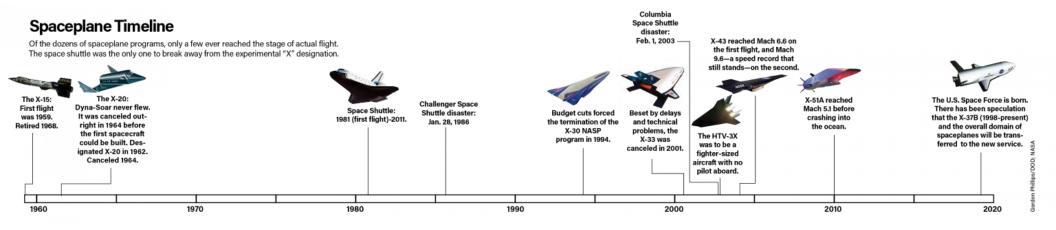
Fundamental Hypersonics

Hypersonic Design History

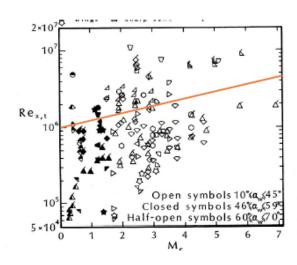


https://www.airandspaceforces.com/article/the-spaceplane-60-years-on/

Dr. Chuck Easttom, M.Ed, MSDS, MBA, MSSE, Ph.D.², D.Sc.

Conceptual Design Methods

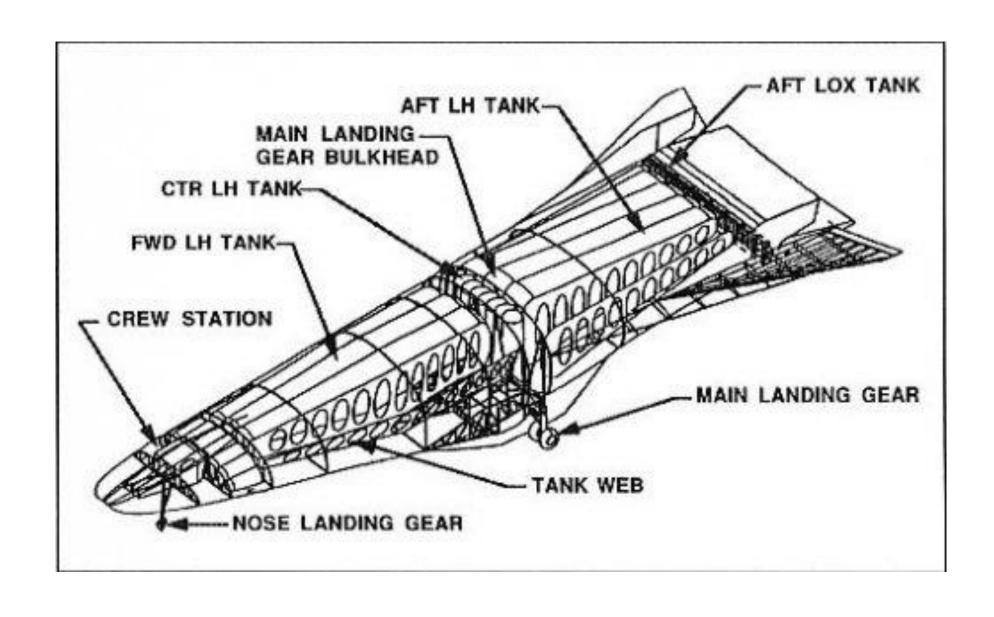
- Methods mostly developed for hypersonics in 1950s and 1960s
- Catalogued into handbook methods such as DATCOM and Missile DATCOM during 1960s and 1970s
- Methods have not changed very much since then
- Example: Missile DATCOM method for boundary layer transition that is used up to Mach 10
- Does not account for current knowledge of multiple transition modes



National Aerospace Plane

- During the late 1980s and early 1990s a large effort was made to design a SSTO air-breathing hypersonic vehicle, NASP (X-30 or "Orient Express")
- Prior to entering Phase III development the NASP program was cancelled for a number of technological reasons:
 - Structure and materials issues
 - · Scramjet engine issues and testing shorfalls
 - Inability to predict boundary layer transition and impact on TPS system
 - Control systems for all flight regimes
 - Lack of maturity of CFD given ground testing limitations
- The uncertainties within these tech areas drove the vehicle into a "design death spiral" that some feared could not be resolved at the time

X-30 NASP 1986





Lockheed Martin SR-72

- Sometimes called the "Son of Blackbird"
- The SR-72 is designed to travel at speeds of Mach 5 to Mach 6—over 4,500 mph, about twice as fast as the SR-71's Mach 3.
- It uses a turbine-based combined-cycle (TBCC) engine.
 This hybrid system starts with a turbine (or turbofan)
 mode for conventional and supersonic speeds, then
 switches to a scramjet (supersonic combustion ramjet)
 mode at hypersonic speed
- Intended for Surveillance
- https://nationalinterest.org/blog/buzz/lockheedmartin-sr-72-son-blackbird-or-darkstar-what-we-knowright-now-208483
- https://nationalsecurityjournal.org/mach-6-sr-72darkstar-summed-up-in-1-word/





Tupolev Tu-2000

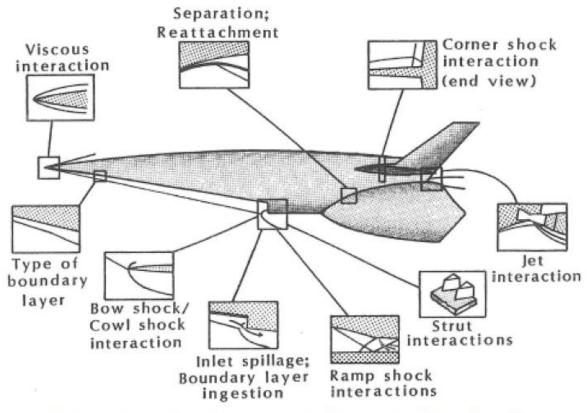
- This was a planned Soviet hypersonic plane.
- The Tupolev Tu-2000 was conceived as a hypersonic, single-stage-to-orbit (SSTO) aerospace plane, developed by the Tupolev Design Bureau in the Soviet Union during the 1980s. It was intended both as a technology demonstrator and as a potential successor to the Tu-360 bomber
- Tupolev planned multiple versions:
 - Tu-2000A: A demonstrator prototype with an estimated take-off mass of 70–90 tons, length of 60 m, capable of Mach 6 near 30 km altitude.
 - Tu-2000B: A long-range hypersonic bomber variant weighing up to 300–350 tons, designed for speeds around Mach 6 with a range near 10,000 km.
 - SSTO Spaceplane: A space-launch version weighing 210–280 tons with a payload capacity of 8–10 tons, aiming for orbital velocity ~ Mach 25 and low Earth orbit altitude of 200–400 km
 - Although cancelled, the Tu-2000 remains one of the most ambitious hypersonic aerospace projects of the late Cold War era.

Mach - Miles Per Hour

Mach	MPH
1	761.2
2	1522.4
3	2283.6
4	3044.8
5	3806
6	4567.2
7	5328.4
8	6089.6
9	6850.8
10	7612
15	11,418.1
20	15,224.1
25	19030.1

Characteristics of Hypersonic Flow: Viscous-Inviscid Interactions

These can be critical design issues for hypersonic airbreathing aircraft



Hypersonic Aerothermodynamics, AIAA Education Series, Bertin, Figure 9.41

Flow

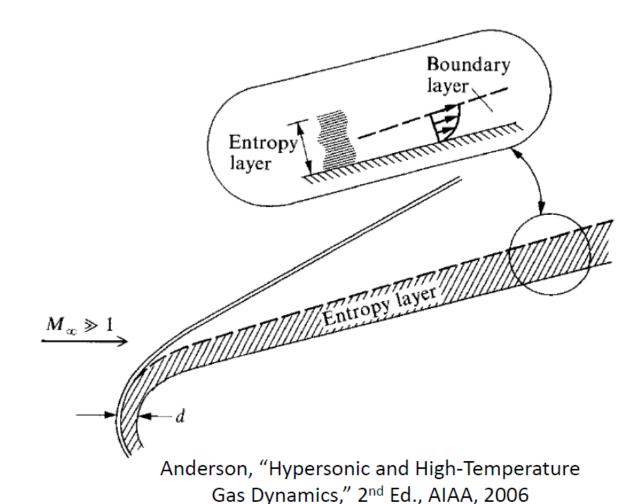
$ \begin{array}{ll} \text{Subsonic} & M_{\infty} < 1 \text{ and } M < 1 \text{ everywhere} \\ \\ \text{Transonic} & \text{case 1: } M_{\infty} < 1 \text{ and } M > 1 \text{ locally } \\ \\ \text{case 2: } M_{\infty} > 1 \text{ and } M < 1 \text{ locally} \\ \\ \text{Supersonic} & M_{\infty} > 1 \text{ and } M > 1 \text{ everywhere} \\ \\ \text{Hypersonic} & \text{supersonic flow with high-} \\ \\ \end{array} $	Incompressible	$M_{\infty} < 0.1$
case 2: $M_{\infty}>1$ and $M<1$ locally $M_{\infty}>1 \ {\sf and} \ M>1 \ {\sf everywhere}$	Subsonic	$M_{\infty} < 1$ and $M < 1$ everywhere
	Transonic	
Hypersonic supersonic flow with high-	Supersonic	$M_{\infty}>1$ and $M>1$ everywhere
temperature effects	Hypersonic	

https://compflow.onlineflowcalculator.com/Anderson/Chapter 1/

Characteristics of Hypersonic Flow: Entropy Layer

Entropy Layer: region of strong gradients

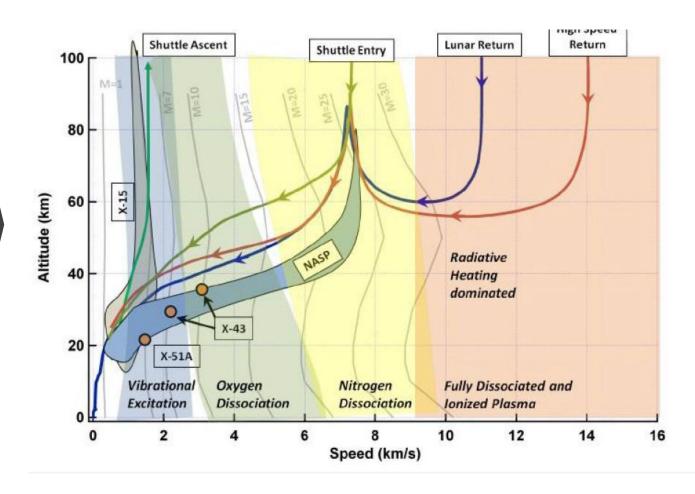
- Downstream of nose, entropy layer interacts with growing boundary layer to cause increase in aerodynamic heating to surface beyond what is predicted without entropy layer
- Can also force boundary layer to become turbulent



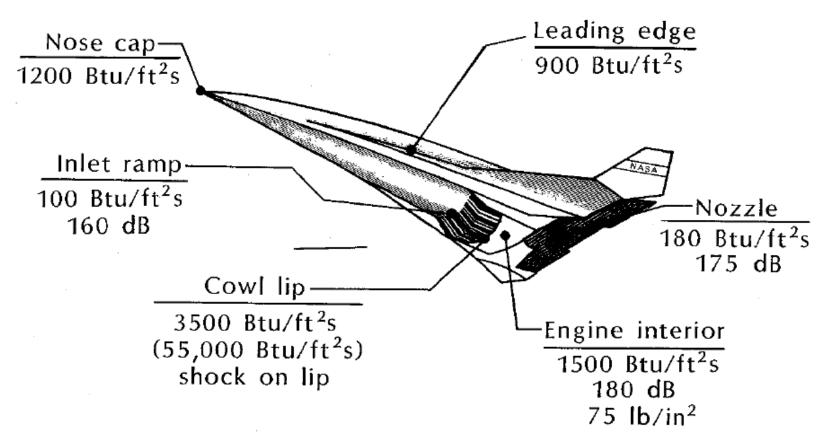
Characteristics of Hypersonic Flow: High Temperature Effects

- High Mach number flow is extremely high energy
- Slowing flow converts kinetic energy to internal energy—fluid temperature proportional to kinetic energy $(V^2/2)$
 - Flow entering boundary layer slowed by effects of friction = high-energy flow
 - Flow behind shock wave slowed = high-temp flow
- High temperatures cause chemical reactions to occur in flow:
 - T > 800 K, molecular vibration
 - T > 2000 K, diatomic oxygen dissociates
 - T > 4000 K, diatomic nitrogen dissociates and forms nitric oxide (NO) and may ionize
 - T > 9000 K, oxygen and nitrogen atoms ionize

Characteristics of Hypersonic Flow: High **Temperature Effects**



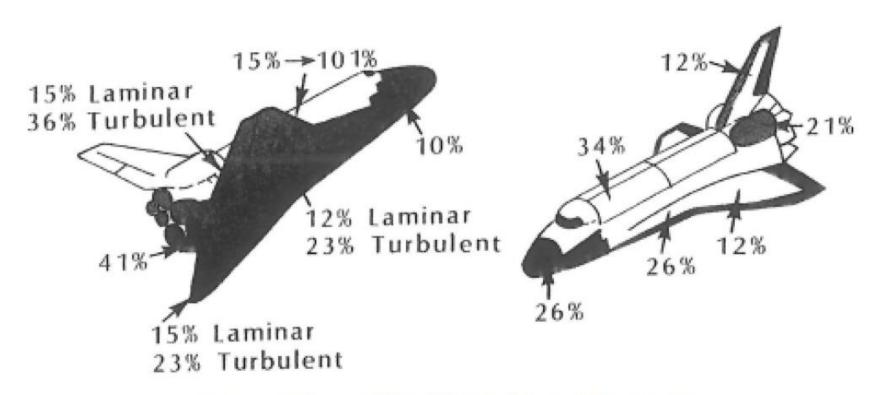
Characteristics of Hypersonic Flow: High Temperature Effects



Hypersonic Aerothermodynamics, AIAA Education Series, Bertin, Figure 10.14

Characteristics of Hypersonic Flow: High Temperature Effects

Uncertainties in pre-flight heating estimates for Shuttle *Orbiter* based on empirical correlations complemented by analytical solutions

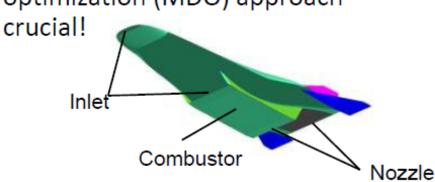


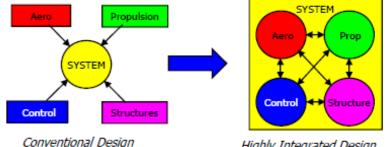
Hypersonic Aerothermodynamics, AIAA Education Series, Bertin, Figure 3.3

Highly Integrated Nature of Hypersonic Vehicles Makes Their Design Challenging

 Conventional airplane design approach does not work well for highly integrated vehicles

 Multidisciplinary design optimization (MDO) approach





Highly Integrated Design

All vehicle components and functions strongly interact

 Good hypersonic aircraft designs blend the body, wings and engine to achieve high performance through shared functionality and synergy

Characteristics of Hypersonic Flow: Low-Density Flow

Not inherent part of hypersonic flow, but hypersonic vehicles frequently fly at very high altitudes and encounter low density conditions

Theodore von Karman called the edge of space 100 km, which we now call The Karman Line

Most aerodynamic theories assume air is continuous medium (continuum) where average distance a molecule moves between successive collisions (mean free path) is small (10-7 ft)

At high altitudes (340,000 ft), mean free path is much larger (1 ft)Need to develop additional aerodynamic theories using concepts from kinetic theory of gasses

Called low-density or rarefied flows

The Karmen Line

- While the Fédération Aéronautique Internationale (FAI) officially recognizes the Kármán Line at 100 km, some agencies use different thresholds:
- U.S. Air Force & NASA (historically): Define space beginning at 50 miles (80.47 km).
- Some modern researchers advocate for redefining the line based on gravitational and atmospheric models.
- In 2018 Jonathan McDowell (Harvard-Smithsonian Center for Astrophysics) and Thomas Gangale (University of Nebraska-Lincoln) published a paper that advocated for the demarcation of space being at 80 km (50 miles; 260,000 feet), citing as evidence von Kármán's original notes and calculations (which concluded the boundary should be 270,000 ft), confirmation that orbiting objects can survive multiple perigees at altitudes around 80 to 90 km, plus functional, cultural, physical, technological, mathematical, and historical factors.

The Armstrong Limit

• The Armstrong limit or
Armstrong's line is a measure of
altitude above which atmospheric
pressure is sufficiently low that
water boils at the normal
temperature of the human body.
This limit is approximately 18–19 km
(11–12 mi; 59,000–62,000 ft) above
sea level. This is named after
AirForce general Harry Armstrong,
not astronaut Neil Armstrong.
General Armstrong was director of
the United States Aeromedical
Research Laboratory



Hypersonic flow

Hypersonic flow refers to a fluid flow regime where the Mach number (the ratio of flow velocity to the local speed of sound) exceeds approximately Mach 5. At these extreme speeds, the physics of fluid motion undergoes substantial changes compared to subsonic or even supersonic regimes. Key effects include:

- Strong shock waves ahead of objects.
- High temperatures that may cause chemical reactions, dissociation, and ionization of air.
- Thin shock layers and intense aerodynamic heating.

Continuum Flow

A continuum flow is one in which the fluid can be treated as a continuous medium, meaning the molecular nature of the gas is negligible. This is valid when the Knudsen number (Kn) is very small (typically < 0.01). The Knudsen number is defined as

$$k_n = \frac{\lambda}{L}$$

Where:

- λ = mean free path of gas molecules,
- L = characteristic length (like object diameter).
- In a hypersonic continuum flow, even though the velocity is extremely high, the density of the gas is sufficient for the continuum approximation to remain valid.

Characteristics of Hypersonic Continuum Flow

Shock-Wave Structure: Bow shocks are very close to the body and are much stronger than in lower Mach number regimes.

Viscous Effects: Boundary layers become thin and may transition to turbulence more quickly due to high Reynolds numbers.

High Temperature Effects: At hypersonic speeds, the gas behind shock waves can reach thousands of degrees Kelvin, resulting in:

- Vibrational excitation,
- Molecular dissociation,
- Ionization (plasma formation),
- Radiation effects.

Real Gas Behavior: The air cannot be treated as a perfect gas due to chemical and thermal non-equilibrium.

Modeling Hypersonic Continuum Flow

Due to extreme conditions, hypersonic continuum flows require specialized computational methods:

Navier—Stokes equations with modifications for high temperature and non-equilibrium effects.

CFD codes that include:

- Finite-rate chemical kinetics,
- Thermodynamic models for real gas behavior,
- Radiation transport models.

CFD codes

CFD codes (Computational Fluid Dynamics codes) are software tools used to simulate and analyze fluid flow, heat transfer, and related physical phenomena using numerical methods. These tools solve the governing equations of fluid dynamics—primarily the Navier-Stokes equations—to model complex flow behaviors in various engineering and scientific applications.

Software	Description
ANSYS Fluent	Industry-standard for a wide range of CFD problems; powerful GUI and solvers.
STAR-CCM+	Developed by Siemens; supports fluid- structure interaction, multiphase flows.
COMSOL Multiphysics	CFD with strong multiphysics coupling (e.g., EM, thermal, chemical).
CFX (ANSYS)	High-fidelity solver optimized for turbomachinery and rotating equipment.

Hypersonic gas models are advanced physical and mathematical formulations used to simulate and analyze gas behavior at hypersonic speeds — typically defined as Mach 5 and above. At these extreme velocities, standard ideal gas assumptions break down due to high temperatures, chemical reactions, molecular dissociation, and non-equilibrium effects.

At hypersonic speeds, kinetic energy is converted into thermal energy, heating the air to thousands of kelvin. This results in:

- Vibrational excitation of gas molecules
- Dissociation of O₂, N₂
- Ionization of atoms at very high temperatures
- Real gas behavior (non-ideal)
- Thermo-chemical non-equilibrium
- Standard perfect gas models (e.g., p=PRT) no longer suffice.

The Perfect Gas Model

- Assumes constant γ, no chemical or vibrational effects.
- Valid for Mach < 3, moderate temperatures (T < 1500 K).
- Used for baseline CFD, wind tunnel predesign.
- Limitations: Not valid at high temperature, no dissociation or internal energy modes.

Thermally Perfect Gas

- Internal energy and specific heats vary with temperature.
- Includes:
 - Translational and rotational modes
 - Vibrational excitation (via models like Einstein or harmonic oscillator)
- Often uses NASA polynomials for property interpolation.
- Suitable for T ≈ 1500–3000 K

Calorically Perfect Gas

- γ is constant, but includes multiple internal energy modes (translational, rotational).
- Specific heats c_p, c_v are constant.
- Still fails at high Temperatures (ignores vibrational and electronic excitation).

Phenomenon	When it Occurs	Modeled By
Vibrational excitation	T > 1000 K	Thermally perfect gas
Dissociation	T > 2000–3000 K	Chemical nonequilibrium
Ionization	T > 8000–10,000 K	Multi-species plasma models
Non-equilibrium	Shock layers, rarefied flows	Finite-rate chemistry

Typical Hypersonic Temperature Ranges

Region on Missile	Temperature Range	Notes
Nose cone (stagnation point)	1200°C – 3000°C (≈ 2200–5400°F)	Highest due to direct compression of air
Leading edges (fins, intakes)	1000°C – 2000°C	Sharp edges focus heat; second hottest area
Body midsection	400°C – 1000°C	Lower heat due to oblique flow and shielding
Base (aft) region	200°C – 600°C	Lower heating but possible backflow issues

Dr. Chuck Easttom, M.Ed, MSDS, MBA, MSSE, Ph.D.², D.Sc.

Mach Number vs Temperature (Stagnation Point)

Mach Number Approx. Stagnation Temperature

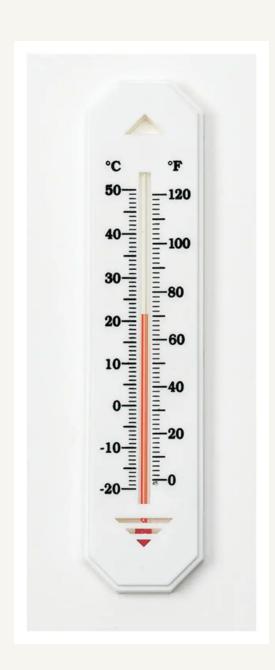
Mach 5 ~1300−1500 K (≈ 1000−1200°C)

Mach 7 ~1800–2200 K (≈ 1500–1900°C)

Mach 10 ~3000−3500 K (≈ 2700−3200°C)

Mach 15 >5000 K (fully ionized flow)

These values assume sea-level equivalent density or moderate altitude. At high altitude, temperatures are lower due to thin air but radiation may dominate.



Factors Influencing Temperature

- Mach number: Higher Mach = greater compression and kinetic-to-thermal energy conversion.
- Altitude: At high altitude (30–60 km), convective heating is lower, but radiative heating may be more dominant at Mach >10.
- Surface material and emissivity: High-emissivity coatings radiate heat away better.
- Ablative materials (e.g., phenolic composites) are used in peak-heating zones.
- Shape of vehicle
 - Blunt shapes increase stagnation temperature.
 - Sharp shapes reduce wave drag but lead to higher heat flux.
- Flight duration
 - Longer flights allow more heat accumulation, requiring thermal protection systems (TPS).

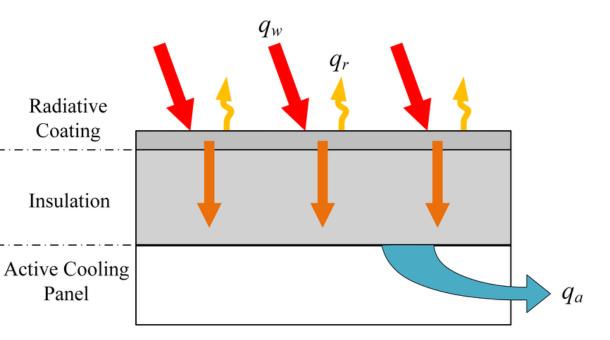
High Temperatures

- "At temperatures above 1700 K (3000 °R), the equilibrium of specific heat (c p) of air depends strongly upon both temperature and pressure because chemical reactions have become important. Hence, γ reaches a value of 1.286 when the formation of nitric oxide (NO) begins. When nitrogen is released during combustion, it combines with oxygen atoms to create NO, which then combines with oxygen to create nitrogen dioxide (NO2). At temperatures above 1700 K, chemical reaction and dissociation become very complicated and cannot be treated with a simple gas model."
- -Musielak, Dora. Scramjet Propulsion: A Practical Introduction (Aerospace Series) (pp. 3-4). Wiley. Kindle Edition.

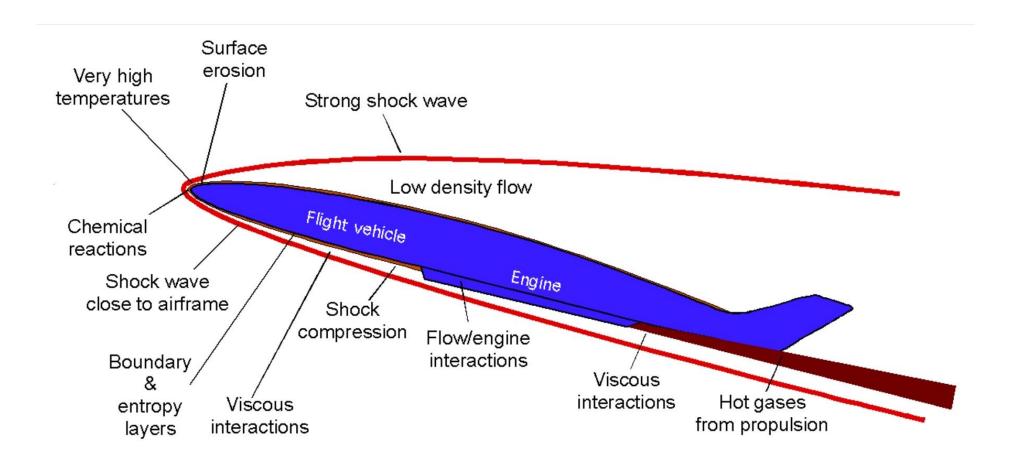
Thermal Protection Systems (TPS)

To survive hypersonic temperatures, missiles employ:

- Ablative materials (e.g., carbo) phenolics): vaporize to absorb heat.
- Ceramic tiles (e.g., silica or alumina-based): insulate and reflect heat.
- Refractory metals (e.g., tungsten, niobium alloys): withstand extreme temperatures structurally.
- Active cooling (e.g., transpiration or film cooling): rare in missiles due to complexity.



Hypersonic Stresses





Hypersonic Gas Dynamics

Hypersonic Gas Dynamics is the branch of fluid mechanics that deals with the behavior of gases when objects move through them at hypersonic speeds, generally defined as speeds greater than Mach 5 (i.e., five times the speed of sound). This field is critical in the design of hypersonic vehicles, such as missiles, spacecraft re-entry vehicles, and advanced aircraft.

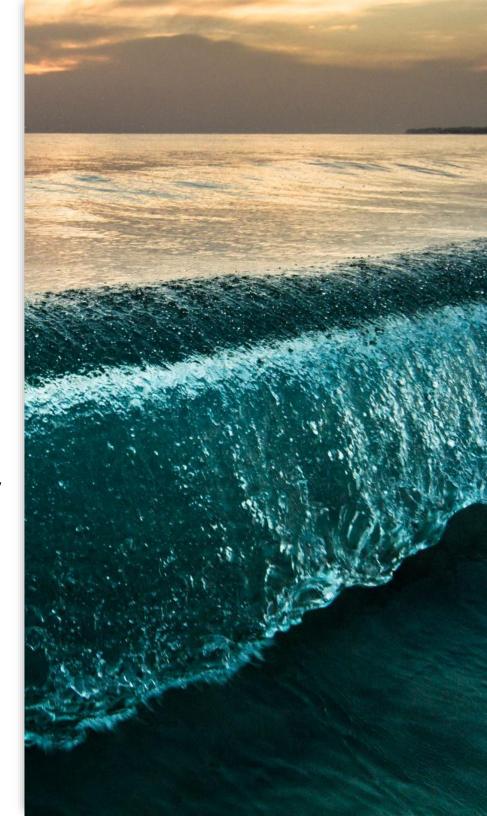
Hypersonic gas dynamics encompasses:

- Shock waves
- Boundary layers
- High-temperature effects
- Chemical reactions and dissociation of gas molecules
- Radiation and real gas effects

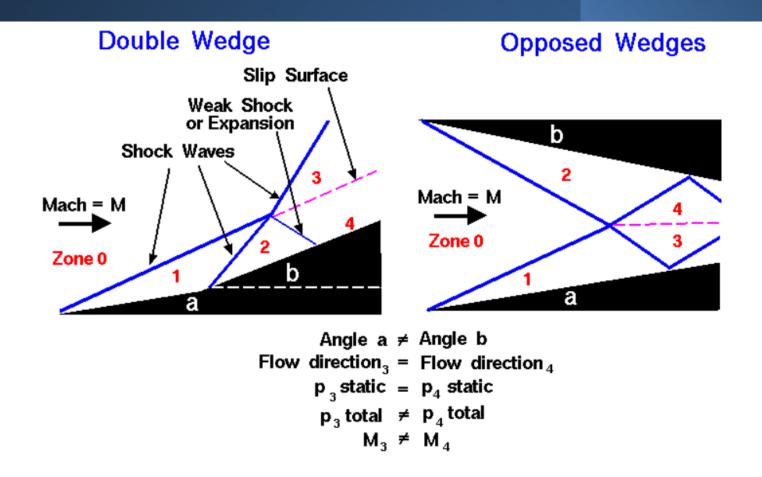
Shock Wave Concepts

In order to understand hypersonics, you need a basic understanding of shock waves. A Static Pressure Disturbance refers to a localized change in the ambient (static) pressure of a fluid—typically a gas like air—without necessarily involving large-scale movement of the fluid itself. It is an essential concept in fluid dynamics, aerodynamics, and acoustics. Static pressure is the pressure exerted by a fluid at rest or in motion independent of its velocity. It is the thermodynamic pressure that you would measure using a pressure tap aligned with the flow and not affected by dynamic pressure. The magnitude and propagation of static pressure disturbances are influenced by:

- Medium properties (e.g., compressibility, density)
- Amplitude of disturbance
- Wave speed (usually speed of sound)
- Source geometry and motion

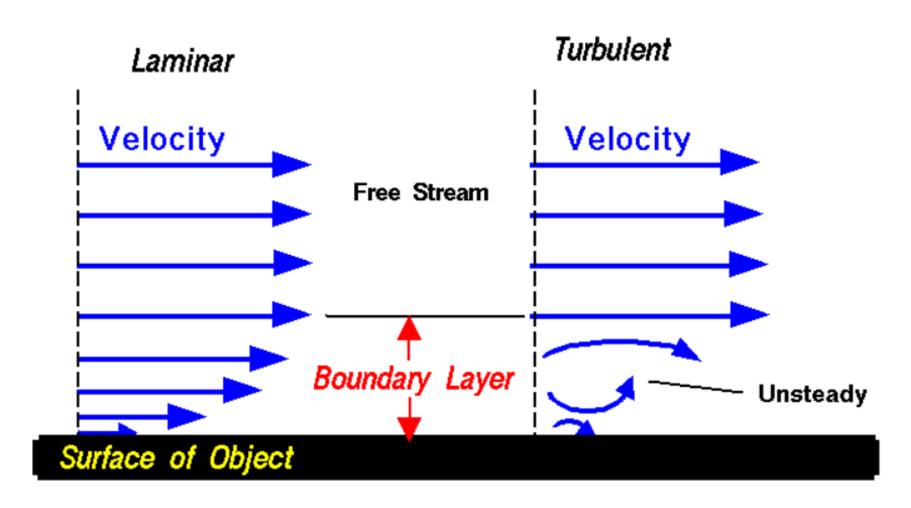


Crossed Shock waves



When an object moves faster than the speed of sound, and there is an abrupt decrease in the flow area, shock waves are generated. Shock waves are very thin regions in the gas where the gas properties change by a large amount. In many flow problems multiple shocks are present. The shocks may intersect with each other and with the surfaces generating them. On this page we present the physics which govern the intersection of two shock waves and include a Java program that you can use to investigate these flow problems. https://www.grc.nasa.gov/WWW/BGH/crosshock.html

Boundary Layers



Velocity is zero at the surface (no - slip)

https://www.grc.nasa.gov/WWW/BGH/boundlay.html

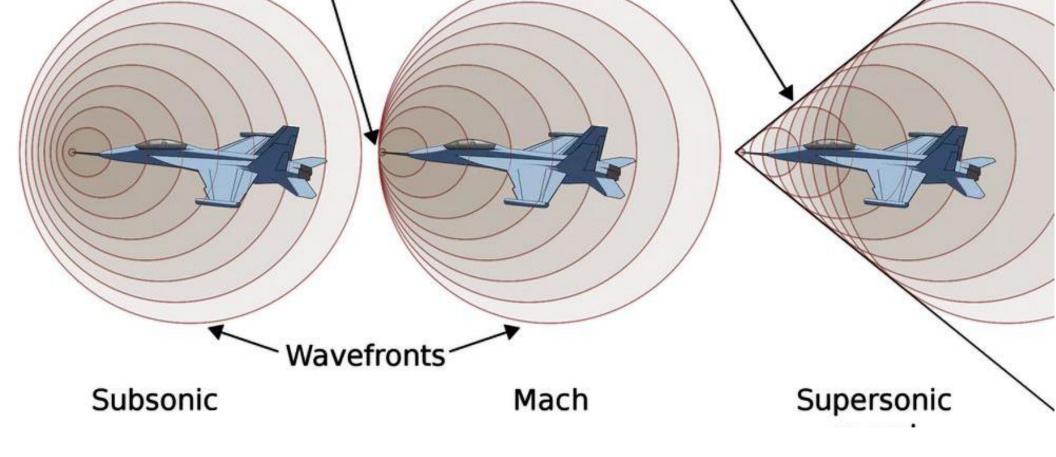
Dr. Chuck Easttom, M.Ed, MSDS, MBA, MSSE, Ph.D.², D.Sc.

Boundary Layer

As an object moves through a fluid, or as a fluid moves past an object, the molecules of the fluid near the object are disturbed and move around the object. Aerodynamic forces are generated between the fluid and the object. The magnitude of these forces depend on the shape of the object, the speed of the object, the mass of the fluid going by the object and on two other important properties of the fluid; the viscosity, or stickiness, and the compressibility, or springiness, of the fluid. To properly model these effects, aerospace engineers use similarity parameters which are ratios of these effects to other forces present in the problem. If two experiments have the same values for the similarity parameters, then the relative importance of the forces are being correctly modeled.

Aerodynamic forces depend in a complex way on the viscosity of the fluid. As the fluid moves past the object, the molecules right next to the surface stick to the surface. The molecules just above the surface are slowed down in their collisions with the molecules sticking to the surface. These molecules in turn slow down the flow just above them. The farther one moves away from the surface, the fewer the collisions affected by the object surface. This creates a thin layer of fluid near the surface in which the velocity changes from zero at the surface to the free stream value away from the surface. Engineers call this layer the boundary layer because it occurs on the boundary of the fluid.

https://www.grc.nasa.gov/WWW/BGH/boundlay.html



Subsonic pressure

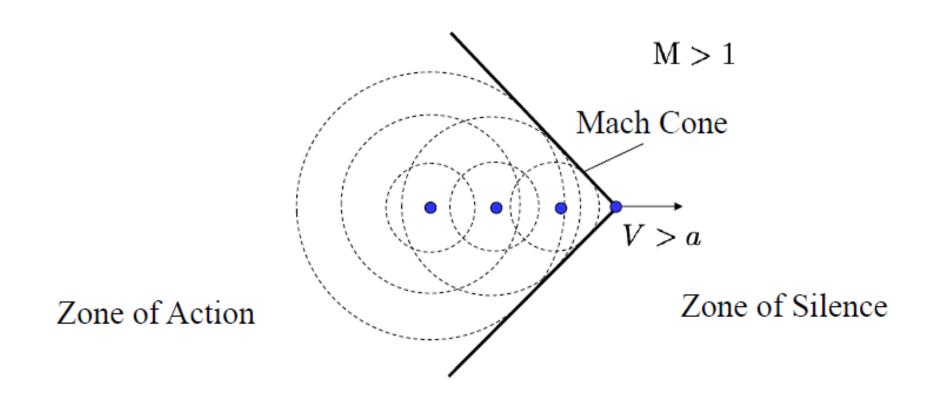
A subsonic pressure disturbance is a localized change in pressure that occurs within a flow field moving at subsonic speeds—that is, less than the speed of sound in the medium (i.e., M<1, where M is the Mach number).

In subsonic flows, pressure disturbances are communicated smoothly and continuously through the fluid, and they propagate in all directions, not just downstream. This is fundamentally different from supersonic conditions, where information (disturbances) can only travel downstream. As air flows around an airfoil, it compresses slightly ahead of the leading edge and expands around the curvature. These changes cause pressure gradients that are smoothly communicated across the flow field. Only the air behind the disturbance is "aware" that the disturbance exists

Supersonic Pressure Disturbance

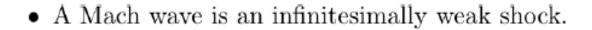
A supersonic pressure disturbance refers to a sudden change in pressure that occurs in a fluid flow when the flow velocity exceeds the local speed of sound (i.e., Mach number M>1). These disturbances behave very differently from those in subsonic flow because of how information and pressure waves propagate. In supersonic flow, pressure disturbances cannot propagate upstream. This is due to the fact that the flow speed is greater than the speed at which pressure (sound) waves can travel.

Supersonic Pressure Disturbance

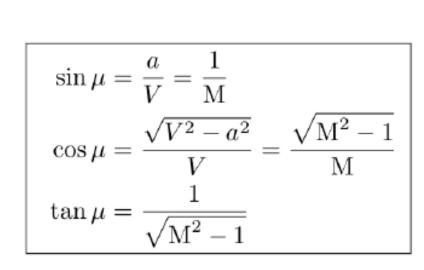


Mach Wave/Mach Angle

 μ



• Flow through a Mach wave is isentropic



+ Shock Waves

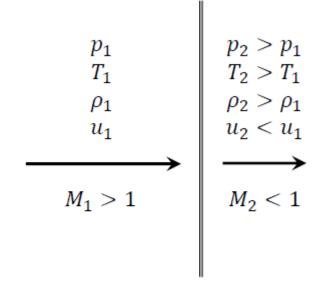
- •Shock waves form around vehicles flying near or above Mach 1
- •Shock waves create abrupt changes (discontinuities) in flow properties (velocity, pressure, density, temperature)
- Types of shocks:
- Normal shock—flow is perpendicular to shock
- Oblique shock—flow is at an angle to shock
- Bow shock—flow crosses shock at various angles

Normal Shock Waves

Across a normal shock:

- Mach number decreases from Supersonic to Subsonic value
- Pressure, temperature, and density increase
- Shocks are adiabatic (no heat
- transfer)
- Shocks are not isentropic (process is not reversible)

Shock theory gives relations for defining the changes across a normal shock which are functions of Mach number and ratio of specific heats



Normal Shock

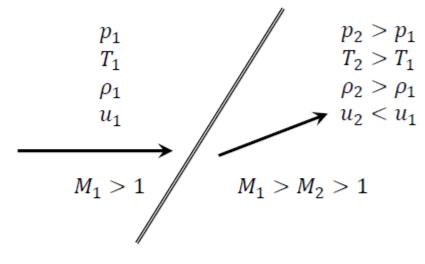
$$rac{T_2}{T_1} = rac{\left(1 + rac{\gamma - 1}{2} M_1^2
ight) \left(rac{2\gamma}{\gamma - 1} M_1^2 - 1
ight)}{M_1^2 \left(rac{2\gamma}{\gamma - 1} + rac{\gamma - 1}{2}
ight)}$$

Oblique Shock Waves

Across an oblique shock:

- Flow turns "toward" shock
- Mach number decreases from Supersonic to lower Supersonic value
- Pressure, temperature, and density increase
- Oblique shocks are adiabatic (no heat transfer)
- Oblique shocks are not isentropic (process is not reversible)

Oblique shock theory presented in charts (graphs) as a function of the turning angle and upstream Mach number

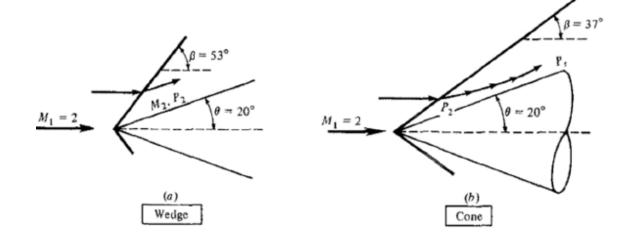


Oblique Shock

Conical Shocks

Cones are 3D flow while wedges are 2D flow

As the flow proceed downstream after the conical shock the properties of the air can change due to "3D pressure relief"



Anderson J. (2016) Fundamentals of Aerodynamics, McGraw-Hill.

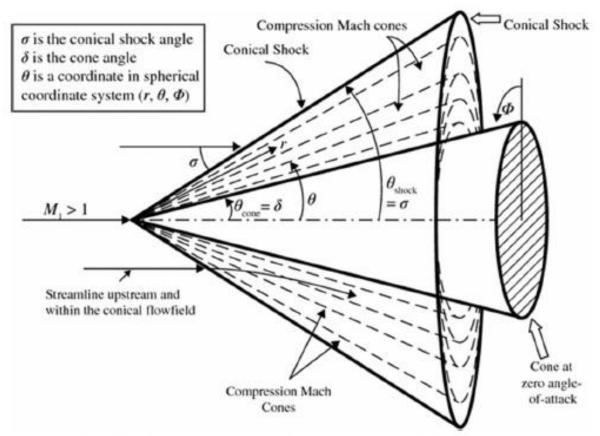
Conical Shocks

Cone theory was developed in the 1930s by G.I. Taylor and J.W. Maccoll for a sharp, right circular

Cone

Derivation uses spherical coordinates and assumes no variations in circumferential Direction

Properties are constant along rays coming from cone tip Results can be obtained for surface of the cone



Vos R., Farokhi S. (2015) Shock-Expansion Theory. In: Introduction to Transonic Aerodynamics. Fluid Mechanics and Its Applications, vol 110, Springer.

Supersonic Pressure Disturbance

Туре	Description
Shock waves	Thin regions across which pressure, temperature, and density increase suddenly.
Expansion fans (Prandtl-Meyer fans)	Smooth, continuous pressure decreases occurring when the flow turns around a convex corner.
Oblique shocks	Angled shocks formed when supersonic flow encounters a wedge or ramp.
Normal shocks	Perpendicular shocks, usually occurring inside supersonic inlets or nozzles.

Viscous Shock-Layer (VSL) code

The Viscous Shock-Layer (VSL) code is a computational tool used to simulate hypersonic flow fields around reentry vehicles and other high-speed aerospace configurations. It provides an efficient way to analyze the aerothermodynamic environment experienced during reentry, particularly in the thin, high-temperature shock layer that forms between the bow shock and the vehicle surface. VSL code solves the Navier-Stokes equations in a simplified form for a viscous, chemically reacting shock layer. It is designed to model:

- Boundary layers
- Shock-layer heating
- Species transport
- Chemical nonequilibrium

It is especially valuable in preliminary design and analysis stages for high-speed vehicles, where full Navier-Stokes simulations may be too computationally expensive.

Viscous Shock-Layer (VSL) code

Element

Equations Solved

Gas Model

Species Transport

Energy Modes

Wall Conditions

Description

Parabolized Navier-Stokes or boundary-layer equations in stagnation streamline

Thermochemical nonequilibrium (including air dissociation and ionization)

Tracks chemical species like N_2 , O_2 , NO, electrons, etc.

Accounts for vibrational, translational, rotational, and electronic energy

Allows specification of thermal boundary conditions (e.g. isothermal, adiabatic)

Langrangian Flow

In the Lagrangian description, you follow individual fluid particles as they move through space and time. Each particle is tagged (conceptually) and tracked, so the description focuses on the trajectory, velocity, and other properties of that particle. Imagine dropping a tracer dye into a fluid and following each molecule of dye as it drifts along with the flow—that's a Lagrangian perspective. A fluid particle is identified by its initial position

X at time t=0

t

Its position at time t is given by a trajectory function:

x=x(X,t)

where:

X = initial particle label (Lagrangian coordinate)

x = actual position at time t

The velocity of the particle is:

 $v(X,t) = \partial x(X,t)/\partial t$

Other quantities like density, pressure, and temperature are expressed as functions of the particle label X and time t

Eulerian Flow

In the Eulerian framework, instead of following individual particles, we focus on fixed points in space and ask: "What is happening to the fluid here as time evolves? Imagine you set up a sensor at a fixed location in a river. The Eulerian description is the record of how velocity, pressure, and temperature change at that location over time. You don't care which particle is passing by, just what the flow field looks like at that point.

We describe the flow field using functions of space and time:

$$u=u(x,y,z,t), p=p(x,y,z,t), \rho=\rho(x,y,z,t)$$

u = velocity vector field

p = pressure field

 ρ = density field

At any point in space (x,y,z), the field tells us the value at time t

The Navier-Stokes equations are a set of partial differential equations that describe the motion of fluids. They are fundamental in fluid dynamics and are used to model a wide range of phenomena, from weather patterns to blood flow. These equations express the conservation of mass and momentum, taking into account both pressure and viscous forces within the fluid

Navier Stokes - History

Despite the fact that the motion of fluids is an exploratory topic for human beings, the evolution of mathematical models emerged at the end of the 19th century after the industrial revolution. The initial appropriate description of the viscous fluid motion was indicated in the paper "Principia" by Sir Isaac Newton (1687), in which he investigated the dynamic behavior of fluids under constant viscosity.

Later, Daniel Bernoulli (1738) and Leonhard Euler (1755) subsequently derived the equation of inviscid flow, which is now expressed as Euler's inviscid equations. Even though Claude-Louis Navier (1827), Augustin-Louis Cauchy (1828), Siméon Denis Poisson (1829), and Adhémar St. Venant (1843) had carried out studies to explore the mathematical model of fluid flow, they overlooked the viscous (frictional) force.

In 1845, Sir George Stokes derived the equation of motion of a viscous flow by adding Newtonian viscous terms, thereby the Navier-Stokes Equations were brought to their final form, which has been used to generate numerical solutions for fluid flow ever since.

In short, the Navier-Stokes equations were derived independently and progressively by Claude-Louis Navier and Sir George Stokes over a span of decades in the 19th century. They were based on applying Newton's second law of motion to fluid flow, taking into account the viscous and pressure effects in order to describe a viscous fluid flow.

https://www.simscale.com/docs/simwiki/numerics-background/what-are-the-navier-stokes-equations/

The Navier–Stokes equations are a set of nonlinear partial differential equations that describe the motion of fluid substances, including liquids, gases, and plasmas. They form the foundation of fluid dynamics and are essential for understanding how fluids flow under various physical conditions — from airflow over a wing to blood flow in arteries, and from ocean currents to the behavior of stars. They express conservation laws for:

Mass (Continuity equation)

Momentum (Newton's second law for fluid motion)

Energy (optional, for compressible or thermally dynamic flows)

Continuity Equation (Mass Conservation)

$$\frac{\partial_P}{\partial t} + \nabla * p(u) = 0$$

ρ: Fluid density

u: Velocity vector

See also https://www.grc.nasa.gov/www/k-12/airplane/nseqs.html

And

https://cfd-online.com/Wiki/Navier-Stokes_equations

Momentum Equation (Navier–Stokes Equation)

$$p\left(\frac{\partial U}{\partial t} + U * \nabla u\right) = -\nabla_p + \mu \nabla^2 \mathbf{u} + \mathbf{f}$$

p: Pressure

μ: Dynamic viscosity

 ∇^2 u: Viscous (diffusion) term

f: External body forces (e.g., gravity)

This is essentially Newton's second law (F = ma) applied to a fluid element

Energy Equation (for Compressible Flows)

$$\frac{\partial}{\partial t}(pe) + \nabla * [(pc + p)U] = \nabla * (k\nabla T) + \overline{\Phi}$$

e: Total energy per unit mass

T: Temperature

κ: Thermal conductivity

Φ: Dissipation function

Rankine-Hugoniot conditions

The Rankine–Hugoniot conditions are a set of relations in fluid dynamics (and shock wave theory) that describe the conservation laws across a shock wave or a discontinuity surface in a compressible fluid.

When a shock wave passes through a medium, physical quantities such as pressure, density, temperature, and velocity change abruptly. The Rankine–Hugoniot relations ensure that the conservation of mass, momentum, and energy holds across that discontinuity. These are also referred to as Rankine–Hugoniot jump conditions or Rankine–Hugoniot relations.

The Rankine–Hugoniot conditions are fundamental in shock wave analysis, combustion theory, and high-speed aerodynamics. They describe how flow properties change across a shock by enforcing conservation of mass, momentum, and energy.

Rankine-Hugoniot conditions

Conservation of mass

 $\rho 1u1 = \rho 2u2$

ρ1,ρ2 :fluid density before and after the shock

u1,u2: fluid velocity relative to the shock

Conservation of momentum

$$p_1 + \rho_1 u_1^2 = p_2 + \rho_2 u_2^2$$

p1,p2: pressure before and after the shock Conservation of Energy

$$h_1+rac{u_1^2}{2}=h_2+rac{u_2^2}{2}$$

h: specific enthalpy

Rankine-Hugoniot conditions

Shock Relations (Derived Forms)

From these conservation laws, one can derive useful relations, such as:

Pressure ratio:

$$rac{p_2}{p_1} = 1 + rac{2\gamma}{\gamma+1}(M_1^2-1)$$

Density ratio:

$$rac{
ho_2}{
ho_1} = rac{(\gamma+1)M_1^2}{(\gamma-1)M_1^2+2}$$

Temperature ratio:

$$rac{T_2}{T_1} = rac{p_2}{p_1} \cdot rac{
ho_1}{
ho_2}$$

where γ is the specific heat ratio and M_1 is the Mach number before the shock.

Dr. Chuck Easttom, M.Ed, MSDS, MBA, MSSE, Ph.D.², D.Sc.

Becker– Morduchow– Libby solution

The Becker–Morduchow–Libby (BML) solution is a classical analytical solution in compressible fluid dynamics, specifically dealing with one-dimensional, steady shock structures in a viscous, heat-conducting gas. When we use the Rankine–Hugoniot conditions, we treat a shock wave as a mathematical discontinuity (infinitely thin surface).

But in reality, shocks have a finite thickness (on the order of the mean free path of molecules) because of viscosity and thermal conduction. To model this *internal shock structure*, researchers developed exact and approximate solutions of the Navier–Stokes equations with transport effects.

Becker (1922): First obtained an exact solution for steady, 1D shock waves assuming constant viscosity and thermal conductivity.

Morduchow & Libby (1949–1951): Extended Becker's work to include realistic transport models and provided closed-form solutions for velocity and temperature distributions.

This combined framework is often referred to as the Becker–Morduchow–Libby solution.

Shock is not a discontinuity – it has a finite thickness set by viscosity and conductivity.

Velocity profile: Smooth transition between pre-shock and post-shock states, approximately hyperbolic tangent shaped.

Temperature profile: Often non-monotonic; it may overshoot the downstream temperature before relaxing back — a well-known prediction of the BML solution.

Validation: The BML solution has been confirmed by molecular simulations (Boltzmann equation, DSMC) for moderate Mach numbers.

Coupled Thin-Layer Navier-Stokes (TLNS)

The Coupled Thin-Layer Navier-Stokes (TLNS) and Parabolized Navier-Stokes (PNS) codes are advanced computational fluid dynamics (CFD) tools used for simulating hypersonic flowfields, especially around reentry vehicles. When used together in a "coupled" approach, these codes provide a detailed and efficient analysis of both the shock layer flow and the boundary layer dynamics, accounting for viscous, thermal, and chemical nonequilibrium effects. By coupling TLNS and PNS, you get the best of both worlds:

- Use TLNS near the stagnation region or in areas with complex interactions (e.g., nose, shock layers)
- Use PNS farther downstream along the cone or body, where the flow is smoother and parabolic
- TLNS is solved in the nose region to capture full viscous and inviscid behavior accurately.
- The output of TLNS is used as an inlet condition for PNS, which is then solved down the body and into the wake.
- The full solution includes temperature, pressure, velocity, species composition, and heat flux along the surface.

Blasius Solution

A Blasius boundary layer (named after Paul Richard Heinrich Blasius) describes the steady two-dimensional laminar boundary layer that forms on a semi-infinite plate which is held parallel to a constant unidirectional flow

The Blasius solution is the similarity solution to the laminar boundary layer equations for a flat plate. It predicts how the velocity profile, boundary layer thickness, and wall shear vary along the plate. It is one of the cornerstone results in boundary layer theory and aerodynamics.

Paul Blasius, a student of Prandtl, solved the boundary layer equations for the special case of steady, 2D, incompressible laminar flow over a flat plate with uniform free-stream velocity.

His solution gives the velocity profile within the boundary layer and predicts boundary layer growth along the plate.

Blasius Solution (Flat plate, steady, incompressible flow)

The Blasius profile is a solution of the Navier Stokes equations for a flow close to a semi-infinite flat plate. Firstly, by assuming that the flat plate is infinitely long along the spanwise direction, we can ignore variations along one-dimension entirely. Secondly, the flow is assumed steady, so that derivatives with respect to time are neglected as well. This allows us to start off with the 2D steady-state Navier-Stokes equations:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right)$$

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right)$$

Falkner-Skan Solution

The Falkner–Skan solution is a similarity solution to the Navier–Stokes equations under boundary layer approximations. It solves the boundary layer flow for external velocity profiles of the form:

$$U_e(x) = K_x{}^m$$

Where:

 $U_{e}(x)$ is the freestream velocity just outside the boundary layer,

m determines the nature of the pressure gradient:

m>0: Favorable pressure gradient (accelerating flow),

m=0: Zero pressure gradient (flat plate — Blasius case),

m<0: Adverse pressure gradient (decelerating flow).

Chorin's Projection

Chorin's Projection is a numerical method used to solve the incompressible Navier–Stokes equations (fluid flow problems) in a way that enforces the divergence-free condition ($\nabla \cdot \mathbf{u} = 0$) for velocity fields without having to solve the coupled velocity–pressure equations directly.

In incompressible fluid dynamics, the Navier—Stokes equations couple velocity and pressure in such a way that solving them directly can be computationally expensive. The main challenge is that the incompressibility constraint means the velocity field must remain divergence-free at all times, and pressure is not given explicitly but instead acts as a Lagrange multiplier to enforce this condition.

Chorin's method, introduced by Alexandre Chorin in 1967, avoids solving for velocity and pressure simultaneously by splitting the computation into separate steps — a technique known as a projection method.

Hypersonic IssuesWave Drag

Wave drag is a specific type of aerodynamic drag that occurs when an object moves at transonic or supersonic speeds, typically above Mach 1. It is caused by the formation of shock waves around the object as it moves through the air.

Wave drag is the drag force resulting from compressibility effects—specifically, shock waves generated due to the high-speed flow of air around a body. It is a major component of total drag in supersonic and hypersonic flight regimes.

At high speeds (Mach 1+), air can no longer move smoothly around a body:

The airflow compresses and creates shock waves, especially at sharp edges (e.g., nose tips, wing leading edges).

These shock waves change pressure and velocity suddenly, causing a loss in total pressure—a direct source of drag.

There are two main forms:

- Bow shock drag: Caused by the leading shock wave ahead of the object.
- 2. Expansion and recompression drag: Due to airflow separating and recombining, especially around curves and transitions.

Hypersonic Issues- Other Drags

Skin friction drag is a type of aerodynamic drag caused by the viscous friction between a solid surface (like a missile or aircraft body) and the air (or fluid) moving over it. It results from the tangential shear stress created as layers of air "rub" against the object's surface. When air flows over a surface:

- 1. Viscosity causes the air molecules in contact with the surface to slow down and stick to it (called the no-slip condition).
- 2. Layers of air just above this slow layer also get slowed down, creating a velocity gradient in a region called the boundary layer.
- This shearing action results in frictional forces, which resist the object's motion — this is skin friction drag.

Factor	Effect on Skin Friction Drag
Surface area	More surface = more drag
Surface roughness	Rougher = higher drag (turbulent boundary layer)
Speed (Reynolds number)	Higher speed = thinner boundary layer = increased drag
Flow type	Laminar flow = low drag; Turbulent flow = high drag
Viscosity of air	Higher viscosity = more friction

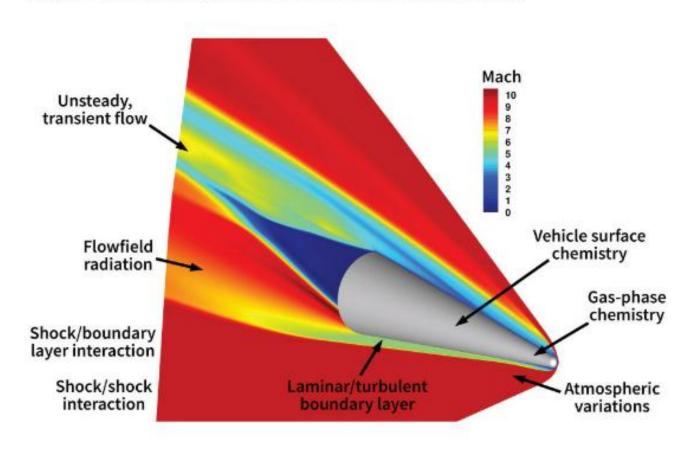
Hypersonics and Heat

Hypersonic weapons experience challenging aerothermal conditions that strain the limits of current guidance, control, and materials technologies. After reentering the atmosphere at speeds of around Mach 20, an intercontinental-range hypersonic glider experiences extreme pressures and vibration modes, as well as temperatures topping 4,000 degrees Fahrenheit. Fahrenheit. In such an environment, the vehicle's surrounding atmosphere dissociates into a plasma, reacting violently with the airframe's surface. Whereas ballistic reentry vehicles experience similar conditions for tens of seconds during atmospheric reentry, hypersonic weapons must survive similar conditions for many minutes.

Karako, Tom; Dahlgren, Masao. Complex Air Defense: Countering the Hypersonic Missile Threat (CSIS Reports) Bloomsbury Publishing.

Hypersonics and Heat

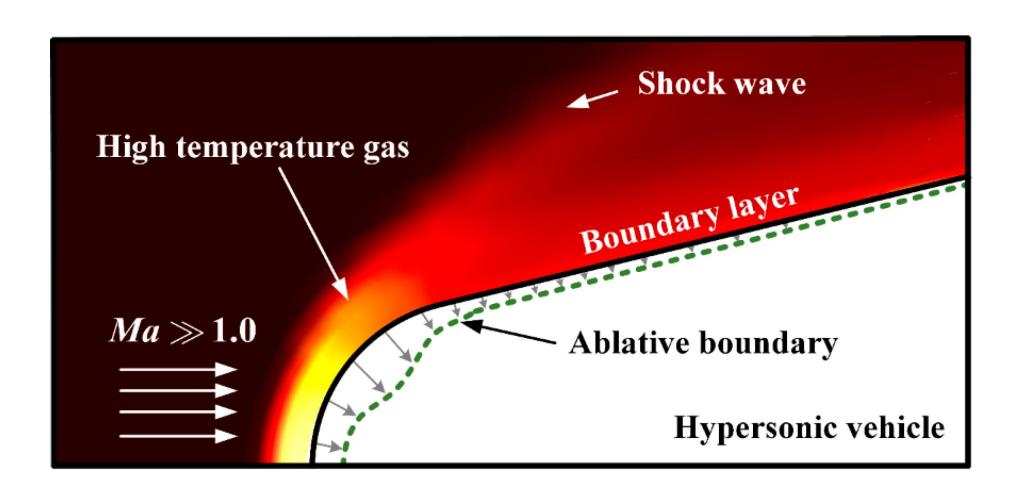
Figure 7: Depicting Hypersonic Flow Phenomena



Karako, Tom; Dahlgren, Masao. Complex Air Defense: Countering the Hypersonic Missile Threat (CSIS Reports) Bloomsbury Publishing.

Dr. Chuck Easttom, M.Ed, MSDS, MBA, MSSE, Ph.D.², D.Sc.

Hypersonics and Boundary Layers



Hypersonic Effects

PHENOMENON	DESCRIPTION	
Thin shock layer	The shrinking distance between the vehicle surface and shockwave at hypersonic speeds can induce additional stress.	
Entropy layer	The layer of high entropy vorticity near the vehicle's leading edges can cause unusual aerodynamic effects, leading to dynamic instability.	
Viscous interaction	The boundary layer—airflow around the vehicle body—thickens, interacts with the shockwave, and can increase heat and turbulence.	
Shock-shock interaction	The interaction between shockwaves of various vehicle features can complicate aerodynamic predictions.	
Boundary layer	This thin layer of air directly interacts with the surface of a hypersonic vehicle.	
Boundary layer transition	Hypersonic vehicles are engineered to maximize laminar boundary layer flows, where air travels in an ordered path over the vehicle surface. Changes in vehicle speed, surrounding air temperature, and others can cause laminar boundary layer flows to become turbulent, where air follows a chaotic path over the vehicle. This phenomenon is known as boundary layer transition. Turbulent boundary layer flows can impose significantly higher heat and vibration loads on the vehicle's surface.	
High-temperature effects	Dissociation of air molecules, plasma formation, internal changes to thermodynamic properties of air molecules, and off-gassing from heat shield materials complicates design of thermal protection systems and can induce electromagnetic interference and other challenges.	
Low-density flow	At high altitudes approaching 100 km, the physical characteristics of the atmosphere change considerably, resembling a series of discrete particles instead of continuous airflow. At the edge of space, air molecules striking a vehicle may never interact with other air molecules striking its surface.	

Karako, Tom; Dahlgren, Masao. Complex Air Defense: Countering the Hypersonic Missile Threat (CSIS Reports) Bloomsbury Publishing.

Dr. Chuck Easttom, M.Ed, MSDS, MBA, MSSE, Ph.D.², D.Sc.

LASTRAC

Langley Stability and Transition Analysis Code (LASTRAC) is a numerical code developed at NASA Langley Research Center for analyzing laminar-turbulent transition in boundary-layer flows. It is designed to solve linear and nonlinear stability equations that describe the evolution of small disturbances in the laminar boundary layer — critical for predicting transition to turbulence.

The main purpose of LASTRAC is to:

- Predict boundary-layer transition (from laminar to turbulent) under a wide variety of flow conditions.
- Analyze the growth of disturbances using linear stability theory (LST) and parabolized stability equations (PSE).
- Support high-speed flow applications such as supersonic and hypersonic vehicles, including transition over cones, wings, or capsules.

LASTRAC

LASTRAC includes several modules and features:

- 1. Linear Stability Theory (LST)
- Solves the Orr–Sommerfeld and Squire equations for 2D and 3D boundary-layer disturbances.
- Determines critical amplification rates (growth of Tollmien–Schlichting waves, crossflow modes, etc.).
- Identifies neutral stability curves (where disturbances neither grow nor decay).
- 2. Parabolized Stability Equations (PSE)
- Extends LST to include streamwise variation of the base flow and disturbances.
- Captures non-parallel effects, especially important in high-speed and spatiallyevolving flows.
- 3. Transition Prediction Methods
- eⁿ method: Empirical approach based on amplification of disturbances.
- Transition is assumed when:
- ſ Amplification Rate · dx≥n
- where n is a user-specified threshold (typically between 9 and 12 for flight conditions).

LASTRAC

Inputs:

- Velocity and temperature boundary layer profiles,
- Free-stream conditions (Mach, Reynolds number, pressure, temperature),
- Wall boundary conditions (temperature, roughness),
- Disturbance types (2D/3D, crossflow, acoustic, etc.).

Outputs:

- Growth rates of disturbances,
- Transition onset location (based on eⁿ),
- Amplification curves and stability contours,
- Sensitivity to freestream disturbances or wall conditions.
- LASTRAC has been validated extensively against:
- Wind tunnel experiments at NASA Langley and other research centers,
- Other computational tools like STABL, NEK5000, or DNS codes,
- Flight test data from programs like X-43A and Space Shuttle missions.

Governing Equations of the LASTRAC code

The nonlinear parabolized stability equations for the Fourier mode (m,n):

$$\hat{A}_{mn}\frac{\partial\psi_{mn}}{\partial x} + \hat{B}_{mn}\frac{\partial\psi_{mn}}{\partial y} + \hat{D}_{mn}\psi_{mn} - \frac{V_{yy}^l}{R_0}\frac{\partial^2\psi_{mn}}{\partial y^2} = \frac{F_{mn}}{A_{mn}}$$

Shape function:

$$\psi_{mn} = (\hat{p}_{mn}, \hat{u}_{mn}, \hat{v}_{mn}, \hat{w}_{mn}, \hat{T}_{mn})$$

Streamwise curvature and transverse curvature may be included in the computation

The left-hand side of the equation contains only linear coefficient matrices

Nonlinear forcing term:

$$F^{n} = -\Gamma^{n} \frac{\partial \phi}{\partial t} - \frac{A^{n}}{h_{1}} \frac{\partial \phi}{\partial x} - B^{n} \frac{\partial \phi}{\partial y} - \frac{C^{n}}{h_{3}} \frac{\partial \phi}{\partial z} - D^{n} \phi + \frac{1}{R_{0}} \left(\frac{V_{xx}^{n}}{h_{1}^{2}} \frac{\partial^{2} \phi}{\partial x^{2}} + \frac{V_{xy}^{n}}{h_{1}} \frac{\partial^{2} \phi}{\partial x \partial y} + V_{yy}^{n} \frac{\partial^{2} \phi}{\partial y^{2}} \right)$$

$$+ \frac{V_{xz}^{n}}{h_{1}h_{3}} \frac{\partial^{2} \phi}{\partial x \partial z} + \frac{V_{yz}^{n}}{h_{3}} \frac{\partial^{2} \phi}{\partial y \partial z} + \frac{V_{zz}^{n}}{h_{3}^{2}} \frac{\partial^{2} \phi}{\partial z^{2}} \right)$$

The equations are simplified for LST and Linear PSE.

Molecular Theory of Gas Flow

The Molecular Theory of Gas Flow is a framework that describes gas behavior based on the motions and interactions of individual molecules, rather than treating the gas as a continuous medium (as in classical fluid dynamics). This theory is especially important in hypersonic and rarefied gas flows, where the mean free path of molecules is comparable to or larger than the characteristic dimensions of the flow (e.g., spacecraft at high altitudes).

Kinetic Theory of Gases:

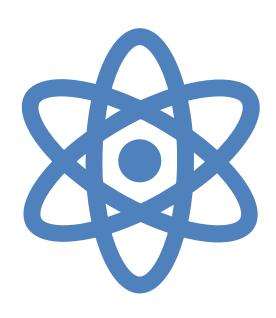
The theory assumes that a gas is composed of a large number of molecules in constant, random motion, colliding elastically with each other and with container walls.

Mean Free Path (λ):

The average distance a molecule travels between collisions:

$$\lambda = rac{k_B T}{\sqrt{2}\pi d^2 p}$$

where kB is the Boltzmann constant, T is temperature, d is molecular diameter, and p is pressure.



Molecular Theory of Gas Flow

The Maxwell-Boltzmann equation, which forms the basis of the kinetic theory of gases, defines the distribution of speeds for a gas at a certain temperature. From this distribution function, the most probable speed, the average speed, and the root-mean-square speed can be derived. The Maxwell-Boltzmann distribution describes the probability of molecular speeds:

$$f(v) = \left(rac{m}{2\pi k_B T}
ight)^{3/2} 4\pi v^2 e^{-rac{mv^2}{2k_B T}}$$

where m is molecular mass and v is molecular speed.

A good resource is

https://chem.libretexts.org/Bookshelves/Physical and Theoretical Chemistry Textbook Maps/Supplemental Modules (Physical and Theoretical Chemistry)/Kinetics/03%3A Rate Laws/3.01%3A Gas Phase Kinetics/3.1.02%3A Maxwell-Boltzmann Distributions

Molecular Theory of Gas Flow

MOLECULAR THEORY OF GAS FLOW

Continuum

$$K = \frac{\lambda}{L}$$

Molecular

Knudsen Regime

Continuum Kn < 0.01

Slip/Transitional 0.01 < Kn < 0.1

Free molecular flow Kn > 10

Navier-Stokes equations

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0$$

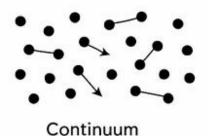
$$\frac{\partial \rho u}{\partial t} + \nabla \cdot (\rho \bar{u}u) + \nabla \rho = \nabla$$

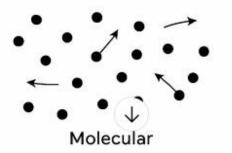
$$\frac{\partial E}{\partial t} + \nabla \cdot ((E + \dot{r})) = \nabla (kT)$$

$$f(v) = rac{m}{\left(rac{2\pi k_3 T}{m v^2}
ight)^{rac{2}{2}}} 4\pi rac{2}{ au}$$

Boltzmann equation

$$\frac{\partial f}{\partial t} + v \cdot \nabla f + \bar{\alpha} \cdot \frac{\partial f}{\partial v}_{\text{(collisions)}}$$





Hypersonic Free Molecular Flow

Free molecular flow occurs when a gas is so rarefied (low in density) that its molecules travel long distances without colliding with each other. In this regime, the mean free path of molecules is much larger than the characteristic length of the ootject in the flow.

When this happens at hypersonic speeds (Mach > 5), the flow is referred to as hypersonic free molecular flow — a combination of extremely high velocity and low-density conditions.

The Knudsen number (Kn) determines the flow regime:

$$Kn = \frac{\lambda}{L}$$

Where:

Λ Mean free path of gas molecules,

L: Characteristic length scale of the object.

Hypersonic Free Molecular Flow

Characteristics of Hypersonic Free Molecular Flow

- No Intermolecular Collisions
 - Molecules do not interact with each other only with surfaces.
 - Conventional fluid dynamics (e.g., Navier– Stokes equations) breaks down.
- Surface-Molecule Interaction Dominates
 - Flow properties are determined by how individual molecules reflect off surfaces.
 - Reflection models: specular, diffuse, or mixed (accommodation coefficients).
- Ballistic Trajectories
 - Molecules follow straight-line paths until they hit a surface.
- Non-continuum Behavior
 - No defined pressure or temperature fields in the classical sense.
 - Local properties must be interpreted statistically.
- Reentry and High-Altitude Relevance
 - Common in orbital reentry and upper atmosphere flight (altitudes above ~90 km).
 - Critical in spacecraft design for heat shield sizing and aerodynamic modeling.

Hypersonic Free Molecular Flow

In the free molecular regime, flow is typically modeled using kinetic theory rather than continuum equations. One important equation is Boltzmann Equation (Simplified):

$$\frac{\partial f}{\partial t} + v + \nabla f = \mathbf{0}$$

Where f(r,v,t): Molecular velocity distribution function.

Collisions are neglected, so the equation describes particles in ballistic motion.

Hypersonic Free Molecular Flow

Free Molecular Heat Flux Equation (for diffuse reflection)

$$q = \frac{1}{4} n\bar{c} (\bar{c}^2 - C_W^2)$$

Where:

n: number density,

 \bar{c} : average molecular speed,

c_w: speed of molecules re-emitted from the wall.

Skin Friction

Skin friction is the shear stress exerted by a fluid on a solid surface due to viscosity. In hypersonic flows, even though the pressure drag often dominates, skin friction contributes significantly to surface heating and can affect boundary layer development and transition to turbulence.

Hypersonic conditions (Mach > 5) involve:

High-temperature effects (dissociation, vibrational excitation),

Real gas behavior,

Thin boundary layers,

Strong pressure and thermal gradients.

These make skin friction estimation more complex than in subsonic or even supersonic regimes.

Skin Friction -Formulas

Laminar Flow - Flat Plate

Assuming ideal gas and constant properties:

$$C_f = \frac{0.664}{\sqrt{\text{Re}_x}}$$

Where

 C_f = Skin friction coefficient,

 Re_x = Reynolds number at distance x

The Reynolds number (Re) is a dimensionless quantity used to predict flow patterns in different fluid flow situations. It represents the ratio of inertial forces to viscous forces and is used to determine whether the flow will be laminar, transitional, or turbulent.

$$Re = \frac{p\mu L}{\mu} = \frac{uL}{v}$$

Where:

ρ: Fluid density (kg/m³),

u: Characteristic flow velocity (m/s),

L: Characteristic length (m),

μ: Dynamic viscosity (Pa·s),

 $v=\mu/\rho$: Kinematic viscosity (m²/s).

Flow regimes by Reynolds Number

Flow Type	Range of Reynolds Number	Description
Laminar Flow	Re<2,300	Smooth, orderly fluid layers
Transitional	2,300 <re<4,000< td=""><td>Unstable; flow may oscillate between laminar and turbulent</td></re<4,000<>	Unstable; flow may oscillate between laminar and turbulent
Turbulent Flow	Re>4,000	Chaotic, vortical motion, mixing

Characteristic Length L based on Geometry

Geometry	L Definition	
Pipe Flow	Pipe diameter D	
Flat Plate	Distance from leading edge x	
Sphere/Cylinder	Diameter D	
Airfoil	Chord length	

Sutherland's Law

In 1893 William Sutherland, an Australian physicist, published a relationship between the dynamic viscosity, µ, and the absolute temperature, T, of an ideal gas. This formula, often called Sutherland's law, is based on kinetic theory of ideal gases and an idealized intermolecular-force potential. Sutherland's law is still commonly used and most often gives fairly accurate results with an error less than a few percent over a wide range of temperatures. Sutherland's Law describes how the dynamic viscosity of a gas changes with temperature. It provides a more accurate model for gas viscosity than assuming a constant value, particularly in high-temperature flows such as those found in hypersonics and combustion. Sutherland's law can be expressed as:

$$\mu = \mu_{ref} \left(\frac{T}{T_{ref}}\right)^{3/2} \frac{T_{ref} + S}{T + S}$$

Where:

T_ref is a reference temperature.

μ_ref is the viscosity at the T_ref r S is the Sutherland temperature https://www.cfd-online.com/Wiki/Sutherland%27s_law

Sutherland, W. (1893), "The viscosity of gases and molecular force", Philosophical Magazine, S. 5, 36, pp. 507-531 (1893).

Sutherland's Law

As with almost all formulas, there are several ways to express Southerland's law, this is another way:

$$\mu=\mu_0\left(rac{T}{T_0}
ight)^{3/2}rac{T_0+S}{T+S}$$

where:

 μ = dynamic viscosity at temperature TTT, μ_0 = reference viscosity at a reference temperature T0T 0T0,

T = absolute temperature (in Kelvin), T_0 = reference temperature (often 273 K or 300 K),

S = **Sutherland's constant** (depends on the specific gas, in Kelvin).

Van Driest Transformation

The Van Driest Transformation is a mathematical method used in compressible fluid dynamics to transform velocity profiles from compressible turbulent boundary layers into an equivalent form that can be compared with incompressible velocity profiles. This transformation accounts for density variations and other compressibility effects that occur at high speeds, particularly in supersonic and hypersonic flows.

In compressible boundary layers, the velocity and temperature profiles are affected by large variations in density and viscosity. The Van Driest transformation adjusts the velocity profile so that it resembles an incompressible turbulent boundary layer when plotted in wall coordinates (law-of-the-wall). It is especially useful in engineering analyses of high-speed aerodynamic surfaces.

Van Driest Transformation

Transformed Velocity

$$u^+ = \int_0^u \left(rac{
ho}{
ho_w}
ight)^{1/2} rac{du}{u_ au}$$

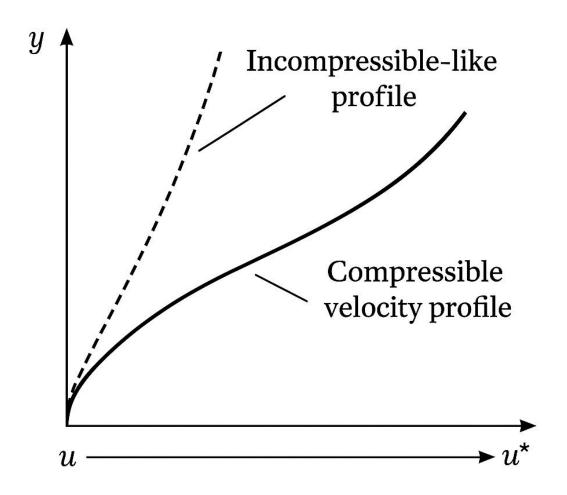
Where:

- ρ , ρ_w : local and wall densities,
- $u_{ au}$: friction velocity = $\sqrt{ au_w/
 ho_w}$

This transformation allows the use of incompressible law-of-the-wall relations in compressible flows

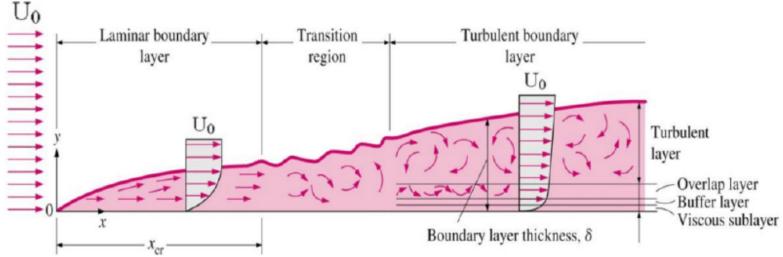
https://arc.aiaa.org/doi/abs/10.2514/3.12259 ?journalCode=aiaaj

Van Driest Transformation



Boundary Layer Concepts

- When air flows over a surface a thin layer is formed near the surface called the boundary layer—this layer contains friction effects
- Boundary layers start as laminar (smooth, layered) and then at some point starts to transition to turbulent (unsteady, chaotic)



https://www.nuclear-power.com/nuclear-engineering/fluiddynamics/boundary-layer/boundary-layer-thickness/

Boundary Layer Transition

In fluid flow over a surface, the boundary layer is a thin region near the surface where viscous effects are significant. The flow in this layer can be:

Laminar: Smooth, orderly motion with low skin friction and heat transfer.

Transitional: Unstable, disturbed flow with mixed laminar and turbulent characteristics.

Turbulent: Chaotic, vortical motion with much higher skin friction and heat transfer.

Transition refers to the process where flow changes from laminar to turbulent

In hypersonic flight (Mach > 5), boundary layer transition affects:

- Surface heating: Turbulent layers cause sharp increases in heat flux.
- Aerodynamic drag: Turbulent layers increase shear forces.
- Thermal protection system (TPS): Accurate transition prediction is vital for sizing and material selection.

Boundary Layer Transition

Transition is governed by the growth of small disturbances in the boundary layer. For hypersonic flows, several instability modes exist:

First Mode (Tollmien-Schlichting)

Similar to subsonic and supersonic instabilities. Viscosity-driven waves.

Second Mode (Mack Mode)

Unique to hypersonic flows.

Dominant at high Mach numbers.

High-frequency acoustic waves trapped in the boundary layer.

Crossflow Instabilities

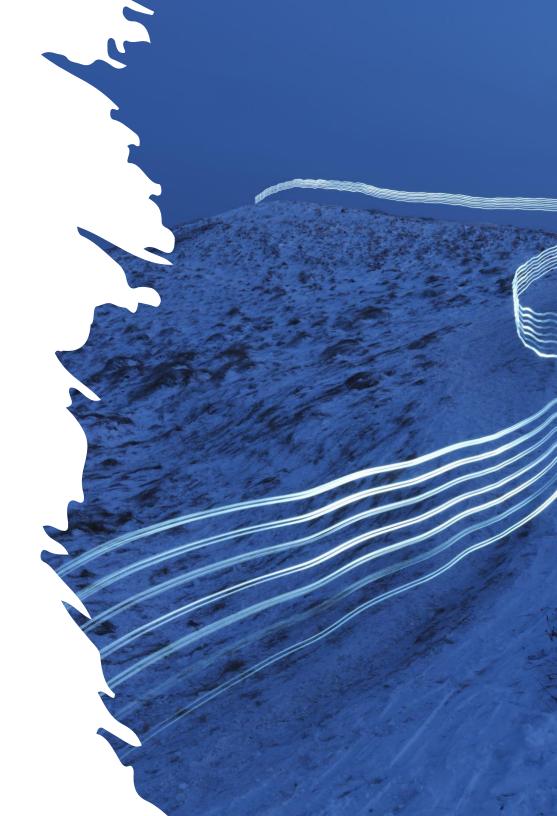
Occur in 3D boundary layers (e.g., swept wings).

Driven by spanwise velocity gradients.

Goertler Vortices

Caused by surface curvature, especially concave surfaces.

Lead to streamwise vortex formation.



Tollmien-Schlichting (T-S) waves

Tollmien—Schlichting waves are viscous instability waves that occur in the laminar boundary layer of a fluid flowing over a surface. They are small-amplitude disturbances that can grow exponentially under the right conditions, eventually triggering transition to turbulence.

They are named after:

Walter Tollmien (1929), who first predicted their existence theoretically.

Hermann Schlichting, who further developed the theory.

These waves are most commonly observed in incompressible or subsonic flows, but are also relevant in low-to-moderate supersonic flows.

T–S waves are generated due to the interaction between:

Viscous shear in the boundary layer, and

Pressure perturbations (usually from external disturbances).

Tollmien-Schlichting (T-S) waves

T–S waves arise from solving the linearized Navier–Stokes equations for a laminar base flow, leading to the Orr–Sommerfeld equation:

$$\left(U-c
ight)\left(\phi''-lpha^2\phi
ight)-U''\phi=rac{i}{lpha Re}\left(\phi''''-2lpha^2\phi''+lpha^4\phi
ight)$$

Where:

 $\phi(y)$: Streamfunction of the disturbance,

U(y): Base velocity profile,

c: Complex wave speed,

α: Streamwise wave number,

Re: Reynolds number,

Primes denote derivatives w.r.t. wall-normal direction y.

If the imaginary part of c > 0, the disturbance grows — signifying instability.

Characteristics of T–S Waves

Feature Description

Type Two-dimensional viscous instability

Wavelength On the order of the boundary layer thickness

Frequency range Typically, a few hundred Hz in low-speed flows

Speed Slightly less than the free-stream velocity

Affected by

Pressure gradients, wall temperature,
Reynolds number

Mack (2nd) Mode

The Mack Second Mode is a high-frequency instability that arises in hypersonic boundary layers (typically Mach > 4). It is named after Leonard Mack, who in the 1960s identified two main instability families in compressible boundary layers:

First Mode: Similar to subsonic Tollmien—Schlichting waves (viscous-shear driven).

Second Mode (Mack Mode): A compressibility-driven instability unique to high Mach number flows.

How it works:

In hypersonic boundary layers, acoustic waves become trapped between the wall and the sonic line (the region where local flow speed = speed of sound).

These trapped acoustic waves reflect between the wall and the sonic line, amplifying under certain flow conditions.

The high-frequency nature of this wave allows it to grow rapidly, triggering transition to turbulence.

Mack (2nd) Mode

Second-mode instability is studied via linear stability theory (LST) and Parabolized Stability Equations (PSE).

Perturbation form:

$$\psi(x,y,t) = \hat{\psi}(y)e^{i(\alpha x - \omega t)}$$

Where:

ψ: Disturbance quantity (velocity, pressure, etc.),

a: Streamwise wave number,

ω: Angular frequency.

Using these in the compressible Navier–Stokes equations leads to an eigenvalue problem. The Mack modes appear as eigenvalues with positive growth rates when conditions are favorable

Mack (2nd) Mode

Key Parameters affecting Mack Mode

Parameter Influence on Mack 2nd Mode

Mach number

Higher Mach → greater compressibility

→ stronger mode

Wall temperature Colder walls promote trapping and

amplification

Reynolds number Higher Re favors instability growth

Pressure gradient Weak effect, but adverse gradients can

enhance growth

Disturbance frequencyNeeds high frequency (typically > 100

kHz)

Crossflow instabilities

Crossflow instabilities are three-dimensional disturbances that develop in boundary layers over swept wings or cones at an angle of attack. These instabilities are driven by spanwise pressure gradients, causing the flow within the boundary layer to develop a crossflow component — a velocity vector not aligned with the external (free-stream) flow.

In flows over 3D geometries (e.g., swept wings):

- 1. The external flow has a component along the spanwise direction (due to sweep).
- 2. Inside the boundary layer, this causes a crossflow velocity component www, in addition to the streamwise component u.
- 3. The result is a helical (corkscrew-like) flow path within the boundary layer.
- 4. If the crossflow is strong enough and the pressure gradient is adverse, stationary or traveling vortices can grow.

Crossflow instabilities – Governing Factors

Factor Influence on Crossflow Instability

Wing sweep angle Greater sweep \rightarrow stronger crossflow

Pressure gradient Adverse pressure gradient promotes instability

Reynolds number Higher Re → earlier transition

Surface roughness

Can initiate stationary crossflow modes

Freestream turbulence Enhances traveling mode growth



Crossflow instabilities

In 3D boundary layers, transition can be dominated by crossflow rather than traditional 2D mechanisms. Here's how it unfolds:

Receptivity: Disturbances (e.g., from roughness or turbulence) enter the boundary layer.

Linear Growth: Crossflow vortices amplify due to the instability in the base flow profile.

Nonlinear Growth: Disturbances become strong and interact.

Breakdown: Leads to turbulent spots and full transition.

Crossflow Mitigation/Control

Method Purpose

Surface suction

Removes energy from growing vortices

Targeted roughness Triggers stable modes early (controlled transition)

Shape optimization Minimizes crossflow velocity

Laminar Flow Control Actively maintains laminar **(LFC)** conditions

Görtler Vortices

Görtler vortices are streamwise-aligned vortical structures that develop in the boundary layer when it flows over a concave surface. They are caused by centrifugal forces acting on fluid elements due to the curvature of the wall. Named after German physicist Ludwig Görtler, who first described this instability in 1940. These vortices are a three-dimensional instability, meaning they require 3D flow disturbances and geometric curvature to develop. These form due to:

- When fluid flows over a concave wall, particles in the boundary layer experience centrifugal acceleration directed outward (away from the surface).
- If the stabilizing effect of viscous diffusion is not strong enough to counter this acceleration, the flow becomes unstable.
- This leads to the formation of counter-rotating streamwise vortices — the Görtler vortices.
- These vortices draw fluid up and down in alternating streaks, resulting in regions of high and low momentum.

Görtler instability

The tendency for Görtler instability is characterized by the Görtler number:

$$G = \frac{Ue\theta}{v} \sqrt{\frac{\theta}{R}}$$

Where:

Ue: Edge velocity of the boundary layer,

θ: Momentum thickness,

v: Kinematic viscosity,

R: Radius of curvature (positive for concave surfaces)

Comparisons

Instability Mode	Dominant Regime	Key Driver
Tollmien- Schlichting	Low-Mach, incompressible	Viscous instability
Mack (2nd) Mode	Hypersonic	Acoustic-trapped waves
Crossflow	3D swept flows	Spanwise pressure gradients
Goertler	Concave surfaces	Centrifugal instability

Transition Prediction eⁿ Method

Uses linear stability theory.

Predicts transition when disturbance amplification reaches a threshold:

$$n = \ln\left(\frac{A}{A_0}\right)$$

Where:

A: Final disturbance amplitude,

A₀: Initial amplitude,

n: Empirical threshold (typically n=9-12n=9-12).

Tangent Wedge Theory

Tangent Wedge Theory is based on the idea that any smooth, slender surface can be locally approximated by a series of flat wedge segments tangent to the actual surface profile.

By applying known shock wave relations (typically oblique shock relations) to each wedge segment, one can estimate:

- Local pressure distribution,
- Surface shear and heat flux,
- Overall aerodynamic coefficients (lift, drag).

This is particularly useful for preliminary analysis in hypersonic flow regimes (Mach > 5), where traditional CFD or experiments may be expensive or unavailable. The basic assumptions are:

- The body is slender (small angle of attack and small body slopes),
- The surface is treated as a series of planar wedges,
- The flow is hypersonic, so shock angles are much smaller than in subsonic or transonic flow,
- The pressure on each segment is determined from oblique shock theory,
- The method assumes inviscid, compressible, steady flow.

Tangent Wedge Theory

Key Steps in Tangent Wedge Theory

Break the body into segments:

Divide the surface into discrete panels or small wedges, each with its local inclination angle theta $\boldsymbol{\theta}$

Apply oblique shock relations:

For each wedge, assume a shock wave forms due to the local deflection θ theta,

Use oblique shock equations to find pressure ratio p2/p1, temperature, and Mach number behind the shock.

Integrate pressure over the surface:

Combine pressures from all wedge segments to calculate net lift, drag, and moment.

Tangent Wedge Theory

While the method is geometric and piecewise in nature, it relies on oblique shock equations, such as:

$$an heta=2\coteta\left(rac{M_1^2\sin^2eta-1}{M_1^2(\gamma+\cos2eta)+2}
ight)$$

Where:

- θ: wedge deflection angle,
- β: shock angle,
- M₁: upstream Mach number,
- γ : specific heat ratio (1.4 for air).

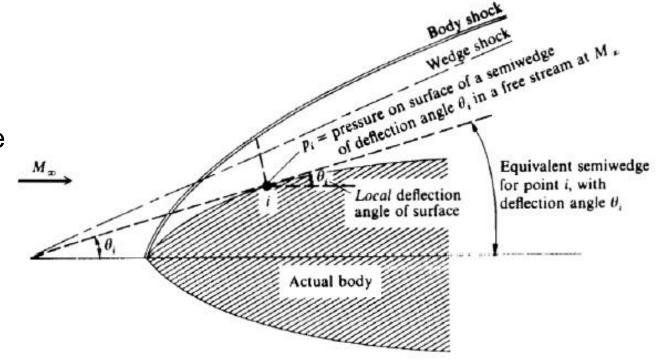
And the pressure ratio:

$$rac{p_2}{p_1}=1+rac{2\gamma}{\gamma+1}\left(M_1^2\sin^2eta-1
ight)$$

Once pressure p_2 is known for each segment, **integrate** over the surface to get total forces.

Tangent Wedge Theory

Assume local pressure coefficient is the same as a 2D wedge with the same local slope



Tangent Cone Theory

Tangent Cone Theory is a simplified aerodynamic analysis method used primarily for axisymmetric slender bodies (like cones or ogives) in hypersonic flow. It approximates a complex surface by a sequence of tangent cones, each of which is analyzed using conical flow theory. It is ideal for sharp, slender, rotationally symmetric shapes traveling at Mach > 5. The fundamental ideas are:

- A curved, axisymmetric body (e.g., an ogive) is decomposed into infinitesimally small cone elements.
- At each point on the surface, the local geometry is approximated by a cone tangent to the surface.
- The shock angle and surface pressure are calculated using conical shock wave relations.

Tangent Cone Theory

Assumptions:

Flow is inviscid, compressible, and steady.

Body is axisymmetric and sharp-nosed.

Valid at **high Mach numbers** (typically M>5).

Only attached shocks are considered.

Viscous effects, heat transfer, and chemical reactions are neglected.

Tangent Cone Theory

For a cone of half-angle θ in hypersonic flow:

- The shock wave is attached and conical, forming an angle β with the cone axis.
- Pressure coefficient on the surface C_p is derived from oblique/conical shock theory:

$$C_p = rac{2}{\gamma M^2} \left(rac{p_2}{p_1} - 1
ight)$$

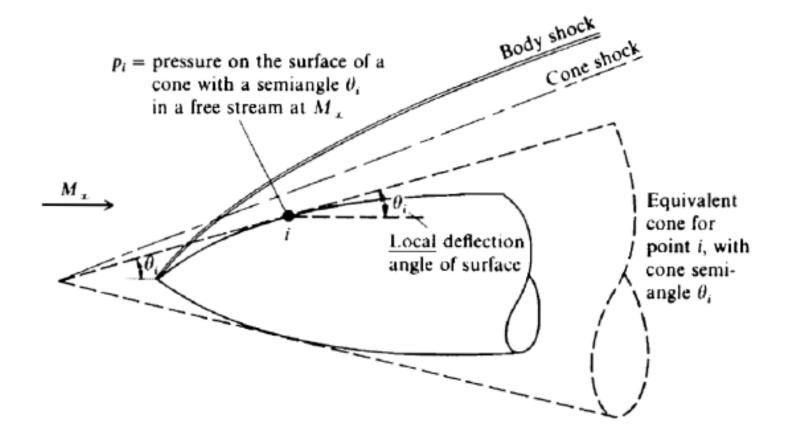
Where:

- M: freestream Mach number,
- p_2/p_1 : pressure ratio across the shock (from conical flow theory),
- γ: specific heat ratio.

Using Taylor-Maccoll equations (for inviscid, axisymmetric supersonic flow), you can compute the flow field around each cone.

Tangent Cone Theory

Assume local pressure coefficient is the same as a cone with the same local slope



Comparison

Feature/Aspect	Tangent Wedge Theory	Tangent Cone Theory	CFD (Computational Fluid Dynamics)
Basic Concept	Approximates curved surfaces as a series of flat wedges	Approximates axisymmetric bodies as a series of tangent cones	Solves Navier-Stokes or Euler equations numerically
Applicable Geometry	2D or quasi-3D slender bodies	Axisymmetric slender bodies	Any geometry (2D or 3D, simple or complex)
Mach Number Range	Hypersonic (typically M > 5)	Hypersonic (typically M > 5)	All Mach regimes (subsonic to hypersonic)
Accuracy	Good for sharp, slender 2D shapes at high Mach	Good for axisymmetric shapes (e.g., cones)	High fidelity; accuracy depends on grid & model
Computational Cost	Very low (analytical/hand-calculable)	Very low (analytical/hand-calculable)	High (requires numerical solvers & hardware)
Flow Physics Captured	Inviscid flow, pressure only (shock relations)	Inviscid, shock-based pressure approximation	Viscous, compressible, turbulent, unsteady, chemistry-capable
Shock Capture	Uses oblique shock theory	Uses conical shock theory	Directly resolves shocks
Viscous Effects?	X No (inviscid only)	× No	Yes (if viscous model used)
Time Dependence?	X No (steady only)	× No	Yes (transient simulations possible)
Use Cases	Hypersonic lifting surfaces (e.g., sharp waveriders)	Sharp-nosed axisymmetric bodies (e.g., cones, ogives)	All applications: design, optimization, heating, flow control
Turnaround Time	Minutes	Minutes	Hours to days

Dr. Chuck Easttom, M.Ed, MSDS, MBA, MSSE, Ph.D.², D.Sc.

Laminar Theory refers to the mathematical and physical modeling of laminar flow, a flow regime where fluid particles move in smooth, orderly layers (or laminae) with minimal mixing across streamlines. It provides foundational tools for analyzing viscous, low-disturbance flows — typically occurring at low Reynolds numbers or near the leading edge of aerodynamic surfaces.

Feature	Description
Streamlines	Smooth, non-intersecting, and parallel to each other
Flow layers	Fluid moves in adjacent layers without cross-stream mixing
Momentum transport	Primarily via molecular diffusion (viscosity), not turbulence
Predictability	High — governed by deterministic PDEs (Navier–Stokes)
Shear stress	Linear relation with velocity gradient (Newtonian assumption)

Laminar theory is rooted in the Navier–Stokes equations, simplified under certain assumptions. For incompressible, steady, 2D laminar flow:

$$ho \left(u rac{\partial u}{\partial x} + v rac{\partial u}{\partial y}
ight) = -rac{\partial p}{\partial x} + \mu \left(rac{\partial^2 u}{\partial x^2} + rac{\partial^2 u}{\partial y^2}
ight)$$

Where:

- ρ: density,
- μ: dynamic viscosity,
- u, v: velocity components,
- p: pressure.

The boundary layer approximation leads to simpler forms for external flows over surfaces.

Laminar Boundary Layer Theory

Laminar Boundary Layer Theory was developed by Ludwig Prandtl (1904). In laminar boundary layers:

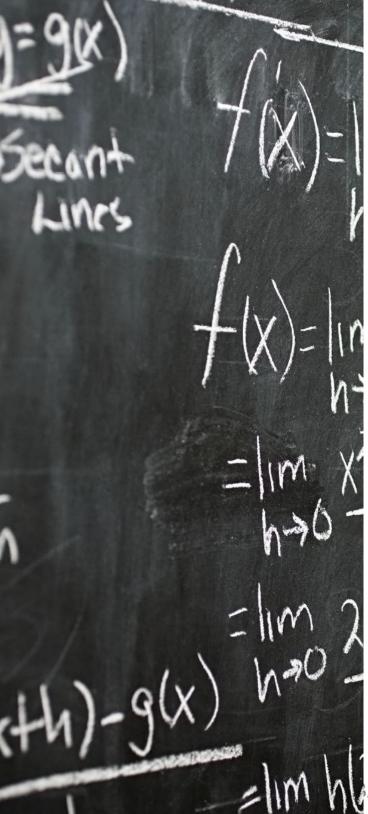
Flow adjacent to a surface slows due to viscous drag,

Velocity increases smoothly from 0 (at wall) to freestream U∞

Laminar flow is sensitive to disturbances (vibrations, roughness, acoustics). When instability thresholds are exceeded, the flow transitions to turbulent, often via:

Tollmien—Schlichting waves (viscous instability),

Crossflow or Görtler vortices (3D instabilities).



Low speed incompressible boundary layer theories: Poiseuille flow and Couette flow, which are exact solutions of the Navier- Stokes equations

• Couette flow solution

$$\frac{u}{U} = \frac{y}{h} + P \frac{y}{h} \left(1 - \frac{y}{h} \right)$$
Where $P = -\frac{h^2}{2\mu U} \frac{dp}{dx}$ is a non-dimensional pressure gradient

A, MSSE, Ph.D.², D.Sc.

- Other more complicated flows require a change of variables
- Blasius solution for flow over a flat plate:

$$ff^{\prime\prime}+f^{\prime\prime\prime}=0$$

Where $\eta = \frac{u_e y}{\sqrt{2 v s}}$ and $f' = \frac{u}{u_e}$ are transformed variables

 Solve the ODE to find boundary layer profiles (β is pressure gradient for Falker-Skan solution)

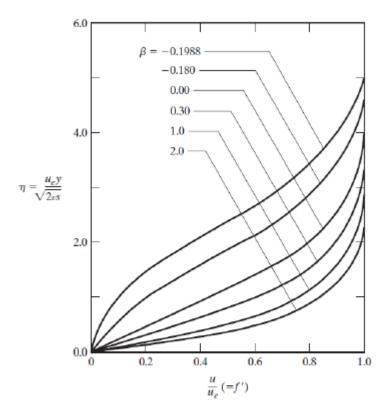


Figure 4.3 Solutions for the dimensionless streamwise velocity for the Falkner-Skan, laminar, similarity flows.

This is a similarity solution that can be used to find various boundary layer parameters:

Boundary layer thickness and displacement thickness

$$\delta = \frac{5.0x}{\sqrt{Re_x}} \qquad \delta^* = \frac{1.72x}{\sqrt{Re_x}}$$

• Momentum thickness and local skin friction coefficient
$$\theta = \frac{0.664x}{\sqrt{Re_x}} \qquad C_f = \frac{0.664}{\sqrt{Re_x}}$$

And most importantly the total skin friction coefficient

$$\overline{C}_f = \frac{1.328}{\sqrt{Re_L}}$$

Allows for prediction of skin friction over vehicles

Laminar Flow Summary

Term	Description
Laminar Flow	Smooth, layered flow with no turbulence
Laminar Theory	Mathematical framework for analyzing viscous laminar flows
Key Equation	Navier–Stokes (simplified in boundary layers)
Key Application	Aerodynamic surfaces, thermal analysis, drag reduction
Transition Trigger	Instability growth or external disturbances

Taylor-Couette instability

The Taylor—Couette flow describes the motion of a viscous fluid contained between two coaxial cylinders, where:

- The inner cylinder rotates,
- The outer cylinder may be stationary or rotating.

As the inner cylinder's rotation rate increases beyond a critical threshold, the previously laminar, purely azimuthal (circular) flow becomes unstable and develops into Taylor vortices — a series of axisymmetric, toroidal (doughnut-shaped) vortices stacked along the axis of the cylinders.

This transition from stable laminar flow to vortex flow is known as the Taylor-Couette instability, and it is caused by a balance between centrifugal forces and viscous damping.

This basic state is known as circular Couette flow, after Maurice Marie Alfred Couette, who used this experimental device as a means to measure viscosity. Geoffrey Ingram Taylor examined the stability of the Couette flow.

Taylor-Couette instability

The main control parameter is the Taylor number (Ta), a dimensionless number that measures the ratio of centrifugal to viscous forces:

$$\mathrm{Ta} = rac{(r_1 \cdot \Omega_1 \cdot d)^2}{
u^2}$$

Where:

r1: radius of inner cylinder,

Ω1: angular velocity of inner cylinder,

d=r2-r1: gap between cylinders,

v: kinematic viscosity.

Instability sets in when the Taylor number exceeds a critical value, typically around Tacrit≈1700 (for the simplest case with stationary outer cylinder).

Taylor-Couette instability

In the basic Couette flow, the fluid experiences no radial motion — only tangential.

At higher rotation speeds, centrifugal forces dominate over viscous damping, making the flow unstable to radial perturbations.

These perturbations evolve into Taylor vortices, which efficiently transport angular momentum outward — similar to convection in rotating fluids.

Taylor vortices

Taylor vortices refer to a specific type of fluid flow instability that occurs in a couette flow setup, which consists of a fluid trapped between two rotating cylinders. The Taylor vortex flow represents a pattern of vortices that form in a rotating fluid when the flow transitions from laminar to turbulent.

These vortices are named after the British mathematician G.I. Taylor, who first described them in 1923 while studying the stability of flow between concentric rotating cylinders. The vortices appear in systems where there is a velocity difference between two concentric cylinders—one is stationary and the other is rotating. The resulting flow pattern can exhibit a variety of states, depending on factors like the rotational speed, fluid viscosity, and Reynolds number.

Taylor vortices

Flow Setup:

Consider two concentric cylinders: the inner cylinder rotates at a constant angular velocity, while the outer cylinder is either stationary or rotates at a different speed.

The fluid trapped between the cylinders experiences a shear flow as the inner cylinder moves faster than the outer cylinder. Formation of Taylor Vortices:

At low to moderate rotational speeds (low Reynolds numbers), the flow remains laminar, and there's no formation of vortices.

As the rotational speed increases, the flow becomes unstable, and the fluid develops into a series of circular vortices (also called Taylor vortices).

These vortices are aligned along the axis of the cylinders and typically form in pairs. The vortices act like "tubes" that rotate in opposite directions, creating a flow pattern that resembles concentric rings of swirling fluid.

Formation Conditions:

Taylor vortices form when the Reynolds number (which is a function of fluid velocity, fluid viscosity, and characteristic length) exceeds a certain critical value.

For flow between concentric cylinders, the onset of Taylor vortices typically occurs when the Reynolds number reaches around 1000–2000, depending on specific system conditions.

The pattern and number of vortices formed can depend on factors like the rotation speed, the radius of the cylinders, and the fluid properties.

Taylor vortices

Flow Structure:

The vortices themselves are usually axisymmetric, meaning they are symmetric around the centerline of the system, and they form in a regular, repeating pattern along the axis of the cylinders.

At higher Reynolds numbers, more complex patterns such as turbulence or secondary flows (such as traveling waves) may develop, but Taylor vortices are often the first sign of instability before the flow becomes fully turbulent.

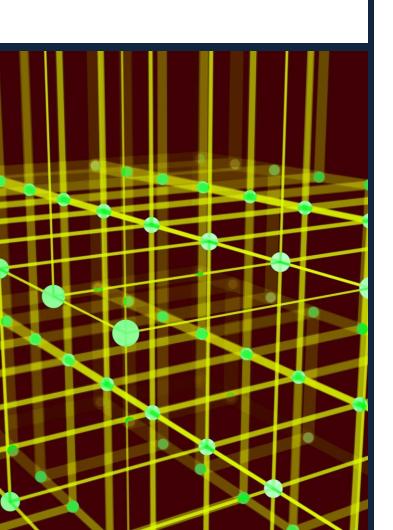
Regimes of Flow:

Laminar Flow: Below the critical Reynolds number, the flow remains smooth and ordered with no vortices.

Taylor Vortex Flow: Between the critical Reynolds number and the onset of turbulence, the flow transitions to Taylor vortices.

Turbulent Flow: At even higher Reynolds numbers, the flow may transition to fully turbulent motion, where chaotic fluctuations dominate

The Lattice Boltzmann Method (LBM)



The Lattice Boltzmann Method (LBM) is a computational fluid dynamics (CFD) technique that simulates fluid flows by modeling the microscopic particle dynamics rather than directly solving the macroscopic Navier–Stokes equations. It's based on kinetic theory — instead of tracking individual molecules (which would be too expensive), LBM tracks particle distribution functions on a discrete lattice in space and time.

LBM starts from the Boltzmann equation — a statistical equation that describes how the probability distribution f(x,v,t)f of particle velocities evolves over time due to streaming and collisions.

It then discretizes:

- 1. Space into a lattice (grid points).
- Velocity space into a small set of discrete directions.
- 3. Time into discrete steps.

The simulation alternates between:

- Streaming step: Particles move along the lattice links to neighboring nodes.
- Collision step: Particle distributions at each node relax toward an equilibrium distribution (often given by the BGK model — Bhatnagar–Gross–Krook).

The Lattice Boltzmann Method (LBM)

In its simplest form (Bhatnagar–Gross–Krook approximation):

$$f_i(\mathbf{x}+\mathbf{c}_i\Delta t,t+\Delta t)=f_i(\mathbf{x},t)-rac{\Delta t}{ au}\left[f_i(\mathbf{x},t)-f_i^{eq}(\mathbf{x},t)
ight]$$

 f_i = particle distribution in direction ii.

c_i = discrete velocity vector for direction ii.

 τ = relaxation time (linked to viscosity).

fi_{eq} = local equilibrium distribution (usually from Maxwell–Boltzmann statistics).

Panel Method

The panel method is an analysis method that can be used to arrive at an approximate solution for the forces acting on an object in a flow. The method is based on inviscid flow analysis, so it is limited to the resultant pressure forces over the surface. The panel method is basically a numerical approximation that relies on using discrete elements on the surface of an object and then prescribing a flow element (such as a vortex or doublet or source or sink) on each element that will satisfy certain boundary conditions (like no flow crosses the surface of the object). The interaction of the elements are accounted for and must also satisfy the condition that far from the object the flow should be equal to the free stream velocity approaching the object. There are a number of books and papers written that describe the method in very general terms and even the inclusion of viscous forces to some degree. More complicated models exist but they all are based on the simplified form presented here.

https://open.oregonstate.education/intermediate-fluid-mechanics/chapter/the-panel-method-an-introduction/

Panel Methods

Panel theory (or panel methods) models the aerodynamic surface (like an airfoil or wing) as being covered with a distribution of simple mathematical flow elements — usually sources, sinks, and vortices — placed on small surface segments called panels.

Each panel induces a velocity field in the surrounding flow.

By superimposing the effects of all panels, we approximate the total flow field around the body.

The strength of each panel's source/vortex is solved so that boundary conditions (like the no-penetration condition at the body surface) are satisfied.

This transforms a difficult partial differential equation problem (Laplace's equation for potential flow) into a linear algebra problem.

Panel Methods

Panel theory is based on potential flow theory:= and has a few assumptions:

- Inviscid (no viscosity)
- Incompressible (often assumed, though compressible extensions exist)
- Irrotational (flow has a velocity potential function)

This means panel theory cannot capture viscous effects like drag from skin friction or boundary-layer separation. It mostly predicts lift and pressure distributions.

Panel Methods

For an airfoil:

Place panels along the surface.

Assign unknown source/vortex strengths.

Apply boundary conditions: no normal velocity through the surface.

Apply the Kutta condition at the trailing edge (enforces smooth flow off the airfoil).

Solve the resulting system of equations for panel strengths.

Skin Friction Methods

Developer	Equation	Accuracy
Prandtl	$C_f = \frac{0.074}{\text{Re}_L^{\frac{1}{5}}}$	±25%
Prandtl/ Schlichting	$C_f = \frac{0.455}{\left(\log \text{Re}_L\right)^{2.58}}$	±3%
Karman/ Schoenherr	$\frac{1}{\sqrt{C_f}} = 4.13 \log(\text{Re}_L C_f)$	±2%
Schultz/ Grunon	$C_f = \frac{0.427}{\left(\log \text{Re}_L - 0.407\right)^{2.64}}$	±7%

from F.M. White, Viscous Fluid Flow, New York: McGraw-Hill, 1974, p. 501

Heat Transfer Prediction

 Typically, the heat transfer coefficient can be related to skin friction using Reynolds analogy

$$C_h = \frac{C_{f,inc}}{2Pr^{2/3}}$$

The Prandtl number is (defined in previous lessons):

$$Pr = \frac{\mu c_p}{k}$$

Where for $\gamma = 1.4$, Pr = 0.690 (which is an assumption that is valid for incompressible flow but which is questionable for hypersonic flow)