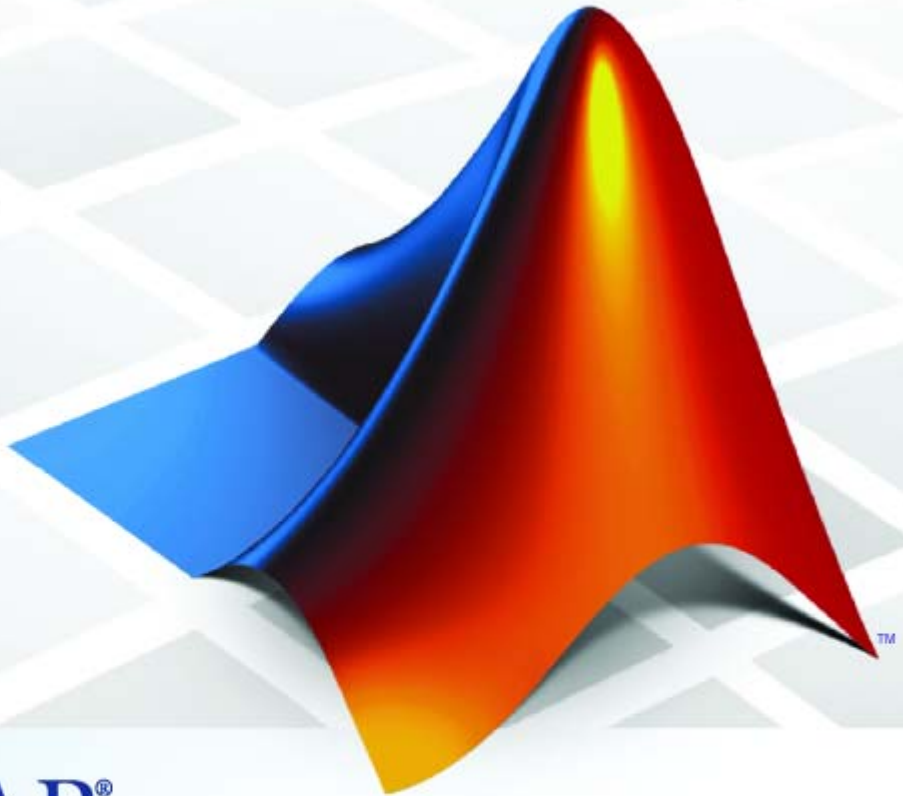


SimMechanics™ Link 1 Reference



MATLAB®

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SimMechanics™ Link Reference

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Linking and Using the SolidWorks Add-In

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Command Reference

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Linking and Using Pro/ENGINEER Toolkit

This chapter describes the procedure for linking SimMechanics™ Link software to the Pro/ENGINEER® computer-aided design (CAD) platform as a Pro/TOOLKIT application and for exporting CAD assemblies from Pro/ENGINEER into a form you can use with SimMechanics software.

For general information on installing and using SimMechanics Link software, as well as demos and examples, see the *SimMechanics Link User's Guide*.

- “Registering and Unregistering with Pro/ENGINEER” on page 1-2
- “Supported Pro/ENGINEER Constraints and Their Corresponding Joints” on page 1-6
- “Configuring SimMechanics Link Settings in Pro/ENGINEER” on page 1-11
- “Exporting Assemblies to Physical Modeling XML Format” on page 1-13
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Registering and Unregistering with Pro/ENGINEER

In this section...

“Requirements Before Registering” on page 1-2

“Creating a Pro/TOOLKIT Registry File” on page 1-2

“Choices for Registering the SimMechanics Link Utility” on page 1-3

“**SimMechanics Link** Menu in Pro/ENGINEER” on page 1-4

“Unregistering the SimMechanics Link Utility” on page 1-4

Requirements Before Registering

After you have installed the SimMechanics Link utility, you must register it with Pro/ENGINEER before you can use it. See “Installing and Linking SimMechanics Link Software” in the *SimMechanics Link User’s Guide*.

You must also have a Pro/TOOLKIT registry file for the SimMechanics Link utility.

Creating a Pro/TOOLKIT Registry File

To complete registration, you must have the following Pro/TOOLKIT registry text available, either as separate text file or as text ready to be pasted into an existing file. Consult the documentation that accompanies your Pro/ENGINEER installation for further details about Pro/TOOLKIT registration.

Pro/TOOLKIT Registry File for Pro/ENGINEER Wildfire 4 and Later Versions

The Pro/TOOLKIT registry file text ensures that Pro/ENGINEER treats the executable SimMechanics Link module as a Pro/TOOLKIT application. Each line of the registry file text consists of one of a predefined set of keywords, followed by a value.

Create the registry file text in the following standard form:

```
name SimMechanics Link
```

```

startup dll
exec_file $matlabroot/bin/arch/cl_proe2sm.dll
text_dir $matlabroot/toolbox/physmod/smlink/cad_systems/proe/
unicode_encoding false
end

```

Replace the variables *\$matlabroot* and *arch* by, respectively, the directory path to your MATLAB® installation and the operating system architecture; for example, for 32-bit Windows®, use win32.

Pro/TOOLKIT Registry File for Pro/ENGINEER Wildfire 3 and Earlier Versions

For the Wildfire 3 and earlier versions of Pro/ENGINEER, you must omit the next-to-last line in the standard registry file. For Wildfire 3 and earlier versions, the complete registry file text is:

```

name SimMechanics Link
startup dll
exec_file $matlabroot/bin/arch/cl_proe2sm.dll
text_dir $matlabroot/toolbox/physmod/smlink/cad_systems/proe/
end

```

Choices for Registering the SimMechanics Link Utility

Before you can register SimMechanics Link as a Pro/TOOLKIT application, you must have already installed Pro/ENGINEER and SimMechanics Link software on your system.

There are two ways to register the SimMechanics Link utility with Pro/ENGINEER. Either way completes the Pro/TOOLKIT registration of the SimMechanics Link utility.

Registering Using a Separate Pro/TOOLKIT Registry File

To register using a separate Pro/TOOLKIT registry file:

- 1 Create a SimMechanics Link Pro/TOOLKIT registry file and save it in an accessible location on your system. See “Creating a Pro/TOOLKIT Registry File” on page 1-2.

- 2 Specify the absolute path of this registry file in the PROTKDAT, PRODEV DAT, and TOOLKIT_REGISTRY_FILE statements of the Pro/ENGINEER configuration file.

When you open Pro/ENGINEER again, it searches for this registry file by this absolute path.

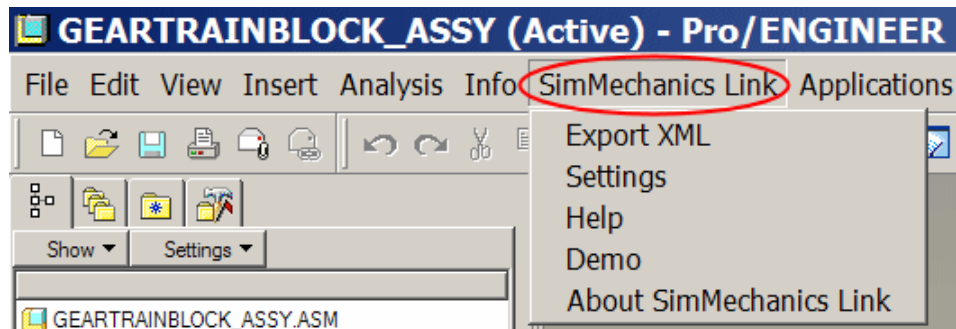
Registering Using a Modified Existing Pro/TOOLKIT Registry File

To register using an existing Pro/TOOLKIT registry file, you must locate and modify the existing Pro/TOOLKIT registry file in your Pro/ENGINEER installation.

Copy the contents of the SimMechanics Link registry file into your existing Pro/TOOLKIT registry file and save the Pro/TOOLKIT registry file.

SimMechanics Link Menu in Pro/ENGINEER

After you register the SimMechanics Link utility with Pro/ENGINEER, the **SimMechanics Link** menu appears in the Pro/ENGINEER menu bar.



Unregistering the SimMechanics Link Utility

How you registered the SimMechanics Link utility with Pro/ENGINEER determines how you unregister it. See “Choices for Registering the SimMechanics Link Utility” on page 1-3.

- If you created a separate SimMechanics Link registry file, remove the absolute path specification of this registry file from the Pro/ENGINEER configuration file.
- If you copied the SimMechanics Link registration text into your Pro/TOOLKIT registry file, delete that text.

Supported Pro/ENGINEER Constraints and Their Corresponding Joints

In this section...
“Degrees of Freedom and Constraints” on page 1-6
“Supported Pro/ENGINEER Constraint Entities” on page 1-7
“Supported Pro/ENGINEER Constraint–Constraint Entity Combinations” on page 1-7
“Supported SimMechanics Joints and Pro/ENGINEER Connections” on page 1-9

Degrees of Freedom and Constraints

In Pro/ENGINEER, an unconstrained part has six degrees of freedom (DoFs). You reduce these DoFs by inserting constraints between bodies. Groups of constraints, called connections, define particular types of motion.

In SimMechanics models, a body has no DoFs until you connect joints to it. Each joint is a combination of these joint primitives.

Primitive	Abbreviation	Motion Type	Number of DoFs
Prismatic	P	Translational	1
Revolute	R	Rotational	1
Spherical	S	Rotational	3
Weld	W	None	0

CAD translation maps the Pro/ENGINEER constraints between parts to SimMechanics joint primitives between bodies. In general, the mapping of constraints to joints is not one-to-one. When you generate a SimMechanics model from a CAD assembly, the primitives are combined into the appropriate Joints.

Supported Pro/ENGINEER Constraint Entities

The SimMechanics Link utility conditionally supports the following Pro/ENGINEER constraint entities, depending on what constraints they are combined with.

Entity	Description
Circle/Ellipse/Arc	Elliptical edge/arc sketch segment*
Cone	Conical face
Cylinder	Cylindrical face
Line	Linear edge/sketch segment/reference axis
Plane	Reference plane or planar face
Point	Vertex/sketch point/reference point

* A circular arc is a special case of an elliptical arc. A complete circle or ellipse is a special case of a circular or elliptical arc.

Supported Pro/ENGINEER Constraint–Constraint Entity Combinations

The SimMechanics Link utility conditionally supports the following Pro/ENGINEER constraint–constraint entity combinations. The combinations are grouped by constraint according to the structure:

Constraint Entity1 — Constraint — Constraint Entity2

Note A combination listed here might be supported, depending on the overall geometry of the assembly. A combination not listed here is not supported.

If the SimMechanics Link exporter cannot translate a constraint–constraint entity combination into a supported SimMechanics joint with DoFs, it converts the combination into a weld (W) primitive.

Align or Mate Constraint Without Offset – Point on Line, Edge on Surface, or Point on Surface Constraint

The paired entities indicated with a plus (+) sign are supported when linked with a one of these constraints:

- Align without offset
- Mate without offset
- Point on Line
- Edge on Surface
- Point on Surface

Mate Entity2	Point	Line	Plane	Cylinder	Cone	Circle/ Ellipse/Arc
Mate Entity1						
Point	+					
Line		+	+			
Plane		+	+			+
Cylinder				+	+	+
Cone				+	+	+
Circle/Ellipse/Arc			+	+	+	+

Insert Constraint

The paired entities indicated with a plus (+) sign are supported when linked with an Insert constraint.

Mate Entity2	Point	Line	Plane	Cylinder	Cone	Circle/ Ellipse/Arc
Mate Entity1						
Point						
Line				+	+	+

Plane			+			
Cylinder		+		+	+	+
Cone		+		+	+	+
Circle/Ellipse/Arc		+		+	+	+

Align or Mate Constraint with Translational Offset

These entity pairs are supported when linked with an Align or Mate constraint with translational offset:

- Point — Point
- Plane — Plane
- Line — Plane or Plane — Line
- Plane — Point or Point — Plane

Align or Mate Constraint with Rotational Offset

These entity pairs are supported when linked with an Align or Mate constraint with rotational offset:

- Line — Line
- Plane — Plane

Supported SimMechanics Joints and Pro/ENGINEER Connections

The SimMechanics Link utility supports the following SimMechanics primitives and primitive combinations. The last column lists the equivalent Pro/ENGINEER connections. The weld (W) is the default target primitive if the exporter is unable to convert a mate–mate entity combination into a moving joint.

Primitive Combination	Description	Equivalent Pro/ENGINEER Connection
P	Prismatic	Slider

Primitive Combination	Description	Equivalent Pro/ENGINEER Connection
PP	Planar: P and P are perpendicular	Planar
PPP	Custom	
PPPR	Custom	
S	Spherical	Ball
R-S	Revolute-spherical massless connector	
R	Revolute	Pin
PR	Cylindrical: P and R are parallel	Cylinder
PPR	In-plane: R is perpendicular to PP	
PPPS	Six-DoF	6DOF
R-R	Revolute-revolute massless connector	
S-S	Spherical-spherical massless connector	
W	Weld: Rigid with no DoFs	Rigid

Tips for Specific Constraints

- You form a spherical primitive with a point-point Align constraint with zero offset.
- A point-point Align constraint with nonzero offset is translated to a spherical-spherical massless connector joint. For the translation to succeed, this constraint must be the only one between the two parts so constrained.

Configuring SimMechanics Link Settings in Pro/ENGINEER

In this section...

“About the SimMechanics Link Settings Dialog Box” on page 1-11

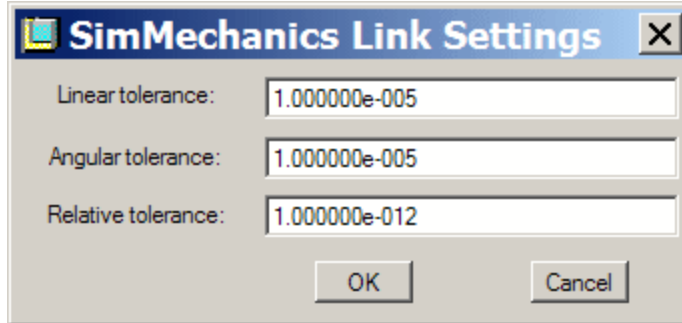
“Reviewing, Changing, and Applying Export Settings” on page 1-11

About the SimMechanics Link Settings Dialog Box

You can access certain export settings from the **SimMechanics Link** menu in your Pro/ENGINEER menu bar. Select **Settings** to open the settings dialog box.

Reviewing, Changing, and Applying Export Settings

The **SimMechanics Link Settings** dialog box contains three active fields where you can specify linear and angular tolerances as well as relative roundoff.



When the Pro/TOOLKIT application creates joints from Pro/ENGINEER constraints, it checks for vector alignments and spacing. Instead of comparing these alignments and spacings with zero, it compares them with the **Linear tolerance** and **Angular tolerance** that you specify in the dialog box. The units are meters (m) and radians (rad), respectively. The **Relative tolerance** specifies the smallest significant relative numerical difference.

- Click **OK** to save the settings and close the **SimMechanics Link Settings** dialog box.

- If you do not want to save your settings, click **Cancel** to close the **SimMechanics Link Settings** dialog box.

Exporting Assemblies to Physical Modeling XML Format

In this section...

“Export Procedure” on page 1-13

“Automatic Translation of a Rigid Subassembly to an Equivalent Rigid Body” on page 1-15

“CAD Assembly Export Errors” on page 1-15

Export Procedure

To export a currently open assembly as Physical Modeling XML:

- 1 From the **SimMechanics Link** menu, select **Export XML**. The **Select Directory** dialog box opens.



- 2 Choose a directory into which you want to save the exported XML file. Click **Open**.
- 3 A message appears in the message window at the bottom of the Pro/ENGINEER interface. The message prompts you to enter a name (without the .xml extension) for the exported XML file. Its default name is *filename.xml*, where *filename* is the name of the current CAD assembly.

- 4 Complete the export by clicking the green check mark button. If you decide to abort the export, clicking the red “X” button.



If no CAD translation errors occur, another message appears indicating that the translation is complete and the name and location of the XML file.

You can use the exported XML file to import as a SimMechanics model or give to another user to import.

STL Files Exported for Each Body

For each body in the CAD assembly, the exporter exports a stereolithographic (STL) file with geometric information about the body’s shape.

Automatic Translation of a Rigid Subassembly to an Equivalent Rigid Body

CAD assemblies in Pro/ENGINEER can have subassemblies without underconstrained components. Each corresponding SimMechanics subsystem, with each subassembly part translated individually, would have all its bodies welded to its subsystem ground. Such a translation would result in an unnecessarily complex SimMechanics model and reduced simulation performance. The subassembly components could not move relative to one another in any case.

The exporter automatically detects each such subassembly without underconstrained components and replaces it with a single equivalent rigid body in the translated model.

CAD Assembly Export Errors

If CAD translation errors occur:

- The specific CAD constraint errors are written to a separate error log file.
- The Pro/ENGINEER message window confirms that the XML file is still exported, and models that you generate from this file are valid. However, these models do not represent the original CAD assembly.

- An error dialog box opens, indicating the name and location of the exported XML file and the error log file.

For supported constraints and joints and for model generation instructions, see “Supported Pro/ENGINEER Constraints and Their Corresponding Joints” on page 1-6.

STL File Export Errors

If the exporter cannot export an STL file for one or more of the CAD assembly’s bodies, the Pro/ENGINEER message window displays a warning indicating for which bodies an STL file could not be created.

Getting Help and Examples

In this section...

“CAD Examples Based on Pro/ENGINEER” on page 1-17

“Getting SimMechanics Link Help and Examples from Pro/ENGINEER” on page 1-17

CAD Examples Based on Pro/ENGINEER

CAD assembly example files are located in the SimMechanics Link demos area of your MATLAB installation.

You can also view demos by opening the MATLAB Demos browser and locating the SimMechanics Link entry.

Getting SimMechanics Link Help and Examples from Pro/ENGINEER

You can open the MATLAB Help browser to the SimMechanics Link documentation or demos. From the Pro/ENGINEER **SimMechanics Link** menu:

- Select **Help** for documentation.
- Select **Demo** for demos.

See “**SimMechanics Link** Menu in Pro/ENGINEER” on page 1-4.

Linking and Using the SolidWorks Add-In

This chapter explains the procedure for linking SimMechanics Link software to the SolidWorks® computer-aided design (CAD) platform as a SolidWorks add-in and for exporting CAD assemblies from SolidWorks into a form you can use with SimMechanics software.

For general information on installing and using SimMechanics Link software, as well as demos and examples, see the *SimMechanics Link User's Guide*.

- “Linking to and Unlinking from SolidWorks” on page 2-2
- “Supported SolidWorks Mates and Their Corresponding Joints” on page 2-5
- “Configuring SimMechanics Link Settings in SolidWorks” on page 2-10
- “Exporting Assemblies to Physical Modeling XML Format” on page 2-13
- “Getting Help and Examples” on page 2-16

Linking to and Unlinking from SolidWorks

In this section...
“Requirements Before Linking” on page 2-2
“Registering SimMechanics Link Software” on page 2-2
“ SimMechanics Link Menu in SolidWorks” on page 2-4
“Unregistering SimMechanics Link Software” on page 2-4

Requirements Before Linking

Before you can register SimMechanics Link with SolidWorks, you must have already installed SolidWorks and SimMechanics Link software on your system. See “Installing and Linking SimMechanics Link Software” in the *SimMechanics Link User’s Guide*.

After you have installed the SimMechanics Link utility, you must register it with SolidWorks before you can use it.

Registering SimMechanics Link Software

You must perform two sets of steps to register the SimMechanics Link utility with SolidWorks, the first from within MATLAB, the second from within SolidWorks.

Steps in MATLAB

To start registration:

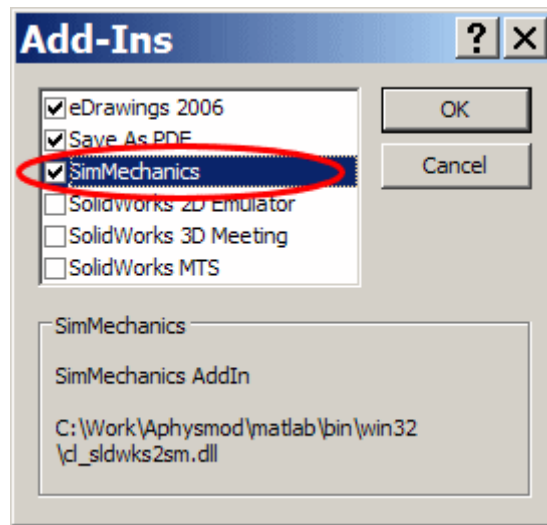
- 1 Start a MATLAB session with the SimMechanics Link utility already installed.
- 2 From the MATLAB command line, enter `smlink_linksw`. It should return a message indicating a successful link.

You can now close MATLAB if you want.

Steps in SolidWorks

To complete registration:

- 1 Start SolidWorks and close any introductory dialog boxes.
- 2 Choose **Add-Ins** from the **Tools** menu, and then select **SimMechanics**. Click **OK** to complete the link to SolidWorks.



Linking to Multiple Installations of SolidWorks

If you have more than one installation of SolidWorks, the SimMechanics Link utility is linked automatically to all installations by a single execution of the `smLink_linksw` command at the MATLAB command line.

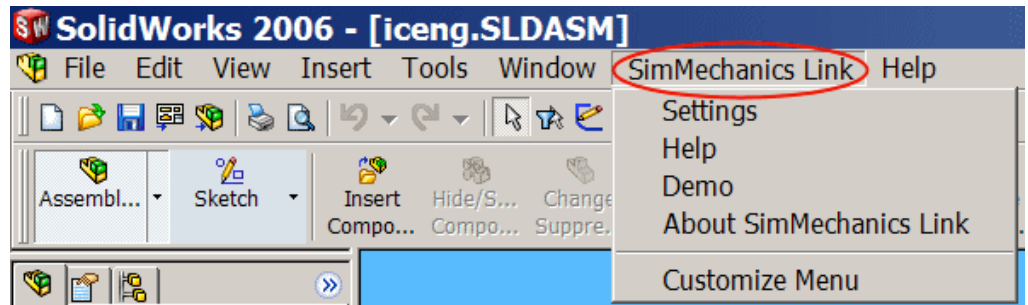
To complete the linking, you must still open each installation of SolidWorks, select **SimMechanics** from the **Tools > Add-Ins** menu, and click **OK**.

Linking Different SimMechanics Link Installations

You can link only one SimMechanics Link installation at a time to a given CAD platform. If you want to change the SimMechanics Link installation that is registered with SolidWorks, you must unlink the existing installation, then link the other installation.

SimMechanics Link Menu in SolidWorks

After linking the SimMechanics Link utility, the **SimMechanics Link** menu appears in the SolidWorks menu bar if you open a SolidWorks assembly file (file with extension .SLDASM).



Unregistering SimMechanics Link Software

To unregister the SimMechanics Link utility from a MATLAB session running on the same computer as your SolidWorks installation, enter the `smLink_unlinksw` command at the MATLAB command line.

Supported SolidWorks Mates and Their Corresponding Joints

In this section...

“Degrees of Freedom, Mates, and Constraints” on page 2-5

“Supported SolidWorks Mate Entities” on page 2-5

“Supported SolidWorks Mate–Mate Entity Combinations” on page 2-6

“Supported SimMechanics Joints” on page 2-8

Degrees of Freedom, Mates, and Constraints

In SolidWorks, an unconstrained part has six degrees of freedom (DoFs). You reduce these DoFs by inserting mates (constraints) between bodies.

A SimMechanics body has no DoFs until you connect joints to it. Each joint is a combination of these joint primitives.

Primitive	Abbreviation	Motion Type	Number of DoFs
Prismatic	P	Translational	1
Revolute	R	Rotational	1
Spherical	S	Rotational	3
Weld	W	None	0

CAD translation maps the SolidWorks mates between parts to SimMechanics joint primitives between bodies. In general, the mapping of mates to joints is not one-to-one. When you generate a SimMechanics model from a CAD assembly, the primitives are combined into the appropriate Joints.

Supported SolidWorks Mate Entities

The SimMechanics Link utility conditionally supports the following SolidWorks mate entities, depending on what mates they are combined with.

Entity	Description
Circle/Arc	Circular edge/arc sketch segment*
Cone	Conical face
Cylinder	Cylindrical face
Line	Linear edge/sketch segment/reference axis
Plane	Reference plane or planar face
Point	Vertex/sketch point/reference point

* A complete circle is a special case of a circular arc.

Supported SolidWorks Mate-Mate Entity Combinations

The SimMechanics Link utility conditionally supports the following SolidWorks mate-mate entity combinations. The combinations are grouped by mate according to the structure:

Mate Entity1 — Mate — Mate Entity2

Note A combination listed here might be supported, depending on the overall geometry of the assembly. A combination not listed here is not supported.

If the SimMechanics Link exporter cannot translate a mate-mate entity combination into a supported SimMechanics joint with DoFs, it converts the combination into a weld (W) primitive.

Coincident Mate

The paired entities indicated with a plus (+) sign are supported when linked with a Coincident mate.

Mate Entity2	Point	Line	Plane	Cylinder	Cone	Circle/Arc
Mate Entity1						
Point	+					
Line		+	+			
Plane		+	+			+
Cylinder				+	+	+
Cone				+	+	+
Circle/Arc			+	+	+	+

Concentric Mate

The paired entities indicated with a plus sign (+) are supported when linked with a Concentric mate.

Mate Entity2	Point	Line	Plane	Cylinder	Cone	Circle/Arc
Mate Entity1						
Point						
Line				+	+	+
Plane			+			
Cylinder		+		+	+	+
Cone		+		+	+	+
Circle/Arc		+		+	+	+

Perpendicular Mate

These entity pairs are supported when linked with a Perpendicular mate:

- Line — Line

- Plane — Plane
- Line — Plane or Plane — Line

Parallel Mate

These entity pairs are supported when linked with a Parallel mate:

- Line — Line
- Plane — Plane
- Cylinder — Cylinder
- Cone — Cone
- Line — Plane or Plane — Line

Distance Mate

These entity pairs are supported when linked with a Distance mate:

- Point — Point
- Plane — Plane
- Line — Plane or Plane — Line
- Plane — Point or Point — Plane

Angle Mate

These entity pairs are supported when linked with an Angle mate:

- Line — Line
- Plane — Plane

Supported SimMechanics Joints

The SimMechanics Link utility supports the following SimMechanics primitives and primitive combinations. The weld (W) is the default target primitive if the exporter is unable to convert a mate–mate entity combination into a moving joint.

Primitive Combination	Description
P	Prismatic
PP	Planar: P and P are perpendicular
PPP	Custom
PPPR	Custom
S	Spherical
R-S	Revolute-spherical massless connector
R	Revolute
PR	Cylindrical: P and R are parallel
PPR	In-plane: R is perpendicular to PP
PPPS	Six-DoF
R-R	Revolute-revolute massless connector
S-S	Spherical-spherical massless connector
W	Weld: Rigid with no DoFs

Tips for Specific Mates

- You form a spherical primitive with a Point-Point Coincident mate.
- A Point-Point Distance mate is translated to a spherical-spherical massless connector joint. For the translation to succeed, this mate should be the only mate between the two parts so constrained.

Configuring SimMechanics Link Settings in SolidWorks

In this section...

“About the SimMechanics Link Settings Dialog Box” on page 2-10

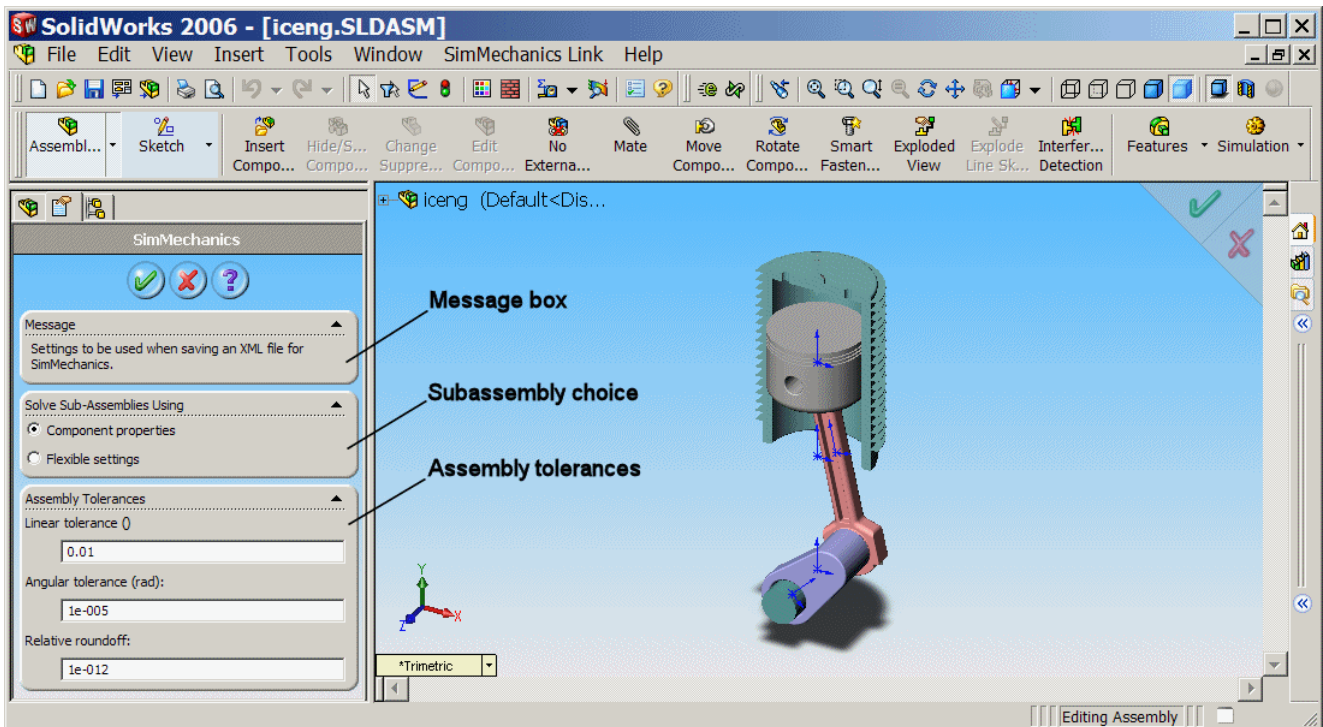
“Reviewing, Changing, and Applying Export Settings” on page 2-10

About the SimMechanics Link Settings Dialog Box

You can access certain export settings from the **SimMechanics Link** menu in your SolidWorks menu bar. Select **Settings** to open the settings dialog box.

Reviewing, Changing, and Applying Export Settings

The settings dialog box contains three active areas.



Message Box

The **Message** box is a description of the dialog box.

Subassemblies Menu

The **Solve Sub-Assemblies Using** menu allows you to choose between rigid and flexible subassemblies.

In SolidWorks, assemblies can have subassemblies, and these in turn can have sub-subassemblies, and so on. When you select a subassembly in the SolidWorks feature tree:

- 1** Right-click the subassembly and select the **Component** menu, then **Properties**. An option appears that allows you to make the component **Flexible** or **Rigid**.
- 2** Select **Rigid** (the default) if you are not interested in the motion of the parts within the subassembly. The subassembly is treated as a single rigid part and is constrained to move as a single body.
- 3** Select **Flexible** if you are interested in the motion of the parts within the subassembly. Its children (parts and mates within the subassembly) are treated as dynamically active and can move relative to one another according to their mates.
 - If you select **Component** properties (the default) in the settings dialog box, then the add-in respects the internal SolidWorks settings, that is, **Rigid** or **Flexible**, accordingly as each subassembly is configured.
 - If you choose **Flexible** settings in the settings dialog box, however, then the add-in assumes that all subassemblies are flexible regardless of the SolidWorks setting. Selecting this option does not change the CAD subassembly settings in the CAD assembly file.

Assembly Tolerances Subdialog Box




The **Assembly Tolerances** subdialog box allows you to specify linear and angular tolerances as well as relative roundoff.

When the add-in creates joints from SolidWorks mates, it checks for vector alignments and spacing. Instead of comparing these alignments and spacings with zero, it compares them with the **Linear tolerance** and **Angular**

tolerance that you specify in the dialog box. The units are meters (m) and radians (rad), respectively.

The **Relative roundoff tolerance** specifies the smallest significant relative numerical difference.

Save, Close, and Help Buttons

Click...	To...
	Save and close the settings dialog box
	Close the settings dialog box without saving your settings
	Open online SimMechanics Link help

Exporting Assemblies to Physical Modeling XML Format

In this section...
“Export Procedure” on page 2-13
“CAD Assembly Export Errors” on page 2-14

Export Procedure

To export a valid assembly, ensure that all the assembly’s parts are fully resolved first, with all model data loaded into memory. No part should be lightweight. Refer to the SolidWorks documentation for additional details.

To export a currently open assembly as Physical Modeling XML:

- 1 From the **File** menu, select **Save As**, and then select **SimMechanics Link (*.xml)**.

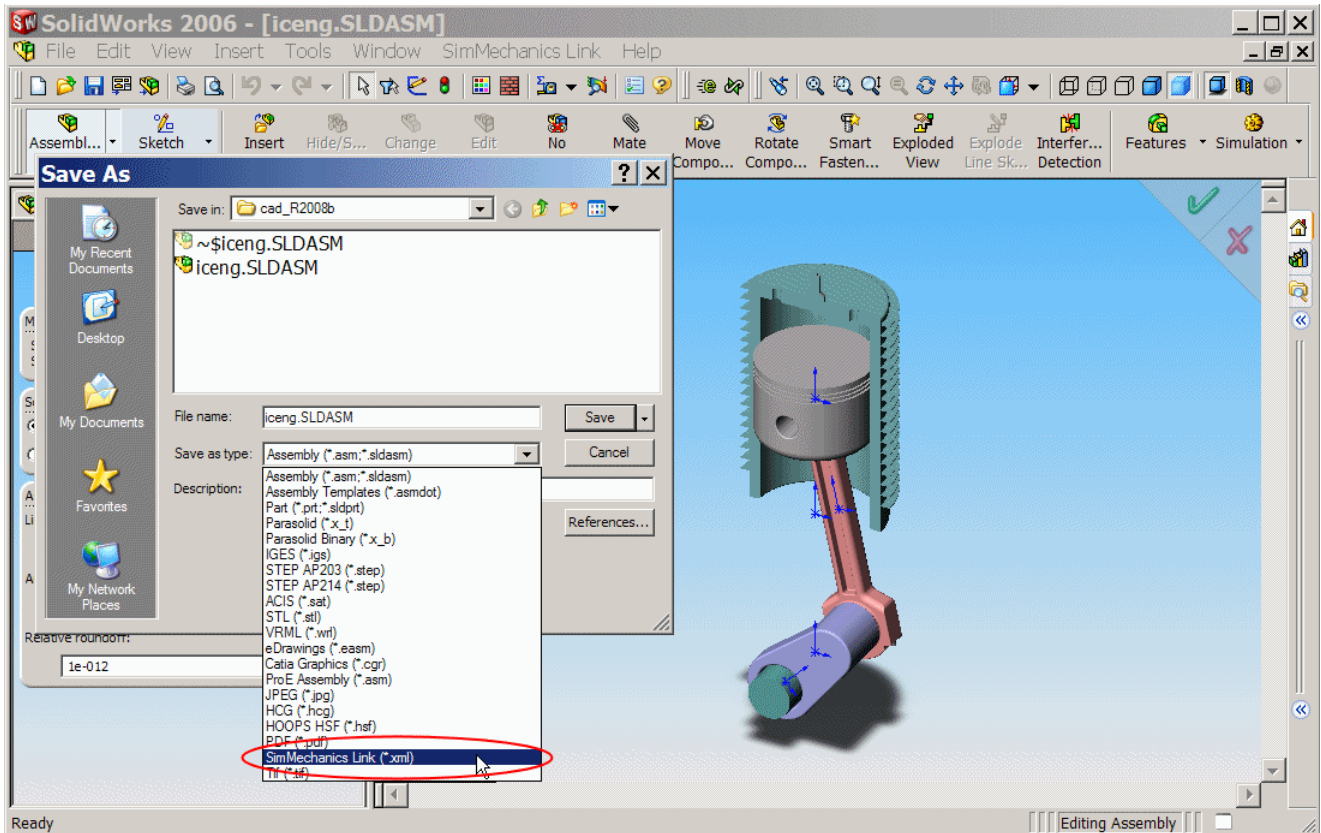
The file’s default name is *filename.xml*, where *filename* is the name of the original CAD assembly. You can change the name of the file at this point, as well as the directory.

- 2 Click **OK**. If no CAD export errors occur, a dialog box opens to indicate that the export is complete and the name and location of the XML file.

You can use the exported XML file to import as a SimMechanics model or give to another user to import.

STL Files Exported for Each Body

For each body in the CAD assembly, the exporter exports a stereolithographic (STL) file with geometric information about the body’s shape.



CAD Assembly Export Errors

If CAD export errors occur:

- The errors are written to a separate error log file.
- The XML file is still exported, and models that you generate from this file are valid. However, these models do not represent the original CAD assembly.
- An error dialog box opens listing the CAD mates that could not be properly translated into joints. The error dialog box also indicates the name and location of the exported XML file and the error log file.

For supported mates and joints information, see “Supported SolidWorks Mates and Their Corresponding Joints” on page 2-5.

STL File Export Errors

If the exporter cannot export an STL file for one or more of the CAD assembly’s bodies, the error dialog box displays a warning indicating for which bodies an STL file could not be created.

Getting Help and Examples

In this section...
“CAD Examples Based on SolidWorks” on page 2-16
“Getting SimMechanics Link Help and Examples from SolidWorks” on page 2-16

CAD Examples Based on SolidWorks

CAD assembly example files are located in the SimMechanics Link demos area of your MATLAB installation.

You can also view demos by opening the MATLAB Demos browser and locating the SimMechanics Link entry.

Getting SimMechanics Link Help and Examples from SolidWorks

You can open the MATLAB Help browser to the SimMechanics Link documentation or demos. From the SolidWorks **SimMechanics Link** menu:

- Select **Help** for documentation.
- Select **Demo** for demos.

See “**SimMechanics Link** Menu in SolidWorks” on page 2-4.

Command Reference

smlink_linksw

Purpose	Register SimMechanics Link software as SolidWorks add-in
Syntax	smlink_linksw
Synopsis	smlink_linksw registers SimMechanics Link software as an add-in to SolidWorks. It receives no input arguments.
Description	You use this command to link SimMechanics Link software to SolidWorks. You need to register SimMechanics Link software with SolidWorks before you can use it to export CAD assemblies from SolidWorks.

Caution

To use this command, you must already have SolidWorks installed on your system, which must be using the Windows operating system.

If you have multiple installations of SimMechanics Link software, you can link only one of these installations at a time to SolidWorks.

If you have multiple installations of SolidWorks, registering SimMechanics Link software with SolidWorks registers it to all of your SolidWorks installations at the same time.

Outputs If registration succeeds, you see a message indicating that the linking has worked, with the location of the add-in module.

If registration fails, you see an error message describing the failure.

See Also smlink_unlinksw

Purpose Unregister SimMechanics Link software as SolidWorks add-in

Syntax smlink_unlinksw

Synopsis smlink_unlinksw unregisters SimMechanics Link software as an add-in to SolidWorks. It receives no input arguments.

Description You use this command to unlink SimMechanics Link software from SolidWorks.

Caution

To use this command, you must already have SolidWorks installed on your system, which must be using the Windows operating system.

SimMechanics Link software must also have already been registered with SolidWorks on your system.

If you have multiple installations of SolidWorks, unregistering SimMechanics Link software from SolidWorks unregisters it from all of your installations of SolidWorks at once.

Outputs If unlinking succeeds, you see a message indicating that the unlinking has worked, with the location of the add-in module.

If unlinking fails, you see an error message describing the failure.

See Also smlink_linksw

smlink_unlinksw

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