Materials Research Proceedings 31 (2023) 564-570 https://doi.org/10.21741/9781644902592-58

Numerical simulation of high-pressure gas-jet impingement on the adjacent equipment

Esam I Jassim[1], Muhammad Asad[1], and Bashar Jasem[2]

[1]Department of Mechanical Engineering, Prince Mohammad Bin Fahd University, Khobar, KSA

[2]Department of Computer Technology, Al-Hadba University College, Mosul, Iraq

Keyword: High-Pressure Gas, Cylinder Rupture, CFD, Jet Impingement, Thermal Stresses

Abstract. Cold jet is a result of a high-pressure leakage through a wall crack, valve stem, or any other opening caused by an accident or failure to a high-pressure device. The phenomena and its impact on the adjacent and surrounding equipment are simulated using Computational Fluid Dynamics (CFD) techniques. For the preliminary study, the working fluid used in the simulation is methane. The simulation is linked to appropriate thermal and fluid property estimation software using parameter-tuned equations of state to predict the real multi-component natural gas flow conditions. Through the simulation, a spot of condensation and/or nucleation was predicted. These spots are crucial since their effect appear on the jet simulation in which multiphase properties must be considered. The results showed that the temperature variation on the adjacent surface is divided into three regions, namely: vicinity of stagnation point, sharp variation region, and mild variation region. Such distribution results in inception of thermal stresses that could cause catastrophic incident if rapture occurs. Hence, wall thickness of the equipment that exposed to cold jet should be cautiously selected to sustain the extra thermal stress as well as localised non-uniform stress distribution as detailed in the Finite Element based coupled thermo-mechanical analysis.

Introduction

Industries with complex equipment that deal with hazardous substances mandate that a Quantitative Risk Assessment (QRA) is implemented. The need of simulation of accidental scenarios to measure the risk quantity requires efficient and accurate model to accommodate all aspects of the scenario such as compressibility effect, interaction with the surroundings, and geometrical complexity. Empirical models pertaining to jet phenomena, however, lack the presentation of jet-obstacles interaction, resulting in overestimation of the damage area. Hence, an accurate and more realistic method for the jet phenomena becomes essential. Computational Fluid Dynamics (CFD) emerged as a state-of-the-art technique aiming to simulate accidental pressurized gas releases in congested industrial environments.

In the present work, a high-pressure methane leakage from a small hole in a congested environment is considered. When high-pressure fluid leaks to the ambient due to tiny hole or crack, supersonic flow is induced and shock would be incepted. Numerous researches have addressed the shock location experimentally [1, 2] as well as numerically [3, 4]. Jassim studied the nozzle geometry impaction on the shockwave location using CFD FLUENT code. He reported that elliptical shape, among six different geometries, predicts the shock farthest from the throat. Jassim et al. studied numerically the effect of real gas behavior in supersonic nozzle on shock location [5] and vorticity potential [6]. The outcome of the simulation revealed that shock inception is significantly altered when an accurate equation of state is employed. Their results also concluded that the vorticity increases rather sharply near the shock.

The phenomenon is complex and for simplicity, it splits into two parts: release and impact. Near the rupture, the released gas is under-expanded and tends to adjust to the ambient conditions through some expansion and compression waves (shock waves).

The resulting flow is supersonic and highly compressible with strong discontinuities of the flow field variables, clearly represented by the presence of a Mach disk: this is the "release" phase. Release and dispersion of pressurized gas have been addressed in many literatures. Wilkening and Baraldi [7] modeled the jet formed from the release of pressurized hydrogen. They realized that a small-time step and dense mesh near the shock inception region are necessary. Deng et al. [8] modified the simulation by splitting the jet release and dispersion phases. Initially, the expansion process is modelled using pseudo-source. Then, it is used as an input to the modeling of dispersion process.

Dharavath et al. [9] numerically investigated two turbulence models. They figured out that k–ε model performed better in predicting the variation of fluid properties. The outcome of the simulation reported that fine meshing is mandatory for accurate predicting near shock region.

Theory

The geometry of the crack considered in the study is nozzle with 2D axisymmetric geometries. The general form of the continuity equation for steady state compressible flow, in deferential form, is given by Eq.1:

$$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}=0 \tag{1}$$

The 2D Eular Equations for compressible, inviscid flow can be simplified to the following expressions:

$$\frac{\partial(\rho u)}{\partial t}+u\frac{\partial(\rho u)}{\partial x}+v\frac{\partial(\rho u)}{\partial y}=-\frac{\partial P}{\partial x} \tag{2}$$

$$\frac{\partial(\rho v)}{\partial t}+u\frac{\partial(\rho v)}{\partial x}+v\frac{\partial(\rho v)}{\partial y}=-\frac{\partial P}{\partial y} \tag{3}$$

Energy equation in deferential form is obtained from Eq 4:

$$\rho\frac{\partial(e)}{\partial t}=\frac{p}{\rho}\frac{d\rho}{dt}+k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right)+\rho\frac{dQ}{dt} \tag{4}$$

CFD modelling

To construct the nozzle geometries, unstructured cells were used. The grid density was higher near the outlet of the nozzle so that the resolution for capturing shocks can be improved. Accuracy of any simulation is significantly dependent on the mesh construction. Structure of supersonic flow is complex due to large speed gradient and sudden drop in the flow velocity across the shock. Hence, art of mesh size, number, and density construction should account for grid resolution and time consumption.

The essence of grid independency is to test different resolutions in a way that the computational grid should be able to capture the actual flow features. Since the accuracy of flow structure and properties of the fluid rely on grid resolution of the entire system, the near exit of the nozzle is chosen for measuring the resolution. Previous studies on shock capturing analysis have shown that the thickness of the shock wave is of order of 0.1 mm when the gas undergoes severe gradients in the thermo-fluid properties [1, 5, 6]. Hence, sufficient cell resolution was maintained in the region where the shock induced. From an optimization perspective, the analysis of grid resolution concludes that 0.5 mm grid spacing is firmly acceptable in the convergent portion while 0.1 mm spacing is more adequate to capture the flow feature (including shock position) in the divergent portion of the nozzle.

Results and Discussion

Jet simulation

A jet of natural gas developing outside a tank hitting a stationary wall some distant apart is simulated using FLUENT code. Figure 1 illustrates the contours of the velocity dispraise of the Jet. Near the impingement region, the flow direction deviates due to the effect of the stagnation point. The altered stream then strikes the wall of the adjacent component (in this case another tank with the same height) leaving unaffected area under less strength. Thus, the temperature of such area becomes relatively higher.

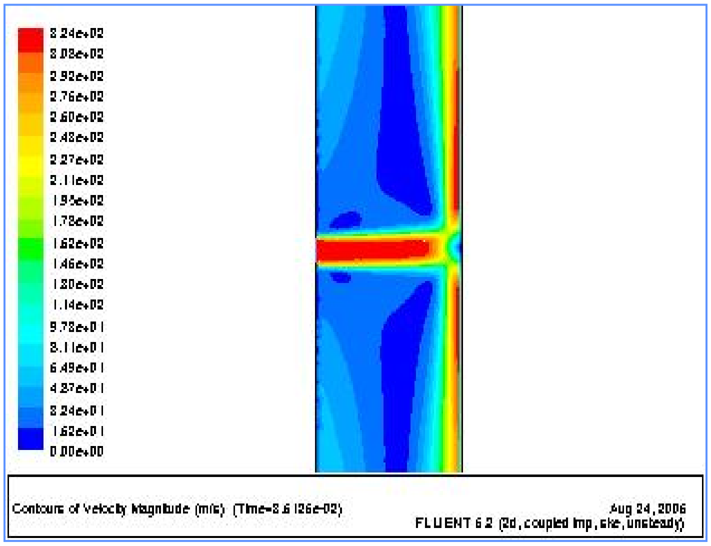
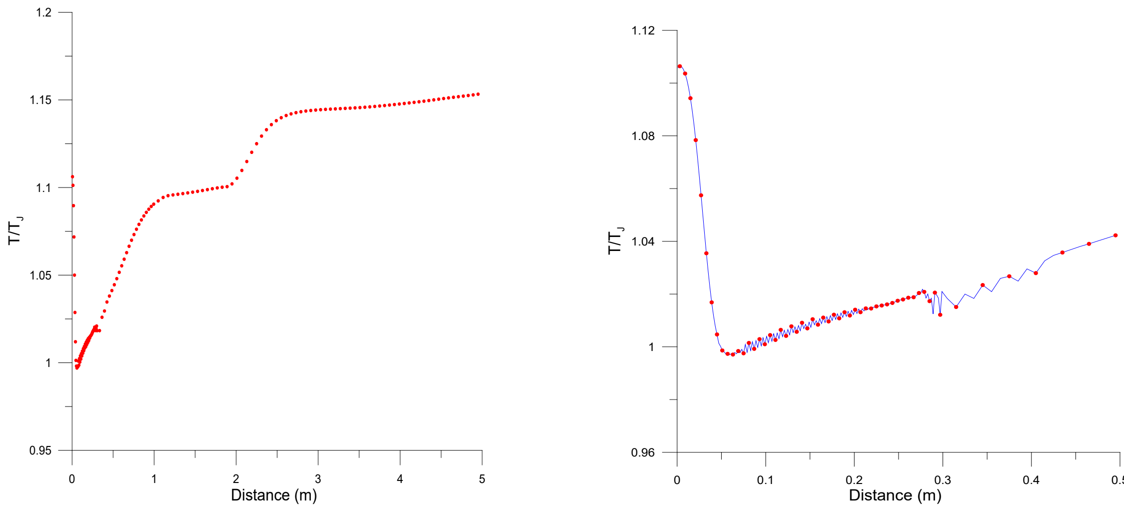


Fig. 1: Velocity contours of jet impingement

Temperature distribution along the adjacent wall is depicted in Fig. 2. The temperature variation can be divided into three sub-regions, namely: stagnation point vicinity, sharp variation region, and mild variation region.

As discussed in the jet velocity profile near the adjacent wall, the flow slightly alters then severely changes its direction to become parallel to the wall as it moves closer towards the surface. Thus there is a certain distance on the wall that stagnation or circulated flow (eddies) are in contact with the wall. Hence the temperature of this region decreases sharply from (stagnation value) to the minimal then increases gently along the rest of distance on the wall.

Materials Research Proceedings 31 (2023) 564-570 https://doi.org/10.21741/9781644902592-58



a) The Three Regions b) Enlargement near stagnant point

Fig. 2: Temperature distribution on the adjacent wall

Distance between the crack and the adjacent tank is also of interest and is addressed in the present work. Three distances are selected between the two vessels (18 cm, 30 cm, and 500 cm).

A dimensionless temperature parameter for the adjacent wall is plotted for three different distances and for the case of very high heat transfer coefficient inside the adjacent tank. As shown in Fig. 3, when the distance between the two adjacent walls is short (i.e. stronger impinging), a non-uniform temperature variation develops. Such variation decays as the distance becomes larger because the impact of the impinging jet is proportion inversely to the distance. Hence, it gradually diminishes.

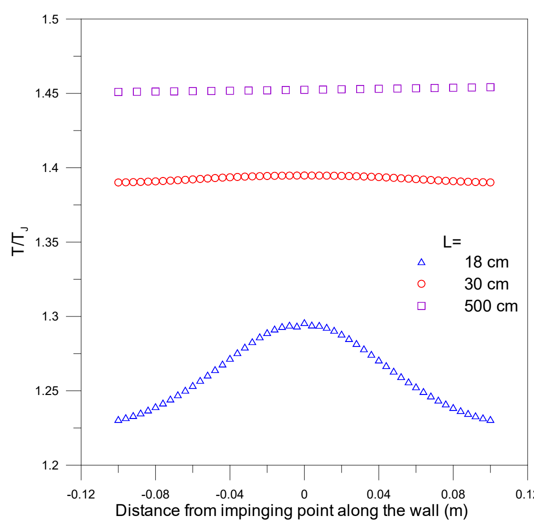


Fig. 3: Temperature Distribution along the Adjacent Wall when the inside Wall Temperature=265K

FE analysis and investigations of adjacent surface

Abaqus implicit software was exploited to perform steady-state coupled temperature displacement calculations to investigate the cold jet effect on an adjacent pressure vessel. The conceived axisymmetric model for pressure vessel (with standard dimensions [10]) was meshed with 4 nodes thermally coupled axisymmetric quadrilateral, bilinear displacement and temperature, reduced integration, hourglass control- CAX4RT elements, as shown in Fig. 4. The thermomechanical properties of the simulated pressure vessel material; 34CrMo4 Chrome Molybdenum steel, are shown in Table 1.

Table 1 Thermomechanical Properties of 34CrMo4 Chrome Molybdenum steel [11]

Properties	Thermal Conductivity	Thermal Expansion (µm/m-°C)	Specific Heat (J/g-°C)	Density (g/cm³)	Modulus of elasticity (GPa)	Poisson's Ratio
	30-60	12	0.46	7.8	210	0.3

Figure 5, shows the von-Mises stresses, nodal temperatures and thermal strains profiles on the pressure vessel with and without considering cold jetting. It can be noticed that by implementing thermal loading in the form of cold jetting the stresses are increased, though still are well below yield strength of the material [12]. While, temperature variation through the cylinder thickness are also evident that has caused thermal strains. In the light of the numerical results, it can be stated that under this cold jetting, though there will not be any considerable effects on the cylinder strength and safety, however a cylinder with lower thickness may result in localized softening of the material leading to rupture and failure.

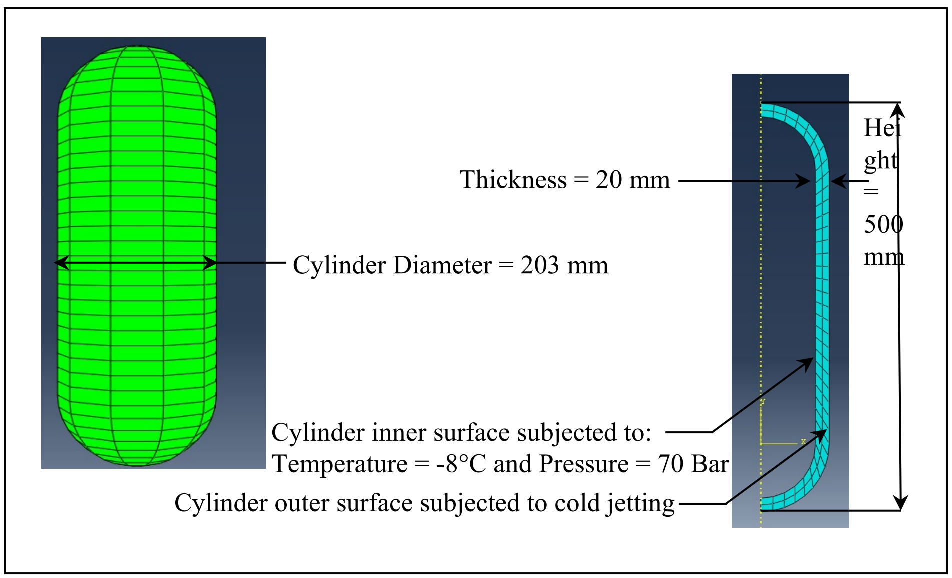


Fig 4: Conceived pressure vessel axisymmetric model geometry and boundary conditions.

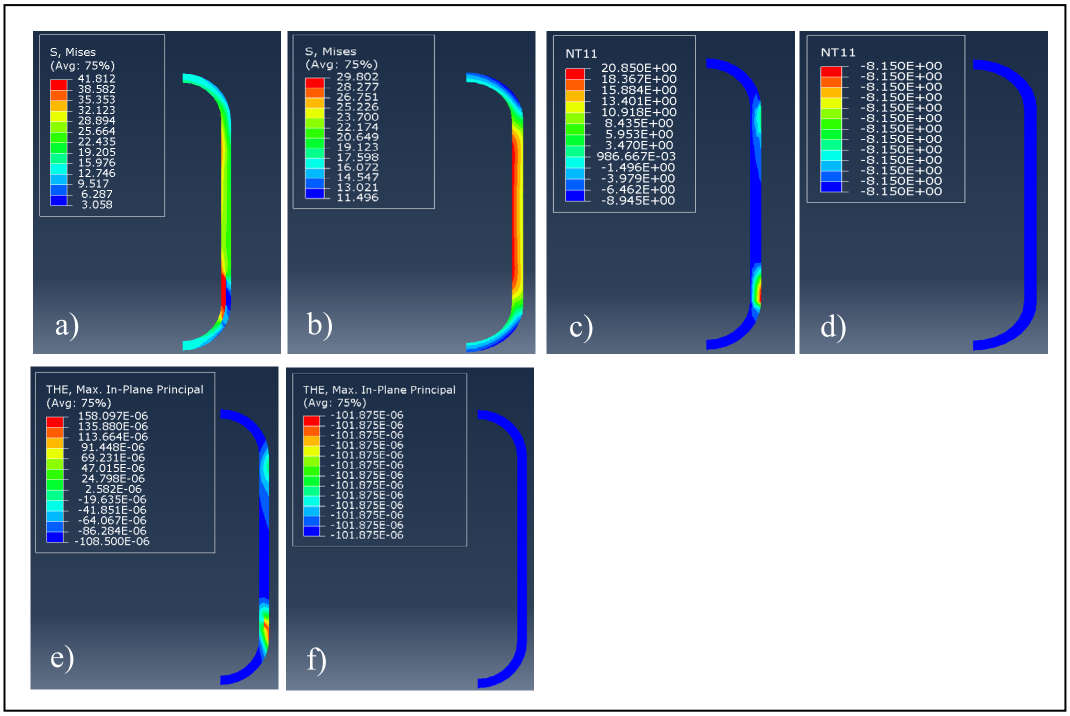


Fig 5: Von-Mises stresses (a, b), nodal temperatures (c, d) and thermal strains (e, f) profiles on pressure vessel with and without considering cold jetting effects, respectively.

Conclusion

In this paper, the jet evolution from ruptured high-pressure gas cylinder is modeled to study its impact on the adjacent high-pressure-vessel. Influence of the distance between the jet source and the adjacent part on the temperature distribution is addressed. Thermal strains of the adjacent surface due to the impingement are also reported. It is concluded that the jet starts supersonic and slows down to subsonic regime. The variation of the surface temperature of the adjacent vessel revealed that three sub-regions can be recognized with steep variation in temperature near the stagnation domain and smooth variation at the area far from the stagnant point. When the distance between the hole rupture and the adjacent surface augments, the sharp variation (the second sub region) diminishes.

Results of numerical simulation revealed that cold jet impingement could incept extra stresses that may result in localized softening of the material leading to rupture and failure. Hence caution must be taken when designing the thickness of the adjacent equipment, particularly if the equipment is a high-pressure container.

References

[1] Esam I. Jassim, 'Geometrical Impaction of Supersonic Nozzle on the Dehumidification Performance During Gas Purification Process: An Experimental Study', Arabian Journal for Science and Engineering, 44 (2019), pp.1057-1067. https://doi.org/10.1007/s13369-018-3340-x

[2] Jassim, E. 'Experimental Study on Dehumidification Performance of Supersonic Nozzle', World Academy of Science, Engineering and Technology, Open Science Index 121, International Journal of Aerospace and Mechanical Engineering (2017), 11(1), 182 - 185.

[3] Jassim, E.I. (2016) 'CFD study on particle separation performance by shock inception during natural gas flow in supersonic nozzle', Progress in Computational Fluid Dynamics, Vol. 16, No. 5, pp.300-312. https://doi.org/10.1504/PCFD.2016.078755

[4] Jassim E.I., Awad, M.M. (2013), 'Numerical investigation of nozzle shape effect on shock wave in natural gas processing', In Proceedings of World Academy of Science, Engineering and Technology (Vol. 78, p. 326). World Academy of Science, Engineering and Technology (WASET).

[5] Jassim E., Abedinzadegan Abdi M., and Muzychka Y., 'Computational Fluid Dynamics Study for Flow of Natural Gas through High Pressure Supersonic Nozzles: Part 1- Real Gas Effects and Shockwave', Journal of Petroleum Science and Technology, Vol. 26, issue 15, 1757-1772, 2008. https://doi.org/10.1080/10916460701287847

[6]. Jassim E., Abedinzadegan Abdi M., and Muzychka Y., 'Computational Fluid Dynamics Study for Flow of Natural Gas through High Pressure Supersonic Nozzles: Part 2- Nozzle Geometry and Vorticity', Journal of Petroleum Science and Technology, Vol. 26, issue (15), 1773-1785, 2008. https://doi.org/10.1080/10916460701304410

[7] H. Wilkening, D. Baraldi, 'CFD modelling of accidental hydrogen release from pipelines', International Journal of Hydrogen Energy, Volume 32, Issue 13, 2007, Pages 2206-2215. https://doi.org/10.1016/j.ijhydene.2007.04.022

[8] Yajun Deng, Hongbing Hu, Bo Yu, Dongliang Sun, Lei Hou, Yongtu Liang, 'A method for simulating the release of natural gas from the rupture of high-pressure pipelines in any terrain', Journal of Hazardous Materials, Volume 342, 2018, Pages 418-428. https://doi.org/10.1016/j.jhazmat.2017.08.053

[9] Dharavath, M., Sinha, P. and Chakraborty, D. (2010) 'Simulation of supersonic base flow: effect of computational grid and turbulence model', Proceedings of IMechE in Journal of Aerospace Engineering, pp.311-319. https://doi.org/10.1243/09544100JAERO600

[10] https://www.yongancylinder.com/13-4L-ISO11439-Standard-High-Pressure-CNG-1-Vehical-Seamless-Steel-CNG-Cylinder-pd43004559.html

[11]https://www.matweb.com/search/DataSheet.aspx?MatGUID=638f2c3469bc46f3b4ef0ebd06867ce6&ckck=1

[12] https://virgamet.com/35hm-34crmo4-1-7220-35crmo4-34cd4-aisi-4135-structural-steel