Mathematical Modelling and Analysis Multiphase flow characteristics of Nucleate boiling and Its Application in Automobile using CFD

Himanshu Tiwari[1], Vardan singh nayak[2], Arvind Gupta[3]

[1]M.tech Scholar Mechanical Engg. Dept. VIST, Bhopal
[2,3]Asst. Professor Mechanical Engg. Dept. VIST, Bhopal

---***---

ABSTRACT - Boiling processes are widely used in technical applications. The chemical industry, power plant technology, and refrigeration engineering are just a few examples. Intensive research on nucleate boiling processes has been undertaken for decades. Nevertheless, the physical phenomena are still not sufficiently understood. This is mainly caused by the large number of influencing factors and the wide range of length and time scales boiling processes act on. Especially through the research on small length scales a better understanding of the underlying physical phenomena should be achieved. The subject of the present thesis is the Numerical investigation of boiling processes with single bubbles and bubble interactions. In a further step, the reliability of the results obtained for a few interacting bubbles is tested on a technical length scale.

Within the course of this study, sub cooled nucleate boiling in a vertical pipe has been modelled using CFD. The modelling has been carried out within the Fluent14.5 and a two-phase Eulerian approach has been chosen. The code can be used to predict the distribution of the local flow parameters, i.e. the void fraction, the bubble diameter, the velocity of both liquid and gas, the turbulent intensity as well as the liquid temperature. Special attention has been devoted to the phenomena which govern the void fraction distribution in the radial direction. Two different solvers have been implemented and the simulations have been performed in two dimensions.

In second part of the study, Modern engine cooling systems use sub cooled boiling as an effective heat transfer form. The significantly higher heat transfer rate achievable with boiling can keep the temperature in key thermal areas within an acceptable level. In this study, a three-dimensional two-fluid model coupled with RPI wall boiling model was adopted to simulate the local two-phase characteristics of boiling at different passages under low-pressure condition. Moreover, a modified bubble departure diameter correlation was developed and implemented in computational fluid dynamics code with user-defined function, and the correlation was verified by two sets of experimental data under different flow conditions. Results showed an improved agreement with the measured local void fractions. Finally, the verified model was applied to a practical cooling passage for a water-cooled diesel engine. The simulation results showed that present two-fluid model could get an accurate temperature field for cylinder head. The distribution of vapour phase in cooling passage was obtained, and it could be used as a reference to evaluate the reliability of local boiling. Moreover, due to the function of inter phase forces, the majority of bubbles were quickly removed from near wall and the void fraction was reduced.

***Keywords*: Nucleate boiling, Two-fluid boiling model, Void fraction , Engine cooling passage, Computational fluid dynamic ,Bubble departure diameter etc.**

1. INTRODUCTION

1.1 GENERAL

Boiling is an extremely important process having vast application in various fields of science, technology & industries. Despite this, the fundamental mechanisms involved in the process are far from being understood. In the complete phase change problem the mass, momentum and energy transport equations must typically include the effects of surface tension, latent heat, interface mass transfer, discontinuous material properties and complicated liquid-vapour dynamics. Over the recent years several studies have been made to clarify and model the interface transport mechanism associated with liquid-vapour phase change process, such as film boiling. In the past several experimental works have been performed for understanding the phenomena of boiling. But those early investigations could not provide suitable physical details needed for understanding bubble formation and the time varying heat transfer characteristics. However experimental studies have resulted in several empirical correlations, which are valid in specific cases.

Boiling heat transfer is used in a wide field of applications: From rather simplistic cooking activities in everyday life to high-tech solutions for the chemical industry, power generation and cooling applications. While boiling research is, of course, not required for cooking, the exact knowledge of the boiling process, its parameters and limitations is compulsive for the optimized technological application. Besides the wide range of different applications, another very interesting characteristic of boiling heat transfer is the wide range of system scales. Boiling is used to cool tiny high-performance electronic components. On the other hand, boiling is also used on a much larger scale in steam generators for power plants. Due to its intensive use, boiling heat transfer has been intensively studied in the past and still is subject of ongoing research activities in many groups all over the world. In spite of the past research, many aspects of boiling are still not well understood. The demand to transfer higher heat fluxes at the same or even at lower wall superheats and to predict the limits of the boiling process is growing and makes the scientific investigation of boiling phenomena inevitable. In particular, the demand for highly efficient small scale heat transfer is growing. In 1965, Moore has predicted that the performance of computer chips is doubled every 18 months. Although the increase of the computational power has slowed down meanwhile, it still is ever increasing. More and more components are packed on the same area and the dissipated heat flux increases tremendously. Boiling is one of the solutions to overcome this cooling problem.

2. LITERATURE

1. Th. Frank*, P. Beckstein*, C. Lifante*, A.D. Burns- The aim of this paper is to present the validation of a newly implemented wall boiling model for the prediction of subcooled nucleate boiling under pressurized and normal conditions, so e.g. in rod bundles and fuel assemblies of PWR and BWR, first made available with full GUI support in ANSYS CFX 12.0 (ANSYS Inc., 2009). The model formulation is based on the so-called RPI wall boiling model published by Kurul & Podowski (1991) but has been extended in ANSYS 12.0 by a special approach regarding the prediction of the liquid subcooling temperature in order to make the model grid independent in the context of a CFD simulation. Furthermore the wall boiling heat partitioning algorithm has been coupled to the prediction of conjugate heat transfer in the solid material of the heated boundary of the fluid domain, allowing for the definition of volumetric energy sources in e.g. the rods of nuclear fuel assemblies heated by nuclear fission reaction. The Paper discusses the theory of the implemented model as well as the application of the model to boiling flow in vertically directed circular annulus with a centralized heated rod, as investigated by Lee et al. (2008). CFD results are compared to experiments of Lee et al. applying a hierarchy of consistently refined meshes and different CFD setup configurations including the prediction of heat transfer in the solid material of the heated rod by CHT and by prescribing a volumetric thermal energy release in the solid material instead of a prescribed wall heat flux or wall temperature at the heated boundary of the fluid flow domain.

2. K. Jagannath, Akhilesh Kotian, S. S. Sharma, Achutha Kini U., P. R. Prabhu- —Boiling process is characterized by the rapid formation of vapour bubbles at the solid–liquid interface (nucleate boiling) with pre-existing vapour or gas pockets. Computational fluid dynamics (CFD) is an important tool to study bubble dynamics. In the present study, CFD simulation has been carried out to determine the bubble detachment diameter and its terminal velocity. Volume of fluid method is used to model the bubble and the surrounding by solving single set of momentum equations and tracking the volume fraction of each of the fluids throughout the domain. In the simulation, bubble is generated by allowing water-vapour to enter a cylinder filled with liquid water through an inlet at the bottom. After the bubble is fully formed, the bubble detaches from the surface and rises up during which the bubble accelerates due to the net balance between buoyancy force and viscous drag. Finally when these forces exactly balance each other, it attains a constant terminal velocity. The bubble detachment diameter and the terminal velocity of the bubble are captured by the monitor function provided in FLUENT. The detachment diameter and the terminal velocity obtained are compared with the established results based on the shape of the bubble. A good agreement is obtained between the results obtained from simulation and the equations in comparison with the established results.

3. OBJECTIVE

The main objective of this study is to assess the enhancement in heat transfer coefficient and critical heat flux/dry-out heat flux using enhanced surfaces. For this, the project is divided into two major portions each with multiple objectives.

The first portion of the study utilizes Nucleate boiling as heat transfer mechanism. This will investigate the effects of wet ability on heat transfer performance. The nobility of this study is that wet ability is induced on a polished silicon surface without altering the surface roughness. Heat transfer coefficient and the critical heat flux will be the focal point of discussion.

The second part of the study is conducted with the intention to create thin-film evaporation similar to micro layer evaporation in bubble nucleation in pool boiling but at higher frequency on a localized high heat flux area. The objectives and goals of the two major portions of the study are listed below.

1. Investigation of the effects of wet ability on heat transfer coefficient and critical heat flux.

2. Surface treatment of robust thin film heater on silicon substrate that can survive up to critical heat flux (CHF)

3. Surface treatment with wet ability ranging from highly hydrophilic to hydrophobic without altering the surface roughness.

4. Quantify the enhancement in CHF and BHT (Boiling heat transfer)

5. Bubble dynamics study at lower heat flux for all testing surfaces Effects of surface orientations

5. Thin-film evaporation of water on enhanced surfaces for high heat flux dissipation

6. Study the effects micro-gaps, system pressure and temperature

4. METHODOLOGY

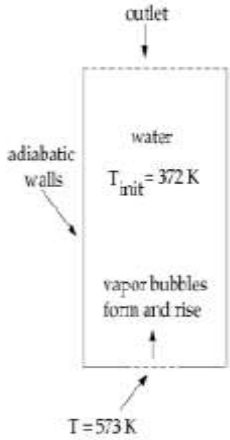


Figure 1: Problem Schematic

4.4.2 CAD Model: Generation of 2d geometry in fluent

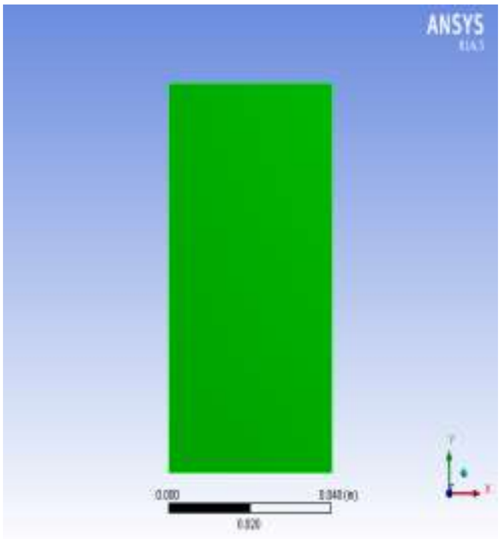


Figure 2: 2D CAD Model

4.2.3 MESH MODEL:

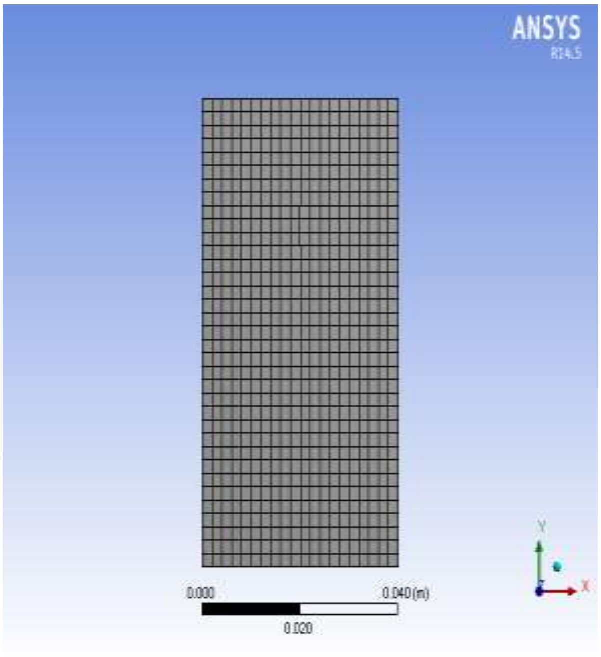
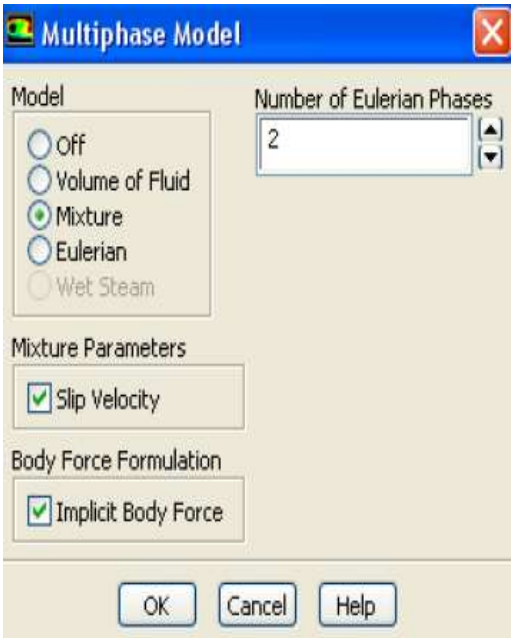


Figure 3: Mesh model

1. General: Transient

2. Models: Multiphase model is used.



3. Energy: Enable the Energy Equation

4. Materials:

i. Select water-liquid (h2o<l>) from the FLUENT Fluid Materials selection list.
ii. Click Copy and close the FLUENT Database Materials dialog box.

[a] Liquid- Material property

Parameters	Value
Density	200 kg/m3
Specific heat	400 j/kg-k
Thermal conductivity	40 w/m-k
Viscosity	0.1 kg/m-s

[b] Similarly For Vapour:

Parameters	Value
Density	5 kg/m3
Specific heat	200 j/kg-k
Thermal conductivity	1 w/m-k
Viscosity	0.05 kg/m-s

5. Phases:

(a) Primary phase is vapour and define Secondary phase is Fluid.
(b)Phase interaction – Surface Tension is 0.1 N/m

6. Cell zone: Condition is Fluid.

7. Boundary Conditions:

(a) Heat-

a. Temperature is 510 k (10 K superheat with respect to the saturation Temperature 500 k.
b. Various time intervals are taken with time step for the transient responses of multiphase model.

(b) Outlet- enters 350 k for Backflow Total Temperature.

(C) Operating Condition- Enable Gravity.
Set the Gravitational Acceleration in the Y direction to -9.81 m/s2.
Enable Specified Operating Density and set the Operating Density to 5 kg/m3.

8. Initialize the solution:

(a)Initialize with pressure and velocity components at zero.
(b) Enter 1 for the fluid Volume Fraction.
(c) Enter 500 k for the Temperature.
(d) Click Initialize.

9. Solution:

Start the calculation.
Run Calculation
(a) Set Time Step Size to 0.0001.
(b) Set the Number of Time Steps to 3000
(c) Click Calculate.

5. RESULTS

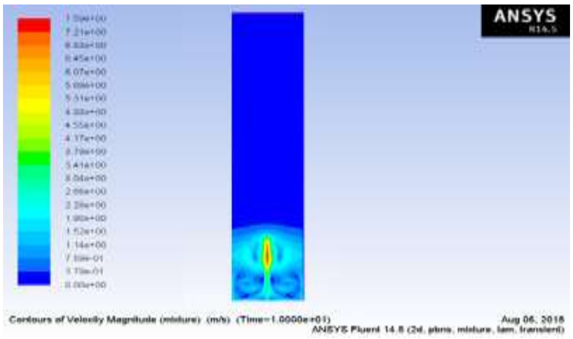


Figure no. 1 Contours of Velocity magnitude (Liquid) in time step bubble formation process

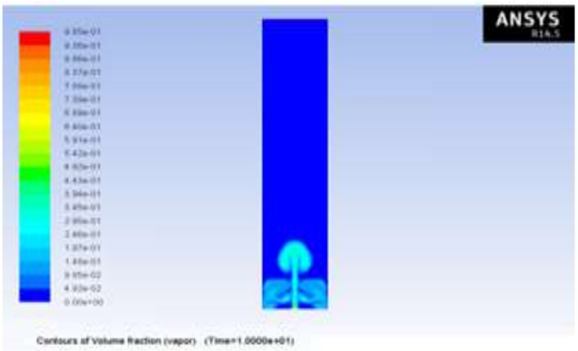


Figure no. 2 Complete bubbles formation in vapour form

APPLICATION OF NUCLEATE BOILING

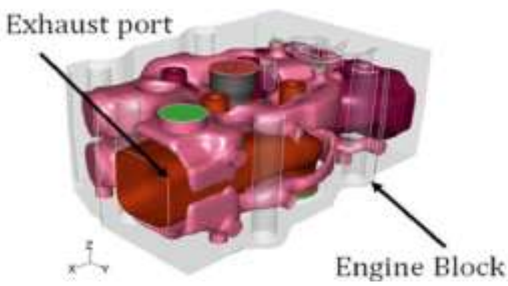


Figure 21 Engine block assembly

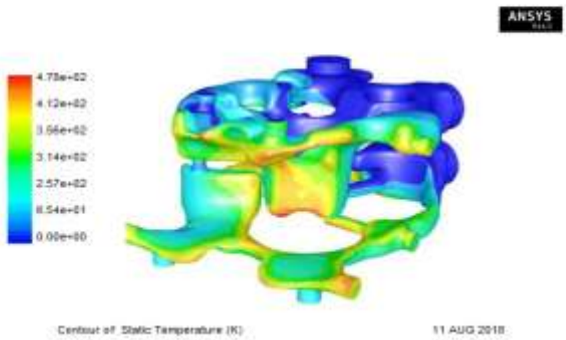


Figure 22 Temperature profile in Engine Surface

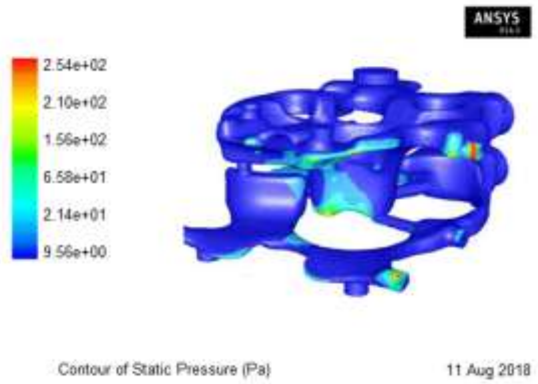


Figure 23 Pressure profile in Engine Surface

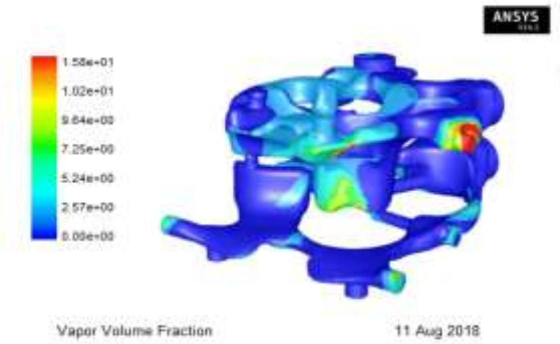


Figure 24 Volume fraction of vapor profile in Engine Surface

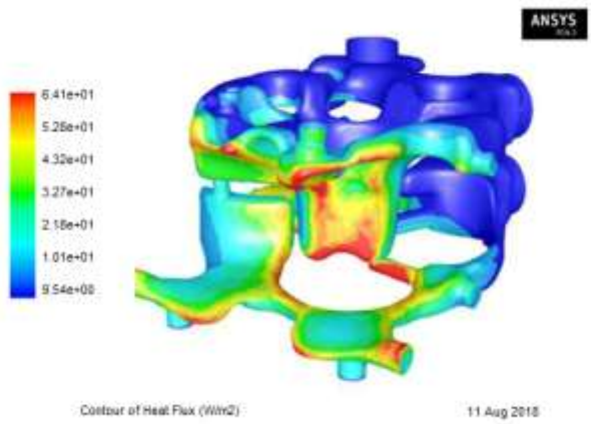


Figure 25 Contour of Heat flux in Engine Surface

6. CONCLUSIONS

In present study we investigate Firstly; isothermal turbulent bubble flow is mechanistically modelled in a solver named my Two Phase Euler Foam Adiabatic. The conservation equations of mass and momentum are solved for the two phases, taking special care in the modelling of the interfacial forces. The turbulence phenomena are described by a classical k-epsilon model in combination with standard wall functions for the near-wall treatment. Furthermore, an interfacial area concentration equation is solved and two different models for its sink- and source terms (corresponding to bubble coalescence and bubble breakup) have been investigated. Secondly, a solver named my Two Phase Euler Foam Boiling has been developed based on the first solver in order to model a heated wall leading to sub cooled nucleate boiling and subsequent condensation in the sub cooled liquid. Additional terms accounting for the phase change have been included in the mass and momentum conservation equations as well as in the interfacial area equation. Assuming the gas phase being at saturation conditions, only one energy equation for the liquid phase needs to be solved.

Cooling plays an important role in optimal performance of an IC engine. Insufficient cooling results into degraded performance, reduced power and, in long run, may result into engine failure due to wear, cracking or warping. The objectives of IC engine cooling jacket design are as follows:

• Keep wall temperatures of different components within acceptable levels to avoid loss of power and failure due to thermal stresses.
• Avoid flow mal distribution in the cooling jacket. Since engine jacket has complex flow passages, there is a possibility of stagnant zone or high velocity zones. While the former may cause hot spots, the latter may result in higher pressure drop
• Minimize the pressure drop

There has been a change in the thermal management of IC engines where engineers now like to harness the superior heat transfer rates available when limited and controlled nucleate boiling is used to remove heat from high temperature zones. Any flaws in the design of such systems, such as uncontrolled boiling that leads to Dry Out situation, can have an adverse effect on the cooling performance. A detailed engineering model of this process would allow engineers to weed out flawed designs early in the design process. In this paper, we proposed and validated a CFD model for this process. A CFD model is built using the commercial CFD solver ANSYS FLUENT. The mixture multiphase model is used to study subcooled nucleate boiling in IC engine cooling jackets. The departure of bubbles enhances heat transfer at walls, which is captured using the empirical correlation. Volumetric mass transfer is modeled using the inbuilt evaporation-condensation model. Results obtained from heat transfer in channels are compared with experimental results available in the literature for a range of operating pressures, different inlet sub-cooling and different inlet flow velocities. The predicted heat fluxes are in good agreement with experimental data. Results from a typical I.C. engine cooling jacket geometry are also presented.

Outcomes

In this Study, the distribution of the velocity field, pressure, and heat transfer coefficient (HTC) in the cooling water jacket of a diesel engine has been investigated through three-dimensional CFD method using ANSYS/Fluent software. Moreover, by presenting mathematical models based on, the numerical simulation of sub cooled boiling in the water jacket has been considered in this study. The significant point is the simultaneous solution of the structure and fluid in this analysis. The results show that:

• Using nucleate boiling flow regime can increase the heat transfer coefficient, which results in dissipating more heat from the surface and provides a more uniform temperature distribution for the engine.
• The temperature at which the nucleate boiling will start increases by increasing water flow velocity.
• The effect of pressure on decreasing or increasing the heat transfer rate is very important when the heat transfer occurs through boiling. In this way, high pressure can delay the start of film boiling.
• The low velocity of the coolant fluid and the lack of design of the vortex path behind the exhaust valves lead to an increase in the fluid temperature in the seat of the exhaust valve and around it, especially the narrow bridge between the two valves.
• The comparison of the thermal flux passing through the wall of the cylinder head with and without considering the boiling phenomenon shows that the occurrence of the boiling phenomenon can significantly increase the heat flux.

7. REFERENCES

[1]. Th. Frank*, P. Beckstein*, C. Lifante*, A.D. Burns-Prediction of Subcooled Wall Boiling in a Heated Annulus Including Conjugate Heat Transfer in the Central Heated Rod 7 th International Conference on Multiphase Flow ICMF 2010, Tampa, FL USA, May 30-June 4, 2010

[2]. K. Jagannath, Akhilesh Kotian, S. S. Sharma, Achutha Kini U., P. R. Prabhu "Investigation of Bubble Growth during Nucleate Boiling Using CFD" World Academy of Science, Engineering and Technology International Journal of Mechanical and Mechatronics Engineering Vol:9, No:6, 2015

[3]. Mohammad Amin Abdollahi1, Abtin Ataei2, Mohammad Heydari3 Water Boiling Heat Transfer in Vertical Jacketed Pipe: A CFD Model , International Conference on Chemical, Civil and Environmental Engineering (CCEE-2015) June 5-6, 2015 Istanbul (Turkey)

[4]. B. Končar E. Krepper Y. Egorov CFD MODELING OF SUBCOOLED FLOW BOILING FOR NUCLEAR ENGINEERING APPLICATIONS International Conference Nuclear Energy for New Europe 2005 Bled, Slovenia, September 5-8, 2005 .

[5]. S. Šarić, B. Basara, S. Santner High-fidelity heat transfer modeling for automotive CFD applications International Multidimensional Engine Modeling User's Group Meeting at the SAE Congress April 9, 2018, Detroit, MI

[6]. Kazuaki KOBAYASHI* Osamu NAKAMURA Yoichi HARAGUCHI- Water Quenching CFD (Computational Fluid Dynamics) Simulation with Cylindrical Impinging Jets NIPPON STEEL & SUMITOMO METAL TECHNICAL REPORT No. 111 MARCH 2016

[7]. Hariswaran Sitaraman, Gilberto Moreno, LOCAL-SCALE SIMULATIONS OF NUCLEATE BOILING ON MICROMETERFEATURED SURFACES, Proceedings of ASME Summer Heat Transfer Conference SHTC2017 July 9-14, 2017, Bellevue, Washington, USA

[8]. Hemant Punekar and Saurish Das, Numerical Simulation of Subcooled Nucleate Boiling in Cooling Jacket of IC Engine, SAE International by ANSYS Inc, Monday, September 14, 2015.

[9]. Hussam Jouhara , Bandar Fadhl , Luiz C. Wrobel , THREE-DIMENSIONAL CFD SIMULATION OF GEYSER BOILING IN A TWO-PHASE CLOSED THERMOSYPHON

[10]. Baozhi Suna, Xiang Yua, Shanghua Liua,b, Jianxin Shia, Longbin Yanga,⁇, Guolei Zhanga, Peng Zhanga

[11]. H.B. Zhang Y.Y. Yan , Y.Q. Zu \,Numerical modelling of EHD effects on heat transfer and bubble shapes of nucleate boiling, Applied Mathematical Modelling 34 (2010) 626–638

[12]. C.D. Moyes*, M.L. Sharma, C. Lyons, S.C. Leary, M. Leon, A. Petrie, S.G. Lund, B.L. Tufts, Origins and consequences of mitochondrial decline in nucleated erythrocytesBiochimica et Biophysica Acta 1591 (2002) 11 – 20

[13]. G.H. Yeoh , J.Y. Tu, Two-fluid and population balance models for subcooled boiling flowAustralian Nuclear Science and Technology Organisation (ANSTO), B40, ANSTO, Private Mail Bag 1, Menai, NSW 2234, Australia b School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, Vic. 3083, Australia

[14]. Zhihui Xie, Lingen Chen⁎ , Fengrui Sun Geometry optimization of T-shaped cavities according to constructal theory Postgraduate School, Naval University of Engineering, Wuhan 430033, China Mathematical and Computer Modelling 52 (2010) 1538–1546

[15]. Vedanth Srinivasan , Kil-Min Moon , David Greif , De Ming Wang , Myung-hwan Kim, Numerical simulation of immersion quenching process of an engine cylinder headNumerical simulation of immersion quenching process of an engine cylinder headAdvanced Simulation Technologies, AVL Powertrain Engineering Inc., Plymouth, MI 48170, USA b Advanced Technology and Analysis Team, R&D Div. for Hyundai Motor Company and Kia Motors Corp., 772-1 Jangduk-Dong Hwaseong, Republic of Korea c AVL Advanced Simulation Technologies (AST), Maribor, Slovenia

[16]. Ci Chu,1 Kun Qu,1 Franklin L. Zhong,2 Steven E. Artandi,2 and Howard Y. Chang Genomic Maps of Long Noncoding RNA Occupancy Reveal Principles of RNA-Chromatin Interactions 1Howard Hughes Medical Institute and Program in Epithelial Biology, Stanford University School of Medicine, Stanford, CA 94305, USA.

[17]. J. Wobst a,b , Apurwa Sharma d , Marc I. Diamond d , Erich E. Wanker b,⇑ , Jan Bieschke The green tea polyphenol ()-epigallocatechin gallate prevents the aggregation of tau protein into toxic oligomers at substoichiometric ratios Heike .

[18]. Polyamines Loı¨c Hamon,* Philippe Savarin, Patrick A. Curmi, and David Pastre´* Rapid Assembly and Collective Behavior of Microtubule Bundles in the Presence of Polyamines Biophysical Journal Volume 101 July 2011 205–216

[19]. PKim Lau Nielsen , Viggo Failure by void coalescence in metallic materials containing primary and secondary voids subject to intense shearing, International Journal of Solids and Structures 48 (2011) 1255–1267

[20]. Baoyu Chen,1 Klaus Brinkmann, The WAVE Regulatory Complex Links Diverse Receptors to the Actin Cytoskeleton, Cell 156, 195–207, January 16, 2014 ª2014 Elsevier Inc