IDEALVENT Deliverable
ACP2-GA-2012-314066-IDEALVENT
Integrated Design of Optimal Ventilation Systems for Low Cabin and Ramp Noise

Deliverable D4.5: WP4 final report (public)
WP4 final report on the simulation of the flow and acoustic interactions in the combined configuration

Estimated number of person-months: 1.0
Actual number of person-months: 1.0

Contributing partners: VKI, DLR, KTH, KUL, SISW, ECL, LTS, NTS

Author: M. Shur (NTS)
Approved by: C. Schram (VKI)
# Table of Contents

1. Introduction ........................................................................................................ 5
2. Developments and achievements made by VKI ..................................................... 6
3. Developments and achievements made by DLR Braunschweig ............................. 10
   3.1 Introduction ....................................................................................................... 10
   3.2 Task 4.3: CAA simulations for tandem diaphragm cases ............................. 10
      3.2.1 Application of PIANO to the diaphragm cases ........................................ 12
      3.2.2 Application of DISCO++ to the diaphragm cases ...................................... 12
      3.2.3 DISCO++ simulation set-up ...................................................................... 12
      3.2.4 FRPM set-up for the diaphragm cases ...................................................... 13
      3.2.5 Evaluation of simulation data ..................................................................... 14
   3.3 Task 4.1: CAA broadband sound simulations for Butterfly Valve ................ 16
      3.3.1 Application of DISCO++ for the Butterfly Valve ...................................... 16
   3.4 Discussion of Performance and Conclusions ................................................... 18
      3.4.1 FRPM performance ................................................................................... 18
      3.4.2 CAA performance .................................................................................... 19
      3.4.3 Final Conclusions ..................................................................................... 19
4. Developments and achievements made by DLR Berlin ........................................ 20
   4.1 Introduction ....................................................................................................... 20
   4.2 Configuration with inflow devices .................................................................. 20
   4.3 Flow simulation ............................................................................................... 20
   4.4 Tonal noise simulations .................................................................................. 21
   4.5 Broadband simulation .................................................................................... 23
      4.5.1 Flow extraction and transformation in 2D ................................................ 23
      4.5.2 Turbulence ............................................................................................... 23
      4.5.3 Setup ....................................................................................................... 24
      4.5.4 Results .................................................................................................... 25
   4.6 Conclusion and outlook ................................................................................... 27
   4.7 Bibliography .................................................................................................... 27
5. Developments and achievements made by KTH .................................................. 29
   5.1 Introduction ....................................................................................................... 29
   5.2 Flow Computations ........................................................................................ 30
   5.3 Acoustic Computations ................................................................................... 30
   5.4 Combining Multi-Ports .................................................................................... 31

This document is property of the IDEALVENT Consortium and has been approved by all partners to be disseminated
9.2.3 Comparison with experiments and simulations made by KUL and EXA

9.2.4 Comparisons with LTS valve noise prediction

9.2.5 Conclusion

9.3 Diaphragm prediction tool

9.4 Fan prediction tool

9.5 Integration of LTS acoustic tools in DYMOLA (global system model)

10 Developments and achievements made by NTS

10.1 General overview of developed computational procedure and considered flows

10.2 Effect of inlet distortions of ducted fan noise

10.2.1 Considered cases

10.2.2 Sample flow visualizations, time-averaged fields, and turbulence statistics

10.2.3 Comparison with experiment on “raw” wall pressure spectra

10.2.4 Extracted acoustic modes

10.3 Computations of flow and noise produced by single and tandem diaphragms

10.3.1 Considered cases

10.3.2 Sample results and comparison with available experimental data

10.4 Conclusions

10.5 Bibliography
1 Introduction

This final report presents major results obtained by IDEALVENT Partners in the course of their work on the Work Package 4 “Simulation of the interactions”. This includes application of the new simulation approaches developed and tested within Work Package 3 of IDEALVENT to the following problems:

- simulation of industrial valve with upstream bend (Task 4.1);
- simulation of aeroacoustic installation effects of a ducted fan with inlet flow distortions (Task 4.2);
- simulation of aeroacoustic installation effects for ducted orifices (Task 4.3).

All of the IDEALVENT Partners participated in at least one of the tasks listed above, and the results obtained are presented in this report in a “partner wise” manner. For a reader convenience, below a “task wise” distribution of the report material is listed as well:

- Task 4.1 results are presented in the contributions of VKI, DLR (Braunschweig), KUL, SISW, LTS;
- Task 4.2 results are presented in the contributions of DLR (Berlin), KTH, SISW, ECL, LTS, NTS;
- Task 4.3 results are presented in the contributions of DLR (Braunschweig), KTH, SISW, LTS, NTS.
2 Developments and achievements made by VKI

The contribution of VKI to WP 4.1 is to perform CFD computations to complement the work made by other partners, mainly focusing on stochastic approaches. It includes RANS computations on the valve case in order to obtain the mean flow properties required as background flow for sound propagation/prediction methodologies. Unsteady computations are performed in this WP to validate these methodologies but also to extract unsteady wall-pressure fluctuations and volumetric sources for direct sound prediction based on analogies by SISW. The IDEALVENT DoW stated that VKI will perform incompressible computations on the valve case. As the experiments performed by KUL, and reported in WP2, showed that the Mach number is nearly 0.5 at the chosen operating conditions, compressible computations are mandatory. In the project timeframe and given the available resources, only a preliminary unsteady compressible computation has been performed.

The CAD of the valve is made available by LTS. A view of the initial CAD is shown in Figure 2-1, including all the geometrical details as in the experiments. The valve opening is fixed to 30 degrees. In order to be able to mesh the geometry, all the unnecessary geometrical details are removed, as they will have only a little influence the computational results. The main surfaces of the valve are kept as well as the multi-step shape of the valve side. The valve side details are important as they may lead to vortex shedding occurring at a different position if the corners are not properly reproduced in the CFD mesh.

The mesh is created using Numeca Hexpress Hybrid 4.2 that generates unstructured hex-dominant conformal meshes for meshing large and complex geometries in a flexible automatic manner, based on an octree meshing algorithm.

Domains having different geometrical sizes are considered for the 3 computations being:

1. Small domain: 5D-5D, D being the duct diameter, first and second numbers being respectively on the upstream and downstream side of the valve. This computational domain is used to determine the mass flow through the valve to obtain a 500 mbar pressure drop, as in the experiments. This is performed using a RANS computation.
2. Large domain: 19D-19D. This extended domain corresponds to the experimental case including microphones for multi-port matrix determination. These microphone locations should be included in the computational domain to have the corresponding mean flow properties for the sound predictions performed by other partners (KUL, DLR).
3. Unsteady domain: 5D-7D. Compared to the small domain, only the downstream part is extended to give larger distance for the flow disturbances to damp before reaching the outlet domain boundary and avoid possible reflections.
All the RANS computations have been performed using rhoSimpleFoam, the compressible steady solver of OpenFoam 2.4.0. The k-ω SST turbulence model is used with second order accurate schemes for all variables. Air is used as fluid with perfect gas properties and the Sutherland model is used for the variation of viscosity with compressible effects. Other properties are kept constant. Mass flow rate is used as inlet condition. As the corresponding value is not directly available from the experiments performed by KUL, but only the pressure drop and the corresponding volume flow rate, computations are performed in a range of inlet mass flow rate. The operating conditions corresponding to this specific pressure drop are then determined through the operating curve results.

For the unsteady computations, the solver Fluent 16.2 is used. The SAS turbulence model is used, that is an improve URANS formulation, which allows the resolution of the turbulence spectrum in unstable regions, resulting in a LES-like behavior in unsteady regions of the flow field. At the same time, it provides standard RANS capabilities in stable flow regions. As for the RANS computations, second order schemes are used for all variables and central bounded second order schemes are used in time. Same fluid properties are used as in the RANS computations. The inlet condition is defined as mass flow rate of 0.205 kg/s, defined from RANS results. Furthermore, general Non-Reflecting Boundary Conditions are used to limit possible reflections at the inlet boundary conditions. At the outlet, a pressure far-field condition is applied that is a non-reflecting boundary condition based on the introduction of Riemann invariants for a one-dimensional flow normal to the boundary. Wall-pressure fluctuations (dipoles) and volumetric velocity data (quadrupoles) are extracted in the complete CFD domain, to be used as input to acoustic computations performed by SISW.
In order to obtain the mass flow rate passing through the valve and corresponding to 500 mbar of pressure drop (as defined as relevant test case in D2.2), several computations have been run, ranging from 0.04 to 0.12 $m^3/s$. This results in an operating curve, that is compared to experiments of KUL in Figure. The pressure drop is evaluated at the same locations than in the experiments, at 1.25D and 2D, upstream and downstream of the valve, respectively. An excellent agreement in found between the RANS computations and the experiments of KUL reported in D2.2, for all the investigated mass flow rates. The resulting volume flow rate of 0.1145 $m^3/s$ obtained for the pressure drop of 500 mbar is then used for the unsteady computation. The corresponding pressure drop of the unsteady computation is also reported in Figure2-2, showing that the operating condition still match the experimental value.
Figure 2-3: Flow topology described by the Q factor and colored by velocity magnitude.

The flow topology of the unsteady SAS computation is represented with the Q criterion, and shown in Figure . As no turbulence is injected at the inlet boundary condition (uniform mass flow rate constant in time) no turbulence structures are observed upstream of the valve. On the upstream beam of the valve, a detachment of flow structures is observed, which almost immediately disappears due to the strong acceleration observed in the valve opening. Due to the blockage created by the small opening between the duct and the valve near the beam location a horseshoe vortex is observed on both sides of the valve. The main turbulent activity is created right after the flow passes the valve opening. On the top part, relatively fine structures are created, due to the high velocity and shear, resulting in a direct transition to turbulence. On the bottom part, larger structures are created that are convected farther away from the valve, before being dissipated in smaller structures as the flow re-homogenized.
3  Developments and achievements made by DLR Braunschweig

3.1 Introduction
Simulations by DLR Braunschweig have been conducted for the tandem diaphragm cases in Task 4.3 and the butterfly valve used in Task 4.1. Due to its high geometrical complexity, Task 4.1 had to be simulated applying the unstructured CAA solver DISCO++ of DLR. The simulations conducted in Task 4.3 have been simulated using both, a structured and an unstructured CAA solver. The simulations in Task 4.1 and 4.3 have been conducted combining the CAA propagation solver with stochastic sound source modeling based on precursor RANS simulation to predict the broadband sound generation process at the different components. The CAA simulations with stochastic sound sources complete the numerical analysis of the butterfly valve in terms of a RANS based broadband noise characterization of this component. Since in terms of geometrical complexity the butterfly valve case of Task 4.1 is much more challenging than the diaphragm cases of Task 4.3, Task 4.1 will be discussed subsequently after Task 4.3. Finally, the 3-D FRPM and CAA performance will be discussed and conclusions will be drawn.

3.2 Task 4.3: CAA simulations for tandem diaphragm cases
The contribution of DLR to task 4.3 comprises aeroacoustic simulations of the tandem diaphragm cases to enable their active characterization based on stochastic sound sources. Two diaphragm cases are considered, i.e. diaphragm separations based on 2 tube diameters distance (2D) and 4 tube diameters distance (4D). The simulation procedure performed by DLR is shown in Figure 3-1. A hybrid simulation procedure is conducted using in the first step a time-averaged CFD solution of the flow field from Reynolds-Averaged Navier Stokes (RANS) simulation, to simulate in a second step unsteady fluctuations over the mean-flow with CAA methods. The DLR TAU code with Reynolds stress turbulence model (RSM-ω) was applied in previous work to predict the mean-flow in agreement with measured flow conditions.

To get for the CAA calculations acoustic sources based on the RANS turbulence statistics (turbulence kinetic energy ‘\(k_t\)’, dissipation rate ‘\(\omega\)’) the approach utilizes a stochastic method (Fast Random Particle-Mesh Method, FRPM).

Two different in-house CAA solves are used in the framework of Task 4.3, viz. the structured multi-block (SMB) CAA code PIANO and the Discontinuous Galerkin solver DISCO++ for unstructured tetrahedral meshes.
In both solvers the same procedure according to Figure 3-1 is applied with the same stochastic source module. In particular, Acoustic Perturbation Equations (APE) solved on the CAA grid for sound propagation over the RANS background flow.

Table 3-1 provides an overview on the simulated cases tackled with both codes. The tandem-diaphragm cases with two different diaphragm separations are simulated, i.e. 2D relative distance and 4D relative distances, where D indicates the tube diameter. Furthermore, variations in the bulk velocity are considered.

As indicated in column ‘FRPM’, there are two source models tested: ‘A’ denotes the simulation of fluctuating velocity with FRPM. In this approach, the gross fluctuations generated by FRPM are treated as a fluctuating vector stream function, the curl of which provides solenoidal (divergence-free) velocity fluctuations. Hence, vorticity fluctuations are obtained from the curl operator applied twice to the initially generated vector stream function.

‘ASV’ (Anisotropic Solenoidal Vorticity) indicates the simulation of anisotropic solenoidal vorticity with FRPM. Here, the output of FRPM is directly treated as a fluctuating velocity vector, the curl of which provides corresponding solenoidal vorticity fluctuations. Essential, the difference of the two approaches is the number of differentiation operations applied to the initial fluctuating field, i.e. two curl operations for case ‘A’ versus one operation in case ‘ASV’. Furthermore, the components of the vector-fluctuations are scaled to reassemble the anisotropic Reynolds stress tensor provided by RANS RSM-modelling.

Fluctuations are generated by filtering a numerical generated white-noise-field with spatial Gaussian filters. The different filter procedures tested are also indicated in column ‘FRPM’. For the filter-step the in-house best practice filter is a recursive Purser filter (‘Purser’), i.e., a fast filter based on a Gaussian filter kernel allowing a spatially variable filter-width (length-scale).

As a studied alternative, ‘diffGauss’ indicates a filter-kernel, which is based on the spatial derivative of a simple Gaussian filter. The performance of a simple Gaussian filter is in general poorer compared to recursive filtering. However, the necessary differentiation step is already included into the filter kernel, which helps to prevent artefacts in the stochastic reconstruction of turbulence due to the numerical differentiation in the FRPM domain with immersed solids present, refer to the discussion in D4.3.

For source option ‘ASV’ (modeling of fluctuating resolved vorticity) a filter-kernel with fixed length scale corresponding to the FRPM mesh resolution is sufficient. Therefore, the recursive constant length-scale filter as proposed by Young & van Vliet ‘van Vliet’ filter is applied here.

The last important option mentioned in the table above is ‘GD’ and its counterpart ‘LD’, standing for a global particle density, respectively local particle density and describing the handling of the particle
density in FRPM module. For global density, the particle density is derived from the global domain extension and number of particles involved. For ‘LD’ the particle density is computed locally in a FRPM cell for each time-step.

3.2.1 Application of PIANO to the diaphragm cases
Figure 3-2 depicts cross-sections of the structured PIANO grid. The mesh is refined in the area where the stochastic source term is imposed and the maximum cell size is defined by a grid spacing resolving the smallest acoustic wave-length with seven points per wavelength. In total 18 tube diameter are resolved in axial direction. Since hanging nodes are currently not available in PIANO and only a simple mesh topology was chosen, overall 11.1 million CAA grid points are finally obtained for the CAA mesh. Note, more advanced mesh topologies involving mesh clamping and/or hanging nodes could lead to significantly reduced numbers of points. The resolution was chosen to resolve up to 5 kHz.

![Figure 3-2: PIANO CAA grid; left, cut at constant axial x-position away from diaphragm; right, part of the refined section at the second diaphragm.]

3.2.2 Application of DISCO++ to the diaphragm cases
In addition to the PIANO computations a further set of simulations addressing the same problem is conducted employing the DLR CAA code DISCO++. This particular solver does not require a structured grid but operates on unstructured tetrahedral meshes, a feature very well suited for applications where a complex geometry is involved and thus local refinement required. DISCO++ uses the same source reconstruction as PIANO via the FRPM module. DISCO++ solves Acoustic Perturbation Equations (APE), thus flow induced effects, i.e. refraction and diffraction, are resolved.

The equations are spatially discretized by means of the Discontinuous Galerkin Method (DGM) - a procedure combining techniques from the finite element (FEM) as well as the finite volume methods (FVM). Values of variables are defined through polynomials at a number of discrete points inside a grid element. In this instance a polynomial order of $p = 3$ is used. Temporal discretization is realized by an explicit forth order accurate Runge-Kutta (RK4) scheme.

3.2.3 DISCO++ simulation set-up
DISCO++ requires a computational mesh consisting solely of tetrahedral elements. Currently two mesh formats are supported, i.e. MSH and NC. The former is the native format of the grid generator
GMSH distributed under the terms of the GNU General Public License (GPL), the latter is a commonly used data format from within the free NETCDF package. Usually the grid generation process is done by utilization of either the GMSH grid generator or the CENTAUR mesh generator by CentaurSoft.

In the present case CENTAUR is used to generate the grid using a CAD model as input. The exact same geometry was applied as for the PIANO setup as well. The maximum resolved frequency for the DISCO++ version of the diaphragm case is chosen to be \( f_{\text{max}} = 10 \text{ kHz} \) thus defining the minimum grid resolution required. Similar to PIANO, a refinement has to be applied in the source reconstruction area, i.e. around the second diaphragm. Figure 3-3 depicts details of the grid.

![Figure 3-3: Section of the CENTAUR grid; (left), local refinement in the source region around the diaphragm and transition to pure acoustic resolution; (right), cut through the grid at second diaphragm position, local refinement in this area as demanded by source resolution requirement.](image)

The final grid for the present case consists of 960k tetrahedral elements, most of them located in the source region. This number is not particular high for the DISCO++ solver as it is highly parallelized and relies on multithreading. Furthermore, test computations revealed that the computational grid used resolves frequencies higher than the estimated limit, i.e. \( f_{\text{max}} = 11 \text{ kHz} \).

**3.2.4 FRPM set-up for the diaphragm cases**

To both CAA codes the stochastic sound source module FRPM is coupled. The simulation settings used for combined PIANO/FRPM and DISCO++/FRPM are the same. Major settings studied are identified in Table 3-1. A more detailed discussion of the statistical evaluation of the stochastically generated fluctuations and a comparison with the RANS statistics aimed to be realized as key target values is given in Deliverable D4.3. Typical use cases at DLR are related to external aircraft flows and it had to verify whether the same standard procedure applied for those cases is properly working for internal flow problems as well. The general set-up of the FRPM source region (source patch) is shown in Figure 3-4. The source patch is mainly located at the position of the second diaphragm where the major noise sources are expected. The stochastic source reconstruction uses an auxiliary Cartesian mesh. Its extension is indicated in Figure 3-4. The source patch has an extension beyond the circular channel walls and realizes internal surfaces (i.e. the second diaphragm) via immersed boundary conditions of the specific stochastic method. This simplifies the generation of the FRPM source patches.

In the current set-up a FRPM patch with \( \Delta x = \Delta y = \Delta z \approx 1\text{mm} \) cell size is used to resolve a box of size 0.24 m x 0.15 m x 0.15 m with about 3.6 Mio. cells. The number of actually used FRPM grid nodes is 2.5 Mio. (the others are located in the ‘exterior’, i.e. in solid regions or outside the channel). On the FRPM patch three million particles are used and the synthetic turbulence resolves length-scales above
2mm size with the maximum size limited to 10mm. Those values are chosen based on the resolution of the source-patch grid and as the most reasonable highest size probably existent within the flow-field behind the diaphragm.

![Image](image.png)

**Figure 3-4: Location of FRPM patch; the flow direction is from left to right.**

![Image](image.png)

**Figure 3-5: Turbulent sound sources from FRPM in DISCO++ simulation.**

### 3.2.5 Evaluation of simulation data

For further post-processing of the simulated data via multi-modal analysis by KTH a large set of virtual microphones (sampling positions) is defined inside the tube. The same set-up of the sampling positions is used for both, PIANO and DISCO++ simulations. Figure 3-6 provides a slice through the simulated tube section containing both diaphragms (showing a snapshot of the pressure field from PIANO). Upstream and downstream from the second diaphragm the areas where sampling probes are located are indicated. Each sampling area consists of 10 \( y-z \) planes. In turn, each of these slices contains five microphone circles.
Figure 3-6: Positions of the regions containing evaluation planes (perpendicular to flow) upstream and downstream of the second diaphragm; exact offsets to the first slices provided for both directions.

The distance from the center point is expressed in terms of the tube diameter $D = 0.15$ m. The evaluation slices are not distributed equidistantly but with a variable offset as indicated in Figure 3-6. In the sketch, the bulk velocity in the tube is denoted by $U_0$.

Figure 3-7: Microphone positions within a slice. Center circle contains only one microphone. Three circles inside the flow with 8 points each. Outermost circle contains 72 wall microphones located at 99% of $r_{\text{max}} = 0.085$ m.

The specific arrangement of the microphone circles is further clarified in Figure 3-7. Starting with one center position at radius $r_1 = 0$ m, three additional circles of increasing radii ($r_2 = 0.01875$ m, $r_3 = 0.0375$ m and $r_4 = 0.05625$ m) are defined, each consisting of eight points. The outermost 4th circle contains 72 near-wall microphones at radius 99% of $r_{\text{max}} = 0.085$ m. Due to the polygon shape used for the grid a closer placement to the wall would lead to interpolation errors.

The virtual microphones record the pressure signal at their position $N$ time steps each. The interval $N$ is chosen such as to satisfy the Nyquist sampling criterion for a maximum frequency of $f_{\text{Nyquist}} = 20$ kHz which is well above the maximum resolvable frequency of the CAA grid.

Inflow spectra measured at the walls inside the tube have been evaluated from PIANO and DISCO++ simulations and compared with experiment, refer to Figure 3-8. The simulations capture the cut-on of
different channel modes at the expected frequencies. However, also visible in the simulations are large fluctuations as a result of the relatively short sampling time. Their amplitudes could be reduced further by increasing the simulation time and thus increase the number of frequency spectra averaged. The PIANO simulations provide correct spectral shape up to frequencies around 3kHz. An over prediction in the level simulated can be observed that might stem from artefacts observed in the specific source model applied, D4.3. For DISCO++ a better resolution of higher frequencies is visible while some deviations occur for lower frequencies.

3.3 Task 4.1: CAA broadband sound simulations for Butterfly Valve

Simulations of the broadband sound generation at the butterfly valve within Task 4.1 have been conducted by DLR Braunschweig using the unstructured CAA code DISCO++ together with the Fast Random Particle-Mesh (FRPM) module to reconstruct turbulent sound sources from an inexpensive precursor CFD simulation with Reynolds Averaged Navier-Stokes (RANS) equations. Please refer to the discussion in the previous section for further details on DISCO++ and FRPM.

An overview on the simulations performed and related within Task 4.1 is given by Table 3-2. In the following details of the simulation set-up will be described.

<table>
<thead>
<tr>
<th>Simulation</th>
<th>Case</th>
<th>Code</th>
<th>FRPM</th>
<th>CAA Mesh</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Butterfly Valve</td>
<td>DISCO++ (0.1.56)</td>
<td>A; posture; LD; Lev</td>
<td>344,000 Elements; 10 KHz; simplified mesh 1</td>
<td>No mean-flow in CAA simulation</td>
</tr>
<tr>
<td>2</td>
<td>Butterfly Valve</td>
<td>DISCO++ (0.1.51)</td>
<td>A; diffGauss; LD; Ugr</td>
<td>325,000 Elements; 10 KHz; simplified mesh 2</td>
<td>No mean-flow in CAA simulation</td>
</tr>
</tbody>
</table>

Table 3-2: Overview on simulations performed with DISCO++ for the Butterfly Valve; simulations vary in settings of source generation module FRPM as well as CAA mesh; both cases based on same RANS solution provided by VKI.

3.3.1 Application of DISCO++ for the Butterfly Valve

The choice to use the still-in-development code DISCO++ over the well-established structured DLR solver PIANO to tackle the Butterfly Valve problem is driven by the extreme complexity of the geometry, which makes a structured multi-block meshing of the geometry almost impossible. Local refinement, as it is needed for geometric details, can easily be incorporated while computational effort can be saved in areas where a coarser resolution is sufficient. Nevertheless, nearly always an additional reduction of the geometrical complexity by suitable simplifications of the CAD model is necessary. Thus, geometrical details that are considered to have minor influence on the acoustic field are altered to reduce the computational effort, refer to Figures 3-9 and 3-10.
To perform the grid generation using CENTAUR, first, the geometry has to be defined. Figure 3-9 depicts the unchanged CAD model as provided by project partner VKI. The model features geometrical details obstructive for the acoustic analysis for mainly two reasons:

- Only internal flow/acoustic is studied – features on the outside are irrelevant
- Features of relatively small size do not affect acoustics (in a relevant manner)

To resolve the problem, the geometry is stripped of all features mentioned above using a CAD software. During this process, geometrical details such as the hexagonal female screw in the middle of the throttle are reduced to simpler shapes. Thus the dimensions of the features remain unaltered, while the effort to discretize those is reduced.

The outside wall of the tube containing the valve is completely removed. Only those parts of the geometry are of relevance, that are “in contact” with the fluid. Solid walls do not need to have any thickness since defined via a solid wall boundary condition right at the surface.

Two grids have been built as highlighted in Table 3-2. For the first, the amount of tetrahedral elements is around 344,000. Due to very small cell sizes caused by the geometry, only a small time step, i.e. 1.6e-05 is possible (dimensionless time units).
A second improved grid is used for simulation Nr. 2 (Table 3-2). The number of tetrahedral elements is reduced, i.e. 329,000, and the time step required is reduced by a factor of 10 by increasing the smallest cell size. Despite the optimization, there are still geometrical features demanding a more than 10 times finer discretization as required to resolve the maximum frequency. These features cannot be easily altered without more significant changes to the geometry. A possible solution would be the application of a local time stepping scheme. Such a feature allows for different time step levels within the CAA domain thus making bigger elements independent of the smallest elements time step.

Despite these areas of improvement that could be identified, it was possible to run successful CAA computations for the butterfly valve with stochastic sound sources based on a RANS simulation provided by VKI, refer to Figure 3-11 for snapshots showing the induced stochastic sound sources in the vicinity of the valve. Based on the actual butterfly valve tube diameter D, the same relative position of sampling microphones as for the tandem diaphragm case has been used to evaluate data for subsequent multi-port analysis.

### 3.4 Discussion of Performance and Conclusions

The general applicability of stochastic sound sources to realistic 3-D system components as studied in the frameworks of WP4 could be demonstrated by successfully running hybrid RANS/CAA simulations for these cases. However, if applied to 3-D problems stochastic methods lose part of the efficiency they have in 2-D¹ (of the order of 3 orders of magnitude compared to scale resolved simulation) and their performance is less obvious. Different areas of improvement could be identified as part of WP4 that have to be addressed in future work. The major findings are summarized with respect to the stochastic method and the various CAA codes applied.

#### 3.4.1 FRPM performance

The FRPM performance that could be achieved for source options ‘A’ and ‘ASV’ (Table 3-1) yields 7.7 μs per point and time-step (131 CPUh for 3.6 Mio mesh points and 0.1 sec real time resolution with 17,000 time steps). Here, the time step is limited by a CFL constraint defined by the flow velocity and the resolved cell size (not the speed of sound). This has been taken into account in the present simulations by updating the FRPM sources only every 10th explicit CAA time step.

¹ Stochastic methods have in particular a high efficiency if applied to reduced 2-D problems e.g. for aeroacoustic design purposes. For example, in the past the hybrid RANS/CAA method has been applied for trend predictions at aircraft high-lift systems (EU TIMPAN), airfoils (EU VALIANT), and quite recently to optimized wind turbine airfoils. Similar to CFD simulations, 2-D simulations (with aeroacoustic 2-D to 3-D correction) can provide predictions of the sound generated by the turbulent flow around long-span wings. For jet noise, axisymmetric versions of the hybrid RANS/CAA method were capable to successfully predict the effect of 3-D nozzle modifications on sound.
In total, the time savings compared to scale resolving simulation (such as the Lattice Boltzmann Method) can be estimated to yield about two orders of magnitude for the stochastic sound source generation alone, which would allow to run the method for a larger number of configurations (aeroacoustic optimization) or on smaller computer platforms.

3.4.2 CAA performance
For the studied cases the computational bottle-neck was clearly given by the computational time needed for the CAA simulations. For the tandem diaphragm cases PIANO simulations on the block structured mesh ran 5000 CPUh for 0.1 sec resolved physical time. The structured mesh had to comprise about 11Mio mesh points. The high number of mesh points was caused by the resolution requirements in the source region, which, as a consequence of the structured mesh topology, yielded unnecessary many points in the remaining parts of the resolved tube. This problem might be removed with hanging nodes allowed for the CAA mesh.

The unstructured mesh had clearly fewer computational nodes. However, especially for the butterfly valve the simulation time was also considerable too high because of very small element sizes caused by the very complex butterfly valve geometry. To circumvent this problem, a local time stepping procedure (asynchronous time stepping) would be needed for DISCO++.

3.4.3 Final Conclusions
A hybrid RANS/CAA approach has the potential to also gain an increased numerical efficiency compared to scale resolving simulation. However, to accomplish full numerical efficiency of the stochastic method, the CAA propagation methods have to be adapted accordingly. For small mesh sizes defined by geometrical constrained, local time stepping methods seem to be necessary. If the small mesh resolution is required by source resolution constrains, an option would be the derivation of equivalent acoustic surface sources from the stochastic method. Since the acoustic resolution requirements are less severe for pure acoustic propagation, this could help to reduce the CAA mesh resolution requirements and CAA computational time. An interesting surface source approach was proposed by SISW during the project.
4 Developments and achievements made by DLR Berlin

4.1 Introduction

DLR’s work performed at the Institute of Propulsion technology, Department Engine Acoustics in Berlin, is concerned with the simulation of fan noise using a hybrid strategy. In the focus of the investigation lie the prediction of the rotor-stator-interaction noise and the impact of inflow devices on that noise source.

The hybrid approach involves three numerical tools: (1) the URANS solver TRACE, (2) the CAA solver PIANO and (3) the turbulence synthesizer fRPM.

At first, a URANS or Harmonic Balance [Frey 2014] (HB) computation of the fan-stage is performed with the in-house CFD solver TRACE. The fluctuation fields corresponding to the periodic part of the flow (incl. the statistics of the turbulence) are saved in the frequency domain for the lowest relevant harmonics. The pressure fluctuations are used to calculate the tonal component of noise.

Second, a CAA computation is carried out with the CAA code PIANO with the objective to predict broadband noise. The mean-flow properties are used as background flow solution for the perturbed field. The turbulence-induced noise sources are at the same time generated in a third step by a turbulence generator - the so-called fast Random Particle Mesh (fRPM) method - in the time domain, which uses the turbulence statistics from the URANS calculation. This gives the broadband fan noise component. As currently no experience is present in performing such large computations in 3D, a simplification is made by using quasi 3D stream-surfaces of the CFD mesh calculated in a 2D CAA domain.

The method is described in Deliverables D3.3, D3.6 / D3.7. In the following the application of the method to the IDEALVENT fan is demonstrated; for further details please refer to D4.2.

4.2 Configuration with inflow devices

The CAD geometry of the fan has been provided by Liebherr and has been cleaned to generate suitable input for the Simulations. The influence of the simplifications has been investigated in D4.2.

The operating condition used for the measurements has been communicated in Deliverable D2.8 as

<table>
<thead>
<tr>
<th>RPM</th>
<th>Volume flow rate</th>
</tr>
</thead>
<tbody>
<tr>
<td>11200</td>
<td>0.53 m³/s</td>
</tr>
</tbody>
</table>

The inflow conditions are taken from the hot wire measurements at Station 4 as defined in the WP2 final report and described in D4.2. The three inflow devices are:

(a) Bell mouth inflow
(b) T-junction inflow
(c) Rectangular to circular transition.

4.3 Flow simulation

The RANS is performed at the aforementioned conditions. The impact on performance is plotted in Figure4-1. The results emphasize that the CAD cleaning is not significantly changing the fan performance and therefore the results of the simulations can be compared to the measurements.
4.4 Tonal noise simulations
The unsteady RANS simulations are performed using the Harmonic Balance method as described in D3.6 & 7. For the bell-mouth configuration all rotor-stator interaction modes are taken into account up to the 4th harmonic in the absolute frame of reference and up to the 6th harmonic in the rotating frame of reference. For the other two configurations we added the interaction modes of the inflow distortions with the rotor up to an azimuthal mode order of m = 20. This includes 8 (T-junction) and 4 (Rectangular) additional harmonics in the rotor domain. Only the harmonic null or mean flow is calculated in the stator domain; as a result we cannot predict the mode field downstream of the stator.

The tonal noise is extracted from the URANS simulation by using the in-house code Connect3D, which performs an extended Triple-Plane-Pressure (XTPP) mode matching to extract the amplitude and phase of the acoustic propagating modes [Wohlbrandt2016, Weckmüller2009]. Figure 4-2 visualizes the positions and the reconstructed instantaneous pressure field for the clean inflow configuration of the cut-on modes up- and downstream of the fan. The blade passing frequency (BPF) is cut-off in this configuration; only the higher harmonics of the BPF produce tones.

![Figure 4-1: Fan performance of the undertaken simulations compared to the measurements.](image)

The sound power levels of the upstream propagating modes of the first radial mode order are shown in Figure. The findings are similar for higher radial mode orders. The over-all sound power levels are summarized in Table4-1 for three configurations. It can be observed that the differences due to the inflow distortions are not exceeding 3 dB. For the rectangular and T-junction configurations the blade passing frequency is cut-on, but much smaller than its harmonics.
Figure 4-2: The dominant mode structures generated by the rotor-stator interaction are visualized, BPF cut-off, 2BPF (left), 3BPF (middle), and 4BPF (right).

<table>
<thead>
<tr>
<th>BPF 1</th>
<th>BPF 2</th>
<th>BPF 3</th>
<th>BPF 4</th>
<th>BPF 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cut off</td>
<td>97.9</td>
<td>90.2</td>
<td>75.2</td>
<td>76.4</td>
</tr>
<tr>
<td>37.8</td>
<td>95.8</td>
<td>87.4</td>
<td>76.0</td>
<td>75.6</td>
</tr>
<tr>
<td>45.7</td>
<td>96.2</td>
<td>87.9</td>
<td>76.7</td>
<td>76.3</td>
</tr>
<tr>
<td>Not computed</td>
<td>98.2</td>
<td>94.8</td>
<td>80.4</td>
<td></td>
</tr>
</tbody>
</table>

Table 4-1: Overall sound power level in dB [re 1e-12 W] for the computed configurations.

Figure 4-3: Sound power levels in dB [re 1e-12 W] for the upstream (top) and downstream (bottom) propagating azimuthal mode orders $m$ of radial mode order $n = 0$. Note that for the T-junction and Rectangular configuration the downstream modes are not computed.
4.5 Broadband simulation

According to our investigations, the influence of the non-steadiness of the inflow distortion on the velocity field is too small to have an impact on broadband noise (see D4.2). Subsequently we will study the effect of the different steady inflow conditions focusing on rotor-stator-interaction noise.

4.5.1 Flow extraction and transformation in 2D

The CAA simulations were performed in 2D space as the method is still in development. A 3D simulation would have been feasible, but was not performed in this project. Following, the flow extraction from the 3D CFD simulation is explained; the numbering refers to Figure. For more details see D4.2 and [WohlbrandtAIAA2015].

1) Starting from the cleaned CAD geometry
2) the CFD simulation for the three configurations are performed as explained in section 4-3.
3) The flow is extracted following a mean line starting at 50% radial position at the stator leading edge and is expanded to a full circle. Note that to get a smooth flow the rotor has been moved 10 mm away from the stator and an additional quasi 3D simulation is performed to get the flow for the CAA simulation. This step is necessary as otherwise the potential field of the stator acts on the passing rotors and the rotor could not be neglected in the CAA simulation.
4) The flow in the rotor domain is transformed from moving frame of reference to the absolute frame of reference and interpolated to the CAA grid. In regions where the CAA domain is larger than the CFD domain the flow is simply extrapolated by nearest-neighbor interpolation.
5) A $\alpha m' \theta$-transformation is used to transform the geometry and flow into 2D space.

![Figure 4-4: Showing the interpolation of the flow from 3D CFD simulation to 2D CAA simulation.](image)

4.5.2 Turbulence

The synthetic turbulence generated by the fRPM method is imposed onto one stator vane at an upstream position. Two different methods are investigated, for further explanations of the methods refer to D3.6 & 7 or to [WohlbrandtAIAA2015]:

1. **Stationary turbulence**: The turbulence of the rotor wake is mixed out in front of the stator (at the mixing interface) and imposed as uniform in the stator block. The values for the turbulence have been extracted using the in-house tool C3D_T2P, which extrapolates the turbulent kinetic energy and integral length scale from the rotor domain to the stator leading edge [Jaron2015]. After extrapolation the turbulent kinetic energy is reduced by a factor 0.44 and the integral length scale by a factor 0.44. Which means using the values directly would have led to an over-prediction of the noise level and a frequency shift to higher frequencies. The values used for the three configurations are summarized in Table 4-2.
2. **Non-stationary turbulence:** The turbulence characteristics are imposed in the rotor frame of reference. In this way the integral length scale and turbulent kinetic energy remain distinguished between inflow turbulence and wake turbulence. The error using the values from the CFD directly is overcome by applying the factors determined in the last item.

Note that considering a periodic background flow is in general possible and implemented, but for the current investigation is hard to accomplish due to the small distance of rotor and stator.

| Table 4-2: Turbulent characteristics used for the stationary broadband simulations |
|---|---|---|
| | Bell | T- | Rectangular |
| Turb. vel | 1.5 m/s | 3.5 m/s | 2.5 m/s |
| TLS Λ | 1.2 mm | 2.65 mm | 1.55 mm |

**4.5.3 Setup**

![Figure 4-5: Setup for the broadband simulations. The whole CAA domain in stream-surface coordinates is shown. The vorticity is generated by the inflow patch (blue) and eliminated by the outflow patch (green). The vorticity (red) is convected by the mean flow and impinges on the vane, generating noise, visible as the instantaneous pressure field in grayscale. The orange dots indicate the position of the microphones.](image)

The setup for the broadband simulations is shown in Figure 4-5. The whole CAA domain in stream-surface coordinates is shown with the instantaneous pressure field in grayscale. The cascade is realized with all 10 stator vanes and a periodic boundary condition in y-direction. In axial direction radiation and outflow condition are combined with a sponge region to minimize reflections. The turbulence is generated by the FRPM method and imposed to the CAA domain using the LEE-relaxation method [Ewert2014] at the inflow patch (blue). Only a single vane is excited assuming uncorrelated noise for adjacent vanes. The turbulence is eliminated by a second patch (green) to reduce spurious noise in the downstream microphones. The orange dots indicate the position of the microphones.
### Table 4-3: Over-all sound power levels for stationary realisations.

<table>
<thead>
<tr>
<th></th>
<th>Bell</th>
<th>T-ji</th>
<th>Rectangular</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turb. vel</td>
<td>1.5 m/s</td>
<td>3.5 m/s</td>
<td>2.5 m/s</td>
</tr>
<tr>
<td>TLS Λ</td>
<td>1.2 mm</td>
<td>2.65 mm</td>
<td>1.55 mm</td>
</tr>
<tr>
<td>PWL</td>
<td>85.1dB</td>
<td>91.2dB</td>
<td>89.8dB</td>
</tr>
<tr>
<td>delta PWL</td>
<td>-</td>
<td>+6.1dB</td>
<td>+4.7dB</td>
</tr>
</tbody>
</table>

#### 4.5.4 Results

##### 4.5.4.1 Stationary turbulence

The results using the stationary turbulence on the inflow patch are shown in Figure 4-6 to Figure 4-8 and compared as over-all sound power levels in Table.

It can be concluded that the simulation reproduces the order of magnitude of the measurements and follows the trends predicted by the analytical solution of PropNoise (a DLR tool dedicated to fan noise prediction). The predictions for the Rectangular configuration diverge, the reason is unclear, but PropNoise uses the whole duct section, while the simulations are based only on the 50% duct height. Also for high frequencies the two methods diverge. Comparing the three configurations to each other the results are consistent in such a way that a higher turbulent velocity results in higher acoustic levels.

![Figure 4-6: Spectral results of the stationary turbulence realisation of the bell mouth configuration compared to analytical results of PropNoise and experimental results of the multi port method.](image-url)
Figure 4-7: Spectral results for the stationary turbulence realisation of the T-junction configuration.

Figure 4-8: Spectral results for the stationary turbulence realisation of the rectangular configuration.
4.5.4.2 Non-stationary turbulence

For the Bell mouth configuration the influence of the background turbulence on the sound emission is investigated. For this purpose the turbulence on the patch in the moving frame of reference is divided into a back ground patch and a wake patch. The spectral results are compared to the stationary case in Figure 4-9. It is obvious that the back ground turbulence for this configuration is negligible; the tabulated results in Table underline this. The comparison of the spectral and over-all results of the stationary and non-stationary cases shows that neglecting the non-stationarity is legitimate. These results correspond to the findings in [WohlbrandtAIAA2015]. Note that the influence of the periodic background flow has not been investigated here.

Table 4-4: Sound power levels for non-stationary realisations.

<table>
<thead>
<tr>
<th></th>
<th>Upstream</th>
<th>Downstream</th>
</tr>
</thead>
<tbody>
<tr>
<td>Background turb. only</td>
<td>76.6dB</td>
<td>75.0dB</td>
</tr>
<tr>
<td>Wake turb. only</td>
<td>80.4 dB</td>
<td>77.8 dB</td>
</tr>
<tr>
<td>Background + wakes</td>
<td>82.0 dB</td>
<td>79.6 dB</td>
</tr>
<tr>
<td>Stationary turb.</td>
<td>82.0dB</td>
<td>82.2dB</td>
</tr>
</tbody>
</table>

Figure 4-9 Comparison of the non-stationary to the stationary results.

4.6 Conclusion and outlook

The fRPM method in application to fans as developed in WP3 has been successfully applied to the IDEALVENT fan in WP4. Comparison to measurements and analytical tools shows that the trends are predicted correctly.

The benefit of this method has been demonstrated to be in the focused study of separated effects of sound generation in the rotor-stator-interaction. These findings are of interest for analytical modelling of sound sources. Here the influence of the background turbulence could be quantified as negligible in the bell mouth case and also the non-stationarity of the turbulent wakes has no significant effect on the resulting spectra. Further investigations of the other inflow configurations as well as flow non-stationarities would be of much interest, but could not be investigated so far.

4.7 Bibliography


5 Developments and achievements made by KTH

5.1 Introduction

Within the DoW 4.3 of the IDEALVENT project, KTH has performed computations of the sound scattering of a single orifice and tandem orifice configuration placed inside the ECS-duct. Two tandem configurations were considered, one with a separation distance between the two diaphragms of 4D (duct diameters) and another one with 2D. The diameter of the ECS-duct was 0.15m and the aperture of the orifice was 0.116m which resulted in an open area ratio of around 0.6. The parameters corresponded to the measurement setup at KTH and VKI and, hence, allowed a direct comparison with the results from DoW 2.2. The scattering for both, a case without duct flow and a case with duct flow of a bulk velocity of 25 m/s were computed. The latter case gave together with the contribution by NTS a full numerical multi-port characterization for the single orifice and both of the tandem configurations.

The computations were carried out solving the Linearized Navier Stokes Equations (LNSE) in the frequency domain using the Finite Element Method (FEM) in Comsol Multi-Physics. This has successfully been applied in earlier work by Kierkengaard et al. [1] on the plane wave mode and was now extended to higher order mode propagation, e.g., a total number of 6 propagating modes was computed upstream and downstream. The full formulation of the Linearized Navier Stokes equations can be seen in Equation 1-3

\[ \rho \omega \rho + \nabla \cdot (\rho \vec{u} + \rho_0 \vec{u}) = \mathcal{M} \]
\[ \rho_0 (i \omega \vec{u} + (\vec{u} \cdot \nabla) \vec{u}_0 + (\vec{u}_0 \cdot \nabla) \vec{u}) - \nabla \cdot \sigma = \mathcal{F} \]
\[ \rho \mathcal{C}_p (i \omega \vec{T} + (\vec{u} \cdot \nabla) \vec{T}_0 + (\vec{u}_0 \cdot \nabla) \vec{T}) + \rho \mathcal{C}_p (\vec{u} \cdot \nabla) \vec{T}_0 \]
\[ - \alpha_0 (\omega \rho + (\vec{u} \cdot \nabla) \rho_0 + (\vec{u}_0 \cdot \nabla) \rho + \alpha_0 \vec{T} (\vec{u}_0 \cdot \nabla) \rho_0 - \nabla \cdot (\mu \nabla \vec{T}) = \mathcal{Q} + \mathcal{Q} \]

\[ \sigma = -\rho \vec{l} + \mu \nabla \vec{u} + (\nabla \vec{u})^T + \left( \mu_b - \frac{2}{3} \mu \right) (\nabla \cdot \vec{u}) \vec{l} \]
\[ \rho = \rho_0 (\beta \rho - \alpha_0 \vec{T}) \]

Where \( \rho \) is the density, \( p \) is the pressure, \( \vec{u} \) is the velocity vector, \( \omega \) is the angular frequency, \( T \) is the temperature, \( \mathcal{F} \) is the force-vector, \( \mathcal{M} \) is the mass-source, \( \mathcal{Q} \) is a heat-source, \( \mu \) is the dynamic viscosity, \( \mu_b \) is the bulk viscosity, \( \mathcal{C}_p \) is the heat coefficient and the index 0 denotes a property of the stationary background flow. Kierkengaard assumed, however, the acoustics wave propagation in ducts to be fully adiabatic which yielded good accuracy of his results. This approach simplifies the problem as it decouples the energy-equation (Equation 3) from the mass-equation (Equation 1) and momentum-equation (Equation 2) and hence decreases the degree of freedom.

The computations were carried out in a hydride approach. In a first step, the stationary, incompressible background flow was computed using RANS with a \( \kappa-\varepsilon \) turbulence model on
a two-dimensional, axisymmetric grid. The result for velocity and pressure where then used as background fields for the LNSE computation in a second step. The flow field simulation was decoupled from the acoustic field, i.e. strong perturbations in the acoustic would not change the flow inside the ECS duct. However, as the stationary flow-field was one-way coupled to the acoustic computation the model generally accounted for one-way interactions between the hydrodynamic and the acoustic field, i.e. acoustic shedding at the edges of the orifice.

5.2 Flow Computations
The solution of the flow field was assumed to be axisymmetric as the orifice is a fully symmetric element. The κ-ε RANS flow computations where carried out on a two-dimensional, axisymmetric grid. A structured grid with quadratic elements with a maximum element size of 0.01D was used. The computational domain was chosen to have a length of 50D on the upstream side to ensure a fully developed flow profile and a length of 10D downstream. The boundary layers were resolved at the duct walls and around the orifice using a boundary layer mesh containing 8 layers. The average inlet flow speed was U=25m/s, yielding a Mach number of M=0.073. An ambient pressure boundary condition was applied to the outlet.

5.3 Acoustic Computations
For the acoustic computation, the adiabatic, linearized Navier Stokes Equations were solved using the acoustic module in COMSOL multi-physics. Two different kinds of models were used

- For the radial modes (the (0,0) and the (0,1) mode ) a two-dimensional axisymmetric model was used
- For the circumferential modes (the (1,0) and the (2,0) mode) a three-dimensional model of a quarter duct was used. That approach simplified the computations, i.e. drastically reduced the degree of freedom and hence the computational costs. However, it assumes no scattering from the radial to the circumferential modes. This assumption was reasonable as the problem appears to be completely axis-symmetric.

The scattering of the single orifice with flow is presented in Figure 5-2 for the (0,0) mode over the entire frequency. We found, that the agreement for low frequencies is very good and the results differ less than 1% up to a frequency of around 1500 Hz. For higher frequencies, the results differ slightly, which is especially visible for the reflection coefficients. This result agrees with the findings in ref. [1] that showed a higher sensibility of the reflection coefficients for uncertainties in LNSE- computations.
5.4 Combining Multi-Ports

5.4.1 Passive part

We consider a system of two passive multi-ports as shown in Figure 5-3 where $p^+$ and $p^-$ are complex vectors of outgoing and incident pressure mode amplitudes and $S$ and $\Phi$ are the scattering matrices such that

\[
\begin{bmatrix}
 p_1^+ \\
 p_2^+ \\
 p_3^+
\end{bmatrix} = S
\begin{bmatrix}
 p_1^- \\
 p_2^- \\
 p_3^-
\end{bmatrix}, \quad S = \begin{bmatrix}
 S_{11} & S_{12} \\
 S_{21} & S_{22}
\end{bmatrix}
\]

\[
\begin{bmatrix}
 p_2^+ \\
 p_3^+
\end{bmatrix} = \Phi
\begin{bmatrix}
 p_2^- \\
 p_3^-
\end{bmatrix} \quad (4)
\]

The aim of this investigation is to combine the two multi-ports to one multi-port with a distinct scattering matrix. We can reformulate Equation 4 to a transfer matrix

\[
\begin{bmatrix}
 p_1^+ \\
 p_2^+
\end{bmatrix} = S' \begin{bmatrix}
 p_2^+ \\
 p_2^-
\end{bmatrix}
\]

\[
\begin{bmatrix}
 p_1^+ \\
 p_2^+ \\
 p_3^+
\end{bmatrix} = S' \Phi' \begin{bmatrix}
 p_1^- \\
 p_1^-
\end{bmatrix} = S' \phi' \begin{bmatrix}
 p_1^- \\
 p_1^-
\end{bmatrix} \quad (5)
\]

Which allows us to combine passive multi-ports in a simple matrix multiplication

\[
\begin{bmatrix}
 p_3^+ \\
 p_3^-
\end{bmatrix} = S' \phi' \begin{bmatrix}
 p_1^+ \\
 p_1^-
\end{bmatrix} = S' \phi' \begin{bmatrix}
 p_1^+ \\
 p_1^-
\end{bmatrix} \quad (6)
\]

$S_c$ is the combined scatter matrix.

We found that the prediction of the scattering of tandem orifice configurations using Equation 6 works well for low Mach-number flows (M=0.1) even for higher order modes, as can be seen in Figure 5-4.
Figure 5-4: Predicted and measured scatter coefficients for the tandem diaphragm with a distance of 4D and low Mach-number flow (M=0.1).

5.4.2 Active Part

We consider a system of two active multi-ports as shown in Figure 5-5 where $p^+$ and $p^-$ are complex vectors of outgoing and incident pressure mode amplitudes, $S$ and $\Phi$ are the scattering matrices of the two multi-ports and $P_{s/\Phi}$ are the (reflection free) source vectors at the inlet and outlet of the multi-port. We can find a set of equations for the coupling between the outlet of $S$ and the inlet of $\Phi$

\[
p_+ = \Phi_{11}p_- + p_{\Phi b}
\]

(7)

\[
p_- = S_{22}p_+ + p_{sb}
\]

(8)

\[
p_+ = (\Phi_{11}p_{sb} + p_{\Phi b})(E - \Phi_{11}S_{22})^{-1}
\]

(9)

\[
p_- = (S_{22}p_{\Phi b} + p_{sb})(E - S_{22}\Phi_{11})^{-1}
\]

(10)

Equation (9) and (10) can be used compute the source vector of the combined multi-port

\[
P_{a+} = S_{21}(E - \Phi_{11}S_{22})^{-1}(\Phi_{11}p_{sb} + p_{\Phi b}) + p_{sa}
\]

\[
= D_{12}(\Phi_{11}p_{sb} + p_{\Phi b}) + p_{sa}
\]

(11)

\[
p_{b+} = \Phi_{21}(E - S_{22}\Phi_{11})^{-1}(S_{22}p_{\Phi b} + p_{sb}) + p_{\Phi a}
\]
It is advantageous to rewrite equation (11) and (12) in terms of cross-power spectra as those spectra contain reflection free acoustic data that is cleaned of hydrodynamic pressure fluctuations:

\[ G_{aa+} = P_{a+}P_{a+}^c = D_{12}(\phi_{11}G_{sbb}\phi_{11}^c + G_{\phi bb})D_{12}^c + 2Re(D_{12}\phi_{11}G_{sba}) + G_{saa} \]  

(13)

\[ G_{ab+} = P_{a+}P_{a+}^c = D_{12}(G_{\phi bb}S_{22}^c + \phi_{11}G_{sbb})D_{21}^c + D_{12}G_{\phi ba} + G_{sab}D_{21}^c \]  

(14)

Where \((\ldots)^c\) denotes the complex conjugated transposed. We can find similar formula for \(G_{bb+}\) and \(G_{ba+}\). In Equation 13 and 14, cross terms between the sources are neglected, i.e. uncoherent sources are assumed. This is a strong assumption which introduces uncertainties if the source mechanism are correlated, e.g. when the sources are too close. This behavior is demonstrated in Figure . We found, that the distance has a very strong effect on the source-spectra of the combined orifice plates. One can see, that the spectrum is highly under-predicted for distances closer than 10D. Characteristic is the periodic pattern that corresponds to the distance of the plates and is due to the neglected (periodic) cross-term \(2Re(\phi_{11}P_{ab}P_{\phi a})\) in Equation 13. The case with longer distances, however, could be predicted with good accuracy (divergence < 2dB for all modes).

### 5.5 Extract Source Data From Compressible Computations

The methods used in WoD 2.2 in order to extract reflection free acoustical source data from measurements are not restricted to measurement data but can be used on sensoring points in an numerical grid. The decomposition of numerical data brings a number of advantages compared to the empirical approach. Admittedly, the computed time samples are still much shorter than data gained from measurements; However, a plethora of sensoring-points is accessible, even inside the channel cross-sections. The decomposition matrix is hence highly overdetermined and guarantees a stable matrix inversion. Furthermore, in computations we can disable all the disturbing effects which usually complicate measurements. We have already introduced a decomposition method in WoD 2.5. However, we further improved the approach, using cross-correlation spectra instead of FFTs which allows more efficient and accurate averaging and stable decompositions.

As we are not restricted to a low number of sensoring points as in measurements, we can decompose the computed source field in p+ and p- waves and hence do not need to extract the reflection coefficients of the inlet and outlet from a separate computation (compare WoD 2.5). We use the multi-port equation:

\[ P_+ = Sp_- + p^s \]  

(15)

Which we solve for the source vector. To separate the acoustic pressure fluctuations from the (strong) hydrodynamic pressure fluctuations, it is beneficial to use cross-correlation pressure spectra instead of pressure [2]. One could rewrite equation 23 in terms of cross-spectra:

\[ G^s = G_+ - S G_- S^c \]  

(16)

Where \(G^s\) is the cross pressure source matrix \(G^s = p_p^c\), \(p_p^c\) is the source vector and \(^c\) denotes the complex conjugated transposed. If we define a decomposition matrix \(M\) such that
Figure 5-5: Active part of a tandem diaphragm for different distances, compared to a predicted spectra from a single orifice measurement.
\[
M^{-1}p = \begin{bmatrix} p^+ \\ p^- \end{bmatrix},
\]

where \( p \) is a set of sample points in a numerical grid, we can decompose the simulated pressure field into pressure source matrices in + and – direction by splitting the computational domain in two sub-domains (domain 1 and domain 2)

\[
M_1^{-1}p_1M_2^{-1}p_2 = M_1^{-1}p_1p_2^cM_2^c = M_1^{-1}G_12M_2^c = \begin{bmatrix} G_+ & G_\pm \\ G_\mp & G_- \end{bmatrix}.
\]

(18)

A difficulty that directly follows from the equations 15 is the appearance of the scattering matrix that has to be determined in order to extract the reflection-free source data. The scattering matrix, however, has to be extracted with considerable effort from measurements or computations. One could simplify equation 16 assuming that the reflections at the inlet and outlet of the computational domain are small. This assumption, however, has to be proven by computing the reflections of the terminations

\[
R = G_+^{-1}G_-
\]

(19)

whereas the absolute values of each element in \( R \) should be <<1. As we found, this assumption might be violated closely after the cut-on frequency of the higher order modes.

If the reflections of the terminations are high, the scattering of the test object has to be taken into account. Apparently, these scattering data cannot be gained from the same computation as the source data (see WoD 2.2). Nevertheless, one could use scattering data that was determined in a simplified computation, i.e., using the Linearized Navier Stokes equations or data from measurements.

5.6 Bibliography


6 Developments and achievements made by KUL

6.1 Introduction

KUL performed in Task 4.1, numerical simulations for the butterfly with bend case with the time-domain RKDG methods for which improved boundary conditions and coupling techniques were developed in WP3. As simulation technique, the impulse response method, developed and validated in D3.5 for an empty duct with and without the presence of a mean flow, is used to determine the passive scattering matrix of the butterfly valve. These simulations are compared with the detailed experimental measurement campaign, discussed in D2.4. A more in-depth discussion of this analysis is presented in [1].

This deliverable focuses on following aspects of the butterfly valve:

- The ability and an accuracy-assessment to model the passive multi-port scatter matrix coefficients with a time-domain RKDG solver using the Linearized Navier-Stokes (LNS)-equations as propagation equations

- Influence of modelling issues on the final accuracy of the passive multi-port characterization including:
  
  - An analysis of the influence of the total simulation time on the final accuracy. As, in the vicinity of the cut-on frequency, the acoustic field is characterized by very small axial wave number, the acoustic waves are propagating very slowly upstream and downstream in the measurement ducts. As such, an in-depth time convergence analysis is needed to determine the total simulation time.
  
  - An analysis of the influence of the geometrical complexity of the butterfly valve on the final accuracy. The butterfly valve can indeed be simplified to a simple flat plate disc geometry, reducing the total number of element and thus also the computational time. Although this simplification cannot be justified from an aerodynamic point of view (due to the small aerodynamic length scales), it might give reasonable acoustic results, especially in the low frequency region were the acoustic waves are characterized by a large acoustic wavelength.

The aspects are considered and discussed in detail in this deliverable for the butterfly valve without bend under quiescent conditions. The fact that the presence of the bend is not considered in this deliverable is motivated by the detailed experimental measurements, discussed in D2.4, where it is shown that the presence of the bend has almost no influence on both the passive and active multi-port scatter matrix coefficients. As a result, the analysis of the butterfly valve has been extended to incorporate the influence of modelling issues on the final accuracy of the simulations.
6.2 Numerical set-up

For the numerical validation of the multi-port characterization an in-house, time-domain Runge-Kutta Discontinuous Galerkin (RKDG) discretization of the Linearized Euler Equations (LEE) is used, discussed in D3.2. Linear elements, allowing the use of an efficient quadrature-free DG formulation, are used throughout the domain except in the vicinity of the curved walls of the circular duct where curved second-order elements are locally used locally around complex geometries [1]. This mixed approach results in an optimal balance between the computational efficiency of the quadrature-free method and the geometrical flexibility of the second-order elements. Due to the small mesh size in the vicinity of the valve the use of quadratic elements in this region is not needed. For the inter-element communication, a Lax-Friedrich formulation is used for the convective fluxes. Boundary conditions are imposed by specifying the fluxes at the boundary faces. The time integration is done using a Runge-Kutta scheme, optimized for the spatial discontinuous Galerkin operator [2].

As discussed in D2.5, an important advantage of a time-domain simulation is the fact that an arbitrary choice of the excitation signal is enabled. The passive multi-port characterization technique assumes a dominant external excitation, for which, in principle any type of signal can be used. For the experimental multi-port characterization a proper choice of excitation signal is needed to increase the signal-to-noise (S/N) ratio of the microphone signals. When using a numerical approach the S/N-ratio is, however, sufficiently large and the choice of the excitation is mainly dominated by the required computational time and the accuracy of the obtained multi-port characteristics. In this framework the preferable approach is an impulse response method [3]; i.e. applying a pulse excitation, thus exciting the full frequency range with only one transient signal, and running the simulation until all or most part of the energy has left the system.

Although this approach has the potential to yield a significant speed-up with respect to a steady state multi-sine technique, the multi-modal character of the multi-port characterization poses a significant issue in comparison to the impulse response method classically used for the determination of the two-port characteristics. At the cut-on frequency, a mode does not propagate in the axial direction since the real part of the axial wavenumber exactly equals zero. With a discrete frequency excitation, these frequencies can be easily avoided by not including them in the excitation signal. However, when using a pulse excitation, all frequencies will be excited by definition. Therefore, also energy is fed into the transversal modes at the cut-off frequency and the energy will only leave the system through numerical damping. This will result in prohibitive computational times to get fully converged transient time signals.

The results at the cut-off frequencies are, however, of no significant importance for the multi-port characterization since the singularity of the modal matrix simply makes it impossible to apply a modal decomposition at these frequencies. Hence, it is sufficient that the energy contained in the other frequencies has left the system to consider the simulation to be converged. A window function can then be applied to the full time-signal before applying the FFT to avoid leakage phenomena to occur.
This approach is applied to both the butterfly valve and a simplified geometry representing the valve as a flat circular plate in a quiescent medium. A 3-dimensional tetrahedral mesh with 71,731 and 44,471 first-order elements is used to properly represent the geometry of, respectively, the butterfly valve (left of Figure 6-1) and flat plate configuration (right of Figure 6-1) both the geometry and positioning of the excitation and monitoring point is identical to the experimental set-up, discussed in D2.4.

![Figure 6-1: Surface discretization of the butterfly valve (left: complete valve geometry; right: simplified flat plate geometry)](image)

In all simulations, a full duct length of 58D is considered. The valve is positioned on the middle with regard to the duct full length. For all cases, a 4th order polynomial is used as a basis function within the RKDG method, which is able to accurately represent the solution for the configuration cases under consideration. A damping layer (along which axial length, equal to 9D, the fluctuating variables are gradually damped, following a 3rd power, to the steady-state reference variables) and characteristic non-reflecting boundary conditions are imposed at the inlet and outlet of the duct. All simulation cases are run for a non-dimensional period of $\Delta t^* = \Delta t \frac{c_0}{D} = 500$.

Similar as for the experimental multi-port characterization, six simulations are carried out, changing the position of the Gaussian pulse. All six Gaussian pulse excitations are located at a distance of 12.5D from the center of the valve, 3 simulations (with pulse locations at a radius equal to 0.3D and varying azimuthally 120°) are carried out with an upstream excitation and three simulations (with identical cross-sectional pulse locations) are carried out with a downstream excitation. The width (standard deviation) of each Gaussian pulse is set to 0.2D.

The time dependent pressure fluctuations are stored at two sets of 54 monitoring positions arranged into nine cross sections, located at, respectively, a upstream and downstream distance of 8.1D, 7.7D, 7.3D, 6.9D, 6.5D, 6.2D, 6.0D, 5.5D and 5.0D, from the center of the valve. At each of the nine cross-sections, six equally distributed monitor points are applied along a circumference radius of 0.375D. In order to minimize sensitivity errors related to the measurement points distribution, the longitudinal spacing ($\Delta z$) between each of the nine cross sections is chosen according to the criteria, suggested in [4], as $0.1\pi < k_0\Delta z < 0.8\pi$. A half-Hanning window is applied to the time-data before applying the FFT algorithm.

As mentioned in D2.4, depending on the number of monitor points and their spatial distribution, the linear system of equation can become ill-posed or singular. To evaluate the
error-sensitivity of the linear system, the condition number of the modal matrix $M$ is calculated, and the obtained values are shown in Figure 6-2. As can be noticed, the condition number values are sufficiently small over the whole frequency range of interest; except, in a narrow-band frequency region close to the cut-on frequency (2372 Hz). Due to the extremely small values of the axial wave number in this frequency range, larger condition numbers are observed.

![Figure 6-2: Condition number of the modal matrix $M$ for the spatial distribution of the 54 monitoring points.](image)

6.2.1 Comparisson with the passive scatter matrix without flow
The magnitude of the different numerically and experimentally obtained transmission and reflection scattering matrix coefficients for the butterfly valve configuration are shown in Figure 6-3. An excellent agreement between the experimental and numerical results can be observed for all modes, except at frequencies slightly above the cut-on frequency where the numerical results show, similar as for the circular duct results, an increased transmission and a decreased reflection. Since both the magnitude and frequency range of these deviation is very similar to the circular duct case, presented in D3.5, it can be expected that this is caused by the fact that, in this frequency region, the sound field in the upstream and downstream ducts is dominated by the azimuthal modes (1,0) and (-1,0) which are (due to the very small axial wavenumber) propagating very slowly in the upstream and downstream directions. On the one hand, this imposes severe difficulties to obtain a series of linear independent measurements, required for the experimental determination of the scatter matrix coefficients. This was evidenced by the poor condition number of the, to be inverted, matrix. On the other hand, the slow propagating acoustic field, makes the solution very sensible to dissipation mechanisms caused by e.g. viscous effects of the medium and damping caused by the duct walls.
Figure 6-3: Comparison of the numerically (full) and experimentally (dashed) reflection (red) and transmission (black) scatter matrix coefficients of the butterfly without flow for the, right from top to bottom, downstream (0,0), (1,0) and (-1,0) mode and the left from top to bottom upstream (0,0), (1,0) and (-1,0) mode.

At low frequencies (up to, approximately, 1500 Hz) in the plane wave region the result shows an excellent agreement with an analytical 1D model of a duct system with a sudden area contraction-expansion with an infinitesimally small length (i.e. similar to a diaphragm) with a contraction ratio equal to $1 - \cos \alpha = 0.134$, $\alpha$ being the valve angle, equal to 30°. In this case, for the low frequency region the transmission coefficient linearly decreases with increasing frequency while the reflection coefficient linearly increases. The slope of the linear decrease and increase of, respectively, the transmission and reflection coefficient is determined by the contraction ratio of the obstacle, in this case the butterfly valve. This phenomenon is clearly predicted by both the numerical simulations and the measurements.

At higher frequencies, but still in the plane wave region, the assumption of an infinitesimally small valve length does not longer hold since the acoustic wavelength becomes of the same order of magnitude as the length of the valve body. In this region, a sudden decrease and afterwards increase of the transmission coefficient is noticed and an opposite phenomenon of the reflection coefficient is observed. Above the cut-on frequency of the first circumferential mode and its complex conjugate, a slight decrease of the transmission coefficient is noticeable with increasing frequency. As no flow is present, a reciprocal behaviour (i.e. a similar behaviour of the upstream and downstream propagating modes) is expected. This behaviour is present below the cut-on frequency in the different reflection and transmission coefficients, shown in Figure 6-3. Above the cut-on frequency, the reciprocal assumption is, however not fully valid anymore, which can be noticed when looking at the reflection coefficients in
This phenomenon, observed both in the experimental and numerical results, might be caused by the fact that the valve is non-compact in this frequency region. Over the whole frequency region both the upstream and downstream propagating (-1,0) and (1,0) modes show an almost identical behaviour, as expected.

6.2.2 Influence of the simulation time

Since a time-domain impulse response method is used to simulate the frequency-dependent scattering matrix coefficients, a thorough analysis of the influence of the total simulation time on the final scatter-matrix determination is carried out. Due to the small axial wavenumbers (i.e. the small propagating velocity of the upstream and downstream propagating waves) in the frequency region close to the cut-on frequency, a convergence analysis is needed to verify that the simulation time is indeed large enough to simulate all physical phenomena, as well as to obtain a sufficient accuracy to determine the scatter matrix coefficients in the vicinity of the cut-on frequency. For this analysis, a different number of non-dimensional time-steps (i.e. 176, 338, 500, 662, and 824) are considered and presented in Figure 6-4.

![Figure 6-4: Influence of the simulation time on the reflection (red) and transmission (black) scatter matrix coefficients of the butterfly without flow for the, from downstream propagating (0,0) - mode (top) and (1,0) mode.](image)

It can be observed in Figure 6-4 that, indeed a sufficiently large computational time is needed to represent the acoustic scatter matrix in the vicinity of the cut-on frequency. Furthermore, it is shown that for the simulation times, containing 370 of 500, 662 and 824 non-dimensional time-steps, the discrepancies between the different simulations are negligible, even in the vicinity of the cut-on frequency.

6.2.3 Influence of the geometry

In order to investigate the necessity to model the full geometrical complexity of the butterfly valve, an additional numerical analysis (under quiescent conditions) is carried out, where the butterfly valve geometry has been simplified to a flat plate (i.e. disc) case with an identical angular position (i.e. 30°) as for the butterfly valve considered in the previous sections. As shown in Figure 6-5, the pattern of the simulated scatter matrix coefficients is very similar to
the corresponding scattering matrix coefficients obtained for the complete valve geometry. It is noted that, over the whole frequency range, the transmission coefficients values are slightly larger in comparison to both the experimental and numerical data of the full valve geometry; while the amplitudes of reflection coefficients is slightly underestimated. This is indicates that the geometrical complexity of the full butterfly valve geometry, and in particular the presence of the circular rod around which the valve is rotating, introduces (as expected) an increased reflection and decreased transmission. However, as the discrepancies of the coefficient values remain small and approximately constant over the whole frequency range under investigation, the additional detailed components of the valve geometry (if compared to the circular, infinitely thin plate geometry) do not contribute significantly to the sound scattering characteristics of the butterfly valve, thus, depending on the required accuracy, the three-dimensional butterfly valve can be simplified by a single plate but keeping the same area contraction ratio (i.e. introducing the vertical rod in the simplified geometry).

Figure 6-5: Comparison of the experimentally (dashed) and numerically (complete valve geometry: full; simplified valve geometry: dotted) reflection (red) and transmission (black) scatter matrix coefficients of the butterfly without flow for the, right from top to bottom, downstream (0,0), (1,0) and (-1,0) mode and the left from top to bottom upstream (0,0), (1,0) and (-1,0)mode.

6.3 Conclusions

KUL successfully applied the time-domain RKDG impulse response method to the butterfly valve case under quiescent conditions. The results of the passive scattering matrix coefficients are in excellent agreement with the previously obtained experimental data. A time convergence analysis was carried out in order to estimate the total simulation time needed to get accurate multi-port data. It is shown, that especially in the close vicinity of the cut-on
frequency, the simulations need to be carried out over a sufficient amount of time steps to get accurate results. A comparison, with simplified flat plate geometry, shows that the geometrical complexity of the butterfly valve does not have a significant impact except for the radial rod which is needed for the angular positioning of the butterfly valve which causes an increased reflection and decreased transmission in both the upstream and downstream directions.

6.4 Bibliography


7 Developments and achievements made by SISW

7.1 Introduction
In the framework of WP4, SISW has contributed with acoustic computations for the cases of the ECS fan with inlet distortions, the valve with upstream bend and the single and tandem diaphragm configurations. Both passive acoustic characterizations (scattering matrix) and active components (due to aeroacoustic sources) have been computed.

The acoustic boundary value problem is solved for all cases with a Finite Element Method with Adaptive Order (FEM AO) [1], available in Siemens PLM software LMS Virtual.Lab. This FE approach uses hierarchical shape functions for the field variables, and the order is automatically adapted as a function of the frequency. Some more details and extensions to the solver developed within IdealVent are described in D3.4 [2]. At the inlet and outlet, non-reflective boundary conditions are applied (Anechoic Duct End, see LMS Virtual.Lab [1]). These are based on a perfectly-matched layer, the size and properties of which are automatically adapted by the software for each frequency to optimize its absorbing behavior.

For the computations of the passive acoustic behavior of each case, a post-processing based on a multiport characterization is applied. The acoustic excitation for the passive acoustic computations is given by specific duct modes prescribed at the inlet or at the outlet.

For the computations of the active contributions, the aeroacoustic sources are based on the compressible CFD data provided by NTS (for the ECS fan cases and for the single and tandem diaphragm cases) and by VKI (for the valve case). Either surface-distributed dipoles based on wall pressure or volume-distributed sources based on velocity fluctuations are applied as a source term. The motivation for investigating such a hybrid approach is that it can predict the acoustic behavior even when the CFD data correspond to an incompressible flow description and thus do not contain any acoustic information, which is often the case for low Mach number flows. More details on the applied source models can be found in deliverables D4.1 [7], D4.2 [5] and D4.3 [6].

In this report, only the most representative results are shown. More details related to the computations and the results can be found in IdealVent deliverables D4.1 (valve case [7]), D4.2 (fan cases [5]) and D4.3 (diaphragm cases [6]).

7.2 Hybrid computations for the single and tandem diaphragm cases
Hybrid aeroacoustic computations were run for the single and tandem diaphragm configurations. The sources are based on the compressible CFD data provided by NTS [6]. Since the CFD is also able to provide the acoustic levels, these are used together with the experimental data as a reference solution to compare the results of the hybrid approach. The results are discussed in the following sections.

The geometry for all three cases is axisymmetric. The ducts have a diameter of D=0.150m, and the inner diameter of the diaphragms is 0.116m. the thickness of the diaphragms is 0.008m. The volume flow rate in each case was \( VFR = 0.42m^3/s \), which yields an average...
Mach number in the duct of around $M=0.07$. For the tandem diaphragm cases, the separation between diaphragms is $2D$ and $4D$, respectively.

Figure 7-1 displays a view of the finite element mesh used for the case of the tandem diaphragm case with separation $2D$, which is representative for the meshes used in this study.

![External view of the mesh for the tandem diaphragm case, with distance between diaphragms $2D$.](image)

7.2.1 Single diaphragm

Figure 7-2 shows the results obtained with dipole sources only on the diaphragm surface. The dipoles sources on the diaphragm are responsible for most of the noise production. Still, both the CFD and the hybrid results show some discrepancies with respect to experimental results [4]. On the one hand, especially at low frequencies, the experimental curve seems to be somewhat affected by reflection effects at the inlet and outlet. On the other hand, as shown by NTS, the deviation in the higher frequency range is likely be a consequence of the effect of the fluctuations of the turbulent boundary layer in experiments, which for the low acoustic levels present in this case competes in magnitude with the acoustic signal; by contrast, the signal given by the computational approaches is purely acoustic.

![Power spectral density upstream of the single diaphragm with turbulent inflow and VFR=0.42 m$^3$/s: Hybrid computations with dipole sources only on the diaphragm (gray line), direct CFD pressure measurements (red dashed line) and experiments (black line).](image)

7.2.2 Tandem diaphragm configurations

In the cases with two diaphragms, the sound production is concentrated around the second diaphragm, due to the effect of the turbulent flow generated behind the first diaphragm impinging on the second diaphragm. As a consequence, the dipole sources on the downstream
diaphragm are the dominant source of sound, and the sound production is less sensitive to the
type of flow (laminar or turbulent) that is injected upstream of the first diaphragm. Therefore,
dipole sources are defined only on the downstream diaphragm in the hybrid computations.

Figure 7-3 and Figure 7-4 show the results of hybrid computations with dipole sources
only on the downstream diaphragm. The acoustic pressure is measured in a section upstream
from the first diaphragm, and the results measured at 20 different azimuthal angles are
averaged. The results are compared to experimental measurements obtained in the framework
of IdealVent [4]. The hybrid computational approach seems to provide an overall good match
with the experimental data and with the direct CFD results, thus confirming that the dipole
sources on the downstream diaphragm are the dominant source of sound. For each case, two
different hybrid computations are carried out: one based on the CFD with clean inflow, and
another based on the CFD with upstream turbulence injection. It can be observed that this
yields some differences at low frequencies, where the flow with the turbulent inflow tends to
produce more noise. By contrast, the flow with the clean inflow seems to produce somewhat
more high-frequency noise.

Figure 7-3: Power spectral density upstream of the tandem diaphragms with 2D separation: Hybrid
computations with dipole sources only on the downstream diaphragm (gray line), direct CFD
pressure measurements (red dashed line) and experiments (black line). Numerical results with
clean inflow (left) and turbulent inflow (right).

Figure 7-4: Power spectral density upstream of the tandem diaphragms with 4D separation: Hybrid
computations with dipole sources only on the downstream diaphragm (gray line), direct CFD
pressure measurements (red dashed line) and experiments (black line). Numerical results with
clean inflow (left) and turbulent inflow (right).
The deviations of the numerical results with respect to experiments may be attributed to two factors, as in the case of the single diaphragm case. On the one hand, especially at low frequencies, the experimental curve is somewhat affected by reflection effects at the inlet/outlet. On the other hand, as shown by NTS, the deviation in the higher frequency range may be a consequence of the effect of the fluctuations of the turbulent boundary layer in experiments.

### 7.3 Passive behavior of the single and tandem diaphragms

Acoustic computations were run to characterize the passive acoustic behavior of the single and tandem diaphragm cases, and a multiport characterization method was applied to compute the scattering matrices.

Figure 7-5 display the amplitudes of the transmission and reflection coefficients for modes \((m,n)=(0,0)\) and \((m,n)=(1,0)\). Computations with different FE order were run (target accuracy of the adaptive order rule can be set to different levels, leading to lower or higher FE orders). The geometry was nevertheless always discretized with linear tetrahedral elements. It can be observed that the transmission amplitude for a given mode \((m,n)\) tends to suffer a sudden drop around the cut-on frequency of mode \((m,n+1)\), that is to say, at the frequency at which the next radial mode of the same azimuthal order is excited.

Figure 7-6 shows the computed transmission and reflection coefficients for one of the tandem diaphragm cases (2D separation), for modes \((0,0)\) and \((1,0)\). In spite of the relatively coarse frequency resolution, it can be seen that the curves present many oscillations, which are a consequence of the multiple reflections over each of the two diaphragms, and are consistent with the behavior obtained through experimental measurements by KTH.

![Figure 7-5: Amplitudes of the transmission (solid, ◦) and reflection (dashed, △) coefficients for the modes \((m,n)=(0,0)\) and \((1,0)\). Symbols correspond to computations run with higher FE order.](image-url)
This section presents the results for the passive acoustic characterization of the ECS fan. The computations were carried out without mean flow (Helmholtz equation). A detail of the envelope of the mesh inside the duct in the vicinity of the fan can be seen in Figure 7-7. The mesh is refined in this region in order to capture the geometrical details of the fan. Geometrical details such as small holes were simplified during the construction of the mesh.

Figure 7-8 shows the amplitudes of the transmission coefficients for some of the cut-on modes below 3600Hz. The negative azimuthal modes are not displayed because the results are practically the same (except for small numerical errors) as those for the positive azimuthal modes. The figures show two sets of computations, with different FE accuracy. In general, the results are robust and the curves for both accuracies are similar, but they show some sensitivity close to frequencies where a sudden drop of transmission is found, for instance corresponding to the resonance frequency of a cavity in the geometry.
Figure 7-8: Amplitudes of the transmission coefficients for the modes $(m,n) = (0,0), (1,0), (2,0), (0,1)$ and $(3,0)$. Standard accuracy (+), fine accuracy (solid line).

The results indicated that the first drop in transmission of the plane wave mode and in the first azimuthal mode is due to the cavity in the stator area, while the second has to do with the cavity in the front of the fan. If we observe the pressure field in the near-field of the fan, this becomes apparent. Figure 7-9 shows color plots of the pressure amplitude on two transversal planes near the fan at frequencies 1500Hz and 2250Hz, which are close to each of the two minima in the transmission coefficient for the plane wave mode. In this case, the duct is excited at the outlet (stator side) with waves corresponding to the plane wave mode. It can be observed that indeed frequencies 1500Hz and 2250Hz seem to be close to eigenfrequencies of the cavity on the stator and rotor side respectively, where the amplitude is higher.
With respect to the experimental data, the computed results show similar qualitative trends. For instance, both experiments and computations predict the drops in transmission due to the cavities at rotor and stator sides at similar frequencies. Nevertheless, the experimental results display in general lower levels of transmission, and less pronounced peaks, thus suggesting the presence of acoustic damping mechanisms that are not modeled in the acoustic computations.

### 7.5 Hybrid computations of the interaction noise of the ECS fan

The acoustic mesh for the computations of the active contribution (with aeroacoustic sources) of the fan cases is similar to the one used in the previous section, but the rotor surface is excluded from the mesh, leaving only the stator, where the sources are defined, in the duct mesh. The surface of the stator vanes is discretized by around 32000 quadratic triangular elements, and around 65000 nodes. Figure 7-10 shows a detail of the interior of the duct, where surface dipoles (based on pressure) are defined on the surface of the stator vanes. Since the rotor is excluded from the model, its effect is only reflected in the flow input used to define the sources on the stator. The mean flow is assumed to be negligible in the acoustic computations.
Three CFD datasets were provided by NTS [5]: one with clean inflow and two with inlet distortions (due to a T-junction and to a rectangular section upstream). Figure 7-11 shows a curve of power spectral density of the pressure on a point of the stator, which has been obtained by processing the CFD data in the way described above. The peak is clearly seen at the blade passing frequency (around 2800Hz in the CFD computations), although it is relatively wide due to the Hanning window applied together with a somewhat big frequency step.

![Figure 7-11: Fan with clean inflow: PSD of the pressure on a point on the stator surface.](image)

Figure 7-12 shows results for the case with clean inflow, where the sources have only been defined at the stator, together with the compressible CFD results. The modal amplitudes have been extracted both upstream and downstream. The CFD pressure has been post-processed in order to extract the modal amplitudes (note that only a relatively rough post-processing of the CFD was applied here, and that small effects of reflections at the inlet/outlet for higher frequencies may be present).

It can be seen that the broadband levels are in very good agreement with the CFD input. This suggests that indeed the dipole sources on the stator vanes are sufficient to characterize the broadband noise produced by the fan. By contrast, the levels at the BPF (around 2800Hz) are significantly higher in the current computations. Since the acoustic mode at the BPF should be cut off, this indicates that there are some inaccuracies in the acoustic results preventing the cancellation of the acoustic peak. It must be pointed out that the acoustic mesh and the CFD mesh were not exactly coincident and may have generated some inaccuracies in the mapping of the flow pressure from the CFD mesh to the acoustic mesh. This numerical effect may be worsened by the close proximity of the BPF to the cut-on frequency of the first radial mode. In any case, the agreement of the broadband levels is quite satisfactory and confirms the validity of the approach to predict broadband fan noise. It is worth noting that the approach is also applicable if the CFD is incompressible and thus unable to provide acoustic predictions.

### 7.6 Computations of the passive and active contributions of the valve

This section shows the results obtained by SISW for the acoustic computations of the valve case (with and without aeroacoustic sources). Figure 7-13 displays a view of the finite element mesh. It has a total of around 194 000 tetrahedral elements. A detail of the mesh
inside the duct in the vicinity of the valve can be seen in Figure 7-14, including the discretization of the surface of the valve.

Figure 7-12: Fan with clean inflow: Modal amplitudes for the modes \((m,n)=(0,0), (1,0), (-1,0),\) and \((0,1)\), upstream (dashed line) and downstream (solid line). The current results with sources only on the stator (black lines) are displayed together with CFD pressure results (red lines).

Figure 7-13: View of the finite element mesh from the exterior of the duct.
Figure 7-14: Detail of the interior of the duct close to the butterfly valve.

Figure 7-15 shows the computed amplitudes of the transmission and reflection coefficients for the cut-on modes below 3500Hz for the case without a mean flow. The values are overall in good agreement with the experimental measurements obtained in task 2.3 of IdealVent [3].

![Figure 7-15: Computed amplitudes of the transmission coefficients (solid line) compared to the experimental values (dashed line) for the modes $(m,n)$=$(0,0)$, $(1,0)$ and $(-1,0)$, for waves traveling from upstream to downstream locations in the no flow case.](image)

Computations were also run with a mean potential flow. In this case, there were some significant deviations with respect to the experiments with flow. Note that these computations only include the effects of the mean flow as a convection term: other effects, such as the influence of shear flow in the propagation, are not taken into account. Indeed, a strong shear layer with a non-negligible Mach number jump is created behind the valve, and may facilitate the conversion of acoustic energy to aerodynamic energy, thus providing a mechanism for sound absorption. These effects may have some importance in the zone of the opening of the valve, where the local Mach number can become relatively high.

The high Mach number of this case also has consequences on the computation of the active part. Initially, computations with dipole sources based on wall pressure, implemented as...
equivalent boundary conditions of the Helmholtz equation, were foreseen. Nevertheless, this approach is applicable only for low Mach number flows. Since the Mach number at the valve opening is higher than initially foreseen (M>0.6), this was found not to be a suitable approach. Indeed, preliminary computations using dipole sources predicted acoustic similar levels upstream and downstream from the valve, in contrast to the experimental measurements performed by KUL [3].

![Figure 7-16: Computed valve sound generation using quadrupoles based in vortex sound theory: Modal amplitudes for the plane wave mode upstream and downstream.](image)

As an alternative, a linearized acoustic propagation equation in potential flow, with volume-distributed sources based on vortex sound theory, was solved. The CFD dataset was provided by VKI [7]. Figure 7-16 shows results of the active part of a multiport model (see D3.5). Both results based on the present computations with volume-distributed sources, as
well as experimental results obtained by KUL are displayed. Unfortunately, the relatively short CFD time series that was available (1000 time steps exported every $1e^{-5}s$, leading to a frequency resolution of 100Hz and only one time window) leads to a relatively coarse resolution and to spurious peaks which could be smoothed out by introducing some averaging. Still, these initial results show some promise. The computational results show a difference between upstream and downstream acoustic levels that is similar to that of experiments, and the overall behavior seems to follow the same trends. This suggests that the selected approach is suitable for computing flow-generated sound for moderate Mach numbers. Additional computations with a longer CFD time series are planned to improve the accuracy of the results.

7.7 Summary and Conclusions
In the framework of IdealVent WP4, SISW has performed finite element acoustic computations of both active and passive acoustic behavior for the valve with upstream bend case, fan cases with inflow distortions and single and tandem diaphragm cases. The computations of the active contribution are based on equivalent sources, either surface-distributed dipoles implemented as equivalent boundary conditions, or volume-distributed sources based on flow velocities. In general, the obtained results are in good agreement with experimental data, thus illustrating the suitability of the investigated methods to compute the sound generated by ECS flows. The applied approach has the advantage that it can provide accurate acoustic predictions based on a low-order description of the flow field that does not contain acoustic information.

7.8 Bibliography
8 Developments and achievements made by ECL

8.1 Context and modelling strategy

Predictions performed in the first part of the project using Amiet’s isolated-airfoil theory and realistic, though not accurate, input data for the turbulence indicated that the dominant source of broadband noise is the impingement of the rotor wakes on the stator vanes (deliverable D2.4). Rotor trailing-edge noise is presumably the secondary contributor whereas stator trailing-edge noise is even weaker because of the smaller relative velocity. In this first-step assessment the duct was not considered and only an indicative free-field power was calculated. Sound levels produced using such an approach are not accurate, especially dealing with the spectral shape. Yet they can be referred to for the ranking of various mechanisms of broadband noise. Therefore the rotor-stator wake-impingement noise was selected as the mechanism to be addressed in the project with advanced analytical modelling techniques. This means that the focus is on the outlet guide vanes as the dominant source of sound.

In essence, Amiet’s theory proceeds in two steps. In the first step the fluctuating loads on blades/vanes are determined from oncoming velocity disturbances described by their statistical properties (model turbulence spectra). These loads are the sources of the generated sound as stated by the acoustic analogy and radiate as equivalent dipoles. In the second step the sound transmission through the duct is calculated from the dipole distributions using the Green’s function of the duct, assumed hard-walled and perfectly annular. Modal acoustic powers for downstream and upstream transmissions are finally obtained. The main underlying assumptions are that blades and/or vanes are assimilated to thin rigid flat plates and that only isolated-airfoil response functions are considered. It has been recognized that this framework suffers from limitations despite its formal simplicity. Indeed the ECS fan investigated in the project is characterized by a stator of high solidity (chord length to vane-to-vane spacing ratio). As long as stator noise is considered a significant cascade effect is expected. The latter is defined as the effect of adjacent vanes on the aero-acoustic response of a single vane to incident disturbances. For a significant overlap between adjacent vanes, the induced unsteady loads on a vane as predicted using a cascade response substantially differ from those produced by an isolated-airfoil response (H. Posson, M. Roger & S. Moreau (2010) - On a uniformly valid analytical rectilinear cascade response function, J. Fluid mech. Vol. 663, pp. 22-52). Of course the former are more reliable than the latter.

This is why the intensive use of Amiet’s theory and its extensions initially planned in Task3 and Task4 was abandoned at the benefit of an alternative approach in which cascade effects are included. The reason for this re-definition of the working plan is that the design parameters of blades/vanes (twist, lean…) that could be accounted for in extended Amiet’s theory are assumed less important than the aforementioned cascade effect, especially in the present case of the ECS fan. The reorientation of the modelling effort has also been motivated by dimensional analysis and consideration of previous/other projects in which ECL was participating. Finally, choosing another analytical approach was not prejudicial to any contribution from other partners in the project.

To account for the cascade effect, ECL developed a new model based on a mode-matching technique. The underlying idea is that the domain of interest, namely the volume including the rotor, the stator and their vicinity, can be split into sub-domains in which the acoustic field is expressed as a sum of normal modes; the modal expressions are matched at the interfaces between the sub-domains according to conservations laws of physical quantities (mass-flow rate and enthalpy for a row of stationary vanes). This matching generates infinite matrix equations that are truncated and solved by...
standard inversion to determine the coefficients of all reflected and transmitted modes. When this is applied to the outlet guide vanes, the sub-domains are the azimuthally-unbounded spaces upstream and downstream of the stator and the series of inter-vane channels, considered as a periodic arrays of bifurcated waveguides. For convenience the approach is therefore referred to as the MMBW technique (for Mode-Matching in Bifurcated Waveguides).

The mathematical background of the MMBW technique for axial-flow fan architectures is inherited from the theory of electromagnetism and optical gratings (R. Mittra & S.W. Lee (1971) - Analytical techniques in the theory of guided waves, MacMillan). It has been developed by ECL in a national project (SEMAFOR) addressing the tonal rotor-stator interaction noise. In IDEALVENT it has been extended to include a Kutta condition and used for broadband-noise predictions in two-dimensional and three-dimensional versions of the stator, which represented a substantial mathematical effort. The two-dimensional implementation performed first was a necessary step for the assessment of the mode-matching procedure. The main limitation at that stage is that the vanes are assimilated to flat plates of zero stagger angle (the vanes are assumed parallel to the fan axis). This is not realistic in view of the true design of the outlet guide vanes but is a necessary condition for mathematical tractability. Setting the stagger angle to a non-zero value with flat-plate vanes is possible (M. Roger & B. François (2016) - Combined analytical models for sound generation and transmission in cambered axial-flow outlet guide vanes, ISROMAC16) but not more realistic; in fact this would correspond to an identical swirling flow upstream and downstream of the vanes. For this reason, the method is presently extended by ECL to curved vanes (bent inter-vane channels) at the price of additional approximations, apart from the IDEALVENT program.

Apart from the account for the cascade effect, the interest of the MMBW technique is that is it only one-step, directly producing the waves emitted from the stator without resorting to the unsteady loads on the vanes. Yet the background is the same as for Amiet’s technique, namely a linearized, inviscid approach as shortly discussed in the next section. The technique can be adapted in any space for which the Helmholtz equation is separable and the modal functions explicitly known in all involved sub-domains. This holds in particular in an annular hard-walled duct, making the technique very attractive for turbomachinery noise investigations. Furthermore the excitation of the waveguide system in the MMBW technique can be of either acoustic or vortical/hydrodynamic nature. This means that both sound-generation and sound-transmission mechanisms can be addressed in a unified approach. It is worth noting that in a rotor-stator arrangement part of the noise generated by the rotor (stator) is transmitted through the stator (rotor), leading to reflection/transmission phenomena. As many of these physical features as possible must be reproduced in analytical modelling strategies for consistency, especially if the predictions must be used for optimization purposes and not only for order-of-magnitude estimates. This very general capability is another advantage of the technique and a reason why it has been preferred to classical Amiet’s theory.

### 8.2 Theoretical Background

The main assumptions and theoretical background are shortly reminded in this section. Linearized, inviscid gas dynamics equations for constant entropy and mean-flow variables are considered, following Chu & Kovácsznay’s analysis. In these conditions two kinds of fluctuating motion are pointed, the radiating acoustic motion (compressible, rotational-free) and the convected vortical motion (pressure-free, incompressible); though the latter is driven by viscosity, viscosity will be ignored in the mathematical formulation and artificially reintroduced if needed. Both motions remain uncoupled, except at rigid walls where their
combined normal velocity vanishes. The velocity field can be split into a rotational, incompressible part and a potential part, as

\[ \mathbf{v} = \mathbf{v}_t + \mathbf{v}_a; \quad \mathbf{v}_t = \nabla \times \mathbf{A}; \quad \mathbf{v}_a = \nabla \phi \]

so that the rotational part is associated with the vortical motion and the potential part with the acoustic motion. In absence of solid boundaries the vortical field is frozen: \( \frac{d}{dt} (\nabla \times \mathbf{v}_t) = 0 \). As prescribed vorticity impinges on an isolated, thin rigid plate, only a potential motion can be generated as a response because of the inviscid assumption (no boundary layer is considered). The radiating part of this potential motion is precisely the generated sound. But because viscosity cannot be ignored at the trailing edge, it is re-introduced as a vortex sheet in the wake via the Kutta condition. The acoustic normal velocity must cancel that of the oncoming vortical motion on the plate and a zero pressure jump is imposed at the trailing-edge and in the wake. These boundary conditions are used explicitly as such in Amiet’s theory.

![Figure 8-1: Schematics of airfoil response (upper plot) and cascade response (lower plot) to incident vortical patterns, for analytical modelling](image)

If now a series of zero-stagger plates is considered (Figure 8-1), the same principles apply but the potential field has to satisfy the rigidity condition on all plates. Furthermore a phase-shift must be maintained between the contributions of adjacent plates according to the excitation. This makes the mathematical problem more complicated. Posson *et al.* (2010) typically solve it with an advanced use of the Wiener-Hopf technique. The MMBW approach is mathematically equivalent (same boundary conditions for the convected Helmholtz equation) but conceptually different, and in fact simpler. Instead of being considered at the walls, the
matching of the vortical and acoustic motions is displaced at the interfaces. First assume that there is no wake (no Kutta condition). The incident vortical field is again considered frozen and extending everywhere irrespective of the plates, and the acoustic field must be deduced from the continuity of the total field at the interfaces. Of course both motions have to satisfy the rigidity conditions on the plates but this is ensured by giving them a convenient expression inside the inter-vane channels, *a priori*. The acoustic pressure is expressed as a sum of cosine modes, as well as the axial component of the vortical velocity, and other variables are deduced from the physical conditions of both motions (incompressibility for the vortical field…). The coefficients of all channel modes are unknown. Those of the vortical field are imposed in such a way that they ensure the frozen continuation of the incident vorticity. Those of the generated acoustic waves are determined from the required continuity of the total field at the interfaces according to general laws of gas dynamics. In the present case, the continuity of the pressure and of the axial velocity is imposed. An infinite system of linear equations on the modal coefficients is obtained. After truncation, the system is solved by classical matrix inversion. This general scheme can be adapted with the same principles if the incident vortical wave is replaced by an acoustic wave to formulate a sound-transmission problem. In this case no frozen vortical field is involved.

The aforementioned procedure is incomplete because free of any Kutta condition. When this condition is added, the pressure difference is forced to zero between both sides of any plate of the cascade just upstream of the trailing edge. The resulting constraint on the (purely acoustic) pressure of the channel waves makes the system over-determined. But an additional vortical field is also added in the wakes, so that the problem is again well-posed and solvable by matrix inversion. All details of the derivations are given in the deliverable 4.2 and in previous reports.

### 8.3 Validation and 2D Modelling

The implementation of the MMBW approach has first been assessed in a two-dimensional context (see Figure 8-2 for the problem reduction) and validated by comparison with alternative mathematical formulations, typically the Wiener-Hopf technique and numerical simulations (*C. Durand & R. Hixon* (2015) - *Comparison of Computational Aeroacoustics Prediction of Vortical Gust Scattering by a 2D stator with Flat Plate Theory, AIAA paper 2015-2842*). This was achieved both in the framework of the IDEALVENT project and during the French program SEMAFOR, by addressing the two following generic problems:

- transmission of an oblique acoustic wave (equivalent to a duct mode in an unwrapped configuration);
- generation of acoustic waves due to a single impinging oblique vortical gust.

When addressing pure sound transmission, the predicted acoustic fields are in complete agreement with those of alternative methods provided that the Kutta condition is implemented, but de-activating this condition leads to significant errors (increasing with increasing Mach number). In absence of Kutta condition, the energy balance for acoustic waves is very accurate. With the Kutta condition activated, part of the energy of the acoustic motion is lost because of conversion into vortical motion. When addressing sound generation
by impinging vorticity, again a full agreement is found between all approaches/methods if the Kutta condition is implemented. This set of preliminary results confirmed that the MMBW approach is robust and reliable. In fact the plate edges are singular points not prejudicial to the results, in the sense that the fields of interest are continuous away from an arbitrary small region around these points (smaller for larger mode numbers in the solving procedure). The convergence of the alternate iterative procedure is generally ensured with about 4 iterations.

![Figure 8-2: True stator architecture (left) and two-dimensional unwrapped representation of a cylindrical cut, in Cartesian coordinates (right). Zero-stagger approximation](image)

The only limitation of the approach is that the axial plane-wave mode cannot be generated in cases where it should be produced in the true fan stage. Indeed the equivalent sources of the scattered field are lift dipoles oriented normal to the axis, because of the zero-stagger assumption. In a realistic stator architecture, this assumption is acceptable from the downstream standpoint but not from the upstream standpoint.

![Figure 8-3: Validation test case of the CAA Workshop Cat.3. Present prediction with the MMBW technique (top) and Durand & Hixon’s (2015) numerical simulation (bottom). Instantaneous pressure patterns](image)
Keeping all these properties in mind, predictions of the broadband wake-interaction noise have been performed first with the two-dimensional model. The set of wake data needed as input was tuned on the dissipation and turbulent kinetic energy extracted from RANS simulations performed by DLR. The tuning assumed locally isotropic and homogeneous turbulent fields and provided values for the integral length scale and turbulent intensity. The actual duct was split into 10 annuli and the model applied with a strip-theory approach to predict acoustic-power spectra both upstream and downstream. A strong overestimate has been found when comparing with the measured power spectra. Though still to be fully understood, this discrepancy is attributed to the difficulty of correctly describing the spanwise correlation of the random loads. Furthermore some approximations must be accepted in the description of the turbulence. Typically in the two-dimensional model the vorticity vector is normal to the unwrapped plane and the radial velocity component is assumed zero. This is needed to exploit the condition of zero divergence in the description of the vortical field inside the inter-vane channels.

8.4 3D Modelling

The simplified stator configuration used for the three-dimensional derivations is shown in Figure 8-4 together with the main notations. A hydrodynamic excitation (three-dimensional gust, see deliverable 4.2) of velocity \(v_i^h\) is prescribed and the scattered potentials are written \(\phi_{R,T,U,D}\). A single channel (in red) is enough to solve the matching equations since all channels have the same response with phase shifts imposed by the obliqueness of the incident field. At the present stage the zero-stagger assumption is the only unrealistic geometrical feature, at least as viewed from an upstream observer, when compared to a true architecture. The attractiveness of the method is that the annular configuration is treated as such without other approximation, directly compatible with the main duct. This would not be the case with alternative methods based on the strip-theory approach.

![Figure 8-4: Sub-domains and reference frames for the three-dimensional statement of the MMBW technique, zero-stagger outlet guide vanes in an annular duct. Prescribed hydrodynamic excitation and acoustic response (no Kutta condition)](image-url)
However the three-dimensional implementation of the MMBW technique faced a problem inherent to it, namely the description of the incident turbulent field. Indeed the zero-divergence equation in three dimensions is not enough to explicitly relate a velocity component to the other two. Consistency issues with isotropic and homogeneous turbulence were also identified, as discussed in deliverable D4.2. This difficulty is not encountered with some alternative approaches such as Amiet’s technique for which the only required quantity is the local fluctuating velocity normal to the vanes; it will require further attention in future works. Further assumptions have been accepted, also addressed in deliverable 4.2.

Despite this open question, acoustic-power spectra predicted with the three-dimensional model are compared to measured spectra in Figure 8-5. The analytical predictions perform as well as some of the alternative/numerical strategies developed by other partners in the project; in particular the sound level at low frequencies is well captured. Yet the predicted spectral shape differs significantly from the measured one in that it underestimates the noise level in the high-frequency range. Though not fully explained, this discrepancy could be attributed to the fact that only one mechanism is considered in the analytical predictions, namely the rotor-stator wake-interaction noise, whereas other broadband noise mechanisms are involved in the experiment (trailing-edge noise, separation noise, tip noise). The aforementioned issues related to turbulence modelling also possibly cause discrepancies.

Figure 8-5: Predicted upstream and downstream broadband noise spectra (acoustic power levels) compared to measured spectra. From deliverable 4.2

8.5 Concluding Remarks

The new analytical model developed and implemented based on the MMBW technique (Mode-Matching in Bifurcated Waveguides) provides a unified approach for sound-generation and sound-transmission problems. It is in essence versatile and compatible with a three-dimensional annular geometry. Therefore it is a fast-running, attractive tool for repeated
parametric calculations in axial-flow fan architectures. The MMBW technique can be used to formulate both tonal and broadband noise mechanisms, but it has been applied to predict rotor-stator wake-interaction broadband noise in task 4. In the ECS fan test case the achieved predictions have been found in a good agreement with the measurements at low frequencies but the model substantially underestimates the sound at high frequencies. This can be explained either by the fact that sources that are not included in the model dominate at high frequencies, on the one hand, or by some remaining issues in the way of describing the turbulence responsible for the sound generation, on the other hand. For this reason, the proposed approach still needs to be assessed.
9 Developments and achievements made by LTS

9.1 Introduction

LTS has worked on different subjects to improve the internal existing tools:

- Valve noise prediction tools;
- Fan noise prediction tools;
- Independent tools transfer to a system tool.

Moreover, as KUL labs could not provide enough pressurized air to run high operating points for the valve, LTS made a similar test with higher operating points and compared it to KUL test and numerical calculations.

All these points are outlined in this report.

9.2 Valve test

9.2.1 Test description

The aim of this test was to characterize the noise coming from an Air Temperature Valve (ATV) according to upstream static pressure and valve position. In this context, the effects on acoustic of a bent-pipe upstream the ATV valve is also investigated.

Tests have been performed on the ATV valve P/N 1394A01, same that the one used in KUL laboratory. This test was performed in the anechoic chamber at Liebherr Aerospace facility in Toulouse.

19 Microphones installed on hemispheric surface were used to measure the sound pressure level and the acoustic power level to ISO standard 3745. The microphone number 20 was not used because it was in the airflow, and thus the acoustic power has been computed with the 19 microphones. The picture in Figure 9-1 shows the microphone setup that was used during the test.

![Microphone setup inside the anechoic chamber](image)

Figure 9-1: Microphone setup inside the anechoic chamber
The two following configurations have been investigated:

- Setup 1: baseline with a “straight” duct upstream the valve;
- Setup 2: configuration with a bent-duct upstream the valve.

Only the duct upstream the valve ATV is changed between test configurations.
Different valve positions and static pressure conditions upstream the ATV ($P_{amont}$) have been tested. The effect of these two parameters has been investigated independently:

- points in the conf.A1 to A5 and A7 (baseline) & conf.B1 to B7 (right-angle setup) are made at fixed valve angle with variation in the pressure ratio $R$ between upstream and downstream pressures;
- points in the conf.A6 (baseline) & B8 (right-angle setup) are made at identical pressure conditions ($R \approx 2$) with variation in the valve position.

The points listed in the following table (Table 9-1) have been performed for the baseline (Setup 1) and the setup with the right-angle upstream the valve (Setup 2).

**Remarks:**

- the position of the valve $\alpha = 90^\circ$ corresponds to the full-opening of the valve;
- the number indicated in the column with the name of the configuration corresponds to the moment the test has been carried-out. It does not correspond to specific operating conditions, i.e. points performed in conf.A1 do not necessarily correspond to the points in conf.B1;
- the accuracy of the position of the ATV is not well known, so the angles given in Table 9-1 only give an indication on the position of the valve. In this report, only the points with identical pressure ratios ($R$) and flow rates ($Q_{kimo}$) have been compared.
Table 9-1: List of the configurations performed with the baseline and the right-angle

<table>
<thead>
<tr>
<th>Angle ATV (°)</th>
<th>Pamb (barg)</th>
<th>Nb PT</th>
<th>Conf. name</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>0.99 - 3.28</td>
<td>12 pts</td>
<td>A4</td>
</tr>
<tr>
<td>30</td>
<td>1.10 - 3.29</td>
<td>13 pts</td>
<td>A2</td>
</tr>
<tr>
<td>40</td>
<td>0.99 - 3.29</td>
<td>12 pts</td>
<td>A3</td>
</tr>
<tr>
<td>43</td>
<td>1.10 - 3.28</td>
<td>12 pts</td>
<td>A7</td>
</tr>
<tr>
<td>70</td>
<td>0.99 - 3.33</td>
<td>14 pts</td>
<td>A5</td>
</tr>
<tr>
<td>90</td>
<td>1.10 - 1.95</td>
<td>6 pts</td>
<td>A1</td>
</tr>
<tr>
<td>Iso-pressure</td>
<td>90 - 20</td>
<td>8 pts</td>
<td>A6</td>
</tr>
</tbody>
</table>

9.2.2 Experimentally observed characteristics for the noise coming from the valve

Results given in this section come from the baseline (configurations A1 to A7). Effect of the bent-duct upstream of the valve (configurations B1 to B8) is discussed in §9.2.2.3.

9.2.2.1 Effect of variation of the valve position

Acoustic power level (SWL) spectra are presented for a selection of points in the following.

SWL for the baseline is given in Figure 9-5 for different valve positions, when the ratio between upstream static pressure and downstream static pressure is kept almost constant ($R \sim 2$). It also corresponds to a supersonic jet at the valve ($M_j \sim 1.0$).
Figure 9-5: Variation of SWL with the valve position for the baseline

N.b.: fluctuations of hydrodynamic nature are indicated by the grey area.

For all valve positions, a broadband noise in the mid-frequency region (above approximately 1500 Hz) is observed on the acoustic spectra. SWL starts to show a significantly different scaling with the valve angle in comparison to the small valve angles ($\alpha \leq 50^\circ$). These variations tend to show that the noise generation changes with the opening of the valve:

- **at small valve angle** ($\alpha \leq 50^\circ$), SWL in the mid-frequency region increases with increasing opening angle;
- **at larger valve angle** ($\alpha > 50^\circ$), SWL does not significantly change with the valve position and thus it can be assumed that the aerodynamic noise generation is not only being caused by the valve itself.

Regarding the low frequency domain, one can observe high SWL at low frequencies at operating points 3 to 7 (i.e. at large valve angles). This phenomenon is characteristic of aerodynamic pressure fluctuations caused by the flow leaving the downstream duct. This trend is only observed in the case of large valve angles (up to the point 7). At these points, the flow rate is highly increased in order to keep an identical pressure drop over the ATV, what causes the pseudo-sounds observed in Figure 9-5.

Low frequency sound pressure level (SPL) variations are given in Figure 9-6 for a microphone that is not perturbed by the flow leaving downstream duct (i.e., the microphone 7).
At this position, hydrodynamic fluctuations in the low frequency domain are no more observed. Moreover, bumps are clearly noticeable at frequencies below the cut-off frequency of the high-order duct modes, which is given for this system by $f_{\text{max}} = 1.84c_0/\pi D \sim 2300$ Hz, where D is the inner diameter of the duct and $c_0 = 346$ m/s is the sound velocity in air at $T \sim 25{}^\circ$C. These frequencies are more pronounced at smaller valve angles (i.e., $\alpha < 50^\circ$).

Such phenomenon is assumed to be generated by longitudinal duct modes and thus should be related to duct length downstream the ATV. Note that at large ($\alpha \geq 50^\circ$) angles, the presence of the valve does not produce significant acoustic reflections and thus resonance phenomena generated by the valve is very limited. For this reason, bumps in the low frequency domain are less pronounced in comparison to the small angle valve positions. Indeed, it can be observed in Figure 9-7 that acoustic spectra in the case $\alpha = 70^\circ$ (points 1/2) are not dominated by installation effects. It can be assumed that, due to the large length of the duct (from injector nozzle to downstream duct outlet), duct resonance do still occur but at very low frequencies.

Duct lengths allow one to calculate the first theoretical longitudinal mode frequencies. Experiment is successfully compared in Table 9-2 to the predictions for a duct with a length $L = 1.16$ m, which is open on one side and close on the other. This length corresponds to the pipe with a constant section between the ATV valve outlet and the exhaust.
Table 9-2: Theoretical & experimental frequencies for the first duct modes

<table>
<thead>
<tr>
<th>Mode n</th>
<th>Predictions (Hz)</th>
<th>Experiment (Hz)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>74</td>
<td>75</td>
</tr>
<tr>
<td>1</td>
<td>223</td>
<td>230</td>
</tr>
<tr>
<td>2</td>
<td>372</td>
<td>380</td>
</tr>
<tr>
<td>3</td>
<td>521</td>
<td>530</td>
</tr>
<tr>
<td>4</td>
<td>670</td>
<td>688</td>
</tr>
<tr>
<td>5</td>
<td>818</td>
<td>844</td>
</tr>
<tr>
<td>6</td>
<td>967</td>
<td>988</td>
</tr>
<tr>
<td>7</td>
<td>1116</td>
<td>1156</td>
</tr>
</tbody>
</table>

9.2.2.2 Effect of variation of the pressure ratio

Variation of SWL with the pressure ratio $R$ at identical valve positions is given hereafter. Cases of $\alpha \geq 50^\circ$ and $\alpha < 50^\circ$ are shown in Figure 9-7 and Figure 9-8, respectively.

Figure 9-7: Variation of SWL with the pressure ratio at large valve angle: $\alpha=90^\circ$ (on the left side) & $\alpha=70^\circ$ (right side)

Figure 9-8: variation of SWL with the pressure ratio at small valve angle: $\alpha=20^\circ$ (on the left side) & $\alpha=30^\circ$ (right side)
As previously discussed, SWL spectra in the case of large valve angles (Figure 9-7) seem to be dominated by very energetic aerodynamic fluctuations (“pseudo-sound”) at low frequencies. This trend even extends to the mid-frequency domain (up to approximately 4 kHz) at high pressure ratios/flow rates. At these positions, spectra are only given for $f \geq 400$ Hz.

It can also be noticed that, in the case of the Figure 9-7 ($\alpha \geq 50^\circ$), SWL spectra show the emergence of a broadband noise at high frequencies with increased pressure ratio. This phenomenon might also be related to the high flow rates with $\alpha \geq 50^\circ$. It could be explained by the fact that the aerodynamic noise generation caused by the jet leaving the downstream end of the duct (exhaust) can become predominant over the noise coming from the valve.

A decay of the SWL with increasing frequency is observed in the mid-frequency region in the case of $\alpha < 50^\circ$ (Figure 9-8). Besides this overall decay of the SWL, the shape of the acoustic spectrum does not change significantly with the velocity, which indicates that the aerodynamic noise generation mechanism do not significantly change when incoming flow velocity is increased.

### 9.2.2.3 Effect of the bent-duct upstream of the valve

Acoustic results in the configurations with the bent-duct upstream of the valve are discussed in the following. For this purpose, experimental OASWL and SWL spectra are compared to the baseline in order to highlight the effect the right-angle on the noise coming from the valve.

The operating points performed in the baseline and the right-angle configurations are compared in the following figure, where both the massflow rate ($Q_{kimo}$) and the static pressure ratio ($R$) are displayed. The points performed are very similar for the both configurations. Results in the case $\alpha = 70^\circ$ are discussed later in the following subsections.

![Figure 9-9: Performance comparisons between the baseline and the configuration with the bent-duct upstream the ATV.](image)

**Remark:** points highlighted in the figures above are compared in the following.
Figure 9-10 shows the variations of the global noise power level (OASWL) versus the pressure ratio in both the right-angle and the baseline configurations. Results are given at three different valve positions.

**Remark:** global acoustic power levels are calculated by averaging the acoustic power in third octave frequency bands between $125\, Hz$ and $16\, kHz$.

![Figure 9-10: Variation of OASWL with the pressure ratio: comparison between the baseline and the right-angle configurations](image)

For most valve positions and incoming flow conditions tested, comparisons with the baseline show that the right-angle does not significantly change the overall noise power level. The only exception is the case with $\alpha = 20^\circ$. At this valve position, differentia in OASWL is approximately $4\, dB$ for identical pressure ratios and mass flow rates (cf. Figure 9-10).

Variations of PSD with the pressure ratio and the position of the valve are presented in the following figures. Results in the baseline configuration are also included in order to highlight the effect of the right-angle on the PSD. Only the points with similar pressure ratios and mass flow rates indicated in Figure 9-9 have been compared. Differentia between the baseline and the configuration with the bent-duct upstream of the valve seem to depend on the position of the valve. Different trends have been observed for cases $\alpha > 20^\circ$ and $\alpha = 20^\circ$.

Acoustic spectra presented in Figure 9-11 correspond to valve positions with $\alpha > 20^\circ$. 

Overall, the right-angle seems to have only a small influence on the shape of the acoustic spectra with $\alpha > 20^\circ$. This is motivated by the fact that:

- At the exception of the case $\alpha = 70^\circ$, PSD are very similar in both configurations. Differentia is barely visible and should be related to operating conditions. Especially, the pressure ratios and flow rates are not exactly identical for the points that are compared in Figure 9-9.

- At the position of the valve $\alpha = 70^\circ$, differentia in PSD could also be related to different in operating conditions. For similar pressure ratios, the massflow rate is about 50 to 100 $g/s$ higher in the case of the bent-duct configuration, what corresponds to approximately $(4 - 5)\%$ increase between the two configurations (see Figure 9-9). Such an increase could be explained by differences in the position of the valve.

- In the high frequency range (above approximately $5kHz$), the right-angle yields a small variation in noise level, with differentia of more or less $1 - 2dB$ depending on the point considered; this effect is more pronounced at low pressure ratios (e.g., see the cases $\alpha = 43^\circ$). At these points, unexplained peaks have also been observed between $6kHz$ and $8kHz$. These have only been observed in the case of the right-angle.
Remark: unexplained peaks in the high frequency range are also observed above $10kHz$ in the case $\alpha \sim 20^\circ$ (in Figure 9-12). However, such frequencies are not observed at valve positions $30^\circ/70^\circ/90^\circ$.

The case of the smaller valve angle performed (i.e., $\alpha \sim 20^\circ$) is presented in Figure 9-12.

![Figure 9-12: PSD comparisons between the baseline and the right-angle in the case $\alpha \sim 20^\circ$](image)

Comparisons at this valve position give slightly different results:

- At low frequencies, the noise level is very similar for both configurations. In this frequency range, the acoustic spectra are dominated by duct modes, as previously discussed in §9.2.2.1.

- At high frequencies (above approximately $2000Hz$), however, PSD in the case of the right-angle are noticeably higher at identical incoming pressure and mass flow rates. This emergence could be related to the incoming flow characteristics, which are changed in the case of the right-angle. Indeed, the flow rate is significantly reduced at the smallest valve angle, so it can be assumed that in this case this emergence is not being hidden in the noise generated by the flow through the valve.

- Here again, unexplained peaks have also been observed above $10kHz$, in the case of the right-angle.

### 9.2.3 Comparison with experiments and simulations made by KUL and EXA

Experiments and numerical simulations have been carried-out by KUL within the scope of IDEALVENT project and by Exa GmbH (subcontracting by LTS), respectively, on the noise
coming from the ATV valve. These studies are briefly described in the following and the link with the test reported in this document is discussed.

9.2.3.1 Measurements by KUL

Acoustic investigations have been carried out by KUL on the noise coming from a butterfly valve. This test has been performed in a semi-anechoic room at KUL (Belgium). The setup is close to the baseline configuration.

This test consisted in directivity measurements on the noise leaving the downstream duct. For this purpose, six microphones have been installed on a radius of 1.5m from the center of the downstream duct. Note that the valve is located inside the semi-anechoic chamber. Only the blower, which is used to create a uniform flow through the valve, is located outside of the semi-anechoic chamber. Various valve positions and pressure conditions upstream and downstream of the ATV have been performed. Examples of acoustic spectra in the cases of $\alpha = 20^\circ$ and $60^\circ$ are presented in Figure 9-13.

![Figure 9-13: Measurement by KUL: sound pressure level for the six different microphones: 60° valve position (top) and 20° valve position (bottom)](image)

Note that the frequency range in the measurements made by KUL is restricted to frequencies below $6250\,Hz$. Below the cut-off frequency of the high-order duct modes (i.e. below $2400\,Hz$ for this system), one can clearly notice that the shape of the acoustic spectrum is much more peaked in the case of a small valve angle ($20^\circ$ in Figure 9-13). These bumps,
which are associated to duct resonances, have also been observed in the acoustic measurements at LTS, as shown in Figure 9-14.

Figure 9-14: example of measurements made by LTS - SPL spectra at two valve positions ($\alpha = 20^\circ/60^\circ$) and four different microphone positions (Mic.1/90°, Mic.8/68°, Mic.15/46° & Mic.19/22°)

Note that operating conditions in the case of the point 6 in conf.A6 differ from the case at the 60° valve position in Figure 9-13. Points with similar operating conditions (angles & massflow rates) are given in Figure 9-15 (KUL) and Figure 9-16 (LTS). Spectra for $Q = 403 m^3/h$ and $411 m^3/h$ in KUL experiments correspond to LTS test points 3 & 4 in the conf.A2, respectively. The position of the microphone (at 45° at KUL and 46° at LTS) is similar in both setups.
9.2.3.2 Numerical simulations performed by ExA

Numerical simulations have been performed by Exa GmbH on the aeroacoustic of a ducted butterfly valve in IDEALVENT project. Calculations use the CFD/CAA solver PowerFLOW developed by Exa Corporation. It is based on the Lattice-Boltzmann Method (LBM). In this simulation, the noise is propagated from the duct exhaust into an anechoic room to microphones located on a semi-sphere of 1m radius. The computational setup is similar to the configuration with a bent-duct upstream of the valve. Inputs for calculations are given in

---

Figure 9-15: Measurement by KUL: SPL for different incoming flow rates with valve position of $30^\circ$.

Figure 9-16: Measurements by LTS: SPL in the conf.A2 with points 3 ($\alpha = 30^\circ$, $Q = 400m^3/h$) and 4 ($\alpha = 30^\circ$, $Q = 413m^3/h$).
Table 9-3. It corresponds to a subsonic point at the valve position that is **similar to the point 1 in the conf.B1**, with $R = 1.29$ and $Q = 0.13 \, \text{kg/s}$.

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet total pressure</td>
<td>125000 Pa</td>
</tr>
<tr>
<td>Inlet total temperature</td>
<td>300 K</td>
</tr>
<tr>
<td>Inlet mass flow</td>
<td>0.12 kg/s</td>
</tr>
<tr>
<td>Room static pressure</td>
<td>100000 Pa</td>
</tr>
<tr>
<td>Room temperature</td>
<td>281.5 K</td>
</tr>
<tr>
<td>Inlet density</td>
<td>1.45005 kg/m³</td>
</tr>
<tr>
<td>Inlet dynamic viscosity</td>
<td>1.8457×10⁻⁶ Pa·s</td>
</tr>
<tr>
<td>Inlet velocity</td>
<td>9.64286 m/s</td>
</tr>
<tr>
<td>Inlet temperature</td>
<td>299.954 K</td>
</tr>
<tr>
<td>Inlet pressure</td>
<td>124932 Pa</td>
</tr>
<tr>
<td>Inlet speed of sound</td>
<td>347.196 m/s</td>
</tr>
</tbody>
</table>

**Table 9-3 : Input in the simulations made by EXA**

Overall, simulations over-estimate OASPL by at least $8\,\text{dB}$. This result is obtained when the whole frequency range is used, i.e. over the frequency range $[100\,\text{Hz} - 20\,\text{kHz}]$. A much better agreement with experiment is observed with the OASPL restricted to the frequency range $[1.5\,\text{kHz} - 20\,\text{kHz}]$: in this case, experiment and simulation give very similar results, with less than 1dB differentia. Simulations mainly differ from experiment at low frequencies, where the noise levels are dominated by the bumps associated with the duct modes.

One-third octave band SPL spectra are given in Figure 9-17 for the microphones 5, 10 & 15.

![Figure 9-17: One-third octave band SPL at three microphone locations - comparison between experiment (“LTS”) and numerical simulation (“EXA”)](image)

Only small variations between the microphone positions can be observed in both the simulation EXA and the experiment LTS. As previously observed, calculations largely over-estimates SPL in the low frequency domain, below approximately $1.5 \, \text{kHz}$. Otherwise, experiment and simulation are in quite good agreement, especially in the mid-frequency range (between approximately $1.5 \, \text{kHz}$ and $7 \, \text{kHz}$).
9.2.4 Comparisons with LTS valve noise prediction

A prediction tool has been developed by Liebherr Aerospace (Toulouse) for the noise coming from butterfly-like valves. It is based on norm series IEC 60534. Aerodynamic and acoustic measurements in the baseline configuration are compared in the following to the simulations performed.

Inputs for calculations correspond to the operating conditions performed. Acoustic pressure fluctuations have been calculated using the IEC standard. Also note that noise evaluation does not account for any reflection on the external or internal surfaces induced by the duct connections, mechanical vibrations or any instability that could occur.

Global acoustic power (OASWL) predictions are presented in Figure 9-18 for three different valve positions. The case of the full opening of the valve is not included here because of aerodynamic perturbations. This case is discussed in the following.

![Figure 9-18: experimental OASWL and predictions according to the pressure ratio (R)](image)

OASWL comparisons should be considered with care. Global noise levels seem in quite good agreement with test results at 20° and 70°, whereas spectra in Figure 9-19, Figure 9-20, and Figure 9-21 are in very poor agreement. This could be explained by the following observations:

- in the case of a very small valve angle ($\alpha = 20^\circ$), the acoustic spectrum is mainly dominated by installation effects (duct modes) at low and mid-frequencies. This phenomenon is not taken into account by the simulations and thus the noise level in this frequency range is largely underestimated;
at large valve angles ($\alpha = 70^\circ$ & $90^\circ$), pseudo-sound is observed in the low frequency domain. This frequency range is dominated by aerodynamic perturbations. Also, noise levels at high frequencies are largely under-estimated. At large angles, SWL in this frequency range are probably related to the noise generated by the jet leaving the downstream duct, rather than to the noise coming from the valve itself. This is especially true in the case of the points A5-PT6 (70°) and A1-PT6 (90°), where the flow leaving downstream duct is at a high mass flow rate ($Q = 1.47 \, kg/s$ and $2.31 \, kg/s$, respectively).

Figure 9-19: Variation of the one-third octave band SWL with the pressure ratio in the case $\alpha = 20^\circ$: Comparison between predictions (in purple lines) and experiments (red lines)

Figure 9-20: One-third octave band SWL at the position $\alpha = 70^\circ$: comparison between predictions (in purple lines) and experiment (red lines)
Figure 9-21: One-third octave band SWL at the full-opening of the valve ($\alpha = 90^\circ$): comparison between predictions (in purple lines) and experiment (red lines).

At the intermediate valve position $\alpha = 43^\circ$ (Figure 9-22), predictions and measurements are in much better agreement, especially in the frequency range $[1 \text{ kHz} - 10 \text{ kHz}]$, where differentia are less than 5 dB at most operating points. Moreover, the shape of the acoustic spectrum is well reproduced by the simulations.

Figure 9-22: Variation of the one-third octave band SWL with the pressure ratio in the case $\alpha=43^\circ$: comparison between predictions (in purple lines) and experiment (red lines).

Peaks in the low frequency range are less pronounced in this case than at $\alpha = 20^\circ$ and predictions are still in quite good agreement with experiments. Also note that at the point 10 pseudo-sounds can be observed below approximately 700 Hz.
Acoustic measurements (global noise level, acoustic spectra) have been compared in this section to predictions. Two different tools developed by LTS have been tested for evaluating the noise coming from butterfly-like valves. Both models have proved to be quite efficient for the prediction of the global noise level at specific operating conditions:

- **IEC predictions** should be considered with care with the present test setup:
  - differentia with experiment at small valve angle should be mainly related to installation effects (duct modes) at low and mid-frequencies;
  - differentia with experiment at large valve angles is probably related to aerodynamic perturbations in the low frequency range and to the noise generated by the jet leaving downstream duct;

- one-third octave band noise levels given by IEC are however in quite good agreement with the experiment at intermediate valve position ($\alpha = 43^\circ$). Especially, the shape of the acoustic spectrum is efficiently reproduced in the mid-frequency range at low and medium incoming pressures;

- regarding comparisons with PacJel predictions:
  - overall predictions are in quite good agreement with experiment ($\Delta < 5\, \text{dB}$) only with a high incoming pressure;
  - Also, spectra provided by PacJel seem to indicate that high frequency fluctuations in the case of high flow rates are mainly caused by the jet leaving downstream duct.

### 9.2.5 Conclusion

The noise coming from the valve ATV (P/N 1394A01) has been experimentally investigated. Various incoming flow rates, pressures and valve positions have been tested, as well as the effect of a bent-pipe located upstream of the valve.

Acoustic results in both the baseline and the configuration with the bent-duct have been presented and discussed. Results are then compared to experiments and numerical simulations, which have been performed by *KUL* and *Exa GmbH* within the scope of IDEALVENT project on the noise coming from butterfly valves.

Finally, a prediction tool developed by *Liebherr Aerospace* for the butterfly valve noise (Norme_IEC_60534) has been tested. Test/prediction comparisons have also been presented. It has been shown that valve noise simulations are able to successfully predict both the overall noise level and the acoustic spectrum at most points performed.

### 9.3 Diaphragm prediction tool

LTS has a diaphragm prediction tool (ORFEO) developed by KTH in 2006. This tool is not used in LTS and needed to be validated.

The first step was to compare results coming from ORFEO with data from literature (Numerical and Experimental Analysis of Sound Generated by an Orifice – R. Arina – AIAA-
This document is an IDEALVENT public report 2007-3404). All experimental data, inputs and results, are available in the paper. ORFEO runs on this simple test and the results are compared in Figure 9-23.

![Figure 9-23: Comparison of ORFEO results and test results](image)

Data are equivalents for comparisons.

We note that ORFEO over estimates the diaphragm noise measured in the reference paper below 2000 Hz. That can be explaining because ORFEO give sound power level just downstream a diaphragm. Measurements in this test has been taken 3 m at the exhaust of the duct as shown in Figure 9-24, it takes into account propagation outside and duct modes.

ORFEO gives good agreement between 2000 and 5000 Hz.

![Figure 9-24: Schematic of the experimentation for ORFEO validation](image)
ORFEO has also been tested on the IDEALVENT diaphragm and compared with VKI tests. In this case, ORFEO have good shape at low frequencies but under estimates of 20 dB the test made in VKI (see Figure 9-25). The IDEALVENT measurements are taken with flush mounted microphones and can record other noise sources (boundary layers, etc.).

![Figure 9-25: Comparison of ORFEO results and IDEALVENT test results](image)

9.4 Fan prediction tool

LTS don’t design “low speed” rotor/stator fan, as the IDEALVENT fan. They are buy to our partners. The specialization of LTS is “high speed” rotor alone fan. The LTS internal tools are developed to predict noise of these last one. It is important for LTS to be able to challenge their partners and have some acoustic tools to predict noise of “low speed” rotor/stator fan. During IDEALVENT project, LTS try to adapt existing tools to “low speed” fan noise.

In a first step, LTS implements a law of similar to predict, from one test on one fan, the noise generated by a similar fan in another operating point, or with another diameter. Predictions are one dimension but can give good first approximations.

\[ L_{W_2}(f_2) = L_{W_1}(f_1) + 40 \log \frac{N_2}{N_1} + 70 \log \frac{D_2}{D_1} + 10 \log \frac{BW_2}{BW_1} \]

\[ f_2 = f_1 \frac{N_2}{N_1} \]

where \( f \) is the 1/3 octave band frequency and \( BW \) the bandwidth center on the \( f \) frequency.
To validate this law, LTS compare the results of this model and experimental data from LTS measurements on a rotor alone fan measured at different operating points (rotational speed – Table 9-4).

The measured spectra of the PT1, corresponding to a rotational speed of 35011 rpm is considered to calculate the spectra of the other operating points that will be compared to the measurements.

<table>
<thead>
<tr>
<th>N_ACM (rpm)</th>
<th>Conf_1_PT1.csv</th>
<th>Conf_1_PT2.csv</th>
<th>Conf_1_PT3.csv</th>
<th>Conf_1_PT4.csv</th>
<th>Conf_1_PT5.csv</th>
</tr>
</thead>
<tbody>
<tr>
<td>35011</td>
<td>Conf_1_PT1.csv</td>
<td>Conf_1_PT2.csv</td>
<td>Conf_1_PT3.csv</td>
<td>Conf_1_PT4.csv</td>
<td>Conf_1_PT5.csv</td>
</tr>
</tbody>
</table>

Table 9-4 : Experimental operating points of the same fan

The agreement between the predicted spectra and the experimental ones is good. The discrepancies are less than 5 dB, as seen in Figure 9-26.
The second step was to use the internal FRACTAL tool, dedicated to prediction of overall fan sound power level to predict an initial fan spectrum.

FRACTAL calculates the mechanical power of the fan to give acoustic fan prediction.

Then FRACTAL is used for the IDEALVENT fan with its input data to have an overall prediction. A spectrum is created from these results and with the addition of expected blade passing frequencies.

In Figure 9-27, the comparison between the rebuilt spectrum and the experimental one is done. The first approximation prediction has a good agreement with the test.

![Figure 9-27: Comparison of the rebuilt spectrum with Idealvent experimental spectra (third octave spectra)](image)

### 9.5 Integration of LTS acoustic tools in DYMOLA (global system model)

One more work done in LTS during IDEALVENT WP4 is the implementation of some acoustic tools in a global system modelisation.

LTS works with DYMOLA. DYMOLA is a commercial modeling and simulation environment based on the open Modelica modeling language. LTS has developed a complete design toolset based on its own experience and also on LTS research and development activities. LTS modeling capabilities covers all components used to build up LTS systems.

The next step is to add noise sources and noise propagation in this tool: ORFEO and FRACTAL was migrated in DYMOLA.
ORFEO in DYMOLA

An acoustic component was created, taking into account the aerodynamic of the flow upstream and downstream the orifice.

The «AcousticOrifice» table is a configurable table which can be use in Excel, and it contains the orifice geometry. The acoustic component must be integrated in a more general component including the aerodynamic of the diaphragm, as shown in the following picture.
In an Excel file, the mass flow and the geometrical data are the only input needed.

<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>constantVFlow</td>
<td>diameter_in</td>
</tr>
<tr>
<td>0.42 m3/s</td>
<td>0.116 m</td>
</tr>
<tr>
<td>optionalComment</td>
<td>diameter_out</td>
</tr>
<tr>
<td></td>
<td>0.15 m</td>
</tr>
<tr>
<td></td>
<td>thickness</td>
</tr>
<tr>
<td></td>
<td>0.006 m</td>
</tr>
</tbody>
</table>

The aerodynamic part gives the good delta of pressure which is the input of the acoustic part. The results are similar between Orfeo initial version and Orfeo implemented in Dymola (Figure 9-28).

![Figure 9-28: Checking of the good implementation of Orfeo in a Dymola component on the test case of Idealvent diaphragm](image)

**FRACTAL (and extensions) in DYMOLA**

The FRACTAL tool and the law of similar which was explained in §9.4 were as well implemented in DYMOLA.
The configuration of the calculations are also made in an Excel file and need inlet fan temperature, rotational speed, diameter, mechanical power and number of blade.

<table>
<thead>
<tr>
<th>Part: fractal_energetique</th>
</tr>
</thead>
<tbody>
<tr>
<td>TTFO: 140 degC</td>
</tr>
<tr>
<td>N: 32000 1/min</td>
</tr>
<tr>
<td>D: 0.152 m</td>
</tr>
<tr>
<td>Wmech: 6000 W</td>
</tr>
<tr>
<td>Nblade: 5</td>
</tr>
</tbody>
</table>
10 Developments and achievements made by NTS

10.1 General overview of developed computational procedure and considered flows

Within WP4, NTS has contributed to an investigation of the effect of inlet distortions on ducted fan noise (Task 4.2) and to an analysis of the noise produced by single and tandem diaphragms installed in a circular duct (Task 4.3).

The approaches developed by NTS in the framework of WP3 for solving these two problems are essentially the same and, in general, consist in direct computation of the noise produced by the fan or diaphragm(s) based on a compressible scale-resolving simulation of turbulence with the use of a hybrid RANS-LES method of the DES-type, namely, the Improved Detached Eddy Simulation (IDDES [1]), which is currently well established for unsteady simulations in both aerodynamic and aeroacoustic applications including engine and airframe noise.

In the presence of “inflow” turbulent content, IDDES performs as Wall-Modeled LES (WMLES). In order to "inject" such turbulent content upstream of the noise-generating region (fan or diaphragm(s)), specially designed distributed volumetric source terms are introduced into the momentum- and turbulent kinetic energy transport equations within a confined (“source”) region. This approach, which was also proposed and validated in the framework of WP3 of IDEALVENT [2] and called Volume Synthetic Turbulence Generator (VSTG), is based on the earlier proposed efficient “surface” STG [3]. Thus, within the developed computational procedure the IDDES is used as effectively a zonal method which functions in the RANS mode upstream of the VSTG source area and in the WMLES mode downstream of this area. Similar to the STG [3], the VSTG parameters are set based on the RANS solution in the corresponding duct section and ensure that the “created” turbulence matches the RANS Reynolds stresses right downstream of the VSTG region. A major advantage of the VSTG over the STG [3] and other surface generators of synthetic turbulence is that with a sufficiently stretched source region, it does not create any noticeable spurious noise, which is of primary importance for aeroacoustic problems. In addition, the VSTG is more flexible in terms of grid-structure and, particularly, allows carrying out zonal RANS-WMLEs computations on a “single” grid, i.e., without using overlapping RANS and LES grid blocks required for the STG [3].

In principle, the outlined approach is capable of predicting both the tonal and the broadband components of the noise produced within the sound source area and its propagation over the ventilation duct and into the surrounding ambient air. However, such computations would require non-affordably large computational resources. For this reason, within current project, the computational domains in which the turbulence-resolving (IDDES) computations of the considered flows are performed are limited to a relatively small part of the duct neighboring the source region (fan or diaphragm(s)).

As a result, the whole computational procedure, in addition to the main (“production”) IDDES computation of the flow in such truncated domain, involves two auxiliary precursor stages necessary for setting inflow and outflow boundary conditions for the production simulation. Specifically, the inflow profiles used in the IDDES stage of the computations are obtained from the steady RANS computation of the duct inlet flow, whereas the information needed for imposing the outflow boundary conditions is obtained from a precursor unsteady RANS (URANS) computation of the fan flow or from steady RANS of the flow in the duct with installed diaphragm(s). Schematics of the outlined computational procedure as applied to the fan and diaphragm flows and also computational domains.
and boundary conditions used in different stages of the simulations are presented in Figure 10-1. A more complete description of these boundary conditions as well as detailed information about the computational grids can be found in the IDEALVENT reports [2], [4] and in recent publications [5, 6].

Another consequence of carrying out scale-resolving simulations in the limited computational domain is that extraction of acoustic signal from such simulations is far from trivial. Indeed, in the vicinity of the noise-generating region (at least downstream of it) the acoustic signal is strongly covered with pressure fluctuations caused by turbulence and, in particular, by impinging of turbulent structures on the duct walls. Other than that, despite employing “non-reflecting” boundary conditions at the inlet and outlet of the IDDES computational domain, some inlet and outlet reflections of the sound waves are still possible. So, in order to extract acoustic signals that are free of reflections and hydrodynamic fluctuations, specialized supplementary acoustic techniques should be used, for which the detailed unsteady information obtained in the course of the scale-resolving IDDES of the flow serves as an input. Within IDEALVENT project this was done by KTH with the use of a methodology for acoustic mode extraction based on latest developments of a linear multi-port model [7]-[11]. Corresponding results are presented below in this section of the report. Another set of acoustic computations for both the ECS fan with inlet distortions and the single and tandem diaphragm configurations on the basis of the IDDES unsteady data obtained by NTS was performed by SISW. These computations were carried out with the use of a Finite Element Method with Adaptive Order (FEM AO) [12], available in Siemens PLM software LMS Virtual.Lab, and their results are discussed section 7 of the present report.

All the elements of the outlined computational approach were implemented in the NTS in-house general purpose code [13], which is a structured finite-volume high-order CFD/CAA code accepting multi-block overlapping grids of Chimera type. It has been intensively used for a wide range of aerodynamic and aeroacoustic problems and shown to be rather reliable and accurate. The code was adapted to the computation of the flows with rotating elements with the use of the approach based on employing different reference frames in different grid blocks combined with the sliding interfaces technique for imposing the boundary conditions on the interfaces between rotating and stationary grid blocks.
10.2 Effect of inlet distortions of ducted fan noise

10.2.1 Considered cases

A specific fan studied experimentally by VKI and computed by NTS is a generic ECS blower installed in a circular duct and having a rotor with 15 blades and a stator with 10 vanes (see Figure 10-2). The duct inner diameter $D$ is equal to 0.15 m, the rotation speed is around 11200 RPM, and the bulk flow velocity is approximately 32 m/s resulting in the Reynolds number of the flow close to that in practical applications.

In order to analyze the effect of the flow distortions upstream of the fan, three different cases were considered. In the first (baseline) case the duct has a smooth bell-mouth inlet resulting in a "clean" flow upstream of the fan with turbulence confined within a thin boundary layer on the duct wall. In the two other cases, a significant non-uniformity of the flow and elevated levels of turbulent fluctuations in the upstream part of the duct away from its walls are created by replacing the bell-mouth inlet by inlets with a rectangular-to-circular transition of the duct cross-section and with a T-junction formed by two circular inlet pipes (see Figure 10-3).

10.2.2 Sample flow visualizations, time-averaged fields, and turbulence statistics

Figure 10-4 which presents a visualization of the fan flow (baseline case of the bell-mouth duct inlet) in the form of instantaneous iso-surface of swirl (magnitude of the second eigenvalue of the velocity gradient tensor) “colored” by the value of streamwise velocity provides a visual representation of “rich” turbulent content captured by the scale-resolving IDDES approach. Other than that, Figures 10-5 to 10-10 give an idea about the effect of the inlet distortions caused by replacement of the smooth bell-mouth duct inlet by the two “distorted” inlets on the instantaneous flow parameters, time-averaged fields, and turbulent statistics.
Particularly, Figure 10-5 visibly reveals a significant difference in turbulence impinging on the front surface of the hub and on the rotor blades in the three considered cases. Namely, in the case of the clean (bell-mouth) inlet, only fine-grained turbulent structures confined within a narrow boundary layer on the duct wall are observed, whereas with the T-junction inlet, presence of large-scale turbulent structures upstream of the fan is evident, and the turbulence “covers” the whole duct cross-section. For the inlet formed by the rectangular-to-circular transition, an intermediate situation takes place. However, as seen from Figure 10-6, this difference in the upstream turbulence does not manifest itself any significantly in the resolved large-scale or fine-grained turbulence within the fan passage and behind the hub base. This can be explained by a generation of much more intense vortical structures inside the fan and in the shear layers separated from the trailing edge of the hub. The effect of the type of the duct inlet on the instantaneous and time-averaged velocity fields (Figure 10-7) turns out to be rather weak as well.

Figure 10-4: Two 3D-views of instantaneous swirl iso-surface $s=12U_0/D$ colored with magnitude of streamwise velocity from IDDES of baseline (bell-mouth) case.

Figure 10-5: Instantaneous fields of vorticity magnitude and streamwise and radial velocity upstream of fan in ducts with the three inlets.

Figure 10-6: Predicted effect of type of duct inlet on 3D views of instantaneous field of vorticity magnitude.
As far as the turbulent statistics is concerned, quite naturally, it turns out to be more sensitive to the presence of the inlet disturbances, which is seen, for example, in Figures 10-8, 10-9. They show that for the two disturbed inlets the flow distortions and turbulence generated by the inlet result in increased levels of both TKE in the core of the flow upstream of the fan and the wall-pressure fluctuations on the “front face” of the hub and on the leading edge of the rotor blades, the effect being much stronger in the case of the T-junction inlet. This, in turn, results in a noticeable alteration of the acoustic pressure field inside the duct, which is illustrated by Figure 10-10 presenting instantaneous distributions of pressure fluctuations over the duct wall for all the three considered cases. Particularly, the figure and (more visibly) corresponding animations reveal an increase of the amplitude of the acoustic waves propagating in the upstream direction in the ducts with the distorted inlets, which is consistent with the higher level of the $\text{rms}$ value of the pressure fluctuations on the fan surface discussed above. Again, the effect is pronounced stronger for the duct with the T-junction inlet.
10.2.3 Comparison with experiment on “raw” wall pressure spectra

A direct comparison of the “raw” wall pressure spectra obtained in the IDDES of the three considered flows and in the experiment for the microphones located both upstream and downstream of the fan module is shown in Figures 10-11 and 10-12.

When analyzing these spectra, the first thing that catches one’s eyes is a striking difference between the predicted and measured amplitude of the tonal noise of the fan corresponding to the 1st blade pass frequency (BPF) order harmonic ($f=2800$ Hz): indeed, in the experiment a strong tone at this frequency is observed, whereas in the simulations this tone is almost completely cut-off. Note, however, that for the considered ducted fan the 1st BPF harmonic cut-off is exactly what has to take place (at least for the case of an axisymmetric bell-mouth inlet) according to the rule of Tyler and Sofrin [15], because the tangential velocity of all the spinning acoustic modes corresponding to this harmonic is subsonic. So the presence of this tone in the measured spectra is, most probably, explained by violation of the flow periodicity in the experiment (e.g., because of presence of a thick power supply cable and some other irregularities of the geometry not taken into account in the numerical problem set-up).
Leaving aside this difference, the agreement of the simulation results with the data can be assessed as quite acceptable. Within the resolved frequency range, for all the three considered duct inlets a significant discrepancy is observed only in the low-frequency end of the spectra of the upstream propagating noise (see Figure 10-11), which at least partly may be caused by stronger reflections of the low frequency sound waves by the duct inlets in the experiment than in the simulations. Note also that the effect of the type of the inlet (Figure 10-12), i.e., the effect of the inlet distortions, which is of primary interest in the present study, is predicted more accurately than the absolute sound power.

10.2.4 Extracted acoustic modes

As mentioned above, extraction of the individual acoustic modes with filtered out inlet/outlet reflections and turbulent fluctuations from both the experimental and computational unsteady pressure fields was performed by KTH with the use of the linear multi-port acoustic approach.

Figure 10-13 presents a side outcome of this exercise, namely the frequency dependencies of inlet/outlet reflection coefficients for different acoustic modes propagating upstream and downstream of the fan in the duct with clean bell-mouth inlet. Results are shown for 6 modes with the lowest cut-on frequencies: the plane wave (mode (0, 0)) propagating at all frequencies, 4 azimuthal modes (m, 0), |m| = 1, 2 with cut-on frequencies at 1330 Hz (m = ±1) and 2220 Hz (m = ±2), and the first radial mode (0, n), n = 1 (cut-on frequency 2780 Hz which is very close to the blade passing frequency of the considered fan). Note that for the two other inlets, the computational reflection coefficients (not shown) are very close to those for the bell-mouth inlet, whereas advanced measurements allowing a modal decomposition were performed only for the bell-mouth inlet at frequencies (500 – 3500) Hz.

The figure demonstrates that the computational reflection coefficient is small everywhere, except for a close neighborhood of cut-on frequencies of corresponding modes and the lowest frequency range, \( f < 350 \) Hz (for the outlet reflections only). This provides a posteriori justification of neglecting scattering of the fan in the modal decomposition of the IDDES unsteady pressure field, except for the longest sound waves propagating downstream. For the latter, the reflections by the outflow sponge layer turn out to be very strong (possible reason(s) of this deficiency of the simulations should yet be understood), which may result in an inaccuracy of the amplitudes of extracted downstream propagating plane waves (at that low frequencies, all the other duct acoustic modes are cut-off). As for the sound waves reflections by the duct terminations in the experiment (both at the inlet and at the outlet), they are rather strong over the whole range of frequencies and for all the considered acoustic
modes, which suggests that accounting for both the reflection and the scattering effects in the modal decomposition of the measured pressure signals is necessary.

Figure 10-13: Inlet/outlet reflection coefficients for different acoustic modes propagating upstream (left) and downstream (right) of the fan in the duct with bell-mouth inlet. Dashed vertical lines correspond to cut-on frequencies.

Figure 10-14: Normalized amplitudes of acoustic modes extracted from experiment (bell-mouth inlet; black lines) and from IDDES of ducts with different inlets (bell-mouth inlet: green; rectangular-circular inlet: blue; T-junction inlet: red). Left: upstream propagating waves; right: downstream propagating waves.
Figure 10-14 compares the computed and measured amplitudes of the extracted acoustic modes for the bell-mouth inlet and illustrates the effect of the inlet type on these amplitudes predicted by the IDDES. Major observations based on this figure are as follows.

First, the agreement of the predictions with the experiment for all the acoustic modes is consistent with the observations concerning the raw wall pressure spectra discussed in Section IV-B above. Namely, similar to the wall pressure spectra (Figure 10-11), the major discrepancy of the simulation and experiment resides in the tonal noise of the fan corresponding to the 1st blade pass frequency order harmonic which shows up in the experiment and is absent in the computed spectra. The computations also reproduce the spectral shapes and levels of the broadband part of the individual acoustic modes, which is encouraging considering that these are rather subtle characteristics of the sound field of the source. Unfortunately, the lack of the experimental data at frequencies lower than 500 Hz does not allow suggesting any specific reasons of the discrepancy between the measured and computed raw pressure spectra of the upstream propagating noise observed in the low frequency range (see Figure 10-11).

As far as the effect of the inlet distortions predicted by the IDDES is concerned, for the individual modes it turns out to be not exactly the same as for the wall pressure spectra (Figure 10-12). In particular, for the individual modes, replacement of the bell-mouth inlet by the T-junction inlet, in a wide frequency range results in a significant (3-5 dB) increase of the noise propagating both upstream and downstream, whereas for the wall pressure spectra the effect for the downstream direction is almost unnoticeable, which is explained by “contamination” of the wall pressure signal at the downstream microphone location by turbulent pressure fluctuations. At low frequencies (f < 300 Hz) the noise penalty of using the T-junction inlet increases for the upstream direction up to ~20 dB. For the rectangular-circular inlet for both, individual acoustic modes and total wall pressure, the effect is marginal, except for the low-frequency noise propagating upstream, which is up to 10dB stronger than in the system with the bell-mouth inlet.

10.3 Computations of flow and noise produced by single and tandem diaphragms

10.3.1 Considered cases

Two tandem configurations were considered with the separation distances between the diaphragms equal to 2 and 4 duct diameters, \( D \). In addition, a baseline case of a single ducted diaphragm was analyzed. Geometrical and flow regime parameters of the considered flows are as follows: the duct diameter is equal to 0.15 m, the diameter of the diaphragms orifice is \( d = 0.116 \) m, which corresponds to \( \approx 60\% \) open area, and the volume flow rate through the duct is \( VFR = 0.42 \) m\(^3\)/s (the bulk velocity is around 25 m/s). These parameters correspond to the conditions of the experimental study of these three configurations (see Figure 10-15), which was carried out by VKI [16].
10.3.2 Sample results and comparison with available experimental data

Major results of the three simulations described above are presented in Figures 10-16 – 10.24 in the form of a 3D view of the instantaneous iso-surface of swirl (magnitude of the second eigenvalue of the velocity gradient tensor - Figure 10-16), snapshots of vorticity magnitude and temperature in the meridian plane of the ducts (Figures 10-17, 10-18), mean velocity fields (Figure 10-19), fields of rms of pressure fluctuations (Figure 10-20), distributions of the azimuthally averaged rms of wall pressure fluctuations along the ducts wall (Figure 10-21) and finally comparison of predictions with the VKI experimental data on power spectra and rms of these fluctuations (Figures 10-22, 10-23, and 10-24).

Figure 10-15: Three diaphragm flow configurations tested in experiments of VKI. The figure shows also RANS and IDDES computational domains and location of VSTG source region used in the simulations.

Figure 10-16: Instantaneous iso-surface of swirl colored by streamwise velocity in the vicinity of the diaphragm from IDDES of single diaphragm flow

Figure 10-17: Instantaneous fields of vorticity magnitude in meridian plane of the ducts
Figure 10-18: Instantaneous fields of temperature in meridian plane of the ducts

Figure 10-19: Time-averaged fields of streamwise velocity in meridian plane of the ducts

Figure 10-20: Fields of root-mean-square fluctuations of pressure in meridian plane of the ducts

Figure 10-21: Distributions of azimuthally-averaged rms of wall-pressure fluctuations along the ducts
Figure 10-22: Induct microphones location with corresponding notations (upper frame) and comparison with experimental data on distribution of rms of wall-pressure fluctuations along the duct and wall-pressure spectra at microphones positions for Tandem-4D case. For the upstream microphones, predicted spectra are corrected with the use of empirical correlation of Goody [14] to account for the contribution of turbulence of attached boundary layer on the duct wall.

Figure 10-23: Same, as in Figure 10-22, for Tandem-2D case.

Figure 10-24: Same, as in Figure 10-22, for single diaphragm case.
Observations and some conclusions that can be made based on the analysis of these figures are as follows.

1). The “shear layer adapted” modification of the subgrid length scale $\Delta = \Delta_{SLA} = \tilde{\Delta}_\theta F_{KH}$ used in the IDDES carried out in the present work (see [4] for more detail) ensures pretty rapid formation of resolved vortical structures in the shear layers separated from the edges of the first (upstream) diaphragm (Figure 10-17), which is the main reason of a much better agreement of the locations of predicted and measured maxima of the pressure fluctuations on the duct wall behind the diaphragm (Figures 10-22 to 10-24) compared to that obtained from the IDDES with the standard definition of the subgrid length scale $\Delta = \Delta_{max}$ (see [4]).

2). In all the three considered configurations, the length of the “inviscid” core of the flow behind the first diaphragm is around 3 duct diameters ($\approx 0.45$ m) – see Figures 10-17, 10-18, and 10-19. As a result, for the tandem-4D configuration the second diaphragm turns out to be located in the region of already fully developed turbulent flow. In contrast, for the tandem-2D configuration, the front wall of the second diaphragm gets into the shear layer separated from the first diaphragm, with more intense turbulent vortices than in the fully developed turbulence. It can be assumed that exactly this is the reason of a somewhat stronger noise emitted upstream by the tandem-2D geometry than by the tandem-4D one (Figures 10-20 and 10-21). Note also that as could be expected, in terms of the upstream noise level both tandem configurations turn out to be much more “noisy” than the single diaphragm.

3). Overall, the predictions of both the wall-pressure spectra, which are a major outcome of the simulations, and of the $rms$ of wall-pressure fluctuations are in good qualitative and acceptable quantitative agreement with experiment. The main discrepancy resides in the low frequency end of the spectra of the upstream propagating noise (100-300 Hz) and is somewhat more pronounced for the single diaphragm case. A cause of this discrepancy is not quite clear so far, but most likely it is associated with noticeable reflections of the low frequency sound waves by the duct inlet in the experiment. Particularly, because of these reflections, the presented comparison of the “raw” spectra from the simulations and experiment cannot be considered as fully consistent.

4). A more consistent comparison was provided by S. Sack of KTH who has extracted from both the experimental and computational pressure signals the individual acoustic modes free of the impact of the turbulence impinging on the duct walls and inlet/outlet reflections of sound waves. The results of this comparison, which can be evaluated as rather encouraging, are shown if Figures 10-25, 10-26 for both the sound waves propagating upstream and downstream of the diaphragm(s).
Figure 10-25: Normalized amplitudes of individual acoustic modes extracted from IDDES of single orifice flow and their comparison with post-processed measurements of VKI. Left: upstream propagating waves; right: downstream propagating waves. Solid lines: experiment; symbols: IDDES. These results and the figure are provided by S. Sack of KTH.

Figure 10-26: Same as in Figure 10-25, for tandem diaphragm with separation 4D.
10.4 Conclusions

The studies of a fan and diaphragms installed in a circular duct carried out by NTS in collaboration with KTH within WP4 of IDEALVENT are in general rather successful and conclusive. First of all, they clearly demonstrate a high potential of hybrid RANS-LES scale-resolving approaches (and the IDDES method, in particular) for prediction of aerodynamics and noise generated by ECS flows. Other than that, they show a high value of invoking a technique of modal decomposition coupled with the acoustic multi-port approach which allows extracting “pure” (with filtered out turbulent fluctuations) acoustic modes propagating under reflection-free conditions and thus makes possible an adequate comparison of predictions and measurements.

10.5 Bibliography


