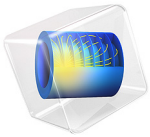


Process Control Using a PID Controller

Introduction

In the chemical process industry it is often important to control a specific process. PID control (proportional-integral-derivative control) is one way to achieve that, but it can be difficult to optimize the parameters in the PID algorithm. This example illustrates how you can implement a PID control algorithm to simulate a process control system and to find the optimal PID parameters.

This application is a generic example but could resemble the environment in a combustion chamber where the concentration at the ignition point is crucial. Two gas streams with different oxygen concentrations are mixed in the combustion chamber. The concentration is measured at the ignition point before complete mixing of the streams is reached. The control algorithm alters the inlet velocity of the gas with the lower oxygen content to achieve the desired total concentration at the ignition point. Since an increased flow of that gas will decrease the concentration, the PID coefficients will have a negative sign for this PID controller.

Model Definition

The model geometry appears in Figure 1. At the upper inlet, a gas stream with high oxygen content enters the reactor at a velocity of 10 mm/s, while a gas with a lower oxygen level enters from the left. The oxygen concentration is measured at a measurement

point, and the inlet velocity of the less concentrated stream is altered by the PID control algorithm to achieve the desired concentration at that point.

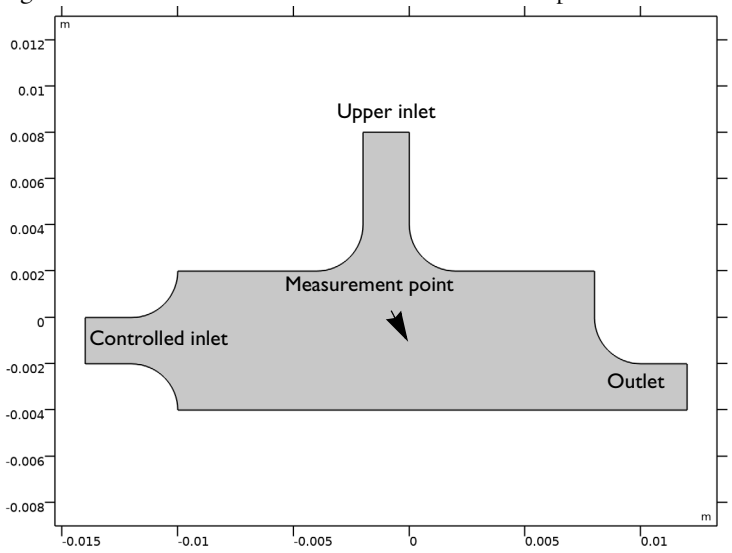


Figure 1: Model geometry.

The model uses the Laminar Flow interface to describe the fluid flow and the Transport of Diluted Species interface for the mass balance. The corresponding equations read (assuming incompressible flow and absence of reactions)

$$\rho\frac{\partial \mathbf{u}}{\partial t} - \nabla \cdot [\eta(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \rho \mathbf{u} \cdot \nabla \mathbf{u} + \nabla p = \mathbf{0}$$

$$\nabla \cdot \mathbf{u} = 0$$

$$\frac{\partial c}{\partial t} + \nabla \cdot (-D\nabla c) = -\mathbf{u} \cdot \nabla c$$

To formulate the boundary conditions for the mass-transport equation, begin by assuming that you know the two inlet concentrations. In addition, assume that the reactant transport at the outlet is mainly driven by convection; that is, neglect diffusion in the main direction

of the convective flow. A no-flux boundary condition describes all walls. The boundary conditions for the mass balance are:

TABLE 1: MASS-BALANCE BOUNDARY CONDITIONS.

BOUNDARY	CONSTRAINT
Upper inlet	$c = c_{\text{in,top}}$
Controlled inlet	$c = c_{\text{in,inlet}}$
Outlet	$\mathbf{n} \cdot (-D\nabla c) = 0$
Walls	$\mathbf{N} \cdot \mathbf{n} = 0$

Here c is the concentration; $c_{\text{in,top}}$ and $c_{\text{in,inlet}}$ are the inlet concentrations (SI unit: mol/m^3) for the upper and controlled inlets, respectively; D is the applied diffusivity (SI unit: m^2/s); and $\mathbf{N}$ is the molar flux (SI unit: mol/(m^2·s)).

The model uses the following boundary conditions for the fluid flow:

TABLE 2: FLUID-FLOW BOUNDARY CONDITIONS.

BOUNDARY	CONSTRAINT
Upper inlet	$\mathbf{u} = (0, -v_{\text{in,top}})$
Controlled inlet	$\mathbf{u} = (u_{\text{in}}, 0)$
Outlet	$p_0 = 0$
Inlet sections	$\mathbf{n} \cdot \mathbf{u} = 0$
Walls	$\mathbf{u} = \mathbf{0}$

Here $\mathbf{u}$ is the velocity vector (SI unit: m/s), $v_{\text{in,top}}$ is the inlet velocity at the top inlet, and u_{in} is the PID-controlled velocity. At the outlet, set the pressure to 0. No Slip boundary conditions describe all walls except the inlet sections where slip conditions apply, allowing for a smooth transition to a laminar velocity profile.

The PID control algorithm used to calculate u_{in} is the following, which is the standard PID control algorithm available in the PID Controller add-in:

$$u_{\text{in}}(t) = k_{\text{P}}(c_{\text{set}} - c(t)) + k_{\text{I}}\int_0^t (c_{\text{set}} - c(\tau))d\tau - k_{\text{D}}\frac{\partial}{\partial t}c(t) \quad (1)$$

with the following parameters:

TABLE 3: PID CONTROLLER PARAMETERS.

PARAMETER	VALUE
c_{set}	0.5 mol/m^3
k_{P}	-0.5 m^4/(mol·s)
k_{I}	-1 m^4/(mol·s^2)
k_{D}	-10^{-3} m^4/mol

As mentioned, the negative values of the coefficients reflect the fact that c_{set} is lower than the concentration at the upper inlet, $c_{\text{in,top}}$, and that the purpose of the gas stream at the controlled inlet thus is to reduce the concentration. In practice, the derivative constant, k_{D}, is set to 0 in most cases as this parameter can be difficult to determine. Moreover, the derivative term may increase the fluctuations in the system because it amplifies noise in the measurement c. The PID Controller add-in includes the option to filter the derivative part, which reduced the influence of high-frequency noise and therefore makes the derivative part more useful.

Results and Discussion

The two plots in Figure 2 show the oxygen concentration and the velocity streamlines in the chamber after 0.05 s and 2 s, respectively. The figures show that the measured concentration depends strongly on the flow field. At startup, when the inlet velocity of the stream entering from the left is very low, the sensor is entirely exposed to the highly concentrated stream, and as the left inlet velocity increases the opposite relation occurs.

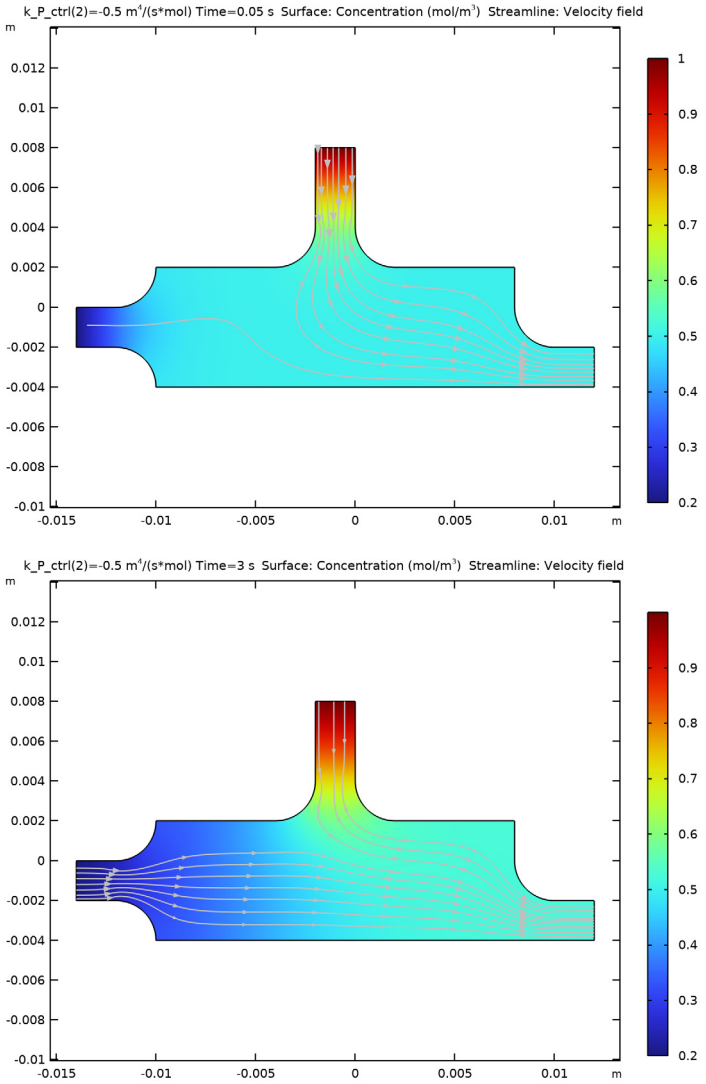


Figure 2: Oxygen concentration and velocity streamlines after 0.1 s (top) and 1.5 s (bottom).

Figure 3 shows the inlet velocity and concentration in the measurement point as a function of time for two different values for the k_P parameter. The green line represents the results for a k_P value of $0.5\ \mathrm{m}^4/(\mathrm{mol}\cdot\mathrm{s})$ while the blue line corresponds to k_P equal to $0.1\ \mathrm{m}^4/$

(mol·s). The results evaluated for the smaller k_P value oscillate more before stabilizing. Thus, it is clear that for this case the higher k_P value yields a more stable process control.

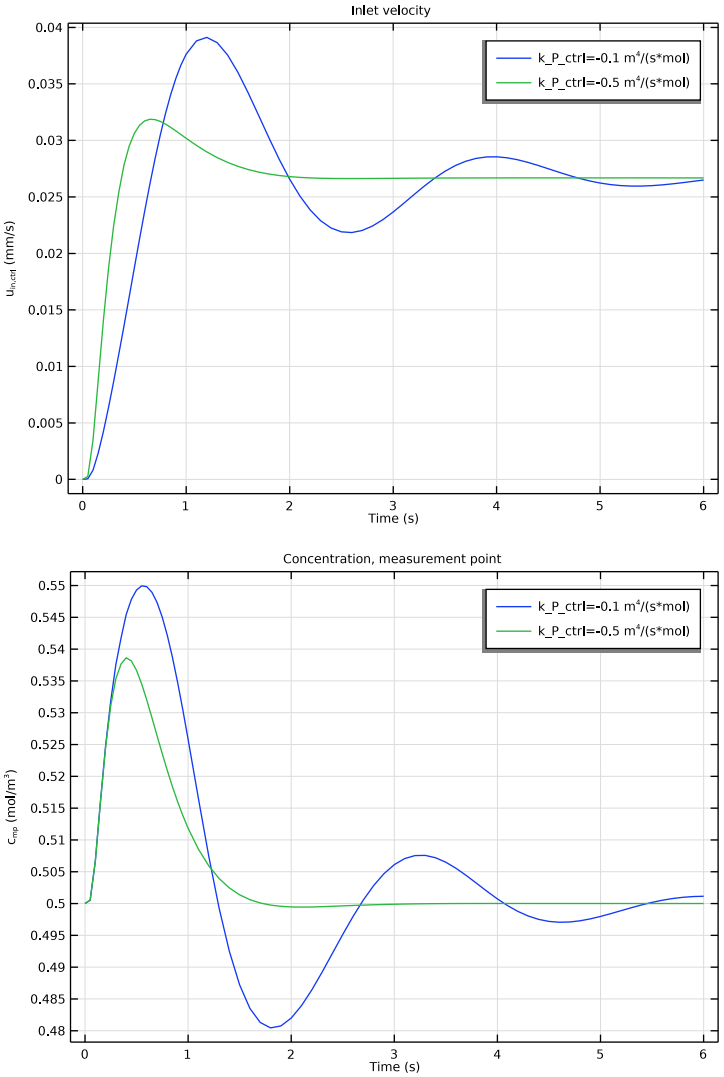


Figure 3: PID-controlled inlet velocity (top) and concentration in the measurement point (bottom) as a function of time for k_P = 0.5 $m^4/(mol{\cdot}s)$ (green) and k_P = 0.1 $m^4/(mol{\cdot}s)$ (blue).

Application Library path: `COMSOL_Multiphysics/Multiphysics/pid_control`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1. In the **Model Wizard** window, click **2D**.
2. In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.
3. Click **Add**.
4. In the **Select Physics** tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **General Studies>Time Dependent**.
8. Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
v_in_top	0.01[m/s]	0.01 m/s	Velocity, upper inlet
c_in_top	1[mol/m^3]	1 mol/m³	Concentration, upper inlet
c_in_inlet	0.2[mol/m^3]	0.2 mol/m³	Concentration, controlled inlet
c00	0.5[mol/m^3]	0.5 mol/m³	Initial concentration, chamber interior
D	1e-4[m^2/s]	1E-4 m²/s	Diffusivity
c_set	0.5[mol/m^3]	0.5 mol/m³	Setpoint concentration
k_P_ctrl	-0.5[m^4/(mol*s)]	-0.5 $m^4/(s \cdot mol)$	Proportional parameter
k_I_ctrl	-1[m^4/(mol*s^2)]	-1 $m^4/(s^2 \cdot mol)$	Integral parameter
k_D_ctrl	-1e-3[m^4/mol]	-0.001 m^4/mol	Derivative parameter

As mentioned in the Model Definition section, the PID parameters are negative because the setpoint concentration is lower than that at the upper inlet.

GEOMETRY 1

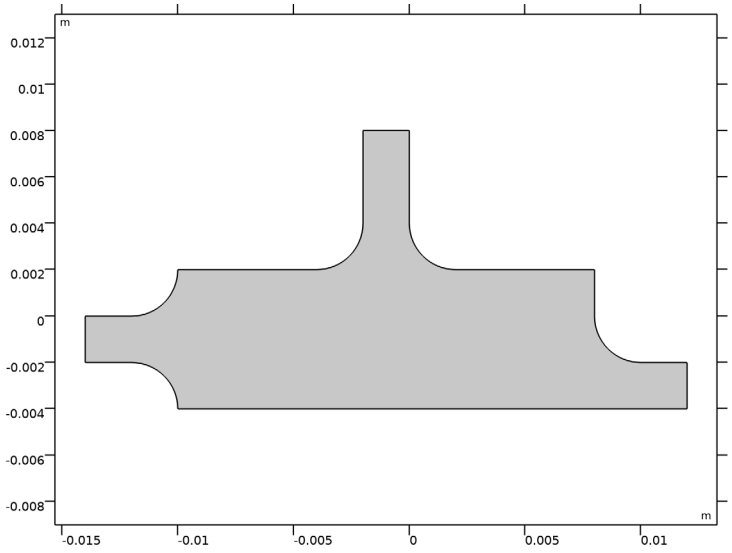
Create the geometry. To simplify this step, insert a prepared geometry sequence.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `pid_control_geom_sequence.mph`.

3 In the **Geometry** toolbar, click  **Build All**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.



DEFINITIONS

Next, add a probe to sample the concentration and its time derivative at the point $x = 0$, $y = -0.002$.

Domain Point Probe 1

1 In the **Definitions** toolbar, click **Probes** and choose **Domain Point Probe**.

2 In the **Settings** window for **Domain Point Probe**, locate the **Point Selection** section.

3 In row **Coordinates**, set **y** to `-0.002`.

Point Probe Expression 1 (ppb1)

1 In the **Model Builder** window, expand the **Domain Point Probe 1** node, then click **Point Probe Expression 1 (ppb1)**.

2 In the **Settings** window for **Point Probe Expression**, type `c_mp` in the **Variable name** text field.

3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Transport of Diluted Species>Species c>c - Concentration - mol/m³**.

Point Probe Expression 2 (ppb2)

1 In the **Model Builder** window, right-click **Domain Point Probe 1** and choose **Point Probe Expression**.

2 In the **Settings** window for **Point Probe Expression**, type `ct_mp` in the **Variable name** text field.

3 Locate the **Expression** section. In the **Expression** text field, type `ct`.

Proceed to import the PID Controller add-in and set up the control algorithm.

4 In the **Home** toolbar, click **Windows** and choose **Add-in Libraries**.

ADD-IN LIBRARIES

1 In the **Add-in Libraries** window, in the tree, select the check box for the node **COMSOL Multiphysics>pid_controller** (if it is not already selected).

2 Click **Done**.

DEVELOPER

In the **Developer** toolbar, click **Add-ins** and choose **PID Controller>PID Controller**.

GLOBAL DEFINITIONS

PID Controller 1

1 In the **Model Builder** window, under **Global Definitions** click **PID Controller 1**.

2 In the **Settings** window for **PID Controller**, locate the **Controller Parameters** section.

3 Enter the following settings:

Parameter	Value
Proportional gain	`k_P_ctrl`
Integral gain	`k_I_ctrl`
Derivative gain	`k_D_ctrl`
Reference value	`c_set`

Keep the default values for the remaining parameters.

4 Click **Create**.

MATERIALS

For the physics setup, you need to specify the density, ρ, and the dynamic viscosity, μ, of the fluid. For this purpose, define a material.

Fluid

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Fluid` in the **Label** text field.

3 Select Domain 1 only.

4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`1.2[kg/m^3]`	kg/m³	Basic
Dynamic viscosity	mu	`3e-5[Pa*s]`	Pa·s	Basic

You are now ready to set up the physics of the model.

LAMINAR FLOW (SPF)

Inlet 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Inlet**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `comp2.u_in_ctrl`. This is the inlet velocity defined by the PID controller.

Inlet 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `v_in_top`.

Outlet 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundary 13 only.

Wall 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Wall**.

2 Select Boundaries 2, 3, 6, and 8 only.

3 In the **Settings** window for **Wall**, locate the **Boundary Condition** section.

4 From the **Wall condition** list, choose **Slip**.

TRANSPORT OF DILUTED SPECIES (TDS)

Transport Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>** **Transport of Diluted Species (tds)** click **Transport Properties 1**.

2 In the **Settings** window for **Transport Properties**, locate the **Convection** section.

3 From the **u** list, choose **Velocity field (spf)**.

4 Locate the **Diffusion** section. In the D_c text field, type `D`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the *c* text field, type `c00`.

Inflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Inflow**, locate the **Concentration** section.

4 In the $c_{0,c}$ text field, type `c_in_inlet`.

Inflow 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Inflow**, locate the **Concentration** section.

4 In the $c_{0,c}$ text field, type `c_in_top`.

Outflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundary 13 only.

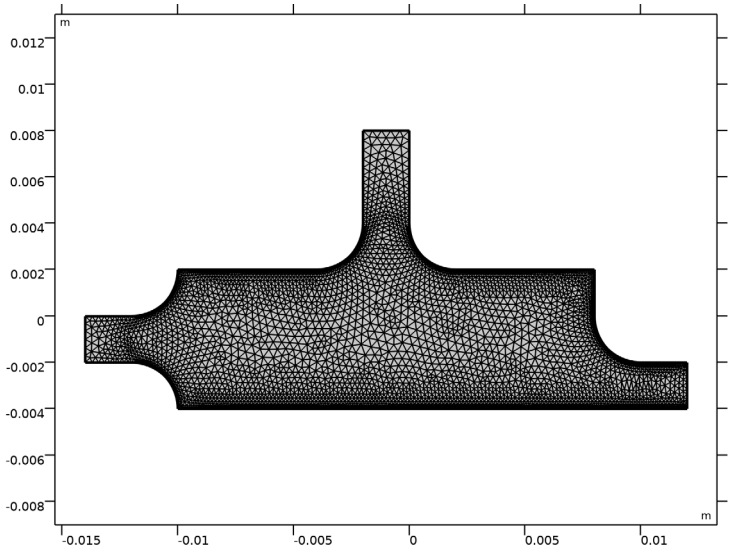
MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.



STUDY 1

Use a parametric sweep to solve for two different values of the proportional parameter, k_P.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list
k_P_ctrl (Proportional parameter)	`-0.1 -0.5`

Step 1: Time Dependent

1 In the **Model Builder** window, click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,0.05,1) range(1.1,0.1,6)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 From the **Method** list, choose **Generalized alpha**.

5 From the **Steps taken by solver** list, choose **Intermediate**.

This forces the solver to take at least one step in each of the time intervals you specified.

6 Click to expand the **Advanced** section. In the **Study** toolbar, click **Compute**.

RESULTS

Concentration (tds)

The default **Concentration** plot group contains a surface plot that shows the concentration at the end of the simulated time span, as well as a streamline plot of the velocity. Study the solution at $t = 0.05$ s and $t = 2$ s.

First, however, adjust the streamline positioning so that the streamline density reflects the velocity magnitude.

Streamline 1

1 In the **Model Builder** window, expand the **Concentration (tds)** node, then click **Streamline 1**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Laminar Flow> Velocity and pressure>u,v - Velocity field**.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Magnitude controlled**.

4 In the **Minimum distance** text field, type `0.005`.

5 In the **Maximum distance** text field, type `0.1`.

6 In the **Concentration (tds)** toolbar, click **Plot**.

Concentration (tds)

1 In the **Model Builder** window, click **Concentration (tds)**.

2 In the **Settings** window for **2D Plot Group**, locate the **Data** section.

3 From the **Time (s)** list, choose **0.05**.

4 In the **Concentration (tds)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

6 From the **Time (s)** list, choose **3**.

7 In the **Concentration (tds)** toolbar, click **Plot**.

Compare the resulting plots with those in Figure 2.

Inlet Velocity

Plot the PID-controlled inlet velocity (Figure 3).

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `Inlet Velocity` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Inlet velocity`.

5 Locate the **Plot Settings** section.

6 Select the **y-axis label** check box. In the associated text field, type `u<sub>in,ctrl</sub> (mm/s)`.

Global 1

1 In the **Model Builder** window, expand the **Inlet Velocity** node, then click **Global 1**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2 (comp2)>Definitions>Variables>comp2.u_in_ctrl - Control variable - m/s**.

3 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Time**.

4 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Description** check box.

5 In the **Inlet Velocity** toolbar, click **Plot**.

Proceed to plot the concentration at the measurement point as a function of time (Figure 3).

Concentration, Measurement Point

1 In the **Model Builder** window, right-click **Inlet Velocity** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Concentration, Measurement Point` in the **Label** text field.

3 Locate the **Title** section. In the **Title** text area, type `Concentration, measurement point`.

4 Locate the **Plot Settings** section. In the **y-axis label** text field, type `c<sub>mp</sub> (mol/m<sup>3</sup>)`.

Global 1

1 In the **Model Builder** window, expand the **Concentration, Measurement Point** node, then click **Global 1**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions> c_mp - Domain Point Probe 1, c - mol/m³**.

3 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Time**.

4 In the **Concentration, Measurement Point** toolbar, click **Plot**.

The resulting plot should look like that in the lower panel of Figure 3.