Convection Heat Transfer and Flow Calculations Suitable for Analytical Modelling of Electric Machines

D.A. Staton
Motor Design Ltd
1 Eaton Court, Tetchill, Ellesmere
Shropshire, SY12 9DA, UK
dave.staton@motor-design.com

A. Cavagnino
Politecnico di Torino
Departimento di Ingegneria Elettrica
Torino, ITALY
andrea.cavagnino@polito.it

***Abstract* – Motor-CAD is a commercial software package dedicated to the optimisation of motor cooling. Its solver is based on thermal lumped circuit analysis. This provides near instantaneous calculations speeds allowing 'what-if' scenarios to be run in real time. The user inputs geometric data for the design they wish to simulate using the graphical radial and cross-section editors. Materials to be used in the machine and the cooling type to be modelled (TENV, TEFC, Liquid Cooling, etc) are selected. All thermal parameters such as conduction, radiation and convection thermal resistances are then calculated by the program and the thermal performance calculated. This article concentrates on the formulations used to predict the convection cooling and flow within the machine.**

I. INTRODUCTION

Convection is the transfer process due to fluid motion. In natural convection, the fluid motion is due entirely to buoyancy forces arising from density variations in the fluid. In a forced convection system movement of fluid is by an external force (fan, blower, pump). If the fluid velocity is large then turbulence is induced. In such cases the mixing of hot and cold air is more efficient and there is an increase in heat transfer. The turbulent flow will however result in a larger pressure drop such that with a given fan/pump the fluid volume flow rate will be reduced.

Proven empirical heat transfer correlations based on dimensionless analysis are used [1-6] to predict the heat transfer coefficient, h [W/m2/C], for all convection surfaces in the machine. Many such correlations are built into the software – the most appropriate formulation for a given surface and flow condition being chosen automatically. This means that the user need not be an expert in heat transfer analysis to use the software effectively.

Forced convection heat transfer from a given surface is a function of the local flow velocity. In order to predict the local velocity a flow network analysis is performed to calculate the flow of fluid (air or liquid) through the machine. Empirical dimensionless analysis formulations are used to predict pressure drops for flow restrictions such as vents, bends, contractions and expansions.

II. DIMENSIONLESS ANALYSIS - CONVECTION

The advantage of using formulations based on dimensionless analysis is that any gas/fluid can be used, even if the original formulation was based on one particular fluid. Altitude has a significant effect on convection cooling and is fully accounted for as the variation in air pressure, density and temperature with altitude are all modelled.

For natural convection the typical form of the convection correlation is:

$$Nu = a\,(Gr\,Pr)^{b}$$

For forced convection the typical form of the convection correlation is:

$$Nu = a\,(Re)^{b}\,(Pr)^{c}$$

where a, b and c are constants given in the correlation. Also:

$$Re = \rho\, v\, L / \mu$$

$$Gr = \beta\, g\, \Delta T\, \rho^{2}\, L^{3} / \mu^{2}$$

$$Pr = c_{p}\, \mu / k$$

$$Nu = h\, L / k$$

Nu -	Dimensionless Nusselt Number
Re -	Dimensionless Reynolds Number
Gr -	Dimensionless Grashof Number
Pr -	Dimensionless Prandtl Number
h -	heat transfer coefficient [W/m2/C]
μ -	fluid dynamic viscosity [kg/s.m]
ρ -	fluid density [kg/m3]
k -	fluid thermal conductivity [W/m/C]
cp -	fluid specific heat capacity [kJ/kg/C]
v -	fluid velocity [m/s]
ΔT -	delta temperature of surface-fluid [C]
L -	characteristic length of the surface [m]
β -	coefficient of cubical expansion, $1/(273+T_{FLUID})$ [1/C]
g -	gravitational force of attraction [m/s2]

The magnitude of Re is used to judge if there is laminar or turbulent flow in a forced convection system. Similarly the Gr.Pr product is used in natural convection systems. The parameter that we are ultimately after is h. Once we know h we can calculate a thermal resistance [R = 1/Ah, A = surface area] to put in the heat transfer network.

Natural convection heat transfer is a primary function of the temperature difference between component and fluid and the fluid properties. Forced convection is a primary function of the fluid velocity and fluid properties and only a secondary function of the temperature because fluid properties are temperature dependent. The mixed heat transfer due to the combination of natural and forced convection is estimated using the formulation [1]:

$$h_{MIXED}^{3} = h_{FORCED}^{3} \pm h_{NATURAL}^{3}$$

where the motor orientation determines the ± sign used, a + sign for assisting and transverse flow and a – sign for opposing flows.

III. NATURAL CONVECTION – TENV COOLING

A few of the housing types suitable for external natural convection (TENV) are shown in Fig. 1. The surface is usually smooth. If fins are used to increase the convection surface they should ideally have radial orientation so not to disturb the air flow. Even with the fin types shown in Fig. 4 that are more suitable for TEFC machines the natural convection heat transfer must be calculated as it can dominate the cooling at low fan speeds.

The following set of simple shape correlations are used to form predictions of the convection heat transfer from the more simple housing structures.

- horizontal and vertical cylinders
- horizontal and vertical flat plates
- horizontal and vertical u-shaped channels

Table I gives values for the a and b coefficients for laminar and turbulent flow for smooth housing surfaces [1-3]. Two set of coefficients are given, one for laminar flow and one for turbulent flow. The GrPr product at which the transition to turbulent flow occurs is also given.

Table I: Natural Convection Correlation Coefficients

Shape	Gr.Pr lam to turb	a lam.	b lam.	a turb.	B turb.
horizontal cylinder	10^9	0.525	0.25	0.129	0.33
vertical cylinder	10^9	0.59	0.25	0.129	0.33
vertical flat plate	10^9	0.59	0.25	0.129	0.33
horiz. plate [upper]	10^8	0.54	0.25	0.14	0.33
horiz. plate [lower]	10^5	0.25	0.25	NA	NA

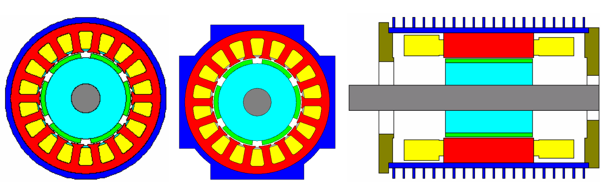

Fig 1: Examples of housing types suitable for TENV cooling

For more complex housing types, that have finned structures, correlations for u-shaped channels are also used. For a U-shaped vertical channel with laminar flow [6]:

$$Nu = (r/L.Gr.Pr)/Z \times (1\text{-}EXP[-Z(0.5/(r/L.Gr.Pr))^{0.75}])$$

$$Z = 24 \times [1 - 0.483\ EXP(-0.17/a)] / [1+a/2]^{3} \times [1+ (1\text{-}EXP(-0.83a)) \times (9.14 . \sqrt{a} . EXP(-465 . S) - 0.61)]^{3}$$

Where S = fin spacing, L = fin depth, a = channel aspect ratio (S/L) and r = characteristic length (hydraulic radius), 2.L.S/[2.(L+S)].

For a U-shaped horizontal channel with laminar flow [7]:

$$Nu = 0.00067 \times Gr.Pr \times [1 - EXP\ \{(-7640/Gr.Pr)^{0.44}\}^{1.7}$$

The fin spacing is used as characteristic length in this case.

The convection correlation chosen for a particular housing section depends on its geometry and orientation. Many correlation types are required to suit the varied housing geometries used in practice. In many cases the housing shape may be so complex that a single correlation does not exit. In such cases separate correlations are used for the parts of the surface that have a shape with a known correlation. An area based average is then carried out for the separate correlations. For example, a radial finned motor housing is a combination of a cylinder and vertical & horizontal fin channels. There is little air circulation at the base of deep narrow axial channels fitted to the sides of the motor – Fig 2 [11]. For such fin structures terms are introduced into the formulation to limit dissipation area to a depth down the fin channel equal to fin spacing. This is required so that the dissipation from axial finned housings is not over-predicted. In deed we get good results for such housings. Fig 3 shows that a good prediction of the natural convection can be achieved using such complex correlations. Here we see both calculated and measured thermal resistance values between housing and ambient for the motors shown in the diagram, the fan being at rest in this case [10].

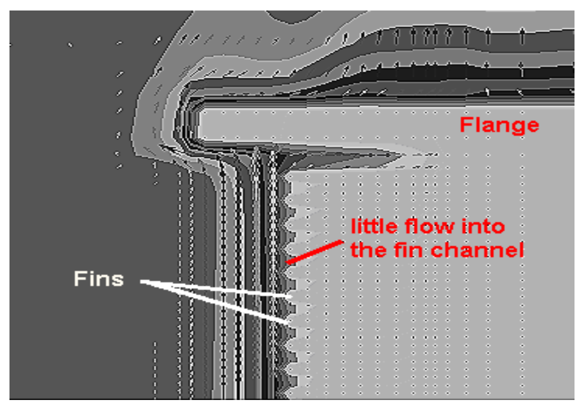

Fig 2: CFD Analysis – radial fins with vertical flange mount

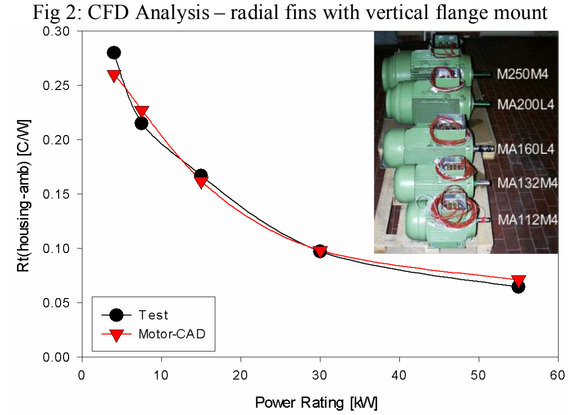

Fig. 3: Natural convection resistance - housing to ambient

IV. FORCED CONVECTION – TEFC COOLING

A few of the housing types optimised for external forced convection (TEFC) are shown in Fig. 4. A fan is usually fitted to the end of the shaft that blows air in an axial direction over the outside of the housing. If the surface is smooth then we can use the well known correlation for flow over a flat plate [1-3]:

Laminar Flow (($Re < 5 \times 10^5$) and ($0.6 < Pr < 50$):

$$Nu = 0.664\ (Re)^{0.5}\ (Pr)^{0.33}$$

Turbulent Flow (($Re > 5 \times 10^5$):

$$Nu = [0.037\ (Re)^{0.8} - 871]\ (Pr)^{0.33}$$

In the TEFC machine type, axial fins are usually included on the housing surface to increase the convection heat transfer. Also, in the majority of TEFC machines the fin channels are semi-open and the common external and internal flow correlations are not directly applicable. A special formulation for semi-open channels can be used. This is based on the extensive testing carried out by Heiles [5] on finned induction motor housings of various size and shape. In the correlation it is assumed that the flow is always turbulent due to the fact that the radial fans and cowlings used in such machines create turbulence. The convection heat transfer coefficient h is calculated using the formulation:

$$h = \rho\ cp\ D\ v\ /\ (4.L) \times [1-\exp(-m)]$$

$$m = 0.1448\ L^{0.946}\ /\ D^{1.16}\ \{k\ /\ (\rho\ cp\ v])\}^{0.214})$$

where, D is the Hydraulic Diameter (4 x channel area / channel perimeter, including open side) and L is the axial length of cooling fins. Heiles recommends the use of a Turbulence Factor to directly multiply h by. His tests indicate typical turbulence factors in the range 1.7 - 1.9 which seem independent of the flow velocity.

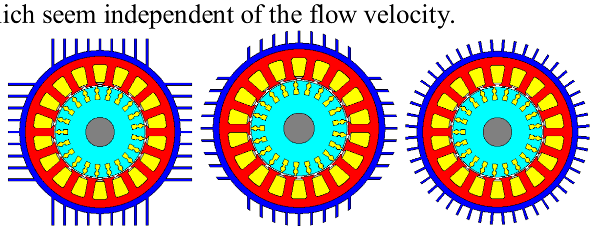

Fig 4: Examples of housing types suitable for TEFC cooling

The inlet velocity to the fin channels must be estimated. We can use empirical data such as that shown in Fig 5. This shows that average velocity of the air in the fin channels as it leaves the fan. The variation in velocity with shaft speed is as expected, a linear relationship. The actual variation in velocity from channel to channel can vary significantly and is a function of the fan direction [10]. Alternatively we may know the volume flow rate. As we know the channel dimensions and the inside diameter of the cowling we can calculate the velocity from the cross-sectional area available for flow.

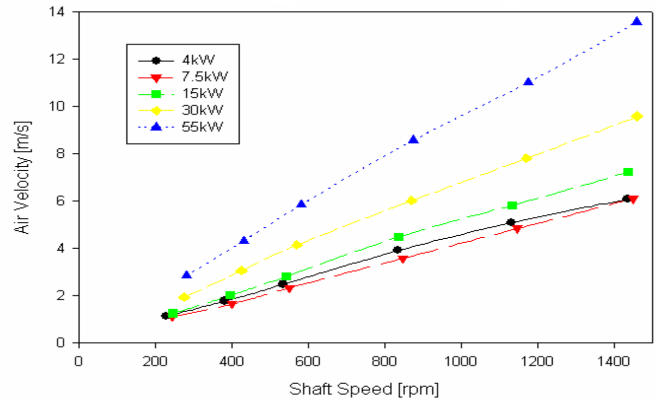

Fig. 5: Variation in fin channel air velocity with RPM (motors in Fig 3)

Typically in a TEFC machine some of the fin channels on the outside of the machine are blocked by bolt lugs and terminal boxes. Another deficiency of TEFC machines is that the air leaks out of the open channels causing the local air velocity to be lower at the drive end than at the non-drive end. The typical form of the reduction in velocity is shown in Fig 6. The prediction of the actual reduction in velocity is a complex function of many factors including the fan, fin and cowling design and rotational speed. A more accurate model is formed if some calibration is performed using testing and/or CFD [10].

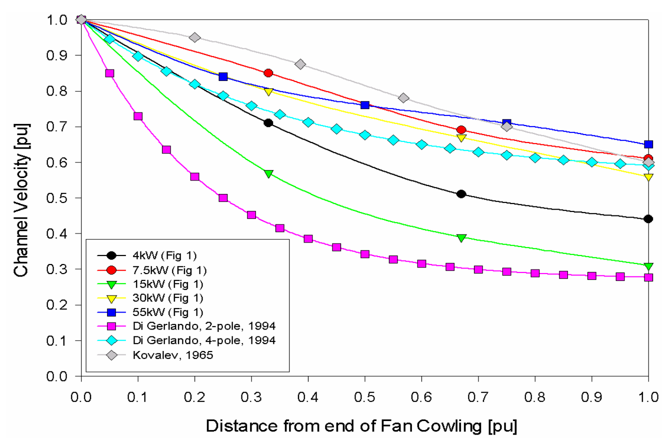

Fig. 6: Fin channel air velocity reduction with distance from fan

V. FORCED CONVECTION – WATER JACKETS

Liquid cooling methods such as spiral grooves and zig-zag arrangements of axial covered channels are often used in highly loaded machines. Fig 2 shows examples of typical liquid cooling ducting systems. Correlations suitable for internal flow are used to calculate the heat transfer coefficient in such cases. The heating effect of the fluid is also taken into account in the formulation.

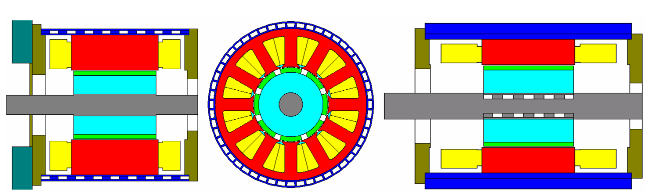

Fig 7: Examples of liquid cooling types

A. Laminar Flow:

For Round Channels [3]:

$$Nu = 3.66 + [0.065\ (D/L)\ Re\ Pr] \ / \ [1 + 0.04\ ((D/L)\ Re\ Pr\ )^{2/3}]$$

For Rectangular Channels [3]:

$$Nu = 7.49 - 17.02\ H/W + 22.43\ (H/W)^2 - 9.94\ (H/W)^3 + [0.065\ (D/L)\ Re\ Pr] \ / \ [1 + 0.04\ ((D/L)\ Re\ Pr\ )^{2/3}]$$

For Concentric Cylinders [3]:

$$Nu = 7.54 + [0.03\ (D/L)\ Re\ Pr] \ / \ [1 + 0.016\ ((D/L)\ Re\ Pr\ )^{2/3}]$$

H/W is the channel height/width ratio. D is the channel hydraulic diameter, i.e. 2 x Gap for concentric cylinders and 4 x channel cross sectional area divided by channel perimeter in round and rectangular channels. The variable part of the above equation is the entrance length correction [8] which accounts for entrance lengths where the velocity and temperature profiles are not fully developed.

B. Turbulent Flow:

For fully developed turbulent flow, i.e. $3000 < Re < 1e6$ [9]:

$$Nu := (f/8)\ (Re\text{-}1000)\ Pr \ / \ [1 + 12.7\ (f/8)^{**}0.5\ (Pr^{2/3} - 1)]$$

where f = Friction Factor and for a smooth wall is:

$$f = [0.790 \times Ln(Re) - 1.64]^{-2}$$

The flow is assumed to be fully laminar when $Re < 2300$ in round & rectangular channels and when $Re < 2800$ in concentric cylinders. The flow is assumed to be fully turbulent when $Re > 3000$ (in practice the flow may not be fully turbulent until $Re > 10000$), A transition between laminar and turbulent flow is assumed for Re values between those given above. Typical results showing transition from laminar to turbulent flow for the enclosed channel correlation is shown in Fig 8. It is seen that the two formulations do not in fact join each other and a small transition zone (taken from critical Re number dependent upon channel shape to 3000) is used to make the two functions join and so give numerical stability. A weighted average (based on Re) is then used to calculate Nu in the transition zone.

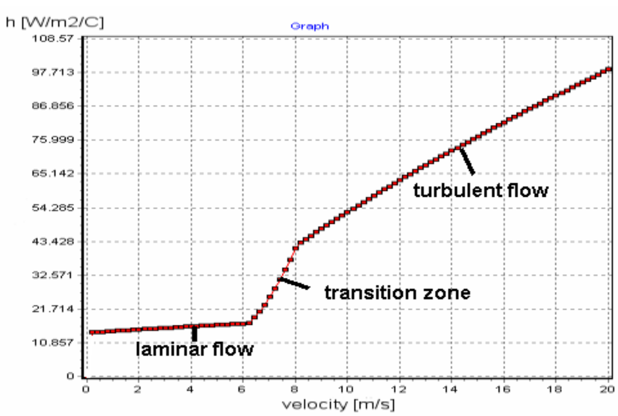

Fig 8: Enclosed channel forced convection

VI. FORCED CONVECTION – END SPACES

Convection for all surfaces within the internal sections of the machine must be modelled, e.g. end-windings. The convection cooling of internal surfaces can be complex as the fluid flow depends on many factors including the shape & length of the end winding, added fanning effects due to wafters (simple fan features included on an induction motors end-rings), simple internal fans & the surface finish of the end sections of the rotor and turbulence. Luckily, several authors have studied such cooling and in general propose the use of a formulation of the form [10]:

$$h = k1 \times [1 + k2 \times (velocity)^{k3}]$$

Where *k1, k2, k3* are curve fit coefficients. Fig 3 shows data taken from existing correlations of the end space cooling. It is reassuring that all the references show much the same trends as shown in Fig 9.

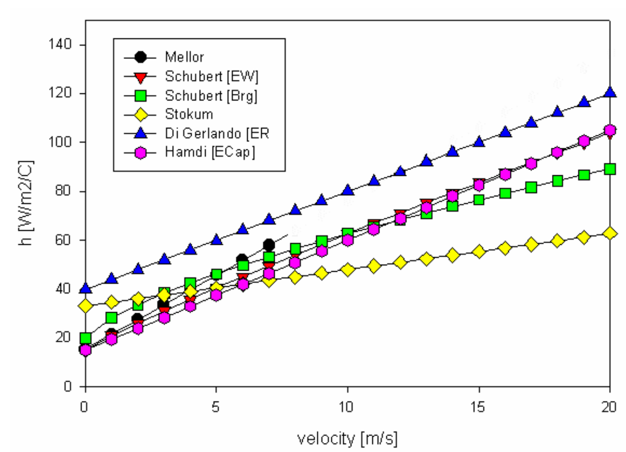

Fig 9: End-space convection correlations

VII. AIRGAP HEAT TRANSFER

The traditional method for accounting for heat transfer across airgaps in electrical machines is to use the dimensionless convection correlation developed from testing on concentric rotating cylinders firstly by Taylor [12] in 1935 and then added to by Gazley [13] in 1958. In the analysis use is made of the Taylor (Ta) number to judge if the flow is laminar, vortex or turbulent:

$$Ta = Re\ (l_g/R_r)^{0.5}$$

Where l_g is the airgap radial length, R_r the rotor radius and $Re = l_g\ v\ /\ \mu$. The flow is laminar if $Ta < 41$. In this case $Nu = 2$ and heat transfer is by conduction only. If $41 < Ta < 100$ the flow takes on a vortex form with enhanced heat transfer:

$$Nu = 0.202\ Ta^{0.63}\ Pr^{0.27}$$

If $Ta > 100$ the flow becomes fully turbulent flow and a further increase in heat transfer results:

$$Nu = 0.386\ Ta^{0.5}\ Pr^{0.27}$$

VIII. THROUGH VENTILATION

In the through ventilation model the air flow through the machine is calculated using flow network analysis [14-17]. Typically there are three parallel flow paths through the machine - stator & rotor ducting and the airgap. Examples of typical ducting types are shown in Fig 10. The total flow through the machine is determined from the intersection between the fan characteristic and system flow resistance characteristic – Fig 11. The flow velocity in each section of the flow circuit is calculated from the local flow rate and cross sectional area. The velocity information is then used to calculate the local heat transfer coefficient and subsequently the thermal resistance.

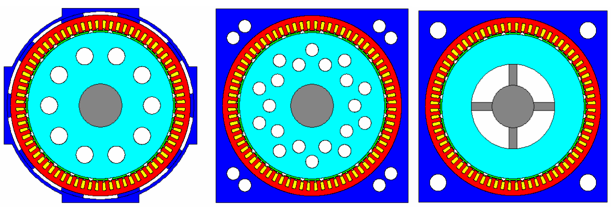

Fig 10: Examples of typical through ventilation duct types

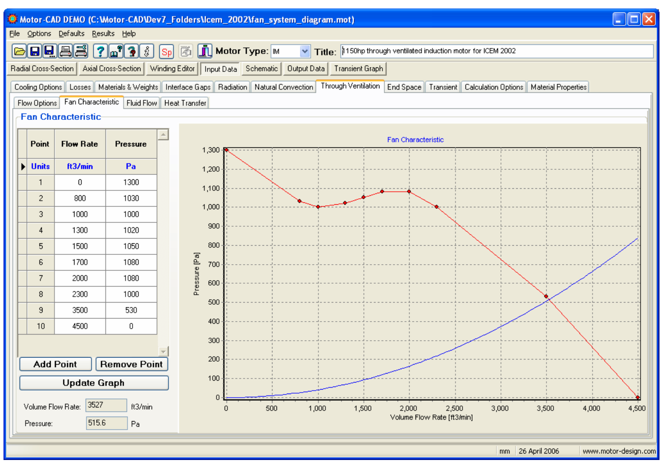

Fig.11: Fan and system resistance characteristics

IX. FLOW NETWORK ANALYSIS

The governing equation that relates pressure drop (P [Pa], flow equivalent of voltage in an electrical system) to volume flow rate (Q [m^3/s], equivalent to electrical current) and resistance (R [kg/m^7]) is:

$$P = R\,Q^2$$

The formulation is in terms of Q^2 rather than Q due to the turbulent nature of the flow. Two types of flow resistance exist. Firstly where there is a change in flow condition – such as expansions and contractions and bends. Secondly due to fluid friction at the duct wall surface - this is usually negligible compared to the first type of resistance due to the comparatively short flow paths. The flow resistance is calculated for all changes in the flow path using the formula:

$$R = k\,\rho / (2\,A^2)$$

Where k is the dimensionless coefficient of local fluid resistance whose value depends upon the local flow condition (obstruction, expansion, contraction, etc). Formulations are used to calculate the k factors for all changes in flow section within the motor - the most appropriate formulation being assigned automatically to the particular flow path component, i.e. a sudden contraction when air enters the stator/rotor ducts, a 90 degree bend where the air passes around the end winding, etc. ρ is the air density (kg/m^3) and A is the area of flow section that relates to the k factor formulation.

Five types of flow resistance are used to model the flow through the machine:

- Inlet Grill/Guard
- Outlet Grill/Guard
- Sharp Bend
- Sudden Expansion
- Sudden Contraction

(A) Inlet and Outlet Grill/Guards

The characteristic shown in Fig 12 is used to calculate pressure drop at entry to system due to a Grill/Filter over the inlet vents. The similar characteristic shown in Fig 13 is used for outlet vents. Both use a combination of data from Woods [15] and Lightband & Bicknell [16]. The arrow in the diagrams relate to which area is used in the flow resistance calculation.

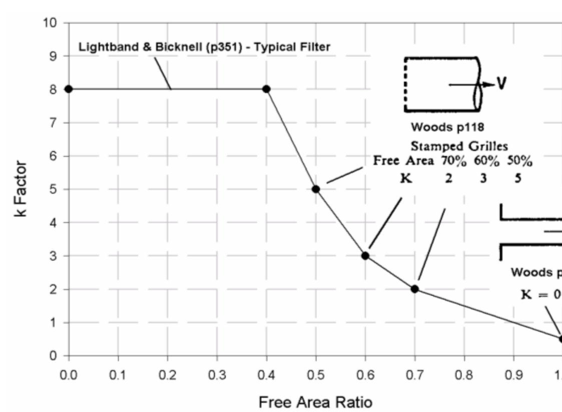

Fig 12: Inlet Grill k factor

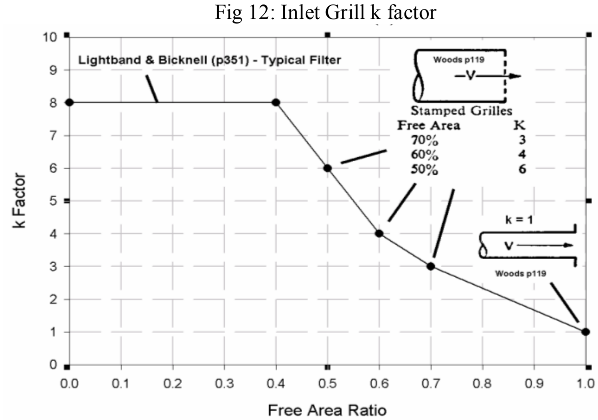

Fig 13: Outlet Grill k factor

(B) Sharp Bend:

The worst case of a right angle bend is assumed in the flow calculation (k = 1). We use the average area at each end of the bend. Fig 14 shows k data for other types of bend [15].

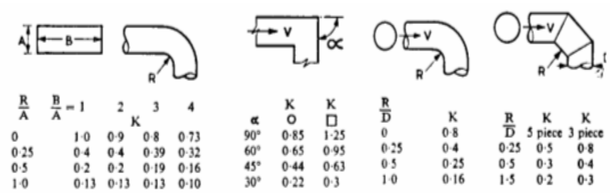

Fig 14 k data for bends [15]

(C) Sudden Expansions and Contractions:

The minor loss factor for a sudden expansion can be calculated using the formulation [15,16]::

$$k = (1 - \text{Area1}/\text{Area2})^2$$

A plot of the formulation is shown in Fig 15. The k-factor for a sudden contraction is shown in Fig 16 [15,16]. Note that the arrow in the diagrams show which area is used in the flow resistance calculation. Modified k-factors are also available for graded expansions and contractions [14-16].

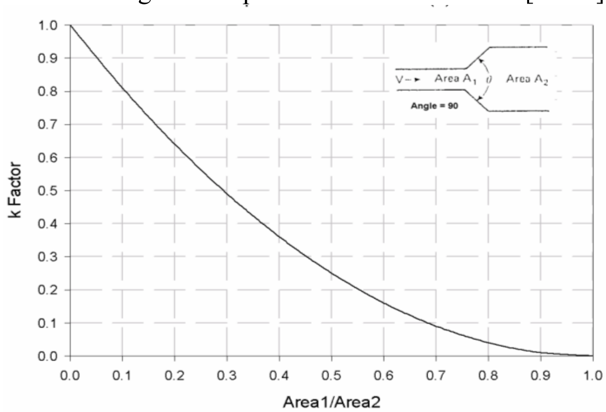

Fig 15 Sudden Expansion k-factor

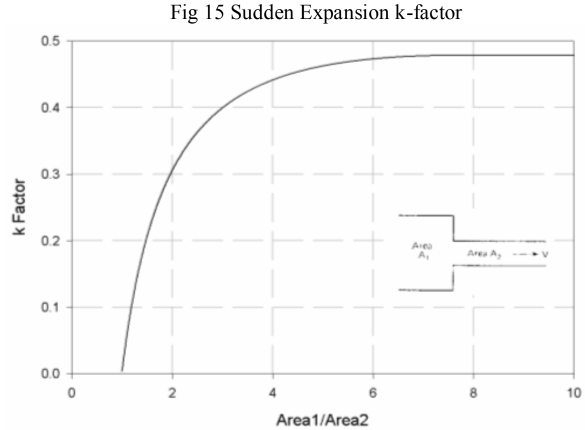

Fig 16: Sudden Contraction k-factor

The k-factor for the rotor duct entry contraction can be adjusted for rotation effects using the equation below [17]:

$$k(rot) = k(static) \times [V(rot)^2 + V(air)^2] / V(air)^2$$

k(rot) = minor loss factor with rotation
k(static) = minor loss factor with no rotation
V(rot) = average peripheral velocity of rotor ducts
V(air) = axial velocity of air through the ducts

The formulation is to account for the increase in pressure drop with rotational speed in rotating ducts. This adjustment is applied to all ducts on the rotor. It is more debatable if such an adjustment should also be applied to the airgap and so a choice can be made.

X. REFERENCES

[1] F.P. Incropera, & D.P. DeWitt, D.P.: Introduction to Heat Transfer, Wiley, 1990.
[2] J.P. Holman, Heat Transfer, McGraw-Hill, 1997.
[3] A.F. Mills, Heat Transfer, Prentice Hall, 1999.
[4] W.S. Janna, Engineering Heat Transfer, Van Nostrand Reinhold (International), 1988.
[5] Heiles, F.: Design and Arrangement of Cooling Fins, Elecktrotecknik und Maschinenbay, Vol. 69, No. 14, July 1952.
[6] Van De Pol, D.W. & Tierney, J.K. : Free Convection Nusselt Number for Vertical U-Shaped Channels, Trans. ASME, Nov. 1973.
[7] Jones, C.D., Smith, L.F. : Optimum Arrangement of Rectangular Fins on Horizontal Surfaces for Free-Convection Heat Transfer, Trans. ASME, Feb 1970.
[8] Edwards, D.K., Denny, V.E., Mills, A.F. : Transfer Processes, 2nd ed., Hemisphere, Washington, D.C. (1979)
[9] Gnielinski, V.: New equations for heat and mass transfer in turbulent pipe and channel flow, Int. Chemical Engineering, 16, pp.359-368 (1976)
[10] Staton, D.A., Boglietti, A., Cavagnino, A.: 'Solving the More Difficult Aspects of Electric Motor Thermal Analysis', IEMDC 2003, Madison, USA, July 2003
[11] Staton, D.A., So, E. : 'Determination of the Optimal Thermal Parameters for Brushless Permanent Magnet Motor Design', IEEE-IAS 33rd Annual Meeting, St.Louis, Missouri, October 1998.
[12] Taylor, G.I.: 'Distribution of Velocity and Temperature between Concentric Cylinders', Proc Roy Soc, 1935, 159, PtA, pp 546-578
[13] Gazley, C.: 'Heat Transfer Characteristics of rotating and axial flow between concentric cylinders', Trans ASME, Jan 1958, pp.79-89.
[14] Idlechik, I.E.: Handbook of Hydraulic Resistance - Coefficients of Local Resistance and of Friction, 1960
[15] Woods Practical Guide to Fan Engineering, Woods of Colchetser Ltd, 2nd Ed, June 1960.
[16] D. A. Lightband, D. A. Bicknell, The Direct Current Traction Motor: Its Design and Characteristics, 1970
[17] J.L. Taylor - Calculating air flow through electrical machines, Electrical Times, 21 July 1960