Investigation into far-field drag calculation methods and pseudo total enthalpy generation for airplane CFD

Kisa MATSUSHIMA*, Ryosuke SHIMIZU* and Kentaro GOSHIMA*
* Department of Mechanical Engineering, University of TOYAMA
Gofuku 3190 Toyama, 930-8555, Japan
E-mail: kisam@eng.u-toyama.ac.jp

Received: 29 March 2019; Revised: 26 May 2019; Accepted: 13 August 2019

Abstract
In this study drag calculations were performed for an airplane cruising at transonic speed in a flow-field using two far-field methods. Subjected flow-fields in the calculations were computational fluid dynamics (CFD) results of Reynolds-averaged Navier–Stokes simulation (RANS) around a wing. The objectives were to establish effective means to utilize far-field drag methods and wake flow phenomena as well as report pseudo total enthalpy which peculiarly appeared in the CFD simulation results. The drag calculation was completed using four wake integration equations. The first originated from the equation of the momentum conservation law itself, the second was based on enthalpy variation, the third was based on entropy variation, and the last was a method for induced drag calculation. Drag values resulting from the integrations were compared to those via a near-field method commonly used in CFD. Through the computation and comparison of wake integration drag values, it was proved that the both far-field methods were able to predict a comparably accurate drag value compared to that of the near-field method. In addition, strange inappropriate total enthalpy (pseudo total enthalpy) generation was found in some regions where a flying airplane never affects flow behavior. In the wake integrations, the pseudo total enthalpy generation seems to compensate for the drag value via the first equation. The pseudo total enthalpy generation depends on far-field boundary locations. In summary, wake integrations are promising to provide useful information for accurate drag prediction.

Keywords: Airplane wake, Computational fluid dynamics, Drag, Far-field method, Wake integration, Total enthalpy variation

1. Introduction

For airplane aerodynamic design, it is important to obtain accurate drag prediction tools. There have been two categories for calculating drag force on an airplane: integrating pressure and skin-friction acting on the airplane surface, termed a “near-field” calculation, and integrating the deficits of momentum flux and pressure over the surfaces of a control volume surrounding the airplane in a flow-field, termed a “far-field” calculation. Several research groups have been actively involved in “far-field” drag prediction for both experimental and computational fluid dynamics (CFD) simulation (Meheut et al., 2008) (Mele et al., 2017) (Yamazaki et al., 2005; 2008). In the field of CFD, near-field calculation is popular to obtain drag forces. However, an alternative method is welcome to obtain more reliable drag values because each drag calculation has its own advantages. For example, a far-field calculation has drag breakdown capability which decomposes total drag into several components, each of which is generated by a different physical factor from that of the others. This capability is particularly useful for aerodynamic design. There have been developments and many studied methods derived from the far-field calculation concept. Among them, it has been found that the integration domain can be reduced to only a downstream (outflow) surface when the control volume is appropriately selected. This reduced form of far-field calculation is termed “wake integration.” It was first developed for wind tunnel experiments by Betz during the early twentieth century. Then, the method was improved and extended to meet the need for accurate lift and drag prediction in wind tunnel experiments (Kusunose, 2005 and References
therein) and numerical simulation (Giles and Cumming, 1999) (Hunt and Giles, 1999). A merit of wake integration is simplicity, because it is performed over a simply shaped wake plane. In this study, we investigated resultant drag values using several types of wake integration equations. The wake integrations were conducted on flow-fields using Reynolds-averaged Navier–Stokes (RANS) CFD simulation. The drag values were compared those of near-field. The effect of a wake plane position along the freestream direction (x-axis) on the predicted drag values was examined. In addition, we discussed the interesting behavior of a spurious total enthalpy variation (ΔH) via CFD computation. Then, the role of the spurious ΔH was considered in terms of wake integration and CFD simulation.

2. Basic Equations of Wake Integration for Drag Prediction
2.1 Near-field Method
Drag via the “near-field” method is expressed as $D_{\text{surface}}$ in this study, which is calculated using the following equation, assuming the freestream flows to the positive direction of the $x$-coordinate.

\begin{equation}
D_{\text{surface}} = \int_{S_0} \left[ P \cdot n_x - \bar{t}_x \cdot \tilde{n} \right] dS
\end{equation}

where $P$ is the pressure, $\tilde{n}$ is a unit normal vector, and $\bar{t}_x$ indicates the $x$-directional component ($\bar{t}_{xx}, \bar{t}_{yx}, \bar{t}_{zx}$) of a viscous stress tensor on the airplane (wing) surface $S_0$ as shown in Figs. 1 and 2.

2.2 Wake Integration Method
Several types of wake integration equations were used to predict a drag value (Kusunose, 2005). The first is the primary equation of Eq. (2).

\begin{equation}
D = \int_{S_2} \left[ \rho u \left( U_\infty - u \right) + \left( P_\infty - P \right) \right] dS
\end{equation}

It is derived from the integral form of the momentum conservation law (i.e. the Navier–Stokes equations). The equation for the $x$ component of the integral form in a steady state condition is

\begin{equation}
0 = \int_S \left[ - \rho u (\bar{V} \cdot \tilde{n}) - P \cdot n_x + \bar{t}_x \cdot \tilde{n} \right] dS
\end{equation}

where $\bar{V}$ is a velocity vector whose $x$, $y$, and $z$ components are $u$, $v$, $w$, respectively.

In a two-dimensional case, the integral path, $S$, consists of $S_2$, $S_C$, $S_{\text{cut}}$, $-S_{\text{cut}}$, and $S_0$ as shown in Fig. 2. $S$ is the enclosed path of fluid area in a control volume. The integrations of $S_{\text{cut}}$ and $-S_{\text{cut}}$ cancel out each other. On $S_0$ path, $\bar{V}$ should be $\bar{0}$. Then, Eq. (2) yield to

\begin{equation}
\int_{S_0} \left[ P \cdot n_x - \bar{t}_x \cdot \tilde{n} \right] dS = \int_{S_2+S_C} \left[ - \rho u (\bar{V} \cdot \tilde{n}) - P \cdot n_x + \bar{t}_x \cdot \tilde{n} \right] dS
\end{equation}

The left-hand side of Eq. (4) corresponds to $D_{\text{surface}}$. The right-hand side (R.H.S.) is transformed into Eq. (5) using the conservation of mass and the fact that the closed path integral of $n_x$ is zero;

\begin{equation}
R.H.S. = \int_{S_2+S_C} \left[ \rho (U_\infty - u) (\bar{V} \cdot \tilde{n}) + \left( P_\infty - P \right) n_x + \bar{t}_x \cdot \tilde{n} \right] dS
\end{equation}

where $U_\infty$ and $P_\infty$ are the freestream velocity and pressure, respectively. If the $S_C$ surface is so far from the wing that flows on the $S_C$ are minimally affected by the existence of the wing, the integration on $S_C$ is negligible. On $S_2$ path, $\bar{V} \cdot \tilde{n} = u$ and the viscous stress term may be neglected because $S_2$ is also sufficiently far from the wing. Then, the R.H.S. yields Eq. (2) whose integration result, $D$, is numerically equals to $D_{\text{surface}}$ (Giles and Cumming, 1999) (Shimizu et al., 2017).

The second, third, and fourth equations are derived from the perturbation form of Eq. (2) (Kusunose, 2005). They are derived in order to decompose drag $D$ into several elements which are enthalpy drag $D_h$, profile drag (entropy drag) $D_p$, and induced drag $D_i$. The definition of $D_h$ of Eq. (6) is different from its original definition as referred to by other literature (Kusunose, 2005). The original equation integrates $-\rho_\infty \Delta H$, but in this study the integrand has no
negative sign. Thus, the $D_h$ value here has an opposite plus or minus sign to that of the original. Thus, in this study, $D_h$ is termed the minus-signed enthalpy drag.

$$D_h = \iint_{S_2} \rho \omega \Delta H \, dS$$  \hspace{1cm} (6)

$$D_p = \iint_{S_2} P \frac{\Delta s}{R} \, dS - \iint_{S_2} \frac{P \omega}{2} \left( \frac{\Delta s}{R} \right)^2 \, dS$$  \hspace{1cm} (7)

$$D_i = \iint_{S_2} \frac{\rho \omega}{2} (v^2 + w^2)\, dS - \iint_{S_2} \frac{\rho \omega}{2} (1 - M^2 \omega) (\Delta u)^2 \, dS$$  \hspace{1cm} (8)

Some symbols used in Eqs. (6)–(8) are shown in Figs. 1 and 2. The $S_2$ plane is perpendicular to the freestream ($U_\infty$) direction. In this study, $S_2$ is alternatively expressed as a wake plane. In Figs. 1 and 2, the $U_\infty$ direction is along the x-axis. $D$ represents the total drag. Eq. (2) is the basic and primary form of the wake integration concept, where $\rho u$ is the x momentum and $P$ is the pressure. $D_h$ is the minus-signed enthalpy drag $\Delta H = H - H_\infty$, where $H$ represents the total enthalpy of the unit volume at any location in a wake plane while $H_\infty$ is the freestream total enthalpy. $H$ is calculated using fluid properties as $H = \gamma / (\gamma - 1) \cdot (P / \rho) + \frac{1}{2} (u^2 + v^2 + w^2)$.

$D_p$ is the drag resulting from entropy variation ($\Delta s = s - s_\infty$) from uniform flow states. $\Delta s$ is expressed as $\Delta s = R \left( \frac{1}{\gamma - 1} \ln \frac{P}{P_\infty} + \frac{\gamma}{\gamma - 1} \ln \frac{\rho}{\rho_\infty} \right)$. $D_i$ is related to lift. $(u, v, w)$ is a velocity vector in the rectangular coordinate (x, y, z). $M_\infty$ indicates the freestream Mach number of a flow-field. $R$ is the gas constant. $\gamma$ is a ratio of specific heats.

From a theoretical point of view, $D_{surface} \equiv D \equiv -D_h + D_p + D_i$. If an airplane has no powered engine, $\Delta H$ is negligible such that $D_h$ yields zero. The target flow-fields for wake integration in this study are those around a simple wing without engines, as shown in the next section. Therefore, the enthalpy drag is supposed to be zero. The wake integration for drag force was conducted expecting the following relations in Eqs. (9) and (10).

$$D_{surface} \equiv D_p + D_i$$  \hspace{1cm} (9)

$$D_{surface} \equiv D$$  \hspace{1cm} (10)
3. Flow-field for Drag Prediction

For the examination of drag prediction, RANS simulation results were prepared. It is a flow field past a rectangular wing whose section shape is NACA0012; the flow speed is Mach 0.82, AOA is 4.86°, and Reynolds number is 3.0 million based on the wing chord length. First, we performed flow simulations around the rectangular wing with two computational spaces of different size. Both computational spaces of the C-H topological mesh system are shown in Fig. 3. The far-field boundary location of the smaller one is ten times the chord length from the wing and the other is twenty times the length. The wing chord length is abbreviated as C. Therefore, the size of the computational domain for the first case (FF10) in each coordinate is from -10C to 11C in the x, from 0 to 15C in the y coordinate and, and -10C to 10C in the z coordinate. That for the second case (FF20) is from -20C to 21C in the x coordinate, from 0 to 30C in the y coordinate, and from -20C to 20C in the z coordinate. The plane of y=0 is the symmetrical boundary for the RANS simulation. The total number of mesh points for each case is 1.16 million and 1.53 million, respectively. Regarding the resolution of the both mesh distributions in the boundary layers, the minimum spacing in the normal direction to the wing wall is 10^6C which results in 4.7 of the averaged y^+ value on the wing and the number of mesh points inside a boundary layer is at least 30.

Typically for efficient CFD simulation during the design process, we would like to adopt as small a computational domain as possible as long as simulation using the domain can produce reliable results. CFD simulation requires less time when a smaller domain is used. Therefore, the far-field boundary was set at 10C or 20C from a wing notwithstanding the location of the far-field boundaries in this study might not be sufficiently distant.

Regarding numerical schemes for the RANS simulation, the Finite Volume Method (FVM) for structured mesh was adopted where advection terms are discretized using the TVD method of HLLEW Riemann solver with Venkatakrishnan limiter. Third order accuracy was attained by using the third-order MUSCL scheme. The Baldwin–Lomax turbulence model was used to evaluate turbulent viscosity. The time integration was completed using the Lower Upper-Symmetric Gauss-Seidel method. Boundary conditions of the inflow, outflow, and far-field boundaries were a Dirichlet type; the density, pressure and momentum in each direction have the same values same as those of uniform flow, respectively.

Figure 4 shows the wing and one of its wake planes which is several chords downstream of the wing and perpendicular to the main flow stream direction. The simulation result of the pressure coefficient (Cp) contours and mesh distribution are presented. In the wake plane, it seems that the influence of the wing on a flow-field weakens when flows are near the far-field boundary such as the planes on z = ±10C. However, at the far-field boundaries, their locations and boundary conditions slightly affect the behavior of flows near the far-field boundaries. Even a slight effect might sometimes lead to substantially deteriorating drag prediction capability.

Figure 5 shows the wake planes upon which wake integrations were conducted. During an actual design work phase, the wake integrations for drag would be calculated over one wake plane. In this study the integrations were completed over numerous planes because we wanted to analyze how a predicted drag value depends on the wake plane position downstream of a wing. For analysis, 30 locations were selected along the x-coordinate direction. The region for integration, $S_z$, in Eqs. (2) and (6)–(8), is the entire area of a wake plane at every location. The relation between the wake plane layout and a CFD mesh is explained when the main flow stream has non-zero angle of attack (AOA) with the wing chord line. There were two mesh systems to realize the AOA. One was to have the main flow stream direction inclined while a wing section chord is parallel with the x axis as shown in Fig. 6. The other was to have a wing inclined while the main flow stream direction is parallel with the x axis as in Fig. 7. In the first case, a wake plane intersects mesh lines and a rather complicated interpolation is required to obtain physical values in the wake integration equations. However, a wake plane is parallel to the mesh (y-z) plane in the second case where no complicated interpolation is required for the wake integrations. We compared the drag prediction accuracy of both cases. There is a less than 1% difference between them in terms of the near-field drag value as well as the wake integration drag prediction capability. In this study, the first case was adopted because the parametric study of changing AOA is more easily conducted.

In addition, to verify the applicability of the wake integration equation for drag, we conducted another CFD simulation adopting a different turbulence model, the Spalart–Allmaras model, using FF20 mesh. We also completed a simulation using refined FF20 mesh. In the x and y directions, every mesh point was halved. Through both the turbulence model and mesh refinement studies, the performance of the wake integration for drag prediction over the CFD results is the same as that over the original results within 1%. This performance is based on to what extent Eqs.
Fig. 3 Two computational mesh spaces are prepared for RANS simulations; the left is the smaller, termed FF10, where the far-field boundary is 10C from a wing while the right is the larger, termed FF20, where the boundary is 20C from it. Both are a half space around a semi-span wing because the CFD simulations were conducted using symmetrical conditions.

Fig. 4 $C_p$ distribution and mesh in the wake flow-field of a semi-span model of a rectangular wing

Fig. 5 Locations of wake planes downstream from the trailing edge of a semi-span wing

Fig. 6 Wake plane layout when a wing is set parallel to the $x$ axis. The main flow direction is not normal to the $z$ axis.

Fig. 7 Wake plane layout when a wing is inclined with AOA relative to the $x$ axis. The main flow direction is normal to the $z$ axis.
(9) and (10) hold and how large is the error between the near-field and wake integration drag. Assessment of the performance is a major objective of this study.

4. Discussion of Wake Integration in Rectangular Wing Simulation

Drag values were transformed to drag coefficients, such that $D$ was transformed to $CD$ and $D_h$ to $CD_h$ while the “near-field” drag was transformed to the $CD_{surface}$. The integral area of a wake plane, $S_2$, was defined as the entire square plane of the computational space. The edges of every $S_2$ plane in the FF10 case range from -10C to 10C in the $z$ direction and from 0 (the symmetrical center line position) to 15C in the $y$ direction at each $x$ location. Those for the FF20 case range from -20C to 20C in the $z$ direction and from 0 (the symmetrical center line position) to 30C in the $y$ direction.

4.1 Profile Drag and Induced Drag

In this section, Eq. (9) is examined. Figure 8 shows $CD_p$, $CD_i$, and the sum of $CD_p$ and $CD_i$ which is denoted by $CD_t$ compared to the near field drag coefficient, $CD_{surface}$. The left graph and table present each $CD$ value of the 30 different wake planes obtained from FF10, the smaller case for CFD computation. The right ones show $CD$ values from FF20, the larger case. The horizontal axis of the graphs is the wake location in the $x$ direction which is the distance from the wing leading edge. The distance is normalized by the wing section chord, $C$. The vertical axis indicates the drag coefficient value. The near-field drag coefficient is also shown with a blue straight solid line. In the vicinity of the wing trailing edge ($x=1.0$), each $CD$ value rapidly changes with an increase in distance. After the distance reaches 2.5, all the $CD$ values remain constant. $CD_t$ agrees with $CD_{surface}$ with less than 1% error. As expected, Eq. (9) holds and the drag decomposition successfully works. The total $CD_t$ obtained by the present drag integration method is irrespective of the far-field boundary location of the CFD computational spaces. The induced drag rate in the total drag is approximately 15%, which is a lower percentage than general passenger airplanes. This is partly because strong shock waves are generated during the wing simulation. Shock waves cause additional profile drag. Thus, the induced drag rate is relatively lower. The result is also because of numerical dissipation given the lack of mesh resolution in the wakes to capture reversible vortices.

Fig. 8 $CD_p$ using Eq. (7) and $CD_i$ using Eq.(8) and the sum of them at different wake plane location compared to the $CD_{surface}$ using Eq.(1). The horizontal axis of each graph indicates a plan location coordinate normalized by $C$ for wake integration. Each drag coefficient value at the wake plane, $x \approx 6C$, is numerically presented in tabular form.
Fig. 9 Profile drag source distributions on the symmetrical half crossflow plane at $x = 6C$ of the entire computational space. For FF10, the plane size is from -10C to +10C in the z direction and 0 to 15C in the y direction. For FF20, it is from -20C to +20C in the z direction and 0 to 19C in the y direction. In the plane area the drag source is nearly zero, $-10^{-4} < \text{profile drag source} < 10^{-4}$.

Fig. 10 Enlarged contours of profile drag source distributions in each enclosed region within a dotted blue rectangle in the maps of Fig. 9. Both the FF10 and FF20 regions contain a substantial amount of drag source.

Fig. 11 Induced drag source distributions on the symmetrical half crossflow plane at $x = 6C$ of the entire computational space. For FF10, the plane size is from -10C to +10C in the z direction and 0 to 15C in the y direction. For FF20, it is from -20C to +20C in the z direction and 0 to 19C in the y direction. In almost the plane area the drag source is nearly zero, $-10^{-4} < \text{induced drag source} < 10^{-4}$.

Fig. 12 Enlarged contours of induced drag source distributions in each enclosed region within a dotted blue rectangle in the maps of Fig. 11. The FF10 and FF20 regions, where a substantial amount of drag source exists, are shown in detail.
The present wake integration shows minimal dependence on the wake plane location as long as it is greater than 1.5C downstream from the wing trailing edge. Precisely observing the graphs, the \( CDp \) values are slightly increasing by several drag counts and the \( CDi \) values are slightly decreasing, as the wake plane location moves down stream, while the total \( CDt \) values remain the same. This is the same tendency as the results of other studies in which \( CDi \) was calculated using Maskell’s equation (Ueno et al., 2013).

As previously mentioned, Eq. (9) is the expression of drag decomposition. Using this expression, we can observe which physical phenomena are responsible for each drag component as well as how much they affect each. First, the profile drag source, which is the sum of the integrands of the two terms in Eq. (7), is shown in Figs. 9 and 10. In Fig. 9, the contour map of the profile drag source distributions is show in the entire domain of integration. The left contour map is of the smaller mesh case, while the right one is of the larger mesh case. To see detailed flow phenomena of the wake plane in Fig. 9, an enclosed area within a dotted blue rectangle on each contour map is zoomed. The area is supposed to contain a substantial amount of drag sources. The enlarged contour maps of Fig. 9 are shown in Fig. 10. The enlarged maps of FF10 and FF20 agree with each other. The profile drag sources are clearly recognized; the viscous shear layer led by the wing boundary layers, the entropy increase due to shock waves, and tip vortices. Thus, the \( Dp \) definition of Eq. (7) includes wave drag. Second, the induced drag source, which is the sum of the integrands of the two terms in Eq. (8), is shown in Figs. 11 and 12. The drag source distributions of the entire domain are shown in Fig. 11. The section where a substantial amount of induced drag source exists is wider than the section for the profile drag. Figure 12 shows the enlarged maps of Fig. 11. A strong contribution of tip vortex to drag generation can be seen. As for the profile and induced drag sources, there is no recognizable difference found in the two cases of FF10 and FF20. The drag calculation using Eqs. (7) and (8) is not affected by the far-field boundary locations of the CFD meshes.

### 4.2 Minus-Signed Enthalpy Drag

In this section, we examine whether \( Dh \) is zero or not. As stated in Section 2.2, aero-thermodynamics predicts it should be zero for real flows. The wake integration of Eq. (6) was conducted on the aforementioned flow-field via CFD simulation. Drags \( Dh \) obtained by Eq. (6) were transformed into non-dimensional values, such as \( CDh \), an enthalpy drag coefficient (strictly speaking it is minus-signed). The \( CDh \) values of wake integration at different wake plane locations in the \( x \) direction are plotted in the two graphs in Fig. 13. The left graph shows FF10 simulation and wake integration results and the right one shows FF20 results. In these graphs, the horizontal axis indicates the wake plane location in the \( x \) coordinate (see Fig. 5) while the vertical axis shows the drag coefficient value. The near-field drag coefficient is presented with a blue straight line. Contrary to expectation based on aero-thermodynamic theory, the \( CDh \) values are not zero for all CFD simulation flows. The FF10 of the smaller mesh space produces enthalpy drag coefficients as much as one-half of the total drag coefficient, \( CDsurface \). The FF20 of the larger mesh generates much lower enthalpy drag coefficients than those of FF10; they are approximately one quarter of the \( CDsurface \).

Because we encounter the curious behavior of \( CDh \), the contour map of the integrand of Eq. (6), \( \rho \omega \Delta H \), is shown in several planes selected from the FF10 and FF20 computations. Figure 14 shows the distribution of \( \rho \omega \Delta H \) on the \( x-z \) plane at \( y = 0 \) (symmetrical center plane) and a crossflow \( (y-z) \) plane of \( x = 6C \) of the two mesh spaces shown in Fig. 3. The smaller size contour maps are of FF10 with the far-field boundary location of 10C while the larger size maps are of FF20 with the far-field boundary location of 20C. A certain amount of \( \rho \omega \Delta H \) can be recognized in the wide region near the far-field boundary, particularly in the FF10 contour maps. For CFD computation, every flow variable is normalized by its value in the free-stream flow-field; then, \( \rho \omega \Delta H = \Delta H \) because \( \rho \omega = 1.0 \). In the FF20 contour maps, the distribution appears much more natural than that in FF10 case; most of the computational space, \( \Delta H \), is less than \( 10^{-5} \). The whole contour map domain of the semi-ellipse and rectangle of FF10, respectively, corresponds to the subdomain inside the blue dotted semi-ellipse and rectangle of FF20’s symmetrical plane (Fig. 14 (c)) and crossflow plane (Fig. 14 (d)). FF10’s semi-ellipse \( \Delta H \) contour map was compared to the semi-ellipse subdomain surrounded by blue dotted lines in the FF20. Though both are geographically the same, their flow physics are quite different. In the same manner, the FF10 rectangular contour map was compared to the rectangular subdomain in the FF20 map. In the subdomain of FF20, there is no total enthalpy variation except the limited area of the wing boundary layers and their trace in the wake, which the authors believe is the real physics. Moreover, the wake integrations conducted on the subdomain in FF20 means the integral interval in the \( z \) direction is from -10 to +10 resulting in zero \( Dh \). The positive \( \Delta H \) and negative \( \Delta H \) cancel out in the boundary layer region near the wing and its intrinsic wake. However, the nonzero \( \Delta H \) in the region neighboring the far-field boundaries is non-physical. Such a \( \Delta H \) (non-physical total
enthalpy variation) is termed a pseudo (or spurious) \( \Delta H \) in this article. Accordingly, non-physical \( D_h \) is termed a pseudo (or spurious) \( D_h \).

![Fig. 13](image)

**Fig. 13** \( C_{D_h} \) values via Eq. (6) at different wake plane locations compared to \( C_{D_{surface}} \) via Eq. (1).

![Fig. 14](image)

**Fig. 14** Contour maps of \( \Delta H \) distributions on the symmetrical \((v = 0)\) and crossflow \((v = 6C)\) planes of the CFD simulation results. The two semi-ellipse contour maps are for the symmetrical plane; the right rectangle maps are for the crossflow plane. The upper semi-ellipse and rectangle show the CFD results using the FF10 mesh and the lower ones those using the FF20 mesh.
To investigate the cause of the pseudo $\Delta H$, its dependency on a turbulence model or mesh resolution was examined using the simulation results of the Spalart–Allmaras turbulence model or the doubly refined mesh mentioned in the third section. Minimal dependence was found in either case. Even for the FF20 case, the pseudo $CD_h$ is not negligible, though the FF20 mesh provides much more reasonable flow phenomena than those of the FF10 mesh. A certain amount of pseudo $\Delta H$ still remained near the far-field boundaries. The upper far-field boundary region has a negative $\Delta H$, while the bottom boundary region has a positive $\Delta H$. For the next examination of pseudo $\Delta H$, we reviewed the Dirichlet type boundary condition which imposes uniform flow properties on far-field boundaries. Strictly speaking, the flow at 20C from the wing is not equal to the uniform flow. Let us consider the velocity, applying a simple circulation theory of lift to the FF20 flow-field. If the wing span were infinite and its lift force per unit span length were $L$, the velocity on the lower far-field boundary would be approximately $U_\infty - L/(2\pi(20C)\rho_\infty U_\infty)$ and that on the upper would be $U_\infty + L/[2\pi(20C)\rho_\infty U_\infty]$. To impose uniform flow properties on the far-field boundaries would yield a positive impulse for flow on the lower boundary but a negative impulse for flow on the upper boundary. The positive impulse causes a positive $\Delta H$ and vice versa.

Then, to inspect the effect of circulation associated with lift force on the pseudo $\Delta H$, a zero-lift simulation was performed using FF20 mesh. Figure 15 shows the distribution of $\Delta H$ on the $x-z$ plane at $y = 0$ (symmetrical center plane) and that of a cross flow plane on $x = 6C$ of the CFD simulation. The contour maps of $\Delta H$ distribution indicate that the spurious $\Delta H$ everywhere is less than $10^{-5}$. Comparing Fig. 15 to the FF20 case $\Delta H$ contour maps, circulation by lift force generation affects the pseudo $\Delta H$. However, for the wake integrations in the zero-lift simulation, a $CD_h$ value occur that is approximately 50 drag counts.

The next examination in this study was far-field boundary location. Additional three simulations were conducted for the same flow-field as that described in the third section. The far-field boundary locations of computational mesh systems used for the three simulations were much further from the wing while their mesh resolution was equivalent to that of FF20. The first was termed FF30 whose far-field boundaries were 30C, the second was FF50 whose far-field boundaries were 50C, and the third was FF100 whose far-field boundaries were 100C from the wing.

In Fig. 16, the contour maps of (a), (b) and (c) show the $\Delta H$ distributions for the FF30, FF50, and FF100 simulation results, respectively. The further the location of the far-field boundaries from a wing, the less pseudo $\Delta H$ occurs. The maximum of the absolute value of the pseudo $\Delta H$ distributed on each contour map of (a), (b), and (c) in Fig. 16 was compared to that of FF20 in Fig. 14 (c) and (d). The ratio of the maximum of the FF30 case to that of FF20 is 0.34, to that of FF50 is 0.067, and to that of FF100 is 0.018. In terms of wake integration results, the $CD_h$ of FF30 is approximately 80% of that of FF20, FF50 is approximately 55%, and FF100 is approximately 50%.

---

**Fig. 15** Contour maps of $\Delta H$ distributions on the symmetrical ($y = 0$) and crossflow ($x = 6C$) planes of the zero-lifted CFD simulation results (AOA = 0). The left semi-ellipse contour map is for the symmetrical plane; the right rectangular map is for the wake of the FF20 case.
Consequently, the location of the far-field boundaries impacts the generation of pseudo $\Delta H$. We believe FF50 is the best choice for the present CFD simulation considering the balance between computational workload and theoretical accuracy. From our limited experience regarding the simulation of the present flow, we also realize other factors remain to cause a pseudo $C_{D_h}$. In fact, numerical schemes to impose boundary conditions would affect it. Some other CFD codes of different schemes generate less (but non-zero) spurious $D_h$.

4.3 Drag from the Momentum Balance Eq. (2) and the Compensating Behavior of the Enthalpy Drag

In this section, we examine Eq. (9) and whether the $D$ of Eq. (2) is equal to $D_{surface}$. For the first discussion, the drag values were transformed into coefficient form, $C_D$ and $C_{D_{surface}}$, respectively. Both coefficients are plotted in the two graphs shown in Fig. 17. $C_D$ values on the 30 different wake planes are plotted in dark blue triangles. The $C_{D_{surface}}$ value is indicated by a blue straight solid line. The left graph shows FF10 results and the other FF20 results. The $C_D$ values decrease as the wake plane location moves further downstream from the wing. For the FF10 case, $C_D$ values are approximately one-half the $C_{D_{surface}}$ values. For the FF20 case, $C_D$ values are approximately three quarters of the $C_{D_{surface}}$ values. There is a reversed tendency in $C_{D_h}$ behavior shown in Fig. 13. Then, an attempt was made to take the sum of $C_D$ and $C_{D_h}$. As shown in Fig. 17, $C_{D_h}$ and the sum were also graphed, indicated by a red circle and a purple triangle at each wake plane location, respectively. The sum values are constant along the vertical axis as well as nearly equal to the $C_{D_{surface}}$ value.
Fig. 17 $C_D$ via Eq. (2), $C_{Dh}$ via Eq. (6), and their sum at different wake plane locations compared to $C_{D_{\text{Surface}}}$ via Eq. (1). Below the graphs, each drag coefficient quantity at the wake plane of $x \approx 6.0$ is presented as a number in a table format. The FF10 results are shown in the left graph and table and the FF20 results on the right.

(a) FF10

| Quantity | Value |
|----------|-------|
| $C_{Dh}$ | 0.029685 |
| $C_D$   | 0.031115 |
| $C_D + C_{Dh}$ | 0.060800 |
| $C_{D_{\text{Surface}}}$ | 0.060834 |

(b) FF20

| Quantity | Value |
|----------|-------|
| $C_{Dh}$ | 0.014123 |
| $C_D$   | 0.046277 |
| $C_D + C_{Dh}$ | 0.060400 |
| $C_{D_{\text{Surface}}}$ | 0.060340 |

Fig. 18 Contour maps of the distribution of three flow variables presented in three pairs. Each pair consists of contour maps on the crossflow ($x = 6C$) planes of the FF10 and FF20 CFD simulation results. The first pair which lies on the left side shows the distribution of the integrand of Eq. (2), the primary drag source. The second pair in the middle shows that of Eq. (6), i.e. $\rho \Delta H$, the enthalpy drag source. The right pair shows that of the sum of both integrands.
There would be no rational background to take the sum at present. However, in fact, \( CD_h \) compensates for the difference between \( CD \) and \( CD_{\text{surface}} \). In the CFD results, a substantial amount of \( CD_h \) exists and the equation, \( CD_{\text{surface}} = CD + CD_h \), holds within 0.1% error. The ratio of \( CD_h \) decreases as the size of the computational mesh space increases. Consequently, \( CD_h \) has some meaning in computational analysis. From our limited experience regarding simulation using other CFD codes, it was also observed that \( CD_h \) compensates for the difference between \( CD \) and \( CD_{\text{surface}} \).

The flow phenomena of the CFD simulation results, which directly relate to the drag quantity, are shown in Fig. 18. The distribution of the integrand of Eq. (2), the primary drag expression for the wake integration method, is shown in the left side pair of the shorter and longer rectangles. The short one is a crossflow plane of the computational mesh space of FF10 and the longer is a crossflow plane of the entire computational mesh space of FF20. The distribution of the integrand of Eq. (6), that is \( \rho \omega \Delta H \), is shown in the middle pair. The distribution of the sum of the primary drag source and that of the total enthalpy are shown in the right pair of contour maps. In the right one, we obtain natural drag source distributions on the crossflow plane. The spurious drag sources recognized in regions near the far-field boundaries of the left and middle pairs of the contour maps cancel each other out; therefore, in the right pair, they completely vanish. The spurious drag sources supposedly originate from the far-field boundary conditions which do not match the real physics of the flows at those locations.

Incidentally, \( D \), the primary drag, was evaluated by the original far-field integration which is the root of Eq. (2). The equation for the original far-field integration is identical to that of Eq. (2) except for the integration area as shown in Eq. (8). The resultant value of \( D \) using Eq. (11) is equal to \( D_{\text{surface}} \) as follows:

\[
D = \int_{S_2 + S_C} \rho \mu (U_\infty - u) + (P_\infty - P) \, dS \tag{11}
\]

where the integration area of \( S_2 + S_C \) indicates the entire outer surface of a control volume around a wing (see Fig. 2).

This fact elucidates that if using the original integration equation, the relation \( D_{\text{surface}} \equiv D \) of Eq. (10) holds. This is exactly what momentum conservation theory states. Therefore, it was found that omitting the integration on the \( S_C \) surfaces results in the difference between \( D_{\text{surface}} \) and \( D \) via Eq. (2) and the difference can be compensated for by adding \( D_h \) to Eq. (6) on a wake plane, \( S_2 \).

5. Conclusions

Drag calculations were conducted for CFD simulation results to examine the performance of wake integration equations derived from a far-field drag concept. The objectives were to establish effective means to utilize far-field drag methods and wake flow phenomena. Four wake integration equations were used. The first (\( D \)) originated from the equation of the momentum conservation law itself, the second (\( D_h \)) was based on enthalpy variation, the third (\( D_p \)) was based on entropy variation, and the last (\( D_t \)) was for induced drag calculation. They were applied to flow-fields by CFD simulation around a rectangular wing using two types of mesh spaces, one smaller and the other larger. Drag values using the wake integration were compared to the near-field drag (\( D_{\text{surface}} \)). In addition, the distribution of the integrand of each wake integration equation was shown on a contour map. Each map clarified what physical property causes each drag component. Through this investigation several key points for present wake integration and pseudo total enthalpy generation were noted.

First, the equality of \( D_{\text{surface}} = D_p + D_t \) in Eq. (9) was confirmed within 1% error. It is the drag decomposition form of the wake integration method. As a whole, both the equality and the simulated wake phenomena were neither influenced by the size of the CFD mesh space nor the location of a wake plane over which the wake integration was completed. Precisely, the \( CD_p \) values slightly increase by several drag counts and the \( CD_t \) values slightly decrease as a wake plane location moves downstream where the mesh spacing coarsens. It was found that a wake plane should be located greater than 1.5 times the wing chord length behind the trailing edge and a wake plane should be within the range where the mesh spacing did not coarsen.

Second, the equality of \( D_{\text{surface}} = D \) in Eq. (10) was examined. It was confirmed when \( D \) was the result of the original far-field integration Eq. (11). However, it did not hold when \( D \) was the result of the wake integration Eq. (2) because the integration on \( S_C \), the control volume surface except the wake surface plane, in Eq. (11) was not negligible.
Then, the amount of the integration on $S_C$ was found to be restored with $D_h$ in Eq. (6), one of the wake integration equations. $D_h$ is the non-physical minus-signed enthalpy drag originating from spurious $\Delta H$, the total enthalpy variation associated with the error of a numerical scheme and boundary condition. It was also found that the location of far-field boundaries as well as the circulation related to the lift force of the wing impacted the generation of spurious $\Delta H$. The further the location of the far-field boundaries is from a wing, the less spurious $\Delta H$ is generated, and the less lift force, the less spurious $\Delta H$.

From the viewpoint of aiming for accurate CFD simulation to suppress non-physical phenomena, this implied that boundary conditions imposed on a far-field boundary should be carefully handled if its location is not sufficiently far from a wing.

In future work, a physical interpretation will be considered given the sum of $D$ of Eq. (2) and spurious $D_h$ were equivalent to $D_{surface}$.

**Acknowledgment**

This work was supported by Japan Society for the Promotion of Science Grants-in-Aid for Scientific Research Grant Number JP16K06884. We appreciate their support.

**References**

Giles, M. B. and Cummings, R. M., Wake integration for three-dimensional flow-field computations, Journal of Aircraft, Vol. 36, No. 2 (1999), pp. 357-365.

Hunt, D. L. and Giles, M. B., Wake integration for three-dimensional flow-field computations - Applications, Journal of Aircraft, Vol. 36, No. 2 (1999), pp. 366-373.

Kusunose, K., A wake integration method for airplane drag prediction, Tohoku University Press, ISBN4-86163-015-0 C3353(2005).

Matsushima, K., Shimizu, R. and, Takahashi, Y., Investigation on wake integration technique airplane drag prediction, Proceedings of 10th International Conference on Computational Fluid Dynamics (ICCFD10) (2018), Paper No. ICCFD10-297.

Meheut, M. and Baily, D., Drag-breakdown methods from wake measurements, AIAA Journal, Vol. 46, No. 4 (2008), pp. 847-862.

Mele, B., Ostieri, M., and Tognaccini, R., Aircraft lift and drag decomposition in transonic flows, Journal of Aircraft, Vol. 54, No. 5 (2017), pp. 1933-1944.

Shimizu, R., Matsushima, K. and Goshima, K., Investigation on numerical drag prediction for airplanes by wake integration. JAXA-SP-17-004, ISSN 1349-113X (2017), pp. 127-132 (in Japanese).

Ueno, M., Yamamoto, K., Tanaka, K. and Murayama, M., Far-field drag analysis of NASA common research model simulation, Journal of Aircraft, Vol. 50, No. 2 (2013), pp. 388-397.

Yamazaki, W., Matsushima, K. and Nakahashi, K., Drag prediction and decomposition based on CFD computations, JSME International Journal Series B., Vol. 48, No. 2 (2005), pp. 235-240.

Yamazaki, W., Matsushima, K. and Nakahashi, K., Drag prediction, decomposition and visualization in unstructured mesh CFD solver of TAS-code, International Journal for Numerical Methods in Fluids, Vol. 57, Issue 4 (2008), pp. 416-436.