Ansys Formula Student Partnership

Ansys partners with hundreds of Formula Student teams around the globe by providing free access to full version industry-leading simulation software, learning resources and technical support. Teams use Ansys simulation software to design their aero packages, lightweight structural components, design electronics, and much more.

▶ To obtain a partnership, visit **ansys.com/teams**

/ LEARNING RESOURCES & SUPPORT

Ansys Innovation Courses provides public access courses developed for Formula teams such as:

- CFD on an FSAE car (full setup, run, and post-processing)
- Chassis torsional stiffness
- Chassis stress under braking
- Chassis stress under frontal impact and aero loading
- Rocker and bell crank
- Upright analysis
- Brake pedal and mounts
- Brake disk
- Anti-intrusion plate
- Composite Monocoque Chassis

This is in addition to hundreds of publicly accessible, free mini courses on physics and engineering topics to help you get started using Ansys products. Courses include lecture videos, handouts, simulation examples and more to support learning.

Additional student team tutorial videos are available to help student team members learn and discover how Ansys simulation can be used for the design and optimization of their racecar. Videos provide step-by-step instructions on topics such as:

- Aerodynamic analysis
- Structural analysis
- Topology optimization
- Chassis analysis
- Thermal modeling

Ansys Learning Hub provides access to in-depth training and support materials with industry-specific application training.
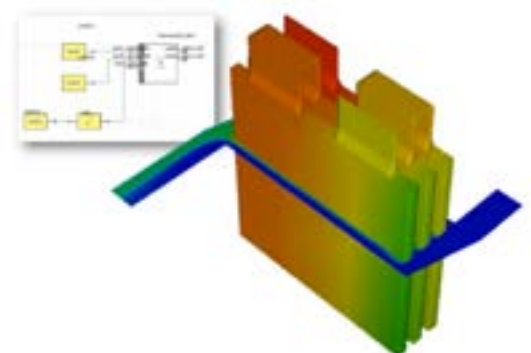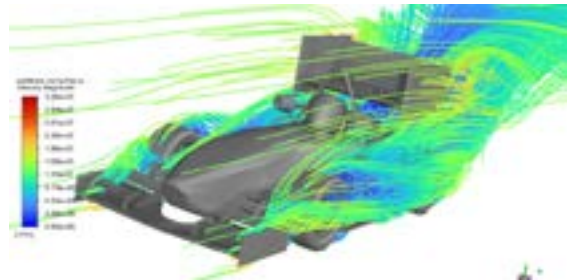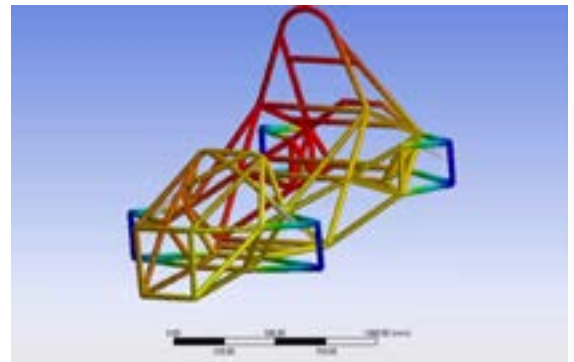
Structures

Often teams using CAD-embedded FEA software think they save time due to the CAD/FEA connection but don't realize:

· Ansys Mechanical can directly connect to your CAD for rapid design iterations.

· The lack of training on FEA best practices leads to inaccurate results, broken parts, lower design points, and the missed chance to lightweight. Ansys, however, has 9+ FSAE tutorials and general FEA training which is publicly accessible.

· Twice the time is spent on FEA by validating results from one software against Ansys.

· Meshing in Ansys is much faster, has less errors, and can automatically refine in areas needed based on the solution leading to better accuracy

· Ansys has mesh models that require less cells through the thickness, meaning it can mesh and run faster.

· Ansys automation for parametric design sweeps is much more robust.

· Teams that use Ansys Mechanical report learning it very quickly.
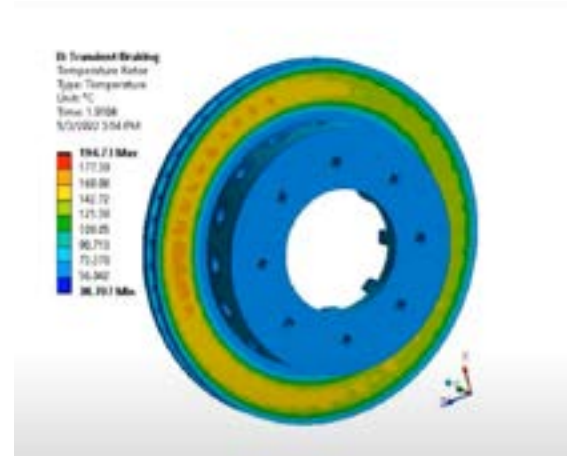
Aerodynamics

Teams often have problems accessing tutorials that explain the full simulation setup process and cannot easily access training that explains best practices. Ansys has publicly accessible content that covers:

· FSAE geometry preparation for CFD

· Meshing best practices and boundary conditions

· Reports, plots, and scenes specific to the needs of FSAE teams

· **Coming soon:** Automated design space exploration, car cornering, adjoint for optimization, radiators, and more
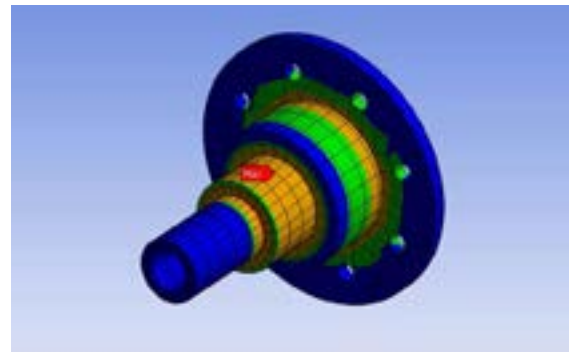
Teams often don't realize that Fluent:

· Is now a cohesive interface with a modernized tabular layout.

· Runs faster in steady state sims than the leading competing CFD tool.

· Utilizes Python instead of a custom version of Java for scripting.
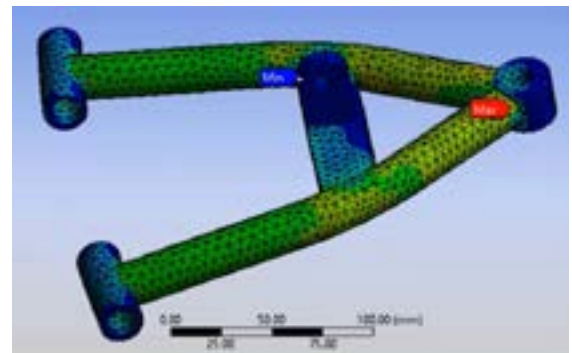
Materials

Ansys Granta Selector can be a valuable tool to help your team:

· Increase design scores by giving you the ability to find optimal materials for any component.

· Reduce material costs while increasing performance.

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

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

· Estimate properties of and get recommendations for composites (which matrix, fiber, fiber volume, ply structure, etc.).

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

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304

ansys.com