Relative roles of computational fluid dynamics and wind tunnel testing in the development of aircraft

S. S. Desai
Computational and Theoretical Fluid Dynamics Division, National Aerospace Laboratories, Wind Tunnel Centre, Belur Campus, Bangalore 560 037, India

Much is talked of Computational Fluid Dynamics (CFD), concerning its role in the possibility of bringing down the costs and turn-around times in the design and development of aircraft. Views have been expressed, at one extreme end, that wind tunnel testing may play a secondary role eventually. This communication takes a balanced view according to which both CFD and wind tunnels are inevitable; there are roles which are exclusive for CFD and wind tunnels and there are roles which are synergistic and complementary. Additionally, both CFD and wind tunnel testing are, after all, only tools whose optimality of usage is always as good as the experience and the relevant design database of the designer of the aircraft.

It is argued here that the efficacy of CFD in aircraft design is to be judged not so much by the number of wind tunnel blow downs that can be saved as it is to be judged by the value addition to the design cycle by its contribution to a superior baseline for final tuning and check-out of configuration through detailed wind tunnel testing and by the guidance obtainable through rapid CFD simulations for better utilization of tunnel tests.

Prior to the mid-seventies, aircraft design was mainly based on approximate theories of fluid flow, on engineering data sheets and relied heavily on vast amount of wind tunnel testing. Even then human ingenuity ensured superb designs such as the Concorde and the Jumbos. Over the last two decades, however, the discipline of Computational Fluid Dynamics (CFD) is fast changing this world of airplane design. A new tool is now available to the designer in the form of a variety of CFD software with varying degree of sophistication and accuracy of flow simulation.

The reputation which this vastly developed discipline has gained over the past decade tends to lend support to an utopian view that aircraft can henceforth be designed by the computer keyboard alone. The much-advertised statements in open literature about the CFD capabilities, may make one believe that the wind tunnels can henceforth be laid to rest in our efforts at designing and developing an aircraft. At the present time it is, for instance, natural for anyone to ask the question: is it really true? Or else, how much of wind tunnel testing have we saved as a result of the CFD efforts? These questions are relevant because of the tremendous support that CFD has enjoyed over the past two to three decades, with a promise of providing a new tool to the designer.

The answer to this question is not simple. While some enthusiasts of CFD might like to insist that sooner or later wind tunnel testing may have to play only second fiddle in the aircraft development, practical reality underscores the importance this wind tunnel testing has been playing, and is certain to continue to do so, in the aeronautical development activities; there are indeed areas of fluid flow which can only be handled effectively by wind tunnel testing. In this exclusive domain it is unlikely that our theoretical understanding of fluid flows, or of modeling them mathematically or their numerically simulation would ever ripen to a stage where wind tunnel testing will become unnecessary. At the same time, there are also areas which are exclusive for CFD. For example, CFD is extremely well suited in the evolution of configurations, particularly for optimized performance at a single design point, such as, for example, the cruise performance for a transport aircraft; the flow domains under such conditions are predominantly well behaved with dominantly attached flows which imply small boundary layer effects. This is an area where CFD finds itself at ease.

There are also areas where CFD simulations can be synergistically used with wind tunnel testing; with the help of detailed flow simulation obtainable by a CFD simulation, it is possible to organize flow diagnostics and metric elements optimally in any wind tunnel testing.

Finally, the role of experience and support from design data bank in the development of superior aircraft configurations cannot be ignored. It is indeed the experience which actually ensures an optimum utilization of the essential design clues, not only from CFD and wind tunnel tests but also from approximate theories, in order to bring about competitive designs within reduced cycle times and with cost benefits. Subtract this experi-
ence, and any of these sophisticated tools become ineffective.

**Essential elements in the aerodynamic development of aerospace configurations**

There are many areas of activity involved in the design and development of aerospace configurations. These include: aerodynamics, structural design and analysis, power plant and its integration with aircraft, system integration and flight testing. In this article, we concentrate only on the activities related to aerodynamic configuration design, which is the first and the foremost step in any aircraft design.

Figure 1 depicts the four elements that are essential in the design and development of any aircraft, or for that matter any aerospace configuration such as missiles, launch vehicles, etc.

It is to be noted here that it is the experience tempered by database, wind tunnel testing and CFD simulations that is at the root of any aircraft design. Experience is of considerable significance in *de novo* designs, and it dominates, particularly in all the derivative designs aimed at improving a given design, either to enhance its performance or to suit it best to the changing market/consumer requirements. It is once again *the experience* at Boeing that made it possible for CFD simulations to play a major role in the final design decisions in the development of the 777 airplane, pushing the role of wind tunnels into the ‘after-the-fact’ design validation.

The experience at Boeing made CFD possible in making the 777 airplane, pushing the role of wind tunnels into the ‘after-the-fact’ design validation.

Figure 1 also shows, by dashed lines, contributions to wind tunnel testing and to data sheets that are becoming possible in recent times owing to latest progress made in more realistic flow simulation through CFD. These were not available some 10–15 years ago.

**Exclusive roles of CFD and wind tunnel testing**

There are appropriate roles of wind tunnel tests and CFD simulations that are exclusive to each activity. The wind tunnels, like the flight tests, dominate the phase of design fine-tuning and design validation.

The edge of CFD simulations over wind tunnel tests lies in their ability to assess swiftly different configurations at one or two design points for better performance. In the pre-CFD era this step was taken by the designer through approximate theories, available proprietary data and through design data sheets as shown in Figure 2. In this mode, wind tunnel tests were not only used to validate the candidate design but also, in a limited way, used in the preliminary design stage of an aircraft. Refinement to a candidate configuration in the preliminary design stage through wind tunnel testing could be prohibitively expensive due to the model making costs and tunnel running costs. Because of both time and cost elements involved in wind tunnel testing, it becomes impractical to study several major configurational changes during the initial design freeze of the shapes. This could have prevented a superior candidate being frozen at the end of the preliminary design phase.

The laudable progress made in the ability of CFD simulations for complex 3D, real-life configurations now gives CFD the prime role in the fast design freeze of the candidate configuration (Figure 3).

It is to the credit of CFD developments that, for example, aerofoil analysis and design and optimization of wing–body combinations can now be carried out with a high level of confidence through CFD. One can now design an aerofoil for laminar or supercritical applica-

---

**Figure 1.** Elements in the design of aerodynamic shapes.

**Figure 2.** Typical aircraft design process during pre-CFD era. Note the essential role of the wind tunnels in the preliminary design phase of an aircraft. This role, essential in the pre-CFD era, is now taken over by CFD (see Figure 3).
tions instead of choosing them from NACA or other experimental databases which were the only source during pre-CFD era. Similarly, by an inverse design it is possible to arrive at, quickly during a couple of CFD simulations, an optimized shape to give some advantageous pressure distribution which is conducive to minimizing shock strength or avoiding stall originating at the wing tips or constraining the behaviour of pitching moment, etc.

The tunnel experiments, on the other hand, have a pivotal role to play in the validation of CFD software. For this purpose carefully planned and specific experiments would be required. There is still a pressing need for basic experiments to provide validational data for CFD in the area of flow stability, transitional flows, hysteresis effects and the vast area of turbulence and its modelling, and for three-dimensional boundary layers and flow separation characteristics.

The wind tunnels hold and continue to hold their supremacy in the design validation of configurations at the off-design conditions. The major issues that do not easily lend themselves to CFD analysis include: off-design conditions which are primarily dominated by medium or large-scale flow separations, flight Reynolds number effects, control-surface gap effects, control-surface effectiveness, installed power-system losses, combined internal and external flows and so on. Figure 3 shows the present-day scenario in which CFD now occupies an important step in the initial design stage. Here, not only can CFD reduce the cycle time in the final design cycle by minimizing the possibility of the changes to baseline, which are typically brought to the surface during proving flight tests, but it can also give the designer an advantage in cost and cycle times through the input of a superior baseline to begin the essential design process. In fact, it is in this manner that CFD has its major contribution.

Figure 4, adapted from Rubbert is more instructive in bringing out the relative roles of CFD and wind tunnels. What is significant is the fact that in a typical aircraft design, CFD may be able to contribute through mere hundreds of simulations; beyond this the cost and time for simulations would be prohibitive. The wind tunnel on the other hand, has to cater to the requirement of millions of aerodynamics simulations that are invariably essential to validate any design before fabrication of the prototypes. It is to be noted here that one simulation consists of one free stream Mach number and one incidence for one geometry configuration. Figure 4, a typical experience of aircraft companies, brings out the essential role of CFD, more importantly in the initial design phase through a small number of flow simulations where it is cost and time-effective, whereas the wind tunnel is expensive in an iterative process for the choice of the baseline configurations. After the initial time and cost of wind tunnel model fabrication, the wind tunnel is more expedient for the large number of simulations typically required in any aircraft design.

Figure 3. Changing world of airplane design with the emergence of CFD as an important design tool, particularly in the preliminary design phase.

Figure 4. Cost and flow time characteristics of wind tunnels and CFD. It is noteworthy that the cost/time of CFD simulations are not affordable beyond 10 s and 100 s of simulations. With the availability of a model and tunnel time slot, wind tunnel simulations are much cheaper.
Areas of uncertainty in flow simulation which are common to both CFD and wind tunnels

In a sense, neither CFD nor wind tunnels are perfect as far as simulation of realistic flow fields is concerned. This is understandable. Not only does each one of them have its own drawbacks, there are also areas common to both CFD and wind tunnels where there are questions of difficulties and of their accuracies. These areas are:

- High Reynolds number flows – Simulation of full-scale Reynolds number \( Re \) of the order of 20–40 million is unusual and out of the question for wind tunnels for reasons like cost of building and operation of the facility. CFD simulation has also a lot of uncertainties here because of grid and turbulence model limitations.
- Drag evaluation – Drag evaluation within 5 to 10% accuracy is what is typically possible in both CFD and wind tunnel simulations.
- Model support and wall interference in wind tunnels and influence of outer boundaries in CFD – These are elements which contribute to interference effects.
- Flow at large incidence; \( C_{\text{max}} \) for a wing, for example – Unduly large blockage, wake and tunnel-wall effects combined with scale and free stream turbulence effects result in uncertainties in tunnel measurements. In CFD, modelling of large-scale separation is still in a nascent state; lack of turbulence models is a big hindrance.
- Gap effects of control surfaces – \( Re \) (scale) effects in gaps can give misleading results in wind tunnel testing. Also, model inaccuracy is a contributing factor in the small sizes of models that are practical for tunnel testing.
- Confluent boundary layers and internal–external flows – Close geometric simulation of internal and external flows is a herculean task in tunnel testing.
- Resolution of viscous effects – Effects of free stream turbulence in tunnels and of turbulence models in CFD simulations contribute to simulation errors.
- Power effects – This poses considerable complications in tunnel testing and is not amenable to CFD.
- Implementational difficulties – The elements of cost and time of model making and building a tunnel facility, its maintenance and difficulty of availability of trained manpower to man the tunnels (in this dominantly Information Technology era) are a big deterrent to tunnel testing. Similarly, the CFD software is hindered by the necessity of long hours of set-up time and of longer apprenticeship needed to develop adequate familiarity with limitations, range of applicability and with capabilities of a CFD software.

Advantages in CFD simulations

The CFD techniques are particularly effective in the following efforts:

i. Aerofoil design and analysis, and optimization of wing–body shapes.
ii. Deciphering in a short time the merits and demerits of several variants considered for a baseline. A few simulations at one design condition can reveal merits of one variant over the other.
iii. Detailed flow field information. There are times where flow field information in specific locations over the configuration may be required (For example, the flow distribution at the inlet plane of an engine.) This is not only too cumbersome in wind tunnels but also, often impractical.
iv. Quicker and less expensive changes to configuration and their numerical simulation.
v. Smoothing of the configuration to reduce pressure-drag levels.
vi. Quick checkouts during after-design stages, where it is not possible to carry out wind tunnel tests for want of timely tunnel slots or logistics.
vii. Assessment of incremental changes quickly in the configuration.
viii. Design of air-data system and determination of suitable location for air-data probes on a given aircraft.
ix. Hypersonic flow and chemical/rarefied-gas effects, for which experiments are impossible to be done.

Advantages of wind-tunnel testing

i. Accurate data generation for validating CFD simulations. CFD has considerably benefited from this role of wind tunnel simulation.
ii. Simulation of complex shapes where grid generation and accuracy of flow simulation pose definitive problems in CFD.
iii. Simulation of separated flows, typical of large incidence aerodynamics. Here limitations of accuracy of turbulence models is a big hindrance in CFD simulations. This area would be dominated by wind tunnels for a long time to come.
iv. Simulation of viscous effects in corners and over gaps in control surfaces.
v. Expediency in store carriage and separation studies.
vi. Simulation of power effects, rotor flows and multi-component flows with relative ease.
vii. In one tunnel-run lasting for less than a minute it is possible to obtain 15–20 simulations, whereas in CFD simulations one run of the code generates only one simulation.

Areas of individual superiority for CFD and wind tunnels

There are areas in flow simulation where wind tunnels and CFD have their own advantages.
Questions of CFD code validation

This is a truly difficult issue. It is to be constantly kept in mind that CFD tools cannot be used as a black-box by anyone without much exposure to the nature of CFD, its capabilities, its weakness and pitfalls that are likely to arise in any blind use.

CFD is presently a fairly mature tool for the analysis and design of 2D shapes and in particular aerofoils. A variety of software can be used for this purpose: transonic full-potential codes with boundary-layer correction, for both analysis of viscous flows even close to stall, or unsteady flows. CFD tools can also be used with fair amount of confidence in the analysis of wing–body shapes or in their optimization. Simple wing-alone cases are amenable to reasonably good CFD simulations.

Extremely complex 3D, real-life shapes continue to pose, of course, considerable challenge to CFD tools. So are also flow regimes dominated by separated viscous flows, confluent viscous flows over multiple components, etc. Even then, it is possible to grossly assess the configuration. Here the panel codes, despite being only limited to inviscid flow simulations, have been the workhorse in the aeronautical industry. The panel codes require simple surface panels and are not impeded by the requirement of three-dimensional volume grids.

It is important to realize that even when quantitative accuracy is not guaranteed for all the flow parameters, the CFD codes have still a role to play. For example, it is, even to date difficult to predict drag accurately from any of the CFD codes, including the most sophisticated ones. However, overall lift characteristics may be reliably obtained from many of the CFD codes. Similarly with some care overall pitching moment behaviour – for example, stable or unstable behaviour – can be surmised by a few other CFD codes (Figure 5). Many RANS codes, for instance, can predict the quantitative pitching moments and hence also the detailed pressure distributions for two-dimensional flows reasonably well, particularly when the configuration and the flow are not prone to large viscous effects or flow separations.

A hierarchy of flow simulation codes is therefore essential for an effective functional use of CFD in any vehicle development work. This is not in any way different from what is true with wind tunnel testing; specialized tunnels are indeed necessary for specific tasks: water tunnels for flow visualization, particularly for vortex-dominated flows; specialized quiet tunnels for laminar-flow investigations; special low-speed tunnels for high-lift investigations; 2D-tunnels for transonic aerofoil analysis; large low-speed tunnels for power-effects and full high-lift configuration, special tunnels for rotary flows and so on.

What is to be specially emphasized here is the following observation: CFD is a tool and the efficacy of its use depends on the user; it is equally possible to obtain wrong results with sophisticated codes as it is possible to obtain smart simulations that help in design decisions from simpler, less-sophisticated codes. Many a times special features may not leap to the eye from computed details of flow, but one needs to closely examine the detailed flow fields simulated, for example, the pressure contours, the surface or field velocity vectors, etc. It is

---

Figure 5. Results for a SARAS wing–body configuration from wind tunnel experiments at NAL in comparison with numerical simulations with an Euler and a panel code from the CTFD Division. This kind of comparison justifies the useful role that different levels of approximations of the Navier–Stokes equation play in the preliminary design phases in the aircraft development.
then possible to choose from an examination of inviscid surface pressures, streamline or velocity vector distributions, those that are less prone to flow separation, formation of strong shocks or favourable pitching moment characteristics, etc. Experience is the key to success in such studies.

**Synergistic use of CFD and wind tunnels**

A synergistic use of CFD and wind tunnels is possible and can be gainfully employed in the understanding and designing of complex hardware such as an aircraft, where any one approach may not yield all the answers. It is also essential to mix the use of many of the novel approximate theories of fluid flow. With due experience a synergy of all these can be utilized to bring out a competitive product.

It is well known that it is actually the wind tunnel tests carefully carried out that provide the background data for validation of any numerical simulation software.

An example is given here, where it was the CFD simulations that helped refinements to certain traditional practices in wind tunnel testing. Till about the early 80s, it used to be a common practice to use a large open area ratio (see Figure 6a for definition) for the transonic testing of aerofoils within slotted/perforated wall test sections. At NAL we were routinely using 8% open area ratio. Figure 6b shows results obtained in a 0.3 m transonic tunnel at NAL for the pressure distribution over a standard BGK (Bauer, Garabedian and Korn) aerofoil. The shock position was obtained at around 20% of the chord for a particular Mach number and incidence (Figure 6b). It may be noted here that the accuracies in model making, the dimensional proportions of the model to the tunnel height (typically, a ratio of chord to tunnel height less than one-third) and the measurement techniques were accurate and consistent with recommended test practices for two-dimensional tests. Moreover, the results were perfectly repeatable. Because of this, one could not have suspected their accuracy of simulation of flow over a given aerofoil.

But then, just around that time it became possible for the first time to obtain numerical simulation of viscous transonic flow over this aerofoil by CFD techniques. The free-air shock location was predicted to be around 65% for the same Mach number and incidence in the numerical simulation. This created panic and was the basis for a CAARC (Commonwealth Advisory Aeronautical Research Council) programme involving India, Australia, Canada and England to investigate the role of the open area ratio in governing the magnitude of blockage interference. It was possible to conclude that most of the tunnels were operating under too open conditions (India 8%,

![Image](image-url)
Australia 20%, England 5%, Canada 12%), leading to vast tunnel interference of open-jet type. Subsequent tests\textsuperscript{16} at NAL established that an optimum open area ratio is indeed around only 2% (see Figure 7 a). As a result of this investigation the importance of the role of the slots or perforations in the tunnel walls was realized for the first time. Indeed, Figure 7 b brings out the dramatic effects of the open area ratio on the shock position for a BGK aerofoil, in specific tunnel tests\textsuperscript{15} which were undertaken to bring out the role of the open area ratio. It is essential to note here that the results with open area ratio of 3% and 8% are actually distorted by tunnel boundary conditions and can never be simulated (see Box 1) by CFD with free-air boundary conditions. All the 2D tests are now done only around this reduced open area ratio\textsuperscript{16}. Figure 8, taken from ref. 15, demonstrates the kind of influence the open area ratio of the tunnel walls can have on the shock location over the much-studied NACA 0012 aerofoil at different free stream transonic Mach numbers at zero incidence.

This is one example where CFD has actually guided and suitably corrected the routine practice in experimental simulations.

An example is now given where experiments and CFD are used together to obtain important design data. This example is taken from Rubbert\textsuperscript{17}. In sizing the control surfaces one should have an idea of the control effectiveness. For an aft-tail configuration, control effectiveness of the elevator is of critical importance. This information is not easy to obtain in wind tunnels because of inadequate Re simulation; neither is it reliably obtainable from CFD simulations owing to difficulties of geometry (gap effects particularly), turbulence modeling and grid generation appropriate to the flight Re. It is however possible to obtain fairly easily inviscid simulations, which correspond to a Re of infinity, by using panel methods. Figure 9 adapted from Rubbert\textsuperscript{17}, combines the infinite Re results with wind tunnel results at two tunnel Re values in a plot of control effectiveness versus inverse of Re. From this graph it is then possible to read off the control effectiveness for the required flight Re.

**Some examples of the use of CFD at NAL\textsuperscript{18}**

It is to the credit of the development in CFD that the software tools developed over the past two decades have found acceptance in many aircraft development activities all over the world. In India, CFD tools are finding increasing use in several design and engineering organizations which include: Hindustan Aeronautics Ltd, Aeronautical Development Agency, Defence Research and Development Laboratory (DRDL), Naval Science and Techno-

![Figure 7.](image-url)

**Figure 7.** Dramatic role of open area ratio in influencing the aerofoil (BGK) characteristics. Results at 3 and 8% open area ratio are distorted by wall interference and can never be simulated by free-air, far-field conditions in a CFD simulation. 
\textbf{a}, Experimental results with 2% open area ratio in the 0.3 m transonic tunnel at NAL. Note the shock position at 65% of chord. Current practice is to use an open area ratio around 2%. \textbf{b}, Dramatic variation in the nature of pressure distributions and in particular shock location as function of open area ratio.

<table>
<thead>
<tr>
<th>Open air ratio</th>
<th>Shock position</th>
</tr>
</thead>
<tbody>
<tr>
<td>8%</td>
<td>20% C</td>
</tr>
<tr>
<td>3%</td>
<td>42% C</td>
</tr>
<tr>
<td>2%</td>
<td>65% C</td>
</tr>
</tbody>
</table>
logical Laboratory, Indian Space Research Organization, etc. Amongst the academic institutions, mention may be made of all the Indian Institutes of Technology and the Indian Institute of Science, Bangalore, where CFD is being used in the analysis of flow fields in research and in sponsored support to other organizations. An overall scenario of status of CFD in India is covered in Desai.

A few examples are presented briefly just to give a flavour of the latest applications of CFD drawn from experience at NAL in the aeronautical and the non-aeronautical domain.

**Aircraft projects**

The Civil Aircraft projects, viz. HANSA and SARAS, at NAL have provided opportunities for the CFD to demonstrate its utility in aircraft design and analysis.

The biggest challenge in aircraft design is to minimize adverse interference effects of one component on the other; obviously, putting together two individually well-designed components may not assure a superior design. This is evidently the case in the mutual interferences of wing and body; independently optimized wing and body may not offer the best wing–body combination.

Figure 10 shows the HANSA configuration as put together by the designer using conventional wisdom. In a CFD simulation, this configuration demonstrated, as in wind tunnel experiments, poor flow aft of the crest of the fuselage and over the boom. Subsequently, a CFD study was undertaken using a panel code to finalize a shape that ensured well-organized flow over the aft fuselage and boom of the configuration. The modified configuration is the one that went through prototype fabrication, flight testing and final type-certification. What is significant is that no wind tunnel tests were carried out after the modifications suggested by CFD.

CFD was again used in the design of wing–body combination for the SARAS aircraft. Figure 11 shows computed streamline pattern for the ‘as-designed’ shape. Modifications were carried out to the wing–body interference region and to the aft-fuselage near the stub wing using an Euler code in an analysis mode repeatedly, to ensure smoothness of streamlines over the configuration. In another representation, Figure 12 shows the isobars by Euler computations for the two wing–body shapes. As can be seen, the isobars smoothly merge into the body isobars in the modified shape.

Figure 13 shows results of Euler computations for the wing–fuselage combination pertaining to the MiG-21 aircraft. This geometry was created from the production drawings and was analysed for pressure distribution at transonic Mach numbers to provide load data for studies.
on fatigue life and life-extension to MiG aircraft undertaken at the Structural Integrity Division (SID), NAL.

Figure 14 shows the overall force, drag and moment coefficients obtained by the numerical simulation using a $Re$-averaged Navier–Stokes solver for the aerofoil used on HANSA aircraft for the entire range of incidences, up to and beyond stall$^{22}$. Comparison with wind tunnel data appears to be quite good for lift and pitching moments even up to and beyond stall. It is remarkable, as seen from Figure 14, that such a simulation is possible with CFD now. This comparison is further strengthened by the comparison of pressure distributions shown in Figure 15.
What was most essential from an application point of view in one instance was information on the movement of the front stagnation point as a function of the aerofoil/wing incidence. Figure 16 shows the data obtained for the HANSA aerofoil. This information, useful in the location of a stall warning system, cannot be easily obtained in a wind tunnel simulation.

Figure 17 shows the results of an effort in which CFD was used to reduce the areas of pressure drag of SARAS aircraft by local geometric changes\(^1\). It is believed that this exercise is valuable and could bring about a reduction of up to 5% in pressure drag. But such local changes are impossible to be studied expeditiously in a wind tunnel simulation.

In the hypersonic flow regime where the wind tunnel simulations begin to become impractical, design help will come increasingly from the CFD simulations. Figure 18 gives the density contours at different sections of air inlet of a Hypersonic Research Vehicle\(^2,3,4\).

Non-aircraft applications

The present-day scenario in project execution is marked by a tendency of the investigators to seek alternatives to tunnel testing. Time and cost of accurate wind-tunnel model design and fabrication, availability and timeliness of tunnel test schedules normally persuade investigators to seek CFD solutions as an alternative.

Figure 19 shows the numerical simulations from a low-speed RANS code\(^5\) for the tethered balloon under a sponsorship. The pressure distributions and the subsequent load information obtained by CFD were the direct inputs to the structural analysis groups at NAL.

In another application, information on pressure distribution and loads was generated for a 13-m diameter weather radome\(^6\) that was to be designed and built by NAL for a sponsor. Figure 20 shows the pressure distributions obtained on the spherical radome mounted on a tall building. The simulations not only provided information about overall load and wind pressure, but also about the level of truncation of the spherical radome to improve the lift-to-drag ratio for the radome. This increased lift-to-drag ratio helped in the structural design of the radome. What is noteworthy in this effort has been that no wind tunnel tests were conducted nor contemplated.

![Figure 11](image1.png)

**Figure 11.** Streamline patterns from Euler computations: wing–body (SARAS) (a) as designed, (b) after modifications suggested by CFD. Notice the smooth pattern of streamlines on the fuselage around and downstream of the wing region.

![Figure 12](image2.png)

**Figure 12.** Isobars from Euler computations: wing–body (SARAS). Notice the smooth merger of wing isobars with the isobars over the fuselage around the wing region.
That is the kind of confidence one can place on CFD simulations; of course, this level depends on the geometric complexity of the shapes and also on the speed regimes of flow.

**Future trends in wind tunnel testing**

Figure 21, adapted from open literature, brings together the tunnel testing requirements for some of the well-known commercial airliners. For example, typically for the development of Boeing 747, wind tunnel hours of the order of about 10,000 were required during the late sixties and the early seventies. The trend in Figure 21 shows a continuous increase in the demand for wind tunnel testing up to mid-eighties. In Figure 21 we have indicated...
Figure 15. Surface pressure distribution for GA(W)-2 aerofoil — N-S computation; OOO, WSU tests; ΔΔΔ, NASA tests. These good comparisons for the pressure distributions are actually behind the good comparisons for pitching moment up to and beyond stall shown in Figure 13.

Figure 16. Stagnation point location for GA(W)-2 aerofoil ($M_\infty = 0.3$, $Re = 2 \times 10^6$). This information may not be obtained easily in wind tunnel tests.

Figure 17. Levels of pressure drag on SARAS aircraft.

Figure 18. RANS computations at Mach 6.5 – density contours at different sections of air inlet of a Hypersonic Research Vehicle.
the possible reduction in the wind tunnel test requirements due to CFD support subsequent to 1985. It is perhaps not unreasonable to realize that even with full support from CFD capabilities the number of hours of tunnel testing may not come down below about 5000 h, for a new design. Of course, for a derivative design, past experience can help in bringing about a drastic reduction in the requirements for tunnel testing.

It is useful to go back to Figure 4 where it is mentioned that a typical aircraft design would entail something like 2.5 million simulations. It is also mentioned that some 10 to 100 CFD simulations might add considerable value to

![Image of aerostat with fins and streamlines](image)

**Figure 19.** Turbulent flow around an aerostat with fins ($Re = 1.5 \times 10^7$).

- Multiblock RANS computation on parallel computer using $300 \times 82 \times 88$ (approximately 2 million) nodes with H-O grid topology.
- Realistic surface pressure distribution and close agreement between measurement data and RANS prediction are observed for $\alpha = 20^\circ$, $\beta = 0^\circ$, $\text{Roll} = 0^\circ$. 

- O–O Grid topology (162 × 52 × 62) used (Reynolds No. = 4.789 × 10^7).
- Realistic surface pressure distribution obtained.
- Flow separation on dome surface followed by a typical longitudinal vortex pair in the wake captured reasonably well.
- Within an error band the present computation gives Lift coefficient (CL) consistent to those from other sources including wind tunnel data.

Figure 20. Turbulent flow around a radome structure.
these large number of simulations. After a wind tunnel model is made, tunnel simulations are fast and less expensive compared to each of the CFD simulations. It is virtually impossible to hope that several hundred simulations could practically be carried out by CFD in an affordable time-frame for an aircraft design. This is evident from Table 1, where some rough estimates are given for the cost of CFD and wind tunnel simulations at NAL.

**Concluding remarks**

The laudable growth over the past three decades in the capability of CFD has convincingly brought home its distinctive role as an adjuvant to the design and development efforts in aerospace. Its role in aeronautical engineering is to be judged by its addition of efficacy to the traditional elements of aircraft design, such as engineering data sheets and experiences, wind tunnel testing and flight testing. It is not to be expected that these traditional methods will be put to disuse because of the mature developments in computational fluid dynamics; it is indeed important to realize that these traditional methods will continue to have their roles to play but now with a synergy from CFD, to arrive at an optimum design sooner and cheaper than in the earlier times.

---

Leprosy is still a major health problem in India. *Mycobacterium leprae*, the organism causing the disease, appears to be genetically invariant. Attempts to identify strain variation in different isolates of *M. leprae* have met with little success. The *M. leprae* 18-kDa heat shock protein is a major T-cell antigen, and this protein belongs to the family of small heat shock proteins. In the present study, we have analysed the *M. leprae* 18-kDa heat shock protein gene in lesions across the leprosy spectra by RT–PCR and also sequenced the PCR amplicons prepared from 25 endemic populations. In addition, transmission patterns and relapse and/or reinfection in reactive patients could be examined using this polymorphism. Expression of the 18-kDa HSP gene in reactive cases indicates that live bacterium might be contributing to the reactional conditions in leprosy.

LEPROSY is a chronic infectious disease caused by the acid-fast bacillus *Mycobacterium leprae*. Leprosy is still a health problem globally, and India has 64% of the global leprosy burden. Leprosy is an immunological disease with defined immunological and clinical parameters, and ranges from tuberculoid (TT) to lepromatous (LL) leprosy with a range of borderline cases between the polar ends. TT patients have robust, cell-mediated immune response that restricts the growth of the pathogen leading to the reactional conditions in leprosy.