Numerical Simulation of a Cross-Flow Heat Exchanger

Cristina Autuori Tomazeti (cristm@fem.unicamp.br)
Carlos Alberto Carrasco Altemani (altemani@fem.unicamp.br)
Energy Department
Mechanical Engineering Faculty
State University of Campinas
PO Box 6122
Zip Code 13083-970 – Campinas – SP Brazil

Abstract. *An essential parameter in the thermal analysis of heat exchangers is the global heat transfer coefficient. For some simple configurations, this coefficient may be obtained from literature correlations for the Nusselt number associated to the duct flows in the heat exchanger. For more complex heat exchanger configurations, the heat transfer coefficients have been measured by experiments and they may be found in the literature. For many other possible configurations, there is no information available in the literature about the friction and the convective coefficients for the flows inside the heat exchanger. One alternative to the experimental tests for each configuration is the numerical simulation of the flow and heat transfer. In this work numerical simulations of each duct flow of a cross-flow heat exchanger were performed to obtain the friction and the convective coefficients. The heat exchanger was assumed with unfinned rectangular ducts, so that the calculated coefficients could be compared with those from analytical correlations available in the literature, for both laminar and turbulent flow regimes. The numerically obtained heat transfer coefficients were employed in the ε-NTU method for the heat exchanger analysis. Since the considered heat exchanger geometry was very simple, numerical simulations were also performed for the entire heat exchanger, encompassing both flows and the separation plate. The heat exchanger results obtained from this new procedure were then compared with those from the previous procedure.*

Keywords: *cross-flow heat exchangers, global coefficient of heat transfer, numerical simulation, laminar flow, turbulent flow.*

1. Introduction

An ubiquitous equipment in many industries, a heat exchanger is an equipment used to promote the heat exchange between two or more fluid flows. Solid walls may separate the fluids or they may exchange heat by direct mixture. The radiators, condensers, economizers and evaporators are examples of heat exchangers.

The purpose of the thermal analysis of a heat exchanger is to determine the rate of heat transfer between the fluids and to evaluate their outlet temperatures. One essential parameter for this analysis is the global heat transfer coefficient (U). It is associated to the total thermal resistance between the hot and cold fluids in the heat exchanger, including the convective resistance of both flows and the conductive resistance of the separating walls. Thus, to obtain (U), the average convective coefficients (h_m) for each fluid flow in the heat exchanger must be known. Due to the complexity of some flow geometries and flow regimes in many heat exchangers, there are no available analytical results or correlations for the convective coefficients. This difficulty is usually solved by experiments in the heat exchangers ducts (Kays, 1955). This procedure however, is restricted only to the configurations tested in a laboratory.

This work was performed with the purpose to develop an alternative to these experimental measurements by means of numerical simulations of the flow and convective heat transfer in the heat exchanger ducts. The selected configuration was that of a cross-flow heat exchanger with unfinned rectangular ducts. This was due to both its simplicity and the availability of analytical results in the literature.

Two independent procedures were used to develop this work. First, the rectangular duct flows were simulated individually assuming isothermal surfaces at the duct walls. The numerical convective and friction coefficients were initially compared with correlations from the literature. Then they were employed with the effectiveness method (ε-NTU) to perform the heat exchanger thermal analysis. Second, the cross-flow of both fluids separated by a wall of the heat exchanger was also simulated. In this case, the imposed conditions for the simulations were the flow rates and the inlet temperature of both fluid flows. The results obtained from this simulation were the rate of heat transfer and the outlet temperatures of both fluids, which were compared with those of the first procedure.

Three distinct flow conditions were tested in the cross-flow heat exchanger rectangular ducts. Configuration "A" refers to laminar flow in both ducts. For configuration "B", the flow was turbulent in both ducts, and for configuration "C" the flow was laminar in one duct and turbulent in the other.

2. Heat Exchanger Thermal Model

Pairs of unfinned rectangular ducts constitute the considered heat exchanger. Air was assumed as the fluid in both ducts, flowing in crossed directions, as indicated in Fig. 1.

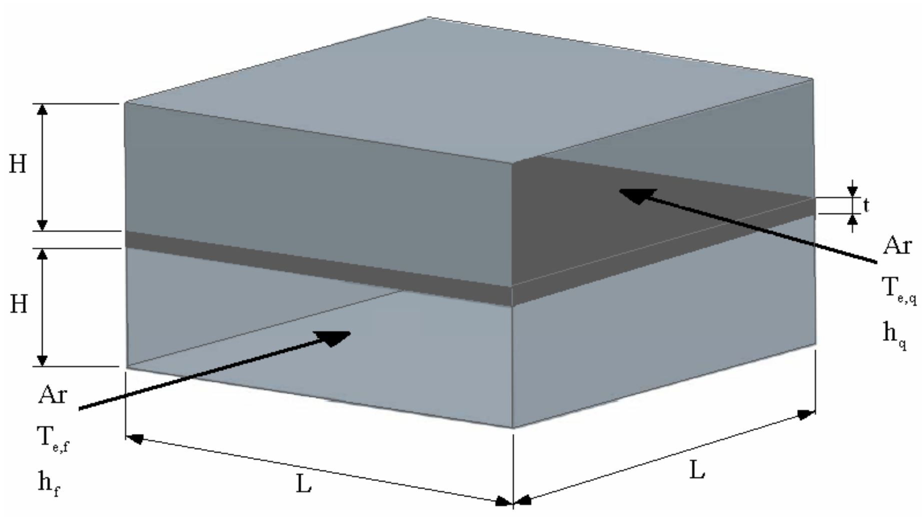

Figure 1 – The considered cross-flow heat exchanger

Neglecting any fouling effect on the ducts surfaces, the global heat transfer coefficient may be obtained from Eq. (1).

$$\frac{1}{UA} = \frac{1}{Ah_{mf}} + \frac{t}{k} + \frac{1}{Ah_{mq}} \tag{1}$$

Where U Global heat transfer coefficient $\left[W / m^2.K\right]$

A Heat transfer area at the fluids interface $[m^2]$

h_{mf} Mean convective coefficient, cold side $\left[W / m^2.K\right]$

h_{mq} Mean convective coefficient, hot side $\left[W / m^2.K\right]$

t Interface wall thickness $[m]$

k Interface wall thermal conductivity $[W / m.K]$

When the values of (h_{mf}) and (h_{mq}) are obtained, the value of (U) from Eq. (1) may be used in the thermal analysis of the heat exchanger through the effectiveness-NTU method (Incropera, 2002). The effectiveness is the ratio of the actual rate of heat exchange (q) to the theoretical maximum ($q_{máx}$).

$$\varepsilon = \frac{q}{q_{máx}} = \frac{C_q\left(T_{e,q} - T_{s,q}\right)}{C_{mín}\left(T_{e,q} - T_{e,f}\right)} = \frac{C_f\left(T_{s,f} - T_{e,f}\right)}{C_{mín}\left(T_{e,q} - T_{e,f}\right)} \tag{2}$$

Where C_q Hot fluid thermal capacity $[W / K]$

C_f Cold fluid thermal capacity $[W / K]$

$C_{mín}$ Minimum of C_q, C_f $[W / K]$

T Fluid temperature $[°C]$

For the fluid temperature, the subscripts “e” and “s” refer respectively to their inlet and outlet of the heat exchanger, while the subscripts “f” and “q” refer respectively to the cold and the hot fluids.

For a heat exchanger

$$\varepsilon = f(NUT, R) \tag{3}$$

In this equation, (R) indicates the ratio of the fluids thermal capacity (C_{min}/C_{max}) and NTU, known as the number of transfer units, is defined by

$$NUT = \frac{UA}{C_{mín}} \tag{4}$$

For a cross-flow heat exchanger with both fluids unmixed, the functional form of equation (3) may be expressed (Incropera, 2002) as

$$\varepsilon = 1 - \exp\left\{\frac{1}{R} NUT^{0,22} \left[\exp\left(-R\, NUT^{0,78}\right) - 1\right]\right\} \tag{5}$$

When the effectiveness is known, the fluids outlet temperatures and the heat exchanger rate of heat transfer may be obtained.

$$T_{s,q} = T_{e,q} - \frac{\varepsilon C_{mín}\left(T_{e,q} - T_{e,f}\right)}{C_q} \tag{6}$$

$$T_{s,f} = T_{e,f} + \frac{\varepsilon C_{mín}\left(T_{e,q} - T_{e,f}\right)}{C_f} \tag{7}$$

$$q = \varepsilon q_{máx} = \varepsilon C_{mín}\left(T_{e,q} - T_{e,f}\right) \tag{8}$$

3. Numerical Modeling

The flow and the convective heat transfer in each duct and also in the cross flow heat exchanger were simulated numerically by the method of the control volumes, using the software PHOENICS. In each case the conservation equations of mass, momentum and energy were solved assuming uniform velocity and temperature at the entrance of each rectangular duct. Since the flows encompassed the entrance region, the simulations were three-dimensional. As described previously, both laminar and turbulent flows were considered in the analysis. In the later case, the LVEL turbulence model (Agonafer, Liao and Splading, 1996) included in the PHOENICS was used.

The convective coefficients (h_m) in each rectangular duct flow were evaluated assuming a uniform wall temperature equal to the inlet temperature of the other fluid. Due to symmetry, the simulated domain comprised only a quarter of the duct cross-section, as indicated by the dashed line in Fig. 2 (a).

For the simulation of the cross-flow heat exchanger, the adopted symmetry was that due to a heat exchanger with several parallel plates. In this case the flow in one duct exchanges heat symmetrically with both neighbor plates. Due to this symmetry, the calculation domain for these simulations was that indicated by dashed lines in Fig. 2 (b), encompassing half height of each duct and the separation plate.

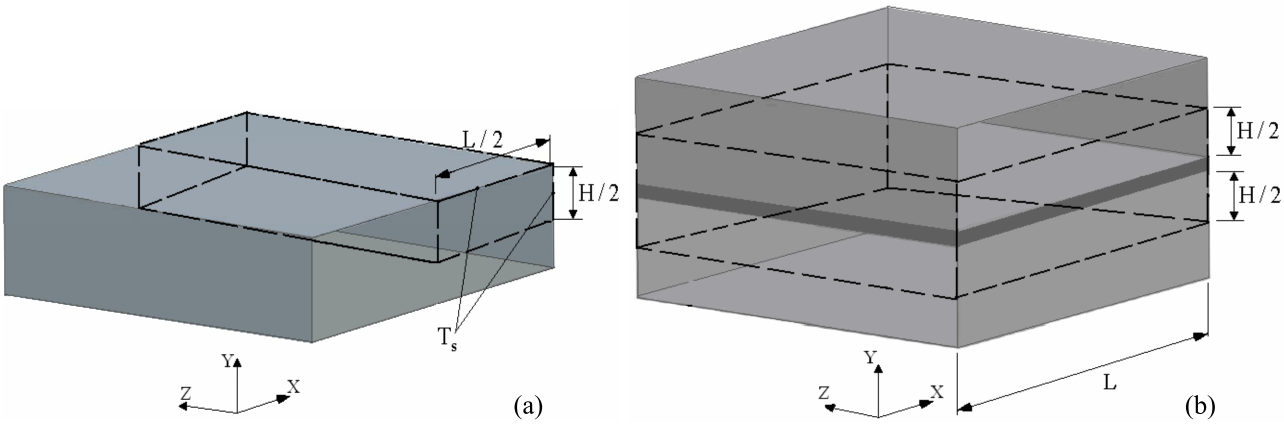

Figure 2 - Numerical domain (a) for each duct (b) for the heat exchanger

4. Results and discussion

Air was assumed as the flowing fluid in both heat exchanger ducts and the separating plates material was Aluminum. With reference to Fig. 1, the dimensions employed are indicated in Table 1.

Table 1. Characteristics of heat exchanges simulated.

$L\ [mm]$	$H\ [mm]$	$t\ [mm]$
200	5	1
Work fluid: Air Plate Materials: Aluminum $(k = 204\ W/m.K)$		

In the first calculation procedure, the values of (h_m) were obtained from the numerical simulations of the hot and cold flows in each rectangular duct. The computational mesh associated to the domain indicated in Fig. 2 (a) is presented in Table 2.

Table 2. Grid used on simulation of each channel separately.

Ducts	**Laminar Flow**			**Turbulent Flow**		
	X	**Y**	**Z**	**X**	**Y**	**Z**
Duct 1 - (Cold Fluid)	19	19	38	19	70	38
Duct 2 - (Hot Fluid)	19	19	38	19	70	38

The computational mesh used for the heat exchanger domain indicated in Fig. 2 (b) is presented in Table 3. The numbers presented in Tables 2 and 3 correspond to the number of control volumes used in the calculation domains, in order to obtain grid independence results.

Table 3. Numerical Grid for the Three Heat Exchanger Configurations.

Heat Exchanger	**Configuration A** Laminar - Laminar			**Configuration B** Turbulent - Turbulent			**Configuration C** Laminar - Turbulent		
	X	**Y**	**Z**	**X**	**Y**	**Z**	**X**	**Y**	**Z**
Duct 1 – Cold Fluid	38	19	38	38	70	38	38	19	38
Interface - Plate	38	2	38	38	2	38	38	2	38
Duct 2 – Hot Fluid	38	19	38	38	70	38	38	70	38

The results from the simulations for each duct with laminar flow are presented in Table 4. The variable (u) refers to the flow uniform entrance velocity in each duct and (Re) is the Reynolds number. The results include the average Nusselt number (Num), the average friction coefficient (f_m) and the duct flow pressure drop (ΔP). The values of (Re) and (Num) are based on the hydraulic diameter of each duct. Table 4 also includes results obtained from literature (Kays, 1993) using correlations valid for the fully developed regime and for the thermal entrance region of rectangular duct flows.

It is observed that the numerical values of the average convective coefficient (h_m) and the friction factor (f_m) are larger than those obtained from the literature for the fully developed flow and for the thermal entrance region. This is to be expected, since the velocity and thermal boundary layer development at the duct entrance was present only in the numerical simulations. The values of the velocity and the temperature at each duct inlet were chosen arbitrarily, considering only that the duct flows should be in the laminar regime. The fluid properties were evaluated at their mean bulk temperature between the entrance and outlet of the duct. This procedure demanded an iterative procedure, since the fluid outlet temperatures were not known. The differences in the presented values of (f_m) for the fully developed flow and for the thermal entrance region are due to distinct values of the bulk temperatures at the duct outlet for these two conditions.

Table 4 – Comparative Results when Both Duct Flows were Laminar.

Duct 1 – Cold Fluid				Duct 2 – Hot Fluid			
Variable	**Developed**	**Thermal Inlet Region**	**Numerical**	**Variable**	**Developed**	**Thermal Inlet Region**	**Numerical**
$u[m/s]$	2	2	2	$u[m/s]$	1	1	1
Re	1200	1193	1187	Re	551	553	554
$T_w[°C]$	60	60	60	$T_w[°C]$	20	20	20
$T_{e,f}[°C]$	20	20	20	$T_{e,q}[°C]$	60	60	60
$T_{s,f}[°C]$	39.4	41.3	43.2	$T_{s,q}[°C]$	29.4	28.6	27.8
Nu_m	7.1	8.1	9.1	Nu_m	7.1	7.5	8.0
$h_m[W/m^2.K]$	18.7	21.5	24.3	$h_m[W/m^2.K]$	19.6	20.8	22.2
f_m	0.01935	0.01946	0.02380	f_m	0.04210	0.04201	0.04640
$\Delta P[Pa]$	3.65	3.67	4.46	$\Delta P[Pa]$	1.89	1.89	2.09

The thermal analysis for the cross-flow heat exchanger corresponding to the duct flows considered in Table 4 was performed and the results are presented in Table 5. The convective coefficients presented in Table 4 were used in the effectiveness method to obtain the results presented in columns (1), (2) and (3) of Table 5. The results obtained from the simultaneous simulation of both cross-flows separated by the plate are presented in column (4). It is noticed that the values in columns (3) and (4) are very close to each other, the values of (q) and (U) are within 3 %, and the bulk temperature changes for both fluids were no more than 2 % from each other.

Table 5. Heat Exchanger Comparative Results - Configuration "A".

Configuration A Flow Laminar - Laminar	**(ε-NUT) Analysis**			**Numerical**
	Column (1)	**Column (2)**	**Column (3)**	**Column (4)**
	Developed	Thermal Inlet Region	Numerical	Numerical
$u_f[m/s]$	2	2	2	2
$u_q[m/s]$	1	1	1	1
Re_f	1240	1238	1236	1235
Re_q	533	536	536	537
$U[W/m^2.K]$	9.6	10.6	11.6	11.9
$T_{e,f}[°C]$	20	20	20	20
$T_{s,f}[°C]$	28.2	28.8	29.3	29.5
$T_{e,q}[°C]$	60	60	60	60
$T_{s,q}[°C]$	42.0	40.9	39.8	39.4
$q[W]$	19.5	20.7	21.9	22.3

The numerical results for the duct flows when both are in the turbulent regime are presented in Table 6. The values predicted by correlations presented in the literature (Kays, 1993) for fully developed turbulent flow are also indicated in Table 6. The value of (Num) was obtained from the Gnielinski correlation and the average friction factor (f_m) was evaluated from the Petukhov equation, as indicated next.

$$Nu_m = \frac{\frac{f_m}{2}\left(Re - 1000\right)Pr}{1 + 12{,}7\left(\frac{f_m}{2}\right)^{1/2}\left(Pr^{2/3} - 1\right)} \tag{9}$$

$$f_m = \frac{\left[0{,}79\ln\left(Re\right) - 1{,}64\right]^{-2}}{4} \tag{10}$$

Similarly to Table 4, the results in Table 6 show that the numerical values of the average convective coefficient (h_m) and the friction factor (f_m) are larger than the corresponding values obtained from the literature correlations. This difference is due to the simultaneous flow and temperature development at the duct entrance, which was considered only in the numerical simulations.

Table 6. Comparative Results for Both Ducts with Turbulent flow

Duct 1 – Cold Fluid			**Duct 2 – Hot Fluid**		
Variable	**Developed.**	**Numerical**	**Variable**	**Developed.**	**Numerical**
$u[m/s]$	20	20	$u[m/s]$	20	20
Re	12290	12182	Re	10474	10562
$T_w[°C]$	60	60	$T_w[°C]$	20	20
$T_{e,f}[°C]$	20	20	$T_{e,q}[°C]$	60	60
$T_{s,f}[°C]$	31.2	34.2	$T_{s,q}[°C]$	48.5	45.3
Nu_m	35.9	47.5	Nu_m	31.4	42.5
$h_m[W/m^2.K]$	94.0	125.1	$h_m[W/m^2.K]$	89.1	120.1
f_m	0.007434	0.008960	f_m	0.007769	0.009152
$\Delta P[Pa]$	142.2	170.6	$\Delta P[Pa]$	135.6	160.5

The results presented in Table 6 were used to perform the thermal analysis of the corresponding cross-flow heat exchanger with both flows in the turbulent regime (configuration “B”), presented in Table 7. In this table, columns (1) and (2) show the results obtained from the effectiveness method with the heat transfer coefficients (h_m) from Table 6. The results obtained from the direct numerical simulations of the cross-flow heat exchanger are presented in column (3). It is noticed again that the results in columns (2) and (3) are very close. The values obtained for (U) and (q) are within less than 3 % from each other, and the bulk temperature changes for both fluid flows are less than 4 % from each other.

Table 7. Heat Exchanger Comparative Results - Configuration “B”

Configuration B Flows Turbulent - Turbulent	**(ε-NUT) Analysis**		**Numerical**
	Column (1)	**Column (2)**	**Coluna (3)**
	Developed	Numerical	Numerical
u_f [m/s]	20	20	20
u_q [m/s]	20	20	20
Re_f	12514	12462	12456
Re_q	10313	10355	10359
U [W/m^2.K]	45,7	61,3	63,1
$T_{e,f}$ [°C]	20	20	20
$T_{s,f}$ [°C]	25,1	26,5	26,6
$T_{e,q}$ [°C]	60	60	60
$T_{s,q}$ [°C]	54,3	52,8	52,6
q [W]	120,5	153,4	157,0

The results obtained from the numerical simulations for each duct, one with laminar flow and the other in the turbulent regime – configuration “C” are presented in Table 8. The values obtained from literature (Kays, 1993) correlations for fully developed flows are also presented in Table 8. Again, the average convective coefficients (h_m) and friction factors (f_m) are larger the corresponding fully developed values, for the same reasons considered before.

Table 8. Comparative Results with Laminar and Turbulent Duct Flows

Duct 1 – Cold Fluid			**Duct 2 – Hot Fluid**		
Variable	**Developed.**	**Numerical**	**Variable**	**Developed.**	**Numerical**
u[m/s]	2	2	u[m/s]	45	45
Re	1200	1187	Re	23468	23539
T_w [°C]	60	60	T_w [°C]	20	20
$T_{e,f}$ [°C]	20	20	$T_{e,q}$ [°C]	60	60
$T_{s,f}$ [°C]	39.4	43.2	$T_{s,q}$ [°C]	50.0	48.9
Nu_m	7.1	9.1	Nu_m	59.1	67.3
h_m [W/m^2.K]	18.7	24.3	h_m [W/m^2.K]	168.0	191.0
f_m	0.01935	0.02380	f_m	0.006279	0.006665
ΔP[Pa]	3.65	4.46	ΔP[Pa]	553.4	588.5

The results for the heat exchanger considering the conditions presented in Table 8 and the effectiveness method are indicated in columns (1) and (2) of Table 9, corresponding to configuration “C”. The results obtained from the direct heat exchanger simulation are indicated in column (3). In columns (2) and (3) the values of (U) and (q) are within 4 % from each other. The cold fluid bulk temperature changes are within 5 % and the hot fluid temperature changes are within 0.1 °C.

Table 9. Heat Exchanger Comparative Results - Configuration “C”

Configuration C Flows Laminar - Turbulent	**(ε-NUT) Analysis**		**Numerical**
	Colunm (1)	**Colunm (2)**	**Colunm (3)**
	Developed	Numerical	Numerical
u_f [m/s]	2	2	2
u_q [m/s]	45	45	45
Re_f	1207	1195	1193
Re_q	22913	22922	22930
U $[W/m^2.K]$	16.8	21,6	23,2
$T_{e,f}$ [°C]	20	20	20
$T_{s,f}$ [°C]	37.4	40,8	41,8
$T_{e,q}$ [°C]	60	60	60
$T_{s,q}$ [°C]	59.2	59,0	58,9
q [W]	40.4	48,0	50,3

5. Conclusions

An analytical and numerical procedure was applied for the thermal analysis of a cross-flow heat exchanger with unfinned rectangular ducts under distinct flow regimes. The global heat transfer coefficient (U) was obtained by two procedures. First, the average convective coefficient (h_m) for each heat exchanger duct flow was obtained numerically, assuming a uniform wall temperature thermal boundary condition. These values were employed in the ε-NTU method of the heat exchanger thermal analysis. The results thus obtained included the global heat transfer coefficient and the heat transfer rate within the heat exchanger, as well as the fluids outlet temperatures. In the second procedure, the numerical simulations were performed for a symmetrical region of the cross-flow heat exchanger encompassing the cross-flows and the separation plate. In this case, the heat exchanger thermal analysis was obtained directly from the numerical simulations, without any need to apply the effectiveness method. The results obtained from both procedures compared very well. For example, the values of (U) and (q) were within 4 % from each other for the three flow configurations considered in the simulations.

A simple cross-flow heat exchanger with unfinned rectangular ducts was considered in this work due to the purpose to perform simulations not only for each single duct, but also for the entire heat exchanger, with a minimum computational effort. The results obtained indicated that the simulations for the single ducts assuming isothermal walls can be applied to predict the cross-flow heat exchanger performance through the effectiveness – NTU method. Next, this approach will be applied to cross-flow heat exchangers with more complex duct geometries.

6. Acknowledgements

The present work was performed with the support of CAPES (Coordination for the Improvement of Higher Education Personnel) in the form of a Scholarship to the first author.

7. References

Kays, W. M., 1993, ”Convective Heat And Mass Transfer”, Ed. MacGraw-Hill, 3ºEd.
Spalding, B.D., 1994, “The Phoenics Encyclopedia”.
Kays, W. M., London, A. L., 1955, ”Compact Heat Exchangers”, Ed. MacGraw-Hill, 2ºEd.
Incropera, F. P., Dewitt, D. P., 2002, “Fundamentos de Transferencia de Calor e de Massa”, Ed. John Wiley & Sons.
Burmeister L. C., 1982, “Convective Heat Transfer”, Ed. John Wiley & Sons.
Agonafer D., Gan-Li L., Spalding B., 1996, "The LVEL turbulence model for conjugate heat transfer at low Reynolds numbers", Application of CAE/CAD Electronic Systems, EEP ASME, vol. 18.
D. Post, X-DO, 2000, “Codes Written by the National and International Computational Physics Community”, http://www.highproductivity.org/026284coverCEEGcodes.pdf