Modeling CANDU Header Conditions: Coupling CFD and Thermal Hydraulic Tools

P. Szymanski
McMaster University, Ontario, Canada

Abstract

Nuclear safety analysis relies on computer modeling tools for demonstrating reactor safety and for determination of safe operating parameters such as neutron flux profiles and coolant flow rates. Use of computational fluid dynamics for solving fluid flow behavior in complex components such as CANDU reactor headers has the potential to increase operating margin and, therefore, profitability of a reactor. This paper briefly describes a proposed project involving the coupling of thermal hydraulics tool CATHENA to the CFD tool STAR-CD. Expected challenges are outlined.

1. Introduction

The nuclear safety and regulatory communities depend heavily on computer codes for complex systems analysis from reactor physics to thermal hydraulics. Accurate representation of physical phenomena is essential for modeling of these complex systems. As the trend in nuclear safety continues to move towards best-estimate and uncertainty analysis over traditional extremely conservative approaches, more accurate modeling techniques are required to satisfy regulatory demands.

Common thermal hydraulics system codes (such as CATHENA or RELAP) often make significant approximations in their treatment of various systems components. Such approximations often consist of correlations which account for three dimensional fluid flow effects. These correlations are sufficient for conservative safety analysis, however, if greater operating margin is desired, greater code accuracy is also necessary. Coupling of computational fluid dynamics (CFD) tools with system thermal hydraulics tools is a logical step in best-estimate and uncertainty safety analysis. Such coupling demands greater computational resources but may provide sufficient operating margin gains to justify cost, especially with continually decreasing processing power cost.

2. Applicability to Best-Estimate and Uncertainty Analysis

As can be seen from figure 1, showing a SOPHT simulation from Novog et al. [1], considerations of reactor inlet and outlet header flow conditions are traditionally lumped into a single average value, assuming the same conditions at junctions to all channel feeders. In reality, the conditions for each fuel channel may be significantly different in terms of coolant flow rate, temperature, and power. An example of uneven pressure distribution from Muhana [2] along a header may be seen in figure 2. Use of CFD code in determining individual channel coolant flow rates could aid in the analysis of numerous scenarios, including transient accidents such as loss of coolant. This is potentially useful in demonstrating compliance with regulatory requirements. For example, in Canada, regulatory guide G-144 of the Canadian Nuclear Safety Commission accepts that fuel damage is unlikely to occur if fuel sheath temperatures do not exceed 600ºC and the duration of post-dryout operation does not exceed 60 seconds [3]. Use of a CFD solver in calculating header flow behaviour could eliminate a degree of uncertainty in the use of a code such as CATHENA for calculating header to header pressure drops and channel flow rates. Channel flow rate is key in determining critical power, and therefore safety margin. Code coupling would be useful in a best estimate and uncertainty approach to reactor safety, since simulation would provide estimates within a more narrow probability and confidence limit than thermal hydraulic tools alone, potentially increasing allowable operating margin of a CANDU type reactor.

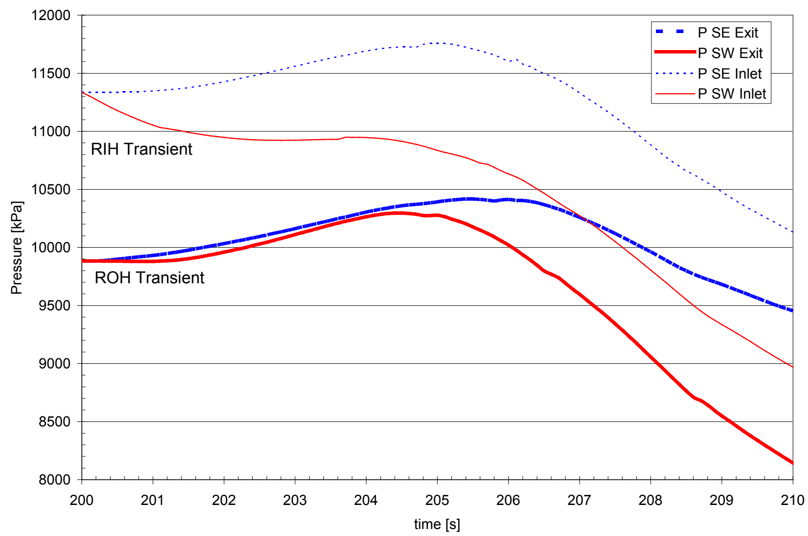

Figure 1: SOPHT treatment of header pressure during accident conditions [Source: Novog et al.] [1]

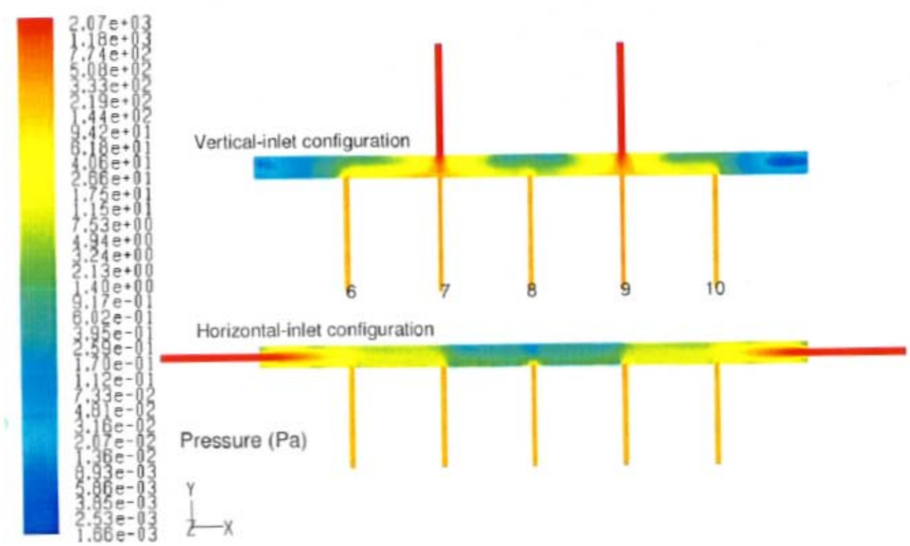

Figure 2: Uneven flow distribution in a header-like geometry [Source: Muhana] [2]

3. Outline of Planned Work

Moffet et.al. [4] used a CFD code (CFDS-FLOW3D) to predict pressure distribution inside a CANDU-6 reactor inlet header. Muhana [2] performed a validation of the FLUENT 6.3.26 CFD tool using experimental data and found it to be an effective tool for header flow distribution prediction. This demonstrates the soundness of the premise of CFD usefulness to model header flow behaviour. Under the supervision of Dr. D. Novog of McMaster University and with the cooperation of Atomic Energy of Canada Ltd. (AECL), a project will be undertaken to couple a Canadian industry standard thermal hydraulics code CATHENA [5] to a CFD code, likely STAR-CD. This project will consist of a number of phases. A reactor geometry will be chosen. This will either be a generic CANDU-6, or RD-14M, a reduced scale CANDU configuration facility with 5 electrically heated channels, from which experimental data should be available for comparison. Initially, only single phase flow modeling will be attempted. A flow chart of the proposed code coupling structure may be seen in figure 3.

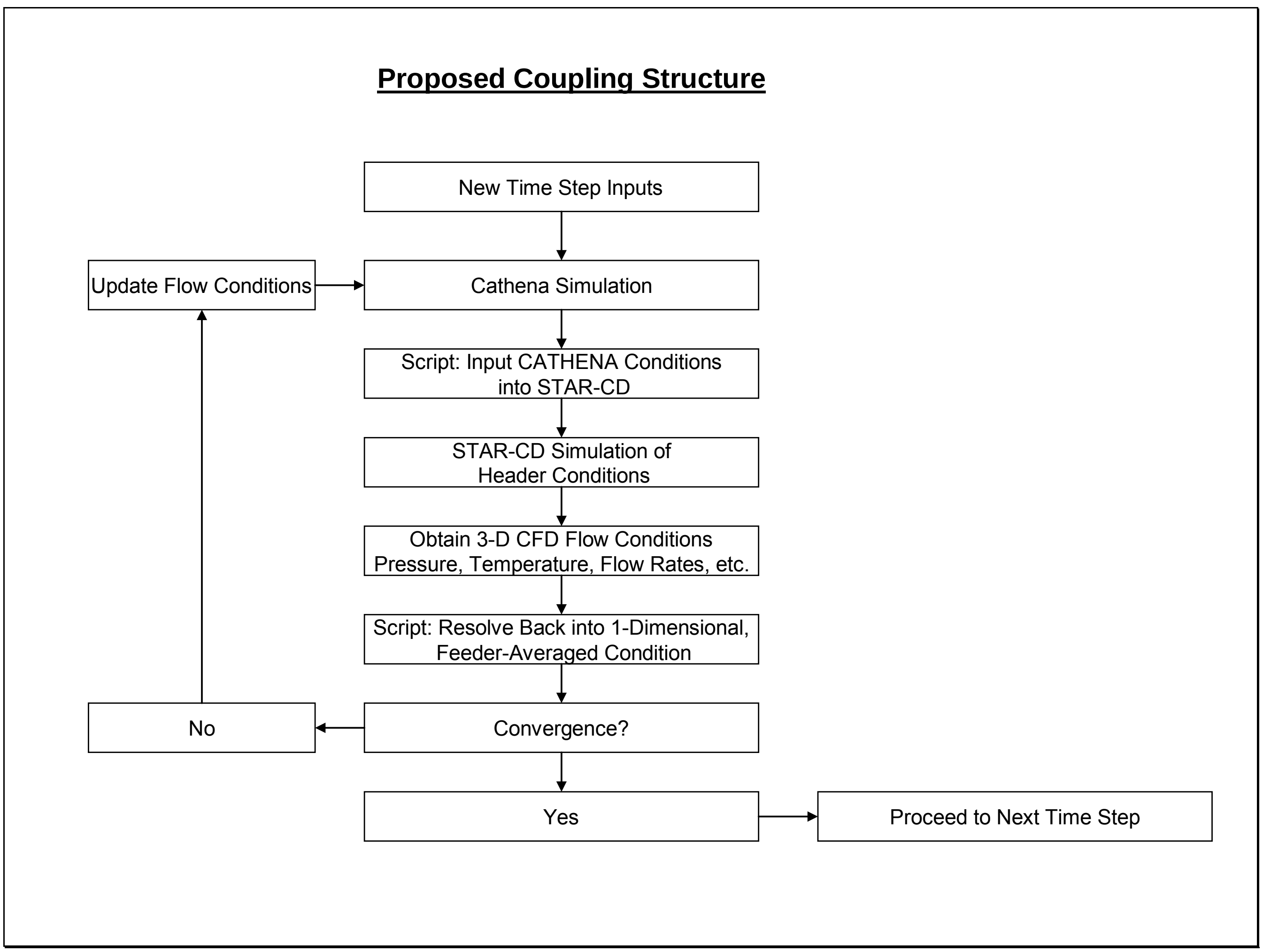

Figure 3: Proposed code coupling structure

Anticipated challenges include selection of turbulence model, treatment of near wall conditions, and matrix solver method, as well as determination of appropriate grid density, and time step. Finally, error analysis will be required to ensure results are within acceptable tolerance.

3. Conclusion

Exploring coupling of thermal hydraulics and CFD tools is a worthwhile pursuit due to the potential for increased operating margin while maintaining safe reactor operation. Modeling of three dimensional fluid flow effects has been successfully performed in the past and should, therefore, be acceptable for integration into nuclear safety analysis programs. It is essential to validate such a coupled tool, and so comparison of output to experimental data from a system such as RD14M would be necessary once the tool development is complete.

4. References

[1] D.R. Novog, F. Bao, J. Luxat, “Best Estimate analysis of Loss of Flow Events at CANDU Nuclear Power Plants”, Proceedings of the 28th Annual Conference of the Canadian Nuclear Society, St. John, New Brunswick, Canada, 2007 June 3-6.

[2] A. Muhana, "Validation of Computational Fluid Dynamic Tools for Single-Phase Flow Distribution in CANDU Headers", M.A.Sc. Thesis, McMaster University, On. 2009.

[3] Canadian Nuclear Safety Commission, Draft Regulatory Guide G-144, Guidelines for Establishment of Shutdown System Trip Parameter Effectiveness, March 2005.

[4] R. Moffet, M. Soulard, G. Hotte, R.A. Gibb, A.O. Banas, "Pressure Distribution Inside a CANDU-6 Reactor Inlet Header", Proceedings of CFD-96, Ottawa, Ontario, June 2-6, 1996.

[5] T. Beuthe, B.N. Hanna, "CATHENA MOD-3.5d/Rev 2 Input Reference", Atomic Energy of Canada Ltd. Chalk River, Ontario. August 2005.