CHALLENGES IN EUROPEAN AEROSPACE
5TH CEAS AIR & SPACE CONFERENCE

An automated CFD analysis workflow in overall aircraft design applications

Xiangyu GU
German Aerospace Center, DLR, Air Transportation Systems
Research engineer
Blohmstraße 18, 21079, Hamburg, Germany
E-mail: Xiangyu.gu@dlr.de
Pier Davide Ciampa (DLR, Integrated Aircraft Design, Research Engineer),
Björn Nagel (DLR, Integrated Aircraft Design, Head of Department)

Keywords*: CPACS, Automated Mesh Generation, CFD, multi-fidelity aircraft design*

ABSTRACT

An automated CFD based analysis process for early aircraft development stages is presented. The robustness of the implemented process, which relies on the knowledge based layer implemented into the pre-processing of the geometrical components, allows taking advantage of high fidelity simulations, also for large explorations of the design space. The well-known aircraft configuration DLR-F6 is applied to verify the automated process. The CFD analysis chain is integrated into the DLR multi-fidelity aircraft design synthesis process, by using the DLR open source distributed framework RCE, and the DLR central data model CPACS. An overall aircraft design synthesis is performed for a conventional passenger transportation aircraft configuration, by making use of variable fidelity methods for the aerodynamic analysis. The results demonstrate the impact of using CFD based analysis into overall aircraft design applications

Nomenclature

b = wing span
C_L = lift coefficient $= \frac{lift}{q_\infty S_{ref}}$
C_D = drag coefficient $= \frac{drag}{q_\infty S_{ref}}$
C_D = drag coefficient $= \frac{drag}{q_\infty S_{ref}}$
C_P = pressure coefficient $= \frac{P - P_\infty}{P_\infty}$
C_{ref} = wing reference chord
S_{ref} = reference area
M_∞ = far field Mach number
Re_c = Reynolds number based on C_{ref}
AOA = angle of attack
η = fraction of wing semi-span

1. Introduction

The increasing demand for commercial aviation, the rising fuel price, as well as the growing environmental concerns have become the key drivers in improving aircraft fuel efficiency. The ICAO Programme of Action on International Aviation and Climate Change, which targets a 2% improvement in global fuel efficiency annually until the year 2050[1], and the ACARE Strategic Research and Innovation Agenda (SRIA), are examples of such ambitious targets.

Unconventional aircraft configurations, such as the Blended Wing Body[2] (BWB) and the strut-braced wing[3], are promising candidates to significantly improve the fuel efficiency. However, unlike for conventional aircraft designs, novel configurations suffer from the lack of empirical knowledge. Hence, due to the high development costs and the economic risks associated with unconventional configurations, from the beginning of the design phase it is necessary to correctly predict the configuration's behaviour, in order to guarantee the promised performance.

Most of the current large commercial aircraft operate in the transonic flight regime at the cruise phase. As a result, accurate wave drag assessment is essential for the design trade-off. Currently, the Vortex lattice method (VLM) is widely used to evaluate the aerodynamic performance in the early design stage of the aircraft. However, even if corrections can be applied, it is not capable to account for the wave drag at cruise condition. On the other hand, the increasing computational efficiency, as well as the improvements into Computational Fluid Dynamics (CFD) techniques over the past decades, allows engineers to make use of CFD in order to accurately predict the flow field, even at the critical transonic conditions and within acceptable analysis time. Hence, it becomes possible to employ CFD to evaluate the aerodynamic performance in early design stages. However, and inevitably, introducing higher modelling complexities [4], labour intensive pre-processing of the geometry and higher computational resources cost.

CFD analysis requires an accurate and water-tight representation of the aircraft wetted surface, or Outer Mould Line (OML). Besides, the generation of CFD meshes, requires extensive, and usually manual pre-processing operations of the geometry components. Further, at the early design stages these operations may be repeated multiple times in order to extensively explore the design space, and to perform large geometry variations. Hence, the automation of geometrical pre-processing operations and of the mesh generation step constitutes the main challenges to employ CFD within overall aircraft design applications. Further, MDAO (Multidisciplinary Analysis and Optimization) techniques are often necessary to capture the interdisciplinary dependencies, demanding for increasing robustness and flexibility of the automated CFD based analysis process.

This paper presents an automated CFD based analysis chain, aiming to improve the prediction of the aerodynamic behaviour in the pre-design stages, and bringing CFD analysis into the overall aircraft synthesis process. The process is based on the DLR Common Parametric Aircraft Configuration Schema (CPACS) data modelling.

The paper is organized as follows. The implementation process is presented in section 2. In this section the geometry representation, the automated mesh generator and the CFD solver used in the analysis process are described. In section 3, the analysis process is employed for the analysis of the DLR-F6 wing body configuration to verify the geometry representation and mesh generation. In section 4, the analysis process is used within the aircraft synthesis of a passenger transportation aircraft configuration. Mission analysis results, such as mission fuel, are compared against the synthesis results when employing only empirical based method, and the ones when the aerodynamics characteristics rely on VLM based methodologies. Conclusions and outlook of the article are provided in the last section.

2. Automated CFD based analysis process

In order to foster the collaboration among disciplinary specialists and the integration of disciplinary expertise into the overall aircraft design process, the centralized data structure CPACS[5] has been developed by DLR over last decade. It contains information on the model, such as its geometry description, and holds process data to control the overall analysis workflow. In order to support the handling of CPACS-described geometries to be progressed to the disciplinary analysis, the dedicated library TiGL [6](TIVA Geometry Library) has being developed by DLR. The TiGL Geometry Library which is based on OpenCASCADE kernel represents the airplane's components geometry by B-spline surfaces, and it can export the geometry as CAD based format for further disciplinary analysis.

The analysis components in this study make use of the CPACS model description, with the objective to link it with automated high fidelity analysis capabilities for early design stages. The overall process, starting with the processing of the geometrical CPACS description, to the results of the aerodynamic solution, has been implemented to be flexible and fully automated for arbitrary configuration input. The robustness of the developed process, which relies on the knowledge based layer implemented into the pre-processing components, allows taking advantage of high fidelity simulations, also for large explorations of the design space, as typically required at the early development stages.

The engineering framework chosen for the implementation of the workflow process in this study is the open source integration distributed engineering environment RCE (Remote Component Environment), developed by DLR[7]. A representation of the implemented workflow is illustrated in Figure 1, and the individual components are described in the following sub-sections.

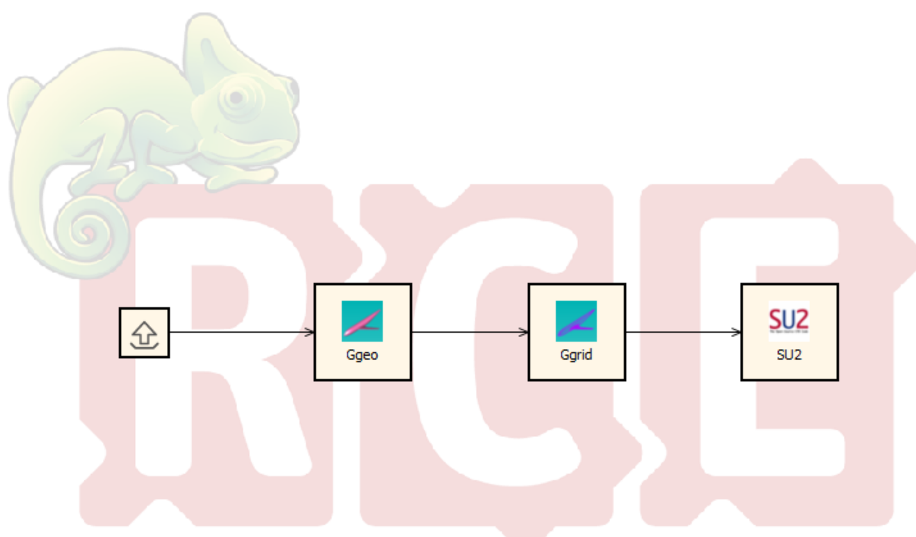


Figure 1: CFD analysis workflow in RCE

2.1 Geometry representation

The first component of the implemented chain, named Ggeo, is responsible to translate the aircraft CPACS description, into a CAD based model suitable for CFD mesh applications. The component automatically generates a CAD models from arbitrary valid CPACS files. The component, which is based on the OpenCascade kernel accessible via API TiGL library, is implemented in Python. Ggeo makes use of the CPACS hierarchical structure in order to identify the aircraft geometry topology, such as the numbers of wings, fuselages and the connectivity between them, then processes the information to the following disciplinary analysis modules.

2.2 Knowledge based Mesh Generator

The mesh generation component, named Ggrid, is an under development Python based tool, which automatically generates macros for the mesh generators, in order to produce Isotropic tetrahedral meshes for inviscid flow simulations, and hybrid or anisotropic tetrahedral meshes for viscous flow simulations. According to the geometrical information incoming from the previous component, the macros will distribute default sources with the pre-implemented knowledge in the “critical positions”, such as the leading edges, the trailing edges of the wings, and the junction of wings and fuselage. In this component, a global factor is implemented to control the grid size settings according to the compromise of computational resources cost and the accuracy of the result. In this study the exported macros are compatible with Pointwise [8] meshing tool.

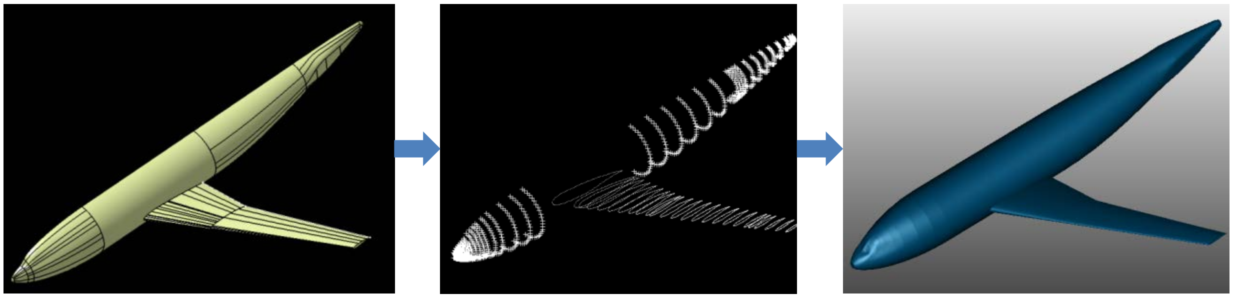
2.3 CFD Solver

As soon as the grid is generated with a suitable format for the CFD solver, it is passed to the CFD solver component. Two solvers are used in this study. The open source CFD solver SU2 [9] is chosen for in-viscous analysis, and ANSYS Fluent[10] solver is adopted for viscous simulation. SU2 is a finite-volume, cell based unstructured CFD solver. In this article, Jameson-Schmidt-Turmel (JST) scheme [11] augmented with arterial dissipation is used for the spatial discretization. In Fluent, a Density based unstructured solver, cell based method was chosen. Second order upwind spatial discretization is used to calculate convective fluxes. For viscous term, one-equation Spalart–Allmaras (SA) turbulent model is used. All the needed input scripts, and settings for the solvers aregenerated by the components as an automated process as well.

3. Verification

3.1 Geometry and computational grids

In order to verify the described process, the implemented chain is initially applied to the well-known test case DLR-F6 wing body configuration. A CPACS file of the DLR-F6 model has been assembled by the extracting the coordinates of the points from the original DLR-F6 IGES CAD file from 2nd AIAA drag predication workshop (DPW)[12]. The process shows in Figure 2.



(a)DLR F6 IGES from AIAA DPW (b) profiles in each slices (c) CPACS configuration in TIGL

Figure 2 : CPACS DLR F6 configuration initialization

The summarized reference data for the DLR-F6 are reported in Table 1.

C_{ref}	141.2mm
$S_{ref}/2$	727,700mm^2
b/2	585.647mm
$\boldsymbol{M_\infty}$	0.75
$\boldsymbol{Re_c}$	3×10^6

Table 1: reference quantities for DLR-F6

In order to investigate the quality of the CAD geometry generated by the CPACS-TiGL implemented process, the RANS analysis performed with aforementioned chain is compared with the solution obtained with the original DPW CAD file. Both of the grids have approximately 2 million of cells to facilitate the results comparison. Figure 3 shows the mesh used in CFD simulation grid generated for the TiGL geometry on the surface and symmetry plane.

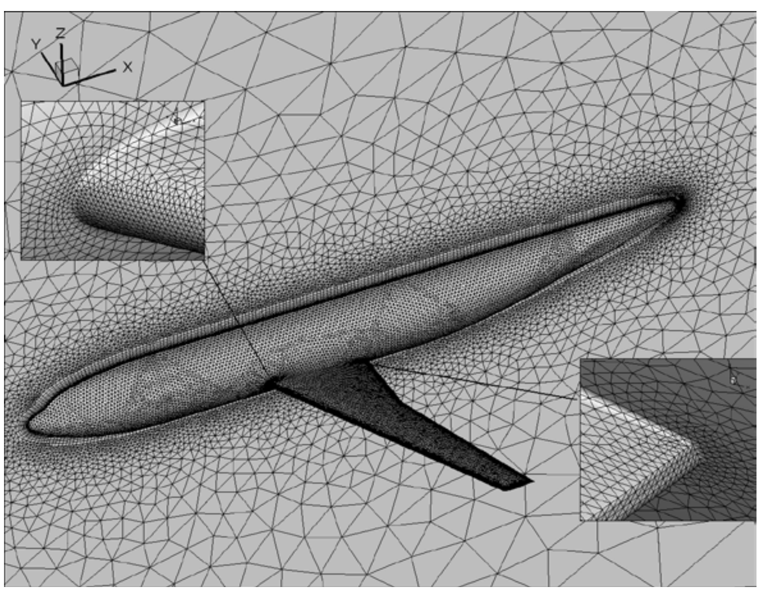


Figure 3: DLR-F6 surface and symmetry mesh

3.2 Verification results

Table 2 reports the C_L, C_D from both simulations at AOA = 0.49, Ma=0.75, $Re_c=3\times e10^6$ Wind tunnel experimental data are also reported as reference values. A distinction is made according to the source of the geometry CAD input file (original DPW provided and the one generated by the CPACS-TiGL based component).

DLR_F6	DPW_RANS*	TIGL_RANS*	EXP
C_L	0.529287	0.532579	0.49
C_D	0.0372347	0.0376182	0.00293

* DPW_RANS and TiGL_RANS are RANS solution for original DPW IGES RANS and TiGL IGES respectively

Table 2: Comparison of C_L and C_D

Overall, the flow solver has predicted very close values of C_L and C_D with two different input geometries. Main source of the difference is the representations of the wing tip between the CPACS based CAD model and the original CAD geometry.

Figure 4 shows the C_P comparison of wind tunnel experimental data against the simulation results for both the input CAD geometries at several wing's span-wise stations. The flow solver can accurately predict the coefficient of pressure at each span-wise section for both the geometries. The discrepancy between the solution for DPW and the TiGL configuration is small, which indicates the CPACS-TiGL based CAD file provides a good representation of the original configuration. Further, the CFD results match well with the experimental data for both the geometries, which suggests the automated mesh generation process provides a good discretization of the configuration, as well as of the flow field.

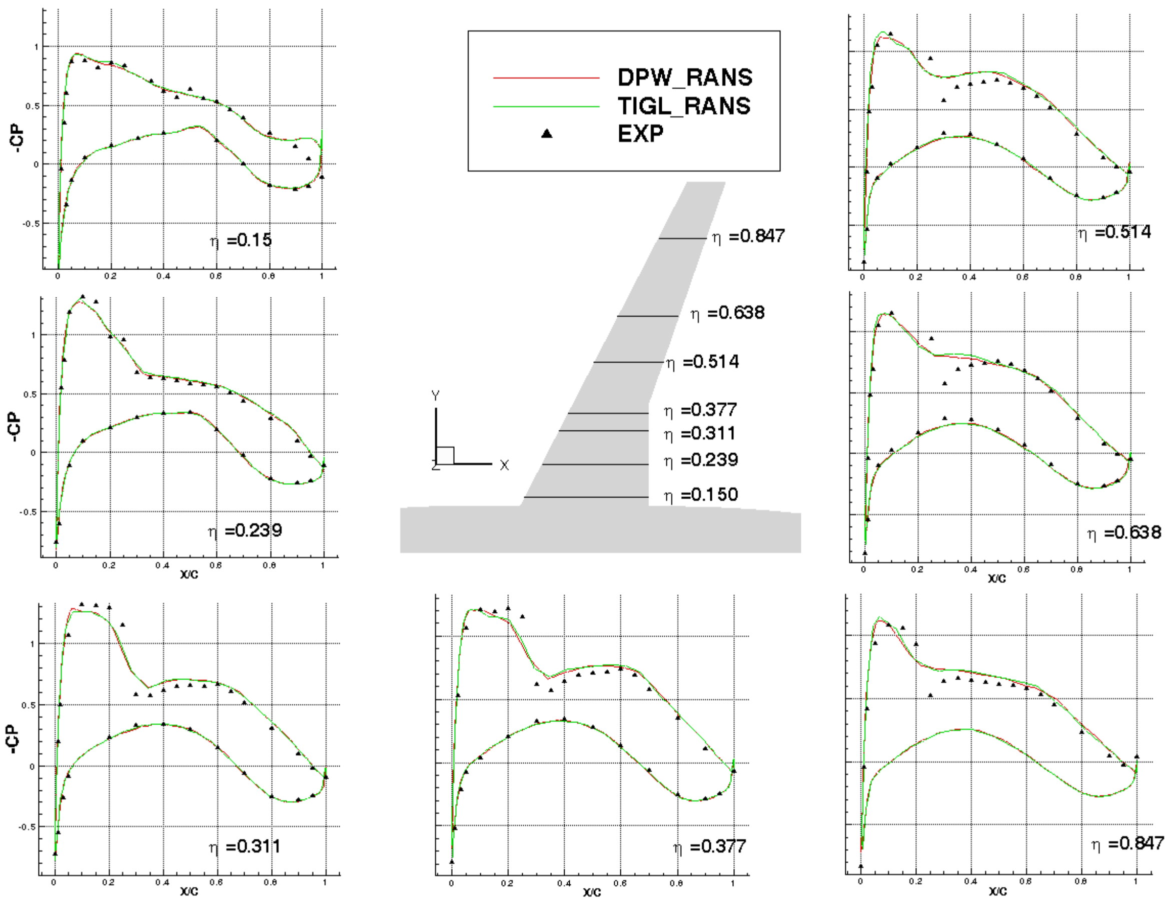


Figure 4: Comparison of wing surface pressure distributions at Ma = 0.75, AOA = 0.49.

4. CFD application in overall aircraft design

In this section the aforementioned CFD analysis chain, is applied to provide aerodynamics performance, within an overall aircraft design task. A short to medium-range transport aircraft is selected as a test case to demonstrate the impact of the implemented CFD automated chain over the aircraft synthesis process. A grid refinement study is made to determine the resolution accuracy of this mesh. The overall fuel burn obtained by the mission analysis making use of the CFD computed polars is compared with the results obtained by using the VLM solution, and against a pure empirical based synthesis.

4.1 Design workflow

The overall aircraft synthesis process setup in this work is based on a multi-fidelity architecture, in order to account for the CFD based analysis process described into the previous sections. The implemented design workflow architecture is shown in Figure 5.

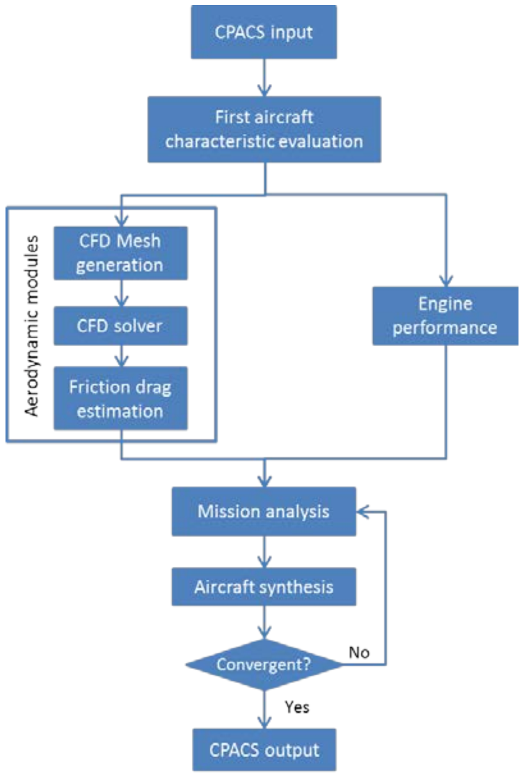


Figure 5: Workflow of the design process

In the design workflow, Top Level Aircraft Requirements (TLAR), are specified for the synthesis. The first module is the conceptual aircraft design tool, VAMPzero [13], which is used as aircraft initializer in order to provide the initial overall synthesis of the aircraft performance, such as the fuel consumption and operating empty mass (OEM). Based on a multi-fidelity architecture, the program allows making use of the aircraft performance values evaluated by external tools. If some aircraft characteristics are already defined in the input dataset, they will be directly inherited and not recalculated by VAMPzero analysis modules. This feature allows integrating the presented CFD based analysis process, within the overall synthesis, instead of using the aerodynamics characteristics estimation available internally to the conceptual tool.

To provide affordable solution, the aerodynamic performance used in this design study is obtained by solving the Euler equations. As a result, the skin friction drag is not accounted into the CFD based results. However, in order to obtain realistic mission fuel values, an estimation of the friction drag is obtained by a method based on the flat plate equivalency for the aircraft components.

An available representative engine is also chosen to provide performance maps of fuel flow and thrust for pre-defined engines depending on the flight conditions, i.e. Mach number, altitude and thrust setting.

For the Mission analysis, FSMS analysis tool is chosen to simulate an aircraft's flight on a given flight mission profile, and to determine the mission block fuel of the design mission depending on the given aerodynamic polars, the engine performance and the aircraft geometry.

After the configuration is initialized by conceptual design module, the resulting model is progressed to the other analysis components in the workflow. The aerodynamic performances are evaluated with the described CFD analysis chain. The design workflow architecture allows using tools with different levels of fidelity, such as a VLM method, in order to evaluate the aerodynamic performance for the synthesis process. Afterwards, the aerodynamics performances are modified by taking into account friction drag estimation. Hence, FSMS is used to calculate an updated, and more detailed, mission fuel mass based on the conceptual results (e.g. for the design masses), and on the CFD analysis (for the aerodynamics). With the updated mission fuel mass, the design is forwarded once more to the synthesis process, in order to account for the updates provided by the aerodynamics and mission modules, and to perform an updated synthesis of the aircraft. Hence, with the updated values of OEM, and MTOW, a new mission analysis is performed. The design loop is executed till the convergence of the design masses (OEM, MTOW, and Fuel Mass). In this way the convergent solution accounts for all the snowball effects in the aircraft synthesis process.

4.2 Test Case

The configuration used in this design case is the D150, which is an A320 like aircraft and has been used as baseline aircraft in previous studies [14] [15].Figure 6 shows the initialized configuration, which is in CPACS format, and visualized by the CPACS geometry interpreter TIGLViewer.

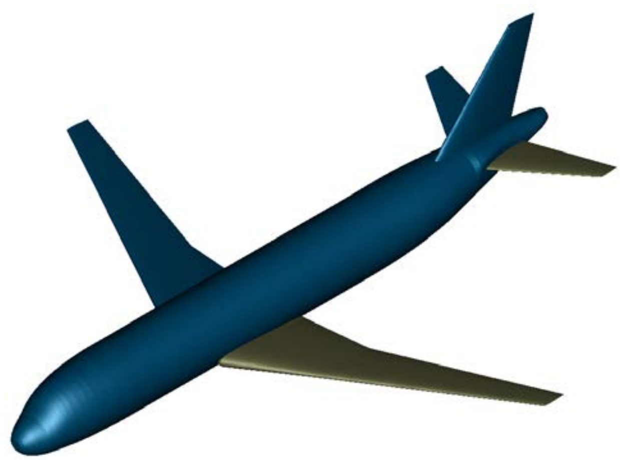


Figure 6: D150, as visualized in TIGLViewer

The main top level aircraft requirements (TLAR) are reported in Table 3.

Parameter	Value
Design range (Km)	4000
PAX	150
Mach cruise	0.78
Altitude (m)	11000

Table 3: TLAR for D150

Before the design case is carried out, a grid refinement study is made. A sequence of three refined grids with grid sizes ranging from 0.9 million cells to 2 million cells, named coarse, medium, and fine respectively is generated by varying the global factor defined in Ggrid.

An angle of attack from -4° to 8° is run for each grid to generate drag polars at the cruise Mach number of Ma=0.78. The polars are shown in Figure 7. It is clear that the coarse grid is not sufficiently resolved to match the other two polars. However, the medium and fine grids are nearly indistinguishable from each other except at the higher lift coefficients.

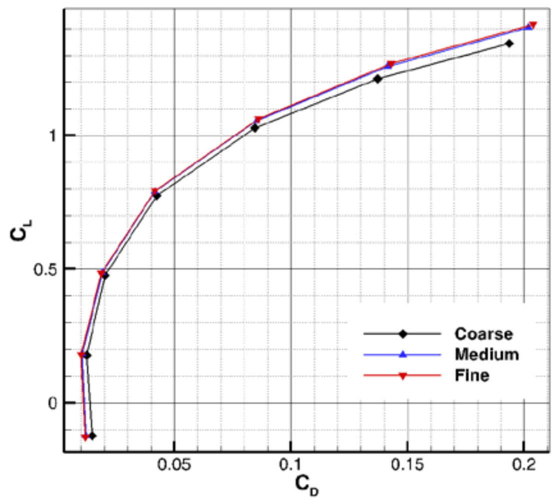


Figure 7: Drag polars for 3 levels of refinement

A description of all grids used in this work, as well as the C_D for each grid at C_L=0.5 in Ma=0.78 are given in Table 4. The medium grid offers substantial computational savings compared to the fine grid and with acceptable accuracy. In order to provide an efficient evaluation of the aerodynamic performance the medium grid is used in the later aircraft synthesis study.

Grid	Surface Cells	Cells	C_D
Coarse	61,335	913,568	0.0221
Medium	100,517	1,368,046	0.0200
Fine	159,511	2,114,627	0.0197

Table 4: Mesh sizes and C_D at C_L=0.5 in Ma=0.78

4.3 Synthesis Results

The overall aircraft synthesis results are compared for three cases:
1) Pure conceptual based synthesis
2) Multi-fidelity synthesis with VLM based aerodynamics
3) Multi-fidelity synthesis with CFD Euler based aerodynamics

Figure 8 shows the comparison of drag polars between VLM method and Euler simulation for a set of Mach numbers which range from 0.2 to 0.78. As expected, the differences between VLM and Euler polars in subsonic regime are relatively small. However, in the cruise condition, due to the wave drag, the difference is substantial. For example, at C_L=0.5, the difference of the drag can up to 100 drag account.

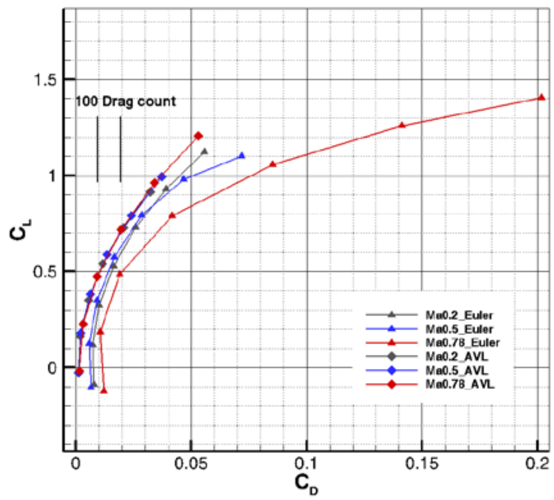


Figure 8: Comparison of the drag polars with VML and Euler simulation

The result of the synthesis process, such as the take-off mass (mTOM), fuel mass (mFM) and operating empty mass (OEM) for three synthesis cases are shown in Table 5.

Mass	Conceptual	VLM*	Euler CFD*
mTOM [kg]	76168	-2.5%	+6.0%
mFM [kg]	13142	-12.9%	+29.5%
OEM [kg]	40527	-0.1%	+0.4%

*respect to conceptual values

Table 5: synthesis results

With extensive available database for conventional configurations, the conceptual synthesis process is calibrated on real aircraft data. As a result, the conceptual design results provide a good reference for comparison. Both of the multifidelity synthesis results, which make use of VLM and Euler CFD simulations, show a difference with the conceptual design case, as shown in Table 5. The main difference is in the fuel consumption at cruise condition, resulting by the difference in the predicted drag, in the three cases values.

Further, due to the snowball effects accounted into the iterative synthesis process, the drag difference results into a different trimming condition at the cruise condition, which is reported in Table 6.

	Conceptual	VLM	Euler CFD
C_L	0.584	0.562	0.623
C_D	0.0304	0.0271	0.0453

Table 6: C_L and C_D in synthesis cruise condition

It is obviously that the under estimation of fuel for VLM case is due to the absence of the wave drag. On the other hand, the Euler case tends to give a higher fuel consumption value. The overestimated wave drag at cruise condition results into an increased fuel consumption, and thrust requirements, leading to a higher C_L values to trim the aircraft at the different mission points. Further, as the wave drag predicted by the Euler simulation is highly sensitive on the wing's geometrical representation, it is crucial to provide suitable input file to the CFD based synthesis process. Hence, the transition from the conceptual to the CFD based analysis, needs to account for an enhancement of the geometry quality as well. The automated process here presented may be further exploited to generate CFD quality shapes, within the multi-fidelity synthesis at the early design stages, and avoid redesign processes at the later stages.

5. Conclusions and Outlook

In this paper, an automated CFD analysis chain is implemented, which aims to improve the prediction of the aerodynamic behaviour for conventional and unconventional aircraft configurations in the pre-design stages, and bringing CFD based analysis into the overall aircraft synthesis process. The chain is verified with the well-known test case DLR-F6. The results shows the chain provide a high quality representation of geometry and a good simulation of the flow field.

With the centralized CPACS data modeling, a multi-fidelity aircraft synthesis process is implemented by making use of automated CFD based analysis process deployed in RCE framework. The design synthesis is performed with different levels of fidelity. As expected, by taking wave drag into account, the synthesis results with Euler simulation shows higher fuel consumption compare with VLM results. Further, by giving an overestimated drag, the CFD simulation results into much higher mission fuel consumption compared with the purely conceptual design method. Source of the drag overestimation is also due to the representation of the wing design, resulting from the conceptual synthesis (as expected). Hence, this study highlights on of the complexities faced by the designer when introducing physics based analysis in the predesign stage, and the need to initialize geometries suitable to the analysis modules.

Nevertheless, in order to have a better understanding on the aircraft characteristics, it is of great meaningful to introduce CFD based analysis into the overall aircraft design, especially for unconventional aircraft configuration, where the flow physics requires deeper investigations. Further, introducing automated CFD based analysis into the early aircraft synthesis, is expected to minimize the re-design activities at the later stages.

In the following studies, it is expected to use aerodynamic shape design to initialize a suitable aircraft configuration for OAD making use of CFD analysis and optimization techniques. Effective methods are reported in this field, which make use of gradient-based optimization algorithm in conjunction with an adjoint method for the computation of the required shape derivatives [16].

6. References

[1] ICAO. Aviation and Climate Change. International Civil Aviation Organization (ICAO) Environmental Report, 2010
[2] R. H. Liebeck, Design of the blended wing body subsonic transport, Journal of Aircraft, 41:10-25, 2004.
[3] Joel M. Grasmeyer, Multidisciplinary design optimization of a strut-braced wing air-craft, Master's thesis, Virginia Polytechnic Institute and State University, Blacksburg, VA, April 1998.
[4]P. D. Ciampa, B. Nagel, P. Meng, M. Zhang, and A. Rizzi, "Modeling for physics based aircraft predesign in a collaborative environment," presented at the 4th CEAS Air & Space Conference, Linköping, Sweden, 2013.
[5] Liersch, C.M., Hepperle, M., A Distributed Toolbox for Multidisciplinary Preliminary Aircraft Design, CEAS Aeronautical Journal, Vol. 2, p. 57 – 68, Springer, 2011.

[6] A. Bachmann, M. Kunde, M. Litz, A. Schreiber, Advances in Generalization and Decoupling of Software Parts in a Scientific Simulation Workflow System, German Aerospace Center (DLR) Simulation and Software Technology, 2010
[7]https://code.google.com/a/eclipselabs.org/p/rce
[8]http://www.pointwise.com
[9]F. Palacios, T. D. Economon, A. Aranake, S. R. Copeland, A. K. Lonkar, T. W. Lukaczyk, D. E. Manosalvas, K. R. Naik, S. Padron, B. Tracey, A. Variyar, and J. J. Alonso, “Stanford University Unstructured (SU2): Analysis and Design Technology for Turbulent Flows,” in 52nd Aerospace Sciences Meeting, National Harbor, Maryland, 13-17 January 2014.
[10]http://www.ansys.com/Products/Simulation+Technology/Fluid+Dynamics/Fluid+Dynamics+Products/ANSYS+Fluent
[11] Jameson, A., Schmidt, W., and Turkel, E., Numerical Solution of the Euler equations by Finite Volume Methods Using Runge Kutta Time Stepping Schemes," 14th AIAA, Fluid and Plasma Dynamics Conference, 1981.
[12] http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw/Workshop2
[13] Böhnke, D., Nagel, B., Gollnick, V., “An Approach to Multi-Fidelity in Conceptual Aircraft Design in Distributed Design Environments”, 32nd IEEE Aerospace Conference, Big Sky, 2011.
[14] Ciampa, P.D., Zill, T., Nagel, B., “Aeroelastic Design and Optimization of Unconventional Aircraft Configurations in a Distributed Design Environment”, AIAA-2012-1925, 53rd AIAA/ASME/ACSE Structures, Structural Dynamics and Materials Conference, Hawaii, 2012
[15] Zill, T., Ciampa, P.D., Nagel, B., “A Collaborative MDO Approach for the Flexible Aircraft”, 54th AIAA/ASME /ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, Boston, 2013
[16] F. Palacios, T. D. Economon, A. D. Wendorff, and J. Alonso. Large-Scale Aircraft Design Using SU2. In 53st AIAA Aerospace Sciences Meeting, AIAA 2015-1946, Kissimmee, Florida, USA, 5–9 January 2015.