



# **International Conference on**

# **Computational**

# **Fluid Dynamics**

# **in Research & Industry 2015**

***Seri Pacific Hotel,***  
***Kuala Lumpur, Malaysia***

**17-19**  
**August**  
**2015**

### Table of Contents

Paper ID	Track	Title	Page
1119	CFD	PARALLELIZATION OF THE MOVING PARTICLE PRESSURE MESH (MPPM) FLUID FLOW SOLVER BY USING OPENMP	- 6 -
1120	CFD	NUMERICAL WORKS ON 2-DIMENSIONAL AND 3-DIMENSIONAL TURNING DIFFUSER WITH BAFFLES	- 6 -
1125	CFD	SOLUTIONS OF POISEUILLE FLOW USING HOMOTOPY PERTURBATION METHOD LINKED WITH PADÉ APPROXIMATION	- 6 -
1131	CFD	NUMERICAL INVESTIGATION OF FIRE SPREAD, EVACUATION AND HAZARD ASSESSMENT IN AN OFFSHORE PETROLEUM PLATFORM BY USING CFD SIMULATION	- 7 -
1133	CFD	PRELIMINARY NUMERICAL SIMULATION OF STEADY-STATE GAS-LIQUID FLOW IN HORIZONTAL T-JUNCTION	- 7 -
1139	CFD	NUMERICAL INVESTIGATION ON THE HYDRODYNAMIC UNDERWATER GLIDER WITH DIFFERENT WING LAYOUTS CHARACTERISTICS OF AN AUTONOMOUS	- 7 -
1141	CFD	THE FLOW PATTERN STUDY OF A CENTRIFUGAL PUMP USING CFD METHODS CONCENTRATING ON VOLUTE TONGUE ROLE	- 8 -
1142	CFD	DISCHARGE AND FLOW COEFFICIENT ANALYSIS IN INTERNAL COMBUSTION ENGINE USING COMPUTATIONAL FLUID DYNAMICS SIMULATION	- 8 -
1144	CFD	CFD SIMULATION FOR CORROSION MODELING AT OIL & GAS TOPSIDES: A PRELIMINARY STUDY	- 8 -
1146	CFD	ARBITRARY LAGRANGIAN-EULERIAN FORM OF FLOWFIELD DEPENDENT VARIATION METHOD FOR FLUID-STRUCTURE INTERACTION APPLICATION	- 9 -
1155	CFD	RECENT PROGRESS ON THE DEVELOPMENT OF A PARTICLE METHOD FOR INCOMPRESSIBLE SINGLE- AND MULTI-PHASE FLOW COMPUTATION	- 9 -
1160	CFD	EFFECTS OF ASYMMETRIC INTAKE VALVE LIFT CONFIGURATION TOWARDS IN-CYLINDER AIR FLOW BEHAVIOR	- 9 -
1162	CFD	VORTEX IN CELL METHOD TO PREDICT FLUTTER PHENOMENON OF 2D BRIDGE DECK MODEL	- 10 -
1163	CFD	NUMERICAL SIMULATION OF FALLING FILM THICKNESS FLOWING OVER HORIZONTAL TUBES	- 10 -
1164	CFD	PREDICTION OF TEMPERATURE AND AIR FLOW IN A DOMESTIC REFRIGERATOR	- 10 -
1165	CFD	UTILIZING OPEN SOURCE SOFTWARE RUNNING IN INEXPENSIVE HIGH PERFORMANCE COMPUTING SYSTEM FOR CFD APPLICATIONS	- 11 -
1166	CFD	COMPARISON BETWEEN ROE FVM AND DAVIS-YEE TVD SCHEME ON FLOW OVER NACA 0012	- 11 -
1178	CFD	STUDY ON THE EFFECT OF WING GEOMETRY ON UNDERWATER GLIDER HYDRODYNAMICS	- 11 -
1183	CFD	THE AERODYNAMICS ANALYSIS FOR THE FLOW PAST THROUGH AN INLET MODEL BY USE OF A FOURTH ORDER RUNGE KUTTA – DAVIS –YEE SCHEME	- 12 -
1186	CFD	POTENTIAL OF VOIDS TO ENHANCE NATURAL VENTILATION IN MEDIUM COST MULTI-STOREY HOUSING (MCM SH) FOR HOT AND HUMID CLIMATE	- 12 -



Paper ID	Track	Title	Page
1187	CFD	AN INTEGRATION OF COMPUTATIONAL INTELLIGENCE TECHNIQUES IN ENGINEERING DATA PROCESSING FOR IMPROVING FORECAST ACCURACY USING CUBIC-SPLINE INTERPOLATION AND ANN MODEL	- 12 -
1196	CFD	COMPARISON FINE SPRAY NOZZLE FOR TWO PHASE FLOW WITHIN EXPERIMENTAL AND SIMULATION	- 13 -
1197	CFD	STUDY THE FLOW FOR THE INJECTOR NOZZLE WITH EFFECT OF DIFFERENT PRESSURE TO ANALYZE SPRAY PATTERN	- 13 -
1201	CFD	HEMODYNAMIC CHARACTERISTIC IN PATENT DUCTUS ARTERIOSUS	- 14 -
1115	HT	NUMERICAL INVESTIGATION ON PITCH DISTANCE, COMPOUND ANGLE AND TURBULENCE INTENSITY OF DOUBLE CYLINDRICAL FILM COOLING HOLE GEOMETRY	- 14 -
1116	HT	EFFECTS OF LOD AND POD IN COMBINED-HOLE FILM	- 14 -
1124	HT	MIXED CONVECTION IN AN INCLINED LID-DRIVEN SQUARE CAVITY WITH SINUSOIDAL HEATING ON TOP LID	- 15 -
1145	HT	DESIGNE OF THIN FILM'S GEOMETRY BY ANNEALING ALGORITHM AND COHERENT FORMULATION	- 15 -
1149	HT	EFFECT OF SINUSOIDALLY HEATING ON MIXED CONVECTION IN SQUARE CAVITY FILLED WITH A POROUS MEDIUM	- 15 -
1121	TA	IMPLEMENTING VIBRATION FRAMEWORK FOR SIMULATION OF VIV ON RIGID PIER BY SPH	- 16 -
1122	TA	PENETRATIVE TURBULENCE ASSOCIATED WITH MESOSCALE SURFACE HEAT FLUX VARIATIONS	- 16 -
1181	TA	CFD ANALYSIS OF SD 7003 AIRFOIL AT LOW REYNOLDS NUMBER WITH A LAIMINAR KINETIC ENERGY BASED TRANSITION MODEL	- 16 -
1118	FSI	ARTERY VIBRATION ANALYSIS IN THE PRESENCE OF ARTIFICIAL MECHANICAL HEART VALVE MECHANICAL HEART VALVE	- 17 -
1128	FSI	VIBRATION ANALYSIS ON ARTERIES IN PATIENTS WITH ARTIFICIAL HEART VALVE	- 17 -
1130	FSI	NUMERICAL INVESTIGATION ON THE EFFECT OF BLOOD FLOW INDUCED VIBRATION ON BILEFLET ARTIFICIAL HEART VALVE BY USING FLUID STRUCTURE INTERACTION TECHNIQUE	- 18 -
1134	FSI	NUMERICAL SIMULATION OF LIQUIDS DRAINING FROM A TANK USING OPENFOAM	- 18 -
1135	FSI	COMPARATIVE STUDY ON ENERGY EXTRACTION FROM VIBRATING SQUARE CYLINDER	- 18 -
1158	FSI	VALIDATION FRAMEWORK FOR FLUID-PARTICLE-STRUCTURE-INTERACTION SOLVER	- 19 -
1168	FSI	FLOW INDUCED VIBRATION OF A SQUARE CYLINDER WITH HIGH SCRUTON NUMBER	- 19 -
1176	FSI	TRANSVERSE VORTEX INDUCED VIBRATION OF SPRING-SUPPORTED CIRCULAR CYLINDER WITH MASS RATIO OF 10 TRANSLATING CLOSE TO A PLANE WALL	- 19 -
1180	FSI	SIMULATION OF TWO DIMENSIONAL CARANGIFORM AND ANGUILLIFORM SWIMMER	- 20 -

Paper ID	Track	Title	Page
1182	FSI	STUDY OF SHEAR RATES IN SPINNING PROCESS OF KAOLIN/POLYETHERSULFONE(PESF) MEMBRANE PRECURSOR: EFFECT ON FIBER MORPHOLOGY	- 20 -
1200	FSI	COMPARATIVE STUDY OF THE COMPUTATIONAL FLUID DYNAMICS AND FLUID STRUCTURE INTERACTION ANALYSIS IN HUMAN AIRWAYS FLOW	- 20 -
1179	CBT	CO-COMBUSTION MODELING OF RICE HUSK AND PLASTIC BAG AS ENERGY SOURCE IN INDONESIA	- 21 -
1126	IAQ	EXPERIMENTAL STUDY ON PERFORMANCE OF WIND CATCHER IN TROPICAL CLIMATE	- 21 -
1175	IAQ	CFD INVESTIGATION OF INDOOR HYGROTHERMAL AND AIRFLOW PROFILE IN ACADEMIC RESEARCH STORAGE ROOM: EFFECT OF LMA ON THERMOHYGRIC BALANCE AND MOULD GROWTH	- 21 -
1177	IAQ	COMPUTATIONAL ANALYSIS ON THE EFFECTS OF FACADE MODIFICATIONS ON WIND-DRIVEN NATURAL VENTILATION PERFORMANCE OF A SINGLE-CELL ROOM	- 22 -
1184	IAQ	CFD INVESTIGATION OF INDOOR HYGROTHERMAL PERFORMANCE IN ACADEMIC RESEARCH STORAGE ROOM: MEASUREMENT AND VALIDATION	- 22 -
1198	IND	EXPERIMENTAL STUDY OF GREASE EMISSION FILTRATION FOR KITCHEN HOOD BY WATER MIST	- 22 -
1127	SA	INVESTIGATE THE BEHAVIOUR OF WATER MIST SPRAY WITH LOW PRESSURE APPLIED FOR KITCHEN VENTILATION EXHAUST (KVE) SYSTEM	- 23 -
1169	SA	A NUMERICAL INVESTIGATION ABOUT THE EFFECTS OF NOZZLE HOLE GEOMETRY ON SPRAY CHARACTERISTICS OF THE PREMIX INJECTOR	- 23 -
1159	AE	NUMERICAL STUDY OF VARIABLE LENGTH EXHAUST PIPE IN SMALL ENGINE.	- 23 -
1190	AE	VEHICLE AERODYNAMICS ANALYSIS OF A MULTI PURPOSE VEHICLE USING CFD	- 24 -
1191	AE	THE EFFECT OF INCREASING THE PERCENTAGE OF HYDROGEN ADDED TO METHANE IN DIRECT INJECTION SPARK IGNITION ENGINE TO THE COMPOSITION OF CO AND CO <sub>2</sub> EMISSIONS	- 24 -
1167	PMF	METHODOLOGY TO OBTAIN THERMAL CONDUCTIVITY, PERMEABILITY AND HEAT TRANSFER COEFFICIENT OF A POROUS COPPER HEAT SINK BASED ON X-RAY MICRO-CT MODELLING	- 25 -
1157	FM	CFD SIMULATION OF FLOW CONTROL WITH ENERGY PROMOTERS IN S-SHAPED DIFFUSER	- 25 -
1161	FM	NUMERICAL STUDY OF VELOCITY AND PRESSURE DROP IN SINGLE FLOW MICROCHANNEL USING WATER AND AL <sub>2</sub> O <sub>3</sub> NANOFLUID	- 25 -
1143	AER	AERODYNAMICS CHARACTERISTICS AROUND SIMPLIFIED HIGH SPEED TRAIN MODEL UNDER THE EFFECT OF CROSSWINDS	- 26 -
1148	AER	VEHICLE AERODYNAMICS ANALYSIS OF A MULTI PURPOSE VEHICLE USING CFD	- 26 -
1192	AER	AERODYNAMICS PERFORMANCES OF PURGE FLOW INJECTION BY THE EFFECT OF SLOT POSITIONS AND ORIENTATIONS IN LINEAR CASCADE OF HIGH-PRESSURE TURBINE	- 26 -
1114	GEN	AN INTEGRATION OF COMPUTATIONAL INTELLIGENCE TECHNIQUES IN ENGINEERING DATA PROCESSING FOR IMPROVING FORECAST ACCURACY USING CUBIC-SPLINE INTERPOLATION AND ANN MODEL	- 27 -

<b>Paper ID</b>	<b>Track</b>	<b>Title</b>	<b>Page</b>
1153	GEN	SIMULATION OF WATER RE-INJECTION IN THE DOWNHOLE WITH PRESENCE OF HYDROCYCLONE SEPARATOR	- 27 -
1173	GEN	AN EFFECT OF STRAIGHT AND SERPENTINE FLOW FIELD DESIGN ON PROTON EXCHANGE MEMBRANE FUEL CELL	- 28 -
1202	GEN	THE FLOW BEHAVIOUR STUDY OF SPLITTING DEVICE FOR HORIZONTAL PIPELINE	- 28 -
1203	CFD	A CFD STUDY OF SWIRL EFFECT ON HYDRODYNAMICS IN A GAS-LIQUID BUBBLY FLOW REACTORS	- 28 -

Paper ID	Track	Title
1177	IAQ	COMPUTATIONAL ANALYSIS ON THE EFFECTS OF FACADE MODIFICATIONS ON WIND-DRIVEN NATURAL VENTILATION PERFORMANCE OF A SINGLE-CELL ROOM

**Abstract** Single-sided ventilated (SSV) buildings have been always assumed to be less efficient in natural ventilation performance compared to cross ventilated building, thus when a good ventilation performance is required in a building the single-sided ventilation strategy has always been ignored as an alternative ventilation strategy to cross ventilation. It is known that the cross ventilation strategy can generally perform better than the single sided ventilation. However, this is not necessarily true in all cases, due to various factors such as wind direction and façade treatment. The objective of this study is to investigate wind-driven natural ventilation performances for a single-cell room with various façade treatment options. This study explores various façade treatments, and the performance of each façade treatment is evaluated. This study uses computational analysis to investigate the ventilation performances. This research methodology is used due to its flexibility and post-processing advantage. This study found that façade treatments such as wing-wall and balcony can significantly influences the natural ventilation performance of a single-cell room.

Paper ID	Track	Title
1184	IAQ	CFD INVESTIGATION OF INDOOR HYGROTHERMAL PERFORMANCE IN ACADEMIC RESEARCH STORAGE ROOM: MEASUREMENT AND VALIDATION

**Abstract** Poor hygrothermal performance exacerbates deterioration risk from mould growth, corrosion and damage to archival materials. Improved microcomputers' computational power has significantly advanced computational fluid dynamics (CFD) models and research developments in indoor airflow, heat transfer and contaminant transport. Nevertheless, numerous uncertainties exit in the CFD experiments which require adequate clarifications for improved results' reliability. This paper presents the measurement and validation of a CFD model for the investigation of the hygrothermal performance in an indoor environment with known cases of microbial proliferations. The room, 5.2 m × 4.8 m × 3.0 m high, is air-conditioned and ventilated by constant air volume (CAV) system controlling the indoor airflow and hygrothermal profiles with ceiling mounted four-way supply diffuser and extract grille for indoor air distribution. A good correlation was established between the predicted and measured indoor hygrothermal profile with less than 10% deviation indicating that the model can be employed for further investigation with high confidence.

Paper ID	Track	Title
1198	IND	EXPERIMENTAL STUDY OF GREASE EMISSION FILTRATION FOR KITCHEN HOOD BY WATER MIST

**Abstract** In this study, water mist spray operation are applied to control the emissios produces from cooking smoke and to reduce the temperature inside of the ductwork. A full scale of experiments were performed to investigate the effect of water mist spray on grease emission filtration in kitchen hood. Continuous measurement of temperature, pressure drop, and generations of TVOC at specific locations were made at average 8 hours and 15 minutes under grease load using grease generator designed based on UL 1046 Standard. For comparison, parameters were measured from 3 different types of test conditions which are without cold mist operation, with cold mist operation, and KSA filter exhaust plenum. The water mist activation have significant effects to filter the grease emissions contains parcticles size more than 10 µm and to decrease the temperature of the kitchen exhaust