An Implementation of Turbulence Models into the CUPID code

I. K. Park*, J. Kim, H. Y. Yoon, H. K. Cho, J. J. Jeong
Korea Atomic Energy Research Institute,1045 Daedeok-daero, Yuseong-gu, Daejon, 305-353,Korea
**Corresponding author: gosu@kaeri.re.kr*

1. Introduction

A component-scale thermal-hydraulics analysis module, CUPID has been being developed for a transient three-dimensional two-phase flow analysis of nuclear reactor components [1]. CUPID is based on a two-fluid, three-field model, which is solved by using an unstructured finite volume method.

At present, turbulence effects are not considered in the CUPID code. However, the turbulence model is not to be neglected in a liquid single phase flow or a subcooled boiling flow even though a turbulence could be less important in other flow regimes of two phase flows. In this paper, the implementations of turbulence models into CUPID are discussed.

2. Mathematical Model and Numerical Method

The governing equations of the two-fluid, three-field model are similar to those of the time-averaged two-fluid model derived by Ishii and Hibiki[2].

$$\frac{\partial}{\partial t}(\alpha_m\rho_m) + \nabla\cdot(\alpha_m\rho_m\underline{u}_m) = \Gamma_m \tag{1}$$

$$\frac{\partial}{\partial t}(\alpha_m\rho_m\underline{u}_m) + \nabla\cdot(\alpha_m\rho_m\underline{u}_m\underline{u}_m) = -\alpha_m\nabla P + \nabla\cdot[\alpha_m\tau_m] + \alpha_m\rho_m\underline{g} + P\nabla\alpha_m + M_m^{mass} + M_m^{drag} + M_m^{VM} \tag{2}$$

$$\frac{\partial}{\partial t}[\alpha_m\rho_m e_m] + \nabla\cdot(\alpha_m\rho_m e_m\underline{u}_m) = -\nabla\cdot(\alpha_m q_m) + \nabla\alpha_m\tau_m : \nabla\underline{u}_m - P\frac{\partial}{\partial t}\alpha_m - P\nabla\cdot(\alpha_m\underline{u}_m) + I_m + Q'''_m \tag{3}$$

where α_m, ρ_m, u_m, p, Γ_m, M_m M_m, and I_m are the m-phase volume fraction, density, velocity, pressure, an interface mass transfer rate, interfacial momentum transfer, and interfacial heat transfer respectively. For a closure of the system of equations, constitutive relations and the equations of states are included.

The semi-implicit ICE scheme used in the RELAP5 code [3] was adapted to a cell-centered unstructured finite volume method. For this, the momentum equations were solved over a non-staggered grid and the velocities at the cell faces were interpolated by using the Rhie-Chow scheme[4].

3. Turbulence Model

2.1. Turbulence model for a single phase flow

The Reynolds' average of the Navier-Stokes equation resulted in the following form:

$$\frac{\partial(\rho\vec{u})}{\partial t} + \nabla\cdot(\rho\vec{u}\vec{u}) = B - \nabla p' + \nabla\cdot\left[\mu_{eff}\left(\nabla\vec{u} + \nabla\vec{u}^T\right)\right] \tag{4}$$

Where $\mu_{eff} = \mu + \mu_t$, $\quad p' = p + \frac{2}{3}\rho k + \frac{2}{3}\mu_t\nabla\cdot\vec{u} \sim p$

where μ_t is eddy viscosity or turbulent viscosity. For a nearly incompressible flow, $\nabla\cdot\vec{u}$ is zero. μ_t can be evaluated by using a turbulence models. Here, a zero equation model and standard $k-\varepsilon$ model were implemented in the CUPID code.

2.1.1. Zero equation model

In a zero equation model, the turbulence viscosity is approximated as:

$$\mu_t = \rho f_\mu U_t l_t \text{ where } l_t = \frac{1}{7}\left(V_D^{1/3}\right) \tag{5}$$

where f_μ, U_t, l_t, V_D are a proportional constant, a maximum velocity of a fluid domain, a length scale, a domain volume.

2.1.2. $k-\varepsilon$ model

In the $k-\varepsilon$ model, the turbulence viscosity is modeled as:

$$\mu_t = C_\mu\rho\frac{k^2}{\varepsilon} \tag{6}$$

where k and ε are the turbulence kinetic energy and dissipation rate. These terms can be obtained by the following transport equations:

$$\frac{\partial(\rho k)}{\partial t} + \nabla\cdot(\rho k\vec{u}) = \nabla\cdot\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\nabla k\right] + P + G - \rho\varepsilon \tag{7}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \nabla\cdot(\rho\varepsilon\vec{u}) = \nabla\cdot\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon\right] + C_{\varepsilon 1}\frac{\varepsilon}{k}\left(P + C_{\varepsilon 3}\max(G,0)\right) - C_{\varepsilon 2m}\rho\frac{\varepsilon^2}{k} \tag{8}$$

where $\sigma_t, \sigma_\varepsilon, P, G$ are a turbulent Prandtl number(~1.0), a dissipation Prandtl number(~1.3), a shear production, and a production due to a body force.

A wall function to describe a logarithmic relation between the dimensionless distance from the wall and the dimensionless near-wall velocity was used for compensating coarse meshes near the wall

2.2. Turbulence model for a two-phase phase flow

The Reynolds' average of the phasic momentum equations results in the following form:

$$\frac{\partial(\alpha_m\rho_m\vec{u}_m)}{\partial t} + \nabla\cdot(\alpha_m\rho_m\vec{u}_m\vec{u}_m) = B_m - \nabla p + \nabla\cdot\left[\alpha_m\mu_{eff}\left(\nabla\vec{u}_m + \nabla\vec{u}_m^{\,T}\right)\right] \tag{9}$$

The zero equation model and the $k-\varepsilon$ model are modified as:

$$\mu_{m,t} = \alpha_m\rho_m f_\mu U_{m,\tan} l_t \tag{10}$$

$$\mu_t = C_\mu\rho\frac{k_m^2}{\varepsilon_m} \tag{11}$$

$$\frac{\partial(\alpha_m\rho_m k_m)}{\partial t}+\nabla\cdot(\alpha_m\rho_m k_m\vec{u}_m)=\nabla\cdot\left[\alpha_m\left(\mu+\frac{\mu_t}{\sigma_k}\right)_l\nabla k_m\right]+\alpha_m P_m$$
$$+\alpha_m G_m-\alpha_m\rho_m\varepsilon_m+\sum_{n=1}^{Np}c_{mn}^{k}(k_n-k_m)+\sum_{n=1}^{Np}(m_{mn}k_n-m_{nm}k_m) \qquad (12)$$

$$\frac{\partial(\alpha_m\rho_m\varepsilon_m)}{\partial t}+\nabla\cdot(\alpha_m\rho_m\varepsilon_m\vec{u}_m)=\nabla\cdot\left[\alpha_m\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)_l\nabla\varepsilon_m\right]$$
$$+C_{\varepsilon1}\alpha_m\frac{\varepsilon_m}{k_m}(P_m+C_{\varepsilon3}\max(G_m,0))-C_{\varepsilon2}\alpha_m\rho_m\frac{\varepsilon_m^2}{k}+\sum_{n=1}^{Np}c_{mn}^{\varepsilon}(\varepsilon_n-\varepsilon_m)+\sum_{n=1}^{Np}(m_{mn}\varepsilon_n-m_{nm}\varepsilon_m) \qquad (13)$$

c_{mn}, m_{mn} are a interfacial k or ε transfer coefficient between m and n, a mass flow rate into m from n phase.

For the wall function, the same strategies as the single phase turbulence model are adopted for the two-phase flow turbulence model.

For the dispersed two phase flow, using the Sato model or declaring the disperse phase to be laminar is a general approach. The effect of a turbulence in the continuous phase on a turbulence in the disperse phase can be modeled by setting the disperse phase viscosity proportional to the continuous phase eddy viscosity:

$$\mu_{t,d}=\frac{\mu_{t,c}}{\sigma_{t,d}} \qquad (14)$$

Alternatively, the effect of turbulence in the continuous phase on the disperse phase can be modeled using the discrete particle model. For non-disperse gas-liquid flows, using the homogenous turbulence model is an alternative. The remaining things are what are the non-disperse gas-liquid flows and how to treat a discontinuity among the flow regimes.

4. Preliminary Calculation for the Implementation Steps of the Turbulence Models

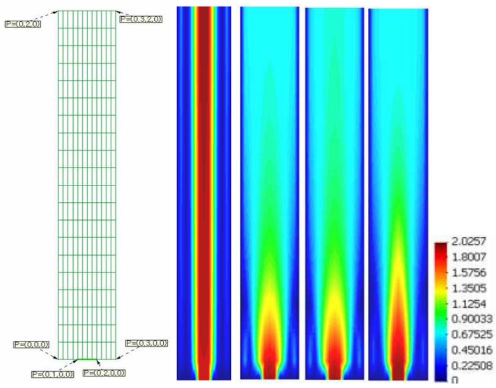

Fig.1 Configuration of the geometrical conditions of the test section and velocities profiles for implementation steps of Turbulence Models: No Model, Zero Equation, Zero Equation with Wall Function, $k-\varepsilon$ Model with Wall Function.

The turbulence models only for a liquid phase are implemented and tested.

The test problem was a 2-dimensional rectangular tube of 0.3 m x 2.0 m filled with subcooled water, and the subcooled water was being injected into its inlet at 0.1m/sec which is located in the center bottom. The outlet was assumed to be a constant pressure boundary of 0.1 MPa. The geometrical conditions and the overall velocity profiles are presented in Fig.1 and the cross-sectional velocity profiles at the exit are shown in Fig.2.

Fig.2 shows that the wall function is essential for the realistic velocity profile, and the zero equation model is considerably useful for the initial stage of the calculation or the numerical test.

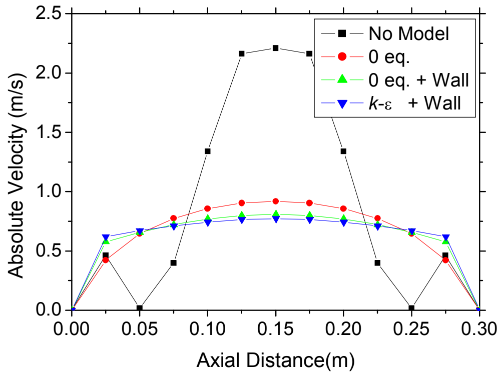

Fig.2 Comparison of the velocity profiles by the implementation of the turbulence models: None, 0 equation, wall Function, $k-\varepsilon$ model.

5. Conclusions

Turbulence models were implemented into the CUPID, a 3-dimensional, 2-fluid, 3-field code. The implemented turbulence models worked as expected and didn't cause any numerical problems for the CUPID code.

ACKNOWLEDGMENTS

This study has been carried out under the nuclear R&D program by the Korean Ministry of Education, Science and Technology.

REFERENCES

[1] J.J. Jeong, H.Y. Yoon, H.K. Cho, J. Kim, I.K. Park, A Semi-implicit numerical scheme for a transient two-fluid three-field model on an unstructured grid, Int. Comm. Heat and Mass Transfer, 35, 597–605, 2008.
[2] M. Ishii, T. Hibiki, Thermo-fluid Dynamics of Two-Phase Flow, Springer, pp. 155-169, 2006.
[3] The RELAP5-3D Code Development Team, RELAP5-3D© Code Manual Volume I: Code Structure, System Models and Solution Methods, Idaho National Engineering and Environmental Laboratory, Feb. 2001.
[4] C. M. RHIE, W. L. CHOW, Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation", AIAA Journal, Vol. 21, No. 11, pp. 1525-1532, 1983.