	こうち　としのり
氏名	河内　俊憲
授与学位	博士（工学）
学位授与年月日	平成17年3月25日
学位授与の根拠法規	学位規則第4条第1項
研究科，専攻の名称	東北大学大学院工学研究科（博士課程）航空宇宙工学専攻
学位論文題目	スクラムジェット内の火炎構造と燃焼器最適化に関する研究
指導教員	東北大学教授　升谷　五郎
論文審査委員	主査　東北大学教授　升谷　五郎　　東北大学教授　中橋　和博 東北大学教授　小林　秀昭　　東北大学教授　佐宗　章弘

論文内容要旨

Airframe-integrated scramjet engines are promising as the engines for space planes. Scramjet engine is one of the air-breathing engines and operates with supersonic combustion. The scramjet engine consists of an inlet, a combustor and a nozzle. Component studies of the engine have been conducted to optimize these configurations. The inflow air is compressed by shock waves in the inlet and it is difficult to reduce flow distortion in the inlet. The inlet distortion might promote the non-uniformity of combustion to reduce the combustion and thrust performance. The intensive combustion generates the flow separation which might induce the inlet unstart. Therefore, the studies of the aerothermodynamic interaction between the combustor and the inlet are important for designing of the configuration and system of the scramjet engine.

Few studies of this interaction have been conducted and the details of the interaction process have not been clarified. To reveal the effects of the inlet / combustor interaction, it is necessary to examine the whole engine flowfield. The details of the flowfield in the engine are difficult to measure in the engine test facility, because of severe environment around the engine. On the other hand, three-dimensional computational fluid dynamics (CFD) solutions can provide detailed insight to the complex hypersonic combustion flowfields. The numerical simulations of a scramjet engine have been expected to be a useful tool to understand the aerothermodynamic process in the engine, especially that of the inlet / combustor interaction. The object of this thesis is to examine the interaction process ruling the engine performance and to indicate the way to improve the engine performance by the numerical simulations.

In Chapter 2 entitled with "Numerical Method and Grid Sensitivity Analysis", the numerical method employed for the internal flow of the scramjet engine was described in the beginning. The computations were performed with the Tohoku University Aerodynamic Simulation (TAS) code, developed by Nakahashi and his coworkers and extended to reactive flow simulations by Kodera in the Japan Aerospace Exploration Agency (JAXA). The code uses a cell-vertex, finite-volume technique with upwind scheme on hybrid unstructured grid to solve the full three-dimensional compressible Reynolds-averaged Navier-Stokes equations with finite rate chemistry.

In this code, the AUSM-DV scheme is applied to approximate the convective flux function. The second order spatial accuracy is realized by a linear reconstruction of the primitive gas dynamic variables with the Venkatakrishnan's limiter. The computational efficiency is greatly improved by the LU-SGS implicit method. The algorithm adopts the finite rate hydrogen-oxygen reaction proposed by Stahl and Warnatz, which consisted of nine species / eighteen elementary reactions. The turbulent eddy viscosity is evaluated by the one-equation turbulence model proposed by Goldberg and Ramakrishnan. The turbulent Prandtl number and Schimdt number were set to be 0.9 and 1.0, respectively.

CFD analyses always require the grid resolution which is sufficient to capture the physically and chemically relevant features. The number of the nodes for the whole scramjet engine grid studied in this thesis is too large to directly examine the grid sensitivity. Therefore, the grid sensitivity analyses for the inlet and combustor sections in the engine were separately conducted, and checked not only the grid-convergence for the wall pressure distributions but also those of the inlet performance, mixing / combustion performance, and transport properties on the wall. The following conclusions were drawn:

- One percent of the total length of the inlet (10 mm or less) was small enough for the grid resolution of the inlet.
- The wall pressure distribution easily converged in the combustor. Especially, the pressure thrust, which was obtained by the integration of the wall pressure, converged even with the coarse grid of 4-mm tetrahedrons.
- The mixing efficiency without combustion was difficult to converge even with the 1-mm tetrahedrons, because the numerical diffusion was greater than the eddy diffusion.
- The mixing and combustion efficiencies with combustion were strongly affected by the change in the flowfield caused by the combustion itself, and the eddy diffusion became larger than that without combustion. As the result, the 2-mm tetrahedrons were sufficient to capture the combustion efficiency.
- Transport properties on the wall such as heat flux and wall shear stress needed the prismatic grid of $y+ < 4$ on the wall.

These grid-convergence studies made the present grids of whole scramjet engine optimized.

In Chapter 3 entitled with "Validation of CFD with Engine Test Data", the validation studies of the numerical simulations of the scramjet engine were performed. The validation study is most important in using the CFD, because it is not sure that turbulence and combustion models are applicable to the supersonic reactive flow even with the grid resolution, which is sufficient to capture the physically relevant features. If the computational results can be calibrated with the experimental data, the CFD can become a powerful tool to help our understanding of the physical processes in the engine and to improve the engine design.

There published only few numerical simulations of the whole scramjet engine with combustion and they were not well validated with the engine test data. So, the scramjet engine grid, which can capture the physical and chemical process in the engine, was generated based on the results of the grid sensitivity analysis in Chapter 2. The scramjet engine, tested in the Ramjet Engine Test Facility (RJTF) in Japan under Mach 6 flight condition, was numerically simulated by the three-dimensional Reynolds averaged Navier-Stokes flow code with finite rate chemistry. Reliability of the numerical simulations was intensively verified by comparing them with the engine test data such as the wall pressure and heat flux distribution, the inlet performance, and the thrust performance etc. The following conclusions were drawn:

- The numerical results of the wall pressure and heat flux distributions in the fuel-off case and the combustion case showed good agreement with the experimental data.
- The accuracy of the numerical results of the wall shear stress was investigated by the comparison of the engine internal drags, because the local wall shear stress was very difficult to measure in the experiments. The numerical result reproduced the experimental data of drag within 10 %.
- The reliability of the air capture ratio was confirmed by the comparison between the numerical and experimental results under various conditions. Then, the additive drag caused by the air spillage of inlet was evaluated from the CFD. The additive drag was found to account for 28% of the internal drag.
- Although the computed thrust failed to duplicate the jump between the two combustion modes observed in the experiments, it can simulated the dependence on the fuel equivalence ratio (F) and the calculated value of thrust increment with combustion at $F = 1$ was in good agreement with measured one.

These comprehensive validation studies show that the present CFD code is sufficiently accurate and the aerothermodynamic process in the scramjet engine can be analyzed with them.

In Chapter 4 entitled with "Heat Flux Prediction for Scramjet Engines", applicability of the Reynolds analogy between skin friction and heat flux in the scramjet was focused. The scramjet engines are exposed to severe thermal environments during flight. Preliminary estimation of heat-transfer in the scramjet engine is important for designing of the engine configuration and the propulsion system. One of the principal goals of the researches on the engine is to design with an engine-cooling requirement equivalent to only a fraction of the total fuel heat sink available. In the preliminary studies of the engines, thermal load to the engine walls have been estimated by the method based on the Reynolds analogy. However, it is not sure that the Reynolds analogy can be applicable to the scramjet internal walls, because there are strong pressure gradients due to the shock waves and combustion. The validity of the Reynolds analogy for the scramjet internal flow has not been quantitatively confirmed.

In the experiments, it is difficult to quantitatively reveal the accuracy of the Reynolds analogy in the engine, because of difficulties in measuring the local skin friction coefficient. The CFD can provide valuable insight to this quantity, too. So, the accuracy of the Reynolds analogy in the scramjet engine was quantitatively investigated by using the numerical results verified in Chapter 3. The following conclusions were drawn:

- The heat flux in the engine was proportional to the 0.8 power of the wall pressure. This result implied that the Reynolds analogy was applicable for the engine walls.
- The local friction coefficient and Stanton number on the engine walls could be calculated by using the mass weighted average flow properties obtained by the three-dimensional computations.
- The Reynolds analogy was applicable for 90% of the wetted area in the engine within ± 10 % error even for the combustion case. As the result, the heat-transfer of scramjets can be easily and accurately estimated by the one-dimensional analysis using the Reynolds analogy.
- The one-dimensional estimations demonstrated that the average heat flux in the engine was 0.5 MW/m2 and that the engine-cooling requirement was equivalent to the available heat sink of the fuel at the Mach 8 flight condition. To extend the operating Mach number range, the flowpath in the combustor should be shortened, because the combustor produced 80 % of the cooling requirement.

In Chapter 5 entitled with "Suppression Mechanism of Engine Unstart by Boundary-Layer Bleeding", the unstart characteristics with combustion were investigated by the numerical simulations of the whole scramjet engine under the Mach 6 flight condition. The airframe-integrated scramjet engine swallows the thick boundary-layer which develops on the forebody of space plane. The scramjet engines must be therefore tested under the condition of boundary-layer ingestion in the ground test. The ingested boundary-layer may be separated by the combustion-generated shock waves. The separation of the ingested boundary-layer promotes the engine stall or unstart. To prevent the engine unstart and to extend the operating range of the engine, a boundary-layer bleeding system was attached to the engine and tested under the Mach 6 flight conditions in the RJTF. Although the boundary-layer bleeding extended the operating range and doubled the net thrust, the mechanism which suppressed the engine unstart by the bleeding was not clarified.

So, the numerical simulations of the scramjet engine with boundary-layer system were performed and revealed the effect of the bleeding on the separation of the ingested boundary-layer on the topwall. At the bleeding region in the computations, flow speed normal to the wall was specified to impose the bleeding flow rate and the other flow properties were determined by zeroth-order extrapolation. The numerical predictions using this simplified bleed condition well reproduced the engine combustion tests with bleeding, and identified the suppression mechanisms of the engine unstart. The following conclusions were drawn:

- The adiabatic wall conditions of 1500 K, which was the maximum possible wall temperature in the experiments, promoted "bubble" type separations. The ingested boundary-layer on the topwall was separated not only around the leading edge of the strut, but also at the impingement location of the cowl shock.
- These separations easily caused engine unstart with fuel injection. For the adiabatic wall condition, the combustion in the separation bubble made itself larger and promoted it expansion toward the inlet. As the result, the engine easily fell into the engine unstart even at very low F of 0.2.

- On the other hand, the cooled wall kept at room temperature of 300 K can keep the engine start condition up to F = 0.5 without the bleeding, because there was no "bubble" type separation of the ingested boundary-layer on the topwall.
- The boundary-layer bleeding prevented the flow separation and extended the start limit. Bleeding 0.6% of the captured air extended the start limit to F = 1.0 for both the cases of adiabatic and cooled wall. This was caused by the drastic decrease of the momentum thickness of the boundary-layer on the topwall with very low bleeding rate.

In Chapter 6 entitled with "Flame Structures and Combustion Efficiency in Scramjet Engines", flame structures in the scramjet engine were examined and the improvement of the combustor design was proposed on the basis of the examination. Early tests of scramjet combustors using direct-connect test facilities achieved high combustion efficiency of 80 ~ 90 %. However, the thrust performance measured in the engine tests installing the similar type of the combustor was not high and its achievement factor was found to be about 50 %. This degradation implied that there was the interaction between inlet and combustor in the engine, but the details have not been clarified. To efficiently improve the engine performance, we must understand the aerothermodynamic processes determining the engine performance and optimize the flowpath to reduce the interaction.

So, the numerical simulations of the engine under Mach 6 condition were conducted. Additionally, the calculations of sector combustor with uniform inflow were performed to simulate the direct connect tests and compared with the engine simulations. These studies revealed flame structures in the engine and how the combustion performance was controlled by them. The following conclusions were drawn:

- Time-wise developments of the reacting zone in the engine showed that the fuel jets near the topwall could not burn at first. Autoignition assisted by the secondary flow induced by combustion-generated high-pressure was the mechanism of flame development to stabilize the flame near the topwall.
- The engine at the Mach 6 condition operated in two combustion modes: One was a weak or reaction controlled combustion mode where the flame anchored downstream of the combustor, and the other was an intensive or diffusion controlled mode where the flame attached to the entrance of combustor. The mode transition was also controlled by the secondary flow due to the combustion.
- Steady-state solutions showed that the combustion flowfield in the engine was consisted of small separated flames around individual jets near the fuel injectors and a large envelope diffusion flame downstream of the combustor. The small-separated flames around individual jets produced 80% of the combustion efficiency achieved in the engine.
- The comparison between the engine and sector combustor simulations showed the large flame structure downstream of the combustor in the engine was caused by the cowl shock and hindered the increase of the combustion efficiency. The alleviation of the flow distortion in the inlet producing the cowl shock is very important to improve the engine performance.
- These flame structures in the engine and the combustion efficiency distributions showed the way to improve the engine design. To improve the combustion performance of the engine, the transverse injection on the strut or the parallel injection from the strut base of the cowl side should be adopted in addition to the transverse injection on the sidewall.

論文審査結果の要旨

スクラムジェットエンジンの燃焼性能試験には，大規模地上設備や飛行実験機が必要であり，容易には行えない。また，実験を行っても，エンジン形状の複雑さや厳しい試験環境などから，内部の流動及び燃焼状態の詳細を知ることは難しく，得られた実験データを，エンジンの運転条件選定や形状設計に反映させることには大きな困難を伴う。そこで，エンジン内部の流動及び燃焼に関する詳細なデータを得ることができる数値シミュレーションに対し，高い期待が寄せられている。しかし，計算コードには乱流や化学反応に関するモデルが組み込まれており，スクラムジェット内の極限的作動条件におけるこれらモデルの妥当性は，エンジンレベルではほとんど検証されていない。本論文は，マッハ6飛行条件においてスクラムジェットエンジン全体の数値シミュレーションを行い，実験結果に基づく十分な検証を行った上で，実験で観察された燃焼器と空気取入れ口の大規模な干渉の機構を解明し，その抑制法や燃焼状態とその改善方法について考察した結果をまとめたもので，全編7章よりなる。

第1章は序論である。

第2章では，数値シミュレーションの計算コードを示し，格子間隔が結果に及ぼす影響を調べて，エンジン全体の計算を行うために妥当な，主流部分及び壁面付近における格子間隔を決定している。

第3章では，前章で決定した格子間隔を用いて行った数値シミュレーション結果を，エンジン試験結果と比較することにより，シミュレーションの妥当性を検証している。比較は，局所的な流れの特性を表す壁圧や壁面熱流束の分布とともに，全エンジンの性能指標である空気捕獲率や抗力と推力についても行われ，いずれも定量的に良い一致を得ている。これにより，本論文で用いている数値シミュレーションの妥当性は十分に検証されている。

第4章では，エンジン内の各断面における質量平均量を主流状態として求めた表面摩擦係数とスタントン数を比較し，多くの圧力波が存在するスクラムジェットエンジン内でも90%以上の領域で，摩擦応力と熱流束の間のレイノルズ・アナロジーが成り立つことを示している。これはスクラムジェットエンジンの熱解析上有用な知見である。

第5章では，壁温及び境界層吸い込みがエンジンの作動状態に及ぼす影響を調べている。壁温が低い場合に境界層の剥離が起こり難いこと，壁温が高い場合でも捕獲空気量の 0.6%程度を境界層から吸い込むことによって，剥離を抑え多量の燃料を燃焼させてもエンジン不始動に陥らず，推力を増大できることを示している。これはエンジンの作動範囲を拡大する上で，非常に重要な成果である。

第6章では，エンジン内の火炎構造について調べ，空気取入れ口の気流の偏りが火炎を大きく変形し，燃焼性能の低下を招くことを明らかにしている。さらに，この性能低下を回避するための，新たな燃料噴射方法を提案している。これらは，スクラムジェットエンジンの空気取入れ口及び燃焼器の設計において極めて有益な結果である。

第7章は結論である。

以上要するに本論文は，スクラムジェットエンジン内の流動と火炎構造を数値シミュレーションによって明らかにし，その結果に基づいてエンジン性能向上のための方策を提案したもので，航空宇宙工学および推進工学の発展に寄与するところ少なくない。

よって，本論文は博士(工学)の学位論文として合格と認める。