ANS MC2015 - Joint International Conference on Mathematics and Computation (M&C), Supercomputing in Nuclear Applications (SNA) and the Monte Carlo (MC) Method · Nashville, Tennessee · April 19–23, 2015, on CD-ROM, American Nuclear Society, LaGrange Park, IL (2015)

INITIAL 1-D SINGLE PHASE LIQUID TRANSIENT VERIFICATION OF COBRA-TF

Chris Dances and Dr. Maria Avramova
Department of Mechanical and Nuclear Engineering
The Pennsylvania State University
137 Reber Building, University Park, PA, 16802, USA
A@cad39@psu.edu; B@mna109@psu.edu

Dr. Vince Mousseau
Computer Science Research Institute
Sandia National Laboratories
1450 Innovation Parkway, Albuquerque, NM 87123, USA
C@vamouss@sandia.gov

ABSTRACT

Abstract . . .

Key Words: List no more than five key words

CONTENTS

Contents 2

List of Figures 2

1 Introduction 3

2 Sine Wave Advection Problem Setup 4

3 Conclusions 5

4 Acknowledgments 5

5 References 5

LIST OF FIGURES

1 The finite volume structure for COBRA-TF 3

2 Enthalpy near the inlet and the analytical solution 4

3 Mass Flow rate near the inlet and the analytical solution 5

1 INTRODUCTION

[1]

For the past several decades, the primary focus in nuclear engineering within the United States has been focused on light water reactors (LWR). Commercially, all nuclear reactors are either boiling water reactors (BWR) or pressurized water reactors (PWR). Correct computation of the thermal hydraulics within the reactor core leads to effi- cient design and accuracy in the safety analysis. A popular subchannel code for modelling the hydrodynamics with in the reactor core is COBRA-TF. This FORTRAN based code solves 8 conservation equations for liquid, entrained droplet, and vapor phases in 3-D dimmensions [2]. The conservation equations analytically reduce into a pressure matrix in a semi-implicit method with rod temperatures solved for explicitly. Because the physics are integrated into the numerical solution, the equations must be linear and the solution method semi-implicit. With a residual formulation, greater flexibility and control over the numerical solution is possible. COBRA-TF was originally written in FORTRAN 77, but over the years has been partially updated to newer versions of Fortran.

The finite volume structure in COBRA-TF in figure 1 is for a one-dimmensional channel in the axial direction with n number of cells. The first and last cells at 0 and n + 1 are ghost cells and act as the boundary conditions for the problem. Pressure, enthalpy, and density are averaged over the cell volume and are located at the center of the cell. Mass flow rate and velocity are located at the faces in between cells. The cells are represented with an index i, and the faces with indexes of $i + \frac{1}{2}$ or $i - \frac{1}{2}$. This project will initially focus on this 1-D configuration. Usually the code is 3-D, with channels connecting to each other in two more dimmensions.

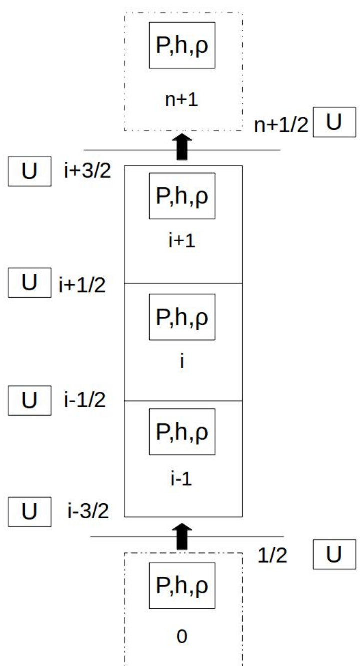

Figure 1. The finite volume structure for COBRA-TF

2 SINE WAVE ADVECTION PROBLEM SETUP

The problem is to transiently vary the inlet enthalpy h and inlet mass flow rate $\dot{m}$ using a smooth trigonometric function so as to keep velocity constant throughout the solution. Using a cosine, the analytical solution for a variable Y at time index j and space index i, where Y_1 is the initial value, Y_2 is the minimum value of the wave, and P is the period of the wave. The time step size dt, axial mesh size dx, and velocity V_o are assumed constant. If $V_o * j * dt > i * x$, then this equation doesn't apply and the value should just equal the initial value Y_1.

$$Y(i,j) = \frac{1}{2}\left((Y_1 + Y_2) + (Y_1 - Y_2)cos\left(\frac{2\pi}{P}\left(j * dt + \frac{i * dx}{V_o}\right)\right)\right) \tag{1}$$

This analytical solution can be applied to mass flow rate $\dot{m}$, density ρ, liquid enthalpy h, and liquid temperature T. Since velocity and pressure are cosntant, mass flow rate will be proportional to density, and enthlapy will be proportional to temperature.

For the inlet condition, a transient data table was generated for enthalpy and mass flow rate and applied at the inlet node. The comparison between the data table and the output in CTF are shown for enthalpy and mass flow rate in figures 2 and 3 respectively. The CTF output was read from hdf5 data files at each point in time, wich omitted the actual ghost cell where these values were applied. The CTF values are located at the nearest node to the inlet, and therefore will be slightly out of phase to the exact values in the figure. This difference is more notable for smaller mesh sizes.

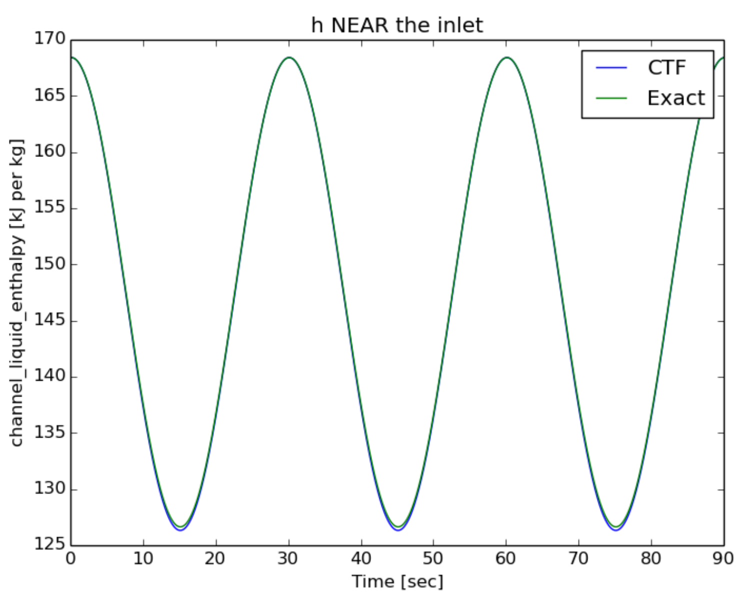

Figure 2. Enthalpy near the inlet and the analytical solution

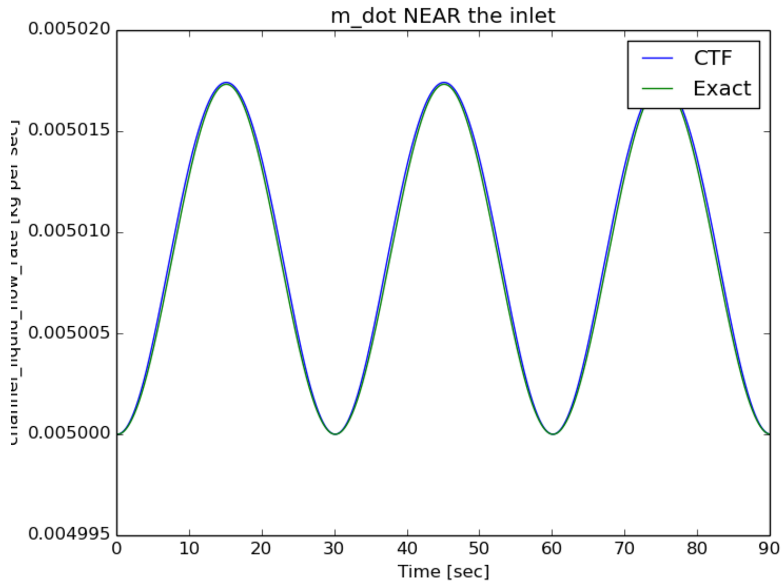

Figure 3. Mass Flow rate near the inlet and the analytical solution

3 CONCLUSIONS

Present your summary and conclusions here.

4 ACKNOWLEDGMENTS

Dr. Vince Mosseau, Dr. Maria Avramova, Dr. Kostadin Ivanov, and Nathan Porter.

5 REFERENCES

[1] C. J. Roy, "Review of Code and Solution Verification Procedures for Computational Simulation," *J. Comput. Phys.*, **205**, *1*, pp. 131–156 (2005).

[2] R. K. Salko, "CTF Theory Manual," The Pennsylvania State University (2014).