



# Case Study

## Windcatcher- Sustainable Cooling using Ansys Fluent®

Developed and curated by the Ansys Academic Development Team

Gautham Varma Raja Kochanattu

[education@ansys.com](mailto:education@ansys.com)

**Ansys Software Used**

This resource uses Ansys Fluent®, the fluid simulation software, and Ansys Discovery™, the 3D Product Simulation Software

**Summary**

Ansys Fluent is a Computational Fluid Dynamics (CFD) software that is used to solve various problems related to fluid flow, heat and mass transfer, chemical reactions, and more. It uses advanced physical models like turbulent modeling, multiphase modeling, battery modeling, combustion, and fluid-structure interactions to solve the given problem to high level of accuracy.

In this case study, Ansys Fluent software is used to understand the functioning of a windcatcher architectural design and how it is used to regulate the temperature inside a room. In this case study, we will also discuss the computational methods used as well as setting up the

**Table of Contents**

1. Introduction..... 3
2. Physics Behind the calculations ..... 3
2.1 Mass Flow in a Windcatcher..... 3
2.2 Cooling Through Convection ..... 4
2.3 Evaporative Cooling..... 4
2.4 Combined Cooling Effect ..... 4
2.5 Buoyancy-Driven Flow ..... 5
3. Geometry and Meshing..... 5
4. Results ..... 6
5. Conclusions..... 8
6. References ..... 8

## 1. Introduction

The windcatcher, or badgir in Persian, is a centuries-old architectural innovation designed to harness wind power for natural ventilation and cooling. Originating in the arid regions of Persia (modern-day Iran), it ingeniously captures wind at various angles and channels it into buildings, providing relief from intense heat without mechanical systems. Often paired with water features like qantas, windcatchers utilized evaporative cooling to create comfortable indoor environments, blending functionality with environmental harmony. Over time, their use spread across the Middle East, North Africa, and parts of South Asia, influencing regional architecture with their energy-efficient design.

In modern times, the windcatcher has inspired advancements in sustainable technology, evolving from a traditional cooling method to a symbol of eco-friendly innovation. Contemporary designs incorporate the principles of windcatchers into renewable energy systems, urban sustainability projects, and modern architecture. Whether as a passive cooling solution or as part of cutting-edge wind harnessing technologies, the windcatcher continues to demonstrate the enduring value of ancient ingenuity in addressing modern environmental challenges.

In this case study, we analyze the airflow and cooling performance of a windcatcher system using Computational Fluid Dynamics (CFD). The study focuses on understanding the velocity distribution, temperature variation, and overall cooling effect inside a basic building structure with a windcatcher.

## 2. Physics Behind the calculations

The mass and cooling effects in a windcatcher can be explained using principles of fluid dynamics and thermodynamics, focusing on airflow (mass flow) and heat transfer (cooling). Below, we provide a detailed explanation with relevant equations:

### 2.1 Mass Flow in a Windcatcher

The airflow (mass flow rate) through the windcatcher is driven by pressure differences caused by wind or buoyancy. The mass flow rate is given by:

$$\dot{m} = \rho \cdot A \cdot v$$

Where:

$\dot{m}$ : Mass flow rate kg/s

$\rho$ : Air density kg/m<sup>3</sup>, which depends on temperature and humidity:  $\rho = P / \{R \cdot T\}$

P: Atmospheric pressure Pa

R: Specific gas constant for air (287 J/kgK)

T: Air temperature (K)

A: Cross-sectional area of the windcatcher opening (m<sup>2</sup>)

v: Air velocity at the opening (m/s)

## 2.2 Cooling Through Convection

The cooling effect involves convective heat transfer as the air passes over surfaces or water features. The rate of heat transfer by convection is given by:

$$Q_{conv} = h \cdot A \cdot \Delta T$$

Where:

$Q_{conv}$  : Convective heat transfer rate (W)

$h$ : Convective heat transfer coefficient (W/m<sup>2</sup>K), determined experimentally or using correlations like:

$$h = \frac{k}{L} \cdot Nu$$

$k$ : Thermal conductivity of air (W/mK)

$L$ : Characteristic length (m)

$Nu$ : Nusselt number (dimensionless), which depends on the flow regime (laminar or turbulent).

$A$ : Surface area in contact with airflow (m<sup>2</sup>)

$\Delta T = T_{surface} - T_{air}$  : Temperature difference between the surface and the incoming air (K).

## 2.3 Evaporative Cooling

If the windcatcher includes a water feature, evaporative cooling enhances the cooling effect. The rate of evaporative cooling is:

$$Q_{evap} = \dot{m}_{water} \cdot h_{fg}$$

Where:

$Q_{evap}$  : Heat transfer due to evaporation (W)

$\dot{m}_{water}$  : Mass flow rate of evaporated water (kg/s)

$h_{fg}$  : Latent heat of vaporization (J/kg), typically around  $2.26 \times 10^6$  J/kg for water

The mass flow rate of water can be related to the rate of evaporation as:

$$\dot{m}_{water} = \rho_{air} \cdot A \cdot v \cdot (\omega_{sat} - \omega_{air})$$

Where:

$\omega_{sat}$  : Saturation humidity ratio (kg vapor/kg air) at the water temperature.

$\omega_{air}$  : Humidity ratio of the incoming air.

## 2.4 Combined Cooling Effect

The total cooling provided by the windcatcher is the sum of convective and evaporative cooling:

$$Q_{total} = Q_{conv} + Q_{evap}$$

Using this combined equation, the cooling effect can be optimized by adjusting parameters like wind speed, surface area, water availability, and air humidity.

## 2.5 Buoyancy-Driven Flow

In the absence of wind, buoyancy drives the airflow. The velocity of air due to buoyancy can be approximated as:

$$v_{buoyancy} = \sqrt{2gH \frac{\Delta\rho}{\rho}}$$

Where:

g: Gravitational acceleration (9.81 m/s<sup>2</sup>)

H: Height of the windcatcher (m)

$\Delta\rho$  : Density difference between indoor and outdoor air (kg/m<sup>3</sup>)

This buoyant airflow contributes to natural ventilation and cooling.

In this Simulation example, we will only consider the simplest of cases, where air is passed through a series of pipes that are kept at the entrance. These pipes are maintained at a lower temperature. This process will help us understand how a series of cold pipes could help us reduce the temperature inside a room through simple convective cooling.

## 3. Geometry and Meshing

The windcatcher features a compact geometry measuring 1m × 1m × 1m. It is equipped with 45° angled louvers and cold pipes, each with a diameter of 0.02m, positioned at the base of the windcatcher. These pipes are arranged in four layers, with a vertical spacing of 0.02m and a horizontal spacing of 0.05m between them. This windcatcher is designed to cool a space measuring 3m × 5m × 5m, which includes a window aligned with the windcatcher openings.

The setup is adapted from the study by (Calautit, 2016) but in this case, no macro-climatic environment is modeled around the entrance. Instead, the focus is on internal simulations, considering only the inlet temperature and velocity. The CAD model was developed using Ansys Discovery software and meshed using Fluent Meshing software. The simulation incorporates inflation regions around the pipes to accurately capture the heat transfer between the pipes and the airflow. The figure below illustrates the resulting mesh.

The windcatcher simulation in this case study uses a detailed mesh consisting of approximately 4.7 million cells. This level of refinement is necessary to capture fine details of the flow dynamics and heat transfer processes accurately, especially around complex geometries like louvers and cooling pipes. However, the Ansys Fluent student version has a limitation of 1.5 million cells, which may restrict running this specific simulation directly.

To visualize the results, students can study the simulation outcomes provided in this case study or modify the mesh to reduce the cell count. While a coarser mesh (e.g., within the 1.5-million-cell limit) might still capture general trends and provide insight, it could compromise the accuracy and resolution of the results, particularly in areas with high gradients like near the cooling pipes.

For those needing to run the full simulation with the existing mesh, a researcher or professional version of the Ansys Fluent software is required. This version supports significantly larger cell counts, enabling more precise modeling of complex phenomena.

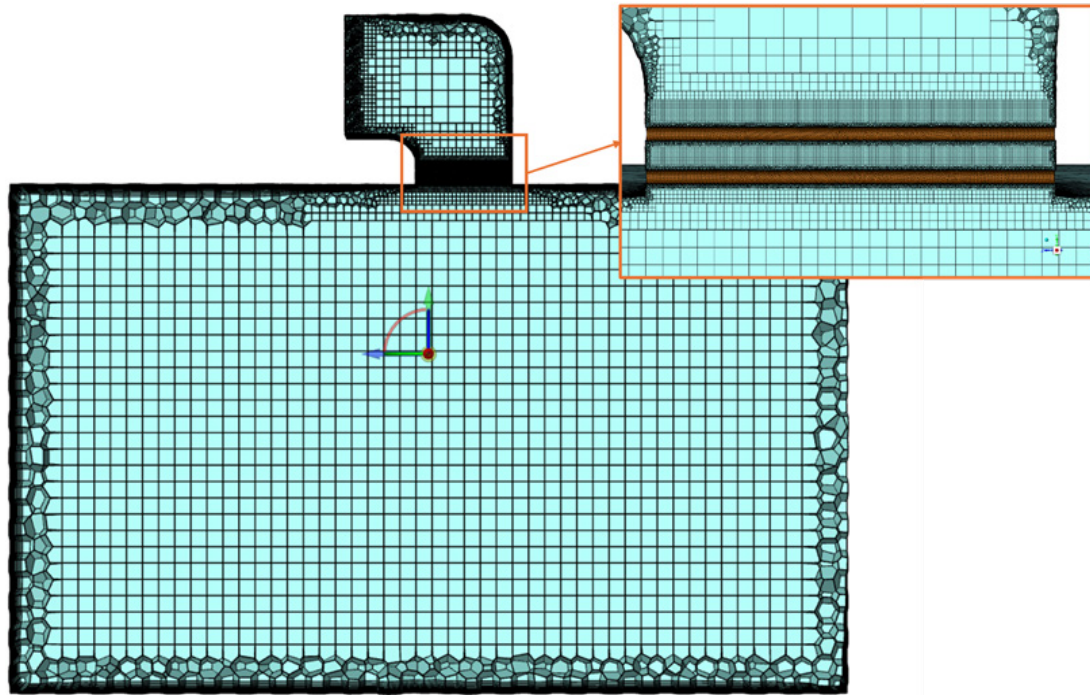


Figure 1: Mesh for the CAD profile where the inflations can be seen in detail around the piping.

## 4. Results

Figure 2 temperature profile provides a 2D contour view of the air temperature as it flows from the windcatcher into the interior space. In this visualization, the red region near the louvers and cooling pipes represents higher temperatures, indicating the incoming air that has not yet undergone significant cooling. As the air descends and spreads into the room, the temperature gradually decreases, transitioning to green and blue zones. This gradient illustrates the cooling effect of the cold pipes, which transfer heat from the air, lowering its temperature. The smooth progression toward cooler colors in the interior space demonstrates the effectiveness of the cooling mechanism as the air disperses further into the room. The average temperature just below the pipes is obtained as 35.04 °C suggesting a good passive cooling of the air entering the area through cool pipes.

The second profile (Figure 3) presents a 3D volumetric rendering of the temperature distribution within the room. This visualization provides a spatial perspective of how the cooled air spreads and interacts with the environment. Warmer regions near the windcatcher, highlighted in red, transition into cooler tones of green and blue as the cooled air moves further into the space. This rendering captures the temperature stratification and flow dynamics, emphasizing how the windcatcher influences the air's thermal behavior throughout the room. The image showcases how the cooling effect extends from the windcatcher and creates a gradual temperature reduction, effectively cooling the interior space. Together, the two profiles offer a comprehensive understanding of the windcatcher's performance and its impact on temperature regulation.

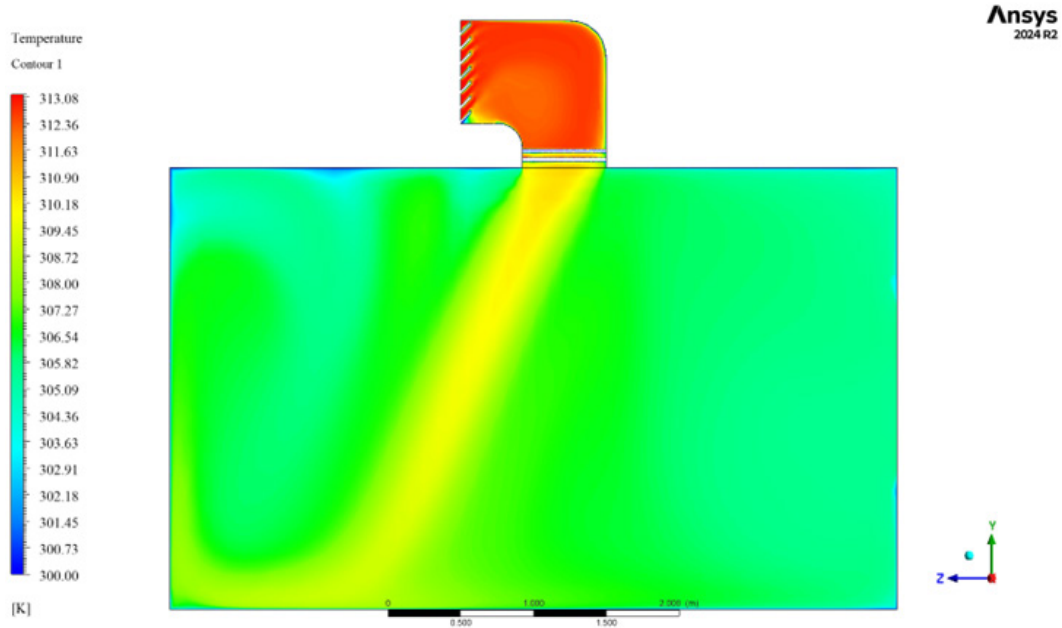


Figure 2: The temperature profile for the YZ plane is shown for the building.

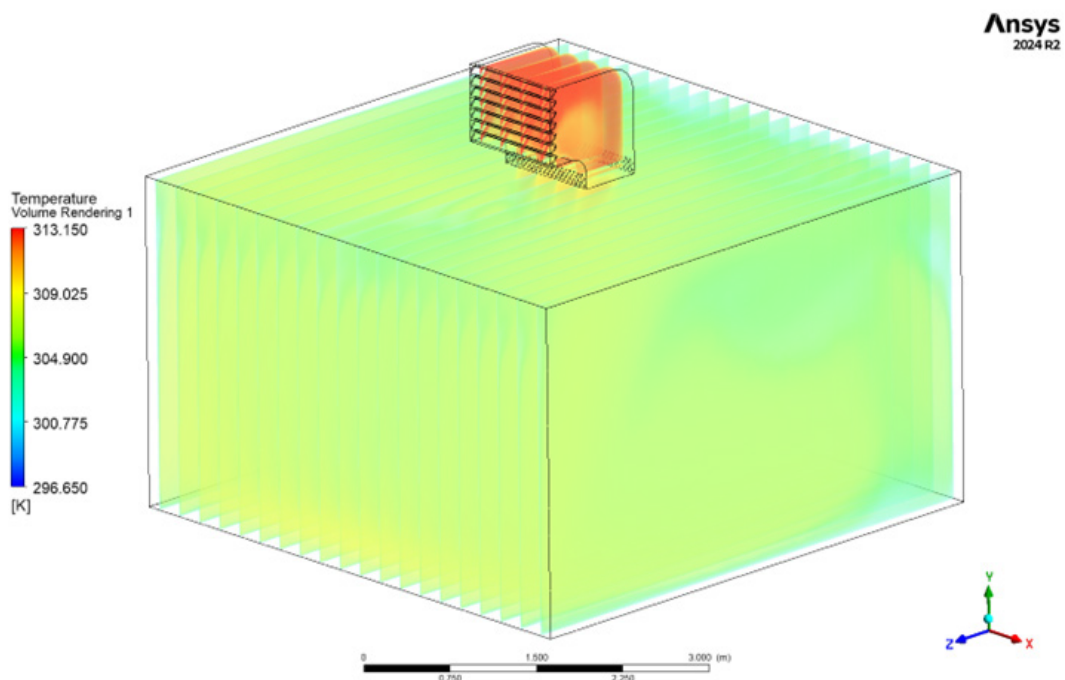


Figure 3: Volume rendering of the entire room to check the temperature reduction inside the room.

## 5. Conclusions

The analysis of the windcatcher system using Ansys Fluent simulation software demonstrates its effectiveness in cooling an enclosed space by leveraging passive cooling techniques. The results show that the integration of angled louvers and cold pipes significantly reduces air temperature as it flows into the room, creating a comfortable thermal environment without the need for active cooling systems. The 2D temperature contours reveal a clear gradient of cooling as the air interacts with the cold pipes, while the 3D volume rendering confirms uniform temperature stratification and effective dispersion of cooled air throughout space.

Overall, the simulations highlight the feasibility of using windcatcher systems as sustainable and energy-efficient cooling solutions for buildings. This design, inspired by (Calautit, 2016), proves capable of delivering significant thermal comfort, reducing reliance on energy-intensive mechanical systems, and contributing to greener building designs. Further optimization and testing under varying external conditions could enhance its applicability across different climates and building layouts.

## 6. References

Calautit, J. K. (2016). A passive cooling wind catcher with heat pipe technology: CFD, wind tunnel and field-test analysis. *Applied Energy* 162, 460-471.



© 2025 ANSYS, Inc. All rights reserved.

## Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

## Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

## Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit [www.ansys.com/education-resources](http://www.ansys.com/education-resources).

## Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

**ANSYS, Inc.**  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit **[www.ansys.com](http://www.ansys.com)** for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2025 ANSYS, Inc. All Rights Reserved.