ABAQUS 2023
SOLIDWORKS ASSOCIATIVE INTERFACE USER'S GUIDE

Contents

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SOLIDWORKS Associative Interface User's Guide

This guide provides information for system administrators on how to install and configure the components of the SOLIDWORKS Associative Interface. The guide also provides information for end users on how to transfer models from SOLIDWORKS to Abaqus/CAE.

This guide is intended for use with SOLIDWORKS Associative Interface Version 2.8. The content was updated in April 2023.

What's New

This page describes recent changes in the SOLIDWORKS Associative Interface.

Abaqus 2023 FD03 (PF.2323)

Support for SOLIDWORKS 2023

This guide is intended for use with Release 2.8 of the SOLIDWORKS Associative Interface add-in.

Release 2.8 of the SOLIDWORKS Associative Interface supports SOLIDWORKS 2023 and supports Abaqus/CAE 2023 or later releases.

To check the release number of the add-in you are using, see *Determining the release number of the SOLIDWORKS Associative Interface add-in*.

Benefits: The updated compatibility improves the overall workflow.

For more information, see *System requirements for the SOLIDWORKS Associative Interface*.

SOLIDWORKS Associative Interface: overview

The SOLIDWORKS Associative Interface allows you to transfer the geometry of a SOLIDWORKS model to Abaqus/CAE.

In this section:

About the SOLIDWORKS Associative Interface

The SOLIDWORKS Associative Interface allows you to transfer the geometry of a SOLIDWORKS model to Abaqus/CAE.

The SOLIDWORKS Associative Interface consists of the following two components:

- The SOLIDWORKS Associative Interface add-in. The add-in is a plug-in for SOLIDWORKS that allows you to save a model in SOLIDWORKS as an assembly (`.eaf`) file. The SOLIDWORKS Associative Interface add-in is delivered on the assembly media (see the SIMULIA Installation Guide for more information). For details on additional delivery methods, search for "How to obtain the Abaqus/CAE Associative Interface" in the **Search our Knowledge** field at *http://www.3ds.com/support*.
- The Abaqus/CAE CAD Connection toolset. The CAD Connection toolset, which is enabled with a SOLIDWORKS Associative Interface license, allows you to import an assembly file that is generated by the SOLIDWORKS Associative Interface add-in. The SOLIDWORKS Associative Interface license is available as an add-on product from your local SIMULIA office.

What is associative import?

Associative import allows you to easily transfer a part or assembly from SOLIDWORKS to Abaqus/CAE. You can subsequently modify the model in SOLIDWORKS and propagate these modifications to Abaqus/CAE without losing any analysis features assigned to the model in Abaqus/CAE.

The SOLIDWORKS Associative Interface is useful when you are iterating on the design of a model in SOLIDWORKS based on the results of analyses conducted in Abaqus. *Figure 1* shows the connection between SOLIDWORKS and Abaqus/CAE using associative import.

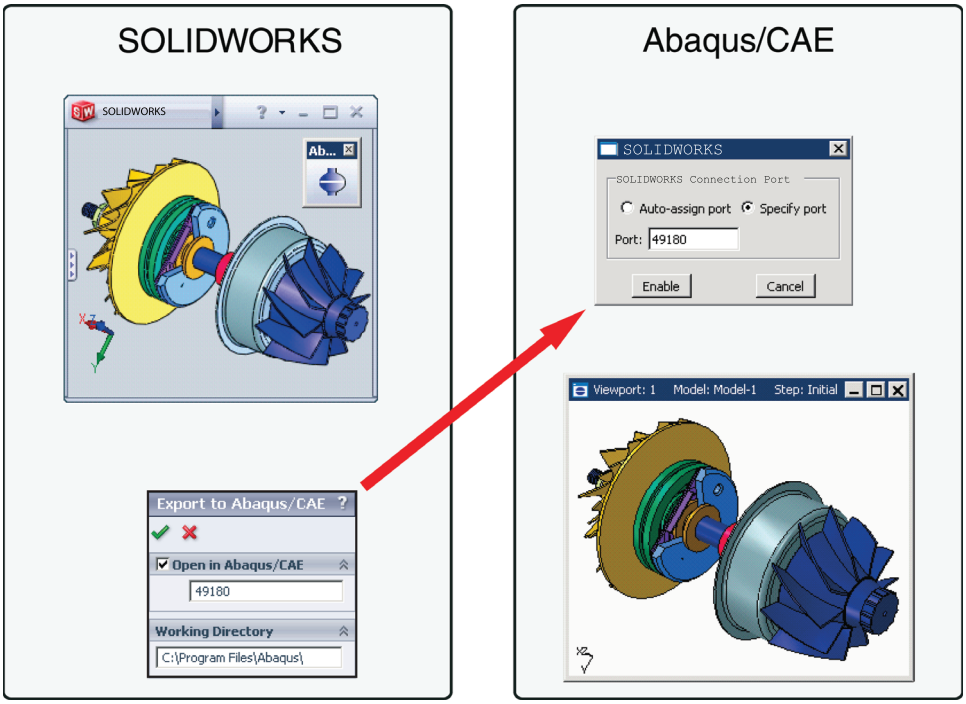

Figure 1: Using associative import to export a model from SOLIDWORKS (left) to Abaqus/CAE (right).

When you use associative import to transfer a model from SOLIDWORKS to Abaqus/CAE, the model appears in the current Abaqus/CAE viewport. The parts and assembly from SOLIDWORKS are stored in the Abaqus/CAE model database and appear in the Model Tree. You can use SOLIDWORKS to modify the parts or to change the position of components in the assembly; when the modifications are complete and the model is ready to be analyzed, perform another associative import. Every time you import the model into Abaqus/CAE, Abaqus/CAE updates the current viewport and the Model Tree to reflect the changes.

In addition, associative import retains any features that you added to the model with Abaqus/CAE. Any of the features that you created in Abaqus/CAE—such as partitions, loads, boundary conditions, sets, and surfaces—are regenerated each time you import the modified model from SOLIDWORKS to Abaqus/CAE. For example, if you partitioned a cell with Abaqus/CAE, the partition is regenerated each time you import the model from SOLIDWORKS into Abaqus/CAE. However, the regeneration of features may fail if the changes that you made with SOLIDWORKS change the topology of the model in ways that fail to fully support the creation of those features.

When you import an assembly, Abaqus/CAE creates parts that correspond to the imported part instances and maintains the relationship between the parts and their instances. Abaqus/CAE does not support subassemblies; if the SOLIDWORKS model includes subassemblies, Abaqus/CAE imports the complete model as a single assembly. In addition, features defined at the assembly level in SOLIDWORKS (such as merges or splits) are not imported into the Abaqus/CAE model.

In most cases Abaqus/CAE retains the names of the parts from the SOLIDWORKS model.

Using the SOLIDWORKS Associative Interface

The SOLIDWORKS Associative Interface offers two associative import methods: automatic associative import and manual associative import.

Automatic associative import

Automatic associative import transfers the geometry of a SOLIDWORKS model to Abaqus/CAE in a single step. Sessions of SOLIDWORKS and Abaqus/CAE must be running simultaneously on the same computer. Using the CAD Connection toolset, you establish a link between Abaqus/CAE and SOLIDWORKS. As soon as an assembly file is created by the SOLIDWORKS Associative Interface add-in, the assembly is imported automatically into Abaqus/CAE. After the model has been transferred, you can continue to make design modifications in SOLIDWORKS and propagate these modifications to the Abaqus/CAE model with a single mouse click. For detailed instructions on performing an automatic associative import, see *Using automatic associative import*.

Manual associative import

Manual associative import is a multistep procedure that can be used to perform an associative import when SOLIDWORKS and Abaqus/CAE are running on different computers or on different platforms. Using the SOLIDWORKS Associative Interface add-in, you can save an assembly (`.eaf`) file to a specified location. Using the CAD Connection toolset, you can import the geometry into Abaqus/CAE or use the assembly file to update an existing model in Abaqus/CAE at a later time. For detailed instructions on performing a manual associative import, see *Using manual associative import*.

System requirements for the SOLIDWORKS Associative Interface

The components of the SOLIDWORKS Associative Interface are supported on the platforms indicated in the table below.

Table 1: Supported platforms for the SOLIDWORKS Associative Interface.

Platform	SOLIDWORKS Associative Interface add-in	Abaqus/CAE CAD Connection toolset
Windows/x86-64	√	√
Linux/x86-64*		√
* The manual associative import method must be used when Abaqus/CAE is running on a Linux platform.		

Release 2.8 of the SOLIDWORKS Associative Interface add-in can be installed with all SOLIDWORKS releases beginning with SOLIDWORKS 2023. For more information about installing the SOLIDWORKS Associative Interface add-in in SOLIDWORKS, see *Installing the SOLIDWORKS Associative Interface add-in*.

The CAD Connection toolset for Abaqus/CAE supports the SOLIDWORKS Associative Interface in Abaqus 6.8 or later.

For the latest information about which versions of SOLIDWORKS and Abaqus/CAE are supported by the SOLIDWORKS Associative Interface, refer to the **Support** page at *www.3ds.com/simulia*.

Units

SOLIDWORKS allows you to change the displayed units of measure in a model; if you change the displayed units, all measurements and parameters in the model are converted and updated to reflect the new units. Abaqus/CAE has no units built into it.

When you use associative import to transfer a model from SOLIDWORKS, Abaqus/CAE imports the displayed numerical measurements without regard for the displayed units.

For example, consider a model created in SOLIDWORKS of a cube that measures 30 millimeters on a side (see *Figure 1*). If the displayed units system in SOLIDWORKS is **MMGS** (millimeter, gram, second), Abaqus/CAE imports a cube with a side length of 30. If the displayed units system in SOLIDWORKS is **CGS** (centimeter, gram, second), Abaqus/CAE imports a cube with a side length of 3.

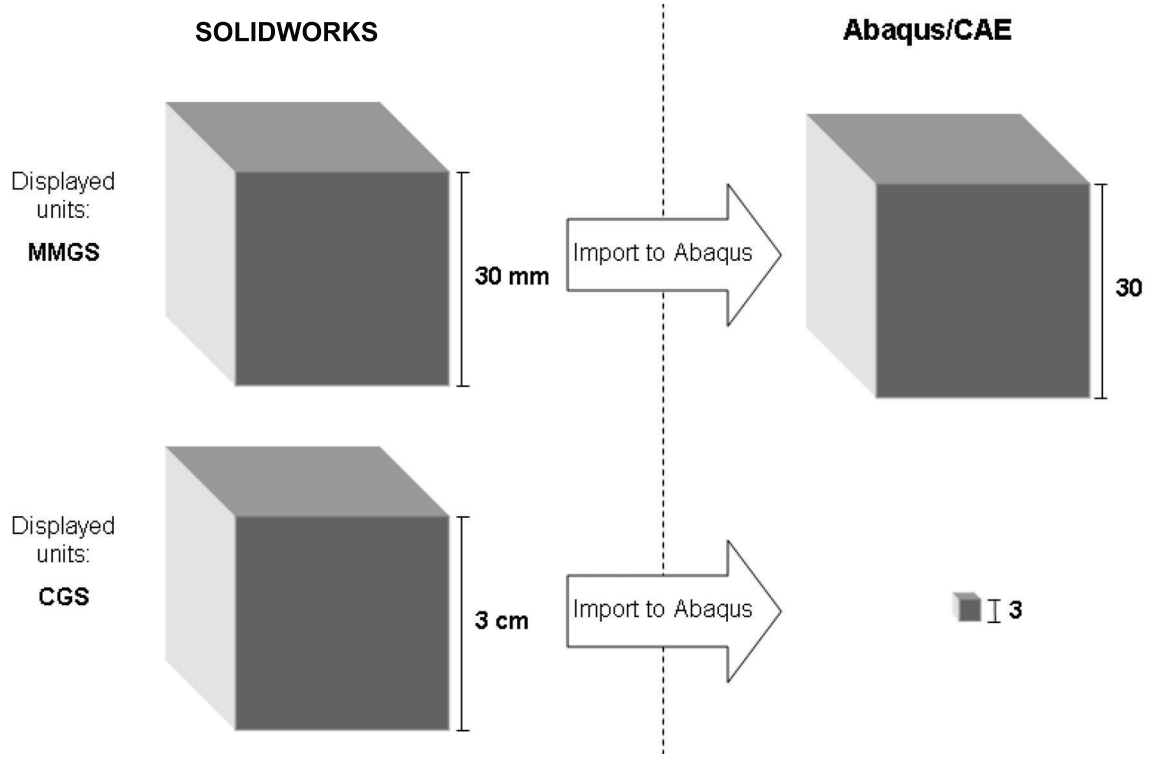

Figure 1: Abaqus/CAE ignores the displayed units when importing models from SOLIDWORKS.

The individual parts in an assembly should use the same units system. When you import an assembly with mixed units into Abaqus/CAE, all of the parts are scaled to a consistent set of units before the import. However, if you subsequently update the Abaqus/CAE assembly by importing an individual part, the units for that part are not scaled and may be inconsistent with the existing dimensions in Abaqus/CAE.

Note: Some model assemblies that are imported using Release 1.1 of the SOLIDWORKS Associative Interface may employ inconsistent units when positioning the assembly components in Abaqus/CAE. To resolve this problem, rewrite the assembly (`.eaf`) file for the model from SOLIDWORKS using Release 1.2 or higher of the SOLIDWORKS Associative Interface add-in, and import the new assembly file into Abaqus/CAE.

Installing the SOLIDWORKS Associative Interface add-in

The SOLIDWORKS Associative Interface add-in is a plug-in component for SOLIDWORKS. The add-in must be enabled after it is installed.

In this section:

Installing the SOLIDWORKS Associative Interface add-in

The following procedure describes how to install the SOLIDWORKS Associative Interface plug-in on Windows systems.

Before you begin:

You need administrative privileges to install the plug-in. SOLIDWORKS must be installed and licensed on your system before installing the plug-in. If you are installing the SOLIDWORKS Associative Interface on an operating system that uses User Access Control (UAC), such as Windows Vista or Windows 7, you must turn off UAC or select the least secure setting.

1. Obtain the SOLIDWORKS Associative Interface zip file from the assembly media (see the SIMULIA Installation Guide for more information).
2. Unzip the SOLIDWORKS Associative Interface zip file and save the contents to a permanent directory; the directory must be accessible to all users of the SOLIDWORKS Associative Interface. The zip file contains the add-in `.dll` file.
3. Start SOLIDWORKS as a Windows administrator in one of the following ways:
   - For instructions on installing the SOLIDWORKS Associative Interface from the command line, see “Installing or removing SIMULIA Abaqus/CAE Associative Interface for SolidWorks via command line,” which can be found by entering the title in the **Search our Knowledge** field at *http://www.3ds.com/support*.
   - From the **Start** menu, **Shift +** right-click **SOLIDWORKS** and select **Run as administrator** from the context menu.
   - From the desktop, **Shift +** right-click **SOLIDWORKS** and select **Run as administrator** from the context menu.

   You must be part of the local administrator group to open SOLIDWORKS as an administrator.
4. Select **File**->**Open** from the main menu bar.

   The **Open** dialog box appears.
5. In the **Files of type** field, select **Add-Ins (*.dll)**.
6. Navigate to the directory that contains the add-in file. Select the add-in `.dll` file, and click **Open**.

   SOLIDWORKS installs the SOLIDWORKS Associative Interface add-in.

Enabling the SOLIDWORKS Associative Interface add-in

The following procedure describes how to enable the SOLIDWORKS Associative Interface add-in in SOLIDWORKS after you have installed it.

1. From the SOLIDWORKS main menu, select **Tools**->**Add-Ins**.

   SOLIDWORKS displays the **Add-Ins** dialog box.

2. In the **Add-Ins** dialog box, the SOLIDWORKS Associative Interface add-in is listed as **Abaqus/CAE** underneath the **Other Add-ins** heading (see *Figure 1*).

   - To enable the SOLIDWORKS Associative Interface add-in in the current SOLIDWORKS session, toggle on the check box in the **Active Add-ins** column.
   - To enable the SOLIDWORKS Associative Interface add-in for future SOLIDWORKS sessions, toggle on the check box in the **Start Up** column.

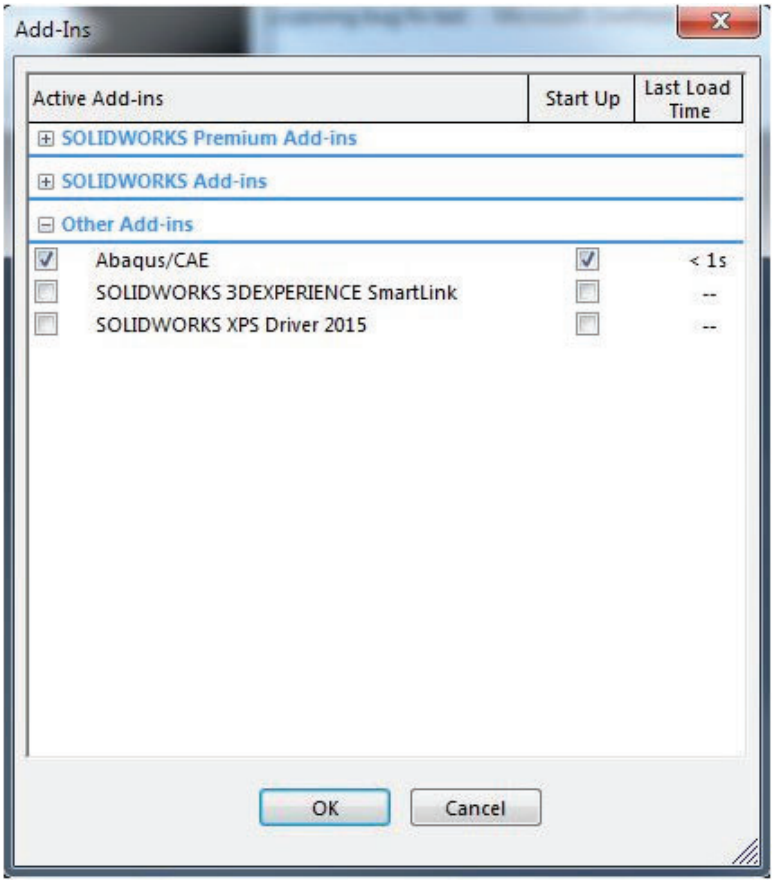

Figure 1: The Add-Ins dialog box.

3. Click **OK**.

   If you toggled on **Active Add-ins** for the SOLIDWORKS Associative Interface add-in, **Abaqus** appears in the **Tools** menu in the SOLIDWORKS main menu bar, and the **Export to Abaqus/CAE** icon appears in the SOLIDWORKS toolbar.

4. To disable the SOLIDWORKS Associative Interface add-in, follow the above steps and toggle off the appropriate check boxes.

Determining the release number of the SOLIDWORKS Associative Interface add-in

Use the procedure below to determine which release of the SOLIDWORKS Associative Interface add-in you are using.

This information may be necessary if you are contacting SIMULIA for troubleshooting issues related to the SOLIDWORKS Associative Interface.

1. From the SOLIDWORKS main menu, select **Tools**->**Abaqus**->**Export to Abaqus/CAE**.

   Export to Abaqus/CAE appears in the SOLIDWORKS PropertyManager.

2. In the PropertyManager, click the question mark  .

   The dialog box that appears contains trademark information and the release number for the SOLIDWORKS Associative Interface add-in.

Uninstalling the SOLIDWORKS Associative Interface add-in

The following procedure describes how to uninstall the SOLIDWORKS Associative Interface add-in.

To uninstall the SOLIDWORKS Associative Interface add-in, run the following command from a command prompt:

```
regsvr32 /c /u full_path_to_dll_file
```

where *full_path_to_dll_file* is the location of the saved add-in `.dll` file on your computer. Solidworks session need to be closed for successful uninstallation of plug-in using this command.

The SOLIDWORKS Associative Interface add-in is disabled in SOLIDWORKS, and the **Abaqus/CAE** item is removed from the **Add-Ins** dialog box.

Note: On Windows Vista, User Account Control (UAC) must be turned off for the administrator account and the user account before uninstalling the SOLIDWORKS Associative Interface add-in.

Importing a part or assembly from SOLIDWORKS to Abaqus/CAE

Associative import through the SOLIDWORKS Associative Interface allows you to transfer model geometry and subsequent design changes from SOLIDWORKS to Abaqus/CAE without losing any analysis features that are defined in Abaqus/CAE.

Associative import can be performed automatically or manually; for an overview of the two import techniques, see *Using the SOLIDWORKS Associative Interface*.

In this section:

Using automatic associative import

You can use automatic associative import to import an assembly from SOLIDWORKS to Abaqus/CAE.

Follow the instructions below to import an assembly from SOLIDWORKS to Abaqus/CAE using automatic associative import.

1. Start Abaqus/CAE, and enter the Assembly module.
2. From the main menu bar, select **Tools**->**CAD Interfaces**->**SOLIDWORKS**.
3. From the **SOLIDWORKS** dialog box that appears, choose **Auto-assign port** and click **Enable**.

   Abaqus/CAE displays the port number that it assigned in the message area. (If desired, you can click **Specify port** and enter the port number.)
4. Start SOLIDWORKS, and load the model to be exported.

   Abaqus appears in the **Tools** menu in the SOLIDWORKS main menu bar. If **Abaqus** does not appear in the **Tools** menu, refer to *Installing the SOLIDWORKS Associative Interface add-in*.
5. If the model includes any new or modified parts, save the SOLIDWORKS model. Current SOLIDWORKS part (. `SLDPRT`) files must exist for each part in the assembly before using the SOLIDWORKS Associative Interface.
6. Select **Tools**->**Abaqus**->**Export to Abaqus/CAE** from the SOLIDWORKS main menu.

   The **Export to Abaqus/CAE** options appear in the SOLIDWORKS PropertyManager.
7. In the SOLIDWORKS PropertyManager, specify the following information:
   a. Toggle on **Open in Abaqus/CAE** (see *Figure 1*).

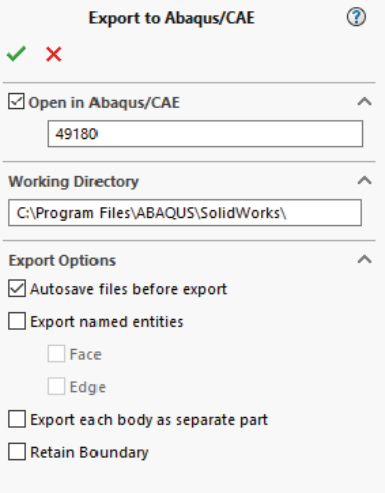

Figure 1: Use automatic associative import to transfer the model directly to Abaqus/CAE.

   b. If necessary, change the default number in the **Open in Abaqus/CAE** field so that it matches the port number displayed by Abaqus/CAE when you enabled the CAD interface.
   c. The SOLIDWORKS Associative Interface saves necessary files (the assembly file and part geometry files) to a working directory during the import. If necessary, change the path in the **Working Directory** field to save these files in a different location.
   d. By default, the SOLIDWORKS Associative Interface automatically saves the current version of the SOLIDWORKS model as part of the export process. To prevent the SOLIDWORKS files from

being saved automatically, toggle off **Autosave files before export**. If you toggle off this option, you must manually save the SOLIDWORKS model before exporting it to Abaqus/CAE.

e. Toggle on **Export named entities** if you want to create sets and surfaces that correspond to a named face or edge. By default, this option is toggled off.

f. The SOLIDWORKS Associative Interface provides the option to export a SOLIDWORKS Multibody Part as separate parts. By default, **Export each body as separate part** is toggled off.

g. The **Retain Boundary** option allows you to retain the intersecting boundaries in Abaqus/CAE. By default, this option is toggled off.

8. Click the green check mark in the SOLIDWORKS PropertyManager.

The model appears in the current Abaqus/CAE viewport. If an assembly is open in SOLIDWORKS, the parts are imported into Abaqus/CAE and an assembly of part instances is created automatically. If a single part is open in SOLIDWORKS, only the part is imported into Abaqus/CAE. You can use the parts and the assembly in an Abaqus/CAE analysis; for example, you can apply sections to the parts, and you can apply loads and boundary conditions to the assembly.

9. If you modify the model in SOLIDWORKS, click the **Export to Abaqus/CAE** icon to update the model in Abaqus/CAE. The modified model is imported immediately into Abaqus/CAE using the settings previously specified in the **Export to Abaqus/CAE** dialog box. You can also select **Tools**->**Abaqus**->**Export to Abaqus/CAE** from the SOLIDWORKS main menu to change the settings and import the updated model.

Note: If you toggled off the **Autosave files before export** in the **Export to Abaqus/CAE** dialog box, you must manually save the modified SOLIDWORKS model before exporting it to Abaqus/CAE.

Abaqus/CAE imports only the modified parts. Any changes that you make to the position of components in the SOLIDWORKS assembly will also be exported to Abaqus/CAE. Any components that you suppress or unsuppress in SOLIDWORKS are suppressed or resumed similarly in Abaqus/CAE. If you hide a component in SOLIDWORKS, the visibility of that part instance in Abaqus/CAE is turned off; use the **Assembly Display Options** in Abaqus/CAE to control the visibility of part instances (for more information, see "Controlling instance visibility" in the Abaqus/CAE User's Guide).

Features (such as loads and sets) and geometric modifications that you created in Abaqus/CAE are updated to account for the changes to the SOLIDWORKS parts. Abaqus/CAE will be unable to regenerate some features if the topology of the part has changed significantly.

10. From the Abaqus/CAE main menu bar, select **Tools**->**CAD Interfaces**->**SOLIDWORKS** to show the port number that is being used or to disable the connection with SOLIDWORKS. For more information, see "Creating a CAD connection" in the Abaqus/CAE User's Guide.

Using manual associative import

You can use manual associative import to export an assembly from SOLIDWORKS to Abaqus/CAE.

Follow the instructions below to import an assembly from SOLIDWORKS to Abaqus/CAE using manual associative import.

1. Start SOLIDWORKS, and load the model to be exported.

   Abaqus appears in the **Tools** menu in the SOLIDWORKS main menu bar. If **Abaqus** does not appear in the**Tools** menu, refer to *Installing the SOLIDWORKS Associative Interface add-in*.

2. If the model includes any new or modified parts, save the SOLIDWORKS model. Current SOLIDWORKS part (. `SLDPRT`) files must exist for each part in the assembly before using the SOLIDWORKS Associative Interface.
3. Select **Tools**->**Abaqus**->**Export to Abaqus/CAE** from the SOLIDWORKS main menu.

   The **Export to Abaqus/CAE** options appear in the SOLIDWORKS PropertyManager.

4. In the SOLIDWORKS PropertyManager, specify the following information:
   a. Toggle off **Open in Abaqus/CAE** (see *Figure 1*).

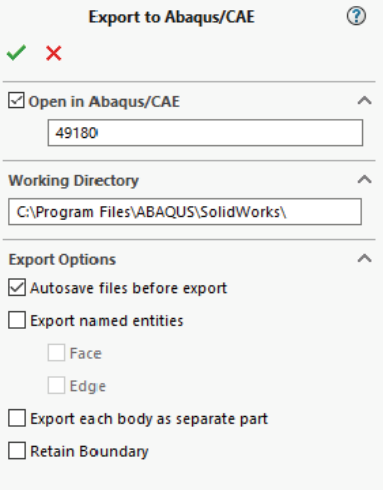

Figure 1: Use manual associative import to save an assembly file.

   b. In the **Working Directory** field, specify a path and file name to which the assembly file will be saved.
   c. By default, the SOLIDWORKS Associative Interface automatically saves the current version of the SOLIDWORKS model before it creates the assembly file. To prevent the SOLIDWORKS files from being saved automatically, toggle off **Autosave files before export**. If you toggle off this option, you must manually save the SOLIDWORKS model before creating the assembly file.
   d. Toggle on **Export named entities** if you want to create sets and surfaces that correspond to a named face or edge. By default, this option is toggled off.
   e. The SOLIDWORKS Associative Interface provides the option to export a SOLIDWORKS Multibody Part as separate parts. By default, **Export each body as separate part** is toggled off.
   f. The **Retain Boundary** option allows you to retain the intersecting boundaries in Abaqus/CAE. By default, this option is toggled off.

5. Click the green check mark in the SOLIDWORKS PropertyManager.

SOLIDWORKS saves the assembly (`.eaf`) file and any necessary part files in ACIS (`.sat`) format. You can now end your SOLIDWORKS session.

6. If necessary, copy the assembly file and all of the ACIS part files to the computer where you will be running Abaqus/CAE.
7. Start Abaqus/CAE, and select **File**->**Import**->**Assembly** from the main menu bar.

   The **Import Assembly** dialog box appears.
8. From the **File Filter** menu at the bottom of the **Import Assembly** dialog box, select **Assembly File (*.eaf*)**.
9. Select the assembly file that was written in Step 5, and click **OK**.
10. From the **Import Assembly from EAF File** dialog box that appears, select the assembly or individual part instances to import, and click **OK**.

    The selected part instances appear in the current Abaqus/CAE viewport. You can use the parts and the assembly in an Abaqus/CAE analysis; for example, you can apply sections to the parts, and you can apply loads and boundary conditions to the assembly.

    For more information about importing assembly files, see "Importing an assembly from an Elysium assembly file" in the Abaqus/CAE User's Guide.
11. If you modify the model in SOLIDWORKS, use the following steps to propagate the changes to the model in Abaqus/CAE:
    a. Repeat Steps 1–5 to create an updated assembly file. The name of the assembly file can change between imports; however, the names of the parts and components in the SOLIDWORKS model must remain the same.
    b. If necessary, copy the assembly file and any ACIS part files that have been modified since the last import to the computer where you are running Abaqus/CAE.
    c. Open the existing model in the current viewport of Abaqus/CAE.
    d. Import the updated assembly file using the procedure in Steps 7–10.

       Abaqus/CAE imports only the modified parts. Any changes that you made to the position of components in the SOLIDWORKS assembly will also be imported to Abaqus/CAE. Any components that you suppressed or unsuppressed in SOLIDWORKS are suppressed or resumed similarly in Abaqus/CAE. If you hid a component in SOLIDWORKS, the visibility of that part instance in Abaqus/CAE is turned off; use the **Assembly Display Options** in Abaqus/CAE to control the visibility of part instances (for more information, see "Controlling instance visibility" in the Abaqus/CAE User's Guide).

       Features (such as loads and sets) and geometric modifications that you created in Abaqus/CAE are updated to account for the changes to the SOLIDWORKS parts. Abaqus/CAE will be unable to regenerate some features if the topology of the part has changed significantly.

Using the SOLIDWORKS Associative Interface scripting interface

You can use the scripting interface for the SOLIDWORKS Associative Interface to iterate on the design of your model.

For example, you can attempt to minimize the weight of your design by running a script that decreases the thickness of a region until Abaqus calculates that a maximum stress level is reached. The scripting interface for the SOLIDWORKS Associative Interface utilizes the SOLIDWORKS VBA macro scripting interface to update the SOLIDWORKS model in Abaqus/CAE and to save SOLIDWORKS parts in assembly (`.eaf`) file format. The scripting interface provides the following four methods:

SetPortNumber(*port*)

You use the `SetPortNumber` method to assign the port that will be used by the CAD Connection toolset. *port* is a long integer specifying the port number.

SetWorkingDirectory(*directory*)

You use the `SetWorkingDirectory` method to specify where the SOLIDWORKS Associative Interface saves files (the part geometry files) during the import. *directory* is a string specifying the path to the working directory.

SetAutoSave(*save*)

You use the `SetAutoSave` method to automatically save the current version of the SOLIDWORKS model in assembly (`.eaf`) file format as part of the export process. *save* is a long integer specifying whether to save the current version of the SOLIDWORKS model. If *save*=0, the model is not saved; if *save*=1, the model is saved.

ExportToCAE(*export*)

You use the `ExportToCAE` method to update the model in Abaqus/CAE. *export* is a long integer specifying whether to update the model. If *export*=0, the model is not updated; if *auto*=1, the model is updated.

Before you run a SOLIDWORKS Associative Interface script for the first time, you must do the following:

1. Start SOLIDWORKS, and select **Tools**->**Macro**->**Edit** from the main menu bar.
2. Open a file containing a script (`.swp` file).
3. From the main menu bar, select **Tools**->**References**.
4. Verify that the following libraries (`.tlb` file) are registered:
   - Visual Basic For Applications
   - SOLIDWORKS Extensibility Type Library
   - OLE Automation
   - SOLIDWORKS *version* Constant Type Library
   - SwAddinTolAnalyst 1.0 Type Library
   - Sw2AbqPlugin 1.0 Type Library
   - SOLIDWORKS *version* Commands Type Library
   - SOLIDWORKS *version* Exposed Type Libraries for add-in use

The following example illustrates the four methods that are provided in the SOLIDWORKS Associative Interface scripting interface. The example updates a SOLIDWORKS part in Abaqus/CAE.

```
Dim mySAIPlugin As ISw2AbqPlugin
Set mySAIPlugin = swApp.GetAddInObject("Sw2AbqPlugin.Sw2AbqPlugin")
Dim port As Long
port = 45001
mySAIPlugin.SetPortNumber(port)
Dim workDir As String
workDir = "C:\wdir\SAI_TMP\"
mySAIPlugin.SetWorkingDirectory(workDir)
Dim auto As Long
auto = 1
mySAIPlugin.SetAutoSave(auto)
mySAIPlugin.ExportToCAE(auto)
```

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