Implementation of a Model of Turbulence into a System Code, GAMMA+

Hyeonil Kim[a*], Hee-Cheon No[b], Hong-Sik Lim[a]
[a]*Korea Atomic Energy Research Institute, Research Reactor Core Design Division, 989-111 Daedeok-daerho Yuseong, Daejeon, 305-353, Rep. of Korea*
[b]*Korea Advanced Institute of Science and Technology, Nuclear and Quantum Engineering 291 Daehak-rho, Yuseong, Daejeon, 305-701, Rep. of Korea*

[*]*Corresponding author: hyeonilkim@kaeri.re.kr*

1. Introduction

The Launder-Sharma model was selected as the best model to predict the heat transfer performance while offsetting the lack of accuracy in even recently updated empirical correlations from both the extensive review of numerical analyses and the validation process [1] [2].

An application of the Launder-Sharma model into the system analysis code GAMMA+ for gas-cooled reactors is presented: 1) governing equations, discretization, and algebraic equations, 2) an application result of GAMMA^T, an integrated GAMMA+ code with CFD capability of low-Re resolution incorporated.

2. Methodology

2.1 Governing equations

Reynolds-Averaged Navier-Stokes (RANS) [3] equations in the conservative were selected as the set of equations governing the conservation of the mass, momentum, and energy for a single phase and single component.

The entire set of numerical methods, except for the model of turbulence, is identical to that implemented in the original system code GAMMA [4] and GAMMA+ [5]. The essentials for the model of turbulence will be explained for a fundamental understanding. Moreover, the details are expanded for a numerical analysis in a formula for a simplified single component instead of a multi-component situation, as in the original code. However, the matrix includes all elements.

The Reynolds stress is expressed in the turbulent shear stress as a product of the turbulent viscosity and mean rate of strain, as proposed by Boussinesq [3]. Options [6] to describe this turbulent viscosity are very wide depending on required level of complexity and cost. Here, the L-S model [7][8] of turbulence with two equations for k-ε was selected according to the issues that arise with the use of a system code.

Compared to the original "standard k-ε" model [9], the added function is a specific function to consider the damping effect near the wall [7]

The quantities of turbulence, i.e., the turbulent kinetic energy and the turbulent kinetic energy dissipation rate, the turbulent thermal conductivity used in the energy equation as equation was obtained using the Reynolds analogy [3] between the momentum transfer and heat transfer.

Approaching the viscous wall layer, this assumes a constant value of 0.87. Frequently, this value is assumed for the entire fully turbulent outer layer (for Pr > 0.5). When Pr is much larger or smaller than unity, this assumption no longer holds. However, because air or helium gas, the working gas in a GCR such as a GFR or a HTGR, has a Prandtl number near unity, the assumption is that a constant turbulent Prandtl number over the entire domain can be used.

2.2. Discretization

The governing equations are discretized in a semi-implicit manner in the staggered layout [10]. For a fast computation, the Revised Implicit Continuous fluid Eulerian (ICE) technique [11] rather than the original ICE technique [12] is adopted to reduce a nN×nN matrix to an 1N×1N pressure difference matrix (n: # of equations, N: size of meshes).

All of the equations for conservation are discretized as follows:

Turbulent kinetic energy:

$$\frac{(\rho k)_i^{n+1}-(\rho k)_i^n}{\Delta t}+\nabla_j\cdot\left(\dot{\rho}^n\dot{k}^n\mathbf{u_j}^{n+1}\right)=\nabla_j\cdot\left[\mu_{eff}{}^n\nabla_j\left(k^n\right)\right]+\left(P+G\right)^n-\left(\rho\varepsilon\right)_i^{n+1}+\left(\frac{\sqrt{k_j{}^n}-\sqrt{k_{j-1}{}^n}}{\Delta y}\right)^2$$

Turbulent kinetic energy dissipation rate:

$$\frac{(\rho\varepsilon)_i^{n+1}-(\rho\varepsilon)_i^n}{\Delta t}+\nabla_j\cdot\left(\dot{\rho}^n\dot{\varepsilon}^n\mathbf{u_j}^{n+1}\right)=\nabla_j\cdot\left[\mu_{eff}{}^n\nabla_j\left(\varepsilon^n\right)\right]+c_1\left(\frac{\varepsilon}{k}\right)_i^{n+1}\left(P+c_3G\right)^n-c_2\left(\rho\frac{\varepsilon^2}{k}\right)^{n+1}+2\left(\frac{\mu\mu_t}{\rho}\right)^n\left[\nabla_j\cdot\nabla_j\left(\bar{u}_z{}^n\right)\right]^2$$

Here, a bar (-) indicates the averaged property and a dot (·) indicates the donored property which depends on the flow direction. In a staggered mesh, 'i' is the index of a scalar cell and 'j' is the index of a momentum cell.

A forward time scheme was used with first-order accuracy in time. In addition, a donor scheme and a centered scheme were selected for convective terms and diffusion terms, respectively: (1) the donor scheme has first-order accuracy in space but feasible performance when considering the inherent characteristics of the phenomenon for the convective term. It is also advantageous for convergence. The (2) centered scheme with second-order accuracy was selected as it is recommended for a flow with a relatively low Reynolds number for stability and accuracy [11]

2.3. Linearization

The set of nonlinear governing equations was linearized by the Newton method [12], which is a powerful technique for solving non-linear equations as in the earlier set of equations.

Scalar variables of dependent variables such as the density, enthalpy, turbulent kinetic energy, and turbulent kinetic energy are treated implicitly and are linearized:

$$\rho^{n+1} \rightarrow \rho^k + \left(\frac{\partial \rho}{\partial P}\right)^k \delta P + \left(\frac{\partial \rho}{\partial T}\right)^k \delta T + \sum_s \left(\frac{\partial \rho}{\partial Y_s}\right)^k \delta Y_s$$

$$H^{n+1} \rightarrow H^k + \left(\frac{\partial H}{\partial T}\right)^k \delta T + \sum_s \left(\frac{\partial H}{\partial Y_s}\right)^k \delta Y_s$$

$$k^{n+1} \rightarrow k^k + \delta k$$

$$\varepsilon^{n+1} \rightarrow \varepsilon^k + \delta \varepsilon$$

2.4. Algebraic equations

By inserting linearized variables ($\rho_i^{n+1}, T_{fi}^{n+1}, k_i^{n+1}, \varepsilon_i^{n+1}$) and $\mathbf{u}_j^{n+1}$ into the discretized scalar equations, and then combining the resulting equations into a linear algebraic form, a 9×9 square matrix (considering 5 members of species) is obtained:

$$\underline{\underline{B}}\,\delta \underline{X} = \underline{b} + \underline{c}\left(\delta P_E\right) + \underline{d}\left(\delta P_W\right) + \underline{e}\left(\delta P_T\right) + \underline{f}\left(\delta P_B\right) + \underline{g}\left(\delta P_N\right) + \underline{h}\left(\delta P_S\right),$$

where $\left[\delta \underline{X}\right]^T = \left[\delta P\ \delta T_f\ \delta Y_1\ \delta Y_2\ \delta Y_3\ \delta Y_4\ \delta Y_5\ \delta k\ \delta \varepsilon\right]$

As a result, the first row becomes a pressure-only N×N matrix with diagonal dominance:

$$-\underline{d}_i'\,\delta P_{i-1} + \left(1 + \underline{d}_i' + \underline{c}_i'\right)\delta P_i - \underline{c}_i'\,\delta P_{i+1} = \underline{b}_i'$$

This pressure matrix can be solved by any direct method, such as Gaussian elimination. The remaining rows regarding the temperature, turbulent kinetic energy, and turbulent kinetic energy dissipation rate are simply expressed as a function of only the pressure:

2.4. Numerical domain, meshes, and boundary conditions

Before presenting the results from the numerical analyses, the numerical domain and the terminology in grid system will be explained and all calculations presented in the following part are based on these definitions.

The numerical calculation domain is a simple pipe in axi-symmetric coordinate system as in Figure 1 with boundary conditions such as uniform inflow, Neumann condition at outflow, axis of symmetry, and no slip boundary condition at wall (imposed as adiabatic wall or heat flux in heat transfer). The distance from inlet to outlet is defined as L, and the diameter is defined as D: since axi-symmetric coordinate is used, radius, D/2 is used for actual numerical domain.

The diameter, D is fixed to be 0.2 m because of simplicity (radius, D/2 is 0.1). And the axial distance, L is controlled according to the entrance length that is required for flow entering a pipe with uniform velocity to be fully developed since boundary condition at outlet is imposed as Neumann condition (gradient boundary condition to normal direction), that is, fully developed condition in fluid flow. The entrance length for fully developed turbulent flow is empirically estimated as $4.4\,\mathrm{Re}_D^{1/6}$ in case that inflow is uniform for smooth walls: the total axial distance from inlet to outlet is dependent on Reynolds number.

And the grid system is constructed as uniform meshes in both axial and radial direction separately. To the meshes in radial direction, special care should be paid in accordance with the numerical model of turbulence, the Launder-Sharma model, which is empowered to resolve viscous sublayer with the viscous damping function. In other words, the key in the L-S model is to solve turbulent boundary layer with consideration about low Re effect very near wall by integrating directly up to the points corresponding to extremely thin viscous layer with mathematical functions asymptotically approximating to the layer. The viscous layer is conventionally defined as below 5 in wall coordinate, $y^+ = u^* y / \nu$: u* is wall velocity (it means shear stress at wall) defined as $u^* = \sqrt{\tau_w / \rho}$, y is distance apart from wall, and ν is dynamic viscosity. Therefore the first point in the radial direction meshes

from wall should be at least located in the range ($y^+ \leq 5$) for correct calculations. In order to express this restriction, the very first point is expressed as y_p and the wall coordinate corresponding to y_p is defined as $y_p^+ = u^* y_p / \nu$.

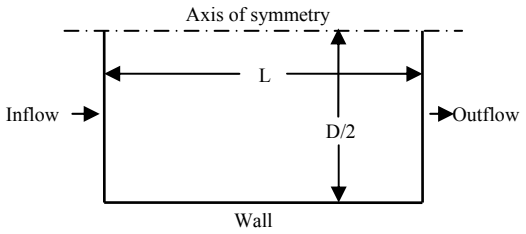

Fig. 1 Numerical domain for calculation by GAMMA^T

3. Validation of the L-S model in GAMMA^T

3.1. Integral parameters

The validation of the L-S model in GAMMA^T for heat transfer is presented here. Both the integral parameters, i.e., the skin friction coefficient for a momentum transfer and the Nusselt number, which serves as clear proof to show whether or not the heat transfer is suitably predicted, were tested.

Focusing on a selected Reynolds number, 5300, the size of the numerical domain, i.e., the axial length and the meshes in the radial direction was controlled according to the wall coordinate y_p^+. However, the meshes were only provided only for about y_p^+=1~2, since the result from the study of the sensitivities showed that meshes to radial direction of approximately y_p^+~2 provided a solution with an acceptable level of accuracy.

3.1.1. Comparison to Correlations

The skin friction coefficient was calculated using the formula by Prandtl, and the Nusselt number was calculated using a widely accepted empirical correlation by Gnielinski [15]

$$Nu_{Gnielinski} = \frac{(f/8)(Re_D - 1000)\,Pr}{1 + 12.7\sqrt{f/8}\,(Pr^{2/3} - 1)}$$

,where $f = (0.79 \ln Re_D - 1.64)^{-2}$

Only 0.3%, 12% of error for the skin friction coefficient and Nusselt number were obtained using the formula as below.

$$Error_{Nu} = (Nu_{LS} - Nu_{Gnielinki}) / Nu_{Gnielinki}$$

3.1.2. Comparison to the results from the FLUENT

Comparing the meshes adopted in Fluent, at y_p^+~1, coarser meshes around y_p^+~2, in the radial direction were used to simulate the same flow in a turbulent convection regime by GAMMA^T.

Integral parameters such as skin friction coefficient and Nusselt number were obtained for heat transfer in turbulent forced convection regime in Table 1 using GAMMA^T with even coarser meshes.

Table 1 Comparison of integral parameters: skin friction coefficient and Nusselt number between the results using Fluent and GAMMAT^T

	$C_f/C_{f,0}$		Nu/Nu_0	
	Fluent	GAMMA^T	Fluent	GAMMA^T
Forced convection at Re_D=5,300	1.02	1.00	0.97	0.99

*All values are normalized to the data from the DNS [14]

3.2. Profiles

Excellent agreement clearly resulted in the profile of the velocity, as shown in Figure 2, in a turbulent forced convection regime by GAMMA^T: all layers including the viscous layer, the transition, and the outer layer were well resolved. Furthermore, even coarser meshes in GAMMA^T were used to obtain the same level of accuracy by the costly commercial CFD, Fluent and the extremely expensive DNS packages. The Reynolds shear stress in Figure 3 by GAMMA^T also showed good agreement with the result from the Launder-Sharma model using Fluent.

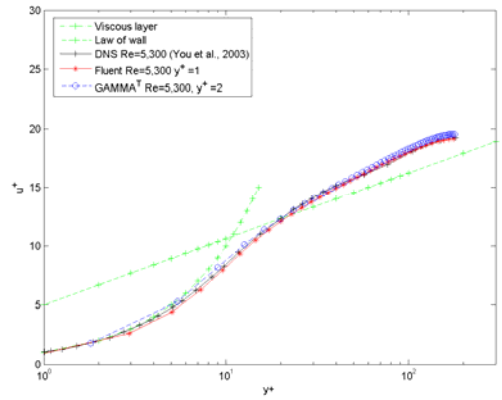

Fig. 2 Comparison of the profile of the velocity in the wall coordinate at Re=5,300 in a turbulent forced convection regime

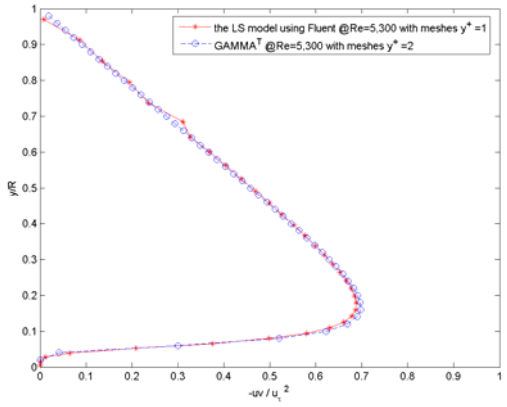

Fig. 3 Comparison of the profile of the Reynolds shear stress at Re=5,300 in a turbulent forced convection regime

3. CONCLUSIONS

With the help of improved computational capability, an application of CFD for nuclear reactor safety has been highlighted in this multi-dimensional heat transfer prediction within turbulent mixed convection regimes.

Hence, an integrated system analysis code with multi-performance capability is preferred to establish firm knowledge of physics and improve the capability of on-line multi-scale cascade analyses.

The numerical foundation was formulated and implemented in a way such that the capability of the L-S model was incorporated into GAMMA+, a system code for gas-cooled reactors, based on the same backbone of the ICE scheme on stagger mesh, that is, the code structure and numerical schemes used in the original code.

The GAMMA^T code, an integrated system code with low-Re CFD capability on board, was suitably verified using an available set of data covering a turbulent flow and turbulent forced convection. In addition, a much better solution with the same quality of prediction with fewer meshes was given. This is a considerable advantage of the application into the system code.

REFERENCES

1. Kim H., 2010, Development of a System Code with CFD Capability for Analyzing Turbulent Mixed Convection in Gas-Cooled Reactors, Ph. D. thesis. KAIST.
2. Kim H., Lee J.I. and No H.C., Thermal hydraulic behavior in the deteriorated turbulent heat transfer regime for a gas-cooled reactor. Nuclear Engineering and Design. 240, pp.783-795.
3. Schlichting Herrmann and Gersten K., 2000, Springer.
4. Lim H. S., 2005, Transient Multi-component Mixture Analysis for Air Ingress Phenomena in a High Temperature Gas-cooled Reactor, Ph. D. thesis. KAIST
5. Lim H. S., 2008, Seminar on GAMMA+ code, KAIST
6. Wilcox D.C., 2002, Turbulence modeling for CFD, 2nd ed., DCW Industries.
7. Launder B.E. and Spalding D.B., 1972, Lectures in Mathematical models of turbulence. Academic Press Inc., London.
8. Launder B.E. and Sharma B.I., 1974, Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. Int. J. Heat Mass Transfer 1, 131-138.
9. Jones W.P. and Launder B.E., 1972, The prediction of laminarization with a two-equation model of turbulence. Int. J. Heat Mass Tr10ansfer 15, 301-314.
10. Patankar, S.V., 1980, Numerical heat transfer and fluid flow, Hemisphere Publishing Corporation.
11. Harlow Francis H. and Amsden Anthony A., 1971, A Numerical Fluid Dynamics Calculation Method for All Flow Speeds. J. Chem. Phys., 8, 197-213.
12. Harlow Francis H. and Amsden Anthony A., 1968, Numerical Calculation of Almost Incompressible Flow. J. Com. Phys., 3, 80-93
13. Gnielinski V., April 1976, New Equations for Heat and Mass Transfer in Turbulent Pipe and Channel flow. International Chemical Engineering, 16, 359-387.
14. You J., Yoo J.Y., and Choi, H., 2003, Direct numerical simulation of heated vertical air flows in fully developed turbulent mixed convection. Int. J. Heat Mass Transfer 46, 1613-1627.