



Install Instructions for Thermal Desktop

This installation process will install Thermal Desktop, which includes SINDA/FLUINT (solver), RadCAD, FloCAD, and TD Direct.

Table of Contents

Before Installing	2
Prerequisite Software	2
Install Build Tools	3
Download Intel Fortran Compiler Classic	3
Install Intel Fortran Compiler	4
Install Intel Fortran Compiler Runtime	5
Download Thermal Desktop Installation Package	5
Install SINDA/FLUINT	6
Install Thermal Desktop	8
Install Ansys License Manager and License File	8
Verify Thermal Desktop Installation and Licensing	9
Supporting Materials	11
Appendix 1: Configure and Start Your Autodesk License Server	12
Appendix 2: Installing Ansys SpaceClaim on the client.....	16

Before Installing

1. Verify your computer meets our [System Requirements](#).
2. Ensure the person performing the installation has administrator rights.
3. Prior to installing, turn off User Account Control through the Windows Control Panel and restart the computer.
4. Verify that AutoCAD 2021 or newer has been installed on the computer. Open it at least once to verify its functionality and that it is licensed properly. **If AutoCAD is not yet installed and you purchased AutoCAD 2024 as part of your Thermal Desktop package, proceed and follow the steps below, installing AutoCAD 2024 as directed.**
 - a. If you are in need of a trial license of AutoCAD, please visit [Autodesk's website](#).
5. In order to use TD Direct, verify that SpaceClaim 2024 R1 is installed on the computer. If SpaceClaim is not installed, download the SpaceClaim package from the [Ansys Customer Portal](#) and install before proceeding. See [Appendix 2](#) for instructions regarding installing and configuring SpaceClaim on client computers.

Prerequisite Software

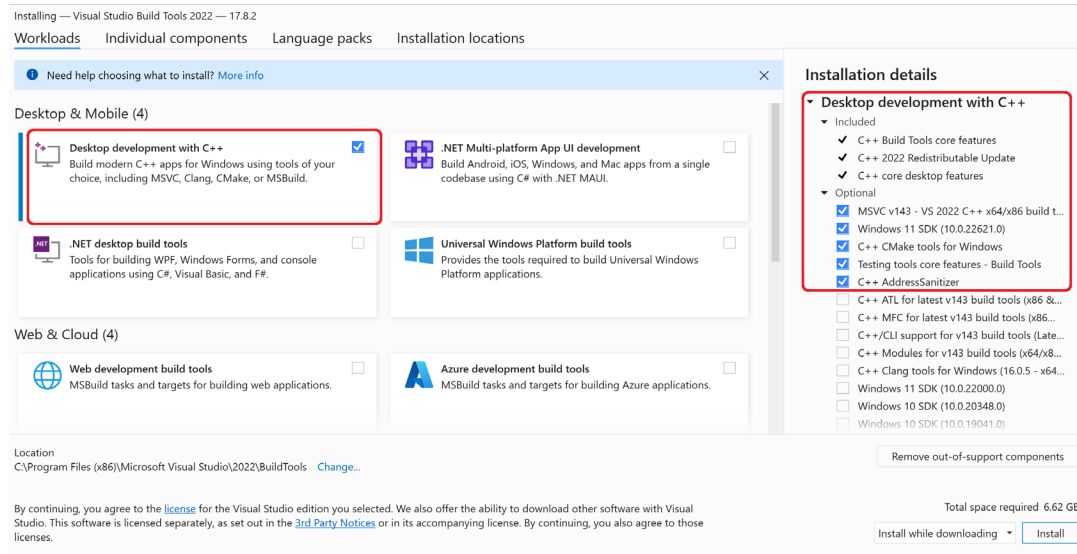
SINDA/FLUINT requires Intel Fortran Compiler Classic to be installed. Intel Fortran Compiler Classic in turn requires Microsoft Build Tools for Visual Studio 2019 or newer.

- If you do not have Build Tools installed, [click here to download it from the Microsoft website](#). This is a direct link to the install download. It will either automatically download or you will see a pop up on your screen asking if you want to save it.

Install Build Tools

You will need to install Build Tools before installing Intel Fortran Compiler Classic.

The default installation of Build Tools does not install support for C++ which is required for CRTech software. During the installation process you will be prompted to select which “Workloads” you wish to install. You must add support for Desktop development with C++ as shown below. Details of this option will appear on the right side of the screen. No additional options need to be selected. Continue with the install of VS Build Tools.



Download Intel Fortran Compiler Classic

Before proceeding with this step, be sure to reboot your computer. Failure to do so could result in the Fortran compiler not integrating properly with Build Tools.

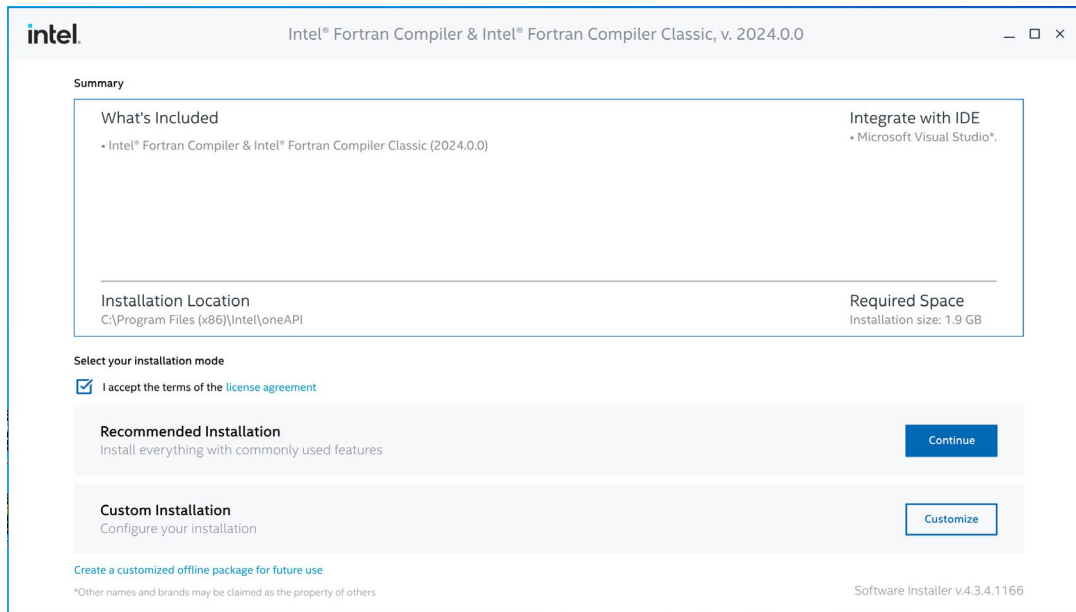
The Intel Fortran compiler can be [downloaded here](#). Be sure to download the version for Windows. Online and offline versions are available, either will work.

2024.0.0 Release

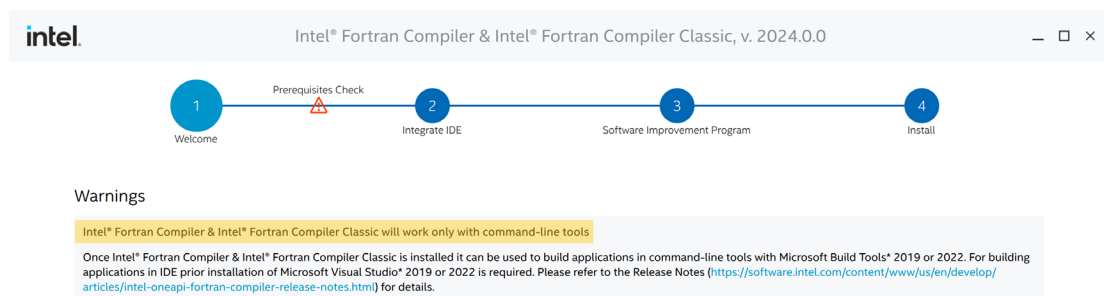
Name (Click to initiate download)	Version	Size	Installer	Date
Intel Fortran Compiler Classic and Intel Fortran Compiler for Linux	2024.0.0	19 MB	Online	Nov. 16, 2023
Intel Fortran Compiler Classic and Intel Fortran Compiler for Linux	2024.0.0	628 MB	Offline	Nov. 16, 2023
Intel Fortran Compiler Classic and Intel Fortran Compiler for Windows*	2024.0.0	14 MB	Online	Nov. 16, 2023
Intel Fortran Compiler Classic and Intel Fortran Compiler for Windows	2024.0.0	470 MB	Offline	Nov. 16, 2023

Install Intel Fortran Compiler

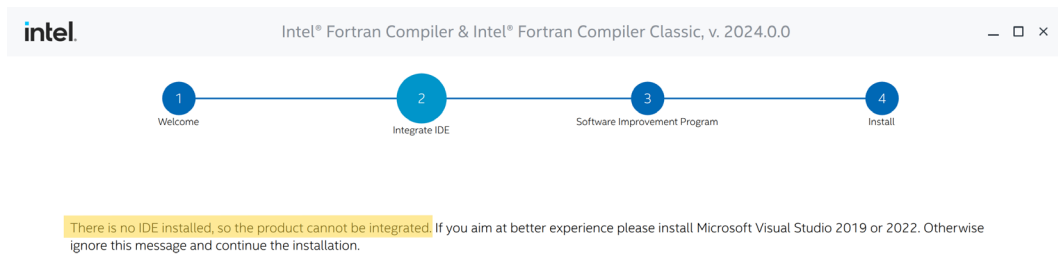
After downloading the installer, run it by double clicking on the install file. Use the recommended installation.



You may see the following warning, which can be ignored. Visual Studio is not required if you have Build Tools installed as previously instructed in this document.



You may see a message stating that “There is no IDE installed, product cannot be integrated”; ignore it as long as you have installed Build Tools.



If you see other warnings, you will need to investigate what the issue is.

After installation of the compiler, edit the Environment variables on the computer. You should have a variable called VS2022INSTALLDIR. It should be set to C:\Program Files (x86)\Microsoft Visual Studio\2022\Build Tools (or whatever you have installed). If this variable does not exist, please create it.

Install Intel Fortran Compiler Runtime

Download and install the Intel® Fortran Compiler Runtime for Windows* 2024.0.0 from the [Intel Fortran Compiler Runtime Library](#)

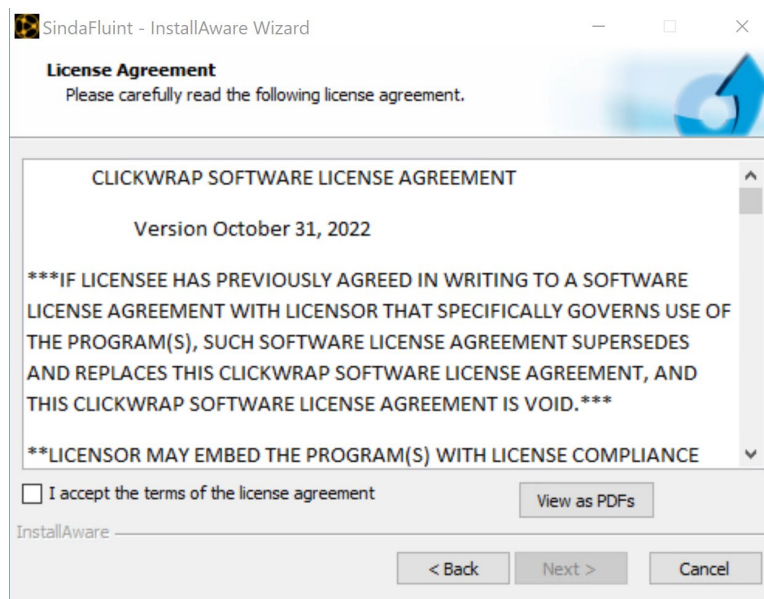
Download Thermal Desktop Installation Package

Download the Thermal Desktop 2024 R1 package from the [Ansys Customer Portal](#). The ZIP file contains the SINDA/FLUINT installer, the Thermal Desktop installer, and the two-part AutoCAD 2024 installer (for use only if AutoCAD was purchased as part of your Thermal Desktop package).

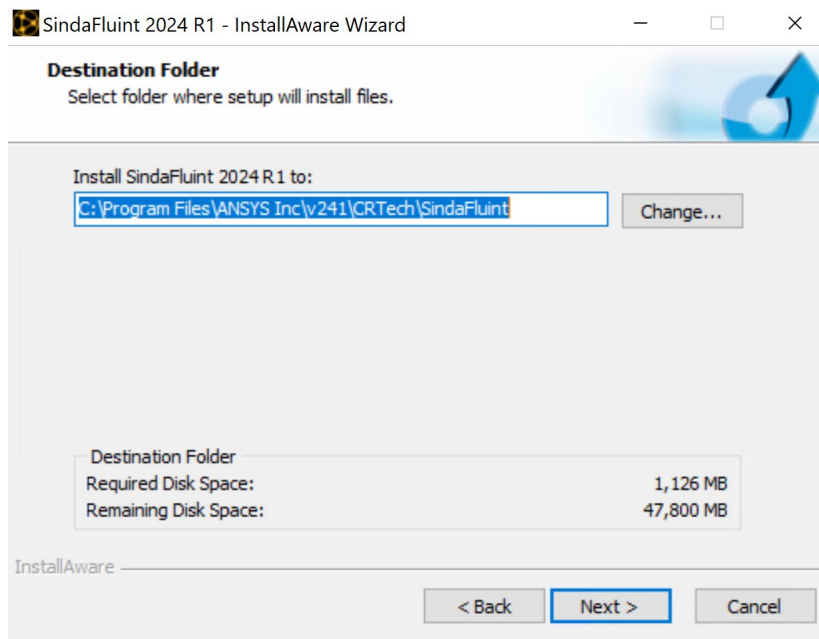
If applicable, install AutoCAD 2024 from this package first, before proceeding with SINDA/FLUINT and Thermal Desktop installation. See [Appendix 1](#) for instructions regarding setting up an Autodesk license server and configuring client computers. AutoCAD license file (.lic) would've been provided by your Ansys Account Manager. Otherwise, ensure that AutoCAD 2021 or newer is already installed before proceeding.

Install SINDA/FLUINT

1. Close any other applications you may have running. Shortly after the SINDA/FLUINT installer starts, it may require a reboot. If this happens, the installer should automatically restart after the computer reboots.
2. Double-click on the SINDA/FLUINT installer to begin installation:
SINDAFLUINT_2024R1_WINX64.exe
3. Review the license agreement. Check the box to accept the agreement and then click “Next.”



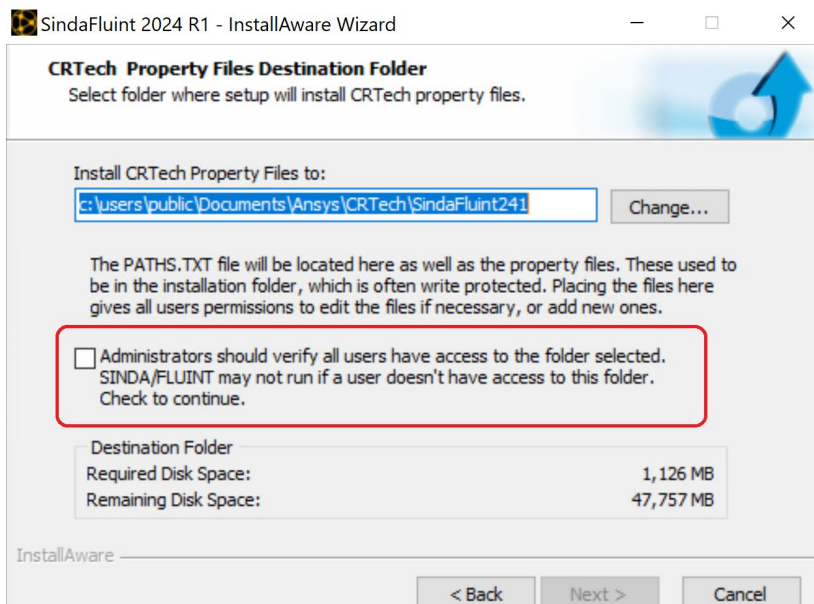
4. The program will prompt you to define the location where SINDA/FLUINT is to be installed.



5. The next screen defines where the fluid property files are to be installed. By default, these files are placed in the following folder, making it accessible to all users:

C:\Users\Public\Public Documents\Ansys\CRTech\SindaFluint

If a different folder is selected during the install, verify it is accessible by all users. Note, this location is stored in the registry, so files should not be moved to a different location after the installation.



6. Continue through the install prompts. When the installer is done, click Finish.

Install Thermal Desktop

1. Close any other applications you may have running.
2. Double click on the Thermal Desktop installer: THERMALDESKTOP_2024R1_WINX64.exe
3. Review the license agreement. Check the box next to "I Accept" and then click "Next".
4. Similar to the SINDA/FLUINT installation, the program will prompt you for directories and paths. The default install directory is:

C:\Program Files\ANSYS Inc\v241\CRTech\Thermal Desktop

And the default location for installing property files is:

C:\Users\Public\Public Documents\Ansys\CRTech\ThermalDesktop

If a different folder is selected during the install, verify it is accessible by all users.

5. The next screen prompts you to define a name for program shortcuts that will point to Thermal Desktop. Confirm your choice, then click Next.
6. When the installer is done, click Finish.

Install Ansys License Manager and License File

NOTE: Users migrating from CRTech licensing to Ansys licensing must delete the CRTECH_LICENSE_FILE Environment Variable before proceeding. To reach the Environment Variables menu, search the term "environment" (or similar) in the Windows taskbar and click on "Edit the system environment variables" menu entry. From the **Advanced** tab of the **System Properties** window that opens, click the "Environment Variables..." button.

NOTE: If you do not already have your Ansys license file (.lic), reach out to your Ansys Account Manager.

1. Download and install Ansys License Manager, available from the [Ansys Customer Portal](#), on the server and all client machines (if you don't already have it installed via installation of other Ansys products).
2. To install, extract ANSYSLICMAN_2024R1_WINX64.zip. Open setup.exe to start the process.
3. Once installation is complete,

On the server:

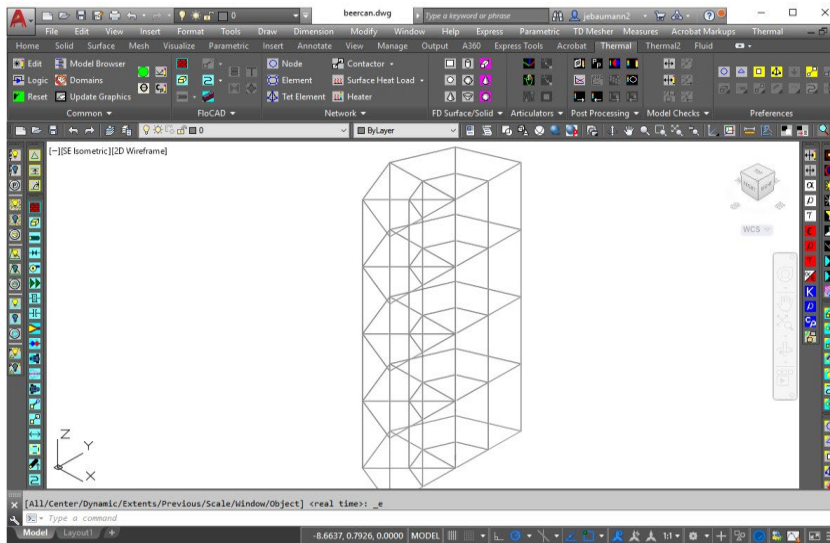
- a. Open **Ansys License Management Center** from the Start Menu and click **Add a License File** under the **License File Options** menu along the left side of the screen.
- b. Navigate to where you saved your Ansys .lic file, then click **Install License File** to upload the license file. The License Manager is automatically started.

On client computers:

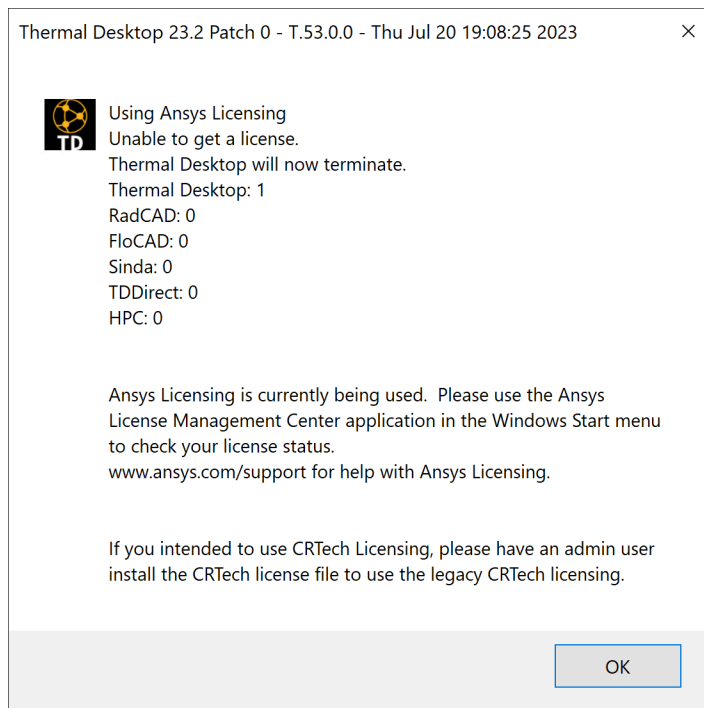
- a. Open the **Ansys Licensing Client Settings** located at:
C:\Program Files\ANSYS Inc\v241\licensingclient\winx64\LicensingSettings.exe,
by right-clicking, then select **Run as Administrator**.
- b. Under the **FlexNet Publisher > License servers** section, flip the toggle to **Enabled**, then enter the Port number and Server Hostname (this information can be found within the .lic file provided by Ansys).
- c. Click **Test** to verify the server connection. A green check mark indicates success. Click **Save**.


Verify Thermal Desktop Installation and Licensing

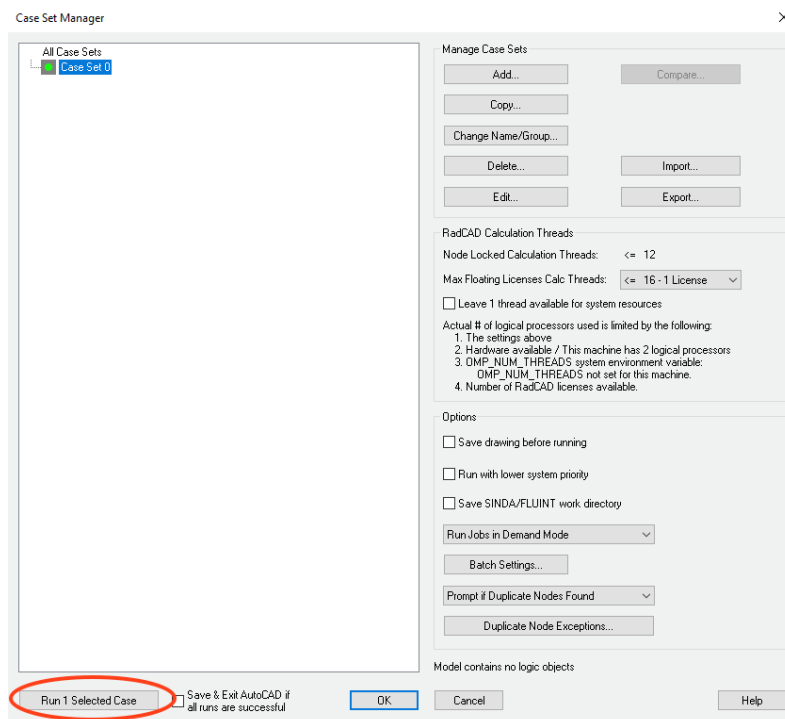
1. Open Windows Explorer and browse to location of the Thermal Desktop install test files (default is C:\Users\Public\Public Documents\Ansys\CRTech\ThermalDesktop).
2. Right click on the file beercan.dwg and select “Open With”, then select “More apps” ↓. Scroll down and click “Look for another app on this PC”. Browse to:
C:\Program Files\ANSYS Inc\v241\CRTech\Thermal Desktop\TdDwgLauncher
Check the box next to “Always use this app to open .dwg files” and click OK.
3. The test model should open in AutoCAD and will appear similar to the graphic below. Depending on your exact version of AutoCAD and Thermal Desktop, the ribbons and toolbars may vary. If you see the half cylinder, continue to the next step.



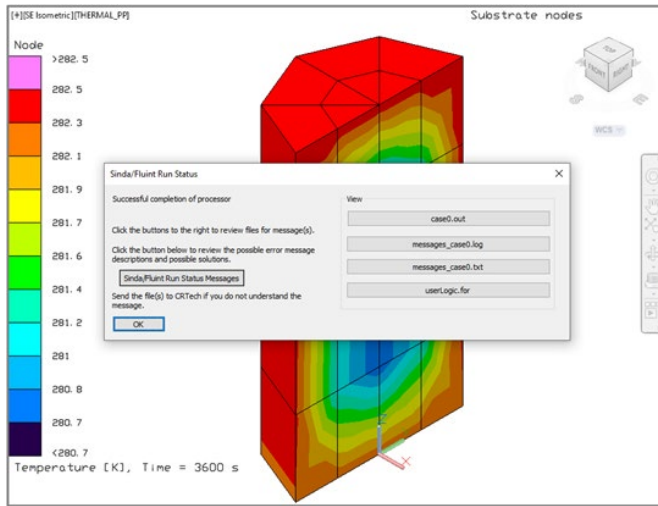
4. If you do not see any error messages, skip to step 6.
5. If you see the error message indicating “Unable to get a license” similar to the graphic shown below, you may not have installed the license properly. Use the **Ansys License Management Center** to configure and check your license status.



6. If the model file opened, click on the green circle icon  (Thermal > Case Set Manager) to open the Case Set Manager dialog shown below. Click on “Case Set 0” in the left panel (it should already be highlighted), then click on the Run 1 Selected Case button in the lower left corner of the window.



- After you click the Run button, a SINDA/FLUINT Processor Status window will appear on the screen followed by the message box (shown below) stating “Successful completion of processor.” If you see this window, Thermal Desktop and SINDA/FLUINT are installed and working properly.



Supporting Materials

View our online [Getting Started Guide](#) for additional learning resources.

Appendix 1: Configure and Start Your Autodesk License Server

Install Autodesk Network License Manager (NLM)

Install NLM on each server that you plan to use for setting up your license server.

1. Download [NLM for Windows](#).
2. Double-click the downloaded file to start the installation wizard.

Note: By default, NLM tools are installed in C:\Autodesk\Network License Manager.

Create a Debug Log File

Debug log files contain Network License Manager (NLM) history that you can review to troubleshoot problems with your license servers.

1. Create a subfolder named Logs under the folder in which you installed NLM.
2. Use a text editor to create a text file in the Logs subfolder.
3. Name the text file anything you'd like, just be sure to change the filename extension from *.txt* to *.log*.

Note: Be sure that users and services have write access to the debug log file or folder.

Save the License File

Save the Autodesk license file (.lic) that you received from Ansys in the folder where you installed the Network License Manager. For example:

- *C:\Autodesk\Network License Manager\Licenses*

Configure the License Server

1. Open the LMTOOLS utility. Click Start > Autodesk > LMTOOLS Utility.
2. From the **Service/License File** tab, verify that the **Configuration Using Services** option is selected.
3. Select the checkbox **LMTOOLS Ignores License File Path Environment Variables**.
4. From the **Config Services** tab, use the **Service Name** drop-down menu to specify a name.
5. Specify paths for these three files:
 - Path to the lmgrd.exe file: Browse to select the file in your NLM folder.
 - Path to the license file: Browse to select the .lic license file you obtained from Ansys.
 - Path to the debug log file: Browse to select the debug log file you created earlier.
Note: Be sure that users and services have write access to the debug log file or folder.
6. Select **Use Services** checkbox at the bottom of the screen. Then select **Start Server At Power Up**. This setting ensures that the license server starts automatically after maintenance or a power outage.
7. Click **Save Service** and confirm.

Start the License Server

After configuration, use the LMTOOLS utility to start the license server.

1. From the **Start/Stop/Reread** tab, verify that the correct server name is highlighted. Then click **Start Server**.
 - a. If the server fails to start and give the following error: "VD is starting, please check vendor daemon's status in debug log", see [Server Fails to Start](#).
2. Verify that the license server has started (within 30 seconds). You should see the message "Server Start Successful" at the bottom of the dialog.

Confirm that the License Server is Running

1. From the **Server Status** tab of the LMTOOLS utility, click **Perform Status Enquiry**.
2. When you see information in the status window, scroll down and verify that:
 - The license server is running the latest version. Check the line that reads something like:
YourServerName: license server UP (MASTER) v11.18.0
 - The license server has successfully read the license file and found no errors. You should see a line that reads something like:
Adskflex: UP v11.18.0

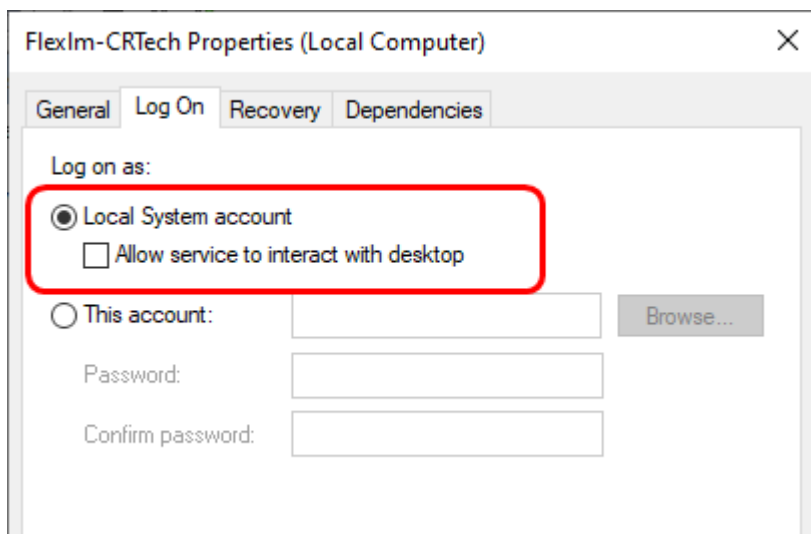
Configure / Change the License on the Client

1. On first open, within the “Let’s Get Started” window, select **Use a network license**. Otherwise, to reach this menu later or to switch from a named user license or device license to a network license:
 - a. Click the **User Account** menu in the upper-right corner of AutoCAD and select **Manage License**.
 - b. In the **License Manager** window, click **Change License Type** for an active license. This command lets you choose an existing license or complete the licensing of a new one. It doesn't remove or deactivate existing device licenses.
 - c. Select **Use a network license**.
2. Select your Server Model/Type (Single, Redundant, or Distributed). In the **Server Name** field, enter the port specified in the Autodesk license file (.lic) — typically port 2080 for Autodesk — and your license server name. *e.g. 2080@server_name*.
3. Click **Done**. If successful, the client should be able to checkout an AutoCAD license from the server and proceed.

Server Fails to Start

When setting up the server licensing, if you receive an error stating “*VD is starting, please check vendor daemon's status in debug log*”, do the following steps:

1. Open Services
 - a. Search the term "services" (or similar) in the Windows taskbar and click on the "Services" app.
2. Locate the Autodesk license service that was set up. Right click the name of the license service and select **Properties**.
3. On the **Log On** tab, ensure that **Local System account** is selected.



4. Click the **General** tab, then click the **Start** button.
 - a. It may take several seconds for the server to start.
5. Return to LMTOOLS. The service should now stop and start normally from the **Start/Stop/Reread** tab in LMTOOLS. After starting, click **Perform Status Enquiry** on the **Server Status** tab to verify that licenses are now available.

Appendix 2: Installing Ansys SpaceClaim on the client

To install Ansys SpaceClaim, follow the steps below.

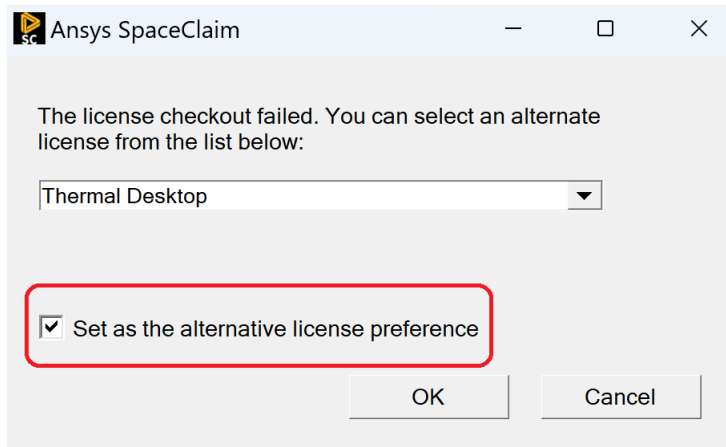
1. Download and extract the SpaceClaim ZIP file from the Thermal Desktop Download Page on the [Ansys Customer Portal](#).
2. In the extracted folder, right-click the setup.exe file and select **Run as administrator**.
3. Click **Install Ansys Products**. The license agreement appears. Select “**I agree**” and click **Next**.
4. Select the directory where you want to install Ansys SpaceClaim. Ansys recommends using the default directory: C:\Program Files\ANSYS Inc.
Choose whether to associate file extensions with Ansys products, then click **Next**.
5. On the **Default Licensing Mode** screen, select **Server Shared License**. Click **Next** to continue the installation.

NOTE: If using Thermal Desktop licensing for access to SpaceClaim, no additional .lic license file is required; the Thermal Desktop license is shared with SpaceClaim.

6. Once installation is complete — and after ensuring that the Ansys License Manager is running on the server and License Settings are configured on the client computer as indicated in the [Install Ansys License Manager and License File](#) section of the install instructions — from the Start menu, click on:

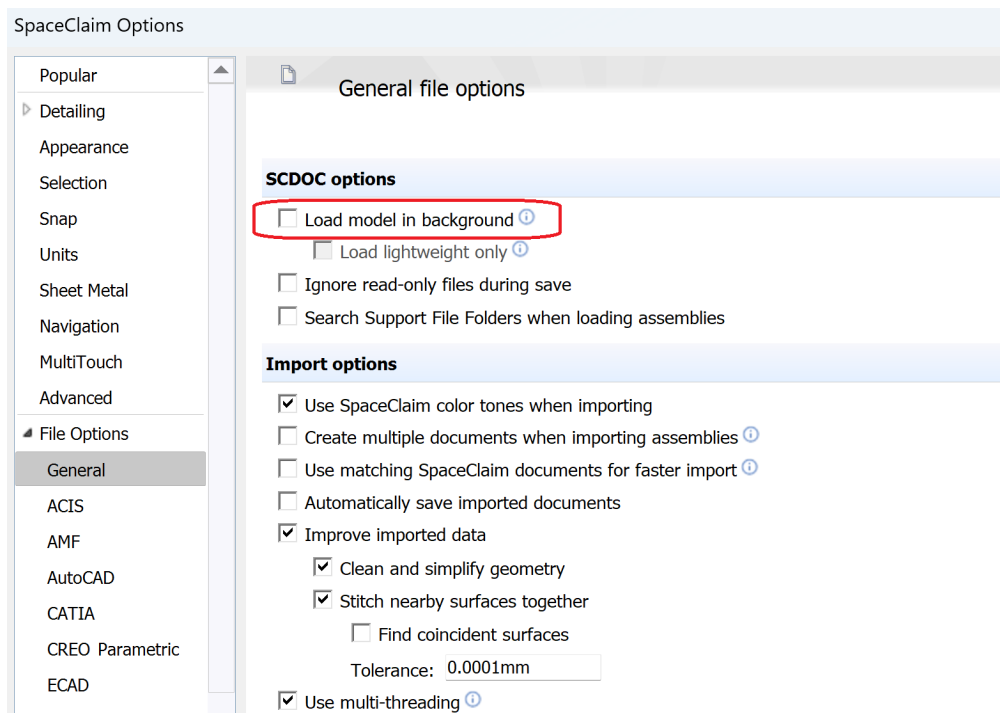
Ansys 2024 R1 > SpaceClaim 2024 R1

If Thermal Desktop is the only license that has been purchased (i.e. if there is not a dedicated Ansys license for SpaceClaim available on the account), the following pop-up window will appear:



With **Thermal Desktop** selected from the drop-down list, click the checkbox next to “**Set as the alternative license preference**” and click **OK**. SpaceClaim will now open and operate normally utilizing the shared Thermal Desktop license.

7. Now that SpaceClaim’s Alternate Licensing has been set to Thermal Desktop, turn off background loading of models.
 - a. Within SpaceClaim, click on **File > SpaceClaim Options**.
 - b. On the **File Options > General** tab, uncheck “Load model in background”.



8. Your installation and configuration of SpaceClaim is now complete.

NOTE: When using shared licensing between Thermal Desktop and SpaceClaim, you cannot have an instance of one of the two programs already open and then separately

open the other program. To use the two programs in tandem, a Thermal Desktop instance must be opened *from within SpaceClaim* via the **TD Connect** link (without otherwise already having an instance of Thermal Desktop open):



Conversely, a SpaceClaim instance must be opened *from within Thermal Desktop* via the **TD Direct** menu links.