Proceedings of the 5th World Congress on Momentum, Heat and Mass Transfer (MHMT'20)
~~Lisbon, Portugal~~ Virtual Conferences – October, 2020
Paper No. ICMFHT 184
DOI: 10.11159/icmfht.184

Numerical Simulation of A Radial Free Surface Liquid Jet Impinging on A Heated Surface

[1]LIKITHA.S, [2]PATRICK.S, [3]ALOK.K, [4]AMINE.B, [5]VINAY.G,
[1,3,5]Ansys Software Pvt. Ltd.
Pune, India
[1]slikitha.siddanathi@ansys.com, [3]alok.khaware@ansys.com, [5]vinaykumar.gupta@ansys.com
[2]Ansys Milton Park
Pune, India
[2]patrick.sharkey@ansys.com
[4]Ansys Germany GmbH
Pune, India
[4]amine.benhadjali@ansys.com

***Abstract* -** Impinging liquid jets have been demonstrated to be an effective means of providing high heat transfer rates, and widely used in designing cooling systems for electronic modules, plastic manufacturing, and many other applications in the industry. It is very important to study the factors which govern the heat transfer rate in the liquid impingement on a heated surface to ensure cooling efficiency. The paper presents a numerical approach to study the convective heat transfer of circular liquid jet impingement on a heated surface where influencing factors like surface tension, gravity, viscosity, surface temperature etc. are considered. Finite volume method (FVM) with pressure based coupled solver implemented in commercial ANSYS Fluent CFD is used to solve Reynolds Averaged Navier Stokes equations. Free surface flow is modelled using Volume of Fluid (VOF) Method along with the compressive scheme and sharp interface modelling which accurately captures interfaces between immiscible fluids. The liquid film formation and the heat transfer phenomenon are examined in detail. The influence of jet velocity profiles on pressure distribution and heat transfer along the heated surface is presented. The results obtained from numerical solution are validated against experiment and previously published work with a close match.

***Keywords*:** Impinging Liquid Jet, Heat transfer, Volume of Fluid