Development of Scaling Approaches for Hydro Generator Cooling Flow

David Noelle
Research Assistant
Technische Universität Dresden
01062, Dresden, Germany

Bastian Diebel
Development Engineer
Voith Hydro Holding GmbH & Co. KG
89522, Heidenheim, Germany

Thilo Dauch
Head of Fluid Mechanics
Voith Hydro Holding GmbH & Co. KG
89522, Heidenheim, Germany

Abstract
This paper presents the development and evaluation of scaling approaches (SAs) for the cooling flow of a large rim ventilation hydro generator. The SAs are intended for designing a model of this generator to validate the machine's pressure rise and drop characteristics.
The research begins with the establishment and selection of similarity numbers that are relevant to the hydro generator cooling flow. This foundational step provides a framework for the development of SAs, ensuring that the chosen parameters accurately reflect the critical aspects of the flow problem. The strategies aim to either match the identified similarity numbers or achieve pragmatic compromises between them. Two of the selected similarity numbers are not established in literature yet. They apply for the rotating parts of the generator. Therefore, the investigations focus on this domain. The flow domain is split into two distinct sections. The first section is the flow into the rotor hub, a large sectioned chamber around the generator shaft. The second section includes the flow in the radial rim channels, the pole gaps, and cooling channels in the stator. This flow section is referred to as the homogeneous section.
The SAs are compared to each other by evaluating CFD simulations of scaled models of both flow sections. The SAs are applied across a range of model scaling factors from 1:5 to 1:20. The evaluation is focused on two metrics. Firstly, the pressure rise and drop as crucial ventilation parameters. Secondly, the power demand is analyzed, as it is essential for it to be within a reasonable range for a model test.

Introduction
The effective cooling of hydroelectric generators is essential to maintain operational temperatures. The cooling is primarily facilitated through air ventilation driven by the rotor. Two primary methods are used for cooling circuit design: classical network calculations and computational fluid dynamics (CFD) simulations. For both methods, a validation is possible through experimental studies, utilizing a down-scaled model of the generator. Despite there being research on the topic of generator ventilation models for serval decades, there is a lack of openly available literature regarding the down scaling process itself.
The majority of research in this field focusses on the numerical investigations of real size generator ventilation [1-3]. A short overview on the existing model based research shall be given here. For research older than 15 years, Boglietti [4] provides a comprehensive historical overview of numerical simulations addressing generator flow problems. However, experimental data from scaled models are only occasionally used to supplement CFD results. Moradnia et al. [5] compare experimental results from a scaled model with CFD simulations, but their success in obtaining accurate quantitative results is limited. Over the past decade, the research institute of Hydro-Québec (IREQ) has been a leading group in the field, operating a 1:4 scaled model of a hydroelectric generator to provide validation data for CFD calculations [6]. Their experimental setup mirrors the conditions in this paper, specifically focusing on cold conditions without heat transfer. This model has also been utilized for advanced measurements, such as particle image velocimetry (PIV) [7]. In contrast, Jamshidi et al. [8] place greater emphasis on model scaling, combining scaled generator model tests with CFD simulations. Recent work by Diebel et al. [10] contributes to the experimental validation of generator flow CFD simulations, though it does not consider scaling in its analysis.
The contribution of this paper is the direct investigation of the scaling itself, that is required to design a ventilation model of a radial rim cooled hydro generator. Since this model is used for pressure rise and drop measurements only, the heat transfer is not included. A top-down nondimensionalization approach is used to derive a unique set of similarity numbers tailored to the specific characteristics of generator cooling by nondimensionalizing the volume forces in fluid mechanics laws. Based on selected similarity numbers, various scaling approaches (SAs) are developed. A SA is a collection of scaling strategies that define different scaling factors of the model. These scaling factors can describe the ratio of geometrical parameters as well as the ratio of flow parameters from original to model. The SAs are compared using CFD simulations in discrete sections of the generator flow.

In a rim ventilation machine, also known as radial-radial ventilation machine the cooling air flows into the rotor hub axially. From there, the air flow follows radial ducts through the rotor rim into the gap between the poles, where the pole windings are cooled. Inside the air gap between rotor and stator the flow loses its tangential component that originates from the rotation. It then enters radial cooling ducts in the stator core. The rotor rotation itself drives the cooling flow, there are no additional fans. For an original generator a heat exchanger follows the stator before the cooling circuit is closed to a loop. The backflow of this loop passes the winding ends of the stator. The heat exchanger is not part of ventilation models that are exclusively built for pressure rise and drop measurements. Instead, the ventilation is an open system where the volume flow is measured upstream or downstream of the scaled model.

1 Similarity Numbers

To obtain the similarity numbers required for developing SAs, a two-step process is applied: Firstly, the flow problem must be described mathematically. Secondly, this description is nondimensionalized, resulting in the definition of similarity numbers that must be satisfied during scaling. Since heat transfer is not considered in the model test, the equation of momentum conservation leads to all relevant similarity numbers. The Navier-Stokes equation is used in its convective form:

$$\rho \frac{D\vec{v}}{Dt} = \rho \left[\frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla)\vec{v}\right] = -\nabla p + \eta \nabla^2 \vec{v} + \left(\xi + \frac{\eta}{3}\right) \nabla(\nabla \cdot \vec{v}) + \vec{F} \tag{1}$$

This equation describes the fluids motion due to conservation of momentum. For further information, see [11]. The flow problem of generator ventilation includes rotating structures, so Eq. 1 is transferred to a rotating reference frame, which is indicated by primed variables. Due to the rotational speed $\vec{\omega}$, the volume force term $\vec{F}$ in Eq. 1 does not only include the gravitational volume force $\vec{F}_g$, but it is extended with terms for the centrifugal acceleration $\vec{F}'_{cf}$ and Coriolis acceleration $\vec{F}'_{Co}$:

$$\vec{F}' = \vec{F}'_g + \vec{F}'_{cf} + \vec{F}'_{Co} = -\vec{g}\rho - \rho\vec{\omega} \times (\vec{\omega} \times \vec{x}') - 2\rho\vec{\omega} \times \vec{v}' = \begin{pmatrix} 0 \\ 0 \\ -g\rho \end{pmatrix} + \begin{pmatrix} \rho\omega^2 r' \\ 0 \\ 0 \end{pmatrix} + \begin{pmatrix} 2\rho\omega v_{\phi'} \\ 2\rho\omega v_{r'} \\ 0 \end{pmatrix} \tag{2}$$

Nondimensionalization is achieved by replacing all dimensional variables with the ratio of a dimensionless equivalent to a characteristic dimensional variable. This method is well-established and will not be elaborated on in this paper. The nondimensionalized form of Eq. 2 in Eq. 1 contains all relevant similarity numbers of the flow. To indicate that a variable is dimensionless, an asterisk (*) is used.

$$\begin{gathered} Sr \frac{\partial \vec{v}^*}{\partial t^*} + (\vec{v}^* \cdot \nabla^*)\vec{v}^* = -Eu\nabla^* p^* + \frac{1}{Re}\nabla^{*2}\vec{v}^* + \left(\frac{1}{3Re} + \frac{1}{Re_b}\right)\nabla^*(\nabla^* \cdot \vec{v}^*) \\ + \frac{1}{Fr}\vec{F}_g^* + Cf\vec{F}_{cf}^* + Ro_r\vec{F}_{Co,r}^* + Ro_\phi\vec{F}_{Co,\phi}^* \end{gathered} \tag{3}$$

The appearing Strouhal number Sr, the Euler number Eu, the Reynolds numbers for shear viscosity Re and for bulk viscosity Re_b and the Froude number Fr are already well established in literature [12].
The Strouhal number is not relevant for stationary flow. The Euler number, representing pressure field similarity, is the result of correct scaling. Therefore, it can only be set as an achievable, not as an input requirement. The bulk Reynolds number is negligible for the cooling air flow, due to the low compression effects. The Froude number is not relevant for low density gradient flow. The radial Rossby number is not considered, as it has little influence on the generator flow due to the radial main flow direction [13]. The Reynolds number is often treated as the most relevant similarity number for low-speed flow problems. One additional similarity number, that originated from the conservation of energy, is taken into account: The Mach number Ma. It becomes relevant as soon as compressibility effects cannot be neglected from $Ma \approx 0.3$. This is not the case for the majority of generator cooling flows. For a down scaled model with an increased flow velocity, Ma can become crucial.
In previous work the relevance of the centrifuge number Cf, the radial Rossby number Ro_r and the tangential Rossby number Ro_ϕ was investigated by the authors [13]. They identified Cf and Ro_ϕ to be crucial for generator model scaling. The scaling investigations in this paper also refer to these two similarity numbers, which is why they are presented in more detail here.
The definition of Cf, caused by the centrifugal acceleration is given in Eq. 4. Its characteristic dimensions are the rotor radius r_0 as characteristic length and v_0 as characteristic velocity.

$$Cf = \frac{\omega^2 r_0^2}{v_0^2} \tag{4}$$

The Centrifuge number represents the pressure rise generated by the rotor, normalized by the dynamic pressure of the flow. It is a specific way to express kinematic similarity, meaning that the flow field's directions and relative velocity are consistent across the entire flow region under centrifugal compression.
The tangential Rossby number Ro_ϕ describes the Coriolis acceleration due to the radial component of the flow velocity. Its defining equation is reminiscent of the atmospheric Rossby number, which is why this name was

adopted. It includes a characteristic length in tangential direction $l_{\phi,0}$ and the flow angle α relative to the radial direction:

$$Ro_{\phi} = \frac{2\omega\cos(\alpha)l_{\phi,0}}{v_0} \tag{5}$$

The relevant similarity numbers selected for the investigations in this paper are therefore the Reynolds number, the Mach number, the centrifuge number, and the tangential Rossby number.

2 Scaling Approaches

Taking all similarity numbers into account, only partial similarity can be achieved. Based on the selection of the four most relevant similarity numbers in section 1, different SAs are developed. Those SAs will be applied on the generator ventilation problem for comparison. Each SA follows defined strategies. The similarity numbers of the scaled model (index M) are either equal to the original (index O), or change with a model scale factor S. The main scale factor is S_r, which is the ratio of the model's rotor radius and the originals rotor radius. Following the strategies, every model parameter has a scale factor from the original. S_V is the scale factor of the characteristic flow velocity, S_l is the scale factor of the characteristic length of each individual section of the flow. S_c is the sonic speed scale factor and S_ν the factor of the fluids kinematic viscosity. All SAs have in common, that the Centrifuge number is matched by scaling the rotational speed ω. This decision affects the tangential Rossby number for certain approaches (SA3, SA5). In the following, five SAs are defined. For each approach there is a description of the applied strategies and the resulting similarity numbers of the model.

SA1: Full geometrical downscaling

The entire geometry is scaled down according to the model's main scale factor S_r. The utilized fluid is air, like in the original generator, but under laboratory conditions. This choice of fluid is the most common and involves the least additional effort. The ambient conditions lead to a slightly different density and kinematic viscosity of the fluid. For ensuring that the Reynolds number is equal to the originals, the velocity factor S_V depends on the characteristic length scale factor S_l and the scale factor of viscosity S_ν:

$$S_V = \frac{S_\nu}{S_l} = \frac{S_\nu}{S_r} \approx \frac{1}{S_r} \tag{6}$$

Among the similarity numbers that are chosen to be most relevant in this paper, the partial similarity only compromises the Mach number:

$$Re_M = Re_O \qquad Cf_M = Cf_O \qquad Ro_{\phi,M} = Ro_{\phi,O} \qquad Ma_M = Ma_O\frac{S_V}{S_c} \approx Ma_O S_V$$

SA2: Keep characteristic lengths

The core strategy of SA2 is to maintain the characteristic lengths equal the originals. For example, the axial and circumferential dimensions of a radial cooling duct in the rim or stator is kept constant, while the total number of cooling ducts in axial and circumferential direction is reduced. This SA also employs the same fluid under laboratory conditions, like SA1. The velocity change from original to model is just in the magnitude of a small percentage, caused by the different ambient conditions of the laboratory compared to the original hydro power plant. The similarity numbers of this approach are:

$$Re_M = Re_O \qquad Cf_M = Cf_O \qquad Ro_{\phi,M} = \frac{1}{S_r}Ro_{\phi,O} \qquad Ma_M = Ma_O\frac{S_V}{S_c} \approx Ma_O$$

SA3: Maintain velocity and Reynolds number regime

SA3 compromises the Reynolds number by setting the velocity equal to that of the original, $V_M = V_O$. This ensures that the Mach number closely approximates the original. The scale factor for the characteristic dimensions S_l in the model can be freely chosen, allowing flexibility in balancing the trade-off between Reynolds number and geometric similarity. Used values are $S_l = 0.5$ and $S_l = 0.25$. This SA also uses the same fluid under laboratory conditions. The similarity numbers of this approach are:

$$Re_M = Re_O\frac{S_l}{S_\nu} \approx Re_O S_l \qquad Cf_M = Cf_O \qquad Ro_{\phi,M} = \frac{S_l}{S_r}Ro_{\phi,O} \qquad Ma_M = Ma_O\frac{1}{S_c} \approx Ma_O$$

SA4: Full geometrical downscaling with water

Replacing air with another fluid as the cooling medium affects the Reynolds number due to changes in viscosity. Water is a common choice for downscaled models. To meet similarity requirements concerning compressibility, both the original and modeled flows must be free of compressibility effects. This means the Mach number for the original flow must not exceed 0.3, while the modeled flow with water must avoid cavitation. The geometry of the model is equal to the geometry of SA1. The similarity numbers of this approach are:

$$Re_M = Re_O \qquad Cf_M = Cf_O \qquad Ro_{\phi,M} = Ro_{\phi,O} \qquad Ma_O \leq 0.3$$

SA5: Keep characteristic length in water
In this SA, the geometry of SA2 is combined with water as fluid. The compressibility considerations are equivalent to SA 4. With this combination, the lowest possible fluid velocities can be achieved.

$$Re_M = Re_O \qquad Cf_M = Cf_O \qquad Ro_{\phi,M} = \frac{1}{S_r} Ro_{\phi,O} \qquad Ma_O \leq 0.3$$

3 CFD-based Scaling Approaches Performance Studies

The SAs developed in section 2 is applied to a generic reference generator and their performance is analyzed by CFD simulations. The rotor diameter of the reference machine is 14 meters. Such a large machine is chosen, because downscaling becomes more challenging with increased size, due to the larger scaling ratios. Since the purpose of the model is to validate the pressure rise and drop characteristics of the machine, heat transfer is not considered in the simulations.
As described above, two sections of the total flow are analyzed separately: the hub inflow and the homogeneous section. Those flow sections are described in more detail in sections 3.2 and 3.3. A series of rotor scale factor was applied, including 1:1, 1:5, 1:10, 1:15, and 1:20. The rotor radius scale factor S_r is generally determining the overall model size. Each part of the machine is scaled according to the strategies of the applied SA. The original machine was also simulated for comparison.
The simulations were conducted using the commercial CFD code STAR-CCM+. Most simulations were based on the steady-state Reynolds-Averaged Navier-Stokes (RANS) model, although transient simulations were used in specific cases, such as the selection of a rotor-stator interface type. The chosen physical models vary depending on the case. For higher Mach numbers, the isothermal segregated flow approach was applied, while many simulations with low Mach numbers used a constant density model. The k-omega SST model was consistently used as turbulence model.
A comprehensive mesh and simulation convergence analysis was conducted for each flow section, resulting in a total of 205 simulations. The result generating main studies comprised 93 simulations, each with 12 to 20 million cells. The number of iterations varied between 3,000 and 20,000 on average. The larger number of iterations are caused by simulation procedures such as the gradual increase of rotational speed or mass flow for simulation stability and the iterative adjustment of the outlet boundary to achieve correct inlet pressure. The simulations were executed on the high performance computing cluster at TU Dresden. A detailed insight into the CFD setup and convergence analysis is not possible within this paper due to space constraints. For the same reason, only a selection of key results is presented to support the main outcome of this paper: a recommendation for the scaling of the most relevant part of large hydro power ventilation flow.
The performance of each scaled flow section in predicting the pressure drop of the cooling flow is evaluated using the Euler number. The Euler number represents the dimensionless total pressure loss in the system:

$$\mathrm{Eu} = \frac{\Delta p_t}{\frac{\rho_{ref}}{2}\left(\omega_{rotor}\, r_{rotor}\right)^2} \tag{7}$$

In Eq. 7 Δp_t is the total pressure loss. The reference pressure is defined as a machine-specific dynamic pressure of the rotor compression. The Euler number of every model is normalized with the Euler number of the original generator simulation:

$$\mathrm{Eu}_N = \frac{\mathrm{Eu}_M}{\mathrm{Eu}_O} \tag{8}$$

3.1 Power Demand

By assuming a proportionality of the ventilation power P_M and the radial compression: $P_M \sim \dot{V} \cdot \Delta p$, the model windage power can be approximated for every scaling approach using the model parameter scale factors. The scale factor of the volume flow $S_{\dot{V}}$ is equal to $S_r S_{NoP} S_l$ and the scale factor for the pressure rise $S_{\Delta p}$ can be replaced by $S_\rho {S_V}^2$. The number of poles (NoP) of original and model must be an integer value, which leads to a rounding function in Eq. 9:

$$P_M = S_{\dot{V}} S_{\Delta p} P_O = S_r S_{NoP} S_l S_\rho {S_V}^2 P_O = S_r \frac{NoP_M}{NoP_O} S_l S_\rho {S_V}^2 P_O = S_r \frac{round\left(NoP_O \frac{S_r}{S_l}\right)}{NoP_O} S_l S_\rho {S_V}^2 P_O \tag{9}$$

The ventilation power demands are listed in Tab. 1. They indicate that SA1 and SA4 both exceed the power demand of the original for most model sizes. This fact makes them unfeasible for any model test and they are not included in the subsequent CFD studies.

Tab. 1: Ventilation power demand of original and scaled models in kW

S_r	original	SA1	SA2	SA3 $S_l = 0.25$	SA4	SA5
1:1	2252	1906	2469.0	1906.0	433.3	433.3
1:5		3811	617.3	476.4	866.6	108.3
1:10		19056	24.7	19.1	4333.0	4.3
1:15		28569	11.0	8.5	6496.0	1.9
1:20		38111	6.2	4.8	8666.0	1.1

3.2 Scaling Approaches Performance for Hub Inflow Section

The first simulated part of the cooling flow is the hub inflow (HI) section, which begins downstream of the winding overhang. This section has four inlets. One main inlet is located at the non-drive end (NDE) and the drive end (DE), respectively. The air gap between rotor and stator is sealed with a gap sealing on both sides. The leakage flow through the two gaps is taken into account by two additional inlets.

The NDE sided compartment is modeled within a rotating reference frame and with counter-rotating moving walls. The DE sided section is non-rotating, and a rotor-stator interface (RSI) is positioned upstream of the hub openings. The flow enters the rotor through the hub openings on both sides. The hub openings lead into the hub chambers between the shaft and the rim, which are divided by hub arms. Depending on the SA, either one or two hub openings and chambers are modeled, before they get repeated by a periodic boundary. The original machine consists of 24 hub chambers, each containing two columns of rim channels.

The rim structure has the shape of a hollow cylinder and holds the poles. The cooling fluid flows through the rim in small radial rim channels. These channels are designed in two different variants, which is typical for large rim ventilated hydro generators. The simulation outlets are located at the end of these channels. The rim channels are not included in the evaluation for this section. However, the flow in these channels is modeled with precision.

Figure 1 depicts the fluid volume for both the original machine and the 1:20 scaled model with the most abstract geometry. In this most simplified model, one hub chamber encompasses the full 360°, so there is no periodic symmetry in the model.

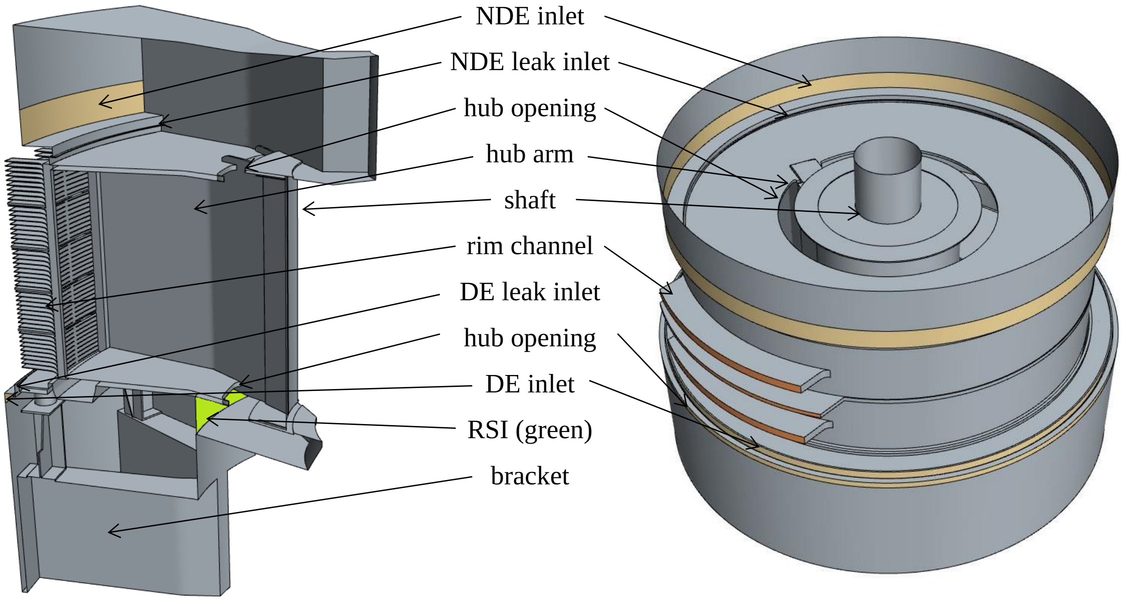

Fig. 1: Hub inflow geometry of original simulation (left) and SA2 or SA5 1:20 model (right).

The evaluation is based on the normalized Euler number. For the rotating simulation domain, the required total pressure loss Δp_t is derived from the rotational stagnation pressure p_{rot}, that is constant in an ideal radial compressor. This is analogous to the rothalpy, which is conserved in a steady, isentropic flow of a rotating machine, as described by Wu [14]:

$$p_{rot,1} = p_{rot,2} + \Delta p_t = p_{t,rel,1} - \varrho \frac{u_1^2}{2} = p_{t,rel,2} - \varrho \frac{u_2^2}{2} + \Delta p_t \tag{10}$$

The variable u is the tangential velocity of the rotation and ϱ the reference density. The relative total pressure $p_{t,rel}$ is obtained by a mass flow averaged total relative pressure report at the rim channel inlet (index 2). Index 1 is at the simulation inlet. In this context, the term relative means relative to the ambient pressure. In the result chart in Fig. 2, the normalized Euler number is obtained using Eq. 7 and Eq. 8.

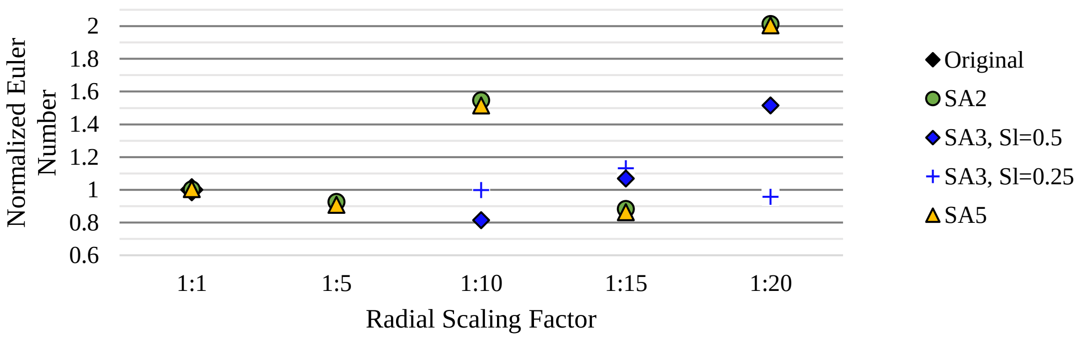

Fig. 2: Normalized Euler number performance of scaling approaches on HI flow section.

For the lowest scale factor of $S_r = 1:5$, SA2 and SA5 perform well. However, for higher scale factors, the results indicate that SA2 and SA5 fail to produce satisfactory results for the HI section due to excessive geometric abstractions. Those geometric abstractions result from the drastic reduction of number of ducts, that is required to keep the characteristic lengths of the model as close to the originals as possible. This geometric abstraction is clearly visible in Fig. 1.
Two implementations of SA3 are included in Fig, 2: one with a characteristic length scale factor of $S_l = 0.5$ and another with $S_l = 0.25$, corresponding to approximately half and a quarter of the Reynolds number, respectively. The results for SA3 with a characteristic length scale of $S_l = 0.25$ are satisfactory, as this approach balances the trade-off between the Reynolds number error due to reduced velocity and the geometric abstraction.

3.3 Scaling Approaches Performance for Homogenous Section

The second flow section is referred to as the "homogeneous section" (HS) due to the repeating pattern of axial stacking, which allows for axial periodic simplification in addition to the circumferential periodicity. The previous HI section volume flow distributions agree with this assumption: In the original simulation, the volume flow rate was nearly constant over the full axial length of the model. The velocity profiles of the flow entering the rim channels are taken from the preceding HI simulations. Downstream of the rim channels, the flow enters the pole gap and then moves through the air gap, which includes the RSI in the middle. The fluid then flows into the stator, which consists of two parts: narrow flow channels due to the stator bars, followed by wider flow channels. Due to the absence of rotation, the lengths of the stator channel parts can be adjusted to a correct pressure drop. A case study that is not included in this paper investigated different stator length options. The model on the right side of Fig. 3, has a long first part (radial length scale = 1) and a short second part (length scale = S_r). This configuration was chosen as the most applicable for a model, because the majority of the pressure drop is already generated in the narrow part, while the wider part would lead to unfeasibly large models. A long outlet section is included in the CFD model to smooth the flow exiting the stator. This improves the compatibility with the pressure outlet boundary condition. Periodicity is maintained in both the tangential and axial directions. Each model includes two pole gaps and two rim channels, as depicted in Fig. 3.

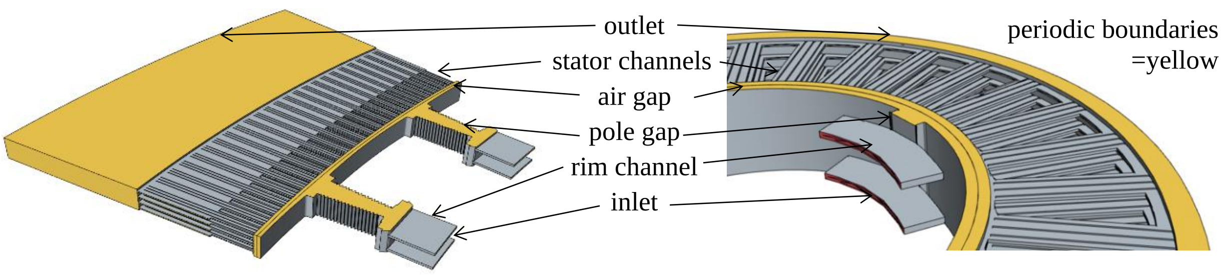

Fig. 3: Homogenous section geometry of original simulation (left) and SA2 or SA5 1:20 model (right).

The evaluation of the HS is once again based on Eqs. 7, 8 and 10. The analysis of the pressure rise in the rotor is separated from the pressure drop in the stator. A combined evaluation of both stator and rotor is not feasible, because the Euler number is calculated with the rotational stagnation pressure in the rotating system. In the stator, Eu is calculated based on the total pressure drop. Additionally, the applied scaling approach may differ for the stator and rotor. As illustrated in the charts in Fig. 4, the performance of SA2 and SA5 is again inadequate, in particular for the rotor. In contrast, SA3 with $S_l = 0.25$ continues to perform the best.

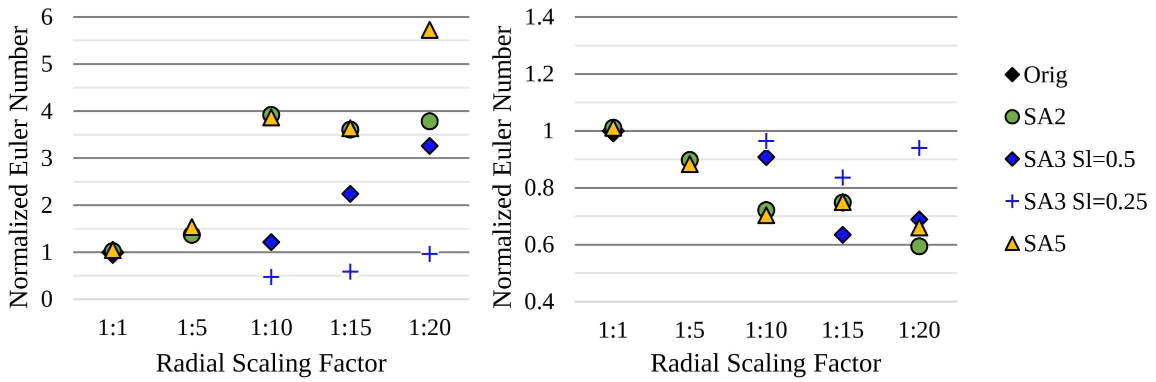

Fig. 4: Euler number performance of scaling approaches on rotor (left) and stator (right) homogenous section flow.

4 Scaling Recommendation

For the investigated large rim ventilated generator in this paper, a scaling recommendation for a ventilation model based on the pressure drop / rise performance and the power demand is given.
For main scale factors larger than 1:5, which is the evaluated range of $1{:}10 \leq S_r \leq 1{:}20$, SA3 with $S_l = 0.25$ provides the best Euler number results while maintaining an acceptable power demand. For larger models with a model size of $1{:}5$ or less, SA2 and SA5 are sufficient as well. However, those large models are not reasonably considerable for machines as large as investigated in this paper.
It is important to note that optimizing the stator channel design based on friction loss estimations generally improves results for all model sizes. This can be achieved by adjusting the channel length. Additionally, manufacturing tolerances are critical, especially for small ducts. If the manufacturing tolerances are significantly large relative to the characteristic parameters of the model, the model geometry should be measured, and the geometrical deviations should be considered. For example the reference pressure for the Euler number can be adjusted for velocity errors due to cross section deviations.

5 Conclusion

The present study is based on the fundamental approach of nondimensionalization applied on the cooling flow of large rim ventilated hydro generators. This leads to similarity numbers that aid the scaling of generator models to validate the cooling flow with pressure measurements. For the rotating sections of the flow, the relevant similarity numbers contained the centrifuge number and the tangential Rossby number. Both are not established in literature yet. Several scaling approaches were derived evaluated through numerical investigations using CFD simulations. The CFD simulations included two sub-sections of the generator flow. Those simulation sections cover the entire rotating domain of the generator, so the benefit of the new introduced similarity numbers is utilized. A series of different model sizes were evaluated, based on the Euler number and power demand. The evaluation of the scaled models indicate that the optimal scaling approach depends on the size of the scaled model. For scaling factors in the range of $1{:}10 \leq S_r \leq 1{:}20$, scaling approach 3 with a characteristic length scale factor $S_l = 0.25$ delivers the most accurate Euler number results while maintaining manageable power demands. These larger scale factors are particularly beneficial for the large reference machine of this study. Further studies should investigate if other generator sizes or cooling concepts can also be scaled with SA3 and if S_l should have the same value or if it should be chosen for each machine individually.

References

1 Chong, Y. C. - Thermal analysis and air flow modelling of electrical machines. PhD thesis, The University of Edinburgh, 2015

2 Connor, P. H. et al. - Stator and rotor vent modelling in a MVA rated synchronous machine, ICEM conference 2016

3 Klomberg, S. et al. - Comparison of CFD analyzing strategies for hydro generators, ICEM conference 2014

4 Boglietti, A. et al. - Evolution and modern approaches for thermal analysis of electrical machines. In IEEE Transactions on industrial electronics, Vol. 56, pp. 871–882, 2009

5 Moradnia, P. et al. - Experimental and numerical investigation of the cooling air flow in an electric generator”. In HEFAT2011, pp. 242–249
6 Toussaint, K., et al. - CFD Analysis of Ventilation Flow for a Scale Model Hydro-Generator. ASME Power Conference, Vol. 2, pp. 627–637, 2011
7 Dang, D.-D., et al., - CFD analysis of turbulent convective heat transfer in a hydro-generator rotor-stator system. Applied Thermal Engineering, Vol. 130, Elsevier BV, pp. 17–28, 2018
8 Jamshidi, H. et al. - Experimental and numerical investigation of hydro power generator ventilation. In IOP Conference Series: Earth and Environmental Science, Vol. 22, IOP Publishing, p. 012007, 2014
9 Bach, E. et al. - PIV Characterization of the Air Flow in a Scale Model of a Hydrogenerator. In ASME 2015 Power Conference 2015
10 Diebel, B. et al. - Validating a CFD simulation approach by ventilation measurements for an air-cooled salient pole model generator. In Advances in Modelling and Control of Wind and Hydrogenerators, IntechOpen, 2020
11 Truckenbrodt, E. - Fluidmechanik. BerlinHeidelberg: Springer, 1980 (German language)
12 Johnson, R. W. - The Handbook of Fluid Dynamics. Springer, 1998
13 Noelle, D. et al. - Similarity Numbers for the Scaling of Hydroelectric Generator Ventilation Flow [Paper presentation]. ICEM conference, 2024, Sept 1-4, Torino, Italy
14 Wu, C.-H., - Discussion of a paper by Smith, Traugott and Wislicenus. ASME Transactions, Vol. 75, 1953, pp. 802-803

The Authors

David Noelle has been a researcher in the field of experimental aerodynamics at the Chair of Flight Dynamics and Control at Technische Universität Dresden, Germany, since 2021. He received his Diploma in Engineering from Technische Universität Dresden in 2017. In 2018, he joined the Experimental Aerodynamics team at Technische Universität Dresden, where he has been involved in designing scaled aerodynamic ventilation models of hydrogenerators for industrial projects.

Bastian Diebel received the master’s degree (M.Sc.) in aerospace engineering at University of Stuttgart, Stuttgart, Germany. In 2016 he joined Voith Hydro Holding GmbH \& Co. KG in Heidenheim, Germany. In the department “Fluid Mechanics” he is responsible for research and development on cooling and ventilation of hydro generators. This includes the development and validation of 1D and 3D thermal design tools.

Dr. Thilo Dauch completed his master’s degree (M.Sc.) in mechanical engineering with a major in turbomachinery and jet engines at RWTH Aachen University, Aachen, Germany. Afterwards he worked as research associate and PhD candidate at the Institute of Thermal Turbomachinery (ITS) at the Karlsruhe Institute of Technology (KIT) in Karlsruhe, Germany. In 2020 he joined Voith as development engineer in the field of hydro generator ventilation. In 2024 he became “Head of Fluid Mechanics” and is now responsible for R\&D activities related to fluid mechanics in hydro generators and turbines.