濟州大學校 尖端技術硏究所 論文集, 第18卷, pp.89-95, 2007. 12
Cheju Nat'l Univ. Res. Inst. Adv. Tech. Jour., Vol.18, pp.89-95, 2007. 12

CFD 기법을 이용한 Once-Through형 증기발생기의 유동장 해석

노만 말릭 무함마드* · 수브라마니안 산토시 쿠마르* · 김 신**

Once-Through Steam Generator Fluid Dynamics Analysis Using CFD Techniques

Nauman Malik Muhammad* · Subramanian Santhosh Kumar* · Sin Kim**

ABSTRACT

In this study, a proposed 3.3 MWth once through steam generator is modeled and analyzed using modern Computational Fluid Dynamics (CFD) techniques deploying CFD package FLUENT for analysis and GAMBIT for modeling purposes. The prime objective being the three-dimensional analysis of flow distribution of primary water within the steam generator and measurement of pressure drop in and across it. Recommendations are also discussed to mitigate the non-uniform flow distribution in the inner and outer modules of the steam generator.

Key Words : Once-Through Steam Generator, Computational Fluid Dynamics, Primary Water, Baffle Plates, Wrapper

Ⅰ. INTRODUCTION

CFD techniques are getting more and more popular for engineering analysis and design because of their peculiar qualities regarding analysis of flow regimes e.g. velocity profiles, pressure contours, turbulence intensities etc and heat transfer, pollutants'emissions and lot of other physical phenomenon. The detailed and precise results are the major motives to use CFD packages for design and analysis of critical plant equipment.

These tools give the confidence in design and save much time and money by simulating the equipment behavior in close proximity to real-time performance thus making one able to design equipment, less prone to failure.

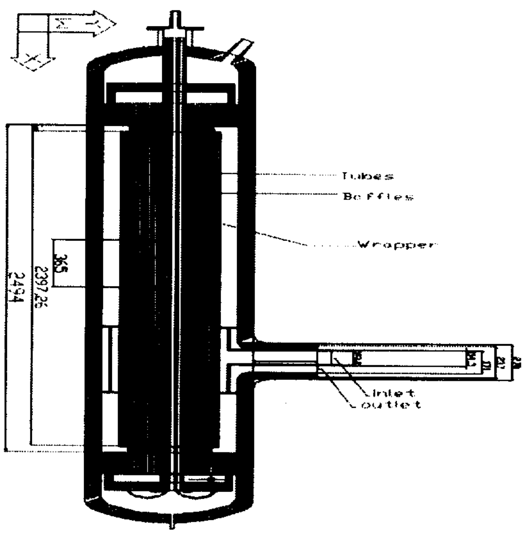
The intended analysis is of a 3.3 MWth

* 제주대학교 대학원
Graduate School, Cheju Nat'l Univ.
** 제주대학교 에너지공학과,첨단기술연구소
Dep't of Nucl. & Energy, Cheju Nat'l Univ. Res. Inst. Adv. Tech.

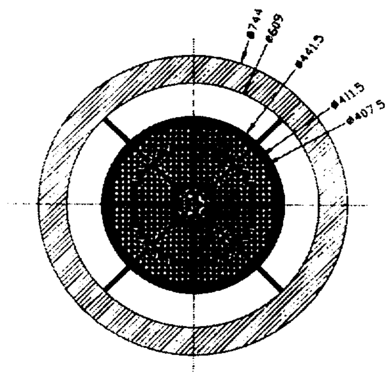
Once-Through Steam Generator which is done using Computational Fluid Dynamics (CFD) techniques. Certain simplifications have been adopted to keep the analysis straightforward but comprehensive, thus various intricate components which do not affect the analysis much are either neglected or modeled in a simplified way as mentioned in section 3.1.

Two modern state of the art CFD software packages i.e. GAMBIT 2.0 and FLUENT 6.0 have been used for the modeling and analysis of the problem. System is modeled in GAMBIT 2.0 with a mesh size suitable for the desired accuracy incorporating a refined mesh size at certain critical regions. For the solution and post-processing FLUENT 6.0.

II. STRUCTURE DESIGN



a



b

Fig. 1. a. Steam Generator Layout
b. Cross-sectional view

The OTSG design is a vertical shell and straight tube type capable of removing 3.3MWth power. The tubes are arranged in square-lattice bundle configuration. The super-heated steam is produced in the tubes which are 416 in number with 8mm outer diameter and 1mm thickness. These tubes are arranged in 16 modules, with each module having 26 tubes. A separation plate is provided between these 16 modules thus resulting in 2 separate concentric sections containing 8 independent modules in each section as shown in Fig. 1.

The hot reactor coolant coming out of the reactor core enters the inlet nozzle on the lower part of the shell. A partition plate is meant to separate the inlet and outlet primary water, the plate is present between the wrapper and the shell. The inlet and outlet nozzles are eccentric. The inlet water ascends between the wrapper and the shell and enters the tube bundle from the top through the gap between the wrapper top end and upper tube sheet. The reactor coolant transfers its heat to the secondary side fluid while flowing downwards around the tubes and then comes out of the tube bundle from bottom through the gap between wrapper end and the lower tube sheet. The feed water enters the feed water header through inlet nozzle. The feed water header distributes the incoming flow into 16 feed water modules through Feed watertubes. Feed water then enters in steam generator tube modules and is converted into super heated steam. The steam is first collected in the steam module and is then collected in the steam header, which leaves the steam generator through steam nozzle.

III. ANALYSIS SCHEME

The general CFD analysis scheme could be broken down into three characteristic stages i.e.:

i) Pre-Processing
ii) Solution
iii) Post-Processing/Results.

The Pre-Processing is carried out using commercially available software package GAMBIT 2.0, while FLUENT 6.0 is deployed for the other two stages.

3.1 Pre-Processing

The steam generator contains 16 modules in a cylindrical geometry. To cater for the GAMBIT's resource limitations and make the calculations and modeling time efficient 1/8th [1] of the total SG is modeled using GAMBIT 2.0 with the notable restriction that the inlet and outlet nozzles are disregarded for simplification as shown in Fig. 2. In addition, another notable modification is that in the separation plate of the two modules. Instead of the original multifaceted separation plate, a simple circular separation plate has been modeled. All the auxiliary components e.g. inlet and outlet nozzles, steam nozzle, steam orifices etc., which does not affect the analysis, are omitted for modeling simplicity and solution efficiency. 52 tubes are modeled [1,2] in two modules, which are separated from each other by a wall. Inlet and outlet are modeled in a very simplified way depicting only entry and exit, thus the whole model is made in a loop which is disconnectedat the inlet-outlet portion. Shell, wrapper, and tubes are modeled in real epitome. The flow is not uniform when it enters the inlet but approaches fully developed flow regime when it just enters the top space between wrapper and tube sheet. In addition, because of the symmetry of geometry, the simplification of modeling 1/8th of the steam generator is fully justified by this fact.

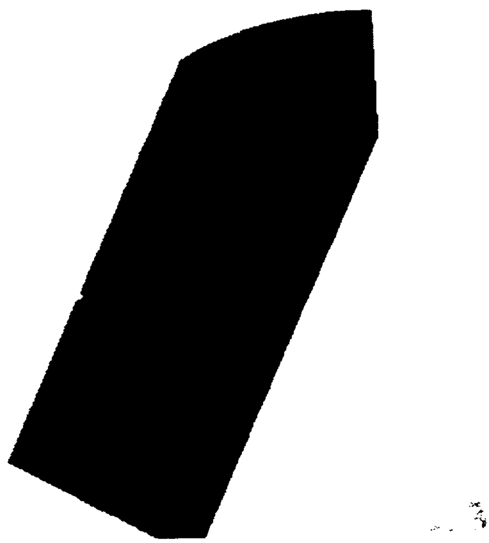


Fig. 2. 1/8th Symmetrical SG Model

3.2 Solution

The solver used is one of the most reliable in CFD industry i.e. FLUENT 6.0. Fluid parameters used are given in the following table:

Table 1. Fluid parameters

Pressure	10 MPa
Mass flow rate (1/8th of 32.26 Kg/s)	4.0325 Kg/s
Temperature	285℃
Density	746.88 Kg/ m^3
Dynamic Viscosity	0.000092558 Pa.sec

The above mentioned mass flow rate is in effect 1/8th of the original mass flow rate i.e. 32.26 Kg/s. Since velocity cannot be taken 1/8th as it remains same at any instant in a specified region and areas have been reduced, consequently mass flow rate will automatically be reduced to 1/8 justifying mass flow rate to be 4.0325 kg/s [1, 2].

3.3 Post-Processing

3.3.1 Velocity

Results have been compiled for the velocity profile at different velocity magnitudes [3] and are given in the Fig. 4 through 6. Velocity vectors emerging from the inlet and entering the tube banks are represented in Fig. 3.

The primary water with a mass flow rate of about 4.0325 Kg/s enters the gap between shell and wrapper. At the inlet, the velocity is about 0.3 m/s which rises to 0.5 m/s before entering the gap between wrapper and upper tube sheet. While passing through the 47mm gap its velocity increases to a maximum of 1.4 m/s.

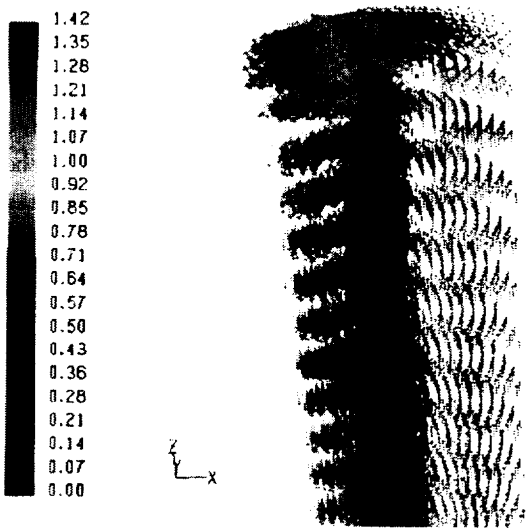


Fig. 3. Velocity Vectors (Enlarged View)

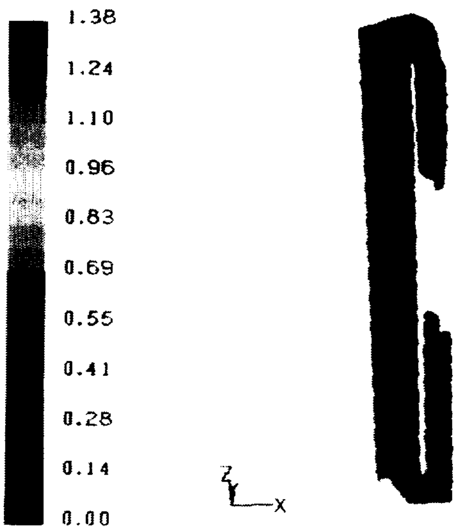


Fig. 4. Regions of SG with Vel= 0.3m/s

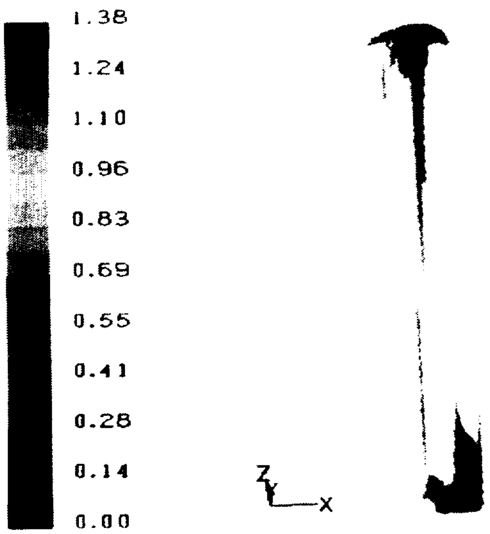


Fig. 5. Regions of SG with Vel= 0.5m/s

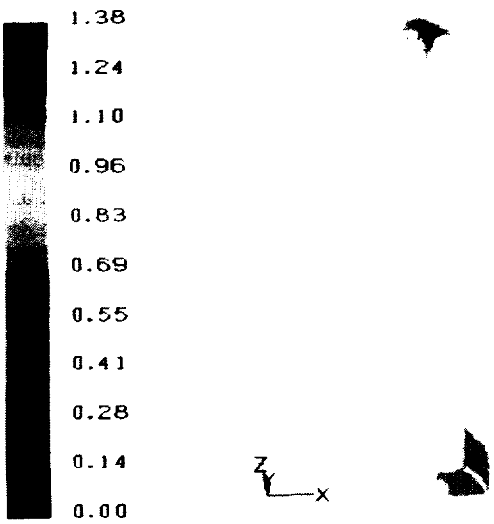


Fig. 6. Regions of SG with Vel= 0.7m/s

The velocity vectors showing this behavior are shown in Fig. 7. From here the flow changes its direction downwards in tube bank and gradually becomes uniform after passing through the first two baffles [4, 5]. The velocity in the tube bank is in the range of 0.4 m/s to 0.6 m/s. This velocity is maintained till the flow enters lower gap of 47mm as mentioned in Fig. 8 through 10. From here it again changes its direction upwards.

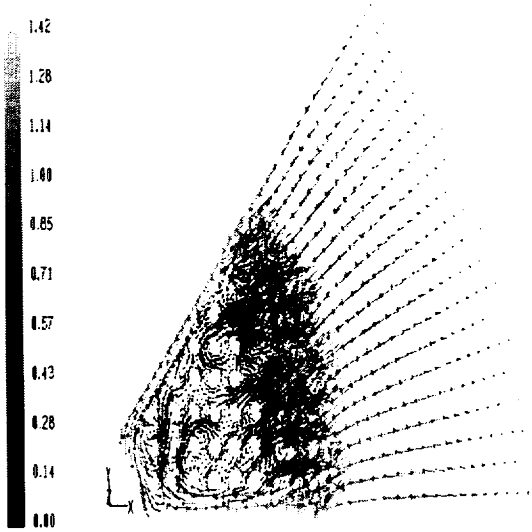


Fig. 7. Velocity Vectors Between Wrapper and Upper Tube Sheet

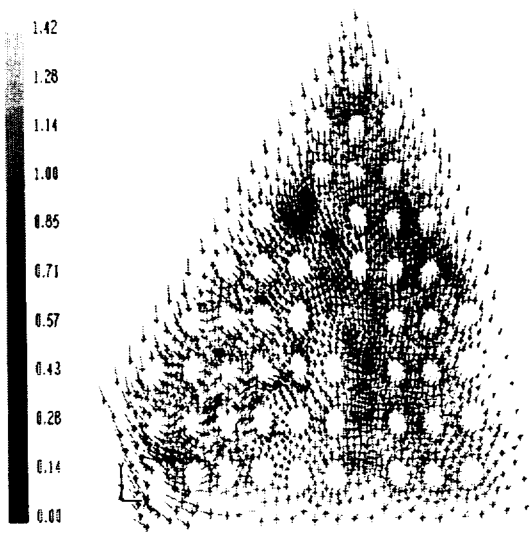


Fig. 8. Velocity Vectors Between 1st and 2nd Baffle Plates

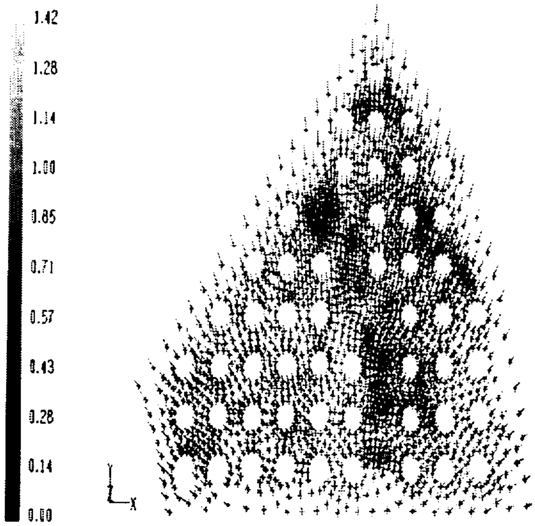


Fig. 9. Velocity Vectors Between 3rd and 4th Baffle Plates

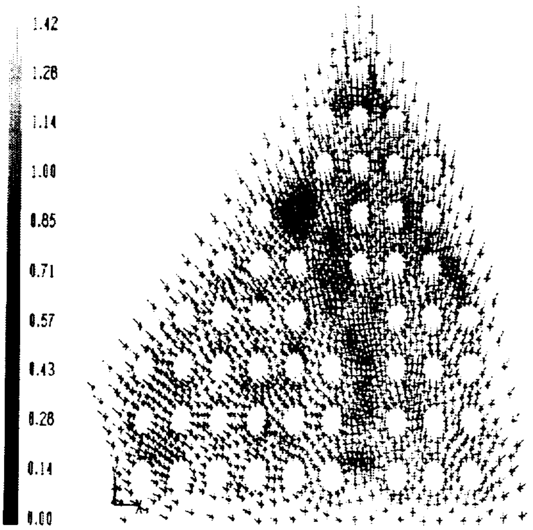


Fig. 10. Velocity Vectors Between 5th and 6th Baffle Plates

Since the turbulence increases in the lower 47mm gap velocity again increases up to a maximum of 1.4 m/s as mentioned in Fig. 11. The velocity then becomes even and reduces to about 0.2 to 0.4 m/s as the flow reaches the outlet.

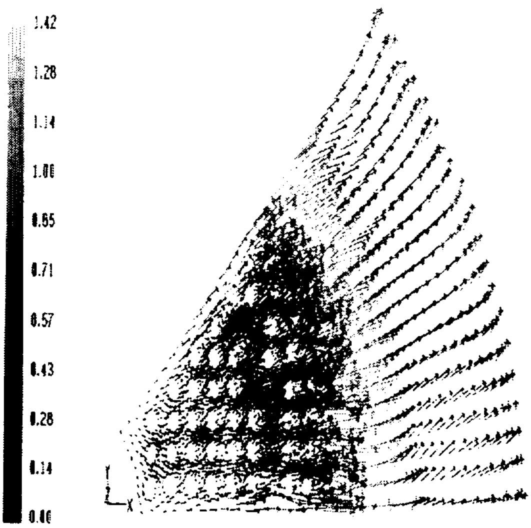


Fig. 11. Velocity Vectors Between Wrapper and Lower Tube Sheet

3.3.2 Pressure

The pressure difference between inlet and the regions, where pressure is minimum, is about 0.05 MPa and the pressure drop found between inlet and outlet is about 0.0002 MPa.

IV. CONCLUSIONS

For uniform flow distribution and heat flux, the flow should be same in each module, which, in fact is not the case as represented in Fig. 12. Because of different flow areas of the inner and outer modules and inappropriate spacing between wrapper and upper tube sheet, the results of CFD analysis show that the flow in the outer module is greater than that in the inner one i.e. in the outer module mass flow rate is about 2.46 kg/s, while it is 1.54 kg/s in the inner module, thus 62% of the total flow passes through outer module while the remaining 38% flows in the inner module. This happens due to uneven flow distribution which can be rectified by adopting either of the following suggestions:

a) The gap of 47mm between wrapper and upper tube sheet should be decreased to produce sufficient thrust to the flow so that enough coolant may reach the inner module for equal distribution between these two modules.
b) Areas of both modules should be same for equal flow distribution.
c) Wrapper at the entry and exit points of shell should be chamfered instead of sharp-edged rectangular shape to avoid the fluid stagnation and wake formations.

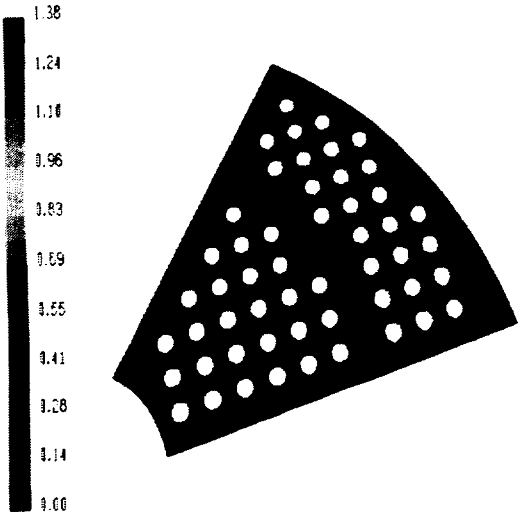


Fig. 12. Cross-Section of SG(XY-Plane) Showing Different Velocities in Two Modules

For further improvements and modifications, there is a need to explorethe matter with further detailed modeling. It is worth mentioning that the time taken, on the available computer with 300 MHz processor, by FLUENT to carry out this analysis was about 150 continuous hours. In addition, more than a month was put away by GAMBIT for the modeling process. Therefore, the available computing resources restrict the complexity of the model.

Acknowledgement

The researchers are supported by the grant from "the 2nd phase BK21 project."

References

[1] C.F. Boyd, K. Hardesty, 2003, CFD Analysis of 1/7th Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident, NUREG, NUREG-1781.

[2] C.F. Boyd, D.M. Helton, K. Hardesty, 2004, CFD Analysis of Full-Scale Steam Generator Inlet Plenum Mixing During a PWR Severe Accident, NUREG, NUREG-1788.

[3] M. Rahimi, S.S. Madaeni and K. Abbasi, 2005, CFD modeling of permeate flux in cross-flow microfiltration membrane, Journal of Membrane Science, Volume 255, pp. 23-31.

[4] Sean X. Liu, Ming Peng, Leland M. Vane, 2005, CFD simulation of effect of baffle on mass transfer in a slit-type pervaporation module, Journal of Membrane Science, Vol. 265, pp.124-136.

[5] R. Ghidossi, D. Veyret and P. Moulin, 2006, Computational fluid dynamics applied to membranes: State of the art and opportunities, Chemical Engineering and Processing, Volume 45, pp.437-454.