



Student Version Fluent Tutorial

Intro:

- This tutorial provides instructions on how to download and install student version of ANSYS software.



Download ANSYS Student

(1) Download **ANSYS Student** from [Ansys Student Versions | Free Student Software Downloads](#).

The screenshot shows the Ansys website's academic page. The browser address bar displays 'ansys.com/academic/students'. The navigation menu includes 'COUNTRIES & REGIONS', 'CONTACT US', 'CAREERS', 'STUDENTS AND ACADEMIC', 'CUSTOMER CENTER', and 'FREE TRIALS'. The main content area features the Ansys logo and a search bar. Below the logo, a paragraph states: 'Used by millions around the world, students can take advantage of our free engineering software for homework, capstone projects and student competitions. Our renewable products can be downloaded at no cost by students across the globe and installed on any supported MS Windows 64-bit machine.' Three software options are presented in a grid:

- Ansys Student**: Includes Ansys Workbench-based bundle of Ansys Mechanical, Ansys CFD, Ansys Autodyn, Ansys Discovery, Ansys SpaceClaim, and Ansys DesignXplorer. A red box highlights the 'FREE DOWNLOAD NOW' button.
- Ansys LS-DYNA Student**: Features a vast array of capabilities using its explicit solver for simulating material response to severe loading and extreme deformation.
- Ansys Electronics Desktop Student**: Labeled as 'NEW', it includes industry gold standard simulators for antenna, RF, Microwave, PCB, IC, and IC package designs.

A chat icon is visible in the bottom right corner of the page.



Download ANSYS Student

(2) Click Download ANSYS Student 2021 R2 to start downloading

ansys.com/academic/students/ansys-student

COUNTRIES & REGIONS CONTACT US CAREERS STUDENTS AND ACADEMIC CUSTOMER CENTER FREE TRIALS

Why Ansys Products & Services Learn

Ansys

Search All Ansys

Home > Students & Academic > Students > Ansys Student Software: Ansys Student

Ansys Student - Free Software Download

Ansys Student offers free access to our Ansys Workbench-based bundle. This bundle includes Ansys Mechanical, Ansys CFD, Ansys Discovery, Ansys SPEOS, Ansys Autodyn, Ansys DesignXplorer, and Ansys SpaceClaim. Used by students across the globe, Ansys Student can be leveraged to enhance your skill set with some of our most-used products.

From 2023R1 on, Ansys Discovery is now included in Ansys Student. Ansys Discovery offers free access to our simulation-driven design tool that combines instant physics simulation, proven Ansys high-fidelity simulation and interactive geometry modelling in a single user experience. This is the perfect product for a student just getting started with simulation—it lets students learn about physics without the time investment required to learn how to use a complex simulation tool.

Terms of Use: Free student downloads are for educational use only and may only be used for self-learning, student instruction, student projects, and student demonstrations.

DOWNLOAD ANSYS STUDENT 2023 R2

(Built-in license valid until 07/31/2024)

For the [free online simulation course from Cornell University](#), Ansys Student 2022 R2 is recommended.

QUICK LINKS

- Learning Forum
- Innovation Courses
- Learning Resources
- Student Teams

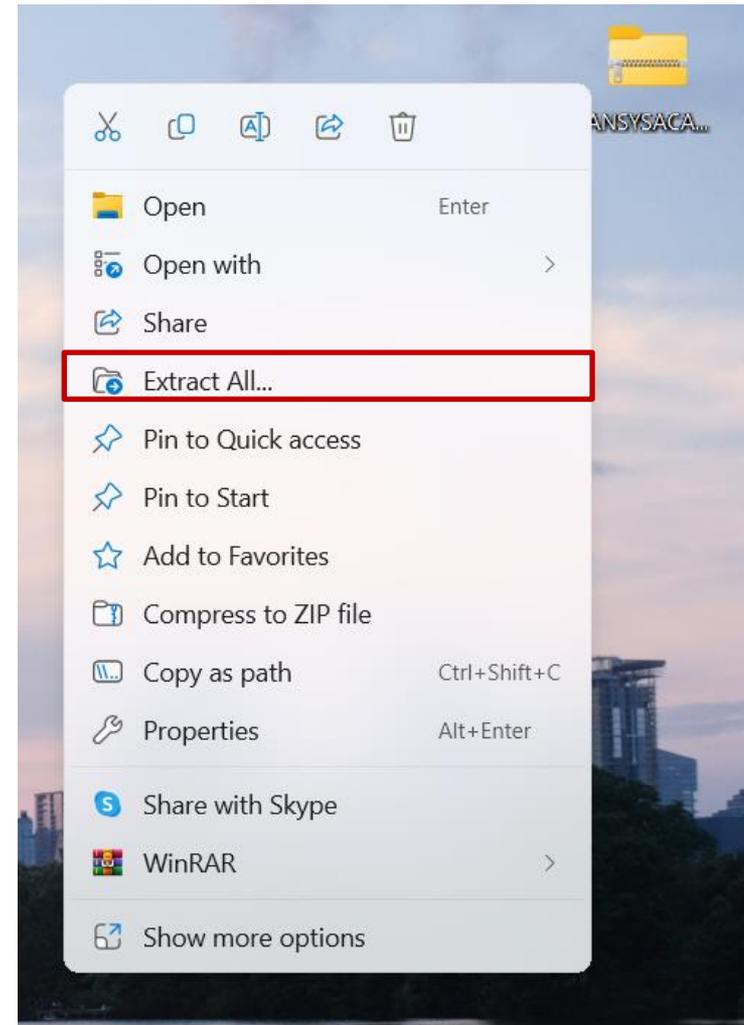
Would you like to speak with a human right now?



Extract installer

(3) Right Click

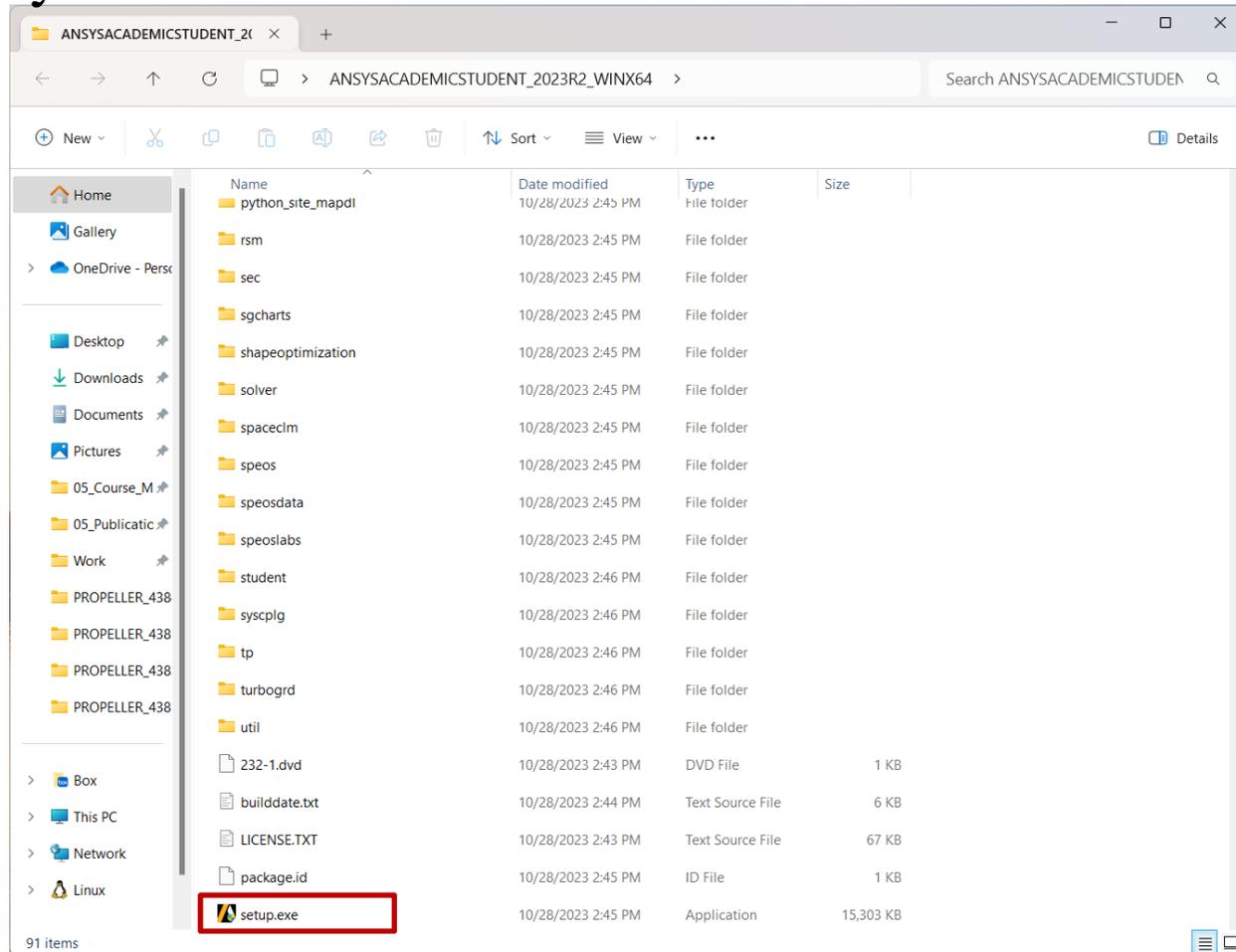
“ANSYSACADEMICSTUDENT_2023R2_WINX64.zip” and click on “Extract All”, as shown in the screenshots.





Install ANSYS Student

(4) Double-click on the setup.exe in the extracted folder to start installing Ansys Student.





Install ANSYS Student

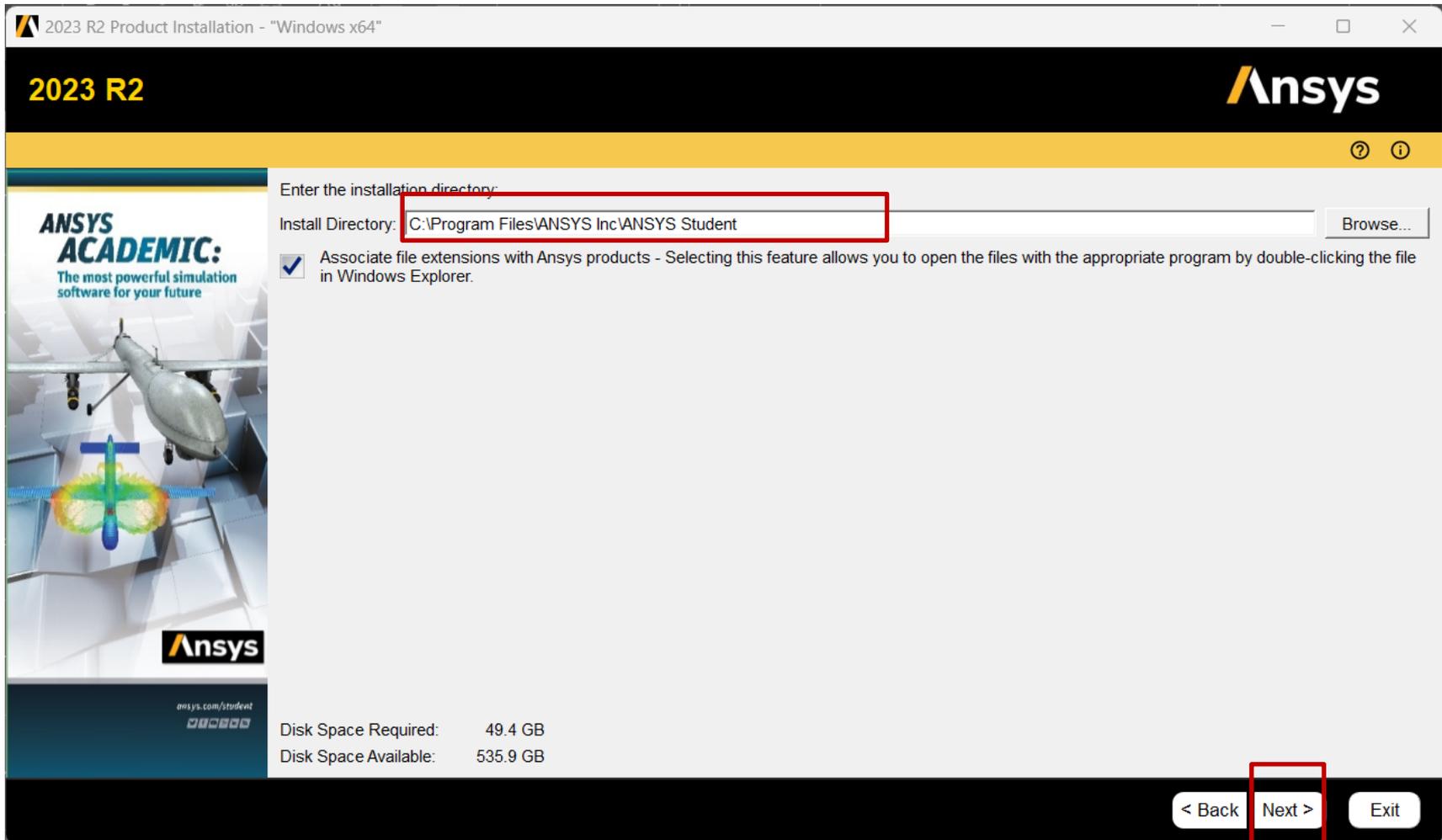
(5) Follow the steps to install Ansys Student on your PC.

The screenshot shows the '2023 R2 Product Installation - "Windows x64"' window. The interface features a yellow header with the '2023 R2' version number and the Ansys logo. On the left, there is a promotional graphic with the text 'SIMULATION IS THE FUTURE OF ENGINEERING' and an image of a smartwatch. The main area displays the 'CLICKWRAP SOFTWARE LICENSE AGREEMENT' for Version October 31, 2022. The agreement text includes terms regarding previous license agreements, embedded compliance technology, and the requirement to read and understand Section 11. Below the text, there are two radio button options: 'I agree to the terms of the license agreement, including, to the extent applicable, the cross-border transfer of data and collection of geographic location data under Section 11.' (which is selected) and 'I do not agree'. At the bottom right, there are three buttons: '< Back', 'Next >' (highlighted with a red box), and 'Exit'.



Install ANSYS Student

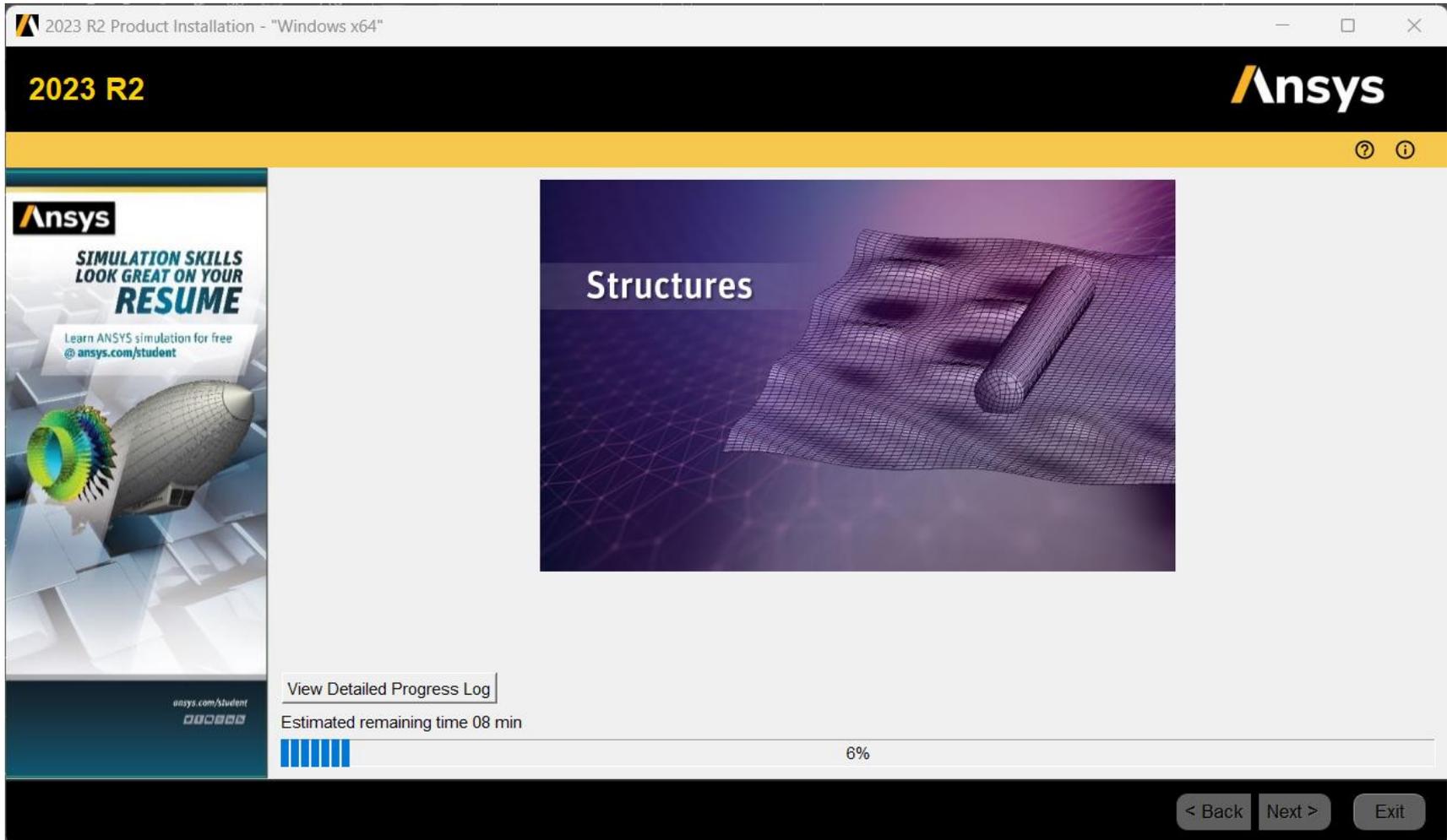
(5) Follow the steps to install Ansys Student on your PC.





Install ANSYS Student

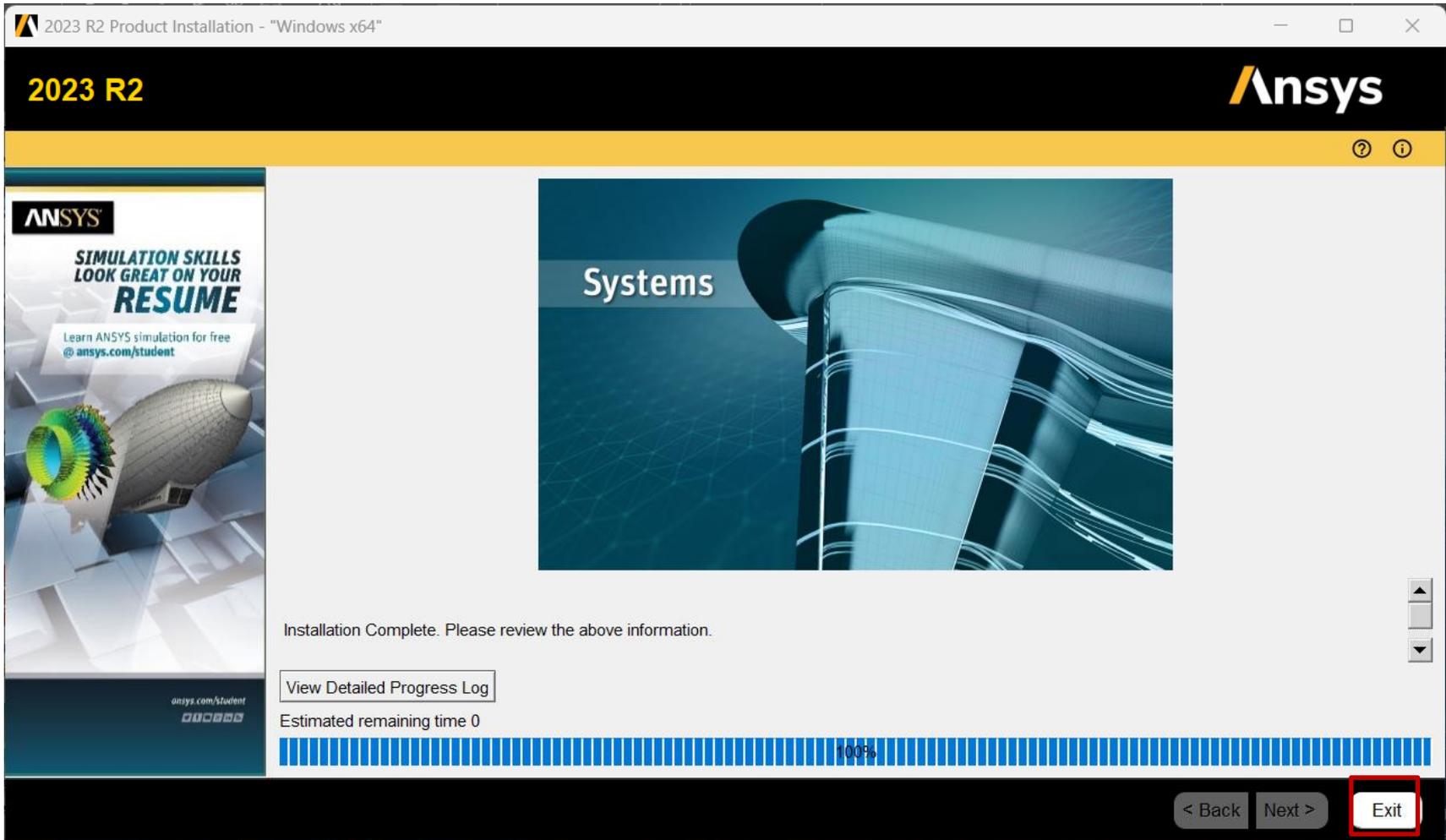
(5) Follow the steps to install Ansys Student on your PC.





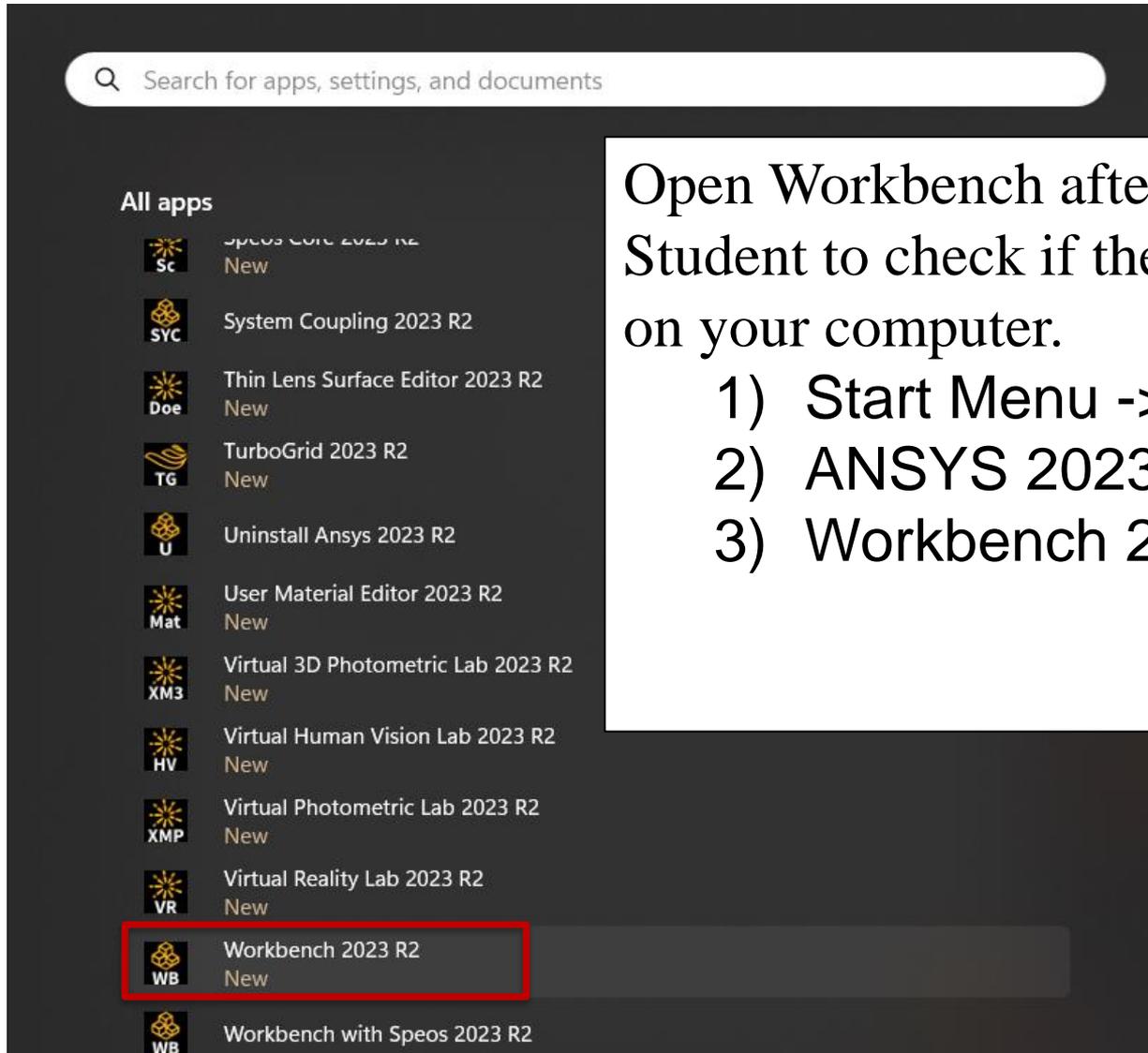
Install ANSYS Student

(5) Click Exit to finish the installation





Open ANSYS Fluent



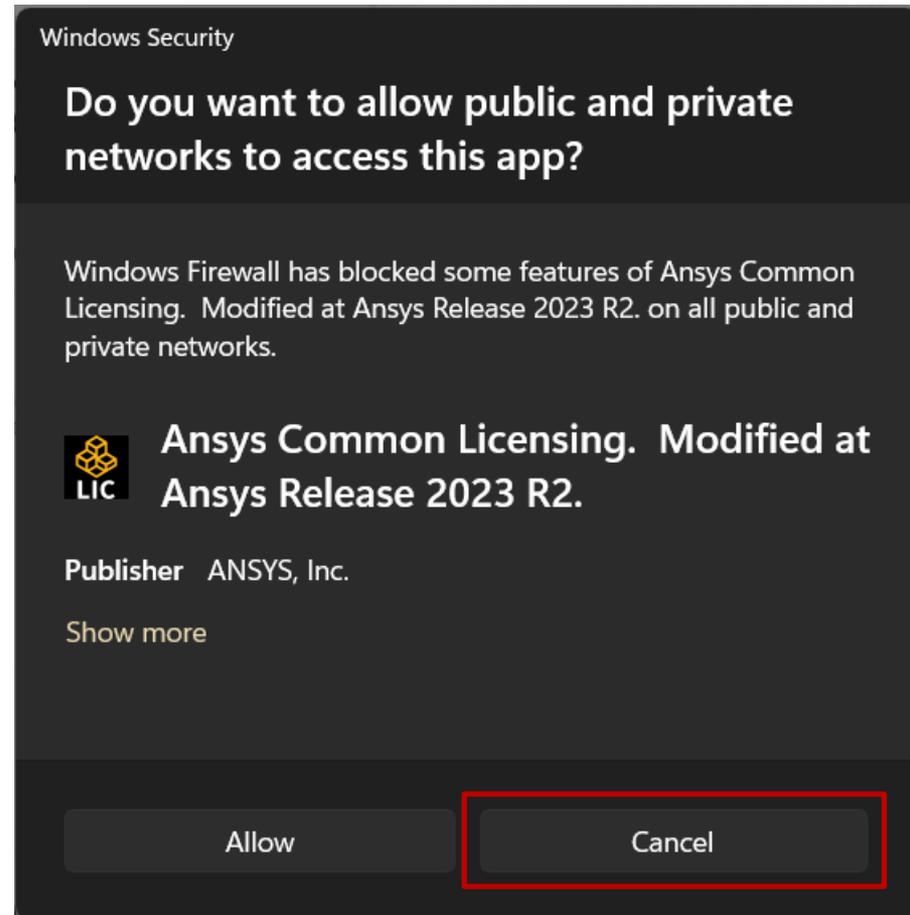
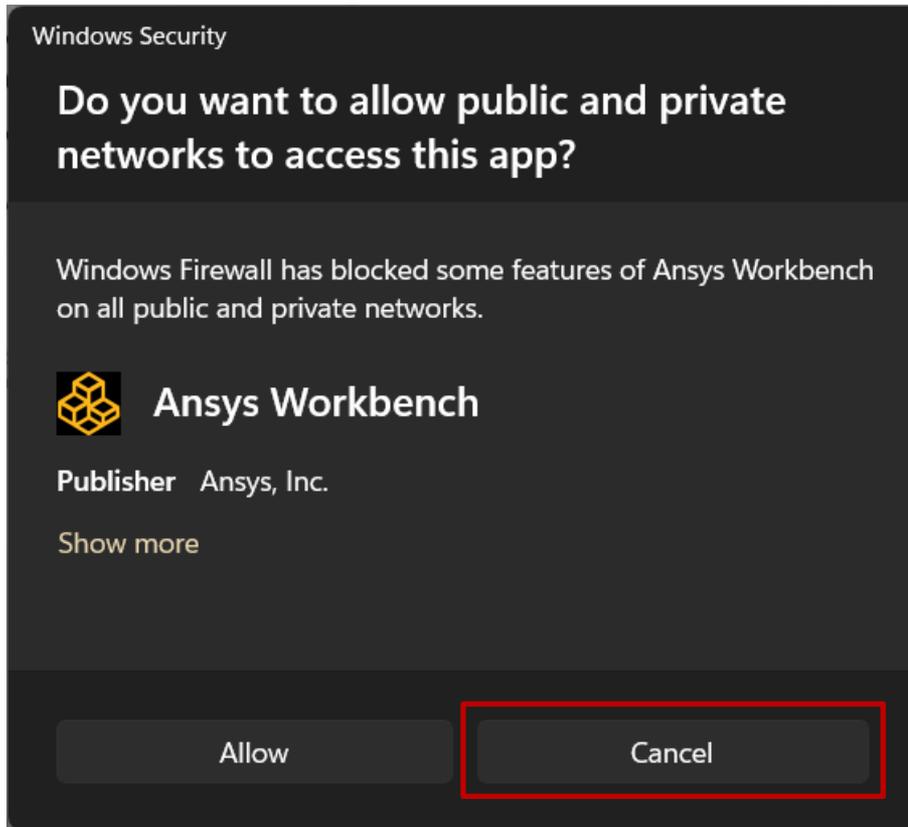
Open Workbench after you install ANSYS Student to check if the software works well on your computer.

- 1) Start Menu ->
- 2) ANSYS 2023 R2 ->
- 3) Workbench 2023 R2



Open ANSYS Fluent

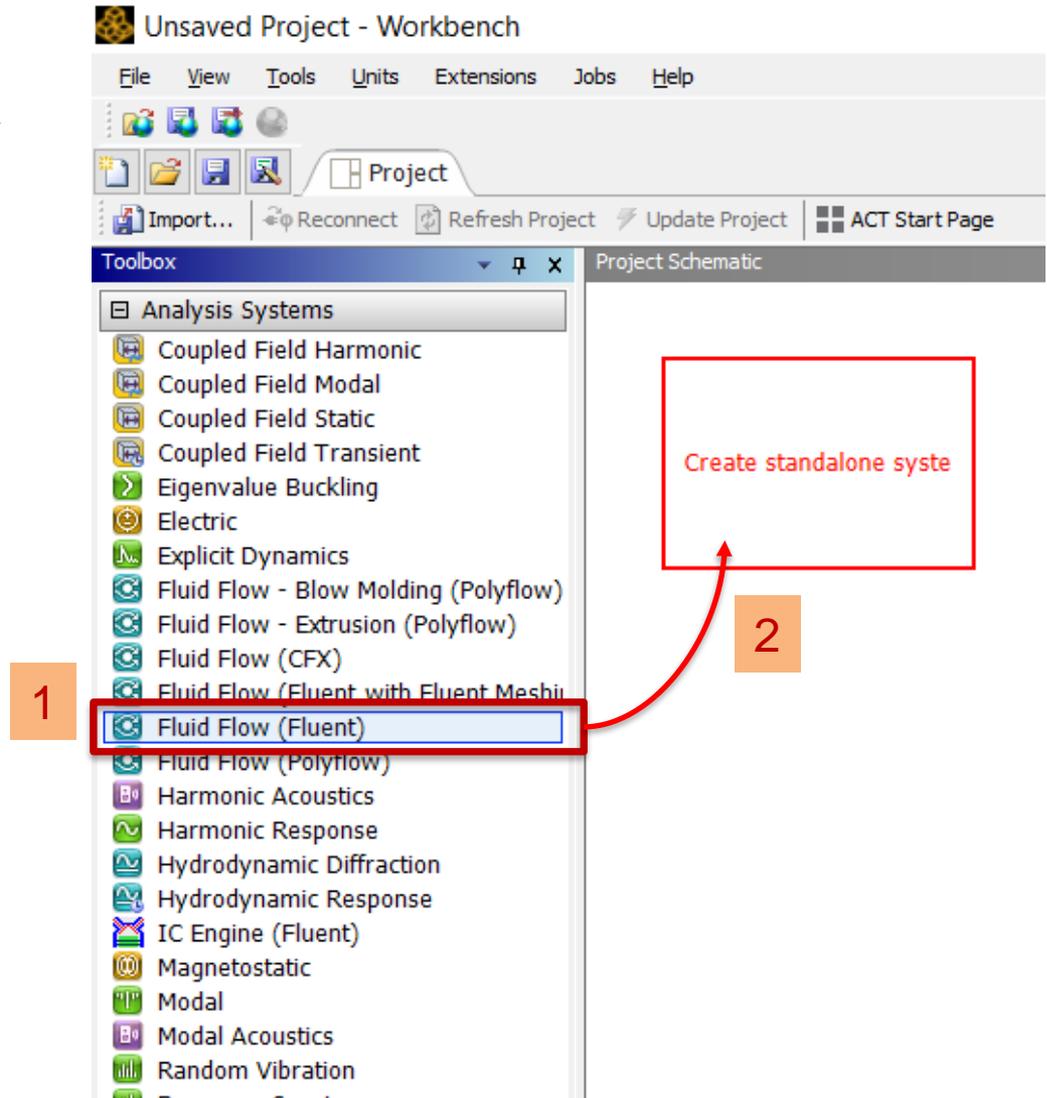
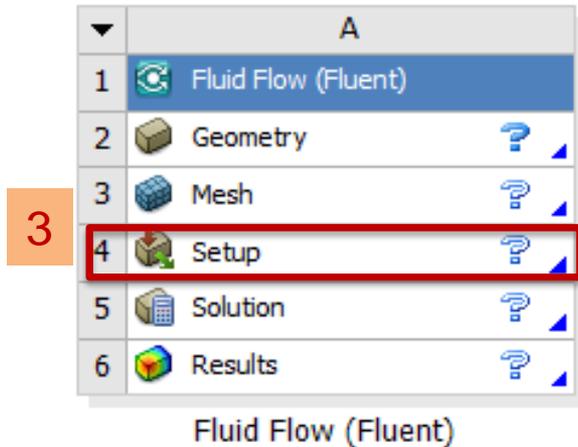
Left click “Cancel” if you see any prompt



Open ANSYS Fluent

Once you open the Workbench you can find Fluent in the left toolbox

- 1) Click and hold the **Fluid Flow (Fluent)**
- 2) Drag it into **Project Schematic**
- 3) Double click on **Setup**





Open ANSYS Fluent

Now you are ready to start Term Project Part A.

The screenshot displays the ANSYS Fluent software interface. The title bar indicates the file path: "A:\Fluid Flow (Fluent) Parallel Fluent@ONGLAISOO [3d, dp, pbns, lam, single-process] [CFD Solver - Level 2, CFD Base, CFD Solver - Level 1]". The main menu includes File, Domain, Physics, User-Defined, Solution, Results, View, Parallel, and Design. The toolbar contains various icons for meshing and simulation. The left sidebar shows the Outline View with a tree structure: Setup (General, Materials), Solution, Results (Surfaces, Graphics, Plots, Dashboard, Animations, Reports), Parameters & Customization, and Simulation Reports. The central Task Page is set to "Mesh" and includes sections for General (Scale..., Check, Report Quality, Display..., Units...), Solver (Type: Pressure-Based, Density-Based; Velocity Formulation: Absolute, Relative), Time (Steady, Transient), and Gravity. The right sidebar shows the Ansys 2023 R2 STUDENT logo and a vertical toolbar. The bottom Console window displays the following text: "MPI Option Selected: intel", "Cleanup script file is C:\Users\Thomas\cleanup-fluent-ONGLAISOO-82984.bat", and a prompt ">".