Second-Order Filter

Implement second-order filter

Library

Simscape / Electrical / Specialized Power Systems / Control & Measurements / Filters

Description

Based on the **Filter type** selected in the block menu, the Second-Order Filter block implements the following transfer function:

Low-pass filter:

$$H(s) = \frac{\omega_n^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}$$

High-pass filter:

$$H(s) = \frac{s^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}$$

Band-pass filter:

$$H(s) = \frac{2\zeta\omega_n s}{s^2 + 2\zeta\omega_n s + \omega_n^2}$$

Band-stop (notch) filter:

$$H(s) = \frac{s^2 + \omega_n^2}{s^2 + 2\zeta\omega_n s + \omega_n^2}$$

s = Laplace operator

ω_n = natural frequency; $\omega_n = 2\pi f_n$

ζ = damping ratio (called Zeta in the block menu)

The key characteristics of the Second-Order Filter block are:

- Input accepts a vectorized input of N signals, implementing N filters. This feature is particularly useful for designing controllers in three-phase systems (N = 3).
- Filter states can be initialized for specified DC and AC inputs.
- It enables you to compute and plot filter response.

Parameters

Filter type

Specify the type of filter: `Lowpass`, `Highpass`, `Bandpass` (default), or `Bandstop (notch)`.

Natural frequency fn (Hz)

Specify the natural frequency of the filter, in hertz. This value must be a scalar or a vector. Default is `120`.

Damping ratio Zeta (Q = 1/(2*Zeta))

Specify the damping ratio of the filter. The damping ratio is typically a value between 0 and 1. Default is `0.707`.

The damping ratio is related to the filter quality factor Q:

$$Q = \frac{1}{2\zeta}$$

For a bandpass or a bandstop filter, the 3 dB bandwidth is given by

$$BW = \frac{f_n}{Q} = 2\zeta f_n$$

Sample time

Specify the sample time of the block, in seconds. Set to 0 to implement a continuous block. Default is `0`.

Initialize filter states

When this check box is selected, filter states are initialized according to the **AC initial input** and **DC initial input** parameters. Default is selected.

AC initial input: [Mag, Phase (degrees), Freq (Hz)]

Specify the magnitude of the initial AC component of the input signal, its phase, in degrees, and its frequency, in hertz. Default is `[0, 0, 60]`.

When the input is vectorized (N signals), specify an N-by-3 matrix, where each row of the matrix corresponds to a particular input.

The **AC initial input** parameter is visible only when the **Initialize filter states** parameter is selected.

DC initial input

Specify the value of the initial DC component of the input signal. When the input signal is vectorized, specify a 1-by-N vector, where each value corresponds to a particular input. Default is `0`.

The **DC initial input** parameter is visible only when the **Initialize filter states** parameter is selected.

Plot filter response

When this check box is selected, the filter step response and its Bode diagram (magnitude and phase of transfer function as a function of frequency) are plotted in a figure. Default is cleared.

Frequency range (Hz): [Start, End, Inc.]

Specify the frequency range for plotting the filter Bode diagram. Specify a vector containing the starting frequency, the end frequency, and the incremental frequency, in hertz. Default is `[0, 500, 1]`.

The **Frequency range** parameter is visible only when the **Plot filter response** parameter is selected.

Characteristics

Direct Feedthrough	Yes
Sample Time	Specified in the **Sample Time** parameter Continuous if **Sample Time** = 0
Scalar Expansion	Yes, of the parameters
States	Two states per filter
Dimensionalized	Yes

Examples

The `power_SecondOrderFilter` example shows the Second-Order Filter block using two **Filter type** parameter settings (`Lowpass` and `Bandstop`).

The model sample time is parameterized with variable Ts (default value Ts = 50e-6). To simulate continuous filters, specify Ts = 0 in the MATLAB® Command Window before starting the simulation.

Introduced in R2013a