

CFD-BASED COLLABORATIVE DESIGN OPTIMIZATION USING eCFD

B.I. Soemarwoto*, A. Nugroho, E.H. Baalbergen*, A. Aribowo*****

***National Aerospace Laboratory NLR, The Netherlands**

****PT Regio Aviasi Industri, Indonesia**

*****National Institute of Aeronautics and Space LAPAN, Indonesia**

bambang.soemarwoto@nlr.nl

Keywords: *CFD design, turboprop, laminar airfoil, adjoint method, aerodynamic optimization*

Abstract

With the advancement of computer technology, CFD has now found its way into the industrial environment and become indispensable for the aerodynamic design of aircraft. It remains, however, that a reliable use of CFD requires an expertise equipped with a sufficiently accurate numerical algorithm and adequate computing power.

Operating specific CFD software on typically high-end hardware may entail a steep learning curve. Consequently, apart from acquiring a software license and high-end hardware, building and maintaining own CFD software capabilities within a company may become undesirably expensive. eCFD, where “e” stands for e-mail, offers a new concept to circumvent this situation. eCFD facilitates a remote and tailored use of CFD software and computing capacity, using e-mail as basic means for controlling the flow simulations.

The concept of eCFD is described in this paper. Its viability is demonstrated in an international successful collaboration setting, involving two geographically distant locations, Indonesia and The Netherlands. The potential of eCFD is illustrated by two collaborations, concerning the analysis of a turboprop aircraft in a high-lift configuration (RAI and NLR) and a laminar airfoil design optimization (LAPAN and NLR).

1 Introduction

Computational Fluid Dynamics (CFD) plays an indispensable role in the aerodynamic design

process of aircraft. Nowadays it has been able to capture essential flow physics, allowing an accurate selection of design candidates in order to isolate the best design.

It is not surprising that there is now ample choice of CFD software in the market. Besides those offered by major commercial vendors, open-source CFD software has also become a serious option. However, the wide availability of software does not mean that one can readily exploit the potentials of CFD after installing the software on their computer.

A reliable use of CFD analysis and design tools requires an expertise equipped with skilled scientific insights into extensive preparation of the flow simulation, covering pre- and post-processing tasks, such as domain modelling, grid generation, specification of boundary conditions, determining the solution procedure, and visualization of the results.

The tasks involved are not trivial, and should not be expected to be borne by a non-expert of CFD. Aerodynamic engineers should be freed from the complexities involved in the above tasks, although the engineers are very knowledgeable in fluid dynamics. Their precious time should instead be devoted to interpret flow simulation results, draw conclusions and make crucial design decisions.

In addition to the abovementioned, operating a specific CFD analysis tools on typically a specific hardware to perform the above tasks may entail a steep learning curve. Consequently, installing, operating and maintaining own CFD software capabilities within a company may become undesirably

expensive, apart from purchasing a software license and high-end hardware.

eCFD, where “e” stands for e-mail, offers a new concept to circumvent the need to acquire, operate and maintain own CFD software. It facilitates a remote and tailored use of CFD software and provides required computing capacity, using e-mail as basic means for controlling the flow simulations.

The new concept promotes a collaboration between (a) a team of experts in the field of CFD who prepares the setup of simulations (henceforth referred to as *the provider*), and (b) a team of aerodynamic engineers (henceforth referred to as *the user*). They are generally located at distantly separated sites.

This paper will first describe the concept of eCFD in terms of specification and implementation. Subsequently, its viability is demonstrated in an international collaboration setting, involving two geographically distant locations, Indonesia and The Netherlands. Two collaborations are presented, concerning the analysis of a turboprop aircraft in a high-lift configuration and a laminar airfoil design optimization.

2 eCFD specification and implementation

The collaboration between *the provider* and *the user* is supported by eCFD as depicted globally in Fig. 1. The collaboration comprises four steps: (1) the user first provides the geometry, the relevant flow conditions, and the design problem, (2) the provider then parameterizes the computation and design problem in collaboration with the user, (3) the provider next sets up the simulation as an *eCFD application*, (4) the user finally performs a series of simulations by using the eCFD application. The blue box in Fig. 1 represents the eCFD system for automated execution of the simulations driven by the parameters defined in step (b).

This section specifies eCFD in terms of its global set-up, the security and budgeting issues involved, and its implementation.

2.1 Specification

In technical terms, eCFD is a software system

that implements NLR CFD simulations *as-a-service* and *in the cloud*. The global set-up of eCFD is depicted in Fig. 2. The solid arrows indicate the flow of data and information. The numbered arrows specify the order in which the exchange of information for a single simulation takes place from the user’s perspective. The details are described in the remainder of this section.

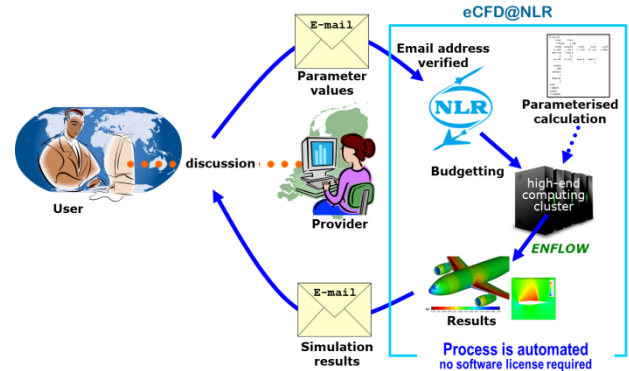


Fig. 1 Collaboration utilizing eCFD.

The provider wraps a usually complex, CPU-power and memory demanding CFD simulation, including a set of input data and parameters, into an easy to operate eCFD application. Two groups of parameters can be distinguished: (i) those with values that are either fixed or must follow certain rules to ensure well-posedness of the physical and numerical problem at hand, and (ii) those with values that may vary depending on the prevailing demand for flow simulation.

The latter group is referred to as the *application parameters*, the values of which may be specified by the user when requesting a simulation. The application parameters are typically but not necessarily limited to such parameters as Mach number, Reynolds number, and angle of attack. When starting an eCFD application, eCFD automatically incorporates the specified values into the wrapped CFD simulation and subsequently runs the simulation to produce the flow solution.

Under the hood, the simulation is driven by an automated tool chain comprising a grid generator, a flow solver (in particular, NLR’s ENFLOW), and post-processing tools e.g. for visual presentation of the flow solution. The tool chain runs on a compute server that

provides sufficient capacity to run the simulations within reasonable response times.

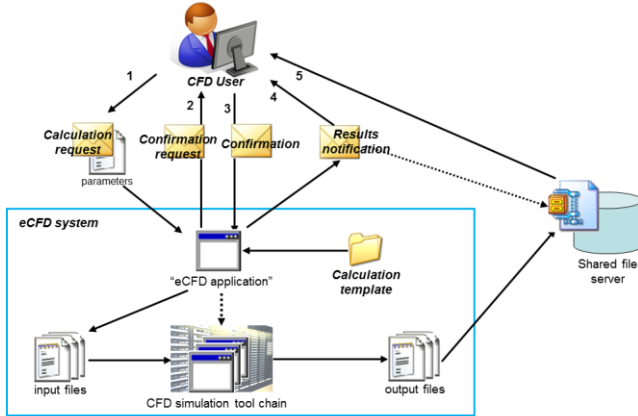


Fig. 2 Global set-up of eCFD.

The eCFD application can be operated via ordinary text-based e-mail messages sent to an e-mail address specifically defined for the application. The interaction between the user and eCFD is represented by the numbered arrows and the four mail messages in Fig. 2.

The user can initiate a simulation by first sending a *simulation request* by e-mail (cf. Fig. 5). The message must specify the values for a usually small subset of the application parameters in a prescribed format. Unspecified parameters have a nominal (i.e., default) value as defined together by the provider and the user. The message may also specify ranges of values for some of the application parameters, to allow several simulations to be specified in a single simulation request (i.e. in a single e-mail message). The ranges enable the user to perform parameter studies in an easy way.

In response to a simulation request, the user receives a *confirmation request* by e-mail (cf. Fig. 6). This message lists the simulation(s) that is (or are) requested in terms of the specified application parameter values. To confirm the simulation, the user simply replies to this message.

Upon receiving the confirmation, eCFD activates the tool chain to perform the requested simulation. After the simulation has finished, the user receives a *results notification* message by e-mail (cf. Fig. 7). The results comprising the flow solution are packed in a ZIP file that is stored on e.g. a secured FTP server. The user

can download the flow solution by clicking the hyperlink included in the results notification.

2.2 Security and budgeting

Since security is an important aspect in today's collaboration (ref. [1]), the usage of the eCFD system is controlled with respect to access and budgeting, and data is exchanged via secured data servers. Since the interaction with eCFD applications is restricted to the provision of input parameter values and the receipt of flow solutions, eCFD does not reveal the intellectual property that is comprised in the methods, codes and data that constitute the true CFD simulation. It also precludes direct usage and visibility of the tools and the tool chain. In addition, operating an eCFD application is restricted to a list of known e-mail addresses.

The combination of having an access list per eCFD application, the need to confirm simulations, and controlling access to the data server through which results are provided, ensures that only authorized users can run simulations and retrieve results.

Budgeting is applied to control the total number of simulations that may be run. Based on the actual cost of a "benchmark case" (i.e., the case where all application parameters have their nominal values), the provider estimates the computing cost for a single simulation. The user can then estimate the total cost required to carry out the analyses and design for the project at hand. As such, an arrangement can be made on the budget for one or more users to perform the required number of simulations.

2.3 Implementation

The eCFD system is implemented as middleware software that can handle several eCFD applications and simulations at a time. It is installed on a system that handles incoming e-mail messages and that can start jobs on a compute server for distinct eCFD applications.

Upon receiving a simulation request for a particular eCFD application, the server performs the following steps: (a) it sends the confirmation request and handles the confirmation, (b) it creates one or more 'instances' of the wrapped CFD simulation by incorporating the application

parameter values as specified in the simulation request, (c) it runs the instanced simulation by managing the execution of the tool chain, and (d) it deals with returning the results (i.e., flow solution) to the user.

To define an eCFD application, the provider must develop a *simulation template*. The template basically consists of the scripts, programs and data files needed for the CFD simulation. It also contains a definition of the application parameters, along with their types, allowable value ranges and nominal values. Examples of such definitions are shown in Table 1 and Table 2. eCFD performs checks to ensure that the values of the application parameters specified in the simulation request are within the allowable ranges. When an instance of the simulation is created, all textual occurrences of the parameters throughout the constituents of the template are replaced by their actual values to enable the simulation to run.

The eCFD application can be operated through a dedicated e-mail address, such as *ecfd-demo@nlr.nl*, to which the user sends the simulation request as well as other eCFD commands. The application also defines a small set of administrative data such as the total budget, remaining budget, a list of e-mail addresses of people that are allowed to request and confirm simulations, and the maximum number of simulations allowed in a single simulation request. The latter number serves mainly to protect the customer from accidentally using large portions of the available budget in a single simulation request.

The other eCFD commands enable the user to cancel requested but not yet confirmed simulations, and to retrieve help information, the definitions of the application parameters, the status of active simulations, and the remaining budget. These commands are available via specific keywords in the e-mail messages.

3 Collaborative project 1: Turboprop aircraft

Two partners are involved in this collaboration utilizing eCFD: Regio Aviasi Industri (RAI) Ltd. in Indonesia as the user, and

National Aerospace Laboratory NLR located in the Netherlands as the provider.

A generic model of an 80-90 passenger turboprop aircraft was developed at RAI to get some understanding of the flow around the aircraft. The model is of a typical high wing, twin engine T-tail configuration. This exercise was part of a configuration development of an Indonesian designed regional turboprop aircraft, to meet the domestic and world's demand of future larger turboprop aircraft in 2018-2038. The configuration optimization includes amongst others selection of the tail location, i.e. whether a low tail or a T-tail, and is part of a complete vehicle multidisciplinary optimization.

One of the most important design aspects is to ensure that the tail location will provide enough stability and control authority for all flight conditions and power settings within the aircraft's operating envelope. The wake of the wing, at high flaps deflection, combined with slipstream effects might impose unfavorable condition to tail effectiveness.

CFD is considered capable to reveal the essential flow physics and is used as a tool to understand non-linear behavior. The trends will be analyzed in conjunction with wind tunnel measurement, with and without power effect.

The wind tunnel and computational method correlation is an effective tool to gain an understanding of the flow, while minimizing the expensive wind tunnel runs. The knowledge of aerodynamic behavior will be updated during the aircraft development including flight test data when it becomes available.

3.3 Geometry modeling

The configuration under consideration is a complete turboprop aircraft with a double-slotted deflected flap in a power-off condition. Any CFD simulation requires a clean air-tight CAD model. CAD modelling is carried out by an engineer at the side of the user. Fig. 3 gives an illustration of the resulting geometry model.

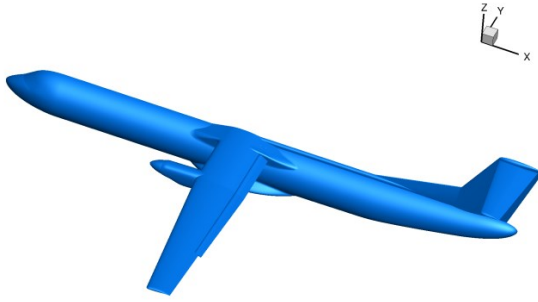


Fig. 3 Impression of the aircraft geometry.

3.3 Computational setup

NLR's ENFLOW [2] is used as the CFD software. The flow modelling is based on the steady-state Reynolds-Averaged Navier-Stokes (RANS) equations equipped with the Explicit Algebraic Reynolds Stress turbulence Model (EARSIM). The system of equations is discretized by means of a central-difference scheme in a finite-volume formulation. The flow domain surrounding the aircraft is discretized by a multi-block structured grid, which is a curvilinear mesh in blocks having a specific mapping to a uniform computational domain.

This mapping process is carried out first by laying out the so-called face decomposition on the surface geometry of the aircraft, where each face represents one of the six faces enclosing a block. One of the key technologies enabling a multi-block domain decomposition around a complex aircraft configuration is a semi-automatic approach exploiting a Cartesian space [3,4].

Each curvilinear face in the physical space is then arranged to have a corresponding computational face in the Cartesian space. Fig. 4 shows how different components of the aircraft are represented distinctly in the Cartesian abstraction. The resulting Cartesian topology facilitates a fast and efficient three-dimensional construction of a multi-block domain topology and high-quality structured grid. This task is performed fully by the provider.

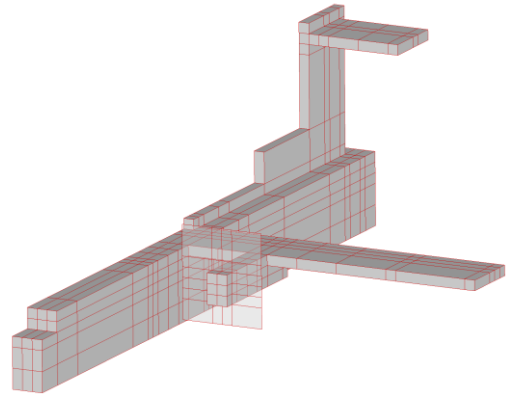


Fig. 4 Cartesian abstraction of the aircraft.

3.3 eCFD application parameters

The prevailing demand has led to the application parameters shown in Table 1. Each parameter has its own range specified together by the user and the provider. A consensus of the range is determined on the basis of, on the one hand, the aircraft design specification and, on the other hand, the applicability area of the CFD method used, which in this case is based on a steady-state flow solver. The default values for the above parameters refer to the design (cruise) condition.

Name	Description	Type
mach	Mach number	real
reyn	Reynolds number	real
velo	Velocity [m/s]	real
alti	Altitude [m]	real
alpha	Angle of attack [deg]	Real
cond	Flow condition option: 1: mach,reyn (default) 2: velo,alti 3: mach,alti	integer

Table 1 Application parameters definition.

It is noted that Mach number, Reynolds number, velocity and altitude are not independent with each other. The parameter **cond** which is of an integer type has a value of either 1, 2, or 3. If cond=1 the CFD simulation will be performed based on the specified Mach number and Reynolds number. Similarly, if cond=2 (cond=3), the CFD simulation will be performed for the specified velocity and altitude (Mach number and altitude).

3.3 eCFD template

The template contains a script that controls various phases of the CFD simulation according to the values of the application parameters specified by the user. This includes pre-processing such as stretching the structured grid properly towards the surface to accurately resolve the turbulent boundary layer including its viscous sub-layer ($y^+ \approx 1$) for the given Reynolds number. Another example is to determine whether or not the low-Mach preconditioning should be turned on based on the given Mach number.

After pre-processing has finished, the script invokes the flow solver to perform the CFD simulation on a cluster computing machine. A typical elapsed time required for one simulation is about 2.5 hours on 192 cores, or about 75 minutes if 384 cores are used, for the grid size of 74.1 million grid cells in 9427 blocks. One polar curve consisting of 10 simulations can be obtained in one day, or overnight if 384 cores are used.

The basic product of a simulation is the flow solution containing values of the density, momentum, and energy (including the turbulence kinetic energy). This is further post-processed to yield: (i) a set of tables and plots of the aerodynamic force and moment coefficients, (ii) convergence history of the numerical solution procedure, (iii) results of a grid convergence study to determine the extrapolated values at the limit of zero mesh size, (iv) wing sectional pressure distribution, (v) flow visualization on the surface and in the flow field, and (vi) files containing numerical data used to produce the flow visualization. All these are collected into a directory named *results* to be packed further by the eCFD application into a ZIP file and made available on an ftp server for download by the user.

3.4 eCFD session and results

An eCFD session from the user's point of view consists of:

(a) Send a *simulation request* e-mail, say to ecfd-turboprop@nlr.nl, containing a sequence of parameter names and values in an embedment bounded by a double minus sign.

Outside this embedment any text are allowed to be present in the e-mail body. Fig. 5 shows an example of a request for a series of simulations defined by a range starting from an angle of attack of 4 degrees, ending at 18 degrees, with an increment of 2 degrees. The subject field of the e-mail is left free for the user to fill in to facilitate a convenient discussion amongst the group of users.

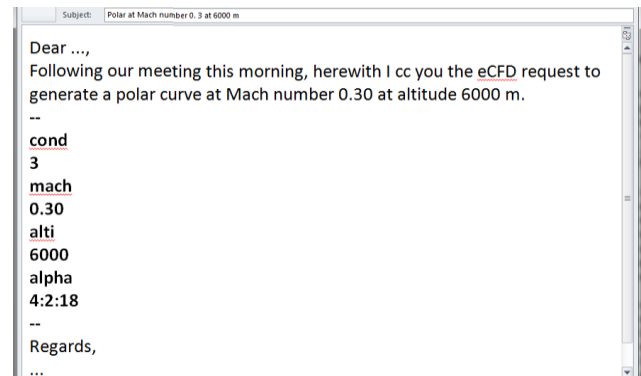


Fig. 5 Example of a request e-mail sent to ecfd-turboprop@nlr.nl

(b) Reply to the *confirmation request* e-mail sent by the server after verifying the intended simulations (see Fig. 6 for example).

(c) Download and discuss the results upon receipt of a *results notification* e-mail, like shown in Fig. 7. After unpacking the ZIP file, all the tables and plots are examined.

One typical task which is not obvious for a person who is not familiar with numerical aspects of a CFD simulation is how to determine the uncertainty due to grid resolution. It is therefore considered rather a task for the provider. Accordingly, the script in the template includes a Richardson extrapolation and a second-order extrapolation procedure.

The term extrapolation refers to extrapolating the value that results from a simulation using a finite number of grid cells (with a certain measure of mesh size) to obtain a value at the limit of infinite number of grid cells (with a zero mesh size). The formal order of the numerical scheme employed by ENFLOW is second-order.

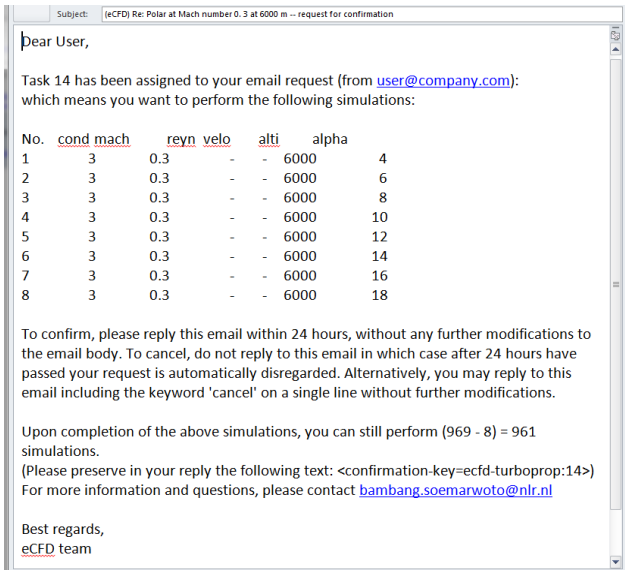


Fig. 6 Example of a confirmation request e-mail received from ecfd-turboprop@nlr.nl

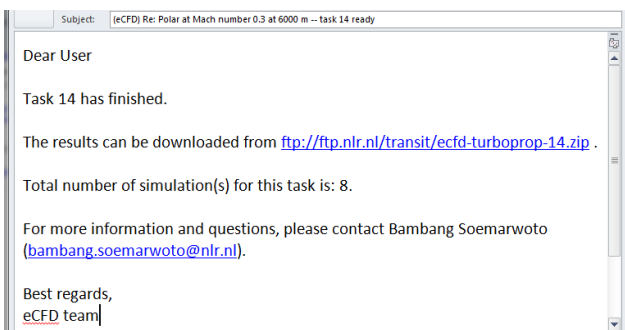


Fig. 7 Example of a results notification e-mail from ecfd-turboprop@nlr.nl

Fig. 8 presents a result of the extrapolation. Using results from the coarse, medium and fine grids, the scheme's order calculated by the Richardson formula is close to second-order. The second order extrapolation uses only results from the medium and fine grids. The difference in the extrapolated values at the limit of zero mesh size is within 2 drag counts. This means that the extrapolated value may be used with high confidence for prediction of the aircraft performance.

Another task which may take some time to fulfill, albeit easy, is to present an XY-plot from a series of simulations to visualize a trend. The script in the template takes this burden away by, for example, automatically generating a lift and pitching moment curve for the assessment of the

stall behavior and longitudinal stability. Fig. 9 shows a distinct non-linear trend near stall.

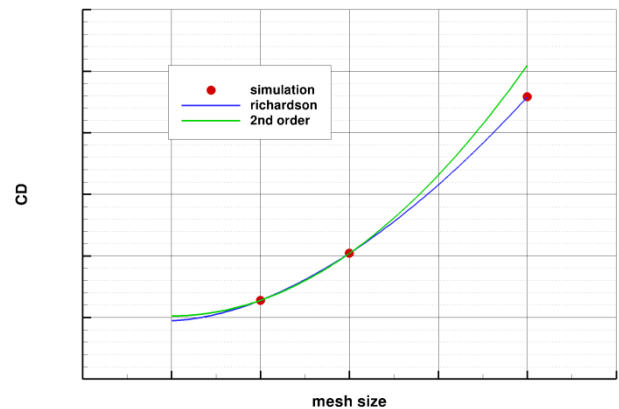


Fig. 8 Typical grid convergence analysis.

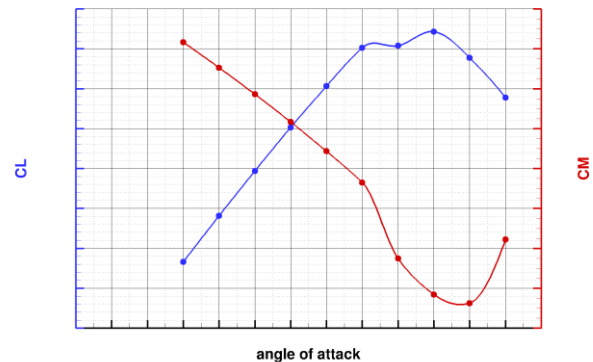


Fig. 9 Typical pitching-moment curve.

Last but not least, the visualization of the flow field to reveal flow structures is in itself an art. The script makes use of customized macros and layouts to produce a flow field visualization such as one shown in Fig. 10. This figure shows how the vortex system develops around the aircraft as the angle of attack increases. The contours with relatively high total pressure losses indicate losses due the turbulent wakes, separated flows and vortical flows. At higher angles of attack, one should observe how and when the control surfaces on the tail planes become submerged in the turbulent wake and/or vortical flows. Along with the numerical data for visualization, the aerodynamic engineers may further investigate the non-linearity observed near stall condition, with their own tool when appropriate.

A typical eCFD session to produce a polar curve may generate several tables and hundreds

of XY and contour plots, with which the trends of the flow can be examined thoroughly.

4 Collaborative project 2: Laminar airfoil

The second project concerns an ultra-low Reynolds number laminar airfoil design, with the Reynolds number in the order of 10^3 , where the flow is assumed to be fully laminar and steady. There is a growing interest for this flow regime as manufacturability of small structures become more feasible. The application range is wide, e.g. small (micro) UAV's, but also in other fields such as cooling fans, pre-swirl stator, flow conditioning devices in small ducts, etc.

The project involves two partners: National Aerospace Laboratory NLR located in the Netherlands and National Institute of Aeronautics and Space LAPAN in Indonesia. This is another example how eCFD facilitates the collaboration in a natural way, where the CFD simulations can be planned and discussed from anywhere using any platform that supports e-mail communication.

4.3 Design problem

The design objective is to minimize the drag of an airfoil in an ultra-low Reynolds number flow subject to constraints on geometric properties of the airfoil: the cross-sectional area, leading edge radius, trailing edge included angle and maximum thickness. The design variables define the airfoil geometry where the C_2 -continuity of the airfoil contour is maintained throughout the optimization process [5].

In general, the human designer has a limited knowledge on how to formulate the design problem, e.g. setting the constraints and their bounds appropriately. Initially these are at best defined by an educated guess. As a result, after examining the optimal solution of an initial problem formulation, one may want to adapt the formulation to obtain a physically more desirable optimal solution in the next design cycle. This situation was well reflected in a design project representing one of the most successful regional jet aircraft [6].

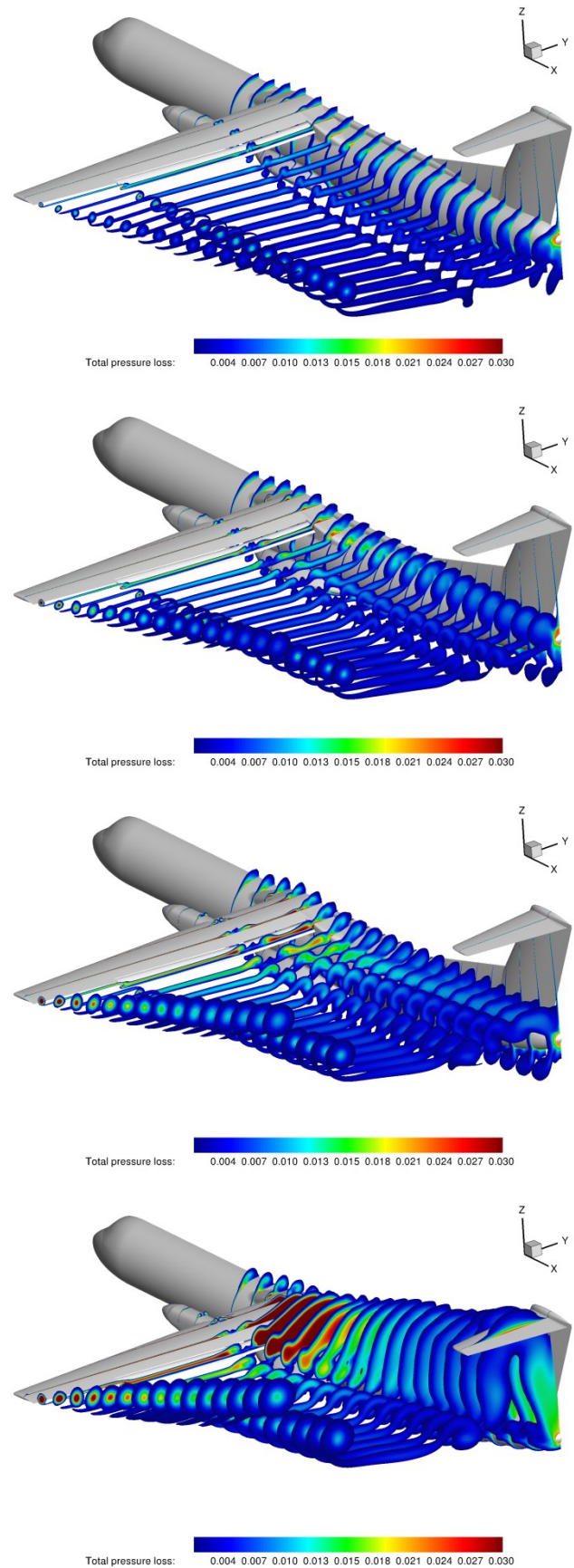


Fig. 10 Flow visualization.

In this collaborative project eCFD facilitates such a successive adaptation of the design problem formulation.

4.1 Optimization using the adjoint method

The design problem is solved by means of a gradient-based optimization method, in which the gradient is determined using the adjoint method. It is widely known that the adjoint method is most efficient for a small ratio between the number of aerodynamic functionals and the number of design variables. In the present case, two aerodynamic functionals are considered, i.e. for the drag and lift coefficient, and there are 41 design variables defining the airfoil geometry to sufficiently cover the design space.

4.3 eCFD application parameters

The optimization problem of the airfoil is parameterized in a form of

*Starting with an initial airfoil [**init**]
Minimize CD at a design CL,
subject to constraints:
 $A_{section}$ [**casec**] [**basec**],
 R_{LE} [**crle**] [**brle**],
 δ_{TE} [**ctea**] [**btea**],
 T_{max} [**ctmax**] [**btmax**].*

where CD and CL the drag and lift coefficient, respectively. $A_{section}$, R_{LE} , δ_{TE} , and T_{max} denotes the cross-sectional area, leading edge radius, trailing edge included angle and maximum thickness. The application parameters are shown above in bold between square brackets, described in Table 2.

A new feature in this eCFD application is the possibility to specify the initial airfoil geometry (coordinates) in a file which is prepared by the user in their own environment and uploaded to an agreed ftp server. The parameter **init** is used to pass the name of the file. The script downloads the file and uses the geometry to start the optimization process.

Name	Description	Type
<i>init</i>	name of the file containing airfoil coordinate uploaded in the ftp server	string
<i>basec</i>	bound for $A_{section}$	real
<i>brle</i>	bound for R_{LE}	real
<i>btea</i>	bound for δ_{TE}	real
<i>btmax</i>	bound for T_{max}	real
<i>casec</i> <i>crle</i> <i>ctea</i> <i>ctmax</i>	0: no constraint ±1: equal to ±2: larger than ±3: smaller than (a negative value will take the properties of the initial airfoil as the constraint bounds)	integer

Table 2 Application parameters definition.

4.4 eCFD session and results

The Reynolds number of the flow is 10^3 based on the airfoil chord. The design lift coefficient is 0.336. Two eCFD sessions were performed, each with a different formulation of the design problem. In both sessions, the Eppler 387 airfoil is selected as the initial airfoil geometry. It should be noted that each session is limited by the same number of flow and adjoint solutions, which are 10 and 16 respectively, determining the unit cost of this application.

In the first formulation, all constraints are released, except the cross-sectional area which has to be equal to that of the initial airfoil, such that $casec = -1$, $crle$, $ctea$, $ctmax = 0$. Starting from the initial airfoil shown in Fig. 11a, this formulation leads to an optimal airfoil in Fig. 11b. Clearly, the trailing edge separation bubble has been reduced significantly, along with a reduction of 49 drag counts. The optimal airfoil has a rather peculiar shape. Also, further investigation reveals that the leading edge radius and maximum thickness have been decreased significantly. As a result, the stall behavior becomes less favorable.

In the second session, the formulation is adapted by including the other constraints, where they must be larger than their initial values ($crle$, $ctea$, $ctmax = -2$), while keeping

$casec = -1$. This results in an optimal airfoil shown in Fig. 11c. The trailing edge separation bubble has also been reduced significantly, with a drag reduction of 40 counts. Clearly, the optimal airfoil features a significant aft loading compared to the initial one. The leading edge radius and trailing edge including angle become larger, but the maximum thickness remains the same. From the practical point of view, the new optimal shape is much more desirable than the initial one.

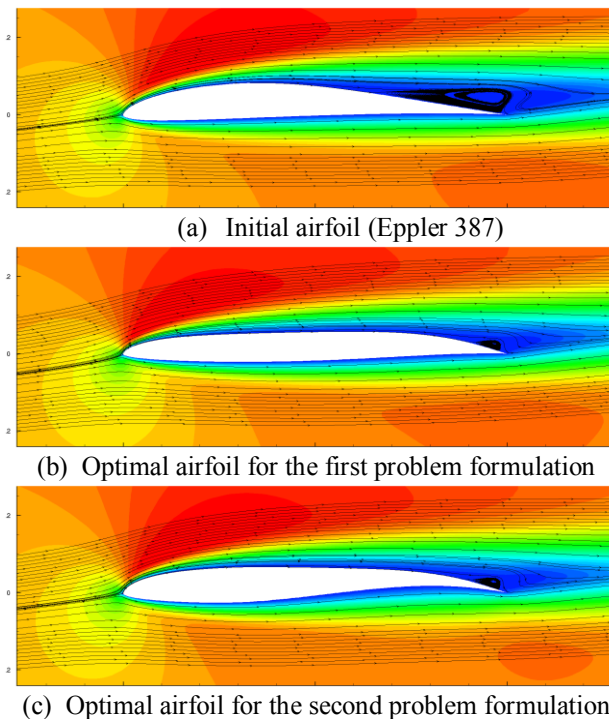


Fig. 11 Velocity magnitude and streamlines.

5 Conclusions and future perspectives

The concept and implementation of the eCFD system have been described. Two Indonesian-Dutch successful collaborative projects have been presented. By the notion of e-mail, eCFD substantiates a low-overhead communicative nature of collaboration involving distantly separated sites.

A complex industrial case, involving a complete turboprop aircraft in a double-slotted high-lift configuration, could be handled effectively. The industry (RAI) was relieved from the burden of formidable specific CFD tasks.

The potential of eCFD to explore the design space in a collaborative design setting has also been demonstrated. The presented case showed that optimization (synthesis) problems can be parameterized in a versatile fashion to facilitate the search for a favorable optimal solution.

As part of the ongoing work within the projects, the eCFD applications may be adapted to the prevailing needs by extending the application templates and parameters. Due to the successive steps of defining the application, the parameters and the template, eCFD promotes the best practice of collaboration based on consensus and mutual understanding, exploiting the CFD technology for robust flow simulations and high confidence results.

Acknowledgment

Thanks are due to Mr. M. Wirata (RAI) for defining the CFD simulations, Mr. B. Nugrohati (RAI) for the CAD modeling, Mr. O.J. Boelens (NLR) for the computational grid, Mr. W.J. Blaauw (NLR) for installing eCFD applications, and Mr. B. Prananta for reviewing the manuscript.

References

- [1] E.H. Baalbergen, J. Kos and W.F. Lammen, Collaborative multi-partner modelling & simulation processes to improve aeronautical product design. *4th CEAS Air & Space Conference*, Linköping, Sweden, September 2013.
- [2] J.W. Boerstoeel, A. Kassies, J.C. Kok and S.P. Spekrijse, ENFLOW, a full-functionality system of CFD codes for industrial Euler/Navier–Stokes flow computations, NLR TP 96286U, 1996.
- [3] O.J. Boelens, CFD analysis of the flow around the X-31 aircraft at high angle of attack, *Aerospace Science and Technology* 20 (2012) 38–51.
- [4] B.B. Prananta, B.I. Soemarwoto, O.J. Boelens, B.S. Kartika and S. Chaedar, Analysis of store separation from a generic twin-engine turboprop aircraft, AIAA paper 2007-4077.
- [5] B.I. Soemarwoto, Airfoil optimization using the Navier-Stokes equations by means of the variational method, AIAA paper 98-32402.
- [6] B.I. Soemarwoto, M. Laban, A. Jameson, A.L. Martins and B. Oskam, Adaptive aerodynamic optimization of regional jet aircraft, AIAA paper 2002-0260.

Copyright Statement

The authors confirm that they, and/or their company or organization, hold copyright on all of the original material included in this paper. The authors also confirm that they have obtained permission, from the copyright holder of any third party material included in this paper, to publish it as part of their paper. The authors confirm that they give permission, or have obtained permission from the copyright holder of this paper, for the publication and distribution of this paper as part of the ICAS 2014 proceedings or as individual off-prints from the proceedings.