ANSYS ICEMCFD

"Practice On" Geometries for Meshing

LearnCAx

Inspire | Educate | Mentor

www.LearnCAx.com

LearnCAx,

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

"Meshing is an art". It's not only about knowing the tools and techniques available in software but also about applying those tools at right time and for right purpose. Understanding the tools, how it work etc. is required, but practicing those tools on various geometries is must for developing mesh expertise.

Here we have given set of clean geometries of only fluid domain which you can practice on. All of the geometries are extract of real life industrial problem. These geometries are ranging from simple to moderately complex configuration.

1 Prerequisites

The main pre-requisite for this project is good understanding of all meshing tools and techniques available in ANSYS ICEMCFD. It is expected that you have gone through the lessons on Unstructured and multi-block structured meshing before you start working on these geometries. It is also advised that you go through tutorials, demos and complete all mini projects before you start working on these geometries. It is strongly advised not to start working on these geometries before you complete all lessons, demos, tutorials and mini project.

2 Problem Definition

All the geometries given are extract of real life industrial problems. There two ways one can mesh these geometries. First, you can do unstructured meshing with triangular, tetrahedral and prism cells. Second, you can create all quadrilateral and hexahedral cells using structured multi-block meshing technique.

It is expected that you mesh all the geometries using both unstructured and structured meshing approaches. This will help you practice and perfect both meshing techniques. When you start working on any real life project, you may choose one of the meshing technique depending on the complexity of geometry, time available for meshing and overall accuracy requirement.

The main idea is to create a representative mesh for all the geometries. The mesh size always depends on the flow physics you want to capture. The inputs for flow conditions are not given here. You can assume any flow condition for calculating the mesh size. May be you can do some literature survey to find a similar device or configuration and extract the flow conditions and use those conditions to calculate the mesh size. As mentioned, idea is to learn how to mesh these geometries rather than crating a mesh for specific flow condition.

It is expected that you create appropriate boundary layer mesh. In case of unstructured mesh, put prism layers with at least 4 layers. In case of structured meshing, create appropriate bunching of nodes near wall.

3 Input Geometry Files

A brief description of geometry and link to download geometry file is given below. All geometry files are given in ANSYS ICEMCFD format (tin). These files are created using ANSYS ICEMCFD 13.0 version and would not work with any lower version. The files are given in compressed zip format. Download and unzip file for each geometry to extract ANSYS ICEMCFD geometry (tin) files.

4 Geometries

4.1 2D Heat Exchanger: Type 1

This geometry is a 2D extract of shell-tube-heat exchanger. For this geometry only shell side fluid mesh is to be created. The main challenge is to capture small gap between tubes.

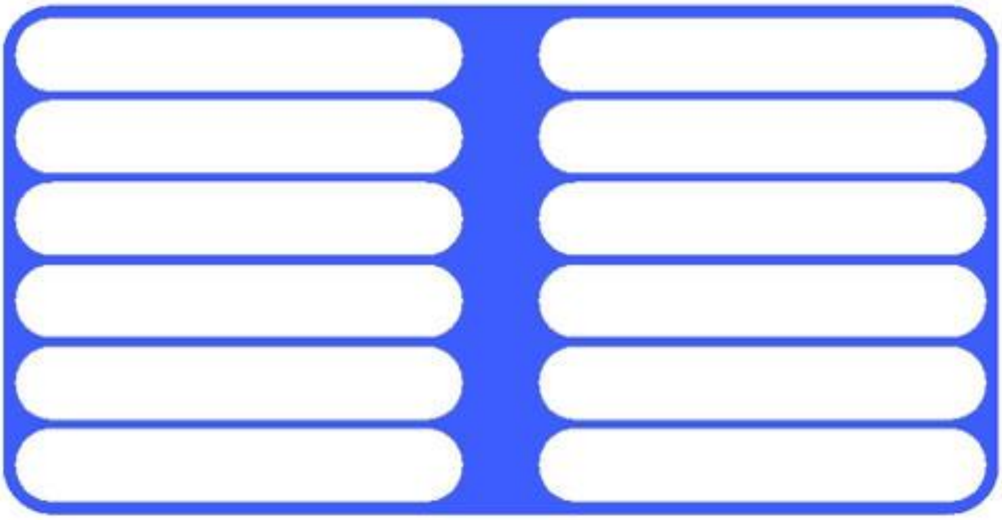


Figure 1: 2D heat exchanger – Type 1

4.2 2D Heat Exchanger: Type 2

This geometry is also a 2D extract of shell-tube-heat exchanger. In this case shell and tube is of circular shape. For this geometry both shell and tube side fluid mesh is to be created.

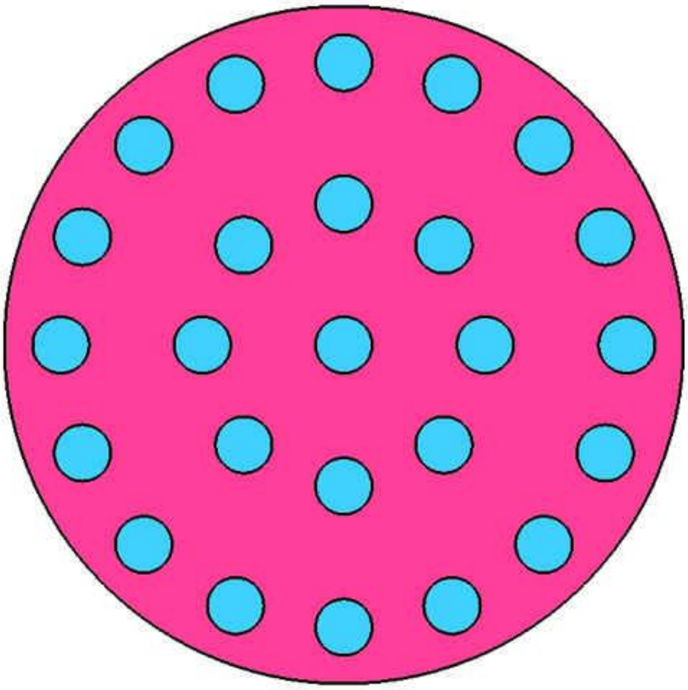


Figure 2: 2D heat exchanger – Type 2

4.3 3D Heat Exchanger

This geometry is a 3D shell-and-tube heat exchanger geometry. The shape of heat exchanger is rectangular and mesh for both shell and tube side fluid is to be created.

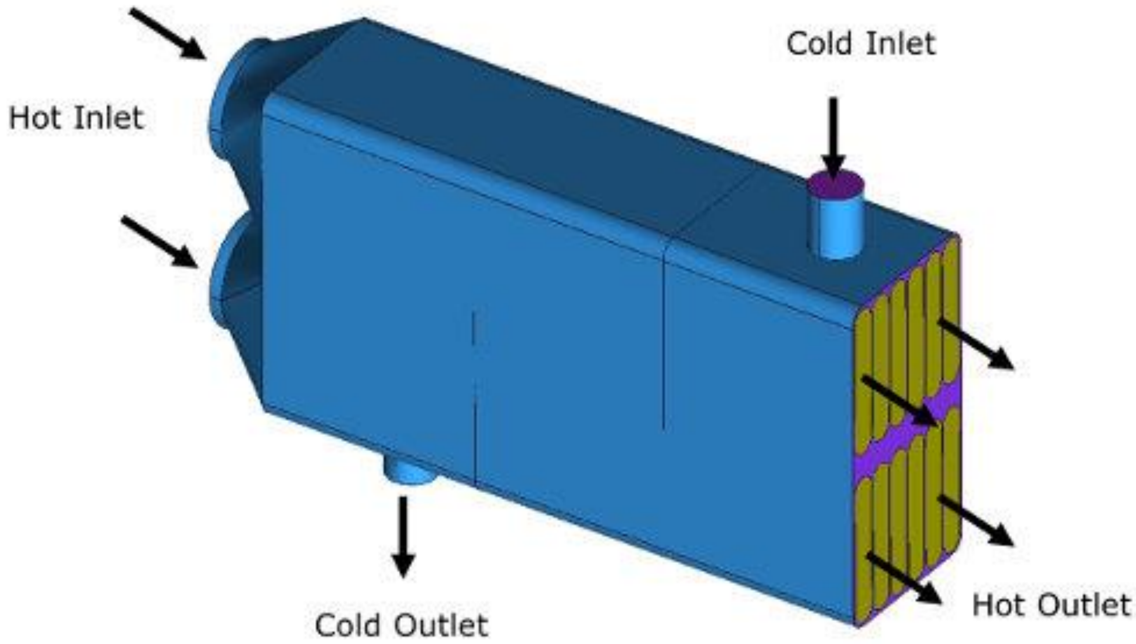


Figure 3: 3D heat exchanger

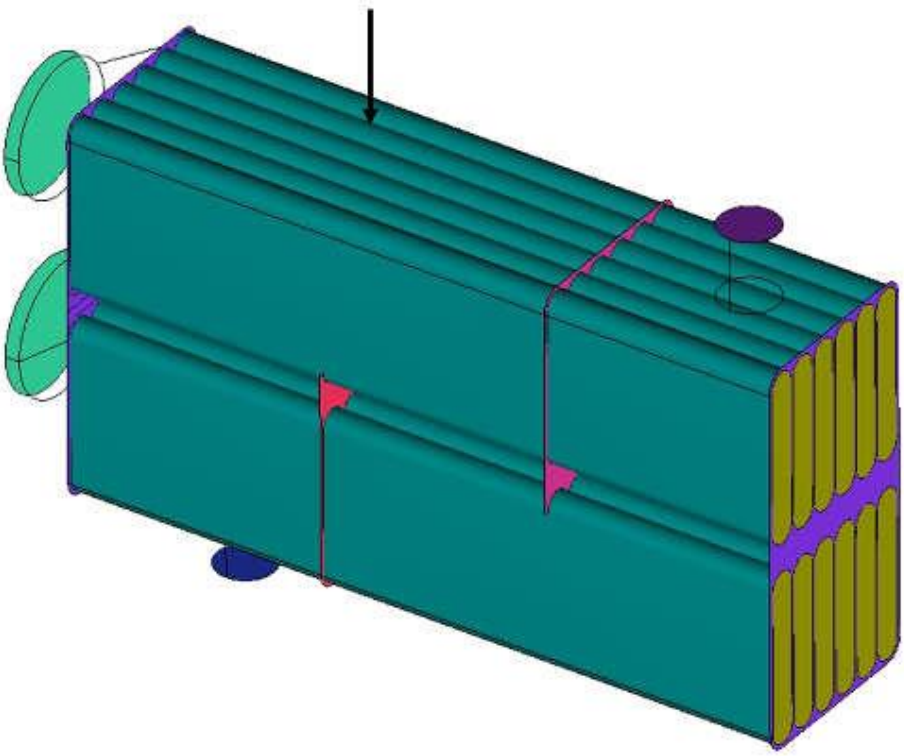


Figure 4: 3D heat exchanger tubes

4.4 Counter Flow Heat Exchanger

This geometry is a 3D shell-and-tube heat exchanger geometry with counter flow arrangement. Both shell and tube side fluid mesh is to be created.

Figure 5: Counter flow heat exchanger

4.5 Cyclone Separator

This is simple cyclone separator geometry. The dust collecting hopper is not included in this geometry and has only top surface on dust collector.

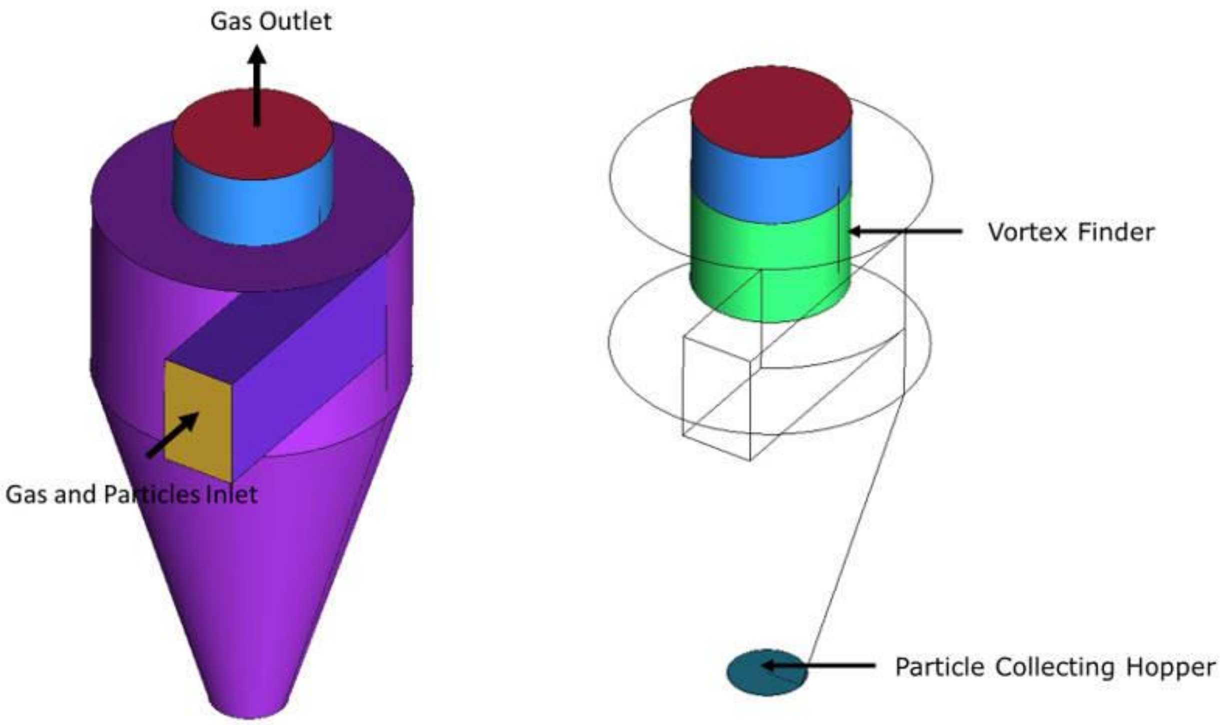


Figure 6: Cyclone Separator

4.6 Office HVAC Duct

A typical building HVAC system contains AHU (Air Handling Unit), supply ducting, supply nozzles and/or diffusers, return grills and return ducting. This geometry is an extract of complete HVAC system for office space. It only includes supply ducting for four office room.

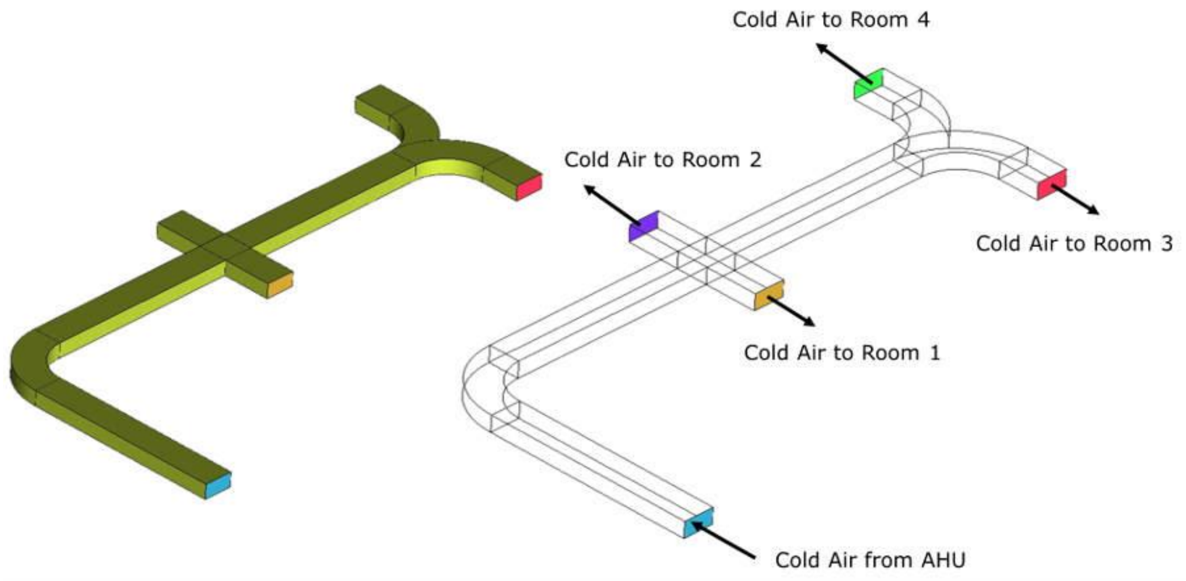


Figure 7: Office HVAC duct

4.7 Engine Silencer

This is a typical geometry of engine silencer. In actual condition, the engine silencer would contain many internal components to attenuate the sound generated by engine. In this geometry all internal components are removed and a plane silencer geometry is given. The objective is to create a mesh for flow analysis inside a silencer.

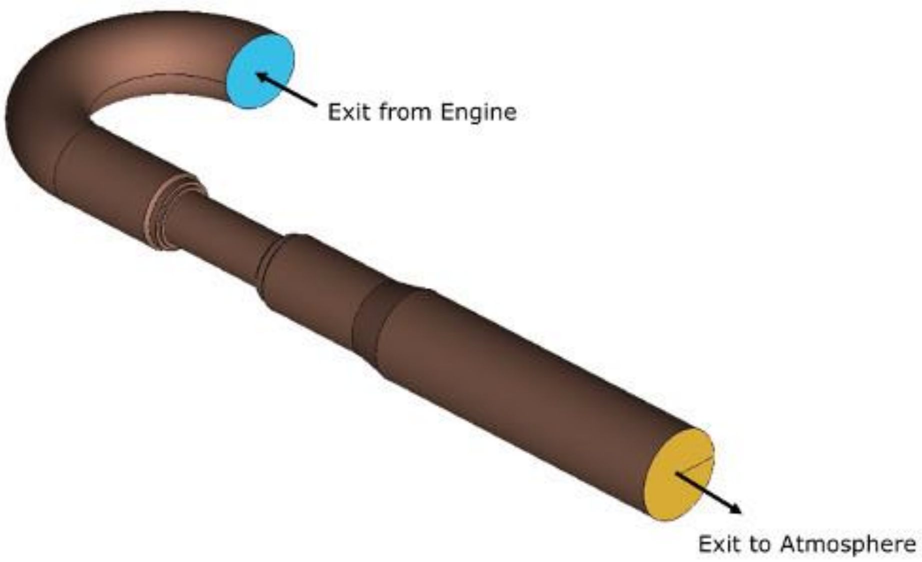


Figure 8: Engine silencer

4.8 Solar Dryer

Solar dryers are utilizes solar energy to dry items like grapes, fish etc. Solar energy is collected using solar collectors, and supplied to air. This heated air is then passed through solar dryer containing food items. The hot air is passed through inlet channel and distributed over drying tray placed in drying section. The number of trays depends on quantity of food to be dried and amount of hot air available. Uniform flow distribution at inlet of drying section is critical factor for optimum performance of dryer.

The given geometry is one of such solar dryer. This geometry contains only inlet channel, drying section and outlet channel. All the trays are modeled with zero thickness walls. The objective is to create a mesh for flow analysis.

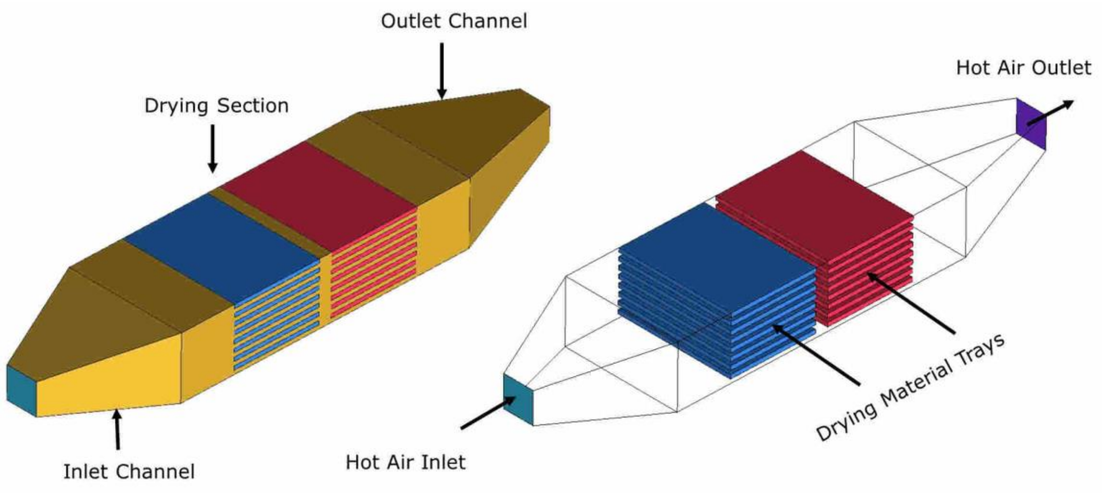


Figure 9: Solar dryer

4.9 T-Section

This is simple T-connection used in typical piping system. It has two inlet branches and one outlet branch.

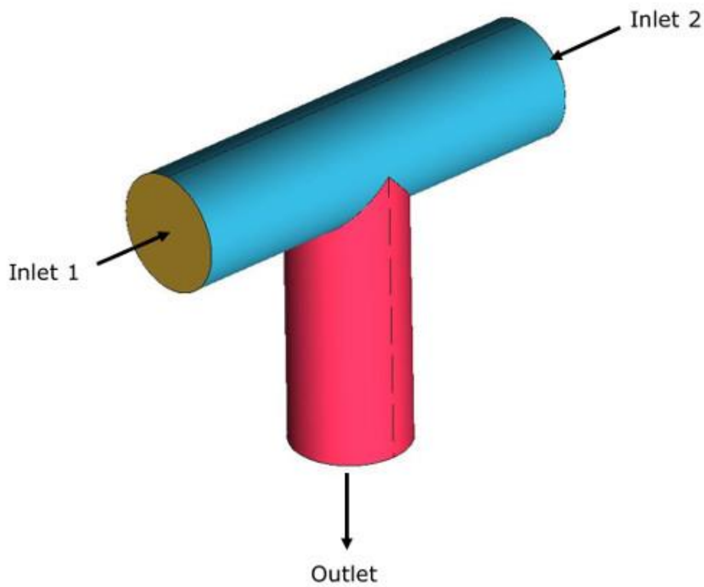


Figure 10: T-connection

5 Download Other Files

Links to download a PDF copy of this document is provided below for your future reference. It's a compressed zip file. Download and unzip the file will get a PDF copy of this document.

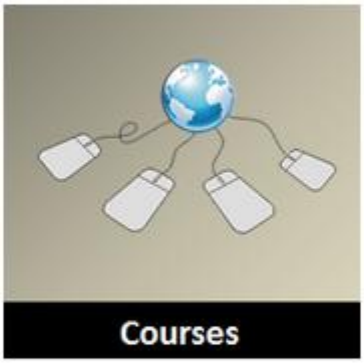
1. PDF copy of this document

You can also download all the input geometry files and PDF copy of this document from "Shared Files" section on lesson page.

6 Results and Discussion

If you have any specific query about meshing of any of the geometry or want to share the meshing results, 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