RECENT PROGRESS ON POWERPLANT / AIRFRAME INTEGRATION AT AEROSPATIALE MATRA AIRBUS

Arnaud HUREZ
Aerodynamic Department, AEROSPATIALE MATRA AIRBUS
316 route de Bayonne (M0142/3) F-31060 Toulouse Cedex 03

Keywords: *aerodynamics, engine integration, nacelle, design process, wind tunnel test*

Abstract

Over the past 20 years, Powerplant / Airframe Integration has become an essential step in the design of any transport aircraft.

For the aircraft manufacturer, it is essential to reply to the program requirements in term of performance of the aircraft, to minimize the risks linked with the aircraft behavior in flight, and to reduce the costs and the development cycle. To achieve these goals, it is important for the Airbus partner in charge of powerplant / airframe integration to master the design of all related parts as pylon and nacelle aerodynamic lines.

This paper describes the experience acquired by AEROSPATIALE MATRA AIRBUS on their products, highlighting the recent progress at each step of the design process. It is developed around the A340-600 program and the new project A3XX.

1 Aerodynamics: the stakes for the aircraft

In the Airbus organization, the Aerodynamics Department of AEROSPATIALE MATRA AIRBUS (AM Airbus) has the responsibility of the design of the external shape of the pylon, of the nacelle external lines, of the nacelle air inlet and more globally is in charge of powerplant / airframe integration. The aerodynamic work is directly linked with some important stakes for the aircraft:

- to reach an optimum performance of the aircraft, in order to reply to its general specifications,
- to reduce at a minimum level, the risks linked with the aircraft behavior in the complete flight domain,
- to minimize the costs and the time cycle of the design process.

Aircraft performance improvement leads the engine manufacturers to increase engine bypass ratio. This results in larger nacelles which makes it always more difficult to both achieve the required cruise performances and comply with low speed requirements. Nacelle becomes a significant part in the overall aircraft performance assessment [1], requiring the designer to minimize isolated nacelle drag as well as to optimize installed propulsion system efficiency [2].

After a description of the aerodynamic aspects of powerplant / airframe integration, this paper describes the recent progress achieved at each step of the design process, and concludes on the replies brought to the 3 stakes above mentioned.

2 Aerodynamic aspects of powerplant / airframe integration

2.1 Global aircraft performance

2.1.1 High speed aircraft performance

The contribution of the engine installation to the total drag of the aircraft is not negligible (see Figure 1). It can be shared in two parts: the friction drag due to the wetted surface of the different components, and the interaction drag. It is mainly on this second part that the aerodynamic optimization can have a significant influence.

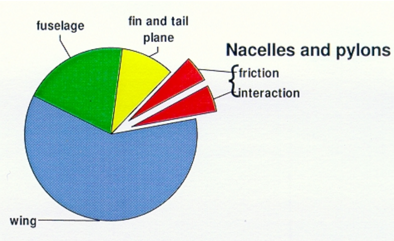

Figure 1: Typical airframe drag breakdown

The aerodynamic phenomenon at the origin of this interaction can be explained by the local modification of the flow leading to a substantial degradation of the local lift on the wing around the pylon, and so to a decreasing of the global lift coefficient (C_L) for a same angle of attack. To recover the necessary lift for the aircraft, this loss of C_L needs to be balanced by an increase of the angle of attack, with consequences on the wing shock and on the induced drag. The objective is to design an integrated set of lines (pylon, nacelle, wing) which minimizes these consequences.

2.1.2 High speed nacelle performance

The nacelle drag divergence Mach number must not be lower than the aircraft one, and sets drag variation limits within a specified range of mass flow and angle of attack covering typical cruise conditions.

2.1.3 Low speed aircraft performance

The low speed performance of the aircraft is linked with the efficiency of the high lift devices: flaps and slats. Due to the kinematics of deployment, and to the sweep angle of the wing, a large gap between the side of the deployed slat and the inboard side of the pylon may occur if no special attention is paid in the design. It is important for the low speed performance, and more particularly at take-off, to design an appropriate pylon shape to obtain a constant and small gap (see Figure 2).

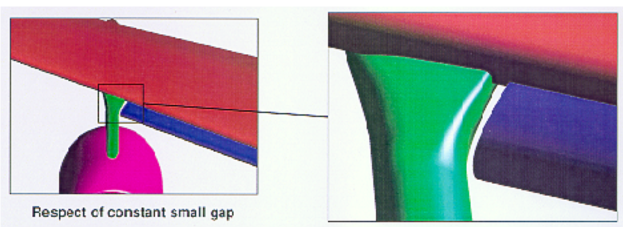

Figure 2: Gap between pylon and deployed slat

2.2 Operational risk

Localized aerodynamic impacts can have consequences on the flight domain limitation of the aircraft.

It is the case of the over-speed which is created on the lower surface of the wing on the inboard side of the outboard pylon. This over-speed is created when the flow skirts the pylon and wing leading edges, as the angle of attack decreases (low C_L). If this over-speed becomes important, it can induce a flow separation which is at the origin of excitations which can lead to wing vibrations, depending on aircraft damping. This phenomenon becomes more critical when the Mach number is more important and so can impact the flight domain of the aircraft at high Mach/ low C_L (rapid descent), reducing the airlines interest for the aircraft.

2.3 Aerodynamic nacelle requirements

These objectives, associated with sometimes conflicting requirements, ask for AM Airbus, who are in charge of powerplant / airframe integration, to master the whole nacelle lines definition. This is done either directly by the lines definition (air inlet, fan cowl lines), or thanks to close relationship with engine manufacturer (nozzle lines).

These nacelle flow lines must be designed to comply with various requirements and objectives, relative to both the aircraft and the engine design.

2.3.1 Aircraft specification

The inlet needs to satisfy engine mass flow demand within acceptable distortion limits throughout the aircraft operating envelope, at low speed. This is to be translated into high Angle of Attack, as well as inlet cross wind requirements. Taking into account the wing induced flow deflection, the aircraft low speed flight envelope translates into inlet high angle of attack operating conditions. In order to prevent adverse impact on wing maximum lift and on 2^{nd} segment drag, an other low speed requirement is to avoid inlet external flow separation at low engine mass flow (see Figure 3).

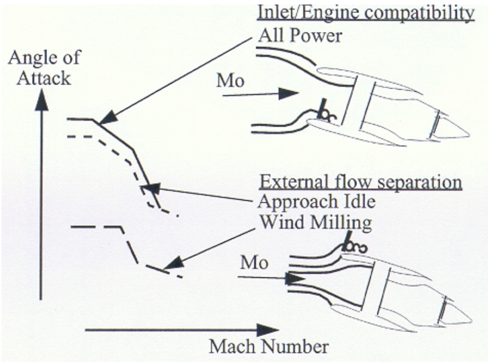

Figure 3: Low speed / high incidence aerodynamic requirements

2.3.2 Engine aerodynamic requirements

Engine requirements refer to engine cycle and performance. The inlet needs to satisfy engine mass flow demand within acceptable distortion limits (inlet/engine compatibility) throughout the whole aircraft and engine operating envelope. On the other side nozzle exhaust areas, in both direct and reverse modes are driven by engine cycle. Engine performance guarantees impose inlet pressure recovery and nozzle flow coefficients objectives (see Figure 4).

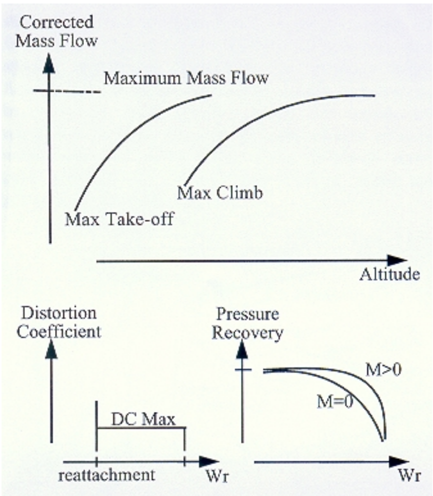

Figure 4: Engine requirements

2.4 Geometrical constraints

2.4.1 Pylon

The installation of an engine pod and a pylon under a wing is first driven by multi-disciplinary constraints as primary structure, mounts and systems to be enveloped in the pylon aerodynamic lines (see Figure 5).

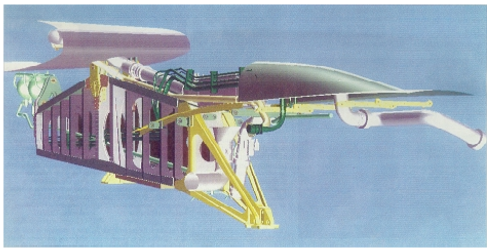

Figure 5: Pylon geometrical constraints

2.4.2 Nacelle

Nacelle hardware requirements refers to engine geometry, i.e. flange diameter or engine length, as well as location and size of equipment's which usually drives nacelle maximum cross section (see Figure 6). Acoustic treatment areas, dictated for noise attenuation, are also to be translated into geometrical constraints, as they may have an impact on both air intake and nozzle duct length.

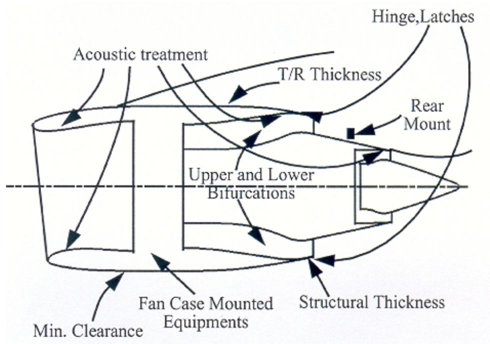

Figure 6: Nacelle hardware requirements

Nacelle flow lines must be compatible with structure design and manufacturing requirement, and are therefore strongly affected by nacelle technology. In addition, the design must be integrated with the rest of the aircraft. It is why nacelle flow lines are generally the result of extensive cooperation between Aircraft, Engine and Nacelle manufacturers.

2.4.3 Nozzle

Basically, a nozzle can first be regarded as a body of revolution. However one cannot ignore that real nozzle geometry is much more complex and, for mechanical reasons, substantial amount of nozzle flow is three-dimensional, as shown on Figure 7.

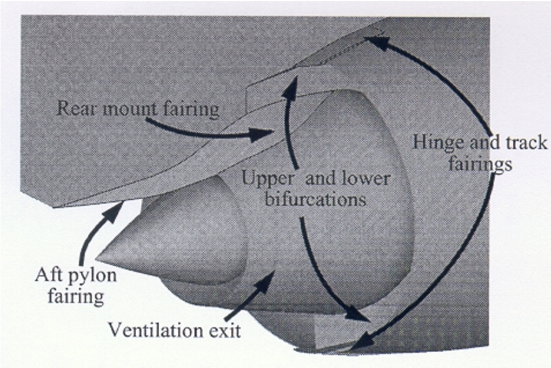

Figure 7: Nozzle and reverser fairings

3 The means: numeric and experimental tools

It can be seen that the quality of the design of the engine integration is essential for the global aircraft performance and for the minimization of the risks. It is why, to complete the experience of the aerodynamicists, a lot of efforts have been made in AM Airbus on tools in order to improve the design result, the time cycle and the costs.

The design process follows different steps which are:

- the aerodynamic shape definition in a Computational Aided Design (CAD) system,
- the mesh generation of the domain around the aircraft,
- the Computational Fluid Dynamics (CFD) analysis of the flow in this domain,
- the wind tunnel test of the most promising shapes,
- the flight test which "crowns" all the process.

These different steps are linked by an iterative process which is described on Figure 8.

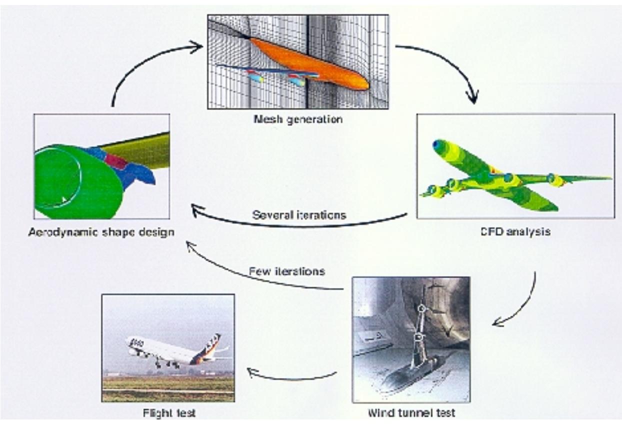

Figure 8: Design process

3.1 Progress in numeric tools

3.1.1 Surface generation

The use of a CAD system gives the possibility to take into account the numerous multi-disciplinary constraints above described.

The ICEMSURF software provides a lot of new possibilities for external aircraft surfaces. The main advantages of this tool are the global deformation allowing the aerodynamic designer to modify locally the geometry of the surfaces, keeping continuity with the neighboring ones, and the ability to analyze in real time the surface modifications.

3.1.2 Mesh generation

The industrial software used at AM Airbus is the result of a ten years co-operation with ICEM Technologies, and makes possible the 3D structured mesh generation of complex aircraft configurations. In parallel to this co-operation, AM Airbus developed their own mesh environment which allows to control, modify (geometry and topology), enrich (Navier-Stokes, adaptation) and publish the mesh.

Three creation modes are used for structured multi-block mesh design. With the interactive mode it takes less than 8 days to create the topology and the mesh around a completely new four-engine aircraft configuration. For similar aircraft geometries, automatic mode can replay an existing mesh on a new geometry (less than one day for a complete aircraft). For local surface modifications, immediate mode only projects the existing mesh done around first surfaces, on the new surfaces (less than one hour for a complete aircraft). The above described modes are used to generate meshes for Euler calculations.

Navier-Stokes (N-S) calculations are also commonly used now for Aircraft design [3]. It is still a challenging task to generate interactively a structured multi-block mesh for N-S calculation, especially to have a good control of node location in the boundary layer region. So AM Airbus have developed their own industrial software to convert fully automatically an Euler mesh into a N-S mesh. For example, the conversion of an Euler mesh around complete four-engine aircraft (1.5 10^6 nodes) into a N-S mesh (5 10^6 nodes) takes less than 1.5 hour.

For more specific studies (aerothermics), internal aerodynamic flow computation are now based on unstructured meshes generated with the required tetrahedrons.

3.1.3 CFD tools

The Computational Fluids Dynamics (CFD) is then used to analyze the flow around the complete aircraft or to modelize the flow in more precise areas.

AM Airbus use in an industrial basis, Navier Stokes / viscous Euler codes developed at ONERA and CERFACS. Their rapid response, even for complex geometry (complete aircraft), gives the possibility to reduce the design cycle up to one cycle per day.

3.1.4 Post processing

A post-processing tool named QUICKVIEW, developed internally, gives facilities for 3D phenomena representation on screen (see Figure 9). It is possible to investigate each aerodynamic values computed in all the domain with different types of representation (surface values, flow lines, cut in the domain, iso surface, 2D cuts …) with an interactive way of working, as the exploitation is very rapid, even for "big" results data.

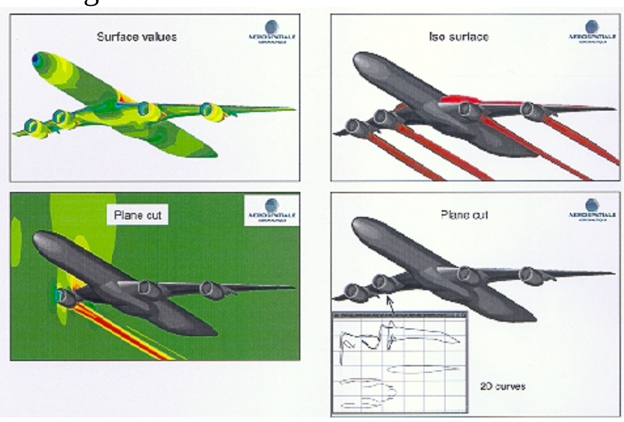

Figure 9: Typical QUICKVIEW views

In fact, this tool can also be qualified of "pre-processing" tool, as it gives to designer facilities to prepare the mesh, to prepare the calculation (boundary conditions definition), to follow the calculation itself (convergence). It is really integrated to all the design process, with possible return links towards surface generation and mesh tools.

3.1.5 Design environment

All tools used for numerical aerodynamic analysis (surface generation, mesh generation, CFD computation, visualization, post-processing) are activated by the Aerodynamic designer through an environment named AEROSTATION [4].

The architecture of this consistent work bench is compliant with CORBA standard for distributed objects. It allows communication between components of different process and management data tools. The complete traceability of the design process is ensured automatically, and the users can work together, sharing data, working methods, and JAVA scenarios.

This frame work will make easier the work done with partners, allowing the communication between the different tools and the management of data exchanges.

3.2 Progress in experimental tools

In order to validate the most promising configurations computed, the next step of the design process is the wind tunnel tests.

3.2.1 Air inlet tests

Low speed tests of air inlet are carried out at ONERA F1-Le Fauga wind tunnel, for both high angle of attack and crosswind performances. The corrected mass flow in the inlet is achieved by the flow between the high pressure air in tunnel and atmospheric air and controlled by venturis.

The model used for cruise speed tests at ONERA S1-Modane is the same one as in Le Fauga. The main objective of S1 wind tunnel test is: to measure inlet total pressure recovery in cruise conditions, and to measure inlet drag variation in a specified range of Mach number, Mass flow and angle of attack.

3.2.2 High speed drag of nacelle

In order to have access to absolute level of drag of the nacelle itself, a new kind of test is performed in ONERA S2 Modane wind tunnel.

The nacelle is mounted on a special long pylon, as it can be seen on Figure 10.

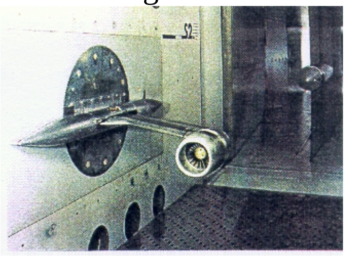

Figure 10: S2 set-up

3.2.3 Nozzle tests

After the optimization of aerodynamic surfaces, thrust bench tests are run so as to validate the performances of the nozzle (see Figure 11). These tests, can be performed either in Fluidyne (USA) or in ONERA Modane BD2 (France). Several configurations are usually tested in order to quantify exhaust area modifications (for example nozzle exhaust trim) effects on nozzle coefficients.

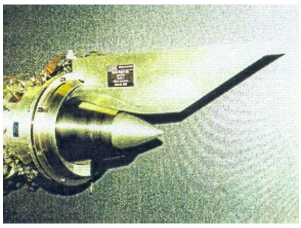

Figure 11: Nozzle test bench set-up

3.2.4 global aircraft performance

For a test dedicated to a global aircraft performance analysis in presence of nacelles and pylons, a full span model is used (see Figure 12) and the simulation of power-plant is made by Through Flow Nacelles (TFN).

Figure 12: Full span model of aircraft

3.2.5 Engine installation

In the case of a test dedicated to engine installation analysis, a bigger model is used (half model, see Figure 13), and a more precise jet representation is required. The exact representation of the jet effect on the complex flow behavior in the region of the pylon / wing junction can be made with a Turbo Powered Simulator (TPS) which is a little engine powered with high pressure air (see Figure 14). This device gives the possibility to have a real jet exhausting the nacelle and interfering the rest of the airframe. The difficulty resulting from this test technique is to have a conception of model with high pressure air coming through the wing and the pylon to the TPS. After a calibration on a static bench of the internal drag of this TPS (in relation with its mass flow rate), it is possible to accurately measure the interaction drag level.

Figure 13: half model for engine installation

In certain cases, AM Airbus also use simple but still representative TFN nacelles which are in fact triple body ones: with fan cowl, core cowl and plug (see Figure 14). It gives relevant and complementary results compared to TPS nacelles, with reduction of model complexity and so reduction of manufacturing cycle and costs.

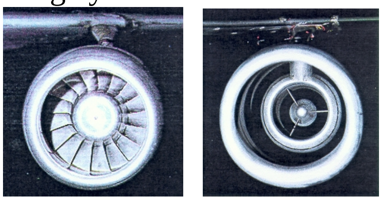

Figure 14: TPS and TFN triple body nacelles

3.2.6 Analysis of flow separation risk

As it was described in § 2.2, one of the potential problems can be the flow separation in the pylon / wing junction region. To have a qualitative idea of the phenomenon, flow visualization can be performed in wind tunnel with oil put on the model surface. But the precise understanding of this phenomenon can only be done with the quantitative values of excitation. This is now deduced from measurements in wind tunnel with unsteady pressure sensors "Kulite". AM Airbus have now developed dedicated experience in treatment of that information, which validates the aerodynamicists' design work in the process of risk mitigation.

AM Airbus master all these above described different tests, and the links between them, to have a global experimental analysis of aerodynamic phenomena.

4 Validation of results and respect of constraints

Before to see which level of performance and risk minimization has been effectively reached within the design process, it is important to validate the results, and to verify that the aerodynamic requirements are satisfied.

4.1 Global aircraft performance

As described in § 2.1, one of the goals of the aircraft manufacturer is to achieve a level of aircraft performance in line with the program requirements.

4.1.1 High speed aircraft performance

The flow around complete aircraft is first analyzed with CFD tools (see Figures 15).

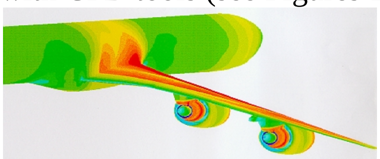

Figure 15: Euler results on complete aircraft

Then the results can be compared with wind tunnel results: a good agreement between the 2 kinds of results can be seen on Figure 16 on pressure measurements on wing.

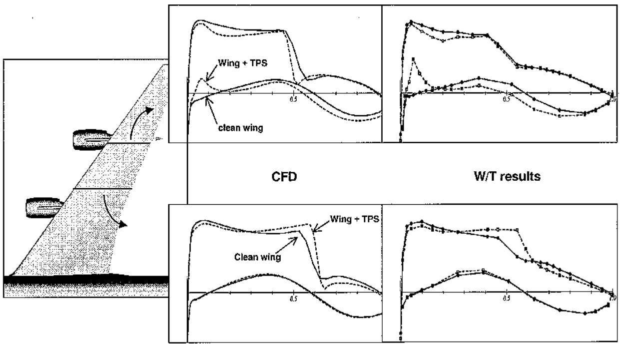

Figure 16: CFD and experimental results comparison

4.1.2 High speed nacelle performance

The nacelle drag divergence Mach number is predicted with an Euler code post-processing (see Figure 17).

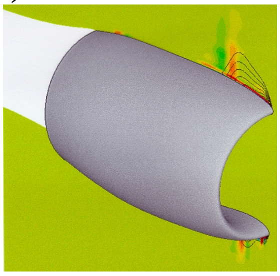

Figure 17: Isolated nacelle shock wave

It is possible to predict drag variation (Cx) with respect to Mach number (Mo), mass flow ratio (ε) and angle of attack (AoA). This good correlation with S1 isolated inlet test results can be seen on Figure 18.

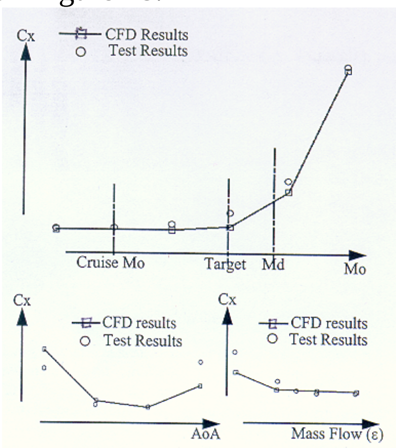

Figure 18: S1 isolated inlet drag results

4.2 operational risks

As described in § 2.2, an important risk for the aircraft is to have vibrations in flight domain, because of flow separation. The lower surface wing over-speed on the inboard side of outboard pylon can be at the origin of such flow separation (see Figure 19). It is why a special attention must be paid in the design of the shapes in this area.

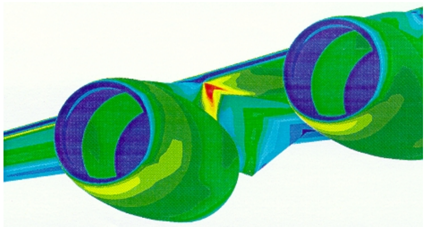

Figure 19: High Mach / Low CL Euler computation of over-speed.

Wind tunnel analysis based on oil visualizations (see Figure 20), and on unsteady pressure signal treatment (kulites) give the data necessary for the validation of the design.

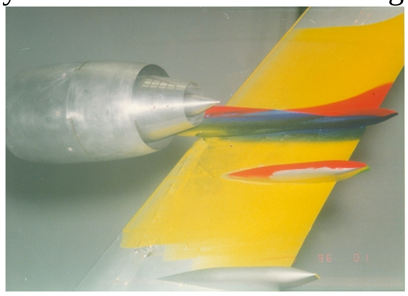

Figure 20: S1 oil visualization

4.3 Aerodynamic nacelle requirements

The 2 aerodynamic goals above described, have to be reached with some geometrical constraints, but also with some aerodynamic requirements, more particularly on nacelle behavior, as explained in § 2.3.

4.3.1 Inlet / engine compatibility at low speed and high angle of attack

For high angle of attack flight conditions, pressure distortion arises from thick boundary layer in the internal keel region. At a specified Mach (Mo)/ Angle of Attack (AoA) condition, maximum inlet flow capacity is limited by flow pressure gradient and shock that may induce separation. Inlet performance prediction (see Figures 21 and 22) results from viscous flow analysis, either Euler/Boundary layer coupling or Navier-Stokes modelisation.

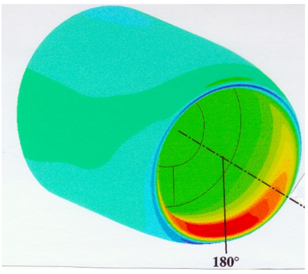

Figure 21: Local Mach number High incidence, high mass flow

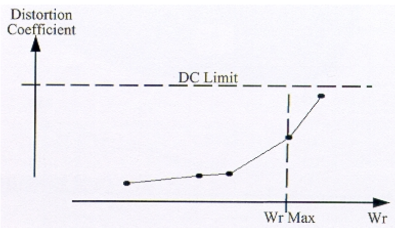

Figure 22: Inlet / Engine compatibility

Then, the final validation of the design (respect of specifications) is made using F1 isolated nacelle test results (see Figure 23).

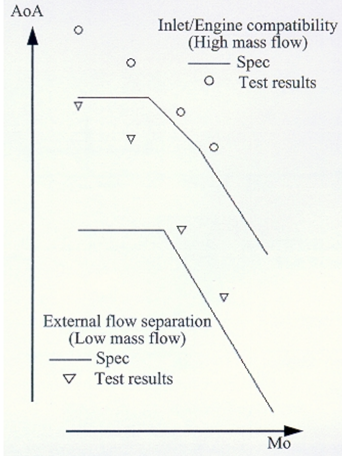

Figure 23: F1 results and specifications

4.3.2 Inlet / engine compatibility in crosswind conditions

Ground operations (i.e. take-off in crosswind conditions) results in the same internal flow analysis at the inlet side. However special care must be taken at low mass flow when adverse pressure gradient usually induce flow separation at typical crosswind conditions. Potential flow coupled with boundary layer analysis has proven to be a reliable design tool, substantiated by wind tunnel test experience. Figure 24 shows the local Mach number and the strong deviation between external flow and friction line induced by the adverse pressure gradient.

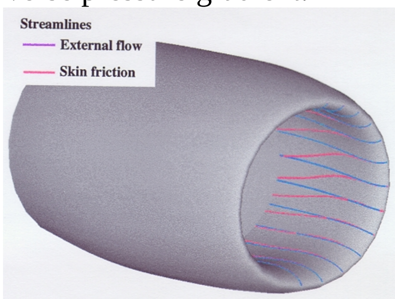

Figure 24: CFD in crosswind conditions

Total pressure recorded in F1 in cross wind conditions (see Figure 25) is in line with the specification, and clearly shows:

- Flow separation induced by adverse pressure gradient at low engine mass flow.
- Separation free air intake at intermediate engine mass flow demand.

- Distortion induced by shock strength at maximum engine mass flow requirement.

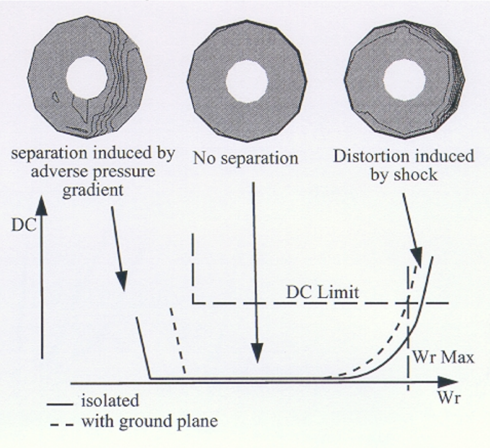

Figure 25: Distortion Coefficient (DC) versus engine mass flow (Wr) in cross wind conditions

4.3.3 External flow separation

Inlet external flow separation must be avoided at low engine mass flow, and at idle airflow (up to the aircraft maximum angle of attack encountered for certification testing). In these conditions, top stagnation point is well inside the inlet, with a high Mach number peak close to the top leading edge (see Figure 26). The mass flow ratio at the specified Altitude / Mach number /AoA conditions and the local curvature near the crown hilite have the strongest influence on inlet performance.

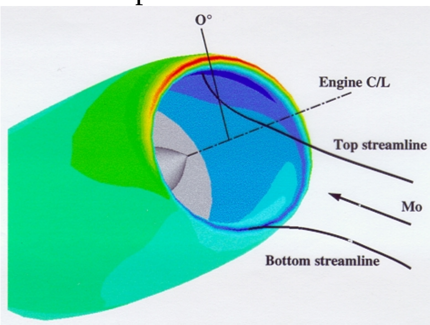

Figure 26: Local Mach number, High incidence, low mass flow

F1 test results (see again Figure 23) allow to demonstrate that the design meets the requirements.

4.4 Nozzle design

CFD codes used at AM Airbus have successfully demonstrated their ability to simulate nozzle flow and predict nozzle flow coefficients. Figure 27 shows Navier-Stokes flow computation results.

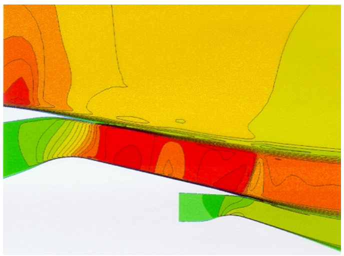

Figure 27: Mach number in the nozzle (Navier-Stokes)

Three dimensional CFD analysis is also carried to minimize impact of all the fairings on nozzle performance (see Figure 28).

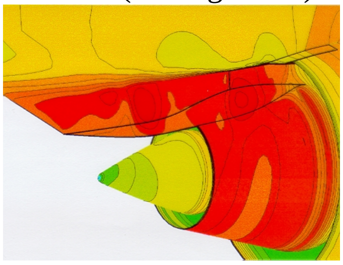

Figure 28: Euler results on 3D nozzle shapes

Figure 29 shows the good agreement between test results and industrial CFD predictions on these shapes

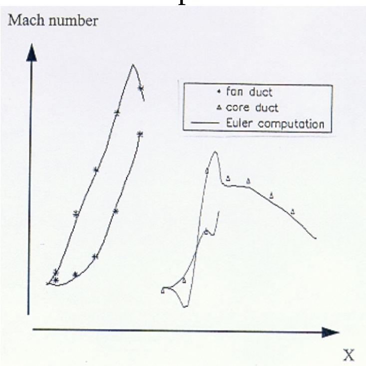

Figure 29: CFD and experimental results in fan duct and core duct

5 The reply to the stakes

Thanks to all the progress made in tools, and associated with expertise acquired on previous programs, it is possible to reach the aerodynamic goals:

5.1 Aircraft performance

The objective of all the work done, is to improve the performance of the aircraft, and in this case, to reduce the interaction drag due to the powerplant installation, by an optimum integration work. On A340-600, compared to A340-300, the engine installation drag, measured in wind tunnel at Mach 0.82, has been improved by about 30% (see Figure 30).

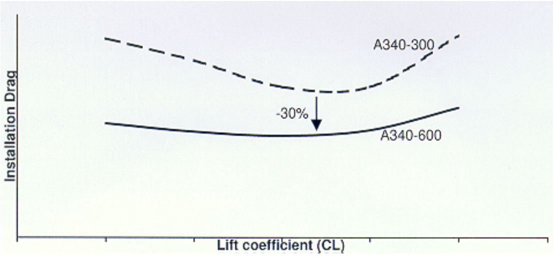

Figure 30: Installation drag of A340-300 and A340-600

5.2 Operational risk minimization

Wind tunnel analysis based on unsteady pressure signal treatment (kulites) has been used for the design of A340-600 pylon in order to push the risk associated with flow separation out of the flight domain (see Figure 31).

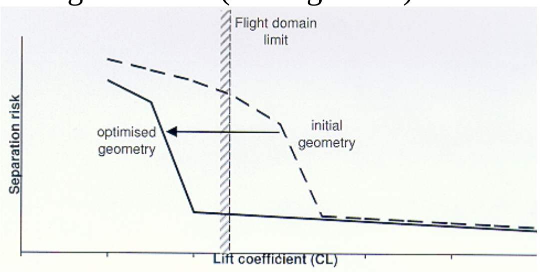

Figure 31: Separation risk control

5.3 Cost and cycle reduction

In order to reply to the previous technical requirements, but with an industrial logic, it is necessary to master the cost and the design loop cycle. A large improvement has been made for the A340-600 development, as, in comparison with A340-300 program, the design studies hours, the wind tunnel model manufacturing costs and the test hours have been reduced by about half (see Figure 32).

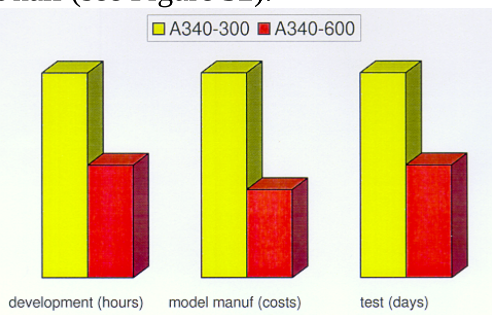

Figure 32: cost and cycle reduction

6 Conclusion

For some years, large progress have been made at each step of the design process in Aerospatiale Matra Airbus.

These improvements can be measured on the quality of design: aircraft performance and minimization of risks, but also on the reduction of design loop cycle, of wind tunnel tests hours and costs. The A340-600 development is already a good example of these improvements.

AM Airbus have also demonstrated their capability to design the Trent 500 nacelle lines for this aircraft in the short timescale required by this program, with validation in Wind tunnel that they meet engine and aircraft specifications in terms of low speed characteristics and high speed performances.

The new A3XX project facing the Airbus partners, is even more challenging (closer position, bigger engine, more rapid aircraft), and no doubt the processes and tools presented here will be put to good use and further enhanced.

Acknowledgments

The author would like to express its appreciation for the technical support provided by all the people involved in the AM Airbus Aerodynamics Department.

References

[1] Mazat E, Pelagatti O and Surply T. Design of A340-600/Trent 500 nacelle lines. *AAAF 7th European propulsion forum – Aspects of engine / airframe integration*, Pau, France, 10-12 March 1999.

[2] Hurez A. Recent advances on engine airframe integration at Aerospatiale. *AAAF 7th European propulsion forum – aspects of engine / airframe integration*, Pau, France, 10-12 March 1999.

[3] Gacherieu C, Collercandy R, Larrieu P, Soumillon S, Tourette L, Viala S. Navier-Stokes calculations at Aerospatiale Matra Airbus for aircraft design. *22nd congress of ICAS*, Harrogate, United Kingdom, 27 August- 1st September 2000.

[4] Casties C, Soulard A, Chaput E, Barrera L, Huchard J. Aerostation - Corba component approach to aerodynamic design framework. *22nd congress of ICAS*, Harrogate, United Kingdom, 27 August- 1st September 2000.