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.

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.

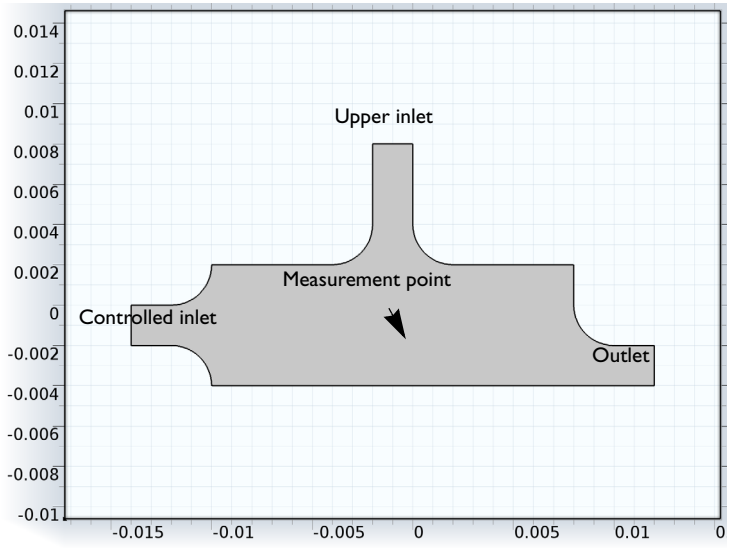

Figure 1: Model geometry.

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:

BOUNDARY	CONSTRAINT
Upper inlet	$c = c_{\text{in,top}}$
Controlled inlet	$c = c_{\text{in,inlet}}$

BOUNDARY	CONSTRAINT
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 (mol/m^3) for the upper and controlled inlets, respectively; D is the applied diffusivity (m^2/s); and $\mathbf{N}$ is the molar flux (mol/(m^2·s)).

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

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 (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

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

with the following 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

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 error $c - c_{\text{set}}$.

Results and Discussion

The two plots in Figure 2 show the oxygen concentration and the velocity stream lines 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 start-up, 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.

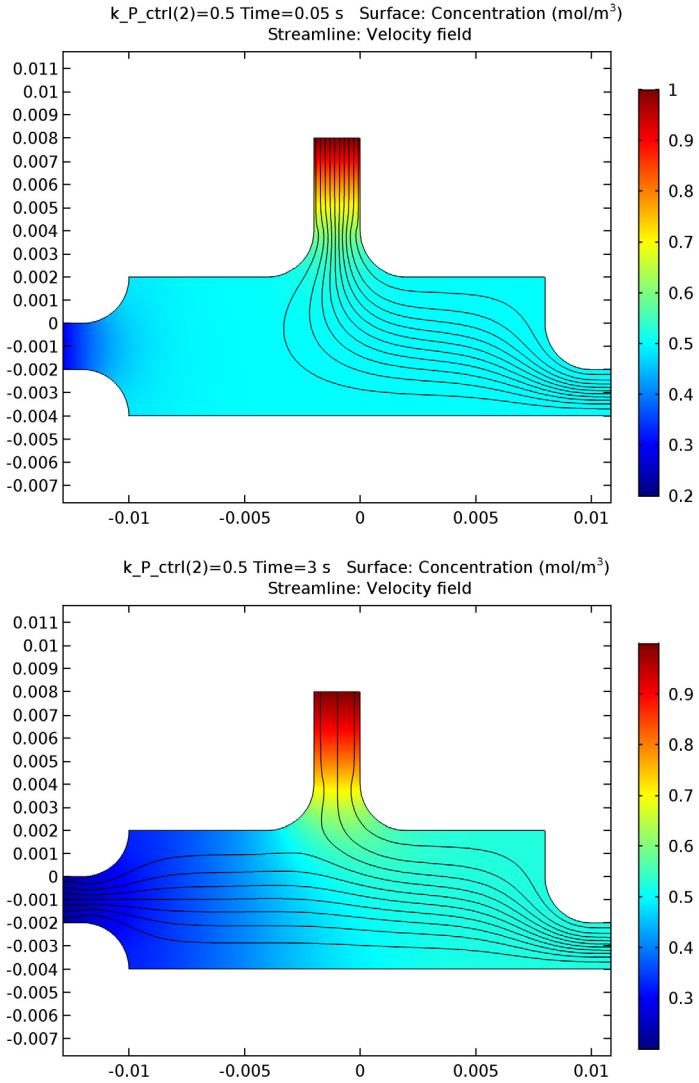

Figure 2: Oxygen concentration and velocity streamlines after 0.1 s (top) and 1.5 s (bottom).

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 solid line represents the results for a k_P value of 0.5 $m^4/(mol \cdot s)$ while the dashed line corresponds to k_P

equal to $0.1\ \mathrm{m}^4/(\mathrm{mol}\cdot\mathrm{s})$. 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.

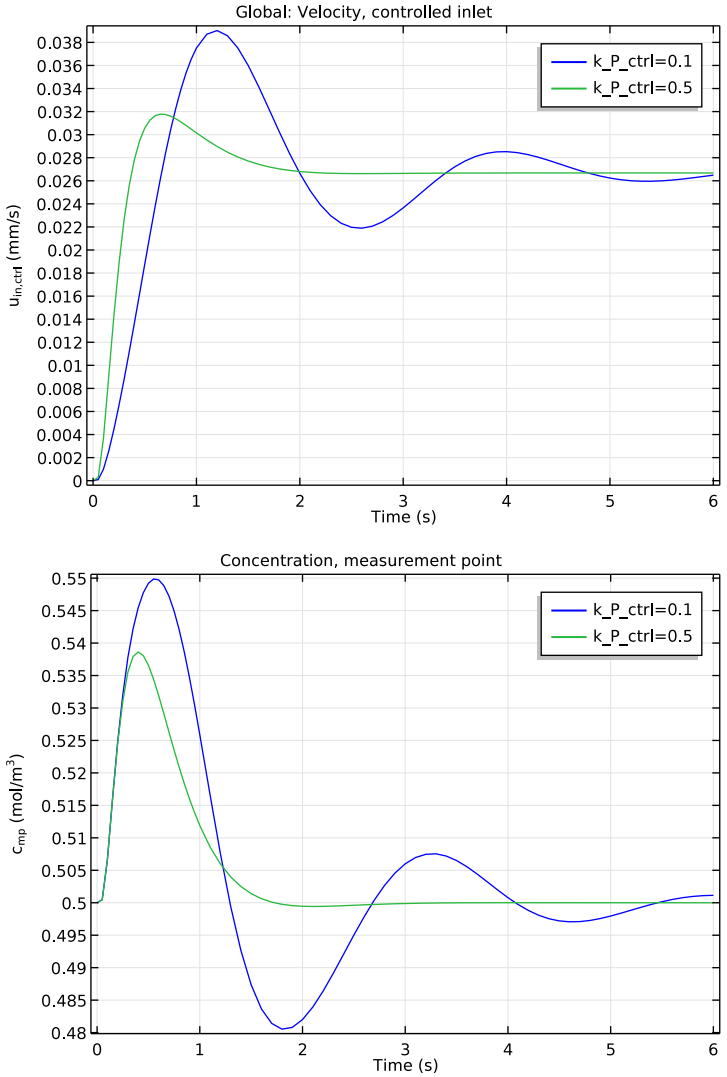

Figure 3: PID-controlled inlet velocity (top) and concentration in the measurement point (bottom) as a function of time for $k_\mathrm{P} = 0.5\ \mathrm{m}^4/(\mathrm{mol}\cdot\mathrm{s})$ (blue) and $k_\mathrm{P} = 0.1\ \mathrm{m}^4/(\mathrm{mol}\cdot\mathrm{s})$ (green).

Application Library path: `COMSOL_Multiphysics/Multiphysics/pid_control`

Modeling Instructions

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

NEW

1 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 In the **Select physics** tree, select **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.

7 Click **Add**.

8 Click **Study**.

9 In the **Select study** tree, select **Preset Studies for Selected Physics Interfaces>Time Dependent**.

10 Click **Done**.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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·mol)	Proportional parameter
k_I_ctrl	1[m^4/(mol*s^2)]	1 m^4/(s²·mol)	Integral parameter
k_D_ctrl	1e-3[m^4/mol]	0.001 m^4/mol	Derivative parameter

GEOMETRY 1

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

1 On the **Geometry** toolbar, click **Insert Sequence**.

2 Browse to the application's Application Library folder and double-click the file `pid_control.mph`.

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

MATERIALS

Material 1 (mat1)

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

2 Select Domain 1 only.

3 In the **Settings** window for Material, locate the **Material Contents** section.

4 In the table, enter the following settings:

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

LAMINAR FLOW (SPF)

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** 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 `u_in_ctrl*(u_in_ctrl>0)`.

Inlet 2

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

2 Select Boundary 13 only.

Wall 2

1 On 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 **Boundary 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 **Diffusion** section.

3 In the D_c text field, type `D`.

4 Locate the **Model Inputs** section. From the $\mathbf{u}$ list, choose **Velocity field (spf)**.

Initial Values 1

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

4 In the **Model Builder** window, click **Transport of Diluted Species (tds)**.

Inflow 1

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

2 Select Boundary 13 only.

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 On 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`.

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

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

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

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

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

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

Variables 1

1 On the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for Variables, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
u_in_ctrl	nojac(k_P_ctrl*(c_mp-c_set)+k_I_ctrl*I[mol*s/m^3]+k_D_ctrl*ct_mp)		Velocity, controlled inlet

The `nojac` operator ensures that the above expression gives no Jacobian contribution. In practice, this means that the control velocity will always be evaluated based on the previous time step. This is necessary to avoid evaluation of an implicit time derivative in the inlet condition, which is not supported in the time dependent solver.

Moreover, `'I'` refers to the time integral in Equation 1, which you define next.

GLOBAL ODES AND DAES (GE)

Global Equations 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Global ODEs and DAEs (ge)** node, then click **Global Equations 1**.

2 In the **Settings** window for Global Equations, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
I	It-(c_mp-c_set)	0	0	Time integral term

MESH 1

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

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

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

4 Click the **Build All** button.

STUDY 1

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

Parametric Sweep

1 On 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	Parameter unit
k_P_ctrl	0.1 0.5	

Step 1: Time Dependent

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

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

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

Solution 1 (sol1)

1 On 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 Locate the **Time Stepping** section. 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. Locate the **Time Stepping** section. Find the **Algebraic variable settings** subsection. From the **Error estimation** list, choose **Exclude algebraic**.

7 On the **Study** toolbar, click **Compute**.

TABLE

Go to the **Table** window.

RESULTS

Velocity (spf)

The presence of the derivative term leads to a warning message from the solver. As already mentioned in the introduction, this term is difficult to determine and also sensitive to noise, so it is often set to 0.

Concentration (tds)

Add a streamline plot of the velocity to the default surface plot that shows the concentration at the end of the simulated time span (Figure 2). Study the solution at t = 0.05 s and t = 2 s.

1. In the **Model Builder** window, under **Results** right-click **Concentration (tds)** and choose **Streamline**.
2. In the **Settings** window for Streamline, locate the **Streamline Positioning** section.
3. From the **Positioning** list, choose **Magnitude controlled**.
4. In the **Density** text field, type `10`.
5. On the **Concentration (tds)** toolbar, click **Plot**.
6. In the **Model Builder** window, click **Concentration (tds)**.
7. In the **Settings** window for 2D Plot Group, locate the **Data** section.
8. From the **Time (s)** list, choose **0.05**.
9. On the **Concentration (tds)** toolbar, click **Plot**.
10. Click the **Zoom Extents** button on the **Graphics** toolbar.
11. From the **Time (s)** list, choose **3**.
12. On the **Concentration (tds)** toolbar, click **Plot**.

1D Plot Group 4

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, locate the **Plot Settings** section.
3. Select the **y-axis label** check box.
4. In the associated text field, type `u<sub>in,ctrl</sub> (mm/s)`.
5. In the **Model Builder** window, expand the **1D Plot Group 4** node, then click **Global 1**.
6. 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>Definitions>Variables>u_in_ctrl - Velocity, controlled inlet**.

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

8 On the **1D Plot Group 4** toolbar, click **Plot**.

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

1D Plot Group 6

1 In the **Model Builder** window, right-click **1D Plot Group 4** and choose **Duplicate**.

2 In the **Settings** window for 1D Plot Group, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Concentration, measurement point`.

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

6 In the **Model Builder** window, expand the **1D Plot Group 6** node, then click **Global 1**.

7 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>Definitions>c_mp - Probe variable c_mp**.

8 On the **1D Plot Group 6** toolbar, click **Plot**.

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