

Instructor Guide- Forced Convection in External Flows



This instructor guide can be used to implement the module “Heat Transfer in Fluids- External Flows” in the classroom.

Module Aim

The overall aim of this module is to support educators introducing topics related to heat transfer in fluids fundamentals with simulation. This module serves as a foundation, providing slides that can easily be expanded to go into detail for various topics depending on specific course curriculum. Details of how simulation can aid visualization and exploration of concepts through images and homework.

It is important to note that this module is meant to supplement materials taught in an introductory course and not to replace the course in its entirety.

Suitable Courses

It is well suited for a heat transfer course; it may also be applicable to a heat exchanger or external flow module.

Contents

- Lesson 1- Intro to Forced Convection
- Lesson 2- Boundary Layers
- Lesson 3- Flat Plate
- Lesson 4- Cross Flow Geometry
- Simulation examples (including simulation files)
 - » Laminar Flow over a Flat Plate
 - » Forced Convective Heat Transfer
- Homework
 - » Turbulent Flow over a Flat Plate
 - » Round Impinging Jet
- Quiz Questions

Module Duration

Approximately 3 hours.

Learning Outcomes

Students are able to:

1. Understand forced convection and its types.
2. Derive dimensionless numbers and correlations.
3. Understand velocity and thermal boundary layers.
4. Solve simple heat transfer problems in fluids using Ansys Fluent

References and additional resources

This content was adapted from the [Ansys Innovation Course Forced Convection in External Flows](#).



Additional Ansys Education Resources of interest:

- [Lecture Unit: Introduction to Heat Transfer with Ansys Discovery](#)
- [Case Study: Thermal Analysis of Heat Sinks with Ansys Discovery](#)
- [Teaching Package: Governing Equations of Fluids](#)
- [How Heat Exchangers Work - ANSYS Innovation Courses](#)