ANSYS FLUENT Continuous Fiber Module Manual

Table of Contents

Using This Manual

This preface is divided into the following sections:

1. The Contents of This Manual

The ANSYS FLUENT Continuous Fiber Model Manual tells you what you need to know to model melt and dry spinning processes with ANSYS FLUENT. In this manual, you will find background information pertaining to the model, a theoretical discussion of the model used in ANSYS FLUENT, and a description of using the model for your CFD simulations.

2. The Contents of the FLUENT Manuals

The manuals listed below form the FLUENT product documentation set. They include descriptions of the procedures, commands, and theoretical details needed to use FLUENT products.

FLUENT Getting Started Guide contains general information about getting started with using FLUENT.

FLUENT User's Guide contains detailed information about using FLUENT, including information about the user interface, reading and writing files, defining boundary conditions, setting up physical models, calculating a solution, and analyzing your results.

FLUENT in Workbench User's Guide contains information about getting started with and using FLUENT within the Workbench environment.

FLUENT Theory Guide contains reference information for how the physical models are implemented in FLUENT.

FLUENT UDF Manual contains information about writing and using user-defined functions (UDFs).

FLUENT Tutorial Guide contains a number of example problems with detailed instructions, commentary, and postprocessing of results.

FLUENT Text Command List contains a brief description of each of the commands in FLUENT's text interface.

FLUENT Adjoint Solver Module Manual contains information about the background and usage of FLUENT's Adjoint Solver Module that allows you to obtain detailed sensitivity data for the performance of a fluid system.

FLUENT Battery Module Manual contains information about the background and usage of FLUENT's Battery Module that allows you to analyze the behavior of electric batteries.

FLUENT Continuous Fiber Module Manual contains information about the background and usage of FLUENT's Continuous Fiber Module that allows you to analyze the behavior of fiber flow, fiber properties, and coupling between fibers and the surrounding fluid due to the strong interaction that exists between the fibers and the surrounding gas.

FLUENT Fuel Cell Modules Manual contains information about the background and the usage of two separate add-on fuel cell models for FLUENT that allow you to model polymer electrolyte membrane fuel cells (PEMFC), solid oxide fuel cells (SOFC), and electrolysis with FLUENT.

FLUENT Magnetohydrodynamics (MHD) Module Manual contains information about the background and usage of FLUENT's Magnetohydrodynamics (MHD) Module that allows you to analyze

the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields.

FLUENT Migration Manual contains information about transitioning from the previous release of FLUENT, including details about new features, solution changes, and text command list changes.

FLUENT Population Balance Module Manual contains information about the background and usage of FLUENT's Population Balance Module that allows you to analyze multiphase flows involving size distributions where particle population (as well as momentum, mass, and energy) require a balance equation.

Running FLUENT Under LSF contains information about the using FLUENT with Platform Computing's LSF software, a distributed computing resource management tool.

Running FLUENT Under PBS Professional contains information about the using FLUENT with Altair PBS Professional, an open workload management tool for local and distributed environments.

Running FLUENT Under SGE contains information about the using FLUENT with Sun Grid Engine (SGE) software, a distributed computing resource management tool.

3. Typographical Conventions

Several typographical conventions are used in this manual's text to facilitate your learning process.

- Different type styles are used to indicate graphical user interface menu items and text interface menu items (for example, **Iso-Surface** dialog box, `surface/iso-surface` command).

- The text interface type style is also used when illustrating exactly what appears on the screen or exactly what you need to type into a field in a dialog box. The information displayed on the screen is enclosed in a large box to distinguish it from the narrative text, and user inputs are often enclosed in smaller boxes.

- A mini flow chart is used to guide you through the navigation pane, which leads you to a specific task page or dialog box. For example,

Models → **Multiphase** → **Edit...**

indicates that **Models** is selected in the navigation pane, which then opens the corresponding task page. In the **Models** task page, **Multiphase** is selected from the list. Clicking the **Edit...** button opens the **Multiphase** dialog box.

Also, a mini flow chart is used to indicate the menu selections that lead you to a specific command or dialog box. For example,

Define → **Injections...**

indicates that the **Injections...** menu item can be selected from the **Define** pull-down menu, and

`display` → `mesh`

indicates that the `mesh` command is available in the `display` text menu.

In this manual, mini flow charts usually precede a description of a dialog box or command, or a screen illustration showing how to use the dialog box or command. They allow you to look up information about a command or dialog box and quickly determine how to access it without having to search the preceding material.

- The menu selections that will lead you to a particular dialog box or task page are also indicated (usually within a paragraph) using a "/". For example, **Define/Materials...** tells you to choose the **Materials...** menu item from the **Define** pull-down menu.

4. Mathematical Conventions

- Where possible, vector quantities are displayed with a raised arrow (e.g., $\vec{a}$, $\vec{A}$). Boldfaced characters are reserved for vectors and matrices as they apply to linear algebra (e.g., the identity matrix, I).

- The operator ∇, referred to as grad, nabla, or del, represents the partial derivative of a quantity with respect to all directions in the chosen coordinate system. In Cartesian coordinates, ∇ is defined to be

$$\frac{\partial}{\partial x}\vec{\imath} + \frac{\partial}{\partial y}\vec{\jmath} + \frac{\partial}{\partial z}\vec{k} \tag{1}$$

∇ appears in several ways:

- The gradient of a scalar quantity is the vector whose components are the partial derivatives; for example,

$$\nabla p = \frac{\partial p}{\partial x}\vec{\imath} + \frac{\partial p}{\partial y}\vec{\jmath} + \frac{\partial p}{\partial z}\vec{k} \tag{2}$$

- The gradient of a vector quantity is a second-order tensor; for example, in Cartesian coordinates,

$$\nabla(\vec{v}) = \left(\frac{\partial}{\partial x}\vec{\imath} + \frac{\partial}{\partial y}\vec{\jmath} + \frac{\partial}{\partial z}\vec{k}\right)\left(v_x\vec{\imath} + v_j\vec{\jmath} + v_z\vec{k}\right) \tag{3}$$

This tensor is usually written as

$$\begin{pmatrix} \dfrac{\partial v_x}{\partial x} & \dfrac{\partial v_x}{\partial y} & \dfrac{\partial v_x}{\partial z} \\[2ex] \dfrac{\partial v_y}{\partial x} & \dfrac{\partial v_y}{\partial y} & \dfrac{\partial v_y}{\partial z} \\[2ex] \dfrac{\partial v_z}{\partial x} & \dfrac{\partial v_z}{\partial y} & \dfrac{\partial v_z}{\partial z} \end{pmatrix} \tag{4}$$

- The divergence of a vector quantity, which is the inner product between ∇ and a vector; for example,

$$\nabla \cdot \vec{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z}$$ **(5)**

– The operator $\nabla \cdot \nabla$, which is usually written as ∇^2 and is known as the Laplacian; for example,

$$\nabla^2 T = \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}$$ **(6)**

$\nabla^2 T$ is different from the expression $(\nabla T)^2$, which is defined as

$$(\nabla T)^2 = \left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2 + \left(\frac{\partial T}{\partial z}\right)^2$$ **(7)**

- An exception to the use of ∇ is found in the discussion of Reynolds stresses in "Modeling Turbulence" in the User's Guide, where convention dictates the use of Cartesian tensor notation. In this chapter, you will also find that some velocity vector components are written as u, v, and w instead of the conventional v with directional subscripts.

5. Technical Support

If you encounter difficulties while using ANSYS FLUENT, please first refer to the section(s) of the manual containing information on the commands you are trying to use or the type of problem you are trying to solve. The product documentation is available from the online help, or from the ANSYS Customer Portal (www.ansys.com/customerportal).

If you encounter an error, please write down the exact error message that appeared and note as much information as you can about what you were doing in ANSYS FLUENT.

Technical Support for ANSYS, Inc. products is provided either by ANSYS, Inc. directly or by one of our certified ANSYS Support Providers. Please check with the ANSYS Support Coordinator (ASC) at your company to determine who provides support for your company, or go to www.ansys.com and select **About ANSYS> Contacts and Locations**. The direct URL is: http://www1.ansys.com/customer/public/supportlist.asp. Follow the on-screen instructions to obtain your support provider contact information. You will need your customer number. If you don't know your customer number, contact the ASC at your company.

If your support is provided by ANSYS, Inc. directly, Technical Support can be accessed quickly and efficiently from the ANSYS Customer Portal, which is available from the ANSYS Website (www.ansys.com) under **Support> Technical Support** where the Customer Portal is located. The direct URL is: http://www.ansys.com/customerportal.

One of the many useful features of the Customer Portal is the Knowledge Resources Search, which can be found on the Home page of the Customer Portal.

Systems and installation Knowledge Resources are easily accessible via the Customer Portal by using the following keywords in the search box: `Systems/Installation`. These Knowledge Resources provide solutions and guidance on how to resolve installation and licensing issues quickly.

NORTH AMERICA

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Toll-Free Telephone: 1.800.711.7199

Fax: 1.724.514.5096

Support for University customers is provided only through the ANSYS Customer Portal.

GERMANY

ANSYS Mechanical Products

Telephone: +49 (0) 8092 7005-55

Email: support@cadfem.de

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

National Toll-Free Telephone:

German language: 0800 181 8499

English language: 0800 181 1565

International Telephone:

German language: +49 6151 3644 300

English language: +49 6151 3644 400

Email: support-germany@ansys.com

UNITED KINGDOM

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Fax: +44 (0) 870 142 0302

Email: support-uk@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

JAPAN

CFX , ICEM CFD and Mechanical Products

Telephone: +81-3-5324-8333

Fax: +81-3-5324-7308

Email: *CFX:* japan-cfx-support@ansys.com; *Mechanical:* japan-ansys-support@ansys.com

FLUENT Products

Telephone: +81-3-5324-7305

Email: *FLUENT:* japan-fluent-support@ansys.com; *POLYFLOW:* japan-polyflow-support@ansys.com; *FfC:* japan-ffc-support@ansys.com; *FloWizard:* japan-flowizard-support@ansys.com

Icepak

Telephone: +81-3-5324-7444

Email: japan-icepak-support@ansys.com

Licensing and Installation

Email: japan-license-support@ansys.com

INDIA

ANSYS Products (including FLUENT, CFX, ICEM-CFD)

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +91 1 800 233 3475 (toll free) or +91 1 800 209 3475 (toll free)

Fax: +91 80 2529 1271

Email: *FEA products:* feasup-india@ansys.com; *CFD products:* cfdsup-india@ansys.com; *Installation:* installation-india@ansys.com

FRANCE

All ANSYS, Inc. Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Toll-Free Telephone: +33 (0) 800 919 225

Email: support-france@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

BELGIUM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +32 (0) 10 45 28 61

Email: support-belgium@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SWEDEN

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +44 (0) 870 142 0300

Email: support-sweden@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

SPAIN and PORTUGAL

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +33 1 30 60 15 63

Email: support-spain@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

ITALY

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://www.ansys.com/customerportal) and select the appropriate option.

Telephone: +39 02 89013378

Email: support-italy@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

xi

Chapter 1: Introduction

The continuous fiber model is provided as an addon module with the standard ANSYS FLUENT licensed software.

Several fiber spinning techniques exist in industrial fiber production. The most common types are melt spinning and dry spinning.

In melt spinning, the polymer is heated above its melting point and extruded in a liquid state through nozzles into a vertical spinning chamber. The molten polymer is processed in an inert gas environment, such as nitrogen, then extruded at high pressure and a constant rate into a cooler air stream, thus solidifying the fiber filaments.

In dry spinning, the liquefaction of the polymer is obtained by dissolving it in a suitable solvent. This technique often is applied to polymers that are destroyed thermally before reaching its melting point or if the production process leads to a solvent/polymer mixture. In the spinning chamber, the solvent vaporizes by drying with a hot air stream. The solidification ensures that the fiber is nearly free of solvent.

ANSYS FLUENT's continuous fiber model allows you to analyze the behavior of fiber flow, fiber properties, and coupling between fibers and the surrounding fluid due to the strong interaction that exists between the fibers and the surrounding gas.

This document describes the ANSYS FLUENT Continuous Fiber model. *Continuous Fiber Model Theory* (p. 3) provides theoretical background information. Instructions for getting started with the model are provided in *Using the Continuous Fiber Module* (p. 19).

1

Chapter 2: Continuous Fiber Model Theory

This chapter presents an overview of the theory and the governing equations for the mathematical model and ANSYS FLUENT's capabilities to predict melt and dry spinning processes.

2.1. Introduction

ANSYS FLUENT's Continuous Fiber Model uses a one-dimensional approach used to predict the flow in fibers and to predict the flow field in the spinning chamber.

In melt spinning, where the extruded molten polymer is sent through the nozzles into the spinning chamber, the velocity of the liquid jet increases due to gravity and the tensile force, which is applied at the take-up point of the fibers. The conservation of mass leads to a decrease in the cross-section of the jet up to the final diameter. The molten polymer is cooled by an air stream until the solidification temperature is reached.

In dry and melt spinning, production of hundreds or thousands of fibers in a spinning chamber leads to strong interaction between the fibers and the surrounding gas, requiring a coupled calculation procedure for the fibers and the fluid flow in the spinning chamber.

2.2. Governing Equations of Fiber Flow

Mass conservation of a fiber element is written as

$$\frac{d}{dz}\left(\rho_f \vec{u}_f \vec{A}_f Y_s\right) = -\pi d_f \dot{m}_s'' \tag{2-1}$$

In this equation ρ_f is the fiber density, $\vec{u}_f$ is the fiber velocity vector, $\vec{A}_f$ is the surface area vector of the fiber surface parallel to the flow direction, Y_s is the mass fraction of the solvent s in the fiber, $\dot{m}_s''$ is the evaporated mass flow rate of the solvent s, and d_f is the fiber diameter. $\dot{m}_s''$ is calculated using a film theory.

$$\dot{m}_s'' = M_s c \beta \ln \left(\frac{1 - \psi_{s,g}}{1 - \psi_{s,I}} \right)$$ (2–2)

The mass transfer coefficient β is estimated from an appropriate correlation, see *Correlations for Momentum, Heat and Mass Transfer* (p. 12). M_S is the solvent's molecular weight, c is the molar concentration of the surrounding gas and $\psi_{s,g}$ is the mole fraction of the solvent vapor in the surrounding gas. At the fiber surface, the mole fraction of the solvent in the gas $\psi_{s,I}$ is related to the solvent mass fraction in the fiber Y_s by the vapor-liquid equilibrium equation given by Flory [1] (p. 49),

$$\psi_{s,I} = \frac{p_{s,vap}}{p} Y_s e^{1 - Y_s + \chi (1 - Y_s)^2}$$ (2–3)

where χ is the Flory-Huggins parameter, p is the absolute pressure in the surrounding flow, and $p_{s,vap}$ is the saturation vapor pressure of the solvent. These equations are used only when dry spun fibers have been selected.

The formation of fibers is based on tensile forces in the fiber that are applied at the take-up point and result in the drawing and elongation of the fiber.

A force balance for a differential fiber element gives the equation of change of momentum in the fiber.

$$\frac{d \left(\rho_f u_f u_f \overline{A}_f \right)}{dz} = \frac{d \overline{F}}{dz} + \overline{F}_{friction} + \overline{F}_{gravitation}$$ (2–4)

The tensile force in the fiber changes due to acceleration of the fiber, friction force with the surrounding gas, and the gravitational forces.

The friction force with the surrounding gas is computed by

$$\overline{F}_{friction} = \frac{1}{2} \rho c_{f,ax} \pi d_f \mid \overline{u}_f - \overline{u}_{par} \mid \left(\overline{u}_f - \overline{u}_{par} \right)$$ (2–5)

where ρ is the gas density, $c_{f,ax}$ is the axial friction factor parallel to the fiber, and $\overline{u}_{par}$ is the gas velocity parallel to the fiber.

The gravitational force is computed from

$$\overline{F}_{gravitation} = \rho_f \frac{\pi}{4} d_f^2 \overline{g} \cdot \overline{n}_f$$ (2–6)

where $\overline{n}_f$ is the direction vector of the fiber element.

The tensile force $\overline{F}$ is related to the components of the stress tensor by

$$\vec{F} = \overline{A}_f (\tau_{zz} - \tau_{rr}) \tag{2–7}$$

Neglecting visco-elastic effects and assuming Newtonian flow one can obtain

$$\tau_{zz} = 2\eta_f \frac{du_f}{dz} \tag{2–8}$$

$$\tau_{rr} = -\eta_f \frac{du_f}{dz} \tag{2–9}$$

leading to

$$\vec{F} = 3\overline{A}_f \eta_f \frac{du_f}{dz} \tag{2–10}$$

The elongational viscosity is estimated by multiplying the zero shear viscosity η_f by three.

The transport of enthalpy in and to a differential fiber element is balanced to calculate the fiber temperature along the spinning line.

$$\frac{d}{dz}\left(\rho_f \vec{u}_f \cdot \overline{A}_f h_f\right) = \frac{d}{dz}\left(\lambda_f \overline{A}_f \frac{dT_f}{dz}\right) + \pi d_f \left(\alpha \left(T - T_f\right) - \dot{m}_s'' h_{s,v}\right) + \tag{2–11}$$

$$\dot{Q}_{viscous\,heating} + \dot{Q}_{radiation,abs} - \dot{Q}_{radiation,emission}$$

where h_f is the fiber enthalpy, λ_f is the fiber thermal conductivity, T_f is the fiber temperature, $h_{s,v}$ is the enthalpy of the solvent vapor, and α is the heat transfer coefficient.

In the case of a melt spinning process, $\dot{m}_s''$ is zero because there is no mass transfer. The term for heat generation due to viscous heating is derived from the fluid mechanics of cylindrical flow to be

$$\dot{Q}_{viscous\,heating} = \frac{\pi}{4}d_f^2\left(4\left(\frac{du_f}{dz}\right)^2 - \frac{2}{3}\dot{m}_s'''^2\right) \tag{2–12}$$

Radiation heat exchange is considered by the last two terms

$$\dot{Q}_{radiation,abs} \quad = \quad d_f \varepsilon_f G \tag{2-13}$$

$$\dot{Q}_{radiation,emission} \quad = \quad \pi d_f \varepsilon_f \sigma T_f^4 \tag{2-14}$$

where G is the thermal irradiation, ε_f is the fiber's emissivity, and σ is the Boltzman constant.

The fiber enthalpy h_f is related to the fiber temperature T_f as follows

$$h_f = \int_{T_{ref}}^{T_f} \left((1 - Y_s) C_{p_p} + Y_s C_{p_s} \right) dT \tag{2-15}$$

It uses C_{p_p} the specific heat capacity of the polymer and C_{p_s} the specific heat capacity of the solvent in the fiber.

The enthalpy of the solvent vapor at a given temperature T_v depends on the heat of vaporization Δh_s, given at the vaporization temperature T_{vap}, and is computed from

$$h_{s,v} = \int_{T_{ref}}^{T_{vap}} C_{p_{s,l}} dT + \Delta h_s \mid_{T_{vap}} + \int_{T_{vap}}^{T_v} C_{p_{s,v}} dT \tag{2-16}$$

where $C_{p_{s,l}}$ is the specific heat capacity of the solvent liquid and $C_{p_{s,v}}$ is the specific heat capacity of the solvent vapor.

2.3. Discretization of the Fiber Equations

The governing *Equation 2–1* (p. 3), *Equation 2–4* (p. 4), and *Equation 2–11* (p. 5) have the form of convection diffusion equations and are discretized using a finite volume scheme of Patankar [5] (p. 49).

The convective terms in *Equation 2–1* (p. 3), *Equation 2–4* (p. 4), and *Equation 2–11* (p. 5) are discretized using first order upwinding, central differencing or the DISC scheme [6] (p. 49). While the first two schemes are well described in the ANSYS FLUENT User's Guide, the reader is referred to [6] (p. 49) to learn more about the DISC scheme. It is the recommended scheme because it provides outstanding numerical stability combined with second order accuracy. The diffusion terms are discretized with second order accuracy. All other terms are treated as source terms and linearized according to Patankar [5] (p. 49).

For more information, please see the following section:

2.3.1. Under-Relaxation

Because the fiber equations are non-linear, the change of the solution variable ϕ has to be controlled. This is achieved by under-relaxation. The new value of ϕ in each cell depends to some degree upon the old value ϕ_{old} and the change in ϕ, $\Delta \phi$. It is computed for a given under-relaxation factor α as follows:

$$\phi = \phi_{old} + \alpha \, \Delta \phi$$

$$(2-17)$$

2.4. Numerical Solution Algorithm of Fiber Equations

Each governing differential equation is discretized into a set of algebraic equations that are solved using the tri-diagonal matrix algorithm. All differential equations for conservation of mass, momentum, energy, and (when appropriate) for solvent in the fiber are solved sequentially (that is, segregated from one another). Because the governing equations are coupled and non-linear, several iterations have to be performed to obtain a converged solution. The solution process consists of several steps outlined below:

1. The fiber properties are updated based on the initialized or the current solution.

2. The friction factors for momentum exchange between the fibers and surrounding fluid are computed based on current values of fiber and fluid velocities.

3. The fiber momentum equation is solved and current values of the mass fluxes in the fiber are used.

4. The heat transfer coefficients are computed using Reynolds numbers from the beginning of the iteration loop.

5. The fiber energy equation is solved.

6. In the case of dry spun fibers, the equation for the mass fraction of the solvent is solved. First, the mass transfer coefficient is updated. The evaporated (condensed) mass is computed based on the vapor liquid equilibrium at the beginning of the iteration loop. Finally, the governing equation is solved.

7. The mass fluxes and the diameter of the fiber cells are updated.

8. A check for convergence of the equation set is made.

These steps are continued until the convergence criteria are met for all equations of the considered fiber or until the number of iterations exceed the given limit.

This solution algorithm is applied to all defined fibers.

2.5. Residuals of Fiber Equations

The solution algorithm for the fiber equations requires a means for checking convergence of the solution. In the fiber model a simple residual is used for this purpose.

The conservation equation for a general variable ϕ at a cell P can be written as

$$a_P \phi_P = \sum_{nb} a_{nb} \phi_{nb} + S_c + S_P \phi_P$$

$$(2-18)$$

where a_P is the center coefficient, a_{nb} are the influence coefficients for the neighboring cells, S_c is the constant part of the source term, and S_P is the linear part of the source term.

The residual R^ϕ computed by the fiber model is the imbalance in *Equation 2–18* (p. 7) summed over all fiber cells.

$$R^\phi = \sum_{fibercells} \left| \sum_{nb} a_{nb}\phi_{nb} + S_c + S_P\phi_P - a_P\phi_P \right|$$ **(2–19)**

This is called the absolute residual. Relative residuals are defined as the change of the absolute residuals between two subsequent iterations divided by the absolute residual.

$$\hat{R}^\phi = \frac{R^\phi_{iterationN} - R^\phi_{iterationN-1}}{R^\phi_{iterationN}}$$ **(2–20)**

2.6. Coupling Between Fibers and the Surrounding Fluid

The fiber model accounts for all mass, momentum, and heat gained or lost by the fibers and can be incorporated into the subsequent calculations of the surrounding fluid performed by ANSYS FLUENT. This means that you can incorporate the effects of the fibers on the surrounding fluid, while the surrounding fluid always impacts the fiber flow. This two-way coupling is achieved by alternatively solving the fiber and the surrounding fluid flow equations until both phases have converged. The interphase exchange of heat, mass, and momentum from the fibers to the surrounding fluid is depicted qualitatively in *Figure 2.1* (p. 9).

For more information, please see the following sections:

2.6.1. Momentum Exchange

Figure 2.1 Fiber Grid Penetrating Grid of the Gas Flow
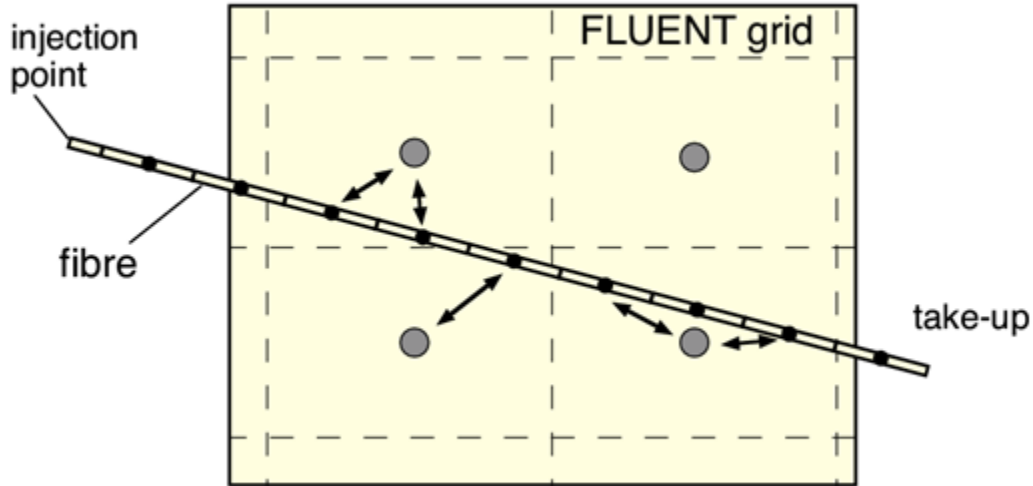
Momentum transfer from the fibers to the surrounding fluid is computed in ANSYS FLUENT by considering the change of momentum of the fiber as it crosses each control volume in the ANSYS FLUENT model. It is computed as

$$\vec{F}_c = \sum_{fibers} \left(\frac{\rho}{2} \pi d_f l_{f,c} \left(c_{f,par} \left(\overline{u}_f - \overline{u}_{par} \right)^2 - c_{f,lat} \left(\overline{u}_f - \overline{u}_{lat} \right)^2 \right) + \overline{u}_f \pi d_f l_{f,c} \dot{m}_s'' \right)$$ **(2–21)**

where

ρ = density of the fluid

d_f = diameter of the fiber

$l_{f,c}$ = length of the fiber f in cell c

$\overline{u}_f$ = velocity of the fiber

$\overline{u}_{par}$ = velocity of the fluid, parallel to the fiber

$\overline{u}_{lat}$ = velocity of the fluid, lateral to the fiber

$c_{f,par}$ = drag coefficient parallel to the fiber

$c_{f,lat}$ = drag coefficient lateral to the fiber

$\dot{m}_s''$ = evaporated mass flow rate of the solvent in the fiber

This momentum exchange appears as a momentum source in the surrounding fluid momentum balance and is taken into account during every continuous phase computation. It can be reported as described in *Exchange Terms of Fibers* (p. 45).

2.6.2. Mass Exchange

The mass transfer of evaporating solvent from the fibers in dry spinning applications is computed in ANSYS FLUENT by balancing the evaporated mass of solvent in every fiber cell volume. It is computed as

$$M_c = \sum_{fibers} \pi d_f l_{f,c} \dot{m}_s''$$

(2–22)

The mass exchange appears as a mass source in the continuity equation of the surrounding fluid as well as a source of chemical species of the solvent vapor defined by you. It is included in every subsequent calculation of the continuous phase flow field and is reported as described in *Exchange Terms of Fibers* (p. 45).

2.6.3. Heat Exchange

The heat transfer from the fibers to the surrounding flow is computed in ANSYS FLUENT by balancing the change of the fiber energy as it crosses each control volume in the ANSYS FLUENT model. It considers sensible as well as latent heat transfer due to evaporation of solvent in dry spinning applications and is computed as

$$Q_c = \sum_{fibers} \left(\pi d_f l_{f,c} \left(\alpha \left(T_f - T \right) + \dot{m}_s'' \left(\frac{1}{2} \bar{u}_f^2 + \int_{T_{ref}}^{T_f} C_{p_{s,v}} dT \right) \right) + \bar{u}_f \vec{F}_f \right)$$

(2–23)

where

α = heat transfer coefficient

T_{ref} = reference temperature

$C_{p_{s,v}}$ = specific heat capacity of solvent vapor

$\vec{F}_f$ = momentum exchange of fiber f

This heat exchange appears as a source in the surrounding fluid energy balance and is taken into account during every continuous phase computation. It can be reported as described in *Exchange Terms of Fibers* (p. 45).

2.6.4. Radiation Exchange

The fibers participate in radiation exchange by absorbing irradiation energy from the surrounding flow irradiation and by emitting irradiation at the fiber temperature. This effect on the irradiation of the surrounding flow is considered by computing the absorbed and emitted energy of the fibers in each cell as

$$G_{abs} \quad = \quad \sum_{fibers} \frac{\varepsilon_f}{4} A_f G \tag{2–24}$$

$$G_{emiss} \quad = \quad \sum_{fibers} \frac{\varepsilon_f}{4} \frac{A_f}{\pi} \sigma T_f^4 \tag{2–25}$$

where

A_f = fiber surface area

σ = Boltzman constant

G = irradiation of the surrounding flow

ε_f = emissivity of the fiber

Important

The transfer of the fiber radiation energy to the surrounding flow is only considered when the single-band Discrete-Ordinate Model is used in the ANSYS FLUENT flow model. While radiation effects on the fibers are taken into account when the P1 model is used, there is no two-way coupling for this model.

2.6.5. Under-Relaxation of the Fiber Exchange Terms

Note that the exchange of momentum, heat, and mass from the fibers is under-relaxed during the calculation, so that

$$\vec{F}_{c,new} = \vec{F}_{c,old} + \alpha \left(\vec{F}_{c,calculated} - \vec{F}_{c,old} \right) \tag{2–26}$$

$$M_{c,new} = M_{c,old} + \alpha \left(M_{c,calculated} - M_{c,old} \right) \tag{2–27}$$

$$Q_{c,new} = Q_{c,old} + \alpha \left(Q_{c,calculated} - Q_{c,old} \right) \tag{2–28}$$

where α is the under-relaxation factor for fibers that you can set in the **Solution Controls** task page. The default value for α is 0.5. This value may be reduced in order to improve the stability of coupled calculations. Note that the value of α does not influence the predictions obtained in the final converged solution.

2.7. Fiber Grid Generation

The fibers penetrate the grid of the surrounding flow field arbitrarily. Both grids are treated distinctly from each other (see *Figure 2.1* (p. 9)).

The fiber grids are generated by defining fiber injections. The fibers are considered to be straight lines between injection points and a take-up point. Each fiber is divided into a number of volume cells.

For the grid generation, the following grid types are available:

equidistant
All cells of the fiber have the same length.

one-sided
The cells are graded near the injection point of the fiber and change their size according to a specified growth factor.

two-sided
In addition to the injection point the cells can also be graded at the take-up point by specifying a second growth factor at the end of the fiber.

three-sided
The second point where the fiber cells are graded at the end can be moved to a local refinement point laying between injection and take-up point. This generates fibers with a mesh graded at the injection point and at the local refinement point.

All grid types, except the **equidistant** grid type, require the specification of a growth factor R, which is the ratio of two subsequent fiber grid cells. It refines the mesh for values larger than 1 and coarsens the mesh for values smaller than 1.

If a finite fiber volume cell spans across several ANSYS FLUENT grid cells, a weighted average is used to estimate the corresponding variables of the surrounding flow. This averaging procedure considers the intersection point of each fiber volume cell with the boundaries of the ANSYS FLUENT grid cell.

When computing the source terms in an ANSYS FLUENT grid cell, only the part of each fiber volume cell that is inside the ANSYS FLUENT grid cell is taken into account. This provides a proper computation of ANSYS FLUENT fiber interactions even in hanging node adapted grids.

Important

If the grid is adapted, the data structures that include information about neighbor cells are not updated automatically. For this you have to reinitialize all fibers to start a new search of the neighboring ANSYS FLUENT grid cells.

2.8. Correlations for Momentum, Heat and Mass Transfer

The fiber model makes use of correlations to compute transfer of momentum, heat, and mass to the fibers. The fibers are subject to several physical effects that can be considered based on experimental correlations. Some of these effects may be lateral or longitudinal oscillations due to the applied take-up system, or to gas flow turbulence in the spinning chamber caused by the gas supply system of the spinning chamber. You can choose to specify a constant for the drag, heat transfer or mass transfer coefficients or use one of the Fluent-provided methods (for example, `kase-matsuo`) to compute the coefficients. These methods are described below. Alternatively, you can specify a custom drag, heat transfer, or mass transfer coefficient using a user-defined function (UDF). See *User-Defined Functions (UDFs) for the Continuous Fiber Model* (p. 38) for details on how to define and use UDFs in the fiber model.

For more information, please see the following sections:

2.8.1. Drag Coefficient

The following options for drag coefficients are available in the fiber model to compute the drag due to flow moving parallel to the fibers:

const-drag
    A constant value for the drag can be specified.

kase-matsuo
    A drag coefficient using the model taken from Kase and Matsuo [3] (p. 49), see *Equation 2–29* (p. 14).

gampert
    A drag coefficient using the model from Gampert [2] (p. 49).

    Gampert provided analytical and numerical solutions for laminar axisymmetric flow of a moving cylinder in stationary air including strong curvature effects in the boundary layer, [2] (p. 49). The drag coefficient and the Nusselt number are shown as dimensionless groups in *Figure 2.2* (p. 13).

    Note that the curvature k is defined as the abscissa in *Figure 2.2* (p. 13). This correlation is recommended in laminar flows.

Figure 2.2 Dimensionless Groups of Drag Coefficient and Nusselt Number
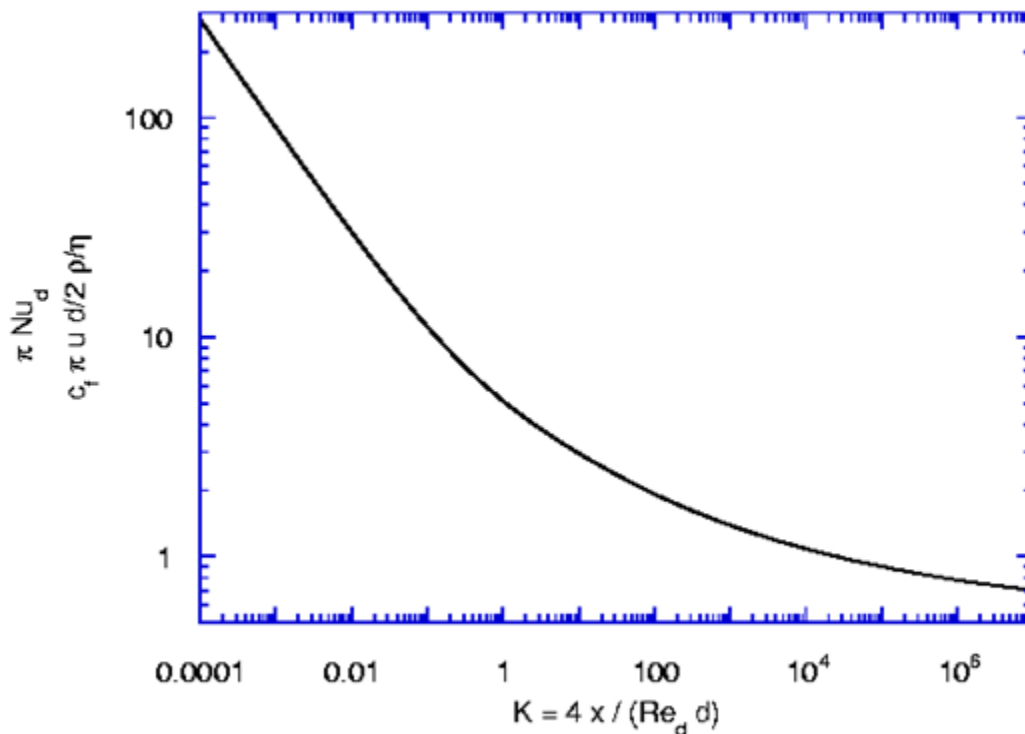
user-defined
    A drag coefficient that you specify in a user-defined function (UDF). See *User-Defined Functions (UDFs) for the Continuous Fiber Model* (p. 38) for more information on using UDFs in the fiber model.

$$c_{f,par} = \frac{1.24}{Re_d^{0.81}}$$ (2–29)

In *Figure 2.2* (p. 13) and *Equation 2–29* (p. 14), the Reynolds number is computed based on the relative velocity of the surrounding flow parallel to the fibers $Re_d = \frac{\rho d \left(u_f - u_{par}\right)}{\eta}$.

Lateral drag due to flow of the surrounding fluid perpendicular to the fibers is computed by a correlation from Schlichting [7] (p. 49)

$$c_{f,lat} = 10^{\left(a_1 + a_2 \log Re_{d,lat} + a_3 \log^2 Re_{d,lat}\right)}$$ (2–30)

In *Equation 2–30* (p. 14) the Reynolds number is computed based on the relative velocity of the surrounding flow perpendicular to the fibers $Re_{d,lat} = \frac{\rho d u_{lat}}{\eta}$.

2.8.2. Heat Transfer Coefficient

The following options for heat transfer coefficients are available in the fiber model to compute the exchange of heat between fibers and surrounding flow:

const-htc
A constant value for the heat transfer coefficient in SI units can be specified.

kase-matsuo-1
A heat transfer coefficient based on a model from Kase and Matsuo [3] (p. 49) that considers pure parallel flow, see *Equation 2–31* (p. 14).

$$Nu_d = \frac{\alpha d_f}{\lambda} = 0.42 Re_d^{0.334}$$ (2–31)

kase-matsuo-2
A heat transfer coefficient based on a model from Kase and Matsuo [3] (p. 49) that also considers cross flow, see *Equation 2–32* (p. 14). Refer to *Momentum Exchange* (p. 9) for definitions of the variables below.

$$Nu_d = \frac{\alpha d_f}{\lambda} = 0.42 Re_d^{0.334} \left[1 + \left(\frac{8 u_{lat}}{u_f - u_{par}} \right)^2 \right]^{1/6}$$ (2–32)

gampert
A heat transfer coefficient based on a model from Gampert [2] (p. 49).

Gampert provided analytical and numerical solutions for laminar axisymmetric flow of a moving cylinder in stationary air including strong curvature effects in the boundary layer, [2] (p. 49). The drag coefficient and the Nusselt number are shown as dimensionless groups in *Figure 2.2* (p. 13). This correlation is recommended for laminar flows.

user-defined

A heat transfer coefficient that you specify in a user-defined function (UDF). See *User-Defined Functions (UDFs) for the Continuous Fiber Model* (p. 38) for more information on using UDFs in the fiber model.

2.8.3. Mass Transfer Coefficient

The mass transfer coefficients are based on the Reynolds analogy applied to the heat transfer coefficients. The following options are available in the fiber model to compute the exchange of mass between fibers and surrounding flow:

const-mtc

If you select this coefficient, you specify the direct transferred mass flow rate in $kg/(m^2 s)$, rather than the mass transfer coefficient.

kase-matsuo-1

A mass transfer coefficient based on a model from Kase and Matsuo [3] (p. 49) that considers pure parallel flow, see *Equation 2–33* (p. 15).

$$Sh_d = \frac{\beta d_f}{D_s} = 0.42 Re_d^{0.334} \tag{2–33}$$

kase-matsuo-2

A mass transfer coefficient based on a model from Kase and Matsuo [3] (p. 49) that also considers cross flow, see *Equation 2–34* (p. 15).

$$Sh_d = \frac{\beta d_f}{D_s} = 0.42 Re_d^{0.334} \left[1 + \left(\frac{8u_{lat}}{u_f - u_{par}} \right)^2 \right]^{1/6} \tag{2–34}$$

gampert

A mass transfer coefficient based on a model from Gampert [2] (p. 49).

Gampert's analytical and numerical solutions for laminar axisymmetric flow of a moving cylinder in stationary air include strong curvature effects in the boundary layer, [2] (p. 49). The Sherwood number is analogous to the Nusselt number as shown as dimensionless groups in *Figure 2.2* (p. 13). This correlation is recommended for laminar flows.

user-defined

A mass transfer coefficient that you specify in a user-defined function (UDF). See *User-Defined Functions (UDFs) for the Continuous Fiber Model* (p. 38) for more information on using UDFs in the fiber model.

Because these correlations are valid only for nearly-zero mass transfer due to the Reynolds analogy, a film theory is used to compute the non-zero mass transfer, see *Equation 2–2* (p. 4).

2.9. Fiber Properties

Most properties needed for the fibers can be specified in ANSYS FLUENT's material dialog box except the ones described in this section.

2.9.1. Fiber Viscosity

Fibers are treated as Newtonian fluids. In elongational flow of Newtonian fluids, the elongational viscosity (or sometimes called Trouton viscosity) is related to the zero shear viscosity by a factor of 3. Because this approach is applied to the computation of melt and dry spun fibers, only the zero shear viscosity is described.

2.9.1.1. Melt Spinning

In melt spinning, the fiber is considered to be liquid until its temperature falls below the solidification temperature $T_{f,solid}$. For the liquid state an exponential approach is used, see *Equation 2–35* (p. 16).

$$\eta_0 = A e^{\left(\frac{B}{C + T_f}\right)} \tag{2–35}$$

Below this temperature the value given in the material dialog box for the fiber polymer material is used. Typically, a high value like $\eta_0 = 1 x 10^8 Pas$ is used for the fiber viscosity to simulate a solid fiber. This value may depend on your polymer and the range of viscosity values in your simulation. You can use every profile available in the materials dialog box except UDF's to describe temperature dependency of the viscosity of the solidified fiber.

The fiber model uses a blending interval for the temperature $\Delta T_{f,bl} = T_{f,liquid} - T_{f,solid}$ to provide a smooth transition of the viscosity between liquid and solid state of the fiber. The viscosity in this blending interval is computed as

$$\eta_0 = \frac{\Delta T_{f,bl}}{\frac{T_{f,liquid} - T_f}{\eta_{f,solid}\left(T_f\right)} + \frac{T_f - T_{f,solid}}{\eta_{f,liquid}\left(T_f\right)}} \tag{2–36}$$

Important

The chosen values of the blending interval may influence the results. Values for the blending interval should be adapted to the rheological data of the polymer.

2.9.1.2. Dry Spinning

In dry spinning, the following approach is used to consider the effect of solvent on the zero shear viscosity:

$$\eta_0 = A B^C \left(1 - Y_s\right)^D e^{\left(E / T_f\right)} \tag{2–37}$$

In this equation, B is considered to be the degree of polymerization as it is used by Ohzawa [4] (p. 49).

2.9.2. Vapor-Liquid Equilibrium

In dry spinning applications, the solvent in the fiber is in thermodynamic equilibrium with the solvent vapor in the surrounding fluid. The saturation vapor pressure of the solvent is computed using an Antoine-type equation

$$p_{s,vap} = e^{\left(A + \frac{B}{T_f + C} \right)}$$

(2–38)

If you enable the vapor-liquid equilibrium given by Flory [1] (p. 49), *Equation 2–3* (p. 4) is used to compute the mole fraction of the solvent in the surrounding gas.

2.9.3. Latent Heat of Vaporization

The latent heat of vaporization of the solvent Δh_s has to be specified at the vaporization temperature T_{vap}. The temperature dependency of the latent heat is not needed because this is treated automatically by the fiber model. Using *Equation 2–16* (p. 6) the following equation can be derived to get the latent heat of vaporization at any temperature $T^\star$:

$$\Delta h_s \left(T^\star \right) = \Delta h_s \left(T_{vap} \right) + \int_{T_{vap}}^{T^\star} \left(C_{p_{s,v}} - C_{p_s} \right) dT$$

(2–39)

2.9.4. Emissivity

If the P-1 or the discrete ordinates radiation model is enabled, the emissivity of the fiber has to be specified. The fiber diameter can approach the order of magnitude of the wavelength of the irradiation. In this parameter range, additional effects like diffraction take place in addition to scattering and transmission. Because these properties are not well known for fibers, diffraction, scattering, and transmission are neglected. They have to be included only in the fiber emissivity.

2.10. Solution Strategies

The fiber model solves sets of differential equations. It provides its own numerical algorithms showing their own numerical difficulties. Therefore it is highly recommended to first achieve a numerical solution for the pure fibers without any coupling to the surrounding flows. If the fiber model does not converge for a given flow field of the surrounding fluid it will not converge for changing fields of the surrounding fluid.

When you start with a fiber simulation, chose the appropriate models needed for the fibers. For the first simulation, disable options like **Include Lateral Drag**, **Fiber Radiation Interaction**, and **Fiber Viscous Heating** to reduce possible interactions.

When specifying the grid of the fibers be sure to refine the grid in the area where large gradients of the velocity appear. This is mainly near the injection point where the fiber is released and near the point of solidification. Because this point is not known a priori, you have to refine the grid during subsequent steps.

If the fiber grid seems to be well suited, you can influence the convergence behavior by starting the iteration with low under-relaxation factors in the **Fiber Solution Controls** dialog box. This may help in most situations where the species and energy equations are strongly coupled (for example, dry spinning applications), or if the solvent has a very high latent heat of vaporization (for example, water).

Important

Be sure to increase the under-relaxation factor of the momentum equation to 1, when doing a melt spinning case to achieve a converged solution. This should be done after a numerically stable solution has been set up.

When the solution process of the pure fiber equations show a numerically stable behavior, you can increase the complexity of the models by activating viscous heating, or radiation interaction, if such effects are important in their application. After this, you can proceed with a coupled solution by solving the fiber equations and the fluid flow equations alternately. If the solution algorithm of the fluid flow equations diverges, you must investigate the source terms computed by the fiber model, see *Exchange Terms of Fibers* (p. 45).

You can damp strong changes of the source terms with a low under-relaxation factor. Another choice is to increase the number of ANSYS FLUENT iterations between two subsequent fiber computations.

If the coupled solution process converges, you can increase the under-relaxation factor of the source terms and decrease the number of ANSYS FLUENT iterations between two subsequent fiber computations.

You may also want to consider underrelaxing the fluid flow equations. This helps especially for the energy and species equations.

Chapter 3: Using the Continuous Fiber Module

The procedure for setting up and solving fiber spinning flows is described in detail in this chapter. Only the steps related to fiber modeling are shown here. Refer to *Continuous Fiber Model Theory* (p. 3) for information about the theory.

Important

Note that the continuous fiber model is available for the pressure-based solver, only.

For information about inputs related to other models used in conjunction with the fiber models, see the appropriate sections for those models in the ANSYS FLUENT User's Guide.

3.1. Installing the Continuous Fiber Module

The continuous fiber model is provided as an add-on module with the standard ANSYS FLUENT licensed software. The module is installed with the standard installation of ANSYS FLUENT in a directory called `addons/fiber` in your installation area. The continuous fiber module consists of a UDF library and a pre-compiled scheme library, which need to be loaded and activated before calculations can be performed.

3.2. Loading the Continuous Fiber Module

The continuous fiber module is loaded into ANSYS FLUENT through the text user interface (TUI). The module can be loaded only when a valid ANSYS FLUENT case file has been set or read. The text command to load the module is

define → models → addon-module

A list of ANSYS FLUENT add-on modules is displayed:

```
 > /define/models/addon-module
 FLUENT   Addon Modules:
      0. None
      1. MHD Model
      2. Fiber Model
      3. Fuel Cell and Electrolysis Model
      4. SOFC Model with Unresolved Electrolyte
      5. Population Balance Model
      6. Adjoint Solver
```

```
      7. Battery Module
Enter Module Number: [0] 2
```

Select the continuous fiber model by entering the module number 2. During the loading process a scheme library containing the graphical and text user interface, and a UDF library containing a set of user-defined functions (UDFs) are loaded into ANSYS FLUENT.

During this process, you will be asked the question

```
  Preset all fiber model specific UDF hooks? [no]
```

If you answer yes the standard fiber source term UDFs will be assigned to *all* fluid zones in your case, and a message will be reported to the console window confirming this:

```
  Assigning standard fiber source terms to all fluid zones.
```

If you answer no to presetting source term UDFs to all fluid zones in the domain, then three options will be available to you when setting up source terms for fluid zones in your fiber model: no source, constant source, or UDF source. Note that it is your responsibility to specify the rest of the settings for a proper fiber simulation. See *Source Term UDF Setup* (p. 21) for details.

If you are loading an existing case file then you should answer the question with no. Otherwise, your saved source term settings will be replaced by a UDF.

If a mixture material has been defined, then you will be asked an additional question

```
  Preset also fiber model UDFs which handle mass exchange source terms? [no]
```

If you intend to conduct a dry spinning simulation, then you should reply yes.

During the loading process the UDF library for the continuous fiber module is loaded in ANSYS FLUENT. This is reported to the console (see below). The UDF library also becomes visible as a new entry in the **UDF Library Manager** dialog box. The basic setup of the continuous fiber model is performed automatically when the fiber module is successfully loaded.

```
  Opening library "/.../addons/fiber"...
  Library "/...addons/fiber/lnx86/3d/libudf.so" opened
  fm_adjust
  fm_src_mass
  fm_src_x_mom
  fm_src_y_mom
  fm_src_z_mom
  fm_src_enthalpy
  fm_src_dom
  fm_on_demand Done.

  Addon Module: fiber...loaded!
```

Before you will be able to use these UDFs in your fiber model, you will need to allocate user-defined memory and hook an adjust function (fm_adjust) to ANSYS FLUENT. This is explained is *User-Defined Memory and the Adjust Function Setup* (p. 21).

Important

Note that user-defined memory locations for the fiber model will not be allocated properly if you do not initialize the flow field. If you are setting up a fiber computation based on a converged case, you *must* re-load the ANSYS FLUENT data file after initializing the solution.

The continuous fiber module setup is saved with the ANSYS FLUENT case file. The module is loaded automatically when the case file is subsequently read into ANSYS FLUENT. Note that in the saved case file, the continuous fiber module is saved with the absolute path. Therefore, if the locations of the continuous fiber module installation or the saved case file are changed, then ANSYS FLUENT will not be able to load the module when the case file is subsequently read. In this situation, you will have to unload the UDF library using the **UDF Library Manager** dialog box after the case file is read, and then reload the continuous fiber module. To unload the UDF library go to the **UDF Library Manager** dialog box

Define → **User-Defined** → **Functions** → **Manage...**

select the fiber library under **UDF Libraries**, and click **Unload**. Previously-saved continuous fiber model setup and parameters will be preserved in this process.

3.3. Getting Started With the Continuous Fiber Module

The continuous fiber model is implemented by user-defined functions (UDFs) and scheme routines in ANSYS FLUENT. A number of UDFs are used to solve the fiber equations. When you loaded the fiber module in the previous step (*Loading the Continuous Fiber Module* (p. 19)), UDF and scheme libraries that are required by the continuous fiber model were *automatically* loaded. Before you can begin the process of defining your fiber model, however, you will need to do perform some additional setup tasks that involve allocating user-defined memory for the UDFs and hooking an adjust UDF to ANSYS FLUENT. Follow the procedure below.

For more information, please see the following sections:
3.3.1. User-Defined Memory and the Adjust Function Setup
3.3.2. Source Term UDF Setup

3.3.1. User-Defined Memory and the Adjust Function Setup

1. Allocate user-defined memory for the model by incrementing the **Number of User-Defined Memory Location** to 8 in the **User-Defined Memory** dialog box.

    Define → **User-Defined** → **Memory...**

    ***

    Important

    Note that you *must* initialize your solution (in the **Solution Initialization** task page) in order for user-defined memory to be allocated properly. If you are setting up a fiber simulation based on a converged case, then you will have to reload the ANSYS FLUENT data file after initializing the solution.

2. Hook the adjust function UDF to ANSYS FLUENT by choosing `fm_adjust::fiber` from the drop-down list for **Adjust** in the **User-Defined Function Hooks** dialog box.

    Define → **User-Defined** → **Function Hooks...**

3.3.2. Source Term UDF Setup

If you answered `no` to presetting all fiber model-specific UDF hooks during the loading process (*Loading the Continuous Fiber Module* (p. 19)) then you will need to set source terms, individually, for each fluid zone in your model. Alternatively, you can leave the default settings (`none` for no source term).

For each fluid zone in your model, specify `none`, `constant`, or UDF for all of the source terms by following the procedure below:

⬍**Cell Zone Conditions**

1. In the **Cell Zone Conditions** task page, select a fluid zone under **Zone** and click **edit...**. This opens the **Fluid** dialog box.

2. In the **Fluid** dialog box, check **Source Terms** and click on the **Source Terms** tab.

3. For each of the source terms in the scroll list (**Mass, X Momentum**, and so on), click the **Edit...** button next to each source term to open the corresponding source dialog box. Leave the default `none`, or choose `constant` or UDF from the drop-down list. Choose the UDF in the drop-down list that corresponds to the particular source term. For example, `udf fm_src_mass` corresponds to the `Mass` source term. Use the table below (*Table 3.1: Source Terms and Corresponding UDFs* (p. 22)) as a reference guide.

Table 3.1 Source Terms and Corresponding UDFs

Mass	`udf fm_src_mass`
X Momentum or Axial Momentum	`udf fm_src_x_mom`
Y Momentum or Radial Momentum	`udf fm_src_y_mom`
Z Momentum	`udf fm_src_z_mom`
Energy	`udf fm_src_enthalpy`
discrete ordinates model	`udf fm_src_dom`

4. Click **OK** when all of the UDFs have been hooked.

5. Repeat this process for the remaining fluid zones in your ANSYS FLUENT model.

> **Important**
>
> If you want to include radiative interaction of the fibers with the discrete ordinate (DO) radiation model, then the appropriate source term UDF (`udf fm_src_dom`) will be hooked *automatically* when you select **Fiber Radiation Interaction** in the **Fiber Model** dialog box. You *must* initialize the solution (which will allocate memory for the DO model) before the fiber model will be ready to accept the fiber radiation interaction data.

3.4. Fiber Models and Options

This section provides information on how to choose the type of fiber model you want to implement as well as select various model options that are available in ANSYS FLUENT's fiber model. The fiber **Model** and **Options** are selected in the **Fiber Model** dialog box (*Figure 3.1* (p. 23)).

⬍**Models** → ≡ **Continuous Fiber Spinning** → **Edit...**

Figure 3.1 The Fiber Model Dialog Box
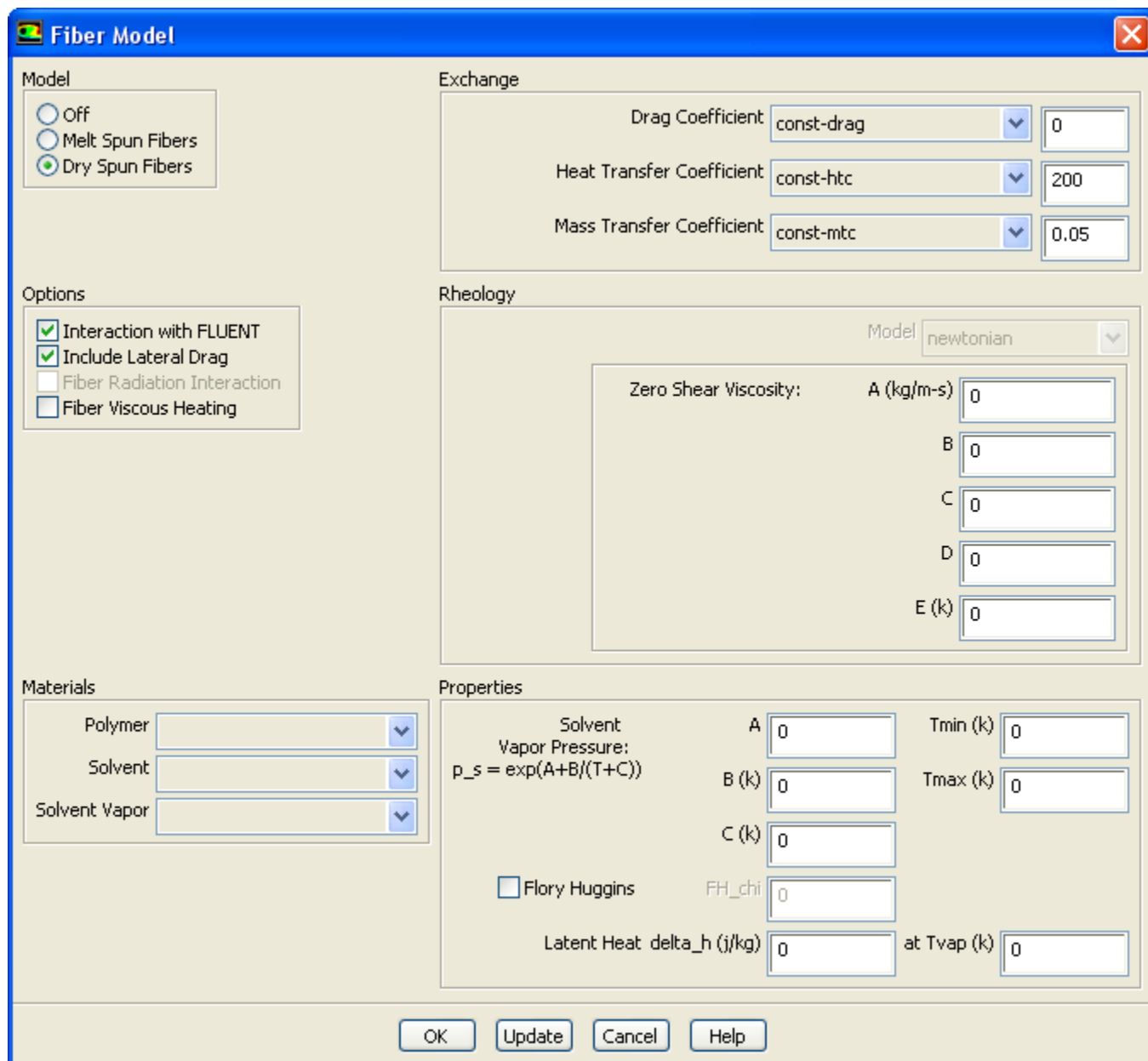
For more information, please see the following sections:

3.4.1. Choosing a Fiber Model

With ANSYS FLUENT's fiber model, you can model two types of fibers: melt spun and dry spun. The model you choose depends on the polymer that is used to draw the fibers. If the fiber polymer can be molten without being destroyed thermally, then a melt spun fiber process is typically used to produce

the fibers. In such cases, select **Melt Spun Fibers** under **Model** in the **Fiber Model** dialog box (*Figure 3.1* (p. 23)).

When the fiber polymer can be liquefied with a suitable solvent, or the fiber polymer's production process involves a solvent, the fibers are formed typically in a dry spinning process. In such cases, select **Dry Spun Fibers** under **Model** in the **Fiber Model** dialog box (*Figure 3.1* (p. 23)).

3.4.2. Including Interaction With Surrounding Flow

If the fibers in your simulation strongly influence the flow of the surrounding fluid and need to be considered, you must select **Interaction with FLUENT** under **Options** in the **Fiber Model** dialog box (*Figure 3.1* (p. 23)). When iterating a solution with ANSYS FLUENT, the fiber model equations are solved alternating with ANSYS FLUENT's flow equations. Source terms are also computed to couple the fiber equations with the fluid flow equations. The calculation of the source terms is performed only during the course of an ANSYS FLUENT computation. See *Coupling Between Fibers and the Surrounding Fluid* (p. 8) for a description of the source terms.

3.4.3. Including Lateral Drag on Surrounding Flow

In typical fiber simulations, the axial drag of the fibers is the most important force acting on the fibers as well as on the surrounding fluid. In some situations, the lateral or cross-flow drag can become important. This is the case when the fibers are mainly cooled or dried in a cross-flow situation. In such cases you can select **Include Lateral Drag** under **Options** in the **Fiber Model** dialog box (*Figure 3.1* (p. 23)). The drag is estimated for a cylinder in cross-flow and is shown in *Equation 2–30* (p. 14). Lateral drag is not considered by the fiber equations and therefore lateral bending of the fibers is not considered.

3.4.4. Including Fiber Radiation Interaction

In some situations radiative heat exchange is important. If you are using ANSYS FLUENT's P-1 radiation model or ANSYS FLUENT's discrete ordinates (DO) radiation model, you can consider the effects of irradiation on the cooling and heating of the fibers. When you are using the DO radiation model, the effects of the fibers upon the DO model is considered as well. If the DO radiation model is enabled, the **Fiber Radiation Interaction** option will be turned on and you will need to enter the fiber's **Emissivity** under **Properties** in the **Fiber Model** dialog box (*Figure 3.1* (p. 23)).

3.4.5. Viscous Heating of Fibers

When high elongational drawing rates are combined with large fiber viscosities, viscous heating of fibers may become important. To consider this effect in the fiber energy equation (*Equation 2–12* (p. 5)), you can select **Fiber Viscous Heating** under **Options** in the **Fiber Model** dialog box (*Figure 3.1* (p. 23)).

3.4.6. Drag, Heat and Mass Transfer Correlations

The effects of the boundary layer of the fiber are modeled in terms of drag, heat transfer, and mass transfer coefficients. These parameters are specified under **Exchange** in the **Fiber Model** dialog box (*Figure 3.1* (p. 23)).

- For the **Drag Coefficient**, you can choose between **const-drag**, **kase-matsuo**, **gampert** and **user-defined** from the drop-down list. If you choose **const-drag**, the constant you enter must be specified as a dimensionless value. See *Drag Coefficient* (p. 13) for a description of these options. User-defined functions (UDFs) are described in detail in *User-Defined Functions (UDFs) for the Continuous Fiber Model* (p. 38).

- For the **Heat Transfer Coefficient** you can choose between **const-htc**, **kase-matsuo-1**, **kase-matsuo-2**, **gampert**, and **user-defined** from the drop-down list. If you choose **const-htc**, the constant you enter must be specified SI units of $W / (m^2 K)$. See *Heat Transfer Coefficient* (p. 14) for a description of these options. User-defined functions (UDFs) are described in detail in *User-Defined Functions (UDFs) for the Continuous Fiber Model* (p. 38).

- For the **Mass Transfer Coefficient** you can choose between **const-mtc**, **kase-matsuo-1**, **kase-matsuo-2**, **gampert**, and **user-defined**. If you choose **const-mtc**, you must enter the mass transfer rate in units of $kg / (m^2 s)$ instead of the mass transfer coefficient. See *Mass Transfer Coefficient* (p. 15) for a description of these options. User-defined functions (UDFs) are described in detail in *User-Defined Functions (UDFs) for the Continuous Fiber Model* (p. 38).

3.5. Fiber Material Properties

For more information, please see the following sections:

3.5.1. The Concept of Fiber Materials

The material properties you specify for the fibers are used for all fibers defined in your model. You cannot consider fibers consisting of different fiber materials in one simulation.

The continuous fiber model makes use of ANSYS FLUENT's material concept for the **Material Type** of the **fluid**. Because not all properties are available in ANSYS FLUENT's **Create/Edit Materials** dialog box for this material type, some additional property information can be provided through the **Fiber Model** dialog box (*Figure 3.1* (p. 23)). The procedure to define the material properties for the fibers in your simulation is as follows:

1. In the **Create/Edit Materials** dialog box, set the **Material Type** as **fluid**. This fluid will be used as the polymer or solvent in the fiber.

2. Enter all data for this material.

3. Select the **Polymer** or **Solvent** in the **Materials** group box of the **Fiber Model** dialog box.

4. For dry spinning simulations, you also have to select the gas phase species that represents the **Solvent Vapor**.

5. Enter any additional data needed for the fiber material in the **Fiber Model** dialog box.

Important

You can use all profiles available in the **Create/Edit Materials** dialog box to define the properties as functions of temperature, except user-defined profiles.

3.5.2. Description of Fiber Properties

The properties that appear in the **Fiber Model** dialog box vary depending on the fiber model type.

The following list describes the properties you may need for a fiber material. For every property listed, the dialog box name is provided where the property can be defined.

Blending Interval

> (**Fiber Model** dialog box) is the temperature interval used to compute an average of the fiber viscosities in liquid and solid state of **Melt Spun Fibers**. This option is only available when the **Melt Spun Fibers** option is selected. See *Fiber Viscosity* (p. 16) for details about how the **Blending Interval** is applied to the fiber viscosity.

Cp

> (**Create/Edit Materials** dialog box) is the specific heat, $C_{p,f}$, of the fiber in units of energy per mass and temperature. In the case of dry spun fibers, a mass average is computed based on the values entered for **Polymer** and its **Solvent**. You can use any of the functions available to define temperature dependency, except user-defined functions.

Density

> (**Create/Edit Materials** dialog box) is the density, ρ_f, of the fiber in units of mass per unit volume. This density is the mass density and not the volume density. In the case of dry spun fibers, a mass average is computed based on the values entered for **Polymer** and its **Solvent**. You can use any of the functions available to define temperature dependency, except user-defined functions.

Emissivity

> (**Fiber Model** dialog box) is the emissivity of fibers in your model, ε_f, used to compute radiation heat transfer to the fibers (*Equation 2–13* (p. 6), *Equation 2–14* (p. 6), *Equation 2–24* (p. 11) and *Equation 2–25* (p. 11)) when the P-1 or discrete ordinates radiation model is active. Note that you must enable radiation to fiber, using the **Fiber Radiation Interaction** option in the **Fiber Model** dialog box.

Flory Huggins

> (**Fiber Model** dialog box) can be enabled to apply *Equation 2–3* (p. 4) to compute the vapor-liquid equilibrium at the fiber surface. When it is enabled, you have to specify an appropriate value for the dimensionless Flory Huggins parameter, χ. This option is only visible when **Dry Spun Fibers** has been chosen.

Latent Heat

> (**Fiber Model** dialog box) is the latent heat of vaporization of the **Solvent** when evaporated from a dry spun fiber. Note that you have to enter the vaporization or reference temperature, T_{vap}, where the specified value of the latent heat has been measured. This vaporization temperature is used to automatically consider the change of latent heat with temperature. See *Equation 2–16* (p. 6) and *Equation 2–39* (p. 17) for more information on how this is achieved. These options are only visible when **Dry Spun Fibers** has been chosen.

Solidification Temperature

> (**Fiber Model** dialog box) is the temperature below which the fiber polymer of a **Melt Spun Fiber** will solidify. It will be used when computing the viscosity of **Melt Spun Fibers**. This option is only visible when **Melt Spun Fibers** has been chosen.

Solvent Vapor Pressure

> (**Fiber Model** dialog box) is the vapor pressure of the solvent evaporating from the fiber surface in dry spinning. You have to enter coefficients for an Antoine-type equation (*Equation 2–38* (p. 17)). Note that the coefficients must be entered in such units that the outcome of the Antoine-type equation is in Pascal. In addition to the coefficients of the Antoine equation, you have to enter the range of validity for the vapor pressure. Below the minimal temperature the vapor pressure at the minimal temperature will be used. Above the maximal temperature, the vapor pressure at the maximal temperature is used.

Thermal Conductivity

> (**Create/Edit Materials** dialog box) is the thermal conductivity, λ_f, of the fiber in units of power per length and temperature. In the case of dry spun fibers, a mass average is computed based on the values

entered for **Polymer** and its **Solvent**. You can use any of the functions available to define temperature dependency, except user-defined functions.

Zero Shear Viscosity
(**Create/Edit Materials** dialog box and/or the **Fiber Model** dialog box, depending on the chosen fiber model) is the fiber viscosity, μ_f, at zero shear rate. It is not the elongational or Trouton viscosity.

For **Melt Spun Fibers**, you have to enter the viscosity of the fiber in solid state in the **Create/Edit Materials** dialog box for the fluid you have selected as the fiber polymer. Typically this value will be very high compared to the liquid fiber viscosity to represent the fibers as solids. For the solid fiber viscosity you can make use of any temperature-dependent function available in the **Create/Edit Materials** dialog box except user-defined functions.

In the **Fiber Model** dialog box, you have to enter the coefficients for the fiber viscosity in liquid state. In the case of **Melt Spun Fibers**, you also have to enter data for the **Solidification Temperature** and the **Blending Interval**. The blending of the viscosities in liquid and solid state will be computed based on *Equation 2–36* (p. 16).

For **Dry Spun Fibers** you only have to enter the coefficients in the **Fiber Model** dialog box. Any value entered in the **Create/Edit Materials** dialog box for viscosities of the fluids used as fiber polymer and fiber solvent will not be considered.

3.6. Defining Fibers

For more information, please see the following sections:
3.6.1. Overview
3.6.2. Fiber Injection Types
3.6.3. Working with Fiber Injections
3.6.4. Defining Fiber Injection Properties
3.6.5. Point Properties Specific to Single Fiber Injections
3.6.6. Point Properties Specific to Line Fiber Injections
3.6.7. Point Properties Specific to Matrix Fiber Injections
3.6.8. Define Fiber Grids

3.6.1. Overview

The primary inputs that you must provide for the continuous fiber model calculations in ANSYS FLUENT are the starting positions, mass flow rate, take up positions, and other parameters for each fiber. These provide the boundary conditions for all dependent variables to be solved in the continuous fiber model. The primary inputs are:

- Start point (x, y, z coordinates) of the fiber.

- Number of fibers in group. Each defined fiber can represent a group of fibers that will be used only to compute the appropriate source terms of a group of fibers.

- Diameter of the fiber nozzle, d_f.

- Mass flow rate per nozzle to compute the velocity of the fiber fluid in the nozzle. The velocity is used as boundary condition for the fiber momentum equation.

- Temperature of the fiber at the nozzle, T_f.

- Solvent mass fraction of the fiber fluid in the nozzle. This value is used as boundary condition for the solvent continuity equation.

- Take-up point (x, y, z coordinates) of the fiber.

- Velocity or force at take-up point to describe the second boundary condition needed for the fiber momentum equation (see *Equation 2–4* (p. 4)).

In addition to these parameters, you have to define parameters for the grid that is distributed between the start position and take-up point of the fibers. On this grid the dependent fiber variables are solved, by discretizing *Equation 2–1* (p. 3), *Equation 2–4* (p. 4), and *Equation 2–11* (p. 5).

You can define any number of different sets of fibers provided that your computer has sufficient memory.

3.6.2. Fiber Injection Types

You will define boundary conditions and grids for a fiber by creating a fiber "injection" and assigning parameters to it. In the continuous fiber model three types of injections are provided:

- single
- line
- matrix (only in 3D)

You should create a single fiber injection when you want to define a single fiber. Create a line injection when the fibers you want to define start from a line and the starting points are located at constant intervals on this line. When the fiber starting points are arranged in the shape of a rectangle you should create a matrix fiber injection.

3.6.3. Working with Fiber Injections

Figure 3.2 The Fiber Injections Dialog Box
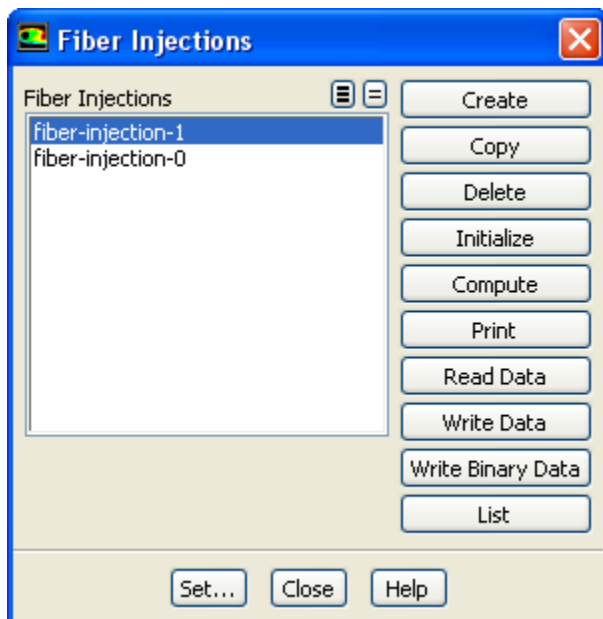
You will use the **Fiber Injections** dialog box (*Figure 3.2* (p. 28)) to create, modify, copy, delete, initialize, compute, print, read, write, and list fiber injections. To access the **Fiber Injections** dialog box, first make sure you enable a fiber model, then go to

Models → **Fiber-Injections** → **Edit...**

3.6.3.1. Creating Fiber Injections

To create a fiber injection, click **Create**. A new fiber injection appears in the **Fiber Injections** list and the **Set Fiber Injection Properties** dialog box opens automatically to enable you to specify the fiber injection properties (as described in *Point Properties Specific to Single Fiber Injections* (p. 34)).

3.6.3.2. Modifying Fiber Injections

To modify an existing fiber injection, select its name in the **Fiber Injections** list and click **Set...**. The **Set Fiber Injection Properties** dialog box opens, and you can modify the properties as needed.

3.6.3.3. Copying Fiber Injections

To copy an existing fiber injection to a new fiber injection, select the existing injection in the **Fiber Injections** list and click **Copy**. The **Set Fiber Injection Properties** dialog box will open with a new fiber injection that has the same properties as the fiber injection you have selected. This is useful if you want to set another injection with similar properties.

3.6.3.4. Deleting Fiber Injections

You can delete a fiber injection by selecting its name in the **Fiber Injections** list and clicking **Delete**.

3.6.3.5. Initializing Fiber Injections

To initialize all solution variables of the fibers defined in a fiber injection, select its name in the **Fiber Injections** list and click **Initialize**. The solution variables will be set to the boundary condition values at the starting points of the fibers.

You can select several fiber injections when you want to initialize several fiber injections at one time.

> **Important**
>
> If you do not select a fiber injection and click **Initialize**, all fiber injections will be initialized.

3.6.3.6. Computing Fiber Injections

To solve the fiber equations of a fiber injection for a number of iterations, select its name in the **Fiber Injections** list and click **Compute**. The solution variables of the fibers defined in this fiber injection will be updated for the number of iterations specified in *Figure 3.13* (p. 43).

You can select several fiber injections when you want to compute several fiber injections at one time.

> **Important**
>
> If you do not select a fiber injection and click **Compute**, all fiber injections will be computed.

3.6.3.7. Print Fiber Injections

To print the fiber solution variables of a fiber injection into a file, select its name in the **Fiber Injections** list and click **Print**. A file name is generated automatically based on the name of the fiber injection and the number of the fiber. The solution variables of each fiber is stored in a separate file. You may use this file for an external post-processing or analysis of fiber data.

You can select several fiber injections when you want to print several fiber injections at one time.

Important

If you do not select a fiber injection and click **Print**, all fiber injections will be printed.

3.6.3.8. Read Data of Fiber Injections

To read the data (properties and solution variables) of a fiber injection previously stored in a file, click **Read Data**. A file selection dialog box is opened where you can select the name of the file in a list or you can enter the file name directly. The names of all fiber injections included in the file are compared with the fiber injections already defined in the model. If the fiber injection exists already in your model, you are asked to overwrite it.

3.6.3.9. Write Data of Fiber Injections

While the settings of the continuous fiber model for numerics and models are stored in the ANSYS FLUENT case file, the data of the fibers and defined injections have to be stored in a separate file. To write the data of a fiber injection to a file, click **Write Data**. A file selection dialog box is opened where you can select the name of an existing file to overwrite it or you can enter the name of a new file.

You can select several fiber injections when you want to store several fiber injections in one file.

Important

If you do not select a fiber injection and click **Write Data**, all fiber injections will be stored in the specified file.

3.6.3.10. Write Binary Data of Fiber Injections

To write the data of a fiber injection in binary format to a file, click **Write Binary Data**. A file selector dialog box is opened where you can select the name of an existing file to overwrite it or you can enter the name of a new file.

You can select several fiber injections when you want to store several fiber injections in binary format in one file.

Important

If you do not select a fiber injection and click **Write Binary Data**, all fiber injections will be stored in binary format in the specified file.

3.6.3.11. List Fiber Injections

To list starting positions and boundary conditions of the fibers defined in a fiber injection, click **List**. ANSYS FLUENT displays a list in the console window. For each fiber you have defined, the list contains the following (in SI units):

- File number in the injection in the column headed `(NO)`.
- x, y, and z position of the starting point in the columns headed `(X)`, `(Y)`, and `(Z)`.

- Fiber velocity in the column headed (U).

- Temperature in the column headed (T).

- Solvent mass fraction in the column headed (SOLVENT).

- Diameter in the column headed (DIAM).

- Mass flow rate in the column headed (MFLOW).

- The number of fibers represented by this fiber group in the column headed (FIBERS).

- The number of fiber grid cells defined for this fiber in the column headed (CELLS).

- A notification whether the starting point of the fiber is located inside or outside the domain in the column headed (IN DOMAIN?).

The boundary conditions at the take up point are also listed. This list consists of the following (in SI units):

- x, y, and z position of the take-up point in the columns headed (X), (Y), and (Z).

- Boundary condition type and its specified value (VELOCITY for given take-up velocity, FORCE for given force in the fiber) in the column headed (BOUNDARY CONDITION).

You can select several fiber injections when you want to list several fiber injections.

Important

If you do not select a fiber injection and click **List**, all fiber injections will be listed.

3.6.4. Defining Fiber Injection Properties

Once you have created an injection (using the **Fiber Injections** dialog box, as described in *Creating Fiber Injections* (p. 29)), you will use the **Set Fiber Injection Properties** dialog box (*Figure 3.3* (p. 32)) to define the fiber injection properties. (Remember that this dialog box opens when you create a new fiber injection, or when you select an existing fiber injection and click **Set...** in the **Fiber Injections** dialog box.)

Figure 3.3 The Set Fiber Injection Properties Dialog Box

The procedure for defining a fiber injection is as follows:

1. If you want to change the name of the fiber injection from its default name, enter a new one in the **Fiber Injection Name** field. This is recommended if you are defining a large number of injections so you can easily distinguish them.

2. Choose the type of fiber injection in the **Fiber Injection Type** drop-down list. The three choices (**single**, **line**, and **matrix**) are described in *Fiber Injection Types* (p. 28).

3. Click the **Injection Point Properties** tab (the default), and specify the point coordinates according to the fiber injection type, as described in *Point Properties Specific to Single Fiber Injections* (p. 34)–*Point Properties Specific to Matrix Fiber Injections* (p. 35).

4. If each of the defined fibers is referring to a group of fibers, enter the number of fibers in **Number of Fibers in Group**. If your nozzle plate has 400 holes and you can simulate them as a line fiber injection with 5 groups, you have to enter a value of 80. This means that only 5 fibers are solved numerically, but each of these fibers stands for 80 fibers to be used to compute the source terms for the surrounding fluid. This enables you to reduce computing efforts while achieving a proper coupling with the surrounding fluid.

Important

Note that the **Number of Fibers in Group** is applied to all fibers, defined in your injection. If the number of fiber groups in your line or matrix injection is not the same for all fibers in this injection, you should split this injection into several fiber injections.

5. Specify the diameter of the nozzle in the **Diameter** field.

6. Enter the mass flow rate for a single nozzle in the **Flow rate per Nozzle** field. This will be used to compute the starting velocity of the fiber fluid.

Important

Note that the value specified refers to one single nozzle and not to the mass flow rate of all fibers defined in this fiber injection.

7. Specify the temperature of the fiber fluid leaving the nozzle in the **Temperature** field.

8. If you are modeling dry spun fibers you also have to enter the solvent's mass fraction at the nozzle in the **Solvent Mass Fraction** field.

Figure 3.4 The Set Fiber Injection Properties Dialog Box With Take-Up Point Properties
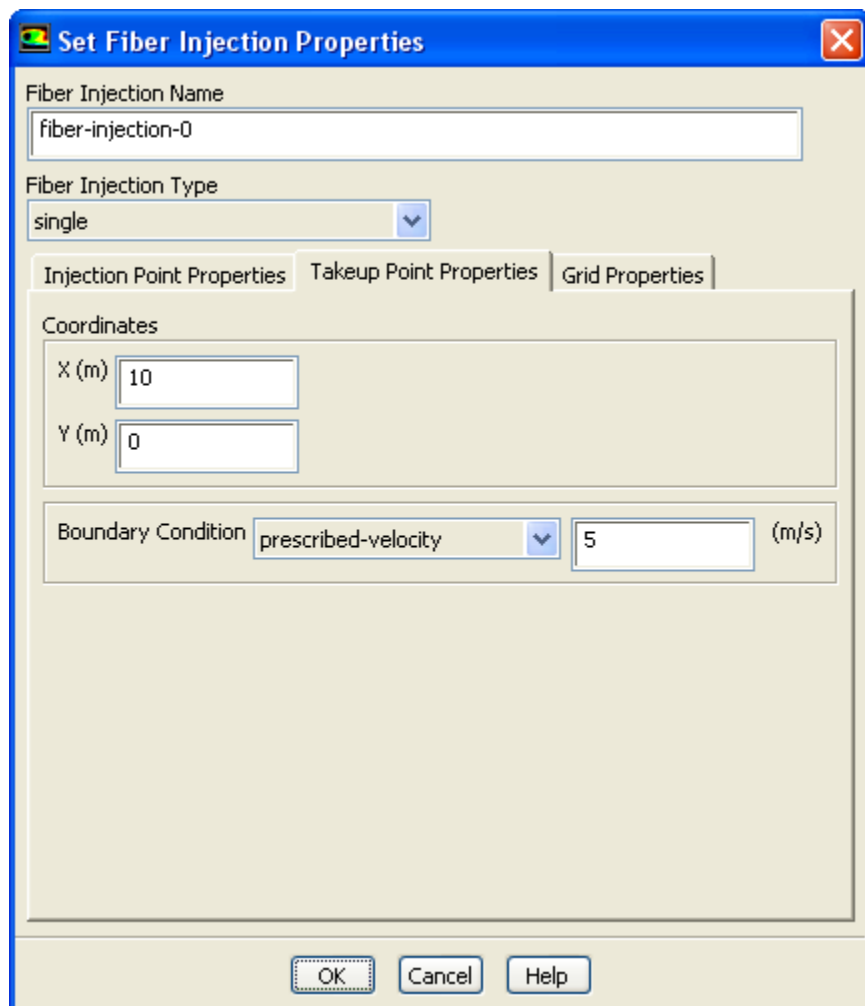
9. Click the **Takeup Point Properties** tab and enter the coordinates of the take-up point (see *Figure 3.4* (p. 33)).

> **Important**
>
> Note that all fibers defined in the fiber injection are collected at the same point. If the fibers of your line or matrix injection vary in this property, you have to define them using several fiber injections.

10. Select the appropriate boundary condition from the **Boundary Condition** drop-down list and specify the value for this boundary condition. Choose **prescribed-velocity** if you know the drawing or take-up velocity. Choose **tensile-force** if you want to prescribe a given tensile force in the fiber at the take-up point.

11. Click the **Grid Properties** tab and enter all data needed to generate the fiber grid as described in *Define Fiber Grids* (p. 35).
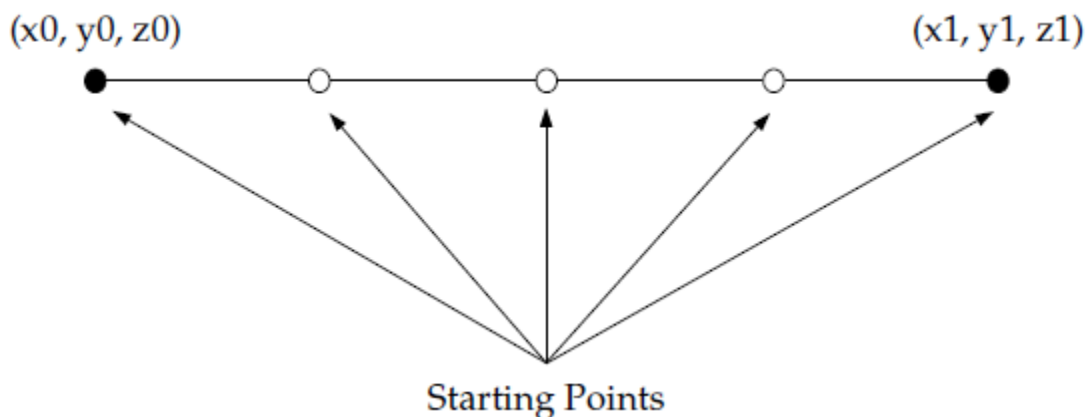
3.6.5. Point Properties Specific to Single Fiber Injections

For a single fiber injection, you have to specify the coordinates of the starting point of the fiber. Click the **Injection Point Properties** tab and set the x, y, and z coordinates in the **x0**, **y0**, and **z0** fields of the **Points** box. (**z0** will appear only for 3D problems.)

3.6.6. Point Properties Specific to Line Fiber Injections

In a line fiber injection the starting points of the fibers are arranged on a line at a constant distance (see *Figure 3.5* (p. 34)). You have to specify the coordinates of the starting point and the end point of this line. Click the **Injection Point Properties** tab in the **Set Fiber Injection Properties** dialog box (*Figure 3.3* (p. 32)). In the **Points** region, set the x, y, and z coordinates in the **x0**, **y0**, and **z0** fields for the starting point of the line and the x, y, and z coordinates in the **x1**, **y1**, and **z1** fields for the end point of the line. (**z0** and **z1** will appear only for 3D problems.)

Figure 3.5 Line Injections

In addition to the coordinates, you have to set the number of fibers defined in the line injection by entering the appropriate value in the **Point Density Edge1** field. Please see *Figure 3.5* (p. 34) for an example of a line fiber injection with a **Point Density Edge1** of 5.

3.6.7. Point Properties Specific to Matrix Fiber Injections

In a matrix fiber injection the starting points of the fibers are arranged in several rows having the shape of a rectangle or a parallelogram. Each row has the same distance to the previous and is divided into equal sections (see *Figure 3.6* (p. 35)).

You have to specify the coordinates of the starting point and the end of point of the first row and the coordinates where the last row should start. Click the **Injection Point Properties** tab in the **Set Fiber Injection Properties** dialog box (*Figure 3.3* (p. 32)). In the **Points** region, set the x, y, and z coordinates in the **x0**, **y0**, and **z0** fields for the starting point and the x, y, and z coordinates in the **x1**, **y1**, and **z1** fields for the end point of the first row of fibers. The x, y, and z coordinates of the starting point of the last row have to be entered in the **x2**, **y2**, and **z2** fields.

At each row, the number of fibers specified in the **Point Density Edge1** field are injected. The number of rows to be injected is specified in **Edge2**.

You can double check the number of fibers computed in this fiber injection if you inspect the **Numbers of Fibers Computed** field.

Important

Note that this fiber injection type is only available for 3D problems.

**Figure 3.6 Matrix Injections**
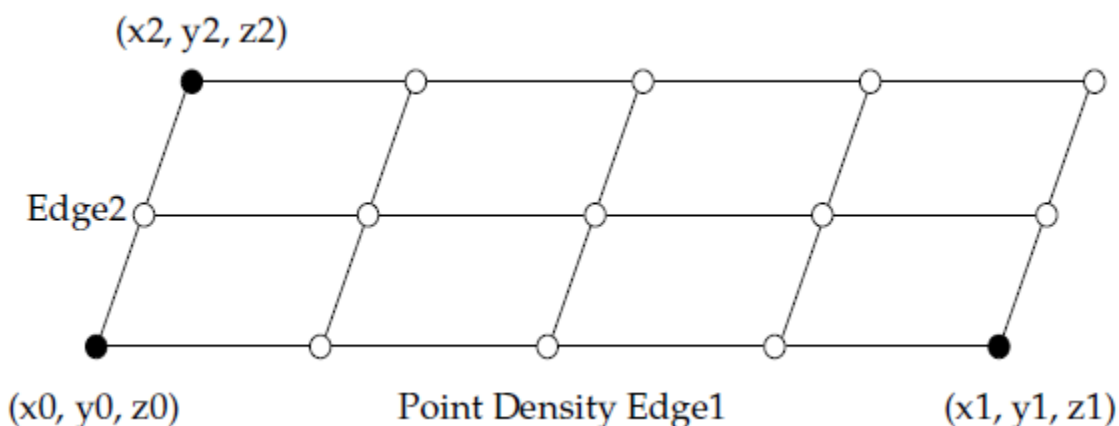
3.6.8. Define Fiber Grids

Each fiber consists of a straight line between the injection point and the take-up point. It is divided into a number of finite volume cells. Every fiber defined in a fiber injection has its own grid, which you can specify if you click the **Grid Properties** tab in the **Set Fiber Injection Properties** dialog box.

3.6.8.1. Equidistant Fiber Grids

To define an equidistant grid for the fibers select **equidistant** from the **Grid Type** drop-down list. Specify the **Number of Cells** into which every fiber of the fiber injection will be divided.

Figure 3.7 Equidistant Fiber Grid
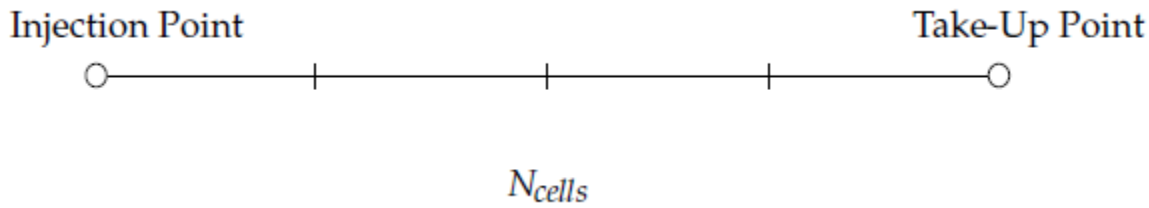
3.6.8.2. One-Sided Fiber Grids

A one-sided grid is graded near the injection point of the fiber. To define a one-sided grid for the fibers select **one-sided** from the **Grid Type** drop-down list. Specify the **Number of Cells** into which every fiber of the fiber injection will be divided. In addition, you have to specify the ratio, R, between each subsequent fiber cell in the **Grid Growth Factor at Injection Point** field. Values larger than 1 refine the grid, while values smaller than 1 coarsen the grid.

Figure 3.8 One-Sided Fiber Grid
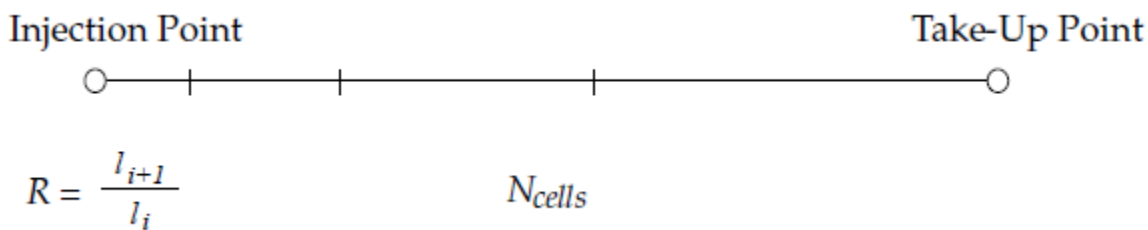
3.6.8.3. Two-Sided Fiber Grids

A two-sided grid is graded near the injection point as well as at the take-up point of the fiber. To define a two-sided grid for the fibers, select **two-sided** from the **Grid Type** drop-down list. Specify the **Number of Cells** into which every fiber of the fiber injection will be divided. You also have to specify the ratio, R, between each subsequent fiber cell at the injection point in the **Grid Growth Factor at Injection Point** field and the ratio, R, near the take-up point in the **Grid Growth Factor at Takeup Point** field. Values larger than 1 refine the grid, while values smaller than 1 will coarsen it.

Figure 3.9 Two-Sided Fiber Grid
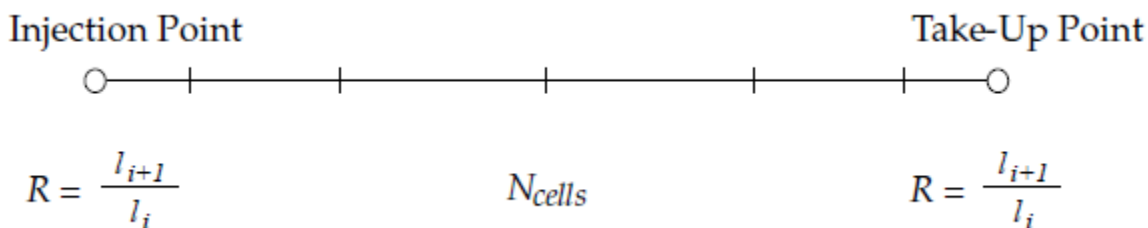
3.6.8.4. Three-Sided Fiber Grids

A three-sided grid consists of three sides where the fiber grid can be graded. The first side is near the injection point. The other two sides are around a refinement point within the fiber grid. Both sides at the refinement point are graded at the same ratio between the fiber grid cell lengths. To define a three-sided grid for the fibers, select **three-sided** from the **Grid Type** drop-down list (see *Figure 3.11* (p. 38)). Specify the **Number of Cells until Grid Refinement Point**, the **Grid Growth Factor at Injection Point**, and the **Grid Growth Factor at Local Grid Refinement Point**. You also have to specify the location of the grid refinement point in the **Location of Local Grid Refinement Point** field in a dimensionless way. The value you have to enter is relative to the fiber length and may be between 0 and 1. Values larger than 1 refine the grid, while values smaller than 1 coarsen the grid.

Click the **Compute** button to estimate the **Number of Cells behind Grid Refinement Point**.
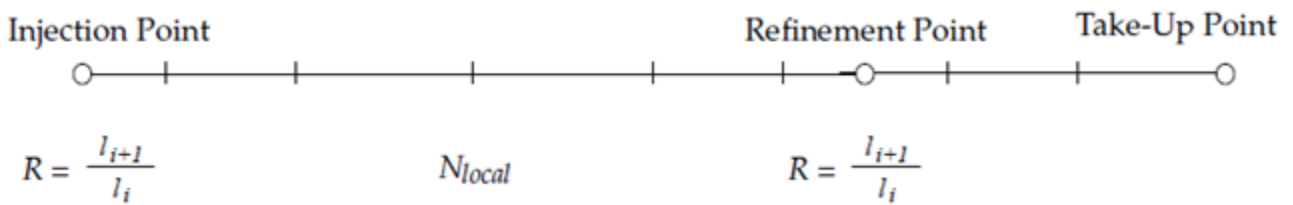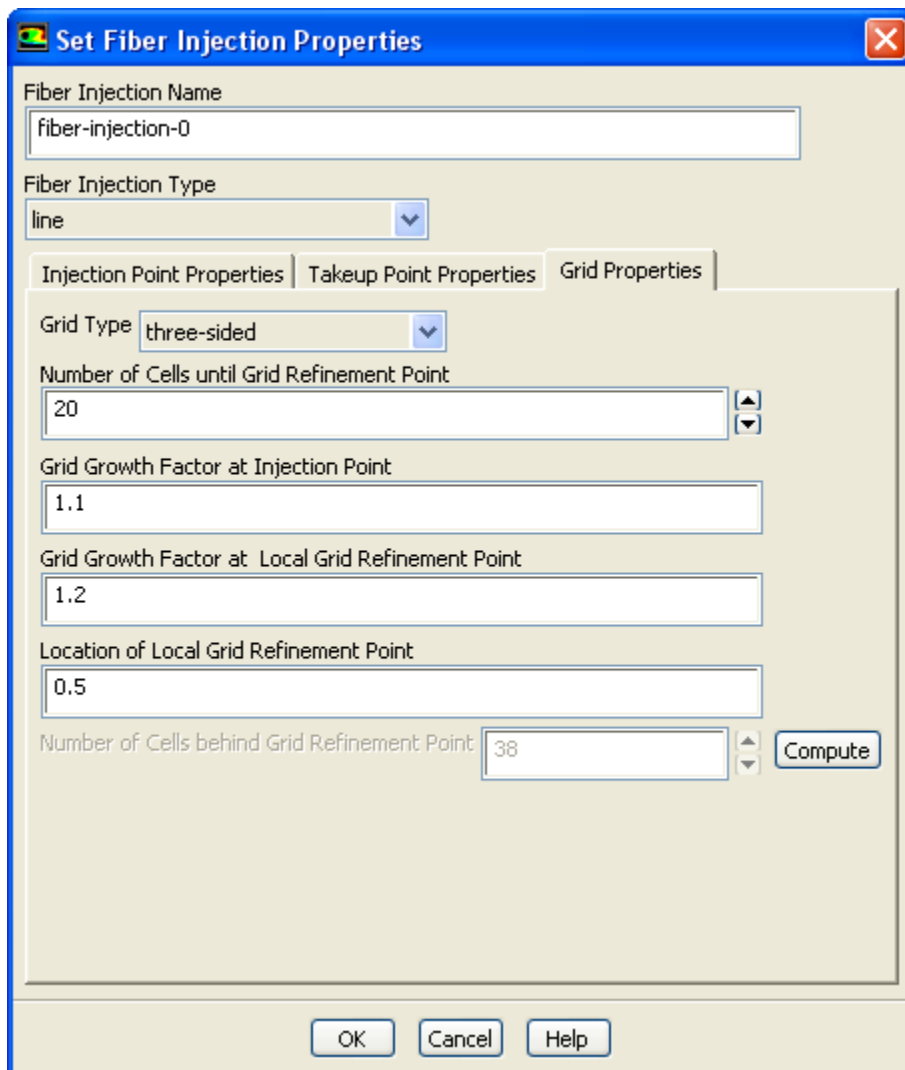
Figure 3.10 Three-Sided Fiber Grid

Figure 3.11 Defining a Three-Sided Fiber Grid Using the Set Fiber Injection Properties Dialog Box

3.7. User-Defined Functions (UDFs) for the Continuous Fiber Model

The continuous fiber model enables you to apply custom correlations for drag, heat transfer, and mass transfer coefficients to the fibers by means of user-defined functions (UDFs). You are provided with a C template file named `fiber_fluent_interface.c` that contains source code for the drag, heat transfer, and mass transfer coefficient UDFs. You will need to modify this UDF source file to suit your application, compile it, and hook the resulting UDF object(s) to the fiber model. This process is described below.

For more information, please see the following sections:
3.7.1. UDF Setup
3.7.2. Customizing the fiber_fluent_interface.c File for Your Fiber Model Application
3.7.3. Compile Fiber Model UDFs
3.7.4. Hook UDFs to the Continuous Fiber Model

3.7.1. UDF Setup

Before you can edit the UDF template and create your custom UDF for drag, heat transfer or mass transfer coefficient, you must copy the `fiber` directory to your working directory. The `fiber` directory contains all of the libraries and support files that are required to compile UDFs for the fiber model and build shared libraries for the architecture that you specify.

3.7.1.1. Linux Systems

Make a local copy of the `fiber` directory by copying it from the path below to your working directory.

`path/ansys_inc/v140/fluent/fluent14.0.0/addons/fiber/`

where `path` is the directory in which you have installed ANSYS FLUENT.

3.7.1.2. Windows Systems

1. Open a Visual Studio `.NET` DOS prompt.

2. Make a local copy of the `fiber` folder (not a symbolic link) by copying it from the folder below to your working folder.

   `path\ANSYS Inc\v140\fluent\fluent14.0.0\addons\fiber\`

   where `path` is the folder in which you have installed ANSYS FLUENT (by default, the path is `C:\Program Files`).

3.7.2. Customizing the fiber_fluent_interface.c File for Your Fiber Model Application

Now that you have copied the `fiber` directory to your working directory, you can edit the UDF template file and customize it to fit your model needs.

1. In your working directory, change directories to `fiber/src`. The `/src` directory contains the UDF template source file `fluent_interface_fiber.c`.

2. In the `/src` directory, use any text editor and edit `fiber_fluent_interface.c`.

3. Scroll down to the bottom of the `fiber_fluent_interface.c` file to the section that contains three concatenated functions for friction factor (drag coefficient), heat transfer coefficient, and mass transfer coefficient, respectively. These are the UDFs that you can modify and customize.

4. Edit the function(s) you desire for your particular application. Save `fiber_fluent_interface.c` and overwrite the existing file.

Important

Do not save the file with another name because it will not be recognized by the system.

> **Important**
>
> The function names of the templates `user_friction_factor`,
> `user_heat_transfer_coefficient`, and `user_mass_transfer_coeffi-`
> `cient` *must not* be altered because they are called by other routines in the continuous
> fiber model.

3.7.2.1. Example Heat Transfer Coefficient UDF

Below is an example of a heat transfer coefficient UDF that is defined in `fiber_fluent_interface.c`.
The function is taken from Kase and Matsuo [3] (p. 49) and is implemented as the `kase-matsuo-1`
option in the fiber model. The function name `user_heat_transfer_coefficient` *must not*, under
any circumstances, be altered because it is called by other functions in the continuous fiber model.

There are two arguments to the `user_heat_transfer_coefficient` UDF: `Fiber` and `Loc-`
`al_Fiber_Data_Type`. `Fiber  *f` is a pointer to the fiber structure that contains information about
the fiber and `Local_Fiber_Data_Type  *fd` accesses temporary variables that are needed during
the calculation of the fiber.

In the sample UDF below, a loop is performed over all fiber grid cells using the macro `FIB_N(f)` which
represents the number of grid cells for the fiber `f`. The Reynolds number is computed based on the
relative velocity, $u_f - u_{par}$ (`ABS(FIB_C_U(f,i)-fd-up[i])`), the fiber diameter; `FIB_C_D(f,i)`,
the density of the surrounding fluid; `fd-rho[i]`, and the viscosity of the surrounding fluid; `fd-`
`vis[i]`. The heat transfer coefficient α is computed from the Nusselt number using the thermal con-
ductivity of the surrounding fluid, λ, (`fd-k[i]`) and is stored in `fd-alpha[i]`.

```
void
user_heat_transfer_coefficient(Fiber *f, Local_Fiber_Data_Type *fd)
{
 int i;
 real Red, Nud;

 /* model from Kase/Matsuo (1967) */
 for (i=0; iFIB_N(f); i++)
 {
   /* compute Reynolds number based on relative velocity */
   Red = ABS(FIB_C_U(f,i)-fd-up[i])*FIB_C_D(f,i)*fd-rho[i]/fd-vis[i];
   Nud = 0.42*pow(Red, 0.334);
   /* store heat transfer coefficient for latter use */
   fd-alpha[i] = Nud*fd-k[i]/FIB_C_D(f,i);
 }
}
```

All variables and macros that are used in `user_heat_transfer_coefficient` are defined in
header files provided with the continuous fiber model. For example, you will find the type definition
`Fiber` and the macros that are used to access variables of a single fiber in the header file `fiber.h`.
Temporary variables used in the type definition of `Local_Fiber_Data_Type` can be found in the
header file `fib-mem.h`.

> **Important**
>
> Note that you must not modify the header files provided with the continuous fiber model.
> Otherwise the compiled library will not be compatible with ANSYS FLUENT and will show
> runtime errors.

3.7.3. Compile Fiber Model UDFs

Once you have customized the UDF template `fiber_fluent_interface.c` for your fiber model application, you are now ready to compile the source file using the text interface. Follow the procedure below for Linux Systems and Windows Systems, respectively.

3.7.3.1. Linux Systems

In the `/src` directory that is under `/fiber` in your working directory, execute the `Makefile` command that will compile your fiber mode UDF and build a shared library for the architecture you are running. To do this, type the following command at the prompt:

```
make -f Makefile-client FLUENT_ARCH=your_arch
```

where `your_arch` is replaced by the architecture of the machine you are running (for example, lnx86).

For example, if your computer architecture is `lnx86` type the following command in a terminal session:

```
make -f Makefile-client FLUENT_ARCH=lnx86
```

To identify the architecture of the machine you are running on, scroll up the ANSYS FLUENT console window to the message that begins with "Starting".

When you execute the `makefile` process, the source code (`fluent_fiber_interface.c`) will be compiled into object code and a shared library will be built for the computer architecture and version of ANSYS FLUENT you are running. Messages about the compile/build process will be displayed on the console window. You can view the compilation history in the log file that is saved in your working directory. Below is an example of console messages for a lnx86 architecture running a 2D version of ANSYS FLUENT.

```
Working...
for d in lnx86[23]*; do \
    ( \
        cd $d; \
        for f in ../../src*.[ch] ../../src/makefile; do \
            if [ ! -f 'basename $f' ]; then \
                echo "# linking to " $f "in" $d; \
                ln -s $f .; \
            fi; \
        done; \
        echo ""; \
        echo "# building library in" $d; \
        make -kmakelog 2&1; \
        cat makelog; \
    ) \
done
# linking to ...  myudf.c in lnx86/2d

# building library in lnx86/2d
make[1]: Entering directory ..../udf_names.c
# Generating udf_names
make[2]: Entering directory ..../fluent_fiber_interface.c
make libudf.so ...
# Compiling udf_names.o ...
# Compiling profile.o ...
# Linking libudf.so ...
make[2]: Leaving directory ..../udf_names.c
make[1]: Leaving directory ..../fluent_fiber_interface.c

You can also see the 'log'-file in
the working directory for compilation history

Done.
```

3.7.3.2. NT/Windows Systems

In the `/src` directory that is under `/fiber` in your working directory, execute the `Makefile` command that will compile your fiber mode UDF and build a shared library for it for the architecture you are running. To do this, type the following command at the prompt:

```
nmake /f makefile_master-client.nt
```

To identify the architecture of the machine you are running on, scroll up the ANSYS FLUENT console window to the message that begins with "Starting".

When you execute the `makefile` process, the source code (`fluent_fiber_interface.c`) will be compiled into object code and a shared library will be built for the computer architecture and version of ANSYS FLUENT you are running. Messages about the compile/build process will be displayed on the console window. You can view the compilation history in the log file that is saved in your working directory.

3.7.4. Hook UDFs to the Continuous Fiber Model

Once you have successfully compiled your continuous fiber model UDF(s), the `user-defined` option will appear in drop-down lists for parameters under **Exchange** in the **Fiber Model** dialog box (*Figure 3.12* (p. 42)). To hook your UDF to a particular fiber model parameter, simply choose `user-defined` from the drop-down list for **Drag Coefficient**, **Heat Transfer Coefficient**, or **Mass Transfer Coefficient** and click **OK**.

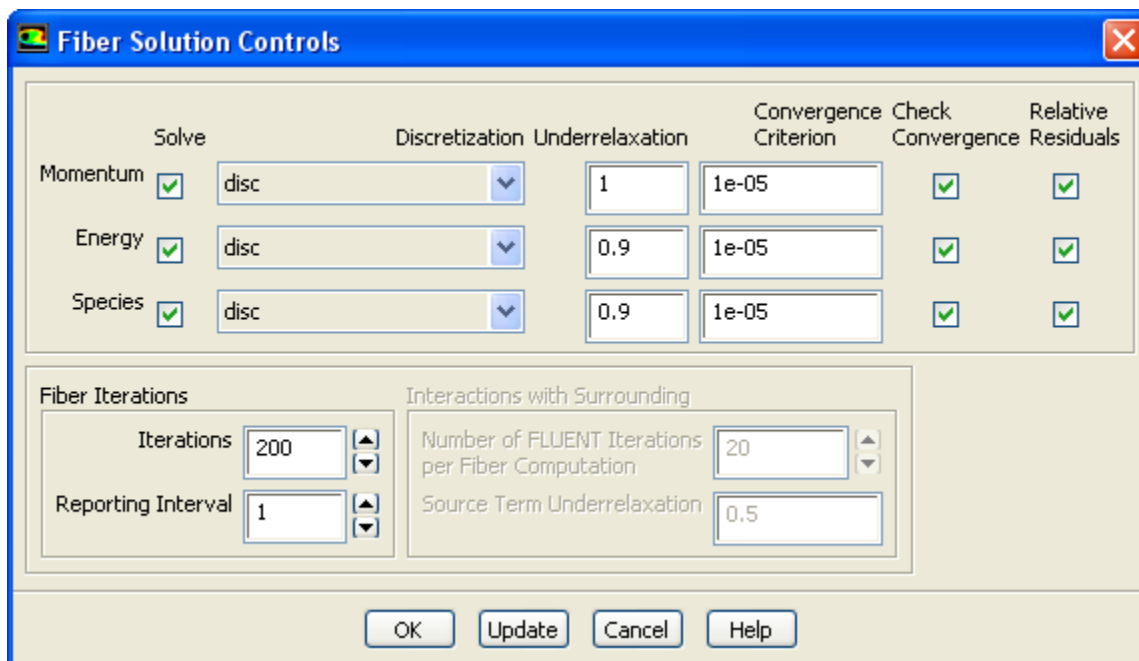**Figure 3.12 The Fiber Model Dialog Box**
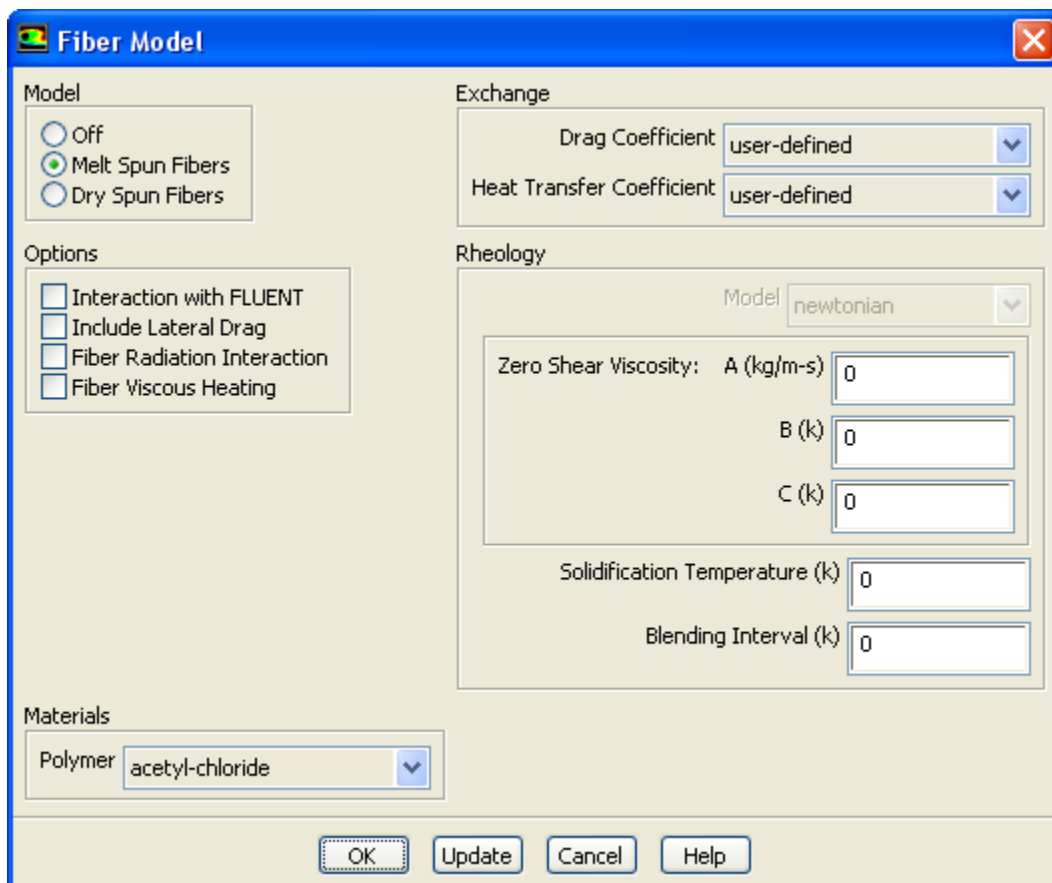
3.8. Fiber Model Solution Controls

To access the **Fiber Solution Controls** dialog box, go to

⬥**Models** → ≣**Fiber-Controls** → **Edit...**

Figure 3.13 Fiber Solution Controls Dialog Box

The **Fiber Solution Controls** dialog box enables you to set common solution parameters for the fiber equations and their coupling with the surrounding fluid.

Solve
    is used to enable and disable the solution of the fiber equations for **Momentum** (*Equation 2–4* (p. 4)), **Energy** (*Equation 2–11* (p. 5)), and **Species** (*Equation 2–1* (p. 3)). When switching off the solution of one of these equations, it is not computed for all fibers defined in your model.

Discretization
    provides a drop-down list where you can assign to each of the fiber equations one of three different discretization schemes explained in *Discretization of the Fiber Equations* (p. 6).

Underrelaxation
    contains all under-relaxation factors for all fiber equations that are being solved in the continuous fiber model. See *Under-Relaxation* (p. 7) for additional background information and see *Solution Strategies* (p. 17) for how to make use of under-relaxation factors in your solution strategy.

Convergence Criterion
    is used to stop the fiber iterations when the residuals of all fiber equations are below the prescribed criteria. You can define a separate convergence criterion for every fiber equation.

Check Convergence
    must be turned on if you want to compare the residuals of the fiber equations with the **Convergence Criterion**. If you turn this option off, the given number of fiber iterations will be computed.

Relative Residuals

are used to compute the change of the residual of two subsequent iterations relative to the residual of the last iteration by applying *Equation 2–20* (p. 8). The result of this is compared to the **Convergence Criterion** to check whether convergence has been achieved.

Iterations

defines the number of **Fiber Iterations** performed every time the fiber equations are updated.

Reporting Interval

sets the number of fiber iterations that will pass before the residuals will be printed.

You can reduce the output to the last fiber iteration by specifying **Reporting Interval** as 0. This is recommended when performing a solution that is coupled with the surrounding flow.

Number of FLUENT Iterations per Fiber Computation

sets the number of FLUENT iterations before the fiber equations are updated in a solution that is coupled with the surrounding flow.

Source Term Underrelaxation

factor is used to under-relax the fiber source term exchange to the surrounding fluid. In a converged solution this value does not influence your predictions.

For additional information on how to set and choose values for the options in the **Fiber Solutions Control** dialog box, see *Solution Strategies* (p. 17).

3.9. Postprocessing for the Continuous Fibers

After you have completed your inputs and performed any coupled calculations, you can display the location of the fibers and source terms of the fibers, and you can write fiber data to files for further analysis of fiber variables.

The following data can be displayed using graphical and alphanumeric reporting facilities:

- Graphical display of fiber locations
- Exchange terms with surrounding fluid
- Fiber solution variables.

For more information, please see the following sections:
3.9.1. Display of Fiber Locations
3.9.2. Exchange Terms of Fibers
3.9.3. Analyzing Fiber Variables

3.9.1. Display of Fiber Locations

When you have defined fiber injections, as described in *Defining Fibers* (p. 27), you can display the location of the fibers using ANSYS FLUENT's **Contour** plot facility (*Figure 3.14* (p. 45)).

⬙**Graphics and Animations** → ☰ **Contours** → **Set Up...**

Figure 3.14 Displaying Fiber Locations Using the Contours Dialog Box
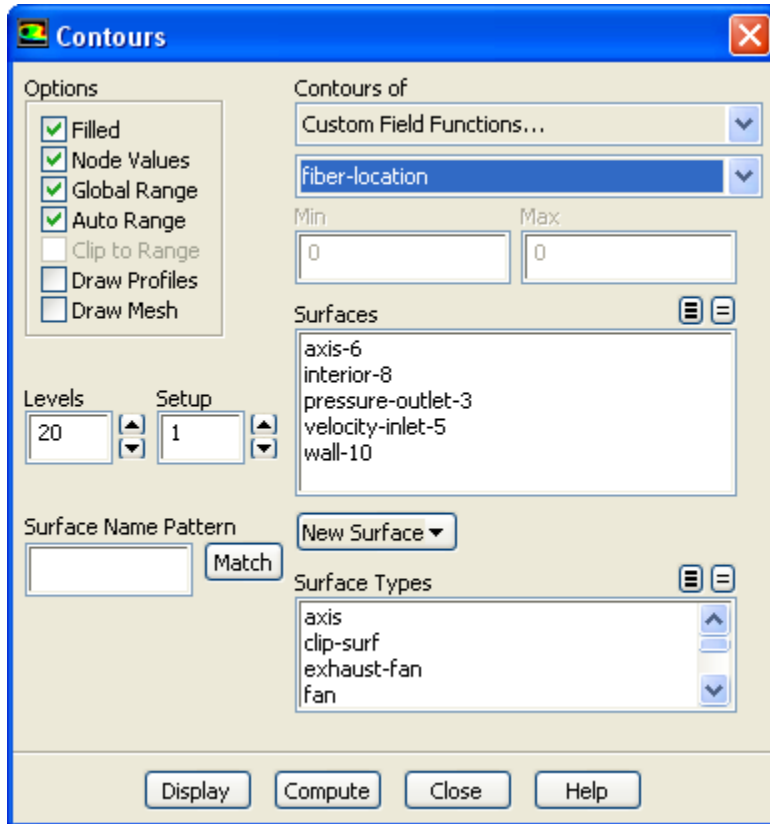
In the **Contours of** drop-down list, select **Custom Field Functions...**, then select **fiber-location**. The values in this field are between zero and the number of fiber cells in an ANSYS FLUENT grid cell.

You may generate iso-surfaces of constant values of **fiber-location** to display the fibers in 3D problems.

3.9.2. Exchange Terms of Fibers

The continuous fiber model computes and stores the exchange of momentum, heat, mass, and radiation in each control volume in your ANSYS FLUENT model. You can display these variables graphically by drawing contours, profiles, etc. They all are contained in the **Custom Field Functions...** category of the variable selection drop-down list that appears in postprocessing dialog boxes:

fiber-mass-source
    defined by *Equation 2–22* (p. 10).

fiber-x-momentum-source
    is the x-component of the momentum exchange, defined by *Equation 2–21* (p. 9).

fiber-y-momentum-source
    is the y-component of the momentum exchange, defined by *Equation 2–21* (p. 9).

fiber-z-momentum-source
    is the z-component of the momentum exchange, defined by *Equation 2–21* (p. 9).

fiber-energy-source
    defined by *Equation 2–23* (p. 10).

fiber-dom-absorption
    defined by *Equation 2–24* (p. 11).

fiber-dom-emission
>   defined by *Equation 2–25* (p. 11).

Note that these exchange terms are updated and displayed only when coupled computations are performed.

3.9.3. Analyzing Fiber Variables

You can use ANSYS FLUENT's **Plot** facilities to analyze fiber solution variables such as fiber velocity, temperature, diameter, etc.

For this you have to generate an xy-file for every variable you want to investigate using the following procedure from the text command interface (assuming that you have already defined a fiber injection):

1. Store an xy-file with the variable of interest.

    ```
    continuous-fiber → print-xy
    ```

2. Enter the fiber injection of interest when asked for

    ```
    Fiber Injection name>
    ```

3. Specify a variable for `x-column`. This will be used as abscissa in the file plot.

4. Specify another variable for `y-column`, which will be used as ordinate in the file plot.

5. Specify a file name when asked for `XY plot file name`.

6. Load the file with ANSYS FLUENT's xy plot facilities, described in the ANSYS FLUENT manual.

The following variables can be entered as `x-column` and `y-column`:

*   axial drag
*   conductivity
*   curvature
*   diameter
*   enthalpy
*   evaporated mass flux
*   heat transfer coefficient
*   lateral drag
*   length
*   mass flow
*   mass fraction
*   mass transfer coefficient
*   Nusselt number
*   Sherwood number
*   Reynolds number
*   specific heat
*   surface mass fraction
*   temperature

- tensile force

- user variable

- velocity

- velocity gradient

- viscosity

If a fiber injection consists of several fibers, the data of all fibers in this fiber injection will be stored in the xy-file.

In addition to this, you can store a fiber history file for a fiber injection as described in *Print Fiber Injections* (p. 29). This can be used to analyze a single fiber using ANSYS FLUENT's xy-plot facilities or by external postprocessing programs.

Bibliography

[1] P. J. Flory. *Thermodynamics of High Polymer Solutions*. *Journal of Chemical Physics*. 10:51–61. 1942.

[2] B. Gampert. *Grenzschichttheoretische Probleme des aerodynamischen Schmelzspinnprozesses*. *PhD Thesis from the Technical University of Berlin*. Berlin, Germany 1973.

[3] S. Kase and T. Matsuo. *Studies on Melt Spinning. II. Steady-State and Transient Solution of Fundamental Equations Compared with Experimental Results*. *J. Applied Polymer Science*. 11:251–287. 1967.

[4] Y. Ohzawa, Y. Nagano, and T. Matsuo. *Fundamental Equations of Dry Spinning with an Example Calculation*. *In Procs of 5th Intern. Congress on Rheology*. pp 393–408. Kyoto. 1968.

[5] S. V. Patankar. *Numerical Heat Transfer and Fluid Flow*. McGraw-Hill. Washington, New York, London 1980.

[6] C. H. Rexroth, H. J. Bauer, and S. Wittig. *DISC - an efficient method for the discretization of convection on unstructured grids*. Aerospace Science and Technology. 6. 1997.

[7] H. Schlichting and K. Gersten. *Grenzschicht-Theorie*. Springer. Berlin, Heidelber, New York, London 1997.

Index