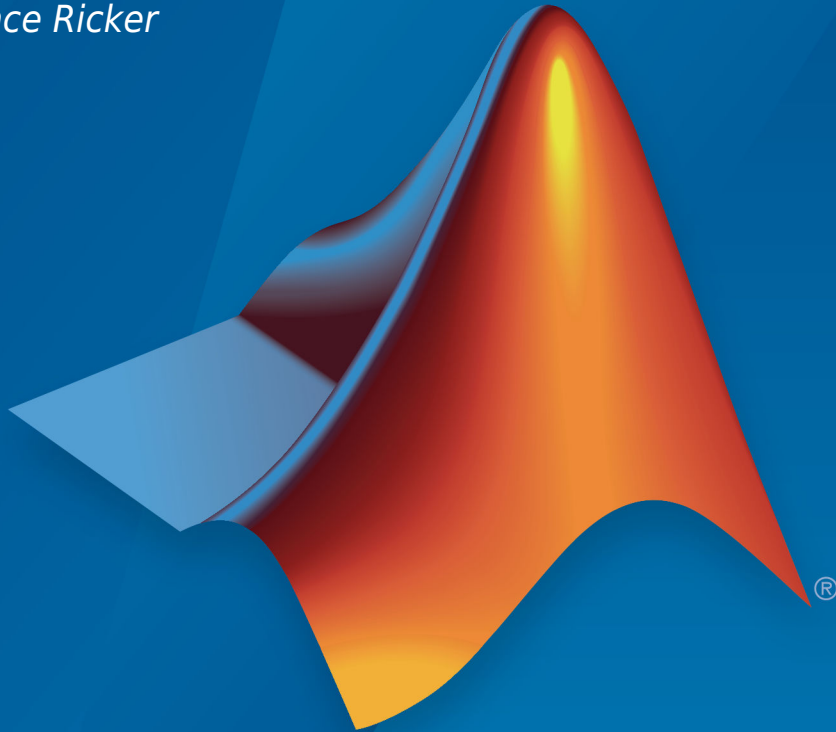


# Model Predictive Control Toolbox™

## Getting Started Guide

*Alberto Bemporad*  
*Manfred Morari*  
*N. Lawrence Ricker*



# MATLAB®

R2018a

 MathWorks®

# How to Contact MathWorks



Latest news: [www.mathworks.com](http://www.mathworks.com)  
Sales and services: [www.mathworks.com/sales\\_and\\_services](http://www.mathworks.com/sales_and_services)  
User community: [www.mathworks.com/matlabcentral](http://www.mathworks.com/matlabcentral)  
Technical support: [www.mathworks.com/support/contact\\_us](http://www.mathworks.com/support/contact_us)



Phone: 508-647-7000



The MathWorks, Inc.  
3 Apple Hill Drive  
Natick, MA 01760-2098

*Model Predictive Control Toolbox™ Getting Started Guide*

© COPYRIGHT 2005–2018 by The MathWorks, Inc.

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from The MathWorks, Inc.

FEDERAL ACQUISITION: This provision applies to all acquisitions of the Program and Documentation by, for, or through the federal government of the United States. By accepting delivery of the Program or Documentation, the government hereby agrees that this software or documentation qualifies as commercial computer software or commercial computer software documentation as such terms are used or defined in FAR 12.212, DFARS Part 227.72, and DFARS 252.227-7014. Accordingly, the terms and conditions of this Agreement and only those rights specified in this Agreement, shall pertain to and govern the use, modification, reproduction, release, performance, display, and disclosure of the Program and Documentation by the federal government (or other entity acquiring for or through the federal government) and shall supersede any conflicting contractual terms or conditions. If this License fails to meet the government's needs or is inconsistent in any respect with federal procurement law, the government agrees to return the Program and Documentation, unused, to The MathWorks, Inc.

## Trademarks

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See [www.mathworks.com/trademarks](http://www.mathworks.com/trademarks) for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

## Patents

MathWorks products are protected by one or more U.S. patents. Please see [www.mathworks.com/patents](http://www.mathworks.com/patents) for more information.

## Revision History

October 2004	First printing	New for Version 2.1 (Release 14SP1)
March 2005	Online only	Revised for Version 2.2 (Release 14SP2)
September 2005	Online only	Revised for Version 2.2.1 (Release 14SP3)
March 2006	Online only	Revised for Version 2.2.2 (Release 2006a)
September 2006	Online only	Revised for Version 2.2.3 (Release 2006b)
March 2007	Online only	Revised for Version 2.2.4 (Release 2007a)
September 2007	Online only	Revised for Version 2.3 (Release 2007b)
March 2008	Online only	Revised for Version 2.3.1 (Release 2008a)
October 2008	Online only	Revised for Version 3.0 (Release 2008b)
March 2009	Online only	Revised for Version 3.1 (Release 2009a)
September 2009	Online only	Revised for Version 3.1.1 (Release 2009b)
March 2010	Online only	Revised for Version 3.2 (Release 2010a)
September 2010	Online only	Revised for Version 3.2.1 (Release 2010b)
April 2011	Online only	Revised for Version 3.3 (Release 2011a)
September 2011	Online only	Revised for Version 4.0 (Release 2011b)
March 2012	Online only	Revised for Version 4.1 (Release 2012a)
September 2012	Online only	Revised for Version 4.1.1 (Release 2012b)
March 2013	Online only	Revised for Version 4.1.2 (Release 2013a)
September 2013	Online only	Revised for Version 4.1.3 (Release 2013b)
March 2014	Online only	Revised for Version 4.2 (Release R2014a)
October 2014	Online only	Revised for Version 5.0 (Release 2014b)
March 2015	Online only	Revised for Version 5.0.1 (Release 2015a)
September 2015	Online only	Revised for Version 5.1 (Release 2015b)
March 2016	Online only	Revised for Version 5.2 (Release 2016a)
September 2016	Online only	Revised for Version 5.2.1 (Release 2016b)
March 2017	Online only	Revised for Version 5.2.2 (Release 2017a)
September 2017	Online only	Revised for Version 6.0 (Release 2017b)
March 2018	Online only	Revised for Version 6.1 (Release 2018a)



	<b>Introduction</b>
<b>1</b>	
	<b>Model Predictive Control Toolbox Product Description . . . . . 1-2</b>
	Key Features . . . . . 1-2
	<b>Acknowledgments . . . . . 1-3</b>
	<b>Bibliography . . . . . 1-4</b>
	<b>Building Models</b>
<b>2</b>	
	<b>MPC Modeling . . . . . 2-2</b>
	Plant Model . . . . . 2-2
	Input Disturbance Model . . . . . 2-4
	Output Disturbance Model . . . . . 2-5
	Measurement Noise Model . . . . . 2-6
	<b>Signal Types . . . . . 2-8</b>
	Inputs . . . . . 2-8
	Outputs . . . . . 2-8
	<b>Construct Linear Time Invariant (LTI) Models . . . . . 2-10</b>
	Transfer Function Models . . . . . 2-10
	Zero/Pole/Gain Models . . . . . 2-10
	State-Space Models . . . . . 2-11
	LTI Object Properties . . . . . 2-12
	LTI Model Characteristics . . . . . 2-16
	<b>Specify Multi-Input Multi-Output Plants . . . . . 2-17</b>

<b>CSTR Model</b> .....	<b>2-20</b>
<b>Linearize Simulink Models</b> .....	<b>2-22</b>
Linearization Using MATLAB Code .....	<b>2-22</b>
Linearization Using Linear Analysis Tool in Simulink Control Design .....	<b>2-25</b>
<b>Linearize Simulink Models Using MPC Designer</b> .....	<b>2-32</b>
Define MPC Structure By Linearization .....	<b>2-32</b>
Linearize Model .....	<b>2-37</b>
Specifying Operating Points .....	<b>2-39</b>
Connect Measured Disturbances for Linearization .....	<b>2-50</b>
<b>Identify Plant from Data</b> .....	<b>2-53</b>
Identify Plant from Data at the Command Line .....	<b>2-53</b>
Working with Impulse-Response Models .....	<b>2-56</b>
<b>Design MPC Controller for Identified Plant Model</b> .....	<b>2-58</b>
Design Controller for Identified Plant Using Apps .....	<b>2-58</b>
Design Controller for Identified Plant at the Command Line .....	<b>2-76</b>
Configure Noise Channels as Unmeasured Disturbances .....	<b>2-81</b>

## Designing Controllers Using MPC Designer

### 3

<b>Design Controller Using MPC Designer</b> .....	<b>3-2</b>
<b>Test Controller Robustness</b> .....	<b>3-24</b>
<b>Design MPC Controller for Plant with Delays</b> .....	<b>3-36</b>
<b>Design MPC Controller for Nonsquare Plant</b> .....	<b>3-44</b>
More Outputs Than Manipulated Variables .....	<b>3-44</b>
More Manipulated Variables Than Outputs .....	<b>3-46</b>

## Designing Controllers Using the Command Line

### 4

<b>Design MPC Controller at the Command Line</b> .....	<b>4-2</b>
<b>Simulate Controller with Nonlinear Plant</b> .....	<b>4-16</b>
Nonlinear CSTR Application .....	<b>4-16</b>
Example Code for Successive Linearization .....	<b>4-17</b>
CSTR Results and Discussion .....	<b>4-19</b>
<b>Compute Steady-State Gain</b> .....	<b>4-23</b>
<b>Extract Controller</b> .....	<b>4-25</b>
<b>Signal Previewing</b> .....	<b>4-28</b>
<b>Update Constraints at Run Time</b> .....	<b>4-30</b>
Update Bounds on Input and Output Signals at Run Time ...	<b>4-30</b>
Update Mixed Input/Output Constraints at Run Time .....	<b>4-31</b>
<b>Tune Weights at Run Time</b> .....	<b>4-33</b>

## Designing and Testing Controllers in Simulink

### 5

<b>Design MPC Controller in Simulink</b> .....	<b>5-2</b>
<b>Test an Existing Controller</b> .....	<b>5-23</b>





# Introduction

---

- “Model Predictive Control Toolbox Product Description” on page 1-2
- “Acknowledgments” on page 1-3
- “Bibliography” on page 1-4

## Model Predictive Control Toolbox Product Description

### Design and simulate model predictive controllers

Model Predictive Control Toolbox provides functions, an app, and Simulink® blocks for designing and simulating model predictive controllers (MPCs). The toolbox lets you specify plant and disturbance models, horizons, constraints, and weights. By running closed-loop simulations, you can evaluate controller performance.

You can adjust the behavior of the controller by varying its weights and constraints at run time. To control a nonlinear plant, you can implement adaptive and gain-scheduled MPCs. For applications with fast sample rates, you can generate an explicit model predictive controller from a regular controller or implement an approximate solution.

For rapid prototyping and embedded system implementation, the toolbox supports automatic C-code and IEC 61131-3 Structured Text generation.

### Key Features

- App for interactive design of MPC controllers
- Runtime adjustment of weights and constraints
- Adaptive and gain-scheduled MPCs for controlling systems with nonlinear dynamics
- Explicit MPC and approximate solution for guaranteed worst-case execution time
- Economic MPC with arbitrary nonlinear cost function and constraints
- Computationally efficient quadratic programming (QP) solver, and support for third-party solvers
- Support for C-code generation (with Simulink Coder™) and IEC 61131-3 Structured Text generation (with Simulink PLC Coder™)

## Acknowledgments

MathWorks would like to acknowledge the following contributors to Model Predictive Control Toolbox.

### **Alberto Bemporad**

Professor of Control Systems, IMT Institute for Advanced Studies Lucca, Italy. Research interests include model predictive control, hybrid systems, optimization algorithms, and applications to automotive, aerospace, and energy systems. Fellow of the IEEE®. Author of the Model Predictive Control Simulink library and commands.

### **Manfred Morari**

Professor at the Automatic Control Laboratory and former Head of Department of Information Technology and Electrical Engineering, ETH Zurich, Switzerland. Research interests include model predictive control, hybrid systems, and robust control. Fellow of the IEEE, AIChE, and IFAC. Co-author of the first version of the toolbox.

### **N. Lawrence Ricker**

Professor of Chemical Engineering, University of Washington, Seattle, USA. Research interests include model predictive control and process optimization. Author of the quadratic programming solver and graphical user interface.

## **Bibliography**

- [1] Allgower, F., and A. Zheng, *Nonlinear Model Predictive Control*, Springer-Verlag, 2000.
- [2] Camacho, E. F., and C. Bordons, *Model Predictive Control*, Springer-Verlag, 1999.
- [3] Kouvaritakis, B., and M. Cannon, *Non-Linear Predictive Control: Theory & Practice*, IEE Publishing, 2001.
- [4] Maciejowski, J. M., *Predictive Control with Constraints*, Pearson Education POD, 2002.
- [5] Prett, D., and C. Garcia, *Fundamental Process Control*, Butterworths, 1988.
- [6] Rossiter, J. A., *Model-Based Predictive Control: A Practical Approach*, CRC Press, 2003.

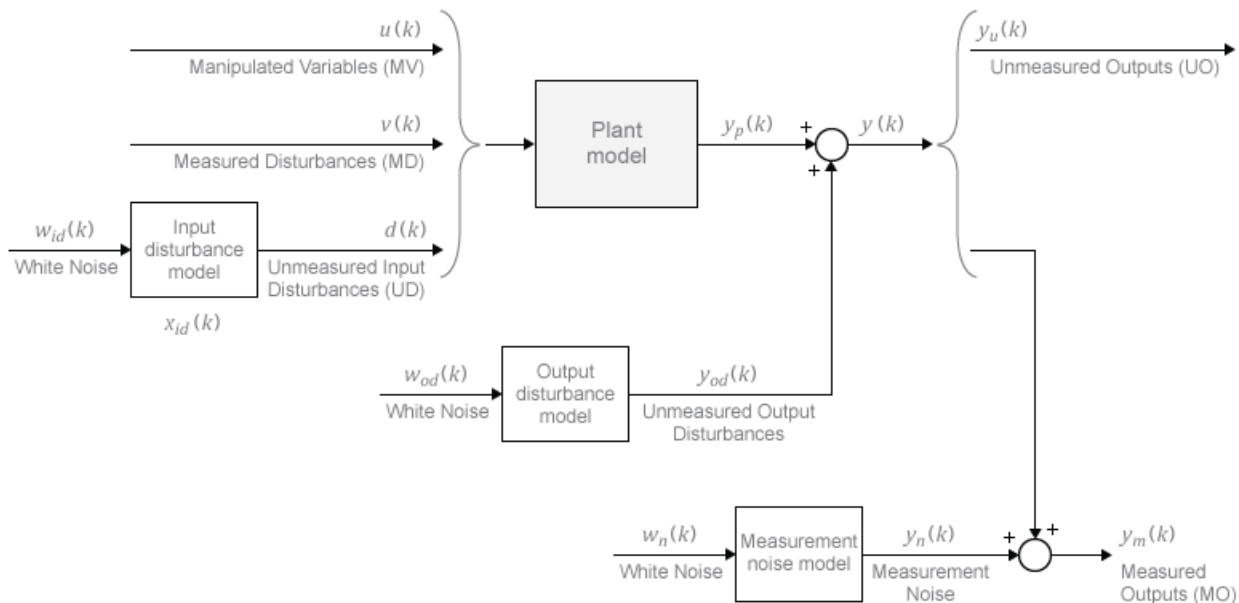
# Building Models

---

- “MPC Modeling” on page 2-2
- “Signal Types” on page 2-8
- “Construct Linear Time Invariant (LTI) Models” on page 2-10
- “Specify Multi-Input Multi-Output Plants” on page 2-17
- “CSTR Model” on page 2-20
- “Linearize Simulink Models” on page 2-22
- “Linearize Simulink Models Using MPC Designer” on page 2-32
- “Identify Plant from Data” on page 2-53
- “Design MPC Controller for Identified Plant Model” on page 2-58

## MPC Modeling

Model predictive controllers use plant, disturbance, and noise models for prediction and state estimation. The model structure used in an MPC controller appears in the following illustration.



### Plant Model

You can specify the plant model in one of the following linear-time-invariant (LTI) formats:

- Numeric LTI models — Transfer function (`tf`), state space (`ss`), zero-pole-gain (`zpk`)
- Identified models (requires System Identification Toolbox™) — `idss`, `idtf`, `idproc`, and `idpoly`

The MPC controller performs all estimation and optimization calculations using a discrete-time, delay-free, state-space system with dimensionless input and output variables. Therefore, when you specify a plant model in the MPC controller, the software performs the following, if needed:

- 1 Conversion to state space — The `ss` command converts the supplied model to an LTI state-space model.
- 2 Discretization or resampling — If the model sample time differs from the MPC controller sample time (defined in the `Ts` property), one of the following occurs:
  - If the model is continuous time, the `c2d` command converts it to a discrete-time LTI object using the controller sample time.
  - If the model is discrete time, the `d2d` command resamples it to generate a discrete-time LTI object using the controller sample time.
- 3 Delay removal — If the discrete-time model includes any input, output, or internal delays, the `absorbDelay` command replaces them with the appropriate number of poles at  $z = 0$ , increasing the total number of discrete states. The `InputDelay`, `OutputDelay`, and `InternalDelay` properties of the resulting state space model are all zero.
- 4 Conversion to dimensionless input and output variables — The MPC controller enables you to specify a scale factor for each plant input and output variable. If you do not specify scale factors, they default to 1. The software converts the plant input and output variables to dimensionless form as follows:

$$x_p(k+1) = A_p x_p(k) + B S_i u_p(k)$$

$$y_p(k) = S_o^{-1} C x_p(k) + S_o^{-1} D S_i u_p(k).$$

where  $A_p$ ,  $B$ ,  $C$ , and  $D$  are the constant state-space matrices determined in step 3, and:

- $S_i$  is a diagonal matrix of input scale factors in engineering units.
- $S_o$  is a diagonal matrix of output scale factors in engineering units.
- $x_p$  is the state vector from step 3 in engineering units. No scaling is performed on state variables.
- $u_p$  is a vector of dimensionless plant input variables.
- $y_p$  is a vector of dimensionless plant output variables.

The resulting plant model has the following equivalent form:

$$x_p(k+1) = A_p x_p(k) + B_{pu} u(k) + B_{pv} v(k) + B_{pd} d(k)$$

$$y_p(k) = C_p x_p(k) + D_{pu} u(k) + D_{pv} v(k) + D_{pd} d(k).$$

Here,  $C_p = S_o^{-1}C$ ,  $B_{pu}$ ,  $B_{pv}$ , and  $B_{pd}$  are the corresponding columns of  $BS_i$ . Also,  $D_{pu}$ ,  $D_{pv}$ , and  $D_{pd}$  are the corresponding columns of  $S_o^{-1}DS_i$ . Finally,  $u(k)$ ,  $v(k)$ , and  $d(k)$  are the dimensionless manipulated variables, measured disturbances, and unmeasured input disturbances, respectively.

The MPC controller enforces the restriction of  $D_{pu} = 0$ , which means that the controller does not allow direct feedthrough from any manipulated variable to any plant output.

## Input Disturbance Model

If your plant model includes unmeasured input disturbances,  $d(k)$ , the input disturbance model specifies the signal type and characteristics of  $d(k)$ . See “Controller State Estimation” for more information about the model.

The `getindist` command provides access to the model in use.

The input disturbance model is a key factor that influences the following controller performance attributes:

- Dynamic response to apparent disturbances — The character of the controller response when the measured plant output deviates from its predicted trajectory, due to an unknown disturbance or modeling error.
- Asymptotic rejection of sustained disturbances — If the disturbance model predicts a sustained disturbance, controller adjustments continue until the plant output returns to its desired trajectory, emulating a classical integral feedback controller.

You can provide the input disturbance model as an LTI state-space (`ss`), transfer function (`tf`), or zero-pole-gain (`zpk`) object using `setindist`. The MPC controller converts the input disturbance model to a discrete-time, delay-free, LTI state-space system using the same steps used to convert the plant model on page 2-2. The result is:

$$\begin{aligned}x_{id}(k+1) &= A_{id}x_{id}(k) + B_{id}w_{id}(k) \\d(k) &= C_{id}x_{id}(k) + D_{id}w_{id}(k).\end{aligned}$$

where  $A_{id}$ ,  $B_{id}$ ,  $C_{id}$ , and  $D_{id}$  are constant state-space matrices, and:



- $x_{id}(k)$  is a vector of  $n_{xid} \geq 0$  input disturbance model states.
- $d_k(k)$  is a vector of  $n_d$  dimensionless unmeasured input disturbances.
- $w_{id}(k)$  is a vector of  $n_{id} \geq 1$  dimensionless white noise inputs, assumed to have zero mean and unit variance.

If you do not provide an input disturbance model, then the controller uses a default model, which has integrators with dimensionless unity gain added to its outputs. An integrator is added for each unmeasured input disturbance, unless doing so would cause a violation of state observability. In this case, a static system with dimensionless unity gain is used instead.

## Output Disturbance Model

The output disturbance model is a special case of the more general input disturbance model. Its output,  $y_{od}(k)$ , is directly added to the plant output rather than affecting the plant states. The output disturbance model specifies the signal type and characteristics of  $y_{od}(k)$ , and it is often used in practice. See “Controller State Estimation” for more details about the model.

The `getoutdist` command provides access to the output disturbance model in use.

You can specify a custom output disturbance model as an LTI state-space (`ss`), transfer function (`tf`), or zero-pole-gain (`zpk`) object using `setoutdist`. Using the same steps as for the plant model on page 2-2, the MPC controller converts the specified output disturbance model to a discrete-time, delay-free, LTI state-space system. The result is:

$$\begin{aligned}x_{od}(k+1) &= A_{od}x_{od}(k) + B_{od}w_{od}(k) \\ y_{od}(k) &= C_{od}x_{od}(k) + D_{od}w_{od}(k).\end{aligned}$$

where  $A_{od}$ ,  $B_{od}$ ,  $C_{od}$ , and  $D_{od}$  are constant state-space matrices, and:

- $x_{od}(k)$  is a vector of  $n_{xod} \geq 1$  output disturbance model states.
- $y_{od}(k)$  is a vector of  $n_y$  dimensionless output disturbances to be added to the dimensionless plant outputs.
- $w_{od}(k)$  is a vector of  $n_{od}$  dimensionless white noise inputs, assumed to have zero mean and unit variance.

If you do not specify an output disturbance model, then the controller uses a default model, which has integrators with dimensionless unity gain added to some or all of its outputs. These integrators are added according to the following rules:

- No disturbances are estimated, that is no integrators are added, for unmeasured plant outputs.
- An integrator is added for each measured output in order of decreasing output weight.
  - For time-varying weights, the sum of the absolute values over time is considered for each output channel.
  - For equal output weights, the order within the output vector is followed.
- For each measured output, an integrator is not added if doing so would cause a violation of state observability. Instead, a gain with a value of zero is used instead.

If there is an input disturbance model, then the controller adds any default integrators to that model before constructing the default output disturbance model.

### Measurement Noise Model

One controller design objective is to distinguish disturbances, which require a response, from measurement noise, which should be ignored. The measurement noise model specifies the expected noise type and characteristics. See “Controller State Estimation” for more details about the model.

Using the same steps as for the plant model on page 2-2, the MPC controller converts the measurement noise model to a discrete-time, delay-free, LTI state-space system. The result is:

$$\begin{aligned}x_n(k+1) &= A_n x_n(k) + B_n w_n(k) \\ y_n(k) &= C_n x_n(k) + D_n w_n(k).\end{aligned}$$

Here,  $A_n$ ,  $B_n$ ,  $C_n$ , and  $D_n$  are constant state space matrices, and:

- $x_n(k)$  is a vector of  $n_{xn} \geq 0$  noise model states.
- $y_n(k)$  is a vector of  $n_{ym}$  dimensionless noise signals to be added to the dimensionless measured plant outputs.
- $w_n(k)$  is a vector of  $n_n \geq 1$  dimensionless white noise inputs, assumed to have zero mean and unit variance.

If you do not supply a noise model, the default is a unity static gain:  $n_{xn} = 0$ ,  $D_n$  is an  $n_{ym}$ -by- $n_{ym}$  identity matrix, and  $A_n$ ,  $B_n$ , and  $C_n$  are empty.

For an mpc controller object, `MPCobj`, the property `MPCobj.Model.Noise` provides access to the measurement noise model.

---

**Note** If the minimum eigenvalue of  $D_n D_n^T$  is less than  $1 \times 10^{-8}$ , the MPC controller adds  $1 \times 10^{-4}$  to each diagonal element of  $D_n$ . This adjustment makes a successful default Kalman gain calculation more likely.

---

## See Also

### More About

- “Controller State Estimation”

# Signal Types

## Inputs

The *plant inputs* are the independent variables affecting the plant. As shown in “MPC Modeling” on page 2-2, there are three types:

### Measured disturbances

The controller can't adjust them, but uses them for feedforward compensation.

### Manipulated variables

The controller adjusts these in order to achieve its goals.

### Unmeasured disturbances

These are independent inputs of which the controller has no direct knowledge, and for which it must compensate.

## Outputs

The *plant outputs* are the dependent variables (outcomes) you wish to control or monitor. As shown in “MPC Modeling” on page 2-2, there are two types:

### Measured outputs

The controller uses these to estimate unmeasured quantities and as feedback on the success of its adjustments.

### Unmeasured outputs

The controller estimates these based on available measurements and the plant model. The controller can also hold unmeasured outputs at setpoints or within constraint boundaries.

You must specify the input and output types when designing the controller. See “Input and Output Types” on page 2-14 for more details.

## **See Also**

### **More About**

- “MPC Modeling” on page 2-2

## Construct Linear Time Invariant (LTI) Models

Model Predictive Control Toolbox software supports the same LTI model formats as does Control System Toolbox software. You can use whichever is most convenient for your application. It's also easy to convert from one format to another. For more details, see the Control System Toolbox documentation.

### Transfer Function Models

A transfer function (TF) relates a particular input/output pair. For example, if  $u(t)$  is a plant input and  $y(t)$  is an output, the transfer function relating them might be:

$$\frac{Y(s)}{U(s)} = G(s) = \frac{s+2}{s^2+s+10} e^{-1.5s}$$

This TF consists of a *numerator* polynomial,  $s+2$ , a *denominator* polynomial,  $s^2+s+10$ , and a delay, which is 1.5 time units here. You can define  $G$  using Control System Toolbox `tf` function:

```
Gtf1 = tf([1 2], [1 1 10], 'OutputDelay', 1.5)
```

Control System Toolbox software builds and displays it as follows:

Transfer function:

$$\exp(-1.5*s) * \frac{s + 2}{s^2 + s + 10}$$

### Zero/Pole/Gain Models

Like the TF, the zero/pole/gain (ZPK) format relates an input/output pair. The difference is that the ZPK numerator and denominator polynomials are factored, as in

$$G(s) = 2.5 \frac{s + 0.45}{(s + 0.3)(s + 0.1 + 0.7i)(s + 0.1 - 0.7i)}$$

(zeros and/or poles are complex numbers in general).

You define the ZPK model by specifying the zero(s), pole(s), and gain as in

```
poles = [-0.3, -0.1+0.7*i, -0.1-0.7*i];
Gzpk1 = zpk(-0.45,poles,2.5);
```

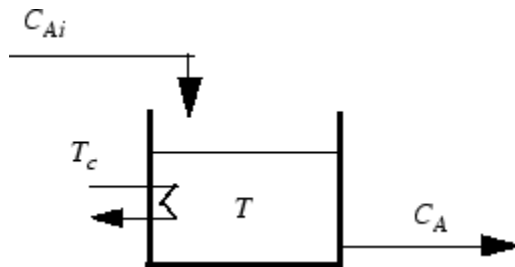
## State-Space Models

The state-space format is convenient if your model is a set of LTI differential and algebraic equations. For example, consider the following linearized model of a continuous stirred-tank reactor (CSTR) involving an exothermic (heat-generating) reaction [1].

$$\frac{dC'_A}{dt} = a_{11}C'_A + a_{12}T' + b_{11}T'_c + b_{12}C'_{Ai}$$

$$\frac{dT'}{dt} = a_{21}C'_A + a_{22}T' + b_{21}T'_c + b_{22}C'_{Ai}$$

where  $C_A$  is the concentration of a key reactant,  $T$  is the temperature in the reactor,  $T_c$  is the coolant temperature,  $C_{Ai}$  is the reactant concentration in the reactor feed, and  $a_{ij}$  and  $b_{ij}$  are constants. See the process schematic in “CSTR Schematic” on page 2-11. The primes (e.g.,  $C'_A$ ) denote a deviation from the nominal steady-state condition at which the model has been linearized.



### CSTR Schematic

Measurement of reactant concentrations is often difficult, if not impossible. Let us assume that  $T$  is a measured output,  $C_A$  is an unmeasured output,  $T_c$  is a manipulated variable, and  $C_{Ai}$  is an unmeasured disturbance.

The model fits the general state-space format

$$\frac{dx}{dt} = Ax + Bu$$

$$y = Cx + Du$$

where

$$x = \begin{bmatrix} C'_A \\ T' \end{bmatrix}, u = \begin{bmatrix} T'_c \\ C'_{Ai} \end{bmatrix}, y = \begin{bmatrix} T' \\ C'_A \end{bmatrix}$$

$$A = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix}, B = \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix}, C = \begin{bmatrix} 0 & 1 \\ 1 & 0 \end{bmatrix}, D = \begin{bmatrix} 0 & 0 \\ 0 & 0 \end{bmatrix}$$

The following code shows how to define such a model for some specific values of the  $a_{ij}$  and  $b_{ij}$  constants:

```
A = [-0.0285  -0.0014
      -0.0371  -0.1476];
B = [-0.0850   0.0238
      0.0802   0.4462];
C = [0 1
      1 0];
D = zeros(2,2);
CSTR = ss(A,B,C,D);
```

This defines a *continuous-time* state-space model. If you do not specify a sampling period, a default sampling value of zero applies. You can also specify discrete-time state-space models. You can specify delays in both continuous-time and discrete-time models.

---

**Note** In the CSTR example, the  $D$  matrix is zero and the output does not instantly respond to change in the input. The Model Predictive Control Toolbox software prohibits direct (instantaneous) feedthrough from a manipulated variable to an output. For example, the CSTR model could include direct feedthrough from the unmeasured disturbance,  $C_{Ai}$ , to either  $C_A$  or  $T$  but direct feedthrough from  $T_c$  to either output would violate this restriction. If the model had direct feedthrough from  $T_c$ , you can add a small delay at this input to circumvent the problem.

---

## LTI Object Properties

The `ss` function in the last line of the above code creates a state-space model, `CSTR`, which is an *LTI object*. The `tf` and `zpk` commands described in “Transfer Function



Models” on page 2-10 and “Zero/Pole/Gain Models” on page 2-10 also create LTI objects. Such objects contain the model parameters as well as optional properties.

### LTI Properties for the CSTR Example

The following code sets some of the CSTR model's optional properties:

```
CSTR.InputName = {'T_c', 'C_A_i'};
CSTR.OutputName = {'T', 'C_A'};
CSTR.StateName = {'C_A', 'T'};
CSTR.InputGroup.MV = 1;
CSTR.InputGroup.UD = 2;
CSTR.OutputGroup.MO = 1;
CSTR.OutputGroup.UO = 2;
CSTR
```

The first three lines specify labels for the input, output and state variables. The next four specify the signal type for each input and output. The designations MV, UD, MO, and UO mean *manipulated variable*, *unmeasured disturbance*, *measured output*, and *unmeasured output*. (See “Signal Types” on page 2-8 for definitions.) For example, the code specifies that input 2 of model CSTR is an unmeasured disturbance. The last line causes the LTI object to be displayed, generating the following lines in the MATLAB® Command Window:

```
A =
      C_A      T
C_A -0.0285 -0.0014
T   -0.0371 -0.1476
```

```
B =
      T_c      C_Ai
C_A -0.085  0.0238
T   0.0802  0.4462
```

```
C =
      C_A      T
T      0      1
C_A    1      0
```

```
D =
      T_c      C_Ai
T      0      0
C_A    0      0
```

Input groups:

Name	Channels
MV	1
UD	2

Output groups:

Name	Channels
MO	1
UO	2

Continuous-time model

### Input and Output Names

The optional `InputName` and `OutputName` properties affect the model displays, as in the above example. The software also uses the `InputName` and `OutputName` properties to label plots and tables. In that context, the underscore character causes the next character to be displayed as a subscript.

### Input and Output Types

#### General Case

As mentioned in “Signal Types” on page 2-8, Model Predictive Control Toolbox software supports three input types and two output types. In a Model Predictive Control Toolbox design, designation of the input and output types determines the controller dimensions and has other important consequences.

For example, suppose your plant structure were as follows:

Plant Inputs	Plant Outputs
Two manipulated variables (MVs)	Three measured outputs (MOs)
One measured disturbance (MD)	Two unmeasured outputs (UOs)
Two unmeasured disturbances (UDs)	

The resulting controller has four inputs (the three MOs and the MD) and two outputs (the MVs). It includes feedforward compensation for the measured disturbance, and assumes that you wanted to include the unmeasured disturbances and outputs as part of the regulator design.

If you didn't want a particular signal to be treated as one of the above types, you could do one of the following:

- Eliminate the signal before using the model in controller design.
- For an output, designate it as unmeasured, then set its weight to zero.
- For an input, designate it as an unmeasured disturbance, then define a custom state estimator that ignores the input.

---

**Note** By default, the software assumes that unspecified plant inputs are manipulated variables, and unspecified outputs are measured. Thus, if you didn't specify signal types in the above example, the controller would have four inputs (assuming all plant outputs were measured) and five outputs (assuming all plant inputs were manipulated variables).

---

For model CSTR, the default Model Predictive Control Toolbox assumptions are incorrect. You must set its `InputGroup` and `OutputGroup` properties, as illustrated in the above code, or modify the default settings when you load the model into **MPC Designer**.

Use `setmpcsignals` to make type definition. For example:

```
CSTR = setmpcsignals(CSTR, 'UD',2, 'UO',2);
```

sets `InputGroup` and `OutputGroup` to the same values as in the previous example. The CSTR display would then include the following lines:

```
Input groups:
  Name      Channels
  Unmeasured  2
  Manipulated  1
```

```
Output groups:
  Name      Channels
  Unmeasured  2
  Measured    1
```

Notice that `setmpcsignals` sets unspecified inputs to `Manipulated` and unspecified outputs to `Measured`.

## LTI Model Characteristics

Control System Toolbox software provides functions for analyzing LTI models. Some of the more commonly used are listed below. Type the example code at the MATLAB prompt to see how they work for the CSTR example.

<b>Example</b>	<b>Intended Result</b>
<code>dcgain(CSTR)</code>	Calculate gain matrix for the CSTR model's input/output pairs.
<code>impulse(CSTR)</code>	Graph CSTR model's unit-impulse response.
<code>linearSystemAnalyzer(CSTR)</code>	Open the Linear System Analyzer with the CSTR model loaded. You can then display model characteristics by making menu selections.
<code>pole(CSTR)</code>	Calculate CSTR model's poles (to check stability, etc.).
<code>step(CSTR)</code>	Graph CSTR model's unit-step response.
<code>zero(CSTR)</code>	Compute CSTR model's transmission zeros.

## References

[1] Seborg, D. E., T. F. Edgar, and D. A. Mellichamp, *Process Dynamics and Control*, 2nd Edition, Wiley, 2004, pp. 34-36 and 94-95.

## See Also

`setmpcsignals` | `ss` | `tf` | `zpk`

## More About

- “Specify Multi-Input Multi-Output Plants” on page 2-17

## Specify Multi-Input Multi-Output Plants

Most MPC applications involve plants with multiple inputs and outputs. You can use `ss`, `tf`, and `zpk` to represent a MIMO plant model. For example, consider the following model of a distillation column [1], which has been used in many advanced control studies:

$$\begin{bmatrix} y_1 \\ y_2 \end{bmatrix} = \begin{bmatrix} \frac{12.8e^{-s}}{16.7s+1} & \frac{-18.9e^{-3s}}{21.0s+1} & \frac{3.8e^{-8.1s}}{14.9s+1} \\ \frac{6.6e^{-7s}}{10.9s+1} & \frac{-19.4e^{-3s}}{14.4s+1} & \frac{4.9e^{-3.4s}}{13.2s+1} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix}$$

Outputs  $y_1$  and  $y_2$  represent measured product purities. The controller manipulates the inputs,  $u_1$  and  $u_2$ , to hold each output at a specified setpoint. These inputs represent the flow rates of reflux and reboiler steam, respectively. Input  $u_3$  is a measured feed flow rate disturbance.

The model consists of six transfer functions, one for each input/output pair. Each transfer function is the first-order-plus-delay form often used by process control engineers.

Specify the individual transfer functions for each input/output pair. For example, `g12` is the transfer function from input  $u_1$  to output  $y_2$ .

```
g11 = tf( 12.8, [16.7 1], 'I0delay', 1.0, 'TimeUnit', 'minutes');
g12 = tf(-18.9, [21.0 1], 'I0delay', 3.0, 'TimeUnit', 'minutes');
g13 = tf( 3.8, [14.9 1], 'I0delay', 8.1, 'TimeUnit', 'minutes');
g21 = tf( 6.6, [10.9 1], 'I0delay', 7.0, 'TimeUnit', 'minutes');
g22 = tf(-19.4, [14.4 1], 'I0delay', 3.0, 'TimeUnit', 'minutes');
g23 = tf( 4.9, [13.2 1], 'I0delay', 3.4, 'TimeUnit', 'minutes');
```

Define a MIMO system by creating a matrix of transfer function models.

```
DC = [g11 g12 g13
      g21 g22 g23];
```

Define the input and output signal names and specify the third input as a measured input disturbance.

```
DC.InputName = {'Reflux Rate', 'Steam Rate', 'Feed Rate'};
DC.OutputName = {'Distillate Purity', 'Bottoms Purity'};
DC = setmpcsignals(DC, 'MD', 3);
```

```
-->Assuming unspecified input signals are manipulated variables.
```

Review the resulting system.

DC

DC =

From input "Reflux Rate" to output...

$$\text{Distillate Purity: } \exp(-1*s) * \frac{12.8}{16.7 s + 1}$$

$$\text{Bottoms Purity: } \exp(-7*s) * \frac{6.6}{10.9 s + 1}$$

From input "Steam Rate" to output...

$$\text{Distillate Purity: } \exp(-3*s) * \frac{-18.9}{21 s + 1}$$

$$\text{Bottoms Purity: } \exp(-3*s) * \frac{-19.4}{14.4 s + 1}$$

From input "Feed Rate" to output...

$$\text{Distillate Purity: } \exp(-8.1*s) * \frac{3.8}{14.9 s + 1}$$

$$\text{Bottoms Purity: } \exp(-3.4*s) * \frac{4.9}{13.2 s + 1}$$

Input groups:

Name	Channels
Measured	3
Manipulated	1,2

Output groups:

Name	Channels
Measured	1,2

Continuous-time transfer function.

## References

[1] Wood, R. K., and M. W. Berry, *Chem. Eng. Sci.*, Vol. 28, pp. 1707, 1973.

## See Also

setmpcsignals | ss | tf | zpk

## Related Examples

- “Construct Linear Time Invariant (LTI) Models” on page 2-10

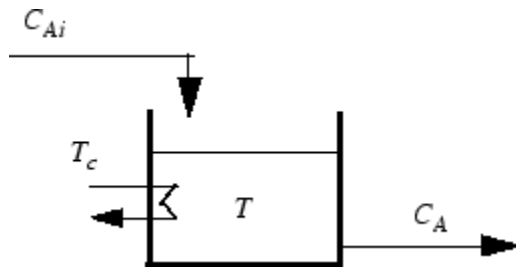
## CSTR Model

The linearized model of a continuous stirred-tank reactor (CSTR) involving an exothermic (heat-generating) reaction is represented by the following differential equations:

$$\frac{dC'_A}{dt} = a_{11}C'_A + a_{12}T' + b_{11}T'_c + b_{12}C'_{Ai}$$

$$\frac{dT'}{dt} = a_{21}C'_A + a_{22}T' + b_{21}T'_c + b_{22}C'_{Ai}$$

where  $C_A$  is the concentration of a key reactant,  $T$  is the temperature in the reactor,  $T_c$  is the coolant temperature,  $C_{Ai}$  is the reactant concentration in the reactor feed, and  $a_{ij}$  and  $b_{ij}$  are constants. The primes (e.g.,  $C'_A$ ) denote a deviation from the nominal steady-state condition at which the model has been linearized.



Measurement of reactant concentrations is often difficult, if not impossible. Let us assume that  $T$  is a measured output,  $C_A$  is an unmeasured output,  $T_c$  is a manipulated variable, and  $C_{Ai}$  is an unmeasured disturbance.

The model fits the general state-space format

$$\frac{dx}{dt} = Ax + Bu$$

$$y = Cx + Du$$

where



$$x = \begin{bmatrix} C'_A \\ T' \end{bmatrix}, u = \begin{bmatrix} T'_c \\ C'_{Ai} \end{bmatrix}, y = \begin{bmatrix} T' \\ C'_A \end{bmatrix}$$

$$A = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix}, B = \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix}, C = \begin{bmatrix} 0 & 1 \\ 1 & 0 \end{bmatrix}, D = \begin{bmatrix} 0 & 0 \\ 0 & 0 \end{bmatrix}$$

The following code shows how to define such a model for some specific values of the  $a_{ij}$  and  $b_{ij}$  constants:

```
A = [-0.0285  -0.0014
      -0.0371  -0.1476];
B = [-0.0850   0.0238
      0.0802   0.4462];
C = [0 1
      1 0];
D = zeros(2,2);
CSTR = ss(A,B,C,D);
```

The following code sets some of the CSTR model's optional properties:

```
CSTR.InputName = {'T_c', 'C_A_i'};
CSTR.OutputName = {'T', 'C_A'};
CSTR.StateName = {'C_A', 'T'};
CSTR.InputGroup.MV = 1;
CSTR.InputGroup.UD = 2;
CSTR.OutputGroup.MO = 1;
CSTR.OutputGroup.UO = 2;
```

To view the properties of CSTR, enter:

```
CSTR
```

# Linearize Simulink Models

Generally, real systems are nonlinear. To design an MPC controller for a nonlinear system, you must model the plant in Simulink.

Although an MPC controller can regulate a nonlinear plant, the model used within the controller must be linear. In other words, the controller employs a linear approximation of the nonlinear plant. The accuracy of this approximation significantly affects controller performance.

To obtain such a linear approximation, you *linearize* the nonlinear plant at a specified *operating point*.

---

**Note** Simulink Control Design software must be installed to linearize nonlinear Simulink models.

---

You can linearize a Simulink model:

- From the command line.
- Using the Linear Analysis Tool.
- Using **MPC Designer**.

## Linearization Using MATLAB Code

This example shows how to obtain a linear model of a plant using a MATLAB script.

For this example the CSTR model, `CSTR_OpenLoop`, is linearized. The model inputs are the coolant temperature (manipulated variable of the MPC controller), limiting reactant concentration in the feed stream, and feed temperature. The model states are the temperature and concentration of the limiting reactant in the product stream. Both states are measured and used for feedback control.

### Obtain Steady-State Operating Point

The operating point defines the nominal conditions at which you linearize a model. It is usually a steady-state condition.

Suppose that you plan to operate the CSTR with the output concentration,  $C_A$ , at  $2 \text{ kmol}/\text{m}^3$ . The nominal feed concentration is  $10 \text{ kmol}/\text{m}^3$ , and the nominal feed

temperature is 300 K. Create an operating point specification object to define the steady-state conditions.

```
opspec = operspec('CSTR_OpenLoop');
opspec = addoutputspec(opspec, 'CSTR_OpenLoop/CSTR', 2);
opspec.Outputs(1).Known = true;
opspec.Outputs(1).y = 2;
```

```
op1 = findop('CSTR_OpenLoop', opspec);
```

```
Operating point search report:
```

```
-----
```

```
Operating point search report for the Model CSTR_OpenLoop.
(Time-Varying Components Evaluated at time t=0)
```

```
Operating point specifications were successfully met.
States:
```

```
-----
```

```
(1.) CSTR_OpenLoop/CSTR/C_A
     x:          2      dx:    -4.6e-12 (0)
(2.) CSTR_OpenLoop/CSTR/T_K
     x:        373      dx:    5.49e-11 (0)
```

```
Inputs:
```

```
-----
```

```
(1.) CSTR_OpenLoop/Coolant Temperature
     u:          299    [-Inf Inf]
```

```
Outputs:
```

```
-----
```

```
(1.) CSTR_OpenLoop/CSTR
     y:          2      (2)
```

The calculated operating point is  $C_A = 2 \text{ kmol/m}^3$  and  $T_K = 373$  K. Notice that the steady-state coolant temperature is also given as 299 K, which is the nominal value of the manipulated variable of the MPC controller.

To specify:

- Values of known inputs, use the `Input.Known` and `Input.u` fields of `opspec`
- Initial guesses for state values, use the `State.x` field of `opspec`

For example, the following code specifies the coolant temperature as 305 K and initial guess values of the C\_A and T\_K states before calculating the steady-state operating point:

```
opspec = operspec('CSTR_OpenLoop');
opspec.States(1).x = 1;
opspec.States(2).x = 400;
opspec.Inputs(1).Known = true;
opspec.Inputs(1).u = 305;

op2 = findop('CSTR_OpenLoop',opspec);
```

```
Operating point search report:
```

```
-----

Operating point search report for the Model CSTR_OpenLoop.
(Time-Varying Components Evaluated at time t=0)

Operating point specifications were successfully met.
States:
-----
(1.) CSTR_OpenLoop/CSTR/C_A
     x:          1.78   dx:      -1.6e-14 (0)
(2.) CSTR_OpenLoop/CSTR/T_K
     x:          377   dx:      1.99e-13 (0)

Inputs:
-----
(1.) CSTR_OpenLoop/Coolant Temperature
     u:          305

Outputs: None
-----
```

### Specify Linearization Inputs and Outputs

If the linearization input and output signals are already defined in the model, as in CSTR\_OpenLoop, then use the following to obtain the signal set.

```
io = getlinio('CSTR_OpenLoop');
```

Otherwise, specify the input and output signals as shown here.

```
io(1) = linio('CSTR_OpenLoop/Coolant Temperature',1,'input');
io(2) = linio('CSTR_OpenLoop/Feed Concentration',1,'input');
```

```
io(3) = linio('CSTR_OpenLoop/Feed Temperature',1,'input');
io(4) = linio('CSTR_OpenLoop/CSTR',1,'output');
io(5) = linio('CSTR_OpenLoop/CSTR',2,'output');
```

### Linearize Model

Linearize the model using the specified operating point, `op1`, and input/output signals, `io`.

```
sys = linearize('CSTR_OpenLoop',op1,io)
```

```
sys =
```

```
A =
```

	C_A	T_K
C_A	-5	-0.3427
T_K	47.68	2.785

```
B =
```

	Coolant Temp	Feed Concent	Feed Tempera
C_A	0	1	0
T_K	0.3	0	1

```
C =
```

	C_A	T_K
CSTR/1	0	1
CSTR/2	1	0

```
D =
```

	Coolant Temp	Feed Concent	Feed Tempera
CSTR/1	0	0	0
CSTR/2	0	0	0

Continuous-time state-space model.

## Linearization Using Linear Analysis Tool in Simulink Control Design

This example shows how to linearize a Simulink model using the Linear Analysis Tool, provided by the Simulink Control Design product.

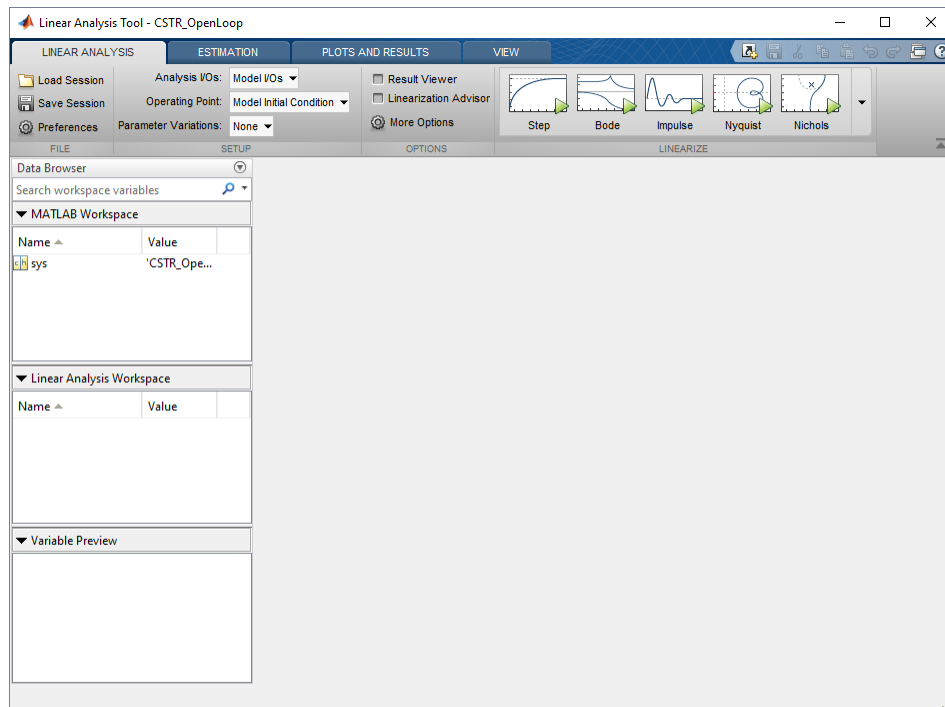
This example uses the CSTR model, `CSTR_OpenLoop`.

### Open Simulink Model

```
sys = 'CSTR_OpenLoop';  
open_system(sys)
```

### Open Linear Analysis Tool

In the Simulink model window, select **Analysis > Control Design > Linear Analysis**.



### Specify Linearization Inputs and Outputs

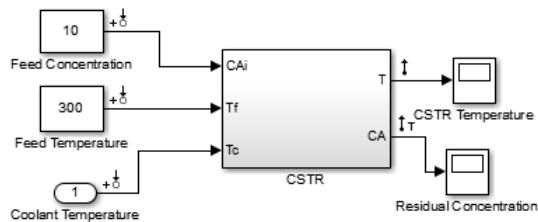
The linearization inputs and outputs are already specified for CSTR\_OpenLoop. The input signals correspond to the outputs from the Feed Concentration, Feed Temperature, and Coolant Temperature blocks. The output signals are the inputs to the CSTR Temperature and Residual Concentration blocks.

To specify a signal as a:

- Linearization input, right-click the signal in the Simulink model window and select **Linear Analysis Points > Input Perturbation**.
- Linearization output, right-click the signal in the Simulink model window and select **Linear Analysis Points > Output Measurement**.

### Specify Residual Concentration as Known Trim Constraint

In the Simulink model window, right-click the CA output signal from the CSTR block. Select **Linear Analysis Points > Trim Output Constraint**.

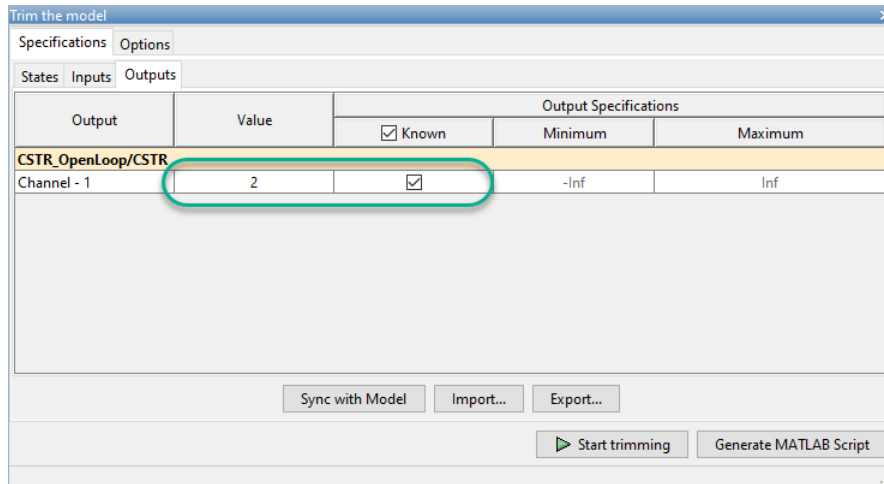


Copyright 1990-2012 The MathWorks, Inc.

In the Linear Analysis Tool, in the **Linear Analysis** tab, in the **Operating Point** drop-down list, select **Trim model**.

In the **Outputs** tab:

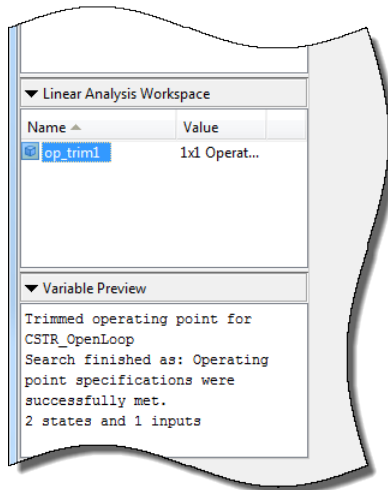
- Select the **Known** check box for Channel - 1 under **CSTR\_OpenLoop/CSTR**.
- Set the corresponding **Value** to 2 kmol/m<sup>3</sup>.



### Create and Verify Operating Point

In the Trim the model dialog box, click **Start trimming**.

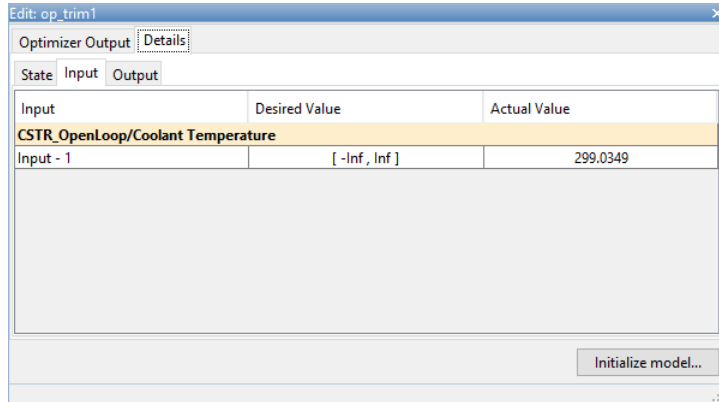
The operating point `op_trim1` displays in the **Linear Analysis Workspace**.



Double click `op_trim1` to view the resulting operating point.



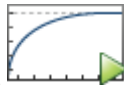
In the Edit dialog box, select the **Input** tab.



The coolant temperature at steady state is 299 K, as desired.

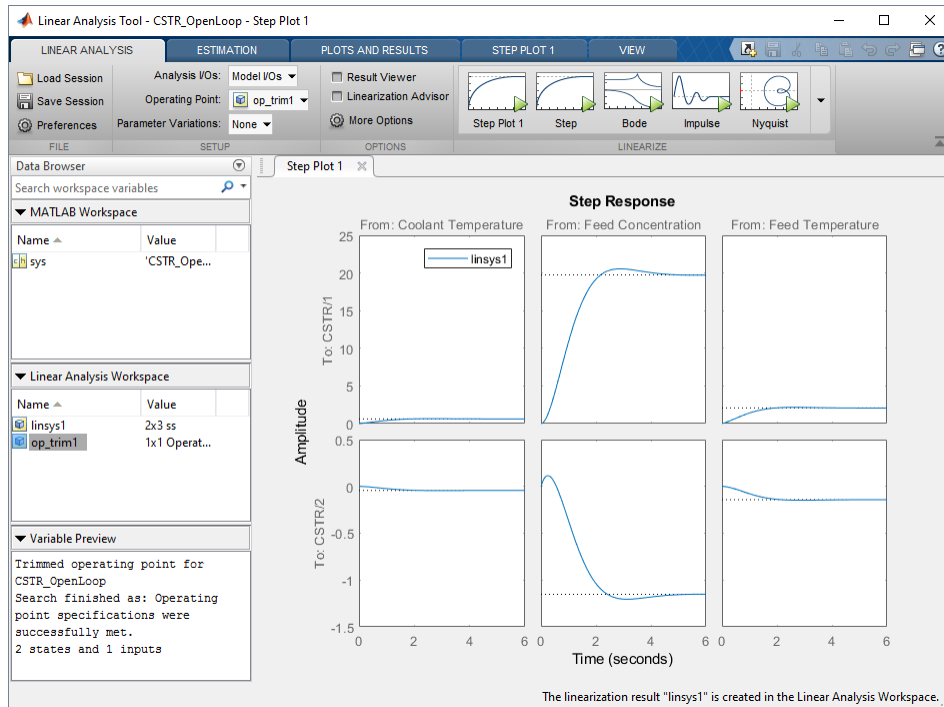
### Linearize Model

In the **Linear Analysis** tab, in the **Operating Point** drop-down list, select `op_trim1`.



Click **Step** to linearize the model.

This option creates the linear model `linsys1` in the **Linear Analysis Workspace** and generates a step response for this model. `linsys1` uses `op_trim1` as its operating point.



The step response from feed concentration to output CSTR/2 displays an interesting inverse response. An examination of the linear model shows that CSTR/2 is the residual CSTR concentration,  $C_A$ . When the feed concentration increases,  $C_A$  increases initially because more reactant is entering, which increases the reaction rate. This rate increase results in a higher reactor temperature (output CSTR/1), which further increases the reaction rate and  $C_A$  decreases dramatically.

### Export Linearization Result

If necessary, you can repeat any of these steps to improve your model performance. Once you are satisfied with your linearization result, in the Linear Analysis Tool, drag and drop it from the **Linear Analysis Workspace** to the **MATLAB Workspace**. You can now use your linear model to design an MPC controller.

## See Also

**Linear Analysis Tool** | `linearize`

## **Related Examples**

- “Design MPC Controller in Simulink” on page 5-2
- “Design Controller Using MPC Designer” on page 3-2
- “Design MPC Controller at the Command Line” on page 4-2

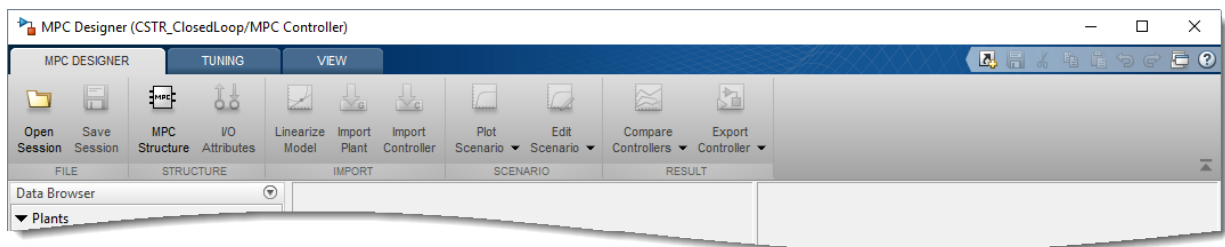
## Linearize Simulink Models Using MPC Designer

This topic shows how to linearize Simulink models using **MPC Designer**. To do so, open the app from a Simulink model that contains an MPC Controller block. For this example, use the CSTR\_ClosedLoop model.

```
sys = 'CSTR_ClosedLoop';
open_system(sys)
```

In the model window, double-click the MPC Controller block.

In the Block Parameters dialog box, ensure that the **MPC Controller** field is empty, and click **Design** to open **MPC Designer**.



Using **MPC Designer**, you can define the MPC structure by linearizing the Simulink model. After you define the initial MPC structure, you can also linearize the model at different operating points and import the linearized plants.

---

**Note** If a controller from the MATLAB workspace is specified in the **MPC Controller** field, the app imports the specified controller. In this case, the MPC structure is derived from the imported controller. In this case, you can still linearize the Simulink model and import the linearized plants.

---

### Define MPC Structure By Linearization

This example shows how to define the plant input/output structure in **MPC Designer** by linearizing a Simulink model.

On the **MPC Designer** tab, in the **Structure** section, click **MPC Structure**.

Define MPC Structure By Linearization

**MPC Structure**

**Controller Sample Time**

Specify MPC controller sample time (default sample time in the MPC block):

**Simulink Operating Point**

Choose an operating point at which plant model is linearized and nominal values are computed: Model Initial Condition

**Simulink Signals for Plant Inputs**

Selected	Type	Block Path
<input checked="" type="radio"/>	Manipulated Variables (MV)	<a href="#">CSTR_ClosedLoop/MPC Controller:1</a>

Select Signals

**Simulink Signals for Plant Outputs**

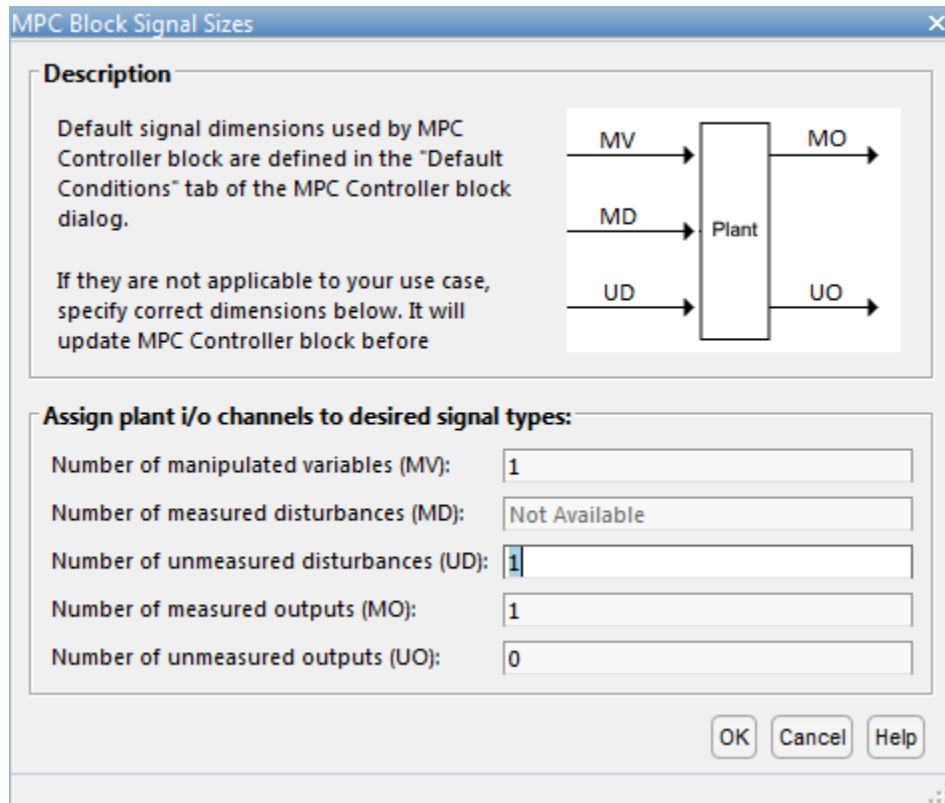
Selected	Type	Block Path
<input checked="" type="radio"/>	Measured Outputs (MO)	<a href="#">CSTR_ClosedLoop/CSTR:2</a>

Select Signals

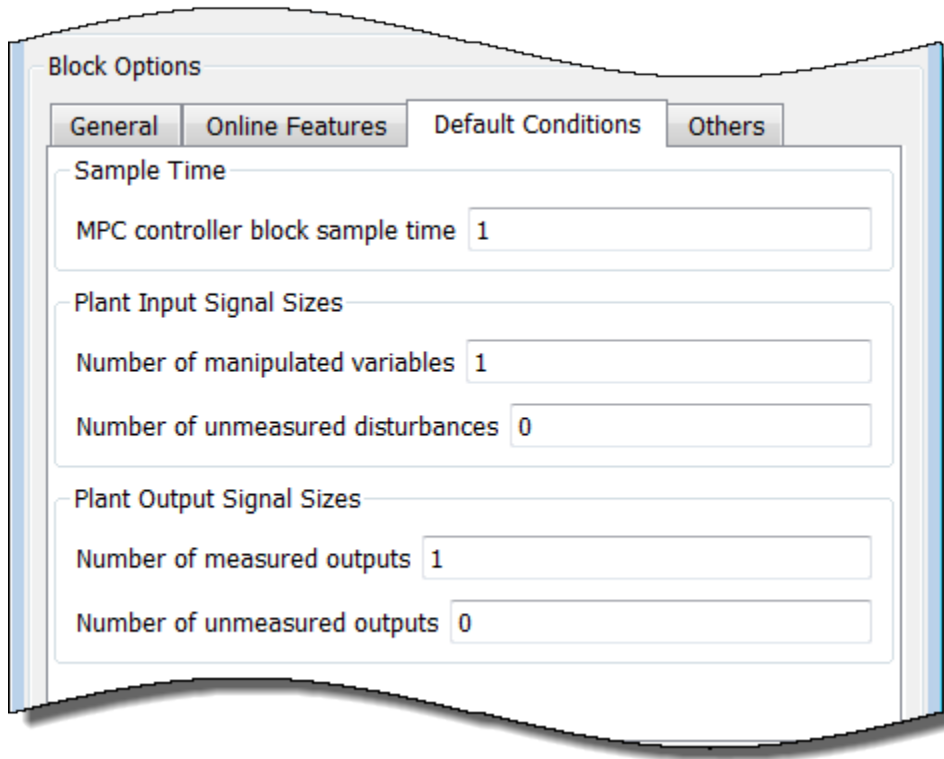
Define and Linearize Cancel Help

## Specify Signal Dimensions

In the Define MPC Structure By Linearization dialog box, in the **MPC Structure** section, if the displayed signal dimensions do not match your model, click **Change I/O Sizes** to configure the dimensions. Any unmeasured disturbances or unmeasured outputs in your model are not detected by the MPC Controller block. Specify the dimensions for these signals.



**Tip** In the MPC Controller Block Parameters dialog box, in the **Default Conditions** tab, you can define the controller sample time and signal dimensions before opening **MPC Designer**.

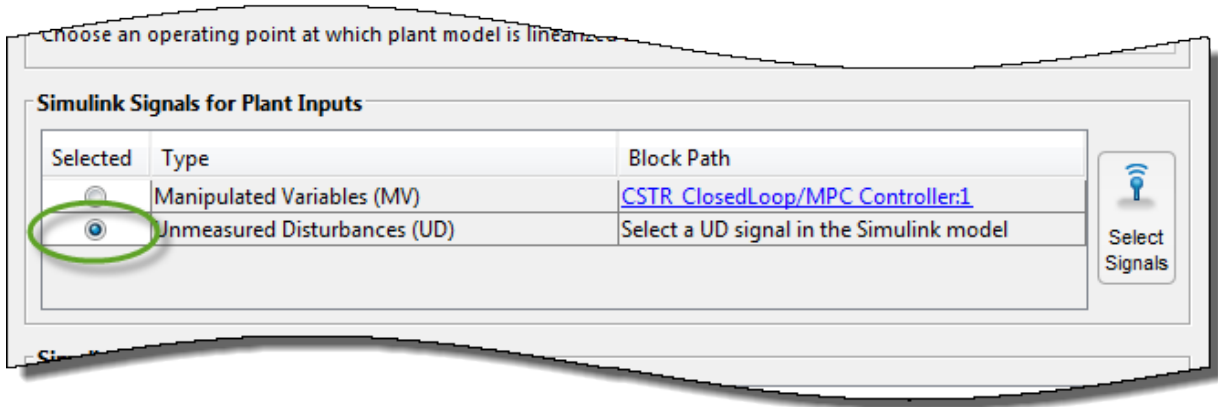


### Select Plant Input/Output Signals

Before linearizing the model, assign Simulink signal lines to each MPC signal type in your model. The app uses these signals as linearization inputs and outputs.

In the **Simulink Signals for Plant Inputs** and **Simulink Signals for Plant Outputs** sections, the **Block Path** is automatically defined for manipulated variables, measured outputs, and measured disturbances. **MPC Designer** detects these signals since they are connected to the MPC Controller block. If your application has unmeasured disturbances or unmeasured outputs, select their corresponding Simulink signal lines.

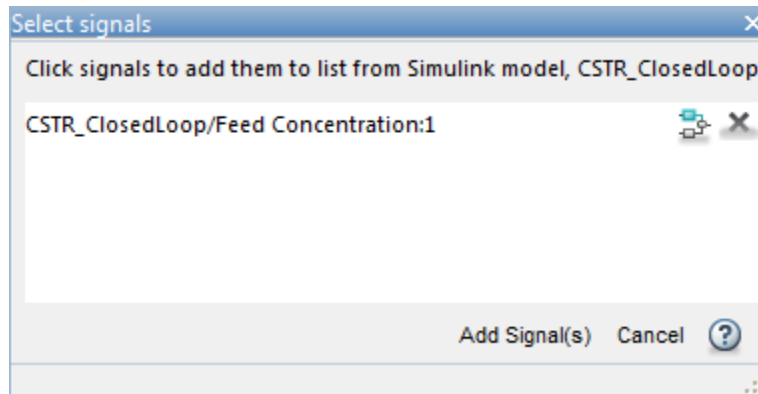
To choose a signal type, use the **Selected** option buttons.



Click **Select Signals**.

In the Simulink model window, click the signal line corresponding to the selected signal type.

The signal is highlighted, and its block path is added to the Select signals dialog box.



In the Select signals dialog box, click **Add Signal(s)**.

In the Define MPC Structure By Linearization dialog box, the **Block Path** for the selected signal type updates.



---

**Note** If your model has measured disturbances, you must connect the corresponding plant inputs to the signal line connected to the md port of the MPC Controller block. For more information, see “Connect Measured Disturbances for Linearization” on page 2-50.

---

### Specify Operating Point

In the **Simulink Operating Point** section, in the drop-down list, select an operating point at which to linearize the model.

For information on the different operating point options, see “Specifying Operating Points” on page 2-39.

---

**Note** If you select an option that generates multiple operating points for linearization, **MPC Designer** uses only the first operating point to define the plant structure and linearize the model.

---

### Define Structure and Linearize Model

Click **Define and Linearize**.

The app linearizes the Simulink model at the specified operating point using the specified input/output signals, and adds the linearized plant to the **Data Browser**.

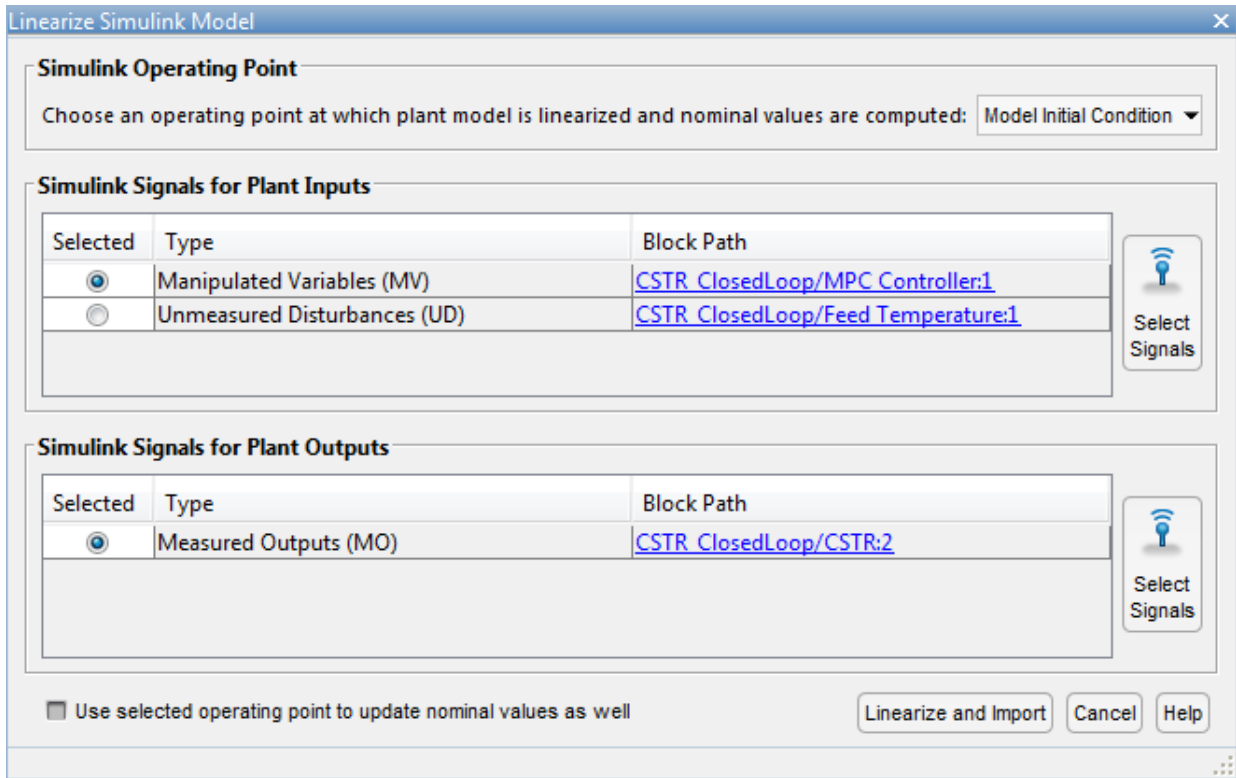
Also, a default controller, which uses the linearized plant as its internal model, and a default simulation scenario are created.

**MPC Designer** uses the input/output signal values at the selected operating point as nominal values.

### Linearize Model

After you define the initial MPC structure, you can linearize the Simulink model at different operating points and import the linearized plants. Doing so is useful for validating controller performance against modeling errors.

On the **MPC Designer** tab, in the **Import** section, click **Linearize Model**.



### Select Plant Input/Output Signals

In the **Simulink Signals for Plant Inputs** and **Simulink Signals for Plant Outputs** sections, the input/output signal configuration is the same as you specified when initially defining the MPC structure.

You cannot change the signal types and dimensions once the structure has been defined. However, for each signal type, you can select different signal lines from your Simulink model. The selected lines must have the same dimensions as defined in the current MPC structure.

### Specify Operating Point

In the **Simulink Operating Point** section, in the drop-down list, select the operating points at which to linearize the model.

For information on the different operating point options, see “Specifying Operating Points” on page 2-39.

### **Linearize Model and Import Plant**

Click **Linearize and Import**.

**MPC Designer** linearizes the Simulink model at the defined operating point using the specified input/output signals, and adds the linearized plant to the **Data Browser**.

If you select the **Use selected operating point to update nominal values as well** option, the app updates the controller nominal values using the operating point signal values.

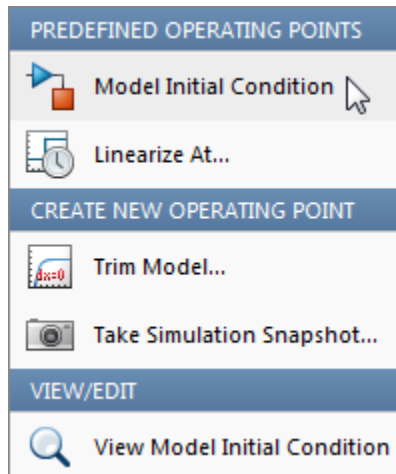
If you select an option that generates multiple operating points for linearization, the app linearizes the model at all the specified operating points. The linearized plants are added to the **Data Browser** in the same order in which their corresponding operating points are defined. If you choose to update the nominal values, the app uses the signal values from the first operating point.

## **Specifying Operating Points**

In the **Simulink Operating Point** section, in the drop-down list, you can select or create operating points for model linearization. For more information on finding steady-state operating points, see “About Operating Points” (Simulink Control Design) and “Compute Steady-State Operating Points” (Simulink Control Design).

### **Select Model Initial Condition**

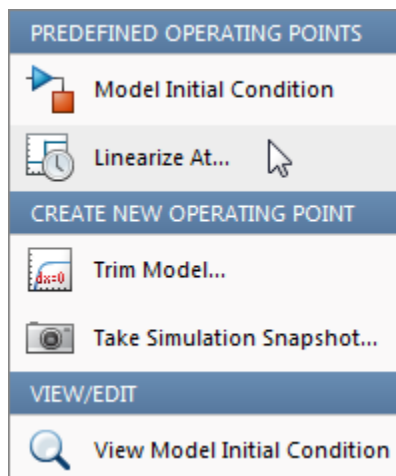
To linearize the model using the initial conditions specified in the Simulink model as the operating point, select **Model Initial Condition**.



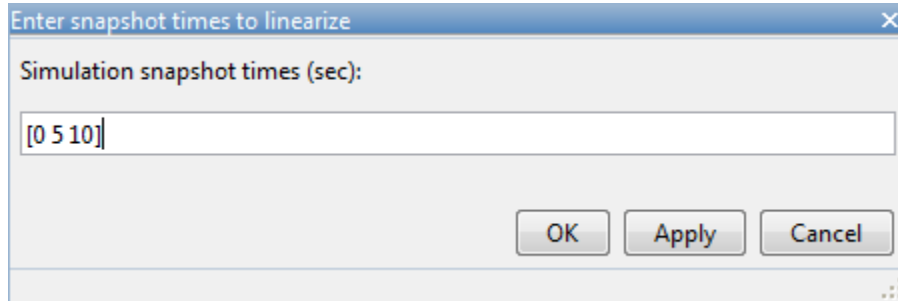
The model initial condition is the default operating point for linearization in **MPC Designer**.

### Linearize at Simulation Snapshot Times

To linearize the model at specified simulation snapshot times, select **Linearize At**. Linearizing at snapshot times is useful when you know that your model reaches an equilibrium state after a certain simulation time.



In the Enter snapshot times to linearize dialog box, in the **Simulation snapshot times** field, enter one or more simulation snapshot times. Enter multiple snapshot times as a vector.



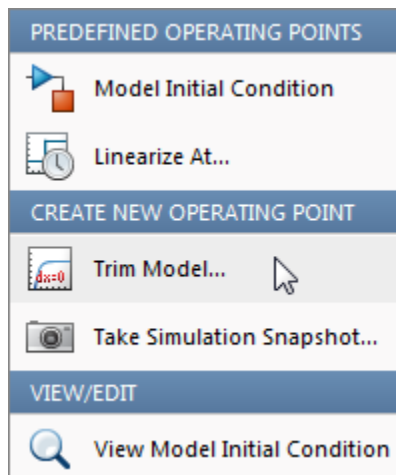
Click **OK**.

If you enter multiple snapshot times, and you selected **Linearize At** from the:

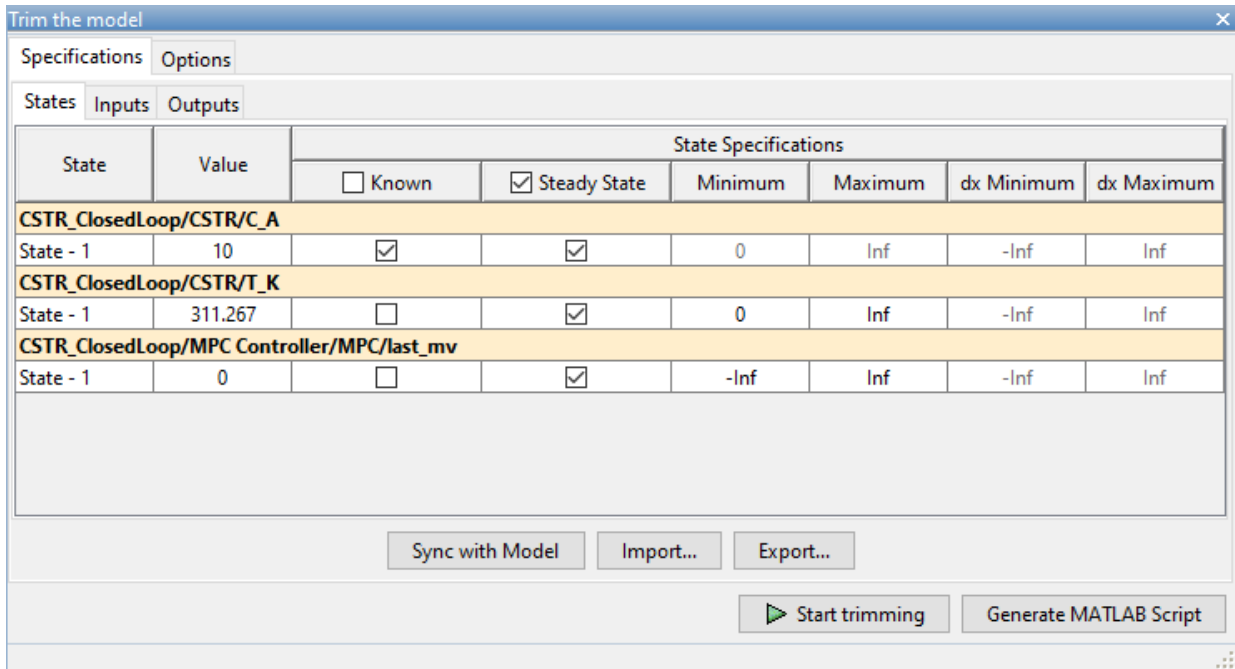
- Define MPC Structure By Linearization dialog box, **MPC Designer** linearizes the model using only the first snapshot time. The nominal values of the MPC controller are defined using the input/output signal values for this snapshot.
- Linearize Simulink Model dialog box, **MPC Designer** linearizes the model at all the specified snapshot times. The linearized plant models are added to the **Data Browser** in the order specified in the snapshot time array. If you selected the **Use selected operating point to update nominal values as well** option, the nominal values are set using the input/output signal values from the first snapshot.

### Compute Steady-State Operating Point

To compute a steady-state operating point using numerical optimization methods to meet your specifications, select **Trim Model**.



In the Trim the model dialog box, enter the specifications for the steady-state values at which you want to find an operating point. You can specify values for states, input signals, and output signals.



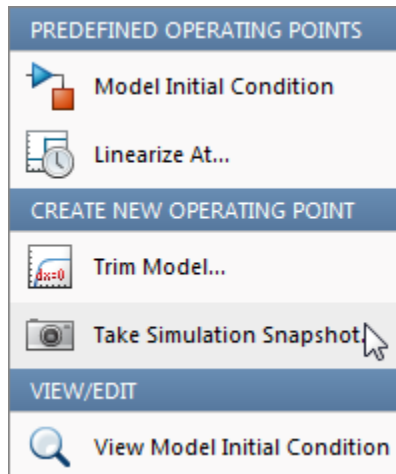
Click **Start Trimming**.

**MPC Designer** creates an operating point for the given specifications. The computed operating point is added to the **Simulink Operating Point** drop-down list and is selected.

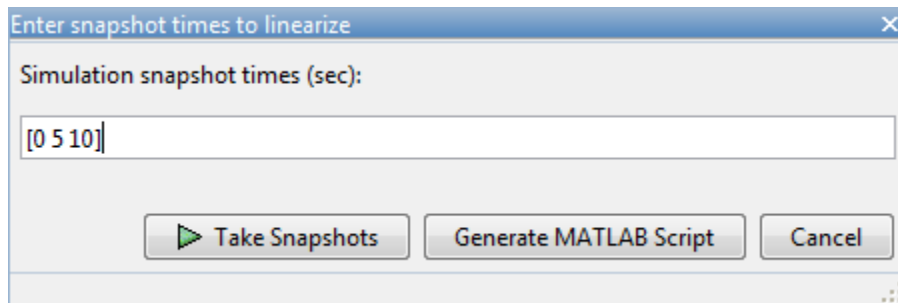
For examples showing how to specify the conditions for a steady-state operating point search, see “Compute Steady-State Operating Point from State Specifications” (Simulink Control Design) and “Compute Steady-State Operating Point from Output Specifications” (Simulink Control Design).

### Compute Operating Point at Simulation Snapshot Time

To compute operating points using simulation snapshots, select **Take Simulation Snapshot**. Linearizing the model using operating points computed from simulation snapshots is useful when you know that your model reaches an equilibrium state after a certain simulation time.



In the Enter snapshot times to linearize dialog box, in the **Simulation snapshot times** field, enter one or more simulation snapshot times. Enter multiple snapshot times as a vector.



Click **Take Snapshots**.

**MPC Designer** simulates the Simulink model. At each snapshot time, the current state of the model is used to create an operating point, which is added to the drop-down list and selected.

If you entered multiple snapshot times, the operating points are stored together as an array. If you selected **Take Simulation Snapshot** from the:

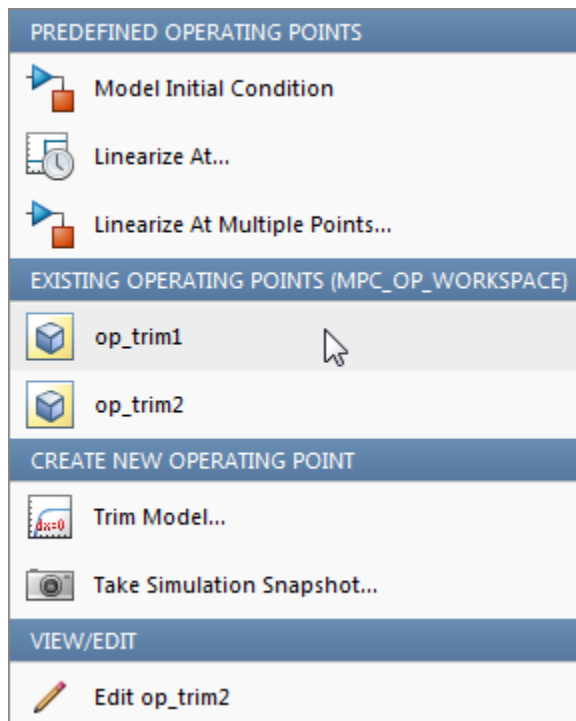


- Define MPC Structure By Linearization dialog box, **MPC Designer** linearizes the model using only the first operating point in the array. The nominal values of the MPC controller are defined using the input/output signal values for this operating point.
- Linearize Simulink Model dialog box, **MPC Designer** linearizes the model at all the operating points in the array. The linearized plant models are added to the **Data Browser** in the same order as the operating point array.

In **MPC Designer**, the **Linearize At** and **Take Simulation Snapshot** options generally produce the same linearized plant and nominal signal values. However, since the **Take Simulation Snapshot** option first computes an operating point from the snapshot before linearization, the results can differ.

### Select Existing Operating Point

Under **Existing Operating Points**, select a previously defined operating point at which to linearize the Simulink model. This option is available if one or more previously created operating points are available in the drop-down list.

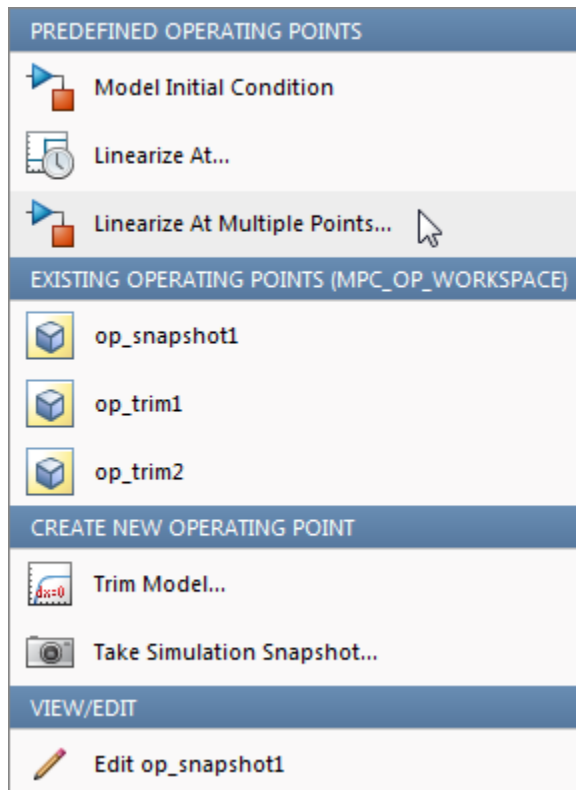


If the selected operating point represents an operating point array created using multiple snapshot times, and you selected an operating point from the:

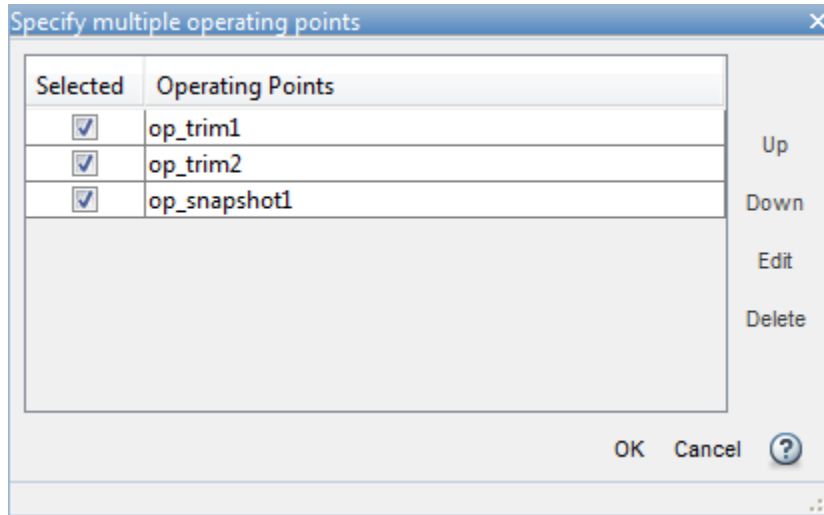
- Define MPC Structure By Linearization dialog box, **MPC Designer** linearizes the model using only the first operating point in the array. The nominal values of the MPC controller are defined using the input/output signal values for this operating point.
- Linearize Simulink Model dialog box, **MPC Designer** linearizes the model at all the operating points in the array. The linearized plant models are added to the **Data Browser** in the same order as the operating point array.

### Select Multiple Operating Points

To linearize the Simulink model at multiple existing operating points, select **Linearize at Multiple Points**. This option is available if there are more than one previously created operating points in the drop-down list.



In the Specify multiple operating points dialog box, select the operating points at which to linearize the model.



To change the operating point order, click an operating point in the list and click **Up** or **Down** to move the highlighted operating point within the list.

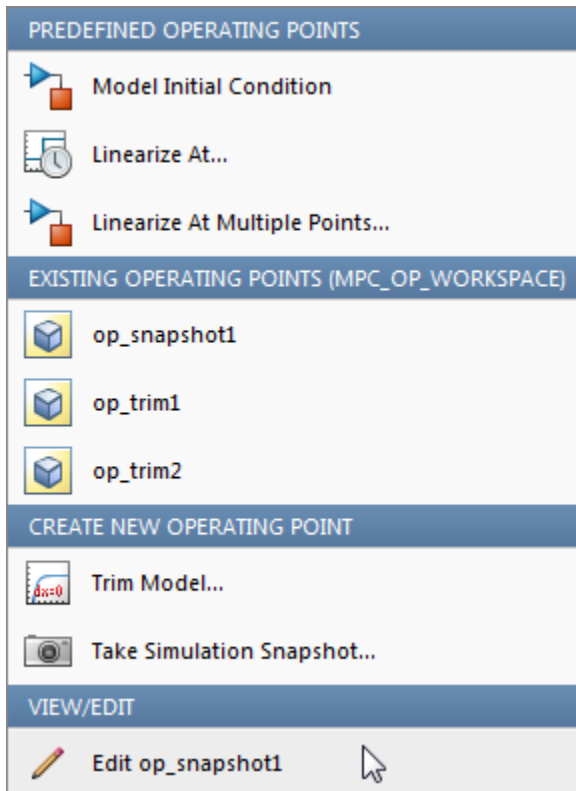
Click **OK**.

If you selected **Linearize at Multiple Points** from the:

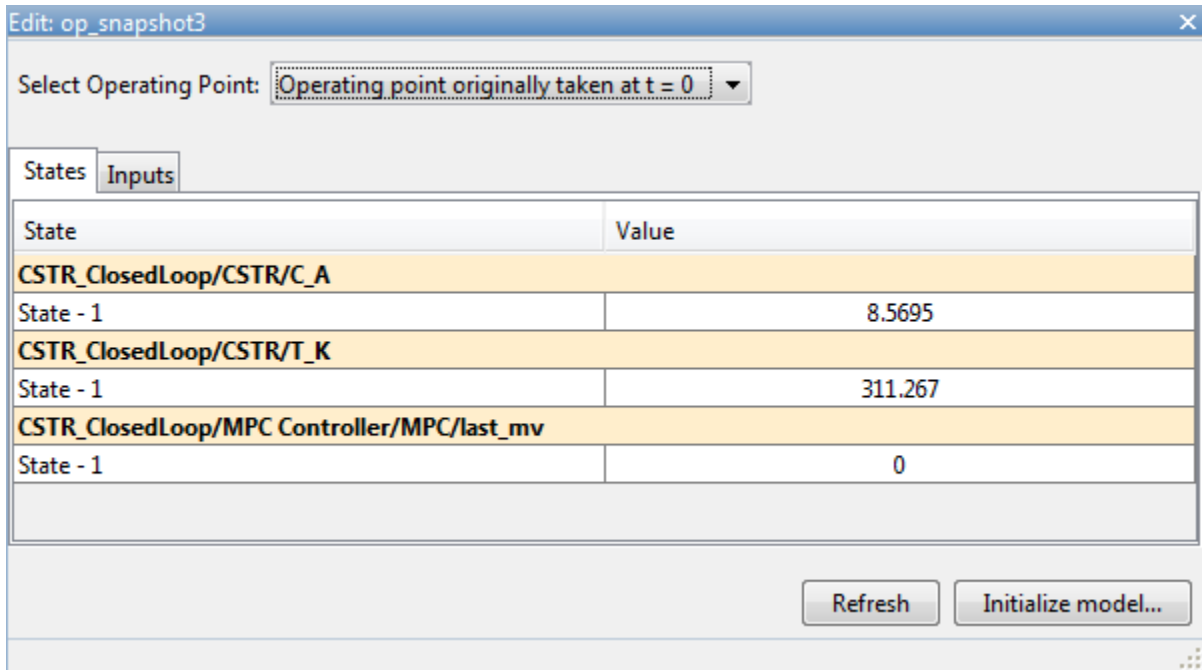
- Define MPC Structure By Linearization dialog box, **MPC Designer** linearizes the model using only the first specified operating point. The nominal values of the MPC controller are defined using the input/output signal values for this operating point.
- Linearize Simulink Model dialog box, **MPC Designer** linearizes the model at all the specified operating points. The linearized plant models are added to the **Data Browser** in the order specified in the Specify multiple operating points dialog box.

### View/Edit Operating Point

To view or edit the selected operating point, under **View/Edit**, click the **Edit** option.



In the Edit dialog box, if you created the selected operating point from a simulation snapshot, you can edit the operating point values.



If the selected operating point represents an operating point array, in the **Select Operating Point** drop-down list, select an operating point to view.

If you obtained the operating point by trimming the model, you can only view the operating point values.

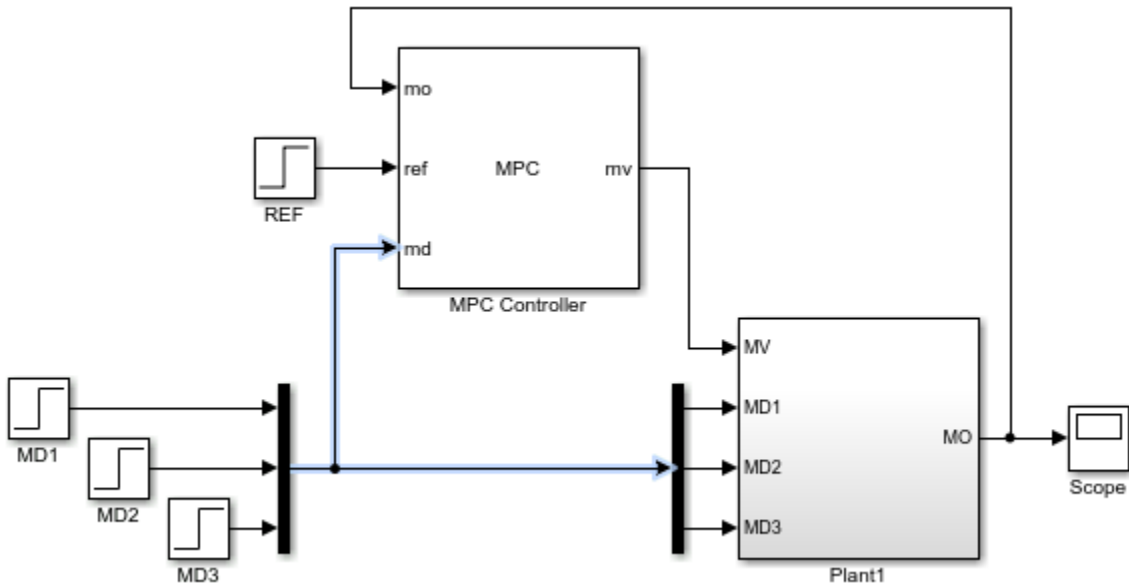
State	Desired Value	Actual Value	Desired dx	Actual dx
<b>CSTR_ClosedLoop/CSTR/C_A</b>				
State - 1	10	10	0	-3.5732e-08
<b>CSTR_ClosedLoop/CSTR/T_K</b>				
State - 1	[ 0 , Inf ]	161.9721	0	4.3283e-07
<b>CSTR_ClosedLoop/MPC Controller/MPC/last_mv</b>				
State - 1	[ -Inf , Inf ]	-298.1209	0	0

To set the Simulink model initial conditions to the states in the operating point, click **Initialize model**. You can then simulate the model at the specified operating point.

## Connect Measured Disturbances for Linearization

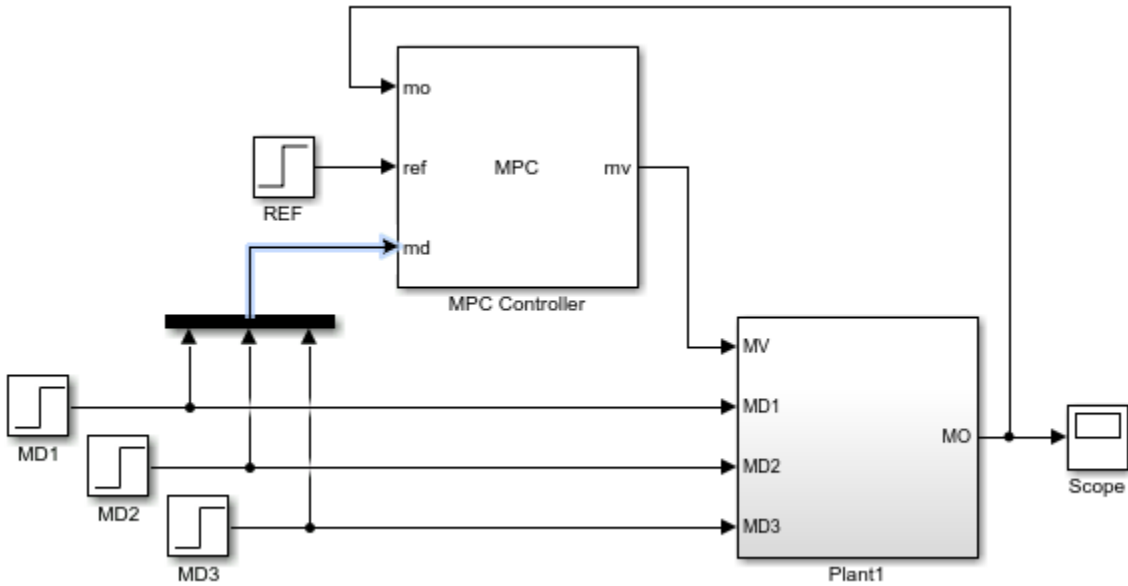
If your Simulink model has measured disturbance signals, connect them to the corresponding plant input ports and to the `md` port of the MPC Controller block. If you have multiple measured disturbances, connect them to the MPC Controller using a vector signal. As discussed in “Define MPC Structure By Linearization” on page 2-32, **MPC Designer** automatically detects the measured disturbances connected to the MPC Controller block and sets them as plant inputs for linearization.

Since the measured disturbances connected to the `md` port are selected as linearization inputs, you must connect the plant measured disturbance input ports to the selected signal line, as shown in the following:



### Correct MD Connection

If you connect the plant measured disturbance input ports to the corresponding signals before the Mux block, as shown in the following, there is no linearization path from the signals at the md port to the plant. As a result, when you linearize the plant using **MPC Designer**, the measured disturbance channels linearize to zero.



**Incorrect MD Connection**

**See Also**

MPC Designer

**Related Examples**

- “Linearize Simulink Models” on page 2-22
- “Design MPC Controller in Simulink” on page 5-2



## Identify Plant from Data

When designing a model predictive controller, you can specify the internal predictive plant model using a linear identified model. You use System Identification Toolbox software to estimate a linear plant model in one of these forms:

- State-space model — `idss`
- Transfer function model — `idtf`
- Polynomial model — `idpoly`
- Process model — `idproc`
- Grey-box model — `idgrey`

You can estimate the plant model programmatically at the command line or interactively using the **System Identification** app.

### Identify Plant from Data at the Command Line

This example shows how to identify a plant model at the command line. For information on identifying models using the System Identification app, see “Identify Linear Models Using System Identification App” (System Identification Toolbox).

Load the measured input/output data.

```
load plantIO
```

This command imports the plant input signal, `u`, plant output signal, `y`, and sample time, `Ts` to the MATLAB® workspace.

Create an `iddata` object from the input and output data.

```
mydata = iddata(y,u,Ts);
```

You can optionally assign channel names and units for the input and output signals.

```
mydata.InputName = 'Voltage';  
mydata.InputUnit = 'V';  
mydata.OutputName = 'Position';  
mydata.OutputUnit = 'cm';
```

Typically, you must preprocess identification I/O data before estimating a model. For this example, remove the offsets from the input and output signals by detrending the data.

```
mydatad = detrend(mydata);
```

You can also remove offsets by creating an `ssestOptions` object and specifying the `InputOffset` and `OutputOffset` options.

For this example, estimate a second-order, linear state-space model using the detrended data. To estimate a discrete-time model, specify the sample time as `Ts`.

```
ssl = ssest(mydatad,2,'Ts',Ts)
```

```
ssl =
```

```
Discrete-time identified state-space model:
```

```
   $x(t+T_s) = A x(t) + B u(t) + K e(t)$ 
```

```
   $y(t) = C x(t) + D u(t) + e(t)$ 
```

```
A =
```

```
          x1      x2
x1  0.8942  -0.1575
x2  0.1961   0.7616
```

```
B =
```

```
      Voltage
x1  6.008e-05
x2  -0.01219
```

```
C =
```

```
          x1      x2
Position  38.24  -0.3835
```

```
D =
```

```
      Voltage
Position      0
```

```
K =
```

```
      Position
x1  0.03572
x2  0.0223
```

```
Sample time: 0.1 seconds
```

```
Parameterization:
```

```
  FREE form (all coefficients in A, B, C free).
```

```
  Feedthrough: none
```

```
  Disturbance component: estimate
```

```
  Number of free coefficients: 10
```

Use "idssdata", "getpvec", "getcov" for parameters and their uncertainties.

```
Status:
Estimated using SSEST on time domain data "mydatad".
Fit to estimation data: 89.85% (prediction focus)
FPE: 0.0156, MSE: 0.01541
```

You can use this identified plant as the internal prediction model for your MPC controller. When you do so, the controller converts the identified model to a discrete-time, state-space model.

By default, the MPC controller discards any unmeasured noise components from your identified model. To configure noise channels as unmeasured disturbances, you must first create an augmented state-space model from your identified model. For example:

```
ss2 = ss(ss1, 'augmented')

ss2 =

A =
           x1          x2
x1  0.8942  -0.1575
x2  0.1961   0.7616

B =
      Voltage  v@Position
x1  6.008e-05   0.004448
x2  -0.01219   0.002777

C =
           x1          x2
Position  38.24  -0.3835

D =
      Voltage  v@Position
Position      0    0.1245

Input groups:
  Name      Channels
Measured    1
Noise      2

Sample time: 0.1 seconds
Discrete-time state-space model.
```

This command creates a state-space model, `ss2`, with two input groups, `Measured` and `Noise`, for the measured and noise inputs respectively. When you import the augmented model into your MPC controller, channels in the `Noise` input group are defined as unmeasured disturbances.

### Working with Impulse-Response Models

You can use System Identification Toolbox software to estimate finite step-response or finite impulse-response (FIR) plant models using measured data. Such models, also known as nonparametric models, are easy to determine from plant data ([1] and [2]) and have intuitive appeal.

Use the `impulseest` function to estimate an FIR model from measured data. This function generates the FIR coefficients encapsulated as an `idtf` object; that is, a transfer function model with only numerator coefficients. `impulseest` is especially effective in situations where the input signal used for identification has low excitation levels. To design a model predictive controller for this plant, you can convert the identified FIR plant model to a numeric LTI model. However, this conversion usually yields a high-order plant, which can degrade the controller design. For example, the numerical precision issues with high-order plants can affect estimator design. This result is particularly an issue for MIMO systems.

Model predictive controllers work best with low-order parametric models. Therefore, to design a model predictive controller using measured plant data, you can:

- Estimate a low-order parametric model using a parametric estimator, such as `ssest`.
- Initially identify a nonparametric model using `impulseest`, and then estimate a low-order parametric model from the response of the nonparametric model. For an example, see [3].
- Initially identify a nonparametric model using `impulseest`, and then convert the FIR model to a state-space model using `idss`. You can then reduce the order of the state-space model using `balred`. This approach is similar to the method used by `ssregest`.

### References

- [1] Cutler, C., and F. Yocum, "Experience with the DMC inverse for identification," *Chemical Process Control — CPC IV* (Y. Arkun and W. H. Ray, eds.), CACHE, 1991.

- [2] Ricker, N. L., "The use of bias least-squares estimators for parameters in discrete-time pulse response models," *Ind. Eng. Chem. Res.*, Vol. 27, pp. 343, 1988.
- [3] Wang, L., P. Gawthrop, C. Chessari, T. Podsiadly, and A. Giles, "Indirect approach to continuous time system identification of food extruder," *J. Process Control*, Vol. 14, Number 6, pp. 603-615, 2004.

## See Also

### Apps

#### System Identification

### Functions

detrend | iddata | ssest

## More About

- "Handling Offsets and Trends in Data" (System Identification Toolbox)
- "Identify Linear Models Using System Identification App" (System Identification Toolbox)
- "Design MPC Controller for Identified Plant Model" on page 2-58

## Design MPC Controller for Identified Plant Model

You can define the internal plant model of your model predictive controller using a linear model identified while using System Identification Toolbox software. You can identify the plant model and design the MPC controller interactively using apps or programmatically at the command line. For more information on identifying plant models, see “Identify Plant from Data” on page 2-53.

### Design Controller for Identified Plant Using Apps

This example shows how to interactively design a model predictive controller using an identified plant model. First, estimate the plant model from data using the **System Identification** app. Then design an MPC controller by importing the identified plant into **MPC Designer**.

#### Load Input/Output Data

Load the input and output data for identification.

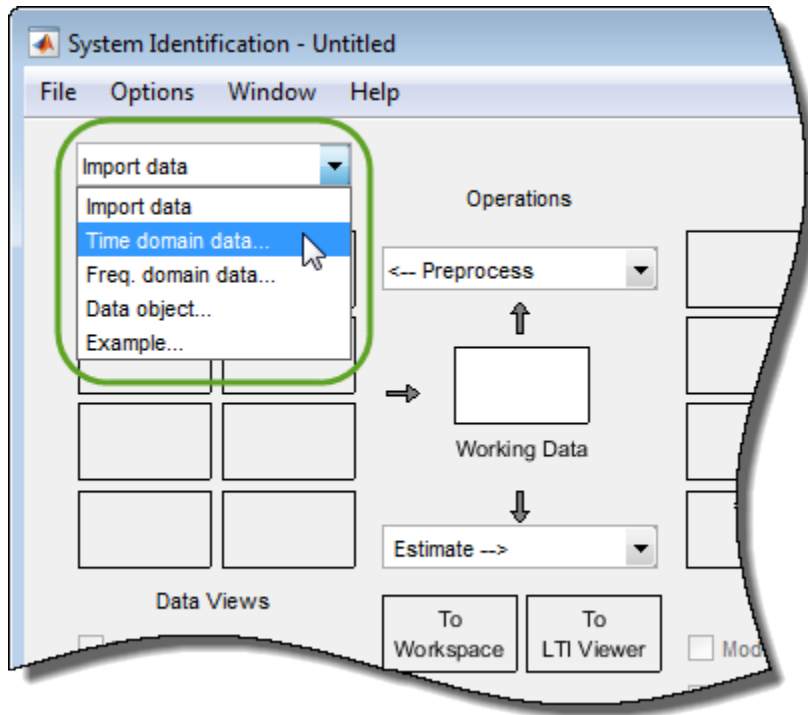
```
load(fullfile(matlabroot, 'examples', 'mpc', 'plantIO'))
```

This command imports the plant input signal, *u*, output signal, *y*, and sample time, *Ts*, to the MATLAB workspace.

Open the **System Identification** app.

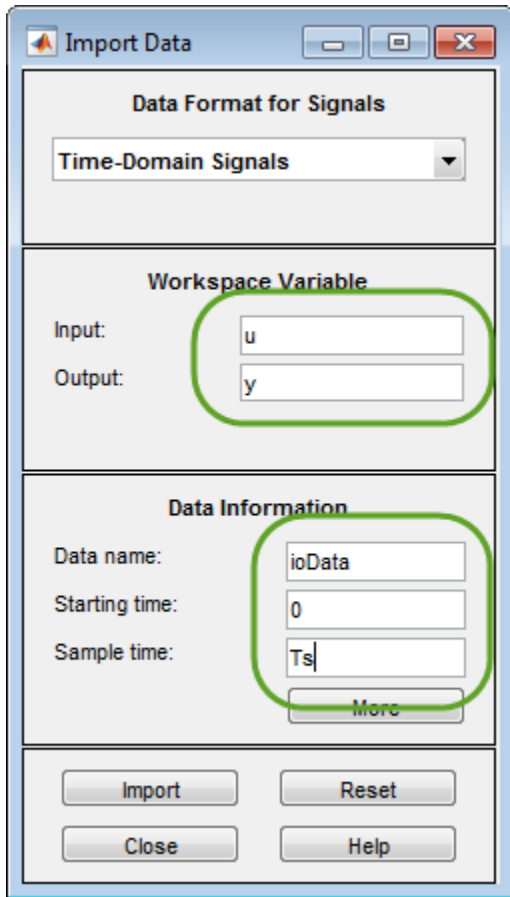
```
systemIdentification
```

In the **System Identification** app, under **Import data**, select Time domain data.



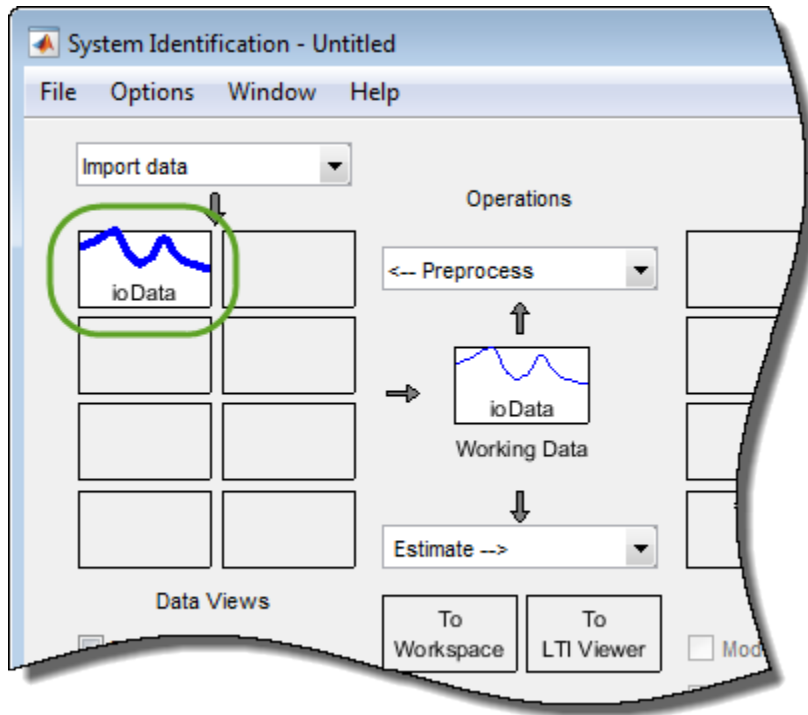
In the Import Data dialog box, specify the **Input**, **Output**, and **Sample time** using the data from the MATLAB workspace.

Also, specify the **Data name** as ioData and **Starting time** as 0.



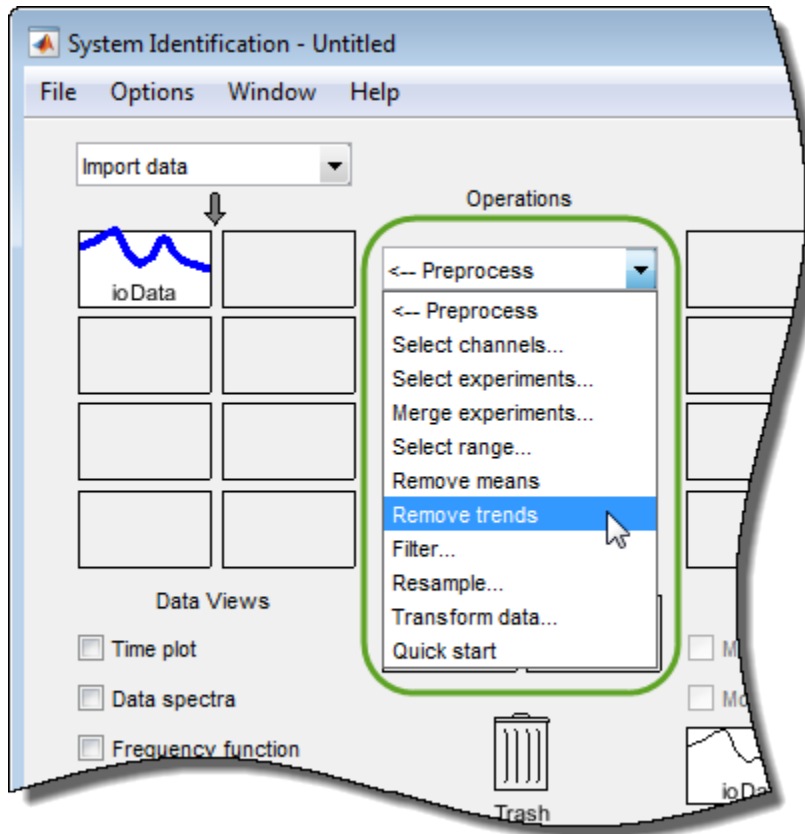
Click **Import**. The app imports the data, creates an `iddata` object with the specified name and signal properties, and adds this object to the **Data Views** area.





### Preprocess Data

Typically, you must preprocess identification I/O data before estimating a model. For this example, remove the offsets from the input and output signals by detrending the data. In the **System Identification** app, under **Preprocess**, select Remove trends.

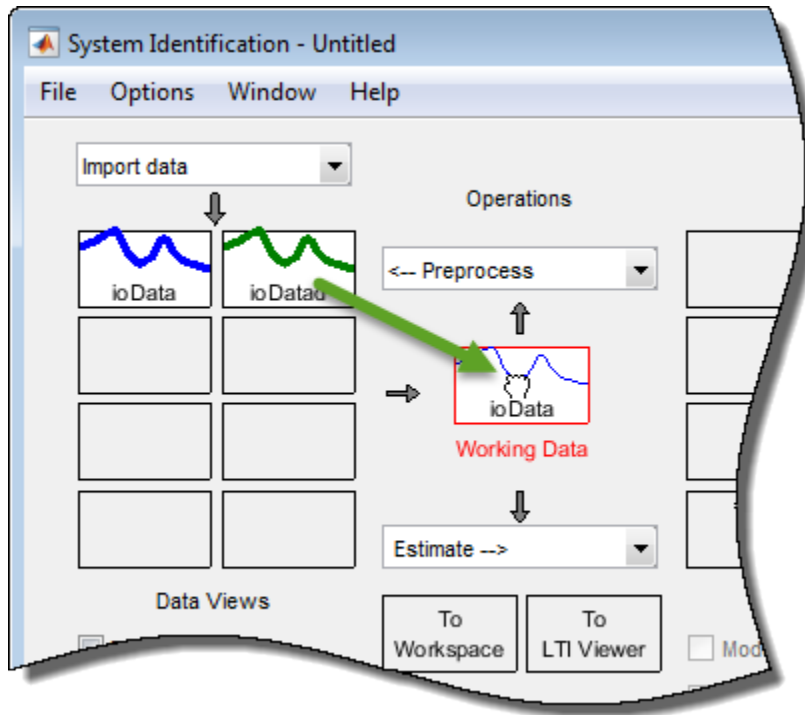


The app creates a data object, `ioData`, using the preprocessed data, and adds this object to the **Data Views** area.

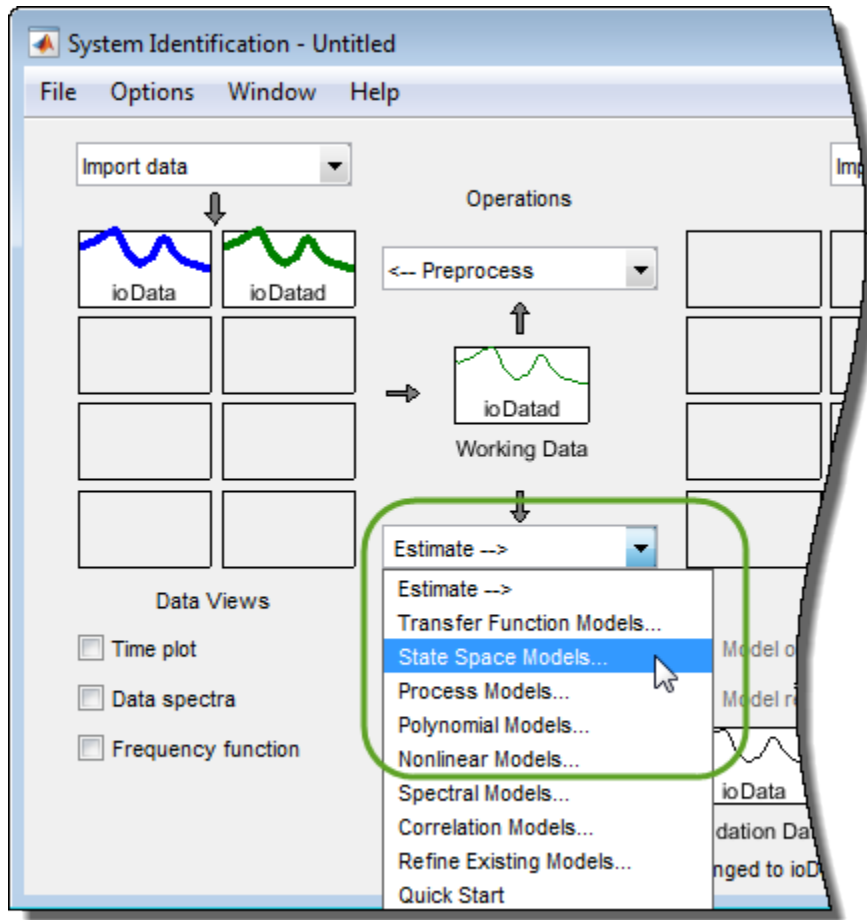
For more information on preprocessing identification data, see “Preprocess Data” (System Identification Toolbox).

### Estimate Linear Model

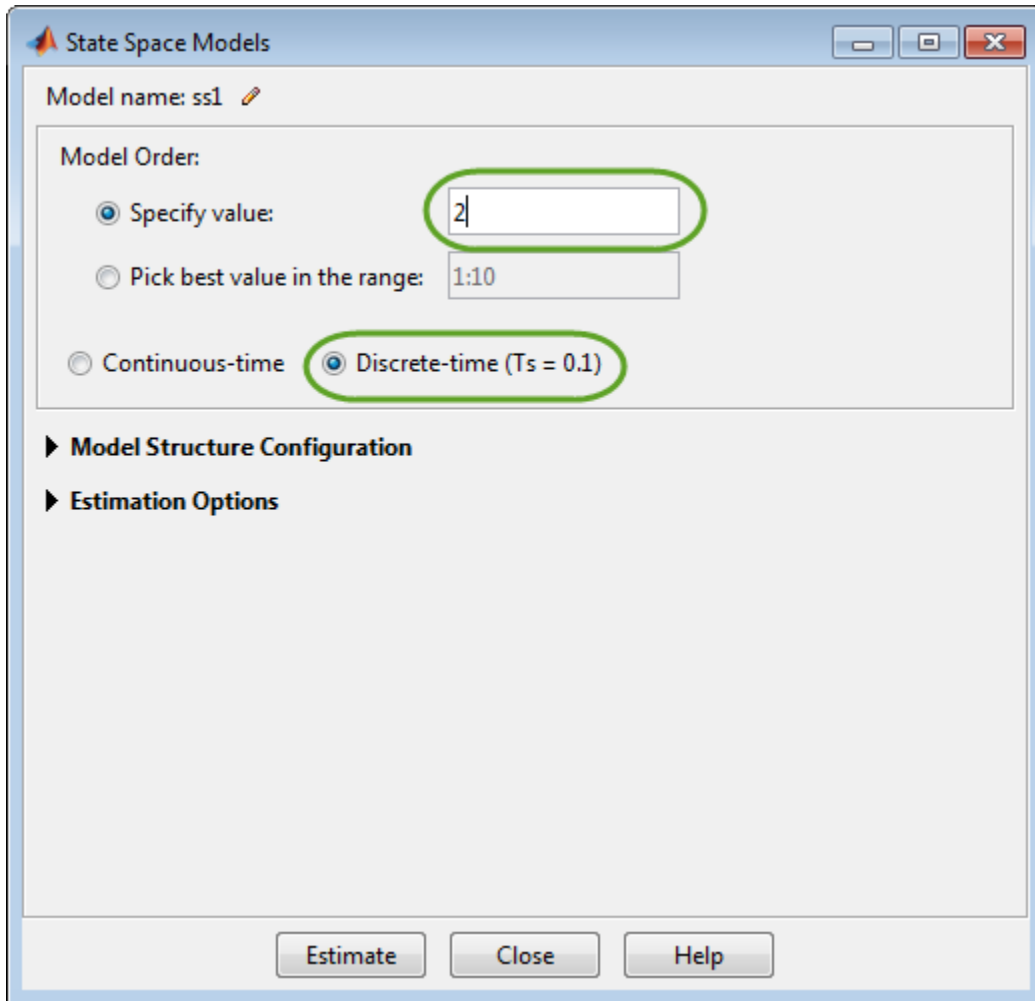
To use the detrended data, `ioData`, for model estimation, first drag the corresponding data object from the **Data Views** area to **Working Data**.



To estimate a state-space model, under **Estimate**, select State Space Models.

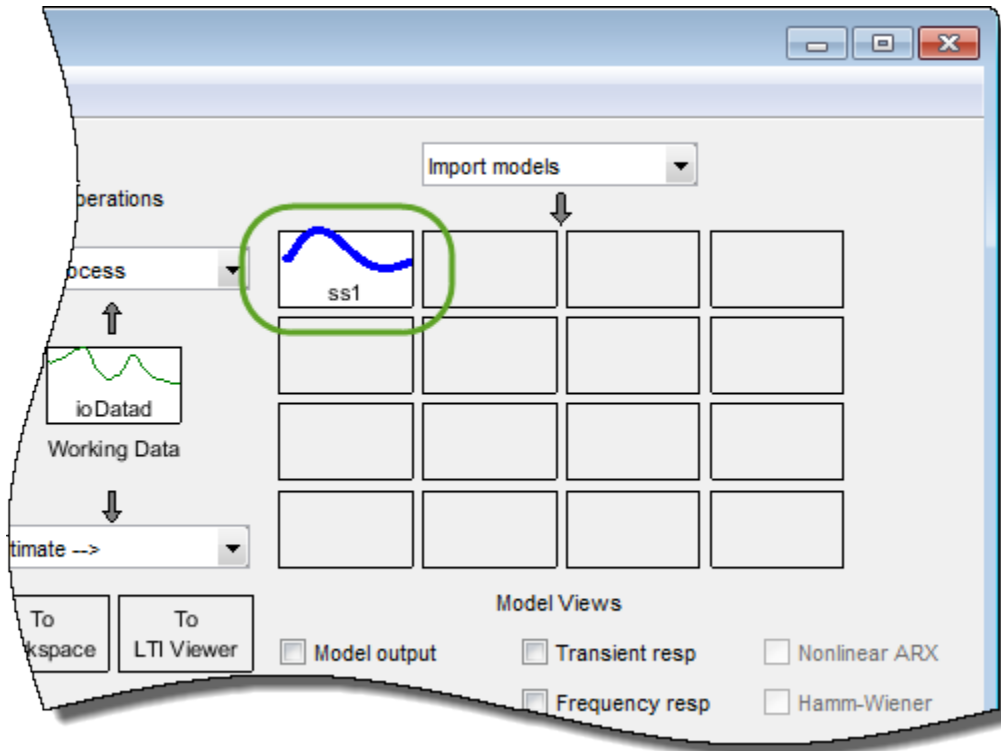


In the State Space Models dialog box, specify the properties of the estimated model and the estimation options. For this example, estimate a second-order, discrete-time model, leaving the other estimation options at their default values.



For more information on estimating state-space models, see “State-Space Models” (System Identification Toolbox).

Click **Estimate**. The app estimates a state-space model, `ss1`, and adds the model to the **Model Views** area.

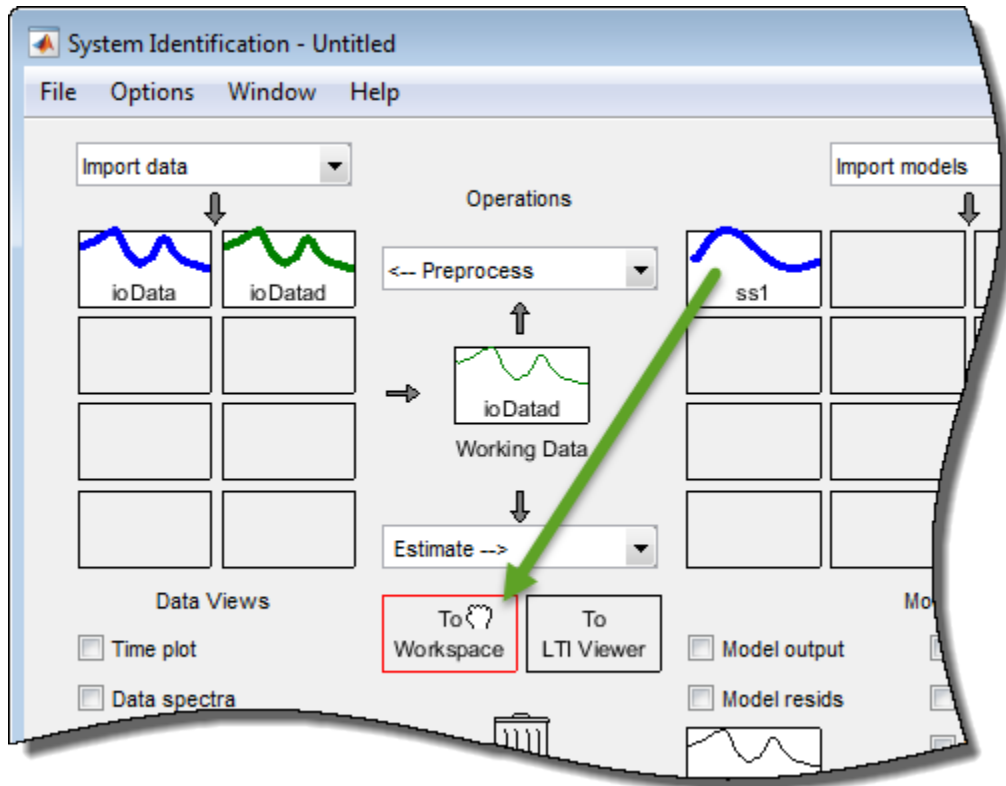


The estimated model has one measured input and one unmeasured noise component.

### **Import Identified Plant to MPC Designer**

To use `ss1` for MPC control design, first export the model to the MATLAB workspace.

Drag `ss1` from the **Model Views** area to **To Workspace**.



Open **MPC Designer**. At the MATLAB command line, type:

```
mpcDesigner
```

To import the identified model, in **MPC Designer**, click **MPC Structure**. In the Define MPC Structure By Importing dialog box, select **ss1** from the table.

Define MPC Structure By Importing

**MPC Structure**

**Select a plant model or an MPC controller from MATLAB Workspace:**

Select	Name	Type	Order	Inputs	Outputs
<input checked="" type="radio"/>	<a href="#">ss1</a>	idss	2	1	1

**Controller Sample Time**

Specify MPC controller sample time:

**Assign plant i/o channels to desired signal types:**

Manipulated variable (MV) channel indices:

Measured disturbance (MD) channel indices:

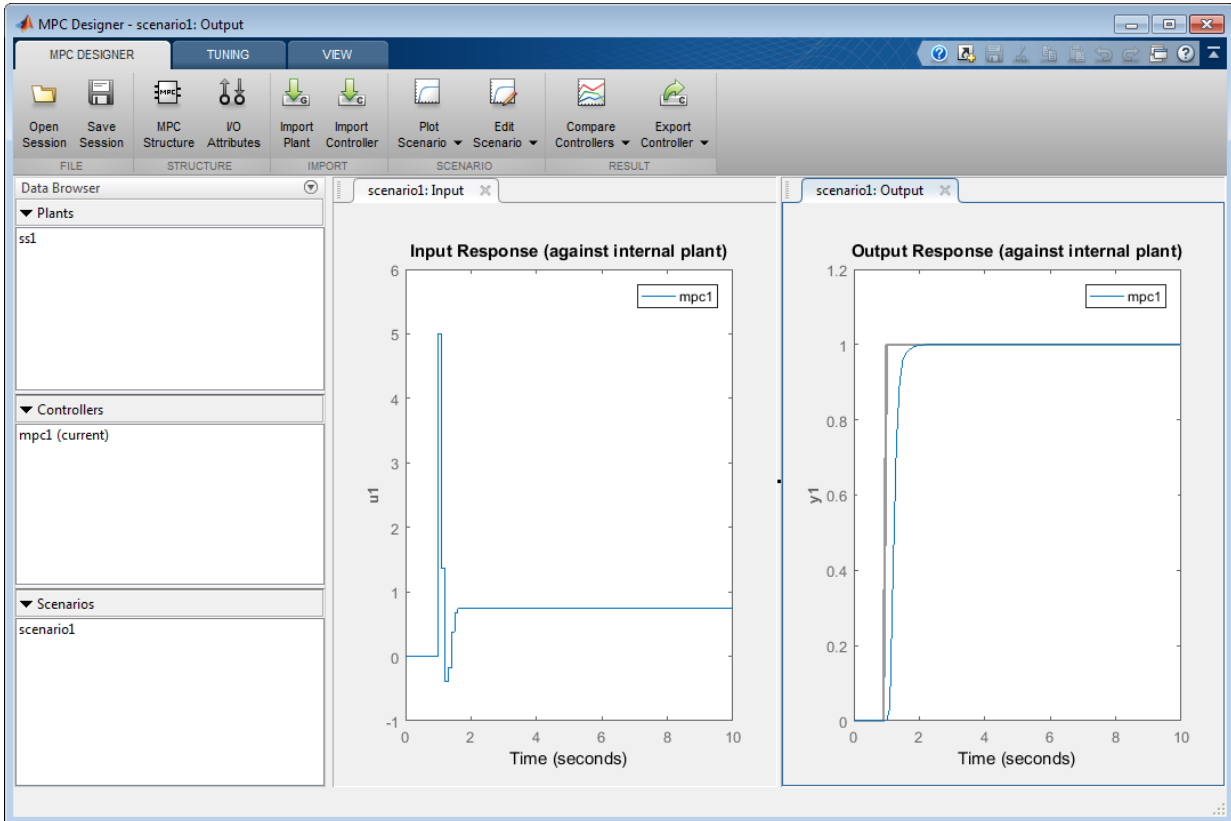
Unmeasured disturbance (UD) channel indices:

Measured output (MO) channel indices:

Unmeasured output (UO) channel indices:



Click **Define and Import**.



**Tip** You can also import the identified model when opening **MPC Designer**.

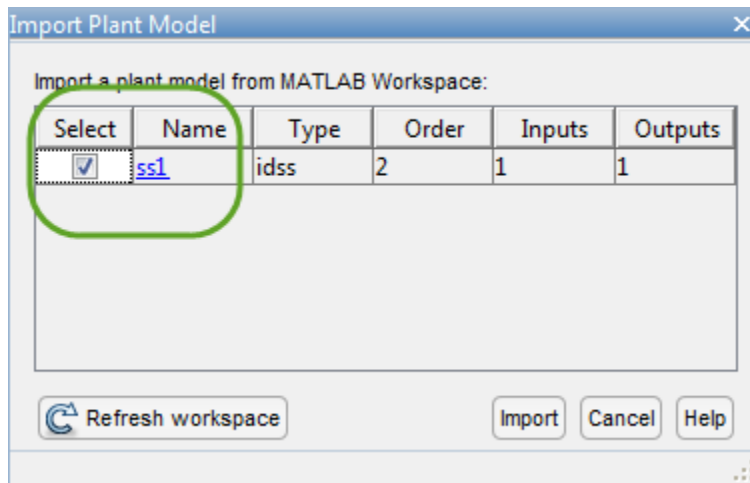
```
mpcDesigner(ss1)
```

The app converts the identified plant to a discrete-time, state-space model, if necessary, and creates a default MPC controller, `mpc1`, in which the:

- Measured input of the identified plant is a manipulated variable.
- Output of the identified plant is a measured output.

By default, the MPC controller discards the unmeasured noise component from your identified model. To configure noise channels as unmeasured disturbances, you must first create an augmented state-space model from your identified model. For more information, see “Configure Noise Channels as Unmeasured Disturbances” on page 2-81.

**Note** You can also import an identified linear model into an existing **MPC Designer** session. In **MPC Designer**, click **Import Plant**. In the Import Plant Model dialog box, select an identified model from the table.



Only identified models with an I/O configuration that is compatible with the current MPC structure are displayed in the Import Plant Model dialog box. If the current MPC structure includes unmeasured disturbances, any noise channels from the identified model are converted to unmeasured disturbances. Otherwise, the noise channels are discarded.

### Specify I/O Attributes

To improve controller performance and simplify controller tuning, specify the following attributes for each input and output signal:

- **Scale Factor** — Scale each signal by a factor that approximates its span, which is the difference between its maximum and minimum values. Scaling simplifies controller

weight tuning and improves the numerical conditioning of the controller. For more information, see “Specify Scale Factors”.

- **Nominal Value** — Apply an offset to each signal that corresponds to the nominal operating conditions under which you collected the identification data; that is the offsets removed by detrending the data. Specifying nominal values places the controller at the same operating point as the plant, which is important when the plant is a nonlinear system.

In **MPC Designer**, on the **MPC Designer** tab, click **I/O Attributes**.

In the Input and Output Channel Specifications dialog box, specify the **Nominal Value** and **Scale Factor** for the input and output signals.

**Plant Inputs**

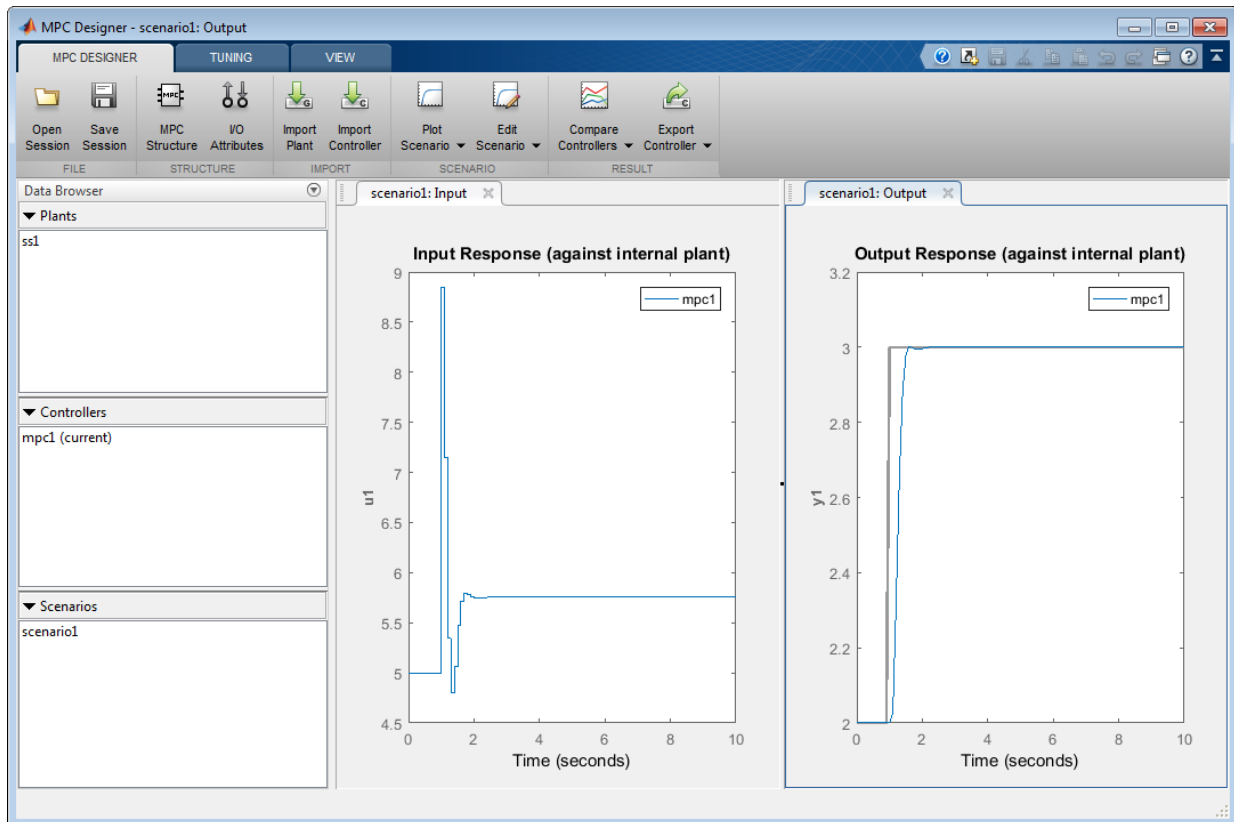
Channel	Type	Name	Unit	Nominal Value	Scale Factor
u(1)	MV	u1		4.99999999999995	3

**Plant Outputs**

Channel	Type	Name	Unit	Nominal Value	Scale Factor
y(1)	MO	y1		2	4.5

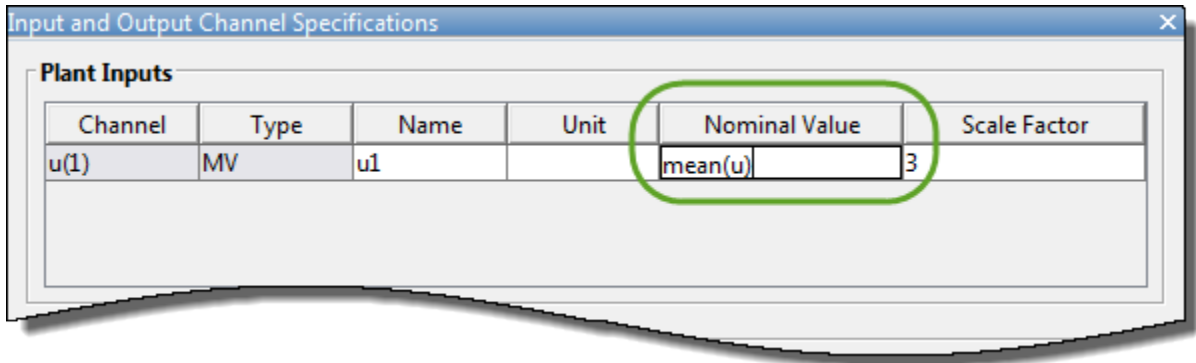
OK Apply Cancel Help

Click **OK**.



The default controller tracks the output reference value well, however the initial controller response is aggressive.

**Tip** You can specify the **Nominal Value** or **Scale Factor** using expressions such as  $\text{mean}(u)$  or  $\text{max}(y) - \text{min}(y)$  respectively, where  $u$  and  $y$  are the I/O signals from the MATLAB workspace.

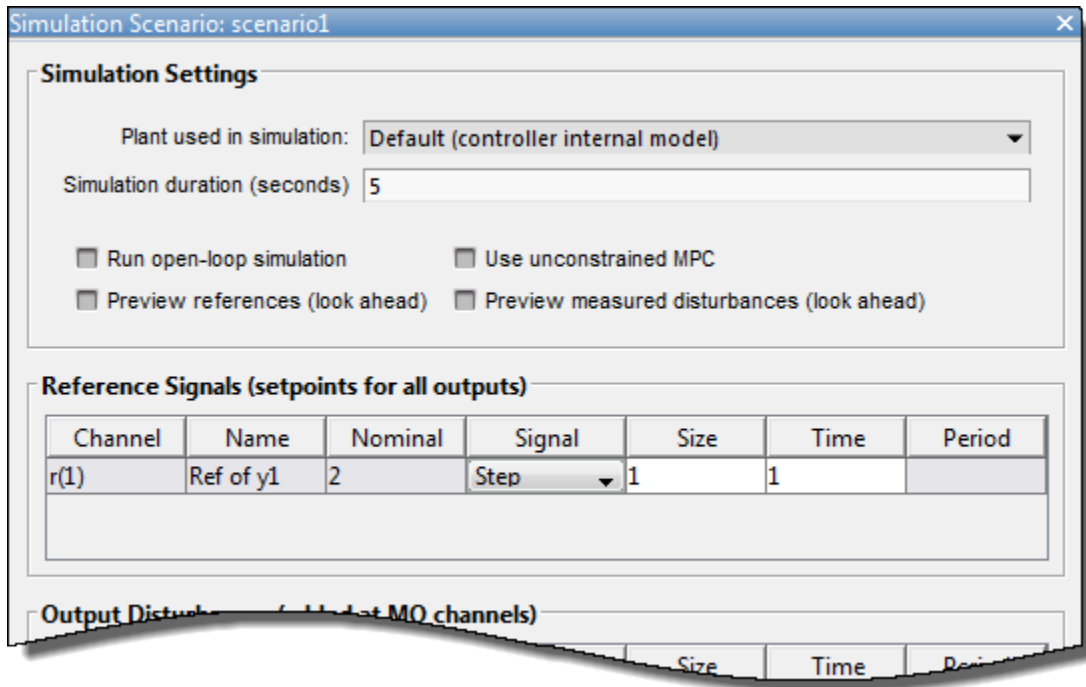


### Configure Simulation Scenario

In **MPC Designer**, on the **Tuning** tab, click **Edit Scenario** > **scenario1**.

In the Simulation Scenario dialog box, specify a **Simulation duration** of 5 seconds.

In the **Reference Signals** section, keep the default step signal.

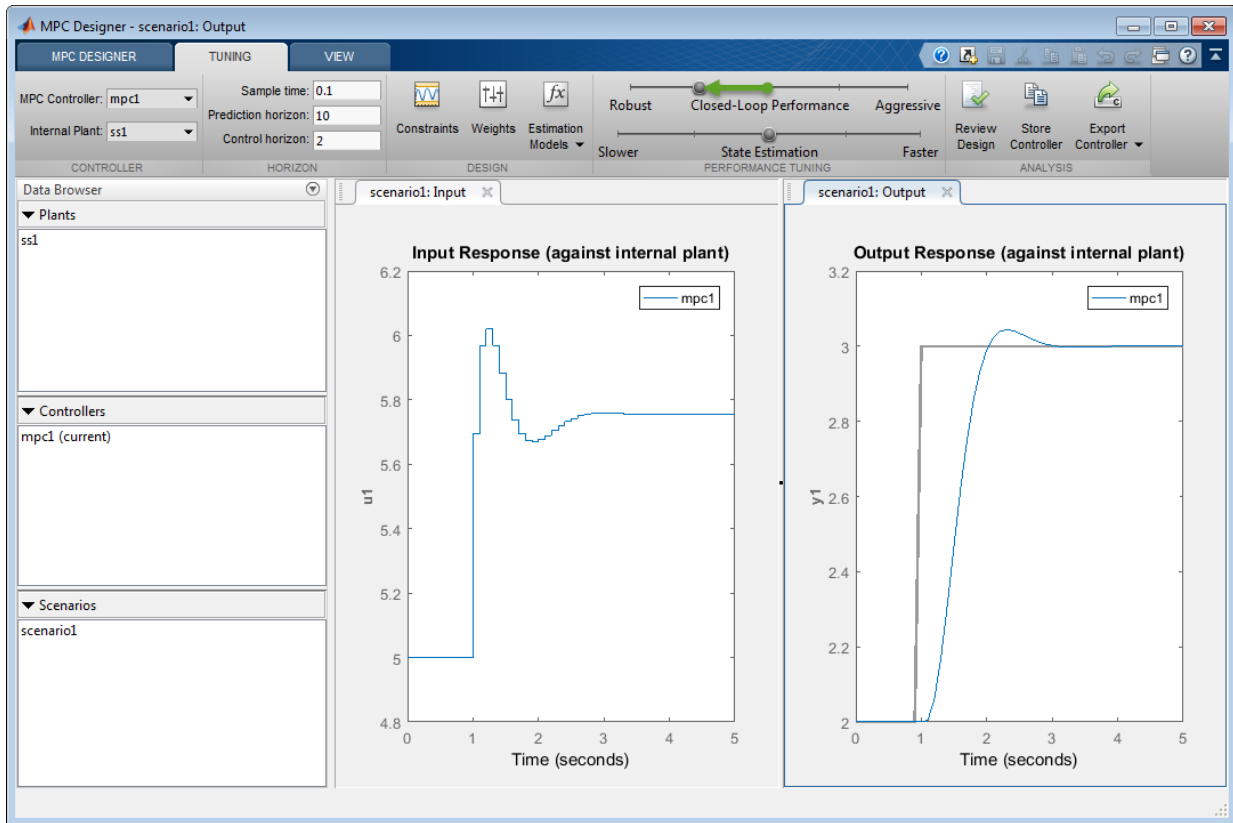


Click **OK**.

### Tune Controller

Before tuning your controller, it is good practice to specify the controller sample time, prediction horizon, and control horizon. Since you identified a discrete-time plant model, the controller automatically derives its sample time from the identified model. For this example, use the default prediction and control horizons. For more information, see “Choose Sample Time and Horizons”.

To make the controller less aggressive, on the **Tuning** tab, drag the **Closed-Loop Performance** slider to the left. Doing so increases the cost function weight on the manipulated variable rate of change, and decreases the weight on the output variable.



The input response is now more conservative. The trade-offs are an increased overshoot and longer settling time.

For more information on tuning controller weights, see “Tune Weights”.

**Note** If your plant has known physical or safety constraints that limit the output range, input range, or input signal rate of change, you can specify these constraints in the MPC

controller. If so, define the constraints before tuning your controller weights. For more information, see “Specify Constraints”.

---

### Design Controller for Identified Plant at the Command Line

This example shows how to design a model predictive controller at the command line using an identified plant model.

Load the input/output data.

```
load plantIO
```

This command imports the plant input signal,  $u$ , plant output signal,  $y$ , and sample time,  $T_s$ , to the MATLAB® workspace.

Create an `iddata` object from the input and output data.

```
mydata = iddata(y,u,Ts);
```

Preprocess the I/O data by removing offsets (mean values) from the input and output signals.

```
mydatad = detrend(mydata);
```

You can also remove offsets by creating an `ssestOptions` object and specifying the `InputOffset` and `OutputOffset` options.

Estimate a second order, linear state-space model using the I/O data. Estimate a discrete-time model by specifying the sample time as  $T_s$ .

```
ss1 = ssest(mydatad,2,'Ts',Ts);
```

The estimated model has one measured input and one unmeasured noise component.

Create a default model predictive controller for the identified model, `ss1`.

```
mpcObj = mpc(ss1);
```

```
-->Converting linear model from System Identification Toolbox to state-space.  
-->The "PredictionHorizon" property of "mpc" object is empty. Trying PredictionHorizon  
-->The "ControlHorizon" property of the "mpc" object is empty. Assuming 2.  
-->The "Weights.ManipulatedVariables" property of "mpc" object is empty. Assuming defa  
-->The "Weights.ManipulatedVariablesRate" property of "mpc" object is empty. Assuming c  
-->The "Weights.OutputVariables" property of "mpc" object is empty. Assuming default 1
```



By default the controller discards the unmeasured noise component from your identified model.

To simplify the tuning process, specify input and output signal scaling factors.

```
mpcObj.MV(1).ScaleFactor = max(u) - min(u);
mpcObj.OV(1).ScaleFactor = max(y) - min(y);
```

Specify the nominal values for the input and output signals. Use the offsets that you previously removed from the I/O data.

```
nominalInput = mean(u);
nominalOutput = mean(y);
mpcObj.Model.Nominal.u = nominalInput;
mpcObj.Model.Nominal.y = nominalOutput;
```

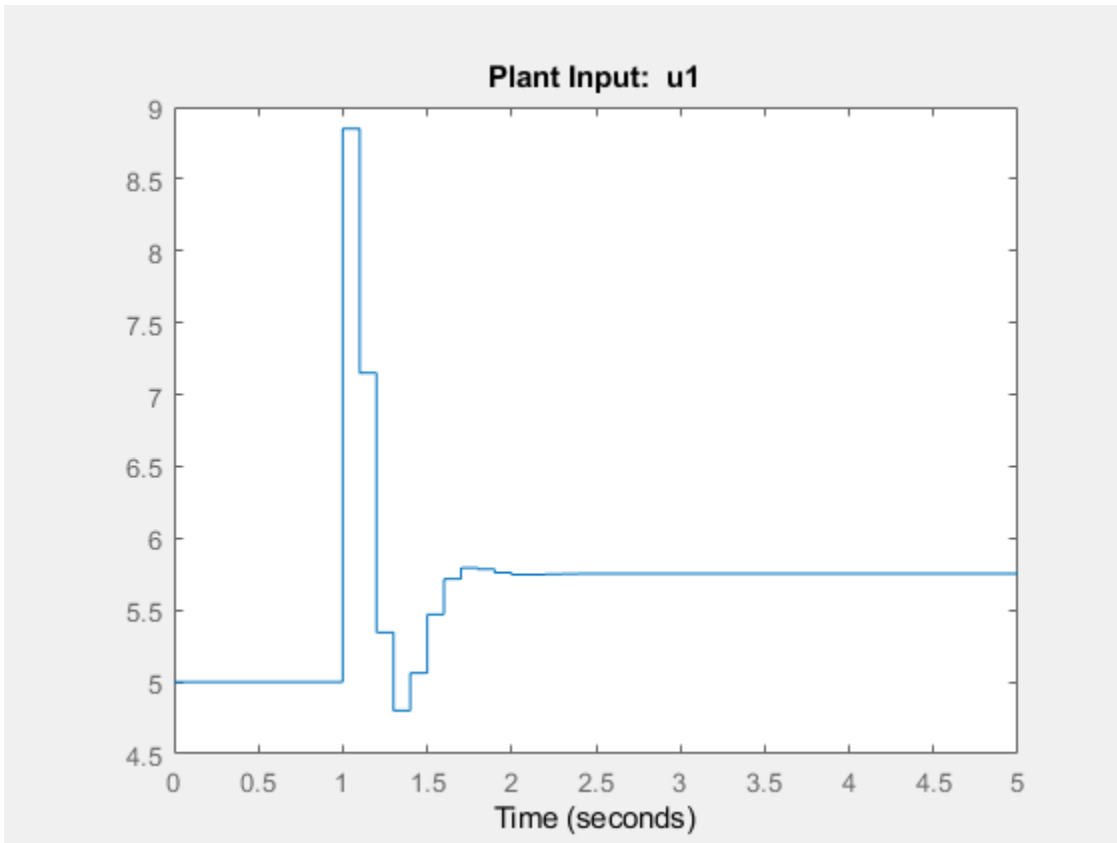
Configure the simulation reference signal. Specify a reference signal with a five-second duration and a unit step at a time of one second. The initial value of the reference signal is the nominal value of the output signal.

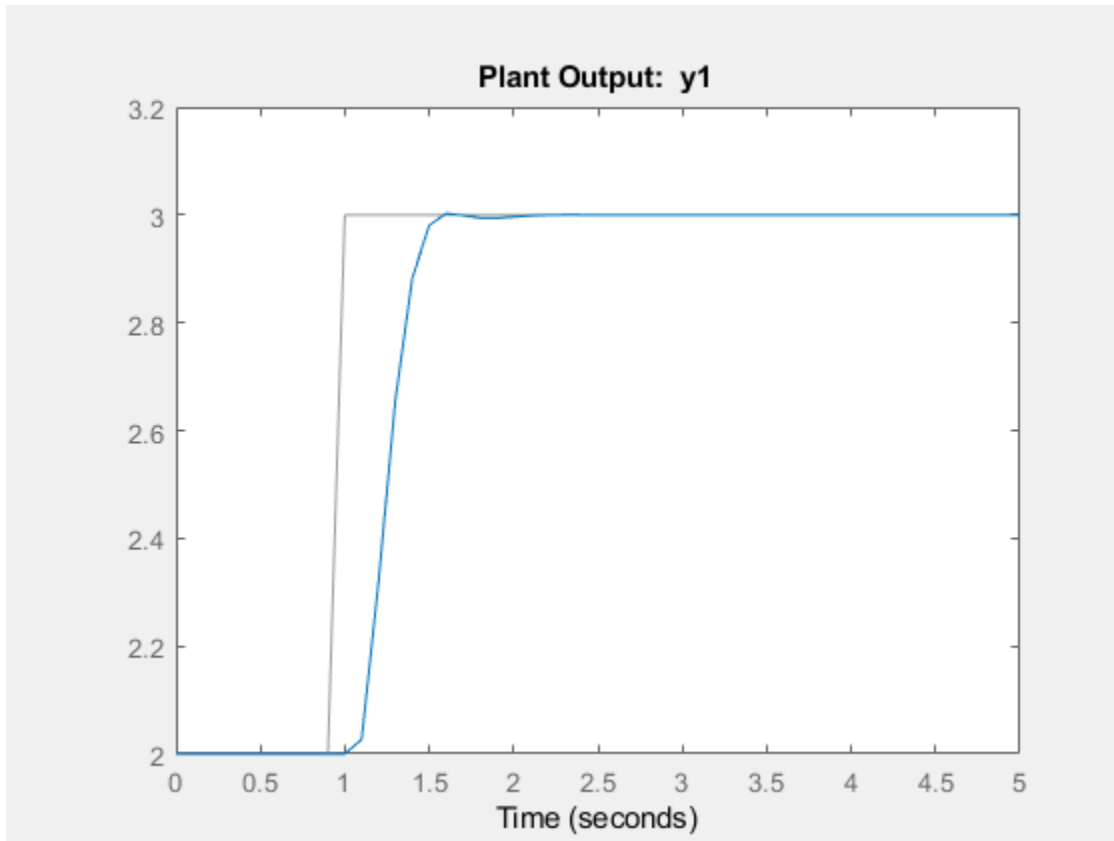
```
outputRef = [nominalOutput*ones(1/Ts,1);
             (nominalOutput+1)*ones(4/Ts+1,1)];
```

Before tuning the controller, simulate the initial controller performance.

```
sim(mpcObj,[],outputRef)
```

```
-->Assuming output disturbance added to measured output channel #1 is integrated white
-->The "Model.Noise" property of the "mpc" object is empty. Assuming white noise on ea
```





The default controller tracks the output reference value well, however the initial controller response is aggressive.

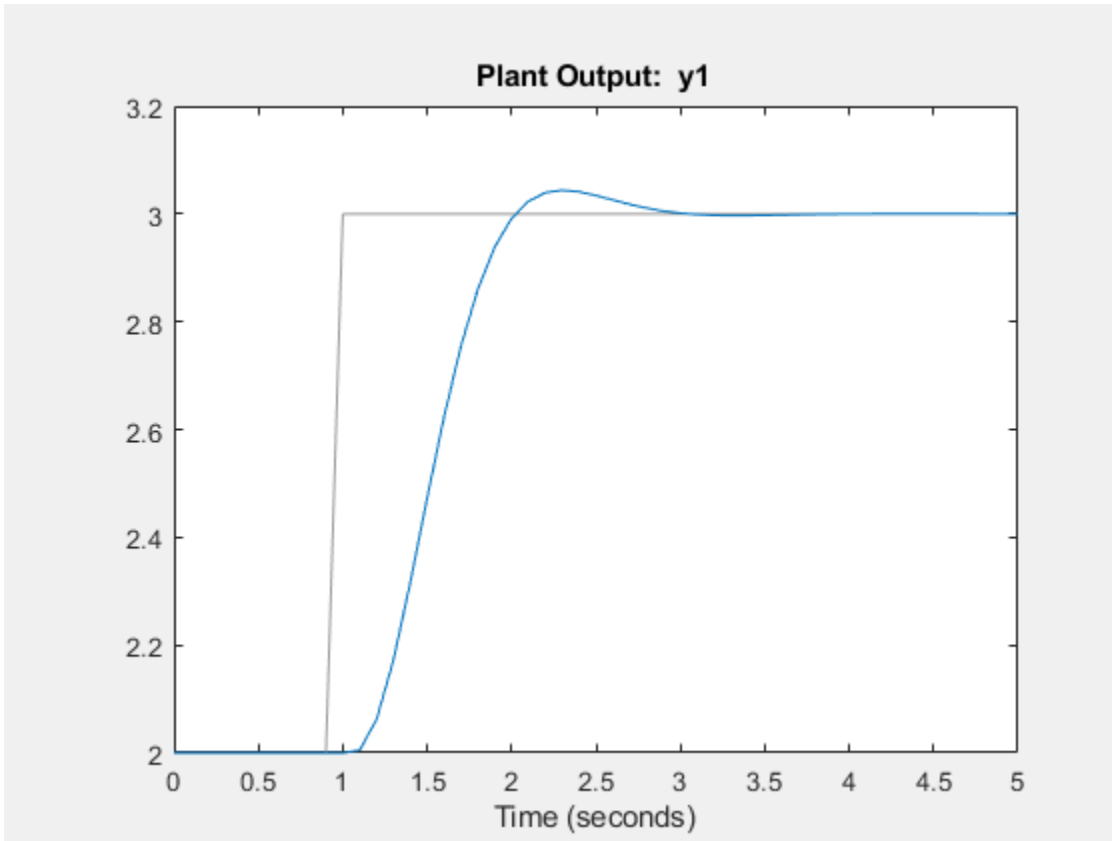
To make the controller less aggressive, simultaneously increase the tuning weight for the manipulated variable rate of change and decrease the tuning weight for the output variable.

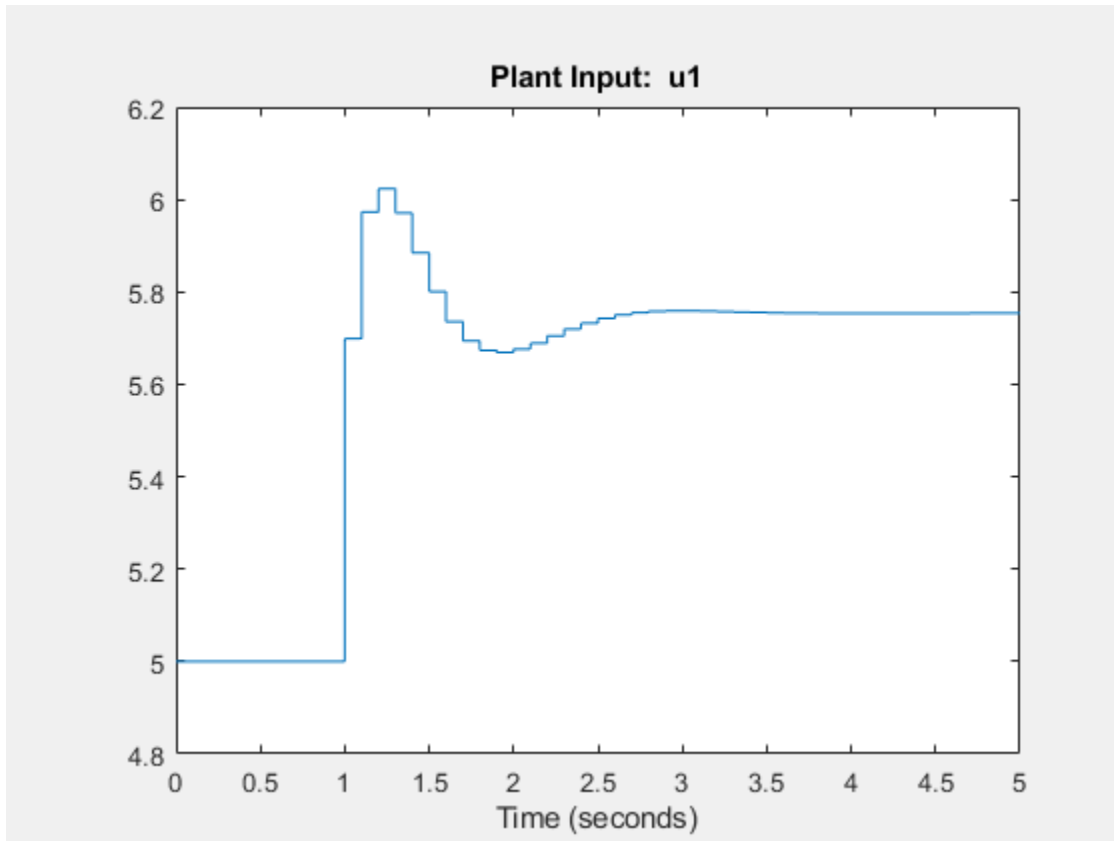
```
beta = 0.37;  
mpcObj.Weights.MVRate = mpcObj.Weights.MVRate/beta;  
mpcObj.Weights.OV = mpcObj.Weights.OV*beta;
```

Simulate the tuned controller response

```
sim(mpcObj, [], outputRef)
```

-->Assuming output disturbance added to measured output channel #1 is integrated white  
-->The "Model.Noise" property of the "mpc" object is empty. Assuming white noise on ea





The input response is now more conservative. The trade-offs are an increased overshoot and longer settling time.

## Configure Noise Channels as Unmeasured Disturbances

When you create an MPC controller using an identified model, the software discards any noise channels from the model by default. You can configure the noise channels as unmeasured disturbances by augmenting the identified model.

### Augment Identified Model with Noise Channels

To convert noise channels to unmeasured disturbances, first convert the identified model, `ss1`, to a state-space model using the 'augmented' option. At the MATLAB command line, type:

```
ss2 = ss(ss1, 'augmented');
```

This option creates a state-space model, `ss2`, with the following input groups:

- **Measured** — The input channels from the identified model.
- **Noise** — The noise channels from the identified model. The number of noise channels matches the number of outputs channels.

---

**Note** The System Identification Toolbox software assumes that the inputs to the **Noise** channels are unit-variance Gaussian noise. Therefore, the augmented model encapsulates any noise dynamics from the identified model, such as integration at the disturbance source.

---

You can then create an MPC controller using the augmented state-space model.

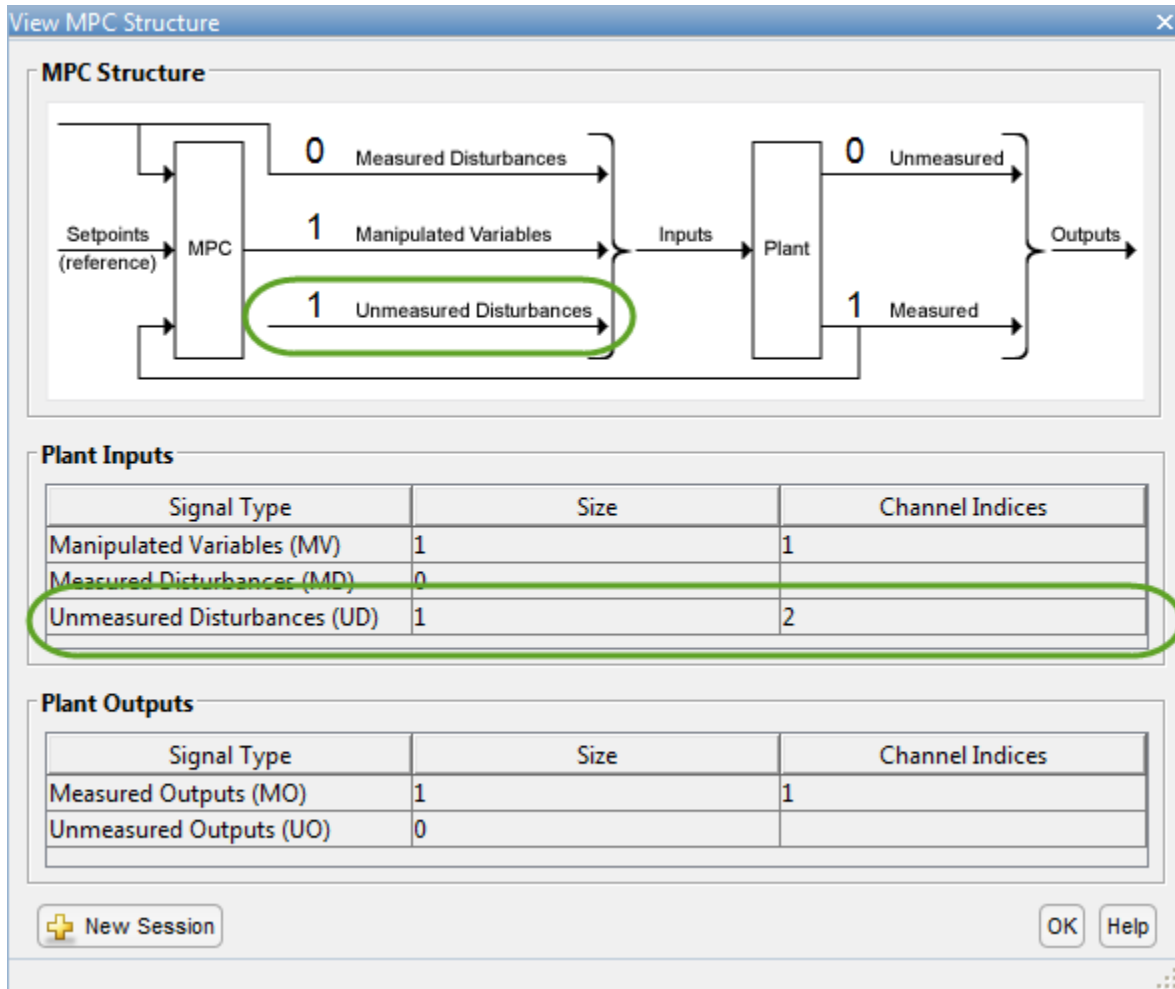
```
mpcObj = mpc(ss2);
```

The software configures the **Measured** inputs as manipulated variables and the **Noise** inputs as unmeasured disturbances.

You can also import the augmented model into **MPC Designer**.

```
mpcDesigner(ss2)
```

To view the MPC signal configuration, in **MPC Designer**, on the **MPC Designer** tab, click **MPC Structure**.



The View MPC Structure dialog box shows the noise channels as unmeasured disturbances.

### Configure Input Disturbance Model

When you convert an identified model to an augmented state-space model, the System Identification Toolbox software assumes that noise sources are unit-variance Gaussian

noise. However, by default, MPC controllers model unmeasured input disturbances as integrated Gaussian noise. When designing your controller, you can:

- Remove the integrators from the input disturbance model, which simplifies the controller. Use this option if the experimental identification data was collected under conditions that closely match the expected plant operating conditions. In this case, the augmented state-space model encapsulates any noise dynamics from the identified system.
- Keep the default integrated white noise input disturbance model, which leads to more aggressive disturbance rejection. Use this option if the experimental identification data was collected under controlled conditions that may not match the expected plant operating conditions. In this case, the controller compensates for noise dynamics that the augmented model does not encapsulate.

---

**Note** When using **MPC Designer**, you can tune your controller disturbance rejection properties by adjusting the **State Estimation** slider. For more information, see “Disturbance Rejection Tuning”.

---

To remove an integrator from an input disturbance model channel, configure that channel as a static unit gain. For example, to remove the integrators from all input disturbance model channels, set the input disturbance model to a static gain identity matrix. At the MATLAB command line, type:

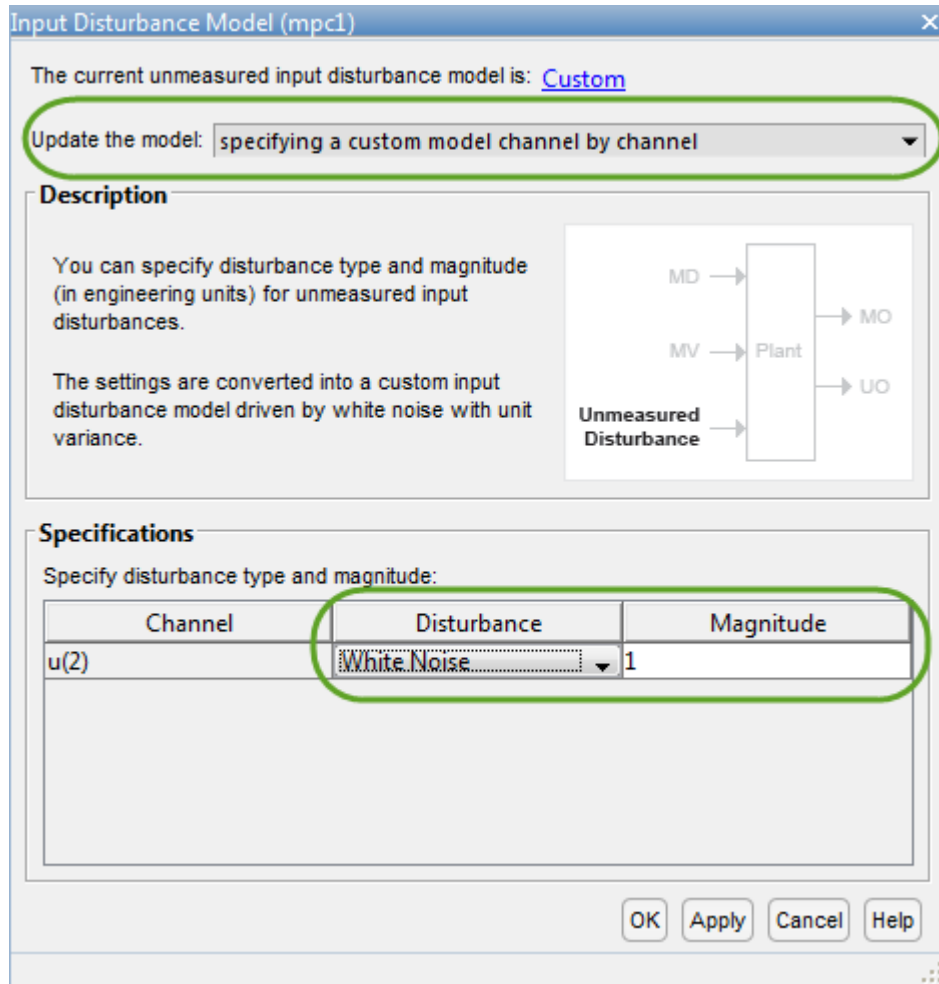
```
setindist(mpcObj,ss(eye(Nd)));
```

where  $N_d$  is the number of unmeasured disturbances.

To set the disturbance model for an unmeasured disturbance channel to a static unit gain using **MPC Designer**:

- 1 On the **Tuning** tab, select **Estimation Models > Input Disturbance Model**.
- 2 In the Input Disturbance Model dialog box, in the **Update the model** drop-down list, select specifying a custom model channel by channel.
- 3 In the **Specifications** table, in the **Disturbance** drop-down list, select **White Noise**.
- 4 Specify a **Magnitude** of 1.





- 5 Repeat steps 3 and 4 for each unmeasured disturbance.
- 6 To apply the changes and update the input disturbance model, click **OK** or **Apply**.

For more information about changing the input disturbance model, see “Adjust Disturbance and Noise Models”.

### Configure Simulation Scenario

You can simulate your MPC controller using unit-variance Gaussian noise unmeasured disturbance signals, as assumed by the System Identification Toolbox software. This scenario emulates the experimental conditions under which the data was collected for identification.

To configure unmeasured disturbance signals, create an MPC simulation option set for your controller using `mpcsimopt`. At the MATLAB command line, type:

```
opt = mpcsimopt(mpcObj);
```

Configure the `UnmeasuredDisturbance` option using `randn`.

```
opt.UnmeasuredDisturbance = randn(T,Nd);
```

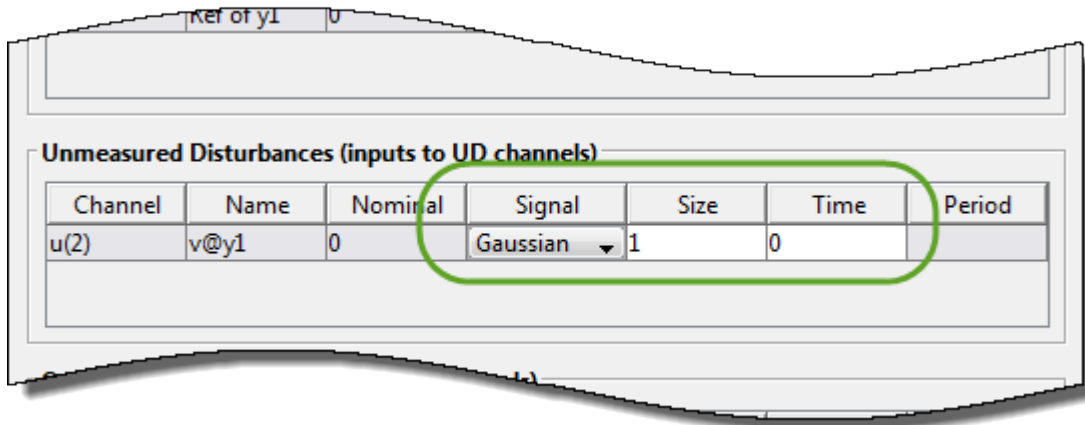
where `T` is the number of simulation steps and `Nd` is the number of unmeasured disturbances.

Simulate the controller using this option set and an output reference signal, `outputRef`.

```
y = sim(mpcObj,T,outputRef,opt);
```

To configure your simulation in **MPC Designer**:

- 1 On the **Tuning** tab, under **Edit Scenario** select the simulation scenario you want to edit.
- 2 In the Simulation Scenario dialog box, in the **Unmeasured Disturbances** section, under **Signal**, select **Gaussian**.
- 3 Specify a **Size** of 1, which corresponds to a unit variance.
- 4 To apply the disturbance from the start of the simulation, specify a **Time** of 0.



- 5 Repeat steps 2-4 for each unmeasured disturbance channel.
- 6 To apply the changes and update the **MPC Designer** response plots, click **OK** or **Apply**.

## See Also

### Apps

**MPC Designer** | **System Identification**

### Functions

mpc | sim | ss | ssest

## More About

- “About Identified Linear Models” (System Identification Toolbox)
- “Identify Plant from Data” on page 2-53
- “Design Controller Using MPC Designer” on page 3-2
- “Design MPC Controller at the Command Line” on page 4-2



# Designing Controllers Using MPC Designer

---

## Design Controller Using MPC Designer

This example shows how to design a model predictive controller for a continuous stirred-tank reactor (CSTR) using **MPC Designer**.

### CSTR Model

The following differential equations represent the linearized model of a continuous stirred-tank reactor (CSTR) involving an exothermic reaction:

$$\frac{dC'_A}{dt} = a_{11}C'_A + a_{12}T' + b_{11}T'_c + b_{12}C'_{Ai}$$

$$\frac{dT'}{dt} = a_{21}C'_A + a_{22}T' + b_{21}T'_c + b_{22}C'_{Ai}$$

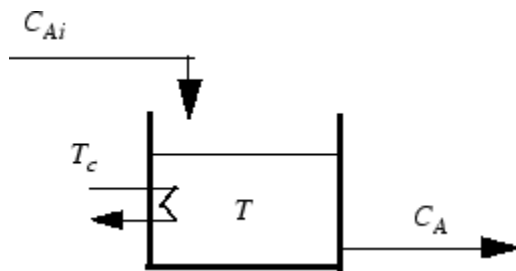
where the inputs are:

- $C_{Ai}$  — Concentration of reagent  $A$  in the feed stream (kgmol/m<sup>3</sup>)
- $T_c$  — Reactor coolant temperature (degrees C)

and the outputs are:

- $T$  — Reactor temperature (degrees C)
- $C_A$  — Residual concentration of reagent  $A$  in the product stream (kgmol/m<sup>3</sup>)

The prime terms, such as  $C'_A$ , denote a deviation from the nominal steady-state condition at which the model has been linearized.



Measurement of reagent concentrations is often difficult. For this example, assume that:

- $T_c$  is a manipulated variable.
- $C_{Ai}$  is an unmeasured disturbance.
- $T$  is a measured output.
- $C_A$  is an unmeasured output.

The model can be described in state-space format:

$$\frac{dx}{dt} = Ax + Bu$$

$$y = Cx + Du$$

where,

$$x = \begin{bmatrix} C'_A \\ T' \end{bmatrix}, u = \begin{bmatrix} T'_c \\ C'_{Ai} \end{bmatrix}, y = \begin{bmatrix} T' \\ C'_A \end{bmatrix}$$

$$A = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix}, B = \begin{bmatrix} b_{11} & b_{12} \\ b_{21} & b_{22} \end{bmatrix}, C = \begin{bmatrix} 0 & 1 \\ 1 & 0 \end{bmatrix}, D = \begin{bmatrix} 0 & 0 \\ 0 & 0 \end{bmatrix}$$

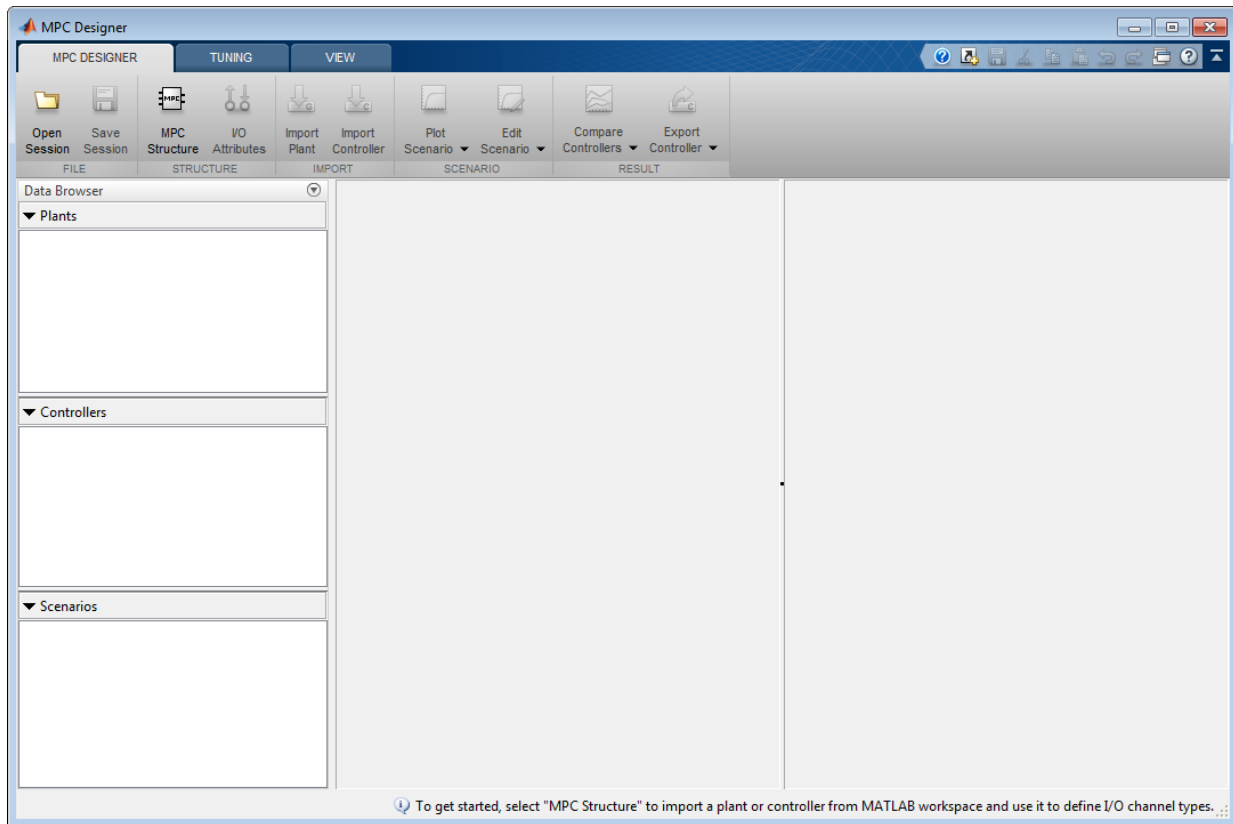
For this example, the coolant temperature has a limited range of  $\pm 10$  degrees from its nominal value and a limited rate of change of  $\pm 4$  degrees per sample period.

Create a state-space model of a CSTR system.

```
A = [-0.0285 -0.0014; -0.0371 -0.1476];
B = [-0.0850 0.0238; 0.0802 0.4462];
C = [0 1; 1 0];
D = zeros(2,2);
CSTR = ss(A,B,C,D);
```

### Import Plant and Define MPC Structure

```
mpcDesigner
```



On the **MPC Designer** tab, in the **Structure** section, click **MPC Structure**.

In the Define MPC Structure By Importing dialog box, in the **Select a plant model or an MPC controller** table, select the CSTR model.

Since CSTR is a stable, continuous-time LTI system, **MPC Designer** sets the controller sample time to  $0.1 T_r$ , where  $T_r$  is the average rise time of CSTR. For this example, in the **Specify MPC controller sample time** field, enter a sample time of 1.

By default, all plant inputs are defined as manipulated variables and all plant outputs as measured outputs. In the **Assign plant i/o channels** section, assign the input and output channel indices such that:

- The first input, coolant temperature, is a manipulated variable.



- The second input, feed concentration, is an unmeasured disturbance.
- The first output, reactor temperature, is a measured output.
- The second output, reactant concentration, is an unmeasured output.

Define MPC Structure By Importing

**MPC Structure**

**Select a plant model or an MPC controller from MATLAB Workspace:**

Select	Name	Type	Order	Inputs	Outputs
<input checked="" type="radio"/>	<a href="#">CSTR</a>	ss	2	2	2

**Controller Sample Time**

Specify MPC controller sample time:

**Assign plant i/o channels to desired signal types:**

Manipulated variable (MV) channel indices:

Measured disturbance (MD) channel indices:

Unmeasured disturbance (UD) channel indices:

Measured output (MO) channel indices:

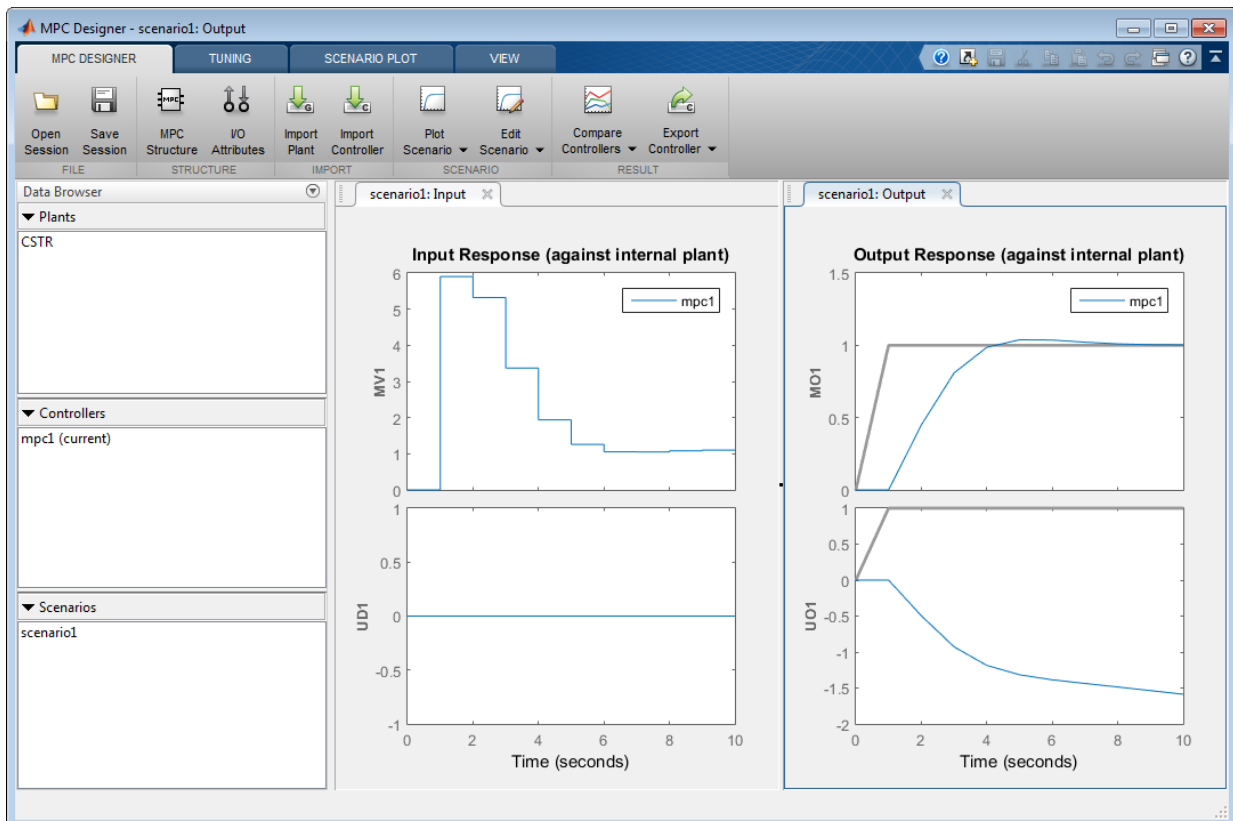
Unmeasured output (UO) channel indices:

Click **Define and Import**.

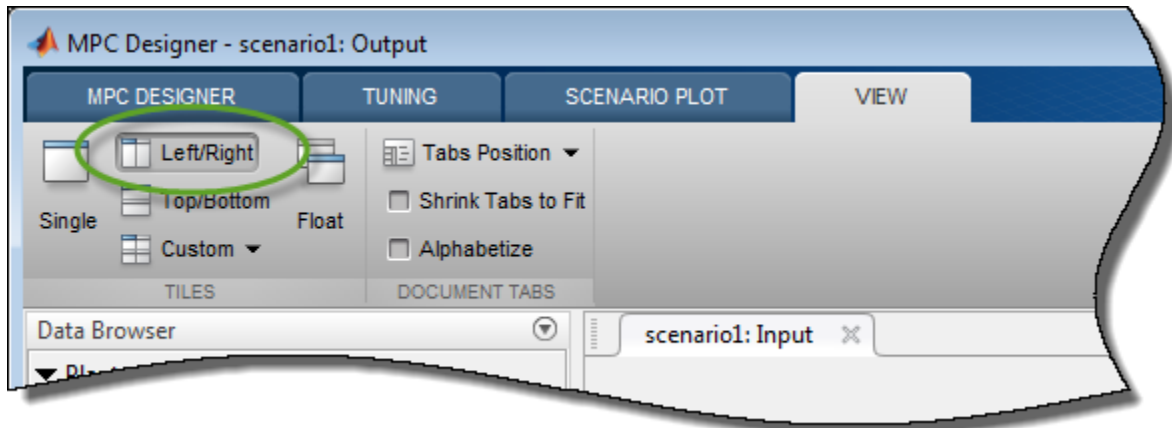
The app imports the CSTR plant to the **Data Browser**. The following are also added to the **Data Browser**:

- `mpc1` — Default MPC controller created using `sys` as its internal model.
- `scenario1` — Default simulation scenario.

The app runs the default simulation scenario and updates the **Input Response** and **Output Response** plots.



**Tip** To view the response plots side-by-side, on the **View** tab, in the **Tiles** section, click **Left/Right**.



Once you define the MPC structure, you cannot change it within the current **MPC Designer** session. To use a different channel configuration, start a new session of the app.

#### **Define Input and Output Channel Attributes**

On the **MPC Designer** tab, select **I/O Attributes**.

In the Input and Output Channel Specifications dialog box, in the **Name** column, specify a meaningful name for each input and output channel.

In the **Unit** column, optionally specify the units for each channel.

Since the state-space model is defined using deviations from the nominal operating point, set the **Nominal Value** for each input and output channel to 0.

Keep the **Scale Factor** for each channel at the default value of 1.

Input and Output Channel Specifications

**Plant Inputs**

Channel	Type	Name	Unit	Nominal Value	Scale Factor
u(1)	MV	Tc	deg C	0	1
u(2)	UD	CAi	kgmol/m <sup>3</sup>	0	1

**Plant Outputs**

Channel	Type	Name	Unit	Nominal Value	Scale Factor
y(1)	MO	T	deg C	0	1
y(2)	UO	CA	kgmol/m <sup>3</sup>	0	1

OK Apply Cancel Help

Click **OK**.

The **Input Response** and **Output Response** plot labels update to reflect the new signal names and units.

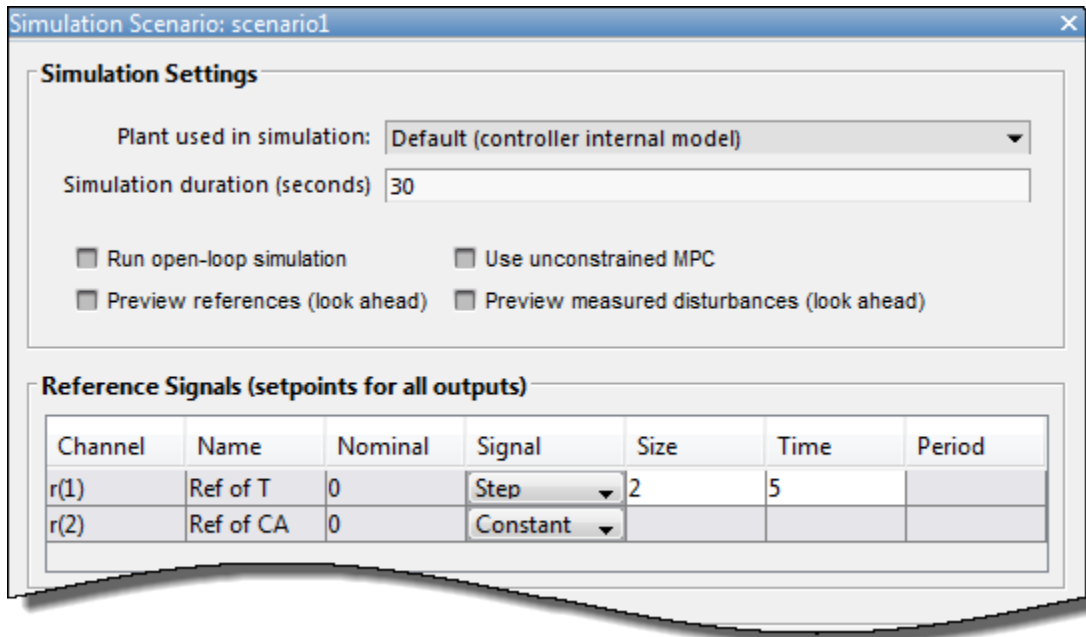
### Configure Simulation Scenario

On the **MPC Designer** tab, in the **Scenario** Section, click **Edit Scenario > scenario1**.

In the Simulation Scenario dialog box, increase the **Simulation duration** to 30 seconds.

In the **Reference Signals** table, in the first row, specify a step **Size** of 2 and a **Time** of 5.

In the **Signal** column, in the second row, select a **Constant** reference to hold the concentration setpoint at its nominal value.

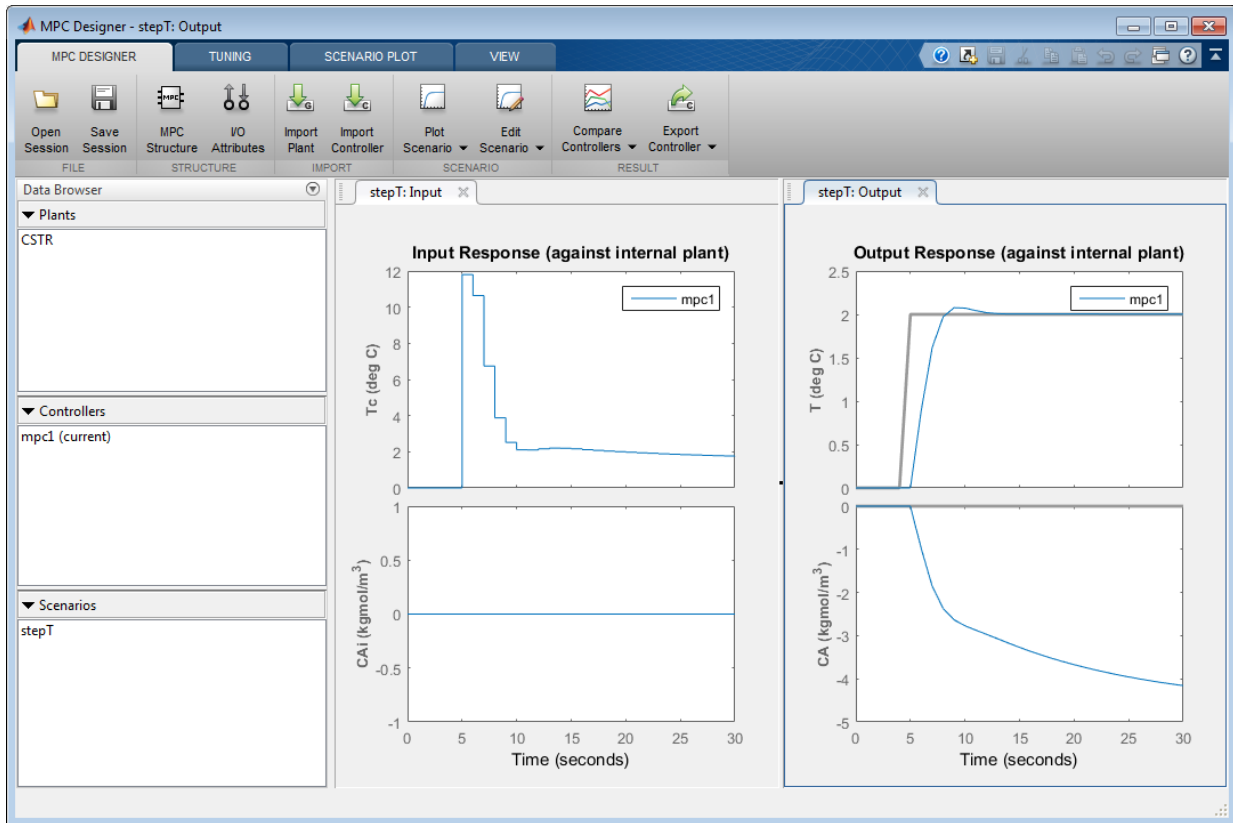


The default scenario is configured to simulate a step change of 2 degrees in the reactor temperature,  $T$ , at a time of 5 seconds.

Click **OK**.

The response plots update to reflect the new simulation scenario configuration.

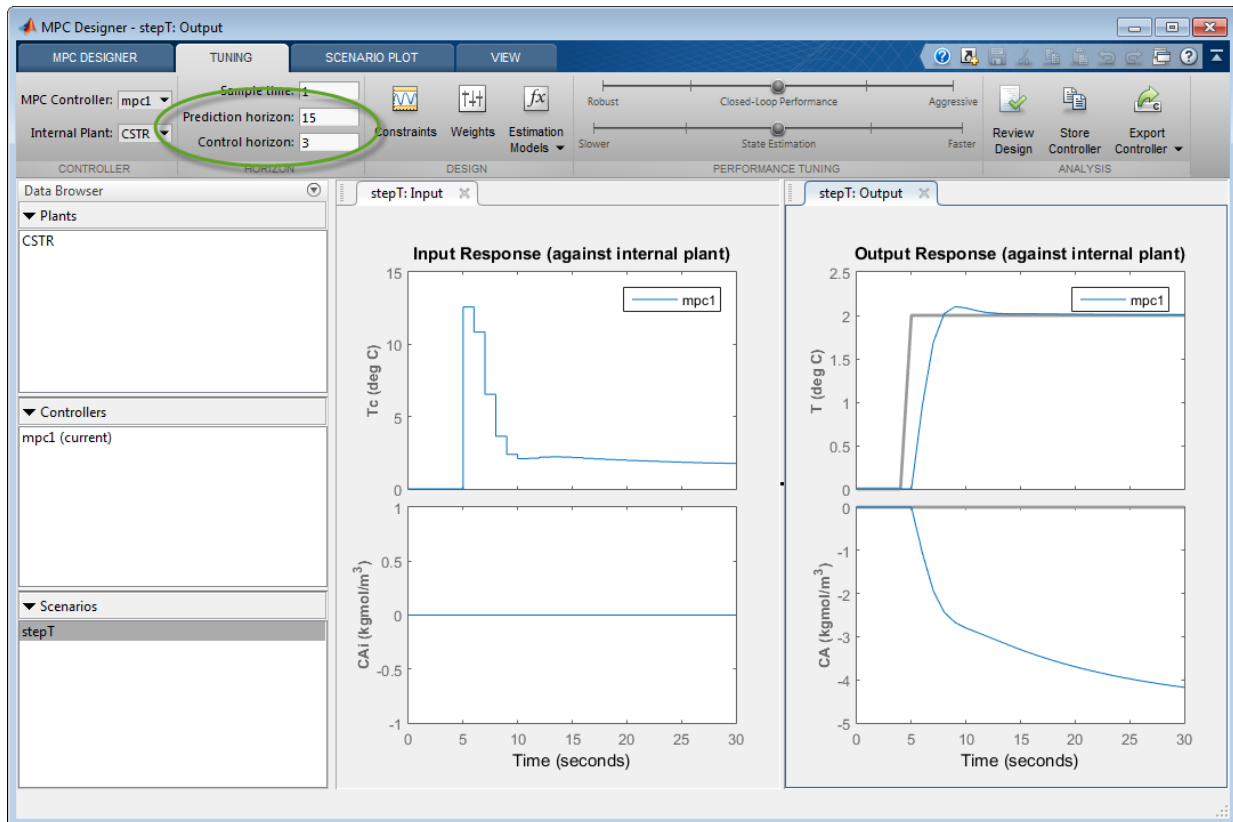
In the **Data Browser**, in the **Scenarios** section, double-click `scenario1`, and rename the scenario as `stepT`.



### Configure Controller Horizons

On the **Tuning** tab, in the **Horizons** section, specify a **Prediction horizon** of 15 and a **Control horizon** of 3.

### 3 Designing Controllers Using MPC Designer



The response plots update to reflect the new horizons. The **Input Response** plot shows that the control actions for the manipulated variable violate the required coolant temperature constraints.

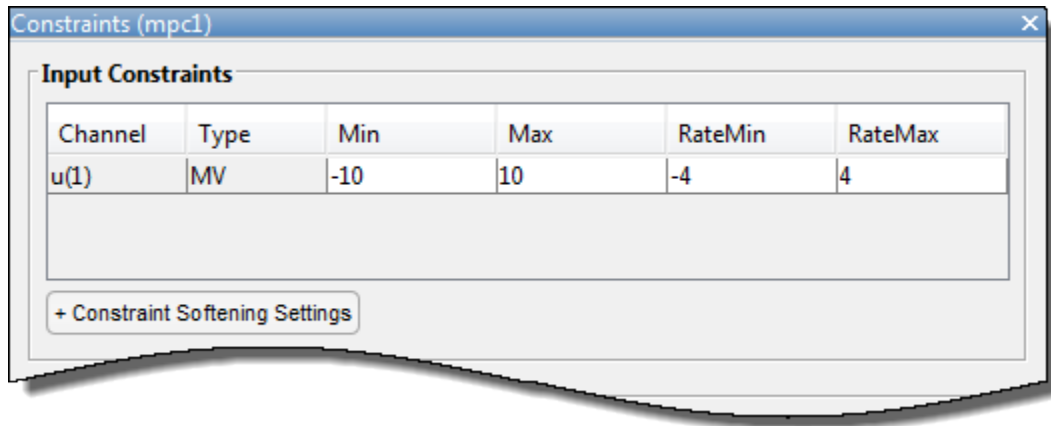
#### Define Input Constraints

In the **Design** section, click **Constraints**.

In the Constraints dialog box, in the **Input Constraints** section, enter the coolant temperature upper and lower bounds in the **Min** and **Max** columns respectively.

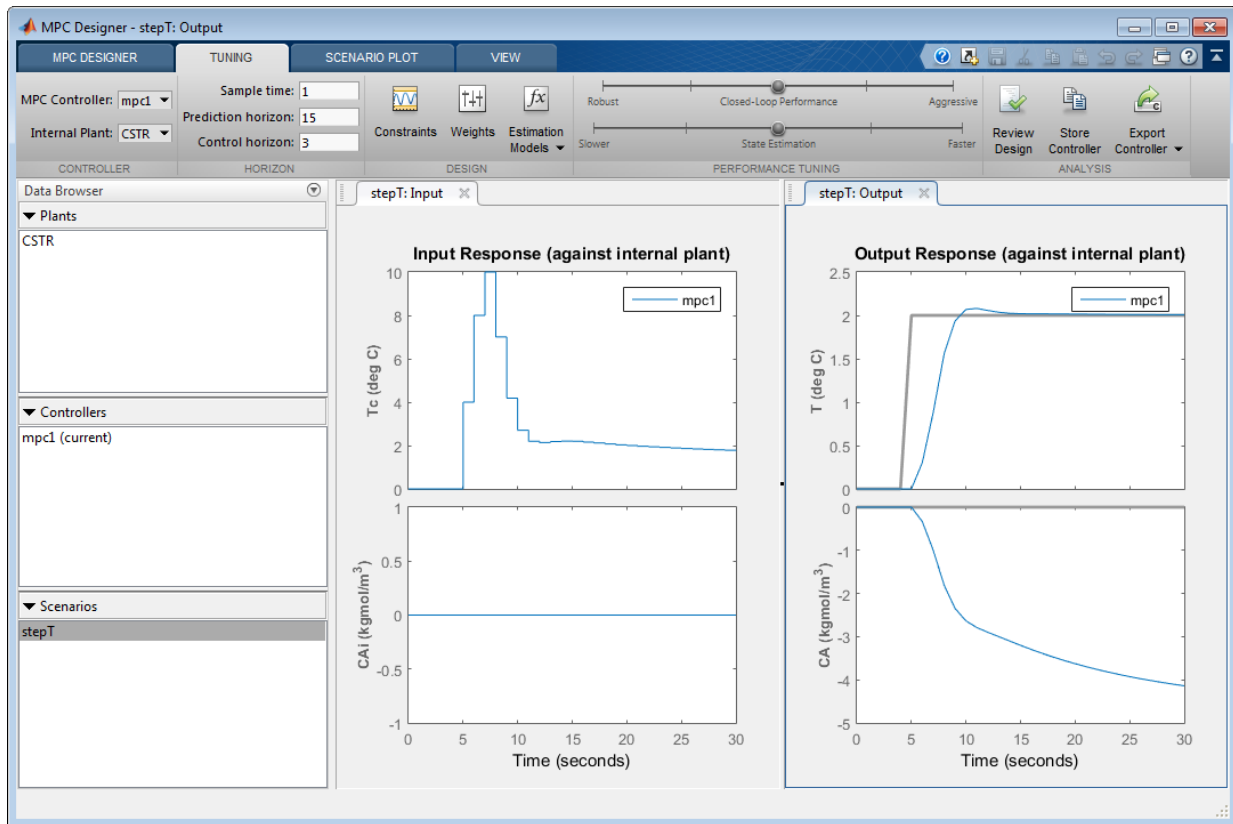
Specify the rate of change limits in the **RateMin** and **RateMax** columns.





Click **OK**.

### 3 Designing Controllers Using MPC Designer



The **Input Response** plot shows the constrained manipulated variable control actions. Even with the constrained rate of change, the coolant temperature rises quickly to its maximum limit within three control intervals.

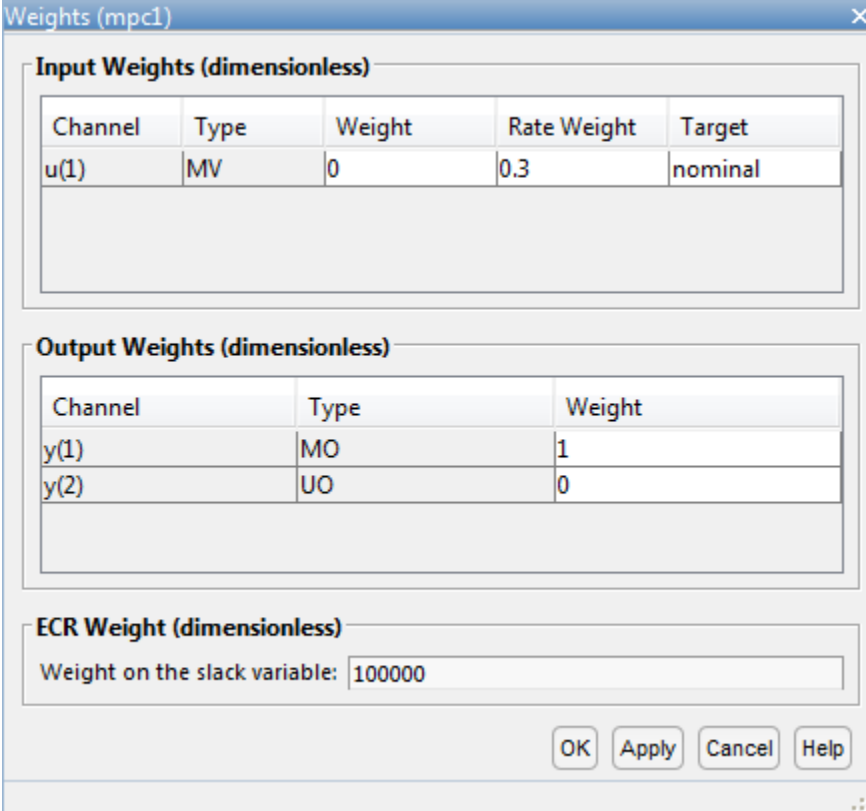
#### Specify Controller Tuning Weights

On the **Tuning** tab, in the **Design** section, click **Weights**.

In the **Input Weights** table, increase the manipulated variable (MV)**Rate Weight** to 0.3. Increasing the MV rate weight penalizes large MV changes in the controller optimization cost function.

In the **Output Weights** table, keep the default **Weight** values. By default, all unmeasured outputs have zero weights.

Since there is only one manipulated variable, if the controller tries to hold both outputs at specific setpoints, one or both outputs will exhibit steady-state error in their responses. Since the controller ignores setpoints for outputs with zero weight, setting the concentration output weight to zero allows reactor temperature setpoint tracking with zero steady-state error.



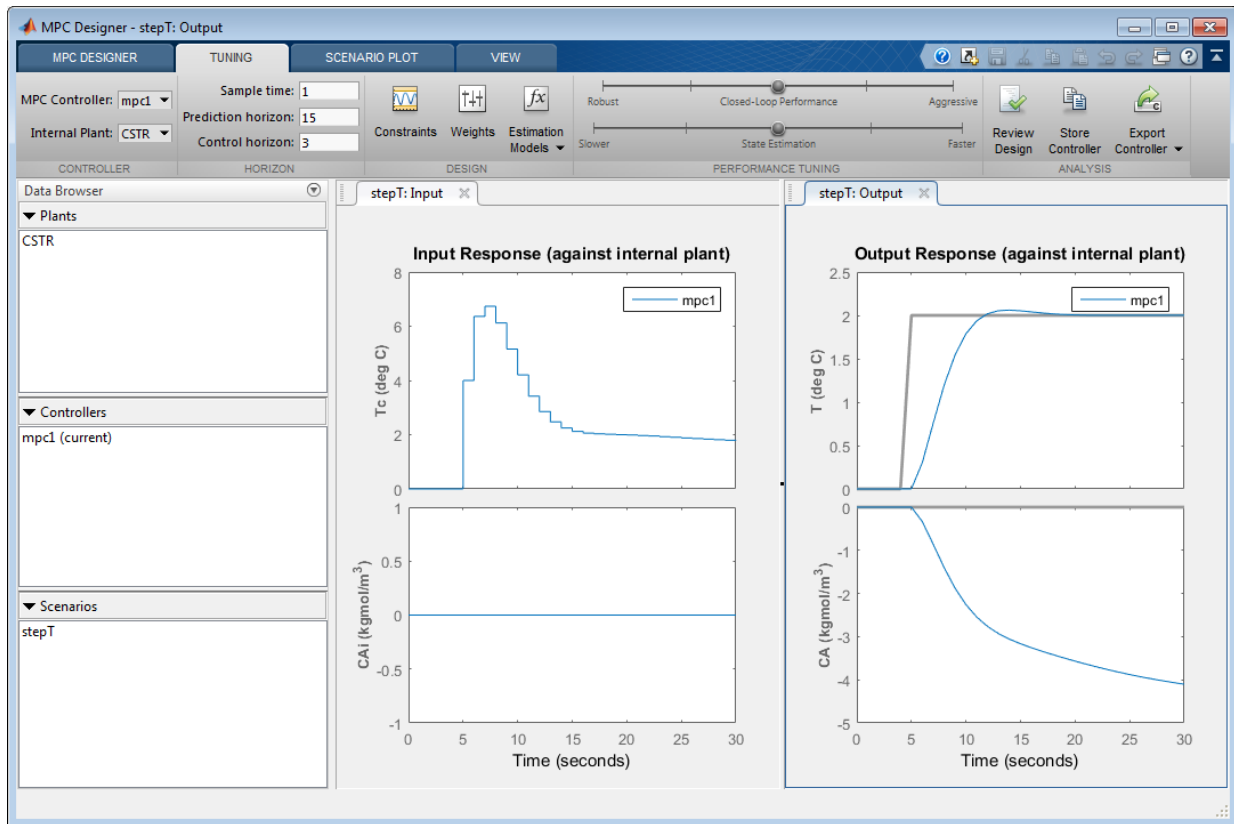
The dialog box titled "Weights (mpc1)" is divided into three sections:

- Input Weights (dimensionless)**: A table with 5 columns: Channel, Type, Weight, Rate Weight, and Target. It contains one row for channel u(1) with Type MV, Weight 0, Rate Weight 0.3, and Target nominal.
- Output Weights (dimensionless)**: A table with 3 columns: Channel, Type, and Weight. It contains two rows: y(1) with Type MO and Weight 1; and y(2) with Type UO and Weight 0.
- ECR Weight (dimensionless)**: A text field labeled "Weight on the slack variable:" with the value 100000.

Buttons for OK, Apply, Cancel, and Help are located at the bottom right of the dialog.

Click **OK**.

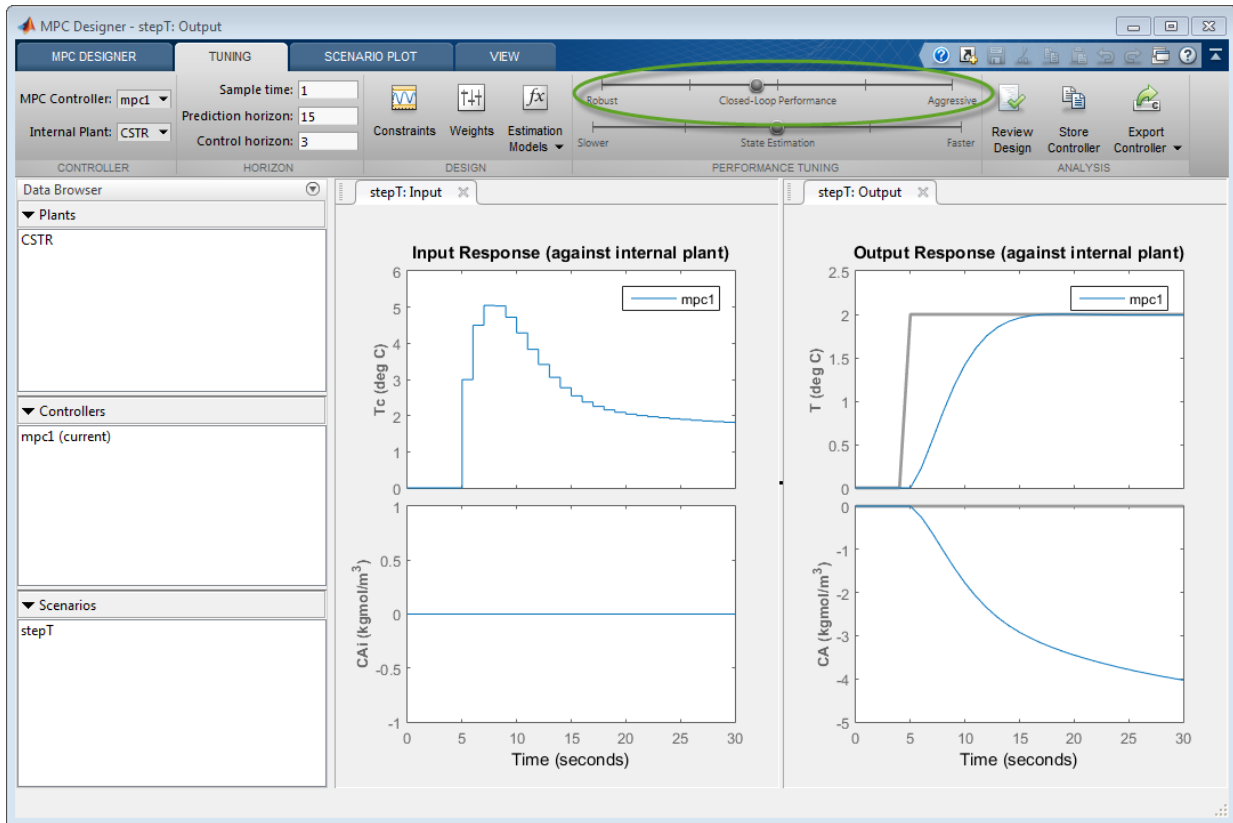
### 3 Designing Controllers Using MPC Designer



The **Input Response** plot shows the more conservative control actions, which result in a slower **Output Response**.

#### Eliminate Output Overshoot

Suppose the application demands zero overshoot in the output response. On the **Performance Tuning** tab, drag the **Closed-Loop Performance** slider to the left until the **Output Response** has no overshoot. Moving this slider to the left simultaneously increases the manipulated variable rate weight of the controller and decreases the output variable weight, producing a more robust controller.



When you adjust the controller tuning weights using the **Closed-Loop Performance** slider, **MPC Designer** does not change the weights you specified in the Weights dialog box. Instead, the slider controls an adjustment factor, which is used with the user-specified weights to define the actual controller weights.

This factor is 1 when the slider is centered; its value decreases as the slider moves left and increases as the slider moves right. The weighting factor multiplies the manipulated variable and output variable weights and divides the manipulated variable rate weights from the Weights dialog box.

To view the actual controller weights, export the controller to the MATLAB workspace, and view the `Weights` property of the exported controller object.

### Test Controller Disturbance Rejection

In a process control application, disturbance rejection is often more important than setpoint tracking. Simulate the controller response to a step change in the feed concentration unmeasured disturbance.

On the **MPC Designer** tab, in the **Scenario** section, click **Plot Scenario > New Scenario**.

In the Simulation Scenario dialog box, set the **Simulation duration** to 30 seconds.

In the **Unmeasured Disturbances** table, in the **Signal** drop-down list, select **Step**.

In the **Time** column, specify a step time of 5 seconds.

Simulation Scenario

**Simulation Settings**

Plant used in simulation: Default (controller internal model)

Simulation duration (seconds) 30

Run open-loop simulation       Use unconstrained MPC

Preview references (look ahead)       Preview measured disturbances (look ahead)

**Reference Signals (setpoints for all outputs)**

Channel	Name	Nominal	Signal	Size	Time	Period
r(1)	Ref of T	0	Constant			
r(2)	Ref of CA	0	Constant			

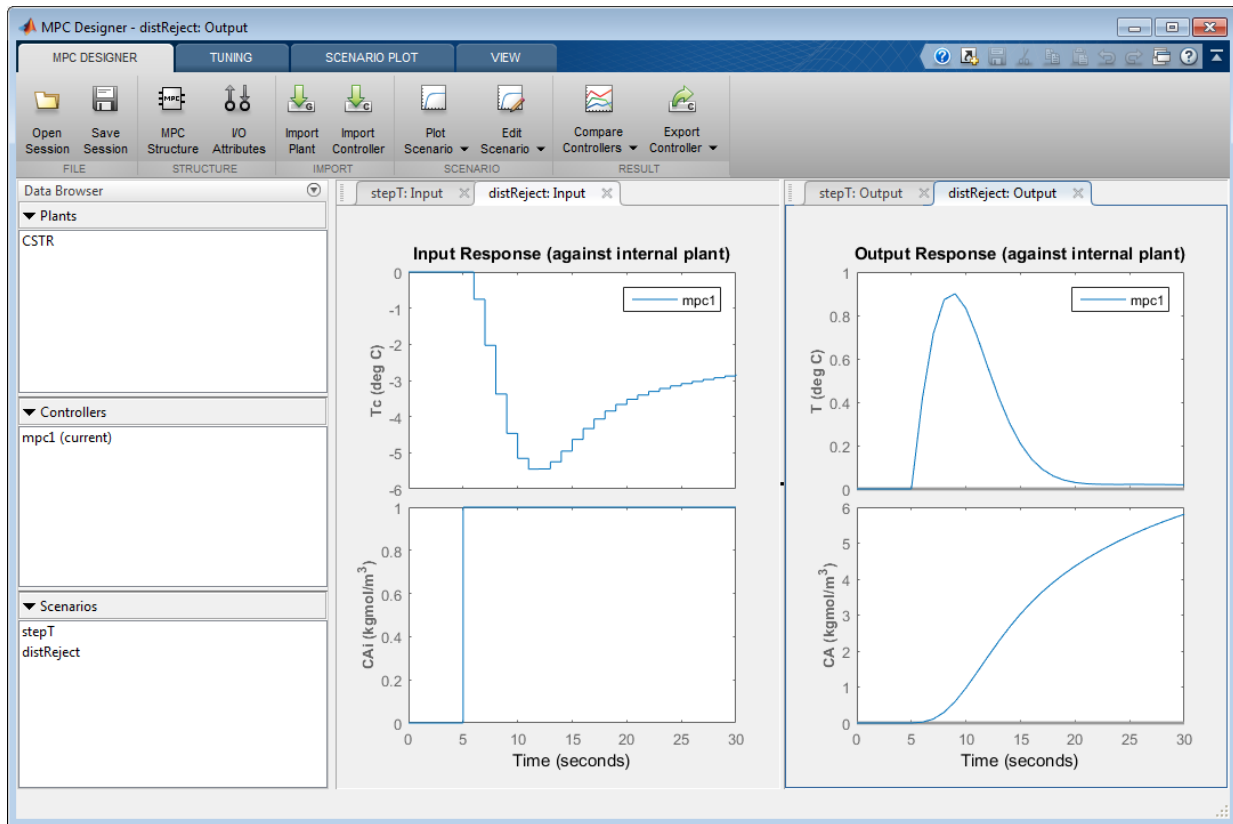
**Unmeasured Disturbances (inputs to UD channels)**

Channel	Name	Nominal	Signal	Size	Time	Period
u(2)	CAi	0	Step	1	5	

Click **OK**.

The app adds new scenario to the **Data Browser** and creates new corresponding **Input Response** and **Output Response** plots.

In the **Data Browser**, in the **Scenarios** section, double-click NewScenario, and rename it distReject.



In the **Output Response** plots, the controller returns the reactor temperature,  $T$ , to a value near its setpoint as expected. However, the required control actions cause an increase in the output concentration,  $CA$  to  $6 \text{ kgmol/m}^3$ .

#### Specify Concentration Output Constraint

Previously, you defined the controller tuning weights to achieve the primary control objective of tracking the reactor temperature setpoint with zero steady-state error. Doing so enables the unmeasured reactor concentration to vary freely. Suppose that unwanted reactions occur once the reactor concentration exceeds a  $3 \text{ kgmol/m}^3$ . To limit the reactor concentration, specify an output constraint.

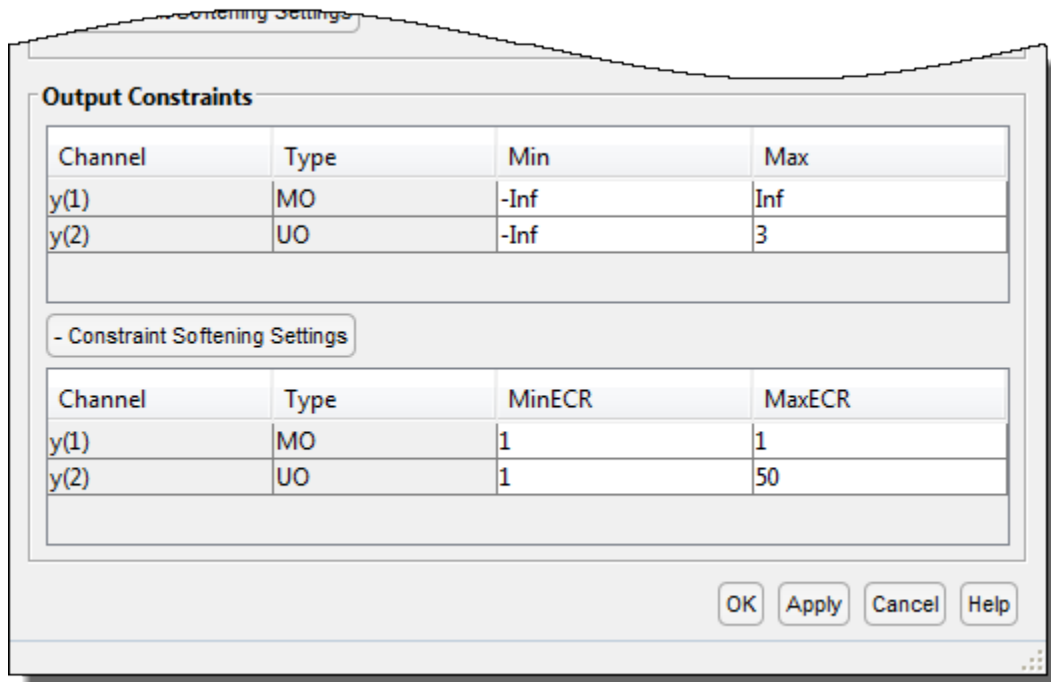
On the **Tuning** tab, in the **Design** section, click **Constraints**.



In the Constraints dialog box, in the **Output Constraints** section, the second row of the table, specify a **Max** unmeasured output (UO) value of 3.

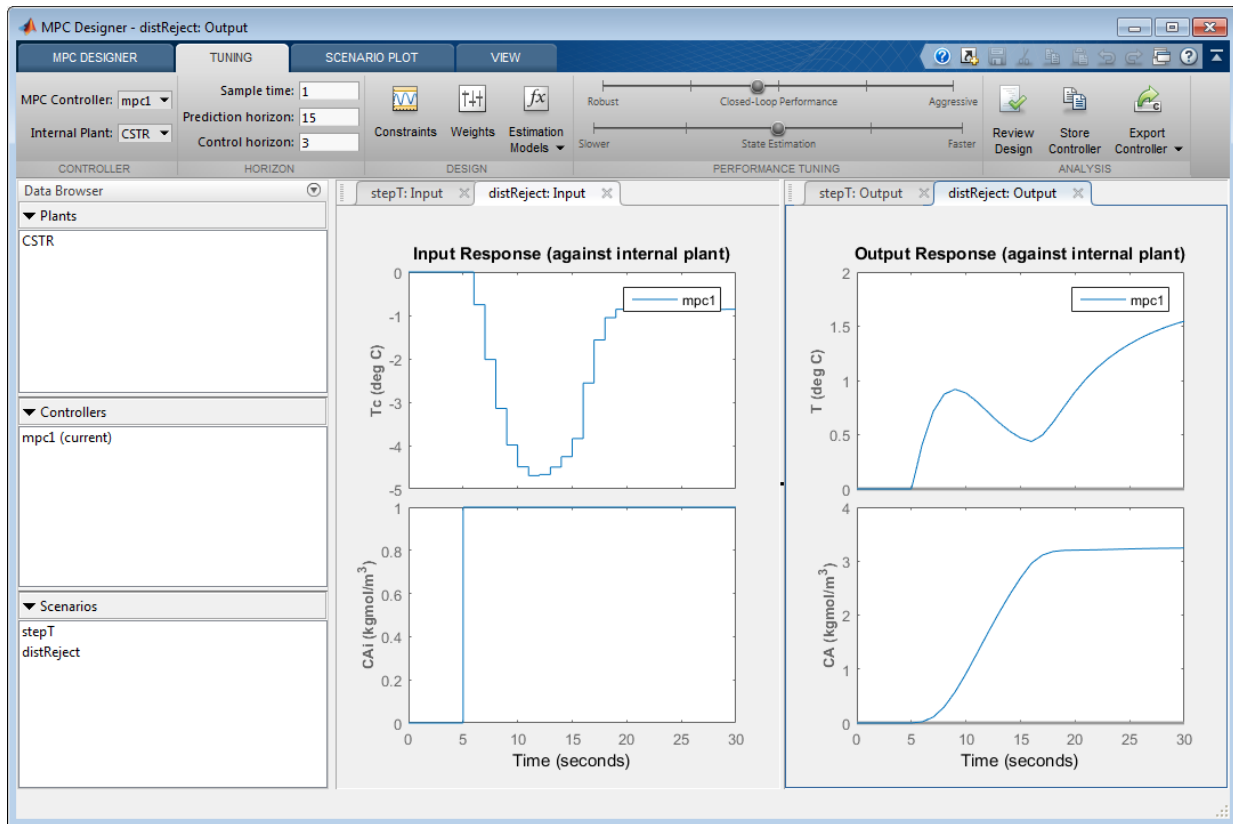
In the **Output Constraints** section, click **Constraint Softening Settings**.

By default, all output constraints are soft, meaning that their **MinECR** and **MaxECR** values are greater than zero. To soften the unmeasured output (UO) constraint further, increase its **MaxECR** value.



Click **OK**.


### 3 Designing Controllers Using MPC Designer



In the **Output Response** plots, once the reactor concentration,  $CA$ , approaches 3 kgmol/m<sup>3</sup>, the reactor temperature,  $T$ , starts to increase. Since there is only one manipulated variable, the controller makes a compromise between the two competing control objectives: Temperature control and constraint satisfaction. A softer output constraint enables the controller to sacrifice the constraint requirement more to achieve improved temperature tracking.

Since the output constraint is soft, the controller maintain adequate temperature control by allowing a small concentration constraint violation. In general, depending on your application requirements, you can experiment with different constraint settings to achieve an acceptable control objective compromise.

## Export Controller

In the **Tuning** tab, in the **Analysis** section, click **Export Controller**  to save the tuned controller, mpc1, to the MATLAB workspace.

## References

[1] Seborg, D. E., T. F. Edgar, and D. A. Mellichamp, *Process Dynamics and Control*, 2nd Edition, Wiley, 2004, pp. 34-36 and 94-95.

## See Also

**MPC Designer**

## More About

- “Specify Constraints”
- “Tune Weights”
- “Design MPC Controller in Simulink” on page 5-2

## Test Controller Robustness

This example shows how to test the sensitivity of your model predictive controller to prediction errors using simulations.

It is good practice to test the robustness of your controller to prediction errors. Classical phase and gain margins are one way to quantify robustness for a SISO application. Robust Control Toolbox™ software provides sophisticated approaches for MIMO systems. It can also be helpful to run simulations.

### Define Plant Model

For this example, use the CSTR model described in “Design Controller Using MPC Designer” on page 3-2.

```
A = [-0.0285 -0.0014; -0.0371 -0.1476];  
B = [-0.0850 0.0238; 0.0802 0.4462];  
C = [0 1; 1 0];  
D = zeros(2,2);  
CSTR = ss(A,B,C,D);
```

Specify the signal names and signal types for the plant.

```
CSTR.InputName = {'T_c', 'C_A_i'};  
CSTR.OutputName = {'T', 'C_A'};  
CSTR.StateName = {'C_A', 'T'};  
CSTR = setmpcsignals(CSTR, 'MV', 1, 'UD', 2, 'MO', 1, 'UO', 2);
```

Open **MPC Designer**, and import the plant model.

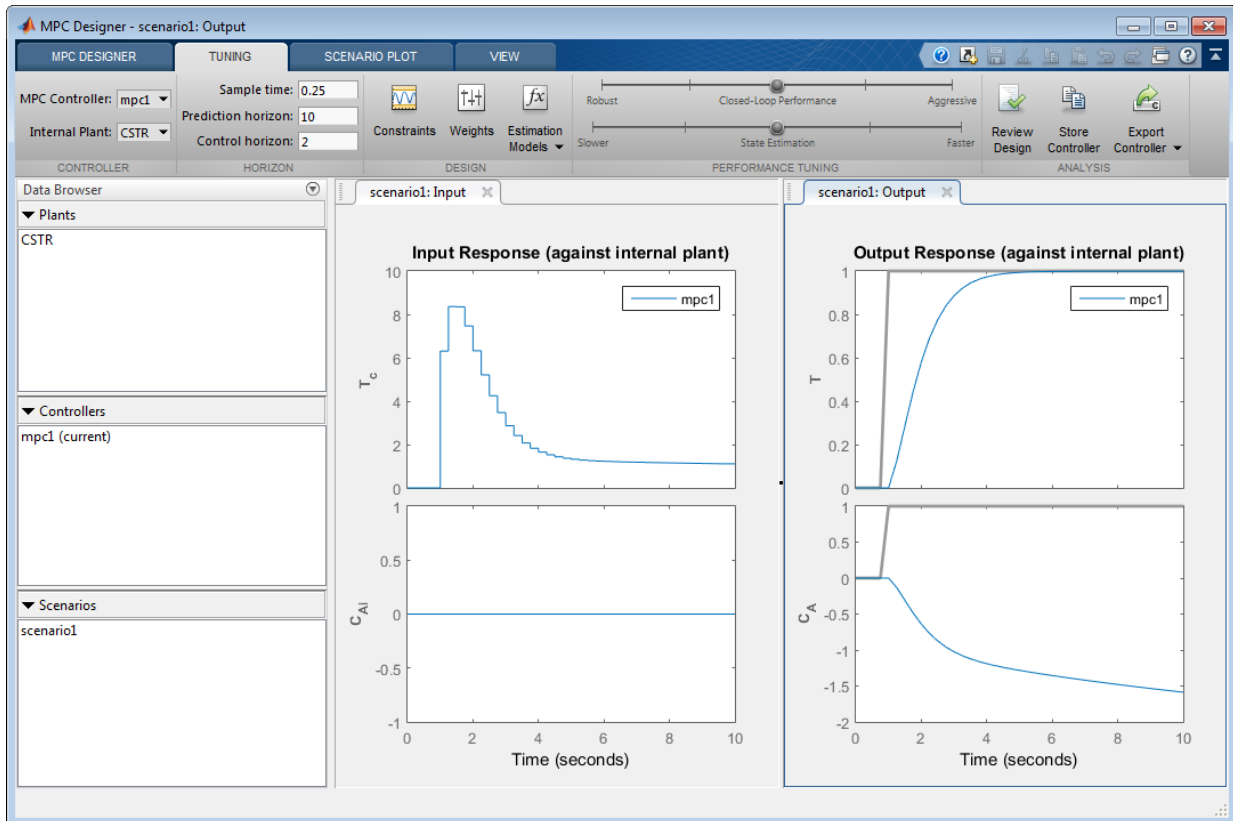
```
mpcDesigner(CSTR)
```

The app imports the plant model and adds it to the **Data Browser**. It also creates a default controller and a default simulation scenario.

### Design Controller

Typically, you would design your controller by specifying scaling factors, defining constraints, and adjusting tuning weights. For this example, modify the controller sample time, and keep the other controller settings at their default values.

In **MPC Designer**, on the **Tuning** tab, in the **Horizon** section, specify a **Sample time** of 0.25 seconds.



The **Input Response** and **Output Response** plots update to reflect the new sample time.

### Configure Simulation Scenario

To test controller setpoint tracking and unmeasured disturbance rejection, modify the default simulation scenario.

In the **Data Browser**, in the **Scenarios** sections, right-click `scenario1`, and select **Edit**.

In the Simulation Scenario dialog box, specify a **Simulation duration** of 50 seconds.

In the **Reference Signals** table, keep the default Ref of  $T$  setpoint configuration, which simulates a unit-step change in the reactor temperature.

To hold the concentration setpoint at its nominal value, in the second row, in the **Signal** drop-down list, select Constant.

Simulate a unit-step unmeasured disturbance at a time of 25 seconds. In the **Unmeasured Disturbances** table, in the **Signal** drop-down list, select Step, and specify a **Time** of 25.

**Simulation Scenario: scenario1**

**Simulation Settings**

Plant used in simulation: Default (controller internal model)

Simulation duration (seconds): 50

Run open-loop simulation       Use unconstrained MPC

Preview references (look ahead)       Preview measured disturbances (look ahead)

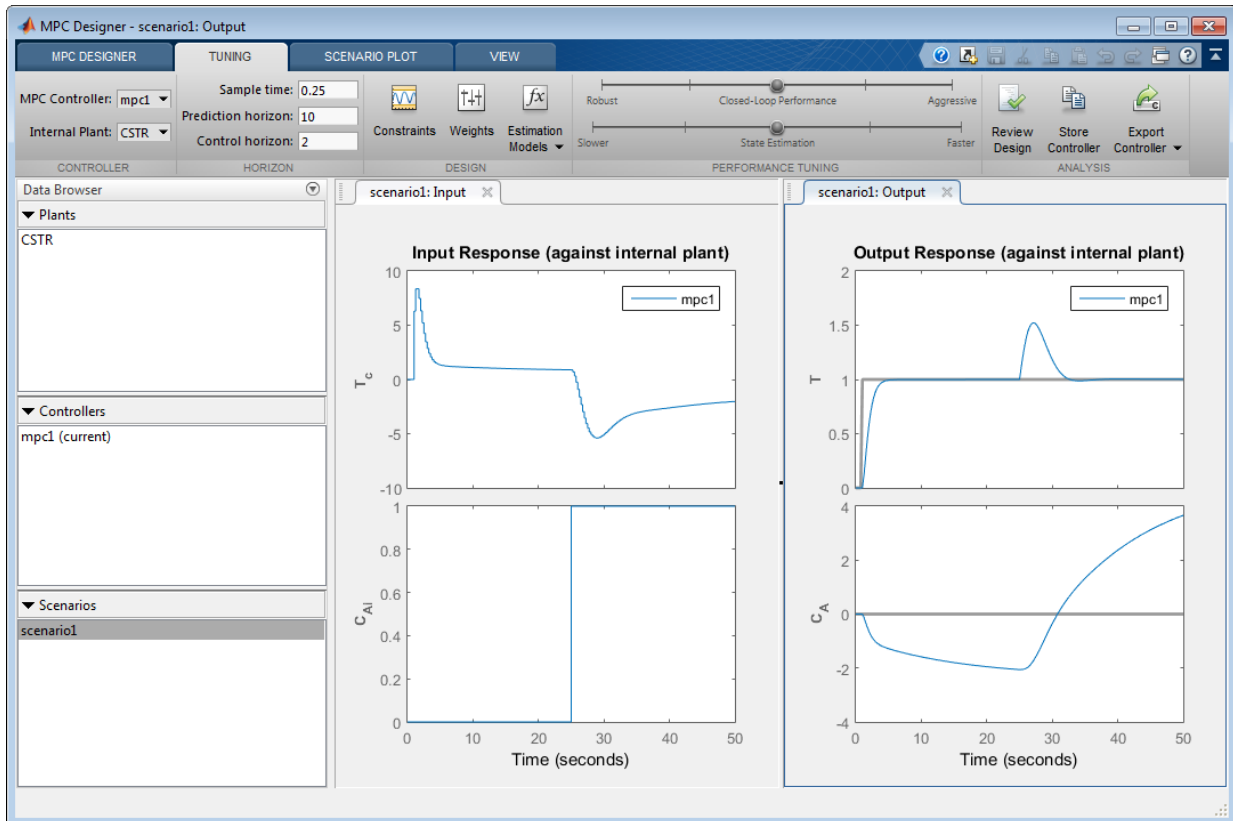
**Reference Signals (setpoints for all outputs)**

Channel	Name	Nominal	Signal	Size	Time	Period
r(1)	Ref of T	0	Step	1	1	
r(2)	Ref of C_A	0	Constant			

**Unmeasured Disturbances (inputs to UD channels)**

Channel	Name	Nominal	Signal	Size	Time	Period
u(2)	C_A_i	0	Step	1	25	

Click **OK**.



The app runs the simulation scenario, and updates the response plots to reflect the new simulation settings. For this scenario, the internal model of the controller is used in the simulation. Therefore, the simulation results represent the controller performance when there are no prediction errors.

### Define Perturbed Plant Models

Suppose that you want to test the sensitivity of your controller to plant changes that modify the effect of the coolant temperature on the reactor temperature. You can simulate such changes by perturbing element  $B(2, 1)$  of the CSTR input-to-state matrix.

In the MATLAB Command Window, specify the perturbation matrix.

```
dB = [0 0; 0.05 0];
```

Create the two perturbed plant models.

```
perturbUp = CSTR;  
perturbUp.B = perturbUp.B + dB;
```

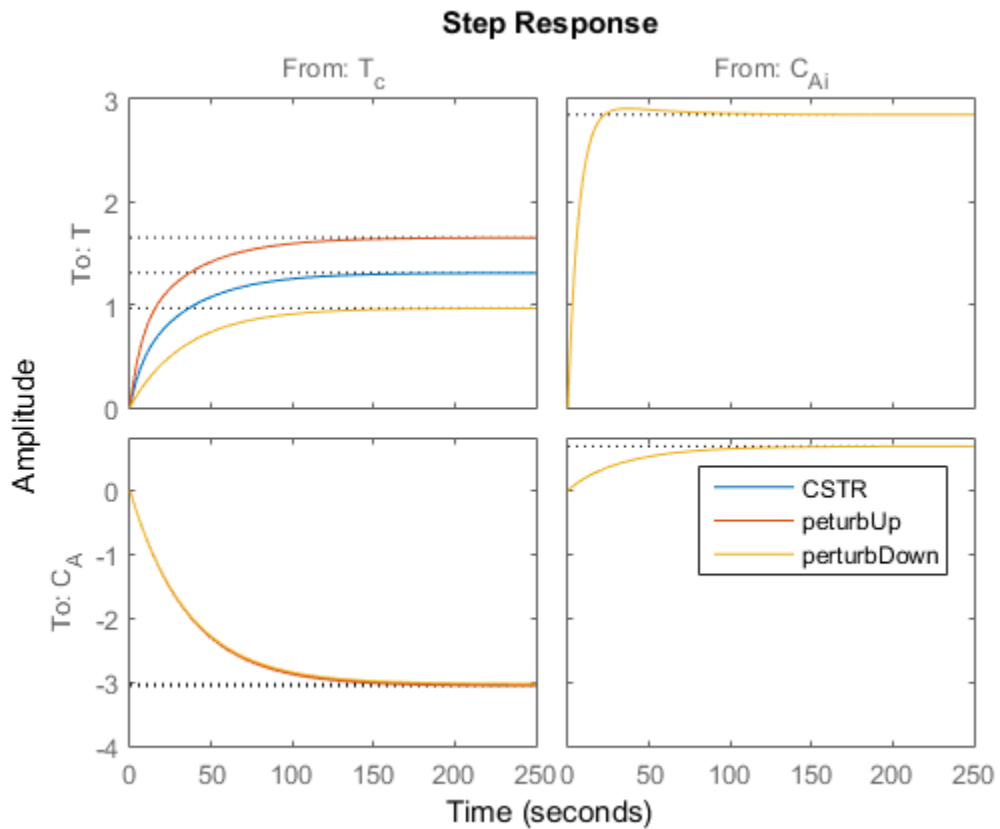
```
perturbDown = CSTR;  
perturbDown.B = perturbDown.B - dB;
```

#### **Examine Step Responses of Perturbed Plants**

To examine the effects of the plant perturbations, plot the plant step responses.

```
step(CSTR, perturbUp, perturbDown)  
legend('CSTR', 'peturbUp', 'perturbDown')
```



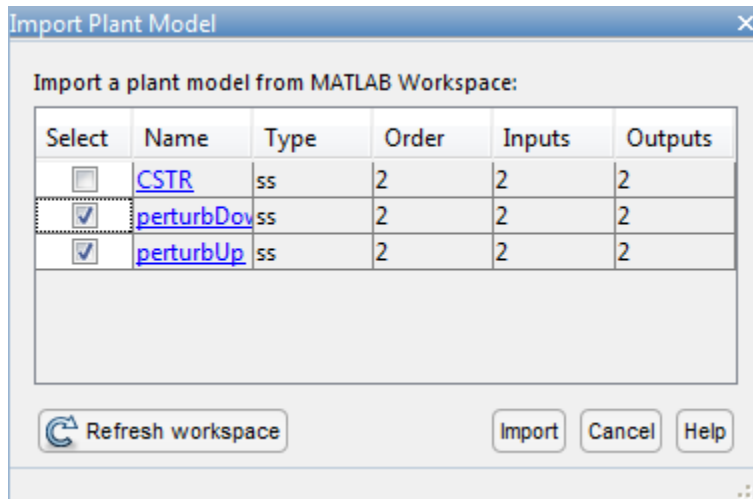


Perturbing element  $B(2, 1)$  of the CSTR plant changes the magnitude of the response of the reactor temperature,  $T$ , to changes in the coolant temperature,  $T_c$ .

### Import Perturbed Plants

In **MPC Designer**, on the **MPC Designer** tab, in the **Import** section, click **Import Plant**.

In the Import Plant Model dialog box, select the perturbUp and perturbDown models.



Click **Import**.

The app imports the models and adds them to the **Data Browser**.

#### **Define Perturbed Plant Simulation Scenarios**

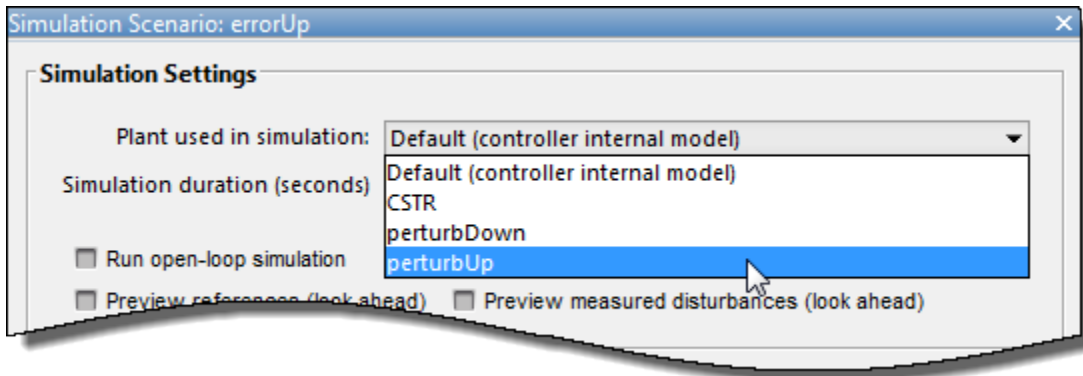
Create two simulation scenarios that use the perturbed plant models.

In the **Data Browser**, in the **Scenarios** section, double-click `scenario1`, and rename it `accurate`.

Right-click `accurate`, and click **Copy**. Rename `accurate_Copy` to `errorUp`.

Right-click `errorUp`, and select **Edit**.

In the Simulation Scenario dialog box, in the **Plant used in simulation** drop-down list, select `perturbUp`.

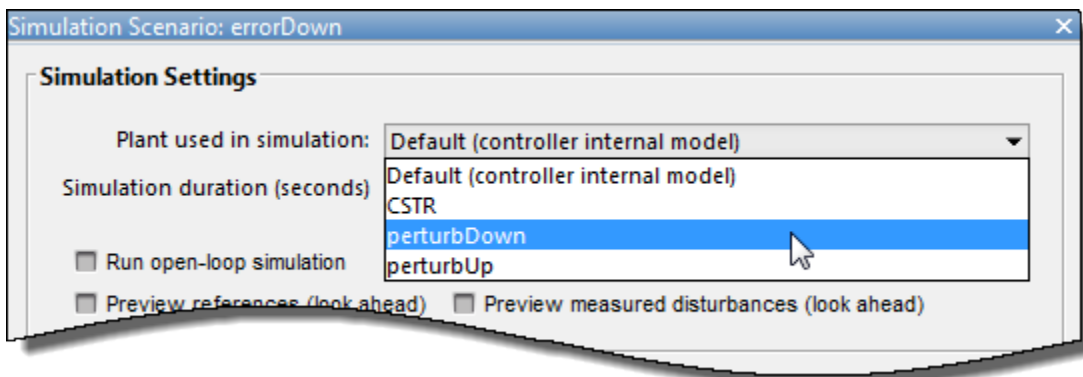


Click **OK**.

Repeat this process for the second perturbed plant.

Copy the accurate scenario and rename it to errorDown.

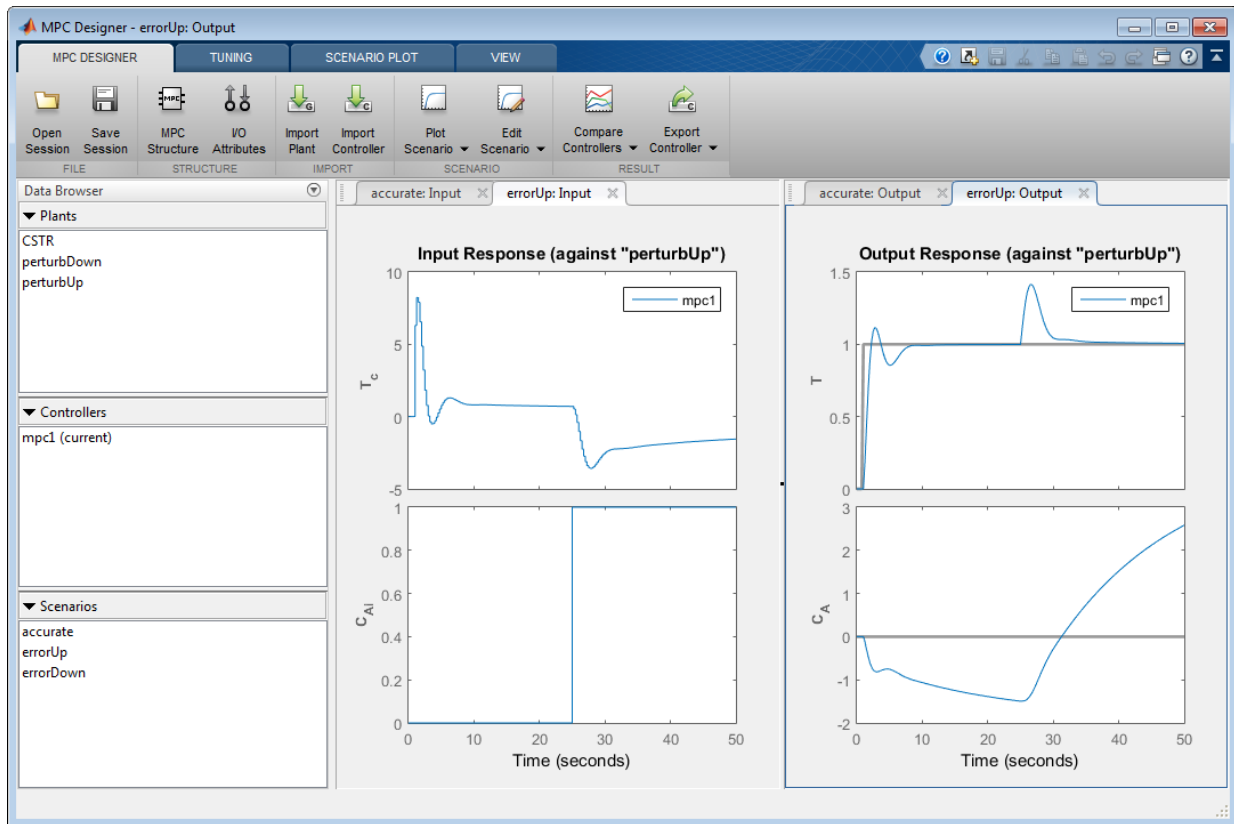
Edit errorDown, selecting the perturbDown plant.



### Examine errorUp Simulation Response

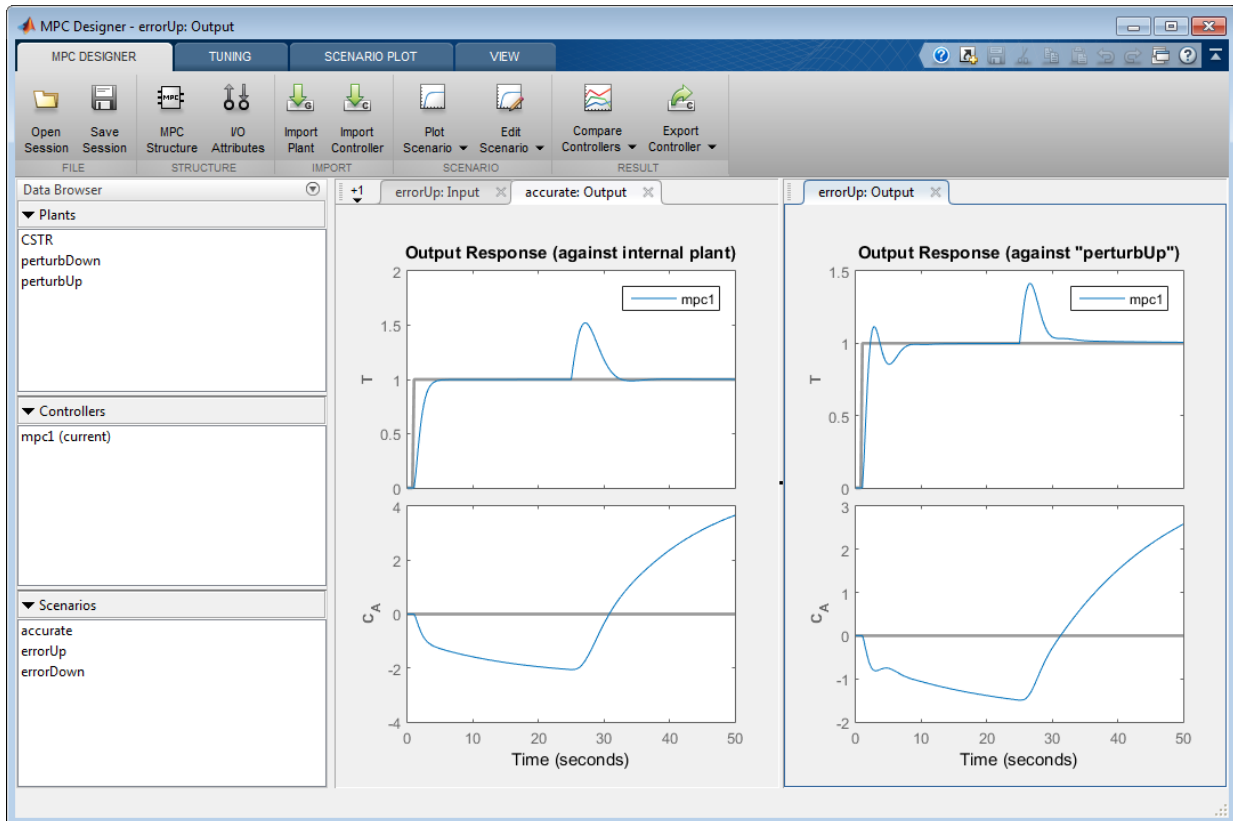
On the **MPC Designer** tab, in the **Scenario** section, click **Plot Scenario > errorUp**.

### 3 Designing Controllers Using MPC Designer



The app creates the **errorUp: Input** and **errorUp: Output** tabs, and displays the simulation response.

To view the accurate and errorUp responses side-by-side, drag the **accurate: Output** tab into the left plot panel.



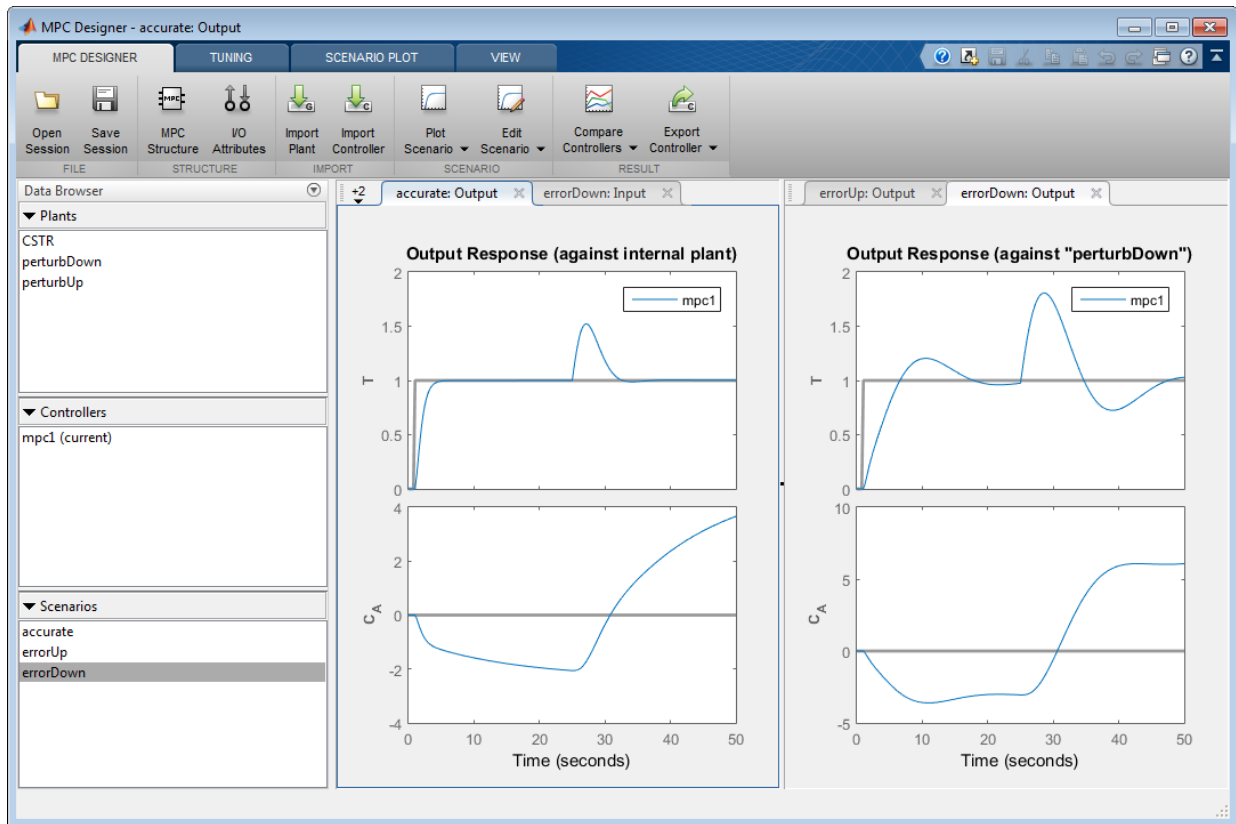
The perturbation creates a plant, `perturbUp`, that responds faster to manipulated variable changes than the controller predicts. On the **errorUp: Output** tab, in the **Output Response** plot, the **T** setpoint step response has about 10% overshoot with a longer settling time. Although this response is worse than the response of the `accurate` simulation, it is still acceptable. The faster plant response leads to a smaller peak error due to the unmeasured disturbance. Overall, the controller is able to control the `perturbUp` plant successfully despite the internal model prediction error.

### Examine `errorDown` Simulation Response

On the **MPC Designer** tab, in the **Scenario** section, click **Plot Scenario** > **errorDown**.

The app creates the **errorDown: Input** and **errorDown: Output** tabs, and displays the simulation response.

To view the accurate and errorDown responses side-by-side, click the **accurate: Output** tab in the left display panel.



The perturbation creates a plant, `perturbDown`, that responds slower to manipulated variable changes than the controller predicts. On the **errorDown: Output** tab, in the **Output Response** plot, the setpoint tracking and disturbance rejection are worse than for the unperturbed plant.

Depending on the application requirements and the real-world potential for such plant changes, the degraded response for the `perturbDown` plant may require modifications to the controller design.

## See Also

**MPC Designer** | `mpc`

## More About

- “Design Controller Using MPC Designer” on page 3-2
- “Test an Existing Controller” on page 5-23

## Design MPC Controller for Plant with Delays

This example shows how to design an MPC controller for a plant with delays using **MPC Designer**.

### Plant Model

An example of a plant with delays is the distillation column model:

$$\begin{bmatrix} y_1 \\ y_2 \end{bmatrix} = \begin{bmatrix} \frac{12.8e^{-s}}{16.7s+1} & \frac{-18.9e^{-3s}}{21.0s+1} & \frac{3.8e^{-8.1s}}{14.9s+1} \\ \frac{6.6e^{-7s}}{10.9s+1} & \frac{-19.4e^{-3s}}{14.4s+1} & \frac{4.9e^{-3.4s}}{13.2s+1} \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix}$$

Outputs  $y_1$  and  $y_2$  represent measured product purities. The model consists of six transfer functions, one for each input/output pair. Each transfer function is a first-order system with a delay. The longest delay in the model is 8.1 minutes.

Specify the individual transfer functions for each input/output pair. For example, g12 is the transfer function from input  $u_2$  to output  $y_1$ .

```
g11 = tf(12.8,[16.7 1],'I0delay',1.0,'TimeUnit','minutes');
g12 = tf(-18.9,[21.0 1],'I0delay',3.0,'TimeUnit','minutes');
g13 = tf(3.8,[14.9 1],'I0delay',8.1,'TimeUnit','minutes');
g21 = tf(6.6,[10.9 1],'I0delay',7.0,'TimeUnit','minutes');
g22 = tf(-19.4,[14.4 1],'I0delay',3.0,'TimeUnit','minutes');
g23 = tf(4.9,[13.2 1],'I0delay',3.4,'TimeUnit','minutes');
DC = [g11 g12 g13;
      g21 g22 g23];
```

### Configure Input and Output Signals

Define the input and output signal names.

```
DC.InputName = {'Reflux Rate','Steam Rate','Feed Rate'};
DC.OutputName = {'Distillate Purity','Bottoms Purity'};
```

Alternatively, you can specify the signal names in **MPC Designer**, on the **MPC Designer** tab, by clicking **I/O Attributes**.

Specify the third input, the feed rate, as a measured disturbance (MD).



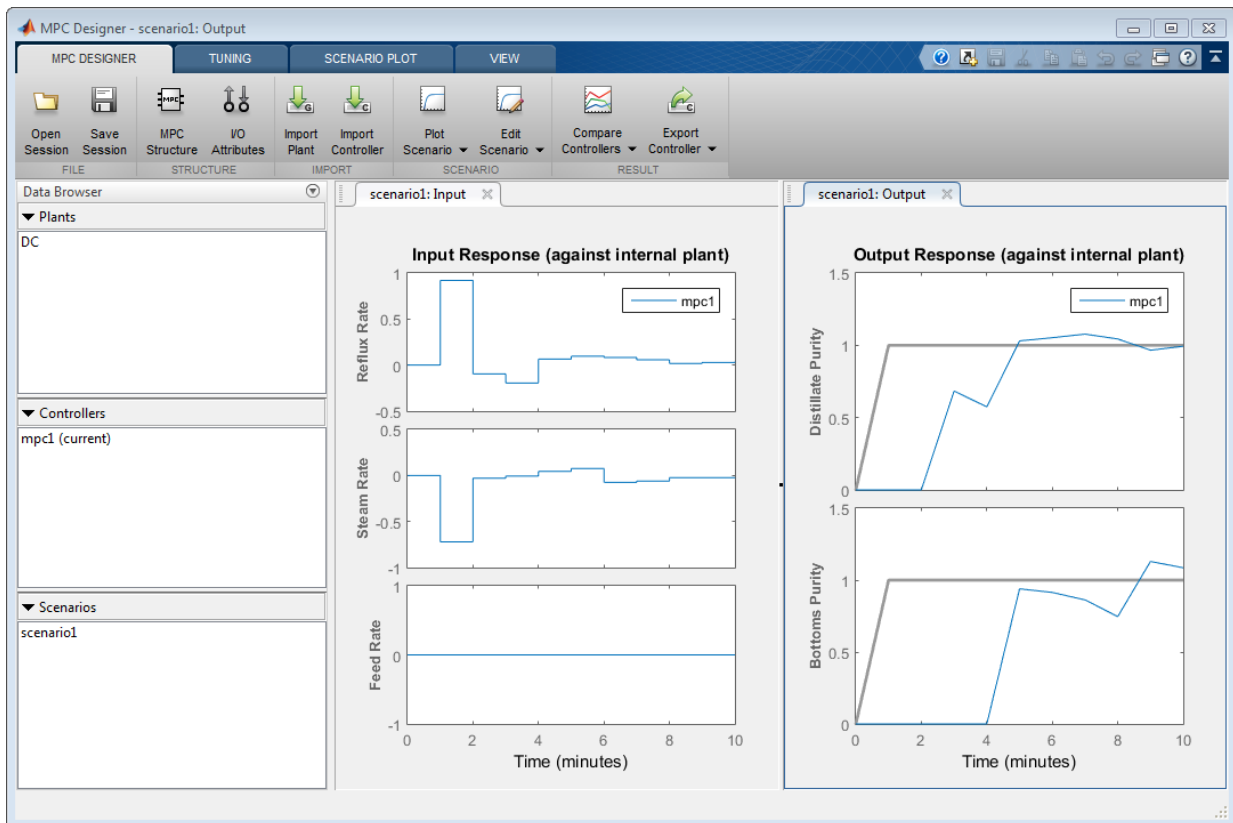
```
DC = setmpcsignals(DC, 'MD', 3);
```

Since they are not explicitly specified in `setmpcsignals`, all other input signals are configured as manipulated variables (MV), and all output signals are configured as measured outputs (MO) by default.

## Open MPC Designer

Open **MPC Designer** importing the plant model.

```
mpcDesigner(DC)
```



When launched with a continuous-time plant model, such as DC, the default controller sample time is 1 in the time units of the plant. If the plant is discrete time, the controller sample time is the same as the plant sample time.

**MPC Designer** imports the specified plant to the **Data Browser**. The following are also added to the **Data Browser**:

- `mpc1` — Default MPC controller created using DC as its internal model.
- `scenario1` — Default simulation scenario.

The app runs the simulation scenario and generates input and output response plots.

#### Specify Prediction and Control Horizons

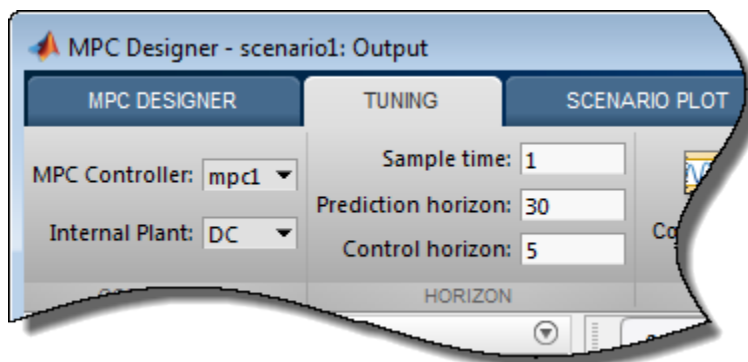
For a plant with delays, it is good practice to specify the prediction and control horizons such that

$$P - M \gg t_{d,max} / \Delta t$$

where,

- $P$  is the prediction horizon.
- $M$  is the control horizon.
- $t_{d,max}$  is the maximum delay, which is 8.1 minutes for the DC model.
- $\Delta t$  is the controller **Sample time**, which is 1 minute by default.

On the **Tuning** tab, in the **Horizon** section, specify a **Prediction horizon** of 30 and a **Control horizon** of 5.



After you change the horizons, the **Input Response** and **Output Response** plots for the default simulation scenario are automatically updated.

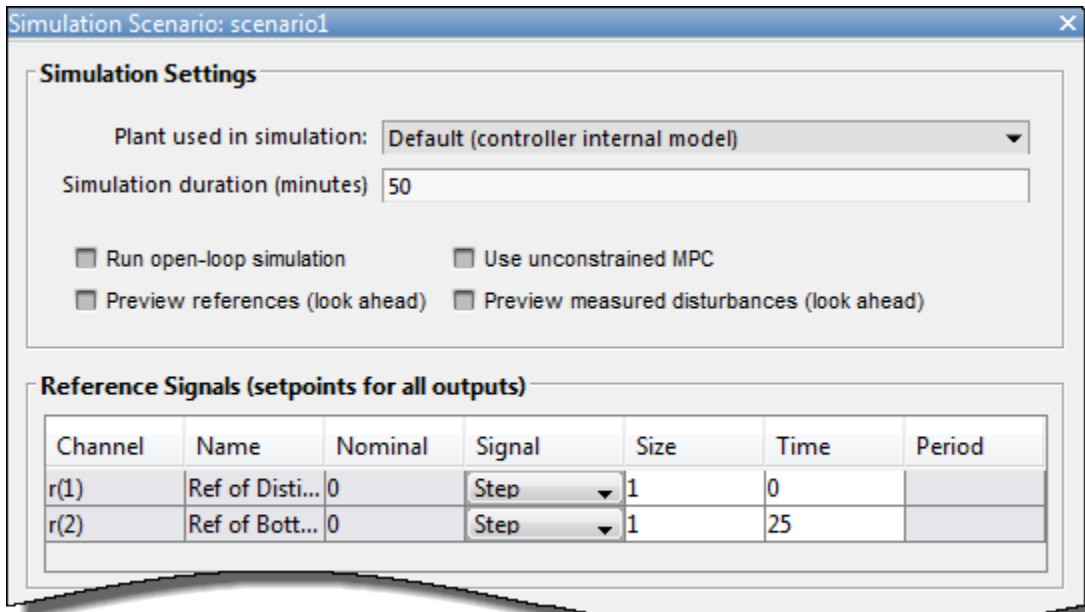
## Simulate Controller Step Responses

On the **MPC Designer** tab, in the **Scenario** section, click **Edit Scenario > scenario1**. Alternatively, in the **Data Browser**, right-click `scenario1` and select **Edit**.

In the Simulation Scenario dialog box, specify a **Simulation duration** of 50 minutes.

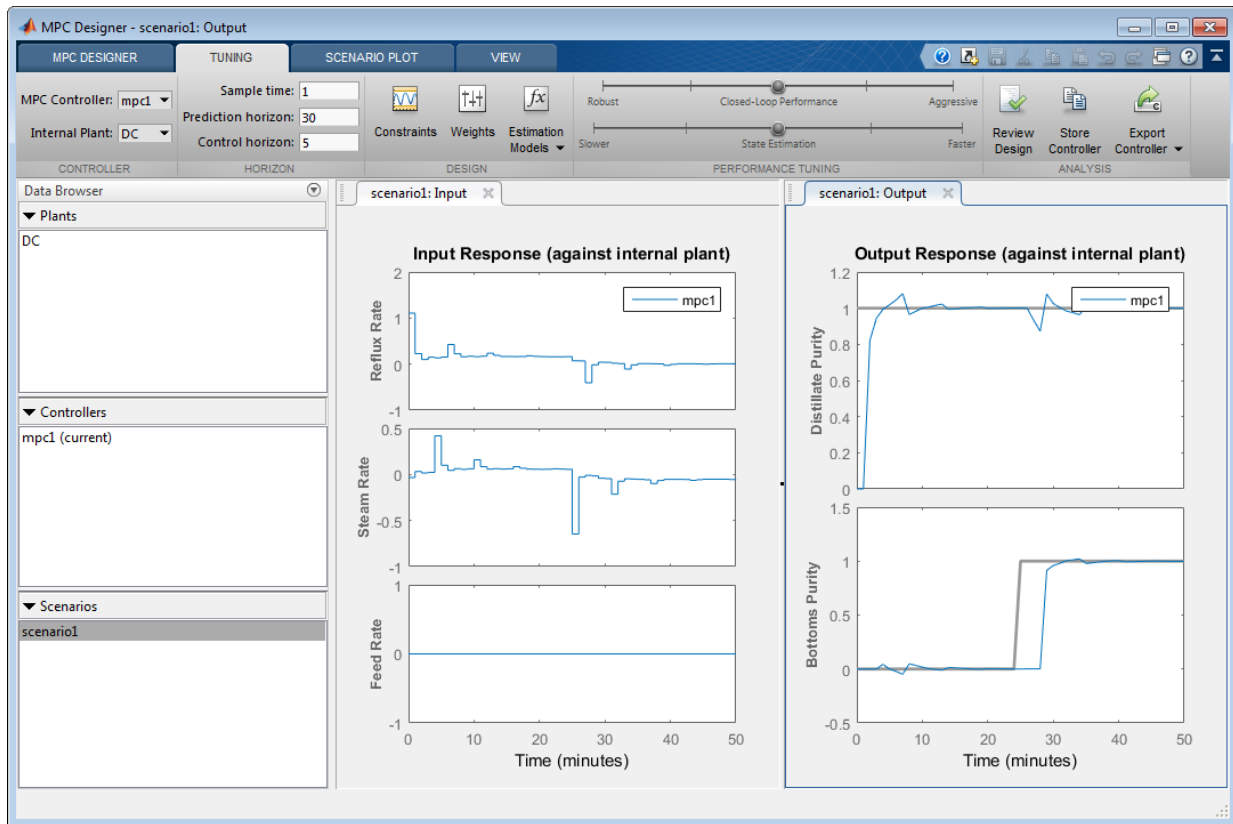
In the **Reference Signals** table, in the **Signal** drop-down list, select **Step** for both outputs to simulate step changes in their setpoints.

Specify a step **Time** of 0 for reference **r(1)**, the distillate purity, and a step time of 25 for **r(2)**, the bottoms purity.



Click **OK**.

The app runs the simulation with the new scenario settings and updates the input and output response plots.



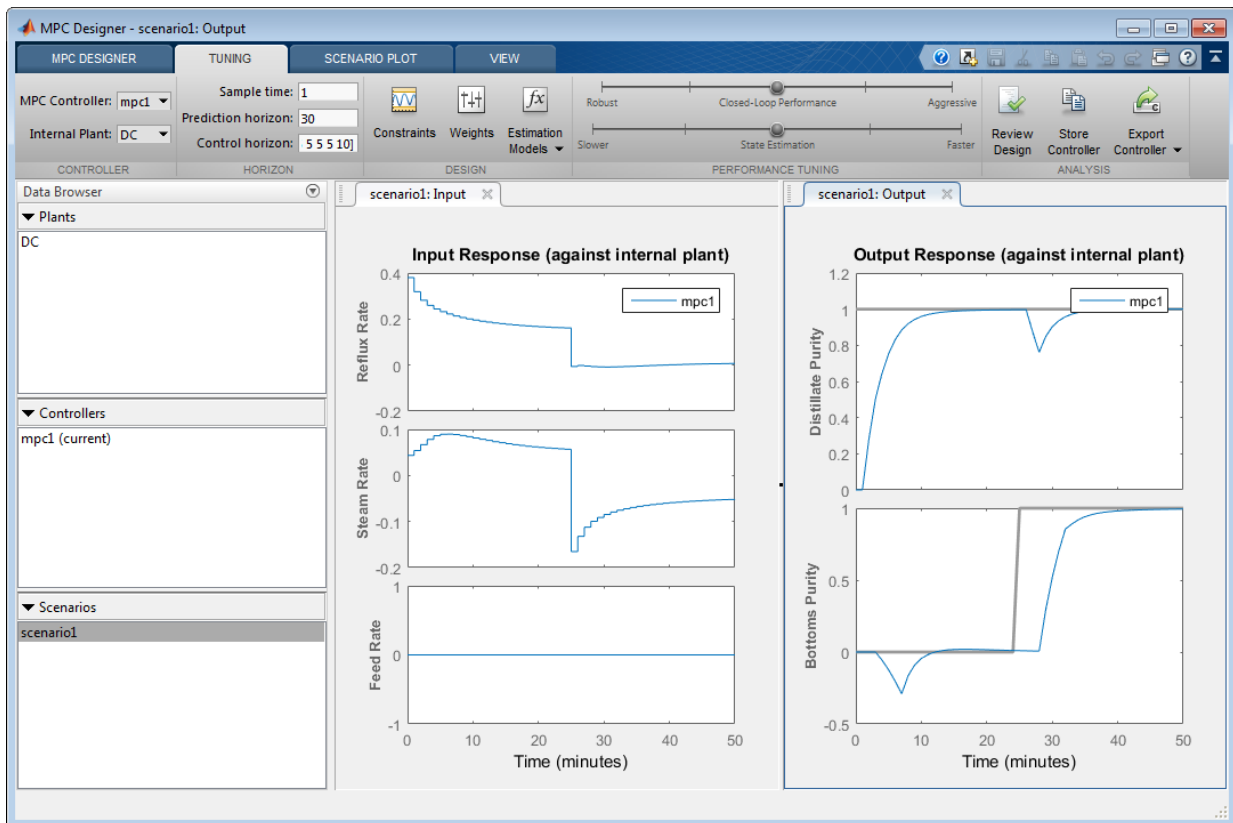
The **Input Response** plots show the optimal control moves generated by the controller. The controller reacts immediately in response to the setpoint changes, changing both manipulated variables. However, due to the plant delays, the effects of these changes are not immediately reflected in the **Output Response** plots. The **Distillate Purity** output responds after 1 minute, which corresponds to the minimum delay from  $g_{11}$  and  $g_{12}$ . Similarly, the **Bottoms Purity** output responds 3 minutes after the step change, which corresponds to the minimum delay from  $g_{21}$  and  $g_{22}$ . After the initial delays, both signals reach their setpoints and settle quickly. Changing either output setpoint disturbs the response of the other output. However, the magnitudes of these interactions are less than 10% of the step size.

Additionally, there are periodic pulses in the manipulated variable control actions as the controller attempts to counteract the delayed effects of each input on the two outputs.

## Improve Performance Using Manipulated Variable Blocking

Use manipulated variable blocking to divide the prediction horizon into blocks, during which manipulated variable moves are constant. This technique produces smoother manipulated variable adjustments with less oscillation and smaller move sizes.

To use manipulated variable blocking, on the **Tuning** tab, specify the **Control horizon** as a vector of block sizes, [5 5 5 5 10].



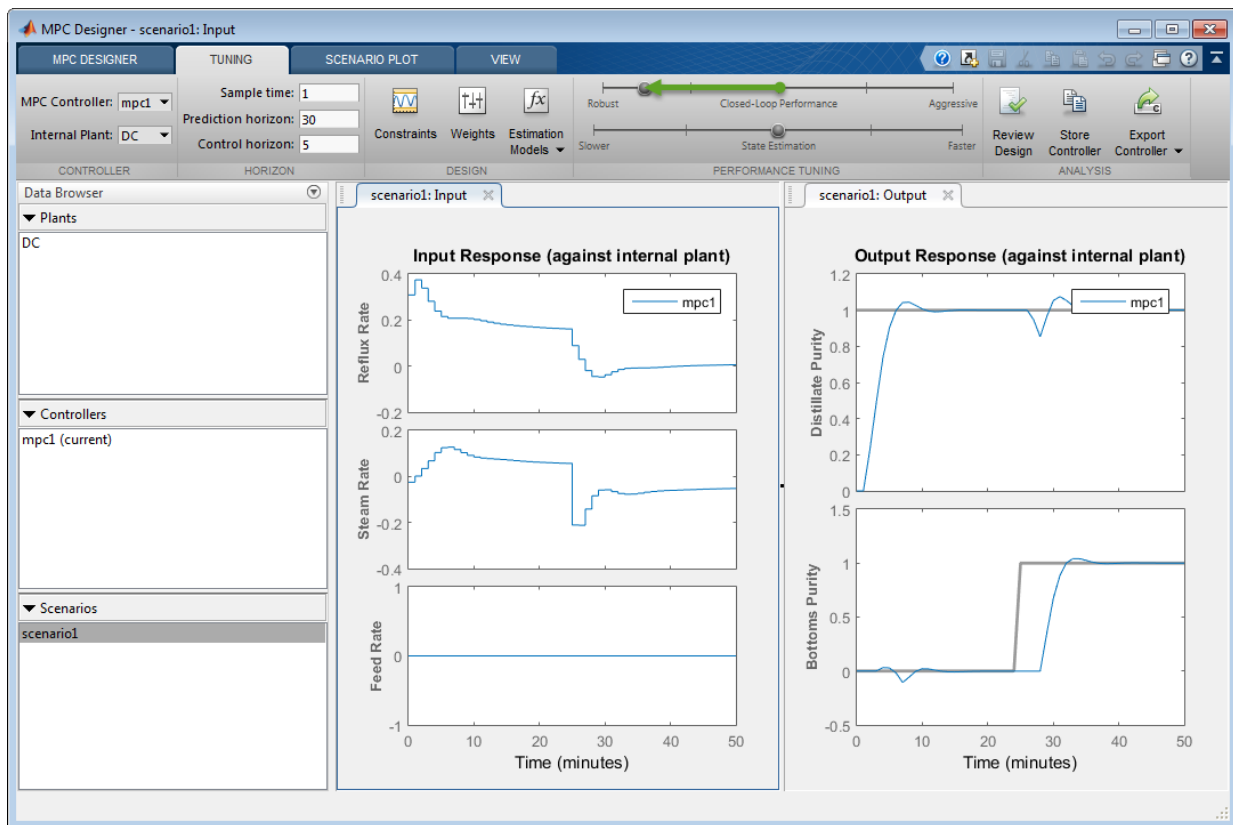
The initial manipulated variable moves are much smaller and the moves are less oscillatory. The trade-off is a slower output response, with larger interactions between the outputs.

#### Improve Performance By Tuning Controller Weights

Alternatively, you can produce smooth manipulated variable moves by adjusting the tuning weights of the controller.

Set the **Control horizon** back to the previous value of 5.

In the **Performance Tuning** section, drag the **Closed-Loop Performance** slider to the left towards the **Robust** setting.



As you move the slider to the left, the manipulated variable moves become smoother and the output response becomes slower.

## References

[1] Wood, R. K., and M. W. Berry, *Chem. Eng. Sci.*, Vol. 28, pp. 1707, 1973.

## See Also

**MPC Designer**

## More About

- “Manipulated Variable Blocking”
- “Design Controller Using MPC Designer” on page 3-2
- “Specify Multi-Input Multi-Output Plants” on page 2-17

## Design MPC Controller for Nonsquare Plant

This topic shows how to configure an MPC controller for a nonsquare plant with unequal numbers of manipulated variables and outputs. Model Predictive Control Toolbox software supports plants with an excess of manipulated variables or plant with an excess of outputs.

### More Outputs Than Manipulated Variables

When there are excess outputs, you cannot hold each at a setpoint. In this case, you have two options:

- Specify that certain outputs do not need to be held at setpoints by setting their tuning weights to zero.

The controller does not enforce setpoints on outputs with zero weight, and the outputs are free to vary. If the plant has  $N_e$  more outputs than manipulated variables, setting  $N_e$  output weights to zero enables the controller to hold the remaining outputs at their setpoints. If any manipulated variables are constrained, one or more output responses can still exhibit steady-state error, depending on the magnitudes of reference and disturbance signals.

Outputs with zero tuning weights can still be useful. If measured, the controller can use the outputs to help estimate the state of the plant. The outputs can also be used as performance indicators or held within an operating region defined by output constraints.

- Enforce setpoints on all outputs by specifying nonzero tuning weights for all of them.

The controller tries to hold all outputs at their respective setpoints. However, due to the limited number of manipulated variables, all output responses exhibit some degree of steady-state error.

You can change the error magnitudes by adjusting the relative values of the output weights. Increasing an output weight decreases the steady-state error in that output at the expense of increased error in the other outputs.

You can configure the output tuning weights at the command line by setting the `Weights.OutputVariables` property of the controller.

To configure output tuning weights in **MPC Designer**, on the **Tuning** tab, in the **Design** section, click **Weights** to open the Weights dialog box.



In the **Output Weights** section, specify the **Weight** for each output variable. For example, if your plant has two manipulated variables and three outputs, you can:

- Set one of the output weights to zero.

The screenshot shows a dialog box titled "Weights (mpc1)" with a close button (X) in the top right corner. It is divided into three sections:

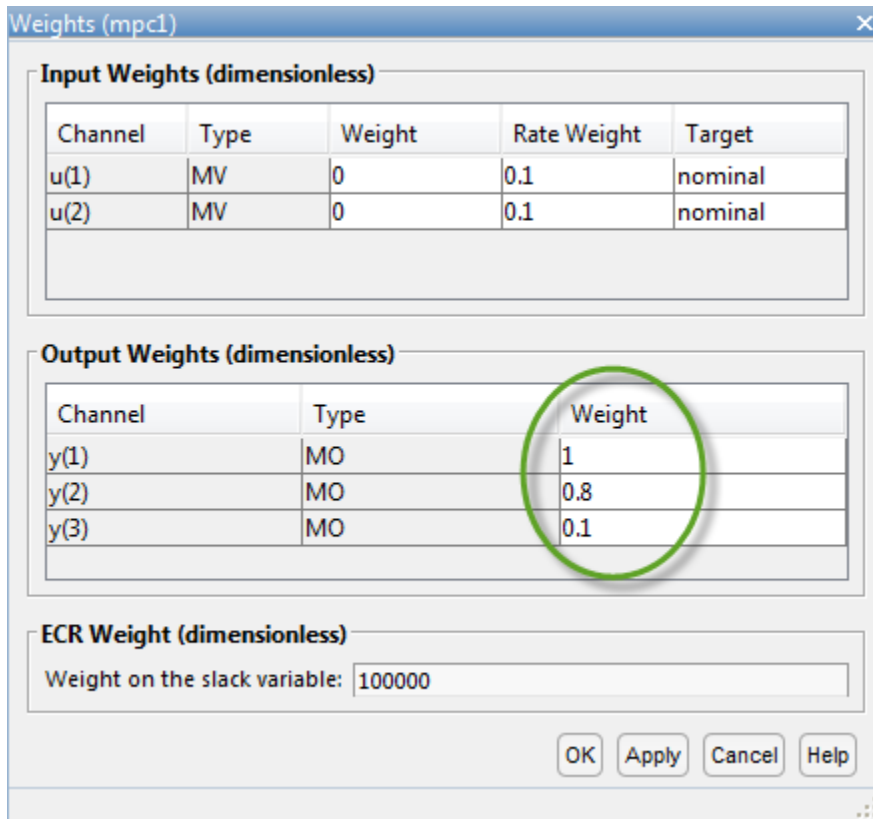
- Input Weights (dimensionless):** A table with 5 columns: Channel, Type, Weight, Rate Weight, and Target.
 

Channel	Type	Weight	Rate Weight	Target
u(1)	MV	0	0.1	nominal
u(2)	MV	0	0.1	nominal
- Output Weights (dimensionless):** A table with 3 columns: Channel, Type, and Weight. The "Weight" column is circled in green.
 

Channel	Type	Weight
y(1)	MO	1
y(2)	MO	1
y(3)	MO	0
- ECR Weight (dimensionless):** A text field labeled "Weight on the slack variable:" with the value "100000".

At the bottom right, there are four buttons: "OK", "Apply", "Cancel", and "Help".

- Set all the weights to nonzero values. Outputs with higher weights exhibit less steady-state error.



## More Manipulated Variables Than Outputs

When there are excessive manipulated variables, the default MPC controller settings allow for error-free output setpoint tracking. However, the manipulated variables values can drift. You can prevent this drift by setting manipulated variable setpoints. If there are  $N_e$  excess manipulated variables, and you hold  $N_e$  of them at target values for economic or operational reasons, the remaining manipulated variables attain the values required to eliminate output steady-state error.

To configure a manipulated variable setpoint at the command line, use the `ManipulatedVariables.Target` controller property. Then specify an input tuning weight using the controller `Weights.ManipulatedVariables` property.

To define a manipulated variable setpoint in **MPC Designer**, on the **Tuning** tab, in the **Design** section, click **Weights**.

In the Weights dialog box, in the **Input Weights** section, specify a nonzero **Weight** value for the manipulated variable.

Specify a **Target** value for the manipulated variable.

The screenshot shows the 'Weights (mpc1)' dialog box with the following sections:

- Input Weights (dimensionless)**: A table with columns Channel, Type, Weight, Rate Weight, and Target.
 

Channel	Type	Weight	Rate Weight	Target
u(1)	MV	0	0.1	nominal
u(2)	MV	0	0.1	nominal
u(3)	MV	0.2	0.1	5
- Output Weights (dimensionless)**: A table with columns Channel, Type, and Weight.
 

Channel	Type	Weight
y(1)	MO	1
y(2)	MO	1
- ECR Weight (dimensionless)**: A text box labeled 'Weight on the slack variable:' with the value '100000'.

Buttons at the bottom: OK, Apply, Cancel, Help.

By default, the manipulated variable **Target** is **nominal**, which means that it tracks the nominal value specified in the controller properties.

---

**Note** Since nominal values apply to all controllers in an **MPC Designer** session, changing a **Nominal Value** updates all controllers in the app. The **Target** value, however, is specific to each individual controller.

---

The magnitude of the manipulated variable weight indicates how much the input can deviate from its setpoint. However, there is a trade-off between manipulated variable target tracking and output reference tracking. If you want to have better output setpoint tracking performance, use a relatively small input weight. If you want the manipulated variable to stay close to its target value, increase its input weight relative to the output weight.

You can also avoid drift by constraining one or more manipulated variables to a narrow operating region using hard constraints. To define constraints in **MPC Designer**, on the **Tuning** tab, in the **Design** section, click **Constraints** to open the Constraints dialog box.

In the **Input Constraints** section, specify **Max** and **Min** constraints values.

## See Also

**Apps**  
**MPC Designer**

**Functions**  
mpc

## More About

- “Tune Weights”
- “Specify Multi-Input Multi-Output Plants” on page 2-17
- “Setting Targets for Manipulated Variables”

# Designing Controllers Using the Command Line

---

- “Design MPC Controller at the Command Line” on page 4-2
- “Simulate Controller with Nonlinear Plant” on page 4-16
- “Compute Steady-State Gain” on page 4-23
- “Extract Controller” on page 4-25
- “Signal Previewing” on page 4-28
- “Update Constraints at Run Time” on page 4-30
- “Tune Weights at Run Time” on page 4-33

# Design MPC Controller at the Command Line

This example shows how create and test a model predictive controller from the command line.

### Define Plant Model

This example uses the plant model described in “Design Controller Using MPC Designer” on page 3-2. Create a state space model of the plant and set some of the optional model properties.

```
A = [-0.0285 -0.0014; -0.0371 -0.1476];  
B = [-0.0850 0.0238; 0.0802 0.4462];  
C = [0 1; 1 0];  
D = zeros(2,2);  
CSTR = ss(A,B,C,D);
```

```
CSTR.InputName = {'T_c', 'C_A_i'};  
CSTR.OutputName = {'T', 'C_A'};  
CSTR.StateName = {'C_A', 'T'};  
CSTR.InputGroup.MV = 1;  
CSTR.InputGroup.UD = 2;  
CSTR.OutputGroup.MO = 1;  
CSTR.OutputGroup.UO = 2;
```

### Create Controller

To improve the clarity of the example, suppress Command Window messages from the MPC controller.

```
old_status = mpcverbosity('off');
```

Create a model predictive controller with a control interval, or sample time, of 1 second, and with all other properties at their default values.

```
Ts = 1;  
MPCobj = mpc(CSTR, Ts);
```

Display the controller properties in the Command Window.

```
display(MPCobj)
```

```
MPC object (created on 26-Feb-2018 19:15:45):
```

```

-----
Sampling time:      1 (seconds)
Prediction Horizon: 10
Control Horizon:   2

Plant Model:

      1 manipulated variable(s)  -->|  2 states  |
      0 measured disturbance(s)  -->|  2 inputs  | --> 1 measured output(s)
      1 unmeasured disturbance(s) -->|  2 outputs | --> 1 unmeasured output(s)
-----

Indices:
(input vector)   Manipulated variables: [1 ]
                  Unmeasured disturbances: [2 ]
(output vector)  Measured outputs: [1 ]
                  Unmeasured outputs: [2 ]

Disturbance and Noise Models:
  Output disturbance model: default (type "getoutdist(MPCobj)" for details)
  Input disturbance model: default (type "getindist(MPCobj)" for details)
  Measurement noise model: default (unity gain after scaling)

Weights:
  ManipulatedVariables: 0
  ManipulatedVariablesRate: 0.1000
  OutputVariables: [1 0]
  ECR: 100000

State Estimation: Default Kalman Filter (type "getEstimator(MPCobj)" for details)

Unconstrained

```

### View and Modify Controller Properties

Display a list of the controller properties and their current values.

```
get(MPCobj)
```

```

          Ts: 1
  PredictionHorizon (P): 10
    ControlHorizon (C): 2
          Model: [1x1 struct]
  ManipulatedVariables (MV): [1x1 struct]

```

```
OutputVariables (OV): [1x2 struct]
DisturbanceVariables (DV): [1x1 struct]
Weights (W): [1x1 struct]
Optimizer: [1x1 struct]
Notes: {}
UserData: []
History: 26-Feb-2020 19:15:45
```

The displayed `History` value will be different for your controller, since it depends on when the controller was created. For a description of the editable properties of an MPC controller, enter `mcpprops` at the command line.

Use dot notation to modify these properties. For example, change the prediction horizon to 15.

```
MPCobj.PredictionHorizon = 15;
```

You can abbreviate property names provided that the abbreviation is unambiguous.

Many of the controller properties are structures containing additional fields. Use dot notation to view and modify these field values. For example, you can set the measurement units for the controller output variables. The `OutputUnit` property is for display purposes only and is optional.

```
MPCobj.Model.Plant.OutputUnit = {'Deg C', 'kmol/m^3'};
```

By default, the controller has no constraints on manipulated variables and output variables. You can view and modify these constraints using dot notation. For example, set constraints for the controller manipulated variable.

```
MPCobj.MV.Min = -10;
MPCobj.MV.Max = 10;
MPCobj.MV.RateMin = -3;
MPCobj.MV.RateMax = 3;
```

You can also view and modify the controller tuning weights. For example, modify the weights for the manipulated variable rate and the output variables.

```
MPCobj.W.ManipulatedVariablesRate = 0.3;
MPCobj.W.OutputVariables = [1 0];
```

You can also define time-varying constraints and weights over the prediction horizon, which shifts at each time step. Time-varying constraints have a nonlinear effect when they



are active. For example, to force the manipulated variable to change more slowly towards the end of the prediction horizon, enter:

```
MPCObj.MV.RateMin = [-4; -3.5; -3; -2.5];
```

```
MPCObj.MV.RateMax = [4; 3.5; 3; 2.5];
```

The -2.5 and 2.5 values are used for the fourth step and beyond.

Similarly, you can specify different output variable weights for each step of the prediction horizon. For example, enter:

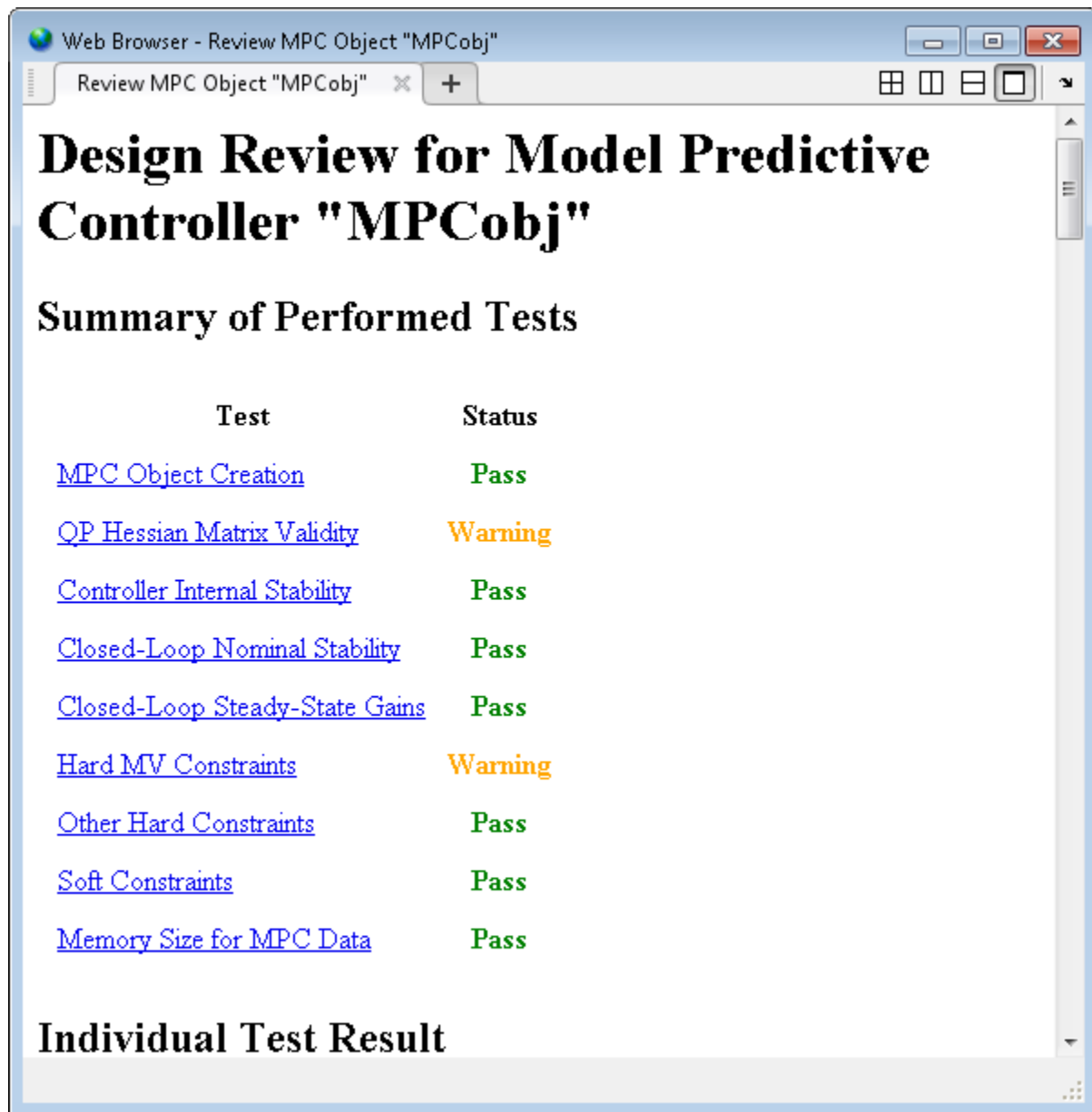
```
MPCObj.W.OutputVariables = [0.1 0; 0.2 0; 0.5 0; 1 0];
```

You can also modify the disturbance rejection characteristics of the controller. See `setEstimator`, `setindist` and `setoutdist` for more information.

### **Review Controller Design**

Generate a report on potential run-time stability and performance issues.

```
review(MPCObj)
```



The screenshot shows a web browser window titled "Web Browser - Review MPC Object 'MPCobj'". The address bar contains "Review MPC Object 'MPCobj'". The main content area features a large heading "Design Review for Model Predictive Controller 'MPCobj'" and a sub-heading "Summary of Performed Tests". Below this is a table with two columns: "Test" and "Status". The table lists ten tests with their corresponding statuses: "MPC Object Creation" (Pass), "QP Hessian Matrix Validity" (Warning), "Controller Internal Stability" (Pass), "Closed-Loop Nominal Stability" (Pass), "Closed-Loop Steady-State Gains" (Pass), "Hard MV Constraints" (Warning), "Other Hard Constraints" (Pass), "Soft Constraints" (Pass), and "Memory Size for MPC Data" (Pass). At the bottom of the page, the heading "Individual Test Result" is visible.

Test	Status
<a href="#">MPC Object Creation</a>	Pass
<a href="#">QP Hessian Matrix Validity</a>	Warning
<a href="#">Controller Internal Stability</a>	Pass
<a href="#">Closed-Loop Nominal Stability</a>	Pass
<a href="#">Closed-Loop Steady-State Gains</a>	Pass
<a href="#">Hard MV Constraints</a>	Warning
<a href="#">Other Hard Constraints</a>	Pass
<a href="#">Soft Constraints</a>	Pass
<a href="#">Memory Size for MPC Data</a>	Pass

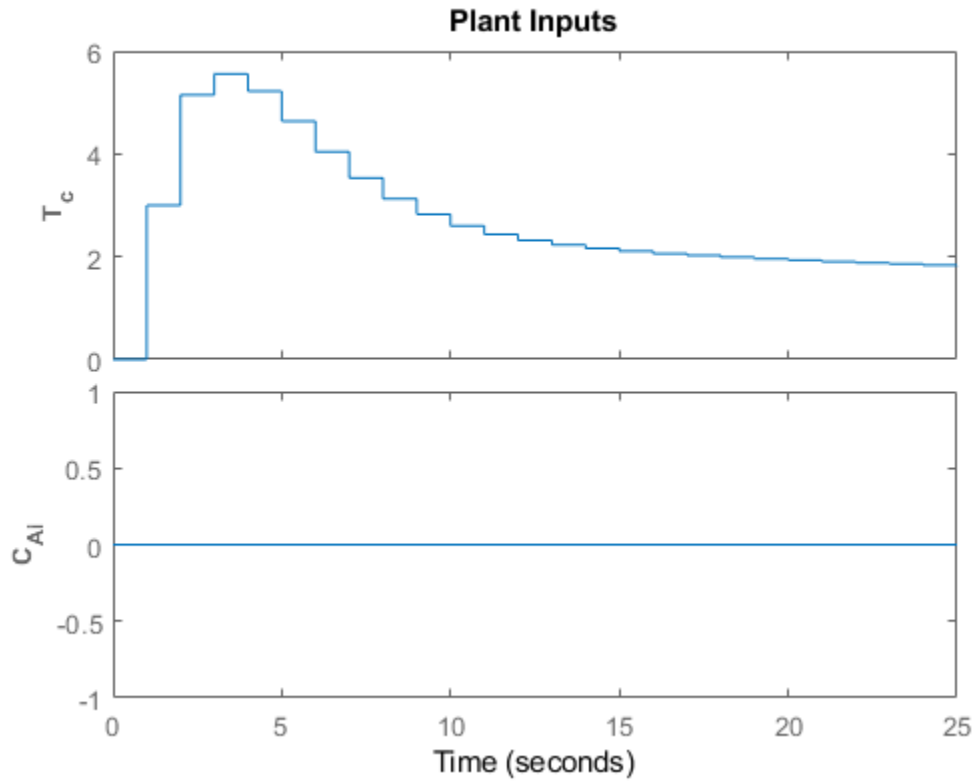
Individual Test Result

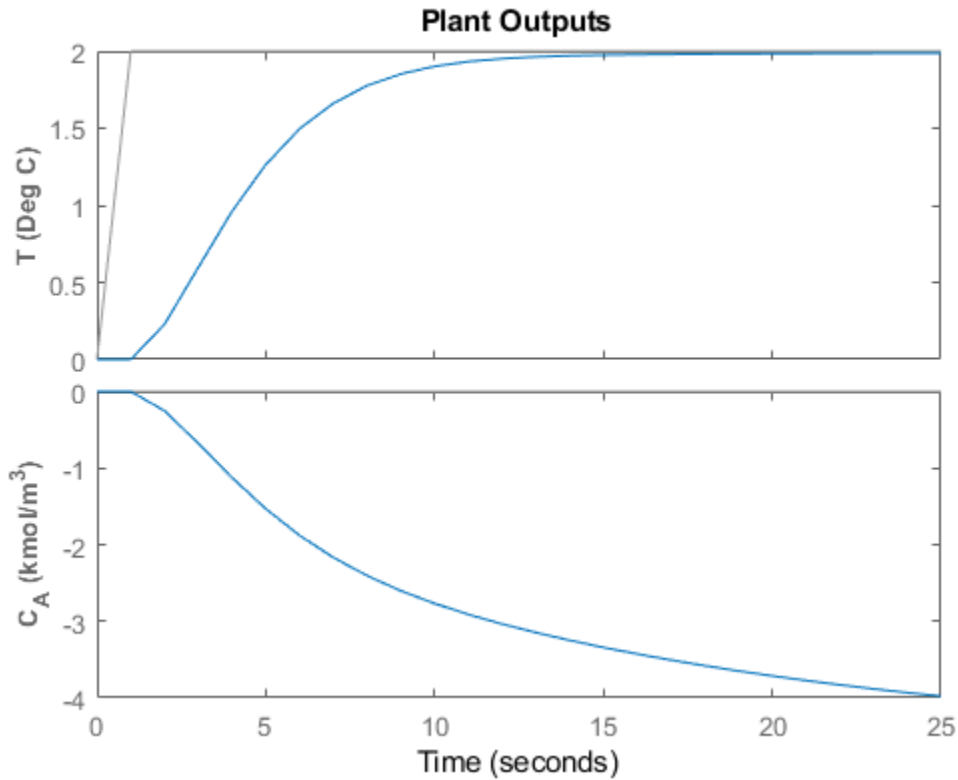
In this example, the `review` command found two potential issues with the design. The first warning asks whether the user intends to have a weight of zero on the `C_A` output. The second warning advises the user to avoid having hard constraints on both `MV` and `MVRate`.

### **Perform Linear Simulations**

Use the `sim` function to run a linear simulation of the system. For example, simulate the closed-loop response of `MPCobj` for 26 control intervals. Specify setpoints of 2 and 0 for the reactor temperature and the residual concentration respectively. The setpoint for the residual concentration is ignored because the tuning weight for the second output is zero.

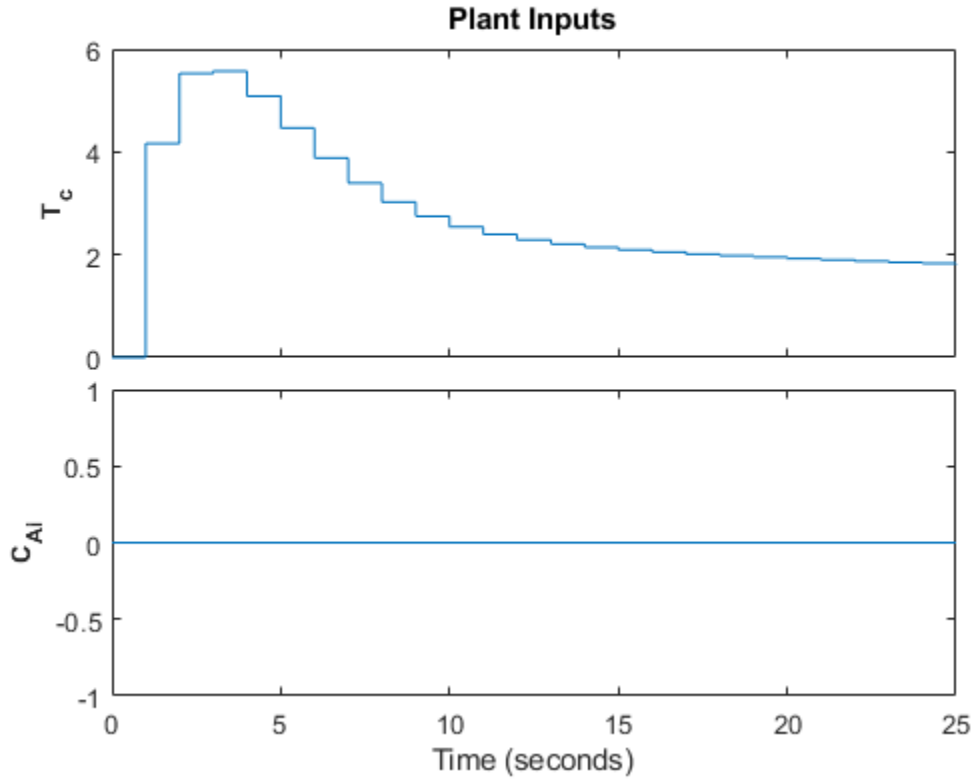
```
T = 26;  
r = [0 0; 2 0];  
sim(MPCobj,T,r)
```

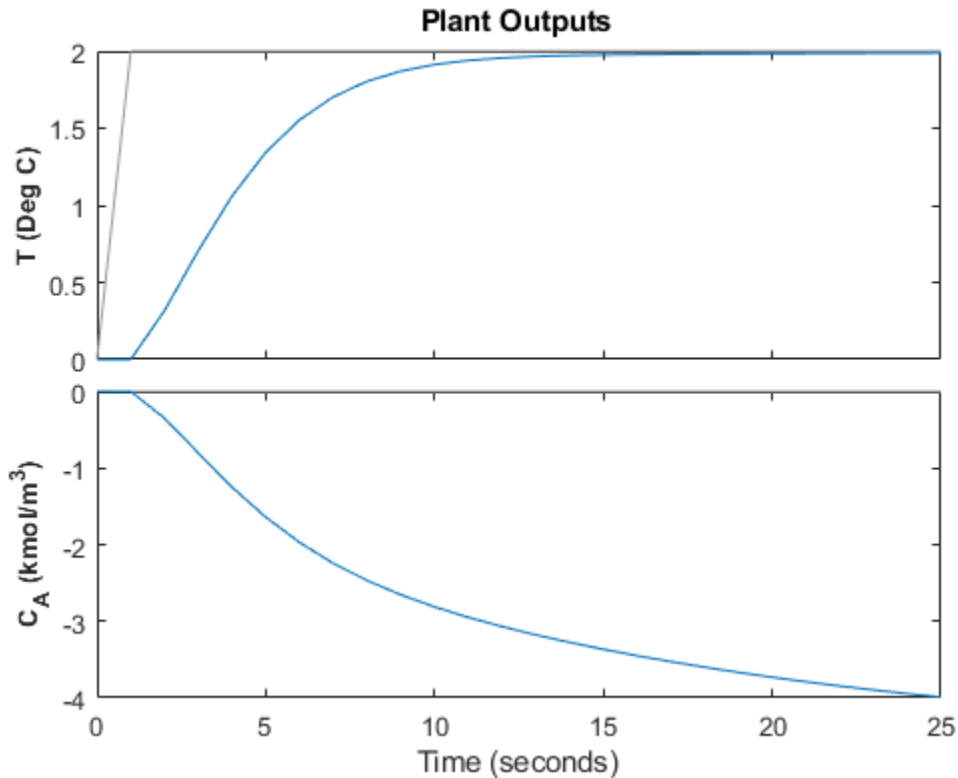




You can modify the simulation options using `mpcsimopt`. For example, run a simulation with the manipulated variable constraints turned off.

```
MPCopts = mpcsimopt;  
MPCopts.Constraints = 'off';  
sim(MPCobj,T,r,MPCopts)
```

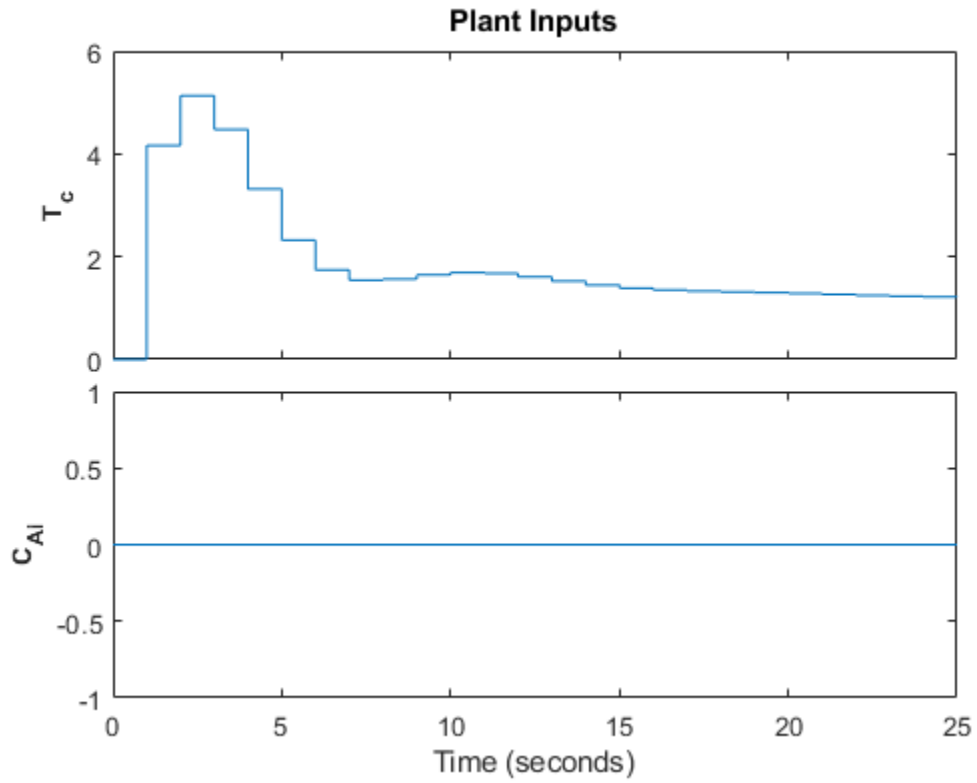




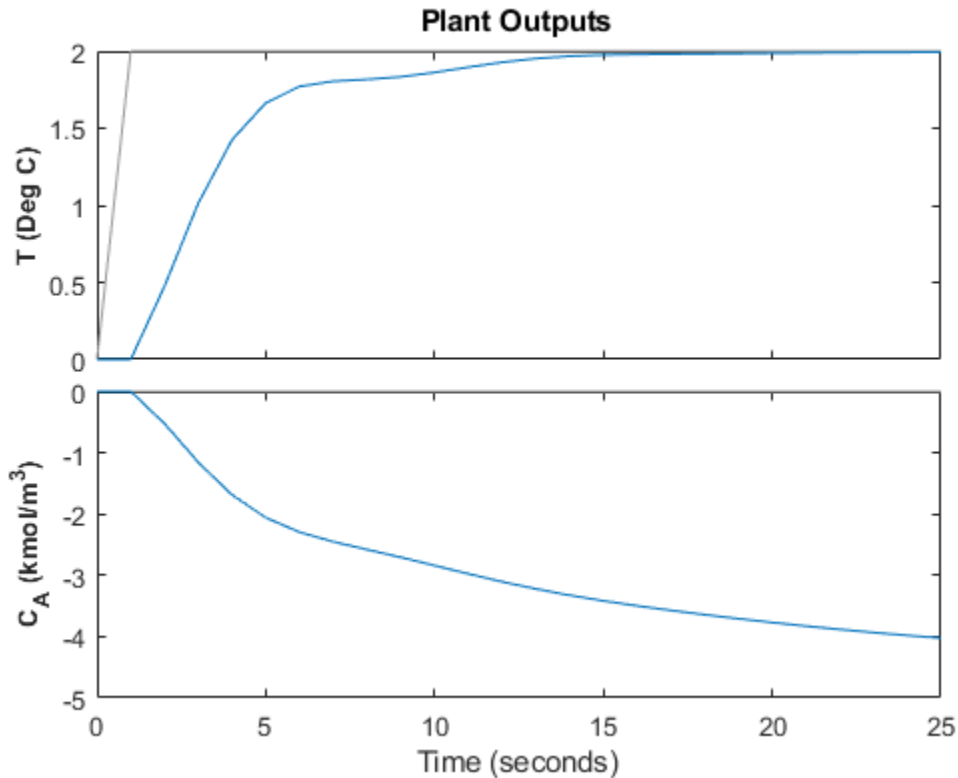
The first move of the manipulated variable now exceeds the specified 3-unit rate constraint.

You can also perform a simulation with a plant/model mismatch. For example, define a plant with 50% larger gains than those in the model used by the controller.

```
Plant = 1.5*CSTR;  
MPCopts.Model = Plant;  
sim(MPCobj,T,r,MPCopts)
```







The plant/model mismatch degrades controller performance slightly. Degradation can be severe and must be tested on a case-by-case basis.

Other options include the addition of a specified noise sequence to the manipulated variables or measured outputs, open-loop simulations, and a look-ahead option for better setpoint tracking or measured disturbance rejection.

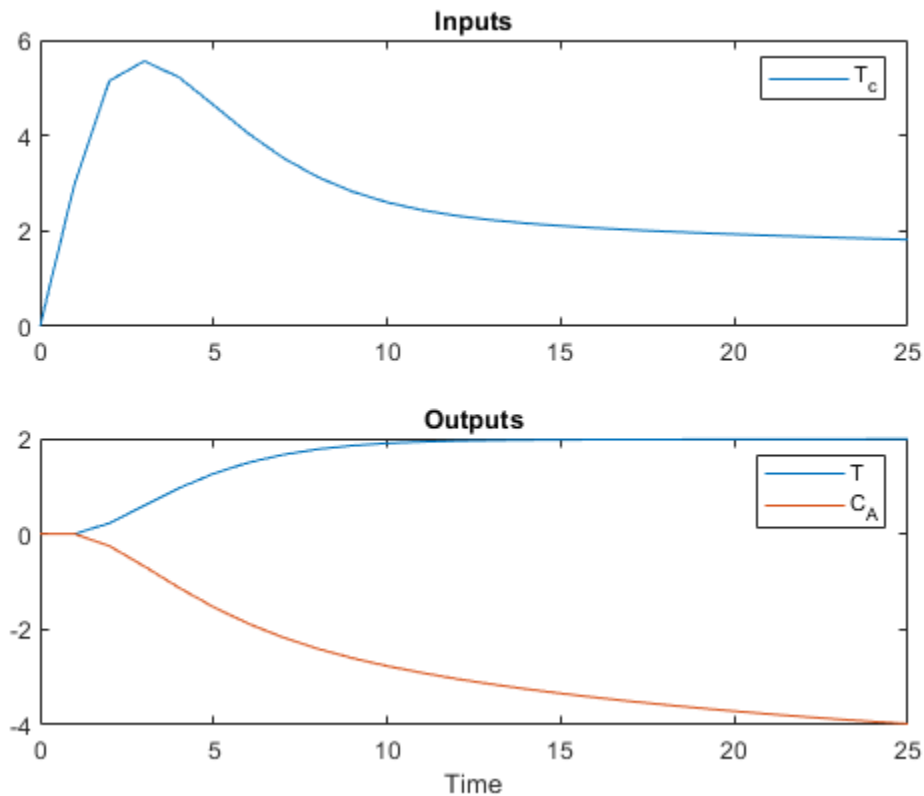
### Store Simulation Results

Store the simulation results in the MATLAB Workspace.

```
[y,t,u] = sim(MPCobj,T,r);
```

This syntax suppresses automatic plotting and returns the simulation results. You can use the results for other tasks, including custom plotting. For example, plot the manipulated variable and both output variables in the same figure.

```
figure
subplot(2,1,1)
plot(t,u)
title('Inputs')
legend('T_c')
subplot(2,1,2)
plot(t,y)
title('Outputs')
legend('T','C_A')
xlabel('Time')
```



Restore the `mpcverbosity` setting.

```
mpcverbosity(old_status);
```

## See Also

`mpc` | `review` | `sim`

## More About

- “MPC Modeling” on page 2-2
- “Design Controller Using MPC Designer” on page 3-2
- “Design MPC Controller in Simulink” on page 5-2

# Simulate Controller with Nonlinear Plant

You can use `sim` to simulate a closed-loop system consisting of a linear plant model and an MPC controller.

If your plant is a nonlinear Simulink model, you must linearize the plant (see “Linearization Using Linear Analysis Tool in Simulink Control Design” on page 2-25) and design a controller for the linear model (see “Design MPC Controller in Simulink” on page 5-2). To simulate the system, specify the controller in the MPC block parameter **MPC Controller** field and run the closed-loop Simulink model.

Alternatively, your nonlinear model might be a MEX-file, or you might want to include features unavailable in the MPC block, such as a custom state estimator. The `mpcmove` function is the Model Predictive Control Toolbox computational engine, and you can use it in such cases. The disadvantage is that you must duplicate the infrastructure that the `sim` function and the MPC block provide automatically.

The rest of this section covers the following topics:

- “Nonlinear CSTR Application” on page 4-16
- “Example Code for Successive Linearization” on page 4-17
- “CSTR Results and Discussion” on page 4-19

## Nonlinear CSTR Application

The CSTR model described in “Linearize Simulink Models” on page 2-22 is a strongly nonlinear system. As shown in “Design MPC Controller in Simulink” on page 5-2, a controller can regulate this plant, but degrades (and might even become unstable) if the operating point changes significantly.

The objective of this example is to redefine the predictive controller at the beginning of each control interval so that its predictive model, though linear, represents the latest plant conditions as accurately as possible. This will be done by linearizing the nonlinear model repeatedly, allowing the controller to adapt as plant conditions change. See references [1] and [2] for more details on this approach.

## Example Code for Successive Linearization

In the following code, the simulation begins at the nominal operating point of the CSTR model (concentration = 8.57) and moves to a lower point (concentration = 2) where the reaction rate is much higher. The required code is as follows:

```
[sys, xp] = CSTR_INOUT([],[],[], 'sizes');
up = [10 298.15 298.15];
u = up(3);
tsave = [];
usave = [];
ysave = [];
rsave = [];
Ts = 1;
t = 0;
while t < 40
    yp = xp;
    % Linearize the plant model at the current conditions
    [a,b,c,d] = linmod('CSTR_INOUT',xp,up);
    Plant = ss(a,b,c,d);
    Plant.InputGroup.ManipulatedVariables = 3;
    Plant.InputGroup.UnmeasuredDisturbances = [1 2];
    Model.Plant = Plant;

    % Set nominal conditions to the latest values
    Model.Nominal.U = [0 0 u];
    Model.Nominal.X = xp;
    Model.Nominal.Y = yp;

    dt = 0.001;

    simOptions.StartTime = num2str(t);
    simOptions.StopTime = num2str(t+dt);
    simOptions.LoadInitialState = 'on';
    simOptions.InitialState = 'xp';
    simOptions.SaveTime = 'on';
    simOptions.SaveState = 'on';
    simOptions.LoadExternalInput = 'on';
    simOptions.ExternalInput = '[t up; t+dt up]';

    simOut = sim('CSTR_INOUT',simOptions);

    T = simOut.get('tout');
    XP = simOut.get('xout');
```

```
YP = simOut.get('yout');

Model.Nominal.DX = (1/dt)*(XP(end,:) - xp(:));

% Define MPC controller for the latest model
MPCobj = mpc(Model, Ts);
MPCobj.W.Output = [0 1];

% Ramp the setpoint
r = max([8.57 - 0.25*t, 2]);

% Compute the control action
if t <= 0
    xd = [0; 0];
    x = mpcstate(MPCobj,xp,xd,[],u);
end

u = mpcmove(MPCobj,x,yp,[0 r],[]);

% Simulate the plant for one control interval
up(3) = u;

simOptions.StartTime = num2str(t);
simOptions.StopTime = num2str(t+Ts);
simOptions.InitialState = 'xp';
simOptions.ExternalInput = '[t up; t+Ts up]';

simOut = sim('CSTR_INOUT',simOptions);

T = simOut.get('tout');
XP = simOut.get('xout');
YP = simOut.get('yout');

% Save results for plotting
tsave = [tsave; T];
ysave = [ysave; YP];
usave = [usave; up(ones(length(T),1),:)]];
rsave = [rsave; r(ones(length(T),1),:)]];

xp = XP(end,:)';

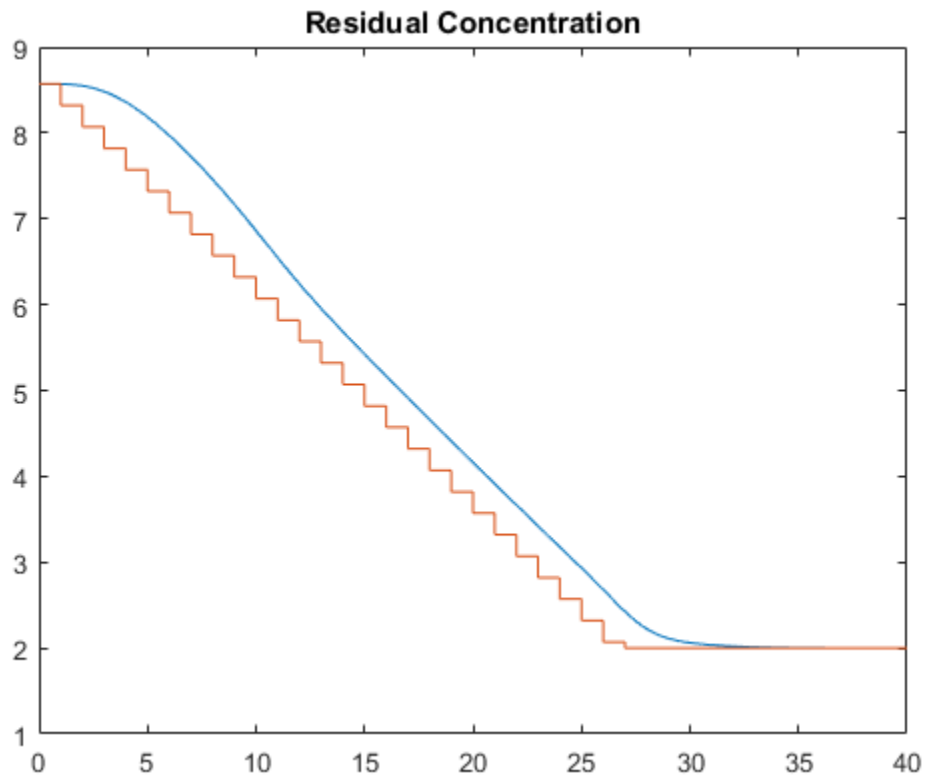
t = t + Ts;
end
```

```
figure(1)
plot(tsave,[ysave(:,2) rsave])
title('Residual Concentration')
figure(2)
plot(tsave,usave(:,3))
title('Coolant Temperature')
```

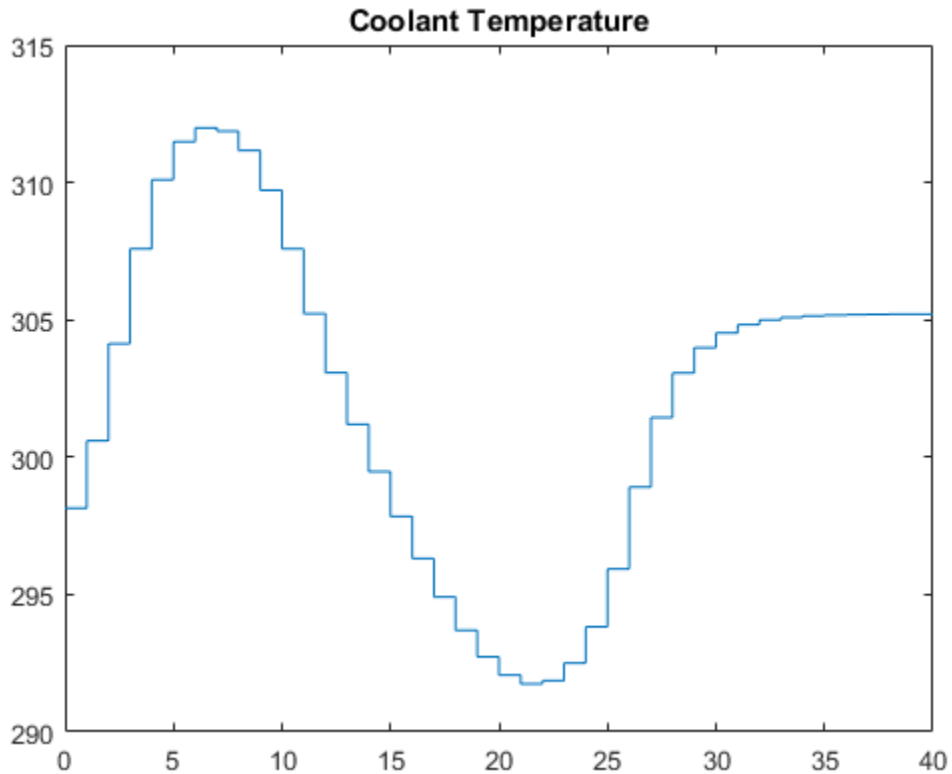
## CSTR Results and Discussion

The plotted results appear below. Note the following points:

- The setpoint is being ramped from the initial concentration to the desired final value (see the step-wise changes in the reactor concentration plot below). The reactor concentration tracks this ramp smoothly with some delay (see the smooth curve), and settles at the final state with negligible overshoot. The controller works equally well (and achieves the final concentration more rapidly) for a step-wise setpoint change, but it makes unrealistically rapid changes in coolant temperature (not shown).
- The final steady state requires a coolant temperature of 305.20 K (see the coolant temperature plot below). An interesting feature of this nonlinear plant is that if one starts at the initial steady state (coolant temperature = 298.15 K), stepping the coolant temperature to 305.20 and holding will not achieve the desired final concentration of 2. In fact, under this simple strategy the reactor concentration stabilizes at a final value of 7.88, far from the desired value. A successful controller must increase the reactor temperature until the reaction “takes off,” after which it must reduce the coolant temperature to handle the increased heat load. The relinearization approach provides such a controller (see following plots).







- Function `linearize` relinearizes the plant as its state evolves. This function was discussed previously in “Linearization Using MATLAB Code” on page 2-22.
- The code also resets the linear model's nominal conditions to the latest values. Note, however, that the first two input signals, which are unmeasured disturbances in the controller design, always have nominal zero values. As they are unmeasured, the controller cannot be informed of the true values. A non-zero value would cause an error.
- Function `mpc` defines a new controller based on the relinearized plant model. The output weight tuning ignores the temperature measurement, focusing only on the concentration.
- At  $t = 0$ , the `mpcstate` function initializes the controller's extended state vector,  $x$ , which is an *mpcstate object*. Thereafter, the `mpcmove` function updates it automatically using the controller's default state estimator. It would also be possible to

use an Extended Kalman Filter (EKF) as described in [1] and [2], in which case the EKF would reset the `mpcstate` input variables at each step.

- The `mpcmove` function uses the latest controller definition and state, the measured plant outputs, and the setpoints to calculate the new coolant temperature at each step.
- The Simulink `sim` function simulates the nonlinear plant from the beginning to the end of the control interval. Note that the final condition from the previous step is being used as the initial plant state, and that the plant inputs are being held constant during each interval.

Remember that a conventional feedback controller or a fixed Model Predictive Control Toolbox controller tuned to operate at the initial condition would become unstable as the plant moves to the final condition. Periodic model updating overcomes this problem automatically and provides excellent control under all conditions.

### References

- [1] Lee, J. H. and N. L. Ricker, "Extended Kalman Filter Based Nonlinear Model Predictive Control," *Ind. Eng. Chem. Res.*, Vol. 33, No. 6, pp. 1530-1541 (1994).
- [2] Ricker, N. L., and J. H. Lee "Nonlinear Model Predictive Control of the Tennessee Eastman Challenge Process," *Computers & Chemical Engineering*, Vol. 19, No. 9, pp. 961-981 (1995).

## Compute Steady-State Gain

This example shows how to analyze a model predictive controller using `clffset`. This function computes the closed-loop, steady-state gain for each output when a sustained, 1-unit disturbance is added to each output. It assumes that no constraints are active.

Define a state-space plant model.

```
A = [-0.0285 -0.0014; -0.0371 -0.1476];
B = [-0.0850 0.0238; 0.0802 0.4462];
C = [0 1; 1 0];
D = zeros(2,2);
CSTR = ss(A,B,C,D);
```

```
CSTR.InputGroup.MV = 1;
CSTR.InputGroup.UD = 2;
```

Create an MPC controller for the defined plant.

```
MPCobj = mpc(CSTR,1);
```

```
-->The "PredictionHorizon" property of "mpc" object is empty. Trying PredictionHorizon
-->The "ControlHorizon" property of the "mpc" object is empty. Assuming 2.
-->The "Weights.ManipulatedVariables" property of "mpc" object is empty. Assuming default
-->The "Weights.ManipulatedVariablesRate" property of "mpc" object is empty. Assuming default
-->The "Weights.OutputVariables" property of "mpc" object is empty. Assuming default 1
    for output(s) y1 and zero weight for output(s) y2
```

Specify tuning weights for the measured output signals.

```
MPCobj.W.OutputVariables = [1 0];
```

Compute the closed-loop, steady-state gain for this controller.

```
DCgain = clffset(MPCobj)
```

```
-->Converting model to discrete time.
-->The "Model.Disturbance" property of "mpc" object is empty:
    Assuming unmeasured input disturbance #2 is integrated white noise.
-->Assuming output disturbance added to measured output channel #1 is integrated white
    Assuming no disturbance added to measured output channel #2.
-->The "Model.Noise" property of the "mpc" object is empty. Assuming white noise on each
```

```
DCgain = 2x2
```

```
0.0000  -0.0000
2.3272  1.0000
```

$\text{DCgain}(i, j)$  represents the gain from the sustained, 1-unit disturbance on output  $j$  to measured output  $i$ .

The second column of  $\text{DCgain}$  shows that the controller does not react to a disturbance applied to the second output. This disturbance is ignored because the tuning weight for this channel is  $\theta$ .

Since the tuning weight for the first output is nonzero, the controller reacts when a disturbance is applied to this output, removing the effect of the disturbance ( $\text{DCgain}(1, 1) = \theta$ ). However, since the tuning weight for the second output is  $\theta$ , this controller reaction introduces a gain for output 2 ( $\text{DCgain}(2, 1) = 2.3272$ ).

### See Also

`cloffset | mpc`

### More About

- “MPC Modeling” on page 2-2

## Extract Controller

This example shows how to obtain an LTI representation of an unconstrained MPC controller using `ss`. You can use this to analyze the frequency response and performance of the controller.

Define a plant model. For this example, use the CSTR model described in “Design Controller Using MPC Designer” on page 3-2.

```
A = [-0.0285 -0.0014; -0.0371 -0.1476];
B = [-0.0850 0.0238; 0.0802 0.4462];
C = [0 1; 1 0];
D = zeros(2,2);
CSTR = ss(A,B,C,D);
```

```
CSTR.InputGroup.MV = 1;
CSTR.InputGroup.UD = 2;
CSTR.OutputGroup.MO = 1;
CSTR.OutputGroup.UO = 2;
```

Create an MPC controller for the defined plant using the same sample time, prediction horizon, and tuning weights described in “Design MPC Controller at the Command Line” on page 4-2.

```
MPCobj = mpc(CSTR,1,15);
```

```
-->The "ControlHorizon" property of the "mpc" object is empty. Assuming 2.
-->The "Weights.ManipulatedVariables" property of "mpc" object is empty. Assuming default.
-->The "Weights.ManipulatedVariablesRate" property of "mpc" object is empty. Assuming default.
-->The "Weights.OutputVariables" property of "mpc" object is empty. Assuming default 1 for output(s) y1 and zero weight for output(s) y2
```

```
MPCobj.W.ManipulatedVariablesRate = 0.3;
MPCobj.W.OutputVariables = [1 0];
```

Extract the LTI state-space representation of the controller.

```
MPCss = ss(MPCobj);
```

```
-->Converting model to discrete time.
-->The "Model.Disturbance" property of "mpc" object is empty:
    Assuming unmeasured input disturbance #2 is integrated white noise.
    Assuming no disturbance added to measured output channel #1.
-->The "Model.Noise" property of the "mpc" object is empty. Assuming white noise on each
```

Convert the original CSTR model to discrete form using the same sample time as the MPC controller.

```
CSTRd = c2d(CSTR,MPCss.Ts);
```

Create an LTI model of the closed-loop system using `feedback`. Use the manipulated variable and measured output for feedback, indicating a positive feedback loop. Using negative feedback would lead to an unstable closed-loop system, because the MPC controller is designed to use positive feedback.

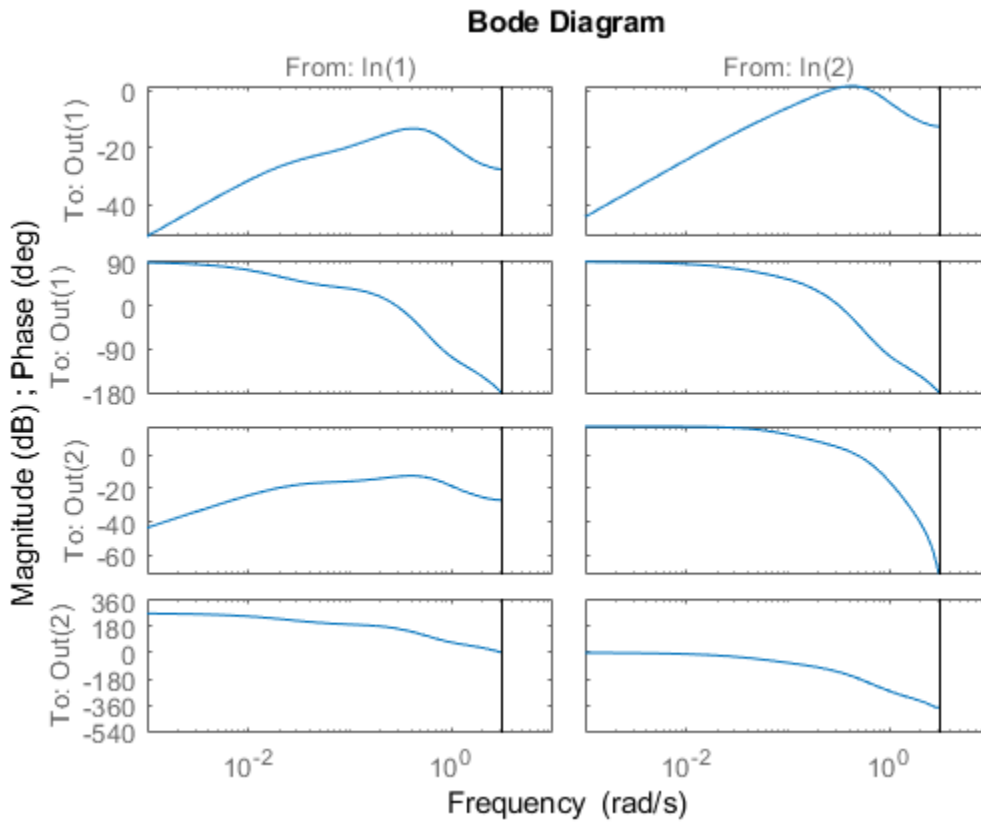
```
CLsys = feedback(CSTRd,MPCss,1,1,1);
```

You can then analyze the resulting feedback system. For example, verify that all closed-loop poles are within the unit circle.

```
poles = eig(CLsys)
poles = 6x1 complex
    0.5513 + 0.2700i
    0.5513 - 0.2700i
    0.6131 + 0.1110i
    0.6131 - 0.1110i
    0.9738 + 0.0000i
    0.9359 + 0.0000i
```

You can also view the system frequency response.

```
bode(CLsys)
```



## See Also

feedback | mpc | ss

## More About

- “Design MPC Controller at the Command Line” on page 4-2

# Signal Previewing

By default, a model predictive controller assumes that the current reference and measured disturbance signals remain constant during the controller prediction horizon. By doing so, the controller emulates a conventional feedback controller.

However, as shown in “Optimization Problem”, these signals can vary within the prediction horizon. If your application allows you to anticipate trends in such signals, an MPC controller with signal previewing can improve reference tracking, measured disturbance rejection, or both.

The following Model Predictive Control Toolbox commands provide previewing options:

- `sim`
- `mpcmove`
- `mpcmoveAdaptive`

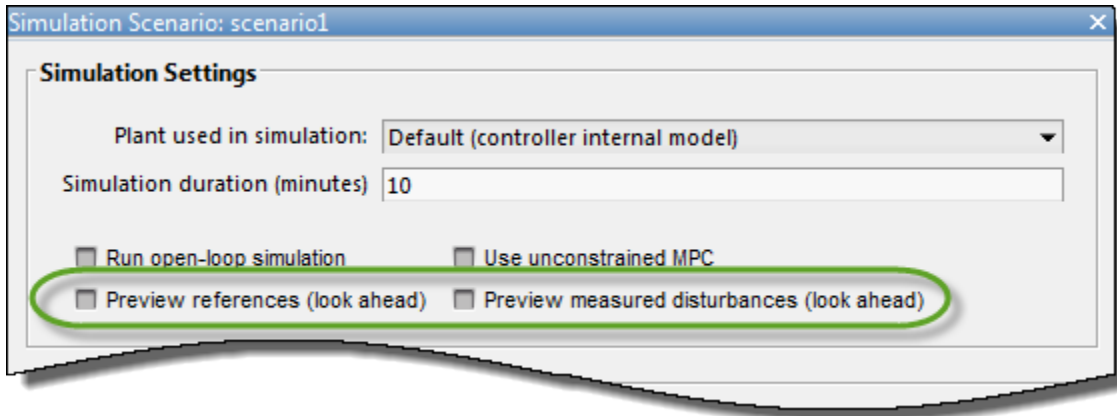
For Simulink, the following blocks support previewing:

- MPC Controller
- Adaptive MPC Controller
- Multiple MPC Controllers

Previewing for explicit MPC controllers will be supported in a future release.

In **MPC Designer**, you can specify whether simulation scenarios use previewing. When editing a scenario in the Simulation Scenario dialog box, select the **Preview references** or **Preview measured disturbances** options.





## See Also

### More About

- "Update Constraints at Run Time" on page 4-30
- "Improving Control Performance with Look-Ahead (Previewing)"

# Update Constraints at Run Time

To compensate for changing operating conditions, you can update constraints on plant inputs and outputs at run time. You can update the saturation limits for input and output signals as well as linear mixed input/output constraints.

Run-time constraint updating supports code generation.

## Update Bounds on Input and Output Signals at Run Time

You can update the bounds on plant input and output signals at run time. To do so, first define initial signal bounds when designing your MPC controller. For more information, see “Specify Constraints”. If you do not specify initial bounds for a given signal, you cannot constrain that signal at run time.

To update signal bounds during a command-line simulation, at each control interval, set the corresponding properties of an `mpcmoveopt` object before calling `mpcmove`, `mpcmoveAdaptive`, or `mpcmoveMultiple`. To update:

- Manipulated variable lower and upper bounds, set the `MVMin` and `MVMax` properties respectively.
- Output variable lower and upper bounds, set the `OutputMin` and `OutputMax` properties respectively.

To update signal bounds during a Simulink simulation, select the **Plant input and output limits** parameter of your MPC Controller, Adaptive MPC Controller, or Multiple MPC Controllers block. Doing so adds the following input ports to the block:

- `umin` — Lower bounds on manipulated variables
- `umax` — Upper bounds on manipulated variables
- `ymin` — Lower bounds on output variables
- `ymax` — Upper bounds on output variables

Connect signals to these ports that specify the run-time values of the bounds for each variable. If there is more than one manipulated variable or output variable, connect a vector signal to the corresponding ports. For example, if there are three output variables, connect a three-element vector signal to the `ymin` and `ymax` ports. If a variable is unconstrained in the controller object, then the connected signal value is ignored.

If you define time-varying constraints in your controller object, the new bounds are applied to the first finite values in the prediction horizon. All subsequent prediction horizon values adjust to maintain the same profile across the prediction horizon; that is, they change by the same amount.

For an example, see “Vary Input and Output Bounds at Run Time”.

## Update Mixed Input/Output Constraints at Run Time

You can update mixed input/output constraints at run time. For more information on these constraints, see “Constraints on Linear Combinations of Inputs and Outputs”. This feature is not supported for gain-scheduled MPC controllers.

You can update the following constraint matrices during your simulation:

- E — Manipulated variable constraint constant
- F — Controlled output constraint constant
- G — Mixed input/output constraint constant
- S — Measured disturbance constraint constant

To do so, first define initial constraints using the `setconstraint` command. You cannot add additional constraints at run time.

To update mixed input/output constraints during a command-line simulation, in each control interval set the `CustomConstraint` property of an `mpcmoveopt` object before calling `mpcmove` or `mpcmoveAdaptive`. Specify `CustomConstraint` as a structure with E, F, G, and S fields. Specify each field as an array with dimensions that match the initial constraint arrays specified using `setconstraint`.

To update mixed input/output constraints during a Simulink simulation, select the **Custom constraints** parameter of your MPC Controller or Adaptive MPC Controller block. Doing so adds E, F, G, and S input ports to the block. The S input port is added only if your controller has measured disturbances.

Connect matrix signals to these ports that specify the run-time values for each array. If you define E, F, G, or S in your MPC controller, you must connect a signal to the corresponding input port, and that signal must have the same dimensions as the array specified in the controller. If an array is not defined in the controller object, use a zero matrix with the correct size.

For an example that updates mixed input/output constraints for an adaptive MPC controller, see “Obstacle Avoidance Using Adaptive Model Predictive Control”.

### **See Also**

`mpcmove` | `mpcmoveAdaptive` | `mpcmoveExplicit` | `setconstraint`

### **More About**

- “Tune Weights at Run Time” on page 4-33
- “Constraints on Linear Combinations of Inputs and Outputs”
- “Vary Input and Output Bounds at Run Time”

## Tune Weights at Run Time

There are two ways to perform tuning experiments using Model Predictive Control Toolbox software:

- Modify your controller object off line (by changing weights, etc.) and then test the modified object.
- Change tuning weights as the controller operates.

This topic describes the second approach.

You can adjust the following tuning weights as the controller operates (see “Tune Weights”):

- Plant output variable (OV) reference tracking,  $w^y$ .
- Manipulated variable (MV) reference tracking,  $w^u$ .
- MV increment suppression,  $w^{\Delta u}$ .
- Global constraint softening,  $\rho_\epsilon$ .

In each case, the weight applies to the entire prediction horizon. If you are using time-varying weights, you must use offline modification of the weight and then test the modified controller.

In Simulink, the following blocks support online tuning:

- MPC Controller
- Adaptive MPC Controller
- Multiple MPC Controllers. In this case, the tuning signals apply to the active controller object, which might switch as the control system operates. If the objects in your set employ different weights, you should tune them off line.

The Explicit MPC Controller and Multiple Explicit MPC Controllers blocks do not support online tuning because a weight change requires a complete revision of the explicit MPC control law, which is computationally intensive.

For command-line testing, the `mpcmove` and `mpcmoveAdaptive` commands include options to mimic the online tuning behavior available in Simulink.

## **See Also**

### **More About**

- “Signal Previewing” on page 4-28
- “Tuning Controller Weights”

# Designing and Testing Controllers in Simulink

---

- “Design MPC Controller in Simulink” on page 5-2
- “Test an Existing Controller” on page 5-23

The Model Predictive Control Toolbox provides a controller block library for use in Simulink.

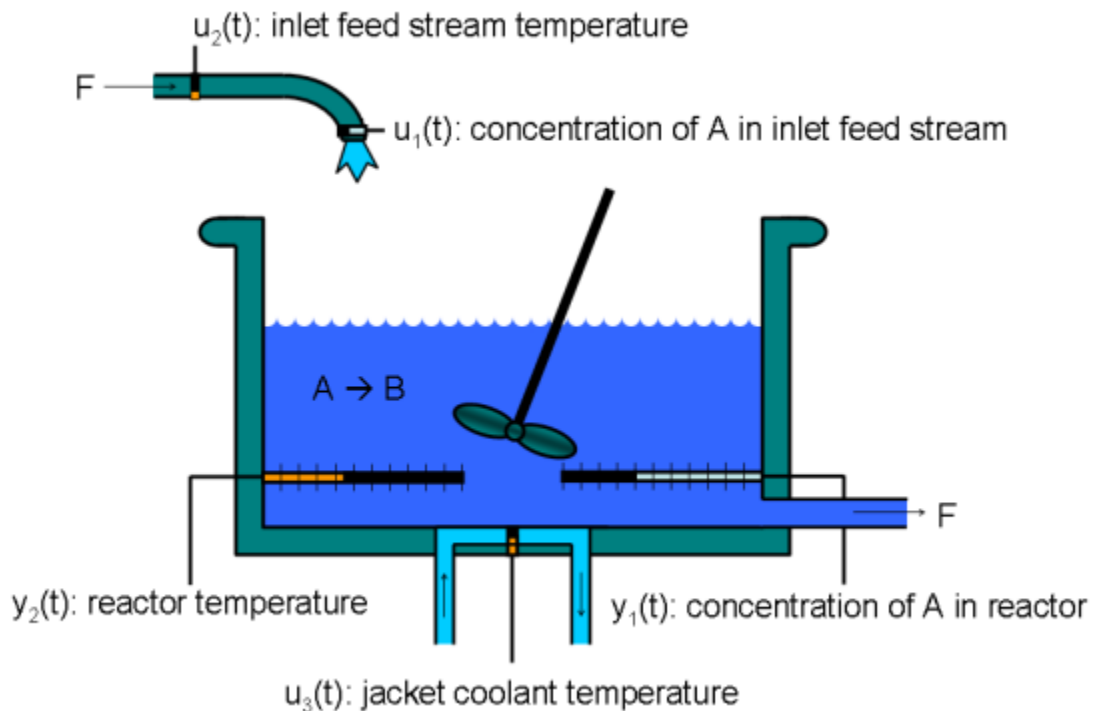
## Design MPC Controller in Simulink

This example shows how to design a model predictive controller for a continuous stirred-tank reactor (CSTR) in Simulink using **MPC Designer**.

This example requires Simulink Control Design software to define the MPC structure by linearizing a nonlinear Simulink model.

### CSTR Model

A CSTR is a jacketed nonadiabatic tank reactor commonly used in the process industry.



An inlet stream of reagent A feeds into the tank at a constant rate. A first-order, irreversible, exothermic reaction takes place to produce the product stream, which exits the reactor at the same rate as the input stream.

The CSTR model has three inputs:



- Feed Concentration ( $CA_i$ ) — The concentration of reagent  $A$  in the feed stream ( $\text{kgmol/m}^3$ )
- Feed Temperature ( $T_i$ ) — Feed stream temperature (K)
- Coolant Temperature ( $T_c$ ) — Reactor coolant temperature (K)

The two model outputs are:

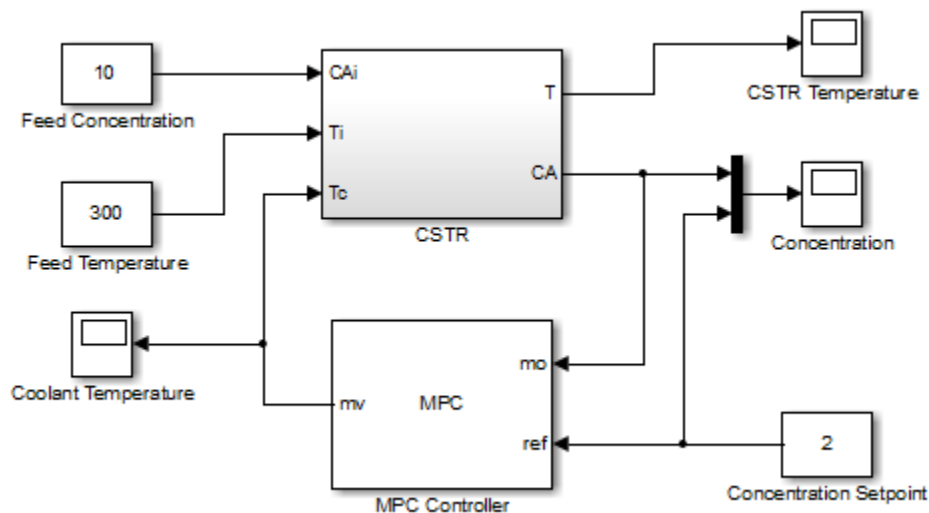
- CSTR Temperature ( $T$ ) — Reactor temperature (K)
- Concentration ( $CA$ ) — Concentration of reagent  $A$  in the product stream, also referred to as the residual concentration ( $\text{kgmol/m}^3$ )

The control objective is to maintain the residual concentration,  $CA$ , at its nominal setpoint by adjusting the coolant temperature,  $T_c$ . Changes in the feed concentration,  $CA_i$ , and feed temperature,  $T_i$ , cause disturbances in the CSTR reaction.

The reactor temperature,  $T$ , is usually controlled. However, for this example, ignore the reactor temperature, and assume that the residual concentration is measured directly.

### Open Simulink Model

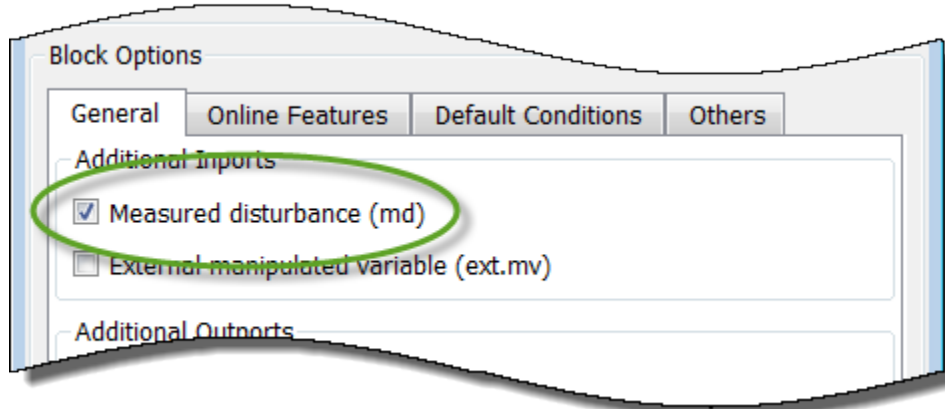
```
open_system('CSTR_ClosedLoop');
```



### Connect Measured Disturbance To MPC Controller Block

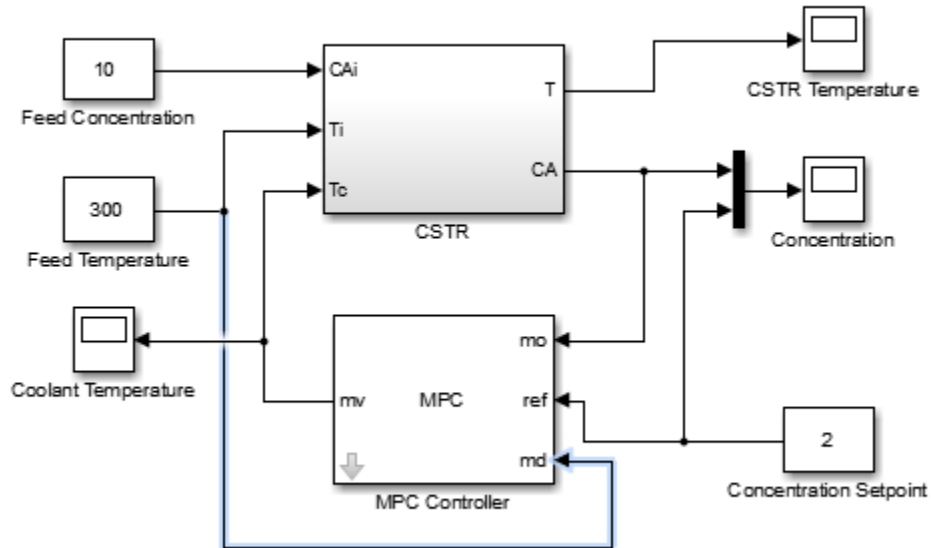
In the Simulink model window, double-click the MPC Controller block.

In the Block Parameters dialog box, on the **General** tab, in the **Additional Inports** section, check the **Measured disturbance (md)** option.



Click **Apply** to add the md inport to the controller block.

In the Simulink model window, connect the Feed Temperature block output to the md inport.



### Open MPC Designer App

In the MPC Controller Block Parameters dialog box, click **Design** to open **MPC Designer**.

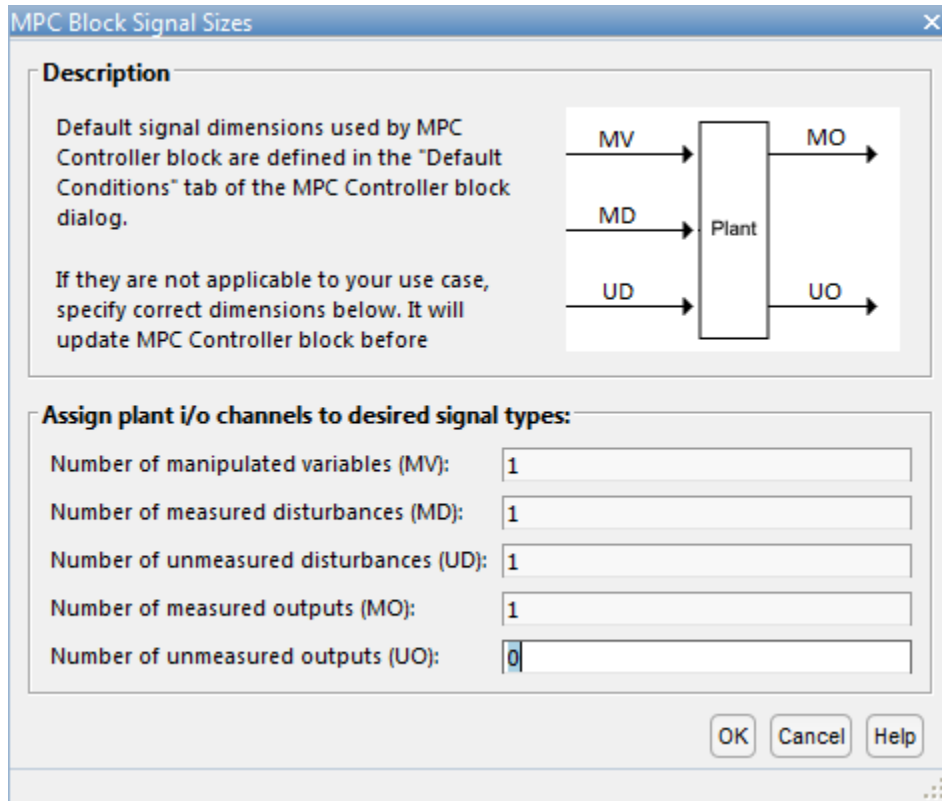
### Define MPC Structure

In **MPC Designer**, on the **MPC Designer** tab, in the **Structure** section, click **MPC Structure**.

In the Define MPC Structure By Linearization dialog box, in the **Controller Sample Time** section, specify a sample time of 0.1.

In the **MPC Structure** section, click **Change I/O Sizes** to add the unmeasured disturbance and measured disturbance signal dimensions.

In the MPC Block Signal Sizes dialog box, specify the number of input/output channels of each type:



Click **OK**.

In the Define MPC Structure By Linearization dialog box, in the **Simulink Signals for Plant Inputs** section, the app adds a row for **Unmeasured Disturbances (UD)**.

Define MPC Structure By Linearization

**MPC Structure**

**Controller Sample Time**

Specify MPC controller sample time (default sample time in the MPC block):

**Simulink Operating Point**

Choose an operating point at which plant model is linearized and nominal values are computed:

**Simulink Signals for Plant Inputs**

Selected	Type	Block Path
<input checked="" type="radio"/>	Manipulated Variables (MV)	<a href="#">CSTR_ClosedLoop/MPC Controller:1</a>
<input type="radio"/>	Measured Disturbances (MD)	<a href="#">CSTR_ClosedLoop/Feed Temperature:1</a>
<input type="radio"/>	Unmeasured Disturbances (UD)	Select a UD signal in the Simulink model

**Simulink Signals for Plant Outputs**

Selected	Type	Block Path
<input checked="" type="radio"/>	Measured Outputs (MO)	<a href="#">CSTR_ClosedLoop/CSTR:2</a>

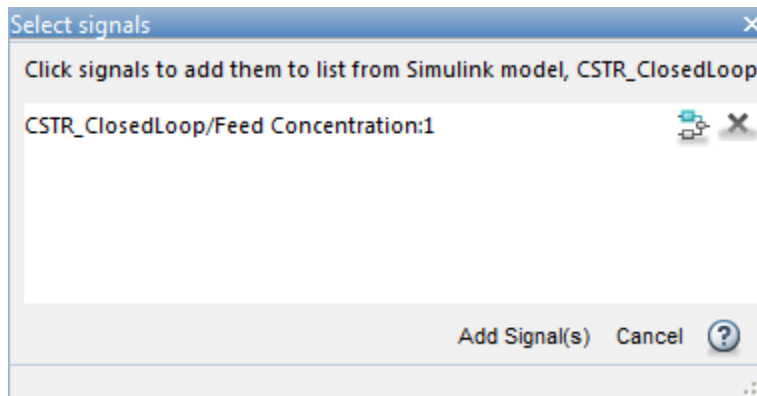
Define and Linearize Cancel Help

The manipulated variable, measured disturbance, and measured output are already assigned to their respective Simulink signal lines, which are connected to the MPC Controller block.

In the **Simulink Signals for Plant Inputs** section, select the **Unmeasured Disturbances (UD)** row, and click **Select Signals**.

In the Simulink model window, click the output signal from the Feed Concentration block.

The signal is highlighted and its block path is added to the Select Signal dialog box.



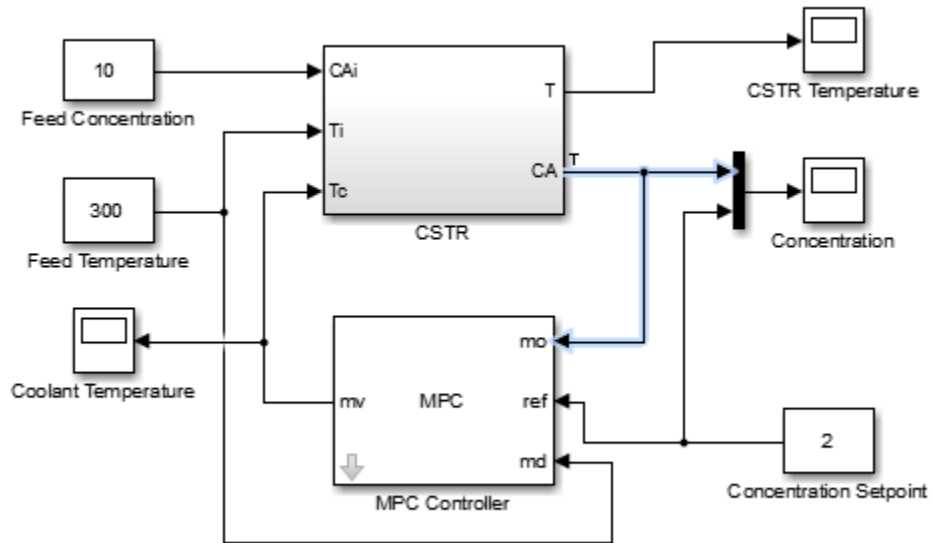
In the Select Signals dialog box, click **Add Signal(s)**.

In the Define MPC Structure By Linearization dialog box, in the **Simulink Signals for Plant Inputs** table, the **Block Path** for the unmeasured disturbance signal is updated.

### Linearize Simulink Model

Linearize the Simulink model at a steady-state equilibrium operating point where the residual concentration is  $2 \text{ kgmol/m}^3$ . To compute such an operating point, add the CA signal as a trim output constraint, and specify its target constraint value.

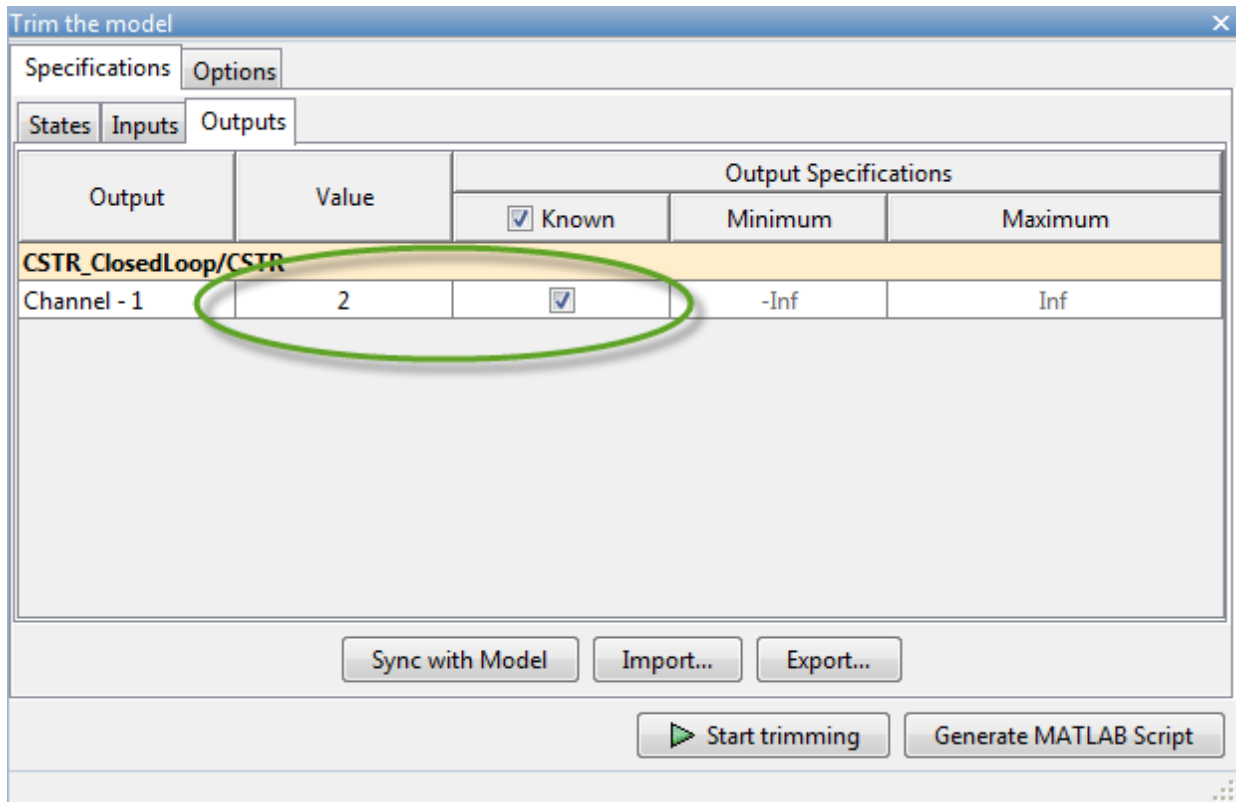
In the Simulink model window, right-click the signal line connected to CA output of the CSTR block, and click **Linear Analysis Points > Trim Output Constraint**.



The CA signal can now be used to define output specifications for calculating a model steady-state operating point.

In the Define MPC Structure By Linearization dialog box, in the **Simulink Operating Point** section, in the drop-down list, under **Create New Operating Point**, click **Trim Model**.

In the Trim the model dialog box, in the **Outputs** tab, check the box in the **Known** column for Channel - 1 and specify a **Value** of 2.



This setting constrains the value of the output signal during the operating point search to a known value.

Click **Start Trimming**.

In the Define MPC Structure By Linearization dialog box, in the **Simulink Operating Point** section, the computed operating point, `op_trim1`, is added to the drop-down list and selected.

In the drop-down list, under **View/Edit**, click **Edit op\_trim1**.



State	Desired Value	Actual Value	Desired dx	Actual dx
<b>CSTR_ClosedLoop/CSTR/C_A</b>				
State - 1	[ 0 , Inf ]	2	0	-4.5972e-12
<b>CSTR_ClosedLoop/CSTR/T_K</b>				
State - 1	[ 0 , Inf ]	373.1311	0	5.4897e-11
<b>CSTR_ClosedLoop/MPC Controller/MPC/last_mv</b>				
State - 1	[ -Inf , Inf ]	299.0349	0	0

In the Edit dialog box, on the **State** tab, in the **Actual dx** column, the near-zero derivative values indicate that the computed operating point is at steady-state.

Click **Initialize model** to set the initial states of the Simulink model to the operating point values in the **Actual Values** column. Doing so enables you to later simulate the Simulink model at the computed operating point rather than at the default model initial conditions.

In the Initialize Model dialog box, click **OK**.

Close the Edit dialog box.

In the Define MPC Structure By Linearization dialog box, click **Define and Linearize** to linearize the model.

The linearized plant model is added to the **Data Browser**. Also, the following are added to the **Data Browser**:

- A default MPC controller created using the linearized plant as an internal prediction model

- A default simulation scenario

### Define Input/Output Channel Attributes

On the **MPC Designer** tab, in the **Structure** section, click **I/O Attributes**.

In the Input and Output Channel Specifications dialog box, in the **Name** column, specify meaningful names for each input and output channel.

In the **Unit** column, specify appropriate units for each signal.

**Plant Inputs**

Channel	Type	Name	Unit	Nominal Value	Scale Factor
u(1)	MV	Tc	deg K	299.034880275325	1
u(2)	MD	Ti	deg K	300	1
u(3)	UD	CAi	kgmol/m <sup>3</sup>	0	1

**Plant Outputs**

Channel	Type	Name	Unit	Nominal Value	Scale Factor
y(1)	MO	CA	kgmol/m <sup>3</sup>	2	1

OK Apply Cancel Help

The **Nominal Value** for each signal is the corresponding steady-state value at the computed operating point.

Click **OK**.

### Define Disturbance Rejection Simulation Scenarios

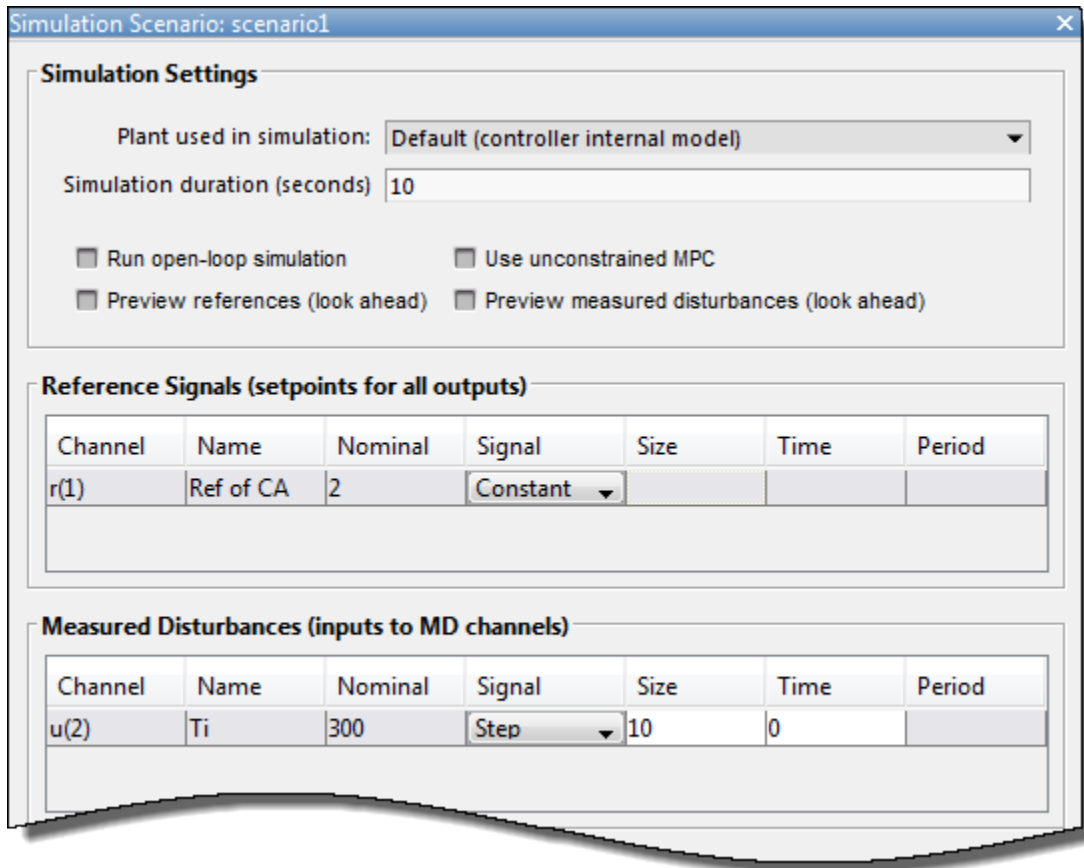
The primary objective of the controller is to hold the residual concentration,  $CA$ , at the nominal value of  $2 \text{ kgmol/m}^3$ . To do so, the controller must reject both measured and unmeasured disturbances.

In the **Scenario** section, click **Edit Scenario > scenario1**.

In the Simulation Scenario dialog box, in the **Reference Signals** table, in the **Signal** drop-down list select **Constant** to hold the output setpoint at its nominal value.

In the **Measured Disturbances** table, in the **Signal** drop-down list, select **Step**.

Specify a step **Size** of  $10$  and a step **Time** of  $0$ .



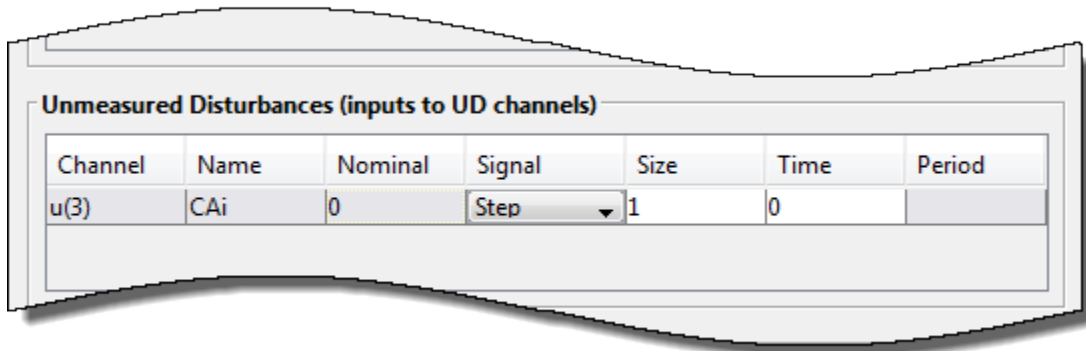
Click **OK**.

In the **Data Browser**, under **Scenarios**, double-click `scenario1` and rename it `MD_reject`.

In the **Scenario** section, click **Plot Scenario > New Scenario**.

In the Simulation Scenario dialog box, in the **Unmeasured Disturbances** table, in the **Signal** drop-down list, select **Step**.

Specify a step **Size** of 1 and a step **Time** of 0.



Click **OK**.

In the **Data Browser**, under **Scenarios**, double-click `NewScenario` and rename it `UD_reject`.

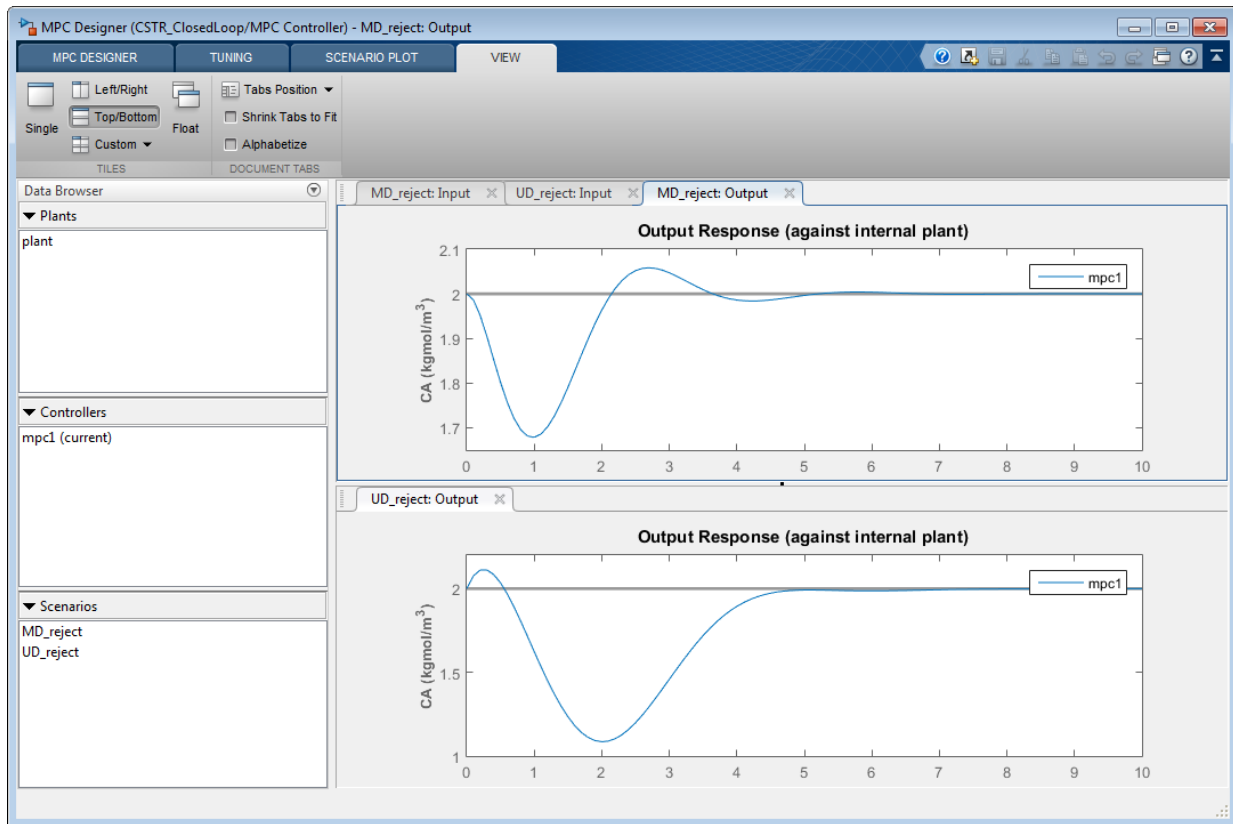
### Arrange Output Response Plots

To make viewing the tuning results easier, arrange the plot area to display the Output Response plots for both scenarios at the same time.

On the **View** tab, in the **Tiles** section, click **Top/Bottom**.

The plot display area changes to display the Input Response plots above the Output Response plots.

Drag the **MD\_reject: Output** tab up to the top plot.



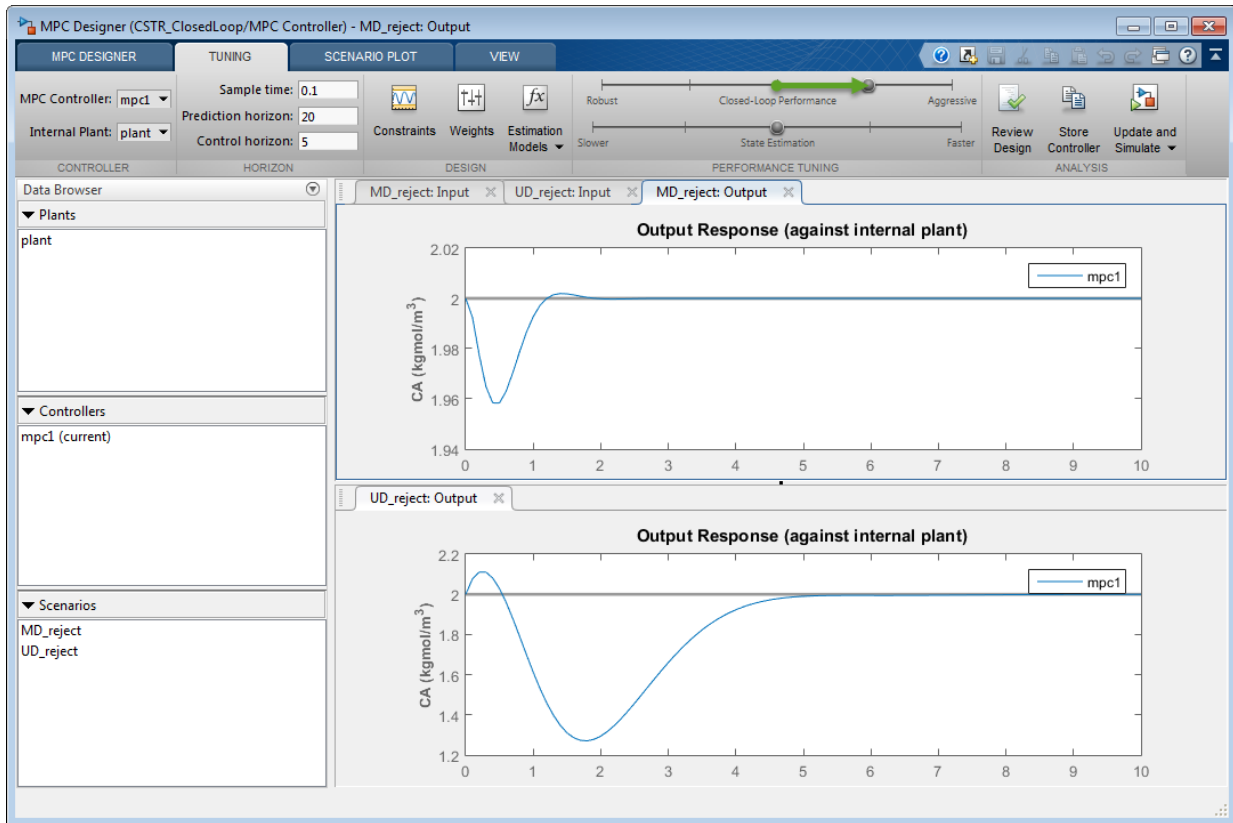
### Tune Controller Performance

In the **Tuning** tab, in the **Horizon** section, specify a **Prediction horizon** of 20 and a **Control horizon** of 5.

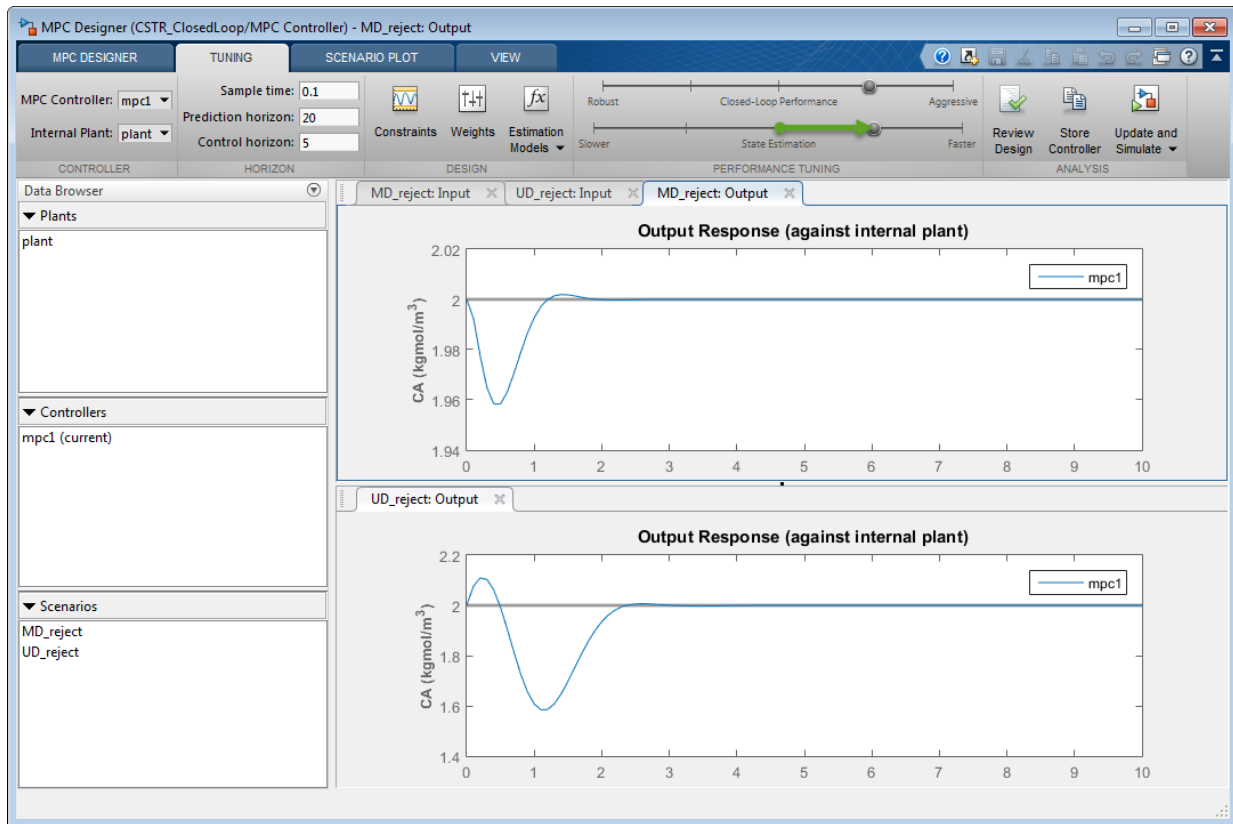
The **Output Response** plots update based on the new horizon values.

Use the default controller constraint and weight configurations.

In the **Performance Tuning** section, drag the **Closed-Loop Performance** slider to the right, which leads to tighter control of outputs and more aggressive control moves. Drag the slider until the **MD\_reject: Output** response reaches steady state in less than 2 seconds.



Drag the **State Estimation** slider to the right, which leads to more aggressive unmeasured disturbance rejection. Drag the slider until the **UD\_reject: Output** response reaches steady state in less than 3 seconds.



### Update Simulink Model with Tuned Controller

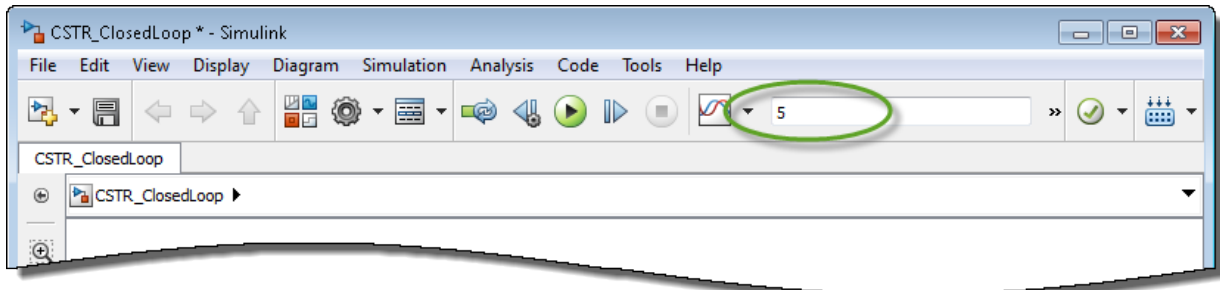
In the **Analysis** section, click the **Update and Simulate** arrow .

Under **Update and Simulate**, click **Update Block Only**. The app exports the tuned controller, mpc1, to the MATLAB workspace. In the Simulink model, the MPC Controller block is updated to use the exported controller.

### Simulate Unmeasured Disturbance Rejection

In the Simulink model window, change the simulation duration to 5 seconds.

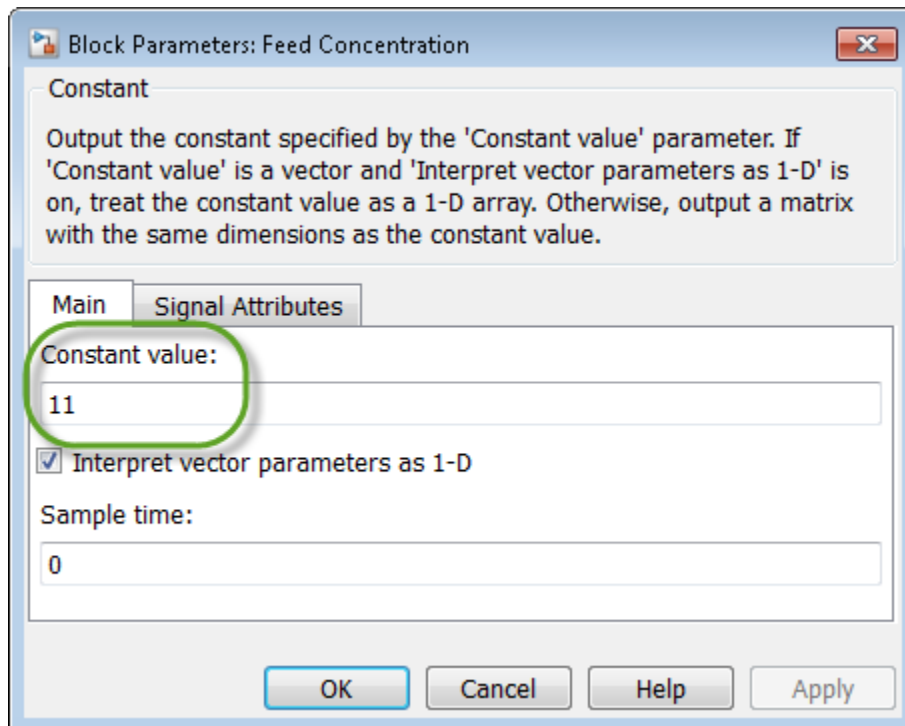




The model initial conditions are set to the nominal operating point used for linearization.

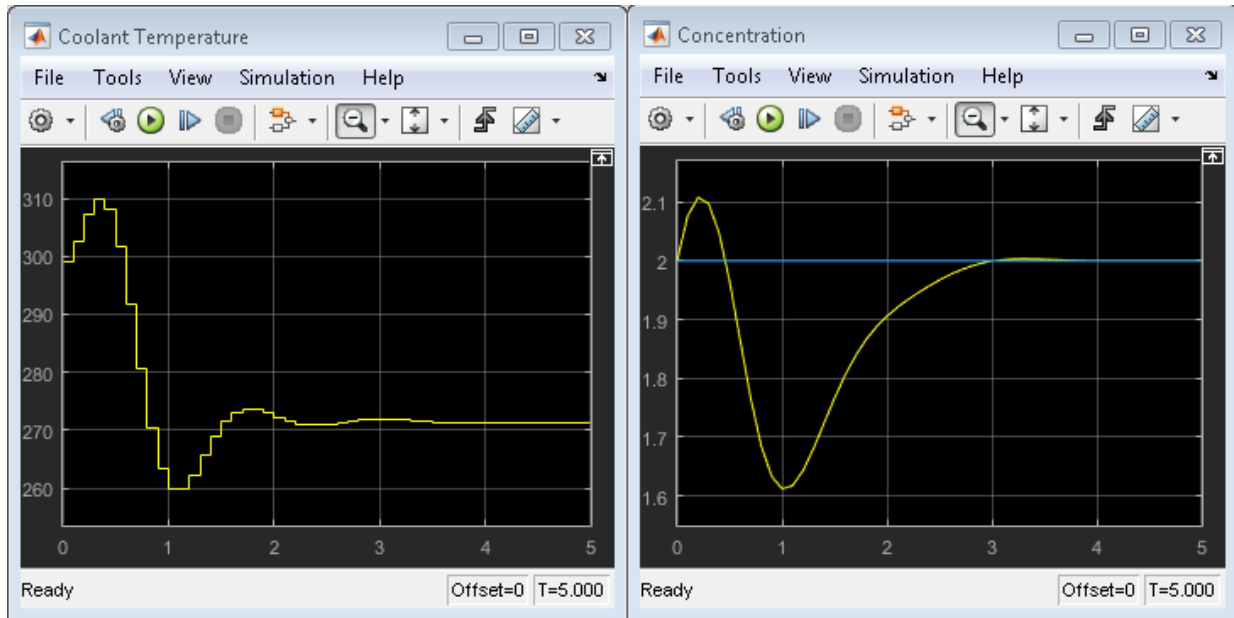
Double-click the **Feed Concentration** block.

In the Block Parameters dialog box, enter a **Constant Value** of 11 to simulate a unit step at time zero.



Click **Apply**.

In the Simulink model window, click **Run**.



The **Concentration** output response is similar to the **UD\_reject** response, however the settling time is around 1 second later. The different result is due to the mismatch between the linear plant used in the **MPC Designer** simulation and the nonlinear plant in the Simulink model.

### Simulate Measured Disturbance Rejection

In the Block Parameters: Feed Concentration dialog box, enter a **Constant Value** of 10 to return the feed concentration to its nominal value.

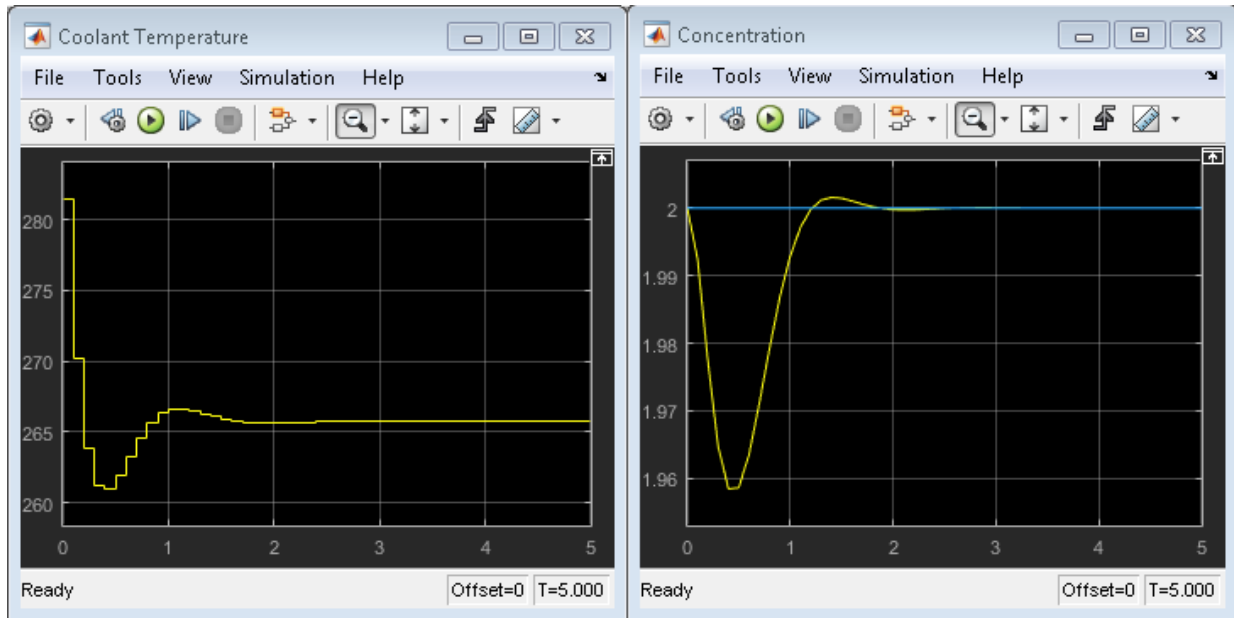
Click **OK**.

In the Simulink model window, double-click the Feed Temperature block.

In the Block Parameters: Feed Temperature dialog box, enter a **Constant Value** of 310 to simulate a step change of size 10 at time zero.

Click **OK**.

In the Simulink model window, click **Run**.



The **Concentration** output response is similar to the **MD\_reject** response from the **MPC Designer** simulation.

## References

- [1] Seborg, D. E., T. F. Edgar, and D. A. Mellichamp, *Process Dynamics and Control*, 2nd Edition, Wiley, 2004, pp. 34–36 and 94–95.

## See Also

MPC Controller | **MPC Designer**

## More About

- “Tune Weights”
- “Linearize Simulink Models” on page 2-22

- “Design Controller Using MPC Designer” on page 3-2

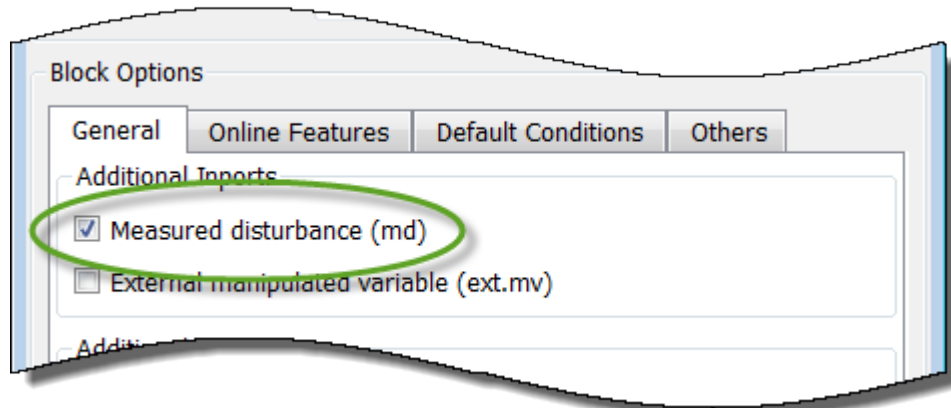
## Test an Existing Controller

This topic shows how to test an existing model predictive controller by adding it to a Simulink model.

- 1 Open your Simulink model.
- 2 Add an MPC Controller block to the model.
- 3 If your controller includes measured disturbances, add the md inport to the MPC Controller block.

Double-click the MPC Controller block.

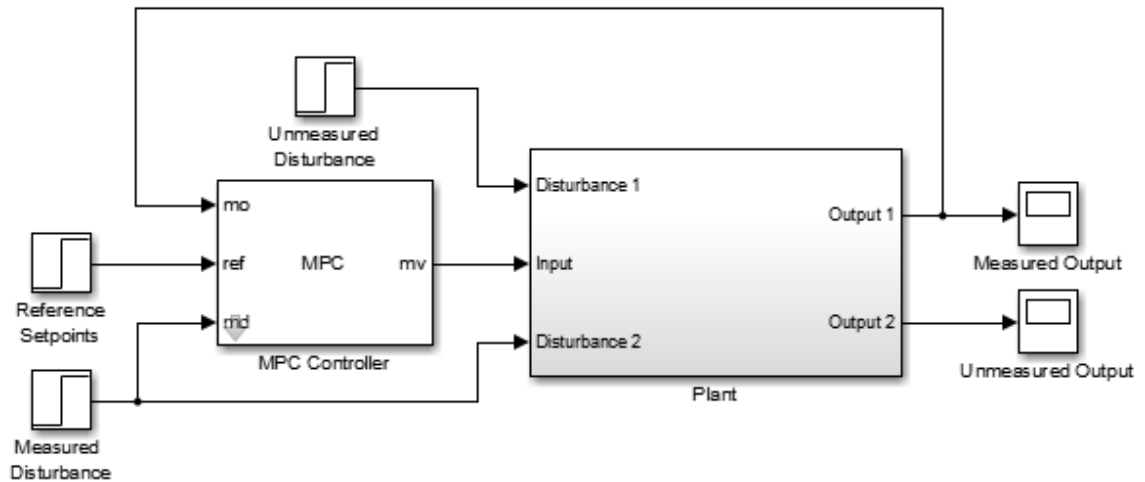
In the Block Parameters dialog box, on the **General** tab, select **Measured disturbance (md)**.



Click **OK**.

- 4 Connect the plant and controller signals in the Simulink model. Connect:
  - The plant inputs to the manipulated variable (mv) inport of the MPC Controller block.
  - The plant measured outputs to the measured output (mo) inport of the MPC Controller block.
  - The measured disturbances, if any, to the plant and to the measured disturbance (md) inport of the MPC Controller block.

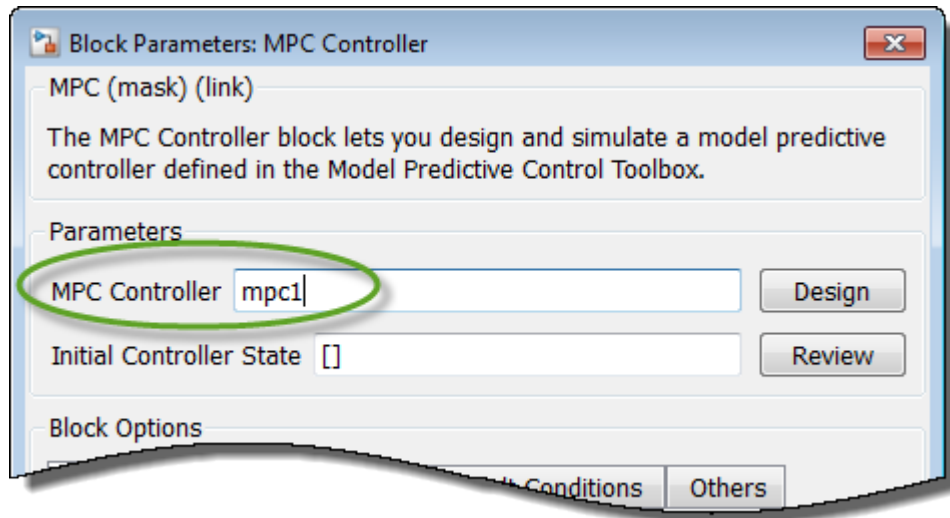
- Any unmeasured disturbances or unmeasured outputs to their corresponding plant inport and outport.
- The reference signals to the reference (ref) inport of the MPC Controller block.



### 5 Specify the controller.

Double-click the MPC Controller block.

In the Block Parameters dialog box, in the **MPC Controller** field, specify the name of an mpc controller from the MATLAB workspace.



Click **OK**.

**6** (Optional) Modify the controller.

After specifying a controller in the MPC Controller block, you can modify the controller:

- Using **MPC Designer**:
  - In the Block Parameters dialog box, click **Design**.
  - In **MPC Designer**, tune the controller parameters.
  - In the **MPC Designer** tab, in the **Result** section, click **Update and Simulate > Update Block Only**.

The app exports the updated controller to the MATLAB workspace.

- Using commands to modify the controller object in the MATLAB workspace.

**7** Run the Simulink model.

---

**Tip** If you do not have a Simulink model of your plant, you can generate one that uses your MPC controller to control its internal plant model. For more information, see “Generate Simulink Model from MPC Designer”.

---

### See Also

MPC Controller | **MPC Designer** | `mpc`

### More About

- “Design MPC Controller in Simulink” on page 5-2
- “Design MPC Controller at the Command Line” on page 4-2
- “Generate Simulink Model from MPC Designer”