SimMechanics™

Getting Started Guide

R2012a

MATLAB® & SIMULINK®

MathWorks®

How to Contact MathWorks

www.mathworks.com	Web
comp.soft-sys.matlab	Newsgroup
www.mathworks.com/contact_TS.html	Technical Support
suggest@mathworks.com	Product enhancement suggestions
bugs@mathworks.com	Bug reports
doc@mathworks.com	Documentation error reports
service@mathworks.com	Order status, license renewals, passcodes
info@mathworks.com	Sales, pricing, and general information

508-647-7000 (Phone)

508-647-7001 (Fax)

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

For contact information about worldwide offices, see the MathWorks Web site.

SimMechanics™ Getting Started Guide

Revision History

March 2012	Online only	New for Release 2012a (Version 4.0)

Contents

Getting Started

1 Introduction to SimMechanics Software

2 Essentials of SimMechanics Software

Getting Started

Introduction to SimMechanics Software

Product Description

Model and simulate multibody mechanical systems

SimMechanics provides a multibody simulation environment for 3D mechanical systems, such as robots, vehicle suspensions, construction equipment, and aircraft landing gear. You model the multibody system using blocks representing bodies, joints, constraints, and force elements, and then SimMechanics formulates and solves the equations of motion for the complete mechanical system. Models from CAD systems, including mass, inertia, joint, constraint, and 3D geometry, can be imported into SimMechanics. An automatically generated 3D animation lets you visualize the system dynamics.

You can parameterize your models using MATLAB® variables and expressions, and design control systems for your multibody system in Simulink®. You can add electrical, hydraulic, pneumatic, and other components to your mechanical model using Simscape™ and test them all in a single simulation environment. To deploy your models to other simulation environments, including hardware-in-the-loop (HIL) systems, SimMechanics supports C-code generation (with Simulink Coder™).

Key Features

- Blocks and modeling constructs for simulating and analyzing 3D mechanical systems in Simulink
- Rigid body definition using standard geometry and custom extrusions defined in MATLAB
- Automatic calculation of mass and inertia tensor
- Simulation modes for analyzing motion and calculating forces
- Visualization and animation of multibody system dynamics with 3D geometry
- SimMechanics Link utility, providing an interface to Pro/ENGINEER®, SolidWorks®, and Autodesk Inventor, and an API for interfacing with other CAD platforms
- Support for C-code generation (with Simulink Coder)

Required and Related Products

In this section...

Required Products

To use SimMechanics software, you must have installed current versions of the following products:

- MATLAB
- Simulink
- Simscape

SimMechanics Visualization Requirements

SimMechanics visualization requires Silicon Graphics OpenGL® graphics support on your system to display and animate mechanical systems.

You can improve your speed and graphics resolution by adding a graphics accelerator hardware card to your system. Animation of simulations is sensitive to central processor and graphics card speed and memory. Experiment with graphics hardware and system settings to find a reasonable compromise between quality and speed for your system.

Support for Recorded Animations

You can record simulation animations in Microsoft Audio Video Interleave® (AVI) format using SimMechanics visualization. Animations are recorded as compressed AVI files with the default Motion JPEG codec. This codec is available on most operating systems.

To play back AVI files, you need an AVI-compatible media application. MATLAB has an internal movie player compatible with AVI. You can also use an external AVI-compatible player.

Note SimMechanics visualization does not support uncompressed AVI recording.

Related Products

You can extend the capability of SimMechanics using other physical modeling products found in the Simscape family. Each physical modeling product gives you a set of block libraries with which you can model common components found in industry and academia: rigid bodies, gears, valves, solenoids, etc.

With the physical modeling products, you can model not only mechanical systems, but also electrical, hydraulic, and power systems. You can model each system separately, and then integrate them into a single multiphysics model where you can analyze their combined performance.

Physical Modeling Product Family

The physical modeling family includes five products:

- SimDriveline™, for modeling and simulating drivetrain systems
- SimElectronics®, for modeling and simulating electronic systems
- SimHydraulics®, for modeling and simulating hydraulic systems
- SimMechanics, for modeling and simulating three-dimensional mechanical systems
- SimPowerSystems™, for modeling and simulating electrical power systems

First and Second Generation Technologies

SimMechanics software contains two technologies: First Generation and Second Generation. These are reflected in the structure of SimMechanics block libraries and documentation, both of which are organized into First Generation and Second Generation categories.

First Generation technology includes the SimMechanics block libraries and computational engine included in all releases prior to R2012a. SimMechanics continues to support First Generation technology, so that you can run and edit any First Generation models that you may have.

Second Generation technology includes a completely redesigned set of block libraries and computational engine. The new blocks are based on a new approach to mechanical modeling centered on coordinate frames. You can connect these blocks to First Generation blocks indirectly through Simulink signals.

This guide is for Second Generation technology *only*. For information on First Generation technology, see the SimMechanics User's Guide for First Generation technology.

SimMechanics Components

Blocks, ports, and connection lines form the backbone of any SimMechanics model.

In this section...

Blocks

What a Block Is

A block is a ported entity that represents a physical component used in a machine, a geometric relationship or constraint, or a dynamic element like force and torque. You can connect blocks to each other using ports and connection lines. For more information, see “Description of SimMechanics Block Libraries” on page 1-9.

Ports

What a Port Is

A port allows you to connect one block to other blocks, establishing a relationship between them.

Types of Ports

In SimMechanics, there are two types of ports, *frame* and *physical signal*:

- Frame ports represent a coordinate frame, identifying a location and orientation in space relative to a reference frame. Frame ports can be:
  - Reference—identifies a local reference frame for a SimMechanics component
  - World—identifies a global reference frame for a whole model

  - Base—identifies the frame which a block transforms to obtain a new (follower) frame
  - Follower—identifies a new frame obtained by transforming a base frame

- Physical signal ports transmit force and motion signals. Physical signal ports can be:
  - *In*ports—accept physical signals as input for actuation
  - *Out*ports—provide physical signals as output for sensing

Connecting Ports

To connect two compatible ports, you draw a connection line between them. You can connect a frame port *only* to another frame port. Likewise, you can connect a physical signal port *only* to another physical signal port. You cannot mix frame and physical signal ports.

Connection Lines

What a Connection Line Does

A connection line connects two or more blocks to each other. Use it to establish a geometric relationship between two blocks or to transfer physical signals between them.

Connection Line Types

SimMechanics software uses two types of connection lines:

- *Frame connection line*—makes two frame ports coincident with each other, sharing the same location and orientation in space. To specify two different frames, see .

Note A frame connection line represents a single frame along its length. When you connect one connection line directly to other connection lines, all lines represent the same frame along their lengths. Their frames share the same origin and orientation in space for all time.

- *Physical signal connection line*—transmits physical signals, like force and motion parameters.

Frame and physical signal connection lines look identical in a model. You identify them by the ports they connect to: a frame connection line always connects two or more frame ports; a physical signal connection line always connects two or more physical signal ports.

Drawing Connection Lines

When you hover the mouse over a port, the arrow assumes a cross-hair shape. Click the port of one block, and hold it while dragging the mouse onto a port on a second block. This draws a *connection line* between the two ports.

Note You can connect a frame connection line *only* to a frame port or to another frame connection line. You can connect a physical signal connection line *only* to a physical signal port or to another physical signal connection line. You cannot mix frame and physical signal connection lines.

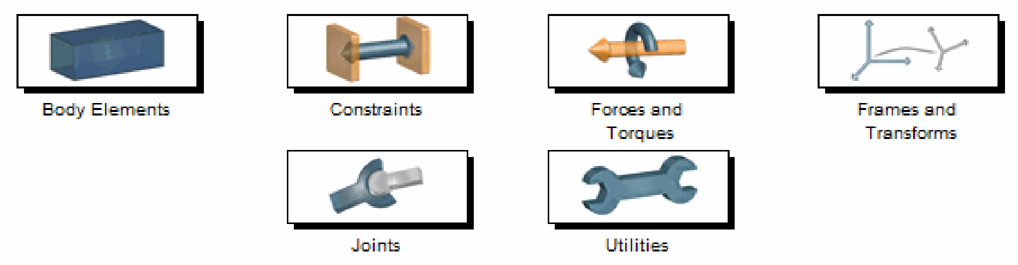
SimMechanics Block Libraries

In this section...

Description of SimMechanics Block Libraries

SimMechanics contains two block libraries: First Generation and Second Generation. This section is about the Second Generation block library. For information about the First Generation block library see "Block Reference" in the SimMechanics First Generation documentation.

The SimMechanics Second Generation block library contains six sublibraries. With the six sublibraries you can define every element in a mechanical model: bodies, joints and constraints, forces and torques, and more.



SimMechanics™ Product Library

Body Elements

The Body Elements library gives you the basic blocks to model a rigid body, including blocks for specifying geometry, inertia, and graphic properties. You can model rigid bodies with simple predefined, solid shapes and colors, or with custom geometry and inertia. You can also rigidly connect multiple solid shapes to form compound rigid bodies.

Constraints

The Constraints library provides blocks to restrict the relative motion between two body frames. You can constrain two body frames to maintain a specified distance or a specified angle between them.

Forces and Torques

The Forces and Torques library provides blocks to actuate and otherwise influence a mechanical model. You can apply different forces and torques to model the different interactions between components, or between a component and the external environment.

Frames and Transforms

The Frames and Transforms library provides blocks that allow you to define and measure spatial relationships between different coordinate frames in your model.

Joints

The Joints library provides blocks that connect bodies to one another in kinematic relationships representing different degrees of freedom between two body frames. These are time-varying relationships that restrict translation and rotation between the bodies to a specified set of axes.

Utilities

The Utilities library contains blocks to configure solver and simulation parameters in the SimMechanics environment.

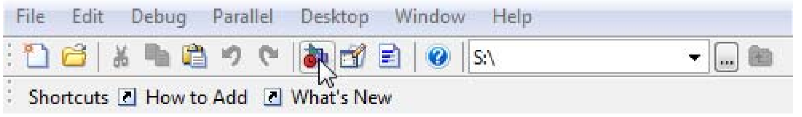
Opening SimMechanics Libraries

You can open the SimMechanics in two ways: by navigating the Simulink Library browser, or at the MATLAB command line.

From the MATLAB Toolbar

You can open the SimMechanics block library directly from the MATLAB window.

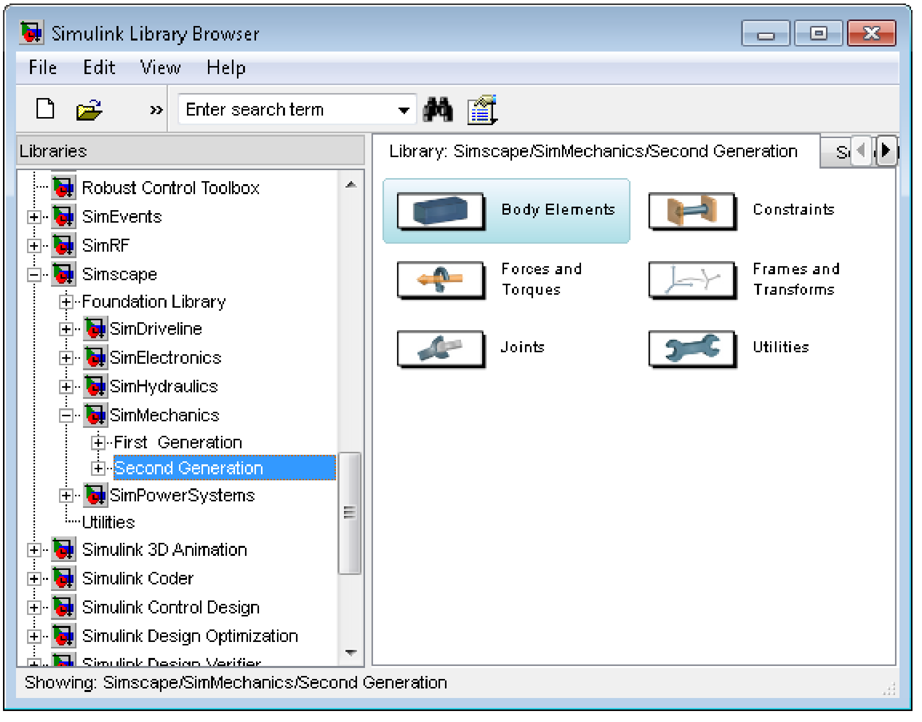
1 On the MATLAB toolbar, click the Simulink icon to open the Simulink Library Browser.



2 In the Libraries section of the Simulink Library browser, click **Simscape > SimMechanics**.

3 Select **First Generation** or **Second Generation** for the respective libraries.

4 Click a sublibrary to view its blocks—e.g. **Body Elements** in the **Second Generation** library.



At the MATLAB Command Prompt

You can open the SimMechanics product library through the MATLAB command window. At the MATLAB prompt, enter `sm_lib` at the command line. The library opens, displaying the six sublibraries. Double-click each sublibrary to open it.

Starting a New SimMechanics Model

In this section...

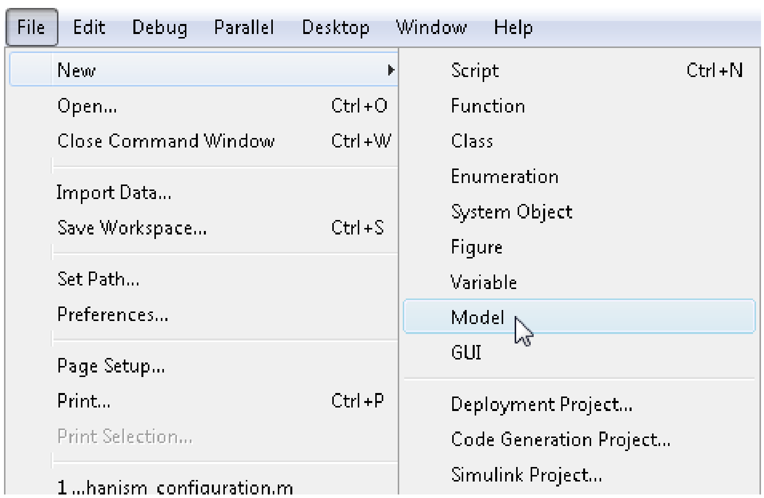
Opening a New Simulink Editor Window

You build a SimMechanics model in a Simulink Editor window. To start a new model, you must first open a new Simulink Editor window.

From the MATLAB Window

To open a new Simulink Editor window from the MATLAB window:

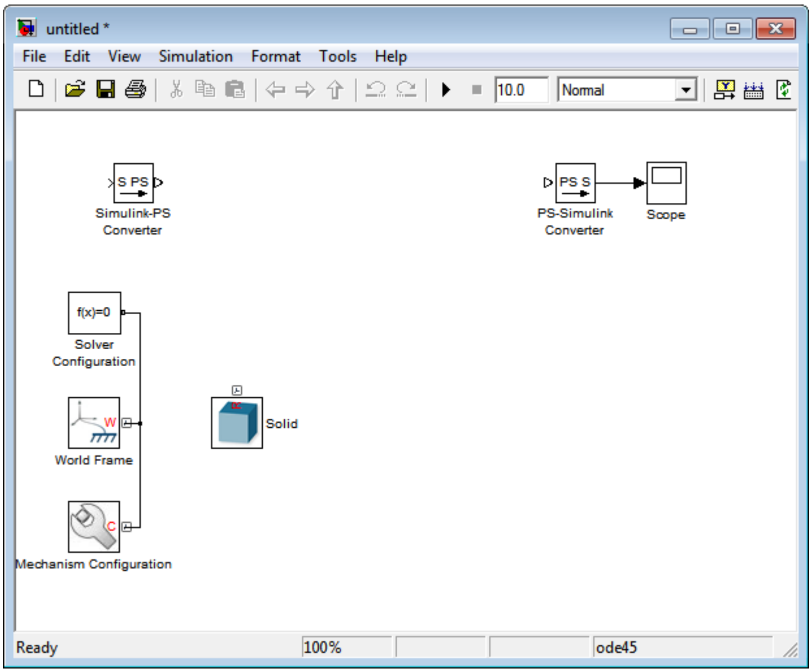
- In the MATLAB toolbar, click **File > New > Model**.



At the MATLAB Command Line

To open a new Simulink Editor window at the MATLAB command line:

- At the MATLAB command line, enter `sm_new`. A new Simulink Editor opens, with some commonly used blocks and the SimMechanics Block library. For more information, see "Opening SimMechanics Libraries" on page 1-10.



Adding a SimMechanics Block to a Model

1 Open the Simulink Library Browser. See "Opening SimMechanics Libraries" on page 1-10.

2 In the Libraries section, select **Simscape→SimMechanics** to access the SimMechanics block libraries.

3 Select one of the SimMechanics block libraries.

4 Select and add a block to the Simulink Editor window containing your model.

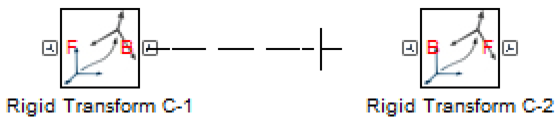
Connecting SimMechanics Blocks

Connecting Two Ports

You can connect two SimMechanics blocks by drawing a connection line between two ports. To do this:

1 Click and hold the appropriate port on the first block you wish to connect.

2 Drag the mouse to a compatible port on the second block you wish to connect.



Note You can only connect a *frame* port to another *frame* port. Likewise, you can only connect a *physical signal* port to another *physical signal* port. For more information, see " Ports" on page 1-6.

An alternate procedure is to:

1 Click the first block you want to connect.

2 On your keypad, press and hold the Ctrl key.

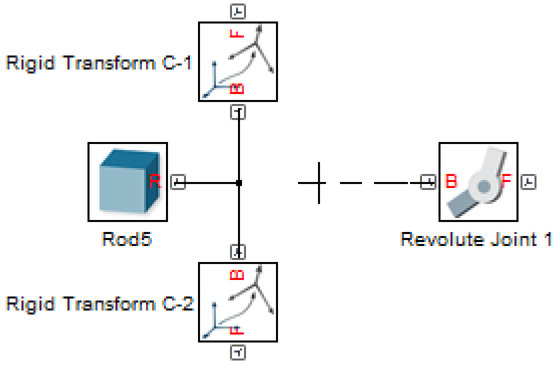
3 Click the second block you want to connect.

Connecting a Port to a Connection Line

You can also connect two blocks by drawing a connection line from a port to a connection line. To do this:

1 Click and hold the appropriate port on the first block you wish to connect.

2 Drag the mouse to the connection line you wish to connect.



Note You can only connect a *frame* port to a *frame* connection line. Likewise, you can only connect a *physical signal* port to a *physical signal* connection line. For more information, see " Ports" on page 1-6.

An alternate procedure is to:

1 Right-click and hold the connection line you wish to connect.

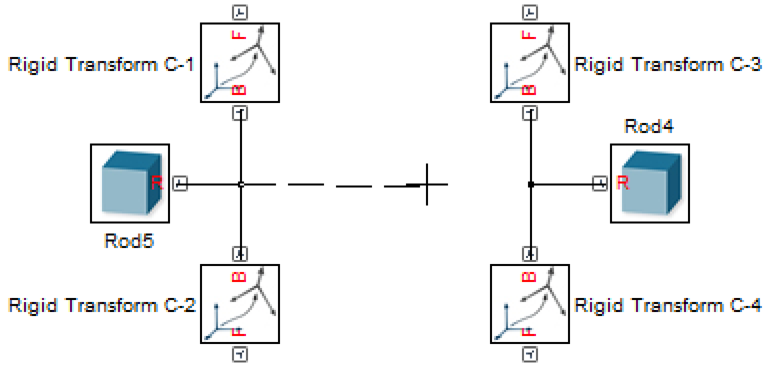
2 Drag the mouse to a compatible port on the block you wish to connect.

Connecting Two Connection Lines

Finally, you can connect two blocks by drawing a connection line between two existing connection lines. To do this:

1 Right-click and hold the first connection line you wish to connect

2 Drag the mouse to the second connection line you wish to connect.

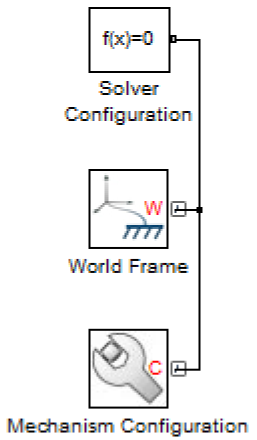


Note You cannot arbitrarily connect a connection line to another connection line. You can only connect a *frame* connection line to another *frame* connection line. Likewise, you can only connect a *physical signal* connection line to another *physical signal* connection line. For more information, see " Ports" on page 1-6.

Essential SimMechanics Blocks

Most SimMechanics models use three essential blocks: World Frame, Mechanism Configuration, and Solver Configuration.

To begin building a SimMechanics model, drag and connect these three blocks in a new Simulink window. When you enter sm_new at the MATLAB command line, SimMechanics automatically inserts and connects the blocks for you in a new Simulink window. For more information see "At the MATLAB Command Prompt" on page 1-12. The following figure shows the three essential blocks connected by a branched connection line.



World Frame

World Frame provides a unique inertial frame in a model, against which all other frames are referenced. Most SimMechanics models employ at least one World Frame block. This block belongs to the SimMechanics Frames and Transforms sublibrary.

Mechanism Configuration

Mechanism Configuration provides a gravity vector and a numerical linearization parameter. Each SimMechanics model can have multiple Mechanism Configuration blocks, but only one per machine. This block belongs to the SimMechanics Utilities sublibrary.

Solver Configuration

Solver Configuration provides solver settings for numerical simulation of a kinematic model. Each SimMechanics model can have multiple Solver Configuration blocks, but only one per machine. This block belongs to the Simscape Utilities sublibrary.

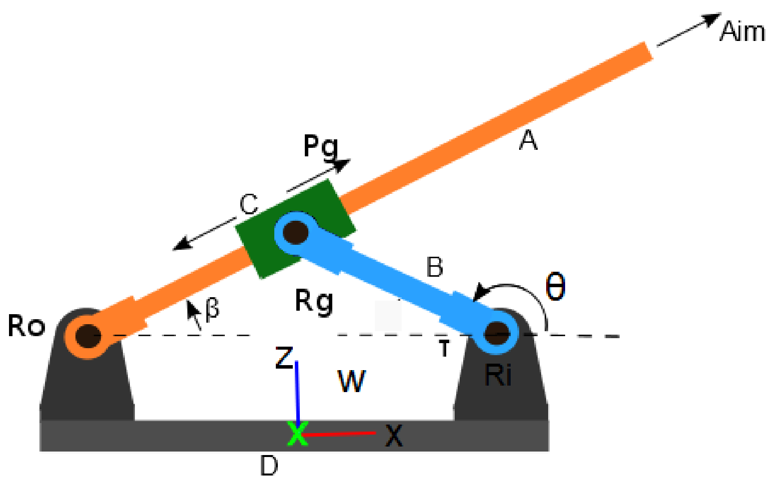
Double Crank Aiming Mechanism Demo

In this section...

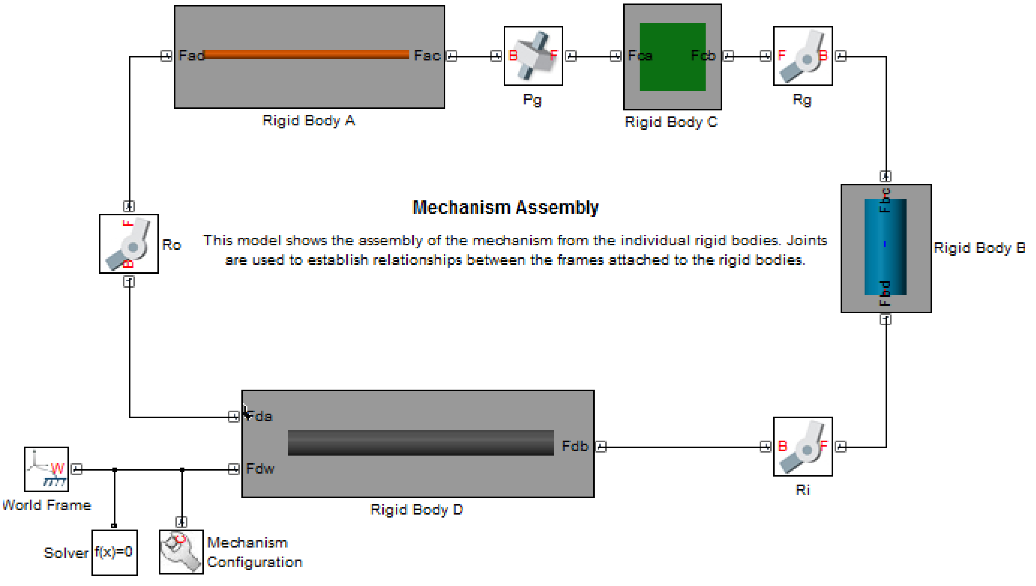
For the suggested workflow to build a model, see the double crank aiming mechanism demo on **How To Build A Model**.

What the Demo Describes

The demo on **How to Build a Model** models and simulates a double-crank aiming mechanism, illustrated in the following figure.

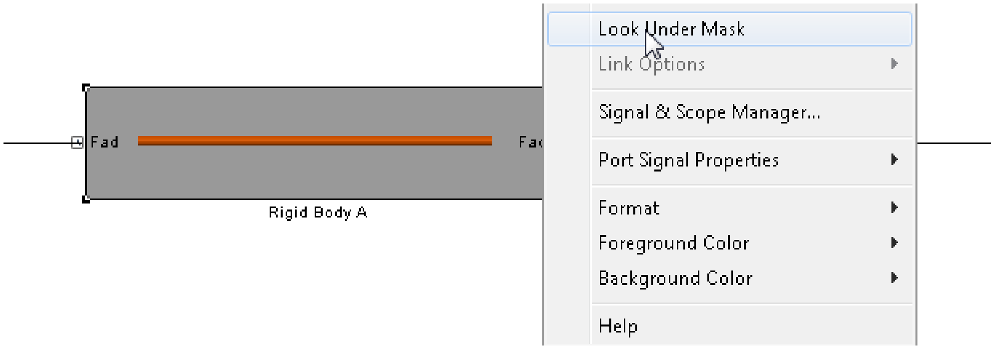


In the model, you can see the rigid bodies and joints that make up the mechanism. There are four links, connected by four joints: three revolute and one prismatic. This system is an instance of a planar four-link mechanism, an important topic in mechanical design.

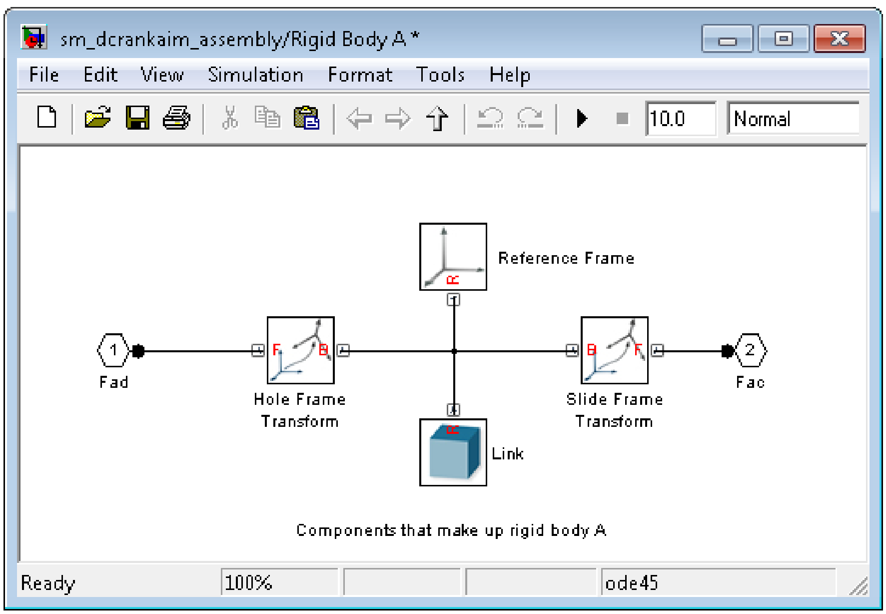


Bodies

Each body is modeled as a self-contained subsystem, composed of other SimMechanics blocks behind a mask. You can inspect and even modify the subsystems in the demo model. To open a subsystem, right click its mask and select **Look Under Mask**.



Inside each subsystem, there are blocks that specify geometry and inertia. There are also blocks that specify any relevant coordinate frames — such as those connected to joints. Subsystems are self-contained, and you can alter them without affecting other subsystems or model architecture. The figure below shows the block diagram for Rigid Body A in the demo model.



Joints

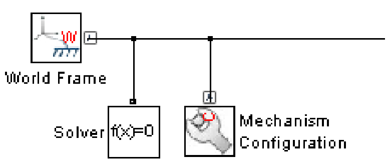
Each joint connects two rigid bodies. The joint confers relative degrees of freedom (DOF) to the two connecting bodies. In the model:

- Prismatic joint **Pg** allows exactly one translational DOF between rigid bodies **A** and **C** — they can move along a single axis relative to each other.
- Revolute joint **Rg** allows exactly one rotational degree of freedom between rigid bodies **B** and **C** — they can rotate about one axis relative to each other.
- Revolute joint **Ri** allows exactly one rotational degree of freedom between rigid bodies **B** and **D** — they can rotate about one axis relative to each other.
- Revolute joint **Ro** allows exactly one rotational degree of freedom between rigid bodies **C** and **D** — they can rotate about one axis relative to each other.

In the joint block dialog box, you can specify initial state conditions. These include position and velocity parameters. You can also actuate the two connecting bodies at the joint with force or torque, and sense their relative motion parameters — position, velocity and acceleration. To open the dialog box, double-click the corresponding joint block.

Essential Blocks

Finally, note the use of the three essential blocks: World Frame, Solver Configuration, and Mechanism Configuration.



World Frame and Solver Configuration are required in every model. The former provides a common ground to all frame networks in a model; the latter specifies model-level simulation parameters. Mechanism Configuration is not required, but important in SimMechanics models — it specifies machine-level simulation parameters, including the gravity vector. For more information, see "Essential SimMechanics Blocks" on page 1-17.

Opening the Demo

To open the demo in a new Simulink window, at the MATLAB command window type `sm_dcrankaim_assembly`.

For a guided discussion of the demo, open its supporting documentation:

- At the MATLAB command prompt, type `showdemo sm_double_crank_assembly`.
- Alternatively, in the Simulink Help browser, navigate to **SimMechanics Second Generation**. Under **Demos**, click **Modeling Techniques**, and then click **How To Build A Model**.

Modeling Workflow

The demo illustrates the suggested workflow for building a model. It is good practice to follow this workflow. The key steps highlighted in the demo are:

1. Identify the rigid bodies and joints that comprise the mechanism to model
2. Build an isolated model for each body
3. Assemble the rigid bodies with joints
4. Refine and troubleshoot assembly using Mechanics Explorer and the Model Report tool
5. Specify the initial configuration using state targets in joint dialog boxes

For more information, refer to the supporting documentation for the demo (see "Opening the Demo" on page 1-23).

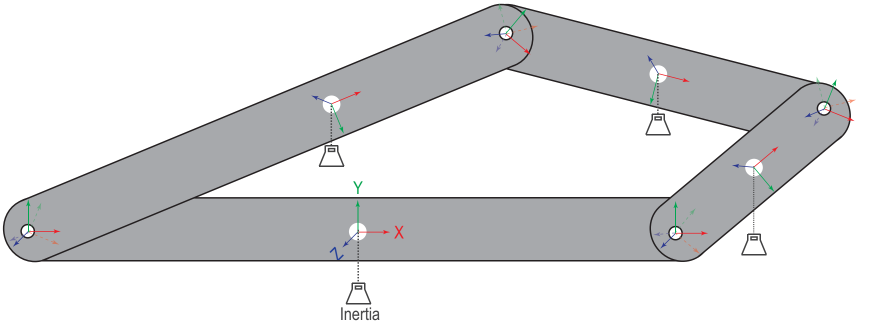
Essentials of SimMechanics Software

A Simple Mechanical Model

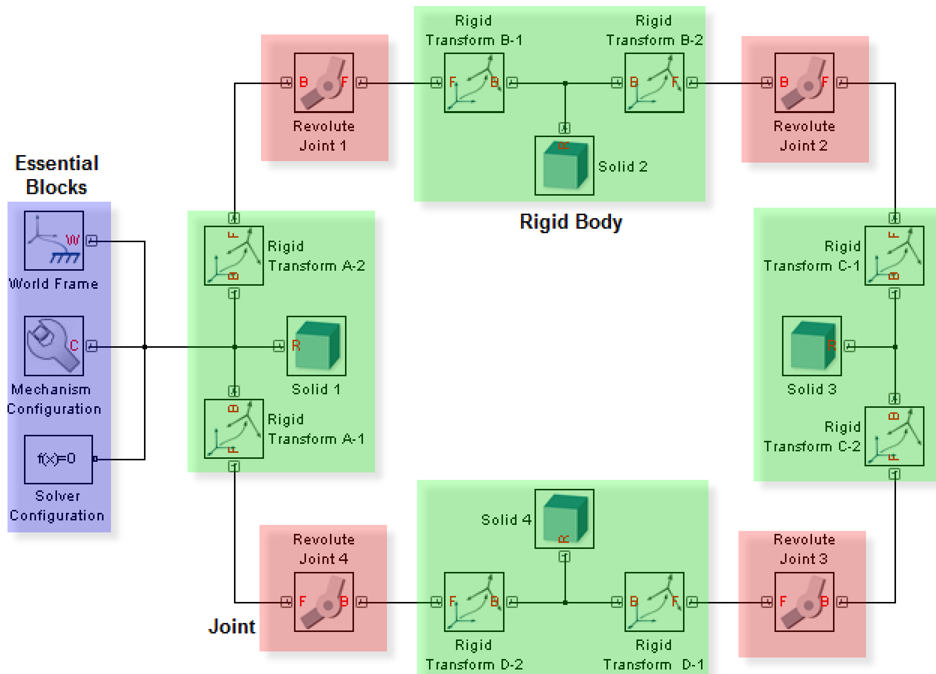
Examining a Four-Bar Mechanism Model

In SimMechanics, you model a mechanical system as a set of rigid bodies connected by joints. One example is the four-bar mechanism illustrated below. This mechanism contains four separate bars, each with a well defined position and orientation, shape and size, mass and inertia.

To model a system like this four-bar mechanism, you must specify a set of properties and coordinate frames for each bar. You do this by adding and connecting SimMechanics blocks on the Simulink Editor window, and providing any relevant parameters through the block dialog box.



The following figure shows an elementary SimMechanics model of the four-bar mechanism. It is a block diagram, with blocks specifying the rigid bodies that make up the mechanism, and the joints used to connect them. Frame ports and connection lines identify a location and orientation for each key point in a rigid body—for example, a joint attachment point. In this model, the constituent bodies are four binary bars (*dyads*), interconnected by four revolute joints.



Each binary bar has two joints, one at each end. In this model, a binary bar is represented by one solid block, specifying its physical properties, and two Rigid Transform blocks, specifying its joint attachment points.

There are three highlighted areas. Each represents a key piece of the model: a rigid body, a joint, and other essential blocks. The rigid body area has two types of blocks:

- Solid
- Rigid Transform

You use these blocks to specify a set of properties and coordinate frames for each rigid body. For more information, see “Rigid Bodies” on page 2-5 . See also “Coordinate Frames” on page 2-12.

The Joint area has one block:

- Revolute Joint

You use this block specify the allowed degrees of freedom between two rigid bodies. There are fourteen different Joint blocks that you can use. For more information, see "Joints" on page 2-21.

The essential blocks area has three blocks:

- World Frame
- Mechanism Configuration
- Solver Configuration

You use these blocks to specify a unique inertial reference frame, a gravity vector, as well as linearization and solver parameters. For more information, see "Essential SimMechanics Blocks" on page 1-17.

The blocks shown above give you the basic tools to build a simple model. With them, you can begin exploring the capability of SimMechanics. You are not limited to these blocks. Add others as needed—to actuate your model, constrain its motion, and sense force or motion parameters.

Rigid Bodies

In this section...

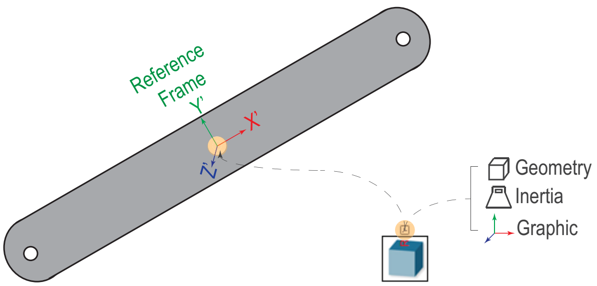
Specifying a Rigid Body with the Solid Block

To specify a rigid body, you use one or more blocks from the Body Elements library. You can use Solid, Inertia, or Graphic blocks. The Solid block combines inertia and graphic options into a single block, and is recommended for new users. This block represents the simplest body that you can define.

Examining the Solid Block

The Solid block provides:

- A single frame port identifying a local reference frame.
- A set of adjustable properties used to define a rigid body.



For solids with simple predefined shapes, the local reference frame has an origin at the center of geometry, and axes aligned with the solid's axes of symmetry. The following table lists the solid properties which you can specify.

Property	What it specifies
Geometry	Shape and size
Inertia	Mass, density, moments and products of inertia
Graphic	Graphical representation

For more information, see the Solid block reference documentation.

Modeling an H-Shaped Bar with the Solid Block

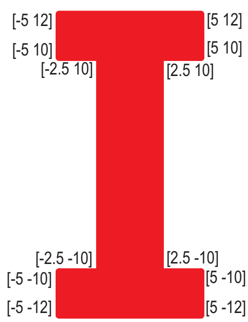
In this section...

H-Bar Example

With the Solid block, you can select a pre-specified geometry (brick, cylinder, or sphere) or a custom geometry with constant cross-section (regular and general extrusion). A more advanced alternative allows you to import a custom geometry using an .STL file.

In this example, you use the Solid block to specify an H-shaped bar. Later, you can use Rigid Transform blocks to specify multiple joint attachment points. For more information, see "Specifying a Joint Attachment Frame with the Rigid Transform Block" on page 2-15.

The following figure illustrates the cross-section of the H-shaped bar modeled in this example.



Selecting and Connecting the Blocks

1 Open a new Simulink Editor window. See “Opening a New Simulink Editor Window” on page 1-13.

2 Make sure the Simulink Library Browser is open. See “Opening SimMechanics Libraries” on page 1-10

3 In the Libraries section of the Simulink Library Browser, select **Simscape > Utilities**.

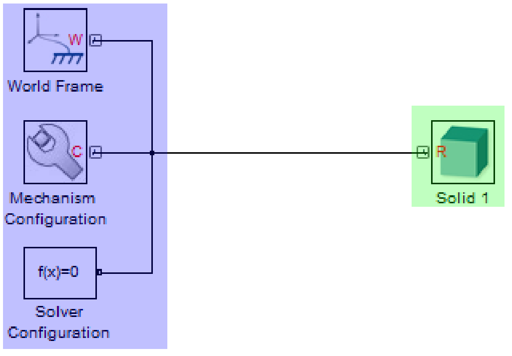
4 In the Utilities library, select one Solver Configuration block and drag it onto the Simulink Editor window.

5 In the Libraries section of the Simulink Library Browser, select **Simscape > SimMechanics > Second generation**.

6 Select and drag one each of the following blocks onto the Simulink Editor window:

Block	Sublibrary
World Frame	Frames and Transforms
Mechanism Configuration	Utilities
Solid	Body Elements library

7 Connect the blocks as shown in the following figure. See “Connecting SimMechanics Blocks” on page 1-15.



Specifying Solid Properties for the H-Shaped Bar

Specifying Geometry

1 In your model, double-click the Solid block to open its dialog box.

2 In the **Geometry** section of the dialog box, select `General Extrusion` from the **Shape** drop-down list.

> **Tip** The General Extrusion option allows you to specify a solid with an arbitrary cross-sectional shape.

3 In the **Cross-section** field, enter a matrix containing the cross-sectional vertices for the H-shaped bar: `[5  12; 5  10; 2.5  10; 2.5 10; 5 10; 5 12;  5 12;  5 10;  2.5 10;  2.5  10;  5  10;  5  12]`.

4 In the unit drop-down list, select `mm` (millimeter).

5 In the **Length** field, enter `200`, and select `mm` for the units.

Specifying Inertia

1 In the Inertia section of the dialog box, leave **Type** as `Calculate from Geometry`.

> **Tip** Leaving **Type** as `Calculate from Geometry` lets SimMechanics compute the moments of inertia for you. When you select this option, you do not need to specify the moments of inertia directly.

2 Check that the **Based on** field is set to `Density`.

3 In the Density field, enter the density of steel, 7750 kg/m^3.

Specifying Graphic

1 In the Graphic section of the dialog box, leave **Type** as `From Geometry`.

2 In the **Visual Properties** field, select `Simple`.

3 Expand the Visual Properties expandable menu.

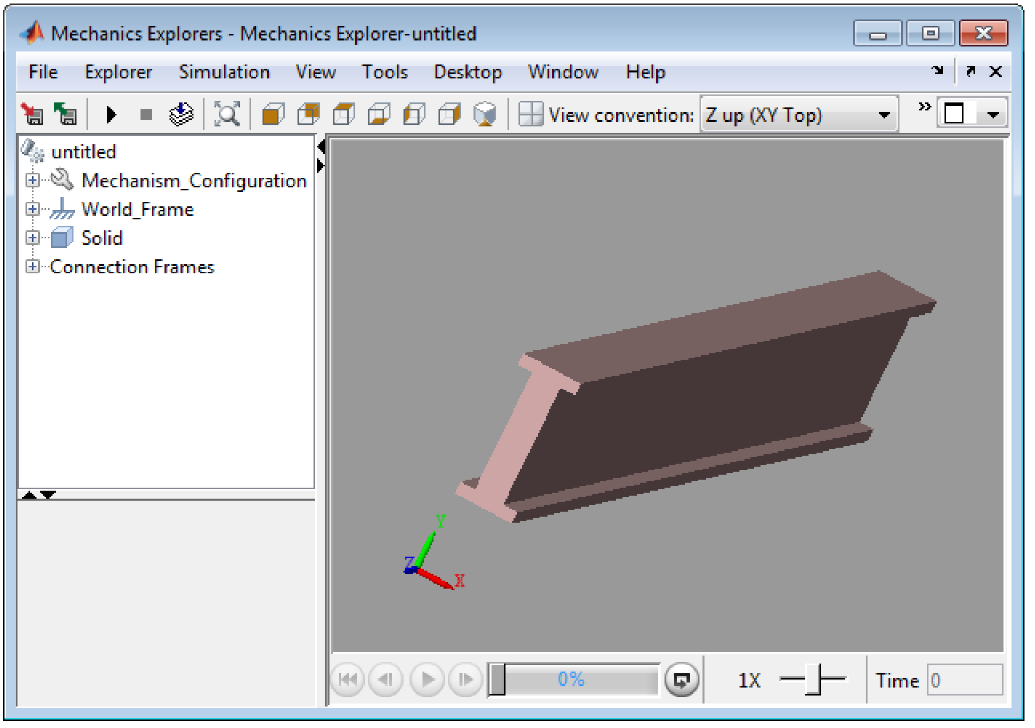
4 In the Color field, select the color you wish to apply to the H-shaped bar. You can also specify the [R G B] color vector, for example [1 0.8 0.8].

5 In the Opacity field, specify a value smaller than 1.0 to make the H-shaped bar transparent.

Visualizing the H-Shaped Bar Model

You can visualize the H-shaped bar model using the Mechanics Explorers window. On your keyboard, press Ctrl+D. Mechanics Explorers opens up, displaying the H-shaped bar you just modeled.

Note Mechanics Explorers is an advanced visualization and animation tool. You can use it to visualize, animate, and optimize your mechanical model.


Mechanics Explorers - Mechanics Explorer-untitled
File
Explorer
Simulation
View
Tools
Desktop
Window
Help
View convention:
Z up (XY Top)
untitled
Mechanism_Configuration
World_Frame
Solid
Connection Frames
Y
Z
X
0%
1X
Time
0

Coordinate Frames

In this section...

Specifying Coordinate Frames with the Rigid Transform Block

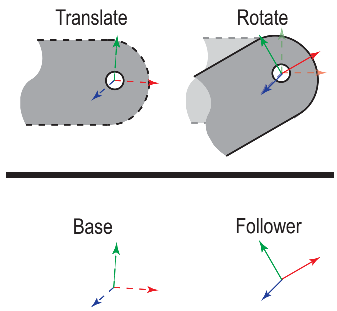
Coordinate frames allow you to specify both position and orientation for any rigid body and joint in your model. This spatial information allows two rigid bodies to connect via a joint placed at the right location and with the proper orientation. It also allows you to exert a force or torque anywhere, and in any direction, on a rigid body. You must define a coordinate frame for every point of interest in a model. To define an arbitrary frame with respect to another frame, use the Rigid Transform block.

The Rigid Transform block operates on one frame (the *base* frame) to define a second frame (the *follower* frame). You can use this block to specify a frame with arbitrary position and orientation. Use multiple Rigid Transform blocks to specify multiple frames in your model.

Note Itself, the Rigid Transform block does not represent a coordinate frame. Instead, it represents the *mathematical operation* that is used to create one frame from another.

The Rigid Transform block performs two operations:

- Translation
- Rotation



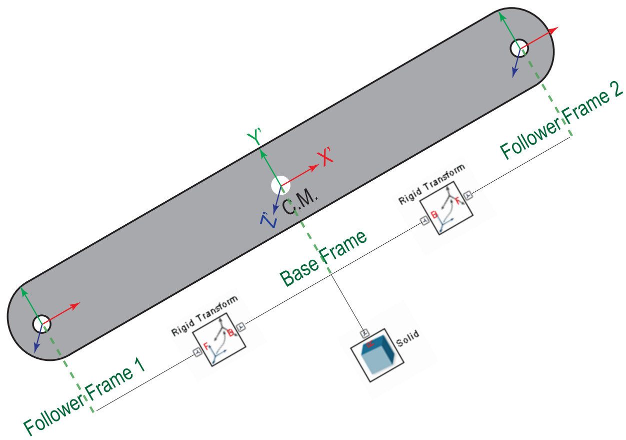
When using a Rigid transform block, you are free to specify translation only, rotation only, or both. When you specify both translation and rotation operations, translation applies first, followed by rotation. If you do not specify a transform operation, the Rigid Transform block behaves as a frame connection line, making the base and follower frames coincident with each other.

Note The follower frame of the Rigid Transform block is *rigidly* defined with respect to the base frame. There is a static relationship between these two frames. They cannot move independently, but only as a set.

Examining the Rigid Transform Block

The Rigid transform block provides two frame ports:

Frame	Frame Port	Connect this port to identify...
Base	B	The frame on which the specified transform operates
Follower	F	The frame which results from the specified transform



The following table lists the properties which you can specify using the Rigid Transform block. These properties correspond to the Rigid Transform operations listed above (translation *along* and rotation *about* a single axis).

Property	Function
Rotation	Specify a method to rotate the follower frame with respect to the base frame
Translation	Specify a method to translate the follower frame with respect to the base frame

For more information, see theRigid Transform block reference documentation.

Specifying a Joint Attachment Frame with the Rigid Transform Block

In this section...

Specifying a Joint Attachment Frame

In SimMechanics, you specify any frame of interest using the Rigid Transform block. Frames of interest include joint, actuation, and sensing points in a rigid body. In this example, you use two Rigid Transform blocks to specify two joint attachment frames in the H-shaped bar modeled above (see "Modeling an H-Shaped Bar with the Solid Block" on page 2-7). Once you've specified the joint attachment points in a binary bar, you can connect them with joints to assemble a four-bar mechanism.

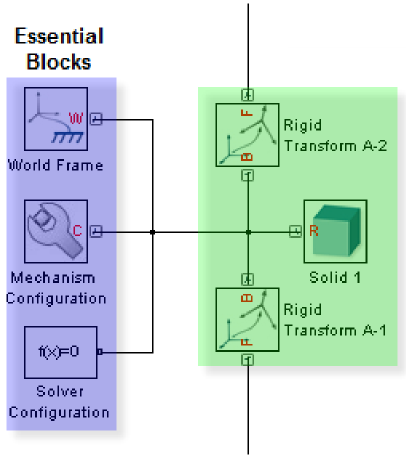
Note This example does not show you how to use the Joint block. For an example showing how to connect two binary bars with a joint block, see "Specifying a Joint Between Two Binary Bars" on page 2-23

Selecting and Connecting the Blocks

1. In the Libraries section of the Simulink Library Browser, select **Simscape > SimMechanics > Second Generation > Frames and Transforms**.

2. Select and drag two Rigid Transform blocks to the Simulink Editor window containing your model.

3. Connect the Base (B) frame ports to the connection line attached to the Reference (R) frame port belonging to the Solid block.



Note You can rotate a block to better position its base frame port. To rotate a block, on your keyboard press Ctrl+R. Each time you use this shortcut, the selected block rotates 90 degrees clockwise.

Recall that a connection line represents a single frame along its entire length. Also, two connection lines connected to each other share the same frame along their entire lengths. In this figure, the two Rigid Transform *base* frames are coincident with the Solid reference frame. The *follower* frames are defined relative to the Solid reference frame.

Specifying Rigid Transform Properties

Rotation

You can rotate the follower frame relative to the base frame. You do this in the Rotation section of the Rigid Transform dialog box. With the graphic markers added to your model, it is easy to visualize the orientation of each frame.

1 In your model, double-click the Rigid Transform block to open its dialog box.

2 In the dialog box, expand the **Rotation** menu.

3 Set the **Rotation Method** to `Aligned Axes`.

4 Specify **Pair 1** and **Pair 2**:

Frame	Base	Follower
Pair 1	-X	+Z
Pair 2	+Z	+X

Translation

You can translate the follower frame relative to the base frame. You do this in the Translation section of the Rigid Transform dialog box. With the graphic markers added to your model, it is easy to visualize the location of each frame.

1 In the Rigid Transform dialog box, expand the **Translation** menu.

2 Set the **Method** field to `Standard Axis`.

3 Set the **Axis field** to `+Z`.

4 In the **Offset** field, specify the distance from the bar's center of geometry to the joint location. In the H-shaped bar example, specify `+100 mm` for the first transform, and `-100 mm` for the second transform.

Adding Visualization Aids

Selecting and Connecting the Blocks

You can add a Graphic block to visualize any frame in your model. This block has no physical properties, and does not affect model dynamics. The Graphic block functions strictly as a visual aid.

1 In the SimMechanics Second Generation library, select **Body Elements**.

2 Select and drag three **Graphic** blocks to the Simulink Editor window containing your model.

3 Connect two **Graphic** blocks to the follower frame port (F) on the **Rigid Transform** blocks.

4 Connect the remaining Graphic block to the frame connection line attached to the solid block.

Specifying Graphic Properties

You can specify different graphic icons to represent a frame. The Frame icon is particularly useful to visualize the relative orientation of different frames. You can select a different color for each frame, and you can specify a size that's convenient for you.

1 Double-click a Graphic block to open its dialog box.

2 Set the **Shape** field to Frame to visualize the attached frame as a coordinate frame.

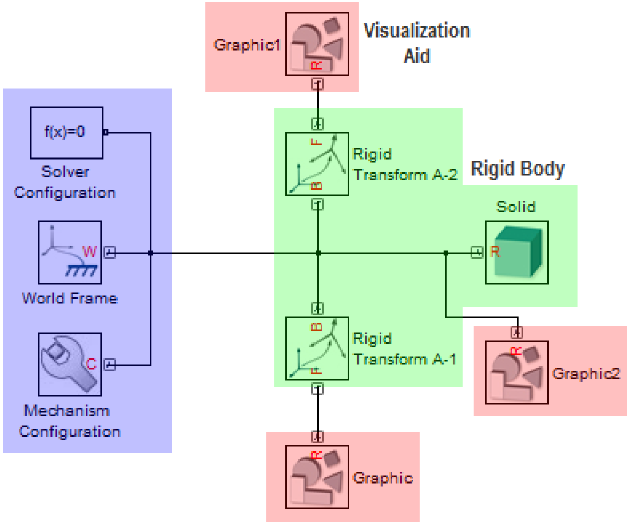
3 Specify a size that's convenient for you, for example 30 pixels.

Note Graphic icon size is defined in pixel units. The size remains constant regardless of the zoom setting in the Mechanics Explorers window.

---

4 In the **Visual Properties** field, select a unique color for your graphic.

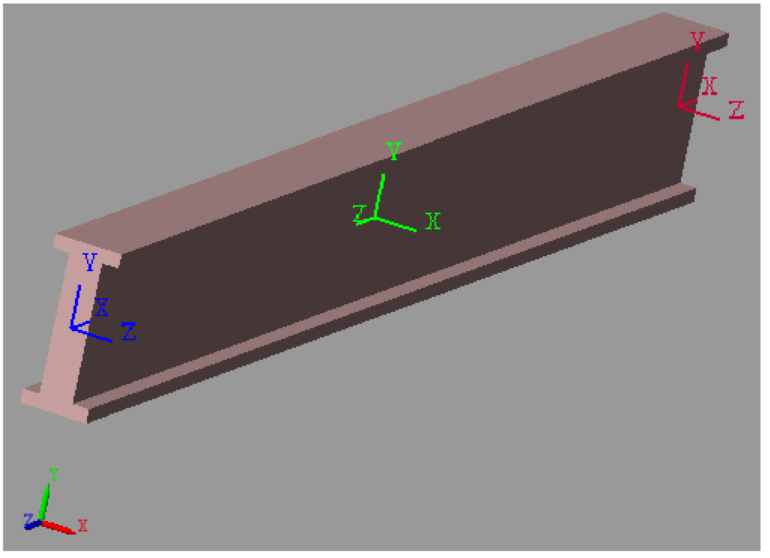
5 Repeat for the remaining Graphic blocks.



Visualizing the H-Shaped Bar Model

You can visualize the coordinate frames in the H-shaped bar model using the Mechanics Explorers window. On your keyboard, press Ctrl+D. A Mechanics Explorers opens up, displaying the H-shaped bar you just modeled. The coordinate frames appear as frame icons, each corresponding to one Graphic block in your model. For more information, see "Model Visualization and Animation" on page 2-48.

Note Mechanics Explorers is an advanced visualization and animation tool. You can use it to visualize, animate, and optimize your mechanical model.


Y
X
Z
Y
Z
X
Y
X
Z
Y
Z
X

Joints

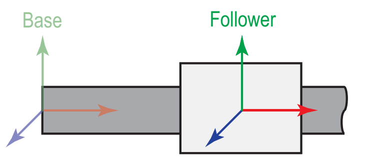
In this section...

Specifying Joints with Joint Blocks

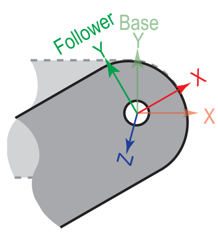
Once you've defined two rigid bodies in a model, you can connect them using a Joint block. Joints specify how two connecting bodies can move. They limit the degrees of freedom between the two connecting bodies, constraining them to move *along* or *about* a limited set of axes.

In SimMechanics, every joint is defined in terms of *joint primitives*: elementary joints, from which all other joints can be built. There are three joint primitives:

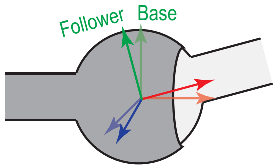
- Prismatic primitive: constrains motion to translation along a single axis.



- Revolute primitive: constrains motion to rotation about a single axis.



- Spherical primitive: constrains motion to rotation about three Cartesian axes.

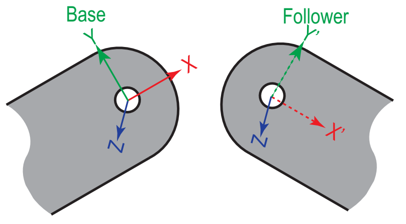


Examining the Joint Blocks

A Joint block specifies a time-varying relationship between two frames, a base and a follower. . This transformation is similar to that in Rigid Transform blocks, except that it is *time dependent*. For more information, see "Examining the Rigid Transform Block" on page 2-13.

Every Joint block has two frame ports, *base* and *follower*:

- *Base*—identifies the frame to be transformed.
- *Follower*—identifies the resulting frame after the transformation.



The Joint block gives you the option to specify an initial state, apply forces and sense motion for each joint primitive.

Property	Function
State Targets	Specify the initial position and velocity of a joint primitive
Internal Mechanics	Specify damping and spring forces acting on a joint primitive
Actuation	Select a mode of actuation for a joint primitive
Sensing	Select the motion parameters you wish to sense in a joint primitive

Specifying a Joint Between Two Binary Bars

In this section...

Specifying a Joint

You can use a Joint block to connect any two rigid bodies with a desired set of degrees of freedom. In this example, you use a Joint block to connect two H-shaped binary bars as defined above (see "Specifying a Joint Attachment Frame with the Rigid Transform Block" on page 2-15.

Selecting and Connecting the Blocks

1 Open the SimMechanics Second Generation sublibrary. See "Opening SimMechanics Libraries" on page 1-10

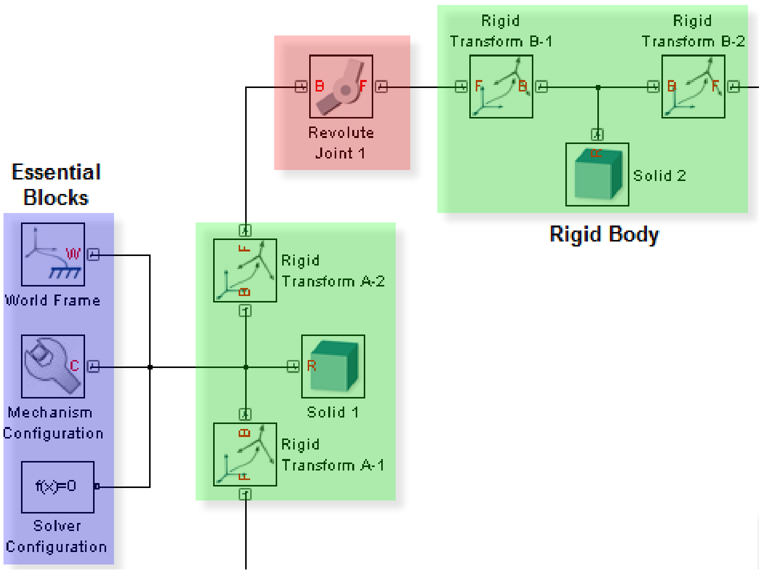
2 In the Joints sublibrary, select and drag a Revolute Joint block into a Simulink Editor window containing your model.

3 In each binary bar, identify the Rigid Transform block which specifies the joint attachment point.

4 Connect the first Rigid Transform block to the *base* frame port on the Revolute Joint block.

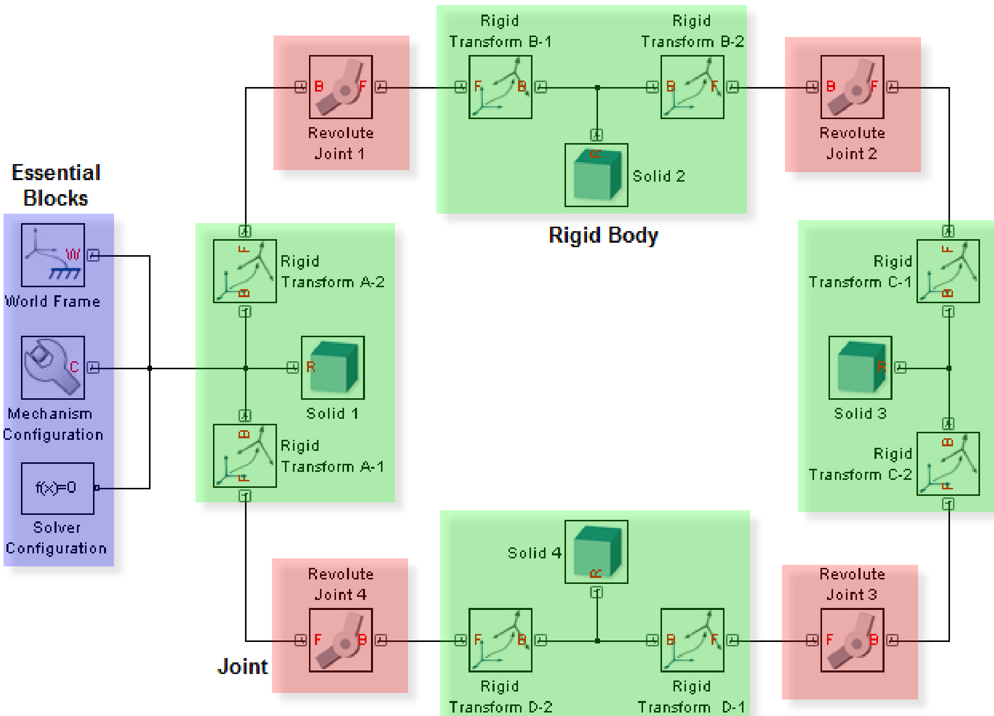
5 Connect the second Rigid Transform block to the *follower* frame port on the Revolute Joint block.

The following figure shows portion of a SimMechanics model. In this model, a revolute joint block connects two rigid body subsystems, giving them a single rotational degree of freedom about a shared axis.



Modeling a Complete Four-Bar Mechanism

Each Joint block connects two and only two rigid bodies. You can use as many joint blocks as needed to model a multi-body system. The following figure shows a complete model of a simple four-bar mechanism.



To build a complete model of a four-bar mechanism:

1 Select, drag and connect the three blocks outlined in "Essential SimMechanics Blocks" on page 1-17 to your model window.

2 Model the four binary bars employed in the four-bar mechanism. For more information, see "Specifying a Rigid Body with the Solid Block" on page 2-5 and "Specifying Coordinate Frames with the Rigid Transform Block" on page 2-12.

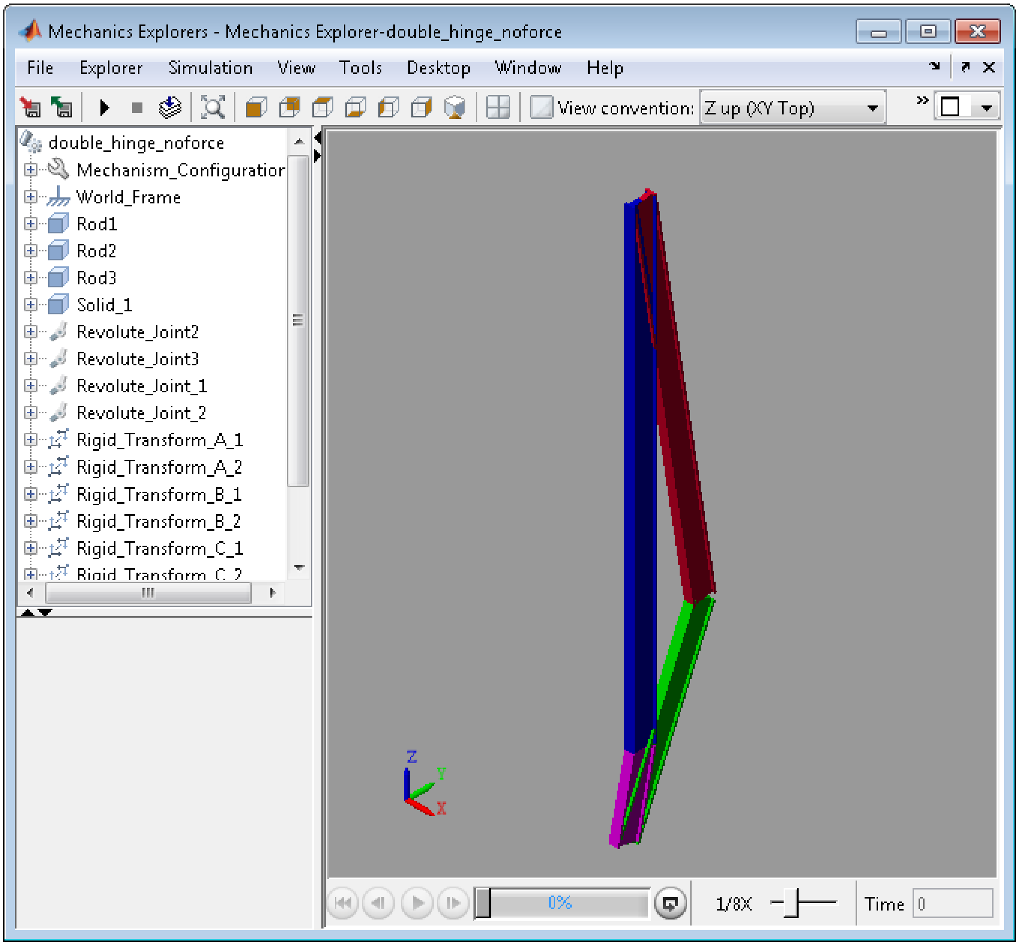
Tip When two rigid bodies are similar to each other, you can often build a model for one body, then copy and paste it to model the second body. You can change any properties as needed by opening the Solid and Rigid Transform block dialog boxes.

---

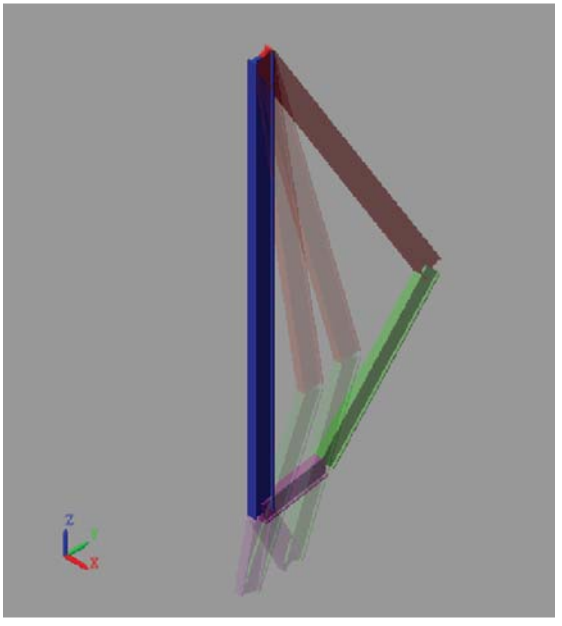
3 Select and drag four Revolute Joint blocks from the SimMechanics Joints library to your model window.

4 Connect each pair of binary bars with a single Revolute Joint box. For more information, see "Specifying Joints with Joint Blocks" on page 2-21.

The following figure shows the model of a four-bar mechanism in its initial state, using the Mechanics Explorers tool.



Once you have a working SimMechanics model, you can simulate and visualize its dynamic behavior using the Mechanics Explorers tool. The following figure illustrates the model of a four-bar mechanism at different points in time during the simulation. For more information, see "Model Visualization and Animation" on page 2-48.



Specifying Joint Initial State Targets

SimMechanics can assemble your model automatically into an initial state configuration. However, you can specify the desired initial state for any joint in your model. The initial state of a joint includes the following parameters:

- Initial position
- Initial velocity

Specifying a joint's initial state is especially important if there are multiple possible initial states. In this case, you can guide SimMechanics assembly into the right state simply by providing joint initial state targets. You do not have to provide exact initial state values for a joint—approximate values suffice.

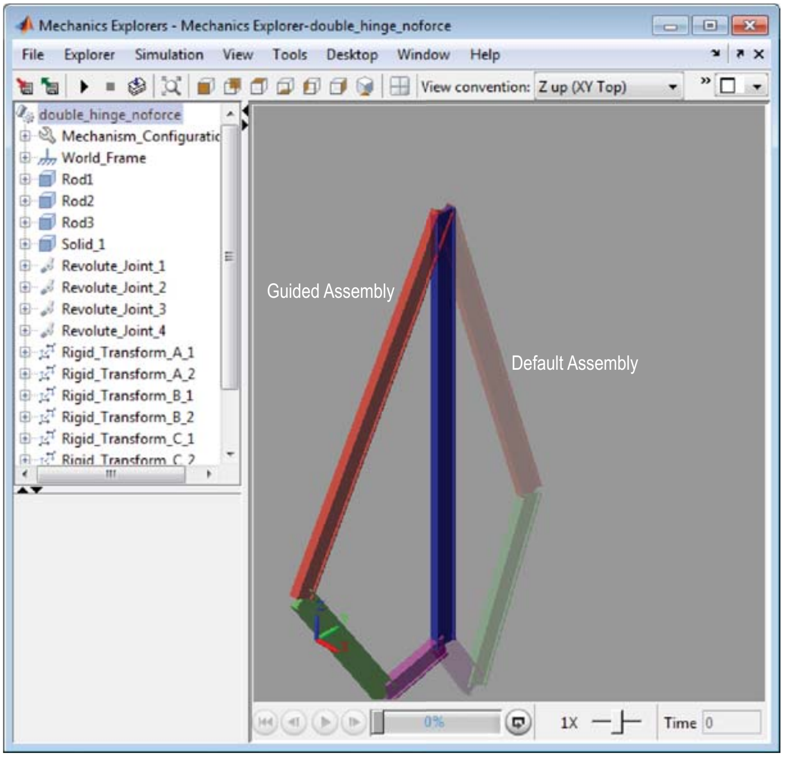
When you specify a joint's initial state targets, there are two priority settings which you can select:

- High—SimMechanics tries to match the specified joint initial state *exactly*.
- Low—SimMechanics tries to match the specified joint initial state *approximately*.

To specify a joint's initial state targets:

1. In your model window, double-click any joint block whose initial state you wish to specify.
2. In the **Properties** area of the dialog box, click the **State Targets** expandable menu to display more options.
3. Check the box for the parameter you wish to specify, Position or Velocity. Additional fields appear.
4. Specify a priority setting for the initial state target: `High` or `Low`.
5. Enter an initial state value and select its physical units.
6. Close the joint block dialog box.

The following figure shows a four bar mechanism in its default assembly configuration, and in a guided assembly configuration.


Mechanics Explorers - Mechanics Explorer-double_hinge_noforce
File Explorer Simulation View Tools Desktop Window Help
View convention: Z up (XY Top)
double_hinge_noforce
Mechanism_Configuratic
World_Frame
Rod1
Rod2
Rod3
Solid_1
Revolute_Joint_1
Revolute_Joint_2
Revolute_Joint_3
Revolute_Joint_4
Rigid_Transform_A_1
Rigid_Transform_A_2
Rigid_Transform_B_1
Rigid_Transform_B_2
Rigid_Transform_C_1
Guided Assembly
Default Assembly
0%
1X
Time 0

Forces and Torques

Forces and Torques in SimMechanics

Forces and torques act to alter the motion state of a rigid body. In mechanical systems, there are different forces of interest, including actuation, damping, and spring forces. In SimMechanics, you can input, output, and sense forces and torques as time-dependent physical signals.

Depending on the application, you can apply a force or torque either at a joint or on a rigid body. Because joint actuation is very common in mechanical systems, SimMechanics Joints blocks are designed to accept actuation inputs. To actuate a rigid body or subsystem instead of a joint, use one of the blocks provided in the Forces and Torques library.

To input and output forces and torques, SimMechanics blocks employ two types of ports, outlined in the following table.

Port Type	What it does	Where you can find it
Physical signal inport ▷|	**Receive a physical signal** specifying one or more forces and torques	External Force and Torque *only*
Frame 	**Identify the frame** subjected to a forces and torques	All Forces & Torques blocks

For more information, see “ Ports” on page 1-6.

Applying a Force or Torque on a Joint

In this section...

Actuation and Internal Forces

Joints blocks accept physical signals specifying forces and torques acting on a joint. You have the option to actuate each joint primitive individually. Each joint primitive is associated with a unique physical signal inport, which receives the actuation signal. Each joint primitive provides a different actuation mode, characteristic of the type of motion it allows, linear or rotational. The following table lists the actuation modes that you can specify for different joint primitives.

Joint Primitive	Actuation Mode
Prismatic	`None`, `Force`
Revolute	`None`, `Torque`
Spherical	`None`, `Torque`

Joints blocks also accept a set of parameters specifying internal forces, including spring and damping forces. You can provide internal force parameters individually for each joint primitive. Internal force parameters that you can provide for each internal force include:

- Spring Force: `Equilibrium Position`, `Spring Stiffness`
- Damping Force: `Damping Coefficient`

Actuating a Joint

To actuate a joint, perform the following steps:

1. In the Simulink window for your model, double-click the joint you wish to actuate.

2 In the joint block dialog box, click the expandable menu for the joint primitive you wish to actuate (e.g. **Z Prismatic Primitive (Pz)**).

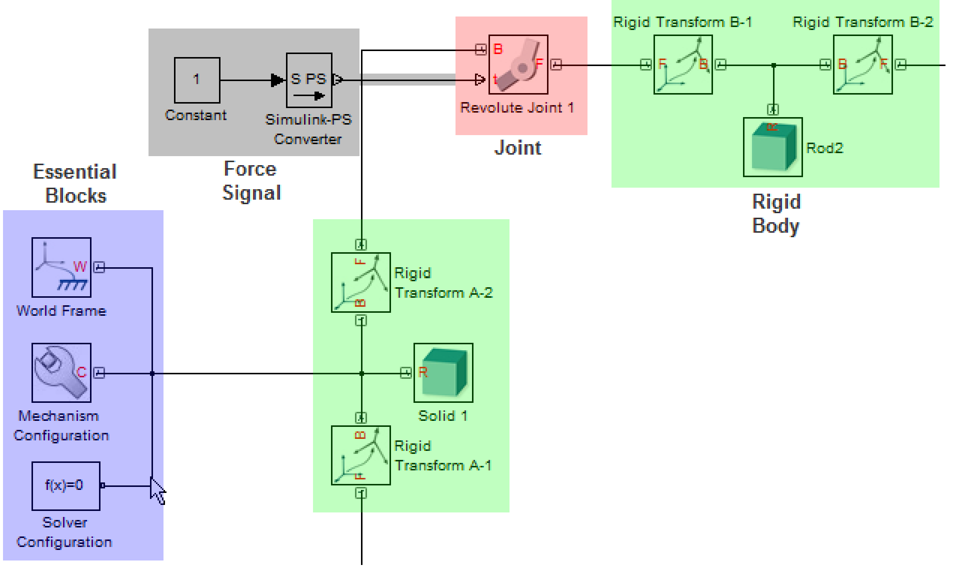
3 Click the **Actuation** expandable menu.

4 In the **Mode** drop-down list, select the appropriate actuation input type to expose a physical signal inport: `none`, `force`, or `torque`.

Note Force actuation mode is present in the Actuation menu for prismatic joint primitives *only*. Torque actuation mode is present in the Actuation menu for revolute joint primitives *only*.

---

5 Attach a connection line to the exposed physical signal inport, specifying a force input. For more information, see "Specifying an External Force or Torque" on page 2-36.



Applying Damping and Spring Forces to a Joint

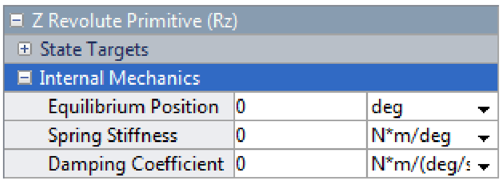
To add damping or spring mechanics to a joint primitive:

1 In the Simulink window for your model, double-click the joint you wish to actuate.

2 In the joint block dialog box, click the expandable menu for the joint primitive you wish to actuate (e.g. **Z Prismatic Primitive (Pz)**).

3 Click the expandable menu for **Internal Mechanics**.

4 Enter the equilibrium position, spring stiffness, and damping coefficient for the joint.



For more information, see the "Joints" reference documentation.

Applying a Force or Torque on a Rigid Body

In this section...

Forces and Torques

Forces and Torques blocks allow you to specify external or internal forces. You can *input* or *output* forces and torques acting on any frame in your model.

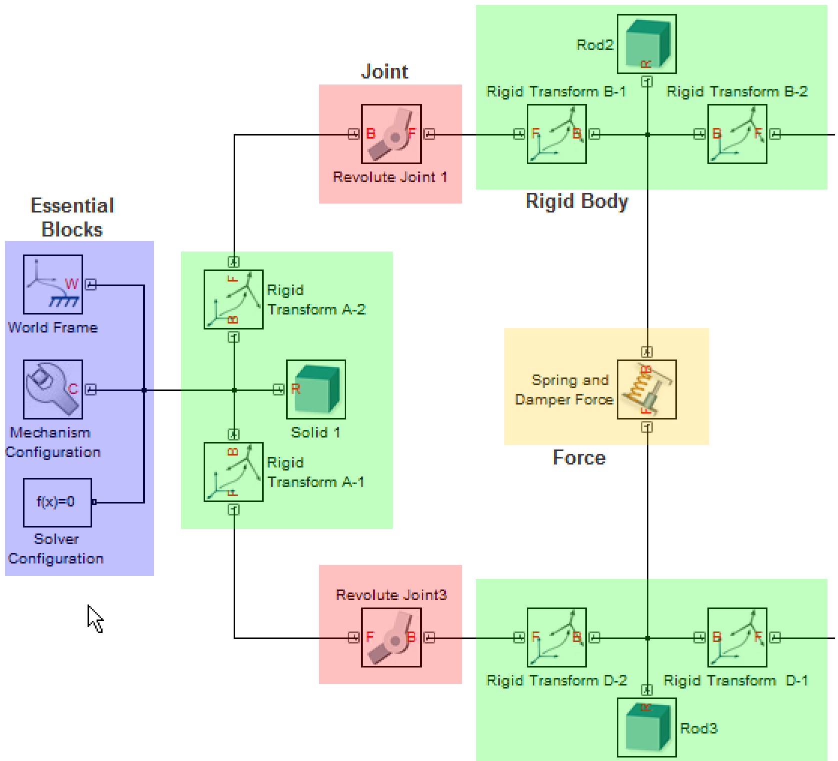
Applying a Force or Torque Between Two Rigid Bodies

To add a force block to a model:

1. In the Simulink Library browser, expand the SimMechanics **Second generation** block library.
2. From the **Forces and Torques** sublibrary, select and drag a force block into the Simulink Editor window containing your model.
3. Connect the follower frame port of the force block to the frame you wish to actuate.
4. Connect the base frame port of the force block to the frame where the force originates.

Note Only two Forces and Torques blocks have a base frame port: Spring and Damper Force, and Inverse Square Law Force. The External Force and Torque block applies a force originating outside the system, and does not need a base frame port.

The following figure shows part of a SimMechanics model containing two rigid bodies connected by a Spring and Damper Force.



Force Parameters and Signals

Force blocks accept either a set of force constants or a physical signal as input. The following table outlines the method used in each block:

Force Block	Specification Method
External Force and Torque	Provide an external physical signal as input.
Spring and Damper Force	Provide a set of force constants.
Inverse Square Law Force	Provide a set of force constants.

Specifying Force Parameters

To specify a force constant:

1 In your model, double-click the force block you wish to specify.

2 In the block diagram box, enter the appropriate force constant and other required force parameters.

Properties

Natural Length	0	m
Spring Stiffness	0	N/m
Damping Coeffi...	0	N/(m/s)
Sense Force	☐	

Specifying an External Force or Torque

The External Force & Torque block provides a physical signal inport, which accepts a force input connection line. To connect a physical signal connection line to the External Force & Torque force block:

1 Double-click the External Force & Torque block and select the force components you wish to specify. The corresponding physical signal outports appear on the affected block.

2 Connect the exposed physical signal outports to a physical signal connection line defining the time-dependent force signal.

You can use Simulink source blocks to specify an arbitrary force or other physical signal. However, Simulink signals are strictly numeric and carry no physical units. To use a Simulink signal in a SimMechanics model, you must first convert it to a physical signal.

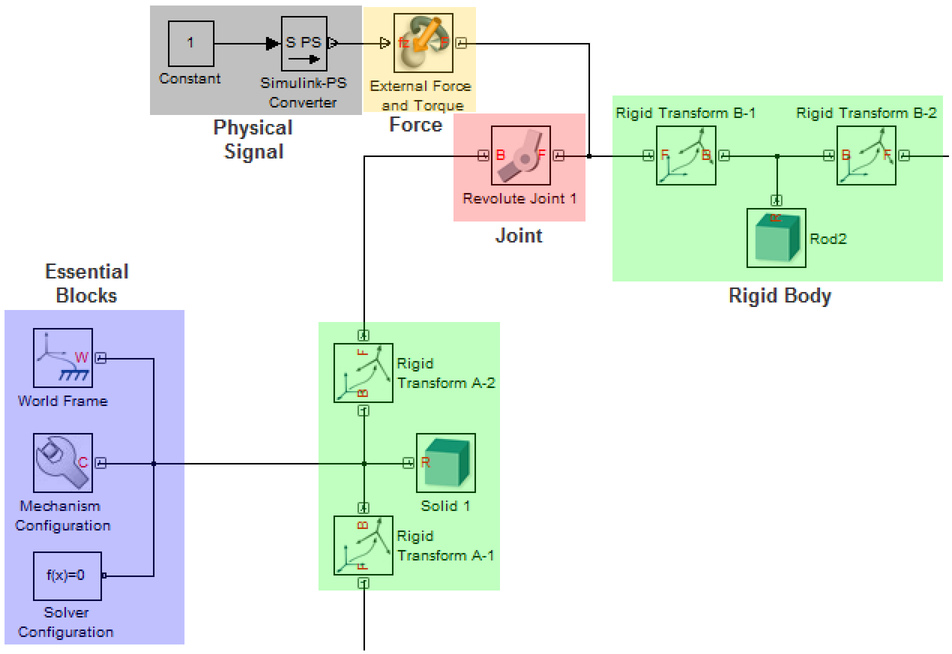
1 Drag a Simulink-PS Converter block to the Simulink Editor window containing your model.

2 Connect its physical signal *outport* to the physical signal *inport* on the force block.

3 Connect its Simulink signal *inport* to the Simulink signal *outport* on the desired Simulink Source block.

4 Double-click the Simulink-PS Converter block and specify the desired physical units.

The following figure shows portion of a SimMechanics model, using a converted Simulink signal to specify a time dependent external force.



Force and Motion Sensing

In this section...

Sensing in SimMechanics

In SimMechanics, you can sense forces acting on a rigid body. When you sense a force, SimMechanics records its value at each instant in time, so that you can plot them in a chart or feed them into a control system. You can only sense forces at points defined by a coordinate frame. Before you sense a force at an arbitrary point, check that you've defined a coordinate frame at that point. For more information, see"Coordinate Frames" on page 2-12.

Two blocks allow you to sense forces:

- Spring and Damper Force
- Inverse Square Law Force

You can only sense forces if your model includes one of these two Forces and Torques blocks.

You can also sense motion parameters like the linear and angular position, velocity, and acceleration at a point in a machine. Like forces, you can only sense motion parameters at points defined by a coordinate frame. You can sense motion parameters through all Joint blocks except the Weld Joint block. You can only sense motion parameters if your model includes one of these Joint blocks.

Sensing Forces and Torques

To sense force in a mechanical model:

1. Confirm that your model uses at least one of the two following blocks:
   - Spring and Damper Force

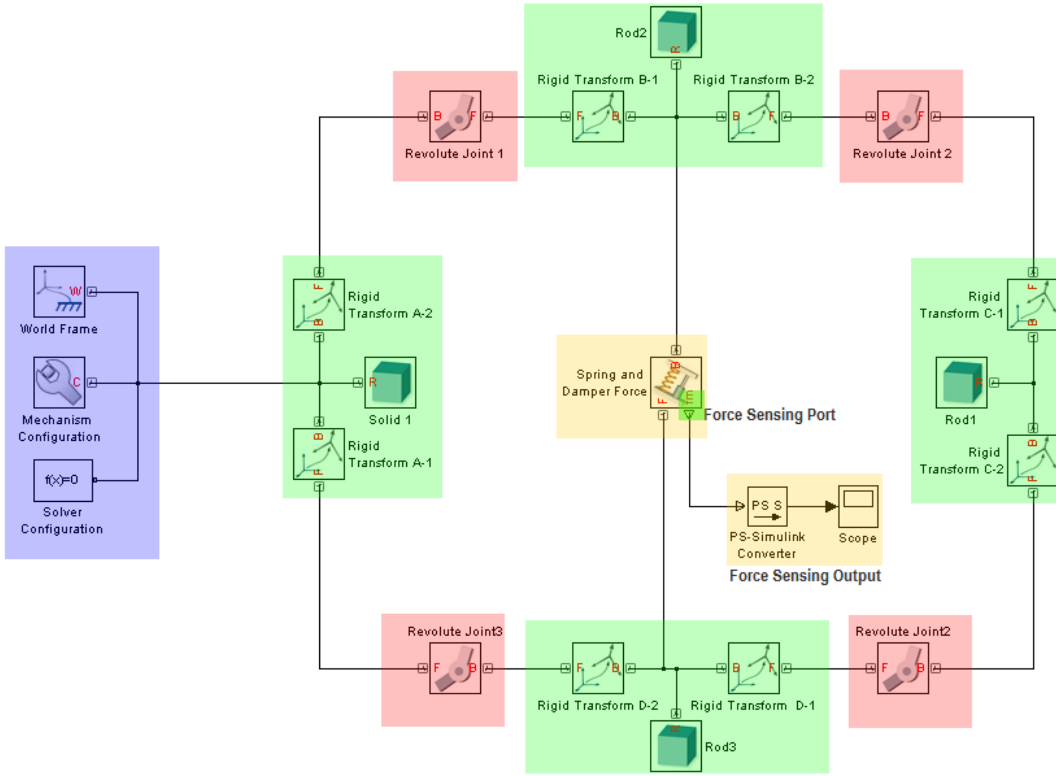
- Inverse Square Law Force

2 Double-click one of the above force blocks in your model.

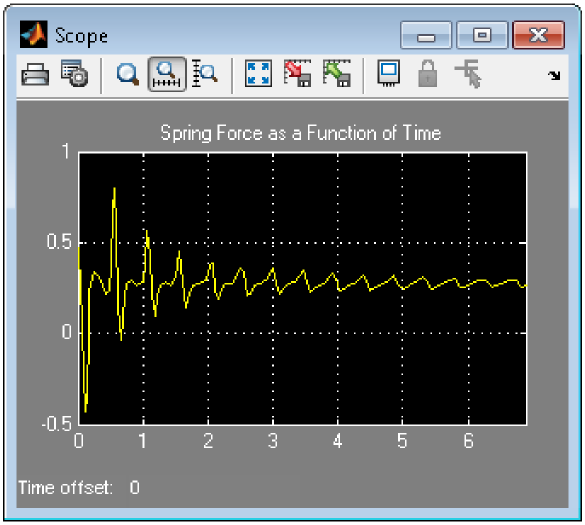
3 In the block dialog box, check the **Sense Force** box. A physical signal port is exposed allowing you to output the force signal. You can plot the force signal using a Simulink Scope or XY Graph block, or you can feed it to a feedback loop used to control your model.

Note To feed a Simscape-based physical signal to a Simulink block, you must first convert it to a Simulink signal using a PS-Simulink Converter block. For more information, see the documentation for the PS-Simulink Converter block.

The following figure illustrates force-sensing in the model of a four-bar mechanism as shown in "Modeling a Complete Four-Bar Mechanism" on page 2-24.



Adding spring and damping forces between two of the binary bars yields a periodic force signal that decays as a function of time. In the model illustrated above, you can double-click the scope block to open a plot of the time-varying force signal between the two binary bars.



Sensing Motion

To sense motion in a mechanical model:

1. Confirm that your model uses at least one Joint block other than Weld Joint.

2. Double-click the Joint block at which you wish to sense motion parameters.

3. In the dialog box, click the expandable menu for the joint primitive you wish to sense—for example **Z Revolute Primitive (Rz)**.

4. click the **Sensing** expandable menu.

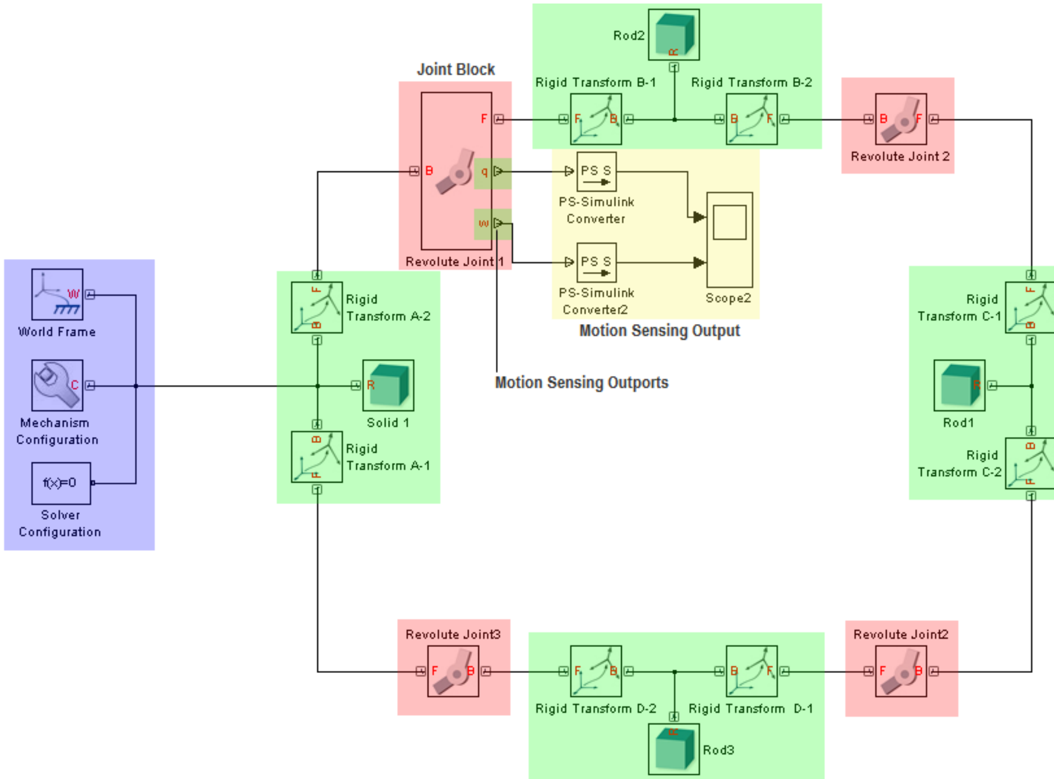
5. Check the motion parameters which you wish to sense. Options include **Position**, **Velocity**, and **Acceleration**.

Note You can sense linear or angular motion parameters. Prismatic primitives support sensing for linear motion parameters. Revolute and spherical primitives support sensing for angular motion parameters.

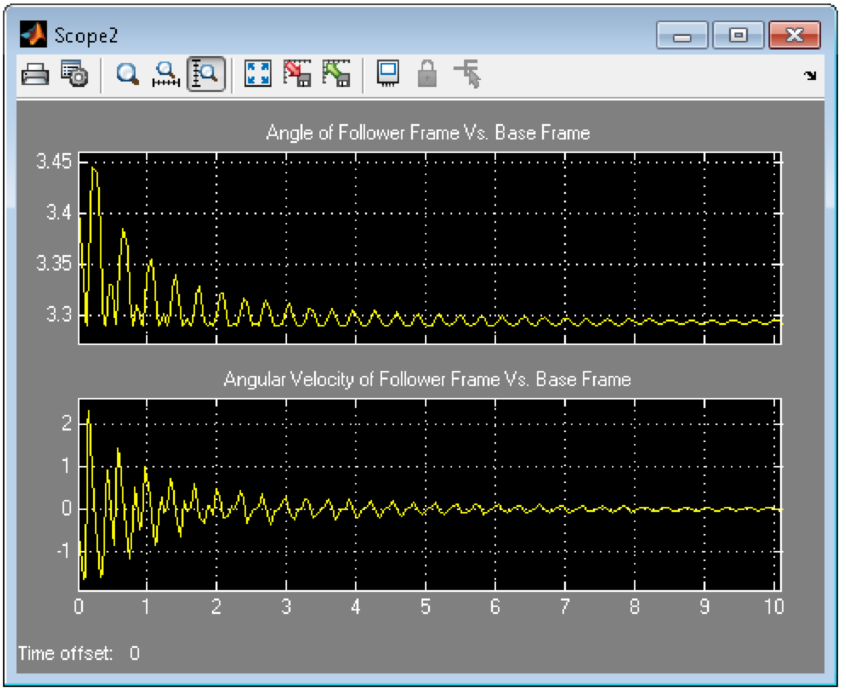
For each motion parameters you choose to sense, a physical signal outport appears on the selected joint block. You can plot each motion parameter in a Simulink Sinks block—e.g. Scope or XY Graph.

Note To feed a Simscape-based physical signal to a Simulink block, you must first convert it to a Simulink signal using a PS-Simulink Converter block. For more information, see the documentation for the PS-Simulink Converter block.

In the following figure, a Revolute Joint Block is configured to sense angular position and angular velocity. Two physical signal outports are present in this block, each connected to a PS-Simulink Converter block which makes them compatible with Simulink Sink blocks. The model shown corresponds to the four-bar mechanism as shown in "Modeling a Complete Four-Bar Mechanism" on page 2-24.



Configuring the Internal Mechanics section of the Joint dialog box to add damping, yields periodic angle and angular velocity signals that decay as a function of time. In the model illustrated above, you can double-click the scope block during simulation to obtain a plot of the time-varying angle and angular velocity signals.


Scope2
Angle of Follower Frame Vs. Base Frame
3.45
3.4
3.35
3.3
Angular Velocity of Follower Frame Vs. Base Frame
2
1
0
-1
0
1
2
3
4
5
6
7
8
9
10
Time offset: 0

Model and Machine Configuration

In this section...

Preparing to Simulate a Model

Before you can simulate and animate a mechanical model, you must verify the following:

- Mechanism Configuration Settings—for each machine in your model.
- Simulation Configuration Parameters—for the whole model.

Configuration Level	Access configuration parameters with...
Model-wide	Simulink Configuration Parameters
Machine-wide	Mechanism Configuration block dialog box

Note A machine is a complete, connected block diagram that represents one mechanical system. It is topologically isolated from any other machine in your model and has at least one World Frame and one Mechanism Configuration block. One model can have multiple machines.

Configuring the Model Environment

The Configuration Parameters dialog box is a standard feature of Simulink. Reset its entries for models in the SimMechanics environment to obtain more accurate simulation results.

1. In the Simulink menu bar, open the **Simulation** menu and click **Configuration Parameters** to open the Configuration Parameters dialog.

2. Select the **Solver** node of the dialog. Under **Solver options**, change **Relative tolerance** to `1e-6` and **Absolute tolerance** to `1e-4`. Change **Max step size** to `0.2`.

   If you want the simulation to continue running without stopping, change **Stop time** to `inf`. The pendulum period is approximately 1.6 seconds.

3. Close the Configuration Parameters dialog box.

Configuring the Machine Environment

A special feature of SimMechanics models is the Mechanism Configuration block.

1. Open Mechanism Configuration block dialog box.

2. Note the default **Gravity** vector, `[0 0 -9.81] m/s`^{2}, which points in the $-Z$ direction. The gravitational acceleration is $g = 9.81$ m/s^2.

3. Note the default **Linearization** value, `0.001`. This value specifies the linearization delta perturbation value to compute the numerical partial derivations for linearization. Change this value as necessary.

4. Close the Mechanism Configuration block dialog box.

Choosing a Simulink Solver

SimMechanics models support all Simulink solvers, depending on the model design.

If Your SimMechanics Model	See...
Is a simple mechanical system with few Simscape blocks	Existing Simulink guidelines when choosing a solver for your mechanical system model. For more information on Simulink solvers in general, see "Choosing a Solver Type" in the *Simulink User's Guide*.
Is a more complex mechanical system with many Simscape blocks	Existing Simscape guidelines for solvers. For more information on these solvers, see "Changing the Simulink Solver and Tolerances" and "Setting Up Solvers for Physical Models" in the *Simscape User's Guide*.

Model Visualization and Animation

When you simulate a model, the Simulink environment displays changes in the model in the model editor. The model editor does not visualize the data from the mechanical animation. To visualize the animation, use the Mechanics Explorer. By default, updating or simulating a SimMechanics model starts the Mechanics Explorer for that model.

The Mechanics Explorer has a number of controls that allow you to control the visualization of the animation. For more information on these controls, see “SimMechanics Visualization”.

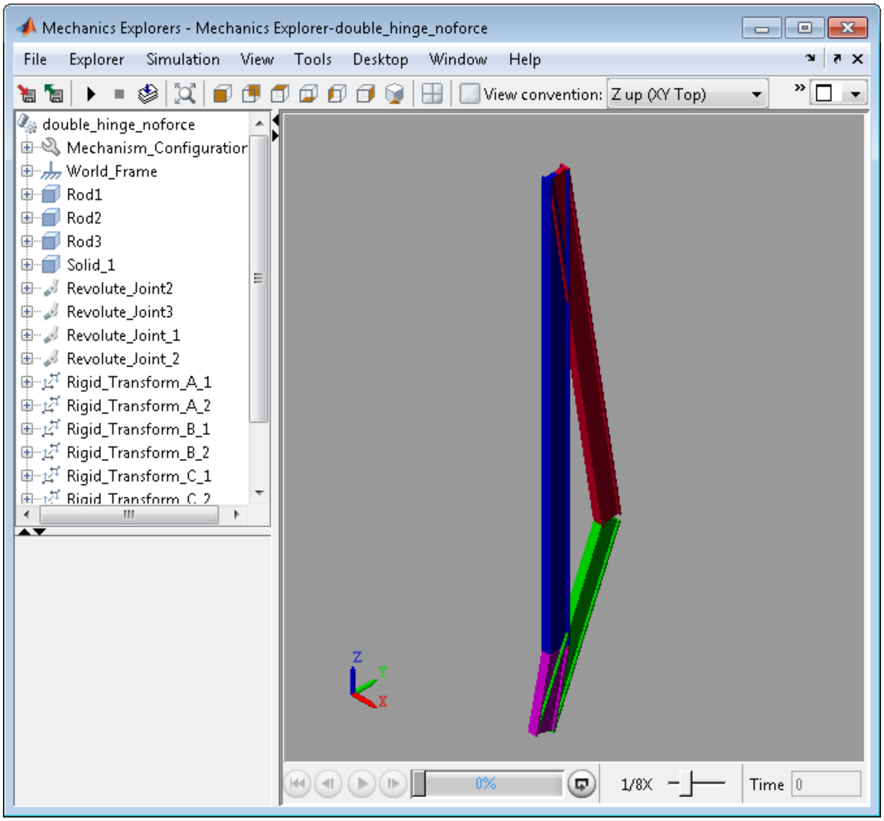
Note If you start to simulate your model, the machine automatically attempts assembly as an intermediate step before simulation itself begins.

In this section...

Assembling and Visualizing a SimMechanics Model in its Initial State

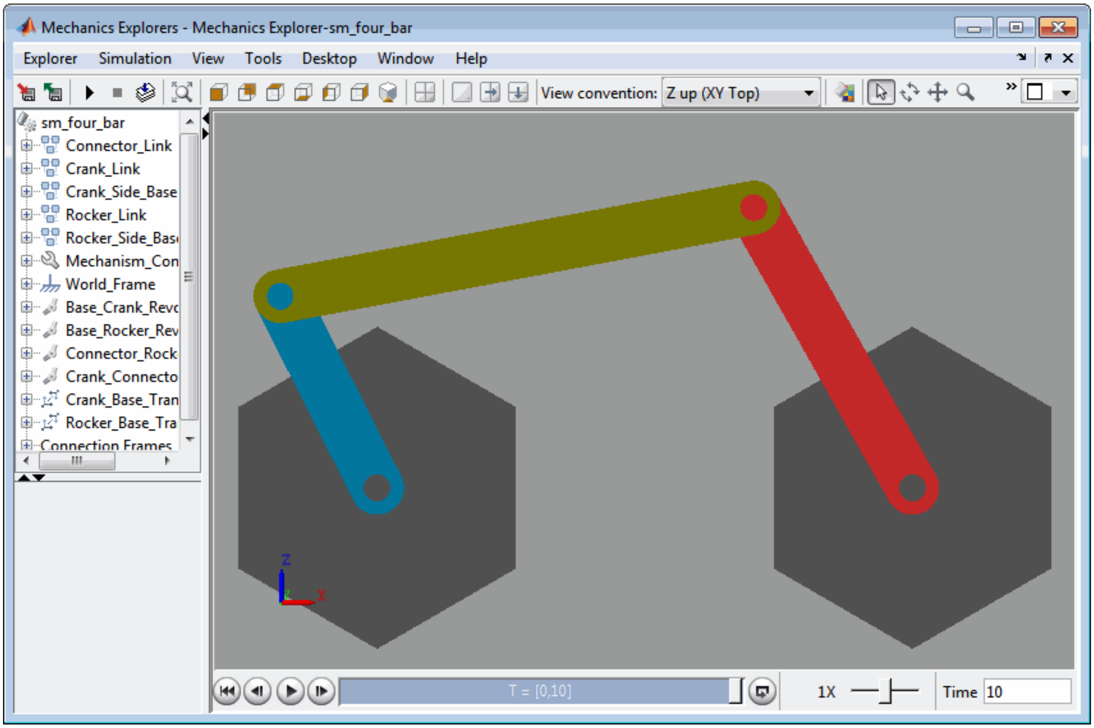
1. Open a Simulink Editor window with the mechanical model you wish to visualize.

2. On your keyboard, press Ctrl+D to assemble and visualize the mechanical model. A Mechanics Explorers window opens, with the mechanical model in its initial state. By default, the Mechanics Explorer is docked in the MATLAB Command Window.



3 Press Ctrl+T to simulate the mechanical model. Alternatively, press the Play button in either the Simulink Editor or the Mechanics Explorers window. The Mechanics Explorers window displays a kinematic animation of your model. For more information about using Mechanics Explorers, see “Configuring Mechanics Explorer for Automatic Start-Up”.

To open an example model of a four-bar mechanism that you can visualize and simulate, at the MATLAB command line, type `sm_four_bar`, or click here. The following figure shows the sm_four_bar model in its initial state.



Simulating and Animating a Mechanical Model

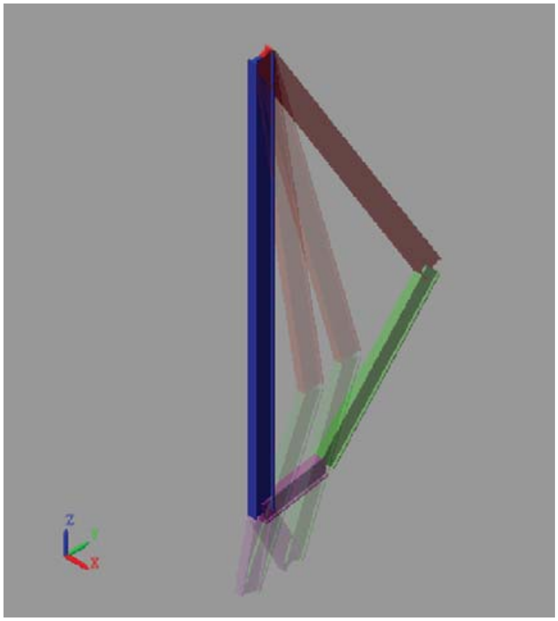
Simulating a SimMechanics Model

To simulate and animate a mechanical model;

1. Check that your model has been properly configured. For more information, see "Model and Machine Configuration" on page 2-45

2. Check that your model has been properly assembled in an initial state. For more information, see "Assembling and Visualizing a SimMechanics Model in its Initial State" on page 2-48

3. In the Mechanics Explorers toolbar, click the **Start** button ▶ to start the simulation. The Mechanics Explorers window plays a kinematic animation of the simulated model.



Note By default, the simulation runs for a time interval of 10 seconds. To change the simulation run time, see Configuration Parameters in the Simulink Reference documentation.

Playing Back a SimMechanics Animation

Each time you simulate a mechanical model, SimMechanics automatically caches the animation data for future replay. This feature allows you to play back an animation without the need to rerun the simulation. To control an animation, use the animation toolbar at the bottom of the Mechanics Explorer window.

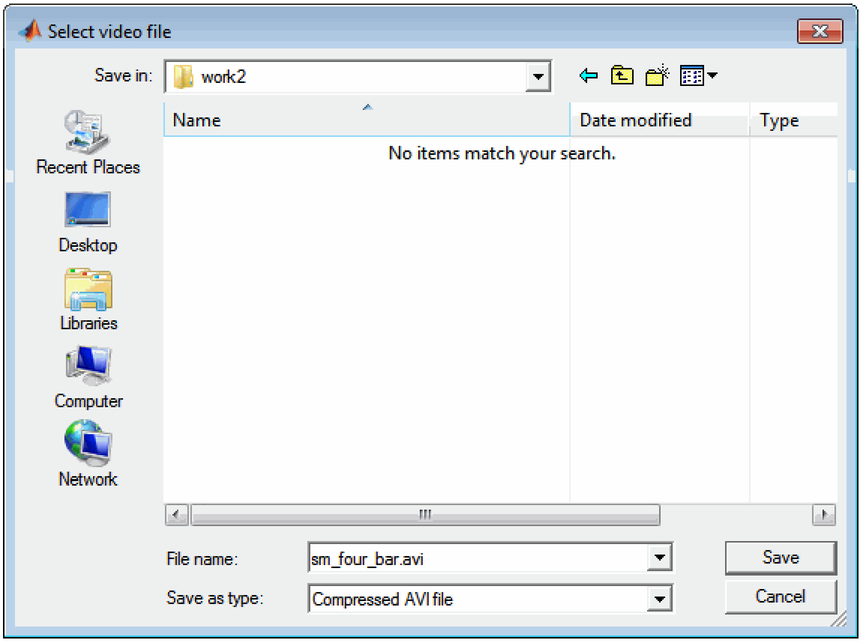
- Click the play button  to animate a model.
- Click the rewind button  to restart an animation from the beginning.

- Click the backward step button or the forward step button, to incrementally move the animation backward or forward in time.
- Click and drag the animation slider bar to the desired point in time, or enter the desired time in the `Time` field to move the animation to the desired point in time.
- Click the loop button to automatically restart the animation from the beginning once it reaches the end.
- Click and drag the animation speed slider bar to adjust the animation speed

Recording a SimMechanics Animation

You can record an animation as an .AVI file, and view it with any media player that supports .AVI files.

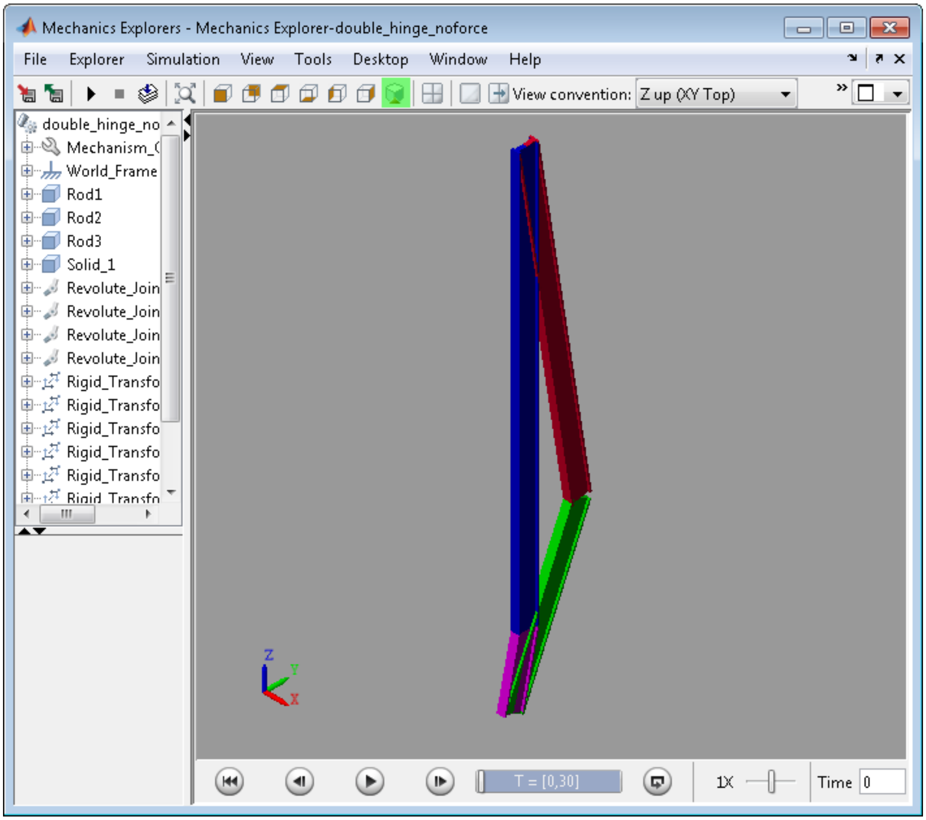
1 To save the animation for later playback, in the Mechanics Explorers toolbar, click the **Save animation** button. The Select video file dialog box is displayed.



2 Confirm the file directory and file name are correct, and click **Save** to save the animation as a .AVI file in the given directory.

The simulation and recording start simultaneously. When the recording is complete, a message is displayed.

Note The software records the animation with the view you select before you start the recording. For example, if you change the view to the isometric view of the model, then record the animation, the viewpoint of the animation is isometric.



3 Navigate to the directory where your animation file was saved, and open it to play the recorded animation.