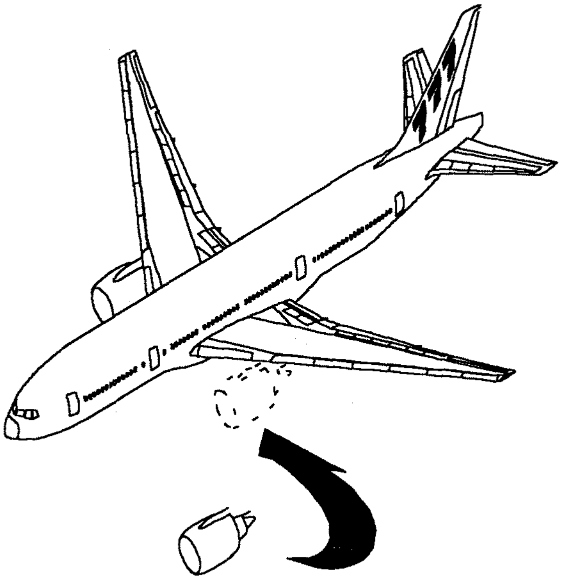
THE BOEING 777 ENGINE/AIRCRAFT INTEGRATION AERODYNAMIC DESIGN PROCESS

ICAS-94-6.4.4

Dennis L. Berry
Boeing Commercial Airplane Group
Seattle, Washington USA



Propulsion System Integration

Abstract

The integration of the propulsion system on the Boeing 777 aircraft presented a significant challenge due to the size and complexity of the powerplants involved. To achieve success required a multi-disciplined approach and a clear understanding of design and customer requirements. Traditional rules, emphasizing aircraft design mainly for best fuel economy, did not apply since in today's environment both fuel costs and cost of ownership are major portions of airline operating expenses. To achieve success required a more open, working-together relationship between the engine companies, airframe manufacturers, vendors and airlines to obtain a design which met customer requirements with the best balance between performance, weight, noise, maintainability, producibility, and cost.

The information presented outlines the processes utilized, examines how non-aerodynamic influences affected the design, and emphasizes those areas where teamwork, process improvements, manufacturability considerations, and response to customer requirements played an important role in the aerodynamic development and validation of the propulsion system installation, with a glimpse at what the future holds.

Introduction

Propulsion system integration is one of the more interesting and challenging aspects of aircraft design. The 777 airplane presented a unique challenge due to the size and complexity of the powerplants involved and concurrent development of installations for all three engine manufacturers, Pratt & Whitney, General Electric, and Rolls-Royce. To achieve success required a multi-disciplined approach involving aerodynamics, structures, loads, noise, weights, propulsion, flutter, and ECS, a clear understanding of design requirements, and a more open, working-together relationship with the engine companies(1) and the airlines to obtain a design with the best balance between performance (drag/TSFC), weight, noise, maintainability, producibility, and cost. Although Boeing had prime design/build responsibility, the engine companies were involved in all aspects of the design and offered designs and suggested improvements for consideration and evaluation.

Since the projected thrust and engine size requirements for the 777 had never before been achieved on a commercial transport, a Special Engine Study Team was established by F. Shrontz (Boeing) and B. Rowe (General Electric) early in the development phase (February-June, 1989) to determine the feasibility and identify "show-stoppers" for installing large diameter engines on the 777 considering all aspects of the installation. Initially the study concentrated on the GE engine but was soon expanded to include designs from all three engine companies - PW, GE and RR. Specialists from all disciplines were co-located to promote the free flow of information and ideas. While no "show-stoppers" were identified, the study did identify key changes required to the initial 777 airplane design, and areas requiring further development, including:

- Increased gear length, body attitude on the ground, and wing shear to provide adequate ground and taxi/runway light clearances for a nacelle up to 160" maximum diameter to accommodate engines of 100,000+ pounds thrust.
- Transportation and ground handling systems for the large engines.
- Research and development of manufacturing capabilities for the large strut-to-wing attachment pins, fittings, and wing skin thicknesses.

- Adoption of the conical core cowl exhaust system concept by all 3 engine manufacturers to achieve low aerodynamic interference drag goals.
- Nacelle geometric trades and minimum aerodynamic and installation parameters were established to provide design guidance for engine sizing, positioning and hardware design. High Reynolds Number testing was identified as a key to developing a suitable inlet design for engine-inoperative operation. The need for a new facility for calibration of large nacelle models was identified and conceptually developed.
- Predicted noise levels indicated a shortfall in meeting 777 noise goals and required further investigation.

Final review of the team results was held with Boeing and engine company management on June 7, 1989 for a go-ahead decision. The results and design effort were then turned over to the main 777 development team for detailed design.

To achieve airplane performance objectives within the tight program schedule required extensive use and development of improved Computational Fluid Dynamics codes. By coupling CFD with selective wind tunnel testing and a knowledge of past successful design practices, configuration options requiring study, hence, design cycle times, were reduced.

Design-build teams (DBT's) were established to provide a communication channel for trading performance improvement concepts against manufacturability, weight, engine TSFC, noise and maintenance considerations. The DBT environment also provided a good forum for Aerodynamics to provide the hardware designers with aerodynamic smoothness trade information, Figure 2, key areas of importance, and guidance on how the hardware design affected drag early in the design process. To reduce manufacturing costs and improve on the aerosmoothness inspection process, a Cross-Functional Aerosmoothness Task Team was formed and successfully identified a new process for reducing the number of non-value-added rejection tags and associated costs, while providing acceptable excrescence drag levels.

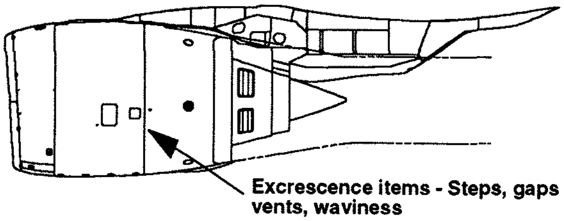


Figure 2. Aerodynamic Smoothness Excrescence Items

The 777 Program *Strategic Initiatives* included 100% digital product definition and digital pre-assembly. To support this goal, a common 3-D design and geometry system, CATIA, a product of Dassault of France, was established to speed the process of communicating information between the various design disciplines and the engine manufacturer, Figure 3. The use of CATIA reduced the need for expensive full-scale mockups, provided up-to-date hardware constraints for aerodynamic contour development, and reduced the number of "surprises" encountered during manufacturing.

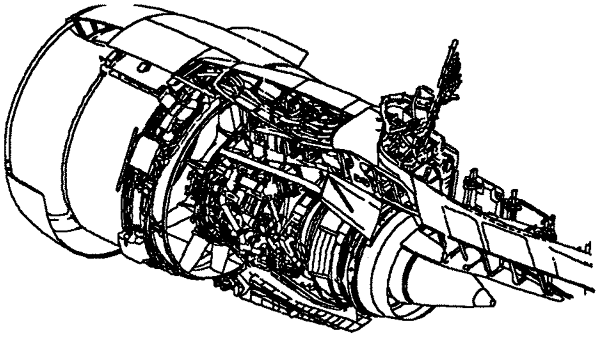


Figure 3. CATIA Drawing of Engine Components

The information provided in this paper summarizes the aerodynamic design requirements and validation criteria, test processes utilized, examines how non-aerodynamic influences affected the design, emphasizes those areas that were key in the aerodynamic development and validation of the 777 propulsion system installation, and provides a brief summary of the primary powered and unpowered test results.

Comparison To Prior Designs

The 777 engines are considerably larger than previously used on the 767 as shown in Figure 4.

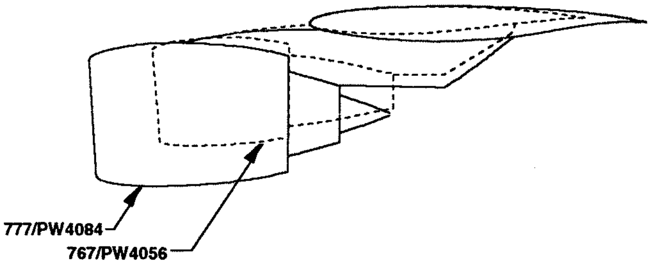


Figure 4. 777 versus 767 Nacelle Installations

Notable design differences are:

- Significant difference in exhaust system design philosophy. Use of a 15 degree conic exhaust system was adopted on the 777 based on lessons learned from the 747/RB211-524, 707/CFM-56 and 737/CFM-56 development efforts and as verified by CFD analyses of the 777 installation using TRANAIR(2,3,4,5). This, in combination with the wing design philosophy, provided benign underwing flow characteristics for the 777 and was a key to the low drag for the installation.

- The 777 PW4084 installation is closely coupled (wing-to-cowl gap) and in a similar location relative to the wing as the 767/PW4056 nacelle.
- The longer heat shield overhanging the core exhaust on the 777 is mainly due to the use of an external plug vs. the 767 internal plug nozzle. Special testing, and tailoring of the heat shield contour, were performed by the 777 Propulsion Group to ensure that hot core flow gases would not expand above the heat shield leading to overheating of the composite strut fairings.
- The 767 uses a cambered strut (not shown) while the 777 strut is symmetrical. Early studies and testing on the 777 showed no significant interference drag benefit for strut cambering relative to the cost of manufacturing "handed struts".

All three engine offerings for the 777 are compared in Figure 5. A similar design philosophy was used for all three designs. Engine location was selected to provide as much common strut structure and fairings as possible to minimize manufacturing costs and inventory requirements. All three designs utilized a ≈15 degree, conic core cowl with external plug nozzle.

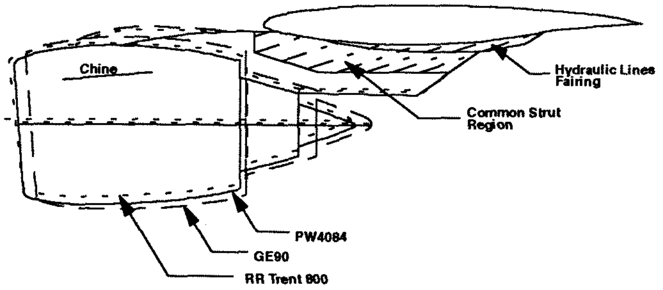


Figure 5. Comparison of PW4084/GE90/RR Trent 800 Engine Installations

Notable design differences are:

- The GE and RR have a fan case front engine mount which required an extended strut forward fairing while the PW core front mount allowed a shorter fairing.
- The RR has fan case mounted accessories (which required cowl cheek bulges) whereas the PW and GE have core mounted accessories.
- The PW and GE nacelles have 4 degrees of inlet droop to align the inlet with the wing upwash at cruise, while RR has 6 degrees to better accommodate the fan case mounted accessories.
- The larger size (bypass ratio = 9) of the GE engine is apparent, resulting in higher nacelle wetted areas.
- The GE operates at a lower fan nozzle pressure ratio (jet velocity) at cruise due to its higher bypass ratio which generated less blowing drag than the PW or RR engines as discussed in detail later.
- All three designs have a nacelle chine on the inboard side to improve low speed performance.

Historical nacelle placement relative to the wing for several aircraft/engine combinations is shown in Figure 6. Aerodynamically, history has shown that forward engine placement is generally beneficial for minimizing installed drag. The conventional wing/nacelle installation boundary delineates close-coupled configurations which usually present challenges to aerodynamicists, with highly tailored configurations often required. On this basis, the locations selected for the 777 engines are not highly close-coupled, but are similar to those used for the 767 and 757 airplane.

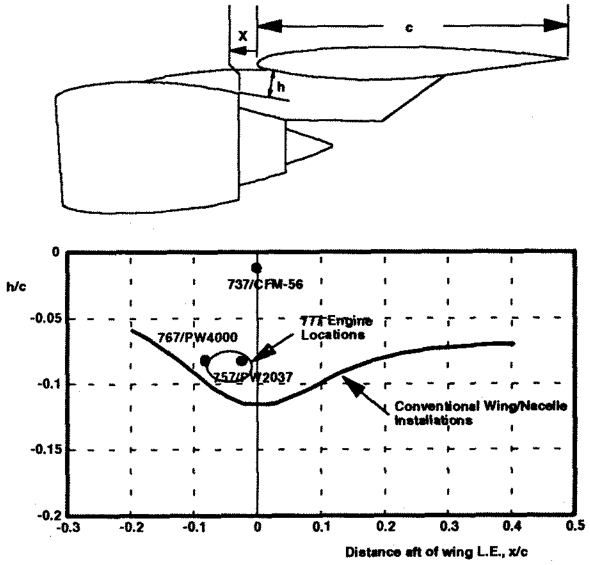


Figure 6. Nacelle/Wing Installation Location

Aerodynamic and Non-Aerodynamic Design Considerations

The design of the powerplant installation involves careful consideration of many factors both aerodynamic and non-aerodynamic(6).

Aerodynamic factors include (Figure 7):

- Overall shape and orientation of the nacelle affects nacelle and wing profile, wave, and induced drag at cruise and engine-inoperative conditions.
- The fore/aft, up/down, and spanwise positioning affect interference drag, wing span loading and induced drag. Spanwise location also influences vertical tail sizing for engine-inoperative control.
- The inlet external contours must provide minimum drag at cruise conditions while satisfying engine-

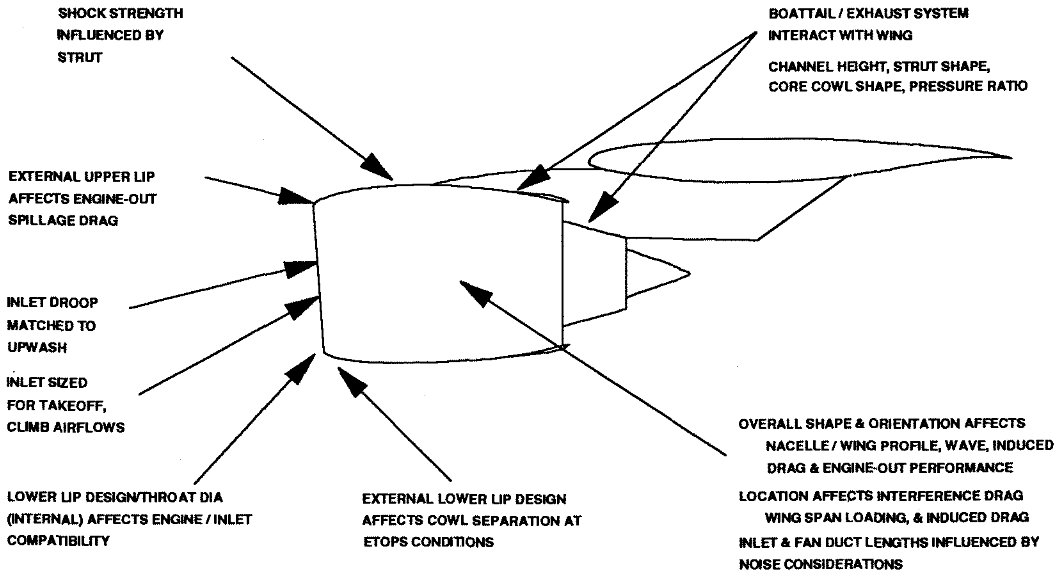


Figure 7. Factors Influencing Nacelle Installation Design

inoperative requirements of separation free flow and minimum spillage drag.

- The inlet internal geometry is designed to provide distortion-free air to the engine at all operating conditions such as takeoff and climb (high airflow, high angle of attack), while maximizing inlet recovery at cruise conditions.
- The strut contours must clear all hardpoints, such as engine mounts and wing attachment fittings, while minimizing adverse flow interactions with the nacelle and wing flow.
- The exhaust system and jet have a strong influence on the installation drag for under-wing mounted engines. Careful design of this component of the nacelle insures a successful design with trades often required between exhaust system TSFC and installed drag.

Non-aerodynamic factors: Some of the non-aerodynamic factors considered during development (Figure 8 - Figure 14) included:

- Engine size and bypass ratio were determined to meet the aircraft thrust requirements for takeoff, climb, cruise, and engine-inoperative conditions while providing efficient cruise operation. The size, location and orientation of the engine had a strong influence on the overall design of the 777.
- Ground, runway and taxi light clearances, allowable wing shear, manufacturing capability, installed drag, and airport gate compatibility (200" passenger entry door sill height limit) influenced the maximum engine size allowed.

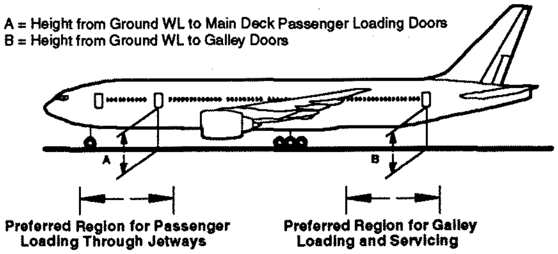


Figure 8. Passenger/Loading Door Sill Height Consideration

- Sufficient aircraft roll clearance was maintained to allow for abnormal conditions such as severe crosswind landings or flat tires.

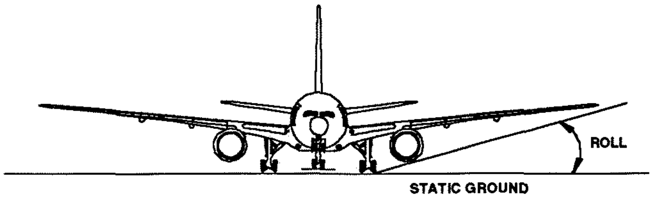


Figure 9. Roll Clearance

- The engine spanwise location was also chosen to satisfy passenger door escape slide, loading ramp and engine inlet flow hazard zone (suction) clearance requirements.

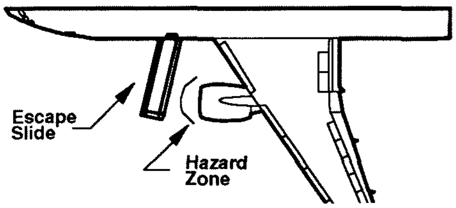


Figure 10. Escape Slide, Loading Ramp & Inlet Hazard Zone Consideration

- Ground clearance criteria also allows sufficient clearance for critical engine components in the event of a collapsed nose gear, although this is an extremely rare event for modern transports.

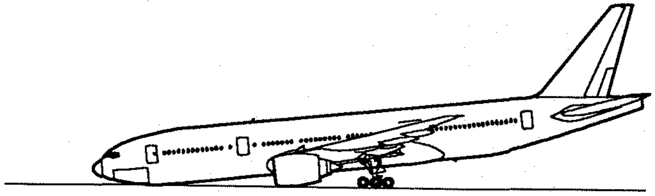


Figure 11. Collapsed Nose Gear

- Turbine disk burst zones were identified and wing dry bays were provided to avoid hazardous conditions in the event of a turbine disk burst. Location of critical aircraft control systems in this region was also avoided. Shielding of the turbine was not practical due to the extremely high energy of a turbine disk burst.

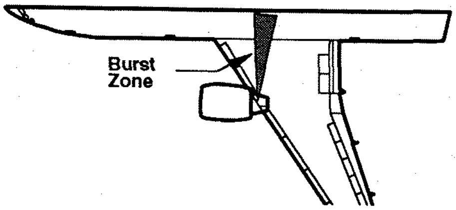


Figure 12. Turbine Disk Burst Zone Consideration

- Spanwise position of the engine was established outside a 22° cone to be free of nose or main gear water spray ingestion zones (based on statistical data) to ensure stable engine operation during takeoff or landing.

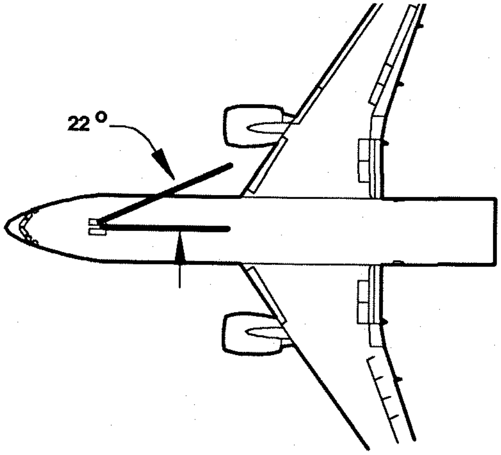


Figure 13. Nose Gear Water Spray Clearance

- Aircraft loads for the large engines pushed the limits of manufacturing capability and imposed weight penalties on the aircraft. During initial studies, limits were at first imposed on engine size due to wing attachment fitting and pin size manufacturing capability.

- Satisfying flutter design requirements limited the options allowed for optimum aerodynamic positioning of the nacelle, especially fore/aft positioning. Wing weight penalties and strut stiffness to satisfy estimated flutter requirements were a continual difficulty until transonic flutter tests were completed which allowed a reduction in the wing flutter weight penalties for the final design.

- Both the airframe and engine contribute to the total noise of the aircraft, with the dominant component dependent on the flight condition examined. At takeoff conditions, engine noise dominates while during approach at low power and high flap deflections, airframe noise may dominate. It is important to consider noise during the development of the engine installation and flap design. Powered model testing is often used to understand the trades involved in minimizing noise due to jet/flap interaction.

Engine noise reduction requires the use of acoustic materials inside the inlet, fan nozzle, and core exhaust (Figure 14). Requirements often dictate that the length of the inlet and fan nozzle extend beyond optimum for minimum aerodynamic drag. Engine efficiency also suffers due to acoustic material surface roughness and leakage losses.

To reduce noise, full-length duct mixed flow exhaust systems are in use on several commercial aircraft at the present. The concept uses pre-mixing of the fan and core flows prior to exiting the nozzle in order to reduce the jet velocity and associated noise. Thermal efficiency gains (engine TSFC) can also be realized due to the mixing, but is generally offset by higher installed nacelle drag and weight. Early studies for the 777/RR engine included a long duct mixed flow nacelle to reduce noise levels. Tested drag penalties for the installation, however, far outweighed the noise benefits.

For the 777 designs, acoustic treatment requirements increased inlet and fan duct lengths beyond that required for minimum cruise drag and inlet recovery for the PW and RR designs, while the higher bypass ratio of the GE engine allowed a somewhat shorter design due to its reduced jet velocity and noise characteristics.

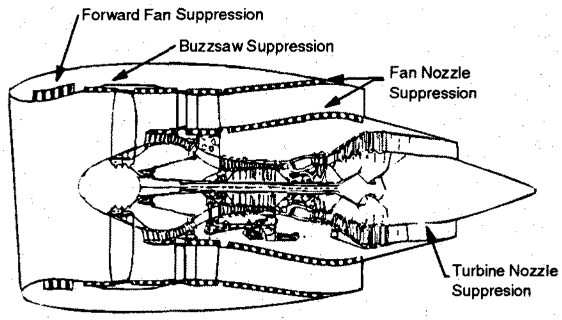


Figure 14. Engine Noise Treatment

Thrust/Drag Bookkeeping

An agreed-to thrust/drag bookkeeping system was employed to insure that all forces were properly accounted for between Propulsion, Aerodynamics, and the engine companies, to avoid double-bookkeeping of forces, and to maintain consistency from model testing up to full scale engine and aircraft testing. The system used during the 777 program (Figure 15) is consistent with traditional methods, wherein the forces and scrubbing losses inside the inlet captured streamtube and inside the fan jet boundary as obtained during nacelle/engine static calibrations are bookkept as thrust, while forces outside this boundary are bookkept as drag.

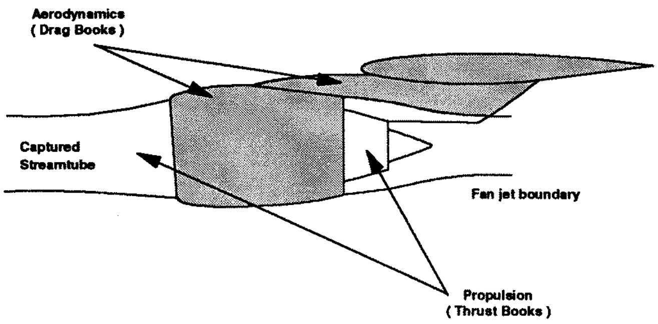


Figure 15. Thrust/Drag Bookkeeping Scheme

While this system simplifies engine calibration procedures (since thrust forces are defined statically), it should be noted that if the exhaust system forces change due to wing or external flow influences, the resultant forces in the exhaust stream are bookkept as drag. The only external flow effect typically accounted for in the thrust definition is the change in fan and core airflow due to flow suppression induced by the external flow or wing influences.

For the 777 engines, traditional full scale calibration facilities, such as the Willgoos altitude facility for PW engines or NGTE for RR engines, could not be used due to the size of the 777 engines. While the large altitude facility at Arnold Engineering Development Center (AEDC), Tullahoma, Tennessee was capable of calibrating the 777 engines, it was an unproven facility for commercial engines. A team composed of representatives from Aerodynamics, Propulsion and all three engine companies was therefore established to work this issue to insure that thrust definition and calibration procedures would result in accurate and consistent results for all three companies. To a significant extent, a "Common PW/GE/RR" thrust method was established relying on sea level static engine calibrations. Several areas were also "discovered" that have contributed to reduced accuracy during past programs, such as, inconsistent fuel flow calculation methods, nozzle area measurement procedures, and fan rake instrumentation. To provide validation of the "Common PW/GE/RR" thrust method, PW engines were also calibrated in AEDC's altitude cell.

Model calibrations were performed to provide airflow and internal force measurements to allow removal of internal forces bookkept in the propulsive books during wind tunnel testing. The standard Flight Simulation Chamber facility, Figure 16, was used for small models while a new larger rig was developed to allow calibration of the larger 777 nacelles, Figure 17. The main difference in the calibration approach is that the FSC provides reduced exhaust pressure whereas the pressure facility elevates the inlet pressure and exhausts to ambient pressure. This results in a different Reynolds Number for the two facilities, hence, cross-stand calibrations of similar nacelles were conducted to ensure consistency in results.

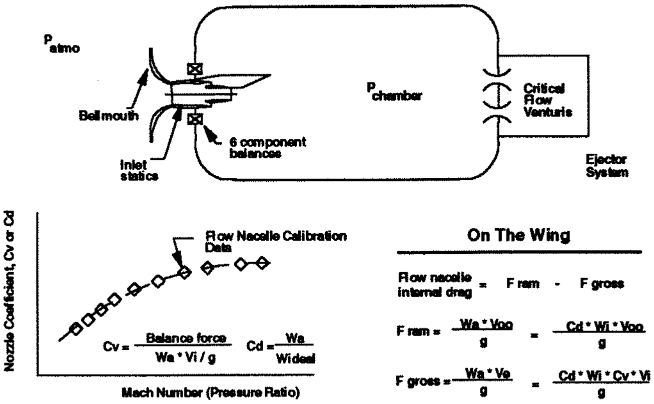


Figure 16 Flight Simulation Chamber (FSC) Nacelle Calibration Facility

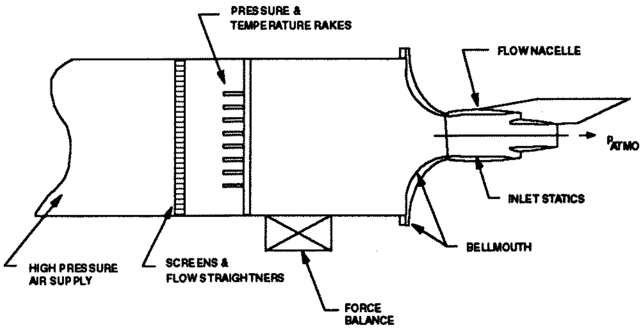
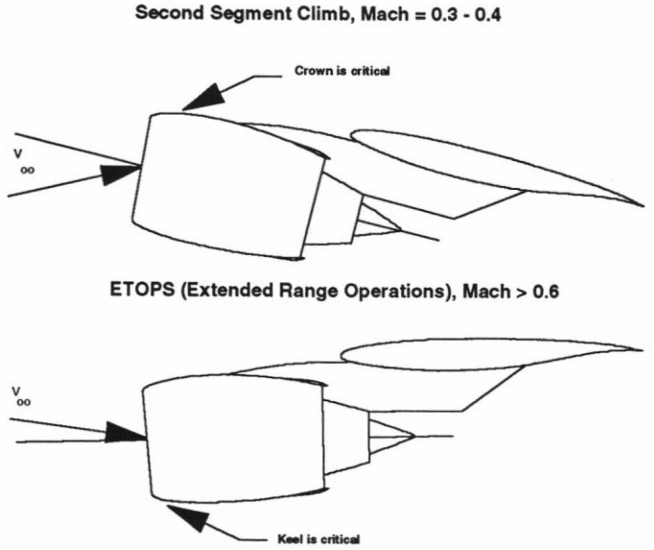


Figure 17 Pressure Rig For Nacelle Calibrations

Aerodynamic Design and Validation of the 777 Nacelle and Installation

Isolated Nacelle Design Philosophy

The aerodynamic design of the inlet and cowling involves compromises between satisfying criteria for takeoff, cruise, descent, and engine-inoperative conditions. For twin-engine aircraft, engine-inoperative requirements at takeoff conditions are more stringent than for 3 or 4 engine aircraft. Additionally, to provide airlines with operational flexibility, modern twin-engined transports strive for Extended Twin-Jet Operations (ETOPS) which imposes additional engine-inoperative design requirements for cruising with a failed engine at high speed for extended times. Each engine-inoperative flight regime imposes different design challenges since the inlet is exposed to a variety of flow, speed, and angle of attack conditions, Figure 18. The successful design is achieved by satisfying design requirement at these conditions while maintaining good cruise performance.


Second Segment Climb, Mach = 0.3 - 0.4

ETOPS (Extended Range Operations), Mach > 0.6

Figure 18 Inlet Design Conditions for Engine-Inoperative Operation

The external nacelle flow environment varies widely depending on engine airflow, speed, and angle of attack, Figure 19. At normal takeoff power conditions the inlet captured streamtube is large and the stagnation point is near the lip producing low external velocities on the cowl. As power is reduced the stagnation streamline moves inside the lip requiring the uncaptured flow to accelerate around the lip. At engine-inoperative conditions the inlet flow is low, causing high velocities and gradients on the external lip. If the gradients are sufficiently steep, flow separation occurs and external drag increases rapidly.

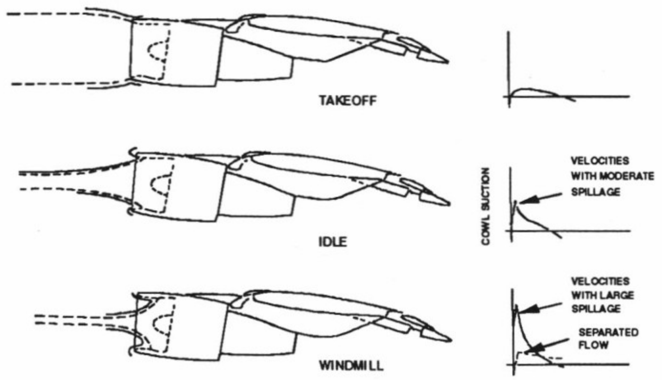


Figure 19 Nacelle Inlet External Flow Environment

For the 777, the nacelle was designed to satisfy stringent engine-inoperative criteria for takeoff and ETOPS conditions. The criteria used were to minimize engine-inoperative spillage drag at second-segment climb conditions to provide maximum performance during takeoff and climb, and at ETOPS conditions to minimize fuel reserve requirements for engine-inoperative operation and to provide acceptable engine-inoperative altitude capability.

Additionally, no external flow separation was allowed within the aircraft normal operational flight envelope which could otherwise cause pre-mature buffeting of the aircraft.

Isolated Nacelle Design - Use Of CFD

Computational Fluid Dynamics codes were used extensively to develop the 777 isolated nacelle design for both cruise and engine-inoperative conditions. An Euler(7) code was used for the majority of isolated cowl design since it provided profile and wave drag assessments which have been shown to give reasonable comparisons with test results. TRANAIR was used to investigate 3-D strut/cowl interactions and for tailoring the strut leading edge blending with the cowl.

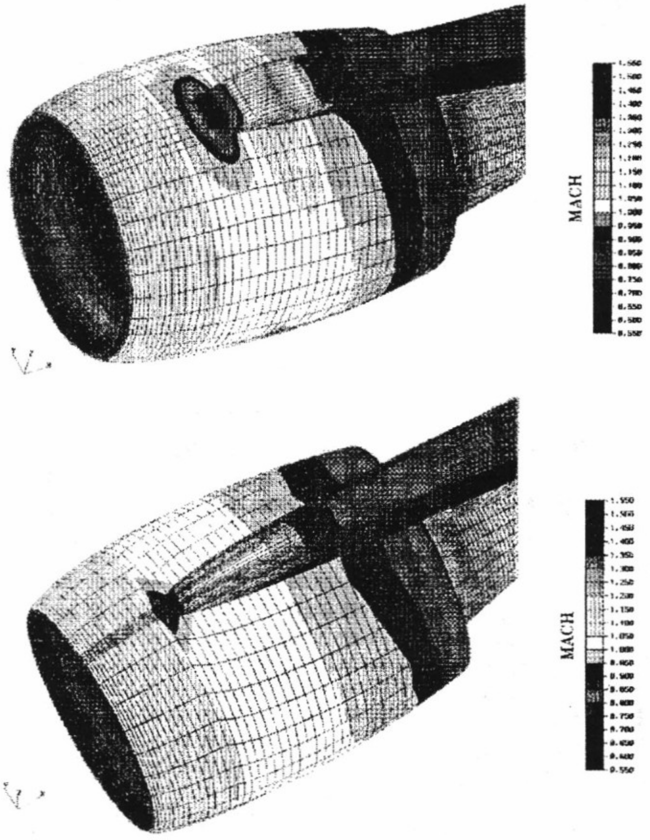


Figure 20. Inlet Design Using CFD

For example, Figure 20 shows how the flow tends to stagnate near the strut intersection with the cowl and accelerate over the top of the strut, reaching supersonic speeds for an early fairing design. By studying the flow patterns, an improved design was developed where the stagnation region is reduced and supersonic flow regions are minimized.

While these CFD codes provided a wealth of information about the flow on the nacelle and interaction with adjoining or adjacent surfaces at cruise conditions, drag prediction was somewhat limited. At engine-inoperative conditions, prediction of external flow properties and flow separation was rather poor based on comparisons with prior test results. Reliable inverse design codes were also not available, so reliance was placed on the designer's ability to judge contour modifications required to achieve the desired pressure distributions.

Thus, to validate the designs and to provide quantitative drag information for aircraft performance assessments, required wind tunnel testing.

Isolated Nacelle Design Validation - Cruise

High speed isolated nacelle wind tunnel testing was conducted in the Boeing Transonic Wind Tunnel for the PW, GE, RR nacelles to verify that the designs met cruise requirements of no shocks or drag rise until beyond Mach 0.83. The rig used, Figure 21, simply allowed testing with and without the nacelle and strut installed to allow assessing isolated nacelle/strut drag.

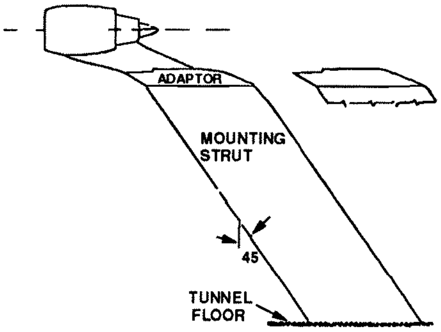


Figure 21 High Speed Isolated Nacelle Test Rig

Results (Figure 22) show that significant drag rise does not occur until beyond the airplane cruise Mach number.

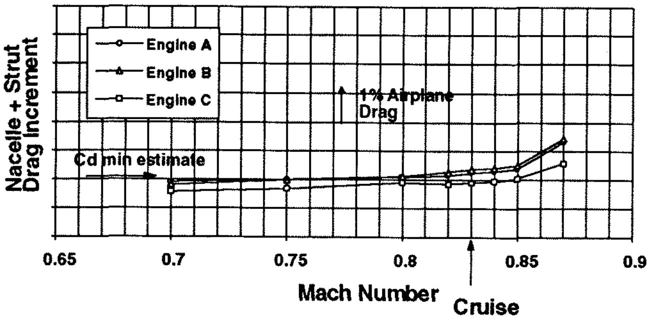


Figure 22. Isolated Nacelle Drag Comparison - Cruise

Isolated Nacelle Design Validation - Engine-Inoperative

Second-Segment Climb - To validate that the inlet/cowl design satisfied the 2nd segment climb design criteria of no external flow separation within the normal flight envelope required wind tunnel testing at high Reynolds Number since flow separation is significantly affected by boundary layer characteristics. Testing was conducted at the Cornell Aeronautical Laboratories (Calspan) using a 0.12 scale isolated nacelle model, Figure 23.

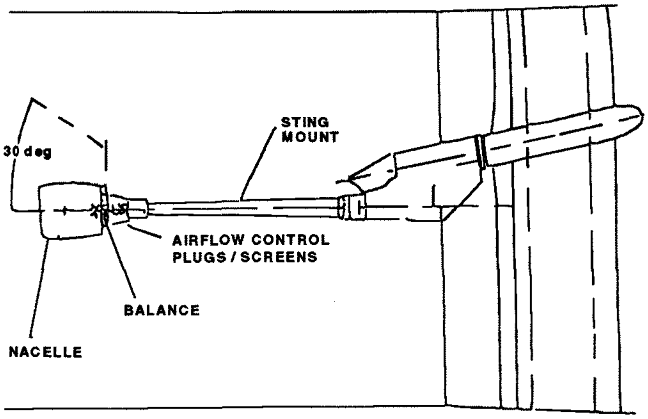


Figure 23 Isolated Test Rig For Engine-out Testing

Airflow, Mach number, angle of attack, and Reynolds Number were varied for all three nacelle designs (PW, GE, and RR). Typical results are shown in Figure 24.

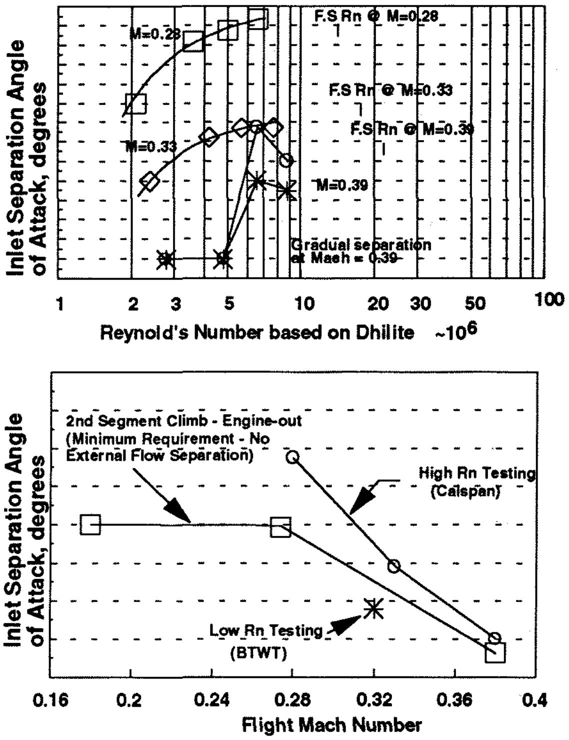


Figure 24. Effect of R_N on Inlet External Flow Separation Angle Of Attack at Second Segment Climb Engine-Inoperative Conditions

A significant dependency on Reynolds Number up to about 50% of full scale is apparent. If testing had only been accomplished at low R_N, as obtained in an atmospheric wind tunnel, this particular design would have been judged inadequate since flow separation was observed within the design flight envelope.

ETOPS Operation - At engine-inoperative conditions during ETOPS operation the inlet experiences higher Mach numbers and lower angles of attack. On the 777, the inlet generally operates at a slightly negative angle of attack at these conditions.

Cowl surface pressure data obtained during the Calspan testing were used to provide flow separation indications (where lip velocities collapse) as shown in Figure 25. Results show that no flow separation occurs on the keel lip at ETOPS conditions until beyond the normal flight envelope (-1 deg). Reynolds Number was limited, however, to about 17% of full scale for ETOPS conditions due to tunnel limitations at Calspan at the higher Mach numbers.

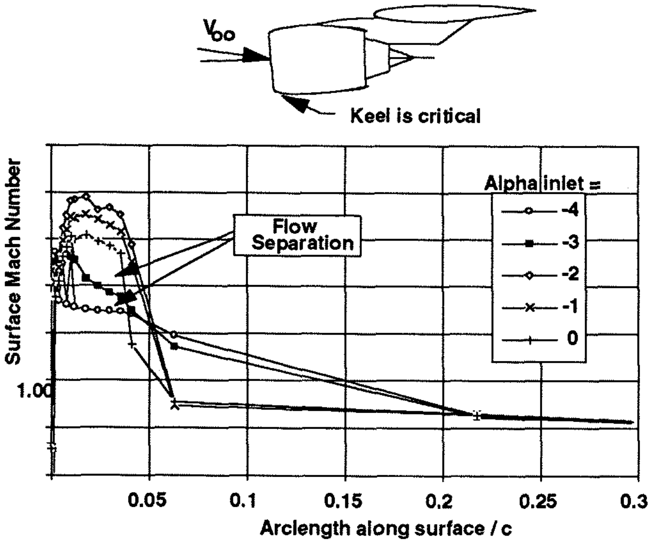


Figure 25. Inlet External Keel Lip Pressures at ETOPS Engine-Inoperative Conditions

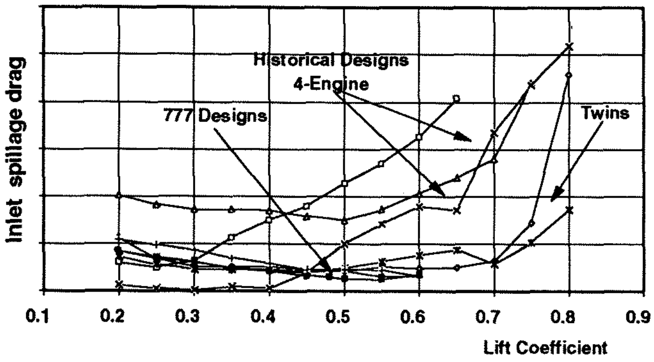


Figure 26. Engine-Inoperative Spillage Drag, M = 0.3

Flaps-up and down engine-inoperative spillage drag data were also obtained during installed nacelle testing for assessing pre-flight engine-inoperative performance characteristics across the C_L and Mach number range where engine-inoperative performance is important. When compared to prior airplane configurations, results compared favorably with historical designs, Figure 26.

Installed Nacelle Design Philosophy

The overall design philosophy used for the 777 nacelle and installation is summarized in Figure 27 and was based on proven successes in prior designs and extensive development using Computational Fluid-Dynamics and wind tunnel testing.

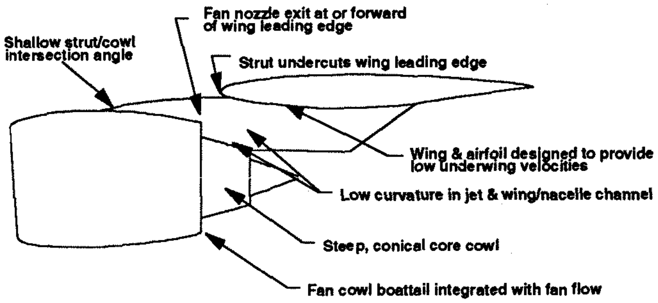


Figure 27. Aerodynamic Installation Design Features

Design features are summarized below:

- Testing and judicious use of CFD were used to provide design guidance during nacelle positioning studies and wing development. While CFD codes were not available to truly design the wing in the presence of the nacelle, adverse nacelle/wing/body flow interactions were minimized by observing the influence of the nacelle installation, tailoring the wing upper and lower surface pressure distributions accordingly, and careful selection of forward, vertical and spanwise positioning of the nacelles. Minimum interference drag was obtained with the fan nozzle exit forward of the wing leading edge.
- The wing and airfoil section were designed to provide a low velocity lower surface pressure distribution (nacelles-off) which allowed easier integration of the nacelle. When combined with the 15° conic exhaust system, a benign underwing flow field resulted which inherently provided lower blowing drag.
- The fan cowl boattail was designed to avoid adverse interactions with the jet flow to minimize flow suppression, plume shape and entrainment effects.
- Slab-sided (low curvature) strut contours were incorporated in the critical region between exhaust system and wing to avoid unnecessary acceleration of the flow.
- A shallow strut leading edge/cowl intersection angle was adopted to avoid stagnation regions, vortex

formation, pre-mature cowl shock formations and resulting drag penalties.

- The strut leading edge undercuts the wing leading edge (aft of wing stagnation point) thereby avoiding overwing strut fairings which have shown significant drag penalties on previous aircraft.

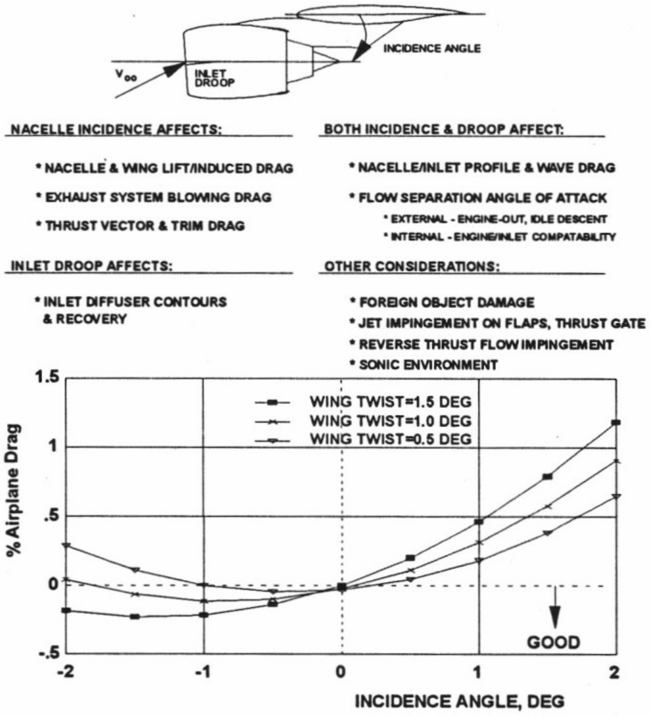


Figure 27. Nacelle Incidence Selection Considerations & Drag Trades

- The nacelle incidence selection was based on a number of factors outlined in Figure 27. Key elements to obtaining low drag were to avoid lift on the nacelle and to adopt an exhaust system and wing design providing benign flow characteristics to allow optimization of incidence. Summing the effects of profile, induced, trim, blowing, and thrust vector drag contributions (Figure 27) provided guidance on final incidence selection. Negative incidence angles were avoided to minimize the possibility of the jet blowing toward the wing or flaps. Testing was used to select the final design.

- Nacelle toe-in was selected to align the nacelle with the wing cross-flow based on TRANAIR studies.

Installed Nacelle Development Using CFD

Computational Fluid Dynamics plays an important role in the development of an efficient engine installation. Different concepts can be explored without the expense and time lag present with wind tunnel testing, especially powered nacelle testing. Disadvantages of CFD at the present include lack of sufficiently accurate drag information which is essential for making aircraft program design decisions. Inadequate validation (test/theory comparisons) of some of the newer codes also hinders applying results directly to assist in design decisions.

Considerable progress has been made in the last few years in simulating the details of the nacelle/strut installation geometry and exhaust flow. Improvements in TRANAIR predictions evolved continually during the 777 development much to the credit of the Boeing Engineering Aerodynamics Research Organization. Due to the highly interactive and complex flows in the jet, however, much work remains before CFD can be reliably used to predict proper trends.

Some of the newer Navier-Stokes and multi-block codes(11,12,13,14) allow design of the wing in the presence of the nacelle, and show promise for further improvements in exhaust system modeling. Unfortunately, these codes were not available during the 777 development effort and much reliance was placed on wind tunnel development with its inherent time lag.

TRANAIR was used extensively to guide development of the 777 nacelle installation, conduct trade studies, and to evaluate nacelle installation effects on the wing upper/lower surfaces and exhaust jet/wing flow interactions, Figure 28.

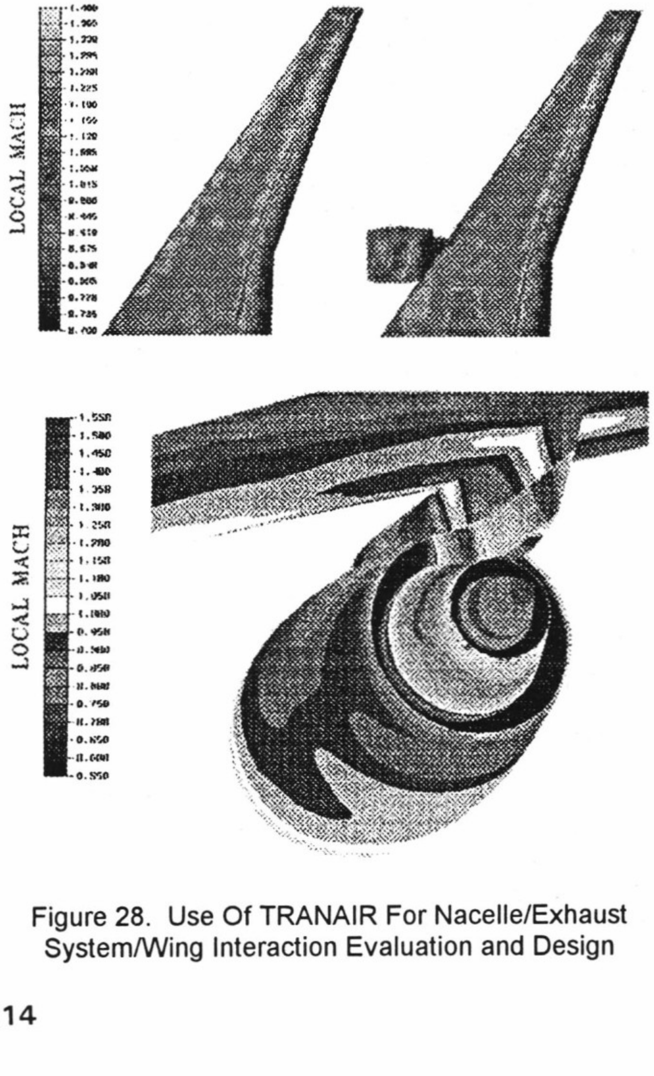


Figure 28. Use Of TRANAIR For Nacelle/Exhaust System/Wing Interaction Evaluation and Design

In working closely with the Propulsion Organization, trades were performed and configurations evaluated to develop the best compromise between installed drag and exhaust system TSFC to obtain the best overall installed performance. Navier-Stokes codes (PARC$^{(8)}$) were used to design and evaluate isolated exhaust system performance which were then evaluated installed under the wing using TRANAIR. Trades were performed between internal and external plug nozzles, core cowl boattail angle, fan duct length, and convergent and convergent-divergent fan nozzles. A mixed flow exhaust nozzle was considered early in the development program, but was eliminated when model testing showed that the installed drag penalty far outweighed the noise and TSFC benefits.

Nacelle positioning and strut cambering studies were also conducted using TRANAIR in combination with wind tunnel flow-through and powered nacelle testing to establish trades for nacelle and exhaust system interference drag against weight, flutter and structural considerations.

Installed Nacelle Testing and Drag Buildup Technique

For final configuration verification, a detailed sequence of model configurations was tested to ensure that all contributions to installed nacelle drag were accounted for in full scale drag buildups and to maintain consistency with the thrust/drag bookkeeping scheme adopted. Since existing model test methods do not allow simultaneous modeling of the complete flow field (inlet and exhaust), a series of models was required for a complete drag buildup, Figure 29. Although Turbo-Powered Nacelles are sometimes used to more closely simulate the inlet and exhaust flows simultaneously, Figure 30, their use is usually limited to cases where the inlet flow field couples with the wing or exhaust flow fields, or for thrust reverser flow simulation.

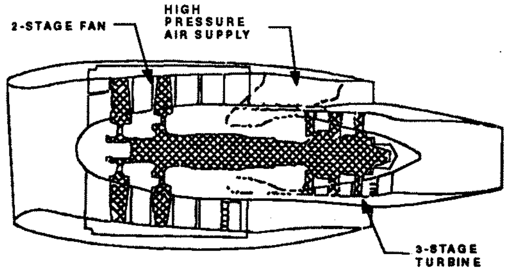


Figure 30 Schematic of Turbo-Powered Nacelle

For the 777, an unpowered flow through nacelle was used to establish a baseline for performance buildup with the inlet flowfield properly simulated. Since this model was un-powered, the proper flowrate through the inlet was obtained by increasing the nozzle area by truncating the primary cowl.

Since the modified primary cowl has a different flow interaction with the wing and external flow relative to the production primary (commonly labeled "Geometry drag"), a second flow-through model was tested with the production primary cowl geometry. This model had a contracted inlet to minimize excessive spillage effects at the lower airflows obtained with the production primary. To use this method required special isolated nacelle testing to

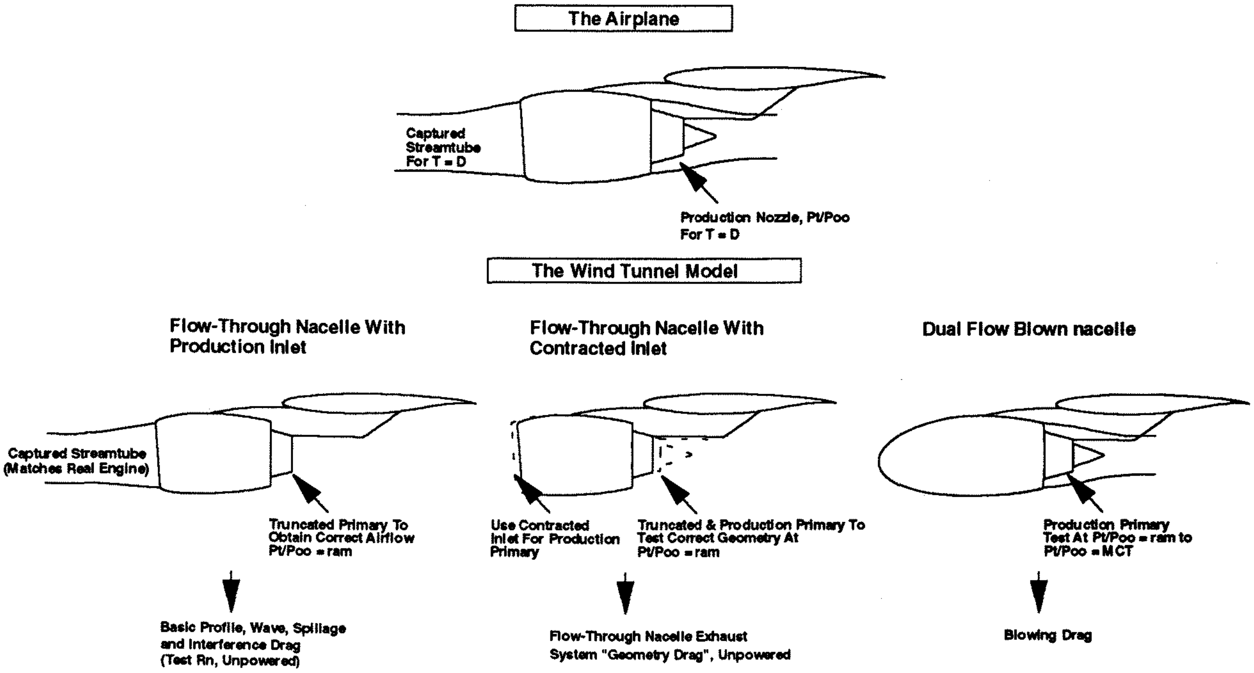


Figure 29. Model Testing Required For Full-Scale Drag Buildup

determine the differences in drag between the contracted and production inlets. While a more traditional method uses a single-flow blown nacelle to obtain the "geometry drag", the contracted inlet method was used on the 777 program to reduce test costs.

Lastly, a powered (blown nacelle) model was used to test the production exhaust system at ram and cruise pressure ratio to obtain jet interference (blowing) drag. Blowing drag is defined as the change in external drag as the jet pressure ratio is increased from ram to cruise conditions,

$$\Delta C_{D_{\text{blowing}}} = C_{D_{\text{cruise pressure ratio}}} - C_{D_{\text{ram pressure ratio}}}$$

where ram pressure ratio is that obtained by a flow-through nacelle's exhaust system. All internal forces (thrust) are removed via nacelle static calibrations (with no external flow). Thus, blowing drag is a measure of the jet's influence on its' surroundings, and the effect of the external flow on the exhaust system performance.

Installed Nacelle Design Validation

Back-to-back installed nacelle half-span model powered and un-powered wind tunnel testing was conducted for the PW, GE, RR nacelles to establish full scale drag buildup parameters and to verify that the designs satisfied cruise requirements. Results, corrected to full scale (Figure 31), show similar drag levels for all three engines and very little drag variation with Mach number until beyond the airplane cruise Mach number (0.83). The drop-off at higher Mach numbers is typical and reflects the favorable effect of the nacelle on the wing polar at higher than design lift coefficients and Mach numbers.

The nacelle installation drag breakdown at cruise conditions for the three engine installations is shown in Figure 32 to provide visibility on the order of magnitude of the various components involved in the drag buildup and differences between engines.

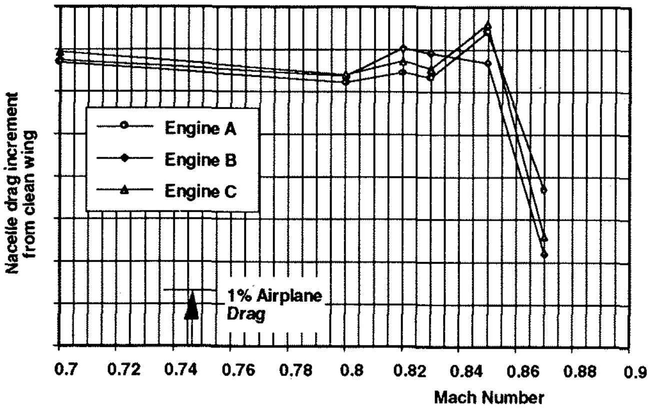


Figure 31. Installed Drag Comparisons, Wind Tunnel Data Corrected to Full Scale

Results show that the estimated flight drag levels, and magnitude of the corrections to full scale, are very similar for all three engines which isn't surprising considering that the same design philosophy was used for all three engine installations.

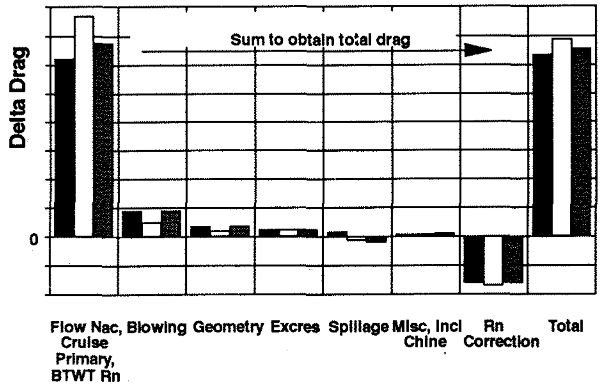


Figure 32. Cruise Drag Buildup For PW4084, GE90, RRTrent Engines

Un-powered nacelle drag differences are mainly due to differences in profile drag (wetted area); other differences are small except for blowing drag. The differences in blowing drag for the three engines is best understood by examining how the jet influences the external flow, Figure 33.

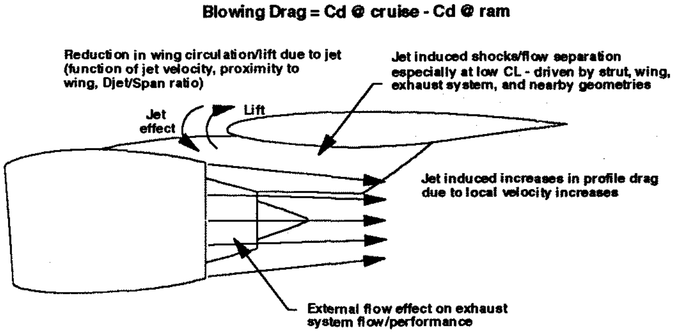


Figure 33. Effect Of Jet Exhaust Flow On Interference Drag

In an underwing mounted engine installation where the wing is lifting, the jet induces a higher velocity which counters the natural circulation around the wing. As the jet velocity increases above freestream levels, the circulation around the wing is reduced resulting in a local loss in lift and a poorer span loading and polar shape. To regain the lift, the wing must fly at a higher angle of attack, regaining the lift with the poorer polar shape. Additional losses are incurred if jet-induced velocities exceed sonic levels appreciably creating shocks or flow separation on the wing or strut. Profile drag also increases due to the higher local velocities outside the jet boundary, but is usually minimal unless flow separation occurs.

Additionally, since exhaust system thrust is defined during static calibrations with no external flow (by convention), any exhaust system losses created by the external flow are bookkept as drag items, and may also contribute to blowing drag.

Minimum blowing drag is achieved when only lift-induced losses remain, Figure 34. By analyzing the measured lift loss, ΔC_L , and converting to equivalent induced drag (as shown below), lift loss effects can be separated from those due to shocks or separated flow.

$$\Delta C_{D_{induced}} = 2 * C_L * (\Delta C_L) / \pi * AR * e$$

In Figure 34, lift losses generally dominate at C_L's higher than cruise where the drag polar is poorer and jet velocities are higher (due to the higher thrust required for level flight). At lower C_L's, wing lower surface shocks develop, especially at higher power settings, contributing additional losses. By converting the lift loss into equivalent induced drag, using the polar shape and equations shown previously, the blowing drag due to the lift loss can be obtained and correlated with the jet-to-freestream velocity ratio, Figure 35.

Since polar shape plays a major role in the magnitude of the blowing drag observed, special care must be taken to design the wind tunnel model, and select the test facility, to insure that the polar shape reflects full scale airplane characteristics, Figure 35.

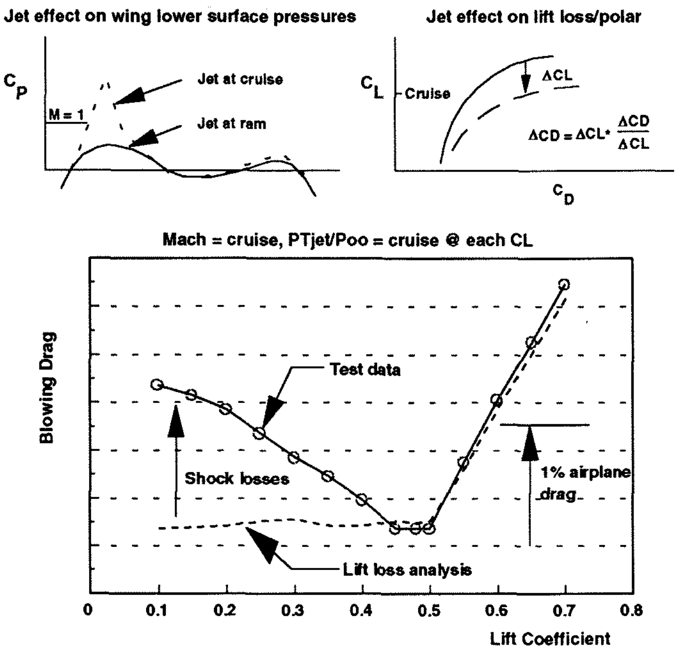


Figure 34. Breakdown of Blowing Drag Into Lift and Shock-Induced Losses

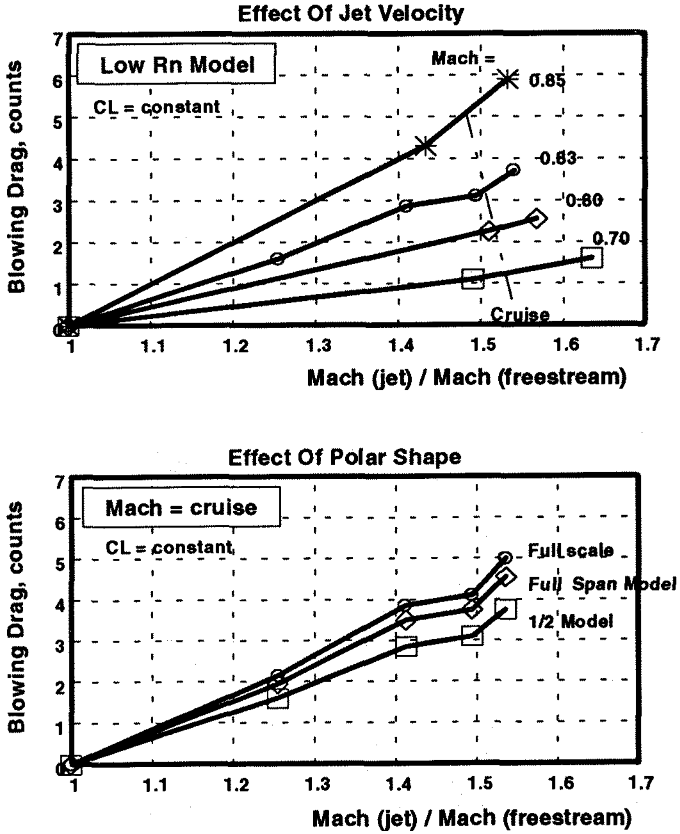


Figure 35. Effect of Jet Velocity & Wing Polar Shape On Blowing Drag Due to Lift-Loss

The importance of these effects are better appreciated by comparing the blowing drag for the 777 conical exhaust system with that achieved with a older, curvilinear exhaust system design, Figure 36. The high levels at cruise and at lower C_L's for the curvilinear exhaust system are caused by the high Mach numbers and localized shock formations created by the uncontrolled external flow turning of the jet coupling with the wing flow field. The magnitude of the shock strength can be correlated with peak underwing velocity, Figure 37.

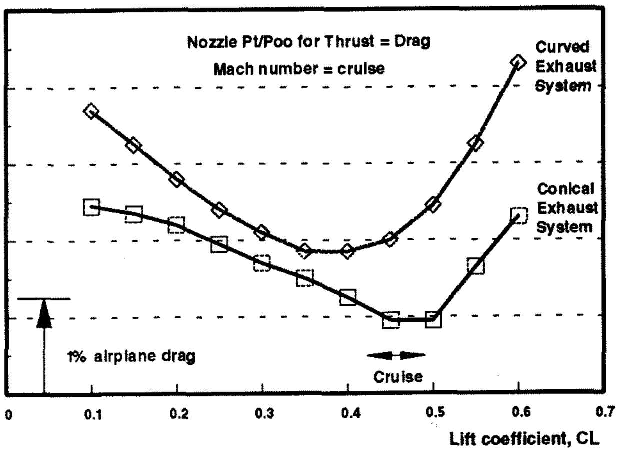


Figure 36. Comparison of Blowing Drag For Curvilinear versus Conical Exhaust System

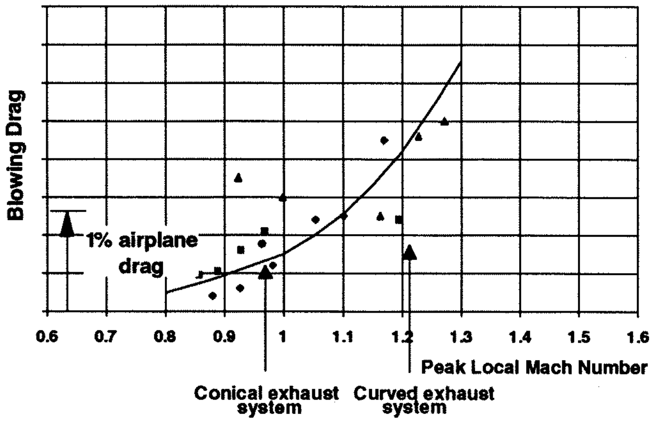


Figure 37. Correlation of Blowing Drag with Peak Local Mach Number

Future Engine Developments

Future engine developments are heading toward larger, higher bypass ratio engines with higher thrust capability due mainly to the economics of large twin-engined or high capacity aircraft, Figure 38.

Different engine manufacturers offer different solutions to this challenge[9], Figure 39, with direct drive fan engines up to BPR 9.0, geared fans to BPR 15-25, unducted ultra-high-bypass-ratio propfans, and counter-rotating fan concepts being studied.

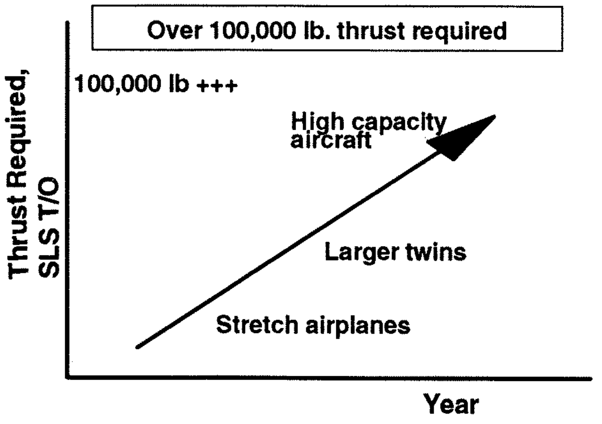


Figure 38. Future Thrust Growth Requirements

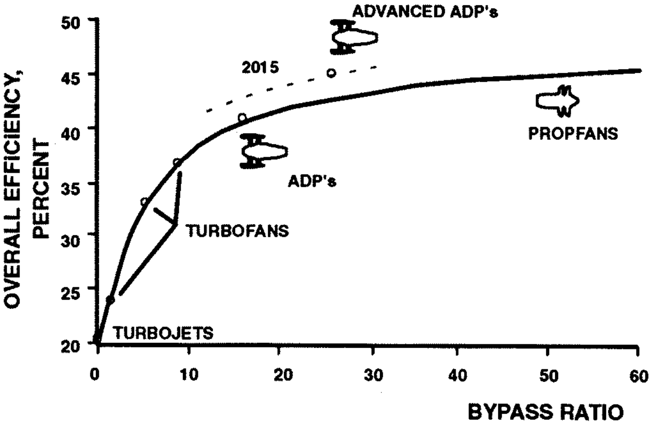


Figure 39. Advanced Engine Efficiencies

While engine TSFC is improved for the higher bypass ratios, the aerodynamic challenge is to install these engine with minimal offsetting drag increases[10] which could reduce the optimum bypass ratio, hence, efficiency, achievable, Figure 40.

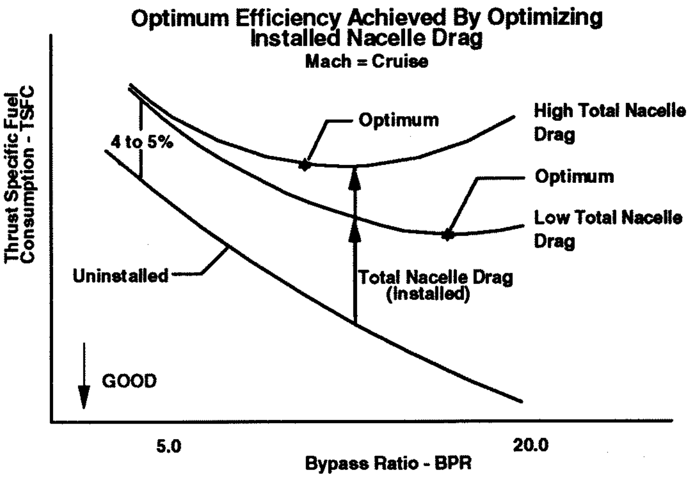


Figure 40. Optimizing Installed Engine Efficiency

Advanced Ducted Prop Engine

One concept employs a variable pitch fan with gearing to keep fan speeds low while allowing higher turbine speeds for greater efficiency, Figure 41.

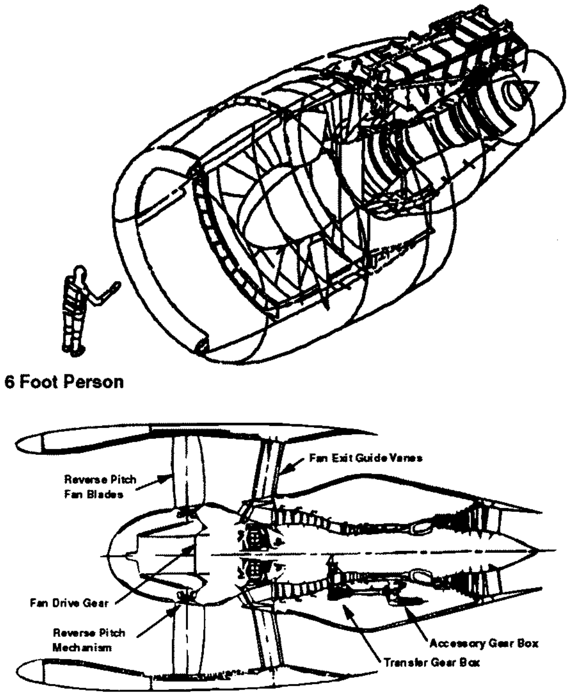


Figure 41. Advanced Ducted Prop Engine

By employing variable pitch fan blades, optimum fan efficiency can be obtained over a wide variety of flight conditions. Additionally, by reversing the fan blades, reverse thrust can be obtained without the complexity and weight of the conventional blocker door/cascade arrangement. However, the challenge lies in getting the flow to reverse through the exhaust system with minimal distortion which could otherwise cause compressor or fan stall.

Conventional vs. Advanced Ducted Prop Installation

For the aerodynamicist, features of an advanced ducted prop nacelle design which are different from conventional engines include (Figure 42):

- Lower jet velocities which allows closer coupled installations and reduced blowing drag.

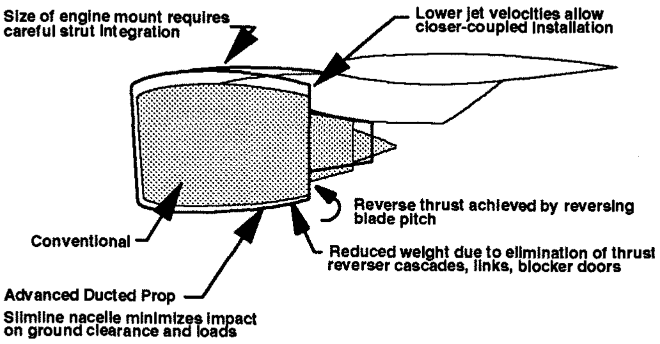


Figure 42. Conventional vs. Advanced Ducted Prop Installation

- Slimmer nacelles are allowed (higher D_{hilite}/D_{max} ratios, Figure 43) due to the higher engine-out airflows possible with a variable pitch fan engine. This would allow a larger engine with higher thrust within a limited space. To take advantage of this aspect of the engine, however, the designer must be assured of a fail-safe gearbox design that provides a low loss blade angle during failed engine operation.

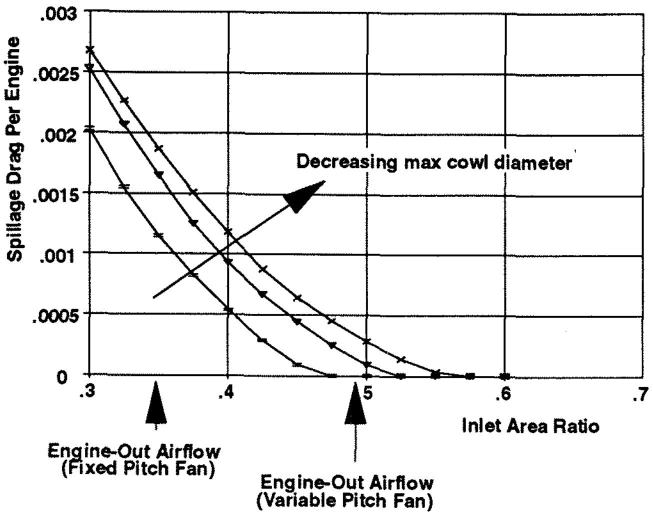


Figure 43. Higher Engine-Out Airflows for ADP Engines Allows Slimmer Nacelles

- Larger engine mounts, strut-to-wing fittings and associated strut design difficulties.
- Significant air-conditioning bleed penalties for the higher bypass ratio engines which suggests that alternate sources of air-conditioning airflows should be investigated (full-time APU, turbo-compressors, etc).

Conclusions

The modern powerplant installation is a complex system requiring close coordination among many disciplines such as Aerodynamics, Propulsion Design, Structures, Loads, and Noise as well as the engine manufacturer to achieve a successful, efficient installation.

The design of the powerplant installation involves careful consideration of many factors both aerodynamic and non-aerodynamic, and can have a significant effect on the overall airplane design.

Aerodynamic design of the nacelle involves compromises between satisfying criteria for takeoff, cruise, descent, and engine-out conditions. Each flight regime imposes different requirements. For twin-engine aircraft, engine-out requirements at takeoff conditions are more stringent than for 3 or 4 engine aircraft. Achievement of design objectives is assisted greatly through the use of high Reynolds Number wind tunnel testing.

Computational Fluid Dynamics plays an important role in developing an efficient nacelle and engine installation with minimal testing. CFD also allows concepts to be studied quickly without the time lag of wind tunnel testing. Disadvantages of CFD at the present are the lack of firm drag information which is essential for making aircraft design decisions, and inadequate validation (test/theory comparisons) which hinders applying results to assist in design decisions. Recent advances in application of Navier-Stokes and inverse design codes show promise in improving the reliability of CFD codes for full configuration evaluation, design of the wing in the presence of the nacelle, and improving the exhaust system modeling with the goal of reducing the design cycle time and cost for configuration development.

Future engine developments are heading toward larger, higher bypass ratio engines with higher thrust capability and improved engine TSFC. The aerodynamic challenge will be to install these engines with minimal offsetting drag increases.

Acknowledgments

The development of the 777 propulsion system installation required true teamwork on the part of many individuals and organizations both within and outside Boeing. Their contributions and dedication were key to the success of the 777 engine installations. Of particular note are the following people and organizations,

777 Aerodynamics:

D. G. Akiyama, M. Boctor, R. B. Carlson, T. P. Campbell, M. M. Curtin, M. I. Goldhammer, M. S. Hoffman, I. S. Reichmanis, J. Runyan, A. Shariatmadar

777 Propulsion:

M. W. Su, J. Piro, J. L. Colehour, J. Vann, B. Neal, R. L. Balzer, D. Foutch, L. J. Herbert, B. L. Koh, M. Piraino, M. L. Sanguin, C. Clark

and many other key individuals in the Boeing Aerodynamic Laboratories, Boeing Aerodynamics Engineering Research Unit, Boeing Propulsion Systems Division, Pratt & Whitney, General Electric, and Rolls-Royce.

References

1. Tiegarden, Fred W., "GE90 Integration With The B777 - A Success Story", Royal Aeronautical Society Paper, RAes Conference, Nottingham, England, October, 1992.

2. Tinoco, E. N., Chen, A. W., "CFD Applications to Engine/Airframe Integration," Numerical Methods for Engine-Airframe Integration, Vol. 102 of Progress in Astronautics and Aeronautics, AIAA, 1986.

3. Bussoletti, J. E.., "TRANAIR Application Notes: The TRANAIR Boundary Layer System," Boeing Document D6-56275, November 1991.

4. Chen, A. W., Curtin, M. M., Carlson, R., Tinoco, E. N., "TRANAIR Applications to Engine/Airframe Integration", AIAA-89-2165CP, August, 1989.

5. Young, D. P., Melvin, R. G., Bieterman, M. B., Johnson, F. T., Samant, S. S., Bussoletti, J. E., "A Locally Refined Rectangular Grid Finite Element Method: Application to Computational Fluid Dynamics and Computational Physics," Journal of Computational Physics, Vol. 92, No. 1., January 1991.

6. Berry, D. L., "Civil Aircraft Propulsion Integration: Current & Future", SAE 932624, AeroTech 93 Conference, Anaheim, Ca, September, 1993.

7. Boeing Document D6-56359, "A588 User's Guide," March 27, 1992

8. Arnold Engineering Development Center Technical Report AEDC-TR-89-15, "PARC Code: Theory and Usage," Cooper,G.K., Sirbaugh, J.R., December 1989

9. Kraus, E. F., Park, P., Raymer, D. P., McMasters, J. H., AIAA Short Course, "Perspectives in Aircraft System Design", February 20-21, 1993, Irvine, Ca

10. Greff, E., Becker, K., Karwin, M., Rill, S. "Integration of high bypass ratio engines on modern transonic wings for regional aircraft", Aeronautical Journal Paper No. 1896, January, 1993.

11. Kao, T. J., Su, T. Y., and Yu, N. J., "Navier-Stokes Calculations for Transport Wing-Body Configurations with Nacelles and Struts," AIAA 93-2945, July, 1993.

12. Smith, L. A., and Campbell, R. L., "Applications of a Direct/Iterative Design Method to Complex Transonic Configurations," NASA TP-3234, September 1992.

13. Huffman, W. P., R. G. Melvin, Young, D. P., Johnson, F. T., Bussoletti, J. E., Bieterman, M. B., and Hilmes, C. L., "Practical Design and Optimization in Computational Fluid Dynamics," AIAA-93-3111, July 1993.

14. Clem, B.C., Elliott, J. K., Tamigniaux, T. L. B., and Tinoco, E. N., "Recent CFD Applications on Jet Transport Configurations," AIAA-92-2658CP, June 1992