REDESIGN OF A CLOSED-LOOP HIGH-SPEED FACILITY TO TEST DISTORTION GENERATORS

Manas Payyappalli
von Karman Institute for Fluid Dynamics
Rhode-Saint-Genèse
Belgium
e-mail: manas.madasseri.payyappalli@vki.ac.be

Elissavet Boufidi
von Karman Institute for Fluid Dynamics
Rhode-Saint-Genèse
Belgium
e-mail: elissavet.boufidi@vki.ac.be

Riccardo Toracchio
von Karman Institute for Fluid Dynamics
Rhode-Saint-Genèse
Belgium
e-mail: riccardo.toracchio@vki.ac.be

Fabrizio Fontaneto
von Karman Institute for Fluid Dynamics
Rhode-Saint-Genèse
Belgium
e-mail: fabrizio.fontaneto@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. Additionally, the present-day aircraft engines also incur inflow distortions which are undesirable during flight. It is important to test and characterize the effect of such inflow distortions 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 distortion screens under desired flow conditions. To achieve this, 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 screens can be tested and characterized. Thus, a compressor test-bench is now modified to act as a wind tunnel for testing distortion screens at engine-like conditions.

NOMENCLATURE

c – contraction ratio
D – Diameter of test section, m
M_a – Mach number at test article location
L – Length of convergent
$\dot{m}$ – Mass flow rate at the test section, kg/s
π_c – Compressor total-to-total pressure ratio
BL – Boundary layer

INTRODUCTION

Inlet distortion is the phenomena that occurs when the flow entering aircraft engine is non-uniform either due to different flying conditions or due to the typical engine architecture. Engine architectures are in the phase of a change to meet the future market needs. In the immediate future, short nacelle layouts will be adopted for the Ultra High Bypass Ratio (UHBR) fan configuration, and in the long term, Hybrid Wing Body or Boundary Layer Ingesting (BLI) engines remain as a promising architecture [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, it 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. That is why distortion tests are made at different steps of the engine design process.

Research related to inlet distortions has been focused on total pressure for many years. Distortion devices such as assembly of wire screens have been designed to recreate total pressure profiles in a laboratory environment. However, what is missing in literature is a method to develop distortion generators that could reproduce realistic flow conditions that are likely to occur at engine inlets. Though one of the recent works describes an interesting method to develop swirl distortion generators [2].

In this context, the EU funded ASTORIA project has the objective to develop a new methodology to design distortion screens replicating combined swirl and total pressure distortion patterns. With these devices, the effect of realistic inlet distortions on the performance and stability of engines can be investigated. In order to test distortion generators, one of the high-speed closed-loop compressor test-facilities at the von Karman Institute for Fluid Dynamics (VKI) has been redesigned. The present paper describes the steps of the design process and provides details on the methods and tools employed. 3D RANS numerical simulations and a 0D lumped parameter model of the facility have been employed to properly quantify the total pressure losses over the whole return loop as well as to confirm the validity of the undertaken design choices. As a last step, a boundary layer control system has been designed to control the boundary layer thickness in the test section and guarantee a flow pattern as uniform as possible at the location of test article in the test section.

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 number that represent real engine conditions. This paper thus outlines the basic design methodology adopted to develop the novel test-facility.

METHODOLOGY AND DESIGN PHILOSOPHY

To realize the objective of this study, the idea is to redesign the high-speed closed-loop compressor facility, R4 at VKI such that its return duct is capable of testing distortion generators at engine representative conditions. A schematic of the test facility is shown in Figure 1. The design methodology is based on thorough literature survey, reduced order models, and CFD.

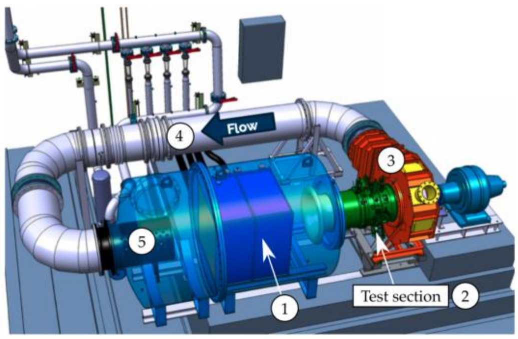


Figure 1. Schematic of the VKI R4 original test facility (1) Plenum (2) Compressor stage (original test section) (3) Collector (4) Return duct (5) Throttle valve

VKI R4 test-facility

VKI R4 original test-facility is a closed-loop compressor test bench (see Figure 1). The compressor stage (in green) is driven by a DC motor that can deliver a maximum power of ~700 kW. One or two gear-boxes are used to raise the motor shaft rotational speed up to the RPM range required by the installed component. The flow from the plenum follows a smooth convergent bell-mouth that gently guides the flow from to the compressor inlet. After the compressor stage, the flow is discharged into a large collector directly connected to the return duct. The flow is then delivered back to the plenum where the rig throttle valve is located. Past 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 barA) or by employing vacuum pumps able to reach pressures as low as 0.3 barA.

Preliminary assessments

The first step in this study was to assess if the desired flow conditions can be achieved using the capabilities of the existing test-facility. This was inferred by estimating the range of target mass flow rates and Mach numbers at the test section.

This analysis concluded that the compressor stage will be able to successfully deliver the required mass flow rates at the test section. However, due to the nature of stall inception in the compressor at the assessed operating conditions and to maintain a sufficiently high stall margin, it was decided to operate the compressor at a larger mass flow rate and bypass the excess mass flow rate that is not required at the test section. To satisfy these conditions, a 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-

Table 1. Parametric analysis for M = 0.60 at test section

π_c	D/D_{duct} [m]	Corrected $\dot{m}$ [kg/s]	Bypass ratio [%]	Stall margin [%]
1.24	0.33	9.29	129.4	14.3
1.24	0.38	9.29	97.8	14.3
1.24	0.41	9.29	82.8	14.3
1.26	0.33	8.81	120.8	6.75
1.26	0.38	8.81	91.4	6.75
1.26	0.41	8.81	77.3	6.75

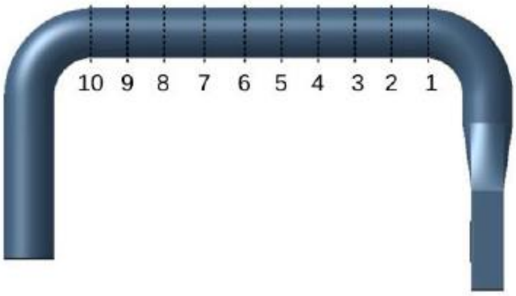
section, while mapping the bypass ratio and the stage stall margin. The output of this design exercise for a Mach of 0.60 at the test section is shown in Table 1.

The conclusions of this assessment are:

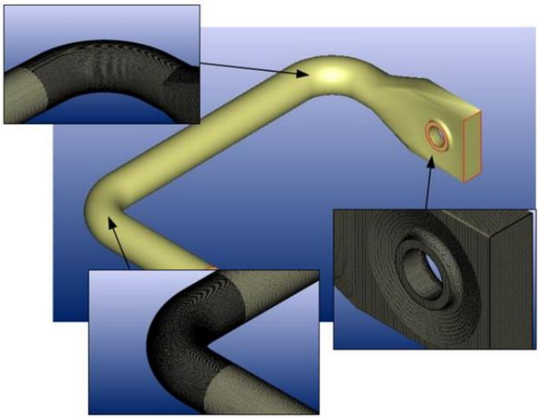
- The DREAM stage operating point should be set in such a way that the compression ratio would be kept in a low range for satisfactory stall margin.
- The critical test section diameter was assessed that would allow a feasible bypass ratio. Under these operating points, the facility can reach the target flow conditions.
- The critical test section diameter also concludes that it is necessary to add a convergent section upstream of the test section and a divergent section downstream of the test section.

Numerical simulations of R4 original test-facility

The next step in the design phase was to carry out a numerical simulation of the original test facility. The numerical domain of the original test facility showing the mesh used for the simulations is shown in Figure 2. The inlet of the numerical domain was specified with total pressure, total temperature, flow angle, and turbulence quantities. 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. Moreover, a turbulence model sensitivity analysis was performed. Despite the different results provided, the major conclusions of this study were not modified employing a different turbulence model. The objective of this exercise was to investigate the total pressure and swirl distortion indices at different streamwise locations in the return duct to identify the most suitable location for the novel test section (see Figure 2 (a)). This exercise also helped to understand how the components upstream of the novel test section will need to be re-designed to ensure uniform flow conditions at the test section. The distortion was characterized by SAE ARP 1420 [3] total pressure and swirl distortion descriptors. These distortion indices were chosen to bring not only information



(a) Numerical domain showing the axial locations investigated



(b) Details of the mesh used for the simulation

Figure 2. Numerical domain and details of mesh of the original return duct of R4

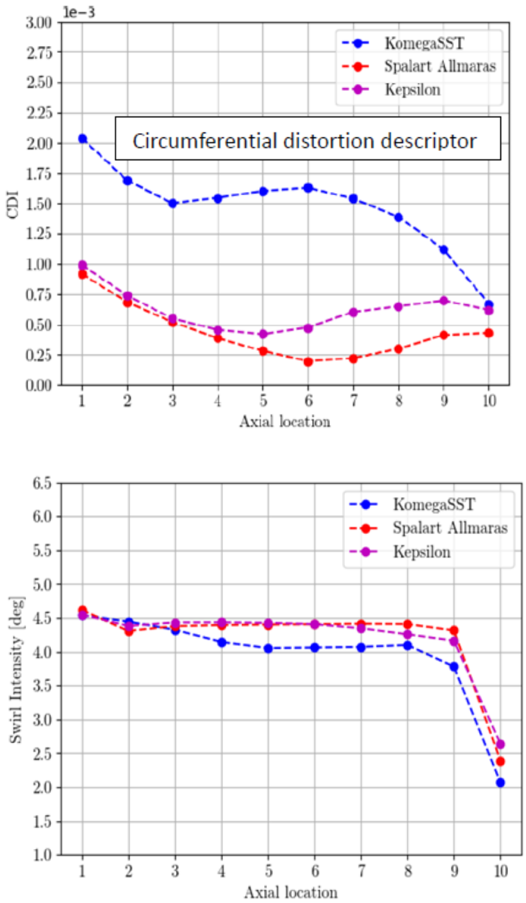


Figure 3. Variation of circumferential distortion descriptor (CDI) and swirl intensity along axial distance

concerning distortion strength and severity, but also to understand pattern and distribution of total pressure and flow angle at the desired location, to support therefore the successive steps of the design process.

The major conclusion of the numerical simulations on the original test facility are:

- As expected, the circumferential distortion indices reduced along the streamwise direction from axial location 1 to axial location 10 due to flow re-organization (Figure 3).
- 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 (Section 10 to avoid being too close to the return-channel bend). The exact location of the test section has been therefore later chosen in view of this study and according to specific design constraints raised later during the design process.

Convergent section

From the critical diameter obtained from the preliminary assessment, it was confirmed that a convergent prior to the test section is necessary in the return duct to accelerate the flow to reach the target Mach numbers in the test section. Since the test section critical diameter was assessed in the preliminary assessment and the pipe diameter is already known, the contraction ratio was calculated. However, the challenge in the design of convergent was to reduce the losses in the component and to decide its length. The main loss source within a convergent coincide with the friction generated on wetted surfaces [4] [5] with possible separations occurring for too aggressive designs, as reported by Chmielewski [6]. Considering the contraction ratio foreseen in the present application and according to the results previously mentioned, it was inferred that the length to diameter ratio should be kept higher than 1.5. With the help of empirical data available as in Figure 4, the length of the convergent section was assessed.

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. Additionally, the semi-aperture angle can

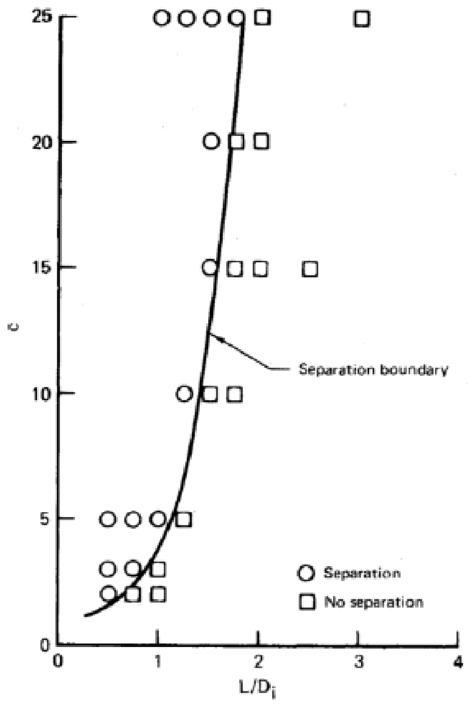


Figure 4. Separation line at varying contraction ratios (c) and length of convergent (L/D$_i$) [6]

also be calculated from the contraction ratio and the length of the convergent.

Boundary layer (BL) control system

The convergent section was able to make the flow at the test article location as uniform as possible. However, the only non-uniformity existed was in the boundary layer regions. This can also be observed from the works of Chmielewski [6] as shown in Figure 5. The boundary layer continues to grow at the exit of the convergent and thus delivers a thick boundary layer at the test article location. Therefore, a boundary layer control system needs to be designed that can further reduce these non-uniformities existing in the boundary layer at the test article location. The basic idea of realizing the BL control system was using the principle of tangential blowing. With this technique, the momentum that is lacking in the boundary layer can be added by using a jet of air that is blown tangentially into the boundary layer. The important steps involved in the design of the BL control system were locating the most suitable position, sizing the control system correctly and finding the angle at which the injection can be realized. The BL control system was simulated 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 BL 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 system gave the best results when it was placed at the exit of the convergent as shown in Figure 6. As shown in Figure 7, the boundary layer displacement thickness at the test article location reduces to approximately 1/3rd as compared to the thickness without injection. The sizing and angle of injection of the BL control system was also simulated parametrically to achieve a suitable case to achieve a realistic design of the system.

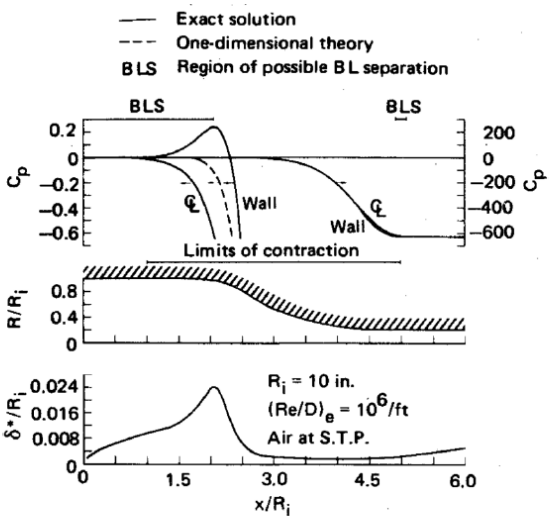


Figure 5: Boundary layer growth in a convergent channel [6]

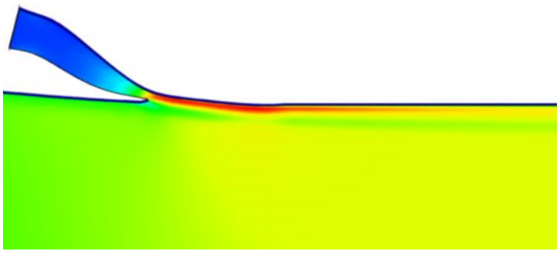


Figure 6. Location of the BL control system showing the velocity contours

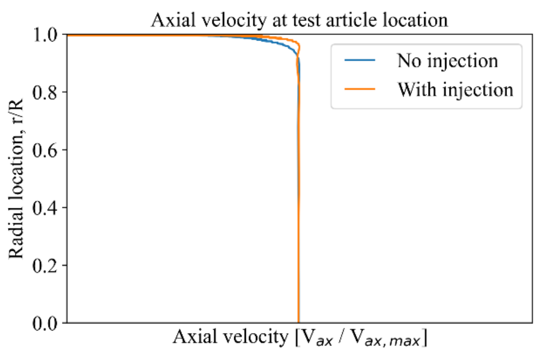


Figure 7. Axial velocity distribution at the test article location

Diffuser and diffusing bend

Diffusers exhibit a high tendency to flow separation because of the adverse pressure gradients generated along its length. The design of an efficient and robust diffuser translates therefore in ensuring the shortest linear displacement (minimal friction) while preserving a high flow separation margin. Three main performance parameters can be found for diffuser in the open literature which are the pressure coefficient, efficiency, and loss coefficient.

In order to employ the performance coefficients within a design approach, a link must be set up between the diffuser geometrical parameters and the expected static pressure rise and/or total pressure drop. Different empirical models are available which can assume a different definition 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. Care was taken in considering only the diffuser geometries for which a loss coefficient correlation was available and in ensuring consistency among the different definitions provided for the various configurations. Loss coefficient definitions have then been calculated using a loss evaluation tool in Matlab, allowing to parametrically compute the efficiency of the diffusers for various geometrical dimensions. A value of the loss coefficient was selected (K = 0.282) and then the most appropriate diffuser with length that will fit in the current facility was chosen. The “cropped diffuser” configuration provides the shortest diffuser length at fixed loss coefficient magnitude, with results in agreement between the two different correlations employed within the study. Given the superior performance when compared to other layouts, the cropped diffuser configuration is therefore selected as the design candidate hereafter.

An interesting solution from the point of view of the facility integration, corresponds to the adoption of a cropped diffuser integrated with a diffusing bend and originally investigated by Miller [7]. Three main configurations were selected (Figure 8) for numerical simulations to finalize the best

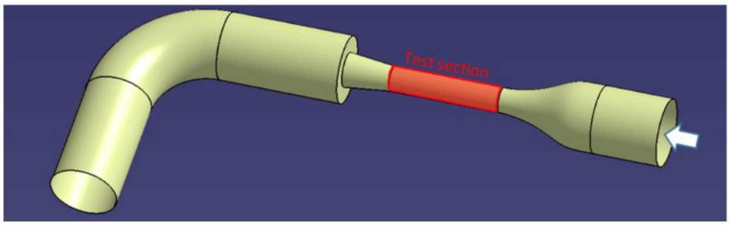
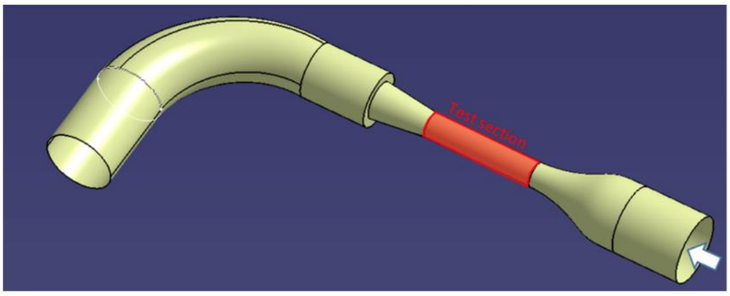
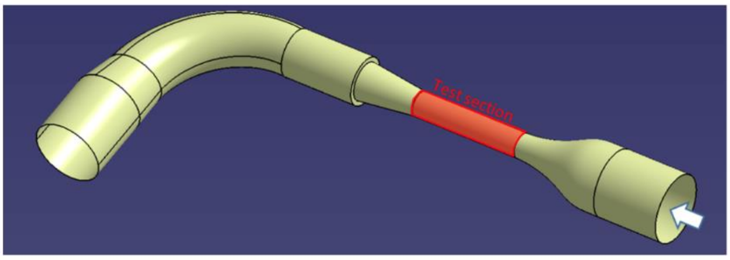
Case	Description
1	*Cropped* diffuser 
2	*Cropped diffuser + diffusing bend (AR = 1.5)* 
3	*Cropped diffuser + diffusing bend (AR = 2)* 

Figure 8. Numerical domain of different diffuser configurations

diffusion 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-channel duct. Among the analyzed solutions, configuration (iii) 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 of the considered configurations and that the main element introducing losses was the cropped section.

Numerical simulations on R4 novel test-facility

As last step of this design procedure, a lumped parameter model was developed in MATLAB, and CFD simulations of the entire facility (see Figure 9 for the numerical domain) were performed to assess the overall loss budget obtained with the presented configuration. This reduced-order model was implemented to be strongly modular to easily modify the architecture of the return channel, and retrieve global parameters as the overall pressure loss, the total pressure reduction introduced by every component, and the Mach number at the inlet of the test-section. For each element, geometrical parameters were explicated following easy syntax rules and pressure loss correlations were implemented to retrieve the loss coefficient by means of specified geometrical parameters. When the loss coefficient is computed, the total pressure drop and therefore the total pressure value at the exit of the component can be estimated. The Mach number value can be obtained by solving the continuity equation iteratively. With this information, all flow variables at a given section can be computed, and these values can be treated as input for the successive component of the return duct.

The main outcome of this analysis is that the overall losses increased considerably with respect to the three design solutions analyzed in the section “Diffuser and diffusing bend”. This is mainly due to the presence of a stronger swirled flow coming from the collector which was not present in the previous analysis. Indeed, considering the entire return-channel, the total pressure drop was more than doubled. These results very well matched the results of the reduced-order model.

With the intent of assessing the overall pressure budged available when a distortion screen is installed, a distortion screen was introduced into the lumped parameter model as a source of loss. Figure 10 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 of 3500Pa, meaning that the facility is fully able 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.

As last point, 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 (which are thought to be way larger than the total pressure loss introduced by the honeycomb screen itself).

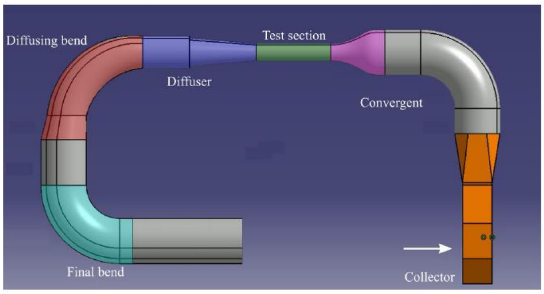


Figure 9. Numerical domain of the R4 novel test-facility

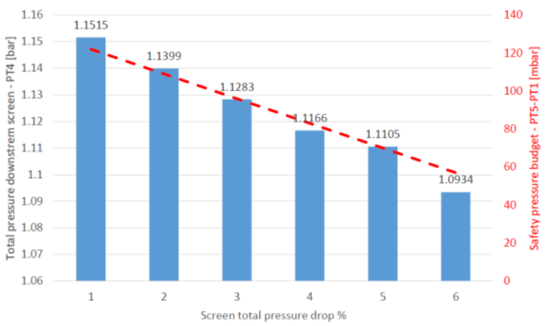


Figure 10: Loss estimation and available pressure budget

Bypass duct

The bypass duct is designed to bleed the mass flow that is not required at the test-section. The bypass is designed such that it is able to fit a flow control valve and a venturi flow meter that is able to measure the mass flow rate. Thus, the sizing of the bypass duct is based on two conditions – first, it is able to bleed-off a range of mass flows that will ensure all ranges of flow conditions are achieved at the test section; second, the pressure drop and the in the venturi flow meter is within specified limits. An anti-surge duct is also designed in parallel to the bypass duct in parallel that can bypass additional mass flow in case of necessity.

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 the desired flow conditions are met at the test section, the following modifications are carried out on the existing facility:

1. 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.
2. 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.
3. 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.
4. 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.
5. The test section is designed as a constant-area cylindrical duct where distortion generators can be placed for testing.
6. Following the test-section, a cropped diffuser along with a diffusing bend is designed which ensures minimum pressure losses and also connects the test section to the rest of the portion of the return duct of the test-facility.

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 testing distortion generators, but as a general guideline to design closed-loop wind tunnels.

ACKNOWLEDGMENTS

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

REFERENCES

[1] Diamantidou, D.E., Hosain, M.L. and Kyprianidis, K.G., 2022. Recent Advances in Boundary Layer Ingestion Technology of Evolving Powertrain Systems. *Sustainability*, *14*(3), p.1731.

[2] Hoopes, K.M. and O'Brien, W.F., 2013. The StreamVane method: a new way to generate swirl distortion for jet engine research. In *49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference* (p. 3665).

[3] SAE S-16 Committee, "Gas Turbine Engine Inlet Flow Distortion Guidelines," ARP 1420, Revision B, Society of Automotive Engineers, February 2002.

[4] Eckert, W.T., Mort, K.W. and Jope, J., 1976. *Aerodynamic design guidelines and computer program for estimation of subsonic wind tunnel performance* (No. A-5944).

[5] Barlow, J.B., Rae, W.H. and Pope, A., 1999. *Low-speed wind tunnel testing*. John wiley & sons.

[6] Chmielewski, G.E., 1974. Boundary-layer considerations in the design of aerodynamic contractions. *Journal of Aircraft*, *11*(8), pp.435-438.

[7] Miller, D.S., 1990. *Internal flow systems* (Vol. 5, pp. 61-67). Bedford, UK: BHRA (Information services).