

Effects of Cavitation and Hydraulic Flip in 3-Hole GDI Injectors

Mathis Bode, Tobias Falkenstein, Marco Davidovic, and Heinz Pitsch
 RWTH Aachen University

Hiroyoshi Taniguchi, Kei Murayama, and Toshiyuki Arima
 Honda R&D Co., Ltd.

Seoksu Moon
 National Institute of AIST

Jin Wang
 Argonne National Laboratory

Akira Arioka
 Keihin Corp.

ABSTRACT

The performance of Gasoline Direct Injection (GDI) engines is governed by multiple physical processes such as the internal nozzle flow and the mixing of the liquid stream with the gaseous ambient environment. A detailed knowledge of these processes even for complex injectors is very important for improving the design and performance of combustion engines all the way to pollutant formation and emissions. However, many processes are still not completely understood, which is partly caused by their restricted experimental accessibility. Thus, high-fidelity simulations can be helpful to obtain further understanding of GDI injectors. In this work, advanced simulation and experimental methods are combined in order to study the spray characteristics of two different 3-hole GDI injectors. First, a simulation approach for computing cavitation and hydraulic flip is presented, which appropriately combines simulations with different levels of abstraction allowing for predictive GDI injector simulations on currently available supercomputers. Next, general resolution requirements and various initial conditions are discussed. Especially the effect of initial bubbles inside the sac on the cavitation formation process is investigated by performing a transient simulation with moving needle. Finally, coupled compressible Large-Eddy Simulations (LES) of the internal nozzle flow with and without models for cavitation and hydraulic flip, Direct Numerical Simulations (DNS) of the resulting primary breakup, and far-field Lagrangian Particle-based LES (LP-LES) are performed in order to achieve a detailed data set of the injection process of two different 3-hole GDI injectors. X-ray measurements of in-nozzle cavitation as well as droplet size distribution obtained by Phase Doppler Particle Analyzer (PDPA) measurements in the far-field are used for validation and understanding the impact of the different effects on the underlying breakup processes. Furthermore, sensitivities between nozzle design features and hydraulic flip are investigated.

CITATION: Bode, M., Falkenstein, T., Davidovic, M., Pitsch, H. et al., "Effects of Cavitation and Hydraulic Flip in 3-Hole GDI Injectors," *SAE Int. J. Fuels Lubr.* 10(2):2017, doi:10.4271/2017-01-0848.

INTRODUCTION

Gasoline Direct Injection (GDI) is one way to increase thermal efficiency of gasoline engines due to stratification of gasolineoxidizer mixtures. Thus, optimizing injector geometries resulting in advantageous fuel-oxidizer mixing has become a very important topic in current GDI engines, since it directly effects their efficiency and emissions. Especially, compressibility effects inside the nozzle such as cavitation and hydraulic flip, which are highly dependent on the injector geometry, and their interplay with the primary breakup in the vicinity of the nozzle are hardly understood yet, but affect multiple other physical

processes downstream [1,2]. Experimental as well as simulation studies of these processes are very challenging due to the overall complexity and the involved small length and time scales [3, 4, 5, 6, 7].

The lack of understanding regarding the impact of the injector design on cavitation and hydraulic flip and the respective influence of these phenomena on the primary breakup behavior is also a big challenge for far-field spray simulations [8, 9, 10, 11, 12, 13, 14, 15]. Since models for primary breakup are not at a predictive stage yet, it is very difficult to initialize commonly performed Lagrangian Particle-based

(LP) simulations. In fact, the initial conditions as well as model parameters often have to be tuned with experimental results and thus fully a-priori predictions of new injector designs are not possible.

In order to overcome these issues, Eulerian models have been developed for internal nozzle flow simulations on modern High Performance Computing (HPC) systems recently. These can be assigned to the two major categories of homogeneous mixture models and multi-fluid models.

In homogeneous mixture models, all phases are assumed to form a mixture with a single velocity and a single pressure. Thus, only one set of governing equations is solved, usually the compressible Navier-Stokes equations. Most models rely on the assumption of thermodynamic equilibrium. The occurrence of cavitation is determined based on a cavitation criterion, in the simplest case just by means of the local pressure in comparison with the vapor pressure. Dabiri et al. [16] compared this simple criterion with an advanced total stress criterion including both pressure and viscous stress. They concluded that viscous stress significantly contributes to the size of cavitating regions and leads to earlier cavitation inception.

The appropriate formulation of an Equation Of State (EOS) that covers all states in the liquid and vapor phase is important for homogeneous mixture models. Goncalves and Patella [17] implemented a stiffened-gas equation of state that is valid for both the liquid and vapor phases and compared it to a sinusoidal barotropic EOS. Even though steady-state results of both models were in good agreement with experiments, they failed to adequately predict unsteady test cases in the context of the used Shear Stress Transport (SST) $k-\omega$ turbulence model. Applying a similar barotropic model, Im et al. [18] found a relationship between nozzle geometry and cavitation intensity, i.e. cavitation occurs only if the inner-nozzle curvature radius is below a critical value. These cavitation models only considered a liquid and a vapor phase. Since non-condensable gas from the combustion chamber is known to be responsible for hydraulic flip, Örley et al. [19] extended the barotropic homogeneous models by the addition of non-condensable gas as an additional phase, ending up with a two-phase two-fluid model, but still relying on the homogeneous mixture assumption.

A slightly adapted type of homogeneous model is the so-called Homogeneous Relaxation Model (HRM). In contrast to homogeneous equilibrium models, the mixture is not assumed to be in thermodynamic equilibrium. Instead, the relaxation of the vapor mass fraction to its local equilibrium value is accounted for by an additional equation based on a specific relaxation time scale [20, 21, 22].

Since the treatment of the different phases in cavitating flows as one mixture simplifies its actual multi-phase behavior, Eulerian multifluid models were developed where each phase is treated separately with an own set of governing equations. The coupling between the phases including mechanical interaction as well as heat and mass transfer is challenging and requires sophisticated strategies. Saurel et al. [23] set up a two-phase cavitation model based on the well-known Baer and Nunziato two-phase flow model. They assumed stiff mechanical relaxation towards

equilibrium, thus the momentum and energy equations of both phases reduce to one equation each for all phases with a single local pressure and local velocity, while still one separate mass conservation equation is solved for each phase. Heat and mass transfer was included through relaxation terms derived from the principle of equilibrium entropy maximization. The theory can also be generalized for flows with more than two phases [24]. Wang et al. [25] adapted the two-phase model to perform simulations of injector-like liquid jets, considering air as an additional component in ideal gas mixture with the vapor phase. However, hydraulic flip was not discussed explicitly.

Habchi [26] adopted the basic multifluid methodology of Saurel et al. [23] and also included an additional phase of non-condensable gas. In contrast to previous multifluid models, phase change was not assumed to happen instantaneously here. 3D computations were performed for multiple test cases. From the simulations results, two different regimes were identified: A gaseous cavitation regime was observed when the static pressure was close to but above the saturation pressure. Once the pressure fell below the saturation pressure, phase change occurred leading to a vaporous cavitation regime.

Wang et al. [27] conducted a numerical study of cavitating flows inside the conical-spray nozzle and found four regimes of cavitation, one of which was named super cavitation regime [28,29]. A flow enters this regime when the ambient pressure is very low thus making the flow independent of the Reynolds number and purely dependent on the cavitation number. Andriotis et al. [30] have investigated diesel injector nozzle for a cavitating flow. They found that cavitating structures originate from pre-existing cavitation sites forming at sharp corners inside the nozzle where the pressure falls below the vapor pressure of the flowing liquid. Dabiri and Sirignano [31] and Vahedi Tafreshi and Pourdeyhi [32] studied hydraulic flip through numerical and experimental work, respectively. It was found that hydraulic flip is observed for flows with high Reynolds numbers through orifices with small length-to-diameter ratios and with sharp corners. It has also been observed that cavitating regions shrink or even disappear when hydraulic flip occurs. Nurick [33] found from an experimental study that hydraulic flip is avoided only if the injector entrances are not perfectly sharp.

Most previous studies focus on in-nozzle flow only and do not consider its influence on primary breakup and processes further downstream. Besides, hydraulic flip is still not considered in many approaches even though its crucial impact on the in-nozzle flow is known. This motivates the incorporation of current models for cavitation and hydraulic flip into a simulation framework for coupled injector simulations in this work. Internal Nozzle Flow Large-Eddy Simulations (INF LES) considering the exact injector geometry, highly resolved Primary Breakup Direct Numerical Simulations (PB DNS) in the vicinity of the nozzle, and Far-Field Lagrangian Particle-based Large-Eddy Simulations (FF LP-LES) downstream are combined in order to achieve detailed data of two GDI 3-hole injectors without tuning the results to experimental data. In combination with X-ray and Phase Doppler Particle Analyzer (PDPA) measurement data, this dataset is used to

understand the importance of cavitation and hydraulic flip for GDI simulations and make reliable conclusions with respect to the underlying physics. Furthermore, the data set is used to estimate sensitivities between nozzle design features and hydraulic flip.

The remainder of this article is organized as follows. The target GDI injectors used in this work are described in the next section. Then, two experimental techniques, applied for this work in order to obtain reference data, are briefly introduced. Next, the coupled simulation approach as well as single numerical methods and models are discussed. In the “Results and Discussion” section, the correct initialization of the coupled injector simulations is investigated and the results of the simulations are compared to measurements in order to understand the impact of the different simulation models on the breakup process. Also, sensitivities between nozzle design features and hydraulic flip are estimated. The paper finishes with conclusions.

CASE DESCRIPTION

This work focuses on two 3-hole GDI injectors operated at the same conditions and with very similar geometries as shown in [Figure 1](#) and [2](#). While the hole arrangement of injector N4X3 is completely symmetric with three inwards (towards the sac) tilted holes, the orientation of each hole is different for injector N145. The body and needle as well as the orientation of hole 2 are very similar for both injectors, but clear differences are given for holes 1 and 3. More precisely, hole 1 of N145 (denoted as N145-1) is straight and N145-3 is tilted outwards. An X-ray visualization of N145-1 made before start of injection can be seen in [Figure 3](#). Also, further nomenclature is introduced there including hole length L , diameter D , and inside (sac side) as well as outside inclination angles ϕ_1 and ϕ_0 . Exact values for these quantities can be found in [Table 2](#).

The operating conditions are governed by five different dimensionless numbers: First, the liquid Reynolds number defined by nozzle outlet diameter, nozzle exit velocity, and kinematic viscosity of the liquid phase. Second, the Weber number of the liquid phase calculated with the nozzle outlet diameter, nozzle exit velocity, density of the liquid, and the surface tension coefficient. Third, the density ratio of liquid and gas phase. Fourth, the dynamic viscosity ratio of liquid and gas phase. Fifth, the cavitation number defined as difference between fuel injection pressure and vapor pressure divided by the difference between fuel injection pressure and ambient pressure. Assuming the same flow rate for all of the three holes, neglecting needle opening and closing effects, and using Bernoulli's equation, the nozzle bulk exit velocity can be estimated as 125.6 m/s and all dimensionless numbers describing both cases can be calculated. The exact injection conditions as well as the resulting characteristic non-dimensional numbers can be found in [Table 2](#).

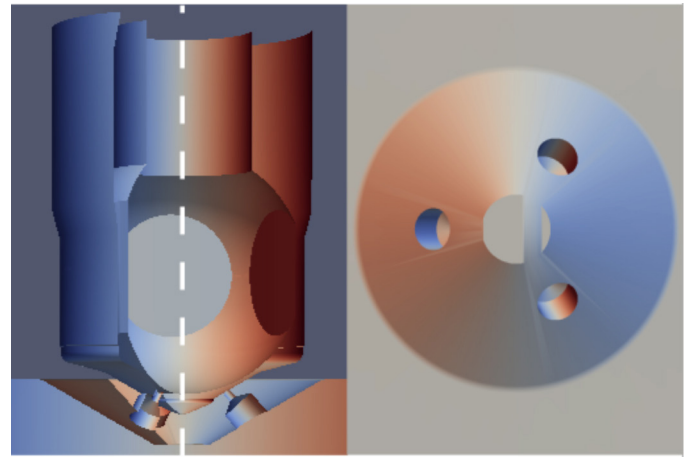


Figure 1. Geometry of N4X3 injector.

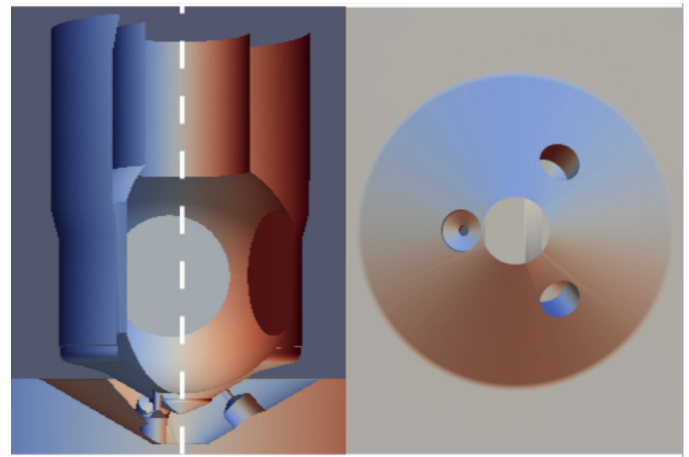


Figure 2. Geometry of N145 injector.

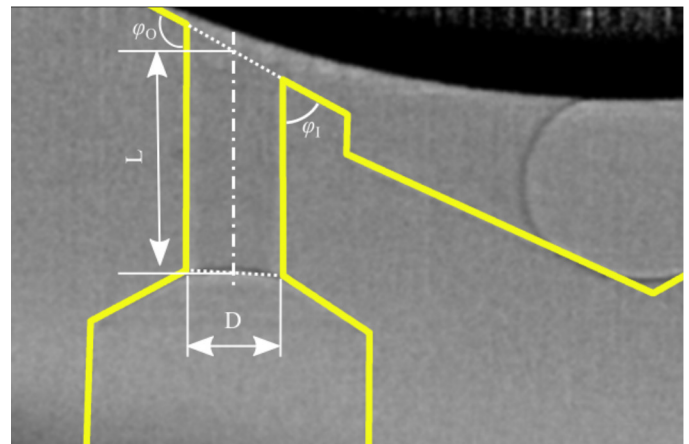


Figure 3. X-ray visualization of N145-1 made before start of injection.

Table 1. Nozzle geometry details.

	D	L/D	ϕ_1	ϕ_0
N4X3-1	0.13 mm	1.08	77°	103°
N4X3-2	0.13 mm	1.08	77°	103°
N4X3-3	0.13 mm	1.08	77°	103°
N145-1	0.13 mm	2.38	60°	120°
N145-2	0.13 mm	0.92	80°	100°
N145-3	0.13 mm	1.31	100°	80°

Table 2. Summary of the case conditions and characteristic dimensionless numbers.

Fuel	n-heptane
Fuel temperature at nozzle exit	295.0 K
Fuel injection pressure	15000000.0 Pa
Fuel density ρ_{liquid}	668.3 kg/m ³
Fuel dynamic viscosity η_{liquid}	410.0*10 ⁻⁶ kg/(m*s)
Ambient gas temperature	295.0 K
Ambient gas pressure	101325.0 Pa
Ambient gas density ρ_{gas}	1.2 kg/m ³
Ambient gas dynamic viscosity η_{gas}	18.3*10 ⁻⁶ kg/(m*s)
Ambient gas velocity	near-quiescent
Surface tension coefficient	20.0*10 ⁻³ kg/s ²
Injection time	2.0 ms
Reynolds number (liquid)	26600
Weber number (liquid)	68500
Density ratio $\rho_{\text{liquid}}/\rho_{\text{gas}}$	558.2
Viscosity ratio $\eta_{\text{liquid}}/\eta_{\text{gas}}$	22.4
Cavitation number	1

The stated fuel density, fuel dynamic viscosity, and surface tension in Table 1 are taken from the “Chemical Properties Handbook” by Yaws [34] as data of n-heptane at 295 K and 15 MPa. The properties of the ambient gas are taken from the NIST Chemistry WebBook [35] as data of air at 295 K and 0.101325 MPa.

EXPERIMENTAL METHODS

Following the combined experimental and simulation approach of this work, highly resolved experimental data were generated in order to gain insight into the mechanism of hydraulic flip and validate simulation results. A Phase Doppler Particle Analyzer (PDPA) technique was used to measure the droplet size distributions at three different downstream locations. Since it is commonly known, further details are not given here. In contrast, the applied X-ray methods are unique and briefly summarized in this section in order to enable correct interpretation of measurement data in the “Results and Discussion” section.

The X-ray measurements of the two-phase flows inside and near the injector nozzles were visualized using the X-ray beam at the Advanced Photon Source (APS). This X-ray beam was used due to the following two reasons: One is the high energy of the X-ray beam that enables the X-ray beam to pass through both metal nozzles and the dense region of spray although some portion of the initial energy is absorbed by the metal. The other is the sub-nanosecond X-ray pulses providing a high temporal resolution of the X-ray images that enables to freeze the motion of high-speed features existing inside and near the nozzles. An X-ray Phase-Contrast Imaging (XPCI) technique was used for the visualization of in- and near-nozzle flow images. Moon et al. [36,37] describe all details of XPCI such as the exact experimental setup.

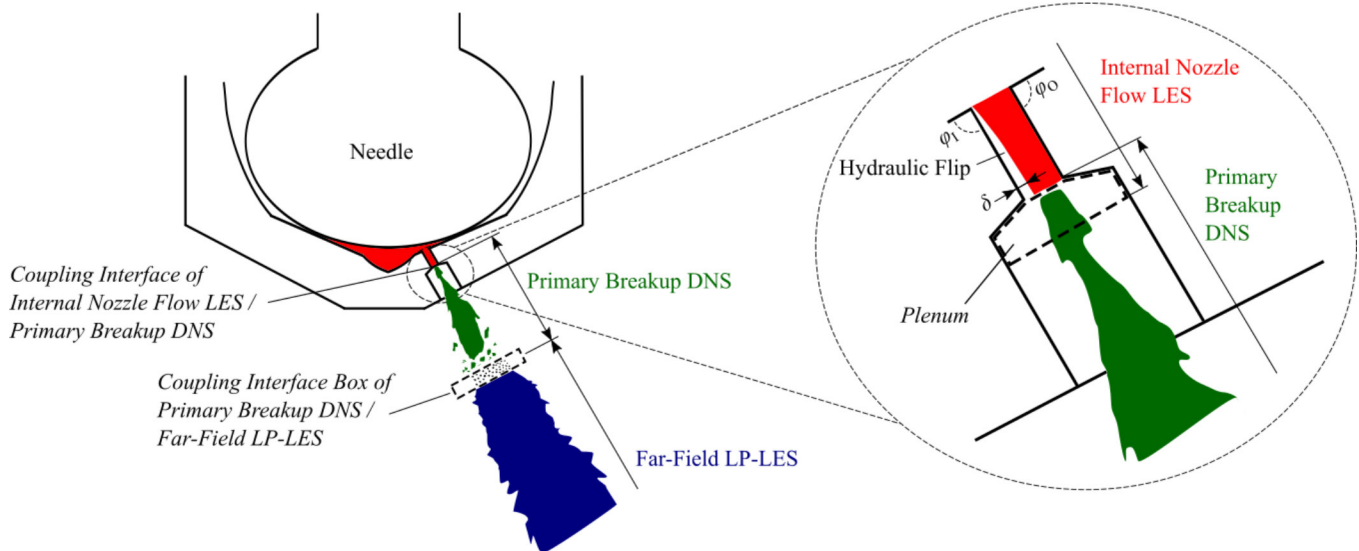


Figure 4. Sketch of the simulation approach including the plenum for the INF LES. Also, the gas height δ is indicated, which measures hydraulic flip.

An X-ray pulse with a 150 ps duration and a 16 mA current was used for the visualization of in- and near-nozzle flow with a high temporal resolution. On the other hand, three X-ray pulses with a 68 ns period were used for the multi-exposed imaging to analyze the dynamics of emerging jet flow by the auto-correlation analysis of the multi-exposed X-ray images. The detailed methodology to analyze the local flow velocity from the multi-exposed X-ray images has been documented by Moon et al. [37].

SIMULATION APPROACH

Simulations of full injection systems are very challenging for multiple reasons. One example is the wide range of time and length scales, which needs to be considered. While the penetration length is of the order of 100 mm, resulting small droplets are of the order of 1 μm and turbulent and boundary layers often require the resolution of even smaller flow structures due to high Reynolds numbers. Additionally, the droplet formation process in the primary breakup area is very difficult due to often high density jumps across a theoretically infinitely small interface. Another problem is the complex interplay of various flow phenomena such as shocks at needle opening, nucleation of cavitation bubbles, dissolved gas in the liquid fuel, or gaseous cavitation inside the nozzle. One way to deal with this issue is to combine simulations with different levels of abstraction in a consistent manner as described by Bode et al. [38,39]. This allows tailoring of the simulation approach in each physically-defined subdomain based on the most important flow features and results in locally accurate simulations while the overall computing cost is reduced to an acceptable level.

An application of Bode et al.'s coupling approach to the GDI injectors considered here is shown in Figure 4. For each injector, a compressible Internal Nozzle Flow Large-Eddy Simulation (INF LES) considering the exact injector geometry and focusing on one of the three holes as target hole is coupled to a Primary Breakup Direct Numerical Simulation (PB DNS) in the vicinity of the nozzle, which transfers droplet information to a Far-Field Lagrangian Particle-based Large-Eddy Simulation (FF LP-LES). For details about the implementation of the coupling interfaces and the consistent transfer of data with different abstraction levels, the reader is referred to Bode et al. [38, 39, 40]. An overview of the resulting coupled simulations is given in the remainder of this section, details about particular important models and numerical methods are given in the next section. More discussions about the resolution requirement for each simulation follow at the beginning of the "Results and Discussion" section.

The INF LES use no-slip boundary conditions at the walls and almost 300 million cells each. Due to the structured mesh, this results in 40 points across the diameter for the target hole in each simulation and less resolution for the other two holes. This is in agreement with the internal flow simulations shown at the third Engine Combustion Network (ECN) meeting [41]. Due to the high speed of sound, the resulting CFL number turned out to be smaller than 0.1. In order to study the impact of cavitation and hydraulic flip, respective models are available for the INF LES, which can be independently switched on in each run. In order to have a well-defined exit boundary condition, the INF LES domain is extended by a counterbore-shaped

plenum as indicated in Figure 4. This avoids exit boundary conditions, which allow gas entrainment as would have been required at the INF LES/PB DNS coupling interface.

The PB DNS is computed as liquid jet into a rectangular plenum of size 2.34 mm $\times$ 1.17 mm $\times$ 1.17 mm and zero gradient boundary conditions. It uses a special interfacial flow solver in order to allow accurate breakup simulations for arbitrary density ratios. On one side of the plenum, the velocity profile of the internal nozzle flow is imposed for simulating the nozzle exit. Experiments by Crua and Heikal [42] showed that the initial interface at the nozzle exit is mushroom shaped for valve-covered orifice nozzles in combination with heavier fuels. This simple shape is corrected by information from the INF LES in order to consider hydraulic flip and then imposed as initial condition within this study. More precisely, the Level Set (LS)- and Volume of Fluid (VoF)-fields are initialized such that they represent this corrected geometry. More than 900 million cubic cells are used for the PB DNS resulting in a resolution of roughly 1.5 μm and a CFL number of about 0.3. This resolution is comparable to the resolution of the experiments and thus sufficient within this study. Also, it is in good agreement with the recommendations by the ECN [43].

Finally, the droplet information such as diameter and velocity of the PB DNS is transferred to the FF LP-LES 2 mm downstream of the nozzle exits. This avoids the typically needed guess of an initial droplet size distribution at the nozzle orifice for commonly used Lagrangian Particle Simulations and tuning of parameters by means of experimental data. The FF LP-LES use a uniform mesh of the size 160 mm $\times$ 80 mm $\times$ 80 mm with roughly 20 million cells each. This results in a resolution of 0.1 mm, which is finer than what is typically used for the Spray G case of the ECN [43], that is comparable to the cases considered here.

Overall, each of the performed GDI simulations resulted in high-fidelity data of the nozzle internal flow, the primary breakup, and the spray characteristics far from the nozzle orifice. Each GDI simulation costs less than 10 million core-h due to the use of the coupled approach and the commonly used approach of calibrating the breakup model to experimental data was not required. Although this is still quite expensive compared with typical Lagrangian spray simulations currently used in industry, the advantage of the present approach is a higher level of predictivity and completeness of the results.

NUMERICAL METHODS AND MODELS

All simulations were performed using an in-house code called CIAO. It solves the Navier-Stokes equations along with the effects of multiphysics. CIAO is a structured, second order, finite difference code [9,38,44, 45, 46, 47] that enables coupling and computation of multiple domains simultaneously. Within the code framework, a fully compressible and an incompressible/low-Mach solver are available. In order to increase the accuracy of the stencil, staggered grid arrangement was employed in the spatial dimensions. The sub-filter model for the momentum equations is an eddy viscosity concept in the form of dynamic Smagorinsky model [48] with Lagrangian averaging along fluid particle trajectories [49]. A similar model for