SWINTON, J., ESHAAFI, T., ASIM, T., IRONS, J. and MISHRA, R. 2017. Effect of manufacturing method of a centrifugal fan hub on its heat dissipation characteristics. In Sherrington, I., Onsy, A., Brooks, H., George, J. and Rao, R. (eds.) *Proceedings of the 30th International conference on condition monitoring and diagnostic engineering management (COMADEM 2017), 10-13 July 2017, Preston and Grange-Over-Sands, UK*. Preston: University of Central Lancashire [online], pages 355-360. Available from: http://clok.uclan.ac.uk/24444/

Effect of manufacturing method of a centrifugal fan hub on its heat dissipation characteristics.

SWINTON, J., ESHAAFI, T., ASIM, T., IRONS, J. and MISHRA, R.

2017

Effect of Manufacturing Method of a Centrifugal Fan Hub on its Heat Dissipation Characteristics

James Swinton[1], Taher Eshaafi[2], Taimoor Asim[*3], John Irons[4], Rakesh Mishra[5]
[1,4]Halifax Fans Ltd, Brookfoot Business Park, Elland Road, Brighouse, HD6 2SD, UK
[2,3,5]School of Computing & Engineering, University of Huddersfield, Huddersfield, HD1 3DH, UK
JamesSwinton@halifax-fan.com[1], Taher.Eshaafi@hud.ac.uk[2], t.asim@hud.ac.uk[3], JohnIrons@halifax-fan.com[4], r.mishra@hud.ac.uk[5]

ABSTRACT

As the process temperature of a fan system increases, the amount of heat that gets transmitted to the bearings and/or motor increases. If this is not accounted for, it can lead to catastrophic failure. The main heat conduction path is through the shaft, and certain mechanisms must be considered when looking for new solutions. These include; how heat is transmitted through the shaft or increasing the thermal resistance of the shaft, and dissipating heat as it is conducted through the shaft. These aspects must always be considered in addition to the impact of the manufacturing complexity. In the present study, an existing heat dissipation arrangement is reviewed and replaced by a new design which reduces the time taken to machine the part, and ultimately the overall cost of the product. Computation Fluid Dynamics (CFD) based techniques have been used to numerically simulate the designs under operating conditions, and the resulting heat transfer through the shaft compared with respect to the heat dissipation properties. The results demonstrate that although the new design is less effective at dissipating heat, it provides a substantial cost reduction compared to the existing design, while substantially reducing the impact of the design on various aspects of production.

Keywords: CFD, Heat Transfer, Centrifugal Fan

1. Introduction

The application of fans in high temperature environments requires the control of heat transfer through its components, in particular through the shaft as this connects the impeller to the prime mover, which moves the high temperature gas through the system, to the sensitive areas such as bearings that lie within a motor. Heat transfer can create problems for equipment operating at high temperatures, if the phenomenon is not understood and equipment is designed incorrectly the outcomes can be less than desirable. Currently a special coupling is utilised for high temperature applications, in order to dissipate heat. The coupling connects the impeller to the motor shaft, the impeller would be subjected to the high temperatures whilst the motor would remain below 70°C. According to Fourier's law heat transfer will take place through the shaft, to what extent depends on material and geometric properties. Should the temperature be too great at the point along the shaft where the motor bearings sit, permeant damage will occur and result in bearing failure. The current design is quite labour intensive to produce, because of this a new design has been created that can be easily machined, thus reducing the overall time spent manufacturing, which in doing so reduces the overall cost of the product. CFD analysis has been carried out at 300°C, the temperature at which the hubs would be exposed to, to analyse if the new design has the same heat dissipation properties as the baseline model.

Researchers have analysed how changes to the shaft speed, fin arrangement and fin thickness affects the transfer of heat through a system and away from a shaft. Zainullin.et al. (1) carried out an experimental investigations of a furnace fan to determine how changing fan speed and enclosing the shaft from its surroundings affects the heat transfer from the shaft. The results indicate that the heat transfer coefficient were 40-60% higher for a shaft that was open and rotating compared to an enclosed stationary shaft. Aziz.et al. (2) looked at solutions for rotating radial fins and how heat is lost to their surroundings, the study analysed theoretically using homotopy analysis to determine how fin thickness affected the results. The results gained when compared directly to numerical solutions to show the accuracy of the theoretical work. Watel.et al. (3) studied the effect of fin cooling experimentally using infrared thermography. The study explores how the rotational speed and fin spacing affect the heat exchanged in a finned tube. Using the results the mean value for the convective heat transfer from the rotating finned tube and the heat transfer coefficient from the cooling process could be calculated. Xie.et al. (4) experimentally studied a rotating and a stationary heat pipe in a condenser to investigate how the temperature in the pipe changes due to the rotation. By measuring the temperature at five points along the shaft using infrared thermocouples, it was possible to determine that when using the rotating shaft the temperature difference between the condenser and the shaft was reduced to almost zero, opposed to the stationary shaft whose temperature was much higher. Mori.et al. (5) present a CFD analysis of heat transfer on rotating blades. The boundary conditions represented real world characteristics that the geometry would be experience in practice. The results were verified experimentally using infrared thermography to study the temperature distribution. The method developed from the work can be used in other rotating machinery examples where accessing the object to gain readings is problematic.

From the review of the published literature there is good knowledge within the field, using apparatus to dissipate heat away from rotating shafts. The majority of examples found use fins in various arrangements and have calculations that are able to support their findings. Where there are potential areas lacking in knowledge would be the direct application of heat dissipation products for use in fan environments. In the present study, numerical simulations have been carried out investigating the differences in the heat transfer of a rotating component, that arise from modifying is geometry.

2. Numerical Modelling

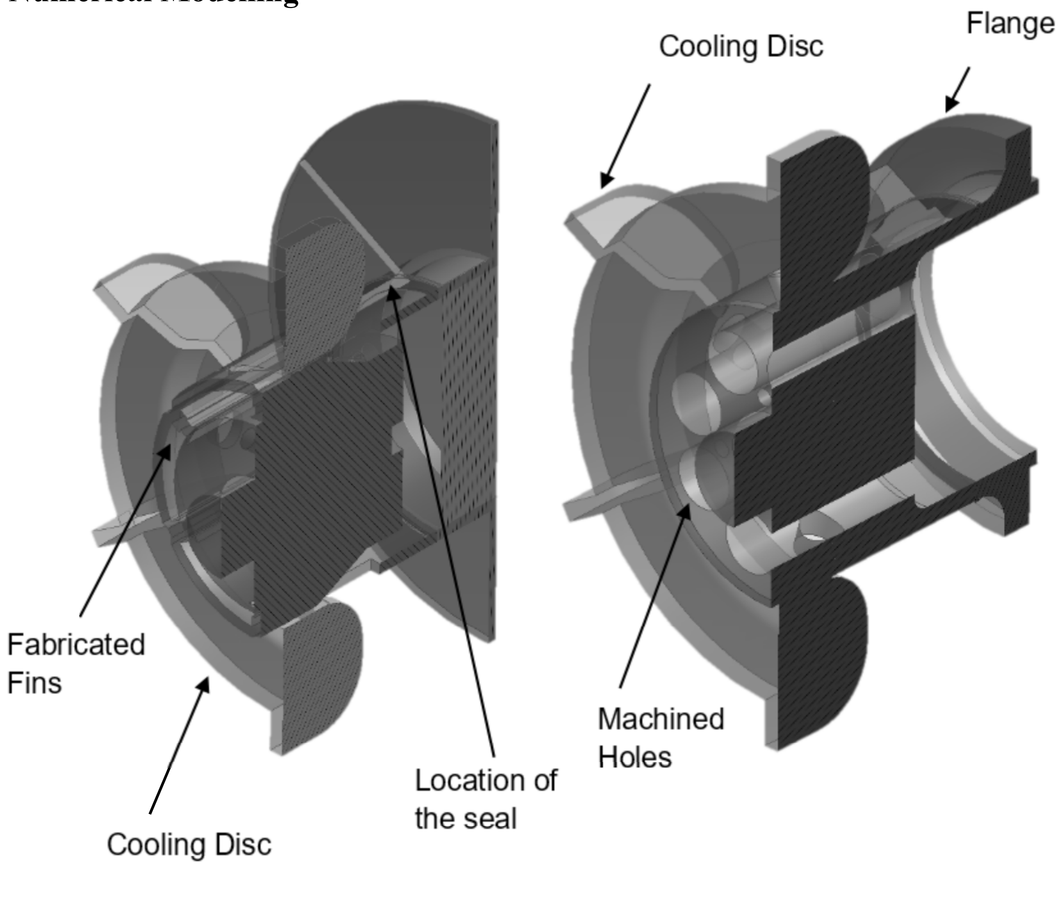

Figure 1. Cross sectional views showing the two geometries

For the numerical solutions, two models using the old and new hub designs made up the , these formed part of a system that included; the shaft with the hub attached, a small gap to simulate the seal between the fan casing and the shaft and an area representing the bearing unit terminating the shaft. The shaft has a diameter of 28mm and the point at which it reaches the bearing unit is 15mm from the hub body. When modelling the hubs the elements kept the same for both models were the overall length, diameter, materials (the hub body is carbon steel and the cooling disc aluminium) and the cooling disc position. The cooling disc is bolted onto the hub and provides a key role in dissipating heat away from the shaft; this is due mainly to its finned design. To simulate the cooling disc and shaft rotation a moving reference frame (MRF) approach has been applied.

Figure 1 show both hub designs, the old hub with the internal fin sections within the main body show the complexity of the design as these have to be fabricated individually to the body. It also shows the new hub design with its simplified construction as a single machined part.

2.2 Meshing of the Flow Domain

For the simulation a hybrid mesh has been used, utilised both hexagonal and polyhedral elements. The domain consisted of 5.5 million elements in total, with 1.1 million for the solids zone and 2.2 million for the fluid zone. It has been noticed that the mesh considered in the present study can be effectively used to capture the complex flow phenomena, and the associated heat transfer, with reasonable accuracy.

2.3 Boundary Conditions

The boundary conditions for the simulation are summarised in table 1.

Table 1. Boundary Conditions

Item	Boundary Type	Value
Flange	Heat	300°C
Seal	Pressure (Total)	2000Pa
Cooling Disc	MRF Zone	1500rpm
Ambient of Domain	Heat	26.85°C

3. Results and Analysis

The heat dissipation coupling models, both the old and the new design have been analysed at a temperature of 300°C. Figures 2 and 3 show the results from the heat transfer analysis and the differences between the models are highlighted and discussed. Figure 2 shows a cross section of the old hub model, the 300°C section can clearly be seen on the flanged section of the hub. The temperature distribution displays how the geometry of the hub affects the temperature through the different parts of the model.

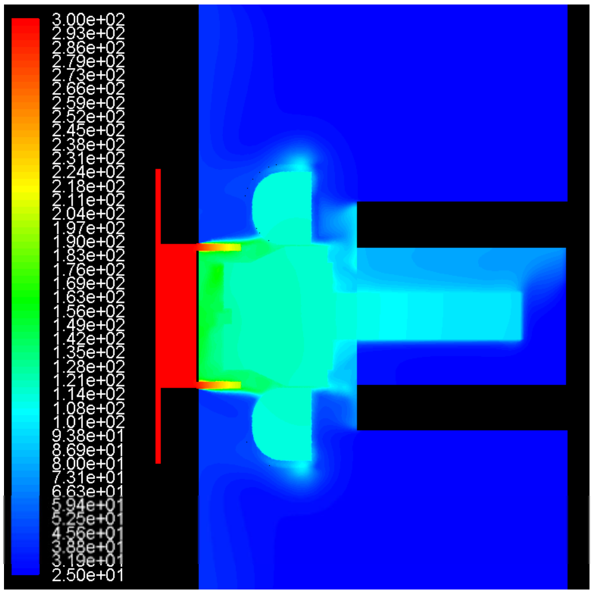

Figure 2. Temperature distribution through cross section of old hub design

Figure 3 shows a section view of the new machined hub, the temperature of 300°C has again been set to the flange section, on this model the flange is smaller. By drawing a direct comparison between the two models it is clear to see that after the seal section of the new hub model the overall temperature is increased.

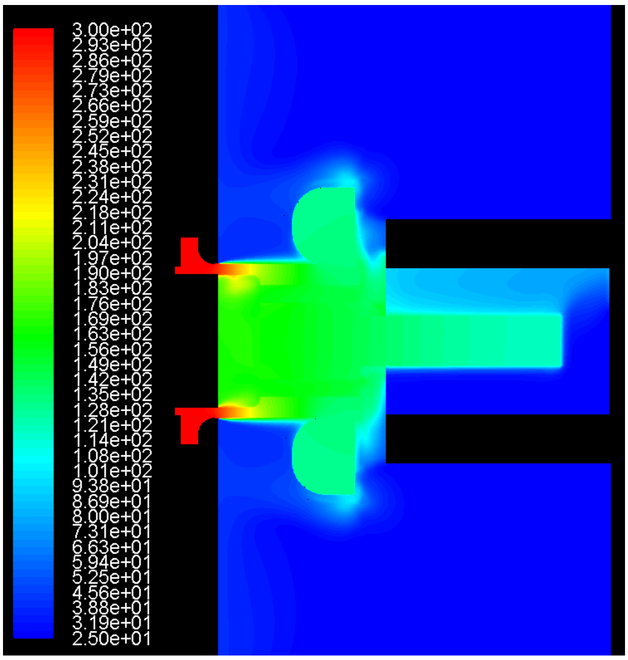

Figure 2. Temperature distribution through cross section of new hub design

In both figures 2 and 3 the boundary condition of 300°C, applied to the section of the models that are in contact with the impeller and located inside the fan casing, is shown in red on the distributions. After the seal point the hub is located outside of the fan case and the temperature distribution through the model reflects this. Immediately after the seal section the colours show a difference between the two models, this is due to the wall thickness of the two geometries, the old hub is 2mm thinner. If a line is taken through both models as shown in figure 4 we can see the temperature distribution through the models.

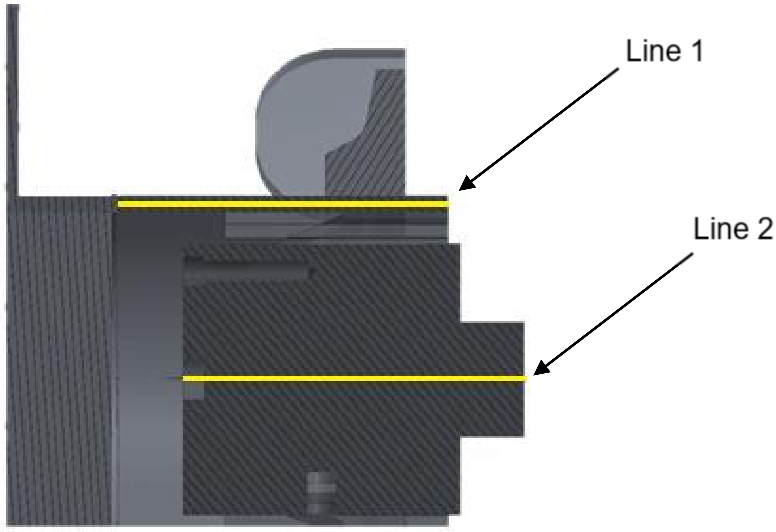

Figure 4. Detail of the two lines used to investigate the temperature distribution through the models

The temperature along the line located within the wall of the hub, (line 1) begins after the seal section in the models, the graph in figure 5 shows the change in temperature through to the front section of the model. This backs up how the difference in wall thickness is causing the overall temperature to be greater in the new design. The graph details how initially after the seal section, the temperature decreases in a linear trend until a point at which the temperature in the new design does not decrease as quickly as the baseline model. The reason for this lies again in the geometry, if a comparison is made between the models in figure 1 it can be seen that there is a noticeable increase in the material area at the centre of the hub.

Once the baseline model reaches around 120°C and the new model 140°C, there is a definitive levelling out of the temperature on the graph. This point within both models is where the cooling disc ends which suggests that for the cooling disc to be at its most effective it should be located as close to the end of the hub as possible.

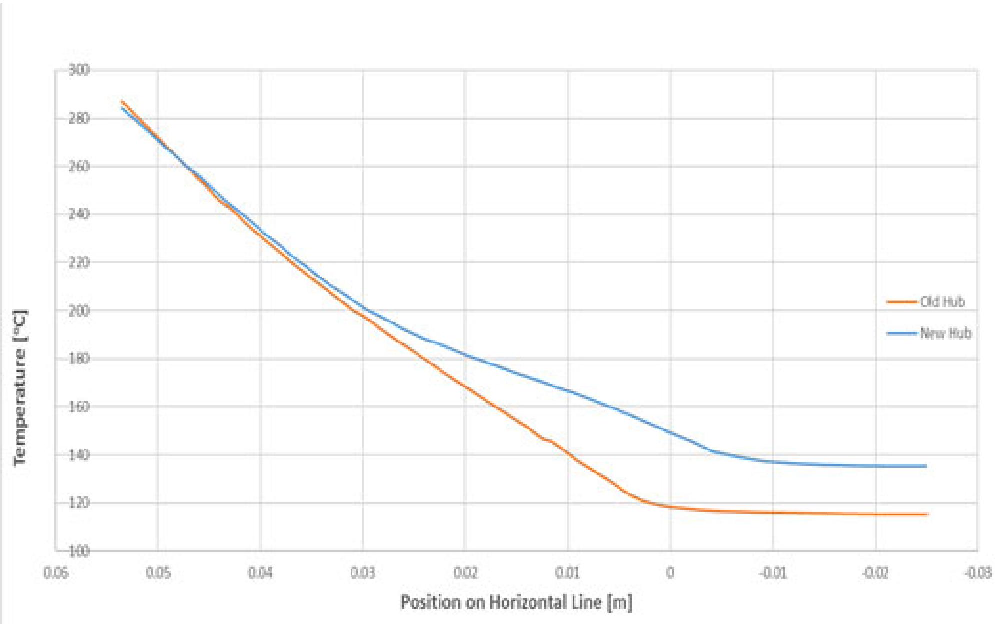

Figure 5. Temperature distribution through line one for both designs

Figure 6 shows the temperature distribution from line 2 in the models, the new model shows a higher overall temperature. This would be due to the thicker geometry sections already discussed which causes a higher temperature at the centre of model. The hub is in use to control the temperature in the shaft so the motor bearing does not become too great, to measure this a point was added where the bearing would sit and is located at the end of line 2. At this point the temperature when using the baseline hub model is 117°C and on the new hub design 135°C, an increase of 13%.

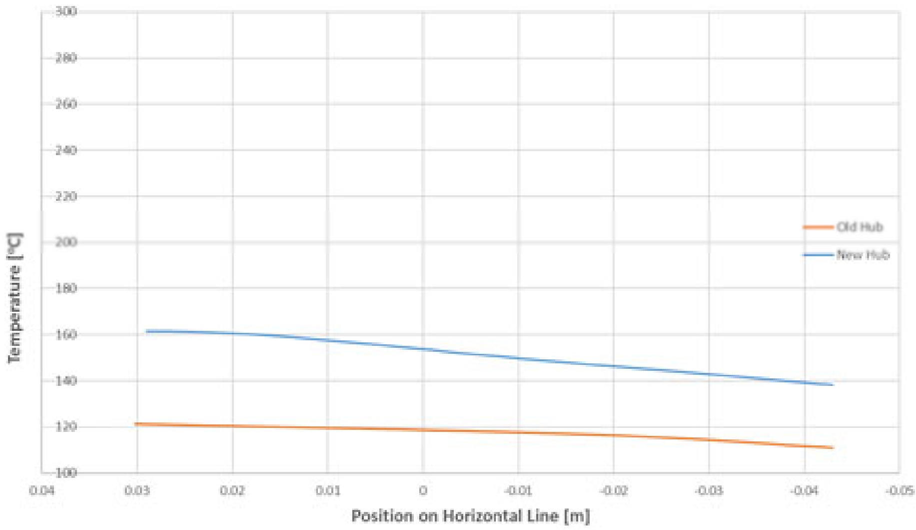

Figure 6. Temperature distribution through line two for both designs

This can pose a potential problem, as the dissipation of heat by new hub is not as effective as the old design, which means that the bearing temperature will increase. The main difference in the geometry is the internal fin arrangement of the baseline design compared to the machined holes of the new design. From the literature reviewed, it was clear to see that existing methods for dissipating heat away from a rotating shaft involve using finned apparatus.

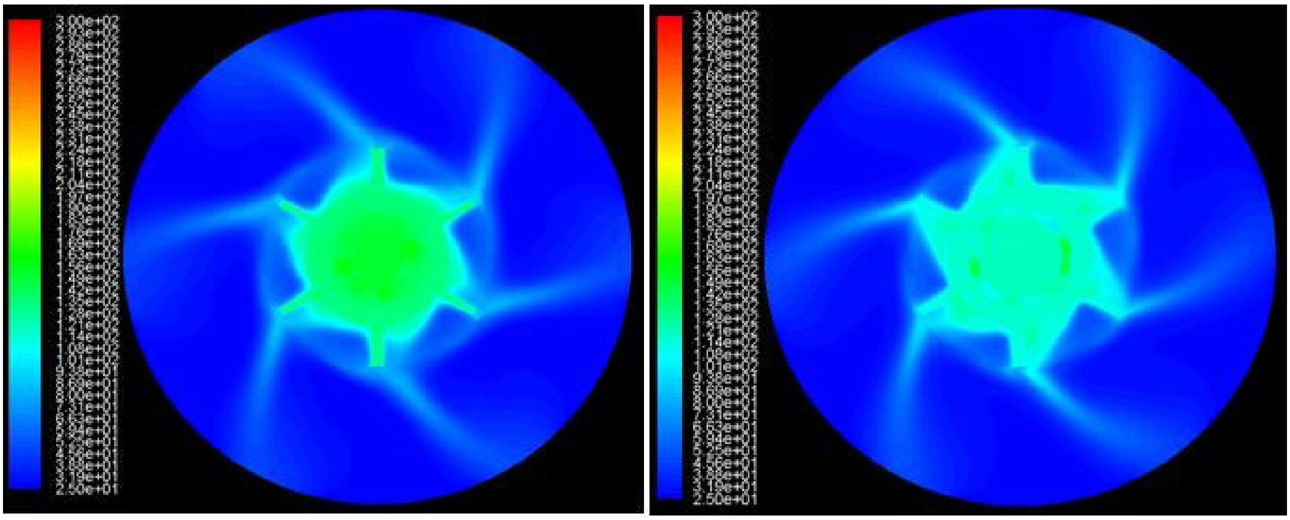

Figure 7. Temperature distribution in the vicinity of the cooling discs

Figure 7, temperature distribution through the cooling disc sections for both designs

Figure 7 depicts a cross section view through the cooling discs showing the temperature contours, the baseline illustrates a lower overall temperature. However by removing material through machining holes the new design has shown it is possible for other methods to be applied to prevent high temperatures travelling through a shaft, even if the temperature reduction is not as great. This difference is shown in the calculated heat transfer coefficient; this was taken at the edge of the cooling disc for both models. It was found to be 118.18 W/m^2k on the baseline model and 31.41W/m^2k on the new hub model. This shows the difference in the amount of heat transfer from the models to the surroundings.

CONCLUSION

Whilst the old hub design performed better from the simulation with respect to the temperature at the point where the motor bearing sits on the shaft. The new design can still be used for high temperature applications, something that is preferable due to its cost effective design, the reduction in its heat dissipating capabilities however will need to be factored into the design limitations of the product.

REFERENCES

[1] L. A. Zainullin, M. V. Kalganov, D. V. Kalganov, V. F. Yarchuk, (2015) Cooling the rotating shaft of a high-temperature furnace fan, Steel in Translation, 45 (9) 646-649

[2] Aziz, A., & Khani, F. (2010). Analytic solutions for a rotating radial fin of rectangular and various convex parabolic profiles. *Communications in Nonlinear Science and Numerical Simulation*, *15*(6), 1565-1574.

[3] Watel, B., Harmand, S., & Desmet, B. (2000). Influence of fin spacing and rotational speed on the convective heat exchanges from a rotating finned tube. *International journal of heat and fluid flow*, *21*(2), 221-227.

[4] Xie, M., Xue, Z., Qu, W., & Li, W. (2015). Experimental investigation of heat transfer performance of rotating heat pipe. *Procedia Engineering*, *99*, 746-751.

[5] Mori, M., Novak, L., & Sekavčnik, M. (2007). Measurements on rotating blades using IR thermography. *Experimental Thermal and Fluid Science*, *32*(2), 387-396.