68th Conference of the Italian Thermal Machines Engineering Association, ATI2013

Numerical Analysis of the Effects of Surface Roughness Localization on the Performance of an Axial Compressor Stage

Nicola Aldi\textsuperscript{a}, Mirko Morini\textsuperscript{b}, Michele Pinelli\textsuperscript{a\*}, Pier Ruggero Spina\textsuperscript{a}, Alessio Suman\textsuperscript{a}, Mauro Venturini\textsuperscript{a}

\textsuperscript{a}Dipartimento di Ingegneria, Università degli Studi di Ferrara, Via G. Saragat 1 - 44122 Ferrara, Italy
\textsuperscript{b}MechLav - Università degli Studi di Ferrara, Via Guercino, 47 - 44042 Cento (FE), Italy

Abstract

In this paper, the most common and experienced source of loss for a gas turbine, i.e. compressor fouling, is modeled and analyzed by means of a three-dimensional numerical approach. In particular, CFD simulations of fouling affecting an axial compressor stage are carried out. To do this, the NASA Stage 37 is considered as compressor model for numerical investigation. The numerical model, validated against experimental data available from literature, is used to simulate the occurrence of fouling by imposing different combinations of added thickness and surface roughness. The results highlighted that the main effect of fouling is the decrease in the flow rate, even if a decrease in the stage compression ratio was also noticed. Therefore, different non-uniform combinations of surface roughness levels on rotor and stator blades were imposed. Simulations showed that the greatest effect on performance is attributable to the rotor, and in particular, by analyzing the effect of roughening pressure surface and suction surface separately, to its suction surface. Moreover, a rotor roughness non-uniformity is considered also in spanwise direction, which causes a significant work redistribution.

© 2013 The Authors. Published by Elsevier Ltd. Open access under CC BY-NC-ND license. Selection and peer-review under responsibility of ATI NAZIONALE

Keywords: axial compressor stage; fouling; surface roughness; CFD simulation

* Corresponding author. Tel.: +39 0532 974889; fax: +39 0532 974889.
E-mail address: michele.pinelli@unife.it
1. Introduction

Gas turbine performance degradation over time is mainly due to the deterioration of compressor and turbine blades, which, in turn, causes a modification of compressor and turbine performance maps. Compressor fouling, in particular, is one of the main causes of gas turbine performance degradation: it can be estimated that 70 - 85% of the
overall gas turbine performance loss due to continuous operation, can be attributed to compressor fouling [1]. Since detailed information about the actual modification of compressor and turbine performance maps is usually unavailable, component performance can be modeled and investigated (i) by scaling the overall performance map [2-3], or (ii) by using stage-by-stage models of the compressor and turbine and by scaling each single stage performance map to account for each stage deterioration or (iii) by performing 3D numerical simulations, which allow to both highlight the fluid-dynamic phenomena occurring in the faulty component and grasp the effect on the overall performance of each component.

In this paper, the most common and experienced source of loss for a gas turbine, i.e. compressor fouling, is modeled and analyzed by means of a three-dimensional numerical approach. In particular, CFD simulations of fouling affecting an axial compressor stage are carried out. To do this, the NASA Stage 37 is considered as compressor model for numerical investigation. The numerical model, validated against experimental data available from literature, is used to simulate the occurrence of fouling by imposing different combinations of added thickness and surface roughness. The results highlighted that the main effect of fouling is the decrease in the flow rate, even if a decrease in the stage compressor ratio was also noticed. Different non-uniform combinations of surface roughness levels on rotor and stator blades were then imposed. Then, the analysis of the effect of roughening pressure surface and suction surface separately is considered. Moreover, a rotor roughness non-uniformity is considered also in spanwise direction.

2. Literature Background

In literature, many interesting papers on the effects of compressor fouling on gas turbine and compressor performance have been presented. A good review of the main mechanisms and of the effects of fouling on the overall performance of gas turbines can be found in [4]. In more detail, some papers deal with the analysis of compressor performance deterioration by means of numerical calculations and only a few of them also report experimental data.

Bammert and Woelk [5] conducted experimental tests on a three-stage compressor with an increasing blade surface roughness \( (k/c = 1.5 \times 10^{-5}, 2.5 \times 10^{-5} \text{ and } 4.5 \times 10^{-5}) \). They noticed: (i) a shift towards lower mass flow rates of the compressor performance maps; (ii) a decrease of the range of mass flow rate at a given rotational speed and (iii) a decrease in compressor efficiency.

The paper by Suder et al. [6] deals with performance deterioration the NASA Rotor 37 due to artificially imposed alterations in terms of surface roughness in the range \((0.508\pm3.18) \mu m\) and airfoil thickness variations. A quasi-3D Navier-Stokes flow solver is used to simulate these alterations and the comparison with the experimental data is reported. Among the numerous and very interesting results, the most significant for this paper are: (i) the maximum mass flow achieved by the altered rotor is lower than that of the baseline rotor at all rotational speeds; (ii) at constant pressure rise, the efficiency loss is in the region of \((2.5\pm6.5) \%\) for the thickened smooth surface and \((6.0\pm8.5) \%\) for the thickened rough surface; (iv) a \(9\ \%\) loss in pressure ratio across the rotor when near the design flow is observed for rough surfaces; (v) regarding non-uniform roughness, pressure surface roughness was found to have little impact on performance degradation, which is triggered by additional thickness and/or roughness at the leading edge. Finally, the authors found that compressor performance characteristics predicted by CFD calculations are sensitive to the addition of surface roughness to the numerical model, but at the same time the roughness model significantly under predicts the performance changes with respect to the measured ones.

Gbadebo et al. [7] presented the results of experiments and numerical simulations performed on a large-scale single-stage low-speed compressor. The experimentally imposed roughness was chosen by scaling roughness measurement values obtained from a turbofan engine after a long period of airline operation, these values being established in the range \(R_s = (1.53\pm2.03) \mu m\). The roughness to be imposed on the test-rig blades was calculated to be about \(25 \mu m\), which corresponds to an equivalent sand grain roughness \( k_s = 160 \mu m\). Regarding numerical simulations, they used a 3D Navier-Stokes code which uses a control volume formulation on a structured mesh, an eddy viscosity mixing length for turbulence modelling and an ad hoc developed simple model to simulate roughness effects. The most interesting results can be summarized as follows: (i) surface roughness typical of real engine operations can lead to a significant reduction in performance due to the effect on 3D separation; (ii) when tested as a stage, the 3D separation induced by roughness causes a significant loss in stage total pressure over a wide range of flow; (iii) the measured overall mass-averaged total pressure rise coefficient reduced by about \(5.4\ \%\) due to
roughness, while the numerical simulations significantly underestimated this reduction (the calculated reduction of the mass-averaged total pressure rise coefficient was about 2.4 %).

Additional recent considerable contributions to this topic are the papers by Syverud et al. [8] and Syverud and Bakken [9]. In [8], test results from a series of accelerated deterioration tests on a GE J85-13 jet engine are presented. The axial compressor (8 stages, pressure ratio equal to 6.5) was deteriorated by spraying atomized droplets of saltwater into the engine intake. They noticed that the roughness of the rotor blade deposits is 50 % of the roughness found on the stator vanes. They also found that surface roughness was not uniform and that the greatest roughness values were found on the pressure side of the vanes. Measured equivalent sand grain roughness values \( k_s \) were in the range \( R_a = (15÷25) \mu m \) for the pressure side and \( R_a = (2÷15) \mu m \) for the suction side, which correspond to relative surface roughness values \( k_s/c \) ranging from \( 0.6 \cdot 10^{-4} \) to \( 11 \cdot 10^{-4} \). Regarding compressor performances, they found: (i) a shift of the compressor operating line to a lower flow rate and a lower pressure ratio; (ii) deterioration reduced both the stage work coefficient and the flow coefficient. In [9], the experimental results gathered in [7] were compared against models based on correlations in terms of loss estimates due to fouling. None of the frictional loss models proved to be effective in predicting the large variation in the measured total pressure coefficient from clean to deteriorated conditions. All the models considered capture the shift in flow coefficient but the degradation effect due to roughness in the stage performance was underpredicted.

3. Geometrical and numerical model

In order to investigate the effects of fouling on a compressor stage, the NASA Stage 37 test case was used as the baseline geometry. The effort was focused on simulating the entire compressor stage (rotor and stator) rather than the rotor alone. This could lead to a lower accuracy of the computational model with respect to modelling the rotor alone, because of the presence of a rotor/stator interface models and of the assumptions inherent in the periodicity of the computational domain. However, the simulation of the entire compressor stage could give thorough information on compressor stage performance deterioration. Hence, since the aim of the paper was to give an insight in the change of compressor stage performance caused by deterioration, the model development and validation focused on reproducing, as accurately as possible, the behaviour – in terms of performance map shape and main fluid dynamic features - of an axial compressor stage rather than perfectly matching the experimental data of the original compressor. Hence, the choices and the considerations in the following paragraphs should be read in this way.

The geometry and performance data of NASA Stage 37 are gathered from [10]. The stage is composed of a 36-blade rotor and a 46-blade stator. The overall performances at design point (corrected mass flow of 20.19 kg/s and rotational speed of 17,188 rpm) are \( \beta = 2.050 \) and \( \eta_u = 0.842 \). To reduce computational effort, only a section of the full geometry is modelled. The computational domain consists of 4 rotor and 5 stator blades. This results in a 40.00° section for the rotor and a 39.13° section for the stator. This also results in a rotor/stator pitch ratio at the interface equal to 1.250 instead of 1.278 of the real geometry (2.2 % difference). The grids used in the calculations are hybrid grids generated by means of ANSYS ICEM CFD [11]. Sketches of the numerical grid are reported in Figs. 1a and 1b. The grids are realized by starting from a tetrahedral mesh core and then by adding prism layers on the surface of the blades to better resolve the boundary layer around the blade. After a grid sensitivity study, the chosen final mesh is composed of about 1,055,000 elements (on a 1/9 section). The grid has 9 prism layers on the surface and the first grid point height away from the blade surface is fixed at 50 \( \mu m \). These choices are discussed in detail in [12]. The numerical simulations were carried out with the commercial CFD code ANSYS CFX [13]. The code solves the 3D Reynolds-averaged form of the Navier–Stokes equations by using a finite-element based finite-volume method. An Algebraic Multigrid method based on the Additive Correction Multigrid strategy was used. A second-order high-resolution advection scheme was adopted to calculate the advection terms in the discrete finite-volume equations. The simulations were performed in a steady multiple frame of reference in order to consider the contemporary presence of moving and stationary domains. In particular, a Mixing Plane approach was imposed at the rotor/stator interface (which was located half-way between the two components) and used for all the simulations. Finally, the approximation of the rotor/stator pitch ratio is handled by the code by scaling the flow that crosses the interface.
3.1. Turbulence and Wall Roughness Model.

In this paper, the RANS eddy-viscosity two-equation $k$-$\varepsilon$ model is used. In the code used for the calculations, the wall functions used are scalable wall functions. Scalable wall functions are based on the analytical-wall-function approach (well documented in [14]), in which a modified turbulent velocity scale $\tilde{u}_e$ is used, which is dependent on the turbulent kinetic energy at the near-wall node $k_p$

$$\tilde{u}_e = C_u k_p^{1/2}$$

and, as a consequence, a modified $y^+$ based on $\tilde{u}_e$ can be obtained:

$$\tilde{y}^+ = \frac{\tilde{u}_e y}{v}$$

The idea behind the scalable wall-function approach implemented in [13] is to limit the computed $\tilde{y}^+$ value used in the logarithmic formulation from falling below 11.06, which is the value assumed for the intersection between the logarithmic and the linear near wall profile.

To account for roughness effects, the near-wall model of the $k$-$\varepsilon$ turbulence model has to be modified. The near-wall functions, as described above, are appropriate when walls can be considered hydraulically smooth. For rough walls, the logarithmic profile still exists, but moves closer to the wall. As an index of the wall roughness, the equivalent sand grain $k_s$ has been widely used by many researchers [15]. The corresponding Center Line Average roughness $R_a$ is related empirically to $k_s$. In this paper, the coefficient proposed in [16] is used, resulting in the relationship $k_s = 6.2 R_a$. Roughness effects can be accounted for by modifying the logarithmic profile as follows:

$$u^+ = \frac{u^+}{u_e} = \frac{1}{\kappa} \ln \left( \frac{\tilde{y}^+}{1 + 0.3 \cdot \tilde{k}^+} \right) + C$$

where

$$\tilde{k}^+ = \frac{k \tilde{u}_e}{v}$$

Equations (3) and (4) are coherent with the definition of the scalable wall functions. In turbomachinery calculations, to distinguish between the hydraulically smooth regime and rough regimes, Koch and Smith [16] proposed a method based on the definition of a roughness Reynolds number $Re_k = k_s W_1 / v$, where $W_1$ is the relative inlet velocity and $v$ is the kinematic viscosity of the fluid. The method states that a surface is hydraulically smooth if $Re_k$ is less than 90.

3.2. Boundary conditions.

The total pressure, total temperature and flow angle were imposed at the inflow boundary. The total pressure and total temperature at the stage inlet were fixed at $p_{01} = 101,325$ Pa and $T_{01} = 288.15$ K, respectively. Regarding
outflow boundary conditions, in the near-choked flow region, an average relative static pressure $p_{r2}$ was imposed at the outflow boundary. The outflow pressure was progressively increased from $p_{rel3} = 0$ Pa until near-stall region is reached. In the near-stall region, a mass flow rate was imposed at the outflow boundary. The mass flow rate was progressively decreased to simulate different working points. Finally, since only a section of the full geometry has been modeled, rotational periodic boundary conditions were applied to the lateral surfaces of the flow domain. The imposed angular velocity refers to the working conditions at 100% design speed [10]. Therefore, the imposed rotational speed for the calculations was set as $n_{CFD} = 17,197$ rpm.

4. Model validation

As a first step, the model was validated against the experimental data reported in [10], which refer to a real non-deteriorated compressor. The calculations focus on the $k-$İ model with the roughness model activated, since this is the model used when blade deterioration is considered in the calculation. For this reason, an equivalent sand grain roughness equal to $k_i = 1 \mu$m ($R_a = 0.16 \mu$m) was set, which is an hydraulically smooth condition ($Re_k = 25$). The calculated and experimental performance maps are reported in Fig. 2. The error bars reported for the experimental data are not the actual experimental uncertainty, but are intended to fix a reference offset to judge the simulation validity. The error bars are equal to $\pm 2\%$ of the reading for $T_{03}/T_{01}$, $\pm 3\%$ of the reading for $p_{03}/p_{01}$, $\pm 3\%$ of the reading for $\eta_s$. The validation established that the shapes of all the experimental performance maps of the compressor stage are correctly reproduced by the numerical code, and the error between the calculated and measured values of the mass flow in choked condition is about 1.4% at 100% rotational speed. It can also be noticed that the numerical values are in good agreement with the experimental data. If the variability of the rotational speed – due to both measurement uncertainty ($\pm 30$ rpm) and experimental repeatability – is considered, the agreement is more than satisfactory. Since the aim of the validation was to obtain a compressor model, the numerical model can be considered reliable.

5. Model application to blade fouling

5.1. Model application to uniform blade surface roughness.

In this section the compressor stage numerical model, which has previously set up and validated, is used to simulate the behaviour of compressor stages when blade fouling occurs. Following the methodology carried out in [17], fouling on compressor stages was simulated by adding roughness and thickness to the blade surface. Roughness and thickness were added to stator and rotor in order to predict the modifications in the performance of the entire stage rather than the rotor alone. Moreover, modifications were imposed simultaneously on all the relevant blade surfaces (suction side, pressure side, leading edge, trailing edge), and on their entire extension from inlet to outlet and from hub to shroud (100% pressure side and suction side surface coverage). The surface roughness was taken into

![Fig. 2. Performance maps. Comparison between experimental data and numerical results.](image-url)
account by means of the roughness model outlined above. In particular, the value of \( k_s \) was imposed equal to 40 \( \mu \text{m} \) by considering a relative roughness values \( k_s/c \) of the same order of magnitude as the values usually found in literature [7,8,18,19]. The magnitude of the added thickness \( \Delta_t \) was chosen in order to: (i) simulate the presence of dust and particles which could have stick on the airfoil surface and, thus, alter the geometry of the airfoil and, (ii) take into account that roughness models can usually significantly underestimate the real effect of surface roughness [7,17]. Added thickness was magnified with respect to the actual values reported in literature, in order to partially compensate for underestimation issues of the roughness models. Hence, the added thickness \( \Delta_t \) was set equal to 0.3 mm. Moreover, the effect of surface roughness was taken into account by differentiating between the suction side and pressure side of the rotor. In these cases, no thickness was added neither to stator nor to rotor. Following these assumptions, different fouling conditions were simulated as reported in Table 1.

<table>
<thead>
<tr>
<th>Rotor ps</th>
<th>Rotor ss</th>
<th>Stator</th>
<th>( k_s/\mu\text{m} )</th>
<th>( \Delta_t/\text{mm} )</th>
<th>( k_s/c )</th>
<th>Re0</th>
</tr>
</thead>
<tbody>
<tr>
<td>SR-SS</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>-</td>
<td>-</td>
<td>1.77 \times 10^5</td>
</tr>
<tr>
<td>t-SR-SS</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>+0.3</td>
<td>-</td>
<td>7.14 \times 10^4</td>
</tr>
<tr>
<td>t-RR-RS</td>
<td>40</td>
<td>40</td>
<td>40</td>
<td>+0.3</td>
<td>-</td>
<td>7.14 \times 10^4</td>
</tr>
<tr>
<td>RR-RS</td>
<td>40</td>
<td>40</td>
<td>40</td>
<td>-</td>
<td>-</td>
<td>7.14 \times 10^4</td>
</tr>
<tr>
<td>RR-SS</td>
<td>40</td>
<td>1</td>
<td>1</td>
<td>-</td>
<td>-</td>
<td>7.14 \times 10^4</td>
</tr>
<tr>
<td>RR(ps)-SS</td>
<td>40</td>
<td>40</td>
<td>1</td>
<td>-</td>
<td>-</td>
<td>7.14 \times 10^4</td>
</tr>
<tr>
<td>RR(ss)-SS</td>
<td>1</td>
<td>40</td>
<td>1</td>
<td>-</td>
<td>-</td>
<td>7.14 \times 10^4</td>
</tr>
</tbody>
</table>

The simulated compressor performance curves are reported in Fig. 3(a). As can be seen, the main effect of fouling is the decrease in mass flow rate. However, if a plausible compressor working line is followed, it is possible to also notice a decrease in the stage compressor ratio.

In Fig. 3(b), the performance curves for the cases Smooth-Rotor/Smooth-Stator (SR-SS), Rough-Rotor/Smooth-Stator (RR-SS) and Rough-Rotor/Rough-Stator (RR-RS) are reported. As can be seen, the roughness on rotor blades is responsible for almost the entire performance deterioration, while the roughness on stator blades has a nearly negligible influence on performance (more details can be found in [20]). Therefore, further analyses were carried out by considering the roughness on only the rotor surfaces. In particular, the cases of roughness added only to the rotor blade pressure surface (RR(ps)-SS) and suction surface (RR(ss)-SS) have been analyzed. As can be seen, most of the performance deterioration can be attributed to the roughness on the suction side, although the effect of the roughness on the pressure side is noticeable.
In Fig. 4(a), the surface static pressure distributions at 50% of the span for the four cases SR-SS, RR-SS, RR(ps)-SS and RR(ss)-SS are reported. This result confirms the evidence of the loading redistribution caused by surface roughness. In fact, the reduction in blade loading is mostly concentrated from leading edge to mid-chord and performance deterioration can be attributed to the roughness on the suction side. A comparison of the flow field in smooth and rough blades is provided by the streamwise Mach number distributions, shown in Fig. 4(b). It has to be preliminary noticed how the averaging does not allow the shock sharpness to be resolved. However, this has little impact on macroscopic performance. The location and strength of the passage shock are almost the same for the two cases. The main difference is in the diffusion downstream of the shock, which results in a higher Mach number at the trailing edge. The different diffusion in the aft part of the blade between the smooth and rough blade is clearly visible and confirmed experimentally [6]. This result is consistent with the reduced pressure rise seen in Fig. 4(a).

Fig. 4. (a) Surface static pressure distributions at 50% of the span and (b) streamwise Mach number distributions.

5.2. Model application to non-uniform blade surface roughness.

The numerical model of the axial compressor stage was also used to simulate the occurrence of blade fouling characterized by a non-uniform spanwise distribution of blade roughness on rotor blades (both suction and pressure surfaces), designated in this paper as Non Uniform Roughness Distribution (NURD), applied on the thickened stage. To perform the simulations carried out in this paper, the blade surfaces were divided into 10 regions along the spanwise direction, to impose the roughness distributions. The number of regions was identified starting from the original number of fluid dynamic sections used for the blade design. The sketch of the computational domain is reported in Fig. 1b, where the different colors allow the identification of the 10 regions where a different roughness level can be imposed. It should be noted that the different regions are characterized by different surface areas. The three cases considered in this paper are reported in Fig. 5. More details and more cases can be found in [21]. The performance maps are reported in Fig. 6(a).

It can be observed that, in all the considered distributions, the shape of the performance maps is similar and in agreement with the baseline case (i.e. t-SR-SS). The NURD 3, which simulates the most severe fouling, clearly leads to the most severe performance drop. The performance maps of NURDs 1 and 2 are superimposed and the performance drop is less marked, since fouling is less severe. The most severe is the NURD 2, probably due to the increase of roughness at blade tip. The pressure ratio can be evaluated as a function of the span height at the rotor outlet section. The trend is shown in Fig. 6(b), which, from a physical point of view, is also representative of rotor work distribution. The curves show that, in NURDs 1 and 3, a significant redistribution of the work occurs, mainly at blade hub and a steeper variation along the span is observed.

The results presented above suggest that, in addition to the local value of surface roughness, the stage performance may be represented by an averaged roughness parameter, which can be defined as

$$k_s = \frac{1}{A_s} \int k_s dA$$

Since the blade surface are divided into a discrete number of surfaces, the parameter $\bar{k}_s$ can be defined as the average of surface roughness values, weighted on the surface area of each sliced region. The stage performance maps were recalculated by using the averaged roughness parameter values, for the same outflow boundary...
conditions. The quantitative comparison of between NURD and averaged roughness cases is reported in Table 2 in terms of RMSE of mass flow rate and pressure ratio on the eight calculated points. It can be observed that the deviation is always lower than 0.19 % and it is almost negligible in most cases. The highest values of RMSE are obtained for NURD 1 and NURD 2, probably because of the high difference between the roughness at hub and the roughness at tip.

![Fig. 5. Non uniform roughness distributions on rotor blades.](image)

![Fig. 6. (a) NURD stage performance and (b) pressure ration vs. span height.](image)

<table>
<thead>
<tr>
<th>NURD vs. averaged</th>
<th>$m$ [kg/s]</th>
<th>$p_{op}/p_{ol}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.022</td>
<td>0.189</td>
</tr>
<tr>
<td>2</td>
<td>0.052</td>
<td>0.074</td>
</tr>
<tr>
<td>3</td>
<td>0.005</td>
<td>0.051</td>
</tr>
</tbody>
</table>

The analysis about the local effect of roughness on blade loading shows that the averaged roughness curve is in practice superimposed in the case of NURD 3 distribution, while a deviation can be highlighted for NURD 1 and 2. Such a deviation can be explained by considering that, at blade tip, the value of $k_t$ of the NURD is remarkably different from the value of $\bar{k}_t$. In conclusion, the use of the averaged roughness parameter $\bar{k}_t$ allows highly accurate prediction of stage performance at macro-scale level (see Table 2), but the local effects can be grasped only by considering the actual distribution of roughness on blade surface.

6. Conclusions

In this paper, numerical simulations of fouling affecting an axial compressor stage were carried out. To do this, the NASA Stage 37 was considered for the numerical investigation, which was performed by means of a commercial CFD code. The code, set up against experimental data taken from literature, allowed a representative model of a realistic compressor stage to be obtained, both in terms of overall performance. The model was used to simulate the
occurrence of fouling by imposing different combinations of added thickness and surface roughness levels in order to estimate the performance modification of axial compressor stages due to fouling. The results highlighted that the main effect of fouling is the decrease of the flow rate, which is higher when fouling condition becomes more severe, but a decrease in the stage compressor ratio was also noticed. Different non-uniform combinations of surface roughness levels on rotor and stator blades were imposed. Since a roughened rotor shows the greatest effect, this case was studied in more detail by roughening pressure surface and suction surface separately. Simulations showed that the main effect on stage performance is due to suction surface roughness. Finally, a rotor roughness non-uniformity was considered also in spanwise direction and a significant work redistribution was highlighted.

References