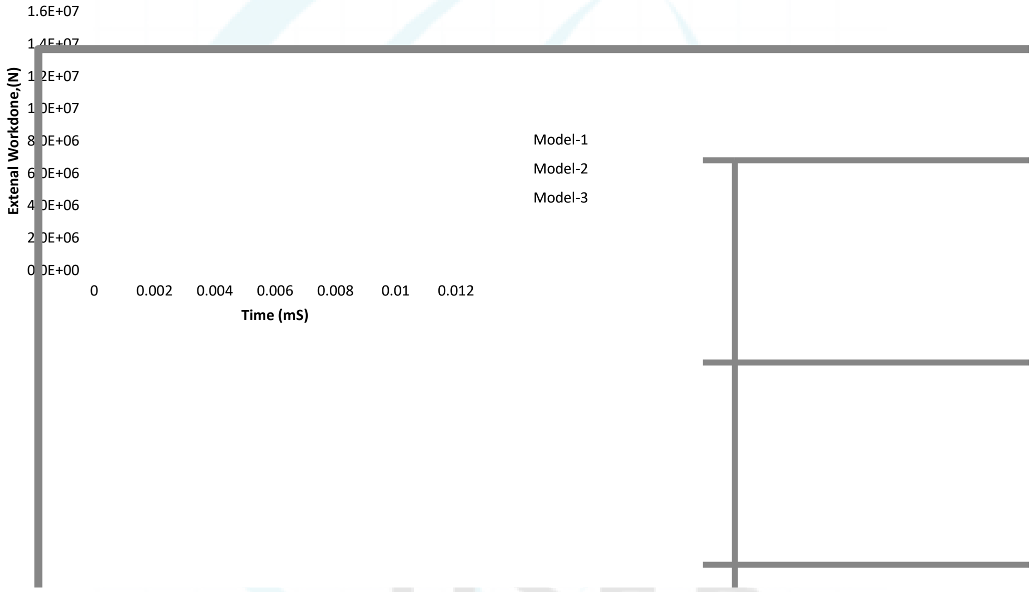


Figure 5.4: Work done comparison for thickness of 3.5 mm

5.3 Internal Energy

The Internal energy is the sum of the recoverable elastic strain energy, the energy dissipated through inelastic processes such as plasticity, the energy dissipated through viscoelasticity or creep, the artificial strain energy, the energy dissipated through damage, the energy dissipated through distortion control, and the fluid cavity energy, Internal energy is plotted for the three designs in the same manner as above. And the three figures below are for the three thickness cases, explaining that the Internal energy of the optimized design is better than the rest two. We can see that the Optimized design is containing more internal energy than the rest two.

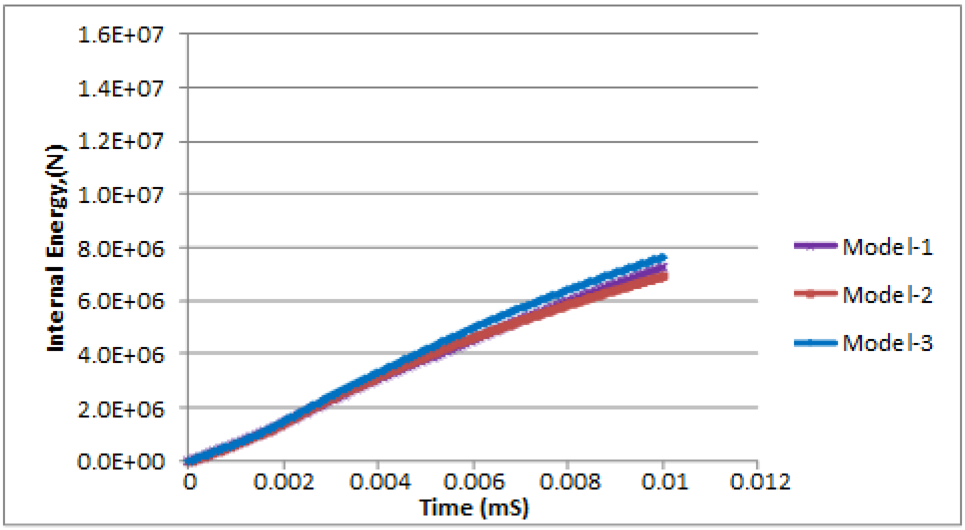


Figure 5.5: Internal Energy comparison for thickness of 2.5 mm

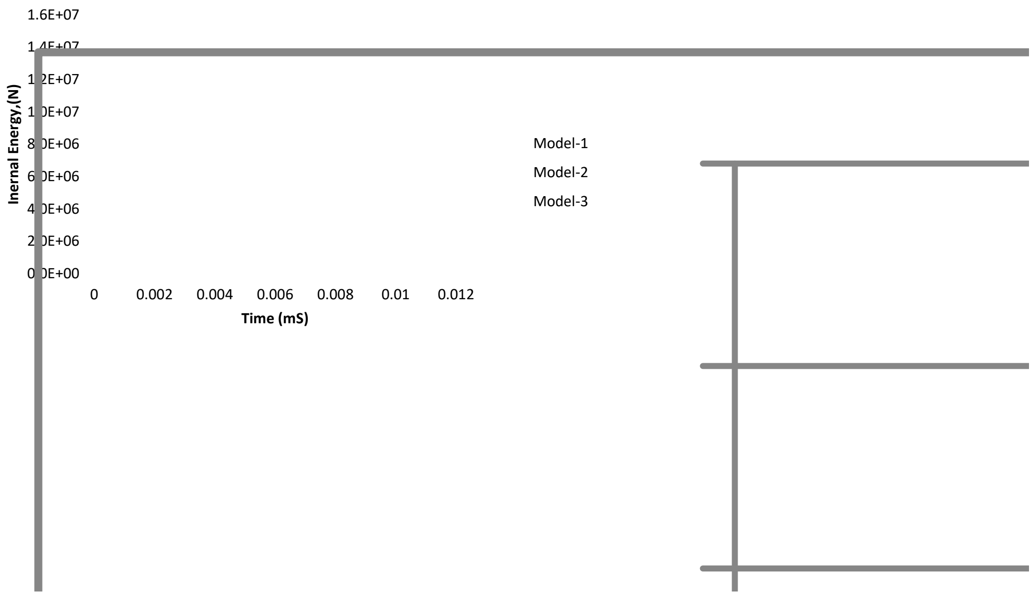


Figure 5.6: Internal Energy comparison for thickness of 3.0 mm

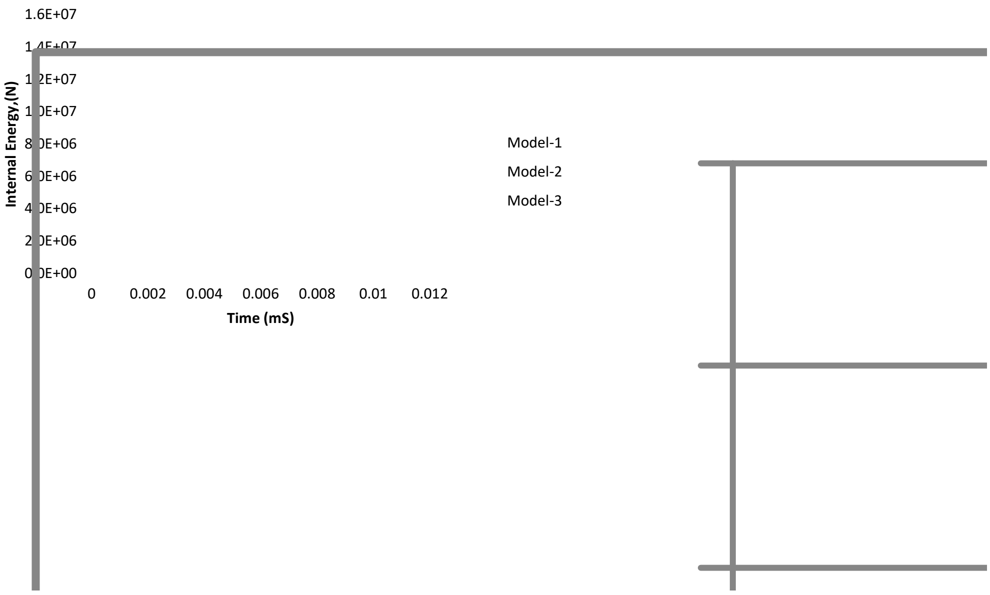


Figure 5.7: Internal Energy comparison for thickness of 3.5 mm

5.4 Force vs Displacement Curve

The Force-Displacement curve explains the force absorbed by the part through its displacement, and also the stiffness of the part is understood based on the curve transformation and the kinks on the curves. The force vs Displacement curve is not directly available in the Abaqus results. Force vs Time and Displacement vs time are obtained in the results and are cross plotted to get Force vs Displacement curve.

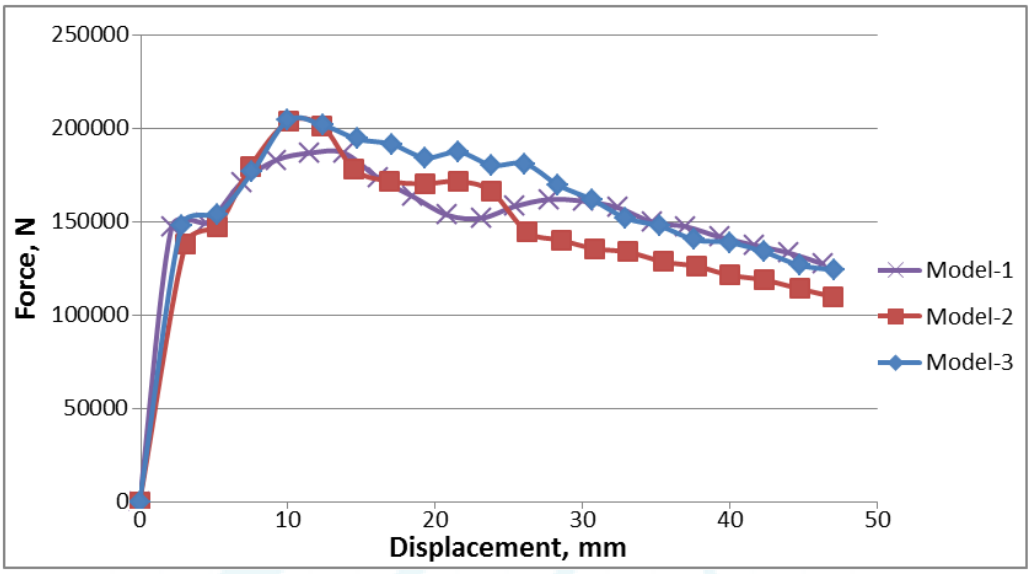


Figure 5.8: Force Vs Displacement for thickness of 2.5 mm

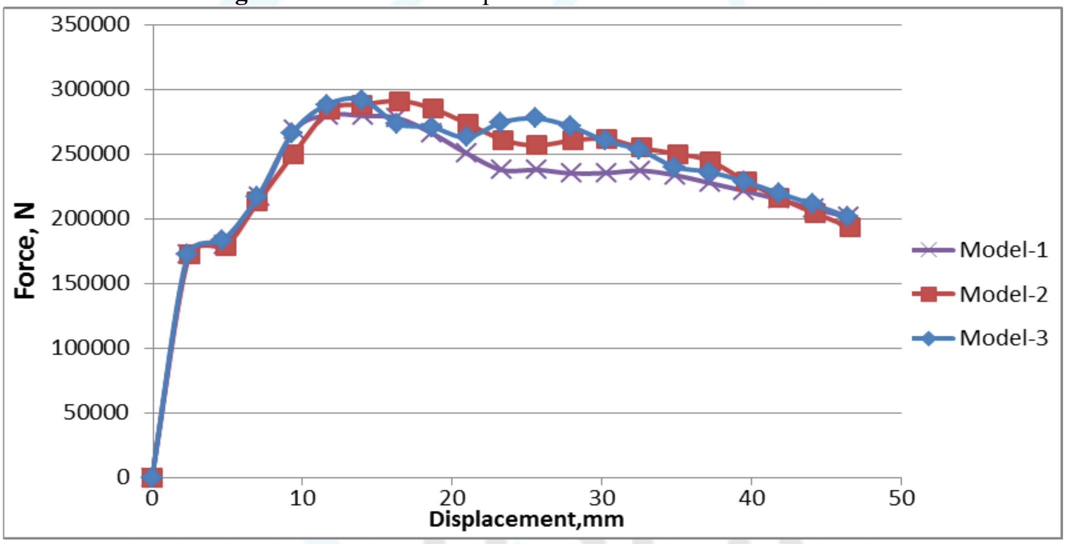


Figure 5.9: Force Vs Displacement for thickness of 3.0 mm

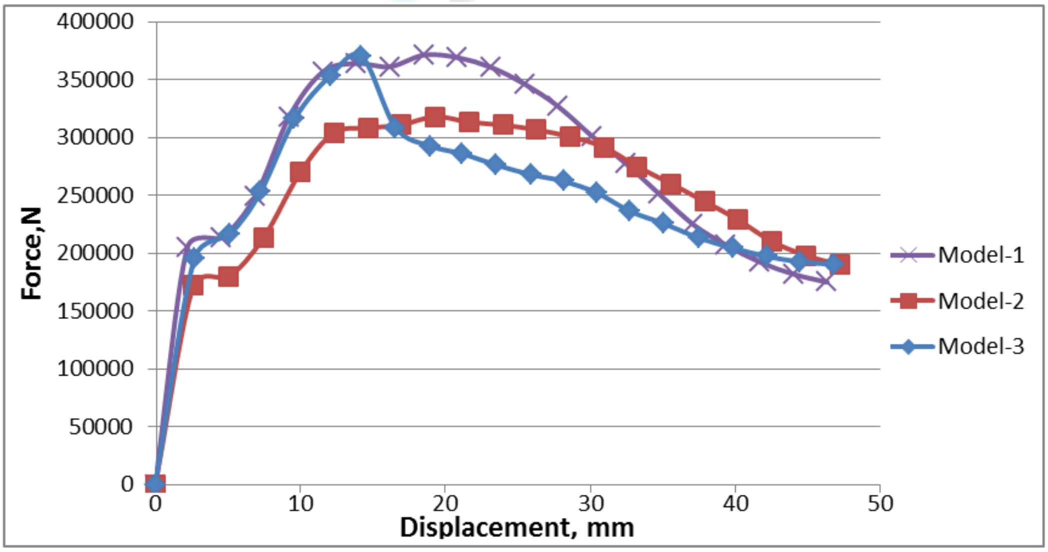


Figure 5.10: Force Vs Displacement for thickness of 3.5 mm

5.5 Deformed Shapes

The Deformed Shapes are recorded by saving the snapshots in the Abaqus Viewer, which is a post processing tool. The result files are loaded and the Deformed shape visualization gives the view of Initial and final Position in a single frame. The deformed shape shows the final displacement of all the FE nodes in the model.

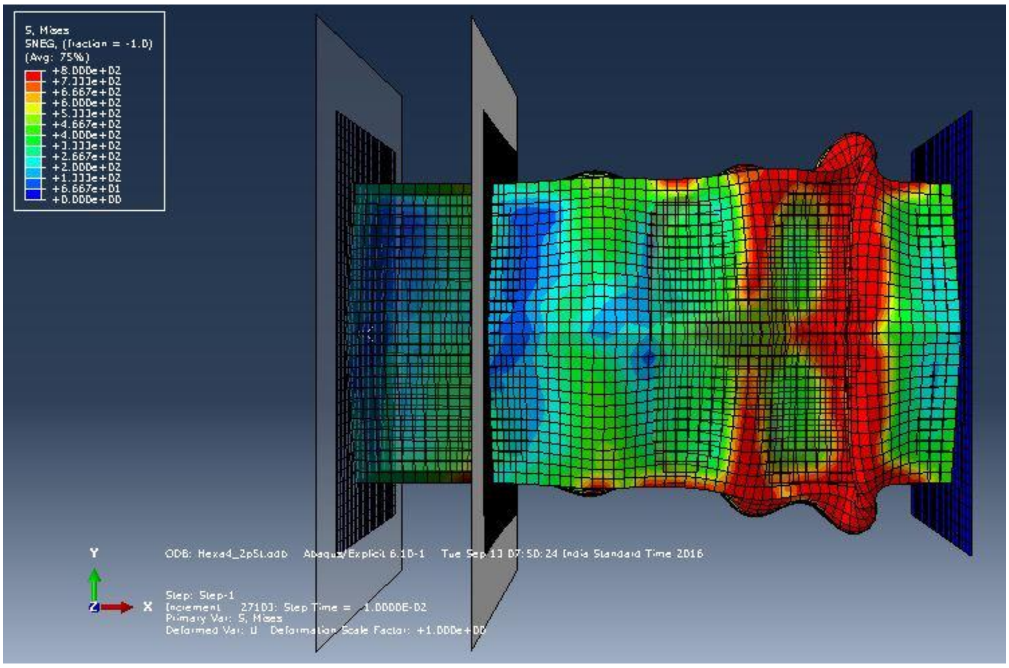


Figure 5.11: Deformation at final step for Model 1

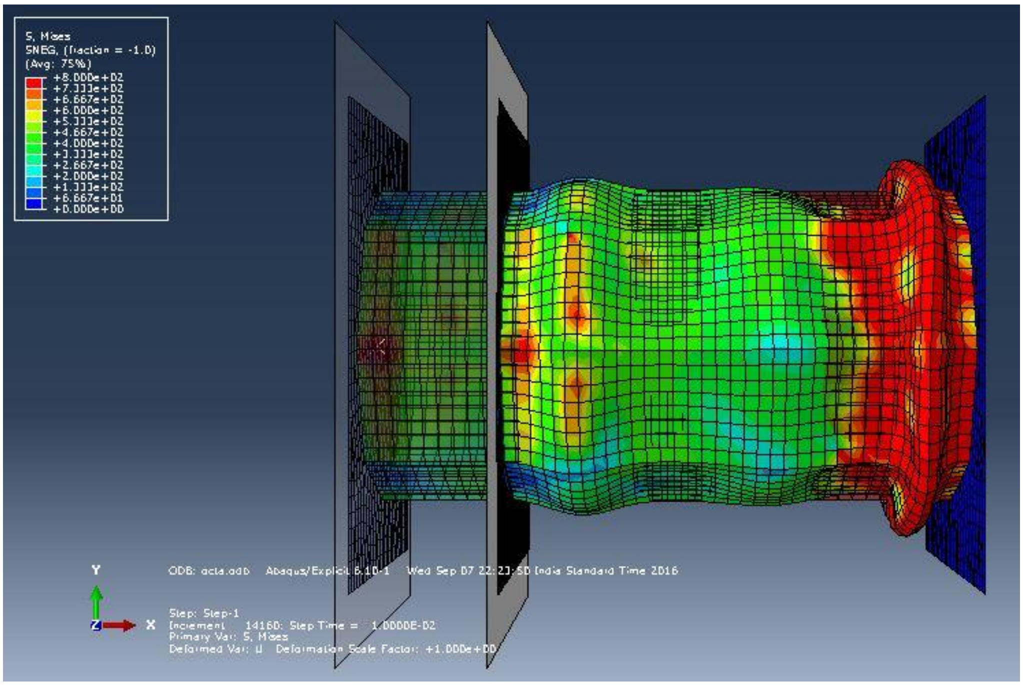


Figure 5.12: Deformation at final step for Model 2

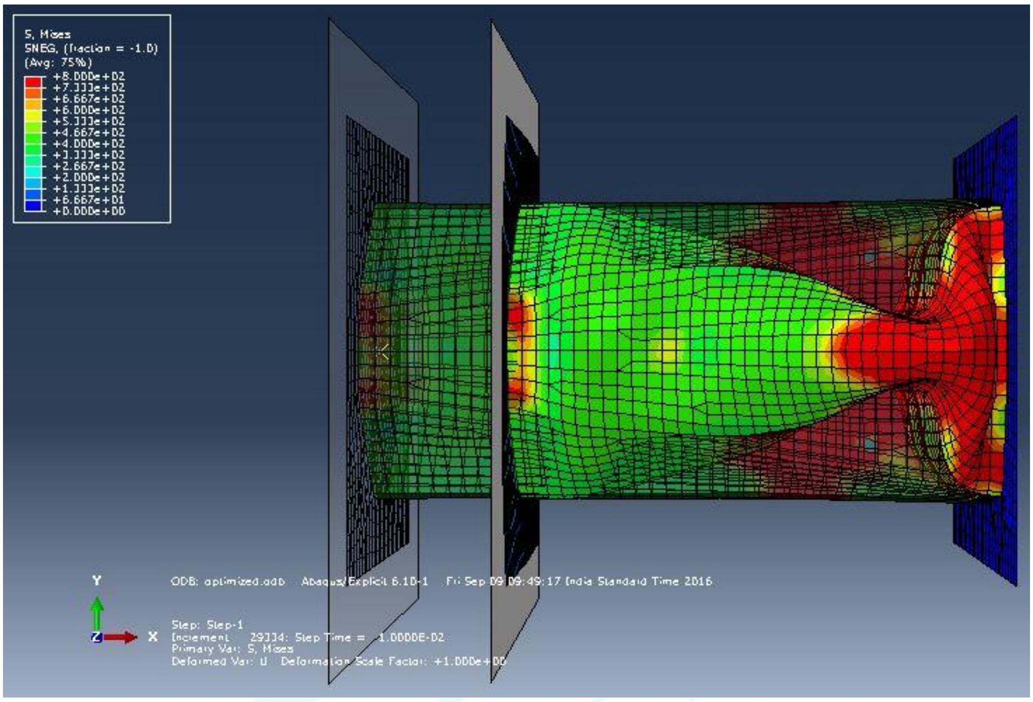


Figure 5.13: Deformation at final step for Model 3

5. Conclusions and Recommendations

Total energy absorption of the New Optimized design and the other regular designs are compared with each other, for the Low speed Crash simulation test conditions. Total Energy, Internal energy, Strain energy, Kinetic Energy, External work done, in each case is compared. Force vs Displacement curve is also plotted in all the cases and it is concluded based on all the results, that the optimized design is more efficient and safest design during the Frontal crash situation.

By implementing the new design in the system, it could serve its purpose well with more efficiency, by absorbing more impact energy during the Frontal Crash situation and serve better purpose for the same material cost and volume.

By comparing the computational results of new design with the other regular standard designs it can be concluded that

- There is considerable increase in the impact energy absorbed
- For the same material and manufacturing cost, the optimized design would serve its purpose to the fullest
- The Optimized design deformation is safest, since its absorption is steady and gradual without any kinks

The following are the future recommendations which can be addressed by using composite material with new design.

- Experimental validation can be done before practical implementation of the Optimized shape in automobile industry.
- Effectiveness of the new design can be improved by using Holes and/or beads for proper folding of the sheet during the Impact conditions.
- Change in the thickness of the sheet can also be considered in the design changes, but it would affect the manufacturing processes.
- Varying area of cross-section, Foam filled methods can also be added to this design, for better results.

References

[1] Yamazaki K, Han J. Maximization of the crushing energy absorption of tubes. StructOptim1998;16:37–46.
[2] Lee S-H, Kim H-Y, Oh S-I. Cylindrical tube optimization using response surface method based on stochastic process. J Mater Process Tech 2002; 130–131: 490–496.
[3] Sheriff NM, Gupta NK, Velmurugan R, Shanmugapriyan N. Optimization of thin conical frusta for impact energy absorption.Thin-WalledStruct2008;46: 653–66.
[4] Avalle M, Chiandussi G. Optimization of a vehicle energy absorbing steel component with experimental validation. Int J Impact Eng2007; 34:843–58.
[5] Hou S, Li Q, Long S, Yang X, Li W. Multi objective optimization of multi-cell sections for the crash worthiness design. Int J Impact Eng2008; 35:1355–67.
[6] Hou S, Li Q, Long S, Yang X, Li W. Design optimization of regular hexagonal thin-walled columns with crashworthiness criteria. Finite Elem Anal Des 2007; 43:555–65.
[7] Lönn D, Fyllingen O, NilssonaL. An approach to robust optimization of impact problems using random samples and meta-modeling. Int J Impact Eng 2010;37:723–34.
[8] Acar E,Guler M A, Gerceker B, Cerit M E, Bayram B. Multi-objective crash- worthiness optimization of tapered thin-walled tubes with axisymmetric indentations. Thin-WalledStruct2011; 49(1):94–105.
[9] Qi C, Yang S, Dong F. Crushing analysis and multi objective crashworthiness optimization of tapered square tube under oblique impact loading. Thin-Walled Struct 2012; 59:103–19.
[10] Liu Y. Crashworthiness design of multi-corner thin-walled columns. Thin-Walled Struct 2008; 46:1329–37.
[11] Yang S, Qi C. Multi objective optimization for empty and foam-filled square columns under oblique impact

loading.IntJImpactEng2013; 54:177–91.
[12] Zarei H R, Kröger M. Multi objective crashworthiness optimization of circular aluminum tubes.Thin-WalledStruct2006; 44:301–8.
[13] Kim C H, Arora Jasbir S. Development of simplified dynamic models using optimization: application to crushed tubes. Computation Method Appl M 2003; 192:2073–97.

ABAQUS Cards Used:

```
*MATERIAL, NAME= STEEL
 *DENSITY
            7.85E-9,
*ELASTIC
203000., 0.3
*PLASTIC
 129.1,      0.
 163.71, 0.00052
 169.04, 0.00102
 169.98, 0.00137
 170.44, 0.00176
 176.53, 0.02687
 265.04, 0.04478
 334.53, 0.06574
 392.49, 0.09142
 425.23, 0.11252
 454.58, 0.13915
 473.96, 0.16732
   800.,    0.17
****************************************************************************
*SHELL SECTION, ELSET=P1;CRUSHCANS, MATERIAL=M1;STEEL
3., 5
** SECTION: SECTION-2-P2;PLATES
*SHELL SECTION, ELSET=P2;PLATES, MATERIAL=M1;STEEL
3., 5                 0.3,        , M12000935;COMPOSITE_MAT_BASE,                    0., A
*ELEMENT, TYPE=R3D4
1321, 19129, 19131, 19132, 19130
*NSET, NSET=RIGID_BARRIER-REFPT_, INTERNAL
 19273,
*ELSET, ELSET=CONSTRAINT, INSTANCE=PART-1-1
*ELSET, ELSET=PLATES, INSTANCE=PART-1-1
*ELSET, ELSET=RIGID_BARRIER, INSTANCE=RIGID_BARRIER-1
 1321,
*NSET, NSET=_REF-PT_RIGID_BARRIER-1_19273, INTERNAL, INSTANCE=RIGID_BARRIER-1
 19273,
****************************************************************************
** INTERACTION PROPERTIES
**
*SURFACE INTERACTION, NAME=I1;FRICTION
*FRICTION
 0.2,
****************************************************************************
** BOUNDARY CONDITIONS
**
** NAME: BC-7 TYPE: DISPLACEMENT/ROTATION
*BOUNDARY
** NAME: IC-1   TYPE: VELOCITY

*INITIAL CONDITIONS, TYPE=VELOCITY
_M16, 1, 4633.
****************************************************************************
** INTERACTIONS
**
** INTERACTION: GENERAL_CONTACT
```

```
*CONTACT, OP=NEW
*CONTACT INCLUSIONS, ALL EXTERIOR
*CONTACT PROPERTY ASSIGNMENT
 I1;FRICTION
****************************************************************************
** STEP: STEP-1
**
*STEP, NAME=STEP-1
*DYNAMIC, EXPLICIT
, 0.01
*BULK VISCOSITY
0.06, 1.2
****************************************************************************
** BOUNDARY CONDITIONS
**
** NAME: DISP-BC-1 TYPE: DISPLACEMENT/ROTATION
*BOUNDARY
_M11, 4, 4
** NAME: DISP-BC-2 TYPE: DISPLACEMENT/ROTATION
*BOUNDARY
_M11, 5, 5
** NAME: DISP-BC-3 TYPE: DISPLACEMENT/ROTATION
*BOUNDARY
_M11, 6, 6
** NAME: DISP-BC-4 TYPE: DISPLACEMENT/ROTATION
*BOUNDARY
_M13, 2, 2
** NAME: DISP-BC-5 TYPE: DISPLACEMENT/ROTATION
*BOUNDARY
_M13, 3, 3
** NAME: VEL-BC-1 TYPE: VELOCITY/ANGULAR VELOCITY
*BOUNDARY, TYPE=VELOCITY
_M12, 1, 1, 4633.
****************************************************************************

** OUTPUT REQUESTS
**
*RESTART, WRITE, NUMBER INTERVAL=1, TIME MARKS=NO
**
** FIELD OUTPUT: F-OUTPUT-1
**
*OUTPUT, FIELD, VARIABLE=PRESELECT
**
** HISTORY OUTPUT: H-OUTPUT-1
**
*OUTPUT, HISTORY, VARIABLE=PRESELECT
*END STEP
****************************************************************************
```