Torregrosa Huguet, AJ.; Olmeda González, PC.; Gil Megías, A.; Cornejo, O. (2015). Computational fluid dynamics assessment of subcooled flow boiling in internal-combustion engine-like conditions at low flow velocities with a volume-of-fluid model and a two-fluid model. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering. 229(13):1830-1839. doi:10.1177/0954407015571674.

Final version published as: *Proc IMechE Part D: J Automobile Engineering* **229**(13) 1830–1839*,* 2015.

CFD assessment of subcooled flow boiling in I.C. engine-like conditions at low flow velocities with Volume-of-Fluid and Two-Fluid models

A. J. Torregrosa, P. Olmeda, A. Gil, O. Cornejo

CMT - Motores Térmicos. Universitat Politècnica de Valencia.
Aptdo. 22012, E-46071 Valencia, Spain

Abstract

The use of subcooled flow boiling is a convenient option for the thermal management of downsized engines, but proper control of the phenomenon requires the accurate prediction of heat transfer at the coolant side, for which the use of computational fluid dynamics is a suitable alternative. While in most of the applications found to engine cooling a single-fluid equivalent method is used, in this paper the performance of a two-fluid method is evaluated in engine-like conditions with special interest in the low velocity range. The results indicate that the description of the process at low velocities provided by the two-fluid method is better than that of a single-fluid model, while model calibration is simpler and more robust and the computational cost is substantially reduced.

Keywords: Subcooled boiling, two-fluid model, engine cooling

Introduction

The dominant trends for improving fuel economy and reducing pollutant emissions from current internal combustion engines are engine downsizing and raising power density. This produces a significant increase in the density of the waste heat generated, and its removal has led to the use of high coolant-flow velocities, with the consequent increase of hydraulic losses in the engine cooling system and the associated penalty in auxiliary power requirement, especially at part-load conditions [1].

Among the different existing possibilities for the solution of this problem, the change in heat-transfer mode represents a suitable approach, as the use of an evaporative cooling system leads to substantial reductions in the power demand of the coolant pump [1]. However, the system operation range should be limited so as to maximize the potential of the system while avoiding excessive vapour generation, which could have catastrophic consequences.

Such limit is achieved when considering the sub-cooled boiling flow regime, which presents the additional advantage that, as bubbles collapse in the outer bulk flow region due to the sub-cooling, vapour is present only in a thin superheated layer close to the wall so that no essential changes in the design of the cooling system are required [2].

However, boiling-based cooling strategies require that heat transfer at the coolant side is accurately predicted, for which computational fluid dynamics (CFD) simulations supplemented by subcooled boiling models constitute a suitable alternative. As most of the existing models have been developed for nuclear technology applications, with flow and boiling conditions differing from those found in internal combustion engines, they cannot be directly applied to the CFD analysis of engine cooling jackets [3].

Most published applications to flow conditions similar to those occurring in internal combustion engines have made use of a single-fluid representation of the liquid-vapour mixture. In the first attempts [4] several nucleate boiling heat transfer models were implemented in order to calculate wall surface heat transfer coefficients that then affected the computed fluid temperatures, so that the computed fluid dynamics was not affected and the coolant fluid properties were assumed to be those of the liquid coolant.

Subsequent contributions make use of an equivalent single-phase fluid representing the liquid-vapour mixture and accounting for the vapour fraction and its spatial and temporal

DOI: 10.1177/0954407015571674

variations. Bo [5] used a simple driving force - time scale representation in order to account for all the mechanisms influencing mass transfer between the two phases, and computed the surface heat flux with the empirical correlation proposed by Rohsenow [6], concluding that the flow field and the temperature distribution are significantly modified. A similar method was used by Fontanesi et al. [7], who found qualitative agreement in flow and temperature distributions in a real coolant jacket. Other application of the Rohsenow correlation in engine-like conditions are that of Li et al. [3], who used it the for the fully developed boiling regime in the frame of a division description method, and Li et al. [8] who used it to compute the surface heat flux in the context of a superposition method.

Some amount of work has also been devoted to the inclusion into single-phase fluid CFD computations of the Chen correlation [9]. This expression provides acceptable results when the hydraulic diameter is uniquely defined, as when reproducing experiments performed in cooling gallery simulator rigs, but poses serious problems when applied to real cooling gallery geometries. Different solutions have been proposed, as by Cardone et al. [10] who defined equivalent approximating ducts for the different parts of the cooling jacket, with acceptable results, and Punekar and Das [11].

To the authors' knowledge, the only application of a two-fluid method to internal combustion engine cooling galleries is that of Mohammadi and Yaghoubi [12], who obtained a fair reproduction of the trends observed in measurements performed in a gallery simulator with an inlet temperature of 90 ºC and an inlet velocity of 0.25 m/s, but with a consistent underestimation of the heat fluxes.

The objective of the present note is to provide additional evidence on the performance of two-fluid multiphase methods for the CFD assessment of subcooled boiling flow at velocities as low as 0.1 m/s in engine-like conditions, considering that of an equivalent single-phase fluid model as a reference for comparison. With that purpose, the Volume of Fluid (VoF) and the two-fluid models implemented in STAR-CCM+ were used to reproduce the results obtained in a cooling gallery simulator rig [13]. The paper is organized as follows: first, the geometry considered is described together with the mesh used. Then, a brief account of the wall heat flux computation used in both models is given, with special focus on the calibration of model parameters. Following, the results obtained are discussed in terms of heat flux prediction and temperature distributions. Finally, conclusions are summarized and future developments are outlined.

Geometry and mesh

The geometry considered is shown in Figure 1, where the different sections used later for the representation of the results are also shown. The system is a straight duct with square cross section, to which the heating surface is attached. All the details of the experimental setup, comprising its instrumentation and control, can be found in [13].

A structured mesh with hexahedral cells was used. Different mesh densities (see in Figure 2 and Table 1) were considered in order to check the mesh-independence of the results.

It was found that the difference between using mesh b and mesh c was only of about 0.75% in the heat flux results, and this in the more critical case of low velocity (0.1 m/s), wall temperature of 141ºC and maximum vapour quality of 9%. Accordingly, mesh (b) was chosen as that providing the best compromise between computation time and accuracy of the results.

DOI: 10.1177/0954407015571674

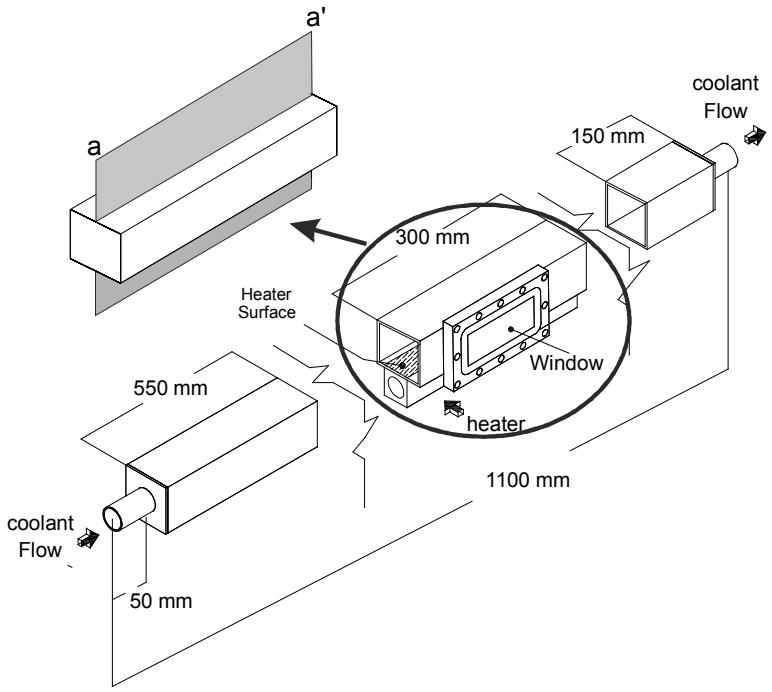

Figure 1. Geometry of the system and section used for representation.

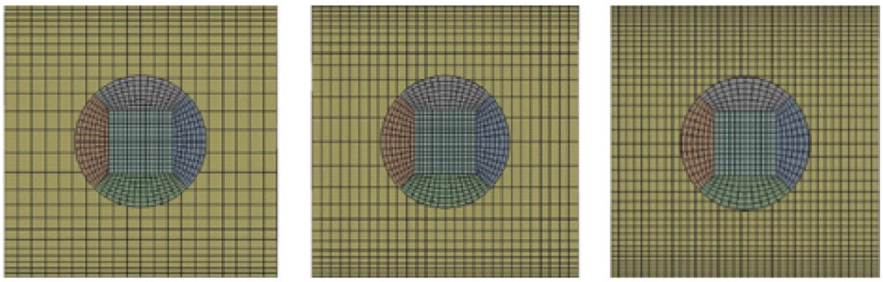

Figure 2. The three meshes considered

Mesh	Elements	Aspect Ratio
a	168,984	2.96
b	283,864	2.94
c	651600	2.6

Table 1: Characteristics of the meshes used.

Modelling

Volume-of-Fluid (VoF) model

The VoF model considers both phases combined on a volumetric basis in an Eulerian frame. It was chosen because it provides a relatively simple treatment when the focus is the computation of the heat flux. No account is thus given for the interaction between phases, and the velocity, pressure and temperature fields are the same for both phases.

DOI: 10.1177/0954407015571674

The flow equations are solved with a pressure-based segregated solver [14], and making use of a high resolution interface capturing scheme (HRIC) based on a compressive interface capturing scheme for arbitrary meshes (CISCAM) [15]. The boundary layer was modelled with hexahedral layers whose width is properly chosen according to the wall treatment adopted. Additionally, in view of the low vapour fraction expected in the subcooled boiling regime it can be assumed that the vapour phase follows the fluctuations of the liquid phase and thus the turbulent stresses are modelled only for the liquid phase making use of the realizable $k-\varepsilon$ two layer all y^+ wall treatment [14].

Evaporation and condensation of the coolant fluid are computed assuming that the temperature of the vapour bubbles is equal to the saturation temperature T_{sat}, and that the temperature of the liquid fluid is close to the mixture temperature T, so that mass exchange between phases is controlled by heat transfer as:

$$\dot{m}_{EC} = C_{HA}(T - T_{sat})/h_{lv} \tag{1}$$

where h_{lv} is the latent heat of vaporization and C_{HA} is the product of the heat transfer coefficient between the bubbles and the surrounding fluid times the interfacial area. This model is turned on when the temperature of the surface in contact with the fluid is higher than the saturation temperature, when bubbles start to appear.

The evaporation at wall boundaries is controlled by the wall heat flux q_{bw}, so that the vapour mass generation rate is given by

$$\dot{m}_{ew} = C_{ew} q_{bw}/h_{lv} \tag{2}$$

where C_{ew} is the fraction of the heat flux related with bubble generation.

Several well-established correlations are available for the estimation of heat transfer in nucleate boiling conditions. STAR-CCM+ incorporates that proposed by Rohsenow [6], which was initially developed for pool boiling and later extended to flow boiling:

$$q_{bw} = \mu_l h_{lv} \sqrt{\frac{g(\rho_l - \rho_v)}{\sigma}} \left(\frac{c_{pl}(T_w - T_{sat})}{C_{qw} h_{lv} \mathrm{Pr}_l^{1.7}} \right)^{3.03} \tag{3}$$

where μ_l, c_{pl}, ρ_l, and Pr_l denote, respectively, dynamic viscosity, specific heat at constant pressure, density and Prandtl number of the liquid phase, and g, ρ_v, σ, and T_w, are gravity acceleration, vapour density, surface tension at the liquid/vapour interface and wall temperature, respectively. The value of the empirical constant C_{qw} depends on the liquid/solid interaction, and accounts for the geometry, the thermal conductivity and the thermal absorption of the surface [16] and for the heating process [3].

Different possibilities exist for the determination of C_{qw}. When experimental data are available, it can be determined from:

$$c_{pl}\left[\frac{T_w - T_{sat}}{h_{lv}\mathrm{Pr}_l^{1.7}}\right] = C_{qw}\left[\frac{q_{bw}}{\mu_l h_{lv}}\sqrt{\frac{\sigma}{g(\rho_l - \rho_v)}}\right]^{1/3} \tag{4}$$

From the experimental data obtained in [13], equation (4) provides the values shown in Figure 3, where it can be observed that the C_{qw} values obtained range approximately from 0.01 to 0.06, which is a considerable variation. Additionally, while the influence of the inlet coolant temperature T_i is relatively clear, the dependencies observed with respect to velocity and surface temperature do not exhibit any clear trends, what poses serious problems to define a suitable value in order to calibrate the model.

DOI: 10.1177/0954407015571674

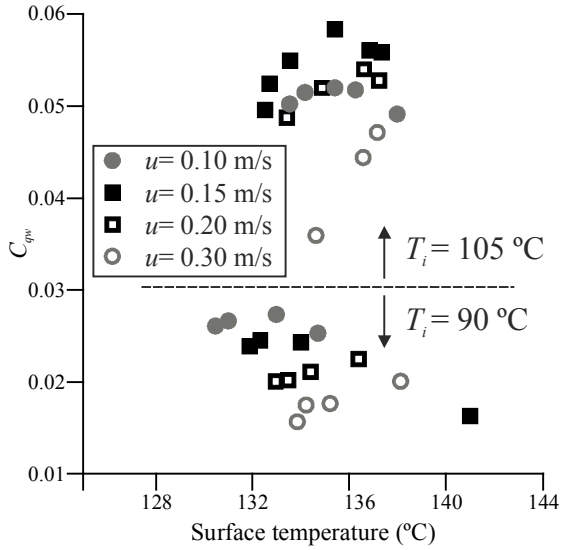

Figure 3. Values of C_{qw} obtained from equation (4) and experimental data from [13].

In view of this, alternative approaches were considered. A different way to obtain C_{qw} was proposed in [17], where it was expressed as a function of the roughness arithmetic average R_a as:

$$C_{qw} = C\{[a\ln(R_a) - b]p_r - c\ln(R_a) + d\} \tag{5}$$

where a, b, c and d are empirical constants obtained for different halocarbon refrigerants and wall rugosities and materials. The fluid/surface material combinations are accounted for by coefficient C, whose values are around 1, ranging from 0.8 to 1.3 [17]. Even if the fluids are different from that considered in this work, the possibility of applying equation (5) to the present case was checked. Taking $C = 0.8$ gives $C_{qw} = 0.05$, which is within the value range shown in Figure 3. This suggests that equation (5) could be a suitable alternative if only a rough estimate of the surface roughness is available.

Another alternative was proposed by Podowski [18], which has the advantage that only pressure is required to compute C_{qw}, as

$$C_{qw} = 0.01[1.058 - 0.0056\,p/p_{atm} + (0.0045\,p/p_{atm})^2 - (0.0037\,p/p_{atm})^3] \tag{6}$$

However, this expression is valid in pool boiling for heat fluxes such that the effect of single-phase convection becomes negligible. For pressures close to atmospheric pressure it is supposed to be applicable to heat fluxes as low as 5·kW/m² or less. When considering the present conditions, a value of 0.011 is obtained which, while being within the range of Figure 3, does not seem representative of the current case, as pressure variations were not so important so as to justify the dispersion observed in C_{qw}.

Therefore, the values shown in Figure 3 were used in the present simulations for each particular condition, just to ensure the best modelling results for each case considered. Of course, this must be bore in mind when discussing the relative merits of the two computational approaches.

Euler-Euler two-fluid model

The model description given here follows closely that provided in [19]. The wall heat partitioning model used by Star-CCM+ to determine the rate of vapour generation is:

$$q_w = q_l + q_Q + q_e \tag{7}$$

DOI: 10.1177/0954407015571674

Here, q_w is the total heat flux from the wall, q_l represents single-phase convection, $q_Q$is the quenching heat flux within the influence area A_e of the bubbles and q_e is the evaporation heat flux. The area A_e is taken to be proportional to the maximum cross-section of a single bubble at departure and to the density number N of active nucleation sites:

$$A_e = F_A(\pi D_D^2/4)N \tag{8}$$

where D_D is the bubble departure diameter and F_A is an empirical constant. The density of nucleation sites is estimated as proposed by Lemmert and Chawla [20]:

$$N = [m(T_w - T_{sat})]^n \tag{9}$$

Here, m and n are empirical constants with values $m = 185$ and $n = 1.805$, and T_w and T_{sat} are the wall and the fluid saturation temperature, respectively, so that the nucleation site density is proportional to the n-th power of the wall superheat. In this model, a bubble contact angle of 46 degrees is assumed, which is consistent with the value provided by a mechanistic bubble detachment model for the case considered in this paper [21].

Pure convection (i.e. out of the area covered by nucleation sites) is represented by:

$$q_l = h_c A_c(T_{wall} - T_l) \tag{10}$$

where T_l is the liquid temperature, the heat transfer coefficient h_c is obtained from the wall function model, and

$$A_c = 1 - A_e \tag{11}$$

The quenching heat flux is that used to heat the liquid replacing the space occupied by a bubble before detachment. It is modelled following Del Valle and Kenning [22] who considered an analogy with transient heat conduction in a semi-infinite medium:

$$q_Q = h_q A_e(T_{wall} - T_l) \tag{12}$$

where the heat transfer coefficient is obtained from the Nusselt number correlation proposed by Ranz and Marshall [23]:

$$N_u = 2 + 0.6\mathrm{Re}_v^{0.5}\mathrm{Pr}_l^{0.3} \tag{13}$$

Finally, the heat flux associated with evaporation is given by [24]:

$$q_e = (\pi D_D^2/6)\rho_d h_{lv} f N \tag{14}$$

Here, f is the bubble departure frequency, which is obtained following Cole [24] from

$$f^2 = (4/3)g(\rho_l - \rho_v)/(D_D\rho_c) \tag{15}$$

and the bubble departure diameter D_D is computed, as proposed in reference [25], as:

$$D_D = D_0\exp(-\Delta T_{sub}/\Delta T_0) \tag{16}$$

where, $D_0 = 0.6$ mm and $\Delta T_0 = 45$ K are model constants, and $\Delta T_{sub} = T_{sat} - T_l$ is the liquid sub-cooling.

From the application of all the models described, it appears that the only model parameter required for model calibration is in this case the bubble influence area. Contrary to what was observed in the C_{qw} coefficient, a single value could be used in all the flow and temperature conditions considered.

Results and discussion

In order to dispose of a comparison reference computation, first the VoF model described in section 3.1 was used. All the duct walls other than the heater surface were assumed to be adiabatic. For each heat flux and flow condition, the values of the calibration constant

DOI: 10.1177/0954407015571674

C_{qw} shown in Figure 3 were used, so that the results obtained should be, in principle, the best achievable with such a model.

The results are shown in Figure 4, for two different inlet temperatures (90 ºC and 105 ºC) and four different inlet flow velocities (0.1, 0.15, 0.2 and 0.3 m/s), which are representative of temperature and velocity conditions occurring in different parts of the engine cooling jacket. Apart from the experimental and CFD results, the modified Chen correlation obtained in reference [13] has also been plotted in order to provide an additional check criterion. It can be observed that, in general, the CFD results agree better with the measurements for the higher heat fluxes corresponding the lowest inlet fluid temperature for all the fluid velocities considered.

It is also apparent from the results that the agreement gets better the higher is the flow velocity. This could be expected, since the suppression effect of the flow is higher the higher is the velocity [9], and therefore the weight of the boiling contribution to the total heat flux is smaller, so that any deviations in the description of the boiling itself should become less apparent. This effect is particularly clear for the lowest velocity considered and an inlet temperature of 105 ºC, where significant deviations are observed.

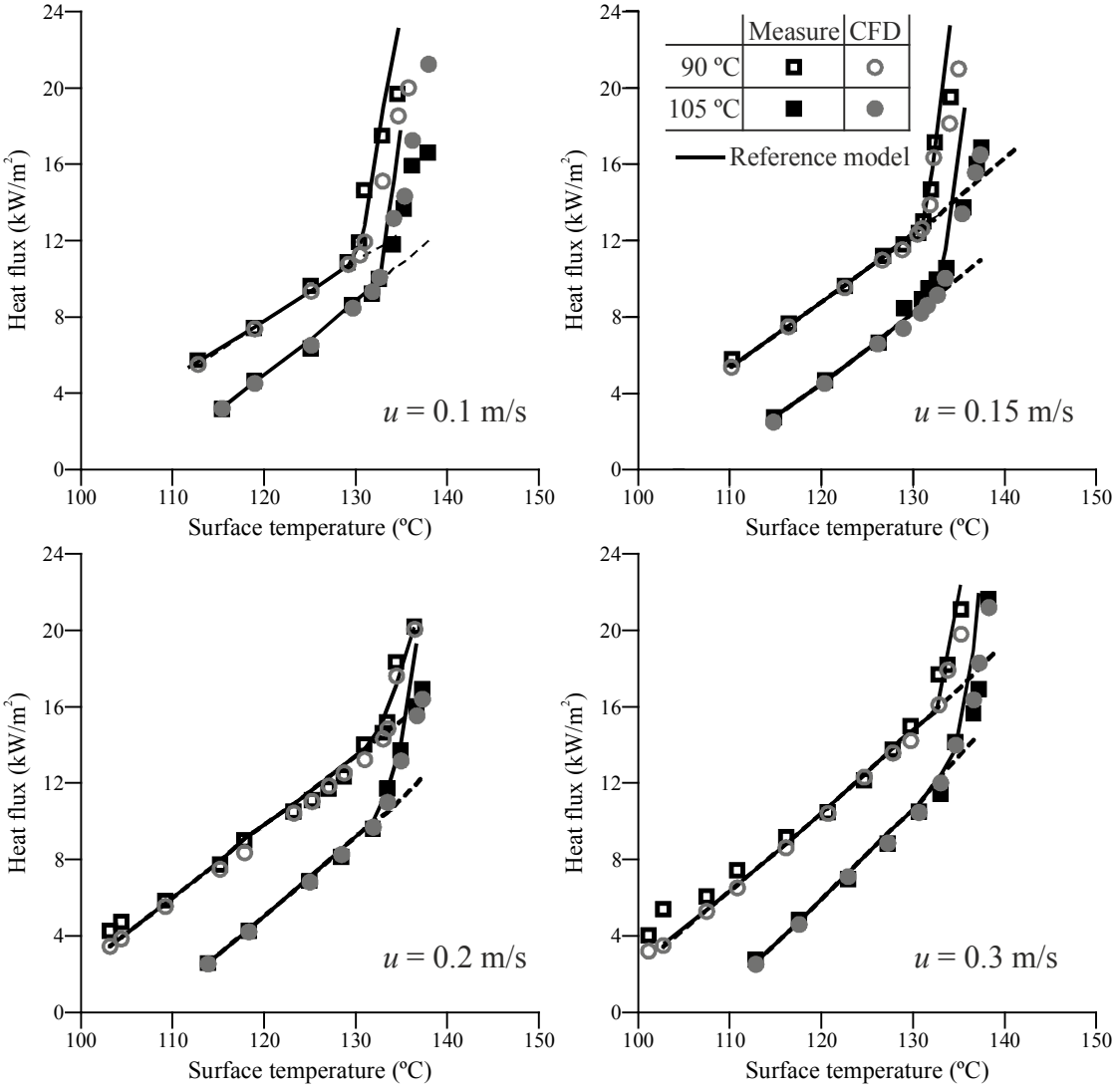

Figure 4. Comparison of measurements and CFD results from the VoF model for different flow velocities and inlet temperatures.

DOI: 10.1177/0954407015571674

In the case of the two-fluid model it was possible, as mentioned in section 3.2, to consider a single value for the empirical constant F_A affecting the bubble influence area. The value finally chosen turned out to have an order of magnitude consistent with the density of nucleation sites estimated in [21] for the same heater surface. The corresponding results are shown in Figure 5, again together with the reference correlation.

It can be observed that, as in the previous case, the results are better for the lowest inlet fluid temperature, and that the agreement improves as the inlet flow velocity increases. However, a significant improvement in the results at low velocities can be observed, which indicates that the representation of the boiling contribution is more accurate than that provided by the VoF model, even if the calibration parameter was in that case adjusted to each condition.

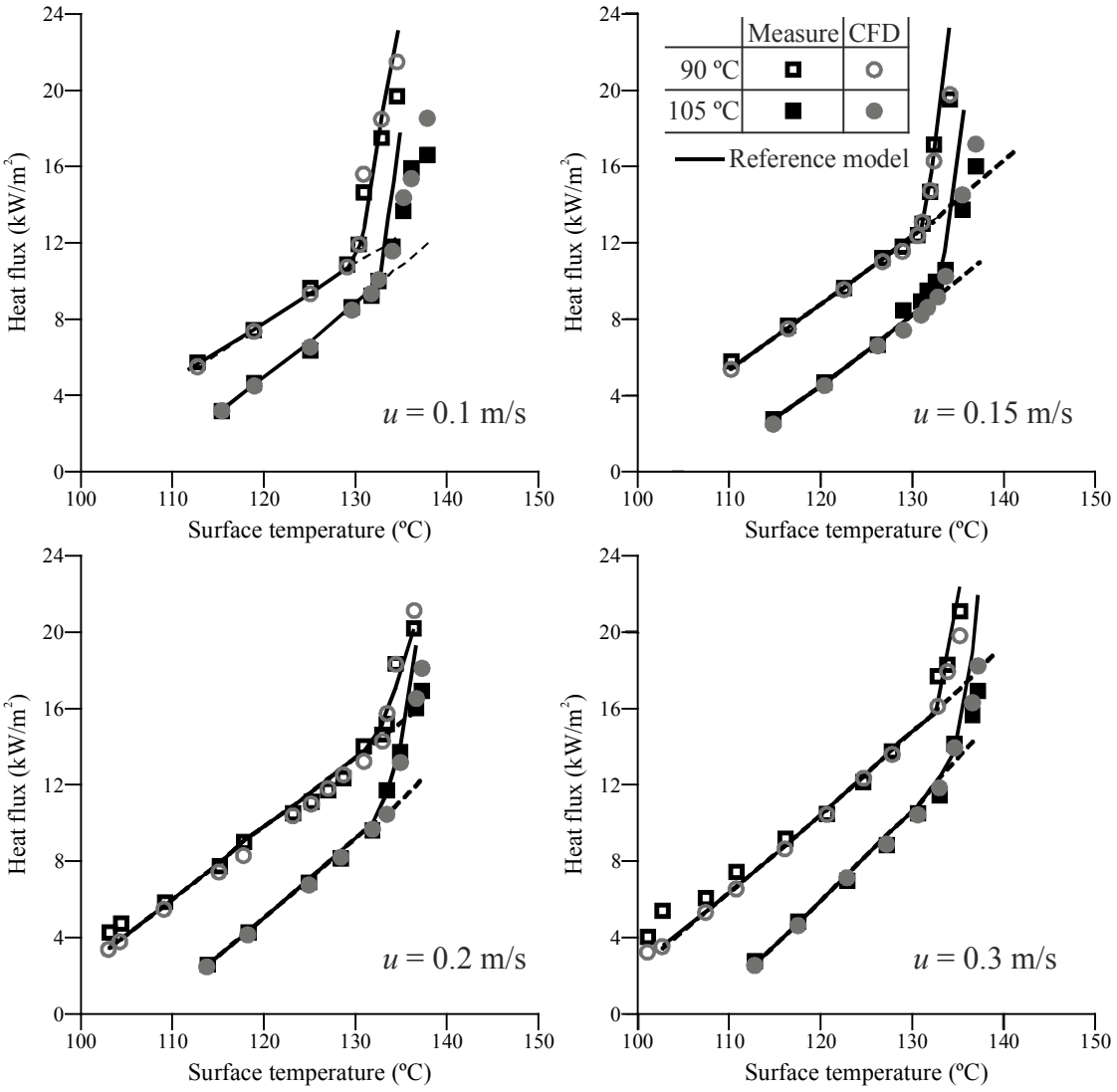

Figure 5. Comparison of measurements and CFD results from the two-fluid model for different flow velocities and inlet temperatures.

This suggests that the two-fluid method considered provides a more comprehensive description of the problem. Additionally, the number of iterations required for the computation to converge in mass flows and bulk fluid temperatures is about half that required by the VoF method. These aspects, together with the fact that a robust calibration

DOI: 10.1177/0954407015571674

of the model is achievable, are relevant when considering the potential extension of the use of two-fluid models to realistic cooling gallery geometries. One should take into account, however, that the validity of this approach is highly dependent on the fact that the vapour fraction is below 1% [14], so that care has to be taken to ensure that only subcooled flow boiling occurs.

As a final check of the consistency of the solutions provided, in Figure 6 the temperature fields computed at section a-a' (see Figure 1) are compared in the case of a coolant velocity of 0.1 m/s, inlet coolant temperature of 90 ºC and wall temperature of 134.7ºC. It can be observed that, in the case of the two-fluid method, the results are consistent with the expected temperature distribution, with a smooth growth of the temperature along the heater surface and into the outlet duct, and an essentially self-similar vertical profile.

However, in the case of the VoF method, it appears that only a thin fluid layer is affected by the presence of the heater surface, and it is only downstream of that surface that a significant vertical temperature gradient is observed. As a consequence, the mean temperature in the outlet duct was underestimated, even if the calibration constant C_{qw} was adjusted to approach the measured heat flux.

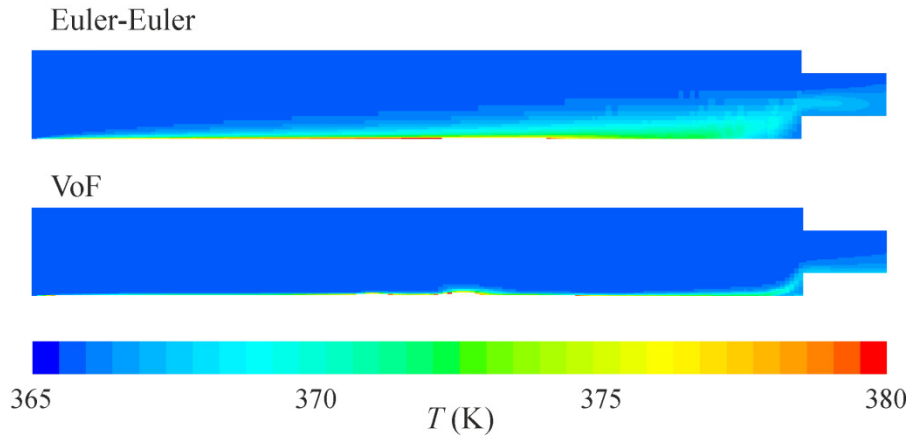

Figure 6. Temperature distribution predicted by the two-fluid model at section a-a'.

Summary and conclusions

The ability of a two-fluid Euler-Euler method for describing subcooled flow boiling at low velocities in conditions similar to those found in cooling galleries of internal combustion engines has been checked. With this purpose, experiments on an engine cooling gallery simulator rig have been reproduced, using the two-fluid method and an equivalent single-fluid VoF method for reference purposes.

The details of the model calibration in both cases have been discussed, showing that the two-fluid method is more robust in this aspect, as it has been possible to find a single value of the calibration constant valid for all the flow and temperature conditions considered.

From the results obtained, it appears that the performance of the two-fluid method is comparable to that of the VoF method for the highest flow velocities considered (0.2 and 0.3 m/s), but significantly better for the lowest velocities. Since it is in these last conditions when the relative importance of the boiling contribution is higher, this indicates that the two-fluid method provides a more accurate description of this contribution. Additionally, the computational cost is significantly lower than that required by the VoF method for similarly converged results.

DOI: 10.1177/0954407015571674

These results indicate the interest of extending the use of the two-fluid method to realistic engine cooling jacket geometries, taking care in any case that the resulting vapour fractions do not surpass the validity range of the method.

Acknowledgement

The equipment used in this work has been partially supported by FEDER project funds "Dotación de infraestructuras científico técnicas para el Centro Integral de Mejora Energética y Medioambiental de Sistemas de Transporte (CiMeT), (FEDER-ICTS-2012-06)", framed in the operational program of unique scientific and technical infrastructure of the Ministry of Science and Innovation of Spain. O. Cornejo is indebted to Senacyt Panamá for their support.

References

1. Pang HH and Brace CJ. Review of engine cooling technologies for modern engines. *Proc IMechE, Part D: Journal of Automobile Engineering* 2004; 218: 1209-1215.
2. Steiner H, Brenn G, Ramstorfer F, et al. Increased cooling power with nucleate boiling flow in automotive engine applications. In: Chiaberge M (ed) *New trends and developments in automotive system engineering*. InTech, 2011, pp. 249-272.
3. Li Z, Huang RH and Wang ZW. Subcooled boiling heat transfer modelling for internal combustion engine applications. *Proc IMechE, Part D: Journal of Automobile Engineering* 2012; 226: 301-311.
4. Hawley JG, Wilson M, Campbell NAF, et al. Predicting boiling heat transfer using computational fluid dynamics. *Proc IMechE, Part D: Journal of Automobile Engineering* 2004; 218: 509-520.
5. Bo T. CFD homogeneous mixing flow modelling to simulate subcooled nucleate boiling flow. SAE paper 2004-01-1512, 2004.
6. Rohsenow WM. A method of correlating heat transfer data for surface boiling. *Trans. ASME* 1952; 74: 969-975.
7. Fontanesi S, Cicalese G, Fantoni S, et al. CFD investigation of the thermo-mechanical behavior of a high performance bike engine. SAE paper 2011-32-0525, 2011.
8. Li GX, Fu S, Liu Y, et al. A homogeneous flow model for boiling heat transfer calculation based on single phase flow. *Energy Convers. Manage.* 2009; 50: 1862-1868.
9. Chen JC, Correlation for boiling heat transfer to saturated fluids in convective flow. *I&EC Process Des. Dev.* 1966; 5: 322-329.
10. Cardone M, Senatore A, Buono D, et al. A model for application of Chen's boiling correlation to a standard engine cooling system. SAE paper 2008-01-1817, 2008.
11. Punekar H and Das S. Numerical simulation of subcooled nucleate boiling in cooling jacket of I.C. engine. SAE paper 2013-01-1651, 2013.
12. Mohammadi A and Yaghoubi M. Two phase flow simulation for subcooled nucleate boiling heat transfer calculation in water jacket of diesel engine. *J. Engine Res.* 2011; 22: 50-61.
13. Torregrosa AJ, Broatch A, Olmeda P, et al. Experiments on subcooled flow boiling in I.C. engine-like conditions at low flow velocities. *Exp. Therm. Fluid Sci.* 2014; 52: 347-354.
14. STAR-CCM+ v8.02. User Guide. CD-Adapco, 2012.

DOI: 10.1177/0954407015571674

15. Hernandez-Perez V, Abdulkadir M and Azzopardi BJ. Grid generation issues in the CFD modelling of two-phase flow in a pipe. *J. Comput. Multiphase Flows* 2011; 3: 13-26.

16. Pioro IL, Rohsenow W and Doerffer SS. Nucleate pool-boiling heat transfer II: Assessment of prediction methods. *Int. J. Heat Mass Transfer* 2004; 47: 5045-5057.

17. Saiz-Jabardo JM. An overview of surface roughness effects on nucleate boiling heat transfer. *Open Transp. Phenom. J.* 2010; 2: 24-34.

18. Podowski MZ. Toward mechanistic modeling of boiling heat transfer. *Nucl. Eng. Technol.* 2012; 44: 889-896.

19. Lo S and Osman J. CFD modeling of boiling flow in PSBT 5×5 Bundle. *Sci. Technol. Nucl. Install.* 2012; 2012: 795935.

20. Lemmert M and Chawla JM. Influence of velocity on surface boiling heat transfer coefficient. In: Hahne E and Grigull U (eds) *Heat Transfer in Boiling.* Academic Press, 1977, pp. 237-247.

21. Torregrosa AJ, Broatch A, Olmeda P, et al. A note on bubble sizes in subcooled flow boiling at low velocities in internal combustion engine-like conditions. *J. Appl. Fluid Mech.* (under review).

22. del Valle MVH and Kenning DBR. Subcooled flow boiling at high heat flux. *Int. J. Heat Mass Transfer* 1985; 28: 1907-1920.

23. Ranz WE and Marshall WR. Evaporation from drops, 1, 2. *Chem. Eng. Prog.* 1952; 48: 141-146, 173-180.

24. Cole R. A photographic study of pool boiling in the region of the critical heat flux. *AIchE J.* 1960; 6: 533-542.

25. Tolubinsky VI and Kostanchuk DM. Vapour bubbles growth rate and heat transfer intensity at subcooled water boiling heat transfer. In: *4th International Heat Transfer Conference* (ed U Grigull), Paris, France, 1970, paper no. B-2.8. Amsterdam : Elsevier.

DOI: 10.1177/0954407015571674