How to Design and Optimize Turbine Blades

Author: Arman Arghavani – University of Waterloo Mechanical Engineering

This guide offers an open-source method for designing, analyzing, and optimizing small horizontal-axis wind turbine blades. The reference testing environment is a 3 ft by 3 ft collegiate wind tunnel, which offers a controlled and repeatable flow field suitable for evaluating small-scale rotors.

The guide covers:

- Tools required
- Outlining the specifications of the turbine
- Generating an airfoil in QBlade
- Creating and understanding a polar map
- Generating the 360° Polar (Polar Extrapolation)
- Generating the Full Blade Geometry
- Running Blade Element Moment Analysis
- Designing the Rotor
- Setting up a CFD environment
- Running and analyzing CFD results

Tools Required

Only three software tools are necessary for a complete aerodynamic and mechanical design workflow. All are free or have academic licenses.

1. QBlade

Used for airfoil creation, low Reynolds number polar generation, blade element momentum analysis, automatic blade generation, and geometry export.

2. Onshape

Used for designing the hub, nacelle, and overall turbine assembly, importing blade geometry from QBlade, preparing STEP files for CFD, and rapid CAD iteration.

3. Ansys Fluent or Ansys CFX

Used for three-dimensional aerodynamic simulation of the turbine, turbulence modeling, torque and power analysis, and comparisons between computational fluid dynamics predictions and wind tunnel measurements.

Together, these tools provide a complete workflow from aerodynamic design to optimization.

Outlining the Specifications of the Turbine

A turbine blade must be designed with respect to its operating environment. The following parameters describe the turbine configuration used throughout this guide.

---

1. Test Environment

- Test section size: 3 ft × 3 ft, or 0.9144 m × 0.9144 m
- Cross-sectional area: 0.836 m^2
- Airspeed in the test section: 5 m/s
- Air density: 1.225 kg/m^3
- Dynamic viscosity: 1.8×10^{-5} Pa × s

These values define the flow properties that determine the blade's operating Reynolds number.

---

2. Rotor Geometry

- Blade span: 0.25 m
- Hub radius: 0.025 m
- Total rotor radius: R = 0.25m + 0.025m = 0.275 m
- Rotor diameter: 0.55 m
- Swept area: $A=\pi R^2 = 0.238$ m^2
- Number of blades: 3

This rotor configuration fits well within the test section while maintaining the necessary aerodynamic clearance.

Why Rotor Size and Clearance Matter in a Wind Tunnel

Airflow inside a wind tunnel is not uniform across the entire test section. The quality of the air entering the rotor strongly determines turbine performance, and certain regions of the flow are less suitable for accurate aerodynamic testing. To ensure reliable and repeatable turbine behavior, the rotor must be sized so that the blades operate only in clean and uniform airflow.

The rotor used in this guide has:

- Rotor diameter:

$$D = 0.55 \text{ m}$$

- Wind tunnel width:

$$W = 0.914 \text{ m}$$

This means the rotor occupies approximately:

$$\text{Fraction of width} = \frac{0.55 \text{ m}}{0.914 \text{ m}} \approx 0.60$$

Therefore, the rotor uses about 60% of the tunnel width. This value lies within widely accepted aerodynamic guidelines, which recommend using no more than 60-70% of the test section width for small wind turbines. The reasons for this guideline are explained below.

1. Boundary Layer Near the Tunnel Walls

As air flows along the tunnel walls, friction at the surface causes the air to slow down and form a boundary layer. This layer has lower velocity and higher turbulence intensity than the core flow in the center. The farther the air travels downstream, the thicker this layer becomes.

If blade tips enter this region:

- Lift decreases because the local airspeed is lower
- Aerodynamic loading becomes uneven
- Tip vortices weaken or destabilize
- Torque and power measurements fluctuate

Ensuring that the rotor only interacts with the central free stream eliminates these inaccuracies.

2. Wall Interference and Tip Vortex Behavior

Blade tips generate strong vortices that influence lift and stability. When a blade tip approaches a solid wall, the wall modifies these vortices in a way that reduces the lift near the tip and increases aerodynamic losses. In severe cases, the wall can cause early or asymmetric stall.

Maintaining a clearance of several chord lengths between the blade tips and the tunnel wall prevents this type of interference. For the rotor in this guide:

$$\text{Side clearance} = \frac{0.914 \text{ m} - 0.55 \text{ m}}{2} \approx 0.182 \text{ m}$$

This provides more than enough spacing to avoid wall effects.

3. Blockage of the Test Section

A rotor that occupies too much of the tunnel area can produce blockage. This occurs when the rotor obstructs the flow and forces air to accelerate around it. The increased velocity artificially inflates the measured power, causing results that do not reflect real world performance.

Blockage is quantified as:

$$\text{Blockage ratio} = \frac{A_{\text{rotor}}}{A_{\text{tunnel}}}$$

For this turbine:

$$\begin{aligned} A_{\text{rotor}} &= \pi(0.275\ \text{m})^2 = 0.238\ \text{m}^2 \\ A_{\text{tunnel}} &= 0.836\ \text{m}^2 \\ \text{Blockage ratio} &= \frac{0.238\ \text{m}^2}{0.836\ \text{m}^2} = 0.284 \end{aligned}$$

Blockage ratios below 0.30 are generally acceptable for low-speed wind tunnel testing.

The chosen rotor stays just under this limit, which ensures that blockage effects remain small and that measured power is realistic.

Summary

These three aerodynamic concepts work together to explain why only part of the wind tunnel can be safely used by the rotor. The central region of the tunnel contains clean, uniform, low turbulence airflow. The airflow near the walls is slower, more turbulent, and less predictable. A rotor that fits entirely inside the clean flow region will produce accurate, repeatable aerodynamic results.

The selected rotor radius of 0.275 meters ensures that:

- All blade sections operate in high quality airflow
- Wall turbulence does not affect the tips
- Blockage remains low
- The turbine behaves predictably in both QBlade simulations and wind tunnel tests

This sizing approach is considered best practice in small wind turbine design and allows others to reproduce the turbine under the same controlled conditions.

---

3. Operating Tip Speed Ratio

A horizontal axis wind turbine operates under the combined influence of two velocities: the axial wind speed and the tangential speed created by the rotor's rotation. The ratio of these two velocities determines how the airfoil sections experience the flow and is known as the tip speed ratio. This parameter controls the angle at which the air meets the blade, the lift that each section generates, and the overall aerodynamic efficiency of the turbine.

The tip speed ratio is defined as:

$$\lambda = \frac{\omega R}{V_\infty}$$

where

- λ is the tip speed ratio
- ω is the rotor angular velocity in radians per second $(\mathrm{rad/s})$
- R is the rotor radius in meters (m)
- V_∞ is the wind speed in meters per second $(\mathrm{m/s})$

Choosing an appropriate tip speed ratio is essential because it establishes the inflow angle along the blade and ensures that the airfoil operates near its optimal aerodynamic angle of attack. Too low of a tip speed ratio causes stall and low torque, while too high of a tip speed ratio produces excessive drag, noise, and tip losses. Three bladed turbines typically operate most efficiently within the range of three to six.

For this turbine, a tip speed ratio of four was selected. This value provides a stable rotational speed for wind tunnel operation, keeps the airfoil within its effective low Reynolds number lift region, and produces consistent aerodynamic loading across the span.

Design calculations are based at seventy percent of the blade radius because this region produces most of the turbine's torque and reflects the most stable and representative aerodynamic conditions. The root is constrained by low tangential speed and structural requirements, while the tip is influenced by strong three-dimensional losses, making seventy percent radius the most reliable reference point.

Using the chosen values:

- $R = 0.275\ \mathrm{m}$
- $V_\infty = 5\ \mathrm{m/s}$
- $\lambda = 4$

the angular velocity becomes:

$$\omega = \frac{4 \times 5\ \mathrm{m/s}}{0.275\ \mathrm{m}} = 72.7\ \mathrm{rad/s}$$

The tangential speed at seventy percent radius is:

$$U_t(0.7R) = 72.7\ \text{rad/s} \times 0.1925\ \text{m} = 14.0\ \text{m/s}$$

The relative velocity seen by the blade at this location is then:

$$V_{rel} = \sqrt{U_t^2 + V_\infty^2} = \sqrt{(14.0\ \text{m/s})^2 + (5\ \text{m/s})^2} = 14.9\ \text{m/s}$$

This velocity directly establishes the operating Reynolds number and sets the aerodynamic conditions used for polar generation and blade design.

---

4. Reynolds Number Determination

Reynolds number is a dimensionless quantity that compares inertial forces in a flow to viscous forces. It is one of the key parameters that determines how an airfoil behaves. For turbine blades, Reynolds number controls whether the flow remains mostly laminar, transitions to turbulent, or separates from the surface. These regimes strongly affect lift, drag, stall, and therefore the efficiency of the rotor.

For a blade section, the local Reynolds number is defined as:

$$Re = \frac{\rho\, V_{rel}\, c}{\mu}$$

where

- Re is the Reynolds number (unitless)
- ρ is the air density in kilograms per cubic meter (kg/m^3)
- V_{rel} is the relative flow speed seen by the section in meters per second (m/s)
- c is the local chord length in meters (m)
- μ is the dynamic viscosity in pascal seconds ($\text{Pa} \cdot \text{s}$)

For this turbine in the wind tunnel:

- Air density:

$$\rho = 1.225\ \text{kg/m}^3$$

- Dynamic viscosity:

$$\mu = 1.8 \times 10^{-5}\ \text{Pa} \cdot \text{s}$$

- Relative velocity at seventy percent radius:

$$V_{rel} = 14.9 \text{ m/s}$$

Substituting these into the definition gives:

$$Re = \frac{1.225 \text{ kg/m}^3 \times 14.9 \text{ m/s} \times c \text{ (m)}}{1.8 \times 10^{-5} \text{ Pa} \cdot \text{s}}$$

Grouping the constants:

$$\frac{1.225 \times 14.9}{1.8 \times 10^{-5}} \approx 1.01 \times 10^6$$

so

$$Re \approx 1.01 \times 10^6 \cdot c$$

with $c$in meters.

This shows that, for this turbine at seventy percent radius, Reynolds number scales linearly with chord length.

Typical chord values for a small rotor in a 3 ft by 3 ft tunnel fall in the range of 0.03 m to 0.05 m. Using the expression above:

- For $c = 0.03$ m:

$$Re \approx 1.01 \times 10^6 \times 0.03 \approx 3.0 \times 10^4$$

- For $c = 0.04$ m:

$$Re \approx 4.0 \times 10^4$$

- For $c = 0.05$ m:

$$Re \approx 5.0 \times 10^4$$

The turbine therefore operates in a low Reynolds number regime between approximately 30,000 and 50,000 at the representative section.

In reality, Reynolds number varies along the span because both the relative velocity $V_{rel}(r)$and the chord $c(r)$change with radius. The root has lower relative velocity and often larger chord, while the tip has higher relative velocity and smaller chord. Using the seventy percent radius section as the reference provides a good approximation of the average aerodynamic conditions because this region contributes a large fraction of the torque and is less affected by root and tip losses.

Operating in the 30,000 to 50,000 Reynolds range has several important implications:

- The flow is very sensitive to laminar separation and small disturbances.
- Conventional high Reynolds number airfoils tend to perform poorly due to large laminar separation bubbles and high drag.
- Thin, moderately cambered airfoils, such as NACA 4408, provide better lift to drag ratio and more predictable stall behavior.
- Polar data must be generated specifically at these Reynolds numbers rather than at much higher values.

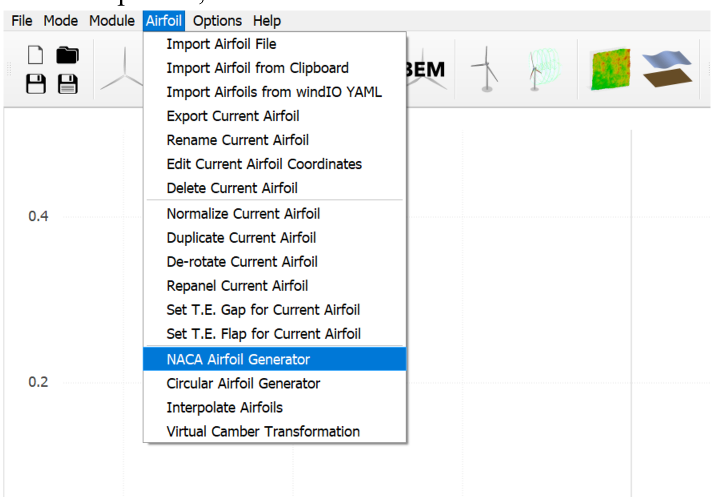
For this reason, when generating airfoil polars in QBlade, it is appropriate to create polar sets at Reynolds numbers such as 30,000, 40,000, and 50,000. These values correspond directly to realistic chord choices at the seventy percent radius section of the blade and will be used in later steps to design the chord and twist distributions and to perform blade element momentum analysis.

Generating an Airfoil in QBlade

Once the operating conditions and target Reynolds number range have been defined, the next step is to generate the airfoil geometry that will be used throughout the blade design process. QBlade includes a built-in generator for NACA four-digit airfoils, which allows precise control over camber, camber location, and thickness. For this turbine, the selected profile is NACA 4408, chosen because its moderate thickness and camber perform well in the 30,000 to 50,000 Reynolds number range.

To generate the airfoil:

1. Open **QBlade**
2. Select **New Project**
3. In the upper left corner, choose the **Airfoil** module
4. In the dropdown, select **NACA Airfoil Generator**



A dialog box will appear prompting for the NACA designation. Enter:

4408

This code corresponds to an airfoil with:

- 4% maximum camber
- Camber located at 40% of the chord
- 8% maximum thickness

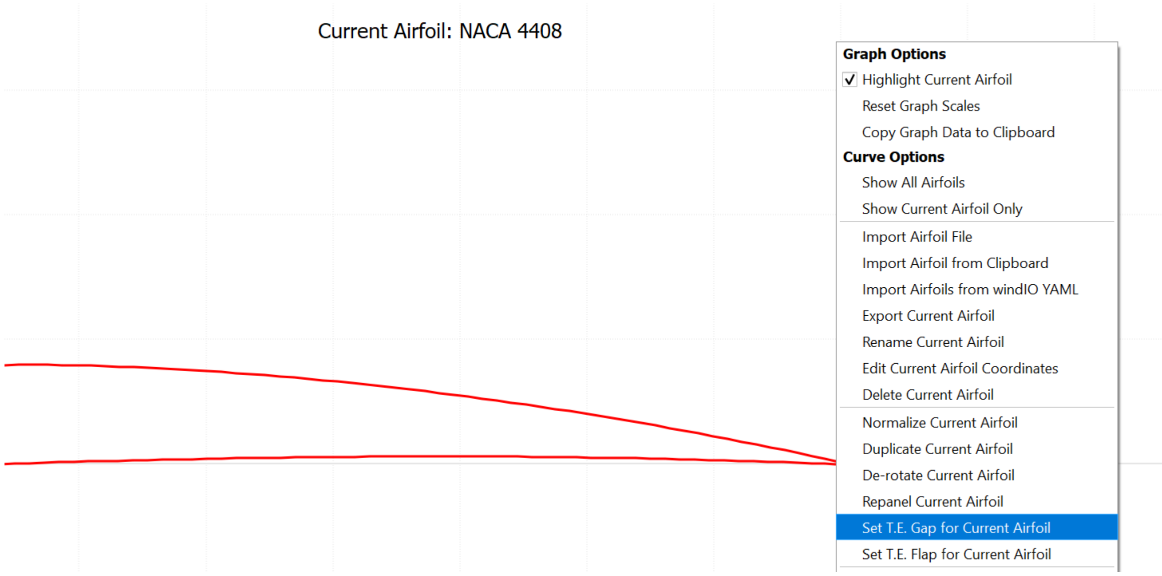
These parameters match the aerodynamic requirements for small rotors operating at low Reynolds numbers.

QBlade also asks for the **number of panels**, which determines how many line segments QBlade uses to approximate the airfoil surface. A higher panel count creates a more accurate airfoil but also increases computation time during polar generation and BEM simulation. The default value of **200 panels** offers a good balance between accuracy and speed, so it will be used for this guide.

After clicking generate, the airfoil will appear on the screen. QBlade highlights the newly created airfoil in red if the trailing edge is open. NACA airfoils are generated with a small numerical gap at the trailing edge, which is acceptable for analysis but must be closed before exporting the airfoil as a three dimensional model for CAD or 3D printing.

To close the trailing edge:

1. Right click on the airfoil
2. Select **Set T.E. Gap for Airfoil**



3. Set **T.E. Gap** to **0**

Once the trailing edge is closed, the airfoil will turn green, indicating that it is watertight and suitable for export to CAD software.

The airfoil is now ready for polar generation, which will be performed next and will establish the lift and drag characteristics used by QBlade to design the blade geometry.

Creating and Understanding a Polar Map in QBlade

Once the airfoil has been generated and its trailing edge has been closed, the next step is to create a polar map. A polar describes how the airfoil performs across a range of angles of attack at a specific Reynolds number. QBlade uses this information when generating blade geometry and running BEM simulations.

Follow these steps to create a polar:

Step 1: Open the Airfoil Analysis Module

1. Click the **Airfoil Analysis Module** icon (the red airfoil symbol).
2. Confirm that your airfoil (NACA 4408) is selected in the **Airfoils** dropdown.

Step 2: Create a New Polar

1. Under **Polar Controls**, click **New**.
2. A window will appear asking for the Reynolds number and other settings.

Step 3: Enter the Reynolds Number

For this turbine, the relevant Reynolds numbers are in the 30,000 to 50,000 range.
Start with:

$$Re = 30000$$

Leave Mach at 0 and keep the transition settings at default values.

Step 4: Set the Angle of Attack Sweep

Under **Analysis Settings**, enter:

- **Start** = -4 degrees
- **End** = 12 degrees
- **Δ** = 0.2 degrees

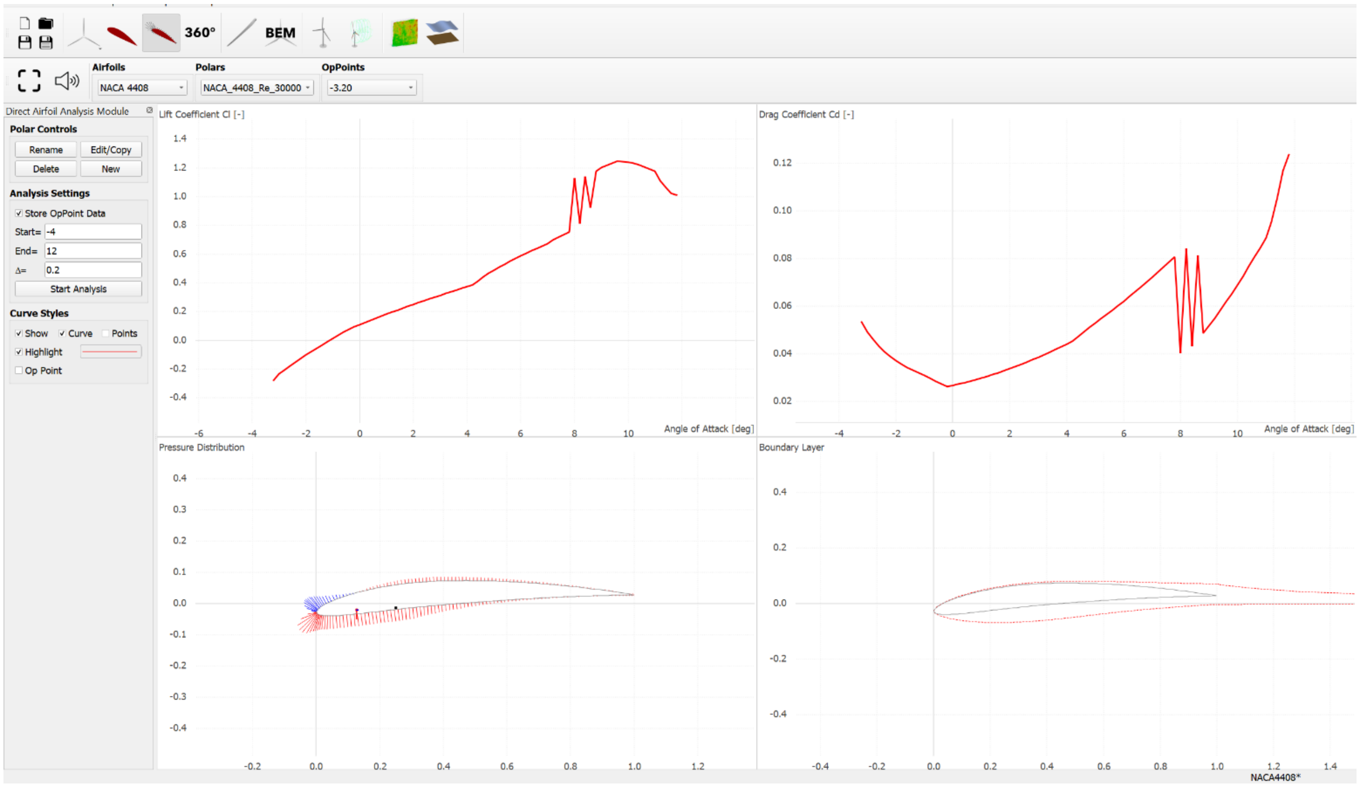
This sweep captures the linear lift region, the peak lift point, and the beginning of stall.

Step 5: Run the Analysis

1. Check **Store OpPoint Data**.

2. Click **Start Analysis**.

3. QBlade will generate the polar and display several key plots.



What the Polar Map Is Used For

A polar map is one of the most important aerodynamic tools in QBlade. Each polar contains the lift and drag characteristics of the airfoil across a range of angles of attack at a specific Reynolds number. This information forms the aerodynamic foundation for the entire blade design process. The polar determines how the airfoil behaves at different operating conditions and provides the data that QBlade uses when calculating forces, power, and blade geometry.

The polar map is used for the following purposes:

1. Determining the optimal angle of attack

The Cl and Cl/Cd curves identify the angle of attack where the airfoil produces high lift with low drag. This defines the target operating angle of attack used later when generating the twist distribution along the blade.

2. Identifying stall behavior and stall margin

The polar clearly shows where lift begins to decrease, indicating the onset of stall. Understanding this region ensures the blade operates safely below stall, improving stability and efficiency.

3. Evaluating aerodynamic efficiency

The Cl/Cd curve highlights how effectively the airfoil converts incoming wind energy into useful lift. Higher efficiency allows for thinner chords and improved turbine performance.

4. Providing lift and drag curves for BEM analysis

Blade Element Momentum (BEM) theory requires accurate lift and drag values at each angle of attack. QBlade uses the polar directly to compute aerodynamic forces, torque, axial induction factors, and total power output.

5. Designing chord distribution

The chord length at each blade radius depends on how much lift the airfoil can produce at the chosen operating angle of attack. The required lift values come directly from the polar.

6. Designing twist distribution

The twist distribution is selected so that each blade section operates at the most efficient angle of attack identified in the polar. This ensures consistent aerodynamic performance along the blade span.

7. Predicting startup performance

Small turbines often struggle to start at low wind speeds. The polar shows how lift behaves at small angles of attack and low Reynolds numbers, which helps determine whether the airfoil can generate enough torque to self-start.

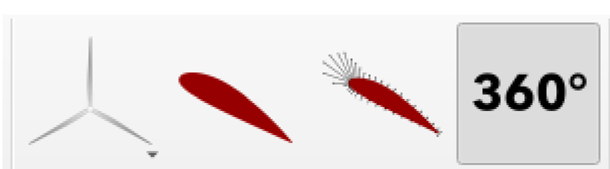
8. Comparing and selecting airfoils

By generating polars for different airfoils, the designer can directly compare lift, drag, stall behavior, and low Reynolds number performance. This helps identify the best airfoil for the turbine's specific operating conditions.

The polar map is therefore an essential step in turbine blade design, providing all the aerodynamic data needed for accurate simulation, optimization, and performance prediction.

Generating the 360° Polar (Polar Extrapolation)

After generating the two-dimensional airfoil polars, the next step is to create a 360° polar so the airfoil data can be used reliably in Blade Element Momentum simulations. The original 2D polar is only valid over a limited range of angles of attack and does not include post-stall behavior. A 360° polar extends this data to angles of attack from −180° to +180° while preserving the trustworthy low-angle aerodynamic information.



To create the 360° polar, navigate to the Polar Extrapolation Module by clicking the "360°" tab, located next to the Airfoil Analysis Module. With the correct airfoil and 2D polar selected, click "New" and choose Montgomerie Extrapolation. This method accounts for stall delay caused by blade rotation and is appropriate for turbines designed to operate below stall.

Before saving the extrapolated polar, limit the range of the original polar so that only the smooth, physically reliable portion of the 2D data is used. The upper limit should be set just before the lift curve becomes jagged or unstable near stall. This prevents numerical noise from being carried into the extrapolation.

The finetuning sliders control how the extrapolated polar blends with the original data. The A and B sliders adjust the smoothness of the transition into stall, ensuring the lift curve does not contain sharp spikes or kinks. The Cd 90 value defines the assumed drag coefficient at very high angles of attack and influences startup and off-design behavior. The St+ and St− values control the angular range over which the extrapolation is decomposed and blended. Small adjustments are usually sufficient, and the goal is a smooth, stable curve rather than a perfectly precise one.

Once the lift, drag, and moment curves appear smooth in the operating region, click Save to generate the 360° polar. QBlade will overlay the original 2D polar and the extrapolated polar by default so the transition can be visually verified. For clarity, the original 2D polar can be hidden, leaving only the final 360° polar displayed.

Repeat the Airfoil Analysis and 360° Polar Extrapolation for Reynolds numbers 40,000 & 50,000. At that point, the airfoil data is ready for use in Blade Element Momentum analysis and blade generation.

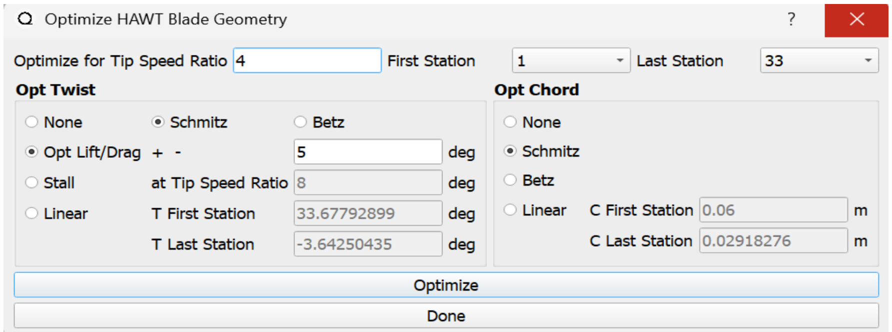
Generating the Full Blade Geometry

With the airfoil polars complete, the full blade geometry is generated using Blade Element Momentum theory. This process determines the chord and twist distribution along the blade so that each radial section operates near the chosen design angle of attack at the target tip speed ratio.

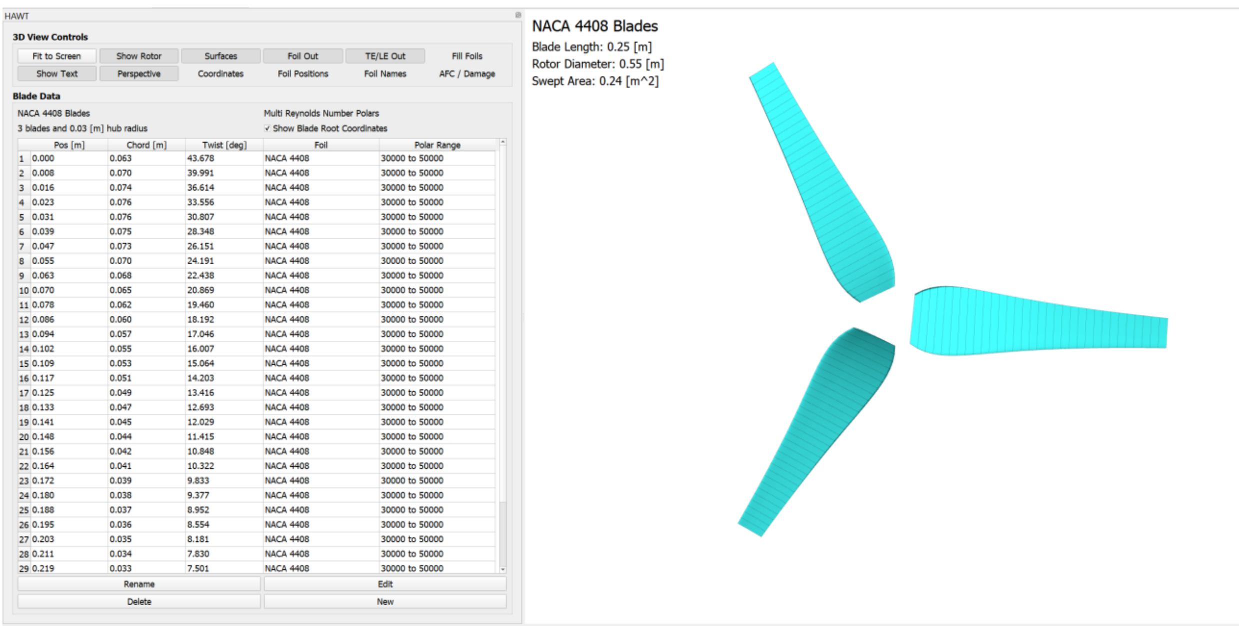
Open the **HAWT Blade Design** tab, then select **Blade Design** (located above the Airfoil Analysis Module), and click **Create New Blade Design**. Set the **hub radius** to **0.025 m** and the **blade length** to **0.25 m**, matching the physical constraints of the turbine.

To allow for a smooth geometric and aerodynamic transition along the blade, increase the blade resolution by adding stations. Use the **Insert Station** button to add stations incrementally until the blade contains **approximately 30 stations**. A good practice is to insert a station **every two existing stations**, which provides sufficient resolution without overcomplicating the model.

Define an initial chord distribution by setting a **base chord length of 0.06 m** and a **tip chord length of 0.02 m**. This linear taper produces a realistic blade shape, avoids excessive root chord growth, and places the mid-span chord in the range required to match the previously calculated Reynolds numbers (approximately 30,000 to 50,000).



With the blade discretized and the initial geometry defined, click on "Optimize" and apply **Schmitz optimization** for both chord and twist, using the selected design tip speed ratio and design angle of attack (approximately 5°). The resulting blade should exhibit a smooth taper in chord and a monotonic decrease in twist toward the tip. Visual inspection is used to confirm that no sudden jumps or nonphysical features are present. Select "show rotor" to view all 3 blades and their relative positions.



	Pos [m]	Chord [m]	Twist [deg]	Foil	Polar Range
1	0.000	0.063	43.678	NACA 4408	30000 to 50000
2	0.008	0.070	39.991	NACA 4408	30000 to 50000
3	0.016	0.074	36.614	NACA 4408	30000 to 50000
4	0.023	0.076	33.556	NACA 4408	30000 to 50000
5	0.031	0.076	30.807	NACA 4408	30000 to 50000
6	0.039	0.075	28.348	NACA 4408	30000 to 50000
7	0.047	0.073	26.151	NACA 4408	30000 to 50000
8	0.055	0.070	24.191	NACA 4408	30000 to 50000
9	0.063	0.068	22.438	NACA 4408	30000 to 50000
10	0.070	0.065	20.869	NACA 4408	30000 to 50000
11	0.078	0.062	19.460	NACA 4408	30000 to 50000
12	0.086	0.060	18.192	NACA 4408	30000 to 50000
13	0.094	0.057	17.046	NACA 4408	30000 to 50000
14	0.102	0.055	16.007	NACA 4408	30000 to 50000
15	0.109	0.053	15.064	NACA 4408	30000 to 50000
16	0.117	0.051	14.203	NACA 4408	30000 to 50000
17	0.125	0.049	13.416	NACA 4408	30000 to 50000
18	0.133	0.047	12.693	NACA 4408	30000 to 50000
19	0.141	0.045	12.029	NACA 4408	30000 to 50000
20	0.148	0.044	11.415	NACA 4408	30000 to 50000
21	0.156	0.042	10.848	NACA 4408	30000 to 50000
22	0.164	0.041	10.322	NACA 4408	30000 to 50000
23	0.172	0.039	9.833	NACA 4408	30000 to 50000
24	0.180	0.038	9.377	NACA 4408	30000 to 50000
25	0.188	0.037	8.952	NACA 4408	30000 to 50000
26	0.195	0.036	8.554	NACA 4408	30000 to 50000
27	0.203	0.035	8.181	NACA 4408	30000 to 50000
28	0.211	0.034	7.830	NACA 4408	30000 to 50000
29	0.219	0.033	7.501	NACA 4408	30000 to 50000

The blade geometry is now ready for performance verification using BEM simulation.