International Journal of Engineering Sciences &Research Technology

(A Peer Reviewed Online Journal)
Impact Factor: 5.164

IJESRT

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

NUMERICAL MODELING OF DIRECT CONTACT CONDENSATION OF VERTICALLY INJECTING STEAM IN STAGNANT WATER

Jibin Y George*[1], K E Reby Roy[2], Sanjeev Krishnan[3] & Satheesh Chandran C.[4]
*Department of Mechanical Engineering, TKM College of Engineering, Kollam, INDIA

DOI: 10.5281/zenodo.2629208

ABSTRACT

Direct Contact Condensation is a multiphase phenomenon where the gaseous phase comes into direct contact with the liquid phase and gets condensed and mixed with the liquid phase. DCC is a natural phenomenon and is being used in many industrial applications like the steam depressurization in boiler, food industry etc. The difficulties in analytical research due to the increased turbulence and other obstructions in the two phase flows, the complete information about the DCC of steam is even now not fully explored. In this paper, a new model for the DCC of steam in stagnant water bath was modeled using commercial software Ansys Fluent. And using this model, all the information and details regarding the DCC of steam can be unveiled. This model is validated against the steam plume shape and length obtained from the experimental results for ensuring its reliability. This model could replicate the other features of DCC of steam like chugging also.

KEYWORDS: Steam condensation, two phase flow, VOF model, steam plume length, chugging.

1. INTRODUCTION

Direct contact condensation is a two phase phenomenon and the condensation occurs at the interface of two phases namely gas and liquid. Through this interface various thermodynamic and physical properties are being exchanged. Through the introduction of DCC, there would be a great advantage that the fouling, corrosion and other effects which increase the maintenance costs can be eliminated and hence the overall energy efficiency can also be maintained. Studies are being happened in this area because of its wild acceptance and interest among the researchers. Since DCC of steam in water is a two phase phenomenon and because of the increased turbulence in the vicinity of condensation, the visualization, understanding of the mechanism, pressure variation, condensation rate, steam plume shape, steam plume length in direct contact condensation of steam is very difficult in nature. As a result most of the details and parametric studies are veiled by these obstructions. There is the need for the introduction of simulation of the DCC case of steam. In order to do a simulation based analysis, it is important to have a model by which the conditions and cases can be validated and get reliable and accurate results. Direct Contact Condensation (DCC) is the phenomenon of phase change from gaseous phase to liquid phase. One of the first experimental studies under taken to find the properties of DCC is the investigation done by Kerney et al. (1972). After that many researchers showed their interest in this field. But the accurate modeling of the multiphase transition is till now out-of-reach because of the complexity in the physical behavior of the phase transition. The inclusion of two different phases in a single system and its vigorous transition at the interface makes the analytical and numerical modeling of the DCC phenomenon as a challenging task. So the experimental study and the numerical modeling of such flows are essential for the better understanding of the flow behavior and to optimize the performance of such engineering devices.

Mainly the previous studies on DCC can be categorized into three. Studies which focuses on the steam jet condensation length and its correlation, studies which focuses on the heat transfer coefficient and its empirical correlation and studies that look into the effect of water bath temperature or rate of sub cooling on DCC.Kerney*et al.* (1972) studied the characteristics of sonic jet in an experiment using a horizontal submerged injection. They developed a correlation for the jet length for pool temperatures ranging from 28°C -85°C and

ISSN: 2277-9655
Impact Factor: 5.164
CODEN: IJESS7

were able to show that the jet length is proportional to the square root of the injection rate and inversely proportional to the sub-cooling. Weimer *et al.* (1973) studied DCC phenomenon in a cylindrical tank with horizontal steam injection and developed correlations for the vapor penetration distance and also corrections for the external expansion of the choked flow beyond the nozzle exit. Chan & Lee (1986) studied the downward steam injection into a sub cooled water bath at low and large steam mass fluxes. Based on the observations, they developed a condensation regime map with the water bath temperature in x coordinate and the steam mass flux which is the driving potential on the y axis. From their observations, an oscillatory jet is observed for high rate of steam injection (>125 kg/m^{2}s) and for low rate of steam injection (<50 kg/m^{2}s), a phenomenon called steam chugging is observed where the periodic reversal flow of pool water into the steam nozzle. Shukla *et al.* (2015) analyzed the DCC phenomenon of steam in water bath numerically with VOF multiphase and standard K-ε turbulence model. They studied the hydrodynamic and thermal characteristics of the pool for varying pool temperature and steam injection velocity. From their work, it is inferred that the interfacial temperature will be decreased by increasing the steam velocity and hence the enhancement of condensation rate. Many researches were carried out to study the chugging phenomenon during the injection of steam into water bath. H. Nariai*et al.* (1983) studied this chugging phenomenon and developed an oscillation classification map considering the steam mass flux and pool temperature. In their work they observed high frequency oscillation (large chugging) for lower pool temperatures. Later in 1986, they demonstrated a detailed linear frequency analysis of oscillation pattern. An analytical model oscillation pattern was also conducted by Ali *et al.* (2007). Recently, due to the complexity of understanding this DCC phenomenon, researches are being carried out numerically using Computational Fluid Dynamics. Meier *et al.* (2000)and G. Yadigaroglu (2000) had numerically analyzed the chugging behavior without incorporating mass transfer. Later Thiele (2010) considered the mass transfer effect on his VOF simulations. Laine*et al.* (2010) numerically analyzed an experiment conducted in Lappeenranta University of Technology (LUT). The analysis done in Euler-Euler multiphase model of CFD resulted with weak heat transfer and condensation rate. They suggested scope for the improvement of this model due to the complexity in bubble formation and detachment. Later Tanskanen (2012) studied the chugging phenomenon of DCC and predicted interfacial heat transfer using Euler-Euler multiphase model in 2D axisymmetric domain. Through this study, he could generate a pattern recognition approach which can interpret the condensation rate from the video material of a test conducted under chugging-condensation model and later on the study of Tanskanen*et al.* (2012) for the bubble size (ellipsoidal for fully inflated bubble) and chugging frequency, it can be inferred that the bubble size and chugging frequency is increased with the increase of pool temperature. On the numerical study of Li *et al.* (2015), they have used the VOF multiphase model with LES turbulence model. They validated the experiment conducted by Chan & Lee (1986) and resulted with the chugging and condensation phenomenon as illustrated in the experiment. With the UDF they have incorporated in the simulation, they have observed that the pressure fluctuation in the system has great dependency on the steam injection velocity, pressure of the pool and rate of condensation.

2. NUMERICAL METHODOLOGY

Model development

In this study, the simulation of steam jet injection through a vertical nozzle into a sub-cooled water bath is done with the commercial software Ansys Fluent 19.

Doing a simulation model for the DCC phenomenon is helpful to obtain many details which can’t be obtained through the experiment due to the multiphase phenomenon in DCC. So the development of such a simulation model has much demand in the present scenario. Since the main objective of this simulation model is to validate the model developed with experiment data and also to conduct the simulation of test conditions which couldn’t done experimentally, the domain chosen is as same as the experiment set up.

Computational Model

In this study, a simulation of a steam jet injected into a stagnant pool of sub cooled water through a vertical pipe is carried out using commercial CFD software ANSYS FLUENT 19. Fig. 1 depicts the geometry of the computational domain. The domain is selected as same as the experimental domain with which it is planned to validate the numerical results. A 2D Axisymmetric numerical model has been developed using commercial software to study the steam injection in a cylindrical pool filled with sub cooled water. Geometry consists of a cylindrical pool of height 435 mm and diameter of 560mm. A blow down pipe is submerged in water pool vertically through the axis of the tank at the top. Height and diameter of blowdown pipe is 185mm and 4mm respectively.

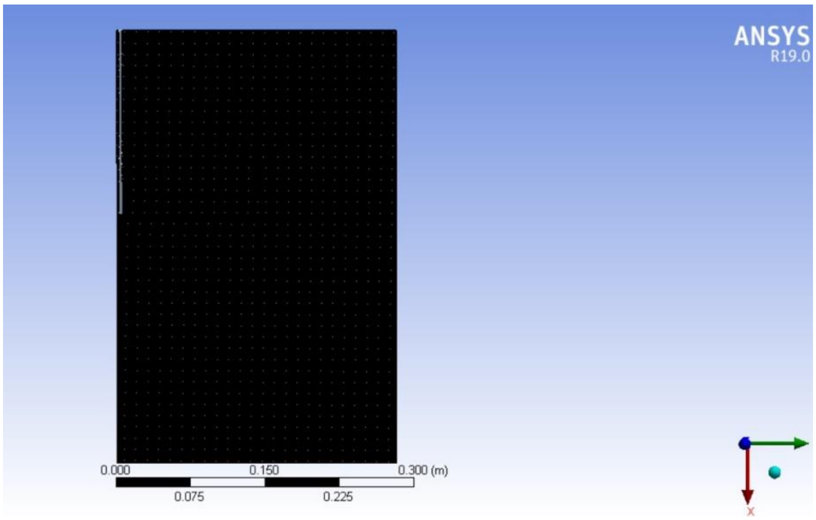

Fig. 1.Axisymmetric fluid simulation model

Simulation settings and boundary conditions

The governing equations used for the development of the model are mass conservation equation, momentum transfer equation, energy conservation equation, turbulence model, condensation and evaporation model, and interfacial heat and mass transfer coefficients. In condensation process, the interfaces keep oscillating and continuously changes. Hence, Eulerian-Eulerian based Volume of Fluid (VOF) technique for two-phase modeling is used to simulate the phenomenon. A detailed description of Eulerian-Eulerian based Volume of Fluid (VOF)multiphase flow model can be referred to Fluent manuals(2019). For simulating the condensation the evaporation condensation model which is based on the Lee model was used. Realizablek–ε turbulent model was used. Simple algorithm was employed for pressure-velocity coupling. Gravity effect was taken into account. Steam was treated as a compressible fluid using idealgas law for calculating its density. The other properties of steam such as viscosity, thermal conductivity, specific heat capacity andso on were taken from Fluent database and assumed to be constant during the simulation.

Boundary conditions

The domain is having one inlet section, the steam jet inlet. In steam inlet, it is enforced with the pressure inlet boundary condition. The temperature of the steam is given as 117° by considering the pressure and saturated steam conditions. Adiabatic condition with heat flux zero is given at the wall boundaries of both blowdown pipe and water bath and considered the wall as stationary wall with no slip condition. The top area of the tank which is opened to the atmosphere is assigned with pressure outlet condition and the pressure is also mentioned at outlet which is the atmospheric pressure.

Method of solution

As the flow phenomenon is dynamic in nature, variation of the flow phenomena both in time and space has been considered. Continuity equation has been discretized by PRESTO. Volume fraction, turbulent kinetic energy and dissipation rate equation are discretized by first order upwind method. While momentum and energy equations are discretized by second order upwind method. SIMPLE algorithm is used for pressure velocity coupling. The momentum and energy were solved using the second order upwind discretization scheme. The simulation is carried out for 10 s with a time step of 0.000005 s and with convergence criteria that residuals falls below 10^{-4}.

3. RESULTS AND DISCUSSION

The results of the simulation study and its validation with the experimentally obtained data are listed in this section. After validation the effect of different parameters on the DCC is evaluated. The parameters studied for this is the pool water temperature. From previous researches, it is evident about the presence of four regions in the DCC of steam in water which are the pure steam plume, interface between steam plume and water, hot water layer and ambient water region. In the simulation case also, the mentioned regions in DCC can be clearly seen. Before the analysis, it is to be noted that the condensation of steam in water is not a sudden process. The interface is the region where the steam and water co exists. Hot water region is in between the ambient water and interface which is relatively at a higher temperature than the ambient.

Fig. 2. shows the experimentally obtained image of direct contact condensation of steam in stagnant water. The image was selected from the stable condensation images out of the images captured during the test. In the test the image capturing frequency was 1000 frames per second and hence the image of condensation can be analyzed in the millisecond range.The mentioned experiment was conducted at LPSC, ISRO, Valiamala, India as a study for the direct condensation of steam in DM water at 40℃ and the steam nozzle exit is 4mm. The pressure noted at the top of the steam injection portion was 1.8 bar and temperature of the plume noted was 117℃.As seen in the image, the steam plume, interface between steam and water, hot water layer, ambient water can be clearly seen from the experimentally obtained image.

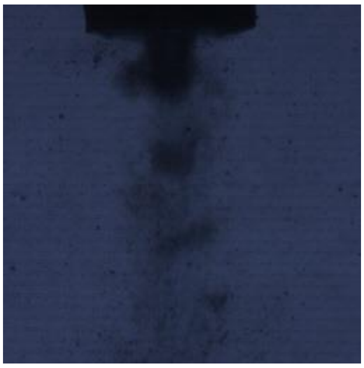

Fig. 2. Experimental image of DCC of steam in water

Fig. 3. Binary image for edge detection

Processed image for the edge detection is shown in the fig.3. The image processing is done with MATLAB by converting the RGB image to 256 level gray scale and then converting to binary mode. By converting to binary mode, the area with luminosity less than the threshold value will be seen as black and the rest in white. Here the black color represents the plume area with .95 and above steam volume fraction and the white area represents the background with DM water.

From the image, the length of plume is obtained using the calculation;
$l = r_p$x N_p;

where l is the steam plume length, r_p is the ratio of real length to the corresponding number of pixels in the image (ie, real length indicated by one pixel in image). r_p is obtained since the outer diameter of the nozzle is known (15 mm) and corresponding number of pixels are obtained from the image. Therefore $r_p$is equalto the ratio of nozzle actual outer diameter to the sum of pixels of the nozzle outer diameter. N_p is number of pixels for the longest line, starting from centre of the nozzle exit plane, in steam plume region.

Simulation Results

The volume fraction contour of steam obtained from the simulation model is shown in the fig. 4. In the3 figure, as the legend shows, the red color represents the steam region and blue color represents the water region. Since the condensation process is not a sudden process, after the interface, the region is with high density of vapor bubbles but the volume fraction of steam in that region is low compared to the pure steam plume. So the red color extends downward but its not pure steam plume. For the better understanding, the contour is clipped to the required volume fraction of steam and the same is shown in the fig.5

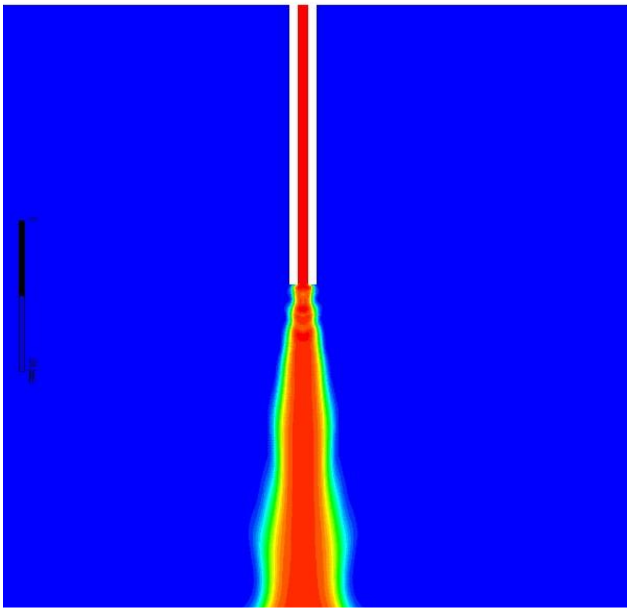

Fig. 4. Volume fraction contour obtained from simulation

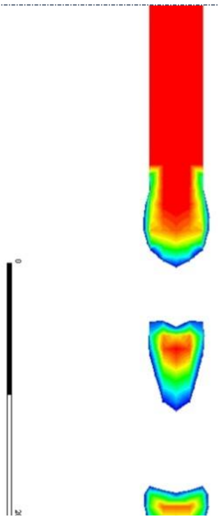

Fig. 5. Volume fraction contour clipped to 0.95 to 1

From fig. 3 and fig. 5, it is clear that the contour obtained from the simulation model under the same conditions of experiment is same as the image obtained from experimental results. Now one more validation that can be done is the validation length of plume in both CFD simulation and experimental image. By observing the pattern in experiment and simulation cases, it is similar about the steam plume detachment that happening intermittently. This observation in both these simulation and experiment cases can be stated as when the steam ejects from the nozzle exit, the low temperature water from surrounding gets inside the steam plume and condensation of steam happens with a reduction in pressure at the condensation regime. Then as the steam ejects out further from nozzle exit, the pressure builds up and then the surrounding water entraps the steam plume and detachment of steam plume happens and the cycle continues.

From the image the length of plume is obtained using the calculation;
$l = r_p x\ N_p$;

where l is the steam plume length, r_p is the ratio of real length to the corresponding number of pixels in the image (ie, real length indicated by one pixel in image). r_p is obtained since the outer diameter of the nozzle is known (15 mm) and corresponding number of pixels are obtained from the image. Therefore $r_p$is equalto the ratio of nozzle actual outer diameter to the sum of pixels of the nozzle outer diameter. N_p is number of pixels for the longest line, starting from centre of the nozzle exit plane, in steam plume region.

From fig. 3, the length of steam plume is obtained as 6.55 mm while from fig. 5, the steam plume length obtained from simulation result is 7.08. i.e., a percentage difference of 7%, which means the simulation result is very close to the experimental result in measurement wise and pattern wise.

One of the main observations with DCC of steam in water is the chugging phenomenon. This phenomenon has been statedby many researchers in their studies. Fig. 6 shows the chugging phenomenon observed experimentally with inlet pressure of 1.8 bar and 4 mm steam nozzle inner dia at 40°C. Chugging is a phenomenon characterized by the frequent oscillation of water column in vent tube. Mainly the chugging is occurred with the low steam mass flux and low temperature of water bath or larger sub-cooling. The chugging phenomenon can be explained as follows. Due to the vigorous condensation of steam plume at the interface between steam and water as it enters the water bath, a negative pressure will be created at the top of the nozzle. As a result the water from the bath enters the blow down pipe or nozzle and when the interface is inside the nozzle, the condensation ceases due to the formation of temperature boundary layer at the water side around the interface inside the nozzle. The interface is then pushed downward towards the water bath by the recovered steam pressure inside the nozzle.

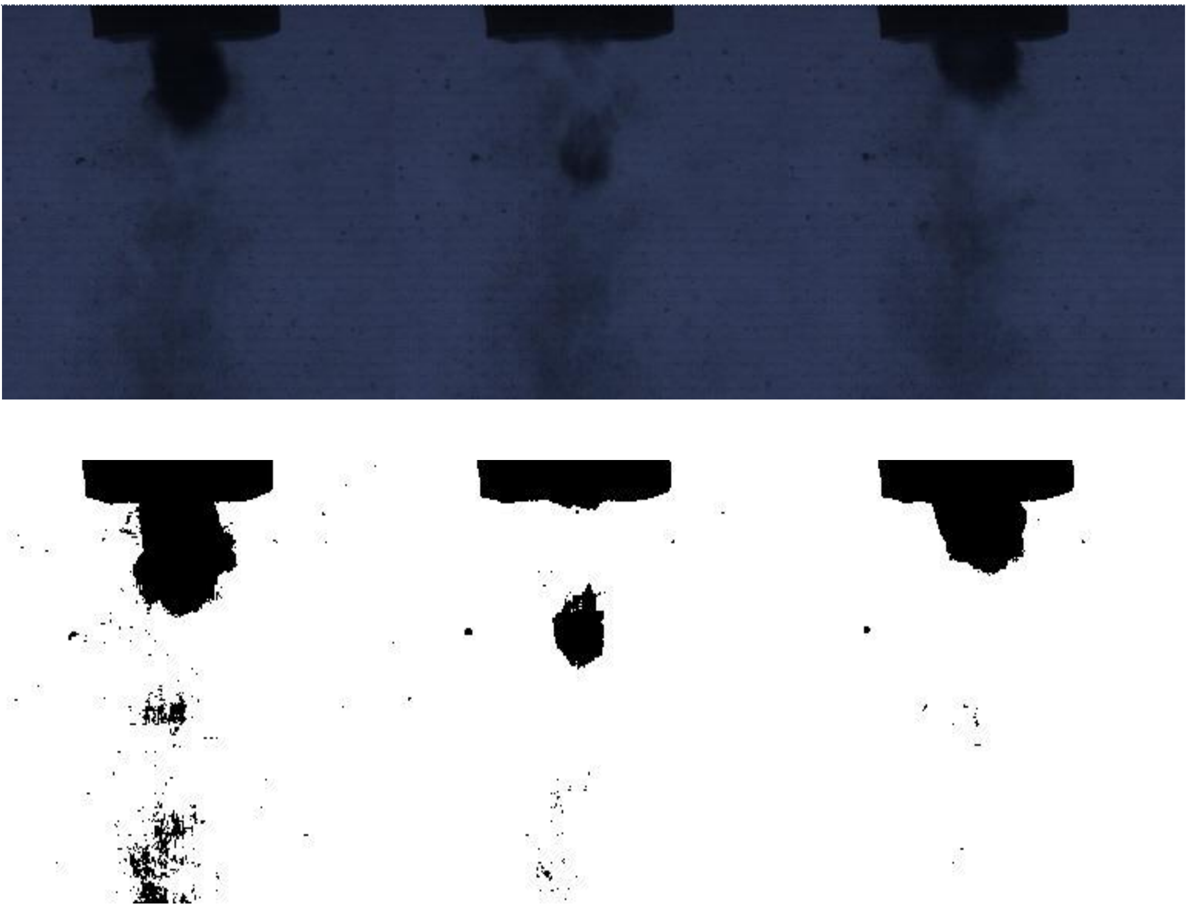

Fig. 6. Chugging phenomenon observed in experiment

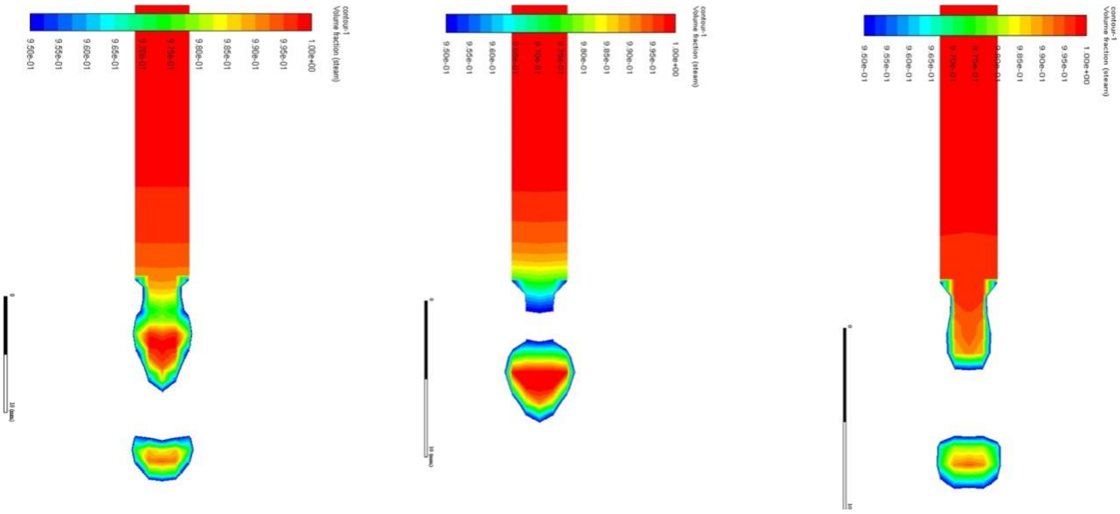

Fig. 7. Chugging phenomenon replicated by simulation

From fig. 6 and Fig. 7, it can be inferred that the experimentally observed chugging phenomenon and the replication of the same through the simulation model holds good match with the shape and pattern. All these similar observations in results obtained from this simulation model with the experimental observations suggest that this simulation model holds good reliability for the direct contact condensation phenomenon of steam in stagnant water bath. So this validation of simulation model with the experimental data confirms the accuracy and reliability of results with various parameters. With the same model, the simulation was done with water bath temperatures of 30°C and 50°C.

The radial temperature distribution in the domain at an axial distance of 188 mm from the top of the cylinder water bath is plotted in the fig. 8.For better clarity, the plot was made with maximum radius of 15 mm from the axis. For rest of the radius also, the temperature measurement of domain is constant. From the figure, it is seen that as the radial position from the point of steam injection (axis of domain) increases, the temperature gradually decreases. This same trend can be seen for all cases with different initial pool temperatures, T_w.

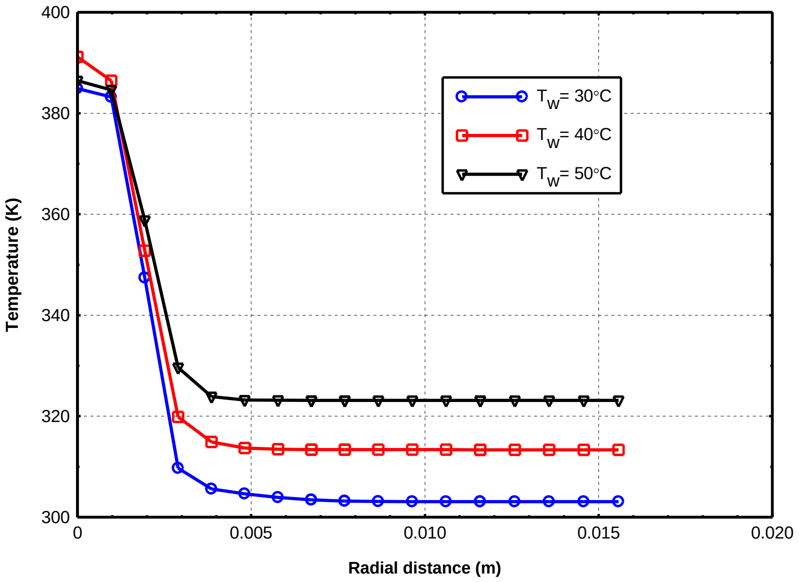

Fig. 8. Radial temperature distribution

4. CONCLUSION

The Simulation model for DCC of steam in stagnant water has been successfully developed using commercial software Ansys Fluent 19. The reliability of the simulation model was confirmed by validating the simulation results with the images captured during experiment under the same conditions. In validation, the simulation was validated with the shape and pattern of steam condensation in stagnant water, steam plume length for the specific condition and the simulation could replicate the chugging phenomenon observed in the experiment. Since its reliability has been confirmed, the same model was used to simulate the conditions with varied bath temperature and the graph for radial temperature distribution was plotted for each condition.

5. ACKNOWLEDGEMENTS

The authors wish to thank Head and Director, Propulsion Research & Studies (PRS) entity LPSC, ISRO, Valiamala, Thiruvananthapuram for permitting this work to be done with the help of computing machines in the high performance computing facility. Authors would like to thank all the members and to all others who have directly or indirectly helped in carrying out this work.

REFERENCES

[1] Kerneyet al., “Penetration characteristics of submerged jet”, in proc. AIChE Journal, 1972
[2] H. Nariai, I. Aya, “Threshold of pressure and fluid oscillations induced by injection of subcooled water into steam flow in horizontal pipe”, in proc. ASME-JSME, 417–424, 1983.
[3] H. Nariai, I. Aya, “Fluid and pressure oscillations occuring at direct contact condensation of steam flow with cold water”, in proc. Nucl. Eng. Des. 95, 35–45, 1986.

[4] S.M. Ali, V. Verma, A.K. Ghosh, ”Analytical thermal hydraulic model for oscillatory condensation of team in presence of air”, in proc.Nucl. Eng. Des. 237, 2025–2039, 2007.
[5] M. Meier, G. Yadigaroglu, M. Andreani, “Numerical and experimental study of large steam-air bubbles injected in a water pool”, in proc. Nucl. Sci. Eng. 136, 363–375, 2000.
[6] R. Thiele, “Modeling of Direct Contact Condensation WithOpenFOAM Master thesis”, KTH, Royal Institute of Technology, Division of Nuclear Reactor Technology, Royal Institute of Technology, Stockholm, Sweden, ISSN: 0280-316X, 2010.
[7] J. Laine, M. Puustinen, H. Purhonen, “CFD modelling of condensation of vapour in the pressurized ppoolex facility”, CFD4NRS-3, 14–16 September, Washington D.C., USA, 2010.
[8] ANSYS Fluent 19 User’s Guide, Fluent Inc., Lebanon, USA, 2019.
[9] V. Tanskanen, CFD modelling of direct contact condensation in suppression pools by applying condensation models of separated flow, in: ActaUniversitatisLappeenrantaensis, 2012, 472. Lappeenranta University of Technology, ISBN: 978-952-265-221-8, ISBN: 978-952-265-222-5 (PDF), ISSN: 1456-4491.
[10] V. Tanskanen, A. Jordan, M. Puustinen, R. Kyrki-Rajamaki, “CFD simulation and pattern recognition analysis of the chugging condensation regime”, Ann. Nucl. Energy 66, 133–143, 2014.
[11] S.Q. Li, P. Wang, T. Lu, “Numerical simulation of direct contact condensation of subsonic steam injected in a water pool using VOF method and LES turbulence model”, Prog. Nucl. Energy 78, 201–215, 2015.
[12] Satyajit Kumar Shukla, A.M. Naveen Samad, Sumana Ghosh, “CFD simulation of steam condensation in a subcooled water pool”, Thermal Science and Engineering Progress 2, 80–86, 2017