January/May

Small ANSYS Report

August 2018

Margarida Giestas Lima

Estrada do Paço do Lumiar, 22
1649-038 LISBOA - Portugal
Tel: +351 210 924 600/1
Fax: +351 217 166 966 online:217166966@fax.ptprime.pt
www.lneg.pt

Contents

Figures Content

1. Introduction

This report presents the work developed during the first three months of 2018 for the NewSol Project. After doing a devoted and necessary State of Arts, in the aim of ANSYS Fluent procedures, the work deals with the development of a thermocline, concerning its design and meshing.

It was decided to design and dimension the geometry in Design Modeler of a Thermocline concrete tank simulation, according to Figure 1.

In the sequence of the design a mesh must be proposed. The purpose of this mesh is allow to define and generate a computational mesh for all the analysis.

To perform both geometry and mesh it was necessary to do a brief revision of these concepts according ANSYS Fluent 17.1[1]

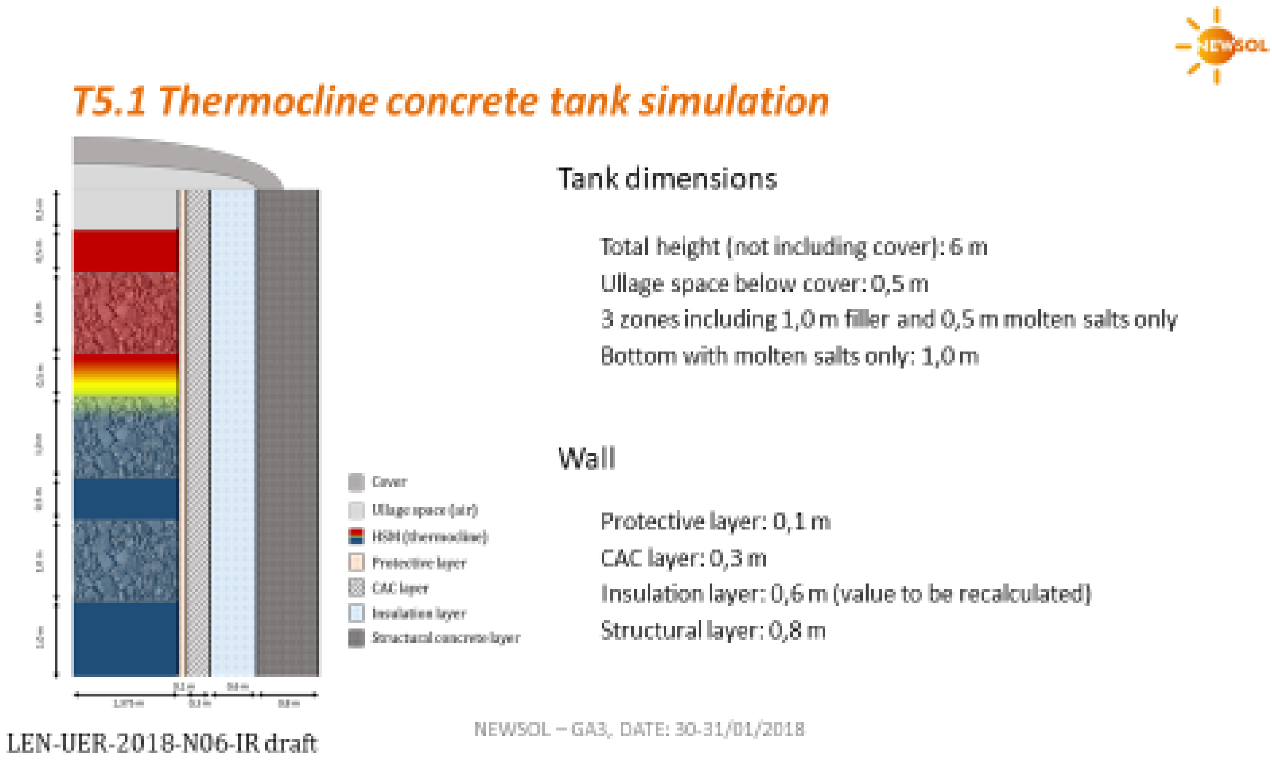

Figure 1- Proposed Thermocline concrete tank simulation: Draft Version

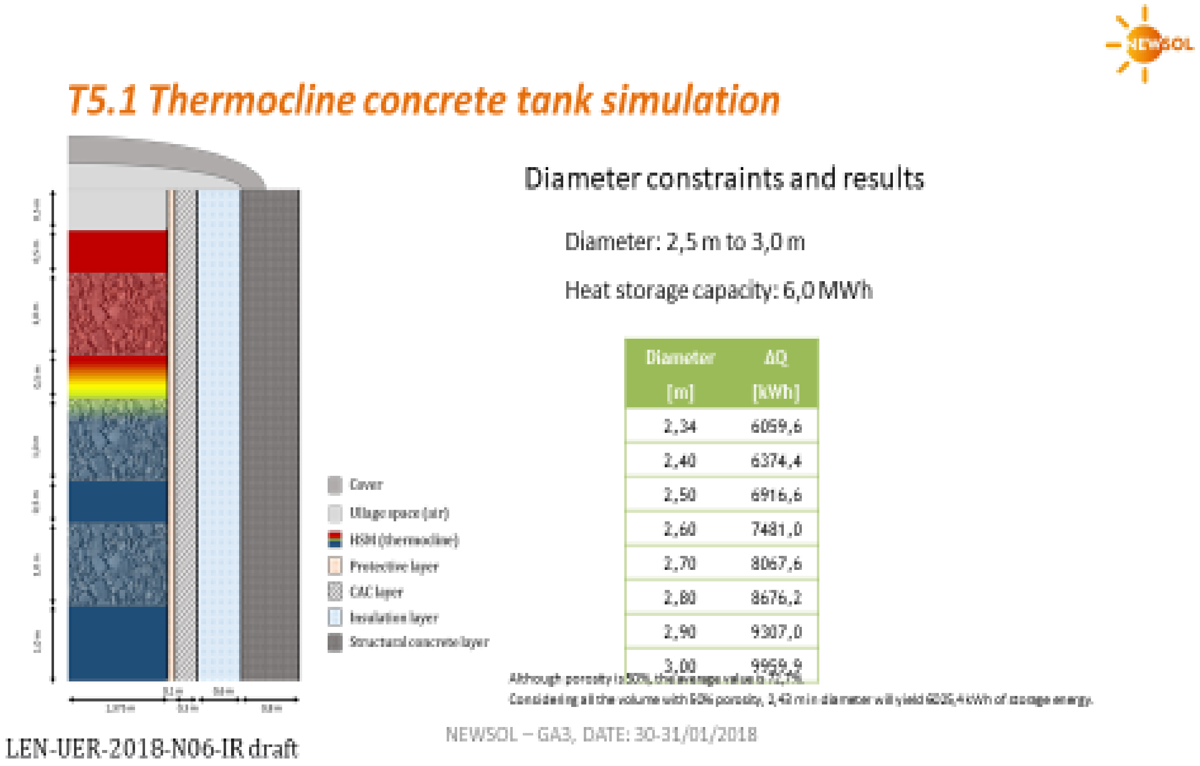

Figure 2- Draft Version: Parameters

1.1 Definition and State of Art

Thermal Energy Storage (TES) is essentially for solar energy applications when the demand is not in phase with the supply of Solar Energy.

There are several applications of TES but in the area of solar collectors the most important is a solar concentric parabolic collector that uses two tanks structure. This solution proved to be expensive and solutions using only one tank with a mixture of molten salt and a filler bed with a less expensive porous medium are a good alternative. To obtain these two-layer stratification (hot fluid on top and cold fluid in the bottom) a gradient (called thermocline) region between the two layers must be maintained thin and with an intense temperature gradient.

The thickness of a TES stratified tanks are able to operate effectively by exploiting the density difference of a single fluid at different temperatures. In this way the overall storage system tank turns to be economically attractive. Another low cost effect in a single tank is the used filler material which is not expensive and acts as the primary thermal storage medium (Price, 2003) [2] (Kearney *et al*, (2001)).[3]

A first concept of thermocline degradation considering a saturated porous media with a fluid was made by Gross (Gross, 1982) [4]. Thermocline degradation in saturated porous media have been studied in first hands by Najem (Al-Najem et al.1993, 1993a) [5] and also by Atkins (Atkins *et al*., 2008) [6]. Several investigators have made studies concerning this topic, proposing the model to model the tank, boundary conditions, porosities and another parameters

The effect of buoyancy forces to help to maintain stable the stratification between hot and cold molten salts in the same tank (Sanderson and Cunningam,1995) [7], (Mawire and McPherson,2009) [8], (Singh, *et al* 2010) [9].

1.2 Proposed Problem

This work presents the design and the draft of the mesh of the 2D model of a real tank that will be constructed at Alentejo in the aims of an international project. This report is only a first draft of that model concerning Design (Geometry) and Mesh based in a draft furnished for the tank. Results here obtained will be changed as another hypothesis will be considered for the tank. Nevertheless, the main structure will be the same. Conduction, convection and turbulence have already been study between boundaries, on the wall, and in the overall packed bed. Several molten salts have been tested as well as prototypes.

The tank that is here proposed is different from the tanks that have been studied in same particularities, and it is our hope to be possible to make a comparison between numerical results and data results that will be taken from the tank through a set of probes that are in the tank.

2. Geometry

The Fluid Flow (Fluent) analysis system allows to perform a Computational Fluid Dynamic complete analysis containing cells that allow to create geometry; generate a mesh; specify settings in Fluent, run the Fluent solver and visualize the results in CFD-Post (a post processor program).

For geometry creation the application of ANSYS accessed is DesignModeler.

This application is composed for a sequence of steps that may be called as Modeling Fundamentals.

2.1 Sketching

Sketching: after deciding if the problem to be design is in 2D or 3D and introducing the respective referential the Draw Menu is chosen. However before tracing lines or circles it is important to define the most appropriate grid. This is done considering the dimensions provided in Figure 1.

Sketching is the basis of the all geometry. It must be always generated in order to be recognized. If a draw is done over a sketch, even correct, if the previous was not well generated the draw is not recognized.

This first step is very important in the geometry. Only after performing the sketch the other menus should be applied.

2.2 Modeling

This part consists in pick all the draws, and choose an appropriate figure that will connect them. The type of this figure is different from 2D to 3D designs. The way that it is chosen must be done according what we want for meshing the design. There are a large variety of hypothesis to choose so this topic must be considered with a lot of care. Using the Draw Menu it is possible to make draws with lines, with curves with squares, circles in a referential. Dimensions may be corrected in the Menu Dimension. To 3D is the same but with surfaces. Sometimes it is difficult to fix the adequate dimension so a grid must be defined according the problem in hands.

The presented Figure 1 was rather complex once it is constituted by nine layers (parts), each one with fluid or molten storage material, that must be divided by a boundary that let the HTF heat transfer fluid enter and get down along the tank providing the respective mixture of materials and fluids. Nevertheless, this boundary must be clearly defined and when modeling is performed it must become nine clear parts and not only a surface.

At first this part was difficult and some previous experiences were done using only two layers instead of nine. See Figure 3. Here two layers appear.

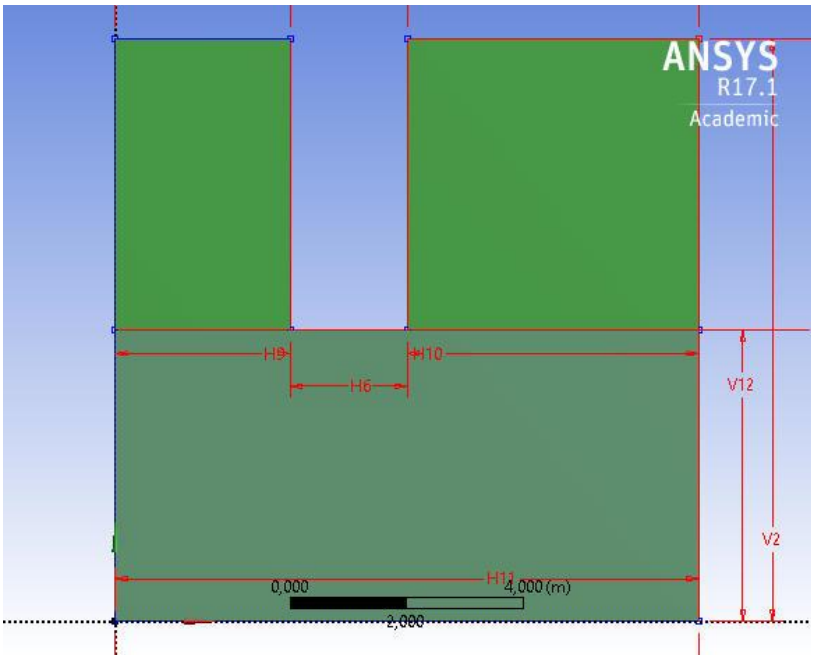

Figure 3. Two layers

The first draft of the tank simulation pretends that both inlet and outlet of the fluid were done by tubes at different heights of the tank. There will be more or less four tubes that can be manipulated (close and/or open) to fill the tank with more fluid. According this the design of the tank will be similar to Figure 4.

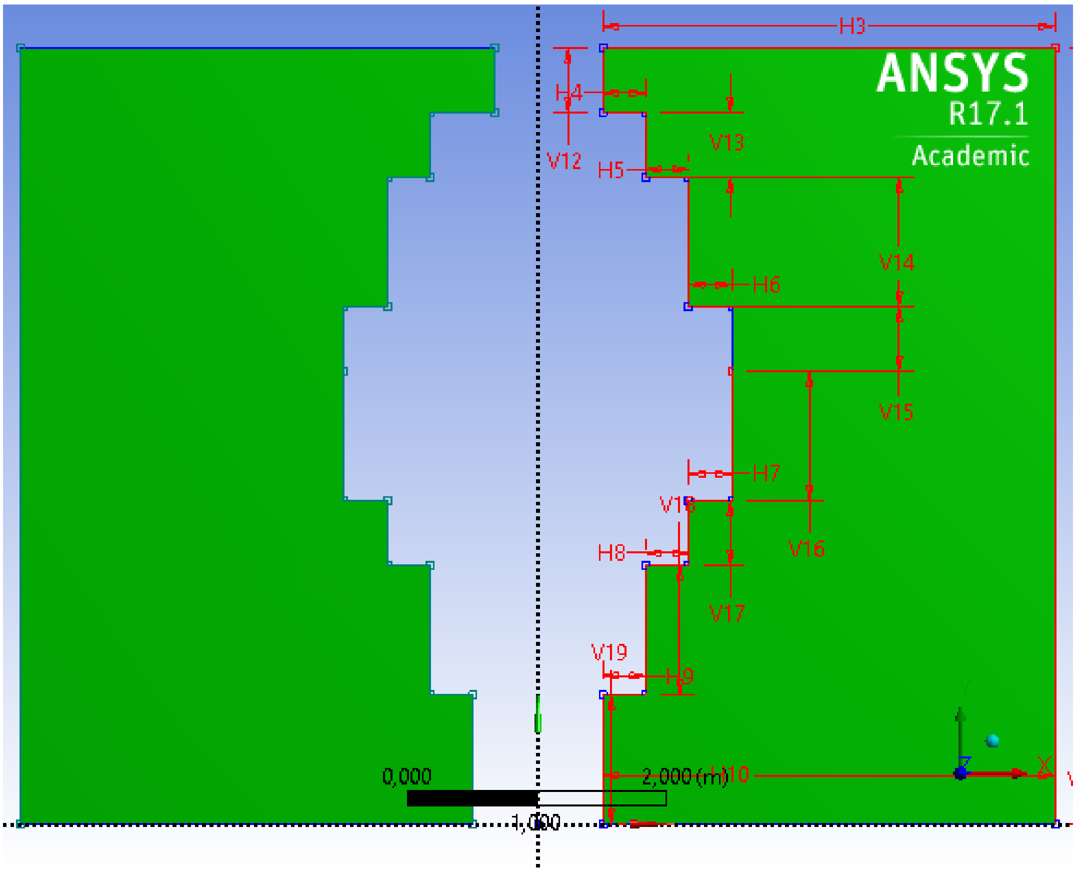

Figure 4 – Injection of HTF at different heights

After the draft of Figure 4 the idea of injection of HTF at different heights was removed and the proposed design was a quadrilateral divided in nine layers. As before each one of this layer has a different material or a fluid. If the material is not yet defined the better option is to consider all as a fluid. In set up material will be properly defined. To connect them the command sketch were applied to each one and a two selection boundary edge were introduced at the top of a layer and superposed to the bottom boundary of the next layer.

The layers must be considered as freezing.

2.3 Save

The geometry must be save in an independent file with a specific name. In this way it can be used later without doing all the process.

The all File is saved as Saved File.

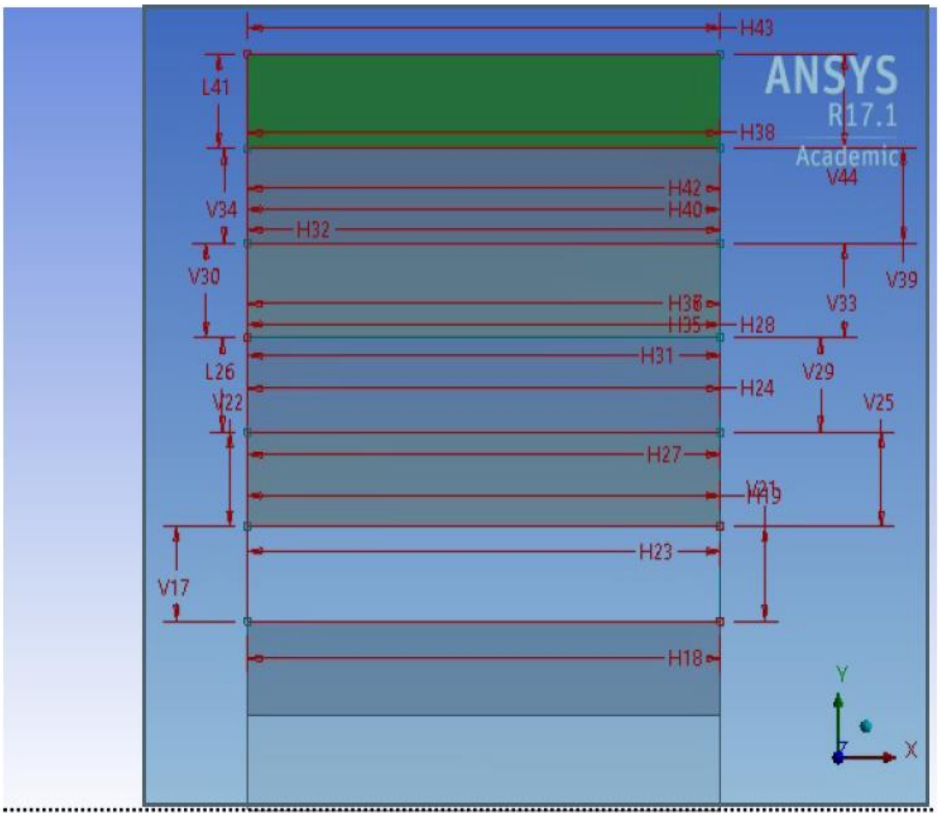

Figure 5 - Sketches not well defined

Figure 5 shows the situation where sketches were not well defined. As a consequence, Surfaces could not be defined. On contrary in Figure 6 the draft of the tank as the sketch well defined.

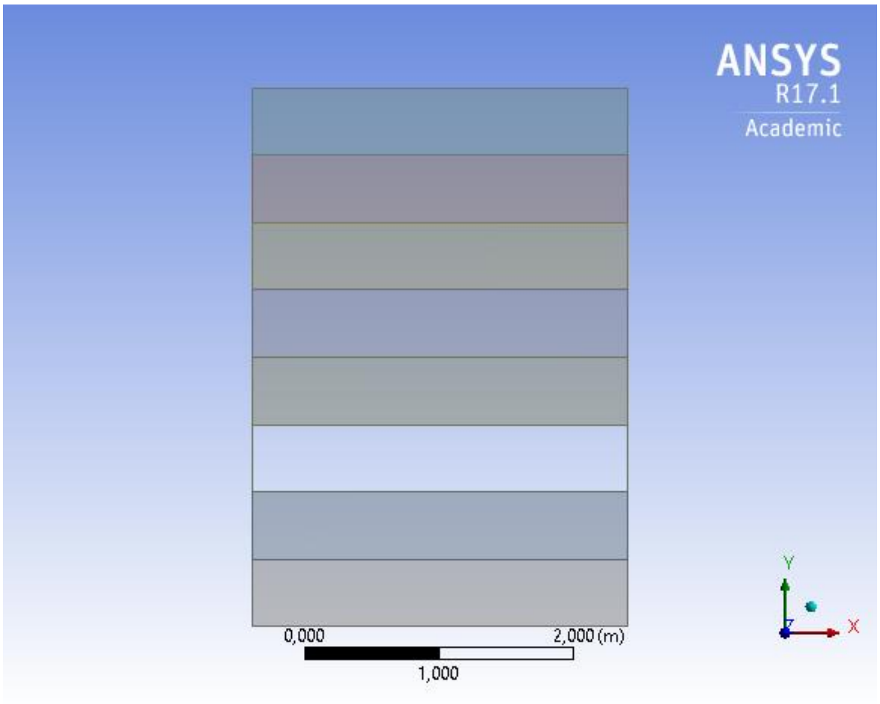

Figure 6 –Layers well defined

3 Mesh

Mesh: objects which are good quality, well-connected. Conformal surface mesh representations of the geometry. Mesh objects may contain multiple volumes with shared faces (in 3D), and they may be created using the Sew, Improve operations, or Build Topology operations.

Having the design properly done in Design Model it is necessary to give names to all the boundaries of the design. In this case these boundaries are edges.

After creating the Geometry of the tank and assigning to all the boundaries the respective Named Selections (the name of the boundaries already assigned in geometry and that mesh will recognize) that must be generated one by one, we need to generate a computational mesh throughout the 2D figure. The Meshing application was used to create a mesh for the CFD analysis.

In the present case the boundaries proposed by Figure 1 are Wall, Symmetry, Inlet, and Outlet.

After opening the application, the geometry previously save is automatically loaded and it is possible to generate a mesh with the several options that are set by default.

Meshing will assume the same boundaries and Create Named Selection could be done if there is anyone missing. We can also change boundary names according to those that will be run in the Set Up. So more appropriated names were given.

Other features were setting as: expanding the inflation nodes near the wall to better understand heat conduction.

When mesh is not tight enough near a part of a layer to solve the problem it is possible to refine the mesh Fluent has an adapt feature that automatically adds grid points when needed for a better resolution There several options: through curvature, velocity gradients or Solve Run Calculation with more iterations. In this case a structured mesh is made with respect to the geometry. Uniform transfer of data between each cells while charging and discharging the tank is the main idea behind the structured mesh.

Sizing function is applied at the wall, symmetry and the interfaces between the parts of the structure dividing the geometry into various equal parts to form a properly structured mesh.

An orthogonal mesh was generated. The statistics of the generation gave indication about mesh quality.

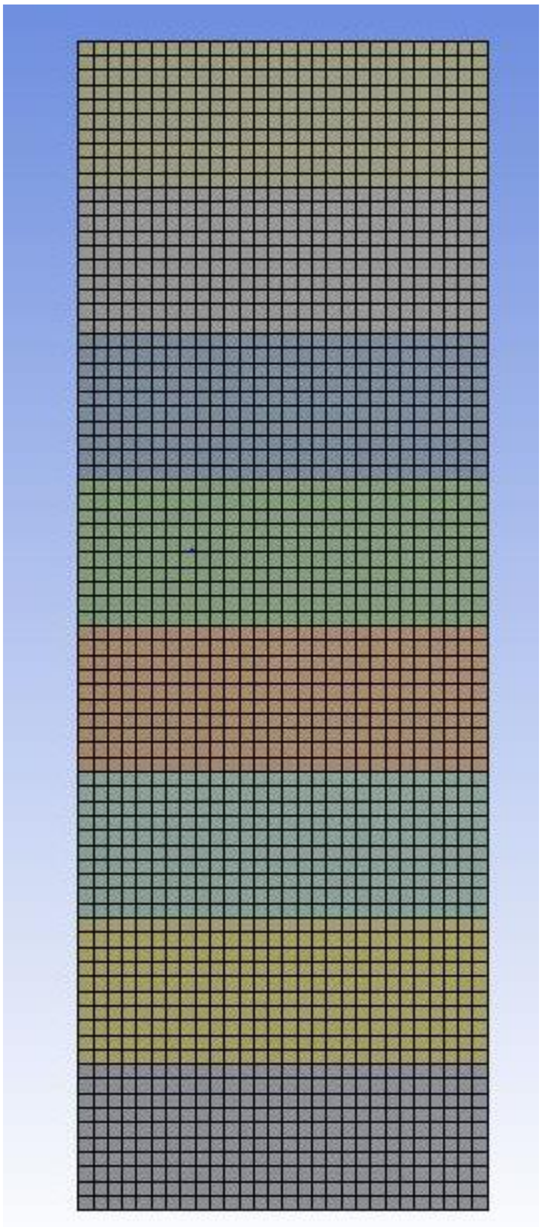

Fig. 7 Mesh

4 Set Up

After making an analysis to the mesh in order to determine which will be the more suitable for the case in study, the next problem is to prepare the set up.

Here materials will be assigned to the project, model, boundaries, with its properties of temperature, momentum and others if it is the case. Models must be considered taking in account the kind of media and properties expected.

In this case we deal with a porous material called Slag and a molten salt, Yara Most. Their properties will be given in a separated document when compiled. Initial velocity is available and inlet temperature is 500º C, outlet 200º C. At that time the study of the steady state of the problem will be made to see if everything is correct. That will be subject of a report from NewSol. The expected movements will be conduction, convection and turbulence. Equations must be of momentum, mass and energy.

The model will give account to the momentum, mass energy (for solid and liquid parts.) It will have 4 equations.

The Initialization method chosen was the standard initialization and the solution is computed from inlet zone. The solution method will be a coupled with a Pressure Velocity scheme. Pressure, momentum turbulent kinetic energy and time must be discretized by a second order upwind scheme. Turbulence dissipation rate must be discretized by a first order upwind scheme. Iterations at each time step will end when the dimensionless residual for all equations dropped below 10^{-5} (accuracy convergence).

4.1 Porous media

The porous media is a field where both liquid and small rocks with a porosiy are mixed together. Properties of this media in the context of a CFD resolution are the following;

- The porous media model in CFD ANSYS [1] assumes that the porosity is isotropic and it can vary with space and time;
- The superficial velocity is based on the volumetric volume rate to ensure the continuity of the flow velocity vectors across the porous media interface;
- The porous media momentum resistance and heat source terms are calculated separately on each phase;
- Are modeled by the addition of a momentum source term to the standard fluid flow equations. Tis source term is composed of two parts, a viscous loss term (Darcy, the first term on the right –hand –side RHS) and an inertial loss term (the second term on the RHS);

- | *v* | is the magnitude of the velocity. this momentum contributes to the pressure gradient in the porous cell creating a pressure drop that is proportional to the fluid velocity (or velocity squared) in the cell.

4.2 Insight on Turbulence Methods

To model turbulence, it is necessary to concern with an unsteady, irregular motion in which transported quantities (mass, momentum, scalar properties) fluctuate in time and space. It is also necessary to pay attention to fluid properties and velocity random variations. Energy transfer is performed from larger eddies to smaller eddies and in smallest eddies turbulent energy is converted to internal energy through viscous dissipation. Transition models used to predict boundary layer development and calculate transition onset; see [10]

4.3 k-ω Wall Turbulence Model

In the class of k-ω Wall Turbulence Models there are Standard, (BSL)-Baseline, and (SST)-Shear Transfer models. All of them have similar transport equations for k and ω. Turbulence damping is available only in with k-ω models. The k-ω of turbulent models have gained popularity mainly because they can be integrated to the wall without using wall functions and because they are accurate and robust for a wide range of boundary layer flows with pressure gradient.
k-ω of turbulent models are divided in: Standard k-ω (SKW) model and (SSTKW) model (Mentor)

4.4 Shear Stress Transport (SST) k-ω Model

This model includes the refinements of the BSL k-ω model and accounts for the transport of the turbulence shear stress in the definition of the turbulent viscosities.

The SST k- ω model uses a blending function to gradually transition from the SKW near the wall to a high Re number version of the k-ε model in the outer portion of the boundary layer and contains a modified turbulent viscosity formulation to account for the transport effects of the principal turbulent shear stress.

As in the Spalart-Allmaras model the concept of wall turbulent viscosity has been adopted. This model will be adopted to model turbulence in the present case.

Conclusion

As this report only concerns Geometry and Mesh the Set Up was only established but any run of the problem was performed although material properties and boundaries were known.

A deep study concerning the more appropriate mesh to be used must be analyzed before, and only after that steady state must be run to know the behavior of heat change and velocity inside the tank and at the boundaries.

References

[1] - ANSYS® Academic Research, Release 17.1, In: Chapter 6.2.3.7

[6] –Atkins M.J., Morrison A.S. and Walmsley M.R., 2008. Stratified Tanks for Heat Storage – Interface Control and Re-establishment. CHEMECA Conference, New Castle, Australia, September

[5] - Al-Najem N, Al-Marafie A.M. and Ezuddin K.Y 1993. Analytical and experimental investigations of thermal stratification in storage tanks. Int. J. Energy Res. Vol. 97, pp. 77.

[4] - Gross R. I. 1982. An experimental study of single medium thermocline thermal-energy storage. ASME Paper 82 HT-53.

[10] - M. Giestas Lima e P. Azevedo (2017), “Available CFD models assessment”, LNEG's internal report LEN-UER- 2017-N04-IR, December, 2017.

[8] - Mawire A., McPherson M., van der Heatkamp RRJ., Mlatho SJP.2010 Simulated performance of storage materials for pebble bed thermal energy storage (TES) systems. Appl. Energy. 2009. 86 .pp.1246-52

[3] - Kearney and Associates, 2001. Engineering evaluation of molten salt HTF in a parabolic through solar field. NREL Contract No NAA-1-30441-04.

[7] - Sanderson, T.M., Cunningham, G. T., 1995. Performance and efficient design of packed bed thermal storage systems. Part I. Appl. Energy. 50,119-132

[9] - Singh H., Saini, R. P., Saini, J. S. 2010. A review on packed bed solar energy solar systems. Renew. Sustain. Energy. Rev. 14(13). Pp. 1059-1069.

[2] - Price H., 2003. A parabolic trough solar power plant simulation model.