

Min Project on ANSYS ICEMCFD Unstructured Mesh Generation for Catalytic Converter



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 deals with meshing of catalytic convertor geometry. Unstructured tetrahedral mesh needs to be created for complete geometry. It is also expected that you create prism layer for capturing boundary layer physics. As this problem demands for modelling porous zone, your understanding of multi zone meshing is also tested. After completing this mini project you will be comfortable in unstructured meshing for simple geometries including multi domain problems.

1 Prerequisites

The main pre-requisite for this mini project is basic understanding of unstructured meshing using ANSYS ICEMCFD. Before taking this test, please make sure that you have gone through our lectures on unstructured surface, volume and prism meshing.

2 Problem Definition

2.1 Domain

Automobile → Emission Control System → Catalytic Converter

The flow field non-uniformities at the inlet of catalytic converters are considered undesirable for its performance. CFD simulations are widely used to study the flow distribution and the conversion efficiency of automotive catalytic converter. These studies help to understand the performance of the catalytic converter in order to optimize the converter design.

2.2 Device Understanding

Catalytic Converter is an important part of automobile's emission control system. The job of the catalytic converter is to convert harmful pollutants (CO, NOx, and HC) into less harmful emissions before they leave the car's exhaust system.

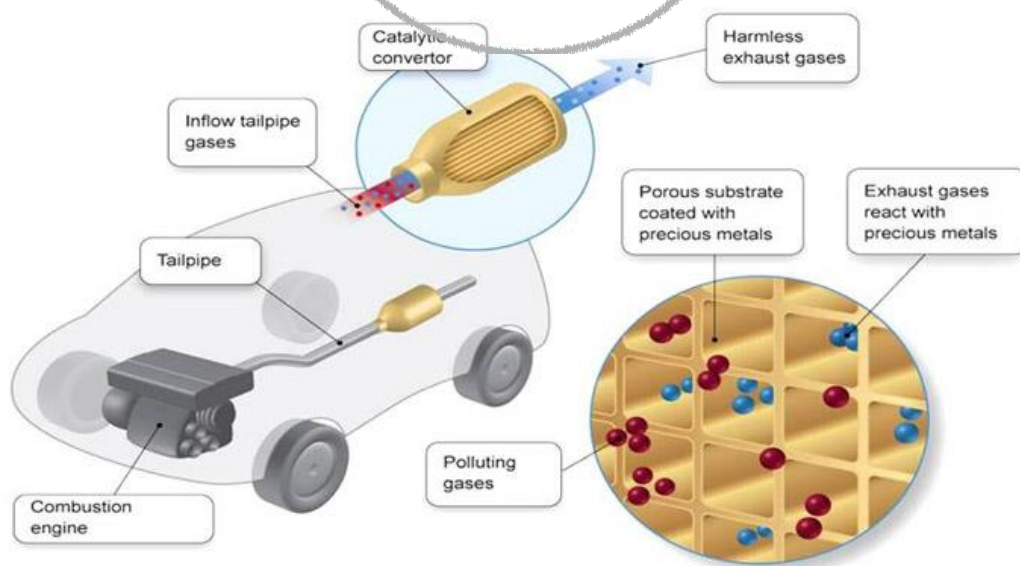


Figure 1: Location of Catalytic Converter in Automotive

Image source:

<http://www.preciousmetals.umicore.com/recyclables/SAC/CatalyticConverter/>

A catalyst is a substance that accelerates a chemical reaction, but is neither reactants nor products of the reaction. In the catalytic converter, there are two different types of catalyst at work, a reduction catalyst to reduce NO_x emissions and an oxidation catalyst to convert unburned hydrocarbons and carbon monoxide to H₂O and CO₂ by oxidation. Both types consist of a ceramic / honeycomb structure coated with a metal catalyst, usually platinum, rhodium and/or palladium. The idea is to create a structure that exposes the maximum surface area of catalyst to the exhaust stream, while also minimizing the amount of catalyst required, as the materials are extremely expensive.



Figure 2: Catalytic Converter Working

Image source:

<http://www.preciousmetals.umicore.com/recyclables/SAC/CatalyticConverter/>

2.3 Problem Definition

The catalytic converter under current study is shown in figure below. Nitrogen flows in through the inlet with a uniform velocity of 22.6 m/s, passes through two substrates which has square shaped channels and then exits through the outlet.

In order to reduce the cell count and complexity of problem, the substrates are modeled as porous media. In order to model the substrate as porous media, one need not to create the internal details of the substrate. The substrate will be modeled as single volume with porous media properties. Flow through substrate is characterized by viscous and inertial loss coefficients in flow direction. The substrate is impermeable in other directions.

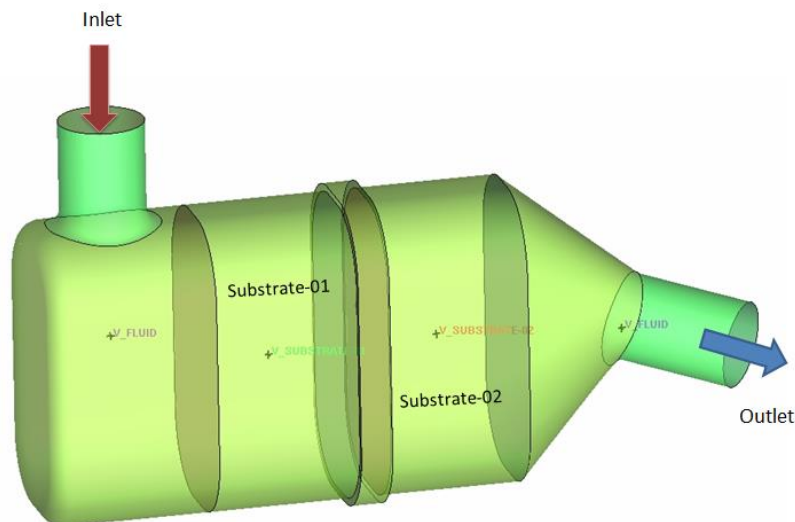


Figure 3: Catalytic Converter Geometry for Current Study

In this mini project you have to create unstructured mesh for the given catalytic converter using ANSYS ICEMCFD. The mesh requirement is as below:

- Appropriate mesh size should be calculated and used for meshing. The mesh size should be sufficient enough to capture the geometry features (curvature etc.) and to capture the flow features (boundary layer, turbulence etc.)
- There should be conformal mesh between all three domain with domain labels V_FLUID, V_Substrate-01, V_Substrate-02. These three volume meshes will be used to provide appropriate volume conditions (fluid/porous media) in ANSYS FLUENT.
- The surfaces between all three domains should have appropriate labels and they should be visible in ANSYS FLUENT. (This requires understanding of multi domain mesh using ANSYS ICEMCFD).
- Quality of mesh: The minimum quality of mesh to pass this test is given below. If you can achieve the mesh quality more than criteria provided below would be great.

SR. No.	Quality Criteria	Minimum Value Expected
1	3x3 Determinant	0.3
2	Angle	23

3 Hints

- You can assume the turbulence model that would be used for simulation is in k-ε with standard wall functions. This turbulence model demands Y+ in the range of 30 to 150. You can use this data to calculate the first cell height.
- Use scan plane to understand the issues in the volume mesh. This will help in improving the quality of mesh

4 Further Reading

- ANSYS FLUENT document on setting up porous media simulations
- ANSYS FLUENT tutorial named "Modeling Flow Through Porous Media"
- <http://www.autocatalyst-recycling.umicore.com/catalyticConverter/>

5 Solver Settings

Although carrying CFD simulation is not a part of this mini project, you can carry out CFD simulation to check the performance of mesh you have created. In this section, the necessary boundary conditions for solver setup using ANSYS FLUENT are given. You can use this boundary condition if you are willing to do the simulation on created mesh.

You can use following data for setting up simulation in ANSYS FLUENT:

- Dimension of geometry : mm
- Inlet gas : Nitrogen (N₂) with velocity 22.6 m/s
- Outlet: Static pressure = 0 Pa (Gauge)
- Material properties of Nitrogen:
 - Density: 1.138 kg/m³
 - Viscosity : 1.663e-05 kg/ms
- Porous media properties:

Direction	Viscous Resistance (1/m ²)	Inertial Resistance (1/m)
Along Flow	3.846e+07	20.414
Perpendicular to Flow	3.846e+10	20414

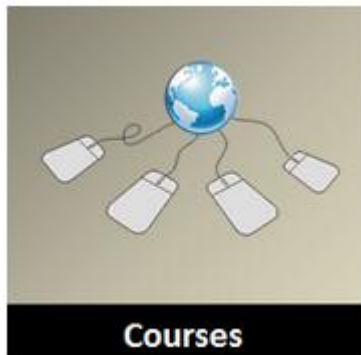
6 Download Input Files

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 ANSYS ICEMCFD format (tin). This file is created using ANSYS ICEMCFD 12.0 version and would not work with any lower version.

1. PDF instructions for this mini project
2. Catalytic convertor geometry files (tin)

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

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](#)