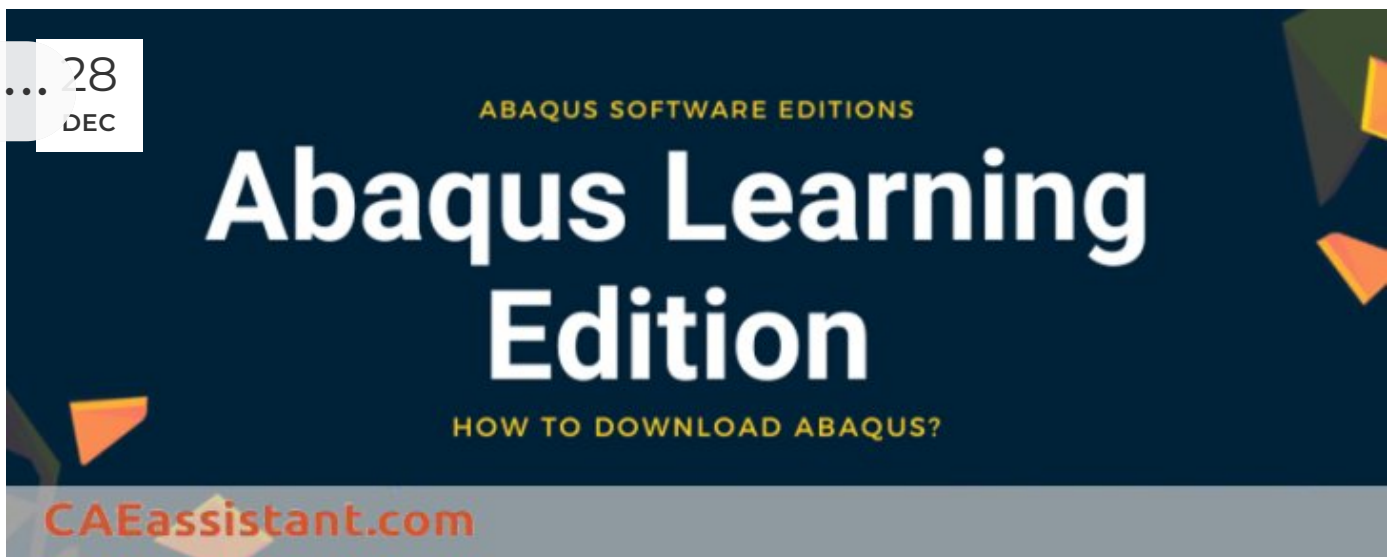# Blog

BLOG

How to download Abaqus?|Abaqus commercial & student edition

Posted by 👤 Training Expert

28 DEC

Abaqus is a finite element software that is used for various simulations in the engineering world. However, there are always a number of questions that are on the minds of those who want to use this software, especially students. where we can find different editions of Abaqus, including Abaqus for students and the commercial version? Where to download it (Abaqus download)? Is Abaqus software free, or do you have to pay for it? If it isn't free, how much would you have to pay? Does it have different types of editions? Abaqus student edition or Abaqus learning edition. In this article we tried to answer all your possible questions about Abaqus editions(Abaqus student version & Abaqus Commercial version), their positive aspects and limitations, where you can download Abaqus and which Abaqus editions is best for you.

Stay with **CAE ASSISTANT** to solve this matter once and for all.

Table of Contents

Contents

Abaqus editions

1. Introduction of Abaqus software editions

2. Abaqus Student Edition | Abaqus for student

3. Abaqus commercial edition

3.1. How much does Abaqus cost?

3.2. What are Abaqus tokens?

4. Minimum requirements to install Abaqus

5. Abaqus Student edition download: Getting ABAQUS Student version?!

6. Steps of installing ABAQUS Student Edition

7. ABAQUS Student Edition (SE) limitations

8. How hard is it to learn Abaqus?

9. What's new in Abaqus 2022?

9.1. Benefits

9.2. Features

10. Temporary subscription license

11. Tips for you

1. Introduction of Abaqus software editions

Abaqus is a powerful engineering simulation software suite based on the finite element method that can solve problems ranging from simple linear analyses to the most difficult nonlinear simulations. Abaqus has a large library of elements that can model almost any geometry. It also has a long list of material models that can simulate the behavior of most common engineering materials; for example, metals, rubber, polymers, composites, reinforced concrete, crushable and resilient foams, and geotechnical materials like soils and rock. Abaqus is a general-purpose simulation tool that can be used to study problems other than structural (stress/displacement). It can simulate heat transfer, mass diffusion, thermal management of electrical components (coupled thermal-electrical analyses), acoustics, soil mechanics (coupled pore fluid-stress analyses), and piezoelectric analysis, among other things.

Today, engineering teams frequently perform product simulations using niche simulation tools from various vendors to simulate various design attributes. Using multiple software vendors creates inefficiencies and increases costs. SIMULIA provides a scalable suite of unified analysis products that enable all users, regardless of simulation expertise or domain focus, to collaborate and share simulation data and approved methods without compromising information fidelity.

There are two types of Abaqus editions: Abaqus Student Edition (SE) and Abaqus commercial edition.

- *Abaqus Student Edition*

Anyone interested in learning more about Abaqus can do so for free by downloading the

ABAQUS Student Edition (Abaqus learning edition). Perform finite element analysis operations on a personal computer using the educational and research versions of the suite, but it has size restrictions, such as 1000/10,000 element nodes, and is governed by a personal use license. The complete documentation collection in HTML format makes this the perfect Abaqus learning tool.

We've got you covered with all the information you need about the Abaqus student version in Section 2. Stick around for the full scoop!

- *Abaqus commercial edition*

The Abaqus commercial edition can offer a strong and complete solution for common and complex engineering issues. This software is not accessible to the general public because it is currently using an evaluation demo license with a single seat. It will be necessary to purchase an academic license if your group wants to use the commercial Abaqus edition. (which incidentally appears very reasonably priced considering the capabilities of this package).

2. Abaqus Student Edition | Abaqus for student

Abaqus Student Edition, Abaqus Learning Edition, Abaqus Student Version, and Abaqus Free Version are all names that users call this particular version. Therefore, if you hear any of these terms, they all refer to the same edition.

Dassault Systèmes offers a free version of Abaqus for students. This version is specifically designed to cater to the needs of students.

The ABAQUS Student Edition is a personal finite element analysis tool for dealing with problems of limited size (1000 nodes for ABAQUS/Standard and ABAQUS/Explicit, and 10,000 nodes for ABAQUS/CFD). It is subject to a personal use license, can be installed on your laptop and home computer, and can be used permanently. The Abaqus learning edition has the complete online ABAQUS documentation set and is compatible with Windows XP/Vista/Windows 7, 8, 10, 11 and also Linux. The ABAQUS Student Edition is fully compatible with the ABAQUS Teaching and ABAQUS Research Editions, making it an ideal complement to your campus ABAQUS installations.

ABAQUS/Standard, ABAQUS/Explicit, ABAQUS/CFD, and ABAQUS/CAE are all included in the ABAQUS Student Edition. Model databases are interoperable with other academically licensed ABAQUS versions, allowing students to perform increasingly sophisticated analyses on models created in Abaqus Student Edition. The complete HTML documentation has made Abaqus for students version a comprehensive, searchable resource for finding detailed information.

The maximum model size for structural analysis and post-processing is 1000 nodes for personal education and 10000 nodes for fluid analysis. There is no access to features that require compilers (user subroutines, ABAQUS Make, C ODB API), parallel execution, or add-on products. ABAQUS/CAE does not support replay or journal files.

As we mentioned before, Maybe you have seen the learning edition in various sources; this is the same Student Edition but with a different name. Anyone interested in learning ABAQUS can download the ABAQUS Learning Edition for free. The ABAQUS Learning Edition is only available for Windows and supports structural models with up to 1000 nodes. ABAQUS is an expensive piece of software. If you work at an academic institution, your department may already have a subscription to the Research edition. Engineering Technical Services currently offers Abaqus Research Edition on a Pay-as-you-Go basis. The Abaqus/Standard, Abaqus/Explicit, and Abaqus/CAE are available with the Abaqus Research Edition. Due to the inclusion of both Abaqus/Standard and Abaqus/Explicit, users can select the best analysis technique for their simulation type or even combine the two techniques. in the research edition, Modeling advanced research materials, complex surface and contact behaviours, and other user-specific features are made easier with user-defined subroutines.

You can download the student version of Abaqus for free from the link below:

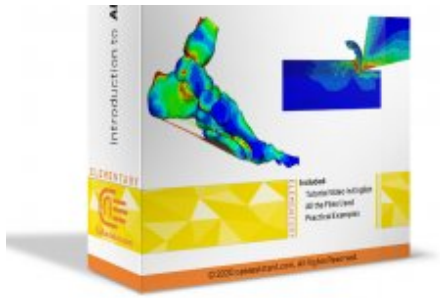https://edu.3ds.com/en/software/abaqus-learning-edition

If you work for a company, they will need to purchase licenses for the Commercial edition if they do not already have any. The good news is that if all you want to do is learn how to use the software, you can do so for free by downloading ABAQUS Student Edition (SE) or Abaqus learning edition.

Although ABAQUS SE was previously limited to students, it is now available for download by anyone for educational purposes. The download also includes the ABAQUS documentation. In the following, there is information about the paid and commercial versions of Abaqus, the minimum system requirements for installing Abaqus, relevant links to download the software, and the steps to install the SE version of Abaqus. Take a break and join us again.

Suppose you've downloaded the Abaqus learning edition and are ready to learn Abaqus; But, do you know where to start? I recommend searching on YouTube or checking these tutorial packages available on **CAE ASSISTANT**.

3. Abaqus commercial edition

As mentioned in the introduction, this version of Abaqus requires payment, and you have to pay different prices according to different analyses. It provides a powerful and comprehensive solution for routine and complex engineering problems, as well as a good collection of Multiphysics and structural-pore capabilities capturing simulation software. If you work for a company, you will need to purchase licenses for the Commercial edition. You can contact Dassault Systems directly if your company has earnings of more than $2 billion or more than 5,000 employees. Otherwise, a local reseller can help.

3.1. How much does Abaqus cost?

Dassault Systèmes, Abaqus' parent company, offers a number of promotional pricing packages throughout the year, but here is an overview of the starting prices for different packages (as of April 2022):

- A quarterly lease of ABAQUS begins at around $7,000

- A year's lease on ABAQUS begins at around $19,000

- The cost of purchasing ABAQUS starts around $31,000, with annual maintenance (updates and support) starting at $6,000 per year.

3.2. What are Abaqus tokens?

One of FEA (Finite Element Analysis) products, Abaqus, is frequently used for simulations in CAE (computer-aided engineering). Dassault purchased Abaqus in 2005, and there are two license managers available: Dassault's own DSLS product and Flexera's Flexnet, which is made to manage token licenses. Due to the nature of FEA software, which frequently runs on multiple processors with no or minimal user intervention, the token system performs impressively for Abaqus. The following factors determine how many tokens are needed for a job:

1. number of simultaneous users

2. the required number of parallel processors (CPUs or GPUs)

3. based on user count, at least 1 token for the CAE pre-processor

Abaqus is licensed through tokens. Tokens provide access to both the Standard and Explicit solvers. Other than the bounds of your computer hardware, these solvers have no element or node limits. Multiple CPUs or CPU-cores generally improve computational throughput for large and time-consuming FEA models. A minimum of 5 analysis tokens are required to run a model on a single CPU core. In Tables 1 and 2 you can see an illustration of token costing for various Abaqus products.

Table 1: The number of tokens required according to the number of analyses

| Concurrent Users | Number of Tokens | |
	CAE Tokens	Analysis tokens
1	1	5
2	2	10
3	3	15
4	4	20
5	5	25

Table 2: The number of tokens required according to the number of CPUs

| Product | Number of CPUs | | | | | | | | | |
| | 1 | 2 | 4 | 8 | 12 | 16 | 24 | 32 | 64 | 128 |
	Number of analysis tokens required									
Abaqus/Standard Explicit and CFD	5	6	8	12	14	16	19	21	23	38
Foundation	3	4	5	7	8	9	11	12	17	23
Aqua Design	6	8	10	14	17	19	22	25	34	46

4. Minimum requirements to install Abaqus

Before you begin the installation, make sure your system meets the following requirements:

- Windows 7/8/8.1/10 must be your operating system. You cannot run ABAQUS directly on Mac OS and must have Windows administrator privileges to complete the installation. However, ABAQUS is unavailable on computers running Mac OSX, but you can gain access to the software via remote access.

- You must have a 64-bit operating system (32-bit will not work). If you're still not sure, open a command prompt window and type "systeminfo" into it. You have a 32-bit operating system if the "System type" field says "X86-based PC." If it says "X64-based PC," you have a 64-bit operating system. It should say something along the lines of a 64-bit operating system.

- The installation will require 8.5 GB of disk space.

The Abaqus Student Edition installation consists of 3 basic sections.

a) Abaqus HTML Documentation Installation (5.8 GB disk space)

b) Abaqus Product Installation (2.7 GB disk space)

c) Abaqus Installation Verification

- Internet Explorer 11.0.11 or higher is required. If you allow Windows to install updates, you probably already have this. The installer requires it.

- You must have administrator access to the machine.

5. Abaqus Student edition download: Getting ABAQUS Student version?!

Follow these steps to download ABAQUS Student Edition:

You can download the Abaqus student version for free from the link below:

https://edu.3ds.com/en/software/abaqus-learning-edition

Then, on the page, click the free download button (see figure 1).

Figure 1: The page containing the download link of ABAQUS SE

After that, you must create an account. However, sign in if you already have an account. Accept the terms of the agreement and click the "Download Now" button.(ABAQUS

Student Edition download start)

Figure 2: Create an account at 3ds.com

6. Steps of installing ABAQUS Student Edition

You can easily obtain the Abaqus student edition download by following the provided instructions for a seamless installation process.

Follow the steps highlighted below:

1. Go to the Abaqus Simulia learning edition page and click "Free Download."

2. Register or sign in if you already have an account.

3. Accept the Terms of Service Agreement and click "Download Now." You'll then be directed to the Download Center.

4. In the Download Center, select the "Download & Instructions" dropdown menu and download the ABAQUS version you require. Extract the downloaded ".zip" file and enter the folder. If you are installing **ABAQUS SE 2019**, If you could not easily go through the installation steps from the text, we showed you the installation steps of Abaqus in a video in section 11 of this post.

5. Run "setup.exe" to complete the installation. You can use the default ABAQUS directory settings or specify a different folder.

6. Once the installation is complete, you can begin using ABAQUS.

Below are step-by-step instructions for each section. The pictures below come from a Windows 10 installation of SIMULIA Abaqus Student Edition 2017.

Step1

Extract the downloaded media .zip file using 7-Zip into a folder/directory of your choice.

Note: This path must be less than 60 characters; for example C:\downloads

This download location differs from the folder/directory where the products will be installed (and may optionally be deleted once the installation is complete).

It is recommended that you use the free 7-Zip utility (V4.57 or higher), which handles the zip format. If you do not have 7-Zip, download the latest version at https://7-zip.org/.

Step2

Run the following executable from the extracted media directory to verify the integrity of the extracted media:

c:\downloads\SIMULIA_Abaqus_Student.Windows64\1\0Data\intel_a\DSYInsMediaCheck.exe

Step3

Double-click the setup.exe executable in the extracted media to begin the installation process:

c:\downloads\SIMULIA_Abaqus_Student.Windows64\1\setup.exe

Step4

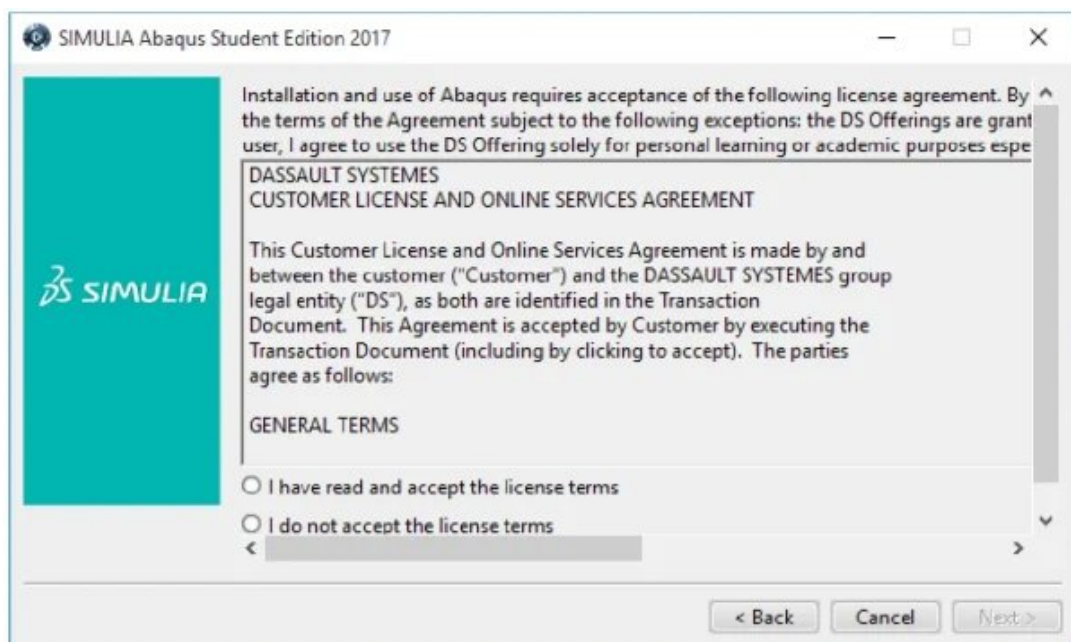Read and accept the terms of the license agreement to continue. Click Next.

Figure 3: Step 4 for installing Abaqus SE (accept the terms of the license agreement)

Step5

You will be asked to provide an installation directory.

NOTE: This location is different from the extracted media folder.

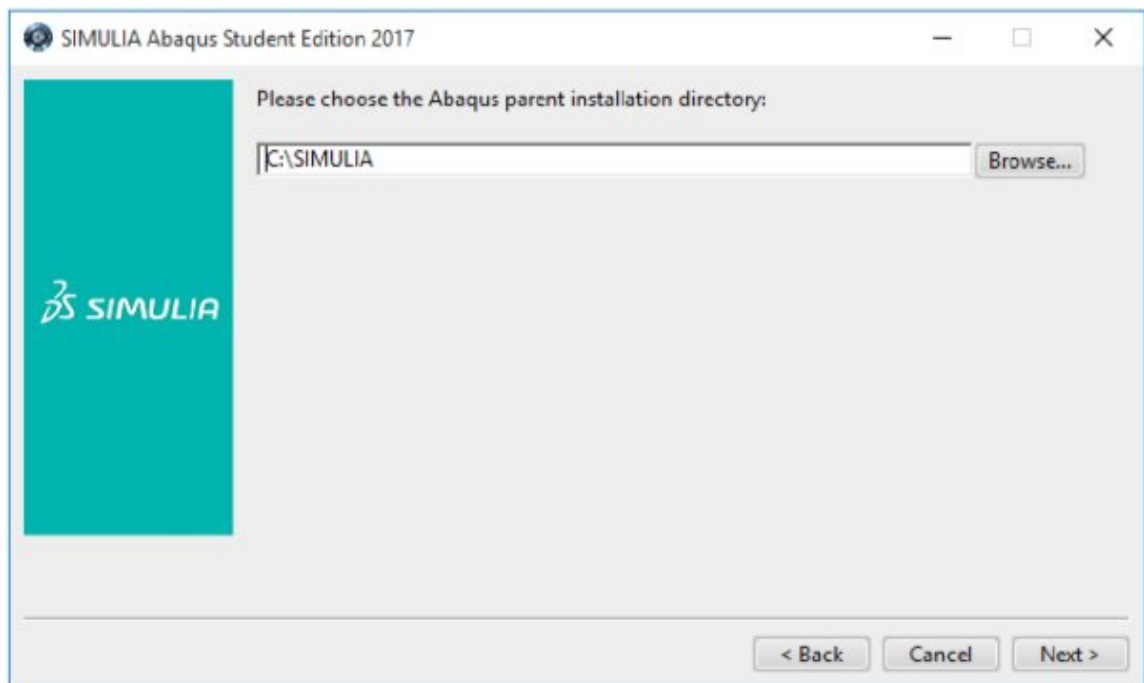Choose your installation directory and click Next.

Figure 4: Choosing installation directory

Step6

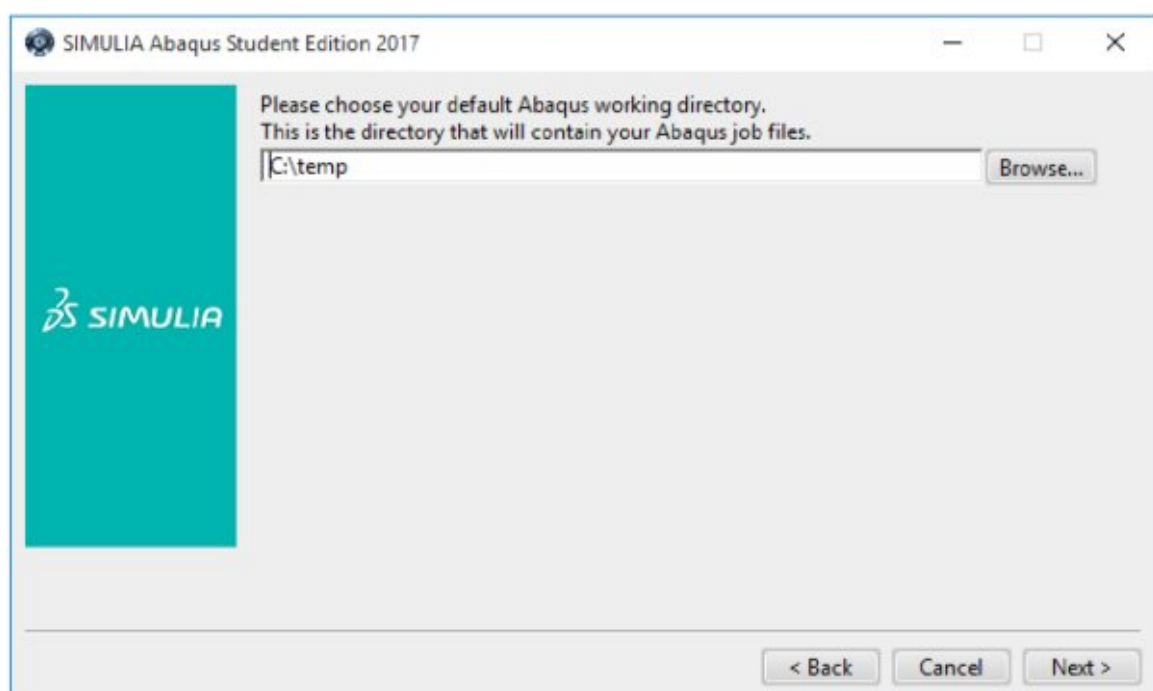Choose the location for your Abaqus job files and click Next.

Figure 5: Choosing the default Abaqus working directory

Step7

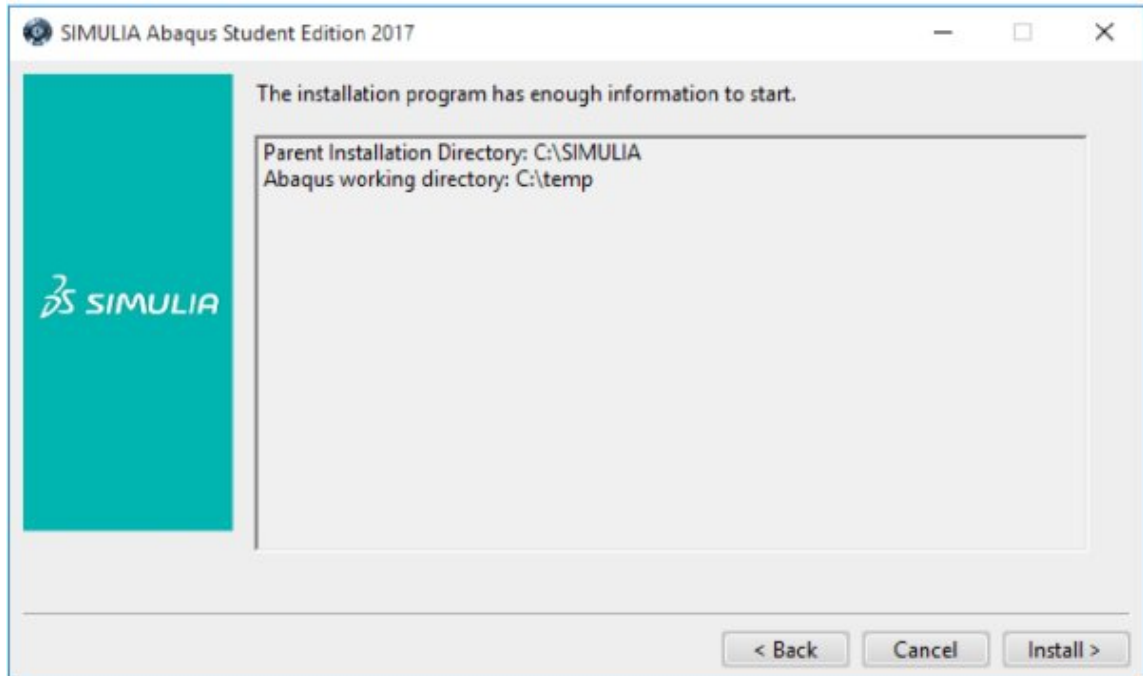Review the information in the pre-installation summary. Click Install when you are ready to proceed.

Figure 6: Pre-installation summary

Step 8

First, the Abaqus Student Edition HTML Documentation is installed. This process may take up to 30 minutes to complete.
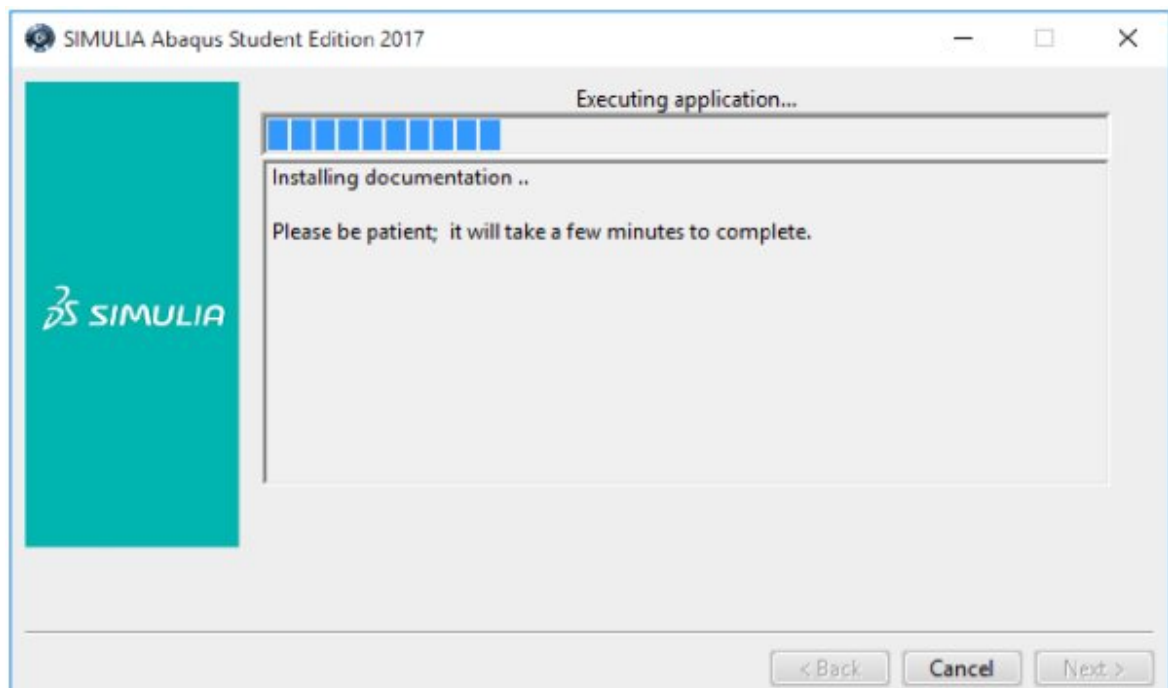
Step9

After the documentation installation is completed, the Abaqus Student Edition product installation will start.
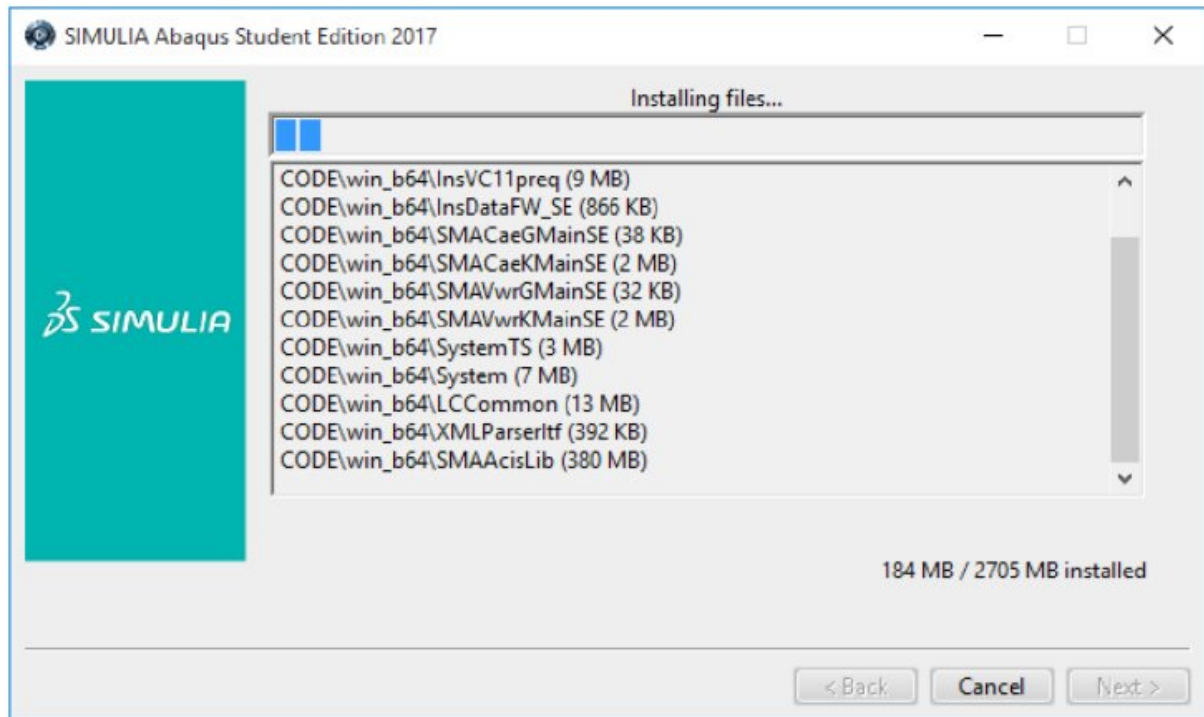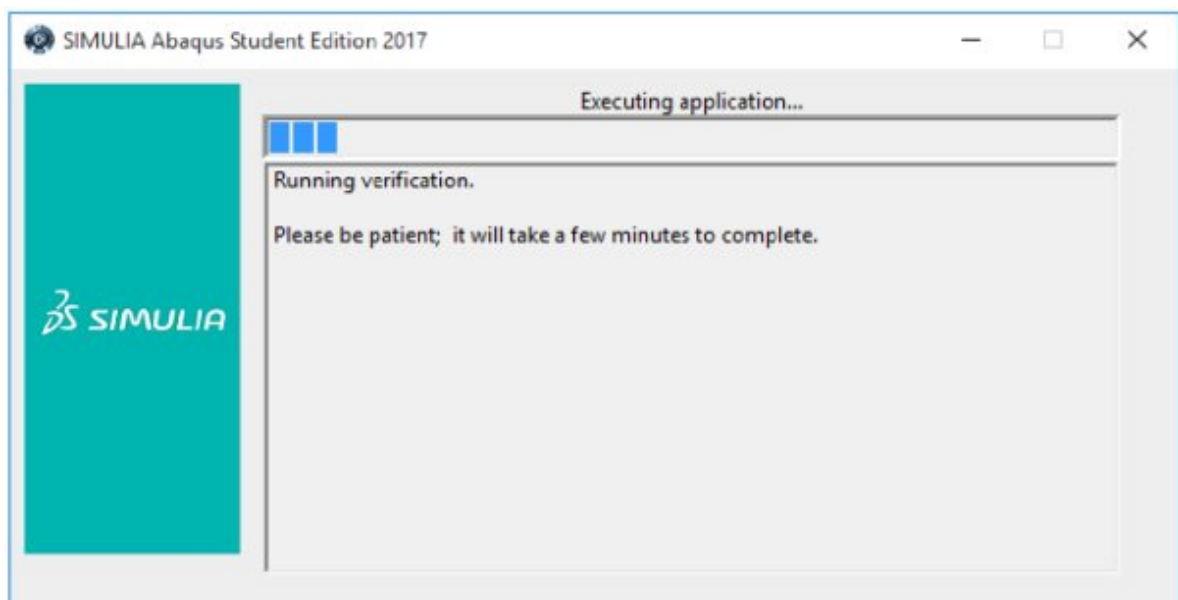

Figure 8: Abaqus SE installation process

Step10

Once the Abaqus product installation has been completed, the product verification begins automatically. This tests the integrity of the installation.
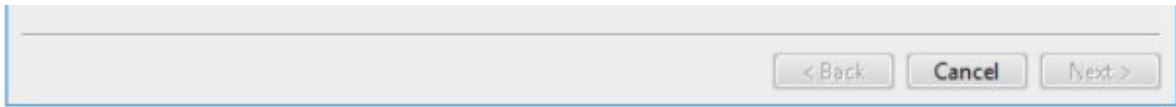
Figure 9: Running verification

Step 11

After verification completes, a panel is displayed, giving you the path to where the verification results are located and information about launching the Abaqus Student Edition software.
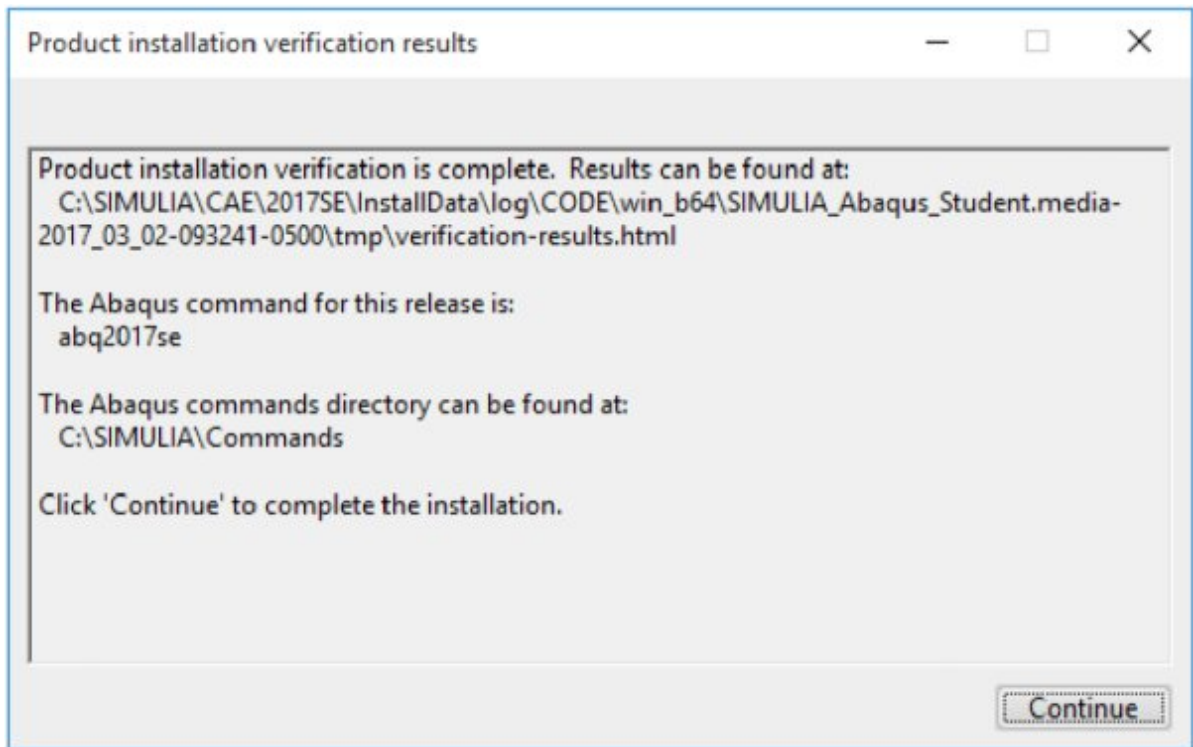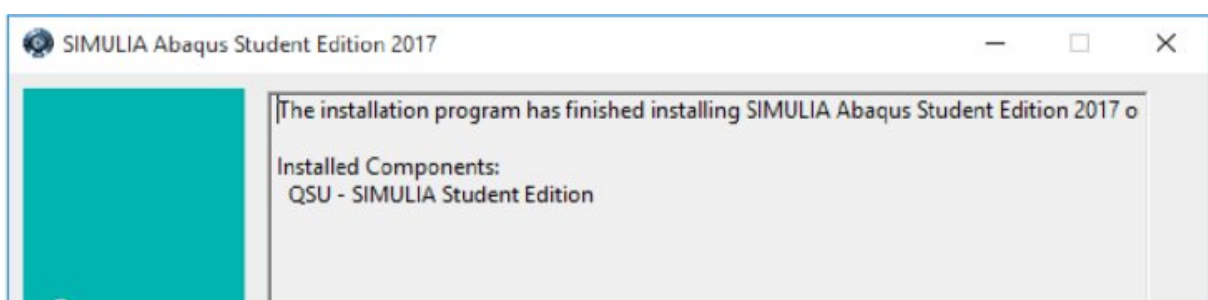

Figure 10: Product installation verification results

Open the verification-results.html file specified in the panel for additional information and troubleshooting.

Step 12

After clicking Continue, the final screen will appear. Clicking Close will finish the installation and exit the window.
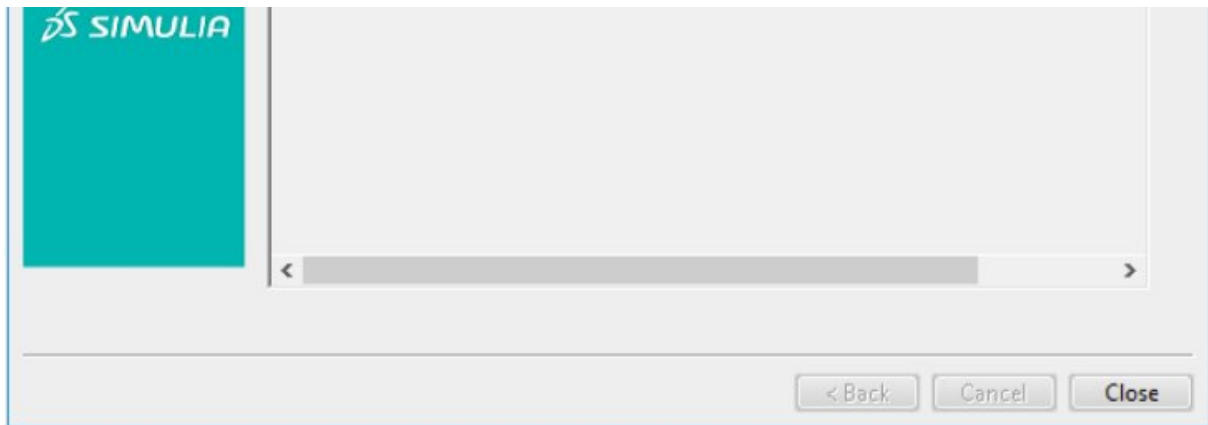
Figure 11: The installation is completed

Execute the newly installed Abaqus Student Edition:

- On Windows 7 and earlier, from the shortcuts in the Start menu under Programs -> Dassault Systemes SIMULIA Abaqus Student Edition 2017.

- On Windows 8 and later, from the icons in the Apps screen under the section Dassault Systemes SIMULIA Abaqus Student Edition 2017.

Known Issues:

- On Windows 10, Microsoft Edge is not supported for User Assistance (documentation). If Microsoft Edge is configured as your default browser, you'll need to change your default browser to either Internet Explorer or Mozilla Firefox in order for the Abaqus User Assistance to launch properly from Abaqus Student Edition.

- Some User Assistance (documentation) contains mathematical notation written in the MathML markup language. The Open Source MathJax javascript package is used to render this notation in HTML output. The fonts that MathJax normally uses are web fonts, included with the documentation and downloadable by MathJax as it performs the rendering.

Reference: https://edu.3ds.com/sites/default/files/2020-08/2020_Abaqus_SE_InstallationGuide%26KnownIssues.pdf

7. ABAQUS Student Edition (SE) limitations

To be clear, ABAQUS SE has some limitations that prevent it from being used for serious commercial work.

- ABAQUS/Standard, ABAQUS/Explicit, and ABAQUS/CAE comprise the ABAQUS Student Edition.

- Complete HTML documentation is provided.

- The maximum model size for structural analysis and post-processing is 1000 nodes, and features requiring compilers are unavailable. This means that you cannot use Fortran user subroutines or the C++ ODB API for post-processing.

- Because parallel execution is not supported, you will be limited to running your analysis jobs on a single CPU in Abaqus for student version.

- There are no add-on products available.

- Academically licensed ABAQUS releases are compatible with ABAQUS Student Edition model databases but not commercially licensed ABAQUS releases.

- Any models created in ABAQUS for students version will not be compatible with commercially version(but they will work in ABAQUS Research Edition).

8. How hard is it to learn Abaqus?

Abaqus is a popular finite element analysis software package. It is a very powerful tool, but it is difficult for beginners to use; moreover, learning how to use the software from the documentation is difficult. If you are new to ABAQUS, the "**Abaqus tutorial**" will assist you in getting started.

This tutorial will walk you through the basics of using ABAQUS as well as the general workflow for creating and running finite element simulations with the software. If you are new to ABAQUS, I strongly advise watching this tutorial before proceeding with the rest of the series.

Also, if you need more examples in fields like Biomechanical, Steel material structures, Concrete structures, etc, click on the links below. You'll find them useful.

Bio-Mechanical Abaqus simulation Full package

Abaqus Soil Modeling Full Tutorial

Abaqus steel material and structures Full Tutorial

Abaqus Crack Growth Full Tutorial

Abaqus Concrete structure Modeling Full Tutorial

9. What's new in Abaqus 2022?

9.1. Benefits

- Accurate prediction of real-world product behavior allows businesses to improve product reliability while lowering warranty costs.

- Significantly reduced the need for costly physical testing.

- Rapid turnaround time allows the simulation to influence decisions early in the product design process, reducing the need for costly re-designs.

- Increase engineering productivity by standardizing on a single complete structural simulation capability.

9.2. Features

- A broad range of nonlinear analysis procedures includes structural stress, explicit dynamics, thermal stress, heat transfer, pore fluid diffusion, and others.

- A modern linear dynamics framework with high-performance Eigen solvers, transient and steady-state dynamics, random response, and substructures.

- Capabilities for advanced material modeling in metals, polymers, composites, concrete, and fracture mechanics.

- Strong general contact capability for simulating complex interactions among large assemblies of parts.

- High-performance solvers that reduce simulation time.

10. Temporary subscription license

In this part of the article, our goal is to introduce you to the Abaqus license seller. You can buy the license you want by communicating with the sales admin.

10.1. Technia

As a Dassault Systèmes PLATINUM Partner, TECHNIA's success has been driven by the insights they have gained through their own use of the products for sale. This allows them to provide their clients with expert engineering services because they understand the all-important user experience first-hand. The result is superior product implementation, services, and training that drive success and deliver the best ROI for the customer.

SIMULA

SIMULIA – Realistic Simulation Solutions SIMULIA makes virtual testing a standard business practice that improves product performance, reduces physical prototypes, and drives innovation. The SIMULIA product portfolio offers the most up-to-date and advanced simulation products,

including Abaqus FEA, Isight, fe-safe and Tosca. Power of the Portfolio Technologies (Extended Packaging): Abaqus Unified FEA → Finite Element Analysis [...]

10.2. Fidelis

Fidelis is a provider of custom CAE solutions and a reseller of Dassault Systèmes' industry-leading simulation software. This site has tried to express all the tips and materials about downloading software(Abaqus download) and their tips in the form of articles and posts in a useful way. You already know that integration of advanced simulation leads to improved products, lower cost to market, and a more streamlined engineering process. Let us help you make the most of it by doing what we do best.

https://www.fidelisfea.com/post/how-much-does-abaqus-cost-and-what-is-included

10.3. CATI

Computer Aided Technology (CATI) is a product development solutions provider empowering innovators in solving America's engineering and manufacturing challenges. They are a team of engineering and manufacturing experts helping their clients expand their capabilities and improve productivity through innovative technology. You can also buy products related to Solidworks software from this site.

> Abaqus Unified FEA

How to Buy Abaqus: 5 Easy Steps

If you are considering acquiring a best-in-class finite element analysis software package such as ABAQUS, you might have wondered about what's involved in that process. Here's a breakdown of the five easy

10.4. FEA Services LLC

FEA Services LLC is an official business partner of Dassault Systemes Americas Corp. Being one of very few Abaqus software business partners of Dassault Systemes Americas Corp., they can offer their customers fully licensed copies of the Abaqus software. You can also buy the software published by Dassault from this site.

> How to Buy

10.5. Ohio Supercomputer Center

The Ohio Supercomputer Center's Summer Institute delivers high-impact science-based learning to high school students across Ohio. OSC's ABAQUS license can only be used for educational, institutional, instructional, and/or research purposes. Only users who are faculty, research staff, or students at the following institutions are permitted to utilize OSC's license:

- The University of Ohio
- University of Toledo
- University of Cincinnati
- University of Dayton
- University of Akron

If you are a user from another academic institution may request to be added to this list per cost by contacting "OSC Help" and "Contact OSC Help" to get access to ABAQUS if you are a commercial user.

10.6. Cambridge University

From 24th June 2022, access to the full ABAQUS academic research suite licenses (seats and tokens) will change. Access will only be available to those users listed, and payment will need to be arranged before users are added to the list.

Access to a license (seats and tokens) costs £120 per year, with the license running from

1st November – 31st October. Licenses bought mid-year will be sold on a pro-rata basis (£10 per month). Users will have access to the general pool of licenses and tokens, so if a license/token isn't available on a rare occasion, they will need to wait until one is.

A teaching license is freely available to all undergraduates and those involved in teaching. There is no cost, but the license doesn't have the greater functionality of the full research suite license.

Abaqus

Accessing a licence Changes: From 24th June 2022 access to the full ABAQUS academic research suite licenses (seats and tokens) will change. Access will only be available to those users listed, and payment will need to be arranged before users are added to the list. Access to license pool for Engineering users: Access to a … Continue reading

Ⓦ Computing help and support   ⏻

10.7. ECN

ECN has recently moved into newly renovated office space in the Duncan Annex of the ECE Building. Abaqus has the same functionality as commercial licenses. They can be used for teaching or academic research.

There are classes that use Abaqus as part of the curriculum. If you are in a class that uses Abaqus but cannot get a license, please email app@ecn.purdue.edu for help.

The Abaqus software is only available to licensed users. Faculty or PIs must buy research licenses through ECN and provide a list of users (usually members of the research group) who are allowed to use them. Classes that use Abaqus must submit a list of students who need access to teaching licenses to ECN. One has to be in an access group to be able to run Abaqus.

https://engineering.purdue.edu/ECN/Support/KB/Docs/Abaquslicensing

The cases listed in this section are just some of the institutions that provide Abacus licenses. If you know another institution or site, be sure to write it in the comments on this post.

11. Tips for you

Now you should be all set to use the ABAQUS Student Edition. But to get the best use of

this edition and learn Abaqus like a pro, I recommend you check these packages out; I hope you find this helpful:

Please, click on the below images...

Additionally, this helpful video provides step-by-step guidance on how to gain access to the Abaqus student version. It covers the download and installation process, ensuring a smooth setup for your convenience.(**Abaqus student edition download**)

< NEWER   ⊞   OLDER >

ONE THOUGHT ON "HOW TO DOWNLOAD ABAQUS?|ABAQUS COMMERCIAL & STUDENT EDITION"

Juan says:

January 15, 2023 at 10:20 am

Hello, thank you for preparing this article, I really needed this installation guide, is it possible to install the commercial version in the same way?

LOG IN TO REPLY

LEAVE A REPLY

You must be logged in to post a comment.

ADDRESSES

⌖ Carrer de Jaume II ,46015,Valencia ,Spain
⌖ REON INTERNATIONAL GROUP LTD, 21 Hill Street, Unit 5, Haverfordwest, Dyfed, United Kingdom, SA61 1QQ (Sales Representative)
⌖ Enviroflex GmbH, Sterngasse 3/2/6 1010, Vienna, Austria (Sales Representative)

CAE Assistant
Your Assistant in CAE Challenges

With our assistance, consider your simulation project is done; we brought together a set of services and tutorial material to meet all your needs in CAE.

LINKS

- About Us
- Portfolio
- Contact Us
- Privacy Policy
- Terms & Conditions
- Cookie Policy
- Join Us
- FAQs

f 🐦 ▶ in