

# EVALUATING MATHEMATICAL HEAT TRANSFER EFFECTIVENESS EQUATIONS USING CFD TECHNIQUES FOR A FINNED DOUBLE PIPE HEAT EXCHANGER.

Ali Hasan

Project Management and Construction Management, KEO Consulting Engineers, Doha,  
Qatar.

## **ABSTRACT**

*Mathematical heat transfer equations for finned double pipe heat exchangers based on experimental work carried out in the 1970s can be programmed in a spreadsheet for repetitive use. Thus avoiding CFD analysis which can be time consuming and costly. However, it is important that such mathematical equations be evaluated for their accuracy. This paper uses CFD methods in evaluating the accuracy of mathematical equations. Several models were created with varying; geometry, flue gas entry temperature, and flow rates. The analysis should provide designers and manufacturers a judgment on the expected level of accuracy when using mathematical modelling methodology. This paper simultaneously identifies best practices in carrying out such CFD analysis.*

*Methodology; CFD software was used to simulate different models. Results were tabulated and graphically presented. The investigated mathematical equations were programmed in a spreadsheet, for data entry.*

*Results and analysis; data obtained from the two methods were compared and differences were recorded. Discussions were included explaining the possible reasons for the deviations that surfaced between the two methodologies.*

*Conclusions; this analysis has shown that although mathematical equations are effective and simple tools in producing results, the results may not reflect the actual physical conditions. The analysis showed that the exhaust gas temperature outlet of a double pipe heat exchanger is actually higher than what were calculated using mathematical equations, and therefore, more heat energy is available for recapturing.*

*k-epsilon RNG turbulence model was found to be the most suitable method in analyzing heat transfer in a finned double pipe heat exchanger.*

## **KEYWORDS**

*CFD, Heat transfer, Double Pipe Heat Exchanger, Energy recovery.*

## **1. INTRODUCTION**

Research in heat transfer and thermo-physical fluid properties in the past three decades have helped develop heat exchangers designs, by relying on; mathematical, experimental, and computer numerical analysis. This has allowed designers to better design heat exchangers and push for better energy recovery systems. Lower energy consumption means lower operating costs and lower carbon emissions. The importance of heat recovery systems in combined heat and power systems have created a demand for effective heat recovery systems capable of recapturing heat energy out of the engine exhaust gas. A good example of this heat recapturing system is a

double pipe heat exchanger. Such a system can easily be fitted as part of an engine exhaust pipe system. The examples discussed in this paper, consider the process of heat energy transfer from the exhaust flue gas to water. This paper focuses on evaluating numerical techniques used to calculate heat transfer effectiveness. Providing designers and manufacturers, an insight on the expected level of accuracy when using mathematical equations associated with a double pipe heat exchanger.

CFD software was used to evaluate the model. Results were tabulated and compared with mathematically driven results. The analysis showed the difference between the results generated by the two different methods. Discussions on possible causes of differences were included supported by graphical CFD images. Best practices in CFD analysis were discussed with the support of similar recently carried out CFD work.

The CFD analysis highlighted the following facts; Mathematical equations are good for an initial analysis, but will require CFD or experimental modeling to obtain more accurate results on heat transfer, the mathematically generated results showed an underestimated temperature output for the exhaust flue gas. This analysis has shown the possibility of further energy recovery from the higher flue gas outlet temperature which would otherwise be wasted heat energy.

This document describes, and is written to conform to, author guidelines for the journals of AIRCC series. It is prepared in Microsoft Word as a .doc document. Although other means of preparation are acceptable, final, camera-ready versions must conform to this layout. Microsoft Word terminology is used where appropriate in this document. Although formatting instructions may often appear daunting, the simplest approach is to use this template and insert headings and text into it as appropriate.

## **2. MATHEMATICAL MODEL FOR DOUBLE PIPE HEAT EXCHANGERS**

Equations for calculating heat exchanger effectiveness are covered in details in ASHRAE Fundamentals (2009) [1], representing various types of heat exchangers.

Reference is made to Appendix 1 for a sample calculations with the conditions involved. Counter flow conditions were assumed in the two techniques; mathematical and CFD models. In certain cases and usually at the end of such equations, a check is required for a possible further iteration. Refer to Appendix 1 for such an example.

The example in Appendix 1 refers to heat transfer across a pipe surface area towards water flowing in the inner pipe. Tables 1 to 3 indicate results computed using the equations shown in Appendix 1. The second column from left of table shows the first attempt of calculations.

Table 1. Double Pipe Heat Exchanger with 16 fins.

Item	Description	CFD method	Mathematical method	Temperature difference (%) between mathematical and CFD methods (*)	After iterations (**)
1	Heat transfer effectiveness	-	0.984714		-
2	Temperature of water – out °C (°F)	42.00 (116)	47.39 (126.8)	13 %	-
3	Temperature of exhaust – out °C (°F)	64.03 (160.1)	42.44 (116.9)	51 %	-

Fin thickness 1 mm (0.04 inch). Fluids temperatures at entry point; water 40 °C (104 °F), & flue gas 200 °C (392 °F). Flue gas mass flow rate 0.12 kg/s (0.264 lb/s).

(\*) Results without iterations carried out on the mathematically generated results. (\*\*) Results after iterations using mathematical methods showing temperature and percentage difference between the CFD and mathematically generated temperatures.

Table 2. Double Pipe Heat Exchanger with 15 fins.

Item	Description	CFD method	Mathematical method	Difference (%) between mathematical and CFD methods (*)	After iterations (**) – temperature / [difference %]
1	Heat transfer effectiveness	-	0.887249		0.884507
2	Temperature of water – out °C (°F)	46.05 (124.1)	66.13 (164.3)	43.6 %	66.33 (164.66) / [6.4 %]
3	Temperature of exhaust – out °C (°F)	83.31 (198.6)	74.95 (181.9)	11 %	75.80 (183.6) / [86 %]

Fin thickness 1 mm. Fluids temperatures at entry point; water 40 °C (104 °F), & flue gas 350 °C (662 °F). Flue gas mass flow rate 0.243 kg/s (0.535 lb/s).

(\*) Results without iterations carried out on the mathematically generated results. (\*\*) Results after iterations using mathematical methods showing temperature and percentage difference between the CFD and mathematically generated temperatures.

Table 3. Double Pipe Heat Exchanger with 13 fins.

Item	Description	CFD method	Mathematical method	Difference (%) between mathematical and CFD methods (*)	After iterations (**) – temperature / [difference %]
1	Heat transfer effectiveness	-	0.887249		0.882163
2	Temperature of water – out °C (°F)	46.36 (124.72)	66.06 (164.12)	42.5 %	66.26 (164.52) / [13.4 %]
3	Temperature of exhaust – out □C (°F)	81.03 (194.06)	75.67 (183.34)	7.1 %	76.52 (185.04) / [9.0 %]

Fin thickness 1 mm. Fluids temperatures at entry point; water 40 □C (104 □F), & flue gas 350 □C (662 □F). Flue gas mass flow rate 0.243 kg/s.

(\*) Results without iterations carried out on the mathematically generated results. (\*\*) Results after iterations using mathematical methods showing temperature and percentage difference between the CFD and mathematically generated temperatures.

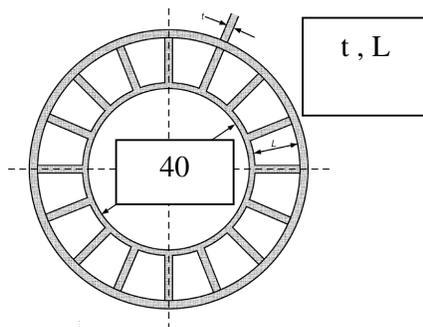


Figure 1. 16 fin cross sectional view, from ASHRAE Fundamentals Handbook (2009) [1].

t = fin thickness  
 L = fin length (radial)  
 40 mm = 1.6 inch

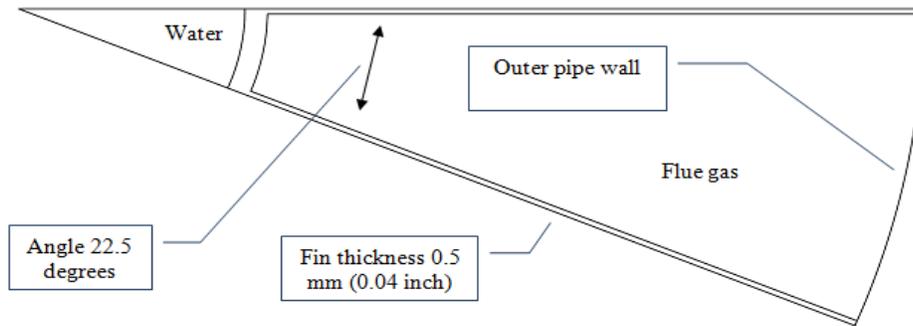


Figure 2. Sectional view example of a segment for a 16 fin double pipe heat exchanger. Graphical results are as shown in Fig 3. The outer pipe wall assumed to have no heat conductance.

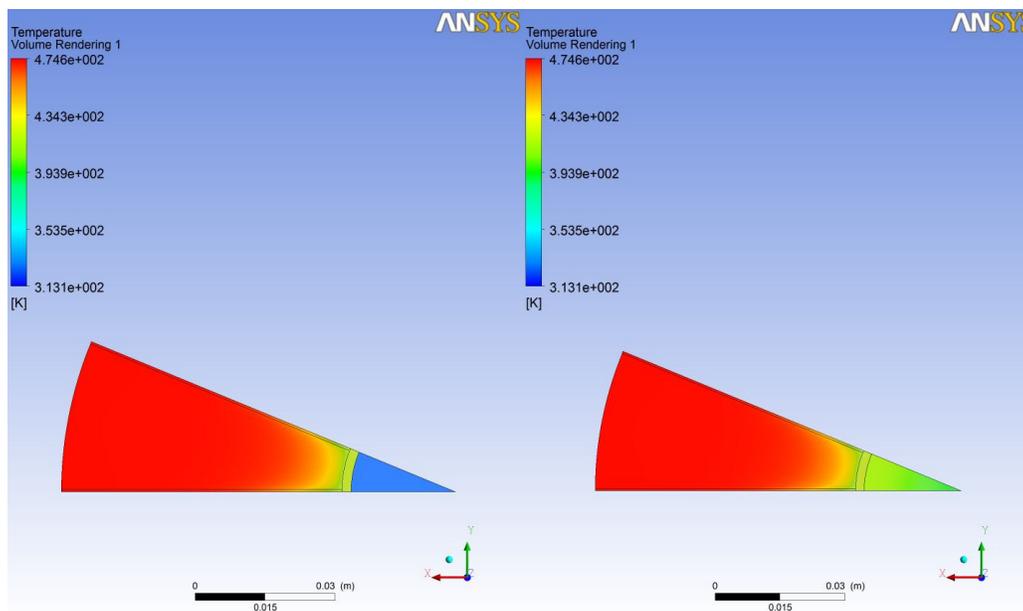


Figure 3. Front view images showing a 22.5 degree segment of pipe. Image to the left shows the water inlet section of the inner pipe. Image to the right shows how the water section of the inner pipe has increased in temperature. Copper pipe wall and copper fins are one continuous body. The lower and higher gas temperatures are clearly visible. Flue gas temperatures along the fin/tube metal surfaces are relatively lower than the middle flue gas volume. This is due to heat energy being transferred across the metal volume and towards the lower temperature body of fluid, water.

Temperatures produced for water and exhaust gas were added to entry temperatures to produce the average temperatures. Using the new average temperature results, the new values for density of water and specific heat capacity for gas were re-entered in the excel sheet to calculate the final temperature outputs, shown in the first column from right. In table 1 for an example the density of water and the specific heat capacity of gas at the new average temperatures were found close to what was originally assumed for, and therefore no further iteration work is required.

ASHRAE Fundamentals Chapter 4 (2009) [2], section on heat exchangers documents the development and experimental work associated with heat exchangers, including complex geometrical shapes. The

Reference is made to tables 1 to 3. Second from left column results were based on initial assumption of specific heat capacity of 1018 J/(kg.K) (0.2431 Btu/lb.F) at 175 °C (347 °F).

First column from right in tables 1 to 3, the specific heat capacity found in standard thermodynamic tables to be 1029 J/(kg.K) (0.2457 Btu/lb.F) at 212 °C (413.6 °F). Therefore iterations were carried out by entering in the mathematical equation 1029 J/(kg.K) (0.2457 Btu/lb.F). Results are as shown in tables 1 to 3.

Equations used in Appendix cater for conditions where temperatures of fluids leaving the heat exchanger are unknown. To avoid trial-and-error calculations, the NTU-  $\epsilon$  method uses three dimensionless parameters: effectiveness  $\epsilon$ , number of transfer units (NTU), and capacity rate ratio  $cr$ . Also, the equations used do not consider the inner pipe wall thickness, but just surface area, unlike CFD analysis carried below.

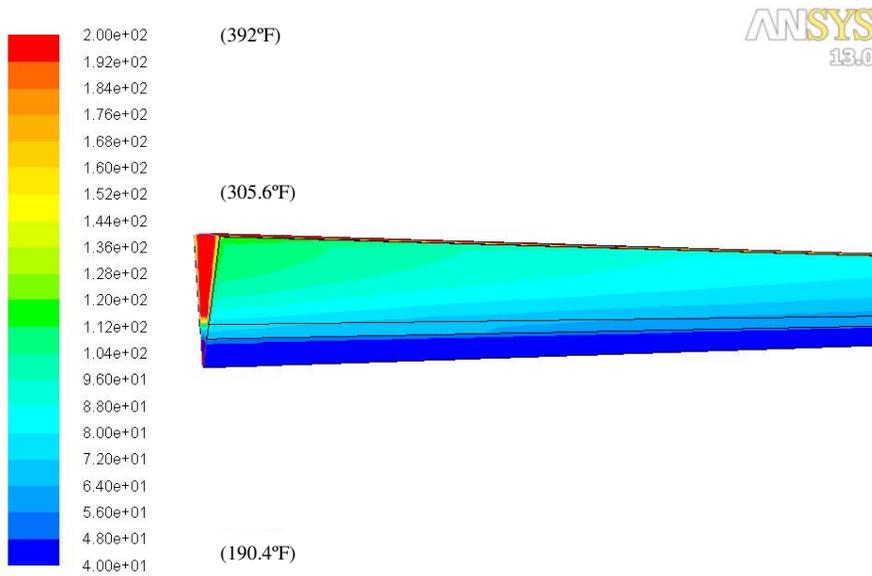
Fay C McQuiston (2000) [4] notes that, precise values are difficult to predict, and experience along with experimental data is often relied on.

In practice, improving heat transfer between two fluids in a heat exchanger usually depends on; construction materials, velocity of fluids, and size of heat exchanger (heat exchange surfaces). According to Fay C McQuiston (2000) [4], the trade-off between first cost (primarily size) and operating cost (primarily due to pressure drop) is a major consideration in heat exchanger analysis and selection.

### **3. CFD ANALYSIS**

Models were developed for a cross sectional segment. AutoCAD 2013 was used for modeling, and exported to ANSYS Fluent version 13, using IGES files. The AutoCAD file drawing in 2D first and then extruded into a 5000 mm (200 inch) length segment creating a 3D model. Walls, boundary conditions were defined, materials, and fluids data were entered. Analysis was run, and data was produces as shown in Figures 3 to 10. Fluids temperature properties were entered in tables 1 to 3 for comparison with the mathematically derived results.

Unlike the equations referred to in section 1, CFD modeling considers the inner pipe wall thickness, the material responsible for transferring heat energy from flue gas to the water body, and the fluid boundary conditions. Thus, CFD tools provide an effective heat transfer simulation between the fluid body and the solid body.



Contours of Total Temperature (c) Aug 31, 2013  
ANSYS FLUENT 13.0 (3d, pbns, ske)

Figure 4. 3D image (104°F) temperature contours for a 22.5 degree segment of the double pipe heat exchanger. Entry point for flue gas shown in red to the left of picture. Water temperature contours are shown as the water exits inner pipe, left of picture.

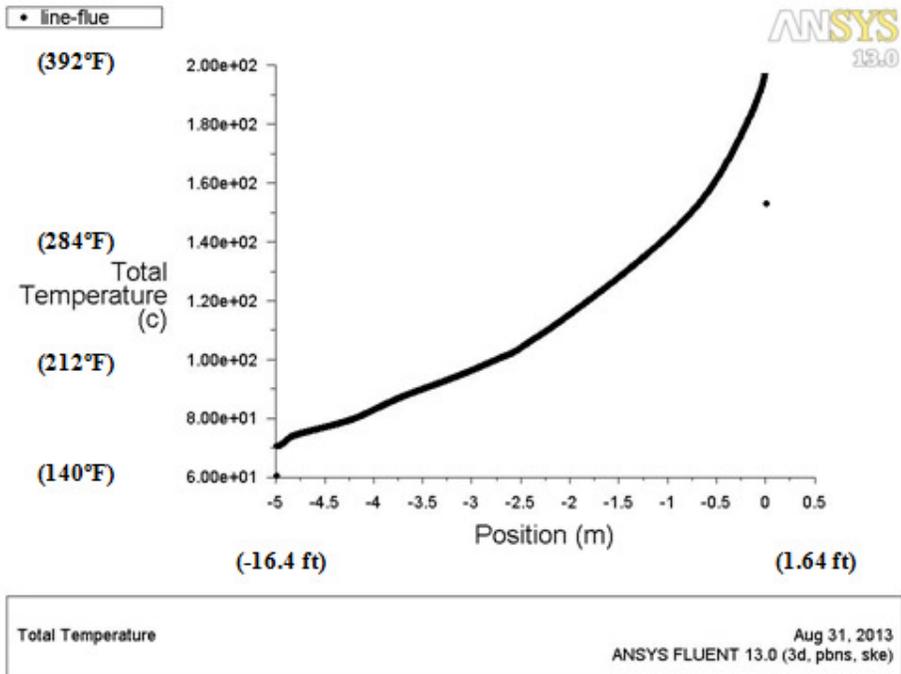


Figure 5. Flue temperature line for 22.5 degree segment.

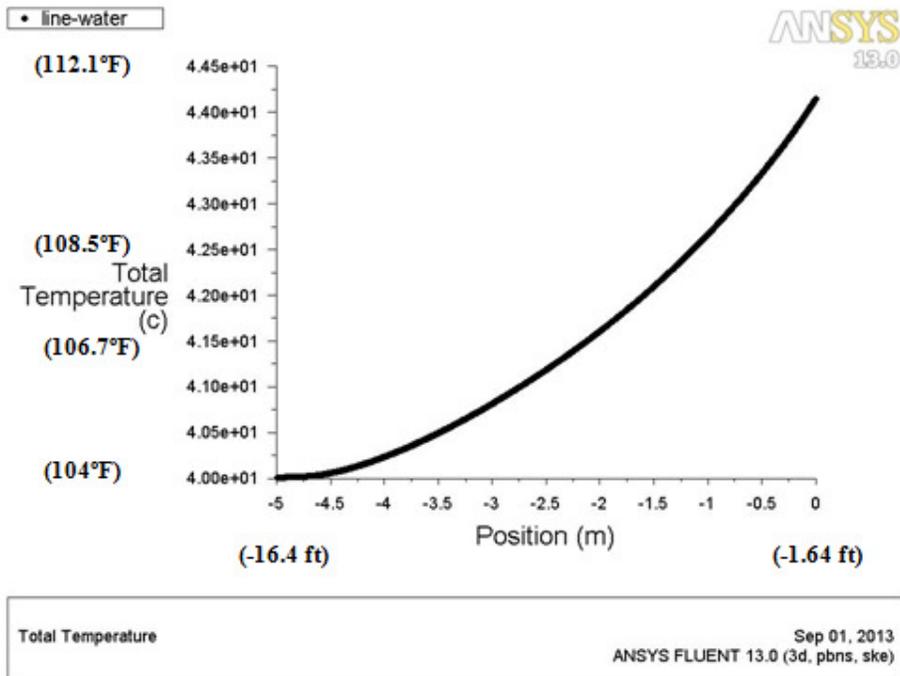


Figure 6. Water temperature line for 22.5 degree segment.

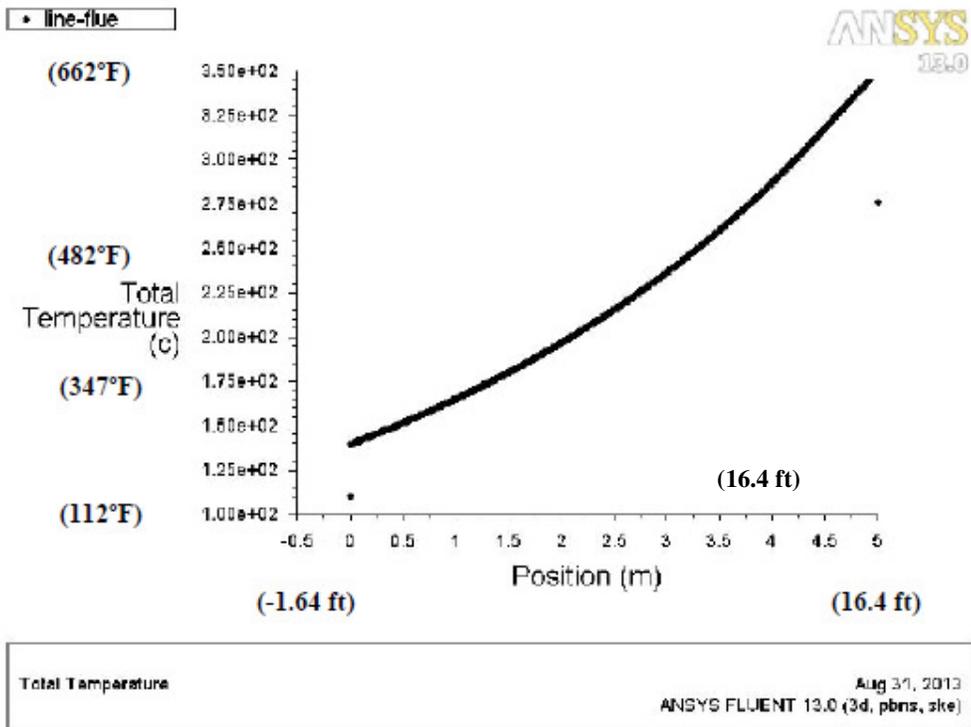


Figure 7. Flue temperature line for 24 degree segment.

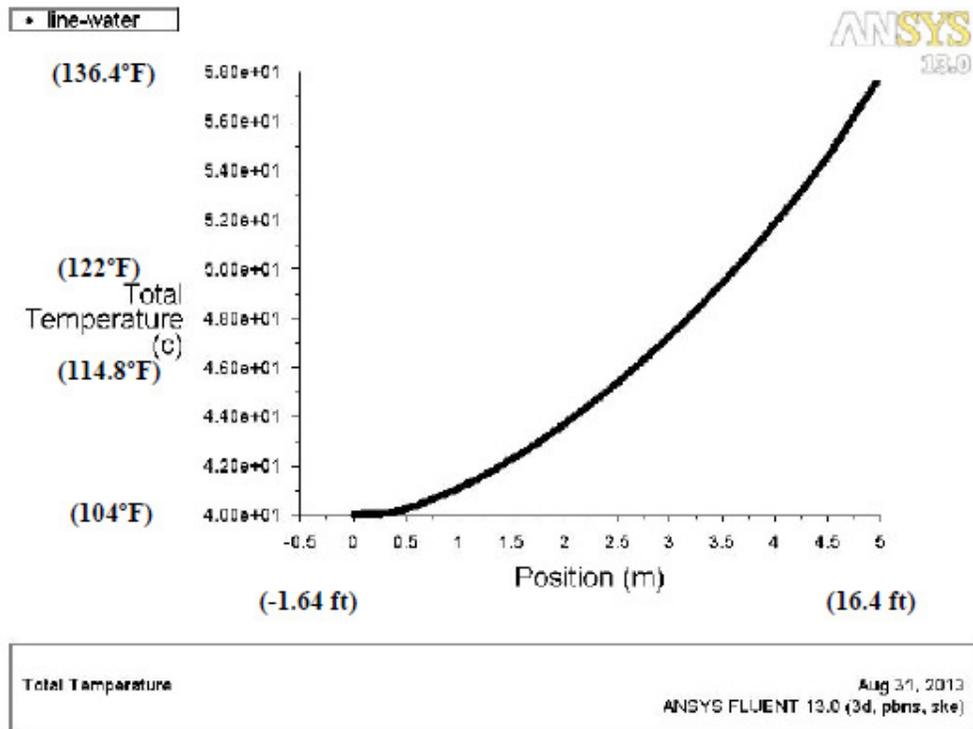


Figure 8. Water temperature line for 24 degree segment.

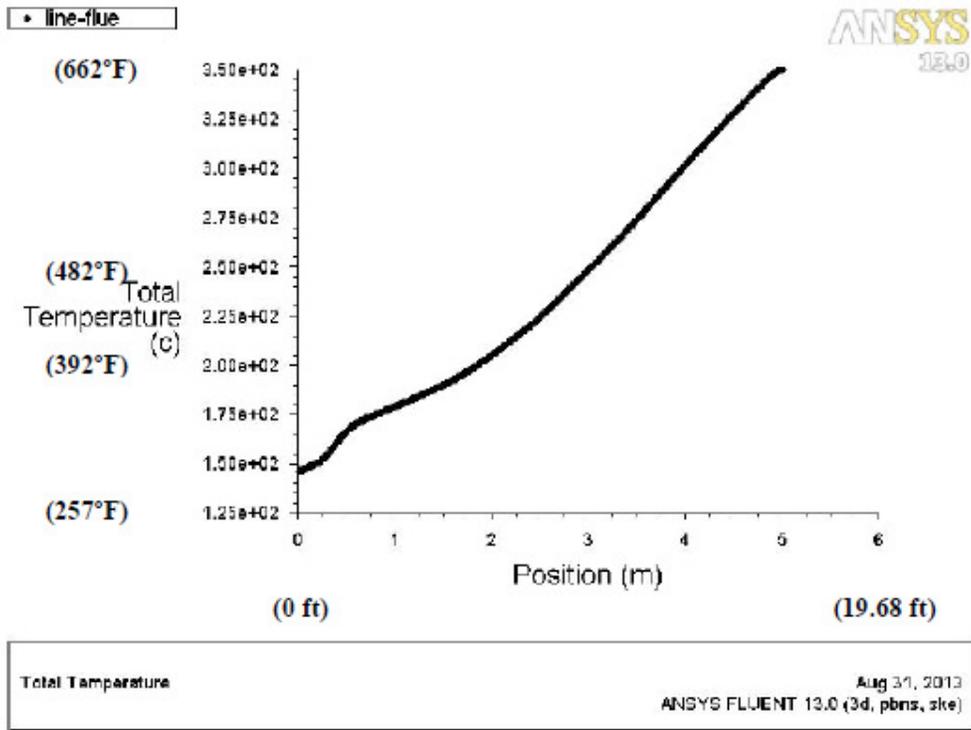


Figure 9.Flue temperature line for 27 degree segment.

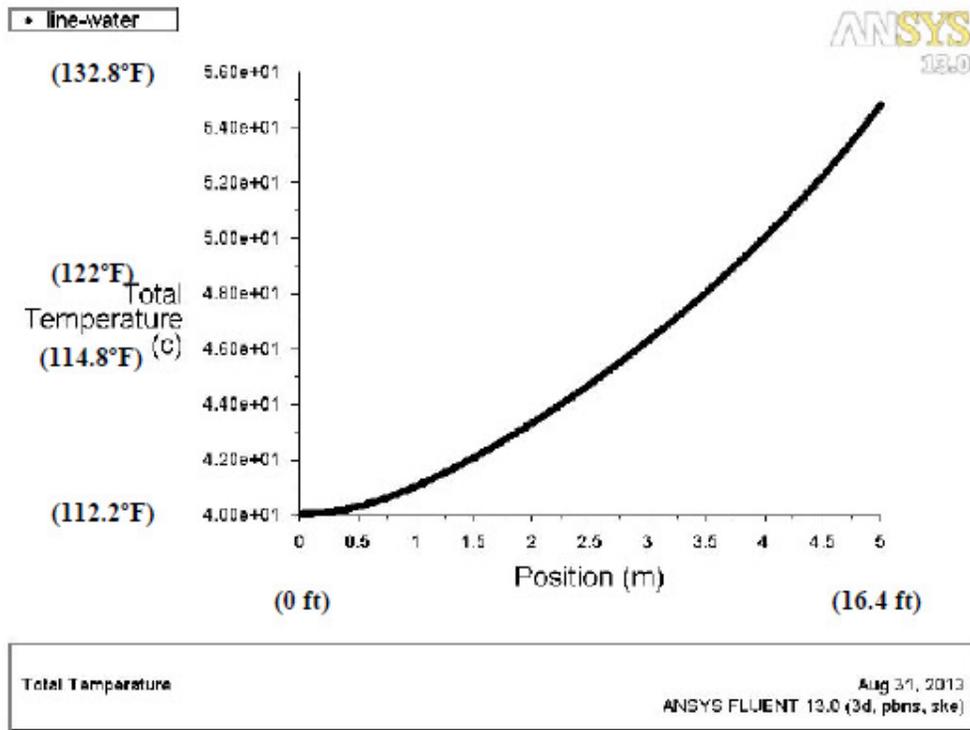
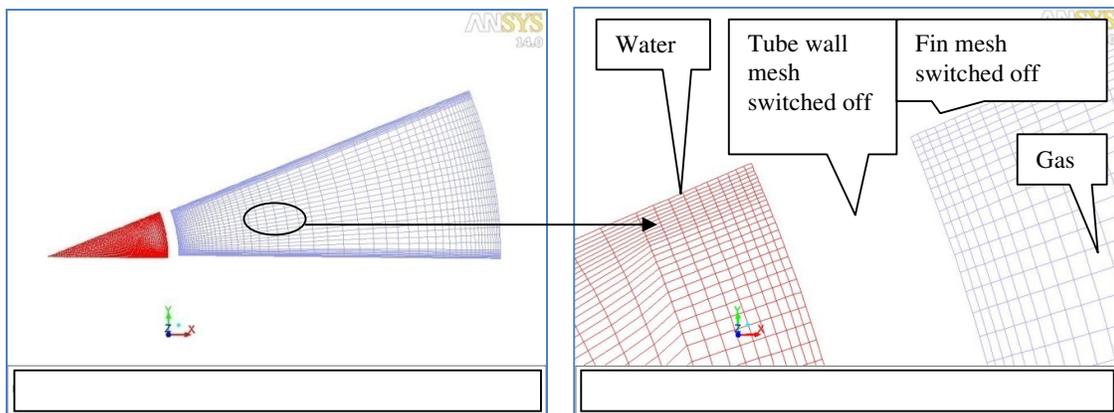


Figure 10. Water temperature line for 27 degree segment.



**Figure 11.** Shows meshing layout for the water inlet and gas outlet heat exchanger section. Geometry details shown in Fig 2. The solid tube and fin sections were switched off to give a clearer view on fluid elements layout. Fluid meshing layers at boundary conditions were inflated to give a more accurate prediction of heat transfer along the copper tube/fin walls. The right hand side image is a zoomed image of the encircled area shown on the left hand side image.

Fay C McQuiston (2000) [4] states, that manual design or simulation of a heat exchanger is an arduous task and seldom done. Computer programs are available to simulate or select heat exchangers for various applications.

#### **Assumptions made:**

- Flue gas as an ideal gas.  
Note: According to E Rathakrishnan (2004) [3], many of the familiar gases such as air, nitrogen, oxygen, hydrogen, helium, argon, neon, krypton, and even heavier gases such as carbon dioxide can be treated as ideal gas with negligible error, often less than 1%. Therefore, air, or gas where referred to in this paper, was assumed as an ideal gas in this CFD analysis.
- Materials, flue gas, copper fins and pipe wall, and water, as per the software standard database.
- K- Epsilon RNG, a common turbulence model selected from the software database. Wall roughness factor constant, 0.5. Refer to Appendix 2 for specific settings. Settings not specified in this paper were kept a software default settings.
- Inflated layers created along the fluid boundary layers in contact with the tube wall, and fin surfaces. This will improve boundary conditions simulations.

## **4. RESULTS AND ANALYSIS**

### **4.1 Results and Analysis discussed separately.**

#### **a. Results**

Results obtained using the mathematical modelling and the CFD analysis were tabulated as shown in tables 1 to 3. The results cover three different scenarios addressing different flow rates, temperatures, fin thickness, and flue gas entering temperatures. The object is to investigate how the mathematical equations perform under different conditions.

#### **b. Analysis**

Impact of fin thickness.

Tables 1 to 3 show that higher heat transfer effectiveness can be achieved with the thinner 1mm thick 16 fins, when compared to the thicker but lower number of fins.

Impact of flue gas entry temperature.

A high entry flue gas temperature of 350 °C (732 °F) produced higher water exiting temperatures, but lower effectiveness value. In accordance with mathematical and CFD derived exiting temperatures.

CFD - flue gas temperature line graphs.

Flue gas line graphs showed a positive slope with curvature (nonlinear) in relation to increasing temperature. Refer to graph Figs 5+.

CFD - water temperature line graphs.

Similar to the flue line graphs, positive slope as temperature increases. Percentage deviations showed better accuracy or lower percentage deviations using thinner fins. Table 1 indicated a deviation of 4.5% for a 1mm (0.04 inch) thick fin. Higher percentage deviations were observed against mathematically generated figures for thicker fins, as shown in tables 2 & 3.

The reasons for the significant differences between the out temperatures calculated using the two different methods, CFD and mathematical equations, are;

- a. The mathematical equation does not take into consideration heat transfer at boundary conditions. Heat transfer in the mathematical equations assumes that the transfer of heat energy between the two fluid bodies is a uniform process, with both fluid volume bodies having a uniform temperature across the body of fluid. In reality and as demonstrated in Figs 3 and 4. The temperature in the flue gas varies from locations along the fin surface and the pipe wall surface. The temperature in the middle of the flue gas volume remains higher than along the pipe wall, and fin surfaces, where heat energy is being conducted through the copper metal and passed towards the water fluid body of relatively lower heat temperature.
- b. The mathematical equations refer to pipe wall surface area, whereas in reality and in this CFD model the pipe wall has a thickness.
- c. Turbulences in the flue gas contribute to the mixing of the cooled gas boundary layers, with the inner middle flue gas volume. The higher the fluid turbulences are the better the heat transfer is between the two different fluid volumes.
- d. K-epsilon RNG turbulence models was found to be the most suitable model. Other models did not converge. This confirms the published work by M Hatami [5]. Stating that the RNG turbulence model converged easily during the processing stage, whereas SST or Shear Stress Transport theory did not easily converge.
- e. B.S. Massey1983 [6] describes flow near the boundary layer may be either laminar or turbulent. Turbulent flow past a solid surface having a random movement of particles perpendicular to the surface. Yet fluid particles cannot pass through an impermeable solid surface, and so, as the surface is approached, these movements perpendicular to it must die out. It follows then that turbulent flow cannot exist immediately in contact with the solid boundary. Thus even when main flow possess considerable turbulence, and even when the greater part of the boundary layer us also turbulent, there is still an extremely thin layer, adjacent to the solid surface, in which the flow has negligible fluctuations of velocity. This region, which may be less than a micrometer in thickness, has frequently been called the laminar sub-layer, but the term viscous sub-layer is now preferred. The viscous sub-layer plays a significant role in heat transfer between a fluid body and a solid surface.

It is this low-Reynolds-number extension for near-wall turbulence catered for by the k-epsilon RNG Near Wall Treatment function which makes the difference. The RNG Near Wall Treatment was designed to work for coarse and fine mesh, however, for better accuracy, mesh layers were inflated as shown in Fig 11.

#### **4.2 Advantages & disadvantages between the two techniques.**

- i) Spread sheets are relatively easier to program and do not require specialist trained personnel to operate, unlike CFD tools.
- ii) CFD techniques as in the example shown above show graphical results, not possible with excel sheets.

- iii) Both numerical techniques can be used to evaluate heat exchanger designs. Minimizing physical experimental techniques, which can be time consuming and costly.
- iv) Experimental techniques used to evaluate established designs, are prone to the introduction of errors due to non-accurate or non-calibrated instruments.
- v) CFD analysis is also cable of calculating pressure drops simultaneously through analysis. Though, pressure drop calculations can easily be programmed in a spread sheet.
- vi) Established CFD tools can be considered as more accurate, when compared with mathematical techniques, where thicker fins are used.

## **5. CONCLUSIONS**

Analyses showed that the water line exit temperature results produced better accuracy for thinner fins. Mathematically derived results differed when compared with CFD results by up to 13 % for a 1mm (0.04 inch) thick fin. Higher deviation between CFD and mathematical equations were observed for thicker fins.

Larger deviations existed between CFD and mathematical modelling on the flue line exit temperature results, as in the case of 1 mm fin thickness. Which is greater than 70 % in deviations on temperature flue gas exits. Therefore, it is important to establish the level of accuracy of mathematical equations under specific conditions; geometrical, and operational.

The CFD analysis has shown that the exhaust heat temperature is actually higher than what have been calculated using mathematical equations.

While for water outlet temperature figures, the mathematically derived figures were found to be higher than the CFD derived results.

In this investigation, it was observed that although the mathematical methods are simpler and easy to use once programmed in a spread sheet, the level of accuracy and how much energy can actually be recovered is a concern. Where justified and accuracy is important CFD and/or experimental investigations are recommended.

This CFD analysis confirms recently published work stating that the k-epsilon RNG turbulence model is the most suitable method in analyzing heat transfer in double pipe heat exchangers.

## **6. RECOMMENDATIONS**

Further research work using; mathematical, experimental and CFD techniques for different fin geometry and flow rates.

## **ACKNOWLEDGEMENTS**

The author would like to thank; Dr.Lik F. Sim for setting up and providing access to the FLUENT software, and MrAbdelkaderBenzamia of Flowpak, Flowpak, Doha, Qatar, [www.flowpak.net](http://www.flowpak.net), for the guidance and input on best practices in CFD mes.

## **REFERENCES**

- [1] ASHRAE Fundamental. F4.22-2009.
- [2] ASHRAE Fundamentals. F4.21: 2009.
- [3] Author E Rathakrishnan, Title; Gas Dynamics. Fifth edition. Publisher PHI. Year 2004.
- [4] Author, Fay C McQuiston. Title, Heating Ventilating and Air Conditioning – Analysis and Design. Fifth edition. Published by John Wiley and sons, Inc USA. Pages 484 and 485. Year 2000.
- [5] Paper by; M. Hatami, M. Jafaryar, D.D. Ganji and M. Gorji-Bandpy. Optimization of finned-tube heat exchangers for diesel exhaust waste heat recovery using CFD and CCD techniques. International Communications in Heat and Mass Transfer, Volume 57, October 2014, Pages 254–263.
- [6] B.S. Massey, Title; Mechanics of Fluids. Fifth edition. Publisher Van Nostrand Reinhold (UK). Year 1982.

## APPENDIX 1

Spreadsheet excel version 2010 was used in programming the equations shown below given in ASHRAE Fundamentals (2009) [2].

Assumptions:

- a. Not heat losses on the outside pipe wall, insulated.
- b. Starting with assumed properties of water and specific heat capacity of flue gas. At mean temperatures. Requirement for iterations can be checked as discussed in section 1 above.
- c. Counter flow conditions equation.  $C \neq 1$ , ASHRAE Fundamentals.

Table 3. Shows data entered in a spreadsheet programmed with the equations mentioned in steps 1 to 4.

Description	Abbreviation	Input	Units
Water in pipe	$t_{ci}$	40	°C (104 °F)
Water Velocity	$v_c$	0.5	m/s (1.64 ft/s) (392 °F) (0.264 lb/s)
Gas enters	$t_{hi}$	200	°C
Mass flow rate	$\dot{m}_h$	0.12	kg/s
Length of heat exchanger	$L_{tube}$	5	m (16.4 ft)
diameter of inner tube	$d$	0.04	m (0.1312 ft)
Fin radial height	$L$	0.06	m (0.1968 ft)
Fin thickness	$t$	0.001	m (0.00328 ft)
Number of fins	$N$	16	
Convective heat transfer coefficient on water side	$h_i$		
Gas side heat transfer coefficient	$h_o$	115	W/(m <sup>2</sup> K)
Surface effectiveness	$\phi_s$	0.641964	
Fin efficiency	$\phi$		
Surface are of non-finned surface	$A_{uf}$	0.5484	m <sup>2</sup>
Fin surface area	$A_f$	9.6	m <sup>2</sup>
$A_{uf} + A_f$	$A_o$	10.1484	m <sup>2</sup>
$\pi d L_{tube}$	$A_i$	0.6284	m <sup>2</sup>

Density of water	$\rho$	990.4	kg/m <sup>3</sup>
Specific heat capacity of water	$c_{pc}$	4183	J/(kg.K)
Dynamic viscosity	$\mu$	5.96E-04	(N.s)/ m <sup>2</sup>
Thermal conductivity water side	$k$	0.6376	W/(m.K)
Prandtl Number	$P_r$	3.91	
Gas specific heat - assumed figures for air	$c_{ph}$	1018	J/(kg.K)

$$\varepsilon = 0.984714$$

$$t_{he} = 42.44 \text{ } \square \text{C} \quad (108.4 \text{ } \square \text{F})$$

$$t_{ce} = 47.39 \text{ } \square \text{C} \quad (117.3 \text{ } \square \text{F})$$

$t_{he}$  = temperature of gas leaving system, &  $t_{ce}$  = temperature of water leaving system.

### Step 1

$$Re = \rho v_c d / \mu$$

$$f_s/2 = [1.58 \ln(Re) - 3.28]^{-2} / 2 = 0.00288$$

$$N_{ud} = [0.00288 \times (33213 - 1000) \times 3.91] \div [1 + 12.7 \times (0.00288)^{0.5} \times (3.91^{1/3} - 1)] = 180.4$$

$$h_i = (180.4 \times 0.6376) / 0.04 = 2876 \text{ W/(m}^2\text{K)}$$

### Step 2

Calculating fin efficiency  $\phi$  and surface effectiveness  $\phi_s$ . For a rectangular fin with the end of the fin not exposed.

$$\phi = [\tanh(mL)] / mL$$

For copper  $k = 401 \text{ W/(mK)}$

$$mL = (2h_i/k)1/2L = [(2 \square 2876)/(401 \square 0.001)]1/2(0.06) = 1.44$$

$$\phi = 0.62$$

$$\phi_s = (A_{uf} + \phi A_f) / A_0 = (0.548 + 0.62 \times 9.6) / 10.15 = 0.64$$

### Step 3

Find heat exchanger effectiveness. For air at an assumed mean temperature of 175°C,  $c_{ph} = 1018 \text{ J/(kg.K)}$ .

$$C_h = c_{ph} = 0.12 \square 1018 = 122.2 \text{ W/K}$$

$$\square_c = \rho v_c \pi d^2 / 4 = (990.4 \times 0.5 \times \pi \times 0.04^2) / 4 = 0.6223 \text{ kg/s}$$

$$C_c = c_{pc} = 0.6223 \times 4181 = 2602 \text{ W/K}$$

$$c_r = C_{min} / C_{max} = 122.2 / 2602 = 0.04696$$

$$UA = [1 / (0.64 \times 115 \times 10.15) + 1 / (2876 \times 0.628)]^{-1} = 528.5 \text{ W/K}$$

$$NTU = UA / C_{min} = 528.5 / 122.2 = 4.32$$

$$\varepsilon = 1 - \exp[-N(1 - cr)] \div [1 - cr \exp[-N(1 - cr)]]$$

$$\varepsilon = 0.983$$

### Step 4

$$q_{max} = C_{min} \times (t_{hi} - t_{ci}) = 122.2 \times (200 - 40) = 19552 \text{ W}$$

$$q = \varepsilon q_{max} = 0.985 \times 19552 = 19255 \text{ W}$$

### Step 5

$$t_{he} = t_{hi} - q / C_h = 200 - (19255 / 122.2) = 42.2 \text{ } \square \text{C} \quad (107.9 \text{ } \square \text{F})$$

$$t_{ce} = t_{ci} + q / C_c = 40 + (19255 / 2602) = 47.4 \text{ } \square \text{C} \quad (117.32 \text{ } \square \text{F})$$

## APPENDIX 2

This section provides a description of the turbulence model equations used in this CFD analysis, boundary inlet & outlet conditions, and details of specific settings used in this CFD analysis. A graphical presentation showing the quality of mesh used can be seen in Fig 11.

### *k-epsilon RNG*

Unlike other turbulence models, the k-epsilon RNG also known as k-ε RNG model focuses on the mechanisms that affect the turbulent kinetic energy. This model is a development on the standard k-ε model. A description of this model given in a paper by M Hatami (2014) [5];

RNG k-ε model thermal effect is considered in the enhanced wall treatment panel. Transport equations for RNG k-ε model in general form are

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{\text{eff}} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \epsilon - Y_M + S_k$$

and

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left( \alpha_\epsilon \mu_{\text{eff}} \frac{\partial \epsilon}{\partial x_j} \right) + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} - R_\epsilon + S_\epsilon$$

where  $G_k$  represents the generation of turbulence kinetic energy due to the mean velocity gradients and  $G_b$  is the generation of turbulence kinetic energy due to buoyancy and  $C_{1\epsilon}=1.42$ ,  $C_{2\epsilon}=1.68$  in RNG k-ε model.

In m Hatami's [2014] [5] paper states that the k-ω SST also known k-omega SST, and k-ε RNG were found most suitable in analyzing a double pipe heat exchanger, in comparison with experimentally derived results. Also, quoted that the RNG model was found easier to converge. These statements were found applicable in this CFD analysis.

The RNG model settings used in this paper were set at default with the exception of the following selections; Select Enhanced Wall treatment, and then from the Enhanced Wall Treatment Effects, select Pressure Gradient Effects and Thermal Effects items from the k-ε RNG model menu. These selections allow the model to address heat transfer and viscous layers at boundary conditions, which are important in this analysis.

For inlets and outlets boundary conditions, Intensity and Viscosity ratio model was used, with the following Specification Methods settings;

- Water inlet, 1% turbulence intensity, and 10 for viscosity ratio.
- Water outlet, 10% turbulence intensity, and 10 for viscosity ratio.
- Gas inlet, 1% turbulence intensity, and 10 for viscosity ratio.
- Gas outlet, 10% turbulence intensity, and 10 for viscosity ratio.

## AUTHORS

Ali Hasan, MSc Engineering 1993 from Sheffield Hallam University, Sheffield UK. Present employer KEO Consulting Engineers, Qatar. Working mainly on HVAC, CFD, and Energy efficiency projects. Place of birth Baghdad, Iraq. Date of birth 03-04-1963.