

Min Project on ANSYS ICEMCFD Multiblock Mesh Generation for 2D Missile Geometry



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 the multiblock mesh generation for 2D missile geometry. This mini project tests your understanding of various multiblock mesh generation options in ANSYS ICEMCFD. After completing this mini project you will be comfortable in creating multiblock structured meshing for simple geometries

1 Prerequisites

The main pre-requisite for this test is basic understanding of multiblock hexahedral meshing using ANSYS ICEMCFD. Before taking this test, please make sure that you have gone through lessons on “Structured Hexahedral meshing”.

2 Problem Definition

This is an extension of previous mini project “Geometry creation for 2D missile geometry”. Missile geometry given in this problem is typical missile geometry used in aerospace industry. The objective of this assignment is to generate a 2D multiblock structured mesh. The mesh is to be generated to carryout external flow CFD simulation. All geometry dimensions are given in Figure 1. You can create the new geometry using given dimensions or use the geometry created in “Geometry creation for 2D missile geometry” mini project. We have also given the geometry as input if you want to start with readymade geometry.

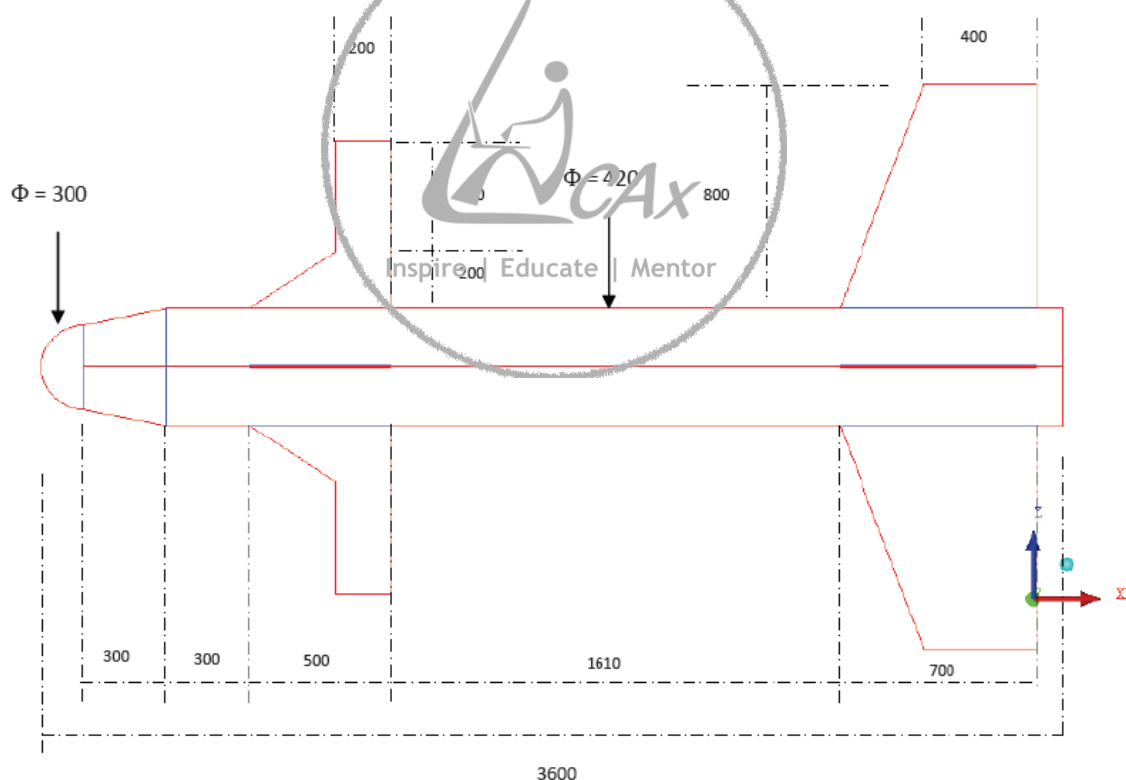


Figure 1: Geometry details (Note: All dimensions are in mm.)

If you wish to create geometry before starting this mini project, make sure that you create domain necessary for external flow analysis as shown in Figure 2. You need to decide what extent of CFD domain should be considering the external flow requirements.

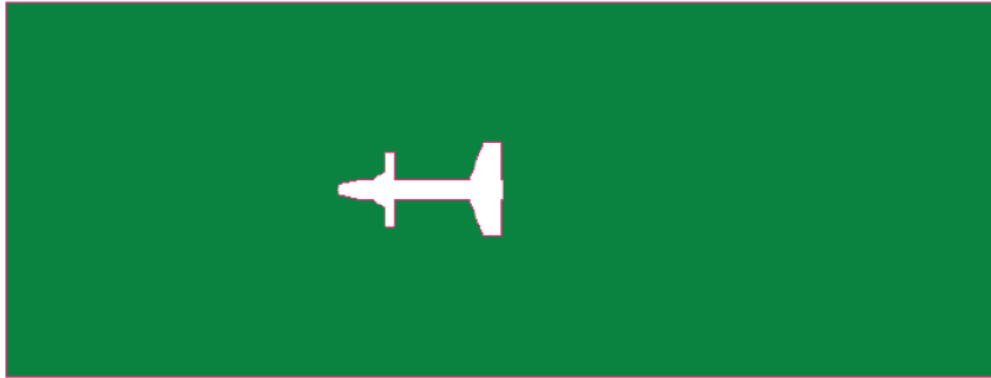


Figure 2: External domain

Once the missile and external domain geometry is created, you need to create following parts to assign boundary conditions:

- Inlet
- Outlet
- Tunnel walls
- Missile walls

3 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 13.0 version and would not work with any lower version.

1. PDF instructions for this mini project
2. 2D missile geometry files (tin)

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

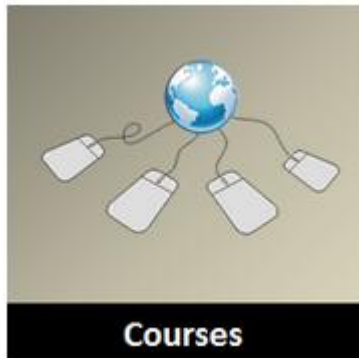
4 Hints

- 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. This is just given as a procedure to be followed. You can apply the same procedure for correct turbulence model for external flow over missile.
- Spend more time on deciding the block topology. Sketch the block topology on a paper roughly before you start creating block topology in ANSYS ICEMCFD.

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](https://www.LearnCAx.com).

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](#)