



Powering Innovation That Drives Human Advancement

Ansys Student Team Partnerships Guide

Contents

- [/ Overview of student team package](#)
- [/ Training and certification](#)
- [/ Unique ways Ansys can help your team](#)
- [/ How to get the software](#)
- [/ Setup guidance](#)
- [/ Standard licensing provided to student teams](#)
- [/ How the licensing works](#)
- [/ Firewall considerations](#)
- [/ Hardware considerations](#)
- [/ Tech support](#)
- [/ Download high-definition logos](#)



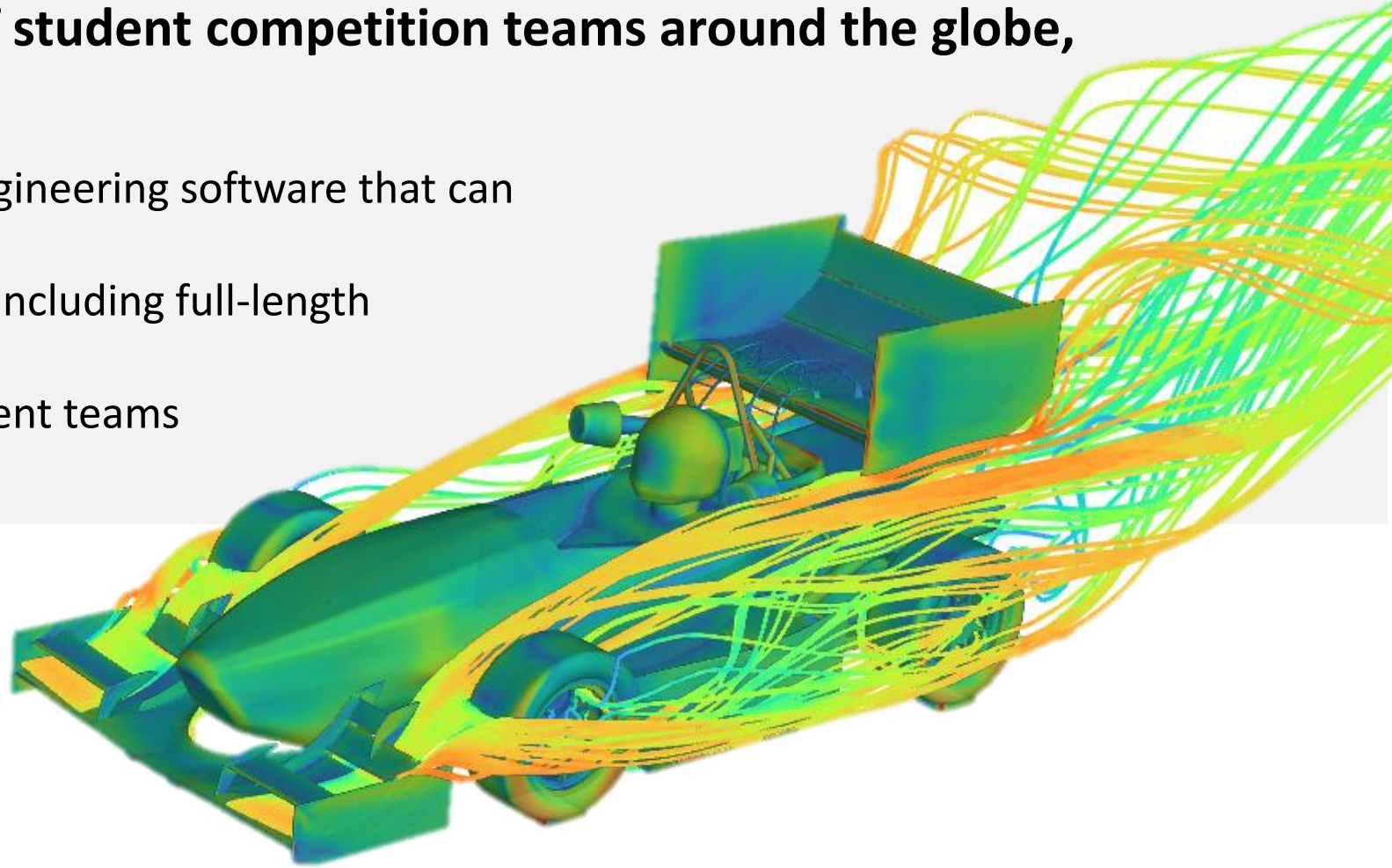
**CLICK FOR
LINKS**



Gain a Competitive Edge as an Ansys Student Team Partner

/ Ansys Partners with hundreds of student competition teams around the globe, offering them the following:

- **Free commercial versions** of our engineering software that can be put on your machines
- Free access to our training content including full-length courses, video tutorials and more
- Resources built specifically for student teams



Learn more at [ansys.com/teams](https://www.ansys.com/teams)

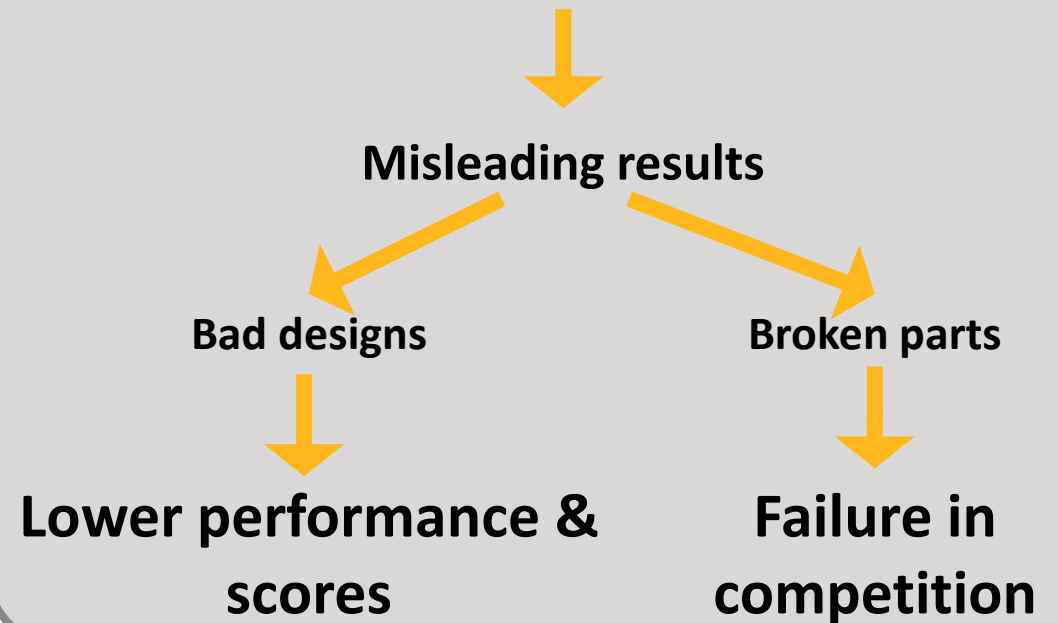
Strong Simulation Training is Essential for Student Teams

Other simulation software usually lacks:

- Concise tutorials for your competition
- Best practices for specific applications
- Training that establishes a fundamental understanding of how to approach simulation (not just the clicks)

The impact:

- Massive time investment (training)
- Improper simulation setup



Ansys Supports Student Team Training

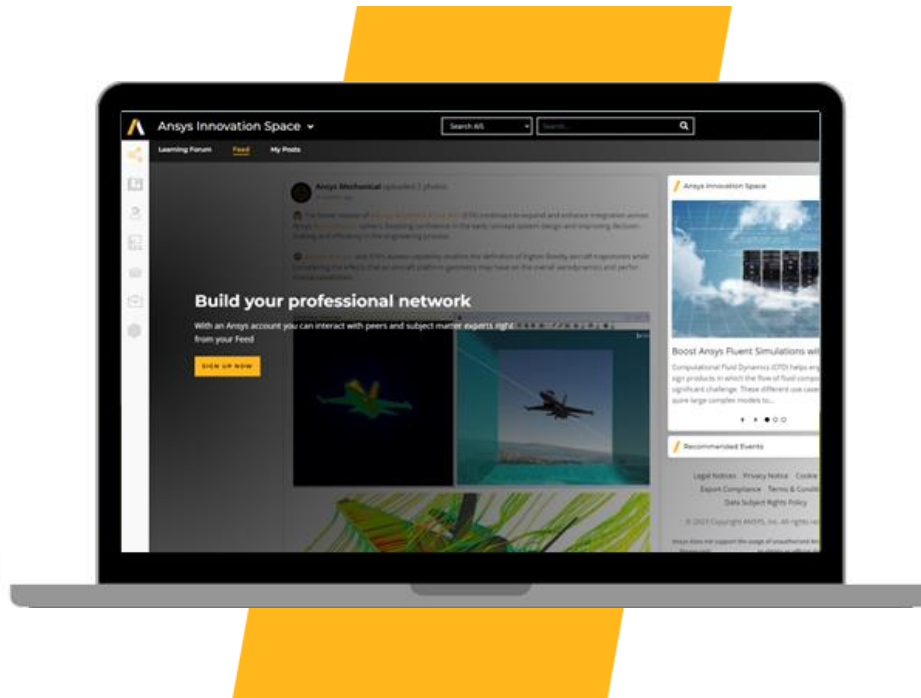
Ansys offers **free, public training content** online through Ansys Innovation Courses.
visit **[Ansys.com/courses](https://www.ansys.com/courses)**

/ Student Team Tutorials

Competition-specific student team courses to get you up and running with Ansys

/ Simulation Fundamentals and Best Practices

Learn the basics of simulation and best practices to implement



/ Basic Ansys Product Training

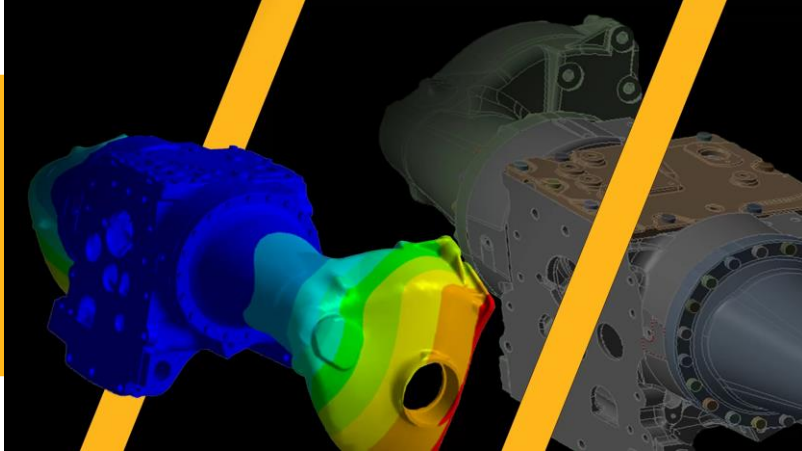
How-to style courses getting you familiar with product interfaces and capabilities

/ Physics Fundamentals

Go back to the basics with courses covering core physics topics

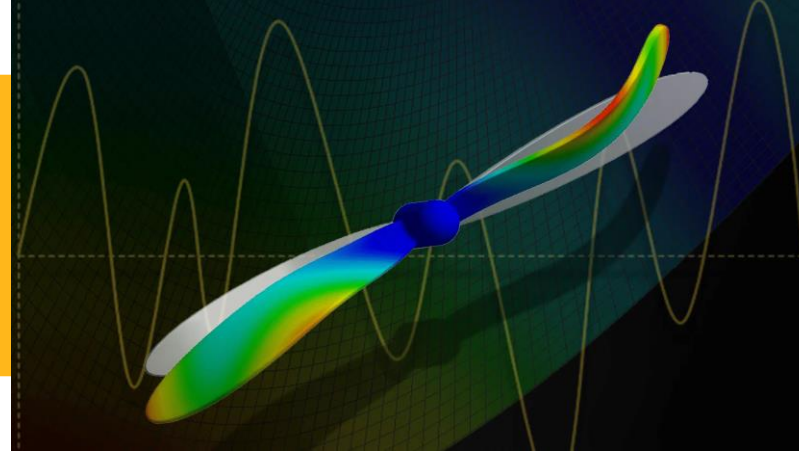
/ And more content on how to leverage simulation to solve engineering problems

Examples Courses on FEA Fundamentals: Ansys Innovation Courses



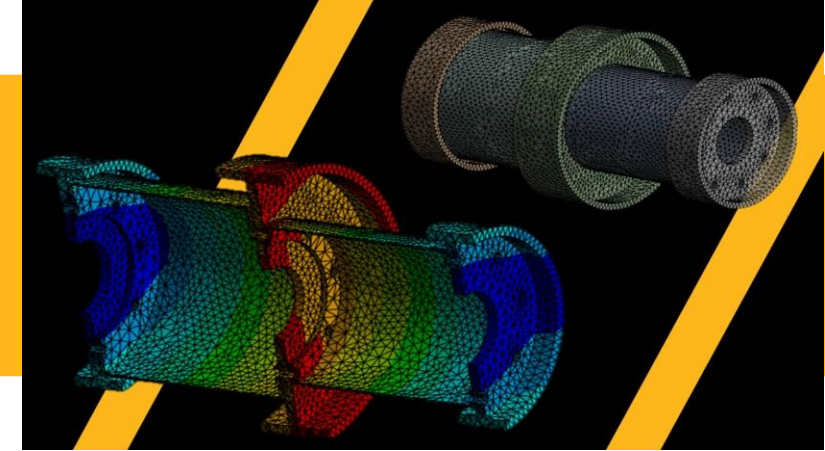
/ Stress Analysis Using Ansys Mechanical

- Geometry representation
- Connecting parts together
- Structural boundary conditions
- Numerically accurate results
- Results insight



/ Linear Dynamics Using Ansys Mechanical

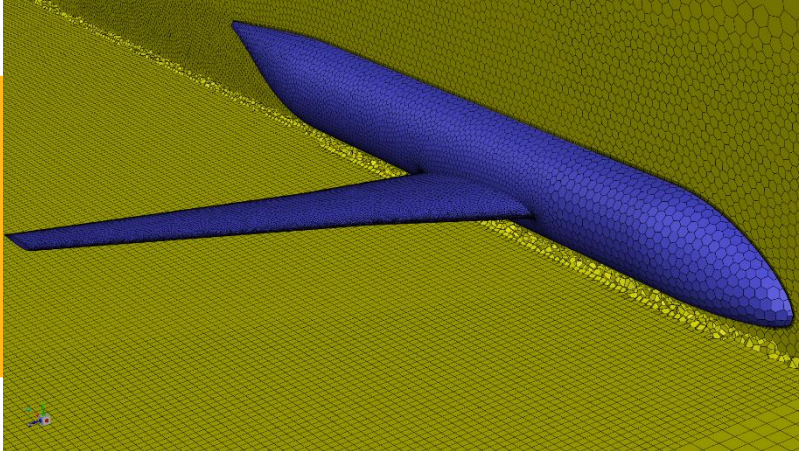
- Modal analysis
- Modal-based methods
- Harmonic response analysis
- Mode superposition transient analysis
- Random vibration analysis
- Single-point response spectrum analysis



/ Pre-loaded Bolted Connections Using Ansys Mechanical

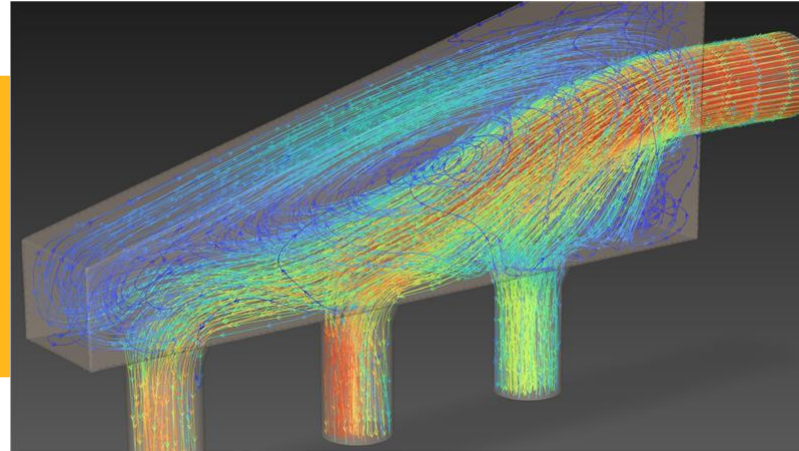
- Pre-loaded bolted joint analysis
- Modeling the bolt and pre-load
- Connecting bolts with assembly
- Best practices and results validation

Examples Courses on CFD Fundamentals: Ansys Innovation Courses



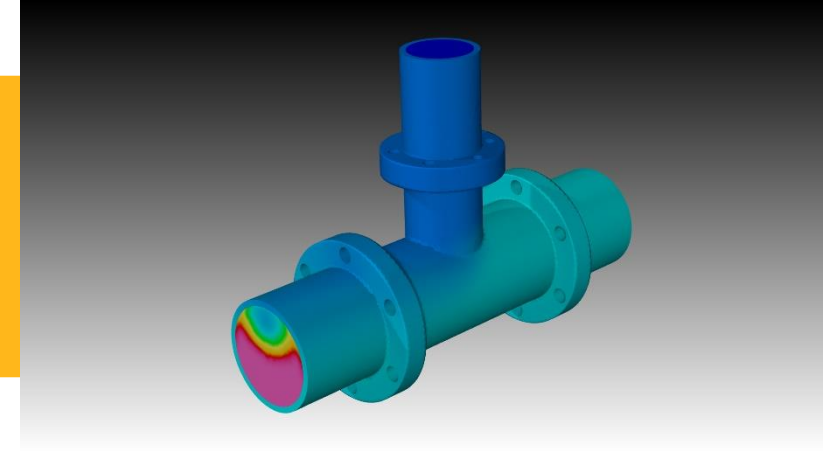
/ Ansys Fluent Meshing Watertight Geometry Workflow

- User interface
- Import geometry
- Add local sizing
- Generate surface mesh
- Describe geometry
- Add boundary layers
- Generate volume mesh



/ Getting Started with Ansys Fluent - Basics

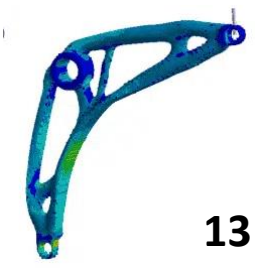
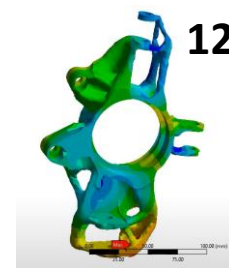
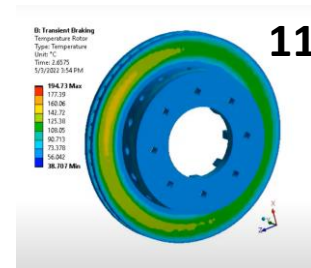
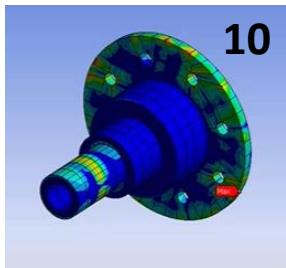
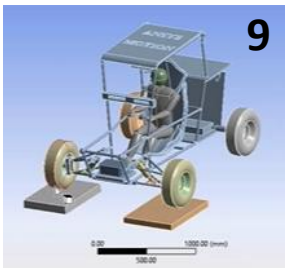
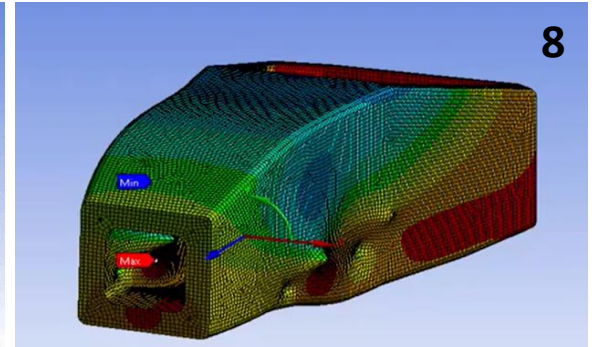
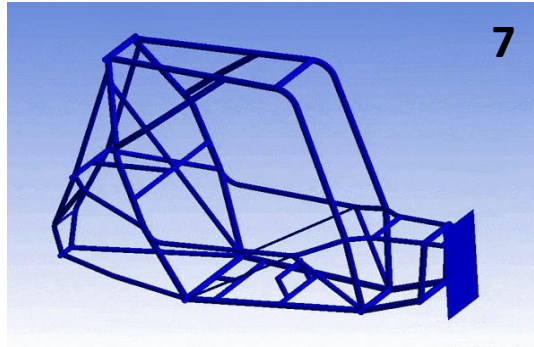
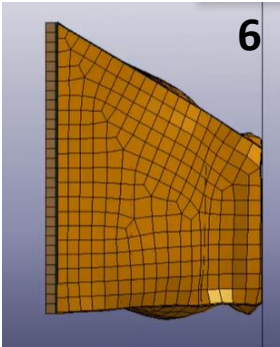
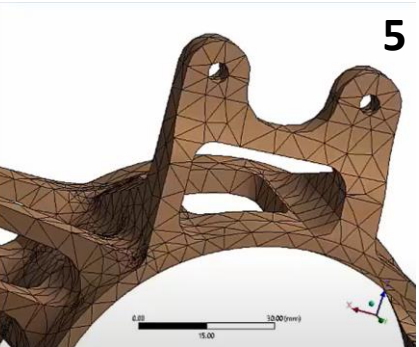
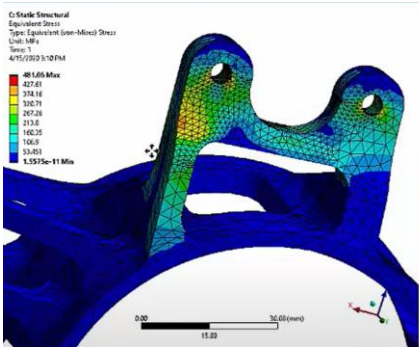
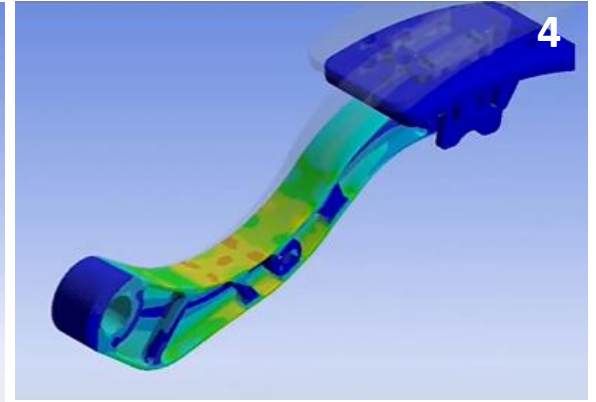
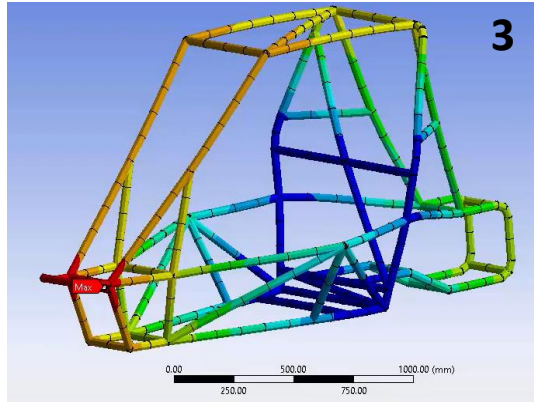
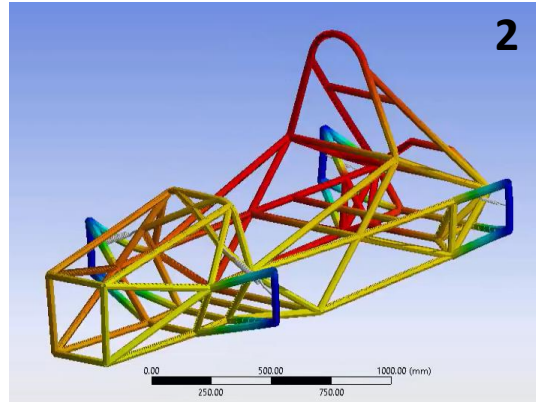
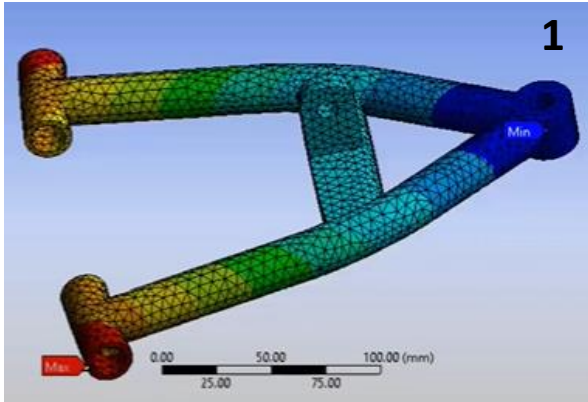
- Intro to Ansys Fluent
- Geometry prep for CFD simulations
- Mesh generation
- Physics setup
- Customization
- Post-processing
- Best practices for CFD simulation (see [webinar](#) for student teams)



/ Getting Started with Ansys Fluent – Beyond the Basics

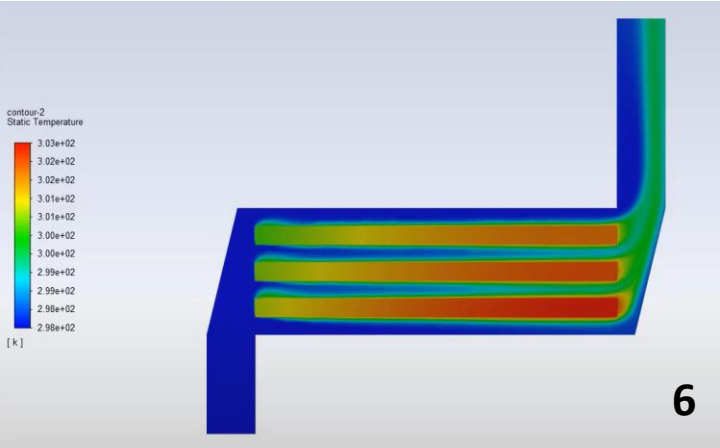
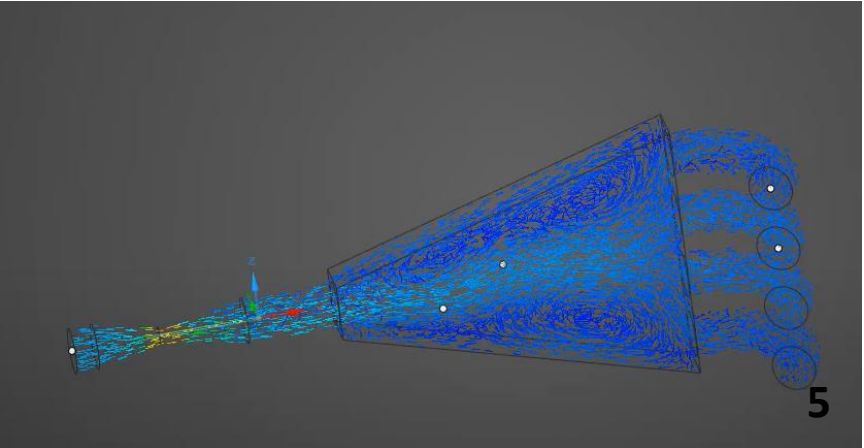
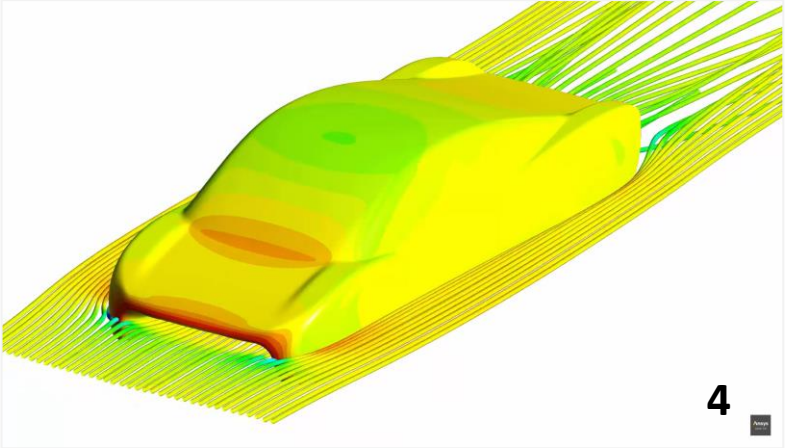
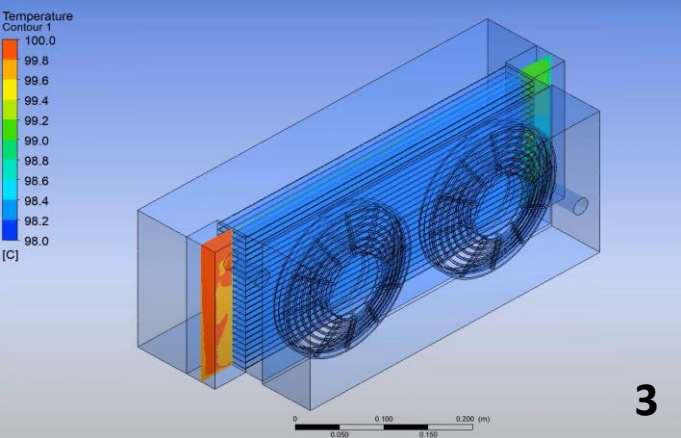
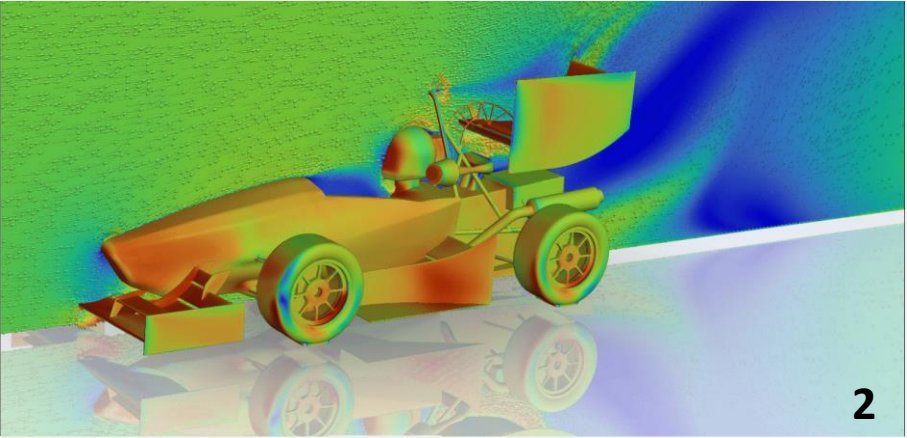
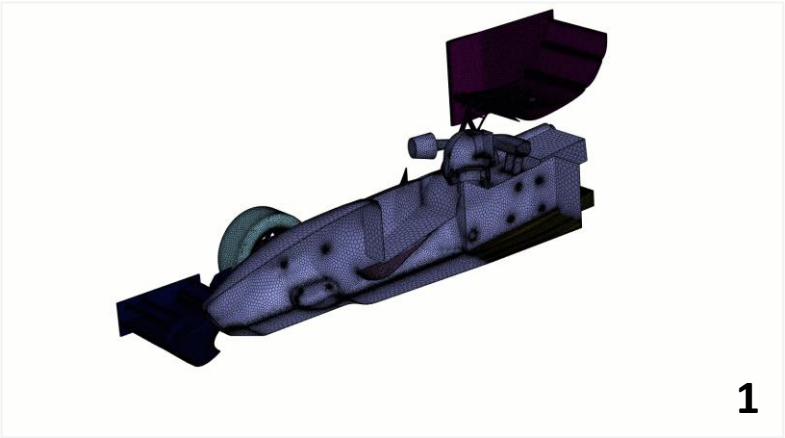
- Turbulence modeling
- Heat transfer modeling
- Transient flow modeling

Sampling of FEA Tutorials for Student Teams



1. Suspension control arm fatigue
2. Formula chassis
3. Baja chassis (impact, modal, torsion)
4. Brake pedal structural analysis
5. Rear upright static structural
6. Impact attenuator
7. Baja dynamic chassis
8. Composite monocoque chassis
9. Front suspension motion
10. Fatigue on an FSAE hub
11. Brake thermo-structural
12. Suspension upright topology optimization
13. Bell crank topology optimization

Sampling of CFD tutorials for Student Teams



- 1. Formula car aerodynamics
- 2. Formula car aerodynamics
- 3. Radiator heat transfer
- 4. Solar car aerodynamics
- 5. Intake manifold
- 6. Battery thermal with equivalent circuit model and reduced order modeling

Ansys Learning Hub: In-Depth Ansys Training

Ansys Learning Hub (ALH) provides users with **on-demand, in-depth learning** on the Ansys product portfolio. This content expands beyond the public content available on Ansys Innovation Courses.

/ Filter in-depth courses

- By product (ex: Mechanical, Fluent, HFSS, Discovery, etc.)
- By application (ex: CFD, FEA, composites, battery design, etc.)
- By course level (introductory or advanced)
- By course discipline (ex: fluids, structures, electromagnetics, materials, etc.)
- See full course list [here](#).

/ Get Access

Contact your academic representative or academic@ansys.com to request access.



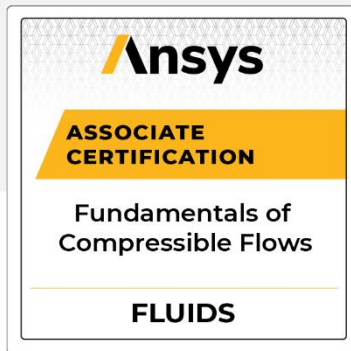
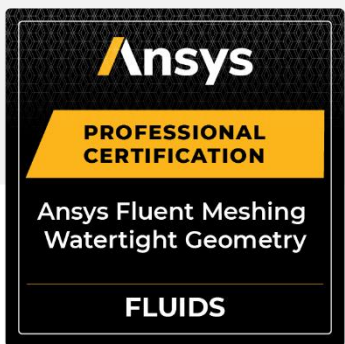
Ansys Certifications and Course Completion Badges

Learning products designed with the power of engineering simulation

Showcase your skills on your online profile and enhance your resume

/ Certifications

- Offered in fluids, structures, and electronics
- Two levels: Associate and Professional Certification
- Successful completion comes with digital certification
- Easily sharable on professional networks and resume



/ Course Completion Badges

- Offered for all Ansys Innovation Courses (AIC)
- Proof of successful completion of AIC and Tracks
- Links to description of credentials earned
- Easily sharable on professional networks and resume



Unique Ways Ansys Can Help: Material Database and Selection

Ansys Granta Selector can:

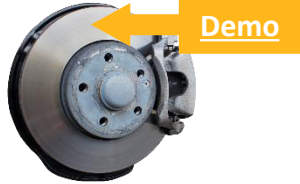



Increase design scores by showing judges a well thought out engineering process that applies engineering goals and constraints to find optimal materials and documents this (report, charts, etc.).

Reduce material costs while increasing performance.

Consider all aspects of material behavior in the real world (ex: Temperature vs strength).

Identify manufacturers of a particular grade of metal to help with price comparisons and shopping.

Transfer verified materials data directly into Ansys simulation software with a click.

Example application	Goals
	<ol style="list-style-type: none"> 1. Min cost & weight 2. Max compressive strength, stiffness, wear resistance, operating temp.
	<ol style="list-style-type: none"> 1. Min cost & weight 2. Max corrosion resistance 3. Min thermal expansion
	<ol style="list-style-type: none"> 1. Min cost & weight 2. Min shear
	<p>Determine which matrix, fiber, fiber volume, ply structure, etc.</p>
<p>See overview webinar for student teams.</p>	

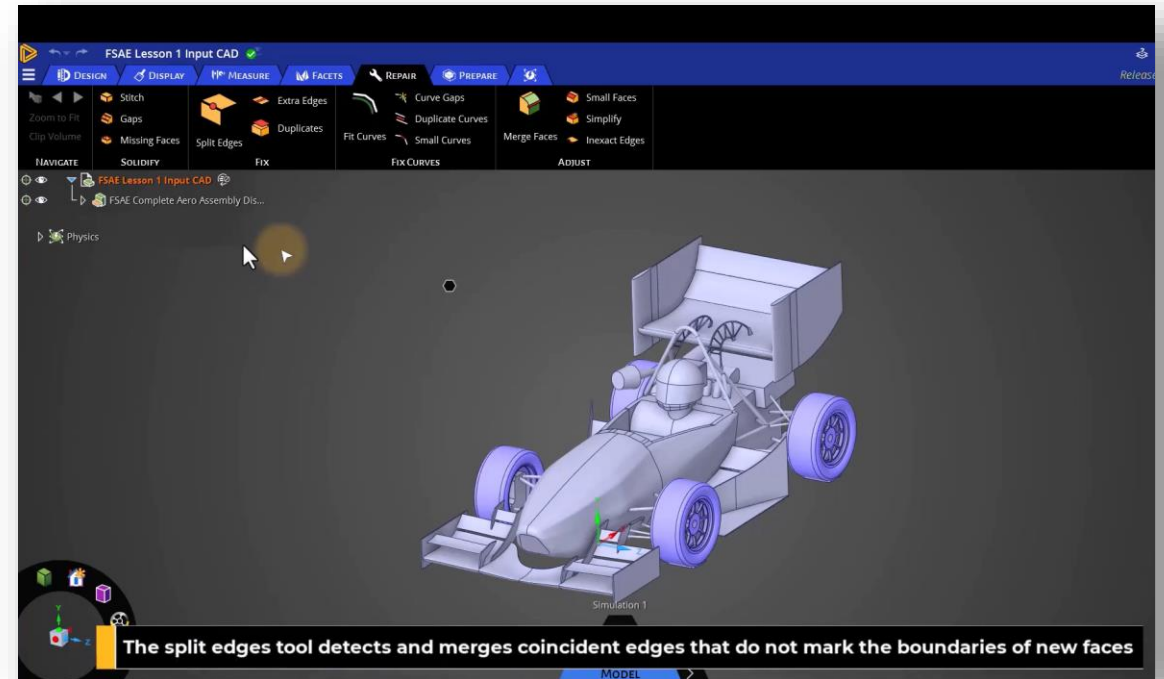
Unique Ways Ansys Can Help: CAD Cleanup

CAD simplifications

in preparation for simulation in history-based CAD can be very time consuming (one issue can cascade into others)

Ansys Discovery

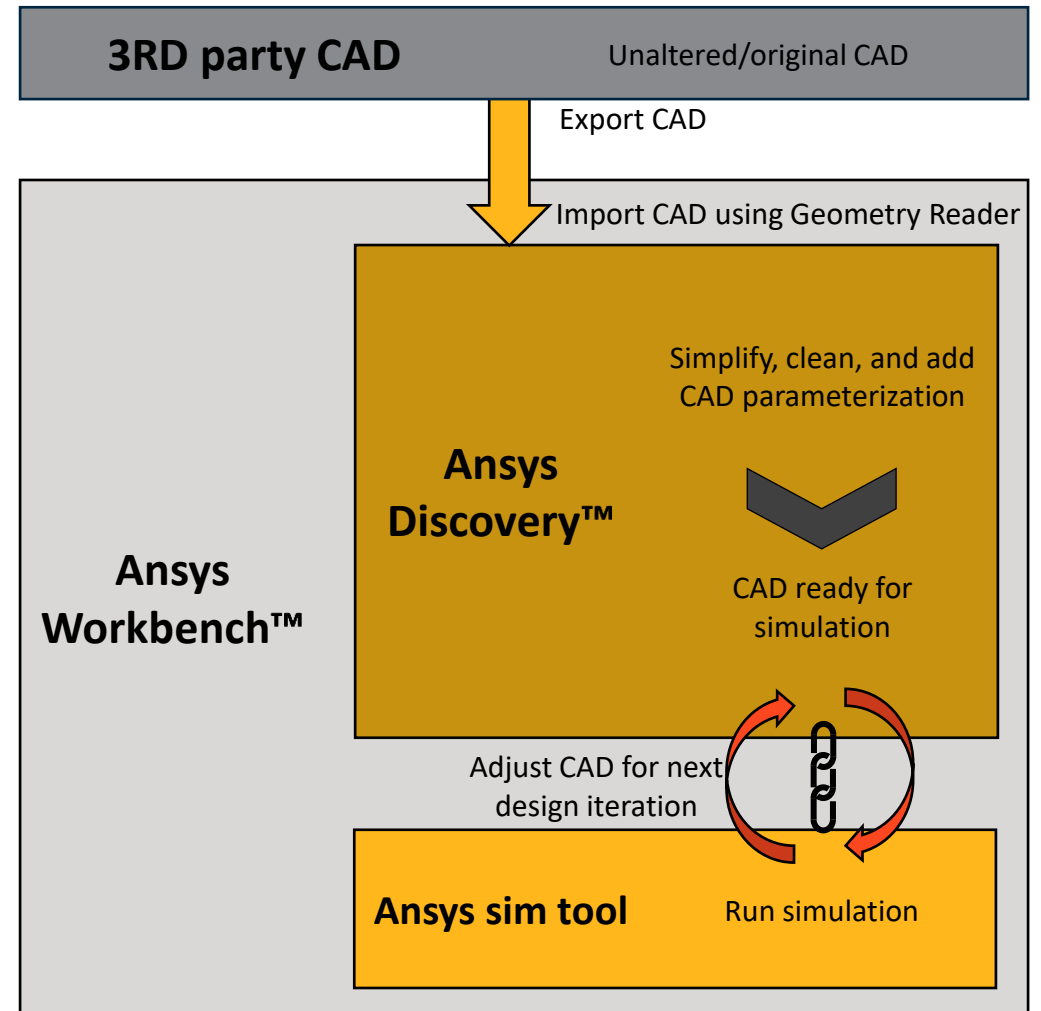
allows for *quick* face-based changes (not tied to CAD history) and surface repair



Unique Ways Ansys Can Help: Rapid Iterations

Rapidly iterate design exploration with our CAD to simulation workflow:

1. Import original CAD to Discovery (there are many CAD import options), then simplify, clean, and parameterize the CAD, build/run simulation in the simulation tool, tweak the cleaned/simplified CAD in Discovery, re-simulate, repeat.
 2. Or follow the same process but simplify the geometry before importing to Ansys.
 3. Or link CAD to Discovery so that changes made to the CAD are updated in both tools (available via Workbench CAD interface)
- When done, export the geometry for the final design back to your CAD package.
 - Mechanical can also plug directly into external CAD package (no CAD export/import).



Unique Ways Ansys Can Help: Battery Pack Designer

Cell technology and full module material selection, arrangement, and power/energy/thermal performance. Start this video at 13:30.

Cell to Module (by performance)

Combine multiple battery cells into a module. This contains several pre-defined configurations:

- Industrial (cylindrical): Rack mounted design, Sheet steel enclosure, Single BMS, Integral heating pads / thermal monitoring.
- Automotive (prismatic): Similar to designs used in vehicles in India, Heavy duty sheet steel enclosure, Single BMS / contactor, Forced air induction cooling, Open design for ease of maintenance, Simple and rugged.
- Shrink wrap (cylindrical): Lightweight design, Shrink wrapped, Single BMS, Integral thermal cutout / fuse, Designed for high volume / low cost.

NOTE: Modules must have integer numbers of cells. If values in a specified range are too close together, some generated modules may have identical numbers of cells. If this is the case, only one record for each instance will be created.

Module

Module name:

Battery cell:

Performance

Should last for at least: min Number of values:

with Current: A Number of values:

and Voltage: V Number of values:

Configuration

Pre-defined module

Pre-defined configuration:

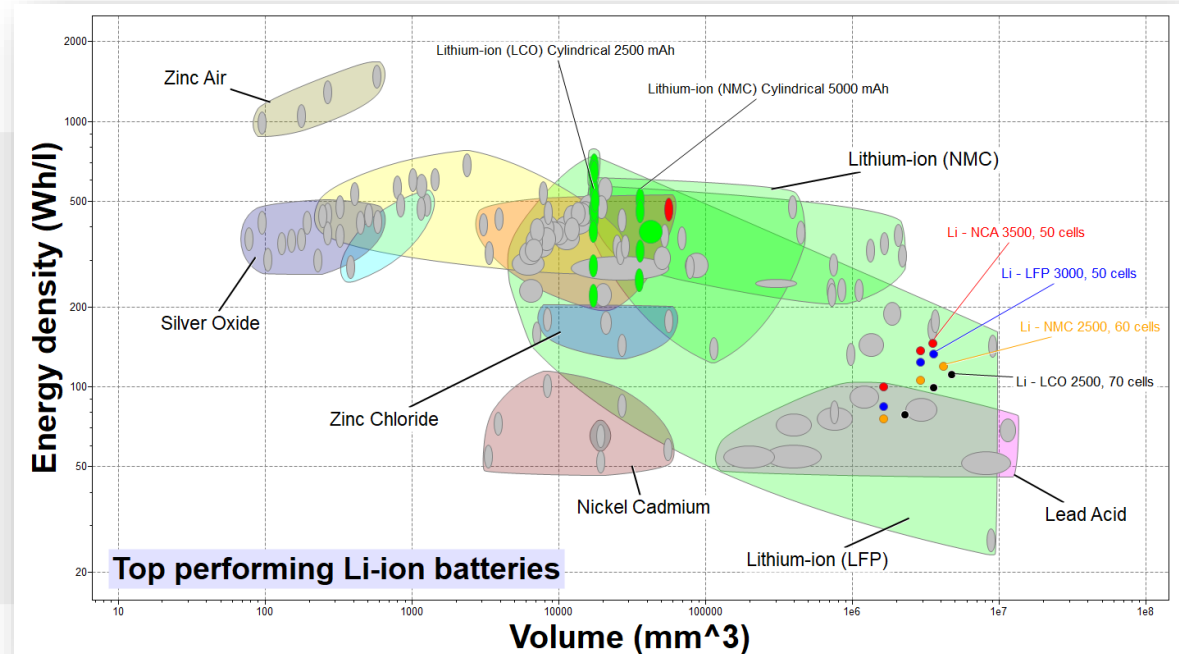
Or custom configuration

Packaging

Thermal management system (TMS)

Cooling system type:

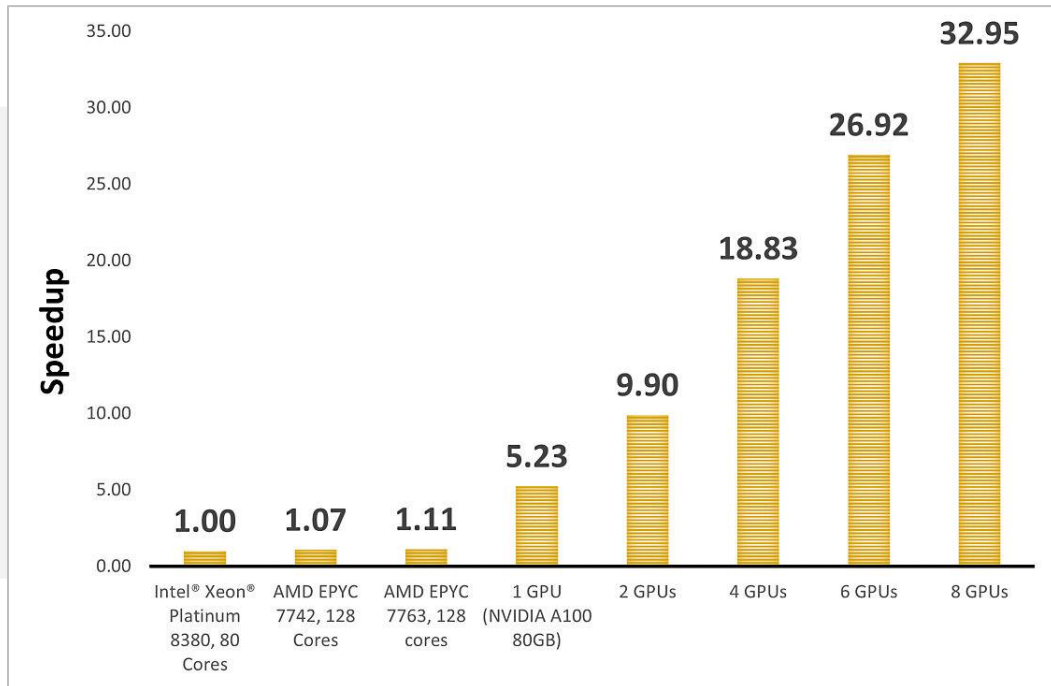
This model will generate 3 records



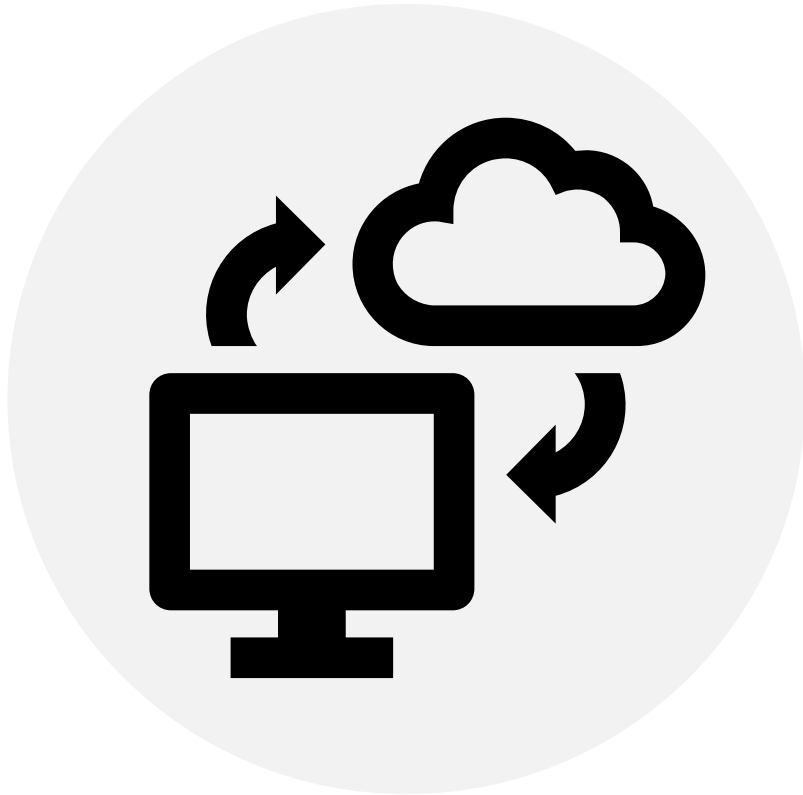
Unique Ways Ansys Can Help: GPU Computing for CFD

Run your CFD simulations **much** faster through GPUs:

1 GPU = 400 CPUs ([more info](#) and [GPU course](#)).



Unique Ways Ansys Can Help: Cloud Computing



Cloud Computing benefits:

- Run simulations much faster
- 3rd party companies have Ansys setup on them
- Easy interface to run sims
- Remote Graphics for 3D visualization and remote desktop
- Some provide free cores hours, storage, and data transfer

Cloud Computing vendors:



Unique Ways Ansys Can Help: Preparing You for the Workforce

The screenshot shows an Indeed search results page. The search criteria are: What: (engineer OR engineering) AND (ansys OR "ansys mechanical" OR Fluent OR CFX), Where: City, state, zip code, or "remote". The search results are sorted by relevance, showing 11,366 jobs. Two job listings are visible:

- Principal Motor Engineer** at Leonardo DRS (3.7★) in Fitchburg, MA. Salary: Estimated \$114K - \$144K a year. Full-time. Requirements: Bachelor of Science degree from a 4 year accredited college or university in Mechanical or Electrical Engineering, or other appropriate degree. Visited 3 minutes ago.
- Cryogenic Engineer / Engineering and Physical Sciences Resea...** at National Security Agency (4.1★) in Fort Meade, MD. Salary: \$86,389 - \$176,300 a year. Monday to Friday +1. Requirements: Relevant experience must be performing professional engineering or physical sciences work and/or conducting academic or independent research in a technical... Visited 1 day ago.

On the right side, a list of companies is shown with the number of job posts each has:

- CANONICAL (271)
- Recruiting from Scratch (261)
- Ansys (236)
- Pratt & Whitney (213)
- Northrop Grumman (129)
- Tesla (100)
- Blue Origin (94)
- Lockheed Martin (91)
- LAUNCH POTATO (91)
- Peter R. Thom and Associates Inc. (88)
- Apple (84)
- WSP (83)
- GE Aviation (71)
- Merkle, Inc. (71)
- Leidos (70)

There are more than **35,000*** job posts that mention "simulation" on Indeed.

*Data collected on 10/16/2023.

Careers Begin with Learning Ansys

Ansys tools are used throughout industry and are valuable to employers. **Ansys and simulation trained students command a premium.**



*Kayla Mennillo, Senior Engineer, Aerothermal Fluids, **Pratt & Whitney***



*Mariana Golden, Mechanical Designer at **Duxion Motors, Inc.***



*Alba Marcelin, Systems Engineer at **GE Aerospace***

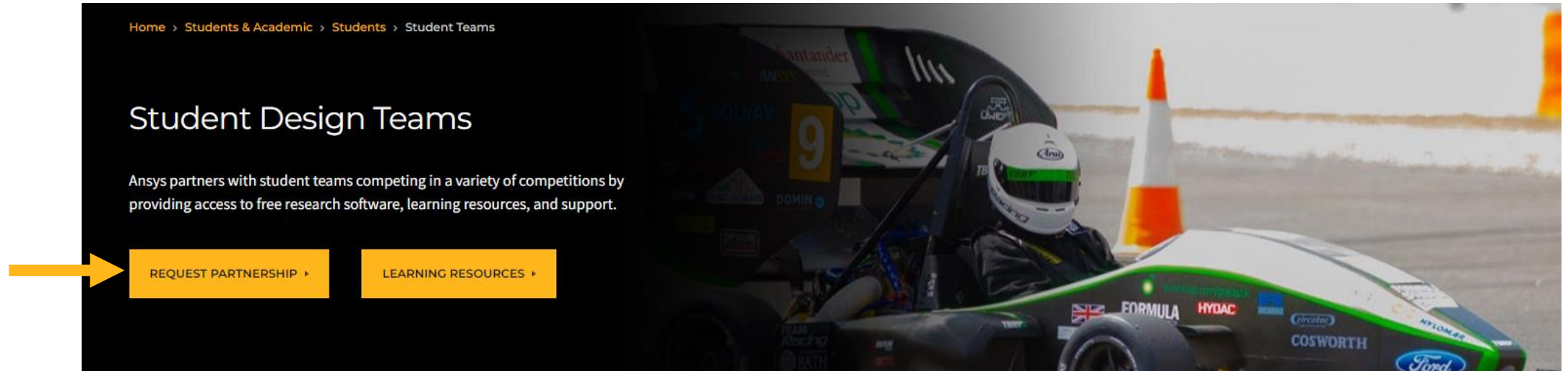


*Christina Peristeri, Application Engineer at **Ansys***

Examples of four women who gained an **Ansys simulation skillset** in their degree courses and on **student teams** and are now successful engineers in industry.

How to Get Access to Commercial Versions of Ansys

Go to ansys.com/teams, select “Request Partnership”



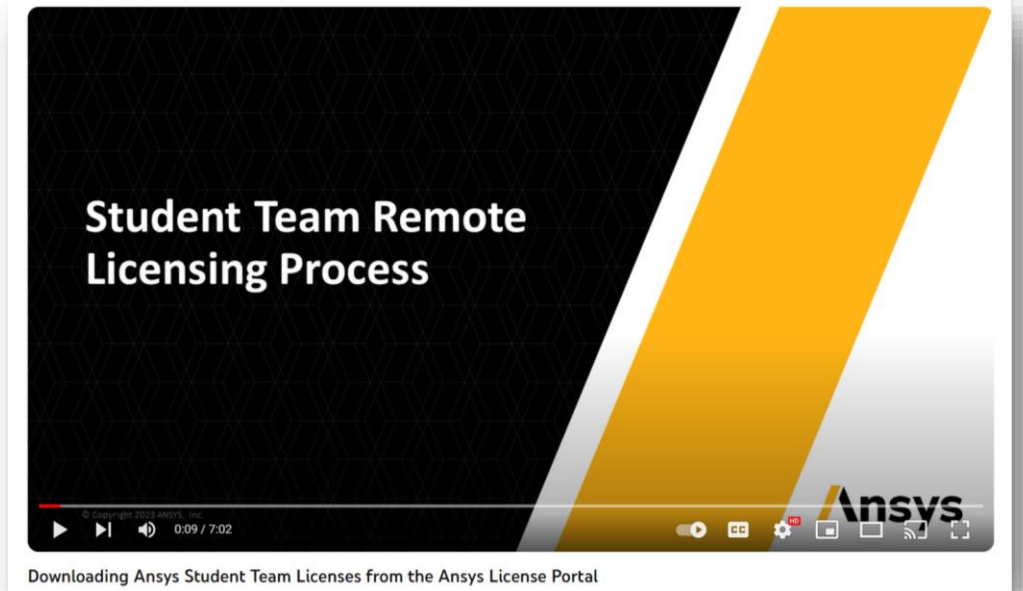
Complete the form - no contract needs to be signed to receive the software.

IMPORTANT: The contact information put into the form will be automatically assigned as the admin on the account, or what we call the Ansys Support Coordinator (ASC).

Tutorial for Setup of Licenses

- What happens after you fill out the form
- Licensing
- Gathering computer info
- Activating/downloading license files for each computer
- Installation first steps

YouTube



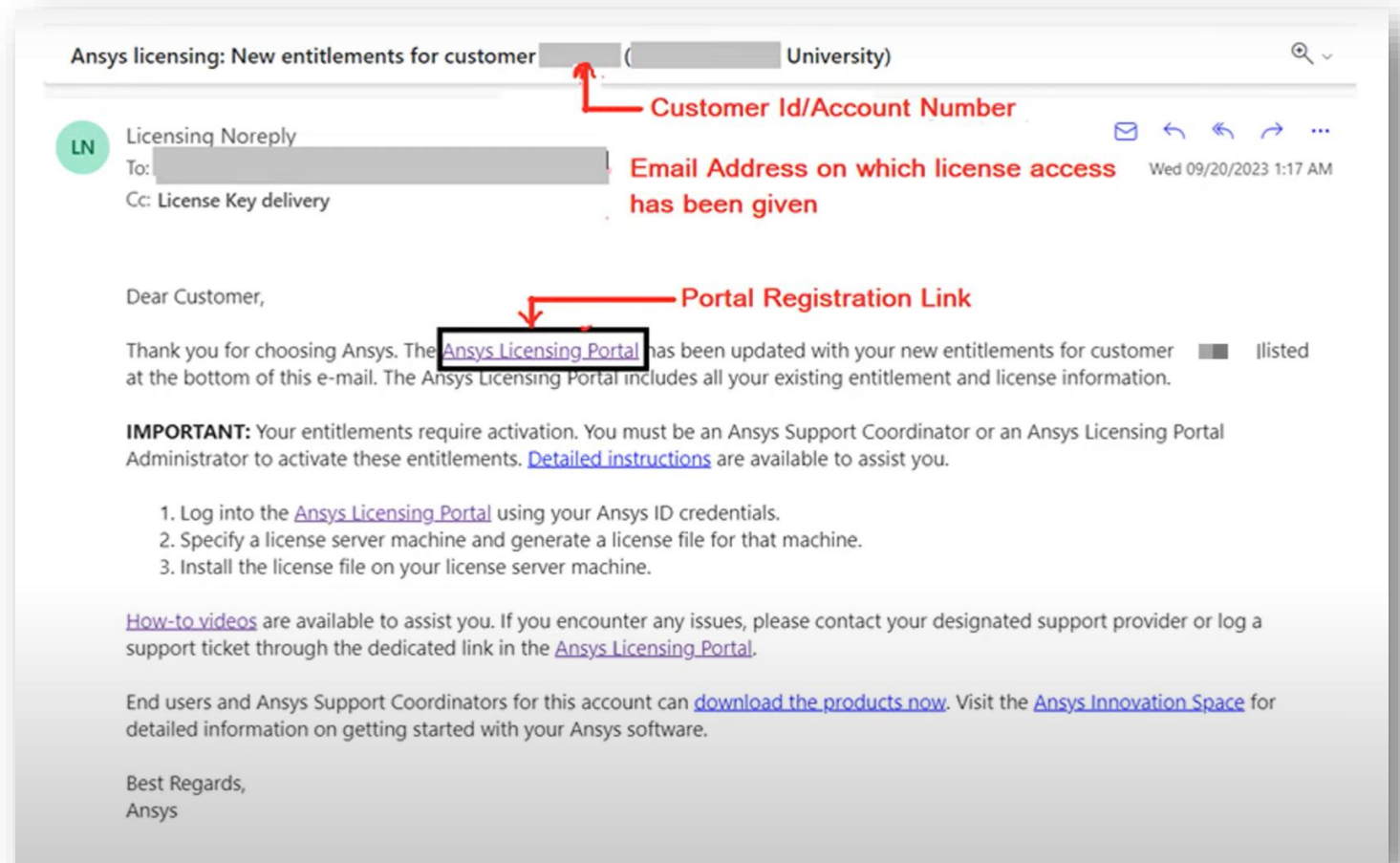
Note: You can also **find the host name, ID, and ID type** by opening the .info file mentioned in the tutorial in Notepad.

Following Completion of the Partnership Form

First, an email will be sent regarding the form completion.

Next, a “new entitlements” email will be sent to the ASC explaining...

- Licensing credentials have been added to your account
- ASC will need to download the license file
- How to setup the software



Summary of the Setup Guidance from the “New Entitlement” Email

Ansys Licensing Portal

Ansys Licensing Portal: enables Ansys Support Coordinators (ASCs, essentially the admins for the account) to download the license file, change server information, activate entitlements, and add/remove other individuals (see [detailed instructions](#) and [how-to videos](#)).

Ansys Customer Portal

The Customer Portal: enables ASCs to access the installation files and guides under [downloads](#). Check the “?” icons for more information on what is contained in each installation file. See [installing Ansys software](#) for documentation, videos, troubleshooting and more. **IMPORTANT:** Do not include "student" in the title when registering.

Ansys Innovation Space

Ansys Innovation Space: has detailed information on how to download and install the software, along with links to our community forum (tech support monitors and responds to these questions), training, class materials, etc.

Ansys Support Coordinator (ASC)

/ Ansys Support Coordinator

ASCs are an administrator on the Ansys Account(s) that have full access to the following:

- [Customer portal](#) for product installation files
- [Ansys Licensing Portal](#) for licensing management (assignment of products to servers, license downloads, etc.) and [adding other ASCs](#)

Contact your or academic@ansys.com if you don't know who the ASCs are or if an ASC should be removed.

NOTE: Group emails cannot generally be used as an ASC.

Standard Licensing Provided to Student Teams

Full commercial versions are provided (no model size limits) through our academic “research” bundles. Standard products include:

25 tasks (a.k.a. simultaneous users or seats) of the following bundles:

- **CFD:** [Fluent](#), [Forte](#), [Chemkin](#), [Discovery](#) and [SpaceClaim](#)
- **Mechanical:** [Mechanical](#), [LS-DYNA](#), [Additive Suite](#) and [Additive Print](#), [Discovery](#) and [SpaceClaim](#)
- More products are included in the bundles, see the above bundles in the [“Ansys academic product bundle reference table”](#)

To adjust what is provided please **contact your academic representative or academic@ansys.com**.

16 **HPC** licenses

Enables sims to run on more cores) locked into each task with 32 additional flexible HPC licenses (can be used in addition to any simulation)

10 tasks of **Thermal Desktop**

An early design stage tool for larger scale multi-system (ex: satellites)

5 tasks of the following bundles

- [Granta Research Selector](#) (materials database)
- **HF (high freq. electromagnetics):** [Icepak](#), [HFSS](#), [Q3D Extractor](#), [Slwave](#)
- **EM (electric motors):** [Icepak](#), [Motor-CAD](#), [Maxwell](#), [Q3D Extractor](#)
- More products are included in the bundles, see the above bundles in the [“Ansys academic product bundle reference table”](#)

Ways to Approach the Licensing



1. **A different license file for each computer** is generated, each team computer serves the licensing to itself.
2. **University IT manages the licensing.** IT may already have licensing you can easily access. If not or if they are missing products, you can obtain this student team license and IT will need to identify a computer that will serve out the licensing to other machines.
3. **The student team serves the licensing from one computer to other computers.** Instead of IT managing the licensing from a single machine, you would manage this, VPN accessibility, and (if applicable) a virtual desktop.

The next slides describe how the licensing works.

Using a Different License File for Each Computer

ASCs create a new license file assigned to each computer, each containing the products needed on that computer

Repeat till there are no bundles left to assign to computers

Team PC 1

License features requested /released on the same computer



License server & user

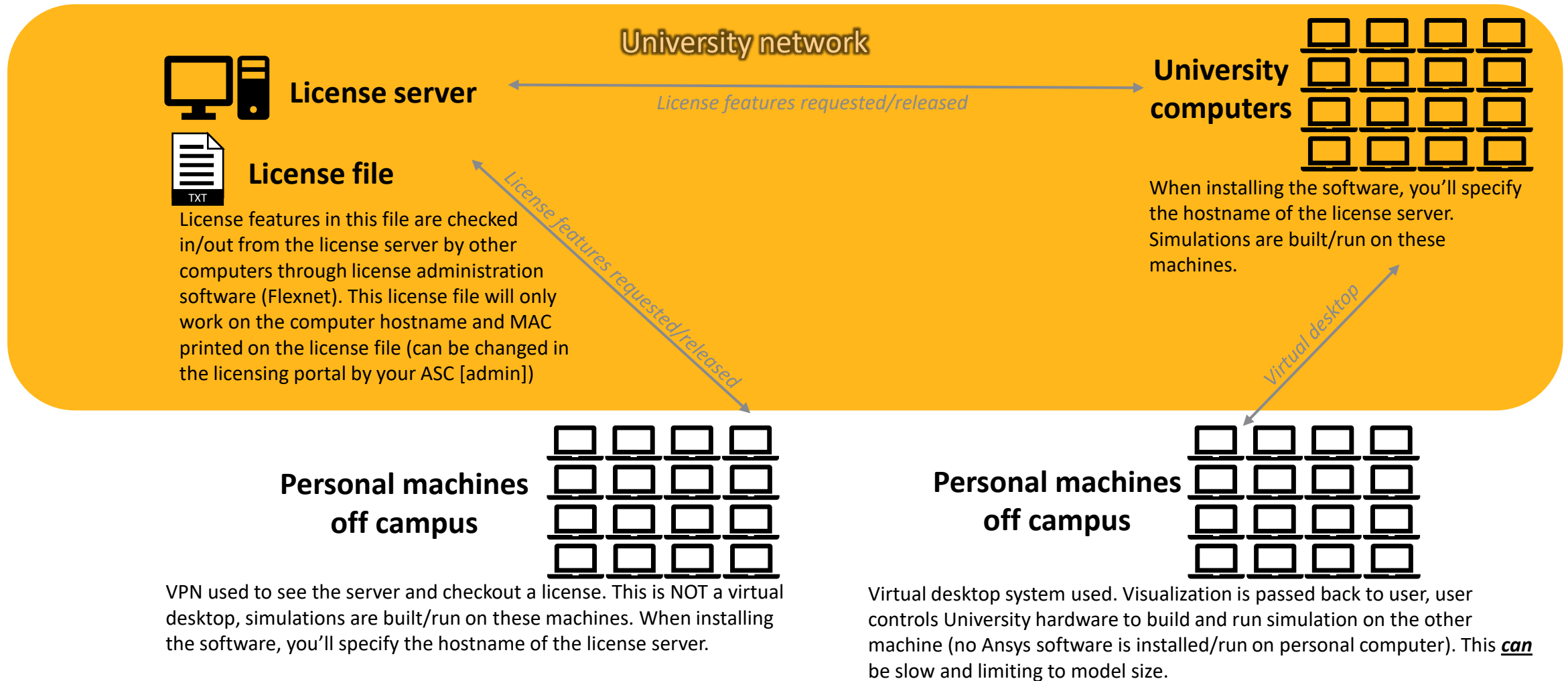
License file assigned to PC 1

License features in this file are checked in/out from the license server which happens to be the same computer the simulations are run on. This is done through license administration software (Flexnet). This license file will only work on the computer hostname and host ID printed on the license file (can be changed in the licensing portal by your ASC [admin])

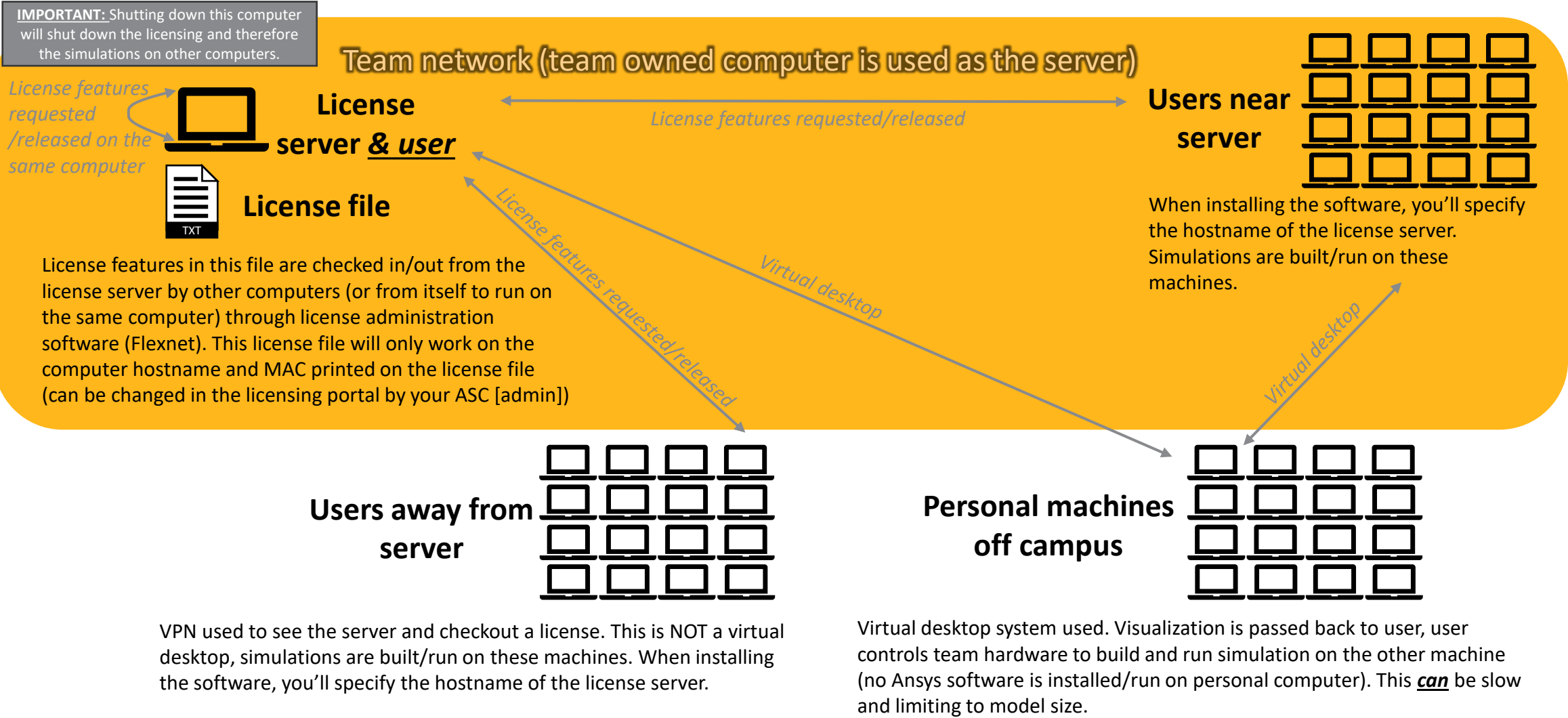
Team PC 2, 3, etc.

Same setup as above, but with a new license file assigned to PC 2, 3, and so on.

Using a Floating License Managed by University IT



Using a Floating License Managed by your Team



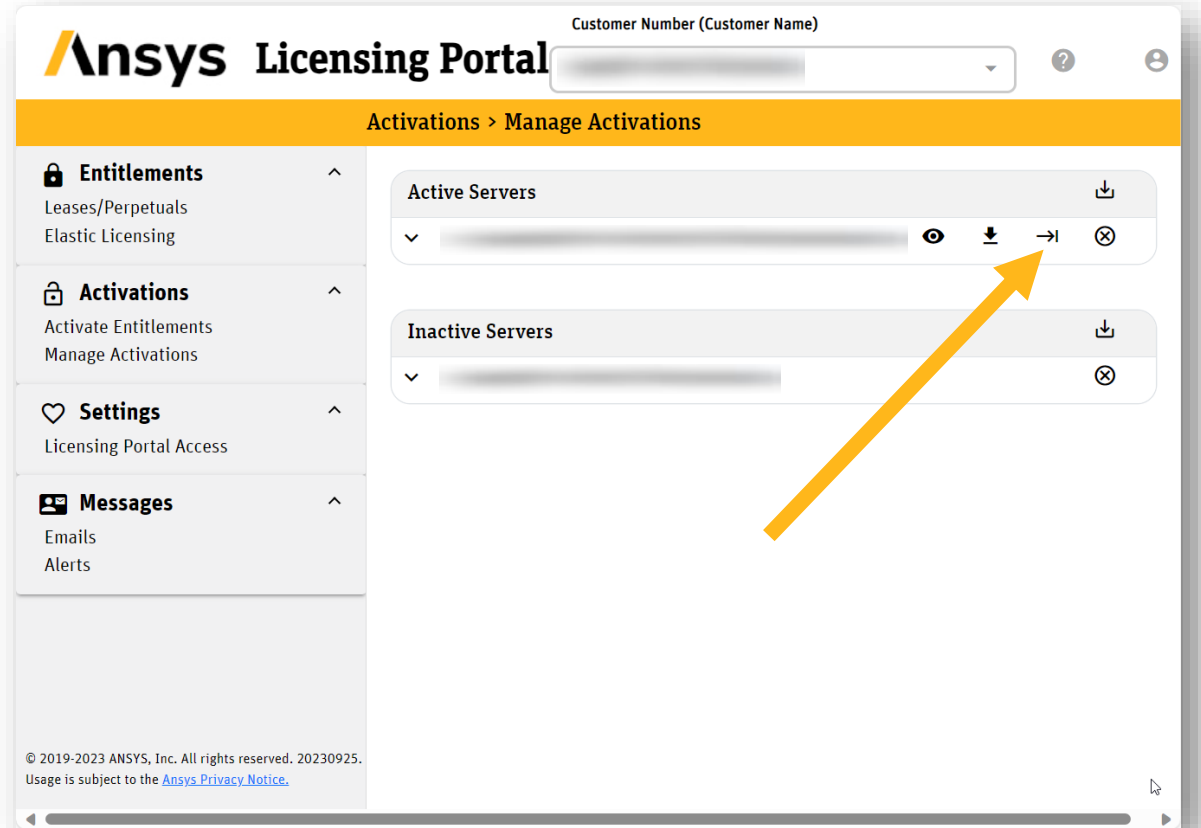
Examples of How to Assign your Licensing Entitlements/Bundles

Need	Action
User wants a license file for their computer. They need to build/run/analyze 5 CFD sims at one time and would like to run on more than 16 cores per session (20 in this case).	Add 5 of the <i>Ansys Academic Research CFD (1 task)</i> and 4 of the <i>Ansys Academic Research HPC</i> to their machine.
User wants a license file for their computer. User needs no more than one session of Granta, Discovery, and Mechanical open at any time.	Add 1 of the <i>Ansys Academic Research Mechanical (1 task)</i> and 1 of the <i>Ansys Granta Research Selector</i> to their machine.
Single server hosts all licensing to other machines.	Select all bundles and add them to that machine.

- See video tutorial on slide 21 that shows how to assign products to certain computers and download the license files.
- Keep in mind what products are in each bundle (refer to slide 25).
- See the [product reference table](#) to understand what is in each bundle.

How to Reassign a License to Another Computer

- In the [Licensing Portal](#) under manage activations, click on the **Rehost** →| icon.
- Enter a new, or existing **Host Name**, **Host ID Type** and **Host ID**. Alternatively, you can populate these fields by clicking the **Upload** button and using the file created from the **Get System Host ID** option available on the **ANSYS License Management Center** or a previous license key for that **Host ID**.
- Click **Save**. The list of entitlements and the counts to be deactivated from the source server and activated on the destination server is displayed.
- 4. After reviewing the changes, click **Ok** to finalize the rehosting procedure. The entitlements are deactivated on the source server (inactive status) and activated on the target server. The license file for the target server is downloaded to the download folder specified in your browser preferences.



Firewall Considerations



When using the software **over a firewall** (this applies to on campus use and at home use with VPN, but not virtual desktop environments), **be sure that all three necessary port numbers are open** to ensure successful communications:

- ✓ 2325 for the Licensing Interconnect
- ✓ 1055 for lmgrd
- ✓ The port you specify (in the license file) for ansyslmd. For example: VENDOR ansyslmd PORT=1056

For more information see:

- [Installing Ansys License Manager on Windows – YouTube](#)
- [Ansys License Manager: Configuring Firewall Exceptions on Windows - YouTube](#)

Hardware Considerations (RAM, CPU, GPU, etc.)

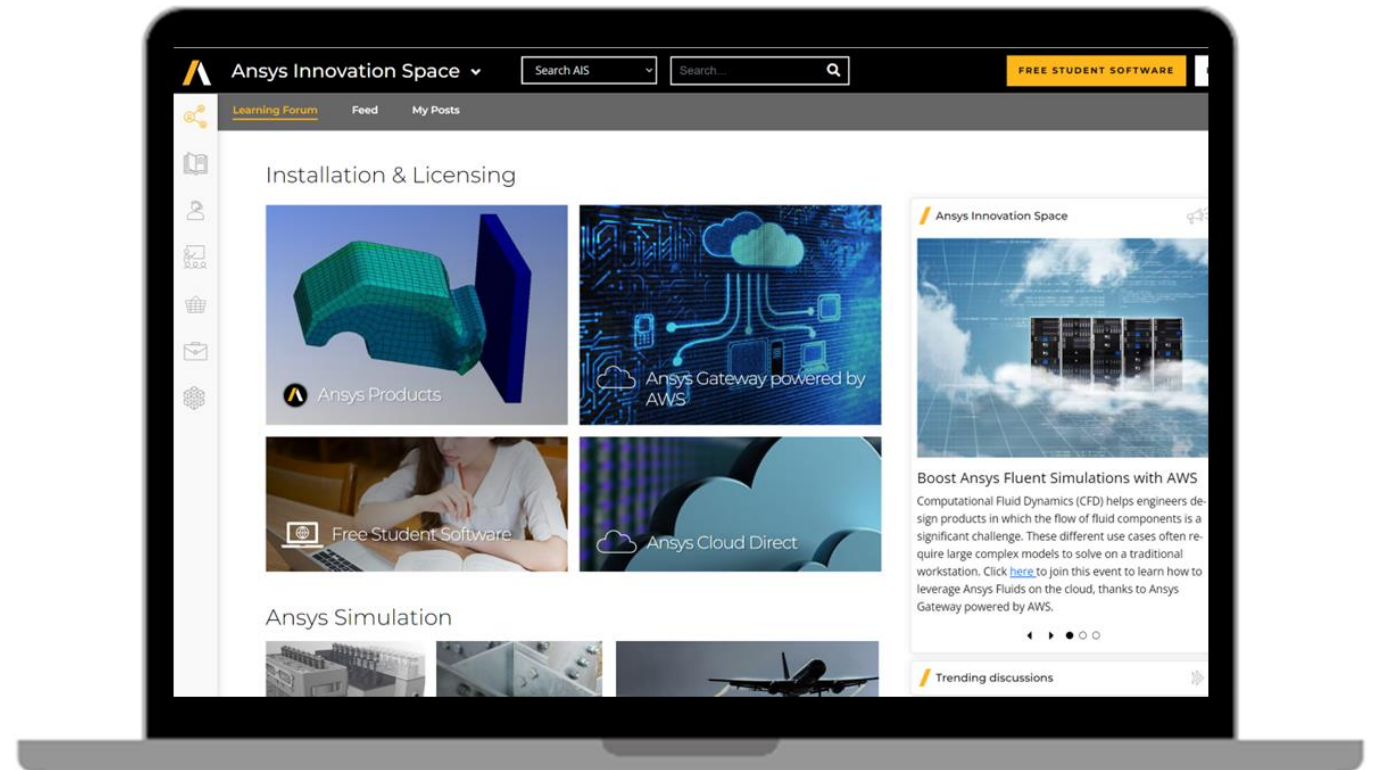
The ASC can go login to the customer portal at support.ansys.com, downloads, installation and licensing help and tutorials, installation, and look for “Ansys Product Hardware Guidelines.”



Visit the Ansys Learning Forum for Tech Support

If the documentation that comes with the software or Ansys Learning Hub are not helping...

- Support provided through the **Ansys Learning Forum**:
 - Search all worldwide support requests and answers ([overview video](#))
 - Ask your own question, responded to by support team
 - Various topics (see examples images)



[Ansys.com/forum](https://www.ansys.com/forum)

Download High-Definition Logos

Flash your partnership with Ansys by downloading the Ansys logo!

Access our logo by looking for the logo icon at the bottom of the student team page:

DOWNLOAD LOGO OPTIONS ▶

[Ansys.com/teams](https://www.ansys.com/teams) (scroll to bottom of page)



The Ansys logo consists of a yellow slanted bar followed by the word "Ansys" in a bold, black, sans-serif font.

