Technical Papers at WCX 2022

1. **CFD Analysis of a Centrifugal Pump Controlling a Vehicle Coolant Hydraulic System: a Comparison between MRF and Transient Approaches**
   2022-01-0780

2. **Numerical Modelling of Coolant Filling and De-aeration in a Battery Electric Vehicle Cooling System**
   2022-01-0775

3. **Transient, 3D CFD, Moving Mesh Simulation of Vehicle Water Wading in a Water Tunnel with Inclined Entry-Exit**
   2022-01-0768

4. **Design Optimization of Centrifugal Pump Using CFD Simulations, Metamodeling and Bayesian Inference**
   2022-01-0787

5. **Numerical Study of an I4 Engine Oil Ejection During an Accidental Cap-off Running Condition for Two Baffle Designs**
   2022-01-0398

6. **3D CFD Simulation of Hydraulic Test of an Engine Coolant System**
   2022-01-0207

7. **Multidimensional CFD Studies of Oil Drawdown in an i-4 Engine**
   2022-01-0397

8. **Heat Transfer Analysis of an Electric Motor Cooled by a Large Number of Oil Sprays Using Computational Fluid Dynamics**
   2022-01-0208

CFD Analysis of a Centrifugal Pump Controlling a Vehicle Coolant Hydraulic System: a Comparison between MRF and Transient Approaches

2022-01-0780

Centrifugal pumps are widely used in different thermal fluid systems in automobile industries. Computational fluid dynamics (CFD) analysis of such a thermal fluid system depends on the accurate component modeling of the system components. This paper presents CFD analysis of a centrifugal pump with two different approaches: Transient (moving grid) and the steady state - Multiple Reference Frame (MRF) methods using a commercial CFD solver Simerics MP+®. In addition, flow and pressure drop data obtained using CFD simulations of a vehicle coolant hydraulic system was compared to results from rig test data. The Transient method incorporates the real motion of the pump blades geometry and temporal flow solutions are obtained for instantaneous positions of the blade geometry. In MRF approach, the flow governing equations for the stationary zone are solved in the absolute/inertial reference frame, whereas flow in the moving zone is solved in the relative/non-inertial reference frame. This study presents a comparison of the Transient and MRF CFD simulation results of a standalone centrifugal pump, for a range of points on the pump curve, and the favorable comparison to the standalone pump tests. Having validated the standalone pump, a complete vehicle coolant hydraulic system incorporating the centrifugal pump is numerically investigated using the MRF approach and compared with the test data. The numerical prediction of flow rates and pressure drops across different components of the coolant system shows close proximity with the test data.

Pages: 7
Event: WCX SAE World Congress Experience
ISSN: 0148-7191
e-ISSN: 2688-3627

Numerical Modelling of Coolant Filling and De-aeration in a Battery Electric Vehicle Cooling System

2022-01-0775

Trapped air bubbles inside coolant systems have adverse effect on the cooling performance. Hence, it is imperative to ensure an effective filling and de-aeration of the coolant system in order to have less air left before the operation of the coolant system. In the present work, a coolant/air multiphase VOF method was utilized using the commercial CFD software Simerics MP+® to study the coolant filling and subsequent de-aeration process in a Battery Electric Vehicle (BEV) cooling system. First, validations of the numerical simulations against experiments were performed for a simplified coolant recirculation system. This system uses a tequila bottle for de-aeration and the validations were performed for different coolant flow rates to examine the de-aeration efficiency. A similar trend of de-aeration was captured between simulation and experimental measurement. Next, the same numerical techniques were further applied to a BEV cooling system to evaluate the efficiency of de-aeration processes. The first step of the process involved a vacuum filling simulation. After the filling process, the air remained in the system is about 10% of total system volume. Different combinations of a multi-position valve and pump on/off cycles strategies were explored to decrease trapped air in the system. It is found that the opening/closing strategy of the multi-position valve plays a crucial role for an effective de-aeration. The devised methodology is observed to be numerically robust and accurate, while having good computational efficiency for modelling the coolant filling and de-aeration process that takes large physical time.

Pages: 9
Event: WCX SAE World Congress Experience
ISSN: 0148-7191
e-ISSN: 2688-3627

Transient, 3D CFD, Moving Mesh Simulation of Vehicle Water Wading in a Water Tunnel with Inclined Entry-Exit

2022-01-0768

Water wading tests are commonly performed for vehicles to ensure the functional integrity of different under-hood components at different water depths. This test has its relevance in both conventional Internal Combustion (IC) engine-based vehicles and Electric Vehicles (EVs). In IC engines, it is important for designing the Air Induction System (AIS), and for EVs, it helps to check the wetting of critical electrical and electronic components. The experimental setup for this test includes a long water tunnel where the car enters and exits the pool of water through a ramp. This work is an extension of the work done by Varshney et al. [6] where the Moving Reference Frame (MRF) technique was used to account for the motion of the car and the rotation of the wheels. The current work uses mesh morphing techniques to account for the motion of the vehicle and the rotation of the wheels that replicates the actual test conditions, including the inclination of the vehicle on the ramp. The entire pre-processing, post-processing, and the 3D CFD simulation have been done using commercial code, Simerics MP+®. Explicit VOF multiphase approach has been followed to predict the air-water interface. The simulation necessitates considering a big computational domain along with the actual geometry of all the under-hood components that make the simulation computationally expensive. Hence, optimization of computational time while ensuring minimal numerical diffusion are vital contributions from the work. Wading analysis has first been done on a DrivAer model [8] and later, a detailed production car model is simulated. Quantification of the level of water in the underhood compartment has been done at different water heights. Results of the wave pattern formed around the car and other under-hood components are also analysed.

Pages: 10
Event: WCX SAE World Congress Experience
ISSN: 0148-7191
e-ISSN: 2688-3627

Design Optimization of Centrifugal Pump Using CFD Simulations, Metamodeling and Bayesian Inference

2022-01-0787

Computational expenses aside, simulating and optimizing pumps operating at pressures near the liquid's saturation pressure needs complete modeling of cavitation physics. This becomes critical in high-temperature applications since the saturation pressure increases with temperature and the pumps become more prone to cavitation. In the present work, the performance of a centrifugal pump was improved by delaying the sudden onset of cavitation at higher flow rates through constrained optimization of impeller geometry. The optimized designs generated over 25% higher head at the operating point and performed better than the baseline design across the range of operation. Constraints were dictated by geometric/ packaging limitations in order to ensure that the optimized impeller can be retrofitted into an existing fluid-power system. A Gaussian Process Regressor (GPR) based metamodel was constructed utilizing a database of designs generated through Latin Hypercube Sampling (LHS). Their respective performances were predicted by CFD simulations using Simerics MP+®, a commercial CFD code. Finally, the optimizer used the statistical insights provided by the metamodel and generated new impeller designs, the performance of which were subsequently evaluated through numerical simulations in Simerics MP+®. Selected designs were fabricated, and experiments were conducted to validate predictions provided by CFD simulations. The optimization process, CFD model, simulation and experiment results are discussed in detail. A good agreement between simulated results and experiments was observed. Finally, through the CFD solution, the internal flow structures were thoroughly analyzed, and a mechanism of performance improvement was established.

Pages: 8
Event: WCX SAE World Congress Experience
ISSN: 0148-7191
e-ISSN: 2688-3627

Numerical Study of an I4 Engine Oil Ejection During an Accidental Cap-off Running Condition for Two Baffle Designs

2022-01-0398

Three-dimensional transient numerical simulations are conducted to study the oil flow in a four-cylinder internal-combustion engine while it operates without its oil filler cap on. The emphasis of the study is on analyzing the consequential oil ejection through the oil-cap open boundary. Navier-Stokes equations are solved together with the multiphase Volume of Fluid (VOF) model and the k-ϵ turbulence model. The engine crank shaft is mechanically connected to two cam shafts through a chain, which operates below the oil-filler duct. A baffle is located between the chain and the duct, shielding the latter to minimize oil ejection and potential spills. The chain geometry and dynamics are captured accurately through volume remesh and conformal mapping techniques. The motion of the four pistons, crank shaft, and two cam shafts is also considered. Retaining all these mechanical and geometrical details in the simulations is essential to obtain accurate oil ejection results. The crank shaft rotates at 1200 RPM, and the study is conducted for two different baffle designs. Quick turn-around-time rolling-average results from numerical simulations are compared with experimental data for baffle designs 1 and 2. Findings demonstrate good agreement both in trend and in magnitude for an application previously considered impractical in Computational Fluid Dynamics (CFD) while using the Volume of Fluid (VOF) method.

Pages: 7
Event: WCX SAE World Congress Experience
ISSN: 0148-7191

3D CFD Simulation of Hydraulic Test of an Engine Coolant System

2022-01-0207

Designing an efficient vehicle coolant system depends on meeting target coolant flow rate to different components with minimum energy consumption by coolant pump. The flow resistance across different components and hoses dictates the flow supplied to that branch which can affect the effectiveness of the coolant system. Hydraulic tests are conducted to understand the system design for component flow delivery and pressure drops and assess necessary changes to better distribute the coolant flow from the pump. The current study highlights the ability of a complete 3D Computational Fluid Dynamics (CFD) simulation to effectively mimic a hydraulic test. The coolant circuit modeled in this simulation consists of an engine water-jacket, a thermostat valve, bypass valve, a coolant pump, a radiator, and flow path to certain auxiliary components like turbo charger, rear transmission oil cooler etc. A commercial CFD software, Simerics-MP+®, is used to simulate the hydraulic test for two different positions of the poppet valve of the thermostat, viz. a closed position and an 50% opening, at different speeds of the engine. In the CFD model, the complete geometrical details of water-jacket, thermostat, and pump are considered. The remaining components are approximated as pipes with flow resistance models to account for flow and pressure drop at different engine speeds. Firstly, the standalone pump performance is validated in the operating regime of interest, followed by the calibration of the resistance models for the simplified components. At the end, complete system level 3D simulations are conducted and validated for the above mentioned two positions of the poppet valve. The flow distribution and pressure drop across different components show good comparison with the hydraulic test data within 7% error band.

Pages: 11
Event: WCX SAE World Congress Experience
ISSN: 0148-7191
e-ISSN: 2688-3627

Multidimensional CFD Studies of Oil Drawdown in an i-4 Engine

2022-01-0397

A computational study based on unsteady Reynolds-Averaged-Navier-Stokes that resolves the gas-liquid interface was performed to examine the unsteady multiphase flow in a 4 cylinder Inline (i-4) engine. In this study, the rotating motion of the crankshaft and reciprocating motion of the pistons were accounted for to accurately predict the oil distribution in various parts of the engine. Three rotational speeds of the crankshaft have been examined: 1000, 2800, and 4000 rpm. Of particular interest is to examine the mechanisms governing the process of oil drawdown from the engine head into the case. The oil distributions in other parts of the engine have also been investigated to understand the overall crankcase breathing process. Results obtained show the drawdown of oil from the head into the case to be strongly dependent on the venting strategy for the foul air going out of the engine through the PCV system. Results also show the dynamic holdup of oil in the steady operation to be highest near the crankshaft and pistons. Results are presented to show how the rotational speed of the crankshaft affects the nature of multiphase flow inside the engine and its influence on the drawdown of oil from the head into the case. The computational study was validated by comparing the computed volume of oil in the sump in steady state operation with the experimental measurements. The computational strategy presented in this study to simulate the crankcase breathing process can be most useful in guiding the design and development of engines.

Pages: 13
Event: WCX SAE World Congress Experience
ISSN: 0148-7191
e-ISSN: 2688-3627

Heat Transfer Analysis of an Electric Motor Cooled by a Large Number of Oil Sprays Using Computational Fluid Dynamics

2022-01-0208

This paper reports on an analytical study of the heat transfer and fluid flow in an electric vehicle e-Motor cooled by twenty five sprays/jets of oil. A three-dimensional, quasi-steady state, multi-phase, computational fluid dynamics (CFD) and conjugate heat transfer (CHT) model was created using a commercial CFD software. The transport equations of mass, momentum, energy and volume fraction were solved together with models for turbulence and wall treatment. An explicit formulation of the volume of fluid (VOF) technique was used to simulate the sprays, a time-implicit formulation was used for the flow-field and three dimensional conduction heat transfer with non-isotropic thermal conductivities was used to simulate the heat transfer in the windings. An important challenge was to formulate an efficient solution algorithm that can rigorously comprehend twenty five sprays in order to accurately predict the temperature distributions in the windings, without thrifting physics, and within reasonable total analysis turn-around times. The latter includes both model set-up times as well as run times. User functions were coded in order to implement the efficient solution methodology. The outcome of this effort was a novel solution technique which due to its economical analysis times of the order of a week or so, can play a key role in virtually optimizing the e-Motor design, reduce dependence on costly and time consuming tests and shorten product development times. The predicted temperatures of the windings show good agreement with thermocouple measurements.

Pages: 14
Event: WCX SAE World Congress Experience
ISSN: 0148-7191
e-ISSN: 2688-3627