Chapter 8: Configuring CAD Products

The connection, geometry interface, and other CAD functionality for all supported CAD products is included with the Ansys release media. Select the CAD systems you want to install during the installation process. If you choose not to install CAD systems during the installation, you can install them at a later date.

To run the connection or geometry interface functionality, you need to:

1. Ensure that the CAD product is correctly installed and licensed.

2. Ensure that you have the correct Ansys, Inc. license(s).

3. Run the installation setup.

For complete information about the files you can import and the platforms supported by the connection capability, see the *Mechanical APDL Connection User's Guide*. For more information about geometry interface information in Ansys Workbench, see the topics **Attach to Active CAD Geometry** and **Import External Geometry File** in the Ansys DesignModeler help.

Caution:

Be sure to install Mechanical APDL and the connection functionality from the same release. If you attempt to run the latest connection functionality on a machine that is running an earlier release of Mechanical APDL, or vice versa, the connection may fail.

The connection for Creo Parametric requires you to run Mechanical APDL, Creo Parametric and the connection for Creo Parametric on the same machine. The connection for NX requires you to run Mechanical APDL, NX, and the connection for NX on the same machine. The connections for CATIA, SAT, and Parasolid do not require any additional CAD installation.

For CAD installation and configuration troubleshooting, see Installation Troubleshooting - CAD Packages (p. 96).

8.1. Using the Product & CAD Configuration Manager

The **Product & CAD Configuration Manager** allows you to configure geometry interfaces for Mechanical APDL and Ansys Workbench. CAD configuration is typically handled during the product installation; however, if you chose to skip those steps, or if you make changes to your local CAD configuration between releases (for example, you install, move, or update your CAD package, or remove it entirely), you can use this utility.

Configuring Geometry Interfaces for the First Time

1. From Windows, right-click the **Start > Ansys 2023 R2> Product & CAD Configuration 2023 R2**.

2.   From the **Product & CAD Configuration Manager**, click **Configure Products & CAD**.

3.   From the product selection screen, verify that the Ansys Geometry Interfaces selection is enabled and click **Next** to continue the configuration.

4.   You are asked to select one of the following two options:

   • Selecting the **Automatically detect my CAD installation and configure it** option and clicking **Next** determines if each CAD system is installed and configured on your computer. The installation detects the CAD installs and displays the geometry interfaces that are configured. Click **Next** to continue the automatic configuration. If the CAD is installed but not configured, this option sets the configuration to appropriate associative interface. If the CAD is not installed, the configuration is set to reader. When the automatic configuration is complete, click **Exit**.

   • Selecting the **I will manually make my selections** option and clicking **Next** walks you through a several manual configuration steps. If you select this option, you can configure the appropriate geometry interface properties by following the steps described below.

5.   After selecting the **I will manually make my selections** option and clicking **Next** (bullet point #2 above), you are presented with a check list of the geometry interfaces. You can manually choose which geometry interfaces you would like to configure by enabling the appropriate check box(es).

   When you have selected the appropriate interfaces, click **Next**.

6.   The next screen the provides you with the option of selecting the reader or associative interface for each enabled geometry interface.

   Configure each geometry interface option and click **Next**.

7.   If you selected the associative interface for **Creo Parametric** or **NX**, a third screen is displayed. In this case follow the instructions described in this step. If you did not select **Creo Parametric** or **NX** the installation continues as described in the step #8 below.

   Creo Parametric

   You may need to specify the Creo Parametric command, and the full Creo Parametric installation path (`C:\Program Files\PTC\Creo 7.0.0.0\Parametric` by default) for an existing Creo Parametric installation.

   NX

   If you choose the associative interface and the UGII environment variables were not set, you may need to specify the NX installation path for an existing NX installation. If you are an administrative user, a file required to load the NX plug-in is placed in the administrative user's Application Data folder by default, which may not be accessible by other, non-administrative users. To allow non-administrative users to run, you will need to define the environment variable **UGII_CUSTOM_DIRECTORY_FILE** prior to performing Product & CAD Configuration Manager functions and specify a location where other users have read access.

   Once you have completed the **Creo Parametric** or **NX** configuration, click **Next**.

8.   When all of your CAD products have been successfully configured, click **Exit**.

Administrative users can review the log file produced by the most recent **Product & CAD Configuration Manager** in Ansys Inc\v232\CADConfigLogs\Latest\CADConfigurationMgr.log, while the non-admin user configuration record is stored in %TEMP%\Ansys\v232\CADConfigLogs\Latest. Historical configurations are maintained in the corresponding 232\<Date>\CADConfigurationM-gr_<Time>.log. Any errors are recorded in CADConfigurationMgr.err located immediately in CADConfigLogs folder. For administrators a log capturing product uninstall is stored in %TEMP%\Ansys\v193\CADConfigLogs\UninstallFinalLog.

> **Important:**
>
> If you change your source from reader to associative plug-in or vice-versa (Creo Parametric, NX, SOLIDWORKS, Autodesk Inventor, SolidEdge, AutoCAD or CATIA V5), you must first un-configure the existing source and then configure using the new source.

8.1.1. Unconfiguring

If you need to unconfigure any of your CAD products, follow the steps above, but choose **Unconfigure Products & CAD** from the **Product & CAD Configuration Manager**.

8.1.2. Running the CAD Configuration Manager in Batch Mode

You can configure Ansys Geometry Interfaces by supplying the CAD Configuration Manager with arguments specific to the CAD sources you want to make available. The following table contains a list of supported arguments.

The command to run the CAD Configuration Manager in batch mode on Windows is:

```
"<installpath>\commonfiles\CAD\bin\winx64\Ans.CadInt.CADConfigurationUtility.exe"
    -arguments
```

Argument	Value	Comment
UNCONFIGURE_SPECIFIED	None	Results in any specified CAD sources being unconfigured. When this flag is absent, the CAD Configuration Manager will attempt to configure all designated CAD sources.
UNCONFIGURE	None	Results in all CAD sources being disabled along with prerequisite libraries.
Either: PE_CONFIG_WB or PE_CONFIG_WBSPATIAL	None	Specify Creo Parametric source as either the associative plug-in or the reader (Spatial, no Creo Parametric install required). When the plug-in is specified, additional arguments PROELOADPOINT and PROE_START_CMD are required.
PROELOADPOINT	Full path to Creo Parametric installation (quotes required on Windows).	Not required with unconfigure operation.
PROE_START_CMD	Full path to command used to	Not required with unconfigure operation.

Argument	Value	Comment
	launch Creo Parametric (quotes required on Windows)	
Either: UG_CONFIG_WB or UG_CONFIG_WBSPATIAL	None	Specify NX source as either the associative plug-in or the reader (Spatial, no NX install required). When the plug-in is specified, the argument UGII_BASE_DIR must also be specified.
UGII_BASE_DIR	Full path to NX installation	This should agree with environment variable **UGII_BASE_DIR**. Not required with unconfigure operation.
Either: CATIA5_READER or CATIA_CAPRI	None	Specify CATIA V5 source as either the Reader (Spatial, no CATIA install required) or CAPRI (CADNexus install required).
OSDM_CONFIG	None	Configure/unconfigure Creo Elements/Direct Modeling.
Either: INVENTOR_CONFIG or INVENTOR_CONFIG_WBSPATIAL	None	Specify Inventor source as either the associative plug-in or the reader (Spatial, no Inventor install required).
Either: SOLIDWORKS_CONFIG or SW_CONFIG_WBSPATIAL	None	Specify SOLIDWORKS source as either the associative plug-in or the reader (Spatial, no SOLIDWORKS install required).
Either: SOLIDEDGE_CONFIG or SOLIDEDGE_CONFIG_SPATIAL	None	Specify Solid Edge source as either the associative plug-in or the reader (Spatial, no Solid Edge install required).
Either: ACAD_CONFIG or ACAD_CONFIG_SPATIAL	None	Specify AutoCAD source as either the plug-in or the reader (Spatial, no AutoCAD install required).
JTOPEN_CONFIG	None	Configure/unconfigure JT.
CATIA4_CONFIG	None	Configure/unconfigure CATIA v4.
CATIA6_CONFIG	None	Configure/unconfigure CATIA v6.

Note:

All arguments require a dash (-) before them in order to be properly recognized by the CAD Configuration Manager. Arguments' values should not have a dash preceding them.

For example, you can configure Creo Parametric and SOLIDWORKS Geometry Interfaces to Ansys Workbench from the command line by using the following:

```
"<installpath>\commonfiles\CAD\bin\winx64\Ans.CadInt.CADConfigurationUtility.exe"
-SW_CONFIG -PE_CONFIG_WB -PROELOADPOINT "C:\Program Files\PTC\Creo 7.0.0.0\Parametric"
-PROE_START_CMD "C:\Program Files\PTC\Creo 2.0\Parametric\bin\parametric.bat"
```

where `installpath` is the same as the value of environment variable **%AWP_ROOT232%**, `platform` is the value of environment variable **%ANSYS_SYSDIR%**, and Creo Parametric is installed to `C:\Program Files\PTC\Creo 7.0.0.0\Parametric`.

To unconfigure the same CAD Interfaces, the command would be:

```
"<installpath>\commonfiles\CAD\bin\winx64\Ans.CadInt.CADConfigurationUtility.exe"
-unconfigure_specified -SW_CONFIG -PE_CONFIG_WB
```

Although the argument order does not matter, an argument value must immediately follow its argument.

8.1.3. Network Considerations

When Workbench is installed to a network location (made available from a mapped network drive), any configuration actions will make the selected geometry interfaces available/unavailable only to the user who is currently logged in. To make changes that apply to all users, the configure/unconfigure action must be run as an administrator logged into the system hosting the Workbench installation. Configuration actions performed as an administrator on this server machine will impact all users on all machines referencing that install, except for users who have overridden the server configuration by configuring for themselves.

For loading associative geometry interfaces into their respective CAD environments, when the Workbench installation is referenced from a network location:

• A local administrator's configuration changes will set up associative CAD interfaces to load into the CAD environment of all local users, when that scope is selected (applicable to AutoCAD, Inventor, SOLIDWORKS, Solid Edge, Creo Elements/Direct and NX).

• The associative geometry interface for Creo Parametric will load into the CAD environment of all users when the CAD installation is writable for that administrative user. Otherwise, it will load only into the CAD environment of the administrator who is currently logged in, and each additional user must be configured individually.

CAD Readers for JT, Creo Parametric, Inventor, NX, CATIAV5, CATIAV6, AutoCAD, Solid Edge and SOLIDWORKS may not fully unconfigure on a client of a network installation, even when this action is attempted by a client's administrator. An administrative user on the server must execute an unconfigure action, using Product & CAD Configuration Manager, in order for these interfaces to be fully deactivated on the client.

There is a limitation in the **Product and CAD Configuration Manager** which prevents you from configuring a different geometry interface type (other than the server specified type) on a network install's client system. To address this issue, run the CAD Configuration Manager on the client system and change the geometry interface to the desired type (Reader vs. Associative).

8.1.4. Uninstalling

> **Warning:**
>
> Do not proceed with an uninstall for CAD-specific unconfigure actions or you will leave some Ansys Workbench products unusable.

8.2. Performing Non-Administrator CAD Configuration

In some scenarios, a non-administrative user may need to make configuration changes to the installed external CAD applications that can be modified. Generally, the external CAD applications that can be modified by a non-administrative user are those that do not require a change to the system registry.

The external CAD applications that cannot be configured by non-administrative users are:

• Solid Edge (associative)

• SolidWorks (associative)

• Creo Elements/Direct

Configuring Non-Administrator CAD Options

1. From Windows, right-click the **Start > Ansys 2023 R2> Product & CAD Configuration 2023 R2**.

2. When the **Product & CAD Configuration Manager** opens, click **Configure Products & CAD**.

3. If available, the next screen the provides you with the option of selecting the reader or associative interface for each installed geometry interface that a non-administrative user can modify.

   Configure each geometry interface option and click **Next**.

4. If you selected the associative interface for **Creo Parametric** or **NX**, an additional screen is displayed. In this case follow the steps below. If you did not select **Creo Parametric** or **NX** the installation continues as described in step #7.

5. **Creo Parametric**

   You may need to specify the Creo Parametric start command, and the full Creo Parametric install-ation path (`C:\Program Files\PTC\Creo 6.0.0.0\Parametric` by default) for an existing Creo Parametric installation.

6. **NX**

   If you choose the associative interface and the UGII environment variables were not set, you may need to specify the NX installation path for an existing NX installation and the NX Custom Directory File Path.

   After entering the information for **Creo Parametric** and/or **NX**, click **Next**.

7. Once you have completed updating all configuration options, or if no selectable CAD options are installed, the **Summary** page displays which CADs are being configured. Click **Next** to start the configuration update process.

8. When the configuration process is complete, click **Exit** to close the **Product & CAD Configuration Manager**.

Unconfiguring Non-Administrator CAD Options

1. From Windows, right-click the **Start > Ansys 2023 R2> Product & CAD Configuration 2023 R2**.

2. When the **Product & CAD Configuration Manager** opens, click **Unconfigure Products & CAD**.

3. You are presented with a check list of the geometry interfaces you can unconfigure as a non-administrative user. You can manually choose which geometry interfaces you would like to unconfigure by enabling the appropriate check box(es).

4. When you have selected the appropriate interfaces to unconfigure, click **Next**.

5. The **Summary** page displays which CADs are being unconfigured. Click **Next** to start the unconfiguration process.

6. When the unconfiguration process is complete, click **Exit** to close the **Product & CAD Configuration Manager**.

8.3. Creo Parametric Configuration

Running the **Product & CAD Configuration Manager** for Creo Parametric performs the following steps to activate the Creo Parametric plug-in:

- Sets the environment variable **PROE_START_CMD232** to the file used to launch Creo Parametric (for example `C:\Program Files\PTC\Creo 7.0.0.0\Parametric\bin\parametric.bat`).

- For administrative installations, adds the Ansys 2023 R2 entry to the `config.pro` file located in `<creo_path>\text`. An example for Creo Parametric would be:

```
PROTKDAT C:\Program Files\PTC\Creo 7.0.0.0\Common Files\text
```

- Updates the `WBPlugInPE.dat` file referenced in the `config.pro` file, so that it contains information for loading the WorkBench Plug-In and the Mechanical APDL Connection.

- Registers the Plug-In file `WBPlugInPECOM.dll` referenced in the `WBPlugInPE.dat` file.

- When Workbench is installed on a network location and the local administrator has write permissions to `<creo_path>\text`, a version-specific `WBPlugInPE232.dat` file will be created and placed in that folder. However, when `<creo_path>\text` is unwriteable, both the `config.pro` and the `WBPlugInPE232.dat` will be placed in either `%HOME%` or `%HOMEDRIVE%%HOMEPATH%`.

If you do not have write access to your Ansys Workbench installation, you may encounter the following error when attempting to import Creo Parametric models without an active CAD session:

No write access, please choose another startup directory for trail file creation.

To prevent this issue, add the following line to your `config.pro` file:

```
trail_dir $TEMP
```

WBPlugInPE.dat File Contents

The `WBPlugInPE.dat` file should look like this example:

```
NAME WB232PluginProWF
EXEC_FILE E:\Program Files\ANSYS Inc\V232\AISOL\CADIntegration\ProE\
    winx64\WBPlugInPECOM.dll
```

```
TEXT_DIR E:\Program Files\ANSYS Inc\V232\AISOL\CADIntegration\ProE\
    ProEPages\Language\<locale>
STARTUP dll
delay_start FALSE
allow_stop TRUE
unicode_encoding FALSE
END

NAME ac4pro232dll
exec_path E:\Program Files\ANSYS Inc\V232\ANSYS\ac4\bin\pro\winx64\ac4pro.exe
text_path E:\Program Files\ANSYS Inc\V232\ANSYS\ac4\data\pro\text
STARTUP dll
delay_start FALSE
allow_stop TRUE
unicode_encoding FALSE
unicode_encoding FALSE
revision 24.0
end
```

Do not delete any of these lines. If you modify this file, do NOT enter a carriage return after the END line. The file may be customized with other information. If these lines are deleted, or if the `WBPlugIn-PE.dat` file is not present in any of the directories in the search path, Creo Parametric will not load Ansys-related CAD interfaces. You should typically never have to edit these files for path information contained within them. Paths are determined by environment variable settings, which are set automatically during installation. If you encounter problems when attempting to run Creo Parametric, use the **Product & CAD Configuration Manager** to reconfigure rather than attempting to edit files directly.

Following an update of an existing Creo Parametric 7.0 installation to a different maintenance release , the Ansys Workbench Associative Plug-In will no longer be configured. To reactivate your plug-in, follow the steps described in .

In order for **Product & CAD Configuration Manager** to successfully perform configuration of the Creo Parametric Associative Geometry Interface it is required that the Windows Operating System's decimal separator be the same symbol when running configuration as it was when the original Creo Parametric installation was performed.

8.3.1. Configuring the Connection for Creo Parametric

All Creo Parametric users must have copies of the `WBPlugInPE.dat` file and the `config.anscon.232` (connection for Creo Parametric) files. The `config.anscon.232` file is placed in the `Program Files\ANSYS Inc\V232\ansys\ac4\data\winx64` directory. `config.anscon.232` must be copied into the user's working directory at the time Creo Parametric is started. The `WBPlugInPE.dat` file is placed in the `Program Files\ANSYS Inc\V232\AISOL\CADIntegration\%ANSYS_PROEWF_VER%\ProEPages\config` directory. This file defines the name of the executable, the path to the executable, the path to the message file, and the current revision of Creo Parametric.

> **Note:**
>
> If the ANSGeom menu in Creo Parametric does not appear correctly, copy the `config.anscon.232` file into your working directory and restart Creo Parametric.

If Mechanical APDL Connection fails to load into Creo Parametric 8.0.0.0, see **AnsysMechanical APDL Connection does not load into Creo Parametric 8.0.0.0** (p. 96).

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80   information of ANSYS, Inc. and its subsidiaries and affiliates.

8.3.1.1. The WBPlugInPE.dat File and config.pro File

Creo Parametric uses the `WBPlugInPE.dat` file to locate related executables such as the connection for Creo Parametric.

Mechanical APDL/Ansys Workbench also requires a `config.pro` file.

- The `config.pro` file resides `<path-to-creo-parametric>\..\Common Files\<Creo Ship Code>\text` for Creo Parametric. The `Creo Ship Code` is a three digit number preceded by a single letter. The ship code is specified in the Creo Parametric installation log, which can be found in `<path-to-creo-parametric>\bin\pim\xml\creobase.xml`. In the cases that the crebase.xml file is missing, the ship code can be determined by looking in the Windows Registry using regedit. For Creo 7.0.0.0 the value is located in HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\PTC\PTC Creo Parametric\7.0.0.0\<DateCode>. <Datecode> is an integer reflecting the date of the Creo build.

The `config.pro` file contains a PROTKDAT line that points to the `WBPlugInPE.dat` file. This file is generated either during the product install or by running the **Product & CAD Configuration Manager** to configure the Mechanical APDL /Ansys Workbench products for Creo Parametric.

The *<proe_platform>* variable is the name that Creo Parametric gives to its platform directories:

WBPlugInPECOM

Hardware Platform	Ansys Platform `<platform>`	Creo Parametric Platform `<proe_platform>`
Win 7 x64	winx64	x86e_win64

8.3.1.2. The config.pro File

A typical `config.pro` file might look like the following example. This example has an entry for Ansys Workbench. You may have other Creo Parametric specific customizations.

```
PROTKDAT E:\Program Files\ANSYS Inc\V232\AISOL\CADIntegration\$ANSYS_PROEWF_VER\
    ProEPages\config\WBPlugInPE.dat
```

8.3.1.3. The config.anscon.232 File

Users who launch Mechanical APDL from Creo Parametric will need the information from the `config.anscon.232` file. This file is installed for all users in the `\ac4\data\winx64` subdirectory. You typically do not need to edit this file. Here is a sample `config.anscon.232` file:

```
ANSYS_CMD %AWP_ROOT232%\ANSYS\bin\%ANSYS_SYSDIR%\ansys232.exe
ANSYS_GRAPHIC_DEVICE WIN32
ANSYS_SOLVER Sparse**
ANSYS_SELECTED_LAYERS 1-256**
ANSYS_GEOMETRY_TYPE Solids Only**
ANSYS_NEUTRAL_FORMAT Yes**
ANSYS_PRODUCT_NAME ANE3FL
```

**These variables are not supported by Creo Parametric and are ignored by Creo Parametric.

See *Setting Mechanical APDL Configuration Parameters* in the *Connection User's Guide* for more information about the `config.anscon.232` file.

8.3.2. Creo Parametric Environment Variables

Most environment variables are set during product installation. In general, you will not need to reset these variables. Check the manuals for the individual CAD products for information about environment variables and their settings.

PROE_START_CMD232

Specifies the name of the `.bat` file which launches Creo Parametric on the system. The path to the executable should be already set if the Creo Parametric installation path has been defined for the **PATH** environment setting.

PROE_START_CMD232 Default = `C:\Program Files\PTC\Creo 7.0.0.0\Parametric\bin\parametric.bat`

8.4. NX Configuration

Running the **Product & CAD Configuration Manager** for NX performs the following steps to activate the NX plug-in:

- If not already existing, sets the environment variable **UGII_CUSTOM_DIRECTORY_FILE** to `%APPDATA%\ANSYS\v232\Unigraphics\custom_dirs.dat`.

- If not already present, adds an entry to the `custom_dirs.dat` file specifying the location of the Plug-In, for example, `C:\Program Files\ANSYS Inc\V232\AISOL\CADIntegration\UnigraphicsNX\winx64`.

- Registers the Plug-In file (`DSPlugInUGCOM.dll`). For the previous example, these files would be located in `C:\Program Files\ANSYS Inc\V232\AISOL\CADIntegration\UnigraphicsNX\winx64\startup`.

8.4.1. Configuring the Connection for NX

User-Defined Environment Variables

You will need to have the following environment variable set if you will be running the connection for NX product from inside NX.

UGII_BASE_DIR

The environment variable **UGII_BASE_DIR** must be defined for proper operation of NX and the Ansys Workbench Plug-In for NX. This environment variable is typically created by the NX installer; however, in rare cases, you may have to create/update it. In these situations, the **NX Installation Location** field will be left blank within the **Product & CAD Configuration Manager**'s NX tab.

When multiple NX versions are installed on a system, the **UGII_BASE_DIR** environment variable must be set to the install path of the NX version to be launched, prior to starting that version of the CAD. Otherwise errors will be encountered during CAD startup and the plug-in will not load.

For example, if switching from NX 11.0 to run NX 12.0, you must update **UGII_BASE_DIR** to the NX 12.0 path before starting NX 12.0.

8.5. Configuring CADNexus/CAPRI CAE Gateway for CATIA V5

The CADNexus/CAPRI CAE Gateway for CATIA V5 product is available for Ansys Workbench. The Ansys Workbench-specific portions are included in the installation. However, to run this product, you must complete the configuration as follows:

1. Install Ansys Workbench according to the instructions in this guide.

2. Install CATIA V5 and DSLS licensing (requires a CATIAV5 license key MD2, HD2 or ME2).

3. From the Ansys customer site, download the CADNexus/CAPRI CAE Gateway for CATIA V5 for your platform to a temporary folder. (Do not download to a folder containing blank spaces in the folder name, for example, Program Files). Follow the download procedures described in Downloading the Installation Files (p. 26).

4. Extract the contents of the zip file using your standard zip utility.

5. Install CADNexus/CAPRI CAE Gateway for CATIA V5 using the appropriate exe for your platform.

6. Follow the instructions on the CAPRI installation screens to complete the installation. When specifying the component to install, you will need to select the desired CatiaV5 release under CAPRI CAD Applications. When asked if CAPRI should set the environment variables, click **Yes**.

7. Run the **Product & CAD Configuration Manager** and select the **Catia V5: CADNexus/CAPRI CAE Gateway** option to complete the configuration as described in Using the Product & CAD Configuration Manager (p. 73).

    > **Note:**
    >
    > Using the **Product & CAD Configuration Manager** to set up the CADNexus/CAPRI CAE Gateway does not require administrative rights, although installing that product (step 5) requires these rights. However, if you want to reconfigure to use the Ansys Workbench Reader for Catia V5, administrative rights are required.

If you want to revert to the standard CATIA V5 interface, run the **Product & CAD Configuration Manager** and select the **Catia V5: Standard Interface** option to complete the configuration as described in Using the Product & CAD Configuration Manager (p. 73).

You cannot run the standard CATIA V5 interface simultaneously with the CADNexus/CAPRI CAE Gateway for CATIA V5 interface.

8.6. Configuring AutoCAD

Running the Product & CAD Configuration Manager for AutoCAD performs the following steps to active the plug-in: