Numerical analysis of internal recirculation into a radiant tube without internal ignition

Revista Soluciones de Postgrado EIA, Número 10. p. 117- 132. Medellín, enero-junio de 2013

José Luis Suárez*, Andrés Amell Arrieta**, Francisco Javier Cadavid ***

* Ingeniero mecánico. M.Sc. en Ingeniería con enfásis combustión y energía térmica. Director del programa de Ingeniería Mecánica de la Escuela de Ingeniería de Antioquia. Correos electrónicos: joseluissuarez@udea.edu.co, ingmecanica@eia.edu.co

** Coordinador Grupo de Ciencia y Tecnología del Gas y Uso Racional de la Energía (GASURE), Universidad de Antioquia. Correo electrónico: anamell@udea.edu.co

*** Profesor Grupo de Ciencia y Tecnología del Gas y Uso Racional de la Energía (GASURE), Universidad de Antioquia. Correo electrónico: fcadavid@udea.edu.co

NUMERICAL ANALYSIS OF INTERNAL RECIRCULATION INTO A RADIANT TUBE WITHOUT INTERNAL IGNITION

José Luis Suárez, Andrés Amell Arrieta, Francisco Javier Cadavid

Abstract

This paper presents a numerical analysis using the CFD Fluent (6.3.26 version) program to identify the effects that can be generated when using radiant tubes with internal recirculation of combustion products, but with a pre-combustion chamber. The numerical results are validated with an experimental assembly based on the outlet deviation of gases temperature. These deviations were less than 5 % and are attributed principality to isolation deficiency in the re-radiant surface.

To independently find the recirculation effect in this tube, the same operational characteristic in the tube were maintained and an analysis using different windows recirculation widths as well as one without windows was done. Through this analysis, it was possible to observe the little importance of recirculation effects when combustion doesn't occur internally in the tube on the uniformity surface temperature on the external tube and the emitted energy, but it decrease the system pressure drop slightly although its magnitude order is very small.

Keywords: Radiant tube, Heat transfer, Internal recirculation, Radiation, CFD

ANÁLISIS NUMÉRICO DE RECIRCULACIÓN INTERNA EN UN TUBO RADIANTE SIN ENCENDIDO

Resumen

Este artículo presenta un análisis numérico sobre los efectos que se generar utilizando tubos radiantes con recirculación interna de los productos de combustión con una cámara de pre-combustión. Este análisis se desarrolló con ayuda de un software comercial de CFD, Fluent (versión 6.3.26). Los resultados numéricos se validan con un montaje experimental basado en la desviación de la temperatura de salida de los gases. Estas desviaciones equivalen a menos del 5% y se atribuyen principalmente a la deficiencia de aislamiento en la superficie re-radiante.

Para encontrar el efecto de recirculación en este tubo, se analizan las variaciones en los resultados considerando la misma característica de funcionamiento pero con diferentes anchos de ventana de recirculación, y particularmente una sin ventanas. A través de este estudio, fue posible observar la poca incidencia de los efectos de recirculación cuando la combustión no se desarrolla internamente en el tubo, particularmente en la temperatura sobre la superficie uniformemente en el tubo externo y la energía emitida, pero por otro lado se observa una leve disminución en la caída de presión del sistema.

Palabras clave: Tubo radiante, Transferencia de calor, Recirculación interna, Radiación, CFD

Numerical analysis of internal recirculation into a radiant tube without internal ignition

José Luis Suárez, Andrés Amell Arrieta, Francisco Javier Cadavid

Recibido 27 de septiembre de 2012. Aprobado 4 de marzo de 2013. En discusion hasta el 30 de junio de 2013
Revista Soluciones de Postgrado EIA, Número 10. p. 117-132. Medellín, enero-junio de 2013

1. Introduction

Infrared heating is a great alternative to solve problems such as uniform heating, inert atmosphere, pollutant emissions, and other problems involved in the industrial sector's competitiveness [1]. Industrial sectors involved include: drying, cured, coating, baking and dehydration, settings dyes, waste treatments and other processes that require heating control and are susceptible to this kind of energy. Within this kind of heating, the use of radiant tube is important because these systems can be coupled with other devices, such as recuperative and regenerative heat exchangers which allow increases in processes efficiencies, and offer "cleaner emission".

Some of these tubes have internal recirculation of flue gases to reduce the temperature peaks generated during the combustion, thus achieving reductions in emissions of NOx and uniform temperature profiles on the tube [1, 2]. However, to preserve the life of the tube, there must be a more direct regulation of the temperature thereof and its relationship to the broadcast to work at low temperature and / or characteristics of special emission, such as a given wavelength, this analysis proposes to evaluate the use of a pre-combustion chamber and observe the effect of keeping the recirculation window and its benefits to homogenize the temperature and promote the emission.

In present work, a numerical study of a commercial radiant tube of Silicon Carbide (SiC) has been done to evaluate the recirculation effect without internal combustion, and it can be validated through comparing it to experimental data from a similar tube using the thermal power's of 23,3 kW and 37,3 kW. Because the commercial tube does not allow access at internal flow, temperature measurements on the tube and in the inlet and outlet of the flue gas are recorded. In a similar way, the radiated heat is found with the simplified method of radiosity [3], while the convective heat is obtained by the difference between input and output power.

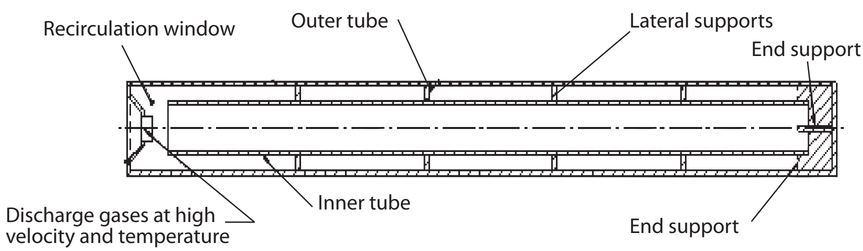
To simulate the actual behavior of the experimental setup, the numerical model contains in addition to the respective boundary conditions, a subroutine to emulate the heat exchange to the surroundings taking into account the heat radiated in an enclosure and external convective heat.

An axisymmetric model is used due to the reduction of the computational requirements and its good results [4], noting that in the evaluated radian tube there are no obstacles in the recirculation window that can generate overestimates of the found data simulation, and that the flue gas inlet is totally axial.

2. radiant tube and experimental setup

The evaluated radiant tube, with the geometry expressed in **Table 1**, is a single ended type with a recirculation window in which the gases (with a mass flow and temperature given) from the combustion chamber are discharged through the inner tube and continue flowing along the outer tube. This recirculation window is usually used to increase the internal mass flow and improve the temperature uniformity in the tube [4, 5]. The scheme of this arrangement is expressed in the **Figure 1.**

Figure 1. Radiant tube scheme



This configuration is localized into an experimental setup (Figure 2) which has a reflection system composed by a "semi-enclosure" completely insulated and only exposed in the place where the load would be, with a width similar to 500 mm. In a similar way, to emulate the ends of the tube that would be inside a furnace but do not interact with the load, these ends are insulated with ceramic blanket in the corresponding areas. Additionally this experimental setup has a perforated ring, which makes a radial scan possible, (positioned in each perforation and properly fastened) installed from the top where and axial scan can be done to carry out the simplified method of radiosity to find the emitted radiant heat.

Two thermal powers were analyzed by operating the tube with natural gas (97% CH4), 23,3 kW and 37,3 kW, with a 60 mm recirculation window, and a nozzle of the combustion chamber 40 mm in diameter. The equipment used for measuring and controlling the experimental procedure were: gas analyzer SICK MAIHAK S700 mark, radiometer SBG01/5 and indicator LI19 Hukseflux mark, thermocouples (corrected by radiation) k type and display CHY820U mark, gas flow meter (OMEGA fma5400) and air flow meter (SIERRA 620s), and the respective refrigeration system. The experimental procedure to find the radiant heat and appropriate selection of the radiometer and restrictor, is analyzed in other future work.

Table 1. Geometrical characteristics of the commercial radiant tube

Part	Outside diameter (mm)	Inside diameter (mm)	Lenght (mm)
Outer tube	114	104	1350
Inner tube	84	76	1150
End Spacer	-	-	100

Figure 2. Experimental setup

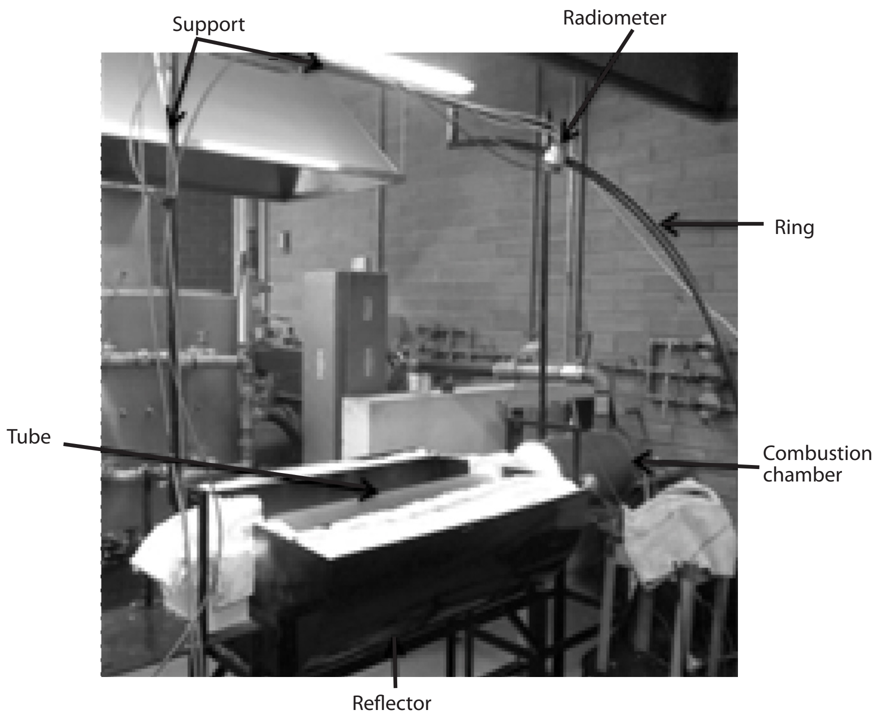


Table 2. Experimental data vs numerical results

	23,3 kW			37,3 kW		
	CFD	Experimental	Difference	CFD	Experimental	Difference
Recirculation %	-27,40	N/A	N/A	-21,30	N/A	N/A
Energy by radiation kW	4,77	4,45	7,2%	9,24	7,67	20,5%
Inlet temperatura	1306,0	1306,0	N/A	1222,0	1222,0	N/A
Energy by natural convection kW	1,80	1,22	47,5%	1,49	1,500	-0,7%
Outlet temperature (°C)	755,0	780,20	-3,2%	823,0	861,04	-4,4%
Maximum temperature on the tube °C	653,4	466,0	40,2%	744,4	569,0	30,8%
Minimum temperature on the tube °C	442,7	430,0	3,0%	591,4	529,0	11,8%
Difference °C	210,7	36,0	485,3%	153,0	40,0	282,5%
Standar deviation	45,2	20,4	121,6%	34,2	20,0	71,2%
Average temperature on the tube (°C)	485,6	453,0	7,2%	622,1	551,3	12,9%

In (**Table 2**) it is possible to observe that the numerical model proposed to evaluate the physical behavior in the tube is corresponds with the experimental results found in a physical model similar to the domain used.

This conclusion is based on the difference between the outlet temperature of gases because in two cases it was less than 5 %, corresponding to deviations in the total energy balance in the same order. On the other hand, the major deviation in each specific heat exchange is due to the assumptions made about the isolate surface on the tube and the external reflector surface, because when decreasing these isolate properties, the exchange in these zones increase and the temperature on the tube wall decreases, and hence the exchange by radiation and natural convection in this length also decrease. Due to this, it is possible to assume that this numerical model is reliable to predict the physic internal and external phenomenon in the tube.

3. Numerical model

With the purpose of evaluating the predictive ability of the numerical model, the experimental data listed in the Table 2 is compared with the results of the proposed model, while the numerical simulations are used to modify the recirculation window to find the effect in the recirculation, and its effect in the temperature profile on the tube and the radiant heat emission.

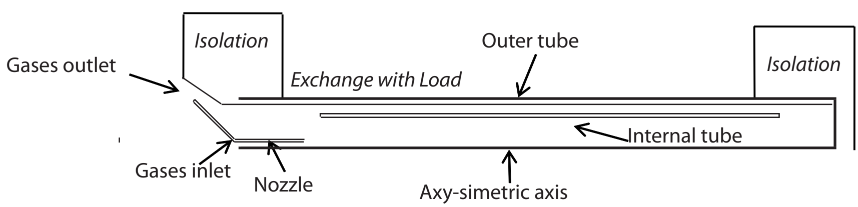
This numerical analysis is performed in the commercial CFD Fluent program.

3.1. Domain

As mentioned above, the proposed computational domain is an axisymmetric model (Figure 3) where the zones of the different boundary conditions, heat exchange and the solid part of the inner tube are represented.

The end spacer used to support this last tube, is not included in the domain, however in this same zone there is no exchange with the surroundings, so the evaluation while retaining the same space is not necessary.

Figure 3. Problem domain.



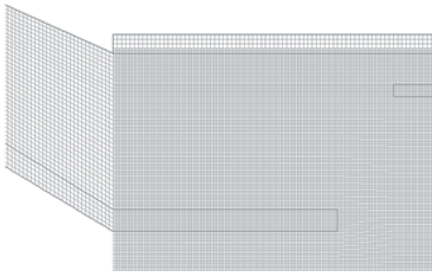
The mesh used for the domain (Figure 4) aims to refine the areas where there are major fluid and dynamic interactions.

Three levels of refinement were carried out where 49659, 171571 and 653498 nodes numbers were obtained, and where the quality characteristics values such as “equisize skew” and the “aspect ratio” were smaller than of 0,32 and 2 respectively.

In **Table 3** the results obtained from different refinement levels of the mesh are shown, in this is possible to observe the influence of the refinement level on recirculation prediction, which is the ratio between the gas flow discharged from the nozzle and the flow into the inner tube.

However, with the second level (171571 nodes) the computational cost is lower than in the last, and the change of the prediction are very few.

Figure 4. Domain meshing



that the emitted energy in this system be higher than the simulated experimental setup. For this, it was necessary to join the outlet gases flow to the inlet gases flow with a numerical subroutine with this external system.

Table 3. Results with the different numbers nodes

Nodes	Recirculation %	By radiation (kW)	Average temperature on the wall (K)	Average temperatura of outlet gases (K)
49659	41%	18,8	960	1082
171571	22%	18,6	958	1095
653498	25%	18,6	957	1098

3.2. Boundary conditions

As shown in **Figure 3**, the system has five boundary conditions type:

Adiabaticity: all parts when there is isolation.

Mass flow inlet: through the area of the nozzle when the gases are discharged, the mass flow inlet of the combustion products is considered, based on combustion characteristic (kind of fuel, power and equivalence ratio) and their temperature. This temperature was found from a model that included a combustion chamber with a heat recuperative system which was not included in the experimental setup, therefore it is expected

Axy-simetric: the problem domain was simplified to a 2D problem, which also has a symmetric axis.

Outlet gases: it was assumed a discharge at atmospheric pressure (100 kPa).

Exchange flux: the heat exchange with the load is on the wall of the outlet tube; this exchange can be estimated analyzing the radiation heat exchange in an enclosure (Figure 5), as expressed in the following equation which considers the different geometric variables that limit the heat exchange. This enclosure has a re-radiant isolated surface which enables all the energy emitted by the tube is directed through the load [6].

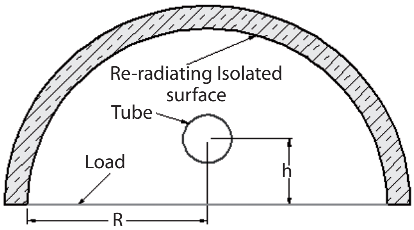


Figure 5. Enclosure model

$$q_T=-q_C=\frac{E_{bT}-E_{bL}}{\frac{(1-\varepsilon_T)}{\varepsilon_T A_T}+\frac{1}{A_T F_{TL}+[(A_T F_{Trr})^{-1}+(A_C F_{Crr})^{-1}]^{-1}}+\frac{(1-\varepsilon_T)}{\varepsilon_L A_L}} \quad (1)$$

$$F_{LT}=\frac{D}{2R}\,Tan^{-1}\left(\frac{R}{h}\right) \quad (2)$$

$$F_{LT}=\left(\frac{2RL}{\pi DL}\right)\frac{D}{2R}\,Tan^{-1}\left(\frac{R}{h}\right)=\frac{Tan^{-1}\left(\frac{R}{h}\right)}{\pi} \quad (3)$$

$$F_{Trr}=1-F_{TL} \quad (4)$$

$$F_{Lrr}=1-F_{LT} \quad (5)$$

Where F_{LT}, F_{TL}, F_{Trr} and F_{Lrr} are the view factors, from the load to tube, from the tube to load [6], from the tube to re-radiant surface and from the re-radiant surface to tube respectively. In this boundary also is included the exchange by natural convection, incorporating therein same subroutine the correlation proposed by Churchill and Chu [6] for the Nusselt number in a cylindrical surfaces.

3.3. Physical model

It is necessary to define the transport equation that can model the different physical problems in the tube without internal combustion.

3.4. Mass conservation (Continuity equation)

The physical principle of mass conservation applied to a control volume fixed in the space, including the different species that could arise, is expressed in conservative form [7, 8] according the Eq. 6 Where ρ is fluid density, u is the average velocity of the differential element in k direction at t time.

$$\frac{\partial \rho}{\partial t}+\frac{(\partial \rho u_k)}{(\partial x_K)}=0 \quad (6)$$

3.5. Second Newton law (Momentum Equation)

The second Newton law applied to a fluid particle with constant mass establishes that linear momentum rate is equal to sum of all force on it in a specific direction. This dynamic can be modeled on a Newtonian fluid in k direction [8] by the Navier and Stokes equations (Eq. 7),

$$\frac{\partial \rho u_k}{\partial t}+\frac{\partial \rho u_k u_j}{\partial x_j}=-\frac{\partial p}{\partial x_k}+\frac{\partial}{\partial x_j}\left[\mu\left[\frac{\partial u_j}{\partial x_k}+\frac{\partial u_k}{\partial x_j}\right]-\frac{2}{3}\,\mu\left(\frac{\partial u_j}{\partial x_j}\right)\delta_{jk}\right]+S_{Mk} \quad (7)$$

This equation represents the linear *momentum* conservation in Cartesian coordinates in conservative form for a compressible gas in laminar flow, where p is the pressure force, μ is the viscosity absolute, and δ_{jk} is the delta *Kronecker.* S_{Mk} is a source term which include the body force like a weight.

3.6. First thermodynamics law (Energy equation)

The first thermodynamics law, applied to a control volume of fixed mas traveling with the fluid, establishes the total energy rate is equal to energy flow of the particle plus the external work performed on it.

The total energy can include the internal, kinetics and gravitational energy. The obtaining of this equation is more difficult than last, but could be consulted in large number of text [7, 8].

The Eq. 8 represents the energy conservation for N number of species.

$$\rho\Sigma_{i=1}^{N}\left(Y_i\frac{\partial}{\partial t}\int_{T_0}^{T}c_{pi}dT\right)+\rho u_j\,\Sigma_{i=1}^{N}\left(Y_i\frac{\partial}{\partial x_j}\int_{T_0}^{T}c_{pi}dT\right)=\frac{\partial}{\partial x_j}\lambda\frac{\partial T}{\partial x_j}-\frac{\partial q_j^R}{\partial x_j}-\Sigma_{i=1}^{N}\left(j_{ij}\frac{\partial}{\partial x_j}\int_{T_0}^{T}c_{pi}dT\right)-\Sigma_{i=1}^{N}(h_i\acute{\omega}_i)+\left(\tau_{jk}:\frac{\partial u_k}{\partial x_j}\right)-\left(p\frac{\partial u_j}{\partial x_j}\right)+\frac{\partial p}{\partial t} \quad (8)$$

Where:

Y_i: is the mass fraction of each species.

c_{pi}: is the specific heat at constant pressure of each species.

T_0: is the reference temperature.

λ: is the thermal conductivity of the mixture.

q_j^R: is the heat flux from other sources like as radiation.

j_{ij}: is the mass diffusive fluxes due to concentration (Fick law) and temperature gradients (Soret effect) for each species.

h_i: is the internal enthalpy of each species.

$\acute{\omega}_i$: is the creation or destruction rate of a particular species. When there is not combustion, this variable is zero.

3.7. Turbulence model

Because the simplicity of analysis, the entire equation model above obeys to laminar fluids, however when the behavioral nature fluctuates in the different variables analyzed (temperature, velocity and species), as in most real problems, it is necessary to evaluate this fluid in a turbulent regimen. These instabilities result from the nonlinear inertial and viscous terms from the Navier and Stokes equations, making problems very rotational, tridimensional, and somehow transient, which involve source terms in remainder equations and the use of average variables.

To include the turbulent problem to the above equation, a great number of models have been developed to predict the behavior from the small to large turbulent structures made. In these models physical problems similar like the radiant tube with great validity [4, 9-11] the *k-ε standar model* have been used, because of robustness, low computational cost and good behavior in turbulent wide range. This is a semi-empirical model based on the transport equation for the turbulent kinetics energy (*k*) and its viscous dissipation (ε). To better predict some physical behavior, it is necessary fixing certain constants. In this case, to predict the recirculation phenomenon or in general the jets expansion [4] it is recommendable to set the Ce1 constant between 1,44 to 1,6.

3.7.1. Models

The models used to carry out the numerical evaluation were selected based on their prediction and computational cost. In general, the solver mode was based on pressure, in steady state, the Green-Gauss method based on cell to calculate the gradient, with absolute formulation of the velocity and the SIMPLE method to couple the pressure and the velocity. The energy equation is activated without chemical reaction because there is not combustion.

Two discretization methods, first and third order, and two radiation models, P-1and discrete ordinates (DOM) to evaluate the internal heat exchange were selected. In this analysis (Table 4) it was possible to observe few incident on results and in computational cost with the discretization grade; however when the P-1 model for radiation is changed to DOM model (8 angular division and 3 in the pixels), the numerical result changes very little but the computational cost grows enormously because iteration time increases two to three times.

In a similar way, the PRESTO model for the pressure was evaluated, verifying the recirculation values and the computational cost. It was possible to observe that the computational cost does not change while the PRESTO model has a little impact in the recirculation grade. Then, is possible use the third order, whit PRESTO model for the pressure and the P-1 model for the radiation.

Table 4. Model analisys

Characteristic	Recirculation %	Heat by radiation (kW))	Average temperature on outer tube (K)	Average temperature in the outlet gases(K)
1st order	39%	18,8	960	1084
3rd order	40%	18,8	960	1083
PRESTO	41%	18,8	960	1082
DOM	41%	18,6	958	1065

3.7.2. Properties

The numerical model to analyze the recirculation problem has the following characteristic:

Fuel: methane (CH_4) 100 %

Thermal power: 30,8 kW

Excess air: 10 %.

In conclusion the adopted models for the properties calculation are show in Table 5. For the Silicon Carbide the properties values were based on 1000 K, like 3160 kg/m^3 for density, 1195 j/kgK for the heat specific at constant pressure, and thermal conductivity of 87 w/mK, equality for the steel in this same order 8030 kg/m^3, 502,48 j/kgK, y 16,27 w/mK, while their emissivity was 0,9 and 0,7 respectively.

Table 5. Models and assumptions for the properties

Properties	Model or assumptions
Density	Incompressible ideal gas
Specific heat at constant pressure	Mixed gas law
Thermal conductivity	Mixed gas law – Ideal gas
Viscosity	Mixed gas law – Ideal gas
Mass diffusivity	Kinetic theory

4. Results

To corroborate the given maximum recirculation grade inside the proposed tube, several recirculation windows width were analyzed: 0, 20, 40, 60, 80, 95, 110 mm. These results are resented in the **Table 6.**

Table 6. Numerical analysis results

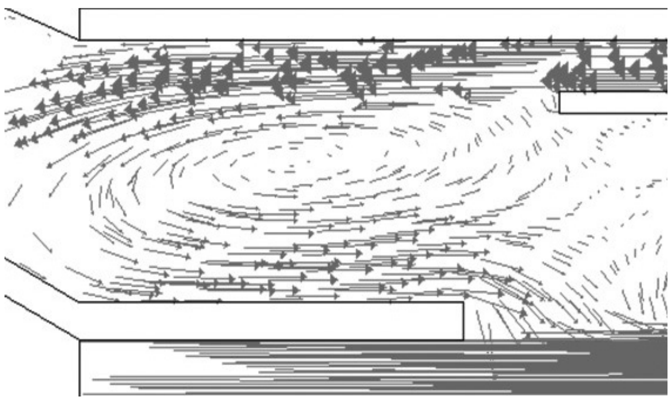
Window (mm)	Average temperature (K)	Standard deviation (K)	Difference bewteen max. and min. (K)	Heat by radiation (kW)	Heat by convection (kW)	Recirculation grade (%)	Pressure drop (Pa)
20	961,41	24,90	125,26	17,38	2,65	27,35	183
40	959,85	26,11	129,56	17,25	2,68	28,33	141
60	958,38	27,72	137,32	17,15	2,67	29,06	98
80	957,65	29,22	142,05	17,13	2,63	31,35	64
95	957,16	30,19	145,16	17,10	2,63	32,98	25
110	959,17	30,82	147,86	17,00	2,60	34,53	30
Without recirculation	960,46	28,55	129,72	17,33	2,64	-	174

It was possible to observe in general, minor changes in the recirculation grade when the window width varied.

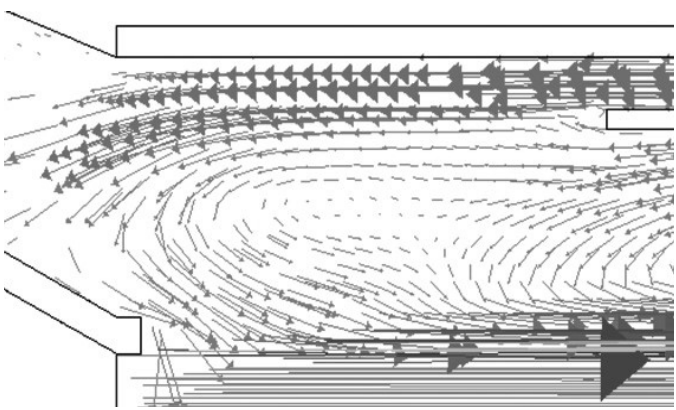
This behavior can be attributed to the low gas velocity in the discharge area in the combustion chamber nozzle compared with gas velocity when they return in the annular region, because as shown in the Figure 6, there are certain fluid streams that are returned immediately to the outlet in the same discharge zone.

Due to high temperatures in the discharge zone (2200 K), these low velocities are attributed solely to the nozzle geometry of the combustion chamber, which suggests that to obtain higher recirculation grades, it is necessary to decrease the nozzle area discharge, which would excessively increase the pressure drops in the system.

Figure 6. Found velocity field to a) window of 20 mm and b) 95 mm.



a)



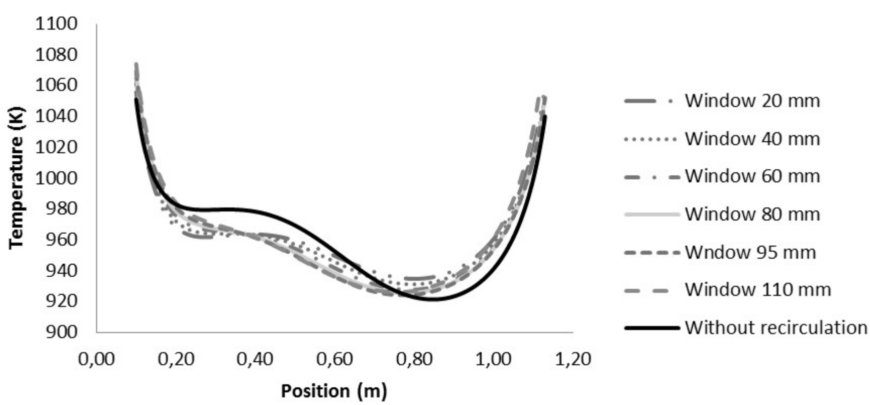
b)

In the **Figure 7** and the Table 6 it is possible to observe the small deviation in the temperature profiles and their uniformity (standard deviation) respect to the average temperature on the outer tube, because when smaller windows were simulated, lower recirculation grades, flatter profiles and higher average temperatures were obtained. This is due mainly to the inlet temperature into the inner tube is greater in the case of lower recirculation grades, and thus increases the temperature in the intermediate zones of both tubes, and as result the internal radiation can attenuate the increased temperature generating and retaining the uniformity profile. In a similar way when there is no recirculation window (Table 6), the values of the heat exchanged with the surroundings (radiation and convection) are very similar to those found with the different window widths.

5. Conclusion

Through the numerical simulation performed using a CFD FLUENT program and using a validated numerical model in an experimental test, the low potential effect of the recirculation when combustion is not developed inside the tube could be found, because the tube operating (except a little variations in pressure drop) was little affected by this phenomenon, including the temperature profile uniformity on the tube, and therefore it can be said that when combustion does not grow in the inner tube the recirculation does not provide significant effect on the thermal performance parameters of the tube as the radiation efficiency or effective radiated power and uniform distribution of temperature on tube, although it may favor some the pressure drop.

Figure 7. Surface temperature profiles on outer tube obtained by CFD



Acknowledgements

The present investigation was performed with the scientific and technological support of the "Grupo de Ciencia y Tecnología del Gas y Uso Racional de la Energía" research group from the Antioquia University.

The financial resources were provided by COLCIENCIAS.

References

C. Baukal, Heat Transfer in Industrial Combustion CRC Press LLC, Florida, 2000.

C. Baukal, Industrial Burners, Handbook, CRC Press LLC, Florida, 2003.

R. Sheridan, Determination of radiant output from infrared tube heaters, American Gas Association Laboratories, G. R. Institute GRI-94/0334, Cleveland, Ohio, May. 1994.

C. Galletti, A. Parente and L. Tognotti, Numerical and experimental investigation of a mild combustion burner, 4, Combustion and Flame, 151, 649-664, 2007.

M. K. Tiwari, A. Mukhopadhyay and D. Sanyal, Parameter optimization through performance analysis of model based control of a batch heat treatment furnace with low NOx radiant tube burner, 13-14, Energy Conversion and Management, 46, 2114-2133, 2005.

F. Incropera and D. D. Witt, Fundamentos de Transferencia de Calor, Prentice Hall, México, 1999.

S. V. Patankar, Numerical Heat Transfer and Fluid Flow, Taylor and Francis, 1980.

K. Versteeg H. and W. Malalasekera, An introduction to computational fluid dynamics. The finite volume method., Longman Scientific and Tehcnical, New York, 1995.

N. Tsioumanis, J. G. Brammer and J. Hubert, Flow processes in a radiant tube burner: Isothermal flow, 1, Fuel, 87, 103-111, 2008.

A. C. Mossi, H. A. Vielmo, F. H. R. França and J. R. Howell, Inverse design involving combined radiative and turbulent convective heat transfer, 11-12, International Journal of Heat and Mass Transfer, 51, 3217-3226, 2008.

M. Bidi, R. Hosseini and M. R. H. Nobari, Numerical analysis of methane-air combustion considering radiation effect, 12, Energy Conversion and Management, 49, 3634-3647, 2008.