Proceedings of the ASME 2016 International Mechanical Engineering Congress and Exposition
IMECE2016
November 11-17, 2016, Phoenix, Arizona, USA

IMECE2016-66999

A CFD STUDY OF WATER CONDENSATION INSIDE THE TUBES OF AN AUTOMOTIVE COMPACT CHARGE AIR COOLER USING LARGE EDDY SIMULATION APPROACH

Robin Cash
Ford Motor Company
Dearborn, MI, USA

Apoorv Talekar
Wayne State University
Detroit, MI, USA

Bashar AbdulNour
Ford Motor Company
Dearborn, MI, USA

ABSTRACT

The usage of compressed air generated by supercharger or turbocharger by automotive Original Equipment Manufacturers (OEM) is growing with the aim to increase engine performance by increasing the density of the air charge being drawn into the cylinder. Denser air coupled with more fuel pulled into the combustion chamber results in increased engine performance. The inlet air is heated during compression which can cause pre-ignition, which leads to reduced engine functionality. The charge air cooler (CAC) is a heat exchanger introduced to extract heat created during the compression process. Previous research developed a 3-D Computational Fluid Dynamics (CFD) model using the k-epsilon turbulent model with near wall treatment to resolve turbulence in the small channels of the CAC. [1] The present research uses a refined computational scheme with a Large Eddy Simulation (LES) model to solve local data as a function of time and location and correlates the result to the experimental measurements, as well as compare to the k-epsilon approach. Using LES resulted in the ability to correlate any portion of the experimental data and take a closer look at local heat transfer between the outside surface of the tube and the cooling air. Large Eddy Simulation for heat transfer gave more information required for design of CACs which is difficult to collect for various operating conditions by experiment. The overall benefit presented is a validated simulation methodology that predicts condensation, which is then used to evaluate and design CACs that function outside the condensate formation zone during various vehicle operation modes.

INTRODUCTION

Turbocharged engines receive pressurized, cool air from the charge air cooler. The CAC is required to maintain inlet air temperature to the engine. This is crucial as the combustion characteristics of the engine depends on the inlet temperature of air and combustion can be unpredictable if the inlet air temperature fluctuates from cycle to cycle and contains water droplets [2, 3]. It is of utmost importance to hold cycle to cycle variation of the engine performance less than 3% [4, 5]. As shown in Figure 1, the air box can prevent unwanted particles from entering the system but humidity is a homogeneous part of the air mixture.

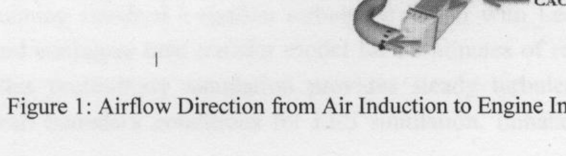

Figure 1: Airflow Direction from Air Induction to Engine Intake

It was noticed in the case of the charge air cooler, which is used with gasoline turbo direct injection (GTDI) EcoBoost internal combustion engine, that at particular operating ranges, CAC develops a layer of water condensate at high rates inside each heat exchanging tube. This water condensate film, when sufficiently thick, gains the momentum from the charge air coming from the turbocharger and then is ingested into the engine manifold. Due to the intricate structure and larger surface area available in the intake manifold of the engine, it becomes more difficult to evaporate this water condensate. Diesel compression ignition internal combustion engine can withstand a defined amount of water ingestion [6] but if there is no control over this water condensate which occurs inside the CACs, it is difficult to predict and take counter measures against it. Condensation formation inside small cross-section of the CAC tubes is highly depends on the thermodynamic conditions, it is, therefore, necessary to define all possible sets of conditions which trigger film formation [7, 8]. This study is in part to find an effective solution to predict condensate formation. Also, it is intended to determine ways to avoid condensation formation and to stop water entering in the engine intake manifold. An experimental rig was setup to detect operation range of condensate formation. A 3D CFD simulation model was also created and validated using experimental results. In this study, two different simulation approaches namely k-epsilon and LES with Eulerian wall film model were used to detect highly unsteady condensation and movement of the water film at elevated pressure and temperature.

EXPERIMENTAL SETUP

The test rig was developed to replicate the operating condition of the vehicle in order to pin point condensation formation. Only one tube of the CAC is used for the experiment with top and bottom surfaces covered with another two tubes to get the exact distance between tubes in the CAC. Pressure, velocity, temperature and relative humidity are controlled at the inlet of the charge inlet to the tube. All experiment were carried out at the same time of the day in temperature and humidity controlled clean room. Inlet chamber is used to collect extra humidity in the charge air before it enters the tube. This chamber has two baffles which help to maintain uniform velocity, turbulence, and relative humidity for each case. Outlet box is also equipped with pressure, temperature, relative humidity measuring probes. Additionally, condensate which accumulates in the outlet chamber is drained and measured hourly.

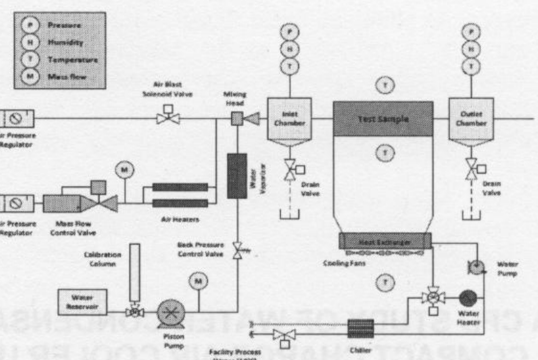

Figure 2: Block Diagram of Experimental Hardware

Three identical fans of capacity 120 CFM were used to create cooling air flow around the CAC tube. This cooling air has critical importance as it takes heat form the CAC tube and depending on the wet bulb temperature inside the tube condensation formation takes place. As cooling air has much lower velocity compared to the charge air inside tube hence surface heat transfer over width of the tube changes. Before each experimental run baseline test is performed. To make sure that the test data is consistent, each test is repeated at least 5 times. Before each test, the rig was run for 60 minutes so that the steady operation condition is achieved prior to data acquisition. Once steady state is reached, the temperature profile is obtained for another 60 minutes and this is repeated for consistency.

Table 1: Limiting Operating Parameters for the Experiment

Parameter	Maximum Operation Limit
Inlet Charge Air Temperature (^{0}C)	80
Pressure (kPa)	353
Inlet Relative Humidity (%)	100
Charge Air Mass Flow Rate (m^3/s)	0.85
Outlet Charge Air Temperature (^{0}C)	80

SIMULATION SETUP

Condensation and evaporation takes place at the boundary layer of the wall. It was therefore necessary to resolve at the viscus sub-layer of the tube wall, hence a very fine mesh was required. For this simulation, two different approaches were selected to analyze the film formation and wetting inside the tube. One approach was based on k-epsilon turbulence modeling of the flow inside the tube with Eulerian wall film model to resolve film momentum. As k-epsilon turbulence model is a time averaged method of transient simulation, it results in average film data. Hence in another strategy, Large Eddy Simulation was coupled with k-epsilon turbulence model to resolve all scales of turbulence. For unsteady condensation and evaporation modeling, LES model was used in both strategies. As heat exchanges between two air streams through Aluminum, if film is present, a thin layer of water condensate, conjugate heat transfer is important and needs spatial resolution. Hence, as shown in Figure 3, both sides of the Aluminium tube have fine mesh with 5 inflation layers with work with the law of wall to resolve boundary layer.

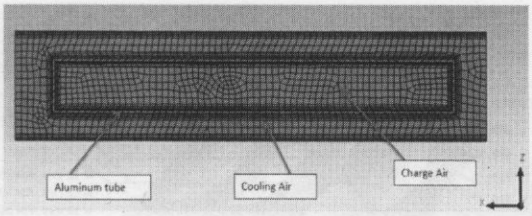

Figure 3: Initial Mesh Cross-section

Figure 3 shows the initial mesh used for the simulation. This mesh is adapted with addition of the another 200,000 cells in the boundary layer regime for each different inlet velocity condition to get fully resolved viscus sub-layer and to maintain y^+ for boundary layer between 0.5 and 1.

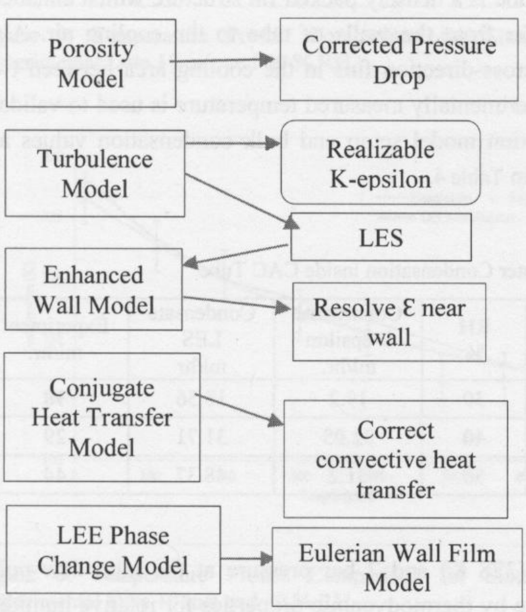

Figure 4: Schematics of the CFD Modeling

Figure 4 shows the overview of the entire CFD simulation modeling and the interaction between different models. It is important to mention that there are two turbulent models which

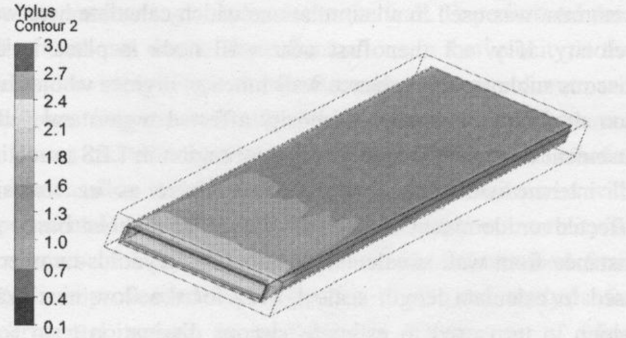

Figure 5: Yplus Criteria for Resolution of Viscous Sub-layer

are compatible with rest of the models without any significant change.

Figure 5 shows the Y plus for the LES cases, though it changes from case to case and also along the length from inlet to outlet, it is fairly constant around 1.6 value. Table 2 shows statistical data for the mesh which is used for this study.

Table 2: Mesh Parameters

CAC Mesh Properties			
Minimum Face Size	9.66e-002 mm	Inflation Growth Rate	1.2
Maximum Face Size	1.0 mm	Maximum Aspect Ratio	10.652
Transition Ratio	0.272	Number of Nodes	2,106,258
Maximum Inflation Layers	5	Number of Elements	2,040,642
y^+	1-3	Inflation Layers	5

$$y^+ = \frac{y}{l^*} \quad (1)$$

where y^+ is the measure of boundary layer.

$$u_* = \sqrt{\frac{\tau_w}{\rho}} \qquad l_w = \frac{\nu}{u_*}$$

According to Pope [9], the boundary layer is divided into three sub-layers and the associated y^+ values are given below,

Viscous sub-layer $0 < y^+ < 5$
Buffer sub-layer $5 < y^+ < 30$
Fully turbulent sub-layer $30 < y^+ < 400 \left(\frac{y}{\delta} = 0.1 - 0.2\right)$

Initial condition for the condensate film was obtained by running standard k-epsilon turbulence model with Lee model and conjugate heat transfer model for 20 minutes of real time. This preliminary simulation provides steady turbulence and wall boundary conditions for LES simulation. Enhanced wall treatment was used in all simulations which calculate near wall velocity. If $y^{+} \approx 1$ then first near wall node is place in the viscous sublayer and enhance wall function divides whole fluid domain into two regimes, viscosity affected region and fully-turbulent region. In the fully turbulent region in LES modeling all intermediate eddies are resolved where as in viscosity affected or dominant region low-Reynolds-number based on distance from wall is calculated. This low-Reynolds-number is used to calculate length scale l_ε [10] for the flow near wall, which in turn used to estimate viscous dissipation term $\mathcal{E}$ as shown in equation (2).

$$\varepsilon = \frac{k^{3/2}}{l_\varepsilon} \tag{2}$$

In this simulation Eulerian-Eulerian approach was used and no discrete droplets of water were considered to initiate condensation but which may form due to stripping of the film. Eulerian wall film model works with LES phase change model and uses mass source term to calculate the height of the film. To accurately predict the film formation sites inside tube Eulerian wall film model is coupled with mixture species transport model. So that a small change in the local relative humidity and velocity can be detected and will affect wall film. The Weber number is used as a switch between film stripping and accumulation and growth of film thickness. [11]

EXPERIMENTAL RESULTS

CAC tube inlet velocities of 12 m/s, 10 m/s, and 8 m/s are chosen for the experiment as these velocities represent the cruising speed of the vehicle. These velocities were observed in the wind tunnel as well as in the full vehicle aerodynamic simulation. Air velocity relative to the vehicle is 70 miles/hour on the outer surface of the vehicle but due to small cross-section area of the front grill and various powertrain cooling-pack components it reduces mostly in the range of the values of 2-4 m/s. Also at these cross-flow velocities condensation is observed in real case scenario and hence these values are intrinsic to the test matrix shown in Table 3. Outlet temperature requirement of the internal combustion engine is 25 ^{0}C and the temperature of inlet air which is outlet of turbocharger is constant at 55 ^{0}C. Test is carried out to closely observe condensation at constant vehicle speed which returns constant cooling air velocity 2 m/s.

Figure 6 explains temperature change inside the CAC tube measured by thermocouples embedded on either side of the tube. This is an average temperature over steady state experiment temperature data. Temperature measured by the thermocouples only gives idea about the bulk air energy. Inside

Table 3: Experimental Cases

Velocity m/sec	Temp K	RH %	Cooling Air m/s	Temp $_{out}$ ^{o}C
10	328	30	2	298
10	328	40	2	298
10	328	50	2	298
12	328	30	2	298
12	328	40	2	298
12	328	50	2	298
8	328	30	2	298

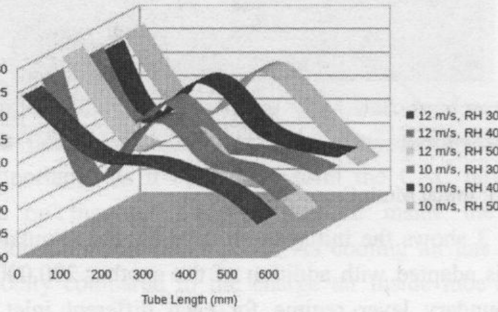

Figure 6: Experimental Temperature Profile along Length of CAC Tube

the CAC tube is a densely packed fin structure which enhances heat transfer from the walls of tube to the cooling air. Also there are cross-direction fins in the cooling area between two tubes. Experimentally measured temperature is used to validate the simulation model setup and bulk condensation values are compared in Table 4.

Table 4: Water Condensation inside CAC Tube

Velocity m/s	RH %	Condensate k-epsilon ml/hr.	Condensate LES ml/hr	Experiment ml/hr.
12	30	19.2	18.56	18
12	40	32.05	31.71	29
12	50	51.2	48.37	44

At 55 ^{0}C (328 K) and 2 bar pressure at the inlet dew point temperature by thermodynamic properties for relative humidity is 31.88 ^{0}C (304 K). As the CAC is designed to get 25 ^{0}C at outlet of tube, the inside surface temperature of the tube is a deciding factor for condensation and also provides site for film accumulation and growth. Experimental data shows in Figure 6 around at the middle of the tube temperature reaches value lower than 305 K. Experimental temperature is recorded at 2 mm inside the surface, this position falls in the turbulent region and does not provide accurate information of near surface temperature.

ANALYSIS

The simulation using two different turbulence models, namely realizable k-epsilon and Large Eddy Simulation (LES) predicted the temperature profile inside the tube and is also validated using experimental data. Figure 7 and Figure 8 show the comparison between experimental and simulated temperature data.

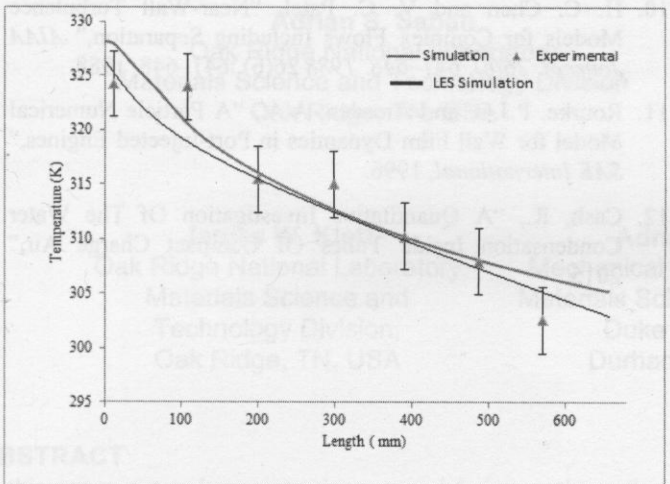

Figure 7: Temperature Profile Comparison for Simulation and Experimental Data 10 m/s and 30 % RH

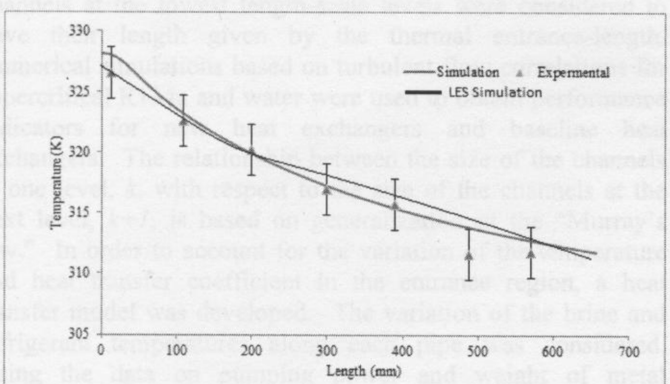

Figure 8: Temperature Profile Comparison for Simulation and Experimental Data 10 m/s and 40 % RH

Large Eddy Simulation is unsteady state turbulence whereas realizable k-epsilon provides statistically steady turbulence. Hence, all LES results are varying in nature but still show good comparison with k-epsilon results. This is due to the use of enhance wall function which uses epsilon correction and resolves laminar region adjacent to the wall. Experimental data varies as shown due to slight fluctuations in the velocity at the inlet and swirling effect of fans. Error bars show maximum and minimum temperature obtained during 60 minutes of test run. As shown in Table 4 using mass-average method total amount of condensation is calculated for one hour and compared with experimental data taken for 12 m/s and relative humidity in increments of 10%. The condensation amount obtained from LES is improvement over k-epsilon. Full error analysis for experimental results is available in Chapter 6 of sited dissertation [12]. Large Eddy Simulation helps predicting unsteady realistic condition and also compensate for the noise factor in the experimentation, whereas k-epsilon provides time-averaged turbulence data and hence useful for less computationally expensive simulation as per the previous study by authors [1].

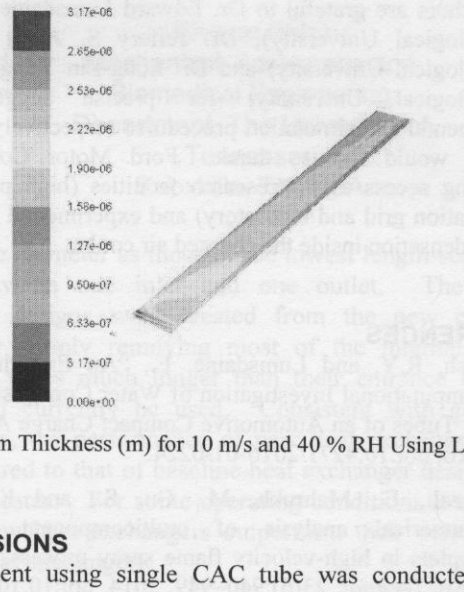

Figure 9: Film Thickness (m) for 10 m/s and 40 % RH Using LES

CONCLUSIONS

1. Experiment using single CAC tube was conducted in a controlled environment to get temperature profile across length of the tube. Also after every test for different case water condensate was measured.

2. As few important aspect of condensation occurrence inside tube like film thickness and film movement are difficult to measure experimentally due to lack of space for instrumentation 3D CFD simulation model is created.

3. 3D CFD model for condensation was validated for two different turbulence schemes namely realizable k-epsilon and Large Eddy Simulation (LES) using experimental temperature profile and amount of condensate collected. Viscous sub layer near wall was fully resolved for turbulent kinetic energy.

4. After validation of 3D model, thickness of water condensate film is modeled and analyzed. K-epsilon and LES both give

satisfactory results in terms of accumulated condensate amount and temperature profile.

5. LES is computationally expensive compared to k-epsilon but gives more accurate results. Also LES calculates for unsteady state film and works with Lee model to simultaneously calculate evaporation and condensation. A detailed model to simulate film development for heat exchanger is created and can be used to design CACs.

Contact Information
Dr. Robin Cash
Ford Motor Company
15011 Commerce Drive
Dearborn, MI 48120
rcash@ford.com

ACKNOWLEDGMENTS

The authors are grateful to Dr. Edward Lumsdaine (Michigan Technological University), Dr. Jeffery S. Allen (Michigan Technological University) and Dr. Song-Lin Yang (Michigan Technological University) for precise suggestions in experimental and simulation procedures respectively. Also, the authors would like to thank Ford Motor Company for providing access to the research facilities (high performance computation grid and laboratory) and experimental test data of the condensation inside the charged air cooler.

REFERENCES

1. Cash, R.Y. and Lumsdaine, E., "An Experimental and Computational Investigation of Water Condensation inside the Tubes of an Automotive Compact Charge Air Cooler," 2016, doi:10.4271/2016-01-0224.
2. Gozali, E., Mahrukh, M., Gu, S., and Kamnis, S., "Numerical analysis of multicomponent suspension droplets in high-velocity flame spray process," *J. Therm. Spray Technol.* 23(6):940–949, 2014, doi:10.1007/s11666-014-0106-1.
3. Talekar, A.P., Lai, M., Zeng, K., and Yang, B., "Simulation of Dual-Fuel-CI and Single-Fuel-SI Engine Combustion Fueled with CNG," ISBN 2016010789, 2016, doi:10.4271/2016-01-0789.Copyright.
4. Contino, F., Masurier, J.B., Foucher, F., Lucchini, T., D'Errico, G., and Dagaut, P., "CFD simulations using the TDAC method to model iso-octane combustion for a large range of ozone seeding and temperature conditions in a single cylinder HCCI engine," *Fuel* 137:179–184, 2014, doi:10.1016/j.fuel.2014.07.084.
5. Selim, M.Y.E., "Effect of engine parameters and gaseous fuel type on the cyclic variability of dual fuel engines," *Fuel* 84(7-8):961–971, 2005, doi:10.1016/j.fuel.2004.11.023.
6. Huo, M., Lin, S., Liu, H., and Lee, C.F.F., "Study on the spray and combustion characteristics of water-emulsified diesel," *Fuel* 123:218–229, 2014, doi:10.1016/j.fuel.2013.12.035.
7. Dalkilic, A.S. and Wongwises, S., "Intensive literature review of condensation inside smooth and enhanced tubes," *Int. J. Heat Mass Transf.* 52(15-16):3409–3426, 2009, doi:10.1016/j.ijheatmasstransfer.2009.01.011.
8. Tang, Y., "The Condensation within a CAC - Thermodynamics Analysis," ISBN 0148-7191, 2011, doi:10.4271/2011-01-1168.
9. Pope, S.B., "Turbulent Flows," Cambridge University press, ISBN 978-0521598866, 2000.
10. H. C. Chen and V. C. Patel, "Near-Wall Turbulence Models for Complex Flows Including Separation," *AIAA Journal. 26(6). 641–648. 1988* 26(6).:641–648, 1988.
11. Rourke, P.J.O. and Amsden, A.A., "A Particle Numerical Model for Wall Film Dynamics in Port-Injected Engines," *SAE International*, 1996.
12. Cash, R., "A Quantitative Investigation Of The Water Condensation Inside Tubes Of Compact Charge Air," 2015.

