



A SIMPLE BARE TUBE CROSS FLOW RECUPERATOR FLOWNEX MODEL

This case study demonstrates the use of Flownex[®] to model simple recuperators using bare tubes in cross flow.



Download the
Flownex network

OIL & PETROCHEMICAL

Challenge:

The main challenge is the application of Flownex® to model simple recuperator designs that are based on bare tubes in cross flow.

Benefits:

One of the main strengths of Flownex® is its ability to model heat transfer to and from piping systems. As a result, it is relatively straightforward to create a Flownex® model for a recuperator which performs the relevant heat transfer and fluid mechanics calculations.

Solution:

A Flownex® recuperator compound component was developed and is presented in this case study. The model is based on single pass bare tubes in cross flow to a rectangular shell. Components can be connected in series (or parallel) to model multi-pass recuperators. The model presented utilizes Flownex®'s gas mixture capabilities to model the shell side fluid which is typically a flue gas. As such, the shell side fluid is only valid for low-pressure applications.

"Flownex® has the unique ability to perform the modelling of complex fluid mechanics and heat transfer interactions, such as those occurring in heat exchangers. The Flownex® Simulation Environment offers all the required building blocks to build advanced and comprehensive models within a short time and with relative ease."

Hannes van der Walt
Principal Thermal Engineer
Gasco (Pty) Ltd

INTRODUCTION

A recuperator is a specific application of a heat exchanger used to recover heat from a hot fluid. Perhaps the most frequent application of a recuperator is to recover heat from hot flue gas before it is discharged into the atmosphere. Several different recuperator designs are used in industry, however one of the simplest and most cost effective designs is a bundle of bare tubes positioned in cross flow to the flue gas. The tubes may be a single pass or multi-pass design. Figure 1 shows a schematic of a six-pass recuperator for a gas turbine.

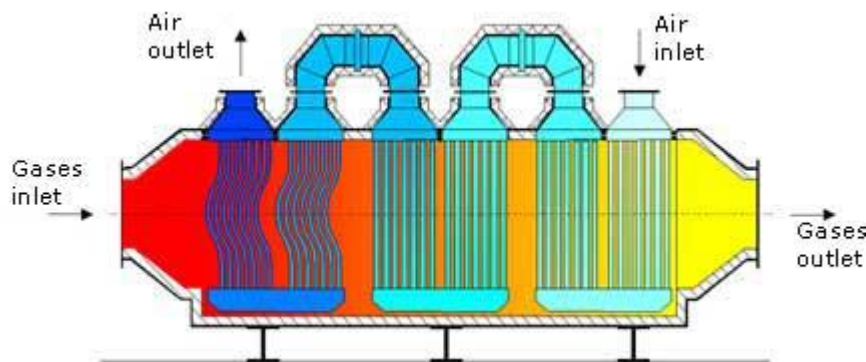


Figure 1: A multi-pass exhaust gas recuperator heating air.

FLOWNEX® MODEL

A compound component was developed in Flownex® to model a single tube pass recuperator based on bare tubes in cross flow to a rectangular shell. The component is set up such that a minimum of inputs are required to assess the performance of the design. Figure 2 shows a simple recuperator model for a single tube pass design. The process inputs are specified at the boundary conditions while the recuperator geometry is specified on the recuperator element itself. The blue rectangle shows most of the available inputs for the recuperator. Figure 3 shows five recuperator bundles in series for the exhaust gas while there are two independent sections; one heating CO₂ and the other heating air. As shown, bundles can easily be connected in series for the exhaust gas, as is usually the case, but they can also be connected in parallel. Combinations of series and parallel bundles can also be specified with ease. Similarly, the connection of the tube-side flow paths can be as simple or complex as the user wants. Figure 3 shows the air outlet bundle consisting of two bundles in parallel.

"Using a relatively straightforward Flownex® compound component, a comprehensive flue gas recuperator model could be constructed. It was shown that the results are in close agreement with other commercial software such as Aspen EDR, however its flexibility goes beyond the capabilities of most commercial software."

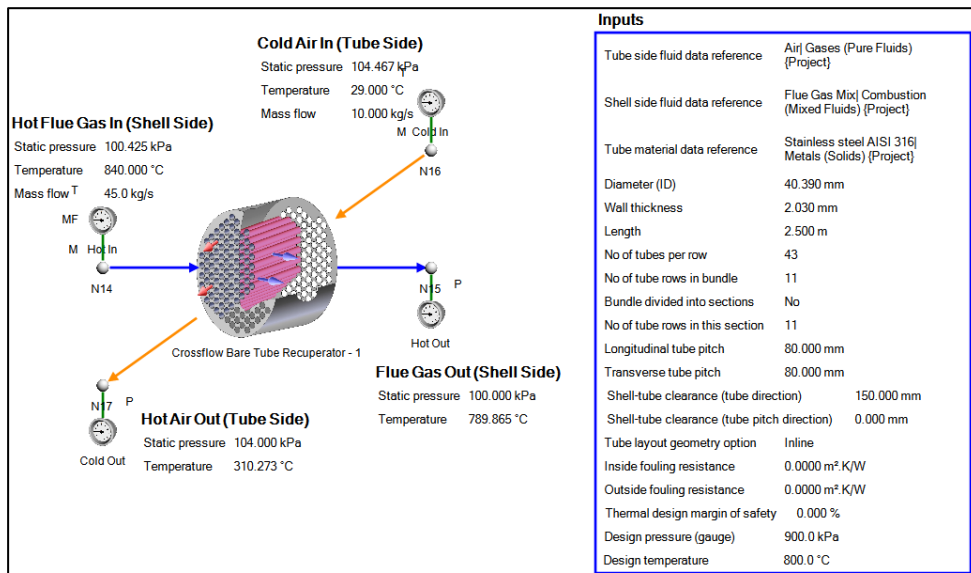


Figure 2: A single-pass exhaust gas recuperator heating acid gas.

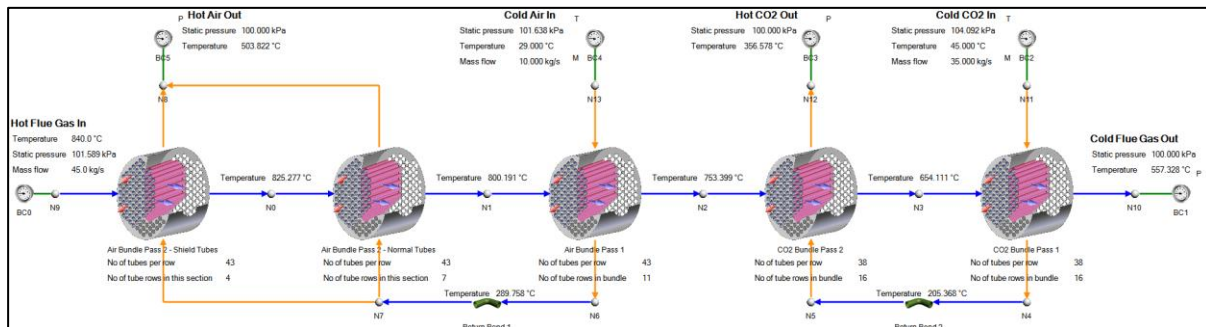


Figure 3: A multi-pass exhaust gas recuperator heating CO₂ and air.

RECUPERATOR COMPOUND COMPONENT INPUTS AND RESULTS

The recuperator compound component inputs property page was laid out to simplify the input data requirements as much as possible and is shown in Figure 4.

In the Fluids & Material group, the shell-side and tube-side fluids and the tube material are specified. The Tube Geometry group specifies data relating to the tube geometry, most of which are self-explanatory. The following items may require further clarification:

- *Length*: this is the tube length per single pass, i.e. it is the length of a single tube visible to the flue gas stream in the shell.
- *No of tubes per row*: this is the number of tubes in the transverse direction, perpendicular to the flow.

- *No of tube rows in bundle*: this is the number of tube rows in the longitudinal direction, i.e. parallel to the flow. The product of the *No of tubes per row* and the *No of tube rows in bundle* represents the total number of tubes in the bundle.
- *Bundle divided into sections*: the model allows for a single bundle (single tube pass section) to be split into more than one Flownex[®] component to allow for more accurate calculations. The tube bundle in the compound component consists of a single Flownex[®] pipe element. Although the pipe element is sub-divided into ten increments, which allows for temperature variations along the tube, no variation in the direction of the flue gas flow is allowed for. Of specific importance is the variation of tube wall temperatures in the direction of flue gas flow. The first few tube rows often referred to as the shock or shield tubes, are usually of a different material capable of withstanding higher temperatures. It is convenient therefore to be able to divide a bundle into two or more sections to obtain better estimates of the tube wall temperatures in the first few tube rows. Therefore, when a bundle is divided as is shown in Figure 3 above, the sub-divided bundle Flownex[®] components are connected in parallel for the

Crossflow Bare Tube Bundle - 1 (Crossflow Bare Tube Bundle) P...	
General	
Identifier	Crossflow Bare Tube Bundle - 1
Solving	☒
Description	
Fluids & Material	
Tube side fluid data reference	Air Gases (Pure Fluids) {Project}
Shell side fluid data reference	Flue Gas Mix Combustion (Mixed...
Tube material data reference	Stainless steel AISI 316 Metals (...)
Tube Geometry	
Geometry option	Specify geometry
Diameter (ID)	40.39 mm
Wall thickness	2.03 mm
Length	2.5 m
Roughness	10 µm
K forward	0
No of tubes per row	43
No of tube rows in bundle	11
Bundle divided into sections	No
Longitudinal tube pitch	80 mm
Transverse tube pitch	80 mm
Shell-tube clearance (tube direction)	150 mm
Shell-tube clearance (tube pitch direction)	0 mm
Tube layout geometry option	Inline
Tube Fouling	
Inside	
Tube inside fouling options	No fouling
Outside	
Tube outside fouling options	No fouling
Thermal Design	
Thermal design margin of safety	0 %
Mechanical Design	
Tensile strength chart	ASTM A312TP316 & A182GrF316...
Design standard option	ASME B31.3
Design pressure (gauge)	900 kPa
Design temperature	800 °C
Corrosion allowance (inside)	0 mm
Corrosion allowance (outside)	0 mm
Under-tolerance	12.5 %
Weld quality factor (joint efficiency)	1 0-1
Weld joint reduction factor	1 0-1

Figure 4: Input data property page

- Furthermore, the *No of tube rows in bundle* for all the subdivided sections must be the same, i.e. 11 in the case shown. The entry field *No of tube rows in this section*, which will appear when a bundle is sub-divided, must be completed so that the totals for all the sub-divided sections will equal the total number of tubes in the bundle. For example, if the bundle consists of eleven rows as shown, and the bundle is subdivided into two sections with the first shield tube section comprised of 4 rows, then the second section must have seven rows.
- The longitudinal and transverse pitches are shown in Figure 7.
- The next two fields, *Shell-tube clearance*, are used to calculate the rectangular shell width and height. The shell width is calculated from the number of tubes per row and the transverse tube pitch, and then the clearance in the tube pitch direction is added. The shell height is simply taken as the tube length plus the *Shell-tube clearance (tube direction)*.
- The user has five options when specifying tube fouling: *No fouling*, *Resistance only*, *Resistance based on thickness*, *Resistance based on conductivity* and *Thickness and conductivity*. For the second option, the user only needs to specify the fouling factor. For the third to fifth options, the user must provide a combination of two variables from the *Resistance* (fouling factor), *Thermal conductivity* of the foulant (fouling material) and the *Thickness* of the fouling layer.
- The Thermal design margin of safety is a safety factor used in design. To ensure the design is 10% over-surface, enter a factor of 10%.
- The model also assists in the mechanical design aspects of the tubes. Design parameters are entered which allows the model to verify the sufficiency of tube wall thicknesses and required flange ratings for tube connections.

The results page is similarly laid out with simplicity in mind. The results groups are briefly as follows:

- The *Performance Results* provides a short summary of the most important heat transfer performance results.
- The *Geometry Results* provides the user with the total number of tubes in the bundle, the heated tube length, shell dimensions and more.
- The *Tube-Side Results* provides *Heat Transfer* results for the tube inside and outside surface as well as *Flow Results*, *Mean Fluid Variables* and *Upstream* and *Downstream* flow element results that are similar to those of a standard Flownex® pipe element.
- *Shell-Side Results* offer the results of mass flow, velocity, pressures, temperature and fluid properties for the shell-side fluid.
- *Mechanical Design Results – Tube* present the user with the results of tube design strength calculations.
- *Mechanical Design Results – Flange* presents the user with the results of flange design strength and rating calculations.

RECUPERATOR COMPOUND COMPONENT

The recuperator compound component design is discussed in the following sections. When double-clicking on any of the bundle compound components shown in Figure 2 or Figure 3, the internals of the component is shown in a separate window (Figure 6) which allows the user to view and edit any of the inputs and results of any of the elements that form part of the compound component.

Although the compound component's network may look rather busy, it is essentially comprised of just three standard Flownex® elements; a pipe element representing the shell side flow, a pipe element representing the tube side flow, and a Composite Heat Transfer (CHT) element representing the heat transfer between the two flow streams. The remaining icons are scripts that perform the necessary calculations for the outside convection coefficient, the shell-side pressure loss, fouling factor calculations, fluid properties, pipe and flange rating calculations and some post-processing calculations. Some of these calculations are discussed in more depth below.

Crossflow Bare Tube Recuperator - 1 (Crossflow Bare Tube Rec...			
General			
Identifier	Crossflow Bare Tube Recuperator - 1		
Description			
Performance Results			
Total heat load	2890.31		kW
Heat flux (average)	17.5034		kW/m²
Uo	27.6179		W/m².K
Ao	165.129		m²
LMTD	637.472		°C
LMTD (corrected)	633.77		°C
Geometry Results			
Total number of tubes	473		
Total tube length (heated)	1182.5		m
Tube outside diameter	44.45		mm
Diagonal tube pitch	89.4427		mm
Shell height (tube direction)	2.5		m
Shell width (tube pitch direction)	3.52		m
Shell length	0.96		m
Shell cross sectional area	8.8		m²
Tube Side Results			
Inside Heat Transfer			
Maximum surface temperature	716.205		°C
Average surface temperature	692.166		°C
Minimum surface temperature	674.6		°C
Prandtl number	0.700328		
Reynolds number	27478.5		
Nusselt number	42.0452		
Convection coefficient	37.5961		W/m².K
Area	150.046		m²
Outside Heat Transfer			
Maximum surface temperature	717.738		°C
Average surface temperature	693.735		°C
Minimum surface temperature	676.12		°C
Prandtl number	0.83046		
Prandtl number at T_s	0.817723		
Reynolds number	10697		
Nusselt number	85.8338		
Convection coefficient	144.747		W/m².K
Area	165.129		m²
Flow			
Mean total pressure	104.401		kPa
Mean static pressure	104.25		kPa
Mean total temperature	170.491		°C
Total pressure loss	0.36182		kPa
Mass flow	10		kg/s
Volume flow	8.67769		m³/s
Maximum velocity	26.6322		m/s
Friction factor	0.0246595		
Mean Fluid Variables			
Density	0.848559		kg/m³
Conductivity	0.0361659		W/m.K
Viscosity	2.47181E-05		kg/m.s
Specific heat	1.0228		kJ/kg.K
Gas constant	0.287004		kJ/kg.K
Upstream			
Total pressure	104.582		kPa
Static pressure	104.469		kPa
Total temperature	29		°C
Area	0.606036		m²
Velocity	13.6926		m/s
Density	1.20508		kg/m³
Downstream			
Total pressure	104.22		kPa
Static pressure	104		kPa
Total temperature	311.607		°C
Area	0.606036		m²
Velocity	26.6322		m/s
Density	0.619575		kg/m³
Shell Side Results			
Mass flow	45		kg/s
Inlet velocity	12.3305		m/s
Maximum velocity between tubes	27.748		m/s
Reynolds number between tubes	10697		
Total pressure loss	0.445005		kPa
Mean total pressure	100.252		kPa
Mean static pressure	100.22		kPa
Mean temperature	814.717		°C
Mean density	0.423486		kg/m³
Mean conductivity	0.0749591		W/m.K
Mean viscosity	4.88293E-05		kg/m.s
Mean specific heat	1.27485		kJ/kg.K
Gas constant	0.217586		kJ/kg.K
Mechanical Design Results - Tube			
Design tensile strength	10400		kPa
Table 304 Y-coefficient	1.97857		0-1
Required pipe nominal thickness	1.84741		mm
Pipe design strength margin	9.88338		%
Mechanical Design Results - Flange			
ASME B16.5 flange class	900		
ASME B16.5 flange MAWP	1050		kPa

Figure 5: Results data property page.

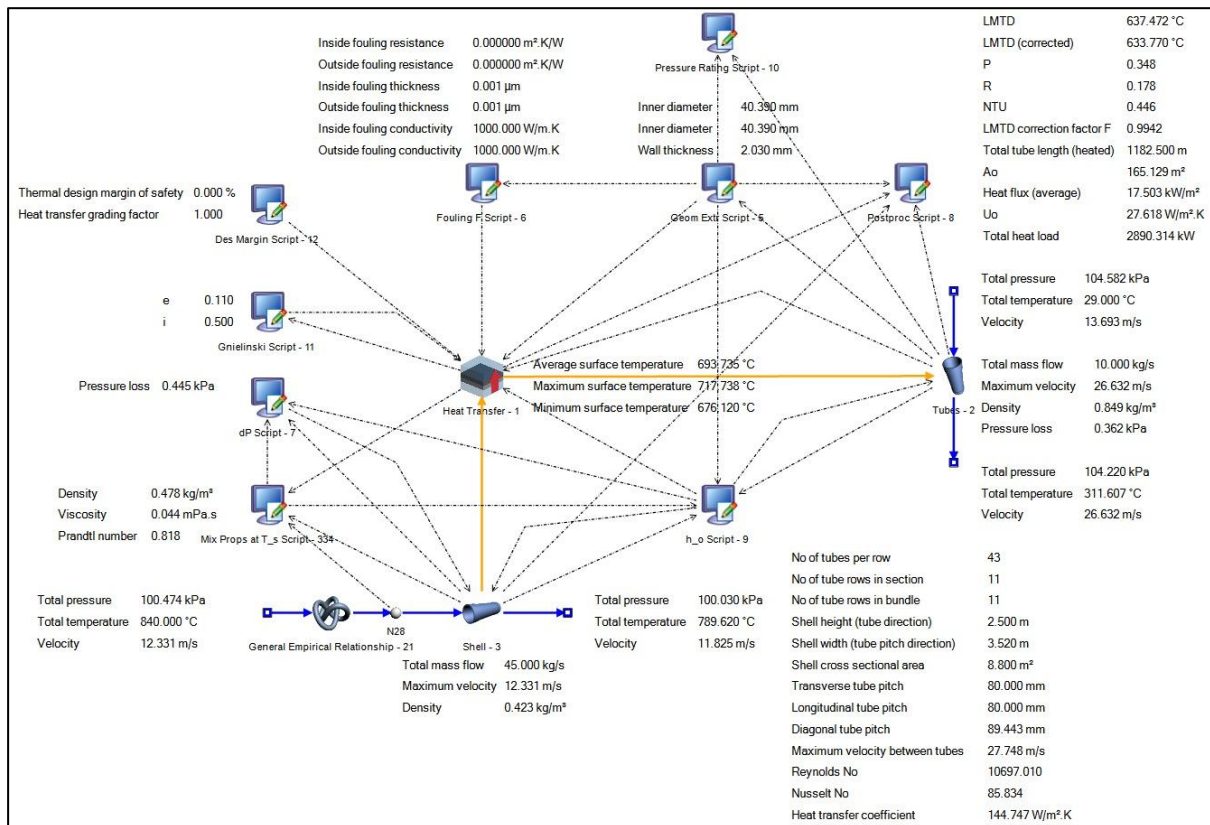


Figure 6: The recuperator compound component.

Note that both the tube-side and shell-side pipe elements are subdivided into ten increments to improve calculation accuracy. Furthermore, the tube pipe element also specifies a number of tubes in parallel equal to the number of tubes in the bundle.

Tube Geometry Extractor (Geom Extr Script)

A script was developed to enable the correct tube diameter and wall thickness to be obtained regardless of whether the user specified the tube geometry in terms of a nominal diameter and a schedule or a simple internal diameter and a wall thickness.

Pressure Pipe & Flange Rating Script (Pressure Rating Script)

This script may be used to perform mechanical strength calculations for the coil pipe and associated flanges. The development of this script is discussed in detail in the following case study available from the Flownex® website [1]:

[Pressure Piping Thickness and Flange Rating Calculation](#)

Outside Convection Script (h_o Script)

This script reads the tube and shell geometries from the tube and shell pipe elements as well as the Tube Geometry Extractor. It then calculates the outside tube convection coefficient using methods developed by Zukauskas [2] which are discussed in more detail below. This convection coefficient is then assigned to the shell-side of the CHT element.

Fouling Factor Script (Fouling F Script)

The script implements fouling factor calculations by modeling the internal and external tube fouling factors as actual layers specified in the CHT element. The user may specify:

- No fouling: When selected, Flownex® will set the layer thickness to a negligibly small value (1E-6 mm) and the layer conductivity shall be fixed to a very high value (1000 W/m.K). This will result in a fouling layer that is negligibly small with a negligible temperature difference across it.
- Resistance only: When selected, the user may specify a value for fouling resistance (m².K/W). Since the actual fouling layers are modeled, Flownex® assumes a hard-coded layer thickness of 0.1 mm. The fouling layer thermal conductivity is then calculated based on these two values.
- Resistance – based on thickness: When selected, the user may specify both the fouling resistance and the fouling layer thickness. As with the previous option, the fouling layer thermal conductivity is then calculated based on these two values.
- Resistance – based on conductivity: When selected, the user may specify both the fouling resistance and the fouling layer thermal conductivity. Similar to the previous two options, these two values are then used to calculate the fouling layer thickness.

A previous case study by this author on the modeling of fouling factors, titled *Fouling Factors in Flownex Heat Transfer Models* [3] may be found here:

[Fouling Factors in Fownex Heat Transfer Models](#)

Mixed Fluid Props at T_s Script (Mix Props at T_s Script)

This script is a copy of script in the Master Database script repository and is used to obtain the shell-side fluid properties at tube surface temperatures.

Pressure Loss Script (dP Script)

This script calculates the shell-side fluid pressure loss. This represents the flue gas flow pressure loss across the tube bank. The script implements a method by Gaddis [4] as published in the VDI Heat Atlas (2010) [4] which is discussed in more detail below. The script allows for both inline and staggered tube arrangements and the specification of transverse and longitudinal tube pitches.

Gnielinski Temperature Correction Script (Gnielinski Script)

The model uses the Gnielinski [5] correlation to calculate the tube inside Nusselt number. The CHT element allows for several correction factors, amongst others for effects where large differences between the tube wall temperature and bulk fluid temperature exist.

Design Margin Script (Des Margin Script)

It is convenient to be able to specify a design margin F (safety factor) when sizing equipment. This script allows for the specification of a design margin which is then assigned to the CHT element as a *Heat transfer grading factor*. This factor F , between 0 and 1, decreases the heat transfer as follows:

$$\dot{Q}_{design} = F \times \dot{Q}_{calculated} \quad \text{Eq. 1}$$

Postprocessing Script (Postproc Script)

This script is used to determine some valuable performance parameters for reporting purposes. These include the LMTD, the corrected LMTD, the tube total outside surface area, the heat flux and the overall heat transfer coefficient U_o .

CORRELATIONS USED IN THE RECUPERATOR MODEL

Several different empirical models are available for the calculation of heat transfer coefficients and pressure losses across tube banks. The tube outside convection coefficient used in this model is based on the Zukauskas [2] correlations. The shell side pressure loss across the tube bundle is based on the method of Gaddis [4]. The fouling factor calculation methods are based on equations presented by this author.

External Convection Coefficients of Tube Banks in Cross Flow

The model implements the updated correlations by Zukauskas [2] as presented by Cengel [6]. The correlations are presented in the following two tables:

$$Nu_D = \frac{hD}{k} = F C Re_D^m Pr^n (Pr/Pr_s)^{0.25} \quad \text{Eq. 2}$$

where:

Nu_D is the Nusselt number based on the tube outside diameter.

h is the tube outside surface convection coefficient, $W/m^2.K$.

D is the tube outside diameter, m.

k is the outside fluid thermal conductivity, $W/m.K$.

F is a correction factor for a bundle with fewer rows than 16 (N_L) as given in Table 2.

C , m and n are constants given in Table 1.

Re_D is the Reynolds number based on the outside tube diameter, m.

Pr is the Prandtl number based on fluid properties at the mean bulk fluid temperature.

Pr_s is the Prandtl number based on fluid properties at the tube surface temperature.

Table 1: Nusselt Number correlations for cross flow over tube banks for $N_L \geq 16$ (from Zukauskas, 1987) (N_L is the number of tube rows in the shell flow direction)

Arrangement	Range of Re_D	Correlation		
		C	m	n
In-line	0 - 100	0.9	0.4	0.36
	100 - 1000	0.52	0.5	0.36
	1000 - 2×10^5	0.27	0.63	0.36
	2×10^5 - 2×10^6	0.033	0.8	0.4
Staggered	0 - 500	1.04	0.4	0.36
	500 - 1000	0.71	0.5	0.36
	1000 - 2×10^5	$0.35 (S_T/S_L)^{0.2}$	0.6	0.36
	2×10^5 - 2×10^6	$0.031 (S_T/S_L)^{0.2}$	0.8	0.36

Table 2 Correction Factor F for $N_L < 16$

N_L	1	2	3	4	5	7	10	13
In-line	0.70	0.80	0.86	0.90	0.93	0.96	0.98	0.99
Staggered	0.64	0.76	0.84	0.89	0.93	0.96	0.98	0.99

Pressure Loss of Flow Across Tube Banks

The pressure loss of the external flow across the tube bank is calculated according to a method by Gaddis [4]. The equations are based on a superposition of equations of Bergelin et al. [7] for laminar flow in a modified form and the equations of Gaddis and Gnielinski [8] for turbulent flows. It is a flexible method which allows for inline and staggered tube configurations as well as variations in the transverse and longitudinal tube pitches. It calculates the ideal flow drag coefficients for the configuration and then corrects these coefficients for variations in temperature and number of rows. Tube bank arrangements and geometric parameters are shown in Figure 7.

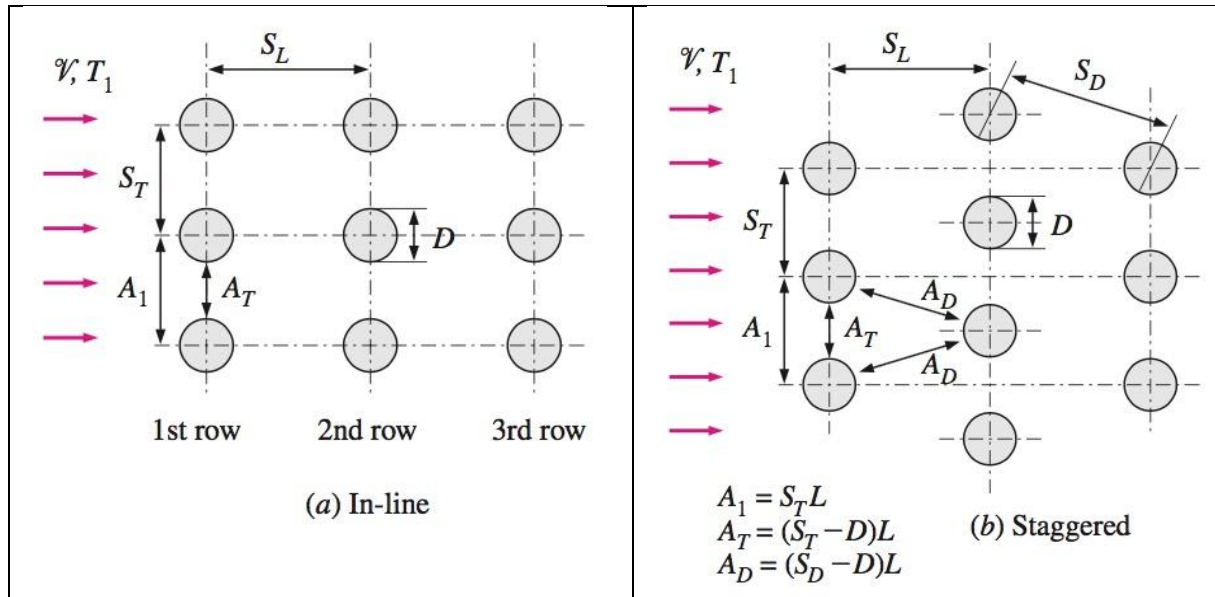


Figure 7: Inline and staggered tube bundle arrangements and dimensions.

The pressure loss across a tube bank is calculated as follows:

$$\Delta P = C_d N_L \frac{1}{2} \rho V_{max}^2 \quad \text{Eq. 3}$$

where:

C_d is the drag coefficient.

N_L is the number of tube rows in the external flow direction.

ρ is the mean density of the external fluid, kg/m^3 .

V_{max} is the maximum fluid velocity between the tubes, i.e. the velocity between the narrowest cross section of the tubes. Note that this could be either A_T or A_D in Figure 7 above, m/s.

All properties are taken at mean conditions, for example, $\rho = (\rho_{in} + \rho_{out})/2$, except where properties are to be taken at the tube outside surface temperature T_s , for example μ_s .

The drag coefficient is a function of the Reynolds number, Re , the transverse pitch ratio a and the longitudinal pitch ratio b , where:

$$Re = \frac{\rho V_{max} D}{\mu} \quad \text{Eq. 4}$$

$$\begin{aligned} a &= S_T/D \\ b &= S_L/D \\ c &= S_D/D \end{aligned} \quad \text{Eq. 5}$$

$$S_D = \sqrt{S_L^2 + \left(\frac{S_T}{2}\right)^2} \quad \text{Eq. 6}$$

S_T , S_L and S_D are the transverse, longitudinal and diagonal tube pitches respectively, in m. To calculate the maximum velocity between the tubes, the minimum cross sectional flow area must be determined:

For inline tube banks:

$$V_{max.i} = \frac{S_T}{S_T - D} \cdot V_{shell.in} \quad \text{Eq. 7}$$

where $V_{shell.in}$ is the shell-side fluid velocity upstream of the tube bank, in m/s.

For staggered tube banks the minimum flow area could be either A_T or A_D as shown in Figure 7. When A_D presents the smallest cross-sectional flow area:

$$V_{max.s} = \frac{S_T}{2(S_D - D)} \cdot V_{shell.in} \quad \left| \quad S_D < \frac{S_T + D}{2} \quad \right| \quad \text{Minimum flow area is } A_D \quad \text{Eq. 8}$$

When A_T presents the smallest cross-sectional flow area (similar to an inline tube arrangement):

$$V_{max.s} = \frac{S_T}{(S_T - D)} \cdot V_{shell.in} \quad \left| \quad S_D \geq \frac{S_T + D}{2} \quad \right| \quad \text{Minimum flow area is } A_T \quad \text{Eq. 9}$$

The shell-side upstream velocity is calculated as follows:

$$V_{shell.in} = \frac{\dot{m}}{\rho A_{shell}} \quad \text{Eq. 10}$$

where:

$\dot{m}$ is the shell side fluid mass flow rate, kg/s.

A_{shell} is the shell side cross sectional area upstream of the tube bank, m².

Definition of an Ideal Tube Bundle

An ideal tube bundle is defined arbitrarily as follows: Physical properties of the fluid is independent of temperature, number of tube rows $N_L \geq 10$, number of tubes per row ≥ 10 , ratio of tube length to tube diameter ≥ 10 , fluid velocity in the free cross section at inlet of tube bundle is uniform and perpendicular to the free cross section and smooth tube surface. Deviations from the ideal situation can be taken into account by means of correction factors.

The drag coefficient C_d for an ideal tube bundle consists of laminar and turbulent components:

$$C_d = C_{d,l} + C_{d,t} \cdot F \quad \text{Eq. 11}$$

where:

$C_{d,l}$ is the laminar component of the drag coefficient

$C_{d,t}$ is the turbulent component of the drag coefficient

F is a Reynolds number dependent correction factor for the turbulent component of the drag coefficient

For inline tube banks, F , $C_{d,l}$ and $C_{d,t}$ are calculated as follows [8]:

$$F_i = 1 - e^{-(Re+1000)/2000} \quad \text{Eq. 12}$$

$$C_{d,l,i} = \frac{f_{l,i}}{Re} \quad \text{Eq. 13}$$

$$C_{d,t,i} = \frac{f_{t,i}}{Re^{0.1(b/a)}} \quad \text{Eq. 14}$$

$$f_{l,i} = \frac{280 \pi [(b^{0.5} - 0.6)^2 + 0.75]}{(4 a b - \pi) a^{1.6}} \quad \text{Eq. 15}$$

$$f_{t,i} = \left[0.22 + 1.2 \frac{(1 - 0.94/b)^{0.6}}{(a - 0.85)^{1.3}} \right] \times 10^{0.47((b/a)-1.5)} + 0.03(a - 1)(b - 1) \quad \text{Eq. 16}$$

where f is a friction factor in the above equations.

For staggered tube banks, F , $C_{d,l}$ and $C_{d,t}$ are calculated as follows [8]:

$$F_s = 1 - e^{-(Re+200)/1000} \quad \text{Eq. 17}$$

$$C_{d,l,s} = \frac{f_{l,s}}{Re} \quad \text{Eq. 18}$$

$$C_{d,t,s} = \frac{f_{t,s}}{Re^{0.25}} \quad \text{Eq. 19}$$

$$f_{l,s} = f_{l,i} (a/c)^{1.6} \quad \left| \quad S_D < \frac{S_T + D}{2} \quad \right| \quad \text{Minimum flow area is } A_D \quad \text{Eq. 20}$$

$$f_{l,s} = f_{l,i} \quad \left| \quad S_D \geq \frac{S_T + D}{2} \quad \right| \quad \text{Minimum flow area is } A_T \quad \text{Eq. 21}$$

$$f_{t,s} = 2.5 + \left(\frac{1.2}{(a - 0.85)^{1.08}} \right) + 0.4 \left(\frac{b}{a} - 1 \right)^3 - 0.01 \left(\frac{a}{b} - 1 \right)^3 \quad \text{Eq. 22}$$

Temperature correction for non-isothermal (non-ideal) flow is applied to the drag coefficients calculated above, as heating or cooling of the tube bundle will lead to changes of the fluid properties in the thermal layer near the tube surface. The drag coefficients are multiplied by these temperature correction factors to account for the temperature influence. For the laminar drag coefficient, the non-isothermal correction factor is [7]:

$$f_{z.l} = \left(\frac{\mu_s}{\mu} \right)^{x_1} \quad \text{Eq. 23}$$

where:

$$x_1 = \frac{0.57}{\left(\left(\frac{4ab}{\pi} - 1 \right) Re \right)^{0.25}} \quad \text{Eq. 24}$$

For the turbulent drag coefficient, the non-isothermal correction factor may be calculated according to Sieder and Tate [9]:

$$f_{z.t} = \left(\frac{\mu_s}{\mu} \right)^{0.14} \quad \text{Eq. 25}$$

Correction factors for small number of tube rows (less than ten rows) for the drag coefficients are required according to Bergelin et al. [7]. For the laminar drag coefficient, the correction factor is very similar to the correction factor for temperature correction as given above.

Therefore, the laminar temperature correction and small number of rows correction factors may be combined, and hence:

$$f_{zn.l} = \left(\frac{\mu_s}{\mu} \right)^{x_1 x_2} \quad \text{Eq. 26}$$

where:

$$\begin{array}{l|l|l} x_2 = \left(\frac{N_L}{10} \right)^{0.25} & N_L < 10 & \text{Small number of tube rows} \\ x_2 = 1 & N_L \geq 10 & \text{Large number of tube rows} \end{array} \quad \text{Eq. 27}$$

where:

$f_{zn.l}$ is the laminar component of the correction factor for the combined effect of the number of tubes rows less than ten as well as correction for temperature.

Note that Eq. 26 and Eq. 27 effectively replace Eq. 23 and Eq. 24.

In the turbulent flow range, the change in the drag coefficient C_{dt} for a tube bundle with change in the number of tube rows between 5 and 10 is small [10] and may be ignored. However, sudden expansion of the flow area from that of the last tube row to the total available free area in the tube bundle leads to an additional pressure drop, which is negligible for a tube bundle with a large number of tube rows but may be significant in a tube bundle with a small number of tube rows. This may be compensated for by adding a correction factor f_{nt} to the drag coefficient C_{dt} in case of an isothermal flow, or to the corrected drag coefficient $f_{zt} C_{dt}$ in the case of a non-isothermal flow [8]:

$f_{n,t} = 0$	$N_L \geq 10$	Large number of tube rows.	Eq. 28
$f_{n,t} = C_{d,0} \left(\frac{1}{N_L} - \frac{1}{10} \right)$	$N_L < 10$	Small number of tube rows.	
$C_{d,0} = \left(\frac{2(c-1)}{a(a-1)} \right)^2$	$S_D < \frac{S_T + D}{2}$	Staggered tube arrangement with A_D the minimum flow area.	Eq. 29
$C_{d,0} = \frac{1}{a^2}$	$S_D \geq \frac{S_T + D}{2}$	Inline or staggered tube arrangement with A_T the minimum flow area.	

General Equation for the Drag Coefficient

The general equation for the drag coefficient for a non-isothermal (non-ideal) tube bundle, as is the case in a heat exchanger, must incorporate the temperature correction factor as well as the correction factor for the number of tube rows. Therefore, Eq. 11 is adapted as follows:

$$C_d = C_{d,l} \cdot f_{zn,l} + (C_{d,t} \cdot f_{z,t} + f_{n,t}) \cdot F \quad \text{Eq. 30}$$

Note that Eq. 30 applies to both inline and staggered tube bank configurations. As such, for inline tube banks Eq. 30 should be written using the variables $C_{d,i}$, $C_{d,li}$, $C_{d,ti}$ and F_i . Similarly, for a staggered arrangement the variables are $C_{d,s}$, $C_{d,ls}$, $C_{d,ts}$ and F_i .

The Modelling of Fouling Factors

In an earlier case study by this author, the modeling of fouling factors was discussed in detail. In this section, only a brief discussion on this topic is repeated in the interest of its implementation in this case study.

Heat transfer is often calculated in terms of a thermal resistance and a temperature difference as follows:

$$\dot{Q} = \frac{\Delta T_{ln}}{R} \quad \text{Eq. 31}$$

where:

R is the thermal resistance [K/W].

Fouling factors are commonly written in terms of thermal resistances and are simply added in series to the other thermal resistance components in the heat transfer problem. The following equation represents the total thermal resistance for heat transfer between two fluids separated by a tube where fouling exists on the inside and outside surfaces:

$$R = \frac{1}{h_i A_i} + \frac{R_{fi}}{A_i} + \frac{\ln(D_o/D_i)}{2 \pi k L} + \frac{R_{fo}}{A_o} + \frac{1}{h_o A_o} \quad \text{Eq. 32}$$

where:

h_i and h_o are the inside and outside surface convection coefficients [W/m².K].

A_i and A_o are the inside and outside surface areas subjected to convection heat transfer [m²].

R_{fi} and R_{fo} are the inside and outside surface fouling factors respectively [$\text{m}^2 \cdot \text{K} / \text{W}$].

D_i and D_o are the inside and outside pipe diameters [m].

k is the pipe material thermal conductivity [W/m.K].

L is the length of the pipe [m].

As mentioned earlier in this document, the user can specify fouling in terms of a resistance, the conductivity of the foulant (fouling material) if it is known and the thickness of the foulant if known. Flownex[®], however, does not offer the capacity to specify fouling in these terms, and as was discussed in the earlier case study, one way to implement fouling is to model a Flownex[®] pipe element with three layers in the Composite Heat Transfer (CHT) element. The first layer represents the tube outside surface fouling layer, the second layer represents the tube wall material while the third layer represents the tube inside surface fouling layer. To enable this layered model to work, the fouling layer thermal conductivity, thermal capacitance and layer thickness must be specified. Under steady state modeling, the thermal capacitance has no influence, thus the required Flownex[®] inputs may be obtained from rearranging Eq. 32.

With a known fouling factor as specified, together with a known foulant, and thus a known or estimated thermal conductivity for the fouling layer, the fouling layer thickness can be calculated. This can be done by equating the fouling resistance in Eq. 32 to the resistance presented by an equivalent solid cylindrical fouling layer:

$$\frac{R_f}{A} = \frac{\ln(D_o/D_i)}{2 \pi k L} \quad \text{Eq. 33}$$

Noting that the diameters in the equation above relate to fouling layer and not the pipe or tube wall in this instance, they may be written in terms of the fouling layer thickness t . Inside and outside fouling thickness may then be written as:

$$\frac{R_{fi}}{\pi D_i L} = \frac{\ln\left(\frac{D_i}{D_i - 2t_i}\right)}{2 \pi k_i L} \quad \text{Eq. 34}$$

and similarly:

$$\frac{R_{fo}}{\pi D_o L} = \frac{\ln\left(\frac{D_o + 2t_o}{D_o}\right)}{2 \pi k_o L} \quad \text{Eq. 35}$$

where:

t_i is the inside fouling layer thickness.

t_o is the outside fouling layer thickness.

Solving for the fouling thicknesses when the thermal conductivities are known, it can be shown that:

$$t_i = \frac{D_i}{2} \left[1 - e^{-(2k_i R_{fi}/D_i)} \right] \quad \text{Eq. 36}$$

$$t_o = \frac{D_o}{2} \left[e^{(2k_o R_{fo}/D_o)} - 1 \right] \quad \text{Eq. 37}$$

Similarly, solving for the foulant thermal conductivities when the fouling layer thicknesses are known, it can be shown that:

$$k_i = \frac{D_i}{2 R_{fi}} \ln \left(\frac{D_i}{D_i - 2t_i} \right) \quad \text{Eq. 38}$$

$$k_o = \frac{D_o}{2 R_{fo}} \ln \left(\frac{D_o + 2t_o}{D_o} \right) \quad \text{Eq. 39}$$

For typical fouling factors used in industry, please refer to the previous case study on fouling factors.

Calculating the Corrected Log-Mean-Temperature-Difference (LMTD)

The heat transfer achieved by a shell-and-tube or cross-flow heat exchanger is often expressed in terms of the following:

$$\dot{Q} = U_o A_o \Delta T_{ln} F \quad \text{Eq. 40}$$

where:

U_o is the overall heat transfer coefficient, [W/m².K].

A_o is the total tube outside surface area, [m²].

ΔT_{ln} is the log-mean temperature difference [LMTD], [K].

F is the LMTD correction factor to correct for the difference between the actual heat exchanger configuration (cross-flow) to an ideal theoretical counter-flow heat exchanger.

The LMTD for a heat exchanger is calculated as:

$$\Delta T_{ln} = \frac{\Delta T_1 - \Delta T_2}{\ln \left(\frac{\Delta T_1}{\Delta T_2} \right)} \quad \text{Eq. 41}$$

where:

$$\Delta T_1 = T_1 - t_2 \quad \text{Eq. 42}$$

$$\Delta T_2 = T_2 - t_1 \quad \text{Eq. 43}$$

T is the shell-side temperature, t is the tube-side temperature, and the subscripts **1** and **2** refer to inlet and outlet conditions respectively.

In determining the correction factor F , the following temperature ratios are commonly used in heat exchanger design:

$$P = \frac{t_2 - t_1}{T_1 - t_1} \quad \text{Eq. 44}$$

$$R = \frac{T_1 - T_2}{t_2 - t_1} \quad \text{Eq. 45}$$

These two parameters are typically used to read the correction factor F from sets of curves that may be found in most heat transfer textbooks. For the single-pass cross-flow heat exchanger presented in this case study however, the following equations (obtained from the CHE Guide, [11]) may be used to determine the correction factor F :

$$P = \frac{1 - e^{-K \cdot R}}{R} \quad \text{Eq. 46}$$

Transposing this equation:

$$K = \frac{\ln(1 - P \cdot R)}{-R} \quad \text{Eq. 47}$$

The number of transfer units **NTU** is used in the well-known effectiveness-NTU calculation method and is related to the factor **K** above as follows:

$$K = 1 - e^{-NTU} \quad \text{Eq. 48}$$

Transposing this equation:

$$NTU = -\ln(1 - K) \quad \text{Eq. 49}$$

The correction factor **F** for a single-pass cross-flow heat exchanger may then be calculated from the following:

$$F = \frac{P}{NTU(1 - P)} \quad \text{when } R = 1. \quad \text{Eq. 50}$$

$$F = \frac{1}{NTU(1 - R)} \cdot \ln\left(\frac{1 - P \cdot R}{1 - P}\right) \quad \text{for all other cases.} \quad \text{Eq. 51}$$

CASE STUDIES

This section compares the results of three case studies using the Flownex[®] model to equivalent calculations using Aspen EDR. The cases are:

- A single pass air-air recuperator with no fouling.
- The same model as above, but with fouling specified and the bundle divided into two sections.
- A multi-pass multi-fluid recuperator extracting heat from flue gas, heating two tube-side streams; air and CO₂.

Case Study 1: A Single Pass Flue-Gas-To-Air Recuperator With No Fouling

The Flownex[®] model of a single pass flue-gas-to-air recuperator is shown in Figure 8. The Flownex[®] compound component offers four fluid connections, two for the flue gas stream and two for the air stream. In the case presented, the pressures are specified at the outlet boundaries while the temperatures and flow rates are specified at the inlet boundaries. For this case study, the input data is presented in Table 3 below.

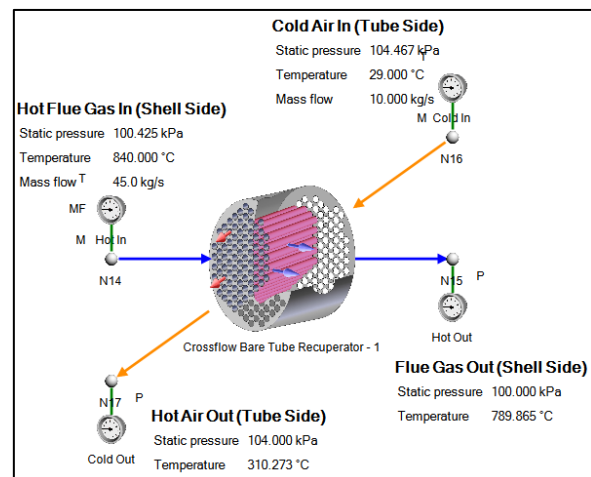


Figure 8: Flownex model of a single pass flue-gas-to-air recuperator.

Table 3: Case Study 1 Input Data

Property	Unit	Value
Tube-side fluid	Air	
Shell-side fluid	Flue gas	
Tube material	316 Stainless Steel	
Tube diameter (ID)	mm	40.39
Tube wall thickness	mm	2.03
Tube length	m	2.5
Number of tubes per row		43
Number of tube rows in bundle		11
Longitudinal tube pitch	mm	80
Transverse tube pitch	mm	80
Shell-tube clearance	mm	0
Tube layout geometry option	Inline	
Tube fouling options (inside/outside)	No fouling	
Thermal design margin of safety	%	0
Tube material	316 SS	
Tube design standard	ASME B31.3	
Tube design pressure	kPa-g	800
Tube design temperature	°C	800
Tube corrosion allowance	mm	0
Tube under-tolerance	%	12.5
Tube type (for weld quality)	Seamless	

The Flownex® results of this case are shown in Figure 5 and are compared with results from a calculation using Aspen Exchanger Design & Rating (EDR) V9 and summarized in the following table. Good agreement is shown between the Flownex® and Aspen results.

Table 4: Case Study 1 Results and Comparison with Aspen EDR

Result	Unit	Flownex®	Aspen EDR
Total heat load	kW	2890	2855
Overall heat transfer coefficient U_o	W/m ² K	27.6	27.6
Heat transfer area	m ²	165.1	165.1
Effective (corrected) LMTD	°C	633.8	625.4
Total number of tubes		473	473
Maximum outside tube wall temperature	°C	717.7	711.9
Average outside tube wall temperature	°C	693.7	692.8
Maximum inside tube wall temperature	°C	716.2	710.0
Average inside tube wall temperature	°C	692.2	690.7
Tube-side pressure loss	kPa	0.362	0.338
Shell-side pressure loss	kPa	0.445	0.482

Case Study 2: A Single Pass Flue-Gas-To-Air Recuperator Split Into Two Sections and with Fouling

The model presented in Figure 9 below is essentially the same as presented in Case Study 1 in Figure 8. For this case however, the tube bundle is split into two sections; the first section representing the first four tube rows, and the second section representing the remaining seven tube rows. This allows the maximum tube wall temperatures for the shield tube rows to be calculated more accurately by not averaging them with all the tubes in the bundle. Furthermore, inside and outside fouling factors of $0.0002 \text{ m}^2\cdot\text{K}/\text{W}$ have also been applied to this case.

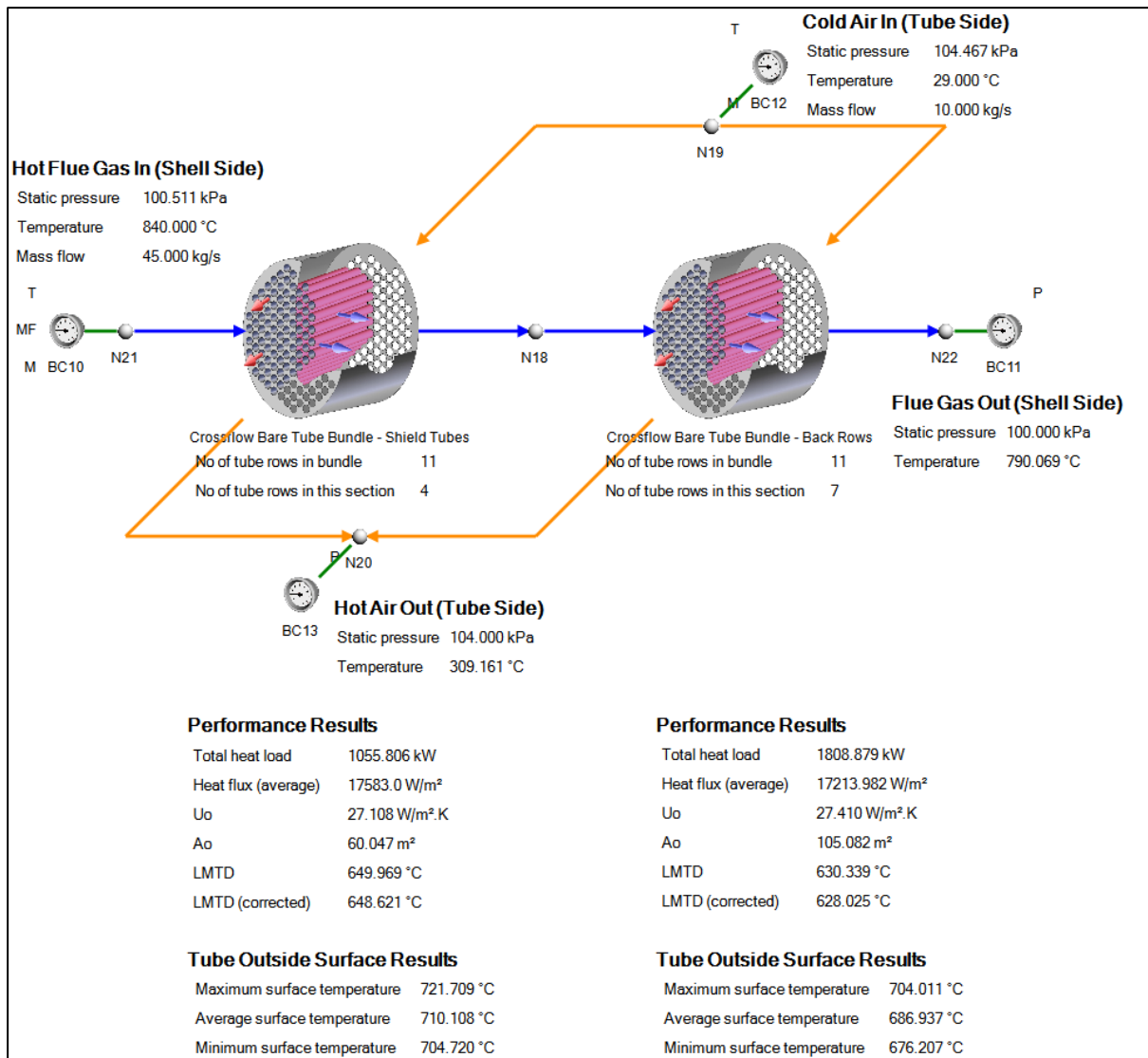


Figure 9: Flownex® model of a single pass flue-gas-to-air recuperator split into two sections.

The Flownex® results for this case are shown in Table 5 and are compared with results from Case Study 1.

Table 5: Case Study 2 Results and Comparison with Case Study 1 and Aspen EDR.

Result	Unit	Case 2	Case 1	Aspen EDR
Total heat load	kW	1056 / 1809	2890	2855
Overall heat transfer coefficient U_o	W/m ² K	27.1 / 27.4	27.6	27.4
Heat transfer area	m ²	60.05 / 105.08	165.1	165.1
Effective (corrected) LMTD	°C	648.6 / 628.0	633.8	625.5
Total number of tubes		172 / 301	473	473
Maximum outside tube wall temperature	°C	721.7 / 704.0	717.7	713.2
Average outside tube wall temperature	°C	710.1 / 686.9	693.7	690.5
Maximum inside tube wall temperature	°C	713.1 / 695.5	716.2	708.1
Average inside tube wall temperature	°C	701.2 / 678.2	692.2	688.4
Tube-side pressure loss	kPa	0.361 / 0.361	0.362	0.338
Shell-side pressure loss	kPa	0.189 / 0.292	0.445	0.482

- The total heat load for Case Study 2 will be lower than for Case Study 1 due to the fouling added to the tubes. This is also reflected in the slightly lower overall heat transfer coefficient for Case Study 2. It is curious, however, that with the application of the inside and outside fouling factors Aspen EDR did not reflect any significant reduction in heat load. The Flownex® model shows an 25 kW heat load reduction.
- The effective LMTD is higher for the shield tubes and lower for the remaining seven rows than was calculated in Case Study 1. For the non-shield tubes, the calculated effective LMTD is in close agreement with the results obtained for Aspen EDR.
- The maximum and average tube temperatures are better differentiated in this case study and results are still in good agreement with Aspen EDR.

Case Study 3: A Multi-Pass Multi-Fluid Recuperator

The first two components of the model presented in Figure 10 below is essentially the same as presented in Case Study 2 in Figure 9. The third component represents the inlet pass of a U-bundle (refer to Figure 1) flue-gas-to-air recuperator. The last two components represent a separate U-bundle for a flue-gas-to-CO₂ recuperator. The model shows the flexibility of the *Cross-Flow Bare Tube Recuperator* compound component presented in this case study where the compound components may be connected in series on the flue gas side and where multiple tube-side fluids are accommodated logically and straightforwardly. Since the outlet of each bundle serves as the inlet for the next, changes in the design of upstream bundles are automatically reflected in the performance of downstream bundles.

Figure 10 also presents most of the bundle inputs and results on the canvas in a convenient layout. This offers the design engineer a single overview of all significant results side-by-side and enables immediate visual feedback of any design changes. The flexibility of the Flownex® model to configure a results layout on the canvas goes beyond the capabilities of most other commercial software such as Aspen EDR.

When modeling of a four-pass recuperator is required, for example, four bundles are simply connected in series for both the shell (flue gas) side as well as the tube side. Furthermore, connections between the tube passes can be accurately modeled, whether return bends or manifold headers or any other configuration. As shown in Figure 10 below, each U-bundle

employs return bends for each U-tube. Note that the return bend element used is of the same dimensions as the tubes it connects and is specified as being the same number in parallel as the total tube count of the bundle.

For a similar model in Aspen EDR, two separate projects, each modeling one bundle with its own tube-side fluid, will have to be created. The design process will then become very iterative as any changes in the upstream bundle's design or operating conditions will result in a different flue gas temperature for the downstream bundle. This temperature must then be manually transferred between the two separate projects.

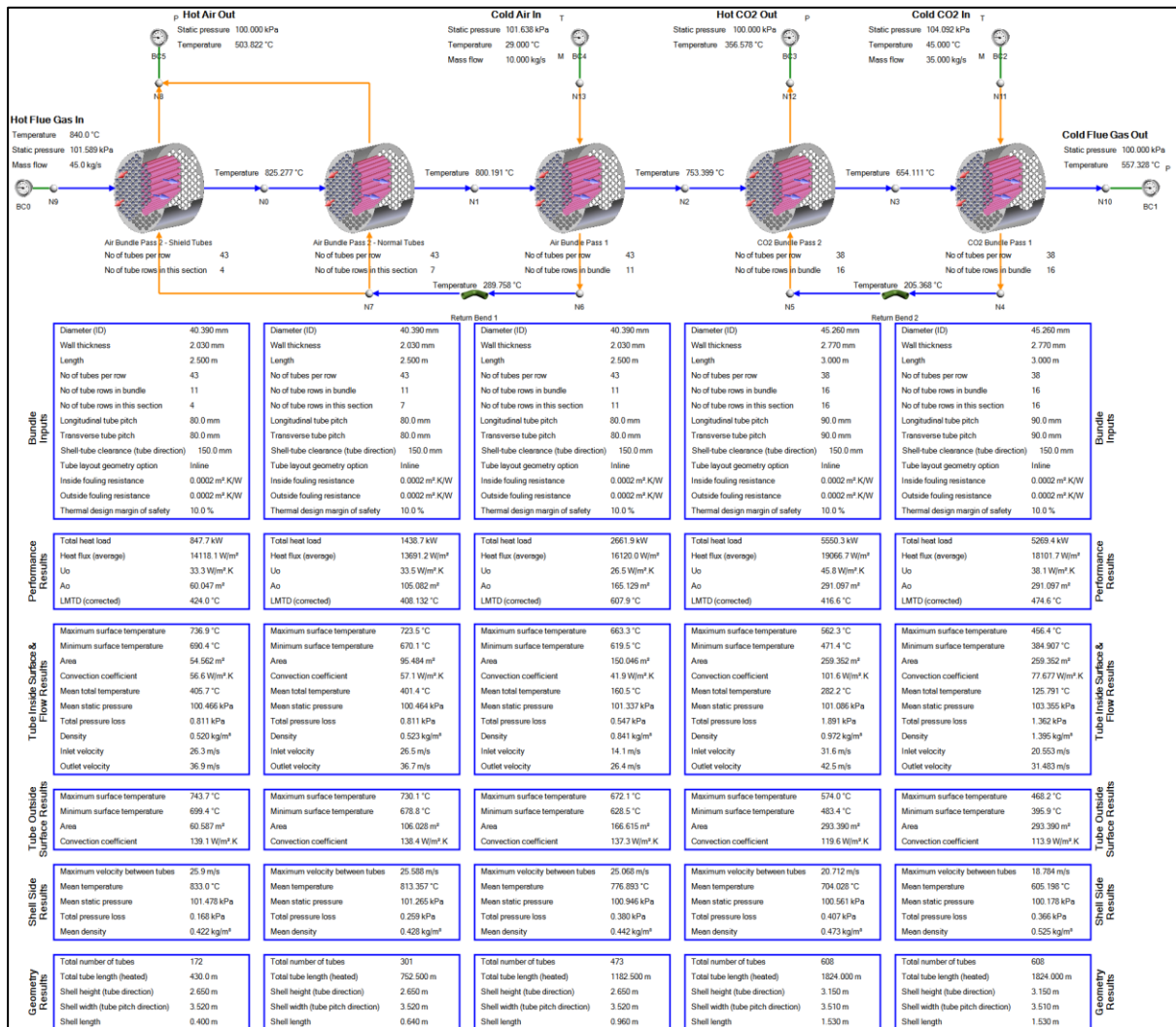


Figure 10: Flownex® model of a multi-pass multi-fluid flue gas recuperator.

Summary

Using a relatively straightforward Flownex® compound component, a comprehensive flue gas recuperator model could be constructed. It was shown that the results are in close agreement with other commercial software such as Aspen EDR. However its flexibility goes beyond the capabilities of most commercial software. It was shown that any number of bundles could be connected in virtually any configuration, such as complex combinations of parallel and series, while flowing multiple fluids tube-side. Furthermore, the model could be extended by the user with relative ease to incorporate more capabilities such as finned surfaces or two-phase fluids.

References

- [1] "Pressure Pipe Wall Thickness and Flange Rating Calculation Using a Script and a Generic 4D Chart", case study available on the Flownex® website at:
<https://flownex.com/case-studies/pressure-piping-thickness-and-flange-rating-calculation/>
- [2] Zukauskas, A (1987), "Heat Transfer from Tubes in Cross Flow." In *Handbook of Single Phase Convective Heat Transfer*, Eds. S. Kakac, R.K. Shah, and Win Aung. New York: Wiley Interscience.
- [3] "Fouling Factors in Flownex® Heat Transfer Models", case study available on the Flownex® website at:
<https://flownex.com/case-studies/fouling-factors-in-flownex-heat-transfer-models/>
- [4] Gaddis E.S. (2010), "Pressure Drop of Tube Bundles in Cross Flow", VDI Heat Atlas (2010), section L1.4.
- [5] Gnielinski, V. (1976), "New Equations for Heat and Mass Transfer in Turbulent Pipe and Channel Flow", *International Chemical Engineering* 16, 1976, pp. 359-368.
- [6] Cengel, Y.A. (2006), "Heat and Mass Transfer, A Practical Approach", Third Ed, McGraw Hill, p.419.
- [7] Bergelin O.P., Colburn A.P. and Hull H.L. (1950), "Heat transfer and Pressure Drop During Viscous Flow Across Unbaffled Tube Banks.", University of Delaware, Engineering Experimental Station, Bulletin No. 2, Newark, Delaware.
- [8] Gaddis E.S. and Gnielinski V. (1983) Druckverlust in Querdurchstroemten Rohrbuendeln. vt "Verfahrenstechnik" 17(7):410–418.
- [9] Sieder E.N. and Tate G.E. (1936) Heat Transfer and Pressure Drop of Liquids in Tubes. *Indl Engg Chem* 28(12):1429–1435.
- [10] Scholz F (1968) Einfluss der Rohrreihenanzahl auf den Druckverlust und Waermeuebergang von Rohrbuendeln bei hohen Reynoldszahlen. *Chemie-Ing.-Tech.* 40(20):988–995.
- [11] CHE Guide (Chemical Engineer's Guide).
<https://cheguide.com/2016/09/lmtd-correction-factor-charts/>