INTRODUCTION
There are many technologies to improve the engine performance in recent years. And turbocharger devices enable the improvement of engine performance efficiently, including fuel economy, emission, and dynamic properties, and are regarded as the second milestone in the history of the development of the internal combustion engine. The compressor performance is critical for the turbocharger performance and to a degree for the engine performance itself.

Quantitative computational fluid dynamic analyses of oil droplets deposition on vaneless diffusor walls of a centrifugal compressor

Liwei Sun1,2 | Chaochen Ma1 | Panpan Song1 | Hang Zhang1

1School of Mechanical Engineering, Beijing Institute of Technology, Beijing, China
2College of Automotive Engineering, Dezhou University, Dezhou, China

Abstract
In the vehicle turbocharging operating process, the diffusor walls are susceptible to oil droplet deposition. This deposition may cause deterioration in the diffusor flow performance, consequently resulting in the deterioration of compressor performance. In this paper, a three-dimensional steady computational fluid dynamic (CFD) numerical investigation was performed to study droplet deposition on the walls of a centrifugal compressor diffusor. Firstly, the theoretical basis of oil droplet distribution and formation on the diffusor were presented. On the basis of these, the distribution rules of the capture efficiency and particle concentration for the diffusor were investigated considering six different droplet diameters (0.25 μm, 0.5 μm, 0.75 μm, 1.00 μm, 1.25 μm, 1.5 μm). The results indicate that the particle mass concentration on the hub side increased gradually, and the concentration on the shroud side decreased inversely. Furthermore, the area of high particle mass concentration focused on the small radius area of the hub side or at the downstream of the channel in the direction of the streamline. And the particle mass concentration on the shroud is equally distributed. Finally, the combination of turbulent diffusion and inertial theory was proposed to analyze the mentioned phenomenon, and the droplet deposition mechanisms on the centrifugal compressor diffusor were summarized.

KEYWORDS
centrifugal compressor, computational fluid dynamic, diffusor, droplet deposition, fouling

1 | INTRODUCTION

After a period of operating the turbocharged engine, especially for those engines with crankshaft ventilation systems, deposition occurs of automobile oil on the walls of the compressor diffusor. Under the influence of the flow stream and high temperature in the compressor, the automobile oil undergoes a series of physical and chemical processes before depositing on the diffusor walls. Deposition accumulation on the diffusor walls would cause narrowing of the diffusor channel and coarsening of the walls. Therefore, the flow resistance in the channel would increase. This may cause
compressor performance deterioration, consequently resulting in deterioration of the engine performance. In order to improve compressor performance, the following studies may be effective: researching the fouling mechanism on the diffusor surfaces of the compressor, determining the diameter range of the droplets which are easily deposited on the diffusor, studying the influence of the compressor operating environment on the droplet deposition, etc.

Of the studies concerning fouling in turbomachinery, few studies have been done about the vehicle compressor, but much research has been conducted on the aero-engine, marine engine, and industrial gas turbine engine. Most of these studies examined turbocharger deposition using the methods of increasing the thickness of the blade, increasing the wall roughness, and changing the local area geometry. Furthermore, to consider the effects of resultant changes in the passage geometry due to the accumulation of deposited particles, a mesh morphing approach was taken to simulate the particle deposition distributions. This work was conducted on an iterative basis to modify the surface topology by altering the locations of surface nodes. Meanwhile, both droplets and solid particles were researched with regard to deposition in the turbomachinery and relative two-phase flow movement.

For the droplet deposition on the turbomachinery walls, particle size is a critical factor. Therefore, the numerical model’s sensitivity for particle size was researched by Forsyth et al. In this research, the discrete random walk (DRW) model and continuous random walk (CRW) model were compared in the number simulation of the deposition in the gas turbine engines. The results showed that the CRW model was more sensitive than the DRW model, especially in the turbulent deposition. Starzmann et al. and Li et al. studied the influence on the droplet deposition rate of droplet diameter. The deposition mechanism was investigated for the blades considering different droplet diameters, with the inertia and turbulent diffusion included in the deposition mechanism as well. In addition, Mishra researched deposition and erosion in the aero gas turbine compressor in a coastal environment and pointed out that the droplets diameter range and the solid particle diameter range caused deposition and erosion.

Some researchers have studied droplet movement following blade impact. Williams et al. introduced a theoretical approach for simulating the movement of liquid water following deposition onto the turbomachine rotor blade. It was proposed that the aerodynamic shear force, but not the pressure force, may influence the water droplet motion. In Sun et al. research, the numerical simulation was carried out on two models of the NASA rotor 37, and the interaction of droplets and walls was analyzed with the theory of spray wall impingement. Two extreme computational models were used. In the spread model, the assumption of parabolic distribution of water film mass at the trailing edge may reflect the accumulation of film due to the centrifugal force. In the splash model, the droplet sizes after breakup were supposed to reduce gradually with the increasing impact angles.

Some reviews described the droplet deposition mechanism about the blades. Crane, Guha, Kurz, and Suman et al. all studied the droplet deposition mechanism of inertia and turbulence in turbomachinery. Crane RI also described the test details and simulation calculations in the steam turbines.

Besides the simulation papers and reviews, there are also several experimental papers on droplet deposition. Eisfeld et al. presented the behavior of air-water two-phase flow in a transonic compressor’s rotor blade passage to understand the detailed processes in wet compression operation. Results show that the fluid phase has little impact on the airflows. The droplet impingement on the leading edge of the blade may cause a splitting into many small droplets due to splashing. Droplets of larger diameter originate from the trailing edge of the blade as consequence of the water accumulating on the blade surface. In the research of Brun et al., the particle image velocity (PIV) testing was carried out in a wind tunnel. These test results may enable the model to be more optical and may present the influence of the particles on the turbomachine more accurately.

As compared with the literature concerning droplet deposition, there exist even more studies about solid particle deposition. Generally, the method to investigate the effects of fouling on the axial compressor blade was to impose different combinations of added thickness and surface roughness or impose different span-wise distributions of surface roughness. The research indicated that fouling may cause a drop in airflow, pressure ratio, compressor efficiency, and compressor performance, resulting in a “rematching” between the gas turbine and compressor, and a subsequent drop in power output and thermal efficiency. The fouling phenomenon in the compressor was caused by many factors, such as the number of particles, particle diameters, particle mass concentration, patterns of particle distribution, and particle mass flow rate. Meanwhile, the design parameters of the axial compressor also influenced the process of fouling formation, such as axial compressor chord length, solidity, roughness, and the number of stages. In addition, studies of the transport process of solid particles in an axial compressor and the kinematic characteristics of particle deposition were helpful in researching the mechanism of solid particle deposition.

In conclusion, most of the studies concerning droplet deposition on the turbomachinery were about the steam turbine, gas turbine, and axial compressor. However, little research has been devoted to the centrifugal compressor. Furthermore, few studies have been done on the vehicle turbocharger concerning droplet deposition. In this paper, we use three-dimensional computational fluid dynamic (3D CFD) steady simulation to investigate the phenomenon...
and mechanism of droplet deposition on the vehicle compressor diffusor. The trend of the capture efficiency and the particle concentration with different droplets diameters are investigated. Meanwhile, the contour plots and histograms of particle concentration are presented, and the distribution features are discussed. The analysis approach of combining the inertia impaction mechanism and the flow field analysis is proposed to investigate the deposition features presented above.

2 METHODOLOGY

2.1 Numerical model and validation

The numerical simulations were carried out by the commercial CFD code ANSYS FLUENT 15.0. Figure 1A shows the geometric model of the compressor. The diameter of the compressor impeller was 52 mm. The impeller was composed of six main blades and six splitter blades. However, in this research, a single passage was modeled as shown in Figure 1B. The purpose of the simulation with a single passage was to reduce the necessary calculations without the influencing accuracy. All the simulations were performed in the steady multiple frame of reference by using a frozen rotor interface. And the computational domains were composed of three domains: two stationary domains (the compressor inlet and the diffusor) and a moving domain (the impeller) as shown in Figure 1B.

The grid type of the model was the multiblock hexahedral grid, which was composed of 1 420 800 elements. The number of grid nodes was 1 489 443. The grid refinements, which were processed in tip clearance near the hub and the shroud, at the leading and the trailing edge of the blade and the hub of the diffusor, were applied to improve the calculation accuracy. Regarding the near walls, the values of γ+ were within 10.

At the inflow boundary, the gauge total pressure and the total temperature were imposed at 0 Pa and 298 K, respectively. The turbulent intensity was 2%, and the hydraulic diameter was 10 mm. At the outflow boundary, the gauge total pressure was imposed at 80 kPa. The frame motion method was selected for the moving domain, and the rotational velocity of the impeller domain was set at 565656 g.

The simulation in this paper was the steady-state calculation in the pressure-based type. The standard k-ε model with the standard wall function was adopted. The SIMPLE scheme was used, and both the turbulent kinetic energy and the turbulent dissipation rate were adopted in the first-order upwind type.

In Figure 2, the simulated and experimental performance maps are shown. The tendency of the simulated performance maps is in fairly good agreement with the experimental ones. The numerical pressure ratio and the total-to-total efficiency always overestimate the experimental data but in a very consistent manner. The numerical model in this paper is carried out without the volute; as a result, the actual loss in the experiment is omitted in the numerical calculation. And in the experiment, there was some measuring error in the performance measurement.

Meanwhile, the clearance leakage loss at the backplate of the compressor impeller was not considered in the calculation, which results in the higher calculated performance. In these terms, the compressor model in this paper can be considered reliable.

2.2 Particle model

The fouling phenomena of the compressor are due to the adherence of substances (liquid and/or solid) on the compressor surfaces, which progressively alter both the shape and roughness of the surface. These phenomena can be described by the following three phases: (a) transport of the particles (discrete phase) by the air (continuum phase), (b) contact and adhesion of the first discrete phase (particle) with the surface, and (c) repeated adhesion of the following particles on those previously deposited on the surface.4

2.2.1 Balance of forces

For studying particle dispersion, spatial distribution, and particle-wall interaction, CFD is a useful tool with either Eulerian or Lagrangian method. Typical numerical calculation methods in the particle-fluid multiphase flow mainly use the Eulerian-Eulerian method and the Eulerian-Lagrangian method. The Eulerian-Eulerian method is widely applied in the gas-solid system or liquid-solid system with dense particles. Regarding the Eulerian-Lagrangian method, the fluid phase is treated as the continuous phase, and the Navier-Stokes equation under the Eulerian coordinate system is used to solve the fluid flow characteristics, heat transfer, and reaction. Meanwhile, the particulate phase is treated as the discrete phase, and Newton’s second law under the Lagrangian coordinate system is applied to solve the particle trajectories in the flow field. The interactions between the continuous and discrete phases obey Newton’s third law, and the law is applied in the equations of continuous phase and discrete phase to realize the effect of the two phases’ coupling. The Eulerian-Lagrangian method directly solves the particle trajectories and reveals the movement of each particle. This method is applied in a gas-solid system or a liquid-solid system with dilute particles.41 In this paper, the solution approach is based on the mathematical model employing the Eulerian-Lagrangian method.
In this approach, the flow field is simulated first, followed by tracking of individual particle trajectories by integrating a force balance equation on the particle. This force balance equates the particle inertia with the forces acting on the particle and can be written as:

\[
\frac{d\vec{u}_p}{dt} = F_D(\vec{u} - \vec{\bar{u}}_p) + \frac{\vec{g}(\rho_p - \rho)}{\rho_p} + \vec{F}
\]

where on the right hand of the equation, \(F_D(\vec{u} - \vec{\bar{u}}_p)\) is the drag force per unit particle mass, \(\frac{\vec{g}(\rho_p - \rho)}{\rho_p}\) is the gravity and the buoyancy contribution, and \(\vec{F}\) is the additional acceleration (force/unit particle mass) term. Where

\[
F_D = \frac{18\mu C_D Re}{\rho_p d_p^2} \frac{\rho}{24}
\]

Here, \(\vec{u}\) is the fluid phase velocity, \(\vec{\bar{u}}_p\) is the particle velocity, \(\mu\) is the molecular viscosity of the fluid, \(\rho\) is the fluid density, \(\rho_p\) is the particle density, and \(d_p\) is the particle diameter. \(Re\) is the relative Reynolds number, which is defined as

\[
Re = \frac{\rho d_p |\vec{u}_p - \vec{\bar{u}}|}{\mu}
\]

And without consideration of the default gravitational acceleration, the value of \(\frac{\vec{g}(\rho_p - \rho)}{\rho_p}\) is zero.

Within the term \(\vec{F}\), the forces included are the Magnus rotating lifting force, Saffman’s lift force, pressure gradient force, thermophoretic force, virtual mass force, Basset force, Van der Waals force, Brownian force and so on. The virtual mass and pressure gradient forces are not important when the fluid density is much lower than the particle density (\(\rho/\rho_p \ll 1\)). In this paper, the fluid is treated as ideal compressible gas, and the air density is about 1.06 kg/m³. Meanwhile, the particle density is about 850 kg/m³.43 Because \(\rho/\rho_p \approx 0.00124 \ll 1\), the virtual mass force and the pressure gradient force should be ignored. Small particles suspended in gas with a temperature gradient experience a force in the direction opposite to that of the gradient. This phenomenon is known as thermophoresis. In this paper, such a temperature gradient exists in the compressor; therefore, the thermophoresis is included in the calculation model. The Brownian force is ignored. One reason is that the effect of Brownian motion can be optionally included in the term of \(\vec{F}\) for submicron particles, and in some calculation cases, the particle diameters are larger than 1 μm. The other reason is that the Brownian force is intended only for laminar simulation and the flow in the compressor should be turbulent.4

When the velocity gradient exists in the flow field, the particles in the flow field suffer from the lift force, even if the particles are not rotated.44 This force is called the Saffman’s lift force and is caused by the shearing action of the viscous fluid. Saffman’s lift force is considered during the simulation. Magnus rotating lifting force is the force acting on the rotating particles in the fluid. In this paper, the assumption is that the particles are not rotated by themselves. Therefore, Magnus rotating lifting force is ignored. When the particles accelerate relative to the fluid, the force acting on the particles owing to the fluid viscous effects is called Basset force. This force only must be included when the particle acceleration is large enough relative to the fluid motion. Basset force is ignored in this calculation. Van der Waals forces exist between the wall and nearby particles and between those particles close together.41 The force is also ignored in the calculation. The volume fraction of the particles was very low (\(\ll 10\%\)); therefore, it is assumed that the particles will not affect the fluid flow. That is one-way coupling.

2.2.2 Near-wall particle behavior

In the analysis of particle deposition, the particle-wall interactions and the particle-particle interactions are both significant. The particle deposition can be explained using the following45: (a) the particle surface and the field forces at direct contact; (b) material bridges between particle surfaces (liquid bridges and solid bridges); and (c) interlocking phenomena provided by macromolecular particle shape effects or by a particular

---

**FIGURE 1** A, Geometric model. B, Computational domain.
particle nature or surface characteristic. These bond effects are directly related to forces and displacements that take place at a microscale level (close to the molecular size). Certainly, the actual deposition process is simplified in the compressor simulation, and the simplified setting will be interpreted in the paper.

Regarding the impact angle function, Nokleberg and Sontvedt suggested the value of $f(\alpha)$ in a piecewise linear manner between three data points for ductile materials as follows: $f(0 \text{ deg}) = 0$, $f(20 \text{ deg}) = 1.0$, and $f(90 \text{ deg}) = 0.3$, and two points linear relationship for brittle materials between $f(0 \text{ deg}) = 0$ and $f(90 \text{ deg}) = 1.0$.46,47

For the restitution, the data were obtained by Whitaker et al.48 In that study, the authors found the restitution coefficients for glass beads impacting the aluminum plate. The restitution coefficient is dependent on the particle impingement angle $\alpha$, and both the perpendicular and tangential components of the restitution coefficient should be considered. Steven et al provided the following correlations for both perpendicular $e_n$, and tangential $e_t$.

$$e_n(\alpha) = 2.5656 - 0.2087\alpha + 0.0088\alpha^2 - 0.0002\alpha^3 + 2e^{-6}\alpha^4 - 6e^{-3}\alpha^5$$

$$e_t(\alpha) = 0.0202 + 0.1754\alpha - 0.013\alpha^2 + 0.0004\alpha^3 - 8e^{-6}\alpha^4 + 7e^{-8}\alpha^5 - 3e^{-10}\alpha^6$$

where the impact angle $\alpha$ is expressed in radians. In this paper, the injection material is automobile oil, and the material of the compressor inner wall is aluminum. The restitution coefficients here are adopted from Whitaker SM’s paper.

In an axial compressor, Jobaidur R. presented that the boundary condition for droplets at the walls can be assigned as either reflected, trapped, or maintaining as a liquid film. All three conditions could occur in areal application, but as FLUENT code only allows one condition to be assigned at a time, this study employs both reflected and filmed boundary conditions at separated cases, comparing the results of each. The results of the trapped condition should be between the results of the reflected and filmed conditions due to its allowance of water droplet evaporation upon surface contact.47 In this paper, the following conditions were adopted for the particle-wall interaction boundary conditions: ideal adherence condition (named trap) on the blade and diffusor surfaces; nonadherence condition (named reflect) on other surfaces of the compressor.

2.2.3 Tracking method and injection

The particles dispersion in the fluid phase can be predicted using a stochastic tracking model. This investigation used the DRW model to simulate the stochastic velocity fluctuations in the airflow.

The injected droplets were assumed spherical without deformation. The droplet material is automobile oil, and the oil density was set to be 850 kg/m$^3$.43 Regarding droplet diameter $d_p$, six values were considered. They are 0.25 μm, 0.5 μm, 0.75 μm, 1 μm, 1.25 μm, and 1.5 μm. There are six main blades and six splitter vanes; however, the compressor model only contains a blade and a splitter vane, making it a single channel model. Therefore, the mass flow rate of the injection of the model in this paper is one-sixth that of the whole compressor. The realistic mass flow rate of the compressor is 1 g/h, and therefore, the mass flow rate in this model is $\frac{1}{6}$ g/h.

3 RESULTS AND DISCUSSION

3.1 Capture efficiency

The analyses of the droplets impacting on the compressor diffusor are presented as follows. Only a portion of particles injected from the compressor inlet of the numerical model impact on the blade and diffusor surfaces. The imposed surface condition on the blades and the diffusor surfaces is...
trap, meaning the contact results are permanent adherences. Therefore, the fouling is caused by the particles’ adherence to the blades and the diffusor surfaces.

For comparison among the studied cases, the capture efficiency $\eta_{hit}$ is used. $\eta_{hit}$ is defined as the ratio between the number of particles that impact the diffusor surface (not including the blade surfaces) and the total number of injected particles. The trend of $\eta_{hit}$ as a function of the droplet diameter $d_p$, in the range of 0.25-1.5 μm is shown in Figure 3. Stokes number $St$ is defined as

$$St = \frac{\rho_p d_p^2 U}{18 \mu d_h}$$

(6)

From Equation (6), Stokes number (calculated at the inlet of the numerical model) is in the range of 0.001-0.041. The trend of Stokes number with the change of the droplet diameter is also presented in Figure 3, where $\mu$ is dynamic viscosity, $U$ is average velocity, and $d_h$ is hydraulic diameter.

For the particles’ movement on reaching the diffusor surface, the following mechanisms could be researched: settling, inertial impaction, interception, diffusion, and electrostatic forces. In this paper, to describe the droplets’ deposition in inertial impaction, interception, diffusion, and electrostatic face, the following mechanisms could be researched: settling, diameter. Let the diameter is also presented in Figure 3, where $d_p$ in the range of 0.001-0.041.

For the particles’ movement on reaching the diffusor surface, the following mechanisms could be researched: settling, inertial impaction, interception, diffusion, and electrostatic forces. In this paper, to describe the droplets’ deposition in the compressor, the turbulent diffusion and inertial theory should be used. Li et al. indicated that the turbulence diffusion deposition is the phenomenon by which the fine particles enter the boundary layer under the action of random pulsation by the turbulent vortex, before moving to the blade surface and depositing onto it. Meanwhile, the inertial deposition is the phenomenon by which the coarse particles are not able to follow the flow stream under the action of inertia and then impact the surface of blade and deposit onto it. From Kurz et al’s inertial deposition mechanism, in accelerated flows, the deviation between the particle trajectory and the streamline is a function of the gas acceleration, the particle size, and the particle density. The particle behavior is captured by Stokes number $St$, reflecting that the flow deflection in a compressor will cause deviation between the particle trajectories and the streamlines. Larger particles with a larger Stokes number will show greater deviation from streamlines.

From Figure 4, the particle trajectories of different diameters may be able to validate above mechanism. In the case, the droplet diameters follow a spectrum of distribution. In the injection properties, diameter distribution is set as rosin-rammler. Referring to Dairene’s literature, one of the peaks of the droplet diameters is between 1 μm and 3 μm. Therefore, the mean diameter is set within this range. Thus, $min$ diameter is set as 0.001 mm, $max$ diameter as 0.015 mm, and $mean$ diameter as 0.00239 mm. And spread parameter is 2.085353, which is calculated by the equation. The following conditions were adopted for the particle-wall interaction boundary conditions: ideal adherence condition (named $trap$) on the diffusor; nonadherence condition (named escape) on other surfaces of the compressor.

The particles of large diameters exist in the inlet zone and blade zone; meanwhile, the small ones exist in the diffusor zone. The reason is that small particles are dominated by the flow structures, whereas the large ones are mainly affected by inertia and centrifugation. Thus, it can be seen that most of the particles impacting the blades are large ones, and the small particles are able to impact the diffusor through the inlet zone and the blade zone.

From Figure 3, it is not difficult to observe that the trend of the capture efficiency on the diffusor is similar to that of the Stokes number. However, there is still some difference between these curves. The Stokes number was observed to increase persistently with droplet diameter, but capture efficiency only increased until the diameter reached 1.25 μm, instead decreasing marginally with a diameter of 1.5 μm. This indicates that the inertia is not the only reason for the impact on the diffusor. Meanwhile, in Figure 5, the curve on the hub side behaves similarly to the trend of Stokes number, that means the impact on the hub side is mostly affected by inertia. On the contrary, the curve on the shroud side does not match, signifying the impact on the shroud side is mostly affected by turbulent diffusion. When the droplet diameter was small, so too was the Stokes number. Because the following feature of the droplets is strong and the inertia of the droplets is weak, the droplets move following the flow streamline. Thus, the movement of droplets is mostly affected by turbulent diffusion. Because of the direction of the flow streamline as shown in Figure 6, most of the droplets impact on the shroud side of the diffusor. Figure 6 also shows that the structure of the shroud side is wedge shaped, explaining the higher capture efficiency on the shroud for the smaller diameter. With an increase in the droplet diameter, both the Stokes number and droplet inertia increase, while the
turbulent diffusion becomes weaker. Therefore, the trend of capture efficiency on the hub side is increasing, while the trend on the shroud side is decreasing. Until the diameter equals to 1.5 μm, the capture efficiency on the hub side is higher than that on the shroud side as presented in Figure 5. In addition, the capture efficiency of droplets on the diffusor may not reflect their deposition accurately, as it might only reflect the movement of the droplets relative to the walls. It is thus necessary to analyze particle concentration in studying the droplet deposition mechanism on the diffusor.

3.2 | Particle concentrations

The capture efficiency discussed previously considers only the statistics for the impact of the droplets on the diffusor, but not their subsequent accumulation. In addition, if the setting mode of the impact on the diffusor is reflect, and the impact velocity of the droplet is less than the critical value of the deposition velocity, the droplets could reflect, not deposit on the diffusor. Therefore, the capture efficiency of droplets on the diffusor may not reflect their deposition accurately, as it might only reflect the movement of the droplets relative to the walls. It is thus necessary to analyze particle concentration in studying the droplet deposition mechanism on the diffusor.

The particle mass concentration term \( C_{DPM} \) models the contaminant concentration on surfaces very close to the diffusor, defined as kg/m³. Although there is an accretion rate provided by the software, the trap condition on the diffusor surface implies unrealistic values of this quantity, in contrast to those obtained from experimental tests reported by Parker and Lee. In particular, the new surfaces were positioned at a constant distance of 20 μm from the diffusor on each side. As there could be some significance to the thickness of the deposition on the walls, it is more accurate to evaluate the presence of contaminants in the portion of fluid that is located very close to the diffusor surfaces.

Figure 7 shows the trend of the total particle mass concentration on the hub surface, the shroud surface, and both diffusor surfaces, respectively, as a function of the droplet diameter \( d_p \). In Figure 7, DPM concentration indicates the total particle mass concentration, and DPM means Discrete Phase Model. The peak of the total DPM concentration occurs at the diameter 0.5 μm. With exception of the diameter 0.5 μm, the observed trend of total DPM concentration was a gradual decrease. The DPM concentration on the hub side was observed to increase gradually, while the concentration on the shroud side decreased inversely. At a droplet diameter of 0.25 μm, the DPM concentration on the hub is lower than that on the shroud side. With an increase in the diameter, the DPM concentration on the hub side is higher than that on the shroud side.
The phenomenon shown in Figure 7 can be explained by consideration of Figure 6. The diffusor in the compressor model is wedge shaped, with the shroud side of the diffusor including both the wedge and parallel parts. As seen from the streamlines, an obvious vortex appears at the trailing edge of the blade. The center of the vortex is located downstream from the export of the impeller. The length of the vortex area is about 1/3 of the contracted part in streamline direction and about 1/5 the span of the channel. There is no vortex in the downstream portion of the contracted part, and the streamlines in the parallel part are parallel with the diffusor walls.\textsuperscript{54}

As mentioned above, droplet deposition on the hub side is due mostly to inertia and deposition on the shroud side to turbulent diffusion. With a diameter of 0.25 μm, the particle inertia is small and the following feature of the particles is high. Most of the particles follow the movement of the flow and thus impact the shroud side of the diffusor due to turbulent diffusion. Therefore, the concentration on the shroud side is higher that that on the hub. With an increase in droplet diameter, more particles impact on the hub side under influence of the centrifugal and inertial forces; as the turbulent diffusion becomes weaker, more particles impact on the hub side. Therefore, the DPM concentration on the hub side increases and the concentration on the shroud side decreases.

Figures 8 and 9 show the contour plot of $C_{\text{DPM}}$ on the hub side and shroud side of the diffusor. In Figures 10 and 11, the strips present thirteen uniform partitions in the radial direction (from the small radial to the large radial). From these contour plots and histograms, it is possible to notice the following phenomena.

1. From Figure 8, the peak of the contaminant concentration on the hub side of the diffusor is located in the small radius area for the six cases with the droplet diameter from 0.25 μm to 1.5 μm. And in the large radius area, the contaminant concentration on the hub side is obviously low.

The particles flow from the flow channels to the diffusor under high velocity. At the same diameter, the particles with high momentum flow to the hub side of the diffusor under centrifugal and inertial forces, placing them under restriction of the diffusor channel. The particles with higher momentum should impact the small radius area of the hub side and deposit on it. Other particles with lower momentum are driven by the flow to the larger radius area. With the influence of the high adverse pressure gradient, the particle momentum decreases and deposits on the larger radius area. With the increase in droplet diameter, the particle following feature becomes weaker, and therefore, the phenomenon is less obvious.

2. The circumferential and radial distribution uniformity on the shroud side is higher than that on the hub side. The phenomenon is consistent with the realistic deposition in Figures 4 and 5 in Uy’s literature.\textsuperscript{50} Also, the distribution of concentration in Figure 11 is consistent with the cloud charts in Figure 9.

Li et al\textsuperscript{14} indicated that when the droplets deposit on the blade mainly by turbulent diffusion, the droplets enter the boundary layer under the action of random pulsation by the
turbulent vortex, moving onto the blade surface and depositing on it. Therefore, the trajectories of droplets are scattered. As the deposition on the shroud side is mainly affected by turbulent diffusion, the distribution uniformity of deposition on the shroud side is therefore higher.

In addition, the pressure on the pressure side of the blade is higher than that on the suction side. Under this pressure difference, part of the flow at the blade tip would move through the leaf peak clearance from one channel to the adjacent, thus moving horizontally. The clearance leakage flow and the mainstream mix in the large area near the shroud side, which enable DPM concentration to be equally distributed on the shroud of the diffusor.

3. In Figure 8, in the direction of channel on the hub side of the diffusor, the contaminant concentration is relatively higher than that of other adjacent areas, as shown in the red arrow direction. The phenomenon is consistent with the realistic deposition in Figure 4 provided by Uy.50

As shown in Figure 8, the high DPM concentration area is downstream of the channel in the direction of the streamline; meanwhile, it is the high-energy flow area as shown in Figure 12. The particles are driven by the flow and have higher momentum. Thus, the particles with higher momentum are
inclined to deposit in this area. The area with less deposition can be explained by the large flow mixing degree in the wake region of the blade trailing. With an increase in droplet diameter, the particles’ inertia increases gradually, and the following feature of the particles becomes weaker. Therefore, the DPM concentration on the hub side of the diffusor increases, and the deposition distribution becomes more and more uniform.

4 | CONCLUSION

In this paper, a study on droplet deposition within a compressor diffusor was carried out. The deposition on the compressor diffusor surfaces of gasoline engine turbochargers reduces diffusor width, which forms a resistance to the air flow and causes deterioration in performance. The automobile oil deposition caused by crankcase ventilation is responsible for the compressor fouling if the engine uses the positive crankcase ventilation system. The authors used the numerical CFD approach to study the droplet deposition mechanism on the compressor diffusor.

The key results can be summarized as follows:

1. The trend of capture efficiency of the diffusor is increased with particle diameters up to 1.25 μm, while the capture efficiency at a diameter 1.5 μm is decreased slightly. The trend of the capture efficiency on the hub side is increasing, and the trend on the shroud side is decreasing.
2. At a droplet diameter equals to 0.25 μm, the DPM concentration on the hub is lower than that on the shroud side. With an increase in the droplet diameter, the DPM concentration on the hub side increases gradually; meanwhile, the concentration on the shroud side decreases inversely. As a result, the DPM concentration on the hub side is higher than that on the shroud side except in the case of the diameter 0.25 μm.
3. The peak of the contaminant concentration on the hub side of the diffusor is located in the small radius area for the six cases with all the droplet diameters considered in this paper. And the circumferential and radial distribution uniformity on the shroud side is higher than that on the hub side. In addition, there are also some special high DPM concentration areas in the direction of the streamline.
4. The particle impact patterns are determined and influenced by the particle diameter, the particle following feature, the shape of flow channels, the pressure gradient, the particular fluid dynamic phenomena such as the leakage flow and so on.

An understanding of the droplets’ fouling mechanism on the diffusor of a vehicular turbocharge is significant to its application and engineering. This paper employs steady calculation using the freezing rotor method. The results describe the deposition feature of the droplets at a particular time and position of the blade relative to the diffusor. In this sense, the next part of this research will focus on the unsteady calculation including the droplets breakage and coalescence, and the deposition accumulation with the time accumulation. This will allow for more accurate research on the deposition mechanism of droplets on the centrifugal compressor diffusor. And such research could have certain significance in academic guidance for future compressor design.

ORCID
Chaochen Ma https://orcid.org/0000-0002-6929-3222

REFERENCES
1. Gao J, Tian G, Sorniotti A. On the emission reduction through the application of an electrically heated catalyst to a diesel vehicle. Energy Sci Eng. 2019. [Epub ahead of print].
2. Emetere ME, Jack-Quincy S, Adejumo A, Dauda O, Adeyemi O. Empirical analysis of biodiesel effect on the automobile properties of diesel engine: a case study of olive and soya biomass. Energy Sci Eng. 2018. [Epub ahead of print].
3. Chang S. Vehicle Power System. Beijing, China: Mechanical Industry; 2006 (in Chinese).
4. Suman A, Kurz R, Aldi N, et al. Quantitative computational fluid dynamics analyses of particle deposition on a transonic axial compressor blade—part I: particle zones impact. J Turbomach. 2014;137:156-158.
5. Mishra RK. Fouling and corrosion in an aero gas turbine compressor. J Fail Anal Prev. 2015;15:1-9.
6. Li Z, Li B, Wang D, Li L. Analysis of sensitivity of compressor performance parameters to fouling. Aeronaut Comput Tech. 2011;41:41-44.
7. Forsyth P, Gillespie DRH, Mcgilvray M, Forsyth P, Gillespie DRH, Mcgilvray M. Development and applications of a coupled particle deposition dynamic mesh morphing approach for the numerical simulation of gas turbine flows. J Eng Gas Turbines Power. 2017;140(2):022603.
8. Brun K, Nored M, Kurz R. Analysis of solid particle surface impact behavior in turbomachines to assess blade erosion and fouling. In: Proceedings of the Forty-First Turbomachinery Symposium; 2012.
9. Xu H, Zeng W, Han Z, Li P, Qian J. Effect of the nonaxisymmetric endwall on wet steam condensation flow in a stator cascade. Energy Sci Eng. 2019;7:557-572.
10. Wang CS, Zhang L. Fluid simulation in a cyclone reverse circulation well washing device based on computational fluid dynamics. Energy Sci Eng. 2019;7:1306-1314.
11. Peng S, Chen Q, Shan C, Wang D. Numerical analysis of particle erosion in the rectifying plate system during shale gas extraction. Energy Sci Eng. 2019;7:1838-1851.
12. Forsyth P, Gillespie DRH, Mcgilvray M, Galoul V. Validation and assessment of the continuous random walk model for particle deposition in gas turbine engines. In: ASME Turbo Expo 2016: Turbomachinery Technical Conference and Exposition; 2016.
13. Starzmann J, Kaluza P, Casey MV, Sieverding F. On kinematic relaxation and deposition of water droplets in the last stages of low pressure steam turbines. In: ASME Turbo Expo 2013: Turbine Technical Conference and Exposition; 2013;V05BT25A028.
14. Li C, Wang X, Wang X, Sun B. Numerical investigation of movement and deposition of water droplet on steam turbine stator cascade. J Xian Jiaotong Univ. 2008;42:1076-1080 (in Chinese).
15. Williams J, Young JB. Movement of deposited water on turbomachinery rotor blade surfaces. J Turbomach. 2007;129:1407-1420.
16. Sun L, Zheng Q, Luo M, Li Y, Bhargava R. On the behavior of water droplets when moving onto blade surface in a wet compression transsonic compressor. J Eng Gas Turbines Power. 2011;133:783-789.
17. Crane RI, Crane RI. Droplet deposition in steam turbines. ARCHIVE Proc Inst Mech Eng Part C J Mech Eng Sci 1989–1996 (vols 203–210). 2004;218:859-870.
18. Kurz R, Brun K. Fouling mechanisms in axial compressors. J Eng Gas Turbines Power. 2012;134:935-946.
19. Guha A. Transport and deposition of particles in turbulent and laminar flow. Fluid Mech. 2008;40:311-341.
20. Suman A, Morini M, Aldi N, Casari N, Pinelli M, Spina PR. A compressor fouling review based on an historical survey of ASME Turbo Expo papers. J Turbomach. 2017;139:041005.
21. Eisfeld T, Joos F, Eisfeld T, Joos F. Experimental investigation of two-phase flow phenomena in transonic compressor cascades. In: ASME Turbo Expo 2009: Power for Land, Sea, and Air; 2009:103-112.
38. Aldi N, Casari N, Dainese D, et al. Quantitative CFD analyses of particle deposition in a heavy-duty subsonic axial compressor. In: ASME Turbo Expo 2017: Turboachinery Technical Conference and Exposition; 2017:V009T027A013.
39. Borello D, Rispoli F, Venturini P. An integrated particle-tracking impact/adhesion model for the prediction of fouling in a subsonic compressor. J Eng Gas Turbines Power. 2012;134:1721-1721.
40. Aldi N, Morini M, Pinelli M, Spina PR, Suman A. An innovative method for the evaluation of particle deposition accounting for the rotor/stator interaction. J Eng Gas Turbines Power. 2017;139:052401.
41. Yuan Z. Gas-particle Two-phase Flow and Numerical Simulation. Nanjing, China: Southeast University Press; 2013 (in Chinese).
42. ANSYS. ANSYS Fluent User Manual 15.0. ANSYS; 2013.
43. Uy D, Storey J, Sluder CS, Barone T, Lewis S, Jagner M. Effects of oil formulation, oil separator, and engine speed and load on the particle size, chemistry, and morphology of diesel crankcase aerosols. NeuroImage. 2016;9:224-238.
44. Pg S. The lift on a small sphere in a slow shear flow. J Fluid Mech. 1965;22:385-400.
45. Rimai DS, Demejo LP, Bowen RC. Mechanics of particle adhesion. J Adhes Sci Technol. 1994;8:1333-1355.
46. Nøkleberg L, Søntvedt T. Erosion of oil & gas industry choke valves using computational fluid dynamics and experiment. Int J Heat Fluid Flow. 1998;19:636-643.
47. Khan JR, Wang T. 3D modeling for wet-compression in a single stage compressor including liquid particle erosion analysis. In: ASME Turbo Expo 2010: Power for Land, Sea, and Air; 2010:205-218.
48. Whitaker SM, Peterson B, Bons JP. Evaluation of coefficients of restitution using high speed particle shadow velocimetry with application to particle separators for gas turbine engines. In: Aerospace Sciences Meeting; 2014.
49. Fuchs NA, Davies CN. Aerosol mechanics. [Book Reviews: The Mechanics of Aerosols]. Science. 1964;146:1033-1034.
50. Uy D, Pranis G, Morelli A, et al. Correlating laboratory oil aerosol coking rig tests to diesel engine tests to understand the mechanisms responsible for turbocharger compressor coking. In: Wcx™ 17: Sae World Congress Experience; 2017.
51. ANSYS. ANSYS Fluent 15.0 Theory Guide; 2013.
52. ANSYS. Fluent 15.0 Tutorial Guide; 2013.
53. Parker GJ, Lee P. Studies of the deposition of sub-micron particles on turbine blades. ARCHIVE Proc Inst Mech Eng 1847–1982 (vols 1–196). 1972;186:519-526.
54. Ma C, Wang H, Lin Y, Zhu Z. Numerical simulation and flow analysis of the vaneless diffuser of centrifugal compressor in low flow condition. ICE Powerplant. 2010;119:13-16 (in Chinese).

How to cite this article: Sun L, Ma C, Song P, Zhang H. Quantitative computational fluid dynamic analyses of oil droplets deposition on vaneless diffusor walls of a centrifugal compressor. Energy Sci Eng. 2020;8:910–921. https://doi.org/10.1002/ese3.561