Flow measurement and simulation of K-DEMO blanket manifold using PIV method

Geon-Woo Kim [a], Hee Su Choe [a], Goon-Cherl Park [a], Hyoung Kyu Cho [a*], Kihak Im [b]
[a]*Dep. of Nuclear Engineering, Seoul National Univ., 1 Gwanak-ro, Gwanak-gu, Seoul 08826*
[b]*National Fusion Research Institute, 169-148 Gwahak-ro, Yuseong-gu, Daejeon 34133*
[*]*Corresponding author: chohk@snu.ac.kr*

1. Introduction

CFD (Computational Fluid Dynamics) codes have been widely used in the design of heat exchangers including conventional manifolds, and related experiments have been conducted [1]. However, there is a limitation to the evaluation of CFD codes using the experimental results of previous research, because there can be large discrepancy in the flow distribution depending on the shape of the branch pipe (size of the pipe, arrangement of branch pipes, etc.). Especially, manifold design of Korean fusion demonstration reactor (K-DEMO)'s blanket has rectangular flow channels and a narrow flow header to prevent neutron leakage and retain TBR (Tritium Breeding Ratio) [2, 3], which may lead to flow mal-distribution in the manifold. In addition, flow separation may occur in a region where the channel area changes suddenly or the channel is bent at a right angle, also, a secondary flow may occur in a channel having a rectangular cross section [4]. In order to ensure the reliability of the thermo-hydraulic analysis in the manifold, it is important to predict those phenomena accurately. This issue also leads to the problem of proper use of the computational mesh and the selection of the turbulence model in the use of CFD. From this point of view, flow measurement data is necessary for the blanket manifold design. Therefore, in this study, a flow measuring experimental apparatus was fabricated and flow distribution in the manifold was measured using PIV method. Then, the experimental result was compared with the simulation result by CFD code. Thereafter, the code validation was performed by evaluating the effect of the computational mesh and the turbulence model.

2. Flow measurement of manifold using PIV method

Experimental apparatus was designed to measure the flow in the manifold and to make validation data for CFD. For the flow measurement, PIV method [5], which is a non-intrusive method, was used. For this, a high-speed camera and laser equipment were used. Experiments were carried out at room temperature and atmospheric pressure using working fluid as water.

2.1 Experimental apparatus

As shown in Fig. 1, the manifold consists of 12 rectangular channels and 2 branch headers connecting them. The fluid entering the lower inlet header branches into 11 channels with different cross-sectional areas and meets in the upper header. Then, it descends through channel 1 and exits the test section. It was made of acrylic material for laser insertion and photographing seed particles in the PIV technique.

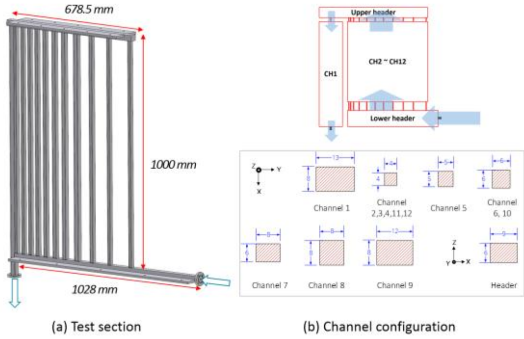

Fig. 1. Manifold test section and its channel configuration

Each of the two and three-axis traverses were installed on both sides of the test section (Fig. 2). A high-speed camera for PIV particle capturing was installed in two-axis one and a laser device for PIV seed particle emission was installed in three-axis one.

The test section was installed in a loop that can circulate the water at room temperature and pressure using a pump (Fig. 3). The pressure difference between inlet and outlet was measured using a differential pressure gauge, and the flow rate into the test section was measured using an electromagnetic flow meter.

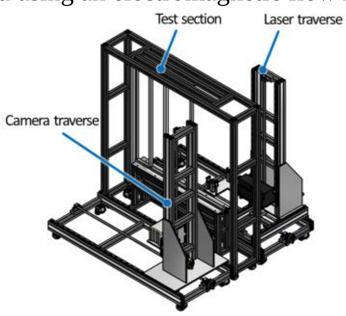

Fig. 2. Configuration of test section and two traverses

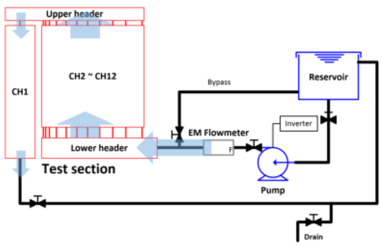

Fig. 3. Configuration of the loop

2.2 Flow measuring technique

In order to measure the flow distribution in the manifold without any disturbance of pressure drop or flow, a non-intrusive flow measurement method is necessary. The PIV technique satisfying that condition is a method of mixing a small size seed to a working fluid, which illuminates with a laser beam at a specific wavelength, and measuring the local velocity of the fluid by irradiating the laser to the fluid and capturing the movement of the particle [5]. Therefore, it is suitable for flow measuring in the manifold because it does not make any resistance in the measuring pipe.

The specific measurement method is as follows. As shown in Fig. 4, the flattened laser through the spherical and cylindrical lenses is transmitted to the side of the channel using two coated mirrors. Then, the laser irradiates the PIV seed particles (3 μm) mixed with the fluid, then they are captured for 5 seconds with a high-speed camera, and the velocity profiles in three cross-sections are measured by cross-correlation method. In order to measure fully developed flow, the flow was measured at the 90D to 225D position in the flow direction.

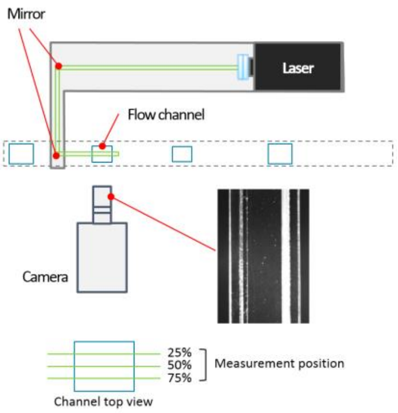

Fig. 4. PIV measurement technique in the manifold geometry

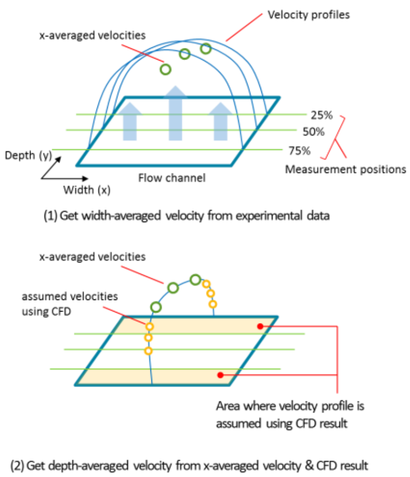

Fig. 5. Procedure to get area averaged velocity from local velocity profile

The local velocity profiles are integrated with respect to the channel area to obtain the area average velocity for each channel. (Fig. 5) In the case of near-wall velocity in the depth direction (y – axis in Fig. 5), it is difficult to obtain the velocity distribution at the near-wall due to laser scattering, so that it is obtained from the CFD data.

2.3 Experimental condition and result

As the working fluid, water at room temperature and pressure was used, and the flow rate at the inlet of the test section was adjusted by using a pump and an inverter. The experimental conditions are shown in Table 1. As shown in Fig. 6, the velocity profiles in the manifold were measured for three depths. In addition, the normalized flow distributions for each experimental condition were obtained (Fig. 7), which show similar results each other. The mass balance in the experiment was evaluated as shown in Table 2, and the maximum error was within 1.86%.

Table 1: Experimental conditions

Parameter	Case 1	Case 2
Inlet flow rate	0.1667 kg/s	0.1333 kg/s
Velocity	3.09 m/s	2.47 m/s
Pressure, temperature	1 bar, 20 °C	

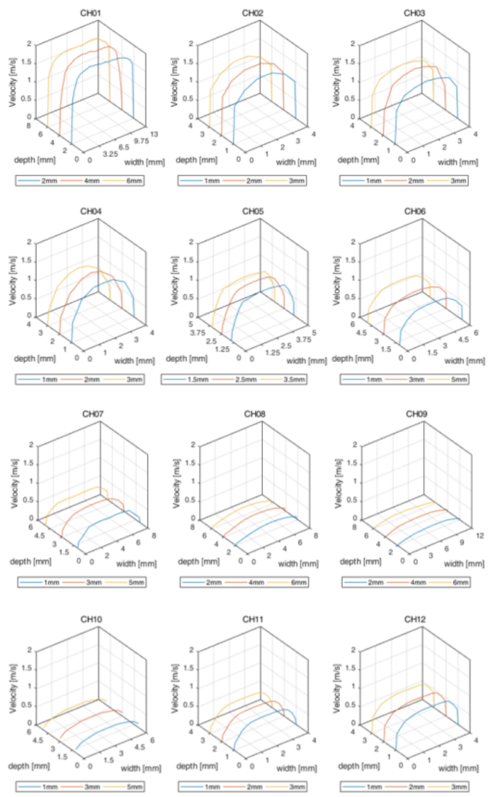

Fig. 6. Local velocity profile in the channels

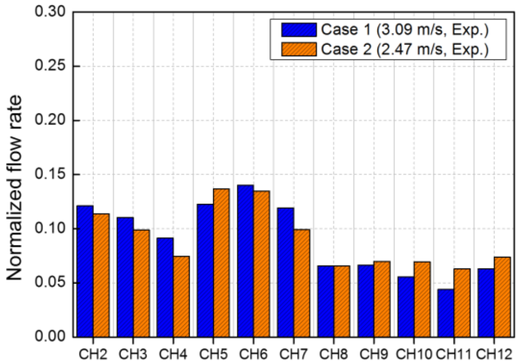

Fig. 7. Flow distribution in the manifold from experiment

Table 2: Mass balance of the experiment

Position	Error	
	Case 1	Case 2
Sum of Ch. 2~12 vs. EM flowmeter	-1.86 %	-0.23 %
Ch. 1 vs. Sum of Ch. 2~12	1.35 %	1.57 %
Ch. 1 vs. EM flowmeter	-0.54 %	1.34 %

3. Flow simulation of manifold using CFD

CFD analysis was carried out for the flow measurement experiment of the manifold to evaluate the reliability of CFD code calculation for the manifold having a small flow header.

3.1 Simulation condition

The fluid part of test section in the experiment is modeled in CFD code. The commercial CFD codes used in this study are ANSYS CFX 17.0 [6] and CD-Adapco STAR-CCM + [7]. For turbulence models, standard k-ε (SKE), realizable k-ε (RKE) and k-ω shear stress transport (SST) were applied. The number of mesh used in SKE and RKE is in the range of 1.2 to 4 million, and for SST, it is in the range of 1.2 to 10 million. For the SST model, the grid convergence was confirmed.

3.2 Turbulence model

The turbulence model used the Eddy viscosity model in the Reynolds-Averaged Navier-Stokes (RANS) model. This model expresses the nonlinear term of the velocity fluctuation as a viscosity form as shown below.

$$-\rho u_i{}'u_j{}' = \mu_t\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)-\frac{2}{3}\delta_{ij}\left(\rho k+\mu_t\frac{\partial U_k}{\partial x_k}\right) \quad (1)$$

where u' is velocity fluctuation, μ_t is turbulent viscosity, U is mean velocity, δ_{ij} is Kronecker delta, and k is turbulent kinetic energy.

The turbulence model is classified according to the modeling method of the term '$-\rho u_i{}'u_j{}'$' in Eq. 1. SKE and RKE calculate two equations for turbulent kinetic energy (k) and turbulent dissipation rate (ε), and SST computes two equations for turbulent kinetic energy (k) and specific dissipation (ω).

The SKE model [6] is widely used turbulence model and it provides reliable analysis results in wide range. However, when the rotating flow exists and the normal Reynolds stress term increases, the calculation accuracy decreases. In addition, there is a limit to apply the wall function.

The RKE model [7] is improved from SKE to predict flow separation phenomenon and the rotational flow, and modifies the dissipation rate (ε) equation based on the vorticity fluctuation.

The SST model [6] considers the transmission of turbulent shear stress and uses a blended function near the wall to use the transformed k-ε and k-ω models properly. Unlike other models, it does not use a nonlinear damping function.

In the CFD analysis of the manifold, ANSYS CFX 17.0 is used for the SKE and SST turbulence models and the CD-Adapco STAR-CCM + is used for the RKE turbulence model. Table 3 shows the near-wall velocity calculation model used for each model.

Table 3: Near-wall velocity modeling of turbulence models [6, 7]

Turbulence models	SKE (CFX)	RKE (STAR-CCM+)	SST (CFX)
Near-wall models	Scalable wall function	High y+ wall treatment	Automatic near-wall treatment

3.3 Simulation result

In the simulation result, the normalized flow distribution in the manifold and pressure drop of the test section were confirmed. Fig. 8 shows comparison results of the flow distribution between the experiment (case 1) and CFD using various turbulence models. In the case of SKE, the rms error was about 12% compared to the experiment. On the other hand, RKE and SST predict better than SKE within error of 8 ~ 10 %. The mesh convergence test was performed for the SST model.

As shown in Table 4, the SST model (10M mesh) showed good agreement in predicting pressure drop within 0.4 % error with experiment. In the local phenomena, a strong rotational flow was observed at the connecting part between channels and the inlet common header, as shown in Fig. 9. Also at the inlet of channel 1, a complex flow including a secondary flow was observed. (Fig. 9) These may lead to an additional pressure drop in the channel. It is considered that accurate prediction of the magnitude and intensity for these flow separations and rotation flows may affect the reliability of the flow distribution results.

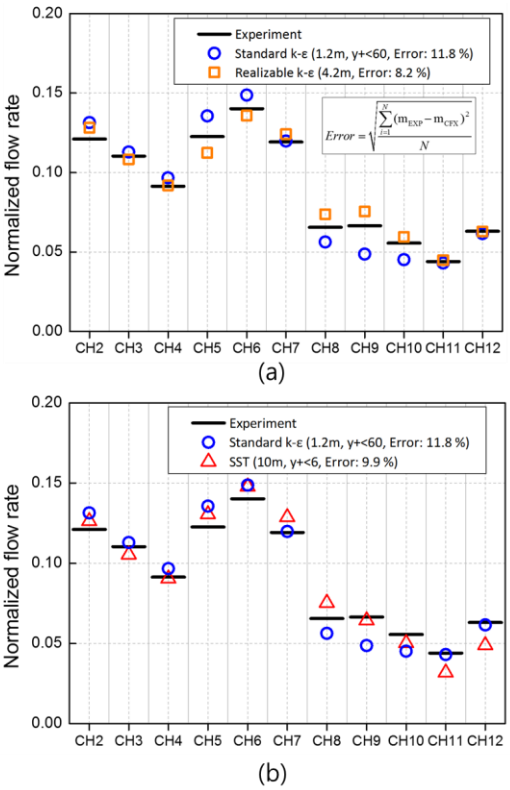

Fig. 8. Comparison result of flow distribution in the manifold between simulation and experiment
(a) SKE and RKE (b) SKE and SST

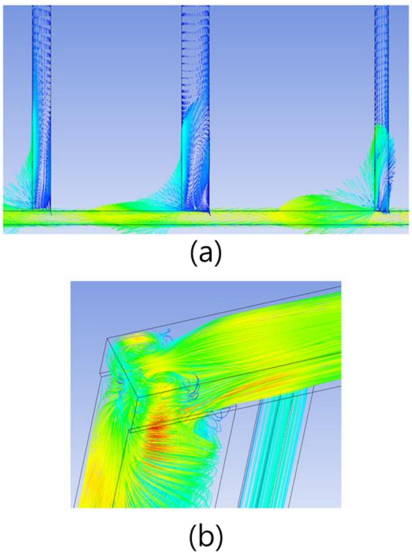

Fig. 9. Local phenomena of the fluid flow in the manifold
(a) Velocity vector in channel connecting part (b) Streamline at the inlet of channel 1

Table 4: Pressure drop results

Cases	Pressure drop [kPa]	Error [%]
Exp. (case 1)	25.3	-
SKE(1.2M)	22.8	-9.9
RKE(4.2M)	25.8	2.0
SST(1.2M)	24.7	-2.4
SST(10M)	25.2	-0.4

4. Conclusions

CFD codes are widely used for thermal hydraulic analysis of K-DEMO blanket piping. For the verification of the code as well as heat exchangers. For validation of CFD, the flow measuring experiment were carried out. For flow measurement, the PIV method was adopted. In result, the local velocity profile and flow distribution in the manifold were obtained. Thereafter, the CFD analysis of the manifold for the three turbulence models (SKE, RKE and SST) was performed and SST model shows good agreement with experiment data.

However, since the K-DEMO blanket uses pressurized water, which has similar condition with PWR, it is necessary to check the effect of the physical property variation in the experiment or analysis condition. It is expected that one can use similarity criterion for fluid between model and prototype.

Acknowledgement

This work was supported by R&D Program through the National Fusion Research Institute of Korea (NFRI) funded by the Government funds.

REFERENCES

[1] A.J. Dalton, Westinghouse AP1000 pressurized water reactor steam generator outlet plenum flow modeling, Rensselaer Polytechnic Institute Hartford, Connecticur, (2013).
[2] J. S. Park et al., Pre-conceptual design study on K-DEMO ceramic breeder blanket, Fusion Eng. Des. 100 (2015) 159-165.
[3] G.W. Kim et al., Development of Thermal-hydraulic Analysis Methodology for Multiple Modules of Water-Cooled Breeder Blanket in Fusion DEMO Reactor, Fusion Eng. Des. 103 (2016) 98-109.
[4] I. Tosun et al., Critical Reynolds number for newtonian flow in rectangular ducts, Ind. Eng. Chem. Res. 27, (1988) 1955-1957.
[5] A. Melling, A. Tracer particles and seeding for particle image velocimetry. Measurement Science and Technology. 8 (1997) 1406–1416.
[6] ANSYS, Inc., ANSYS CFX-Solver theory guide, Release 17.0, 2016.
[7] CD-adapco, STAR-CCM+ documentation: user guide version 10.04, 2015.