Tutor/s

Dr. Ricard Torres Castillo
Department of Chemical Engineering

Dra. Alexandra E. Bonet Ruiz
Department of Chemical Engineering

Treball Final de Grau

Heat transfer simulation in a submerged body using ANSYS® Fluent.

Alba Monné Pompido

June 2019

Cualquier tecnología suficientemente avanzada es indistinguible de la magia.

Arthur Charles Clarke

En primer lugar, quiero agradecer a mi pareja, familia y amigos por toda su confianza durante estos cuatro años, en los cuales me han animado y ayudado a seguir hacia adelante.

A continuación, quiero agradecer a mis tutores, Ricard Torres y Alexandra Bonet, por toda su dedicación, paciencia y tiempo dedicado durante estos meses.

Finalmente, quiero agradecer a todo el profesorado por todo lo que me han enseñado.

CONTENTS

SUMMARY

This project consists in studying the transmission of heat produced in a cylinder through the ANSYS® simulation program in academic version.

First, a brief explanation is presented about the operation of ANSYS® program and the steps followed to carry out the simulation. It is also verified that the program, in said version, is sufficiently capable of obtaining the desired results, carrying out a study on the meshing of the cylinder. This study shows that an error higher than 1.00% is obtained when using a mesh equal to or less than 418 nodes.

Then, the dependency that shows the temperature as a function of time and dimensions is analysed, therefore obtaining the temperature profiles, the distribution and the time necessary to reach the steady state, this being 207 minutes.

Later, the values obtained in the simulation are compared with the empirical equations and the laboratory practice performed in *experimentation in chemical engineering I*. The simulated model shows a good fit with the empirical model but a greater difference with the laboratory practice. This difference may be due to the fact that when the solid submerged, the water could have entered inside the cylinder causing an erroneous reading in the temperature, as shown by the values obtained during the experiment.

Finally, the different simulation has been carried out to obtain the temperature range in which the laboratory practice could be performed, this being between 323.15 and 338.15 K.

Keywords: ANSYS, conduction, cylinder.

RESUMEN

Este proyecto consiste en estudiar la transmisión de calor que se produce en un cilindro mediante el programa de simulación ANSYS® en versión académica.

Primero se presenta una pequeña explicación sobre el funcionamiento del programa ANSYS® y los pasos a seguir para realizar dicha simulación. También se verifica que el programa, en dicha versión, sea suficientemente capaz de obtener los resultados deseados realizándose un estudio sobre el mallado del cilindro. Este estudio muestra que se obtiene un error superior al 1,00% al usarse una malla igual o inferior a 418 nodos.

Después, se analiza la dependencia que muestra la temperatura en función del tiempo y las dimensiones obteniéndose así los perfiles de temperatura, la distribución y el tiempo necesario para alcanzar el estado estacionario siendo este de 270 minutos.

También se comparan los valores obtenidos en la simulación con las ecuaciones empíricas y la práctica de laboratorio realizada en experimentación en ingeniería química I. El modelo simulado muestra un buen ajuste con el modelo empírico pero una mayor diferencia en cuanto a la práctica de laboratorio. Esta diferencia puede ser debida al hecho de que al sumergir el sólido en el baño pueda haber entrado agua dentro del cilindro provocando una lectura errónea en la temperatura, tal y como muestran los valores obtenidos durante el experimento.

Finalmente se han realizado diferentes simulaciones para obtener el rango de temperaturas en el cual se podría realizar la práctica de laboratorio siendo este entre 323,15 y 338,15 K.

Palabras clave: ANSYS, conducción, cilindro

1. INTRODUCTION

Heat transfer is the transmission of energy from one region to another by means of a difference in temperature between them, this transfer can occur through three different mechanisms: conduction, radiation and convection.

The transmission of heat is very important in the chemical industry for the design of reactors, heat exchangers, insulators and in many of the unit operations such as evaporation or condensation.

Heat conduction is the mechanism that takes place in solid bodies and in resting fluids, when the velocity vector is zero, this occurs when there is a temperature gradient in a solid medium and the heat flows in the opposite direction to that gradient. The energy is transmitted by the movement of atoms, molecules, ions and electrons, which constitute the substance, without apparent movement.

One of the most important properties in the conduction is the thermal conductivity, it indicates the amount of heat that flows per unit of time through a unit area when the temperature gradient is unitary, that is describes the ability to conduct heat through them. Thermal conductivity depends on the material and its physical state. The bodies can be:

- Isotropic: they don't present privileged directions in the conduction of heat and conductivity doesn't vary with the position.
- Anisotropic: privileged directions appear in the conduction of heat and conductivity is function on the position.

Throughout the project it has been considered that the material is isotropic.

1.1. PROJECT DESCRIPTION

The project consists of analysing the heat transmission by conduction of a solid submerged in a liquid. It is considered that the liquid is perfectly stirred at a constant temperature and that it does not present heat generation, to analyse only the conduction and to consider that the convection coefficient is infinite.

The properties and conditions of the project have been selected based on the experiment carried out in the subject of *experimentation in chemical engineering I*. The analytical resolution is detailed in the section 1.2

1.2. ANALITICAL RESOLUTION

The Fourier law[2] defines the thermal conductivity of the substance, depending upon the material which it is made.

$$\vec{q} = -k \, \nabla T \tag{1}$$

Where q (W/m^2) is the heat flux density, k (W/(m·K)) the thermal conductivity and ∇T the temperature gradient.

The microscopic balance of energy[2], in the non-stationary state of a solid, with constant density and heat capacity, is:

$$\rho \, C_p \, \frac{\partial T}{\partial t} = -\vec{\nabla} \vec{q} + Q''' \tag{2}$$

Where Q''' is the generation of heat, that is considered null in this study. Applying the Fourier law, equation 1 and considering the thermal conductivity constant this equation will be:

$$\frac{\partial T}{\partial t} = \alpha \, \nabla^2 T \tag{3}$$

With α (thermal diffusivity):

$$\alpha = \frac{k}{\rho \, C_p} \tag{4}$$

The resolution of equation 3, with the appropriate limiting conditions, gives information about the variation of temperature with position and time for a simple geometry with a unidirectional conduction of heat.

The analytical result of the dimensionless temperature at a point of a finite cylinder is made by applying the Newman rule, that is, the intersection of an infinite cylinder with an infinite sheet:

$$Y_{FC}\ (r,x) = Y_C\ (r) \cdot\ Y_S\ (x) \tag{5}$$

Where Y_{FC} is the dimensionless temperature of a finite cylinder, Y_c for an infinite cylinder and Y_s for an infinite sheet.

The analytical resolution for an infinite cylinder and an infinite sheet is presented in the following sections.

1.2.1. Analytical resolution for an infinite cylinder

A cylinder of infinite height and radius r_0 that is initially at a temperature T_0 submerged in a constant temperature fluid T_e. The limit conditions, in this case, for the resolution in cylindrical coordinates are:

Table 1. Limit condition for a cylinder of infinite height

Time	**Radius**	**Limit condition**
$t = 0$	$0 < r < r_0$	$T = T_0$
$t > 0$	$r = 0$	$\partial T/\partial r = 0$
$t > 0$	$r = r_0$	$h\ (T_e - T_w) = -k\ (\partial T/\partial r)$

For the resolution, the dimensionless numbers will be:

$$Y_C = \frac{T_e - T}{T_e - T_0} \tag{6}$$

$$n = \frac{r}{r_0} \tag{7}$$

$$F_o = \frac{\alpha \cdot t}{r_0^2} \tag{8}$$

Where:

Y_C = dimensionless temperature for an infinite cylinder

n = dimensionless distance

F_0 = Fourier number (dimensionless time)

The equation that provides the variation of the dimensionless temperature profile[3] is:

$$Y_C = 2\sum_{i=1}^{\infty}\frac{1}{\beta_i}\cdot\frac{J_1(\beta_i)}{J_0^2(\beta_i)+J_1^2(\beta_i)}\cdot J_0(\beta_i\, n)\cdot exp(-\beta_i^2\, F_o) \qquad (9)$$

Where β_i is the infinite solution of:

$$\frac{\beta_i\cdot J_1(\beta_i)}{J_0(\beta_i)} = Bi = \frac{h\cdot r_0}{k} \qquad (10)$$

Where J_0 and J_1 are the Bessel functions of the first kind and the values of β_i are the solutions of $J_0(x) = 0$ when the convection coefficient (h) is infinite, and therefore the Biot (Bi) is also.

1.2.2. Analytical resolution for an infinite sheet

An infinite sheet of thickness e that is initially at a temperature T_0 located in a medium at constant temperature, T_e. The limit conditions, in this case, for the resolution in cartesian coordinates and with heat transmission in the x-axis is:

Table 2. Limit condition for an infinite sheet

Time	Radius	Limit condition
t = 0	0 < x < e	$T = T_0$
t > 0	r = 0	$\partial T/\partial x = 0$
t > 0	r = e	$h\,(T_e-T_w) = -k\,(\partial T/\partial x)$

For the resolution, the dimensionless numbers will be:

$$Y_S = \frac{T_e-T}{T_e-T_0} \qquad (11)$$

$$n = \frac{x}{e} \qquad (12)$$

$$F_o = \frac{\alpha\cdot t}{e^2} \qquad (13)$$

The equation that provides the variation of the dimensionless temperature profile[3] when Biot's number is infinite, when the temperature of the medium is the same as in the solid wall:

$$Y_S = 4\sum_{i=1}^{\infty} -\frac{(-1)^i}{(2i-1)\pi}\cdot cos\left(\frac{2i-1}{2}\,\pi\cdot n\right)\cdot exp\left[-\left(\frac{2i-1}{2}\,\pi\right)^2\cdot F_o\right] \qquad (14)$$

2. OBJECTIVES

The aim of the project is to perform a CFD (Computational Fluid Dynamics) analysis through the ANSYS® program of a solid body, with a cylindrical geometry, submerged in a liquid. This study has the following objectives:

- To learn of the ANSYS® program (in academic version).
- To study the transmission of heat by conduction in solids.
- To determine the number of minimum nodes to be able to take for granted the results obtained in the simulation.
- To compare the results obtained in the simulation, in the geometric centre of the solid, with the experimental values.
- To compare the results obtained in different points of the solid with the calculated values with the analytical equations for a finite cylinder.

3. MATERIALS AND METHODS

3.1. LABORATORY PRACTICE

The laboratory practice carried out in *experimentation in chemical engineering I* consists of determining the thermal conductivity of a solid. The experimental device, Figure 1, consists of:

- Stirred thermostatic bath.
- Two temperature probes.
- Different nylon bodies with a hole in their geometric centre to introduce the probe.

Figure 1. Experimental device

The experimental procedure is:

a) Fill the bath and connect the heater.
b) Once the temperature of the bath is stable, one of the probes is introduced into the solid, recording said temperature, and immersed in the bath.
c) The temperature of the solid is recorded every 2 minutes until obtaining a Y less than 0.3 (equation 6).

The properties of the solid, to be studied in this project, and the conditions of these are shown in Table 3 and 4.

Table 3. Properties of solid[4]

Geometry	Cylindrical
Material	Nylon
ρ, kg/m^3	1,165
C_p, J/(kg·K)	1,287 + 7.267·T[ºC]
k, W/(m·K)	0.30

The heat capacity will be considered constant for the laboratory practice and the analytical calculation. This has been calculated as follows:

$$\left(C_p\right)_m = \frac{\int_{T_0}^{T_W} C_p \cdot dT}{\int_{T_0}^{T_W} dT} = 1{,}582 \quad (15)$$

Table 4. Conditions of solid

r_o, m	0.065
L, m	0.130
T_0, K	296.15
$T_e = T_w$, K	331.25

3.2. ANSYS® FLUENT

In this project, the ANSYS® FLUENT software will be used in an academic version to find a solution to the proposed problem. This is an engineering support program, through simulation software, that predicts the operation and reaction of an element. This program principally consists of three main tools:

- **Pre-processing:** establishes the model by constructing the geometry (sketch) and creating the mesh over it.
- **Simulation:** solves a set of differential equations, using the finite element method (FEM), that is, converts the derivatives in increments. It is necessary to define the materials used in the geometry, establish the resolution model and define the limit conditions
- **Post-processing:** visualizes of the results obtained in the simulation from different options such as plots and contours.

The accuracy of the results obtained through the simulation depends on the appropriate construction and choice of the models. In the following subsections the steps followed during the simulation process performed are explained.

3.2.1. Pre-processing

Before the creation of geometry, the module to be used for the entire resolution must be decided. In this case the Fluid Flow (Fluent) has been chosen, which is used for processes with fluids, flows, heat transfers or chemical reactions.

3.2.1.1. Geometry

The first thing to do is the construction of the geometry, in this case the ANSYS® program gives multiple options for its creations, in two or three dimensions. This can be created from the tools of the program: DesignModeler or SpaceClaim. The program also allows import the geometry for CAD extensions.

The geometry was created using the DesingModeler tool in three dimensions. The centre of the solid has been established as the origin of coordinates. To ensure the result and to be able to work with more node numbers, symmetry has been created in the XY plane (radial plane). In the Figure 2 can see the geometry created.

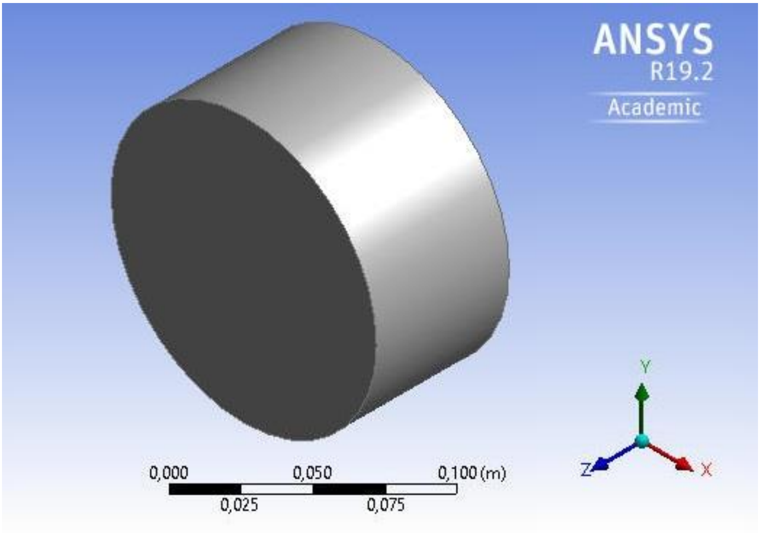


Figure 2. Geometry of solid

3.2.1.2. Mesh

Next, it is necessary to create the meshing of the geometry, this factor is very important in any simulation since it will influence the accuracy of the results obtained and the necessary time for its resolution. It is important to create a mesh that is suitable for each problem giving more priority to certain elements or areas. Subsequently, the influence of said factor on the results will be studied.

A meshing has been made, with the maximum number of elements allowed in the academic version, on the cylinder giving more priority to the edges, these have more nodes or elements than the rest of the geometry, since in these points is where there is greater variation of temperature over time, this has been done through inflation tool.

Firstly, sections have been created in the geometry to simplify the work later in setup tool. The sections that have been created are the cylinder body defined as a solid and the symmetric plane XY (radial plane). Next, the parameters that have been modified within the meshing tool are shown:

- In sizing → Use adaptive sizing: Yes
- In quality → Smoothing: High
- In definition on body sizing → Element size: 2.5E-3 m
- In inflation → Use automatic inflation: Program controlled
- In inflation → Maximum layers: 10

The static of mesh is:

- Nodes → 112,265
- Elements → 499,339

The final mesh created is:

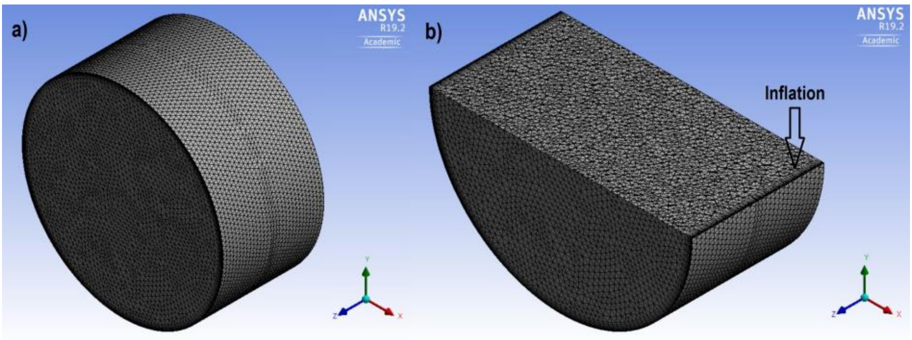


Figure 3. a) Meshing with symmetry; b) Interior view of the mesh with symmetry

3.2.2. Simulation

In this step, the conditions for the resolution of the model created and previously meshed will be established. It is very important to correctly establish the model, properties of the system and resolution method to obtain a convergent and reliable solution.

3.2.2.1. Setup

The simulation is done from the setup tool, that consists of different sections where different system properties are specified. The choice of the model has been made, establishing the properties and boundary conditions of the system. These are:

- Model: energy → ON
- Materials: the properties of density, heat capacity and thermal conduction of the nylon, detailed in Table 3, have been introduced.
- Cell zone conditions: the solid material has been selected.
- Boundary condition 1: condition of symmetry in the XY plane (radial plane) has been established.
- Boundary condition 2: material and temperature properties of the wall of the solid have been established, being 331.25 K and without heat generation rate.

3.2.2.2. Solution

In solution, the chosen method is:

- Pressure-velocity coupling → Simple
- Gradient → Green-Gauss Node Based
- Pressure → Second Order
- Momentum → Second Order Upwind
- Energy → Second Order Upwind
- Transient Formulation → First Order Implicit

Once the resolution model is established, the solution is found through iterations. The residuals of each iteration have a resolution criterion in which the program continues to iterate until it reaches at least that residual value, if enough iterations are established by time step. These values are shown in Table 5.

Table 5. Residual absolute criteria

Residual	Continuity	x-velocity	y-velocity	z-velocity	Energy
Absolute Criteria	10^{-3}	10^{-3}	10^{-3}	10^{-3}	10^{-6}

Finally, it is necessary to indicate the initial conditions in the initialization section. In this case, the initial condition is the temperature of the solid being 296.15 K. To finish, in run calculation the resolution has been made with the following properties of time and iterations:

- Time step size → 120 s
- Number of time steps → 150
- Max iterations for time step → 20

4. RESULTS

One of the objectives of this study is to verify the reliability of the results obtained through the ANSYS® FLUENT program. This will be done from the comparison of these values with the empirical equations and the laboratory practice carried out. The temperature obtained by simulation as a function of time and position will also be displayed.

For the analysis of the results, certain points have been created to compare the results. The position of said points and the name with which it will refer during the presentation of the results are:

Table 6. Points created in the cylinder.

Name	Radius [m]	Length [m]
Point 1	0.000	0.000
Point 2	0.000	0.022
Point 3	0.000	0.044
Point 4	0.022	0.000
Point 5	0.022	0.022
Point 6	0.044	0.000
Point 7	0.044	0.044

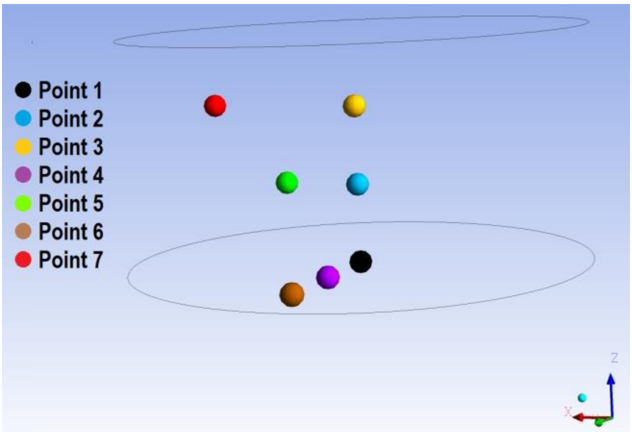


Figure 4. Distribution of points.

4.1. ANSYS® RESULTS

The results obtained from the simulation, described in section 3, will be shown in form of contours and graphs.

In this subsection, the influence of time and position on the temperature will be analysed. Different simulation will be carried out, varying the mesh to determine how it influences the results.

4.1.1. Mesh influence

In this section we will observe the influence that meshing has on the resolution of the system through simulation with ANSYS® program. As mentioned above, meshing is an important part in the resolution since it indicates how many points the equations will be solved by iteration.

The different simulations have been carried out varying only the number of nodes and elements. Only point 1 has been analysed because it is the one that will suffer the greatest deviation due to the meshing previously established, in which the edges of the cylinder are given more priority, therefore the most central points will be those that contain more meshing big.

Table 7 shows the values of nodes and elements, simulated in each case together with their temperature value, in the geometric centre of the solid, and the deviation they present with the maximum number of nodes allowed. The temperature value obtained with the maximum number of nodes in this point is 309.82 K.

From 418 nodes it can be established that the mesh is not enough to consider the values obtained in the simulation correct since they present a deviation higher than 1.00%. This error has occurred using a 0.17% of the number of allowed elements.

Table 7. Influence of mesh

Nodes	Elements	T [K]	Deviation T
6,053.00	18,406.00	309.76	0.02%
3,080.00	9,151.00	310.16	0.11%
1,565.00	3,833.00	310.61	0.25%
798.00	1,766.00	310.99	0.38%
631.00	1,422.00	311.23	0.46%
555.00	1,221.00	311.39	0.51%
454.00	925.00	311.57	0.57%
418.00	859.00	314.26	1.43%
391.00	797.00	314.45	1.49%

The graphic representation of results is:

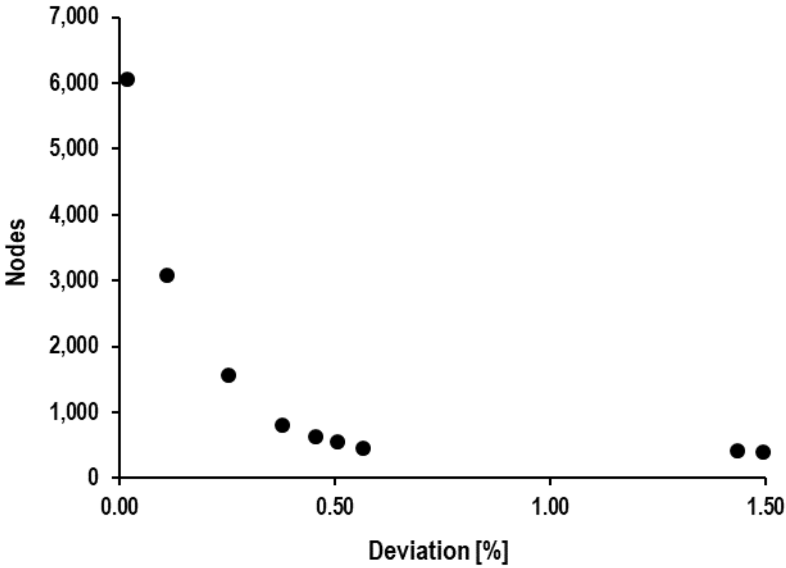


Figure 5. Influence of mesh

It is observed that a large decrease in meshing practically does not affect the results. For this case it is observed that the number of nodes, that the academic version of ANSYS® allows, is more than enough having a high margin in the number of nodes to work. In this way, through the program, in this version, heat transfer problems could be simulated by driving much more complex geometries.

4.1.2. Evolution over time

Figure 6 shows the profile of the temperature at the geometric centre of the solid. This evolution shows three well differentiated sections:

- First section, constituted by the first twenty minutes, the temperature remains practically constant. This is due to the low heat flow that reaches this point.
- Second section: the evolution of temperature shows a linear trend with time up to 92 minutes approximately.
- Third section: the evolution of temperature is getting slower as it approaches the temperature of the wall of solid, in this case, 331.25 K.

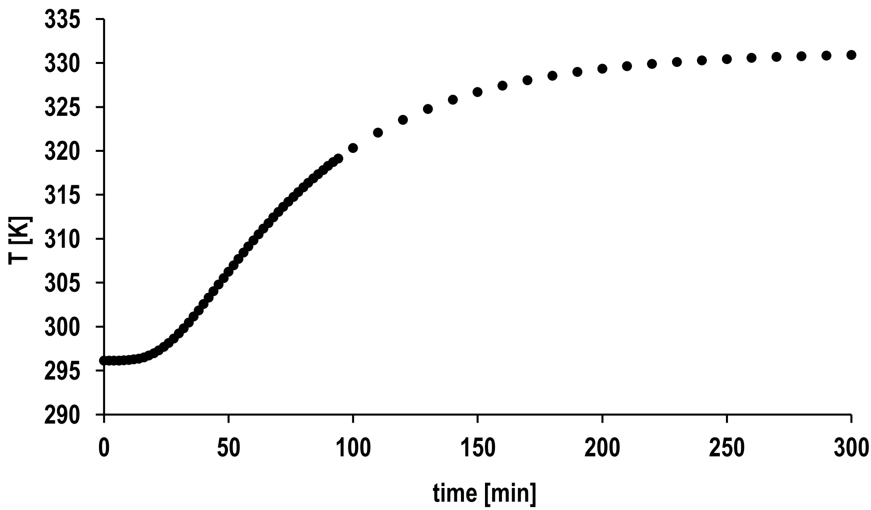


Figure 6. Temperature evolution in point 1

The time required to reach the steady state is infinite, as can be seen in the graphical representation where temperature practically does not vary in the last points. It has been analysed in this specific point since it will be the last to reach this temperature.

Since the necessary time is infinite and considering that the steady state is reached when the temperature at the coldest point is al 1.00% of the wall temperature, the necessary time will be 207 minutes.

Figure 7 shows the evolution of temperature at different pints to see if the trend following by the central point (point 1) is fulfilled throughout the cylinder. To facilitate the visualization of Figure 7, points 4 and 6 are not shown. In Appendix 1 the figure with all points studied is presented.

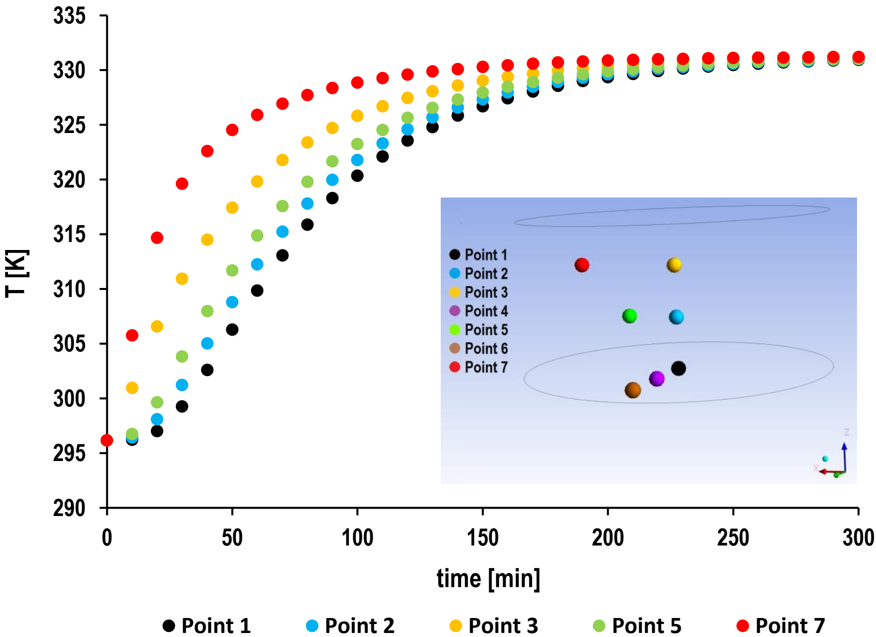


Figure 7. Temperature evolution in different points

As the points move closer to the cylinder wall, the first section is reduced until disappearing in the closest points such as points 3 and 7.

As for the second and third sections, they all show this part, but the second section is reduced, arriving before the third. It is also observed how the slope of the linear stretch increases with the position, so the speed at which the temperature increases in the points closest to the wall is greater.

4.1.3. Evolution with position

It has been analysed how the temperature varies with the position in different points. In Figure 8 the variation of the XY plane (radial plane) as a function of the radius is shown.

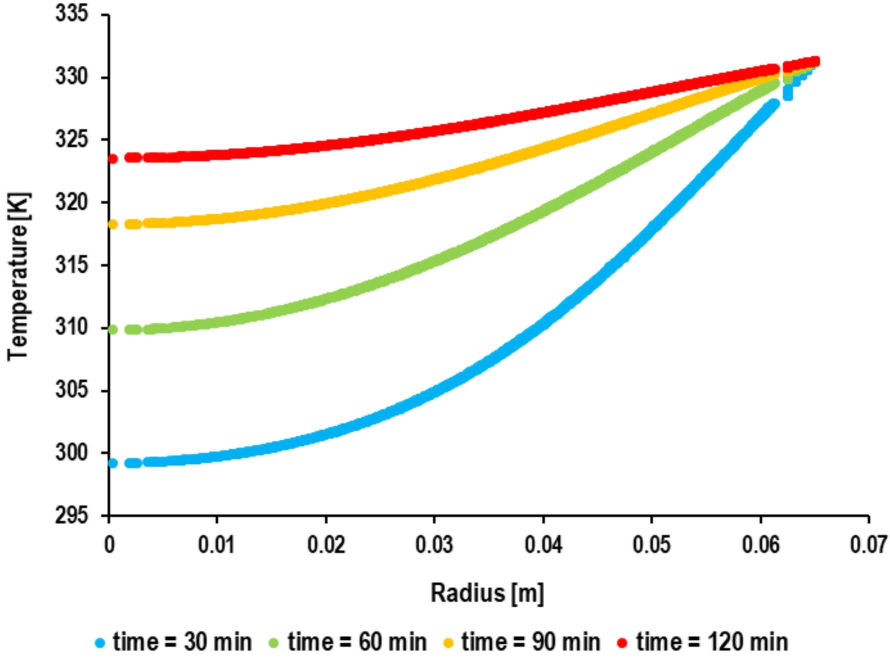


Figure 8. Temperature evolution over radius

From the graphical representation it is observed that the temperature has not a determined tendency based on the radius, this depends on the time. At small times the temperature presents a large difference and distribution as a function of the position, becoming increasingly uniform, until reaching the “steady state” temperature, this being that of the wall at zero time.

This representation in the XY plane (radial plane) is shown in Figure 9 with the contour tool, where the symmetry has been created, which the temperature is only a function of the radius.

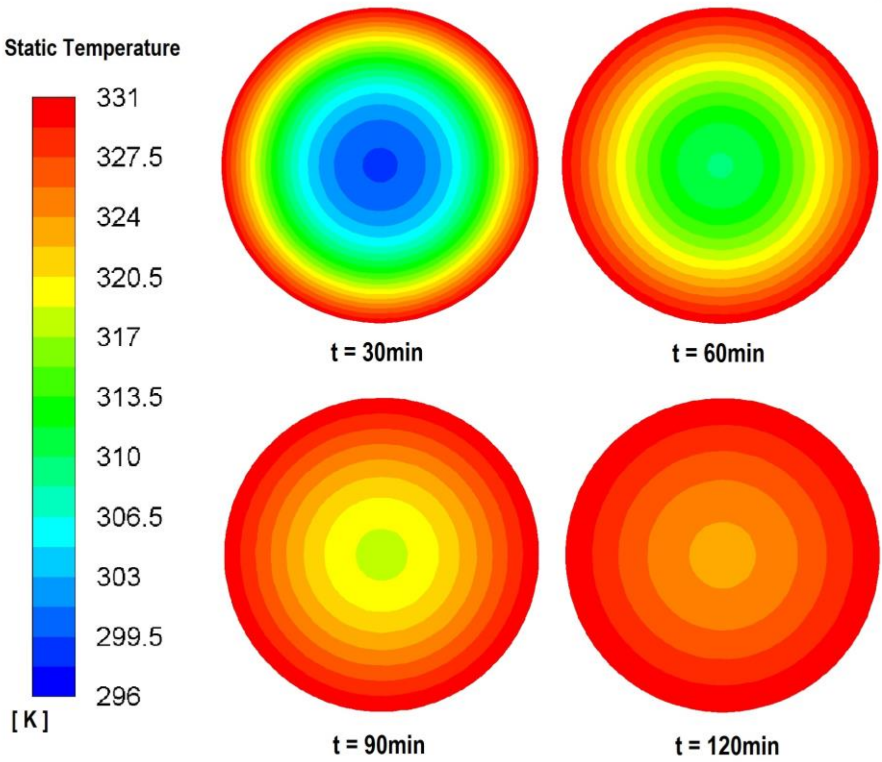


Figure 9. Contours in radial plane at different times

To see the influence of the radius with length of the cylinder, Figure 10 shows the contour in the YZ plane (axial plane). The Y-axis refers to the radius of the geometry and the Z-axis to the length of it.

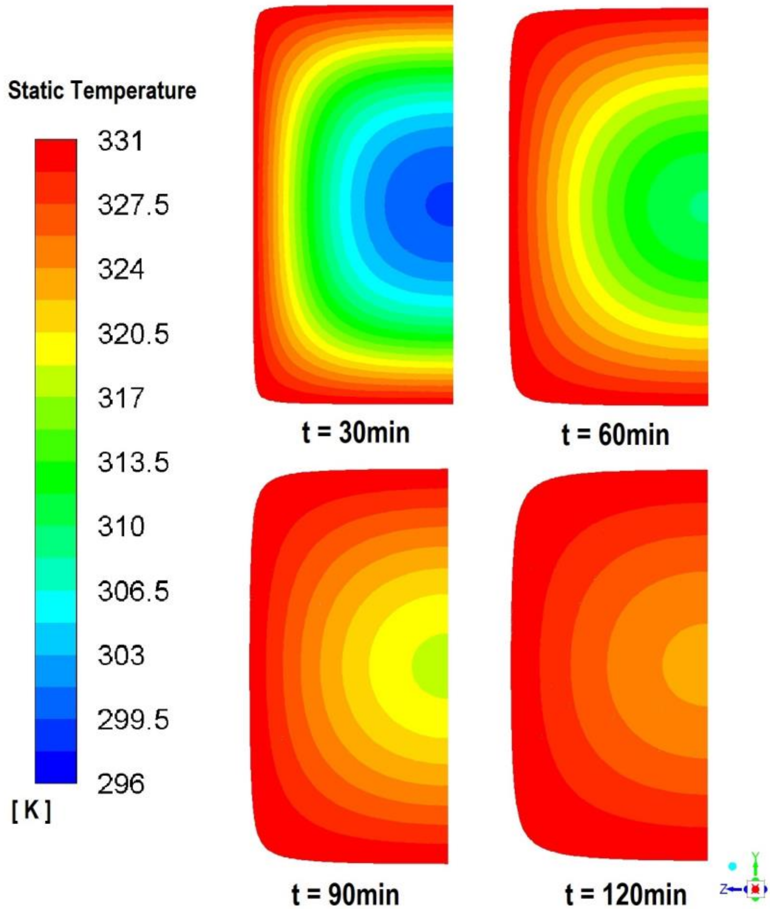


Figure 10. Contours in axial plane at different times

Analysing the distribution of temperature as a function of radius and length it can be observed, that all distributions are very similar. The influence of both dimensions, radius and length, are also very similar. This is because the diameter is equal to the length, if it increases this difference between both dimensions the distribution will vary considerably as shown in Figure 11. In this figure the contours created are a cylinder with the previous diameter 0.013 m and a length of 0.026 m.

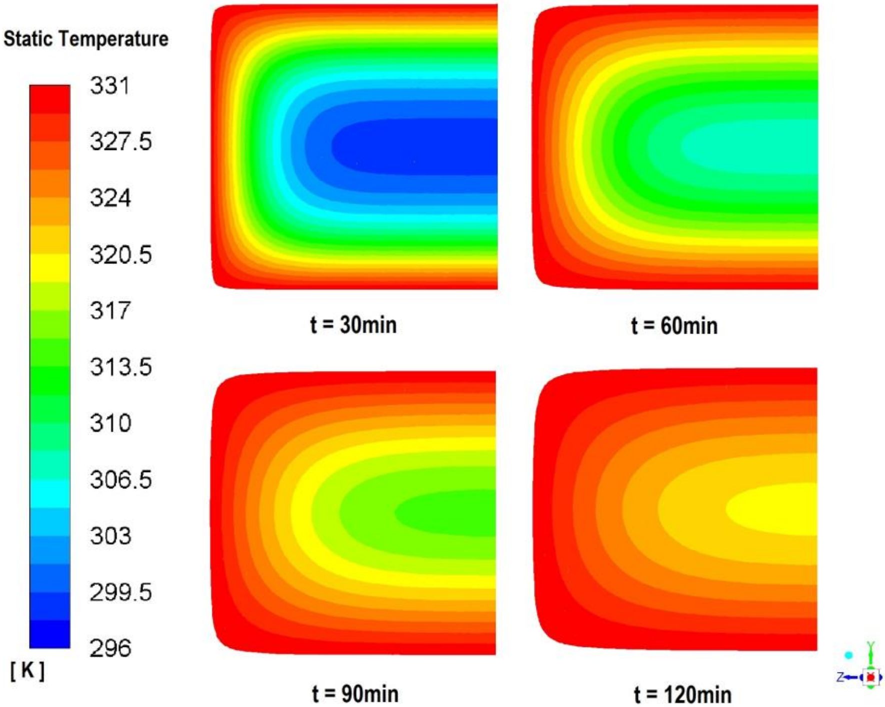


Figure 11. Contours in axial plane at different times with L=2D

In this figure it is observed that the contours, in comparison with the previous one, flattens out having a less radial distribution. As the length increases in relation to the diameter, the contour changes being the heat flow transferred by the body of the cylinder becoming more important than the bases of this.

4.2. COMPARISON WITH ANSYS®

In this subsection it will compare the results obtained in ANSYS® with the values found by the empirical equations and the experimental values realized in the practice of thermal conduction of solids in the subject of *experimentation in chemical engineering I*. It is also going to analyse the possibility of perform the experimental practice at other temperatures.

4.2.1. Comparison with analytical model

The analytical calculation has been made from the equations detailed in section 1.2, considering an average value of the heat capacity of the solid shown in equation 15. The comparison was made in seven points of the cylinder after 30, 60 and 90 minutes. The results with their difference and deviation are:

Table 8. Analytical and simulated temperature values.

Point	Time [min]	Analytical T [K]	ANSYS® T [K]	Difference [K]	Deviation
Point 1	30	298.44	299.22	0.78	0.26%
	60	309.43	309.82	0.39	0.13%
	90	318.41	318.28	0.13	0.04%
Point 2	30	301.42	302.12	0.70	0.23%
	60	312.64	312.89	0.26	0.08%
	90	320.51	320.30	0.21	0.06%
Point 3	30	313.14	312.75	0.39	0.13%
	60	321.26	320.92	0.33	0.10%
	90	325.61	325.27	0.34	0.10%
Point 4	30	300.48	301.11	0.63	0.21%
	60	312.02	312.12	0.10	0.03%
	90	320.21	319.88	0.33	0.10%
Point 5	30	310.80	311.43	0.63	0.20%
	60	319.93	320.13	0.20	0.06%
	90	324.95	324.89	0.06	0.02%
Point 6	30	303.28	304.09	0.81	0.27%
	60	314.84	315.14	0.30	0.10%
	90	321.94	321.84	0.10	0.03%
Point 7	30	319.96	319.62	0.34	0.11%
	60	326.07	325.91	0.16	0.05%
	90	328.46	328.34	0.12	0.04%

From the results obtained, it is observed that the analytical and simulation models give very similar results, obtaining a maximum difference of 0.81 K and 0.27% of deviation. In general, a tendency is detected in which values at a shorter time show more deviation between both models. This can be caused by the fact that in the analytical calculation a constant value has been considered for the heat capacity instead of using its dependence on the simulation with ANSYS®.

4.2.2. Comparison with experimental practice

The results obtained, during the practice of thermal conduction of solids made in the subject of *experimentation in chemical engineering I*, will be compared with the values obtained in the simulation in the geometrical centre of the solid. To make the comparison, the same temperature conditions have been established, both initial as well as water temperature, and the properties of the material established in Table 3. In Appendix 2, the table with the experimental values obtained and their values found in the simulation. The representation of both data set during the time that the practice was carried out, that is 94 minutes, is shown in Figure 12.

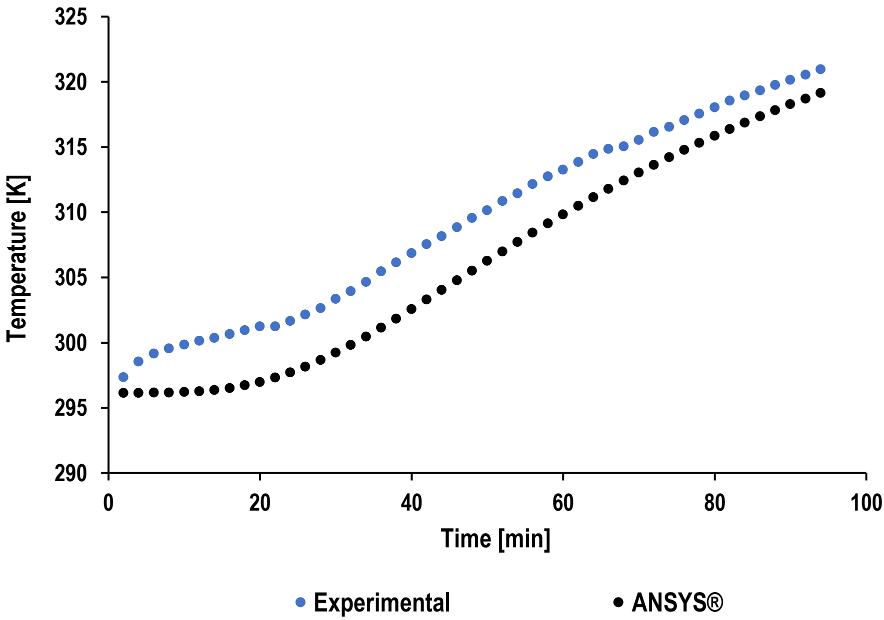


Figure 12. Temperature evolution in point 1 (centre)

A great difference is observed between both procedures, especially in the initial values, with a maximum deviation of 1.44% and a difference of 4.3 K. This deviation is caused because the solid used in the laboratory has a hole in the geometric centre to introduce the temperature probe and when the cylinder is submerged, the water penetrated inside the body giving erroneous temperature values at initial moment of the experiment. When performing the practice, it must be avoided that the water penetrates inside the solid.

Despite the deviation of the temperature, both curves show the same tendency with an initial stretch where the temperature increases slowly followed by a linear stretch. Figure 13 shows the linear section to compare the slope of both lines, obtaining a slope of 0.325 and 0.344, for the experimental and simulated cases, respectively. The deviation between both models is 5.52%.

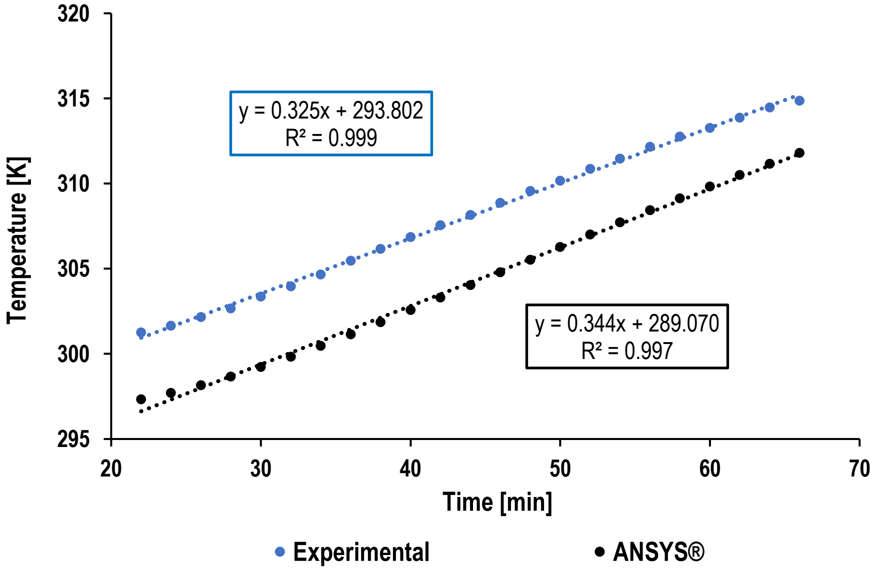


Figure 13. Linear stretch of temperature evolution in point 1 (centre)

The case of performing the laboratory practice at a different temperature has been studied to see if this could be carried out during the time available in the practices of *experimentation in chemical engineering I*. Taking into account that each day the practice was carried out with two different geometries, the maximum time available is 120 minutes and it is considered finished when 15.00% is missing to reach the steady state.

Different cases have been simulated in which the temperature limit condition in the wall of the solid has been modified and 296.15 K has been established as initial temperature. The results obtained are:

Table 9. Necessary values of time

Temperature [K]	**Time [min]**
323.15	104
328.15	110
333.15	114
338.15	118
343.15	122

As observed in the Table 9, the time increases with temperature in the solid wall. With laboratory time available, the experiments could be carried out up to a water bath temperature of 338.15 K. The temperature cannot be much lower than 323.15 K to obtain enough results for the subsequent calculation.

5. CONCLUSIONS

Based on the results obtained and their subsequent comparison, the following conclusions can be drawn.

- From the study of the influence of meshing, it is concluded that the ANSYS® FLUENT program, in an academic version, has enough capacity to simulate conduction heat transfer problems in simple geometries.
- The temperature profile evolves over time. This shows three different sections and their influence varies with the position. As it approaches the cylinder wall, the first section decreases, the speed in the second section increases and the third section, of constant speed, becomes more important.
- The time necessary to reach the steady state, in the case where the bath temperature is 331.25 K, is 207 minutes.
- The flow of the heat transferred to the cylinder is on the same order for the transfer produced by the body and the bases, in the case of studied, where the diameter is equal to length. When increasing one of the dimensions, the temperature distribution varies, producing more heat flow from the cylinder body (for L>>D) or from the bases of this (for L<<D).
- From the comparison with the empirical model, it is concluded that model is adjusted and provides accurate results obtaining a maximum difference of 0.81 K.
- From the comparison with the laboratory practice, it is concluded that the simulated model with ANSYS® shows the same tendency but does not adjust at all, obtaining a maximum difference of 4.3 K. This may be due to the fact that during the realization of the experiment enter water within the cylinder and alter the values obtained.
- The laboratory practices carried out in experimentation in chemical engineering It can be carried out within the range of temperature between 323.15 and 338.15 K.

6. FUTURE WORKS

The project can be extended or modified in different ways, such as:

- Solve the system in case that convection coefficient, h, is not infinite by performing the 2D simulation to facilitate the introduction of limit conditions and their resolution or if it is possible to solve it in 3D.
- Solve the system with different geometries made in the practice in *experimentation in chemical engineering I* and make a small script to be able to implement this resolution in the laboratory practices.

REFERENCES AND NOTES

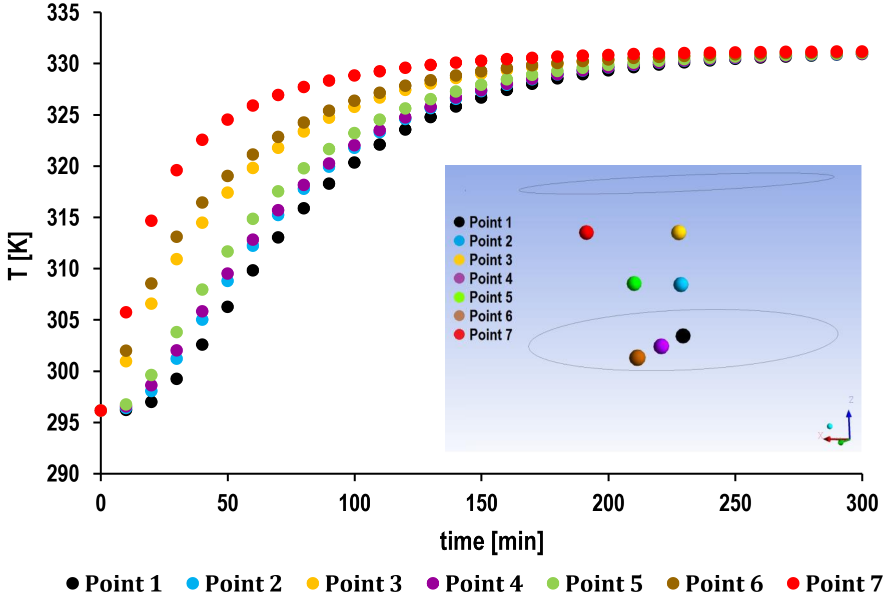
1. Sigalés, B.; *Transferencia de calor técnica*. Barcelona: Reverté, **2003** – Vol.1
2. Kreith, F; Bohn, M. S; *Principios de transferencia de calor*. 6ª ed; Madrid: Thomson, **2002**
3. Esplugas, S; Chamarro, E; *Fundamentos de transmisión de calor*. Barcelona: Ub, **2005**
4. Perry, R. H; *Perry's Chemical Engineers' Handbook*. 7th ed; McGraw-Hill, **1999**
5. Carslaw, H. S; Jaegar, J. C; *Conduction of Heat in Solids*. 2nd ed; Oxford, **1959**
6. ANSYS, Inc.; *ANSYS Meshing User's Guide*. Release 18.0, **2017**
7. ANSYS, Inc.; *ANSYS Fluent User's Guide*. Release 15.0, **2013**
8. ANSYS *Fluent Tutorial Guide*. Release 16.0, **2015**

ACRONYMS

Bi	Biot number, dimensionless
C_p	Heat capacity, J/(kg·K)
e	Thickness, m
F_o	Fourier number, dimensionless
h	Convection coefficient, W/(m^2·K)
J_i	Bessel function, dimensionless
k	Thermal conductivity, W/(m·K)
n	Dimensionless distance, dimensionless
$\vec{q}$	Heat flux density, W/m^2
Q'''	Generation of heat, J/(m^3·s)
r	Radius, m
T	Temperature, K
t	Time, s
x	Distance, m
Y	Dimensionless temperature, dimensionless
α	Thermal diffusivity, m^2/s
ρ	Density, kg/m^3

APPENDICES

APPENDIX 1: TEMPERATURE EVOLUTION IN ALL POINTS



APPENDIX 2: VALUES EXPERIMENTAL AND ANSYS®

Experimental		ANSYS	Difference	Deviation	Experimental		ANSYS	Difference	Deviation
t [min]	T [K]	T [K]			t [min]	T [K]	T [K]		
0	296.2	296.2	0.0	0.00%	48	309.6	305.5	4.0	1.32%
2	297.4	296.2	1.2	0.41%	50	310.2	306.3	3.9	1.27%
4	298.6	296.2	2.4	0.81%	52	310.9	307.0	3.9	1.26%
6	299.2	296.2	3.0	1.01%	54	311.5	307.7	3.7	1.21%
8	299.6	296.2	3.4	1.14%	56	312.2	308.4	3.7	1.21%
10	299.9	296.2	3.6	1.23%	58	312.8	309.1	3.6	1.17%
12	300.2	296.3	3.9	1.31%	60	313.3	309.8	3.4	1.11%
14	300.4	296.4	4.0	1.34%	62	313.9	310.5	3.4	1.08%
16	300.7	296.5	4.1	1.39%	64	314.5	311.2	3.3	1.06%
18	301.0	296.7	4.2	1.42%	66	314.9	311.8	3.1	0.98%
20	301.3	297.0	4.3	1.44%	68	315.1	312.4	2.6	0.84%
22	301.3	297.3	3.9	1.32%	70	315.6	313.0	2.5	0.80%
24	301.7	297.7	3.9	1.33%	72	316.2	313.6	2.5	0.80%
26	302.2	298.2	4.0	1.34%	74	316.6	314.2	2.3	0.74%
28	302.7	298.7	4.0	1.34%	76	317.1	314.8	2.3	0.72%
30	303.4	299.2	4.1	1.38%	78	317.6	315.3	2.2	0.71%
32	304.0	299.8	4.1	1.38%	80	318.1	315.9	2.2	0.69%
34	304.7	300.5	4.2	1.39%	82	318.6	316.4	2.2	0.69%
36	305.5	301.1	4.3	1.43%	84	319.0	316.9	2.1	0.66%
38	306.2	301.8	4.3	1.43%	86	319.4	317.4	2.0	0.63%
40	306.9	302.6	4.3	1.42%	88	319.8	317.8	1.9	0.61%
42	307.6	303.3	4.3	1.40%	90	320.2	318.3	1.9	0.59%
44	308.2	304.0	4.1	1.35%	92	320.6	318.7	1.8	0.58%
46	308.9	304.8	4.1	1.34%	94	321.0	319.1	1.8	0.57%