

The Application of Rapid Aerodynamic Prediction Techniques to Supersonic Aircraft

Nathanael P. Matthews*, Romain Seppey, Simon A. Prince[†] and Zeeshan A. Rana[‡]
Cranfield University, Cranfield, England, MK43 0AL, United Kingdom

When considering new designs of supersonic aircraft, the ability to assess many different configurations with relative accuracy is of the utmost importance. Since the 1950s, simple component flows for axisymmetric bodies, wings and other lifting surfaces have been widely used in a technique known as the component build-up approach. Each component is separately modelled using methods designed for specific scenarios where their accuracy is high. The effect of each component on the others is then calculated and they are summed. This gives, typically, lift and pitching moment. This technique has remained relatively unchanged and, with the advent of higher order methods, the lower order methods have often been overlooked. This study uses the component build-up approach to predict lift and presents a new technique for drag acting upon a full aircraft. The aim of this paper is to demonstrate the accuracy and speed of these methods. The X-15 research aircraft was chosen as a test case due to its wealth of supersonic full-scale data. The presented methods predict lift to within 3.7% and drag within 6.3% of experimental data for a range of Mach numbers from 1.9 to 6. When comparing this low fidelity method with high fidelity steady Reynolds-averaged Navier-Stokes simulations, both computational data sets match well.

I. Nomenclature

C_D	=	Drag coefficient $\frac{D}{\frac{1}{2}\rho U_\infty^2 S}$
C_{DC}	=	Cross-flow drag coefficient
C_f	=	Coefficient of friction
C_N	=	Coefficient of normal force
C_p	=	Pressure coefficient $\frac{\frac{P}{P_\infty} - 1}{M_\infty^2 \frac{\gamma}{2}}$ for theory or $\frac{P - P_\infty}{\frac{1}{2}\rho U_\infty^2}$ for experiments
D	=	Diameter of sphere [m]
M	=	Mach number
IPO	=	In presence of
r	=	Radius of conical frustum
Re	=	Reynolds number based on length of body
S	=	Reference area for coefficients [m ²]
U	=	Velocity [m s ⁻¹]
α	=	Angle of attack [°]
β	=	$\sqrt{M_\infty^2 - 1}$
γ	=	Ratio of specific heats
δ	=	Flow inclination angle [°]
η	=	Drag correction factor for finite length
ρ	=	Density [kg m ⁻³]
Φ	=	Circumferential angle, $\Phi = 0^\circ$ in windward plane, $\Phi = 180^\circ$ in leeward plane [°]

*Research Assistant in High Speed Aerodynamics, Centre for Aeronautics, Young Professional AIAA.

[†]Professor of Aerodynamics, Centre for Aeronautics, Senior AIAA Member.

[‡]Associate Professor in Mechanical Engineering, Department of Mechanical Engineering, Prince Mohammad Bin Fahd University, Al-Khobar, KSA, Member AIAA.

Subscripts

b	=	Base conditions
c	=	Chord length
l	=	Total length of body
w	=	Wall conditions
∞	=	Free stream conditions
0	=	Total conditions

II. Introduction

Low order modelling simplifies the aerodynamic problem at hand, accounting for only what is necessary for a specific application. This directly contrasts conventional computational fluid dynamics (CFD) where, generally, the entire flow field is modelled in an attempt to capture everything to a relevant degree. As such, low order methods are simpler to understand and quicker to compute but also much more constrained than higher order methods such as Navier-Stokes or Euler. Low order methods do have great benefits such as the rapid exploration of design spaces, opening up the potential for completing multi-disciplinary studies at an increased speed. Low order modelling has a rich history and many of these techniques have been instrumental in the designs of groundbreaking aircraft. There is, however, a trend towards using high order CFD where it is not needed, wasting time, power and resources. It is important that the capability and knowledge of low order methods are not lost, as they can be invaluable when applied in the right context. These factors are especially applicable to supersonic flow due to the reduced influence of the body upstream. Low order computations can easily be incorporated into other systems and are incredibly powerful when their potential is realised. Additionally, with modern computing there is another layer that can be explored: where some low order methods were deemed too computationally costly or complex, they can now be computed in a fraction of the time opening new avenues and combinations of methods. Modern improvements in the understanding of fluid principles and the discovery of new relationships between fluids and structures have typically only been applied to higher order CFD. This study explores the application of low order methods to this field, introducing a new drag model and utilising a well-known lift model to create lift and drag polars of a full-scale aircraft. These results can then be compared to a test case to determine their effectiveness.

This method utilises a component build-up approach, with the aircraft being split into several simplistic components such as the body, wing, tail and fin. Each are modelled separately and then combined. One issue with this is that the lift from one component will directly impact the lift of another. For example, the low pressure region created by the wings will bleed onto the body and so increase its lift contribution slightly, or the wing tip vortices may impinge on the tail above or below and severely affect the tails performance. To take these ‘in presence of’ (IPO) effects into account, a method developed by Pitts, Nielsen and Kaattari [1] known as the PNK method is utilised. This is a series of calculations to determine the effect of one component on each other component. These modified component lifts are then summed to create a total lift variable. The drag of a component body, however, does not have such a method associated with it and so one is introduced here. The wave drag and pressure drag acting on the body is calculated using the second order shock-expansion theory (SOSE) and its viscous drag calculated by a combination of a DeJarnette [2] method and one from Kelly [3]. For the lifting surfaces, the wave and pressure drags are calculated in an inviscid panel code. The base drag is calculated for the body using a relationship developed by Gabeaud [4] and on the fin using simple shock-expansion calculations. The skin friction is approximated using a one equation coefficient, and excrescence drag is taken to be a fraction of all the viscous drags summed together. As the drag contributions of all the above are relatively independent there is no need to calculate an equivalent IPO set of drag coefficients. Instead, they can simply be summed to create the total drag acting on the body.

In order to validate this method, a suitable test case is needed. There are few supersonic aircraft with any flight test data available to the public, with the exception of the X-15. The X-15 is a research aircraft developed by NASA and the USAF to explore high-Mach flow regimes and has a wealth of available flight test data. All data above Mach 1.8 is selected for comparison. The X-15 has been re-constructed in CAD for this project based on the technical drawings and reports that detail the aircraft’s geometry.

III. Methodology

A. Fuselage

Several options exist to model an axisymmetric fuselage at supersonic speeds, for example, slender body theory, the method of characteristics, or Newtonian impact theory. Second order shock-expansion (SOSE) was chosen as the preferred method in this case due to its higher order and greater accuracy than most, while still being a rapid method. SOSE divides the body into a cone and a series of conical frustums all tangent to the body as shown in Fig. 1. The initial cone is solved using Taylor-Maccoll conical flow theory so that conditions at position 1 in Fig. 2 are known. Then at each 'corner' a Prandtl-Meyer type expansion calculation can determine the flow conditions at position 2, followed by a calculation of a second order pressure drop across the frustum such that flow conditions at position 3 can be calculated. This process is then repeated for each frustum until the end of the body is reached. The results of such a calculation

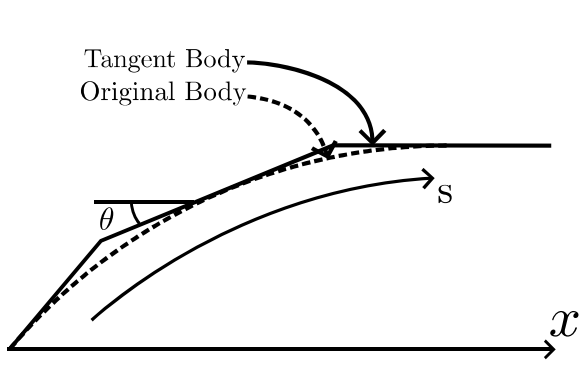


Fig. 1 Approximate geometry used in SOSE; continuous ogive converted into conical frustums

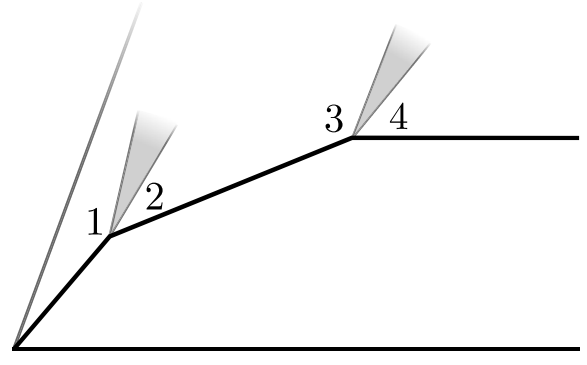


Fig. 2 Shocks and expansion waves present on a tangent body

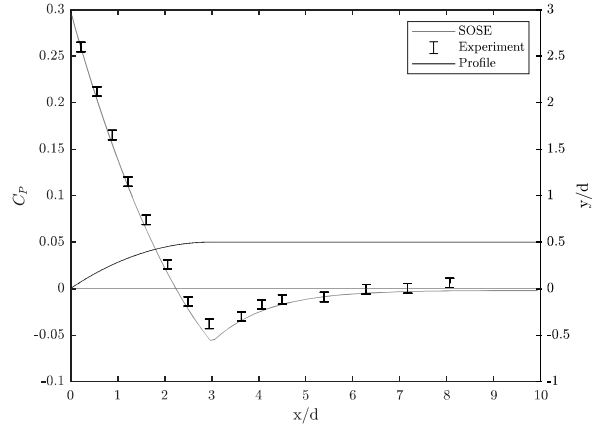


Fig. 3 Pressure distribution along a cylindrical fuselage with tangent ogive fore-body $M = 1.98$.

can be validated using readily available data sets, for example in [5] where an experimental data set of a tangent ogive cylindrical fore-body is presented at a Mach number of 1.98. The pressure distribution as predicted by SOSE compared to this data set can be seen in Fig. 3 where there is a good agreement between the predicted theory and experimental dataset. One limitation of SOSE is that it can only predict the pressure on a body at no incidence. Due to this limitation, DeJarnette developed a simple one equation model to predict the difference in pressure experienced around the body [2].

$$\Delta C_p = \sin 2\alpha \sin 2\delta \cos \Phi + \sin^2 \alpha \cos^2 \delta \left[\left(2 - \frac{1}{\beta} \right) \left(1 - \tan^2 \delta \right) - \left(2 + \frac{2}{\beta} \right) \sin^2 \Phi \right] \quad (1)$$

Equation (1) can then be added to the SOSE result at each position along the body and calculated at each circumferential angle. Again, using the tangent ogive at Mach 1.98, the circumferential pressure at $\alpha = 5^\circ$ can be plotted and compared to the experiment, as shown in Fig. 4. This demonstrates that, for small α , the circumferential pressure distribution can be accurately calculated for an axisymmetric body. As α increases on this type of body, a smooth surface separation occurs as a rolling up of the boundary layer into two lee side vortices which generate significant non-linear lift on the body. This is not predicted by SOSE or the DeJarnette alteration, and the separation and viscous effects present in these higher α s is demonstrated in Fig. 5. Here, viscous effects and separation are visible in the experiment but not predicted in the theory. Instead another method is added which likens the body to an infinite span cylinder and calculates the local normal force generated by subjecting a cylinder to the same cross-flow that is experienced by the body [3]. Utilising the cross-flow drag coefficient of a cylinder and a correction factor for a finite length cylinder, Eq. (2) is an approximation that can be calculated along the body and integrated to find the total normal force.

With all these methods in place, the total lift of a sharp axisymmetric fore-body and fuselage can be found. Figure 6 shows the combination of SOSE, DeJarnette's alteration and Kelly's cross-flow. From this graph and the surrounding literature we can conclude that this method is entirely adequate for aerodynamic force prediction on sharp axisymmetric bodies.

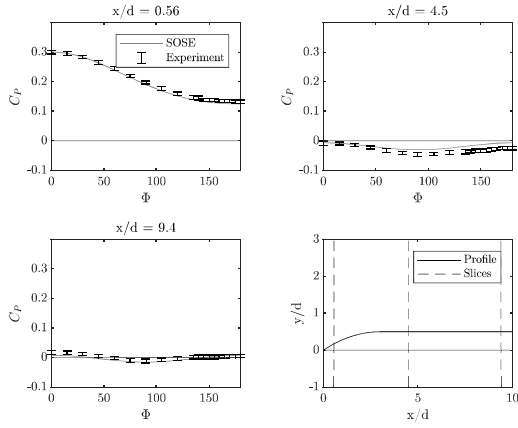


Fig. 4 Circumferential pressure distribution at three stations along the body at $\alpha = 5^\circ$ $M = 1.98$.

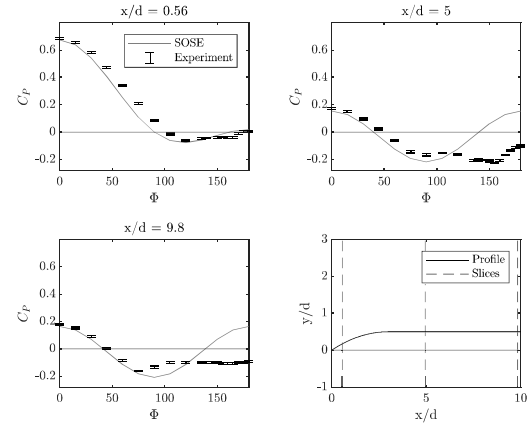


Fig. 5 Circumferential pressure distribution at three stations along the body at $\alpha = 20^\circ$ $M = 1.98$.

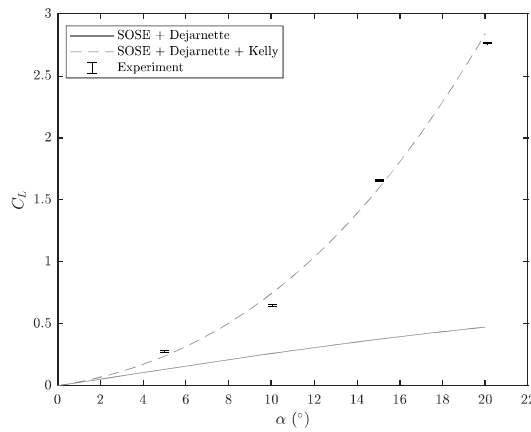


Fig. 6 Lift force for a range of angles of attack $M = 1.98$.

$$\Delta C_N = \frac{2 \sin^2 \alpha}{S} \int_0^l r \eta C_{Dc} dx \quad (2)$$

B. Lifting surfaces

The wings, tail plane, and fairing were all modelled using an early panel code developed by Boeing as a precursor to their PANAIR program [6]. This program has no name and is often only referred to as the NASA AMES WingBody panel code. Panel methods solve the Prandtl-Glauert (PG) equation for linear, inviscid, irrotational flow at either subsonic or supersonic free stream velocities. The PG equation is derived from the Navier-Stokes equations through a series of assumptions which dictate the type of flow that can be modelled. First, viscous and heat transfer terms are neglected, then, the flow is assumed to be irrotational and finally all non-linear terms are disregarded, leading to the limitation of inviscid, irrotational, linear flow. These assumptions mean that separation and viscous effects, such as skin friction, are not predicted, but lift induced drag and wave drag are included. These assumptions create an analytical solution made up of source, doublet and vorticity singularities. A panel method takes these singularities and superimposes them over an approximated version of some geometry which is made up of small, traditionally quadrilateral panels. The panel code used in this instance is a 'low order' panel code, meaning it contains constant strength singularities on each panel and all the panels are flat. There do, however, exist more advanced panel codes [7] with more complex interactions which are often described as being 'higher order'. The PG equation is written as:

$$\nabla^2 = (1 - M_\infty^2)\phi_{xx} + \phi_{yy} + \phi_{zz}$$

where ϕ is the perturbation velocity potential in a certain axial direction. Occasionally this equation is multiplied by -1 for supersonic flow, creating hyperbolic equations. This means that disturbances have a specific region downstream they can influence. Essentially, the PG equation creates a large system of simultaneous equations all to be solved using aerodynamic influence coefficients [AIC] which are determined by the normal component of the velocities at a number of control points.

The basic process is summarised as follows [8]:

- 1) Create geometry of panels from specified shape
- 2) Use panels to create source-doublet distribution in terms of singularity parameters $\{\lambda\}$, whose values are to be determined
- 3) From influence-coefficient expressions for the potential and/or velocity at each of N panel control points owing to the source-doublet distribution of any panel; for supersonic flow the zones of silence must be accounted for
- 4) Use the influence coefficient expressions to enforce boundary conditions at N points, giving the NxN system of equations

$$[AIC] \{\lambda\} = \{b\}$$

- 5) Solve the singularity parameters $\{\lambda\}$ using the known set of boundary conditions $\{b\}$, then compute the potential and velocity at any point of interest
- 6) Knowing the velocity, compute the panel pressure distributions
- 7) Integrate pressure distributions to obtain forces and moments

C. Combination

The component build-up approach is based on the concept of modelling individual parts of a configuration such as the body, wing and tail separately, then modelling the interference each component has on the other and summing the parts. Each wing or wing type component will cause lift to be transmitted to the body via the pressure differences drifting over the body downstream of the leading edge of the wing. The wing vortices near the tips of a forward wing will continue to be evident downstream and if another wing type component is in the path of this vortex then the implication can be a positive or negative lift being transmitted to the downstream component. The primary source for this method is [1], which contains extensive test cases and various wing body tail type configurations. This method, called the Pitts Nielsen Kaattari (PNK) method has derivations based on slender body theory and potential flow theory, with separate equations for subsonic and supersonic cases. Transonic flows and configurations are not addressed in PNK due to the complexity of the flow and the inadequacy of low order methods when addressing the transonic regime. The summary of lift components of wing-body configurations detailed by PNK is as follows:

Lift on. . .

- 1) body nose
- 2) wing in presence of body
- 3) body due to wing
- 4) tail in presence of body (neglecting wing vortices)
- 5) body due to tail (neglecting wing vortices)

- 6) tail section due to wing vortices
- 7) after-body due to wing vortices

D. Drag

The method as described above is tried and proven for lift, however, drag has no such silver bullet. A new method has been developed by the authors in order to attempt bridging this gap. While the method remains simple, the effectiveness will be demonstrated. Drag has so far been calculated through pressure on the body and waves on the bow. This is not the full story, as there are other effects such as skin friction, drag on non-lifting surfaces and excrescence drag.

First, skin friction. This component is highly dependant on whether the composition of the boundary layer is laminar or turbulent and the location of transition. This is not, however, information which can be found for arbitrary shapes without highly involved testing. With this in mind, the approximate friction coefficient equation based on experiments from [9] will be used.

$$C_f = \frac{0.02667}{Re_c^{0.139}}$$

Second, drag on non-lifting surfaces. As an example, take the vertical fins which do not contribute to the lifting of the aircraft and so far have been left out of the low fidelity model. The fins do contribute to drag. This is a simple calculation as the fins are 10° wedges and so can be modelled using 2D shock-expansion theory [10]. To do this, first the pressure coefficient must be calculated on the forward facing side which requires the pressure gradient across the oblique shock wave at the front. For a 10° wedge the flow deflection will be 5° and the pressure difference calculated as follows: Check whether shock is weak or strong, i.e. $\theta < \theta_{max}$

$$\beta_{max} = 0.5 * \cos^{-1} \left[\frac{(\gamma - 1) M_\infty^2 + 4 - \sqrt{16(\gamma + 1) + 8(\gamma^2 - 1) M_\infty^2 + (\gamma + 1)^2 M_\infty^4}}{2\gamma M_\infty^2} \right]$$

$$\theta_{max} = \tan^{-1} \left\{ \frac{2 \cot(\beta_{max}) [M_\infty^2 \sin^2(\beta_{max} - 1)]}{M_\infty^2 [\gamma + \cos(2\beta_{max})] + 2} \right\}$$

For a simple solution $\theta < \theta_{max}$, therefore, weak shock ($k=1$)

$$\lambda = \sqrt{(M_\infty^2 - 1)^2 - 3 \left(1 + \frac{\gamma - 1}{2} M_\infty^2 \right) \left(1 + \frac{\gamma + 1}{2} M_\infty^2 \right) \tan^2(\theta)}$$

$$\chi = \frac{(M_\infty^2 - 1)^3 - 9 \left(1 + \frac{\gamma - 1}{2} M_\infty^2 \right) \left(1 + \frac{\gamma - 1}{2} M_\infty^2 + \frac{\gamma + 1}{4} M_\infty^4 \right) \tan^2(\theta)}{\lambda^3}$$

$$\beta = \tan^{-1} \left\{ \frac{M_\infty^2 - 1 + 2\lambda \cos \left[\frac{4\pi k + \cos^{-1}(\chi)}{3} \right]}{3 \left[1 + \frac{\gamma - 1}{2} M_\infty^2 \right] \tan[\theta]} \right\}$$

Here λ and χ are only intermediate calculations. Now calculate pressure gradient using shock angle β

$$\frac{P}{P_\infty} = \frac{2\gamma}{\gamma + 1} M_\infty^2 \sin^2(\beta) - \frac{\gamma - 1}{\gamma + 1}$$

This pressure gradient can be used to calculate pressure coefficient

$$C_P = \frac{\frac{P}{P_\infty} - 1}{M_\infty^2 \frac{\gamma}{2}}$$

Use this to calculate drag coefficient

$$C_D = \frac{C_P A}{S}$$

Also include the back surface where $P \approx 0$.

Third, the pressure acting on the base of the body must be taken into account. A common approximation is to set

$P \approx 0$, however, for the body with a boat tail, a better method has been provided by [4] showing that the base pressure coefficient can be calculated using:

$$C_{pb} = \frac{2}{\gamma M^2} \frac{2\gamma M^2 - (\gamma - 1)}{\gamma + 1} \left(\frac{2}{\gamma + 1} \right)^{1.4} \frac{1}{M^{2.8}} - \frac{2}{\gamma M^2}$$

This is then scaled by the ratio of base area to fuselage cross sectional area to find the axial force coefficient for the base taking into account boat tailing:

$$C_{Ab} = -C_{pb} \left(\frac{S_b}{S} \right)$$

Lastly, excrescence drag. This component of drag is the most difficult to predict as it depends on the real aircraft and will include drag on instruments such as pitot probes, small shocks on rivets, any extra resistance encountered through gaps in sheet metal or control surfaces, the list goes on. Based on research [11] [12] [13] and an understanding of supersonic flow characteristics, a value of half the sum of all viscous drag is added to account for this unknown. This is the best estimate for all supersonic geometries when intricate detail on each and every component and rivet is unknown. Techniques do exist, published by ESDU which are helpful predictors of excrescence drag for transonic aircraft [14]. Unlike with lift, there is no traditional method for combining drag values and they are relatively independent, therefore, they are all added together without modification.

IV. Test Case

To assess this methodology, a full-scale aircraft with flight test data is required. The X-15 research aircraft is a perfect fit, being the only full-scale supersonic aircraft with enough publicly available lift and drag data at supersonic speeds. Tests were conducted with a Reynolds number (Re_l) range of 9.8×10^6 to 140×10^6 , and a Mach number range of 0.6 to 6 [15]. One of the greatest attributes of the X-15 is that, unlike many other research aircraft created at that time, it was funded by the USAF but was never intended to be a combat vehicle. This difference in attitude meant that great swathes of data could be collected, published and, in time, made available to the public. The X-15 saw three iterations of aircraft design and two were built and tested. The propulsion system of the X-15 was the first throttleable rocket motor and had a burn time of just over 90s. After burnout the aircraft would return, slamming back into the atmosphere and shedding its velocity as heat (seen in Fig. 7). This heat was an incredible challenge and new materials were specially designed so this aircraft could cope with such extreme conditions. The X-15 paved the way for the space shuttle and pushed the boundaries of possibility.

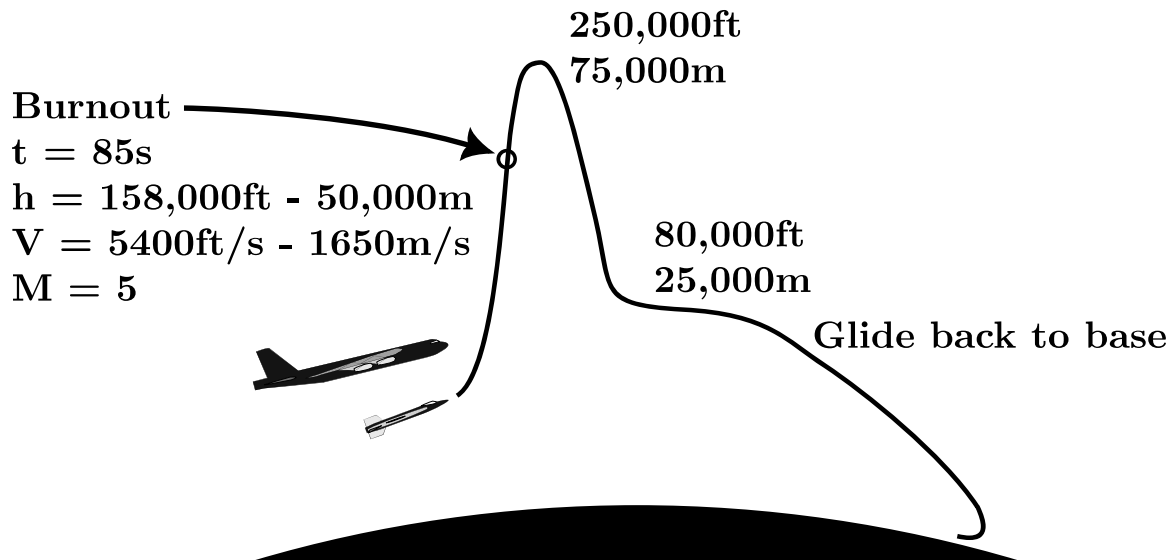


Fig. 7 Typical mission of the X-15 research aircraft, picturing B-52 mother-ship and X-15 with flight path [16]

A. Geometry

The X-15 is composed of six main sections, which are shown in Fig. 9. The geometry and associated CAD model have been made using references from the NASA archives detailing the X-15 and its geometry [17] [18] [19] [20] [21] [22] [23] [24] [25] [26].

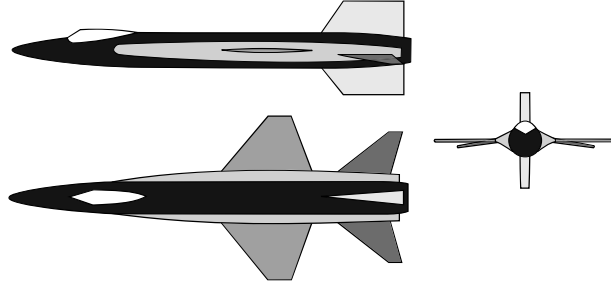


Fig. 8 Three-view of the X-15

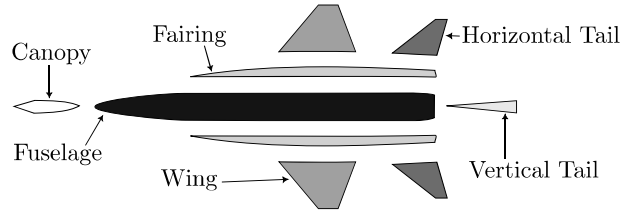


Fig. 9 Top down ‘exploded-view’ of X-15 with labelled sections

While much is recorded about the X-15 and its geometry, some aspects of its design seem to have been lost to time. Regarding the wing profile, for example, all references state that the X-15 sports a modified NACA 66005 profile on the wing and tail, however, details of the modification are unknown and not listed in any of the explored references. As such the CAD for CFD and low fidelity simulations take an unmodified NACA 66005 profile as the profile of the wing and tail. There is also a lack of detail concerning the shape of the canopy and its exact position on the aircraft. The CAD model canopy has been constructed using three-view diagrams and photographs. The impact of the exact position of the canopy is not likely to be of any consideration, and the low fidelity model does not model the canopy at all.

Canopy	- A small raised section of the fuselage with two front facing windows and a sloping rear
Fairing	- An extension of the main cylindrical fuselage which carries feed lines and purge tanks
Fuselage	- Axisymmetric cylinder with tangent ogival nose, a small rounded point, a slight boat tail toward the base and a total length of $15.5m$
Wing	- Tapered wings with a span of $6.8m$, a modified NACA 66005 profile, root chord of $4.5m$ and a taper ratio of 0.2
Horizontal Tail	- Tapered wings with a span of $5.5m$, a modified NACA 66005 profile, root chord of $3.1m$, a taper ratio of 0.2 and a dihedral of -15°
Vertical Tail	- Blunt based wedges with a half angle of 5° and a span of $2.7m$

B. CFD

While there is plenty of data pertaining to the full configuration of the X-15, there is no experimental force data for individual components of the aircraft. To validate the forces generated by each component, a CFD model has been

computed. The grid consists of two sections: an inner box which starts 0.1 body length in front of the model and extends to 10 body lengths behind the nose and 5 body lengths above and below. An outer box which is 10 times the size of the inner box in each respective direction, shown in Fig. 10. The nose section of the mesh is highly refined in order to capture the bow shock wave which is at its strongest in that region Fig. 11. A grid convergence study has shown that the mesh is converged and adequate for capturing the forces acting on the X-15. The cell count of the chosen mesh is 54 million, it was solved on STAR-CCM+ [27] using steady RANS coupled flow, MUSCL 3rd order with AUSM+ FVS, Spalart-Allmaras turbulence model and a real gas equilibrium air physics model.

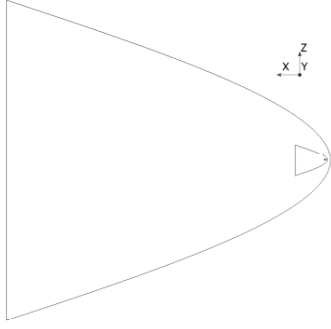


Fig. 10 RANS CFD fluid domain.

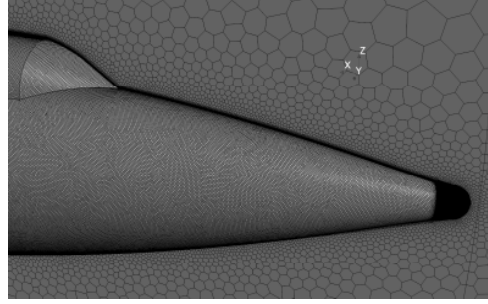


Fig. 11 RANS CFD Mesh close up at the nose.

V. Results

Figure 12, which shows the total lift coefficient at Mach 4.3, displays good agreement between all data above 4° and both computational data sets are consistent with each other at all α . The discrepancy at low α is likely due to the calculation of the experimental results. As the force on the aircraft decreases at low α the errors in the accelerometers are larger proportionally compared to the recorded acceleration data, therefore, the error in the experimental dataset increases at low α . This is the reason for the computational data sets remaining within the experimental error region despite the deviation from the recorded data points.

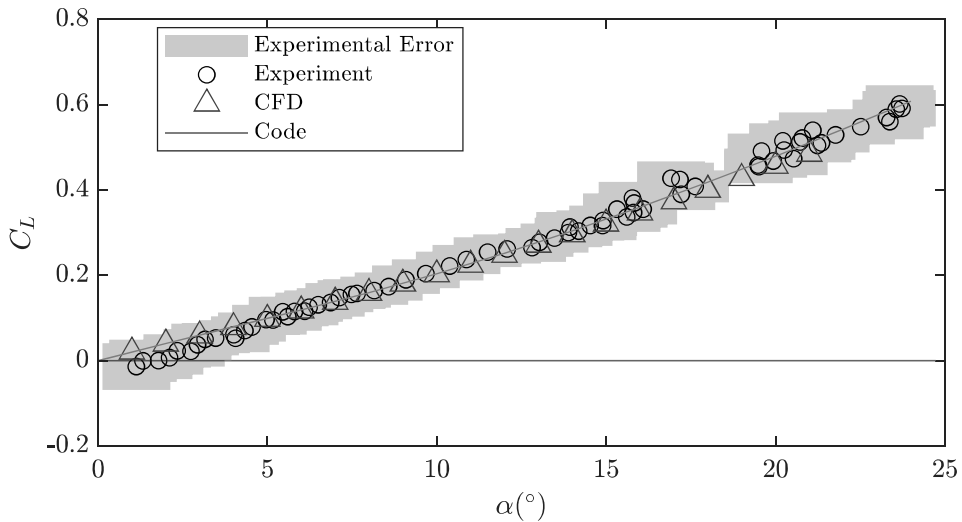


Fig. 12 Total lift coefficient comparison, $M = 4.3$, CFD $Re_l = 140 \times 10^6$

No component lift data exists for experimental data. Although, considering the close match in experimental and CFD datasets, a comparison can be made of each component using CFD instead. By splitting the mesh into separate parts, force data can be extracted from the simulations for each component. As the low fidelity computational method calculates by component already, a direct comparison can be made. This is displayed in Fig. 13. As can be seen in this figure, there is good correlation between the CFD and component build-up method. Each component is accurately represented displaying a viable solution for lift.

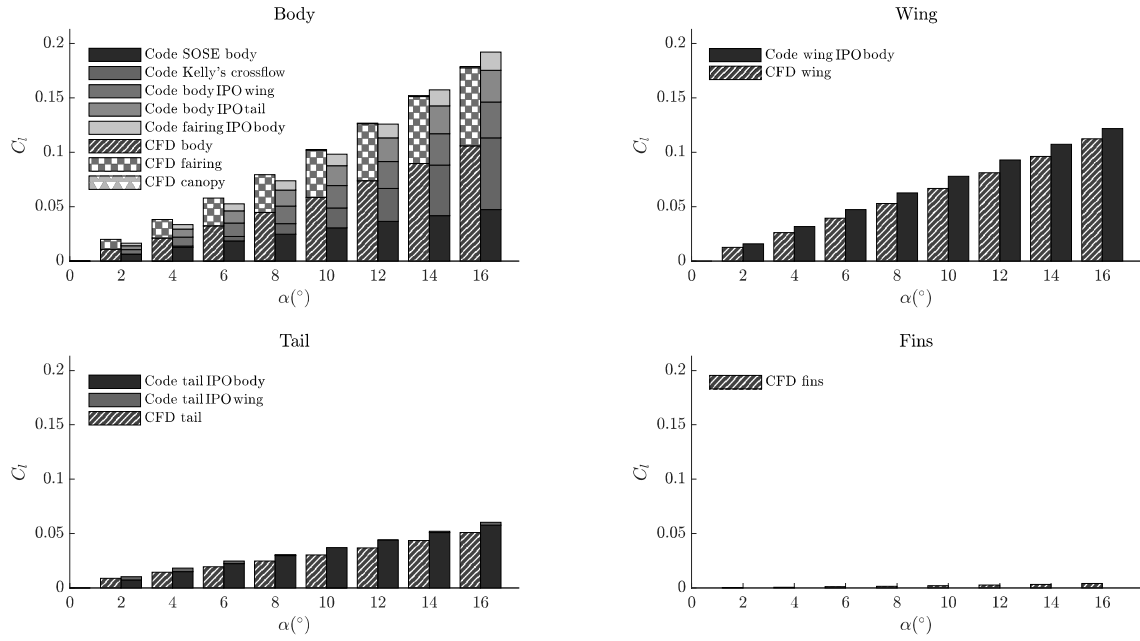


Fig. 13 Component lift coefficient (IPO = in presence of), $M = 4.3$, $CFD Re_l = 140 \times 10^6$

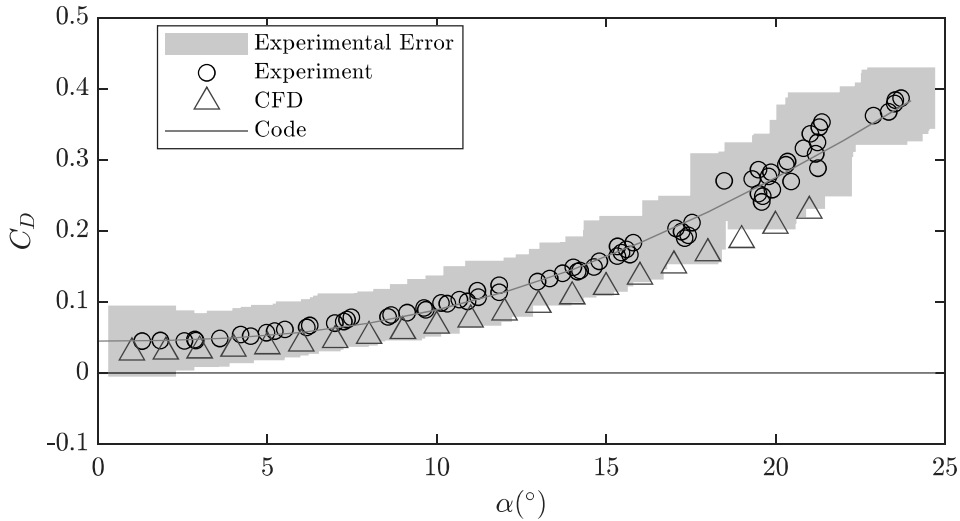


Fig. 14 Total drag coefficient comparison, $M = 4.3$, $CFD Re_l = 140 \times 10^6$

Figure 14 shows a drag polar at Mach 4.3. The drag prediction of the low fidelity method, with the unmodified addition of each component, matches the experimental data. This matching is consistent across all the tested Mach

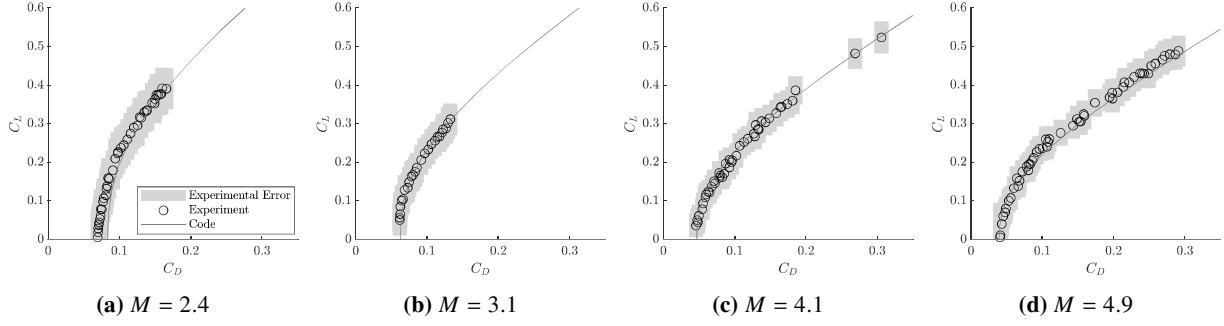


Fig. 15 C_L vs C_D graphs at several Mach numbers

numbers, a selection of which are displayed in Figs. 15a to 15d. Accuracy increases with Mach number as shown in Table 3. The drag prediction is suited to this specific case, however, as the body is slender and deviating from this body shape is likely to result in a less accurate representation of reality. The low fidelity model predicts the experimental data with an average coefficient of determination (R^2) of 0.963 for C_L and 0.937 for C_D when datasets with less than 20 data points are disregarded. This is an accuracy of 96.3% and 93.7% respectively, as shown in Table 3.

Mach Number M	No. Data Points n	Mean Square Error $\sigma^2 (\times 10^{-3})$		Coefficient of Determination R^2	
		C_L	C_D	C_L	C_D
1.9	28	0.766	0.553	0.953	0.262
2.4	38	1.664	0.0723	0.896	0.934
2.6	32	1.513	0.0244	0.858	0.953
3.1	26	0.196	0.00414	0.973	0.993
3.3	55	0.922	0.0799	0.949	0.972
3.5	36	0.594	0.0516	0.977	0.990
3.7	49	0.655	0.0895	0.977	0.988
3.9	37	0.543	0.0693	0.978	0.991
4.1	35	0.884	0.0301	0.947	0.991
4.3	71	0.275	0.176	0.992	0.984
4.5	31	0.257	0.0700	0.98	0.987
4.7	45	0.509	0.0901	0.974	0.988
4.9	30	0.453	0.0950	0.984	0.986
5.1	26	0.337	0.0963	0.985	0.975
5.3	32	0.592	0.128	0.986	0.993
5.5	37	0.874	0.241	0.974	0.978
5.7	40	0.169	0.0837	0.987	0.956
5.9	18	0.442	0.0424	0.606	-0.964
6	6	1.011	0.00253	-4.098	-0.0109

Table 3 Table displaying statistical analysis through coefficient of determination (R^2) of the model when applied to experimental data

VI. Conclusion

A low fidelity method for the aerodynamic forces acting upon a supersonic aircraft is presented and compared to experimental and computational datasets. A test case of the X-15 is chosen as it has a wealth of experimental data, has been rigorously documented, and is available to the public without restriction. Compared to experimental and high fidelity CFD the low fidelity method matches well. The method has an average accuracy within $\pm 3.7\%$ for lift and $\pm 6.3\%$ for drag when compared to the full configuration flight test data for supersonic Mach numbers between 1.9 and 5.7. This covers the entire range of statistically usable and relevant experimental data. This method is many orders of magnitude faster than high fidelity CFD and does not need specialist equipment or software licences. Its accuracy is also well within what would be required for design space exploration. The presented drag method fits well with the current test case. With continuing work on the topic, more complex drag characteristics could be captured and integrated into the system. This specific technique is limited in its ability to model many configurations, but the possible uses of low fidelity modelling has by no means been exhausted. To reduce cost, these methods could be implemented into the design process prior to proceeding with time- and resource-costly high fidelity studies. It is imperative that this methodology is not swept away in the rising tide of ‘RANS solves all’.

References

- [1] Pitts, W. C., Nielsen, J. N., and Kaattari, G. E., “Lift and Center of Pressure of Wing-Body-Tail Combinations at Subsonic, Transonic, and Supersonic Speeds,” TR 1307, NACA, 1957.
- [2] DeJarnette, F. R., Ford, C. P., and Young, D. E., “A new method for calculating surface pressures on bodies at an angle of attack in supersonic flow,” *12th Fluid and Plasma Dynamics Conference*, AIAA, 1979. <https://doi.org/10.2514/6.1979-1552>, URL <https://arc.aiaa.org/doi/abs/10.2514/6.1979-1552>.
- [3] Kelly, H. R., “The Estimation of Normal-Force, Drag, and Pitching-Moment Coefficients for Blunt-Based Bodies of Revolution at Large Angles of Attack,” *Journal of Aeronautical Sciences*, Vol. 21, No. 8, 1954.
- [4] Gabeaud, A., “Base pressures at supersonic velocities,” *Journal of the Aeronautical Sciences*, 1950, pp. 525–526.
- [5] Perkins, E. W., and Jorgensen, L. H., “Comparison of experimental and theoretical normal-force distributions (including Reynolds number effects) on an ogive-cylinder body at Mach number 1.98,” TN 3716, NACA, 1956.
- [6] Woodward, F. A., Tinoco, E. N., and Larsen, J. W., “Analysis and Design of Supersonic Wing-Body Combinations, Including Flow Properties in the Near Field. Part I – Theory and Application,” Contractor Report 73106, NASA, 1967.
- [7] Saaris, G. R., Tinoco, E. N., Lee, J. L., and Rubbert, P. E., “A5021 User’s manual PAN-AIR technology program for solving problems of potential flow about arbitrary configurations,” Tech. Rep. D6-54703, Boeing, Arlington, 1992.
- [8] Erickson, L. L., “Panel methods: An introduction,” techreport TP-2995, NASA, Washington D.C., 1990.
- [9] Schlichting, H., and Gersten, K., *Boundary-Layer Theory*, 9th ed., Springer, 2017. <https://doi.org/10.1007/978-3-662-52919-5>.
- [10] Field, M., “Equations, Tables, and Charts for Compressible Flow,” Tech. rep., NASA, Washington, D.C., 1953.
- [11] Pfeiffer, N. J., and Lednicer, D., “A study of airplane excrescence drag,” *14th AIAA Aviation Technology, Integration, and Operations Conference*, 2014, p. 2866.
- [12] Young, A. D., Paterson, J. H., and AGARD., “Aircraft excrescence drag,” AGARD, 1981.
- [13] Kundu, A., Morris, W., Raghunathan, S., Kundu, A., Morris, W., and Raghunathan, S., “CFD verification of excrescence drag,” *35th Aerospace Sciences Meeting and Exhibit*, 1997, p. 354.
- [14] ESDU, “Excrescence drag levels on aircraft,” Tech. Rep. 94044, Accuris, London, 2007.
- [15] Salzman, E. J., and Garringer, D. J., “Summary of full-scale lift and drag characteristics of the X-15 airplane,” TN D-3343, NASA, 1966.
- [16] Dana, W. H., “A history of the X-15 program,” *NASA Flight Research Center Edwards*, 1987.
- [17] Osborne, R. S., and Stafford, V. C., “Basic pressure measurements on 0.0667-scale model of north american x-15 research aircraft at transonic speeds,” Tech. Rep. TM-X344, NASA, Washington, D.C., 1962.

- [18] Banas, R. P., "Comparison of Measured and Calculated Turbulent Heat Transfer in a Uniform and Nonuniform Flow Field on the X-15 Upper Vertical Fin at Mach Numbers of 4.2 and 5.3," Tech. Rep. TM-X1136, NASA, Washington, D.C., 1965.
- [19] Pyle, J. S., "Flight measured wing surface pressures and loads for the X-15 airplane at Mach numbers from 1.2 to 6.0," Tech. Rep. TN-D-2602, NASA, Washington, D.C., 1965.
- [20] Pyle, J., "Flight pressure distributions on the vertical stabilizers and speed brakes of the x-15 airplane at mach numbers from 1 to 6," Tech. Rep. TN-D-3048, NASA, Washington, D.C., 1965.
- [21] Franklin, A. E., and Lust, R. M., "Investigation of the Aerodynamic Characteristics of a 0.067-Scale Model of the X-15 Airplane (Configuration 3) at Mach Numbers of 2.29, 2.98, and 4.65," Tech. Rep. TM-X-38, NASA, Washington, D.C., 1959.
- [22] Finch, T. W., and Matranga, G. J., "Launch, low-speed and landing characteristics determined from the first flight of the North American X-15 research airplane," Tech. Rep. TM-X-195, NASA, Washington, D.C., 1959.
- [23] Palitz, M., "Measured and calculated flow conditions on the forward fuselage of the X-15 airplane and model at Mach numbers from 3.0 to 8.0," Tech. Rep. TN-D-3447, NASA, Washington, D.C., 1966.
- [24] Saltzman, E. J., "Preliminary full-scale power-off drag of the X-15 airplane for Mach numbers from 0.7 to 3.1," Tech. Rep. TM-X-430, NASA, Washington, D.C., 1961.
- [25] Hodge, L., and Burbank, P., "Pressure distribution of a 0.067 scale model of the X-15 airplane for an angle of attack range of 0 to 28 degrees at Mach numbers of 2.3, 2.88 and 4.65," Tech. Rep. TM-X-275, NASA, Washington, D.C., 1960.
- [26] Saltzman, E., and Garringer, D., "Summary of full-scale lift and drag characteristics of the X-15 airplane," Tech. Rep. TN-D-3343, NASA, Washington, D.C., 1966.
- [27] Siemens Digital Industries Software, "Simcenter STAR-CCM+, version 2024.6," , 2024.



DSPACE

<https://dspace.org/>

The application of rapid aerodynamic prediction techniques to supersonic aircraft

Matthews, Nathanael P.; Prince, Simon A.; Seppey, Romain; Rana, Zeeshan A.

2025-01-06

American Institute of Aeronautics and Astronautics (AIAA)

<https://doi.org/10.2514/6.2025-2831>

Matthews NP, Prince SA, Seppey R, Rana ZA. (2025) The application of rapid aerodynamic prediction techniques to supersonic aircraft. In: AIAA SCITECH 2025 Forum, 6-10 January 2025, Orlando, FL, USA. Paper number AIAA 2025-2831