Simcenter™ Flotherm™ Background Theory Reference Guide

Software Version 2021.1

SIEMENS

Support Center: support.sw.siemens.com
Send Feedback on Documentation: support.sw.siemens.com/doc_feedback_form

Table of Contents

Chapter 1
Background of Computational Fluid Dynamics (CFD) 9

Overview 9
Mathematical Representation 9
The Solution Method 10
Grid Definition 12
Interpretation of Results 12

Chapter 2
Finite Volume Equations 15

Derivation of Finite-Volume Equations 15
Electrical Potential Equations 18

Chapter 3
Wall Functions 21

Friction Effects at Smooth Walls for all Turbulent Options 21
Heat Transfer at Smooth Walls 23
Treatment of Rough Walls 24

Chapter 4
Turbulence Viscosity Model 25

Turbulent Viscosity Estimation 25
Level of Turbulence Viscosity Estimation 26
The Formula Chosen for the Estimation of the Level of Turbulence Viscosity 26
Turbulence Models 29
Automatic Algebraic Turbulence Model 29
Revised Algebraic Turbulence Model 29
LVEL K-Epsilon Turbulence Model 29
Near Wall Treatment 30
References 31

Chapter 5
Radiation Model 33

Radiative Exchange 33
Radiating and Blocking Objects 36
View Factor 39
Calculation of View Factors 39
Costs and Recommendations 40
Generating a View Factor Log 41

Chapter 6
Further Reading 43

Index

List of Figures

Figure 1-1. Layout and Velocities for a Grid. 10
Figure 1-2. Subdivided Grid 11
Figure 1-3. Staggered Vectors 13
Figure 3-1. Turbulent and Laminar Flow at Smooth Walls. 22
Figure 3-2. Roughness Height 24
Figure 4-1. Tree of Turbulence Modeling 27
Figure 5-1. Radiation Emitted from Surface 1 Reaching Surface 2 33
Figure 5-2. View Factor. 39
Figure 5-3. Example View Factors Output 42

List of Tables

Table 5-1. Radiating Objects 36
Table 5-2. Blocking Objects 37

Chapter 1
Background of Computational Fluid Dynamics (CFD)

A brief description of the CFD solution techniques used in Simcenter™ Flotherm™ software.

Overview 9
Mathematical Representation 9
The Solution Method 10
Grid Definition 12
Interpretation of Results 12

Overview

The solution procedures in Simcenter Flotherm are based on CFD techniques. CFD is concerned with the numerical simulation of fluid flow, heat transfer and related processes such as radiation.

The objective of CFD is to provide the engineer with a computer-based predictive tool that enables the analysis of the air-flow processes occurring within and around electronics equipment, with the aim of improving and optimizing the design of new or existing equipment.

The CFD techniques that are in use today evolved from techniques developed in response to the stimuli of the ‘high-technology’ aerospace and nuclear power industries. Today CFD applications are to be found in many industries, research institutes and universities.

Mathematical Representation

The mathematical simulation of fluid flow and heat transfer phenomena involves the solution of a set of coupled, non-linear, second order, partial differential equations.

Simcenter Flotherm uses what is known as the primitive variable treatment in that the field variables that it solves are:

- u, v and w, the velocity resolutes in cartesian coordinate directions x, y and z,
- p the pressure,
- T the temperature of the fluid and/or solid materials.

These variables are functions of x, y, z and time.

The differential equations that these field variables satisfy are referred to as conservation equations. For example *u*, *v* and *w* satisfy the momentum conservation equations in the three coordinate directions. Temperature satisfies the conservation equation of thermal energy. The pressure does not itself satisfy a conservation equation, but is derived from the equation of continuity which is a statement in differential form of the conservation of mass.

Domain of Integration

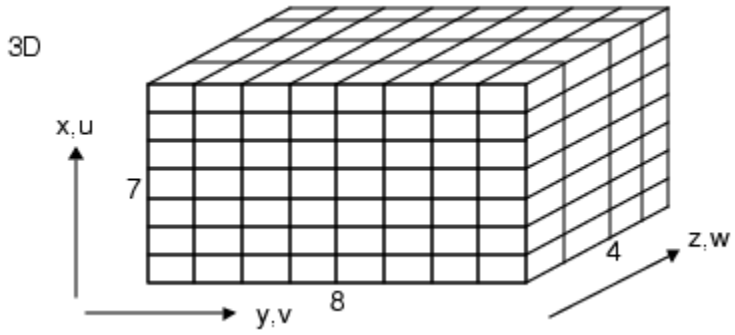
The domain of integration (or solution domain) is the region of space within which the differential equations are to be solved. In addition to setting the extent of the domain, problem specification involves the provision of boundary conditions, namely, a specification of the values (or fluxes) of the variables at these boundaries. Problem specification also requires the properties of the fluid to be described, namely, its conductivity, density, viscosity, specific heat, and expansivity.

The Solution Method

The conservation equations and their associated boundary conditions do not have a general analytical solution. There are particular solutions of the equations for simple situations (for example, laminar flow in a channel). But for the vast majority of cases of practical interest, the equations can only be solved by means of numerical integration. CFD provides the means of numerical integration.

In the CFD technique used in Simcenter Flotherm, the conservation equations are discretized by sub-division of the domain of integration into a set of non-overlapping, contiguous finite volumes over each of which the conservation equations are expressed in algebraic form. These finite volumes are referred to as ‘grid cells’, ‘control cells’, or as ‘cells’. The layout for the grid and velocities for a grid having a total of 224 (= 7×8×4) cells is shown in Figure 1-1.

Figure 1-1. Layout and Velocities for a Grid



The discretization results in a set of algebraic equations, each of which relates the value of a variable in a cell to its value in the nearest-neighbor cells.

For example letting T denote the temperature, the algebraic equation connecting T in a cell to its value in its six neighboring cells (denoted T_1,T_2,T_3,T_4,T_5,T_6) and its value at the old time step (denoted T_0) is written:

$$T = \frac{(C_0T_0 + C_1T_1 + C_2T_2 + C_3T_3 + C_4T_4 + C_5T_5 + C_6T_6 + S)}{(C_0 + C_1 + C_2 + C_3 + C_4 + C_5 + C_6)}$$

where C_n denotes the coefficients that link the in-cell value to each of its neighbor-cell values. S_n denotes the terms that represent the influences of the boundary conditions (if any), for example, a source of heat.

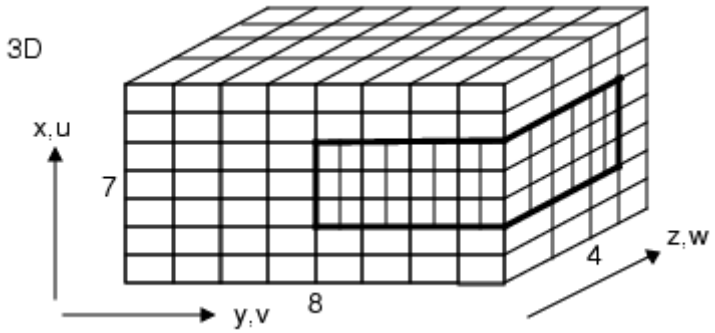
If there are a total of n cells in the integration domain, there are n algebraic equations to solve for each of the field variables T, u, v, w, p. That is, there are 5n equations to solve. Thus, for example, for a grid of 50000 cells there are 250000 equations to be solved, or more if the KE turbulence model is in use.

Expressed in the above form, the equations appear to be linear, but they are not, because the coefficients (C_n) are themselves functions of T, u, v, w and p. This appearance of linearity, however, is exploited as follows: at each *outer iteration* the coefficients are calculated once only and then taken as constant while the resulting algebraic equations are solved by means of *inner iteration*.

Localized Grid

For localized grids, a section of the base grid is subdivided as shown in Figure 1-2.

Figure 1-2. Subdivided Grid



The cell values for a localized grid are solved in exactly the same way as for the base grid, with the following exceptions.

Cells in the base grid which are overlapped by the localized grid are not solved for. Cells in the base grid adjacent to the localized grid feel the effect of the localized grid through extra terms and coefficients: any of the terms C_i, T_i (i = 1, 6) are replaced by a sum over coefficients and temperatures from the adjacent localized grid cells in the above equation for T.

Cells in the localized grid on its boundary with the base grid are affected by adjacent cells in the base grid through *halo cells* which have the same boundary cross-section as the localized grid cells, and the normal width of the adjacent base grid cell. These cells provide the neighbor T_i values required, which are interpolated from the base grid values.

These extra relationships are all included in the inner iteration algebraic solution. The same relationships exist between a localized grid nested in another localized grid, and its parent.

Summary of the Algorithm Used

The algorithm is summarized by the following example for a 3D simulation of flow and heat transfer:

1. Initialize the fields of pressure, temperature and velocities
2. Increase outer iteration count by 1
3. Set up coefficients (that is, the Cs) for temperature field, T
4. Solve linearized algebraic equations for the value of T in each cell by performing a number of inner iterations
5. Repeat 3 and 4 for field variables u, v and w
6. Solve the continuity equations in a similar manner and make any associated adjustments to pressure and velocities
7. Check for convergence and return to 2 if required

Grid Definition

The finer the grid used (that is, the greater the number of grid cells) the closer the algebraic equations approximate to the differential equations from which they originated.

Normally more grid is used in regions of the domain where gradients of the variables are expected to be greatest. However, grid-independence of the solution alone is not enough to ensure that the solution obtained simulates what in reality occurs because other modeling assumptions (for example, the accuracy of the boundary condition information supplied, the adequacy of the turbulence model) may be determining factors in the agreement of the simulation with physical reality.

Consequently, the outcome of a CFD simulation must be inspected carefully with a view to assessing the physical realism of the results obtained.

Interpretation of Results

Grid cell locations of discretized values of scalar and vector field variables.

Scalars

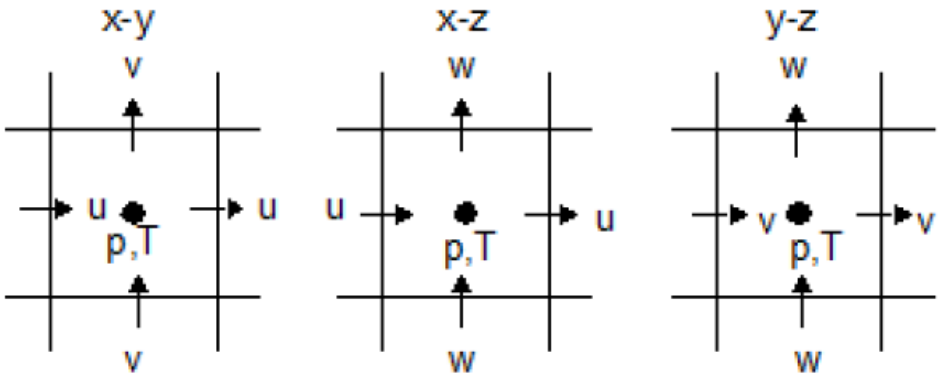
The discretized values of scalar field variables like temperature and pressure are located at the centers of the grid cells.

Staggered Vectors

However, the discretized values of the components *u*, *v* and *w* of the velocity vector field are located at the faces of the grid cells.

This arrangement is depicted schematically in the X-Y, X-Z and Y-Z planes for a single cell:

Figure 1-3. Staggered Vectors



This displacement from the cell centers of the locations of the values of the velocity components is known as staggering and is a recurrent theme in CFD methodologies employing finite-volume representation.

When displaying results in Simcenter Flotherm, you have the option of viewing the velocity values either at the cell faces or interpolated to the cell center.

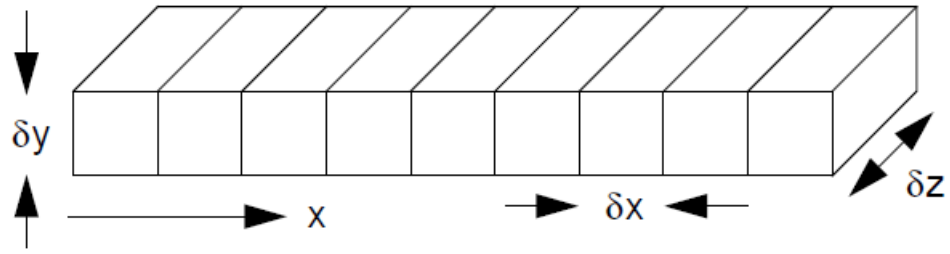
Chapter 2
Finite Volume Equations

The derivation of the finite-volume equations used by Simcenter Flotherm to solve the flow variables. Both the continuity and temperature equations are covered.

Derivation of Finite-Volume Equations 15
Electrical Potential Equations 18

Derivation of Finite-Volume Equations

Consider a transient, one-dimensional, uniform grid in the X-direction.



The finite volume-equations in differential form for:

- Continuity:

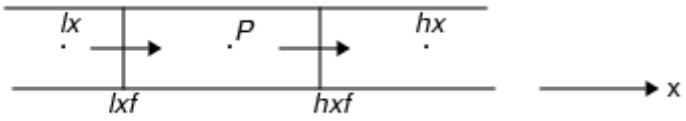
$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} = 0$$

- Temperature:

$$\frac{\partial (\rho C_p T)}{\partial t} + \frac{\partial (\rho u C_p T)}{\partial x} - \frac{\partial}{\partial x}\left(\frac{\lambda \partial T}{\partial x}\right) = S$$

Grid Cell Nomenclature:

Finite-volume equations are derived by volume integration over each grid cell:



Cell volume:

$V_p = \delta x\ \delta y\ \delta z$

x-direction face area:

$A_x = \delta y\ \delta z$

Continuity Equation

$$\iiint_{z\,y\,x} \frac{\partial \rho}{\partial t} dx\ dy\ dz = \left(\frac{\rho_p - \rho_t}{\partial t}\right)\delta x\ \delta y\ \delta z = \left(\frac{\rho_p - \rho_t}{\delta t}\right)V_p$$

$$\iiint_{z\,y\,x} \frac{\partial (\rho u)}{\partial x} dx\ dy\ dz = \iint_{z\,y} [(\rho u)_{hxf} - (\rho u)_{lxf}] dy\ dz$$

$$= (\rho u)_{hxf}\delta y\ \delta z - (\rho u)_{lxf}\delta y\ \delta z$$

$$= (\rho u)_{hxf}A_x - (\rho u)_{lxf}A_x$$

So the full equation is:

$$\left(\frac{\rho_p - \rho_t}{\delta t}\right)V_p + (\rho u)_{hxf}A_x - (\rho u)_{lxf}A_x = 0$$

that is:

Rate of increase of mass in cell + *Difference between outflow and inflow* = 0

or, for steady state or constant density:

$$Mass\ outflow - Mass\ inflow = 0$$

Temperature Equation

Transient term:

$$\iiint_{z\,y\,x} \frac{\partial(\rho C_p T)}{\partial t} dx\ dy\ dz = \left[\frac{(\rho C_p T)_p - (\rho C_p T)_t}{\partial t}\right] V_p$$

(that is, rate of increase of heat content of cell)

Convection term:

$$\iiint_{z\,y\,x} \frac{\partial(\rho u C_p T)}{\partial x}\ dx\ dy\ dz = \iint_{z\,y} [(\rho u C_p T)_{hcf} - (\rho u C_p T)_{lxf}]\ dy\ dz$$

$$= [(\rho u C_p T)_{hxf} A_x - (\rho u C_p T)_{lxf} A_x]$$

For u positive:

$T_{hxf} = T_p$, $\rho_{hxf} = \rho_p$

$T_{lxf} = T_{lx}$, $\rho_{lxf} = \rho_{lx}$

(and vice versa)

So convection term = $[\rho_p C_p u_{hxf} T_p A_x - \rho_{lx} C_p u_{lxf} T_{lx} A_x]$

(that is, outflow − inflow of heat by convection)

Conduction term:

$$\iiint_{z\,y\,x} \frac{-\partial}{\partial x}\lambda\left(\frac{\partial T}{\partial x}\right) dx\ dy\ dz = -\iint_{z\,y}\left[\lambda\left(\frac{\partial T}{\partial y}\right)_{hxf} - \lambda\left(\frac{\partial T}{\partial y}\right)_{lxf}\right] dy\ dz$$

$$= -\left[\lambda\left(\frac{T_{hx} - T_p}{\delta x}\right) - \lambda\left(\frac{T_p - T_{lx}}{\delta x}\right)\right] A_x$$

(that is, outflow − inflow of heat by conduction)

Source term:

$$\int_z \int_y \int_x S \, dx \, dy \, dz = S_p V_p$$

The full equation is:

$$T_p \left[\frac{\rho_p C_p V_p}{\delta t} + \rho_p C_p u_{hxf} A_x + \frac{\lambda A_x}{\delta x} + \frac{\lambda A_x}{\delta x} \right]$$

$$- \left[\frac{\rho_p C_p V_p T_t}{\delta t} + \left(\rho_{lx} C_p u_{lxf} A_x + \frac{\lambda A_x}{\delta x} \right) T_{lx} + \frac{\lambda A_x}{\delta x} T_{hx} \right] = S$$

or:

$$a_p T_p = a_t T_t + a_{lx} T_{lx} + a_{hx} T_{hx} + S$$

Note that using continuity equation:

$$a_p = a_t + a_{lx} + a_{hx}$$

The temperature equation can be written as:

$$a_p T_p - (a_t T_t + a_{lx} T_{lx} + a_{hx} T_{hx} + S) = 0$$

(that is, outflow − inflow − source of heat = 0)

Extension to 3D leads to equation for general variable, φ, of the form:

$$a_p \varphi_p = a_{lx} \varphi_{lx} + a_{hx} \varphi_{hx} + a_{ly} \varphi_{ly} + a_{hy} \varphi_{hy} + a_{lz} \varphi_{lz} + a_{hz} \varphi_{hz} + a_{lt} \varphi_{lt} + S$$

Electrical Potential Equations

Equations used to solve Joule heating.

Electrical potential as a 3D field is defined by:

$$\frac{\partial \frac{1}{r}\left(\frac{\partial \varphi}{\partial x}\right)}{\partial x} + \frac{\partial \frac{1}{r}\left(\frac{\partial \varphi}{\partial y}\right)}{\partial y} + \frac{\partial \frac{1}{r}\left(\frac{\partial \varphi}{\partial z}\right)}{\partial z} = S$$

where:

φ = Potential (volts)

r = Electrical Resistivity (ohm.m). Assumed to be isotropic but may be a function of temperature.

S = Source (amps) and is non-zero only at boundaries.

This equation is solved at each iteration and has the same form as the heat equation (conduction only).

The (vector) current density in A/m^2 is obtained from the potential:

$$I = \left(\frac{1}{r}\left(\frac{\partial \varphi}{\partial x}\right), \frac{1}{r}\left(\frac{\partial \varphi}{\partial y}\right), \frac{1}{r}\left(\frac{\partial \varphi}{\partial z}\right)\right)$$

where I = Current vector.

The power density in W/m^3 is:

$Q = r \times I^2$

Chapter 3
Wall Functions

The wall function formulae used to model smooth and rough walls.

Friction Effects at Smooth Walls for all Turbulent Options 21
Heat Transfer at Smooth Walls 23
Treatment of Rough Walls 24

Friction Effects at Smooth Walls for all Turbulent Options

Considerations for turbulent and laminar flow.

The formulae are expressed in terms of:

- The 'friction velocity':

$$u_\tau = \left(\frac{\tau_w}{\rho}\right)^{\frac{1}{2}}$$

and non-dimensional distance from wall (can also be viewed as a local Reynolds number):

$$y^+ = \frac{u_\tau y \rho}{\mu}$$

The formulae are based on the well-established 'universal' relationships:

For $y^+ > 11.5$ (turbulent flow):

$$\frac{u}{u_\tau} = \frac{1}{0.435} ln(9 y^+)$$

(the 'log-law')

For $y^+ \leq 11.5$ (laminar flow):

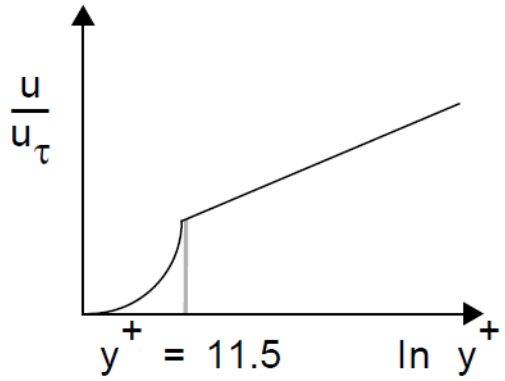
$$\left(\frac{u}{u\tau} = y^+\right)$$

and:

$$\left(\text{i.e. } \tau_w = \frac{\mu u}{y}\right)$$

See Figure 3-1.

Figure 3-1. Turbulent and Laminar Flow at Smooth Walls



From the calculated value of velocity u_p at a distance y from the wall surface, the wall shear stress τ_w is deduced from the above formulae. For turbulent conditions, this involves iteration, which is combined in the overall iterative solution in Simcenter Flotherm. The calculated value of wall shear stress is then applied as a source (negative) of momentum of the near-wall cell.

Note on Laminar Option

If the turbulence flag is set to laminar, then:

$$\tau_w = \frac{\mu u}{y}$$

unconditionally, that is, the laminar formulae for wall shear stress is always used. Therefore, as no wall functions are employed, an explicit resolution of the laminar boundary layer is required. This necessitates a finer near wall grid density.

Heat Transfer at Smooth Walls

Calculation of heat transfer at smooth walls.

The heat transfer rate per unit area at the wall $\dot{Q}''_w$ is related to the Stanton number, *St*, as follows:

$$St = \frac{\dot{Q}''_w}{\rho u C_p (T - T_w)}$$

Note that wall heat transfer coefficient is related to Stanton number by:

$h = St.\rho\, uC_p$

For laminar flow ($y^+ \leq 11.5$):

$$St = \frac{\lambda}{RePr} \quad [\text{ie.} \quad \dot{Q}''_w = \frac{\lambda}{y}(T - T_w)]$$

For turbulent flow ($y^+ > 11.5$) via generalized form of 'Taylor-Prandtl analogy' (heat flux proportional to momentum flux):

$$St = \frac{s}{0.9\left(1 + s^{\frac{1}{2}} P_j\right)}$$

where:

$$s = \frac{\tau_w}{\rho u^2}$$

0.9 is the 'turbulent Prandtl number', and P_j is Jayatilleke's sub-layer resistance function:

$$P_j = 9\left(\frac{Pr}{0.9} - 1\right)\left(\frac{0.9}{Pr}\right)^{\frac{1}{4}}$$

From calculated values of u_p and τ_w (hence s) and T_p, $\dot{Q}''_w$ is deduced. This is then applied as a heat source at near-wall cells.

Treatment of Rough Walls

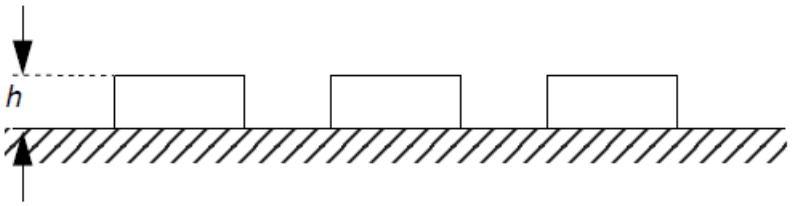
Calculation of heat transfer at smooth walls. A roughness Reynolds number is based on a prescribed roughness height.

The Reynolds number for rough walls is calculated from:

$$Re_r = \frac{h\mu_\tau \rho}{\mu}$$

where h is the roughness height as shown in Figure 3-2.

Figure 3-2. Roughness Height



The treatment is then exactly the same as for smooth walls, except that for $Re_r > 3.3$, the following formula is used to deduce u_τ and hence surface friction τ_w:

$$\frac{u}{u_\tau} = \frac{1}{0.435} ln\left(\frac{29.7y}{h}\right)$$

This leads to increased friction.

These formulae are then used to evaluate wall heat transfer rate. Because of the increased friction, this is also increased as compared with smooth walls.

Chapter 4
Turbulence Viscosity Model

The Simcenter Flotherm model used to represent the turbulence within the circulating fluid (normally air).

Turbulent Viscosity Estimation 25
Level of Turbulence Viscosity Estimation 26

Turbulent Viscosity Estimation

For the representation of turbulence within the fluid, it is necessary to estimate or evaluate values of turbulent viscosity.

The turbulent conductivity is then evaluated from the following formula:

$$\lambda_t = \frac{\mu_t C_p}{Pr_t}$$

where:

λ_t = turbulent conductivity

μ_t = turbulent viscosity

C_p = specific heat capacity

Pr_t = turbulent Prandtl number, taken as 0.9

Level of Turbulence Viscosity Estimation

Calculation of turbulence viscosity.

The Formula Chosen for the Estimation of the Level of Turbulence Viscosity **26**
Turbulence Models **29**
Automatic Algebraic Turbulence Model 29
Revised Algebraic Turbulence Model 29
LVEL K-Epsilon Turbulence Model 29
Near Wall Treatment **30**
References **31**

The Formula Chosen for the Estimation of the Level of Turbulence Viscosity

To model a turbulent flow, the temporal terms of the conservation equations would have to have a time step (dt) small enough to capture all turbulent fluctuations on even the smallest time scales.

The same applies to all physical dimensions of the control volume cells (dx_i) terms. They would have to be as small as that known as the Kolmogarov scale, which decreases non-linearly with an increase in the Reynolds number.

To overcome these limitations, variables are split into a mean and fluctuating component, that is:

$$U = \overline{U} + u'$$
$$H = \overline{H} + h' \qquad (4\text{-}1)$$

These are then substituted back into the instantaneous momentum equation producing the following:

$$\frac{\partial}{\partial x_j}(\rho \overline{U_i U_j}) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \overline{U_i}}{\partial x_j} - \rho \overline{u'}_i \overline{u'}_j\right) + g_i(\rho - \rho_0) \qquad (4\text{-}2)$$

This is known as the time averaged momentum equation. A similar equation exists for the enthalpy equation:

(4-3)

$$\frac{\partial}{\partial x_i}(\rho \overline{U}_i \overline{H}) = \frac{\partial}{\partial x_i}\left(\lambda \frac{\partial \overline{T}}{\partial x_i} - \rho \overline{u'}_i \overline{h'}\right)$$

The extra terms are produced by this substitution are:

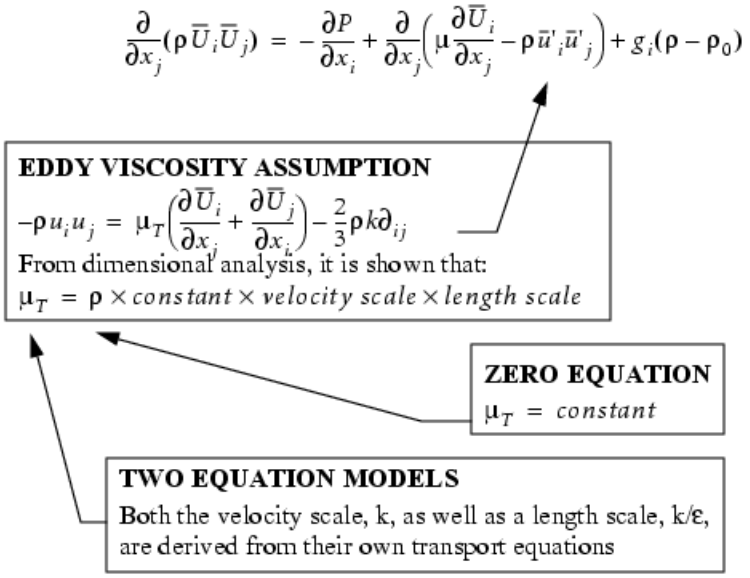
Reynolds stress = $-\rho \overline{u'}_i \overline{u'}_j$

Reynolds flux = $-\rho \overline{u'}_i \overline{h'}$

A turbulent flow is characterized by the dominance of diffusion due to the Reynolds stresses and fluxes over the diffusion due to the laminar viscosity or laminar diffusivity of the fluid.

The introduction of the Reynolds stresses and fluxes after decomposition of the turbulent fluctuating variables means that the equation set is now not closed. Some form of closure is required to model these fluxes and stresses. There have been a wide range of methods used to do this, varying from the most simple zero-equation models to the much more complex Reynolds stress transport equations. Figure 4-1 shows how these turbulence models relate to each other.

Figure 4-1. Tree of Turbulence Modeling



At the center of the zero- and two-equation models lies the analogy that where a laminar stress exists, then so can an equivalent turbulent stress (that is, Reynolds stress). A laminar shear stress is defined as:

(4-4)

$$\tau = \mu \frac{\partial U_i}{\partial x_j}$$

So, if a fluid can have a laminar viscosity, μ, then a turbulent flow should have a turbulent or eddy viscosity, μ_T. By using the eddy viscosity hypothesis, which Boussinesq proposed, we can relate the Reynolds stress to the mean strain by:

(4-5)

$$-\rho u_i u_j = \mu_T \left(\frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \partial_{ij}$$

A zero equation turbulence model either sets a constant value of the eddy viscosity or deduces it as an algebraic function of flow parameters. The zero equation turbulent model will therefore be referenced as an algebraic turbulent model.

The two equation model uses two differential transport equations to predict the eddy viscosity on a cell by cell basis.

The main limitation imposed at this stage by 4-5 is that the eddy viscosity is the same in all directions at any point. Now, where this may be true of laminar viscosity, which is a property of the fluid, it may not be true of turbulent viscosity, which is effectively a property of the flow. Therefore, this eddy viscosity can have differing values in relation to differing Reynolds stresses. This occurs when the turbulence is said to be anisotropic. Conditions that under certain circumstances may cause anisotropy, and thus could invalidate the isotropic assumption of 4-5, include extreme streamline curvature, swirl, adverse pressure gradients, and buoyancy.

Turbulence Models

You have a choice of turbulence models.

Automatic Algebraic Turbulence Model 29
Revised Algebraic Turbulence Model 29
LVEL K-Epsilon Turbulence Model 29

Automatic Algebraic Turbulence Model

Calculation of Automatic Algebraic turbulence.

The Automatic algebraic turbulence model (Ref. 1) removes the need for any user-defined velocity or length scale. A velocity scale is taken to be the cell velocity.

A length scale is calculated from the following equation:

$$D = \sqrt{(|\nabla\phi|^2 + 2\phi)} \tag{4-6}$$

where $\nabla\phi = -1$ with $\phi = 0$ at a wall.

These length and velocity scales (a scale for each grid cell) are then used in conjunction with classical boundary layer wall functions (see “Near Wall Treatment” on page 30) to determine cell turbulent viscosities.

This model, unlike the algebraic model, results in a turbulent viscosity that varies from cell-to-cell in the bulk flow.

Revised Algebraic Turbulence Model

The revised algebraic turbulence model is the same as the Automatic Algebraic model but with a cap.

$$Cap\ \mu_T = 0.01 \times density \times velocity\ scale \times length\ scale$$

LVEL K-Epsilon Turbulence Model

This turbulence model calculates two variables; the kinetic energy of turbulence (k) and the dissipation rate of k (denoted ε).

The eddy viscosity is defined from dimensional analysis as:

$$\mu_T = C_\mu \rho \frac{k^2}{\varepsilon} \tag{4-7}$$

The transport equations for k and ε are:

$$\frac{\partial \rho \bar{U}_i k}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + P + G - \rho\varepsilon \tag{4-8}$$

$$\frac{\partial \rho \bar{U}_i \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_T}{\sigma_k}\right)\frac{\partial \varepsilon}{\partial x_i}\right) + C_1\frac{\varepsilon}{k}(P + C_3 G) - C_2\rho\frac{\varepsilon^2}{k} \tag{4-9}$$

where P is the shear production defined as:

$$P = \mu_{eff}\frac{\partial \bar{U}_i}{\partial x_j}\left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) \tag{4-10}$$

G is the production of turbulence kinetic energy due to buoyancy, and is given by:

$$G = \frac{\mu_{eff}}{\sigma_T}\beta g_i \frac{\partial T}{\partial x_i} \tag{4-11}$$

C_μ	0.09
C_1	1.44
C_2	1.92
C_3	1.0
σ_k	1.0
σ_ε	1.217

This model has been tried and tested for a whole range of engineering applications. It is simple, but more importantly, it is *stable*. Only two extra differential equations are introduced and the convergence process is less prone to divergence than other, higher order turbulence models.

Near Wall Treatment

Corrections are made to the turbulent viscosity so as to damp its value from the bulk value outside the wall boundary layer to zero at the wall surface. Fluid velocity at a wall surface is zero, which is known as the no-slip condition. The type of flow between the wall and the bulk flow is known as a shear layer, in this case a wall boundary layer.

The boundary layer is a very complex region of high velocity gradient and diffusion dominated development. To model it precisely would necessitate an extremely fine grid. An empirical relationship is therefore used to describe the shape of the boundary layer so that only one grid cell near the wall is required. This empirical relationship describes the shape of the boundary layer in non-dimensional terms. Two non-dimensional terms are formulated. These are the friction velocity:

$$u_\tau = \left(\frac{\tau_w}{\rho}\right)^{\frac{1}{2}} \tag{4-12}$$

and a non-dimensionalized distance from the wall (which can be viewed as a local Reynolds number):

$$y^+ = \frac{u_\tau y \rho}{\mu} \tag{4-13}$$

These formulae are based upon the well-established ‘universal’ relationships:

For $y^+ > 11.5$ (turbulent):

$$\frac{u}{u_\tau} = \frac{1}{0.435}\ln(9y^+) \tag{4-14}$$

For $y^+ < 11.5$ (laminar):

$$\frac{u}{u_\tau} = y^+ \tag{4-15}$$

All that is required to deduce the wall shear stress from the near wall velocity is therefore the distance from the near wall cell center to the wall itself. Variation of the eddy viscosity itself in the wall boundary layer is deduced from similar relationships.

References

Turbulence model references.

There is increasing experience in the use of this turbulence representation for Simcenter Flotherm calculations which confirm the validity of the approach for a range of applications, from full system-level analyses (for example, Ref. 4) to detailed analyses of packages and heat sinks (Ref. 5).

1. D. Agonafer, L. Gan-Li and D. B. Spalding, “The LVEL Turbulence Model for Conjugate Heat Transfer at Low Reynolds Numbers”, *EEP-Vol. 18*, Application of CAE/CAD Electronic Systems. ASME 1996.
2. B. E. Launder and D. B. Spalding, “The Numerical Computation of Turbulent Flows”, Appendix D of *Computer Methods in Applied Mechanics and Engineering 3* (1974) 269-289. North-Holland Publishing Company.
3. S. Goldstein, *Modern Developments in Fluid Dynamics*, Dover Publications Inc. Volume 2, 1965.
4. “Telecommunications Exchange System”, *FloTHERM Update*, Winter 1990.
5. S. Gopalakrishna, “Numerical and Experimental Study of Forced Convection Over Power Supply. Heat Sinks”, *ASME Winter Annual Meeting*, Atlanta, Dec. 1991.

Chapter 5
Radiation Model

The treatment of thermal radiation.

Radiative Exchange 33
Radiating and Blocking Objects 36
View Factor 39

Radiative Exchange

Calculation of radiative exchange.

Assumptions

The physical model assumes that:

- There is no dependence of the radiative exchange upon the frequency of the radiation, that is, the “gray” approximation.
- All reflections of radiation from radiating objects are “diffuse”, that is, that radiation is reflected in equal proportions in all directions with no dependence upon the angle of incident radiation.

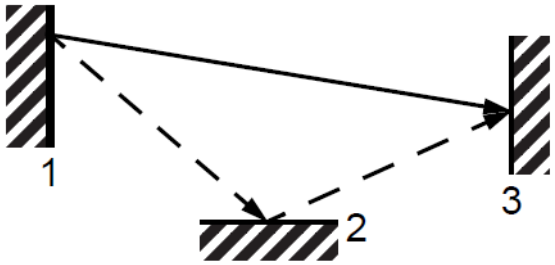
The model is fully conservative, that is, no heat is lost from the system other than that which is correctly lost to the exterior environment.

Radiative Exchange

Simcenter Flotherm considers the effects of both direct and reflected radiation.

Consider Figure 5-1, showing radiation emitted from Surface 1 reaching Surface 2.

Figure 5-1. Radiation Emitted from Surface 1 Reaching Surface 2



The proportion of the radiation emitted by Surface 1 that falls directly on Surface 3 (shown by the unbroken arrow) is F_{13}, that is, the view factor, but only a proportion noted by:

$$\varepsilon_3 F_{13}$$

is absorbed, the rest being reflected.[1] However, a proportion of the radiation emitted from Surface 1 is reflected from Surface 2, that is:

$$F_{12}(1 - \varepsilon_2)$$

and of this a fraction F_{23}, that is:

$$\varepsilon_3 F_{23} F_{12}(1 - \varepsilon_2)$$

is absorbed by Surface 3.

However, the sums of terms 1 and 2 does not comprise all of the radiation that passes from Surface 1 to Surface 3. There are additional routes to consider: in fact there are an infinite number of routes to consider, that is those that result from multiple reflections.

In order to avoid the consideration of the infinity of routes, it is convenient to define a quantity G_{ij} which is:

> the proportion of radiative energy emitted by Surface *j* that makes its way to Surface k by *all* possible routes and is absorbed.

So, in our example, the quantity G_{13} is the sum of the proportion of the radiation emitted from Surface 1 that falls directly on Surface 3 and the fraction of the radiation that travels directly from Surface 1 to every other surface, is reflected, and then finds its way to Surface 3, that is:

$$G_{13} = \varepsilon_3 F_{13} + F_{11} G_{13}(1 - \varepsilon_1) + F_{12} G_{23}(1 - \varepsilon_2) + F_{13} G_{33}(1 - \varepsilon_3)$$

where each reflection term is made up as follows, for example for the second reflection term:

- F_{12} - the fraction of energy emitted by Surface 1 that falls directly on Surface 2
- $F_{12}(1 - \varepsilon_2)$ - the fraction of that energy that is reflected
- $G_{23} F_{12}(1 - \varepsilon_2)$ - the fraction of the reflected energy that finds its way by all paths, to Surface 3 and is then absorbed.

1. The gray approximation tells us that the absorptivity $\alpha = \varepsilon$, the emissivity.

For a general system of N radiant surfaces, the formula for G_{ik} becomes:

(5-1)

$$G_{ik} = \varepsilon_k F_{ik} + \sum_{j=1}^{N} F_{ij} G_{jk} (1 - \varepsilon_j)$$

Given the view factors, that is, all F_{ik}s, the coefficients G_{ik} can be calculated by solving the N set of N linear equations (5-1).

Radiative Heat Flux

The radiative heat flux incident upon a Surface k that is absorbed may now be written as:

$$Q_{k,in} = \varepsilon_k A_k \left(\sum_{j=1}^{N} G_{jk} \sigma T_j^{\,4} \right)$$

and the outgoing radiation is:

$$Q_{k,out} = \varepsilon_k A_k \sigma T_k^{\,4} \sum_{j=1}^{N} G_{jk}$$

so that the net heat-flux arriving at the surface is:

$$Q_k = \varepsilon_k A_k \sum_{j=1}^{N} G_{jk} \sigma (T_j^{\,4} - T_k^4)$$

Note

A simplifying assumption has been made. T_k and T_j are the cell temperatures, not the surface temperatures. This will have little effect unless any of the following conditions occur:

- Material conductivity is small.
- Surface has a surface-fluid thermal resistance attached via a surface attribute.
- A collapsed cuboid is placed on the surface.

Radiating and Blocking Objects

How objects are considered when modeling radiation.

Objects That Radiate

Table 5-1. Radiating Objects

Object	Requirements	Mode
Solid Cuboid	Conducting option selected in attached thermal attribute.	Active
	Fixed Flux option selected in attached thermal.	Inactive
	Fixed Temp. option selected in attached thermal.	Source Only
Collapsed Cuboid	Collapsed cuboids on an external boundary radiates to the external environment.[1]	Active
	Fixed Temp. option selected in attached thermal.	Source Only
Prism	Conducting option selected in attached thermal attribute.	Active
	Fixed Flux option selected in attached thermal.	Inactive
	Fixed Temp. option selected in attached thermal.	Source Only
Tet	Conducting option selected in attached thermal attribute.	Active
	Fixed Flux option selected in attached thermal.	Inactive
	Fixed Temp. option selected in attached thermal.	Source Only
Inverted Tet	Conducting option selected in attached thermal attribute.	Active
	Fixed Flux option selected in attached thermal.	Inactive
	Fixed Temp. option selected in attached thermal.	Source Only
Open Boundary[2]	Radiates unconditionally using radiant temperatures attached to ambients. If no ambients are attached, then uses the External Radiant Temperature set in the Global System Settings dialog box.	Source Only

Table 5-1. Radiating Objects (cont.)

Object	Requirements	Mode
Planar Resistance[3]	Radiates unconditionally using radiant temperatures attached to ambients. If no ambients are attached, then uses the External Radiant Temperature set in the Global System Settings dialog box.	Source Only
SmartParts (all except Compact Component	Radiates from all constituents parts.	Active
Compact Component	Has single radiating surface on the top surface.	Active

1. The radiant temperature of the external environment is set either to the defined value in the Global System Settings dialog box, or the value in any attached ambient.
2. This presumes that an appropriate radiative attribute has been attached.
3. Emissivity is taken to be 1 (one).

The modes are:

- Active − the object emits radiation and that the temperature of the object reacts to absorbed radiation.
- Source Only − the object emits and absorbs radiation but does not react to absorbed radiation.
- Inactive − takes no part in the radiative exchange.

Objects that Block Radiation

Table 5-2 enumerates the objects that block radiation and explains the circumstances under which they block radiation.

Table 5-2. Blocking Objects

Object	Comments
Solid Cuboid	Always
Prism	Always
Resistance	Only when there is resistance to flow and Transparent to Radiation is deactivated for the attached resistance.
Fan	No effect
Fixed Flow	No effect
Recirculation Device	No effect

Table 5-2. Blocking Objects (cont.)

Object	Comments
PCB	PCB blocks as a single collapsed solid cuboid blocks when the system level model is selected, otherwise it blocks as below.
Other SmartParts	These block radiation in so much as their constituent objects block radiation.

View Factor

Improving the accuracy of the radiative model.

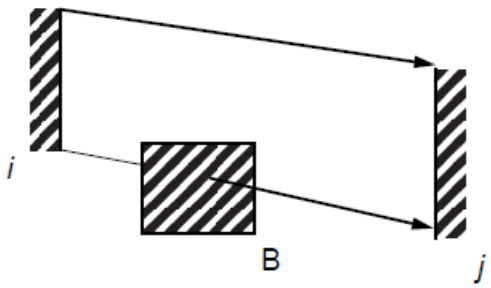
Calculation of View Factors **39**
Costs and Recommendations **40**
Generating a View Factor Log **41**

Calculation of View Factors

The calculation of the view factors takes into account the presence of objects that obstruct the passage of radiation between surfaces.

Consider Figure 5-2, in which Blockage B partially obstructs the radiative path between Surfaces *i* and *j*.

Figure 5-2. View Factor



The view factor calculation is based on the Monte Carlo method.

The radiation heat transfer is modeled by following the progress of discrete amounts (bundles) of energy. As a result, some of the bundles leaving Surface *i* that would reach Surface *j* if Blockage B were not there, will hit Blockage B and thus not contribute to the view factor, hence reducing it. The accuracy of the calculated view factors depends on the number of bundles fired.

Two accuracy levels may be used with the options:

- Radiation On, and
- Radiation On high accuracy.

The Radiation On high accuracy option takes about four times as long as the Radiation On option for the calculation of view factors. Typical accuracies are 1% for the high accuracy method and 5% otherwise.

Costs and Recommendations

The calculation of view factors can be a costly process, and so it is worth discussing briefly the costs involved, and then, based on the costs, to make recommendations.

Cost of View Factor Calculations

It is easy to see that for a system in which there are N radiant surfaces the number of view factors to be computed will be:

$$(1/2)N(N-1)$$

So when there are many radiant surfaces in the system, doubling the number will roughly quadruple the amount of work to be done and the computer time required to do it.

The calculation of the view factors and of the coefficients, G_{ik}, is performed prior to beginning the CFD calculation and the results of the calculation are retained as part of the solution of the Project to be reused in subsequent calculations.

Interaction with Symmetry Boundaries

The view factor calculation will correctly deal with symmetry boundaries with the following rules:

1. The radiation algorithm will correctly handle symmetry boundaries on the overall domain boundary only, not on cutouts. Symmetry conditions on cutouts are ignored.
2. The software can only handle the symmetry conditions on either the high or low overall boundaries in a defined case, not both in the same problem. This means that 1/2, 1/4 or 1/8 symmetry problems are handled correctly.
3. For cases involving radiation where symmetry is defined at two parallel opposing overall boundaries, the software will not activate radiation for those object faces which “see” the symmetry domain face. Internal radiation exchange from internal object faces is unaffected. A warning message will be issued under these circumstances.

Recommendations

The following practices may be recommended:

- Take care to make solid surfaces radiant only when it is really necessary.
- Where possible, avoid changes to a project that will cause the view factors and the G_{ik} coefficients to be recalculated.

References

1. Ellison, G., *Thermal Computations for Electronic Equipment*, Van Nostrand, 1984, Chapter 3.
2. Siegel, R. & Howell, J.R., *Thermal Radiation Heat Transfer*, Hemisphere Publishing Corporation, 3rd Edition, 1992, Chapter 7 (with particular reference to section 7-4.2).

Generating a View Factor Log

By default, the view factor values are not available for display. However, you can request Simcenter Flotherm to write out a log of the view factors as they are generated to an ASCII text file, which can then be viewed using any normal text editor.

Restrictions and Limitations

- Enabling the View Factor Log can seriously lower the performance of the Exchange Factor Generator.

Prerequisites

- Make sure there is sufficient disk space as this file can be very large.

Procedure

1. Set the environment variable.

   ```
   set OUTPUTEFGLOG=1
   ```

   Ensure there are no blank spaces after the 1.

2. Run Exchange Factor Calculations

   Click the Exchange Factors tool icon  or select **Solve > Exchange Factors** to start the exchange factors calculation.

Results

The results of the exchange factor calculation are written to the following file:

.../<solution>/<project_dir>/DataSets/BaseSolution/Exchange/viewf.log

To disable the View Factor Log, either set OUTPUTEFGLOG=0 or remove the OUTPUTEFGLOG environment variable. For information on setting up environment variables, consult your operating system documentation.

The file heading details the number of radiating surfaces and is followed by tabulations:

- Surface Positions — location of radiating surfaces in terms of physical coordinates and emissivity.

Columns for the surface number followed by the coordinates defining the surface points (four for a rectangular surface, three for a triangular surface) and the name of the object to which the surface belongs.

- Group Analysis

  Lists the surfaces associated with the individual isolated radiation groups. Surfaces belonging to the same group exchange radiation.

- View Factors

  Indicates the isolated groups and tabulates the view factors for each surface pair. The first column indicates the surface and the remaining columns give the exchange surfaces and their view factors, followed by the summation of all the view factors for the surface.

- Exchange Factors

  Lists the exchange factors for each group in the same format as for the View Factors.

Examples

In Figure 5-3, the view factor from Surface 1 to Surface 2 is 2.0605e-001 and the total view factor from Surface 1 is 9.9365e-001.

Figure 5-3. Example View Factors Output

```
View-factors:

Group   1 (   6)

   1    2  2.0605e-001    3  1.9531e-001    4  1.9678e-001
   1    5  1.9531e-001    6  2.0020e-001
   1  SUM  9.9365e-001

   2    1  2.0605e-001    3  2.0020e-001    4  2.0166e-001
   2    5  1.9678e-001    6  2.0166e-001
   2  SUM  1.0063e+000

   3    1  1.9531e-001    2  2.0020e-001    4  2.0605e-001
   3    5  1.9531e-001    6  1.9678e-001
   3  SUM  9.9365e-001

   4    1  1.9678e-001    2  2.0166e-001    3  2.0605e-001
   4    5  2.0020e-001    6  2.0166e-001
   4  SUM  1.0063e+000

   5    1  1.9531e-001    2  1.9678e-001    3  1.9531e-001
   5    4  2.0020e-001    6  2.0605e-001
   5  SUM  9.9365e-001

   6    1  2.0020e-001    2  2.0166e-001    3  1.9678e-001
   6    4  2.0166e-001    5  2.0605e-001
   6  SUM  1.0063e+000
```

Chapter 6
Further Reading

Recommended books and papers.

For a thorough description of the fundamentals of fluid flow and heat transfer written for engineers and applied scientists, *Transport Phenomena* by Bird, Stewart, and Lightfoot (Wiley) is recommended. This also contains useful descriptions of friction factors and heat transfer coefficients and an appendix contains tabulations of conversion factors. The chapter on radiation contains graphs on view factors.

For a background on the type of CFD used by Simcenter Flotherm, the reader can consult *Numerical Heat Transfer and Fluid Flow* by Patankar (McGraw-Hill). This book provides detailed derivations of the equations and their methods of solution with useful asides explaining why some numerical representations are better than others.

Lectures in Mathematical Models of Turbulence by Launder and Spalding (Academic Press) is a good introduction to the world of turbulence models. *A First Course in Turbulence* by Tennekes and Lumley (the MITPress) provides more on the physical nature of turbulence.

There are several good books which specifically address thermal cooling of electronic equipment, namely:

Thermal Computations for Electronic Equipment by Ellison (Van Nostrand). The sections on radiation, fans, and heat sinks are especially good.

Thermal Analysis and Control of Electronic Equipment by Kraus and Bar-Cohen (McGraw-Hill).

Cooling Techniques for Electronic Equipment by Steinberg (Wiley).

Hot Air Rises and Heatsinks: Everything you know about Cooling Electronics is wrong by Tony Kordyban (ASME International).

More Hot Air by Tony Kordyban (ASME International).

A good discussion of thermal issues is also to be found in *The Handbook of Electronic Packaging* by Pecht (Marcel Dekker).

For the state of the art of thermal cooling the reader can consult the quarterly *Journal of Electronic Packaging* (American Society of Mechanical Engineers).

Compact Heat Exchangers by Kays and London (3rd edition) (McGraw-Hill) contains a comprehensive source of experimental and design data for component heat exchangers

including those suitable for the cooling of electronic equipment. *The Handbook of Single-phase Convective Heat Transfer* by Kakac, Shah, and Aung (Wiley) contains an abundance of correlations of heat transfer coefficients and skin friction factors. *Flow Resistance: a Design Guide for Engineers* by Fried and Idelchik (Hemisphere) contains perhaps the most comprehensive list of friction factors, for example, Chapter 8 provides resistance coefficients of grids, screens, porous layers, and packings.

For an introduction to numerical experimental design techniques, refer to the following documents:

- Sachs, J., W.J. Welch, T.J. Mitchel, and H.P. Wynn (1989), *Design and Analysis of Computer Experiments*, Statistical Science 4, 409-435.
- Montgomery D.C. (1984), *Design and Analysis of Experiments (Second Edition)*, John Wiley & Sons, New York.

The proceedings of the following FloTHERM International User Conferences include applications of Simcenter Flotherm and a variety of electronic problems:

- 1st International — 19-20th September 1991, Bramley Grange, Guildford, Surrey, UK
- 2nd International — 11-12th May 1993, Boston, USA
- 3rd International — 22-23rd September 1994, Bramley Grange, Guildford, Surrey, UK
- 4th International — 4-5th October 1995, San Jose, USA
- 5th International — 18-19th September 1996, Paris, France
- 6th International — 13-16th October 1997, Cambridge, Massachusetts, USA
- 7th International — 3-4th September 1998, Verona, Italy
- 8th International — 13-14th May 1999, Las Vegas, USA
- 9th International —18-19th October 2000, Orlando, USA
- 10th International —14-15th May 2001, Amsterdam, Netherlands

For a full list of published application examples of the use of Simcenter Flotherm, visit:

https://www.mentor.com/products/mechanical/flotherm/flotherm

Index

— A —
Algorithms for Solution, 10
Automatic Algebraic Turbulence Model, 29

— C —
Calculate
  Calculation Domain, 9
Cells, 10
Computational Cells, 10
Conservation Equations, 10
Continuity Equation, 16

— D —
Differential Equations, 9
Domain (see Solution Domain)
  Domain of Integration, 9

— F —
Friction
  effects at rough walls, 24

— G —
Gridding
  grid definition, 12

— H —
Heat Transfer
  at rough walls, 24
  at smooth walls, 23

— J —
Jayatilleke's Resistance Function, 23

— L —
Localized Grid, 10
LVEL
  K-Epsilon Turbulence Model, 29

— M —
Mathematical Modeling, 9

— O —
OUTPUTEFGLOG, 41

— P —
Prandtl Number, 23

— R —
Results
  staggered vectors, 12
Revised Algebraic Turbulence Model, 29
Reynolds Number, 21
Roughness, 24

— S —
Smooth Walls, 21, 23
Solution
  algorithm, 10
  method, 10
  solution domain, 9
Stagger, 12
Stanton Number, 23

— T —
Taylor-Prandtl Analogy, 23
Temperature
  equation, 17

— V —
Variables
  location, 12
View Factor, 39
  View Factor Log, 41