| 京都大学 | 博士（　工学　） | 氏名 | 温　健 |
|---|---|---|---|
| 論文題目 | Numerical Study of Liquid Fuel Atomization, Evaporation and Combustion<br>（液体燃料の微粒化，蒸発および燃焼に関する数値解析） | | |

（論文内容の要旨）


In this thesis, the detailed mechanisms of liquid fuel atomization, evaporation and combustion are investigated by means of a volume of fluid (VOF) simulation and a large-eddy simulation (LES). This thesis consists of 5 chapters, as follows.

Chapter 1 presents the background, motivation, objectives, and outline of this thesis.

In Chapter 2, the Eulerian-Lagrangian method is proposed for spray atomization simulation. In the Eulerian framework, both liquid and gas phases are solved and treated as incompressible fluids, and the liquid-gas interface is captured by using a coupled levelset and volume of fluid (CLSVOF) method. On the other hand, in the Lagrangian framework, the dispersed droplets generated by the atomization process are considered as point particles, and the evaporation of droplets is considered by employing a non-equilibrium Langmuir-Knudsen evaporation model. A two-way coupling is utilized to consider the interaction between gas phase and fuel droplets by employing the particle-source-in-cell (PSI-cell) method. The Eulerian framework and Lagrangian framework are coupled by an Eulerian-Lagrangian tagging method such that the Eulerian components could be directly transformed into Lagrangian droplets. Several additional criterions are employed to judge the droplet topologies to effectively transform spherical Eulerian components to Lagrangian droplets. Also, the collision of Lagrangian droplets on the liquid surface is considered. The atomization-evaporation process of a liquid fuel jet in the vicinity of a fuel nozzle in several different steady crossflows under the elevated pressure condition is investigated by using the proposed method. The liquid jet trajectory, atomization behavior, droplet size distribution, vortex development and morphology, and evaporation effect under different breakup regimes (i.e., different aerodynamic Weber number conditions) are investigated. The results show that the liquid jet trajectory is strongly affected by the liquid initial velocity distribution and the atomization behavior of the liquid jet is reproduced under different breakup mode regimes, namely the bag mode, multimode, and shear mode regimes. With an increasing aerodynamic Weber number ($We$ = 10 to 200), the dominant breakup behavior changes from the bag breakup to the shear breakup, leading to the smaller size of the generated droplets in the downstream region. It is also found that the droplet behavior is strongly affected by the circulation flows generated behind the liquid jet, namely the bag breakup vortex (BBV) and shear breakup vortex (SBV), which are responsible for the bag breakup and the shear breakup, respectively. The SBV, which is stronger than the BBV and appears just behind the leeward of the liquid jet and near the nozzle, tends to trap many droplets and entrain fuel vapor, and then causes the suppression of the evaporation in the regions just around the SBV. Meanwhile, the SBV acts to promote the evaporation downstream because of the enlarged droplet residence time and the raised temperature owing to mixing with hot air flow.

In Chapter 3, the influence of combustion instability due to the thermoacoustic on the liquid jet atomization-evaporation process is investigated by using the proposed Eulerian-Lagrangian method. The pressure oscillating induced by the combustion instability is simplified by employing an oscillating air inflow condition. Therefore, the numerical simulation of liquid fuel jets in crossflows is performed under an oscillating air inflow condition with a frequency of 500 Hz in the multimode regime. The liquid fuel atomization and evaporation behaviors are compared with those under the steady air inflow condition in Chapter 2. The physical

properties of the liquid fuel and carrier crossflow are the same as those under the steady crossflow. The liquid jet trajectory, droplet size, vortex development and morphology, and fuel vapor distributions are investigated. The results show that the liquid jet trajectory, generated droplet size and fuel concentration in the downstream region vary with time at the same frequency as the oscillating crossflow velocity. Moreover, the Sauter mean diameter and arithmetic mean diameter of droplets in the downstream region show converse changes. It is also found that the SBV is enhanced at the higher velocity phase of the oscillation, which traps more circulating droplets, resulting in a higher evaporation rate. In the downstream region, the fuel vapor concentration has a half-period time delay to the oscillating crossflow velocity, because the weakened SBV at lower velocity sends more fuel vapor, which is entrained by the enhanced SBV at the higher velocity phase. Further, the time-averaged values of the droplet size and the fuel vapor concentration in the downstream region are smaller and larger than those under the steady air inflow condition, respectively. This is attributed to the fact that the enhanced SBV traps more droplets and promotes their evaporation downstream, compared to those under steady crossflow.

In Chapter 4, the detailed combustion mechanism in a dense spray region is investigated by conducting LESs by using a flamelet combustion model of a turbulent dense spray flame coupled with a detailed high-resolution VOF simulation of liquid fuel atomization. Computations are validated against the measurements performed by using the Sydney Burner, which generates both dilute and dense spray flames by varying the recess distance from the liquid fuel jet nozzle to the pilot inlet. In the Eulerian framework of VOF simulation, a liquid jet along with the co-axial air flow is performed to acquire the detailed information of the atomization process. Acetone is used as the liquid fuel and air is used as the carrier gas for atomization, and the physical parameters such as aerodynamic Weber number, and Reynolds number of both liquid and gas phases are given as the same as the experiment. The Eulerian components of dispersed liquid droplets are transformed into Lagrangian droplets at a certain downstream cross-section (i.e., sampling cross-section), whose information (i.e., the position, size, and velocity of each droplet) is stored in the database. Then, the combustion process is solved by the LES by adopting the pre-stored database of Lagrangian droplets as the inlet boundary conditions. The validation of the Eulerlian-Lagrangian tagging method is checked, and the droplets size distribution at various radial locations as well as the droplet volume flux are investigated to confirm the effectiveness of the VOF simulation. The predicted gas temperature distributions by the LES employing different databases of droplets with different gas inlet boundary conditions at different downstream locations are compared with the experimental results to investigate the significance of the coupling of the detailed beforehand atomization process. The results of the VOF simulations show that the fuel spray atomization fluctuates temporally and spatially. Furthermore, the breakup period is in good agreement with an empirical correlation. Three factors, namely the location of the sampling cross-section, sampling time, and threshold value for an Eulerian-Lagrangian transformation, are found to strongly affect the properties of the Lagrangian droplets. It is also observed that the gas temperature of dilute spray flames, predicted by the LES, is in good agreement with the experiments, whereas that of the dense spray flames is overpredicted at the downstream locations. The overestimation occurs because the present Lagrangian approach fails to consider further droplet breakup such as secondary breakup and the influence of the vulnerable droplets in the dense spray region on the turbulence development and flame evolution.

Chapter 5 summarizes all the investigations carried out in this study. Moreover, it provides recommendations for possible future extensions of the present study.

| 氏　名 | 温 健 |
|---|---|

（論文審査の結果の要旨）

本論文は，液体燃料の微粒化，蒸発および燃焼に関する研究の結果をまとめたものであり，得られた主な成果は以下の通りである．

1. クロスフロー型液体燃料の微粒化および蒸発に関する気液二相流の数値解析により，液体燃料液柱の軌跡は流入速度分布の影響を強く受けること，また，液柱の微粒化の特徴は気相 Weber 数の増加に伴い bag breakup から shear breakup へと変化し，生成される液滴サイズは小さくなることを明らかにした．さらに，bag breakup と shear breakup の遷移条件（マルチ条件）および shear breakup 条件においては，液柱の背後に形成される再循環流（shear breakup vortex（SBV））が強くなり，これが微粒化された液滴を捕獲して滞留時間を延伸させるため，蒸発を促進する働きがあることを見出した．
2. クロスフロー型液体燃料の微粒化および蒸発に関する気液二相流の数値解析（マルチ条件）を，燃焼振動を模擬する目的で気流速度に周期変動与えて実施することにより，液体燃料液柱の軌跡，微粒化液滴サイズおよび燃料蒸気濃度の変動の周波数は，気流変動の周波数と一致することを明らかにした．また，気流変動を与えない場合に比べて，与えた場合の微粒化液滴サイズは小さく，燃料蒸気濃度は増加することを見出した．これらの現象は，気流流速が最大の時に形成される強い SBV により，液滴蒸発が促進されることによって生じることを明らかにした．
3. 液体燃料の微粒化と乱流燃焼の数値解析を連携させて解くことにより，液体燃料の微粒化・蒸発・燃焼過程を再現可能な数値解析手法を提案した．また，本手法を既存の噴霧噴流火炎に適用することにより，本手法が燃料希薄条件下における火炎の温度分布を良好に再現可能であることを示した．さらに，本手法は燃料過濃条件下における火炎温度を下流領域で過大評価する傾向があるが，これは，乱流燃焼の数値解析において非球形液滴が流れ場に及ぼす影響を無視していることに起因しており，その影響のモデル化が更なる高精度化に有効であることを示した．

以上，本論文は，噴霧燃焼の素過程となる液体燃料の微粒化，蒸発および燃焼現象を高精度に予測可能な数値解析手法を確立するとともに，噴霧燃焼メカニズムの一端を明らかにしたものであり，学術上，実際上寄与するところが少なくない．よって，出願者が博士後期課程学位取得基準を満たし，本論文が博士（工学）の学位審査の請求に値するものと認める．また，令和３年１２月１３日，論文内容とそれに関連した事項について試問を行って，申請者が博士後期課程学位取得基準を満たしていることを確認し，合格と認めた．