RES MILITARIS
REVUE EUROPEENNE D ETUDES EUROPEAN JOURNAL OF MILITARY STUDIES
Social Science Journal

Thermal Analysis of Emergency Container in Steel Industries Using FEA

By

Srihari Palli
Mech. Engg Dept, Aditya Institute of Technology and Management, Tekkali, India

Rakesh Chandmal Sharma**
Mech. Engg Dept, Graphic Era (Deemed to be University) Dehradun, India
Email[**]: rcsharmaiitr@gmail.com

R.Sivasankar Raju
Mech. Engg Dept, Aditya Institute of Technology and Management, Tekkali, India

Sono Bhardawaj*
Mech. Engg Dept, Shivalik Group of Institutions, Dehradun, India
Email[*]: sonupanditsharma59@gmail.com

Abstract

Traditional evacuation vessels used in the steel industry have a limited capacity and required to be lined with refractory material, which is an expensive and time-consuming procedure. Additionally, cast iron has been replaced with spheroidal graphite iron as the material for the emergency container. The purpose of this study is to replace the traditional emergency container with a modified version in order to avoid using refractories, enhance capacity, and create a design that is optimal for user-friendliness. The suggested one might be subjected to extremely high thermal loads, so before construction, a thermal analysis of the container is done. To simulate the model, we utilise ANSYS 14.0 software. This study analyses whether the altering emergency container can meet the intensity need, providing the company with significant theory gist.

Keywords: Backup Container, Refractory Material, Transient Flow, Thermal Analysis, Steel, FEA

1. Introduction

A steel mill's melting steel debris is stored in a safety tank, a sealed tank. When steel is made from liquid metal, a waste element known as slag is created. Cast iron, because it is more damage durable while being brittle, is often used to construct emergency storing tanks in industry. The traditional vessel versions shown in Fig.1 typically have a capability to hold 6-8 tonnes and are of tetrahedral structure that prohibits them from being raised by machines, making it challenging to manage slag within the vessel. At elevated heats, refractory compounds must retain their structural and molecular characteristics. They must be able to withstand extreme temperatures, be chemically neutral, and fall within narrow parameters for heat conductance and heat expansion, all of which change with the working conditions. The most common compounds employed to create refractories are aluminium, and magnesium (magnesia) [2, 3]. Over time, the container's temperature distribution is evaluated, and if necessary, more treatment is authorised.

The refractory layer, which is employed to protect the crucible against high - temperature waves, must be created well before tank is filled by heated material and again afterwards the tank is emptied. To overcome these challenges, a better prototype has to be developed [4]. A proposal was done by typical Steel industry to modify the design of container stating more advantages over the conventional one.The modified crucible has provision for hooks and thus easily handled with cranes.The objective of this present work is to determine whether the modified containeris suitable to address the steel plant's requirement, and a conclusion is made whether the container can sustain such large thermal stresses in comparison with the real time operating conditions.To find the temperature distribution of the container at different time intervals is also necessary factor to analyse the container design. Analysis is done assuming the practical case i.e. pouring the slag in layer form so that accuracy in results is obtained. Analysis is done using ANSYS software by applying conduction, convection and radiation heat losses of the container. Researchers in the past had carried out various structural and thermal analysis using numerical simulation and FEA techniques [5-19].

Fig.1. *Conventional emergency container (Courtesy: Visakha Steel Plant)*

2. Literature Review

Palli et al., 2021 [10] studied the dynamical responsiveness of a component underneath the operation of certain broad time-dependent stresses may be determined by a method known as time dependent dynamical assessment. The reloading period range is long enough that the inertial or absorption impacts are taken into account. The current study utilises finite element analysis to undertake a period - record assisment of a conventional locomotives carriage under Indian railway circumstances. After modelling a "bump in the track's surface" in the shape of an "ellipsoid" and assuming that an automobile travelling at that speed would cross the bump in 0.144 seconds, the authors mapped the time-dependent alteration in movement at several crucial regions on frameworks of trucks and cars subjected to the same loading settings. Because of the uneven weight arrangement, the forward and back of the locomotives wagon and carriage chassis are more sensitive to wheel resonance than the middle.

Dontikurti et al., 2020 [18] lessen the amount of emissions released by gas-powered vehicles. Typically, a three-way catalytic rectifier is installed in the emissions circuit to minimise emissions by reacting with dangerous pollutants including carbon dioxide, nitrous oxide, particulate emissions, etc. So, in addition to the three-way catalytic connector, gypsum, sandstone, is added to the effluent line to lower the pollution level. At the end of a battery of experiments conducted at the emission monitoring facility, the emissions levels without and with the addition of sediments to the outflow circuit are compared. This results in a significant reduction in the hazardous pollutants discharged via the emission line because to the limestone's ability to soak them. A graphic plot is then created which indicates the amount of hazardous gas is decreased using limestone.

Rao et al., 2020 [20] used "Friction Stir Welding" (FSW) to join the two different types of aluminium composition sheets to analyse the impact of energy production during the joining operation. Using sensor, author were able to determine that the joining process caused the workpiece's advanced face to heat up more than the retracted face. Three different tool settings and four different feed rates were employed in the FSW simulation. The Taguchi OA was used to plan the experiments. Particle dimensions are somewhat smaller at the HAZ for similar weld joints, but much greater for dissimilar weld joints. Welding region energy production was shown to be increased by increasing the process velocity while maintaining the same feed rate. If the feed rate is increased while the rotational speed of the tool is kept constant, the quantity of heat produced in the welding area may be reduced. The optimal thermal range and heating production setting was found by "Analysis of Variance" (ANOVA) for the advancing face and for the retreating face. The thermal profile of the whole material was mapped out using thermodynamic modeling.

Sharma 2017 [17] investigated the kinematics of a three-wheeled automobile prototype with nine degrees of freedom that are connected vertically and laterally. PSD horizontal and vertical acceleration of the sprung weight of a three-wheel automobile are simulated and compared with the equivalent observed from real experimentation, validating the analytical concept. Naturally occurring rates of the system's primary rigid - body are calculated using MATLAB computations. The essential velocity of the automobile is calculated, and the factors that affect the crucial velocity are examined, all in an effort to better understand the car's responsive equilibrium.

3. Modeling

The preliminary approach to carry out the simulation includes building an appropriate container model. The drawing of the container was obtained from a typical steel industry and is presented in the Fig. 2. below. The CAD model is initially developed and further translated into ANSYS, finite element analysis software.

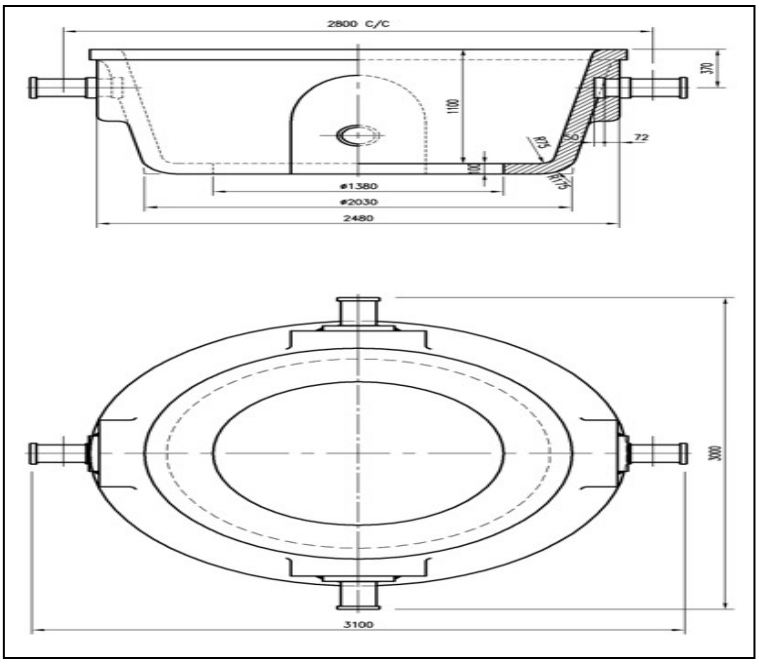


Fig.2. *Container Drawing*

NX10, a top-tier CAD/CAM/CAE programme, was used to create the conceptual framework, and it has a feature that allows the user to export the layout to a variety of file types. The finished tank layout is obtained in IGES template, making it simple to exporting into ANSYS. The tank is shaped like a flat cone, with average diameter of 2030 mm at the base and 2550 mm at the top; it is 100 mm thick and stands 1200 mm tall; and it is equipped with four lifting bails that allow it to be hoisted by a crane.

Fig.3. *3D model of container*

3.1. Limitations of Model

The NX modeling meets our needs but running it in ANSYS is prohibitively time-consuming because of the complexity and size of its many individual components. This leads to the model being explained as a symmetrical structure, and then a quarter of the model being chosen for analysis [2]. Fig.4 shows a cross-section of the container, representing a fourth of the whole, which is then transformed to a 2D axisymmetric design and solved as a plane stress potential issue.

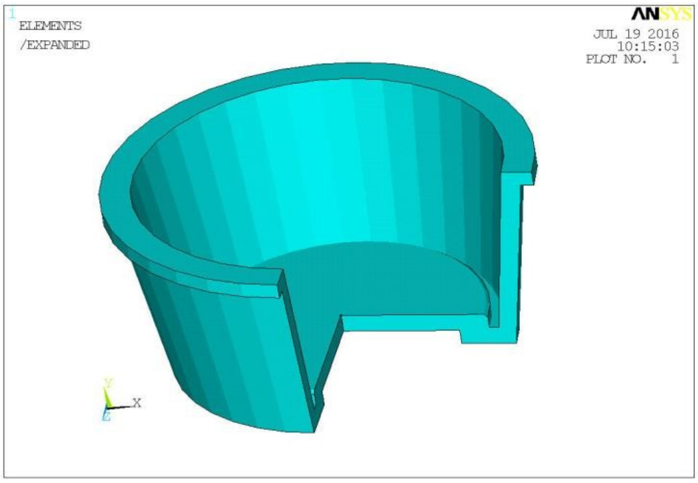


Fig.4. *Portion of container for reduction into Axis symmetric model*

3.2. Reduced Fe Model

The 2-dimensional inputs are a homogeneous 3-dimensional issue due to rotary symmetrical, and the investigation is straightforward due to the axially symmetrical architecture so specified for it [20]. The axisymmetric area considered for final transient thermal analysis in FEA package ANYS is shown in Fig.5. Above figure is the axisymmetric portion of the emergency container which is two dimensional in x and y axis. The dimensions of the model donot change in axisymmetric structure and thus analysis does not bring any difference in 2D and 3D.

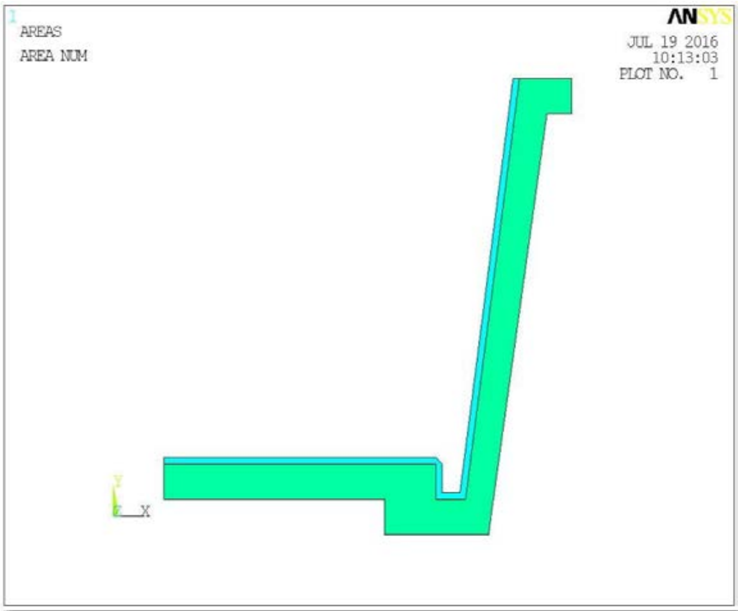


Fig.5. *Axisymmetric model of container*

3.3. Discretization Of Container Model

Making a mesh of nodes, which are grid points, is the definition of meshing. The software's numerous features and options are used to complete the task. At each node of the mesh, the applicable governing equations are numerically solved to determine the outcomes. The FEM technique is often used to evaluate the controlling partial differential equations [3]. The FEmesh used to discretize the container model consists of PLANE55 for thermal and SURF151 for radiation element types and real constant considered for SURF151 is Stephan Boltzmann constant. Mesh is the most crucial component of analysis and can gauge its usefulness and efficiency. The generated auto mesh of the axisymmetric container model is shown in Fig.6.

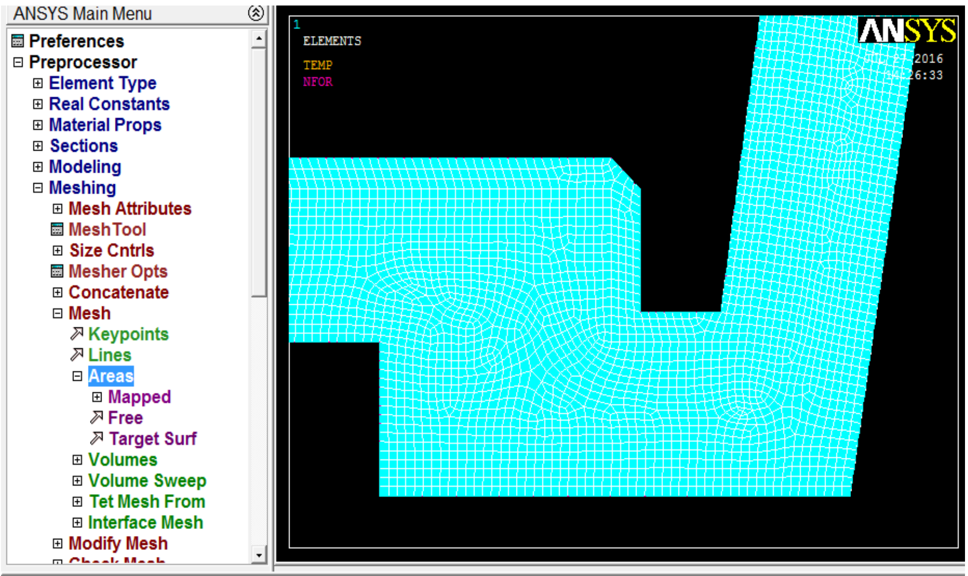


Fig.6. *Meshing axis symmetric model of container*

3.4. *Material Attributes and Boundary Conditions*

The crucial step in building an emergency container is selecting the material. When slag is poured into the crucible, the vessel may be susceptible to significant impact [4]. The tank should be able to transport slag at elevated temperature levels, and the interior heat should be dissipated as soon as allowed via the structure's walls. This means that the heat is affected by the movement of the airflow on the layer. In order to rapidly convey energy across metal, a substance with elevated conductivity is preferred. "Spheroidal graphite iron" is the superior element for vessels due to its excellent mechanical properties and good heat transmission. The container material is spheroidal graphite iron, and its density value is taken as 2100 kg/m^3.

Taking into account the axially symmetrical nature of the durable vessel and presuming the slag is put to a specific altitude within the tank, it is assumed that the heat on the inner wall will move via the vessel surface as energy exchange towards the outer-wall, where it will be exposed to convective thermal energy exchange. The temperature is applicable parallel to the wall of the vessel, and the slag is assumed to be at a 1723K. The vessel underwent a time - dependent thermodynamic study. This assessment accounted for radiation dissipation by using the "Stephen Boltzman constant". Figures 7 and 8 indicate the boundary conditions the container is subjected to for running the transient analysis.

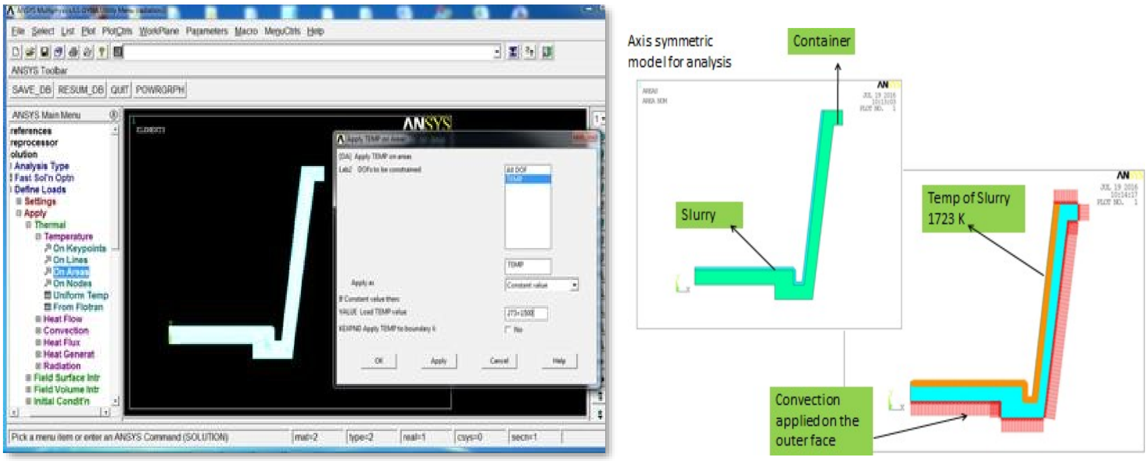


Fig 7. *Applying temperature to the slurry and Boundary conditions applied for convection*

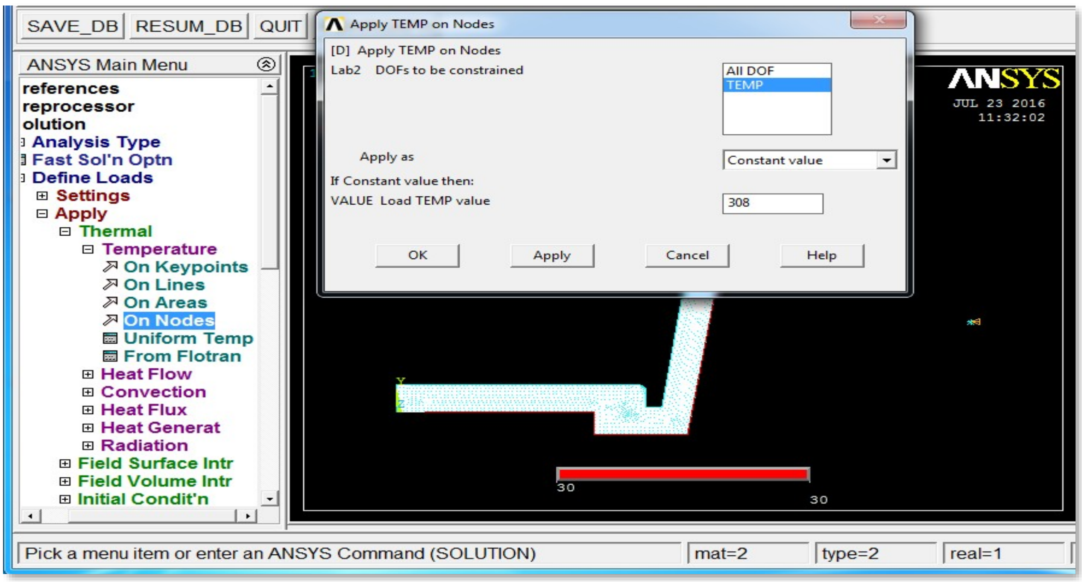


Fig.8. *Applying radiation at temperature 308 K*

4. Results And Discussion

The findings are shown over the course of an hour under the following conditions: the slurry is dropped from a specific elevation over the layer of the vessel, convective heat flow is performed across the top layer of the vessel, and an irradiation component is generated to account for irradiation emissions. Following 2 seconds of exposure to the heated slag, the inside face of the vessel reaches a peak temperature of 1773K, while the outside region cools to a standard rate of 308K.

After 10 seconds, there is a little variation in temperature on the outside surface of the vessel, and it goes from 308K to 470K.

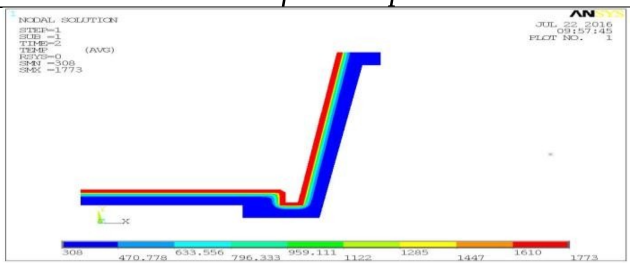
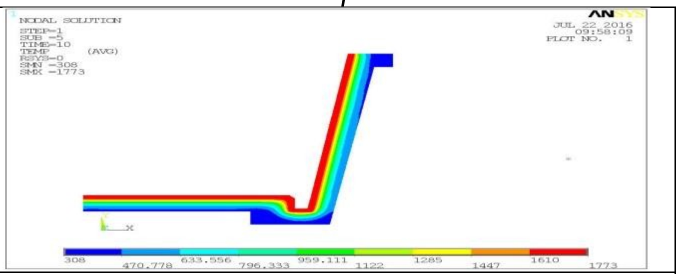
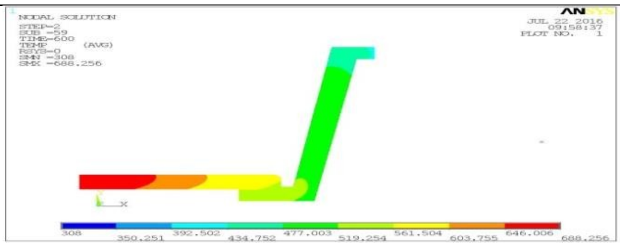
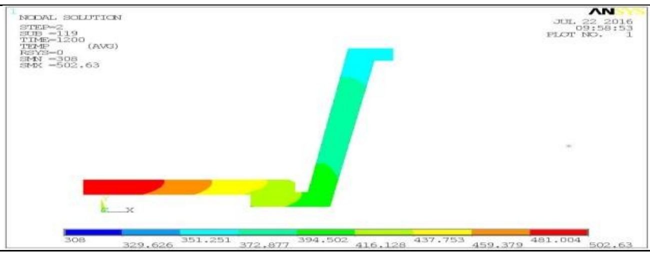
In this case, the peak temp is uniformly spread throughout the vessel after 600 seconds, with readings of 688.256K and 519.25K measured at the centre bottom and outside surface, respectively.

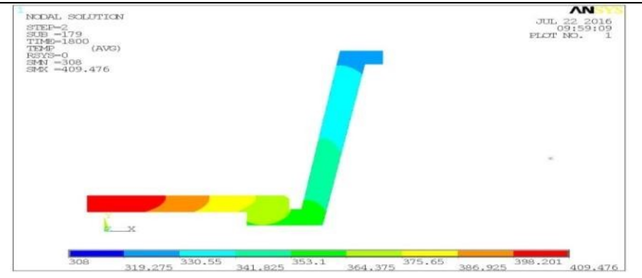
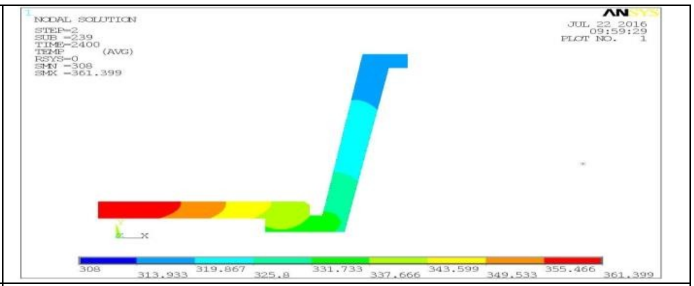
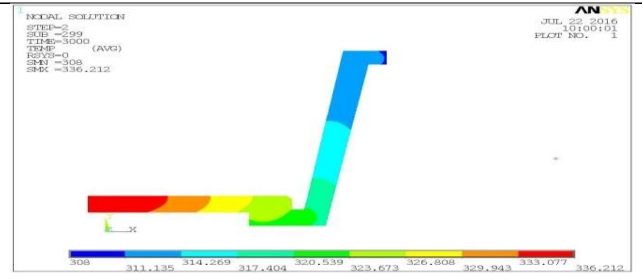
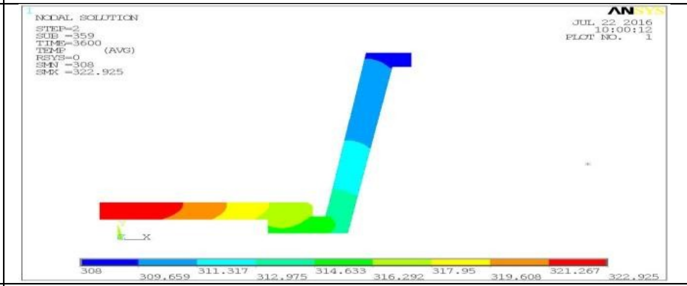
The vessel cools after 1200 seconds, reaching a temp of 394K on the outside walls and a minimum of 302.63K in the centre. After 1800 seconds, the structure's centre bottom reaches a high of 409.47K and its top edge reaches a lowest of 330.55K.

After 2400 seconds, the temp remains to drop, reaching a low of 319.86K and a high of 361.39K.

After 3600 seconds, or an hour, the outside border of the vessel is 308K, while the surface temp is 308.65K to 314.63K in the structure's centre. The temperature decreases to 317.95 K at the structure's bottom boundary, from 322.92 K in the centre. After one hour, the vessel reaches a high of 48.850C and a lowest of 340C, both of which are comfortably ambient. The plot results of the temperature distribution under transient heat transfer are presented in table 1.

Table 1. *results of the temperature distribution under transient heat transfer*

	
Temperature distribution of the container after 2seconds	Temperature distribution of the container after 10 seconds
	
Temperature distribution of the container after 600 seconds	Temperature distribution of the container after 1200 seconds

	
Temperature distribution of the container after 1800 seconds	Temperature distribution of the container after 2400 seconds
	
Temperature distribution of the container after 3000 seconds	Temperature distribution of the container after 3600 seconds

4.1. Comparison Of Computational Model with Real Time Container

The results obtained from the numerical simulation are compared with that of the results experimentally obtained for the modified emergency container installed in typical steel industry in north coast of Andhra Pradesh. Table 2 indicates the comparative values of simulated as well as actual experimental temperature distributions of the container walls

Table2. *Comparison of computational and experimental temperature distributions for one hour*

Simulation Model	Experimental Model	Simulation Model	Experimental Model
At 2 second		**At 1800 second**	
Internal Wall-temp = 1773 K	Internal Wall-temp = 1773 K	Internal Wall-temp = 353 K	Internal Wall-temp = 333 K
Outside Wall-temp = 308 K	Outside Wall-temp = 308 K	Outside Wall-temp = 341 K	Outside Wall-temp = 321 K
At 10 second		**At 2400 second**	
Internal Wall-temp = 1773 K	Internal Wall-temp = 1773 K	Internal Wall-temp = 331 K	Internal Wall-temp = 311 K
Outside Wall-temp = 470 K	Outside Wall-temp = 470 K	Outside Wall-temp = 325 K	Outside Wall-temp = 315 K
At 600 second		**At 3000 second**	
Internal Wall-temp = 519 K	Internal Wall-temp = 510 K	Internal Wall-temp = 317 K	Internal Wall-temp = 310 K
Outside Wall-temp = 519 K	Outside Wall-temp = 500 K	Outside Wall-temp = 314 K	Outside Wall-temp = 311 K
At 1200 second		**At 3600 second**	
Internal Wall-temp = 510 K	Internal Wall-temp = 495 K	Internal Wall-temp = 311 K	Internal Wall-temp = 310 K
Outside Wall-temp = 477 K	Outside Wall-temp = 450 K	Outside Wall-temp = 309 K	Outside Wall-temp = 308 K

5. Conclusions

According to the simulation results, the container's highest and minimum temperatures throughout one hour do not reach the metal's melting point, which is 1422.15 K for spheroidal graphite iron. Due to the strong heat conductivity of the container's metal, the 1723 K slag temperature quickly spreads throughout the container and escapes into the atmosphere. Following the application of hot slag to the container for a period of one hour, the temperature of the container returns to normal. As a result, we can say that the modified emergency is secure when used as intended. The FEA numerical results are in good agreement with the practical values suggesting that the installed container which is a modified version of the conventional one is good to go for long term. However, further investigations are required to study its reliability and durability. By taking the object's radiated and convective emissions into account, ANSYS's thermodynamic evaluation provides findings that are in line with the real world.

Acknowledgements

The authors are grateful to Mr. G. Veerababu, Deputy General Manager, Steel Melting Shop, VisakhaSteel Plant for his guidance and inputs.

References

Shan Li, Yuqi Wang,Bo GAO and Lei Wang, 2012,Finite element analysis to the reformation structure of 75 ton iron ladle based on ANSYS-workbench. Advanced Materials Research Vols.479-481, PP 2120-2123, Trans Tech Publications.

Tian Hong-yu,CHEN Fu-rong,XIERui-jun,LIshi-hui, 2010, Finite Element Analysis of 100 t Hot Metal Ladle in Process of Tipping, Journal of Iron and Steel Research International, 179110,19-23.

KazumichiShimizu, YaerXinba,Masahibto Tanaka and Hideki Shudai, 2009, Mechanical Properties of Spheroidal Graphite Cast Iron Made by Reduced PressureFrozen Mold Casting Process .Materials Transactions,Vol.50, No.5, pp.1128 to 1134,The Japan Institute of Metals.

Jianping Wang, Chenglong Liu, Peng Lu, Fugui Ma. 2011, ANSYS-based Finite Element Analysis on Automotive Panel Drawing Die.Advanced Materials Research Vols 314-316 pp 554-557,Trans Tech Publications,https://doi.org/10.4028/www.scientific.net/AMR.314-316.554

Swathi K, Srihari Palli. Heat transfer enhancement ina tube using rectangular strip inserts. Int J Appl Eng Res. 2015; 10: 41532–41544

Rao CJ, Siva Sankara Raju R, Sreeramulu D, Sagar Y. Fabrication and Evaluation of Optimized Parametric Condition During EDM Machining of Al-CSA Composite Using Taguchi Orthogonal Array. In: Lionel G, Karthikeyan R, Muralidharan V (eds) Adv. Des. Therm. Syst. Proc. ETDMMT 2020, 1st ed. Springer Singapore, pp 411–420

Sharma RC, Palli S, Koona R. Stress and vibrational analysis of an Indian Railway RCF bogie. Int J Veh Struct Syst. 2017; 9: 296–302

Palli S, Koona R, Sharma RC, Muddada V. Dynamic analysis of indian railway integral coach factory bogie. Int J Veh Struct Syst. 2015; 7: 16–20

Sharma RC, Palli S, Sharma N, Sharma SK. Ride behaviour of a four-wheel vehicle using h infinity semi-active suspension control under deterministic and random inputs. Int J Veh Struct Syst. 2021; 13: 234–237.

Palli, S., Dontikurti, R., Sharma, R.C., Sharma, N. Analysis of dynamic response of a railway locomotive using ansys. International Journal of Vehicle Structures and Systems, 2021, 13(2), pp. 250–255.

Satyanarayana, V.S.V., Sharma, R.C., Sateesh, B., ...Mohan Rao, N., Palli, S. Optimum Response of a Nonlinear Passive Vehicle Suspension System under Random Road Excitations. SAE International Journal of Commercial Vehicles, 2022, 16(1).

Bhardawaj, S., Sharma, R.C., Sharma, S.K. Analysis of frontal car crash characteristics using ANSYS. Materials Today: Proceedings, 2020, 25, pp. 898–902.

Sharma, SK. Sharma, RC, Lee, J, Jang, H-L. Numerical and Experimental Analysis of DVA on the Flexible-Rigid Rail Vehicle Carbody Resonant Vibration. Sensors, 2022, 22(5), 1922.

Sharma, RC, Palli, S, Koona, R. Stress and vibrational analysis of an Indian Railway RCF bogie. International Journal of Vehicle Structures and Systems, 2017, 9(5), pp. 296–302.

Sharma, R.C., Sharma, S., Sharma, S.K., Sharma, N., Singh, G. Analysis of bio-dynamic model of seated human subject and optimization of the passenger ride comfort for three-wheel vehicle using random search technique Proceedings of the Institution of Mechanical Engineers, Part K: Journal of Multi-body Dynamics, 2021, 235(1), pp. 106–121.

Bhardawaj, S., Sharma, R.C., Sharma, S.K. Development of multibody dynamical using MR damper based semi-active bio-inspired chaotic fruit fly and fuzzy logic hybrid suspension control for rail vehicle system. Proceedings of the Institution of Mechanical Engineers, Part K: Journal of Multi-body Dynamics, 2020, 234(4), pp. 723–744.

Sharma, Rakesh Chandmal. "Ride, eigenvalue and stability analysis of three-wheel vehicle using Lagrangian dynamics." International Journal of Vehicle Noise and Vibration 13, no. 1 (2017): 13-25.

Dontikurti, Raghuveer, R. Sivasankara Raju, Yogesh MR Gowda, and P. Srinivasa Rao. "Reduction of Pollution on Engine Exhaust System using Sedimentary Rock." (2020).

V.M.M. Thilak, R. Krishnaraj, Dr. M.Sakthivel, K.Kanthavel, Deepan Marudachalam M.G, R.Palani, 2011, Transient Thermal and Structural Analysis of the RotorDisc of Disc Brake. International Journal of Scientific & Engineering Research Volume 2, Issue 8, ISSN 2229-5518

Rao, C. J., Satish Kumar Aadapa, and Sagar Yanda. "Predication of temperature distribution and strain during FSW of dissimilar Aluminum alloys using Deform 3D." Materials Today: Proceedings 59 (2022): 1760-1767.