Numerical investigation of cryogen re-gasification in a plate heat exchanger

Ziemowit Malecha, Paweł Pluszka, Arkadiusz Brenk
Wrocław University of Science and Technology, Department of Cryogenic, Aeronautic and Process Engineering, Wyb Wyspianskiego 27, 50-370 Wrocław, Poland
E-mail: ziemowit.malecha@pwr.edu.pl

Abstract. The efficient re-gasification of cryogen is a crucial process in many cryogenic installations. It is especially important in the case of LNG evaporators used in stationary and mobile applications (e.g. marine and land transport). Other gases, like nitrogen or argon can be obtained at highest purity after re-gasification from their liquid states. Plate heat exchangers (PHE) are characterized by a high efficiency. Application of PHE for liquid gas vaporization processes can be beneficial. PHE design and optimization can be significantly supported by numerical modelling. Such calculations are very challenging due to very high computational demands and complexity related to phase change modelling. In the present work, a simplified mathematical model of a two phase flow with phase change was introduced. To ensure fast calculations a simplified two-dimensional (2D) numerical model of a real PHE was developed. It was validated with experimental measurements and finally used for LNG re-gasification modelling. The proposed numerical model showed to be orders of magnitude faster than its full 3D original.

1. Introduction
Plate heat exchangers (PHE) are commonly known for their high effectiveness and compactness. PHEs are widely utilized in industry and air-conditioning systems. It has been stated that the evaporation heat transfer coefficient of refrigerant in the plates of PHE is much higher than in the case of heat exchangers with circular pipes. Additionally, a vapour quality is considerably easier to control in the case of PHE [1]. However, the advantages of using PHE are accompanied by the significantly enhanced complexity of the flow established in PHE. This is caused by the very complicated pattern of PHE channels.

Flow maldistribution was observed to commonly occur in PHE. This means that some parts of PHE are more active than others. This clearly influences the overall heat transfer of PHE and the evaporation processes. Heat transfer performance is dependent on both flow arrangement in the channels of PHE as well as outlet superheat conditions [2]. The authors of [1] report that various levels of superheating may have a significant influence on the characteristics of the evaporation processes.

As we know from PHE experimental studies that have been performed so far, it is necessary to conduct numerical investigations of fluid flow behaviour in PHE. CFD analysis could be a useful tool for investigating PHE functionality in re-gasification systems. The main challenge in the efficient calculation of flows with a phase change is found in the very high demand for high quality numerical mesh and the necessity of very small time steps. This makes it nearly
impossible to perform calculations in a full size PHE. A trustworthy and relatively fast numerical technique could save a significant amount of time and costs that are related to the geometrical optimization of PHE. This could be helpful in the reduction of undesirable effects, such as flow maldistribution.

2. Simplified numerical model of PHE

Typical PHE consists of several dozens of metal sheet plates. Set of the plates forms a complex pattern of channels for both fluids. The plates are not flat surfaces but shaped e.g. in a chevron pattern, to increase heat transfer area and enhance turbulence of the flow.

The work [1] shows the re-gasification of R410A fluid experiment conducted in a real PHE geometry. The geometry of one pair of plates used in this experiment is shown on the left side of the figure 1. It illustrates a complexity of a chevron plate design commonly used in PHE. The chevron pattern of plates introduces a necessity of a significant mesh refinement in near-wall regions. The right side of the figure 1 shows a cross-section of one of the channels and a minimal quality computational mesh needed for a trustworthy numerical modelling of a flow in the PHE.

Figure 1. Left: CAD model of a pair of PHE plates with chevron pattern; Right: zoomed cross-section of a single channel and numerical mesh.

The work [1] shows the re-gasification of R410A fluid experiment conducted in a real PHE geometry. The geometry of one pair of plates used in this experiment is shown on the left side of the figure 1. It illustrates a complexity of a chevron plate design commonly used in PHE. The chevron pattern of plates introduces a necessity of a significant mesh refinement in near-wall regions. The right side of the figure 1 shows a cross-section of one of the channels and a minimal quality computational mesh needed for a trustworthy numerical modelling of a flow in the PHE.

Figure 2 shows the simplified 2D model proposed in the current studies. Its dimensions and heat transfer area are correlated with the real geometry shown on the figure 1. It consists of one flow channel created by two neighbouring plates. The crucial dimensions are the duct width $W$, depth $D$ and longitudinal height of plates $L_{HT}$. In order to keep heat transfer area consistent with the real PHE, the third dimension $D$, has the same size as the real PHE. Even thou, the third dimension is defined, the numerical calculations are 2D because there is only one row of cells in the third dimension and all of the derivatives in this direction are set to zero.
Figure 2. 2D simplification of the 3D PHE shown on figure 1 and used in [1]

Figure 3. The 2D mesh used in the current calculations. The figure shows a small part of the domain in vertical direction and half of the channel in the width direction (from the wall to the channel center).

The figure 3 shows the numerical mesh used in the presented calculations. The size and structure of the mesh were chosen after performing a grid independence study. Its structure is consistent with the 3D mesh presented in figure 1 but it has only ~ 50000 computational cells.

A critical aspect of 3D to 2D transition was a preservation of the complex trajectory of a fluid caused by the chevron shape. It was achieved by introducing a number of flat, smooth and zero thickness mixing baffles inside the 2D flow channel. They were placed alternately on both plates as shown on figure 2. It was motivated by [5] where similar technique was used to enhance mixing. The figure 4 shows the influence of the mixing baffles on the evaporation process by comparing the computational results of the flow with and without mixing baffles.

3. Mathematical model and numerical correction
In the current study the Lee phase model was used [4]. Its main simplifying assumptions are: phase change occurs in constant pressure and a quasi-thermo equilibrium state, additionally a mass transfer mainly depends on saturation temperature $T_{sat}$. It is also assumed that interfacial temperature between phases is equal to $T_{sat}$.

In the Lee model appropriate rates of mass exchange between liquid and vapour phases are added to the continuity equations:

$$\frac{\partial (\rho \alpha v)}{\partial t} + \nabla (\rho u \alpha v) = R_{\text{evap}} - R_{\text{cond}}$$  \hspace{1cm} (1)

$$\frac{\partial (\rho l \alpha l)}{\partial t} + \nabla (\rho u \alpha l) = R_{\text{cond}} - R_{\text{evap}}$$  \hspace{1cm} (2)

where the rates of mass exchanged are defined as follows:

$$R_{\text{cond}} = C_c \rho_v \alpha_v (T_{sat} - T) \quad R_{\text{evap}} = 0 \quad T < T_{sat}$$  \hspace{1cm} (3)

$$R_{\text{evap}} = C_e \rho_l \alpha_l (T - T_{sat}) \quad R_{\text{cond}} = 0 \quad T > T_{sat}$$  \hspace{1cm} (4)
Figure 4. Comparison of the results for 2D geometry with and without the mixing baffles for Re = 1500. The geometry and location of the baffles are shown on figure 2.

Figure 5. Profile of the wall temperature described by eq. (6), extracted from the experimental data [1] and used as the boundary condition.

where \( \rho \), \( \alpha \) and \( T \) stands for density, phase volume fraction and temperature respectively. The indexes \( v \) and \( l \) refer to the vapour and liquid states. Parameters \( C_c \) and \( C_e \) are numerical constants. It was stated in [4] that large values of \( C_c \) and \( C_e \) may cause issues with simulation convergence, while too small values could be a reason of significant deviation between the interfacial temperature and the saturation temperature.

A sensitivity analysis regarding coefficients \( C_c \) and \( C_e \) were performed in terms of our case geometry. It showed that their influence on the results was up to 10%. It showed that in case of considered problem these parameters had a minor influence on the overall two-phase flow behaviour.

As it was stated before, the aim of the paper was to develop a numerical method which would be a complimentary tool to optimize the PHE used in [1]. The validation process required to perform calculations in a typical range of Re numbers of PHE. The values of Re = 500, 1000, 1500 were chosen to be consistent with [1].

For the numerical calculations the OpenFOAM CFD toolbox was used (release 1612+) [6]. It has the Lee model implemented along with incompressible Navier-Stokes equations as \texttt{interCondensatingEvaporatingFoam} solver. The solver does not have a surface tension model implemented. In practice, it allows for much faster calculations but omits some potentially important physics.

Initial numerical results showed significant deviations form the experimental results. The evaporation process was too weak for considered Re numbers. To improve the considered model and adjust the numerical results to experimental measurements a crude numerical modification was proposed. It was noticed that the major changes in the numerical results were caused by alteration of the latent heat of vaporization. Following this observation a numerical coefficient \( \eta \) was introduced:

\[
h_{vnum} = \eta \ h_{vreal} \tag{5}
\]

where \( h_{vnum} \) and \( h_{vreal} \) are the numerical and real values of the enthalpy of vaporization respectively. Values of \( \eta \) for each considered flow were adjusted to meet consistency between the numerical and experimental results within 10%.

In the numerical model the energy delivered by the heating medium was modelled as a
properly designed wall temperature profile, figure 5. This profile was extracted from the experimental results. The experiment from [1] was designed to evaporate and superheat the R410A refrigerant to the value of $T_{\text{out},\text{exp}} = 11.1 \, \text{C} \quad (\Delta T = T_{\text{out},\text{exp}} - T_{\text{in},\text{exp}} = 11.1 - 1.1 = 10)$, regardless from the Re number. As a consequence the equation for the wall temperature profile has a piece-wise character and combines heat transfer processes of evaporation and superheating respectively:

$$T_w(y) = \begin{cases} T_{w\text{trans}} - T_{\text{w trans}} - T_{\text{r sat}}/m_{\text{ex}}W_{\text{cold}}(1 - e^{-m_{\text{ex}}k_{\text{ex}}2w(y_{\text{trans}} - y)}) & y \leq y_{\text{trans}} \\ T_{\text{w in}} - T_{\text{w in}} - T_{\text{r out}}/m_{\text{sup}}W_{\text{cold}}(1 - e^{-m_{\text{sup}}k_{\text{sup}}2w(L_{HT} - y)}) & y > y_{\text{trans}} \end{cases}$$

(6)

where $T_{\text{w trans}}$ is the heating fluid temperature for which transition from saturated to superheated conditions occurs, $y_{\text{trans}}$ refers to the plate vertical coordinate for which it occurs, $k$ is overall heat transfer coefficient and $m = 1/W_{\text{cold}} - 1/W_{\text{hot}}$ where $W$ is heat capacity rate.

The value of $y_{\text{trans}}$ was extracted from the experimental results of [1]. It was defined as area-weighted average of polygonal chain by extracting the vertices’ locations from IR thermal images, see figure 6. It is important to notice that the value of this coordinate is independent from the Re number. The same wall temperature profile for every Re number was compensated by an increase of the heating liquid mass flow.

![Figure 6](image)

**Figure 6.** Left: Thermal images of the flow in PHE from [1] (courtesy of the authors of [1]), from left: Re=500, 1000, 1500; Right: determination of averaged $y_{\text{trans}}$ coordinate.

4. Numerical results

The figure 7 shows the numerical results for considered range of the Re numbers. Left plot shows the temperature distribution along the height of the PHE. It can be noticed that similarly as in the experiment the superheat of $\Delta T \simeq 10$ at the PHE outlet was achieved. Additionally, the legend of the plot shows the values of $\eta$ used in the individual case.

The right plot from figure 7 shows the changes of the liquid mass fraction along the height of the PHE. The blue line marks the beginning of experimentally achieved superheat region. An agreement between numerical results and experimental data can be considered as satisfactory. It is important to mention that the satisfactory numerical reproduction of the height of the
beginning of the superheat region was possible only if the mixing baffles were present in the 2D geometry.

The fluctuations (zig-zag pattern) visible on the plots of figure 7 are related to the method of data sampling. They were extracted from the vertical line located on the symmetry plane of the flow channel. The presence of the baffles caused the flow stream to change its direction. This is why the data was sampled sometimes along and sometimes across the flow stream. In general, the concentration of the vapour was higher at the sides of the stream rather than in the middle of it, as a result of being in direct contact with the walls.

As it was explained before, the values of $\eta$ for each case were adjusted to satisfy the criteria of results consistency between numerical and experimental outlet temperature values in the range of 10%. Following that, additional numerical calculations were performed for $Re = 250$ and $Re = 3000$ to investigate the changes of $\eta$ parameter with $Re$ number. The same criteria of results validity were used as before. The low $Re$ number case was particularly important because such flows are commonly present in the case of PHE.

Figure 8 presents the functional dependency of the correction parameter $\eta$ from the $Re$ number along with approximating function. The black circles refer to numerical calculations with $Re$

| $Re$  | $\eta$  | $T_{out}$, °C |
|-------|----------|--------------|
| 250   | 1.05     | 11.31        |
| 500   | 0.55     | 12.01        |
| 1000  | 0.33     | 11.02        |
| 1500  | 0.25     | 11.56        |
| 3000  | 0.18     | 11.11        |
numbers used in the experiment, the blue triangles refer to the additional Re numbers used only in numerical calculations. It can be noticed that for the higher Re numbers the correction parameter $\eta$ tends to be Re independent. Table 1 shows the numerical values of $\eta$ and obtained outlet temperature $T_{out}$ (notice that $T_{out, exp} = 11.1 \degree C$).

The approximating function from the figure 8 is $\eta = 38.287 \cdot Re^{-0.683}$ and is characterized by the squared correlation coefficient $R^2 = 0.953$.

![Figure 8. LNG liquid mass fraction along PHE for Re = 500, 1000, 1500.](image)

**Figure 9.** LNG liquid mass fraction along PHE for Re = 500, 1000, 1500. The whole LNG evaporated in the lower part of PHE, below 0.05 m.

Next the validated model was used to study the LNG re-gasification. The general concept was to examine the behaviour of the LNG evaporation process for the same Re numbers and the same thermal conditions (the same wall temperature profile). The saturation pressure and $\eta$ coefficients were also not changed. In case of LNG the saturation temperature was 145 K. Comparing to R410A liquid, LNG had much lower saturation temperature and nearly twice higher heat of evaporation.

The LNG distribution along channel height is presented on figure 9. It can be seen that the LNG re-gasification process was much more dynamic comparing to R410A results. It was related to much larger temperature difference between the liquid LNG and the PHE walls. Table 2 shows values of LNG outlet temperature $T_{out}$ in function of considered Re number and corresponding $\eta$ values. In can be seen that $T_{out}$ did not depend on the $\eta$ parameter. It was in accordance with previous results and general setup of the problem, where the same wall temperature distribution was used for every Re number. It suggests that the introduction of the numeric correction parameter $\eta$ to the considered mathematical model would serve the purpose in case of LNG modelling as well.

Most likely, the values of $\eta$ for LNG evaporation modelling would differ from the values received for R410A. Nevertheless, once the appropriate values have been chosen, based on experimental results, the proposed methodology could result in the creation of a trustworthy model for LNG re-gasification in PHE. This could be particularly useful for the geometrical optimization of PHE for reducing the flow maldistribution problem.

Additionally, it can be deduced that the wall temperature profile (which is directly related to the mass flow of the heating medium) was able to deliver enough heat to evaporate the considered amount of LNG and still assure that the wall temperature remain above zero Celsius. This suggests that solidification should not occur if the heating medium were to be, e.g., a liquid with properly chosen mass flow.

| Re   | $\eta$ | $T_{out}$, $\degree C$ |
|------|--------|------------------------|
| 500  | 0.55   | 17.94                  |
| 1000 | 0.33   | 17.02                  |
| 1500 | 0.25   | 16.02                  |

![Table 2. Values of $\eta$ and resulting outlet temperature for LNG.](image)
5. Conclusions
The conducted numerical investigation has shown that the considered mathematical model of a two-phase flow with phase change can potentially be used in PHE modelling and re-gasification processes.

It was shown that in the case of a full size PHE modelling, which typically consists of O(100) plates, a robust 2D numerical model can be essential. Besides the proper definition of geometrical dimensions and flow conditions, additional geometrical features to model the complexity of PHE flow can also be crucial. In the case of the current investigation, passive mixing baffles were used and they were shown to be necessary to achieve a satisfactory comparison with the experimental data.

It was shown that the Lee phase change model might be insufficient for the proper modelling of re-gasification in PHE. To adjust the Lee model with experimental results, a crude numerical modification was proposed. It assumed the introduction of the numerical correction parameter $\eta$. Its functional dependency on the Re number was shown. This revealed that, for the considered flow, the correction parameter was not necessary for a smaller Re number, as its importance seemed to be crucial only for higher Re numbers. In the case of the 2D PHE model, this could be related to difficulties in the reproduction of flow complexity caused by the structure of real PHE channels. In general, the flow becomes more complex with higher Re numbers.

The proposed mathematical model and the introduced simplification procedure of PHE geometry can be especially useful in investigating a flow in a full-size PHE. This can be particularly important in the case of a flow maldistribution problem. The proposed methodology makes it possible to optimize the geometry with relatively fast calculations to see its effects. It can help to answer the question of whether the flow maldistribution problem is more hydrodynamically driven or if it is more related to the phase change of the considered fluid.

New design proposals can be compared to the referential numerical results by the means of relative case-by-case assessment. The strategy of simulation-based design is an effective instrument to decrease the number of manufactured prototypes and can also be very valuable in terms of time reduction or cost savings.

6. References
[1] Lin Y-H, Li G-C, Yang C-Y 2014 An experimental observation of the effect of flow direction for evaporation heat transfer in plate heat exchanger Appl. Therm. Eng. 88 425
[2] Pawar N and Maurya R S 2012 Flow maldistribution in a simplified plate heat exchanger model - a numerical study Appl. Mech. Mater 110-116 2529
[3] Yang C-Y, Lin Y-H, Lin F-C 2013 Effect of flow direction for the heat transfer performance of refrigerant R-410A evaporation in a plate heat exchanger Heat Transfer Eng. 34 1133
[4] De Schepper S C K, Heynderickx G J and Marin G B 2009 Modeling the evaporation of a hydrocarbon feedstock in the convection section of a steam cracker
[5] Malecha Z M and Malecha K 2014 Numerical analysis of mixing under low and high frequency pulsations at serpentine micromixers Chem. Process Eng. 35 369
[6] Weller H G, Tabor G, Jasak H and Fureby C 1998 A tensorial approach to computational continuum mechanics using object-oriented techniques Comput. Phys. 12 620

Acknowledgements
The presented work has been undertaken in the framework of the Poland – Taiwan scientific cooperation, as a part of the project ”Development of plate heat exchangers for liquid inert gas vaporization, and the modelling of the two-phase flow in heat exchangers” (PHEVAP), financed by The National Center for Research and Development, Poland, No. PL-WTIII/7/2016.

The work has been partly supported by statutory funds from Polish Ministry for Science and Higher Education for the year of 2017.