PAPER • OPEN ACCESS

Redesign of a closed-loop high-speed facility to test distortion generators

To cite this article: Manas Payyappalli *et al* 2023 *J. Phys.: Conf. Ser.* **2511** 012019

Journal of Physics: Conference Series **2511** (2023) 012019 doi:10.1088/1742-6596/2511/1/012019

Redesign of a closed-loop high-speed facility to test distortion generators

Manas Payyappalli, Elissavet Boufidi, Riccardo Toracchio and Fabrizio Fontaneto

von Karman Institute for Fluid Dynamics, Rhode-Saint-Genèse, Belgium

E-mail: manas.madasseri.payyappalli@vki.ac.be

Abstract. Aircraft engine architectures are currently in the phase of a change to meet future market demands. Certain such novel architectures force engines to operate under distorted inflow conditions, which are undesirable during flight. It is important to characterize the effect of such inflow distortions in order to understand the impact on the performance of engine components. This paper discusses the design and development of a novel test-facility that has the capability to test combined total pressure and swirl distortion generators under desired flow conditions. The return duct of the high-speed closed-loop compressor rig R4 at the von Karman Institute for Fluid Dynamics (VKI) has been redesigned to incorporate a test-section where distortion generators can be tested and characterized. Thus, a compressor test-rig is modified to act also as a wind tunnel capable of testing distortion generators at engine-like conditions.

1. Introduction

Inlet distortion is the phenomenon that occurs when flow entering an aircraft engine is non-uniform either due to different flying conditions or due to the typical engine architecture. In the immediate future, Ultra High Bypass Ratio (UHBR) fan configuration, and in the long term, Hybrid Wing Body or Boundary Layer Ingesting (BLI) concepts remain promising aircraft engine architectures [1]. These engine configurations will be subjected to even much higher levels of distortion at the fan inlet than the current configurations, with possible time variations of the distortion patterns. If these inlet distortions are not properly considered during the engine design process, they can drastically reduce the fan and compressor stall margin and increase the risk of engine failure. Quantifying distortion and its impact on engine performance is a continuous source of concern for designers. Therefore, distortion tests are performed at different steps of the engine design process.

The European Union (EU) funded ASTORIA project has the objective to develop a new methodology to design distortion generators replicating combined swirl and total pressure distortion patterns. Figure 1 shows an example of a total pressure distortion generator. Using such devices, the effect of realistic inlet distortions on the performance and stability of engines can be investigated. The ability of these devices to reproduce the target flow conditions has to be understood to characterize the flow conditions downstream. The present paper describes the steps involved in the redesign of a facility enabling it to test distortion generators and provides details on the methods and tools employed. The paper first discusses a preliminary assessment carried out to check the capability of the facility to deliver the required flow conditions at the

Journal of Physics: Conference Series **2511** (2023) 012019 doi:10.1088/1742-6596/2511/1/012019

Figure 1: A total pressure distortion generator

proposed test article (distortion generator) location. The paper later discusses the steps involved in the design of the novel facility and the major modifications carried out. The entire design exercise is targeted to guarantee a uniform inlet flow pattern at the test article location along with the objective of reducing the losses in the facility as much as possible. In brief, the objective of this study is to develop a test-facility that can test distortion generators at a wide range of Mach numbers and desired Reynolds numbers that represent real engine conditions. This paper thus outlines the basic design methodology adopted to develop the novel test-facility with a constant-area cylindrical test-section.

2. Methodology and design philosophy

To realize the objective of this study, the idea is to utilize the closed-loop compressor facility, R4 at VKI such that its return duct is capable of testing distortion generators at engine representative conditions. The return channel was chosen for the novel test-section installation owing to easy modification and access to the facility, and to allow good control on the flow conditions at the test-section inlet. A schematic of the R4 original test facility is shown in Figure 1.

2.1. Evaluation of the original test facility

2.1.1. VKI R4 test facility The VKI R4 test-facility is a high-speed closed-loop compressor test-rig (see Figure 2). The compressor stage is driven by a DC motor that can deliver a maximum power of $\tilde{7}00$ kW and the shaft rotational speed is controlled by gear boxes. The flow from the plenum follows a smooth convergent bell-mouth that gently guides the flow to the compressor

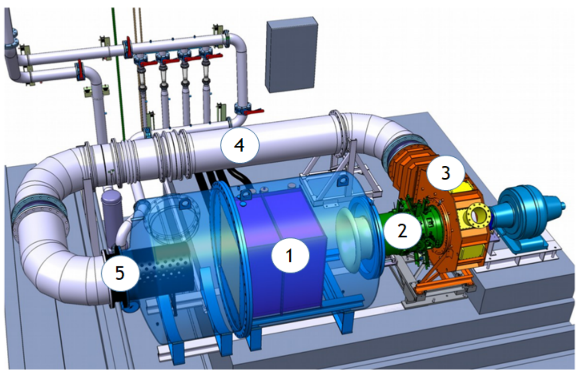


Figure 2: Fig. 1. Schematic of the VKI R4 original test facility – 1. Plenum 2. Compressor stage (original test-section) 3. Collector 4. Return duct 5. Throttle valve

Table 1: Parametric analysis for M = 0.60 at test article location

$\pi_c/\pi_{c,\,design}$	$D/D_{return\ duct}$	Bypass ratio	Stall margin
0.99	0.33	129.4	14.3
0.99	0.38	97.8	14.3
0.99	0.41	82.8	14.3
1.01	0.33	120.8	6.75
1.01	0.38	91.4	6.75
1.01	0.41	77.3	6.75

inlet. After the compressor stage, the flow is discharged into a collector directly connected to the return duct. The flow is then delivered back to the plenum where the rig throttle valve is located. After the expansion in the throttle valve, the flow enters a heat-exchanger with a honeycomb structure, serving the purpose of both controlling the stage inlet temperature and damping any residual swirl component. Due to its closed-loop arrangement, the R4 facility allows to finely adjust the loop mean pressure level. This is achieved by either pressurizing the system (up to 3 bar) or by employing vacuum pumps able to reach pressures as low as 0.3 bar.

2.1.2. Preliminary assessment As specified earlier, the test article is planned to be installed in the return duct (Component 4 in Figure 2) of the facility. However, a preliminary assessment to check the capability of the compressor to deliver the required mass flow at the novel test-section location need to be performed. Total quantities at the outlet of the compressor were imposed as inlet conditions to the novel test-section by means of inviscid assumptions, and isentropic relations were employed to retrieve the required mass flow for the specified Mach number. The mass flow rate required at the novel test-section is here compared with the mass flow delivered by the compressor through a bypass ratio parameter:

$$Bypass\ ratio = \frac{\dot{m}_{compressor}}{\dot{m}_{test-article}} \times 100 \tag{1}$$

A bypass ratio lower than 100% is representative of a condition in which the test-article would need larger mass flow than the one that the compressor can provide at that operating condition, and represents therefore a non-feasible condition for the facility.

The preliminary feasibility assessment was carried out by parametrically varying the stage total-to-total compression ratio, the diameter of the test-section, and the inlet Mach number at the test-section, while mapping the bypass ratio and the stage stall margin. The output of this design exercise for a Mach of 0.60 at the test article location is shown in Table 1.

One of the outcomes of this assessement is that the compressor stage operating point should be set for satisfactory stall margin. The maximum test-section diameter that would allow a feasible bypass ratio with good measurement resolution for optical measurement techniques was also obtained. Yet, it is observed that to achieve the desired target Mach numbers at the test article location, the diameter must be lower than the original diameter of the return duct. Thus, this assessment checks if the return duct delivers the required flow conditions and if any modifications need to be carried out on the facility.

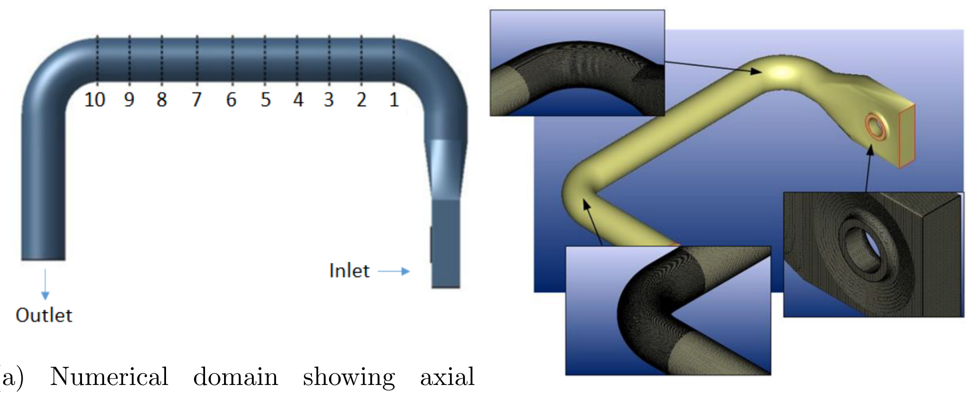
2.1.3. Numerical simulations of R4 original facility The next step in the design phase was to carry out a numerical simulation of the original test facility. The numerical simulations would give an idea about the homogeneity of flow in the return duct and suggest the optimal axial location for the new test-section targeting a flow condition as clean as possible at the test-section inlet. The numerical domain of the original test facility showing the mesh used for the simulations is presented in Figure 3. The numerical domain reproduces the real geometry of the facility from the outlet of the compressor to the inlet of the plenum, and contains therefore the collector and return duct presented in Section 2.1.1. The inlet of the numerical domain was specified with total pressure, total temperature, flow angle, and turbulence quantities, all measured at the stage outlet. At the outlet of the numerical domain, the respective mass flow rate was imposed as the boundary condition. The simulations were carried out with a grid-independent mesh of 35.5 million cells, ensuring a well resolved boundary layer with $y+$ everywhere smaller than 2. The total pressure and swirl distortion indices [2] at different streamwise locations in the return duct were examined to identify the most suitable location for the novel test-section (see Figure 3a). This study also helped to understand how the components upstream will need to be redesigned to ensure uniform flow conditions at the novel test-section. The non-uniformity in the flow was characterized by SAE ARP 1420 [2] total pressure and swirl distortion descriptors. These distortion indices were chosen to bring not only information concerning distortion strength and severity, but also to understand the pattern and distribution of total pressure and flow angle at the desired location, to support therefore the successive steps of the design process.

The numerical simulations concluded that the circumferential distortion indices reduced along the streamwise direction from axial location 1 to axial location 10 due to flow re-organization (Figure 4). The total pressure distortion levels were significantly low whereas the swirl distortion levels were found to be relatively high. Therefore, solutions to reduce swirl levels need to be adopted during the design of the novel test-facility. It is therefore also clear that the location where the smaller distortion intensity present is between section 3 and 6 (such that it also allows to be sufficiently far from the 90° bends of the return duct).

However, it should be noted that the exact location of the test-section is later chosen according to specific constraints raised during the detailed design process.

2.2. Redesign of the novel test facility

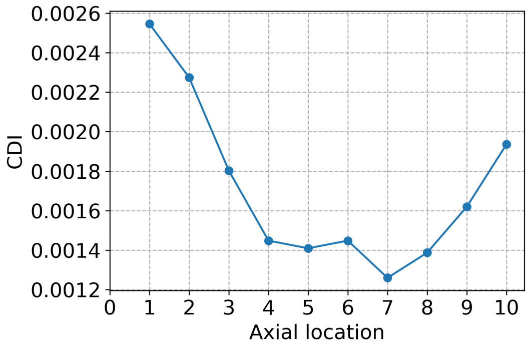
In view of the outcomes obtained from the preliminary assessment and the numerical simulations, it was decided to redesign the original compressor rig to include a bypass duct, flow conditioners, and a convergent channel. Since the convergent reduces the diameter of the return duct, it was essential to also add a diffusing region to take the return duct diameter back to the original diameter after the test-section. This section explains the methodology adopted to design the novel test facility. In brief, the design makes use of available literature, a compressible lumped parameter loss assessment model, and numerical simulations (Figure 5). This section will



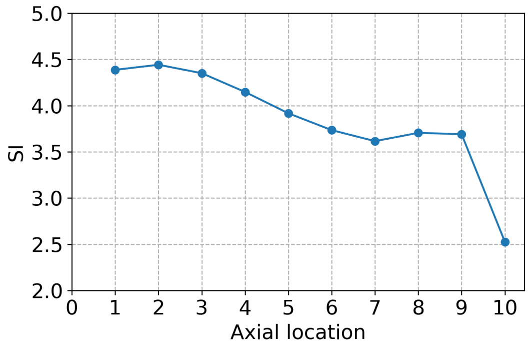
(a) Numerical domain showing axial locations investigated

(b) Details of mesh

Figure 3: Numerical domain and details of mesh of the original return duct of R4



(a) Circumferential distortion intensity



(b) Swirl intensity

Figure 4: Variation of distortion indices along streamwise direction

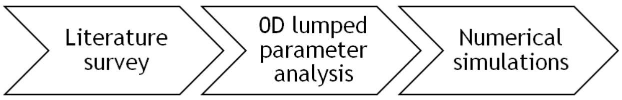


Figure 5: Approach to design of novel test facility

focus on the design of the convergent section, diffusing section, and a final assessment through numerical simulations and the compressible loss estimation tool.

2.2.1. Convergent section The test-section diameter needed to accelerate the flow and reach the desired Mach number was computed in the preliminary design, and thus the contraction ratio of the convergent is directly determined. However, the design of the convergent was to reduce the losses in the component and fix its length. The main loss source within a convergent is due to the boundary layer development [3] [4] with possible separations occurring for too aggressive designs, as reported by Chmielewski [5]. From the calculated contraction ratio and available empirical data (Figure 6), the minimum length of the convergent required to avoid separations was inferred. Chmielewski [5] suggests to employ an inlet contour radius equal to 60% of the outlet one; by doing so, the axial extent of the adverse pressure region can be limited, in favor of an improved efficiency. From the length and contraction ratio, the semi-aperture angle of the convergent can be calculated. This angle will lead to the estimation of the losses of the convergent as seen in Figure 7.

The presence of the convergent section should enhance the flow uniformity at the test article location. However, it is likely that non-uniformity could still exist owing to the thickness of the boundary layer. This is also reported by Chmielewski [5] as shown in Figure 8. This led to adopting a boundary layer control system to deliver a uniform profile at the test article location. The basic idea of realizing the boundary layer control system was using the principle of tangential blowing. With this technique, the momentum deficient in the boundary layer can be added by using a jet of air that is blown tangentially into the boundary layer. The steps involved in the design of the boundary layer control system were to find the most suitable position for injection, to size the blowing slot, and to find the injection angle.

The boundary layer control system was numerically simulated by placing the injector at three different locations - at the inlet of the convergent section, at the inflection point within the convergent section, and at the exit of the convergent section. It is interesting to note that the injection system placed at the inlet and at the inflection point of the convergent were not able to successfully control the boundary layer at the test article location. However, the injector placed

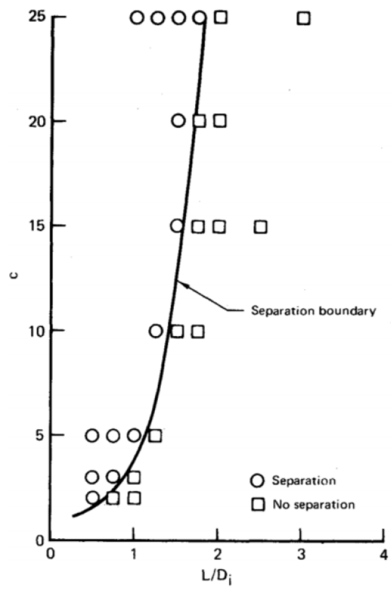


Figure 6: Separation lines at varying contraction ratios (c) and length of convergent (L/D$_i$) [5]

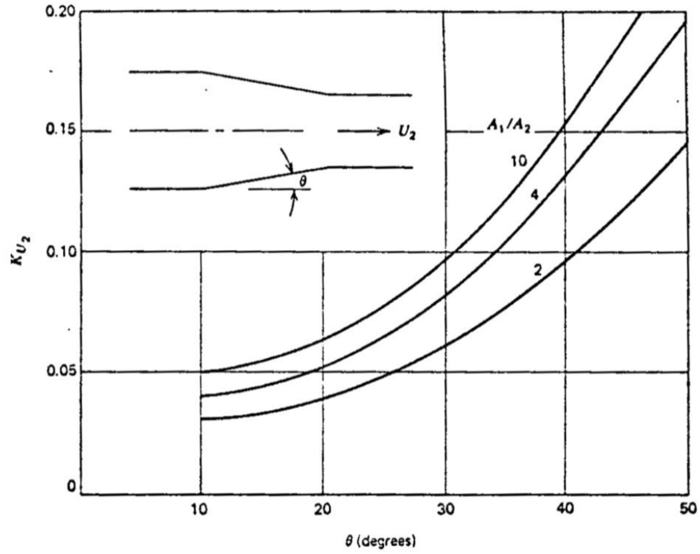


Figure 7: Loss estimation of a convergent [5]

at the exit of the convergent considerably enhanced the uniformity of the velocity profile at the test-article location, as shown in Figure9. In this configuration, the boundary layer displacement thickness at the test article location reduces to approximately 1/3 as compared to the thickness without injection (Figure 10). The sizing and angle of injection of the boundary layer control system were also parameterically varied and simulated to achieve a realistic design of the system, also keeping in mind the manufacturing constraints that could arise for a tangential injection case.

2.2.2. Diffuser and diffusing bend Since the test-section diameter is smaller than the original diameter of the return duct, a diffuser is essential to recover the static pressure and to connect the

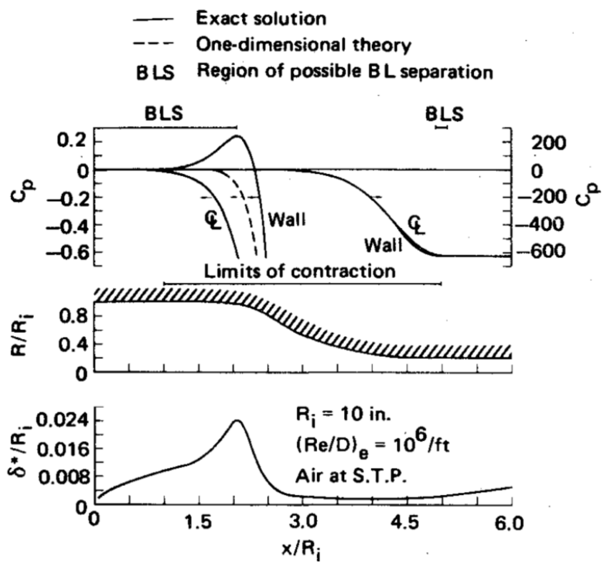


Figure 8: Boundary layer growth in a convergent [5]

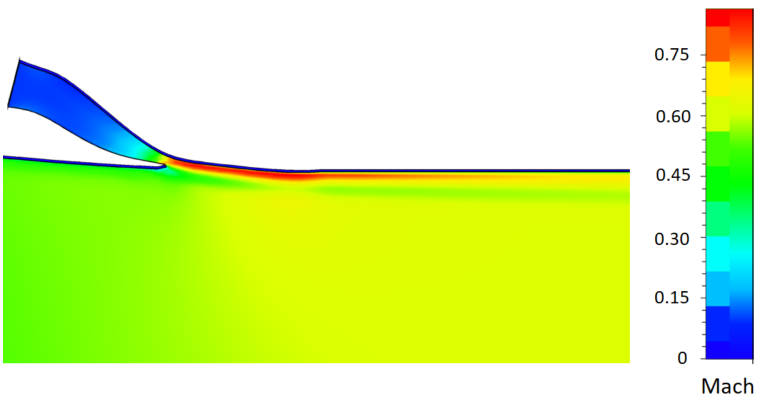


Figure 9: Location of the boundary layer control system. The figure is colored with Mach contours.

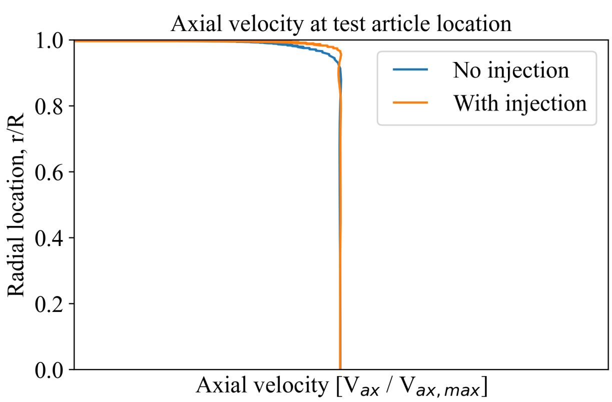
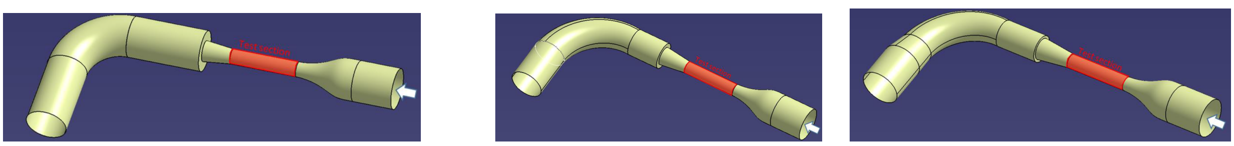


Figure 10: Axial velocity distribution at test article location

test-section back to the original return duct. Designing a diffuser is challenging because these devices exhibit a high tendency of flow separation because of the adverse pressure gradients generated along its length. The design of an efficient and robust diffuser therefore challenges to ensure the shortest length while preserving a high flow separation margin. Different empirical models are available according to the considered geometry, layout and, for some cases, the thermodynamic conditions of the flow. Even though care has been taken in selecting the most appropriate modelling for the case under consideration, the empiricism of the available low-order models forces a more precise evaluation of the diffuser performance by CFD validation. Given the large number of diffuser layouts available in literature, a preliminary analysis was carried out with the objective of comparing the performance of different geometries at given flow conditions. Only the diffuser geometries with available loss coefficient correlations were considered for the assessment. Loss coefficient definitions have then been calculated using the compressible loss evaluation tool, allowing to parametrically compute the efficiency of the diffusers for various geometrical dimensions. The cropped diffuser configuration provides the shortest diffuser length at a fixed loss coefficient magnitude, with results in agreement between the different correlations employed in literature. Given the superior performance when compared to other layouts, the cropped diffuser configuration is therefore selected as the final design candidate.

An interesting solution from the point of view of the facility integration corresponds to the adoption of a cropped diffuser coupled with a diffusing bend which was originally investigated by Miller [6]. Thus, three main candidates were selected (Figure 11) for numerical simulations to finalize the best configuration – (i) cropped diffuser (ii) cropped diffuser + diffusing bend (aspect ratio = 1.5) (iii) cropped diffuser + diffusing bend (aspect ratio = 2). The criteria selected to identify the best design solution was to check the configuration with minimum total pressure loss introduced in the return-duct. Among the analyzed cases, configuration (iii) cropped diffuser + diffusing bend (aspect ratio = 2) was found to clearly outperform the other two configurations in terms of the overall losses. Thus, a cropped diffuser connected to a diffusing bend of aspect ratio 2 was chosen as the final candidate. It is important to specify that as a result of the previous design step, the convergent section was kept constant among all considered configurations and the main element introducing losses was the cropped section 2.

2.2.3. Loss assessment model and numerical simulations of novel facility To assess the pressure losses along the novel facility, a compressible lumped parameter reduced-order model was developed, and CFD simulations of the entire facility were performed. The reduced order model and the CFD were used to assess the overall pressure budget obtained with the novel facility. Pressure budget is the total pressure available at the inlet of the throttle valve to safely expand the flow to the plenum chamber. The reduced-order model was implemented to be strongly dynamic to easily modify the architecture of the return channel, and retrieve global parameters such as the overall pressure loss, the total pressure reduction introduced by every component, and the Mach number at the inlet of the novel test-section. For each component of the novel return-duct, the geometrical parameters were specified and pressure loss correlations



(a) Cropped diffuser (b) Cropped diffuser + diffusing bend (AR = 1.5) (c) Cropped diffuser + diffusing bend (AR = 2.0)

Figure 11: Investigated diffuser geometries

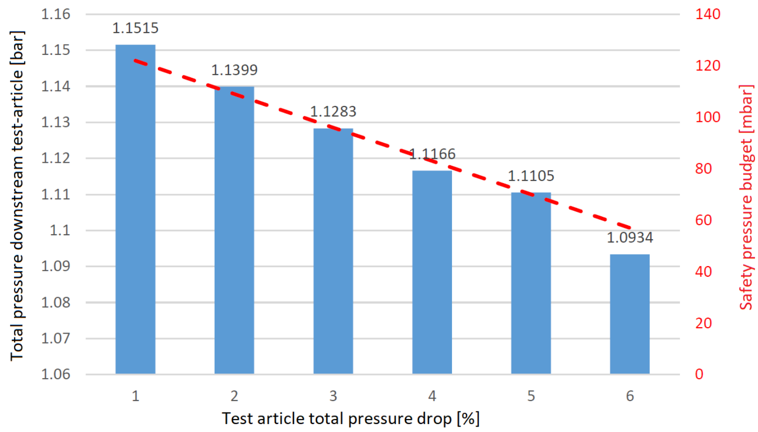


Figure 12: Loss estimation and available pressure budget

Table 2: Compressible total pressure loss comparison between CFD and the loss model

Component	CFD	Model
Collector	0.38	0.38
First bend	0.21	0.20
Convergent	0.01	0.01
Cropped diffuser	0.30	0.14
Diffusing bend	0.27	0.48
Second bend	0.18	0.19

were implemented such that the tool retrieves the loss coefficient of each component, and thus for the entire facility. This calculates the pressure drop across each component and therefore, all flow variables at a given section can be computed, and these values can be treated as input for the successive component of the facility.

The novel test facility is also assessed through CFD simulations maintaining similar conditions that were used in Section 2.1.3. One of the outcomes of this analysis is that the overall losses increased considerably with respect to the three design solutions analyzed in the Section 2.2.2. This is attributed to the presence of a strong flow with swirl coming from the collector which was not present in the previous analysis. Indeed, considering the entire return-duct, the total pressure drop was more than doubled as shown in Table 2.

Additionally, with the intent of assessing the overall pressure budget available when a distortion screen is installed, a loss source was introduced into the lumped parameter model which depicts the loss across the distortion screen. Figure 12 reports the total pressure budget available considering different levels of losses introduced by the distortion screen. Among the other values, the target total pressure loss introduced by the screen (equal to 5%) generates a safety-margin above acceptable limits, meaning that the facility is capable to work properly under the specified distorted conditions. Despite a worst-case scenario of 5% loss introduced by the screen, the facility would be able to operate correctly even with a screen that introduces 6-7% loss. The large losses deriving from the presence of a strong swirl coming from the inlet collector would suggest the installation of a honeycomb which would act positively on the flow homogenization and on the reduction of the total pressure loss in the facility.

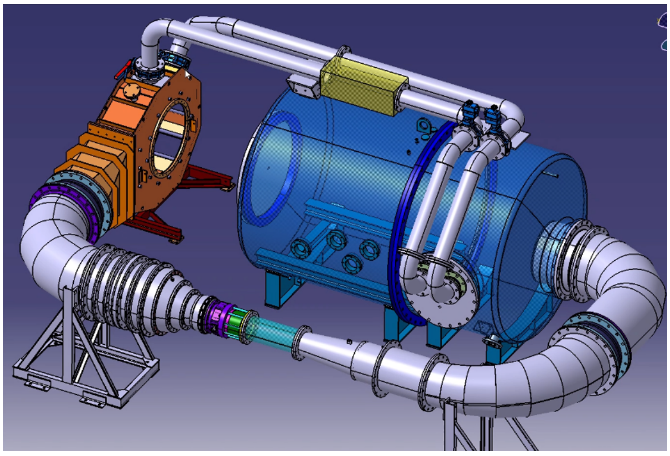


Figure 13: Schematic of VKI R4 novel test facility

2.2.4. R4 novel test facility In summary, the novel facility consists of a bypass duct and an additional duct to reduce the risk during the operation of the novel facility. The flow rate through the bypass duct is measured using a venturi flow meter and can be finely controlled using a flow control valve. The flow from the collector passes through the flow settling unit (honeycombs and screens) before approaching the convergent. The convergent takes the flow to the test-section, and the boundary layer control system additionally helps in maintaining a uniform velocity profile in the test-section. The flow in the test-section is taken back to the original diameter of the return duct through the diffuser and the diffusing bend. The schematic of the R4 novel test-facility is shown in Figure 13. The new facility can thus also act as a wind tunnel that can test distortion generators at engine-realistic conditions.

3. Conclusions

The objective of this work was to develop a test facility that is capable of testing distortion generators at wide ranges of Mach and Reynolds numbers. To realize this, the VKI R4 high-speed compressor test-facility was redesigned to install a novel test-section. To ensure that the desired flow conditions are met at the test-section, the following modifications are carried out on the existing facility:

- The facility is equipped with a bypass duct, the mass flow through which can be individually controlled. This allows an additional control on the mass flow rate passing through the test-section and thus on the Mach and Reynolds numbers.
- Prior to the location of the test-section, a flow settling unit consisting of a series of honeycombs and flow straighteners is proposed that ensures a uniform flow at the exit of this unit.
- Since the test-section has to also operate at different Mach numbers, a convergent section is designed which takes the flow at the exit of the settling unit to the desired Mach number in the test-section. The convergent section delivers a flow that is homogenous at a prescribed location in the test-section.
- To further address the non-uniformity in the boundary layer in the test-section, a boundary layer control system using the idea of tangential air injection is designed.
- Following the test-section, a cropped diffuser along with a diffusing bend is designed which ensures minimum pressure losses and connects the test-section to the remaining portion of the return duct.

Careful consideration is given during the design phase to ensure that the pressure losses across all the components of the novel facility are minimal and falls within the pressure budget available to run the compressor of the closed-loop facility. In summary, this methodology can be used as not limited to designing test-sections for distortion generators, but as a general guideline to design closed-loop wind tunnels.

Nomenclature

m	Mass flow rate at the test-section, kg/s
π_c	Compressor total-to-total pressure ratio
c	Contraction ratio
CDI	Circumferential Distortion Intensity
CFD	Computational fluid dynamics
D	Diameter of test-section, m
L	Length of convergent, m
M	Mach number at test article location
SI	Swirl Intensity

Acknowledgments

This project has received funding from the European Union's Horizon 2020 research and innovation programme under grant agreement No. 864831.

The authors thank the project partners CENAERO, Cadence Design Systems Belgium, and Safran Aircraft Engines for their collaboration and constructive suggestions since the beginning of this project. The authors also thank Giuseppe Anaclerio for supporting the numerical simulations.

References

[1] D. E. Diamantidou, M. L. Hosain, and K. G. Kyprianidis, "Recent advances in boundary layer ingestion technology of evolving powertrain systems," *Sustainability*, vol. 14, no. 3, p. 1731, 2022.
[2] S. S.-. Committee, 2002. Gas Turbine Engine Inlet Flow Distortion Guidelines, ARP 1420 B.
[3] W. T. Eckert, K. W. Mort, and J. Jope, "Aerodynamic design guidelines and computer program for estimation of subsonic wind tunnel performance," tech. rep.
[4] J. B. Barlow, W. H. Rae, and A. Pope, *Low-speed wind tunnel testing*. John wiley & sons, 1999.
[5] G. Chmielewski, "Boundary-layer considerations in the design of aerodynamic contractions," *Journal of Aircraft*, vol. 11, no. 8, pp. 435–438, 1974.
[6] D. S. Miller and D. S. Miller, *Internal flow systems*, vol. 5. BHRA (Information services) Bedford, UK, 1990.