



Comparison of measurements and computations of isothermal flow velocity inside HyperVapotrons



A. Sergis^{a,*}, K. Resvanis^a, Y. Hardalupas^a, T. Barrett^b

^a The Department of Mechanical Engineering, Imperial College London, London SW7 2AZ, UK

^b CCFE, Culham Science Centre, Abingdon, Oxon OX14 3DB, UK

HIGHLIGHTS

- A popular HHF device is the HyperVapotron (HV).
- HVs employ a heat transfer mechanism called the Vapotron effect.
- Experimental quantitative visualisation of the device has not been performed.
- CFD tools used in the past to analyse HVs did not benchmark the coolant flows.
- A PIV experiment is used to create benchmark data and compare them to literature.

ARTICLE INFO

Article history:

Received 23 September 2014

Received in revised form 16 February 2015

Accepted 8 April 2015

Available online 25 April 2015

Keywords:

HyperVapotron

Particle image velocimetry

CFD

uRANS

RANS

High heat flux

ABSTRACT

HyperVapotrons are two-phase water-cooled heat exchangers able to receive high heat fluxes (HHF) by employing a cyclic phenomenon called the “Vapotron Effect”. HyperVapotrons have been used routinely in HHF nuclear fusion applications. A detailed experimental investigation on the effect giving rise to the ability to sustain steady state heat fluxes in excess of 10 MW/m² has not yet been possible and hence the phenomenon is not yet well understood. The coolant flow structures that promote the effect have been a major point of interest, and many investigations based on computational fluid dynamic (CFD) simulations have been performed in the past. The understanding of the physics of the coolant flow inside the device may hold the key to further optimisation of engineering designs. However, past computational investigations have not been experimentally evaluated. Isothermal flow velocity distribution measurements of the fluid flow in HyperVapotron optical models with high spatial resolution are performed in this paper. The same measurements are subsequently calculated via commercial CFD software. The isothermal CFD calculation is compared to the experimental velocity measurements to deduce the accuracy of the CFD investigations carried out. This unique comparison between computational and experimental results in HyperVapotrons will direct future efforts in analysing similar devices.

© 2015 The Authors. Published by Elsevier B.V. This is an open access article under the CC BY license (<http://creativecommons.org/licenses/by/4.0/>).

1. Introduction

HyperVapotrons (HVs) are HHF devices which seem to be a strong candidate as a viable primary or secondary heat exchanger for the International Thermonuclear Experimental Reactor (ITER). They employ the Vapotron effect, a complicated periodic phenomenon where vapour forms inside the grooved part of the device and then condenses inside the freestream. Sustained heat fluxes of more than 30 MW/m² have been already demonstrated [1]. Not a lot is known about the Vapotron effect and how to optimise the device.

Experimental research in ion beam test beds provides overall thermal performance characteristics [2,3]. However, there has been no experiment to document, in high resolution, the Vapotron effect inside the device. Visualisation studies have been attempted in the past, however, the technology employed did not succeed in capturing quantitatively the sequence of events dictated as the Vapotron effect [2,4]. Theoretical and numerical studies, usually two-phase investigations, have been performed to try to bridge the gap of physical knowledge around the HV devices [5–9].

The simulations are usually fine-tuned on demand to reproduce bulk thermal performances recorded experimentally in HV HHF experiments conducted on ion beam test beds. However, these types of numerical studies which attempt to predict the detailed velocity of the coolant have not yet been verified experimentally.

* Corresponding author.

E-mail address: a.sergis09@imperial.ac.uk (A. Sergis).

Hence it becomes questionable whether their predictions can be trusted for design and optimisation purposes.

An experimental method, based on particle image velocimetry (PIV), has been selected to quantify the coolant velocity flow field inside a HV model under isothermal conditions in high spatial resolution. The spatial resolution of the measured velocity flow field is superior to the spatial resolution of any simulation found in the literature. The isothermal case is compared to two popular CFD tools with similar or higher grid resolutions than those used in various multiphase CFD investigations found in the literature and their accuracy will be discussed by comparing their results to the actual benchmarking PIV data. This isothermal comparison, even though at the basic single-phase isothermal coolant velocity flow field state, will indicate whether the multiphase CFD investigations with heat transfer under the same grid resolutions that are performed so far are sufficient to describe the physics of flow involved. The comparison with the literature is performed on the merits of the solver and mesh types used and not on their coolant flow velocity and heat transfer performance data as these are not relevant in the current study.

2. Methodology

2.1. Experimental rig

A transparent 5 groove, full size HV model from the Mega Amp Spherical Tokamak (MAST) variant (free stream channel of 3.4 mm in height, 48 mm length with 1.5 mm longitudinal grooves on the sides and 4 mm deep, 3 mm wide fins) has been manufactured to allow for optical access for the laser diagnostics to obtain velocity measurements. The model was placed inside a closed water circuit arrangement and additional components are manufactured to ensure a repeatable top-hat velocity flow field delivery at the entrance of the grooved part of the model. Variable volumetric flow deliveries are possible. More information can be found in a paper by Sergis and Hardalupas [10]. Normalised vector data are presented throughout this study achieved by dividing the velocity vector magnitude by the free stream velocity (6 m/s) used in the experiments. A 2D plane at the geometrical centre of the model which is perpendicular to the grooves is used to evaluate the velocity comparison process throughout this paper.

2.2. Particle image velocimetry (PIV)

The PIV method is a well-established quantitative visualisation method used widely in aerodynamic and combustion applications to measure flow velocity distribution on planes defined by a laser sheet. Small tracer particles are suspended in the coolant flow (1 μm Al_2O_3 particles) in very small volumetric concentrations (<0.00025%). The tracers are assumed small enough so that their slip velocity with the water molecules is negligible while their small concentration ensures that they do not alter the properties of the fluid they are suspended into (e.g. viscosity). The particles are illuminated with a pulsed laser sheet and their positions are imaged on a PIV CCD camera (a set of two frames is obtained a few ms apart). Comparison of the tracer locations inside the two frames using commercial PIV software yields a velocity vector field with high spatial resolution (30 μm). The process is repeated thousands of times to obtain statistics of the operation of the HV model under testing [10,11].

2.3. Computational fluid dynamics (CFD)

Two popular CFD packages have been used to reproduce the 3D HV geometry model isothermal mean velocity flow field and

compare them with those measured by the PIV method at the plane of interest. The first package is commercial but less advanced software which uses a Reynolds Averaged Navier–Stokes (RANS) solver. The build in mesh generation tool was used which employs geometry adaptive cubic cells that comprise the mesh lattice. The second package is the advanced CFD open source code openFOAM which uses an unsteady Reynolds Averaged Navier–Stokes (uRANS) and a large eddy simulation (LES) solver with a polyhedral mesh cell lattice, which adapts to both flow geometry and, progressively, according to the calculated velocity flow field throughout the simulation. Various grid resolutions are used; a coarse one, which is in par with the mesh grid resolution used widely in the field [5–9] and finer ones (see Table 1 for more information) to examine mesh size solution independence. Standard temperature and pressure (STP) are used for all simulations. The extra-fine mesh resolution is used for the LES calculation only. The solvers are chosen to keep in par with the solvers usually used in the literature.

2.3.1. SolidWorks – RANS solver

Commercial CAD software (SolidWorks CAD, Student Edition Versions 2008 to 2012) was used to reconstruct the model geometry used for the PIV experiments. RANS simulations were performed with the embedded simulation suite tool (SolidWorks Flow Simulation package). The software has been found to be adequate to perform similar type of simulations in the literature [12]. Mirroring of the domain solution is followed along the geometrical symmetry plane of the model to reduce computational time and memory occupied by the calculation. A no-slip condition was applied for the wall boundaries and a turbulence intensity of 2% at the inlet. Velocity and pressure boundary conditions are imposed at the inlet and outlet. More information on the simulation mesh properties can be found in Table 1.

2.3.2. openFOAM – uRANS and LES solver

The Version 2.1 of the open source CFD software, OpenFOAM was used to perform a uRANS and LES investigation of the HV isothermal operating condition. Meshing of the domain was performed with the software package's meshing utility, with three different mesh densities employed. The coarsest mesh of these employed six equal sized elements to span the HV groove similar to earlier investigations [5–9]. For the finer meshes, a greater number of elements was used to span the groove, whilst at the same time employing increased wall refinement; the size of elements nearest to the wall boundaries were smaller than those at the mid span location. As the meshes were refined progressively there was a reduced requirement for prismatic elements to ensure satisfactory skewness and orthogonality criteria of the domain. For all the cases investigated, the whole domain was modelled and the boundary conditions were as follows: Dirichlet for the bulk velocity at the inlet, pressure at the outlet, von Neumann for the pressure at the inlet and the velocity at the outlet. A fluctuating velocity of 5% was prescribed to the inlet. While all the remaining boundaries, being wall boundaries, had the no-slip condition imposed. For the uRANS and the LES calculations the standard $k-\epsilon$ and one equation eddy models were employed to provide closure. Similarly, for the treatment of the walls, the uRANS investigation employed the regular wall treatments and the LES employed the Spalding wall model. All quantitative results given in Table 2 are after 30 flow-through times (defined as the time required for the bulk velocity to cross the whole domain in the streamwise direction) had been calculated. For all cases, the timestep was set to 10 ns, which was required to observe the non-stationary features of the flow in the uRANS cases but more importantly to ensure that the Courant number does not exceed the value of a quarter.

Table 3
Absolute error in the vortex centroid location and max vortex velocity between the CFD and experimental data.

Mesh		Absolute error % ^a			
		Vortex centroid location		Max vortex velocity	
		Horizontally	Vertically		
Solidworks	Coarse	15	41	177	
	Fine	23	41	156	
openFOAM	Coarse	54	53	20	
	Fine	38	41	0	
	Extra fine	uRANS	31	12	0
		LES	31	53	40

^a Please note that the morphological differences of the vortex geometry (apart from centroid location) are not accounted for in this table.

independence has not been achieved even in the finest mesh results, where more than a triple mesh resolution has been used compared to the coarse mesh. Both tools deliver a poor prediction regarding the location and morphological features of the mean vortical velocity flow field established inside the middle groove of the model. Additionally, the CFD calculations overestimate the mean flow speeds encountered, especially of the coolant flow near the boundaries of the groove. An important aspect in most CFD investigations (bar DNS or very well resolved LES) is that some form of wall modelling must be employed. The LES investigation approximates the actual features of the velocity field more accurately. However; there are still discrepancies which suggest further sensitivity analysis is required. An interesting feature captured by the LES, which is observed experimentally but underestimated by uRANS, is the movement/oscillation of the vortex core, which in turn might have an effect on the net heat transfer from the boundaries. Moreover, given that the velocity field inside the groove is sensitive to the flow field around the groove (flow at the side longitudinal grooves as well as free stream flow), it also becomes questionable whether the CFD simulations are effective in calculating accurately the global velocity flow field, which in turn affects the local in-groove velocity field. Even though this was an isothermal investigation, caution must be taken when advancing to two-phase flows where both the flow speeds near the wall boundaries as well as the established flow structures will be playing a significant role to the heat transfer dynamics involved. The discrepancies might arise from failure to model accurately the wall boundaries of the geometry (mesh resolution and solver accuracy near the wall) as well as small flow features arising from the highly unstable flow observed inside the HV which appear to be significant in describing the mean as well as instantaneous operation of the devices and are not captured by the calculation.

In the light of these evidences, both CFD tools, under the isothermal cases examined, fail to predict the true velocity flow field expected inside the typical HV geometry tested. It is believed that the failure is mainly arising from the low mesh resolution used, which is inadequate to capture the full features of the flow that induce the characteristic vortical patterns observed.

However, it is noted that the current mesh resolution is better than has been reported in the literature in the past. Extreme caution should be taken when advancing to two-phase CFD simulations with an applied heat flux while using the same mesh resolutions to describe the performance of HVs.

4. Conclusions

An isothermal high spatial resolution benchmarking velocity measurement was performed to map the flow structures developing inside the groove of a typical HV geometry. Two popular

CFD packages – a commercial and an open source code – were used to numerically reproduce the velocity fields established in the geometry of the experimental study. Three mesh resolutions were used; a coarse, which describes the usual resolution found in 2-phase CFD studies in the literature, a fine and an extra-fine employed for the purpose of this study. Both CFD tools failed to predict the isothermal velocity field developing inside the geometry when compared to velocity measurements. The failure seems to arise from the inability to reach mesh independence. Extreme caution must be taken when two-phase CFD studies are performed with similar mesh resolutions. More experiments are required in a non-isothermal HV device to conclusively decipher the Vapotron effect and provide optimisation advice for this type of devices.

Acknowledgements

This work was part-funded by the RCUK Energy Programme (under grant EP/I501045). To obtain further information on the data and models underlying this paper please contact PublicationsManager@ccfe.ac.uk. The views and opinions expressed herein do not necessarily reflect those of the European Commission.

References

- [1] H.D. Falter, E. Thompson, Performance of HyperVapotron beam-stopping elements at JET, *Fusion Technol.* 29 (1996) 584–598.
- [2] G. Cattadori, G. Gaspari, G. Celata, M. Cumo, A. Mariani, G. Zummo, HyperVapotron technique in subcooled flow boiling CHF, *Exp. Therm. Fluid Sci.* 7 (1993) 230–240.
- [3] F. Escourbiac, Experimental optimisation of a HyperVapotron® concept for ITER plasma facing components, *Fusion Eng. Des.* 66–68 (2003) 301–304.
- [4] J. Milnes, D. Ciric, M. Forner, S. Gee, D. Martin, M. Nightingale, et al., HyperVapotron design for the long pulse upgrades on MAST NBI, in: 20th IEEE/NPSS Symposium on Fusion Engineering, San Diego, California, 2003, pp. 478–481.
- [5] J.H. Bullock, D.L. Youchison, M.A. Ulrickson, A comparison of two-phase computational fluid dynamics codes applied to the ITER first wall HyperVapotron, *IEEE Trans. Plasma Sci.* 38 (2009) 1704–1708.
- [6] P. Domalpalay, E. Rizzo, CFD analysis of flow boiling in the ITER first wall, *Fusion Eng. Des.* 87 (2012) 556–560.
- [7] S. Pascalribot, A. Saroli, M. Grandotto, P. Spitz, F. Escourbiac, 3D numerical simulations of HyperVapotron cooling concept, *Fusion Eng. Des.* 82 (2007) 1781–1785.
- [8] D.L. Youchison, Prediction of critical heat flux in water-cooled plasma facing components using computational fluid dynamics, *Fusion Sci. Technol.* 60 (2011) 177–184.
- [9] D.L. Youchison, Effects of HyperVapotron geometry on thermohydraulic performance, *IEEE Trans. Plasma Sci.* 40 (2012) 653–658.
- [10] A. Sergis, Y. Hardalupas, T.R. Barrett, Potential for improvement in high heat flux HyperVapotron element performance using nanofluids, *Nucl. Fusion* (2013) 53.
- [11] A. Sergis, Y. Hardalupas, T.R. Barrett, Isothermal velocity measurements in two HyperVapotron geometries using particle image velocimetry (PIV), *Exp. Therm. Fluid Sci.* 61 (2015) 48–58.
- [12] A.R. Wheeler, A.C. Klein, Modeling and analysis of a heat transport transient test facility for space nuclear systems, *Nucl. Technol.* 188 (2014) 45–62.