UTHM
Universiti Tun Hussein Onn Malaysia

Prince of Songkla University

Abstract Book

2nd International Conference on Computational Fluid Dynamics in Research and Industry CFDRI 2017

" Streamlining Technological Shift "

3rd - 4th August 2017 | Songkhla, Thailand

Track 1:
 # CFD Methods # Heat Transfer # Fluid Structure Interaction # Electronics # Fluid Machines

Track 2:
 # Fluid Flow Characterisation # Turbulence Analysis # Automotive Engineering # Porous Media Flow # Aerodynamics

Track 3:
 # Heat Exchangers # Spray Analysis # Industrial Environment # Indoor Air Quality # Combustion

Paper ID	**Track**	**Title**
13-457	Track 1	EXPERIMENTAL AND NUMERICAL SIMULATION OF A MOVING CARRIER (FX63-137) THROUGH AN AIR TUNNEL

Abstract This study investigated the movement of a carrier through an air tunnel using numerical simulation and experimental approach conducted in UTHM laboratories. The carrier was designed to have a particular trajectory moves along the air tunnel. The body of the main carrier has a shape of cube and rectangular wing of aspect ratio 2 is set in the middle of the body which has a shape of airfoil FX63-137 as its airfoil section. Experimental result of the carrier movement was investigated and compared with the simulation result obtained by Fluent's software. The experiment was conducted using a flow speed of 19.59 m/s at the inlet station. In the simulation, the flow is considered as an internal flow and has a constant temperature at the entry station. The pressure ratio between the inlet and outlet of the air tunnel is 235 N/m^2. In addition, that the flow is considered as turbulent flow and the present study used k-ε as its turbulence modelling in the numerical simulation. Comparison result between experiment and Fluent software in term of its trajectory and the speed of the carrier movement are found in a good agreement.

Paper ID	**Track**	**Title**
13-458	Track 1	THREE-DIMENSIONAL FINITE VOLUME MODELLING OF BLOOD FLOW IN SIMULATED ANGULAR NECK ABDOMINAL AORTIC ANEURYSM

Abstract An abdominal aortic aneurysm (AAA) is considered a deadly cardiovascular disease that defined as a focal dilation of blood artery. The healthy aorta size is between 15 and 24 mm based on gender, body weight, and age. When the diameter increased to 30 mm or more, the rupture can occur if it is kept growing or untreated. Moreover, the proximal angular neck of aneurysm is categorized as a significant morphological feature with prime harmful effects on endovascular aneurysm repair (EVAR). Flow pattern in pathological vessel can influence the vascular intervention. The aim of this study is to investigate the blood flow behaviors in angular neck abdominal aortic aneurysm with simulated geometry based on patient's information using computational fluid dynamics (CFD). The 3D angular neck AAA models have been designed by using SolidWorks Software. Consequently, CFD tools are used for simulating these 3D models of angular neck AAA in ANSYS FLUENT Software. Eventually, based on the results, we summarized that the CFD techniques have shown high performance in explaining and investigating the flow patterns for angular neck abdominal aortic aneurysm.

Paper ID	Track	Title
13-460	Track 1	OPTIMAL PISTON CREVICE STUDY IN A RAPID COMPRESSION MACHINE

Abstract Multi-dimensional effects such as vortex generation and heat losses from the gas to the wall of the reactor chamber have been an issue to obtaining a reliable RCM data. This vortex initiates a flow in the relatively cold boundary layer, which may penetrate the core gas. This resulting non-uniformity of the core region could cause serious discrepancies and give unreliable experimental data. To achieve a homogenous temperature field, an optimised piston crevice was designed using CFD modelling (Ansys fluent). A 2-Dimensional computational moving mesh is assuming an axisymmetric symmetry. The model adopted for this calculation is the laminar flow model and the fluid used was nitrogen. To get the appropriate crevice volume suitable for the present design, an optimisation of the five different crevice volume was modelled which resulted to about 2-10% of the entire chamber volume. The use of creviced piston has shown to reduce the final compressed gas temperature and pressure in the reactor chamber. All the crevice volumes between 2-10% of the chamber volume adequately contained the roll up vortexes, but the crevice volume of 282 mm3 was chosen to be the best in addition to minimising the end gas pressure and temperature drop. The final pressure trace from experiment shows a reasonable agreement with the CFD model at compression and post compression stage.

Paper ID	Track	Title
13-464	Track 1	THE STUDY OF FLOW AND HEAT TRANSFER CHARACTERISTICS OF IMPINGING JET ARRAY MOUNTING AIR-INDUCED DUCT

Abstract Impinging jet is widely employed in thermal industrial applications due to having high heat transfer coefficient in impingement region. One method to increase heat transfer on an impingement surface is to increase turbulence intensity in jet flow. The mounting of an air-induced duct at nozzle outlet is a passive method to increase entrainment air resulting on increasing turbulence intensity. The aim of this research is to study flow and heat transfer characteristics of array of impinging jets mounting air-induced ducts. The investigation model was jets discharging from pipe nozzle having an inner diameter of d=17.2 mm and a length of 200 mm. Nozzle arrangement were inline configuration having 5 rows x 5 columns. A jet-to-jet distance (S) was S=6d, 8d and a jet-to-plate distance (H) was H=6d. The inner diameter (D) and the length (L) of the air-induced ducts were D=4d and L=4d, respectively. The Reynolds number was fixed at Re=20,000. In addition, the impinging jets without mounting the air-induced ducts were also investigated for benchmarking with the case of mounting the air-induced ducts. In the study, a thin foil technique was used to measure heat transfer on the impingement surface, and a computational fluid dynamic (CFD) using ANSYS, Fluent (V.15.0) was also adopted. The results showed that the effect of mounting air-induced duct can enhance entrainment air into the jet flow resulting on increasing of heat transfer of impinging jets on target surface, and the effects of mounting air-induced duct on increasing heat transfer in case of larger jet-to-jet distance (S/d=8) was more effective than that of smaller jet-to-jet distance (S/d=6).

Paper ID	Track	Title
13-465	Track 1	THE EFFECT OF CONICAL DIMPLE SPACING ON FLOW STRUCTURE AND HEAT TRANSFERCHARACTERISTICS OF INTERNAL FLOW USING CFD

Abstract In the present study, heat transfer and flow characteristics simulations over the surface of conical dimple were investigated. Single dimple row with the inline arrangement was formed on the internal surface of the 3-D rectangular wind tunnel model. The air flow was perpendicular to the centre line of every dimple and the printed diameter of dimples on the surface was D=26.4mm. The depth of dimple on

the surface of wind tunnel was H/D=2. The space between dimple-to-dimple was varied for S/D=1.125, 1.25,1.5, and 2. The Reynold number based on the hydraulic diameter of internal air flow was 20,000 depending on the wind tunnel hydraulic diameter. The numerical computation was applied with a Shear Stress Transport (SST) k- turbulence model. The average Nusselt number for the S/D=1.125 case is the highest. When the spacing becomes increase, the value of average Nusselt number tends to decrease.

Paper ID	Track	Title
13-496	Track 1	A CFD STUDY ON TURBULENT FORCED CONVECTION FLOW OF AL2O3-WATER NANOFLUID IN SEMI- CIRCULAR CORRUGATED CHANNEL

Abstract The performance of heat exchangers especially for single phase flows can be enhanced by many augmentation techniques. One of the most popular method used is a passive heat transfer technique. Researchers have been quite active in the search of novel ways on heat transfer augmentation techniques using various types of passive techniques to increase heat transfer performances of heat exchanger. Computational Fluid Dynamics (CFD) simulations of heat transfer and friction factor analysis in a turbulent flow regime in semi-circle corrugated channels with Al2O3-water nanofluid is presented in this paper. Simulations are carried out at Reynolds number range of 10000-30000, with nanoparticle volume fractions 0-6% and constant heat flux condition. The results for corrugated channels are examined and compared to those for straight channels. Results show that the Nusselt number increased with the increase of nanoparticle volume fraction and Reynolds number. The Nusselt number was found to increase as the nanoparticle diameter decreased. Maximum Nusselt number ratio 2.07% at Reynolds number 30,000 and volume fraction 6%.

Paper ID	Track	Title
13-500	Track 1	PARAMETRIC STUDY AND SHRINKAGE MODELLING OF NATURAL RUBBER SHEET DRYING USING COMSOL MULTIPHYSICS

Abstract Natural rubber is one of the major exporting cash crops in Thailand. Shrinkage in natural rubber sheet during drying creates uneven stresses in the rubber, hampers the quality and the water activity predisposes it to microbial activation. Hence, the effect of drying parameters on shrinkage has been considered in this work. The finite element concept coupled with the arbitrary Lagrangian-Eulerian (ALE) method was used to solve the two-dimensional; one way coupled fluid structure interaction in the rubber sheet drying chamber and account for the shrinkage effect. The spatial domain (drying chamber), material domain (rubber sheet) was combined. An isotropic linear elastic model was assumed for the rubber sheet for analysis. Three Case studies of different velocity, temperature, relative humidity and shrinkage coefficient were considered in the numerical study using COMSOL Multiphysics. It is concluded that increase in the operating parameters increases the shrinkage of the rubber. Therefore, rubber should dry at relatively lower operating parameters to improve its quality but not too low as to increase microbial activity in the rubber.

Paper ID	Track	Title
13-513	Track 1	NUMERICAL SIMULATIONS ON FLOW AND HEAT TRANSFER IN RIBBED TWO-PASS SQUARE CHANNELS UNDER ROTATIONAL EFFECTS

Abstract The main objective of this research is to study the flow and heat transfer characteristics in a rotating two-pass square channel with ribbed walls. In this study, the channel length-to-hydraulic diameter ratio of the rotating two-pass square channel (L/Dh), the rib height-to-hydraulic diameter ratio (e/Dh), rib angle of attack and the rib pitch-to-height (p/e) ratio are fixed at 11.33, 0.13, 60o and 10, respectively. The test fluid is air having the flow rate in terms of constant Reynolds number (Re) of 10,000. The rotation numbers (Ro) are varied from 0.1 to 0.4. The details of the local heat transfer distribution and the flow field of the rotating two-pass square channel are numerically studied by using commercial software ANSYS ver.15.0 (Fluent). The results from this study show that the ribbed walls enhance the heat transfer rate significantly. Under rotation, the average Nu ratio in the first pass with radial outward flow is increased while that in the second pass is decreased, and also found that maximum heat transfer rate is observed for rotation number of 0.4 which is higher than about 10-20% when compared with the other rotation number cases.

Paper ID	**Track**	**Title**
13-516	Track 1	EFFECT OF INCLINED ANGLE OF PIN ARRAYS ON FLOW AND HEAT TRANSFER CHARACTERISTICS IN FLOW CHANNEL WITH COMPUTATIONAL FLUID DYNAMICS

Abstract The aim of this research is to study flow and heat transfer characteristics on an internal surface of rectangular channel by mounting inclined pin arrays on heat transfer surface, ANSYS ver.15.0 (Fluent) to simulate 3-D steady flow and heat transfer (incompressible flow). The model of fluid flow and heat transfer were analyzed using the realizable turbulence model. The cylindrical of pin arrays which having diameter of D=10 mm was mounted on heat transfer surface in the wind tunnel with staggered arrangement. The pin-to-pin distance was fixed at Sy=2D in spanwise direction and at Sx=2.5 in streamwise direction. The effect of pin inclined angle was investigated at =60o, 90o, 120o, and 135o. For all simulations, the Reynolds number of airflow was fixed at Re=32,860. The results show that the pin inclined angle with = 120o and 135o can enhance the heat transfer on the upstream and downstream of the pin when compared to the case of pin angle =90o. While, the pin inclined angle with =60ogives the lowest heat transfer rate overall surface. The mechanism of heat transfer enhancement for inclined pin can be explained with increasing velocity and reduced wake flow behind the pins.

Paper ID	**Track**	**Title**
13-524	Track 1	EFFECT OF TURNING ANGLE ON PERFORMANCE OF 2-D TURNING DIFFUSER VIA ASYMPTOTIC COMPUTATIONAL FLUID DYNAMICS

Abstract The present work aims to numerically investigate the effect of varying = 30° - 90° on the performance of 2-D turning diffuserϕturning angle, and to develop the performance correlations via integrating the turning angle using Asymptotic Computational Fluid Dynamics (ACFD) technique. 1.1 appeared as the≈Standard k-ε adopting enhanced wall treatment of y+ best validated model to represent the actual cases with deviation of ±4.7%. Results show that the pressure recovery, Cp and flow uniformity, out are distorted of respectively 37% and 28% with the increment ofσ turning angle from 30° to 90°. The flow separation starts to emerge within the inner wall, S=0.91Lin/W1 when 45° turning diffuser is applied and its scale is enlarged by further increasing the turning angle. The performance correlations of 2-D turning diffuser are successfully developed with deviation to the full CFD solution approximately of ±7.1%.

Paper ID	**Track**	**Title**
13-532	Track 1	COMPARATIVE STUDY OF ROE, RHLL AND RUSANOV FLUXES FOR SHOCK-CAPTURING SCHEMES

Abstract Computational Fluid Dynamics provides approximations through iterative calculation to simulate and predict the solution and has been extensively used in the investigation of shockwave. The paper aims to compare the capability of Roe, RHLL and Rusanov flux function in capturing shock phenomena. Two cases have been presented in the paper; one-dimensional case and two-dimensional cases. In general, all the considered flux function capable to capture general pattern of the shockwave

phenomena. However, the results shows that it is important to pay great attention to the diffusive component of the fluxes. Strong diffusive component of the flux will helps to bring stability in the solution but at the same time prevent the fluxes to capture small changes of the fluid properties.

Paper ID	**Track**	**Title**
13-548	Track 1	THE EFFECTS OF JET-MAINSTREAM VELOCITY RATIO ON FLOW CHARACTERISTICS AND HEAT TRANSFER ENHANCEMENT ON DOWNSTREAM SURFACE OF OPEN FLOW

Abstract The aim of this research was to study the effects of jet-mainstream velocity ratio on flow and heat transfer characteristics on downstream surface of open flow. The jet from pipe nozzle with inner diameter of D=14 mm was injected perpendicularly to open surface. The surface was blown by mainstream with uniform velocity profile. The velocity ratio (jet to mainstream velocity) was varied at VR=0.25 and 3.5 by fixing velocity of mainstream at 10 m/s. For heat transfer measurement, a thin foil technique was used to measure temperature distributions by using infrared camera, and flow characteristics were simulate by using a computational fluid dynamics (CFD) with commercial software (ANSYS, Fluent, V.15.0). The results showed that the enhancement of heat transfer along downstream direction for the case of VR=0.25 was from the effect of jet stream whereas for the case of VR=3.5 was from the effect of mainstream.

Paper ID	**Track**	**Title**
13-562	Track 1	PERFORMANCE MEASUREMENT OF A NEW CONCEPT RECIPROCATING PISTON EXPANDER (RPE) USING A NEWLY DEVELOPED SMALL-SCALE DYNAMOMETER UNIT

Abstract This paper presents the progress of a small-scale dynamometer prototype development for performance measurement of a reciprocating piston expander (RPE). Since the available dynamometer systems in the market are limited to specific applications that require for the customization, their price normally very expensive. Since the current study on the RPE required a dynamometer unit, therefore, a new and cheaper dynamometer prototype that was suitable for RPE application has been developed. Using air as RPE working fluid, a case study has been carried out to measure its performance at different inlet fluid conditions, i.e., within 20℃−140℃ and 3−5 bars. The results observed that the performance of RPE was proportionally increased to the increase of inlet fluid pressure and temperature. The maximum brake power produced was 27 Watt when the RPE operated at 140℃, 5 bars and the speed of 820 rpm. It also revealed that the changes in the pressure of inlet fluid can give significant change on the performance of the RPE due to its direct relation to the RPE actual rotating force. Although the RPE and dynamometer seems good being adapted to each other, both of them require some improvements to ensure both systems well operated and reliable.

Paper ID	**Track**	**Title**
13-564	Track 1	NUMERICAL ANALYSIS ON CENTRIFUGAL COMPRESSOR WITH MEMBRANE TYPE DRYER

Abstract Moisture content is a common phenomenon in industrial processes especially in oil and gas industries. This contaminant has a lot of disadvantages which can lead to mechanical failure DEC (Deposition, Erosion & Corrosion) problems. To overcome DEC problem, this study proposed to design a centrifugal compressor with a membrane type dryer to reduce moisture content of a gas. The effectiveness of such design has been analyzed in this study using Computational Fluid Dynamics (CFD) approach. Numerical scheme based on multiphase flow technique is used in ANSYS Fluent software to evaluate the moisture content of the gas. Through this technique, two kind of centrifugal compressor, with and without membrane type dryer has been tested. The results show that the effects of pressure on dew point temperature of the gas change the composition of its moisture content, where high value lead more condensation to occur. However, with the injection of cool dry gas through membrane type dryer in the centrifugal compressor, the pressure and temperature of moisture content as well as mass fraction of H_2O in centrifugal compressor show significant reduction.

Paper ID	**Track**	**Title**
13-566	Track 1	COMPUTATION OF TABS AND WING GENERATOR MODIFICATION ON TEMPERATURE DISTRIBUTION AND FLOW OF FREE JET

Abstract The mixing process is important improve the performance of jet-engineering equipment. The purpose research, aim to study temperature distribution might be simulate mixing layer of jet flow with surrounding fluid. In this study measure the temperature at the position the jet exit. Z =1D, 2D, 4D and 6D. Calculate the temperature coefficient and a computational fluid dynamic (CFD) using ANSYS, Fluent (V.15.0) was selected to this computation. The investigation model was jets discharging from pipe nozzle installing wing generator having an inner diameter d=28.15 mm and wall fitted to the outlet of the jet. Promoters were attached at the nozzle exit. 4 types of turbulence promoters were triangular tab with tip angle of 45o and 90o and vortex generators with attack angle of 45o and 60o. In addition, the conventional pipe nozzle was studied to comparing as base results and modifications. Installing at 2 and 4 positions give the measuring temperature. The Reynolds number of test fluid was constant at Re=29,500. Increase the strength of jet mixing was found the case of vortex generator installed with 4 positions, attack angle of 60o gives the strongest mixing in jet when comparing with the other cases.

Paper ID	**Track**	**Title**
13-573	Track 1	THERMAL RADIATION HEAT TRANSFER IN PARTICIPATING MEDIA BY FINITE VOLUME DISCRETIZATION USING COLLIMATED BEAM INCIDENCE

Abstract The main objective of this paper is to study the heat transfer rate of thermal radiation in participating media. For that, a generated collimated beam has been passed through a two dimensional slab model of flint glass with a refractive index 2. Both Polar and azimuthal angle have been varied to generate such a beam. The Temperature of the slab and snells law has been validated by (RTE)Radiation Transfer Equation in OpenFOAM(Open Field Operation and Manipulation), a CFD software which is the major computational tool used in Industry and research applications where the source code is modified in which radiation heat transfer equation is added to the case and different radiation heat transfer models are utilized. This work concentrates on the numerical strategies involving both transparent and participating media. Since(RTE) Radiation Transfer Equation is difficult to solve, the purpose of this paper is to use existing solver "buoyantSimpleFoam" to solve radiation model in the participating media by compiling the source code to obtain the heat transfer rate inside the slab by varying the Intensity of radiation. The FVM(finite volume Method) is applied to solve the RTE(Radiation Transfer Equation)governing the above said physical phenomena.

Paper ID	**Track**	**Title**
13-575	Track 1	NUMERICAL INVESTIGATION OF FLOW PARAMETERS FOR SOLID RIGID SPHEROIDAL PARTICLE IN A PULSATILE PIPE FLOW

Abstract Quantifying, forecasting and analysing the displacement rates of suspended particles are essential while discussing about blood flow analysis. Because blood is one of the major organs in the body, which enables transport phenomena, comprising of numerous blood cells. In order to model the blood flow, a flow domain was created and numerically simulated. Flow field velocity in the stream is solved utilizing Finite

Volume Method utilizing FVM unstructured solver. In pulsatile flow, the effect of parameters such as average Reynolds number, tube radius, particle size and Womersley number are taken into account. In this study spheroidal particle trajectory in axial direction is simulated at different values of pulsating frequency including 1.2 Hz, 3.33 Hz and 4.00 Hz and various densities including 1005 kg/m^3 and 1025 kg/m^3 for the flow domain .The analysis accomplishes the interaction study of blood constituents for different flow situations which have applications in diagnosis and treatment of cardio vascular related diseases.

Paper ID	**Track**	**Title**
13-630	Track 1	PURIFICATION OF CRUDE GLYCEROL FROM TRANSESTERIFICATION REACTION OF PALM OIL USING DIRECT METHOD AND MULTISTEP METHOD

Abstract Crude glycerol which produced from transesterification reaction has limited usage if it does not undergo purification process. It also contains excess methanol, catalyst and soap. Conventionally, purification method of the crude glycerol involves high cost and complex processes. This study aimed to determine the effects of using different purification methods which are direct method (comprises of ion exchange and methanol removal steps) and multistep method (comprises of neutralization, filtration, ion exchange and methanol removal steps). Two crude glycerol samples were investigated; the self-produced sample through the transesterification process of palm oil and the sample obtained from biodiesel plant. Samples were analysed using Fourier Transform Infrared Spectroscopy, Gas Chromatography and High Performance Liquid Chromatography. The results of this study for both samples after purification have showed that the pure glycerol was successfully produced and fatty acid salts were eliminated. Also, the results indicated the absence of methanol in both samples after purification process. In short, the combination of 4 purification steps has contributed to a higher quality of glycerol. Multistep purification method gave a better result compared to the direct method as neutralization and filtration steps helped in removing most excess salt, fatty acid and catalyst.

Paper ID	**Track**	**Title**
13-695	Track 1	COMPUTATIONAL MODELLING OF FLOW AND TIP VARIATIONS OF AORTIC CANNULAE IN CARDIOPULMONARY BYPASS PROCEDURE

Abstract Aortic cannulation has been the gold standard for maintaining cardiovascular function during open heart surgery while being connected onto the heart lung machine. These cannulation produces high velocity outflow which may lead to adverse effect on patient condition, especially sandblasting effect on aorta wall and blood cells damage. This paper reports a novel design that was able to decrease high velocity outflow. There were three design factors of that was investigated. The design factors consist of the cannula type, the flow rate, and the cannula tip design which result in 12 variations. The cannulae type used were the spiral flow inducing cannula and the standard cannula. The flow rates are varied from three to five litres per minute (lpm). Parameters for each cannula variation included maximum velocity within the aorta, pressure drop, wall shear stress (WSS) area exceeding 15 Pa, and impinging velocity on the aorta wall were evaluated. Based on the result, spiral flow inducing cannulae is proposed as a better alternatives due to its ability to reduce outflow velocity. Meanwhile, the pressure drop of all variations are less than the limit of 100 mmHg, although standard cannulae yielded better result. All cannulae show low reading of wall shear stress which decrease the possibilities for atherogenesis formation. In conclusion, as far as velocity is concerned, spiral flow is better compared to standard flow across all cannulae variations.

Paper ID **Track** **Title**

13-697 Track 1 EFFECT OF TIP CLEARANCE ON WALL SHEAR STRESS OF ANAXIAL LVAD

Abstract Wall shear stress is a crucial parameter used for blood damage analysis, and typically a value of 400 Pa is set as an upper limit. Tip clearance is a major factor contributing to hemolysis and pump efficiency. In this study, different tip gap configurations are used to analyse the wall shear stress developed on the blade surface of a constant thickness blade design, and a varying thickness blade design using CFD analysis. For a particular geometry, as the clearance gap reduces, the high wall shear stress area is found to extend, whereas volume owing over the high wall shear stress span decreases. For both designs, the optimum clearance gap is iteratively attained, keeping the maximum WSS as a limiting factor. Thus a better pump designs is obtained, whose leakage flow patterns are lower than that of the initial design, hence leading to higher pump efficiency.

Paper ID **Track** **Title**

13-703 Track 1 STUDY OF CONVERSION OF AMMONIA FROM UREA WATER SOLUTION DROPLETS USING CFD

Abstract This paper deals with the numerical analysis of the spray behaviour of urea water solution (UWS) droplets used in Selective catalytic reduction (SCR) process. The study uses the ANSYS- Fluent-14 as a basic framework for numerical simulation. The evaporation modelling is based on multi-component droplet evaporation approach along with the consideration of Stefan flow. The urea decomposition treated as direct thermolysis approach, where the modelling is based on the single kinetic rate approach by the proper fitting of pre- exponent. This developed model was used for spray simulation of UWS droplet evaporation to determine the urea to ammonia conversion efficiency. The obtained spray simulation results were compared with the available experimental data. The comparison shows the pre- exponent developed in the study is suitable for getting promising results for the spray simulations of the urea-water solution of the direct thermolysis approach to determine the ammonia conversion efficiency.

Paper ID **Track** **Title**

13-715 Track 1 COMPUTATIONAL FLUID DYNAMIC (CFD)ANALYSIS FOR ALUDRA SR-10 UAV WITHPARACHUTE RECOVERY SYSTEM

Abstract Parachute recovery system recently use to replace belly landing method which require a large area and longer runway for landing purpose. Parachute use to slow down flying or falling objects like UAV to a safe landing by opening the canopy to increasing aerodynamic drag. This paper describe Computational Fluid Dynamic (CFD) analysis on ALUDRA SR-10 3-dimensional model with two different condition that is with parachute and the UAV itself. The objective of this analysis is to compare the aerodynamic characteristics of UAV during landing before and after parachute install. The computational simulation was carried out by using ANSYS 16.0 Fluent. The finding of this research show the increasing of drag coefficient from 0.01 to 1.57 as it parachute use to perform improving for aerodynamic characteristic of the UAV.

Paper ID **Track** **Title**

13-761 Track 1 COMPUTATION OF TABS AND WING GENERATOR MODIFICATION ON TEMPERATURE DISTRIBUTION AND FLOW OF FREE JET

Abstract The mixing process is important improve the performance of jet-engineering equipment. The purpose research, aim to study temperature distribution might be simulate mixing layer of jet flow with surrounding fluid.In this study measure the temperature at the position the jet exit. Z =1D, 2D, 4D and 6D. Calculate the temperature coefficient and a computational fluid dynamic (CFD) using ANSYS, Fluent (V.15.0) was selected to this computation. The investigation model was jets discharging from pipe nozzle installing wing generator having an inner diameter d=28.15 mm and wall fitted to the outlet of the jet. Promoters were attached at the nozzle exit. 4 types of turbulence promoters were triangular tab with tip angle of 45o and 90o and vortex generators with attack angle of 45o and 60o. In addition, the conventional pipe nozzle was studied to comparing as base results and modifications. Installing at 2 and 4 positions give the measuring temperature. The Reynolds number of test fluid was constant at Re=29,500. Increase the strength of jet mixing was found the case of vortex generator installed with 4 positions, attack angle of 60o gives the strongest mixing in jet when comparing with the other cases.

Paper ID	Track	Title
13-773	Track 1	OPTIMAL PISTON CREVICE STUDY IN A RAPID COMPRESSION MACHINE

Abstract

Multi-dimensional effects such as vortex generation and heat losses from the gas to the wall of the reactor chamber have been an issue to obtaining a reliable RCM data. This vortex initiates a flow in the relatively cold boundary layer, which may penetrate the core gas. This resulting non-uniformity of the core region could cause serious discrepancies and give unreliable experimental data. To achieve a homogenous temperature field, an optimised piston crevice was designed using CFD modelling (Ansys fluent). A 2-Dimensional computational moving mesh is assuming an axisymmetric symmetry. The model adopted for this calculation is the laminar flow model and the fluid used was nitrogen. To get the appropriate crevice volume suitable for the present design, an optimisation of the five different crevice volume was modelled which resulted to about 2-10% of the entire chamber volume. The use of creviced piston has shown to reduce the final compressed gas temperature and pressure in the reactor chamber. All the crevice volumes between 2-10% of the chamber volume adequately contained the roll up vortexes, but the crevice volume of 282 mm3 was chosen to be the best in addition to minimising the end gas pressure and temperature drop. The final pressure trace from experiment shows a reasonable agreement with the CFD model at compression and post compression stage.

Paper ID	Track	Title
14-456	Track 2	FLAME SPREAD BEHAVIOR OVER COMBUSTIBLE THICK SOLID OF PAPER, BAGASSE AND MIXED PAPER/BAGASSE

Abstract

Flame spread behavior on combustible solid is one of important research related to Fire Safety Engineering. Now, there are a lot of combustible solid composed from mixed materials. In this study, experiments have been conducted to investigate flame spread behavior over combustible solid composed by paper, bagasse and mixed paper/bagasse. Experimental data is captured by using video recording and examined flame spread shape and rate. From the results obtained, shows that the different materials produce different flame spread shape and rate. Different flame shape is seen between all types of samples. Flame spread rate of 100% paper is faster than the one of 100% baggase. Based m the result, it is also inferred that the material composition can be influenced on the flame spread shape and flame spread rate of mixed paper/bagasse.

Paper ID	Track	Title
14-461	Track 2	EFFECT OF GROUND PROXIMITY ON THE FLOW OVER STOL CH750 MULTI-ELEMENT AIRFOIL

Abstract

The paper presents the influence of ground distance on aerodynamic characteristics of the flow over short take-off and landing STOL CH750 aircraft multi-element airfoil at various flap angle of deflection. The angle of attack is kept constant throughout thecomputations. ANSYS is used for the grid generation and the computational calculation.Further investigation suggests the best range of deflection angle where

the aerodynamic performance of the airfoil increases in the presence of ground. Furthermore, a cushion of high pressure air region between the airfoil pressure side and the ground surface also emerges.

Paper ID	**Track**	**Title**
14-467	Track 2	FLOW BEHAVIOUR IN NORMAL AND MENIERE'S DISEASE OF ENDOLYMPHATIC FLUID INSIDE THE INNER EAR

Abstract Meniere's disease is a rare disorder that affects the inner ear which might be more severe if not treated. This is due to the fluctuating pressure of the fluid in the endolymphatic sac which causing the vestibular membrane to start stretching. However, the pattern of the flow re-circulation in endolymphatic sac is still not fully understood. Thus, this study aims to investigate the behavior of flow re-circulation for both normal and distended endolymphatic sac. Three dimensional model of endolymphatic sac is modeled using computer aided design (CAD) software. Two different shapes of endolymphatic sac is also considered in this study; model A and B. the normal condition of endolymphatic sac is also modeled as a basis of comparison. Computational fluid dynamics (CFD) method is used to predict the behavior of flow in endolymphatic sac which might be influenced instability system. Endolymphatic sac fluids are assumed as a Newtonian fluid, incompressible flow and no-slip conditions. From the observed, flow re-circulation has been formed for all shapes of endolymphatic models. However, both model A and B show the increase of pressure distributions in endolymphatic wall about 15 percent and 30 percent, respectively as compared to normal endolyphatic model. In conclusion, model B of endolymphatic model shows the highest distribution of pressure and velocity which can be predicted more severe instability in vestibular system.

Paper ID	**Track**	**Title**
14-501	Track 2	RECENT ADVANCES IN FLUIDIZED BED DRYING

Abstract Fluidized bed drying are very well known to yield high heat and mass transfer and hence adopted to many industrial drying processes particularly agricultural products. In this paper, recent advances in fluidized bed drying were reviewed and focus is given to the drying related to the usage of Computational Fluid Dynamics (CFD). It can be seen that usage of modern computational tools such as CFD helps to optimize the fluidized bed dryer design and operation for lower energy consumption and thus better thermal efficiency. Among agricultural products that were reviewed in this paper were oil palm frond, wheat grains, olive pomace, coconut, pepper corn and millet.

Paper ID	**Track**	**Title**
14-517	Track 2	NUMERICAL SIMULATION OF TANGENTIAL INLET CONFIGURATION FOR PLENUM CHAMBERS

Abstract Swirling fluid motion in enclosed chambers was studied using Computational Fluid Dynamics. Using the tangential inlet configuration as the basic design, 3 swirl generator models was created using Computer Aided Design software. The aim was to see whether a modified design from the original configuration could provide a reduction in the backflow effect that is constantly present in swirling flows. Simulations show that swirl generator inlets at different angles from the original tangential position results in a change in velocity profiles across the flow cross section. From the simulations run, it was found that the swirl generator model with inlets set to 45 degrees produced the least backflow compared to other models.

Paper ID	**Track**	**Title**
14-520	Track 2	PRESSURE RECOVERY PERFORMANCE OF 2-D TURNING DIFFUSER BY VARYING AREA RATIOS AND INFLOW REYNOLDS NUMBERS

Abstract The paper aims to investigate the effects of varying area ratio, AR = 1.2 and 4.0 and inflow Reynolds number, Rein = 5.478 x 10^4 - 1.547 x10^5 on the performance of 90o two-dimensional turning diffuser. The optimum configuration area ratio and Rein to produce good pressure recovery is determined. The rig was developed to produce fully developed entrance flow by adopting arrangement of mesh net and sufficient hydrodynamic entrance length, Lh, turb= 28 Dh. Digital manometer was used to measure the inlet and outlet static pressures and Particle Image Velocimetry (PIV) to visualize the flow structure. The present results were compared with empirical solution of Asymptotic Computational Fluid Dynamics (ACFD) results to give acceptable deviation of ±7.4%. The AR=4.0 produces pressure recovery 20% more than AR=1.2 when applies low Rein<1.40 x 10^5. However, it is subjected to severe flow separation and circulation at Rein>1.40 x 10^5 that considerably disturbs the recovery. Therefore, turning diffuser of AR=1.2 is optimum applied for high Rein>1.40 x 10^5.

Paper ID	**Track**	**Title**
14-526	Track 2	STUDY ON CHARACTERISTICS OF GASOLINE SPRAY INJECTION

Abstract Gasoline injectors are one of the fuel injection components in gasoline engine. Nowadays, fuel injection systems are not only limited to vehicle but have been used widely on motorcycles. The injectors are playing major role in order to provide exact amount of fuel as needed for a stoichiometric combustion. Stoichiometric combustion will produce a good engine performance and emits lower exhaust emissions. It is well known that electronic fuel-injection system has improved fuel consumption, produced high power, and low emissions characteristics compared to conventional fueling system. This study was carried out to characterize qualitative and quantitative of the gasoline spray formation from two different fuel injector which are two-hole and four-hole orifice injector in terms of spray angle, penetration and size of droplet. The fuel injector was taken from motorcycles which are Modenas 118EFI and Honda Wave 125i.

Paper ID	**Track**	**Title**
14-527	Track 2	STUDY ON AIRFLOW CHARACTERISTICS OF REAR WING OF F1 CAR

Abstract The paper aims to investigate CFD simulation is carried out to investigate the airflow along the rear wing of F1 car with Reynold number of 3 x 106 and velocity, u = 43.82204 m/s. The analysis was done using 2-D model consists of main plane and flap wing. Both of the aerofoil is placed inside a box of 350mm long and 220mm height according to regulation set up by FIA. The parameters for this study is the thickness and the chord length of the flap wing aerofoil. The simulations were performed by using FLUENT solver and k-kl-omega model. The wind speed is set up to 43 m/s that is the average speed of F1 car when cornering. This study uses NACA 2408, 2412, and 2415 for the flap wing and be50 for the main plane. Each cases being simulated with a gap between the aerofoil of 10mm and 50mm when the DRS is activated. Grid independence test and validation was conducted to make sure the result that has been obtained is acceptable. The goal of this study is to investigate aerodynamic behavior of airflow around the rear wing as well as to see how the thickness and the chord length of flap wing influence the airflow around the rear wing. The results shows that increasing in thickness of the flap wing aerofoil will decreases the downforce. The results also show that although the short flap wing has lower downforce than the big flap wing, but the drag force can be significantly reduced as the short flap wing has more change in angle of attack when it is activated. Therefore, the type of aerofoil for the rear wing should be decided

according to the circuit track so that it can be fully optimized.

Paper ID	Track	Title
14-530	Track 2	NUMERICAL STUDY OF CANISTER FILTERS WITH ALTERNATIVES FILTER CAP CONFIGURATIONS

Abstract Air filtration system and filter play an important role in getting a good quality air into turbo machinery such as gas turbine. The filtration system and filter has improved the quality of air and protect the gas turbine part from contaminants which could bring damage. During separation of contaminants from the air, pressure drop cannot be avoided but it can be minimized thus helps to reduce the intake losses of the engine [1]. This study is focused on the configuration of the filter in order to obtain the minimal pressure drop along the filter. The configuration used is the basic filter geometry provided by Salutary Avenue Manufacturing Sdn Bhd. and two modified canister filter cap which is designed based on the basic filter model. The geometries of the filter are generated by using SOLIDWORKS software and Computational Fluid Dynamics (CFD) software is used to analyse and simulates the flow through the filter. In this study, the parameter of the inlet velocity are 0.032 m/s, 0.063 m/s, 0.094 m/s and 0.126 m/s. The total pressure drop produce by basic, modified filter 1 and 2 is 292.3 Pa, 251.11 Pa and 274.7 Pa. The pressure drop reduction for the modified filter 1 is 41.19 Pa and 14.1% lower compared to basic filter and the pressure drop reduction for modified filter 2 is 17.6 Pa and 6.02% lower compared to the basic filter. The pressure drops for the basic filter are slightly different with the Salutary Avenue filter due to limited data and experiment details. CFD software are very reliable in running a simulation rather than produces the prototypes and conduct the experiment thus reducing overall time and cost in this study.

Paper ID	Track	Title
14-536	Track 2	AIR BEARING DEVELOPMENT USING COMPUTATIONAL FLUID DYNAMICS (CFD)

Abstract Air bearing, aerostatic bearing is a bearing that operate by floating the shaft with supplying the air pressure to the air film. Air film is a clearance between the air bearing and the shaft. The air pressure occupied the air film and the flow of air pressure causes the shaft to float. The purpose of the study was to study the load characteristic of air bearing. The four different models of air bearing was developed with vary some of bearing parameter to see the effect of it towards bearing performance. The study was carried out through simulation software, ANSYS 16.1.The turbulence model that been using was k-epsilon. The supply pressure varied at 0.3MPa, 0.4MPa, 0.5MPa and 0.6MPa. The simulation was divided into two major parts which is static condition and dynamic condition. Body sizing meshing had applied to the geometry in order to get a proper meshing skewness. The study proved that the air bearing parameter affected the air bearing load characteristic. Besides that, air bearing load characteristic also affected by the rotation speed of the rotor system. The findings of study shows that air bearing with diameter of 18mm have a potential of carrying load capacity until 60N.

Paper ID	Track	Title
14-549	Track 2	CFD SIMULATION OF FATTY ACID METHYL ESTER PRODUCTION IN BUBBLE COLUMN REACTOR

Abstract Non-catalytic transesterification are one of the method that was used to produce the fatty acid methyl ester (FAME) by blowing superheated methanol bubbles continuously into the vegetable oil without using any catalyst. This research aimed to simulate the production of FAME from palm oil in a bubble column reactor. Computational Fluid Dynamic (CFD) simulation was used to predict the distribution of fatty acid methyl ester and other product in reactor. The fluid flow and component of concentration along the reaction time was investigated and the effects of reaction temperature (523 K and 563 K) on the non-catalytic transesterification process has been examined. The study was carried out using ANSYS CFX 17.1. The finding from the study shown that increasing the temperature leads to higher amount of fatty acid methyl ester can be produced in shorter time. On the other hand, concentration of the component such as triglyceride (TG), glycerol (GL) and fatty acid methyl ester (FAME) can be known when reaching the optimum condition.

Paper ID	Track	Title
14-551	Track 2	DEVELOPMENT OF LOW COST CLUSTER COMPUTER FOR NUMERICAL ANALYSIS

Abstract In this study, two units of computer were successfully networked together to form a small scale cluster. Each of the processor involved are multicore processor which has four cores in it, thus made this cluster to have eight processors. Here, the cluster incorporate Ubuntu 14.04 LINUX environment with MPI implementation (MPICH2). Two main tests were conducted in order to test the cluster, which is communication test and performance test. The communication test was done to make sure that the computers are able to pass the required information without any problem and were done by using simple MPI Hello Program where the program written in C language. Additional, performance test was also done to prove that this cluster calculation performance is much better than single CPU computer. In this performance test, four tests were done by running the same code by using single node, 2 processors, 4 processors, and 8 processors. The result shows that with additional processors, the time required to solve the problem decrease. Time required for the calculation shorten to half when we double the processors. To conclude, we successfully develop a small scale cluster computer using common hardware which capable of higher computing power when compare to single CPU processor, and this can be beneficial for research that require high computing power especially numerical analysis such as finite element analysis, computational fluid dynamics, and computational physics analysis.

Paper ID	Track	Title
14-552	Track 2	THREAD ANGLE DEPENDENCY ON FLAME SPREAD SHAPE OVER KENAF/POLYESTER COMBINED FABRIC

Abstract Understanding flame spread behavior is crucial to Fire Safety Engineering. It is noted that the natural fiber exhibits different flame spread behavior than the one of the synthetic fiber. This different may influences the flame spread behavior over combined fabric. There is a research has been done to examined the flame spread behavior over kenaf/polyester fabric. It is seen that the flame spread shape is dependent on the thread angle dependency. However, the explanation of this phenomenon is not described in detail in that research. In this study, explanation about this phenomenon is given in detail. Results show that the flame spread shape is dependent on the position of synthetic thread. For thread angle, θ = 0°, the polyester thread is breaking when the flame approach to the thread and the kenaf thread tends to move to the breaking direction. This behavior produces flame to be ‘V’ shape. However, for thread angle, θ = 90°, the polyester thread melts while the kenaf thread decomposed and burned. At this angle, The distance between kenaf threads remains constant as flame approaches.

Paper ID	Track	Title
14-553	Track 2	STUDY OF THE FLUID FLOW PATTERN IN A BUBBLE COLUMN REACTOR FOR BIODIESEL PRODUCTION

Abstract The applications of bubble columns are very important as multiphase reactors in process industry. The advantages of bubble column are low operating cost and maintenance due to the compactness and no moving part. It is important to

understand the nature of hydrodynamics and operational parameters to characterize their operation including gas superficial velocity, bubble rise velocity, etc., to do the design and scale-up process. Computational fluid dynamics (CFD) can be used to evaluate the performance of bubble column at lower cost compared to experimental setup. In this work, a commercial CFD software, FLUENT 14.0 was used for modeling of gas-liquid flow in a bubble column. Multiphase simulations were performed using an Eulerian-Eulerian two-fluid model. In this study, the simulation were conducted by using three different temperature, which are 523 K, 543 K and 563 K. The CFD result predicts the turbulent kinetic energy, gas hold-up and the liquid velocity fairly well, although the results seem to suggest that further improvement on the interfacial exchange models and possibly further refinement on the two-fluid modeling approaches are necessary especially for the liquid velocity and turbulent kinetic energy.

Paper ID	**Track**	**Title**
14-554	Track 2	EFFECT OF SPARK PLUG AND FUEL INJECTOR LOCATION ON MIXTURE STRATIFICATION IN A GDI ENGINE - A CFD ANALYSIS

Abstract The mixture preparation in gasoline direct injection (GDI) engines operating at stratified condition plays an important role in deciding the combustion, performance and emission characteristics of the engine. In a wall-guided GDI engine, with a late fuel injection strategy, piston top surface is designed in such a way that the injected fuel is directed towards the spark plug to form a combustible mixture at the time of ignition. In addition, in these engines, location of spark-plug and fuel injector, fuel injection pressure and timing are also important to create a combustible mixture near the spark plug. Therefore, understanding the mixture formation under the influence of the location of the spark plug and fuel injector is essential for the optimization of the engine parameters.

Paper ID	**Track**	**Title**
14-555	Track 2	EFFECT OF PISTON PROFILE ON PERFORMANCE AND EMISSION CHARACTERISTICS OF A GDI ENGINE WITH SPLIT INJECTION STRATEGY – A CFD STUDY

Abstract Gasoline direct injection (GDI) engines have gained popularity in the recent times because of lower fuel consumption and exhaust emissions. But in these engines, the mixture preparation plays an important role which affects combustion, performance and emission characteristics. The mixture preparation in turn depends mainly upon combustion chamber geometry. Therefore, in this study, an attempt has been made to understand the effect of piston profile on the performance and emission characteristics in a GDI engine. The analysis is carried out on a four-stroke wall guided GDI engine using computational fluid dynamics (CFD). The spray breakup model used is validated with the available experimental results from the literature to the extent possible. The analysis is carried out for four piston profiles viz., offset pentroof with offset bowl (OPOB), flat piston with offset bowl (FPOB), offset pentroof with offset scoop (OPOS) and inclined piston with offset bowl (IPOB) fitted in an engine equipped with a six-hole injector with the split injection ratio of 30:70. All the CFD simulations are carried out at the engine speed of 2000 rev/min., with the overall equivalence ratio of about 0.65±0.05. The performance and emission characteristics of the engine are compared while using the above piston profiles. It is found that, the OPOB piston is preferred compared to that of the other pistons because it has better in-cylinder flow, IMEP and lower HC emissions compared to that of other pistons.

Paper ID	**Track**	**Title**
14-557	Track 2	POTENTIAL REDUCTION OF ENERGY CONSUMPTION IN PUBLIC UNIVERSITY LIBRARY

Abstract Efficient electrical energy usage has been recognized as one of the important factor to reduce cost of electrical energy consumption. Various parties have been emphasized about the importance of using electrical energy efficiently. Inefficient usage of electrical energy usage lead to biggest factor increasing of administration cost in Universiti Tun Hussein Onn Malaysia. With this in view, a project to investigate potential reduction electrical energy consumption in Universiti Tun Hussein Onn Malaysia was carried out. In this project, a case study involving electrical energy consumption of Perpustakaan Tunku Tun Aminah was conducted. The scopes of this project are to identify energy consumption in selected building and to find the factors that contributing to wastage of electrical energy. The MS1525:2001, Malaysian Standard - Code of practice on energy efficiency and use of renewable energy for non-residential buildings was used as reference. From the result, 4 saving measure had been proposed which is change type of the lamp, install sensor, decrease the number of lamp and improve shading coefficient on glass. This saving measure is suggested to improve the efficiency of electrical energy consumption. Improve of human behavior toward saving energy measure can reduce 10% from the total of saving cost while on building technical measure can reduce 90% from total saving cost.

Paper ID	**Track**	**Title**
14-561	Track 2	*COMPUTATIONAL FLUID DYNAMICS SIMULATION OF PRESSURE AND VELOCITY DISTRIBUTION INSIDE MENIERE'S DISEASED VESTIBULAR SYSTEM*

Abstract Meniere's disease or known as endolymphatic hydrops is an incurable vestibular disorder of the inner ear. This is due to the excessive fluid build-up in the endolymphatic sac which causing the vestibular endolymphatic membrane to start stretching. Although this mechanism has been widely accepted as the likely mechanism of Meniere's syndrome, the reason for its occurrence remains unclear. Thus, the aims of this study to investigate the critical parameters of fluid flow in membranous labyrinth that is influencing instability of the vestibular system. In addition, to visualise the flow behaviour between a normal membranous labyrinth and dilated membranous labyrinth in Meniere's disease in predicting instability of the vestibular system. Three dimensional geometry of endolymphatic sac is obtained from Magnetic Resonance Images (MRI) and reconstructed using commercial software. As the basis of comparison, the two different model of endolymphatic sac is considered in this study which are normal membranous labyrinth for model I and dilated membranous labyrinth for model II. Computational fluid dynamics (CFD) method is used to analyse the behaviour of pressure and velocity flow in the endolymphatic sac. The comparison is made in term of pressure distribution and velocity profile. The results show that the pressure for dilated membranous labyrinth is greater than the normal membranous labyrinth. Due to abnormally pressure in the vestibular system it leads to the increasing value of the velocity at dilated membranous labyrinth while at the normal membranous labyrinth the velocity values decreasing. By changing the parameters which are pressure and velocity can significantly affect to the instability of vestibular system for Meniere's disease.

Paper ID	**Track**	**Title**
14-570	Track 2	NUMERICAL INVESTIGATION OF A THICK PLATE RESTRICTION ORIFICE ON THE PRESSURE DROP PERFORMANCE

Abstract This paper presents a numerical study on the thick plate restriction orifice on the pressure drop performance due to various orifice ratio, B which is defined as pipe internal diameter, D over the constriction diameter, d. The restriction orifice was investigated using commercial software package namely, ANSYS. The restriction orifice was modelled using built-in modeler and simulated using Fluent module. The orifice ratio, B was varied in the range of 0.5 to 0.75. Various flow velocities were applied from 50 to 420 m/s. The fluid flow in the constriction was hydrocarbon in vapour phase. The preliminary results of discharge coefficients were compared with literature and theoretical values between a sharp-edged and thick plate orifice to show a consistent trend. The results yielded that as the Reynolds number, Re increases, the pressure drop performance increases exponentially. This is more prominent at high Re of 8.75×106 where the pressure drop increases by 63% from the baseline of Re number, 1.08×106 based on orifice ratio of 0.75. For a hydrocarbon with low rheological properties and high Re number, the orifice ratio, B is best at higher range of 0.75 which show a pressure drop of 0.18 MPa.

Paper ID	Track	Title
14-611	Track 2	A STUDY ON A SWIRL TYPE WITH MULTI-HOLE NOZZLE FUEL INJECTOR FOR A PFI SYSTEM TO IMPROVE AIR-FUEL MIXTURE FORMATION

Abstract This paper described the development and simulation works that were conducted on an injector that has a combined swirl spray and multi-hole injection functions. It was expected that those two functions would provide beneficial effects to the fuel injector, in order to provide better fuel and air mixing, especially for Port Fuel Injection (PFI) application. The combination of both functions was aimed for achieving high quality spray formation, which might be useful in cases when the combustion time is limited. Swirl spray pattern is known for fine droplets production with relatively smaller SMD and bigger coverage area in the combustion chamber. On the other hand, a multi-hole injector has the advantages of simplicity in manufacturing and operation. Preliminary results on the simulation works that were done separately, demonstrated the ability of the injector's design to produce swirl spray as well as injecting near-symmetrical sprays from the four holes at the injector's end. Nevertheless, these findings must be improved further in the future, especially on simulating the benefits of having a combined swirl and multi-hole functions inside a single injector. Furthermore, detailed assessment on manufacturability and cost effectiveness of this injector must also be addressed.

Paper ID	Track	Title
14-671	Track 2	NEW APPROACH TO REDUCING WATER CONSUMPTION IN COMMERCIAL KITCHEN HOOD

Abstract Water mist sprays are used in wide range of application. However it is depend to the spray characteristic to suit the particular application. The modern commercial kitchen hood ventilation system was adopted with the water mist nozzle technology as an additional tool to increase the filtration efficiency. However, low level of filtration effectiveness and high water consumption were the major problems among the Commercial Kitchen Ventilation specialist. Therefore, this study aims to develop a new mist spray technology to replacing the conventional KSJB nozzle. At the same time, an appropriate recommended location to install the nozzle in kitchen hood system was suggested. An extensive simulation works were carried out to observe the spray characteristics, ANSYS (FLUENT) was used for simulation wise. In the case of nozzle studies, nozzles were tested at 1 bar pressure of water and air. In comparison with conventional nozzles configuration, this new approach suggested nozzle configuration was reduce up to 50% of water consumption, which by adopted 3 numbers of nozzles instead of 6 numbers of nozzles in the commercial kitchen hood system. Therefore, this nozzle will be used in industry for their benefits of water consumption, filtration efficiency and reduced the safety limitations.

Paper ID	Track	Title
14-679	Track 2	CFD SIMULATION OF FLOW THROUGH AN ORIFICE PLATE

Abstract In this present paper, the commercial Computational Fluid Dynamics (CFD) is used to predict the flow features in the orifice flow meter. Outcomes of the CFD simulations in terms of profiles of velocity and pressure are discussed in detail. It is observed that the flow is jet-like flow in the core region and the presence of recirculation, reattachment and shear layer regions flow features downstream the orifice. The location of vena-contracta was also estimated from CFD simulations. These results are consistent with other published data.

Paper ID	Track	Title
14-688	Track 2	INVESTIGATION OF WIND BEHAVIOUR AROUND HIGH-RISE BUILDINGS

Abstract A study on the investigation of wind behaviour around the high-rise buildings is done through an experiment using a wind tunnel and computational fluid dynamics. High-rise buildings refer to buildings or structures that have more than 12 floors. Wind is invisible to the naked eye; thus, it is hard to see and analyse its flow around and over buildings without the use of proper methods, such as the use of wind tunnel and computational fluid dynamics software.The study was conducted on buildings located in Presint 4, Putrajaya, Malaysia which is the Ministry of Rural and Regional Development, Ministry of Information Communications and Culture, Ministry of Urban Wellbeing, Housing and Local Government and the Ministry of Women, Family, and Community by making scaled models of the buildings. The parameters in which this study is conducted on are, four different wind velocities used based on the seasonal monsoons, and wind direction. ANSYS Fluent workbench software is used to compute the simulations in order to achieve the objectives of this study. The data from the computational fluid dynamics are validated with the experiment done through the wind tunnel. From the results obtained through the use of the computation fluid dynamics, this study can identify the characteristics of wind around buildings, including boundary layer of the buildings, separation flow, wake region and etc. Then analyses is conducted on the occurance resulting from the wind that passes the buildings based on the velocity difference between before and after the wind passes the buildings.

Paper ID	Track	Title
14-711	Track 2	NUMERICAL STUDY OF FLOW PAST A SOLID SPHERE AT HIGH REYNOLDS NUMBER

Abstract The present study gives a detail description of separation flow and its effect under high Reynolds number. The unsteady three dimensional flow simulation around sphere using numerical simulation computational fluid dynamics for high Reynolds number between 300 000 < Re < 600 000 is discussed. The separation angle and drag coefficient are also presented. The results show that the increasing Reynolds number affecting the formation of vortex shedding, separation point and drag coefficient. The agreement was good, confirming the reliability of the predicted data from computational fluid dynamic in flow analysis around sphere at high Reynolds number.

Paper ID	Track	Title
15-468	Track 3	A NUMERICAL ANALYSIS OF FLAT FAN AERIAL CROP SPRAY

Abstract Spray drift mitigation, in the agriculture aerial spraying literature, and spray quality in the application of plant protection products, still continues as two critical components in evaluating shareholder value. A study on off-target drift and ground deposit onto a 250 m strip were simulated through series of Computational Fluid

Dynamic (CFD) simulations. The drift patterns for evaporating droplets were released from a constant aircraft velocity at 30 m/s (60 mph) carrying 20 m swath width spray boom with 12 fan-type nozzles at released height from the ground ranging from 3.7 m to 4.7 m. Droplet trajectories are calculated from the given airspeed with a Lagrangian model for particle dispersion excluding any wind effect perturbation. The proposed CFD's model predictions agreed well with cited literatures for a wide range of atmospheric stability values. The results revealed that there is considerable increased in spray drift and droplets trajectories with the increased in spray released height. It suggested that a combination of low aircraft spray released height with low airspeed is essential to improve spray quality and maximizing uniform deposition on the target area are significant in minimizing spray drift risks.

Paper ID	**Track**	**Title**
15-542	Track 3	HEAT TRANSFER EFFICIENCY OF DIFFERENT COOLING AGENTS IN SHELL AND TUBE HEAT EXCHANGER

Abstract A shell and tube heat exchanger is simulated using CFD software. The simulation is based on the cooling process of the hot water in the tubes by using three different cooling agents. The agents are water, ethylene glycol and air. One simulation test is conducted for each cooling agent. The objective of the simulation is to investigate and compare the heat transfer efficiency of the three cooling agents. After that, the best cooling agent is chosen to be suggested for the application of heat exchanger in industry. The simulation is conducted by using solver ANSYS Fluent, which the model of the heat exchanger is sketched using Solid work Software (x64 edition.lnk). The model is imported to the mesh for meshing process, and as a result of the mesh, the shape of model consisted 806712 number of Tetrahedra, 26680 number of Hexahedra and 6960 wedges. Each face parts of the heat exchanger are named. Then, boundary conditions of the cell zones of the heat exchanger are set. Both hot water and the cooling agent for each test are set to have same velocity and pressure, which is 1 ms^(-1)(inlet) and 10000 Pa (outlet) respectively. The number of iterations used for the simulation is 500 times to ensure a better reading. As all the simulations for the three cooling agents are conducted, it is found that water cools hot water better than ethylene glycol and air. The analysis results show that water, ethylene glycol and air has the heat transfer rate of 55.72 kW, 55.495 kW and 55.465 kW respectively. Thus, water is proposed to be the most suitable cooling agent in the cooling system of heat exchanger.

Paper ID	**Track**	**Title**
15-545	Track 3	COMPARISON BETWEEN SEVEN, FIVE AND THREE TUBES OF HEAT EXCHANGER TEMPERATURE AND PRESSURE DROP

Abstract Heat exchangers has been used in many industries around the world as a cooling devices or heat engines. The heat exchanger heat transfer rate is the most essential parameters in the heat exchangers (Fakheri, 2008). This paper objectively conducts a case studies stimulate the pressure drops and temperature drops in tubes of each different numbers of tubes which is three, five and seven tubes in heat exchangers models using the Computational Fluid Dynamics methods with the help of ANSYS-Fluent as a simulation medium. The results are shown and parallel with the theories from the literature reviews. Results shows larger temperature drops and pressure drops on the third heat exchanger model with 7 tubes while the lesser temperature drops and pressure drops happens on first heat exchanger model with 3 tubes. The results positively fulfill the objective.

Paper ID	Track	Title
15-550	Track 3	FLOW CHARACTERISTIC OF WATER SPRINKLER

Abstract This study investigated fluid flow in water sprinkler. The water flow in a common rotatory external vane sprinkler is simulated with ANSYS CFX. Various changes are made to the original model include enlarging nozzle diameter from 4mm to 8mm, changing vane angle from 70 degrees to 45 degrees, and changing vane curvature angle from 10 to 5 degrees. The new models are simulated to investigate how the variation in geometry affects the flow of water and the performance of sprinkler head. Performance of water sprinkler is compared to original model in terms of watering distance, area of spray and velocity of water jet in air. The result of this study shows that enlarge the nozzle diameter have a positive effect on the velocity of water jet in air and the area covered by water jet but it drastically decreases the watering distance of sprinkler. Besides that, changing the angle of vane from 70 degrees to 45 degrees decrease the watering distance slightly and it concentrates the water into a fine jet that cover a small area. To reduce the effect, grooves can be added to the vane to increase the divergence of water spray. Reducing the angle of curvature from 10 degrees to 5 degrees improves the watering distance. The angle of curvature can be reduced more to increase the watering distance further.

Paper ID	Track	Title
15-556	Track 3	STUDY OF BUILDING ENERGY INDEX, BEI IN TUNKU TUN AMINAH LIBRARY, UTHM

Abstract As world getting modern, our beloved country, Malaysia also inseparable from advances forward of achieving a modern country in line with “Transformasi Nasional 50” or TN50. This ideology has affected the whole country as the energy demand become more intensive than before. The building sector in Malaysia among the major energy consumer as it monopoly almost 48% of the total energy usage. Unfortunately, this worldwide phenomenon also hit University Tun Hussein Onn, Malaysia (UTHM), as the rapid growth of UTHM has led to incline of electrical energy usage. This paper presents the findings from a study of building energy index, BEI of Tunku Tun Aminah Library building, to compare the BEI with suggested value and to propose several approaches to minimize energy consumption. To obtain basic information and data, the analysis performed based on the desktop data collection, field data collection, on-site survey and qualitative assessment of the building and its systems. Building Energy Index (BEI) used for comparing energy consumption in buildings for one year and measured in kilowatts hours divided by the gross floor area of the building in square meters. Based on BEI MS 1525:2007, the suggested value is 136 kWh/m2/year. It estimated the result will not satisfy the MS 1525 Malaysian Standard, but this result will able to create awareness of building energy index among building owner and users.

Paper ID	Track	Title
15-559	Track 3	HEAT TRANSFER EFFICIENCY ON DIFFERENT COOLING AGENTS IN SHELL AND TUBE HEAT EXCHANGER

Abstract Shell and tube heat exchanger has been used a decades to serve the energy conservation especially in heavy industry. The efficiency of the heat transfer is depending on the cooling agents being used in the heat exchanger. Thus, this study aims to investigate the heat transfer efficiency for different cooling agents. Three different cooling agents used in this study are water, ethylene glycol and air. One simulation test is conducted for each cooling agent. The objective of the simulation is to investigate and compare the heat transfer efficiency of the three cooling agents. After that, the best cooling agent is chosen to be suggested for the application of heat exchanger in industry. The simulation is conducted by using solver ANSYS Fluent, which the model of the heat exchanger is sketched using Solid work Software.

The model is imported to the mesh for meshing process. Each face parts of the heat exchanger are named. Then, boundary conditions of the cell zones of the heat exchanger are set. Both hot water and the cooling agent for each test are set to have same velocity and pressure, which is 1 ms^(-1)(inlet) and 10000 Pa (outlet) respectively. The number of iterations used for the simulation is 500 times to ensure a better reading. As all the simulations for the three cooling agents are conducted, it is found that water cools hot water better than ethylene glycol and air. The analysis results show that water, ethylene glycol and air has the heat transfer rate of 55.72 kW, 55.495 kW and 55.465 kW respectively. Thus, water is proposed to be the most suitable cooling agent in the cooling system of heat exchanger.

Paper ID	**Track**	**Title**
15-560	Track 3	APPLICATION OF DESIGN FOR SIX SIGMA METHODOLOGY ON PORTABLE WATER FILTER USING MEMBRANE FILTRATION SYSTEM: A PRELIMINARY STUDY

Abstract Portable water filter has grown significantly in recent years. The used of water bottles as a water drink stuff using hand pump water filtration unit has been suggested to be used to replace water bottled during outdoor recreational activities and for emergency supplies. However, quality of water still the issue related to contaminated water due to the residual waste plants, bacteria, and so on. Based on these issues, the study was carried out to design a portable water filter that uses membrane filtration system by applying Design for Six Sigma. Design for Six Sigma methodology consists of five stages which is Define, Measure, Analyze, Design and Verify. There were several tools have been used in each stage in order to come out with a specific objective. In the Define stage, questionnaire approach was used to identify the needs of portable water filter in the future from potential users. Next, Quality Function Deployment (QFD) tool was used in the Measure stage to measure the users' needs into engineering characteristics. Based on the information in the Measure stage, morphological chart and weighted decision matrix tools were used in the Analyze stage. This stage performed several activities including concept generation and selection. Once the selection of the final concept completed, detail drawing was made in the Design stage. Then, prototype was developed in the Verify stage to conduct proof-of-concept testing. The results that obtained from each stage have been reported in this paper. From this study, it can be concluded that the application of Design for Six Sigma in designing a future portable water filter that uses membrane filtration system is a good start in looking for a new alternative concept with a completed supporting document.

Paper ID	**Track**	**Title**
15-563	Track 3	DETERMINE SPRAY DROPLETS ON WATER SENSITIVE PAPER (WSP) FOR LOW PRESSURE DEFLECTOR NOZZLE USING IMAGE J

Abstract In this study, determine of spray droplets size (SMD) using water sensitive paper (WSP) at low fluid pressure with deflector nozzle or tangential flow nozzle model Delavan AL75 and New Design Nozzle with two different type of swirl (ND2.5 A1.0 & ND2.5 B1.0). These three deflected flat sprays have used at different liquid mixing ratio. These liquid mixture ratios are pure water, 10% of lime juice + 90% of water (L10W90) and 30% of lime juice + 70% of water (L30W70). WSP is used to collect the spray droplets from nozzles. The operational liquid pressure of each nozzle is 3 bar, while air operational pressures are 3 bar and 6 bar. Then, the WSP were scanned using scanner then it was analyzed using ImageJ software. ImageJ can be used for determining the diameter of droplets size on the WSP. As the results from an experiment, the AL75 nozzle recorded the lowest Sauter mean diameter which is 193.69µm at 6 bar of pressurized air while ND2.5 A1.0 recorded the highest Sauter mean diameter which is 353.61µm at 3 bar of pressurized air. Summary from the experiment shows that the higher of droplet size is because of the lower air pressure

(3 Bar). Then, increasing of liquid viscosity also increase the SMD. The orifice diameter for New Design nozzle (ND-2.5) is smaller than AL75, which are 2.5mm and 2.8mm respectively. The different nozzle design also gives effect the SMD. WSP is an alternative method to determine SMD for spray droplets with the low cost if compared to Phase Doppler Anemometry (PDA).

Paper ID	**Track**	**Title**
15-565	Track 3	DEVELOPMENT AND TESTING OF A SOLAR POWERED ICE-MAKING MACHINE

Abstract This paper reports on a design, develop and test a solar powered ice making machine project. Utilization of the solar thermal energy is selected as the power source for the refrigeration cycle due to its abundance and also low cost. Adsorption refrigeration system could be used when electrical energy is scarce, such as in rural community and is space systems. It could be powered by solar thermal energy. It is simple in construction, corrosion resistant and could utilise wide range of heat sources. Activated carbon was used as the adsorbent due to its high porosity and can withstand higher temperature. The refrigerant used for the system was methanol, which has high latent heat of evaporation and environment friendly. The system consist of three parts; generator, condenser and evaporator which consist of no moving parts except two ball valves for charging process. The goal of the system is to produce 1 kg of ice. The system was exposed to sunlight and tested for its cooling power. Results showed that the cooling process lasted for 2 hours producing cool water at 16°C. The main reason for the decrease in cooling capacity is due to the generator design, which had caused a problem in mass transfer of methanol vapour during adsorption.

Paper ID	**Track**	**Title**
15-571	Track 3	DETERMINATION OF PARTICLE-BOUND POLYCYCLIC AROMATIC HYDROCARBONS EMITTED FROM CO-PELLETIZATION COMBUSTION OF LIGNITE AND RUBBER WOOD SAWDUST

Abstract Determination of particle-bound Polycyclic Aromatic Hydrocarbons (PAHs) emitted from co-pelletization combustion of lignite and rubber wood sawdust in a horizontal tube furnace is investigated using High Performance Liquid Chromatography with coupled Diode Array and Fluorescence Detection (HPLC-DAD/FLD). In the present study, the particle-bound PAHs are likely abundant in the fine particles. More than 70% of toxicity degree of PAHs falls into PM1.1 while more than 80% of mass concentration of PAHs fall into PM2.5. The addition of lignite amount in the co-pelletization results in the increasing concentration of 4-6 aromatic ring PAHs. Furthermore, the rubber wood sawdust pellets (0% lignite pellets) emit high mass concentration of PAHs whereas the lignite pellets (100% lignite pellets) emit high toxicity degree of PAHs. By co-pelletized rubber wood sawdust with lignite (50% lignite pellets) has significant effect to reduce the toxicity degree of PAHs by 70%.

Paper ID	**Track**	**Title**
15-629	Track 3	ANALYSIS OF HIGH INJECTION PRESSURE AND AMBIENT TEMPERATURE ON BIODIESEL SPRAY CHARACTERISTICS USING COMPUTATIONAL FLUID DYNAMICS

Abstract Efficiency of combustion engines are highly affected by the formation of air-fuel mixture prior to ignition and combustion process. This research investigate the mixture formation and spray characteristics of biodiesel blends under variant in high ambient and injection conditions using Computational Fluid Dynamics (CFD). The spray characteristics such as spray penetration length, spray angle and fluid flow were observe under various operating conditions. Results show that increase in injection pressure increases the spray penetration length for both biodiesel and diesel. Results also indicate that higher spray angle of biodiesel can be seen as the injection pressure increases. This study concludes that spray characteristics of

biodiesel blend is greatly affected by the injection and ambient conditions.

CEiES
UTHM

Organiser:
Centre for Energy & Industrial Environment Studies,
Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia