**Proceedings of ICFDP9:**
**Ninth International Congress of Fluid Dynamics & Propulsion**
**December 18-21, 2008, Alexandria, Egypt**

# ICFDP9-EG-235

## CENTRIFUGAL COMPRESSOR IMPELLER OPTIMIZATION USING GENETIC ALGORITHM

**Wahba W.** Lecturer of mechanical engineering

**Elnashar A.** Lecturer of mechanical engineering

**Abdelrahman M.** Prof. of Aeronautical Engineering, Cairo University

### ABSTRACT

A design optimization tool for centrifugal compressor impellers was developed. The tool was applied to a radial bladed centrifugal compressor impeller available in the open literature. This selected test case is characterized by three-dimensional viscous turbulent flow structure. The optimization target was to maximize the total-to-total adiabatic efficiency, and pressure ratio of the impeller at the design point, considering constant mass flow rate, rotational speed, and nearly constant torque. The optimizer was authorized to change the geometrical dimensions within 20% of their original values in the hub-to-shroud plane, where in the blade-to-blade plane, the authority of changing blade angles were within 100% of their original values. The optimization tool was applied to both blade-to-blade and hub-to-shroud planes individually, then to all of them together. The aerodynamic analysis was performed using (CFX BladeGen) commercial software. This software solves the three-dimensional turbulent Navier-Stokes flow equations, with zero-equation turbulence model using finite volume method. A software validation was made by comparing the computed results with an ASME published experimental data. In that experimental work, yaw probes distributed along the impeller channel were used to determine total and static pressures for hub, shroud, and blade surfaces. GAlib software was used to apply the genetic algorithm for handling of the optimization problem. The optimal impeller configuration that corresponds to maximum efficiency, and maximum pressure ratio, keeping the same mass flow rate and rpm was obtained. An efficiency improvement of 8.872% and pressure ratio increase of 0.237% were achieved at only 0.378% violation of the original torque value. Comparison between original and optimized impellers was made, which revealed the causes for efficiency and pressure ratio improvements.

### KEYWORDS:

Centrifugal Compressor Impellers, Multi-Objective Optimization, Genetic Algorithm, Computational Fluid Dynamics.

## Nomenclature

**Latin**

D = diameter
m = mass flow rate
m = meridional
r = Radius, Polar-coordinate in radial direction

**Greek**

α = absolute flow angle
β = relative flow angle
$\eta_i$ = impeller efficiency
$\varphi = \frac{C_{2r}}{U_2}$ (Flow coefficient)

**Subscripts**

0 = upstream conditions
1 = compressor inlet
2 = compressor outlet
cal = calculated
des = design value
exp = experimental
h = hub
o = total conditions
p = polytropic
s = static, shroud
θ = peripheral direction
ω = angular speed

**Abbreviations**

2-D = two-dimensional
3-D = three-dimensional

GIL = grid Inflation Layer
GRF = grid refinement factor
CFD = computational fluid dynamics
LE = leading edge
TE = trailing edge
Torq = Impeller total torque

## INTRODUCTION

Centrifugal compressors have wide military and civilian applications. They are being used in the industrial field for pharmacological, nutrition industry, and gas liquefaction. In addition, many turbomachinery applications are utilizing them, especially the small gas turbines. Finally, natural gas transmission lines are using them to overcome the pressure drop through the lines. Raising the compressor efficiency has a direct impact on the economy of such applications. Designing an efficient compression system requires high skills, whereas modifying the design of an operating compression system requires more skills and efforts. Modifying an operating compression system design can achieve the required objectives with consideration of many constraints, like maintaining the same mass flow rate, angular speed, and nearly the same torque values (to not affect the turbine operation). In addition, the modified design should be kept far enough from the surge limit. Even small value of efficiency improvement means a lot saving. Due to high cost of the experimental work needed for compressor development, the researchers are being pushed towards implementing software codes for optimization (besides the experimental work), that is to shrink the required experimental work as possible.

The nature of the flow inside the centrifugal compressor is very complex. It is a three-dimensional, viscous, and turbulent flow. In addition, many secondary flows and violent velocity gradients are present (especially the jet-wake pattern often exists at its exit). The previous mentioned points represent a challenge to the designer of such part. They explain the difficulties associated with its design process, especially when minimizing losses is aimed. Usually, the design process starts by using the simple basic fluid equations to layout the basic dimensions. The design process continued using 2-D and 3-D computational fluid dynamics solvers, to determine the losses accurately. During the last decades, the design of the centrifugal compressor has been improved using empirical, theoretical, and experimental approaches. Today, CFD plays an essential role in the design process of the centrifugal compressors.

Bonaiuti. et al. [9] classified the optimization methods into three main techniques. First technique is the gradient-based optimization; second one is the exploratory-based optimization; last technique is the function-approximation optimization. Every technique has its advantages and disadvantages, which determines its suitableness in certain applications. The choice of best suitable technique is based on the nature of problem under investigation. The gradient-based technique is suitable for functions characterized by convex and continuous domain. However, it tends to stuck to the nearest local optimum, which is considered its main disadvantage. Exploratory technique, including genetic algorithm and simulated annealing, can deal with multi-peak problems. It searches the whole space under investigation, and moving towards the optimum solution using statistical data achieved from every step during the domain searching. In this technique, evolution is fulfilled as a result of statistical data exploitation. However, this technique needs a big number of iterations until reaching the optimum solution. The third technique (such as neural-network and design of experiments) is based on correlating the inputs to the objective function. These methods use certain approximated functions designed especially for the required correlation. The main draw back of this technique is the deviation arising from using the approximated functions. The main advantage of this method is its relatively short time needed for achieving the solution.

Lots of work has been devoted for optimization of axial and centrifugal compressor's blades using direct and indirect design methods. Yiu et al. [11], defined the indirect method as a method in which velocity and pressure distributions are first addressed, and their impact on the performance is investigated. The design process is then completed by determining the blade geometry corresponds to the prescribed distribution, using correlations between flow field distributions and the blade geometries. A method for 3-D automatic optimization of turbomachinery blades, using the indirect method is proposed in [11]. In that method, two objective functions defined by entropy loss and aerodynamic blockage were examined. On the other hand, direct method starts by building the blade geometry at first, and then examining its effect on the objectives. The process is repeated until the required objectives are achieved. Wahba [10], Used single-objective and multi-objective genetic algorithm coupled with a 2-D fluid flow solver for optimization of centrifugal impeller pump blades. That author used sixteen geometric parameters to describe the blade shape. The population size and the number of generations used were thirty and one hundred respectively. Benini. [5], proposed a method for a 3-D multi-objective design optimization of a transonic compressor. That author parameterized the rotor using fourteen parameters for camber line and nine parameters for thickness distribution. Thus, a total of twenty-three parameters were used to describe the rotor. A 3-D viscous CFD solver coupled with, an evolutionary algorithm was used. A population size of twenty individuals, and one hundred generations were used to accomplish that optimization process. Two thousands run time hours (using a four-processor Alpha Server ES40) were consumed to achieve the results. That author has made optimization for achieving maximum pressure ratio, and the maximum efficiency. Summers [12], Proposed a method for optimizing centrifugal compressor train with recycle control. Lian. [3] performed Multi-objective optimization on the NASA rotor-67 transonic compressor blade (axial compressor stage). The objectives were to maximize the stage pressure ratio, as well as to minimize the compressor weight. Pazzi [6], proposed a method for optimizing the centrifugal compressor impellers. That author used an automatic searching technique, coupled to an in-house CFD fluid flow solver, for optimization of a low flow coefficient and cylindrical blades impeller. The impeller was described using twenty-five geometrical parameters. The optimization goal was to maximize the polytropic efficiency, with more or less constant flow coefficient and polytropic head. The optimization was carried out on a cluster consists of eight PCs.

The objective of the present work is to provide a fuel saving mean, through increasing of centrifugal type compressor's pressure ratio, and improvement of their efficiency. Saving fuel is the imputes, which has a direct impact on the economy, and indirect impact on the army forces capabilities (Longer equipment ranges for the same fuel quantities). A software combining, 3-D viscous fluid flow solver with an optimizer is being used to accomplish this mission. This paper is organized as follows: First, summary of the influence of different geometric parameters is explained. Second, description and application of the used CFD software are presented. Third, application of the method on a test case, to improve its efficiency and pressure ratio, with some constraints is presented. The developed tool was first applied on the hub-to-shroud plane, and then on the blade-to-blade plane, finally the method was applied on both planes simultaneously. The three cases were optimized using the same cost function, and GA parameters.

## INFLUENCE OF BLADE SHAPE ON THE IMPELLER PERFORMANCE

The impeller could be described through the definition of two main groups, hub-to-shroud, and blade-to-blade plane groups, see Fig. (1). First group consists of:- 1)Blade inlet hub diameter 2)Blade inlet shroud diameter 3)Blade exit hub and shroud diameters 4)Exit blade height 5)The equation describing hub and shroud curves in the axial radial direction. These group elements affect the impeller behavior as follows: Inlet hub and shroud diameters affect the inducer inlet area, and the value of the Mach number at the shroud, which should be kept as low as possible to minimize losses in this part. The blade exit diameter affects directly the amount of work transferred to the impeller, as appeared from Euler turbomachinery equation. Therefore, it will affect the required torque to derive the compressor (for the same rpm value). The exit blade height affects the possible diffusion through the impeller, consequently the exit static pressure, and the tendency of surge. Last element in this group is the equation describing the hub and shroud blade shapes in the meridional plane. This parameter is very effective and dangerous. It affects the pressure gradient on the hub and shroud, and so the flow separation. It is impossible to manipulate this parameter using one-dimensional relations; it should be manipulated using 3-D CFD solvers, to account for the boundary layer growth and its consequences. On the other hand, second group for impeller description is the blade-to-blade plane group. It consists of the following:- 1)The spacing between blades 2)Blade inlet angle 3)Blade exit angle 4)The equation describing the mean camber line 5)Blade thickness distribution along meridional coordinate 6)Leading edge and trailing edge radii. These group elements affect the impeller behavior as follows: the first element affects the flow passage area and consequently the relative Mach number; it depends on the number of blades and the radius at this location. This element together with the second and fifth elements determines the chocking conditions. The second element (blade inlet angle) affects directly the velocity triangle at the inlet and so the inlet relative Mach number. This parameter could be easily manipulated using 1-D flow calculations, to determine an optimum angle, which corresponds to maximum mass flow rate with minimum inlet shroud relative Mach number. Whitfield [15] explained a method for optimizing this parameter using simple one-dimensional relationships. However, the incidence flow angle stills a matter of optimization to complete the choice of inlet blade angle. Finally, the exit blade angle affects directly the work of the compressor, through affecting the circumferential absolute exit velocity component. Exit blade angle effects are discussed in references [15, and 16].

The actual behavior of the impeller is affected by the combination of the previously stated geometrical parameters together, not by every one alone. That limits the ability of trusting empirical, and one dimensional basic equations result. In addition, it forces the designers towards the obligatory 3-D CFD solution, which deals with the actual geometry as one unit, combining all the previously mentioned points.

## THE FLUID FLOW SOLVER

In the present work, the impeller geometry is manipulated using the commercial software (CFX BladeGen). Whereas the flow field solution inside the impeller is performed using the commercial software (CFX BladeGenPlus version 4.1). First, the geometry is described, and then an unstructured grid is generated. Finally, boundary conditions treatment, and the complete CFD solution is performed by the software. CFX-BladeGenPlus solves the Reynolds-averaged Nervier-Stokes equations. In this package version, finite volume method is the used discretization method, and the zero-equation is the used turbulence-model. The boundary conditions treatments are as follows: - At the Inlet, total pressure, and total temperature should be assigned, in addition to the flow direction. At the exit, static pressure value should be given; otherwise mass flow rate should be assigned. Finally, the periodic boundary condition is imposed by the program, where the solution is performed around one blade of the set only. Other required parameters are rpm, surface roughness, and fluid properties. In addition, the target residual and maximum number of iterations should be addressed. The physical time step could be chosen by the user, where a value equals ($1/\omega$) is recommended, and it is the program default value.

## SENSITIVITY ANALYSIS OF CFX SOFTWARE

Before using the program in this work, the program was investigated, to determine its appropriate parameters' values to be used during this case. The following parameters were investigated:- 1) grid density in the whole domain, 2) grid density around blade solid walls, 3) the target residual. The effect of the previous variables on the impeller efficiency, exit static pressure, and exit flow coefficient were investigated.

1-Grid Density

The grid is completely specified in the CFX software using two parameters (grid refinement factor GRF, and grid inflation layer GIL). Clustering is possible around LE and TE. The grid refinement factor indicates the grid density to be used in the whole field. A number between 0.01 and 10.0 could be used. The grid inflation layer determines the level of clustering to be imposed on the grid around the blade surface only; it is used to handle the boundary layers in this area more accurately (if required).

2-Target Residual

The target residual TR is the maximum value accepted as a difference in certain parameters (like velocity components, or pressure) between any iteration and the next one. The solver automatically stops once the target residual has been met for all variables. A value of target residual equals 5E-5 or lower is recommended.

It is known that, higher values of GRF, GIL, and lower values of TR will lead to better results; but the solution will need a longer time. A sensitivity analysis was done by Elnashar et al. [1, 2] involving the preceding parameters, it results in recommending values of 7, 10, 1*10E4 for grid refinement factor, grid inflation layer, and target residual respectively. In the present work, a group of values recommended by the program and activated by a button called "Grid fine" was used. These values are GRF=2, and GIL=5. In addition, a value of 5*E-5 was used for the target residual. These values are recommended because they give reasonable accuracy with a short running time.

## APPLICATIONS OF CFX SOFTWARE

Benini. [8] compared between the flow field solution using CFX software and a published measurements for the same cases, he reported a good agreement between measured and calculated results. Khalil et al. [7] used this software for solving the flow field around an axial fan, then he compared between measured and calculated results using CFX BladeGenPlus software for the same case. Finally, that author reported a good agreement between measured and calculated values. Mizuki et al. [17, and 18], published a detailed measurements of a three centrifugal compressors (named A, B, and C) using yaw probes for measuring total and static pressure along the channel between blades. In spite of that, it is an old case, but due to its accuracy, and large data released about it, it was decided to be used as a test case. Impeller (A) was chosen for validation and optimization because it is addressed as inefficient impeller, so optimizing such a virgin case is promising. The impeller detaild geometry is given in (17, 18). This case was solved using the CFX software. A comparison between measured and calculated values was made by [1]. Measurements of [18] showed that, the area of shroud at the position of deflection from axial to radial direction had a high relative velocity. The down stream area following is characterized by low velocity. Fig. (2a) coincides with these measurements, which also insists on the accuracy and validity of the (CFX BladeGenPlus) software.

## THE OPTIMIZATION PROBLEM

The optimization procedure starts by parameterizing the geometry, then handling the parameters via an optimizer (suggesting a combination). Last step is to determine the CFD solution for the proposed geometry and feedback the optimizer by the degree of fitness of such combination. That guides the optimizer to determine the next suggested combination. In the present work, a free published library called GALib245 [13] was used as an optimizer.

### (1) Geometry Parameterization

The impeller geometry should be casted in a suitable form to facilitate its handling via the optimizer. A good parameterization means perfectly describing the impeller by the minimum possible number of parameters. The impeller parameterization in the hub-to-shroud plane is done by using a four points Bezier curve for each of the hub and shroud surfaces, each point is defined by two coordinates (z, r). The input channel points' axial coordinates have equal fixed values, which make the input channel length equals about one third of the blade length in the meridional direction. The input channel inner/outer diameter takes the same value of the hub/shroud first point diameters, which assures the inlet channel straightness. The exit channel hub/shroud radii have a fixed value equals the average impeller exit radius plus approximately one third of blade length in the meridional direction. The exit channel hub/shroud axial locations take the values of the impeller hub/shroud last points' axial coordinates. That is to say, only sixteen variables are used to define the impeller in the hub-to-shroud plane. This number of Bezier curve points should be increased in case of implementation of tangency condition as a constraint on the hub/shroud impeller surface and hub/shroud of the inlet and exit channel surfaces. The impeller parameterization in the blade-to-blade (m-Θ) plane is done through defining the blade mean line by a six points Bezier curve. The meridional coordinates of these six points are fixed at (0%, 20%, 40%, 60%, 80%, and 100%) of the meridional distance from leading to trailing edges. The theta coordinate values are the variables, that is to say we have only six new variables to define the (m-Θ) plane completely.

After parameterization the impeller it looks like a paste, where choosing the values of only twenty-two variables, in addition to the coordinates of four points for inlet and outlet (dependant variables) will immediately draw a new impeller. Then it will be the optimizer turn to manipulate these variables and determine their optimum values.

## (2) Genetic Algorithm Optimizer

Genetic Algorithm is an optimization tool, which casts the variables in groups (genes and chromosomes), then dealing with the variables through some operators (selection, crossover, and mutation). The optimization process starts by randomly combining parents (for the first generation only) from given set called population, then forming an offspring collecting fetchers from the selected parents. The process continued through the next generations (mating parents to produce offspring). However, the difference in the next generations is that, the chance of every parent to meet another partner is depending on its rank. The rank of every member of the population is determined according to the degree of matching to certain given cost function. That is to say, members of higher ranks have more opportunities to have children than the others have. That coincides with the concept of survival of the fittest. The evolution continues from generation to the other using GA operators until a termination criterion is met.

### (a) Operators

The genetic algorithm operators are selection, crossover, and mutation. Selection is the process of choosing parents for mating. It starts in a random manner in the first generation, and continued depending on the members' ranks in next generations. Crossover is a process of forming an offspring by combining a part of the first parent, with a compliment completes its length from another parent. The percentage of crossover is commonly chosen between 50% and 90%. Mutation is a change of only small percentage of the child genes (replacing zero with one and vice versa). Hessner et al. [14] suggested a value of 1% of the offspring to be mutated. Wahba [10] also used a value of 1% as a mutation rate, in a similar case of optimizing centrifugal pump impeller. The mutation operator works as a fine tuner; where high mutation values may lead to overshooting of nearby goals, very small values will make the solution converges lazily. The process is to be terminated when a predefined criterion is met. The termination criterion may be simply certain number of iterations. Other criterion may be reaching certain saturation, that is to say, when the difference between the maximum cost function value and its correspondence in the next generation is less than certain tolerance. In the present work, the number of iterations is chosen as a termination criterion.

### (b) Cost Function

The cost function is the objective of the optimization process to be met. It is used as a measure of fitness of members. If the cost function is directly the efficiency of the impeller, and no matter what will happen to the other parameters (like pressure ratio, or torque); then gaining efficiency by changing impeller geometry may be on the expense of increasing its driving torque, or losing its pressure ratio. In the present work, the cost function is build carefully so that, gaining efficiency and pressure ratio is obligatory. Meanwhile, increasing or decreasing torque is allowed, but with high penalty. This function is suitable for modifying an operating compression system, where violating original torque value should be minimum, to not spoil the turbine operation.

$$Cost = \frac{a * \Delta\eta\% * \Delta \Pr\%}{\left(\varepsilon + (\Delta Torq\%)^4\right)} \quad ------------(1)$$

The proposed cost function is:

### Where

| | |
|---|---|
| $\Delta\eta\%$ | the percentage increase/decrease in the efficiency |
| $\Delta \Pr\%$ | the percentage increase/decrease in the compression ratio |
| $\Delta Torq\%$ | the percentage increase/decrease in the torque value |
| a | variable equals (+1 or -1) |
| ε | arbitrary constant |

### Notes

- The percentage variations was used instead of the absolute values to include the attitude as a sign (either positive or negative changes)
- The fourth power of the torque variation represents the high penalty imposed on the function in case of violating the original torque value. In addition, it assures the positive value of the torque variation contribution. The positive value of the torque contribution is necessary as increasing or decreasing the torque will lead to the same result (violation of turbine operating conditions), which is not desired.
- The variable (a) was introduced to protect the function against having positive values, in case of having negative efficiency and negative pressure variations symultainously.
- The number (ε) in the dominator was introduced to avoid dividing by zero or having giant values for the cost function (in case of no torque violation). This value is working like a focusing element, where only comparative values of $\Delta Torq\%$ will be contributive to the cost function. That assures elimination of GA buzz by the very small torque variations. In the present work, a value of one was used for this focusing element.

### Application of the method

The proposed method was applied to a twelve radial bladed impeller of Mizuki [17, 18]. The optimization target was to increase its efficiency and pressure ratio, as much as possible. Constant mass flow rate and constant rpm were applied as constraints, where torque violation is allowed, but hopefully it will not exceed 1%. The used population size, number of generations, mutation rate, and crossover rate were 25, 100, 0.01, and 0.6 respectively. The machine used was AMD Athlon 5200, 2.6 GHZ Dual core, 2 M L2cash, 2G RAM. The optimization was performed three times; first, for blade-to-blade plane; second one for hub-to-shroud plane; the last optimization was performed for both planes simultaneously. The optimizer needed about 1200 run along 120 hour for each optimization process to achieve the optimum solution. A comparison between vector plotting and, contours of relative Mach number, for both original and modified impellers was shown in Figures (2, 3, 4, and 5). Comparison between variations in efficiency and pressure ratio, in addition to cost function values are shown in table (1).

## RESULTS AND DISCUSSION

Figures (2, 3, 4, and 5) revealed the causes of efficiency, and pressure-ratio improvement as a result of the optimization process. The optimizer found the best geometrical configuration that minimized the flow separation, vortices, and reduces the high Mach number gradients. In the following paragraph, the impact of optimization process on the flow structure and its consequences is discussed. These figures show comparison between $M_{rel}$ distribution and vector plotting before and after optimization of z-r plane alone, m-Θ plane alone, and both planes together. Figures 2, and 3 show comparisons at stations 25%, and 75% of the distance between periodic surfaces in the z-r plane respectively. Figures 4, and 5 show comparisons at stations 10%, and 90% of the distance between hub and shroud in the m-Θ plane respectively.

AT station 25**%** of the distance between periodic surfaces, the original impeller (Fig. 2a) shows a high relative Mach number zone at the inducer shroud inlet. Whitfield et al. [15] discussed the bad effect of this case, and described it as a big source of deficiency. Figure (2b, 2c, and 2d) show that after optimization, the inlet relative Mach number has been reduced in both z-r, and z-r-m-Θ optimizations, meanwhile it was not affected in the m-Θ optimization. In addition, Fig. (2a) shows a partially stagnant zone at the shroud outlet (the blue spot) neighbored by a relatively higher speed (at the hub side). The high deference in velocity represents the jet wake pattern, which is addressed as a main source of deficiency for both impeller and diffuser. The consequences of this problem are clearer in the vector plotting, a flow separation followed by reattachment appeared in the typical area Fig. 2a, and it represents a source of energy dissipation in non-useful work. After optimization of z-r and m-Θ planes separately, no big improvement occur, but z-r-m-Θ optimization improved the situation a lot.

At station 75% of the distance between periodic surfaces, there is no obvious problems in the flow structure of the original case (Fig. 3a). In addition, no big variations occurred between original and optimized cases figures 3b, 3c, and 3d.

At station 10% of the distance between hub and shroud, no big difference could be noticed, between the original case (Fig. 4a) and optimized cases (4b, 4c, and 4d). The only issue is the slightly high Mach number at the inducer leading edge (Fig. 4a), which is reduced in z-r and z-r-m-Θ optimizations only.

At station 90% of the distance between hub and shroud, two problems appear in the original design (Fig. 5a). The first problem is the relatively high inlet $M_{rel}$. Whereas the second problem is the relatively low $M_{rel}$ at the TE of the blade upper surface, and moderate $M_{rel}$ at the TE lower surface. This distribution represents the jet-wake pattern in this plane. A big vortex exists at the end of the upper blade surface. The flow structure is as bad as possible in the whole impeller. For the optimized geometry (figures 5b, 5c, and 5d), the situation differs, the z-r optimization improves the LE $M_{rel}$ distribution, where the z-r-m-Θ optimization improved both problems and no such pattern exists, which means an improvement in the efficiency and pressure ratio.

# CONCLUSION

A tool for optimizing centrifugal compressor impellers was developed. The tool combines GALIB245 software as a genetic algorithm optimizer, with CFX-BladeGen software as a three-dimensional viscous fluid flow solver. The optimization problem was to maximize the efficiency and pressure ratio, while keeping constant rpm, and mass flow rate. The original torque value was kept nearly constant. This treatment for the torque value is recommended when modifying impeller of an operating compression system. The method was applied to Impeller (A) of reference [17]. The results show an efficiency improvement of 8.872%, and pressure ratio increase of 0.237%, with a torque violation of only 0.378%. The small improvement in the pressure ratio is caused by the constraint on torque violation. The requirements of the paper has been satisfied, to have better efficiency, and pressure ratio, with the lowest possible torque violation. In addition, it was concluded that the contribution of the hub-to-shroud plane optimization dominates over the contribution of the blade-to-blade plane optimization for this case.

## ACKNOWLEDGMENTS

The author would like to express his appreciation to Wall, M. the author of GAlib245 software, and EAE Technology (The owner company of CFX-BladeGen software). These packages were powerful tools, without which this work would never have been accomplished.

## REFERENCES

[1] Elnashar A., Abdelrahman M. “Design Optimization Of Centrifugal Compressor Impeller” Ph. D. thesis, cairo university Jan 2008.

[2] Elnashar A., Wahba W., Abdelrahman M. “Three Dimensional Multi-Objective Design Optimization Of

Centrifugal Compressor Impellers" ASAT, 168 May 2007.
[3] Lian Y Liou M "Multi-Objective Optimization of Transonic Compressor Blade Using Evolutionary Algorithm" JOURNAL OF PROPULSION AND POWER Vol. 21, No. 6, November–December 2005
[4] Oana, M. O., Kawamoto, H. Ohtani, and Yamamoto, Y., "Approach to High-Performance Transonic Compressor Design". AIAA. journal Of Propulsion And Power Vol. 20, No. 1, January–February (2004).
[5] Benini E., "Three dimentional Multi-objective Design Optimization of a transonic compressor rotor". AIAA, jurnal of propulsion and power VOL. 20, No. 3, May-June (2004).
[6] Pazzi S., Martelli F., Energetica D., "Automatic CFD-Based Procedure For The Optimization Of A Centrifugal Impeller", ISABE-2003-1168.
[7] Khalil, M. K. , Alzahaby A."Automation of wind Tunnel Operation", Thesis of Master of Science Cairo 2003.
[8] Benini E. "Optimal Navier–Stokes Design of Compressor Impellers Using Evolutionary Computation" International Journal of Computational Fluid Dynamics, October 2003 Vol. 17(5), pp. 357–369
[9] Bonaiuti, D. Arnon, A. Ermini M, "Analysis and optimization of Transonic Centrifugal Compressor Impeller Using The Design of experiment Technique" ASME June 3-6 (2002).
[10] Wahba, W. A., and Tourlidaks, A.,"A Genetic Algorithm to Design a Blade Profiles for Centrifugal Pump Impellers", AIAA Paper 2001-2582 (2001).
[11] Yiu K.F.C. and Zangeneh,M. "Three Dimensional Automatic Optimization Method For Turbomachenery Blade Design", AIAA, journal of propulsion and power Vol 16 No.6 November-December (2000).
[12] Summers, S. M., "Developing Centrifugal Compressor Train Optimization Models for performance Evaluation", ASME, 322/Vol. 120. April (1998).
[13] Wall M., "GAlib: A C++ Library of Genetic Algorithm Components", version 2.4, Massachusetts Institute of Technology, USA, (1996)
[14] Hessner J. and Männer R., "Choosing Optimal Mutation Rates", in Proceedings of the First Workshop on Parallel Problem Solving from Nature (Lecture Notes in Computer Science, Vol. 496); P. Schwefel and R. Männer (Eds.); Springer- Verlag: Berlin, 1991, pp 23-31, (1991).
[15] Whitfield, A., Banes N.C., "Design of radial turbomachines", Longman Group UK limited (1990).
[16] Ecardt D. ,"Flow Field Analysis of Radial and Back swept Centrifugal Compressor Impellers Part 1: Flow Mesurements Using a Laser Velocimeter" symposium on performance Predection of Centrifugal Pumps & Compressors, (1980).
[17] Mizuki S., Ariga I., Watanabe I., "A Study of Flow Mechanizm within Centrifugal Impeller Channel" ASME T4-GT-14, December (1975).
[18] Mizuki S., Ariga I., Watanabe I., "Investigations Concerning the Blade Loading of Centrifugal Impellers" ASME T4-GT-143, December 1974.

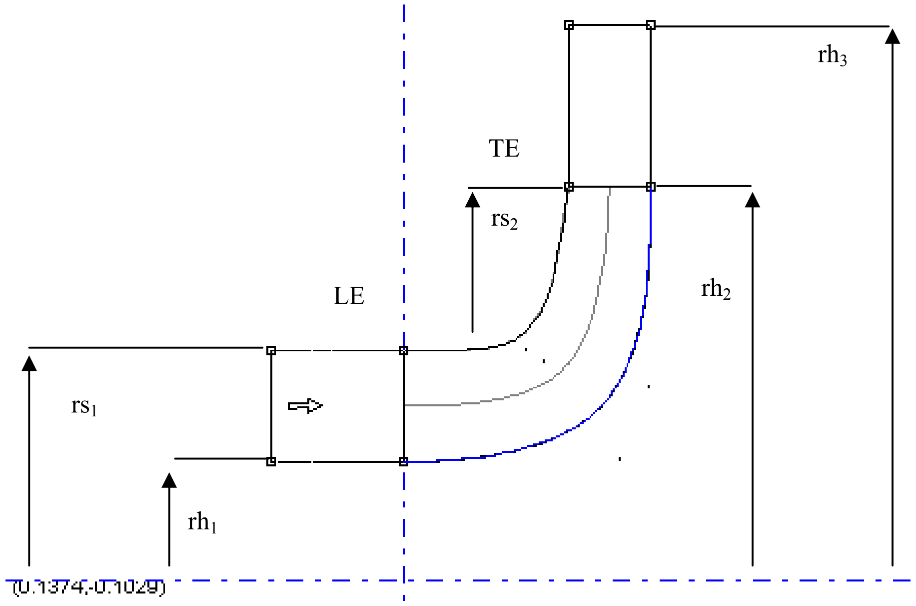



Figure (1a), Hub-to-shroud plane group for description of the centrifugal compressor impeller geometry

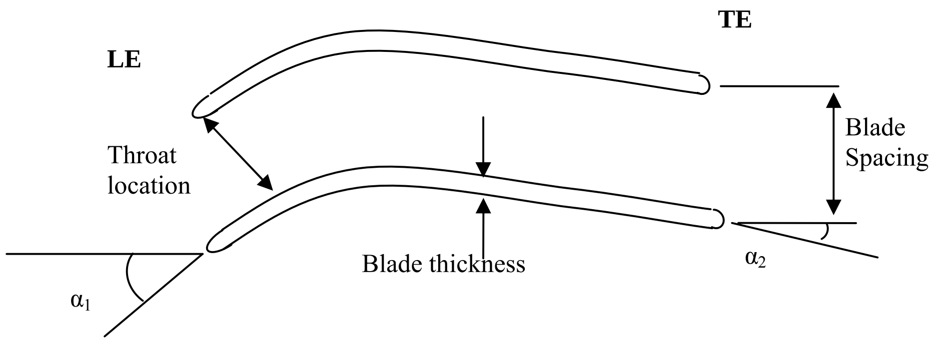
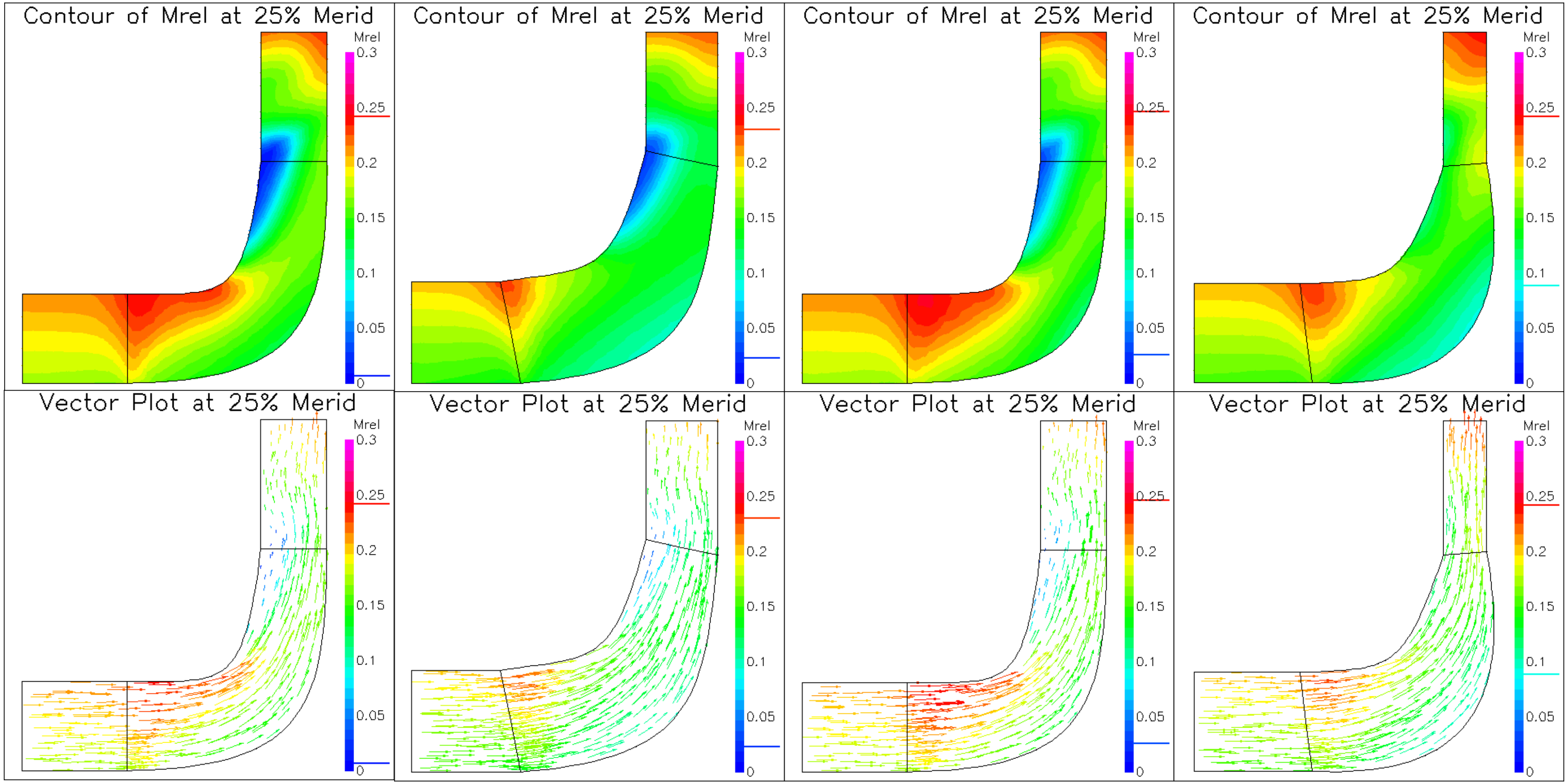
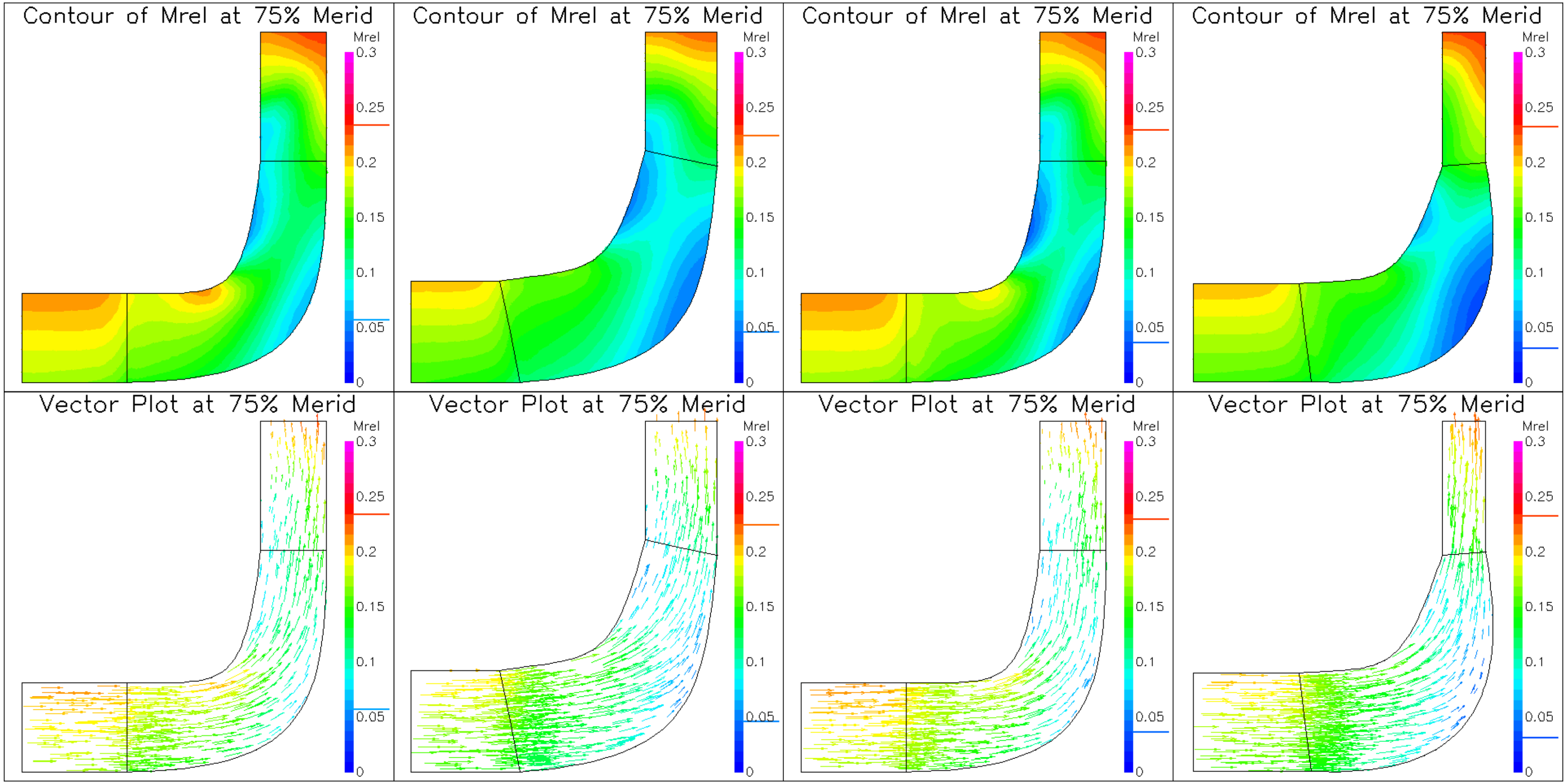



Figure (1b) Blade-to-blade plane group for description of the centrifugal compressor impeller geometry



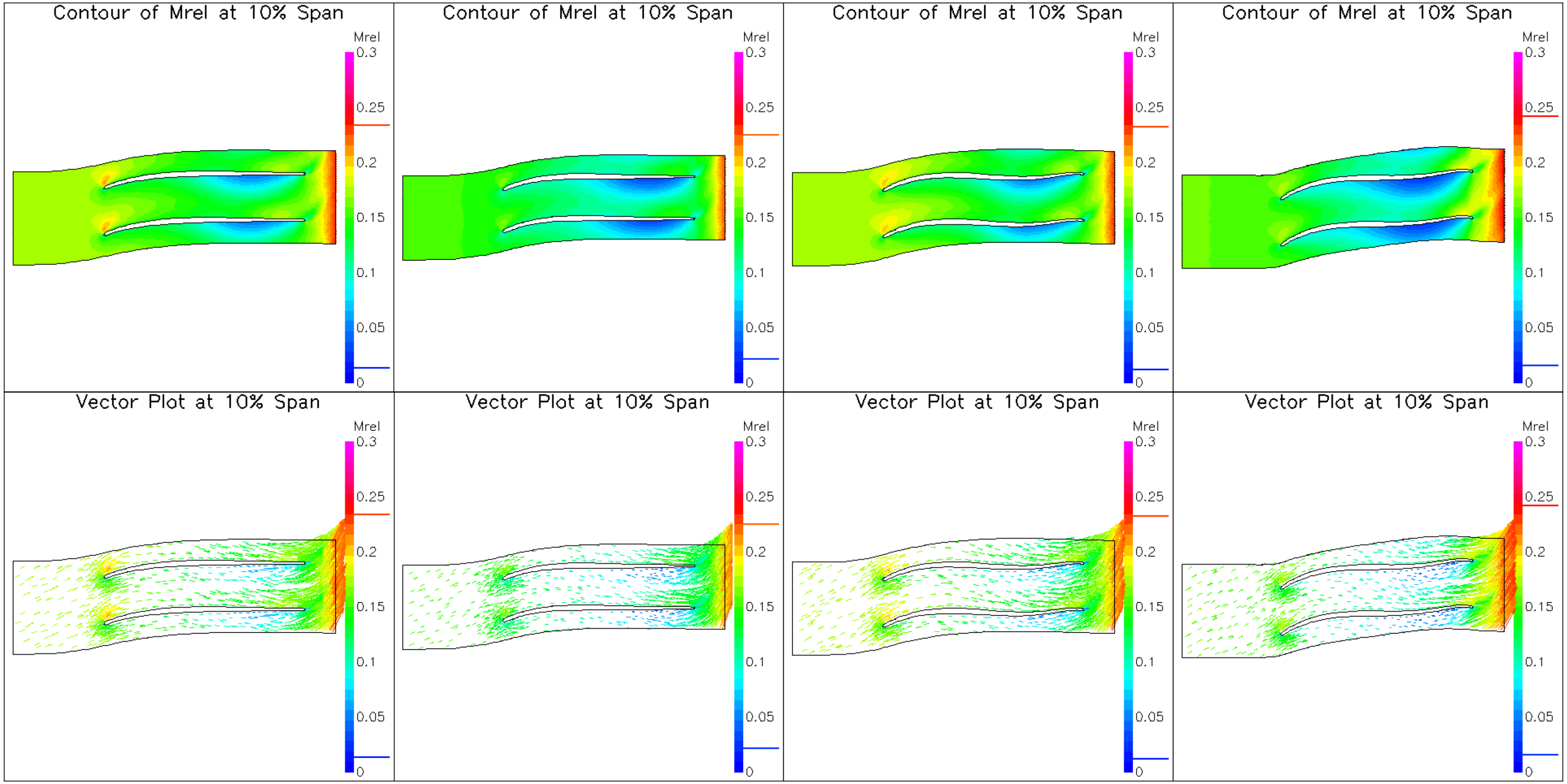

(a) Original case (b) z-r optimized case (c) m-Θ optimized case (d) z-r-m-Θ optimized case

Figure (2), Comparison between relative Mach number distribution and vector plotting for original, z-r optimized, m-Θ optimized, and z-r-m-Θ optimized cases respectively, at station 25% of the distance between periodic surfaces.



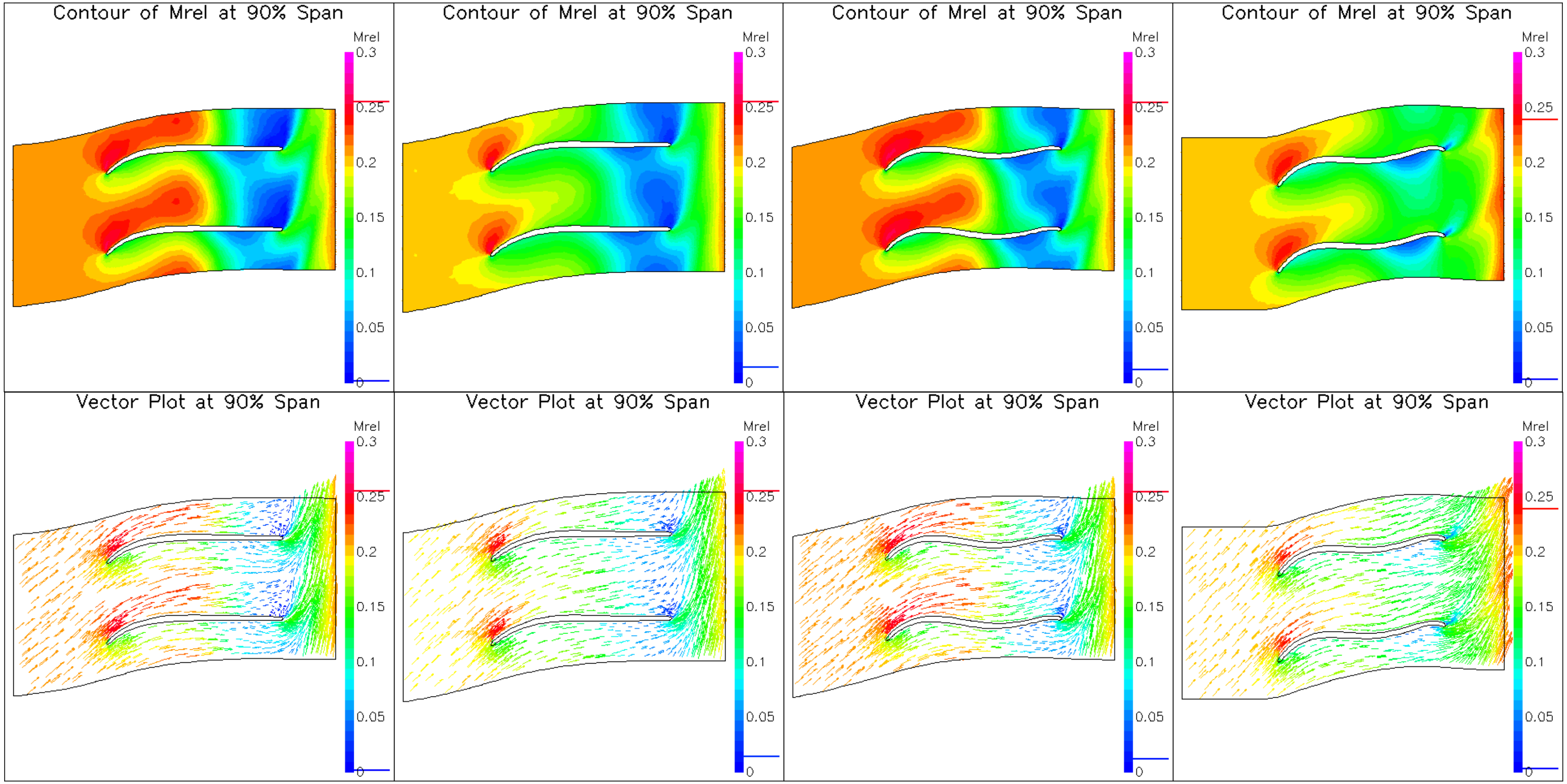

(a) Original case (b) z-r optimized case (c) m-Θ optimized case (d) z-r-m-Θ optimized case

Figure (3), Comparison between relative Mach number distribution and vector plotting for original, z-r optimized, m-Θ optimized, and z-r-m-Θ optimized cases respectively, at station 75% of the distance between periodic surfaces.




(a) Original case (b) z-r optimized case (c) m-Ө optimized case (d) z-r-m-Ө optimized case

Figure (4), Comparison between relative Mach number distribution and vector plotting for original, z-r optimized, m-Ө optimized, and z-r-m-Ө optimized cases respectively, at station 10% of the distance between hub and shroud surfaces.




(a) Original case (b) z-r optimized case (c) m-Θ optimized case (d) z-r-m-Θ optimized case

Figure (5), Comparison between relative Mach number distribution and vector plotting for original, z-r optimized, m-Θ optimized, and z-r-m-Θ optimized cases respectively, at station 90% of the distance between hub and shroud surfaces.

Table (1), Initial and optimized impellers, (optimization for max $\eta_i$ , max Pr , and $\Delta$Torq< 1 %.)

| | Efficiency | Pressure ratio | Torque | Cost function |
|---|---|---|---|---|
| | | | | |
| Original impeller values | 0.842 | 1.062 | 8.2377 N.m | 3.473 E-5 |
| | | | | |
| Hub-to-shroud optimized impeller percentage variation | + 6.917 % | + 0.282 % | + 0.481 % | 1.85 |
| Blade-to-blade optimized impeller percentage variation | + 2.749 % | + 0.066 % | + 0.683 % | 0.15 |
| Blade-to-blade and Hub-to-shroud optimized impeller percentage variation | + 8.872 % | + 0.237 % | + 0.378 % | 2.062 |