Mesh-converged adaptive simulations of the NASA Rotor 37 transonic compressor

Frédéric Alauzet
GAMMA Team, Inria Saclay Ile-de-France, Palaiseau, France

Dimitrios Papadogiannis†
Safran Tech, Modelling and Simulation, Magny-les-Hameaux, France

This paper demonstrates the efficiency of metric-based unstructured anisotropic mesh adaptation techniques for turbomachinery applications. It shows that mesh-independent numerical solutions are obtained thanks to anisotropic mesh adaptation and that it is possible to run high-fidelity CFD on fully unstructured adapted meshes composed only of tetrahedra. This paper emphasizes how mesh adaptation, thanks to its automation, is able to generate meshes that are extremely difficult to envision and practically impossible to generate manually, leading to highly accurate numerical solutions. The chosen test case is the NASA Rotor 37 axial flow compressor in transonic regime. This study compares feature-based error estimate based on the standard multi-scale $L^p$ interpolation error estimate and goal-oriented error estimate using an adjoint state to control the error on turbomachinery output functionals.

I. Introduction

In modern Reynolds-Averaged Navier-Stokes (RANS) numerical simulations, the mesh generation and CAD discretization is known to be one of the main bottlenecks for many applications. This is particularly the case for the generation of suitable meshes for turbomachinery geometries which remains a difficult task. Turbomachinery flows are wall-bounded flows that are characterized by multiple privileged directions. Such directions can be found at the near-wall boundary layers across the blades, hub and casing, at the wakes as well as at the shocks in the cases with transonic operating points. The impact of the near-wall boundary layers and wakes in particular on the overall predictions, combined with the necessity for periodic boundary conditions in the azimuthal direction (to simulate only a single blade per row), renders the simulations of turbomachinery on structured meshes very effective, as such meshes allow for high resolution on those areas. It is usually addressed by the mean of multi-block structured meshes composed only of hexahedra.

Traditional processes rely on the experience and intuition of a skilled engineer to predict and adapt the mesh prescription to the flow. The generation of multi-block structured meshes requires a careful management but nowadays automatic processes have been set-up and work very well when a simple geometry like a blade is considered. However, a standard structured mesh will likely be of insufficient resolution at a shock or around the secondary flows developing across the blade passage, phenomena crucial for the performance and the stability of turbomachinery. Additionally, convecting correctly the wakes far from the blade would require a very large number of cells. Furthermore, the current trend is towards more realistic turbomachinery simulations that include complex technological effects (cavities, seals, squealers, fillets, cooling etc) as these effects, previously usually taken into account via correlations, are shown to impact considerably the flow field. Meshing such complex geometries using structured meshes is very difficult, while the flows in such effects are no longer characterized by the privileged directions observed across a blade, for example. As a result, the mesh generation process is slowed down considerably, leading to a prohibitive cost in pre-processing time of the numerical simulation pipeline. This lack of automation is an impediment for many applications.

*Senior Researcher, Frederic.Alauzet@inria.fr
†Research Engineer, dimitrios.papadogiannis@safrangroup.com
These constraints imply that introducing anisotropic mesh adaptation techniques with unstructured meshes can be an attractive solution to achieve improved predictions, as it can combine easy initial mesh generation with the ability to follow the flow’s preferred directions while being capable of refining around all relevant flow phenomena. This work proposes a metric-based anisotropic mesh adaptation strategy based on unstructured meshes composed only of tetrahedra which is able to automatically manage complex geometries and technological effects, the periodicity of the domain, and adapt the mesh in size and in direction to the flow.

Metric-based anisotropic mesh adaptation is a well known topic for external flows. We have demonstrated that high-fidelity prediction for viscous flows can be obtained using fully unstructured meshes, which until now was considered impossible in the community. Starting from an initial coarse non-adapted mesh which is generated without any a priori knowledge on the solution, it consists in iteratively modifying the computational mesh in order to better capture all the flow features and to obtain an improved numerical solution for a given number of degrees of freedom. The size and the orientation of the elements of the mesh is given by an error estimate which evaluates the error in the computation of the solution due to the discretization. This process can be combined with a convergence study by incrementing gradually the size of the mesh, and thus giving the capability to check the independence of the numerical solution to the domain discretization.

In this paper, we illustrate the efficiency of this technology to deliver mesh-converged solutions for the full NASA Rotor 37 compressor characteristic. This is of utmost importance to, in second step, analyze the impact of the considered geometric and/or physical models such as the turbulence models and/or complex technological effects.

II. Anisotropic mesh adaptation algorithm with mesh-convergence analysis

Mesh adaptation is a non-linear problem where the couple formed by the mesh and the solution needs to be converged at the same time. Therefore an iterative process is required which is usually achieved by means of a mesh adaptation loop starting from an initial mesh $H_0$, an initial solution $W_0$, an initial adjoint state $W^*_0$ if goal-oriented mesh adaptation is considered, and a given mesh complexity $C$ (the continuous counterpart of the mesh size).

At each step of the mesh adaptation loop, a metric tensor $M_i$ is computed from the triple $(H_i, W_i, W^*_i)$ and the given mesh complexity $C$, using the selected error estimate. Metric tensor field $M_i$ contains information on sizes and directions of the elements of the adapted mesh we seek. This information is then used by the remesher to generate a new adapted mesh $H_{i+1}$. Then $W_i$ is interpolated on $H_{i+1}$ to obtain $(W^0)_i$. In the case of goal-oriented mesh adaptation, the adjoint state $W^*_i$ can be also interpolated on the new mesh $H_{i+1}$ to obtain $(W^{*,0})_i$ which is used as a restart for the next adjoint solution. Restart solutions are important to not waste time in the adaptive process and reuse the previous done work. This iterative process is depicted by the step 1 while loop in Algorithm 1.

The convergence criteria of step 1(f) is up to the user, it specifies when the couple mesh/solution is considered as converged in the process. In this work, for turbomachinery applications, we consider that the solution is converged at the given complexity if the mass flow, the pressure ratio and temperature ratio are not varying by a given percentage $\epsilon$ on three consecutive iterations. Here, we choose $\epsilon = 0.01$, i.e., 1%.

In the context of a mesh convergence analysis this adaptation loop has to be repeated for several increasing mesh complexities $\{C^j\}$. An efficient strategy consists in converging the couple mesh/solution for a given complexity and reuse the final mesh, solution and adjoint state to initialize the next computations at an increased mesh complexity. Such a process enables a multiscale resolution of the flow by solving large scale features on coarse adapted meshes (at the smallest complexities) and the fine scale features of the flow on fine adapted meshes (at the largest complexities). This acts like a "multigrid effect" and enables faster convergence on fine adapted meshes. This process is represented by the outer while loop in Algorithm 1. At each outer loop iteration, the complexity is increase by a factor $\alpha$. In this work, we have set $\alpha = 2$ to multiply the mesh size by a factor 2 when increasing the complexity. We have found that it is very advantageous to converge on the smallest complexities because of lot of work is done in converging the solution and these iterations are inexpensive in comparison to the largest complexities.

Algorithm 1 can be used as it in most of the cases. But, for the NASA Rotor 37 compressor, we simulate...
Algorithm 1 General mesh adaptation algorithm with mesh-convergence analysis

Initial mesh $H^0_0$, solution $W^0_0$, adjoint $W^{*,0}_0$, and complexity $C^0$

//--- Outer loop to perform the convergence study

while $C^j \leq C^{\text{max}}$ do

//--- Inner loop to converge the mesh adaptation at fixed complexity

1. while $i \leq n_{\text{adap}}$ do

(a) Compute optimal metric for the considered error estimate and complexity $\Rightarrow M^i_{j-1}$
(b) Generate new adapted mesh $\Rightarrow H^i_j$
(c) Interpolate primal and adjoint states on the new mesh $\Rightarrow (W^0)^j_i$ and $(W^{*,0})^j_i$
(d) Compute primal state $\Rightarrow W^j_i$
(e) Compute adjoint state $\Rightarrow W^{*,j}_i$
(f) if (convergence check) then

\[ i = n_{\text{adap}} + 1 \]
else

\[ i = i + 1 \]
fi

done

2. $H_{0}^{j+1} = H_{n_{\text{adap}}}^{j}$; $W_{0}^{j+1} = W_{n_{\text{adap}}}^{j}$; $(W^{*,})_{0}^{j+1} = (W^{*})_{n_{\text{adap}}}^{j}$; $C^{j+1} = \alpha \cdot C^j$

done

the whole characteristic until the compressor stall by increasing the applied outlet/inlet pressure ratio. For high mass flow rate - i.e., up to 98% of the choke mass flow - we can also use it as is. In such cases, each pressure ratio configuration can be run independently in parallel which is very attractive. However, when a rotor’s full characteristic is studied, a new difficulty occurs when we increase the pressure ratio in the flow solver boundary conditions, in other words, when we simulate low mass flow rate functioning point (below 98% of the choke mass flow). For these cases, this algorithm cannot be applied directly. Indeed, as one can see in the results section, for a given pressure ratio boundary condition prescription, the simulated operating point change with the mesh size until mesh convergence is reached. For the NASA Rotor 37 compressor, the larger the mesh size the higher the mass flow rate. We immediately see that the smaller the mesh size, the earlier the stall occurs for the pressure ratio boundary conditions. We are not thus able to run high pressure ratios on the coarser adapted meshes (lower complexity). To deal with such cases, at a given complexity, we have to start from the previous pressure ratio and compute with a higher one until stall occurs. In this case, the global algorithm is modified as described in Algorithm 2 where $\{P^k\}_k$ are the prescribed flow solver boundary conditions pressure ratio. This time each complexity can be run independently in parallel but not each pressure ratio prescription which is less efficient. The modified initialization using a previous ratio final mesh, solution and adjoint state is done at step 1(b).

A. Flow solver

Wolf is a vertex-centered (flow variables are stored at vertices of the mesh) mixed finite-volume - finite-element Navier-Stokes solver on unstructured meshes composed of triangles in 2D and tetrahedra in 3D.

The convective terms are solved by the finite-volume method on the dual mesh composed of median cells. It uses the HLLC approximate Riemann solver to compute the flux at the cell interface. Second order space accuracy is achieved through a piecewise linear interpolation based on the Monotonic Upwind Scheme for Conservation Law (MUSCL) procedure which uses a particular edge-based formulation with upwind elements. A specific low dissipation scheme is considered using a combination of centered (edge gradient)
Algorithm 2 General mesh adaptation algorithm with mesh convergence analysis and pressure ratio study

//--- Outer loop to perform the convergence study
while $C^j \leq C_{\text{max}}$ do

Initial mesh $H^{j,0}$, solution $W^{j,0}$, adjoint $W^*_{0}^{j,0}$, and pressure ratio $P^0$; set complexity $C^j$

//--- Second loop to compute all the pressure ratio

1. while $P^k \leq P_{\text{max}}$ do

   //--- Inner loop to converge the mesh adaptation at fixed complexity
   (a) while $i \leq n_{\text{adap}}$ do

      i. Compute optimal metric for the considered error estimate and complexity $\Rightarrow \mathcal{M}^{j,k}_{i-1}$
      ii. Generate new adapted mesh $\Rightarrow H^{j,k}_i$
      iii. Interpolate primal and adjoint states on the new mesh $\Rightarrow (W^{0})^{j,k}_i$ and $(W^*)^{j,k}_i$
      iv. Compute primal state $\Rightarrow W^{j,k}_i$
      v. Compute adjoint state $\Rightarrow W^*_{i}^{j,k}$
      vi. if (convergence check) then

         $i = n_{\text{adap}} + 1$
      else
         $i = i + 1$
      fi
      done
   (b) if (stall occurs) then
      break
   else
      $H^{j,k+1}_0 = H^{j,k}_{n_{\text{adap}}}$; $W^0_i^{j,k+1} = W^{j,k}_{n_{\text{adap}}}$; $(W^*)^0_i^{j,k+1} = (W^*)^{j,k}_{n_{\text{adap}}}$; set $P^{k+1}$
   fi
   done

2. $C^{j+1} = \alpha \cdot C^j$

done

and upwind gradients (element gradient). A dedicated slope limiter is employed to damp or eliminate spurious oscillations that may occur in the vicinity of discontinuities. The viscous terms are solved by the $P^1$ Galerkin finite element method (FEM) which provides second order accuracy.

The implicit temporal discretization considers the backward Euler time-integration scheme. At each time step, the linear system of equations is approximately solved using a Symmetric Gauss-Seidel (SGS) implicit solver and local time stepping to accelerate the convergence to steady state. A Newton method based on the SGS relaxation is very attractive because it uses an edge-based data structure which can be efficiently parallelized. From our experience, we have made the following - crucial - choices to solve the compressible Navier-Stokes equations.

- the SGS relaxation iterates until the residual of the linear system is reduced by two orders of magnitude
- the Breadth-first search renumbering proves to be the more effective for the convergence of the implicit method and the overall efficiency
- we found very advantageous to fully differentiate the HLLC approximate Riemann solver and the FEM viscous terms
• to achieve high efficiency, automation and robustness in the resolution of the non-linear system of algebraic equations to steady-state, it is mandatory to have a clever strategy to specify the time step.

For the turbulence model, the negative Spalart-Allmaras (SA-neg) is loosely-coupled to the mean-flow equations, where the mean-flow and turbulence model equations are relaxed in an alternating sequence. The flow solver Wolf is thoroughly detailed in 3,26 with all the associated bibliography.

As regards the adjoint state computation, needed for goal-oriented error estimates, the matrix of the flow solver is exactly differentiated. To solve the adjoint system, we use a restarted GMRES preconditioned with LUSGS relaxation. Note that, it is important to solve the adjoint linear system to machine precision to obtain an accurate adjoint state for mesh adaptation.

B. RANS error estimates

In the mesh adaptation process, the metric field \( (M(x))_{x \in \Omega} \) used to prescribe the new adapted mesh \( \mathcal{H} \) is automatically deduced from the actual solution with different error estimates.\(^4\) The goal is to find the optimal mesh \( \mathcal{H}_{Opt} \) which minimizes the given error model \( E(\mathcal{H}) \) for a fixed number of elements \( C(\mathcal{H}_{Opt}) = N \):

\[
\mathcal{H}_{Opt} = \arg\min_{C(\mathcal{H})=N} E(\mathcal{H}).
\]

This problem can be analytically solved by recasting it in the continuous mesh theoretical framework 20,21,22,23 the goal is to find the optimal continuous mesh \( M_{Opt} \) which minimizes the given continuous error model \( E \) for a fixed continuous mesh complexity \( C(M) = N \):

\[
M_{Opt} = \arg\min_{C(M)=N} E(M).
\]

The continuous mesh complexity is the continuous counterpart of the discrete mesh size (number of points or elements) and is used to prescribed the mesh size during the adaptation process. It is given by relation 4,19

\[
C(M) = \int_{\Omega} \sqrt{\det M} \, d\Omega.
\]

There is a direct relationship between the prescribed metric complexity and the number of elements of the generated mesh. If a unit mesh \( \mathcal{H} = \bigcup_i K_i \) (where the \( K_i \) are the tetrahedra of mesh \( \mathcal{H} \)) is generated with respect to \( (M(x))_{x \in \Omega} \), then the continuous mesh complexity and the mesh size are linked by:

\[
C(M) \approx \sum_K \sqrt{\det M_K} |K| \approx \sum_K \frac{\sqrt{2}}{12} = \frac{\sqrt{2}}{12} \times nt,
\]

where \( |K| \) is the volume of \( K \), \( M_K \) is the average metric at element \( K \), and \( nt \) is the number of elements of the mesh. In this work, we only consider feature-based error estimates based on a control of the interpolation error in \( L^p \)-norm.

FEATURE-BASED MESH ADAPTATION. The most natural and straightforward approach is to control the interpolation error of a sensor field \( u = f(W) \)\(^{10,13,19}\) which is defined from solution \( W \). Given a continuous sensor \( u \), it is represented by its discrete nodal values on the mesh \( u_i = u(x_i) \) and its piecewise linear representation \( \Pi_h u \) on mesh \( \mathcal{H} \). The \( L^p \)-norm of the interpolation error of the sensor field \( u \) is stated as

\[
E(\mathcal{H}) = \left( \int_{\Omega} |u - \Pi_h u|^p \right)^{1/p}.
\]

Feature-based mesh adaptation enables to control the global interpolation error of the given sensor field. Under certain assumptions, we can prove that this approach also controls the approximation error.\(^{24}\) The analytical expression of the optimal continuous mesh \( M_{Opt} \) in \( L^p \)-norm is given by:\(^3\)

\[
M_{L^p}(x) = N^{\frac{d}{2}} \left( \int_{\Omega} \det(|H_u(x)|) \frac{1}{|H_u(x)|} \, d\Omega \right)^{-\frac{1}{2}} \det(|H_u(x)|)^{-\frac{1}{|H_u(x)|}} |H_u(x)|,
\]

where \( N \) is the complexity, \( d \) is the space dimension and \( H_u \) is the Hessian of the sensor \( u \) computed using a double \( L^2 \)-projection method.\(^3,11\)
GOAL-ORIENTED MESH ADAPTATION. A goal-oriented error estimate based on an *a priori* error analysis has initially been proposed for the inviscid Euler equations in. The main idea was to translate the error on the considered output functional into a weighted interpolation error estimate. Weights are given by derivatives of the adjoint state and interpolation errors are on the Euler fluxes. As we are left with weighted interpolation errors, we can use the continuous mesh framework to obtain an analytical expression of the optimal metric field. An extension of this goal-oriented error estimate has been proposed for the laminar Navier-Stokes equations in. The main advantage of these error estimates in comparison to other goal-oriented error estimates is that the anisotropy of the mesh appears naturally. From the analysis of the behavior several error estimates for the Reynolds Averaged Navier-Stokes equations, we came up with the following new error estimate for RANS using integration by part and linearization:

$$\| J(W) - J(W_h) \|_{L^1(\Omega_h)} \approx \int_\Omega |W^*| \left| \nabla \cdot (F^E(W) - F^E(\Pi_h W)) - \nabla \cdot (F^V(W) - F^V(\Pi_h W)) \right| d\Omega$$

where $J$ is the considered output functional, $W$ is the conservative variables vector, $W^*$ the associated adjoint state, $F^E = (F_i)_i$ the convective fluxes, $F^V = (S_i)_i$ the viscous fluxes, $K_{ij}$ the viscous terms under matrix form and $H(W^*)$ the hessian of the adjoint state. The error estimate is a weighted sum of $L^1$ interpolation error on the conservative variables where the weights depend on the gradient and the hessian of the adjoint state and on the convective and viscous fluxes. Therefore, we can directly apply the continuous mesh framework to obtain an analytical expression of the optimal metric field.

C. Cavity-based adaptive remesher

Feflo.a is a generic purpose adaptive mesh generator dealing with 2D, 3D and surface mesh generation. It belongs to the class of metric-based mesh generators and aims at generating a unit mesh with respect to a prescribed metric field $\mathcal{M}$. A mesh is said to be unit when composed of almost unit-length edges and unit-volume element.

The adaptive remesher is based a combination of generalized standard operators (insertion, collapse, swap of edges and faces). The generalized operators are based on recasting the standard operators in a cavity framework. Additional modifications on the cavity allow to either favor a modification, that would have been rejected with the standard operator, or to improve the final quality by combining automatically many standard operators at once. In addition, the CPU time is also improved and becomes independent of the current modification. The unit speed is around 20,000 points inserted or removed per second on Intel i7 architecture at 2.7 GHz. For robustness purpose, both the surface and the volume mesh are adapted simultaneously, and each local modification is checked to verify that a valid mesh is obtained. For the volume, the validity consists in checking that each newly created element has a strictly positive volume. For the surface, the validity is checked by ensuring that the deviation of the geometric approximation with respect to a reference surface mesh remains within a given tolerance.

The generation of a unit mesh is decomposed into two steps:

1. Generate a unit-mesh : the mesh modification operators are used in the goal to optimize the length of the edges in $\mathcal{M}$.

2. Optimization: the mesh modification operators are used to improve the quality $Q_\mathcal{M}$.

During surface remeshing, either a P3 background surface is used or direct CAD queries are used. The CAD kernel is based on EGADS and OpenCascade.
III. Numerical results

In this section, the developed mesh adaptation process is evaluated on a realistic 3D test case, the NASA Rotor 37. This test case is particularly challenging as it is a transonic compressor with complex phenomena such as shocks, shock/boundary layer interaction and flow separation. Additionally, LES results have shown that transition occurs almost immediately at the leading edge due to the leading edge shocks, hence removing the need for RANS transition modeling. The results are obtained for different operating points and are compared to available experimental data. The full compressor characteristic has been simulated. Two operating points are of particular interest: the near peak efficiency point where mass flow is 98% of the value at choke conditions, as most experimental measurements are at this operating point, and the near-stall point where mass flow is 93% of the value at choke conditions, as it is the last stable point before surge occurs.

1. Description of the test case

The 3D turbomachinery test case selected is the transonic Rotor 37 from NASA, designed in 70’s as part of the Stage 37 and retested in the 90’s with the rotor alone. It is a configuration that is openly available and has been extensively investigated numerically in the past. It is also the reference test case for most mesh adaptation efforts for turbomachinery that are found in the literature. The full experimental annulus contains 36 blades. In this work, a single blade sector is simulated (10 degrees) with periodicity imposed in the azimuthal direction. The rotational speed is 17188.7 rpm, leading to relative tip and hub Mach numbers of 1.48 and 1.13 respectively. The experimental measurements consisted of the classic compressor performance maps, radial profiles downstream the rotor as well as Laser Doppler Anemometry.

To generate the blade CAD, instead of relying on the coordinates file available in the literature, it was chosen to regenerate it using the original blade design code employed at the time by NASA, which is openly available. The hub fillet, whose characteristics are provided in the blind test case description, is also included. This was done in an effort to have higher CAD precision, as the blade coordinates in the literature include few significant digits, leading to non-physical oscillations and curvature discontinuities on the blade surface. Such oscillations can considerably hinder the mesh adaptation, as the process will naturally detect them and refine the mesh accordingly, hence exacerbating the CAD errors. The drawback of this CAD regeneration is that the produced geometry is the cold one, while the coordinates reported in the literature are for the hot geometry (produced using a FEM simulation of the test bench). As rerunning a FEM analysis of the test bench with the new CAD is impossible, some differences with the experimental measurements are expected. It was reported, notably, in that the cold geometry is expected to have a reduced choked mass flow, as the impact of the centrifugal forces tends to untwist the blade at high radius and increase the effective flow passage.

2. Study parameters

We present the results obtained with the solution-adaptive platform on the NASA Rotor 37 cold geometry. The rotation speed is 1800 rad/s and we study a sector of 10°. We choose the following reference state to initialize the computation:

\[
\rho^\text{ref} = 1.225 \text{ kg/m}^3, \\
\|\mathbf{u}^\text{ref}\| = 400 \text{ m/s}, \\
p^\text{ref} = 101325 \text{ Pa}, \\
T^\text{ref} = 288.15 \text{ K}, \\
\mu^\text{ref} = 1.789 \times 10^{-5} \text{ kg/m/s}.
\]

For the inlet boundary condition, we impose weakly \( P^\text{in} = 101325 \text{ Pa}, \quad T^\text{in} = 288.15 \text{ K} \) using Riemann invariant. For the outlet boundary condition, we impose weakly \( P^\text{out} = \beta P^\text{in} \) using Riemann invariant.

To study the whole characteristic of this compressor rotor, we have run 16 functioning points with the following \( \beta \) values :

\[\{1, 1.025, 1.05, 1.075, 1.1, 1.125, 1.15, 1.175, 1.2, 1.225, 1.24, 1.25, 1.26, 1.27, 1.28, 1.29\}.\]
The mass flow rate is varying from the choke condition $\beta = 1$ to the near-stall condition for $\beta = 1.29$ and the peak efficiency is close to $\beta = 1.2$. For each operating point, we perform a mesh convergence study with the solution-adaptive platform. Depending on the mesh size, more operating points having be added to enhance the characteristic representation.

For the mesh adaptation, we consider the feature-based approach controlling the $L^1$ norm of the interpolation error on the local Mach number variable and the goal-oriented method focusing on the pressure ratio functional. The initial mesh is a uniform isotropic mesh composed of 158,390 vertices and 851,932 tetrahedra. The mesh convergence study is performed using 5 complexities:

$$\{160\,000, 320\,000, 640\,000, 1\,280\,000, 2\,560\,000, 5\,120\,000\},$$

leading to adapted meshes the number of vertices of which is approximatively

$$\{360\,000, 720\,000, 1\,400\,000, 2\,800\,000, 5\,400\,000, 11\,000\,000\}.$$

For each complexity, we can perform up to 20 mesh adaptation iterations. After each iteration, we analyze the convergence of debit ratio, total pressure ratio and total temperature ratio on the last 3 adapted meshes. If the variations of these 3 functionals is less than 1% on the last 3 adapted meshes, then we assume we are mesh converged for the current complexity, we break the mesh adaptation loop, and run the next mesh adaptation loop with an increased complexity. In practice, we observe that for the majority of the cases, we only perform 4 mesh adaptations iterations (mainly because we analyse the convergence on the last 3 meshes). This analysis suggests that two mesh adaptations could be sufficient, except on very coarse meshes. It is worth noting that every CFD simulation in the mesh adaptation process runs until full convergence (reduction of at least 5 orders of magnitude of the residuals and at least 10 orders of magnitude of the log-residuals).

Performing a mesh convergence study using the mesh adaptation loop for all the functioning points and both error estimates represents more than 1,000 runs.

3. Plots results analysis

The statistics for the main operating points are given in Table 1 for the feature based mesh adaptation (top) and the goal-oriented mesh adaptation (bottom). In both cases, we are giving the results obtained on adapted meshes composed of 5 millions vertices (i.e. a mesh theoretical complexity of 2,560,000). The particular operating points are the following:

- the choke point with the highest mass flow rate
- the experimental near peak efficiency point which was obtained for a mass flow rate ratio of 0.98. The mass flow rate ratio being the ratio between the mass flow rate and the choke mass flow rate.
- the experimental near stall point which was obtained for a mass flow rate ratio of 0.93
- the CFD near peak efficiency point which is given by the highest isentropic efficiency obtained
- the CFD max pressure ratio which is given by the highest pressure ratio obtained (frequently considered as the surge/stall point in RANS simulations)
- the CFD last stable point which is given by highest $P_{ratio}$ prescription for which simulations converged.

Mesh adaptation strategy. The first observation is that for the lower $P_{ratio}$ (i.e., $P_{ratio} \leq 1.150$), we were able to use the classical mesh adaptation algorithm and start from the lower complexity without any issue. But, for very low complexities (40,000 and 80,000), the simulations start to diverge (typical indication of compressor instability/surge in steady-state simulations) at a relatively low $P_{ratio}$ prescription ($P_{ratio} \approx 1.175$) thus impeding the increase of the $P_{ratio}$. This is why the mesh adaptation algorithm has been modified and higher $P_{ratio}$ prescriptions are applied, when the complexity is increased, starting from a lower $P_{ratio}$. As the goal-oriented mesh adaptation provides higher accuracy than the feature-based mesh adaptation, we notice that higher $P_{ratio}$ prescription can be achieved on coarser adapted meshes. For instance, the $P_{ratio} = 1.280$ prescription can be run with 720,000 vertices using the goal-oriented approach while meshes composed of 1,400,000 vertices are required for the feature-based one.
Mesh convergence analysis. We analyze the mesh convergence of each functional for each prescribed operating point in the flow solver. Figures 1, 2, 3 and 4 show respectively the mesh convergence of the outlet mass flow, total pressure ratio, total temperature ratio and isentropic efficiency functionals. For each figure, the left picture gives the results obtained with the feature-based mesh adaptation and the right picture with the goal-oriented approach. The top picture gives the results for the lower $P_{ratio}$ (from 1.000 to 1.175) and the bottom picture for the higher $P_{ratio}$ (from 1.200 to 1.290).

Similar conclusions arise for the outlet mass flow, the total pressure ratio and the isentropic efficiency. We observe that mesh convergence occurs quickly, with only a few hundred thousands of vertices in the mesh, for the operating points between the choke and the near peak efficiency point ($P_{ratio} \approx 1.200$). However, as we move towards higher pressure ratios, mesh convergence occurs for a higher number of vertices and at the highest $P_{ratio}$, further increasing the number of points appears necessary for full mesh convergence. In other words, the lower the $P_{ratio}$ the faster the mesh convergence. The potential reasons for this will be analyzed in a following section.

We also observe that the mesh convergence occurs earlier (early capturing) with the goal-oriented mesh adaptation than the feature-based mesh adaptation pointing out the benefits of using the adjoint to adapt the mesh. Note that, nonetheless, both approaches are converging toward the same values.

With regards to the total temperature ratio, we achieve mesh convergence for all operating points very quickly, with only half a million vertices. This suggests that this functional is easier to converge. However, it is worth reminding that for the goal-oriented approach, the objective function is the total pressure ratio. Switching to an alternative objective function that is more directly dependent on the total temperature ratio, such as the isentropic efficiency, could potentially alter this behavior.

In conclusion, we achieve mesh convergence for at least half of the operating points (lower pressure ratios) and we are almost mesh converged for remaining ones. Therefore, we are pretty confident on the obtained results as the solution-adaptive process has reduced drastically the discretization error uncertainty on the obtained numerical solutions. The predictions are the ones given purely by the considered geometric and physical models: the cold geometry and the RANS equations with the Spalart-Allmaras turbulence model.

Table 1. NASA Rotor 37. Computations statistics for the feature-based (top) and the goal-oriented (bottom) mesh adaptation runs for adapted meshes composed of 5 millions vertices. In this table, the mass flow rate is denoted MFR and the choke mass flow rate $MFR^*$. In the experiments (XP), the peak efficiency is considered for a mass flow rate ratio $MFR / MFR^*$ of 0.98 and the near stall condition for 0.93. For the numerical simulations, the near peak efficiency is considered for the highest isentropic efficiency and the near stall condition for the highest $P_{ratio}$ prescription in the flow solver.

<table>
<thead>
<tr>
<th>Feature-Based</th>
<th>Wolf $P_{ratio}$</th>
<th>MFR</th>
<th>MFR / MFR*</th>
<th>Pres. Ratio</th>
<th>Isen Eff.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Choke</td>
<td>1.000</td>
<td>20.58</td>
<td>1.000</td>
<td>1.904</td>
<td>0.861</td>
</tr>
<tr>
<td>XP Near Peak Eff.</td>
<td>1.225</td>
<td>20.17</td>
<td>0.979</td>
<td>2.099</td>
<td>0.870</td>
</tr>
<tr>
<td>XP Near Stall</td>
<td>1.296</td>
<td>19.18</td>
<td>0.930</td>
<td>2.140</td>
<td>0.850</td>
</tr>
<tr>
<td>CFD Near Peak Eff.</td>
<td>1.175</td>
<td>20.46</td>
<td>0.993</td>
<td>2.058</td>
<td>0.874</td>
</tr>
<tr>
<td>CFD Near Peak Pres. Ratio</td>
<td>1.290</td>
<td>19.36</td>
<td>0.939</td>
<td>2.141</td>
<td>0.854</td>
</tr>
<tr>
<td>CFD Near Stall</td>
<td>1.300</td>
<td>18.86</td>
<td>0.915</td>
<td>2.131</td>
<td>0.841</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Goal-Oriented</th>
<th>Wolf $P_{ratio}$</th>
<th>MFR</th>
<th>MFR / MFR*</th>
<th>Pres. Ratio</th>
<th>Isen Eff.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Choke</td>
<td>1.000</td>
<td>20.66</td>
<td>1.000</td>
<td>1.909</td>
<td>0.865</td>
</tr>
<tr>
<td>XP Near Peak Eff.</td>
<td>1.225</td>
<td>20.27</td>
<td>0.981</td>
<td>2.104</td>
<td>0.874</td>
</tr>
<tr>
<td>XP Near Stall</td>
<td>1.296</td>
<td>19.18</td>
<td>0.929</td>
<td>2.140</td>
<td>0.850</td>
</tr>
<tr>
<td>CFD Near Peak Eff.</td>
<td>1.175</td>
<td>20.54</td>
<td>0.995</td>
<td>2.062</td>
<td>0.877</td>
</tr>
<tr>
<td>CFD Near Peak Pres. Ratio</td>
<td>1.290</td>
<td>19.44</td>
<td>0.941</td>
<td>2.144</td>
<td>0.857</td>
</tr>
<tr>
<td>CFD Near Stall</td>
<td>1.305</td>
<td>18.35</td>
<td>0.888</td>
<td>2.111</td>
<td>0.827</td>
</tr>
</tbody>
</table>
Characteristics analysis. Figures 5, 6 and 7 show the total pressure ratio, isentropic efficiency and total temperature ratio characteristics. For each figure, the feature-based mesh adaptation is presented on the left-hand side and the goal-oriented mesh adaptation on the right-hand side. The top picture shows the characteristic with respect to the mass flow rate and the bottom picture the characteristic with respect to the normalized mass flow, i.e., the mass flow rate divided by the choke mass flow rate. For better characteristic representation in the near stall region, at higher mesh complexities \( C \geq 1280000 \), in addition, we have run 7 functioning points with the following \( \beta \) values:

\[ \{1.285, 1.295, 1.296, 1.297, 1.298, 1.299, 1.300\} \]

This represents a total of 23 functioning points simulated at the highest complexities.

The plots of the characteristics for all the considered mesh complexities clearly highlight that the numerical solutions are mesh converged. Indeed, we note that the curves are superimposed for high mass flow rate illustrating the early mesh convergence for low \( P_{ratio} \). For low mass flow rates, the mesh convergence is visible with the reduction of the gap between two curves when the mesh complexity increases. We are not perfectly mesh converged but we are very close.

We observe that for the considered geometric and physical models:

- the total pressure ratio characteristic is above the experimental data. As we are mesh converged, we think that this is mainly due to the fact that we are running the cold geometry instead of the hot geometry. Our results are not in accordance with the literature which generally shows curves below the data. In our opinion, this points to two potential sources of difference: they run the hot geometry and their results are potential far from being mesh converged as they consider meshes that are too coarse.

- the isentropic efficiency characteristic is below the experiment data for high mass flow rate ratio but fits with the experiments in the near stall region. This means that we don’t have the same slope. Note that the goal-oriented adaptation predicts higher efficiency than the feature-based one, and appears to fit better to the experiments. When we compare our results to the literature, we observe that the predicted isentropic efficiency is a lot lower than our results. Again, we think this is potentially due to a lack of resolution of the considered meshes and the geometry differences.

- for the total temperature ratio characteristic the same conclusions as for the total pressure ratio characteristic arise.

Radial Profiles. To further validate the predicted flow fields with mesh adaptation, the radial profiles of the total pressure and total temperature ratios downstream the rotor are plotted in Figure 8 for the two main operating points: near peak efficiency (top) and near stall (bottom). They are compared with the available experimental measurements. Only the results from the simulations with complexity 2560000 are presented as it was previously shown that at this complexity the predictions are practically mesh converged. It is readily observed that both feature-based and goal-oriented simulations have a very good qualitative agreement with the experimental results, with the curves following very accurately the form of the experimental profiles. This indicates that the principal flow phenomena, notably the secondary flows, are well predicted. The quantitative agreement is also reasonable. Some discrepancies on the levels are present but that is to be expected due to the mass flow discrepancy between the cold geometry and the hot geometry that was measured experimentally. Finally, it is well established in the literature that the cavity under the rotor, not modeled here as its actual geometry is not available, impacts the aerodynamic losses near the hub, explaining the slight over prediction observed in that area.

4. Mesh and results analysis

Near peak efficiency. We now present solutions and adapted meshes obtained in this study to point out the high-fidelity obtained thanks to anisotropic unstructured mesh adaptation. We show results for the feature-based and the goal-oriented methods on the five millions vertices adapted mesh obtained for \( P_{ratio} = 1.225 \) corresponding (see Table 1) to a mass flow rate ratio equal to 0.98 (near peak efficiency operating point). For the associated solutions, we show the relative Mach number (i.e., in the rotating frame) because it provides more information about the physics.
Figure 9 presents the adapted surface mesh and the associated solution. These figures emphasize the adaptation of the periodic surface and the numerous shock waves that are crossing this surface leading to complex mesh adaptation patterns. Periodic mesh adaptation is critical to capture properly the physics involved in this test case. Mesh adaptation that cannot handle the periodicities in this transonic case can, for example, lead to unphysical shock reflections that pollute the flow in the passage or simply to excessive smearing of the shocks. We also notice the high resolution of the surface mesh on the blade and, in particular, where the shock boundary layer interaction occurs.

The feature-based and the goal oriented mesh adaptations have a different behavior in certain areas. The feature-based adaptation puts more refinement near the inflow and in the shock regions. The goal oriented adaptation, on the other hand, focuses more on no-slip (rotating or not) surfaces.

Now, we look inside the volume. Three cut planes along the passage are shown: one close to the hubwall in Figure 10 ($y = -0.195$), one at the blade midspan in Figure 11 ($y = -0.213$) and one near the tipwall in Figure 12 ($y = -0.230$).

First, we notice the great complexity of the physics for this transonic case with many shock waves, shock wave - shock wave interactions, shock boundary layer interaction with separated boundary layer, tip vortices, wakes and so on. These rich physics are automatically captured by the mesh adaptation process leading to a very accurate numerical solution with high mesh resolution around all flow phenomena of importance. It is impossible to manually design such meshes and we can understand why manual meshing will lead to discretization errors and non mesh-converged solutions. Moreover, in the close-up views, we notice no influence of the periodic surface in the mesh adaptation showing that the periodic adaptation is working remarkably well.

The mesh adaptation of the tip clearance region is emphasized in Figure 13 where we show the refinements at different axial cuts along the blade. Like for the surface, the feature-based and the goal oriented mesh adaptations behave slightly differently with minor modifications of the different vortices, as well as the mesh density on the shocks near the tip. The anisotropy of the mesh adaptation around the vortices is, as expected, relatively limited due to allow a fine capturing of their motion. To analyze the impact of this small differences on the tip clearance structures, helicity thresholds near the tip region are plotted for the two different mesh adaptation approaches in Figure 14 (feature-based on the right and goal-oriented on the left). Positive helicity (red colored) indicates a clockwise vortex, while negative helicity (blue colored) indicates a counter clockwise vortex, thus allowing to finely determine the origin of each observed vortex. For both approaches, the flow picture is similar: near the leading edge two main vortices are generated, the tip leakage vortex (positive helicity), due to the pressure gradient across the tip, and an induced vortex that rotates the opposite way. Further downstream, the main tip leakage vortex is pushed towards the neighboring blade, while the induced vortex rolls around it. However, leakage flow continues, forming the tip separation vortices and a second induced vortex appears. Finally, towards the end of the blade and after the shock, most of the vortices have dissipated. Between the two approaches, we can see that the goal-oriented solution focuses a bit more on the induced vortices, in agreement with the mesh views in Figure 13, highlighting the importance of the secondary flows for accurate predictions of compressor performances.

One of the very interesting features of mesh adaptation is its consistency with the boundary layer physics. Figure 15 depicts the $y^+$ value on the rotating blade for four adapted mesh sizes from 360 354 to 2 820 611 vertices obtained with the feature-based mesh adaptation. We immediately notice that, on the coarsest adapted mesh, we reach $y^+ = 1$ before the shock. This is a typical meshing best practice in the industry and the mesh adaptation process appears to automatically respect it. After the shock where the boundary layer is detached, mesh adaptation sets a $y^+$ around 2. We only notice larger $y^+$ value at the corner between the trailing edge and the hub, which is normal as the corner secondary flows in the area render the $y^+$ criterion and computation unreliable. The impressive point is that the same $y^+$ before and after the shock is prescribed on the finest adapted mesh stating clearly that the mesh adaptation converge in that region and do not over refine the boundary layer. The only region where the predicted physics are changing is where vortices interact with the boundary layer near the tip passage and in the region mention above.

**Near stall operating point.** Having looked at the flow field at the near-peak efficiency operating point, we now switch our attention to the near-stall point, defined as 93% of the choked mass flow. A compressor rotor will typically go into stall either through an instability occurring near the tip (spike, rotating instabilities, ...) or through a corner separation, occurring at the junction between the hub and the blade. However, these phenomena are highly unsteady, hence predicting the exact entry of the rotor into
stall with steady-state RANS is impossible. Nonetheless, steady RANS should be able to highlight the flow areas that are the more critical to lead to stall. The objective here is to evaluate how the compressor is more prone to enter in stall and potentially draw some conclusions on why mesh convergence is more difficult for this operating point.

Starting with the tip clearance vortices, Figure 16 depicts the helicity plots at the tip clearance region for the two mesh adaptation approaches (feature-based on the right and goal-oriented on the left) and for the near-stall operating point. Comparing to the near peak efficiency point (Figure 14), it is observed that all vortices appear to become more important in size and strength, occupying a larger part of the passage. Their position, however, remains similar and there are no particular signs on the vortices of a developing instability, for example potential spillage of the tip leakage flow of the adjacent blade passage ahead of the leading edge. Furthermore, past the shock the structures appear more dissipated, similarly to the near peak-efficiency operating point.

To get more insights on the flow past the shock, an axial cut of the relative Mach number for the two approaches and the two operating points, near peak efficiency and near stall, are presented side by side in Figure 17. For both mesh adaptation methods, the conclusions are similar: between the two operating points, near the hub there are very limited differences and a similarly sized shock-induced separation is predicted. This highlights that hub corner separation is unlikely to be the factor that leads to stall. On the other hand, near the tip, we can see that at the near-stall operating point (right blade on the two images), the size of the separation has increased and an additional separation appears at higher blade heights near the tip suction side. Additionally, the low velocity area around the tip, linked to the leakage flow, expands in the azimuthal direction while a small area low momentum fluid appears also on the pressure side at approximately 90% height. These findings hint that the stall comes near the tip, potentially through a tip clearance flow spillage towards the trailing edge of the neighboring blade. The increased flow complexity and surface covered by the leakage flow implies that an increased mesh density is likely necessary to cover the area, even with mesh adaptation. Furthermore, as the flow in that area does not have strong privileged directions the algorithm has to refine this region with more isotropic elements, hence explaining the increased difficulty in getting full mesh convergence for these operating points.

IV. Conclusion

In this work, the mesh-adaptive solution strategy for periodic domain has been successfully applied to the NASA Rotor 37 case using feature-based mesh adaptation based on a control of the interpolation error in $L^4$-norm and using goal-oriented error estimates where turbomachinery output functionals are controlled. The numerical results clearly illustrate the benefits provided by the mesh adaptation:

- The process is independent of the initial mesh, thus we can start from a very coarse inviscid-like uniform mesh. This greatly simplifies the generation of the first mesh.
- The process is fully automatic and free of any human intervention in the meshing process, which is potentially a source of discretization errors.
- Early capturing of the output turbomachinery functionals is achieved in goal-oriented mesh adaptation, meaning that accurate predictions are obtained on relatively coarse adapted meshes compared to the feature-based one.
- Mesh convergence and validation against experimental measurements is demonstrated. As a result, numerical solutions should be no more polluted by discretization error. We should then able to analyze the impact of the turbulence model on the predictions and we are able to analyze the impact of technological effects on the performances (cavities, seals, squealers, fillets, ...) with a major source of uncertainty removed.

The perspectives of this work are to compare the predictions of different turbulence models and to apply the mesh-adaptive solution strategy to more complex turbomachinery geometries.

Acknowledgements

The authors want to acknowledge Adrien Loseille for providing software Feflo.a.
This work was partially supported by funding from Safran Tech with technical monitors Frédéric Feyel and Vincent Brunet.

References


Figure 1. NASA Rotor 37. Mesh convergence of the debit at the outlet for the feature-based (left) and the goal-oriented (right) mesh adaptation processes for all the imposed $P_{\text{ratio}}$ in the flow solver.
Figure 2. NASA Rotor 37. Mesh convergence of the pressure ratio during for the feature-based (left) and the goal-oriented (right) mesh adaptation processes for all the imposed $P_{\text{ratio}}$ in the flow solver.
Figure 3. NASA Rotor 37. Mesh convergence of the temperature ratio during the feature-based (left) and the goal-oriented (right) mesh adaptation processes for all the imposed $P_{ratio}$ in the flow solver.
Figure 4. NASA Rotor 37. Mesh convergence of the isentropic efficiency $\eta$ during the feature-based (left) and the goal-oriented (right) mesh adaptation processes for all the imposed $P_{\text{ratio}}$ in the flow solver.
Figure 5. NASA Rotor 37. Mesh convergence of the pressure ratio characteristic (top) and "normalized" pressure ratio characteristic (bottom) during the feature-based (left) and the goal-oriented (right) mesh adaptation processes while increasing the adaptive meshes complexity.
Figure 6. NASA Rotor 37. Mesh convergence of the isentropic efficiency characteristic (top) and "normalized" isentropic efficiency characteristic (bottom) during the feature-based (left) and the goal-oriented (right) mesh adaptation processes while increasing the adaptive meshes complexity.
Figure 7. NASA Rotor 37. Mesh convergence of the temperature ratio characteristic (top) and "normalized" temperature ratio characteristic (bottom) during the feature-based (left) and the goal-oriented (right) mesh adaptation processes while increasing the adaptive meshes complexity.
Figure 8. NASA Rotor 37. Radial profiles of the total pressure and total temperature ratio downstream of the rotor for the two main operating point: near peak efficiency (top) and near-stall (bottom). Both feature-based and goal-oriented results are for complexity $2560000$. 
Figure 9. NASA Rotor 37. Adapted surface meshes and associated solution (Mach number in rotating frame) for the feature-based (left) and the goal-oriented (right) mesh adaptation processes. Top, view of the top periodic surface. Bottom, view of the blade and the bottom periodic surface.
Figure 10. NASA Rotor 37. Adapted volume meshes and associated solution (Mach number in rotating frame) for the feature-based (left) and the goal-oriented (right) mesh adaptation processes with a cut plane close to the hubwall (plane equation $y = -0.195$). The domain is duplicated to highlight the effectiveness of the periodic adaptation. Top, global view, and bottom, zoom on the blades.
Figure 11. NASA Rotor 37. Adapted volume meshes and associated solution (Mach number in rotating frame) for the feature-based (left) and the goal-oriented (right) mesh adaptation processes with a cut plane at midspan (plane equation \( y = -0.213 \)). The domain is duplicated to highlight the effectiveness of the periodic adaptation. Top, global view, and bottom, zoom on the blades.
Figure 12. NASA Rotor 37. Adapted volume meshes and associated solution (Mach number in rotating frame) for the feature-based (left) and the goal-oriented (right) mesh adaptation processes with a cut plane close to the tipwall (plane equation $y = -0.230$). The domain is duplicated to highlight the effectiveness of the periodic adaptation. Top, global view, and bottom, zoom on the blades.
Figure 13. NASA Rotor 37. Adapted volume meshes for the feature-based (left) and the goal-oriented (right) mesh adaptation processes. Cut planes orthogonal to the blade in order to emphasize the high mesh density to capture accurately the tip vortices.
Figure 14. NASA Rotor 37. Helicity thresholds near the tip clearance region for the two mesh adaptation approaches, near-peak efficiency: feature-based (left) and goal-oriented (right). Positive (red-coloured) helicity indicates a vortex rotating clockwise while negative helicity (blue-colored) indicates a counter-clockwise vortex.

Figure 15. NASA Rotor 37. $y^+$ provided by the feature-based mesh adaptation processes for different mesh sizes.
Figure 16. NASA Rotor 37. Helicity thresholds near the tip clearance region for the two mesh adaptation approaches, near-stall: feature-based (left) and goal-oriented (right). Positive (red-coloured) helicity indicates a vortex rotating clockwise while negative helicity (blue-colored) indicates a counter-clockwise vortex.

Figure 17. NASA Rotor 37. Relative mach number at an axial cut close to the leading edge for the near peak efficiency (left blade) and near stall point (right). Feature-based (left image) and goal-oriented (right image) approaches.