

SimMechanics

For Use with Simulink®

- Modeling
- Simulation
- Implementation

How to Contact The MathWorks:



www.mathworks.com Web
comp.soft-sys.matlab Newsgroup



support@mathworks.com 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. Mail
3 Apple Hill Drive
Natick, MA 01760-2098

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

SimMechanics CAD Translator Guide

© COPYRIGHT 2003 – 2005 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.

MATLAB, Simulink, Stateflow, Handle Graphics, Real-Time Workshop, and xPC TargetBox are registered trademarks of The MathWorks, Inc.

Other product or brand names are trademarks or registered trademarks of their respective holders.

Revision History:

May 2003	Online only	Special for Version 2.1 (Release 13+)
September 2003	Online only	Revised for Version 2.1.1 (Release 13 SP1)
June 2004	Online only	Revised for Version 2.2 (Release 14)
October 2004	Online only	Revised for Version 2.2.1 (Release 14SP1)
March 2005	Online only	Minor update for Version 2.2.2 (Release 14SP2)

Physical Modeling Software Acknowledgment

The following applies to the graphviz code used within the Physical Modeling import utility import_physmod.

This product contains certain software code or other information (“AT&T Software”) proprietary to AT&T Corp. (“AT&T”). The AT&T Software is provided to you “AS IS”. YOU ASSUME TOTAL RESPONSIBILITY AND RISK FOR USE OF THE AT&T SOFTWARE. AT&T DOES NOT MAKE, AND EXPRESSLY DISCLAIMS, ANY EXPRESS OR IMPLIED WARRANTIES OF ANY KIND WHATSOEVER, INCLUDING, WITHOUT LIMITATION, THE IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE, WARRANTIES OF TITLE OR NON-INFRINGEMENT OF ANY INTELLECTUAL PROPERTY RIGHTS, ANY WARRANTIES ARISING BY USAGE OF TRADE, COURSE OF DEALING OR COURSE OF PERFORMANCE, OR ANY WARRANTY THAT THE AT&T SOFTWARE IS “ERROR FREE” OR WILL MEET YOUR REQUIREMENTS.

Unless you accept a license to use the AT&T Software, you shall not reverse compile, disassemble or otherwise reverse engineer this product to ascertain the source code for any AT&T Software.

© AT&T Corp. All rights reserved. AT&T is a registered trademark of AT&T Corp.

About This Guide

Using This Guide	iv
Ways to Get Help Online	v
Getting Online CAD Translator Help	v
Getting Online SimMechanics Help	v
For Further Help and Feedback	vi
Mechanical Units	vii
Mechanical Conventions and Abbreviations	viii
Right-Hand Rule Is Assumed	viii
Vector Multiplication	viii
Common Abbreviations	viii

Modeling with Computer-Aided Design

Introducing CAD Translation	2
Installing the SimMechanics CAD Translator	3
Uninstalling the SimMechanics CAD Translator	5
Exporting CAD Assemblies into Physical Modeling XML ...	6
Building a CAD Assembly for SimMechanics	6
Translating CAD Assemblies into XML	10
Getting Help in the CAD Translator Window	12
Troubleshooting Assembly Export Problems	13

Generating Models from Physical Modeling XML	14
Generating Body-Joint CAD-Based Models	14
Special Features of CAD-Based Models	15
Editing and Completing CAD-Based Models	16
Troubleshooting CAD-Based Models	18
Exporting a Part	20
Viewing the CAD Assembly	20
Exporting the CAD Assembly	21
Generating the SimMechanics Model	21
Designing and Exporting Mates	22
Block Diagram Structure of Two-Part Assemblies	23
Creating a Six DoF Joint	24
Creating a Prismatic Joint	25
Creating a Revolute Joint	29
Creating an Inplane Joint	30
Creating a Spherical-Spherical Massless Connector	31
Creating a CAD-Based Robot Arm Model	34
Viewing the Robot Arm Assembly	34
Exporting the Robot Arm Assembly	36
Generating and Completing the Robot Arm Model	36
Modeling a Stewart Platform	41
Viewing the Stewart Platform Assembly	42
Exporting the Stewart Platform Assembly	43
Generating the Stewart Platform Model	43
Visualizing the Stewart Platform Motion	46

Index

About This Guide

This guide is intended to help you use the SimMechanics computer-aided design (CAD) translator with your CAD platform. From your CAD assemblies, the translator generates XML representations ready for use with SimMechanics. But the translator is independent of SimMechanics, as well as MATLAB® and Simulink®.

Using This Guide (p. iv)

Overview of how to find help

Ways to Get Help Online (p. v)

Finding online help for the CAD translator and SimMechanics

Mechanical Units (p. vii)

Physical units used in CAD translation

Mechanical Conventions and Abbreviations (p. viii)

Mechanical assumptions and abbreviations implicit in this user's guide

Using This Guide

This guide helps you to work with the SimMechanics computer-aided design (CAD) translator from The MathWorks. This translator enables you to translate CAD assemblies from your CAD platform into a Physical Modeling XML file compatible with SimMechanics. You can then generate a SimMechanics block diagram model from this file to simulate the dynamics of the CAD assembly in the Simulink environment.

This guide covers both aspects of CAD translation, exporting the CAD assembly and generating the model.

- Exporting a CAD assembly into a Physical Modeling XML file requires only your CAD platform and the translator. It does not require MATLAB or any MATLAB component such as SimMechanics.
- Generating a SimMechanics block diagram model requires MATLAB, Simulink, SimMechanics, and the Physical Modeling XML file for your assembly. It does not require the CAD platform from which you exported the assembly.

The CAD translator is designed to work with your CAD platform. The translator and its associated help files are independent of MATLAB and any MATLAB component.

Note External references in this guide refer to sections of the SimMechanics User's Guide. For complete documentation of SimMechanics, including CAD translation and related features, consult the user's guide.

Ways to Get Help Online

Online help is available for both the CAD translator and for SimMechanics.

Getting Online CAD Translator Help

In addition to this guide, there are two sources of online help while you are using the CAD translator.

Obtaining Help During the CAD Translator Installation

When you install the CAD translator for your CAD platform, the installer displays a README file containing basic information about the translator, including how to uninstall it.

Obtaining Help from the CAD Translator Window

Once you start using the CAD translator in your CAD platform, you can obtain online help in two ways:

- Open the **SimMechanics** menu in your CAD platform's menu bar, then select **Help**.
- Click the ? symbol in the translator window.

After either step, a Web browser opens to display an HTML help page specific to your CAD platform.

Getting Online SimMechanics Help

There are a number of ways to get online help while you work with MATLAB, Simulink, and SimMechanics.

- *Help Browser* — There are several ways to open the Help browser:
 - Select **Full Product Family Help** from the MATLAB **Help** menu.
 - Select **Help** from the MATLAB **View** menu.
 - Enter `doc mech` at the command line.

Use the **Contents** pane on the left of the Help browser to find a section or chapter. Use the **Search** and **Index** features to find specific words.

- *Block Library Browser* — Click **Help** on the SimMechanics library menu bar to open online Help on Simulink, blocks, shortcuts, S-functions, and demos.

- *Context-sensitive help* — To access the help for a block, right-click the block or click **Help** on the block's dialog box.
- *Command line* — Enter `doc('block name')` at the command line to access the help for a block with the name `block name`. Spaces and capitalization in the block name are ignored.
If the same block name appears in other blocksets, MATLAB returns an `Overloaded methods` warning in the command window to flag those instances.
- *Complete SimMechanics block reference* — Expand the **SimMechanics** entry in the **Contents** pane of the Help browser, and select **Blocks – By Category** or **Blocks – Alphabetical List**.
- *Help desk (via the Web)* — Use a Web browser or the Help browser to connect to the MathWorks Web site at www.mathworks.com. Follow the **Documentation** link on the **Support** Web page for remote access to the documentation.

For Further Help and Feedback

The MathWorks hopes that you find SimMechanics and its CAD translator powerful and easy to use. Your suggestions and comments are welcome.

support@mathworks.com	Technical support
suggest@mathworks.com	Product enhancement suggestions
bugs@mathworks.com	Bug reports
doc@mathworks.com	Documentation error reports

For more contact and program information, visit the MathWorks Web site at www.mathworks.com.

Mechanical Units

SimMechanics and its CAD translator accept any mixture of MKS (SI), cgs, and English units.

Quantity	MKS (SI)	cgs	English
Length	meter (m)	centimeter (cm)	inch (in) foot (ft)
Mass	kilogram (kg)	gram (g)	slug (slug)
Angle	radian (rad) degree (deg)	radian (rad) degree (deg)	radian (rad) degree (deg)
Inertia	kilogram-meter ² (kg-m ²)	gram-centimeter ² (g-cm ²)	slug-foot ² (slug-ft ²)

Mechanical Conventions and Abbreviations

Right-Hand Rule Is Assumed

For rotational motion and vector cross products $\mathbf{a} \times \mathbf{b}$, the right-hand (RH) rule is always assumed.

Vector Multiplication

Scalar-vector products and matrix-vector multiplication are denoted by $\mathbf{a} * \mathbf{b}$ and $M * \mathbf{v}$, respectively.

Common Abbreviations

These are the abbreviations of mechanical terms most commonly used in this guide.

Abbreviation	Meaning
CG	Center of gravity
CS	Coordinate system
DoF	Degree of freedom
RF	Reference frame

Modeling with Computer-Aided Design

Using SimMechanics with computer-aided design (CAD) extends your mechanical modeling and simulation capabilities, allowing you to create block diagram models from CAD assemblies.

Introducing CAD Translation (p. 2)	Overview of installing and using the SimMechanics CAD translator to export assemblies and generate models from them
Exporting CAD Assemblies into Physical Modeling XML (p. 6)	Translating computer-aided design assemblies into the Physical Modeling XML format
Generating Models from Physical Modeling XML (p. 14)	Generating Body-Joint SimMechanics models from the Physical Modeling XML format and manually completing the models with other blocks
Exporting a Part (p. 20)	Modeling and translating a part into a body
Designing and Exporting Mates (p. 22)	Modeling and translating parts with degrees of freedom into bodies and joints
Creating a CAD-Based Robot Arm Model (p. 34)	Generating and completing a full SimMechanics model from an exported robot arm assembly
Modeling a Stewart Platform (p. 41)	Constructing and translating a six-degree-of-freedom Stewart platform assembly

Introducing CAD Translation

Computer-aided design (CAD) tools allow you to model machines geometrically as assemblies. Although this approach is excellent for geometric modeling, it is difficult to incorporate controllers and perform dynamic simulations. Simulink and SimMechanics use a block diagram approach to model control systems around mechanical devices and simulate their dynamics, but without full geometric information.

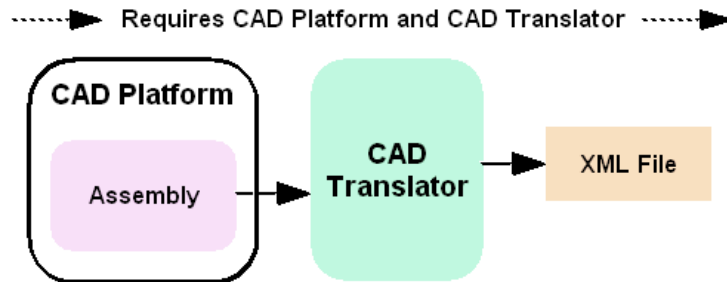
This section explains how to combine the power of CAD and SimMechanics with a CAD-to-SimMechanics translator. The translator transforms geometric CAD assemblies into Simulink block diagram models. The intermediary between a CAD assembly and its SimMechanics model is an XML file in a special Physical Modeling format.

Before starting to use CAD with SimMechanics, you must first

- 1 Install your CAD platform.
- 2 Install the SimMechanics CAD translator appropriate to your CAD platform, using the CAD translator installer. See “Installing the SimMechanics CAD Translator” on page 3.

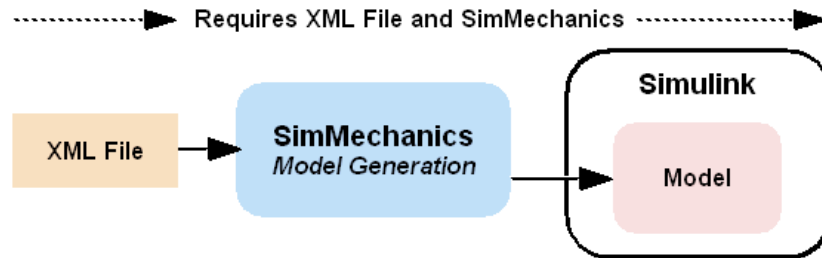
Using the translator with a CAD assembly involves two steps:

- 1 You export the assembly from the CAD platform into a Physical Modeling XML file that you can later use with SimMechanics. This step requires the CAD platform and the appropriate translator, but not MATLAB. See “Exporting CAD Assemblies into Physical Modeling XML” on page 6.



Exporting a CAD Assembly into a Physical Modeling XML File

- 2 You then convert the Physical Modeling XML file into a SimMechanics model in Simulink. This step requires the XML file and SimMechanics, but not the CAD platform or the translator. See “Generating Models from Physical Modeling XML” on page 14.



Importing a Physical Modeling XML File into SimMechanics

Installing the SimMechanics CAD Translator

You do not need any MATLAB component to install the SimMechanics CAD translator, but the target CAD platform needs to be installed.

If you have more than one installation of a particular CAD platform, you must install a translator for each CAD installation separately.

Installing the CAD Translator on Your System

To install the translator,

- 1 Obtain the self-extracting translator installer specific to your CAD platform and operating system by either downloading it or locating it in your SimMechanics installation under the directory

`toolbox/physmod/pmimport/standalone/`

CAD Platform	Installer	Uninstaller
SolidWorks® (Windows only)	Install_SimMechanics_Translator_for_SolidWorks.exe	Uninstall SimMechanics Translator for SolidWorks

- 2 Run the installer executable. A dialog appears. Select the **Setup** option.

A system command-line window opens and displays the installer's search for your CAD platform.

- 3 While the CAD platform search proceeds, you can either
 - Wait for the search to find an installed CAD platform executable
 - Press the spacebar to stop the search and then manually enter a full or partial path to your CAD platform executable
- 4 Once the installer locates your CAD installation directory, it prompts you to confirm that the translator is to be installed there. If you accept this choice, the installation proceeds in that directory.

If you refuse a location, the installer returns to ask you to enter another full or partial path and continues its search for a CAD installation in the new location.

- 5 After the translator files are copied into the proper directory, the installer prompts you to exit and open a README file. Once you close this file, the installation is complete.

The SimMechanics CAD translator files are installed, relative to the top directory of your CAD platform, in `/Applications/TheMathWorks/SimMechanics/`. The subdirectory `/help/` contains special help files for your particular CAD platform, while the `/examples/` subdirectory provides preconstructed assemblies to learn with.

Linking the CAD Translator to Your CAD Platform

To configure your CAD platform to work with the translator,

- 1 Start your CAD platform and open the add-ins or API manager menu. If you are unfamiliar with this menu, locate these settings in your CAD menu bar and documentation.
- 2 Check the box or entry corresponding to SimMechanics.

After you open a CAD assembly file, a SimMechanics menu appears in the CAD platform's menu bar. The SimMechanics CAD translator is now ready to work with your CAD platform.

Uninstalling the SimMechanics CAD Translator

You can remove the CAD translator link from your CAD platform without uninstalling it. You can also uninstall the translator completely. The uninstaller is located in the /Applications/ directory, relative to the top of your CAD platform's installation.

Unlinking the CAD Translator from Your CAD Platform

To unlink the CAD translator from your CAD platform,

- 1 Locate your CAD platform's add-ins or API manager menu.
- 2 Clear the entry or box corresponding to SimMechanics in this menu.
- 3 Close your CAD program.

When you reopen the CAD program, the CAD translator link no longer appears, but the CAD translator remains installed on your system.

Uninstalling the CAD Translator from Your System

To uninstall, you must first unlink the CAD translator from your CAD platform.

Then, in the /Applications/ directory, relative to the top of your CAD platform's installation, locate and double-click the uninstaller icon for the SimMechanics translator. Allow the uninstall to proceed until completion.

If you have a particular CAD platform installed multiple times on your system, this uninstall procedure removes the SimMechanics translators from all installations of this particular CAD platform.

The uninstallation removes all translator-associated files except for the uninstaller itself, which is deleted the next time you reboot your system.

Exporting CAD Assemblies into Physical Modeling XML

The SimMechanics computer-aided design (CAD) translator extends the functionality of your CAD software platform. The translator converts an existing CAD assembly into a Physical Modeling XML file that is portable and independent of SimMechanics. From this XML file, you can generate a SimMechanics model, as discussed in “Generating Models from Physical Modeling XML” on page 14.

Note You do not need MATLAB or any MATLAB component to use the SimMechanics CAD translator, but you need to have the target CAD platform installed.

When you export from your CAD platform, you must export a complete assembly into XML, not just a part.

This section explains how to export CAD assemblies into the Physical Modeling XML format.

- “Building a CAD Assembly for SimMechanics” explains the requirements to generate a working SimMechanics model from your CAD assembly.
- “Translating CAD Assemblies into XML” on page 10 walks you through the steps to export the XML file representing the CAD assembly.
- “Getting Help in the CAD Translator Window” on page 12 shows you how to get online help while you are working with the CAD translator.
- “Troubleshooting Assembly Export Problems” on page 13 examines some of the problems you might encounter when you export a CAD assembly into a Physical Modeling XML representation.

Building a CAD Assembly for SimMechanics

SimMechanics is a dynamic environment for simulating body motions. The CAD translator creates a Physical Modeling XML file that represents the assembly’s parts as bodies and maps the assembly’s mates into a corresponding

set of joints. You need to specify enough information in your CAD assembly for SimMechanics to construct a dynamically meaningful model from the XML file.

CAD Assembly Component	Corresponding SimMechanics Blocks
Part	Body
Mates	Joints
Fundamental Root	Ground – Root Weld – Root Body
Subassembly	Subsystem
Subassembly Root: [[<i>trunk</i>]] – mate(s) – <i>subassembly</i>	[[Root Body – Root Weld – Fixed Body]] – Joint – <i>subsystem</i>
Fixed Part (in a subassembly)	Root Body – Weld – Body

Roots, Subassemblies, and Hierarchies

Every CAD assembly has a single *fundamental root*, a fixed point that does not move. The positions and orientations of all parts refer directly or indirectly to this fundamental root. The CAD translator converts the fundamental root into a unique Ground — Weld — Body combination in SimMechanics.

A *root body* is a zero-mass, zero-inertia body used in the generated SimMechanics model to represent a CAD root. A root body is always welded to ground, so that its zero mass and zero inertia do not affect the model's dynamics.

You can isolate a collection of CAD components (parts and mates) into a *subassembly*. The CAD translator converts subassemblies into subsystems in SimMechanics.

In general, you can make a subassembly either *rigid* or *flexible*.

- A *rigid* subassembly has distinct parts constrained to move as a single body.
- A *flexible* subassembly has distinct parts that can move relative to one another according to their mates.

See also “Configuring Subassemblies” on page 10.

The main assembly is like the trunk of a tree, and its subassemblies are like the branches of the tree. There can be subassemblies of subassemblies, and so

on. This tree is the assembly's *hierarchy*. Each CAD subassembly has its own *subassembly root*, which attaches it to the rest of the assembly through one or more mates.

A *fixed part* of a CAD subassembly is a part that is welded to the subassembly root. It cannot move relative to the subassembly root.

See “Creating a CAD-Based Robot Arm Model” on page 34 for an example.

Optimizing Your Assembly with Subassemblies

Use subassemblies to organize your assembly hierarchically. This will simplify your subsequent SimMechanics model by grouping blocks into corresponding subsystems. Follow these guidelines to ensure that your CAD assembly is translated into a functioning SimMechanics model.

- You must have at least one fixed part inside each subassembly. (See “Roots, Subassemblies, and Hierarchies” on page 7 for more on fixed parts.)
- Put as many welded components as you can inside rigid subassemblies.
If your assembly has a group of parts that do not move relative to one another, put them inside a subassembly and make it rigid. The SimMechanics translator treats the entire subassembly as a single part, thereby eliminating unnecessary body and joint blocks from your subsequent SimMechanics model.
- Avoid creating mates between a subassembly part and geometrical abstractions such as the x - y , y - z , and x - z planes of the subassembly. Instead, create mates between the part and a fixed part in the subassembly.

Specifying Mass Properties of Assembly Parts

The CAD assembly's parts need to have masses and inertia tensors. When you generate the SimMechanics model, this mass property information is used to specify the properties of the SimMechanics Body block corresponding to each assembly part.

Usually, your CAD platform computes masses and inertia values from the mass density and geometry of the assembly parts. Otherwise, you must specify the mass and inertia tensor with respect to the part's center of gravity. The SimMechanics CAD translator computes the center of gravity of each part automatically.

See “Exporting a Part” on page 20 for an example.

Specifying Mate Geometries

Your CAD assembly's mates restrict how parts can move with respect to each other. Without any connecting mates, a pair of CAD parts can move with six unrestricted *degrees of freedom* (DoFs) relative to one another. Mates between pairs of parts reduce the six to fewer DoFs. In SimMechanics, joints express DoFs between bodies, because bodies by themselves carry no DoFs. You must specify the mate geometry in the CAD assembly consistently and in enough detail for SimMechanics to reconstruct the assembly's DoFs as joints.

The relationship between mates and joints is not, in general, a simple mapping. Some SimMechanics joints have only one DoF, while others represent more than one DoF. The translator often combines multiple DoFs into one joint.

Each joint is connected to each of two bodies at a body coordinate system (CS). The mate geometry determines the joints into which the translator transforms the mates and controls the position and orientation of the body CSs. Each of these body CSs has an origin and axis triad fixed relative to its body. The translator creates body CSs on the bodies as necessary for connecting joints.

See “Creating a CAD-Based Robot Arm Model” on page 34 for an example of configuring mates.

Avoiding Redundant Mates

Keep mates simple and few enough to avoid creating unnecessary joints in your SimMechanics model.

For example, consider three parts, P1, P2, and P3, in an assembly. Suppose P1 and P2 are mated so that there is no movement possible between them. When you attach P3, you could put, for example, one mate between P3 and P1 and the other between P3 and P2. But this leads to a redundant joint in the SimMechanics model, making it harder to understand and debug than if you created only one mate. In this example, it is better to create a mate just between P3 and P2, since P2 cannot move with respect to P1 anyway.

Specifying Initial Conditions of Motion

The initial condition of motion for the whole assembly is just its geometrical configuration. All initial velocities are assumed to be zero. SimMechanics begins dynamic simulation of the corresponding model in this state, unless you change the model after generating it. See “Setting the Model's Initial Conditions” on page 17 for more details.

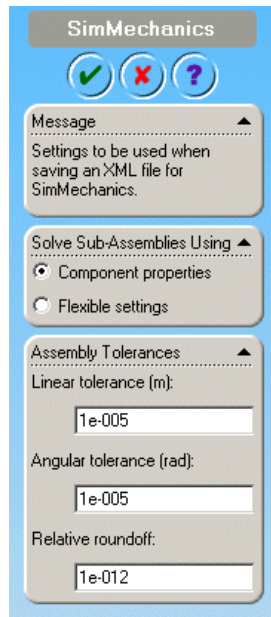
Translating CAD Assemblies into XML

To translate the CAD assembly into a form that SimMechanics can use, you must check and configure the translator settings, then save the assembly through the translator into the special Physical Modeling XML format.

Opening the SimMechanics Translator Settings

In your CAD menu bar, select **SimMechanics**, then **Settings**. The **Settings** area appears in your CAD window.

To apply your settings, click the green check mark at the top. To cancel your settings session at any time, click the red **X** at the top.



SimMechanics Translator Settings Area

Configuring Subassemblies

Here you choose how the CAD translator treats subassemblies. You can simplify the translated SimMechanics model if you treat subassemblies as rigid, because each is translated as a single body. To simulate the motion of individual subassembly parts, treat translated subassemblies as flexible.

- Choose **Component properties** if you want the translator to treat each subassembly according to its individual rigid or flexible setting in the CAD platform.
- Choose **Flexible settings** if you want the translator to treat all subassemblies, regardless of their individual settings, as flexible. Selecting this option does not change the CAD subassembly settings in the CAD assembly file.

The default is **Component properties**.

Configuring Tolerances

You can configure one or more of the translation tolerances. Geometrical and numerical differences smaller than the tolerances are treated as zero. The entries implicitly have the same units selected in your CAD platform settings.

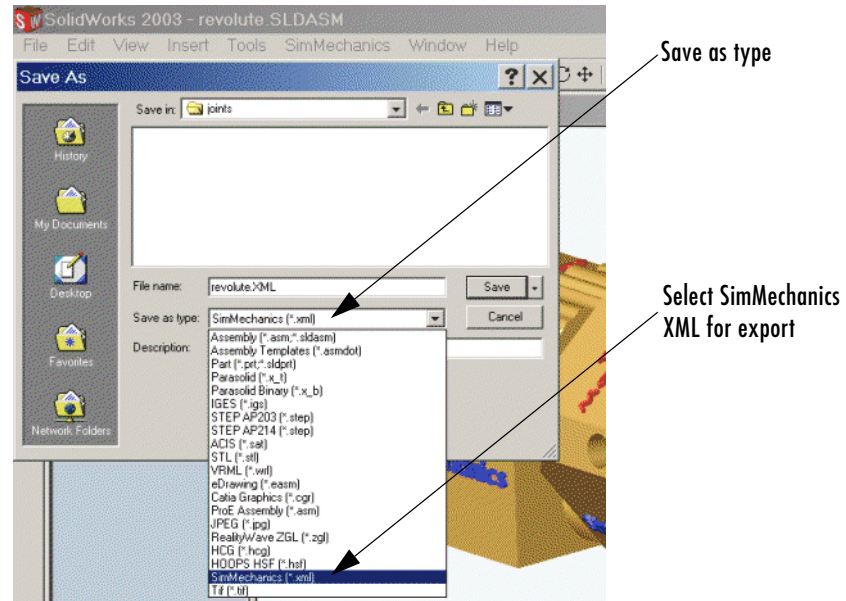
- **Linear tolerance** specifies the smallest significant difference in length.
- **Angular tolerance** specifies the smallest significant difference in angle.
- **Relative roundoff** specifies the smallest significant numerical difference.

Applying the Settings and Creating the XML File

To translate the assembly into a Physical Modeling XML file,

- 1 After entering your SimMechanics settings, click the green check mark at the top to confirm them. The CAD hierarchy window returns.
- 2 If you changed the assembly or any subassemblies, you need to rebuild the assembly and resave it in its native format before exporting it to XML.
- 3 With your assembly open, go to **File**, then **Save As**. The **Save As** dialog opens. For **Save as type**, select **SimMechanics (.xml)**. The default name is

the same as that of the CAD assembly file. You can change the XML filename and directory at this point.



4 Click OK.

The assembly is saved in the new form as an XML file.

Note To use the exported XML file to automatically generate a SimMechanics model, you need to move or copy the file into a MATLAB working directory. See “Generating Models from Physical Modeling XML” on page 14.

Getting Help in the CAD Translator Window

The CAD translator installation for your platform includes special online help that is independent of the MATLAB Help system. These help files are located, relative to the top of your CAD platform directory, in

Applications/TheMathWorks/SimMechanics/help/

Getting HTML Help

To obtain online CAD translator help in your CAD platform, follow either step:

- In the CAD menu bar, select **SimMechanics**, then **Help**.
- In the **Settings** window, click the ? (question mark) symbol.

Your default Web browser opens and displays an HTML help file specific to your CAD platform.

Getting PDF Help

You can also access a short CAD translator guide in PDF format by looking in your CAD platform's special SimMechanics directory.

Troubleshooting Assembly Export Problems

The CAD translator can encounter difficulties when it attempts to represent your assembly in the XML file as SimMechanics bodies and joints.

Mate Translation Errors

Mate translation errors occur when you specify a mate in your CAD assembly that is not supported for export. The mates supported for a particular CAD platform are listed in that platform's special HTML help page (see "Getting HTML Help" preceding).

If the translator fails to map one or more mates into joints, it issues one or more error dialogs and logs the errors into a special text file. The error dialog indicates the name of this error log file, which is located in the same directory as the exported Physical Modeling XML file. The XML file is generated regardless of mate translation errors.

The XML file itself contains the same errors, each paired with the corresponding failed joint. The mates that failed to translate properly are converted instead into welds. These errors reappear as MATLAB command-line warnings if you generate a SimMechanics model from the XML file.

Subassembly Configuration Errors

Be sure that you configure your flexible subassemblies' positions and orientations to be consistent with the main assembly's configuration *before* your export the assembly.

Generating Models from Physical Modeling XML

A CAD assembly can specify part and mate information sufficient to generate a SimMechanics model consisting of Body and Joint blocks and representing the assembly. Once you have exported a CAD assembly into a Physical Modeling XML file, you can generate a SimMechanics block diagram model with this file. Although the generated model will run, you often need to manually simplify and complete it in order to fully represent the original dynamic system.

- “Generating Body-Joint CAD-Based Models” shows how to import the XML file into SimMechanics using the command `import_physmod`, which constructs the model.
- “Special Features of CAD-Based Models” on page 15 explores the special characteristics of an automatically generated CAD-based model.
- “Editing and Completing CAD-Based Models” on page 16 explains when and how to edit your model and expand it with additional blocks, such as Constraints, Drivers, Actuators, and Sensors.
- “Troubleshooting CAD-Based Models” on page 18 points out major model-generation and simulation errors and problems that can arise from CAD-based models, and techniques for solving them.

Note This section assumes that you have SimMechanics installed locally or remotely, and that you have the XML file representing a CAD assembly in your current MATLAB directory.

To generate the SimMechanics model, you do not need the CAD platform from which the XML file was exported or the SimMechanics CAD translator.

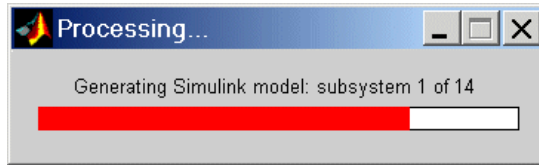
Generating Body-Joint CAD-Based Models

Move or copy the Physical Modeling XML file you want to use into your MATLAB working directory.

The `import_physmod` command creates a SimMechanics block diagram model from a Physical Modeling XML file. To start the model generation, enter

```
import_physmod('cad_assembly.xml')
```

at the command line. A progress bar appears:



The bar is updated as the model is imported.

A Simulink model window opens. The model is populated by Bodies and Joints corresponding to the assembly parts and mates saved in `cad_assembly.xml`, with the corresponding connections and hierarchy explained in “Exporting a Part” on page 20. The name of the generated model is the name of the original assembly file, regardless of the name chosen for the XML file.

If your model needs additional Joints, as well as Constraints, Drivers, Actuators, and Sensors, you must manually add these to the model from the SimMechanics block libraries and connect them. You also might want to manually simplify a model. Consult “Editing and Completing CAD-Based Models” on page 16 for more.

Changing the Appearance of a CAD-Based Model

You can change the appearance of your generated model by using optional import settings. See the `import_physmod` command reference for more details.

For example, entering

```
import_physmod('cad_assembly.xml', 'FontSize', 18)
```

at the command line generates a model with a block label font size of 18 pixels.

Special Features of CAD-Based Models

Most of the properties of models that you generate from an XML file are the same as the default properties of all Simulink models. See the Simulink user’s guide for general information on working with Simulink models.

Models generated from a CAD-based XML file do have certain special features:

- Exactly one Ground block and connected Machine Environment block
- Fundamental root, represented by Ground — Root Weld — Root Body
- Subassembly roots, represented by Root Body — Root Weld — Fixed Body

- Joints with degrees of freedom (DoFs), represented by Custom Joints, containing the correct joint primitives for the translational and rotational DoFs between any two bodies
- Joints without DoFs, represented by Welds

Joints that directly connect a subsystem's Body to the next higher level in the hierarchy appear in that next higher level, not within the subsystem. This means that the boundary between subassembly and main assembly in the CAD world is drawn differently from the boundary between subsystem and top-level system in SimMechanics. This difference does not change the degrees of freedom of the model. They are the same in a CAD assembly and in the SimMechanics model generated from the assembly.

Editing and Completing CAD-Based Models

The `import_physmod` command generates a model containing only Machine Environment, a Ground, Bodies, and Joints. A complete SimMechanics model typically contains other blocks, including Constraints, Drivers, Sensors, and Actuators (from the `mechlib` block library), as well as blocks from Simulink, such as Scopes.

Creating a complete SimMechanics model requires inserting and connecting these additional blocks in your generated model. Use the relevant sections of the “Modeling Mechanical Systems” chapter in the SimMechanics documentation to learn how to use these blocks.

When you first generate a model from Physical Modeling XML, you might want to save this original CAD-based model before you create later versions by eliminating unnecessary blocks and adding new ones.

Deleting Unnecessary Blocks

The CAD mate-to-SimMechanics joint mapping creates unnecessary blocks in order to isolate all the degrees of freedom (DoFs) correctly. You can delete these unnecessary blocks to simplify your model without affecting its dynamics, as long as you reconnect the remaining blocks properly.

The unnecessary blocks consist of nonmoving zero-mass/zero-inertia Bodies and redundant Welds.

In particular, the fundamental root of a CAD assembly translates to a RootGround — RootWeld — RootPart combination, which is then connected to

other Joint and Body blocks. The root body has zero mass and inertia, so you can remove it and the RootWeld, then reconnect the rest of the model to the Ground.

Not all Welds are redundant. Some are needed to connect a Body that would otherwise be isolated to the rest of the machine.

Constraining and Driving Degrees of Freedom

Constraints reduce the number of independent degrees of freedom in a machine by preventing certain movements. Drivers affect DoFs, not by eliminating them completely, but by forcing their motions to follow an external time-dependent signal. A constrained or driven DoF is not free to respond to forces and torques independently.

These sections in the SimMechanics documentation discuss how to add constraints and drivers to your model:

- “Modeling Constraints and Drivers” on page 4-35
- “Actuating a Driver” on page 4-53

Actuating Bodies and Joints with Motions and Forces

SimMechanics gives you the ability to actuate bodies and joints in your models with external forces and motions, and to set up internal forces between bodies. You can also vary the mass or inertia (or both) of a body in time. Consult these sections in the SimMechanics documentation for more details.

- “Actuating a Body” on page 4-42
- “Varying a Body’s Mass and Inertia Tensor” on page 4-45
- “Actuating a Joint” on page 4-47
- “Modeling Force Elements” on page 4-58

Setting the Model’s Initial Conditions

When you import a CAD assembly into SimMechanics, the XML file determines the initial geometric configuration of the model’s bodies. By default, the initial velocities of the bodies are zero.

If you want to change the initial positions and/or velocities of the bodies in your model to be different from the CAD-determined configuration, you need to add initial condition actuators, as discussed in “Specifying Initial Positions and Velocities” on page 4-53 in the SimMechanics documentation.

Sensing Forces and Motions

To detect the motions of bodies and joints and measure the forces acting on or through them, you can add sensors to your model. See “Modeling Sensors” on page 4-63 in the SimMechanics documentation.

Checking Model Validity

You can find general instructions and requirements for building valid SimMechanics models in “Modeling Machines” on page 4-2 and “Checking Model Validity” on page 4-68 in the SimMechanics documentation.

Troubleshooting CAD-Based Models

These are some problems that might arise in your generated SimMechanics model if your CAD assembly was not constructed or exported properly.

General Guidelines

- See “Building a CAD Assembly for SimMechanics” on page 6 for instructions on constructing a CAD assembly specifically for SimMechanics.
- See “Translating CAD Assemblies into XML” on page 10 for details about exporting a CAD assembly.

Troubleshooting Errors During Model Generation

Errors in the Physical Modeling XML file appear as warnings at the MATLAB command line during model generation.

These warnings arise from CAD mate translation errors encountered when the XML file was originally exported. Such errors occur when the CAD translator fails to map one or more CAD mates, which restrict the degrees of freedom between parts, into their corresponding SimMechanics joints. The CAD translator warns you at the export step if such errors occur.

The failed Joint appears in your generated model as a Weld. You can fix such errors in two ways:

- In the generated model, manually replace the Weld with the proper Joints.
- Return to the original CAD assembly, reconfigure the mates, and export it again.

Consult “Mate Translation Errors” on page 13 for more about export errors.

Troubleshooting Model Dynamics Errors

Certain problems with CAD-based models appear only when you run the model.

- You must “fix” at least one part in every CAD subassembly by mating it to the subassembly root. Otherwise, the massless root body is dynamically active and experiences infinite acceleration when forces or torques are applied to it. See “Roots, Subassemblies, and Hierarchies” on page 7 and “Optimizing Your Assembly with Subassemblies” on page 8 for more about fixing subassemblies.
- If you find constraints are violated while SimMechanics is running your model, try the following:
 - Examine your original CAD assembly for redundant mates.
 - Check and possibly increase the assembly’s assembly tolerances at the original CAD export step.
 - Check and possibly increase the translated model’s assembly tolerances in the machine’s Machine Environment block, in the **Parameters** pane.
 - On the **Constraints** panel in the Machine Environment dialog, select the **Use robust singularity handling** check box.
- Never *decrease* assembly tolerances in a CAD-based SimMechanics model. Instead, decrease the assembly tolerances at the original CAD export step.

Troubleshooting SimMechanics and Simulink Problems

You might also encounter general Simulink- or SimMechanics-related problems while running your model.

- For problems specific to SimMechanics, see “Troubleshooting Simulation Errors” on page 5-16 in the SimMechanics documentation.
- For general Simulink problems, consult the Using Simulink documentation.

Exporting a Part

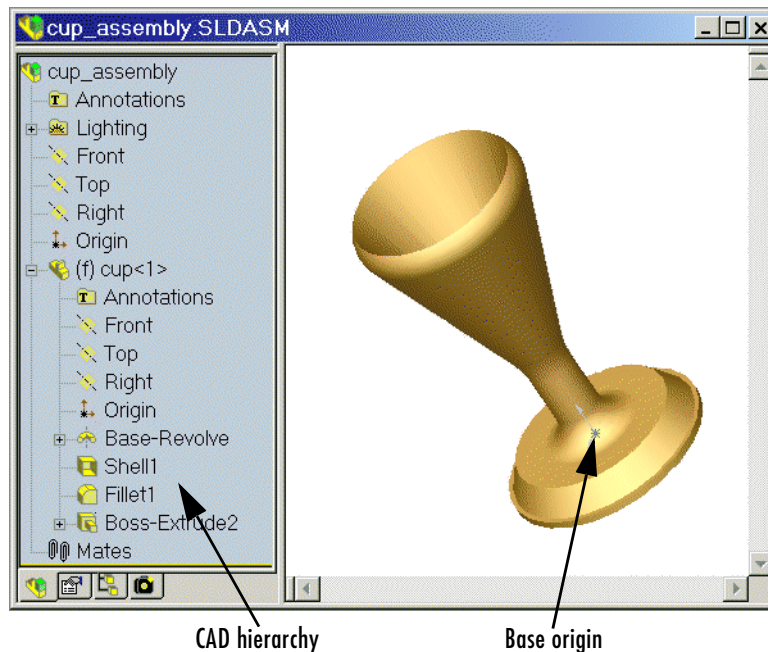
In this example, you export an assembly with one part, a cup, and no mates. After you install your CAD translator, the two CAD files for the cup are located in the `Applications/TheMathWorks/SimMechanics/examples/cup/` directory, relative to the top of your CAD platform installation. The two files are

- The full assembly file, `cup_assembly.ASSEMBLYFILETYPE`
- The cup itself, a part called `cup.PARTFILETYPE`

Although it has only one part, you must export the full assembly into XML, not just the cup part.

Viewing the CAD Assembly

Open the cup assembly file and check its geometry and mass properties.



Cup Assembly in a CAD Platform

Property	Value
Volume	0.0001 cubic meters (m ³)
Surface area	0.0381 square meters (m ²)
Density	3.0 grams/cm ³ = 3000 kg/m ³
Mass	0.2906 kilograms (kg)
Principal moments of inertia at the center of gravity	$I_x = 0.00015$, $I_y = 0.00067$, $I_z = 0.00067 \text{ kg}\cdot\text{m}^2$

The inertia tensor is computed with the origin at the center of gravity and the coordinate axes aligned with base-origin axes, indicated in the figure “Cup Assembly in a CAD Platform” on page 20. The x -axis is the cup’s axis of symmetry, and the y - and z -axes point across the cup.

Exporting the CAD Assembly

Now export the assembly into Physical Modeling XML format. Under **File**, choose **Save As**, and for **Save as type**, select SimMechanics (*.xml). Click **OK**. The XML file cup_assembly.xml appears in your working CAD directory.

Generating the SimMechanics Model

You need to move or copy the exported XML file into a MATLAB working directory to generate a SimMechanics model from the file. See “Generating Models from Physical Modeling XML” on page 14 to learn how to generate models from XML files.

If you generate such a SimMechanics model, the model will have six blocks, a Machine Environment — Ground — Weld — Root Body — Weld — Cup combination inside a subsystem, called cup_assembly, representing the entire assembly. The Root Body is the special zero-mass/zero-inertia body inserted between ground and the cup. The second joint is a Weld, because the CAD assembly has no degrees of freedom. Deleting the Root Body and one of the Welds does not physically change the model, as long as you reconnect the remaining blocks.

Designing and Exporting Mates

In “Exporting a Part” on page 20, you create and export an assembly composed of a single part. The SimMechanics computer-aided design (CAD) translator converts the part into a body, with a mass, an inertia tensor, and body coordinate systems (CSs). Because there are no other parts in that CAD assembly, the SimMechanics body is welded to ground and has no degrees of freedom (DoFs).

Restricting Degrees of Freedom with Mates

CAD platforms normally assume that two parts with no constraints (mates) between them have the complete six relative DoFs possessed by any rigid body relative to another body. You restrict the DoFs between parts by connecting them with mates in the CAD assembly. Mates restrict relative body motion and reduce the number of relative DoFs between body pairs. There is always one assembly part welded to ground.

This section presents a set of complete CAD assemblies, with both parts and mates. Each example assembly consists of two instances of the same part file, representing two identical cubes. In different assemblies, the two cubes are constrained with different mate combinations in order to create different relative DoFs between the cubes. You can typically represent a set of DoFs with a large number of different mate combinations. Each mate combination here, in general, is *not* the unique way to create the corresponding set of DoFs.

- “Creating a Six DoF Joint” on page 24 assembles the two cubes with no mates, so that the cubes have the full six degrees of freedom relative to one another.
- “Creating a Prismatic Joint” on page 25 shows how to constrain the two cubes in two different ways so as to produce the same result, a single prismatic (translational) DoF between them.
- “Creating a Revolute Joint” on page 29 constrains the two cubes so as to allow only a single revolute (rotational) DoF between them.
- “Creating an Inplane Joint” on page 30 constrains the two cubes so as to allow two prismatic (translational) DoFs between them.
- “Creating a Spherical-Spherical Massless Connector” on page 31 constrains the cubes so as to allow relative spherical joint motion, with the two cubes separated by a nonzero distance.

Locating the Example Assembly Files

The CAD assembly files used as examples for this section are located, relative to the top of your CAD platform installation, in the directory

```
Applications/TheMathWorks/SimMechanics/examples/joints/
```

The cube part is contained in `magic_cube.PARTFILETYPE`.

Block Diagram Structure of Two-Part Assemblies

All the models that you generate in this section from the example CAD assemblies have a common structure, because each assembly has a fundamental root and two moving parts. Each model has eight blocks.

- *The assembly's fundamental root.* As in any generated CAD-based model, the four-block combination Machine Environment — RootGround — RootWeld — RootPart represents the assembly's fundamental root. The RootPart is a nonmoving, zero-mass/zero-inertia body. See “Building a CAD Assembly for SimMechanics” on page 6 and “Special Features of CAD-Based Models” on page 15.
- *The moving bodies.* The bodies representing the assembly's parts are `magic_cube-1` and `magic_cube-2`.
- *The joints.* In all the models, the first cube is connected by a Weld to RootPart and cannot move. The second cube is connected to RootPart by a Custom Joint (except in the last example, “Creating a Spherical-Spherical Massless Connector”).

Depending on the degrees of freedom (DoFs) in question in a particular assembly, the CAD translator configures the Custom Joint to represent different DoFs with combinations of prismatic, revolute, and spherical primitives. The second cube can move with respect to the first with the DoFs represented by the Custom Joint.

Some of the blocks in the generated models are redundant. You can manually edit and simplify the models without changing their physical properties. See “Editing and Completing CAD-Based Models” on page 16.

Creating a Six DoF Joint

The simplest assembly with two parts has no mates between the parts. The parts can move with respect to one another with all six degrees of freedom (DoFs).

Viewing and Exporting the Assembly

To see an example of such an assembly and translate it into a SimMechanics model,

- 1 Open the assembly `sixDOF.ASSEMBLYFILETYPE`. The two parts are `magic_cube-1` and `magic_cube-2`.

Note, in the CAD hierarchy, that the Mates node has no entries. Therefore, relative to one another, the cubes are unconstrained in their motion and have six relative DoFs.

- 2 Now translate this CAD assembly into Physical Modeling XML format. From the **File** menu, select **Save As**. In the **Save As** dialog, keep the **File** name as `sixDOF`, but change **Save as type** to `SimMechanics (*.xml)`. Then click **OK**.

The XML file is saved in your current working CAD directory.

Generating the Model

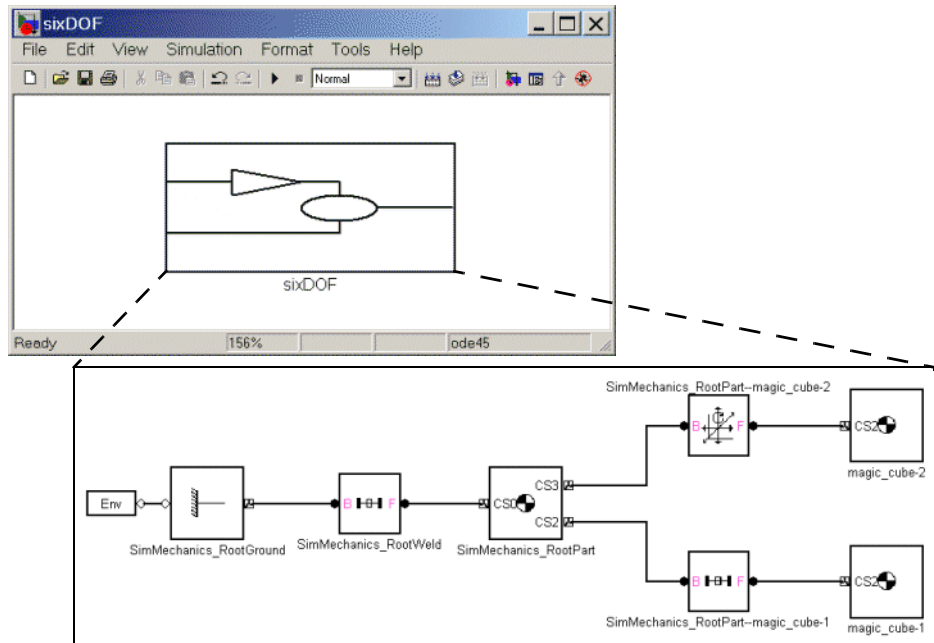
Now you can generate a model based on this assembly.

- 1 Move or copy the XML file to a working MATLAB directory. Then open MATLAB in that directory.
- 2 At the MATLAB command line, enter `import_physmod('sixDOF')`. SimMechanics automatically generates a model, `sixDOF.mdl`, based on `sixDOF.xml`.

The entire assembly is translated into a subsystem, also called `sixDOF`, within the model.

- 3 Open the subsystem. There are eight blocks, arranged in the common structure described in “Block Diagram Structure of Two-Part Assemblies” on page 23.

The Custom Joint represents the six DoFs between the two cubes with one spherical and three prismatic primitives. You can replace this Custom Joint with a Six-DoF Joint block.



Creating a Prismatic Joint

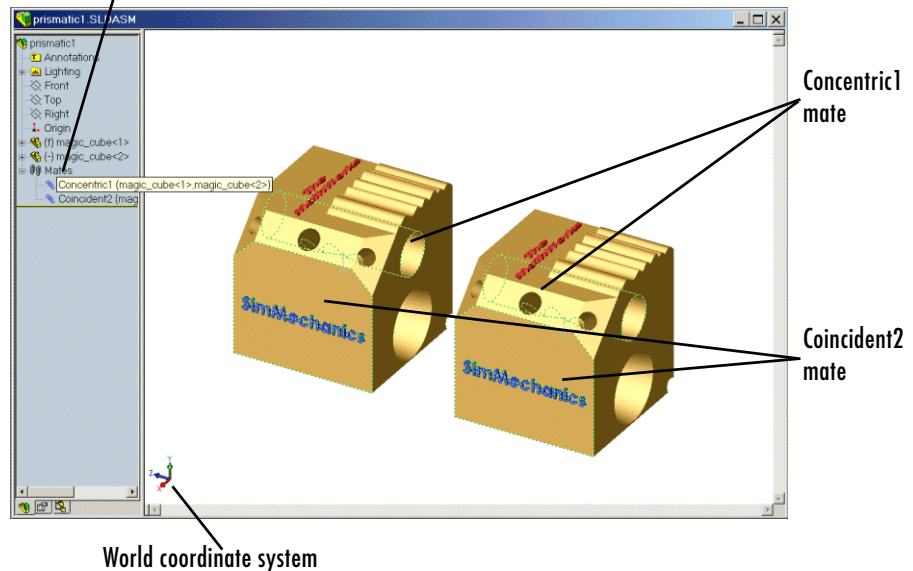
The following two assemblies use mates to constrain the two cubes to have only a single translational degree of freedom (DoF) between them. These are two ways to accomplish this; you can experiment with other mates to find more. In the translated SimMechanics models, this single DoF is a prismatic joint.

Prismatic as a Plane Mate and a Cylindrical Mate

Open the assembly file `prismatic1.ASSEMBLYFILETYPE` and examine the CAD hierarchy. Locate and expand the Mates node. There are two mates constraining the two cubes.

- 1 Highlight the first mate, Concentric1. The mate geometry is highlighted in the assembly. This mate allows the two cubes only to slide along and rotate about the z -axis running through the center of the parallel and concentric upper holes of each cube.
- 2 Highlight the second mate, Coincident2. The mate mate geometry is highlighted in the assembly. This mate allows the two cubes to slide along the y - z plane, with the two sides marked “SimMechanics” sharing a common plane, representing two translational DoFs. It also allows the two cubes to rotate about the x -axis. The cubes are not allowed to rotate about any other axis, or to translate perpendicular to the y - z plane.

Mates node and mates



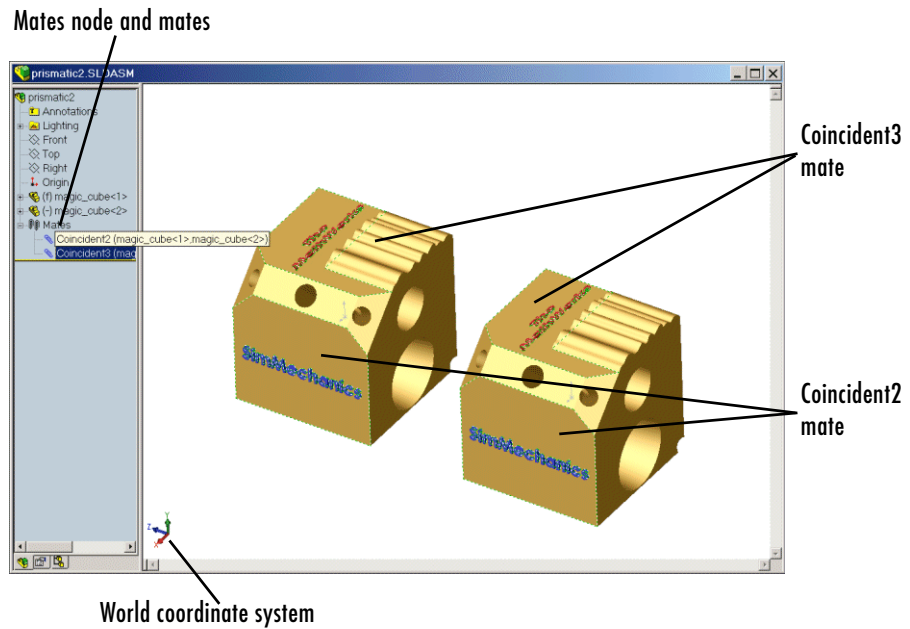
Plane and Cylindrical Mates Constraining Two Cubes

The two mates together constrain the two cubes such that they can only slide along the z -axis common to the two upper concentric holes. The second mate prevents rotation about this axis, leaving the whole assembly with only one translational DoF.

Prismatic as Two Orthogonal Plane Mates

Open the assembly file `prismatic2.ASSEMBLYFILETYPE` and examine the CAD hierarchy. Locate and expand the Mates node. There are two mates constraining the two cubes.

- 1** Highlight the first mate, Coincident2. The mate geometry is highlighted in the assembly. This mate allows the two cubes to slide along the y - z plane, with the two sides marked “SimMechanics” sharing a common plane, representing two translational DoFs. It also allows the two cubes to rotate about the x -axis. The cubes are not allowed to rotate about any other axis, or to translate perpendicular to the y - z plane.
- 2** Highlight the second mate, Coincident3. The mate geometry is highlighted in the assembly. This mate allows the two cubes to slide along the x - z plane, with the two sides marked “The MathWorks” sharing a common plane, representing two translational DoFs. It also allows the two cubes to rotate about the y -axis. The cubes are not allowed to rotate about any other axis, or to translate perpendicular to the x - z plane.



Two Plane Mates Constraining Two Cubes

The two mates together constrain the two cubes such that they can only slide along the z -axis common to the two planes $y-z$ and $x-z$, leaving the whole assembly with only one translational DoF.

Exporting the Assemblies and Generating SimMechanics Models

Export the two assemblies using the **Save as type** option SimMechanics (*.xml) under **File** → **Save As**. The saved XML files are prismatic1.xml and prismatic2.xml. Copy or move them to a MATLAB working directory.

At the MATLAB command line, generate SimMechanics models using `import_physmod`. In both models, the assemblies are translated into the prismatic1 and prismatic2 subsystems, respectively. Each subsystem has eight blocks, arranged in the common structure described in “Block Diagram Structure of Two-Part Assemblies” on page 23.

The Custom Joint represents the single translational DoF between the two cubes with one prismatic primitive along the z -axis. You can replace this Custom Joint with a Prismatic Joint block.

Creating a Revolute Joint

The following assembly uses mates to constrain the two cubes to have only a single rotational degree of freedom (DoF) between them. In the translated SimMechanics model, this single DoF is a revolute joint.

Viewing and Exporting the Assembly

Open the assembly file `revolute.ASSEMBLYFILETYPE` and examine the CAD hierarchy. Locate and expand the Mates node. There are two mates constraining the two cubes.

- 1 Highlight the first mate, Concentric1. The mate geometry is highlighted in the assembly. This mate allows the two cubes to slide along and rotate about the z -axis running through the center of the parallel and concentric upper holes of each cube.
- 2 Highlight the second mate, Coincident1. The mate geometry is highlighted in the assembly. This mate allows the two cubes to slide along the x - y plane, with the parallel sides sharing a common plane. It also allows the two cubes to rotate about the z -axis. The cubes are not allowed to rotate about any other axis, or to translate perpendicular to the x - y plane.

The two mates together constrain the two cubes such that they can only rotate about the z -axis orthogonal to the x - y plane, leaving the whole assembly with only one rotational DoF.

Export the assembly using the **Save as type** option SimMechanics (*.xml) under **File** → **Save As**. The saved XML file is `revolute.xml`. Copy or move it to a MATLAB working directory.

Generating the Model

At the MATLAB command line, generate a SimMechanics model using `import_physmod`. The assembly is translated into an eight-block subsystem called `revolute`, arranged in the common structure described in “Block Diagram Structure of Two-Part Assemblies” on page 23.

The Custom Joint represents the single rotational DoF between the two cubes with one revolute primitive about the z -axis. You can replace this Custom Joint with a Revolute Joint block.

Creating an Inplane Joint

The following assembly uses mates to constrain the two cubes to have only two translational degrees of freedom (DoFs) between them. In the translated SimMechanics model, these two DoFs are two prismatic joints.

Viewing and Exporting the Assembly

Open the assembly file `inplane.ASSEMBLYFILETYPE` and examine the CAD hierarchy. Locate and expand the Mates node. There are two mates constraining the two cubes.

- 1 Highlight the first mate, Coincident2. The mate geometry is highlighted in the assembly. This mate allows the two cubes to slide along the y - z plane, with the two sides marked “SimMechanics” sharing a common plane. It also allows the two cubes to rotate about the x -axis. The cubes are not allowed to rotate about any other axis, or to translate perpendicular to the y - z plane.
- 2 Highlight the second mate, Parallel1. The mate geometry is highlighted in the assembly. This mate allows the two cubes to slide parallel to the x - z plane, with the two sides marked “The MathWorks” parallel but not necessarily in the same plane. It also allows the two cubes to translate perpendicular to the x - z plane and to rotate about the y -axis. The cubes are not allowed to rotate about any other axis.

The two mates together constrain the two cubes such that they can only slide in the y - z plane, leaving the whole assembly with only two translational DoFs.

Export the assembly using the **Save as type** option SimMechanics (*.xml) under **File** → **Save As**. The saved XML file is `inplane.xml`. Copy or move it to a MATLAB working directory.

Generating the Model

At the MATLAB command line, generate a SimMechanics model using `import_physmod`. The assembly is translated into a subsystem, `inplane`,

having eight blocks, arranged in the common structure described in “Block Diagram Structure of Two-Part Assemblies” on page 23.

The Custom Joint represents the two translational DoFs between the two cubes with two prismatic primitives, along the y -axis and the z -axis. You can replace this Custom Joint with an In-Plane Joint block.

Creating a Spherical-Spherical Massless Connector

The following assembly use mates to constrain the two cubes to have six rotational degrees of freedom (DoFs) between them, represented by two spherical primitives. The spherical primitives pivot independently about two pivot points at a fixed relative distance. In the translated SimMechanics model, a spherical-spherical massless connector represents these six DoFs.

Viewing and Exporting the Assembly

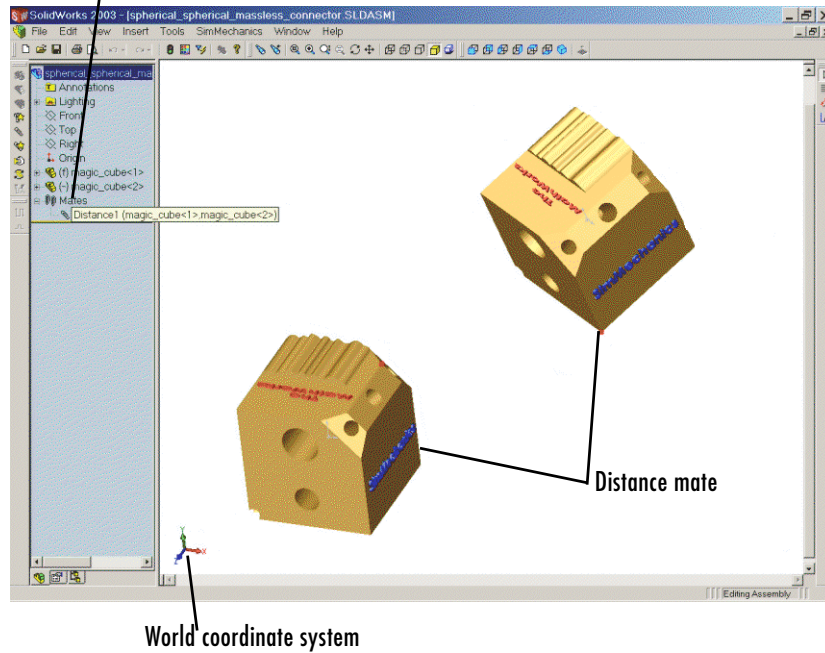
Open the assembly file

```
spherical_spherical_massless_connector.ASSEMBLYFILETYPE
```

and examine the CAD hierarchy. Locate and expand the Mates node. There is one mate constraining the two cubes.

Highlight this mate, Distance1. The two spherical pivot points are highlighted as small red or green squares, one on each cube. These points are the endpoints of the rigid massless connector. The cubes can move such that the distance between these two points (the length of the massless connector) does not change. The mate allows the two cubes to pivot independently about their connector endpoints.

Mate node and mate



A Distance Mate Constraining Two Cubes

Export the assembly using the **Save as type** option SimMechanics (*.xml) under **File** → **Save As**. The saved XML file is

```
spherical_spherical_massless_connector.xml
```

Copy or move it to a MATLAB working directory.

Generating the Model

At the MATLAB command line, generate a SimMechanics model using `import_physmod`. The assembly is translated into an eight-block subsystem called `spherical_spherical_massless_connector`, arranged in the common structure described in “Block Diagram Structure of Two-Part Assemblies” on page 23.

The Spherical-Spherical massless connector Joint block represents the two spherical primitives, each with three rotational DoFs, independently pivoting at each end of the massless, rigid connector connecting the two cubes.

Creating a CAD-Based Robot Arm Model

The example of this section is based on a more complex CAD assembly, a robot arm. It includes multiple parts, multiple mates, and a subassembly.

- “Viewing the Robot Arm Assembly”
- “Exporting the Robot Arm Assembly” on page 36
- “Generating and Completing the Robot Arm Model” on page 36

After you install your CAD translator, the eleven CAD files for the robot arm are located in the directory

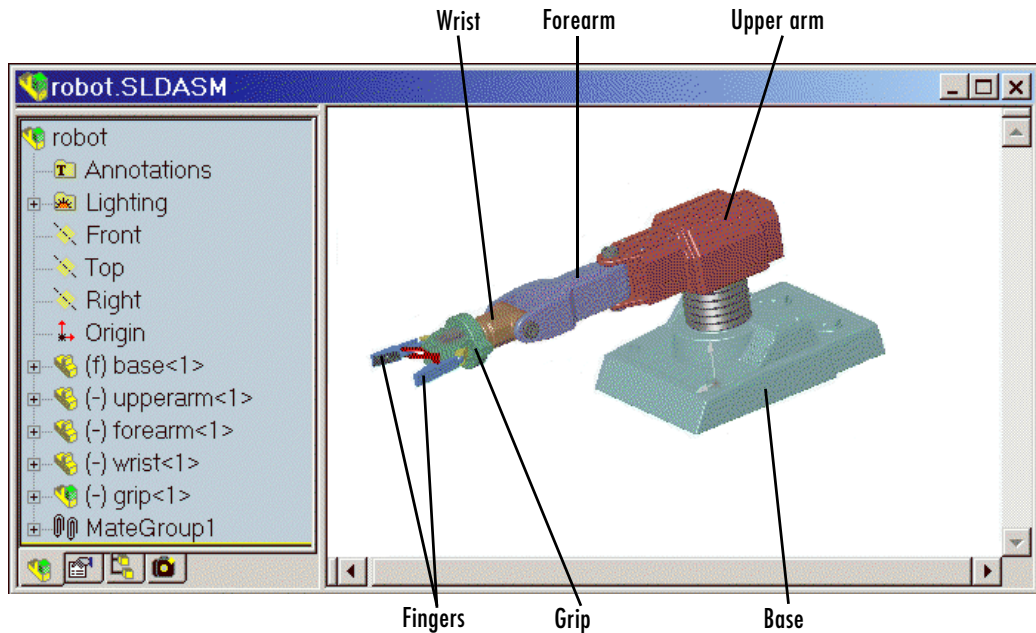
Applications/TheMathWorks/SimMechanics/examples/robot/

relative to the top of your CAD platform installation. The eleven files are

File Name	CAD File Type
robot.ASSEMBLYFILETYPE	Assembly
grip.ASSEMBLYFILETYPE	Subassembly (flexible)
base.PARTFILETYPE	Parts (main assembly)
forearm.PARTFILETYPE	
upperarm.PARTFILETYPE	
wrist.PARTFILETYPE	
fingertips.PARTFILETYPE (twice)	Parts (subassembly)
firstfingerlink.PARTFILETYPE	
firstfingerlinkL.PARTFILETYPE	
metacarpal.PARTFILETYPE	
secondfingerlink.PARTFILETYPE (twice)	

Viewing the Robot Arm Assembly

Open the assembly file for the whole robot.



Robot Arm Assembly in a CAD Platform

Then examine the CAD hierarchy.

- Five of the part files are grouped into the subassembly grip. The subassembly uses two instances each of fingertips and secondfingerlink.
- The subassembly has its own group of 18 mates, MateGroup1. Two mates, Angle1 and Angle2, are not active DoFs. If they were active, they would lock the grip fingers into the open position. Here, each grip finger can move separately.
- The other four part files are separate and grouped into the main assembly.
- The main assembly has its own MateGroup1, consisting of seven mates.

The whole assembly has eight DoFs. The grip subassembly alone contains two, allowing each finger to open and close separately. The main assembly has six DoFs:

- The upper arm can move relative to the base by pitching, yawing, and rolling (three DoFs).
- The forearm can yaw relative to the upper arm (one DoF).
- The wrist can pitch relative to the forearm (one DoF).
- The grip can rotate about its symmetry axis (one DoF).

Exporting the Robot Arm Assembly

If you opened the **Settings** window, apply the settings and close it. If you changed the assembly or any subassemblies, you need to rebuild the assembly before exporting it to XML.

Now export the assembly into Physical Modeling XML. Under **File**, choose **Save As**, and for **Save as type**, select **SimMechanics (*.xml)**. Click **OK**. The XML file `robot.xml` appears in your working CAD directory.

Generating and Completing the Robot Arm Model

Now you can generate a model for a robot arm based on the file `robot.xml`. You can use the preconfigured demo file or export your own version of the XML file from the robot arm CAD assembly, then copy or move the XML file to your MATLAB working directory.

See “Generating Models from Physical Modeling XML” on page 14 to learn more about how to generate models from XML files.

Generating the Initial Model

The preconfigured `robot.xml` file is located in the `toolbox/physmod/mech/mechdemos` directory of your MATLAB installation. Generate the model by entering

```
import_physmod('robot')
```

at the command line. The status bar appears and indicates the progress of model generation.

A model window, named **robot**, opens and is populated with blocks. Save this initial body-joint model, and note these properties:

- The whole robot arm assembly is contained in the subsystem `robot`.
- The top level of the assembly has 13 blocks and the `grip-1` subsystem.

- The grip-1 subsystem has 18 blocks.

The original robot arm assembly has eight DoFs, with two in the grip subassembly and six at the top level. These translate into eight DoFs in the SimMechanics model:

- Six DoFs occur at the top level. These include the upper arm relative to the base, the forearm relative to the upper arm, the wrist relative to the forearm, and the grip relative to the wrist.
- Two DoFs occur in the subsystem. These are the rotational DoFs of the two grip fingers.

There are eight revolute primitives in the subsystem. But they occur in two closed loops as two independent four-bar mechanisms. Each four-bar mechanism actually has only one independent DoF, because each four-bar loop closes on itself. (See “A Four Bar Mechanism” on page 2-36 and “Counting Degrees of Freedom” on page 4-71 in the SimMechanics documentation.)

Obtaining Simulink and Additional SimMechanics Blocks

To complete the robot arm model, you need blocks from the SimMechanics and Simulink block libraries. Open these libraries by entering `mechlib` and `simulink`, respectively, at the command line.

You can also open the Simulink library from the MATLAB window menu or toolbar.

Editing the Model

Some of the blocks in the generated robot arm model are redundant. You can remove them without affecting the model’s dynamics, as long as you properly reconnect the remaining blocks.

- At the top level, the `SimMechanics_RootPart` block is a zero-mass, zero-inertia Root Body. You can delete it, along with the `SimMechanics_RootWeld` block, then reconnect the Root Ground to the base through the `SimMechanics_RootPart--base-1 Weld` block.
- In the grip-1 subsystem, you can delete the grip-1 Root Body block and the `grip-1--grip-1 | metacarpal-1 Root Weld` block, because they are unnecessary. You can also delete the associated body coordinate system on the `metacarpal-1 Body` block.

The non-Weld Joint blocks, those that carry DoFs, are Custom Joints configured with the proper primitives to represent the original CAD assembly's DoFs. You can replace them with appropriate blocks from the SimMechanics Joints library.

- All the Custom Joints, except for one, are configured to represent a single revolute primitive. You can replace each with an equivalent Revolute block.
- The exception is the base-1--upperarm-1 Custom Joint, the first non-Weld Joint encountered as you move away from the Ground block. This Joint is configured to represent a spherical primitive (three rotational DoFs). You can replace this block with an equivalent Spherical block.

If you replace a Custom Joint with a different, equivalent Joint, be sure to reproduce the axis of action and reference coordinate system information from each old block to each new one.

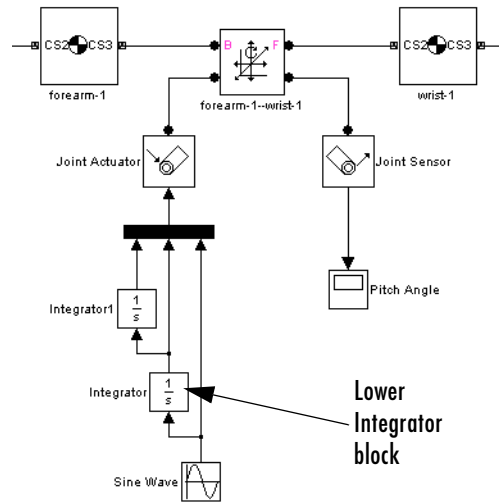
Save this intermediate model as robot2.

Adding an Actuator and a Sensor

You can motion-actuate the wrist relative to the forearm.

- 1** Double-click the forearm-1--wrist-1 Joint. Change the **Number of sensor/actuator ports** to 2.
- 2** Click **OK**. Two new ports appear on the Joint.
- 3** From the SimMechanics Sensors & Actuators library, drag, drop, and attach a Joint Actuator and a Joint Sensor to these new ports.
- 4** Configure the Joint Actuator to accept motion signals. Be sure the angular units are deg (degrees).
- 5** From the Simulink library, drag and drop a Sine Wave, a Mux, two Integrator, and one Scope block. Connect them to the previous blocks as shown. Rename the Scope block to Pitch Angle.

Consult the Simulink documentation for more about these Simulink blocks.



- 6 In the Sine Wave block, set the **Amplitude** to $60 \cdot \pi \cdot \pi$ and the **Frequency** to 60. Leave all other defaults unchanged.
- 7 In the lower Integrator block, set **Initial condition** to $-60 \cdot \pi$. Leave all other defaults unchanged.

Configuring Tolerances

The original robot arm CAD assembly requires looser tolerances than the SimMechanics defaults, and its motion can lead to singularities. To avoid simulation errors or severe slowdown, you need to reconfigure the assembly tolerances and constraint solver.

- 1 Open the Machine Environment block.
- 2 On the **Parameters** panel, reset the **Linear assembly tolerance** to $1e-2$ m (meters) and the **Angular assembly tolerance** to $1e-1$ rad (radians).
- 3 On the **Constraints** panel, select the **Use robust singularity handling** check box. Leave all other defaults. Click **OK**.
- 4 Resave your finished model as robot3.

Observing the Arm Motion

You can now run robot3 and examine its motion.

To use the motion sensor,

- 1 Double-click the Pitch Angle block to open a scope.
- 2 Click the **Start simulation** button. The scope plot displays a trace of the pitch angle motion.

To visualize the body motions,

- 1 From the **Simulation** menu, select **Configuration Parameters**, then the **SimMechanics** node.
- 2 Select **Update machine visualization on update diagram** and **Animate machine during simulation**. Click **OK**.
- 3 Select **Update Diagram** from the **Edit** menu. The SimMechanics visualization window opens.
- 4 In the special **SimMechanics** menu of the visualization window, select **Machine Display**, then **Ellipsoids**. The display now shows the robot arm's component bodies as ellipsoids.
- 5 Click the **Start** button. The simulation begins. Observe the robot arm motion in the SimMechanics window.

Modeling a Stewart Platform

This section introduces a very complex computer-aided design (CAD) assembly that models the Stewart platform, a six-degree-of-freedom (DoF) mechanical system used for accurate positioning applications.

- “Viewing the Stewart Platform Assembly” on page 42
- “Exporting the Stewart Platform Assembly” on page 43
- “Generating the Stewart Platform Model” on page 43
- “Visualizing the Stewart Platform Motion” on page 46

Note The Stewart platform assembly of this section is an advanced example of computer-aided design. You should work through the previous examples of this chapter before attempting to work with this assembly.

After you install your CAD translator, the 45 CAD files for the Stewart platform are located in the directory

```
Applications/TheMathWorks/SimMechanics/examples/stewart/
```

relative to the top of your CAD platform installation. The master assembly file is `stewart_platform.ASSEMBLYFILETYPE`.

What the Stewart Platform Does

The Stewart platform consists of two plates connected by six mobile and extensible legs. The lower or base plate is immobile. The upper or mobile plate has six degrees of freedom, three rotational and three translational. The platform is highly stable and easy to control. Originally designed for flight simulators, the Stewart platform is also used for many transportation and construction applications that require precise positioning and orienting.

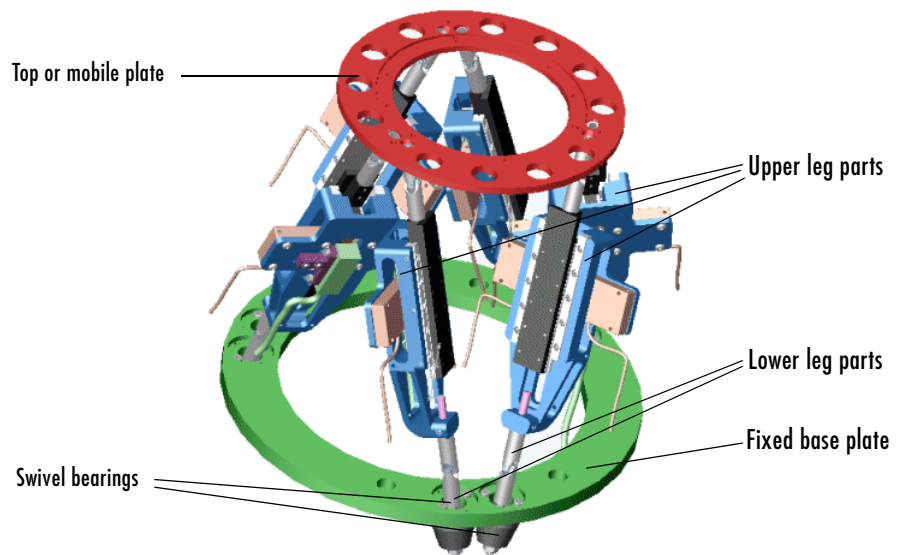
The platform’s six legs each have two parts, an upper and a lower leg, with a piston-like cylindrical DoF between each pair of parts. The legs are connected to both the base plate and the top plate by universal joints located at each end of each leg. (These universals are not just sets of abstract DoFs. Each also contains a spider-like body, while also having two DoFs.) The upper part of each leg can slide into and out of the lower leg, allowing each leg to be varied

in length. The position and orientation of the mobile platform (top plate) varies depending on the lengths to which the six legs are separately adjusted.

The entire Stewart platform assembly, once the top is connected to the legs, has six independent DoFs, the same as the top plate would have if it were disconnected. You can think of these independent DoFs as the six adjustable leg lengths or, equivalently, as the six DoFs of the mobile plate. See “Counting Degrees of Freedom” in the “Modeling Mechanical Systems” chapter of the SimMechanics documentation.

Viewing the Stewart Platform Assembly

Open the master assembly file, `stewart_platform.ASSEMBLYFILETYPE`. Grab the assembly and rotate it about to view the top and bottom plates and the legs.



Stewart Platform CAD Assembly

The CAD hierarchy for the Stewart platform contains assemblies for the top and base plates, as well as assemblies for the six legs. All the mates of the assembly are grouped into one mate group, containing 30 mates. There are 448 component parts and 38 subassemblies, which you can open individually to examine the separate parts.

The base plate is about 24 centimeters (cm) in diameter, the top plate about 16.5 cm. When centered and oriented flat, the top plate is about 20 cm above the base. The assembly models the platform material as aluminum (about 2.7 grams per cubic cm).

Exporting the Stewart Platform Assembly

If you opened the **Settings** window, apply the settings and close it. If you changed the assembly or any subassemblies, you need to rebuild the assembly before exporting it to XML.

Now export the assembly into Physical Modeling XML by using **Save As** under **File** and **SimMechanics (*.xml)** for **Save as type**.

Because the assembly is so complex, the export process takes longer than it does for simpler assemblies. As the export proceeds, the translator highlights various parts and subassemblies. When the highlighting stops, the export is finished.

The exported model appears as the XML file `stewart_platform.xml` in your working CAD directory.

Generating the Stewart Platform Model

Now move or copy the `stewart_platform.xml` file into your working MATLAB directory.

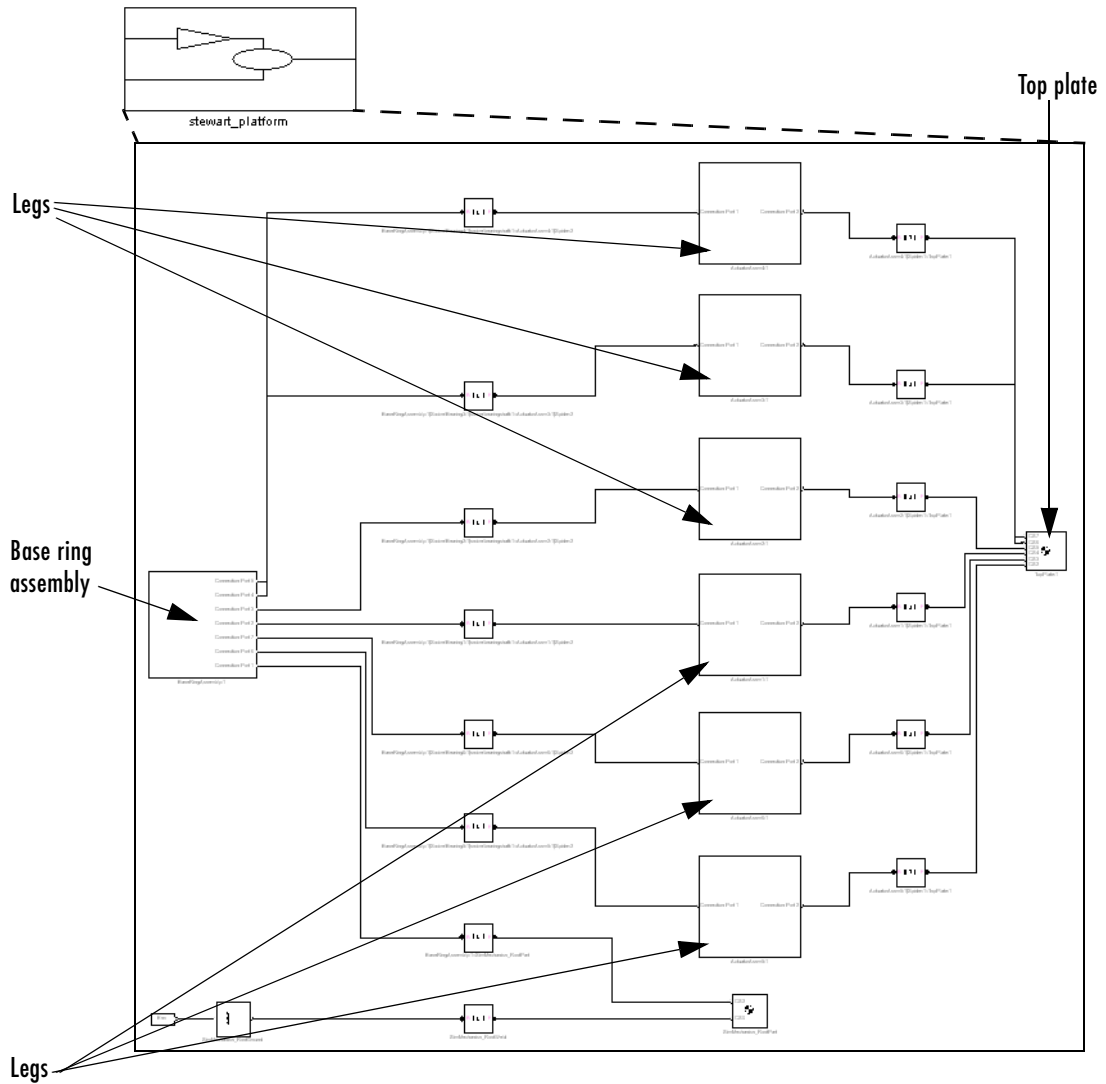
At the MATLAB command line, enter

```
import_physmod('stewart_platform')
```

and wait for SimMechanics to complete the generation of the new Simulink model `stewart_platform`.

Inspecting the Generated Model and Counting Its DoFs

The entire Stewart platform model is contained in a subsystem, also called `stewart_platform`. This subsystem itself contains seven subsystems.



The Stewart Platform Model: Base, Legs, and Top Plate

The subsystems correspond to the subassemblies of the original CAD assembly: one is the base plate, the other six are the platform legs.

- The base plate subassembly `BaseRingAssembly - 1` contains six subassemblies, modeling a base swivel bearing for each leg.
- The six leg subassemblies, `ActuatorAssm`, model the upper and lower halves of each leg and represent part of their DoFs.

For each leg, there are six DoFs. Two pairs of revolute associated with each leg represent the two universal joints connecting each leg to the top and base plates, respectively. Each of these universals has two DoFs.

- At the top level, there are two revolute, one attached to either end of a leg subassembly, connecting each leg to the base and top plates, respectively.
- Within each leg subassembly, there are two other revolute, each one connecting the leg to the top and base plates, respectively.

One of the revolute inside the leg subassembly pairs with one of the revolute outside the leg assembly to make up a two-DoF universal. These pairs occur twice on each leg, one connecting the leg to the top plate, the other connecting the leg to the base plate.

- Within each leg subassembly, there is one prismatic, representing the leg's freedom to expand or contract along its shaft.
- Within each swivel bearing subassembly, itself located within the base ring assembly, is another revolute representing each leg's freedom to rotate about its shaft.

In all, each leg has six DoFs. But the constraints imposed by attaching each leg to fixed points on the base and top plates, respectively, reduce these DoFs to two independent DoFs for each leg: the freedom to rotate about its shaft, and the freedom to expand or contract along its shaft. The rotational DoFs associated with the universals at the attachment points are completely dependent on the leg's prismatic DoF.

Editing the Generated Model

The generated model contains a large number of redundant `RootWeld` and zero-mass `RootPart` blocks. You can delete these and not affect the model's dynamics, if you take care to reconnect the remaining bodies properly after deleting each `Weld`.

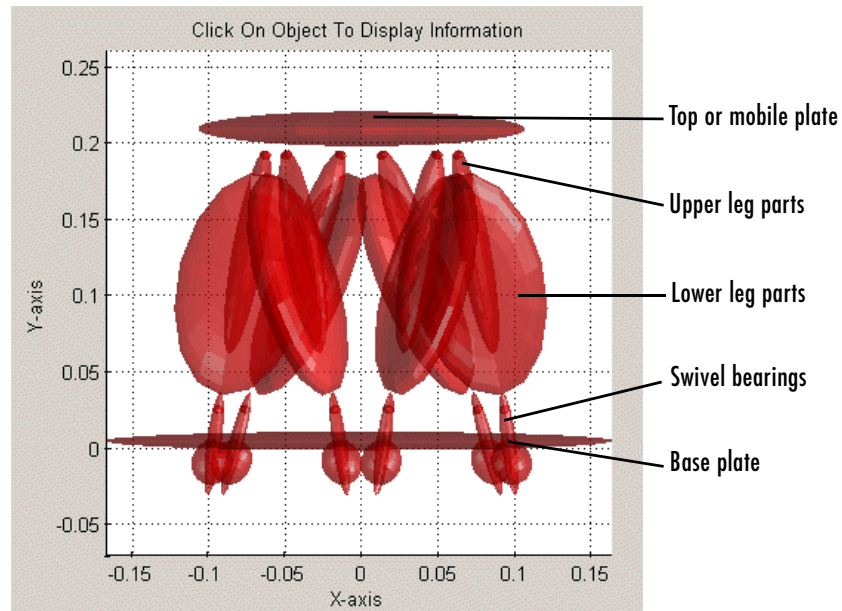
If you want the motion of the platform to be controlled by something other than gravity, you need to add the appropriate Actuators to the model. To quantify the model's motion, you need to make precise measurements with Sensors. You can drive the actuators with external control signals to model an open-loop controller for the Stewart platform. If you introduce feedback from the sensors to the actuators, you can model a closed-loop controller.

You can learn more about using actuator and sensor blocks in the “Modeling Mechanical Systems” chapter in the SimMechanics documentation.

Visualizing the Stewart Platform Motion

Without any external forces acting, apart from gravity, the platform collapses under its own weight. You can verify this by running and visualizing your Stewart platform model.

- 1** From the **Simulation** menu, select **Configuration Parameters**. The **Configuration Parameters** dialog opens. Choose the **SimMechanics** node.
- 2** Select **Update machine visualization on update diagram** and **Animate machine during simulation**. Click **Apply** or **OK**.
- 3** From the **Edit** menu, select **Update Diagram**. The SimMechanics visualization window opens with the special SimMechanics controls. The window displays the Stewart platform in its initial position.



- 4 Start the simulation by clicking the **Start** button in the toolbar of either the visualization window or the model window.

The mobile plate falls under its own weight and reaches the base plate in about 0.2 seconds. Because there is nothing to stop the legs or the top plate, the platform continues to collapse: the mobile plate falls below the base plate, and the upper and lower parts of each leg come apart.

Find more information about SimMechanics visualization in the “Visualizing and Animating Machines” chapter.

A

- assembly
 - CAD
 - See* computer-aided design

C

- CAD
 - See* computer-aided design
- CAD translator 2
- computer-aided design (CAD)
 - building assemblies 6
 - editing models derived from 16
 - exporting assemblies into XML 10
 - generating CAD-based models 14
 - getting CAD translator help 12
 - importing assemblies into SimMechanics 14
 - installing CAD translators 3
 - models derived from 15
 - troubleshooting assembly export errors 13
 - troubleshooting models derived from 18
 - uninstalling CAD translators 5

E

- exporting CAD assemblies
 - See* computer-aided design
 - See* XML

I

- importing CAD assemblies
 - See* computer-aided design
 - See* XML

T

- troubleshooting
 - See* computer-aided design

X

- XML
 - exporting from CAD 10
 - importing into SimMechanics 14
 - Physical Modeling format 2
 - See also* computer-aided design

