Simulation of laminar pulsations in a rectangular duct containing a rod

A. Chettle, M. Lightstone and S. Tullis

Department of Mechanical Engineering, McMaster University, Ontario, Canada

Abstract

Flow pulsations in subchannel geometries are important in the homogenization of fluid temperatures within a fuel rod bundle cross-section. Recent experimental work by Gosset and Tavoularis has indicated that flow pulsations can occur under laminar conditions. This paper presents the results of computational fluid dynamics investigation of laminar flow pulsations in a single rod geometry matching the experimental geometry and conditions of Gosset and Tavoularis. A gap height to rod diameter ratio of 0.3 and Reynolds numbers of 718 and 900 were considered. Rough agreement between the simulations and the experimental results was obtained for the measured frequency, and the simulations revealed highly complex flow patterns throughout the duct cross-section.

1. Introduction

Experimental and numerical investigation of the fluid flow within nuclear rod bundles is important for nuclear safety analysis. A greater understanding of the fluid flow and temperature distribution under both normal operating and accident conditions ultimately could lead to enhanced safety and an improved design for the fuel rods and the reactor core. Previous research [1,2] has indicated that unsteady flow pulsations occur in the smaller interstitial areas of the fuel rod bundles, known as gaps. These pulsations lead to greater mixing and therefore greater heat transfer, in comparison to fully axial flow. The larger interstitial areas of the rod bundles, or subchannels, can be triangular or square, with wall bounded subchannels forming a subset of the square type.

The nuclear safety computer codes employed in the analysis of the heat transfer and axial cooling flow often use a subchannel approach, rather than solving for the fine details of the flow throughout the domain. As a result of the "coarse" subchannel grid, empirical models are introduced to account for effects such as friction, wall heat transfer and fluid mixing. As early as the 1950's, higher mixing rates between the subchannels were found than could be accounted for by purely axial flow [2], which led to a push for further investigation into the mixing so as to gain an appreciation of the physics behind the phenomenon.

2. Background

In order to isolate the phenomenon of subchannel mixing, much of the research has been conducted in greatly simplified geometries. For example, flow pulsations have been observed by Meyer & Rehme [3] in geometries consisting only of two large rectangular ducts connected by a thin rectangular duct to represent a twin subchannel geometry. These simplified geometries gave way to ducts containing four or five rods in a row [4], triangular arrays [5] and also a reduced fuel rod bundle [6]. Throughout these experiments, a range of simulated gap sizes (pitch to diameter ratios, P/D), Reynolds numbers and working fluids have been used.

To the authors' knowledge, all numerical investigation of subchannel mixing have been for turbulent flow, and since 2005, almost all the published results have considered unsteady simulations. These investigations have tried to understand the relationship between gap height (related to P/D), Reynolds number and the frequency of the pulsations, mainly through simplified geometries. Different turbulence models have been used in the simulations including Reynolds Stress Models [7,8,9,10,11], an anisotropic k-ε transport equation model [12] and a one equation Spalart-Almaras model [13]. All of the above models found reasonably good agreement with experimental results, but the optimal turbulence model for this application has not been found [2].

While most experimental results are for fully turbulent conditions, a number of researchers ([14]-[16]) have considered Reynolds number ranges that extend into the laminar regime. Gosset and Tavoularis [14] used dye streak visualization to show the existence of laminar pulsations in a rectangular channel containing a single rod. From the flow visualization, they were able to estimate the frequency and period of the pulsations. Lexmond et al. [15] and the follow up study by Mahmood et al. [16] considered water flow in a twin subchannel geometry for a range of Reynolds numbers. They also found the presence of pulsations for laminar flow, however, the majority of their detailed results from the PIV and mass transfer measurements were provided for turbulent conditions.

Figure 1 shows a comparison between some of the turbulent results [3] and laminar results [14], plotting Strouhal number against Reynolds number. The Strouhal number is defined as fD/U_b, where U_b is the bulk fluid velocity, D is a characteristic diameter and f is the frequency. In the turbulent results, the trend mostly shows that a decrease in gap size for a given gap width results in a higher frequency. However, the opposite trend holds true for the largest gap width in the turbulent results as well as for the laminar results. It is thought that these opposing trends show two sides of a curve which compares frequency to gap size. As one increases gap size from zero, initially, the flow resistance decreases, until the frequency becomes a maximum. After this point, increasing the gap size will decrease the strength of the driving mechanism behind the pulsations.

Simulation of laminar conditions removes the complicating factor of turbulence and turbulence modeling and may be useful in understanding the underlying physics of the pulsations and the effects of the various parameters that influence the pulsations. The purpose of this paper is to show that a laminar flow simulation with a gap geometry predicts these pulsations, and to compare the predicted pulsation characteristics with the existing experimental data [14]. Further, the CFD simulations provide a detailed description of the flow throughout the gap and duct geometry.

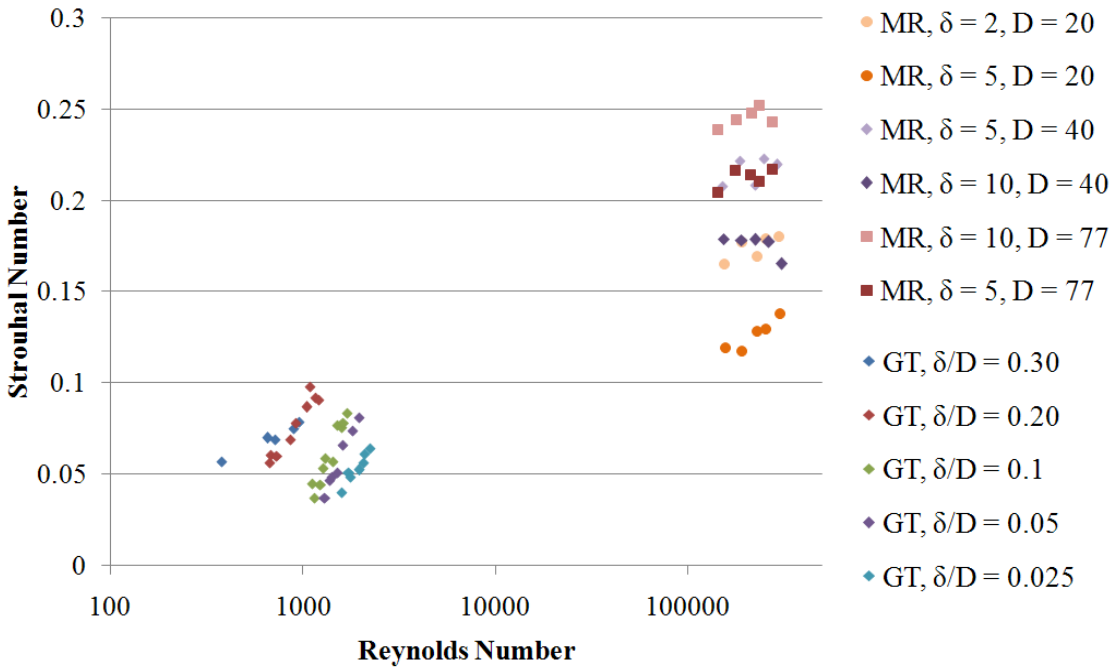

Figure 1: Strouhal number vs. Reynolds number for the combined data of Gosset and Tavoularis (GT) [14] and Meyer and Rehme (MR) [3]. The Strouhal number for both cases is based on the *D*, the diameter of the rod for Gosset and Tavoularis, and the width of the gap for Meyer and Rehme. Gap height is denoted by δ for both cases. The Reynolds number for both cases is based on the bulk velocity and hydraulic diameter of the experimental domain. The Reynolds number reported by Meyer and Rehme was modified to fit the above definitions. Dimensions are in mm.

3. Methodology

The simulations are for a geometry identical to the experiments of Gosset and Tavoularis [14]. Their air flow rig consisted of a rectangular duct containing a rod (D = 13.08 mm), which could be positioned at various distances (δ) from the duct wall. The width of the duct was three times the diameter of the rod and the height was two times the diameter of the rod. A cross section of their experimental domain can be seen in figure 2.

A dimensionless gap height of δ/D = 0.30 and Reynolds numbers, based on the bulk velocity and hydraulic diameter of the entire channel, of 718 and 900 are considered. These cases represent a small subset of the results of Gosset and Tavoularis [14].

A structured grid was made using ICEM CFD 12.0.1, a meshing tool which aids in constructing more complex structured grids. A cross-section of the mesh is shown in figure 3, and each cross sectional slice has 2772 nodes. The grid in the axial (z) direction was uniform, with an element size of 0.8 mm used. The grid was chosen so as to give good refinement in the gap region, without being unnecessarily fine in the far channel regions, yet still having fairly smooth transitions between different mesh regions. A timestep of 0.05 s was used.

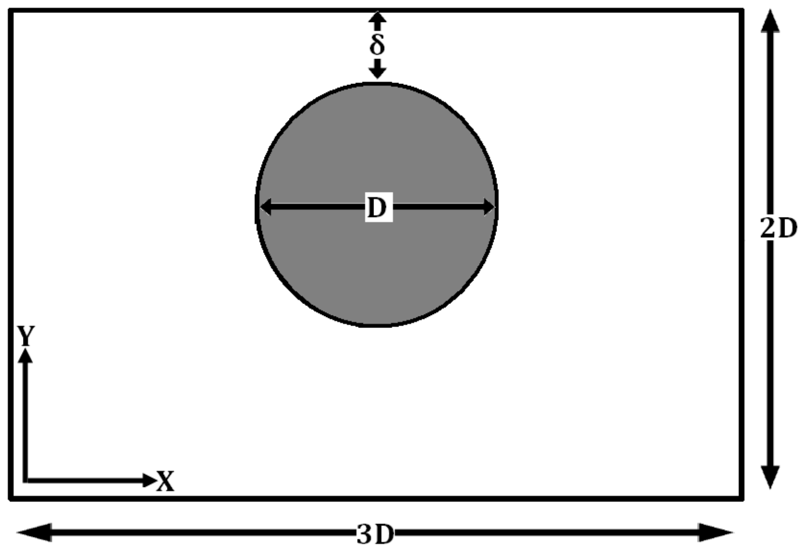

Figure 2: Cross section of numerical domain, which matches the experiments of G&T. The dark grey represents the rod, which has diameter D. The rod is centred horizontally within the domain, and can move vertically to create a simulated gap. The smallest gap between the rod and the walls of the domain is denoted by δ. The origin is at the bottom left corner, with the mean flow parallel to the z-axis into the page.

The boundary conditions were no-slip walls everywhere except the inlet and outlet, which had periodic conditions in the axial direction, with a specified mass flow rate. Simulations were performed using a domain with a length equal to that of the experimental rig, L = 1168 mm. The grid contained 4,047,120 nodes. This grid was used for all numerical work reported in this paper.

Each transient case ran for at least 110s, which is approximately three times the time required for fluid moving at the bulk velocity to flow through the full experimental domain. This was chosen to convect the initial conditions out of the domain and to allow sufficient data to be collected for frequency analysis. Grid and timestep independence studies were successfully conducted. One of the main criteria for these sensitivity studies was RMS values of the spanwise velocity at eleven discrete points in the gap region. The maximum percent difference for the grid independence study was 3.5% and the average percent difference was 1.7% between the grid used and one with half the total number of control volumes. The time independence test showed a maximum percent difference of 2.9%, with an average percent difference of 1.3% between the timestep used (0.05 s) and one twice as long.

A high-resolution advection scheme was used for spatial discretization and a second order backwards Euler scheme was used for temporal discretization. This high-resolution scheme is a second order accurate scheme which calculates the transported quantity between nodes using a blending factor and the spatial gradient. The blending factor is calculated to be as close to 1.0 as possible, without introducing non-physical overshoots or undershoots. The commerical software, ANSYS CFX 12.1, which utilizes a finite volume method, was used for the simulations.

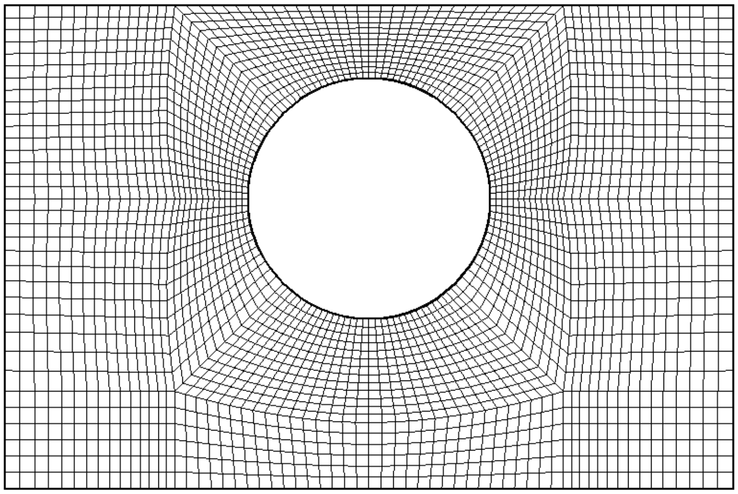

Figure 3: Cross section of a sample grid used in the simulations. The walls and rod are outlined in thick black lines, with the rod being entirely white. The flow direction is normal to the page. This grid is a structured grid, with expanding meshes used to give good refinement in the gap region.

4. Results

Flow on the mid-gap cross-plane for Re=718 is shown in figure 4. Gap pulsations are clearly seen in the laminar numerical simulation results. The spanwise velocities associated with these pulsations at the gap centre have an RMS value that is approximately ten percent of the bulk velocity, and a maximum of approximately two thirds of the bulk velocity. It can be noted that the pulsations appear as pairs of alternating gap spanwise flows, but that flows are not symmetric within a pair, and that successive pairs can differ substantially in magnitude.

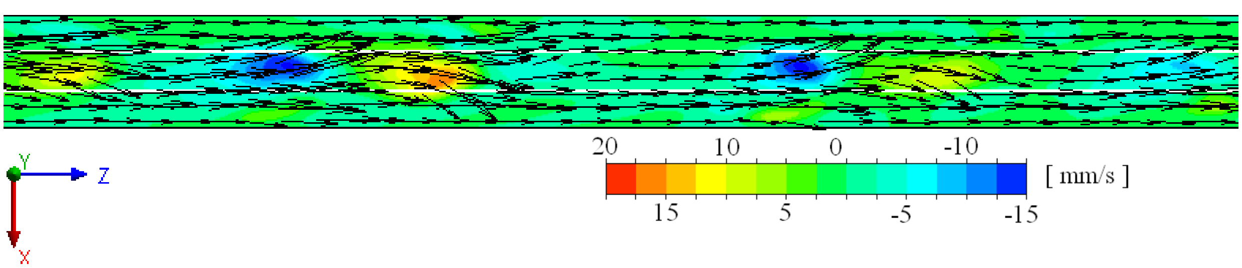

Figure 4: Spanwise (u) velocity contour plot overlaid with a velocity vector plot for Re = 718 at a single time snapshot. The red regions of the contour plot show a positive u-velocity (down), and the blue regions show a negative u-velocity (up). The plane used for this visualization is located in the middle of the gap, at δ/2, and only about ⅓ of the full length is shown. The white lines indicate the position of the rod within the domain. Flow is from left to right.

The Q-criterion [17] can be used to visualize vortices and coherent structures in the flow. It is based on the second invariant of the velocity gradient, Q, which can be defined as:

$$Q = -\frac{1}{2}\frac{\partial u_i}{\partial x_j}\frac{\partial u_j}{\partial x_i}$$

Figure 5 shows an isosurface (Q=0.75 s^{-2}) in the duct. This three dimensional representation illustrates the extremely complex nature of this laminar flow – a regular periodic gap vortex street is not apparent, and vortex structures are seen throughout the duct. The vortex structures are generally elongated axially with a degree of alternating wrap around the rod.

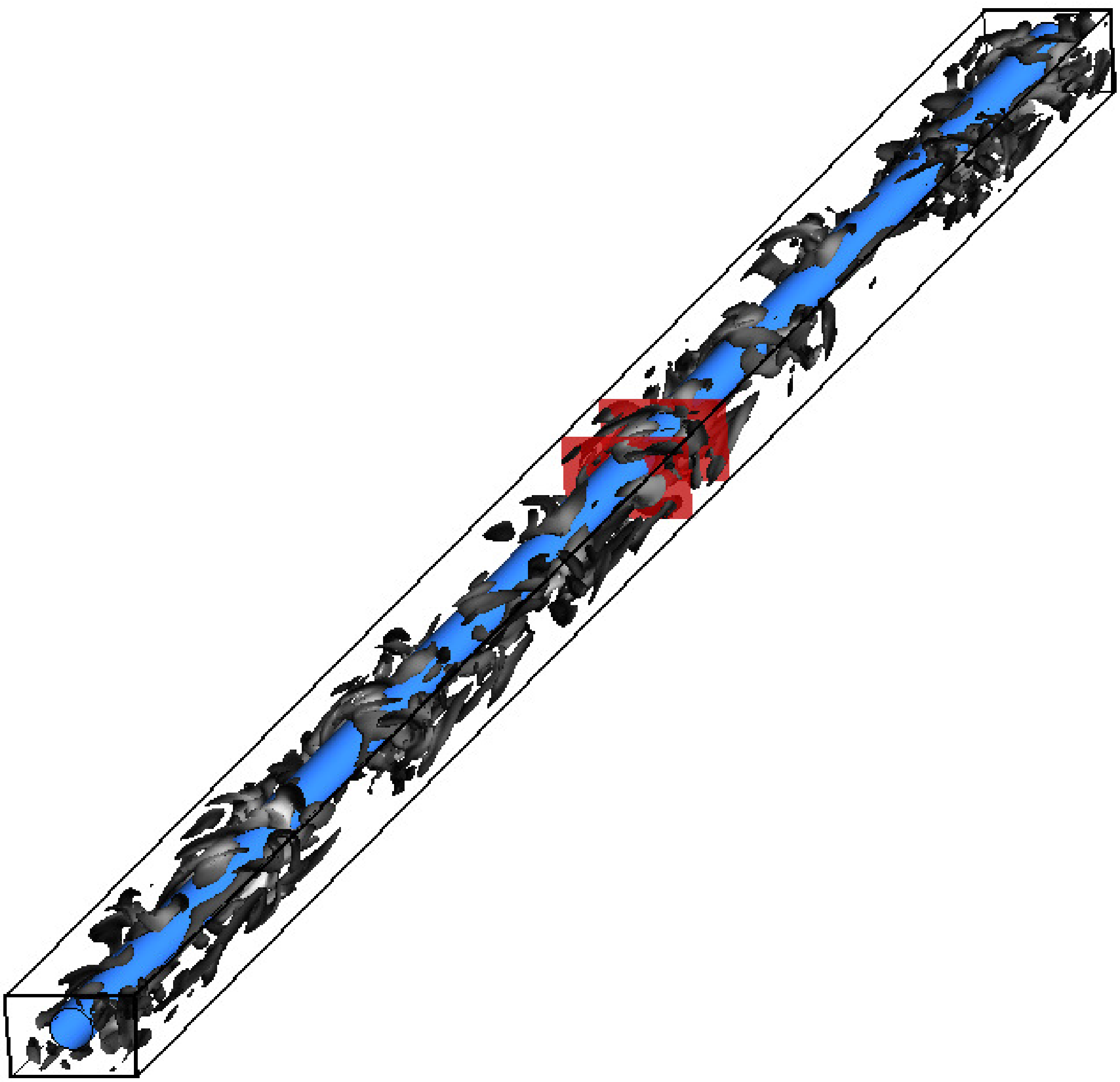

Figure 5: Isosurface plot of Q-criterion for Re=718 at t = 110 s. A level of Q=0.75 s^{-2} is shown. The bulk flow is from bottom left to top right. The two planes highlighted are the locations of the upstream and downstream cross-sections shown in figure 6.

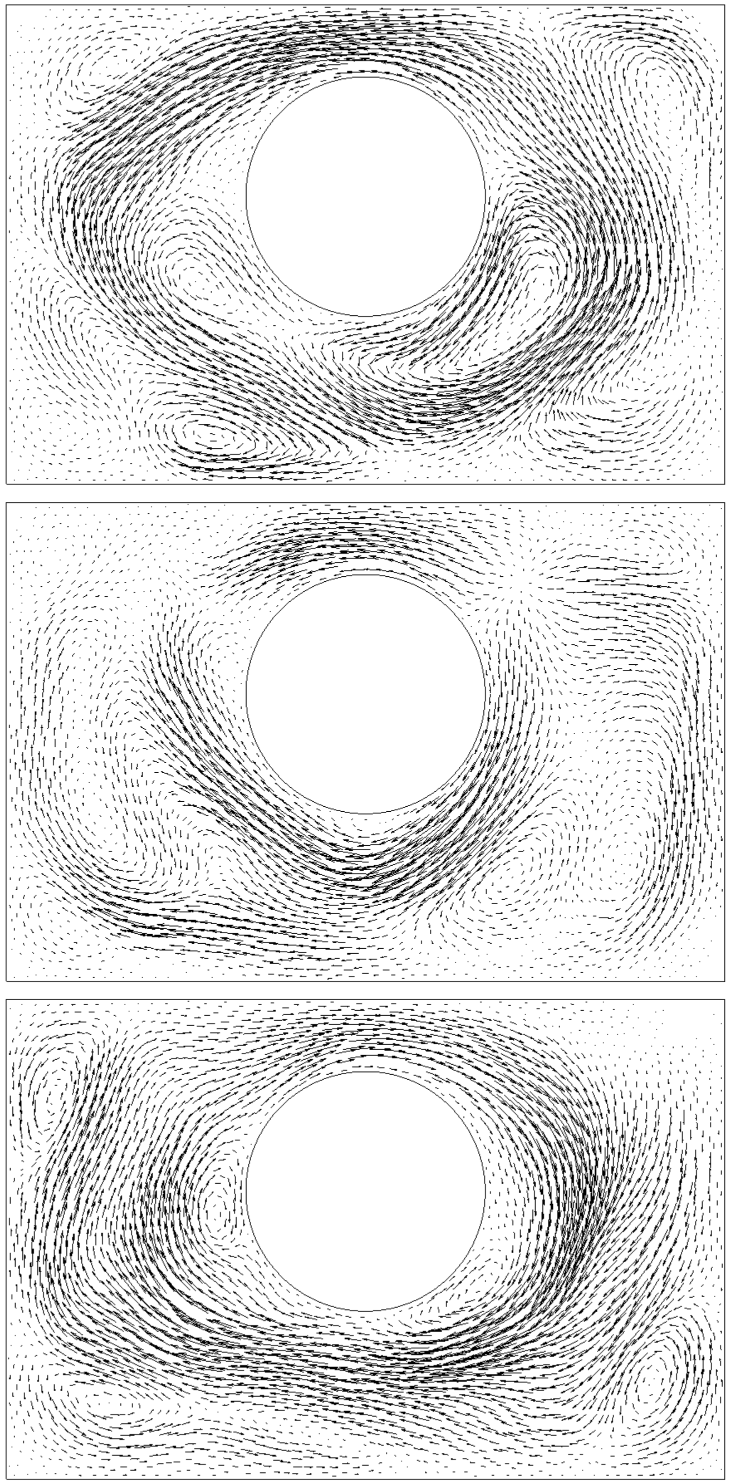

Figure 6: Vector plot showing cross-plane velocities (u,v) at three cross-sections through the Re=718 rectangular duct all taken at a single time interval (t=110s). The planes are each separated axially by 22mm, and progress downstream from the top figure to the bottom figure. The bulk flow is into the page.

Cross-sectional flows between the two cross-planes highlighted in figure 5 are shown in figure 6. At the upstream plane there is a large clockwise flow, while at the most downstream plane there is a large counter-clockwise flow, showing pulsations in the flow direction through the gap. These two locations are separated by 45mm (only about 3.5 rod diameters), which is roughly half of the wavelength of the most significant pulsations (from following frequency analysis). The flow on the intermediately located plane shows the transition between these two different gap and duct flows. A paired source and sink of cross-sectional velocities occur on either side of the gap, and these can be well matched with the corresponding negative and positive axial velocity gradients. This agrees with previous gap edge vortex street hypotheses [10,13,14] as driving mechanisms for the pulsations.

However, the interaction of the gap flows with subchannel bulk flow clearly extends well beyond the subchannel regions adjacent to the gap where there are mean axial velocity gradients that could drive such a vortex street flow. Most significantly, a flow around the other side of the rod is also driven, and contributes to the large circulatory flows around the rod seen. Given the width of this alternative flow path (0.7*D*), it is not yet entirely clear how significant a role it plays in driving the flow (based on examination of the cross-sectional velocities and the axial velocity gradients, it appears minimal). These figures not only demonstrate the pulsating nature of the flow, they also show the complexity of subchannel flows, even under laminar conditions.

To gain an estimation of the frequency and wavelength of these experimentally observed pulsations, Gosset and Tavoularis [14] released dye into the centre of the gap. The frequency was estimated by counting the number of times the dye streak changed direction within a given timeframe. The standard deviation of the estimated frequencies for a given gap size and Reynolds number was reported to be 20% by Gosset and Tavoularis [14].

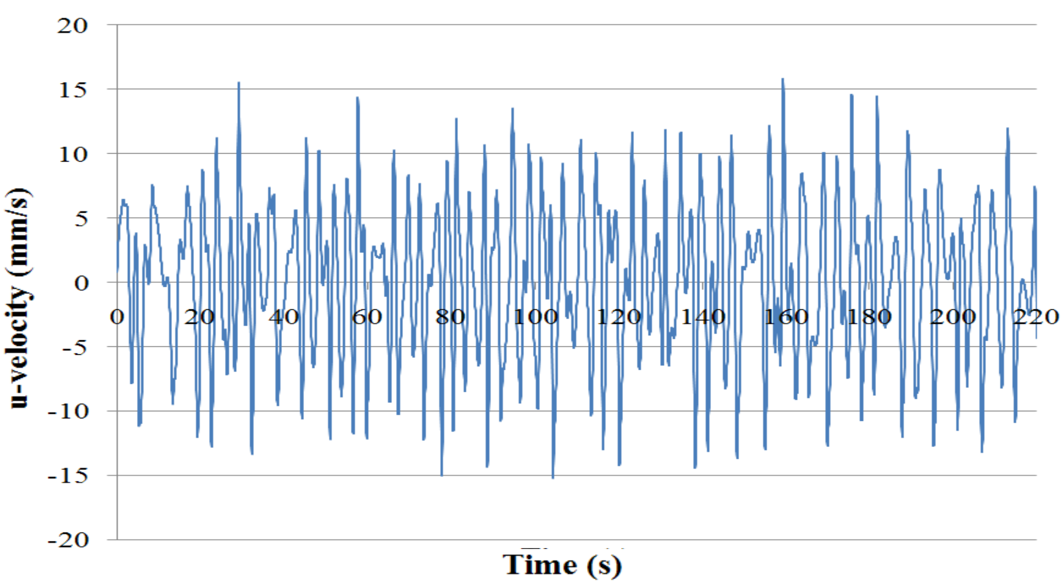

Figure 7: Time trace of spanwise (u) velocity for domain with Re = 718 at a point horizontally and vertically centred within the gap region.

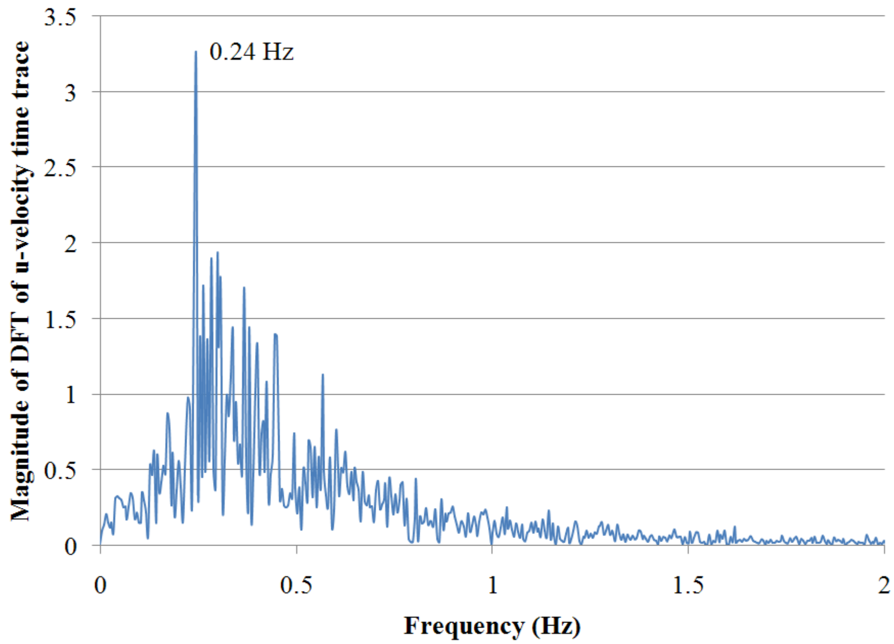

Figure 8: Discrete Fourier transform of the spanwise (u) velocity time trace for Re = 718 at the gap centre.

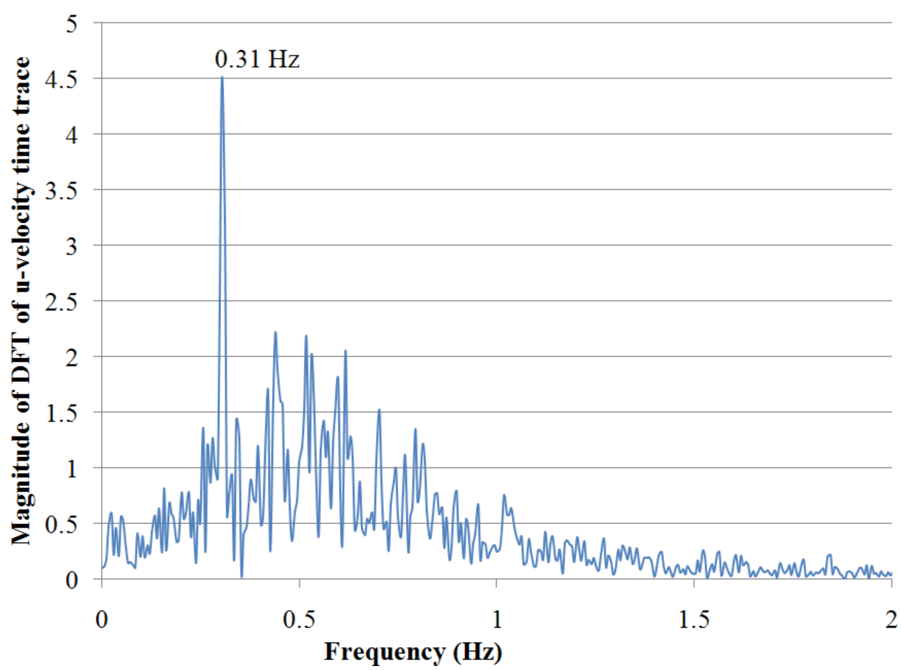

Figure 9: Discrete Fourier transform of the spanwise (u) velocity time trace for Re = 900 at the gap centre.

Predicted pulsation frequencies from the CFD simulation were found through Fourier transforms of velocity and pressure transients at discrete locations in the domain. Figure 7 shows a sample time trace of spanwise velocity from a point horizontally and vertically centered in the gap for Re=718. Frequency analysis of that data is shown in figure 8. The predicted peak frequency of 0.24 Hz (St=0.091) is larger than the experimental value of 0.18 Hz, but is within about 1.6 experimental standard deviations. Similar results are seen in figure 9 for the higher Reynolds number case (Re=900), where a predicted dominant frequency of 0.31 Hz (St=0.092) is compared to the experimental result of 0.25 Hz.

The above figures show the calculated dominant frequency for each Reynolds number case by use of a discrete Fourier transform of a velocity time trace signal. The error in the Fourier transform method is about 1.6% and the error in the numerical calculation of the velocities is less than 4% based on the grid and timestep independence studies. While there is one clear dominant frequency, the background of the FFT indicates that the dominant pulsations are not purely regular.

5. Conclusion

This work has used CFD to predict laminar flow in a rectangular channel containing a single rod. Results were compared to experiments and rough agreement with measured pulsation frequencies was obtained. These simulations have clearly shown that the phenomenon of subchannel flow pulsations can be predicted for laminar flow and have demonstrated the practicality of utilizing a laminar approach to begin to understand subchannel mixing. The predictions revealed the complex nature of the flow throughout the duct cross-section.

6. References

[1] G. Arvanitis, "Simulation of flow pulsations in gap geometries using unsteady Reynolds averaged Navier-Stokes modelling", M.A.Sc. Thesis, McMaster University, Hamilton, Ontario, Canada, 2008

[2] L. Meyer, "From discovery to recognition of periodic large scale vortices in rod bundles as source of natural mixing between subchannels -- A review", *Nuclear Engineering and Design*, Vol. 240, 2010, pp 1575 – 1588

[3] L. Meyer and K. Rehme, "Large-scale turbulence phenomena in compound rectangular channels", *Experimental Thermal and Fluid Science*, Vol. 8, 1994, pp 286 – 304

[4] W.J. Seale, "Turbulent diffusion of heat between connected flow passages", *Nuclear Engineering and Design,* Vol. 54, 1979, pp 183 – 195

[5] B. Kjellstrom, "Studies of turbulent flow parallel to a rod bundle of triangular array", *AB Atomenergie, Studsvik, Sweden,* AE-RL-1344, 1972

[6] A.C Trupp and R.S Azad, "The structure of turbulent flow in triangular array rod bundles", *Nuclear Engineering and Design*, Vol. 32, 1975, pp 47-84

[7] D. Chang and S. Tavoularis, "Unsteady numerical simulations of turbulence and coherent structures in axial flow near a narrow gap ", *Journal of Fluids Engineering*, Vol. 127, 2005, pp 458 - 466

[8] D. Chang and S. Tavoularis, "Convective heat transfer in turbulent flow near a gap", *Journal of Heat Transfer,* Vol. 128, 2006, pp 701 - 708

[9] D. Chang and S. Tavoularis, "Numerical simulation of turbulent flow in a 37-rod bundle", *Nuclear Engineering and Design,* Vol. 237, 2007, pp 575 - 590

[10] D. Chang and S. Tavoularis, "Simulations of turbulence, heat transfer and mixing across narrow gaps between rod-bundle subchannels ", Vol. 238, 2008, pp 109 - 123

[11] F. Abbasian, S.D. Yu and J. Cao, " Experimental and numerical investigations of three-dimensional turbulent flow of water surrounding a CANDU simulation fuel bundle structure inside a channel", *Nuclear Engineering and Design*, Vol. 239, 2009 pp 2224 - 2235

[12] E. Merzari, H. Ninokata, E. Baglietto, "Numerical simulation of flows in tight-lattice fuel bundles", *Nuclear Engineering and Design*, Vol. 238, 2008, pp 1703 - 1719

[13] D. Home, G. Arvanitis, M. Lightstone, M. Hamed, "Simulation of flow pulsations in a twin rectangular subchannel geometry using unsteady Reynolds averaged Navier-Stokes modelling", *Nuclear Engineering and Design*, Vol. 239, 2009, pp 2964 - 2980

[14] A. Gosset and S. Tavoularis, "Laminar flow instability in a rectangular channel with a cylindrical core", *Physics of Fluids*, Vol. 18, Issue 4, 2006, 044108 pp 1-8

[15] A.S. Lexmond, R.F. Muddle, T.H.J.J van der Hagen, "Visualisation of the vortex street and characterisation of the cross flow in the gap between two sub-channels.", Proceedings of the 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics, Avignon, France, 2005 October 2-6.

[16] A. Mahmood, M. Rohde, T.H.J.J. van der Hagen, R.F. Mudde, " Contribution of large-scale coherent structures towards the cross flow in two interconnected channels", Proceedings of the 13th International Topical Meeting on Nuclear Reactor Thermal Hydraulics, Kanazawa City, Ishikawa Prefecture, Japan, 2009 September 27-October 2.

[17] J.C.R. Hunt, A.A. Wray, P. Moin, “Eddies, streams, and convergence zones in turbulent flows”, Centre for Turbulence Research Proceedings of the Summer Program, 1988, pp. 193-208.