สัญญาเลขที่ RTA4980004

รายงานวิจัยฉบับสมบูรณ์

โครงการ

งานวิจัยขั้นสูงทางวิศวกรรมการปรับอากาศ และการทำความเย็น

Advanced Research on Refrigeration and Air-Conditioning Engineering

ผู้วิจัย

ศ. ดร. สมชาย วงศ์วิเศษ

ภาควิชาวิศวกรรมเครื่องกล คณะวิศวกรรมศาสตร์

มหาวิทยาลัยเทคโนโลยีพระจอมเกล้าธนบุรี

91 สุขสวัสดิ์ 48, ทุ่งครุ, กทม. 10140

โทรศัพท์: 02-470-9115 โทรสาร: 02-470-9111

E-Mail: somchai.won@kmutt.ac.th

สนับสนุนโดยสำนักงานกองทุนสนับสนุนการวิจัย

ผลงานโดยสรุป

งานที่ผ่านมาของกลุ่มวิจัยคือเน้นการศึกษาคุณลักษณะทางการไหล (เช่น รูปแบบการไหล ความดันที่ลดลง) และการถ่ายเทความร้อนและมวลของการไหลสองสถานะของสารทำความเย็นในขณะเปลี่ยนสถานะที่เกิดขึ้นในระบบท่อ ทั้งในเชิงการทดลองและเชิงทฤษฎี ได้สร้างองค์ความรู้ใหม่โดยมีผลงานทางวิชาการที่ก้าวหน้าทันสมัย ผลงานได้ถูกนำไปอ้างอิงโดยนักวิทยาศาสตร์ในต่างประเทศ สามารถพัฒนาจากงานวิจัยพื้นฐาน สู่งานวิจัยประยุกต์ซึ่งใช้กับอุตสาหกรรมในประเทศได้ ได้สร้างเครือข่ายงานวิจัยทั้งในประเทศและต่างประเทศอันจะมีประโยชน์ต่อคนรุ่นหลังในการแลกเปลี่ยนวิทยาการที่ก้าวหน้า กลุ่มวิจัย ได้นำศาสตร์ของ การถ่ายเทความร้อนและมวล กลศาสตร์ของไหล การไหลสองสถานะ และเธอร์โมไดนามิกส์ มาประยุกต์ เพื่อการเพิ่มประสิทธิภาพและลดการใช้พลังงานของระบบปรับอากาศและระบบทำความเย็น ความรู้ใหม่ต่างๆที่ได้จากงานวิจัยพื้นฐานเหล่านี้ สามารถนำไปประยุกต์ใช้ในการออกแบบอุปกรณ์แลกเปลี่ยนความร้อนต่างๆให้มีสมรรถนะสูงขึ้นได้ ซึ่งเป็นประโยชน์ต่ออุตสาหกรรมโดยตรง

โดยปกติกระบวนการแลกเปลี่ยนความร้อนระหว่างของไหลสองชนิดที่มีอุณหภูมิแตกต่างกัน สามารถทำได้โดยใช้อุปกรณ์ที่เรียกว่า อุปกรณ์แลกเปลี่ยนความร้อน ถ้าอุปกรณ์แลกเปลี่ยนความร้อนถูกออกแบบมาอย่างเหมาะสม การถ่ายเทความร้อนก็จะเป็นไปอย่างมีประสิทธิภาพ การเพิ่มความสามารถในการถ่ายเทความร้อน ถือเป็นสิ่งหนึ่งที่จะช่วยเพิ่มสมรรถนะของอุปกรณ์แลกเปลี่ยนความร้อน ซึ่งก็เท่ากับเป็นการประหยัดพลังงานหรือเป็นการใช้พลังงานอย่างคุ้มค่า กลุ่มวิจัยได้ศึกษาวิธีพัฒนา คอนเดนเซอร์ และ อีวาโปเรเตอร์ ซึ่งถือเป็นอุปกรณ์แลกเปลี่ยนความร้อนที่อยู่ในวงจรการปรับอากาศและการทำความเย็น ให้สามารถแลกเปลี่ยนความร้อนระหว่าง สารทำความเย็นกับอากาศได้ดียิ่งขึ้น การศึกษาสามารถทำทั้งจากการทดลองและจากการสร้างแบบจำลองทางคณิตศาสตร์ ตัวอย่างของงานวิจัยได้แก่ การออกแบบอุปกรณ์แลกเปลี่ยนความร้อนแบบใหม่ๆซึ่งอาจเป็นแนวทางในการนำไปประยุกต์ใช้ต่อไปในอนาคต การศึกษาถึงผลของ รูปร่างของครีบที่ใช้ในการถ่ายเทความร้อน ความหนาของครีบ ระยะห่างระหว่างครีบ วัสดุที่ใช้ ลักษณะการวางท่อ ต่อสมรรถนะทางการถ่ายเทความร้อน และการลดลงของความดันของอากาศขณะไหลผ่านอุปกรณ์แลกเปลี่ยนความร้อน ทั้งที่ภาวะที่ผิวท่อด้านนอกของคอนเดนเซอร์มีสภาพแห้ง และที่ภาวะที่ผิวท่ออีวาโปเรเตอร์เปียกด้วยน้ำที่เกิดจากการควบแน่น

กลุ่มวิจัยยังสนใจศึกษาในส่วนอุปกรณ์ระเหยซึ่งถือเป็นอุปกรณ์ที่สำคัญอีกตัวหนึ่งในวงจรการปรับอากาศและทำความเย็น อาทิ การนำอีเจ็กเตอร์ และ การนำท่อคาปิลลารีทั้งแบบแอเดียแบติกและ แบบ นอนแอเดียแบติก มาใช้เป็นอุปกรณ์ระเหย โดยสามารถพัฒนาแบบจำลองทางคณิตศาสตร์เพื่อจำลองการไหลของสารทำความเย็นในท่อคาปิลลารี ซึ่งมีการการไหลสองสถานะเป็นกลไกสำคัญ แบบจำลองดังกล่าวสามารถนำไปใช้ในการออกแบบหาขนาดท่อคาปิลลารีที่เหมาะสมที่ใช้กับสารทำความเย็นประเภทต่างๆ ที่ทำงานที่ภาวะต่างๆตามแต่กำหนด และเพื่อให้สะดวกต่อการใช้งานใน

ภาคสนาม คณะวิจัยสามารถพัฒนา แผนภูมิ เพื่อใช้เลือกขนาดท่อคาปิลลารี ที่เหมาะสมกับสารทำความเย็นและภาวะการใช้งานจริง ในส่วนของสารทำงานในวงจรการปรับอากาศและการทำความเย็นซึ่งเราเรียกว่าสารทำความเย็นกลุ่มวิจัยได้ศึกษาคุณลักษณะเฉพาะในการไหลของสารทำความเย็นทางเลือกใหม่ทั้งจากการทดลองและจากการพัฒนาแบบจำลองทางคณิตศาสตร์ ทั้งในด้านการถ่ายเทความร้อนและด้านกลศาสตร์ของไหล ในขณะที่สารทำความเย็นระเหย และควบแน่นภายในท่อเรียบและท่อที่ผิวภายในไม่เรียบ โดยศึกษาทั้งกับสารทำความเย็นบริสุทธิ์และกับสารทำความเย็นที่มีน้ำมันหล่อลื่นผสมซึ่งตรงกับสภาพการใช้งานจริง ผลลัพธ์จากการศึกษาทางด้านการถ่ายเทความร้อนจะอยู่ในรูปของสัมประสิทธิ์การถ่ายเทความร้อน สำหรับในเชิงกลศาสตร์ของไหลจะเสนอผลในรูปของ รูปแบบการไหลสองสถานะซึ่งเป็นไปได้หลากหลายรูปแบบการไหล แฟกเตอร์ความเสียดทาน และ ความดันลด สหสัมพันธ์ที่พัฒนาได้ทั้งสัมประสิทธิ์การถ่ายเทความร้อนและแฟกเตอร์ความเสียดทาน สามารถนำไปใช้ในการออกแบบอีวาโปเรเตอร์ และ คอนเดนเซอร์ ได้โดยตรง นอกจากนั้นแบบจำลองทางคณิตศาสตร์ที่พัฒนาขึ้นมาสามารถนำไปเป็นแนวทางและใช้ดัดแปลงเพื่อการออกแบบอุปกรณ์แลกเปลี่ยนความร้อนที่มีลักษณะแตกต่างออกไปได้

Research summary

The research work of our research group includes two-phase flow, heat transfer enhancement and thermal system design. Generally, the process of heat exchange between two kinds of fluid of different temperatures can be accomplished with equipment called a heat exchanger. A well-designed heat exchanger can transfer heat effectively. Heat transfer enhancement can increase the performance of a heat exchanger through the saving or cost-effective use of energy. Since Thailand is situated in a tropical zone, air conditioners and refrigerators are considered necessary appliances in Thai daily life. Additionally, Thailand has become an important production base for air conditioner and refrigerator manufactoring, as well as a top exporter of these products to the world market. Therefore, research and development, as well as the preparation of specialists - in this field is necessary.

Our research group has applied the sciences of heat and mass transfer, fluid mechanics, and thermodynamics in developing higher-performance heat exchangers, the main focus being on increasing the efficiency and reducing the power consumption of air conditioners and refrigerators. The new body of knowledge gained from this basic research can be applied in designing high-performance equipment that is directly useful in the industry. The research team has tried to study and develop condensers and evaporators, which are heat exchangers used in the cycle of refrigeration and air conditioning, to enable better heat exchange between the refrigerant and surrounding air. These studies covered both real experiments and simulation with mathematical models. One example of these studies is the design of new heat exchangers that can later be applied in the industry. Another important example is the study of the effects that various fin shapes, fin thickness, fin spacing, used materials and tube arrangement, used in heat exchangers, have on the efficiency of the heat transfer and pressure drop of air while flowing through heat the exchangers - both in the condition of the outer surface of the condenser being dry and in the condition of the surface of the evaporator being wet due to condensation.

Moreover, the research team studied the expansion device, which is another important part functioning in the refrigeration and air conditioning cycles. The research used ejectors and both adiabatic and non adiabatic capillary tubes as expansion devices. Mathematical models were developed to simulate the flow of refrigerant in capillary tubes in which the two-phase flow was the key mechanism. These models have been proved reliable as results obtained from these models were similar to those presented in many published research articles. These models could be utilized in designing the appropriate sizes of capillary tubes used with various kinds of refrigerant in different working conditions. To facilitate field use, the research team also developed a selection chart enabling easy selection of capillary tube size suitable for chosen kinds of refrigerant and working conditions.

It is already known that even though the use of refrigerant is suitable for engineering aspects, it has some drawbacks in environmental aspects. In the near future, some refrigerants used nowadays will be banned. Thus, the research team has also studied the flow characteristics of alternative refrigerants both in terms of heat transfer and in terms of fluid mechanics. The flow characteristics were studied both while the refrigerants were flowing and evaporating, and while the refrigerants were flowing and condensing inside smooth pipes and non-smooth pipes. The studied refrigerants included both pure refrigerant and a mixture of refrigerant and lubricant oil. These studies covered both real experiments and developments of mathematical models. Results of heat transfer studies were presented in the form of a heat transfer coefficient, whereas results of fluid mechanics studies were presented in the form of a pressure drop plus the flow pattern occurring during phase changes. The developed correlations - both in terms of heat transfer coefficient and pressure drop - could be directly applied to the design of evaporator and condenser. The developed mathematical models could also be directly utilized or modified in the design of different heat exchangers.

ทุนเมธีวิจัยอาวุโส สัญญาเลขที่ RTA4980004

งานวิจัยขั้นสูงทางวิศวกรรมการปรับอากาศและการทำความเย็น

สรุปรายงาน

หัวหน้าโครงการ : ศาสตราจารย์ ดร. สมชาย วงศ์วิเศษ

รายงานในช่วงวันที่ 20 ก.ค. 2549 ถึง 24 มิ.ย. 2553

1. <u>การดำเนินงาน</u>

ได้ดำเนินการตามแผนที่วางไว้ทุกประการ โดยทำการวิจัยในหัวข้อต่อไปนี้

1.1 Micro-Channel Heat Exchanger used for Refrigeration and Air Conditioning

1.2 Heat and Mass Transfer Characteristics of Fin-and-Tube Evaporators used in Air Conditioning system

1.3 Saturated Pool Boiling Characteristics of New Alternative Fluorine-Based Refrigerants

1.4 Pool Boiling Characteristics of New Alternative Refrigerants "Nano-Fluids"

1.5 Hydrocarbon Mixtures to Replace Refrigerants used in an Automotive Air Conditioning System

1.6 Heat Transfer, Pressure Drop and Effect of Flow Direction of Working Fluids flowing Through a Plate Heat Exchanger Simulated as Evaporator and Condenser

1.7 Characteristics of Refrigerant HFC-134a Flowing Through Micro-Scale Short Tube Orifices

1.8 Vertical Upward Gas-Liquid Flow in Sinusoidal Wavy Channels

1.9 Heat Transfer and Pressure Drop Characteristics of Refrigerant HFC-134a During Evaporation and Condensation in Annulus Coil Evaporator and Condenser

1.10 Two-Phase Flow Patterns, Void Fraction and Pressure Drop in a Narrow Concentric Annuli

1.11 Performance of a Vapor Compression Refrigeration Cycle Using a Two-Phase Ejector as an Expansion Device

1.12 Experimental and Theoretical Investigations of Two-Phase Flow in Micro-Channels

1.13 Effects of Electrohydrodynamics on the In-Tube Condensation and Boiling Heat Transfer Enhancement of a New Alternative Refrigerant in Smooth and Micro-Fin Tubes

1.14 Two-Phase Flow Patterns and Pressure Drop in a Small Circular Tube Vertical Bend used in Refrigeration and Air conditioning

1.15 Forced Convective Heat Transfer Characteristic of New Alternative Refrigerants "Nano-fluid"

1.16 Selection Charts for Fluorine-Based Capillary Tubes Used in Refrigerator and Air-Conditioner

1.17 Effect of Geometric Parameters on the Air-Side Performance of Crimped Finned Tube Condenser used in the Air-Conditioning System

1.18 Heat Transfer Performance of a Helically Coiled Heat Exchanger

1.19 Performance of New Alternative Refrigerants Flowing Through Non-Adiabatic Capillary Tubes Used as An Expansion Device in Refrigeration and Air Conditioning System

1.20 Combined Refrigeration and Solar Energy Systems

2. ผลงานตีพิมพ์ในวารสารระดับนานาชาติ

1. Wongwises S, Naphon P. (2006) Heat transfer characteristics of a spirally coiled finned tube heat exchanger under dry-surface conditions. *Heat Transfer Eng* 27(1) : 25-34. อยู่ในโครงการปี 46

2. Kongtragool B, Wongwises S. (2006) Thermodynamic analysis of a Stirling engine including dead volumes of hot space, cold space and regenerator. *Renew Energy* 31 (3): 345-359. 52

3. Naphon P, Wongwises S. (2006) A review of flow and heat transfer characteristics in curves tubes. *Renew Sust Energy Rev* 10 (5): 463-490. อยู่ในโครงการปี 46

4. Chinnarasri C, Wongwises S. (2006) Flow patterns and energy dissipation over stepped chutes. *J Irrigation Drainage Eng – ASCE* 132(1): 70-76. × อยู่ในโครงการปี 46

5. Wongwises S, Kamboon A, Orachorn B. (2006) Experimental investigation of hydrocarbon mixtures to replace HFC-134a in an automotive air conditioning system. *Energy Convers Manage* 47 (11-12): 1644-1659. × อยู่ในโครงการปี 46

6. Pirompugd W, Wongwises S, Wang CC. (2006) Simultaneous heat and mass transfer characteristics for wavy fin-and-tube heat exchangers under dehumidifying conditions. *Int J Heat Mass Transfer* 49 (1-2): 132-143. 49

7. Kaensup W, Kulwong S, Wongwises S. (2006) A small scale pneumatic conveying dryer of rough rice. *Drying Technol* 24(1) : 105-113. × อยู่ในโครงการปี 46

8. Wongwises S, Polsongkram M. (2006) Evaporation heat transfer and pressure drop of HFC-134a in a helically coiled concentric tube-in-tube heat exchanger. *Int J Heat Mass Transfer* 49 (3-4): 658-670. 23

9. Wongwises S, Pipathanakul M. (2006) Flow pattern, pressure drop and void fraction of two-phase gas-liquid flow in an inclined narrow annular channel. *Exp Therm Fluid Sci* 30 (4): 345-354. 32

10. Wongwises S, Naphon P. (2006) Thermal performance of a spirally coiled finned tube heat exchanger under wet surface conditions. *J Mech Sci Technol* 20 (2): 212-226. × อยู่ในโครงการปี 46

11. Kuwannarat T, Wang CC, Wongwises S. (2006) Effect of fin thickness on the air-side performance of wavy fin-and-tube heat exchangers under dehumidifying conditions. *Int J Heat Mass Transfer* 49 (15-16): 2587-2596. × อยู่ในโครงการปี 46

12. Wongwises S, Kamsanam W, Wang CC. (2006) Flow patterns and pressure gradient of two-phase gas-liquid flow in vertical U-type return bends. *J Thermophysics Heat Transfer* 20 (3): 624-627. 34

13. Nilpueng K, Wongwises S. (2006) Flow pattern and pressure drop for vertical upward gas-liquid flow in sinusoidal wavy channels. *Exp Therm Fluid Sci* 30 (6): 523-534. 31

14. Pirompugd W, Wongwises S. (2006) Capillary tube sizing charts for fluorine-based refrigerants. *ASHRAE Transactions* 112 : 680-689. × อยู่ในโครงการปี 46

15. Assavapusitkul W, Joraden J, Wongwises S. (2006) CFD modeling of a gas-liquid mixing in T-junction. *Asian J Energy Envi* (in press) × อยู่ในโครงการปี 46

16. Kaensup W, Kulwong, S., Wongwises S. (2006) Comparison of drying kinetics of paddy using a pneumatic conveying dryer with and without a cyclone. *Drying Technol* 24(8) : 1039-1045. × อยู่ในโครงการปี 46

17. Laohalertdecha S, Wongwises S. (2006) Effects of EHD on heat transfer enhancement and pressure drop during two-phase condensation of pure R-134a at high mass flux in a horizontal micro-fin tube. *Exp Therm Fluid Sci* 30(7): 675-686. 15

18. Wongwises S, Polsongkram M. (2006) Condensation heat transfer and pressure drop of HFC-134a in a helically coiled concentric tube-in-tube heat exchanger. *Int J Heat Mass Transfer* 49(23-24): 4386-4398. 18

19. Trisaksri V, Wongwises S. (2007) Critical review of heat transfer characteristics of nanofluids. *Renew Sust Energy Rev* 11 (3): 512-523. 13

20. Kongtragool B, Wongwises S. (2007) Performance of low-temperature differential Stirling engine. *Renew Energy* 32 (4): 547-566. 53

21. Duangthongsuk W, Wongwises S. (2007) A critical review of convective heat transfer of nanofluids. *Renew Sust Energy Rev* 11(5): 797-817. 2

22. Laohalertdecha S, Naphon P., Wongwises S. (2007) A review of electrohydrodynamic enhancement of heat transfer. *Renew Sust Energy Rev* 11 (5): 858-876. 5

23. Pirompugd W, Wang CC, Wongwises S. (2007) Finite circular fin method for heat and mass transfer characteristics for plain fin-and-tube heat exchangers under fully and partially wet surface conditions. *Int J Heat Mass Transfer* 50 (3-4): 552-565. 29

24. Laohalertdecha S, Wongwises S. (2007) Effect of EHD on heat transfer enhancement during condensation of R-134a at high mass flux in a horizontal smooth tube. *Heat and Mass Transfer* 43 (9): 871-880.

25. Rattanadecho P, Wongwises S. (2007) Simulation of freezing of water-saturated porous media in a rectangular cavity under multiple heat sources with different temperature using a combined transfinite interpolation and PDE methods. *Computer and Chemical Engineering* 31 (4): 318-333.

26. Muangnoi T, Asvapoositkul W, Wongwises S. (2007) An exergy analysis on the performance of counterflow wet cooling tower. *Applied Thermal Engineering* 27 (5-6): 910-917.

27. Bovornsethanan S, Wongwises S. (2007) Drying parawood with superheated steam. *American Journal of Applied Science* 4 (4):215-219.

28. Pirompugd W, Wang CC, Wongwises S. (2007) A fully wet and fully dry tiny circular fin method for heat and mass transfer characteristics for plain fin-and-tube heat exchangers under dehumidifying conditions. *ASME J Heat Transfer* 129 (9): 1256-1267.

29. Pirompugd W, Wang CC, Wongwises S. (2007) A new reduction method for simultaneous heat and mass transfer characteristics for wavy fin-and-tube heat exchangers under dehumidifying conditions. *AIAA J Thermophysics Heat Transfer* 21 (2): 361-371.

30. Chaiwongsa P, Wongwises S. (2007) Effect of throat diameter of the ejector on the performance of the refrigeration cycle using a two-phase ejector as an expansion device. *J Refrigeration* 30(4): 601-608.

31. Kongtragool B, Wongwises S. (2007) Performance of a twin power piston low temperature differential Stirling engine powered by a solar simulation. *Solar Energy* 81 (7) 884-895.

32. Wongwises S. (2007) Engineering Studies in Thailand: A critical look at the past, Important lessons for the future. *Science Asia* 33 (1): 47-50.

33. Laohalertdecha S, Wongwises S. (2007) A comparison of the effect of EHD on the condensation heat transfer of HFC-134a inside smooth and micro-fin tube. *Journal of Mechanical Science and Technology* 21(12): 2168-2177.

34. Chaiwongsa P, Wongwises S. (2008) Experimental study on R-134a refrigeration system using a two-phase ejector as an expansion device. *Applied Thermal Engineering* 28 (5-6): 467-477.

35. Saisorn S, Wongwises S. (2008) A review of two-phase gas-liquid adiabatic flow characteristics in micro-channels. *Renew Sust Energy Rev* 32 (3): 748-760.

36. Chinnarasri C, Wongwises S. (2008) Closure to "Flow patterns and energy dissipation over various stepped chutes *ASCE Journal of Irrigation and Drainage Engineering* 134 (1) :117. 11

37. Rattanadecho P, Wongwises S. (2008) Moving boundary-moving mesh analysis of freezing process in water-saturated porous media using a combined transfinite interpolation and PDE mapping methods. *ASME Journal of Heat Transfer* 130 (1)012601-10. . 40

38. Saenmart, K., Cheowuttikul, P., Suriyawong, A., Wongwises S. (2008) Onset of flooding in a small diameter tube. *Int. Comm. Heat Mass Transfer* 35 (4): 458-465. 42

39. Saisorn S, Wongwises S. (2008) Flow pattern, void fraction and pressure drop of two-phase air-water flow in a horizontal circular micro-channel. *Experimental Thermal and Fluid Sciences* 32 (3): 748-760. 6

40. Aschariyaphotha N., Wongwises P., Wongwises S., Humphries UW., You XB., (2008) Simulation of seasonal circulations and thermohaline variabilities in the Gulf of Thailand. *Ad*
vances in Atmospheric Sciences 25 (3): 489-506 48

41. Saechan P, Wongwises S. (2008) Optimal configuration of cross flow plate finned tube condenser based on the second law of thermodynamics. *International Journal of Thermal Science* 2008; 47 (11): 1473-1481. . 43

42. Kongtragool B, Wongwises S. (2008) A four power piston low temperature differential Stirling engine by using simulated solar energy as a heat source. *Solar Energy* 82 (6):493-500. 3

43. Muangnoi T, Asvapoositkul W, Wongwises S. (2008) Effects of inlet-relative humidity and inlet-temperature on the performance of a counterflow wet cooling tower based on exergy analysis. *Energy Conversion and Management* 49 (10): 2795-2800. 21

44. Srisawas, K., Wongwises S. (2008) Heat transfer characteristics of a new helically coiled crimped spiral finned tube heat exchanger *Heat and Mass Transfer* 45 (4):381-391. 36

45. Pirompugd W, Wang CC, Wongwises S. (2008) Finite circular fin method for wavy fin-and-tube heat exchangers under fully and partially wet surface conditions. *Int. J Heat Mass Transfer* 51 (15-16): 4002-4017. 30

46. Kongtragool B, Wongwises S. (2008) Testing of a low-temperature differential Stirling engine by using actual solar energy. International Journal of Green Energy 5 (6):491-507. 50

47. Dalkilic, A.S., Laohalertdecha, S, Wongwises, S. (2008) Two-phase friction factor in vertical downward flow at high mass flux region of refrigerant HFC-134a during condensation, *International Communications in Heat and Mass Transfer* 35 (9):1147-1152. 53

48. Duangthongsuk, W., Wongwises, S. (2008) Effect of thermophysical properties models on the prediction of the convective heat transfer coefficient for low concentration nanofluid, *Internationl Communications in Heat Mass Transfer* 35 (10): 1320-1326. ๑๗

49. Saisorn, S., Woñgwises, S. (2008) An inspection of viscosity model for homogeneous two-phase flow pressure drop prediction in a horizontal circular micro-channel, *International Communications in Heat and Mass Transfer* 35 (7): 833-838. ๑๐

50. Dalkilic, A.S., Laohalertdecha, S., Wongwises, S. (2008) Effect of void fraction models on the two-phase friction factor of R134a during condensation in vertical downward flow in a smooth tube, *International Communications in Heat and Mass Transfer* 35: 921-927. ๒๐

51. Srisawas, K., Wongwises S. (2009) Heat transfer characteristics of a new helically coiled crimped spiral finned tube heat exchanger *Heat and Mass Transfer* 45 (4):381-391. ๓๖

52. Daungthongsuk, W., Wongwises, S. (2009) Heat Transfer Enhancement and Pressure Drop Characteristics of TiO_2-Water Nanofluid in a Double-Tube Counter Flow Heat Exchanger, International Journal of Heat and Mass Transfer 52 (7-8): 2059-2067. ๓๓

53. Dalkilic, A.S., Yildiz, S., Wongwises, S. (2009) Experimental investigation of convective heat transfer coefficient during downward laminar flow condensation of R134a in a vertical smooth tube, *Int J Heat Mass Transfer* 52 (1-2): 142-150. ๒๕

54. Trisaksri V, Wongwises S. (2009) Nucleate Pool Boiling Heat Transfer of an Alternative Refrigerant with Nanoparticle Suspension, *International Journal of Heat and Mass Transfer* 52 (5-6): 1582-1588. ๔๑

55. Pirompugd W, Wang CC, Wongwises S. (2009) A review on reduction method for heat and mass transfer of fin-and-tube heat exchangers under dehumidifying conditions, *International Journal of Heat and Mass Transfer* 52 (9-10): 2370-2378. ๒

56. Kaew-on, J., Wongwises, S., (2009) Experimental investigation of evaporation heat transfer coefficient and pressure drop of R-410A in a multiport mini-channel, *International Journal of Refrigeration* 32 (1): 124-137. ๒๖

57.Saisorn S, Wongwises S. (2009) An experimental investigation of two-phase air-water flow through a horizontal circular micro-channel, *Experimental Thermal and Fluid Sciences*, 33 (2):306-315. ๑

58. Dalkilic, A.S., Laohalertdecha, S, Wongwises, S. (2009) Effect of void fraction models on the film thickness of R134a during downward condensation in a vertical smooth tube, *International Communications in Heat and Mass Transfer* 36 (2): 172-179. ๑๙

59.Daungthongsuk, W., Wongwises, S. (2009) Measurement of temperature-dependent thermal conductivity and viscosity of TiO_2-water nanofluids, *Experimental Thermal and Fluid Science* 33 (4): 706-714.

60. Nilpueng, K, Wongwises, S. (2009) Experimental investigation of flow characteristics of HFC134a through short tube orifices, *International Journal of Refrigeration* 32 (5): 854-864.

61.Dalkilic, A.S., Wongwises, S., (2009) Intensive Literature Review of Condensation inside Smooth and Enhanced Tubes, *International Journal of Heat and Mass Transfer*, 52 (15-16):3409-3880.

62. Kaewnai S, Chamaoot M, Wongwises S. (2009) Predicting performance of radial flow type impeller of centrifugal pump using CFD code. *Journal of Mechanical Science and Technology*, 23 (6): 1620-1627.

63. Posew K., Laohalertdecha, S., Wongwises S. (2008) Evaporation heat transfer enhancement of R-134a flowing inside smooth and micro-fin tubes using the electrohydrodynamic technique. *Energy Conversion and Management*, 50 (7):1851-1861.

64. Naphon, P., Klangchart, S., Wongwises, S. (2009) Numerical investigation on the heat transfer and flow in the mini-fin heat sink for CPU, *International Communications in Heat and Mass Transfer* 36 : 834-840.

65 Nilpueng, K, Wongwises, S., (2009) Flow pattern, mass flow rate, pressure distribution, and temperature distribution of two-phase flow of HFC-134a inside short-tube orifices, *International Journal of Refrigeration* 32 (8): 1864-1875.

66. Dalkilic, A.S., Laohalertdecha, S., Wongwises, S. (2009) Experimental investigation of heat transfer coefficient of R134a during condensation in vertical downward flow at high mass flux in a smooth tube, *International Communications in Heat and Mass Transfer* 36 : 1036-1043.

3. <u>บทความทางวิชาการลักษณะอื่น ๆ</u>

<u>สิทธิบัตรที่ยื่นจดกับกรมทรัพย์สินทางปัญญา</u>

1. วิธีการหาความยาวของท่อคาปิลลารีที่ใช้ในระบบปรับอากาศและระบบทำความเย็น
   (Patent no. 21195)

2. เครื่องยนต์สเตอร์ลิงแบบสองลูกสูบ
   (คำขอรับสิทธิบัตรเลขที่ 0601003084)

3. เครื่องยนต์สเตอร์ลิงแบบลูกสูบกำลังคู่
   (คำขอรับสิทธิบัตรเลขที่ 0601003085)

4. เครื่องยนต์สเตอร์ลิงแบบลูกสูบกำลังสี่ลูกสูบ
(คำขอรับสิทธิบัตรเลขที่ 0601003086)

5. ระบบการผลิตกำลังด้วยเครื่องยนต์สเตอร์ลิงพลังแสงอาทิตย์
(คำขอรับสิทธิบัตรเลขที่ 0601003087)

6. อุปกรณ์แลกเปลี่ยนความร้อนแบบท่อขดแบบสปริงติดครีบ
(คำขอรับสิทธิบัตรเลขที่ 0601003088)

7. แคลอริมิเตอร์สำหรับวัดความเข้มรังสีอาทิตย์จากตัวรวมรังสีอาทิตย์
(คำขอรับสิทธิบัตรเลขที่ 0601003089)

4. <u>กิจกรรมอย่างอื่นที่เกี่ยวข้องกับงานวิจัย</u>

- เป็น Reviewer พิจารณาโครงการให้ ฝ่ายวิจัยเชิงวิชาการ สกว
- เป็น Chair person การประชุมประจำปีนักวิจัยรุ่นใหม่พบเมธีวิจัยอาวุโส
- เป็นกรรมการบัญญัติคำศัพท์ทางวิศวกรรมเครื่องกลของราชบัณฑิตยสถาน
- เป็นกรรมการพิจารณาผลงานเพื่อเข้าสู่ตำแหน่งทางวิชาการให้มหาวิทยาลัยต่าง ๆ
- เป็น reviewer พิจารณาบทความให้กับวารสารระดับชาติและระดับนานาชาติ
- เป็นวิทยากรให้องค์กร และ มหาวิทยาลัยต่าง ๆในเรื่องที่เกี่ยวกับงานวิจัย
- เป็นบรรณาธิการหนังสือเฉลิมพระเกียรติจัดทำโดย สกว.
- ได้รับการเชิดชูเกียรติให้เป็นอาจารย์ดีเด่นด้านวิชาการและการวิจัยในโอกาสครบรอบ 48 ปี แห่งการสถาปนามหาวิทยาลัยเทคโนโลยีพระจอมเกล้าธนบุรี

5. <u>ท่านได้พบอุปสรรคในการดำเนินงานหรือไม่ ถ้าพบกรุณาระบุว่ามีอะไรบ้างและได้แก้ไขอย่างไรบ้าง</u>..(ไม่มี)...................................

6. <u>ความเห็นและข้อเสนอแนะ</u>..............................(ไม่มี)...................................

(ศ. ดร. สมชาย วงศ์วิเศษ)
หัวหน้าโครงการ

Available online at www.sciencedirect.com

SCIENCE DIRECT

Renewable Energy 31 (2006) 345–359

RENEWABLE ENERGY

www.elsevier.com/locate/renene

Technical note

Thermodynamic analysis of a Stirling engine including dead volumes of hot space, cold space and regenerator

Bancha Kongtragool, Somchai Wongwises*

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, 91 Suksawas 48, Bangmod, Bangkok 10140, Thailand

Received 7 November 2004; accepted 22 March 2005
Available online 19 May 2005

Abstract

This paper provides a theoretical investigation on the thermodynamic analysis of a Stirling engine. An isothermal model is developed for an imperfect regeneration Stirling engine with dead volumes of hot space, cold space and regenerator that the regenerator effective temperature is an arithmetic mean of the heater and cooler temperature. Numerical simulation is performed and the effects of the regenerator effectiveness and dead volumes are studied. Results from this study indicate that the engine net work is affected by only the dead volumes while the heat input and engine efficiency are affected by both the regenerator effectiveness and dead volumes. The engine net work decreases with increasing dead volume. The heat input increases with increasing dead volume and decreasing regenerator effectiveness. The engine efficiency decreases with increasing dead volume and decreasing regenerator effectiveness.

Keywords: Stirling engine; Hot-air engine; Regenerative heat engine

* Corresponding author. Tel.: +662 470 9115; fax: +662 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0960-1481/$ - see front matter © 2005 Elsevier Ltd. All rights reserved.
doi:10.1016/j.renene.2005.03.012

Available online at www.sciencedirect.com

SCIENCE DIRECT

ELSEVIER

International Journal of Heat and Mass Transfer 49 (2006) 2587–2596

International Journal of HEAT and MASS TRANSFER

www.elsevier.com/locate/ijhmt

Effect of fin thickness on the air-side performance of wavy fin-and-tube heat exchangers under dehumidifying conditions

Thirapat Kuvannarat [a], Chi-Chuan Wang [b], Somchai Wongwises [a,*]

[a] *Department of Mechanical Engineering, Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab (FUTURE), King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

[b] *Energy and Resources Laboratory, Industrial Technology Research Institute, Hsinchu 310, Taiwan, ROC*

Received 11 July 2005; received in revised form 9 January 2006
Available online 20 March 2006

Abstract

This study investigated the effect of fin thickness on the air-side performance of wavy fin-and-tube heat exchangers under dehumidifying conditions. A total of 10 samples were tested with associated fin thickness (δ_f) of 0.115 mm and 0.25 mm, respectively. For a heat exchanger with two rows ($N = 2$) and fin pitch F_p of 1.41 mm, the effect of fin thickness on the heat transfer coefficient is more pronounced. The heat transfer coefficients for $\delta_f = 0.25$ mm is about 5–50% higher than those for $\delta_f = 0.115$ mm whereas the pressure drop for $\delta_f = 0.25$ mm is about 5–20% higher. The unexpected difference in heat transfer coefficient subject to fin thickness is attributable to better interactions between the directed main flow and the swirled flow caused by the condensate droplet for $\delta_f = 0.25$ mm. The maximum difference in heat transfer coefficients for $N = 2$ and $F_p = 2.54$ mm subject to the influence of fin thickness is reduced to about 20%, and there is no difference in heat transfer coefficient when the frontal velocity is above 3 m/s. For $N \geqslant 4$ and $F_p = 2.54$ mm, the influence of fin thickness on the heat transfer coefficients diminishes considerably. This is because of the presence of tube row, and the unsteady/vortex shedding feature at the down stream of wavy channel. Based on the present test results, a correlation is proposed to describe the air-side performance for wavy fin configurations, the mean deviations of the proposed heat transfer and friction correlations are 7.9% and 7.7%, respectively.

Keywords: Air-side performance; Wavy fin-and-tube heat exchangers; Colburn factor; Friction factor; Dehumidifying conditions

1. Introduction

Plate fin-and-tube heat exchangers are a basic type of heat exchanger employed in many industrial applications such as air conditioning, refrigeration and other thermal processes. In application, the dominated thermal resistance on the air-side of the heat exchanger normally limits the heat transfer rate. One way to enhance the heat transfer on the air-side of the heat exchanger is to modify the fin geometry. Among the enhanced fin configurations, herringbone wavy fin-and-tube heat exchangers are currently widely used because the corrugated fins can provide additional surface area and lengthen the mixing length of the airflow. In their application as evaporators, the surface temperature of the fins is generally below the dew point temperature. As a result, the moist air is condensed on the fins, giving rise to a significant change of the air-side performance of the heat exchanger. Development of the numerical simulation of fin-and-tube heat exchangers is generally difficult due to the complexity of the moist airflow pattern across the heat exchangers, so experimental study is needed.

The air-side performance of wavy fin-and-tube heat exchangers have been studied by a number of researches, see [1–5]. However, the effect of fin thickness on the air-side performance has received little attention in literature [6–8].

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0017-9310/$ - see front matter
doi:10.1016/j.ijheatmasstransfer.2006.01.020

Available online at www.sciencedirect.com

SCIENCE DIRECT

ELSEVIER

Experimental Thermal and Fluid Science 30 (2006) 675–686

Experimental Thermal and Fluid Science

www.elsevier.com/locate/etfs

Effects of EHD on heat transfer enhancement and pressure drop during two-phase condensation of pure R-134a at high mass flux in a horizontal micro-fin tube

Suriyan Laohalertdecha [a], Somchai Wongwises [b,*]

[a] *The Joint Graduate School of Energy and Environment (JGSEE), King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

[b] *Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

Received 4 November 2005; received in revised form 19 January 2006; accepted 22 January 2006

Abstract

Effects of electrohydrodynamic (EHD) on the two-phase heat transfer enhancement and pressure drop of pure R-134a condensing inside a horizontal micro-fin tube are experimentally investigated. The test section is a 2.5 m long counter flow tube-in-tube heat exchanger with refrigerant flowing in the inner tube and cooling water flowing in the annulus. The inner tube is made from micro-fin horizontal copper tubing of 9.52 mm outer diameter. The electrode is made from cylindrical stainless steel of 1.47 mm diameter. Positive high voltage is supplied to the electrode wire, with the micro-fin tube grounded. In the presence of the electrode, a maximum heat transfer enhancement of 1.15 is obtained at a heat flux of 10 kW/m^2, mass flux of 200 kg/m^2 s and saturation temperature of 40 °C, while the application of an EHD voltage of 2.5 kV only slightly increases the pressure drop. New correlations of the experimental data based on the data gathered during this work for predicting the condensation heat transfer coefficients are proposed for practical application.

Keywords: Electrohydrodynamic; Condensation heat transfer; Heat transfer enhancement; Micro-fin tube

1. Introduction

Micro-fin tubes have been successfully implemented in the air-conditioning and refrigeration industries for effectively improving tube-side performance. This success is because of their ability to significantly improve the heat transfer coefficient with only a moderate increase of the friction penalty. A heat transfer enhancement technique utilizing electrohydrodynamic (EHD) can be achieved by utilizing the interaction between the electric field and fluid flow in a dielectric fluid medium. This interaction can result in the increase of fluid motion, which leads to a higher heat transfer coefficient.

The physical basis of the electrically enhanced condensation and boiling are due to the EHD force, (f_e), generated by an electric field and is given by the following Eq. (1) [1]:

$$f_e = qE - \frac{1}{2}E^2\nabla\varepsilon + \frac{1}{2}\nabla\left\langle E^2\left(\frac{\partial\varepsilon}{\partial\rho}\right)_T\rho\right\rangle \tag{1}$$

Eq. (1) can be further written in a more detailed form for non-polar fluids as follows in Eqs. (2) and (3):

$$f_e = f_1 + f_2 + f_3 + f_4 \tag{2}$$

$$f_e = qE - \frac{1}{2}E^2\nabla\varepsilon + \frac{1}{6}\varepsilon_0(\kappa-1)(\kappa-2)\nabla E^2 + \frac{1}{6}\varepsilon_0 E^2\nabla\langle(\kappa-1)(\kappa-2)\rangle \tag{3}$$

The first term on the right of Eq. (3), known as the electrophoretic force, is the Coulomb force acting on the free

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0894-1777/$ - see front matter
doi:10.1016/j.expthermflusci.2006.01.003

Available online at www.sciencedirect.com
SCIENCE DIRECT
Renewable and Sustainable Energy Reviews 10 (2006) 463–490
RENEWABLE & SUSTAINABLE ENERGY REVIEWS
www.elsevier.com/locate/rser

A review of flow and heat transfer characteristics in curved tubes

Paisarn Naphon, Somchai Wongwises*

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Received 7 May 2004; accepted 17 September 2004

Abstract

The performance of heat exchangers can be improved to perform a certain heat-transfer duty by heat transfer enhancement techniques. In general, these techniques can be divided into two groups: active and passive techniques. The active techniques require external forces, e.g. electric field, acoustic or surface vibration, etc. The passive techniques require fluid additives or special surface geometries. Curved tubes have been used as one of the passive heat transfer enhancement techniques and are the most widely used tubes in several heat transfer applications. This article provides a literature review on heat transfer and flow characteristics of single-phase and two-phase flow in curved tubes. Three main categories of curved tubes; helically coiled tubes, spirally coiled tubes, and other coiled tubes, are described. A review of published relevant correlations of single-phase heat transfer coefficients and single-phase, two-phase friction factors are presented.

Keywords: Curved tube; Heat transfer characteristics; Flow characteristics; Helically coiled tube; Spirally coiled tube; Pressure drop; Heat transfer coefficient; Friction factor

* Corresponding author. Tel.: +66 2470 9115; fax: +66 2470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

1364-0321/$ - see front matter © 2004 Elsevier Ltd. All rights reserved.
doi:10.1016/j.rser.2004.09.014

Available online at www.sciencedirect.com
SCIENCE DIRECT
Experimental Thermal and Fluid Science 30 (2006) 523–534
Experimental Thermal and Fluid Science
www.elsevier.com/locate/etfs

Flow pattern and pressure drop of vertical upward gas–liquid flow in sinusoidal wavy channels

Kitti Nilpueng [a], Somchai Wongwises [b,*]

[a] *Department of Mechanical Engineering, South East Asia University, Bangkok 10160, Thailand*
[b] *Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

Received 1 July 2005; accepted 16 October 2005

Abstract

Flow patterns and pressure drop of upward liquid single-phase flow and air–water two-phase flow in sinusoidal wavy channels are experimentally studied. The test section is formed by a sinusoidal wavy wall of 1.00 m length with a wave length of 67.20 mm, an amplitude of 5.76 mm. Different phase shifts between the side walls of the wavy channel of 0°, 90° and 180° are investigated. The flow phenomena, which are bubbly flow, slug flow, churn flow, and dispersed bubbly flow are observed and recorded by high-speed camera. When the phase shifts are increased, the onset of the transition from the bubbly flow to the churn flow shifts to a higher value of superficial air velocity, and the regions of the slug flow and the churn flow are smaller. In other words, the regions of the bubbly flow and the dispersed bubbly flow are larger as the phase shift increases. The slug flow pattern is only found in the test sections with phase shifts of 0° and 90°. Recirculating gas bubbles are always found in the troughs of the corrugations. The recirculating is higher when the phase shifts are larger. The relationship between the two-phase multipliers calculated from the measured pressure drops, and the Martinelli parameter is compared with the Lockhart–Martinelli correlation. The correlation in the case of turbulent–turbulent condition is shown to fit the data very well for the phase shift of 0° but shows greater deviation when the phase shifts are higher.

Keywords: Two-phase flow; Flow pattern; Pressure drop; Sinusoidal wavy channels

1. Introduction

Two-phase gas–liquid flow through a confined gap is encountered in several engineering applications including boiling behavior in mini-channels, cooling systems of various types of equipment such as high performance micro-electronics, supercomputers, high-powered lasers, medical devices, and high heat-flux compact heat exchangers in spacecraft and satellites, etc. It can be expected that the restriction of the bubble space in the mini-channel is the cause of the differences in the two-phase flow characteristics from those in conventional channel geometries. This may also affect heat-mass transfer characteristics during the change of phase.

It is not possible to understand the two-phase flow phenomena without a clear understanding of the flow patterns encountered. It is expected that the flow patterns will influence the two-phase pressure drop, holdup, system stability, exchange rates of momentum, and the heat and mass during the phase-change heat transfer processes. The ability to accurately predict the type of flow is necessary before relevant calculation techniques can be developed.

Two-phase flow characteristics in small circular tubes have been studied by a number of researchers while those in mini-non-circular channels have received comparatively little attention in literature.

Zhao and Rezkallah [1] proposed a system parameter to elucidate the effects of inertia and surface tension in

* Corresponding author. Tel.: +662 470 9115; fax: +662 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0894-1777/$ - see front matter
doi:10.1016/j.expthermflusci.2005.10.004

Available online at www.sciencedirect.com
SCIENCE DIRECT
International Journal of Heat and Mass Transfer 49 (2006) 3757
International Journal of HEAT and MASS TRANSFER
www.elsevier.com/locate/ijhmt

Corrigendum

Corrigendum to "Simultaneous heat and mass transfer characteristics for wavy fin-and-tube heat exchangers under dehumidifying conditions" [Int. J. Heat Mass Transfer 49 (2006) 132–143]

W. Pirompugd [a], S. Wongwises [a,*], C.C. Wang [b]

[a] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand
[b] Energy and Resources Laboratory, Industrial Technology Research Institute, Hsinchu 310, Taiwan, ROC

Available online 14 June 2006

Unfortunately several printing errors slipped the authors attention during proof reading.

Corrections are as follows:

Page 133: The unit of $h_{d,o}$ should read kg m^{-2} s^{-1}.

Page 133: The unit of all enthalpies should read J kg^{-1}.

Page 135: Left column, Line 15–17 should read: (namely, the energy balance condition, $|\dot{Q}_w - \dot{Q}_a|/\dot{Q}_{avg}$, is less than 0.05, where $\dot{Q}_a$, $\dot{Q}_w$ and $\dot{Q}_{avg}$ are the air-side heat transfer rate, water-side heat transfer rate and average heat transfer rate, respectively).

Page 135: Table 1, The unit of fin thickness, S_p, D_c, P_t and P_l should read m.

Page 141: Eq. (27) should read:

$$j_h = 0.171e^{0.377N} Re_{D_c}^{(-0.0142N-0.478)} \left(\frac{S_p}{D_c}\right)^{(0.00412N-0.0217)} \times \left(\frac{A_o}{A_{p,o}}\right)^{(-0.114N+0.440)} \tag{27}$$

Page 141: Eq. (28) should read:

$$j_m = 0.315e^{-0.441N} Re_{D_c}^{(0.0580N-0.475)} \left(\frac{S_p}{D_c}\right)^{(-0.00471N+0.0216)} \times \left(\frac{A_o}{A_{p,o}}\right)^{(0.00223N+0.223)} \tag{28}$$

Page 141: Eq. (29) should read:

$$\frac{h_{c,o}}{h_{d,o}C_{p,a}} = 0.490e^{0.792N} Re_{D_c}^{(-0.0714N+0.00361)} \left(\frac{S_p}{D_c}\right)^{(0.00867N-0.0425)} \times \left(\frac{A_o}{A_{p,o}}\right)^{(-0.107N+0.203)} \tag{29}$$

Page 142: Conclusions, in topic 2 the sentence should read "Both the heat and mass transfer performances are increased when the fin spacing is reduced."

DOI of original article: 10.1016/j.ijheatmasstransfer.2005.05.043
* Corresponding author. Tel.: +662 470 9115; fax: +662 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

doi:10.1016/j.ijheatmasstransfer.2006.03.032

Heat transfer characteristics of a spirally coiled, finned-tube heat exchanger under dry-surface conditions

Author(s): Wongwises S, Naphon P

Source: HEAT TRANSFER ENGINEERING **Volume:** 27 **Issue:** 1 **Pages:** 25-34 **Published:** JAN-FEB 2006

Times Cited: 0 **References:** 19 Citation Map

Abstract: In the present study, the experimental and theoretical results of the heat transfer characteristics and performance of a spirally f coiled, finned-tube heat exchanger under sensible cooling conditions are investigated. The test section is a spiral-coil heat exchanger, which consists of a steel shell and a spirally coiled finned-tube unit. The spiral-coil unit consists of six layers of concentric, spirally coiled, finned tubes. Each tube is fabricated by bending a 9.6 mm-diameter straight copper tube into a spiral-coil of four turns. The innermost and outermost diameters of each spiral-coil are 145.0 and 350.4 mm, respectively. Aluminium annular fins with a thickness of 0.6 mm and outer diameter of 28.4 mm are placed helically around the tube. The edge of each fin at its inner diameter is corrugated. Air and water are used as working fluids in the shell side and tube side, respectively. A mathematical model based on mass and energy conservation is developed to simulate the heat transfer characteristics and performance of the heat exchanger. The results obtained from the present model show reasonable agreement with the experimental data.

Document Type: Article

Language: English

KeyWords Plus: TRANSFER COEFFICIENTS; PRESSURE-DROP; LAMINAR-FLOW; COMPACT; PIPES; CONVECTION

Reprint Address: Wongwises, S (reprint author), King Mongkuts Univ Technol, Dept Mech Engn, Thermal Engn & Multiphase Flow Res Lab, FUTURE, Bangkok 10140, Thailand

Addresses:
1. King Mongkuts Univ Technol, Dept Mech Engn, Thermal Engn & Multiphase Flow Res Lab, FUTURE, Bangkok 10140, Thailand

E-mail Addresses: somchai.won@kmutt.ac.th

Publisher: TAYLOR & FRANCIS INC, 325 CHESTNUT ST, SUITE 800, PHILADELPHIA, PA 19106 USA

Subject Category: Thermodynamics; Engineering, Mechanical; Mechanics

Web of Science®

Flow patterns of two-phase flow in vertical U-type return bends

Author(s): Wongwises S (Wongwises, Somchai), Kamsanam W (Kamsanam, Wason), Wang CC (Wang, Chi-Chuan)

Source: JOURNAL OF THERMOPHYSICS AND HEAT TRANSFER **Volume:** 20 **Issue:** 3 **Pages:** 624-627 **Published:** JUL-SEP 2006

Times Cited: 0 **References:** 12 Citation Map

Document Type: Article

Language: English

KeyWords Plus: SMALL-DIAMETER TUBES; AIR-WATER; R-410A; PHASE

Reprint Address: Wongwises, S (reprint author), King Mongkuts Univ Technol Thonburi, Dept Mech Engn, Fluid Mech Thermal Engn & Multiphase Flow Res Lab, Bangkok 10140, Thailand

Addresses:
1. King Mongkuts Univ Technol Thonburi, Dept Mech Engn, Fluid Mech Thermal Engn & Multiphase Flow Res Lab, Bangkok 10140, Thailand
2. Ind Technol Res Inst, Energy & Environm Res Lab, Hsinchu 310, Taiwan

E-mail Addresses: somchai.won@kmutt.ac.th

Publisher: AMER INST AERONAUT ASTRONAUT, 1801 ALEXANDER BELL DRIVE, STE 500, RESTON, VA 22091-4344 USA

Subject Category: Thermodynamics; Engineering, Mechanical

IDS Number: 068ZL

ISSN: 0887-8722

<< Back to results list   Record 61 of 108   Record from **Web of Science®**

Web of Science®

Comparison of drying kinetics of paddy using a pneumatic conveying dryer with and without a cyclone

Author(s): Kaensup W (Kaensup, Weerachai), Kulwong S (Kulwong, Sittidej), Wongwises S (Wongwises, Somchai)

Source: DRYING TECHNOLOGY **Volume:** 24 **Issue:** 8 **Pages:** 1039-1045 **Published:** 2006

Times Cited: 1 **References:** 11 Citation Map

Abstract: The objective of the present work is to find the possibility of reducing the high initial moisture content of wet paddy using a small-scale, low-cost pneumatic conveying dryer that can be provided for each farming household. The dryer without a cyclone equipped at the exit of the dryer is studied and the data obtained from this system is compared with those obtained previously from the dryer with a cyclone. Parametric effects of the following variables are examined: velocity of drying air from 20 to 30 m/s, feed rate of rough rice from 150 to 350 kg/h, and drying air temperature from 35 to 70 degrees C. From the experimental results it is found that the drying process with and without a cyclone are able to lead to very rapid drying without any grain quality problems such as cracks in the rice kernel. For the same experimental conditions, the cyclone-equipped dryer gives around 1% higher decrease of moisture content, 2 degrees C higher average surface temperature of paddy, 3-4% higher average percentage of head rice yield, and 2 kg/h higher average evaporation rate. However, the energy consumption per evaporated mass of water is 20-30% lower than the non-cyclone-equipped dryer.

Document Type: Article

Language: English

Author Keywords: drying; hot air; moisture content; paddy; pneumatic conveying dryer

KeyWords Plus: MOISTURE-CONTENT; TEMPERATURE; QUALITY; RICE

Reprint Address: Wongwises, S (reprint author), King Mongkuts Univ Technol Thonburi, Dept Mech Engn, Bangkok 10140, Thailand

Addresses:
1. King Mongkuts Univ Technol Thonburi, Dept Mech Engn, Bangkok 10140, Thailand

E-mail Addresses: somchai.won@kmutt.ac.th

Publisher: TAYLOR & FRANCIS INC, 325 CHESTNUT ST, SUITE 800, PHILADELPHIA, PA 19106 USA

Cited by: 1

This article has been cited 1 times (from Web of Science).

Rajan KS, Srivastava SN, Pitchumani B, et al. Experimental study of thermal effectiveness in pneumatic conveying heat exchanger APPLIED THERMAL ENGINEERING 28 14-15 1932-1941 OCT 2008

[view all 1 citing articles]

Create Citation Alert

Related Records:

Find similar records based on shared references (from Web of Science).

[view related records]

References: 11

View the bibliography of this record (from Web of Science).

Suggest a correction

If you would like to improve the quality of this product by suggesting corrections, please fill out this form.

Web of Science®

Thermal performance of a spirally coiled finned tube heat exchanger under wet-surface conditions

Author(s): Wongwises S, Naphon P

Source: JOURNAL OF MECHANICAL SCIENCE AND TECHNOLOGY **Volume:** 20 **Issue:** 2 **Pages:** 212-226 **Published:** FEB 2006

Times Cited: 0 **References:** 21 Citation Map

Abstract: This paper is a continuation of the authors' previous work on spiral coil heat exchangers. In the present study, the heat transfer characteristics and the performance of a spirally coiled finned tube heat exchanger under wet-surface conditions are theoretically and experimentally investigated. The test section is a spiral-coil heat exchanger which consists of a steel shell and a spirally coiled tube unit. The spiral-coil unit consists of six layers of concentric spirally coiled finned tubes. Each tube is fabricated by bending a 9.6 mm diameter straight copper tube into a spiral-coil of four turns. The innermost and outermost diameters of each spiral-coil are 145.0 and 350.4 mm, respectively. Aluminium crimped spiral fins with thickness of 0.6 mm and outer diameter of 28.4 mm are placed around the tube. The edge of fin at the inner diameter is corrugated. Air and water are used as working fluids in shell side and tube side, respectively. The experiments are done under dehumidifying conditions. A mathematical model based on the conservation of mass and energy is developed to simulate the flow and heat transfer characteristics of working fluids flowing through the heat exchanger. The results obtained from the present model show reasonable agreement with the experimental data.

Document Type: Article

Language: English

KeyWords Plus: TRANSFER COEFFICIENTS; PRESSURE-DROP; LAMINAR-FLOW; COMPACT; PIPES; CONVECTION

Reprint Address: Wongwises, S (reprint author), King Mongkuts Univ Technol Thonburi, Dept Mech Engn, Fluid Mech Thermal Engn & Multiphase Flow Res Lab, FUTURE, Bangkok 10140, Thailand

Addresses:
1. King Mongkuts Univ Technol Thonburi, Dept Mech Engn, Fluid Mech Thermal Engn & Multiphase Flow Res Lab, FUTURE, Bangkok 10140, Thailand

E-mail Addresses: somchai.won@kmutt.ac.th

Publisher: KOREAN SOC MECHANICAL ENGINEERS, KSTC NEW

A small-scale pneumatic conveying dryer of rough rice

Author(s): Kaensup W, Kulwong S, Wongwises S

Source: DRYING TECHNOLOGY **Volume:** 24 **Issue:** 1 **Pages:** 105-113 **Published:** 2006

Times Cited: 2 **References:** 10 Citation Map

Abstract: This article studies the possibility of reducing the high initial moisture content of wet rough rice using a small-scale low-cost pneumatic conveying dryer as a first stage dryer. The parameters investigated are final moisture content, surface temperature of rough rice, head rice yield, drying rate, power consumption per unit mass of evaporated water, and physical characteristics of rice. Parametric effects of the following variables are examined: velocity of drying air from 20 to 30 m/s, feed rate of rough rice from 150 to 350 kg/h, initial moisture content from 22 to 26% (wet basis), and drying air temperature from 35 to 70 degrees C. From the experimental results, it is found that this drying method can be used for fresh rough rice with an initial moisture content of over 24% (wet basis). The drying process is able to lead to very rapid drying without any grain quality problems such as cracks in the rice kernel. The moisture content can be reduced to approximately 18% (wet basis) or about 5-6% of the initial moisture content within 3-4 s. The optimal drying air temperature is in the range of 50 to 60 degrees C. A comparison of pneumatic conveying drying data obtained from the present study with fluidized bed drying data reported in the open literature is also discussed.

Document Type: Article

Language: English

Author Keywords: drying; paddy; rough rice; pneumatic conveying dryer

KeyWords Plus: MOISTURE-CONTENT; PADDY; TEMPERATURE; QUALITY

Reprint Address: Wongwises, S (reprint author), King Mongkuts Univ Technol Thonburi, Dept Mech Engn, 91 Pracha Uthit Rd, Bangkok 10140, Thailand

Addresses:
1. King Mongkuts Univ Technol Thonburi, Dept Mech Engn, Bangkok 10140, Thailand

E-mail Addresses: somchai.won@kmutt.ac.th

Publisher: TAYLOR & FRANCIS INC, 325 CHESTNUT ST, SUITE

Cited by: 2

This article has been cited 2 times (from Web of Science).

Rajan KS, Srivastava SN, Pitchumani B, et al. Experimental study of thermal effectiveness in pneumatic conveying heat exchanger APPLIED THERMAL ENGINEERING 28 14-15 1932-1941 OCT 2008

Kaensup W, Kulwong S, Wongwises S Comparison of drying kinetics of paddy using a pneumatic conveying dryer with and without a cyclone DRYING TECHNOLOGY 24 8 1039-1045 2006

[view all 2 citing articles]

Create Citation Alert

Related Records:

Find similar records based on shared references (from Web of Science).

[view related records]

References: 10

View the bibliography of this record (from Web of Science).

Suggest a correction

If you would like to improve the quality of this product by suggesting corrections, please fill out this form.

Available online at www.sciencedirect.com
SCIENCE DIRECT

International Journal of Heat and Mass Transfer 49 (2006) 132–143

International Journal of HEAT and MASS TRANSFER

www.elsevier.com/locate/ijhmt

Simultaneous heat and mass transfer characteristics for wavy fin-and-tube heat exchangers under dehumidifying conditions

Worachest Pirompugd [a], Somchai Wongwises [a,*], Chi-Chuan Wang [b]

[a] *Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

[b] *Energy and Resources Laboratory, Industrial Technology Research Institute, Hsinchu 310, Taiwan, ROC*

Received 3 February 2005; received in revised form 8 May 2005
Available online 3 October 2005

Abstract

The present study proposes a new reduction method to calculate the heat and mass transfer characteristics of the wavy fin-and-tube heat exchangers under dehumidifying conditions. For fully wet conditions, the sensible heat transfer and mass transfer characteristics are relatively insensitive to the inlet relative humidity. The heat and mass transfer performances show appreciable influence of fin spacing at 1-row configuration. Both the heat and mass transfer performances increase when the fin spacing is reduced. However, the difference becomes less noticeable when $Re_{Dc} > 3000$. For 1-row configuration, larger wave height shows much larger difference with the fin spacing. However, the effect of inlet conditions and geometrical parameters on the heat and mass performance becomes less significant with the rise of number of tube rows. Test results show that the heat and mass transfer analogy is roughly applicable (the ratios of $h_{c,o}/h_{d,o}C_{p,a}$ are in the range 0.6–1.1, and is insensitive to change of fin spacing). The correlations are proposed to describe the heat and mass transfer characteristics. These correlations can describe 94.19% of the j_h factors within 15% and 83.72% of the j_m factors within 15%. Correspondingly, 93.02% of the ratios of $h_{c,o}/h_{d,o}C_{p,a}$ are predicted by the proposed correlation within 15%.

Keywords: Wavy fin-and-tube heat exchangers; Dehumidifying; Heat transfer; Mass transfer

1. Introduction

The plate fin-and-tube heat exchangers are the most widely used heat exchangers in association with the application of air-conditioning and refrigeration systems. The heat exchangers can be used for condensers where surface is dry and evaporators in which surface may be wet provided the fin temperature is below the dew point temperature. In regard to the wet surface, simultaneous heat and mass transfer occurs along the fin surfaces. In general, the complexity of the moist air flow pattern across the fin-and-tube heat exchangers under dehumidifying conditions makes the theoretical

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0017-9310/$ - see front matter
doi:10.1016/j.ijheatmasstransfer.2005.05.043

ADVANCES IN ATMOSPHERIC SCIENCES, VOL. 23, NO. 6, 2006, 901–908

Carbon Monoxide Emission and Concentration Models for Chiang Mai Urban Area

Surachai SATHITKUNARAT[1], Prungchan WONGWISES*[,1],
Rudklao PAN-ARAM[2], and ZHANG Meigen[3] (张美根)

[1] *The Joint Graduate School of Energy and Environment at King Mongkut's University of Technology Thonburi, Bangkok* 10140, *Thailand*

[2] *Electricity Generating Authority of Thailand, Nonthaburi* 11130, *Thailand*

[3] *Institute of Atmospheric Physics, Chinese Academy of Sciences, Beijing* 100029

(Received 9 January 2006; revised 13 June 2006)

ABSTRACT

An emission inventory containing emissions from traffic and other sources was complied. Based on the analysis, Carbon Monoxide (CO) emissions from traffic play a very important role in CO levels in Chiang Mai area. Analysis showed that CO emissions from traffic during rush hours contributed approximately 90% of total CO emissions. Regional Atmospheric Modeling System (RAMS) was applied to simulate wind fields and temperatures in the Chiang Mai area, and eight cases were selected to study annual variations in wind fields and temperatures. Model results can reflect major features of wind fields and diurnal variations in temperatures. For evaluating the model performance, model results were compared with observed wind speed, wind direction and temperature, which were monitored at a meteorological tower. Comparison showed that model results are in good agreement with observations, and the model captured many of the observed features. HYbrid Particle And Concentration Transport model (HYPACT) was used to simulate CO concentration in the Chiang Mai area. Model results generally agree well with observed CO concentrations at the air quality monitoring stations, and can explain observed CO diurnal variations.

Key words: carbon monoxide, emission, concentration, Chiang Mai, RAMS, HYPACT

doi: 10.1007/s00376-006-0901-9

1. Introduction

Carbon Monoxide (CO) is a highly toxic gas, which is considered a dangerous asphyxiant. Combined with haemoglobin of the blood it reduces the blood's ability to carry oxygen to cell tissues; high CO mixing ratio can directly affect human health. The oxidation of CO initiates photochemical reactions, which result in Ozone (O_3) production on a regional scale (Novelli et al., 1998). In cities, almost all of the CO is anthropogenic due to vehicle emission and fossil fuel combustion. Natural sources, such as forest fire and biomass burning, also produce huge quantities of CO. Pochanart (2003) studied surface CO variations from measurements during 1997–2000 in rural Thailand. The CO mixing ratios in Thailand show a strong seasonal variation with a maximum in the late dry season (February–March) and a minimum in the mid-wet season (July–August). In this study CO emission and dispersion on a local scale in Chiang Mai (18°78′N, 98°98′E, 312 meters above sea level) are studied. Chiang Mai is an important city for the regional administration, business, education, medical and is well known as a tourist and cultural city. Due to economic growth, Chiang Mai is facing environmental problems, especially the air pollution problem. Because of traffic congestion most vehicles emit many types of toxic gasses. Pollutants from the internal combustion process consist of Carbon Monoxide, Sulfur Dioxide, Nitrogen Dioxide, Hydrocarbons, and Lead etc. This study concentrated on CO which was emitted from traffic, industrial processes, airport, railway station, forest fire, fuel consumption and open burning. There are only 2 ambient air pollution monitoring stations, one in an urban area and the other in a suburban area, which do not cover the whole study area.

*E-mail: prungchan.won@kmutt.ac.th

Available online at www.sciencedirect.com
SCIENCE DIRECT
International Journal of Heat and Mass Transfer 49 (2006) 658–670
International Journal of HEAT and MASS TRANSFER
www.elsevier.com/locate/ijhmt

Evaporation heat transfer and pressure drop of HFC-134a in a helically coiled concentric tube-in-tube heat exchanger

Somchai Wongwises *, Maitree Polsongkram

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Received 1 June 2005; received in revised form 18 August 2005
Available online 18 October 2005

Abstract

The two-phase heat transfer coefficient and pressure drop of HFC-134a during evaporation inside a smooth helically coiled concentric tube-in-tube heat exchanger are experimentally investigated. The test section is a 5.786-m long helically coiled tube with refrigerant flowing in the inner tube and heating water flowing in the annulus. The inner tube is made from copper tubing of 9.52 mm outer diameter and 7.2 mm inner diameter. The heat exchanger is fabricated by bending a straight copper tube into a spiral coil. The diameter of coil is 305 mm. The test run are done at average saturated evaporating temperatures ranging between 10 and 20 °C. The mass fluxes are between 400 and 800 kg m^{-2} s^{-1} and the heat fluxes are between 5 and 10 kW m^{-2}. The inlet quality of the refrigerant in the test section is calculated using the temperature and pressure obtained from the experiment. The pressure drop across the test section is directly measured by a differential pressure transducer. The effects of heat flux, mass flux and, evaporation temperature on the heat transfer coefficients and pressure drop are also discussed. The results from the present experiment are compared with those obtained from the straight tube reported in the literature. New correlations for the convection heat transfer coefficient and pressure drop are proposed for practical applications.

Keywords: Two-phase; Evaporation; Heat transfer coefficient; Pressure drop; Refrigerant; Helically coiled tube

1. Introduction

Results from many researches show that the ozone layer is being depleted. The general consensus for the cause of this event is that the free chlorine radicals removes ozone from the atmosphere and later the chlorine atom is continued to convert more ozone to oxygen. The presence of chlorine in the stratosphere is the result of migration of chlorine-containing chemicals. The chlorofluorocarbons (CFCs) is a large class of chemicals which behaves in this manner. These chemicals have many unusual properties for example, nonflammability, low toxicity, and material compatibility that have led to their common widespread use, both consumers and industries around the world as refrigerants, solvent, and blowing agents for foams. Since the depletion of the earth's ozone layer has been discovered, many corporations have been forced to find alternative chemicals to CFCs. Because the thermophysical properties of HFC-134a are very similar to those of CFC-12. Refrigerant HFC-134a is receiving the supporting from the refrigerant and air-conditioning industry as a potential replacement for CFC-12. However, even the difference in properties between both refrigerants is small but it may result in significant differences in the overall system performance. Therefore, the properties of HFC-134a should be studied in detail before it is applied.

Heat transfer and pressure drop characteristics of refrigerants have been studied by a large number of researchers, both experimentally and analytically, mostly in a horizontal straight tube. The study of the heat transfer and pressure drop inside helicoidal tube has received comparatively little attention in the literature.

* Corresponding author. Tel.: +662 470 9115; fax: +662 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0017-9310/$ - see front matter
doi:10.1016/j.ijheatmasstransfer.2005.08.017

Available online at www.sciencedirect.com

SCIENCE DIRECT

Energy Conversion and Management 47 (2006) 1644–1659

ENERGY CONVERSION & MANAGEMENT

www.elsevier.com/locate/enconman

Experimental investigation of hydrocarbon mixtures to replace HFC-134a in an automotive air conditioning system

Somchai Wongwises [a,*], Amnouy Kamboon [b], Banchob Orachon [b]

[a] *Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

[b] *Department of Mechanical Technology Education, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

Received 8 October 2004; received in revised form 31 March 2005; accepted 29 April 2005
Available online 4 October 2005

Abstract

This paper presents an experimental study on the application of hydrocarbon mixtures to replace HFC-134a in automotive air conditioners. The hydrocarbons investigated are propane (R290), butane (R600) and isobutane (R600a). The measured data are obtained from an automotive air conditioning test facility utilizing HFC-134a as the refrigerant. The air conditioner, with a capacity of 3.5 kW driven by a Diesel engine, is charged and tested with four different ratios of hydrocarbon mixtures. The experiments are conducted at the same surrounding conditions. The temperature and pressure of the refrigerant at every major position in the refrigerant loop, the temperature, flow rate and humidity of air, torque and engine speed are recorded and analyzed. The parameters investigated are the refrigeration capacity, the compressor power and the coefficient of performance (COP). The results show that propane/butane/isobutane: 50%/40%/10% is the most appropriate alternative refrigerant to replace HFC-134a, having the best performance of all the hydrocarbon mixtures investigated.

Keywords: Automotive air conditioner; HFC-134a; Propane; Butane; Isobutane; Hydrocarbon mixture

1. Introduction

Stratospheric ozone absorbs the sun's high energy ultraviolet rays and protects both humans and other living things from exposure to ultraviolet radiation. Results from many researches show that this ozone layer is being depleted. The general consensus for the cause of this event is that free chlorine radicals remove ozone from the atmosphere, and later, chlorine atoms continue to convert more ozone to oxygen. The presence of chlorine in the stratosphere is the result of the migration of chlorine containing chemicals. The chlorofluorocarbons (CFCs) and hydrochlorofluorocarbons (HCFCs) are a large class of chemicals that behave in this

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0196-8904/$ - see front matter
doi:10.1016/j.enconman.2005.04.013

Available online at www.sciencedirect.com

SCIENCE DIRECT

Experimental Thermal and Fluid Science 30 (2006) 345–354

ELSEVIER

Experimental Thermal and Fluid Science

www.elsevier.com/locate/etfs

Flow pattern, pressure drop and void fraction of two-phase gas–liquid flow in an inclined narrow annular channel

Somchai Wongwises *, Manop Pipathattakul

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Received 1 July 2005; accepted 16 August 2005

Abstract

Two-phase flow pattern, pressure drop and void fraction in horizontal and inclined upward air–water two-phase flow in a mini-gap annular channel are experimentally studied. A concentric annular test section at the length of 880 mm with an outer diameter of 12.5 mm and inner diameter of 8 mm is used in the experiments. The flow phenomena, which are plug flow, slug flow, annular flow, annular/slug flow, bubbly/plug flow, bubbly/slug–plug flow, churn flow, dispersed bubbly flow and slug/bubbly flow, are observed and recorded by high-speed camera. A slug flow pattern is found only in the horizontal channel while slug/bubbly flow patterns are observed only in inclined channels. When the inclination angle is increased, the onset of transition from the plug flow region to the slug flow region (for the horizontal channel) and from the plug flow region to slug/bubbly flow region (for inclined channels) shift to a lower value of superficial air velocity. Small shifts are found for the transition line between the dispersed bubbly flow and the bubbly/plug flow, the bubbly/plug flow and the bubbly/slug–plug flow, and the bubbly/plug flow and the plug flow. The rest of the transition lines shift to a higher value of superficial air velocity. Considering the effect of flow pattern on the pressure drop in the horizontal tube at low liquid velocity, the occurrence of slug flow stops the rise of pressure drop for a short while, before rising again after the air velocity has increased. However, the pressure does not rise abruptly in the tubes with $\theta = 30°$ and $60°$ when the slug/bubbly flow occurs. At low gas and liquid velocity, the pressure drop increases, when the inclination angles changes from horizontal to 30° and 60°. Void fraction increases with increasing gas velocity and decreases with increasing liquid velocity. After increasing the inclination angle from horizontal to $\theta = 30°$ and $60°$, the void fraction appears to be similar, with a decreasing trend when the inclination angle increases.

Keywords: Two-phase flow; Mini-channel; Narrow annular channel; Flow pattern; Pressure drop; Void fraction

1. Introduction

Two-phase gas–liquid flow through a confined gap is encountered in several engineering applications including boiling behavior in minichannels, cooling systems of various types of equipment such as high performance microelectronics, supercomputers, high-powered lasers, medical devices, high heat-flux compact heat exchangers in spacecraft and satellites etc. It can be expected that the restriction of the bubble space in the minichannel is the cause of the differences in the two-phase flow characteristics from those in conventional channel geometries. This may also affect heat-mass transfer characteristics during the change of phase.

It is not possible to understand the two-phase flow phenomena without a clear understanding of the flow patterns encountered. It is expected that the flow patterns will influence the two-phase pressure drop, holdup, system stability, exchange rates of momentum, heat and mass during the phase-change heat transfer processes. The ability to accurately predict the type of flow is necessary before relevant calculation techniques can be developed.

* Corresponding author. Tel.: +662 470 9115; fax: +662 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0894-1777/$ - see front matter
doi:10.1016/j.expthermflusci.2005.08.002

Thermal Performance of a Spirally Coiled Finned Tube Heat Exchanger Under Wet-Surface Conditions

Somchai Wongwises*, Paisarn Naphon
Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

This paper is a continuation of the authors' previous work on spiral coil heat exchangers. In the present study, the heat transfer characteristics and the performance of a spirally coiled finned tube heat exchanger under wet-surface conditions are theoretically and experimentally investigated. The test section is a spiral-coil heat exchanger which consists of a steel shell and a spirally coiled tube unit. The spiral-coil unit consists of six layers of concentric spirally coiled finned tubes. Each tube is fabricated by bending a 9.6 mm diameter straight copper tube into a spiral-coil of four turns. The innermost and outermost diameters of each spiral-coil are 145.0 and 350.4 mm, respectively. Aluminium crimped spiral fins with thickness of 0.6 mm and outer diameter of 28.4 mm are placed around the tube. The edge of fin at the inner diameter is corrugated. Air and water are used as working fluids in shell side and tube side, respectively. The experiments are done under dehumidifying conditions. A mathematical model based on the conservation of mass and energy is developed to simulate the flow and heat transfer characteristics of working fluids flowing through the heat exchanger. The results obtained from the present model show reasonable agreement with the experimental data.

Nomenclature

A : Area
C_p : Specific heat, kJ/(kg ℃)
D : Tube diameter, m
De : Dean number
E_h : Enthalpy effectiveness
E_w : Umidity effectiveness
G : Mass flux, kg/(m^2 s)
h : Heat transfer coefficient, W/(m^2 ℃)
h_r : Combined conductance through tube surface and water inside tube, W/(m^2 ℃)
h_D : Mass transfer coefficient, kg/(m^2 s)
i : Enthalpy, kJ/kg
i_{fg} : Enthalpy of condensation, kJ/kg
J : Colburn j factor
j : Number of segments
k : Thermal conductivity, W/(m ℃)
Le : Lewis number
M : Mass flow rate per coil, kg/s
m : Total mass flow rate, kg/s
n : Number of coil turns
Nu : Nusselt number
Pr : Prandtl number
Q : Heat transfer rate, W
r : Tube radius, m
R_{min} : Minimum coil radius, m
Re : Reynolds number
R_n : Average radius of curvature, m
T : Temperature, ℃
U : Overall heat transfer coefficient, W/(m^2 ℃)
δ : Thickness, m
α : Rate of radius change per radian, m/radian
ω : Humidity ratio

* Corresponding Author,
E-mail : somchai.won@kmutt.ac.th
TEL : +662-470-9115; **FAX** : +662-470-9111
Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand. (Manuscript **Received** September 1, 2005; **Revised** December 20, 2005)

ELSEVIER

Available online at www.sciencedirect.com

SCIENCE DIRECT

International Journal of Heat and Mass Transfer 49 (2006) 4386–4398

International Journal of HEAT and MASS TRANSFER

www.elsevier.com/locate/ijhmt

Condensation heat transfer and pressure drop of HFC-134a in a helically coiled concentric tube-in-tube heat exchanger

Somchai Wongwises *, Maitree Polsongkram

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Received 17 November 2005; received in revised form 13 May 2006
Available online 17 July 2006

Abstract

The two-phase heat transfer coefficient and pressure drop of pure HFC-134a condensing inside a smooth helically coiled concentric tube-in-tube heat exchanger are experimentally investigated. The test section is a 5.786 m long helically coiled double tube with refrigerant flowing in the inner tube and cooling water flowing in the annulus. The inner tube is made from smooth copper tubing of 9.52 mm outer diameter and 8.3 mm inner diameter. The outer tube is made from smooth copper tubing of 23.2 mm outer diameter and 21.2 mm inner diameter. The heat exchanger is fabricated by bending a straight copper double-concentric tube into a helical coil of six turns. The diameter of coil is 305 mm. The pitch of coil is 35 mm. The test runs are done at average saturation condensing temperatures ranging between 40 and 50 °C. The mass fluxes are between 400 and 800 kg m^{-2} s^{-1} and the heat fluxes are between 5 and 10 kW m^{-2}. The pressure drop across the test section is directly measured by a differential pressure transducer. The quality of the refrigerant in the test section is calculated using the temperature and pressure obtained from the experiment. The average heat transfer coefficient of the refrigerant is determined by applying an energy balance based on the energy rejected from the test section. The effects of heat flux, mass flux and, condensation temperature on the heat transfer coefficients and pressure drop are also discussed. It is found that the percentage increase of the average heat transfer coefficient and the pressure drop of the helically coiled concentric tube-in-tube heat exchanger, compared with that of the straight tube-in-tube heat exchanger, are in the range of 33–53% and 29–46%, respectively. New correlations for the condensation heat transfer coefficient and pressure drop are proposed for practical applications.

Keywords: Two-phase flow; Condensation; Heat transfer coefficient; Pressure drop; Refrigerant; Helically coiled tube

1. Introduction

The presence of chlorine in the stratosphere is the result of migration of chlorine-containing chemicals. The chlorofluorocarbons (CFCs) are a large class of chemicals which behaves in this manner. These chemicals have many unusual properties for example, nonflammability, low toxicity, and material compatibility that have led to their common widespread use, both consumers and industries around the world as refrigerants, solvent, and blowing agents for foams. Since the depletion of the earth's ozone layer has been discovered, many corporations have been forced to find alternative chemicals to CFCs. Because the thermophysical properties of HFC-134a are very similar to those of CFC-12. Refrigerant HFC-134a is receiving the supporting from the refrigerant and air-conditioning industry as a potential replacement for CFC-12. However, even the difference in properties between both refrigerants is small but it may result in significant differences in the overall system performance. Therefore, the properties of HFC-134a should be studied in detail before it is applied.

Heat transfer and pressure drop characteristics of refrigerants have been studied by a large number of researchers, both experimentally and analytically, mostly in a horizontal straight tube. The study of the heat transfer and pressure drop inside helicoidal tube has received comparatively little attention in the literature.

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0017-9310/$ - see front matter
doi:10.1016/j.ijheatmasstransfer.2006.05.010

ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

International Journal of Refrigeration 30 (2007) 601–608

REVUE INTERNATIONALE DU FROID
INTERNATIONAL JOURNAL OF
refrigeration

www.elsevier.com/locate/ijrefrig

Effect of throat diameters of the ejector on the performance of the refrigeration cycle using a two-phase ejector as an expansion device

Praitoon Chaiwongsa, Somchai Wongwises*

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Received 9 May 2006; received in revised form 7 November 2006; accepted 7 November 2006
Available online 12 January 2007

Abstract

This paper is a part in a series that reports on the experimental study of the performance of the two-phase ejector expansion refrigeration cycle. In the present study, three two-phase ejectors are used as an expansion device in the refrigeration cycle. The effects of throat diameter of the motive nozzle, on the coefficient of performance, primary mass flow rate of the refrigerant, secondary mass flow rate of the refrigerant, recirculation ratio, average evaporator pressure, compressor pressure ratio, discharge temperature and cooling capacity, which have never before appeared in open literature, are presented. The effects of the heat sink and heat source temperatures on the system performance are also discussed.

Keywords: Refrigeration; Ejector system; Experiment; Performance; Parameter; Geometry; Nozzle

Effets des diamètres des tuyères des éjecteurs sur la performance du cycle frigorifique à l'aide d'un éjecteur diphasique utilisé comme dispositif de détente

Mots clés : Froid ; Système à éjecteur ; Expérimentation ; Performance ; Paramètre ; Géométrie ; Tuyère

1. Introduction

Throttling loss is one of the thermodynamic losses in a conventional vapor compression refrigeration cycle. In order to reduce this loss, various devices and techniques have been attempted to use instead of the conventional devices. Ejector is a device that uses a high-pressure fluid

* Corresponding author. Tel.: +662 470 9115; fax: +662 470 9111.

E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0140-7007/$35.00 © 2006 Elsevier Ltd and IIR. All rights reserved.
doi:10.1016/j.ijrefrig.2006.11.006

ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

Solar Energy 81 (2007) 884–895

SOLAR ENERGY

www.elsevier.com/locate/solener

Performance of a twin power piston low temperature differential Stirling engine powered by a solar simulator

Bancha Kongtragool, Somchai Wongwises *

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, 91 Suksawas 48, Bangmod, Bangkok 10140, Thailand

Received 7 July 2005; received in revised form 23 October 2006; accepted 27 November 2006
Available online 26 December 2006

Communicated by: Associate Editor Manuel Romero-Alvarez

Abstract

This paper provides an experimental investigation on the performance of a low-temperature differential Stirling engine. In this study, a twin power piston, gamma-configuration, low-temperature differential Stirling engine is tested with non-pressurized air by using a solar simulator as a heat source. The engine testing is performed with four different simulated solar intensities. Variations of engine torque, shaft power and brake thermal efficiency with engine speed and engine performance at various heat inputs are presented. The Beale number, obtained from the testing of the engine, is also investigated. The results indicate that at the maximum simulated solar intensity of $7145\ W/m^2$, or heat input of 261.9 J/s, with a heater temperature of 436 K, the engine produces a maximum torque of 0.352 N m at 23.8 rpm, a maximum shaft power of 1.69 W at 52.1 rpm, and a maximum brake thermal efficiency of 0.645% at 52.1 rpm, approximately.

Keywords: Stirling engine; Hot-air engine; Solar-powered heat engine; Solar simulator

1. Introduction

The low temperature differential (LTD) Stirling engine is a type of Stirling engine that can operate with a low temperature heat source. There are many low temperature heat sources available including solar energy. The LTD Stirling engine then provides the possibility of direct conversion of solar energy to mechanical work.

A LTD Stirling engine can be run with small temperature difference between the hot and cold ends of the displacer cylinder (Rizzo, 1997). LTD Stirling engines provide value as demonstration units, but they immediately become of interest when considering the possibility of power generation from many low-temperature waste heat sources in which the temperature is less than 100 °C (Van Arsdell, 2001). Some characteristics of the LTD Stirling engine are as follow (Rizzo, 1997):

(1) Displacer to power piston swept volumes ratio is large.
(2) Diameters of displacer cylinder and displacer are large.
(3) Displacer length is short.
(4) Effective heat transfer surfaces on both end plates of the displacer cylinder are large.
(5) Displacer stroke is small.
(6) Dwell period at the end of the displacer stroke is rather longer than the normal Stirling engine.
(7) Operating speed is low.

The LTD Stirling engine has been studied by many researchers. Many studies in the field of LTD Stirling

* Corresponding author. Tel.: +66 2 4709115; fax: +66 2 4709111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0038-092X/$ - see front matter
doi:10.1016/j.solener.2006.11.004

ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

Renewable Energy 32 (2007) 547–566

RENEWABLE ENERGY

www.elsevier.com/locate/renene

Performance of low-temperature differential Stirling engines

Bancha Kongtragool, Somchai Wongwises*

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Received 1 March 2006; accepted 2 March 2006
Available online 18 April 2006

Abstract

In this paper, two single-acting, twin power piston and four power pistons, gamma-configuration, low-temperature differential Stirling engine are designed and constructed. The engine performance is tested with air at atmospheric pressure by using a gas burner as a heat source. The engine is tested with various heat inputs. Variations of engine torque, shaft power and brake thermal efficiency at various heat inputs with engine speed and engine performance are presented. The Beale number obtained from testing of the engines is also investigated. The results indicate that, for twin power piston engine, at a maximum actual heat input of 2355 J/s with a heater temperature of 589 K, the engine produces a maximum torque of 1.222 N m at 67.7 rpm, a maximum shaft power of 11.8 W at 133 rpm, and a maximum brake thermal efficiency of 0.494% at 133 rpm, approximately. For the four power pistons engine, the results indicate that at the maximum actual heat input of 4041 J/s with the heater temperature of 771 K, the engine produces a maximum torque of 10.55 N m at 28.5 rpm, a maximum shaft power of 32.7 W at 42.1 rpm, and a maximum brake thermal efficiency of 0.809% at 42.1 rpm, approximately.

Keywords: Stirling engine; Hot-air engine; Regenerative heat engine

1. Introduction

The Stirling engine is a simple type of external-combustion engine. The Stirling engine was the first invented regenerative cycle heat engine. Robert Stirling patented the Stirling

*Corresponding author. Tel.: +662 4709115; fax: +662 4709111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0960-1481/$ - see front matter
doi:10.1016/j.renene.2006.03.003

Available online at www.sciencedirect.com
ScienceDirect
Renewable and Sustainable Energy Reviews
11 (2007) 797–817

RENEWABLE
& SUSTAINABLE
ENERGY REVIEWS

www.elsevier.com/locate/rser

A critical review of convective heat transfer of nanofluids

Weerapun Daungthongsuk, Somchai Wongwises*

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Received 3 May 2005; accepted 1 June 2005

Abstract

A nanofluid is a suspension of ultrafine particles in a conventional base fluid which tremendously enhances the heat transfer characteristics of the original fluid. Furthermore, nanofluids are expected to be ideally suited in practical applications as their use incurs little or no penalty in pressure drop because the nanoparticles are ultrafine, therefore, appearing to behave more like a single-phase fluid than a solid–liquid mixture. About a decade ago, several published articles focused on measuring and determining the effective thermal conductivity of nanofluids, some also evaluated the effective viscosity. There are only a few published articles on deriving the forced convective heat transfer of nanofluids. The purpose of this article is to summarize the published subjects respect to the forced convective heat transfer of the nanofluids both of experimental and numerical investigation.

Keywords: Nanofluids; Ultrafine; Effective thermal conductivity; Effective viscosity; Forced convective heat transfer

Contents

*Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

1364-0321/$ - see front matter
doi:10.1016/j.rser.2005.06.005

ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

Applied Thermal Engineering 27 (2007) 910–917

APPLIED THERMAL ENGINEERING

www.elsevier.com/locate/apthermeng

An exergy analysis on the performance of a counterflow wet cooling tower

Thirapong Muangnoi [a], Wanchai Asvapoositkul [b,*], Somchai Wongwises [b]

[a] *The Joint Graduate School of Energy and Environment, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*
[b] *Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

Received 23 February 2006; accepted 29 August 2006
Available online 18 October 2006

Abstract

Cooling towers are used to extract waste heat from water to atmospheric air. An energy analysis is usually used to investigate the performance characteristics of cooling tower. However, the energy concept alone is insufficient to describe some important viewpoints on energy utilization. In this study, an exergy analysis is used to indicate exergy and exergy destruction of water and air flowing through the cooling tower. Mathematical model based on heat and mass transfer principle is developed to find the properties of water and air, which will be further used in exergy analysis. The model is validated against experimental data. It is noted from the results that the amount of exergy supplied by water is larger than that absorbed by air, because the system produces entropy. To depict the utilizable exergy between water and air, exergy of each working fluid along the tower are presented. The results show that water exergy decreases continuously from top to bottom. On the other hand, air exergy is expressed in terms of convective and evaporative heat transfer. Exergy of air via convective heat transfer initially loses at inlet and slightly recovers along the flow before leaving the tower. However, exergy of air via evaporative heat transfer is generally high and able to consume exergy supplied by water. Exergy destruction is defined as the difference between water exergy change and air exergy change. It reveals that the cooling processes due to thermodynamics irreversibility perform poorly at bottom and gradually improve along the height of the tower. The results show that the lowest exergy destruction is located at the top of the tower.

Keywords: Cooling tower; Exergy analysis; Exergy destruction

1. Introduction

A cooling tower is a heat rejection device. Its main function is to extract waste heat from warm water to the atmosphere. Heat rejection in cooling tower is specified as convection between the fine droplets of water and the surrounding air, and also as evaporation which allows a small portion of water to evaporate into moving air. Therefore, the process involves both heat and mass transfer. Cooling towers are widely used in most power plants, refrigeration and air conditioning industries, etc [1]. They can be classified by the movement of water and air as counterflow and crossflow types. Moreover, they can also be classified by means of air flow into mechanical draft and natural draft types.

The analysis of cooling tower performance has been studied and developed over the last century. Investigations on the performance and its factors have been widely studied. Heat and mass transfer are the core principles in these analyses. Bahaidarah [2] stated that the method generally used for cooling tower calculation was developed by Markel over 70 years ago. The equation was presented in a differential form known as Markel Mathematical Modeling and was used for describing the distributions of water- and air-conditions along the cooling tower. However, an obvious disadvantage of Markel equation was based on the assumptions that evaporation of water flow was neglected in energy balance and saturated air was at the exit. These assumptions made the results inaccurate.

* Corresponding author. Tel.: +662 470 9338; fax: +662 470 9111.
E-mail address: wanchai.asv@kmutt.ac.th (W. Asvapoositkul).

1359-4311/$ - see front matter
doi:10.1016/j.applthermaleng.2006.08.012

A fully wet and fully dry tiny circular fin method for heat and mass transfer characteristics for plain fin-and-tube heat exchangers under dehumidifying conditions

Author(s): Pirompugd W (Pirompugd, Worachest), Wang CC (Wang, Chi-Chuan), Wongwises S (Wongwises, Somchai)

Source: JOURNAL OF HEAT TRANSFER-TRANSACTIONS OF THE ASME **Volume:** 129 **Issue:** 9 **Pages:** 1256-1267 **Published:** SEP 2007

Times Cited: 4 **References:** 26 Citation Map

Abstract: This study proposes a new method, namely the "fully wet and fully dry tiny circular fin method," for analyzing the heat and mass transfer characteristics of plain fin-and-tube heat exchangers under dehumidifying conditions. The present method is developed from the tube-by-tube method proposed in the previous study by the same authors. The analysis of the fin-and-tube heat exchangers is carried out by dividing the heat exchanger into many tiny segments. A tiny segment will be assumed with fully wet or fully dry conditions. This method is capable of handling the plain fin-and-tube heat exchanger underfully wet and partially wet conditions. The heat and mass transfer characteristics are presented in dimensionless terms. The ratio of the heat transfer characteristic to mass transfer characteristic is also studied. Based on the reduced results, it is found that the heat transfer and mass transfer characteristics are insensitive to changes in fin spacing. The influence of the inlet relative humidity on the heat transfer characteristic is rather small. For one and two row configurations, a considerable increase of the mass transfer characteristic is encountered when partially wet conditions take place. The heat transfer characteristic is about the same in fully wet and partially wet conditions provided that the number of tube rows is equal to or greater than four Correlations are proposed to describe the heat and mass characteristics for the present plain fin configuration.

Document Type: Article

Language: English

Author Keywords: fully wet and fully dry tiny circular fin method; dehumidifying conditions; heat transfer characteristic; mass transfer characteristic; plain fin-and-tube heat exchangers

KeyWords Plus: PERFORMANCE

Reprint Address: Wongwises, S (reprint author), King Mongkuts Univ Technol, Dept Mech Engn, Fluid Mech Thermal Engn & Multiphase Flow Res Lab, Bangkok 10104, Thailand

Cited by: 4

This article has been cited 4 times (from Web of Science).

Liu YC, Wongwises S, Chang WJ, et al. Airside performance of fin-and-tube heat exchangers in dehumidifying conditions - Data with larger diameter INTERNATIONAL JOURNAL OF HEAT AND MASS TRANSFER 53 7-8 1603-1608 MAR 2010

Pirompugd W, Wang CC, Wongwises S Correlations for wet surface ratio of fin-and-tube heat exchangers INTERNATIONAL JOURNAL OF HEAT AND MASS TRANSFER 53 1-3 568-573 JAN 15 2010

Pirompugd W, Wang CC, Wongwises S A review on reduction method for heat and mass transfer characteristics of fin-and-tube heat exchangers under dehumidifying conditions INTERNATIONAL JOURNAL OF HEAT AND MASS TRANSFER 52 9-10 2370-2378 APR 2009

Related Records:

Find similar records based on shared references (from Web of Science).

References: 26

View the bibliography of this record (from Web of Science).

Heat and mass transfer characteristics for finned tube heat exchangers with humidification

Author(s): Pirompugd W (Pirompugd, Worachest), Wang CC (Wang, Chi-Chuan), Wongwises S (Wongwises, Somchai)

Source: JOURNAL OF THERMOPHYSICS AND HEAT TRANSFER **Volume:** 21 **Issue:** 2 **Pages:** 361-371 **Published:** APR-JUN 2007

Times Cited: 6 **References:** 28 Citation Map

Abstract: The present study proposes a new reduction method for analyzing the heat and mass transfer characteristics of wavy fin-and-tube heat exchangers under dehumidifying conditions. The analysis of the fin-and-tube heat exchangers is carried out via dividing it into many tiny segments. The proposed reduction method is applicable to the wavy fin-and-tube heat exchanger under fully wet and partially wet conditions. From the experimental results, it is found that the sensible heat transfer and mass transfer characteristics are slightly sensitive to changes in the inlet relative humidity and fin spacing. For high Reynolds numbers where partially wet condition takes place, a considerable increase of the mass transfer characteristic is encountered. The ratios of h(c,o)/h(d,o)C(p,a) are in the range 0.6-1.2. Correlations are proposed to describe the heat and mass transfer characteristics. These correlations can describe 99.18% of the heat transfer characteristic within 20% and 86.89% of the mass transfer characteristic within 20%. Correspondingly, 88.93% of the ratios of h(c,o)/h(d,o)C(p,a) are predicted by the proposed correlation within 20%.

Document Type: Article

Language: English

KeyWords Plus: HERRINGBONE WAVY FIN; FRICTION CHARACTERISTICS; DEHUMIDIFYING CONDITIONS; FLOW; PERFORMANCE

Reprint Address: Wongwises, S (reprint author), King Mongkuts Univ Technol Thonburi, Thermal Engn & Multiphase Flow Res Lab, Bangkok 10140, Thailand

Addresses:
1. King Mongkuts Univ Technol Thonburi, Thermal Engn & Multiphase Flow Res Lab, Bangkok 10140, Thailand
2. Burapha Univ, Dept Mech Engn, Fac Engn, Chon Buri 20131, Thailand

E-mail Addresses: somchai.won@kmutt.ac.th

Cited by: 6

This article has been cited 6 times (from Web of Science).

Huang CH, Chung YL A NONLINEAR FIN DESIGN PROBLEM IN ESTIMATING THE OPTIMAL SHAPES OF LONGITUDINAL AND SPINE FULLY WET FINS NUMERICAL HEAT TRANSFER PART A-APPLICATIONS 57 10 749-776 2010

Pirompugd W, Wang CC, Wongwises S Correlations for wet surface ratio of fin-and-tube heat exchangers INTERNATIONAL JOURNAL OF HEAT AND MASS TRANSFER 53 1-3 568-573 JAN 15 2010

Huang CH, Chung YL An Optimal Fin Design Problem in Estimating the Shapes of Longitudinal and Spine Fully Wet Fins CMES-COMPUTER MODELING IN ENGINEERING & SCIENCES 44 3 249-279 APR 2009

[view all 6 citing articles]

Create Citation Alert

Related Records:

Find similar records based on shared references (from Web of Science).

[view related records]

References: 28

View the bibliography of this record (from Web of Science).

Suggest a correction

If you would like to improve the

Journal of
Mechanical
Science and
Technology

Journal of Mechanical Science and Technology 21 (2007) 2168~2177

A comparison of the effect of the electrohydrodynamic technique on the condensation heat transfer of HFC-134a inside smooth and micro-fin tubes

Suriyan Laohalertdecha and Somchai Wongwises*

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

(Manuscript Received October 23, 2006; Revised June 13, 2007; Accepted June 18, 2007)

Abstract

The results of the condensation heat transfer enhancement and pressure drop of HFC-134a by using the electrohydrodynamic (EHD) technique are presented. The test section is a horizontal tube-in-tube heat exchanger where the refrigerant flows in the inner tube and water flows in the annulus. The outer tube is a smooth copper tube having outer diameter of 21.2 mm. Two types of inner tubes, smooth and micro-fin copper tubes, are tested. The outer diameter and length of both inner tubes is 9.52 mm and 2.5 m, respectively. A stainless steel cylindrical electrode of 1.47 mm in diameter is placed in the center of the tube. Experiments are conducted under conditions providing mass flux of 400 kg/m^2s, saturated temperature of 40 °C, heat flux of 20 kW/m^2 and applied voltage of 2.5 kV. The experimental results indicate that the EHD enhancements of the smooth tube are higher than those of the micro-fin tube over the range of average quality. The maximum heat transfer enhancements for smooth and micro-fin tubes are 1.1 times and 1.08 times, respectively. For a smooth tube, the pressure drop induced by EHD is considerably small. However, the application of EHD in a micro-fin tube can lead to 10 % increase in the pressure drop.

Keywords: Electrohydrodynamic; Condensation; Heat transfer enhancement; Smooth tube; Micro-fin tube

1. Introduction

Chlorofluorocarbons (CFCs), developed in the 1930s, are widely used as working fluids in many applications such as air conditioning and refrigeration because they are non toxic and non flammable. Unfortunately, chlorine molecules from CFCs destroy the ozone layer. HFCs consisting of carbon, hydrogen and fluorine are being used as alternatives to CFCs as they do not contain chlorine and are absolutely harmless to the ozone layer. Thus, HFCs can be used instead of CFCs because their thermo physical properties are very similar to those of CFCs. However, even though the difference in properties between both refrigerants is small, it may result in significant differences in the overall system performance. Therefore, the properties of HFCs should be studied in detail before they are applied.

The improvement in heat transfer performance of heat exchangers enables a considerable decrease in their size. In general, heat transfer enhancement techniques can be divided into two types: active and passive. This paper concentrates on the active techniques, especially the electrohydrodynamic (EHD) technique. The EHD technique is an active heat transfer enhancement technique that can induce secondary fluid motion which can enhance heat and mass transfer in single and two-phase flows. This interaction can result in the increase of fluid motion leading to a higher heat transfer coefficient. The general expression of EHD force is shown in Eq. (1).

*Corresponding author. Tel.: +662 470 9115, Fax.: +662 470 9111
E-mail address: somchai.won@kmutt.ac.th

ELSEVIER

Available online at www.sciencedirect.com
ScienceDirect
Renewable and Sustainable Energy Reviews
11 (2007) 858–876

RENEWABLE
& SUSTAINABLE
ENERGY REVIEWS

www.elsevier.com/locate/rser

A review of electrohydrodynamic enhancement of heat transfer

Suriyan Laohalertdecha[a], Paisarn Naphon[b], Somchai Wongwises[b,*]

[a]The Joint Graduate School of Energy and Environment (JGSEE), King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

[b]Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology, Thonburi, Bangmod, Bangkok 10140, Thailand

Received 1 April 2005; accepted 19 July 2005

Abstract

The heat transfer duty of heat exchangers can be improved by heat transfer enhancement techniques. In general, these techniques can be divided into two groups: active and passive techniques. The active techniques require external forces, e.g. electric field, acoustic or surface vibration etc. The passive techniques require fluid additives or special surface geometries. Electrohydrodynamic (EHD) techniques have been introduced as one of the types of active heat transfer enhancement techniques. This paper presents a review of research works on electrohydrodynamic heat transfer enhancement. This paper can be used as the first guideline for the researcher in using EHD techniques for heat transfer enhancement.

Keywords: Electrohydrodynamic; Condensation heat transfer; Boiling heat transfer; Heat transfer enhancement

Contents

*Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: Somchai.won@kmutt.ac.th (S. Wongwises).

1364-0321/$ - see front matter
doi:10.1016/j.rser.2005.07.002

Heat Mass Transfer (2007) 43:871–880
DOI 10.1007/s00231-006-0170-2

ORIGINAL

Effect of EHD on heat transfer enhancement during two-phase condensation of R-134a at high mass flux in a horizontal smooth tube

Suriyan Laohalertdecha · Somchai Wongwises

Received: 25 March 2005 / Accepted: 4 July 2006 / Published online: 5 August 2006

Abstract In this study, effect of electrohydrodynamic (EHD) on the condensation heat transfer enhancement and pressure drop of pure R-134a are experimentally investigated. The test section is a 2.5 m long counterflow double tube heat exchanger with refrigerant flowing in the inner tube and cooling water flowing in the annulus. The inner tube is made from smooth horizontal copper tubing of 9.52 mm outer diameter. The electrode is made from stainless steel wire of 1.47 mm diameter. The test runs are performed at average saturated temperatures ranging between 40 and 60°C, mass flux ranging between 200 and 600 kg/m^2 s, heat flux ranging between 10 and 20 kW/m^2 and applied voltage at 2.5 kV. For the presence of the electrode, the experimental results indicate that the maximum heat transfer enhancement ratio is around 30% while the maximum increase in pressure drop is about 25%.

List of symbols

Cp	specific heat at constant pressure (J/kg K)
E	electric field strength (V/m)
f_e	EHD force density (N/m^3)
G	mass flux (kg/m^2 s)
h	heat transfer coefficient (W/m^{2}K)
i	specific enthalpy (J/kg)
L	tube length (m)
m	mass flow rate (kg/s)
T	temperature (°C)
Q	heat transfer rate (W)
q	electric field space charge density (C/m^3)
q''	average heat flux (W/m^2)
x	quality (kg/kg)

Greek letters

ε	permittivity of the liquid (F/m)
ε_0	permittivity of free space (8.854×10^{-12}) (F/m)
κ	relative permittivity $\left(\frac{\varepsilon}{\varepsilon_0}\right)$
ρ	condensate density (kg/m^3)

Subscripts

avg	average
e	with EHD
o	without EHD
out	outlet
ref	refrigerant
fg	vaporization latent quantity
in	inlet
ph	pre-heater
sat	saturation
TS	test section
w	water

S. Laohalertdecha · S. Wongwises (✉)
Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE),
Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi,
Bangmod, Bangkok 10140, Thailand
e-mail: somchai.won@kmutt.ac.th

1 Introduction

The heat transfer duty of heat exchangers can be improved by heat transfer enhancement techniques. In

Available online at www.sciencedirect.com
ScienceDirect
International Journal of Heat and Mass Transfer 50 (2007) 552–565
International Journal of HEAT and MASS TRANSFER
www.elsevier.com/locate/ijhmt

Finite circular fin method for heat and mass transfer characteristics for plain fin-and-tube heat exchangers under fully and partially wet surface conditions

Worachest Pirompugd [a,b], Chi-Chuan Wang [c], Somchai Wongwises [b,*]

[a] Department of Mechanical Engineering, Faculty of Engineering, Burapha University, Saensook, Muang, Chonburi 20131, Thailand
[b] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, 91 Suksawas Road, 48 Bangmod, Thungkru, Bangkok 10140, Thailand
[c] Energy and Resources Lab., Industrial Technology Research Institute, Hsinchu 310, Taiwan, ROC

Received 14 December 2005; received in revised form 15 July 2006
Available online 2 October 2006

Abstract

This study proposes a new method, namely the "finite circular fin method" (FCFM), to analyze the performance of fin-and-tube heat exchangers having plain fin configuration under dehumidifying conditions. The analysis is done by dividing the heat exchanger into many tiny segments (number of tube rows × number of tube passes per row × number of fins). The tiny segments are distinguished into three types: the fully dry, partially wet or fully wet surface conditions. The proposed method is capable of handling fully and partially wet surfaces. From the test results, it is found that the sensible heat transfer performance and the mass transfer performance are insensitive to changes of fin pitch. The influence of inlet relative humidity on the sensible heat transfer performance is small, and is almost negligible when the number of tube rows is above four. For one and two row configurations, considerable increase of mass transfer performance is encountered when partially wet condition takes place. The sensible heat transfer coefficient is about the same for those in fully wet and partially wet conditions provided that the number of tube row is equal or greater than four. Correlations applicable for both fully wet and partially wet conditions are proposed to describe the heat and mass performance for the present plain fin configuration.

Keywords: Finite circular fin method; Fin-and-tube heat exchangers; Dehumidifying conditions; Sensible heat transfer performance; Mass transfer performance

1. Introduction

Fin-and-tube heat exchangers are widely used in applications of air-conditioning and refrigeration systems. They can be applicable to condensers and evaporators. In evaporators, which typically use aluminum fins with the surface temperature generally being below the dew point temperature. As a result, simultaneous heat and mass transfer occurs along the fin surfaces.

Many studies have been published on the heat and mass transfer characteristics of fin-and-tube heat exchangers under dehumidifying conditions. For instance, McQuiston [1,2] presented experimental data for five plate fin-and-tube heat exchangers, and developed a well-known heat transfer and friction correlation for both dry and wet surfaces. Mirth and Ramadhyani [3,4] investigated the heat and mass transfer characteristics of wavy fin heat exchangers. Their results showed that the Nusselt number was very sensitive to changes of the inlet dew point temperature, and that the Nusselt number decreased with an increase of dew point temperatures. Similar results were reported by Fu et al. [5] in dehumidifying heat exchangers having a louver fin configuration. They reported a pronounced decrease of the wet sensible heat transfer coefficients with the rise of the inlet relative humidity. Contrary to this, the experimental

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0017-9310/$ - see front matter © 2006 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ijheatmasstransfer.2006.07.017

Available online at www.sciencedirect.com
ScienceDirect
Computers and Chemical Engineering 31 (2007) 318–333
Computers & Chemical Engineering
www.elsevier.com/locate/compchemeng

Simulation of freezing of water-saturated porous media in a rectangular cavity under multiple heat sources with different temperature using a combined transfinite interpolation and PDE methods

P. Rattanadecho [a,*], S. Wongwises [b]

[a] *Faculty of Engineering, Thammasat University (Rangsit Campus), Pathumthani 12121, Thailand*
[b] *King Mongkut University of Technology Thonburi, 91 Suksawas 48, Rasburana, Bangkok 10140, Thailand*

Received 11 January 2006; received in revised form 17 July 2006; accepted 21 July 2006
Available online 22 September 2006

Abstract

A numerical study is made of the freezing of water-saturated porous media in a rectangular cavity subjected to multiple heat sources with different temperature. Focus is placed on establishing a computationally efficient approach for solving multi-dimensional moving boundary problem in a two-dimensional structured grids. Preliminary grids are first generated by an algebraic method, based on a transfinite interpolation method, with subsequent refinement using a PDE mapping (parabolic grid generation) method. A preliminary case study indicates successful implementation of the numerical procedure. A two-dimensional freezing model is validated against available analytical solution and experimental results.

Keywords: Freezing; Porous media; Transfinite interpolation; Moving boundary

1. Introduction

Transient heat transfer problems involving melting or solidification are generally referred to as "phase change" or "moving boundary" problems. They are an important topics which spans a broad spectrum of scientific and engineering disciplines such as the freezing or thawing of soil, ice formation, crystal growth, aerodynamic ablation, casting of metal, food processing and numerous others. Generally, the solution of moving boundary problem with phase transition has been of special interest due to the inherent difficulties associated with the non-linearity of the interface conditions and the unknown locations of the arbitrary moving boundaries. The some up to date reviews of these problems are available (Charn-Jung & Kaviany, 1992; Chellaiah & Viskanta, 1988; Frivik & Comini, 1982; Hasan, Mujumdar, & Weber, 1991; Hashemi & Sliepcevich, 1973; Murray & Landis, 1959; Sparrow & Broadbent, 1983; Weaver & Viskanta, 1986).

In the past, a variety of conventional numerical techniques have been developed for solving these problems, including the enthalpy (Crowley, 1978; Shamsundar & Sparrow, 1976), apparent heat capacity (Bonacina, Comini, Fasano, & Primicerio, 1973), isotherm migration (Crank & Gupta, 1975), and coordinate transformation methods (Cheung, Chawla, & Pedersen, 1984; Hsu, Sparrow, & Patankar, 1981; Sparrow & Chuck, 1984; Sparrow, Ramadhyani, & Patankar, 1978; Rattanadecho, 2004a, b). These methods have been introduced by researchers mainly to overcome the difficulties in handling moving boundaries. Previous works on multi-dimensional moving boundary problems include Duda, Malone, Notter, and Vrentas (1975), Saitoh (1978), Gong and Mujumdar (1998), Cao, Huang, and Russell (1999), Khillarkar, Gong, and Mujumdar (2000), Chatterjee and Prasad (2000), Beckett, MacKenzie, and Robertson (2001), and Ratanadecho, Aoki, and Akahori (2002).

Conventionally numerical methods have been widely used due to easy to handle numerical algorithms for phase change problem. However, in numerical approximations used in this method with discontinuous coefficients, often the largest numerical errors are

* Corresponding author.
E-mail address: ratphadu@engr.tu.ac.th (P. Rattanadecho).

0098-1354/$ – see front matter © 2006 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compchemeng.2006.07.016

Available online at www.sciencedirect.com
ScienceDirect
Renewable and Sustainable Energy Reviews
11 (2007) 512–523

RENEWABLE
& SUSTAINABLE
ENERGY REVIEWS

www.elsevier.com/locate/rser

Critical review of heat transfer characteristics of nanofluids

Visinee Trisaksri[a], Somchai Wongwises[b,*]

[a]*Energy Division, The Joint Graduate School of Energy and Environment, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*
[b]*Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

Received 12 January 2005; accepted 24 January 2005

Abstract

Researches in heat transfer have been carried out over the previous several decades, leading to the development of the currently used heat transfer enhancement techniques. The use of additives is a technique applied to enhance the heat transfer performance of base fluids. Recently, as an innovative material, nanometer-sized particles have been used in suspension in conventional heat transfer fluids. The fluids with these solid-particle suspended in them are called 'nanofluids'. The suspended metallic or nonmetallic nanoparticles change the transport properties and heat transfer characteristics of the base fluid. The aim of this review is to summarize recent developments in research on the heat transfer characteristics of nanofluids for the purpose of suggesting some possible reasons why the suspended nanoparticles can enhance the heat transfer of conventional fluids and to provide a guide line or perspective for future research.

Keywords: Nanofluid; Nanoparticle; Suspension; Heat transfer enhancement

Contents

*Corresponding author. Tel.: +66 2470 9115; fax: +66 2470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

1364-0321/$ - see front matter © 2006 Published by Elsevier Ltd.
doi:10.1016/j.rser.2005.01.010

International Communications in Heat and Mass Transfer 35 (2008) 1147–1152

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

Two-phase friction factor in vertical downward flow in high mass flux region of refrigerant HFC-134a during condensation ☆

A.S. Dalkilic [a,*], S. Laohalertdecha [b], S. Wongwises [b,*]

[a] *Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University, Yildiz, Istanbul 34349, Turkey*
[b] *Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

ARTICLE INFO

Available online 15 July 2008

Keywords:
Condensation
Two-phase pressure drop
Friction factor
Vertical downward flow
HFC-134a
Equivalent Reynolds number model

ABSTRACT

The two-phase pressure drop of the pure refrigerant HFC-134a during condensation inside a vertical tube-in-tube heat exchanger was investigated. The double tube test section was 0.5 m long with refrigerant flowing in the inner tube and cooling water flowing in the annulus. The inner tube was constructed from smooth copper tubing of 8.1 mm inner diameter and 9.52 mm outer diameter. The test runs were performed at average condensing temperatures of 40–50 °C. The mass fluxes were between 260 and 515 kg m^{-2} s^{-1} and the heat fluxes between 11.3 and 55.3 kW m^{-2}. The quality of the refrigerant in the test section was calculated using the temperature and pressure obtained from the experiment. The pressure drop across the test section was directly measured by a differential pressure transducer. A new correlation for the two-phase friction factor of R134a flow is proposed by means of the equivalent Reynolds number model. The effects of heat flux, mass flux and condensation temperature on the pressure drop are also discussed.

1. Introduction

The heat transfer and pressure drop characteristics of refrigerants have been studied by a large number of researchers. However, study of the pressure drop of refrigerants during downward condensation in small diameter vertical tubes has received comparatively little attention in the literature. A brief summary of pressure drop studies of downward condensation is given as follows:

Goodykoontz and Dorsch [1] investigated the local condensation heat transfer coefficients and pressure distribution of R113 for the mass fluxes of 21–455 kg m^{-2} s^{-1} in a 7.4–15.9 mm i.d. vertical tube. Kim and No [2] developed a turbulent film condensation model including pressure drop for high pressure steam in a 46 mm i.d. vertical tube. Ma et al. [3] studied the two-phase friction factors of downward flow of R113 for the mass fluxes of 400–800 kg m^{-2} s^{-1} in a 20.8 mm i.d. smooth and micro-fin tubes.

Akers et al. [4] developed a two-phase multiplier by assuming that two-phase flows are similar to single-phase flows. Their correlation predicts a frictional two-phase pressure drop by means of a multiplying factor, which uses the same rationale as the Lockhart–Martinelli multiplier [5]. Their model is known as the "equivalent Reynolds number model". It can be used for an annular flow regime. According to this model, the equivalent all liquid flow produces the same wall shear stress as that of the two-phase flow. Several researchers have used this model in their work, for example, Moser et al. [6].

To the best of the authors' knowledge, there has been insufficient work dealing with the two-phase pressure drop during downward condensation in small diameter tubes. Although some information is currently available on the two-phase friction factor in annular flow, there still remains room to further discuss whether it gives reliable predictions of friction factors in the high mass flux region. Therefore, the main aim of this study was to extend the existing pressure drop data for R134a during downward condensation to the high mass flux region. The results from the large amount of experimental data were correlated with the equivalent Reynolds number and presented as a new empirical correlation to predict the two-phase friction factor. The effects of various relevant parameters on pressure drop are also discussed.

2. Experimental apparatus and method

A schematic diagram of the test apparatus is shown in Fig. 1. The refrigerant loop consists of a pre-heating loop, test section, cooling loop and chilling loop. The refrigerant is circulated by a gear pump controlled by an inverter. The refrigerant flows in series through a filter/dryer, a sight glass tube, a refrigerant flow meter and a pre-heater and enters the test section. A spiral counter-flow double tube heat exchanger is designed to supply heat to control the inlet quality of the refrigerant before entering the test section. After exiting the test section, the chilling loop condenses and sub-cools the refrigerant and removes the heat input from the pre-heater and test section, and ejects it into the surroundings. After leaving the chilling loop, the

☆ Communicated by W.J. Minkowycz.

* Corresponding authors.
E-mail addresses: dalkilic@yildiz.edu.tr (A.S. Dalkilic), somchai.won@kmutt.ac.th (S. Wongwises).

0735-1933/$ – see front matter © 2008 Published by Elsevier Ltd.
doi:10.1016/j.icheatmasstransfer.2008.06.002

ADVANCES IN ATMOSPHERIC SCIENCES, VOL. 25, NO. 3, 2008, 489–506

Simulation of Seasonal Circulations and Thermohaline Variabilities in the Gulf of Thailand

Nitima ASCHARIYAPHOTHA*[1,2], Prungchan WONGWISES[1], Somchai WONGWISES[3], Usa Wannasingha HUMPHRIES[2], and YOU Xiaobao[4] (游小宝)

[1] *The Joint Graduate School of Energy and Environment, King Mongkut's University of Technology Thonburi, Bangkok* 10140, *Thailand*

[2] *Department of Mathematics, King Mongkut's University of Technology Thonburi, Bangkok* 10140, *Thailand*

[3] *Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangkok* 10140, *Thailand*

[4] *Institute of Atmospheric Physics, Chinese Academy of Sciences, Beijing* 100029

(Received 20 March 2006; revised 20 April 2007)

ABSTRACT

Based on the Princeton Ocean Model (POM), the seasonal thermohaline feature and the ocean circulation in the Gulf of Thailand (GoT), situated between 6°N to 14°N latitude and 99°E to 105°E longitude, were studied numerically with 37×97 orthogonal curvilinear grid and 10 vertical sigma levels conforming to a realistic bottom topography. A spin-up phase of the first model run was executed using wind stress calculated from climatological monthly mean wind, restoring-type surface heat and salt, and climatological monthly mean fresh water flux data. In this paper, the temperature and salinity fields taken from Levitus94 data sets and the calculated temperature and salinity from the model run for 12-month mean and for each season are presented where the winter, summer, rainy, and end of the rainy seasons of Thailand are represented by the months January, April, July, and October, respectively. The simulated circulations are also described. The results show that the temperature in the GoT is warmer than the temperature of the other parts connected to the South China Sea (SCS). At any depth of inflow from SCS into the GoT, the salinity is high, but in the outflow from the GoT at the surface, the salinity is low. The strong circulations are clockwise during summer and the rainy seasons of Thailand, which are the East Asian monsoon periods, northeasterly and southwesterly during summer. They occur near Pattani and Narathiwat provinces during summer and in the central GoT during the rainy seasons. Sensitivity experiments were designed to investigate the effects of wind forcing and open boundary conditions. Wind forcing is shown to be the important factor for generating the circulation in the GoT. The lateral velocity at the open boundaries is of considerable importance to current circulation for the rainy and end of the rainy seasons, with insignificant effect for the winter and summer seasons of Thailand.

Key words: seasonal circulation, Gulf of Thailand, spin-up phase, thermohaline, curvilinear grid

DOI: 10.1007/s00376-008-0489-3

1. Introduction

The Gulf of Thailand (GoT) is located in Southeast Asia immediately to the west of the South China Sea (SCS). The Gulf is a semi-enclosed sea that measures approximately 400-km by 800-km, covering an area of about 320000 km^2. Its location in the global map is between 6°–14°N and 99°–105°E surrounded by the Kingdom of Cambodia, Malaysia, the Kingdom of Thailand and the Socialist Republic of Vietnam (Fig. 1). It is a part of the Sunda Shelf, which is a submerged connection between Southeast Asia, Malaysia, Sumatra, Java, and Borneo, and is relatively shallow. The mean and maximum depths in the central part in the GoT are about 45 m and 80 m, respectively. The Chao Phraya, Tha Chin, Mae Klong, and Bang

*Corresponding author: Nitima ASCHARIYAPHOTHA, nitima.asc@kmutt.ac.th

Web of Science®

Testing of a Low-Temperature Differential Stirling Engine by Using Actual Solar Energy

Author(s): Kongtragool B (Kongtragool, Bancha)[1], Wongwises S (Wongwises, Somchai)[1]

Source: INTERNATIONAL JOURNAL OF GREEN ENERGY **Volume:** 5 **Issue:** 6 **Pages:** 491-507 **Published:** 2008

Times Cited: 0 **References:** 21 Citation Map

Abstract: In this article, the construction and performance of a small low-temperature differential Stirling engine is presented. The engine is connected to a small DC generator and tested with actual solar energy by using a parabolic-dish concentrator as a heat source. Engine performance tested with actual solar energy is reported. Variations of engine-generator power output, combined engine and generator efficiency, and overall efficiency with direct normal intensity are presented.

Document Type: Article

Language: English

Author Keywords: Stirling engine; Low-temperature differential Stirling engine; Hot-air engine; Solar-powered heat engine; Parabolic-dish concentrator

KeyWords Plus: DESIGN CONSIDERATION; PERFORMANCE

Reprint Address: Wongwises, S (reprint author), King Mongkuts Univ Technol Tornburi, 126 Prachautit Rd, Bangkok 10140, Thailand

Addresses:
1. King Mongkuts Univ Technol Thonburi, Fluid Mech Thermal Engn & Multiphase Flow Res Lab, Dept Mech Engn, Fac Engn, Bangkok 10140, Thailand

E-mail Addresses: somchai.won@kmutt.ac.th

Funding Acknowledgement:

Funding Agency	Grant Number
Joint Graduate School of Energy and Environment (JGSEE)	
Thailand Research Fund (TRF)	

[Show funding text]

Publisher: TAYLOR & FRANCIS INC, 325 CHESTNUT ST, SUITE

Moving boundary-moving mesh analysis of freezing process in water-saturated porous media using a combined transfinite interpolation and PDE mapping methods

Author(s): Rattanadecho P (Rattanadecho, P.)[1], Wongwises S (Wongwises, S.)[2]

Source: JOURNAL OF HEAT TRANSFER-TRANSACTIONS OF THE ASME **Volume:** 130 **Issue:** 1 **Article Number:** 012601 **Published:** JAN 2008

Times Cited: 0 **References:** 32 Citation Map

Abstract: This paper couples the grid generation algorithm with the heat transport equations and applies them to simulate the thermal behavior of freezing process in water-saturated porous media. Focus is placed on establishing a computationally efficient approach for solving moving boundary heat transfer problem, in two-dimensional structured grids, with specific application to an undirectional solidification problem. Preliminary grids are first generated by an algebraic method, based on a transfinite interpolation method, with subsequent refinement using a partial differential equation (PDE) mapping (parabolic grid generation) method. A preliminary case study indicates successful implementation of the numerical procedure. A two-dimensional solidification model is then validated against available analytical solution and experimental results and subsequently used as a tool for efficient computational prototyping. The results of the problem are in good agreement with available analytical solution and experimental results.

Document Type: Article

Language: English

Author Keywords: solidification; transfinite interpolation; PDE mapping; moving boundary

KeyWords Plus: FINITE-ELEMENT-METHOD; GRANULAR PACKED-BED; MIXED CONVECTION; HEAT-TRANSFER; CONFIGURATIONS; LIQUID; FLOW

Reprint Address: Rattanadecho, P (reprint author), Thammasat Univ, Fac Engn, Rangsit Campus, Pathum Thani 12121, Thailand

Addresses:
1. Thammasat Univ, Fac Engn, Pathum Thani 12121, Thailand
2. King Mongkuts Univ Technol, Dept Mech Engn, Bangkok 10140, Thailand

E-mail Addresses: ratphadu@engr.tu.ac.th

Available online at www.sciencedirect.com
ScienceDirect
Applied Thermal Engineering 28 (2008) 467–477
APPLIED THERMAL ENGINEERING
www.elsevier.com/locate/apthermeng

Experimental study on R-134a refrigeration system using a two-phase ejector as an expansion device

Praitoon Chaiwongsa, Somchai Wongwises *

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Received 8 May 2006; accepted 14 May 2007
Available online 2 June 2007

Abstract

This paper is a continuation of the authors previous work. In the present paper, the performance of the refrigeration cycle using a two-phase ejector as an expansion device is experimentally investigated. Refrigerant R-134a is used as working fluid. Motive nozzles having three different outlet diameters are tested. New experimental data that have never been seen before are presented on the effects of the external parameters i.e. heat sink and heat source temperatures on the coefficient of performance and various relevant parameters i.e. primary mass flow rate of the refrigerant, secondary mass flow rate of the refrigerant, recirculation ratio, average evaporator pressure, compressor ratio, discharge temperature and cooling capacity. The effects of size of the motive nozzle outlet on the system performance are also discussed.

Keywords: Two-phase ejector; Refrigeration; Liquid-recirculation; Coefficient of Performance; Motive nozzle; Expansion device

1. Introduction

In vapor refrigeration cycle, in order to accommodate the low temperature evaporation process, a saturated or sub-cooled liquid produced from condensation at high temperature must be expanded through an expansion device to low-pressure. Throttling process or isenthalpic process is the process occurring through an expansion device. This process produces the thermodynamic losses and causes a larger amount of the refrigerant to flash into a vapor than in the isentropic process.

In order to reduce the thermodynamic loss in the expansion process, various researchers have attempted to look for other appropriate devices. Due to the low cost, no moving parts and ability to handle two-phase flow without damage, an ejector is considered as an attractive expansion device in the refrigeration system.

To the best of the authors' knowledge, information on the application of an ejector as an expansion device in a refrigeration cycle is still limited.

Kornhauser [1] investigated the thermodynamic performance of the ejector expansion refrigeration cycle by using R-12 as a refrigerant under standard conditions, −15 °C and 30 °C for evaporator and condensor temperatures, respectively. A theoretical COP improvement of up to 21% over the standard cycle was found. This result was based on ideal cycle and constant mixing pressure in the ejector. Harrell et al. [2] used a R-134a two-phase ejector and the test rig to estimate the COP of the refrigeration cycle. It was found that the COP improvement ranged from 3.9% to 7.6%. Menegay et al. [3] developed a bubbly flow tube to reduce the thermodynamic non-equilibrium in the motive nozzle with R-12 as the refrigerant. This device was installed upstream of the motive nozzle. The COP of the system using the bubbly flow tube can be improved up to 3.8% over the conventional cycle under standard conditions. However, they reported that the result was not as good as was expected and study of the ejector expansion

* Corresponding author. Tel.: +662 470 9115; fax: +662 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

1359-4311/$ - see front matter © 2007 Published by Elsevier Ltd.
doi:10.1016/j.applthermaleng.2007.05.005

International Communications in Heat and Mass Transfer 35 (2008) 921–927

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

Effect of void fraction models on the two-phase friction factor of R134a during condensation in vertical downward flow in a smooth tube ☆

A.S. Dalkilic [a,*], S. Laohalertdecha [b], S. Wongwises [b,*]

[a] Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University, Yildiz, Istanbul 34349, Turkey
[b] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Available online 23 May 2008

Keywords:
Heat transfer
Void fraction
Friction factor
Condensation
Downward flow
R134a

ABSTRACT

The void fraction of R134a condensing inside a vertical smooth tube is experimentally investigated in this study. The vertical test section is a 0.5 m long countercurrent flow double tube heat exchanger with refrigerant flowing down the inner tube and cooling water flowing upward in the annulus. The inner tube is made from smooth copper tubing and has an 8.1 mm inner diameter. The test runs are done at an average saturated condensing temperature of 40 °C. The average qualities are between 0.8 and 0.99 while the mass fluxes are 300 kg $m^{-2}s^{-1}$ and the heat fluxes are between 22 and 39.8 kW m^{-2}. The void fractions are indirectly determined using relevant measured data together with various void fraction models and correlations reported in the open literature. The friction factors obtained from various void fraction models and correlations are compared with each other and also with those determined from graphical information provided by Bergelin et al. The effect of void fraction alteration on the momentum pressure drop is also presented.

1. Introduction

Gas–liquid two-phase separated flow is commonly analyzed using the slip flow model. In this model, it is assumed that the two phases are seperated with different uniform velocities. Void fraction, defined as the cross-sectional area occupied by the vapor in relation to the area of the flow channel, is an important parameter and is always used to determine the flow pattern transition, heat transfer coefficient and two-phase pressure drop.

Over the years, many studies have been done on the modelling of void fraction and can be divided in several groups: slip ratio void fraction models and correlations [1–14], Lockhart Martinelli parameter based void fraction models and correlations [15–23], flow regime based void fraction models and correlations [24–26], and $K\varepsilon_H$ parameter based void fraction and correlations [27–29] and general void fraction models and correlations [30–34]. A summary of these works is shown in Table 1.

Heat transfer and pressure drop characteristics, annular flow conditions along the tube length including convective condensation occur in many real applications. Nusselt [35] suggested the first theoretical solution for predicting heat transfer coefficients. He assumed laminar film flowing down with no entrainment on a vertical plate, without waves or the interfacial shear effect between the phases, having a linear temperature profile through the film.

Akers et al. [36] developed a two-phase multiplier based correlation with assumption that two-phase flows were similar to single phase flows. He predicted frictional two-phase pressure drop by means of a multiplying factor which was the same as the rationale behind the Lockhart-Martinelli two-phase multiplier [15]. His model is known as the "equivalent Reynolds number model" in the literature. It has been used for the annular flow regime which could be replaced by an equivalent all liquid flow that produces the same wall shear stress as that of the two-phase flow according to this model. There are many researchers who have studied this model such as Moser et al. [37] and Ma et al. [38].

Although void fraction models or void fraction correlations have been extensively developed in the past, no attention was paid to the comparison among the various correlations or models. As a consequence, the main concern of this study is to determine void fraction from several sets of basic heat transfer data using various void fraction models and correlations. The different void fraction values obtained are then discussed. The effect of void fraction on the momentum pressure drop is also shown in this paper.

2. Experimental apparatus and procedure

A schematic diagram of the test apparatus is shown in Fig. 1. The refrigerant loop consists of a pre-heating loop, test section, cooling

☆ Communicated by W.J. Minkowycz.
* Corresponding authors.
E-mail addresses: dalkilic@yildiz.edu.tr (A.S. Dalkilic), somchai.won@kmutt.ac.th (S. Wongwises).

0735-1933/$ – see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2008.04.001

ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

Solar Energy 82 (2008) 493–500

SOLAR ENERGY

www.elsevier.com/locate/solener

A four power-piston low-temperature differential Stirling engine using simulated solar energy as a heat source

Bancha Kongtragool, Somchai Wongwises *

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Received 21 August 2007; received in revised form 3 December 2007; accepted 14 December 2007
Available online 11 January 2008

Communicated by: Associate Editor Robert Pitz-Paal

Abstract

In this paper, the performances of a four power-piston, gamma-configuration, low-temperature differential Stirling engine are presented. The engine is tested with air at atmospheric pressure by using a solar simulator with four different solar intensities as a heat source. Variations in engine torque, shaft power and brake thermal efficiency with engine speed and engine performance at various heat inputs are presented. The Beale number obtained from the testing of the engine is also investigated. The results indicate that at the maximum actual energy input of 1378 W and a heater temperature of 439 K, the engine approximately produces a maximum torque of 2.91 N m, a maximum shaft power of 6.1 W, and a maximum brake thermal efficiency of 0.44% at 20 rpm.

Keywords: Stirling engine; Hot-air engine; Solar-powered heat engine; Solar simulator

1. Introduction

The low-temperature differential (LTD) Stirling engine is a type of Stirling engine that can run with a small temperature difference between the hot and cold ends of the displacer cylinder. The LTD Stirling engine is therefore able to operate with various low-temperature heat sources.

Some characteristics of the LTD Stirling engine are as follows:

(1) Displacer to power-piston swept volumes ratio or compression ratio is large.
(2) Diameters of displacer cylinder and displacer are large.
(3) Displacer length is short.
(4) Effective heat transfer surfaces on both end plates of the displacer cylinder are large.
(5) Displacer stroke is small.
(6) Dwell period at the end of the displacer stroke is slightly longer than the normal Stirling engine.
(7) Operating speed is low.

While the Stirling engine has been studied by a large number of researchers, the LTD Stirling engine has received comparatively little attention. Many studies related to solar-powered Stirling engines and LTD Stirling engines have been reviewed in the authors' previous works (Kongtragool and Wongwises, 2003a). Some of these works are described as follows:

Haneman (1975) studied the possibility of using air with low-temperature sources. This led to the construction of an unusual engine, in which the exhaust heat was still sufficiently hot to be useful for other purposes.

A simply constructed low-temperature heat engine modeled on the Stirling engine configurations was patented by White (1983). White suggested improving performance by pressurizing the displacer chamber. Efficiencies were

* Corresponding author. Tel.: +662 4709115; fax: +662 4709111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0038-092X/$ - see front matter © 2008 Published by Elsevier Ltd.
doi:10.1016/j.solener.2007.12.005

International Communications in Heat and Mass Transfer 35 (2008) 1320–1326

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

Effect of thermophysical properties models on the predicting of the convective heat transfer coefficient for low concentration nanofluid ☆

Weerapun Duangthongsuk, Somchai Wongwises *

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Available online 5 August 2008

Keywords:
Nanofluids
Convective heat transfer Coefficient
Thermophysical properties

ABSTRACT

The term of nanofluid refers to a solid–liquid mixture with a continuous phase which is a nanometer sized nanoparticle dispersed in conventional base fluids. In order to study the heat transfer behavior of the nanofluids, precise values of thermal and physical properties such as specific heat, viscosity and thermal conductivity of the nanofluids are required. There are a few well-known correlations for predicting the thermal and physical properties of nanofluids which are often cited by researchers to calculate the convective heat transfer behaviors of the nanofluids. Each researcher has used different models of the thermophysical properties in their works. This article aims to summarize the various models for predicting the thermophysical properties of nanofluids which have been commonly cited by a number of researchers and use them to calculate the experimental convective heat transfer coefficient of the nanofluid flowing in a double-tube counter flow heat exchanger. The effects of these models on the predicted value of the convective heat transfer of nanofluid with low nanoparticle concentration are discussed in detail.

1. Introduction

A decade ago, development of thermal engineering stood at the point of increase in heat transfer performance of heat exchangers. Heat transfer fluids such as water, mineral oils and ethylene glycol play an important role in many industry processes including power generation, chemical processes, heating and cooling processes, transportation, microelectronics and other micro-sized applications. The poor heat transfer properties of these fluids are a limitation in improving the heat transfer augmentation and compactness of the heat exchangers. There are many researchers reporting different methods for heat transfer enhancement. An innovative way of improving the heat transfer performance of common fluids is to suspend various types of small solid particles, such as metallic, non-metallic and polymeric particles, in conventional fluids to form colloidal. However, suspended particles of the order of μm or even mm may cause some severe problems in the flow channels, increasing pressure drop, causing the particles to quickly settle out of suspension. In recent years, modern nanotechnology has been discovered. Particles of nanometer dimensions dispersed in base liquids are called nanofluids. This term was first introduced by Choi in 1995 at the Argonne National Laboratory. The ultra fine nanoparticles are normally smaller than 100 nm and have remarkably higher thermal conductivity than base liquids. Compared with millimeter or micrometer sized particle suspensions, nanofluids have shown a number of potential advantages such as better long-term stability and rheological properties, and can have significantly higher thermal conductivities.

In spite of these nanofluids have interesting potential and can be used to replace common fluids for advanced thermal applications, however they are still in the early stages of development. (Mansour et al. [1]) A number of experimental studies were performed to measure the thermal conductivity and the viscosity of nanofluids in approximately a decade ago. There have only been a small number of studies carried out on other aspects of nanofluids, such as phase change and convective heat transfer behavior, which was reported in the published papers of Trisaksri and Wongwises [2], and Daungthongsuk and Wongwises [3], respectively.

A number of researchers have studied and reported the correlations for predicting the thermal conductivity, density, viscosity and specific heat of the nanofluids. Understanding the physical and thermal properties of nanofluid is essential before using nanofluids in practical applications. There are a few important correlations for predicting the thermophysical properties of nanofluids which are frequently cited by a number of researchers. Their works have both experimentally and theoretically investigated the heat transfer behavior of the nanofluids and are summarized as follows:

Pak and Cho [4] studied the heat transfer performance of γ-Al_2O_3 and TiO_2 nanoparticles dispersed in water flowing in a horizontal circular tube. The alumina (Al_2O_3) and titanium dioxide (TiO_2) nanoparticles with diameters of 13 nm and 27 nm respectively were

☆ Communicated by W.J. Minkowycz.

* Corresponding author.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0735-1933/$ – see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2008.07.015

Energy Conversion and Management 49 (2008) 2795–2800

Contents lists available at ScienceDirect

Energy Conversion and Management

journal homepage: www.elsevier.com/locate/enconman

Effects of inlet relative humidity and inlet temperature on the performance of counterflow wet cooling tower based on exergy analysis

Thirapong Muangnoi [a], Wanchai Asvapoositkul [b,*], Somchai Wongwises [b]

[a] The Joint Graduate School of Energy and Environment, King Mongkut's University of Technology Thonburi, Bangmod, Thungkru, Bangkok 10140, Thailand
[b] Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Thungkru, Bangkok 10140, Thailand

ARTICLE INFO

Article history:
Received 21 January 2007
Received in revised form 23 August 2007
Accepted 24 March 2008
Available online 6 May 2008

Keywords:
Exergy analysis
Wet cooling tower
Exergy change
Second law efficiency

ABSTRACT

The purpose of this study is to analyze the influence of the ambient temperature and humidity on the performance of a counterflow wet cooling tower according to the second law, exergy analysis, of thermodynamics. First, the properties of water and air flow through the tower were predicted and validated by the experiment. Exergy analysis then has been carried out for investigating the cooling tower performance with various inlet air conditions, relative humidity and dry bulb temperature, while the water side condition is kept constant. According to the analysis in this paper, the similar result in terms of required dry air flow rate, exergy change of water and that of air, exergy destruction and second law efficiency were obtained for the various inlet air conditions. The exergy change of water Δx_w is higher than that of air Δx_{air}, since Δx_w is the available energy of water to supply to air throughout the tower while Δx_{air} is the available energy of air to recover or utilize that supplied by water. It reveals that Δx_{air} is dominated by the exergy change of air due to evaporative heat transfer. In addition, it gives a clearer explanation of the cooling tower performance and gives clear trends for optimization.

1. Introduction

Wet cooling tower is essentially a semi-enclosed evaporative cooling device [1] undergoing a psychrometric process which is considered as a control-volume system. The working fluids are water and moist air. The cooling tower utilizes the modes of evaporative heat and mass transfer and convective heat transfer to cool warm water to approach the inlet wet bulb temperature, by mean of direct contact between water droplets falling from top and the countercurrent moist air stream.

The analyses of thermal systems such as cooling tower are generally based on the concept of the first law of thermodynamics. The energy conservation has been used to determine the amount of energy transfer between system and environment. The idea is that energy cannot be created or destroyed. On the other hand, energy entering is conserved until leaving the system. However, the energy analysis alone provides no information of energy transfer from the best possible. It shows only a quantity of energy transfer. Thus, it is insufficient to indicate some aspects of energy utilization and, thus, may be misleading. On the contrary, the second law of thermodynamics gives a powerful concept of exergy to fulfill of this incompleteness. Exergy is a measure of the usefulness, quality or potential of energy to cause change, and it appears to be an effective measure of the potential of system to impact the environment [2].

Bejan [3] indicated a method of exergy analysis when the process or system undergoes a control-volume pyschrometric process. It is noted that total exergy is determined as the sum of thermomechanical exergy and chemical exergy where the latter is a major component of total exergy involving mixture, such as moist air at different temperature and composition [4]. Shukuya and Hammache [5] informed that thermomechanical exergy and chemical exergy play an important role in assessing the true thermodynamics merit of psychrometric process application. Furthermore, they outlined the determination of exergy of moist air and water existing in psychrometric process. Lu et al. [6] investigated an exergy change of one cooling tower with ozonation system from various inlet dry bulb temperatures with fixed inlet humidity. An analysis assumed a saturated exit air, and the result showed that the exergy change increased as the increase of inlet dry bulb temperature. However, the analysis neglected chemical exergy component. Chengqin et al. [7] gave the principles of exergy analysis for evaluating HVAC system, with a suggestion of dead state selection in the calculation. Liley [8] tabulated the exergy of moist air for various temperature and pressure. Qureshi and Zubair [9] summarized an analytical framework of exergy and the analysis of cooling tower performance based on second law efficiency. The analysis of second law efficiency varies for two inlet parameters; inlet wet bulb temperature and inlet water temperature. The results

* Corresponding author. Tel.: +662 470 9338; fax: +662 470 9111.
E-mail address: wanchai.asv@kmutt.ac.th (W. Asvapoositkul).

0196-8904/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.enconman.2008.03.019

ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

International Journal of Heat and Mass Transfer 51 (2008) 4002–4017

International Journal of HEAT and MASS TRANSFER

www.elsevier.com/locate/ijhmt

Finite circular fin method for wavy fin-and-tube heat exchangers under fully and partially wet surface conditions

Worachest Pirompugd [a,b], Chi-Chuan Wang [c], Somchai Wongwises [b,*]

[a] *Department of Mechanical Engineering, Faculty of Engineering, Burapha University, Saensook, Muang, Chonburi 20131, Thailand*
[b] *Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*
[c] *Energy and Environment Research Laboratory, Industrial Technology Research Institute, Hsinchu, Taiwan 310, ROC*

Received 4 May 2007; received in revised form 15 November 2007
Available online 5 March 2008

Abstract

The present study proposes the finite circular fin method for analyzing the heat and mass transfer characteristics of wavy fin-and-tube heat exchangers under fully and partially wet surface conditions. The analysis is carried out by dividing the wavy fin-and-tube heat exchanger into many tiny segments. The tiny segments can be analyzed based on surface conditions, i.e. fully wet, fully dry or partially wet surface condition. From the experimental results, it is found that the heat and mass transfer characteristics are insensitive to the inlet relative humidity but the effect of relative humidity on mass transfer characteristic become more pronounced when the partially wet surface condition takes place. The heat transfer characteristic is independent of the fin spacing. Effect of fin spacing on mass transfer characteristic is small when fin spacing is larger than 2.5 mm. However, at smaller fin spacing, the mass transfer characteristic slightly decreases when the relative humidity increases. The ratios of $h_{c,o}/h_{d,o}C_{p,a}$ are in the range of 0.6–1.2. Correlations are proposed to describe the heat and mass transfer characteristics. These correlations can describe 95.63% of the heat transfer characteristic within 15% and 95.14% of the mass transfer characteristic within 20%. Correspondingly, 94.68% of the ratios of $h_{c,o}/h_{d,o}C_{p,a}$ are predicted by the proposed correlation within 20%.
Published by Elsevier Ltd.

Keywords: Finite circular fin method; Fully dry surface condition; Fully wet surface condition; Heat transfer characteristic; Mass transfer characteristic; Partially wet surface condition; Wavy fin-and-tube heat exchangers

1. Introduction

The heat exchange process between two fluids that are at different temperature and separated by a solid wall may be found in air-conditioning, refrigeration, power production and many other thermal processing applications. The most widely used device used to implement the heat exchange process in association with the application of air-conditioning and refrigeration system is called a fin-and-tube heat exchanger which is actually an air-cooled heat exchanger with dominant resistance usually being in the air side. Hence enhanced surfaces are usually adopted for improving the overall heat transfer performance. Among the fin surfaces, the wavy fin-and-tube heat exchangers are one of the most popular fin surfaces applicable to both condenser and evaporator. In the evaporators, the temperatures of outside tube and fin surfaces are generally below the dew point temperature, leading to simultaneous heat and mass transfer outside tube and fin surfaces. In general, the complexity of the moist air flowing across the wavy fin-and-tube heat exchangers in dehumidifying conditions makes the theoretical simulations very difficult.

The air side performance of wavy fin-and-tube heat exchangers had been studied by many researchers (Beecher and Fagan [1], Yan and Sheen [2], Wang et al. [3–6], Lin

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0017-9310/$ - see front matter Published by Elsevier Ltd.
doi:10.1016/j.ijheatmasstransfer.2007.11.049

ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

Renewable and Sustainable Energy Reviews
12 (2008) 824–838

RENEWABLE
& SUSTAINABLE
ENERGY REVIEWS

www.elsevier.com/locate/rser

A review of two-phase gas–liquid adiabatic flow characteristics in micro-channels

Sira Saisorn[a], Somchai Wongwises[b,*]

[a]*Energy Division, The Joint Graduate School of Energy and Environment, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*
[b]*Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

Received 7 July 2006; accepted 13 October 2006

Abstract

A literature review of recent research on two-phase flow in micro-channels is provided in this article. Researches on the micro-hydrodynamics concerned with two-phase gas–liquid adiabatic flow characteristics in both circular and non-circular micro-channels are discussed. This review aims to survey and identify new findings obtained from this attractive area, which may contribute to optimum design and process control of high performance miniature devices comprising extremely small channels. The results obtained from a number of previous studies show that the flow behaviors in the micro-channels deviate significantly from those in ordinarily sized channels. A clear understanding of the flow behaviors encountered is necessary to answer the fundamental questions related to the two-phase flow phenomena. Similar to the ordinarily sized channels, it is expected that the flow pattern, pressure drop and void fraction will influence the two-phase pressure drop, holdup, system stability, exchange rates of momentum, heat and mass during the phase-change heat transfer processes. It is obvious that publications available on micro-channels are relatively small compared to those for ordinarily sized channels and further systematic studies of two-phase gas–liquid adiabatic flow in micro-channels are required.

Keywords: Two-phase flow; Micro-channel; Flow characteristic

*Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

1364-0321/$ - see front matter © 2006 Elsevier Ltd. All rights reserved.
doi:10.1016/j.rser.2006.10.012

Available online at www.sciencedirect.com

ScienceDirect

Experimental Thermal and Fluid Science 32 (2008) 748–760

Experimental Thermal and Fluid Science

www.elsevier.com/locate/etfs

Flow pattern, void fraction and pressure drop of two-phase air–water flow in a horizontal circular micro-channel

Sira Saisorn [a], Somchai Wongwises [b,*]

[a] *Energy Division, The Joint Graduate School of Energy and Environment (JGSEE), King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

[b] *Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

Received 4 June 2007; received in revised form 11 September 2007; accepted 19 September 2007

Abstract

Adiabatic two-phase air–water flow characteristics, including the two-phase flow pattern as well as the void fraction and two-phase frictional pressure drop, in a circular micro-channel are experimentally studied. A fused silica channel, 320 mm long, with an inside diameter of 0.53 mm is used as the test section. The test runs are done at superficial velocity of gas and liquid ranging between 0.37–16 and 0.005–3.04 m/s, respectively. The flow pattern map is developed from the observed flow patterns i.e. slug flow, throat-annular flow, churn flow and annular-rivulet flow. The flow pattern map is compared with those of other researchers obtained from different working fluids. The present single-phase experiments also show that there are no significant differences in the data from the use of air or nitrogen gas, and water or de-ionized water. The void fraction data obtained by image analysis tends to correspond with the homogeneous flow model. The two-phase pressure drops are also used to calculate the frictional multiplier. The multiplier data show a dependence on flow pattern as well as mass flux. A new correlation of two-phase frictional multiplier is also proposed for practical application.

Keywords: Two-phase flow; Micro-channel; Flow pattern; Void fraction; Pressure drop

1. Introduction

Over the years, studies on two-phase flow and heat transfer characteristics in micro-channel flow passages have become more necessary due to the rapid development of micro-scale devices used for several engineering applications including medical devices, high heat-flux compact heat exchangers, and cooling systems of various types of equipment such as high performance micro-electronics, supercomputers, high-powered lasers.

In addition to the attractive phenomena caused by the significant role of the surface effects, this emerging field enables us to develop powerful miniature devices for operation in a domain which seemed to be unfeasible in the past. However, it is obvious that a comprehensive understanding is still lacking on the trends and parameters dominating the flow behavior in mini- and micro-channels. Thus, fundamental heat transfer and flow characteristics problems encountered in the development and processing of micro-electronic-mechanical-systems (MEMS) have become a significant challenge for the design and control of these micro-systems. According to the requirement of new solutions for dissipating heat and keeping uniform temperature distributions in modern devices, a clear understanding of the major mechanisms affecting micro-scale heat transfer is essential for optimum design and process control of micro-systems.

The two-phase flow and heat transfer characteristics in small channels such as micro-channels and mini-channels are likely to be strongly dependent on the surface tension effects in addition to viscosity and inertia forces resulting

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0894-1777/$ - see front matter © 2007 Elsevier Inc. All rights reserved.
doi:10.1016/j.expthermflusci.2007.09.005

Meteorol Atmos Phys (2008) 102:113–121
DOI 10.1007/s00703-008-0013-4

ORIGINAL PAPER

Numerical simulation of terrain-induced mesoscale circulation in the Chiang Mai area, Thailand

Surachai Sathitkunarat · Prungchan Wongwises · Rudklao Pan-Aram · Meigen Zhang

Received: 13 October 2006 / Accepted: 16 October 2008 / Published online: 5 November 2008

Abstract The regional atmospheric modeling system (RAMS) was applied to Chiang Mai province, a mountainous area in Thailand, to study terrain-induced mesoscale circulations. Eight cases in wet and dry seasons under different weather conditions were analyzed to show thermal and dynamic impacts on local circulations. This is the first study of RAMS in Thailand especially investigating the effect of mountainous area on the simulated meteorological data. Analysis of model results indicates that the model can reproduce major features of local circulation and diurnal variations in temperatures. For evaluating the model performance, model results were compared with observed wind speed, wind direction, and temperature monitored at a meteorological tower. Comparison shows that the modeled values are generally in good agreement with observations and that the model captured many of the observed features.

S. Sathitkunarat (✉) · P. Wongwises
The Joint Graduate School of Energy and Environment,
King Mongkut's University of Technology Thonburi,
Bangkok 10140, Thailand
e-mail: surachai_james@yahoo.com; surachai@nstda.or.th

R. Pan-Aram
Electricity Generating Authority of Thailand,
Nonthaburi 11130, Thailand

M. Zhang
State Key Laboratory of Atmospheric Boundary Layer Physics and Atmospheric Chemistry, Institute of Atmospheric Physics, Chinese Academy of Sciences, 100029 Beijing,
People's Republic of China

1 Introduction

In recent years Chiang Mai, a large city in northern Thailand, has suffered from air pollution problems; particulate matter with an aerodynamic diameter of less than 10 μm (PM_{10}) is the most critical pollutant. Burning of agricultural residues in adjacent areas, open burning of rubbish, and traffic exhausts are the main source of PM_{10} in this area. Observations show that the PM_{10} concentration exceeded the standard for 10 and 32 days in 2001 and 2002, respectively. On 11 February 2005, the PM_{10} concentration was recorded as high as 206.9 μg m^{-3} (the Thai standard for PM_{10} for 24 h is 120 μg m^{-3}). Besides the mentioned emission sources, Chiang Mai is a mountainous area where the urban area is located at the bottom of a valley between two mountain ranges. Urban pollution is therefore trapped by surrounding mountains except when the wind is approximately oriented along the valley axis. Moreover, pollution problems are, generally, associated with low-wind conditions, when significant stratification develops an inversion layer, reducing exchange to meteorological scales (Anquetin et al. 1999).

The topographic complexity and the variety of physicogeographic characteristics in the basin are expected to induce local circulation, such as anabatic/katabatic flows, valley winds, lake effects in mountainous terrain, and cold pool formation in basins (Triantafyllou and Kassomenos 2002). Radiative cooling of hill slopes on clear calm nights, and subsequent cooling of the air in contact with them by conduction and convection, results in cold air drainage that follows the terrain to its lowest sections, where it may become slow moving or stagnant. Knowledge of these atmospheric boundary layer flows is important for predicting dispersion of air pollution from near-surface sources in complex terrain (Arya 1999). Besides knowledge about

ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

International Journal of Thermal Sciences 47 (2008) 1473–1481

International Journal of Thermal Sciences

www.elsevier.com/locate/ijts

Optimal configuration of cross flow plate finned tube condenser based on the second law of thermodynamics

Patcharin Saechan *, Somchai Wongwises

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Received 2 July 2007; received in revised form 17 December 2007; accepted 17 December 2007

Available online 28 January 2008

Abstract

This paper presents a mathematical model for determining optimal configuration of plate finned tube condensers widely used in air conditioning system. The condenser investigated in this work is a cross-flow air-refrigerant type with aluminium plain fin geometry. Refrigerant HFC 134a flows inside while air flows outside the copper tubes. Minimum entropy change of air and refrigerant is used as the criterion for analysis. This entropy change resulting from losses due to heat transfer and pressure drop can be represented in a form of entropy generation number, N_S. The proposed mathematical model is divided into three regions: superheated, two-phase and sub-cooled regions. Effects of heat transfer and pressure drop of air side and refrigerant side on entropy generation rate are simultaneously analyzed through the implicit method. Results from the mathematical model show that entropy generation number decreases with increasing fin pitch and decreasing number of rows and tube diameter. In addition, this work also presents an optimal configuration of condenser. The optimal tube diameter and the number of fin per inch are 6.35–12.7 mm ($\frac{1}{4}$–$\frac{1}{2}$ inch) and 11–12, respectively.

Keywords: Cross flow plate finned tube condenser; Second law of thermodynamics; Entropy generation number

1. Introduction

Heat exchangers are widely used for heat transfer between hot and cool fluid in air-conditioning system, industrial plants and electrical plants. A well-designed, highly efficient heat exchanger can help in saving a large amount of energy. At present, heat exchangers are studied in many aspects such as characteristics of heat transfer, efficiency, pressure drop during heat exchange, various configurations that enhance heat transfer, or even economic analysis to find an optimal size, price and weight for investment. One of the common ways in studying heat exchanger is to use the second law of thermodynamic in analyzing heat exchanger. This method is increasingly popular because it takes into account the usable potential or the quality of energy which is not conserved but destroyed by irreversibility and which can be transferred into or out of the system—the situation similar to actual system.

The analysis of heat exchanger with the second law of thermodynamics has extensively been studied. Bejan [1], who studied shell-and-tube counter flow heat changers for gas-to-gas, found that the irreversibility or loss occurred in heat exchangers was caused by pressure drop from friction and heat transfer due to differences in temperature. He proposed a design of minimum irreversibility in a form of entropy generation number (N_s). Later, Ogulata et al. [2] conducted a similar study to that of Bejan but they used plate type counter flow heat exchangers without mixing two types of fluids which had the same heat capacity. Saboya et al. [3] also studied in a similar manner, but with an additional focus on configurations. They used counter flow, parallel flow and cross flow heat exchangers in their analysis without considering irreversibility from pressure drop. On the other hand, Dejong et al. [4] studied capacity of condensers, heat exchangers used in vapor-compression air conditioning systems, with a sole focus on the air side. However, these studies were all dealt only with configurations and

* Corresponding author.
E-mail addresses: psyui@hotmail.com (P. Saechan), somchai.won@kmutt.ac.th (S. Wongwises).

1290-0729/$ – see front matter © 2007 Published by Elsevier Masson SAS.
doi:10.1016/j.ijthermalsci.2007.12.009

ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

International Communications in Heat and Mass Transfer 35 (2008) 458–465

International Communications in HEAT and MASS TRANSFER

www.elsevier.com/locate/ichmt

Onset of flooding in a small diameter tube ☆

Krittapas Saenmart, Panachat Cheowuttikul, Adirek Suriyawong, Somchai Wongwises *

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

Available online 20 November 2007

Abstract

The main concern of this work is to obtain and investigate the experimental data of the onset of flooding (OSF) for air and water in a small diameter tube. Flooding experiments are carried out in a tube having an inner diameter of 8.75 mm and the length of 1 m at inclination angles of 0, 30, 45, 60° from the vertical. The curves of the OSF are built and shown as a function of the dimensionless superficial velocity. The effects of the inclination angles of the small diameter tubes on the OSF are discussed. The present results are compared with existing limited data reported in literature.

Keywords: Onset of flooding; Countercurrent flow limitation; Small diameter tube

1. Introduction

Onset of flooding (OSF) or countercurrent flow limitation (CCFL) refers to the limiting condition at which the flow rates of both the gas and the liquid phase cannot be increased further. A further increase will cause the liquid to be carried by the gas. This is a subject of engineering interest, particularly in the design of two-phase heat and mass transfer processes.

The OSF has been extensively studied by a large number of researchers in the past decades, both experimentally and analytically. In recent years, there is a substantial interest in two-phase gas-liquid countercurrent flow in small channels, mostly for effectively designing or improving the performance of various kinds of compact heat exchangers. Although the information on that issue is needed, the OSF in a small-gap channel has received comparatively very little attention in the literature. Some of the earliest work was performed by Vlachos et al. [1]. They reported new air-water flooding data obtained from vertical narrow rectangular channels. Two rectangular test sections: 150 mm × 5 mm, 150 mm × 10 mm at an equal length of 700 mm, were used in the experiments. The experimental results showed that the gap size had a significant effect on the onset of flooding.

Fiedler et al. [2] investigated the effect of inclination angle on the OSF and on the heat transfer coefficient during reflux condensation in a pipe having the diameter of 7 mm, and the length of 500 mm. Refrigerant R134a and steam were used as working fluid for the flooding experiments. Only R134a was used as working fluid for the heat transfer

☆ Communicated by W.J. Minkowycz.

* Corresponding author.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0735-1933/$ - see front matter © 2007 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2007.09.001

International Communications in Heat and Mass Transfer 35 (2008) 833–838

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

An inspection of viscosity model for homogeneous two-phase flow pressure drop prediction in a horizontal circular micro-channel☆

Sira Saisorn [a], Somchai Wongwises [b,*]

[a] Energy Division, The Joint Graduate School of Energy and Environment (JGSEE), King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand
[b] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Available online 29 April 2008

Keywords:
Two-phase flow
Micro-channel
Flow characteristic

ABSTRACT

This study analyzes several well-known two-phase viscosity models in order to address the appropriate correlations among them for application to micro-channel. Pressure drop data is obtained from adiabatic two-phase air–water flow experiments. A fused silica channel, 320 mm long, with an inside diameter of 0.53 mm is used as the test section. The measured data is compared with the homogeneous flow predictions calculated using the existing viscosity models and detailed comparisons are discussed.

1. Introduction

Over the years, studies on two-phase flow and heat transfer characteristics in micro-channel flow passages have become more necessary due to the rapid development of micro-scale devices used for several engineering applications including, medical devices, high heat-flux compact heat exchangers, and cooling systems of various types of equipment, such as high performance micro-electronics, supercomputers, and high-powered lasers.

In addition to the attractive phenomena caused by the significant role of the surface effects, this emerging field enables us to develop powerful miniature devices for operation in a domain which seemed to be unfeasible in the past. However, it is obvious that a comprehensive understanding is still lacking on the trends and parameters dominating the flow behavior in mini- and micro-channels. Thus, fundamental heat transfer and flow characteristic problems encountered in the development and processing of micro-electronic-mechanical-systems (MEMS) have become a significant challenge in the design and control of these micro-systems. According to the requirement of new solutions for dissipating heat and keeping uniform temperature distributions in modern devices, a clear understanding of the major mechanisms affecting micro-scale heat transfer is essential for optimum design and process control of micro-systems.

In the past decade, there have been a relatively small number of publications available in the area of mini-channels and micro-channels compared to those done for ordinarily sized channels. In comparison to flow characteristics in ordinarily sized channels, the hydrodynamic and transport phenomena in mini-channels and micro-channels are less understood due to the effects of the limited confined space.

It is not possible to understand the two-phase flow phenomena without a clear understanding of the flow patterns encountered. It is expected that the flow patterns will influence the two-phase pressure drop, holdup, system stability, exchange rates of momentum, and the heat and mass during the phase-change heat transfer processes. The ability to accurately predict the type of flow is necessary before relevant calculation techniques can be developed. The recent works associated with two-phase flow in small channels are outlined in the following paragraphs.

Adiabatic two-phase air–de-ionized water (DI water) flow characteristics in micro-channels with hydraulic diameter ranging from 1.1 to 1.5 mm were experimentally studied by Triplett et al. [1,2]. The flow patterns observed were bubbly, slug, churn, slug-annular and annular. The measured void fraction and two-phase pressure drop in the relevant flow regimes were also investigated.

Chen et al. [3] experimentally studied the characteristics of adiabatic two-phase flow of nitrogen–water in circular micro-channels with two different diameters, 1.0 and 1.5 mm. Four common flow patterns including bubbly flow, slug flow, churn flow and annular flow were observed and the so-called bubble-train slug flow, previously reported in a large channel under micro-gravity effects (Zhao and Rezkallah [4]), was also observed and discussed in this work.

Serizawa et al. [5] investigated the visualization of the two-phase flow pattern in circular micro-channels. The flowing mixture of air and

☆ Communicated by W.J. Minkowycz.

* Corresponding author.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0735-1933/$ – see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2008.03.014

International Communications in Heat and Mass Transfer 36 (2009) 172–179

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

Effect of void fraction models on the film thickness of R134a during downward condensation in a vertical smooth tube ☆

A.S. Dalkilic [a,*], S. Laohalertdecha [b], S. Wongwises [b,*]

[a] Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University, Yildiz, Istanbul 34349, Turkey

[b] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Available online 16 December 2008

Keywords:
Heat transfer
Void fraction
Film thickness
Condensation
Downward flow
R134a

ABSTRACT

In the present study, the void fraction and film thickness of pure R-134a flowing downwards in a vertical condenser tube are indirectly determined using relevant measured data together with an annular flow model and various void fraction models reported in the open literature. The vertical test section is a countercurrent flow double tube heat exchanger with refrigerant flowing down in the inner tube and cooling water flowing upward in the annulus. The inner tube is made from smooth copper tubing of 9.52 mm outer diameter with a length of 0.5 m. The experimental runs are carried out at average saturated condensing temperatures of 40 and 50 °C, and mass velocities are around 456 kg m^{-2} s^{-1}, over the vapour quality range 0.82–0.93, while the heat fluxes are between 45.60 and 50.90 kW m^{-2}. Analysis based on simple void fraction models of the annular flow pattern are presented for forced convection condensation of pure R134a, taking into account the effect of the different saturation temperatures at high mass flux conditions. The comparisons of calculated film thickness show that the void fraction models of Spedding and Chen, and Chisholm and Armand are the most accurate ones with the experimental data due to their low deviation with Whalley's annular flow model over 35 void fraction models presented in this paper.

1. Introduction

Heat exchangers are devices that are commonly used to transfer heat between two or more fluids of different temperatures. They are used in a wide variety of applications, e.g. refrigeration and air-conditioning systems, power engineering and other thermal processing plants.

One of the most important contributors to the depletion of the ozone layer is hydrochlorofluorocarbon refrigerants. Since the depletion of the Earth's ozone layer has been discovered, many corporations have been forced to find alternative chemicals to CFCs. Refrigerant HFC-134a is receiving support from the refrigerant and air-conditioning industry as a potential replacement for CFC-12, because the thermophysical properties of HFC-134a are very similar to those of CFC-12.

A large number of empirical and semi-empirical models have been developed over the last sixty years. The studies consider experimental endeavours to measure the void fraction of fluids, theoretical endeavours to model a two-phase phenomenon and a combination of them. According to the slip flow model, it is assumed that the two phases are separated with different uniform velocities, unlike the homogeneous model. The void fraction, defined as the cross-sectional area occupied by the vapour in relation to the area of the flow channel, is an important parameter and is always used to determine the two-phase heat transfer coefficient and pressure drop and flow pattern transitions.

In the technical literature, numerous studies have been carried out on the modelling of the void fraction and can be divided into several groups: general void fraction models and correlations [1–7], $K\varepsilon_H$ parameter-based void fraction models and correlations [8–10], flow regime-based void fraction models and correlations [11–13], Lockhart–Martinelli parameter-based void fraction models and correlations [14–22] and slip ratio void fraction models and correlations [23–34]. A brief and comprehensive overview of these works is presented in Table 1.

Annular flow conditions along the tube length include convective condensation, which occurs for many applications inside tubes. The annular two-phase flow is one of the most important flow regimes and is characterized by a phase interface separating a thin liquid film from the gas flow in the core region. The two-phase annular flow occurs widely in film heating and cooling processes, particularly in power generation and especially in nuclear reactors. This flow regime has received the most attention, both analytically and experimentally, because of its practical importance and the relative ease with which analytical treatment may be applied. Nusselt in 1916 [35] suggested the first theoretical solution

☆ Communicated by W.J. Minkowycz.

* Corresponding authors.
E-mail addresses: dalkilic@yildiz.edu.tr (A.S. Dalkilic), somchai.won@kmutt.ac.th (S. Wongwises).

0735-1933/$ – see front matter © 2008 Published by Elsevier Ltd.
doi:10.1016/j.icheatmasstransfer.2008.10.015

International Journal of Heat and Mass Transfer 52 (2009) 142–150

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Experimental investigation of convective heat transfer coefficient during downward laminar flow condensation of R134a in a vertical smooth tube

A.S. Dalkilic [a,*], S. Yildiz [a], S. Wongwises [b,*]

[a] Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University, Yildiz, Istanbul 34349, Turkey
[b] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

A R T I C L E I N F O

Article history:
Received 12 February 2008
Received in revised form 31 May 2008
Available online 15 August 2008

Keywords:
Condensation
Heat transfer coefficient
Downward flow
Laminar flow

A B S T R A C T

This paper presents an experimental investigation of laminar film condensation of R134a in a vertical smooth tube having an inner diameter of 7–8.1 mm and a length of 500 mm. Condensation experiments were performed at mass fluxes of 29 and 263 kg m^{-2} s^{-1}. The pressures were between 0.77 and 0.1 MPa. The heat transfer coefficient, film thickness and condensation rate during downward condensing film were determined. The results show that an interfacial shear effect is significant for the laminar condensation heat transfer of R134a under the given conditions. A new correlation for the condensation heat transfer coefficient is proposed for practical applications.

1. Introduction

Annular flow along a tube length, including convective condensation, occurs in many real applications. It is one of the most important flow regimes and is characterised by a phase interface separating a thin liquid film from the gas flow in the core region. Because of its practical importance and the relative ease with which analytical treatment may be applied, this flow regime has received the most attention both analytically and experimentally.

Nusselt [1] proposed the first theoretical solution for predicting heat transfer coefficients. He assumed a linear temperature profile through a laminar film flowing downwards without entrainment on a vertical plate. Waves and an interfacial shear effect between the phases were not considered. Under these conditions, it is possible for the Nusselt-type analysis to be used for convective condensation in round tubes.

Rohsenov [2] and Dukler [3] developed a model to predict momentum transfer for turbulent film flow. In addition to this, Levich [4] and Blangetti et al. [5] used a model to estimate the local heat transfer coefficients of the film. An empirical relationship between the Fanning friction factor and vapour Reynolds number for an annular flow regime was proposed by Bergelin et al. [6]. They studied the pressure drop of air–water and several organic vapours for the downward turbulent flow through a vertical 25.4 mm i.d. tube. Their diagram was used by several researchers such as Blangetti et al. [5] and Maheshwari et al. [7] for various refrigerants. The diagram takes account of the effect of mass transfer by including a correction factor developed by Bird et al. [8], considering the effect of suction in condensation. Krebs and Schlunder [9] investigated mass transfer coefficients in the turbulent gas and film flow of a vertical condenser tube in the presence of non-condensing gases. Kuhn et al. [10,11] investigated local condensation heat transfer in the presence of non-condensable gases inside a vertical tube connecting a passive containment cooling system (PCCS). They studied the degradation factor method, diffusion layer theory and mass transfer conductance model. Recently, Maheshwari et al. [7] followed the same path and adopted the model, using the analogy between heat and mass transfer, to the PCCS in nuclear reactors. They considered non-condensable gas with a wide range of Reynolds numbers in their investigation. The film waviness effect on the gas/vapour boundary layer, the suction effect due to condensation, and the developing flow and property variation of the gas were also considered in their study. Local film heat transfer coefficients were multiplied by a factor of 1.28 for the wave effect between the phases in the high mass flux region. In addition to this, Oh and Revankar [12] studied the vertical passive condenser (PCCS) for complete condensation in nuclear reactors. They used a similar analysis model as [2–11]. Their model includes a modified Nusselt theory with a Mc Adams correction factor of 1.2 [13], the Blangetti model [5] and interfacial shear from the Couette flow analysis [14] for small film Reynolds numbers and small interfacial shear conditions.

* Corresponding authors. Tel.: +902122597070; fax: +902122616659 (A.S. Dalkilic); tel.: +6624709115; fax: +6624709111 (S. Wongwises).
E-mail addresses: dalkilic@yildiz.edu.tr (A.S. Dalkilic), Somchai.won@kmutt.ac.th (S. Wongwises).

0017-9310/$ - see front matter © 2008 Published by Elsevier Ltd.
doi:10.1016/j.ijheatmasstransfer.2008.05.035

International Communications in Heat and Mass Transfer 36 (2009) 1036–1043

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

Experimental investigation of heat transfer coefficient of R134a during condensation in vertical downward flow at high mass flux in a smooth tube ☆

A.S. Dalkilic [a,*], S. Laohalertdecha [b], S. Wongwises [b,*]

[a] Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University, Yildiz, Besiktas, Istanbul 34349, Turkey
[b] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

A R T I C L E I N F O

Available online 23 July 2009

Keywords:
Condensation
Heat transfer coefficient
Downward flow
R134a
Refrigerant

A B S T R A C T

The two-phase heat transfer coefficients of pure HFC-134a condensing inside a smooth tube-in-tube heat exchanger are experimentally investigated. The test section is a 0.5 m long double tube with refrigerant flowing in the inner tube and cooling water flowing in the annulus. The inner tube is constructed from smooth copper tubing of 9.52 mm outer diameter and 8.1 mm inner diameter. The test runs are performed at average saturation condensing temperatures between 40–50 °C. The mass fluxes are between 260 and 515 kg $m^{-2}s^{-1}$ and the heat fluxes are between 11.3 and 55.3 kW m^{-2}. The quality of the refrigerant in the test section is calculated using the temperature and pressure obtained from the experiment. The average heat transfer coefficient of the refrigerant is determined by applying an energy balance based on the energy transferred from the test section. The effects of heat flux, mass flux and condensation temperature on the heat transfer coefficients are also discussed. Eleven well-known correlations for annular flow are compared to each other using a large amount of data obtained from various experimental conditions. A new correlation for the condensation heat transfer coefficient is proposed for practical applications.

1. Introduction

Heat exchangers are devices that are commonly used to transfer heat between two or more fluids of different temperatures. They are used in a wide variety of applications, e.g., refrigeration and air-conditioning systems, power engineering and other thermal processing plants. In a refrigeration equipment, condenser cools and condenses the refrigerant vapor discharged from a compressor by means of a secondary heat transfer fluid such as air and fluid. The heat transfer failure occurs in the condenser due to design error, effectiveness of condenser should be examined and improved carefully.

Since the depletion of the earth's ozone layer has been discovered, many corporations have been forced to find alternative chemicals to CFCs. Because the thermo-physical properties of HFC-134a are very similar to those of CFC-12. Refrigerant HFC-134a has received intensive support from the refrigerant and air-conditioning industry as a potential replacement for CFC-12. However, even though the difference in properties between the two refrigerants is small, it may result in significant differences in the overall system performance. Therefore, the properties of HFC-134a should be studied in detail before it is applied.

Heat transfer and pressure drop characteristics of refrigerants have been studied by a large number of researchers, both experimentally and analytically, mostly in a horizontal straight tube. The study of the heat transfer and pressure drop of CFCs inside a small diameter vertical tube for downward condensation has received comparatively little attention in the literature. In addition to this, there are few studies on condensation of R134a during downward flow in vertical micro-fin tubes. Briggs et al. [1,2] have used large diameter tubes of approximately 20.8 mm with CFC113. Shah correlation [3] has been compared by researchers commonly for turbulent condensation conditions and is considered to be the most predictive condensation model for the annular flow regime in a tube. Up to 40 mm i.d. smooth horizontal, inclined and vertical tubes were used in his experiments. Oh and Revankar [4] performed theoretical calculations using modified Nusselt theory [5] for the PCCS condenser, which has 47.5 mm i.d. and 1.8 m long vertical tube and validated by experimental study. Maheshwari et al. [6] studied on the downward condensation presence of non-condensable gas in a 42.77 mm i.d. vertical tube which simulates PCCS condensers used in water-cooled reactors preparing a computer code to predict heat transfer considering mass transfer along the tube length.

Annular flow conditions along the tube length include convective condensation which occurs for many applications inside tubes. Annular two-phase flow is one of the most important flow regimes and is characterized by a phase interface separating a thin liquid film from the gas flow in the core region. Two-phase annular flow occurs widely in film heating and cooling processes, particularly in power

☆ Communicated by W.J. Minkowycz.

* Corresponding authors.
E-mail addresses: dalkilic@yildiz.edu.tr (A.S. Dalkilic), somchai.won@kmutt.ac.th (S. Wongwises).

0735-1933/$ – see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2009.06.017

International Journal of Heat and Mass Transfer 52 (2009) 3409–3426

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Review

Intensive literature review of condensation inside smooth and enhanced tubes

A.S. Dalkilic [a,*], S. Wongwises [b]

[a] Heat and Thermodynamics Division, Department of Mechanical Engineering, Yildiz Technical University, Yildiz, Istanbul 34349, Turkey
[b] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Article history:
Received 21 November 2008
Received in revised form 10 January 2009
Accepted 10 January 2009
Available online 18 April 2009

Keywords:
Condensation heat transfer
Condensation pressure drop
Heat transfer enhancement
Review
Smooth tube
Enhanced tube

ABSTRACT

This paper presents a detailed review of research work on in-tube condensation in the literature due to its significance in refrigeration, air conditioning and heat pump applications. The heat transfer performance of heat exchangers can be improved by heat transfer enhancement techniques, such as active and passive techniques. Passive techniques requiring fluid additives or special surface geometries are mentioned in depth, by comparison with active techniques requiring external forces, e.g. electrical field, acoustic or surface vibration, etc., in the paper due to their common usage in condensation applications. In addition, the importance of usage of hydrocarbons instead of fluorocarbons is emphasised. This paper can not only be used as the starting point for the researcher interested in in-tube condensation process, but it also includes new investigations on condensation inside tubes.

Contents

* Corresponding author. Tel.: +90 212 2597070; fax: +90 212 2616659.
E-mail address: dalkilic@yildiz.edu.tr (A.S. Dalkilic).

0017-9310/$ - see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ijheatmasstransfer.2009.01.011

Springer

Journal of Mechanical Science and Technology

Journal of Mechanical Science and Technology 23 (2009) 1620~1627

www.springerlink.com/content/1738-494x
DOI 10.1007/s12206-008-1106-1

Predicting performance of radial flow type impeller of centrifugal pump using CFD†

Suthep Kaewnai, Manuspong Chamaoot and Somchai Wongwises*

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

(Manuscript Received August 31, 2006; Revised April 20, 2008; Accepted November 17, 2008)

Abstract

The main objective of this work is to use the computational fluid dynamics (CFD) technique in analyzing and predicting the performance of a radial flow-type impeller of centrifugal pump. The impeller analyzed is at the following design condition: flow rate of 528 m^3/hr; speed of 1450 rpm; and head of 20 m or specific speed (N_s) of 3033 1/min in US-Units. The first stage involves the mesh generation and refinement on domain of the designed impeller. The second stage deals with the identification of initial and boundary conditions of the mesh-equipped module. In the final stage, various results are calculated and analyzed for factors affecting impeller performance. The results indicate that the total head rise of the impeller at the design point is approximately 19.8 m. The loss coefficient of the impeller is 0.015 when $0.6 < Q/Q_{design} < 1.2$. Maximum hydraulic efficiency of impeller is 0.98 at $Q/Q_{design} = 0.7$. Based on the comparison of the theoretical head coefficient and static pressure rise coefficient between simulation results and experimental data, from previous work reported in the literature [Guelich, Kreiselpumpen, Springer, Berlin, 2004], it is possible to use this method to simulate the performance of a radial-flow type impeller of a centrifugal pump.

Keywords: Centrifugal pump; CFD; Impeller; Radial flow

1. Introduction

The use of CFD plays an important role in fluid mechanics. Due to the progress of numerical methods and computer capability, the impeller design for centrifugal pumps nowadays has been analyzed by using 3D-Navier stokes program or CFD software to predict impeller performance in advance. However, both the prediction and design cannot be done easily without sufficient information and experience. If the result from the CFD is much different from the designed value, the impeller could be redesigned before being put into real production.

Over the years, the application of CFD in fluid mechanics and turbomachinery has been studied by a number of researchers [1-8]. However there are few studies applying the CFD for predictions involving a radial flow type impeller. To the best of our knowledge to date, there have been only two works carried out by [9] and [10], in dealing with this issue.

[9] analyzed about 30 pump impellers having specific speeds between 12 to 160 (metric units) using a commercial 3D-Navier Stokes program for a standard k-ε turbulence model. The head and efficiency obtained from the software were compared to those obtained from the measurement. A set of rather stringent rules for the CFD calculations was developed in order to reduce the effect of grid and calculation parameters on the results.

[10] performed a 3D-CFD simulation of the impeller and volute of a centrifugal pump having a specific speed of 32 (metric units) and an outside impeller radius of 204.2 mm, by using CFX codes. A 3D flow simulation for the impeller with a structured grid was

† This paper was recommended for publication in revised form by Associate Editor Seungbae Lee

*Corresponding author. Tel.: +662 470 9115, Fax.: +662 470 9111
E-mail address: somchai.won@kmutt.ac.th

INTERNATIONAL JOURNAL OF REFRIGERATION 32 (2009) 124–137

available at www.sciencedirect.com

ScienceDirect

journal homepage: www.elsevier.com/locate/ijrefrig

Experimental investigation of evaporation heat transfer coefficient and pressure drop of R-410A in a multiport mini-channel

Jatuporn Kaew-On[a,b], *Somchai Wongwises*[b,*]

[a]*The joint Graduate School of Energy and Environment, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

[b]*Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand*

ARTICLE INFO

Article history:
Received 20 March 2008
Received in revised form
17 June 2008
Accepted 29 June 2008
Published online 9 July 2008

Keywords:
Heat exchanger
Microchannel
Experiment
Heat transfer
Evaporation
Pressure drop
R-410A

ABSTRACT

The evaporation heat transfer coefficient and pressure drop of R-410A flowing through a horizontal aluminium rectangular multiport mini-channel having 3.48 mm hydraulic diameter are experimentally investigated. The test runs are performed at mass flux ranging between 200 and 400 kg/m^2 s. The heat fluxes are between 5 and 14.25 kW/m^2 and the saturation temperatures range between 10 and 30 °C. The pressure drop across the test section is directly measured by a differential pressure transducer. The effects of the imposed wall heat flux, mass flux, vapour quality, and saturation temperature on the evaporation heat transfer and pressure drop are also discussed. The results from the present experiment are compared with those obtained from the existing correlation. New correlations for the evaporation heat transfer coefficient and pressure drop of R-410A flowing through a multiport mini-channel are proposed for practical applications.

Coefficient de chaleur et chute de pression lors de l'évaporation de R-410A dans un minicanal à plusieurs orifices : étude expérimentale

Mots clés : Échangeur de chaleur ; Microcanal ; Expérimentation ; Transfert de chaleur ; Évaporation ; Chute de pression ; R-410A

* *Corresponding author.* Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0140-7007/$ – see front matter © 2008 Elsevier Ltd and IIR. All rights reserved.
doi:10.1016/j.ijrefrig.2008.06.010

International Journal of Heat and Mass Transfer 52 (2009) 2059–2067

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Heat transfer enhancement and pressure drop characteristics of TiO_2–water nanofluid in a double-tube counter flow heat exchanger

Weerapun Duangthongsuk, Somchai Wongwises *

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Article history:
Received 12 December 2007
Received in revised form 25 October 2008
Available online 26 December 2008

Keywords:
Heat transfer coefficient
Pressure drop
Friction factor
Nanofluid
Heat exchanger

ABSTRACT

This article reports an experimental study on the forced convective heat transfer and flow characteristics of a nanofluid consisting of water and 0.2 vol.% TiO_2 nanoparticles. The heat transfer coefficient and friction factor of the TiO_2–water nanofluid flowing in a horizontal double-tube counter flow heat exchanger under turbulent flow conditions are investigated. The Degussa P25 TiO_2 nanoparticles of about 21 nm diameter are used in the present study. The results show that the convective heat transfer coefficient of nanofluid is slightly higher than that of the base liquid by about 6–11%. The heat transfer coefficient of the nanofluid increases with an increase in the mass flow rate of the hot water and nanofluid, and increases with a decrease in the nanofluid temperature, and the temperature of the heating fluid has no significant effect on the heat transfer coefficient of the nanofluid. It is also seen that the Gnielinski equation failed to predict the heat transfer coefficient of the nanofluid. Finally, the use of the nanofluid has a little penalty in pressure drop.

1. Introduction

New energy-efficient heat transfer equipment stands at the point of a miniature increase in heat flux on one hand and an astronomical one on the other. Heat transfer fluids such as water, minerals, oil and ethylene glycol play a vital role in many industry processes, including power generation, chemical processes, heating and cooling processes, transportation, microelectronics and other micro-sized applications. The poor heat transfer properties of these fluids compared with those of most solids are the primary hindrance of high compactness and the effectiveness of the heat exchanger. The key idea is to exploit the very high thermal conductivities of solid particles that can be several hundreds of times greater than all of the conventional fluids combined. As a result, an important need still exists to develop new strategies in order to improve the effective heat transfer behaviours of conventional fluids. An innovative way of improving the thermal conductivities of common fluids is to suspend small solid particles in the fluids. Various types of particles, such as metallic, non-metallic and polymeric, can be added into fluids to form slurries. However, slurries with suspended particles of the order of micrometre or even millimetre may cause some severe problems. The abrasive action of the particles causes clogging of flow channels, eroding of pipelines and a reduction in their momentum transfer increase in pressure drop in practical applications [1]. Furthermore, they suffer from stability and rheological problems. In particular, the particles tend to settle quickly out of the suspension. Although the slurries have higher thermal conductivities than those of conventional heat transfer fluids, they are still not appropriate to use as heat transfer fluids for practical applications.

A novel approach to engineering fluids with better heat transfer properties is based on the rapidly emerging field of nanotechnology. The use of particles of nanometre dimensions first materialized in a series of studies conducted at the Argonne National Laboratory about a decade ago, and Choi [2] was probably the first to call the fluids with suspended particles of nanometre dimensions "nanofluids", which has gained popularity.

Compared with millimetre or micrometre-sized particle suspensions, nanofluids possess better long-term stability and rheological properties, and can have dramatically increased thermal conductivities. Over the past 10 years, many researchers have studied the heat transfer characteristics of the various nanofluids. Focusing on the forced convective heat transfer experimentally, several existing published articles which involve the use of nanofluids are discussed in the following sections.

Pak and Cho [3] studied the heat transfer performance of γ-Al_2O_3 and TiO_2 nanoparticles dispersed in water flowing in a horizontal circular tube. Alumina (Al_2O_3) and titanium dioxide (TiO_2) nanoparticles with diameters of 13 nm and 27 nm, respectively, were used in their study. They found that the Nusselt number of nanofluids increased with an increase in the Reynolds number as

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0017-9310/$ - see front matter © 2009 Published by Elsevier Ltd.
doi:10.1016/j.ijheatmasstransfer.2008.10.023

Experimental Thermal and Fluid Science 33 (2009) 706–714

Contents lists available at ScienceDirect

Experimental Thermal and Fluid Science

journal homepage: www.elsevier.com/locate/etfs

Measurement of temperature-dependent thermal conductivity and viscosity of TiO_2-water nanofluids

Weerapun Duangthongsuk, Somchai Wongwises *

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, 126 Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Article history:
Received 24 November 2008
Received in revised form 16 January 2009
Accepted 22 January 2009

Keywords:
Nanofluids
Thermal conductivity
Viscosity
Transient hot-wire apparatus

ABSTRACT

Nanofluid is an innovative heat transfer fluid with superior potential for enhancing the heat transfer performance of conventional fluids. Many attempts have been made to investigate its thermal conductivity and viscosity, which are important thermophysical properties. No definitive agreements have emerged, however, about these properties. This article reports the thermal conductivity and dynamic viscosity of nanofluids experimentally. TiO_2 nanoparticles dispersed in water with volume concentration of 0.2–2 vol.% are used in the present study. A transient hot-wire apparatus is used for measuring the thermal conductivity of nanofluids whereas the Bohlin rotational rheometer (Malvern Instrument) is used to measure the viscosity of nanofluids. The data are collected for temperatures ranging from 15 °C to 35 °C. The results show that the measured viscosity and thermal conductivity of nanofluids increased as the particle concentrations increased and are higher than the values of the base liquids. Furthermore, thermal conductivity of nanofluids increased with increasing nanofluid temperatures and, conversely, the viscosity of nanofluids decreased with increasing temperature of nanofluids. Moreover, the measured thermal conductivity and viscosity of nanofluids are quite different from the predicted values from the existing correlations and the data reported by other researchers. Finally, new thermophysical correlations are proposed for predicting the thermal conductivity and viscosity of nanofluids.

1. Introduction

Heat transfer fluids such as water, oil, and ethylene glycol have poor heat transfer properties compared with those of most solids, especially thermal conductivity, and they are widely used in many industries including power generation, chemical processes, heating and cooling processes, transportation, microelectronics and other micro-sized applications. As described above, this behaviour was the main obstacle to the high effectiveness and compactness of the heat exchangers. Therefore, many researchers have been trying to create new kinds of heat transfer fluid in order to increase the heat transfer performance of the common fluids. In general, most solids have thermal conductivity several hundred times higher than that of conventional heat transfer fluids. Thus, an innovative way is to suspend small solid particles in the base liquids to enhance their thermal performance. Various types of particles such as metallic, non-metallic and polymeric can be dispersed into liquids to form colloidal solutions. As much as a decade ago, when nanotechnology was rapidly developing, particles of nanometer size were suspended in common liquids; the resulting liquid was named 'Nanofluid' by Steve Choi in 1995.

In order to study the heat transfer performance of nanofluids and use them in practical applications, however, it is necessary first to study their thermal conductivity and viscosity, which are important thermophysical properties. The sample investigations which reported the thermal conductivity and viscosity of nanofluids are as follows:

Masuda et al. [1] studied the thermophysical properties of the metallic oxide particles (Al_2O_3 and TiO_2) dispersed in water. The transient hot-wire method was used for measuring the thermal conductivity of nanofluids. They reported that the thermal conductivity of nanofluids was significantly larger than the base liquid. For example, the thermal conductivity of Al_2O_3-water nanofluids and TiO_2-water nanofluids at a 4.3 vol.% were approximately 32% and 11% higher than that of base liquid, respectively.

Pak and Cho [2] reported an experimental study of the heat transfer performance of γ-Al_2O_3-water and TiO_2-water nanofluids dispersed in water flowing in a horizontal circular tube under constant heat flux boundary conditions. Alumina (Al_2O_3) and titanium dioxide (TiO_2) nanoparticles with mean diameter of 13 nm and 27 nm, respectively, were used in their study. The Brookfield viscometer was used to measure the viscosity of nanofluids at room temperature. As regards the viscosity of the nanofluids, the results showed that it increased with increasing volume concentration. Pak and Cho also found that the relative viscosity of TiO_2-water

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0894-1777/$ - see front matter © 2009 Published by Elsevier Inc.
doi:10.1016/j.expthermflusci.2009.01.005

International Communications in Heat and Mass Transfer 36 (2009) 834–840

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

Numerical investigation on the heat transfer and flow in the mini-fin heat sink for CPU☆

Paisarn Naphon [a,*], Setha Klangchart [a], Somchai Wongwises [b]

[a] Thermo-Fluids and Heat Transfer Enhancement Lab. (TFHT), Department of Mechanical Engineering, Faculty of Engineering, Srinakharinwirot University, 63 Rangsit-Nakhornnayok Rd., Ongkharak, Nakhorn-Nayok, 26120, Thailand

[b] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, 91 Suksawas 48, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Available online 2 July 2009

Keywords:
Heat sink
Electronics cooling
CPU

ABSTRACT

Fluid flow and heat transfer in the mini-rectangular fin heat sink for CPU of PC using de-ionized water as working fluid are numerically investigated. Based on the real PC operating conditions, the three-dimensional governing equations for fluid flow and heat transfer characteristics are solved using finite volume scheme. The standard k–ε turbulent model is employed to describe the flow structure and behavior. The predicted results obtained from the model are verified by the measured data. There is a reasonable agreement between the predicted results and experiments. The results of this study are expected to lead to guidelines that will allow the design of the cooling system with improved cooling performance of the electronic equipments increasing reliable operation of these devices.

1. Introduction

The effective removal of the generated heat is an important role of the cooling system to ensure that electronic devices are at a safe operating temperature increasing reliable operation of these devices. The developments of the miniaturized technology, mini and micro-components have been introduced as one of the heat transfer enhancement techniques for these devices. Therefore, the heat transfer and pressure drop in the mini and micro-channel have been widely studied by researchers. Guglielmini et al. [1] experimentally studied the pool boiling heat transfer from finned copper surfaces immersed in a saturated dielectric fluid. Twelve test sections with different geometrical configurations were tested. Zhao and Lu [2] presented an analytical and numerical study effect of porosity on the thermal performance of a micro-channel heat sink. Honda and Wei [3,4] reviewed the researches concerning the boiling heat transfer enhancement for the micro-channel heat sink immersed in dielectric liquids. Feng and Xu [5] developed a three-dimensional analytical solution using the method of Fourier expansion for determination of the spreading thermal resistance of a cubic heat spreader for electronic cooling applications. Kobus and Oshio [6] theoretically and experimentally studied the thermal performance of a pin-fin heat sink with various geometrical configurations. Bhowmil [7] studied the steady-state convective heat transfer of water from in-line four electronic chips in a vertical rectangular channel. Li et al. [8] determined the thermal performance of heat sinks with confined impingement cooling. The effects of the impinging Reynolds number, the width and the height of the fins, the distance between the nozzle and the tip of the fins on the thermal resistance were investigated. Zhang et al. [9] reported the study of a single-phase heat transfer of micro-channel heat sink for electronic packages. Dogruoz et al. [10] experimentally and numerically studied the hydraulic resistance and heat transfer of in-line square pin-fin heat sinks. Chein and Chuang [11,22] studied the micro-channel heat sink performance with and without thermoelectric using nanofluids as coolants. Yu [12] studied the thermal performance of a plate pin-fin heat sink and a plate fin heat sink. Peles et al. [13] investigated the heat transfer and pressure drop over a bank of micro pin-fins. Kosar and Peles [14,15] experimentally studied the single-phase heat transfer, boiling heat transfer and pressure drop of refrigerant over a bank of shrouded micro pin-fins. Yakut et al. [16] considered the effects of heights, widths of the hexagonal fins on thermal resistance and pressure drop characteristics. Chiang and Chang [17] developed an effective procedure of response surface methodology for designing parameters of a pin-fin type heat sink. Mohamed [18] investigated the air-cooling characteristics of an electronic device heat sink with various square modules array. Launay [19] experimentally studied the boiling heat transfer from hybrid micro-nano structure. Didarul [20] investigated the heat transfer and fluid flow characteristics of finned surfaces. Yang [21] experimentally studied the heat transfer coefficient of pin-fin heat sinks with different cross sections. Twelve pin-fin heat sinks with

☆ Communicated by W.J. Minkowycz.

* Corresponding author.
E-mail address: paisarnn@swu.ac.th (P. Naphon).

0735-1933/$ – see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2009.06.010

INTERNATIONAL JOURNAL OF REFRIGERATION 32 (2009) 854–864

ELSEVIER

available at www.sciencedirect.com

ScienceDirect

journal homepage: www.elsevier.com/locate/ijrefrig

refrigeration

Experimental investigation of two-phase flow characteristics of HFC-134a through short-tube orifices

Kitti Nilpueng, Somchai Wongwises*

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Article history:
Received 16 May 2008
Received in revised form
4 December 2008
Accepted 7 December 2008
Published online 24 December 2008

Keywords:
Refrigerant
R134a
Experiment
Two-phase flow
Tube
Geometry
Flow
Pressure
Temperature

ABSTRACT

The two-phase flow characteristics of HFC-134a, including flow pattern, mass flow rate, pressure distribution and temperature distribution through short-tube orifices are experimentally investigated. Short tube diameters ranging between 0.605 and 1.2 mm with length-to-diameter ratios ranging between 8.3 and 33 are used in the experiments. The test runs are performed at upstream pressure ranging between 900 and 1300 kPa, downstream pressure ranging between 300 and 400 kPa, and degree of subcooling ranging between 1 and 12 °C. Two groups of short-tube orifices are used in the experiment. The first is used to visualise the flow pattern. The second is used to measure temperature and pressure distributions along the tube. The results from the present experiment show that metastable flow and choked flow phenomena exist inside the short-tube orifices over the whole range of experimental conditions. The metastable liquid flow region increases with increasing degree of subcooling and upstream pressure. The mass flow rate is directly proportional to upstream pressure and degree of subcooling. The results of pressure distribution inside the short-tube orifices indicate that accelerational pressure drop at the inlet and outlet has a significant effect on the total pressure drop across the short-tube orifice.

Etude expérimentale sur les caractéristiques d'écoulement diphasique du HFC-134a à travers des orifices de tubes courts

Mots clés : Frigorigène ; R134a ; Expérimentation ; Écoulement diphasique ; Tube ; Géométrie ; Débit ; Pression ; Température

1. Introduction

An expansion device used in vapour-compression refrigeration systems normally serves two purposes. One is to reduce the pressure of the refrigerant from the condenser pressure to the evaporator pressure, and the other is to regulate the mass flow rate of refrigerant flowing from the condenser to the evaporator. Basically, there are two types of expansion devices, i.e. variable-restriction type and constant-restriction type. In the former, the flow area can be varied depending on

* *Corresponding author.* Tel.: +662 470 9115; fax: +662 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).
0140-7007/$ – see front matter © 2008 Elsevier Ltd and IIR.
doi:10.1016/j.ijrefrig.2008.12.003

INTERNATIONAL JOURNAL OF REFRIGERATION 32 (2009) 1864–1875

available at www.sciencedirect.com

ScienceDirect

journal homepage: www.elsevier.com/locate/ijrefrig

Flow pattern, mass flow rate, pressure distribution, and temperature distribution of two-phase flow of HFC-134a inside short-tube orifices

Kitti Nilpueng[a], Chietta Supavarasuwat[b], Somchai Wongwises[a,*]

[a] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand
[b] C Aircon Tech Co., Ltd., Bangkae, Bangkok 10160, Thailand

A R T I C L E I N F O

Article history:
Received 30 April 2009
Received in revised form 27 July 2009
Accepted 1 August 2009
Published online 6 August 2009

Keywords:
Refrigerant
R134a
Experiment
Two-phase flow
Tube
Geometry
Orifice-flow
Pressure drop
Temperature

A B S T R A C T

New experimental data on the influence of short-tube orifice configuration, including diameter, length, length-to-diameter ratio (*L/D*), and orientation on the flow pattern, mass flow rate, and pressure distribution of HFC-134a inside the short-tube orifice are presented. Short-tube orifice diameters ranging between 0.605 and 1.2 mm with *L/D* ranging between 1.87 and 33 are used in the experiments. Three different forms of the metastable liquid flow, which are metastable liquid core flow, conical metastable liquid core flow, and full metastable liquid flow are visually observed. The short-tube orifice diameter has a significant effect on the increase in the flow rate. Conversely, the change in the orientation of the test section has no significant effect on the flow rate. The choke flow phenomenon disappears inside the short-tube orifice when *L/D* is less than 2.91. Based on the present data, a correlation for predicting the mass flow rate through short-tube orifices is proposed.

Configuration de l'écoulement, débit massique, distribution de la pression et de la température et écoulement diphasique du HFC-134a à l'intérieur d'orifices de tubes courts

Mots clés : frigorigène ; R134a ; expérimentation ; écoulement diphasique ; tube ; géométrie ; orifice ; débit ; chute de pression ; température

* *Corresponding author*. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).
0140-7007/$ – see front matter © 2009 Elsevier Ltd and IIR.
doi:10.1016/j.ijrefrig.2009.08.002

Energy Conversion and Management 50 (2009) 1851–1861

Contents lists available at ScienceDirect

Energy Conversion and Management

Journal homepage: www.elsevier.com/locate/enconman

Evaporation heat transfer enhancement of R-134a flowing inside smooth and micro-fin tubes using the electrohydrodynamic technique

Kittiporn Posew, Suriyan Laohalertdecha, Somchai Wongwises *

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Article history:
Received 29 September 2007
Received in revised form 6 July 2008
Accepted 10 February 2009
Available online 9 April 2009

Keywords:
Electrohydrodynamic
Evaporation
Smooth tube
Micro-fin tube
R-134a
Heat transfer enhancement

ABSTRACT

Heat transfer enhancement using the electrohydrodynamic (EHD) technique during evaporation inside horizontal smooth and micro-fin tubes is experimentally investigated. The test section is a counter-flow concentric tube-in-tube heat exchanger with R-134a flowing inside inner tube and heating water flowing in the annulus. Smooth and micro-fin inner tubes are used as test section. The outer diameter and length of inner tubes are 9.52 mm and 2.50 m, respectively. The outer tube is a smooth copper tube having inner diameter of 27 mm. The electrode is made from stainless steel rod of 1.47 mm diameter. Experiments are performed at average saturation temperatures ranging between 10 and 20 °C, heat fluxes ranging between 10 and 20 kW/m^2 at mass flux of 400 kg/m^2 s and applied voltage of 2.5 kV. The maximum heat transfer enhancement ratios are 1.25 and 1.15 for smooth tube and micro-fin tube, respectively. New correlations based on the data gathered during this work for predicting the evaporation heat transfer coefficient in smooth tube and micro-fin tube are proposed for practical applications.

1. Introduction

The global warming and energy shortage are major issues for this decade. Some chemical substance, for example CFCs, HCFCs, CO_2 can cause the greenhouse effect to increase in the atmosphere. CFCs and HCFCs also destroy the ozone in the stratosphere (15–20 km above the earth's surface). The ozone is necessary to filter out the sun's short wavelength ultraviolet (UV) radiation. The UV radiation from the sun is main cause of serious sunburn, skin cancer and eye disorders. Now, the refrigeration industry has switched over to "ozone-friendly" refrigerants such as R-134a. The CO_2 released during production of power to run the refrigerant device can also be reduced by increasing heat transfer rate in heat exchanger. Heat transfer enhancement techniques such passive method as surface modification and such active ones as mechanical vibration, injection, and electric field have been developed by many researchers. EHD technique is one of active methods which can be achieved by the interaction between electric field and the flowing of dielectric fluid medium. This interaction creates additional fluid motion leading to a higher heat transfer coefficient. The electric body force density acting on the molecules of dielectric fluid in the presence of an electric field can be expressed as:

$$f_e = qE - \frac{1}{2}E^2\nabla\varepsilon + \frac{1}{2}\nabla\left\langle E^2\left(\frac{\partial\varepsilon}{\partial\rho}\right)_T\rho\right\rangle \quad (1)$$

The effect of EHD on heat transfer enhancement has been studied by several researchers. Laohalertdecha and Wongwises [1] investigated the effects of EHD on the condensation heat transfer and pressure drop of pure R-134a inside a horizontal micro-fin tube. The test section was a 2.5 m long concentric counter-flow tube-in-tube heat exchanger with refrigerant flowing in the inner tube and cooling water flowing in the annulus. A micro-fin inner tube was used as the test section. The electrode was made from cylindrical stainless steel of 1.47 mm diameter. The results showed that the maximum heat transfer enhancement ratio is 1.15 while the application of an EHD voltage of 2.5 kV only slightly increases the pressure drop. In their second paper, Laohalertdecha and Wongwises [2] studied the effect of EHD on heat transfer enhancement during two-phase condensation of R-134a at high mass flux in a horizontal smooth tube. The experimental results indicated that the maximum heat transfer enhancement ratio is around 30% while the maximum increase in pressure drop is about 25%. Laohalertdecha and Wongwises [3] also compared the effect of electrohydrodynamic on the condensation heat transfer of R-134a inside smooth and micro-fin tubes. The results demonstrated that the maximum heat transfer enhancement ratios for smooth and micro-fin tubes are 1.1 and 1.08, respectively.

The application of EHD to the pool boiling heat transfer enhancement has been demonstrated by several researchers. Berghmans [4] theoretically studied the effect of electrostatic fields

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0196-8904/$ - see front matter © 2009 Elsevier Ltd. All rights reserved.
doi:10.1016/j.enconman.2009.02.003

Springer

Journal of Mechanical Science and Technology

Journal of Mechanical Science and Technology 23 (2009) 1620~1627

www.springerlink.com/content/1738-494x
DOI 10.1007/s12206-008-1106-1

Predicting performance of radial flow type impeller of centrifugal pump using CFD †

Suthep Kaewnai, Manuspong Chamaoot and Somchai Wongwises*

Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

(Manuscript Received August 31, 2006; Revised April 20, 2008; Accepted November 17, 2008)

Abstract

The main objective of this work is to use the computational fluid dynamics (CFD) technique in analyzing and predicting the performance of a radial flow-type impeller of centrifugal pump. The impeller analyzed is at the following design condition: flow rate of 528 m^3/hr; speed of 1450 rpm; and head of 20 m or specific speed (N_s) of 3033 1/min in US-Units. The first stage involves the mesh generation and refinement on domain of the designed impeller. The second stage deals with the identification of initial and boundary conditions of the mesh-equipped module. In the final stage, various results are calculated and analyzed for factors affecting impeller performance. The results indicate that the total head rise of the impeller at the design point is approximately 19.8 m. The loss coefficient of the impeller is 0.015 when $0.6 < Q/Q_{design} < 1.2$. Maximum hydraulic efficiency of impeller is 0.98 at $Q/Q_{design} = 0.7$. Based on the comparison of the theoretical head coefficient and static pressure rise coefficient between simulation results and experimental data, from previous work reported in the literature [Guelich, Kreiselpumpen, Springer, Berlin, 2004], it is possible to use this method to simulate the performance of a radial-flow type impeller of a centrifugal pump.

Keywords: Centrifugal pump; CFD; Impeller; Radial flow

1. Introduction

The use of CFD plays an important role in fluid mechanics. Due to the progress of numerical methods and computer capability, the impeller design for centrifugal pumps nowadays has been analyzed by using 3D-Navier stokes program or CFD software to predict impeller performance in advance. However, both the prediction and design cannot be done easily without sufficient information and experience. If the result from the CFD is much different from the designed value, the impeller could be redesigned before being put into real production.

Over the years, the application of CFD in fluid mechanics and turbomachinery has been studied by a number of researchers [1-8]. However there are few studies applying the CFD for predictions involving a radial flow type impeller. To the best of our knowledge to date, there have been only two works carried out by [9] and [10], in dealing with this issue.

[9] analyzed about 30 pump impellers having specific speeds between 12 to 160 (metric units) using a commercial 3D-Navier Stokes program for a standard k-ε turbulence model. The head and efficiency obtained from the software were compared to those obtained from the measurement. A set of rather stringent rules for the CFD calculations was developed in order to reduce the effect of grid and calculation parameters on the results.

[10] performed a 3D-CFD simulation of the impeller and volute of a centrifugal pump having a specific speed of 32 (metric units) and an outside impeller radius of 204.2 mm, by using CFX codes. A 3D flow simulation for the impeller with a structured grid was

† This paper was recommended for publication in revised form by Associate Editor Seungbae Lee

*Corresponding author. Tel.: +662 470 9115, Fax.: +662 470 9111
E-mail address: somchai.won@kmutt.ac.th

International Journal of Heat and Mass Transfer 52 (2009) 2370–2378

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

A review on reduction method for heat and mass transfer characteristics of fin-and-tube heat exchangers under dehumidifying conditions

Worachest Pirompugd [a], Chi-Chuan Wang [b], Somchai Wongwises [c,*]

[a] Department of Mechanical Engineering, Faculty of Engineering, Burapha University, Saensook, Muang, Chonburi 20131, Thailand
[b] Energy and Environment Research Lab., Industrial Technology Research Institute, Hsinchu 310, Taiwan, ROC
[c] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Article history:
Received 13 March 2008
Received in revised form 23 October 2008
Available online 30 December 2008

Keywords:
Dehumidifying condition
Fin-and-tube heat exchanger
Heat mass and momentum transfer
Reduction method

ABSTRACT

This paper presents a review of data reduction method for heat and mass transfer characteristics of fin-and-tube heat exchangers with dehumidification. There are many reduction methods for fin-and-tube heat exchangers under dehumidifying conditions. The data reduction methods being reviewed includes the original Threlkeld method, direct method, the equivalent dry-bulb method, tube by tube method, the fully wet and fully dry tiny circular fin method, and the finite circular fin method. Among these methods, the original Threlkeld method, direct method, the equivalent dry-bulb method are lumped method while others can divide the fin-and-tube heat exchangers into small segments for more accuracy in handling the surfaces to be fully dry, fully wet, or partially wet. In addition, the mass transfer characteristics can be obtained from the modified process line equation incorporated with the preceding methods. It should be noted that the conventional assumption of constant ratio ($h_{c,o}/h_{d,o}C_{p,a} \approx$ constant) is actually incorrect. This present paper can be used as the first guideline for the researcher for reducing the experimental data for fin-and-tube heat exchangers under dehumidifying conditions.

1. Introduction

Fin-and-tube heat exchangers are widely used in applications of HVAC&R (heating, ventilating, air-conditioning, and refrigeration) systems. They include a group of fins arranged parallel to themselves at a constant spacing. The configuration of fins can be either continuous (plain or wavy) or interrupted (louver, slit or the like). One or more tubes which take the forms of either circular, flat, or oval configuration are arranged perpendicularly to the direction of air flow. The fin-and-tube heat exchangers can be applicable to condensers and evaporators. In evaporators, which typically use aluminum fins with the surface temperature generally being below the dew point temperature result in simultaneous heat and mass transfer outside tube and fin surfaces. In practice, the fin surfaces may be fully wet or partially wet depending on the local difference between fin surface temperature and dew point temperature. Many researchers presented the mathematical model for predicting the heat and mass transfer characteristics of moist air for fin-and-tube heat exchanger. It is therefore the objective of this review article to present relevant reduction method concerning the heat, mass, and momentum transfer characteristic of fin-and-tube heat exchangers.

2. Experimental apparatus

Many published literatures had experimentally investigated the heat and mass transfer characteristics of the fin-and-tube heat exchanger under dehumidification. A typical schematic diagram of the experimental air circuit assembly is shown in Fig. 1. It consists of a closed-loop wind tunnel in which air is circulated by a variable speed centrifugal fan. The dry-bulb and wet-bulb temperatures of the inlet-air are controlled by an air-ventilator. The air flow-rate measurement station is an outlet chamber set up with multiple nozzles. It is recommended that the setup is constructed following the ASHRAE 41.2 standard [1] and applied a differential pressure transducer is used to measure the pressure difference across the nozzles. It is also recommended that the air temperatures at the inlet and exit zones across the sample heat exchangers are measured by two psychrometric boxes based on the ASHRAE 41.1 standard [2]. The energy imbalance should satisfy the ASHRAE [3] requirements (namely, the energy balance condition, $|\dot{Q}_r - \dot{Q}_a|/\dot{Q}_{avg}$, is less than 0.05, where $\dot{Q}_r$ is the water side heat transfer rate and $\dot{Q}_a$ is the air-side heat transfer rate) are considered in the final analysis.

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0017-9310/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ijheatmasstransfer.2008.10.019

Heat Mass Transfer (2009) 45:381–391
DOI 10.1007/s00231-008-0441-1

ORIGINAL

Heat transfer characteristics of a new helically coiled crimped spiral finned tube heat exchanger

Kwanchanok Srisawad · Somchai Wongwises

Received: 26 March 2007 / Accepted: 27 August 2008 / Published online: 27 September 2008

Abstract In the present study, the heat transfer characteristics in dry surface conditions of a new type of heat exchanger, namely a helically coiled finned tube heat exchanger, is experimentally investigated. The test section, which is a helically coiled fined tube heat exchanger, consists of a shell and a helical coil unit. The helical coil unit consists of four concentric helically coiled tubes of different diameters. Each tube is constructed by bending straight copper tube into a helical coil. Aluminium crimped spiral fins with thickness of 0.5 mm and outer diameter of 28.25 mm are placed around the tube. The edge of fin at the inner diameter is corrugated. Ambient air is used as a working fluid in the shell side while hot water is used for the tube-side. The test runs are done at air mass flow rates ranging between 0.04 and 0.13 kg/s. The water mass flow rates are between 0.2 and 0.4 kg/s. The water temperatures are between 40 and 50°C. The effects of the inlet conditions of both working fluids flowing through the heat exchanger on the heat transfer coefficients are discussed. The air-side heat transfer coefficient presented in term of the Colburn J factor is proportional to inlet-water temperature and water mass flow rate. The heat exchanger effectiveness tends to increase with increasing water mass flow rate and also slightly increases with increasing inlet water temperature.

K. Srisawad · S. Wongwises (✉)
Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand
e-mail: somchai.won@kmutt.ac.th

List of symbols

A	area (m^2)
C_p	specific heat [kJ/(kg K)]
d	tube diameter (m)
D	diameter of the curvature (m)
D_c	diameter of the coil (m)
f	friction factor
F	correction factor
G	mass flux [kg/(m^2 s)]
h	heat transfer coefficient [W/(m^2 K)]
i	enthalpy (kJ/kg)
I_o	modified Bessel function solution of the first kind, order 0
I_1	modified Bessel function solution of the first kind, order 1
j	Colburn j factor
K_o	modified Bessel function solution of the second kind, order 0
K_1	modified Bessel function solution of the second kind, order 1
k	thermal conductivity [W/(m K)]
L	tube length (m)
m	mass flow rate (kg/s)
Nu	Nusselt number
P	pitch of the helical coil (m)
Pr	Prandtl number
Q	heat transfer rate (W)
r	tube radius (m)
Re	Reynolds number
T	temperature (°C)
U	overall heat transfer coefficient [W/(m^2 K)]
V	average velocity (m/s)
δ	thickness (m)
η	fin effectiveness
η_o	overall surface effectiveness

Experimental Thermal and Fluid Science 33 (2009) 306–315

Contents lists available at ScienceDirect

Experimental Thermal and Fluid Science

journal homepage: www.elsevier.com/locate/etfs

An experimental investigation of two-phase air–water flow through a horizontal circular micro-channel

Sira Saisorn [a], Somchai Wongwises [b,*]

[a] Energy Division, The Joint Graduate School of Energy and Environment (JGSEE), King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand
[b] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Lab. (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology, Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Article history:
Received 30 May 2008
Received in revised form 2 September 2008
Accepted 19 September 2008

Keywords:
Two-phase flow
Micro-channel
Flow pattern
Void fraction
Pressure drop

ABSTRACT

This paper is a continuation of the authors' previous work. Two-phase air–water flow experiments are performed in a horizontal circular micro-channel. The test section is made of a fused silica tube with an inner diameter of 0.15 mm and a length of 104 mm. The flow phenomena, which are liquid/unstable annular alternating flow (LUAAF), liquid/annular alternating flow (LAAF), and annular flow, are observed and recorded by a high-speed camera mounted together with a stereozoom microscope. A flow pattern map is presented in terms of the phase superficial velocities and is compared with those of other researchers obtained from different working fluids. Image analysis is performed to determine the void fraction, which increases non-linearly with increasing volumetric quality. It is revealed that the two-phase frictional multiplier data show a dependence on flow pattern rather than mass flux. Based on the present data, a new pressure drop correlation is proposed for practical applications. According to the present study, in general the data for the two-phase air–water flow characteristics are found to comply with those of working fluids other than air–water mixtures.

1. Introduction

Two-phase flow and heat transfer mechanisms in mini- and micro-channels are found to become highly prevalent in recent years. The attractive phenomena appearing in the very small flow passages are necessary for the development of the modern technology applications such as medical devices, high heat-flux compact heat exchangers, cooling systems for high performance micro-electronics and so on.

Two-phase flow and heat transfer characteristics in small channels such as micro- and mini-channels are likely to be strongly dependent on surface tension effects in addition to viscosity and inertia forces, resulting in significant differences in two-phase flow phenomena between ordinarily sized channels and small channels. Several investigators have proposed criteria to address the definition of a micro-channel. The proposed channel classifications are often based on different dimensionless parameters. For instance, arbitrary channel classifications based on the hydraulic diameter D_h have been proposed. Recently, Mehendale et al. [1] employed the hydraulic diameter as an important parameter for defining heat exchangers and Kandlikar [2] proposed criteria for small flow channels used in engineering applications.

As we know, however, a clear understanding of the micro-scale effects on two-phase flow and heat transfer characteristics is still lacking. Thus, further investigations are essential for optimum design and process control of micro-systems. In the present study, the characteristics of air–water flow instead of N_2–deionised water flow, which have never been seen before, are investigated in a horizontal circular micro-channel with a diameter of 0.15 mm.

As compared with the reported two-phase flow mechanisms in ordinarily sized channels, which are available in a relatively large number of publications, hydrodynamic and transport phenomena in mini- and micro-channels tend to show different behaviours due to the effects of the limited, confined space. Recent studies associated with flow patterns, void fractions and pressure drops of two-phase flows in small channels are outlined in the following paragraphs.

1.1. Flow pattern

It is not possible to understand two-phase flow phenomena without a clear understanding of the flow patterns encountered. It is expected that flow patterns will influence the two-phase pressure drop, holdup, system stability, exchange rate of momentum, heat, and mass during the phase-change heat transfer process. The ability to accurately predict the type of flow is necessary before relevant calculation techniques can be developed.

* Corresponding author. Tel.: +662 470 9115; fax: +662 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0894-1777/$ - see front matter © 2008 Elsevier Inc. All rights reserved.
doi:10.1016/j.expthermflusci.2008.09.009

International Journal of Heat and Mass Transfer 52 (2009) 1582–1588

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ijhmt

Nucleate pool boiling heat transfer of TiO_2–R141b nanofluids

Visinee Trisaksri [a], Somchai Wongwises [b,*]

[a] Energy Division, The Joint Graduate School of Energy and Environment, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

[b] Fluid Mechanics, Thermal Engineering and Multiphase Flow Research Laboratory (FUTURE), Department of Mechanical Engineering, King Mongkut's University of Technology Thonburi, Bangmod, Bangkok 10140, Thailand

ARTICLE INFO

Article history:
Received 12 February 2008
Received in revised form 28 July 2008
Available online 22 October 2008

Keywords:
Nanofluid
Nanoparticle
Suspension
Nucleate pool boiling heat transfer

ABSTRACT

Nucleate pool boiling heat transfer of a refrigerant-based-nanofluid was investigated at different nanoparticle concentrations and pressures. TiO_2 nanoparticles were mixed with the refrigerant HCFC 141b at 0.01, 0.03 and 0.05 vol%. The experiment was performed using a cylindrical copper tube as a boiling surface. Pool boiling experiments of nanofluid were conducted and compared with that of the base refrigerant. The results indicate that the nucleate pool boiling heat transfer deteriorated with increasing particle concentrations, especially at high heat fluxes. At 0.05 vol%, the boiling heat transfer curves were suppressed. At high pressures of 400 and 500 kPa, the boiling heat transfer coefficient at a specific excess temperature was almost the same.

1. Introduction

In a refrigeration system, the optimum design of the evaporator depends on the correct evaluation of the nucleate boiling heat transfer of the refrigerant. In recent years, environmental concerns over the use of CFCs have led to the development of alternative fluids to replace CFC refrigerants. An innovative technique in improving heat transfer is to suspend the nanometer-size solid particles in base fluids, resulting in a substance that was called "nanofluid" by Choi [1]. Several recently published articles reported the substantial enhancement of thermal conductivity. Eastman et al. [2] also reported on the significance of thermal conductivity enhancement. They achieved up to a 60% increase in the thermal conductivity at 5 vol% of CuO nanoparticles in water. Murshed et al. [3] measured the thermal conductivity of TiO_2–water nanofluid. The thermal conductivity was enhanced by up to 33%.

Since nanofluids have a higher thermal conductivity than base fluids, the heat transfer properties of nanofluids are expected to be higher than those of the base fluids, which makes them more attractive for heat transfer applications, especially in the case of pool boiling heat transfer.

Das et al. [4,5] carried out an experiment to evaluate pool boiling heat transfer using a horizontal heater tube and nanofluids with 1%, 2% and 4% volume fractions of Al_2O_3 nanoparticles suspended in water. The results were unexpected: nanofluids were expected to enhance the heat transfer characteristics during pool boiling, however, the boiling curves of nanofluids indicated that the boiling heat transfer of the water had in fact deteriorated with the addition of nanoparticles. The resulting deterioration was dependent on the tube roughness and the increase in particle volume fraction. Furthermore, the deterioration of heat transfer performance was stronger with a smoother surface.

The deterioration in nucleate boiling heat transfer of Al_2O_3–water nanofluid was also observed in the work of Bang and Chang [6]. In this study, a very smooth horizontal flat surface was used as the boiling surface, and critical heat flux enhancement was observed.

Controversial results were reported by Wen and Ding [7], who used some surfactants and electrostatic stabilization methods. The nucleate pool boiling heat transfer of Al_2O_3–water nanofluid on a horizontal flat surface was enhanced by up to 40% at a particle concentration of 1.25% by weight.

You et al. [8] conducted an experimental study to determine the boiling curve and critical heat flux in pool boiling from a flat square polished copper heater immersed in Al_2O_3–water nanofluid. Various nanoparticles with volume fractions of Al_2O_3 that ranged from 0.001 g/l to 0.05 g/l were tested and compared with pure water. In the nucleate boiling regime of the boiling curves of the nanofluids, heat transfer enhancement and degradation were not observed. However, the critical heat fluxes of the nanofluids were significantly increased to about 200% higher than pure water when the particle volume fractions were 0.005 g/l.

Zhou [9] conducted an experiment to study the effect of acoustical parameters, nanofluid concentration and fluid subcooling on boiling heat transfer characteristics of copper-acetone nanofluid. He found that without an acoustic field, the boiling heat transfer of nanofluid was reduced. With an acoustic field, on the other hand, heat transfer enhancement was observed and the boiling hysteresis disappeared. However, the heat transfer enhancement

* Corresponding author. Tel.: +66 2 470 9115; fax: +66 2 470 9111.
E-mail address: somchai.won@kmutt.ac.th (S. Wongwises).

0017-9310/$ - see front matter © 2008 Elsevier Ltd. All rights reserved.
doi:10.1016/j.ijheatmasstransfer.2008.07.041