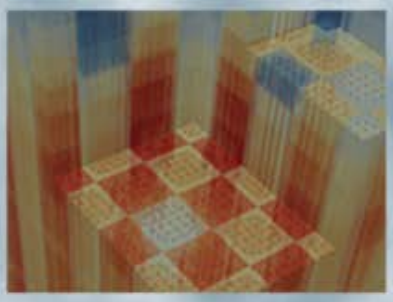
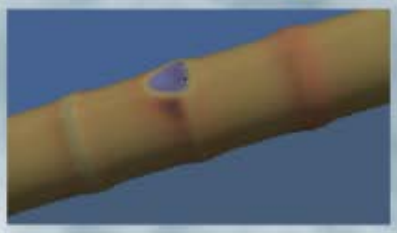
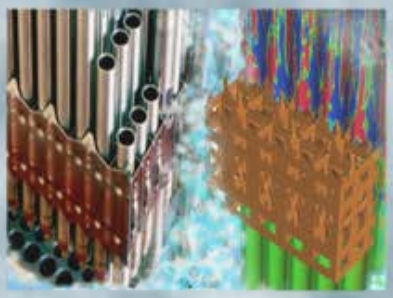
CASL-U-2015-0241-000

CASL CFD-Informed Spacer Grid Models in CTF







Taylor Blyth and Maria Avramova
The Pennsylvania State University

Robert Salko
Oak Ridge National Laboratory

July 7, 2015

U.S. DEPARTMENT OF ENERGY | Nuclear Energy

CASL-U-2015-0241-000

CFD-Informed Spacer Grid Models in CTF

Taylor Blyth[1], Maria Avramova[1], Robert Salko[2]

[1]Pennsylvania State University [2]Oak Ridge National Laboratory

Introduction/Background

Objects placed within the path of a flow will impart deviations on the flow's path. Changes in flow area, such as that experienced while passing through a spacer grid, affect fluid properties and local velocities. The disruption of the boundary layer brought by the imposition of the grid affects the heat transfer ability of the fluid. These effects generally dissipate as distance downstream of the object increases. Taking the overall impact of the object and grouping its combined effects on the flow into various components is an important step in modeling spacer grids in a subchannel code.

CTF contains models and correlations to account for some general spacer grid effects. There are additional effects which are generalized in the source code or simply not modelled at all. In order to enhance CTF's ability to handle spacer grids, it is worth enlisting the capabilities of a Computational Fluid Dynamics (CFD) code.

CFD codes can pick up the fine details in the flow surrounding spacer grids. Patterns from these details can be applied as modifications within the source of CTF to better account for the presence of grids. These updated or new correlations offer users a better and more validated method for modeling subchannels with grid blockages in CTF. There are four models currently developed or under development:

- Grid-directed crossflow model
- Grid-enhanced turbulent mixing
- Grid-induced heat transfer enhancement
- Detailed pressure loss effects

Spacer grids

- Used within fuel assemblies to hold the fuel rods in place
  - Also increase fluid turbulence and heat transfer from the rods to the coolant
- Act as blockages in the flow and disrupt the typical axial flow patterns
- Subchannel codes need to model their placement and geometry in order to capture their influence on the surrounding fluid

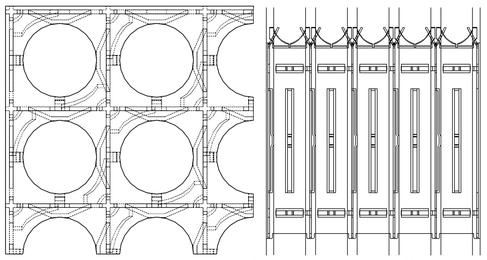


Figure 1: PSBT Mixing Vane Grid (MVG) geometry as a CAD file

Subchannel code: CTF

- Originally developed by Pacific Northwest National Laboratory in 1980
  - COolant Boiling in Rod Arrays – Two Fluid (COBRA-TF)
- Since then, it has been improved and upgraded at several locations:
  - Westinghouse
  - The Pennsylvania State University (PSU)
  - Oak Ridge National Laboratory (ORNL)
  - Other academic and research institutions world-wide
- Currently, PSU's Reactor Dynamics and Fuel Management Group (RDFMG) manages a version: CTF
  - CTF has been adopted into CASL's VERA code suite
- CTF is a Light Water Reactor (LWR) Thermal/Hydraulic (T/H) code used to simulate the core vessel using a two-fluid, three-field approach

CFD code: STAR-CCM+ by CD-adapco

- STAR-CCM+ offers several differences in modeling techniques compared with CTF:
  - Ability to import and mesh user-supplied CAD files and 3D parasolids
  - Graphical User Interface (GUI) well-suited for pre- and post-processing
  - Utilize a geometric mesh not bounded by subchannel boundaries
  - Direct visual representation of many relevant parameters
- Like CTF, STAR-CCM+ can:
  - Determine fluid properties along a defined geometry
  - Export results as 3D data for analysis

By utilizing actual spacer grid data, such as importing directly from CAD files, CFD is able to perform high-fidelity calculations and provide detailed results to the users. The ability to accurately model and simulate results in a subchannel code using CFD-supplied data would provide a quick method to perform subchannel analysis which could be expanded to the required geometry. This method utilizes the advantages of both types of code—the computational speed of CTF and the detailed results of STAR-CCM+.

Implementation of Previously-developed Models

Grid-enhanced turbulent mixing:

This model accounts for the lateral mixing of mass between adjacent subchannels due to turbulence. CFD simulations included a 2x1 subchannel model (Figure 3) with a slight inlet temperature difference to determine a multiplier for the two-phase mixing coefficient, β_{tp}. The grids with larger vane angles produced more turbulent mixing, as expected. This two-phase mixing coefficient is used in the determination of the transverse mass flow caused by turbulent mixing:

$$W_{ij}^{M} = \frac{\beta_{tp}\bar{G}}{\bar{\rho}_{mix}}(G_{iz} - G_{jz})A_{gap}$$

The calculation of the mixing coefficient involves utilizing volume- and surface-averaged values from the CFD simulation to produce a multiplier value at each of the axial nodes. These values are then read into CTF from an external data file and used in the calculation when the model is called.

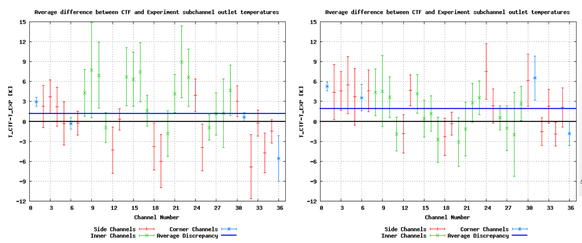


Figure 2: CE 5x5 subchannel exit temperatures with grid-directed crossflow model on and off

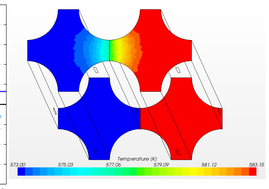


Figure 3: 2x1 Turbulent diffusion model

Grid-directed crossflow model:

This model adds a source term to the liquid transverse momentum equation in order to account for the additional crossflow caused by the geometry of the grids. A lateral convection factor is defined using CFD results from a 2x2 subchannel simulation. This factor is a function of the lateral and axial velocities as well as distance downstream of the grid.

$$f_{lat,SG}(z - z_g) = \frac{V_{z-z_g}^{CFD}}{U_{inlet}^{CFD}}$$

These factors were originally only derived for internal-internal subchannel connections. Therefore, they are not applied to any other type of gaps within the CTF model geometry. They are vane-angle and downstream distance depended. The results of a Combustion Engineering (CE) 5x5 bundle with 36 subchannels are shown in Figure 2. There are 21 experimental test cases which recorded subchannel outlet temperatures to which the CTF results are compared. The main effect is to redistribute the flow across the bundle in a more realistic fashion. This can be seen as the green lines (corresponding to the internal subchannels) show better predictions with the directed crossflow model turned on. The corner and side subchannels are not intended to be improved with the current model.

Future work for this previously-developed model is to include different types of subchannel connections if the lateral convection factors are different enough among the various connections.

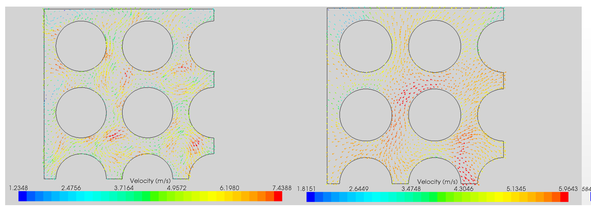


Figure 4: Velocity magnitudes at the PSBT MVG tips and at 0.045 m downstream of the tips

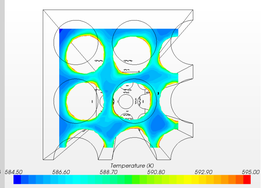


Figure 5: Temperature distribution 0.045 m downstream of a PSBT MVG

Reasoning for CFD data within CTF

Figures 4 and 5 show the level of detail from spacer grid simulations in STAR-CCM+ that is not directly captured in CTF simulations. The mesh size of the CFD code is much smaller than that of CTF and therefore more details are captured in what would be a single CTF mesh volume. Since the volumes of CTF cannot be broken down any more without major modification to the code, the best balance of both codes is to use patterns from the CFD results and place them into models as multipliers or new correlations which can be related to grid design.

Proposed/Ongoing Work

Grid-induced Heat Transfer Enhancement

Altering the geometry of the flow area by means of introducing grids causes differences in the flow around the grid. A spacer grid impacts the downstream heat transfer by the following means:

- Boundary layer (BL) disruption due to additional surfaces—both the hydrodynamic/velocity BL and the thermal BL
- Skin friction along the grids which alters the local axial and lateral velocities
- Thermal radiation from the rod surface to the grid through the fluid
- Fin cooling effect of the rod surface being in contact with the spacer grid
- Upstream of the grid, the flow is accelerated causes variation in rod surface temperatures
- Diversion of the flow around the non-negligible grid strap thickness

For the most part, the current heat transfer models implemented in CTF do not take the effects of spacer grids into account. Many of the heat transfer coefficients (HTCs) are functions of the Reynolds and Prandtl number. In typical LWR conditions, the Prandtl number will not vary by much as it is a relation of viscosity and thermal conductivity. The larger effect of having spacer grids in the path of the fluid will be shown in the change in the local Reynolds number. The Nusselt number is often used in heat transfer calculations and is essentially the Dittus-Boelter equation as found within CTF.

The current option in CTF to specify a grid enhancement of heat transfer utilizes the Yao-Hochreiter-Leech formula. This uses blockage ratios and experimentally-derived constants to provide a ratio of the grid-modified Nusselt number to the non-gridded case.

$$\frac{Nu}{Nu_o} = \left[1 + C_1\varepsilon^2 e^{C_2\left(\frac{z}{D}\right)}\right]\left[1 + A^2 tan^2(\Phi) e^{C_2\left(\frac{z}{D}\right)}\right]^{C_4}$$

This is a general equation which requires the users to provide details on the grid's geometry directly to CTF. Grids with different types of vanes have very different effects on the downstream flow but could have similar blockage ratios. For this reason it is important to take the exact grid geometry and results from CFD calculations into account when modifying the CTF heat transfer models. Patterns in the lateral and secondary flows need to be analyzed and likely applied as local velocity multipliers which will affect local Reynolds number values and therefore local heat transfer rates. If needed, turbulent kinetic energies may also be used in order to provide more details on the flow.

Detailed Pressure-Loss Effects

The current pressure loss model in CTF uses velocity head losses. The grid loss coefficient term is used to calculate a term in the momentum conservation equation. This is in the form of affecting the interfacial drag term. CTF allows users to specify a per-subchannel grid loss coefficient which is used to calculate the pressure drop. This grid loss coefficient may be calculate within CTF if the user has detailed information on the grid's spacer and spring blockage ratios as well as the pressure loss coefficient multiplier.

$$\Delta P = \frac{1}{2}\zeta_{grid}\rho u^2$$

This model does not account for other effects of the mixing vanes on the pressure—only the blockage as seen by the flow. Since CFD codes offer the ability to model with exact grid geometry and not certain values used to characterize the grid, the exact grid loss coefficients can be calculated on a per-grid basis. This would require to the user to only supply a properly-formatted data file.

Methodology

These proposed models can be updated/improved as necessary to provide appropriate results and best suit the needs of CTF users. Current validation of these CFD-informed models utilizes the PSBT Benchmark and its results to ensure the models are properly implemented within CTF. Expansion with more detailed experimental data is planned as relevant datasets are identified.

These methods would allow users to supply their own CAD files and grid data to be used by CTF. Since the data are not placed directly into CTF and rather are read from these external formatted tables, users are not forced to modify files to produce more-generic grid designs. The idea is to eventually provide some typical grid designs from which the user may choose within CTF's source, but still allow proprietary data on grid characteristics to be supplied as necessary.

Why use CTF at all when CFD calculations seem to be more accurate and precise?

- Computational time is still a large factor in the decision
  - CFD runtimes are improving, but will not match those of CTF
    - Updates to CTF's parallelization schemes are planned as well
- If a user needs to run a scoping study or a sensitivity analysis
  - 1000+ runs with parameter variation in CTF gives the user a good set of results
- Modifications to CTF's source can be implemented as desired
  - Models impacted by spacer grids can be easily tested and tuned

This research is supported by and performed in conjunction with the Consortium for Advanced Simulation of Light Water Reactors (http://www.casl.gov), an Energy Innovation Hub (http://www.energy.gov/hubs) for Modeling and Simulation of Nuclear Reactors under U.S. Department of Energy Contract No. DE-AC05-00OR22725.

OAK RIDGE NATIONAL LABORATORY
MANAGED BY UT-BATTELLE FOR THE DEPARTMENT OF ENERGY

U.S. DEPARTMENT OF ENERGY

CASL-U-2015-0241-000