AERONAUTICAL ENGINEERING

Engineering Design Project Syllabus

An Engineering Design Project is a comprehensive, hands-on initiative where students apply scientific and engineering principles to develop innovative solutions to real-world problems. The project emphasizes the entire design process, including problem identification, research, conceptualization, modeling, prototyping, testing, and iteration. It develops technical skills, creativity, teamwork, and project management capabilities, enabling students to design and develop functional products or systems that address societal, industrial, or environmental needs.

1. Title: AI-Powered Crack Detection in Composite Aircraft Panels

Objective:

To develop an AI-based system that automatically detects cracks or defects in composite aircraft panels from images or sensor data, improving inspection accuracy and reducing manual effort in aircraft maintenance.

Problem Statement:

Composite materials are widely used in modern aircraft structures due to their high strength-to-weight ratio. However, **detecting tiny cracks or delaminations** in composites is challenging with traditional visual inspection methods and requires skilled labor. Missed cracks can lead to structural failure. An AI-powered crack detection system that uses computer vision and machine learning can **automate the inspection process**, improving safety and reducing inspection costs.

Scope:

Automate inspection of composite panels using AI image recognition for defects**.**

Features:

CNN image classifier, crack/no-crack output, defect location highlighting.

Tools and Technologies:

- **Programming:** Python
- **ML/DL Libraries:** TensorFlow, Keras, or PyTorch
- **Computer Vision:** OpenCV for image pre-processing
- **Dataset:** Public or lab-generated images of composite panels with cracks/defects (or simulate defects using samples)
- **Annotation Tools:**LabelImg or Roboflow for image labeling
- **Hardware:** PC with GPU (optional) for faster training, camera/scanner for real data

Workflow:

Collect and annotate panel images → Preprocess images → Train AI model to detect cracks → Test and validate accuracy.

Expected Outcomes:

- A trained AI model that can identify cracks in composite panel images with good accuracy.
- A simple user interface or script to analyze new images and highlight defect areas.
- Performance metrics: detection accuracy, false positives/negatives, inference time.
- Documentation and demonstration video of the detection pipeline.

Future Enhancements:

- Use thermal or ultrasonic scan data to detect sub-surface cracks.
- Deploy the trained model on drones or robotic crawlers for real-time inspection.
- Develop a mobile app for field technicians to capture and analyze images instantly.
- Expand the model to detect other defects like delamination or corrosion.

SDG Goals: SDG 9, SDG 12, SDG 13, SDG 4

2. Title: Computer Vision-Based Autonomous UAV Landing System

Objective:

To design and prototype a vision-based system that enables an unmanned aerial vehicle (UAV) to detect a designated landing pad and perform safe, precise autonomous landings using computer vision and AI techniques.

Problem Statement:

One of the main challenges in UAV operations is accurate and safe autonomous landing, especially in environments without GPS reliability or under dynamic conditions. Manual landings rely heavily on pilot skill and visibility, which limits UAV operations in remote or hazardous areas. By equipping a UAV with a computer vision system, it can recognize visual markers or landing pads, calculate its position and orientation in real time, and adjust its descent path automatically - paving the way for safer and more reliable UAV missions in delivery, surveillance, or rescue applications.

Scope:

Enable drones to recognize landing spots visually and auto-land with computer vision.

Features:

OpenCV marker detection, integration with autopilot, landing alignment logic.

Tools and Technologies:

- **Programming Languages:** Python or C++
- **Computer Vision Libraries:** OpenCV, TensorFlow/Keras for deep learning if needed
- **Hardware:** UAV with onboard camera (drone with Raspberry Pi or Jetson Nano for edge processing)

- **Simulation Software:** Gazebo, ROS (Robot Operating System) for simulation and control
- **Optional:** AprilTags or ArUco markers for landing pad detection
- **Control Algorithm:** PID controller or basic path-planning logic

Workflow:

Program camera to detect landing marker → Guide UAV to align and land autonomously.

Expected Outcomes:

- A functional **prototype system** capable of detecting a landing pad visually and guiding the UAV to land autonomously.
- Demonstration videos showing marker detection and landing sequence.
- Performance metrics: landing precision, processing time, and system robustness under varying conditions.
- Project report covering hardware, software, test results, and recommendations.

Future Enhancements:

- Replace marker-based detection with deep learning-based object detection (YOLO, SSD) for landing in unprepared sites.
- Add obstacle detection using LiDAR or depth cameras for safer landing in cluttered areas.
- Improve vision robustness under low light, shadows, or occlusion.
- Integrate GPS-denied landing capability for indoor or remote missions.
- Extend system for moving platform landing (e.g., ship or vehicle).

SDG Goals: SDG 9, SDG 11, SDG 13, SDG 4.

3. Title: AI-Driven Fault Detection System for Aircraft Engines Using Sensor Data

Objective:

To **develop an AI-powered system** that can automatically detect early signs of faults or anomalies in aircraft engines by analyzing real-time or recorded sensor data, thus improving safety, reliability, and maintenance planning.

Problem Statement:

Aircraft engines operate under extreme conditions and are equipped with numerous sensors measuring parameters like temperature, vibration, pressure, RPM, and exhaust gas temperatures. **Manual inspection and scheduled maintenance alone may not detect early signs of faults or unusual trends** that lead to critical failures or costly downtime. By using AI and machine learning techniques, we can build a **predictive model** that continuously monitors sensor data, identifies patterns that indicate potential faults, and alerts operators or maintenance teams — enabling **predictive maintenance** rather than reactive repairs.

Scope:
Use AI to detect engine faults early using vibration or thermal sensor signals.

Features:
Python-based ML pipeline, dashboard for monitoring, real-time alert system.

Tools and Technologies:

- **Programming Languages:** Python (preferred for ML libraries)
- **Machine Learning Libraries:** Scikit-learn, TensorFlow, or PyTorch
- **Data Analysis Tools:** Pandas, NumPy, Matplotlib for data preprocessing and visualization
- **Dataset:** Engine performance datasets (publicly available aviation datasets or simulated data)
- **Hardware:** Laptop/PC with GPU (optional for faster training)
- **Optional:** Cloud platforms like Google Colab or AWS for training larger models

Workflow:
Collect or simulate sensor data → Train ML model → Detect and classify faults in real time.

Expected Outcomes:

- A working **AI model** capable of identifying abnormal patterns in aircraft engine sensor data.
- A clear demonstration of how AI/ML can support **predictive maintenance** and enhance safety.
- A prototype dashboard or output interface showing real-time fault status or anomaly alerts.
- A final report covering data preparation, model training, testing, and results.

Future Enhancements:

- Integrate more advanced deep learning models (e.g., LSTM for sequential time-series prediction).
- Deploy on an embedded system or IoT platform for real-time engine monitoring.
- Extend to multi-sensor fusion combining vibration, thermal, and acoustic data.
- Build a predictive system that not only detects faults but estimates remaining useful life (RUL).
- Implement cloud-based remote monitoring for large fleet management.

SDG Goals: SDG9, SDG 12, SDG 13, SDG 4

4. Title: Design of a Subsonic Wind Tunnel Test Section

Objective:

To **design, fabricate, and test** a small-scale **subsonic wind tunnel test section** for **basic aerodynamic experiments**, such as testing aerofoils, wings, or UAV components under controlled airflow conditions.

Problem Statement:

Most educational institutions and small labs lack **affordable, compact, and customizable wind tunnels** for hands-on aerodynamics experiments. Commercial wind tunnels are often expensive, large, and not easily modifiable for student-level research. This project addresses the need for an **in-house, low-cost experimental setup** that allows students to study fundamental aerodynamic concepts practically.

Scope:

Create an affordable, working wind tunnel for small aero experiments in labs.

Features:

Clear test section, fan-driven airflow, basic flow straighteners, slots for sensors.

Tools and Technologies:

- **CAD Software** (e.g., SolidWorks, CATIA) — for designing the test section and duct profiles.
- **CFD Software** (optional, e.g., ANSYS Fluent) — to verify flow uniformity before fabrication.
- **Measurement Instruments** — pitot tube, manometer, or anemometer to measure flow speed and pressure.
- **Basic Electrical Components** — for fan control and power supply (variable speed fan or blower).

Workflow:

Design the tunnel in CAD → Build test section and fan setup → Test flow speed and visualize flow.

Expected Outcomes:

- A fully functional, low-speed, subsonic wind tunnel test section suitable for **student lab experiments**.
- Experimental validation of flow characteristics — velocity profile uniformity, turbulence levels, and flow speed control.
- Demonstration of basic aerodynamic testing such as lift and drag measurement for small models.
- Documentation of the design, build process, and experimental results to serve as a **lab manual** for future batches.

Future Enhancements:

- **Flow Visualization:** Add smoke generator or tufts for flow pattern visualization.
- **Digital Instrumentation:** Integrate digital sensors for real-time velocity and pressure data logging.

- **Variable Speed Control:** Upgrade fan/blower with precise speed regulation to cover a wider Reynolds number range.
- **Closed-Loop Circuit:** Convert to a closed-loop tunnel for improved efficiency and reduced noise.
- **Modular Test Section:** Design interchangeable test sections for different experiments (wings, diffusers, scale UAVs).

SDG Goals: SDG 4, SDG 9 SDG 12

5. Title: Fuel Consumption Forecasting and Optimization Using AI

Objective:

To **develop an AI-based model** that forecasts fuel consumption for an aircraft or UAV under various flight conditions, and suggests **optimal flight profiles** to minimize fuel burn and emissions.

Problem Statement:

Fuel consumption is one of the largest operational costs for airlines and UAV operators. Small improvements in fuel efficiency can save significant costs and reduce **CO_2 emissions**. However, fuel burn depends on multiple factors: aircraft type, weight, weather conditions, altitude, speed, and routing. Traditional estimation methods are often rigid and do not adapt to real-time data. Using **AI models**, students can learn how to handle **big operational data, train predictive models, and optimize mission planning**, which aligns with the aviation industry's push for digital transformation and sustainable operations.

Scope:

Use ML to predict and minimize fuel use under various flight conditions.

Features:

Python regression or ML model, charts, suggested optimal flight profile.

Tools and Technologies:

- **Programming Languages:** Python (preferred) or MATLAB for AI model development.
- **Libraries:** Scikit-learn, TensorFlow, Keras, or PyTorch for machine learning.
- **Data Sources:** Simulated or publicly available aircraft operational datasets (fuel burn, weather, weight).
- **Optional:** Jupyter Notebook for interactive modeling and visualization.

Workflow :

Collect or simulate flight and fuel data → Preprocess and clean data → Train AI regression or time-series model → Predict fuel burn → Recommend optimized flight profiles.

Expected Outcomes:

- A working AI model that predicts fuel consumption based on input conditions (altitude, weight, route, wind).
- Forecast plots comparing actual vs. predicted fuel usage.
- Analysis of how different flight parameters affect fuel burn.
- Example optimized flight plan or speed-altitude profile that minimizes fuel consumption.

Future Enhancements:

- Integrate real-time weather data for live fuel forecasting.
- Develop a simple decision support tool for pilots or UAV operators.
- Combine AI with **route optimization algorithms** for flight path suggestions.
- Scale up to fleet-level fuel prediction and CO_2 impact estimates.
- Deploy as a cloud-based dashboard for airline or UAV operators.

SDG Goals: SDG 9, SDG 13, SDG 12, SDG 4

6. Title: Fatigue Life Analysis of Aircraft Wing Spar under Cyclic Loading

Objective:

To **analyze and estimate the fatigue life** of an aircraft wing spar subjected to cyclic loading conditions, using theoretical calculations and numerical simulation tools to predict when fatigue cracks may initiate and propagate, ensuring safer and more efficient structural designs.

Problem Statement:

Aircraft wing spars are **critical load-bearing structural members** that experience repeated cyclic loading during every flight due to lift, turbulence, and maneuvers. Over time, these fluctuating stresses can lead to **fatigue failure**, which is a major cause of unexpected structural damage. This project aims to help students understand how to **predict fatigue life**, apply S-N curves, and use computational tools to validate theoretical estimations — an essential skill for preventing catastrophic failure and improving aircraft safety.

Scope:

Analyze fatigue strength, stress cycles, and endurance for realistic wing spar loading.

Features:

CAD spar model, cyclic stress simulation, S-N diagram, fatigue life prediction.

Tools and Technologies:

- **CAD Software** (e.g., CATIA, SolidWorks) — for modeling the wing spar geometry.
- **FEA Software** (e.g., ANSYS Workbench, Abaqus) — for stress analysis under cyclic loading.
- **MATLAB or Python** — to plot S-N curves and perform fatigue life calculations.
- **Material Property Data** — fatigue strength, endurance limit, modifying factors for the selected spar material (e.g., aluminum alloy).
- **Basic Lab Setup** *(optional)* — UTM (Universal Testing Machine) if physical testing of a sample coupon is possible.

Workflow:

Model wing spar → Apply cyclic loads in FEA → Generate S-N curve and predict life.

Expected Outcomes:

- A clear estimate of the expected fatigue life of the wing spar under defined operational conditions.
- FEA stress distribution plots showing high-stress regions.

- An understanding of how design parameters and material properties affect fatigue performance.
- A detailed project report with theoretical and simulation data, plots, and conclusions.

Future Enhancements:

- **Variable Amplitude Loading:** Extend the study to realistic flight load spectrums with cumulative damage models (Miner's Rule).
- **Crack Propagation Study:** Use fracture mechanics to simulate crack growth and remaining life.
- **Physical Validation:** Conduct small-scale physical fatigue testing of coupons to compare with theoretical predictions.
- **Material Alternatives:** Evaluate advanced composite spars for higher fatigue resistance.
- **AI Integration:** Implement machine learning to predict fatigue life from large datasets of stress histories.

SDG Goals: SDG 9, SDG 12, SDG 4

7. Title: CFD Analysis of Air Intake Distortion in Jet Engines

Objective:

To **simulate and analyze airflow distortion** inside a jet engine air intake using Computational Fluid Dynamics (CFD) to understand its effects on engine performance and stability.

Problem Statement:

Modern jet engines rely on a **smooth, uniform airflow** into the compressor for optimal performance and to avoid compressor stalls or surges. However, complex aircraft geometries, high angles of attack, crosswinds, or distorted intakes (e.g., S-ducts in stealth aircraft) can cause **non-uniform flow**, known as **inlet distortion**. This project helps students understand how **inlet flow distortion impacts pressure recovery, swirl, and turbulence**, which are critical for safe and efficient engine operation.

Tools and Technologies:

- **CAD Software:** SolidWorks, CATIA, or Fusion 360 (for modeling the intake geometry).
- **CFD Software:** ANSYS Fluent, OpenFOAM, or STAR-CCM+ for flow simulations.
- **Meshing Tools:** ANSYS Meshing, ICEM CFD, or equivalent.
- **Post-processing Tools:** CFD-Post, ParaView, or Tecplot for results visualization.

Workflow:

Model the air intake → Generate mesh → Define boundary conditions → Run CFD simulation → Analyze flow distortion and pressure fields.

Expected Outcomes:

- Detailed velocity and pressure distribution plots inside the air intake.
- Quantification of total pressure loss, swirl angles, and distortion index.
- Identification of high swirl or low-pressure regions that could affect engine stability.
- Recommendations to minimize intake distortion through design improvements.

Future Enhancements:

- Analyze variable flight conditions (different angles of attack or crosswind scenarios).
- Compare multiple intake shapes (straight duct, S-duct, offset intake).
- Validate CFD results with wind tunnel or scaled experimental data.
- Extend study to investigate the effect on fan/compressor blade performance.

SDG Goals: SDG 9, SDG 13, SDG 4

8. Title: Design of Multi-Cellular Wing Structures for Weight Reduction

Objective:

To **design and analyze a multi-cellular wing structure** that minimizes structural weight while maintaining strength and stiffness, enabling more efficient and sustainable aircraft designs.

Problem Statement:

Modern aircraft wings must balance **light weight with high strength and stiffness** to withstand aerodynamic loads during flight. Traditional single-cell wing boxes are structurally simpler but may not offer optimal weight efficiency for larger wings. **Multi-cellular wing structures** divide the internal wing into multiple box sections (cells), distributing loads more effectively and enabling the use of **thinner skins and spars**, which reduces weight without compromising strength. This project helps students understand **advanced structural concepts** like thin-walled multi-cell analysis, shear flow, and torsional rigidity.

Scope:

Design a lightweight, structurally sound multi-cell wing structure for modern aircraft.

Features:

CAD wing box, stress analysis, weight vs. strength comparison.

Tools and Technologies:

- **CAD Software:** CATIA, SolidWorks, or Fusion 360 for 3D wing box modeling.
- **FEA Software:** ANSYS Mechanical, Abaqus, or NASTRAN for stress, deflection, and buckling analysis.
- **Analytical Tools:** MATLAB or Python to solve shear flow and twist equations manually for validation.
- **Reference Standards:** Aircraft structural design handbooks or textbooks for design constraints and material selection.

Workflow:

Design the wing structure → Divide into multiple cells → Analyze stresses, shear flows, and torsional stiffness → Optimize for weight and strength.

Expected Outcomes:

- A detailed **CAD model** of a multi-cellular wing box showing spars, ribs, and skins.
- Analytical calculations of shear flow and torsion for comparison with FEA results.
- Weight comparison between single-cell and multi-cell configurations.
- Recommendations on material choice and cell layout for maximum weight savings.

Future Enhancements:

- Extend analysis to include composite materials for further weight reduction.
- Add aerodynamic loading (lift distribution) for realistic stress conditions.
- Prototype a scale wing box section for static or vibration tests.
- Explore smart structures with embedded sensors for health monitoring.

SDG Goals: SDG 9, SDG 13, SDG 12, SDG 4

9. Title: Investigation of Boundary Layer Control Using Passive Devices

Objective:
To **study and demonstrate** how passive boundary layer control devices (like vortex generators, turbulators, or riblets) can delay flow separation and improve aerodynamic performance on airfoils or wings.

Problem Statement:
In aircraft aerodynamics, **boundary layer separation** causes increased drag and reduced lift, negatively affecting performance and fuel efficiency. Controlling the boundary layer is crucial, especially at high angles of attack or low Reynolds numbers (e.g., UAVs, small aircraft). Passive devices — simple, fixed structures that energize the boundary layer — offer an effective, low-cost solution without the complexity of active flow control systems. This project helps students understand practical boundary layer behavior and test how small design interventions can make big performance differences.

Scope:
Test how simple add-ons delay separation and improve lift at higher angles of attack.

Features:
CFD with/without control devices, flow visualization, performance comparison.

Tools and Technologies:

- **CAD Software:** SolidWorks, CATIA, or Fusion 360 to model the aerofoil with and without control devices.
- **CFD Software:** ANSYS Fluent, OpenFOAM, or STAR-CCM+ for flow simulation and visualization.
- **Wind Tunnel (optional):** Small low-speed wind tunnel for experimental validation.
- **Measurement Tools:** Smoke generator or tufts for flow visualization, pitot tube or pressure taps for lift/drag measurement.

Workflow:

Design aerofoil → Add passive control devices → Run CFD simulations or wind tunnel tests → Compare flow separation, lift, and drag.

Expected Outcomes:

- Comparative **CFD or experimental results** showing flow behavior with and without passive devices.
- Visualization of boundary layer reattachment, delayed separation, or vortex formation.
- Quantified lift and drag coefficient changes demonstrating improved aerodynamic performance.
- Recommendations on optimal placement and shape of passive devices.

Future Enhancements:

- Test different device shapes (e.g., micro vortex generators, riblets, surface roughness strips).
- Study combined passive and active flow control approaches.
- Scale up findings for real aircraft wings or UAV applications.
- Investigate performance at multiple angles of attack and Reynolds numbers.

SDG Goals: SDG 9, SDG 13, SDG 12, SDG 4

10. Title: Static and Dynamic Load Analysis of an Aircraft Wing Box

Objective:
To **model, analyze, and compare** the structural behavior of an aircraft wing box under **static and dynamic loads**, ensuring the wing's strength, stability, and vibration safety during flight.

Problem Statement:
The wing box is the **primary load-bearing structure** of an aircraft wing, designed to carry aerodynamic lift forces, fuel loads, and inertia loads. It must withstand **static loads** (steady aerodynamic lift, fuel weight) and **dynamic loads** (gusts, turbulence, flutter, and vibration). Failure to accurately predict these loads can cause catastrophic structural failures. This project helps students understand **how to evaluate stresses, deflections, natural frequencies, and mode shapes**, using theoretical calculations and finite element methods.

Scope:
Evaluate how wing boxes handle real flight loads to ensure structural safety.

Features:
FEA stress plots, deflection contours, natural frequency check (dynamic).

Tools and Technologies:

- **CAD Software:** CATIA, SolidWorks, or Fusion 360 for wing box geometry modeling.
- **FEA Software:** ANSYS Mechanical, Abaqus, or NASTRAN for static and modal/dynamic analysis.
- **Analytical Tools:** MATLAB or Python for hand calculations and mode shape validation.
- **Material Properties:** Aerospace-grade aluminum or composite material data.

Workflow
Design wing box → Define static lift and fuel loads → Apply dynamic load cases → Run FEA → Analyze stress, deflection, and vibration response.

Expected Outcomes:

- Detailed **stress and deflection plots** under maximum static loading conditions.
- Modal analysis results: natural frequencies and mode shapes.
- Identification of critical areas prone to high stress or deflection.
- Comparison of static vs dynamic behavior and its implications for design safety.

Future Enhancements:

- Perform **transient dynamic analysis** to simulate gust load response.
- Extend study to include aeroelastic effects like **flutter**.
- Analyze wing box made from advanced composite materials.
- Build a scaled physical model to validate modal frequencies experimentally.
- Integrate optimization algorithms to reduce weight while maintaining safety margins.

SDG Goals: SDG 9, SDG 12, SDG 13

11. Title: Optimization of Winglet Designs for Drag Reduction

Objective:
To **design, analyze, and optimize different winglet shapes** to reduce induced drag on an aircraft wing, thereby improving fuel efficiency and overall aerodynamic performance.

Problem Statement:
Winglets are upward or outward extensions at the wingtips that help control wingtip vortices, which are a primary source of **induced drag**. Reducing induced drag directly increases lift-to-drag ratio, leading to significant **fuel savings** over an aircraft's life cycle. The shape, cant angle, sweep, and curvature of a winglet greatly affect its aerodynamic performance. The project aims to help students understand how to **design and test different winglet configurations**, compare their effects on drag reduction, and select the most efficient design using computational tools.

Scope:
Reduce drag and fuel burn by optimizing winglet shape for lift-induced drag control.

Features:
CFD drag plots, lift coefficient data, best shape recommendations.

Tools and Technologies:
- **CAD Software:** SolidWorks, CATIA, or Fusion 360 for wing and winglet modeling.
- **CFD Software:** ANSYS Fluent, OpenFOAM, or STAR-CCM+ for aerodynamic simulations.
- **Optimization Tools:** Parametric design features in CAD, or design of experiments (DOE) methods for comparing different shapes.
- **Post-Processing:** CFD-Post, ParaView, or Tecplot for lift and drag coefficient analysis.

Workflow:
Model baseline wing → Design multiple winglet shapes → Simulate airflow → Compare drag coefficients → Identify optimum winglet design.

Expected Outcomes:
- 3D CAD models of multiple winglet configurations (e.g., blended, raked, split scimitar).
- CFD results showing flow visualization, lift and drag coefficients, and wingtip vortex behavior.
- Quantitative comparison of drag reduction percentages for each design.
- Recommendation of the best-performing winglet shape for maximum drag reduction.

Future Enhancements:
- Validate CFD results with **wind tunnel testing** of scaled wing/winglet models.
- Apply shape optimization algorithms to further refine the winglet geometry.
- Study the impact of winglets at different flight conditions (take-off, cruise, landing).
- Integrate active or morphing winglet concepts for adaptive performance in real-time.
- Extend to fuel burn calculations to estimate operational savings for an entire flight profile.

SDG Goals: SDG 4, SDG 9, SDG 12, SDG 13

12. Title: Experimental Study of Vortex Generators on an Aerofoil

Objective:
To **investigate how vortex generators (VGs)** influence boundary layer behavior on an aerofoil, with the aim to **delay flow separation**, enhance lift, and reduce drag through practical wind tunnel experiments.

Problem Statement:
At higher angles of attack, an aerofoil experiences **boundary layer separation**, which leads to a sudden drop in lift and a sharp increase in drag — a critical factor in stall behavior. **Vortex generators** are small aerodynamic devices mounted on the surface that create controlled vortices, energizing the boundary layer to keep it attached longer. This project helps students understand **how passive flow control devices work in practice** and quantify their effects through real data.

Scope:
Test how vortex generators improve boundary layer behavior and lift.

Features:
Physical models, wind tunnel tests, lift curve with/without VGs.

Tools and Technologies:

- **Aerofoil Model:** NACA aerofoil (e.g., NACA 2412) with and without vortex generators attached.
- **Wind Tunnel:** Small low-speed wind tunnel for experimental tests.
- **Vortex Generators:** Small triangular or rectangular tabs made from plastic or metal.
- **Measurement Tools:**
    - Smoke generator or tufts for flow visualization
    - Pitot-static tube or multi-hole probe for velocity measurements
    - Lift and drag balance for force measurement
    - Digital angle of attack indicator

Workflow:
Test clean aerofoil in wind tunnel → Attach vortex generators → Repeat tests → Compare lift, drag, and flow separation data.

Expected Outcomes:

- Experimental lift and drag curves for the aerofoil with and without vortex generators.
- Flow visualization images or videos showing delayed separation and vortex formation.
- Quantitative evidence of increased maximum lift coefficient (Clmax) and possible drag penalty at certain angles.
- Analysis report with plots, discussions, and recommendations for VG placement.

Future Enhancements:

- Test various VG sizes, shapes, and spacing to find the optimal configuration.
- Combine vortex generators with other passive devices (e.g., Gurney flaps).
- Validate results with CFD simulations to correlate experimental and numerical data.
- Explore the use of **active vortex generators** for adaptive flow control in future work.
- Extend study to multi-element aerofoils or full wing models.

SDG Goals: SDG 4, SDG 9, SDG 12, SDG 13

13. Title: Aerodynamic Shape Optimization of Low Reynolds Number Aerofoils

Objective:
To **design, analyze, and optimize aerofoil shapes** specifically for low Reynolds number conditions (e.g., UAVs, gliders, or small wind turbines) to achieve **maximum lift-to-drag ratio** for improved aerodynamic efficiency.

Problem Statement:
Aerofoils operating at low Reynolds numbers (typically below **500,000**) face unique challenges: **laminar separation bubbles**, early flow separation, and lower lift-to-drag

performance compared to conventional aerofoils used on full-scale aircraft. UAVs, micro-air vehicles, and small-scale wind turbines all require aerofoils that perform well in these conditions. This project lets students understand **how shape parameters (camber, thickness, leading edge radius, trailing edge angle)** affect performance — and how **numerical tools and shape optimization** can refine designs for specific missions.

Scope:
Develop airfoils for small UAVs, focusing on slow-speed aerodynamic efficiency.

Features:
Shape variations, CFD output, lift/drag plots for different Re.

Tools and Technologies:
- **CAD Software:** XFLR5, SolidWorks, or other aerofoil design tools.
- **CFD Software:** ANSYS Fluent, OpenFOAM, or XFOIL for analysis of lift, drag, and pressure distribution.
- **Optimization Tools:** MATLAB or Python scripts for parametric studies, or built-in optimizers in XFOIL/XFLR5.
- **Validation Data:** NACA or Eppler low-Re aerofoil databases for baseline comparison.

Workflow:
Define baseline aerofoil → Run CFD/XFOIL analysis → Modify shape parameters → Optimize for lift-to-drag ratio → Validate final design.

Expected Outcomes:
- CAD or XFOIL-generated aerofoil shapes showing baseline and optimized designs.
- Lift and drag polars for different Reynolds numbers.
- Pressure coefficient plots highlighting separation bubble control or delayed stall.
- Recommendation of an aerofoil suitable for a UAV, small drone, or micro wind turbine.

Future Enhancements:
- Prototype the optimized aerofoil for **wind tunnel testing** and experimental validation.
- Integrate multi-point optimization to ensure robust performance across various angles of attack and flight speeds.
- Study the effect of **surface roughness** and manufacturing tolerances on real-world performance.
- Apply AI/ML algorithms for advanced shape parameter optimization.
- Extend to 3D wing planforms (e.g., optimized winglets for the same Reynolds regime).

SDG Goals: SDG 9, SDG 12, SDG 13

14. Title: Design and Analysis of a Blended Body Wing

Objective:
To **design and analyze a blended wing-body (BWB) concept**, studying its aerodynamic performance and structural behavior to understand how this advanced configuration can improve lift-to-drag ratio and fuel efficiency.

Problem Statement:

The traditional tube-and-wing aircraft configuration has limitations in aerodynamic efficiency and payload integration. The **blended wing-body** design merges the fuselage and wing into a single lifting surface, offering significant improvements in lift distribution, reduced wetted area, and lower drag. However, it introduces **complex aerodynamic interactions** and structural challenges due to its non-conventional shape. This project helps students learn the fundamentals of **BWB design**, aerodynamic analysis using CFD, and preliminary structural evaluation.

Scope:

Test how integrated wing-fuselage designs improve lift-to-drag ratios.

Features:

3D BWB CAD, CFD Mach plots, stress/weight estimates.

Tools and Technologies:

- **CAD Software:** CATIA, SolidWorks, or Fusion 360 to model the BWB geometry.
- **CFD Software:** ANSYS Fluent, OpenFOAM, or STAR-CCM+ for aerodynamic flow simulation.
- **FEA Software:** ANSYS Mechanical or Abaqus for basic structural load checks.
- **Optional Tools:** MATLAB or Python for lift distribution calculations or parametric studies.

Workflow:

Create BWB CAD model → Run CFD to analyze lift, drag, and flow behavior → Perform basic structural stress and deflection analysis → Interpret results.

Expected Outcomes:

- A detailed 3D model of a conceptual blended wing-body configuration.
- Aerodynamic performance results: lift coefficient (Cl), drag coefficient (Cd), and lift-to-drag ratio (L/D).
- Flow visualization plots showing pressure distribution and vortex patterns.
- Basic structural analysis identifying stress concentrations and bending behavior.
- Design recommendations for improving aerodynamic and structural efficiency.

Future Enhancements:

- Study various BWB planforms with different aspect ratios and sweep angles.
- Optimize internal structural layout for fuel storage or payload integration.
- Incorporate aeroelastic analysis to study wing bending and flutter.
- Validate simulation results with wind tunnel testing of scaled BWB models.
- Investigate hybrid or distributed propulsion integration for BWB aircraft.

SDG Goals: SDG4, SDG 9, SDG 12, SDG 13

15. Title: Design of a Toroidal Propeller for UAV

Objective:

To **design and analyze a toroidal (closed-loop) propeller** for an Unmanned Aerial Vehicle (UAV), focusing on **noise reduction**, aerodynamic efficiency, and improved thrust performance compared to conventional open-blade propellers.

Problem Statement:

Recent experimental research and prototypes (e.g., MIT Lincoln Lab's toroidal propeller concept) show that toroidal propellers can **significantly reduce noise** and vortices at the blade tips, which is critical for urban air mobility and small UAVs operating in populated areas. Traditional open-blade propellers create strong tip vortices that contribute to **aerodynamic noise and energy loss**. Designing an efficient toroidal propeller for UAVs could help address urban noise restrictions, improve stealth in defense applications, and boost overall aerodynamic performance.

Scope:

Test a new propeller type to reduce noise and tip vortices in drones.

Features:

CAD blade ring, CFD thrust/noise results, side-by-side comparison.

Tools and Technologies:

- **CAD Software:** SolidWorks, CATIA, or Fusion 360 for modeling the toroidal blade geometry.
- **CFD Software:** ANSYS Fluent, OpenFOAM, or STAR-CCM+ to simulate airflow, vortex behavior, and thrust.
- **Optimization Tools:** MATLAB, Python, or parametric design tools for adjusting blade curvature, pitch, and ring geometry.
- **3D Printing (Optional):** For prototyping and lab-scale thrust or noise testing.

Workflow :

Design toroidal propeller → Simulate aerodynamic performance in CFD → Compare with standard open propeller → Optimize for thrust and noise.

Expected Outcomes:

- 3D CAD model of a toroidal propeller tailored for a small UAV application.
- CFD results showing thrust, power consumption, tip vortex behavior, and predicted noise levels.
- Performance comparison charts between toroidal and conventional propellers.
- Design report recommending optimal geometry for efficiency and noise reduction.

Future Enhancements:

- Prototype and **experimentally test** the propeller using a small thrust stand.
- Conduct noise measurements in an anechoic chamber or quiet outdoor setup.
- Explore hybrid blade-ring shapes to balance weight, manufacturability, and performance.
- Integrate the design with electric propulsion for urban air mobility (UAM) drones.
- Develop AI/ML models to optimize propeller design parameters automatically.

SDG Goals: SDG4, SDG 9, SDG 11, SDG 13

16. Title: CFD Analysis of a Convergent-Divergent (CD) Nozzle

Objective:

To **simulate and analyze the flow characteristics** through a **convergent-divergent nozzle**, focusing on **supersonic flow development**, shock formation, and the impact of nozzle geometry on thrust and flow expansion.

Problem Statement:

The **convergent-divergent (de Laval) nozzle** is a critical component in rocket engines, supersonic wind tunnels, and gas turbines. Its design enables subsonic flow to accelerate to supersonic speeds as it passes through the throat and divergent section. Predicting how changes in nozzle area ratio, throat design, and back pressure affect flow behavior is crucial for achieving desired thrust and preventing unwanted shocks or flow separation. This project helps students understand **compressible flow, shock waves, expansion fans, and flow choking**, and apply **CFD tools** to visualize these phenomena.

Scope:

Study how nozzle expansion affects Mach number, shocks, and thrust.

Features:

Mach contour plots, shock visual, thrust output.

Tools and Technologies:

- **CAD Software:** SolidWorks, CATIA, or Fusion 360 to design the CD nozzle geometry.
- **CFD Software:** ANSYS Fluent, OpenFOAM, or STAR-CCM+ for compressible flow simulation.
- **Post-Processing:** ParaView, Tecplot, or CFD-Post for Mach number contours, pressure distribution, and shock pattern visualization.
- **Optional:** MATLAB/Python for analytical comparison using isentropic flow relations.

Workflow:

Design CD nozzle → Define boundary conditions and mesh → Run compressible flow simulation → Analyze Mach number, pressure, and shock structures.

Expected Outcomes:

- Clear Mach number distribution showing subsonic, sonic (throat), and supersonic regions.
- Pressure and temperature contours validating isentropic flow assumptions.
- Identification of normal shocks or expansion fans inside or outside the nozzle.
- Comparison of calculated thrust with theoretical values.
- Report linking area ratio and back pressure to flow behavior.

Future Enhancements:

- Study effects of varying nozzle expansion ratios and throat shapes.
- Simulate off-design conditions to visualize shock diamonds or flow separation.
- Extend the analysis to multi-phase flow (e.g., liquid rocket propellant injection).
- Validate CFD results with lab-scale supersonic wind tunnel tests or schlieren photography.

- Implement AI-based optimization to find the best nozzle shape for specific mission profiles.

SDG Goals: SDG4, SDG 9, SDG 13

17. Title: Static Analysis of a Tapered Wing

Objective:

To **analyze the static structural behavior** of a tapered aircraft wing under lift and weight loads — focusing on how taper ratio affects **bending stress, shear stress, deflection, and structural efficiency**.

Problem Statement:

Most modern aircraft wings are **tapered** to improve aerodynamic performance and structural weight efficiency. Tapering changes the lift distribution along the span, reduces tip chord, and helps minimize bending moments at the root. However, the changing cross-section complicates stress analysis and requires students to understand non-uniform load distribution and spanwise variations in bending and shear. This project helps students apply beam theory and finite element analysis to evaluate the static performance of a tapered wing.

Scope:

Evaluate how tapering affects structural strength and wing deflection.

Features:

Spanwise stress plot, tip deflection, comparison with untapered wing.

Tools and Technologies:

- **CAD Software:** SolidWorks, CATIA, or Fusion 360 to create a 3D tapered wing model.
- **FEA Software:** ANSYS Mechanical, Abaqus, or NASTRAN for static structural analysis.
- **Optional:** MATLAB or Python for analytical calculation of lift distribution and bending moment diagram.

Workflow:

Model tapered wing → Apply distributed lift and fuel/weight loads → Perform FEA → Analyze bending stress, shear stress, and tip deflection.

Expected Outcomes:

- A detailed 3D CAD model of the tapered wing planform with defined airfoil cross-sections.
- Stress distribution plots (bending and shear) along the span and chord.
- Deflection plot showing maximum tip displacement under static lift loading.
- Comparison of stress and deflection with analytical estimates (if done).
- Insight into the benefits of tapering versus a constant chord wing.

Future Enhancements:

- Compare different taper ratios to find optimal balance of lift distribution and structural weight.
- Extend to analyze swept tapered wings for more realistic designs.
- Include wing structural elements like spars and ribs for more detailed internal load paths.
- Validate results with a simple scaled experiment using cantilever beam tests.
- Integrate aeroelastic considerations such as lift-curve slope and twist due to bending.

SDG Goals: SDG4, SDG 9, SDG 12, SDG 13

18. Title: Design and Testing of a Parachute Recovery System for Drones

Objective:

To **design, fabricate, and test** a lightweight and reliable **parachute recovery system** for drones to safely recover the UAV during emergencies such as power failure, loss of control, or critical system faults.

Problem Statement:

Small and medium UAVs are increasingly used for delivery, surveillance, mapping, and research missions. In case of engine failure or control loss, these drones risk uncontrolled crashes, causing **damage to property, risk to people, and loss of expensive equipment**. A parachute recovery system offers a simple, passive solution to reduce descent rate and impact energy, improving **flight safety and regulatory compliance**. This project helps students learn about **aerodynamic drag devices, deployment mechanisms, and practical testing**.

Scope:

Develop a simple recovery system to reduce crash damage during UAV failures.

Features:

Canopy CAD, simple deployment system, test results (descent speed).

Tools and Technologies:

- **CAD Software:** SolidWorks, Fusion 360 for designing the parachute canister and deployment system.
- **Materials:** Lightweight ripstop nylon or polyester fabric, spring-loaded or pyrotechnic deployment mechanism (model-scale), lightweight container.
- **Testing Tools:** Small UAV or drone platform, drop test setup, altimeter, stopwatch, camera for recording descent.
- **Simulation (Optional):** MATLAB or Python to estimate descent rate and terminal velocity.

Workflow:

Design parachute canopy and deployment system → Fabricate using lightweight materials → Integrate with drone → Perform drop tests → Measure descent rate and impact force.

Expected Outcomes:

- Parachute canopy design with suitable area to achieve desired descent rate for given UAV weight.

- Prototype deployment system attached to the drone.
- Experimental data showing descent rate, deployment reliability, and impact velocity.
- Final design recommendations for real-world drone recovery systems.

Future Enhancements:

- Test with automatic deployment triggered by flight controller failsafe or GPS-based altitude threshold.
- Use **AI-based fault detection** to trigger deployment autonomously.
- Optimize canopy shape (e.g., cross canopy vs. ribbon) for stability during descent.
- Perform multiple drop tests under varying wind conditions.
- Develop modular recovery units adaptable to different UAV sizes.

SDG Goals: SDG4, SDG 9, SDG 11, SDG 13

19. Title: Static Load Testing of an Aircraft Wing Rib

Objective:

To design a simple aircraft wing rib (e.g., made from plywood or composite sheet), apply static loads, and measure deflection to verify the rib's strength and stiffness.

Problem Statement:

Wing ribs are essential structural members that shape the wing's airfoil and transfer aerodynamic loads to the spars and skin. Testing a rib under bending and shear helps students understand basic aircraft structural principles and how load paths work in a real wing box.

Scope:

Test how a basic wing rib bears bending loads to validate design assumptions.

Features:

Physical rib, simple test rig, deflection vs. theory report.

Tools and Technologies:

- Basic CAD (SolidWorks/AutoCAD) for 2D rib design
- Materials: plywood, balsa, or foam board
- Simple load setup: weights or water bottles
- Measurement tools: scale, ruler, dial gauge

Workflow:

Design → Fabricate rib → Apply known static loads → Measure deflection → Compare with beam theory.

Expected Outcomes:

Deflection and stress results that verify if the rib meets expected design limits.

Future Enhancements:

Extend to multiple ribs, join with spars, or test full wing box.

SDG Goals: SDG 4, SDG 9

20. Title: Noise Measurement of Small UAV Propellers

Objective:
To measure and compare the noise produced by different small UAV propeller sizes or shapes at various RPMs.

Scope:
Quantify how blade size/pitch affects drone propeller noise output.

Features:
Noise meter readings, RPM vs. dB plots, comparison chart.

Problem Statement:
Noise pollution is a growing challenge for drones operating in urban areas. Simple acoustic tests teach students how blade shape, speed, and tip design affect propeller noise.

Tools and Technologies:
- Small electric drone motor with ESC and propellers (different sizes/pitches)
- Sound level meter or smartphone sound app
- Tachometer or RPM meter
- Basic stand to secure motor

Workflow:
Mount motor → Run propeller at different RPMs → Measure sound level at set distance → Compare results.

Expected Outcomes:
Noise levels for different propeller designs, with a simple chart and analysis.

Future Enhancements:
Add simple ducting, test with toroidal propeller, or study effect of blade tip shape.

SDG Goals: SDG 11, SDG 13