Installing Abaqus, Isight, Tosca, fe-safe 2022 - 2017 to run off the Central License Server

Installing Abaqus, Isight, Tosca, fe-safe 2022 - 2017 to run off the Central License Server

On this page:

- Requirements
- Installation
  - Use these Online Program Directory links if you don't have enough disk space to install the optional Program Directory:
  - For Installation help, download one of these PDF guides.
  - Downloads
    - **Optional Local Install Program Directory**
  - Windows installation
  - Linux installation
- See Also

Requirements

This documentation and software are licensed under an academic license agreement between MIT and Dassault Systemes Simulia Corp. It may be used only by MIT students, faculty, and staff (Authorized Users) for non-profit academic research and teaching. The right to use the materials for any other purpose, including commercial purposes, is expressly prohibited. Authorized Users must stop using the software and delete it (and any accompanying license key) from any non-MIT-owned computers they have installed it on when they cease to be affiliated with MIT. By clicking on the links below, you agree that you are an Authorized User and that you will only use the software for the purposes listed above.

You must be on the network during installation, as well as later when you use the software. If you're on campus and using wireless, you must be on the **MIT SECURE** network. If you're off-campus, you need to be running the VPN.

You must launch Abaqus from a writable folder and/or launch it as a user with administrative privileges. Otherwise, you will receive an error referring to:

```
IOError ... Permission denied...
```

This installation procedure is only authorized for MIT student/faculty/staff users. It requires a 64-bit operating system (Windows or Linux). See these pages for more details. You will need a recent Java plugin installed in your browser to search the documentation. Please note that many current browsers no longer support Java plugins; you may need to go to some effort to find one that is compatible.

Note:

- You will not need to register with the vendor or download anything from their web site.
- You will not need to update the annual license file, this is handled on the server.

Installation

Since Abaqus 2017; and newer, have been released, they now embody a unified installer for Abaqus, Isight, Tosca and fe-safe. We no longer have separate installers and installation instructions for the different components. Please do not attempt to install the license server, it won't work and is irrelevant to end-users.

When you get to the part of the installation where the documentation is installed, you can either select the **enter a file path for locally installed documentation** option, or use the central documentation on the Dassault site.

Use these Online Program Directory links if you don't have enough disk space to install the optional Program Directory:

- SIMULIA Online Program Directory 2023
- SIMULIA Online Program Directory 2022
- SIMULIA Online Program Directory 2021

- SIMULIA Online Program Directory 2020
- SIMULIA Online Program Directory 2019
- SIMULIA Online Program Directory 2018
- SIMULIA Online Program Directory 2017

It is highly recommended that you install components to the default locations. The Abaqus suite contains many interrelated components that make assumptions about where other components are located.

For Installation help, download one of these PDF guides.

- SIMULIA Installation Guide 2023 pdf
- SIMULIA Installation Guide 2022 pdf
- SIMULIA Installation Guide 2021 pdf
- SIMULIA Installation Guide 2020 pdf
- SIMULIA Installation Guide 2019 pdf
- SIMULIA Installation Guide 2018 pdf
- SIMULIA Installation Guide 2017 pdf

Downloads

You will need the following items (access limited to students/faculty/staff):

Windows 10 build 1703 and later:

- Abaqus 2023 All OS Win64/Linux64 software
- Abaqus 2022 All OS Win64/Linux64 software
- Abaqus 2021 All OS Win64/Linux64 software
- Abaqus 2020 All OS Win64/Linux64 software
- Abaqus 2019 All OS Win64/Linux64 software
- Abaqus 2018 All OS Win64/Linux64 software
- Abaqus 2017 software

Optional Local Install Program Directory

- Abaqus 2023 Program Directory (optional)
- Abaqus 2022 Program Directory (optional)
- Abaqus 2021 Program Directory (optional)
- Abaqus 2020 Program Directory (optional)
- Abaqus 2019 Program Directory (optional)
- Abaqus 2018 Program Directory (optional)
- Abaqus 2017 Program Directory (optional)

Windows releases prior to Windows 10 build 1703/Linux 64 bit ONLY:

- Abaqus 2017 software
  - 2017 doit.bat

Installation of the Program Directory (Abaqus_20##.PDir_SIMULIA_EstPrd.1-1_Prd.zip archive) is optional. Unzip it in some local disk location `<install_location>` (Cloud Storage Space will fail). It contains release notes, system requirements and general information about the current Abaqus release. The starting point for browsing the information is

- `<install_location>\PDir_SIMULIA_EstPrd\1\default.htm` (Windows)
- `<install_location>/PDir_SIMULIA_EstPrd/1/default.htm` (Linux)

We have repackaged the separate `tar` archives mentioned in the installation manual in a single zip file. When you unzip it, you will have a complete installation tree and can pick up the instructions after step 2. in the **Extracting the Media** section of the manual.

> ⚠ Please install all Abaqus components for a completely functional installation. Isight, Tosca and fe-safe are optional and should only be installed if required.

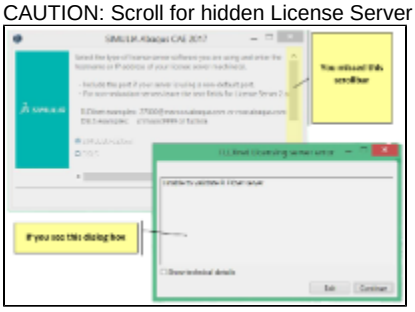
In certain usage modes, Abaqus requires access to system C, C++ and FORTRAN compilers. Installation and configuration of these are beyond the scope of this document; please consult the Abaqus documentation for further information.

Note: Do not install **Tosca Fluid** unless you have at least one of the required fluid dynamics solvers **Fluent** or **STAR_CCM+** installed on your computer, otherwise the installation will fail.

Windows installation

⚠ A bug prevents the correct Abaqus 2017 installation on Windows 10 build 1703 and later from the standard installation files. A patched set in an alternate installation archive works around this. Installation of a short batch file **doit.bat** is needed to work in conjunction with the patched archive. See download link above for the **doit.bat** batch file.

Please note that installation will require about 26 Gb temporarily/14 Gb permanently for a complete install of all products and documentation.

1. Unzip **Abaqus_20##AllOS.zip** in a temporary local disk location, cloud storage spaces will fail (after the extraction you can delete the zip file to save space).
    - For the two 2017 versions, unzip **abaqus_2017.zip** (Windows releases prior to Windows 10 build 1703) or **abaqus-win1703_2017.zip** (Windows 10 build 1703 or later) in a temporary local disk location, cloud storage spaces will fail (after the extraction you can delete the zip file to save space).
    - The recommended application for this is 7-Zip.
2. Abaqus 2017 on Windows 10 build 1703 or later only: copy file **doit.bat** to **%TEMP%** directory.
    - This can be achieved from a terminal window by typing **copy doit.bat %TEMP%** or by manually dragging/dropping the **doit.bat** file to C:\Temp
3. Follow the instructions in .pdf file found in `<unzip location>/AM_SIM_Abaqus_Extend.AllOS/1/SimuliaInstallationGuide.pdf`; skip the **Installing the DS License Server** section at the top. Reminder that only the Abaqus components should be installed unless the others are required.
    - Whenever the installers ask what type of licensing is in use, select **FlexNet** and/or **Network Licensing**.
      CAUTION: Scroll for hidden License Server entry field.
      
      Enter the address found in this file in the **License Server 1** dialog (leave the other entries blank)
    - If Windows firewall alerts pop up during installation, select **Allow Access**
4. After the installation is complete and verified you can delete the contents of the extracted zip file to save space

In order to view and search the documentation, **Internet Explorer** is the best option. The **Microsoft Edge** browser will not display the documentation correctly.

To launch Abaqus from a command prompt, type **abaqus**. To launch the GUI, type **abaqus cae**. To launch the viewer, type **abaqus viewer**.

Linux installation

The only supported Linux distributions are **Red Hat Enterprise Linux**, **Centos** and **SuSE Linux Enterprise Server**. You will likely get an error message about "unsupported Linux distribution" if you attempt to install it on others, in particular **Ubuntu**, and the installation will fail.

You may be able to get around this by editing a line in each of the `Linux.sh` files located in the installation tree (there are 8 occurrences); change the line

```
DSY_OS_Release=`lsb_release --short --id |sed 's/ //g'`
```

to

```
DSY_OS_Release="RedHatEnterpriseWorkstation"
```

before running the installer.

Please note that installation will require about 27.2 Gb temporarily/16.2 Gb permanently for a complete install of all products and documentation. You will also need to have the **openmotif** and **compat-libstdc++-33** packages installed on Red Hat systems.

1. Unzip **abaqus_2017.zip** in a temporary local disk locationm, as cloud storage locations will fail (after the extraction you can delete the zip file to save space)
2. Follow the instructions in `<unzip location>/AM_SIM_Abaqus_Extend.AllOS/1/SimuliaInstallationGuide.pdf`; skip the **Installing the DS License Server** section at the top.
    - Whenever the installers ask what type of licensing is in use, select **FlexNet** and/or **Network Licensing** and **single server**; enter **academic research** for licensing type. Enter this in the **License Server 1** dialog (leave the other entries blank)
3. After the installation is complete and verified you can delete the contents of the extracted zip file to save space

Abaqus is configured to use the Intel FORTRAN compiler by default; it is usually possible to use gfortran instead by editing `/usr/DassaultSystemes/SimulationServices/V6R2017x/linux_a64/SMA/site/lnx86_64.env` as follows:

1. Change `ifort` to `gfortran` on the `fortCmd` line and make sure that `g++` is on the `cppCmd` line
2. Remove gnu-incompatible compiler flags from the `compile_fortran` line (`-V, -auto, -mP2OPT_hpo_vec_divbyzero=F, -extend_source, -fpp, -WB`)
3. Add `-lgfortran` to the `link_sl` and `link_exe` lines

If you select the default command locations:

- The Abaqus launch command is **abaqus** (or **abaqus cae** for the GUI, **abaqus viewer** for the viewer) in the `/var/DassaultSystemes/SIMULIA/Commands` directory.
- The fe-safe launch command is **fe-safe**, in the `/usr/SIMULIA/fe-safe/2017` directory.
- The Isight launch command is **gateway**, in the `/usr/SIMULIA/Isight/2017/linux_a64/code/command` directory.
- The Tosca Structure launch command is **ToscaStructureGui.sh**, in the `/usr/SIMULIA/Tosca/2017/linux_a64/code/command` directory.
- The Tosca Fluid launch command is **ToscaFluidGui.sh**, in the `/usr/SIMULIA/Tosca/2017/linux_a64/code/command` directory.



You may need to set the **XLIB_SKIP_ARGB_VISUALS** environment variable to 1 in the launch terminal first if the Abaqus windows are washed out and translucent. You may run into other windowing artifacts unless you log in with a basic graphical session that disables advanced visual effects. Details for how to do this are beyond the scope of this article.

See Also

- Tosca Landing Page
- Abaqus Landing Page