Preliminary results for validation of Computational Fluid Dynamics for prediction of flow through a split vane spacer grid

A. Rashkovan and D.R. Novog
McMaster University, Ontario, Canada
(novog@mcmaster.ca)

Abstract

This paper presents the results of the CFD simulations of turbulent flow past spacer grid with mixing vanes. This study summarizes the first stage of the ongoing numerical blind exercise organized by OECD-NEA. McMaster University along with other participants plan to submit a numerical prediction of the detailed flow field and turbulence characteristics of the flow past 5x5 rod bundle with a spacer grid equipped with two types of mixing vanes. The results will be compared with blind experimental measurements performed in Korea. Due to the fact that a number of the modeling strategies are suggested in literature for such types of flows, we have performed a series of tests to assess the mesh requirements, flow steadiness, turbulence modeling and wall treatment effects. Results of these studies are reported in the present paper.

1. Introduction

Computational Fluid Dynamics (CFD) is being increasingly utilized in the design and licensing of nuclear power stations. Internationally CFD has been used to assess mixing vane spacer designs, to examine boron dilution problems, t-junction induced aging and other localized phenomena. In Canada, CFD has been applied to primary heat transport system header geometries to study pressure and temperature gradients, flow and turbulence generation inside fuel bundles, and in safety and licensing applications related to moderator flow and temperature distributions. CFD applications continue to increase in the nuclear safety field. In the light water reactor (LWR) and heavy water reactor (HWR) communities there are a variety of safety issues where application of CFD is expected to provide insight and assist in closing the issues.

Prediction of subchannel flows, even in isothermal conditions, is very challenging. Complicated flow structures, mixing in the gap region, and even unsteady pulsing type behaviour, prove challenging for predictive models. This is made more complicated by the presence of grid spacers in LWR assemblies, or endplates in CANDU fuel, which cause rigorous mixing as well as greatly increasing the local turbulence levels. In many historical studies, subchannel thermalhydraulic codes such as COBRA or ASSERT-PV have been used to predict flow and enthalpy distributions within fuel bundles. However these subchannel codes rely on empirically derived mixing coefficients, hydraulic loss factors, shear stress and heat transfer relationships to close the system of equations. The advantage of a CFD code for subchannel predictions is that it does not rely so heavily on geometrically dependent mixing factors and empiricisms, rather they rely on more generically applicable turbulence models. Hence CFD results have the potential for wider applicability, notwithstanding their needs for adequate validation and testing. The study of CFD code applicability and accuracy has been the topic of a large number of validation exercises as well as international benchmarks.

Presently, the Nuclear Energy Agency (NEA) of the Organization for Economic Cooperation and Development (OECD) has organized a new benchmark to study CFD applications to bundle flows and turbulence predictions. This benchmark involves taking new experimental data which will be hidden from all participants until after participants have submitted their computational results. Thus the CFD predictions will be "blind" from this respect. The benchmark team has released the specifications, geometries and boundary conditions for these simulations with a proposed submission deadline of May 2012. After submissions, the experimental data will be released and participants will have a chance to present updated results at a closure meeting in the fall of 2012.

Flow and heat transfer through vaned spacer grids was studied both experimentally [1-7] and numerically [8-10] in the past. Although most cases reported are proprietary and the exact geometry and flow conditions are not readily available, some of the results presented can be useful in the task of modeling the MATIS-H benchmark. Up to our knowledge, all the numerical studies reported on flow through vaned spacer grids were steady simulations utilizing either two-equation or RSM turbulence models. The general conclusion of a number of studies is that SST-kω and different versions of k-ε model were found to outperform the RSM models used. While Holloway *et al* [8] used Fluent and found SST-kω with all-y+ wall treatment resulted in the closest comparison with the experiment, Conner *et al* [10] used STAR-CCM+ found RNG k-ε model with high-y+ grid was better than its counterparts. In [9] used the standard k-ε model in CFX and reported good agreement with experimental results on flow through grid with split vanes. No single model is suggested in the literature to perform better in the case of the flow through spacer grids with mixing vanes. Hence, this paper uses separate effect modelling and available literature to examine a wide range of treatments in order to ascertain the best possible modelling options for simulating the MATIS-H benchmark experiment.

McMaster University, sponsored by the CNSC, will be submitting results for consideration in this blind benchmark. Participants are required to submit their solutions along with the details of the computational scheme. Of particular importance will be the size of the computational mesh used in these simulations. While some participants such as the DOE Nuclear Energy Advanced Modeling and Simulation (NEAMS) are planning to submit cases with billions of mesh points and requiring millions of CPU hours, our approach is to develop the most accurate solution possible with a reasonable mesh size and computational times more relevant to engineering and design calculations today. That is to say, for the CFD tool to be useful today it is still necessary to use a somewhat limited number of mesh points such that solutions can be obtained on small to intermediate clusters and within a reasonable time frame. Hence we are performing studies prior to the OECD-NEA submission with an objective to produce accurate results using a computational scheme that could be adopted in industry today. The goal of these tests is to provide insight on the best path to be used for our May 2012 submission. The paper below describes our results from these studies.

2. Examination of the Small Scale Features of the Flow past Spacer Grid

As discussed, we are performing tests of various CFD configurations against existing literature prior to performing the actual benchmark studies. Figure 1 presents schematically the geometry of the spacer grid with the mixing vanes used in [6,7] which we will be using in this work. Although the vanes geometry is different, the experimental conditions and the measurement techniques are similar to those supplied in the OECD-NEA benchmark documentation. The detailed geometry description and the experimental set-up can be found in [11].

The results reported here were calculated using STAR-CCM+ software with standard modelling coefficients, e.g. turbulent Prandtl numbers and other constants were at their default values (no attempt at tuning was performed). The verification studies mainly include mesh sensitivity simulations using transient and steady-state modelling with different turbulence models and wall treatments. In particular a large number of grid topologies and turbulence models (k-ε and variants, k-ω and variants and RSM) are examined. A significant amount of additional work was done to examine the differences between these steady runs and unsteady simulations (URANS). The focus of this study will be on two-equation RANS and their unsteady URANS counterparts.

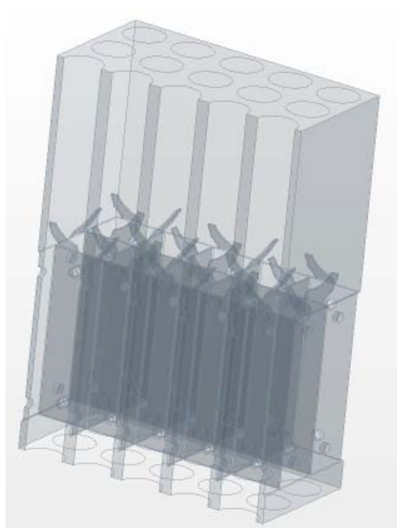

Figure 1. Half of the 5x5 rod bundle used in [6,7].

The geometry in such a grid spacer is very complex. First there is a large number of rods in the array with turbulent interactions between each of the subchannels and gaps. Second, the rods a held in place by small centralizing buttons within the grid spacer with approximately two buttons on each quadrant of each rod. Thirdly as flow exits the grids and buttons used for spacing, there are vanes at the downstream end to promote mixing and enhance heat transfer. Prior to simulating the vaned grid, simulations are performed with a number of simplified geometries, or subdomains, so that the separate effects of these geometrical features could be studied. Each subdomain geometry was selected to study specific aspects of the flow field that may occur in the full geometry. For example, flow around the centralizing buttons located on the grid plates themselves was simulated with a geometry presented in Figure 2.

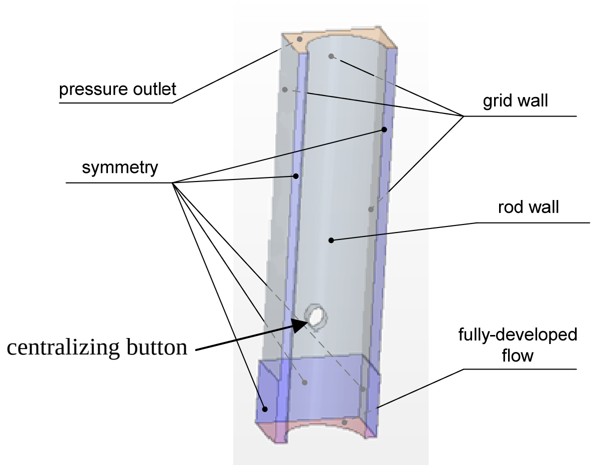

Figure 2. Geometry for studying flow over the centralizing buttons.

The reason for simulating this region separately is due to the possible instabilities of the flow over the buttons. For this subdomain we used a mesh of about 100.000 nodes and simulated a single button. By selecting a smaller region it allowed us to study a large number of mesh possibilities, turbulence models, and steady vs. unsteady behaviour. The instabilities are of similar nature to those appearing in flow over a cylinder with vortex shedding effect. These effects are present starting from relatively low Reynolds number flows and persist for very high values of Reynolds number, although with varying properties. This vortex shedding phenomena is well studied for un-constrained flow around round cylinders but little research is available on such small cylinders within a confined space. Although the geometry of the buttons is that of the cylinder confined between the convex and the flat walls of the rod and the grid respectively, the resulting frequency of vortex shedding predicted by STAR-CCM+ turned out to be of the same order as that expected in the classical situation, 80 instead of 60Hz in the classical treatment. This unsteadiness can be observed in examining the wall shear stress predictions from STAR-CCM+ as shown in Figure 3 for flow past a single button.

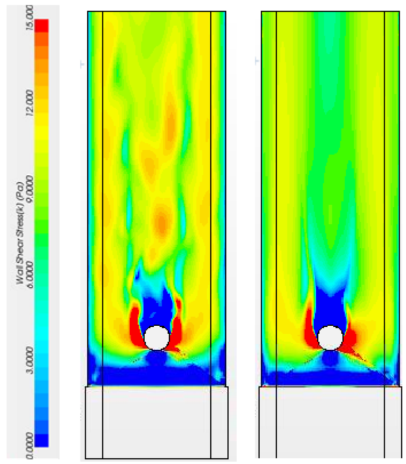

Figure 3. Wall shear stress in steady simulations. SST-kω – left, RKE – right[1].

While resolving the flow instability could eventually result in more accurate flow field predictions, such effects may get overwhelmed by the presence of a large number of buttons and the mixing vanes. If the further validated results of the fully resolved transient solution using say SST-kω show little improvement over those achieved using steady RKE model, it is probably advantageous to submit results for the more efficient model. The same can be argued about the mesh resolution (which also plays a role in damping the unsteady behaviour within the grid structure). These types of questions could be answered only after validation with the existing experimental data on the flow in similar conditions, e.g. published measurements in KAERI experiment [6,7]. Hence we examined these other additional effects prior to proceeding to the MATiS benchmark.

While the buttons give rise to the instabilities and this was numerically verified, the flow through the mixing vanes region was also studied. The model chosen for simulating flow around the vanes is that of two-subchannels from an infinite bundle lattice – that is with symmetry conditions on the external gaps at the edges of the two subchannels as shown in Figure 4.

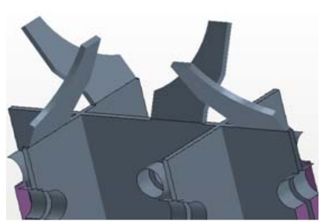

Figure 4. Infinite sub-bundle geometry.

[1] Note: It has to be mentioned that the steady flow solution for the RKE model appears converged and moving to transient simulation does not cause any significant instabilities. On the other hand, while steady solution of the SST-kω model appears to be unstable and not converged, its transient counterpart shows solid basis for the vortex shedding phenomena for a number of time steps and meshes checked, giving the same, i.e. verified shedding frequency. Figure 3 presents the spacer grid wall shear stress for the steady calculations.

Although no finite bundle effects can be incorporated in the geometry presented in Figure 4 (i.e., the side wall effects in the actual bundle are not included) and the domain is quite small it still represents the main features of the flow over the vaned grids. The selection of this subdomain for studying the fine features of the flow allowed us to examine a very large number of mesh topologies, a large number of turbulence models and solution parameters and quickly observe the effects on the solution. In particular one issue to be examined was the role of these vanes in destroying the unsteady structures created by the buttons. The geometry of the vanes is as close as it could be to the published KAERI experiment data [6,7].[2]

Steady RKE simulations of the infinite sub-bundle revealed that the solution in the vicinity of the vane tips is unstable with fine mesh topologies (base size 0.4mm, BL 1st row 0.092mm, growth rate 1.728 with 4 rows, surface mesh size 0.3mm). Qualitatively, simulation with the coarse mesh (base size 1mm, BL 1st row 0.47mm, growth rate 1.0 with 3 rows, surface mesh size 1mm) using RKE showed much more stable and converged solution, though slight instabilities were also recorded.

Figure 5 presents the wall shear stress on the spacer walls for the two meshes described above. Figure 6 demonstrates the solution instability in terms of the minimum lateral velocity on a line segment one hydraulic diameter downstream of the mixing vanes. It is seen that the fine mesh solution fluctuates, while the coarse one is stable in the course of the iterations. Hence one possibility is that the coarser mesh acts as a filter which removes the unsteady features from the flow field. Table 1 summarizes the stability issues of the steady simulations done with the fine mesh. Here we adopt the common technique of examining the convergence characteristics of steady simulations in an attempt to examine the presence and features of possible unsteady flow. Subsequent simulations using unsteady CFD treatment is then focused on cases where we observe quasi-non-steady behaviour in the steady solution convergence characteristics. An example is given below.

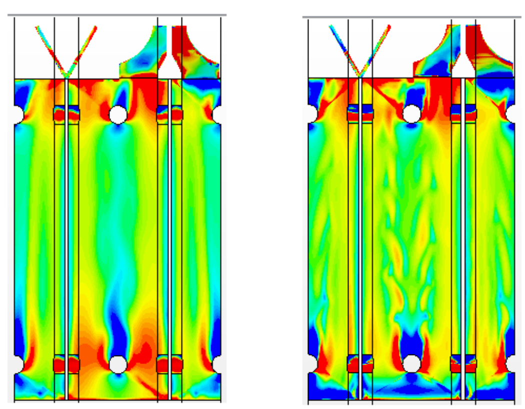

Figure 5. Streamwise wall shear stress. Steady RKE with coarse (left) and fine (right) meshes.

[2] It has to be mentioned that no information concerning the way the rods were centralized in the mentioned KAERI work is provided in the papers. Buttons were assumed to be used as in the case of the benchmark setup for this sub-geometry so that we could study the combined effect of multiple centralizing buttons and mixing vanes.

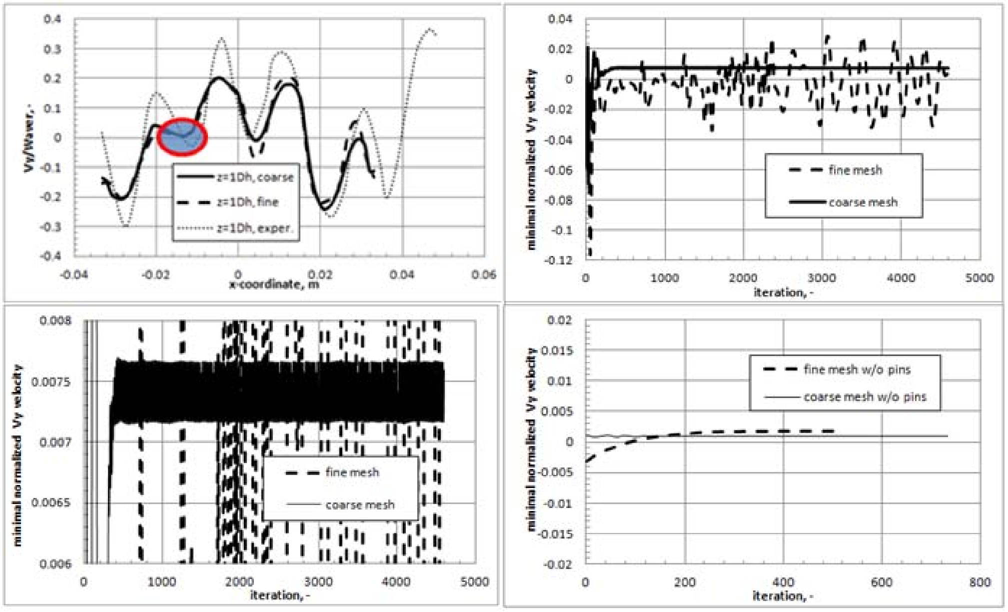

Figure 6. Solution convergence. Top left shows the line segment whose minimal velocity is presented on the rest of the plots in the figure. Top right – minimal velocity convergence as a function of iterations. Bottom left – ordinate axis magnification of the top right plot. Bottom right shows converged solution for the "no-buttons" infinite bundle geometry.

Table 1 shows the matrix of test conducted in examining the impact of the positioning bundles on the steadiness of the flow field as well as the observations of the convergence characteristics. The terms fluctuating and converged indicate the observations of the convergence behaviour in the steady runs. We can see that for the infinite sub-bundle studied here, there are no fluctuating components when the centralizing buttons are not present. Whereas we observe these fluctuations in most other cases as shown Table 1. We are also currently examining the effect of mesh size on observed convergence behaviour as well.

Table 1. Solution convergence for the fine grid, steady state simulations for different geometries.

	RKE	SST-kω
One button (Figure 2)	Converged	Fluctuating
Infinite bundle (Figure 4) with buttons	Fluctuating	Fluctuating
Infinite bundle w/o buttons	Converged	Converged

Note: The same, infinite sub-bundle geometry was calculated without the buttons and steady converged solution was obtained with both RKE and SST-kω models. Keep in mind simulations using RKE with the simplified button geometry and fine grid did converge in steady model – see Figure 3 right[3]. The fact that the same model converged in the one-button geometry but did not converge in the infinite bundle geometry is under investigation, though the instability in the infinite bundle could be caused by the additional complexity in the structure e.g. by the more asymmetric and complicated geometry as compared to the "clean" one button case.

Transient simulations of both RKE and SST-kω models with infinite bundle geometry are now being performed. Based on the results presented so far, it can be concluded that the most probable reason for the solution instability is the vortex shedding in the flow over the buttons upstream of the vanes.

3. Validation Study against Reported Experimental Data

We have obtained preliminary comparisons with the published experimental studies from the same experimental facility, i.e., full 5x5 bundle, though with a slightly different geometry than in the benchmark [6, 7]. Figure 7 presents the results of the steady simulations using realizable k-ε model with all-y+ wall treatment with coarse mesh (approximately 3M nodes), as it was previously defined. The "error bars" represent the product of the gradient of the dependent variable and the mesh size in the vicinity of the point that is approximately 1mm. Figure 7a shows that the axial velocity profiles at 1 and 2 hydraulic diameters from the vanes' tips are captured quite acceptably, while the one at diameters downstream is inconsistent with the measurements in terms of the min/max positions. Similar conclusion can be drawn from Figure 7b where the lateral velocity profiles are presented. Normal Reynolds stresses appear to be captured except those along lines C1 and C2. It can be concluded that the main flow features are captured with the STAR-CCM+ simulations (although some discrepancies exist they may results from fine differences in the actual experiment to those assumed in our studies). Since we did not have the exact geometry from the literature, these differences could not be explored further. Though the comparison is encouraging, further validation is planned for mesh/model optimization toward the final submission.

[3] The small asymmetry of the presented shear stress field, see Figure 3 right, could be attributed in a slight asymmetry in the mesh.

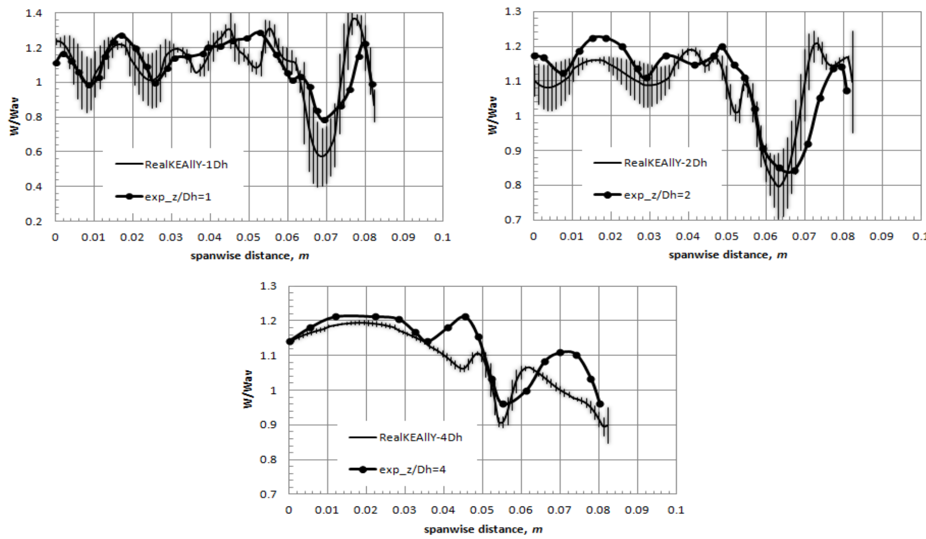

Figure 7a. Normalized streamwise velocity plots at different distances from the vane tips (1 Dh, 2Dh and 4 Dh from the vane tips).

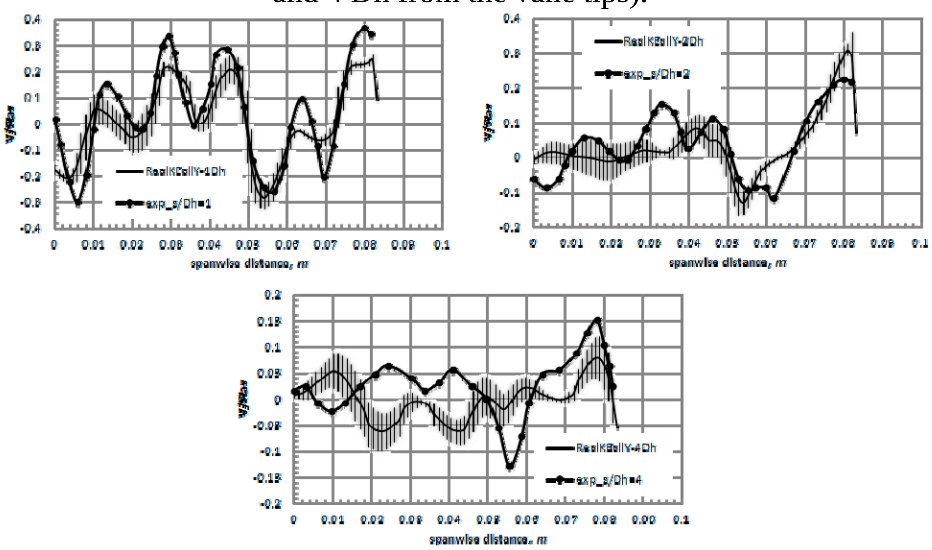

Figure 7b. Normalized lateral velocity plots at different distances from the vane tips.

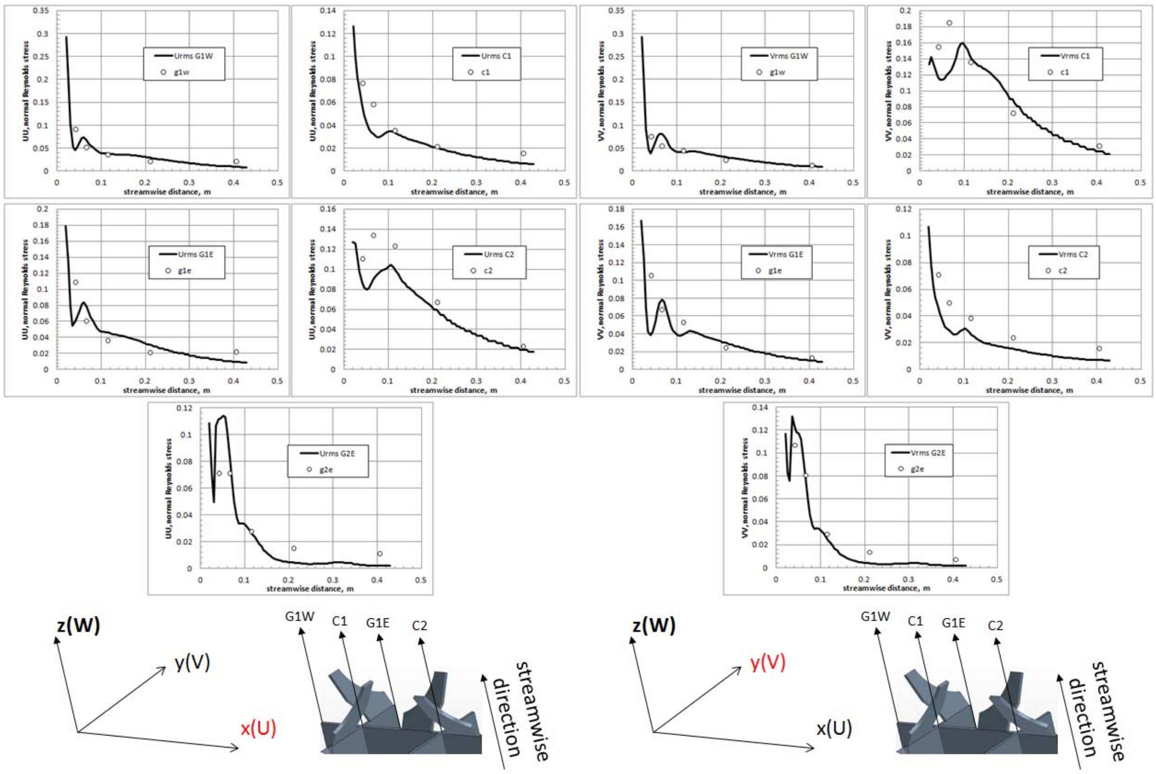

Figure 7c. $\overline{u'u'}$ (left) and $\overline{v'v'}$ (right) Reynolds stresses.

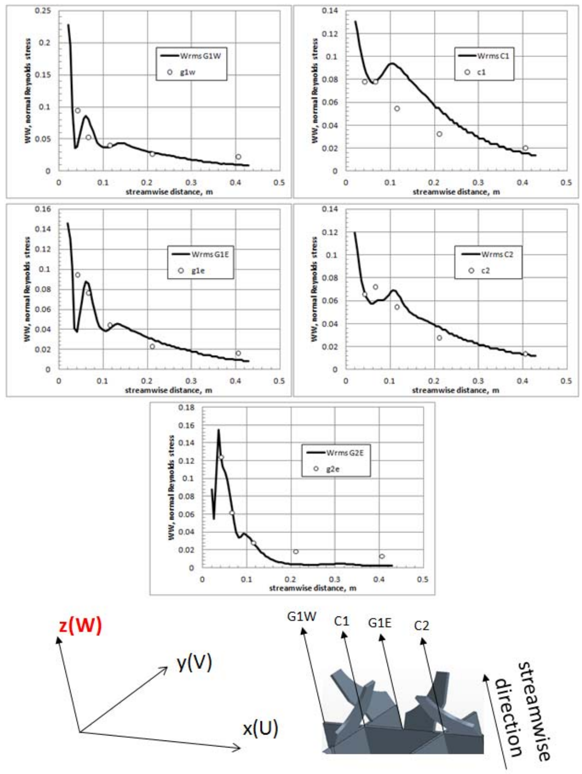

Figure 7d. $\overline{w'w'}$ Reynolds stress.

4. Preliminary Conclusions

The results of our studies have shown several unique and challenging features for CFD applications in rod bundles with mixing grid spacers. In particular we have observed many small scale unsteady features which result from the centralizing buttons within the grid geometries and propagate through the vane tips. We have also studied the turbulent wash downstream of the mixing vanes in an effort to observe the effects of mesh and turbulence modelling. The final stage of our preliminary studies is underway where we are using URANS approaches to study the unsteadiness in the entire 5x5 geometry available in literature. Once complete, a comparison of the steady and unsteady runs for the full geometry will allow us to down-select the best possible solution method which we ultimately apply to the benchmark submission in May 2012.

5. Acknowledgements

The authors would like to acknowledge the direct financial support for this work provided by the Canadian Nuclear Safety Commission (CNSC), and in particular to Jacek Szymansky for his excellent insights. We would also like to acknowledge CD-ADAPCO, developer of STAR-CCM+ for their generous license support and their high quality and responsive user support. Finally, we would like to acknowledge SHARCNET for the use of their High Performance Computing platform.

6. References

[1]. Holloway M.V., Fluid Dynamics and Heat Transfer of Turbulent Flow in Rod Bundle Subchannels, PhD Thesis, Clemson University, 2005.

[2]. Holloway M.V., Conover T.A., McClusky H.L., Beasley D.E. and Conner M.E., The effect of support grid design on azimuthal variation in heat transfer coefficient for rod bundles, Journal of Heat Transfer, V. 127, pp. 598-605, 2005.

[3]. McClusky H.L., Holloway M.V., Conover T.A., Beasley D.E. Conner M.E. and Smith III L.D., Mapping of the lateral flow field in typical subchannels of a support grid with vanes, Journal of Fluids Engineering, V. 125, pp. 987-996, 2003.

[4]. Holloway M.V., McClusky H.L., Beasley D.E. and Conner M.E., The effect of support grid features on local single-phase heat transfer measurements in rod bundle, Journal of Heat Transfer, V. 126, pp. 43-53, 2004.

[5]. McClusky H.L., Holloway M.V., Beasley D.E. and Conner M.E., Development of swirling flow in a rod bundle subchannel, Journal of Fluids Engineering, V. 125, pp. 747-755, 2002.

[6]. Chang S.K., Moon S.K., Baek W.P. and Choi Y.D., Phenomenological investigation on the turbulent flow structures in a rod bundle array with mixing vanes, Nuclear Engineering and Design, 238(2008), 600-609.

[7]. Chang S.K., Moon S.K., Baek W.P. and Chun T.H., The experimental study on mixing characteristics in a square subchannel geometry with typical flow deflectors, Heat Transfer Engineering, 29(2008), 695-703.

[8]. Holloway M.V., Beasley D.E. and Conner M.E., Investigation of swirling flow in rod bundle subchannels using computational fluid dynamics, ICONE14-89068, Proceedings of ICONE14, Miami, Florida, USA, July 17-20, 2006.

[9]. In W.K., Numerical study of coolant mixing caused by the flow deflectors in a nuclear fuel bundle, Nuclear Technology, V. 134, pp. 187-195, 2001.

[10]. Conner M.E., Baglietto E.. and Elmahdi M.E., CFD methodology and validation for single-phase flow in PWR fuel assemblies, OECD/NEA & IAEA Workshop, Grenoble, France, 10-12 September 2008.