CFD theory Sessions

Víctor Martínez Viol

October 5, 2016

Abstract

This paper has been written with the effort on getting comfortable when writing using LaTeXso please don't hate. It includes some notes taken in *Application of CFD to engineering problems*. NANDO NANDO

1 Session 4

1.1 Using Salome Meca

The first hour consisted on building the geometry of an Y-pipe with a software called *Salome Meca*. A complete guide on video (with an older version of the program) can be found in:
`http://caelinux.org/wiki/downloads/docs/Pipe2007/PipeGeom.htm`

We open Salome by writting the following line on a terminal window '/opt/salome_meca/appli_V2016/runAppli'

1.2 Constructing the mesh

We want to make the mesh uing paraview. First we have to create a case. Look ffor a similar tutorial as a basis for our mesh (and of course similar to the case we want to run). For the pipe problem we can start with the motorbike case

We must copy the motorbike folder, change its name to Ypipe, in the system folder there is a file named `BlockMeshDict`. We must modify the block geometry to put the pipe inside it. The new values for the block are: See david's notes

the next steps involve refining the mesh. To do so we are going to use **snappyHexMesh**(See reference 1). We must define the level 0 of the mesh..... See Drugas notes.

The next step is to take `SnappyHexMesh` dictionary. Rather than create it is better to take it from `openfoam4/applications/uilities/mesh/generation/snappyhexmesh` and paste it to the `case/system` folder. Inside the file there are a lot of comments and explanations about the mesh configuration and parameters.

It is possible to define an upper level of discretization inside our blockMesh using searchablebox

SnappyHexMesh is one of the only meshers that can perform a parallel mesh of the domain (using multiple cores). The parameter `maxLocalCells` and `maxGlobalCells` define the maximum number of cells per core and the maximum number of cells on the global mesh respectively.

....

the parameter `LocationInMesh` let us decide wether we want to mesh inside or outside our geometry. the input has to be a point inside of the object

inside the folder `Ypipe/trisurface` we must have the four surfaces .stl files.

References