

Development and Preliminary Validation of A Thermal Analysis Method for Hydrocarbon regenerative-cooled Supersonic Combustor

Yang Lu¹, Xinzhu Wang, Long Li, Di Cheng, Tao Yuan, Wei Yao, Xuejun Fan²

LHD, Institute of Mechanics, Chinese Academy of Sciences, Beijing, 100190, P. R. China

In this research, a thermal analysis method has been developed to analyze the heat transfer process associated with endothermic hydrocarbon regenerative-cooled structure of a combustor. The complex heat transfer processes relevant to such cooling structure exposed in severe heat environment are modeled by three coupled processes: hot side boundary condition specification, flow and convective heat transfer of fuel within cooling channels and heat transfer in combustor structure with cooling channels embeded. To speed up the simulation process while achieving good accuracy, efforts are made in several aspects: first, hot side heat environment of combustor is obtained either by measurement results from upgraded heat flux sensors developed based on the principle of Gardon heat-flux gauge or by quasi 1-D analysis of combustor using static pressure distribution as input; second, to quantitatively describe flow and heat transfer behavior of hydrocarbon coolant when heavy cracking happens, a five-component surrogate model is developed and together used with a thermal cracking model consisting of 18 species and 24 reactions; third, to account for multiple effects happening in redistribution channels, a special method for flow rate redistribution prediction is developed based on characteristic time scale analysis. The validness of this integrated analysis method is tested by comparing simulation results with measurement data from lab tests of a supersonic model combustor. The overall correctness of exit coolant fuel temperature and wall temperature distribution prediction is within 5% and 10% separately. The fast speed and decent accuracy of this method developed make it very promising to be put into use for hydrocarbon regenerative-cooling analysis.

Nomenclature

A	=	area
S	=	perimeter
q	=	heat flux
ρ	=	density
h	=	enthalpy
htc	=	heat transfer coefficient
T_w	=	temperature of wall , T_f = temperature of fuel
τ_w	=	friction of wall
C_f	=	friction coefficient
S_t	=	Stanton number
N_u	=	Nusselt number
P_r	=	Prandtl number

¹ Assistant Professor, National Key Laboratory of High Temperature Gas Dynamics, luyang@imech.ac.cn.

² Professor, National Key Laboratory of High Temperature Gas Dynamics, xfan@imech.ac.cn.

I. Introduction

Hypersonic (typically exceed Mach 5) vehicles, no matter reusable or not, and the related propulsion techniques draw continuous research interests for their latent commercial travel, defense, transportation or space exploring use. However, such high speeds introduce corresponding high heat loads and raise great challenge for effective heat protection. Regenerative cooling has been proposed¹ to be a promising thermal management technique to solve the problem, in which onboard fuel is used to cool part of the flying vehicle before injected into the combustor to burn and gain thrust. It means the fuel plays another role as coolant. Therefore, the selection for such a suitable working substance is of great importance. As shown by Lander and Nixon[2], a Mach 8 scramjet engine could require fuel heat-sink level of 3.5 MJ/kg, which fortunately could be obtained[3] either from sensible heating of liquid hydrogen or from endothermic hydrocarbon fuels when deliberate use of bulk reaction (e.g., thermal cracking, dehydrogenation) are involved. In fact, several researches[4,5] have proved that a few different types of hydrocarbon fuels can meet these heat sink needs by additive effect of both their sensible heating ($C_p\Delta T$) and “chemical heating” with endothermic (heat-absorbing) reactions. Furthermore, as heavy hydrocarbons will produce lighter species that are easier to mix and ignite when thermal degradation begins, it may be possible for them to complete the combustion process before dashing out of the combustion chamber exit. This will not only help to fully realize the potential of heat releasing to increase thrust but also help to reduce the length and weight of combustor. Additionally, due to the greater density and relatively easier handling [3], hydrocarbon fuels are considered good candidates for hypersonic aircraft practical use for Mach number below 8. Moreover, if air-breathing designs were combined used with regenerative cooling, the complexity and total weight of hypersonic vehicle can be further reduced and it would be more cost-effective. Research program [6] following these ideas was forwarded step by step to design hydrocarbon scramjet engines for hypersonic propulsion. Furthermore, recent success of 240s-duration(longest in hypersonic flight history so far) test flight of X-51 has demonstrated the reliability of the concept of regenerative cooling using endothermic hydrocarbon fuels to be a practical long-time comprehensive thermal management solution for hypersonic vehicles.

Despite all the merits and bright prospects, trying to understand thoroughly and finally realizing the regenerative cooling technique for hypersonic air-breathing propulsion is no easy task. The heat transfer processes related to an endothermic hydrocarbon regenerative cooling system can be very complex, which here are divided into three coupled processes: (1) heat from combustion and aerodynamic heating transferred to the inner wall, (2) heat conducted through the engine structure, and (3) heat absorbed by coolant fuel when flowing and reacting within cooling channels. The last two processes are sometimes combined studied and termed as “conjugate heat transfer” [7]. Any of these heat transfer processes is in fact rather complicated and requires close examination. Firstly, look at the coolant fuel. On one hand, endothermic hydrocarbon fuels for practical use are often mixtures of hundreds of species. It is inevitable for researches or engineers to find a suitable surrogate model for use. On the other hand, the fuel temperature varies significantly in the cooling channels. Under expected working pressures, the fuel can change from pressured liquid state to supercritical state [8,9] and finally to partially cracked[10-12] or even to largely cracked state[13]. Consequently, dramatic changes in the thermo-physical, transport properties and especially chemical effects of the endothermic fuel should be considered simultaneously with flow properties, such as friction effect, in the coolant flow analysis. Secondly, mechanisms of turbulent mixing and combustion in high-speed flows, as well as their interactions with shocks in the engine remain not fully understood. Moreover, as the regenerative cooling system works in close-loop mode, the combustion characteristics of liquid, supercritical or cracked fuel can be of notable difference [14,15] and may be hard to capture and describe using simple models. As a result, the predictions of thermal environment near hot-side wall along the engine’s internal flow passage using even advanced CFD tools still need improvement or are too time-consuming. Thirdly, the heat transfer within the three-dimensional combustor structure need treatment with care. Temperature dependent thermal conductivities of different materials should be addressed because there are usually coating layers or different materials for different engine components. Besides, as there are usually (see figure.1) geometry divergent sidewall cooling panels and other functioning structures, e.g. cavities, flow redistribution channels are always embedded from space to space where necessary to re-distribute coolant flows. Heat transfer and flow in these specific local structures should be examined carefully.

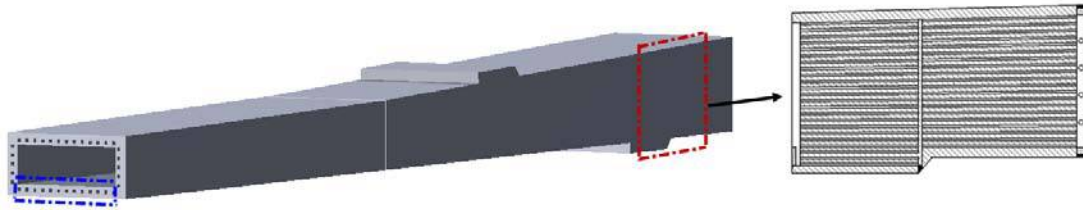


Figure 1. Schematic drawing of combustor(left) and part of its side cooling structure near cavity(right)

In order to aid in analyzing, designing and optimizing actively cooling system, in addition to above-mentioned fundamental researches, substantial efforts have been made over the years in developing integrated analyzing tools. Traci et al. [16] established a systematical thermal analysis code named VITMAC for active cooling in RBCC system. However, it was primarily developed for rocket system cooling panel applications using cryogenic fluids. Cracked model for hydrocarbon fuels was not included and cooling panels were only simulated as a one-dimensional structure. Gamble et al. [17] developed a new heat exchanger analysis tool, SRHEAT, for scramjet/ramjet active cooling use. It improved the structure simulation by using 2-D heat conduction model but the treatment for fuel chemical reaction effect was still missing. Besides, it used heat transfer correlations based on experimental data of water and may need further validation of applicability for endothermic fuels. Bouchez et al. [18] brought in a semi-empirical code named NANCY for active cooling analysis in dual-mode ramjets, in which coolant fuel reactivity is taken into account and analyzed. However, they only used one single species (n-dodecane) kinetic model to compute the pyrolyzed coolant fuel temperature, which was a good start but may not be adequate.

To make full use of heat sink capabilities of endothermic fuels for hypersonic thermal management, further studies for regenerative cooling related heat transfer phenomena are still in need. The aim of this research is to provide such an integrated and rapid thermal analysis method that important aspects in analyzing hydrocarbon regenerative-cooled combustor structure are carefully examined and experimentally verified. The main characteristic of this method is the special ways to estimate thermal environment along flow direction within combustor. Besides, by employing a newly-developed thermal cracking model[13] for endothermic hydrocarbon fuels, it is now possible to analyze the behavior of largely cracked(conversion rate more than 50%) fuel. Details are given respectively in the following sections. The validness and overall correctness of this analysis tool is satisfactory as simulation results showed good coherence with preliminarily combustor lab test experimental results (under both aerodynamic heating and combustion conditions).

II. Overview of analysis method and iteration process

Figure 2 gives a schematic drawing of a cross section of fuel-cooled heat exchange structure for analysis, which amplifies the blue-colored part in Fig.1(left). Subscript 1 represents variables of combustor inner hot wall. Subscript 2 represents variables of the cooling channel wall. Subscript 3 represents variables of combustor outer wall. h in Fig.1 and 2 denotes heat transfer coefficient.

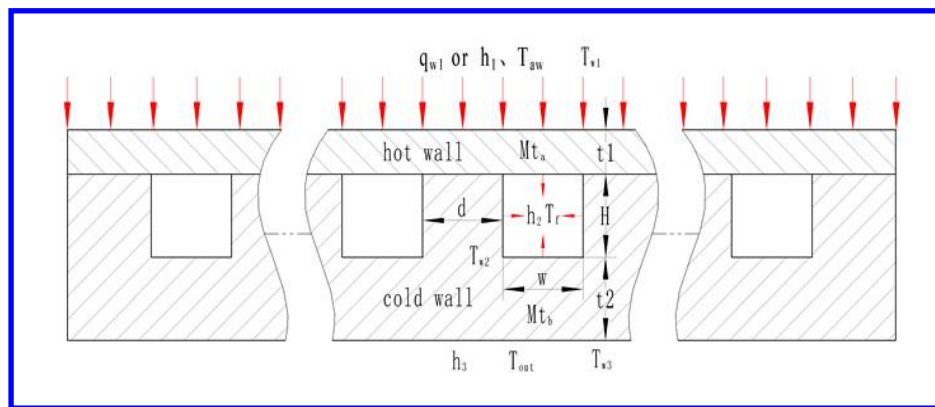


Figure 2. Schematic drawing of the cross section of fuel-cooled structure analyzed

Firstly, with heat flux experimental data or performing 1-D analysis with static pressure distribution as input data and assumed inner hot wall temperature(T_{w1}) distribution, hot wall heat flux q_{w1} distribution along flow direction can be calculated. Secondly, with assumed cooling channel wall temperature(T_{w2}) distribution, the cooling channels analysis module can be invoked and it provides distributions of convection coefficient h_2 and coolant fuel temperatures T_f within every cooling channels. After that, heat conduction analysis module is invoked and temperature distribution within cross section of cooling structure can be obtained. Then, T_{w2} distribution, as well as T_{w1} distribution(if 1-D analysis involved), can be updated and used for next round analysis. Iteration is carried on until final convergence is reached. Figure 3 shows the whole computational process. Details and sub-models of each module are given in the next part.

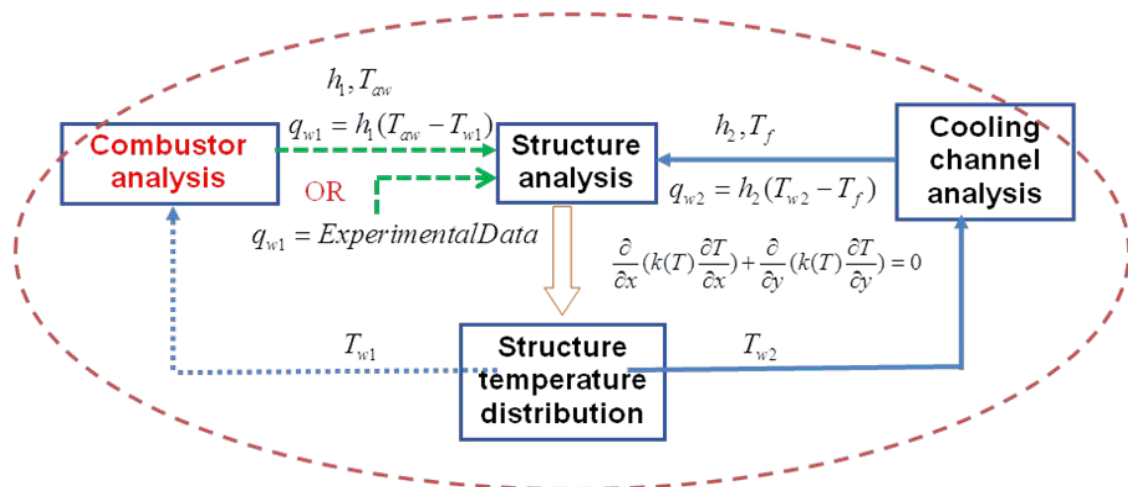


Figure 3. Relation of computation modules and iteration process

III. Detailed description of the analysis method

A. Thermal environment prediction and inner hot wall boundary condition specification

One key feature of this research is the inner hot wall boundary condition specification. As explained in the introduction, the high-precision predictions of thermal environment of engine hot-side wall is very complex and of great challenge, especially when fast analysis is in need. Here the specification can be achieved in two ways: one is to obtain highly-accurate heat flux distribution with specially-designed high-temperature heat flux gauges and related techniques developed by authors' group, details of which can be found elsewhere[19-21]. Although this method is great with its directness and simplicity in reasoning, sometimes it is not easy to implement due to installation difficulties or other limitations. The other is by using static pressure distribution as input of 1-D combustor analysis to give a feasible estimation. The merits of doing this are manifold: it can reflect the effect of complicated combustion in high-speed flows while avoid many unconfirmed assumptions about flows and combustion, the measurement of static pressure is a mature technology and it is much faster compared with CFD simulations. Generalized one-dimensional aerodynamic integral equations, in which effects of varied geometry, friction, heat and mass addition, are jointly used to analyze the flow and heat transfer in scramjet combustor. Figure 4 is the schematic of computation element.

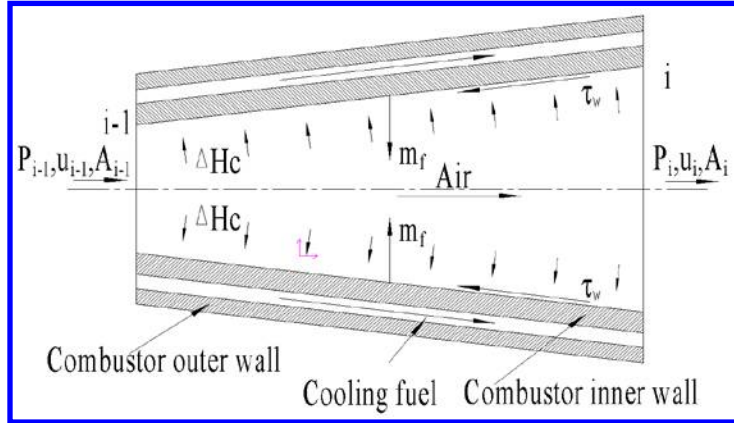


Figure 4. Computation element of the quasi-one-dimensional combustor model

In continuity equation, mass addition effect of fuel injection is accounted for:

$$\dot{m}_i = \rho_i u_i A_i + \dot{m}_{f,i} \quad (1)$$

$$\dot{m}_{f,i} = \begin{cases} \dot{m}_f & x_j - \Delta x / 2 < x_i < x_j + \Delta x / 2 \\ 0 & \text{elsewhere} \end{cases} \quad (2)$$

In the momentum equation, friction is considered and static pressure distribution along flow direction is used:

$$\dot{m}_{i-1} u_{i-1} + p_{i-1} A_{i-1} + \int_{i-1}^i p dA = \dot{m}_i u_i + p_i A_i + \int_{i-1}^i \tau_w dA_{inner} \quad (3)$$

Wall friction shear stress is calculated as:

$$\tau_w = \frac{C_f}{2} \rho u^2 \quad (4)$$

In which friction coefficient is modeled by Reynolds analogy:

$$C_f = 2St \cdot Pr^{2/3} \quad (5)$$

Energy equation considering heat addition is used to estimate combustion efficiency:

$$\dot{m}_{i-1} \left(h_{i-1} + \frac{1}{2} u_{i-1}^2 \right) = \dot{m}_i \left(h_i + \frac{1}{2} u_i^2 \right) + \Delta H_{c,i} \quad (6)$$

To describe convective heat transfer near combustor inner wall, the convective heat flux is calculated by turbulent flat-plate boundary layer convective heat transfer equation and the refined reference enthalpy method proposed by Meador and Smart [22] is used, in which superscript star represents variables at reference point:

$$q_{w1} = St^* \rho^* u (h_{aw} - h_{w1}) \quad (7)$$

$$h^* = \frac{h + h_w}{2} + 0.16r \frac{u^2}{2} \quad (8)$$

Thermal properties of high temperature gas in the combustor are calculated based on NASA database [23] and perfect gas law.

It should be pointed out that here the radiation heat flux in the combustor is not taken into account, which may lead to a little underestimation of q_{w1} .

B. Flow, cracking and heat transfer of endothermic fuel within cooling channels

(i) Development of fuel surrogate model

As the very first step to study numerically heat transfer process of endothermic hydrocarbon fuel in cooling channels, a surrogate model for coolant fuel is in need. The selection of candidates for the species pool is based on chromatography and mass spectrography results[24,25] of the hydrocarbon fuel used in the experiments as well as SUPERTRAPP species database. Table 1 shows the species pool.

Table 1. Species pool

species name	formula
decane	C ₁₀ H ₂₂
undecane	C ₁₁ H ₂₄
dodecane	C ₁₂ H ₂₆
tridecane	C ₁₃ H ₂₈
1-cis,2-trans,4-trimethylcyclohexane	C ₉ H ₁₈
n-butylcyclohexane	C ₁₀ H ₂₀
propylbenzene	C ₉ H ₁₂
butylbenzene	C ₁₀ H ₁₄
methylnaphthalene	C ₁₁ H ₁₀

The determination of species and the weight for individual species for the final surrogate model is not arbitrarily chosen but is derived focusing on the problems – the convective heat transfer and coolant fuel temperature. Start from the classic D-B formula:

$$Nu = 0.024Re^{0.8}Pr^{0.4} = \frac{htc \cdot D}{\kappa} \quad (9)$$

$$htc = 0.023 \cdot \left(\frac{4Q}{\pi D} \right)^{0.8} \cdot \frac{1}{D} \cdot \rho^{-0.4} c_p^{0.4} \kappa^{0.6} \nu^{-0.4} \quad (10)$$

$$\frac{dhtc}{htc} = 0.4 \frac{dc_p}{c_p} + 0.6 \frac{d\kappa}{\kappa} - 0.4 \frac{d\nu}{\nu} - 0.4 \frac{d\rho}{\rho}$$

From the equation above, it can be known that heat transfer coefficient is related simultaneously with density, specific heat at constant pressure, thermal conductivity and kinematic viscosity. It means the propagation of error of all these properties should be considered. In other words, if decent accuracy is desired to describe heat transfer process in cooling channels, the development of a surrogate model should at least account for all these important properties. Following this idea, how to find a good surrogate model is in fact a multiple targets optimization problem. The target optimization function F is here chosen as:

$$F^2 = \left[W_{c_p} \cdot \frac{1}{N_{c_p}} \sum_{i=1}^{N_{c_p}} \text{abs} \left(\frac{c_{p,Cal,i} - c_{p,Exp,i}}{c_{p,Exp,i}} \right) \right]^2 + \left[W_{\kappa} \cdot \frac{1}{N_{\kappa}} \sum_{i=1}^{N_{\kappa}} \text{abs} \left(\frac{\kappa_{Cal,i} - \kappa_{Exp,i}}{\kappa_{Exp,i}} \right) \right]^2 \quad (11)$$

$$+ \left[W_{\nu} \cdot \frac{1}{N_{\nu}} \sum_{i=1}^{N_{\nu}} \text{abs} \left(\frac{\nu_{Cal,i} - \nu_{Exp,i}}{\nu_{Exp,i}} \right) \right]^2 + \left[W_{\rho} \cdot \frac{1}{N_{\rho}} \sum_{i=1}^{N_{\rho}} \text{abs} \left(\frac{\rho_{Cal,i} - \rho_{Exp,i}}{\rho_{Exp,i}} \right) \right]^2$$

in which $W_z, z \in \{c_p, \kappa, \nu, \rho\}$ represents the weight for each of the four physical properties. The calculated value in equation (11) is obtained using SUPERTRAPP. The task is to find a weighted group from the original species pool to get a minimum value of F possible. Now it is time to determine the unknown W_z . Comparing equations (10) and (11), it is natural to set $W_{c_p}^0 = 0.4, W_{\kappa}^0 = 0.6, W_{\nu}^0 = 0.4, W_{\rho}^0 = 0.4$. However, as c_p is crucial to determine the coolant fuel temperature, $W_{c_p}^0$ is adjusted to 1.0 instead.

Now the whole problem can be summarized as following:

$$\min_x F(x) \begin{cases} c(x) \leq 0 \\ ceq(x) = 0 \\ A \cdot x \leq b, \\ Aeq \cdot x = beq \\ lb \leq x \leq ub \end{cases}$$

in which c equations represent non-linear restrictions, A equations represent linear restrictions and the last is variable restrictions. First linear restriction is the sum of all mole fractions of species should be 1. Other restrictions are listed in table 2 and table 3.

Table 2. Species boundaries (mole fraction)

Category	Restrictive condition	Species name	Lower bound	Upper bound
linear paraffin	Lower bound: 0.40 Upper bound: 0.70	decane	0.00	0.70
		undecane	0.00	0.70
		dodecane	0.00	0.70
		tridecane	0.00	0.70
cyclanes	Lower bound: 0.20 Upper bound: 0.50	1-cis,2-trans,4-trimethylcyclohexane	0.00	0.50
		n-butylcyclohexane	0.00	0.50
aromatics	Lower bound: 0.07 Upper bound: 0.25	propylbenzene	0.00	0.25
		butylbenzene	0.00	0.25
		methylnaphthalene	0.00	0.25

Table 3. Other restrictions

Restrictive condition	Lower bound	Upper bound
average C atom number	10.00	12.00
critical pressure/bar	22.00	26.00
critical temperature/K	625.00	665.00

After all the above preparation, it is finally to obtain a surrogate model for the coolant fuel used. Table 4 shows the resulting five component optimized surrogate model:

Table 4. Optimized surrogate model

Category	Mole fraction	Species name	Mole fraction
linear paraffin	0.67	decane	0.4087
		undecane	0.00
		dodecane	0.00
		tridecane	0.2613
cyclanes	0.20	1-cis,2-trans,4-trimethylcyclohexane	0.0856
		n-butylcyclohexane	0.1144
aromatics	0.13	propylbenzene	0.1300
		butylbenzene	0.00
		methylnaphthalene	0.00

(ii) Analysis of coolant flow and convective heat transfer

Now the surrogate model specially optimized for heat transfer calculation is developed, the next step is to account for general flow and heat transfer process of coolant fuel. Considering that practically-used cooling channels are usually long and narrow, convective heat transfer process of fuel in such cooling channels is here approximated to be one dimensional. Figure 5 shows the control volume for analysis:

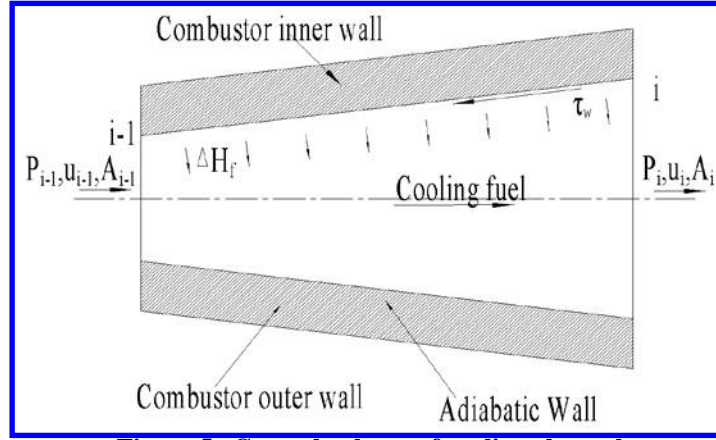


Figure 5. Control volume of cooling channel

To speed up simulation, integral governing equations other than Navier-Stokes equations are used:

$$\begin{aligned}\dot{m}_f &= \rho_i u_i A_i = \rho_{i-1} u_{i-1} A_{i-1} \\ \dot{m}_f u_{i-1} + P_{i-1} A_{i-1} - \int_{i-1}^i P dA &= \dot{m}_f u_i + P_i A_i + \int_{i-1}^i \tau_w dA_f \\ \dot{m}_f \left(h_{i-1} + \frac{u_{i-1}^2}{2} \right) + \Delta H_{f,i-1} &= \dot{m}_f \left(h_i + \frac{u_i^2}{2} \right)\end{aligned}$$

\dot{m} is the mass flow rate of the coolant fuel in a cooling channel. $\Delta H_{f,i-1}$ denotes the heat absorbed by coolant from the wall of the cooling channel and its formula is:

$$\Delta H_f = \dot{q}_{w,f} S_f \Delta x, \quad \dot{q}_{w,f} = htc_f (T_{w2} - T_f), \quad htc_f = \frac{Nu_f \kappa}{D_{eh}}$$

S_f is the local perimeter of the cooling channel cross section and D_{eh} is the hydraulic diameter of the cooling channel. Distributions of cooling channel wall temperature - T_{w2} , which are assumed here, and the heat transfer coefficient of cooling channel wall - htc_f will be used in structure analysis later. The assumed distribution of cooling channel wall temperatures will be replaced after the first-time run of heat transfer analysis of cooling structure (discussed in the next section). Then it will be updated in each round of iteration until convergence is achieved.

The whole cooling channel is divided into lots of subsections to insure the difference of properties of two adjacent sections is small and the following approximations can be used:

$$\int_{i-1}^i P dA \approx \frac{P_i + P_{i-1}}{2} (A_i - A_{i-1}), \quad \int_{i-1}^i \tau_w dA_{ch} \approx \frac{\tau_{w,i-1} S_{f,i-1} + \tau_{w,i} S_{f,i}}{2} \Delta x$$

By definition for Newton fluid, we have :

$$\tau_w = \frac{C_f}{2} \rho u^2$$

And Reynold analogy is used to get friction coefficient:

$$\frac{C_f}{2} = St Pr^{\frac{2}{3}}, \quad St = \frac{Nu}{Re Pr}$$

The merit of doing this is that not only friction effect is considered but also pressure drops in cooling channels are no need to be estimated by using empirical formulas in after-calculation analysis as some studies have done, but rather are calculated during iteration and coupled within the governing equations. It adds the coherence of the whole analysis method.

As mentioned in the introduction, there is no justification for arbitrarily using formulas developed for water or any other matter other than hydrocarbon fuel to describe coolant heat transfer in regenerative cooling system. Suitable formulas are required for describing hydrocarbon fuel heat convection process. These formulas should be directly experimentally obtained with hydrocarbon fuel as working substance or at least are accepted to be applicable to such fluid, such as Gnielinski formula[26]:

$$Nu = 0.012(\text{Re}_f^{0.87} - 280) \text{Pr}^{0.4} \left(\text{Pr}_f / \text{Pr}_w \right)^{0.11}, 1.5 \leq \text{Pr}_f \leq 500, 3000 < \text{Re}_f < 10^6$$

Using all abovementioned equations and formulas, local velocity of fuel can be obtained iteratively. Fuel temperature and other important physical variables, which are considered functions of local fuel temperatures and pressures, can also be deduced. It should be emphasized that functions of SUPERTRAPP code are used again during computation to obtain these state and flow variables since hydrocarbon fuels are more complicated than water:

$$h_f = h_f(P, T), \dots, \rho_f = \rho_f(P, T)$$

(iii) Implementation of thermal cracking mechanism

When coolant fuel temperature is further rising, thermal cracked mechanism should be taken into account. In some previous researches[10-12], several experimental results based lumped global models for describing hydrocarbon cracking were introduced. However, these lumped mechanisms are often short of secondary reactions and have similar limitations as the precondition they can be used is that the fuel can only be mildly cracked or the conversion rates should not exceed 20% or so. While in some other researches[27,28], only one species is chosen to represent mixture fuel for cracking study, which may be not adequate and detailed kinetics models containing hundreds of species are involved in the models, which are too time-consuming for computing.

In this research, to extend the scope of applicability and speed up calculation, a recently developed modified molecular reaction model[13] consisting of 18 species and 24 reactions(1 primary reaction and 23 secondary reactions) is used for cracking simulation of coolant mixture fuel, which is able to describe thermal cracking of supercritical hydrocarbon fuel with conversion rate up to 50% or even higher while the computing time is largely reduced. The comparisons of predicted and experimental results can be found in reference 13 and the average absolute relative error is equal to 9.2%.

In order to implement this mechanism to describe coolant cracking behavior, a steady, one-dimensional pseudoisothermal plug flow reactor model is used to analysis each small segment of cooling channel. So we have:

$$\frac{dF_j}{A \cdot dl} = \sum_{r=1}^{n_r} (v_{rj} r_r) \quad (12)$$

$$r_r = \begin{cases} k_r C_i \\ k_r C_i C_j \end{cases} \quad (13)$$

To solve this problem, fourth order Runge-Kutta method is used here. The characteristic temperature used for reaction rate calculation is estimated by the arithmetic average of inlet and outlet temperature of coolant fuel of this small segment. Exit temperature as well as compositions of each small sector of cooling channels can be iteratively obtained and then by using SUPERTRAPP, properties of cracked coolant fuel can be deduced.

C. Heat transfer analysis within cooling structure

Considering that the cooling panel usually long but thin and the temperature gradient in flow direction is smaller than that in the cross section plane, the problem are here reduced to a series of two-dimension heat conduction problems within every cross plane along flow direction by neglecting heat conduction between two adjacent cross planes. Moreover, benefitting from this simplification, this part can be conveniently programmed in parallel, which turned out to be a very efficient method to perform on multi-core multi-threads desktops to speed up calculation.

It is noteworthy that many previous researches treated the problem by simply analyzing only one channel within the structure with periodic boundary condition. However, as shown in Fig. 2, in this study the whole cooling panel instead of one single cooling cell is used for heat analysis of rectangular cross section cooling structure.

The controlling equation for two-dimensional heat conduction is:

$$\rho c \frac{\partial T}{\partial t} = \frac{1}{\partial x} \left(\lambda \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(\lambda \frac{\partial T}{\partial y} \right) + S$$

A standard practice to solve this equation is to use finite volume method. However, when multiple channels and lots of cross sections along flow direction are involved, the cost of computation can be fairly large. In this research, only steady state situation is considered. Therefore, the left hand side and the source term of above equation are zero. Then, a thermo-resistance formula is employed and an algebra expression[29] for $T_{i,j}$ can be used:

$$T_{i,j} = \frac{\frac{T_{i+1,j}}{R_1} + \frac{T_{i,j-1}}{R_2} + \frac{T_{i-1,j}}{R_3} + \frac{T_{i,j+1}}{R_4}}{\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \frac{1}{R_4}}$$

$$\text{With } \begin{cases} R_1 = \frac{\Delta X_{i+1,j}}{\Delta Y_{i+1,j}} \frac{1}{2} \left(\frac{1}{K_{i,j}} + \frac{1}{K_{i+1,j}} \right), R_2 = \frac{\Delta Y_{i,j-1}}{\Delta X_{i,j-1}} \frac{1}{2} \left(\frac{1}{K_{i,j}} + \frac{1}{K_{i,j-1}} \right) \\ R_3 = \frac{\Delta X_{i-1,j}}{\Delta Y_{i-1,j}} \frac{1}{2} \left(\frac{1}{K_{i,j}} + \frac{1}{K_{i-1,j}} \right), R_4 = \frac{\Delta Y_{i,j+1}}{\Delta X_{i,j+1}} \frac{1}{2} \left(\frac{1}{K_{i,j}} + \frac{1}{K_{i,j+1}} \right) \end{cases}$$

R_i ($i=1\dots$) denotes thermo-resistance from a specific direction. $K_{i,j}$ denotes thermal conductivity coefficient, which varies with temperature as $K(T) = AT + B$ in a wide range. A and B are material-related constants and can be retrieved from material database. For rectangular cross-section channels in rectangular coordinates (as shown in Fig.1):

$$\Delta X_{i+1,j} = \Delta X_{i-1,j} = \Delta X_{i,j+1} = \Delta X_{i,j-1} = \Delta x$$

$$\Delta Y_{i+1,j} = \Delta Y_{i-1,j} = \Delta Y_{i,j+1} = \Delta Y_{i,j-1} = \Delta y$$

Free convection or adiabatic boundary conditions are applied to all walls except the inner hot wall in that regenerative cooling system is exposed in the open air. The hot wall boundary condition is obtained using method discussed in section A. The coolant fuel temperature and heat transfer coefficient are calculated from analysis of convective heat transfer of fuel in section B. Now, boundary conditions specifications are completed and heat transfer process of cooling structure can be analyzed.

Besides, as mentioned in the introduction part, there are usually (see Fig.1) flow redistribution channels embedded from space to space in cooling structure where necessary due to functioning purposes or manufacturing needs. Flow and heat transfer characteristics in these special local structures should be closely examined. First, the respective characteristic time of flow, heat transfer and pressure propagation in the flow redistribution channel will be analyzed and estimated. Then base on these characteristic time data, two assumptions will be made and a method is developed to calculate the redistributed flow rate of each cooling channel where redistribution channel located.

a) characteristic flow time

For a typical case, the flow rate of one single cooling channel is 1-6g/s. The density of heated coolant hydrocarbon is 0.1-0.7g/cm³. The hydraulic diameter is 0.1-0.4mm. As a result, the average speed of coolant fuel is 2-10 m/s. On the other hand, the characteristic length of redistribution channel is 0.01m along flow direction while 0.1m in cross flow direction. Take sudden expansion effect of redistribution channel into consideration, the characteristic flow time is correspondingly 0.01-0.1 s.

b) characteristic heat transfer time

Under typical working conditions, coolant fuel will become turbulent soon after entrance into the coolant channel. As turbulent Prandtl number is around 1.0, the characteristic heat transfer time will be close to the characteristic flow time, which is between 0.01-0.1 s.

c) characteristic pressure propagation time

Pressure propagate at the speed of sound. For coolant fuel at typical working status, speed of sound is larger than 200 m/s. It is mentioned above that the characteristic length of redistribution channel is 0.1m. So the characteristic pressure propagation time will be less than 0.005 s, which is much smaller than the characteristic flow time or the characteristic heat transfer time.

Base on above time scale analysis, two assumptions can be made when describing what happen in the flow redistribution channel when coolant flows through:(1) the pressure will influence one another among multiple cooling channels;(2) coolant temperature of one cooling channel will only be affected by nearby cooling channels. Following these two points, treatment for pressure and temperature for the flow redistribution channel is that: downstream pressure for all cooling channels is the average of all the pressures of upstream cooling channels while

the downstream temperature for one cooling channel is the average of temperatures of two most near upstream cooling channels. Furthermore, a method to estimate redistributed flow rates of every cooling channel can be developed. Flow resistance concept is introduced to help in analysis and the details are given below. Figure 6 shows the schematic drawing of analysis cell between two flow redistribution channels

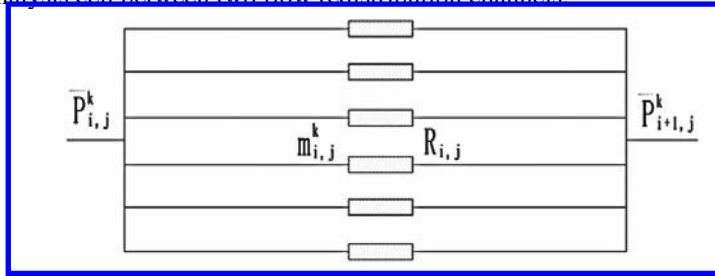


Figure. 6 Schematic drawing of the analysis cell between two flow redistribution channels

The initial guess value of the flow rate of the j th cooling channel of total ncc cooling channels is given by:

$$\dot{m}_{i,j}^k = \frac{\dot{m}}{ncc_i}$$

The inlet pressure for the analysis cell i is the same and known for all cooling channels and is denoted as $\bar{p}_{i,j}^k$:

$$\bar{p}_{i,j}^k = \sum_{j=1}^{ncc_{i-1}} p_{i,j}^k / ncc_{i-1}$$

Then using the method described in section III-B, the pressure drop and the exit pressure for the j th cooling channel can be estimated, which is denoted as $p_{i+1,j}^k$. Now the flow resistance $R_{i,j}^k$ can be defined as:

$$R_{i,j}^k \equiv \frac{\bar{p}_{i,j}^k - p_{i+1,j}^k}{\dot{m}_{i,j}^k}$$

Then the flow rate of the j th cooling channel can be revised as below:

$$\tilde{m}_{i,j}^k = \dot{m}_{i,j}^k + \frac{1}{R_{i,j}^k} (p_{i+1,j}^k - \bar{p}_{i+1,j}^k)$$

Furthermore, the steady state flow rate for the j th cooling channel should be normalized and updated:

$$\dot{m}_{i,j}^{k+1} = \left(\frac{\tilde{m}_{i,j}^k}{\sum_{j=1}^{ncc_i} \tilde{m}_{i,j}^k} \right) \cdot \dot{m}$$

The whole process above is carried on till exit pressure of every cooling channel is identical. Now the updated flow rate for each cooling channel is the final equilibrium redistributed flow rate.

IV. Results and discussion

The integrated analysis method is verified by comparing computational results to experiment results obtained under the same working conditions.

To investigate the validness of this analysis method in different situations, firstly, one typical working condition without combustion is chosen. The experiment is performed on a direct connect supersonic model combustor facility designed and developed by authors' group. The high temperature vitiated air is created by burning configured amount of H_2 , O_2 and air and then accelerated to desired speeds of Mach numbers by nozzles to simulate designed flight conditions. Figure 7 gives a schematic drawing of the combustor part(starting from the isolator), which is 2000 mm total in length.

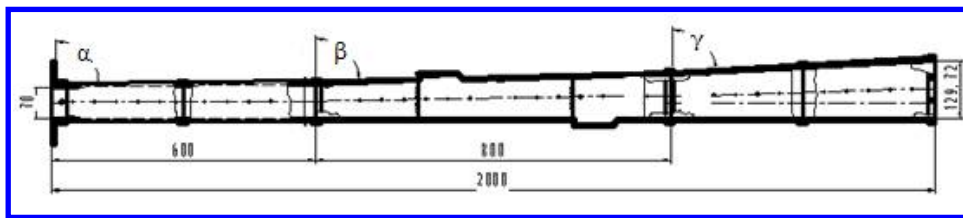


Figure 7. Schematic drawing of the hydrocarbon cooled supersonic model combustor for experiment

The inflow condition for the test case is listed here: total temperature is 1573K, total pressure is 0.938 MPa, flow rate is 2.936Kg/s and the inlet Mach number at the isolator entrance is 2.5. As stated in section III-A, the measured static pressure distribution is used to predict the heat environment within the combustor. However, as shown in Fig.8, the pressure near the exit(back pressure is 1 atm) runs up due to shock. Here the pressure after the sudden rising point is treated using isotropic expanding assumption as pictured in Fig. 8. Then the average static pressure of both side walls is submitted to the analysis program as input data.

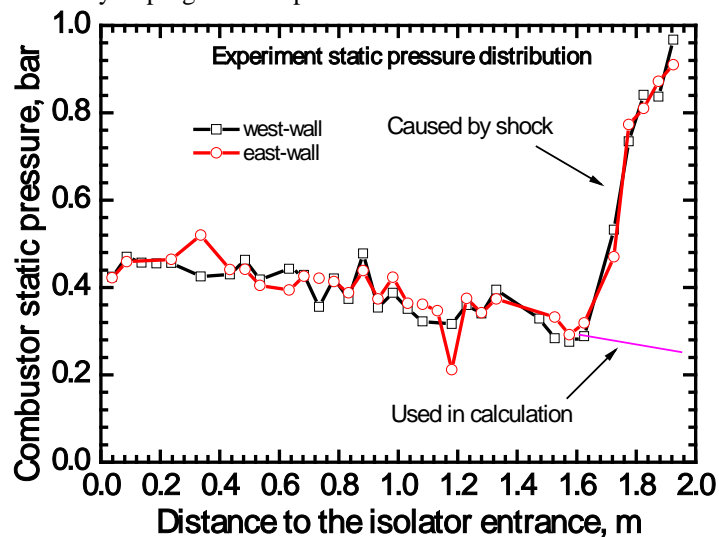


Figure 8. Measured static pressure distributions of the hydrocarbon cooled combustor for analysis

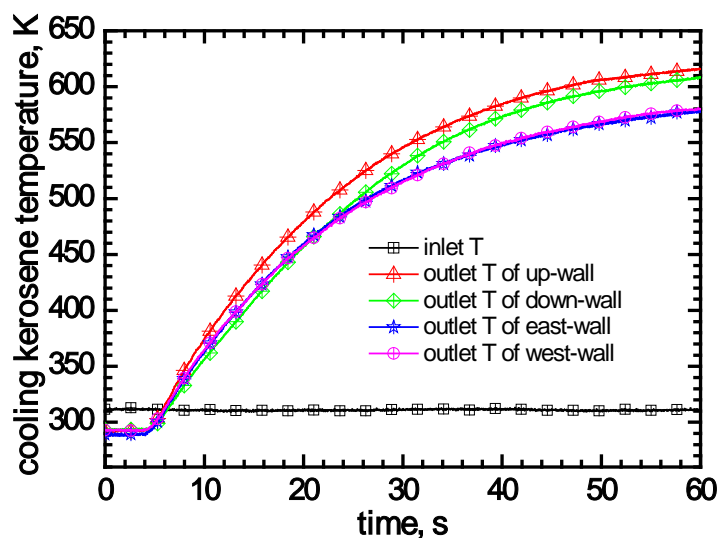


Figure 9. Time evolutions of measured exit coolant fuel temperatures of the combustor

Figure 9 gives the time evolutions of experimental measured coolant exit temperatures of the four cooling panels of the combustor. It can be seen after 60 s the coolant fuel temperatures almost reached steady values implying the thermal equilibrium for the whole system.

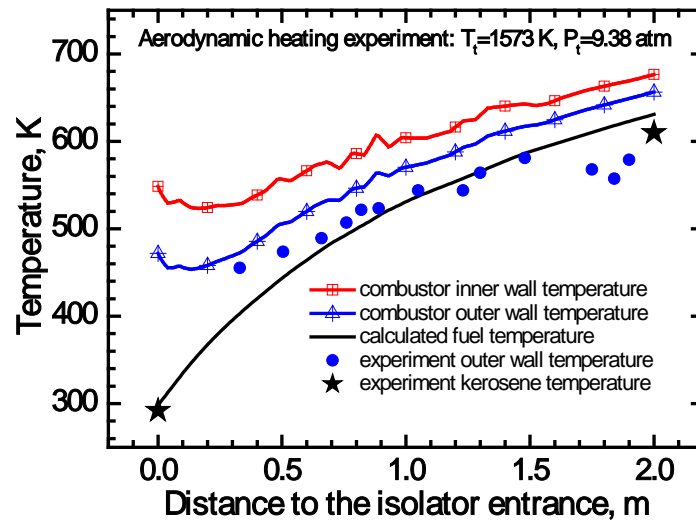


Figure 10. Calculated and measured distributions of the coolant fuel and combustor wall temperatures

In Figure 10, the calculated distributions of the average temperatures of coolant and combustor outer wall are compared with their measured counterparts. The calculated exit coolant fuel temperature is 3.3% higher than the measured. The calculated outer wall temperature is overall 8.1% higher than the experiment data with 6.5% higher in the area before the shock point while 13.3% higher after the shock point. Note that after the shock point, the static pressure used for simulation is treated using isotropic expanding assumption. It will lead to higher speed of the vitiated air flow and overestimate convective heat transfer, so the calculated wall temperature and exit coolant temperature will be higher than the experimental data. For this aerodynamic heating case without combustion, the overall correctness of the analysis method developed is within 10%.

The second case is to testify the validness of this integrated thermal analysis method under supersonic combustion condition with large coolant cracking rate. The experiment was carried out on the same test facility system described above. The inlet Mach number at the isolator entrance was 3.0. The kerosene-cooled supersonic combustion experimental parameters were: $T_t=1558$ K, $P_t=1.4$ MPa, $Q_t=2.363$ Kg/s, and the equivalent ratio of fuel is about 1.4.

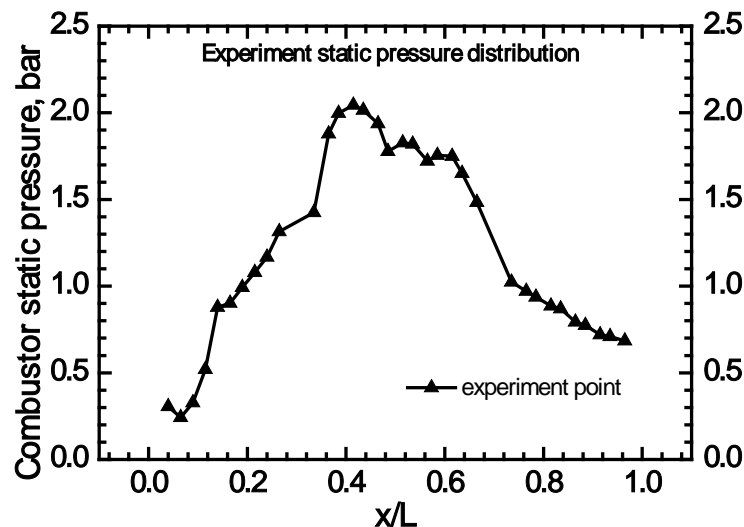


Figure 11. Measured static pressure distribution of the hydrocarbon cooled combustor for calculation

The combustion lasted for about 70 seconds, which was long enough for the whole system to reach thermal equilibrium state. Measured data then are suitable to be compared with simulation results. The measured combustion state static pressure distribution when equilibrium achieved, as depicted in Fig.11, is submitted to program as input data for thermal analysis.

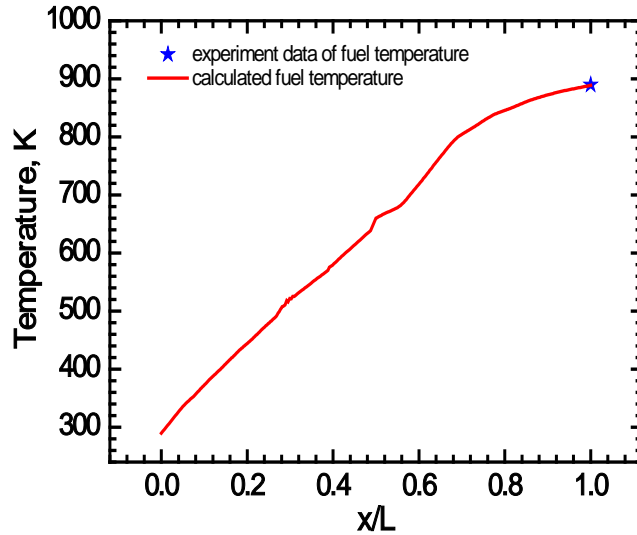


Figure 12. Calculated and measured coolant fuel temperatures

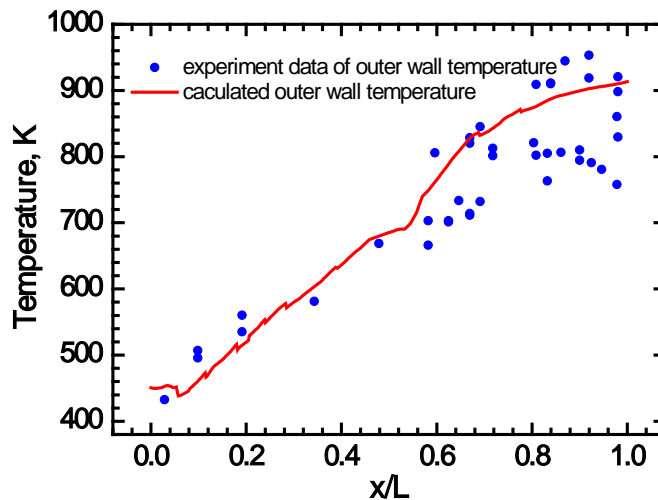


Figure13. Calculated and measured distributions of combustor outer wall temperatures

Figure 12 and 13 compare separately the calculated temperature distributions of coolant fuel and combustor outer wall along with their measured counterparts. It is shown in Fig. 12 that the coolant fuel temperature calculated (889K) is very close to that measured (892K) with a slightly lower of 3 K. The mass fraction of gas phase products (carbon atom number no more than 4) reaches as high as nearly 30%. In Fig. 13, it can be found several experimental measured temperature data points for one identical flow direction position. The reason is that the calculated temperature at one position in flow direction displayed in this figure is an averaged value along the outside wall line perpendicular to the flow direction while the measured data are obtained by thermocouples placed

in several locations along this outside wall line. Generally, the calculated outer wall temperature distribution also agrees very well with the experimental data.

It is encouraging to find out that these distributions agree with the experimental data even better than the abovementioned no-combustion case. The excellent agreement in both test cases between the measured and estimated data quantitatively demonstrated the validness as well as exactness of the integrated analysis method developed.

V. Conclusion and future work

As presented previously, this newly developed integrated heat analysis method is valid under both combustion or no combustion conditions with deviation less than 5% for coolant fuel temperature and no more than 10% for wall temperature, performing especially well when estimating the combustion case in which coolant is largely cracked. Besides, the computation cost is relatively low (several hours for a typical case). So this method is quite promising in engineering practice.

In the future, more work should be done to testify this method with experiment data over a wider range of working conditions. As the static pressure distribution is the crucial input data for this analysis method, an efficient and proper way to obtain such a pressure distribution before large-scale high-cost experiments performed is in need. Then this integrated method can be used not only for fast analysis but also for fast design and optimization of hydrocarbon regenerative cooling hypersonic systems.

Acknowledgments

Current research program at the LHD, CAS was supported by the National Natural Science Foundation of China under Contractor No. 91016005, 10621202. The authors would like to acknowledge Mr Linjin Meng, Mr Peng Huang, Mr Jinquan Wang and Mr. Y. Li for their technical support.

References

- 1 Wagner, W.R. and Shoji, J.M., "Advanced Regenerative Cooling Techniques for Future Space Transportation Systems," AIAA-75-1247, 1975.
- 2 Lander, H. and Nixon, A.C. Endothermic fuels for hypersonic vehicles. *J. Aircraft*, 8(4), 200, 1971.
- 3 T. Edwards, Liquid fuels and propellants for aerospace propulsion: 1903-2003. *Journal of Propulsion and Power*, Vol. 19, No. 6, 2003, pp. 1089-1107.
- 4 Huang, H., Sobel, D.R., and Spadaccini, L.J. Endothermic Heat-Sink of Hydrocarbon Fuels for Scramjet Cooling. AIAA Paper 2002-3871.
- 5 X.J. Fan, F.Q. Zhong, G. Yu, et al. Catalytic Cracking and Heat Sink Capacity of Aviation Kerosene Under Supercritical Conditions. *Journal of Propulsion and Power*, Vol. 25, No. 6, 2009, pp. 1226-1232.
- 6 Powell, O.A., Edwards, J.T., Norries, R.B., Numbers, K.E. and Pearce, J.A., "Development of Hydrocarbon-Fueled Scramjet Engines: The Hypersonic Technology (HyTech) Program," *Journal of Propulsion and Power*, Vol. 17, No. 6, 2001, pp. 1170-1176.
- 7 L.-L. Wang, Chen, Z., Hua, H. Meng., Numerical study of conjugate heat transfer of cryogenic methane in rectangular engine cooling channels at supercritical pressures. *Applied Thermal Engineering* 54 (2013) 237-246.
- 8 D.L. Linne, M.L. Meyer, T. Edwards, D.A. Eitman, Evaluation of Heat Transfer and Thermal Stability of Supercritical JP-7 Fuel, NASA Technical Memo-randum 107485, AIAA-97-3041.
- 9 F. Zhong, X. Fan, G. Yu, J. Li, C. J. Sung, Heat Transfer of Aviation Kerosene at Supercritical Conditions, *Journal of Thermal Physics and Heat Transfer*, Vol. 23, No. 3, 2009, pp. 543-550.
- 10 Ward, T. A., Ervin, J. S., Striebich, R. C., and Zabarnick, S., "Simulations of Flowing Mildly Cracked Normal Alkanes Incorporating Proportional Product Distributions," *Journal of Propulsion and Power*, Vol. 20, No. 3, 2004, pp. 394-402.
- 11 Zhong F Q, Fan X J, Yu G, et al. Thermal cracking of aviation kerosene for scramjet applications. *Sci China Ser E-Tech Sci*, 2009, 52(9): 2644-2652.
- 12 W. Bao, X. L. Li, et al. Efficient utilization of heat sink of hydrocarbon fuel for regeneratively cooled scramjet. *Applied Thermal Engineering* 33-34 (2012) 208-218.
- 13 Rongpei Jiang, Guozhu Liu, Xiangwen Zhang. Thermal Cracking of Hydrocarbon Aviation Fuels in Regenerative Cooling Microchannels. *Energy & Fuels*, 2013, 27 (5), pp 2563-2577.
- 14 Fan, X. J., Yu, G., Li, J. G., Zhang, X. Y., and Sung, C. J., Investigation of Vaporized Kerosene Injection and Combustion in a Supersonic Model Combustor, *Journal of Propulsion and Power*, Vol. 22, No. 1, 2006, pp. 103-110.
- 15 Fan, X., Yu, G., Li, J. G., Lu, X. N., Zhang, X. Y., and Sung, C. J., Combustion and Ignition of Thermally Cracked Kerosene in Supersonic Model Combustors, *Journal of Propulsion and Power*, Vol. 23, No. 2, 2007, pp. 317-324.
- 16 M. Traci R, L. Farr J, Jr., et al., "A Thermal Management Systems Model for the NASA GTX RBCC Concept," NASA/CR-2002-211587, 2002.

- 17 Gamble E J, Gutierrez J, Giet D, et al., "Development of a Scramjet/Ramjet Heat Exchanger Analysis Tool," AIAA 2008-4614, 2008.
- 18 Bouchez M, Dufour E, Daniau E., "Semi-empirical and CFD Analysis of Actively Cooled Dual-mode Ramjets:2006 status," AIAA 2006-8073, 2006.
- 19 Long Li, Xuejun Fan, et al., "Measurements of Wall Heat Flux and Temperature in a supersonic Model Combustor," AIAA-249406-323, 2011.
- 20 Li L,Wang J,Fan XJ. Development Of Integrated High Temperature Sensor For Simultaneous Measurement Of Wall Heat Flux And Temperature[J].Review Of Scientific Instruments,2012,83(7):074901.
- 21 Lu Yang., Li Long., Fan, X. J. A Thermal Analysis Method for Fuel-Cooled Combustor Structure Based on Heat Flux Measurement. AIAA Paper 2012-5878.
- 22 William E. Meador and Michael K. Smart. Reference Enthalpy Method Developed from Solutions of the Boundary-Layer Equations. AIAA Journal , Vol. 43, No. 1, 2005, pp. 135-139.
- 23 Ely J. F. a. H., M. L. NIST Standard Reference Database 4-NIST Thermophysical Properties of Hydrocarbon Mixtures Database: Version 3.2[M]. National Institute Of Standards, Gaithersburg, MD. 1990.
- 24 Sun, Q. M.; Mi, Z.T.; Zhang, X. W. Determination of critical properties(T_c , P_c) of endothermic hydrocarbon fuels RP-3 and simulated JP-7. J. of Fuel Chem. Technol.2006, 34(4),466-470.
- 25 Dawei Tang. Measurement report of thermal physical properties of jet fuel. Institute of Engineering Thermophysics, Chinese Academy of Sciences, 2009.
- 26 V. Gnielinski., "New equations for heat and mass transfer in turbulent pipe and channel flow," *Internatinal Chemical Engineering*, Vol. 16, No. 2, 1976, pp. 359-367.
- 27 E. Daniau, M. Bouchez, R. Bounaceur, F. Battin-Leclerc, P.-M. Marquaire, R. Fournet, "Contribution to Scramjet active cooling analysis using n-dodecane decomposition model as a generic endothermic fuel".AIAA-2003-6920.
- 28 Herbinet, O., Marquaire, P.M.. Thermal decomposition of n-dodecane: Experiments and kinetic modeling, Journal of Analytical and Applied Pyrolysis, Vol. 78, No. 2, 2007, pp.419-429.
- 29 Earl A. Thornton., "A Finite Element Program for Steady-state Thermal Analysis of Convectively Cooled Strutures," NASA/CR-145069,1976.