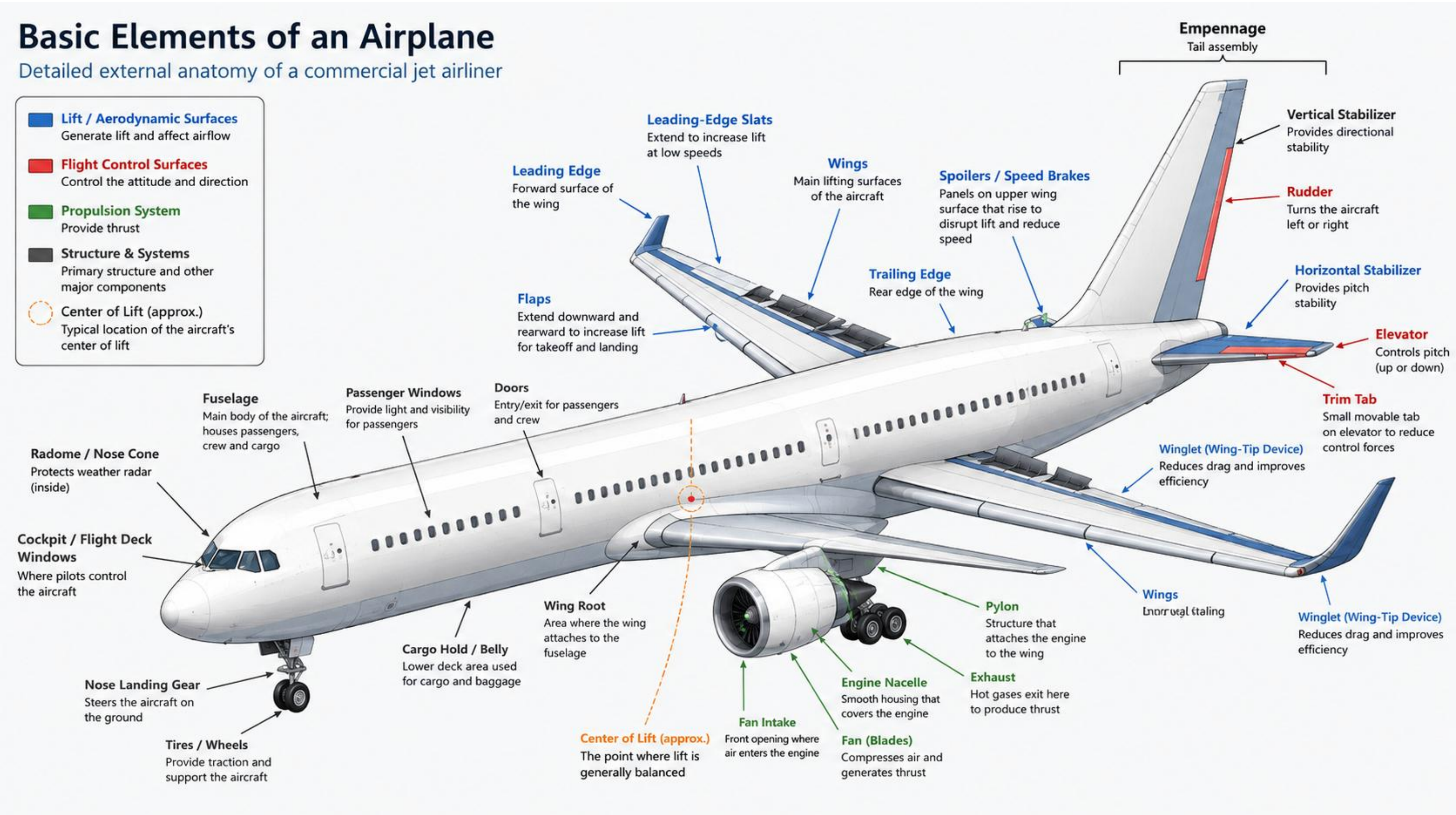
Elementary Aerospace Engineering for Hypersonics

Lesson 2

Dr. Chuck Easttom



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

Components

- An aileron is a hinged flight control surface located on the trailing edge of an aircraft’s wing, usually near the wingtip. Its main purpose is to control the aircraft’s roll (rotation around the longitudinal axis, which runs nose to tail).
  - When the pilot moves the control stick (or yoke) to the right, the right aileron goes up and the left aileron goes down.
  - This decreases lift on the right wing (aileron up) and increases lift on the left wing (aileron down), causing the aircraft to roll right.
- A leading-edge slat is a movable aerodynamic surface installed on the front (leading edge) of an aircraft wing. Its purpose is to improve the wing’s performance at low speeds, especially during takeoff and landing, when lift is critical.
- A rudder is a primary flight control surface located on the vertical stabilizer (tail fin) of an aircraft. Its main function is to control the aircraft’s movement around the yaw axis (nose left or right), which is the vertical axis running through the center of the plane.

Basic Elements of a Hypersonic Aircraft

Detailed external anatomy of a high-speed air-breathing hypersonic vehicle (conceptual)

- **Lift / Aerodynamic Surfaces** — Generate lift and control airflow
- **Flight Control Surfaces** — Control attitude and direction
- **Propulsion System** — Provide thrust
- **Structure & Systems** — Primary structure and other major components
- **Center of Lift (approx.)** — Typical location of the vehicle's center of lift / center of pressure

Tail / Aft Control Surfaces

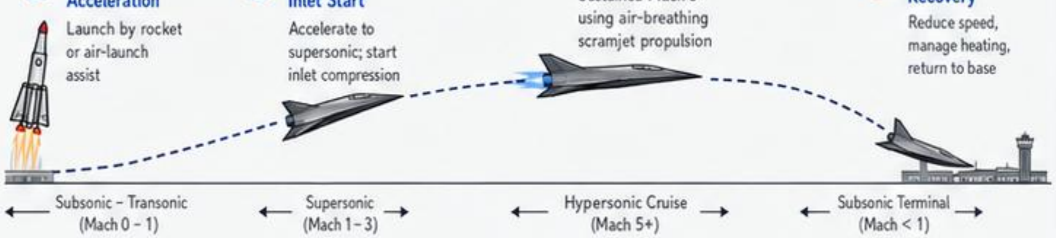
Rudders
On canted vertical tails for yaw control

Elevons
On trailing edge of wings for pitch and roll control

Body Flap (Aft Control Flap)
On upper aft fuselage for pitch trim and stability control

Typical Hypersonic Flight Regime

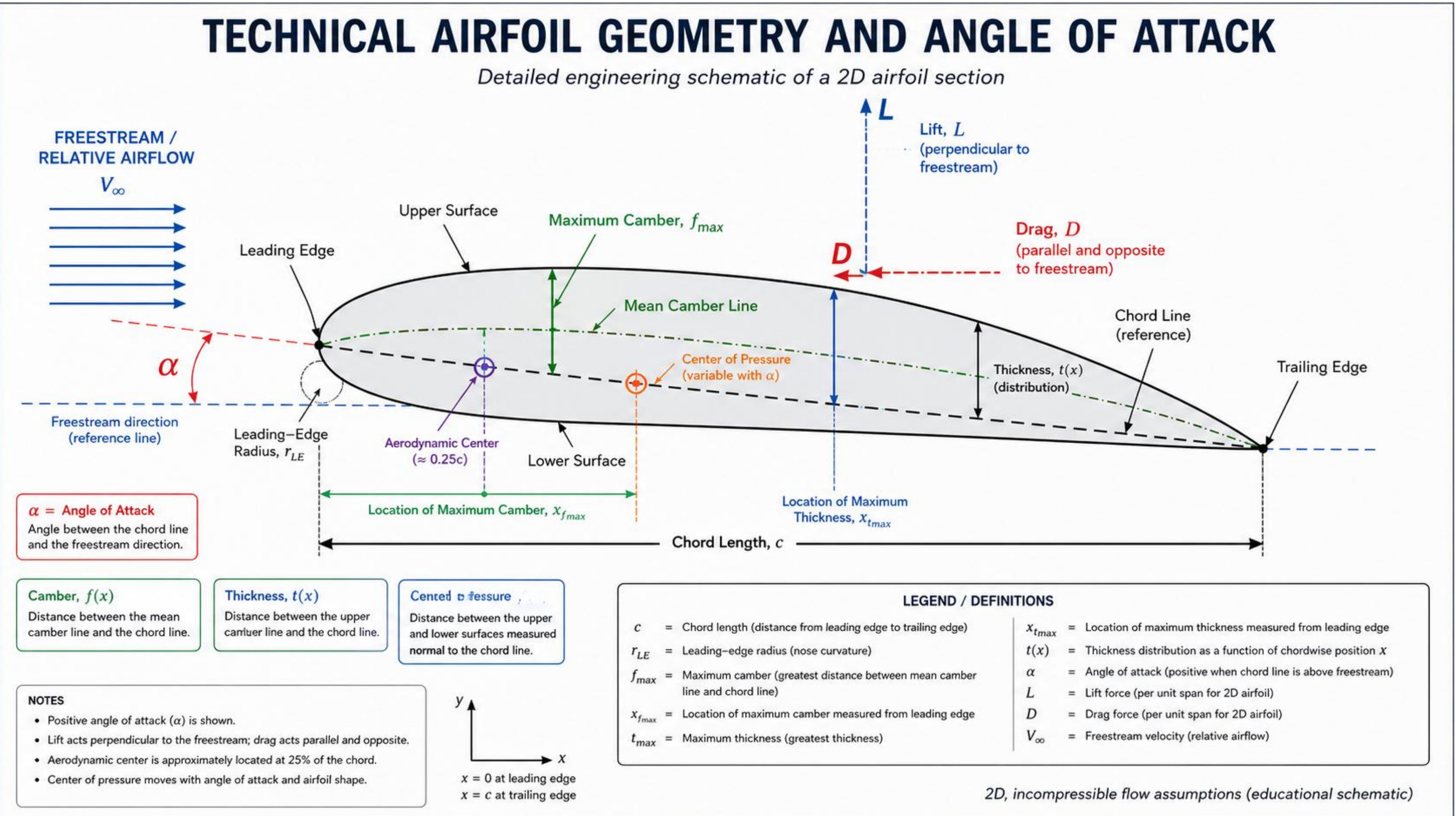
1. **Boost / Acceleration** — Launch by rocket or air-launch assist
2. **Climb and Inlet Start** — Accelerate to supersonic; start inlet compression
3. **Hypersonic Cruise** — Sustained Mach 5+ using air-breathing scramjet propulsion
4. **Descent / Recovery** — Reduce speed, manage heating, return to base



Key Characteristics (conceptual)

- **Flies at Mach 5+** — Extremely high speeds for rapid global reach
- **Requires stable inlet airflow for engine operation** — Inlet unstart must be avoided
- **Experiences intense aerodynamic heating** — Surface temperatures can exceed 1,500°C
- **Integrated air-breathing propulsion system** — Scramjet enables efficient high-speed sustained flight
- **Uses compression lift and high-temperature materials** — Minimizes drag and withstands extreme environments
- **Designed for sustained high-speed flight** — Optimized for range, payload, and mission flexibility

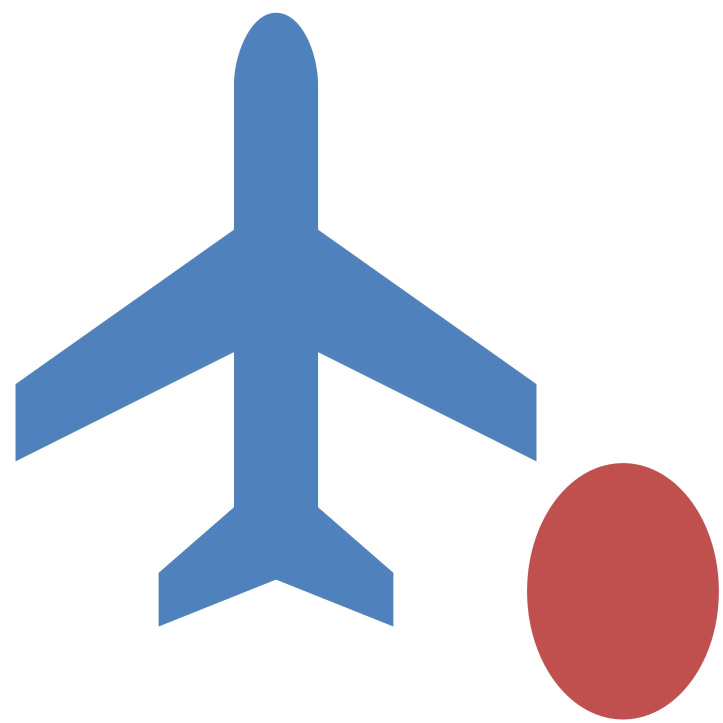
Airfoil



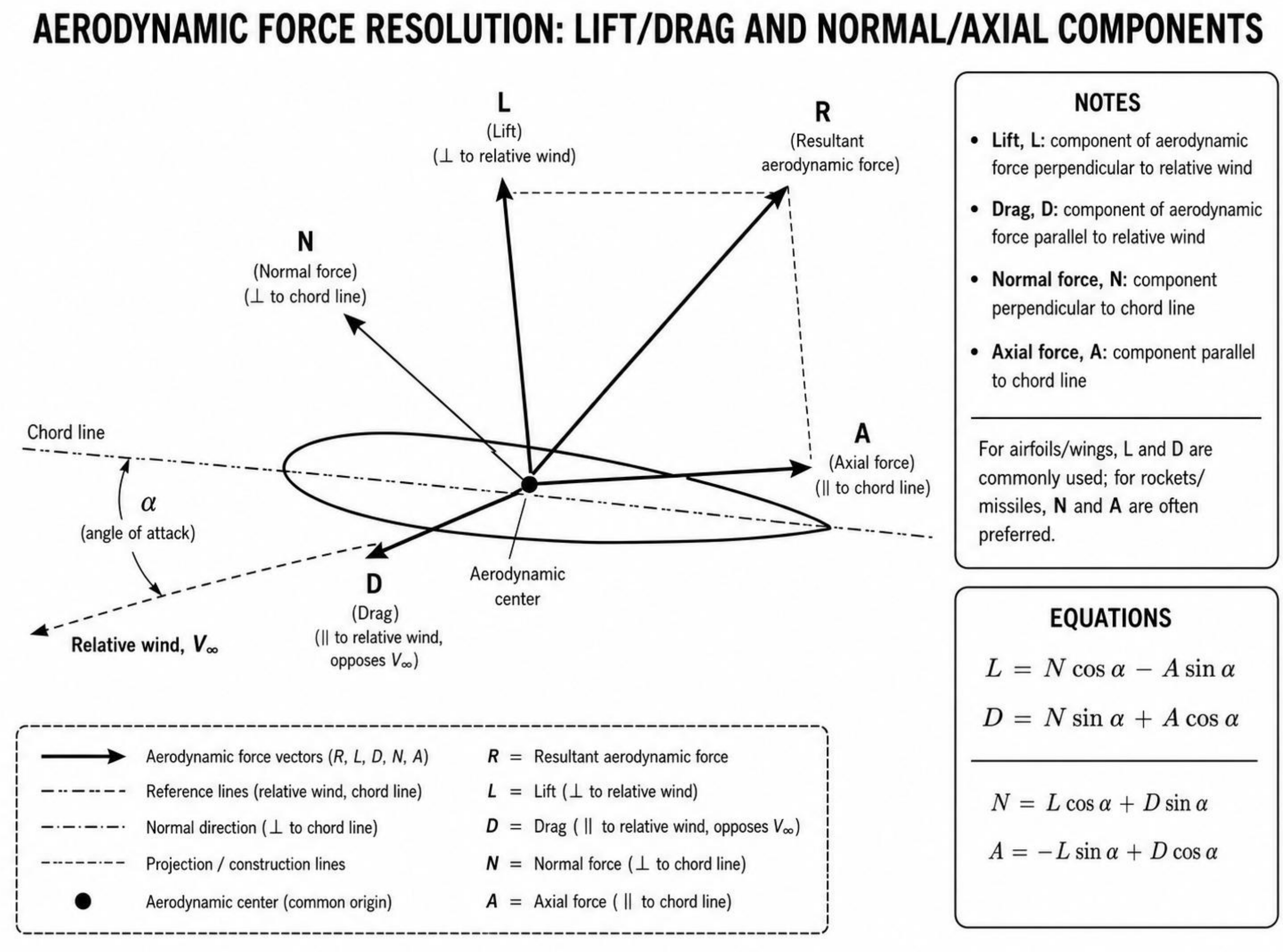
TECHNICAL AIRFOIL GEOMETRY AND ANGLE OF ATTACK
Detailed engineering schematic of a 2D airfoil section
FREESTREAM / RELATIVE AIRFLOW
V_∞
L
Lift, L (perpendicular to freestream)
Upper Surface
Maximum Camber, f_{max}
D
Drag, D (parallel and opposite to freestream)
Leading Edge
Mean Camber Line
Chord Line (reference)
α
Center of Pressure (variable with α)
Thickness, $t(x)$ (distribution)
Trailing Edge
Freestream direction (reference line)
Leading–Edge Radius, r_{LE}
Aerodynamic Center (≈ 0.25c)
Lower Surface
Location of Maximum Camber, $x_{f_{max}}$
Location of Maximum Thickness, $x_{t_{max}}$
Chord Length, c
α = Angle of Attack
Angle between the chord line and the freestream direction.
Camber, $f(x)$
Distance between the mean camber line and the chord line.
Thickness, $t(x)$
Distance between the upper caniuer line and the chord line.
Distance between the upper and lower surfaces measured normal to the chord line.
LEGEND / DEFINITIONS
c = Chord length (distance from leading edge to trailing edge)
r_{LE} = Leading–edge radius (nose curvature)
f_{max} = Maximum camber (greatest distance between mean camber line and chord line)
$x_{f_{max}}$ = Location of maximum camber measured from leading edge
t_{max} = Maximum thickness (greatest thickness)
$x_{t_{max}}$ = Location of maximum thickness measured from leading edge
$t(x)$ = Thickness distribution as a function of chordwise position x
α = Angle of attack (positive when chord line is above freestream)
L = Lift force (per unit span for 2D airfoil)
D = Drag force (per unit span for 2D airfoil)
V_∞ = Freestream velocity (relative airflow)
NOTES
• Positive angle of attack (α) is shown.
• Lift acts perpendicular to the freestream; drag acts parallel and opposite.
• Aerodynamic center is approximately located at 25% of the chord.
• Center of pressure moves with angle of attack and airfoil shape.
y
x
x = 0 at leading edge
x = c at trailing edge
2D, incompressible flow assumptions (educational schematic)

Angle of Attack

- The angle of attack is the angle between the oncoming airflow and the chord line of the wing (the straight line between the leading and trailing edges). A higher angle of attack increases lift—up to a point. Beyond a critical angle, the airflow separates from the surface of the wing, leading to a stall, where lift dramatically decreases, and the aircraft may lose altitude.



THE AERODYNAMIC FORCE ON A WING



Wing Design

- **Camber** is a key design feature of airfoils that affects lift. Camber refers to the curvature of the airfoil from the leading edge to the trailing edge. A highly cambered airfoil has a pronounced curve, which increases lift by enhancing the pressure differential. However, increased camber also tends to increase drag, which can reduce the efficiency of the aircraft at higher speeds. Modern aircraft typically use a moderate camber that balances lift and drag for efficient cruising speeds.
- **Thickness** is another important element in airfoil design. The thickness is the maximum distance between the upper and lower surfaces of the airfoil. A thicker airfoil creates more lift at lower speeds because it increases the surface area and accelerates the airflow over the wing. However, at high speeds, thick airfoils can cause excessive drag, so designers must find a balance that allows for good lift generation without creating too much drag.
- The **angle of attack** is the angle between the airfoil's chord line and the oncoming airflow. As the angle of attack increases, the amount of lift generated by the airfoil also increases—up to a point. There is a critical angle beyond which the airflow over the top of the wing begins to separate from the surface, leading to a stall. In a stall, the airfoil loses much of its lift and the aircraft can experience a rapid loss of altitude. Airfoil design, particularly the leading-edge shape, helps manage the airflow at higher angles of attack to delay stalling and maintain smooth airflow. Laminar flow airfoils are specifically designed to maintain smooth

- Peterson, Alex. Aerospace Engineering Step by Step: Fundamentals of Aircraft Design, Structures & Systems: From Theory to Practice (Step By Step Subject Guides)

Symmetrical Airfoils

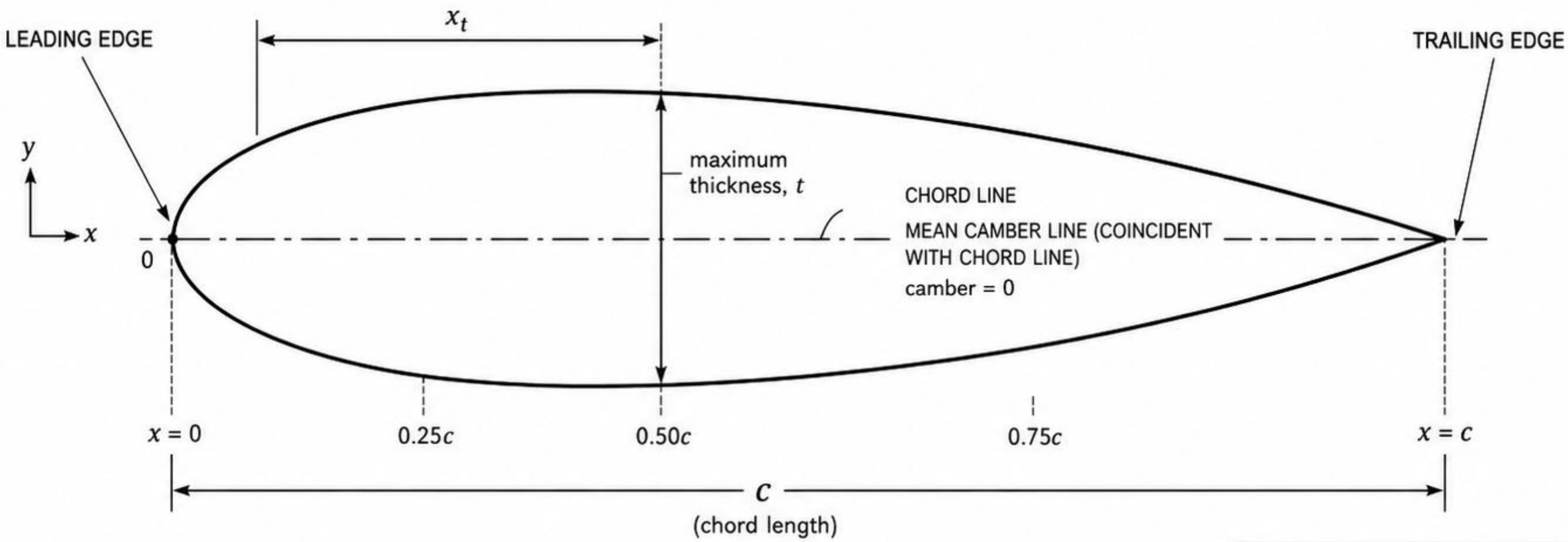
Airfoils with identical upper and lower surfaces — no camber. These have the following characteristics:

- Zero lift at zero angle of attack.
- Lift increases symmetrically with angle of attack.
- No pitching moment (ideal for control surfaces).

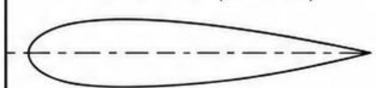
- Used in:
- Aircraft control surfaces (rudders, elevators)
- Aerobatic aircraft (can fly equally well inverted)
- Helicopter rotor blades

SYMMETRICAL AIRFOIL

Geometry, defining features, and typical characteristics



PROFILE NOTE (EXAMPLE)



Example: NACA 0012

DEFINING CHARACTERISTICS

- Zero camber (camber = 0)
- Mean camber line coincides with chord line
- Symmetric pressure/geometry at zero angle of attack ($\alpha = 0$)
- Lift coefficient at $\alpha = 0$ is approximately 0
- Quarter-chord pitching moment is approximately 0
- Similar upright and inverted performance

COMMON USES

- Aerobatic aircraft
- Tailplanes (horizontal stabilizers)
- Some rotor/propeller sections
- Control surfaces (elevators, ailerons, rudders)

KEY PARAMETERS

c = chord length

t = maximum thickness

t/c = thickness ratio

x_t/c = location of maximum thickness

Mean camber = 0

RATIOS (DIMENSIONLESS)

Thickness ratio $= \dfrac{t}{c}$

Location ratio $= \dfrac{x_t}{c}$

NOTATION: c = chord length, t = maximum thickness, x_t = distance from leading edge to location of maximum thickness.

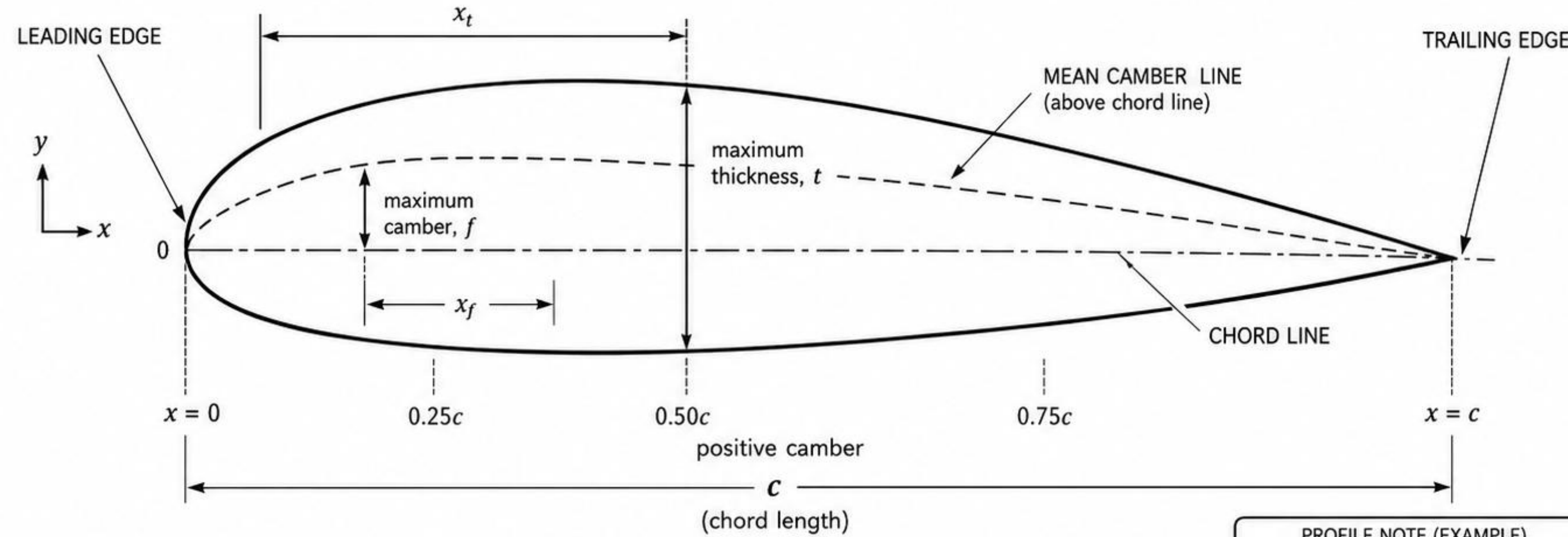
$$\boxed{\text{Thickness ratio} = \frac{t}{c}}$$

Cambered Airfoils

- Airfoils with curved upper surface and flatter lower surface — positive camber. These airfoils have the following characteristics:
- Generate lift at zero angle of attack.
- More efficient at subsonic speeds.
- Often used for general aviation and commercial aircraft.
- Used in:
- General purpose aircraft
- Commercial jets
- Training aircraft

CAMBERED AIRFOIL

Geometry, defining features, and typical characteristics



PROFILE NOTE (EXAMPLE)

Example: NACA 2412

DEFINING CHARACTERISTICS

- Nonzero positive camber
- Mean camber line above chord line
- Produces lift at $\alpha = 0$
- Generally higher lift coefficient at low angle of attack than a symmetric airfoil
- Typically exhibits a nose-down quarter-chord pitching moment
- Widely used when lift efficiency is prioritized

COMMON USES

- General aviation wings
- Gliders
- Low-speed aircraft
- UAV wings
- Flaps and high-lift sections

KEY PARAMETERS

c	=	chord length
t	=	maximum thickness
f	=	maximum camber
t/c	=	thickness ratio
f/c	=	camber ratio
x_t/c	=	location of maximum thickness
x_f/c	=	location of maximum camber

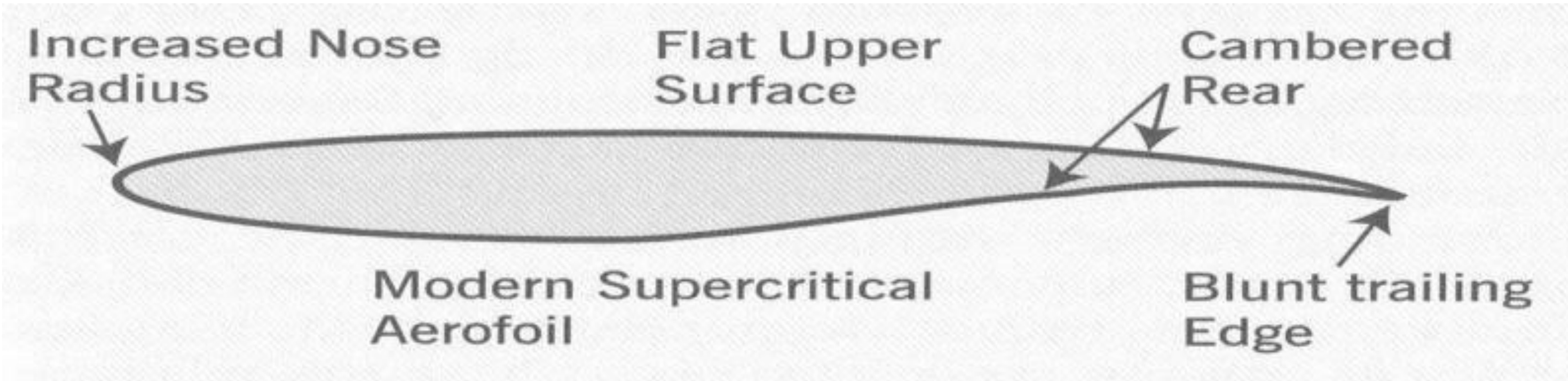
RATIOS (DIMENSIONLESS)

Thickness ratio $= \dfrac{t}{c}$

Camber ratio $= \dfrac{f}{c}$

Thickness location $= \dfrac{x_t}{c}$

Camber location $= \dfrac{x_f}{c}$



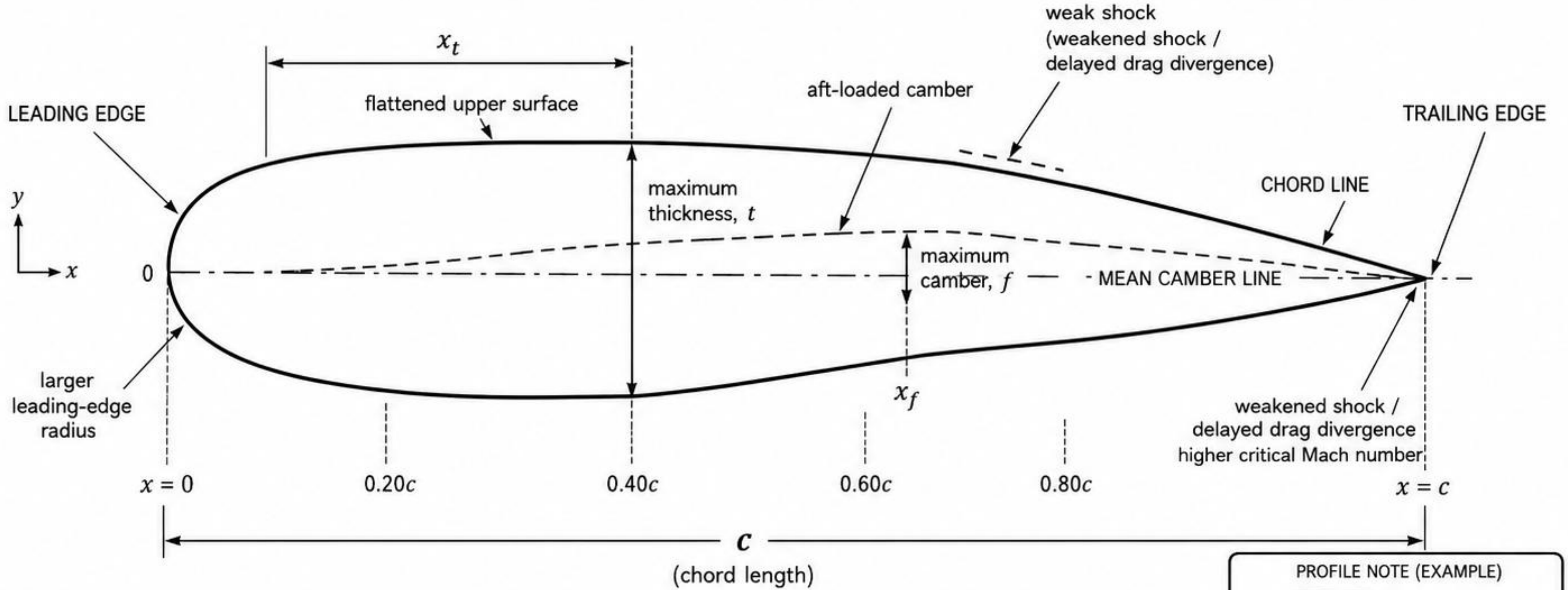
Supercritical Airfoils

Airfoils optimized for high-subsonic speeds (Mach 0.7–0.9). These airfoils are characterized by:
•Flat upper surface with a cusp-like trailing edge.
•Delays shock wave formation and reduces wave drag.
•Reduces drag rise near transonic speeds.

Used in:
•Modern jetliners (Boeing, Airbus)
•Transonic business jets

SUPERCRITICAL AIRFOIL

Geometry, defining features, and transonic characteristics



PROFILE NOTE (EXAMPLE)

Example: NASA SC(2)-0612

DEFINING CHARACTERISTICS

- Optimized for transonic flight
- Flatter upper surface reduces peak suction
- Weakens shock waves and delays drag divergence
- Thicker section can be maintained with favorable transonic performance
- Typically used near cruise Mach numbers of transport aircraft

COMMON USES

- Commercial jet wings
- Transonic transport aircraft
- High-subsonic aircraft
- Advanced wing sections

KEY PARAMETERS

c	=	chord length
t	=	maximum thickness
f	=	maximum camber
t/c	=	thickness ratio
f/c	=	camber ratio
M_{crit}	=	critical Mach number (shock control geometry)

ADDITIONAL RELATIONSHIP

Higher M_{crit} and lower wave drag in transonic flow

NOTATION: c = chord length, t = maximum thickness, f = maximum camber,
x_t = distance from leading edge to location of maximum thickness, x_f = distance from leading edge to location of maximum camber.

Higher M_{crit} and lower wave drag in transonic flow

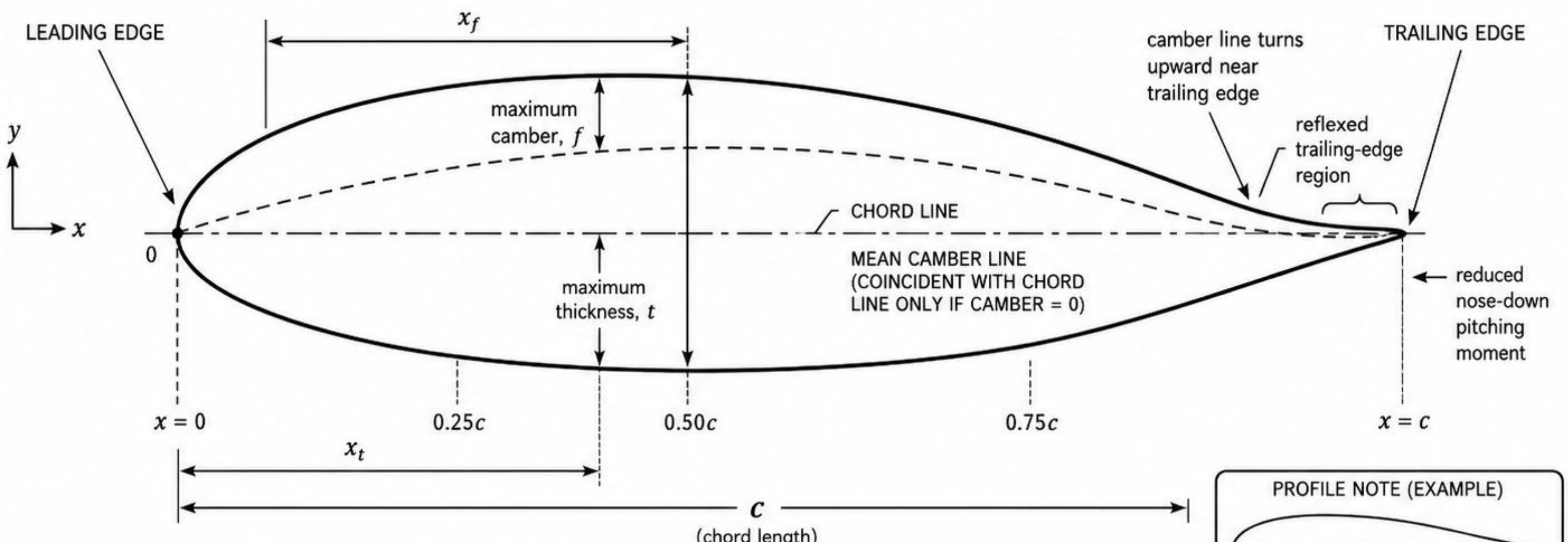
Reflexed Airfoils

- Airfoils with a curve that bends upward near the trailing edge. These are characterized by:
- Produces positive pitching moment, useful for tailless configurations.
- Allows longitudinal stability without a tail.

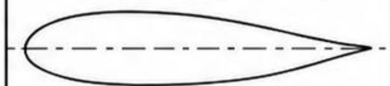
- Used in:
- Flying wings (e.g., B-2 Spirit)
- Drones with no tail
- Control-line model aircraft

REFLEXED AIRFOIL

Geometry, defining features, and typical characteristics



PROFILE NOTE (EXAMPLE)



Example: reflexed section for tailless aircraft

DEFINING CHARACTERISTICS

- Positive camber over the forward/mid chord
- Camber line bends upward near trailing edge (reflexed)
- Reflex reduces the magnitude of negative pitching moment and can make C_m closer to zero
- Suitable where tail-down balancing moment must be minimized
- Often chosen for tailless / flying-wing configurations

COMMON USES

- Flying wings
- Tailless aircraft
- Some UAVs
- Stabilizerless configurations
- Model aircraft

KEY PARAMETERS

c	=	chord length
t	=	maximum thickness
f	=	maximum camber
t/c	=	thickness ratio
f/c	=	camber ratio
x_t/c	=	location of maximum thickness
x_f/c	=	location of maximum camber

Reflex controls moment coefficient.

RATIOS (DIMENSIONLESS)

Thickness ratio $= \dfrac{t}{c}$

Camber ratio $= \dfrac{f}{c}$

Thickness location $= \dfrac{x_t}{c}$

Camber location $= x_f$

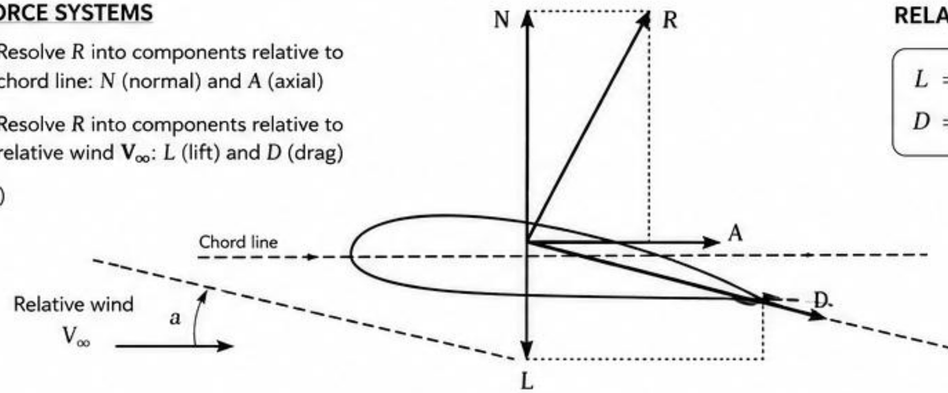
AERODYNAMIC FORCE RESOLUTION ON AIRFOILS

EGEND

= Resultant aerodynamic force
= Normal force ($\perp$ to chord line)
= Axial force ($\parallel$ to chord line)
= Lift ($\perp$ to relative wind, V_∞)
= Drag ($\parallel$ to relative wind, V_∞)
= Angle of attack (between V_∞ and chord)

FORCE SYSTEMS

- Resolve R into components relative to chord line: N (normal) and A (axial)
- Resolve R into components relative to relative wind $\mathbf{V}_\infty$: L (lift) and D (drag)



RELATIONSHIPS: $(\theta = a)$

$$L = N\cos a - A\sin a$$
$$D = N\sin a + A\cos a$$

INVERSE RELATIONSHIPS

$$N = L\cos a + D\sin a$$
$$A = -L\sin a + D\cos$$

$$R = \sqrt{N^2 + A^2} = \sqrt{L^2 + D^2}$$

$\tan\phi = \dfrac{N}{A}$ (ϕ = angle of R above chord line)

$\tan\delta = \dfrac{L}{D}$ (δ = angle of R above relative wind)

FLAT-PLATE AIRFOIL

(SYMMETRICAL, ZERO CAMBER, ZERO REFLEX)

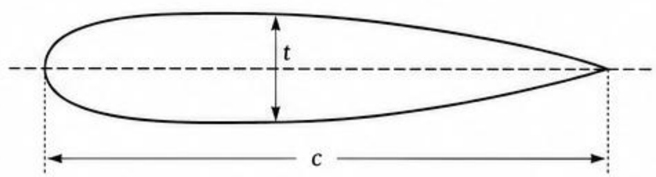
EOMETRY

Upper surface = lower surface

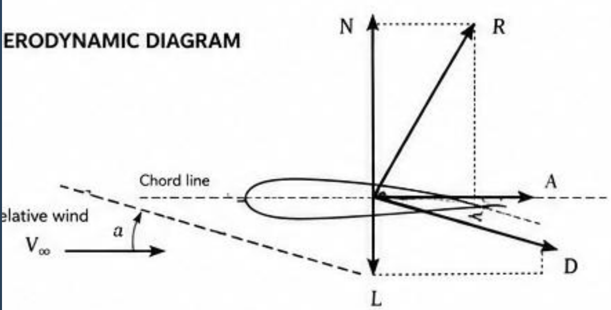
Zero camber: mean camber line coincides with chord line

Thickness (t) constant or distribution symmetric about chord line

Zero geometric angle at trailing edge



ERODYNAMIC DIAGRAM



CHARACTERISTICS

- At $a = 0°$: $L \approx 0,\ N \approx 0,\ A \approx D$ (skin friction drag only)
- Lift curve is linear for small a: $C_L \approx 2\pi a$ (per radian)
- Symmetric lift about $a = 0°$
- Zero pitching moment about quarter-chord (ideal)
- Used as a reference for airfoil data and calibration

AT SMALL ANGLES (a in radians)

$C_L \approx 2\pi a$

$C_D \approx C_{D0} + k\,C_L^2$

(C_{D0} = profile drag at $C_L = 0$)

TYPICAL USES

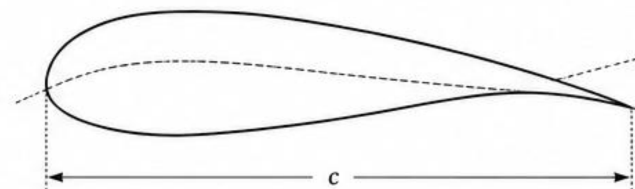
- Control surfaces (elevators, ailerons, rudders)
- Symmetric flight (inverted / upright)
- Baseline for wind tunnel testing

REFLEXED AIRFOIL

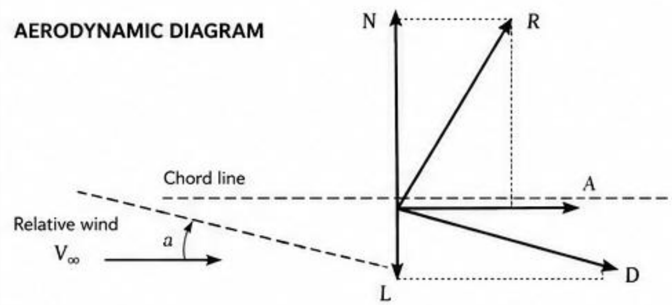
(CAMBERED WITH TRAILING EDGE REFLEX)

GEOMETRY

- Mean camber line curves upward over chord
- Trailing edge deflected downward (reflex angle, $\delta_r < 0°$)
- Provides negative pitching moment and reduced trim drag



AERODYNAMIC DIAGRAM



CHARACTERISTICS

- Produces positive lift at a = ($C_{L0} > 0$)
- Lower pitching moment ab quarter-chord ($C_{m,ac} < 0$)
- Higher lift-to-drag ratio a cruise conditions
- Common in tails and con surfaces for trim efficien

TYPICAL EFFECTS

- Increases C_{L0} (positive lift at zero a)
- Decreases $C_{m,ac}$ (more negative)
- Improves L/D in cruise

COMMON APPLICATIONS

- Horizontal stabilizers
- Elevator and flap surfaces
- Some main wings (e.g., sailplanes, UAVs)

Note: All forces act at the aerodynamic center ($\approx 1/4$ chord for subsonic airfoils). Sign convention: Positive L is upward; positive D is rearward (opposite V_∞); positive N is toward upper surface; positive A is toward trailing edge.

Different Airfoils

Airfoil Type	Lift at AoA=0	Unique Feature	Common Use Cases
Symmetrical	None	Same shape top/bottom	Control surfaces, aerobatics
Cambered	Yes	Curved top, flat bottom	General aviation, commercial jets
Laminar Flow	Yes	Low drag, delayed transition	Gliders, UAVs
Supercritical	Yes	Flat upper, rounded lower	High-subsonic jets
Reflexed	Yes	Upturned trailing edge	Flying wings, tailless aircraft
Flat Plate	Limited	Simplest form, used in models	RC planes, small drones
High-Lift Devices	Enhanced	Uses slats/flaps	STOL aircraft, heavy lifters

Newton's Laws

1st Law: Object at rest, remains at rest, unless outside forces act upon

2nd Law: Force = Mass x Acceleration

3rd Law: Every action has an equal and opposite reaction

Newton Flow Theory

Newton's model of fluid flow refers to his conceptualization of viscous behavior in fluids, leading to the definition of a Newtonian fluid. It is based on the observation that the internal frictional force (shear stress) in a fluid is proportional to the rate of strain (or velocity gradient) between adjacent layers. This concept is essential for understanding how fluids resist deformation when subjected to shear forces.

The mathematical expression of Newton's model is:

$$\tau = \mu \left(\frac{du}{dy} \right)$$

Where:

T: Shear stress (Pa or N/m^2),

μ: Dynamic viscosity of the fluid (Pa·s),

$\frac{\mathrm{d}u}{\mathrm{d}y}$: Velocity gradient perpendicular to the direction of flow (s^{-1}).

Newton Flow Theory

Fluid Type	Behavior	Example
Newtonian Fluid	Shear stress ∝ velocity gradient (linear)	Water, air, ethanol, engine oil
Non-Newtonian Fluid	Non-linear or variable relationship	Blood, ketchup, toothpaste

Basic Formulas

Specific impulse, usually written as **Isp**, is a measure of rocket or jet-propulsion efficiency. It tells you how much thrust a propulsion system produces per unit weight-flow rate of propellant consumed.
In practical terms, a higher specific impulse means the engine gets more useful thrust from a given amount of propellant.
The standard formula is:

$$I_{sp} = \frac{F}{\dot{m} g_0}$$

Where:

$$I_{sp}$$

= specific impulse, in seconds

$$F$$

= thrust, in newtons

$$\dot{m}$$

= propellant mass flow rate, in kilograms per second

$$g_0$$

= standard gravity, approximately:

$$9.80665\ m/s^2$$

An equivalent form uses **effective exhaust velocity**:

$$I_{sp} = \frac{v_e}{g_0}$$

Where:

$$v_e$$

= effective exhaust velocity, in meters per second.
So, if a rocket engine has an effective exhaust velocity of 3,000 m/s:

$$I_{sp} = \frac{3000}{9.80665} \approx 306\ s$$

.

Basic Formulas

Propellant mass fraction is the fraction of a vehicle's total initial mass that consists of usable propellant. It is an important measure in rocketry because most of a rocket's launch mass is normally propellant.
The basic formula is:

$$\text{Propellant } \textit{Mass Fraction} = \frac{m_p}{m_0}$$

Where:
m_p = propellant mass
m_0 = initial total vehicle mass, including structure, engines, payload, and propellant.
It can also be written as:

$$\zeta = \frac{m_0 - m_f}{m_0}$$

Where:
ζ = propellant mass fraction
m_0 = initial mass before propellant burn
m_f = final mass after propellant burn
Since:

$$m_p = m_0 - m_f$$

the formula becomes:

$$\zeta = \frac{m_p}{m_0}$$

Example:
If a rocket has an initial mass of 10,000 kg and carries 8,000 kg of propellant:

$$\zeta = \frac{8000}{10000} = 0.8$$

So, the propellant mass fraction is:

$$0.8 = 80\%$$

- In general, rockets require high propellant mass fractions because reaching high velocity demands a large amount of propellant relative to payload and structure.

Rocket propulsion and mission sizing

Quantity	Formula	Notes
Specific impulse	$I_{sp} = \frac{T}{\dot{m} g_0}$	Thrust efficiency measure
Effective exhaust velocity	$c = I_{sp} g_0$	Sometimes written v_e
Thrust	$T = \dot{m} v_e + (p_e - p_a) A_e$	Momentum thrust plus pressure thrust
Ideal rocket equation	$\Delta V = I_{sp} g_0 \ln\left(\frac{m_0}{m_f}\right)$	Tsiolkovsky equation
Mass ratio	$MR = \frac{m_0}{m_f}$	Initial mass over final mass
Propellant mass fraction	$\zeta = \frac{m_p}{m_0}$	Propellant mass over initial mass
Structural mass fraction	$\epsilon = \frac{m_s}{m_s + m_p}$	Dry structure fraction of stage
Payload fraction	$\lambda = \frac{m_{payload}}{m_0}$	Payload divided by initial mass
Burn time	$t_b = \frac{m_p}{\dot{m}}$	Total propellant over mass flow rate
Thrust-to-weight ratio	$T/W = \frac{T}{m g_0}$	Important for launch vehicles
Acceleration from thrust	$a = \frac{T - D}{m} - g$	Simplified vertical ascent form

Aerodynamics

Quantity	Formula	Notes
Dynamic pressure	$q = \frac{1}{2}\rho V^2$	Central aerodynamic loading term
Lift	$L = qSC_L$	S is reference area
Drag	$D = qSC_D$	Drag force
Moment	$M = qScC_M$	Usually pitching moment
Lift-to-drag ratio	$L/D = \frac{C_L}{C_D}$	Aerodynamic efficiency
Drag polar	$C_D = C_{D0} + kC_L^2$	Parabolic drag model
Induced drag factor	$k = \frac{1}{\pi e AR}$	e is Oswald efficiency
Aspect ratio	$AR = \frac{b^2}{S}$	Wing span efficiency metric
Reynolds number	$Re = \frac{\rho V L}{\mu}$	Viscous similarity parameter
Mach number	$M = \frac{V}{a}$	Speed relative to speed of sound
Speed of sound	$a = \sqrt{\gamma RT}$	Ideal gas approximation
Wing loading	W/S	Weight per wing area
Power loading	W/P	Weight per engine power

Aircraft performance

Quantity	Formula	Notes
Stall speed	$V_s = \sqrt{\frac{2W}{\rho S C_{L,max}}}$	Minimum steady flight speed
Level-flight thrust required	$T_R = D$	For steady, level flight
Level-flight lift condition	$L = W$	Basic cruise equilibrium
Rate of climb	$ROC = \frac{P_A - P_R}{W}$	Excess power over weight
Climb gradient	$\sin\gamma \approx \frac{T - D}{W}$	Jet aircraft approximation
Load factor	$n = \frac{L}{W}$	Maneuvering load
Turn radius	$R = \frac{V^2}{g\sqrt{n^2 - 1}}$	Coordinated level turn
Turn rate	$\omega = \frac{g\sqrt{n^2 - 1}}{V}$	Radians per second
Jet range, Breguet	$R = \frac{V}{c_T}\frac{L}{D}\ln\left(\frac{W_i}{W_f}\right)$	Jet aircraft range
Propeller range, Breguet	$R = \frac{\eta_p}{c_P}\frac{L}{D}\ln\left(\frac{W_i}{W_f}\right)$	Propeller aircraft range
Jet endurance	$E = \frac{1}{c_T}\frac{L}{D}\ln\left(\frac{W_i}{W_f}\right)$	Time aloft for jets

Orbital mechanics

Quantity	Formula	Notes
Circular orbital velocity	$v_c = \sqrt{\frac{\mu}{r}}$	Speed for circular orbit
Escape velocity	$v_e = \sqrt{\frac{2\mu}{r}}$	Minimum escape speed
Orbital period	$T = 2\pi\sqrt{\frac{a^3}{\mu}}$	Kepler's third law
Specific orbital energy	$\epsilon = \frac{v^2}{2} - \frac{\mu}{r}$	Energy per unit mass
Vis-viva equation	$v = \sqrt{\mu\left(\frac{2}{r} - \frac{1}{a}\right)}$	General orbital speed
Hohmann transfer first burn	$\Delta V_1 = \sqrt{\frac{\mu}{r_1}}\left(\sqrt{\frac{2r_2}{r_1 + r_2}} - 1\right)$	Transfer from lower to higher circular orbit
Hohmann transfer second burn	$\Delta V_2 = \sqrt{\frac{\mu}{r_2}}\left(1 - \sqrt{\frac{2r_1}{r_1 + r_2}}\right)$	Circularization burn
Gravitational parameter	$\mu = GM$	Standard gravity parameter

Stability and control

Quantity	Formula	Notes
Static margin	$SM = \frac{x_{NP} - x_{CG}}{\bar{c}}$	Stability measure
Pitching moment coefficient	$C_M = \frac{M}{qS\bar{c}}$	Non-dimensional moment
Yawing moment coefficient	$C_N = \frac{N}{qSb}$	Non-dimensional yaw moment
Rolling moment coefficient	$C_l = \frac{L_{roll}}{qSb}$	Non-dimensional roll moment
Neutral stability condition	$\frac{dC_M}{d\alpha} = 0$	Neutral pitch stability
Longitudinal static stability	$\frac{dC_M}{d\alpha} < 0$	Stable aircraft condition

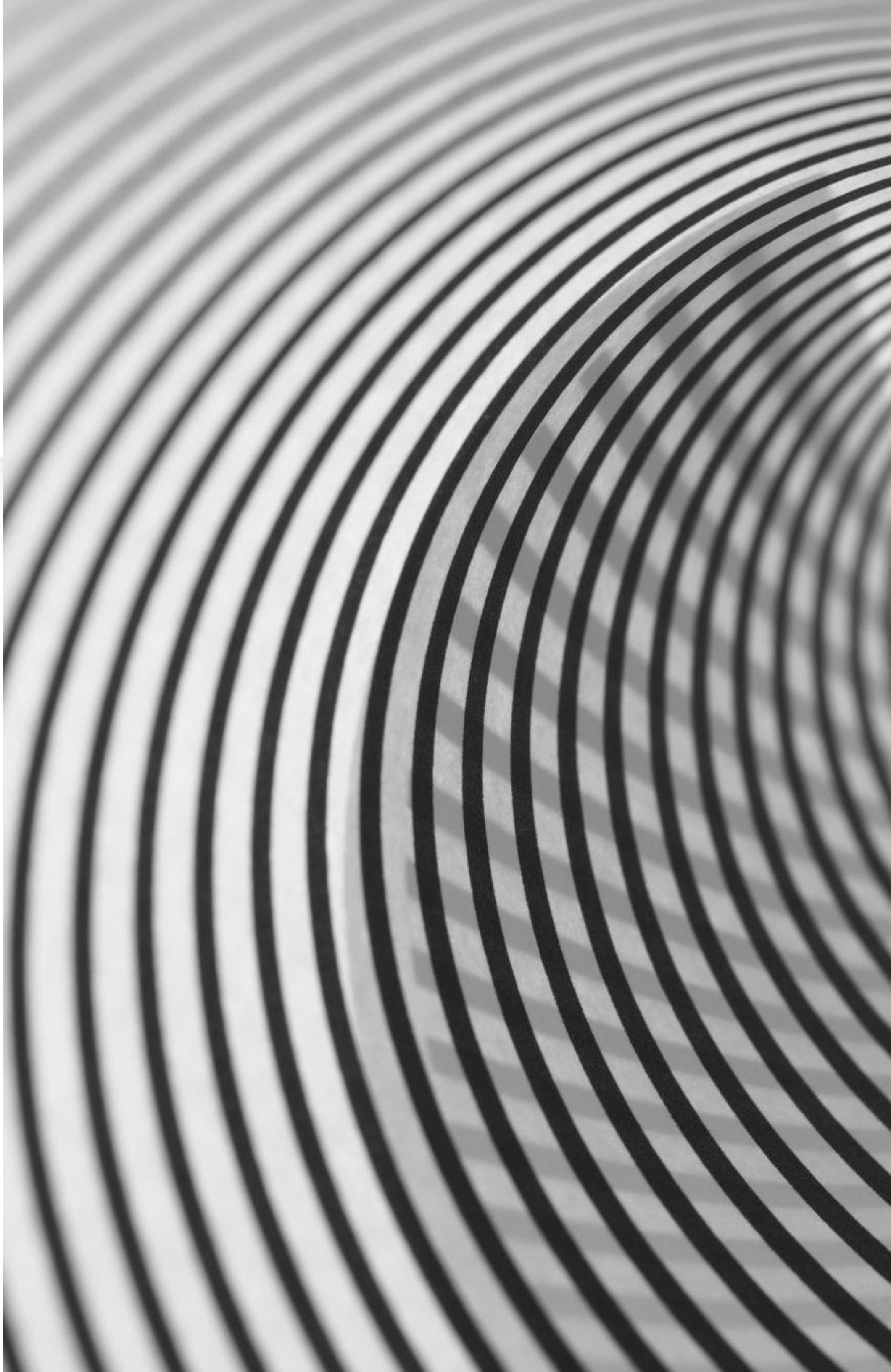
Structures and loads

Quantity	Formula	Notes
Normal stress	$\sigma = \frac{F}{A}$	Axial loading
Shear stress	$\tau = \frac{V}{A}$	Simplified shear
Bending stress	$\sigma = \frac{My}{I}$	Beam bending
Torsional shear stress	$\tau = \frac{Tr}{J}$	Circular shaft approximation
Factor of safety	$FS = \frac{\text{failure load}}{\text{working load}}$	Structural margin
Load factor	$n = \frac{\text{lift}}{\text{weight}}$	Also used in structures
Buckling load	$P_{cr} = \frac{\pi^2 EI}{(KL)^2}$	Euler column buckling

Basic Thermodynamics

- Three thermodynamic properties: pressure, density, and temperature
- Units for temperature must use an "absolute" scale (a scale that starts at "absolute zero" → Kelvin (SI) or Rankine (English)
- Pressure and density have units that combine other consistent units
- Pressure is force per unit area
- $\boldsymbol{p} = \boldsymbol{F/A} == \boldsymbol{N/m2}$
- $\boldsymbol{N/m2}$ is also called a Pascal
- Density is mass per unit volume
- $\boldsymbol{\rho} = \boldsymbol{m/V}$
- $\boldsymbol{kg/m3}$
- Enthalpy is heat of formation and has units of work per unit mass
- $\boldsymbol{h} = \boldsymbol{e} + \boldsymbol{p/\rho}$; $\boldsymbol{e}$ is internal energy of gas (high enthalpy means HOT!)

Speed of Sound



- The speed of sound (acoustic speed) in air depends on:
- Temperature
- Gas properties

For a Perfect Gas the speed of sound is $a = \sqrt{\gamma R T}$

$\boldsymbol{\gamma}$ is the ratio of specific heats (the heat capacity); $\boldsymbol{\gamma} = \mathbf{1.4}$ for air at "normal" temperatures

Remember to use absolute temperatures

Speed of Sound Example

What is the speed of sound at an altitude of 20km on a standard day?

First find the temperature at that altitude using the Standard Atmosphere table We must use an absolute temperature, so convert:

- -56.50℃ +273
- $=216.5K$

Properties of the U.S. Standard Atmosphere (SI Units)[a]

Altitude (m)	Temperature (°C)	Acceleration of Gravity, g (m/s^2)	Pressure, p [N/m^2(abs)]	Density, ρ (kg/m^3)	Dynamic Viscosity, μ (N·s/m^2)
−1,000	21.50	9.810	1.139 E + 5	1.347 E + 0	1.821 E − 5
0	15.00	9.807	1.013 E + 5	1.225 E + 0	1.789 E − 5
1,000	8.50	9.804	8.988 E + 4	1.112 E + 0	1.758 E − 5
2,000	2.00	9.801	7.950 E + 4	1.007 E + 0	1.726 E − 5
3,000	−4.49	9.797	7.012 E + 4	9.093 E − 1	1.694 E − 5
4,000	−10.98	9.794	6.166 E + 4	8.194 E − 1	1.661 E − 5
5,000	−17.47	9.791	5.405 E + 4	7.364 E − 1	1.628 E − 5
6,000	−23.96	9.788	4.722 E + 4	6.601 E − 1	1.595 E − 5
7,000	−30.45	9.785	4.111 E + 4	5.900 E − 1	1.561 E − 5
8,000	−36.94	9.782	3.565 E + 4	5.258 E − 1	1.527 E − 5
9,000	−43.42	9.779	3.080 E + 4	4.671 E − 1	1.493 E − 5
10,000	−49.90	9.776	2.650 E + 4	4.135 E − 1	1.458 E − 5
15,000	−56.50	9.761	1.211 E + 4	1.948 E − 1	1.422 E − 5
20,000	−56.50	9.745	5.529 E + 3	8.891 E − 2	1.422 E − 5
25,000	−51.60	9.730	2.549 E + 3	4.008 E − 2	1.448 E − 5

Ideal Gas Law

The Ideal Gas Law (Sometimes called the Perfect Gas Law), is a fundamental equation in thermodynamics that relates the pressure, volume, temperature, and amount of gas in a system. It assumes that the gas behaves ideally, meaning the gas particles have negligible volume and no intermolecular forces.

PV=nRT

SYMBOL	MEANING	UNITS (SI)
P	Pressure	Pascals (Pa)
V	Volume	Cubic meters (m^3)
n	Amount of substance	Moles (mol)
R	Ideal gas constant	8.314 J/mol·K
T	Temperature (absolute)	Kelvin (K)

Charles' Law (Temperature-Volume Law)

- Gas volume varies directly with temperature at a constant pressure
$V_1/T_1=V_2/T_2$
Put another way "At constant pressure, the volume of a fixed amount of gas is directly proportional to its absolute temperature (measured in Kelvin)"

Pressure law- Gay Lussac law

"For a fixed mass of gas at a constant volume, the pressure is directly proportional to the temperature"

P is proportional to T or P / T = K

Avogadro's hypothesis

Equal volumes of gases, under the same conditions of temperature and pressure, contain equal numbers of molecules.

One mole of a gas

Oxygen (O2) = 32

32 grams of oxygen will have 6.022 * 10^{23} atoms

6 DOF

6-DOF stands for Six Degrees of Freedom — the six independent ways an object can move in 3D space.

These include:

- Three translational degrees: Motion along the X, Y, Z axes.
- Three rotational degrees: Rotation about the X, Y, Z axes (also called roll, pitch, yaw

6-DOF forces and moments provide a complete and realistic model of motion for aerospace systems operating in complex environments. Understanding and simulating 6-DOF behavior is essential for control design, flight stability analysis, and autonomous navigation systems.

6 DOF

Degree of Freedom	Type	Axis	Description
Surge	Translation	X-axis	Forward/backward movement (longitudinal)
Sway	Translation	Y-axis	Side-to-side movement (lateral)
Heave	Translation	Z-axis	Up/down movement (vertical)
Roll	Rotation	X-axis	Rotation around the longitudinal axis
Pitch	Rotation	Y-axis	Rotation around the lateral axis
Yaw	Rotation	Z-axis	Rotation around the vertical axis

6 DOF Equations

Translational Motion

$$m \cdot \frac{d\vec{v}}{dt} = \vec{F}$$

Where:

- m: Mass of the object,
- $\vec{v}$: Velocity vector,
- $\vec{F}$: Net force vector in body or inertial frame.

6 DOF Equations

- Rotational Motion

$$\mathbf{I} \cdot \frac{d\vec{\omega}}{dt} + \vec{\omega} \times (\mathbf{I} \cdot \vec{\omega}) = \vec{M}$$

- Where:
- I: Moment of inertia tensor,
- $\vec{\omega}$: Angular velocity vector,
- $\vec{M}$: External moment (torque) vector

6 DOF

6-DOF forces refer to the external forces and torques (moments) acting on a body that cause motion in each of the six degrees of freedom.

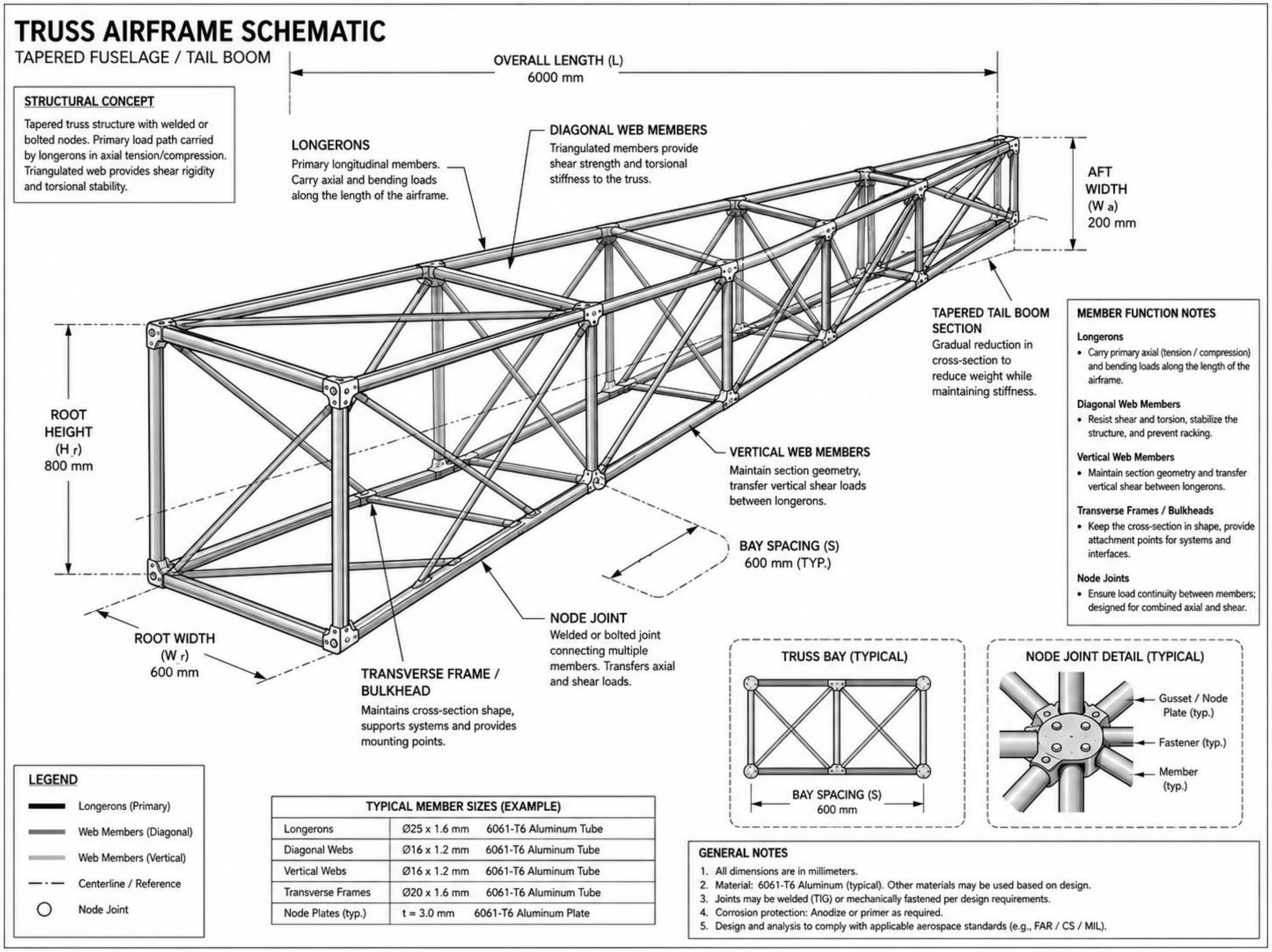
- Forces (F_x, F_y, F_z) cause translation.
- Moments (M_x, M_y, M_z) cause rotation.

These forces are typically caused by:

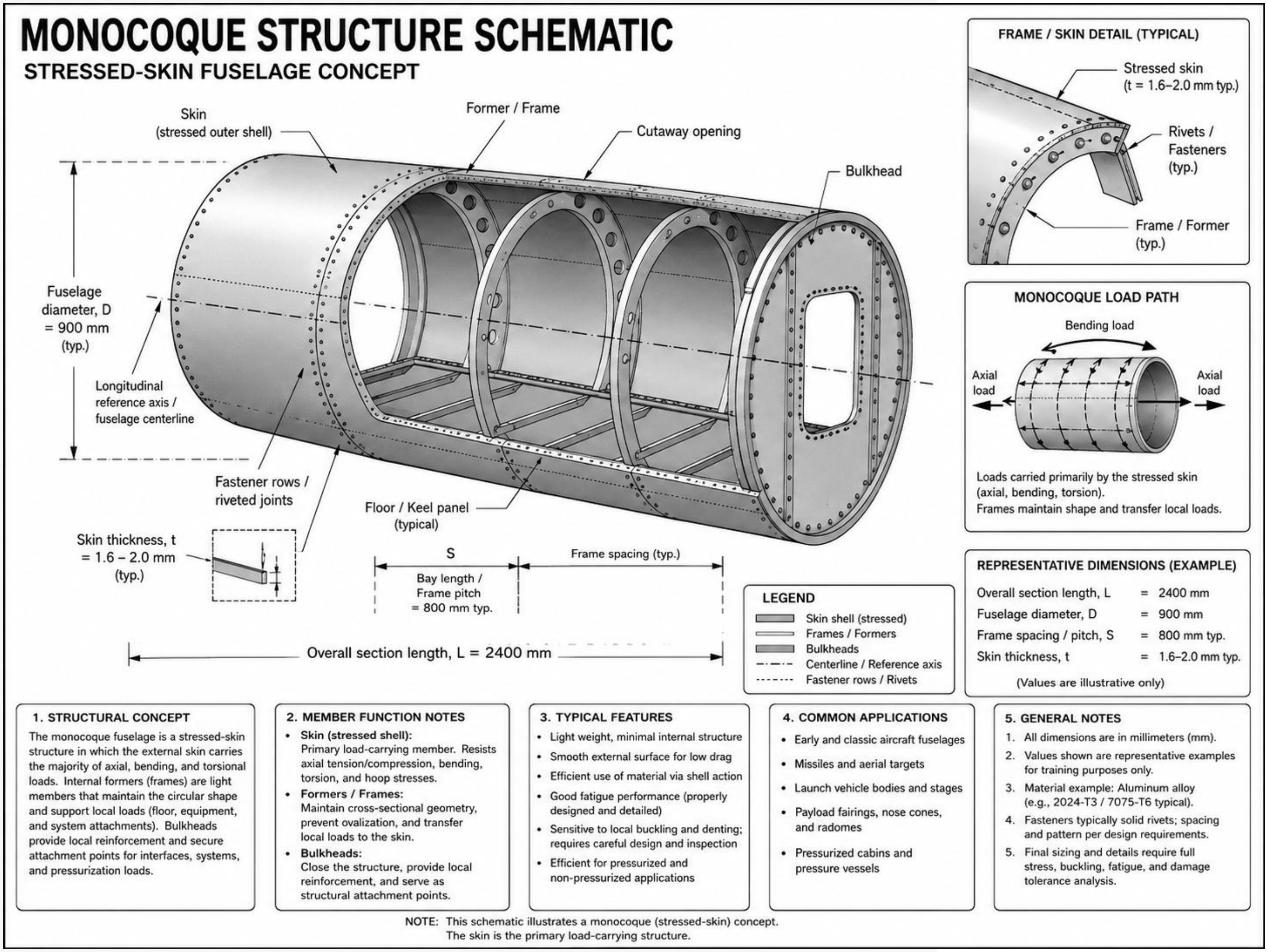
- Aerodynamics (lift, drag, side force),
- Propulsion (thrust vectoring, jet forces),
- Control surfaces (elevator, rudder, ailerons),
- Gravity, buoyancy, and inertial coupling.

Truss Airframe

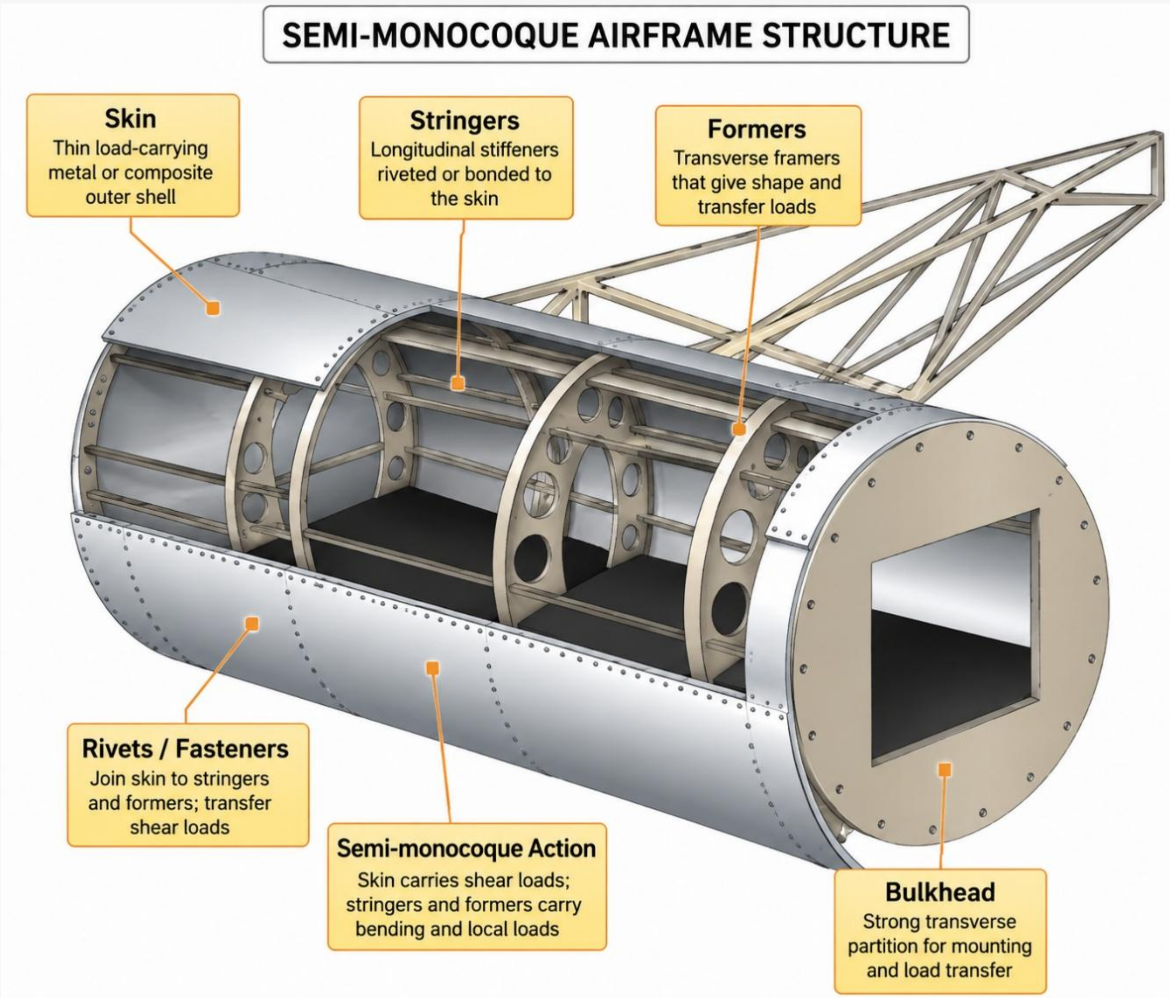
A truss structure airframe is an aircraft framework built from a network of triangular units (trusses) such as steel or aluminum tubes. These triangles distribute loads evenly, making the structure lightweight yet strong. The truss acts like a skeleton, and fabric or thin metal skin is often stretched over it for aerodynamic covering. Eventually replaced by monocoque and semi-monocoque stressed-skin structures as speeds increased and drag reduction became critical.



Monocoque structure



A monocoque structure is a type of construction in which the external skin or shell bears all or most of the structural load, rather than relying on an internal frame. The term comes from the French word "monocoque", meaning "single shell." Used in early aircraft and modern composite fuselage sections. Examples include WWII fighters like the De Havilland Mosquito and modern UAVs. However, even today, rocket and missile bodies often use monocoque or semi-monocoque designs to save weight.



Semi-monocoque

- A semi-monocoque structure is a type of aircraft or vehicle construction that combines elements of both monocoque and truss-type construction. It utilizes a combination of a stressed skin and an internal framework (like stringers and frames) to distribute loads and maintain the structure's shape. This differs from a true monocoque, which relies primarily on the skin for strength, or a truss structure, which relies on the frame

Monocoque vs semi- monocoque

Feature	**Monocoque**	**Semi-Monocoque**
Load carrier	Outer shell only	Shell + internal support (frames/stringers)
Structural redundancy	Minimal	High redundancy; better damage tolerance
Repairability	Difficult	Easier
Used in	Small aircraft, missiles	Most modern aircraft fuselages

Weight and Thrust

Weight is the downward force due to gravity, acting through the object's center of gravity.

$W=mg$

Where:

m = mass

g = gravitational acceleration (~9.81 m/s^2 on Earth)

Thrust is the forward force produced by a propulsion system (engine, rocket, jet).

Drag

Drag is the resistance force that acts opposite to the direction of motion. It is caused by air particles colliding with the object.

$L = ½ \rho V^2 C_D A$

Where:

C_D = drag coefficient

Type	Description
Skin friction drag	Due to air sliding along surfaces
Form (pressure) drag	Due to shape and flow separation
Wave drag (supersonic/hypersonic)	Caused by shock waves
Induced drag	From generating lift (especially at low speeds)

Drag Coefficient

The drag coefficient (C_d) is a dimensionless number that quantifies the drag or resistance of an object as it moves through a fluid, such as air or water. It is an essential parameter in fluid dynamics and aerodynamics, providing a measure of how streamlined or aerodynamic an object is.

$$F_d = \frac{1}{2} C_d \rho v^2 A$$

where:

- F_d = drag force (N),
- C_d = drag coefficient (dimensionless),
- ρ = density of the fluid (kg/m³),
- v = velocity of the object relative to the fluid (m/s),
- A = reference area (m²), typically the frontal area of the object.

A lower Cd value means the object moves more easily through the fluid, experiencing less resistance.

A higher C_d indicates a less aerodynamic shape, which generates more drag.

Factors Affecting the Drag Coefficient

Shape of the Object
- Streamlined shapes (e.g., airplane wings) have much lower drag coefficients than blunt shapes (e.g., a flat plate).

Surface Roughness
- A smoother surface reduces frictional drag, while rough surfaces increase it.

Reynolds Number
- The flow regime (laminar or turbulent) influences CdCd. At different Reynolds numbers, the drag coefficient can vary significantly.

Angle of Attack
- For objects like wings or airfoils, the angle between the object and the fluid flow can drastically change C_d

Basic Concepts –Types of Drag

Type	Description
Parasite Drag	Due to friction and pressure differences; includes form and skin-friction drag
Induced Drag	Caused by lift generation (vortex formation at wingtips)
Wave Drag	From shock waves in transonic and supersonic flow
Base Drag	Caused by low-pressure wake at the rear of a blunt body
Viscous Drag	Internal friction within the boundary layer

Drag

At supersonic and hypersonic speeds, aerodynamic efficiency becomes even more complex. The primary concern shifts to managing wave drag, which occurs when shock waves form as the aircraft travels faster than the speed of sound. In these regimes, traditional airfoil shapes are inefficient, and engineers must design with much sharper and thinner profiles to minimize drag. Delta wing designs are often used in supersonic aircraft to reduce drag and maintain stability at high speeds.

Peterson, Alex. Aerospace Engineering Step by Step: Fundamentals of Aircraft Design, Structures & Systems: From Theory to Practice (Step By Step Subject Guides) (p. 33). Kindle Edition.

Lift

Lift is the upward force that opposes weight (gravity). It is generated by air moving over a surface, usually wings or fins. Caused by pressure differences between the upper and lower surfaces of an airfoil (wing or fin). Faster airflow over the curved top creates lower pressure, while slower flow underneath creates higher pressure.

$L = \frac{1}{2} \rho V^2 C_L A$

Where:

- ρ = air density
- V = velocity
- C_L = lift coefficient (depends on airfoil shape and angle of attack)
- A = reference area

Lift

Lift is the force that opposes the weight of an aircraft and allows it to rise into the air. It is generated by the movement of air over the aircraft's wings. To understand lift, you have to look at Bernoulli's principle, which states that as the speed of a fluid increases, its pressure decreases. When air flows over a wing, the shape of the wing (an airfoil) causes the air on top to move faster than the air below. This difference in airspeed creates a pressure difference, where lower pressure exists on top of the wing and higher pressure below. This pressure imbalance pushes the wing upward, creating lift. However, the amount of lift produced depends on several factors, including the angle of attack, airspeed, and the wing's surface area.

- Peterson, Alex. Aerospace Engineering Step by Step: Fundamentals of Aircraft Design, Structures & Systems: From Theory to Practice (Step By Step Subject Guides).

Generation of Lift

Lift is a complex subject relating to fluid dynamics and thermodynamics

There are still varying theories and misconceptions

It is often said that the lift on a wing is generated because the flow moving over the top surface has a longer distance to travel and therefore needs to go faster. This common explanation is actually wrong.

There is a cause and effect between velocity and pressure on the aerofoil

- Explanations include:
  - **Newton's Third Law of action and reaction** says that Lift occurs when a moving flow of gas is turned by a solid object
    - The flow is turned in one direction, and
    - the lift is generated in the opposite direction
  - **Bernoulli's principle** states that
    - An increase in speed must accompany any reduction in pressure; and
    - a decrease in speed must accompany any pressure increase
- Each of these explanations are partly right but incomplete and complementary

Lift Coefficient

The lift coefficient is influenced by several factors, including the airfoil shape, the angle of attack, and the Reynolds number, which relates to the airflow's viscosity. A higher C_L indicates that the airfoil is more efficient at generating lift for a given set of conditions. The lift coefficient increases with angle of attack, up to a critical point. As the angle of attack rises, the airflow over the wing generates more lift by increasing the pressure differential between the upper and lower surfaces.

-Peterson, Alex. Aerospace Engineering Step by Step: Fundamentals of Aircraft Design, Structures & Systems: From Theory to Practice (Step By Step Subject Guides)

Relationship Between Lift and Drag

- Both forces are generated by pressure differences and shear stresses on the surface of a body.
- The Lift-to-Drag Ratio (L/D) is a critical performance metric:
- $\frac{L}{D} = \frac{C_L}{C_D}$

- Higher L/D→better aerodynamic efficiency (important in gliders, cruise missiles, long-range aircraft).
- In hypersonics, minimizing drag is often more important than maximizing lift (for heat control and structural reasons).

Lift – Drag Balance

Achieving the right balance between lift and drag is central to designing aircraft with high aerodynamic efficiency. Aerodynamic efficiency, represented by the lift-to-drag ratio (L/D), determines how well an aircraft can convert lift into forward motion without generating excessive drag. A high L/D ratio is desirable because it means the aircraft generates enough lift to sustain flight while minimizing drag, which directly impacts fuel efficiency, speed, and overall performance.

Peterson, Alex. Aerospace Engineering Step by Step: Fundamentals of Aircraft Design, Structures & Systems: From Theory to Practice (Step By Step Subject Guides)

Basic Concepts

Lift is the aerodynamic force that acts perpendicular to the direction of the relative airflow (also called freestream velocity).

- $L = ½\ pV^2SC_L$

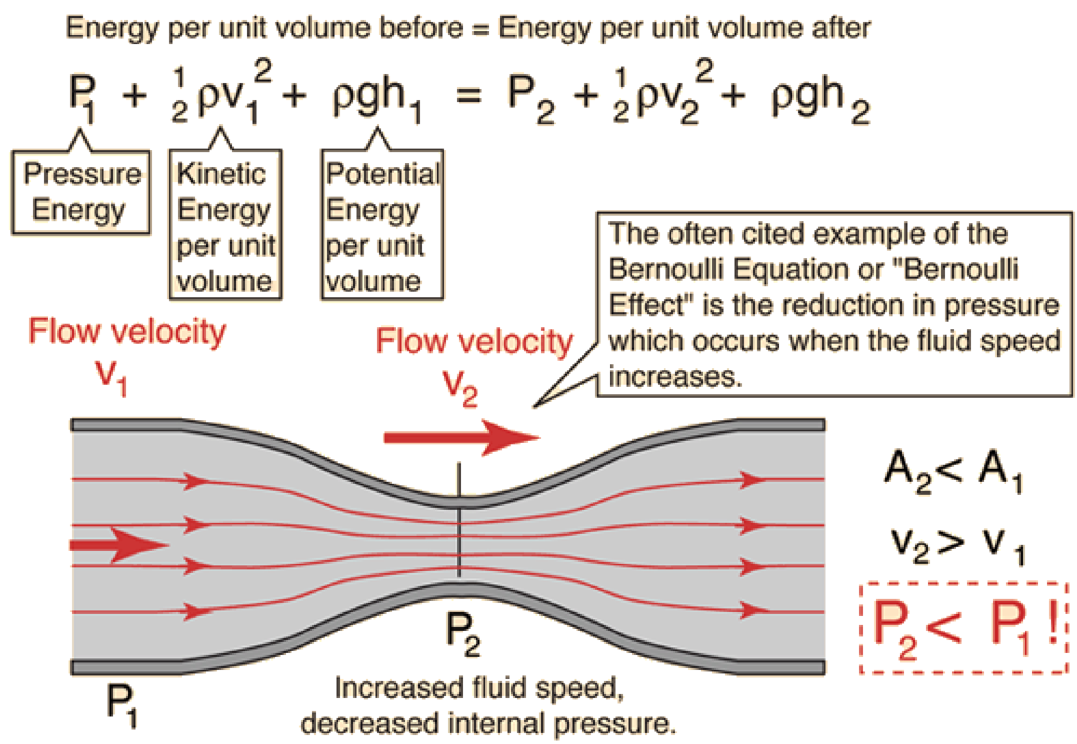
- Where:
- L: Lift force (N)
- ρ: Air density (kg/m³)
- V: Velocity of the object relative to air (m/s)
- S: Reference area (usually wing area) (m²)
- C_L: Lift coefficient (dimensionless, depends on shape, angle of attack)

Factors Affecting Lift:

- Angle of attack (α): Higher α generally increases lift (up to stall).
- Airfoil shape: Curved upper surface creates pressure difference (Bernoulli's principle + circulation).
- Velocity: Faster speeds produce more lift.
- Altitude: Lower air density at high altitudes reduces lift.

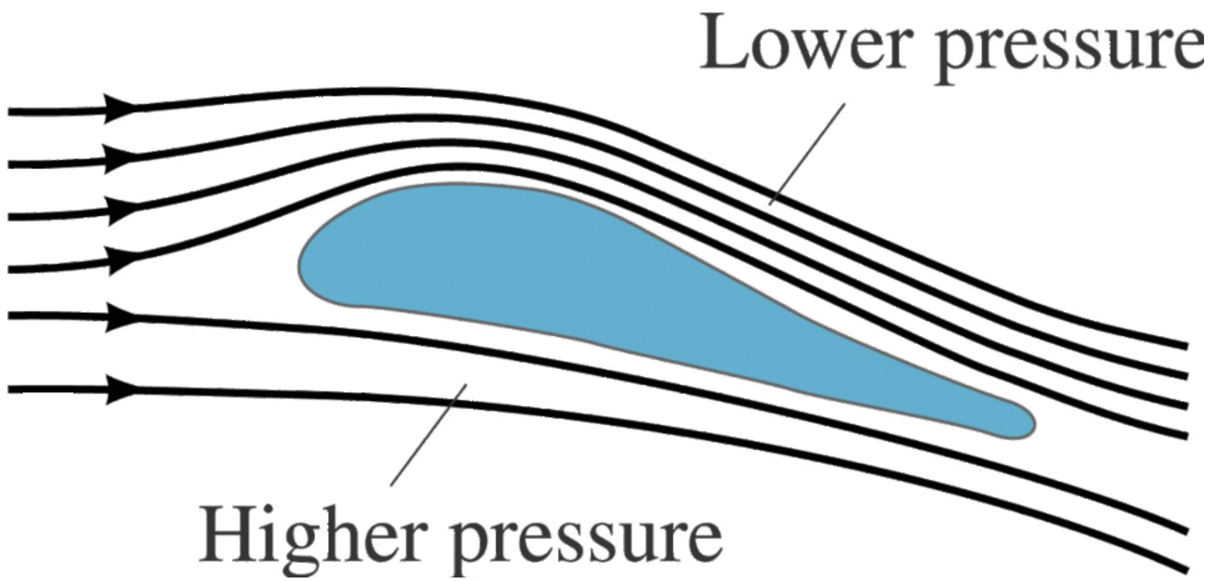
Bernoulli’s Principle

Bernoulli’s principle states that for a steady, incompressible, frictionless flow, an increase in fluid velocity is accompanied by a decrease in static pressure. In aerospace engineering, this helps explain why pressure changes occur around an airfoil: air moving faster over a surface can have lower static pressure than slower-moving air elsewhere. The simplified Bernoulli equation is $p + \frac{1}{2}\rho V^2 = constant$, where p is static pressure, ρ is fluid density, and V is flow velocity. In real aircraft aerodynamics, Bernoulli’s principle is used together with Newton’s laws, circulation, viscosity, and compressibility effects to describe lift accurately.



Bernoulli's Principle and the Airfoil

- The airfoil is a cut off portion of Bernoulli's tube
- An airfoil is any surface that provides aerodynamic force when interacting with a stream of air



Bernoulli’s Principle

In a steady, incompressible flow of an ideal fluid, an increase in the fluid's speed occurs simultaneously with a decrease in its pressure or potential energy. For an incompressible, frictionless fluid, Bernoulli’s equation along a streamline is:
$P+1/2\rho v^2+\rho gh=\text{constant}$

Where:

- P = static pressure (Pa)
- ρ = fluid density (kg/m^3)
- v = flow velocity (m/s)
- g = gravitational acceleration (9.81 m/s^2)

Bernouli's Principle and Lift

The most common use of Bernoulli's Principle is in explaining lift on an aircraft wing (airfoil):

- The upper surface of the wing is curved.
- As air flows over the wing, it travels faster over the curved top than the flatter bottom.
- By Bernoulli's Principle, faster air = lower pressure.
- This creates a pressure difference, with higher pressure beneath the wing pushing upward — this is lift.

Note: While Bernoulli's Principle plays a role, Newton's Third Law (downwash creating reaction lift) is also essential for a full explanation of lift.

Bernoulli's Equation (Ideal Approximation)

Bernoulli's equation can provide insights into pressure changes:

$$P + \frac{1}{2}\rho u^2 + \rho g h = \text{constant}$$

P: Static pressure (Pa),

ρ: Density of the fluid (kg/m^3), note this is the rho Symbol not p

v: Flow velocity (m/s),

g: Acceleration due to gravity (9.81 m/s^2),

h: Elevation above a reference level (m).

In a streamline along the jet:

- Higher velocity near the wall = lower pressure, which draws the jet toward the surface.
- This helps explain the entrainment of ambient fluid and jet adherence qualitatively.

Bernoulli's Equation (Ideal Approximation)

- Static Pressure (P): The pressure exerted by the fluid at a given point when it's not moving (or perpendicular to flow direction). Reflects potential to do work due to pressure forces.

- Dynamic Pressure ($1/2\rho v^2$) :Represents kinetic energy per unit volume. Increases as flow speed increases.

- Hydrostatic Pressure (ρgh) :Represents potential energy due to elevation in a gravitational field. Higher at lower points in a fluid due to weight above

Misunderstandings of the Bernoulli Principle

- Misinterpretations are common:

- Airplane wings: Some textbooks wrongly explain lift by saying “air on top of the wing moves faster because it has farther to travel, so Bernoulli makes pressure lower.” That’s an incorrect justification. The real reason is Newton’s third law + circulation + Bernoulli together, not just "longer path = faster air."

- Spraying atomizers / carburetors: Sometimes people oversimplify, ignoring viscosity, turbulence, or compressibility.

- Suction misunderstanding: Bernoulli doesn’t mean "higher speed always equals lower pressure" in every situation. It only holds when the assumptions are valid.

The Coandă effect

This effect was named after a Romanian aircraft designer Henri Coandă, after an aircraft he designed went up in flames as a consequence of this effect.

Generally pronounced koh-AHN-duh

In English aerospace contexts, you will usually hear “koh-AHN-duh effect.”

The Coandă effect

The Coandă effect is the tendency of a fluid jet to attach itself to a nearby surface and flow along it, even when the surface curves away from the initial direction of the jet. Named after Romanian aerodynamicist Henri Coandă, who observed the phenomenon in 1910, this effect plays a crucial role in many fluid dynamics applications.

At its core, the Coandă effect is a result of viscous interaction between a high-speed fluid jet and the surrounding slower or stationary fluid near a surface. Here's how it works:

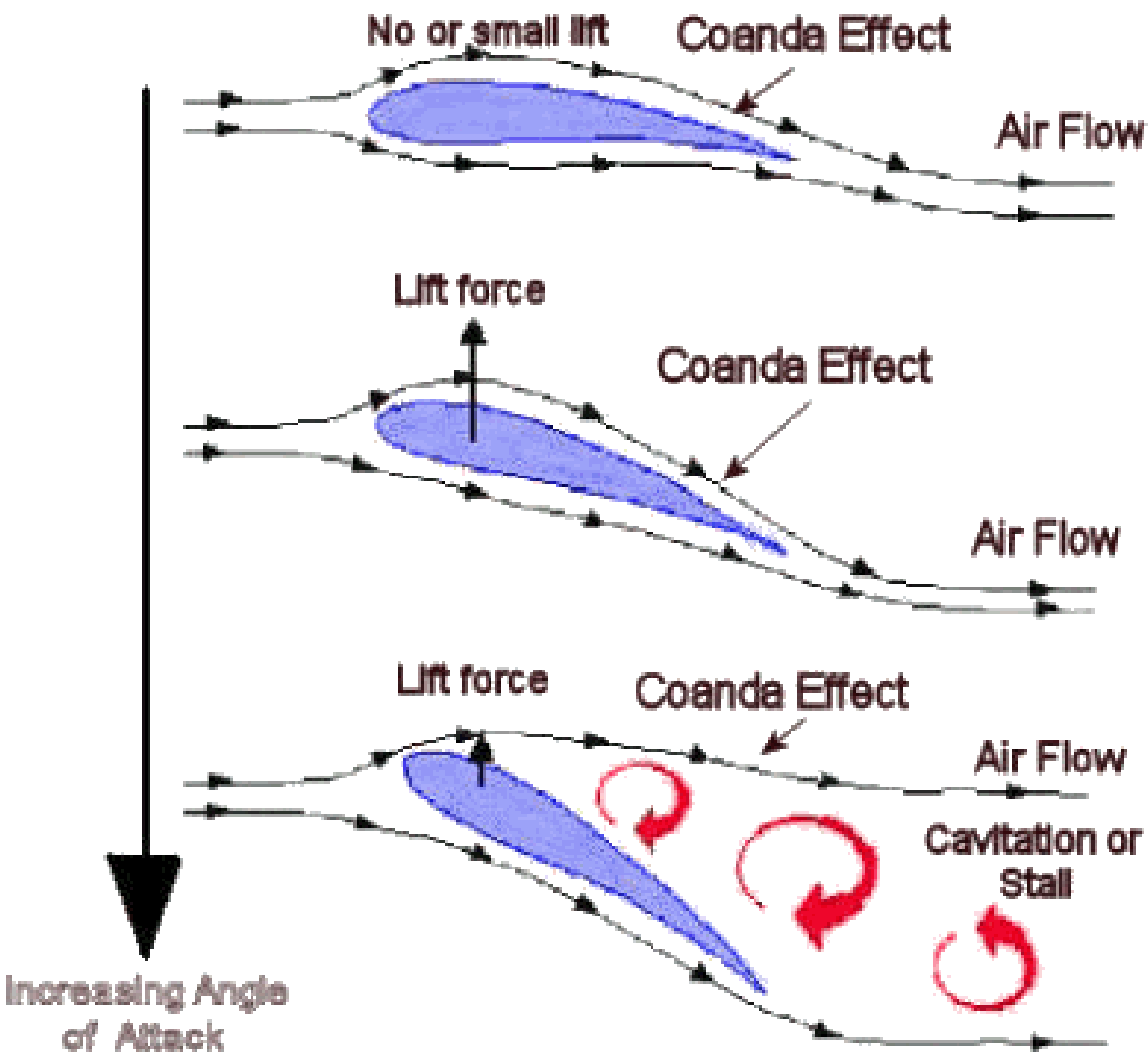
- As a jet of fluid exits a nozzle and passes near a solid surface, some of the surrounding fluid is entrained into the jet due to viscosity.
- This entrainment creates a low-pressure region between the jet and the surface.
- The pressure difference causes the jet to bend toward the surface and stick to it, continuing along its curvature.
- If the surface curves smoothly, the jet continues to follow it due to the continued pressure gradient and entrainment.

The Coandă effect

- **Aircraft control surfaces**: Some experimental and modern aircraft use the Coandă effect to enhance lift or replace traditional ailerons/flaps.
- **Circulation control wings**: Air is blown over curved surfaces to increase lift via the Coandă effect, particularly at low speeds
- The Coandă effect is not Bernoulli's principle, though both involve pressure and velocity changes in fluids.
- It does not explain all lift in aircraft; it contributes in certain designs, but traditional wings use pressure differences and circulation theory.
- It requires controlled conditions; highly turbulent or non-cohesive flows may not exhibit the effect clearly.

- https://www.youtube.com/watch?v=6Q8HssqWDDE

WHY DOES LIFT CURVE BEND OVER?



NO LIFT

FLIGHT

STALL

Source: http://www.formula1-dictionary.net/coanda_effect.html

Stall

- A stall is caused by the separation of airflow from the wing's upper surface
- The result is the loss in production of lift with a exponentially proportional increase in drag
- An airfoil will ALWAYS stall when reaching its Critical AoA
- Stalls will always occur regardless of airspeed

Static stability

Static stability refers to the immediate response of an aircraft after it experiences a small disturbance, such as a gust of wind or a slight change in pitch or yaw. The question we ask is: "What happens right after the disturbance?" In other words, static stability determines whether the aircraft tends to correct itself, keep moving in the disturbed direction, or become even more unstable. There are three types of static stability: positive static stability, neutral static stability, and negative static stability.

- Positive static stability means the aircraft has a natural tendency to return to its original state after being disturbed. Imagine you're flying straight and level, and a gust of wind pitches the nose upward. If the aircraft has positive static stability, it will naturally try to bring the nose back down toward level flight. It's like a pendulum—when pushed away from its resting position, it swings back toward the center.
- Neutral static stability means that when disturbed, the aircraft will remain in its new position without returning to its original state or moving further away. So, if that same gust of wind pitches the nose up, the aircraft will stay at that new pitch angle without trying to correct itself or go further off course. It's stable in the sense that it's not getting worse, but it's not going back to its original state either.
- Negative static stability occurs when the aircraft moves even further away from its original state after a disturbance. This is the opposite of positive static stability. If a gust of wind pitches the nose up, the nose will continue to rise further without any tendency to return to level flight. This can be dangerous if not corrected by the pilot or automatic control systems.

- Peterson, Alex. Aerospace Engineering Step by Step: Fundamentals of Aircraft Design, Structures & Systems: From Theory to Practice (Step By Step Subject Guides)

Dynamic stability

While static stability tells us how an aircraft initially responds to a disturbance, dynamic stability describes what happens over time. Does the aircraft's motion grow more stable or unstable as time passes? Dynamic stability is all about the aircraft's long-term behavior following a disturbance. There are three main types of dynamic stability: positive dynamic stability, neutral dynamic stability, and negative dynamic stability.

- Positive dynamic stability means that not only does the aircraft initially try to correct itself after a disturbance, but over time, it fully returns to its original state. Imagine the aircraft being pushed into a slight pitch up. With positive dynamic stability, the aircraft might oscillate a few times, nose up and nose down, but each oscillation becomes smaller and smaller until the aircraft returns to level flight.
- Neutral dynamic stability means the aircraft oscillates after a disturbance, but the oscillations do not increase or decrease over time. So, the aircraft will keep oscillating back and forth at the same amplitude without returning to its original state or getting worse. In practical terms, this might be uncomfortable for passengers but not necessarily dangerous.
- Negative dynamic stability is when the oscillations grow larger over time. After the initial disturbance, instead of settling back to level flight, the aircraft's nose pitches up higher and higher (or down lower and lower) with each oscillation, eventually leading to a dangerous situation if not corrected. This condition often requires immediate intervention by the pilot or automated systems to regain control.

- Peterson, Alex. Aerospace Engineering Step by Step: Fundamentals of Aircraft Design, Structures & Systems: From Theory to Practice (Step By Step Subject Guides)

Laminar flow

In laminar flow the fluid particles stay in the same "lamina," whereas in turbulent flow a constant intermingling of the particles in neighboring laminas occurs, which can be easily visualized by adding dye particles to the flow. For example, it is found that the laminar flow velocity profile in a tube is parabolic, whereas the turbulent profile is much flatter. On the tube walls the flow velocity must be zero, referred to as the "no-slip" condition. Hence, the velocity gradient adjacent to the surface is much steeper in turbulent flow than in laminar flow. This condition applies to any surface and, consequently, the friction loss on a wing surface is much greater in a turbulent flow. The drag generated by the attached flow over an airfoil or wing, therefore, is called the skin friction drag and it is strongly dependent on the Reynolds number.

- Agrawal, Brij N.; Platzer, Max F.. Standard Handbook for Aerospace Engineers, Second Edition . McGraw Hill LLC.

Laminar Flow

Figure 3.17 shows that over the airfoil's leading edge the air particles move downstream in smooth and regular trajectories without appreciable mixing between different layers of air. This type of flow is known as a laminar boundary layer. The nondimensional group, which heavily influences the development of any boundary layer, is the Reynolds number, based on the surface distance from the origin of the boundary layer to the point in question. When working with boundary layer flows it is convenient to use a body-fitted (or curvilinear) coordinate system in which the x-direction is taken to represent the distance traveled along the surface and the y-direction is taken normal to the surface. Figure 3.18 illustrates the variation in local velocity within a laminar boundary layer; this variation is referred as a velocity profile.

Agrawal, Brij N.; Platzer, Max F.. Standard Handbook for Aerospace Engineers, Second Edition. McGraw Hill LLC.

Transitions

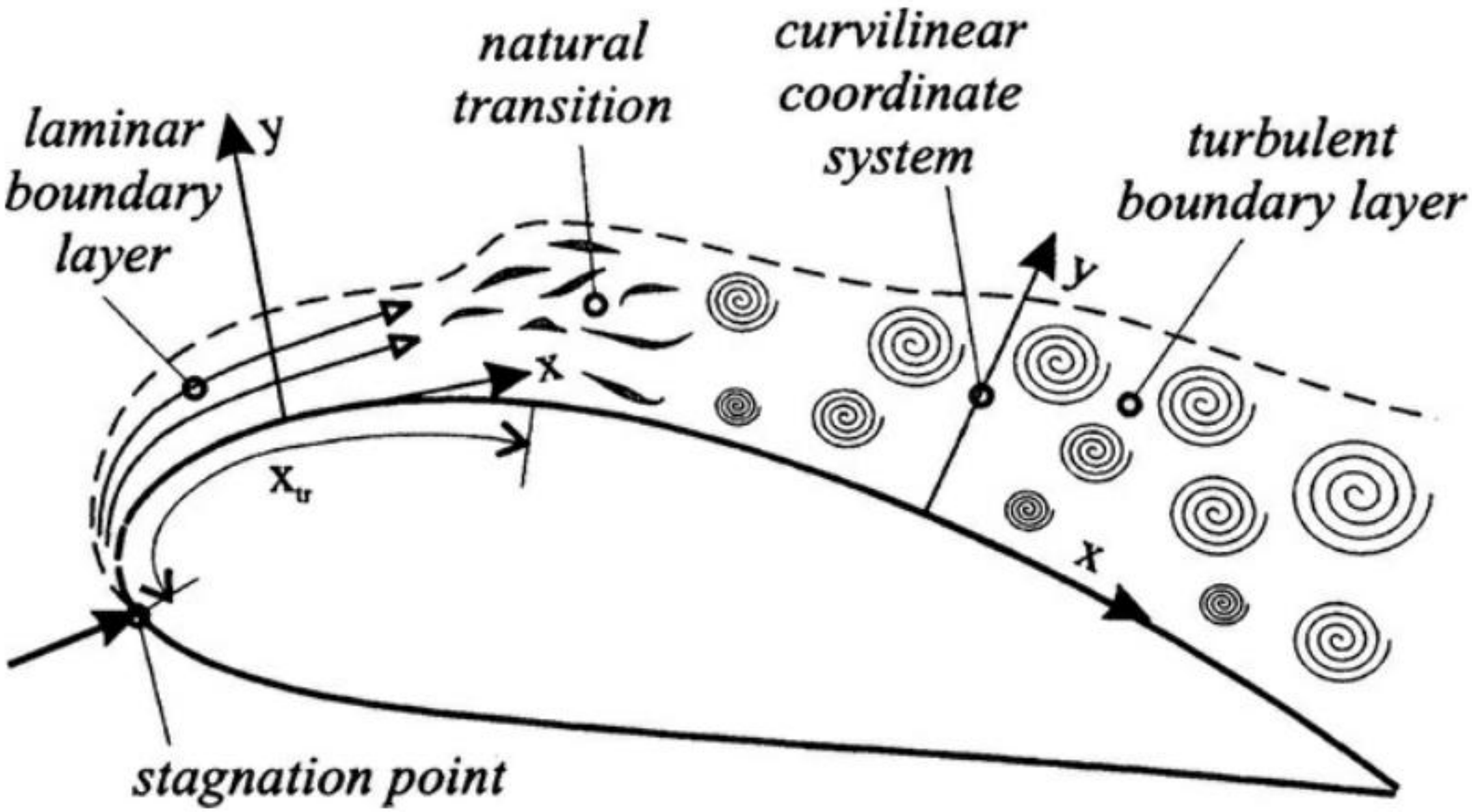


FIGURE 3.17 Typical boundary layer phenomena. Note that the boundary layer thickness is vastly exaggerated for illustrative reasons.

Agrawal, Brij N.; Platzer, Max F.. Standard Handbook for Aerospace Engineers, Second Edition. McGraw Hill LLC.

Transition Demonstration

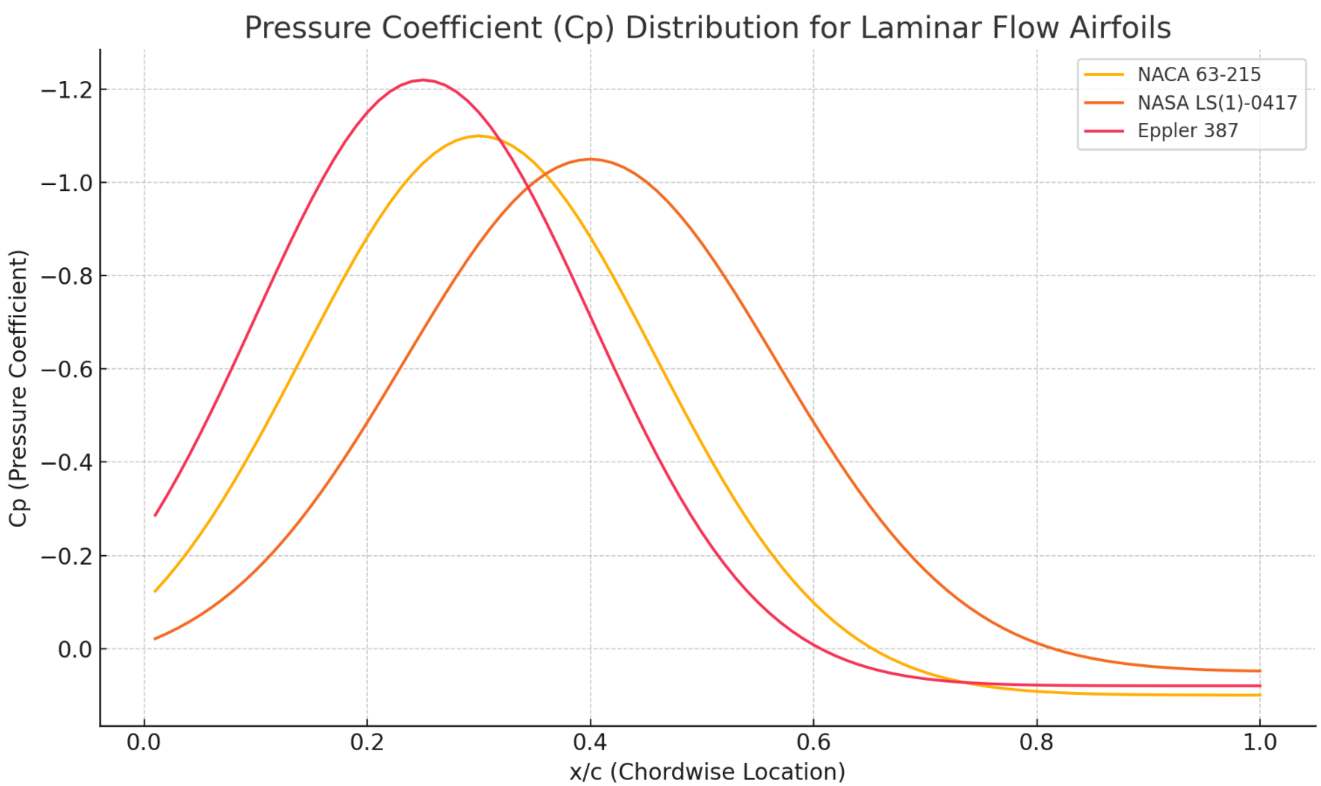
https://www.youtube.com/watch?v=calMxpcEfUo

Feature	Description
Thin leading edge	Helps maintain smooth flow longer
Gentle pressure gradients	Designed to avoid early flow separation
Low camber	Reduces pressure spikes
Transition control	Tailored surface shapes that promote stable laminar flow

A laminar flow airfoil is an airfoil specifically designed to maximize the region of laminar flow over its surface, especially on the leading edge. These airfoils are engineered to delay the transition from laminar (smooth, orderly) flow to turbulent (chaotic) flow — which reduces skin friction drag and improves aerodynamic efficiency

Laminar Flow Airfoil

Pressure Coefficients



X-axis: Location along the chord (from leading edge to trailing edge)
Y-axis (Cp): Pressure coefficient — lower Cp = higher suction pressure
The plot is inverted, as is standard in aerodynamics (higher suction appears lower on the graph).

Mach Number

M= $\frac{V}{a}$

Where :

V = Velocity of the object (m/s)

a = Speed of sound in the medium (m/s), a=$\sqrt{\gamma RT}$

Speed of sound depends on:

γ= Ratio of specific heats (typically 1.4 for air)

R = Specific gas constant

T = Absolute temperature (K)

Stagnation pressure

Stagnation pressure is a key concept in fluid dynamics, especially in aerospace and compressible flow applications. It represents the maximum pressure a moving fluid can attain when it is brought to a complete stop (zero velocity) isentropically — i.e., without any losses due to friction, shock waves, or heat transfer.

The stagnation pressure for Incompressible flow P_0 is given by:

$$P_0 = P + \frac{1}{2}\rho V^2$$

For compressible flow it is:

$$P_0 = P\left(1 + \frac{\gamma - 1}{2} M_2\right)^{\frac{\gamma}{\gamma - 1}}$$

Total Stagnation

- Total (Stagnation) Temperature and Pressure
  - Total Temperature
- $T_0 = T\left(1 + \frac{\gamma-1}{2} M\right)$
  - b) Total Pressure
- $P_0 = p\left(\frac{1-\gamma}{2} M^2\right)^{\frac{\gamma}{\gamma-1}}.$

- γ Ratio of specific heats (Cp/Cv)
- T2, T1 Static and stagnation temperature
- P2, P1 Static and stagnation pressure
- M Mach number

Stagnation Temperature

The stagnation temperature (also called total temperature) is the temperature a moving fluid (like air) would reach if it were brought to rest isentropically (i.e., without any heat transfer or frictional losses).

Normal Shock Relations

Downstream Mach Number (after a normal shock)

$$M_2 = \sqrt{\frac{(\gamma - 1)M_1^2 + 2}{2\gamma M_1^2 - (\gamma - 1)}}$$

Pressure Ratio across Shock

$$\frac{P2}{P1} = 1 + \frac{2\gamma}{\gamma + 1}(M_1^2 - 1)$$

Temperature Ratio across Shock

$$\frac{T_2}{T_1} = \frac{[2\gamma M_1^2 - (\gamma - 1)][(\gamma - 1)M_1^2 + 2]}{(\gamma + 1)^2 M_1^2}$$

γ	Ratio of specific heats (Cp/Cv)
µ	Dynamic viscosity
T2, T1	Static and stagnation temperature
P2, P1	Static and stagnation pressure
R	Specific gas constant
M	Mach number

Heat Transfer (Stagnation Point Heating Rate

$$\dot{q}_s = C \cdot \frac{\rho_e^{0.5} V_e^3}{\sqrt{r_n}}$$

Where:

$\dot{q}_s$ = Stagnation heat flux (W/m²)

$\rho_{e1} V_e$ = freestream density and velocity

r_n = Nose radius of the vehicle

C = Empirical constant

Reynolds Number

Helps predict boundary layer behavior (laminar vs. turbulent). At hypersonic speeds:

Re is high, leading to thin, high-shear layers.

Transition to turbulence affects heating significantly.

Whether flow is laminar or turbulent

$$Re = \frac{\rho v d}{\eta}$$

Where
Re – Reynold's number
ρ – density of fluid
v – velocity of fluid
d – diameter of tube and
η – viscosity of fluid

Reynold's number

Reynold s number of 2000 – borderline

When

Re < 2000 – laminar

Re > 2000 – turbulent

These are rough approximate numbers. You will see other estimates that differ, but not extremely.

- ✓ Viscosity is the important property of laminar flow
- ✓ Density is the important property of turbulent flow
- ✓ Reynold's number of 2000 delineates laminar from turbulent flow

Specific Impulse (for Hypersonic Propulsion)

$$I_{sp} = \frac{F}{\dot{m} g_0}$$

F = thrust

$\dot{m}$ = mass flow rate

g_0 = gravitational acceleration ($9.81\ m/s^2$)

Used to assess scramjets, ramjets, or rocket-boosted systems.

$$\Delta s = R \ln\left(\frac{P_2}{P_1}\left(\frac{\rho_1}{\rho_2}\right)^{\gamma}\right)$$

Entropy Across Shock

Entropy increases across a shock due to irreversibility and is a key loss mechanism in hypersonics.

P_2, P_1	Static and stagnation pressure
R	Specific gas constant
γ	Ratio of specific heats (Cp/Cv)
ρ	Density

Applying formulas to Russia's Avangard

Applying hypersonic flight formulas to the Russian Avangard involves estimating key parameters such as Mach number, stagnation conditions, shock heating, and dynamic pressure. While exact data for Avangard is classified, open-source estimates allow for reasonably accurate engineering approximations.

, Ph.D.[2], D.Sc.

Applying formulas to Russia's Avangard

Stagnation Temperature

$$T_0 = T\left(1 + \frac{\gamma - 1}{2} M^2\right) = 250\left(1 + \frac{0.4}{2} \cdot 625\right) = 250(1 + 125) = 31,500\,\mathrm{K}$$

This is a theoretical stagnation temperature—well above material limits. Real vehicles experience radiative cooling and ablation to survive.

Applying formulas to Russia's Avangard

Dynamic Pressure q

Assuming:
Air density at 40 km = $\rho = 4 \times 10^{-3}\,\mathrm{kg/m^3}$

$$q = \frac{1}{2}\rho V^2 = \frac{1}{2} \cdot 0.004 \cdot (7,925)^2 \approx 125,400\,\mathrm{Pa}\,(\mathrm{or}\ 125.4\,\mathrm{kPa})$$

This dynamic pressure drives aerodynamic heating, control force loads, and structural stress.

Applying formulas to Russia's Avangard

Heat Flux at Stagnation Point

Using $$\dot{q}_s = C \cdot \frac{\rho^{0.5} V^3}{\sqrt{r_n}}$$

Assuming C ~ 1.83×10^{-4}

$$\dot{q}_s = 1.83 \times 10^{-4} \cdot \frac{(0.004)^{0.5} \cdot (7925)^3}{\sqrt{0.5}} \approx 1.83 \times 10^{-4} \cdot \frac{0.063 \cdot 4.97 \times 10^{11}}{0.707} \approx 8.0 \times 10^6 \,\mathrm{W/m^2}$$

So, heating at the nose tip may approach 8 MW/m², requiring specialized ablative materials and TPS architecture.

Newton's second law

Translational Motion (for aircraft, spacecraft, missiles):

$\vec{F} = m\vec{a}$

Where:

- F : net force vector (N)
- m: mass (kg)
- $\vec{a}$: acceleration vector (m/s²)

For forces in aerospace:

- $\vec{F} = \vec{T} + \vec{L} - \vec{D} - \vec{W}$
- T: thrust
- L: lift
- D: drag Note drag is negative in vector form
- W: weight

Euler’s Buckling Load

When a long, slender column is subjected to axial compressive force, it may suddenly deflect sideways (buckle) at a certain load, rather than just compressing. This critical load is lower than the material's yield strength, making buckling a geometric and stability problem, not purely a strength issue.

Euler’s Buckling Load is a classical solution in structural engineering and solid mechanics that predicts the critical axial load at which a slender column (or beam) will buckle. Buckling is a form of instability that can cause structural failure, even if the material is still within its elastic range.

Euler's Buckling Load

$$p_{cr} = \frac{\pi^2 EI}{(KL)^2}$$

Where:

P_{cr}: Critical (buckling) load (N)

E: Young's modulus of the material (Pa)

I: Minimum area moment of inertia of the column's cross-section (m^4)

L: Actual length of the column (m)

K: Effective length factor, depends on end conditions (dimensionless)

Note: Youngs Modulus is a mechanical property of solid materials that measures the tensile or compressive stiffness when the force is applied lengthwise

Euler's Buckling Load

Assumptions in Euler's Theory

- Column is perfectly straight and slender
- Material is homogeneous, linearly elastic
- Load is axially applied and centered
- No initial imperfections or eccentricity

Euler's Buckling Load

Example Calculation

Let's say:

•A steel column with:

- E= 200×10^9 Pa
- I=8×10^{-6} m^4
- L=2.0 m
- Pinned-pinned: K=1.0

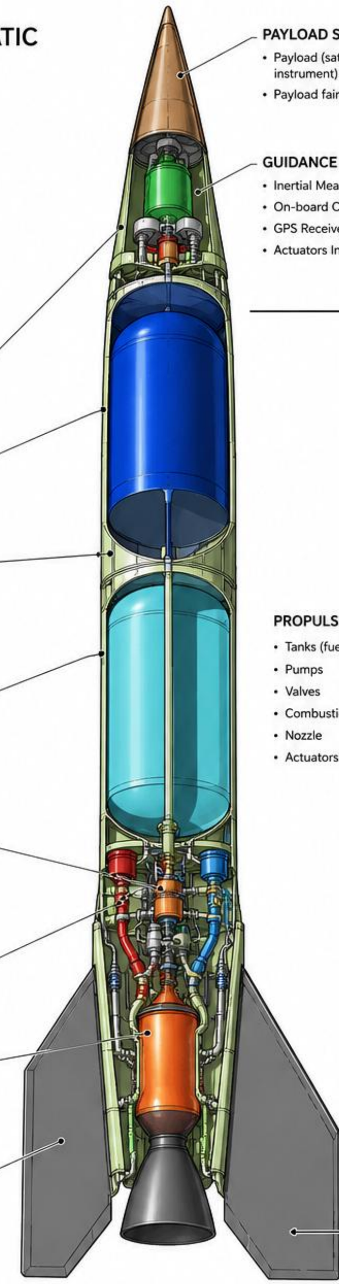
$$P_{cr} = \frac{\pi^2 (200 \times 10^9)(8 \times 10^{-6})}{(1 \cdot 2.0)^2} = 987,000\,\mathrm{N}$$

So, the column will buckle at ~987 kN, even if the material has a much higher yield strength.

Rocket Components

ROCKET VEHICLE ENGINEERING SCHEMATIC

OVERVIEW	
Type	Single-stage, Liquid Propellant Rocket
Configuration	Finned, Conical Nose
Diameter (max)	D
Length (overall)	L
Mass (gross)	m_0
Propulsion	Liquid Bipropellant (LOX / RP-1)
Application	Launch Vehicle / Suborbital / Tactical



NOSE CONE / NOSE CODE
- Aerodynamic nose cone
- Radome (if applicable)
- Houses sensors / antenna

FUEL TANK
- Stores fuel (e.g., RP-1)
- Pressurized / unpressurized
- Baffles to control slosh

INTERTANK / FRAME
- Structural load transfer
- Separates fuel and oxidizer tanks
- Provides mounting points for systems

OXIDIZER TANK
- Stores oxidizer (e.g., LOX)
- Insulated to minimize boil-off
- Pressurized / unpressurized

PUMPS
- Fuel Pump
- Oxidizer Pump
- Driven by Turbopump
- Increase propellant pressure to chamber

COMBUSTION CHAMBER
- Mixing of fuel and oxidizer
- Sustains combustion
- High-temperature alloy chamber liner

NOZZLE
- Expands hot gases
- Converts thermal energy to kinetic energy
- Nozzle exit diameter d_e

FIN SYSTEM
- Provides aerodynamic stability
- Control (if movable) or fixed
- Attached to frame structure

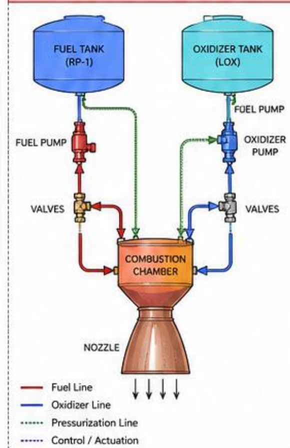
PAYLOAD SYSTEM
- Payload (satellite / warhead / instrument)
- Payload fairing (if used)

GUIDANCE SYSTEM
- Inertial Measurement Unit (IMU)
- On-board Computer (OBC)
- GPS Receiver / Antenna
- Actuators Interface Unit

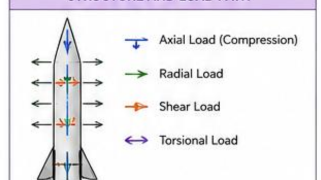
PROPULSION SYSTEM
- Tanks (fuel / oxidizer)
- Pumps
- Valves
- Combustion Chamber
- Nozzle
- Actuators / TVC

GUIDANCE AND CONTROL DETAILS
- Antenna / Radome
- GPS Receiver
- IMU
- On-board Computer
- Power Supply
- Actuator Interface

PROPULSION SYSTEM DETAILS



STRUCTURE AND LOAD PATH



TYPICAL SPECIFICATIONS (EXAMPLE)	
Diameter (D)	1.0 m
Length (L)	10.0 m
Gross Mass (m_0)	5,000 kg
Propellant Mass	4,000 kg
Payload Mass	200 kg
I_{sp} (Vacuum)	~ 300 s
Thrust (Sea Level)	~ 80 kN
Burn Time	~ 60 s

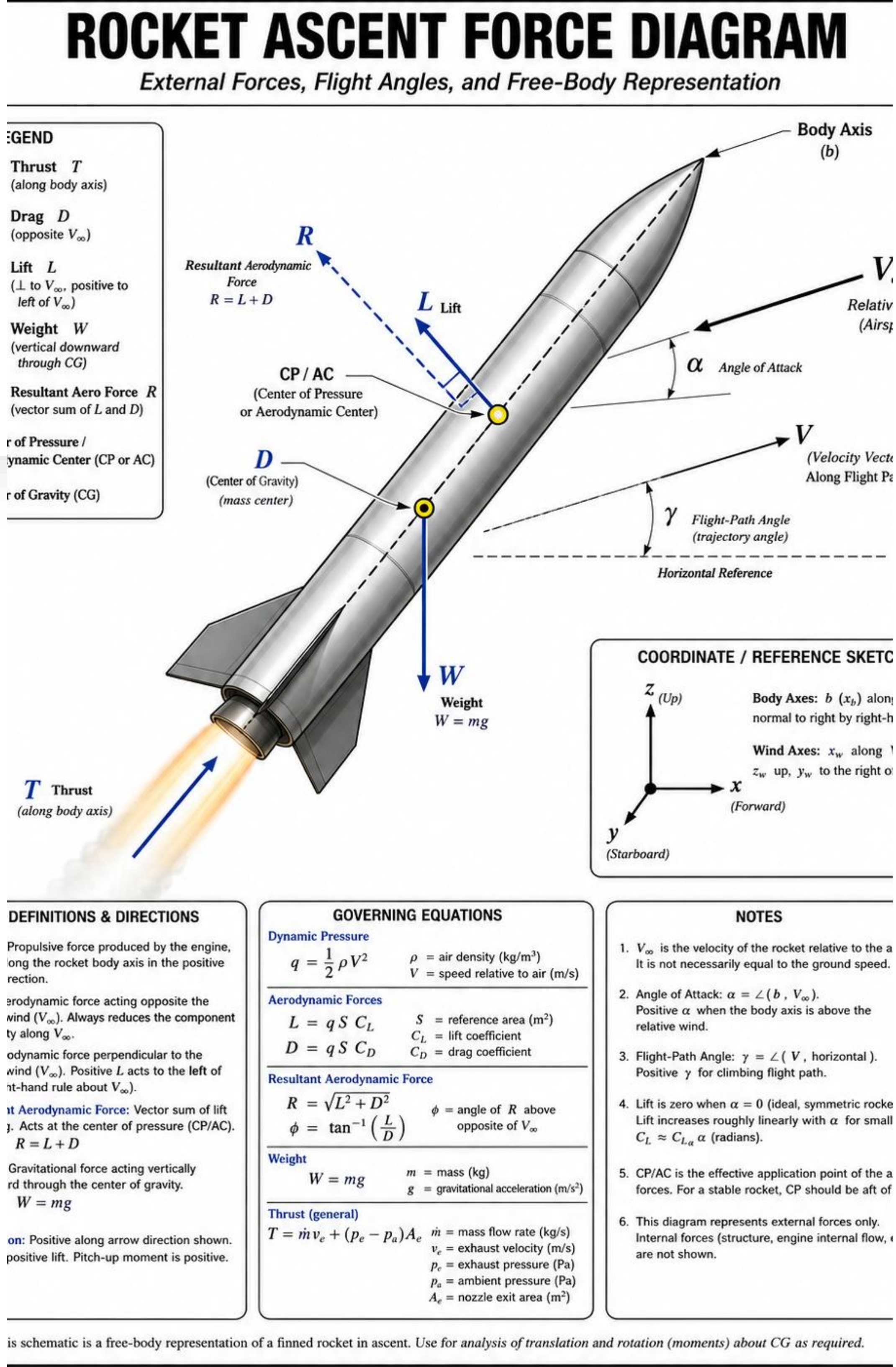
LEGEND (COMPONENT GROUPS)
- Structure / Frame
- Propellant Tanks
- Propulsion Components
- Avionics / Guidance
- Aerodynamic Surfaces
- Payload

NOTES
- Schematic is not to scale.
- Components shown are typical; actual configuration varies by vehicle.
- LOX = Liquid Oxygen
- RP-1 = Refined Petroleum (Rocket Propellant-1)
- TVC = Thrust Vector Control

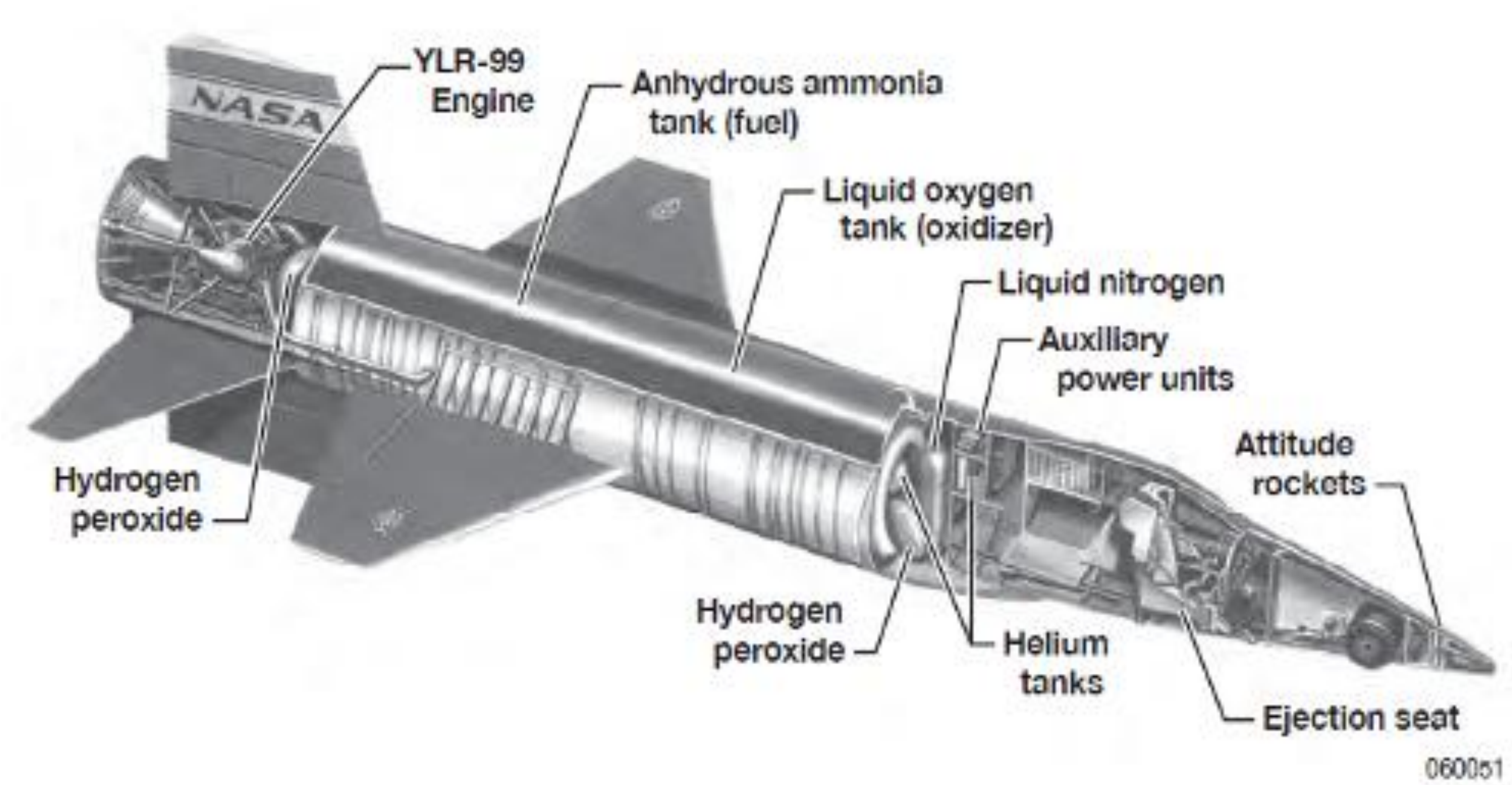
Forces on Rocket in Flight

Four Forces

- Weight
- Thrust
- Drag
- Lift



Hypersonic Vehicle



Structural Elements in Missiles (including hypersonics)

Component	Function	Failure Modes
Fins	Stabilization/control	Flutter, fracture
Interstages	Connect missile sections	Buckling, joint failure
Brackets & Pylons	Mounting to launcher or airframe	Shear, vibration, fatigue
Frames & Stringers	Internal structure	Buckling, crushing

Key Loads Acting on Missile Support Structures

Axial Load: from engine thrust or aerodynamic deceleration

Lateral Load: due to wind gusts, maneuvering

Vibrational Load: from motor vibration, airflow, etc.

Thermal Stress: high heating at hypersonic speeds

Shock Loads: during stage separation or launch release

Material Comparisons

Material	Strength	Temperature Resistance	Weight
Aluminum Alloy	Moderate	Low (~150°C)	Light
Titanium Alloy	High	Moderate (~600°C)	Medium
Carbon Composites	Very High (anisotropic)	Good (depends on resin)	Very Light

Relevant Concepts to Keep In Mind

- **SUBSONIC Gas Flow**
  - Flow velocity is less than the speed of sound (< ~700 MPH)
  - As the area through which the gas is flowing **decreases**, the flow velocity increases and the pressure decreases
  - The convergent section of a nozzle increases the velocity of the subsonic flow

- **SONIC Gas Flow**
  - Where the flow velocity is equal to the speed of sound (= ~700 MPH)
  - By design, this condition occurs at the throat (smallest cross-sectional area in the nozzle)

- **SUPERSONIC Gas Flow**
  - Flow velocity is greater than the speed of sound (> ~700 MPH)
  - As the area through which the gas is flowing **increases**, the flow velocity increases and the pressure decreases
  - The divergent section (a.k.a. exit cone) of the nozzle increases the velocity of the supersonic flow

Thrust Equation

$$\text{Thrust} = ((\text{Mass}_{\text{Propellant}} * \text{Velocity}_{\text{Exhaust}}) / \text{Time}) + (P_e - P_a) * A$$

From this equation, we can see there are several ways to increase the thrust generated by a rocket motor:

1. Increase the total mass that is ejected from the motor
2. Burn the propellant faster (reduce the time period over which the motor burns)
3. Design a nozzle that maximizes the velocity of the gas that is ejected from the motor

•

Increasing the Mass Being Ejected from the Rocket Motor

There are two ways to increase the mass is to use a more dense propellant, or increase the amount of propellant.

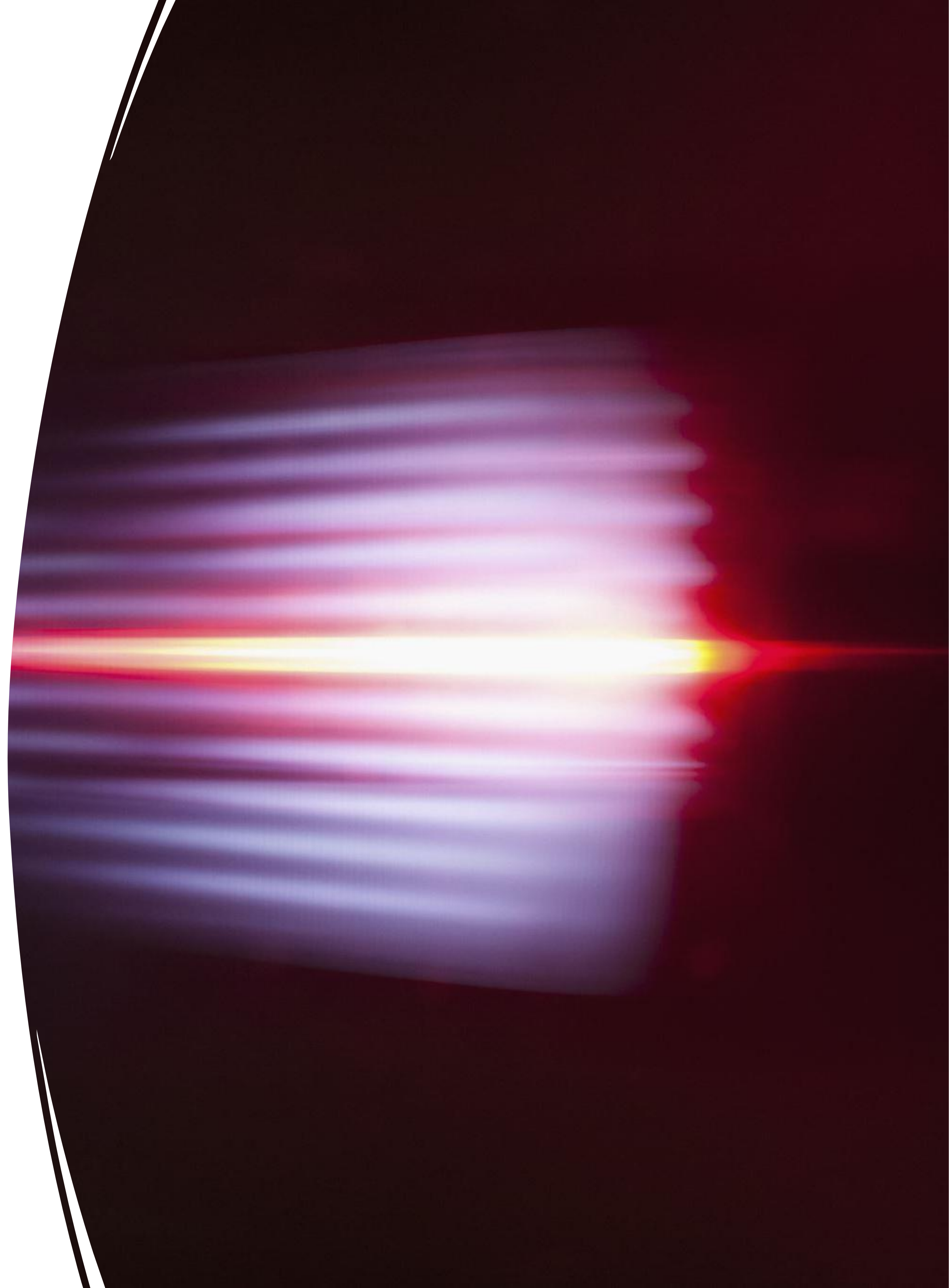
- Higher density propellant, or adding more propellant, causes the mass of the rocket motor to increase. While more mass will be ejected and thrust would increase, the rocket motor will be heavier, and F=ma says the rocket will not accelerate as much. Any gain in thrust will generally be offset by the reduction in final velocity due to the lower acceleration.

- More fuel makes for a more complex motor, and certainly a more expensive one.

- This is not the solution to maximizing the efficiency of a given rocket motor

Burning Propellant Faster

- We can dump more mass out the nozzle faster by burning the propellant quicker.
- This sounds great, but we have a limited amount of propellant, so as the motor burns faster, the burn time will be shorter.
- This sounds great, but we have a limited amount of propellant, so as the motor burns faster, the burn time will be shorter.
- Faster burning propellant will increase acceleration, but since the motor will produce the acceleration over a shorter period, the final velocity will generally not be any higher
- Burning all the propellant at once is also know as an explosion.

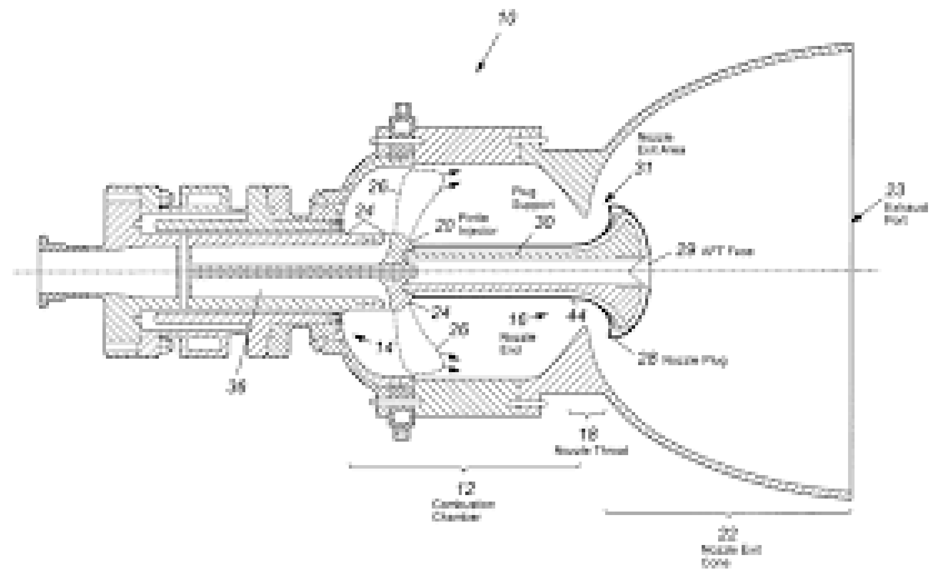


Increasing the Exhaust Exit Velocity

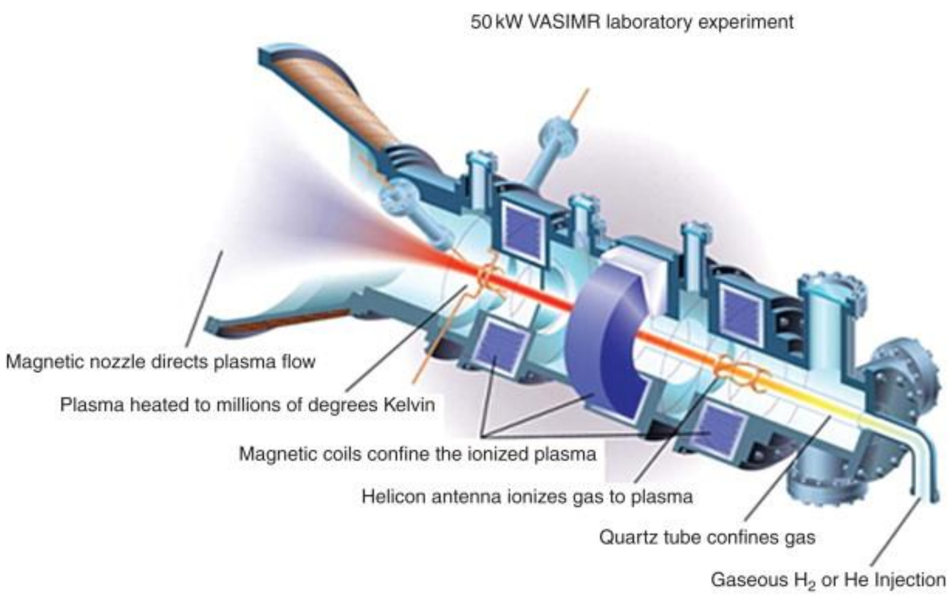
- The only option left is to put a nozzle on the back of the motor to accelerate the exhaust gasses leaving the rocket motor.
- To do this, we need to apply the earlier concepts of subsonic and supersonic gas flows, and design an efficient nozzle that perfectly expands the exhaust gas.
- The next slides will examine rocket nozzles.

Nozzle Geometry

A fixed nozzle geometry refers to a nozzle where the exit area cannot be adjusted or changed during operation. Fixed geometry nozzles are simpler in design and construction but may not be as efficient across a wide range of operating conditions as variable geometry nozzles. The defining feature is that the physical size and shape of the nozzle exit remain unchanged during use. While simple, fixed nozzles may not be as efficient as variable geometry nozzles across a wide range of operating conditions, particularly when there are significant changes in mass flow rate or pressure.



Nozzle Geometry



Variable geometry nozzles are engineered to adjust their shape and size to optimize performance under varying operating conditions, particularly in situations involving changes in airflow or pressure. This adaptability leads to enhanced efficiency and performance in diverse applications, from jet engines to turbochargers and even spray systems. Variable geometry nozzles incorporate mechanisms that allow for changes in the nozzle's exit area or shape. This can involve moving vanes, sliding plates, or other adjustable components.

Nozzle Shapes

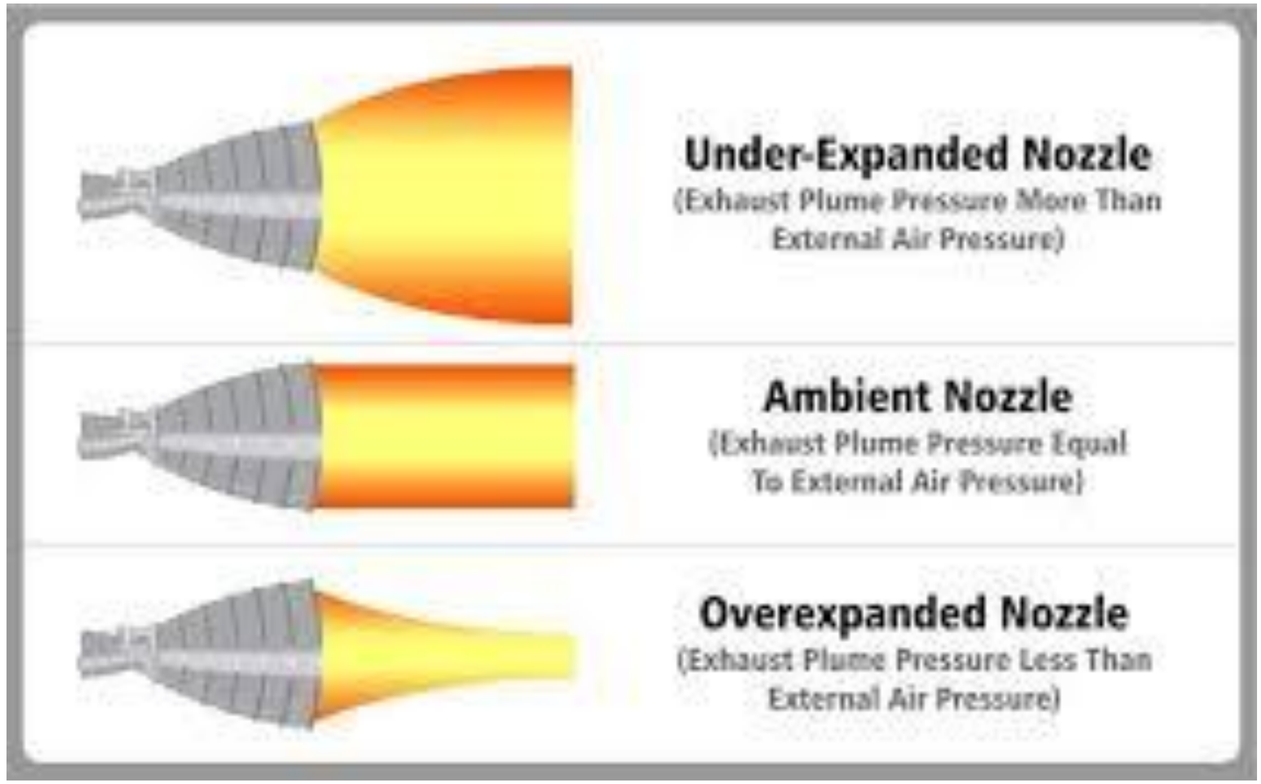
Conical: The simplest design, often with a 15-degree half-angle for efficiency, but often less efficient than other shapes.

Bell Nozzle: A more complex, contoured shape that offers higher efficiency compared to conical nozzles, though they are more expensive to manufacture. The bell shape is often generated using parabolic approximations or the method of characteristics.

Plug/Aerospike Nozzle: These are altitude-compensating nozzles, meaning they maintain efficiency at varying altitudes. They can be annular or linear.

Expansion-Deflection Nozzle: Another type of altitude-compensating nozzle, often shorter than other enclosed nozzles, making them suitable for smaller, inexpensive thrusters.

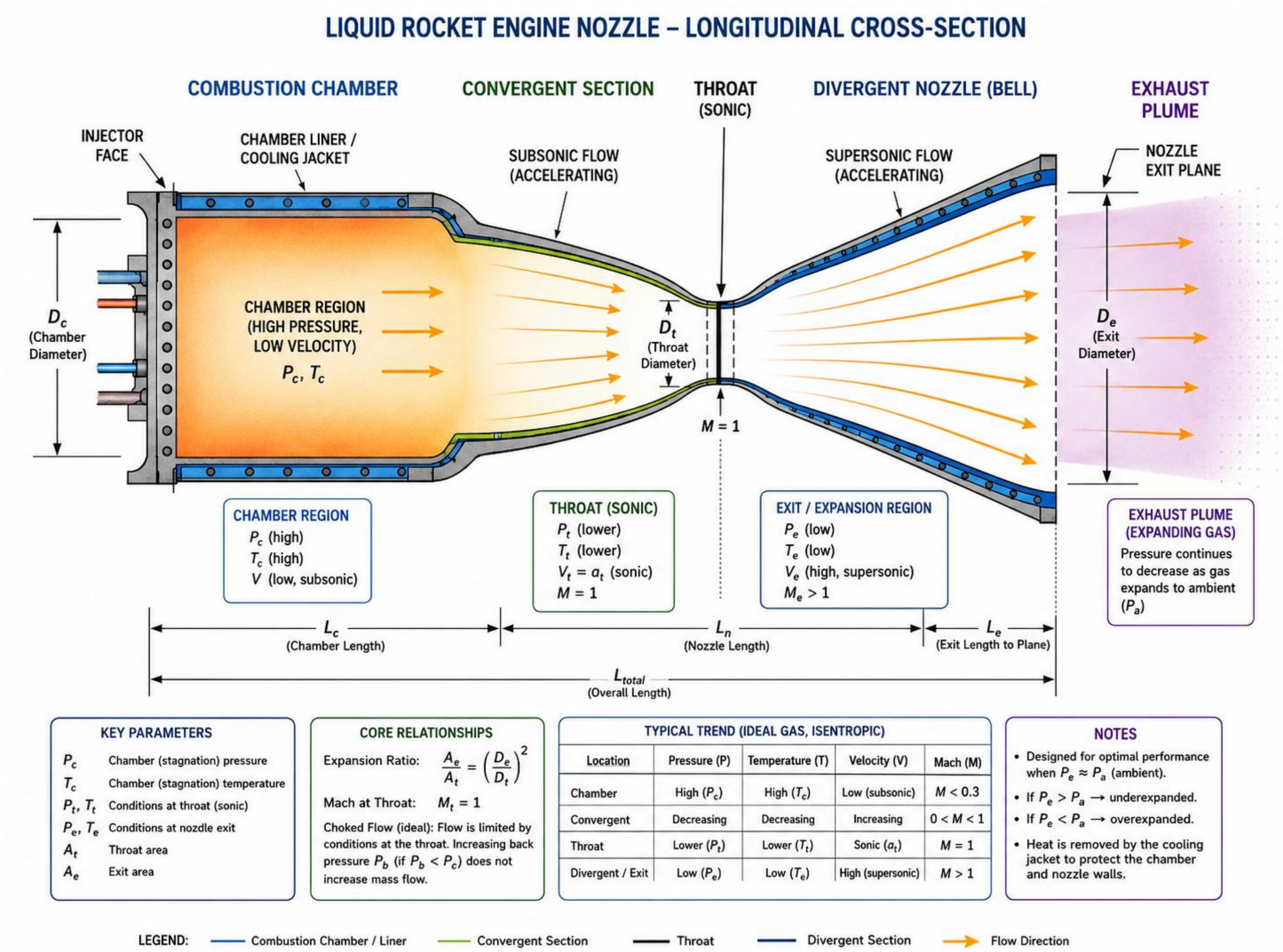
Nozzle expansion



Convergent-Divergent Nozzle

A convergent-divergent nozzle is also called a CD nozzle, con-di nozzle, or deLaval nozzle. The converging-diverging nozzle traces back to Venturi tubs created by Giovanni Battista Venturi. A Swedish Engineer named Gustaf de Laval uses his own converging diverging design in his impulse turbine engine in 1888. Later, Robert Goddard used Laval's nozzle in his rocket.

Elements of the Rocket Nozzle



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

Computational Fluid Dynamics (CFD)

Due to the complexity of scramjet flows, CFD simulations are vital:

Turbulence modeling: RANS (Reynolds-Averaged Navier-Stokes), LES(Large Eddy Simulation), or hybrid models

Combustion models: Finite-rate chemistry, flamelet models, or CMC (Conditional Moment Closure)

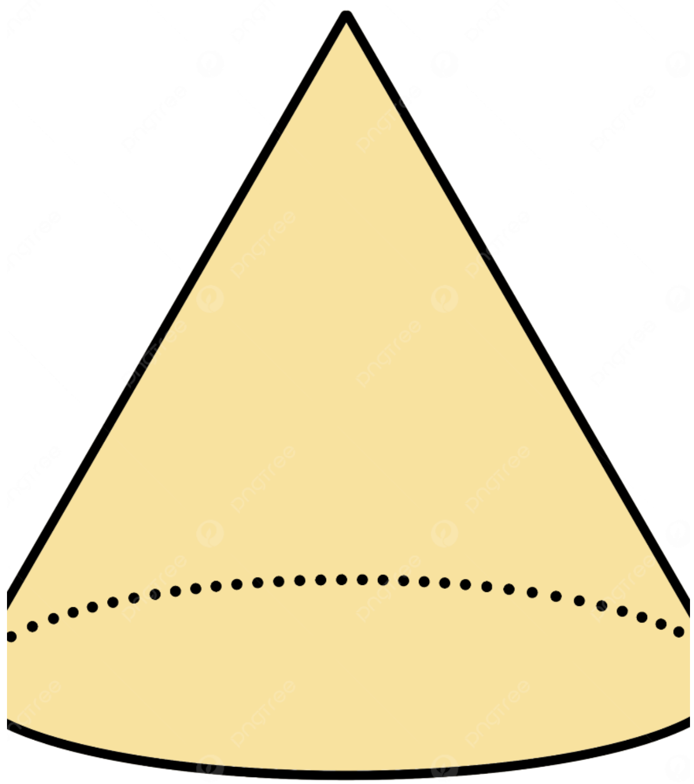
Shock-capturing schemes: Roe's scheme, TVD (Total Variation Diminishing) methods

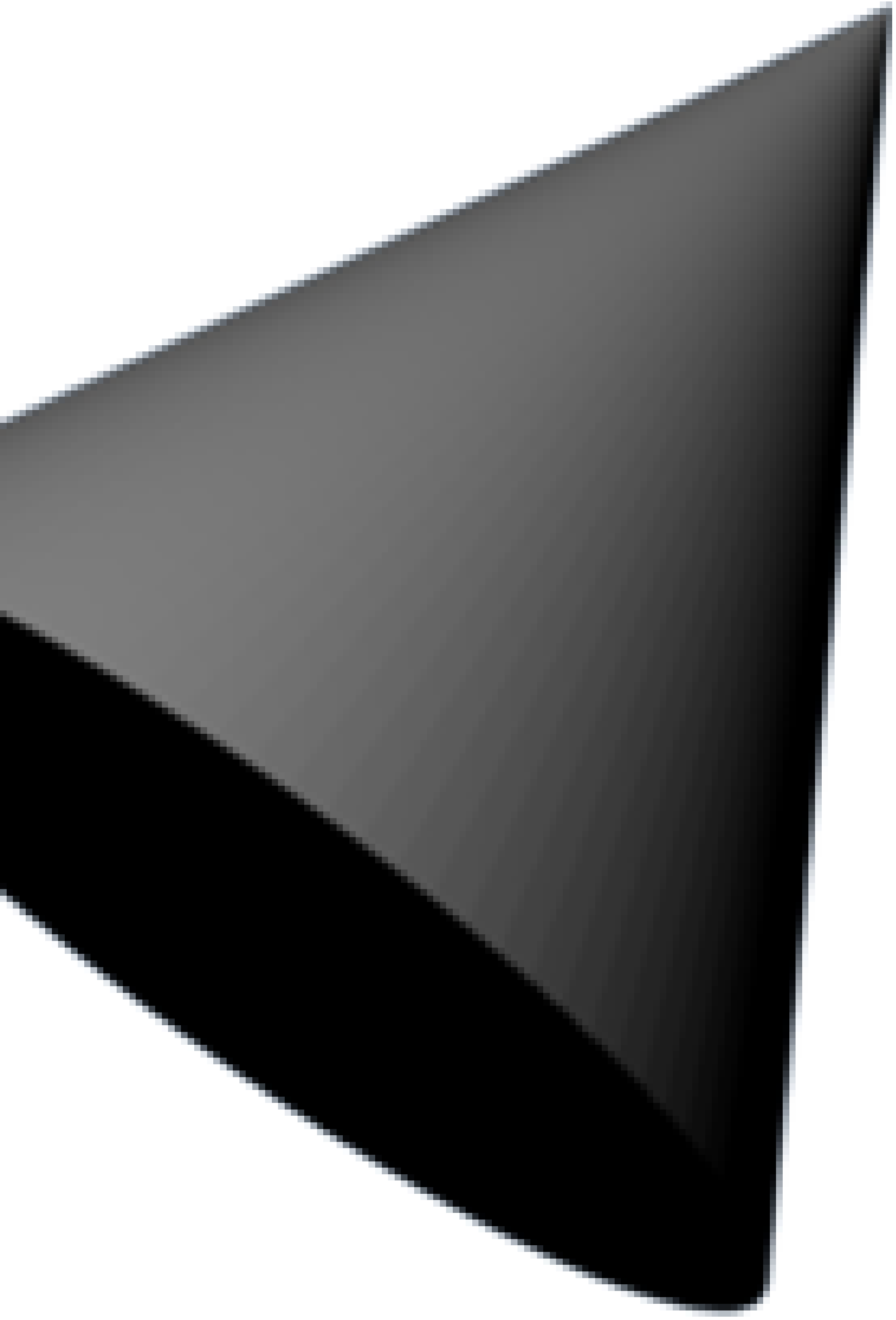
Grid resolution: High refinement near shocks and flame zones

Conical Flow

- Conical flow is a special class of compressible fluid flow in which the flow field variables (like velocity, pressure, and density) depend only on the angle from a point (the vertex of a cone), and not on the distance from that point. It is widely used in high-speed aerodynamics, particularly in problems involving supersonic flow over cones or pointed bodies.

- In conical (self-similar) flow, all streamlines emanate from a single point (usually the origin or a cone vertex), and the flow variables are constant along rays from this point, depending only on the cone angle (θ).



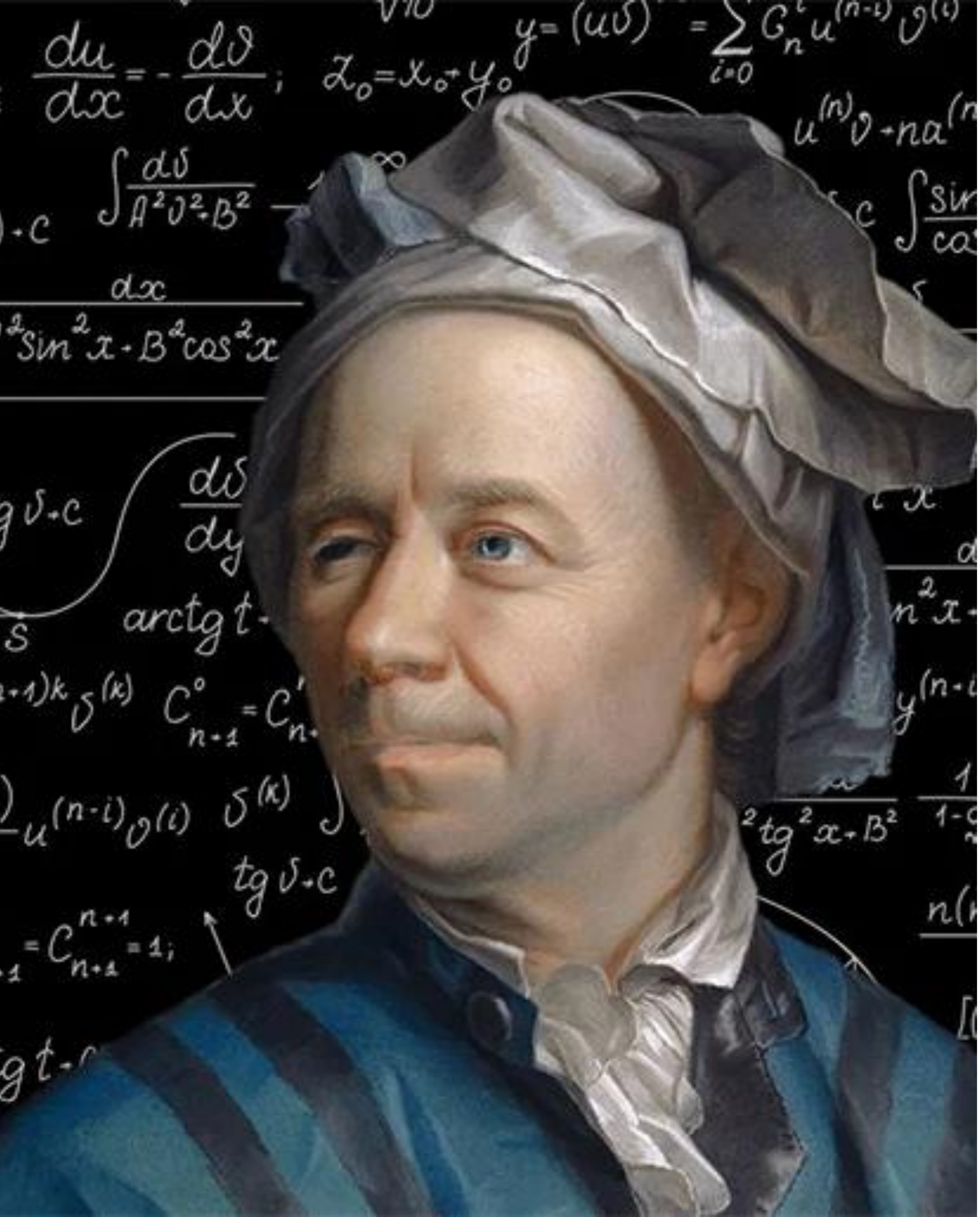


Conical Flow

- Key Assumptions
- Steady flow (no time dependence)
- Compressible, usually supersonic
- Axisymmetric or planar symmetry
- No body forces (e.g., gravity ignored)
- Flow properties vary only with angle θ, not with radial distance r

Conical Flow Fields

Aspect	Description
Flow type	Self-similar, compressible, supersonic
Dependent variable	Cone angle θ only
Main application	Flow around sharp cones, hypersonic bodies
Equations	Reduced Euler equations, Taylor–Maccoll ODE
Shock geometry	Conical (emanates from tip)
Used for	Simplified analysis of supersonic/hypersonic vehicles



Reduced Euler Equations

- The Reduced Euler Equations refer to a simplified form of the compressible Euler equations under specific assumptions, such as symmetry, self-similarity, or steady flow conditions, that allow partial differential equations (PDEs) to be reduced to ordinary differential equations (ODEs) or fewer variables. The Euler equations govern the motion of an inviscid (non-viscous), compressible fluid

Reduced Euler Equations

In conservative form, they are:

Continuity (Mass Conservation) $$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0$$

Momentum Equation $$\frac{\partial (\rho \vec{V})}{\partial t} + \nabla \cdot (\rho \vec{V} \vec{V} + p\mathbf{I}) = 0$$

Energy Equation $$\frac{\partial E}{\partial t} + \nabla \cdot ((E + p)\vec{V}) = 0$$

Where:

ρ: density
$\vec{V}$: velocity vector
p: pressure
$E=\rho e+1/2|\vec{V}|^2$
|2: total energy per unit volume

Reduced Euler Equations

Why are they called 'reduced'? It is because they are obtained when applying assumptions such as:

- Symmetry or Self-similarity
  - 1D flow: All properties vary only along one direction (e.g., x)
  - Axisymmetry: No variation with azimuthal angle
  - Conical symmetry: Properties vary only with polar angle θ, not with radius r (common in supersonic flow over cones)

- Neglect of Certain Terms
  - No body forces (gravity, magnetics)
  - No heat transfer or viscosity (purely inviscid)
- These assumptions convert the original 3D PDE system into a simplified set of equations, often 1D or even ODEs, much easier to solve analytically or numerically.

Reduced Euler Equations

Reduced Euler Equations in Conical Flow
For inviscid, compressible, steady, axisymmetric, conical flow, the equations reduce due to self-similarity:
Let the velocity vector in spherical coordinates be:

$$\vec{V} = V_r(r,\theta)\hat{r} + V_\theta(r,\theta)\hat{\theta}$$

Assuming conical self-similarity:

$Vr=v_r(\theta)$
$V\theta=v_\theta(\theta)$
$\rho=\rho(\theta)$, $p=p(\theta)$

Then the Euler equations reduce to ODEs in θ only, forming the Taylor–Maccoll equation for supersonic flow over a cone.

.

Taylor–Maccoll Equation (for Axisymmetric Conical Flow)

Derived for supersonic flow around a sharp cone, this equation relates:

- Flow deflection angle (θ)
- Mach number and pressure gradients
- Velocity components (radial and angular)

$$\frac{\mathrm{d}^2 V_\theta}{\mathrm{d}\theta^2} + f\left(V_{r_1} V_\theta, \theta, \gamma\right) = 0$$

Where:

Vr: radial velocity

Vθ: tangential velocity

γ: specific heat ratio

Kármán–Moore theory

The Kármán–Moore theory is a framework used to describe the transition from laminar to turbulent flow in fluid dynamics. In the Kármán–Moore theory, the transition from laminar to turbulent flow is studied in the context of boundary layers and particularly in fluid flows around bodies, such as airfoils or in pipes. The theory builds on the observation that flow can transition from laminar to turbulent based on certain factors like the Reynolds number, which measures the relative importance of inertial forces to viscous forces in the fluid. The key factors are:

- **Reynolds Number (Re)**: The theory posits that at higher Reynolds numbers, the likelihood of flow becoming turbulent increases. For boundary layers, the transition to turbulence typically occurs when Re exceeds a certain threshold, often around 500,000 in practical applications (though it can vary depending on other factors like surface roughness and flow conditions).
- **Instability in the Flow**: As the flow moves along a surface, small disturbances or instabilities in the laminar flow can grow and eventually lead to turbulence. The theory explores how these instabilities develop, evolve, and result in a fully turbulent state.
- **Bluff Bodies and Streamlines**: In the presence of bluff bodies (objects with large, irregular shapes like a flat plate or a cylinder), flow often experiences a separation point where the flow detaches from the surface and forms large-scale vortices. These vortices are associated with turbulent flow.

Prandtl Meyer

As an object moves through a gas, the gas molecules are deflected around the object. If the speed of the object is much less than the speed of sound of the gas, the density of the gas remains constant and the flow of gas can be described by conserving momentum and energy. As the speed of the objects increases towards the speed of sound, we must consider compressibility effects on the gas. The density of the gas varies locally as the gas is compressed by the object. When an object moves faster than the speed of sound, and there is an abrupt decrease in the flow area, shock waves are generated. If the flow area increases, however, a different flow phenomenon is observed. If the increase is abrupt, we encounter a centered expansion fan

The Prandtl-Meyer function describes the expansion process of a supersonic flow around a convex corner. It's a fundamental concept in compressible flow and gas dynamics, especially relevant in hypersonic and supersonic aerodynamics. When supersonic flow turns around a convex corner, it accelerates and expands isentropically (no heat exchange or entropy change). The flow turns through a finite angle using a continuous fan of expansion waves, forming what's called a Prandtl-Meyer expansion fan. This contrasts with oblique shocks, which occur when flow turns into itself (concave corner), causing compression.

Heat Flux

Heat flux is a fundamental concept in thermodynamics and aerospace engineering, especially critical in hypersonic flight and reentry vehicle design. It refers to the rate at which heat energy is transferred per unit area, typically at a surface exposed to a hot gas or plasma.

Heat Flux (q"): The amount of thermal energy transferred through a surface per unit time and area.

Units:

- SI: Watts per square meter (W/m²)
- CGS: cal/cm²/s

$$q'' = \frac{\mathrm{d}Q}{A * \mathrm{d}t}$$

q" = heat flux

dQ = differential heat transfer (Joules)

A = area (m²)

dt = time interval (seconds)

Factors Affecting Heat Flux

Parameter	Effect on Heat Flux
Velocity	Heat flux increases dramatically with speed (~velocity^3)
Atmospheric density	Higher density increases convective heating
Nose radius	Smaller radius = higher stagnation point heating
Flow regime	Turbulent flow enhances heat transfer
Material properties	Affects conduction and ablation response

Prandtl

The concept of a thin, viscous layer next to a body over which a fluid is flowing is due to Prandtl.1 According to Prandtl, the fluid velocity relative to the surface of the body increases from zero at the surface to its maximum value away from the surface in a thin region called the boundary layer. This concept is well established and has become a fundamental postulate of fluid dynamics. The concept has been directly confirmed by careful measurements of velocity distribution through the boundary-layer region. Indirect confirmation is provided by the excellent agreement with measurements of solutions to the boundary-layer equations—a set of differential equations derived from the more general equations of motion using the boundary-layer concept to drop certain terms from the more general equations.

Dorrance, William H.. Viscous Hypersonic Flow: Theory of Reacting and Hypersonic Boundary Layers (Dover Books on Engineering) . Dover Publications.

Prandtl Meyer

As an object moves through a gas, the gas molecules are deflected around the object. If the speed of the object is much less than the speed of sound of the gas, the density of the gas remains constant, and the flow of gas can be described by conserving momentum and energy. As the speed of the objects increases towards the speed of sound, we must consider compressibility effects on the gas. The density of the gas varies locally as the gas is compressed by the object. When an object moves faster than the speed of sound, and there is an abrupt decrease in the flow area, shock waves are generated. If the flow area increases, however, a different flow phenomenon is observed. If the increase is abrupt, we encounter a centered expansion fan.

There are some marked differences between shock waves and expansion fans. Across a shock wave, the Mach number decreases, the static pressure increases, and there is a loss of total pressure because the process is irreversible. Through an expansion fan, the Mach number increases, the static pressure decreases, and the total pressure remains constant. Expansion fans are isentropic.

The calculation of the expansion fan involves the use of the Prandtl-Meyer function. This function is derived from conservation of mass, momentum, and energy for very small (differential) deflections. The Prandtl-Meyer function is denoted by the Greek letter nu (ν) on the slide and is a function of the Mach number M and the ratio of specific heats gam (γ) of the gas:

Prandtl Meyer

The function is shown here

$$\nu(M) = \sqrt{\frac{\gamma+1}{\gamma-1}} \cdot \tan^{-1}\left(\sqrt{\frac{\gamma-1}{\gamma+1}(M^2-1)}\right) - \tan^{-1}\left(\sqrt{M^2-1}\right)$$

Symbol	Meaning
ν(M)	Prandtl-Meyer angle (expansion function) in radians or degrees
M	Mach number (must be > 1)
γ	Ratio of specific heats (1.4 for air)

ν(M) gives the angle through which a supersonic flow must turn to reach Mach MM from sonic conditions (M=1).

The difference $\theta=\nu(M_2)-\nu(M_1)$gives the turn angle needed to go from Mach M_1 to M_2 in an expansion.

Prandtl Meyer

Valid only for supersonic flow (M > 1).

Expansion is isentropic (no energy loss).

Flow properties vary smoothly across the fan.

Common in nozzle design, supersonic intakes, and high-speed external aerodynamics.

Scramjet Flow

Scramjet flow is characterized by:

High Mach number (>5)

Supersonic airflow throughout (including in the combustor)

Strong shock interactions

Real gas effects at high temperatures

Unlike ramjets, scramjets do not decelerate flow to subsonic speeds in the combustor — instead, fuel burns in supersonic flow, making the combustion process time-critical and physically demanding.

Scramjet Flow

Inlet (Compression Region)

Function: Compress and slow incoming high-speed air using shock waves (no moving parts)

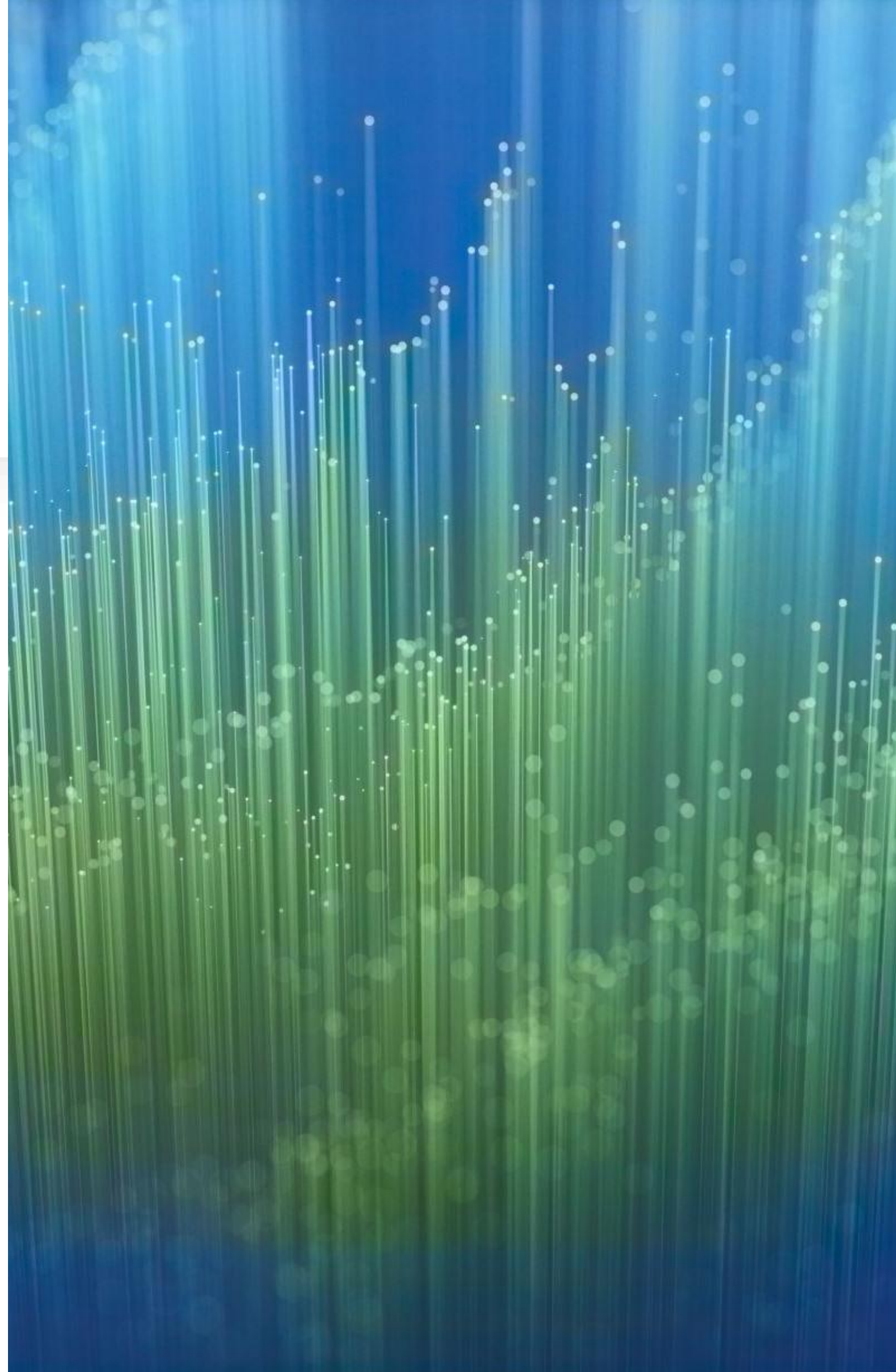
Key phenomena:

Oblique shock waves: Produced by angled surfaces; compress air with minimal entropy increase.

Shock-shock interactions: Complex patterns from multiple shock reflections.

Boundary layer development: Can interact with shocks causing flow separation.

Flow stagnation: Air is compressed and slightly slowed but still supersonic



Scramjet Flow

Nozzle (Expansion Region)

Function: Expand and accelerate hot exhaust gases to produce thrust

Key phenomena:

Supersonic expansion: Via a converging-diverging nozzle

Flow over-expansion or under-expansion: Affects nozzle efficiency at off-design conditions

Nozzle flow separation: Caused by boundary layer interaction

Scramjet Flow

High-Temperature Effects

At Mach >5, air experiences **real gas behavior**:

Vibrational excitation of O_2, N_2 molecules

Dissociation of molecules (especially >2000 K)

Ionization at very high speeds (>Mach 10)

Nonequilibrium chemistry dominates — equilibrium assumptions fail

Summary of Scramjet Physics

Region	Dominant Physics	Modeling Requirements
Inlet	Shock compression, boundary layers	Compressible, viscous CFD
Combustor	Supersonic combustion, mixing	Coupled turbulent-reactive flow
Nozzle	Supersonic expansion, thrust	Nozzle dynamics + heat transfer
Entire Flow	High-speed, high-enthalpy flow	Non-equilibrium gas dynamics

WGS-84 Model

WGS-84 stands for World Geodetic System 1984, and it is the standard coordinate reference system used by the Global Positioning System (GPS). It defines the shape, size, and orientation of the Earth, as well as a reference frame for specifying locations on the planet.
Key Components of WGS-84

- Ellipsoidal Earth Model
    - WGS-84 approximates Earth as an oblate spheroid (flattened at the poles).
    - It defines the equatorial radius and flattening factor:
    - Equatorial radius (semi-major axis), a=6,378,137.0a=6,378,137.0 meters
    - Flattening, f=1/298.257223563f=1/298.257223563
- Geodetic Coordinates
    - Latitude (ϕ): angle from equator
    - Longitude (λ): angle from Prime Meridian
- Height (h): above the reference ellipsoid
    - Earth-Centered, Earth-Fixed (ECEF) Cartesian Coordinates
    - 3D coordinates: (X, Y, Z) centered at Earth's center of mass.
- Useful for satellite positioning and orbital calculations.
- Geoid Separation
    - WGS-84 includes models that relate the ellipsoid to the geoid, a more "true" representation of sea level and gravity-based vertical reference.

WGS-84 Model

Rotating Earth Model The WGS-84 oblate, rotating earth (ORE) model [1] is likely the most widely used by the air and missile defense community and is provided in some detail here. The WGS-84 model accounts for rotation effects such as tangential and centripetal acceleration and oblateness effects like geodetic versus geocentric positioning and nonuniform gravity. Moreover, the ORE model includes rotationally induced forces and effects encountered by a missile in flight.

Boord, Warren J.; Hoffman, John B.. Air and Missile Defense Systems Engineering (p. 322). CRC Press. Kindle Edition.

Parameter	Value
Equatorial radius (a)	6,378,137.0 meters
Flattening (f)	1 / 298.257223563
Polar radius (b)	≈ 6,356,752.3 meters
Reference frame origin	Earth's center of mass
Coordinate systems	Geodetic (ϕ, λ, h), ECEF (X, Y, Z)

ERIC & LCIC

ECIC stands for Earth-Centered Inertial Coordinates. It is a non-rotating, geocentric reference frame, meaning it is fixed with respect to the distant stars and not rotating with the Earth. Describes satellite orbits, ballistic missile trajectories, and spacecraft orientation

Origin: Center of the Earth (Earth's center of mass)

Axes:

X-axis: Points toward the vernal equinox (intersection of equator and ecliptic).

Y-axis: 90° east in the equatorial plane.

Z-axis: Aligned with Earth's mean rotation axis (toward the celestial north pole).

Inertial: Not rotating with Earth — suitable for describing orbital mechanics and spacecraft trajectories.

LCIC stands for Launch-Centered Inertial Coordinates. It is a local inertial frame fixed at the launch point or missile origin and does not rotate with the Earth.. LCIC is commonly used in missile guidance, initial trajectory modeling, and booster phase calculations

EGM 96

There are multiple EGM models. EGM96 stands for the Earth Gravitational Model 1996.

•It is a mathematical representation of Earth’s gravity field developed jointly by:

•NASA Goddard Space Flight Center,

•the U.S. National Imagery and Mapping Agency (NIMA, now NGA), and

•Ohio State University.

•Released in 1996, it was the most accurate global geopotential model at that time.

EGM2008 = Earth Gravitational Model 2008, released by the U.S. National Geospatial-Intelligence Agency (NGA).

EGM2020 is to be a new release (still not released as of July 2025) with the same structure as EGM2008, but with improved accuracy by incorporating newer data.

EGM2020 is to be a new release (still not released as of July 2025) with the same structure as EGM2008, but with improved accuracy by incorporating newer data.

ECEF

ECEF stands for Earth-Centered, Earth-Fixed — a 3D Cartesian coordinate system fixed to the rotating Earth. Unlike inertial systems (like ECIC), the ECEF frame rotates with the Earth, making it ideal for representing locations on or near the Earth's surface in real-time. Axes Orientation

X-axis: Passes through the intersection of the Equator and Prime Meridian (0° latitude, 0° longitude).

Y-axis: Lies in the equatorial plane, 90° east of the X-axis (i.e., toward 90°E longitude).

Z-axis: Points toward the North Pole (along Earth's rotation axis).

ECEF coordinates are given as (X, Y, Z) in meters.

To convert geographic coordinates (latitude ϕ, longitude λ, height h) to ECEF:

$$N = \frac{a}{\sqrt{1 - e^2 \sin^2 \phi}}$$

$$X = (N + h) \cos \phi \cos \lambda$$

$$Y = (N + h) \cos \phi \sin \lambda$$

$$Z = \left[(1 - e^2)N + h\right] \sin \phi$$

Where:

- a = Earth’s equatorial radius (from WGS-84: 6,378,137 m)
- e = Earth’s eccentricity (≈ 0.0818)
- N = radius of curvature in the prime vertical

Python code

This converts LLH (lat, long, height) to ECEF:

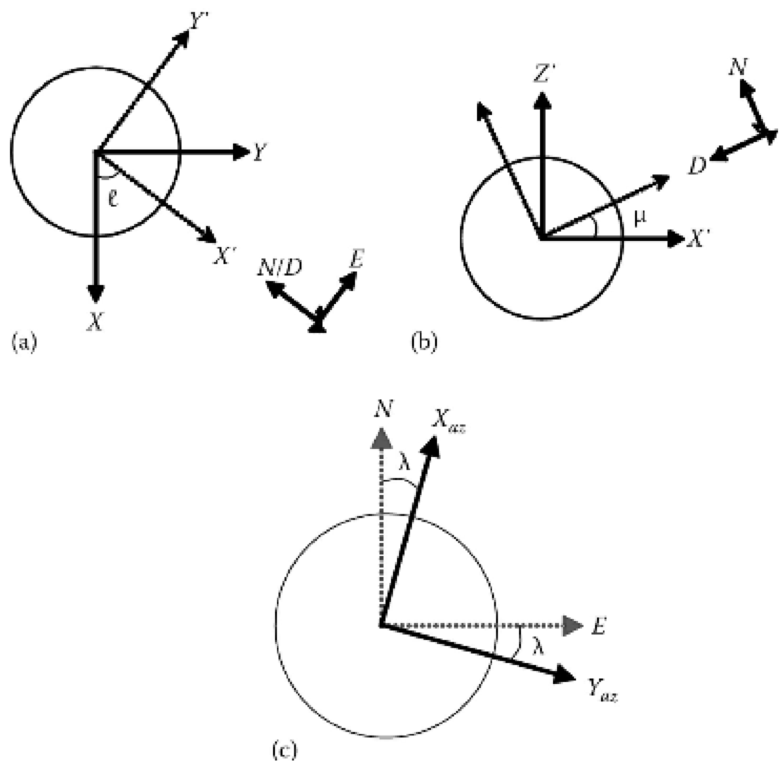
```
import math

def llh_to_ecef(lat_deg, lon_deg, h):
    # WGS-84 ellipsoid constants
    a = 6378137.0              # Semi-major axis (meters)
    f = 1 / 298.257223563      # Flattening
    e2 = f * (2 - f)           # Square of eccentricity

    # Convert degrees to radians
    lat = math.radians(lat_deg)
    lon = math.radians(lon_deg)

    # Radius of curvature in the prime vertical
    N = a / math.sqrt(1 - e2 * (math.sin(lat) ** 2))

    # ECEF coordinates
    X = (N + h) * math.cos(lat) * math.cos(lon)
    Y = (N + h) * math.cos(lat) * math.sin(lon)
    Z = (N * (1 - e2) + h) * math.sin(lat)

    return X, Y, Z

# Example usage
latitude = 37.7749       # San Francisco latitude (degrees)
longitude = -122.4194    # San Francisco longitude
(degrees)
height = 30              # Approximate height in meters

x, y, z = llh_to_ecef(latitude, longitude, height)
print(f"ECEF Coordinates:\nX = {x:.2f} m\nY = {y:.2f} m\nZ =
{z:.2f} m")
```

Transformation

Transformation between any of these systems requires a multiplication by a transformation matrix that relates the orientation of each frame to one another and an addition of the distance between the origins of each reference frame being transformed. For example, to transform from LCIC to ECIC frames, a multiplication by the transformation matrix is required, along with the addition of distance between the two frame origins in Cartesian coordinates.

- Boord, Warren J.; Hoffman, John B.. Air and Missile Defense Systems Engineering (p. 323). CRC Press. Kindle Edition.

Transformations



$$\begin{bmatrix} X' \\ Y' \\ Z' \end{bmatrix} = B_\ell \begin{bmatrix} X \\ Y \\ Z \end{bmatrix}; \quad B_\ell = \begin{bmatrix} \cos(\ell) & \sin(\ell) & 0 \\ -\sin(\ell) & \cos(\ell) & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{9.1}$$

$$\begin{bmatrix} N \\ E \\ D \end{bmatrix} = B_\mu \begin{bmatrix} X' \\ Y' \\ Z' \end{bmatrix}; \quad B_\mu = \begin{bmatrix} -\sin(\mu) & 0 & \cos(\mu) \\ 0 & 1 & 0 \\ -\cos(\mu) & 0 & \sin(\mu) \end{bmatrix} \tag{9.2}$$

$$\begin{bmatrix} X_{az} \\ Y_{az} \\ Z_{az} \end{bmatrix} = B_\lambda \begin{bmatrix} N \\ E \\ D \end{bmatrix}; \quad B_\lambda = \begin{bmatrix} \cos(\lambda) & \sin(\lambda) & 0 \\ -\sin(\lambda) & \cos(\lambda) & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{9.3}$$

ECIC to LCIC transformation matrix rotation sequence: (a) First rotation: Earth top view (Equation 9.1); (b) Second rotation: Earth side view (Equation (Equation 9.2); and (c) Third rotation: Above launch point (Equation 9.3).

-Boord, Warren J.; Hoffman, John B.. Air and Missile Defense Systems Engineering (p. 324). CRC Press

Aeroelasticity

Aeroelasticity is the study of the mutual interaction between the aerodynamic forces acting on a flexible body (like a wing or fuselage) and the body's elastic (structural) and inertial response. It determines how structures deform under airflow and how those deformations, in turn, affect the flow and resulting forces.

Aeroelasticity

There are 4 types of aeroelasticity:

A. Divergence
- Static aeroelastic instability.
- Occurs when aerodynamic loads cause twisting of a lifting surface (like a wing), which increases angle of attack and leads to runaway deformation.
- Critical at high speeds, especially with thin, flexible wings.
- Critical divergence speed: the airspeed at which divergence occurs.

B. Flutter
- Dynamic instability.
- Involves the interaction between bending and torsional modes of vibration.
- At a certain speed (called the flutter speed), oscillations build up rapidly, potentially leading to catastrophic failure.
- Flutter is influenced by:
  - Mass distribution,
  - Stiffness,
  - Aerodynamic damping.
- Important: Flutter must be avoided in all flight envelopes.

C. Buffeting
- Forced vibration caused by unsteady aerodynamic loads.
- Common near shock waves, wake regions, or control surface interactions.
- Typically random and non-resonant, but can lead to fatigue damage.

D. Control Reversal
- A condition where the intended effect of a control surface is reversed due to elastic deformation.
- Especially critical in lightweight, high-speed aircraft.
- Example: Aileron deflection causes wing twist in the opposite direction, reducing or reversing roll response

Key Elements of Aeroelasticity

Force Type	Description
Aerodynamic	Forces from fluid flow (lift, drag, pressure)
Elastic	Internal structural resistance to deformation
Inertial	Resulting from mass acceleration

Ballistic Limit Theory

Ballistic Limit Theory is used to calculate the threshold (limit) at which a projectile will perforate a target. It's crucial in designing protective shielding, especially in space environments, where even millimeter-sized particles can cause catastrophic damage due to their high velocities.

Ballistic Limit (BL): The minimum velocity at which a projectile just perforates a target. Depends on:

- Material properties (density, strength)
- Geometry (thickness, layering)
- Impact angle
- Projectile properties (mass, velocity, shape)

Impact Regimes

- Sub-ballistic: Projectile is stopped by the target.
- Ballistic limit: Projectile barely penetrates.
- Super-ballistic: Projectile perforates and may cause internal damage (secondary ejecta).

Whipple Shield equations

NASA's Whipple Shield equations are empirical formulas used to predict the performance of spacecraft shielding against hypervelocity impacts, such as from micrometeoroids or orbital debris (MMOD). The Whipple shield is one of the most effective and widely used multi-layer protective systems in space applications.

A Whipple shield consists of:

- Bumper (or sacrificial layer) – A thin outer layer that fragments the projectile.
- Standoff distance (gap) – Space between layers allowing the fragments and debris cloud to spread.
- Rear wall (main structure) – The actual component or internal wall to be protected.

When a hypervelocity object hits the bumper:

- It vaporizes or shatters.
- The resulting debris cloud spreads and disperses energy over a larger area.
- The rear wall absorbs the lower-density impact.

NASA developed several forms of empirical equations over time; one widely used is the Christiansen equation, used to determine ballistic limit — the minimum velocity at which the projectile will penetrate the shield.

Whipple Shield Equations (Christiansen Formula)

$$V_{bl} = C \cdot \left(\frac{\rho_t}{\rho_p}\right)^a \cdot d_p^b \cdot t_t^c \cdot S^d$$

Where:

V_{bl} = Ballistic limit velocity (km/s)

ρ_t, ρ_p = Density of target and projectile, respectively

d_p = Projectile diameter

t_t = Rear wall thickness

S = Standoff distance

C,a,b,c,d = Empirical constants (vary by configuration and material)

Riblets

Riblets are small, streamwise-aligned grooves etched or applied to a surface, typically resembling fine parallel ridges. They are engineered to interfere with the structure of turbulent flows in a way that reduces skin friction drag on aerodynamic surfaces.

They were inspired by nature — specifically, the scales of fast-swimming sharks, which have similar microscopic grooves that reduce drag in water.

In turbulent boundary layers:

- Fluid near the wall moves in random, vortical structures called streamwise streaks and quasi-streamwise vortices.
- These vortices lift and mix high-momentum fluid down to the wall, increasing skin friction.

When riblets are applied:
The ridges disrupt the lateral movement of near-wall turbulent structures,
They suppress cross-flow, which reduces momentum transfer toward the wall,
This leads to reduced wall shear stress, hence drag reduction.

Riblets

“Energy conservation and aerodynamic efficiency are the driving forces behind research into methods to reduce turbulent skin friction drag on aircraft fuselages. Fuselage skin friction reductions as small as 10 percent provide the potential for a 250 million dollar per year fuel savings for the commercial airline fleet. One passive drag reduction concept which is relatively simple to implement and retrofit is that of longitudinally grooved surfaces aligned with the stream velocity. These grooves (riblets) have heights and spacings on the order of the turbulent wall streak and burst dimensions. The riblet performance (8 percent net drag reduction thus far), sensitivity to operational/application considerations such as yaw and Reynolds number variation, an alternative fabrication technique, results of extensive parametric experiments for geometrical optimization, and flight test applications are summarized.”

-https://ntrs.nasa.gov/citations/19880005573

Another good paper on this topic is

https://ntrs.nasa.gov/api/citations/20120001342/downloads/20120001342.pdf

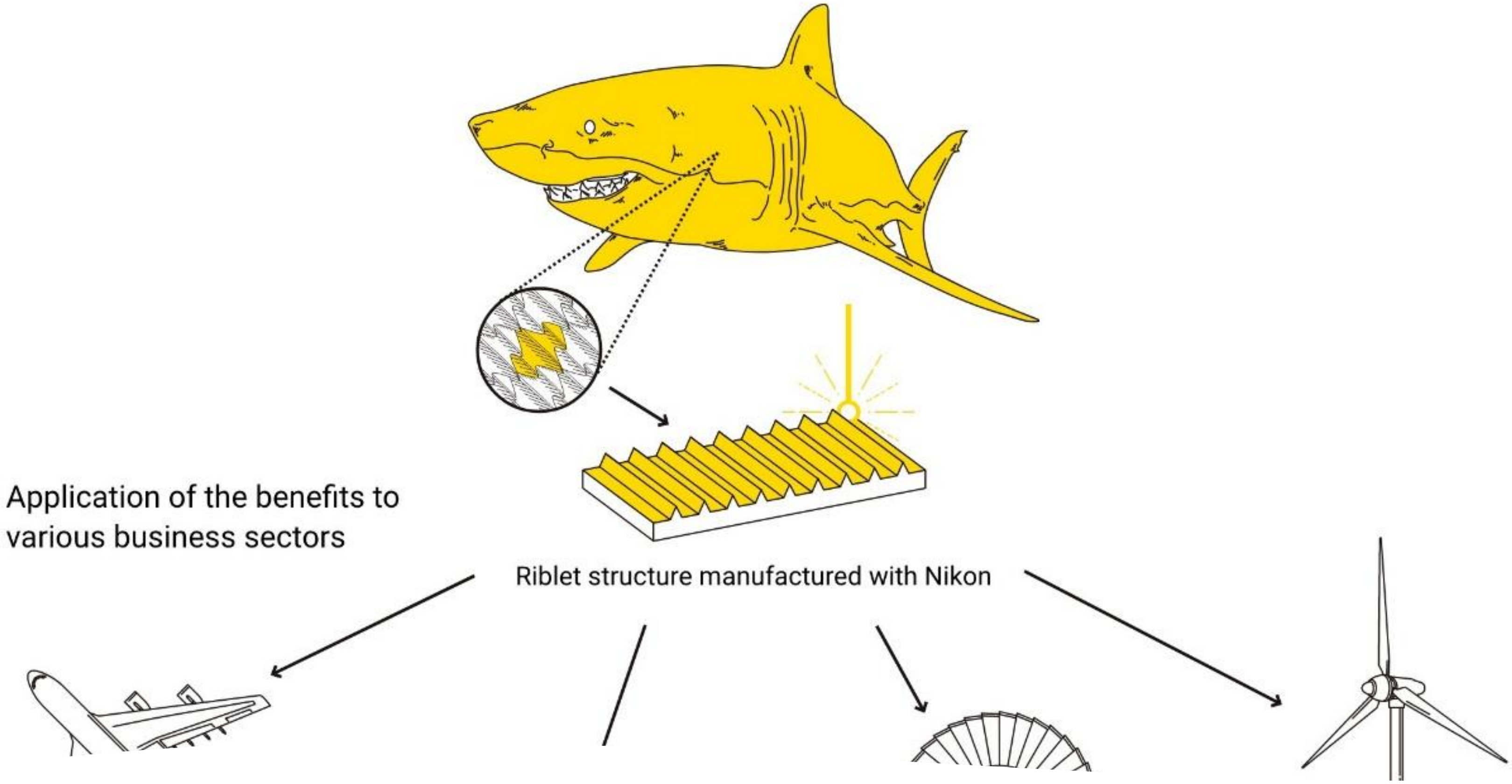
For the physics of riblets

https://encyclopedia.pub/entry/50643

Riblets

“Riblets are drag reduction devices whose effectiveness has been demonstrated by many experiments and reliable numerical simulations. Recent improvements in manufacturing technologies make these devices at a technological level high enough to be applied to the next generation aircraft. Even if riblets effectiveness is ensured, their performance on practical aeronautical configuration are difficult to be computed by numerical analyses due to their dimensions (microns for aviation applications) that would require Direct Numerical Simulation (DNS) still unfeasible for complex shapes at high Reynolds numbers. In the recent years some models for Reynolds Averaged Navier Stokes (RANS) methods or Large Eddy Simulations (LES) have been developed that are able to analyze riblets effect also on full aircraft configurations. In the present paper two models developed by the authors are analyzed and compared also presenting recent developments. The sensitivity to different solver is also discussed. Aircraft configuration analyses are presented showing some interesting and unexpected effect of riblets.”

- https://arc.aiaa.org/doi/10.2514/6.2023-3748



Riblets

Riblets are artificial microstructures that resemble shark skin. By applying riblets to aircraft fuselages, turbine blades of wind turbines, gas turbines, and jet engines, and other applications, it is expected to improve energy efficiency, lower costs, and reduce CO_2 emissions.
Nikon contributes to realizing a sustainable society by utilizing its proprietary laser-processing technology to apply riblets to a wide variety of products.

https://www.nikon.com/company/sustainability/highlight/2306_riblet/

Riblets - Types

Riblet Type	Description
V-groove	Most common; triangular cross-section
Blade-type	Sharp ridges, better for low-speed flows
Sawtooth	More robust, easier to manufacture
Trapezoidal	Stronger, more durable for flight

Riblets

To be effective, riblets must:

Have height and spacing on the order of viscous sublayer thickness (typically 20–50 μm),

Be aligned perfectly with flow direction,

Be durable under erosion, temperature, and contamination.

A non-dimensional design metric is the riblet spacing in wall units:

$$s^{+} = \frac{s u_{\tau}}{\nu} \approx 10 - 15$$

Where:
s: Riblet spacing,
u_τ: Friction velocity,
v: Kinematic viscosity.