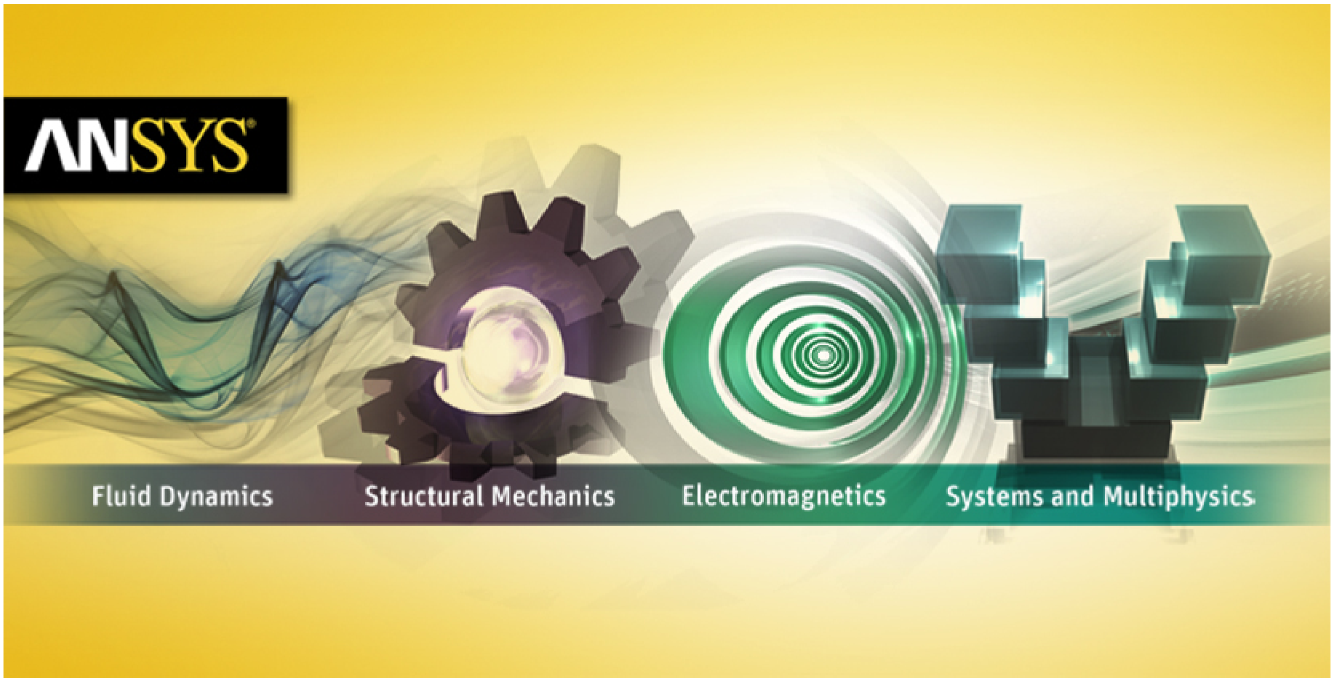


ANSYS FLUENT V2F Turbulence Model Manual

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 14.0
November 2011

ANSYS, Inc. is certified to ISO 9001:2008.

Published in the U.S.A.

Table of Contents

Chapter 1: Introduction

Successful modeling of the separation of fluid from a curved surface (for example, the suction side of an airfoil) depends on the ability to correctly predict the stall angle. For such cases, eddy-viscosity turbulence models, such as the k- ε models, are not satisfactory because they can sometimes overpredict the turbulence kinetic energy and are not sensitive to the interaction between streamline curvature and turbulence anisotropy. The Reynolds-stress model (RSM), on the other hand, accounts for several turbulence features that are not well predicted by eddy-viscosity models, but is substantially more complex and sometimes is numerically unstable.

The $v^2 - f$ model (V2F), based on Durbin's k- ε- v^2 model [2] (p. 17), is an alternative to eddy-viscosity models and the RSM. The $v^2 - f$ model is similar to the standard k- ε model, but incorporates near-wall turbulence anisotropy and non-local pressure-strain effects. It is a general low-Reynolds-number turbulence model that is valid all the way up to solid walls, and therefore does not need to make use of wall functions. Although the model was originally developed for attached or mildly separated boundary layers, it also accurately simulates flows dominated by separation.

This document describes the ANSYS FLUENT $v^2 - f$ model. *V2F Model Theory* (p. 3) provides theoretical background information. *Problem Setup Using the V2F Model* (p. 7) describes how to set up a problem using the $v^2 - f$ model. *Solution Strategies for the V2F Model* (p. 13) describes the solution procedure for problems involving the $v^2 - f$ model, and *Postprocessing for the V2F Model* (p. 15) provides information about postprocessing options.

Chapter 2: V2F Model Theory

The $v^2 - f$ model is a four-equation model based on transport equations for the turbulence kinetic energy (k), its dissipation rate (ε), a velocity variance scale ($\overline{v^2}$), and an elliptic relaxation function (f).

The distinguishing feature of the $v^2 - f$ model is its use of the velocity scale, $\overline{v^2}$, instead of the turbulent kinetic energy, k, for evaluating the eddy viscosity. $\overline{v^2}$, which can be thought of as the velocity fluctuation normal to the streamlines, has shown to provide the right scaling in representing the damping of turbulent transport close to the wall, a feature that k does not provide.

The $v^2 - f$ turbulence model theory is described in the following sections:

2.1. Transport Equations for the V2F Model

The turbulence kinetic energy, k, its rate of dissipation, ε, the velocity variance scale, $\overline{v^2}$, and the elliptic relaxation function, f, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = P - \rho\varepsilon + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + S_k \tag{2–1}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{C'_{\varepsilon 1}P - C_{\varepsilon 2}\rho\varepsilon}{T} + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + S_\varepsilon \tag{2–2}$$

$$\frac{\partial}{\partial t}\left(\rho\overline{v^2}\right) + \frac{\partial}{\partial x_i}\left(\rho\overline{v^2}u_i\right) = \rho k f - 6\rho\overline{v^2}\frac{\varepsilon}{k} + \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \overline{v^2}}{\partial x_j}\right] + S_{\overline{v^2}} \tag{2–3}$$

$$f - L^2\frac{\partial^2 f}{\partial x_j^2} = (C_1 - 1)\,\frac{\frac{2}{3} - \overline{v^2}/k}{T} + C_2\frac{P}{\rho k} + \frac{5\overline{v^2}/k}{T} + S_f \tag{2–4}$$

where

$$P = 2\mu_t S^2, S^2 \equiv S_{ij} S_{ij}, S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{2–5}$$

The turbulent time scale T and length scale L are defined by

$$T' = \max\left[\frac{k}{\varepsilon}, 6\sqrt{\frac{v}{\varepsilon}}\right] \tag{2–6}$$

$$T = \min\left[T', \frac{\alpha}{\sqrt{3}} \frac{k}{\overline{v^2} C_\mu \sqrt{2S^2}}\right] \tag{2–7}$$

$$L' = \min\left[\frac{k^{3/2}}{\varepsilon}, \frac{1}{\sqrt{3}} \frac{k^{3/2}}{\overline{v^2} C_\mu \sqrt{2S^2}}\right] \tag{2–8}$$

$$L = C_L \max\left[L', C_\eta \left(\frac{v^3}{\varepsilon}\right)^{1/4}\right] \tag{2–9}$$

In the above equations, α, C_1, C_2, $C_{\varepsilon 1}$, $C'_{\varepsilon 1}$, $C_{\varepsilon 2}$, C_η, C_μ, and C_L are constants. σ_k and σ_ε are the turbulent Prandtl numbers for k and ε. S_k, S_ε, $S_{\overline{v^2}}$, and S_f are user-defined source terms and v is the kinematic viscosity (μ/ρ).

The variable f is the solution to the elliptic relaxation equation (*Equation 2–4* (p. 3)). Here, the $v^2 - f$ model uses an elliptic operator to compute a term analogous to the pressure-strain correlation of the RSM. Ellipticity is characterized by a modified Helmholtz operator, which introduces wall effects via a linear differential equation.

2.2. Modeling the Turbulent Viscosity

The turbulent (or eddy) viscosity, μ_t, is defined as follows:

$$\mu_t = \rho C_\mu \overline{v^2} T \tag{2–10}$$

2.3. Model Constants

The model constants have the following default values [*1*] (p. 17), [*3*] (p. 17):

$$\alpha = 0.6, C_1 = 1.4, C_2 = 0.3, C_{\varepsilon 1} = 1.4, C_{\varepsilon 2} = 1.9, C_\eta = 70,$$

$$C_\mu = 0.22, C_L = 0.23, \sigma_k = 1, \sigma_\varepsilon = 1.3, C'_{\varepsilon 1} = C_{\varepsilon 1}\left(1 + 0.045\sqrt{k/\overline{v^2}}\right)$$

Chapter 3: Problem Setup Using the V2F Model

To include the $v^2 - f$ turbulence model in your ANSYS FLUENT simulation, you need to activate the model and options, and supply turbulent boundary conditions. These inputs are described in the following sections:

3.1. Enabling the V2F Model

The following is a description of the procedure for setting up a $v^2 - f$ problem.

Note

This procedure includes only the steps necessary for the turbulence model itself; you will need to set up other models, boundary conditions, etc. as usual. See the ANSYS FLUENT User's Guide for details.

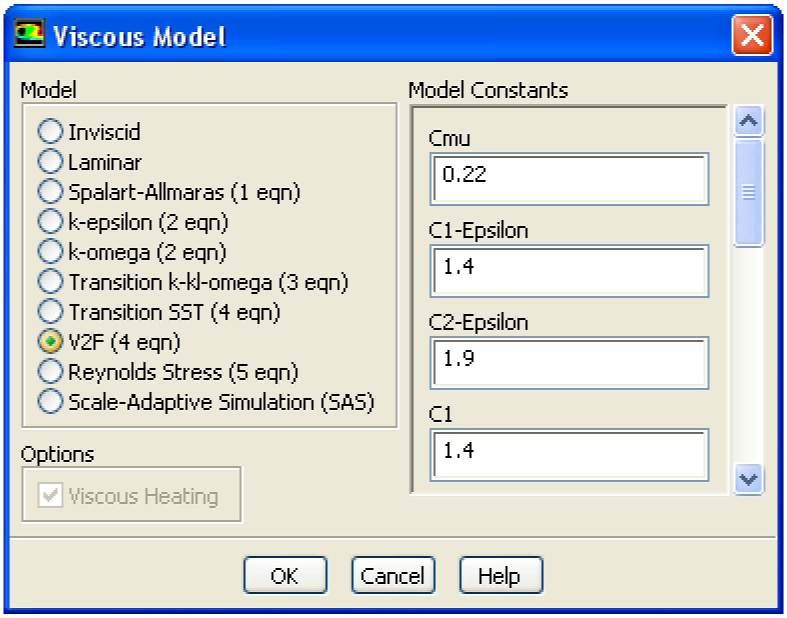
1. To enable the selection of the $v^2 - f$ model, enter the following Scheme command in the main menu of the ANSYS FLUENT console:

```
(allow-v2f-model)
```

2. To activate the $v^2 - f$ model, select **V2F** under **Model** in the **Viscous Model** dialog box (*Figure 3.1* (p. 8)).

Models → **Viscous** → **Edit...**

Figure 3.1 The Viscous Model Dialog Box with the V2F Option Available



3. Specify or confirm the **Model Constants** used in the $v^2 - f$ transport equations.

Cmu
: is the constant C_μ in *Equation 2–7* (p. 4), *Equation 2–8* (p. 4), and *Equation 2–10* (p. 4).

C1-Epsilon
: is the constant $C_{\varepsilon 1}$.

C2-Epsilon
: is the constant $C_{\varepsilon 2}$ in *Equation 2–2* (p. 3).

C1
: is the constant C_1 in *Equation 2–4* (p. 3).

C2
: is the constant C_2 in *Equation 2–4* (p. 3).

Ceta
: is the constant C_η in *Equation 2–9* (p. 4).

Cl
: is the constant C_L in *Equation 2–9* (p. 4).

Alpha
: is the constant α in *Equation 2–7* (p. 4).

TKE Prandtl Number
: is the effective Prandtl number for transport of turbulence kinetic energy σ_k. This effective Prandtl number defines the ratio of the momentum diffusivity to the diffusivity of turbulence kinetic energy via turbulent transport.

TDR Prandtl Number
: is the effective Prandtl number for transport of turbulence dissipation rate σ_ε. This effective Prandtl number defines the ratio of the momentum diffusivity to the diffusivity of turbulence dissipation via turbulent transport.

Important

After you have selected **V2F** and clicked **OK**, ANSYS FLUENT will check to make sure that a license is available for this module. After the module has been enabled successfully, the item **v2f** will appear in the list of models in square brackets in the title bar of the ANSYS FLUENT console.

4. Specify the boundary conditions for the solution variables.

   Boundary Conditions

   See the section that follows (*Defining V2F Boundary Conditions* (p. 9)) and the ANSYS FLUENT User's Guide for details.

5. Specify the initial guess for the solution variables.

   Solution Initialization

   See the ANSYS FLUENT User's Guide for details.

3.2. Defining V2F Boundary Conditions

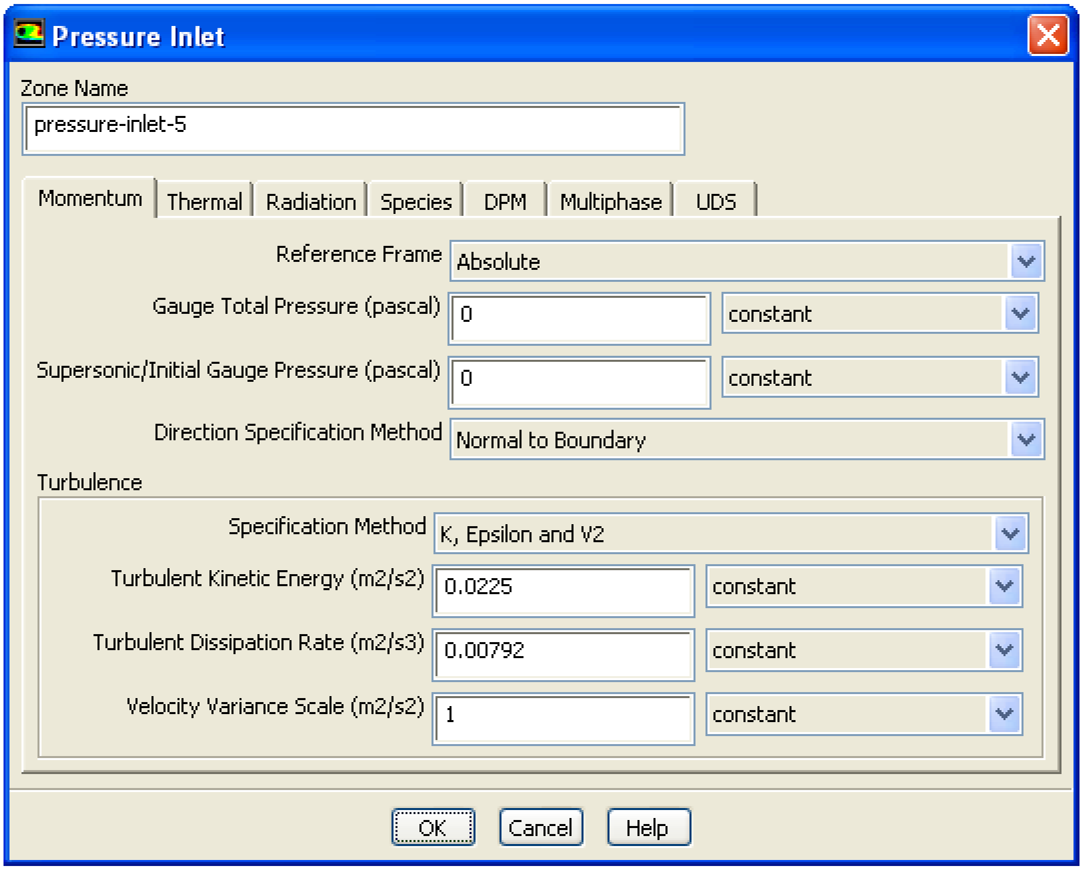
When you are modeling turbulent flows in ANSYS FLUENT using the $v^2 - f$ model, you must provide the boundary conditions for k, ε, and $\overline{v^2}$ in addition to other mean solution variables. The boundary conditions for k, ε, and $\overline{v^2}$ at the walls are internally taken care of by ANSYS FLUENT, which obviates the need for your inputs. You must supply ANSYS FLUENT with boundary condition inputs for k, ε, and $\overline{v^2}$ at inlet boundaries (velocity inlet, pressure inlet, etc.). In many situations, it is important to specify correct or realistic boundary conditions at the inlets, because the inlet turbulence can significantly affect the downstream flow.

To define inlet boundary conditions specific to the $v^2 - f$ model, use the following procedure:

1. Open the appropriate boundary condition dialog box (e.g., *Figure 3.2* (p. 10)).

   Boundary Conditions

Figure 3.2 Specifying Inlet Boundary Conditions for the V2F Model



2. Make a selection from the **Specification Method** drop-down list in the **Turbulence** group box.
   - If you select **K, Epsilon and V2**, specify values for the **Turbulent Kinetic Energy**, **Turbulent Dissipation Rate**, and **Velocity Variance Scale**, as appropriate.
   - If you select any of the other options (e.g., **Intensity and Viscosity Ratio**), ANSYS FLUENT will automatically set the value of $\overline{v^2}$ to $\frac{2}{3}k$ at the inlet.

Important

Note that ANSYS FLUENT automatically assumes a zero-gradient boundary condition for the variable f at inlets. You can change the value of f when you initialize a solution, but the default value of 1 is acceptable in most cases.

See Section 7.2.2 in the ANSYS FLUENT User's Guide for more information about specifying the boundary conditions for k and ε at the inlets.

3.3. Providing an Initial Guess for k, ε, and the Velocity Variance Scale

For flows using the $v^2 - f$ model, the converged solutions or (for unsteady calculations) the solutions after a sufficiently long time has elapsed should be independent of the initial values for k, ε, and the velocity variance Scale $\overline{v^2}$. For better convergence, it is beneficial to use a reasonable initial guess for k, ε, and $\overline{v^2}$.

In general, it is recommended that you start from a fully-developed state of turbulence, using the following guidelines.

- If you were able to specify reasonable boundary conditions at the inlet, it may be a good idea to compute the initial values for k, ε, and $\overline{v^2}$ in the whole domain from these boundary values. (See Section 26.15 in the ANSYS FLUENT User's Guide for details.)
- For more complex flows (e.g., flows with multiple inlets with different conditions) it may be better to specify the initial values in terms of turbulence intensity. 5–10% is enough to represent fully-developed turbulence. k can then be computed from the turbulence intensity and the characteristic mean velocity magnitude of your problem ($k = 1.5\left(Iu_{avg}\right)^2$).

  You should specify an initial guess for $\overline{v^2}$ so that the resulting eddy viscosity ($\rho C_\mu \overline{v^2} T$) is sufficiently large in comparison to the molecular viscosity. In fully-developed turbulence, the turbulent viscosity is roughly two orders of magnitude larger than the molecular viscosity. From this, you can compute $\overline{v^2}$. Alternatively, you can use the approximation $\overline{v^2} = \frac{2}{3}k$, or, if you have experimental measurements, you can enter a profile for $\overline{v^2}$. (See Section 7.26 in the ANSYS FLUENT User's Guide for details about boundary profiles.)

Chapter 4: Solution Strategies for the V2F Model

For a simulation involving the $v^2 - f$ model, you should use the following procedure to achieve full convergence:

1. Start by converging a flow simulation using one of the k- ε models (for example, realizable k- ε model).
2. In the **Viscous Model** dialog box, change the **Model** to **V2F**.

   Models → **Viscous** → **Edit...**

3. Define a custom field function (for example, v2) as $\frac{2}{3}k$, where k is the turbulence kinetic energy. See Section 11.13.1 in the ANSYS FLUENT User's Guide for information about other custom field functions that may be useful for turbulence. For more general information about custom field functions, see Section 30.5.

   Define → **Custom Field Functions...**

4. Patch a value for the velocity variance scale in all fluid zones using the field function created in the previous step (for example, v2).
5. In the **Solution Methods** task page, make sure that the **Velocity Variance Scale** and the **Elliptic Relaxation Function** have the same **Spatial Discretization** schemes as the **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**.

   Solution Methods

6. Continue running the calculation using the $v^2 - f$ model.

Chapter 5: Postprocessing for the V2F Model

ANSYS FLUENT provides postprocessing options for displaying, plotting, and reporting various turbulence quantities, which include the main solution variables and other auxiliary quantities.

Turbulence quantities that can be reported for the $v^2 - f$ model are:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Velocity Variance Scale (v2)**
- **Elliptic Relaxation Function**
- **Production of k**
- **Turbulent Viscosity**
- **Subgrid Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**

Bibliography

[1] M. Behnia, S. Parneix, Y. Shabany, and P. A. Durbin."*Numerical Study of Turbulent Heat Transfer in Confined and Unconfined Impinging Jets*". *International Journal of Heat and Fluid Flow*. 20. 1-9. 1999.

[2] P. A.Durbin. "*Separated Flow Computations with the k-epsilon-v2 Model*". *AIAA Journal*. 33(4). 659–664. 1995.

[3] S. Parneix, P. A. Durbin, and M. Behnia."*Computation of a 3D Turbulent Boundary Layer Using the V2F Model*". *Flow Turbulence and Combustion*. 10. 19–46. 1998.