Two Coupled Analysis Strategies for Melt-Ablation Modeling of Thermal Protection Material in Supersonic Gas-Particle Two-Phase Impingement Flow

Chong-You Lai * and Tzong-Shyng Leu *

Department of Aeronautics and Astronautics, National Cheng Kung University, No. 1, University Road, Tainan 701, Taiwan

* Correspondence: switch3624@gmail.com (C.-Y.L.); tsleu@mail.ncku.edu.tw (T.-S.L.)

Abstract: In this study, two analysis strategies were used to investigate the melt-ablation process of the copper specimen plate in the scaled ducted launcher. The reliability of the simulation results of the two analysis strategies was confirmed by comparing the two-way fluid-thermal-ablation coupled analysis (two-way FTA-CA) strategy with two-way fluid-thermal-ablation loosely coupled analysis (two-way FTA-LCA) strategy. Then, the accuracy of the FTA-LCA strategy was validated by comparing the simulation results of the FTA-LCA strategy and experimental results from the experimental scale test. Finally, the FTA-LCA strategy used in this study can not only estimate the impingement surface heat flux into the copper plate when the inverse heat conduction analysis (IHCA) is not possible, but also provide the analysis solution accuracy and the considerable gain in computational efficiency for predicting the long-duration ablation problem in supersonic gas-particle two-phase exhaust plume impingement flow.

Keywords: two-way fluid-thermal-ablation coupled analysis strategy; two-way fluid-thermal-ablation loosely coupled analysis strategy; gas-particle two-phase flow; supersonic exhaust plume impingement flow; melt-ablation; solid propellant rocket; thermal protection material

1. Introduction

In the past, the most common, affordable, and reliable way to perform a test which could partially simulate the severe hyperthermal environment of a rocket motor was based on the use of an oxy-acetylene torch (OAT) [1,2]. This was due to the combustion products of an oxy-acetylene torch more closely resembling the environment generated in rocket motors, but this test method is more applicable to screening materials. In order to correctly study these insulation materials in an environment nearly identical to that one present in a full-scale solid rocket motor (SRM) in terms of temperature, pressure, convective, and radiative heat transfer, chemical species of the combustion products, viscous shear, and alumina particle impact. Martin [3,4] developed the laboratory-scale SRM to observe the ablation behavior of the test material. The degradation behavior of the ablative material samples exposed to SRM environment, i.e., polybenzimidizole/nitrile butyl rubber and carbon fiber/ethylene propylene diene monomer, can be observed and recorded by real-time X-ray radiography (X-ray RTR). At the same time, the in-depth thermal response of the ablative material samples can be monitored by embedding multiple micro-thermocouples at different depths within the samples to produce temperature histories related to the given subsurface locations. The heat fluxes into the surface of the ablative material samples were determined from the measured transient-temperature profiles by IHCA. Guan et al. [5] designed a novel multifunctional deposition experiment engine (MDEE) that allows the use of a real-time X-ray radiography system to obtain images of
deposits evolution images and a high-temperature-resistant plug calorimeter to measure deposition heat flux. Guan’s team recorded the chamber pressure histories, the temperature histories within the plug, and the evolution images of deposits, through these experimental data, to understand the physical and chemical process involved in the insulator ablation of solid rocket motor (SRM) under slag deposition condition. After obtaining the temperature histories at different depths within the plug calorimeter, Guan and his coworkers mathematically calculated the total heat flux using Beck’s non-linear estimation method for inverse heat conduction problem (IHCP). Additionally, the IHCP-estimated heat flux in the test section is the sum of three parts consisting of convection, thermal radiation, and droplets heat increment. By combining the IHCP-estimated heat flux history with the evolution images of deposits, it can be observed that the gas-phase convective heat flux and the discrete-phase thermal radiation flux are the main contributors to the total heat flux when the deposition phenomenon is not yet formed (i.e., the initial operation period of MDEE). After the deposition phenomenon is formed, the total heat flux is mainly provided by thermal increment of deposits.

Generally, the measuring instruments used for measure heat flux are made of copper and graphite [1–5]. This is because the calorimeter must continually store all of the energy it absorbs throughout the duration of the test in order for the total heat flux to be accurately measured. Martin’s team [3,4] and Guan’s team [5] used graphite as the material for the heat flux gauge due to its thermal stability and erosion resistance. Graphite slabs from both the short- and long-duration subscale SRM test firings did not exhibit significant degradation or erosion, which means that phase-change and oxidation phenomena occurring at the surface of the graphite could be neglected [1,3]. However, several studies pointed out that graphite is damaged in the SRM environment [6,7], which may lead to the inability of the graphite heat flux gauge to estimate the heat flux into the graphite surface by IHCA. A similar thing happened in Lin’s team [8]. From their study results, it is known that the weight of copper specimen plate changes with time during SMR firing, which may be caused by mechanical erosion and phase-change. This made it impossible for the Lin’s team to calculate the heat flux through the material surface by solving the IHCP. For this reason, this study attempted to simulate the ablation behavior of a copper plate in the scaled ducted launcher by coupling the melt-ablation model with the flow analysis model of the small-scale test device of Lin’s team [8] when the IHCA is not possible. In this way, the impingement surface heat flux of copper plate is estimated by simulation.

So far, the small solid rocket motor devices were developed by many research groups [3–5], which successfully observed the ablation process and evaluated the insulating capability of materials through thermocouple-measured temperature history data and real-time X-ray radiography of test material. However, developing mathematical models to describe the physicochemical properties of these test materials is very difficult. Therefore, the authors wanted to develop a supersonic exhaust plume jet impingement flow model and couple it with the physicochemical mechanism model of the thermal protection material to simulate the ablation process of the thermal protection material. The accuracy of the mathematical model is confirmed by comparing the experimental measurement results with the simulation results.

In order to understand the interaction between the solid rocket motor exhaust gas field and thermal protection materials, many researchers studied the complex physical and chemical mechanism of thermal protection materials, and carried out numerical simulation studies coupled with the rocket motor exhaust gas field. These numerical simulation models are used to predict of surface recession depth, rocket motor performance, and the service life of thermal protection materials, etc. Ko [9] used a discrete phase model to simulate the supersonic high-temperature two-phase plume impingement flow field of solid rocket and discussed the effect of chemical reaction and radiation effect of the plume on the impingement surface heat flux distribution. His results showed that the radiation effect was not significant, but the chemical reaction effect led to change in the impingement surface heat flux distribution. Li and Xiang [10] used the NASA Chemical
Equilibrium with Applications (CEA) program to calculate the equilibrium composition of AP/HTPB/Al composite propellant in the chamber. They then solved 2D Navier–Stokes equations with additional terms describing turbulence, chemical kinetics, radiation, and gas-particle (Al2O3) interaction for numerical simulations of the solid rocket motor exhaust plume at 10 km altitude. Their results showed that the exhaust gases generated by the combustion of the composite propellant primarily consist of HCl, CO, and H2. Because of the high temperature and the shear layer, chemical reactions occurred between exhaust and atmospheric species that affect the flow field. These studies’ results indicate that the chemical reaction (i.e., afterburning reaction) mechanism of the plume changes the flow field structure.

Li et al. [11] developed an axisymmetric 2D numerical model to simulate an aluminized solid rocket motor (SRM) flow field. The combustion temperature and the gas/particle composition in their study were calculated using thermodynamic software CEA, and the particle size distribution injected into the solid rocket motor flow field was based on the experimentally measured particle size distribution. The comparison of their numerical and experimental thrust coefficients were within ±5% relative error in their study. York et al. [12] simulated the three-dimensional ducted launcher system flow fields and considered the effect of afterburning chemistry and particulate non-equilibrium. They regarded Al2O3 as non-equilibrated particulates which interacted with the gas-phase via interphase drag and heat transfer source terms. Additionally, they assumed that each particle came to rest as it struck the plate, while all of the kinetic energy was converted to heat energy. In their study, the ablation process of the HAVEG 41N ablation material plate was simulated by one dimensional thermochemical ablation program ABLATE. Their predicted ablation depth profile showed similar trends to the experimental data. Both Li’s team and York’s team used the experimentally measured particle size distribution as the injection condition for the simulated flow field, and their simulations were similar to the experimental results. This proved the validity of the method.

Many study groups developed multiphysics coupled models to simulate the mechanical erosion, thermal ablation, chemical corrosion, and other behaviors of thermal protection materials in solid rocket motor environment or hybrid rocket motor environment [12–14]. Some coupled models were also used to simulate the thermal ablation and chemical ablation behavior of thermal protection materials in hypersonic environment [15]. Xu et al. [14] constructed the insulation thermal decomposition model, thermochemical ablation model and the mechanical denudation model to simulate the thermal decomposition, thermochemical ablation, and mechanical erosion process of EPDM. Zhang et al. [13] developed a coupled thermal–mechanical–chemical model that considered wall regression caused by non-homogeneous chemical reactions at the wall, convective heat transfer, fluid–structure interaction, and mechanical erosion. The results showed that both chemical ablation and mechanical erosion dominated the wall regression rate at the downstream of the converging section and upstream of the throat while the other location was only dominated by chemical ablation. Meng et al. [15] coupled the gas and solid regions at the surface by appropriate energy and mass balance, and implemented the mesh movement algorithm to achieve surface recession. They implemented a mesh movement approach to simulate the recession behavior of thermal protection materials in order to predict the erosion/ablation depth profile of the material surface. These studies considered the surface recession behavior of the material and obtained good research results.

From some studies on rocket motor throat [7,12,13], it can be found that the particles were injected from the nozzle inlet or propellant surface and moved rapidly toward the nozzle exit or downstream. The residence time in the computational domain was not long. The number of tracked particles in the computational domain was not high enough to affect the total computation time. However, when simulating the experimental device of Lin and his coworkers [8], the author found that the particle residence time in the computational domain was very long. The increase in the number of tracked particles made our computational cost increase. This was coupled with the splashing behavior of liquid
particles when they hit the impingement surface [16], and partially or fully melting behavior of solid particles when they hit the impingement surface [17]. This led to a sharp increase in the number of tracked particles in the calculation domain, which increased the computational cost.

For this reason, many research teams used the loosely coupled analysis strategies to reduce computational costs while maintaining a certain accuracy of calculation results. Sun et al. [18] simulated the ablation behavior of carbon/carbon composites in a hypersonic environment through a fluid-thermal-ablation loosely coupled analysis strategy. They argued that the loosely coupled analysis strategy can avoid detailed transient aerodynamic flow analysis and, thus, reduce a large amount of computational effort, and at the same time, they also argued that the fluid-thermal-ablation coupling model can be used to provide some reference for the design of thermal protection systems. Zhang et al. [19] proposed a loosely coupled analysis strategy with the adaptive coupling time stepsize to effectively predict the conjugate heat transfer (CHT) problem in hypersonic flows. They proposed the adaptive coupling time stepsize approach based on a proportional-integral-derivative (PID) controller and embedded into the loosely coupled strategy to further improve efficiency.

In view of the study results of these researchers, the authors wanted to develop a supersonic exhaust plume jet impingement flow model and couple it with the physicochemical mechanism model of the thermal protection material to simulate the ablation process of the thermal protection material. The accuracy of the mathematical model was confirmed by comparing the experimental measurement results with the simulation results. The primary objective of the present study was to reduce the computational cost through FTA-LCA strategy and to obtain a sufficiently reliable predicted ablation depth profile. The simulation results of the FTA-LCA strategy will be compared with the experimental data from the small-scale test device of Lin [8] to confirm the accuracy of the FTA-LCA strategy. Additionally, in order to avoid the inaccurate predicted ablation depth profile due to the excessive variation of impingement surface heat flux, the author coupled the discrete phase model and the transient heat transfer analysis model in the first step of the FTA-LCA strategy to obtain the impingement surface heat flux distribution at the steady state/quasi-steady state of the flow field.

2. Numerical Investigation

To explore the relationship between the supersonic exhaust plume jet impingement flow and the thermal protection material, two analysis strategies were used to simulate the melt-ablation behavior of the copper plate in the scaled ducted launcher. The geometry of the scaled ducted launcher simulated in this study was based on a small-scale test device from the experiment of Lin [8]. Figure 1a shows a scaled ducted launcher for ablative testing in Lin’s team experiment, where the red boxed region is the simulation zone for this study. Figure 1b shows the geometry of the scaled ducted launcher from the Lin’s experiment, where the gray and bronze parts are the fluid zone and the solid zone, respectively. The geometric parameters of the solid propellant rocket motor nozzle in the fluid zone are shown in Table 1.

Two analysis strategies used in this study were two-way FTA-CA and two-way FTA-LCA strategy. The former uses the discrete phase model to simulate the two-phase gas-particle exhaust plume impingement flow field to obtain the heat flux distribution of the impingement surface, and then obtains the ablation depth profile of the copper impinging plate by using the Stefan condition. In the latter, after obtaining the heat flux distribution of the impingement surface, the solid zone is simulated by the “solidification/melting model” to obtain the ablation depth profile of the copper impinging plate. To verify the accuracy of the simulation results of the two analysis strategies, these simulation results were compared with the experimental results of Lin [8].
Figure 1. (a) Schematic diagram of the scaled ducted launcher in the experiment of Lin et al. [8], where \( r \) is radial direction coordinate from the center of the copper specimen plate, \( R_N \) is the supersonic nozzle exit radius (16.25 mm); (b) Geometry of the scaled ducted launcher.

Table 1. The geometric parameters of the supersonic nozzle.

<table>
<thead>
<tr>
<th>Nozzle Geometry</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Throat radius (mm)</td>
<td>6.61</td>
</tr>
<tr>
<td>Nozzle exit radius (mm)</td>
<td>16.25</td>
</tr>
<tr>
<td>Length from the throat to the exit (mm)</td>
<td>53.32</td>
</tr>
<tr>
<td>Divergence half-angle</td>
<td>11°</td>
</tr>
</tbody>
</table>

2.1. Gas-Particle Two-Phase Exhaust Plume Flow

To simulate the supersonic gas-particle two-phase exhaust plume flow field of a solid propellant rocket, the following assumptions were made:

1. In the discrete phase model, the two-way coupling between the gas phase and the discrete phase (i.e., particle phase) is achieved through the interaction between the gas phase and the particle phase (such as drag and heat exchange);
2. The interaction between the particles is ignored;
3. The phase change of the particle phase is ignored;
4. The gaseous substances of plume is compressible ideal gas;
5. The particle phase is in thermal equilibrium with surrounding gas phase at the nozzle inlet;
6. The chemical reaction between the particle phase and the gas phase is not considered;
7. The thermal radiation effect is ignored.

2.2. Governing Equations

The scaled ducted launcher for ablative testing simulated in this study was divided into a fluid zone and a solid zone. The Euler–Lagrange model (i.e., discrete phase model) was used in the fluid zone to simulate the supersonic gas-particle two-phase exhaust plume flow field of the solid propellant rocket. The exhaust plume of solid propellant rocket motor consisted of the gaseous combustion products and the condensed combustion products (CCPs). In the discrete phase model, the gaseous combustion products were treated as the continuous phase (i.e., Eulerian gas phase) and the condensed combustion products were treated as the discrete phase (i.e., Lagrangian particle phase). The gas phase can exchange mass, momentum, and energy with the particle phase. Sections 2.2.1, 2.2.2, 2.2.3, 2.2.5, and 2.2.6 are the governing equations describing the physical behavior of the Eulerian gas phase. Sections 2.2.7 and 2.2.8 are the governing equations describing the physical behavior of the Lagrangian particle phase. When the discrete phase is deposited on the impingement surface to form a liquid film, the physical behavior of the liquid film is described by the conservation of mass, momentum and energy for liquid film in wall-film theory (see Sections 2.3.1, 2.3.2, and 2.3.3).

Sections 2.4 and 2.5 introduce two numerical models that are used in the solid zone, namely, the transient heat transfer analysis model and the solidification/melting model. The transient heat transfer analysis model is divided into the transient fluid-solid conjugate heat transfer model (see Sections 2.4.1 and 2.2.4) and transient heat transfer model (see Section 2.4.2). The former was coupled with the discrete phase model in the first step of the FTA-LCA strategy to obtain the impingement surface heat flux distribution at the steady state/quasi-steady state of the flow field. The latter was used for the FTA-CA strategy to obtain the temperature distribution within the copper plate and the melt-ablation depth profile of the copper plate. Additionally, the solidification/melting model was used in FTA-LCA strategy to obtain the temperature distribution within the copper plate and the melt-ablation depth profile of the copper plate.

The above equations will be introduced one by one. More detailed variables and parameters for the momentum equation, shear-stress transport $k-\omega$ turbulence equation, transport equation, and energy equation can be found in the ANSYS FLUENT 19 Theory Guide.

2.2.1. Continuity Equation

\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0 \]  

(1)

where $t$ denotes time, $x_i$ denotes the spatial coordinates, $\rho$ denotes the density of the continuous phase, $u_i$ means the direction of the time averaged velocity.

Two-way coupling was assumed in the present study. In other words, discrete phase (i.e, particle phase) was affected by continuous phase in momentum and energy transfer. In order to account for such coupling, the standard RANS (Reynolds Averaged Navier–Stokes) equations [20,21] and energy equation [21] must be modified by including the momentum source term and the energy source term, respectively. The momentum equations and energy equation including the source term for two-way coupling are as follows:
2.2.2. Momentum Equation

\[
\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu_{\text{eff}} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j} \left( -\frac{2}{3} \rho \kappa \delta_{ij} \right) + \rho g_i + S_{M,i}
\]

(2)

In the momentum equation [21], the index \( i, j \) and \( k \) indicate the Cartesian components, \( g_i \) means the direction of gravity (when the effect of gravity is not considered, the term \( \rho g_i \) can be ignored), \( \delta_{ij} \) means three order identity tensor (\( \delta_{ij} \) is the “Kronecker delta” symbol), \( \kappa \) is turbulence kinetic energy, \( S_{M,i} \) means the \( i \) component of the momentum exchange term with the discrete phase and represents the effect of discrete phase on the carrier phase (i.e., continuous phase) flow equations.

\[
\mu_{\text{eff}} = \mu + \mu_t
\]

(3)

where \( \mu_{\text{eff}} \) represents the effective turbulent viscosity, \( \mu \) denotes the laminar viscosity, \( \mu_t \) denotes the turbulent viscosity.

2.2.3. Energy Equation in Fluid Regions

\[
\frac{\partial (\rho E)}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho u_i (E + p) \right) = \frac{\partial}{\partial x_i} \left( k_{\text{eff}} \frac{\partial T}{\partial x_i} - \sum_{l=1}^{N} h_l J_{i,l} + u_i \tau_{eff} \right) + S_h + S_E
\]

(4)

In the energy equation [21], \( p \) represents the pressure, \( E \) represents the total energy, \( J_{i,l} \) is the diffusion flux of species \( l \), \( \tau_{eff} \) represents the stress tensor under the influence of turbulence with effective viscosity, \( k_{\text{eff}} \) represents the effective thermal conductivity \( (k_{\text{eff}} = k_{f,\text{cond}} + k_{f,\text{turbulent cond}}) \), \( k_{f,\text{cond}} \) denotes the thermal conductivity. The turbulent thermal conductivity \( (k_{f,\text{turbulent cond}}) \) is defined as \( k_{f,\text{turbulent cond}} = C_p \mu_t / \Pr_t \), where \( C_p \) is heat capacity for continuous phase, \( \Pr_t \) is turbulent Prandtl number, \( S_E \) means the energy exchange term with the discrete phase, \( S_h \) means the energy source due to chemical reaction.

\[
E = h - \frac{p}{\rho} + \frac{|u_i|^2}{2}, \quad h = \sum_{l=1}^{N} Y_l h_l, \quad h_l = \int_{T_{\text{ref},l}}^{T} C_{p,l} dT, \quad T_{\text{ref},l} = 298.15 \, K.
\]

(5)

Here, \( h \) is sensible enthalpy for ideal gases, \( Y_l \) is the mass fraction of species \( l \).

2.2.4. Energy Equation in Solid Regions

\[
\frac{\partial (\rho_s h)}{\partial t} + \frac{\partial}{\partial x_i} \left( u_{i,s} (\rho_s h) \right) = \frac{\partial}{\partial x_i} \left( k_{\text{cond,s}} \frac{\partial T}{\partial x_i} \right) + S_v
\]

(6)

\[
h = \int_{T_{\text{ref}}}^{T} C_{p,s} dT, \quad T_{\text{ref}} = 298.15 \, K
\]

(7)

where \( \rho_s \) is density of thermal protection material, \( h \) is sensible enthalpy, \( k_{\text{cond,s}} \) is thermal conductivity of thermal protection material, \( T \) is temperature, \( S_v \) is volumetric heat source. The velocity field \( u_{i,s} \) is computed from the motion specified for the solid zone.
2.2.5. Shear-Stress Transport (SST) $k - \omega$ Model

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho ku_i) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + \bar{G}_k - Y_k
\]

(8)

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho \omega u_i) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega
\]

(9)

In shear-stress transport model [21,22], $k$ represents the turbulence kinetic energy, $\omega$ represents the specific dissipation rate, $\bar{G}_k$ represents the generation of turbulence kinetic energy due to mean velocity gradients, $G_\omega$ represents the generation of $\omega$, $Y_k$ and $Y_\omega$ represent the dissipation of $k$ and $\omega$ due to turbulence, $D_\omega$ represents the cross-diffusion term. $\Gamma_k$ and $\Gamma_\omega$ represent the effective diffusivity of $k$ and $\omega$, respectively.

\[
\mu_i = \frac{b k}{\omega} \max \frac{1}{\alpha \tau_s} \frac{SD_\tau}{\mu_i \sigma_s}
\]

(10)

\[
\sigma_k = \frac{1}{F_1/\sigma_{k,1} + (1 - F_1)/\sigma_{k,2}}, \quad \sigma_\omega = \frac{1}{F_1/\sigma_{\omega,1} + (1 - F_1)/\sigma_{\omega,2}}
\]

(11)

$S$ is the strain rate magnitude, $\alpha^*$ is damps the turbulent viscosity causing a low Reynolds-number correction, $F_1$ and $F_2$ are the blending functions.

2.2.6. Species Transport Equation

\[
\frac{\partial}{\partial t} (\rho Y_i) + \frac{\partial}{\partial x_j} (\rho u_j Y_i) = -\frac{\partial}{\partial x_j} (J_i) + R_i + S_i
\]

(12)

In species transport equation [21], $Y_i$ is mass fraction of species $I$, $R_i$ is the net rate of production of species $I$ by chemical reaction, $S_i$ is the rate of creation by addition from the dispersed phase plus any user-defined sources.

\[
J_i = -(\rho D_i + \frac{\mu_t}{S_{c_i}} \frac{\partial Y_i}{\partial x_j})
\]

(13)

where $D_i$ is the mass diffusion coefficient for species $I$ in the mixture, $S_{c_i}$ is the turbulent Schmidt number ($S_{c_i} = \mu_t / \rho D_i$ where $\mu_t$ is the turbulent viscosity and $D_i$ is the turbulent diffusivity).

2.2.7. Particle Trajectory Equation

In Lagrangian reference frame, the trajectory is predicted by integrating the force balance on it while the particle phase is treated as a discrete phase. This force balance equates the particle inertia with the forces acting on the particle described as below [20,21,23–25]:

\[
\frac{d u_{p,i}}{dt} = F_{bj}(u_i - u_{p,i}) + g_i \left( \frac{\rho_i - \rho}{\rho_i} \right) + F_{other}
\]

(14)

where $u_{p,i}$ refers to particle velocity. The first term on the right hand side of the particle trajectory equation indicates the drag force on the particles. The second term represents the gravitational force acting on the particle due to its mass. The third term is the other forces ($F_{other}$) that are considered like Saffman lift force, Brownian force, force due to thermophoresis effect, force due to lift motion, force due to virtual mass and force due to pressure gradient per unit mass [21,26,27].
Here, $F_D$ for spherical particles can be written in the following form [25]:

$$F_D = \frac{18 \mu}{\rho_p d_p^2} \frac{C_D \text{Re}_p}{24}$$

(15)

where $d_p$ and $\rho_p$ are the particle diameter and particle density, respectively.

The relative Reynolds number $\text{Re}_p$ is defined as [25]:

$$\text{Re}_p = \frac{\rho d_p |u_{p,i} - u_i|}{\mu}$$

(16)

The drag coefficient is calculated as follows:

$$C_D = c_1 + \frac{c_2}{\text{Re}_p} + \frac{c_3}{\text{Re}_p^2}$$

$$C_D = 24.0 / \text{Re}_p \quad \text{for} \quad \text{Re}_p < 0.1$$

$$C_D = 3.69 + 22.73 / \text{Re}_p + 0.0903 / \text{Re}_p^2 \quad \text{for} \quad 0.1 < \text{Re}_p < 1.0$$

$$C_D = 1.222 + 29.1667 / \text{Re}_p - 3.8889 / \text{Re}_p^2 \quad \text{for} \quad 1.0 < \text{Re}_p < 10.0$$

$$C_D = 0.6167 + 46.5 / \text{Re}_p - 116.67 / \text{Re}_p^2 \quad \text{for} \quad 10.0 < \text{Re}_p < 100.0$$

$$C_D = 0.3644 + 98.33 / \text{Re}_p - 2778 / \text{Re}_p^2 \quad \text{for} \quad 100.0 < \text{Re}_p < 1000.0$$

$$C_D = 0.357 + 148.62 / \text{Re}_p - 47500 / \text{Re}_p^2 \quad \text{for} \quad 1000.0 < \text{Re}_p < 5000.0$$

$$C_D = 0.46 - 490.546 / \text{Re}_p + 57870 / \text{Re}_p^2 \quad \text{for} \quad 5000.0 < \text{Re}_p < 10000.0$$

$$C_D = 0.5191 - 1662.5 / \text{Re}_p + 5416700 / \text{Re}_p^2 \quad \text{for} \quad 10000.0 < \text{Re}_p < 50000.0$$

where $c_1$, $c_2$ and $c_3$ are constants that apply to smooth spherical particles [21,28].

2.2.8. Particle Energy Equation

$$m_p c_p \frac{dT_p}{dt} = h_p A_p (T - T_p)$$

(18)

where $m_p$, $c_p$, $A_p$, $T$ and $T_p$ are the mass of the particle, the heat capacity for discrete phase, the particle surface area, local temperature of the continuous phase and particle temperature, respectively.

The heat transfer coefficient $h_p$ is given by Ranz and Marshall correlation [29]:

$$Nu_p = \frac{h_p d_p}{k_{f,cond}} = 2.0 + 0.6 \frac{\text{Re}_p^{1/2}}{\text{Pr}^{1/3}}$$

(19)

where $k_{f,cond}$ denotes thermal conductivity of the continuous phase, $\text{Pr}$ denotes the Prandtl number of the continuous phase, $\text{Re}_p$ denotes particle Reynolds number based on the particle diameter and the relative velocity.

The momentum exchange term and energy exchange term are calculated as follows [29]:

$$F_i = \sum \left( \frac{18 \mu}{\rho_p d_p^2} \frac{C_i \text{Re}_p}{24} (u_{p,i} - u_i) + \frac{g(\rho - \rho_p)}{\rho_p} \right) m_p \Delta t$$

(20)

$$Q = \left[ \frac{\bar{m}_p}{m_{p,0}} c_p \Delta T_p \right] m_{p,0}$$

(21)
where $F$ is the momentum transfer from the continuous phase to the discrete phase, $Q$ is the heat transfer from the continuous phase to the discrete phase, $\dot{m}_p$ is mass flow rate of the particles, $\Delta t$ is time step, $m_{p0}$ is average mass of the particle in the control volume, $m_{p0}$ is initial mass of the particle, $c_p$ is heat capacity of the particle, $\Delta T_p$ is temperature change of the particle in the control volume, $\dot{m}_{p0}$ is initial mass flow rate of the particle injection tracked, $np$ is number of particles within a cell volume, $\delta V$ is cell volume.

2.3. Conservation Equations for Wall-Film

Wall film mass variation occurs due to splash and rebound when the incoming droplets impinge on the wall. Therefore, the wall film theory is used to describe the behavior of the wall film [16,21,30,31].

2.3.1. Wall-Film Mass Equation

$$\frac{\partial \rho}{\partial t} + \nabla_s \left[ \rho \left( \vec{u}_p - \vec{u}_w \right) h_{\text{film}} \right] = \dot{M}_{\text{imp,at}}$$  \hspace{1cm} (24)

In wall-film mass equation [31], $\rho$ is liquid density (assumed constant), $h_{\text{film}}$ is the film height, $\vec{u}_p$ and $\vec{u}_w$ are mean liquid velocity and wall velocity, respectively. $\nabla_s$ is the gradient operator restricted to the surface, $\dot{M}_{\text{imp,at}}$ is the impingement mass source.

2.3.2. Wall-Film Momentum Equation

$$\rho \left( \frac{d}{dt} \vec{u}_p \right) + h_{\text{film}} \left( \nabla_s p_f \right)_{\alpha} = \tau_g \vec{t}_g + \tau_w \vec{t}_w + \dot{P}_{\text{imp,at}} - \dot{M}_{\text{imp,at}} \vec{u}_p + \dot{F}_{u,a} + gh_{\text{film}}$$  \hspace{1cm} (25)

In the wall-film momentum equation [16,21,30], $\alpha$ denotes the current face on which the particle resides, $h_{\text{film}}$ is the current film height at the particle location and $gh \left( \vec{g} - \vec{a}_w \right)$ is the body force term. $p_f$ is the pressure on the surface of the film, given by [21,30].

$$p_f = P_{\text{cell}} - \dot{P}_{\text{imp,at}} \cdot \vec{n} + M_{\text{imp,at}} \vec{u}_p \cdot \vec{n}$$  \hspace{1cm} (26)

On the right side of wall-film momentum equation, $\tau_g$ denotes the magnitude of the shear stress of the gas flow on the surface of the film, $\tau_w$ denotes the magnitude of the stress that the wall exerts on the film, $\vec{t}_g$ is the unit vector in the direction of the relative motion of the gas and the film surface, $\vec{t}_w$ is the unit vector in the direction of the relative motion of the film and the wall, $\mu_l$ is the liquid viscosity, $\dot{P}_{\text{imp,at}}$ denotes the impingement pressure on the film, $\dot{F}_{u,a}$ is the force necessary to keep the film on
the surface as determined by \( \mathbf{u}_p \cdot \mathbf{n}_\alpha = 0 \), where \( \mathbf{n}_\alpha \) is the unit normal to the wall surface on face \( \alpha \).

### 2.3.3. Wall-Film Energy Equation

To obtain an equation for the temperature in the film, energy flux from the gas side as well as energy flux from the wall side must be considered. The assumed temperature profile in the liquid is bilinear, with the surface temperature \( T_s \) being the maximum gas temperature \( T_g \) at the film height. The temperature for the film surface is equal to the gas temperature \( T_g \), but is limited by the boiling temperature of the liquid. Furthermore, the boiling point of the liquid and the wall temperature will be the maximum of the wall face temperature \( T_w \), and will be the same boiling temperature as the liquid.

An energy balance on a film particle yields [21,30].

\[
\frac{d}{dt} \left( m_p c_p T_p \right) = Q_{\text{cond}} + Q_{\text{conv}}
\]

(27)

where \( Q_{\text{cond}} \) is the conduction from the wall, given by [21,30]

\[
Q_{\text{cond}} = \frac{k_f}{h_{\text{film}}} A_p (T_w - T_p)
\]

(28)

where \( k_f \) is the thermal conductivity of the liquid and \( h_{\text{film}} \) is the film height at the location of the particle.

The convection from the top surface \( Q_{\text{conv}} \) is given by [21,30]

\[
Q_{\text{conv}} = h_f A_p (T_g - T_p)
\]

(29)

\[
Nu_x = \frac{h_f x}{k_f} = \begin{cases} 0.332 \operatorname{Re}_x^{1/2} \operatorname{Sc}^{1/3} & \text{Re}_x < 2500, 0.6 < \operatorname{Sc} < 50 \\ 0.0296 \operatorname{Re}_x^{4/5} \operatorname{Sc}^{1/3} & \text{Re}_x > 2500, 0.6 < \operatorname{Sc} < 60 \end{cases}
\]

(30)

where \( h_f \) is the film heat transfer coefficient given by equation (30), \( k_f \) is the thermal conductivity of film, \( A_p \) is the area represented by a film particle, \( x \) denotes a representative length derived from the face area, \( \operatorname{Re}_x \), \( \operatorname{Sc} \) and \( Nu_x \) are Reynolds number, Schmidt number and Nusselt number, respectively.

### 2.4. Transient Heat Transfer Analysis Model

The transient heat transfer analysis model was divided into two heat transfer model, as described below.

#### 2.4.1. Transient Fluid-Solid Conjugate Heat Transfer Model

The transient fluid-solid conjugate heat transfer model was solved by solving Equation (6) to solve for the temperature distribution within the solid region. It should be noted that the user-defined function (UDF) code is used to control the maximum temperature of the fluid-solid conjugate interface at the surface of the solid-side, which is the melting point of the material in the solid region.

#### 2.4.2. Transient Heat Transfer Model

The transient heat transfer model uses the explicit finite difference method to discretize the heat transfer equation in virtual solid zone to obtain the temperature distribution within impinging plate. The heat transfer governing equation used by the transient heat transfer model in the solid zone is discretized as follows:
when material is solidified material out of the domain constant where material

the conservation of momentum can be written as \[ \frac{dE}{dt} = q_{in} - q_{out} \] (31)

where \( \frac{dE}{dt} \) is rate of change of total energy in control volume, \( q_{in} \) is rate of heat transfer into control volume, \( q_{out} \) is rate of heat transfer out control volume. It should be noted that the UDF code is used to control the maximum temperature of the fluid-solid interface in transient heat transfer model, which is the melting point temperature of the material in the solid region.

The transient heat transfer model obtains surface recession rate of the fluid-solid interface by solving the Stefan condition. The surface recession behavior of the material is implemented by the mesh movement algorithm. The Stefan condition as follows:

\[
\frac{d\delta}{dt} = \frac{q_f - q_s}{\rho_l H_f} = \frac{q_{jump}}{\rho_l H_f} 
\] (32)

\[
d\delta = \frac{d\delta}{dt} \times \Delta t 
\] (33)

where \( \frac{d\delta}{dt} \) is interface velocity (the direction of motion of the interface is moving from the fluid side to the solid side), \( q_{jump} \) is the conductive heat flux jump across the interface, \( q_s \) is heat flux at solid side of the fluid-solid interface, \( q_f \) is heat flux at fluid side of the fluid-solid interface (i.e., \( q_f = q_w \)), \( \rho_l \) is liquid density, \( H_f \) is the latent heat (or Enthalpy of fusion), \( d\delta \) is interface movement distance, \( \Delta t \) is computational time step.

2.5. Solidification/Melting Model

For solidification/melting problem of thermal protection materials in solid regions, the conservation of momentum can be written as [21,32]:

\[
\frac{\partial}{\partial t} (\rho_s v_{i,s}) + \frac{\partial}{\partial x_j} (\rho_s v_{i,s} v_{j,s}) = \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial v_{i,s}}{\partial x_j} + \frac{\partial v_{j,s}}{\partial x_i} \right) \right] + \tilde{S}_A \] (34)

where \( \rho_s \) is density of thermal protection material (i.e., copper), \( v_{i,s} \) is velocity of liquid thermal protection material, \( p \) is pressure related to liquid thermal protection material, \( \mu \) is viscosity of liquid thermal protection material.

The source term \( \tilde{S}_A \) is defined as:

\[
\tilde{S}_A = \frac{(1 - \beta_f)^2}{(\beta_f + \epsilon)} A_{mush} (v_{i,s} - v_{pl,s}) 
\] (35)

where \( \epsilon \) is a small number (0.001) to prevent division by zero, \( A_{mush} \) is the mushy zone constant, \( \beta_f \) is liquid volume fraction, \( v_{pl,s} \) is the solid velocity due to the pulling of solidified material out of the domain.

The liquid volume fraction is zero when material is totally solid (\( \beta_f = 0 \)) and is one when material is totally liquid (\( \beta_f = 1 \)).

\[
\beta_f = \begin{cases} 
0 & \text{if } T < T_{solidus} \\
1 & \text{if } T > T_{liquidus} \\
\frac{T - T_{solidus}}{T_{liquidus} - T_{solidus}} & \text{if } T_{solidus} < T < T_{liquidus} 
\end{cases} 
\] (36)
where $T_{\text{solidus}}$ and $T_{\text{liquidus}}$ are the solidus temperature and liquidus temperature, respectively.

The energy equation solved in solidification/melting model is [21,32]:

$$
\frac{\partial (\rho_s h)}{\partial t} + \frac{\partial}{\partial x_1} \left( v_{i,s} (\rho_s h) \right) = \frac{\partial}{\partial x_1} \left( \alpha_s \frac{\partial h}{\partial x_1} \right) + \tilde{S}
$$

(37)

where $\alpha_s$ is thermal diffusivity of thermal protection material, $\alpha_s = k_{s,\text{cond}} / C_{p,s}$.

The source term $\tilde{S}$ is defined as [21,29,32,33]:

$$
\tilde{S} = \frac{\partial (\rho \Delta H)}{\partial t} + \frac{\partial}{\partial x_1} \left( v_{i,s} (\rho \Delta H) \right)
$$

(38)

The latent heat content $\Delta H$ can be calculated as:

$$
\Delta H = \beta_L L
$$

(39)

where $L$ is the latent heat of the thermal protection material.

2.6. Geometry and Computational Grid

The study results of Lin and his coworkers [8] showed that the temperature of impingement surface in the impact center area was higher than the melting point of the copper, which implies that the copper plate underwent melt-ablation. Moreover, their experimental results (see Figure 2) showed that the shape of the ablation zone of the impinging plate was close to an axisymmetric shape, which implies that the bilaterally symmetrical scaled ducted launcher had an ablation zone of approximately axisymmetric shape.

![Figure 2. The surface depth profile of impinging plate. Left: The orange geometric shape is an approximately axisymmetric shape. Right: The copper specimen plate after the ablative test by Lin et al. [8].](image)

In order to understand the effect of bilaterally symmetrical flow field and axially symmetric flow field on the heat flux distribution on the impingement surface of the impinging plate, we used three-dimensional quarter-symmetric flow field and two-dimensional axisymmetric flow field to investigate the relationship between the plume flow field and heat flux distribution on the impingement surface of an impinging plate.

The three-dimensional quarter-symmetric computational grid for the scaled ducted launcher simulation and two-dimensional axisymmetric computational grid for the scaled ducted launcher simulation are shown in Figure 3. The simulated cases using the 3D quarter-symmetric grid and the 2D axial-symmetric grid are called, respectively, 3D cases and 2D cases, for the convenience of discussion. It is worth noting that the scaled ducted launcher did not have a solid zone (i.e., virtual solid zone), as observed on the left side of Figure 3b, which uses the transient heat transfer model to calculate the temperature distribution within the copper plate and obtains the surface recession rate of the fluid–solid interface by calculating the Stefan condition. Moreover, the scaled ducted launcher had a
solid zone, as seen on the right side of Figure 3b, which uses a transient fluid-solid conjugate heat transfer model for the temperature distribution within the copper plate.

Figure 3. (a) The three-dimensional quarter-symmetric computational grid; (b) the two-dimensional axisymmetric computational grid. (Left, the computational grid for transient heat transfer model. Right, the computational grid for transient fluid-solid conjugate heat transfer model).

2.7. Boundary Conditions

The schematic diagram of the boundary conditions between 2D cases and 3D cases is shown in Figure 4, where Figure 4a is the schematic diagram of the boundary conditions for 2D case used in the FTA-CA strategy, Figure 4b is the schematic diagram of the boundary conditions for 2D case used in FTA-LCA strategy, and Figure 4c is the schematic diagram of the boundary conditions for 3D case used in FTA-LCA strategy. Figure 4b, c show that coupling the fluid zone (i.e., the discrete phase model) with the solid zone (i.e., transient fluid-solid conjugate heat transfer model) to obtain the impingement surface heat flux distribution at the steady state/quasi-steady state of the flow field.

The species mass fractions at nozzle inlet and outlet for the 2D cases and 3D cases are shown in Table 2. The boundary conditions of gas phase and particle phase for these four simulation cases are shown in Tables 3 and 4, respectively.
Figure 4. The schematic diagram of the boundary conditions for (a) 2D cases with transient heat transfer model; (b) 2D cases with transient fluid-solid conjugate heat transfer model; (c) 3D cases.

Table 2. The species mass fractions at nozzle inlet and outlet.

<table>
<thead>
<tr>
<th>Species</th>
<th>Outlet</th>
<th>Nozzle Inlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plume NRF</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>( \text{N}_2 )</td>
<td>0.78</td>
<td>0</td>
</tr>
<tr>
<td>( \text{O}_2 )</td>
<td>0.22</td>
<td>0</td>
</tr>
</tbody>
</table>

2D reactive flow case and 3D reactive flow case

<table>
<thead>
<tr>
<th>Species</th>
<th>Outlet</th>
<th>Nozzle inlet</th>
</tr>
</thead>
</table>


### Table 3. The boundary conditions for gas phase at nozzle inlet and outlet in 2D cases and 3D cases.

<table>
<thead>
<tr>
<th></th>
<th>Outlet</th>
<th>Nozzle Inlet</th>
<th>Outlet</th>
<th>Nozzle Inlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature (K)</td>
<td>300</td>
<td>3500</td>
<td>300</td>
<td>3500</td>
</tr>
<tr>
<td>Mass Flow Rate (kg/s)</td>
<td>-</td>
<td>0.4704</td>
<td>-</td>
<td>0.1176</td>
</tr>
<tr>
<td>Pressure (Pa)</td>
<td>101300</td>
<td>6374323</td>
<td>101300</td>
<td>6374323</td>
</tr>
</tbody>
</table>

**2D non-reactive flow case**

**3D non-reactive flow case**

### Table 4. The boundary conditions for particle phase at nozzle inlet and outlet in 2D cases and 3D cases.

<table>
<thead>
<tr>
<th>Particle Size (µm)</th>
<th>Number Percentage (%)</th>
<th>Mass Fraction (%)</th>
<th>Mass Flow Rate (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.25</td>
<td>2</td>
<td>0.238</td>
<td>0.000213248</td>
</tr>
<tr>
<td>2.75</td>
<td>12</td>
<td>2.602</td>
<td>0.002331392</td>
</tr>
<tr>
<td>3.5</td>
<td>52</td>
<td>23.244</td>
<td>0.02082662</td>
</tr>
<tr>
<td>5</td>
<td>27</td>
<td>35.186</td>
<td>0.03152666</td>
</tr>
<tr>
<td>7</td>
<td>5</td>
<td>17.88</td>
<td>0.01602048</td>
</tr>
<tr>
<td>10</td>
<td>2</td>
<td>20.85</td>
<td>0.0186816</td>
</tr>
<tr>
<td><strong>Total</strong></td>
<td>100%</td>
<td>100%</td>
<td>0.0896</td>
</tr>
</tbody>
</table>

**2D non-reactive flow case and 2D reactive flow case**

**3D non-reactive flow case and 3D reactive flow case**
2.8. Thermophysical Properties Copper and the Composition and Chemical Reaction Mechanism of the Plume

The plume consists of gaseous substances, coarse agglomerate residues containing oxide and non-consumed metallic aluminum, relatively coarse oxide particles formed after full burnout of agglomerates, original aluminum particles, and oxide fine smoke particles composed of nanosized spherules. In this study, two types of plume were used, which had different thermophysical properties. One was used to simulate a two-phase gas-particle non-reactive flow (i.e., plume NRF) and another was used to simulate a two-phase gas-particle reactive flow (i.e., plume RF). The chemical model for the plume for reacting flow (plume RF) involved twelve chemical species (including CO, CO₂, Cl, Cl₂, H, HCl, H₂, H₂O, N₂, O, OH, and O₂) and the seventeen chemical reactions of Arrhenius expression. The seventeen chemical reactions of the finite-rate chemistry model (Arrhenius kinetics) [21] are shown in Table 5.

Table 5. Chemical reaction model, where A is pre-exponential factor, n is temperature exponent.

<table>
<thead>
<tr>
<th>Reaction</th>
<th>A</th>
<th>n</th>
<th>E(J/kg-mol)</th>
</tr>
</thead>
<tbody>
<tr>
<td>H₂+O₂ ⇌ OH+O</td>
<td>1.915×10⁰</td>
<td>0</td>
<td>7.03×10⁷</td>
</tr>
<tr>
<td>O+H₂ ⇌ H+OH</td>
<td>5.080×10⁴</td>
<td>2.67</td>
<td>2.63×10⁷</td>
</tr>
<tr>
<td>OH+H₂ ⇌ H₂+O</td>
<td>2.160×10⁵</td>
<td>1.51</td>
<td>3.43×10⁷</td>
</tr>
<tr>
<td>OH+OH ⇌ O+H₂O</td>
<td>1.506×10⁶</td>
<td>2.02</td>
<td>1.34×10⁷</td>
</tr>
<tr>
<td>H+H+N₂ ⇌ H₂+N₂</td>
<td>4.577×10⁶</td>
<td>−1.4</td>
<td>1.04×10⁸</td>
</tr>
<tr>
<td>H+OH+N₂ ⇌ H₂O+N₂</td>
<td>1.912×10⁹</td>
<td>−1.83</td>
<td>1.18×10⁸</td>
</tr>
<tr>
<td>H+O+N₂ ⇌ OH+N₂</td>
<td>9.880×10⁹</td>
<td>−0.74</td>
<td>1.02×10⁸</td>
</tr>
<tr>
<td>O+O+N₂ ⇌ O₂+N₂</td>
<td>4.515×10⁹</td>
<td>−0.64</td>
<td>1.19×10⁸</td>
</tr>
<tr>
<td>CO+OH ⇌ CO₂+H</td>
<td>9.420×10⁶</td>
<td>2.25</td>
<td>−2.35×10⁶</td>
</tr>
<tr>
<td>CO+O₂ ⇌ CO₂+O</td>
<td>2.499×10⁶</td>
<td>0</td>
<td>4.77×10⁷</td>
</tr>
<tr>
<td>CO+O+N₂ ⇌ CO₂+N₂</td>
<td>6.170×10⁶</td>
<td>0</td>
<td>3.00×10⁸</td>
</tr>
<tr>
<td>H+HCl ⇌ H₂+Cl</td>
<td>1.692×10⁹</td>
<td>0</td>
<td>4.14×10⁸</td>
</tr>
<tr>
<td>H+Cl₂ ⇌ HCl+Cl</td>
<td>8.551×10⁹</td>
<td>0</td>
<td>1.17×10⁹</td>
</tr>
<tr>
<td>HCl+OH ⇌ H₂O+Cl</td>
<td>2.710×10⁶</td>
<td>1.65</td>
<td>−2.2×10⁸</td>
</tr>
<tr>
<td>HCl+O ⇌ OH+Cl</td>
<td>3.370×10⁹</td>
<td>2.87</td>
<td>3.51×10⁸</td>
</tr>
<tr>
<td>Cl+Cl+N₂ ⇌ Cl₂+N₂</td>
<td>4.675×10¹¹</td>
<td>0</td>
<td>−1.8×10⁸</td>
</tr>
<tr>
<td>H+Cl+N₂ ⇌ HCl+N₂</td>
<td>1.192×10⁻⁵</td>
<td>−2</td>
<td>0</td>
</tr>
</tbody>
</table>

The solid zone is the impinging plate of copper specimen which is placed above of small solid propellant rocket nozzle, while the thermophysical properties of copper are shown in Table 6. These thermophysical parameters were fed into the transient heat transfer model and the solidification/melting model for solving the temperature distribution within impinging plate and the melt-ablation process of the impinging plate. In addition, these thermophysical parameters were also fed into the transient fluid-solid conjugate heat transfer model for solving the temperature distribution within impinging plate.
Table 6. Thermophysical parameters of copper.

<table>
<thead>
<tr>
<th>Temperature (K)</th>
<th>Density (kg/m³)</th>
<th>Specific Heat (J/kg-K)</th>
<th>Thermal Conductivity (W/m-K)</th>
<th>Viscosity (kg/m·s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>298</td>
<td>8978</td>
<td>381.6</td>
<td>387</td>
<td>-</td>
</tr>
<tr>
<td>900</td>
<td>-</td>
<td>457.61</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>1000</td>
<td>-</td>
<td>468.92</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>1100</td>
<td>8719.9</td>
<td>480.22</td>
<td>336.18</td>
<td>-</td>
</tr>
<tr>
<td>1200</td>
<td>8684</td>
<td>491.53</td>
<td>328.67</td>
<td>-</td>
</tr>
<tr>
<td>1300</td>
<td>8646.1</td>
<td>502.83</td>
<td>321.16</td>
<td>-</td>
</tr>
<tr>
<td>1356</td>
<td>8623.8</td>
<td>509.16</td>
<td>316.96</td>
<td>-</td>
</tr>
<tr>
<td>1358</td>
<td>8181.3</td>
<td>633.99</td>
<td>157.18</td>
<td>0.00403</td>
</tr>
<tr>
<td>1400</td>
<td>8160.4</td>
<td>-</td>
<td>159.37</td>
<td>0.00374</td>
</tr>
<tr>
<td>1450</td>
<td>8136.7</td>
<td>640.90</td>
<td>161.86</td>
<td>0.00346</td>
</tr>
<tr>
<td>1500</td>
<td>8113.1</td>
<td>-</td>
<td>164.35</td>
<td>0.00321</td>
</tr>
<tr>
<td>1550</td>
<td>8089.7</td>
<td>-</td>
<td>166.83</td>
<td>0.003</td>
</tr>
<tr>
<td>1600</td>
<td>8066.4</td>
<td>-</td>
<td>169.32</td>
<td>0.00281</td>
</tr>
<tr>
<td>1700</td>
<td>8020.3</td>
<td>628.69</td>
<td>174.30</td>
<td>0.0025</td>
</tr>
<tr>
<td>1750</td>
<td>7997.4</td>
<td>-</td>
<td>176.79</td>
<td>0.00237</td>
</tr>
<tr>
<td>1800</td>
<td>7974.6</td>
<td>624.07</td>
<td>179.27</td>
<td>0.00225</td>
</tr>
<tr>
<td>1900</td>
<td>7929.5</td>
<td>621.4</td>
<td>184.25</td>
<td>0.00205</td>
</tr>
<tr>
<td>1950</td>
<td>7907.2</td>
<td>-</td>
<td>186.74</td>
<td>0.00196</td>
</tr>
<tr>
<td>2000</td>
<td>7884.9</td>
<td>619.01</td>
<td>189.23</td>
<td>-</td>
</tr>
</tbody>
</table>

2.9. Two-Way Fluid-Thermal-Ablation Coupled Analysis Strategy

Two-way fluid-thermal-ablation coupled analysis strategy was used to obtain the ablation depth profile of the copper specimen plate. Figure 5 shows the melt-ablation profile of the copper specimen plate at a burn time of 0.6 s. The results demonstrate copper plate surface recession due to melting, but it was not significant. This may be because this time point was still in the stage of initial melting. Therefore, the melt-ablation depth of impingement surface was not deep.

Figure 5. Simulation result of the two-way fluid-thermal-ablation coupled analysis strategy with a burn time of 0.6 s.
2.10. Two-Way Fluid-Thermal-Ablation Loosely Coupled Analysis Strategy

The two-way FTA-LCA strategy uses the discrete phase model for simulating the supersonic gas-particle two-phase exhaust plume field of the solid propellant rocket to obtain the impingement surface heat flux distribution. The impingement surface heat flux distribution was then used as the boundary condition for solidification/melting model. The solidification/melting model was used to simulate the melting behavior of the copper specimen plate. After obtaining the melt-ablation depth profile of the copper specimen plate (i.e., the shape of the ablation zone of the copper specimen plate) and the temperature distribution at that profile, we redrew the scaled ducted launcher with an impingement surface having a surface recession depth profile. The discrete phase model was then used to simulate the supersonic gas-particle two-phase exhaust plume field to obtain impingement surface heat flux distribution required for next coupling time step, where the above-mentioned temperature distribution was used as the boundary condition. After obtaining the heat flux distribution at the impingement surface again, the solidification/melting model was used to obtain the surface recession depth profile for the next coupling time step and the temperature distribution at that profile. When we keep repeating this process, we can obtain the surface recession depth profile which after long-duration of ablation.

The Solution Sequence of Two-Way Loosely Coupled Analysis Strategy

The loosely coupled analysis strategy can be described as illustrated in Figure 6. The detailed implementation procedure can be summarized as follows:

1. At the initial time \( t = t_0 \), the discrete phase model and the transient fluid-solid conjugate heat transfer model are coupled to obtain impingement surface heat flux distribution. At the time \( t \neq t_0 \), the scaled ducted launcher with an impingement surface having a surface recession depth profile \( S_{depth} \) is redrawn, and the temperature distribution \( T_w \) at the surface recession depth profile \( S_{depth} \) is used as the boundary condition. The impingement surface heat flux distribution is then obtained by the discrete phase model.

2. The impingement surface heat flux distribution \( q_w \) for the steady solution/quasi-steady solution of the flow field is obtained.

3. Taking the impingement surface heat flux distribution \( q_w \) as the boundary condition of the solid zone (i.e., the copper specimen plate), the melting behavior of the copper specimen plate is simulated through the solidification/melting model.

4. The solidification/melting model is used to simulate the melting process of copper specimen plate undergoing one coupling time step \( \Delta t \) to obtain the surface recession depth profile and the temperature distribution of that surface depth profile.

5. Repeat steps (1)–(4) to obtain the surface recession depth profile and the temperature distribution of that surface depth profile for the next coupling time step.

Note:

- Supersonic exhaust plume jet impingement flow analysis refers to the discrete phase model for simulate the supersonic gas-particle two-phase exhaust plume impingement flow field of a small scaled solid propellant rocket.
- Transient fluid-solid conjugate heat transfer analysis refers to the transient fluid-solid conjugate heat transfer model used to simulate the heat transfer behavior in the solid zone.
- Transient melt-ablation analysis refers to solidification/melting model used to simulate the melt-ablation process of the copper specimen plate.
Figure 6. Loosely coupled analysis strategy for the melt-ablation problem in supersonic exhaust impingement flow, where $\Delta t_i$ is computational time stepsize for supersonic exhaust plume jet impingement flow analysis, $\Delta t_j$ is computational time stepsize for melt-ablation analysis, $\Delta t_c$ is coupling time stepsize.

3. The Scaled Ducted Launcher for Ablation Test

Sections 3.1, 3.2, and 3.3 will briefly describe the experimental device (i.e., the scaled ducted launcher) and the results of Lin’s study [8].

3.1. The Characteristics of Solid Propellant Motor

The characteristics of the small solid propellant motor used in the test from Lin’s team [8] are shown in Table 7, where the aluminum powder accounts for 17% of the total weight of the powder. Additionally, the aluminum oxide formed after reacting with oxygen accounted for 32.1% of the total weight of the powder. However, about half of the alumina was in the gaseous state, and the rest coalesced into particles of a few microns to hundreds of microns, and so, the mass flow rate of the particle state (liquid or solid) alumina particles was about 16% of the total mass flow rate ($0.56 \times 16\% = 0.0896 \text{ kg/s}$).

Table 7. The characteristics of small solid propellant motor [8].

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Total Temperature (K)</td>
<td>3500</td>
</tr>
<tr>
<td>Chamber Pressure (kgf/cm²)</td>
<td>~65</td>
</tr>
<tr>
<td>Mass Flow Rate (kg/s)</td>
<td>0.56</td>
</tr>
<tr>
<td>Burn Time (sec)</td>
<td>3.6</td>
</tr>
</tbody>
</table>

3.2. The Scaled Ducted Launcher

The test device (i.e., the scaled ducted launcher) of Lin’s team was designed with reference to the experimental framework of Miller [34], and its structure and dimensions are shown in Figure 1. The outer shell of the test device was made of carbon steel. The steel plate had two holes with a diameter of 6 mm, which facilitated the pulling out of the thermocouple wire from the back of the test material to the outside of the test device.
During the test, the solid propellant motor was fixed below the plenum chamber so that the exhaust plume was ejected upward into the plenum chamber and directly impacted the test material. The exhaust plume then flowed through the exhaust ducts directly out the sides of the plenum chamber.

3.3. The Position of the Temperature Measurement Point

A total of 25 backside measurement points were planned for this test. Considering that only two sides of the plenum chamber had exhaust ducts, this will lead to asymmetry of the internal flow field within device. Therefore, the measurement position was divided into four directions. Its location diagram is shown in Figure 7. Except for one measurement point at the central of the specimen plate, the rest of the measurement points were 90 degrees apart on the six circles, and the radius of the circles from the inside to the outside of the position was 8 mm, 20 mm, 30 mm, 43 mm, 58 mm, and 73 mm.

The circle with a radius of 30 mm was expected to be within the impact range of the plume jet. Although the other measurement points were not directly affected, their values still had reference value for the non-impingement area. Additionally, the reason why Lin’s team did not use Martin’s measurement method [3,4] was because the thermocouple may be damaged when the surface of the test material is degraded, making it impossible to calculate the heat flux through the surface of the material through IHCA. Therefore, Lin’s team wanted to use simulation to estimate the heat flux through the surface of the test material.

Figure 7. Schematic diagram of the measurement positions on the backside of the specimen plate [8].

3.4. The Test Results of Test Material within The Scaled Ducted Launcher

The results from Lin’s team [8] showed that the max depths of impingement surface profiles near the center of the specimen plate were similar and the depth values of the surface profiles in different directions were also similar (see Table 8 and Figure 8). This is in line with Miller’s initial purpose in setting up the test device [34], which Miller mentioned in his study: “In order to test multiple materials within the device during each motor firing, flow symmetry about the plume centerline was maintained.” It is worth noting that although the flow field inside the scaled ducted launcher was asymmetric, the design of the test device focused on the need to maintain the flow symmetry of the plume centerline. For this reason, this study attempted to use a two-dimensional axis-symmetry...
model to estimate the heat flow into the surface of the test material and to reduce the computational effort of the simulation.

Table 8 shows the depth data of the surface profile along 0°, 90°, 180°, and 270° directions. These depth data of the impingement surface profile were obtained from the experimental results of Lin [8], which were measured after removing the copper specimen plate after the static firing test (note: the burn time of the solid propellant motor was 3.6 s in the experiment of Lin et al.). A curve was fitted to these depth data and the fitted curve was used to draw the scaled ducted launcher with an ablation zone shape, and the mathematical expression (i.e., mathematical function) of the curve was polynomial. The parameters of the polynomial are shown in Table 9. Figure 8 shows the surface profile depth of the specimen plate, which contained the profiles along four directions in Table 8 and the fitting curve in Table 9. Figure 8 demonstrates that the shape of the ablation zone of the impinging plate was close to an axisymmetric shape.

Table 8. The surface profile depth data along 0°, 90°, 180°, and 270° directions from the experimental results of Lin et al. [8], where ① is radial direction coordinate from the center of the copper specimen plate, ② is the supersonic nozzle exit radius.

| Radial Direction Coordinate from the Center of the Copper Specimen Plate (mm) | The Surface Depth Profile along Different Direction (mm) |
|---|---|---|---|---|
| 0 | 0 | 5.84 | |
| 2 | 7.14 | 6.59 | 6.06 | 5.23 |
| 4 | 7.91 | 7.25 | 5.65 | 5.60 |
| 6 | 8.85 | 7.83 | 6.41 | 5.76 |
| 8 | 9.47 | 8.51 | 7.88 | 7.26 |
| 10 | 10.26 | 8.80 | 9.11 | 8.97 |
| 12 | 11.13 | 8.30 | 9.48 | 11.96 |
| 14 | 12.22 | 9.34 | 9.58 | 13.18 |
| 16 | 12.47 | 12.10 | 9.50 | 13.51 |
| 18 | 12.97 | 12.86 | 9.86 | 13.66 |
| 20 | 13.77 | 12.50 | 11.33 | 13.52 |
| 22 | 13.24 | 12.32 | 13.09 | 13.75 |
| 24 | 12.71 | 11.61 | 14.03 | 13.97 |
| 26 | 14.45 | 11.04 | 12.75 | 13.41 |
| 28 | 12.12 | 9.02 | 8.68 | 11.46 |
| 30 | 9.52 | 8.10 | 6.93 | 10.49 |
| 32 | 7.32 | 6.59 | 5.83 | 8.83 |
| 34 | 2.55 | 4.33 | 5.64 | 1.27 |
| 36 | 0 | 0 | 1.54 | 0 |
| 38 | 0 | 0 | 0 | 0 |
| 40 | 0 | 0 | 0 | 0 |

Note: Exhaust duct direction is 0° and 180°.

Table 9. Polynomial parameters used to the fitting curve.

\[ Y = M_0 + M_1 \times x + \ldots + M_5 \times x^5 + M_6 \times x^6 \]

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( M_0 )</td>
<td>5.9437</td>
</tr>
<tr>
<td>( M_1 )</td>
<td>-0.1661</td>
</tr>
<tr>
<td>( M_2 )</td>
<td>0.12195</td>
</tr>
</tbody>
</table>
\[
\begin{array}{cc}
M_3 & -0.012122 \\
M_4 & 0.00064919 \\
M_5 & -1.7796 \times 10^{-5} \\
M_6 & 1.8111 \times 10^{-7} \\
R & 0.96578 \\
\end{array}
\]

Figure 8. The surface profile depth of the copper specimen plate.

4. Results and Discussion

In this study, the ablation depth profiles of the copper specimen plate were obtained by using two analysis strategies. The size of the coupling time stepsize used in the FTA-LCA strategy affected the simulation results. For this reason, the results of the two-way FTA-LCA strategy were compared with those of the two-way FTA-CA strategy to confirm the reliability of the simulation results of the two-way FTA-LCA strategy. Finally, the accuracy of loosely coupled analysis strategy was confirmed by comparing the ablation depth profiles obtained from the FTA-LCA strategy with the experimental results of Lin [8].

4.1. The Effect of Impingement Surface Profile on Impingement Surface Heat Flux

Figure 9 shows the simulation results of the scaled ducted launcher for 3D cases, which contained the impingement surface heat flux distribution for both gas-particle non-reactive flow and reactive flow cases. Figure 9a shows that the peak locations of heat flux distribution for the non-reactive flow case were similar to the reactive flow case. Figure 9b shows that the heat flux distributions for 3D cases were affected after the impingement surface recession. Additionally, the peak locations of heat flux distribution for the non-reactive flow case and reactive flow case were very close to each other.

Figure 10 shows the simulation results of the scaled ducted launcher for 2D cases, which contained the impingement surface heat flux distribution for both non-reactive
flow and reactive flow cases. Figure 10a shows that the peak locations of heat flux distribution for the non-reactive flow case were similar to the reactive flow case. Figure 10b shows that although 2D cases were influenced by the recession of the impingement surface making the peak positions far from the axis, the peak locations of heat flux distribution for the non-reactive flow case and reactive flow case were very close to each other.

Figure 11a shows the heat flux distributions for 2D and 3D cases. The heat flux curve for 3D cases was the average of the heat flux distribution curves along three selected lines. The selected lines were the straight line along the direction of the exhaust duct (i.e., 0°) and the two straight lines at an angle of 45° and 90° from the direction of the exhaust duct, the schematic diagram of the three lines are shown in Figure 11b. Figure 11a shows that the distance between the peak positions of the heat flux distribution for the 2D non-reactive flow case and for the 3D non-reactive flow case was only 1.2 mm. The distance between the peak positions of the heat flux distribution for the 2D non-reactive flow case and for the 3D reactive flow case was only 2.25 mm. Furthermore, the relative error between the peak heat flux of the 2D non-reactive flow case and the 3D non-reactive flow case was 28.83%. The relative error between the peak heat flux of the 2D non-reactive flow case and the 3D reactive flow case was 11.45%. Although there was still a gap between the 2D non-responsive flow case and the 3D cases, it can be seen from Figure 11a that the peak value of impingement surface heat flux and the peak location of impingement surface heat flux for the 2D non-reactive flow case was very close to 3D cases.
Figure 9. (a) The impingement surface heat flux distribution of the scaled ducted launcher without ablation zone for 3D cases. Left, 3D non-reactive flow case. Right, 3D reactive flow case. (b) The impingement surface heat flux distribution of the scaled ducted launcher with ablation zone. Left, 3D non-reactive flow case. Right, 3D reactive flow case. The symbol “e” means “×10” in legend, which means that “1e+003” means “1×10³”.
Figure 10. (a) The impingement surface heat flux distribution of the scaled ducted launcher without ablation zone for 2D cases. Left, 2D non-reactive flow case. Right, 2D reactive flow case. (b) The impingement surface heat flux distribution of the scaled ducted launcher with ablation zone. Left, 2D non-reactive flow case. Right, 2D reactive flow case.
4.2. Comparison of Transient Heat Transfer Model and Transient Fluid-Solid Conjugate Heat Transfer Model

In order to confirm the transient heat transfer model can accurately calculate the temperature distribution within the impinging plate, we compared the temperature distribution within the impinging plate between the transient heat transfer model and the transient fluid-solid conjugate heat transfer model. From Figure 12a, it can be seen that the simulation results calculated by the transient heat transfer model and the transient fluid-solid conjugate heat transfer model were very close to each other. In addition, Figure 12b shows that the impingement surface temperature distributions for both heat transfer models were nearly the same at simulation time $t = 0.01$ sec (i.e., burn time was 0.01 sec). The result was expected. This was due to two models differ only in the discretization method and numerical method. Therefore, the results of the two models should be similar or even identical. After confirming that the simulation results of the two models were the same, we calculated Equation (33) through the UDF code and simulated the melt-ablation behavior of the copper plate by using the dynamic mesh technique. This method is called the two-way FTA-CA strategy in this study.
4.3. Comparison of Ablation Depth Profiles for 2D and 3D Cases

In order to reduce the computational cost, we chose to use the two-dimensional non-reactive flow case to obtain the ablation process of copper plate in the scaled ducted launcher. Two analysis strategies were used in the study, they were two-way FTA-CA strategy and two-way FTA-LCA strategy. The former obtained the ablation depth profile of the copper specimen plate by coupling the discrete phase model and the transient heat transfer model. The latter obtained the ablation depth profile of the copper specimen plate by coupling the discrete phase model and the solidification/melting model. The reason for choosing the 2D non-reactive case is that the heat flux distribution of the 2D non-reactive case is similar to the 3D reactive case and 3D non-reactive case. More importantly, the peak value of heat flux and the peak location of impingement surface heat flux distribution for 2D non-reactive case were close to the 3D reactive case, both in the scaled ducted launcher with or without ablation zone. Therefore, we tried to use the simulation results of the 2D non-reactive case to obtain the ablation depth profile of the copper specimen plate after long-duration of ablation.

Figure 13a shows the ablation depth profiles for the 2D and 3D cases in two-way FTA-LCA strategy, where the burn time was 0.6 s. We can observe from Figure 13a that the ablation profiles of the 3D reactive and 3D non-reactive cases were very close to each other, and the difference between their maximum depths was only 0.4 mm. The ablation profile for 2D case was slightly smaller than that for 3D non-reactive case, and the difference between their maximum depths was only 1.2 mm.

Figure 13b shows the ablation depth profiles for the 2D non-reactive flow case and 3D reactive flow case, where the burn time was 0.6 s. It can be observed from Figure 13b that the maximum of ablation depths of 2D non-reactive flow case were close for two coupled analysis strategies. Additionally, the ablation depth profiles of two analysis strategies were also very similar. Figure 13b also shows that the ablation depth for 3D reactive flow case was deeper than 2D non-reactive flow case in the range of $r / R_w = 0.5 \sim 1$. This was
due to the peak heat flux position of 3D reactive flow case being farther away from the axis than that of 2D non-reactive flow case and the heat flux value in the range of \( r / R_N = 0.5 \sim 1 \) for 3D reactive flow case being higher than 2D non-reactive flow case. Therefore, the 3D reactive flow case using the loosely coupled analysis strategy had a deeper ablation depth in the range of \( r / R_N = 0.5 \sim 1 \) compared to 2D non-reactive flow case.
**Figure 13.** (a) The ablation depth profiles predicted for 2D and 3D cases in two-way fluid-thermal-ablation loosely coupled analysis strategy, where the ablation depth profiles for 3D cases are the average of the ablation depth profile along orientation of three selected lines, burn time is 0.6 sec. (b) The ablation depth profiles predicted for 2D non-reactive flow case and 3D reactive flow case in two coupled analysis strategies.


Although the numerical model of the two-way FTA-LCA strategy can obtain the surface recession profile after long-duration of ablation, the coupling time stepsize and the accuracy of the simulation results of the loosely coupled analysis strategy need to be confirmed by the simulation results of the two-way FTA-CA strategy. The reliability of the simulation results of the FTA-LCA strategy can be said to be based on the results of the fluid-thermal-ablation coupled analysis strategy. Therefore, we need to compare whether the profiles of two strategies are close to each other.

Figure 14 shows the ablation profiles of the two-way FTA-CA strategy and the two-way FTA-LCA strategy with burn times of 0.3, 0.6, and 1.2 s. It can found that the ablation profiles of the two analysis strategies were very close to each other at 0.3, 0.6, and 1.2 sec. This demonstrates the reliability of the simulation results of the loosely coupled analysis strategy.

**Figure 14.** Variation of melt-ablation depth profile of copper specimen plate with time for two analysis strategies.

4.5. **Comparison of Simulation Results and Experimental Data**

Since the computational cost of the FTA-CA strategy was still huge, we compared the predicted results of the FTA-LCA strategy with the experimental results of Lin et al. after confirming the accuracy of the loosely coupled analysis strategy.
Figure 15a shows the ablation depth profiles for two-way FTA-LCA strategy, where the ablation depth profile for 3D reactive flow case is the average of the ablation depth profile along orientation of three selected lines. The ablation depth profile for 2D non-reactive flow case and 3D reactive flow case at burn time of 3.6 s is close to the experimental results of Lin’s team [8]. Additionally, we compared the simulation result and the study results of Lin and his coworkers. From Figure 15b, it can be seen that the temperature distributions of the backside surface of copper impinging plate for two-way FTA-LCA strategy were slightly higher than the simulation result of Lin’s team and their experimental data, but still similar to their experimental results.
Figure 15. (a) The ablation depth profiles predicted by two-way fluid-thermal-ablation loosely coupled analysis strategy for coupling time stepsize $\Delta t_c = 0.6$ s, where the depth data for the four directions from Lin [8] and the fitting curve is plotted that based on these data. $r$ is radial direction coordinate from the center of the copper specimen plate; (b) temperature distribution of the backside surface of copper plate, where burn time is 3.6 s, the temperature distribution profile for 3D case is the average of the temperature distribution profile along orientation of three selected lines which on the backside of the copper plate.

4.6. Comparison of the Total Computation Time of Two Coupled Analysis Strategies

Figure 16 shows the total computation time for two coupled analysis strategies, where the coupling time stepsize was 0.6 s for the loosely coupled analysis strategy, the burn time of both coupled analysis strategies was 1.2 s. It can be observed from Figure 16 that the loosely coupled analysis strategy can avoid the detailed transient supersonic exhaust plume jet impingement flow analysis, thereby reducing a large amount of computational effort (i.e., reducing a large amount of the computational cost) and reducing the total computational time. Total computation time difference between two analysis strategies was tens of times. It is worth noting that when the number of grids increased, the computational effort to solve the Eulerian gas phase model will increase. Additionally, when the number of tracked particles in the computational domain increases, the computational effort to solve the Lagrangian particle phase model will increase. Both of these problems cause an increase in the execution time for each time step during the simulation (i.e., an increase in the computational cost). This results in an increase in the total computational time as well.

Figure 16. Total computation time for different coupled analysis strategies.

5. Conclusions

In this paper, we used a two-way FTA-LCA strategy to effectively predict the melt-ablation process of copper specimen plate in a supersonic gas-particle two-phase exhaust plume impingement flow. For the long-duration melt-ablation problem of thermal protection material, the loosely coupled analysis strategy provided the analysis solution accuracy and the considerable gain in computational efficiency for predicting the long-duration ablation problem in supersonic gas-particle two-phase exhaust plume impingement flow.

In this study, two analysis strategies were established and the reliability of the loosely coupled analysis strategy was confirmed by comparing the FTA-CA strategy with the FTA-LCA strategy. Finally, the predicted ablation depth profile of the loosely coupled analysis strategy was similar to the experimental results of Lin’s team. Moreover, the temperature distribution of the backside surface of copper plate is also close to the experimental results of Lin’s team.

**Author Contributions:** Conceptualization, C.-Y.L. and T.-S.L.; Methodology, C.-Y.L. and T.-S.L.; Software, C.-Y.L.; Validation, C.-Y.L.; Investigation, C.-Y.L.; Resources, T.-S.L.; Writing – original
draft, C.-Y.L.; Writing – review & editing, T.-S.L.; Visualization, C.-Y.L.; Supervision, T.-S.L.; Project administration, T.-S.L.; Funding acquisition, T.-S.L. All authors have read and agreed to the published version of the manuscript.

**Funding:** The study was partially supported by the funding from National Science and Technology Council, R.O.C under the contracts of MOST 110-2221-E-006-103-MY3 and MOST 108-2221-E-006-080-MY3.

**Data Availability Statement:** The final data presented in this study are available on request from the corresponding author.

**Conflicts of Interest:** The authors declare no conflict of interest.

**References**


Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.