DOI: https://doi.org/10.36094/sc.v89.2025.Role_of_Computation_of_Fluid_Flows_in_the_Design.Puttam.126

ARTICLE

Sci. and Cult. 91 (1–2) : 126-132 (2025)

ROLE OF COMPUTATION OF FLUID FLOWS IN THE DESIGN OF AEROSPACE VEHICLES

JYOTHI KUMAR PUTTAM, SHARANAPPA V. SAJJAN*, K. MADHU BABU, C. ARSHAD SHAMEEM, T.N. VENKATESH AND V. RAMESH

Aerodynamic characterization of the configurations is crucial in designing any airborne vehicle, and this paper explains the role of fluid flow simulation in the design process. In recent years, computational fluid dynamic analysis has become an integral part of the aircraft design, right from the preliminary phase to certification and beyond. The basics of fluid dynamics and different simulation methodologies are presented in this paper while explaining the experience of CSIR-NAL in SARAS Mk II aircraft configuration design as a case study. The importance of CFD in aiding design choices and reducing wind-tunnel campaigns is high-lighted.

Keywords: *Aircraft, Aerodynamics, Fluid flow simulations, CFD, RANS*

Introduction

Since time immemorial, humankind has been fascinated with flying in the air like birds, soaring high, and experiencing floating sensations. Efforts to achieve flight have always been made, and accounts of trials have existed since around 400 BCE. Stories of such attempts are plenty in mythology, such as Pushpaka Vimana with infinite cabin space (Indian) and flying with attached wings made up of feathers and wax by Daedalus and Icarus (Greek). Attempts in the past 200 years, for which historical records exist, include Ornithopters, hot air balloons, gliders, etc. Finally, with the first-ever successful powered flight by Wright Brothers in 1903, the history of human flight started. Since then, many aircraft have been designed by continuously improving the aerodynamic configuration, materials, structural design, engine type, flight controls, etc. This has enhanced civilian and fighter segments' performance, capacity, comfort, and capabilities. This process of evolution is ongoing and will continue in future.

Computational and Theoretical Fluid Dynamics Division, CSIR-National Aerospace Laboratories, Bengaluru 560017, India

* Corresponding Author: svsajjan@nal.res.in

The problem of heavier-than-air flight in the air can be described as follows: An object needs a force on it to counter the gravitational force[1]. This could be provided by expelling mass at high velocities, as is done by rockets. The other method is to cleverly use the reaction of the surrounding air to the forward motion of an object: this is the field of aerodynamics[2]. This approach is more energy efficient; therefore, most modern transport aircraft make use of this. In the following sections, the relevant aspects of aerodynamics are described and shown how they are used in the design of aircraft.

For designing any new aircraft or in order to incorporate any changes in existing aircraft, aerodynamic characterization of the proposed configuration is the first and foremost step. Aerodynamic characterization includes the study of flow behavior at every possible condition which will be encountered by the vehicle in flight and needs to be simulated to understand the difficulties. Based on the results, configuration modifications take place in the preliminary design phase. Various design iterations will occur from the initial configuration to the final one, fulfilling all the aerodynamic requirements. Then only, the detailed design starts. Hence, aerodynamic analysis becomes the foundation of aircraft design.

Aerodynamics

Aerodynamics is the branch of fluid dynamics that studies air flow past objects. Such, studies are required in many problems other than aerospace vehicles like natural atmospheric flows for weather prediction to the construction of big structures, human-induced flows in sports (flow over cricket ball, golf ball) for better performance to design of chimneys, etc.

Such air flows are studied mainly in two ways apart from the approach of monitoring the actual flows, which is costly, complex, time-consuming, etc. or even impossible in a few cases. The first method involves experiments on a scaled/full model in wind tunnels, while the second method involves the physical problem being modelled mathematically, and flow is simulated by solving the system of mathematical equations. It is the second approach that is explained in this paper.

Mathematical Modelling of Fluid Flows and Solution Approaches

For mathematical modelling of significant practical flows other than at rarefied conditions, the fluid, in the present case, air, is considered as a continuum. Even though air consists of molecules, the large number makes the statistical description of air as a continuous fluid valid for the range of speeds and densities normally interested in. The imposition of the "law of conservation of mass" to fluid flows results in the continuity equation. When, Newton's second law of motion is applied to fluids, the momentum equation is obtained. This set of coupled partial differential equations is called the Navier-Stokes Equations (NSE), and it is in memory of the inventors Claude-Louis Navier and George Gabriel Stokes. The "law of conservation of energy" yields the energy equation for fluid flows.

These governing equations are 2nd-order non-linear partial differential equations, which need to be solved either theoretically or numerically to simulate the flow past any object. Most of the fluid flows are governed by the above three equations; whether the flow is over a small fly or a big aircraft. The solution of these equations with the appropriate boundary conditions and equation of state of the flow medium gives the complete information in the flow domain consisting of velocity, pressure, density and temperature. Through post-processing, this solution, any other quantity of importance, viz. various forces and moments exerted on the body (like lift, drag, pitching moment, etc.), vorticity, mass flow rate, heat transfer, etc., can be deduced.

Theoretical Approach: In this approach, the governing equations are solved by analytical methods to get exact closed-form solutions. Even advanced mathematical methods can only solve elementary fluid flow problems. Hence, for solving practical fluid flow problems, resorting to a numerical approach is a viable option using computers.

Numerical Approach: The numerical approach of solving the fluid flow governing equations is called as Computational Fluid Dynamics (CFD). CFD is a branch of fluid dynamics that simulates the flow in a region (computational domain) around a body within certain boundaries (far-field) at discrete locations in the domain (grid) by solving the governing equations in discretized form numerically[3,4]. Depending upon the flow regime, different CFD simulation techniques are adapted as follows[5]:

Laminar Flow Simulations: Navier-Stokes equations are solved as they are since the flow is very orderly without eddies; hence, modelling is not required. With a reasonable grid resolution, all the laminar flow features are captured. Though the solution accuracy increases with grid resolution, the simulation can be completed in a few hours.

Turbulent Flow Simulations: Turbulent flows are highly irregular and involve different time and length scales involving eddies of different sizes varying randomly both specially and temporally. Majority of practical flows are turbulent in nature. Different simulation methods are adapted based on the required level of accuracy and availability of resources and time as follows:

Direct Numerical Simulations (DNS): All the turbulent scales are resolved with a very fine grid up to the size of the smallest eddies in the flow field without modelling any feature. Hence, governing equations are solved as they are. This approach leads to the most accurate solution at the expense of substantial computational resources and time, even for simple flow problems, as the size of the grid is very huge. DNS is like a virtual experiment conducted in computer, which requires hundreds of millions of core hours for even simple flows.

Large Eddy Simulations (LES): In this approach, only large eddies having size above a certain limit are resolved with the grid, and all the smaller eddies are modelled through filtering. Hence, the grid size drastically comes down compared to DNS. LES simulations of simple, practical flows are feasible nowadays due to the advancement of computational resources. The accuracy of LES is sufficient for many sophisticated problems like aero-acoustics, combustion, etc.

Reynolds Averaged Navier Stokes Simulations (RANS): This simulation approach offers moderate accuracy of solutions to practical flow problems by finding the time-averaged turbulent flow quantities. Upon substituting the splitting of flow quantities into time averaged and fluctuating components into NSE results in RANS equations with a requirement of closure on fluctuating quantities. Many models are available for this closure, which are popularly called turbulent models in CFD. The accuracy of RANS simulations is sufficient for most of the engineering problems and, hence, is widely used for engineering and industrial applications.

With this background of simulations of fluid flows using CFD, a case study of the role of CFD in the configuration design of SARAS Mk II aircraft[6] at CSIR–NAL is presented in the next section.

Role of CFD in SARAS Mk II Aircraft Design

During the initial stages of the configuration design, selecting the airfoil is a crucial task. In the absence of wind tunnel data, CFD plays a very important role in selecting the airfoil. Various candidate airfoils are simulated, and their aerodynamic characteristics are analyzed to find the most suitable airfoil for the current design. After selecting the airfoil, various parameters required for the wing design, like planform, twist, taper ratio and aspect ratio, are estimated and then optimized to achieve desirable stall characteristics while maintaining the minimum possible drag. With this wing, along with all other components of the aircraft, an initial configuration of the aircraft is built, which is called conceptual configuration. A series of preliminary configurations of the aircraft are conceived in the design refinement process through various iterations to meet the top-level requirements and airworthiness standards through a large number of complete aircraft CFD simulations. Some of these configurations are also tested in wind tunnels for confirmation purposes. One such preliminary configuration of the SARAS Mk II aircraft is shown in Fig. 1.

Once major requirements are met, various optimization studies are performed through CFD to improve the performance. After that, the final frozen configuration goes on to detailed design. Next, metal cutting and assembly of full aircraft takes place. During flight testing and certification, many unforeseen problems may arise. Once again, CFD is utilized for investigation and correction purposes. This is not the end of the role of CFD in aircraft design as it continues to be used to solve problems of aircraft encountered during the operation by end users also. In a nutshell, CFD is used as an inseparable analysis tool throughout the life cycle of aircraft, i.e. during design, flight testing and certification and operations.

In the following few sections, the role of CFD is highlighted in the design process of SARAS Mk II aircraft for aerodynamic characterization, reducing drag, meeting control power and hinge moments of control surfaces, power-on simulations, flap design, engine nacelle optimization, winglet optimization, etc.

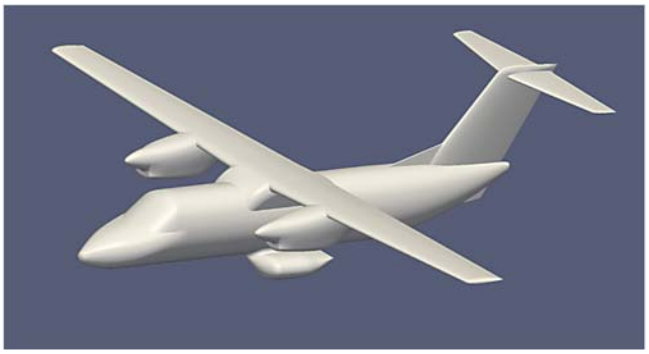

Fig. 1: Preliminary configuration of SARAS Mk II aircraft

Total Aircraft Aerodynamic Characterization: The aerodynamic analysis of the total aircraft is performed using CFD with each configuration modification at every design iteration. The set of studies for complete analysis consists of the following cases, in general:

1. Basic aerodynamic analysis set on a test matrix of angle of attack and side-slip angle sweep at sea level and cruise conditions
2. Various control surface (aileron, elevator, rudder, spoilers, trim tabs, etc.) deflection studies for control power, hinge moments
3. Power-on studies at various thrust levels to include the inference effects due to propeller slip-stream
4. Various flap deflection configurations for checking the achieved maximum lift coefficient ($C_{L\ max}$)
5. Ground effect studies of landing and take-off flap deflections, etc.
6. A set of structural load cases at various points of flight envelope for accessing structural loads. Sometimes the same data is used for designing the wind-tunnel model itself.

Depending upon the modification of configuration few or all of the above cases with at least the set of basic aerodynamic analysis are simulated using CFD. Figure 2, shows the surface pressure distribution on particular candidate configuration at sea level conditions obtained through a simulation performed as part of basic aerodynamic analysis. The suction pressure present on the wing upper surface is seen very clearly along with flow stagnation at nose, wind shield, leading edge of horizontal tail (HT) and vertical tail(VT).

Fig. 2: Surface pressure distribution on a candidate SARAS Mk II aircraft configuration at sea level

Studies to Reduce the Aerodynamic Drag: Once a CFD simulation is performed on any configuration, the total information of the whole computational domain is available, and the same can be utilized for any post-processing and diagnostic purposes. One such diagnostic study was performed to reduce the aerodynamic drag of SARAS Mk II aircraft based on the results of basic aerodynamic analysis. The component-wise drag break-up of different components, as shown in Figure 3 with different colors, is calculated using the results of the angle of attack sweep. It is found that the total drag can be reduced by streamlining some of the components. Hence, all such component modifications, viz. nose of the fuselage, blister fairing, wing fairing, nacelle, etc., are tried through various iterations using CFD to achieve an optimum minimum drag of total aircraft.

Sizing of Control Surfaces and Actuators: The stability and controllability of aircraft are the most important parameters for the safe flight, which needs to be taken care while designing. These parameters depend on the size of the control surface, placement, flow behavior at various conditions, aerodynamic interference with parent member on which it got installed, etc. The primary control surfaces of an aircraft include: rudder, elevator and aileron, which are respectively attached to the vertical tail, horizontal tail and wing near the tip as shown in Fig. 4. These control surfaces are used for directional, longitudinal and lateral controls respectively.

The designers size these control surfaces to provide enough control power to the pilots to perform desired maneuvers. In SARAS Mk II aircraft design, aircraft control-derivatives are extensively used for sizing the control surfaces through a series of iterations. As a result (i) the final configuration of all the control surfaces are fitted with horn balance and (ii) avoided sluggish response of control surfaces in simulator tests by pilots. Hinge moment is a vital information for choosing the hydraulic actuators used to deploy the control surface. Apart from actuator sizing, by calculating the hinge moments over a span of deflections and flight conditions possible rudder lock is also detected and corrected for SARAS Mk II aircraft. The surface pressure distribution and set of streamlines passing the rudder are shown from one such CFD analysis on SARAS Mk II at rudder deflection of 30° in Fig. 5. Near rudder, deflection of streamlines towards pilot left side is seen clearly as the rudder is deflected towards left.

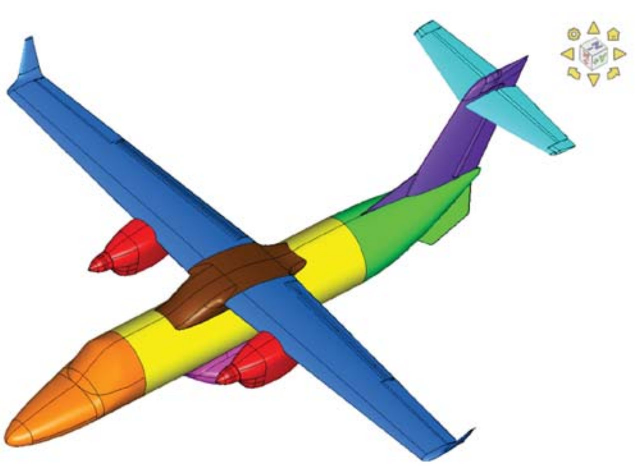

Fig. 3: Different components of aircraft, whose drag break-up is used for reducing the drag of total aircraft

Fig. 4: Different control surfaces of a preliminary configuration of SARAS Mk II aircraft are highlighted in magenta color.

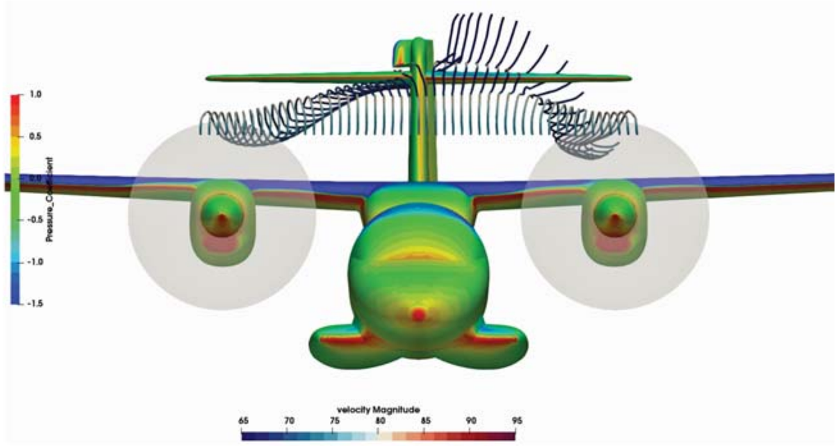

Fig. 5: Surface pressure distribution and set of streamlines passing the rudder of SARAS Mk II at a rudder deflection of 30°

One Engine In-operative Power-on Simulations: The propellers mounted on the aircraft produce thrust, which is the force used to overcome the drag of an aircraft. Thrust is also helpful for aircraft to accelerate, climb and sometimes to perform various maneuvers. The Saras aircraft has two engines mounted on the wing. For the compliance of airworthiness standards, controllability of aircraft during one engine in-operative situation is demonstrated at various conditions and thrust levels through CFD simulations. Performing the same in wind-tunnel experiment is difficult and dangerous in flight test. Fig. 6 shows the simulation result of one such one engine in-operative test case at sea-level and zero-degree angle of incidence with 20° flap deflection configuration in terms of surface pressure distribution and a set of streamlines. As the streamlines are coloured with velocity, the acceleration of fluid on left propeller is seen clearly along with more suction on left wing because left propeller alone is there in power-on mode.

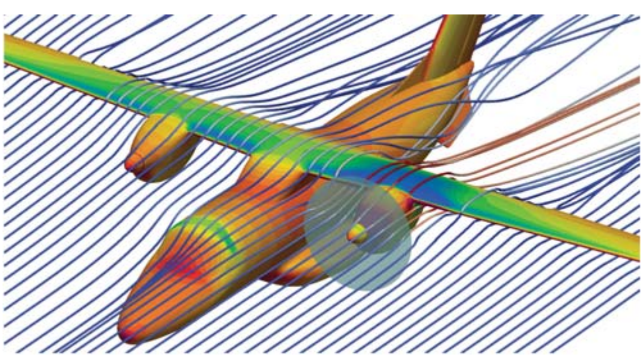

Fig. 6: Surface pressure distribution and set of streamlines on SARAS Mk II aircraft with landing flap deflection and one engine in-operative simulation at sea-level

Design of Optimum Flap Track for SARAS Mk II Aircraft: Flaps are high lift devices normally embedded as trailing edge portion of the wing (as shown in Fig. 4), which are deflected downwards on demand to produce extra lift during take-off and landing (at lower forward speed) by increasing the effective camber of the wing. Upon deployment of flap, drag also increases along with lift and stall speed reduces.

In general, while deploying the flap, backward translation is also needed along with downward rotation in such a way that certain gap is created between main wing and flap such that high pressure air leaks from wing lower surface to upper surface in a form of jet, which energies the fluid on flap upper surface. The gap created between wing and flap plays vital role in achieving optimum performance of flap. Hence, the flap deployment track is the key parameter, which needs to be optimized. At CSIR-NAL such optimization study is performed first in 2D and later in 3D using CFD for SARAS Mk II aircraft. Fig. 7, shows the surface pressure distribution on SARAS aircraft in power-on mode with landing flap deflection while landing gears deployed along with cavity doors open. Prominent suction on the flap upper surface is seen clearly along with propeller slip-stream induced enhanced suction.

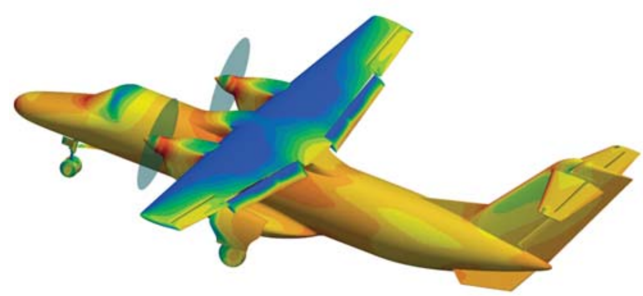

Fig. 7: Surface pressure distribution on SARAS aircraft with flap deflected for take-off

Nacelle Design and Optimization: The optimization of engine nacelle is performed through the approach of Design of Experiments (DOE) by morphing the existing baseline geometry to generate sample space by using a commercial morphing tool. Later, CFD simulations are performed on all the sample designs to find the optimized nacelle, which resulted in reduction of drag by a few counts. Fig. 8. shows the comparison of surface pressure distribution on baseline nacelle and optimized nacelle. The aft portion of the optimized nacelle is split and fastened to the flap such that it leads to better pressure recovery, which is clear from fig. 8.

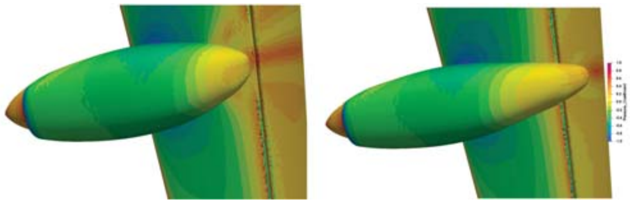

Fig. 8: Nacelle shape optimization: surface pressure distribution on baseline configuration (left) and optimized configuration (right)

Winglet Design and Optimization: Winglets are the devices used to reduce the induced drag through weakening the strength of wingtip vortices, thereby improving aircraft performance. Hence, results in lower fuel consumption and ultimately reduces aircraft operation cost considerably. The baseline winglet design used in the optimization for SARAS Mk II aircraft is shown in Figure 9. The design variables considered for winglet optimization are span, sweep, twist and cant angle. The objective function considered for optimization is maximization of L/D ratio of total aircraft. Around 50 sample geometries of winglet are generated and analyzed to obtain the optimum winglet design. Fig. 10 shows the pressure distribution on three of sample winglets. This optimization study yielded an optimum winglet design for Saras aircraft with a reduction of drag by 0.4% and improvement of aerodynamic efficiency (L/D) by 1.3%.

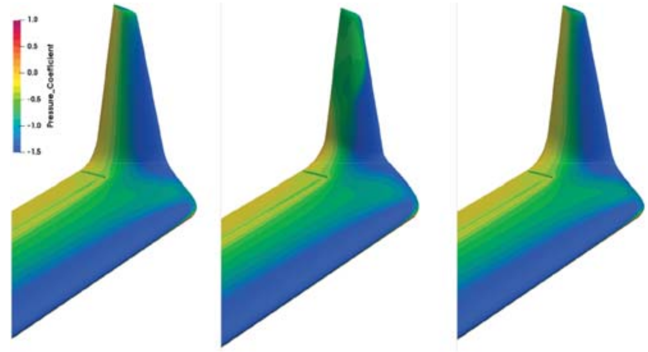

Fig. 10: Pressure distribution on three of candidate configurations of winglet optimization study

Fig. 9: SARAS Mk II aircraft configuration with baseline winglets at the tip of the wing

Aerodynamic Loads for Structural Design: Aerodynamic loads are the part of the total load coming on the aircraft structure, whereas the remaining part comes from inertial, thrust and ground loads. In detailed design process all load bearing structural members needs to be designed to bare these loads at various critical points of flight envelope such that they are safe at any other point of envelope. Aerodynamic loads originating from the total wetted surface are borne by structural members through aircraft skin. From CFD solution, aerodynamic loads are deduced from all the cells/nodes of aircraft surface mesh, which is very fine when compared to the resolution of mesh required for structural analysis to design structural members. The resolution of CFD mesh used on HT and VT is shown in Fig. 11. Hence, for transferring this high resolution aerodynamic loads to component based low resolution structural mesh, a set of python scripts are developed and utilized extensively in SARAS Mk II aircraft design process.

Fig. 11: CFD mesh used on HT and VT surfaces of in a zoomed view

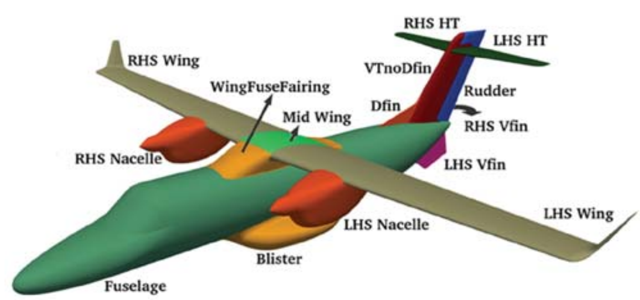

Fig. 12: Separation of different components of Saras aircraft used for structural analysis

Since the structural design is very involved task, different teams design different components. Hence, once the critical cases are simulated using CFD, loads on each component of aircraft viz. LHS wing, RHS wing, fuselage, LHS horizontal tail, RHS horizontal tail, vertical tail, nacelle, etc. as shown in Fig. 12 needs to be separately extracted for structural analysis. A representative mesh used for structural analysis on total aircraft is shown Fig. 13, though the structural analysis is carried out on different components separately as shown in Fig. 12.

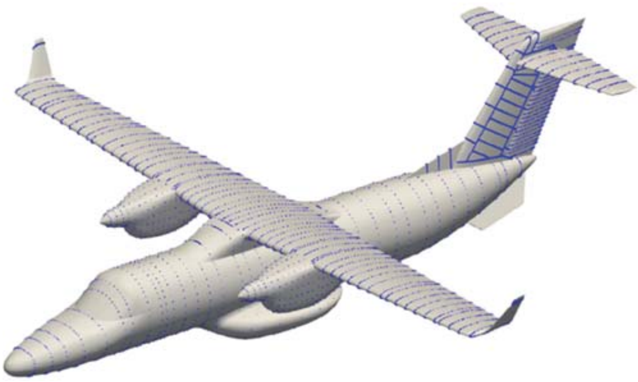

Fig. 13: Mesh on various components of Saras aircraft used for structural analysis

Summary

The process of aircraft design is quite complex and requires the interaction of a large number of teams from various disciplines to perform the required theoretical calculations, numerical simulations, optimizations, physical tests and experiments to achieve a paper design. It is only after this, that the manufacture of a prototype starts. Of the various aspects, aerodynamics plays a fundamental role as this provides the basic force which keeps an aircraft flying in the air. The aerodynamic configuration drives much of the other design. In recent times, the aerodynamics depends heavily on CFD for exploring the various design options and coming up with quantitative estimates of the various parameters. The experience at CSIR-NAL during the design of SARAS Mk II aircraft, has been that CFD plays a crucial role in all aspects for the aircraft design, not only during the initial configuration studies, but also at various stages of the design and beyond.

Acknowledgments

The authors gratefully acknowledge Director, CSIR-NAL for his constant guidance, encouragement and support. Authors also acknowledge the RNCAC design team for their continuous interaction during the course of work. Authors also would like to thank the CSIR-4PI team for providing the HPC facilities and support during the work. SU2 and ParaView development teams are greatly acknowledged for providing these open-source tools. ❒

References

1. A.C. Kermode, Flight Without Formulae (Pearson, India, 2007).
2. E.L. Houghton and P.W. Carpenter, Aerodynamics for Engineering Students (Butterworth-Heinemann Ltd, 2016).
3. J.D. Anderson, Computational Fluid Dynamics-The Basics with Applications, (McGraw-Hill Series in Aeronautical and Aer, 1995).
4. J.C. Tannehill, D.A. Anderson, R.H. Pletcher, Computational Fluid Mechanics and Heat Transfer (CRC Press, 2012).
5. https://jigyasa-csir.in/nal/n18-t3-a1/.
6. https://www.nal.res.in/en/technology/saras-mk2-19-seaterpassenger-aircraft.