



Ansys Fluent Student Versions

- Introduction:

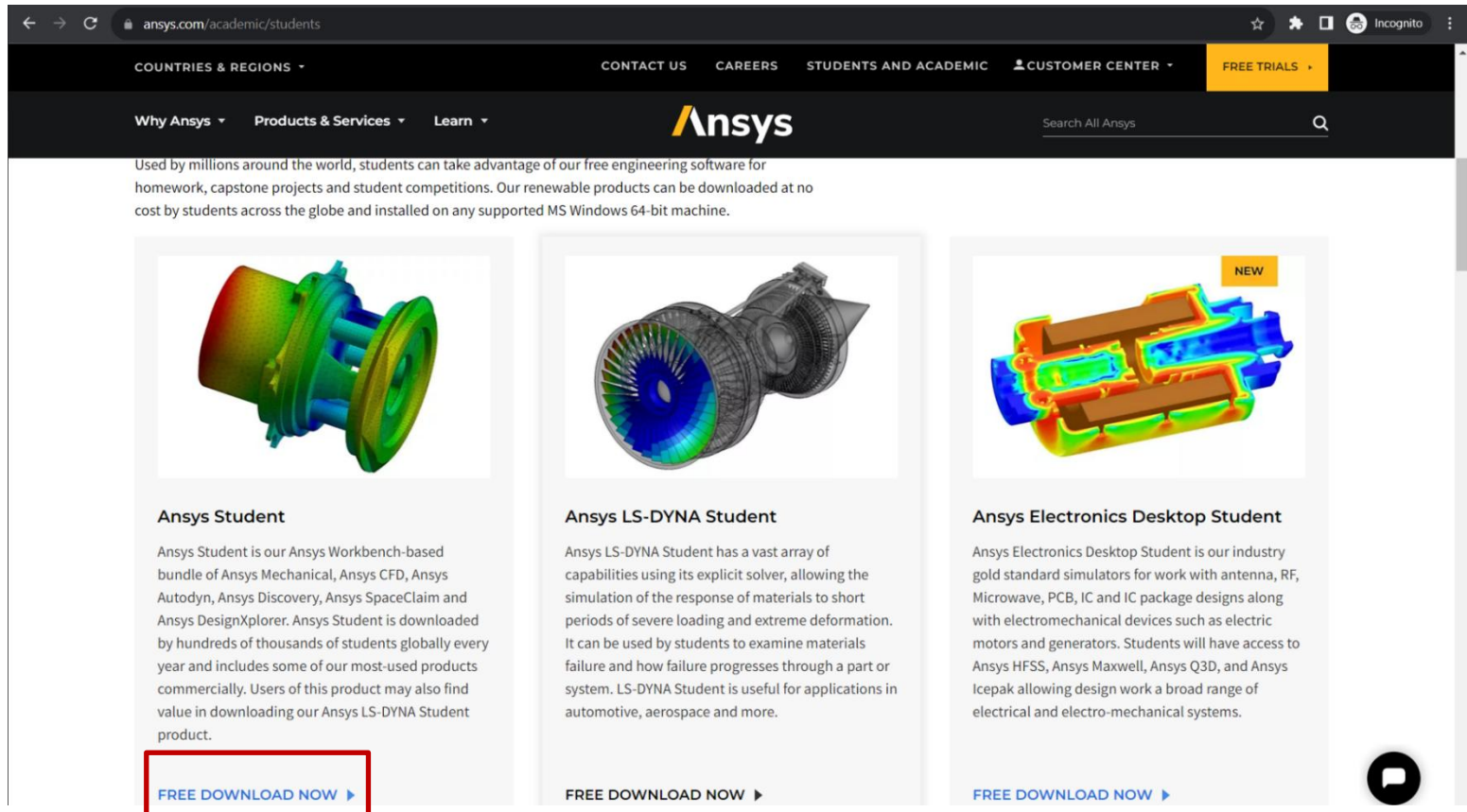
This tutorial provides step-by-step instructions on how to download and install the free student version of ANSYS Fluent.

- Requirements:

1. Windows 10/11 64-bit - Minimum 4GB RAM
2. 50GB+ free disk space
3. Internet connection for download and verification

How to Download Ansys Student

- (1) Visit the official Ansys website: [Ansys Student Versions | Free Student Software Downloads](https://www.ansys.com/academic/students).
- (2) Click on "FREE DOWNLOAD NOW"



The screenshot shows the Ansys website's academic page. The header includes navigation links like "COUNTRIES & REGIONS", "CONTACT US", "CAREERS", "STUDENTS AND ACADEMIC", "CUSTOMER CENTER", and a "FREE TRIALS" button. The main content area features three software options:

- Ansys Student**: Described as a Workbench-based bundle of Ansys Mechanical, Ansys CFD, Ansys Autodyn, Ansys Discovery, Ansys SpaceClaim, and Ansys DesignXplorer. It is downloaded by hundreds of thousands of students globally every year. A red rectangle highlights the "FREE DOWNLOAD NOW" button.
- Ansys LS-DYNA Student**: Described as having a vast array of capabilities using its explicit solver, allowing the simulation of the response of materials to short periods of severe loading and extreme deformation. It can be used by students to examine materials failure and how failure progresses through a part or system. The "FREE DOWNLOAD NOW" button is visible at the bottom.
- Ansys Electronics Desktop Student**: Described as an industry gold standard simulator for work with antenna, RF, Microwave, PCB, IC and IC package designs along with electromechanical devices such as electric motors and generators. Students will have access to Ansys HFSS, Ansys Maxwell, Ansys Q3D, and Ansys Icepak. The "FREE DOWNLOAD NOW" button is visible at the bottom.



How to Download Ansys Student

(3) Click on the “DOWNLOAD ANSYS STUDENT 2025 R2” button to download the installer

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

Why Ansys ▾ Products & Services ▾ Learn ▾

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 Zemax OpticStudio, Ansys Autodyn, Ansys optiSLang, Ansys Rocky, and much more (see “What’s Included” below for a complete list). Used by students across the globe, Ansys Student can be leveraged to enhance your skill set with some of our most-used products.

From 2025R2 on, Ansys Zemax OpticStudio is now included in Ansys Student. Ansys Zemax OpticStudio Student offers free access for students to gain hands-on experience in designing and analyzing optical systems, preparing them for future careers in optics. Students will have access to an intuitive interface and powerful tools with extensive materials and optical elements libraries, enabling them to simulate real-world scenarios accurately, aiding their understanding of optics fundamentals, and enhancing their problem-solving skills.

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 2025 R2 ▸

(Built-in license valid until 07/31/2026)

For the [free online simulation course from Cornell University](#), Ansys Student 2025 R2 is recommended.

Ansys Rocky Add-ins can be downloaded from [here](#).

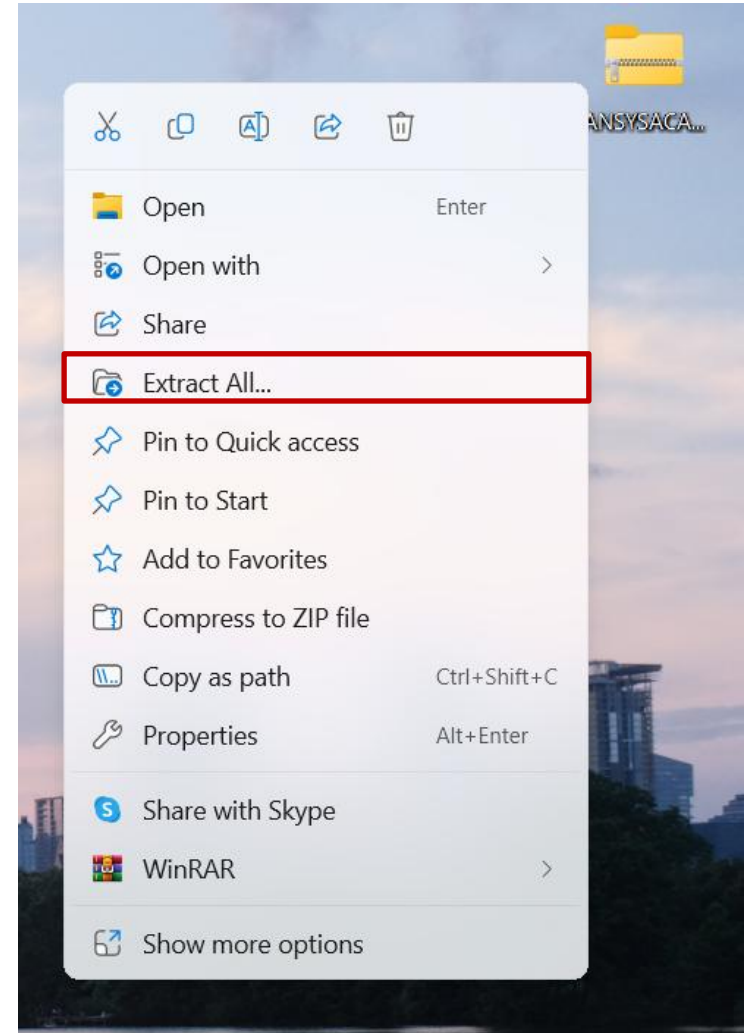
QUICK LINKS

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

Hello! I'm here to assist you. What can I help you find?

Extract Ansys installer

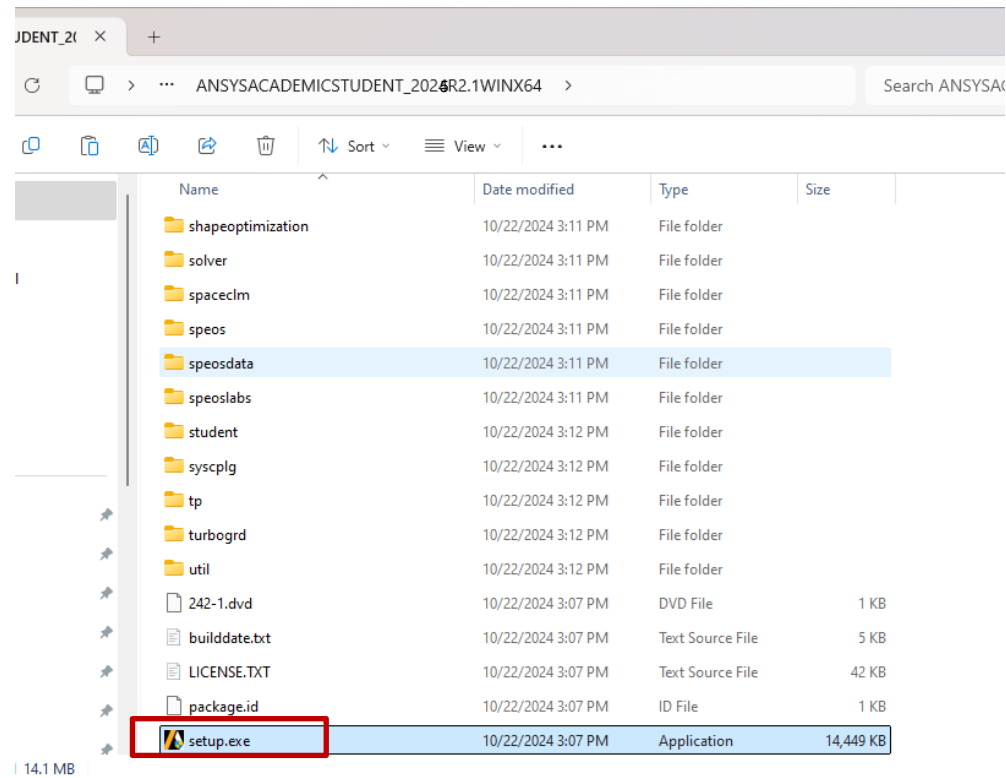
- (1) Locate the downloaded file
- (2) Extract the files:
 - Right-click on the zip file
 - Select "Extract All..." from the context menu





Install Ansys Student

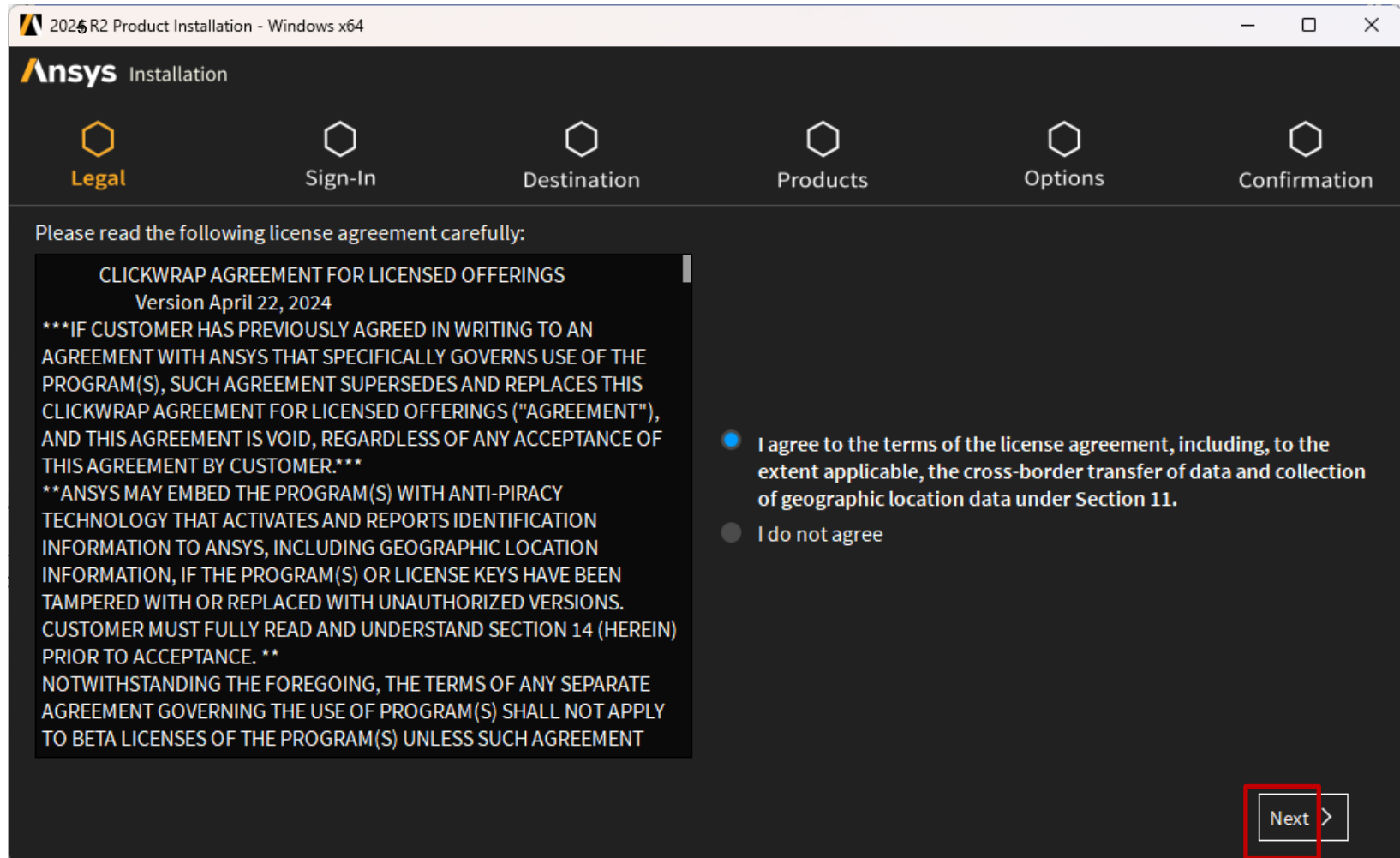
- (1) Navigate to extracted folder
- (2) Locate "setup.exe" file
- (3) Right-click and select "Run as administrator"
 - If prompted by Windows security, click "Yes"





Install Ansys Student

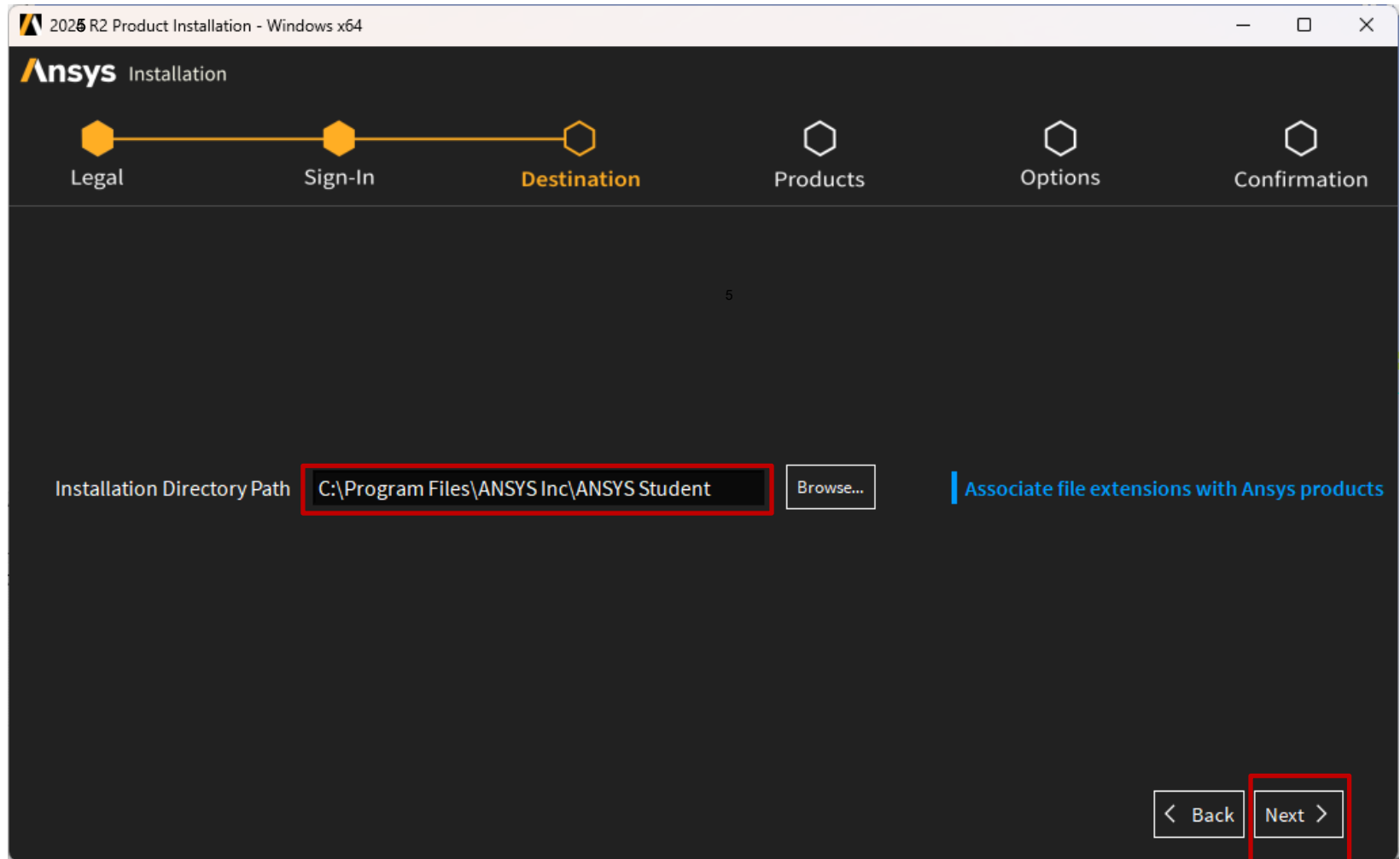
(4) Follow the steps to install Ansys Student on your PC.





Install Ansys Student

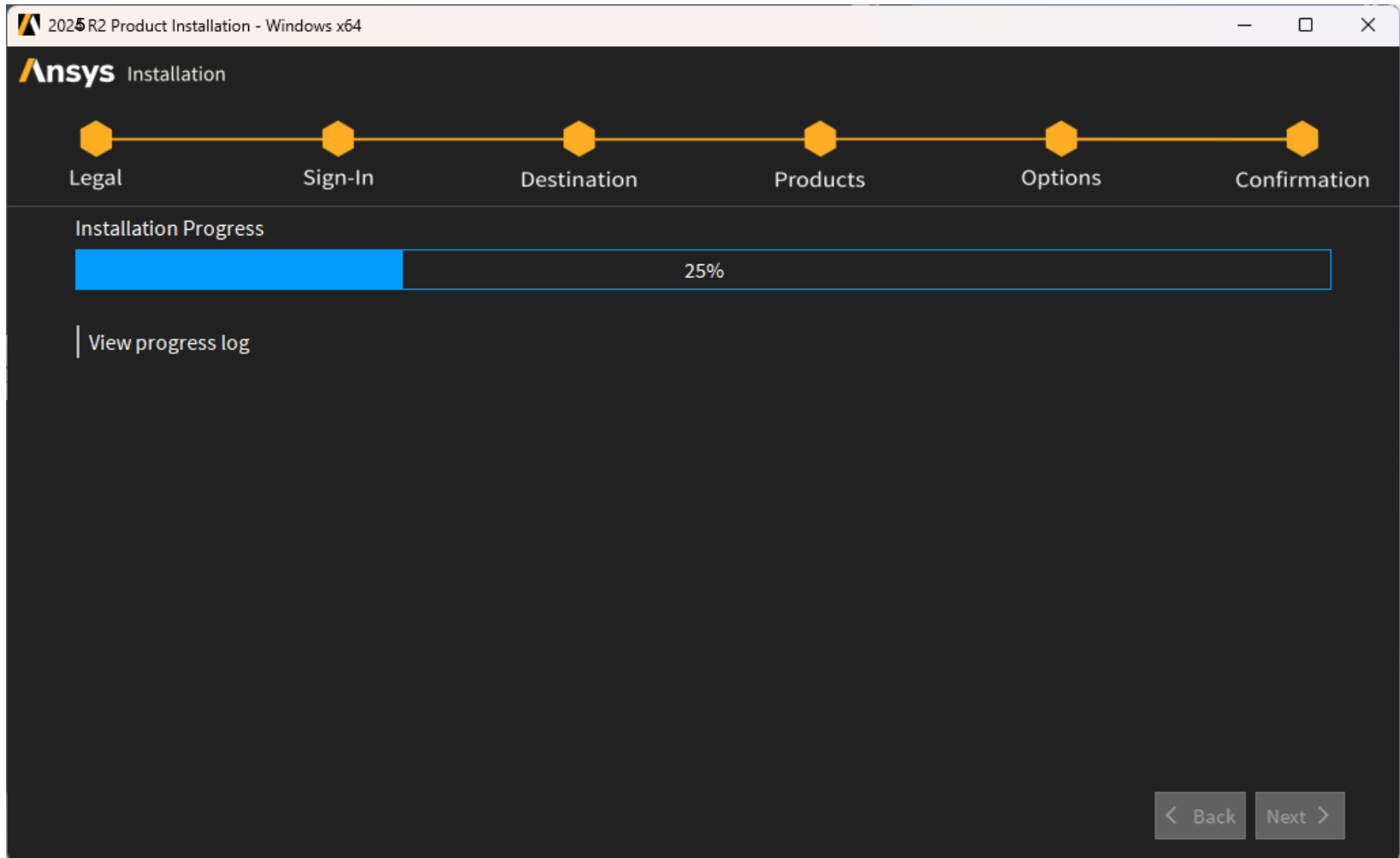
(5) Follow the steps to install Ansys Student on your PC.





Install Ansys Student

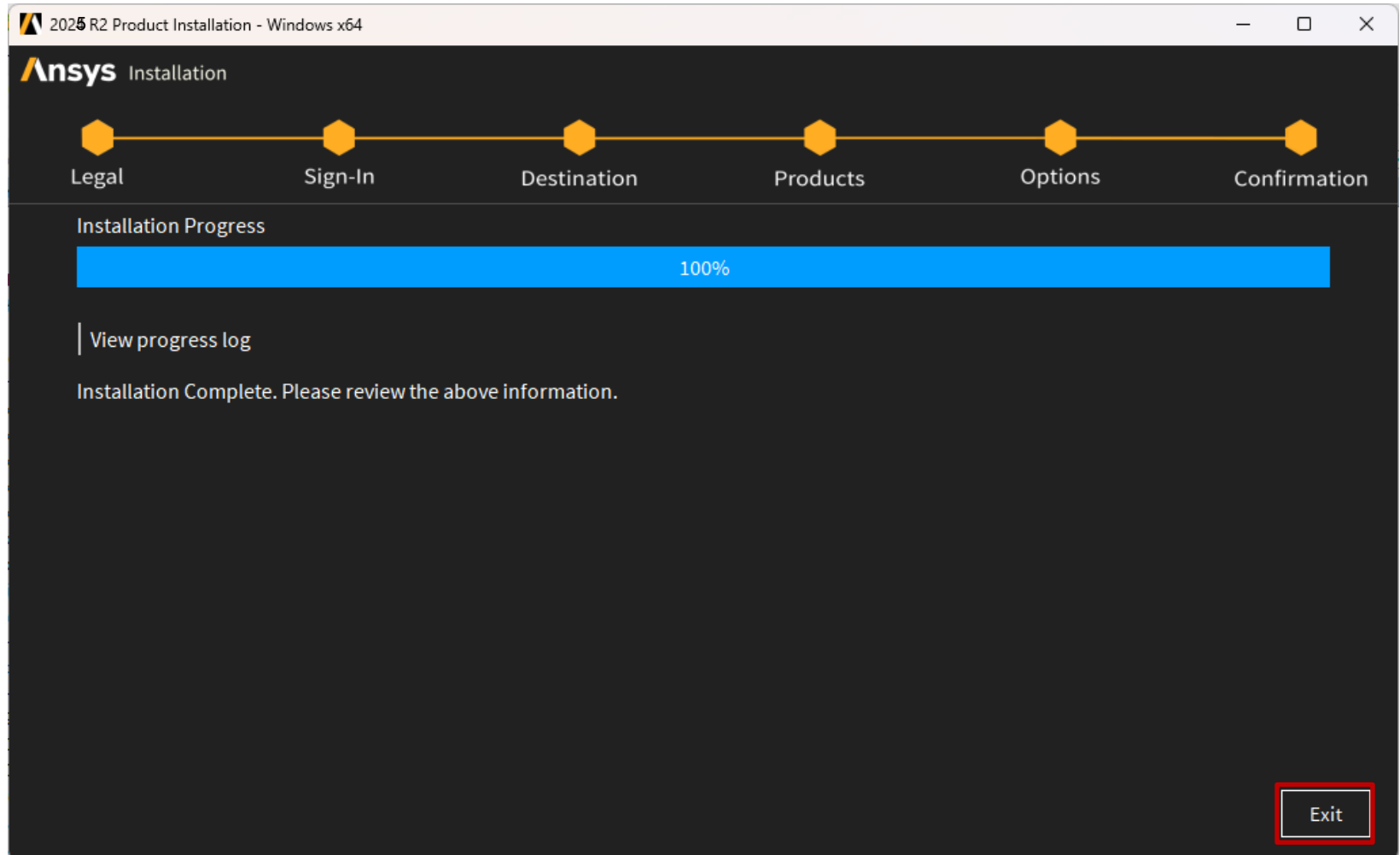
(6) Follow the steps to install Ansys Student on your PC.



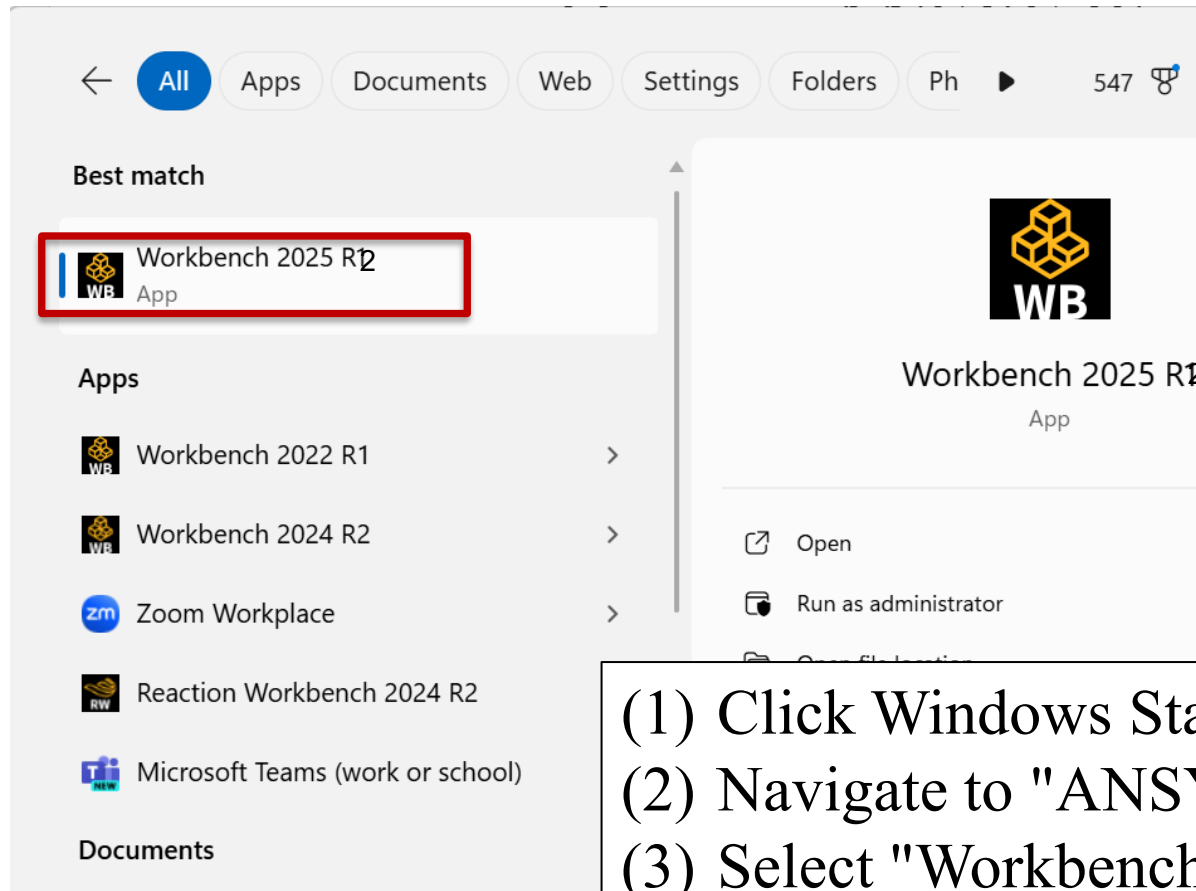


Install Ansys Student

(7) Click Exit to finish the installation



Opening ANSYS Workbench

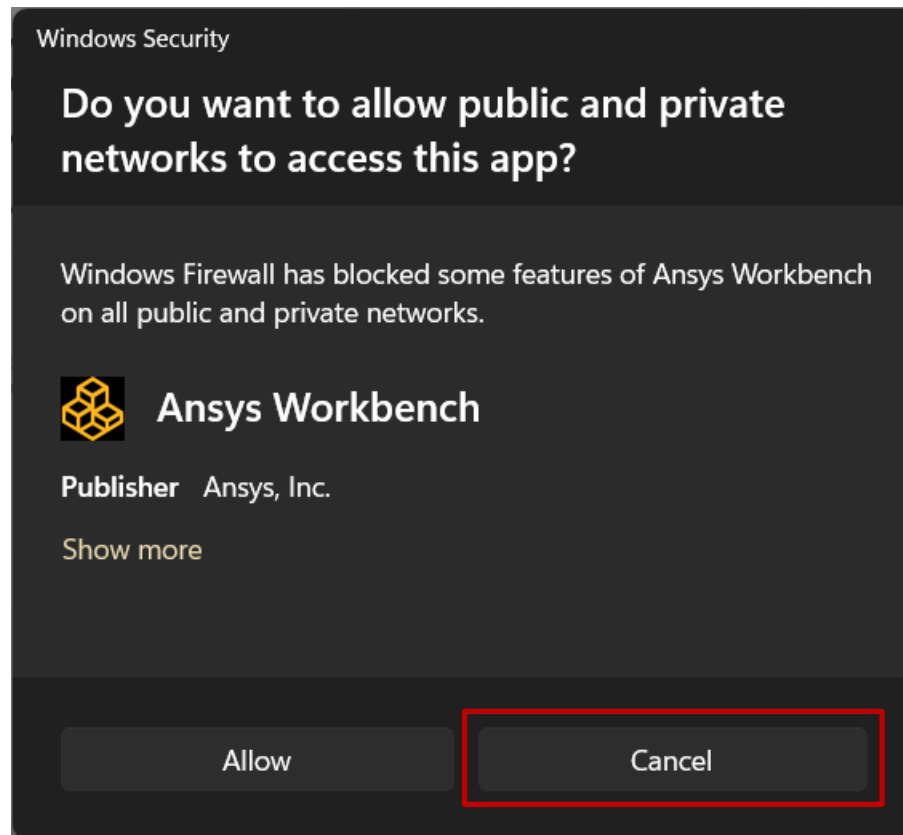


- (1) Click Windows Start Menu
- (2) Navigate to "ANSYS 2025 R2" folder
- (3) Select "Workbench 2025 R2"
 - Initial loading may take 2-3 minutes



Opening ANSYS Workbench

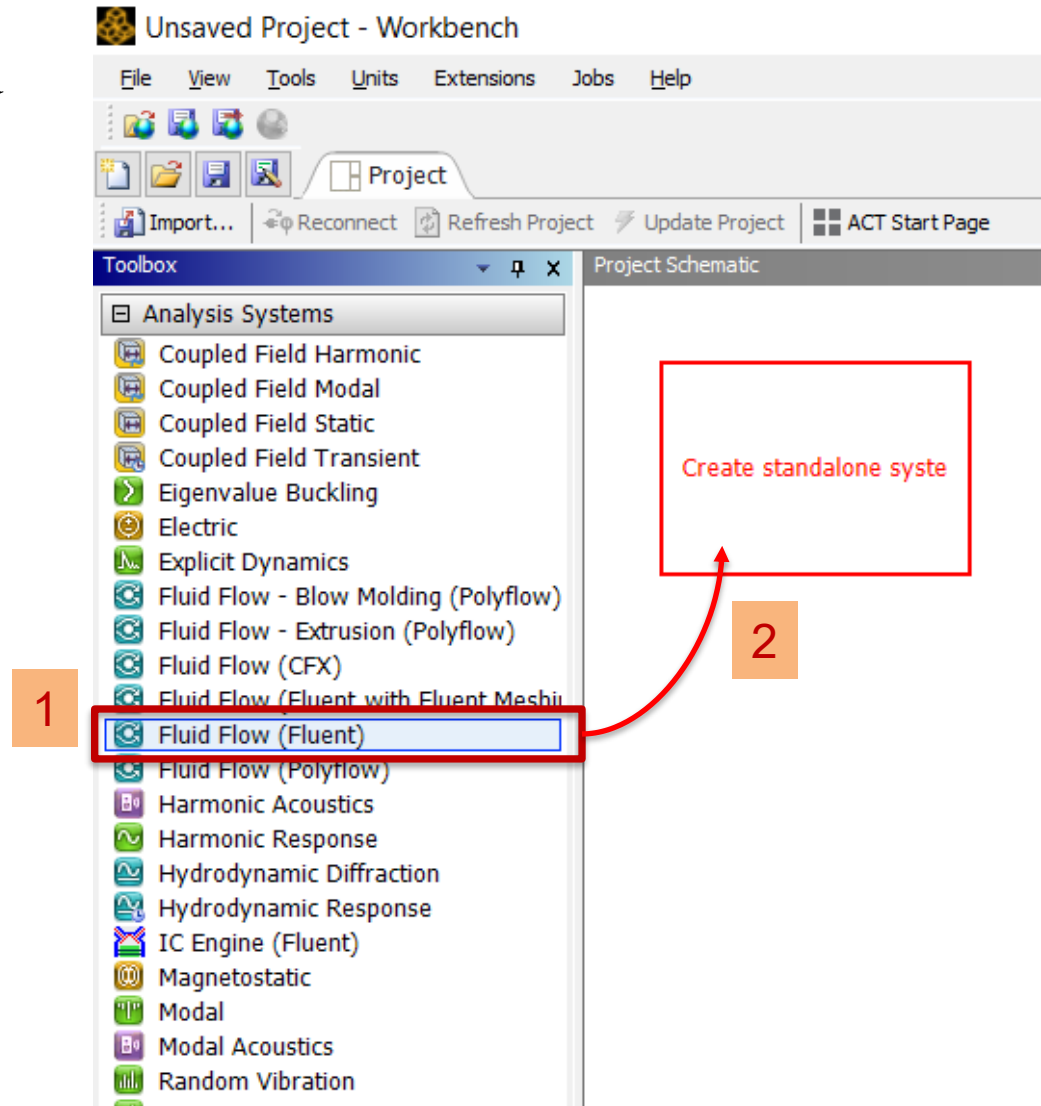
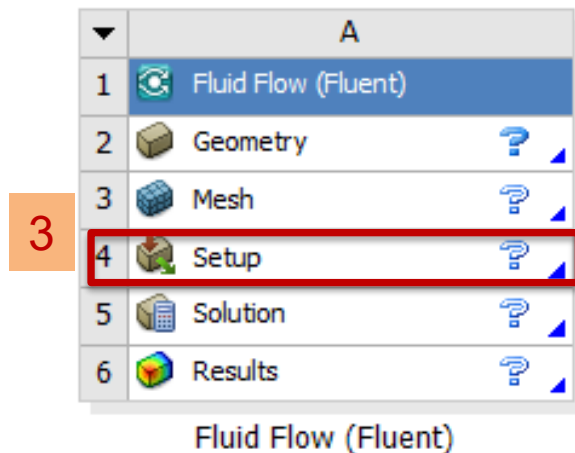
Left-click “Cancel” if you see any Windows Security prompt



Opening 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**





Ready to Begin!

Installation complete! You're now ready to start Term Project Part A.

AsFluid Flow (Fluent) Parallel Fluent@ONGLAISOO [3d, dp, pbns, lam, single-process] [CFD Solver - Level 2, CFD Solver - Level 1, CFD Base]

File Domain Physics User-Defined Solution Results View Parallel Design Learning and Support Quick Search (Ctrl+F) Ansys

Mesh Display... Info Units... Check Quality Make Polyhedra Scale... Transform

Zones Combine Separate Adjacency... Delete... Deactivate... Activate... Append Replace Mesh... Replace Zone...

Interfaces Mesh... Overset...

Mesh Models Dynamic Mesh... Gap Model...

Turbomachinery Turbo Models Turbo Create... Turbo Performance... Spectral Content Turbo Workflow Turbo Topology... Periodic Instanting... Adapt Surface

Outline View Filter Text

- Setup
 - General
 - Materials
- Solution
- Results
 - Surfaces
 - Graphics
 - Plots
 - Dashboard
 - Animations
 - Reports
- Parameters & Customization
- Simulation Reports

Task Page

General

Mesh Scale... Check Report Quality Display... Units...

Solver

Type ☒ Pressure-Based ☐ Density-Based

Velocity Formulation ☒ Absolute ☐ Relative

Time ☒ Steady ☐ Transient

☐ Gravity

User Window 1

Ansys 2024 R2 STUDENT

0 selected all

Console

```
Host spawning Node 0 on machine "ONGLAISOO" (win64).

-----
ID      Hostname  Core  O.S.      PID   Vendor
-----
n0      ONGLAISOO  1/16  Windows-x64  43156  11th Gen Intel(R) Core(TM) i7-11800H @ 2.30GHz
host    ONGLAISOO           Windows-x64  47280  11th Gen Intel(R) Core(TM) i7-11800H @ 2.30GHz

MPI Option Selected: intel

-----

Cleanup script file is C:\Users\Thomas\cleanup-fluent-ONGLAISOO-47280.bat
```