Automated runner blade design optimization process based on CFD verification

By Jacek Swiderski, P. Eng. Swiderski Engineering, Ottawa, Canada
Joseph N. Martin, Norcan Hydraulic Turbine Inc., Carleton Place, Canada
Richard Norrena, Norcan Hydraulic Turbine Inc., Carleton Place, Canada

ABSTRACT

Norcan Hydraulic Turbine Inc. has introduced Computational Fluid Dynamics (CFD), as a foundation for the development of new hydraulic technologies. Recently, research and development (R&D) efforts have been focused on the enhancement of turbine runner design methods, resulting in the creation of runner blade shape optimization algorithms, in an attempt to maximize the hydraulic efficiency of the entire turbine. The optimization procedure links a series of geometry manipulation programs with a CFD module, which provides verifications of each iteration step. The CFD module has become a central object of this algorithm. This paper presents a general outline of a blade shape optimization algorithm, theoretical basis for experimentation and selected examples of applications.

Preface

This article presents the theory and methodology behind an automated runner blade design process, focusing primarily on the initial stages of development conducted by the authors. The goal of this project is to produce runner blade designs, which when analysed will create desired flow patterns. This process is also referred as “Results Orientated Solution” (ROS). Utilizing previous experience in water turbine design, through a process, which relies greatly on the use of Computational Fluid Dynamics (CFD) results, the various steps and procedures within this complicated process were itemized, allowing the entire process to be mapped out, and in the end combined through numerous algorithms.

Historically, the prevailing approaches used in blade design (i.e. [1], [2]), appeared to be focused primarily on the verification of existing blade design methods, which do inherently appreciate the importance of CFD techniques and results.

The "Results Orientated Solution "

The primary focus in hydraulic turbine design has been water flow passage design.
The geometry of the water flow passage has the greatest impact on the efficiency and the power produced by the runner.

The numerous published design algorithms (i.e.: [1], [2]) establish a proof of the importance of this task. However the solvability of the available algorithms is highly dependant on certain assumptions, which greatly simplify the values associated with the natural physical phenomenon involved with this process. In previous methods, these assumptions were integral to the solving of the above-mentioned algorithms known to the authors.

Assuming that the CFD models a real flow in a highly satisfactory way [5], [6], [7], [8], the CFD results should be accepted as a final verification of the performances of hydraulic flow passages. Based on the foregoing assumptions, the authors have attempted to extrapolate and perfect the design method, based on the back-loop principle of CFD results and shape correction, which by making alterations based on pressure and velocity distribution, gives the desirable change of the flow.

The First Approach

The main objective of this presentation is to demonstrate the ability of solving the following problem: to find the shape of the flow passage geometry (de facto: the "blade shape") which enables the user to achieve the desirable parameters of the flow pattern at the runner exit, with minimal levels of energy losses in the blade-to-blade space.
For clarity of this presentation, we will demonstrate the method, which allows the user to automatically achieve the following objectives:

a) the desired vortex at the draft tube inlet (runner exit).
b) no-shock entry to the runner

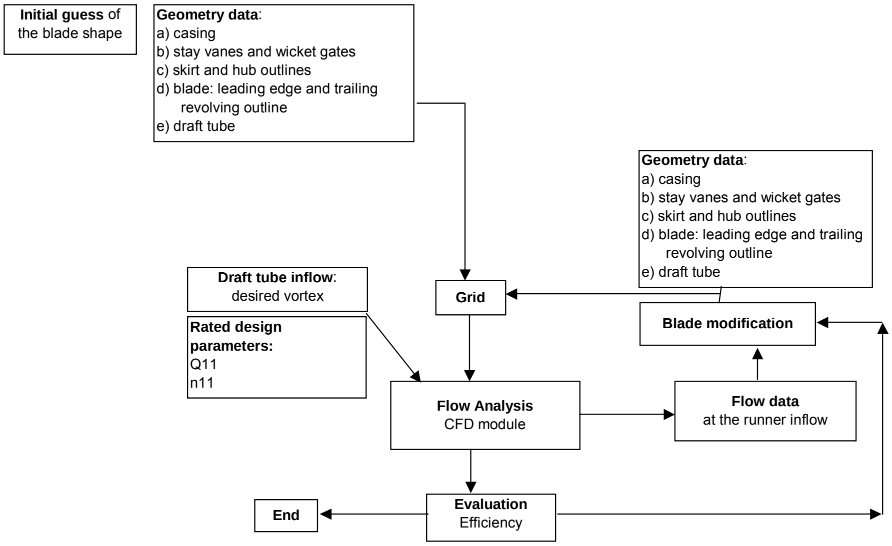


Figure 1.The general scheme of the method based on the back-loop principle - CFD-shape

A Blade Kaplan’s Turbine Designing In a Classical Configuration

Four bladed Kaplan runner should be designed for the following rated operating parameters:

Q11 = 2.1
n11 = 125

The number of the runner blades Z=4
The following are the input geometry data sets required for analysis
a) The hydraulic outline of the runner (Figure 2)
b) Geometry of the distributor, including the wicket gate angle (α_{WG})
c) Draft tube shape

The desired draft tube inlet vortex was assumed as per Figure 3.

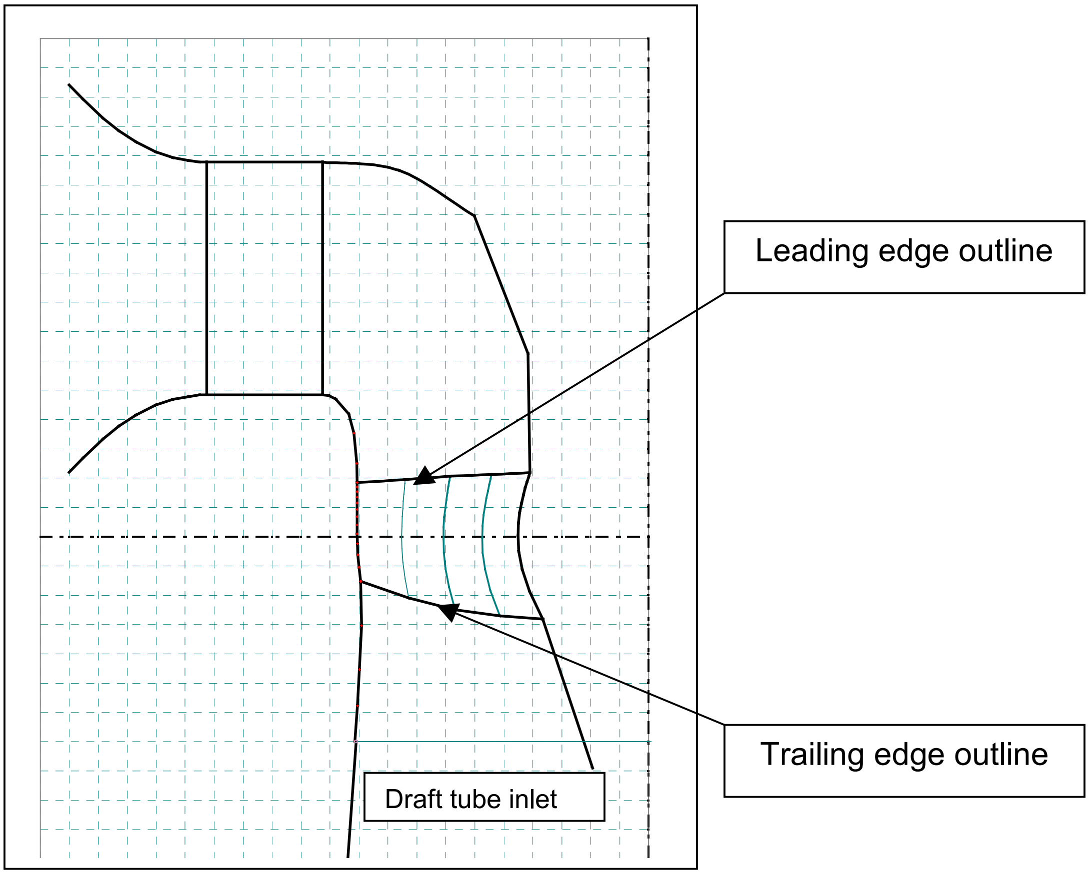


Figure 2. Hydraulic outline of the runner

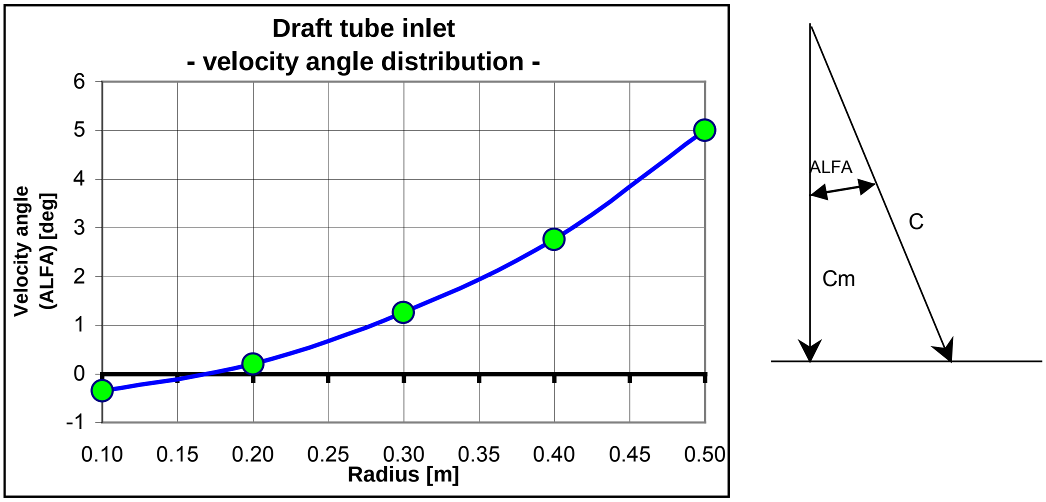


Figure 3. Distribution of the velocity angles at the draft tube inlet (draft tube inlet vortex)

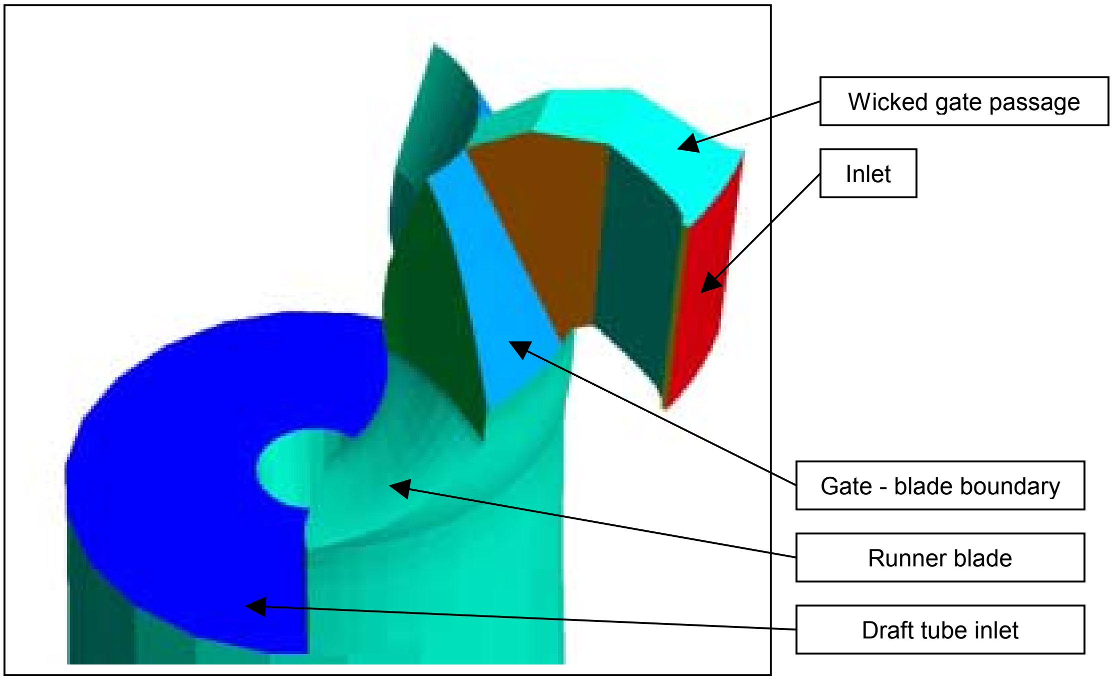
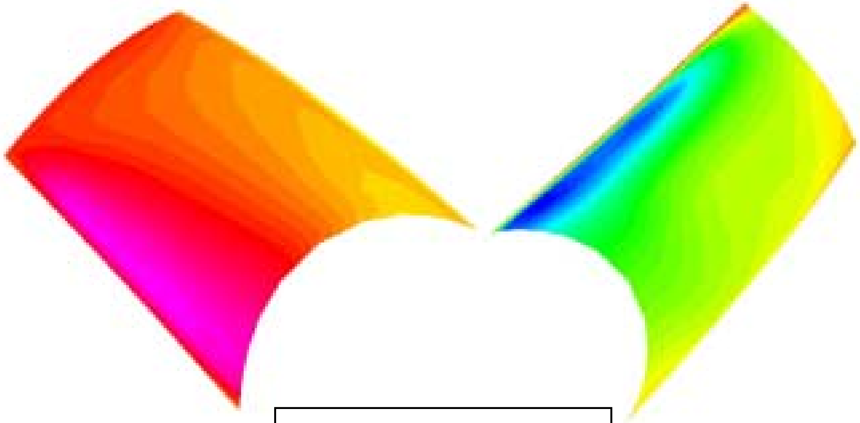
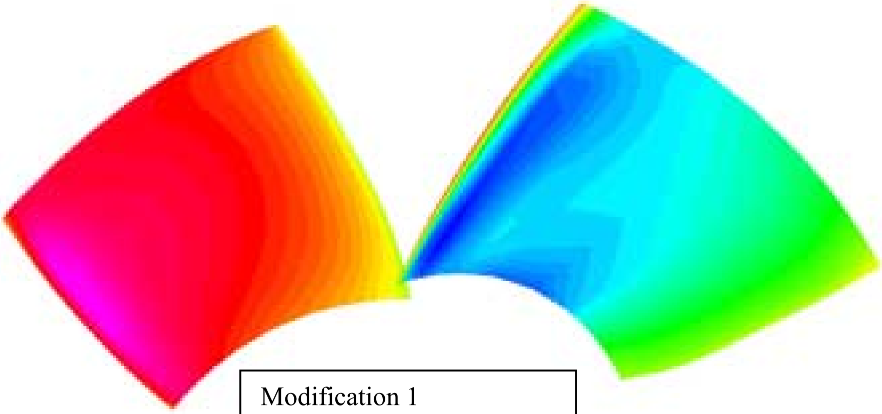


Figure 4. Grid used to test the blade design optimization algorithm

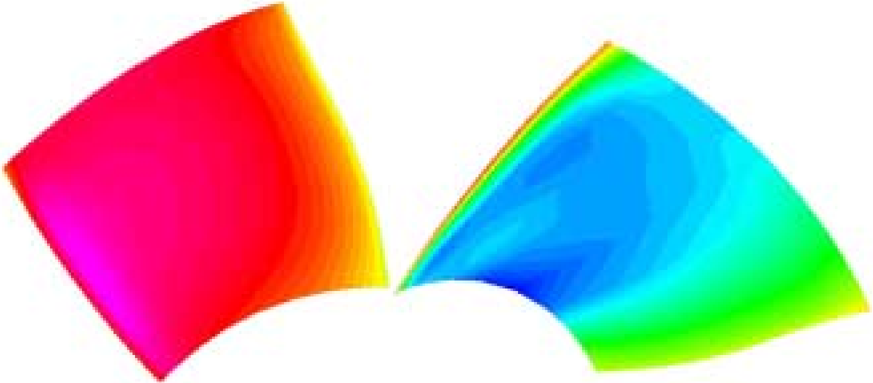
The calculations involved in this iteration process, were conducted for the single blade-to-blade passage modeled by the 47x29x25 grid. The presented here results of the iteration steps are dealing with a very simple Initial Guess geometry: flat plate blade.



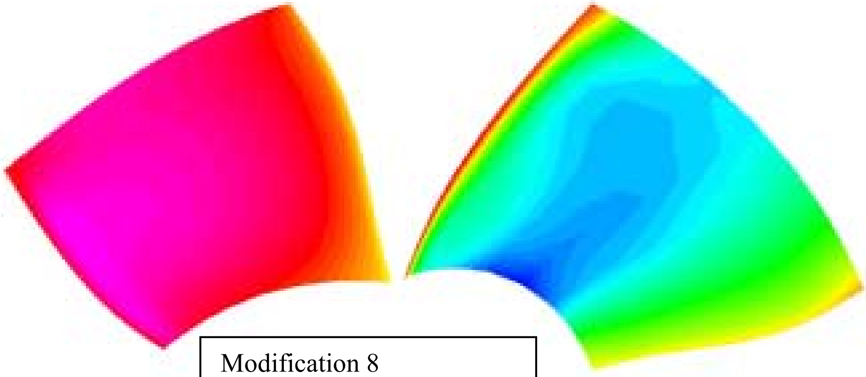
Flat plane blade
$\eta = 67\%$
$\sigma_{cr} = 2.5$



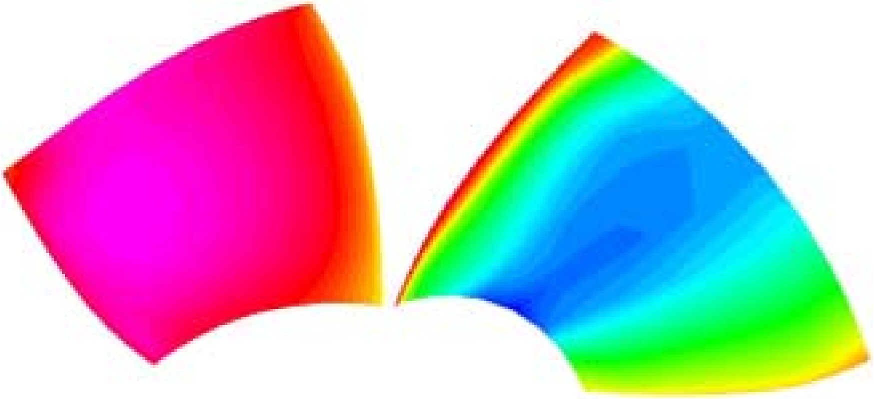
Modification 1
$\eta = 78\%$
$\sigma_{cr} = 1.75$



Modification 3
$\eta = 82\%$
$\sigma_{cr} = 1.62$



Modification 8
$\eta = 86\%$
$\sigma_{cr} = 1.1$



Modification 14
$\eta = 91\%$
$\sigma_{cr} = 0.9$

Figure 5. Results of the flow analysis for selected iteration steps of the design optimization

Observations and Conclusions

In general, the proposed iteration process (Figure 1.) demonstrates good convergence for the relaxation coefficient ζ =0.75.

At the nominal operating parameters, the blades created by a presented algorithm achieve:

a) expected distribution of static pressure on both (pressure and suction) blade surfaces,
b) expected draft tube inlet vortex (as per Figure 3.)

A presented method, as was shown, is based on relatively large amount of primary assumptions - mostly dealing with the geometry.

The Authors, when analyzed their own design methods (CFD based), were trying to determine sub-processes, which can be solved without a control of the designer - in an automatic way by a computer program. One of them is the presented process of a blade shape optimization, which in Authors' opinion is a valuable component of optimization the whole process of the water turbine design.

The problems that are still left to be automatized still remain and need to be algorymized:

a) the outline of the leading edge (in a described method they are assumed) (Figure 1)
b) the shape of the blade section profiles or in other words: the blade thickness distribution

Where turbine geometry data are known (and for practical purposes should not be considered for modification), the design methodology is very consistent with the presented method.

Recently completed work on the design of Francis runner blades, using the method described above as a component in the system of projecting turbine performances, show very promising results.

References::

[1] Krzemianowski Z.: Two-dimensional and three-dimensional models of the pump-turbine, Hydroforum 2000 (in Polish)

[2] Rogalsky T., S. Kocabiyik, R.W. Derksen: Differential Evolution in Aerodynamic Optimization, Canadian Aeronautics and Space Journal, vol. 46, No. 4, December, 2000.

[3] Krzyzanowski W.: Water Turbines – Design and Regulation, WNT, Warsaw, 1977. (in Polish)

[4] Nechleba M.: Hydraulic Turbines, Their Design and Equipment, ARTIA, Prague, 1957.

[5] CFX - TASCflow Computational Fluid Dynamics Software, Version 2.10 – Manuals, AEA Technology, Engineering Software, 2000.

[6] Swiderski J. , J. Martin : Practical implementations of Computational Fluid Dynamics in a design practice - virtual hydraulic laboratory, Hydroforum 2000, Poland

[7] Swiderski J, J. Martin : High Power Francis runner – upgrade with a new design runner, Norcan Hydraulic Turbine Inc. report, 1999

[8] Bennett K., J. Swiderski : Application of CFD Turbine Design for Small Hydro Elliott Falls, A Case Study, Waterpower XII, Salt lake City, 2001