JA217

Preliminary Design and Evaluation of High Bypass Nacelle Inlets Using 3-D Navier-Stokes CFD Simulation

By Joel Frank
Senior Staff Aerodynamicist
Aerostructures Division, Goodrich Corporation
Chula Vista, California

The use of computer simulation significantly reduces the time required to evaluate inlet performance and preliminary design for the engineers at Goodrich Corporation's Aerostructures division. Engineers can design, modify and evaluate the performance of a large array of inlet configurations by closely coupling GAMBIT, FLUENT and CATIA. In the past, inlet design was a lengthy iterative process that could last months prior to finalizing a design followed by expensive inlet performance tests on the engine thrust stand and in flight. Today, Computational Fluid Dynamics (CFD) analysis using FLUENT software has not eliminated those inlet tests, but has drastically reduced their number and cost. By weeding out the least promising inlet designs by means of CFD the Goodrich engineers can focus on only the best inlet candidates and reduce the number of flight tests and avoid potential surprises such as shocks, or shock induced separation. These two phenomena can be potentially detrimental to inlet total pressure recovery and to the structural dynamics of the inlet. Goodrich engineers have mastered CFD techniques to identify these undesirable features during the preliminary design stage and can potentially eliminate them.

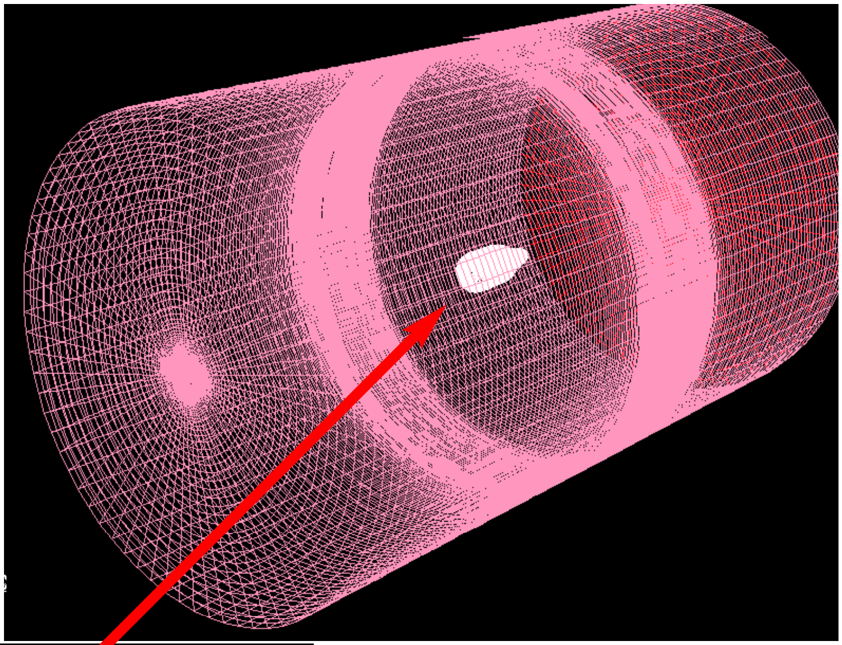
In a generic example, Goodrich engineers successfully used CFD to simulate the airflow through and over a generic nacelle inlet under a range of different operating conditions and identified areas of aerodynamic improvement. The engine supplier often sets the initial inlet design based on engine hard-point constraints, engine operability and inlet total pressure recovery. Goodrich's role is to scrutinize the inlet initial design for airloads and performance. Although an inlet may be adequately designed by an engine supplier to provide uniform flow to the engine and maximize inlet performance, Goodrich pays great attention to minimizing the inlet airloads and vibrations without compromising inlet total pressure recovery. Inlets that are properly designed by the engine supplier to maximize cruise and take-off performance sometimes may be less effective in other flight conditions such as the beginning of descent when potentially high unsteady loads generate shock induced separation. This can be avoided in the preliminary design phase by developing a more robust inlet construction. Goodrich's preliminary design techniques using CFD assure that inlets that may potentially be aerodynamically or structurally unsound do not make it to final production. Ultimately, CFD has allowed Goodrich to fully optimize its inlets from an engine performance, structural, weight, and cost standpoint. This approach was largely driven by Goodrich's Lean Process Deployment (LPD) policy that encourages engineers to strive for the best product design in a cost efficient manner.

Goodrich Engineers were able to use CFD to design inlets that have very mild shock strengths throughout the majority of the flight envelope without compromising engine performance and engine inlet recovery. By analytically computing the location and magnitude of the potential shock strength, the engineers were able to eliminate these phenomena in the iterative preliminary design process.

Copyright © 2005 Fluent Inc.

Goodrich inlets are designed to exceed the specification requirement loading conditions but occasionally can undergo a re-design in order to save additional weight thanks to innovative lighter construction materials and techniques. It is at this stage that Goodrich aerodynamicists re-evaluate the inlet aerodynamics using CFD in order to ensure that the inlet flow field is sound. Under certain flight envelope conditions, an inlet may have transonic flow around its lip, provided that the Mach number is gradually diffused or provided that the shock-strength does not exceed certain proprietary guidelines. Goodrich aerodynamicists have identified the right balance between inlet performance, structural loading and weight so that an inlet is optimally designed. An inlet that is overly designed ultimately will have unnecessary weight and so will cost more to fly. A well-designed inlet will gradually diffuse the Mach number over its surface so as to avoid the sudden change in static pressure, temperature and density that may lead to a potential shock wave and shock induced separation at all areas of the flight envelope.

While a sudden rise in static pressure concentrated over a very small region of the inlet may cause relatively high loading, the unsteady oscillating nature of a shock and its interaction with the turbulent boundary layer eddies can exacerbate the shock strength and lead to shock induced separation which can be far more damaging. The eddy vortices get amplified by several orders of magnitude as they interact with the boundary layer. The mechanical vibration of the nacelle structure exacerbates the problem. At certain levels the potential acoustic field downstream of the shock generated by this interaction may cause inlet fatigue damage. 3-D CFD has allowed the Goodrich engineers to obtain a better understanding of conditions under which these adverse phenomena occur so as to prevent associated high loadings.



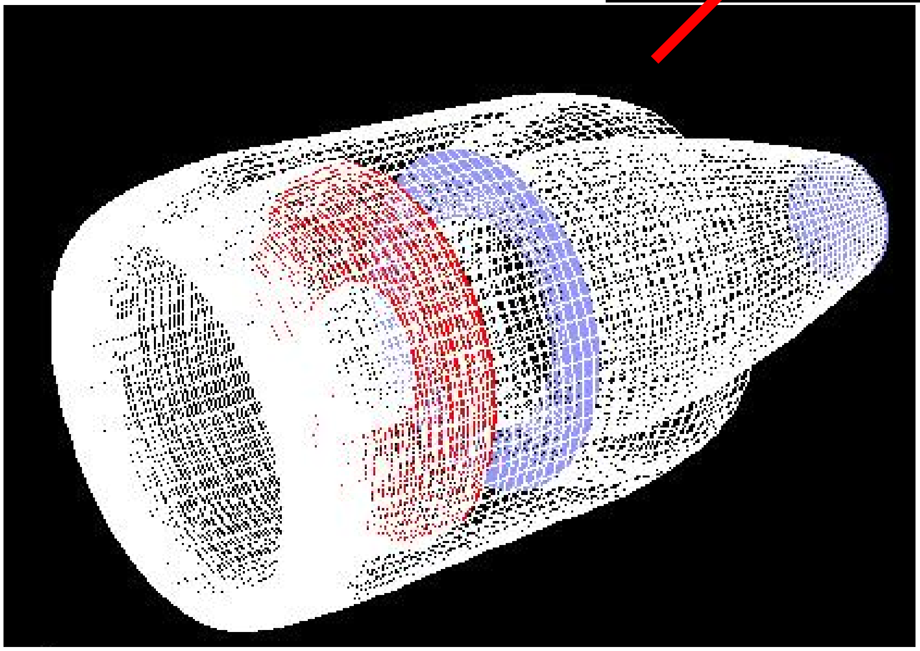


Figure 1 (top) and Figure 2 (bottom): boundary layer and nacelle meshes

As part of Goodrich’s continuous improvement process and research and development, engineers have extensively studied these phenomena by using CFD supported by flight testing in the spirit of ensuring a better, safer and more competitive product.

Inlet aerodynamic optimization: flight testing vs. CFD

CFD can give a reasonably good understanding of the steady state inlet flowfield. FLUENT allows the inlet aerodynamicist and the designer to avoid potentially undesirable unsteady phenomena by designing them

Copyright © 2005 Fluent Inc.

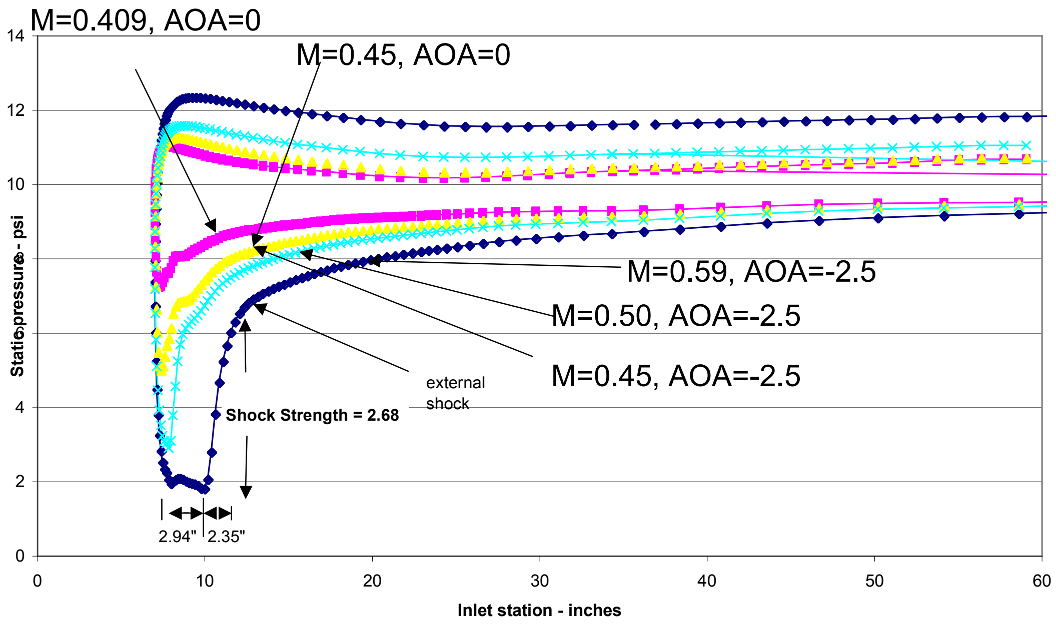


Figure 3: Static pressure distribution at bottom dead center for mid-descent conditions

out of the concept. Without FLUENT, inlet designers and aerodynamicists would have to rely solely on wind tunnel and flight testing in order to ensure a shock-free product but design changes would often have to be made late made late in the program. By ensuring that the CFD calculated inlet flowfield is free of any unsteady behavior and free of shock-induced separation the engineer no longer needs to rely solely on flight testing. In the past, the only way to ensure an aerodynamically sound inlet was through wind tunnel and flight-testing. Wind tunnel testing only provides an approximate evaluation of the inlet aerodynamics since the correct full-scale flight Reynolds number cannot be duplicated. The nacelle industry has traditionally relied on flight tests where technicians drilled holes through the inlets of the nacelles in order to insert the dynamic and static pressure instrumentation in an inlet installed on a flying test

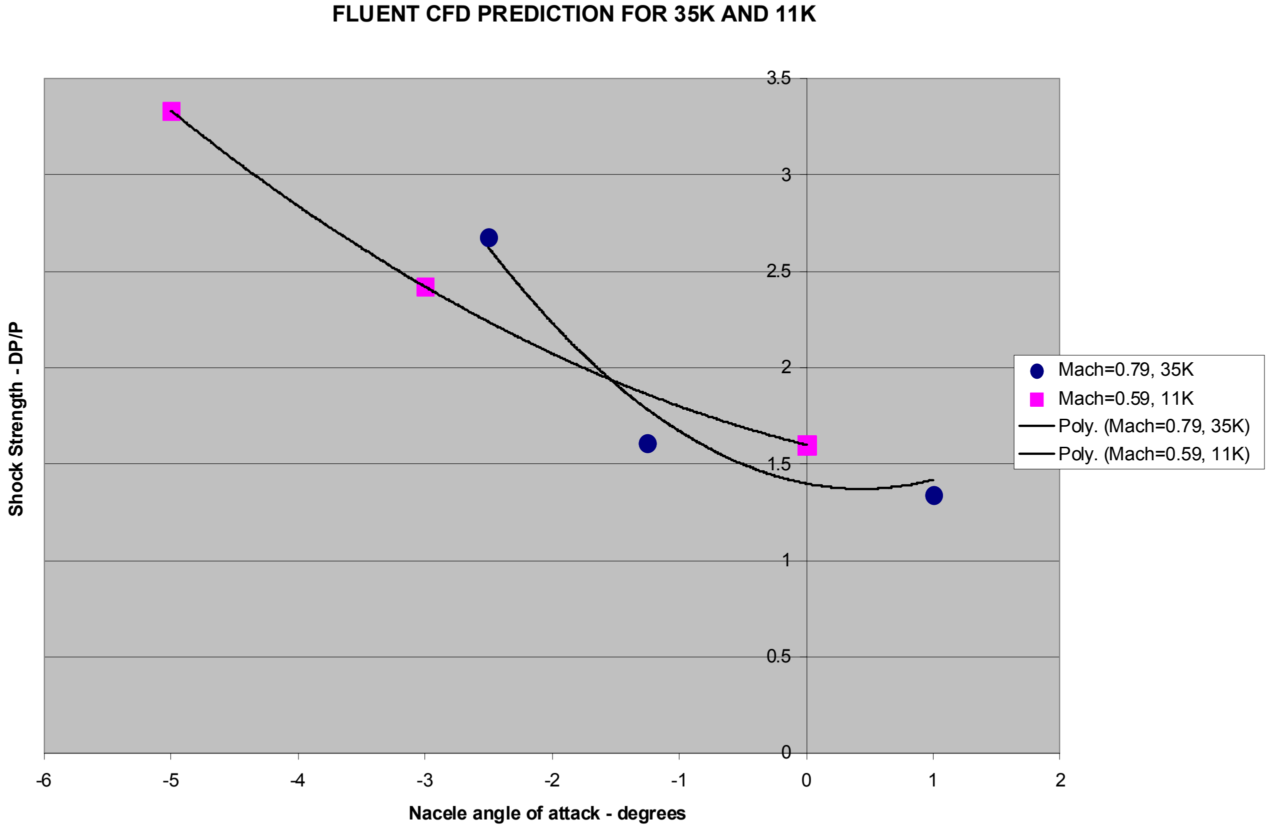


Figure 4: Sensitivity of shock to angle of attack

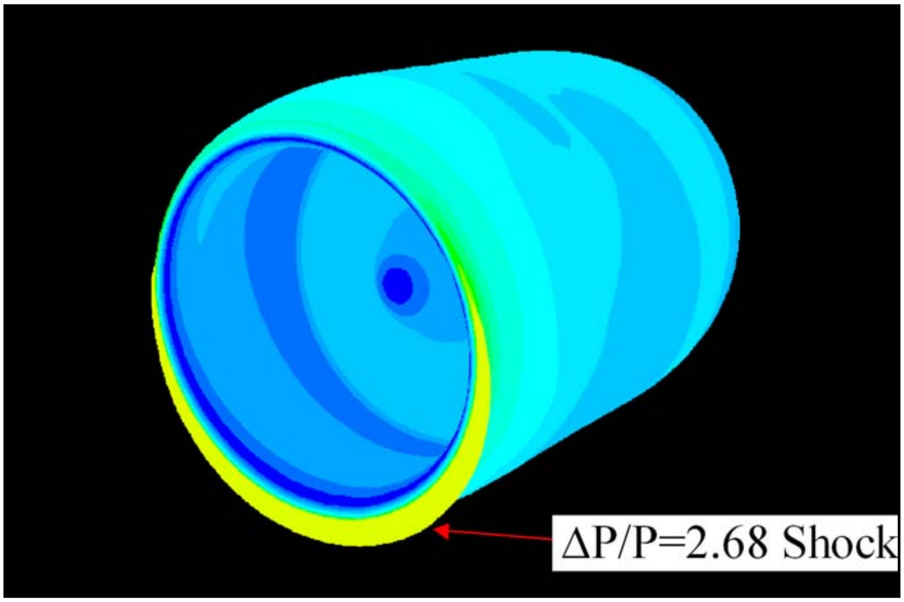


Figure 5: Contours of Mach number illustrate the shock for mid-descent case for generic nacelle: Mach 0.59, 11,000 feet, angle of attack 2.5 degrees

bed or on a test airplane. Then the airplane is flown through a matrix of flight conditions. The problem with this approach is that it may miss the flight condition at which a potential shock may occur or miss the exact location or strength of a potential shock. CFD can cover a much greater number of flight cases and engine mass flow, inlet angle of attack and altitude combinations, thereby homing-in to a successful inlet aerodynamic design far more rapidly and reliably than testing would ever allow. Flight testing is not eliminated but its cost and scope is significantly reduced.

Goodrich began using 2-D CFD to study the potential eventuality of transonic shocks within nacelles about 15 years ago. A CFD simulation provides fluid pressure, temperature, and density and other calculated outputs at the surface and throughout the computational domain for problems with complex geometries and boundary conditions. As part of the CFD analysis, an engineer may parametrically vary the geometry of the inlet or the boundary conditions, and observe the effect of the changes on the flowfield or pressure distributions. In the late '80s Goodrich (then known as Rohr Industries) Aerostructures engineers began working with two-dimensional Navier Stokes solvers and then moved to three-dimensional Euler codes. These packages are typically able to provide rough information on the locations of shock, thereby reducing the amount of physical testing required. In 1995 the company leased a powerful commercial CFD code, FLUENT, from Fluent Incorporated, Lebanon, NH, that makes it possible to more accurately model the nacelle inlet and transonic physics. With this code, engineers can, within limited amount of time and resources, generate fine hexahedral mesh models and run them to generate results that indicate the location and strength of the shock. They can then change the nacelle angle of attack, Mach number and inlet mass flow often without restarting the case, and determine the impact on shock strength and location due minor changes to flight aircraft flight attitude.

Simulation procedure

In this application, Goodrich engineers began by exporting a CATIA step CAD model of generic nacelle and inlet geometry, and importing it into Gambit, the FLUENT preprocessor. They meshed the geometry (Figures 1 and 2) using 3.6 million hexahedral cells with a high mesh density around and aft of the inlet lip. The mesh in the boundary layer

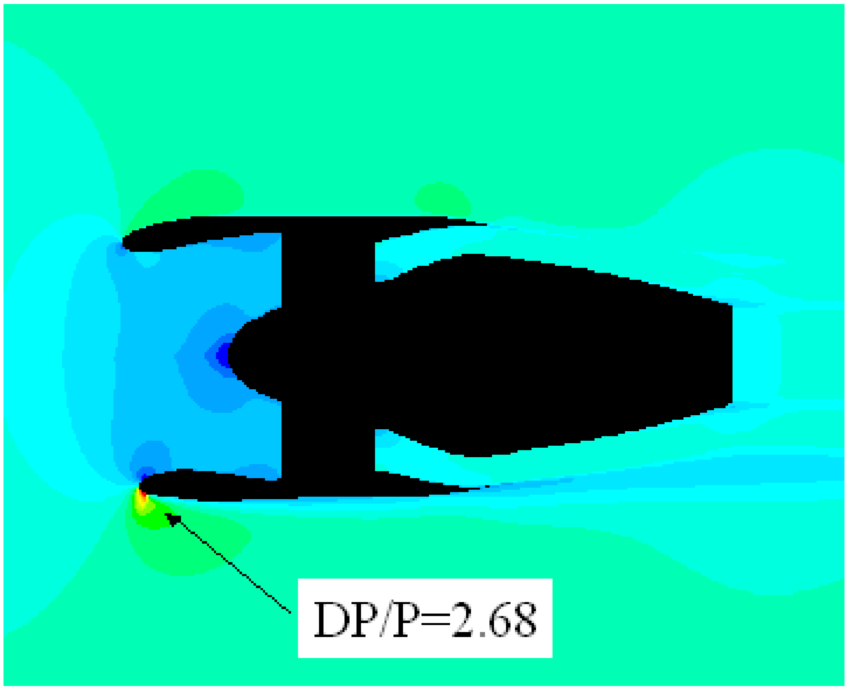


Figure 6: A 2-D cut view of the Mach contours, which clearly shows the presence of a shock on the lower inlet lip

Copyright © 2005 Fluent Inc.

region was sufficiently fine as to capture the viscous boundary layer effects. The mesh was then extruded very gradually so as to avoid any mesh size jumps. The aerodynamic analysis was performed using the realizable k-ε turbulence model with viscous heating enabled. Turbulent boundary layer effects were modeled using standard wall functions. Only half the nacelle was modeled taking advantage of a plane of symmetry about the x-z plane. Over 20 parametric cases were run with the 3D model to ensure inlet shock-free operation throughout the flight envelope. When a shock is discovered in an initial design the focus of the engineer is on its shock strength. The strength of the shock is generally calculated by taking the ratio of the static pressure difference across the shock wave to the upstream static pressure. As an example, the generic inlet studied revealed a potentially damaging shock at 11K, Mach 0.59 and AOA= -2.5 degrees angle of attack. Goodrich research and testing has shown that it is a in this particular flight condition that the inlet design must be carefully scrutinized. The following figure (Figure 3) compares the computed static pressure distributions at bottom dead center (BDC) on the generic nacelle at different Mach number conditions. The CFD data showed that the shock is strongest between 135 and 180 degrees. It rapidly diminishes in strength and in extent closer to the top centerline (TCL = 0 degrees).

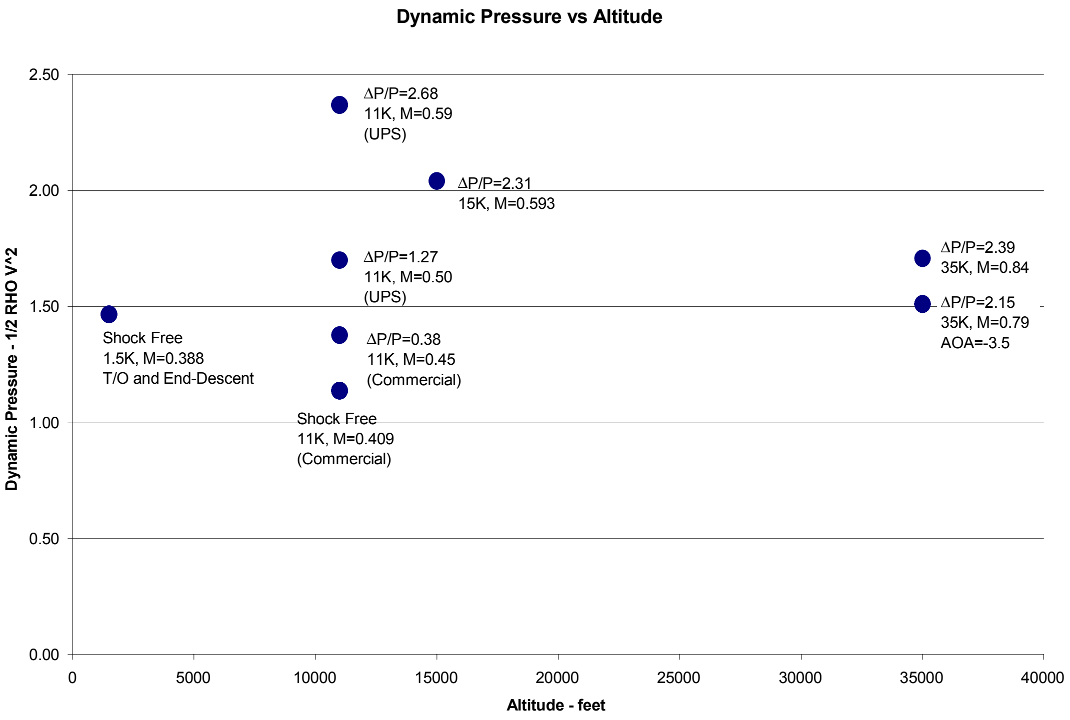


Figure 7: Dynamic Pressure vs. Altitude with associated shock strength. Note that some of the strongest shock strength occurs at the highest dynamic pressure/AOA combination

Figure 3 shows the variation in shock strength with circumferential location. The distance of the shock from the highlight and the shock extent versus circumferential location is also shown. The attached Figures (Figure 5 and 6) show the 3-D Mach contours and the shock strength and location. A 2-D cut view of the Mach contours which clearly shows the presence of a shock on the lower inlet lip (Figure 6) for normal descent conditions. Figure 8 shows a 2-D cut of Mach Number Contours for a cruise condition. This example illustrates how CFD may be used to reveal the potential presence of a shock on an inlet at the begin descent condition that otherwise would have gone unnoticed at the cruise design condition.

The same generic preliminary design inlet was also run at the typical 35,000 feet, steady and level flight, and maximum cruise power case. Although the untrained eye would dismiss this as a relatively benign shock, experience tells us that the begin descent case is more critical to the inlet structural design.

The shock circumferentially extends around the entire external inlet but rapidly diminishes in extent and moves toward the rear and top centerline. A split or discontinuity is visible in the shock wave at 45 degrees. The internal inlet shock wave spans the full circumference of the inlet and is located approximately at the inlet throat. The inner inlet barrel shock is strongest between 45 and 0 degrees with its peak strength at 0 degrees. This generic example illustrates how an inlet design that may have been suitable for engine cruise performance can

Copyright © 2005 Fluent Inc.

feature adverse aerodynamic characteristics at other regions of the flight envelope, in this instance begin descent, and should therefore be avoided. Inadequate inlet designs that do not meet the requirements at all regions of the flight envelope can be screened and identified using CFD in the early preliminary design phase.

The CFD results showed that the typical inlet shock is primarily caused by flow spillage around the inlet lip during the flight idle descent condition at slightly negative incidence. A more generous inlet lip contour as well as a larger nacelle may partially diffuse the shock and lower its strength to more acceptable levels but in general shock avoidance is recommended.

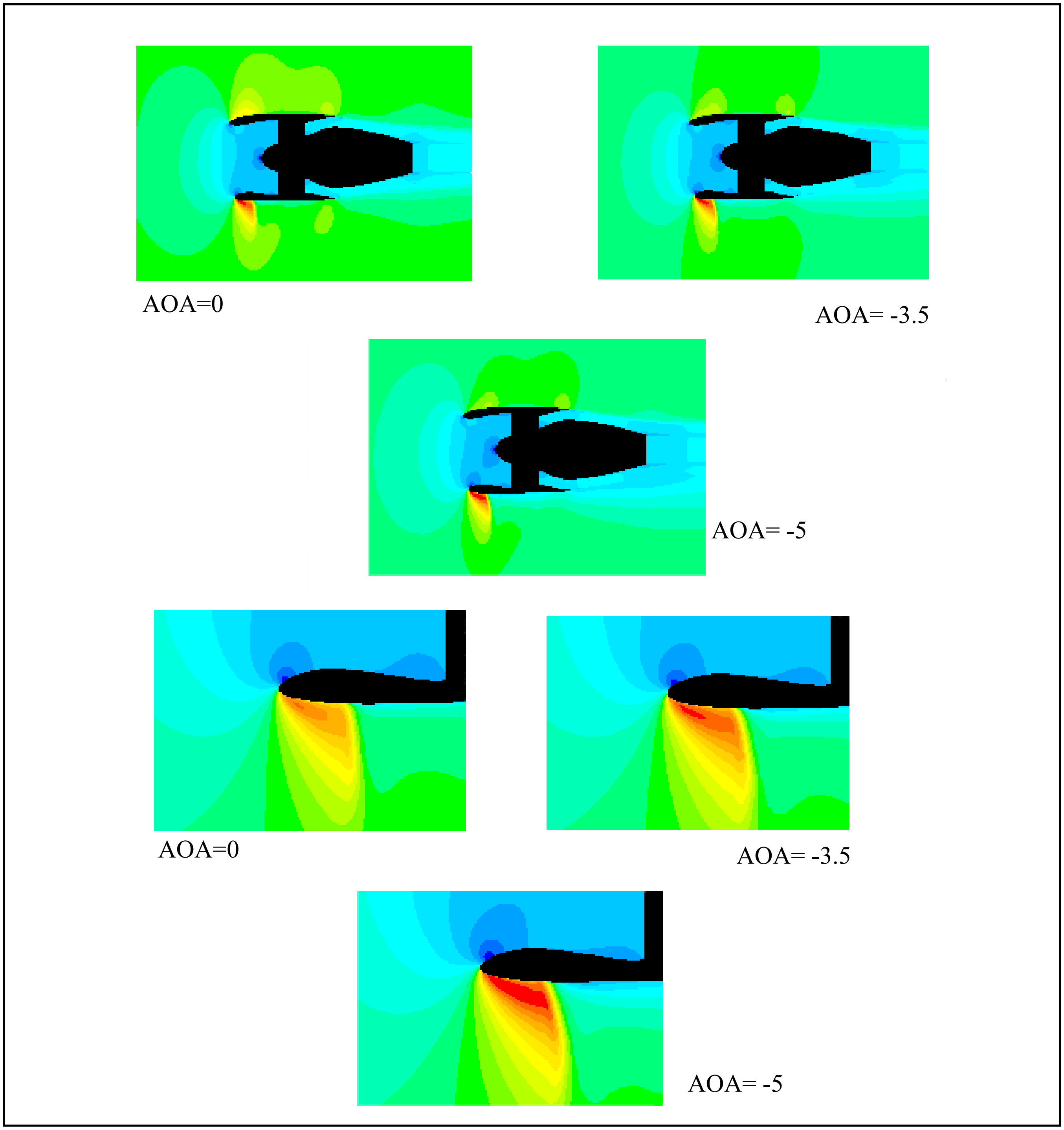


Figure 8: Contours of Mach Number on a 2-D plane for the Cruise conditions, 35K, Mach 0.8 at different angles of attack

Copyright © 2005 Fluent Inc.