Aerodynamic Drag Reduction on a Blunt Body Using Aero-Disks and a Spike

Prachi Patel[1],M. Akshaya Varma[2] ,D. Anitha[3]

[1] Undergraduate Student, Department of Aeronautical Engineering
[2] Undergraduate Student, Department of Aeronautical Engineering
[3] Assistant Professor, Department of Aeronautical Engineering
Institute of Aeronautical Engineering, Dundigal, Hyderabad-500043
[1]prachipatel.1915@gmail.com, [2]akshaya.varma.m@gmail.com, [3]d.anitha@iare.ac.in

Abstract— **Aerodynamic drag reduction is crucial for improving the performance of blunt-bodied vehicles. This study investigates the effect of aero-disks, combined with a spike, on the drag characteristics of a blunt body. Simulations were performed for the baseline geometry and configurations with one and two aero-disks. The computational approach was validated by comparison with data from a previously published study, showing good agreement and confirming the reliability of the results. The introduction of a single aero-disk reduced the drag coefficient from 0.45 (baseline) to 0.30, a 33% decrease, while the addition of a second aero-disk further lowered it to 0.16, achieving an overall 64% reduction. These findings demonstrate that increasing the number of aero-disks significantly modifies the flow field around the blunt body, resulting in enhanced aerodynamic performance. The study provides a quantitative assessment of drag mitigation strategies and offers guidance for the design of high-performance blunt body configurations.**

Index Terms— **Aerodynamic Drag, Blunt Body, Aero-Disks, Spike Configuration, Drag Reduction Techniques, Computational Fluid Dynamics (CFD)**

I. Introduction

Blunt bodies are widely employed in supersonic aerospace applications such as missiles, projectiles, and re-entry vehicles due to their structural robustness and favorable volumetric efficiency. However, at supersonic Mach numbers, blunt geometries generate strong detached bow shocks and large separated flow regions ahead of the forebody, resulting in high pressure drag and reduced aerodynamic efficiency. At moderate supersonic conditions such as Mach 2.2, the interaction between bow shocks, separated shear layers, and reattachment shocks plays a dominant role in determining forebody pressure drag, making drag mitigation a critical aerodynamic design challenge.

One of the earliest and most effective passive drag reduction techniques proposed for blunt bodies is the use of aerospikes, which modify the upstream flow field by displacing the bow shock away from the blunt nose and replacing it with a weaker oblique shock structure. Early experimental investigations demonstrated that aerospikes significantly reduce stagnation pressure on the forebody by creating a large recirculation region between the spike tip and the blunt nose, resulting in substantial reductions in pressure drag [1,2]. These foundational studies established the feasibility of spike-based flow control for supersonic and hypersonic applications.

Subsequent research introduced disk-tipped aerospikes, also known as aero-disks, to further enhance drag reduction. The addition of a disk at the spike tip was shown to stabilize the separated shear layer, enlarge the recirculation zone, and improve aerodynamic shielding of the blunt forebody [3,4]. Both experimental and numerical studies reported that the effectiveness of aero-disks is strongly dependent on geometric parameters such as spike length, disk diameter, and disk-to-body diameter ratio [5]. These studies highlighted the potential of disk-based configurations to achieve greater drag reduction than conventional pointed spikes.

With advancements in Computational Fluid Dynamics (CFD), several researchers employed Reynolds-averaged Navier–Stokes (RANS) solvers to investigate the detailed flow physics associated with aerospike-equipped blunt bodies, including shock–shear layer interactions, recirculation dynamics, and pressure distribution on the forebody [6]. Among these works, the study by Srinivasan and Chamberlain (2004) [7] is particularly significant, as it presented a validated CFD analysis of a spiked blunt body at Mach 2.2, with numerical predictions compared against experimental data. Their work demonstrated reliable prediction of forebody drag and established a robust numerical framework that has since been widely used as a benchmark for CFD validation in aerospike studies. Consequently, this study serves as the validation reference for the present numerical investigation.

Building upon validated single-disk aerospike concepts, later studies explored multiple-disk aerospike configurations to achieve further drag and thermal reduction. Experimental and numerical investigations showed that multiple disks divide the recirculation region into layered zones, weaken reattachment shocks, and improve flow stability downstream of the spike [8]. Numerical studies on double-disk aerospike arrangements further demonstrated reductions in both pressure drag and surface heat flux compared to single-disk configurations, particularly at high-speed flow conditions [9].

More recently, a comprehensive numerical investigation by Zhang et al. (2023) [10] examined spiked blunt bodies equipped with multiple flat aero-disks, analyzing the effects of disk number and geometry on drag and heat reduction. This work provides detailed insight into the aerodynamic mechanisms associated with multi-disk systems and serves as the base paper motivating the present study.

Despite these advances, systematic CFD investigations of dual aero-disk aerospike configurations at moderate supersonic Mach numbers, particularly Mach 2.2, remain limited. Moreover, many studies do not explicitly demonstrate numerical validation prior to extending the analysis to more complex geometries.

In the present work, a CFD-based investigation is conducted to analyze aerodynamic drag mitigation on a blunt body at Mach 2.2 using aerospike-mounted aero-disk configurations. The numerical methodology is first validated by reproducing established results for a baseline blunt body and spiked configuration reported by Srinivasan and Chamberlain [7]. Following validation, simulations are performed for a single-disk aerospike configuration to establish drag reduction trends. The analysis is then extended to a dual-disk aerospike configuration to assess the potential for further drag reduction and to elucidate the associated flow mechanisms, including shock structure modification, pressure distribution on the forebody, and recirculation behavior. The objective of this study is to quantify the aerodynamic benefits of dual aero-disk systems at Mach 2.2 using a validated CFD framework and to provide physical insight relevant to the design of passive drag reduction devices for supersonic applications.

II. METHODOLOGY

This project is carried out using a single software tool and is divided into two major phases. In the first phase, the geometric model is created within ANSYS Fluent. The second and more critical phase involves the analysis of the model , where simulations are performed in ANSYS Fluent under well defined boundary conditions and the required results are obtained.

ANSYS SOFTWARE:

ANSYS is a leading engineering simulation software that enables designers and engineers to analyse and optimize complex product designs across a range of industries, including aerospace, automotive, electronics, energy, and biomedical. Founded in 1970, ANSYS provides a comprehensive suite of tools for various physics simulations, such as structural mechanics, fluid dynamics (CFD), electromagnetics, heat transfer, and even Multiphysics simulations, where several types of physical phenomena interact.

One of the key features of ANSYS software is its ability to handle complex geometries and large-scale simulations with high accuracy. The software's advanced meshing tools create high-quality computational grids, essential for precise simulations. ANSYS also integrates powerful post-processing capabilities to visualize and interpret results, making it easier to understand and improve designs. Through these advanced features, ANSYS helps companies reduce the need for costly physical prototypes, shorten development cycles, and enhance product performance and reliability.

MODELLING

The wing geometry is modeled directly within ANSYS Fluent in Design Modeller, and the flow domain is created in the same environment using Fluent Meshing. A computational domain is defined around the blunt body to represent the surrounding airspace, which is essential for accurately simulating airflow behavior. For the analysis, a rectangular domain with one edge shaped as a semicircle is commonly used. This domain configuration is well suited for aerodynamic studies, as it aligns with the flow direction and effectively captures shock wave formation around the blunt body.

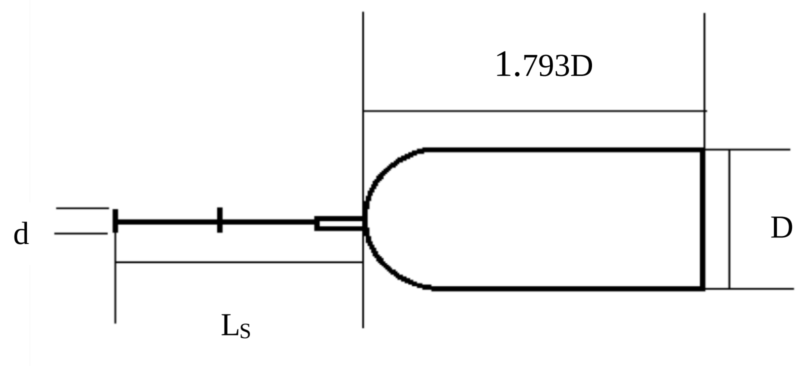

Figure 1: Geometry

Table 1 DIMENSIONS OF THE GEOMETRY

Modified Geometry Dimension	Value
Number of Disks	2
D (Diameter)	0.205m
d	0.194D
Radius of hemisphere	0.5D
Ls/D for disks	0.5

MESHING:

Our next step involves creating a computational mesh in ANSYS Fluent which is essential for performing high-quality simulations in flow analysis. This involves the discretization of the geometry into small cells that can represent complex fluid dynamics. To perform a detailed fluid flow analysis, this geometry serves as the foundation for studying flow-induced vortices and acoustic effects. Next, select a meshing strategy that meets both computational and accuracy requirements. After the meshing is done, conduct thorough quality checks on the mesh, examining aspects like skewness, orthogonality, and aspect ratio. High-quality mesh cells are vital for accuracy, as poorly shaped cells can skew results, leading to incorrect predictions. This ensures the fidelity of simulations, enabling precise analysis of complex flow.

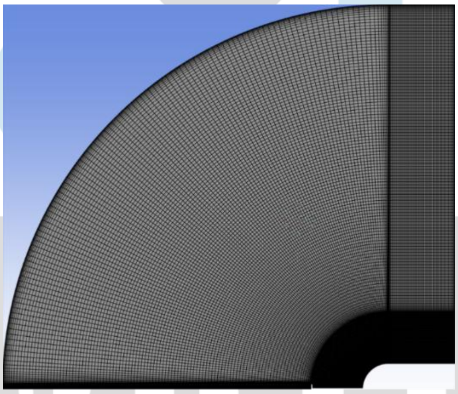

Figure 2: Mesh generation for blunt body with 1 aerodisk

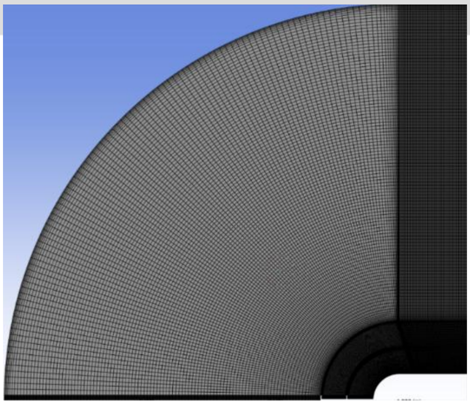

Figure 3: Mesh generation for blunt body with 2 aerodisks

SOLVER SETUP

In this analysis, the SST k–ω turbulence model is employed within a density-based, steady-state CFD framework to accurately capture turbulent boundary-layer behavior under supersonic flow conditions Mach 2.2. The model combines the near-wall accuracy of the k–ω formulation with the free-stream robustness of the k–ε model, making it well suited for predicting flow separation and transition over complex aerodynamic geometries. Simulations are carried out using a pressure far-field boundary condition at an operating pressure of 101.325 kPa, with no-slip isothermal walls maintained at 300 K, ensuring reliable prediction of separation effects that are critical for aerodynamic performance and drag reduction analysis.

GOVERNING EQUATIONS : CONTINUITY , MOMENTUM AND ENERGY

The governing equations determining the flow behaviour are the continuity, momentum and energy equations. These equations form the basis to understand the flow properties around the blunt body. The integral forms of the equations are given below.

Continuity:

$$\frac{\partial}{\partial t}\iiint \rho d\forall + \iint \rho dV dS = 0$$

Momentum:

In direction of x:

$$\frac{\partial}{\partial t}\iiint \rho u d\forall + \iint (\rho V. dS).u = \iiint \rho f d\forall - \iint pdS + F$$

In direction of y:

$$\frac{\partial}{\partial t}\iiint \rho v d\forall + \iint (\rho V. dS).v = \iiint \rho f d\forall - \iint pdS + F$$

Energy: :

$$\iiint q\rho d\forall + \dot{Q}_{vis} - \iint pV dS + \iiint \rho f V d\forall + \dot{W}_{vis} = \frac{\partial}{\partial t}\iiint \rho\left(e + \frac{V^2}{2}\right) d\forall + \iint \rho V\left(e + \frac{V^2}{2}\right) d\, S$$

Where:

- ρ is density
- f is body force per unit mass
- V is velocity
- v is component of velocity in y direction
- u is component of velocity in x direction
- ∀ is volume
- $\dot{Q}_{vis}$ is rate of viscous heat
- $\dot{W}_{vis}$ is rate of work due to viscous force
- p is pressure force
- F is viscous force
- E is internal energy

VALIDATION

The numerical results obtained for the single aero disk configuration were validated against published research to assess the accuracy of the present computational approach. The drag coefficient predicted in the present study was Cd ≈0.16, which is in excellent agreement with the value reported in the reference literature.

This close quantitative agreement confirms the validity of the numerical methodology, including the adopted geometry, boundary conditions, turbulence model, and solver settings. Consequently, the present CFD framework is considered reliable for further investigations involving modified aero disk configurations.

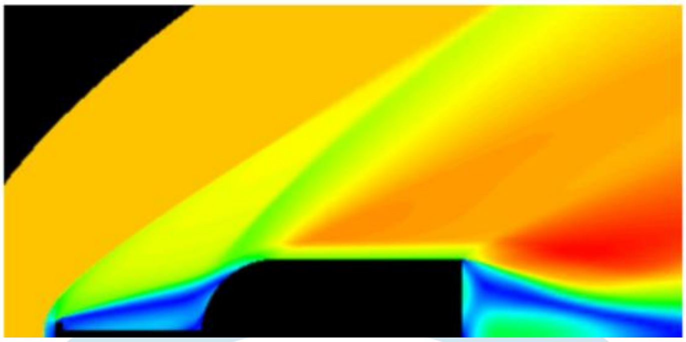

Figure 4: Mach Contour [7]

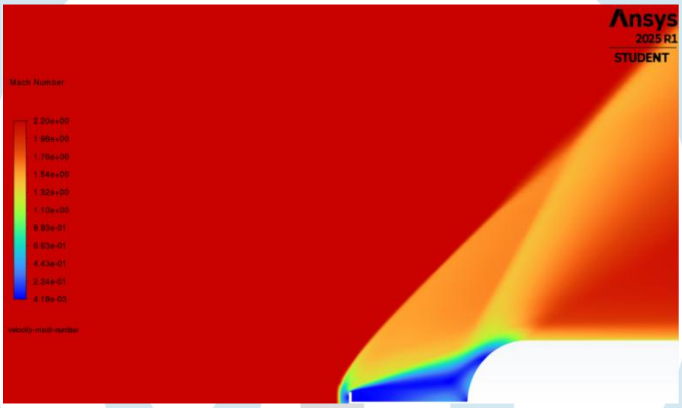

Figure 5: Computed Mach Contour

III. Results and Discussion

The introduction of a single aerodisk to the baseline geometry resulted in a significant reduction in aerodynamic drag, with the drag coefficient decreasing to 0.30. Furthermore, the addition of a second aerodisk led to a further improvement in aerodynamic performance, reducing the drag coefficient to 0.16. These results clearly indicate that increasing the number of aerodisks enhances drag reduction by effectively modifying the flow behaviour around the geometry.

Mach Number Variation:

The single-aerodisk case shows partial deceleration of the incoming flow with a localized low-Mach region near the body. The two-aerodisk configuration enlarges this low-Mach region, resulting in greater flow deceleration and reduced post-shock Mach numbers around the blunt body.

Drag Reduction Performance:

The drag coefficient decreases to approximately 0.30 with a single aerodisk, representing a significant improvement over the baseline case. The two-aerodisk configuration achieves a further reduction to about 0.16, confirming its superior aerodynamic efficiency.

Pressure Characteristics:

The single aerodisk reduces peak stagnation pressure on the body surface compared to the baseline geometry. The two aerodisks further smooth the pressure distribution, lowering pressure gradients and reducing pressure drag more effectively.

Shock Wave Behavior:

The single-aerodisk configuration produces a clearly defined bow shock with moderate stand-off distance from the blunt body. In contrast, the two-aerodisk configuration further displaces and diffuses the shock, increasing the stand-off distance and significantly weakening shock intensity.

Density Distribution:

With a single aerodisk, high-density regions remain concentrated near the stagnation zone, though reduced compared to the baseline case. The addition of a second aerodisk leads to a smoother density gradient and lower peak density near the body, indicating reduced compressibility effects and improved flow expansion.

STATIC TEMPERATURE DISTRIBUTION:

In the single-aerodisk configuration, elevated temperatures are observed near the stagnation region due to shock-induced heating. The two-aerodisk configuration exhibits lower peak temperatures and more uniform thermal gradients, demonstrating improved mitigation of aerodynamic heating.

OVERALL AERODYNAMIC EFFECTIVENESS:

While the single aerodisk provides noticeable drag and thermal reduction, the two-aerodisk configuration offers enhanced flow control, greater shock attenuation, and improved overall aerodynamic performance, making it the more effective configuration for blunt body drag reduction.

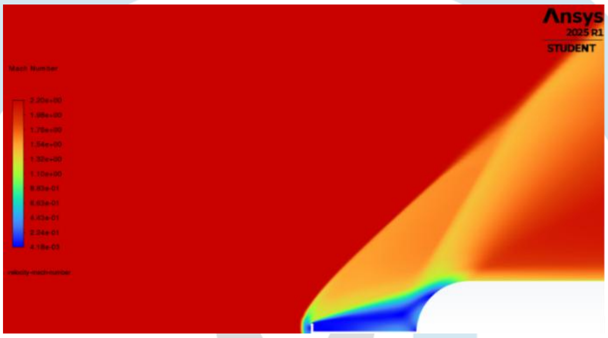

Figure 6: Mach Number Contour-1 disk

Figure 7: Static Pressure Contour-1 disk

Figure 8: Density Contour-1 disk

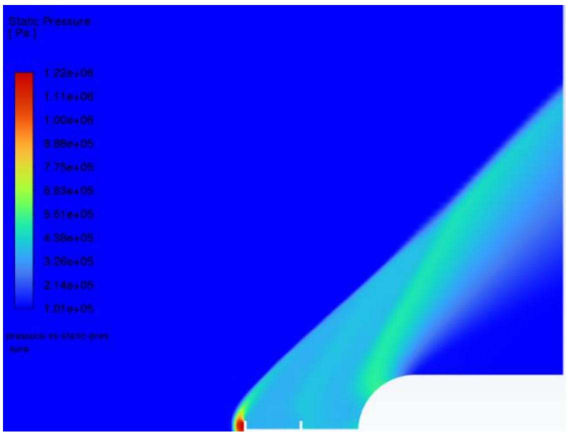

Figure 9: Static Pressure Contour-2 disk

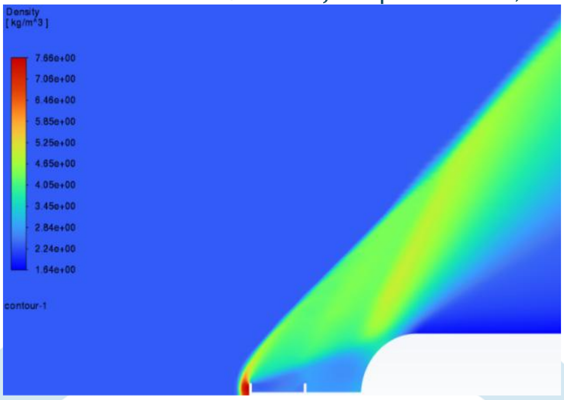

Figure 10: Density Contour-2 disk

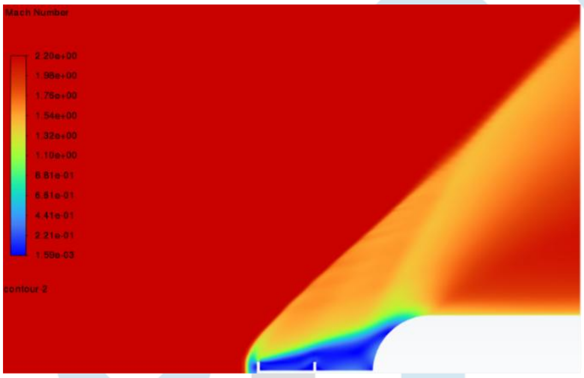

Figure 11: Mach Number Contour-2 disk

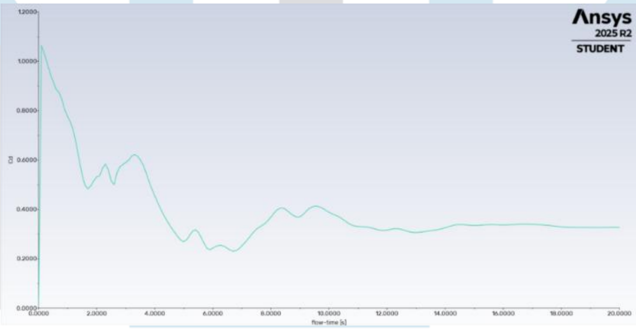

Figure12: cd for -1 disk

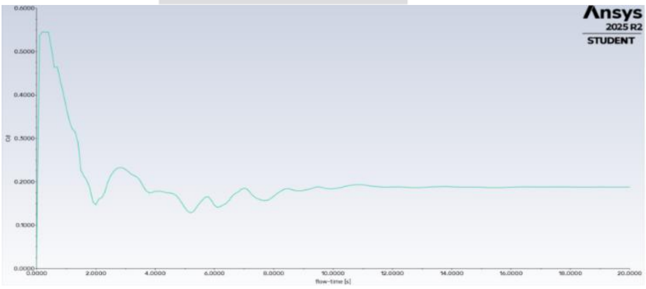

Figure 13: cd for -2 disk

References

[1] Love, E. S., and Woodling, M. J., "A Study of the Effects of a Nose Spike on the Drag of Blunt Bodies at Supersonic Speeds," NACA Technical Report No. 1358, National Advisory Committee for Aeronautics, 1958.

[2] Maull, D. J., "The Reduction of Drag on Blunt Bodies by Means of a Nose Spike," Aeronautical Quarterly, Vol. 11, No. 3, 1960, pp. 185–196. https://doi.org/10.1017/S0001925900002097

[3] Crawford, D. H., "Investigation of Disk-Tipped Nose Spikes for Drag Reduction," NACA Technical Note TN-D-1180, National Aeronautics and Space Administration, 1962.

[4] Ahmed, M. Y., and Qin, N., "Influence of Aerodisk Geometry on Drag Reduction of Spiked Blunt Bodies," Journal of Spacecraft and Rockets, Vol. 48, No. 4, 2011, pp. 600–611. https://doi.org/10.2514/1.51132

[5] Koopaee, M., and Gazor, S., "Numerical Study of Flow Control Using Disk-Tipped Aerospikes," Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering, Vol. 226, No. 4, 2012, pp. 493–504. https://doi.org/10.1177/0954410011418556

[6] Zhong, X., and Wang, J., "Numerical Investigation of Aerospike Length Effects on Supersonic Blunt Bodies," Acta Astronautica, Vol. 69, Nos. 9–10, 2011, pp. 830–841. https://doi.org/10.1016/j.actaastro.2011.05.020

[7] Srinivasan, G. R., and Chamberlain, R. R., "Drag Reduction of Spiked Missile by Heat Addition," Journal of Spacecraft and Rockets, Vol. 41, No. 4, 2004, pp. 623–630. https://doi.org/10.2514/1.1377.

[8] Narayana, P. A. A., Srinivasan, G. R., and Reddy, K. P. J., "Experimental Investigation of Double Aerodisk Spikes for Hypersonic Drag Reduction," AIAA Journal, Vol. 46, No. 11, 2008, pp. 2734–2742. https://doi.org/10.2514/1.37063

[9] Yadav, R., and Guven, U., "Aerothermodynamics of a Hypersonic Projectile with a Double-Disk Aerospike," The Aeronautical Journal, Vol. 117, No. 1195, 2013, pp. 985–1004. https://doi.org/10.1017/S0001924000008402

[10] Zhang, Z., Xu, W., Ye, W., and Li, Z., "Study on Hypersonic Drag and Heat Reduction of a Spiked Blunt Body with Multiple Flat Aerodisks and Rear Opposing Jets," Physics of Fluids, Vol. 35, No. 2, 2023, Article 026104. https://doi.org/10.1063/5.0136609.

[11] Anderson, J. D., "Modern Compressible Flow: With Historical Perspective," 3rd ed., McGraw–Hill, New York, 2003.

[12] Anderson, J. D., "Hypersonic and High-Temperature Gas Dynamics," 2nd ed., AIAA Education Series, Reston, VA, 2006.

[13] Liepmann, H. W., and Roshko, A., "Elements of Gasdynamics," Dover Publications, New York, 2001.

[14] Bertin, J. J., and Cummings, R. M., "Aerodynamics for Engineers," 6th ed., Pearson Education, Upper Saddle River, NJ, 2014.

[15] Zucrow, M. J., and Hoffman, J. D., "Gas Dynamics," Vol. 1, John Wiley & Sons, New York, 1976.