Centrifugal pump design: Three benchmark problems for many-objective optimization

Rasmus K. Ursem

Structural and Fluid Mechanics, R&T
Grundfos Management A/S
Poul Due Jensens Vej 7, DK-8850 Bjerringbro, Denmark
ursem@daimi.au.dk

Technical report no. 2010-01

Table of contents

1 Introduction

The design of centrifugal pumps at Grundfos is performed in three stages. First, the main geometrical parameters are found by means of a so-called 1-dimensional simulator. Second, the main geometry is extended to 3D and simulated using computational fluid dynamics (CFD). Third, a prototype is printed in plastic or metal and tested in a test-rig. Expenses rise rapidly through these stages. A 1D design takes less than a second to simulate, a 3D simulation takes approx 2-4 man-hours and 10-20 simulation hours, and a prototype costs around 5000 Euro and also at least 6-8 hours of testing time. Often these activities are iterated and sometimes it is even necessary to step back in the process. Thus, time and money can be saved if the 1D design is highly accurate and incorporates "rules-of-thumb" normally used in 3D design.

This technical report outlines three 1D pump design problems - two single stage pumps and a multi stage pump. Certain details are deliberately left out for reasons of business confidentiality. However, the report should contain enough theory, equations and pointers to other literature for you to implement a 1D pump simulator yourself. Alternatively, Concepts NREC offers a commerical simulator called Pumpal.

2 Basic pump theory

The purpose of this section is to describe the theoretical foundation of energy conversion in a centrifugal pump. Despite advanced calculation methods which have seen the light of day in the last couple of years, there is still much to be learned by evaluating the pump's performance based on fundamental and simple models. The section only provides a quick introduction. Please refer to e.g. Gülich [1] or Tuzson [2] for a more elaborate description.

Energy is added to the shaft in the form of mechanical energy when the pump is turned on. The impeller converts the mechanical energy to internal (static pressure) and kinetic energy (velocity). The process is described through Euler's pump equation, see section 2.2. By means of velocity triangles for the flow in the impeller in- and outlet, the pump equation can be interpreted and a theoretical loss-free head and power consumption can be calculated. Velocity triangles can also be used for prediction of the pump's performance in connection with changes of e.g. speed, impeller diameter and width.

2.1 Velocity triangles

For fluid flowing through an impeller, it is possible to determine the absolute velocity (C) as the sum of the relative velocity (W) with respect to the impeller and the tangential velocity of the impeller (U). These velocity vectors are added through vector addition, forming velocity triangles at the in- and outlet of the impeller. The relative and absolute velocity are the same in the stationary part of the pump.

An example of velocity triangles is shown in figure 1. Here U describes the impeller's tangential velocity while the absolute velocity C is the fluid's velocity compared to the surroundings. The relative velocity W is the fluid velocity compared to the rotating impeller. The angles α and β describe the fluid's relative and absolute flow angles respectively compared to the tangential direction.

The vectors in the velocity triangles can be calculated from the actual flow (Q), the rotation speed (n) and the geometric properties of the impeller. In short, the flow (Q) must pass through the cross section areas at leading edge A_1 and at trailing edge A_2, which are calculated by revolving the leading and trailing edges around the axial centerline. Assuming no pre-rotation ($C = C_{1m}$), the vectors are:

$$C_1 = C_{1m} = \frac{Q}{A_1} \quad (1)$$

$$U_1 = 2\pi r_1 \cdot n = r_1 \cdot \omega \quad (2)$$

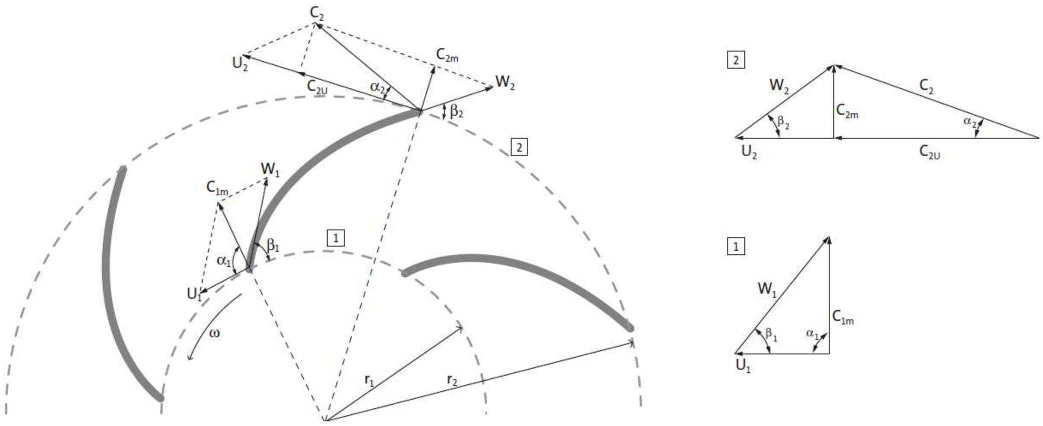

Figure 1: Velocity triangles.

where r_1 is the radius of the leading edge and ω is the angular velocity. Likewise, for trailing edge.

$$C_{2m} = \frac{Q}{A_2} \tag{3}$$

$$U_2 = 2\pi r_2 \cdot n = r_2 \cdot \omega \tag{4}$$

In the beginning of the design phase, $\beta_{2,flow}$ is assumed to have the same value as the blade angle β_2. The relative velocity (W_2) and tangential part of the absolute velocity (C_{2u}) can then be calculated as:

$$W_2 = \frac{C_{2m}}{\sin\beta_2} \tag{5}$$

$$C_{2u} = U_2 - \frac{C_{2m}}{\tan\beta_2} \tag{6}$$

Having these, it is straight-forward to derive the remaining vectors.

2.2 Euler's pump equation

Euler's pump equation is the most important equation in connection with pump design. The equation expresses the theoretical loss-free head, i.e., the maximal head that may be achieved with a given geometry. Applying the cosine relations the velocity triangles, Euler's pump equation can be written as the sum of the three contributions:

- Static head as consequence of the centrifugal force (U_1 and U_2).
- Static head as consequence of the velocity change through the impeller (W_1 and W_2).
- Dynamic head (C_1 and C_2).

Euler's pump eqution is then:

$$H_{th} = \frac{U_2 - U_1}{2g} + \frac{W_1 - W_2}{2g} + \frac{C_2 - C_1}{2g} \tag{7}$$

where g is the gravitational acceleration.

In the derivation of Euler's pump equation it is assumed that the flow follows the blade. In reality this is, however, not the case because the flow angle usually is smaller than the blade angle. This condition is called slip. Nevertheless, there is close connection between the flow angle and blade angle. An impeller has an infinite number of blades which are infinitly thin, then the flow lines will have the same shape as the blades. The flow will not follow the shape of the blades completely in a real impeller with a limited number of blades with finite thickness. The tangential velocity out of the impeller as well as the head is reduced due to this. In the impeller design phase, you have to include the difference between flow angle and blade angle. This is done by including empirical slip factors in the calculation of the velocity triangles. Numerous slip models have been suggested in the literature, see for example [1, 2]. It is important to emphasize that slip is not a loss mechanism but just an expression of the flow not following the blade.

The theoretical hydraulic power ($P_{hyd,th}$) can be calculated from the theoretical head H_{th} (equation 7):

$$P_{hyd,th} = Q \cdot H_{th} \cdot \rho \cdot g \tag{8}$$

where ρ is the viscosity of the fluid, and g is the gravitational acceleration.

In a loss-free pump, the theoretical shaft power $P_{2,th}$ equals the theoretical hydraulic power $P_{hyd,th}$. The shaft power is the power transferred from the motor to the impeller and the hydraulic power is the power transferred from the impeller to the fluid.

2.3 Head and power losses

Obviously, no pump is loss-free. In short, the real head H is calculated from the theoretical Euler head H_{th} by subtracting various losses such as wall friction, shock losses, recirculation losses, etc. Likewise, the real shaft power P_2 is calculated by adding disc friction and bearing friction to the theoretical shaft power $P_{2,th}$. In addition to this, a leak flow near the impeller inlet affects both. Leak flow is a small additional flow the impeller has to pump and it occurs as a consequence of small seal gap between the rotating impeller and stationary pump house. First step is to calculate a leak-flow including theoretical head and shaft power. This may be done from the equations in section 2.1 using a leak including flow $Q = Q_{leak} = Q_{th} + Q_{loss}$, where Q_{th} is the flow to simulate at and Q_{loss} is the leak flow predicted by a leak model. The loss-including head H and shaft power P_2 is calculated as:

$$H = H_{leak} - \sum_{i=1}^{|losses|} H_{loss,i} \tag{9}$$

$$P_2 = P_{2,leak} + \sum_{i=1}^{|losses|} P_{loss,i} \tag{10}$$

where H_{leak} and $P_{2,leak}$ are the theoretical head and power including leak flow. Having the loss-including head, the corresponding hydraulic power may be calculated as:

$$P_{hyd} = Q_{th} \cdot H \cdot \rho \cdot g \tag{11}$$

Notice that Q_{th} is used here and Q_{leak} is used in calculating P_2. This models the additional loss caused by leak flow. It should also be noted that many of the loss models depend on the velocity triangles. Consequently, the calculation must be done iteratively starting with zero leak. In the next iteration, the leak flow model may alter the velocity triangles a bit and thus the losses, which calls for another iteration. After typically 10-20 iterations, the losses have settled and the simulation is converged. After convergence, the hydraulic efficiency may be calculated as:

$$\eta_{hyd} = \frac{P_{hyd}}{P_2} \tag{12}$$

As seen, the hydraulic efficiency express the percentage of the power transferred from the shaft to the fluid. A full pump curve may be simulated by calculating H, P_2, η_{hyd}, etc. for different flows Q_{th} and then compose the performance curves from these values.

In this simulation, the challenging part is to choose the loss models that correctly represent the pump being modeled. As mentioned in the introduction, these details must be left out for business confidentiality. However, the Grundfos simulator consist largely of text book loss models along with a number of internally developed models. It is possible to find a sufficient number of loss models in the literature. In short, you will need:

- A slip model.
- A leak flow model.
- A disc friction model.
- A number of wall friction models.
- A shock model.
- A diffusion model.
- a number of expansion models.

3 Centrifugal pump design problems

In addition to the aforementioned performance curve calculations, the Grundfos simulator also incorporates two rules-of-thumb for 3D simulation. First, hub- and shroud-plates of the impeller and the crossover guiding vane are modelled in 2D as splines. This allows minimization of curvatures since high curvatures are known to result in recirculation zones and thus losses. Second, the smoothness of the cross-section areas seen by the fluid is also calculated. This smoothness should be maximized since abrupt changes are known to cause expansion losses and recirculation zones. The Grundfos simulator is thus not strictly 1D since it incorporates some elements of 3D simulation.

3.1 Objectives and constraints

The pump optimization cases are based on the constraints $C_1 - C_6$ and the objectives $F_1 - F_8$ of which all are maximization except F_5. In this, BEP is the "best efficiency point" (maximal η_{hyd} on the curve) and NFP is "number of flow points in curve".

$$F_1 = \eta_{hyd,BEP} \quad \text{Efficiency at BEP.} \tag{13}$$

$$F_2 = \frac{1}{NFP}\sum_{i=1}^{NFP} \eta_{hyd,i} \quad \text{Average efficiency on curve.} \tag{14}$$

$$F_3 = H_{BEP} \quad \text{Head at BEP.} \tag{15}$$

$$F_4 = Q_{BEP} \quad \text{Flow at BEP.} \tag{16}$$

$$F_5 = \begin{cases} P_2[\text{la}] - P_2[\text{la-1}] & \text{if mx = la} \\ P_2[\text{la}] - 0.99 \cdot P_2[\text{mx}] & \text{if } P_2[\text{la}] > 0.99 \cdot P_2[\text{mx}] \\ 0 & \text{otherwise} \end{cases} \tag{17}$$

$$F_6 = \text{CS}_{smooth} \quad \text{Smooth cross-section areas.} \tag{18}$$

$$F_7 = \frac{1}{\max(C_{shr})} \quad \text{Radius of curvature on shroud plate.} \tag{19}$$

$$F_8 = \frac{1}{\max(C_{hub})} \quad \text{Radius of curvature on hub plate.} \tag{20}$$

The 'la' and 'mx' in objective F_5 denotes the index of the last flowpoint and the index of the flowpoint having the maximal shaft power P_2. The objective F_5 ensures that the power curve does not rise at the end.

In objective F_6, the smoothness measure CS_{smooth} represents how ideal the cross section areas develop through the impeller or crossover guiding vane. In 3D simulation, it is known that sudden changes in cross section area will introduce recirculation zones and thus losses. The optimal design is achieved by a gradual and smooth change in cross section areas. Hence, CS_{smooth} represents the deviation from an ideal linear change in cross section area from impeller inlet to impeller exit (or likewise for the crossover guiding vane). The maximal smoothness is 100% and a good impeller has typically a smoothness above 90%. Figure 2 displays three lines in green (left impeller) that examplifies how the cross section areas are calculated. Each line is revolved around the axial centerline creating a truncated cone of which the curved surface represents the cross section area.

The objectives F_7 and F_8 captures another phenomena in 3D design. It is desireable if the fluid does not have to make sharp turns in it's passage through the pump. A sharp turn will most likely create a recirculation zone as sometimes seen in streams after sharp bends. Figure 2 illustrates the meridional cuts of two impellers. The left impeller has low radius of curvature on the shroud plate. Here, the fluid makes a smooth 90° turn, which will most likely not create a recirculation zone in 3D. In contrast, the impeller on the right is not as high in the axial direction which may be a design constraint. This design is more likely to create a recirculation zone after the sharp turn near the leading edge.

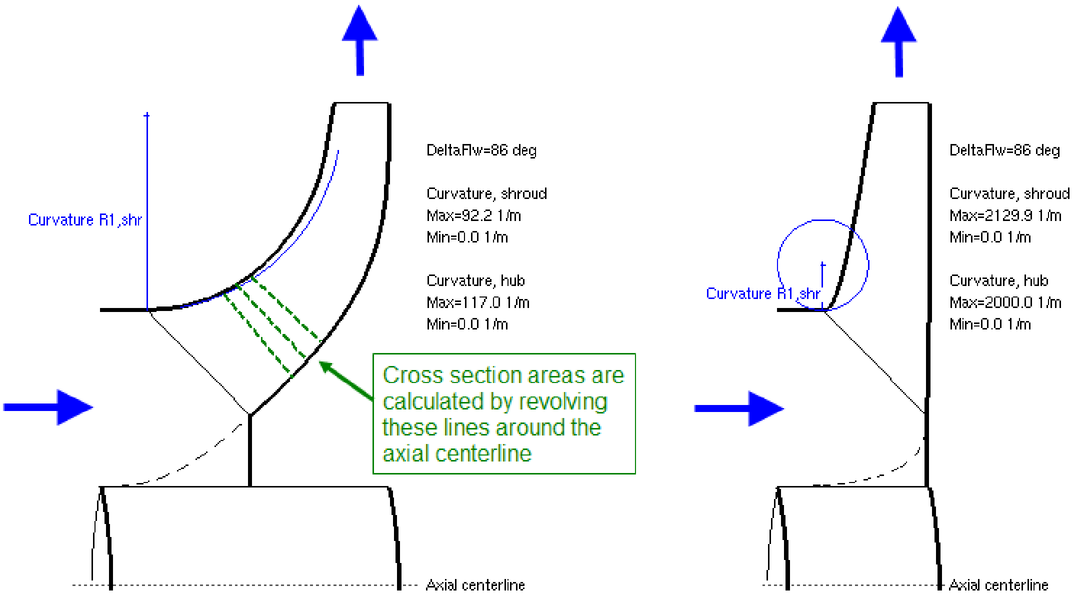

Figure 2: Meridional cuts of two impellers.

The shared constraints are:

$$
\begin{array}{lcll}
C_1 & : & Q_{min} \leq Q_{BEP} \leq Q_{min} & \text{BEP in range.} \\
C_2 & : & 0.5 \leq \delta \leq 9 & \text{Diffusor opening angle valid.} \\
C_3 & : & \min(C_{shr}) \geq 0 & \text{Shroud-plate valid.} \\
C_4 & : & \min(C_{hub}) \geq 0 & \text{Hub-plate valid.} \\
C_5 & : & \max(P_2) \leq 100 & \text{Limit shaft power.} \\
C_6 & : & \dfrac{P_{2,Qmax}}{P_{2,_{BEP}}} \leq 0.99 & \text{Drop in } P_2 \text{ at } Q_{max}.
\end{array}
$$

The constraint C_2 respesents a rule-of-thumb regarding opening angle of conical diffusors. In single-stage pumps, the exit part of the volute is typically a conical diffusor and C_2 ensures the design does not produce a poorly performing diffusor. The constraints C_3 and C_4 ensures that sign of the curvature on the shroud and hub plates is always positive. In other words, that the plates only bend "one way" or is convex. Finally, C_6 models the same phenomena as objective F_5 and is introduced to put extra selection pressure on this.

3.2 Case 1: Single-stage pump with $Q_{BEP} = 5.0$

Test case 1 represents a single stage pump with inline inlet, bladed impeller and a volute with rectangular cross section. The project objective is to find pumps having different heads. The tradeoff is here between max head and hydraulic efficiency. The problem has 27 design parameters, 6 objectives $F = [F_1, F_2, F_3, F_6, F_7, F_8]$, and 4 constraints $C = [C_1, C_2, C_3, C_4]$ with $Q_{min} = 4.8$ and $Q_{max} = 5.2$. The parameters are all related to the geometry, i.e, radii, angles, and cross section areas, of the inlet (3 parameters), the impeller (21 parameters), and the volute (3 parameters).

3.3 Case 2: Multi-stage pump with $Q_{BEP} = 50.0$

Test case 2 represents a multistage pump with bladed impeller (I) and crossover guiding vane (C). The project objective is to maximize the head per stage since this would reduce the number of stages needed to produce the total pump head. For example, 100m head may be achieved with 5 stages of 20m each or 10 stages with 10m each. Fewer stages result in a lower price, but each stage may have a lower efficiency as a result. Thus, the trade-off is between price and efficiency. The problem has 63 parameters, 9 objectives $F = [F_1, F_2, F_3, F_6(I), F_7(I), F_8(I), F_6(C), F_7(C), F_8(C)]$, and 5 constraints $C = [C_1, C_3(I), C_4(I), C_3(C), C_4(C)]$ with $Q_{min} = 49$ and $Q_{max} = 51$. The parameters are all related to the geometry, i.e, radii, angles, and cross section areas, of the the impeller (21 parameters) and the crossover guiding vane (42 parameters).

3.4 Case 3: Single-stage pump with $P_2 \leq 100$W

Test case 3 represents a single stage pump with inline inlet, bladed impeller and a volute with rectangular cross section. The project objective here is to find a number of pumps that exploit 100W in different ways. The problem has 28 design parameters, 8 objectives $F = [F_1, F_2, F_3, F_4, F_5, F_6, F_7, F_8]$, and 6 constraints $C = [C_1, C_2, C_3, C_4, C_5, C_6]$ with $Q_{min} = 4.8$ and $Q_{max} = 5.2$. The parameters are all but one related to the geometry, i.e, radii, angles, and cross section areas, of the inlet (3 parameters), the impeller (21 parameters), and the volute (3 parameters). The last parameter is the desired design flow.

4 Conclusions

This technical report provides a quick introduction to pump theory and how to implement a one-dimensional solver. Following this, three many-objective benchmark problems are described. Unfortunately, I am not allowed to provide full details of the simulator as this would violate requirements on business confidentiality. However, the report should provide enough material to construct a simulator that can support the many-objective problems. Alteratively, a commercial simulator called Pumpal is available from Concepts NREC. However, this simulator does not model the curvature and cross sections of the impeller and guding vanes.

References

[1] J. F. Gülich, *Kreiselpumpen*, 2nd edition. Springer, 2004, ISBN 3-540-40587-9.

[2] J. Tuzson, *Centrifugal pump design.* John Wiley & Sons, 2000, ISBN 0-471-36100-3.