Power Division

Simulation of Critical Heat Flux in Vertical Tubes

Interim Report

Prepared By:
Michael Seibert, Ph.D.

June 2020

UNITED STATES ARMY FUTURES COMMAND (AFC), COMBAT CAPABILITIES DEVELOPMENT COMMAND (CCDC) C5ISR CENTER, COMMAND, POWER, & INTEGRATION (CP&I) DIRECTORATE, ABERDEEN PROVING GROUND, MD 21005-1852

This page is intentionally left blank.

CONTENTS

INTRODUCTION 4
PURPOSE 4
SIMULATION SETUP 4
SIMULATION RESULTS 4
OBSERVATIONS & CONCLUSION 6
APPENDIX 1: List of Settings 6
APPENDIX 2: Material Properties 7

REFERENCES:

1. Hoyer, N. "Calculation of Dryout and Post-Dryout Heat Transfer for Tube Geometry," Int. J. of Multiphase Flow 24(2) 1998 pp319-334
2. Li, H., Vasquez, S.A., Punekar, H., Muralikrishnan, R., "Prediction of Boiling and Critical heat Flux Using An Eulerian Multiphase Boiling Model," Proceedings of the ASME 2011 International Mechanical Engineering Congress and Exposition, November 2011 Denver, CO, IMECE2011-65539
3. Conversations with Surya Deb and John Ibrahim of ANSYS Technical Support

INTRODUCTION

Boiling/two phase heat transfer is an attractive cooling mechanism due to the possibility of very high heat transfer coefficients, giving high heat flux at low temperature difference. The drawback is the possibility of dry-out. This occurs when too much of the liquid is vaporized, and the remaining liquid can't rewet the surface to absorb the heat. The heat transfer coefficient suddenly decreases by an order of magnitude, causing the surface temperature to rapidly rise. This critical heat flux (CHF) condition damages the systems being cooled. Location of CHF is a function of geometry, heat flux, and fluid flow rate. Increasing the fluid flow rate can insure that critical heat flux does not occur; however, doing so increases the required pumping power and fluid mass. Correctly predicting the CHF location gives the minimum weight to meet cooling requirements.

Critical Heat Flux is modeled in ANSYS FLUENT and compared against two published papers [1,2] to confirm that the correct multiphase flow and heat transfer settings are being used. Hoyer used experiments and developed his own model[1] while Li validated the FLUENT Critical Heat Flux submodels against Hoyer's experiments[2].

PURPOSE

The purpose of these simulations is to confirm the correct settings used to compute the location of critical heat flux, which is necessary for electronics and laser cooling.

SIMULATION SETUP

The experiments to be modeled were conducted on a 7 m vertical tube. Heat was applied in either a uniform heat flux or as a function of height. The present simulations focus on the cases of uniform heat flux. The working fluid is water at 7 MPa. Properties of water used in the simulations are saturation properties at 7 MPa, 285°C (Appendix 2). Subcooling was not clearly specified by either Hoyer or Li, so the present work used 10°C.

Li stated that a two-dimensional, 32,000 cell mesh was grid independent. She did not state axial vs radial cell counts. In this study, the mesh used 16 cells radially and 2,000 cells axially. In computational fluid dynamics, it is commonly necessary to demonstrate grid independence using a finer and finer computational grid. This is not the case using FLUENT's critical heat flux model[3]. The model uses a sub-grid calculation to determine the volume fraction of gas in the cell adjacent to the wall. Wall cells are treated differently than interior fluid cells. If the wall cell is too small, the amount of vapor mass injected into it will cause numerical instability[3].

The single phase solution is solved first by setting the wall heat flux to zero. This solution is used as the initial condition when the wall heat flux is set to the desired level. Solving liquid only first is much more numerically stable than solving directly for the boiling solution.

SIMULATION RESULTS

Three different flow conditions are simulated (Table 1). The first case shows the best agreement between the present simulations and [1,2]. The temperature increases sharply at 4 m, as shown in Figure 1. While there is good agreement on the location of critical heat flux, the present simulations overpredict both wall and fluid temperature in the dry-out region. This may be due to different material properties for superheated steam, as mentioned above. Arguably, predicting the location of critical heat flux is more important than the temperature after it occurs.

	Mass flux	Heat flux	Diameter	Flux ratio	Critical Heat Flux height (m)		
Case	kg/s-m2	kW/m2	mm	kJ/kg	Hoyer	Li	Present CFD
1	1495	797	10	1492	4	4	4
2	1002	863	14.9	1618	4.2	4.1	3.7
3	497	350	10	1971	5.3	---	3.2

Table 1. Summary of case conditions

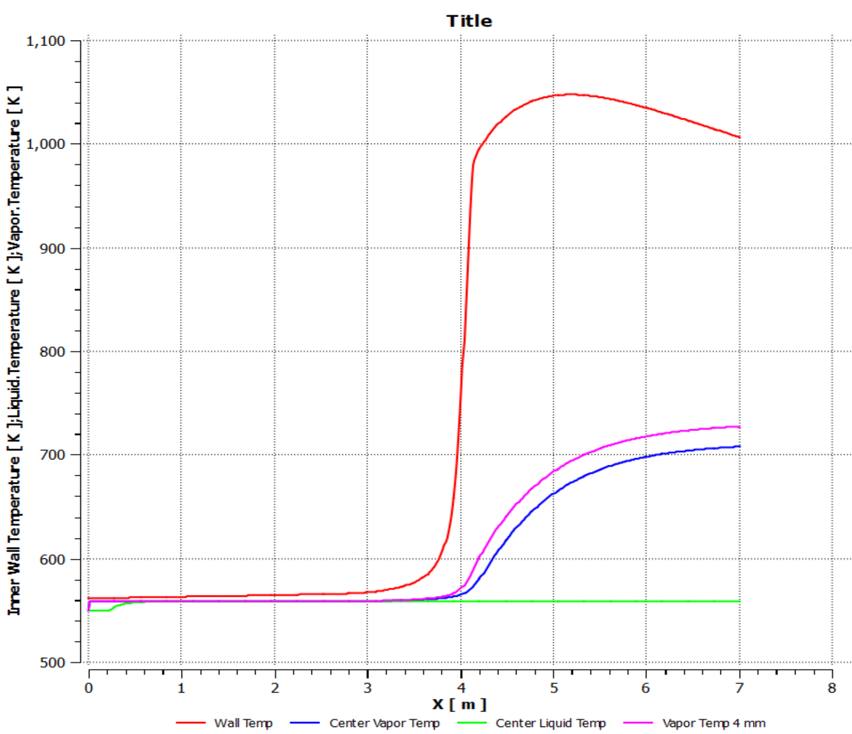

Figure 1. Temperature profiles for Case 1, 797 kW/m2, 1495 kg/m2-s

This flow rate in a 10 mm diameter tube produces a vapor Reynolds number of 786,000. The textbook correlations for turbulent flow produce Nu=1335, h=9078 W/m2-K, giving Twall-Tvapor=87 K. The present CFD gives Twall-Tvapor=300 K. It's possible that the coarse wall mesh necessary for the Critical Heat Flux model is too coarse to accurately capture the boundary layer for single phase convection.

For a given wall heat flux, increasing the mass flow rate moves CHF further downstream. For a given mass flow rate, increasing heat flux moves CHF upstream (sooner). Two cases can be compared using the ratio of heat flux to mass flux in kJ/kg. Higher heat to mass ratio moves critical heat flux sooner.

Cases 2 and 3 used higher heat to mass ratio, which moved the critical heat flux sooner in the present simulations. Curiously, both Li and Hoyer show it occurring later than Case 1 in both situations.

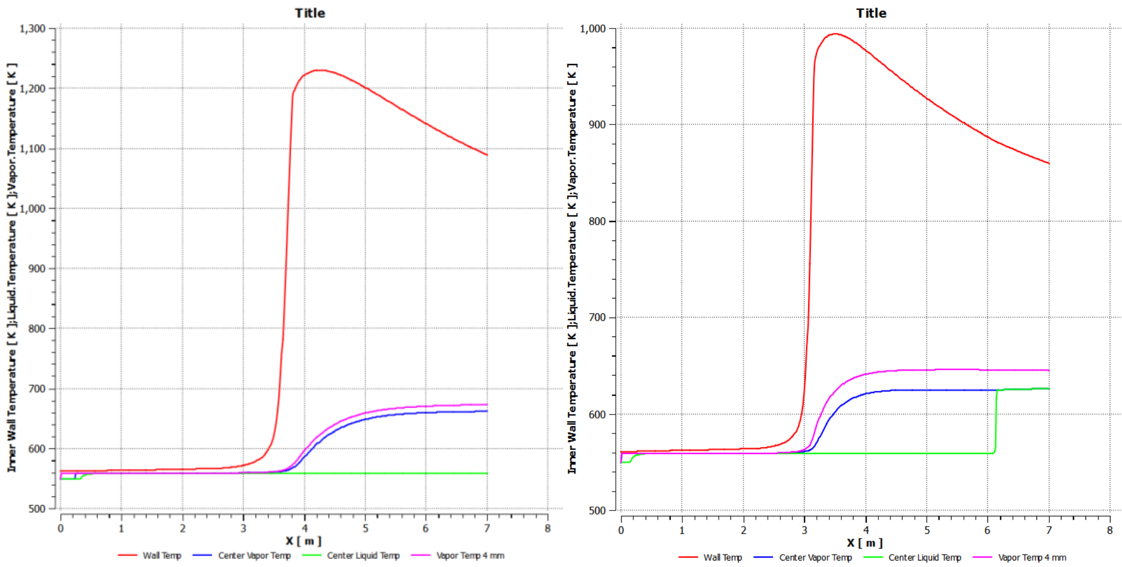

Figure 2. Case 2 and 3 temperature profiles.

OBSERVATIONS & CONCLUSION

The present simulations appear to accurately predict the location of Critical Heat Flux. However, post dry-out heat transfer and temperature modeling is not yet validated. The modeling approach has been validated for a vertical 10 mm diameter tube at high flow rate and Reynolds number. Further investigation is necessary to determine the range of applicability for the model. One likely limit is microchannels where the size of the geometry approaches the bubble departure diameter.

Other reports of experimental CHF measurements are available in the literature. If they contain sufficient detail to duplicate the geometry and operating conditions in CFD, the modeling approach can be tested against them, expanding the range of validated conditions. Simulations of devices in the C5ISR Center Laboratories will be compared to the device experimental performance.

APPENDIX 1: List of Settings

Operating conditions 7MPa, gravity -9.81 in X direction (flow direction is up)

Steady state, pseudo transient ("Pseudo transient" is selected under the Solution Methods button on the Solving tab.) Pseudo transient is numerically more stable but can take longer to converge. Set to timescale factor to 0.01. If the simulation terminates with errors, it may be necessary to reduce the timescale factor. Often a small timescale factor can be increased after 100 or 1000 iterations.

Viscous model: k-omega SST, production limiter, Turbulence Multiphase Model: Mixture

Multiphase: Eulerian, Boiling Model, CHF, Implicit volume fraction formulation

Phase Interactions:

Virtual mass modeling unchecked, coefficient 0.5

Drag: Ishii

Lift: Tomiyama (may have been a later addition)

Wall lubrication: None

Turbulent Dispersion: Lopez de Bertodano (constants) Without this setting enabled, gas stays at the wall and doesn't mix with the liquid. This creates problems, particularly when the gas temperature is fixed at saturation.

Turbulence Interaction: Troshko-Hassan

Heat: Two-resistance: Ranz-Marshall, Lavieville et. Al. Using Lavieville enabled vapor temp to increase above saturation. Options that fix vapor temperature at saturation are suitable for bubbly flow, but not convective boiling or droplet/mist flow.

Mass: Boiling "From liquid" and "to vapor." Set saturation temperature appropriate for conditions. When initially setting up the simulation, the default is "from phase 1" and "to phase 1." If this is not changed, the simulation will have errors which require a ctrl+alt+delete to exit FLUENT. The error messages do not obviously identify this as the problem.

Surface tension: Continuum surface force, 0.0176 N/m (Based on saturated water at 7 MPa)

Interfacial area: ia-symmetric

APPENDIX 2: Material Properties

Properties of Saturated Water at 7 MPa, 285°C

	Liquid	Vapor	
Density	740.2	36.54	kg/m3
Specific Heat	5000	4700	J/kg-K
Thermal Conductivity	0.57	0.068	W/m-K
Viscosity	0.000094	0.000019	kg/m-s (Pa-s)
Molecular Weight	18.0152	18.0152	kg/kmol
Standard State Enthalpy	22805000	49897000	J/kmol
Ref Temp	559	559	K

[Later conversations with ANSYS indicated that using the ideal gas law for vapor density is more stable. I have not compared the effect of ideal gas law density and density listed in saturation tables. Ideal gas law is not particularly accurate near saturation conditions; however, compressibility improves numerical stability].