Compressor integration study for a pulse detonation engine

Bayindir H. Saracoglu*
von Karman Institute for Fluid Dynamics, Rhode-Saint-Genese, Belgium

Guillermo Paniagua †
Purdue University, West Lafayette, IN

The successful implementation of detonative combustors requires specific research on turbomachinery, to ensure the integration with the large flow variations induced by the combustor. The rapid increase in pressure and temperature induced by the detonation waves implies a substantial blockage to the preceding compression system. These large amplitude fluctuations at the compressor outlet may lead to stall and eventually surge. Consequently, the operation range of the engine is significantly penalized. In order to characterize the impact of the combustor blockage on the performance of a typical radial compressor, series of unsteady numerical simulations were performed. The finite volume flow solver NUMECA Fine/Turbo with second order accuracy in space and time was used for the analysis. The impeller geometry was composed of 15 main and 15 splitter blades, delivering a pressure ratio of 4 at design conditions. The outlet of the diffuser was exposed to 5 equally spaced circumferentially distributed disturbances to mimic the detonation wave combustors. Pressure was imposed to vary 70% from the nominal outlet pressure level. The time varying chocking of the flow passages led to unsteady separation over the impeller blades and overall performance abatement in the machine. Furthermore, the detailed analysis revealed the unsteady unstarting of the supersonic passages. The influence of such pulsating disturbances, propagating upstream, resulted in major efficiency variations at high frequency.

Nomenclature

LE	=	*Leading edge*
M	=	*Mach number*
MB	=	*Main blade*
P_{01}	=	*Inlet total pressure*
P_s	=	*Static pressure*
R	=	*Mesh refinement ratio*
SB	=	*Splitter blade*
T_{01}	=	*Inlet total temperature*
TE	=	*Trailing edge*
URANS	=	*Unsteady Reynolds Averaged Navier-Stokes Simulations*
V	=	*Velocity*

I. Introduction

Pressure gain combustors offer a significant efficiency augmentation for future power generation units. In pulsed detonation combustors the fuel is consumed through a detonation wave, which results in a drastic increase of temperature and pressure, which allows theoretically higher efficiencies than in the conventional Brayton cycle. However, the flow regimes rapidly alternates between subsonic and supersonic conditions. The turbomachinery is exposed to severe pressure waves, therefore the turbomachinery integration remains an open challenge. Particularly, the compressors which may be prone to surge and stall issues. Abrupt changes in the compressor flow field may trigger partial or complete flow separation over some the airfoils. Persistent occurance of such phenomena results in stall in the compressor. However, the implications for the detonation based engines to stall and surge has not been addressed in the literature.

The current work aims to understand the detrimental effects of the detonation waves created by pulse detonation tubes on the compressor located just upstream of the combustor. The problem was numerically studied over a generic radial compressor exposed to large spatial and temporal pressure variations at compressor outlet. First, the

---

* Senior Research Engineer, Aeronautics and Aerospace Department, Chaussee de Waterloo 72, AIAA Member
† Associate Professor of Mechanical Engineering, Zucrow Labs, 500 Allison Road, AIAA Associate Fellow

baseline flow field was investigated to understand the undisturbed flow in the compressor. Then, the downstream perturbations were imposed at the outlet plane of the diffuser and the unsteady impact on the impeller and diffuser flow was reviewed.

II. Methodology

A. Solver description

FINE^{TM}/Turbo 9, finite volume solver was used to perform the numerical simulations. The three dimensional Unsteady Reynolds-Averaged Navier Stokes (URANS) equations were solved with second order accuracy in space and fourth order accuracy in time. The solver has a structured multi-block approach which allows to handle complex geometries. Combining this approach with a multigrid convergence acceleration method, the code covers a wide range of flow regimes. The solver offers as well the option to simulate Quasi-Steady rotor-stator interactions. Regarding the spatial discretization, central scheme was chosen. For temporal discretization, an explicit fourth-order Runge-Kutta scheme was applied. The Spalart–Allmaras (S–A) one-equation model was used for the turbulence closure.

B. Computational domain and grid generation

The radial compressor geometry under investigation is modified version of DDA's 404-III compressor with an impeller consisting of 15 main blades (MB) and 15 splitter blades (SB) (Fig. 1-left). Blades are backward curved (50 º) and unshrouded. Tip gap existent at the impeller is 0.1524 mm at the leading edge (0.23 % of the span of the MB at the leading edge) and 0.2032 mm at the trailing edge (1.19 % of the span of the blades at the trailing edge (Fig. 1-right). The diffuser had 24 two-dimensional wedge vanes, which were then removed, with the leading edge located at a radius of 0.147 m relative to the outlet plane. The inlet height of the impeller is 0.21 m and the exit radius of the diffuser is 0.325 m. The diffuser dumps directly into a 90 degrees annular bend.

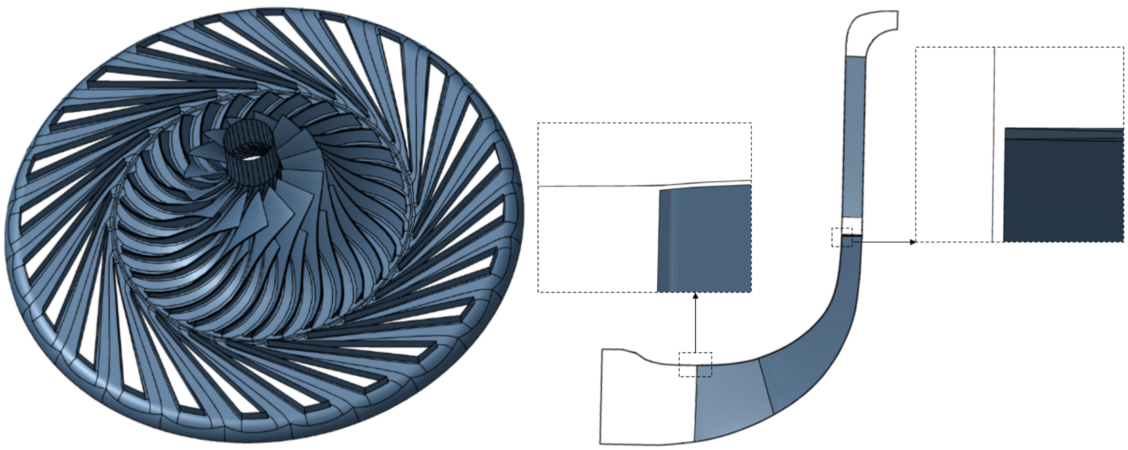

Figure 1. Vaned compressor geometry (left), meridional view of the compressor with the tip gap (right)

The modified compressor geometry excluded the vanes in the diffuser space and the diffuser hub was free to rotate together with the impeller. The annular bend at the end of the diffuser was eliminated since the direction of the outlet flow was imposed to be radially outwards. The final geometry under investigation is depicted in Fig. 2.

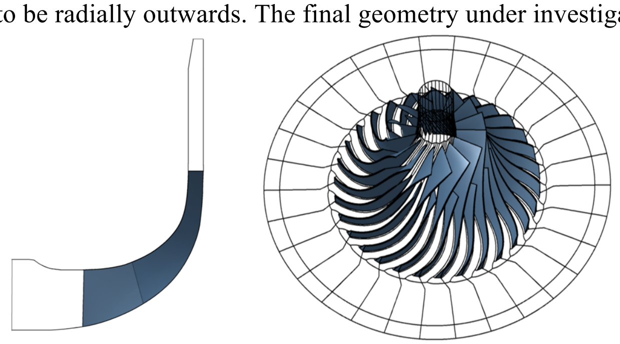

Figure 2. Final vaneless geometry with radial outlet

C. Grid sensitivity analysis

A grid independency study was carried out in order to achieve a computational domain of highest mesh quality and accuracy with considerable run time for 3D simulations. A single blade passage, consisting of a main blade and a splitter blade, was selected and mesh refinement was gradually applied in the domain, with a particular focus on the leading and the trailing edges of the blades (Fig. 3). The procedure outlined by Celik et al. was folloed for the grid convergence study. Four meshes with different grid densities were created: N1 with 4.453.456 structured elements, N2 with 3.546.256, N3 with 2.509.400 and N4 with 617.616.

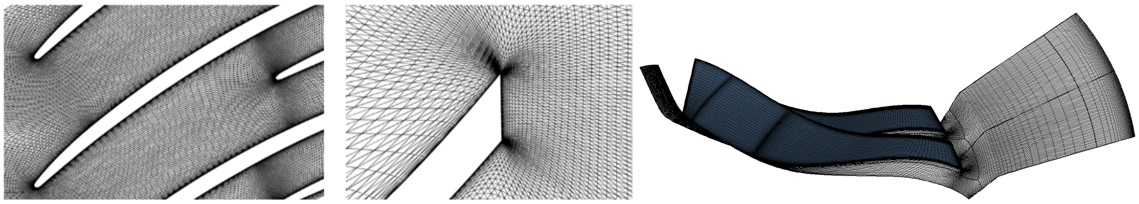

Figure 3. Blade to blade view of the mesh at midspan (left), trailing edge of a blade (center), final vaneless geometry with radial outlet (right)

Calculations were made at the nominal rotational speed (21,789 rpm) of the impeller with atmospheric inlet conditions. The simulations were run to converge until the average residuals reduce 6 orders of magnitude. The absolute total pressure at the outlet was chosen as the figure of merit for this analysis. Results of the study are summarized in Table 1.

Table 1: Grid convergence analysis

Grid	Number of Elements	ϕ (Pa)	R	ε	Error	CPU hours
N1	4453456	416409				133:17:04
			1,2558	716	0,17 %	
N2	3546256	415693				88:50:40
			1,4132	2012	0,48 %	
N3	2509400	413681				72:13:04
			4,063	11652	2,82 %	
N4	617616	402029				44:28:00

The computational grid that represented the best performance in terms of accuracy and the computational time was N2. Henceforth, it was chosen to construct the final numerical domain for further studies. The chosen grid had high boundary layer mesh quality with y^+ value mainly lower than 1. The y^+ distribution around the impeller blades is depicted in Fig. 4. The final computational domain was constructed from 6 flow passages between 3 MB and 3 SB as depicted in Fig. 5-right and consists of more than 10 million structured elements.

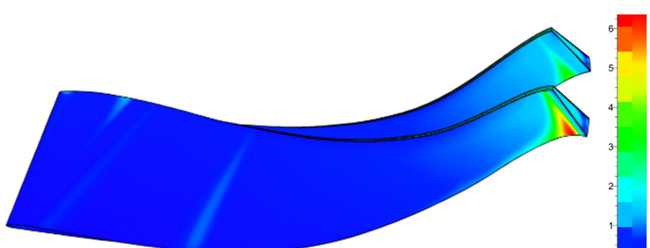

Figure 4. View of the y+ distribution

D. Outlet distortions

The detonation waves that initiated at the outlet of the compressor induce abrupt pressure distortions towards the compressor. In order to mimic the impact of back pressure disruptions on the compressor flow field, unsteady static pressure variations were imposed at five equally spaced, circumferentially distributed locations on the compressor outlet (Fig. 5). Since the effects of the distortions were going to be equally observed at every 72 degrees section of the outlet, the computational domain was simplified to the one fifth of the complete annular section to optimize the computational burden. Three complete flow passages (3 main blades and 3 splitter blades) were simulated with a single outlet distortion region covering half of the outlet as depicted in Fig. 5-right to take into account the effects emanated to the neighboring flow passages. The other half of the outlet (marked with blue in Fig. 5-right) were kept at the constant back pressure (P_s = 278046 Pa). Whereas a temporal variation of pressure of the detonation region were imposed to have pressure augmentation of 2 bars at the peak. The frequency of the distortion was fixed at 80 Hz. Hence the period of the distortion is 0.0125 s (Fig. 6).

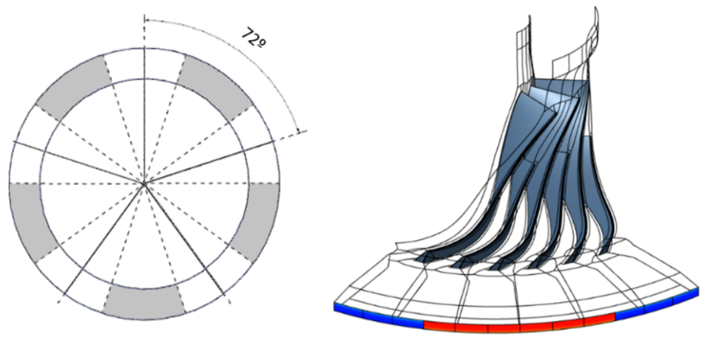

Figure 5. Circumferential position of outlet distortions (left). Computational domain divided in zones (right)

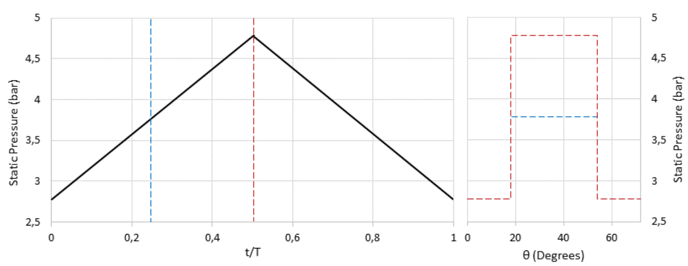

Figure 6. Imposed pressure distortion at the outlet in time (left) and in space (right)

E. Operating and boundary conditions

The atmospheric total quantities were imposed at the inlet of the compressor for all simulations (P_{01}=101353 Pa and T_{01}=288.16 K) with the flow entering to axially (V_R = 0 V_θ=0). The working fluid was assumed to be air with real gas properties and both C_P and γ are defined by means of a 5^{th} order polynomial function of the temperature. The gas constant R is set to be 287 J/kgK.

III. Results

A. Compressor characterization without detonation

The compressor with vaneless diffuser was first investigated without the downstream distortions at design and off-design conditions at fixed rotational speed and various mass-flow rates (Fig. 7). The mass flow through the compressor was varied through the surge line and choke conditions until no convergence in the numerical simulations was obtained. The design point of the compressor (π=4.72) was considered for the baseline flow field analysis.

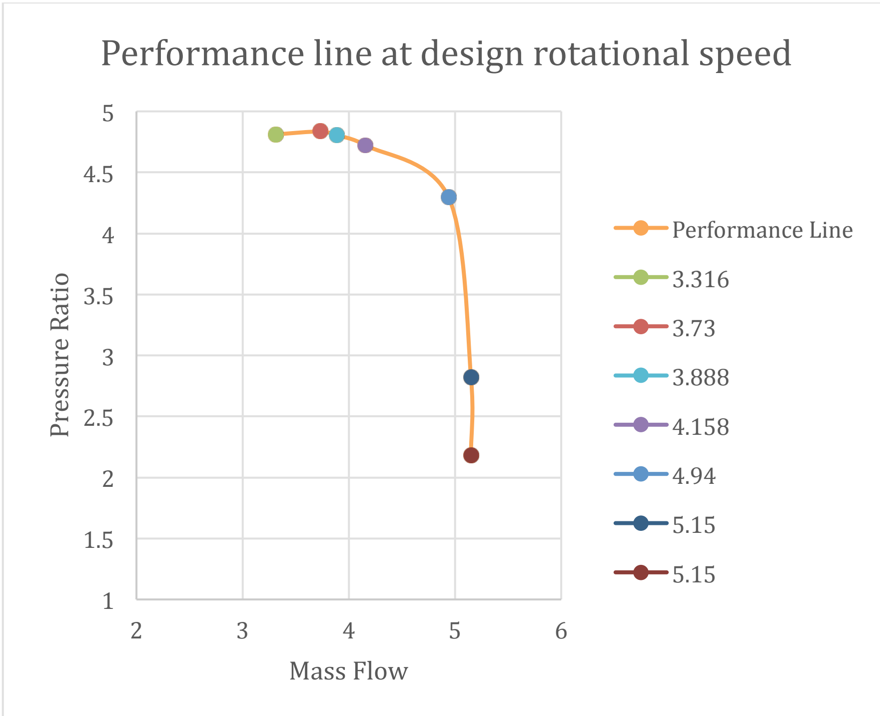

Figure 7. Performance of the compressor with uniform outlet pressure

B. Flow field charactristics of undisturb outlet condition

The flow field of the vanless compressor at the design conditions without outlet distortions were investigated in detail. The incoming stream at the atmospheric conditions shows a gradual increase of pressure throughout the impeller flow passages and the diffuser and eventually increases its static pressure up to 378 kPa (Fig. 8). Fig. 9 shows that the flow entering to the main blade passage exhibits a rapid accelatation just after the leading edge. However, suddenly after the localized acceleration, the flow remains majorly high subsonic in the relative frame of reference of the impeller. Inside the splitter blade passages, the flow starts decelerate, gradually and slow down to low subsonic velocities around 0.3 towards the trailing edge. Finally, the flow reaccelerates while exiting the impeller.

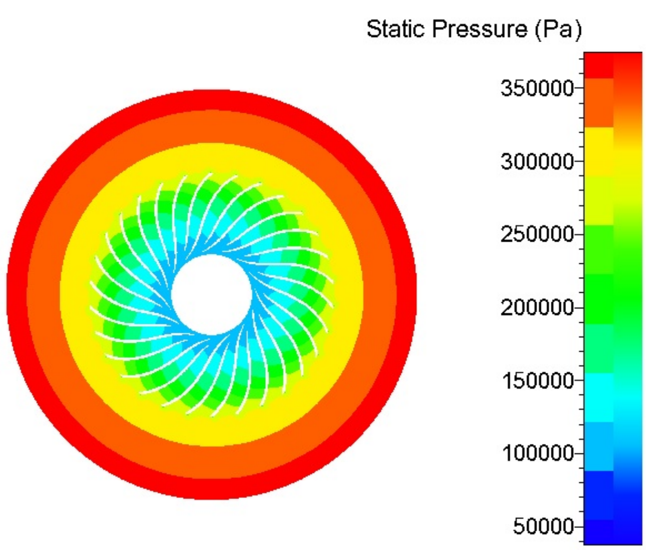

Figure 8. Static pressure augmentation throughout the compressor

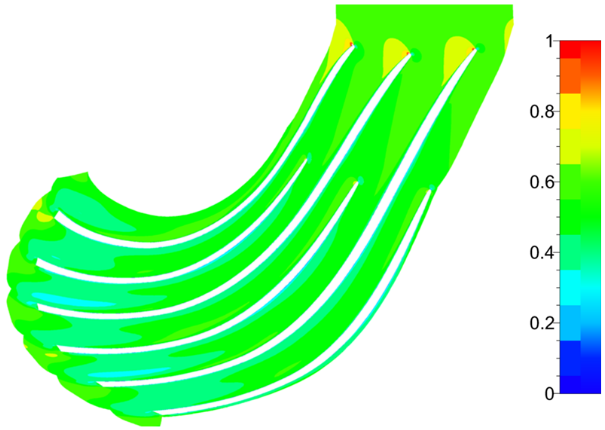

Figure 9. Relative Mach number distribution at the midspan of the impeller with uniform outlet pressure

C. Upstream effects of detontation waves

The static and total pressure field through the diffuser midspan is depicted in Fig. 10. The effects of the periodic pressure distortions coming from the diffuser outlet plane is quite visible in both pressure fields. A large, low pressure zone, circumferentially distributed, rotates in the direction of compressor spin and changes its size with varying amplitude of the outlet pressure variation. The largest pressure deficit is observed at the end of the first quarter of the outlet distortion cycle and gradually decays in the downstroke of the blockage. The extend of the pressure abatement propages towards the impeller. Fig. 11 shows the relative Mach number distribution close to the shroud of the blades (90% of the blade span). A strong deceleration is first felt towards the outlet of the flow passages at the beginning of the cycle. When the increase in the outlet pressure reaches 32% of the maximum pressure rise (Fig. 11-b), the flow deceleration dominates all splitter blade passages towards the shroud. A detached shock wave is depicted just upstream of the splitter blade leading edge (enlareged in Fig. 12) and increases its strength while the back pressure increases. It completely covers main blade passage while the compressor chokes. Furthermore, the low momentum flow extends over the most of the shroud region when the maximum detonation pressure is reached. Meanwhile, the effect is partially felt over the lower radii of the impeller, i.e. through 1 MB and 1 SB flow passages our of 6 simulated confinements. Finally, the upstream effect of the detonation chambers recesses at the end of the cycle while an oblique shoch wave originating from the MB leading edge impacts over the neighboring main blade surface.

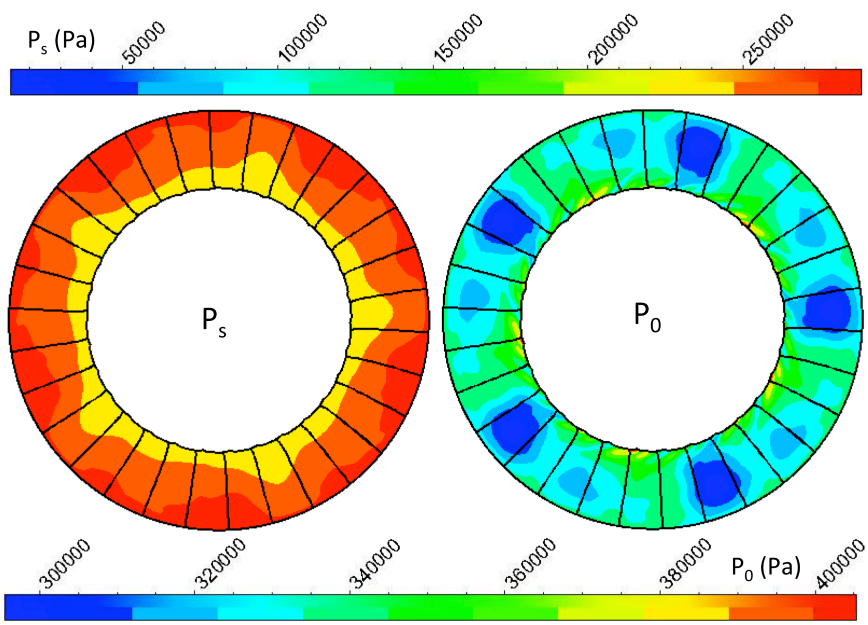

Figure 10. Static and total pressure variation in diffuser with outlet pressure distortion

During the detonation cycle the total-to-total compressor efficiency was recorded to be varying 5% and mass flow fluctuations reach almost as high as 2 kg/s.

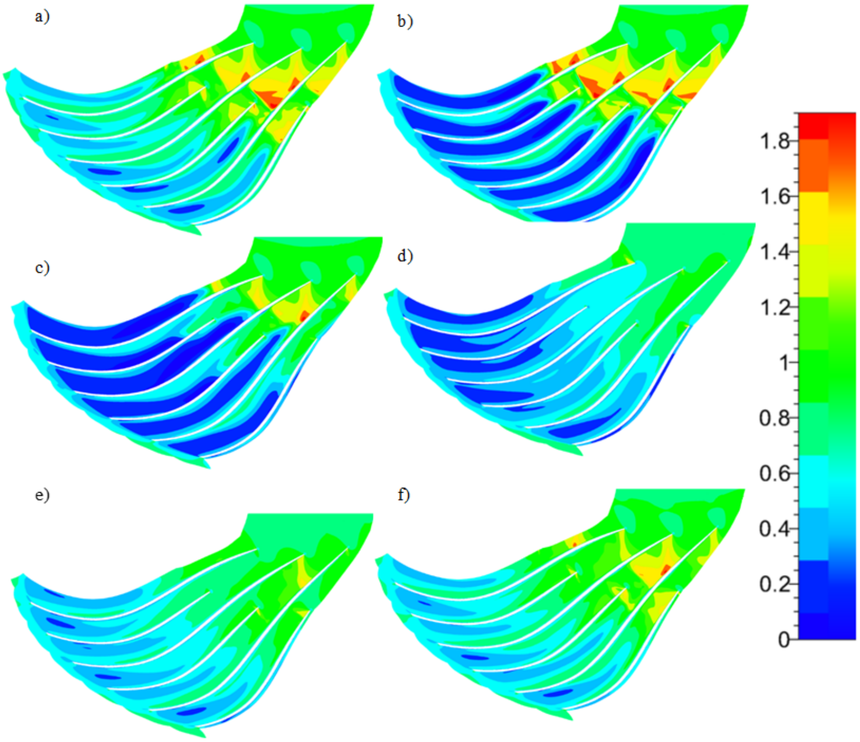

Figure 11. Relative Mach number (shroud) at different time steps: a) t/T = 0.01, b) t/T = 0.16, c) t/T = 0.26, d) t/T = 0.56, e) t/T = 0.82, f) t/T = 0.94

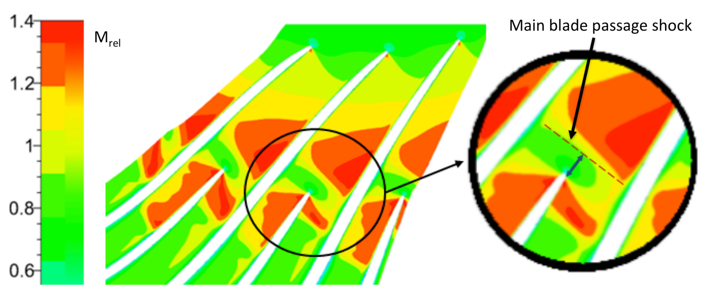

Figure 12. Close-up view of the detached shock wave in the main blade passage

IV. Conclusion

A series of unsteady numerical simulations were performed to understand the effects of the detonation waves over the upstream turbomachinery. The characterization of a generic compressor with a vaneless diffuser was accomplished to draw the performance curve at design speed. The base line flow field was investigated to better analyse the disturbed conditions. In order to simulate the detonation chambers' upsteam effects, five circumferentially distributed zones of unstedy varying back pressure were imposed at the outlet of the compressor. A static pressure variation 70% of the nominal outlet level was impletemented. The preliminary results showed strong unsteady changes in both diffuser and impeller flow path. The compressor is observed to periodically run into the choke conditions while a large pressure variation is in charge at the partial outlet zones. Consequently, a significant efficiency deduction was detected from the base line case while the variation of the efficiency may reach 5% in amplitude.

Acknowledgement

This work was performed within the Tangential Impulse Detonation Engines (TIDE) project supported by the EU within the 7th Framework Program, under the grant agreement number 335091. Consequently the authors would like to express their gratitude to the European Commision for the financial support during the execution of this research. The authors would also like to acknowledge the support of Ms. Nuria Minguez.