Development of a Parametric 3D Turbomachinery Blade Modeler

R.C.W. de Koning
Development of a Parametric 3D Turbomachinery Blade Modeler

MASTER OF SCIENCE THESIS

For obtaining the degree of Master of Science in Aerospace Engineering at Delft University of Technology

R.C.W. de Koning

28-08-2015

Faculty of Aerospace Engineering · Delft University of Technology
The undersigned hereby certify that they have read and recommend to the Faculty of Aerospace Engineering for acceptance a thesis entitled “Development of a Parametric 3D Turbomachinery Blade Modeler” by R.C.W. de Koning in partial fulfillment of the requirements for the degree of Master of Science.

Dated: 28-08-2015

Chairman: ____________________________
Dr.Ir. Piero Colonna

Supervisor: ____________________________
Dr.Ir. Matteo Pini

Supervisor: ____________________________
Ir. Salvatore Vitale

Thesis registration number: 041#15#MT#FPP
Nowadays Organic Rankine Cycle (ORC) power systems are of paramount importance to exploit waste heat and renewable energy sources. Standard design rules and empirical models are mostly available for steam/gas turbines and can not be directly applied for ORC. Because of this, a redefinition of the design strategy is needed, starting from the turbine concept, passing through dedicated preliminary design optimization and eventually arriving at a complete new redefinition of the optimal blade profiles through advanced optimization methodologies. To fill the gap between (zero-dimensional) mean-line analysis and 3D fluid-dynamic analysis a Turbomachinery Blade Modeler (TBM) is required. The modeler not only gives direct control of the blade geometry but also provides valuable feedback of the design. This allows the user to construct a good initial design before refining it with more computationally expensive methods.

The TBM is developed using the python API within the framework of the open source software FreeCAD. Furthermore, it is also tightly coupled to two mesh generators, an in-house one UMG (Unstructured Mesh Generator) and to the open source Salome. This link guarantees the quasi-automatic generation of high quality CFD meshes for any kind of blade design.

The approach to construct a variety of turbomachinery blades is based partially on state-of-art parametrization techniques and uses fundamental design variables such as metal blade angles, chord length and the stagger angle. The geometry is purely build up with NURBS curves and surfaces which has the benefit that sharp edges are avoided and high smoothness of the profile shape is guaranteed. NURBS include control point position, weight and curve degree which allow a flexible control of the shape without introducing many variables, which is beneficial in optimization routines. The TBM allows for the design of any kind of blade: these include axial, centrifugal, centripetal, radial rotors/impellers and mixed blades. Moreover, to aid the designer the flow passage area distribution can be visualized run time.

The TBM has been already successfully tested for the design of a high loaded centrifugal rotor. Additionally, a complex twisted and flared axial compressor, the NASA Rotor 67 was reconstructed using the TBM. The small differences between the reference geometry and the reconstructed one were evaluated with 2D CFD simulations. Finally, a design of a radial-inflow turbine was reproduced and meshed for future analysis.

The parametric 3D turbomachinery blade modeler has proven to be a very powerful tool for designing turbine and compressor stator/rotors. Moreover, after the consolidation of the algorithms and a direct coupling with the CFD solver SU2, the tool will be ready to be used as a turbomachinery optimization environment.
Acknowledgements

I wish to thank my supervisor Salvatore Vitale for all his advice, ideas and help constructing the Turbomachinery Blade Modeler. His enthusiasm about the project and the weekly to almost daily meetings were the main drive to keep me going.

I’d like to thank Matteo Pini for guiding me throughout the thesis to head for the right direction. Furthermore I thank Antonio Ghidoni for the use of his mesh generator UMG and his help with coupling it to the blade modeler, and Raynold Tan for the construction and validation of the NASA Rotor 67 test case and the coupling to the external mesh generator Salome. Also I appreciated the help I got from the users and developers of FreeCAD at the FreeCAD forum. Valuable to me were also my family, friends and fellow students who were all very enthusiastic about the project and kept me in a good mood. Thank you all for making my last year as a student a good one!

Delft, The Netherlands

28-08-2015

R.C.W. de Koning
Contents

Summary v
Acknowledgements vii
List of Figures xii
List of Tables xiii
Nomenclature xvi

1 Introduction 1
1.1 Background ................................................. 1
1.2 Scope ..................................................... 1
1.3 Outline ................................................... 2

2 Basic Concepts and Principles 5
2.1 Turbomachinery fundamentals ................................ 5
2.2 Organic Rankine cycle turbines .............................. 7
2.3 Blade design in turbomachinery ............................. 7
2.4 Turbomachinery mean-line design ........................... 8
2.5 NURBS curves and surfaces .................................. 8
2.5.1 NURBS definition ....................................... 9
2.5.2 Control points and curve order .......................... 9
2.5.3 Knots and basis functions ................................ 10
2.5.4 Rational NURBS ......................................... 10
2.5.5 Interpolation curve ....................................... 11

3 Design of Axial, Centrifugal and Centripetal Blades 13
3.1 Blade modeler design procedure ............................ 13
3.2 Program structure .......................................... 13
3.3 Camberline construction .................................... 16
3.4 2D profile construction ..................................... 17
3.4.1 Parametrization ........................................ 17
3.4.2 Fitting algorithm ....................................... 20
3.4.3 Area distribution ....................................... 22
3.4.4 2D CFD domain ........................................ 22
3.5 3D blade construction ...................................... 26
3.5.1 Blade surface and solid .................................. 26
3.5.2 Flaring ................................................. 27
3.5.3 3D Area distribution ................................... 28
3.5.4 3D CFD domain ........................................ 29

4 Design of Radial Rotors and Impellers 31
4.1 Program structure .......................................... 31
4.2 2D Meridional channel definition .......................... 33
4.3 Camberline definition ...................................... 33
4.4 3D blade construction ..................................... 35
4.5 3D area distribution ....................................... 38
4.6 3D CFD domain ............................................ 38

5 Applications 41
5.1 Design of a centrifugal rotor ............................... 41
## Contents

5.1.1  2D design and CFD analysis ........................................ 41  
5.1.2  3D CFD analysis ..................................................... 43  
5.2  Reconstruction of a 3D axial compressor blade ....................... 44  
5.2.1  Design methodology .................................................. 44  
5.2.2  2D meshes ............................................................ 48  
5.2.3  2D CFD analysis ...................................................... 48  
5.3  Reconstruction of a radial-inflow turbine ............................. 53  

6  Conclusions and Recommendations ........................................ 55  
6.1  Conclusions .............................................................. 55  
6.2  Recommendations ....................................................... 56  

References ............................................................................. 57  

A  NASA Rotor 67 Design ......................................................... 59  
A.1  Design parameters .......................................................... 59  
A.2  UMG2 input files ............................................................ 59  
  A.2.1  Options file .............................................................. 59  
  A.2.2  Spacing and control file ............................................. 60  
  A.2.3  Topology file ............................................................ 61  
  A.2.4  Geometry file ........................................................... 62  
A.3  SU2 configuration file ...................................................... 66  

B  Radial-Inflow Turbine Input Files ........................................... 75  
B.1  Stator input file ............................................................. 75  
B.2  Rotor input file ............................................................. 77
# List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1</td>
<td>Axial compressor</td>
<td>2</td>
</tr>
<tr>
<td>1.2</td>
<td>Radial rotor and impeller</td>
<td>2</td>
</tr>
<tr>
<td>1.3</td>
<td>Centrifugal blades</td>
<td>3</td>
</tr>
<tr>
<td>1.4</td>
<td>Centripetal blades</td>
<td>3</td>
</tr>
<tr>
<td>2.1</td>
<td>Velocity triangles of a turbine rotor row</td>
<td>6</td>
</tr>
<tr>
<td>2.2</td>
<td>Preliminary blade design phase procedure</td>
<td>8</td>
</tr>
<tr>
<td>2.3</td>
<td>3rd order NURBS curve</td>
<td>10</td>
</tr>
<tr>
<td>2.4</td>
<td>Uniform basis functions</td>
<td>10</td>
</tr>
<tr>
<td>2.5</td>
<td>Quadratic rational NURBS curve representing a circular arc</td>
<td>11</td>
</tr>
<tr>
<td>2.6</td>
<td>Regular and interpolated NURBS curve</td>
<td>11</td>
</tr>
<tr>
<td>3.1</td>
<td>Blade modeler structure</td>
<td>14</td>
</tr>
<tr>
<td>3.2</td>
<td>UML class diagram of the axial/centrifugal/centripetal package</td>
<td>15</td>
</tr>
<tr>
<td>3.3</td>
<td>Axial camberline parametrization</td>
<td>17</td>
</tr>
<tr>
<td>3.4</td>
<td>Centrifugal camberline parametrization</td>
<td>17</td>
</tr>
<tr>
<td>3.5</td>
<td>Control point positioning parameters</td>
<td>18</td>
</tr>
<tr>
<td>3.6</td>
<td>Effect of $w_1$ on camberline shape</td>
<td>18</td>
</tr>
<tr>
<td>3.7</td>
<td>Effect arctangent parameter on the u-distribution</td>
<td>19</td>
</tr>
<tr>
<td>3.8</td>
<td>Construction of the 2D profile using NURBS curve</td>
<td>19</td>
</tr>
<tr>
<td>3.9</td>
<td>Effect $w_{le}$ on circular arc leading edge shape</td>
<td>20</td>
</tr>
<tr>
<td>3.10</td>
<td>Ordering of points</td>
<td>21</td>
</tr>
<tr>
<td>3.11</td>
<td>View of the TBM interface during the design of an centrifugal rotor</td>
<td>23</td>
</tr>
<tr>
<td>3.12</td>
<td>CFD domain for an axial blade</td>
<td>24</td>
</tr>
<tr>
<td>3.13</td>
<td>CFD domain for a centrifugal blade</td>
<td>24</td>
</tr>
<tr>
<td>3.14</td>
<td>2D inviscid mesh of an axial blade</td>
<td>25</td>
</tr>
<tr>
<td>3.15</td>
<td>2D hybrid mesh of a centrifugal blade</td>
<td>25</td>
</tr>
<tr>
<td>3.16</td>
<td>Closeup on boundary layers at the trailing edge of a hybrid mesh</td>
<td>26</td>
</tr>
<tr>
<td>3.17</td>
<td>Controlling and interpolated profiles</td>
<td>28</td>
</tr>
<tr>
<td>3.18</td>
<td>Effect of interpolating profiles on the shape of a NASA Rotor 67 slice at $h_\pi = 0.1935$</td>
<td>28</td>
</tr>
<tr>
<td>3.19</td>
<td>Flaring parametrization</td>
<td>28</td>
</tr>
<tr>
<td>3.20</td>
<td>Translated and rotated pitch</td>
<td>29</td>
</tr>
<tr>
<td>3.21</td>
<td>3D CFD domain bottom view</td>
<td>30</td>
</tr>
<tr>
<td>3.22</td>
<td>3D CFD domain side view</td>
<td>30</td>
</tr>
<tr>
<td>3.23</td>
<td>Surface mesh of the NASA Rotor 67</td>
<td>30</td>
</tr>
<tr>
<td>4.1</td>
<td>UML class diagram of the radial blades package</td>
<td>32</td>
</tr>
<tr>
<td>4.2</td>
<td>Meridional channel parametrization</td>
<td>33</td>
</tr>
<tr>
<td>4.3</td>
<td>2D radial camberline parametrization</td>
<td>34</td>
</tr>
<tr>
<td>4.4</td>
<td>Blade angle transformation on a radial-inflow turbine [1]</td>
<td>35</td>
</tr>
<tr>
<td>4.5</td>
<td>Camberline surface and camberlines at hub, mid and tip of a radial-inflow turbine</td>
<td>36</td>
</tr>
<tr>
<td>4.6</td>
<td>Camberline surface and 3D profiles at hub, mid and tip of radial-inflow turbine, zoomed on LE</td>
<td>36</td>
</tr>
<tr>
<td>4.7</td>
<td>Impeller with splitter blades</td>
<td>37</td>
</tr>
<tr>
<td>4.8</td>
<td>3D area distribution of a radial-inflow turbine</td>
<td>38</td>
</tr>
<tr>
<td>4.9</td>
<td>Flow passage area section evaluated in a radial-inflow turbine</td>
<td>39</td>
</tr>
<tr>
<td>4.10</td>
<td>CFD domain of a radial-inflow turbine</td>
<td>40</td>
</tr>
<tr>
<td>4.11</td>
<td>CFD domain of a radial-inflow turbine, bottom view</td>
<td>40</td>
</tr>
<tr>
<td>Figure</td>
<td>Description</td>
<td>Page</td>
</tr>
<tr>
<td>--------</td>
<td>-----------------------------------------------------------------------------</td>
<td>------</td>
</tr>
<tr>
<td>4.12</td>
<td>Inviscid surface mesh of a radial-inflow turbine</td>
<td>40</td>
</tr>
<tr>
<td>4.13</td>
<td>Inviscid surface mesh of a radial-inflow turbine, close-up on the hub 'TE'</td>
<td>40</td>
</tr>
<tr>
<td>5.1</td>
<td>Centrifugal rotor 2D profile [2]</td>
<td>42</td>
</tr>
<tr>
<td>5.2</td>
<td>Flow passage area (top) and thickness (bottom) distribution of the 2D centrifugal rotor [2]</td>
<td>42</td>
</tr>
<tr>
<td>5.3</td>
<td>Mach contour of the designed centrifugal rotor [2]</td>
<td>43</td>
</tr>
<tr>
<td>5.4</td>
<td>Top view of 3D stream lines of the flow solution with ( \frac{\Omega}{\Omega_0} = \frac{1}{3} ) [2]</td>
<td>44</td>
</tr>
<tr>
<td>5.5</td>
<td>Downstream view of 3D stream lines of the flow solution with ( \frac{\Omega}{\Omega_0} = \frac{1}{3} ) [2]</td>
<td>44</td>
</tr>
<tr>
<td>5.6</td>
<td>TBM NASA Rotor 67 blade</td>
<td>45</td>
</tr>
<tr>
<td>5.7</td>
<td>Original NASA Rotor 67 blade</td>
<td>45</td>
</tr>
<tr>
<td>5.8</td>
<td>NASA Rotor 67 constructed with 2D profiles</td>
<td>45</td>
</tr>
<tr>
<td>5.9</td>
<td>NASA Rotor 67 design parameters extracted with the TBM</td>
<td>45</td>
</tr>
<tr>
<td>5.10</td>
<td>2D flow passage areas at the NASA Rotor 67</td>
<td>46</td>
</tr>
<tr>
<td>5.11</td>
<td>2D area distributions at NASA Rotor 67 spanwise positions ( z_{hub} = 0.09563, n_{blades} = 22 )</td>
<td>46</td>
</tr>
<tr>
<td>5.12</td>
<td>Slice 1565 contours</td>
<td>47</td>
</tr>
<tr>
<td>5.13</td>
<td>Slice 1865 contours</td>
<td>47</td>
</tr>
<tr>
<td>5.14</td>
<td>Mesh of 2D profile 1565</td>
<td>48</td>
</tr>
<tr>
<td>5.15</td>
<td>Blade loading of slice 1565</td>
<td>51</td>
</tr>
<tr>
<td>5.16</td>
<td>Mach contours of slice 1565</td>
<td>51</td>
</tr>
<tr>
<td>5.17</td>
<td>Blade loading of slice 1865</td>
<td>52</td>
</tr>
<tr>
<td>5.18</td>
<td>Mach contours of slice 1865</td>
<td>52</td>
</tr>
<tr>
<td>5.19</td>
<td>Blade angle distributions from TBM and reference</td>
<td>53</td>
</tr>
<tr>
<td>5.20</td>
<td>Wrap angle distributions from TBM and reference</td>
<td>53</td>
</tr>
<tr>
<td>5.21</td>
<td>Turbine rotor and stator</td>
<td>54</td>
</tr>
<tr>
<td>5.22</td>
<td>Stator data points fitted</td>
<td>54</td>
</tr>
</tbody>
</table>
## List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.1</td>
<td>Geometrical parameters for the design of the rotor blade</td>
<td>42</td>
</tr>
<tr>
<td>5.2</td>
<td>Inputs and results of the blade to blade simulations [2]</td>
<td>43</td>
</tr>
<tr>
<td>5.3</td>
<td>SU2 input parameters slice 1565</td>
<td>49</td>
</tr>
<tr>
<td>5.4</td>
<td>SU2 input parameters slice 1865</td>
<td>49</td>
</tr>
<tr>
<td>5.5</td>
<td>Results slice 1565</td>
<td>50</td>
</tr>
<tr>
<td>5.6</td>
<td>Results slice 1865</td>
<td>50</td>
</tr>
<tr>
<td>A.1</td>
<td>NASA Rotor 67 spanwise parameters extracted from the blade modeler</td>
<td>59</td>
</tr>
</tbody>
</table>
# Nomenclature

## Latin Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>( m )</td>
<td>Mass flow</td>
<td>([\text{kg/s}])</td>
</tr>
<tr>
<td>( A )</td>
<td>Flow passage area</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( C )</td>
<td>Curve function</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( c )</td>
<td>Chord length</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( f )</td>
<td>Flaring parameter</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( h )</td>
<td>Blade height</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( k )</td>
<td>Single knot of knot sequence</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( M )</td>
<td>Mach number</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( m )</td>
<td>Meridional channel length</td>
<td>([\text{m}])</td>
</tr>
<tr>
<td>( m )</td>
<td>Multiplicity</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( N )</td>
<td>Basis function</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( n )</td>
<td>Number of poles</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( o )</td>
<td>Throat width</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( P )</td>
<td>Power</td>
<td>([\text{J/s}])</td>
</tr>
<tr>
<td>( P )</td>
<td>Pressure</td>
<td>([\text{N/m}^2])</td>
</tr>
<tr>
<td>( p )</td>
<td>Degree of B-Spline</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( R )</td>
<td>Blade radius</td>
<td>([\text{m}])</td>
</tr>
<tr>
<td>( R )</td>
<td>Gas constant</td>
<td>([\text{J/kg/K}])</td>
</tr>
<tr>
<td>( T )</td>
<td>Temperature</td>
<td>([\text{K}])</td>
</tr>
<tr>
<td>( t )</td>
<td>Parameter for control point positioning</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( t_c )</td>
<td>Concave tolerance</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( tcl )</td>
<td>Tip clearance</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( U )</td>
<td>Tangential flow velocity</td>
<td>([\text{m/s}])</td>
</tr>
<tr>
<td>( V )</td>
<td>Absolute flow velocity</td>
<td>([\text{m/s}])</td>
</tr>
<tr>
<td>( W )</td>
<td>Relative flow velocity</td>
<td>([\text{m/s}])</td>
</tr>
</tbody>
</table>
$w$  Weight of control point  

**Greek Symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td>Absolute flow angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Blade angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\delta$</td>
<td>Flaring angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Specific heat ratio</td>
<td>[-]</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Stagger angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\Lambda$</td>
<td>Sweep angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>Cone angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Rake angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Radius used at leading/trailing edge</td>
<td>[-]</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Camberline circumferential position</td>
<td>[deg]</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Pitch (translational/rotational)</td>
<td>[-]/[deg]</td>
</tr>
<tr>
<td>$\zeta$</td>
<td>Kinetic energy loss coefficient</td>
<td>[%]</td>
</tr>
</tbody>
</table>

**Abbreviations**

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>API</td>
<td>Application Programming Interface</td>
</tr>
<tr>
<td>BL</td>
<td>Boundary Layer</td>
</tr>
<tr>
<td>CAD</td>
<td>Computer-Aided Design</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>FD</td>
<td>Fluid Dynamic</td>
</tr>
<tr>
<td>GUI</td>
<td>Graphical User Interface</td>
</tr>
<tr>
<td>LE</td>
<td>Leading edge</td>
</tr>
<tr>
<td>NURBS</td>
<td>Non-Uniform Rational Basis Spline</td>
</tr>
<tr>
<td>ORC</td>
<td>Organic Rankine Cycle</td>
</tr>
<tr>
<td>SU2</td>
<td>Stanford University Unstructured</td>
</tr>
<tr>
<td>TBM</td>
<td>Turbomachinery Blade Modeler</td>
</tr>
<tr>
<td>TD</td>
<td>Thermodynamic</td>
</tr>
<tr>
<td>TE</td>
<td>Trailing edge</td>
</tr>
<tr>
<td>UMG2</td>
<td>Unstructured Mesh Generator 2-dimensional</td>
</tr>
<tr>
<td>UMG3</td>
<td>Unstructured Mesh Generator 3-dimensional</td>
</tr>
</tbody>
</table>
1

Introduction

1.1 Background

Nowadays Organic Rankine Cycle (ORC) power systems are of paramount importance to exploit waste heat and renewable energy sources. Standard design rules and empirical models are mainly available for steam/gas turbines and cannot be directly applied to ORC. Because of this, a redefinition of the design strategy is needed, starting from novel turbine concepts, passing through dedicated preliminary design optimization, Pini et al. (2013) [3]; Casati et al. (2014) [4], and eventually realizing uncommon blade profiles through advanced optimization methodologies, Persico et al. (2013) [5]; Pini et al. (2014) [6]. To fill the gap between (zero-dimensional) mean-line design and 3D fluid-dynamic analysis a Turbomachinery Blade Modeler (TBM), aimed at constructing the blade geometry, is required. The modeler gives direct control of the blade geometry as well as provides valuable outputs of the geometry such as the area distribution, e.g. by analyzing the flow passage area distribution a first guess of the Mach distribution along the channel can be obtained. This allows to speed up the initial design before refining it with more computationally expensive methods, such as shape optimization techniques.

Some applications of turbomachinery blades are axial compressors and turbines used in the aviation sector (Figure 1.1), radial rotors/impellers often used in the automotive industry (Figure 1.2), centrifugal (Figure 1.3) and centripetal (Figure 1.4) blades mainly used nowadays in ORC power systems.

1.2 Scope

Due to the aforementioned interest in the development of design methodologies for ORC expanders, the blade modeler was initially devised for turbine blades parametrization. However, because of the generality of the implemented parametrization techniques and the similarity of the design problem itself, the blade modeler can fulfill parametrization of compressor blades as well. The tool is able to design any turbomachinery blade architectures (axial, centrifugal, centripetal and mixed). Furthermore, also mixed machines
may be prototyped, e.g. a multi-stage centrifugal turbine followed by one or two axial stages. The TBM is implemented using the Python API of the Open Source CAD FreeCAD, Falck and Collette (2012) [7]. It has also a graphical interface embedded into the FreeCAD GUI. Moreover, the tool is directly connected with an in-house highly automated unstructured mesh generator (UMG) in order to accelerate the design process from the geometry construction to the fluid dynamic analysis and optimization.

The objective of this research can ultimately be stated as follows:

_The objective of the research is to develop an effective parametrization to model 3D turbomachinery blades with a major focus on the design of ORC turbines._

### 1.3 Outline

The structure of the report is the following: In chapter 2 the basic concepts and principles of turbomachinery are explained and an overview of the definition and construction of NURBS curves is given. Chapter 3 covers the design of axial, centrifugal and centripetal blades. Thereafter in chapter 4 the design techniques to model radial rotors and impellers are explained. The design methods are applied on several applications in chapter 5, including the design of a centrifugal rotor and the reconstruction of an axial compressor blade and of a mixed-flow turbine. Finally in chapter 6 the conclusions and recommendations are drawn.

**Figure 1.1:** Axial compressor  
**Figure 1.2:** Radial rotor and impeller
Figure 1.3: Centrifugal blades

Figure 1.4: Centripetal blades
Introduction
Basic Concepts and Principles

The purpose of this chapter is to gain some insights in turbomachinery fundamentals such as velocity triangles, work extraction, and fluid dynamic efficiency. Furthermore, a brief summary of the peculiarity of organic Rankine turbines is provided. A typical blade design procedure is presented after which a short explanation of mean-line design is provided. Finally the use of NURBS curves and surfaces are explained.

2.1 Turbomachinery fundamentals

Turbines are used to extract energy from a flow stream and convert it into mechanical energy. The inlet guides the flow to the stator vanes where the flow is turned in tangential direction. Thereafter the rotor blades turn the flow in the opposite direction to extract the energy. In order to develop the required power, the pressure has to decrease throughout the stage, hence the flow passage area should be convergent. For compressors it works with the opposite mechanism and conversion from velocity to pressure takes place in a divergent flow channel [8],[9].

The fluid velocity at the inlet/outlet of a compressor/turbine rotor blade can be decomposed in two components: a tangential and the axial/radial one. The former is the one that causes the increase/decrease in the tangential momentum and energy of the fluid flow, while the latter guarantees the transportation of the mass-flow through the blade. Typically in turbomachinery a velocity diagram as shown in Figure 2.1 is used to visualize these velocity components. The absolute velocity $V$ is inclined with the absolute flow angle $\alpha$ from the axial direction using a non-rotating frame of reference. It contains a component $W$ that represents the relative flow velocity inclined at the blade angle $\beta$. The triangle shape is strictly related to the geometry of the stator and rotor. The other component $U$ is the tangential velocity due to the rotation of the shaft and the subscripts 2 and 3 denote the inlet and outlet section of the rotor, respectively.
The generated/absorbed power can be described using Euler’s equation for turbomachinery:

\[ P = \dot{m} \left( V_{u3} U_3 - V_{u2} U_2 \right) \quad (2.1) \]

Where \( \dot{m} \) is the mass flow and \( V_{u2} \) the tangential component of the absolute flow velocity.

In order to estimate Mach numbers for compressible flows throughout the flow channel without using computational time-expensive CFD calculations simple relations can be used assuming the flow is isentropic. Starting from the continuity equation:

\[ \dot{m} = \rho V_{ax} A \quad (2.2) \]

Using the Mach number definition, the perfect gas law and the ratios between total-to-static temperature and pressure,

\[ \frac{T_0}{T} = 1 + \frac{\gamma - 1}{2} M^2 \]
\[ \frac{P_0}{P} = \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{\frac{\gamma}{\gamma - 1}} \quad (2.3) \]

formula 2.4 is derived to calculate the Mach number. That is, if the mass flow, flow area and stagnation state of the fluid is known, Whitfield and Baines (1990) [10].

\[ \frac{\dot{m} \sqrt{RT_0/\gamma}}{AP_0} = M \left( 1 + \frac{\gamma - 1}{2} M^2 \right)^{-\frac{\gamma + 1}{2(\gamma - 1)}} \quad (2.4) \]
To improve the evaluation of the preliminary design loss models need to be implemented to account for complicated aerodynamic phenomena like profile losses, secondary flows and wakes. Widely used empirical correlations to predict loss coefficients are the model of Craig&Cox (1971) [11] and Traupel (1977) [12]. Profile loss is the loss of stagnation pressure across a blade row due to the growth of the boundary layer and separation at the trailing edge. These effects have an important role in the design of a 2D blade profile. For example, the adverse pressure gradient that steepens the boundary layer depends on the blade loading which in turn depends on the number of blades. A higher solidity leads to lower blade loading and increases the wetted area, therefore the friction losses. The optimal solidity for the blade design is therefore a trade-off among these two phenomena.

2.2 Organic Rankine cycle turbines

Alternatively to standard Rankine cycles, organic Rankine cycles make use of organic fluids to convert thermal energy into mechanical work. This brings several advantages but also complicates turbine design. ORC fluids typically have a higher molecular complexity and mass, which allows to cost-effectively scale down the Rankine power cycle to few kW, i.e. few stages turbine, sometimes also single-stage, are normally used in an ORC turbine. However, the use of these fluids entails the presence of supersonic flows and related detrimental phenomena such as shock waves and their interaction with the boundary layer along the blade. Design challenges arise in small power capacity turbines where the expansion ratio is even higher and manufacturing constraints more stringent. Achieving a high efficiency for the turbine is even more important for small-ORC system, whereby cost-effectiveness is a daunting issue.

2.3 Blade design in turbomachinery

The typical design procedure for compressor and turbine stator/rotor blades consists of three main steps: the mean-line preliminary design, blade construction with CAD models and fluid dynamic and/or structural optimization. For a given thermodynamic cycle (working fluid, power output, pressure and temperature levels, etc.) and geometrical constrains a 1D design of a turbine is constructed using mean-line code, Pini et al. (2013) [3]; Casati et al. (2014) [4]. Basic parameters, such as metal blade angles, chords, blade height etc., resulted by the mean-line design are input data for the TBM. The latter will provide a first attempt of the blade design, which will be later refined by means of advanced 2D and 3D CFD analysis and optimization. The design procedure is visualized by means of the flowchart in Figure 2.2 and more detailed descriptions of the mean-line analysis and parametrization techniques used to model blades are found in section 2.4 and chapters 3 and 4, respectively.
2.4 Turbomachinery mean-line design

In turbomachinery, the mean-line approach is a simplified model that by using 1D conservations equations (mass, energy, momentum or entropy), fluid-dynamic loss and deviation correlations performs a preliminary design of a turbomachinery. Main geometrical quantities, velocity triangles and efficiency estimation are common outcomes. Because of the lack of design methodologies and experimental data in the field of ORC these kind of tools play a major role in the development and prototyping of ORC power units.

At TU Delft an in-house tool is available which is able to design axial, radial-inflow, radial-outflow and Ljungstrom turbines. The code was specifically conceived for ORC turbines and is tightly coupled with external software for accurate thermodynamic calculations. The input for this mean-line tool consists of operating conditions of the TD cycle and fluid-dynamic and geometrical constraints. First an initial isentropic design is computed by assuming values for the rotational speed, outlet geometric angles, throat dimension, chord lengths and the reaction degree. Then the design is optimized for total-to-static efficiency maximization and the outcome includes velocity diagrams, the meridional channel shape and performance parameters such as efficiency and loss coefficients, Casati et al. (2014) [4]. For a thorough description of the mean-line tool the reader is referred to Pini et al. (2013) [3].

2.5 NURBS curves and surfaces

In computer aided design Non-Uniform Rational Basis Splines (NURBS) have become the standard for curve and surface descriptions, Les and Wayne (1997) [13]. FreeCAD, which uses the OpenCasCade geometry kernel, contains the API to construct these NURBS, hence they could easily be implemented in the TBM. Using NURBS, the design parameters for parametric turbomachinery blade construction are not only fundamental (blade angles, chord length etc.) but can also consist of parametric curve parameters (curve degree or
control point weight). Furthermore they provide geometric continuity which avoids the creation of sharp edges. This is extremely beneficial since sharp edges are especially undesired in turbomachinery because they induce supervelocities leading to shockwaves and hence a less efficient design. It is therefore useful to get a thorough understanding of the way these curves and surfaces are created and can be implemented. The following subsections focus on the creation of curves, as surfaces are constructed in a similar manner but more difficult to visualize.

### 2.5.1 NURBS definition

A NURBS curve is defined by its order, knot vector and a list of control points each with their own weight. A NURBS curve of \( p \)th degree (order-1) is defined by equation 2.5, Les and Wayne (1997) [13].

\[
C(u) = \frac{\sum_{i=0}^{n} N_{i,p}(u)w_i P_i}{\sum_{i=0}^{n} N_{i,p}(u)w_i} \quad a \leq u \leq b \tag{2.5}
\]

Where \( P_i \) are the control points, \( u \) is a parameter of the curve typically between 0-1, \( N_i \) are the basis functions and \( w_i \) are the weights corresponding to the control points. Each parameter will be explained in the subsections below.

### 2.5.2 Control points and curve order

The control points of a curve have the most influence on the overall shape. Each point on the curve is computed by taking a weighted average of a number of control points according to the order of the curve. Figure 2.3 shows a NURBS curve with its control points denoted by \( P_i \). The curve is generated as follows: Imagine a particle follows the trajectory of the curve starting at \( P_0 \) and completes it within a certain time interval \( 0 < t < 1 \). In this case the order is three hence only three control points affect the particle at a time. A point on the curve \( (P_{2,0}) \) is therefore an interpolation between the points on the lines connecting the CP \( (P_{1,0}, P_{1,1}) \). These points move within a time span (in the direction indicated by the arrows) which can be seen as the weight of a control point increasing with respect to another in time, as explained in the next paragraph. Note that at \( t = 0 \), \( P_{2,0} \) coincides with \( P_0 \) and \( P_{2,1} \) coincides with \( P_1 \). If the order of the curve increases, \( P_3 \) becomes active from the start and therefore an additional interpolation is performed. This is the interpolation between a particle moving between \( P_{2,0} \) and \( P_{2,1} \) in time. When more CPs are used higher order curve are possible but this requires additional interpolations, which slow down the process of curve generation. Although the smoothness of the curve increases with higher orders, the local control of the curve shape is reduced since a CP effects a larger part of the curve. Due to these two side effects of higher order curves it is recommended to use curves of the order 2-4.
2.5.3 Knots and basis functions

Each CP is associated with a basis function $N_i$ which specifies the weight of the CP on the location of the particle within the time range, see Figure 2.4. The basis functions show that at each time instant only three functions contribute to the location of the particle (order 3). Because the curve starts at $P_0$ the contribution of $N_0$ is 1.0 and the others is 0.0 at $t = 0$. The contribution of $N_0$ decreases to 0.0 at $t = 0.5$ at which the last control point becomes active. The interval time of a basis function determines how long a CP is active and this is corresponds to the knot vector, which is $[0, 0, 0, 0.5, 1, 1, 1]$ in this particular case. The length of this vector always equals $p + n_{cp} - 1$ where $p$ is the degree of the curve (order-1) and $n_{cp}$ is the number of control points. If this knot vector starts and ends with a full multiplicity (number of knot duplicates), is followed by simple (unique) knots and the values are equally spaced then the curve is called uniform. Without changing the position of the CP the shape of the curve can be altered by modifying the knot sequence. However, the effect of the knots on the shape of the curve is less straightforward than altering the CP, the order of the curve or the weight of the CP. Therefore the TBM focuses on the latter three parameters and uses only uniform NURBS curves.

![Figure 2.3: 3rd order NURBS curve](image)

![Figure 2.4: Uniform basis functions](image)

2.5.4 Rational NURBS

A NURBS curve is rational if the weight of one or more control points does not equal 1.0. By increasing the weight of an individual CP its influence on the curve shape increases with respect to other points hence the curve is 'pulled' towards that point. This is a useful aspect of NURBS since it allows an exact representation of a circular, elliptic or hyperbolic arc by adjusting only one parameter. An example is a circular or elliptical arc constructed with three points for the construction of the camberline as shown in Figure 2.5. The weight of the middle CP equals $e/f$ which results in $\frac{\sqrt{2}}{2}$ for quarter of a circle. The effect of the weight is also visualized for an axial camberline in Figure 3.6.
2.5 NURBS curves and surfaces

2.5.5 Interpolation curve

Additional to the techniques presented above to construct NURBS with control points it is also possible to use those points as interpolation points where the curves passes through. This has the benefit that when specifying for example a thickness distribution, the points specifying the thickness curve actually represent the thickness. Furthermore, when creating a surface using two or more NURBS curves it is assured that the surface passes through each individual curve, keeping the valuable curve information. This is not the case for simple lofting operations often found in CAD where the control points of all curves are used as control points for the surface. Using this operation only the first and last curve remain intact. This is similar to the first and last CP of a single curve representing the start and end points of the curve, respectively. However, due to the fact that the calculated interpolation curve has to preserve first and/or higher order derivatives, undesired behavior can occur. If the control points in Figure 2.6 would be used to specify the thickness distribution of a 2D blade, the maximum thickness is lower in case the points are used as CP and higher if they are used as interpolation points. The latter creates the undesired effect of a decreasing thickness between $P_3$ and $P_4$ which is against the intuition of the designer. This effect can occur if more than three CPs are used and therefore it is recommended to apply regular NURBS curves and surface in those situations. Les and Wayne (1997) [13] describe the algorithms behind interpolation curves and, among others, Koini et al. (2008) [14] implemented them in a tool for turbomachinery blade design. Because the algorithms are embedded in most CAD software, including FreeCAD, further detail about the creation of NURBS is not necessary.

---

**Figure 2.5:** Quadratic rational NURBS curve representing a circular arc

**Figure 2.6:** Regular and interpolated NURBS curve
This chapter describes the methodology developed to design turbomachinery axial, centrifugal and centripetal blades. Initially the structure of the blade modeler is explained and afterward the methods used to construct the 3D blades are elaborated. The 3D blade design can be broken down into three major steps: the definition of the camberline, the construction and analysis of the 2D profile, and finally the generation of a 3D blade.

### 3.1 Blade modeler design procedure

The TBM is constructed using the Python API of the open-source software FreeCAD, Falck and Collette (2012) [7]. It has a built-in GUI into the FreeCAD interface, but it can also be run in batch mode. The former may be useful for example in the case where a blade needs to be designed from scratch. The user through the GUI can interactively design the blade, and preliminary verify its consistency using ad hoc algorithms such as the feedback on the blade channel shape. Figure 3.1 shows exactly this two-fold approach. On top of the flow chart a blade is constructed using the GUI either from scratch using the output of a mean-line code, or by fitting an existing blade. On the bottom a shape optimization loop is shown where the TBM is used in batch mode. The modeler is tightly connected with a mesh generator UMG (Unstructured Mesh Genertator) and an open-source CFD solver SU2. A robust coupling between these three tools is of paramount importance when a shape optimization problem has to be solved. However, since the output files of the TBM are standard CAD formats (STEP, IGES, etc.) any mesh generator and CFD solver can be coupled.

### 3.2 Program structure

The Python program structure used to model axial, centrifugal and centripetal blades is shown through an UML class diagram in Figure 3.2. A clear distinction is made between
2D and 3D construction on the lower and upper side of the diagram, respectively. Both a camberline and thickness distribution are required to build a 2D profile surface. The area distribution and CFD domain are also created on 2D level to check the quality of the 2D profile. The 3D blade is then constructed by stacking 2D profiles.
3.2 Program structure

**Blade3D**
- Creates surfaces
- Creates 3D blade

**StackProfiles**
- stackMid : boolean
- n_intP : integer
- Stacks to mid profile
- Creates intermediate profiles

**Blade3DSpecs**
Collects data

**AreaDist3D**
- n_blades : integer
- n_eval : integer
- semi : boolean
- Calculates blade-to-blade area distribution

**SimDom3D**
- n_blades : integer
- p_in : integer
- p_out : integer
- c_t : float
- r_hub : float
- r_te : float
- Creates simulation domain

**Profile2D**
- w_te : percentage
- w_te : percentage
- t_c : float
- f_x : float
- f_y : float
- - Creates suction and pressure side
- Applies flaring

**AreaDist2D**
- θ : angle
- n_eval : integer
- semi : boolean
- Calculates blade-to-blade area distribution

**SimDom2D**
- θ : angle
- p_in : integer
- p_out : integer
- w_fp : float
- α_out : angle
- semi : boolean
- Creates simulation domain

**Camberline**
- optCp : integer
- p : integer
- t_1 : percentage
- t_2 : percentage
- w_1 : percentage
- Constructs 2D camberline

**CambDef**
- c_ax : float
- γ : angle
- β_in : angle
- β_out : angle
- LE : vector
- Calculates TE
- Calculates chord

**CpDist**
- optCp : B-Spline/CircArc
- Specifies u-distribution

**ThickDist**
- th_dist : vectorlist
- n_cp : integer
- th_f : integer
- n_intP : integer
- optP : Equi/ArcTan
- p : integer
- par_at : float
- Collects thickness data

**Figure 3.2:** UML class diagram of the axial/centrifugal/centripetal package
3.3 Camberline construction

The parametrization for 3D axial, centrifugal and centripetal blades starts by defining 2D profiles at various radii. Typically a minimum amount of three 2D profiles are needed (at hub, mid-span and tip) to design a twisted axial blade, while only one is sufficient for an untwisted centripetal/centrifugal blade, Vitale et al. (2015) [2]. For all the three blade architectures mentioned, the 2D profile is constructed by first defining a camberline using inflow/outflow and stagger angles, the axial/radial chord length and the position of the leading edge. The latter is also used to translate the entire 2D profile in order to control the stacking of multiple profiles in case of an axial blade, or to control the radius location in case of a centripetal/centrifugal blade. After the LE location is specified the TE x and y coordinates are calculated. For an axial camberline this is done using equation 3.1, and the middle control point of the NURBS curve is obtained using the intersection of two lines defined by the inflow and outflow angle as visualized in Figure 3.3.

\[
x_{te} = x_{le} + c_{ax} \\
y_{te} = y_{le} - c_{ax} \tan(\gamma)
\]

The approach for centrifugal and centripetal blades is mostly similar, except for the fact that the angles have to be converted to a polar frame of reference. The inflow and outflow angles are measured with respect to the line originating from the center of rotation as shown in Figure 3.4. The TE is finally obtained by calculating the intersection between the chord line and the circle defined by the radius at the outlet, see equations 3.2 and 3.3. Note that for a centrifugal blade the outlet radius is calculated by adding the radial chord value to the inlet radius while for a centripetal blade the chord value is subtracted.

\[
x_{te} = \frac{-b + \sqrt{b^2 - 4ac}}{2a} \\
y_{te} = y_{le} + m(x_{te} - x_{le})
\]

where:

\[
a = 1 + m^2 \\
b = -2m(mx_{le} - y_{le}) \\
c = m^2x_{le}^2 + y_{le}^2 - 2mx_{le}y_{le} - r_{out}^2 \\
m = \tan\left(\tan^{-1}\left(\frac{y_{le}}{x_{le}}\right) - \gamma\right) \\
r_{out} = r_{in} - c_{rad} \\
r_{in} = \sqrt{x_{le}^2 + y_{le}^2}
\]

A Bézier curve with three control points (LE, mid, TE) can be used to define the camberline, Vitale et al. (2015) [2]; Verstraete (2010) [15]. However, using a NURBS curve has the benefit that the shape can be easily adjusted without introducing many other additional variables. For example the weight of the middle control point, \(w_1\), can be adjusted to create hyperbolic or elliptic camberlines as suggested by Koini et al. [14]. Figure 3.6 visualizes the different camberline shapes that are obtained by varying the \(w_1\) parameter. Values lower than one results in an elliptic or circular shape and values higher than one
give a parabolic shape. An option to construct the camberline with four control points is also available. As depicted in Figure 3.5 two additional parameters, denoted by \( t_1 \) and \( t_2 \), are initiated which affect the interpolation between the LE, mid and TE control points. The location of these points, \( P_1 \) and \( P_2 \), are determined by equation 3.4.

\[
\begin{align*}
    x_{p_1} &= x_{le} + t_1(x_{mid} - x_{le}) \\
    y_{p_1} &= y_{le} + t_1(y_{mid} - y_{le}) \\
    x_{p_2} &= x_{mid} + (1 - t_2)(x_{te} - x_{mid}) \\
    y_{p_2} &= y_{mid} + (1 - t_2)(y_{te} - y_{mid})
\end{align*}
\]  

(3.4)

Now that the middle CP is replaced by the two new control points, also the degree of the curve can be varied between 2 and 3 to create different camberline shapes.

The pitch (distance between two contiguous blades) is also brought directly into the camberline definition. For axial blades the pitch is measured as a distance in \( y \)-direction, while for the centrifugal and centripetal one is an angle measured around the machine center of rotation. Using the pitch it is possible to calculate the area distribution of the channel between two camberlines which gives a quick insight of the area distribution of the channel without pressure/suction sides.

3.4 2D profile construction

This section describes the parametrization of the 2D profile, the definition of the channel area distribution, and finally the construction of the 2D CFD domain.

3.4.1 Parametrization

To construct a 2D blade profile a camberline and given thickness distributions for the pressure and suction side are required. The thickness distribution is defined by giving a
list of points out of which a 2nd degree B-Spline curve is constructed. The x values of the points have a range from 0.0-1.0 where 0.0 and 1.0 correspond to the LE and TE position of the camberline, respectively. The y values represent the thickness. Optional parameters are the number of points that control the thickness distribution and the thickness factor. The latter multiplies the y values of the thickness points in order to scale up or down the overall thickness.

In order to create the pressure and suction side first a number of points are distributed on the camberline which will later be translated in perpendicular direction to the camberline to create the thickness. The points on the camberline can either be equally spaced from each other, or an arctangent function can be specified to position more points near the leading and trailing edge. In case the latter function is applied an additional parameter becomes active; the stretching factor. This parameter stretches the arctangent function in order to control the spacing as visualized in Figure 3.7. Note that u represents a parameter on the camberline curve length were 0.0 and 1.0 corresponds to the LE and TE respectively. A larger stretching factor results in relatively more points located near the leading and trailing edge which gives more control of the thickness in those areas.

The spaced points on the camberline are offset perpendicular to the camberline according to the thickness distribution as shown in Figure 3.8. The thickness distribution is specified separately for the pressure and suction side using a number of points defining a B-Spline curve. As the first control point of the suction and pressure side are both located perpendicular to the inflow angle, G1 continuity is preserved at the leading edge (e.g. the pressure and suction side share a common tangent direction at the join point). The distance of the control points to the camberline for suction and pressure side are completely decoupled, hence the part of the curves near the LE/TE do not have to share a common center of curvature (G2 continuity is not secured).

For the construction of the 2D blade additional parameters are available to control the
3.4 2D profile construction

Figure 3.7: Effect arctangent parameter on the $u$-distribution

shape of the leading and trailing edge. The shape of the LE and TE can be either the B-Spline using the radii as mentioned above or a circular arc which is constructed by an additional B-Spline curve of three control points. This is visualized in Figure 3.9 where the weight of the middle control point corresponds to the weight parameter which controls

Figure 3.8: Construction of the 2D profile using NURBS curve
the sharpness/bluntness of the edge. To avoid discontinuities the circular arc is connected to the pressure and suction side respecting 2nd order derivatives.

When testing various 2D profiles the problem occurred that for very concave profiles some pressure side control points intersected each other (e.g. x value of nth control point is lower than x value of (n-1)th control point). This results in discontinuities and can be solved using the concave tolerance factor, $t_c$, which removes a control point if the concavity exceeds the tolerance. Furthermore, using the interactive interface it is possible to manually adjust the position of a single control point to avoid this discontinuity.

In order to design multiple stages the blade modeler allows the user to copy the current blade design which can then be modified as well. This option is also implemented on 3D level but is already very useful for 2D centrifugal and centripetal blades. Additionally to the duplication of the current blade the new blade can automatically be transformed from rotor to stator orientation or vice versa. This implies a simple translation in x-direction with 1.1 times the axial chord length and sign conversions for the inflow, outflow and stagger angle. For centrifugal turbines this would mean that multiple stages can be visualized quickly by starting from the inner most blade.

### 3.4.2 Fitting algorithm

This section describes the fitting algorithm used to obtain the fundamental blade parameters from existing axial, centrifugal and centripetal blades. To recreate existing 3D blades the user has to provide several amount of 2D slices containing a set of data points which can be fit with the interactive interface. In this way the NASA Rotor 67, an axial compressor, was created, see section 3.5. The fitting algorithm involves some manual handling in order to be general and able to fit any kind of 2D profile. The fitting consist of five main steps, each of them explained below.

**Initialize**

First of all the user has to provide data files containing a list of points representing the pressure and suction side. These do not have to be ordered since often when slicing a 3D
blade the points generated are completely random. The format of the data needs to be specified so that the TBM knows how to use the data, for example ‘xy’ or ‘yzz’. Then the flow direction has to be indicated because the blade modeler always assumes flow in positive x-direction. After specifying the format the data points are loaded.

**Order**
The next step is to order the random points. This is done with an ordering algorithm based on the derivatives of adjacent points. Figure 3.10 illustrates how another point is added to the ordered list of points, which is described as follows:

- A starting point and an adjacent point have to be specified, and whether the second point is clockwise from the first point.
- The derivative of the second point with respect to the first point is calculated where after a virtual point \( P \) is projected with this derivative.
- The algorithm selects the closest point to this virtual point as the next ordered point (3).

Even if two adjacent points on the suction side are widely separated (2 and 3) they can be ordered without accidentally selecting a point on the pressure side. However, if point 3 is too far away the method will not be able to find it. Therefore the user has to choose wisely where starting the ordering (preferably at points close to each other).

![Figure 3.10: Ordering of points](image)

**Extract main information**
Before extracting main blade design parameters the points representing the LE and TE have to be indicated. Now information about the camberline position, size and angles can be automatically extracted, operations that may additionally need a limited manual intervention. Furthermore, points are now separated in pressure and suction side and also ordered in each side from the LE to the TE. This last step is needed as the TBM, as mentioned before, constructs 2D blades using two curves; one for the pressure and one for the suction side.
Adjust camberline
Before fitting the blade thickness the camberline can be adjusted manually. The leading and trailing edges are now coinciding with one the data points but this does not always have to be the case. The actual LE can for example be located between two adjacent points, especially if not many data points were extracted around the LE of the original blade. Therefore some parameters are introduced to translate the control points of the LE and TE.

Optimize thickness fit
The thickness of pressure and suction side can now be fit using simple curve fitting routines. From the ‘optimize’ package of the Python SciPy library the Simplex and Sequential Least Squares Programming (SLSQP) algorithms are used. A least squares cost function is used when analyzing the difference between the y-values of one of the data points and a point on the suction/pressure curve with the same x-value.

3.4.3 Area distribution
To gain insight of the 2D blade properties the flow passage area distribution is calculated. Depending on the type of blade, axial or centrifugal/centripetal, a second blade is generated and offset from the initial blade according to the pitch specification and definition. A camberline is constructed and translated/rotated to the middle of the channel. Now to construct a channel shape distribution the approximation is made that a flow path in the middle of the channel follows the camberline. The area distribution is then obtained by spreading a user specified number of points on the flow path, equally spaced in terms of curve length, and taking the vector perpendicular to the flow path and calculating the distance to both the upper and lower blade. The area distribution during the design of a centrifugal rotor is shown in Figure 3.11. The distribution can both be visualized in terms of x-position or channel length. Due to the thickness of the blade leading and trailing edges, which is constraint by manufacturing limitations, it is inevitable to have discontinuities in the area distribution. As demonstrated in chapter 5, if a transonic blade has to be designed, a proper control of the channel shape can avoid undesired supersonic flow bubbles which may induce detrimental phenomena such as flow separation.

When more information is available, e.g. the flow path extracted from the results of CFD blade-to-blade simulations, the approach used for the calculation of the channel shape can be verified and if necessary adjusted accordingly.

3.4.4 2D CFD domain
The 2D CFD domain is constructed by creating two additional camberlines which are translated (axial) or rotated (centrifugal/centripetal) with half pitch up and down, indicated by ‘Periodic-2’ in Figures 3.12 and 3.13. Furthermore, the inflow and outflow boxes, marked by ‘Inflow’, ‘Periodic-1’, ‘Periodic-3’ and ‘Outflow’ are created using the length of the axial chord to determine the distance form the leading/trailing edge. Both this distance and the position of the trailing edge of the upper and lower camberline can be modified.
Using the in-house mesh generator UMG2 the spacing of the mesh can be controlled for each curve and periodicity is taken into account. The names shown near each curve in Figures 3.12 and 3.13 indicated a different type of spacing. With a local refinement algorithm the spacing near the LE and TE is reduced to create a very dense mesh in the region where shockwaves are expected. This can be clearly seen in Figure 3.14 which shows the inviscid mesh corresponding to the axial CFD domain. In order to increase the mesh density near the stagnation point at the leading edge a circular control volume is specified with radius r, the center at the LE position and a smaller spacing. The spacing of the control volume overwrites the other spacing in that area. All spacing is provided in dimensionless form and with certain default values. Finally a high quality mesh for any blade shape/size is automatically generated.
**Figure 3.12:** CFD domain for an axial blade

**Figure 3.13:** CFD domain for a centrifugal blade
Figure 3.14: 2D inviscid mesh of an axial blade

Figure 3.15: 2D hybrid mesh of a centrifugal blade
For viscous flow simulations a hybrid mesh containing a very dense structured mesh at the boundary layer is required. An example of such a mesh is shown in Figure 3.15 which contains the hybrid mesh corresponding to the previously shown CFD domain for a centrifugal blade. The width of the boundary layer, the amount of mesh layers and the width of the first layer are user-specified parameters. A closeup of the boundary layer near the trailing edge is shown in Figure 3.16.

![Figure 3.16: Closeup on boundary layers at the trailing edge of a hybrid mesh](image)

### 3.5 3D blade construction

#### 3.5.1 Blade surface and solid

The 3D blade is constructed by generating a B-Spline surface which is uniform in \( \nu \) direction, given that the \( \nu \) direction is in spanwise direction. In order to construct the B-Spline surfaces for pressure and suction side, and one or two extra surfaces in the case of a circular arc LE/TE, it is required that all curves have the same amount of control points and same option for circular arc or B-Spline LE/TE. An algorithm was written to interpolate the control points to ensure they are in equal number along the span. However, this leads to (slightly) different curves hence it is recommended to use the same amount of control points to create the pressure and suction side for all controlling profiles along the span. Furthermore, the type of curve for LE/TE of the hub profile overwrites all other types when creating the surface. This is also the case for the degree of all spanwise...
3.5 3D blade construction

B-Spline curves, which have to be equal. In this way also the knot sequence for each stacked curve is the same. The available B-Spline surface parameters in spanwise direction (degree, multiplicities and knots) are dependent on the degree in order to control the continuity of the spanwise surface and the accuracy with which the surface approximates the individual profiles (a larger degree means higher continuity but lower accuracy). The number of knots is equal to \( n + p + 1 \) where \( n \) is the number of poles and \( p \) the degree of the B-Spline. The multiplicity is now defined as:

\[
m_v = \begin{cases} 
(p + 1), & 1, \ldots, 1, \underbrace{(p + 1)}_{n-p-1} 
\end{cases}
\] (3.5)

The non-dimensional uniform knot sequence is related to the multiplicity, degree and number of poles by:

\[
k = \begin{cases} 
0.0, & k_{i=0}, \ldots, k_{i=m_0} \\
1.0, & \frac{(n-p-1)k_{m_0}+k_l}{n-p}, k_{i=m_0+1}, \ldots, k_{i=l-m_1} \\
& k_{i=l-m_1+1}, \ldots, k_{i=l} 
\end{cases}
\] (3.6)

Where \( l \) indicates the last index of either the multiplicities or the knot sequence. Additional to the controlling profiles an option is implemented to insert intermediate profiles between the controlling profiles. This is useful, for example, to ensure smoothness between two controlling profiles. The more interpolated profiles are used, the smaller the error between B-Spline surface and B-Spline curves. However, the surface will never actually pass through the specified 2D profiles. The additional profiles are interpolated using the specified spanwise distributions of all design parameters (blade angles, chord length, leading edge position, thickness etc.). A visualization of the stacked profiles before creating the surfaces is shown in Figure 3.17. Using this general approach any kind of twisted, translated or tapered 3D blade is possible to create, including leaned and bowed blades.

The effect of increasing the number of interpolated profiles on a 2D slice of the NASA Rotor 67 is shown in Figure 3.18. The 2D profile contains the controlling B-Splines at 0.1935 height of the blade. 13 controlling profiles were used to fit the 3D blade but without using the interpolated profiles a small error in the shape is visible. This error can be reduced by increasing the number of interpolated profiles as depicted in the figure.

The construction of the solid blade is simply done by connecting the spanwise surfaces together with the hub and tip profiles. This generates a closed contour which is easily transformed to a solid. An option was implemented to automatically generate a disc that has the form of the hub profile. The disc consist of a B-Spline curve that is revolved around the x-axis and is fully adjustable by means of the GUI. The shape of the disc therefore has a large degree of freedom so than any kind disc can be generated. The shape can be used to cut the rotor blade to ensure a cylindrical hub.

3.5.2 Flaring

For 3D axial, centrifugal and centripetal cascade blades the flaring at the hub and tip is the ratio between the outlet and inlet blade height of a given row. This ratio is characterized by the flaring angle \( \delta \) visualized in Figure 3.19 which can be applied at the 2D profile level.
The figure shows a flaring with respect to the x-axis (e.g. along the profile axial chord). This can be used when for example the hub disc has a conical shape, e.g. the radius of the disc varies in axial direction. The flaring can furthermore be applied with respect to the y-axis to create a cylindrical shaped hub and tip, found in axial blades. To realize this non-linear flaring distribution, additional parameters can be added between the LE and TE, denoted as $f_1$ and $f_2$ in Figure 3.19. These parameters form additional control points for the B-Spline flaring distribution and can be increased in amount (equally distributed along the x/y chord length) and translated in z-position to locally increase/reduce the flaring.

The three types of flaring combined with intermediate control points for non-linear flaring allow 2D profiles to twist and bend in multiple directions which is necessary to fit certain tip and hub profiles and makes the TBM very flexible. Furthermore, flaring in y-direction can also directly provide a circular tip and hub such that cutting with cylindrical surfaces is not necessary for axial blades. However, when the hub and/or shroud contours are known the TBM still allows the user to cut the blade with these shapes if desired.

### 3.5.3 3D Area distribution

The 3D area distribution is approximated by taking the average of 2D area distributions along the span. In case of an axial machine the 2D profile at each spanwise position...
is translated by $z \cdot \sin(\theta)$, where $z$ is the height of the profile and $\theta$ the pitch angle corresponding to the number of blades, see Figure 3.20.

![Figure 3.20: Translated and rotated pitch](image)

Due to this translation instead of rotation small errors will exist, which are larger near the hub due to the higher radius of curvature. With the method used to calculated the 2D area distribution is it also not possible to evaluated the area for flared profiles. Despite these drawbacks the obtained 2D area distributions along the span give meaningful results. This is shown in section 5.2.1 where the flow passage area distribution of the NASA Rotor 67 was analyzed. Although these observations can provide some insights about the designed blade it is recommended to calculate the actual 3D area distribution. This is a challenge for complex twisted and flared blades for obvious geometrical reasons.

### 3.5.4 3D CFD domain

In case of non-flared centrifugal or centripetal blades the 3D CFD domain is simply an extruded version of the 2D domain which can be created on 2D level (including tip clearance). For blade shapes varying in spanwise direction a new CFD domain has to be defined. First a camberline surface is generated out of the stacked camberlines which is then rotated with half pitch around the axis of rotation (x-axis for axial and z-axis for centrifugal/centripetal blades). This surface is extended on both sides to create the inflow and outflow region, indicated by the red sides in Figure 3.21. Similar to the inlet and outlet of the 2D CFD domain their respective length can be adjusted. The three sides are then connected and revolved with pitch angle around the axis of rotation to create the CFD domain around the 3D blade. The tip clearance is visualized in Figure 3.22 which shows the side view of a centrifugal blade. For complicated blades like the NASA Rotor 67 this method takes only a fraction of a second and is useful to quickly check in the interface how the CFD domain looks like.

The method described is useful when the blade is designed from scratch and no information about the hub and shroud contours is known. The TBM allows the user to cut the generated CFD domain with objects specifying the shape of the disc or the shroud to
allow to represent the real flow channel through complex turbomachinery (e.g. axial turbofans).

The 3D simulation domain is directly coupled to the in-house mesh generator UMG3 [16] and the open-source platform Salome to automatically create hybrid or inviscid surface/volume meshes. The surface mesh of the NASA Rotor 67 is shown in Figure 3.23.
This chapter describes the parametrization, flow passage area calculation and CFD domain of radial rotors and impellers.

4.1 Program structure

The parametrization of radial rotors and impellers differs from axial, centrifugal and centripetal blades, hence a new package was constructed for these type of blades. A UML class diagram is shown in Figure 4.1 to illustrate the structure of the program. The classes defining the thickness distribution (ThickDist and CpDist) are defined in such a general way that they can be used for all the type of blades. The aggregation link (white diamond), means that all members of the aggregation class can live autonomously and thus also be visualized separately. Note that the Profile3D class requires both a Camb3D and the CambSurf class and a minimum amount of two 3D camberlines are needed. Finally the rotor/impeller disc only requires the 2D hub camberline defined in the Camb2D class.
Blade
- \( p \) : integer
- \( \text{method} \) : norm/int
  - Creates 3D blade

AreaDist
- \( n_{\text{blades}} \) : integer
- \( n_{\text{eval}} \) : integer
  - Calculates blade-to-blade area distribution

Profile3D
  - Creates suction and pressure side

SimDom
- \( n_{\text{blades}} \) : integer
- \( \rho_{\text{up}} \) : integer
- \( \rho_{\text{lo}} \) : integer
- \( \rho \) : float
- \( \rho_{\text{hub}} \) : float
- \( \rho_{\text{le}} \) : float
  - Creates simulation domain

CambSurf
- \( p \) : integer
- \( \text{method} \) : norm/int
  - Generates camberline surface

Profile3D
  - Creates suction and pressure side

Camb3D
- \( \beta \) : list of floats
- \( \text{type} \) : turbine/compressor
- \( \phi \) : angle
- \( \text{pos} \) : angle
  - Transforms 2D camberline to 3D

CpDist
  - Specifies u-distribution

ThickDist
- \( \text{th}_{\text{dist}} \) : vectorlist
- \( n_{\text{cp}} \) : integer
- \( \text{th}_{f} \) : integer
- \( \text{intP} \) : integer
- \( \text{opt}_{\text{cp}} \) : Equi/ArcTan
- \( p \) : integer
- \( \text{par}_{\text{at}} \) : float
  - Collects thickness data

Camb2D
- \( h/h_{\text{up}} \) : percentage
- \( \text{opt}_{\text{cp}} \) : integer
- \( \rho \) : integer
- \( t_{1} \) : percentage
- \( t_{2} \) : percentage
- \( w_{1} \) : percentage
  - Constructs 2D camberline

CambSurf
- \( p \) : integer
- \( \text{method} \) : norm/int
  - Generates camberline surface

MeriDef
- \( r_{\text{up}} \) : float
- \( r_{\text{lo}} \) : float
- \( x_{\text{hub}} \) : float
- \( m \) : float
- \( \Lambda \) : angle
- \( \alpha_{\text{up}} \) : angle
- \( h_{\text{up}} \) : float
- \( \alpha_{\text{lo}} \) : angle
- \( h_{\text{lo}} \) : float
- \( \lambda \) : angle
  - Calculates hub and tip LE and TE

Figure 4.1: UML class diagram of the radial blades package
4.2 2D Meridional channel definition

The most common parametrization for both radial rotors and impellers starts by defining the 2D meridional contour of a single blade [1, 17, 15, 18, 19]. An overview of all design parameters used to construct the inlet and outlet of the channel is presented in Figure 4.2. The main parameters are the radii and heights (or width) of the inlet and outlet, denoted by $r$ and $h$ respectively. Additionally for mixed flow configurations the meridional absolute flow component $\alpha$, the cone angle $\lambda$ and the sweep angle $\Lambda$ can be used. As the definition of the meridional channel is used for both radial-inflow turbines and centrifugal compressors the subscripts upper and lower denote the inlet and outlet, or vice versa. The position of the entire channel along the rotation axis is controlled by the upper hub control point, indicated in red. Note that the shape of the hub and tip is not determined at this point, only their first and last control point.

![Meridional channel parametrization](image)

**Figure 4.2:** Meridional channel parametrization

4.3 Camberline definition

The camberlines are defined using NURBS curves in a similar matter to the definition of axial, centrifugal and centripetal camberlines explained in section 3.3. As a first design attempt the 2D hub camberline can be elliptical and the tip camberline elliptical/circular, as suggested by Glassman (1976) [20]. In this case three control points are enough to define the curves, however if more control is needed again more control points can be used as already seen in section 3.3 and as shown in Figure 4.3. As radial rotors/impellers are often
slightly bend in circumferential direction between hub and tip additional camberlines may
be required in between. A general method is implemented to insert camberlines at a
certain distance \( \frac{h}{h_{up}} \) from the hub upper control point, as shown in Figure 4.3. The lower
end of the camberline is set to the same height ratio between the lower hub and tip control
point. With this approach an unlimited amount of camberlines may be used to design the
blade. Notice that increasing the number of camberlines allow to a more flexible design
of the blade, especially for a better control of secondary flows, Glynn (1982) [21].

![Figure 4.3: 2D radial camberline parametrization](image)

The blade camberlines now lay in the 2D meridional plane. Before transforming each
camberline to 3D a rake angle is introduced. This angle rotates the leading edge of the
camberline in circumferential direction which is typically applied on the tip camberline in
radial-inflow turbines, Lüdecke et al. (2012) [22]. In the TBM this parameter is used for
each individual camberline. Now the complete camberlines are transformed to 3D using
a blade angle (\( \beta \)) distribution as indicated in Figure 4.4.

The transformation is performed by calculating the wrap angle \( \theta \) (circumferential position)
of points on the camberline using their radius \( R \), blade angle and camberline length \( dm \)
as shown in equation 4.1.

\[
\theta = \int \frac{\tan \beta}{R} dm 
\]

(4.1)

The blade angle distribution is defined by a B-Spline curve with a number of control points
equally spaced on the meridional channel length as shown in Figure 4.4. The height of
each CP approximates the blade angle at that meridional channel position. Usually from
mean-line code only the inflow (\( \beta_0 \)) and discharge (\( \beta_4 \)) angles are known and a linear
distribution in between can be assumed for the initial design, Mueller et al. (2012) [1].
Note that when applying equal blade angle distributions to the tip and hub their wrap
angles are different along the meridional channel, because the radius changes along the span and usually the camberline length is different as well. Due to the integral form of equation 4.1 only the starting circumferential position can be maintained for equal blade angles. Since the leading edge is usually aligned in circumferential position the transformation is always applied from leading to trailing edge, hence for radial-inflow turbines it starts at the upper radius position, see Figure 4.4, and for radial-outflow compressors it starts at the lower radius. Typical turbine blade outlet angles are around $60^\circ$ at the tip and $40^\circ$ at the hub, Sauret (2012) [23], which more or less aligns the tip and hub at the outlet.

Another way to apply the camberline distribution is by defining it directly as a function of the wrap angle $\theta$, Abidat et al. (1992) [24]. This has the advantage that the spanwise wrap angles can be kept constant since they are independent of the blade angle. However, the wrap angle has less physical meaning than the blade angle and, in contrary to inlet/outlet blade angles, it is usually not known from mean-line analysis. Therefore the $\beta$ transformation as explained above is applied in the TBM.

A B-Spline surface is constructed using all 3D camberlines as visualized in Figure 4.5. This is mandatory for the definition of the pressure and suction side and will both be used to calculate the flow passage area distribution and to construct the 3D CFD domain. One option to construct the surfaces is to use the regular B-Spline surface creation (lofting) where the the control points of the individual B-Splines act as control points for the surface. Another way is to use an interpolated B-Spline surface which goes through all the B-Spline curves, as proposed by Koini et al. (2008) [14] for axial blades.

### 4.4 3D blade construction

The thickness is applied in a similar manner as the parametrization of axial thickness. However, instead of using normals to the camberline, normals to the camberline surface are used to specify the thickness. At any point on the surface the tangent vectors in $u$ and $v$ direction are obtained using the FreeCAD API and their cross product gives the vector...
perpendicular to the surface. An example of 3D profiles at the hub, mid and tip is shown in Figure 4.6. The number of control points for the thickness distribution are again user specified and a reasonable initial guess is to keep a constant thickness distribution of 4% of the outlet blade height, as proposed by Glassmann (1976) [20].

The sides of the blade are created by defining B-Spline surfaces using the pressure and suction side curves. Together with the hub and tip surfaces these form a closed contour which is used to create the solid blade.

The complete rotor or impeller is generated by duplicating the blades around the x-axis according to the number of blades specified. The rotor disc shape is generated by revolving the hub 2D camberline around the x-axis.

In the design of radial impellers often splitter blades are used to improve the aerodynamic performance of the machine. This smaller splitter blade is positioned at half pitch to guide the flow at the outlet section where the flow passage area between regular blades is relatively large. Figure 4.7 gives an example of an impeller including splitter blades of half length with the same blade angles as the regular blades. For the parametrization the same meridional channel and 2D camberline definition was used to ensure similar scaling with the regular blades. The splitter blade camberline is then created by segmenting the 2D camberline at a percentage of the impeller length, which ensures camberlines at different span are starting at the same x-position. When transforming to 3D camberlines...
not only the wrap and rake angles are applied but also the half pitch. This introduces a new parameter to rotate the entire camberline, and so the entire blade, around the x-axis which is not only useful for positioning the splitter blade but also for the regular blade in case provided data points want to be fitted.

The rest of the parametrization procedure is equal to that of the regular blade, including the option to specify as many spanwise camberlines as desired. In the interactive interface an option was implemented to automatically add splitter blades of similar shape to the current design. Apart from the hub 2D camberline shape the design is completely free and for example also the height can be adjusted by changing the height of the individual camberlines. This option was already available for the regular blade camberlines but was restricted to keep the definition of the meridional channel inlet and outlet height.

Figure 4.7: Impeller with splitter blades

4.5 3D area distribution

The calculation of the 3D area distribution starts by defining a second blade rotated of a pitch with respect the first one and constructing a camberline surface at half pitch between these blades. The hypothesis is made that the flow path in the middle of the channel follows the camberline surface, hence the area is found perpendicular to this surface as shown in Figure 4.8. It is obtained with the following procedure:

- First normal vectors are created in the middle of the camberline surface at half span, e.g. \( v = 0.5 \). The vectors are equally distributed along the B-Spline representing the middle of the camberline surface. The amount of vectors determines the quality of the area evaluation, hence at least a number of 20 vectors is recommended.
• With these vectors 2D planes are constructed which are used to find the intersection with both blades.

• A surface that determines the area at a certain meridional channel length is visualized in Figure 4.9. It consists of four edges: an intersection with the pressure side of the first blade, one with the suction side of the second blade, and a tip and hub area curve constructed with a B-Spline interpolated through the top/bottom of the camberline surface.

For example at a radial-inflow turbine the curve connecting the hubs at the leading edge approximates a circular arc in circumferential direction. The curves further down the meridional channel (as depicted in Figure 4.9) also have a component in axial direction which is why the general approach of construction with interpolated B-Splines was used.

An example of a 3D area distribution for a radial-inflow turbine is shown in Figure 4.8 which is the same turbine validated in section 5.3. The flow passage area is evaluated 100 times and the colors correspond to the areas of the surfaces where green represents the largest area, red the smallest and white indicates the throat. Also from the corresponding graph it can easily be noticed that the throat is located at the outlet of the rotor and there is a considerable deceleration zone that should be avoided in turbine design. The parameters that have the most influence on the area distribution are blade height at inlet and outlet, the pitch and the blade angle distribution.

![Figure 4.8: 3D area distribution of a radial-inflow turbine](image)

### 4.6 3D CFD domain

The 3D CFD domain of radial rotors and impellers is constructed in a similar manner as the simulation domain of 3D axial, centrifugal and centripetal blades, explained in section 3.5.4. The CFD domain for a radial-inflow turbine consisting of 11 blades is shown in Figures 4.10 and 4.11 and is constructed with the following procedure:
The camberline B-Spline surface is revolved around the x-axis to create the domain around the blade.

Tip clearance is generated by translating the control points of the tip 3D camberline in tangential v-direction to create a new B-Spline curve representing the shroud. With this method it is ensured that the clearance is the same along the meridional channel.

The blade can be extended at the hub in a similar way to the axial 3D blade in order to cut it with the rotor disc.

The sides of the inflow section (indicated in red) are created by using the B-Spline representing the first edge of the camberline B-Spline surface and creating an additional B-Spline at an offset tangential to the surface. Using the two curves a new B-Spline surface is created with is revolved with pitch angle to create the inflow section of the domain.

The same approach is used for the outflow section and the lengths of the inflow/outflow sections are defined by the specified offset which is a percentage of the meridional channel length.

Using the tangents of the surface to create the offset allows this method to be applied on both turbines and compressors with any kind of blade angles.
An inviscid surface mesh of a radial-inflow turbine with tip clearance created with UMG3 is visualized in Figure 4.12. The increased mesh density near the TE and around the blade contour is visualized in Figure 4.13.
In this chapter some applications of the TBM are presented. As a first case a complete design, from mean-line analysis to 3D CFD analysis, of a centrifugal rotor is presented. Secondly the designs of an already existing blade and one available in the open literature are reconstructed using the TBM. While the former demonstrates the ability of the tool to design a new blade from scratch, the reconstruction of an existing blade design with different tools represents a verification of the generality of the algorithms implemented in the TBM.

5.1 Design of a centrifugal rotor

This section presents the design of a highly loaded centrifugal turbine blade for mini-ORC applications. The optimal shape of the first rotor of a 5-stage small-scale turbine is determined iteratively using the blade modeler and 2D turbulent CFD simulations. Finally the design is verified with a 3D CFD analysis. For a thorough description of the design and analysis of the five-stage configuration the reader is referred to Vitale et al. (2015) [2].

5.1.1 2D design and CFD analysis

The rotor blade was designed with the methodology presented in section 3.4. The rotor main design parameters, as provided by the mean-line calculation, were computed in Casati et al. (2014) [4] and are shown in table 5.1. These parameters are necessary but not sufficient input data for the blade construction; the designer has additional degrees of freedom such as the thickness distribution of pressure and suction side and the stagger angle. The area distribution given by the TBM, was an essential information to design this blade, i.e. an unexpected throat location and consequently supersonic flow bubbles were avoided. The final blade design was derived by iteratively changing the design parameters shown in table 5.1 and test each design with 2D viscous simulations. This resulted in a stagger angle of 22.5 degrees and a thickness distribution shown in the lower part of Figure 5.2. The final design is visualized in Figure 5.1 and the area distribution is shown
in the top part of Figure 5.2. It is worth noting that the uncustomary, non-monotonic area trend is due to the radial evolution of the blades and can not be avoided. The blade shape is characterized by a thin leading edge which avoids the creation of the throat in the semi-blade inlet region. The blade is relatively thick in the middle part in order to guide the flow and prevent separation. In order to reduce wake and mixing losses the trailing edge thickness is as small as manufacturing limits allow.

Table 5.1: Geometrical parameters for the design of the rotor blade

<table>
<thead>
<tr>
<th>Input parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$R_{in}$</td>
<td>1.88 cm</td>
</tr>
<tr>
<td>$c_{rad}$</td>
<td>0.5 cm</td>
</tr>
<tr>
<td>$\beta_{in}$</td>
<td>66.0 deg</td>
</tr>
<tr>
<td>$\beta_{out}$</td>
<td>-74.1 deg</td>
</tr>
<tr>
<td>$N_{blades}$</td>
<td>38 [-]</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Design parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$o_{out}$</td>
<td>0.1 cm</td>
</tr>
<tr>
<td>$t_{ch}$</td>
<td>100 µm</td>
</tr>
</tbody>
</table>

Figure 5.1: Centrifugal rotor 2D profile [2]

Figure 5.2: Flow passage area (top) and thickness (bottom) distribution of the 2D centrifugal rotor [2]

Table 5.2 shows the inputs and results of the blade-to-blade simulations and the Mach contour of the final blade design is presented in Figure 5.3. As expected shock waves arise on the rear suction side near the throat. The first shock is generated by the impinging of the expansion fan on the suction side and its intensity is amplified by the diverging shape of the flow channel in radial direction. The two shock waves have a large effect on the profile losses. Furthermore it is observed that no shockwaves are present in the semi-blade inlet region which confirms that the use of the area distribution calculated by the TBM is very useful. Finally it is appreciated that no flow separation occurs on both
suction and pressure sides.

**Table 5.2: Inputs and results of the blade to blade simulations [2]**

<table>
<thead>
<tr>
<th>Inputs</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid</td>
<td>D4</td>
</tr>
<tr>
<td>$T_{t,in}$</td>
<td>305.79 °C</td>
</tr>
<tr>
<td>$P_{t,in}$</td>
<td>3.66 bar</td>
</tr>
<tr>
<td>$P_{s,out}$</td>
<td>1.655 bar</td>
</tr>
<tr>
<td>$\beta_{flow,in}$</td>
<td>66.0 deg</td>
</tr>
<tr>
<td>$N$</td>
<td>19000.0 rpm</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Results</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$\dot{m}_l$</td>
<td>1.64 kg/s/m</td>
</tr>
<tr>
<td>$M_{out}$</td>
<td>1.05 [-]</td>
</tr>
<tr>
<td>$\beta_{out,flow}$</td>
<td>74.49 deg</td>
</tr>
<tr>
<td>$\zeta_{is}$</td>
<td>3.9 [-]</td>
</tr>
</tbody>
</table>

**Figure 5.3:** Mach contour of the designed centrifugal rotor [2]

### 5.1.2 3D CFD analysis

As a first design solution a 3D centrifugal blade is obtained by simply loft a 2D profile. A blade height of 0.43 cm was calculated by dividing the expected mass flow of 0.266 kg/s with the linear mass flow resulted by the blade-to-blade CFD analysis. To point out secondary effects, typically rotor blades are first simulated without tip-clearance and imposing first slip and then no-slip endwalls. Thereafter the effect of tip clearance is investigated by varying the tip clearance parameter. Figures 5.4 and 5.5 show the 3D streamlines of the flow solution where the ratio between tip clearance and blade height is set to $\frac{1}{140}$. Despite the small tip clearance a relatively large vortex is generated which covers around a quarter of the flow channel. As shown in the top view this vortex is caused by the flow passage from pressure to suction side over the top of the blade due to the large pressure difference. The supersonic flow through this small gap causes a large distortion of the flow at the tip TE, and consequently a lower effective outflow angle which decreases the amount of work extracted. This effect is emphasized when increasing the tip clearance. Due to the complexity of the fluid-dynamic phenomena such as shock waves and their interaction with the tip clearance vortex, optimization methods are required to further improve the design. However, as shown the TBM is a powerful tool for preliminary blade design.
5.2 Reconstruction of a 3D axial compressor blade

This section describes the fitting of a complex twisted and flared transonic axial compressor, the NASA Rotor 67. CFD analysis was performed on several 2D slices to identify the effect of small geometry deviations.

5.2.1 Design methodology

The reference 3D blade was sampled in thirteen equally spaced 2D slices from the hub to the tip. Each of the 2D slices was fitted by a 2D profile using the fitting algorithms presented in section 3.4.2. The 2D profiles were then used to create B-Spline surfaces required to construct the 3D blade. The geometry constructed with the TBM and the original geometry are shown in Figures 5.6 and 5.7, respectively. Figure 5.8 indicates the 2D profiles used to construct the geometry with the TBM. Note that the 2D profiles are equally spaced in spanwise direction at the trailing edge so that only flaring had to be applied on the hub and tip. The 2D profiles indicated in red, at height 0.1565 and 0.1865, were meshed and fluid-dynamic analysis was performed using SU2, Palacios (2013) [25]. The design parameters of each 2D profile created by the TBM are presented in Figure 5.9 and table A.1. Note that these design parameters were not known beforehand and solely the TBM was used to retrieve this information. Typically for an axial compressor the chord length decreases towards the tip to reduce bending moments. Furthermore, the difference between inflow and outflow angles reduces spanwise according to the vortex-free design specification. The flaring angle at the hub equals 15° and at the tip 27.7°.

The area distribution of the NASA Rotor 67 was approximated by taking several slices at equal spanwise distance from each other and compute the 2D area distribution at that location. The 2D translational pitch at each spanwise position equals $z \cdot \sin(\theta)$. Although
5.2 Reconstruction of a 3D axial compressor blade

Figure 5.6: TBM NASA Rotor 67 blade

Figure 5.7: Original NASA Rotor 67 blade

Figure 5.8: NASA Rotor 67 constructed with 2D profiles

Figure 5.9: NASA Rotor 67 design parameters extracted with the TBM

this gives a rough approximation, the calculated area distribution shown in Figure 5.11 gives meaningful results. First of all it is clear that the areas at all spanwise positions are increasing as foreseeable in a compressor cascade. The throat (i.e. the minimum section), shown by the white line in Figure 5.10 is located near the inlet as expected. Furthermore, it is observed that the area ratio between inlet and outlet decreases towards the tip which means the amount of air compressed decreases towards the tip. This is to lower the loads at the tip in order reduce its weight and so lower the mass moment of inertia.

The constructed geometry was validated using CFD. Calculations were carried out on 2D slices extracted from the created 3D blade. These were compared with the constructed 2D profiles and the reference geometry at the same span. Using only four control points
for the pressure and four for the suction side, the overall 2D shapes were fitted properly. However, it was observed that the three geometries, as depicted in Figures 5.12 and 5.13, deviate in LE shape. The deviations with respect to the reference case can be reduced by locating the LE more accurately, applying a better value for the first CP of the pressure and suction side (controlling the LE radius) or by inserting more CPs in the thickness distribution. In the next sections the effects of these geometry deviations is explained further.
5.2 Reconstruction of a 3D axial compressor blade

Figure 5.12: Slice 1565 contours

Figure 5.13: Slice 1865 contours
5.2.2 2D meshes

The 2D profiles of sections 1565 and 1865 were meshed with UMG2. The input files to construct the hybrid mesh of 2D profile section 1565 are shown in appendix A.2 and the mesh, consisting of 91421 elements, is visualized in Figure 5.14. To accurately capture the fluid-dynamic characteristics in the boundary layer the BL is 0.007 thick and consists of 28 structured layers where the first layer thickness is $2e^{-5}$ in order to get $y+ < 1.0$. Additional control volumes were applied at the leading and trailing edge to locally increase the mesh density. The length of the inflow and outflow sections are set to 1.5 times the axial chord length to ensure spurious reflections from the boundary conditions can affect the solutions, Giles (1990) [26].

![Figure 5.14: Mesh of 2D profile 1565](image)

5.2.3 2D CFD analysis

2D CFD turbulent calculations were performed on the three different geometries for the two slices. The main input parameters for the viscous SU2 simulations are provided in tables 5.3 and 5.4 for slice 1565 and 1865 respectively.
The complete SU2 input file for slice 1565 is provided in appendix A.3. The resulting Mach contours are presented in Figures 5.16 and 5.18 and the corresponding blade loading is visualized in Figures 5.15 and 5.17. The loading for the reference case is not smooth, because in contrast to the B-Spline used for the TBM profiles, the extracted data points were connected by a spline and the amount of data points was relatively low (68 points per side). However, the results were considered acceptable for a rough validation of the design approach implemented in the TBM. A shockwave is formed at the LE due to the high pressure gradient. At approximately half chord this shockwave interacts with the suction side creating an additional weaker shockwave. This interaction induces the separation of the boundary layer on the suction side which clearly grows towards the TE. This causes a reduction of the efficiency of the compressor. Comparing the 2D fitted and 3D sliced profiles of case 1565 it is observed that the magnitude of the shockwaves is very similar and the deflection of the second shockwave is located at a corresponding chordwise position. However, the magnitude of the shocks of the reference case are clearly lower than the other two. When looking at the respective leading edges one can see that reference LE has a more constant radius of curvature than the other two, which induces lower adverse pressure gradients, hence weaker shockwaves. Therefore the BL separation occurs further downstream resulting in a slightly smaller wake.

Looking at the results of slice 1865 it is observed that again the reference case has a weaker shockwave but the differences are smaller due to the more similar leading edge radius. However, the 2D fitted profile shows a larger pressure drop on the suction side after the LE, which increases at $x = 0.025$. This causes a weak shockwave early on the suction side after which the pressure distribution becomes similar to the other two cases.

Conclusively, the stagnation pressure loss coefficient $C_{P_0}$, calculated by equation 5.1 and the effective outflow angle were compared for the three cases, as shown in tables 5.5 and 5.6 for slice 1565 and 1865 respectively. For slice 1565 the outflow angles are almost equal for the three cases. However, the stagnation pressure loss coefficient is 6.6% higher for the 3D slice profile compared to the other two. This was also observed for the slice 1865 but the differences are lower (4.9%).

$$C_{P_0} = \frac{P_{T_1} - P_2}{P_{T_1} - P_1}$$ (5.1)
On the basis of the obtained outcomes, it is concluded that the TBM is capable of reproducing existing geometries using only few design parameters. This is already a trustworthy starting point for further optimizations. However, especially for compressor blades, it is advised to accurately fit the leading edge. To realize this a significant amount of data points should be extracted from the reference blade and more control points can be used for the thickness distribution. If possible it would even be better to cut a section of the reference blade in FreeCAD so that the surface geometry is represented by B-Splines resulting in smooth pressure distributions.

<table>
<thead>
<tr>
<th></th>
<th>$C_{P_0}$</th>
<th>$\beta_{out_{eff}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D Fitted</td>
<td>0.13075</td>
<td>-37.451</td>
</tr>
<tr>
<td>3D Sliced</td>
<td>0.13957</td>
<td>-37.429</td>
</tr>
<tr>
<td>Reference</td>
<td>0.13098</td>
<td>-37.394</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th></th>
<th>$C_{P_0}$</th>
<th>$\beta_{out_{eff}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D Fitted</td>
<td>0.08112</td>
<td>-51.228</td>
</tr>
<tr>
<td>3D Sliced</td>
<td>0.09055</td>
<td>-50.985</td>
</tr>
<tr>
<td>Reference</td>
<td>0.08629</td>
<td>-51.183</td>
</tr>
</tbody>
</table>
5.2 Reconstruction of a 3D axial compressor blade

![Graph of Blade Loading](image)

**Figure 5.15**: Blade loading of slice 1565

![Mach Contours](image)

**Figure 5.16**: Mach contours of slice 1565
Figure 5.17: Blade loading of slice 1865

Figure 5.18: Mach contours of slice 1865
5.3 Reconstruction of a radial-inflow turbine

In order to validate the capabilities of the TBM, a rotor and stator design for a small, high pressure ratio radial turbine was reconstructed. The geometry dimension were taken from appendix A from Sauret (2012) [23] which is a reproduction itself from the design presented by Jones (1996) [27].

Figure 5.21 shows the stator and rotor configuration constructed with the TBM. The input parameters used to construct the stator and rotor are found in appendices B.1 and B.2, respectively. The stator was modeled with inflow and outflow angles of $-26^\circ$ and $-28^\circ$, a stagger angle of $27.5^\circ$ and axial chord length of $26.5\text{mm}$. The thickness specified by Sauret was fitted using simple curve fitting algorithms using five control points on each side and the 2D stator is shown in Figure 5.22. After lofting the 2D profile the 19 nozzles were created using a polar array around the x-axis. The rotor was designed using the 2D meridional channel dimensions after that the blade angle distribution was applied. Only five blade angle parameters per spanwise profile were needed to reach a high level of fitting accuracy as shown in Figure 5.19. The resulting wrap angle distribution is compared with the reference values in Figure 5.20. The small deviation is caused partly by the deviation in blade angles and partly by the definition of the wrap angle distribution in the TBM. The latter always starts at zero but can be adjusted using the rake angle in order to overcome the difference. The thickness was fitting in a similar matter, but the specified thickness distribution by Sauret (2012) [23] was divided by 2 to obtain the required thickness for pressure and suction side. Furthermore, it was taken into account that the thickness distribution is scaled with the meridional channel length in the TBM. For an accurate $<2\%$ error distribution six control points were used for the hub and four for the tip.

![Figure 5.19: Blade angle distributions from TBM and reference](image1)

![Figure 5.20: Wrap angle distributions from TBM and reference](image2)

To validate the radial rotor/impeller design capabilities of the TBM it is recommended to perform CFD calculations on the reconstructed turbine and compare the results with Sauret (2012) [23].
Figure 5.21: Turbine rotor and stator

Figure 5.22: Stator data points fitted
Conclusions and Recommendations

6.1 Conclusions

ORC power systems are of paramount importance to exploit waste heat and renewable energy sources. Due to lack of experimental data available for these unconventional configurations and the fact that standard design rules for turbomachinery blade design can not be applied there is need for parametric blade modeler that uses a general approach to design turbomachinery cascades. This is achievable by developing a dedicated blade modeler using only open-source and in-house software.

The turbomachinery blade modeler is capable of generating a variety of compressor and turbine blades including axial, centrifugal, centripetal, mixed blades and radial rotors/impellers. The modeler is built with an object-oriented programming philosophy and a knowledge based engineering approach was used to maintain valuable fundamental design variables. The TBM is directly coupled to (hybrid) mesh generators in order to speed-up the grid generation process. Furthermore, the tool allows the user to freely select a desired number of design variables in order to be flexible in blade shape definition.

An interactive interface is built to ease both the design of new blades and the reconstruction of existing ones, gaining control and feedback of the designed blade and provide the option of designing multiple stages. Through the design of a highly loaded centrifugal rotor for mini-ORC applications it was demonstrated that a thorough understanding of the flow physics within the blade passage is attainable with the information provided by the TBM such as the flow area distribution. Expensive optimization procedures are anyhow required to reduce efficiency losses due to complex fluid-dynamic phenomena such as shockwaves and vortex interactions.

The capabilities of the blade modeler were also tested by reconstructing a transonic axial compressor: the NASA Rotor 67. CFD simulations performed on several 2D sections indicated that the design constructed with the TBM is very close to the reference geometry, thus demonstrating the accuracy of the devised fitting procedure. Finally, the design of a radial turbine was carried out by reconstructing a design from open literature.

In conclusion, in terms of blade design capabilities it can be stated that the developed
turbomachinery blade modeler is competitive with existing tools. However, as explained in the next section, it is recommended to further improve the TBM algorithms for a wide range of applications.

6.2 Recommendations

Although the turbomachinery blade modeler has proven to be an effective and powerful tool several improvements can be made:

- To improve the robustness and usability it is recommended to validate the TBM with a variety of test cases.
- The TBM was tested with Python 2.7 coupled to FreeCAD 0.14 and 0.15 (both on Windows and Linux), but it is worth to test its usability on other software versions.
- An implementation of dependency tracking would be helpful to avoid unnecessary recomputations which can speed up the design process.
- The algorithms and GUI should be consolidated. For example an algorithm to create the 3D CFD domain for axial and radial blades without the need of cutting operations could speed up the process. Additionally, to improve the 3D blade fitting, an algorithm can be developed that adjusts the 2D profiles in order to fit a given 3D blade or one which creates a B-Spline surface through the profile curves.
- The GUI can be improved by extending FreeCAD with a blade modeler module to create a GUI environment purely focused on turbomachinery.
- A direct coupling with a CFD solver like SU2 offers the opportunity to use advanced shape optimization techniques which are required to improve the initial design.
- Important is also to cover the structural aspect during blade design, hence it is recommended to link the TBM with software for structural computations, opening up the possibility for multidisciplinary design optimization.

The TBM is then ready to form the central part of a turbomachinery preliminary design phase.
References


A.1 Design parameters

Table A.1: NASA Rotor 67 spanwise parameters extracted from the blade modeler

<table>
<thead>
<tr>
<th>$h$</th>
<th>$\beta_{in}$</th>
<th>$\beta_{out}$</th>
<th>$c$</th>
<th>$\gamma$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.00000</td>
<td>-38.2457</td>
<td>13.60203</td>
<td>0.08959</td>
<td>14.74083</td>
</tr>
<tr>
<td>0.08333</td>
<td>-43.0753</td>
<td>10.65439</td>
<td>0.08635</td>
<td>19.95663</td>
</tr>
<tr>
<td>0.16667</td>
<td>-45.0866</td>
<td>-2.97562</td>
<td>0.08306</td>
<td>26.24434</td>
</tr>
<tr>
<td>0.25000</td>
<td>-46.7763</td>
<td>-17.9591</td>
<td>0.07908</td>
<td>32.28043</td>
</tr>
<tr>
<td>0.33333</td>
<td>-49.6079</td>
<td>-28.9792</td>
<td>0.07439</td>
<td>37.74182</td>
</tr>
<tr>
<td>0.41667</td>
<td>-51.5131</td>
<td>-36.8675</td>
<td>0.06967</td>
<td>42.31340</td>
</tr>
<tr>
<td>0.50000</td>
<td>-53.2648</td>
<td>-43.2056</td>
<td>0.06459</td>
<td>46.22770</td>
</tr>
<tr>
<td>0.58333</td>
<td>-55.4005</td>
<td>-47.8424</td>
<td>0.05989</td>
<td>49.54580</td>
</tr>
<tr>
<td>0.66667</td>
<td>-56.9120</td>
<td>-51.4202</td>
<td>0.05631</td>
<td>52.49531</td>
</tr>
<tr>
<td>0.75000</td>
<td>-57.6948</td>
<td>-54.0238</td>
<td>0.05335</td>
<td>54.97714</td>
</tr>
<tr>
<td>0.83333</td>
<td>-59.2398</td>
<td>-55.8933</td>
<td>0.05017</td>
<td>57.40637</td>
</tr>
<tr>
<td>0.91667</td>
<td>-60.8615</td>
<td>-58.4139</td>
<td>0.04728</td>
<td>59.75904</td>
</tr>
<tr>
<td>1.00000</td>
<td>-66.7571</td>
<td>-54.8421</td>
<td>0.04077</td>
<td>62.94178</td>
</tr>
</tbody>
</table>

A.2 UMG2 input files

The following input file were used to create the unstructured hybrid mesh for a NASA Rotor 67 slice at $z = 0.1565$.

A.2.1 Options file

The options file contains information and constraints for the mesh generator such as number of boundary layers and a periodic tolerance.
fmt name
'grd' '1565'
optimization
1
max element deformation
1.0
layer of the background grid
3
Periodic geometry
.True. 1e-06
Scaling for SU2 file
0.1
number of boundary layers BL thickness
28 0.007
Graph for hybrid mesh construction
.False.
Kind of radial basis function (1-10)
8
Support radius for compact basis functions
0.11

A.2.2 Spacing and control file

The spacing and control file defines the minimum and maximum allowed spacing \( (h) \) at each edge, the pitch and has the option to create a control volume with locally increased mesh density. The latter is useful to increase the density near the LE/TE.

\[
\begin{array}{cccc}
thk\_bl & n\_BC & GEOM & CV \\
2e-5 & 5 & axl & 1 \\
\end{array}
\]

\[
\begin{array}{cccc}
PITCH & xc & yc \\
0.447 & 0.0000000 & 0.0000000 \\
\end{array}
\]

1 INFLOW h_min h_max Node per RadCRv
0.015 0.015 5.

3 OUTFLOW h_min h_max Node per RadCRv
0.015 0.015 5.

4 PERIO h_min h_max Node per RadCRv
0.0035 0.015 5.

5 PERIO h_min h_max Node per RadCRv
0.0035 0.015 5.

8 BLADE h_min h_max Node per RadCRv
0.001 0.002 5.

NZONES
2
RADIUS XC YC h
A.2 UMG2 input files

0.01 0.059837 0.054523 0.0005
RADIUS XC YC h
0.01 0.803756 -0.5213 0.0005

A.2.3 Topology file

The topology file specifies the type of each curve and the periodicity or symmetry of one curve with another curve.

<table>
<thead>
<tr>
<th>curve type</th>
<th>periodic curve</th>
<th>Modifiable curve</th>
</tr>
</thead>
<tbody>
<tr>
<td>8</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>8</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>6</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>8</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>7</td>
<td>0</td>
</tr>
<tr>
<td>4</td>
<td>10</td>
<td>0</td>
</tr>
<tr>
<td>5</td>
<td>9</td>
<td>0</td>
</tr>
</tbody>
</table>

Number of ZONES
1
ZONE 1
1
2
3
4
5
6
7
8
9
-10

1 far-field/inflow
2 symmetry
3 outflow
4 periodic1
5 periodic2
6 projected1
7 projected2
8 wall1
9 wall2
10 wall3
11 wall4
12 wall5
13 wall6
A.2.4 Geometry file

The geometry file contains the NURBS information of the simulation domain. 'N' represents a NURBS curve with order, knot sequence and control points with weight. 'S' represents a simple straight line.

<table>
<thead>
<tr>
<th>Number of curves</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>dim</th>
<th>ord</th>
<th>np</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>3</td>
<td>24</td>
</tr>
</tbody>
</table>

knot sequence

<p>| |</p>
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
</tr>
<tr>
<td>0.0</td>
</tr>
<tr>
<td>0.047619047619</td>
</tr>
<tr>
<td>0.0952380952381</td>
</tr>
<tr>
<td>0.142857142857</td>
</tr>
<tr>
<td>0.190476190476</td>
</tr>
<tr>
<td>0.238095238095</td>
</tr>
<tr>
<td>0.285714285714</td>
</tr>
<tr>
<td>0.333333333333</td>
</tr>
<tr>
<td>0.380952380952</td>
</tr>
<tr>
<td>0.428571428571</td>
</tr>
<tr>
<td>0.47619047619</td>
</tr>
<tr>
<td>0.52380952381</td>
</tr>
<tr>
<td>0.571428571429</td>
</tr>
<tr>
<td>0.619047619048</td>
</tr>
<tr>
<td>0.666666666667</td>
</tr>
<tr>
<td>0.714285714286</td>
</tr>
<tr>
<td>0.761904761905</td>
</tr>
<tr>
<td>0.809523809524</td>
</tr>
<tr>
<td>0.857142857143</td>
</tr>
<tr>
<td>0.904761904762</td>
</tr>
<tr>
<td>0.952380952381</td>
</tr>
</tbody>
</table>

control points

<p>| |</p>
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1.0</td>
</tr>
<tr>
<td>1.0</td>
</tr>
<tr>
<td>1.0</td>
</tr>
<tr>
<td>1.0</td>
</tr>
</tbody>
</table>

14 wall7
15 wall8
16 wall9
17 wall10
<table>
<thead>
<tr>
<th>x</th>
<th>y</th>
<th>w</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.05983786028</td>
<td>0.05452371103</td>
<td>1.0</td>
</tr>
<tr>
<td>0.0634203455884</td>
<td>0.0575717904428</td>
<td>1.0</td>
</tr>
<tr>
<td>0.0676452785153</td>
<td>0.0533431908278</td>
<td>1.0</td>
</tr>
<tr>
<td>0.0728367594745</td>
<td>0.0481771723384</td>
<td>1.0</td>
</tr>
<tr>
<td>0.0793599312882</td>
<td>0.0417320290731</td>
<td>1.0</td>
</tr>
<tr>
<td>0.0877832600751</td>
<td>0.0334836405228</td>
<td>1.0</td>
</tr>
<tr>
<td>0.0990351362977</td>
<td>0.0225924012294</td>
<td>1.0</td>
</tr>
<tr>
<td>0.114717555394</td>
<td>0.0076459124132</td>
<td>1.0</td>
</tr>
<tr>
<td>0.137767216869</td>
<td>-0.0138533902329</td>
<td>1.0</td>
</tr>
<tr>
<td>0.173846974936</td>
<td>-0.046465318189</td>
<td>1.0</td>
</tr>
<tr>
<td>0.23365395577</td>
<td>-0.0980156581093</td>
<td>1.0</td>
</tr>
<tr>
<td>0.331231521751</td>
<td>-0.17641881721</td>
<td>1.0</td>
</tr>
<tr>
<td>0.458579839957</td>
<td>-0.270926952527</td>
<td>1.0</td>
</tr>
<tr>
<td>0.570383457652</td>
<td>-0.351348134865</td>
<td>1.0</td>
</tr>
<tr>
<td>0.646549253282</td>
<td>-0.405255629466</td>
<td>1.0</td>
</tr>
<tr>
<td>0.695807261176</td>
<td>-0.43976732321</td>
<td>1.0</td>
</tr>
<tr>
<td>0.728725883251</td>
<td>-0.462684010635</td>
<td>1.0</td>
</tr>
<tr>
<td>0.751815630145</td>
<td>-0.478689804501</td>
<td>1.0</td>
</tr>
<tr>
<td>0.768744609539</td>
<td>-0.49038978135</td>
<td>1.0</td>
</tr>
<tr>
<td>0.781623145665</td>
<td>-0.499270737876</td>
<td>1.0</td>
</tr>
<tr>
<td>0.791720335776</td>
<td>-0.50622192204</td>
<td>1.0</td>
</tr>
<tr>
<td>0.799835032646</td>
<td>-0.511800858989</td>
<td>1.0</td>
</tr>
<tr>
<td>0.806491335577</td>
<td>-0.51637212007</td>
<td>1.0</td>
</tr>
<tr>
<td>0.8037566344</td>
<td>-0.5213099639</td>
<td>1.0</td>
</tr>
</tbody>
</table>

```
'N'
dim  ord  np
task 2 3 24
knot sequence
       0.0
       0.0
       0.0
       0.0
0.047619047619
0.095238095238
0.142857142857
0.190476190476
0.238095238095
0.285714285714
0.33333333333
0.380952380952
0.428571428571
0.47619047619
0.52380952381
0.571428571429
0.619047619048
```
0.666666666667
0.714285714286
0.761904761905
0.809523809524
0.857142857143
0.904761904762
0.952380952381

<table>
<thead>
<tr>
<th>x</th>
<th>y</th>
<th>w</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.05983786</td>
<td>0.05452371</td>
<td>1.0</td>
</tr>
<tr>
<td>0.05625537</td>
<td>0.05147563</td>
<td>1.0</td>
</tr>
<tr>
<td>0.06103042</td>
<td>0.04767749</td>
<td>1.0</td>
</tr>
<tr>
<td>0.06516730</td>
<td>0.04154897</td>
<td>1.0</td>
</tr>
<tr>
<td>0.07039945</td>
<td>0.03390519</td>
<td>1.0</td>
</tr>
<tr>
<td>0.07721567</td>
<td>0.02412725</td>
<td>1.0</td>
</tr>
<tr>
<td>0.08642266</td>
<td>0.01122597</td>
<td>1.0</td>
</tr>
<tr>
<td>0.09944072</td>
<td>-0.00645639</td>
<td>1.0</td>
</tr>
<tr>
<td>0.11894232</td>
<td>-0.03183199</td>
<td>1.0</td>
</tr>
<tr>
<td>0.15026722</td>
<td>-0.07014866</td>
<td>1.0</td>
</tr>
<tr>
<td>0.20404054</td>
<td>-0.13013609</td>
<td>1.0</td>
</tr>
<tr>
<td>0.29567412</td>
<td>-0.21951986</td>
<td>1.0</td>
</tr>
<tr>
<td>0.42126909</td>
<td>-0.32218202</td>
<td>1.0</td>
</tr>
<tr>
<td>0.53879559</td>
<td>-0.39916803</td>
<td>1.0</td>
</tr>
<tr>
<td>0.62185752</td>
<td>-0.44497443</td>
<td>1.0</td>
</tr>
<tr>
<td>0.67654263</td>
<td>-0.47193010</td>
<td>1.0</td>
</tr>
<tr>
<td>0.71343408</td>
<td>-0.48883473</td>
<td>1.0</td>
</tr>
<tr>
<td>0.73945442</td>
<td>-0.50018002</td>
<td>1.0</td>
</tr>
<tr>
<td>0.75860043</td>
<td>-0.50823660</td>
<td>1.0</td>
</tr>
<tr>
<td>0.77320157</td>
<td>-0.51422003</td>
<td>1.0</td>
</tr>
<tr>
<td>0.78466993</td>
<td>-0.51882450</td>
<td>1.0</td>
</tr>
<tr>
<td>0.79389991</td>
<td>-0.52247026</td>
<td>1.0</td>
</tr>
<tr>
<td>0.80147771</td>
<td>-0.52542475</td>
<td>1.0</td>
</tr>
<tr>
<td>0.80375663</td>
<td>-0.52130996</td>
<td>1.0</td>
</tr>
</tbody>
</table>

' S '  
\[
\begin{array}{ccc}
\text{dim} & \text{np} \\
2 & 2 \\
\end{array}
\]

' S '  
\[
\begin{array}{ccc}
\text{dim} & \text{np} \\
-1.0560403009 & -0.16892628897 \\
-1.0560403009 & 0.27797371103 \\
\end{array}
\]
A.2 UMG2 input files

2 2
x y
1.91963479558 -0.2978599639
1.91963479558 -0.7447599639

'S'
dim np
2 2
x y
0.05983786028 -0.16892628897
-1.0560403009 -0.16892628897

'S'
dim np
2 2
x y
-1.0560403009 0.27797371103
0.05983786028 0.27797371103

'S'
dim np
2 2
x y
1.91963479558 -0.7447599639
0.8037566344 -0.7447599639

'S'
dim np
2 2
x y
0.8037566344 -0.2978599639
1.91963479558 -0.2978599639

'N'
dim ord np
2 2 3
knot sequence
0.0
0.0
0.0
1.0
1.0
1.0
control points
x y w
0.05983786028 -0.16892628897 1.0
0.323438513409 -0.47874284531 1.0
A.3 SU2 configuration file

The following configuration file was used to perform SU2 simulations on slice 1565 of the NASA Rotor 67.
% Mathematical problem (DIRECT, ADJOINT, LINEARIZED)
MATH_PROBLEM = DIRECT
%
% Restart solution (NO, YES)
RESTART_SOL = NO
%
% -------------------------- COMPRESSIBLE FREE-STREAM DEFINITION --------------------------
%
% Mach number (non-dimensional, based on the free-stream values)
MACH_NUMBER = 0.2
%
% Angle of attack (degrees, only for compressible flows)
AoA = -49.6079
%
% Free-stream pressure (101325.0 N/m^2 by default, only Euler flows)
FREESTREAM_PRESSURE = 190000.3375
%
% Free-stream temperature (273.15 K by default)
FREESTREAM_TEMPERATURE = 288.15
%
% Free-stream temperature (1.2886 Kg/m^3 by default)
FREESTREAM_DENSITY = 1.2886
%
% Free-stream option to choose if you want to use Density (DENSITY_FS)
% or Temperature (TEMPERATURE_FS) to initialize the solution
FREESTREAM_OPTION = TEMPERATURE_FS
%
% Free-stream Turbulence Intensity
FREESTREAM_TURBULENCEINTENSITY = 0.05
%
% Free-stream Turbulent to Laminar viscosity ratio
FREESTREAM_TURB2LAMVISCRATIO = 100.0
%
% Reynolds number (non-dimensional, based on the free-stream values)
REYNOLDS_NUMBER = 6.0E5
%
% Init option to choose between Reynolds (default) or
% thermodynamics quantities for initializing the solution (REYNOLDS, TD_CONDITIONS)
INIT_OPTION = TD_CONDITIONS

% -------------------------- REFERENCE VALUE DEFINITION --------------------------
%
% Reference origin for moment computation
REF_ORIGIN_MOMENT_X = 0.25
REF_ORIGIN_MOMENT_Y = 0.00
REF_ORIGIN_MOMENT_Z = 0.00
%
% Reference length for pitching, rolling, and yawing non-dimensional moment
REF_LENGTH_MOMENT= 1.0
%
% Reference area for force coefficients (0 implies automatic calculation)
REF_AREA= 1.0
%
% Reference pressure (101325.0 N/m^2 by default, only for compressible flows)
%REF_PRESSURE= 101325.0
%
% Reference temperature (273.15 K by default, only for compressible flows)
%REF_TEMPERATURE= 273.15
%
% Reference density (1.2886 Kg/m^3 by default, only for compressible flows)
%REF_DENSITY= 1.28
%
% Reference element length for computing the slope limiter epsilon
REF_ELEM_LENGTH= 0.1
%
% ---- IDEAL GAS, POLYTROPIC, VAN DER WAALS AND PENG ROBINSON CONSTANTS -----%
%
% Different gas model (STANDARD_AIR, IDEAL_GAS, VW_GAS, PR_GAS)
FLUID_MODEL= IDEAL_GAS
%
% Ratio of specific heats (1.4 default and the value is hardcoded
% for the model STANDARD_AIR)
GAMMA_VALUE= 1.4
%
% Specific gas constant (287.058 J/kg*K default and this value is hardcoded
% for the model STANDARD_AIR)
GAS_CONSTANT= 287.058
%
% Critical Temperature (273.15 K by default)
CRITICAL_TEMPERATURE= 273.15
%
% Critical Pressure (101325.0 N/m^2 by default)
CRITICAL_PRESSURE= 101325
%
% Critical Density (1.2886 Kg/m^3 by default)
CRITICAL_DENSITY= 1.2886
%
% Acentri factor (0.035 (air))
ACENTRIC_FACTOR= 0.592
%
% ---------------------- VISCOSITY MODEL -----------------------%
%
% Viscosity model (SUTHERLAND, CONSTANT_VISCOSITY).
%VISCOSITY_MODEL= CONSTANT_VISCOSITY
% Molecular Viscosity that would be constant (1.716E-5 by default)
MU_CONSTANT= 1.0563E-5
%
% Sutherland Viscosity Ref (1.716E-5 default value for AIR SI)
MU_REF= 1.716E-5
%
% Sutherland Temperature Ref (273.15 K default value for AIR SI)
MU_T_REF= 273.15
%
% Sutherland constant (110.4 default value for AIR SI)
SUTHERLAND_CONSTANT= 110.4
%
% -------------------------------- THERMAL CONDUCTIVITY MODEL ---------------------------------%
%
% Conductivity model (CONSTANT_CONDUCTIVITY, CONSTANT_PRANDTL).
CONDUCTIVITY_MODEL= CONSTANT_PRANDTL
%
% Molecular Thermal Conductivity that would be constant (0.0257 by default)
KT_CONSTANT= 0.025275
%
% ---------------------- BOUNDARY CONDITION DEFINITION -----------------------%
%
% Navier-Stokes wall boundary marker(s) (NONE = no marker)
MARKER_HEATFLUX= ( wall1, 0.0)
%
% Inlet boundary type (TOTAL_CONDITIONS, MASS_FLOW)
INLET_TYPE= TOTAL_CONDITIONS
%INLET_TYPE= MASS_FLOW
%
% Inlet boundary marker(s) with the following formats (NONE = no marker)
% Total Conditions: (inlet marker, total temp, total pressure, flow_direction_x, 
% flow_direction_y, flow_direction_z, ... ) where flow_direction is 
% a unit vector.
%
% Mass Flow: (inlet marker, density, velocity magnitude, flow_direction_x, 
% flow_direction_y, flow_direction_z, ... ) where flow_direction is 
% a unit vector.
MARKER_RIEMANN= (inflow, TOTAL_CONDITIONS_PT, 210111.6973, 354.9064919, 0.648015, 
-0.76163, 0.0, outflow, STATIC_PRESSURE, 162491.0374, 0.0, 0.0, 0.0, 0.0)
%
% Outlet boundary marker(s) (NONE = no marker)
% Format: ( outlet marker, back pressure (static), ... )
%MARKER_OUTLET= ( outflow, 1.909E+05 )
%MARKER_OUTLET= ( outflow, STATIC_PRESSURE, 1.01E+05, 0.0, 0.0, 0.0, 0.0)

% Periodic boundary marker(s) (NONE = no marker)
% Format: ( periodic marker, donor marker, rotation_center_x, rotation_center_y,
% rotation_center_z, rotation_angle_x-axis, rotation_angle_y-axis,
% rotation_angle_z-axis, translation_x, translation_y, translation_z, ... )
MARKER_PERIODIC= ( periodic1, periodic2, 0.0, 0.0, 0.0, 0.0, 0.0, 0.0, 0.0, 0.0447, 0.0)
%
% Marker(s) of the surface to be plotted or designed
%MARKER_PLOTTING= ( airfoil )
%
% Marker(s) of the surface where the functional (Cd, Cl, etc.) will be evaluated
MARKER_MONITORING= ( wall1)
%MARKER_SYM= ( periodic1, periodic2)
% ------------------------ DYNAMIC MESH DEFINITION -------------------------%
%
% Dynamic mesh simulation (NO, YES)
GRID_MOVEMENT= NO
%
% Type of dynamic mesh (NONE, RIGID_MOTION, DEFORMING, ROTATING_FRAME,
% MOVING_WALL, FLUID_STRUCTURE, AEROELASTIC, EXTERNAL)
GRID_MOVEMENT_KIND= MOVING_WALL
%
% Motion mach number (non-dimensional). Used for initializing a viscous flow
% with the Reynolds number and for computing force coeffs. with dynamic meshes.
%MACH_MOTION= 0.35
%
% Coordinates of the motion origin
MOTION_ORIGIN_X= 0.0
MOTION_ORIGIN_Y= 0.0
MOTION_ORIGIN_Z= 0.0
%
% Angular velocity vector (rad/s) about the motion origin
ROTATION_RATE_X = 0.0
ROTATION_RATE_Y = 0.0
ROTATION_RATE_Z = 0.00
% 1250 RPM --> 130.89969389957471
% 2500 RPM --> 261.79938779914943
%
% Translational velocity (m/s) in the x, y, & z directions
TRANSLATION_RATE_X = 0.0 0.0
TRANSLATION_RATE_Y = 313.32 313.32
A.3 SU2 configuration file

TRANSLATION_RATE_Z = 0.0 0.0
% -------------- COMMON PARAMETERS DEFINING THE NUMERICAL METHOD --------------% 
% 
% Numerical method for spatial gradients (GREEN_GAUSS, WEIGHTED_LEAST_SQUARES)
NUM_METHOD_GRAD= WEIGHTED_LEAST_SQUARES 
%
% Courant-Friedrichs-Lewy condition of the finest grid
CFL_NUMBER= 10.0
%
% CFL ramp (factor, number of iterations, CFL limit)
%CFL_RAMP= ( 1.0, 50, 3.0 )
%
% Number of total iterations
EXT_ITER= 7500
%
% Linear solver for the implicit formulation (BCGSTAB, FGMRES)
LINEAR_SOLVER= FGMRES
%
% Min error of the linear solver for the implicit formulation
LINEAR_SOLVER_ERROR= 1E-4
%
% Max number of iterations of the linear solver for the implicit formulation
LINEAR_SOLVER_ITER= 50

% -------------- MULTIGRID PARAMETERS -------------------------------%
%
% Multi-Grid Levels (0 = no multi-grid)
MGLEVEL= 0
%
% Multigrid pre-smoothing level
MG_PRE_SMOOTH= ( 1, 2, 3, 3 )
%
% Multigrid post-smoothing level
MG_POST_SMOOTH= ( 0, 0, 0, 0 )
%
% Jacobi implicit smoothing of the correction
MG_CORRECTION_SMOOTH= ( 0, 0, 0, 0 )
%
% Damping factor for the residual restriction
MG_DAMP_RESTRICTION= 0.75
%
% Damping factor for the correction prolongation
MG_DAMP_PROLONGATION= 0.75

% -------------- FLOW NUMERICAL METHOD DEFINITION -------------------------------%
%
% Convective numerical method (JST, LAX-FRIEDRICH, CUSP, ROE, AUSM, HLLC,
CONV_NUM_METHOD_FLOW = ROE

% Spatial numerical order integration (1ST_ORDER, 2ND_ORDER, 2ND_ORDER_LIMITER)
SPATIAL_ORDER_FLOW = 1ST_ORDER
SLOPE_LIMITER_FLOW = VENKATAKRISHNAN
LIMITER_COEFF = 0.5

AD_COEFF_FLOW = (0.15, 0.5, 0.02)

TIME_DISCRE_FLOW = EULER_IMPLICIT

CONV_NUM_METHOD_TURB = SCALAR_UPWIND

SPATIAL_ORDER_TURB = 2ND_ORDER_LIMITER
SLOPE_LIMITER_TURB = VENKATAKRISHNAN

TIME_DISCRE_TURB = EULER_IMPLICIT

CFL_REDUCTION_TURB = 1.0

KIND_ADAPT = PERIODIC

% -------------------------- GRID ADAPTATION STRATEGY --------------------------
% Kind of grid adaptation (NONE, PERIODIC, FULL, FULL_FLOW, GRAD_FLOW, FULL_ADJOINT, GRAD_ADJOINT, GRAD_FLOW_ADJ, ROBUST, FULL_LINEAR, COMPUTABLE, COMPUTABLE_ROBUST, REMAINING, WAKE, SMOOTHING, SUPersonic_SHOCK, TWOPHASE)

KIND_ADAPT = PERIODIC

% -------------------------- PARTITIONING STRATEGY --------------------------
% Write a tecplot/paraview file for each partition (NO, YES)
%VISUALIZE_PART= YES

% **************************** CONVERGENCE PARAMETERS ****************************
%
% Convergence criteria (CAUCHY, RESIDUAL)
% CONV_CRITERIA= RESIDUAL
% RESIDUAL_FUNC_FLOW= RHO_ENERGY
%
% Residual reduction (order of magnitude with respect to the initial value)
% RESIDUAL_REDUCTION= 6
%
% Min value of the residual (log10 of the residual)
% RESIDUAL_MINVAL= -16
%
% Start convergence criteria at iteration number
% STARTCONV_ITER= 10
%
% Number of elements to apply the criteria
% CAUCHY_ELEMS= 100
%
% Epsilon to control the series convergence
% CAUCHY_EPS= 1E-6
%
% Function to apply the criteria (LIFT, DRAG, NEARFIELD_PRESS, SENS_GEOMETRY,
% SENS_MACH, DELTA_LIFT, DELTA_DRAG)
% CAUCHY_FUNC_FLOW= DRAG
%
% Epsilon for full multigrid method evaluation
% %FULLMG_CAUCHY_EPS= 1E-4

% **************************** INPUT/OUTPUT INFORMATION ****************************
%
% Mesh input file
% MESH_FILENAME= mesh_out.su2
% MESH_FILENAME= su2mesh.su2
%
% Mesh input file format (SU2, CGNS, NETCDF_ASCII)
% MESH_FORMAT= SU2
%
% Divide rectangles into triangles (NO, YES)
% DIVIDE_ELEMENTS= NO
%
% Convert a CGNS mesh to SU2 format (YES, NO)
% %CGNS_TO_SU2= NO
% Mesh output file
MESH_OUT_FILENAME= mesh_out.su2
%
% Restart flow input file
SOLUTION_FLOW_FILENAME= restart_flow.dat
%
% Restart adjoint input file
SOLUTION_ADJ_FILENAME= solution_adj.dat
%
% Output file format (PARAVIEW, TECPLLOT, STL)
OUTPUT_FORMAT= TECPLLOT
%
% Output file convergence history (w/o extension)
CONV_FILENAME= history
%
% Output file restart flow
RESTART_FLOW_FILENAME= restart_flow.dat
%
% Output file restart adjoint
RESTART_ADJ_FILENAME= restart_adj.dat
%
% Output file flow (w/o extension) variables
VOLUME_FLOW_FILENAME= flow
%
% Output file adjoint (w/o extension) variables
VOLUME_ADJ_FILENAME= adjoint
%
% Output objective function gradient (using continuous adjoint)
GRAD_OBJFUNC_FILENAME= of_grad.dat
%
% Output file surface flow coefficient (w/o extension)
SURFACE_FLOW_FILENAME= surface_flow
%
% Output file surface adjoint coefficient (w/o extension)
SURFACE_ADJ_FILENAME= surface_adjoint
%
% Writing solution file frequency
WRT_SOL_FREQ= 500
%
% Writing convergence history frequency
WRT_CON_FREQ= 1
Radial-Inflow Turbine Input Files

B.1 Stator input file

--3D Blade specs--
No.ofProfiles 2
betaInDistType bspline
betaOutDistType bspline
chordDistType bspline
staggerDistType bspline
suctionDistType bspline
pressureDistType bspline
nIntProfiles 10
stackToMidProfile False

--camberline specs--
LE -0.0054 0.07422 0.0
betaIn -26.0
betaOut -28.0
chordAx 0.0265
stagger 27.5
w1 80
degree 2
optCp 1
t1 0.5
t2 0.5
pitch 0.02

--profile2D specs--
Suction cps

<table>
<thead>
<tr>
<th>x</th>
<th>y</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0000</td>
<td>5.0000</td>
</tr>
<tr>
<td>25.0000</td>
<td>20.14180</td>
</tr>
</tbody>
</table>
50.00000 16.87800
75.00000 -1.00000
100.00000 2.00000

Pressure cps

\[
\begin{array}{cc}
0.00000 & 8.00000 \\
30.00000 & 11.54915 \\
50.00000 & 9.23606 \\
75.00000 & 1.00000 \\
100.00000 & 2.00000 \\
\end{array}
\]

thFactorSuc  50
thFactorPres  80
pSuc  2
pPres  2
nintPSuc  10
nintPPres  10
CpDistSuc  EquiSpaced
CpDistPres  EquiSpaced
paramArcTanSuc  1.0
paramArcTanPres  1.0
optLe  BSpline
optTe  BSpline
wLe  80.0
wTe  2.0
concaveTol  2.0
xflaringAngle  0.0
intxflaring
yflaringAngle  0.0
intyflaring
cflaringAngle  0.0
intcflaring

--camberline specs--
LE  -0.0054 0.07422 0.00635
betaIn  -26.0
betaOut  -28.0
chordAx  0.0265
stagger  27.5
w1  80
degree  2
optCp  1
t1  0.5
t2  0.5
pitch  0.5

--profile2D specs--
Suction cps
 x  y
 0.00000  5.00000
 25.00000 20.14180
 50.00000 16.87800
 75.00000 -1.00000
100.00000  2.00000

Pressure cps
 x  y
 0.00000  8.00000
 30.00000 11.54915
 50.00000  9.23606
 75.00000  1.00000
100.00000  2.00000

thFactorSuc  50
thFactorPres  80
pSuc          2
pPres         2
nintPSuc      10
nintPPres     10
CpDistSuc     EquiSpaced
CpDistPres    EquiSpaced
paramArcTanSuc 1.0
paramArcTanPres 1.0
optLe        BSpline
optTe        BSpline
wLe          80.0
wTe          2.0
concaveTol   2.0
xflaringAngle 0.0
intxflaring  0.0
yflaringAngle 0.0
intyflaring  0.0
cflaringAngle 0.0
intcflaring  0.0

B.2 Rotor input file

-- blade 3D specs --
nProfiles    2
nBlades      16
upPar        50.0
loPar        50.0
tipClearance 10.0
hubPar 10.0

-- meridional channel specs --
rotorType turbine
xLe 0.0
dIn 0.0582
dOut 0.026
length 0.0389
hIn 0.00635
hOut 0.0216
sweep -10.3 deg
alphaOut 0 deg
alphaIn 0 deg
coneAngle 0 deg

-- camberline surface specs --
method interpolation
vdegree 2

-- 3D camberline specs --
height 0.0
degree 2
optCp 2
t1 60.0
t2 45.0
w1 100
rake 0 deg
betas -1.02576055139 4.68663936752 -3.4344269969 27.9375723039 40.1927587963

-- suction side specs --
thickness cps
   x   y
  0.0000  4.0000
  20.0000 10.0000
  39.0000 25.0000
  75.0000 10.0000
 100.0000  5.0000
thFactor 250.0
degree 2
nintP 10
optCpDist EquiSpaced
paramArcTan 1.0

-- pressure side specs --
thickness cps
   x   y
  0.0000  4.0000

B.2 Rotor input file

```
20.0000  10.0000
39.0000  25.0000
75.0000  10.0000
100.0000  5.0000
thFactor    250.0
degree      2
nintP       10
optCpDist   EquiSpaced
paramArcTan 1.0

-- 3D camberline specs --
height      100.0
degree      2
optCp       2
t1           46.0
t2           58.0
wl          100
rake        0 deg
betas       0.460582871371 4.61827830753 20.0290265068 57.8271424167 61.470232531

-- suction side specs --
thickness cps
   x    y
   0.0000  2.5000
  25.0000  4.0000
  50.0000  4.0000
  75.0000  3.0000
 100.0000  2.0000
thFactor    250.0
degree      2
nintP       10
optCpDist   EquiSpaced
paramArcTan 1.0

-- pressure side specs --
thickness cps
   x    y
   0.0000  2.5000
  25.0000  4.0000
  50.0000  4.0000
  75.0000  3.0000
 100.0000  2.0000
thFactor    250.0
degree      2
nintP       10
optCpDist   EquiSpaced
paramArcTan 1.0
```