

Min Project on ANSYS ICEMCFD

Geometry Cleanup for Computer Cabinet with Fan



LearnCAx

Inspire | Educate | Mentor

www.LearnCAx.com

Usage Terms

All material in this document is, unless otherwise stated is the property of LearnCAx. Copyright and other intellectual property laws protect this materials. Reproduction or retransmission of the materials, in whole or in part, in any manner, without the prior written consent of LearnCAx, is a violation of copyright law.

This tutorial document is made available for your personal learning purpose. Reproduction of any content of this document with any kind of modification is not allowed. Posting of this document, tutorial inputs files on any other website, social media, forums etc. is not allowed. You are open to share the original link of this tutorial with the community. Usage of this tutorial for any commercial purpose is strictly prohibited. Violation of any of these terms will call for legal action and blacklisting of your account from LearnCAx website.

LearnCAx,
1 Akshay Residency, 50, Anand Park, Aundh, Pune, 411007, India

This mini project about geometry cleanup and fluid domain extraction for computer cabinet with fan geometry. The geometry of empty computer cabinet with inlet grill is to be created. Complete fan geometry is provided as input which needs to be cleaned and assembled on cabinet. The objective of project is create assembly of cabinet and fan for flow analysis. The input fan geometry contains all components including solid parts. This geometry is to be cleaned for flow analysis.

1 Prerequisites

The main pre-requisite for this mini project is good understanding of various geometry operations available ANSYS ICEMCFD. It is also expected that you have understanding of what is fluid domain and what geometry details are required in case if only flow analysis is to be performed. Before taking this project, please make sure that you have gone through lesson on “Geometry Creation” and “Geometry cleanup for control valve tutorial”.

2 Geometry Cleanup – Why?

In an application engineer’s point of view, CFD is a three step process viz. pre-processing, solution and post-processing. The CFD process starts with defining the domain of interest within which governing equations are to be solve. Domain is defined by shape and size of all its confining boundaries. In almost all industrial cases the domain definition is created using CAD software and transferred to meshing software using neutral file formats like IGES, STEP etc.

Traditionally, the most time consuming part in complete CFD process is creating a good quality mesh. Considering the advancements in meshing techniques available in today’s meshing tools, lot of automation of meshing algorithm is done. The main bottleneck is making CAD geometry mesh ready or converting dirty CAD to clean mesh-able CAD.

All meshing algorithms needs a water tight geometry (defined within the constraints of the tolerance considered by meshing software). Creating a clean watertight geometry from dirty CAD is sometimes challenging and in most of the cases very time consuming job.

The geometry cleanup mainly depends on physics under consideration and computational power available for solution. If only flow analysis is to be done, the geometry of only “wet surfaces” should considered and all other components should be removed. In ideal situation, all geometry features should be captured in mesh. But that would result into huge cell count for most of the industrial problems. So the cleanup of small feature is required to be done to reduce the total cell count at the same time not to affect the flow physics considerably. Engineer should have good understanding of flow physics so that he/she can decide what sort of cleanup can be done.

In this project, the input CAD geometry of fan “RAW” CAD data. This CAD include many unwanted parts if only flow analysis is to be carried out. This CAD also contains many small features which should be removed.

3 Problem Definition

3.1 Objective

Axial fan is the device used where one needs large volume of fluid with moderate pressure rise. Axial fans have broad industrial and commercial users. They are used to introduce air or other gases into the process reactors for the chemical and pharmaceutical industries; to assist combustion in furnace; to dry agricultural and manufactured products; in air handling units for cleanrooms used in pharmaceutical, biotechnology, hospital and micro-electronic industries; for roof and wall-mounted power ventilators; in commercial cooking exhaust fans; for dust collection and many more.

Prediction of flow field and performance of axial fan is an important aspect during design, modification and trouble-shooting. The commercial CFD tools are developed to predict accurate flow physics for such rotating components.

There are many standard axial fans available in the industry for different applications. In this mini project, axial fan used in Personal Computer (PC) for cooling purpose is considered. The purpose of this project is to clean provided RAW CAD data for flow analysis, to assemble the clean fan geometry with cabinet and to extract fluid domain for meshing.

Following are the steps to be completed in this project:

1. Create empty cabinet geometry with inlet grill
2. Clean RAW CAD data of fan
3. Assemble the clean fan data with cabinet

The main objective is to extract clean fluid domain for flow analysis.

3.2 Geometry Details

The layout of geometry is shown in Figure 1.

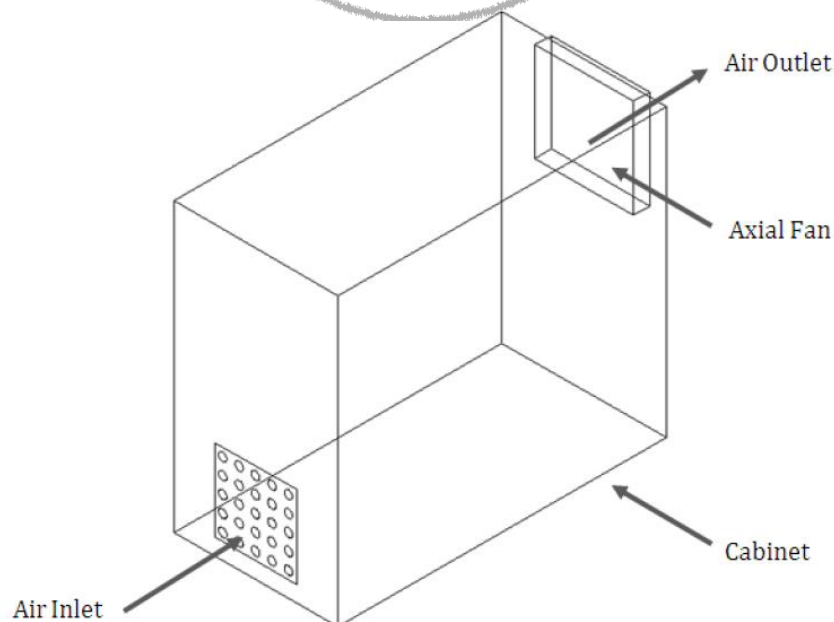


Figure 1: Geometry layout

Details of cabinet geometry are given in Figure 2.

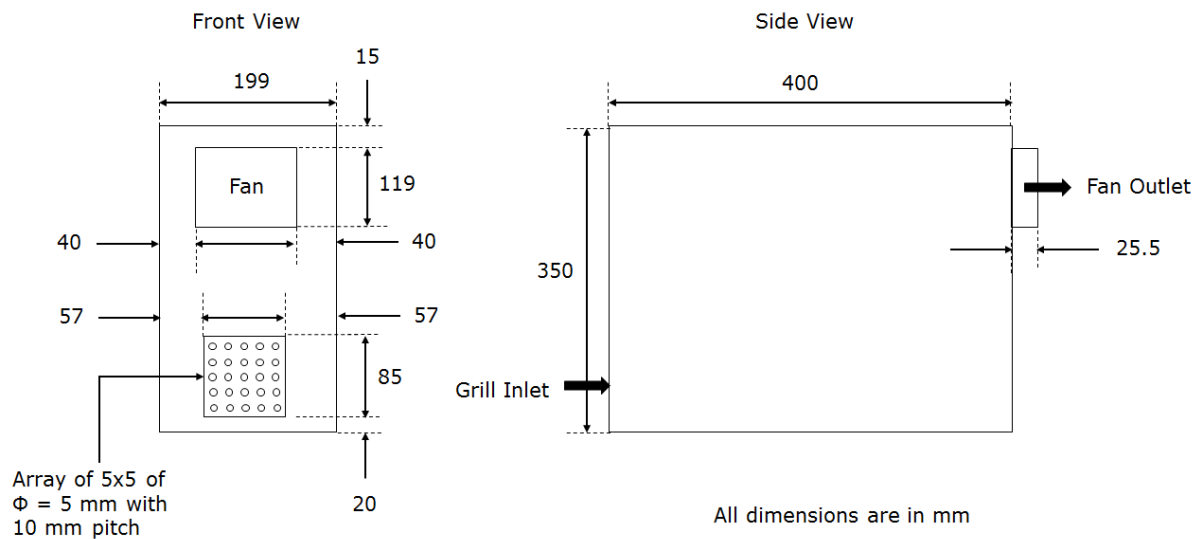


Figure 2: Cabinet geometry details

The grill is of size 85 x 85 mm located at the bottom. Fan is attached at top. The details of fan are given in Figure 3.

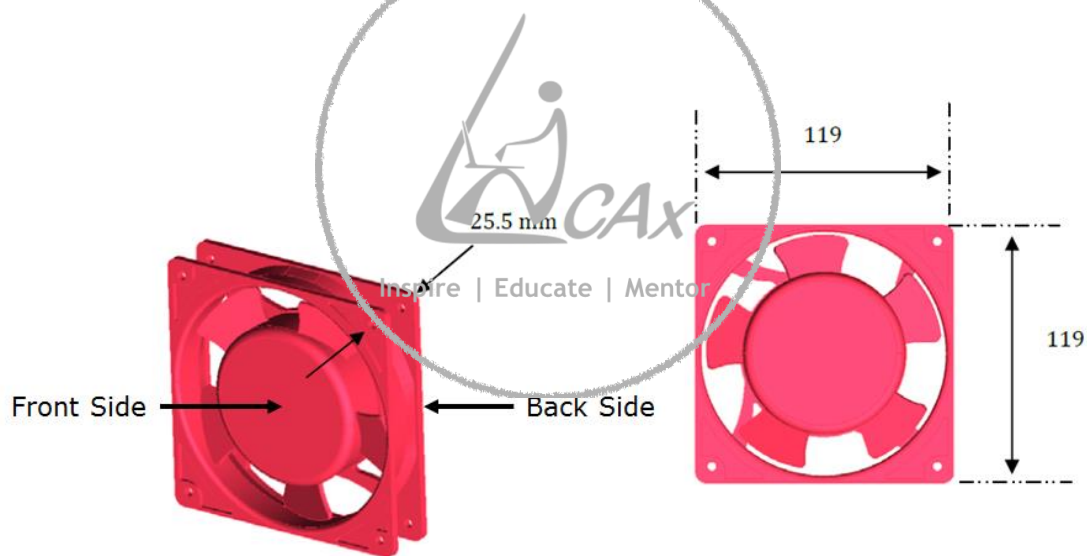


Figure 3: Fan dimensions

Fan is required to be attached to cabinet such that the back side of fan is attached to cabinet and front side is towards the outlet.

3.3 Expected Tasks and Sequence

Following is the list of tasks involved in this project. They are listed in the sequence they should be carried out.

1. Create the cabinet geometry with given dimensions
2. Import fan geometry
3. Move fan geometry to its correct location as shown in Figure 2
4. Clean the fan geometry to have only wet surfaces required for flow analysis

5. Remove small features of fan geometry to simplify geometry for meshing
6. Create inlet grill

4 Download Input Files

Links to download all necessary inputs files are given below. They are compressed zip files. Download them in one folder and unzip the files. This would create all necessary inputs files along with PDF copy of this project details. The geometry files is given in STEP format.

1. PDF instructions for this mini project
2. Fan geometry file (STEP)

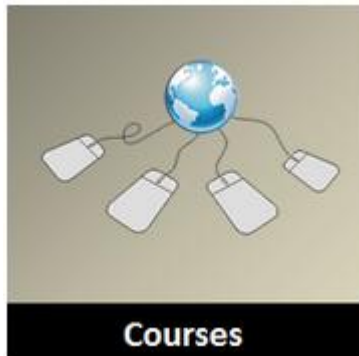
You can also download both the files from “Shared Files” section on lesson page.

5 Results and Discussion

If you have any specific query about the mini project or want to share the results of this project, please post them on [course discussion forum](#).



LearnCAx Knowledge Base



Courses offered by LearnCAx are designed to meet all your needs. It has range of FREE and PREMIUM courses which is designed to meet the industry requirements. All the courses are self-contained with video lectures, quiz, assignments and projects. Every course comes with FAQs and discussion forums where you can get answers to all your questions. Each course contains live assistance from LearnCAx faculty where faculties will guide you through online sessions and desktop sharing.

[View Courses](#)



Blogs is the place where our coaches share their knowledge through articles. This includes best practices, advance techniques, and recent development in respective field. LearnCAx is backed with strong industrial consultancy team. This team does projects for industries. As LearnCAx main focus is “from academics to industry”, blogs gives us an opportunity to share details about our industrial work. It’s not only about what is done, but also about how the project is done. The objective is to give student’s more knowledge about industrial project so that they feel connected to the industry.

[View Blogs](#)



No matter what is the form of learning, an interaction with experts is an inevitable part of every learning process. LearnCAx faculty conducts webinars to share the knowledge with you. Let it be knowhow of software, introduction to a particular topics or discussing fundamentals of a subject, all webinars are targeted towards sharing the knowledge and getting feedback about what your training needs are? Webinar is also a place for our consultancy team to share their work with you. All these live sessions would give an opportunity to you to talk to the experts in the domain.

[View Webinars](#)

Create FREE **LearnCAx** account to access all the knowledge base

[Create Account](#)