Min Project on ANSYS ICEMCFD

Multiblock Mesh Generation for Heat Exchanger Geometry

LearnCAx

Inspire | Educate | Mentor

www.LearnCAx.com

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

This mini project deals with meshing of tube in tube heat exchanger geometry. Multiblock hexahedral mesh needs to be created for complete geometry. After completing this mini project you will be comfortable in multiblock structured meshing for simple geometries. It is also expected that you calculate fist cell height using boundary condition material properties information given and create a mesh satisfying first cell height.

1 Prerequisites

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

2 Problem Definition

In this project, a typical tube-in-tube heat exchanger geometry is to be meshed. The objective of this project is to apply your understanding of multiblock structured mesh on sample heat exchanger geometry. The geometry considered is just an extract of real life tube-in-tube heat exchanger. In reality, there would be large number of tubes and baffles.

The geometry considered for this project is shown in Figure 1.

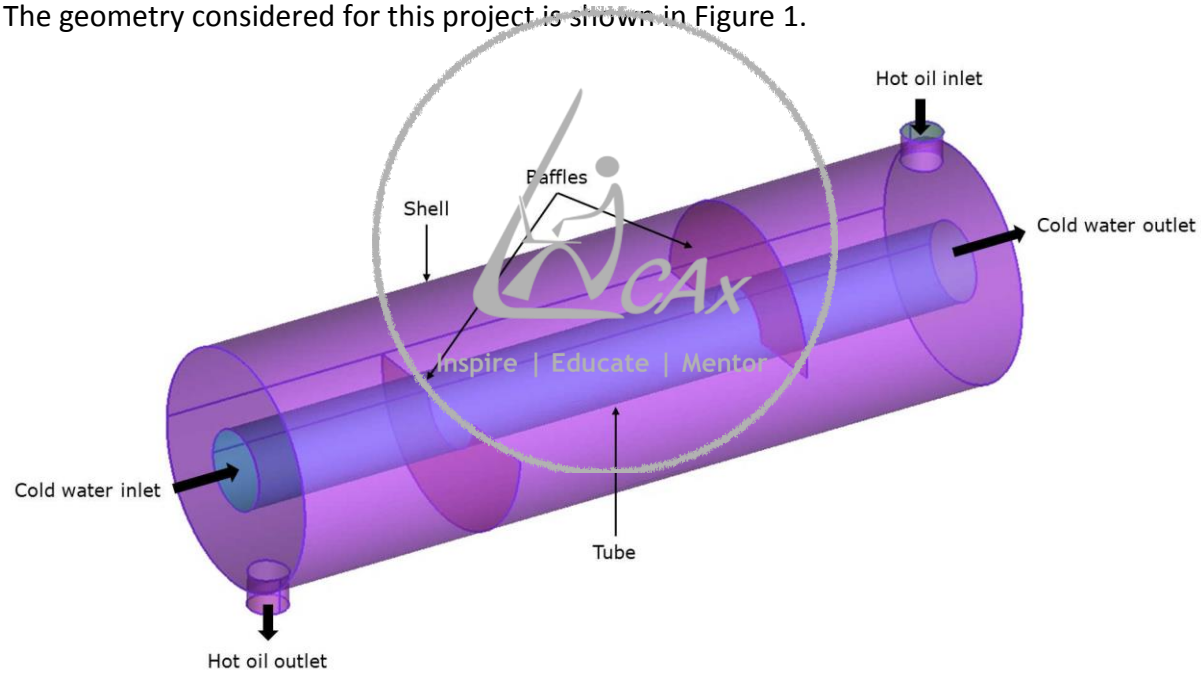


Figure 1: Tube-in-tube heat exchanger geometry

It is also expected that you should create boundary layer mesh with appropriate Y+ values. You can assume k-ε turbulence model with standard wall functions for simulation. This turbulence model demands Y+ in the range of 30 to 150. You can use data given in Table 1 and Table 2 for designing your mesh.

	Cold Water	Oil Inlet
Mass flow rate (kg/s)	0.5	2
Temperature (°C)	45	80

Table 1: Inlet conditions

	Cold Water	Oil Inlet
Density (kg/m^3)	990	852
Conductivity (W/m°C)	0.637	0.138
Prandtl number	3.91	490
Kinematic viscosity (m^2/s)	0.602 x 10-6	37.5 x 10-6

Table 2: Material properties

In this project you have to create hexahedral mesh for the given tube-in-tube heat exchanger geometry using multi-block method in ANSYS ICEM CFD.

- There should be conformal mesh between two domain with domain labels water and oil. These three volume meshes will be used to provide appropriate volume conditions (material properties) in ANSYS FLUENT.
- The surfaces between two domains should have appropriate labels and they should be visible in ANSYS FLUENT. (This requires understanding of multi domain mesh using ANSYS ICEM-CFD).
- 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 blow would be considered in grading scheme.

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

Table 3: Mesh quality

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. Tube-in-tube heat exchanger geometry files (tin)

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

4 Hints

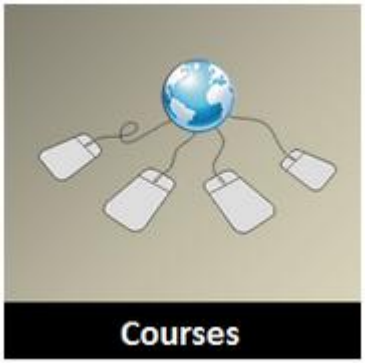
- You can assume the turbulence model that would be used for simulation is in k-ε with standard wall functions. This turbulence model demands Y+ in the range of 30 to 150. You can use this data to calculate the first cell height.
- In case of multi-domain mesh, the face between two blocks of respective volume domain will be projected on closest surface by default. This projection will carry the label of the surface on which it is projected.
- Spend more time on deciding the block topology. Sketch the block topology in a paper roughly for two or more cut/cross section of geometry

- Use scan plane to understand the issues in the volume mesh. This will help in improving the quality of mesh

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