Heat transfer enhancement inside clean room robots mechanical elements

PhD. Eng. Alin Ristea[1, a]

[1]53 Tower Court, CB7 4XS, Ely, Cambridgeshire, England, UK

[a]alin.ristea@gmail.com

Keywords: Industrial robot, Clean room, Heat removal

Abstract. Industrial robots design poses challenges from a multidisciplinary perspective. This paper covers a few aspects regarding heat related problems in industrial robots mechanical elements. Heat management is very important as temperature is one of the key factors to be accounted at the design stage. This problem is approached for industrial robots working in clean room environments, where heat removal combined with maintaining the required negative pressure inside the robot body is proposed. The optimum position of the inlet for the flow is shown as well.

Introduction

Many applications require industrial robots that would suit operating in clean room environments. For this purpose, the manufacturers had to come up with solutions that will solve the problem of particles being generated by the moving parts of the robot contaminating the environment.

One of the solutions used in the industry is applying negative pressure to the robot body, thus no particles generated inside the robot can be released in the environment where the robot operates. One of the challenges of using this solution is that it requires an additional installation, like shown in [1]. This means an additional expense in terms of cost and space required. If other advantages would yield from such a solution, then it could become more attractive, given that it would be very safe to use it in terms of particle generation risk.

This paper is proposing the usage of this solution for this purpose, as well as for cooling the motors inside the robot body, by removing the heat generated by the motors if a convenient flow rate can be provided. Heat management in industrial robots can poses design challenges, like shown in [2] and [3].

Simulation setup and results

The model was setup as one robot element, containing one electric motor. The shapes have been kept as simple as possible, to avoid long time runs for solution convergence, given that Computational Fluid Dynamics was used. A volumetric heat of 10[W] was applied to the motor. The walls of the case have been considered to be adiabatic. A volumetric flow rate of 20[LPM] was used. The fluid was considered to be air at 298.2[K]. Figure below shows the setup diagram.

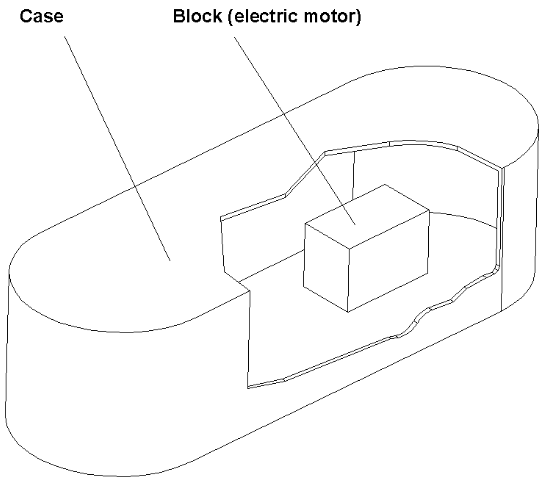

Figure 1. Setup diagram

The dimensions between the two centers was chosen as 350[mm], while the height was 100[mm]. For the simulation, the motor was arranged in vertical position, following the trend of most of the designs in industrial robots. That means the axes of both ends of the case, are parallel with the axis going through the centre (shaft) of the motor. The size of the motor was chosen as 50[mm] for the cross section square, and 60[mm] for the length of its body.

Two runs have been performed. The first run had the inlet on the opposite side of the motor position. The outlet was always kept near the motor, given in a real world scenario that would always be the case, as the flow would need to go through the robot joints. The results from the first simulation are shown in the figures below.

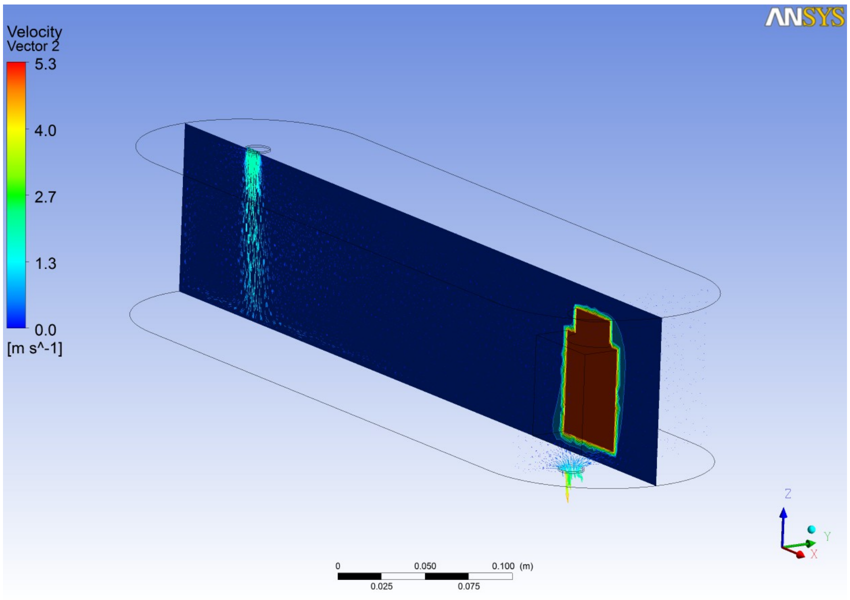

Figure 2. First simulation shows the velocity vectors in section view

The temperature at steady state was also determined, as seen in the figure below.

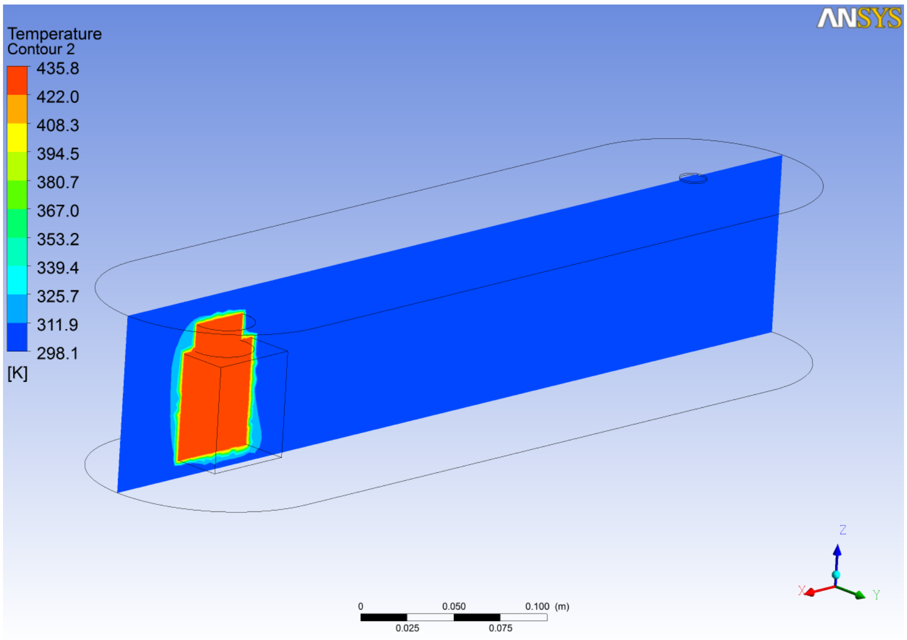

Figure 3. First simulation shows temperature distribution at steady state

The temperature at steady state was determined as being 435.8[K]. The second simulation was run with the inlet position changed, placed above the motor. The picture below shows the velocity vector plot.

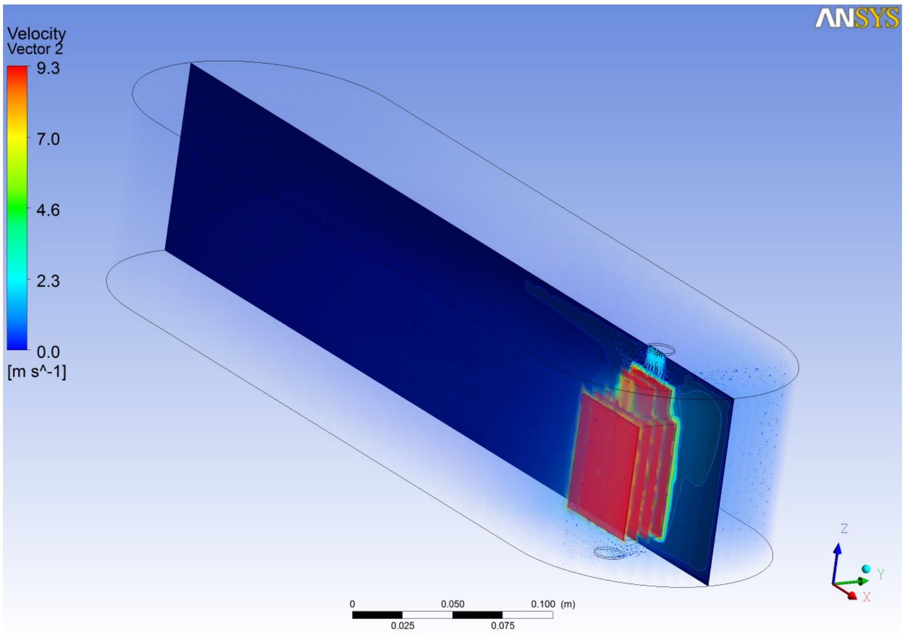

Figure 4. Second simulation shows the velocity vectors in section view

The velocity vector field plot is shown in section view. The air stream flows perpendicularly on the motor. The round shape above the motor emulated the presence of an encoder or electromechanical brake, which is the case in most of the robots used. The shape gives a more realistic representation of the motor, which has an impact to the airflow.
The temperature distribution plot for the second run is show in the figure below.

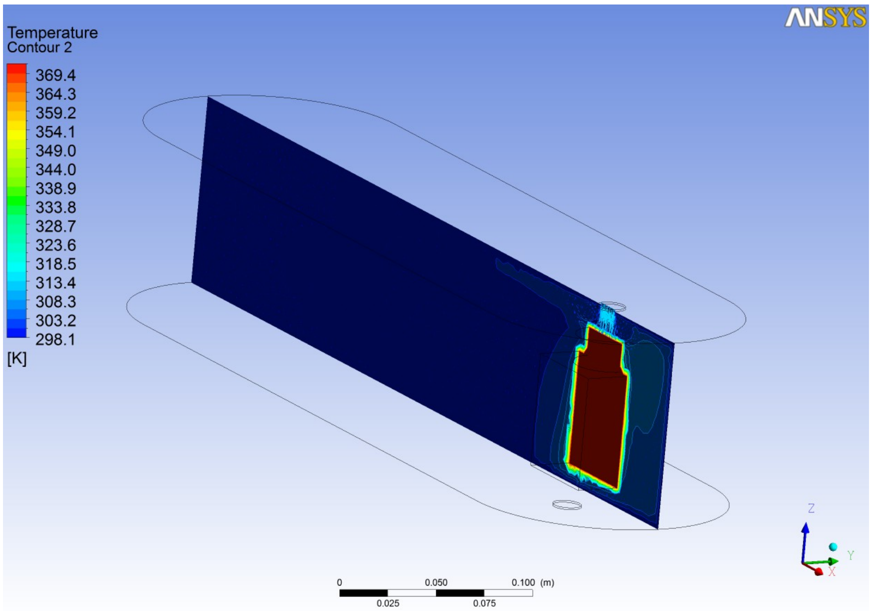

Figure 5. Second simulation shows temperature distribution at steady state

The temperature seen in the second run is lower, as expected, given most of the mass flow rate will participate on the heat transfer between the motor and the air stream, in this case. The value seen is 369.4[K]. The reference ambient temperature used was 298.3[K]. So the temperature in the second case, where the air stream flows directly over the motor will be lower with roughly 50%, which represents a very significant improvement, given that a lower temperature means a higher lifetime of the motor and presents other advantages as well.

Summary

The study shows the benefits of using the negative pressure created inside a robot for clean room purpose for heat removal, when the shown flow rates are permitted. Also, the suction mechanism position is shown as being very important, given the temperature in the motor can be reduced by a significant amount, if all the air stream is guided and used to participate on the heat transfer process.

Acknowledgement

I would like to take this opportunity for thanking Mr. Gavin Butcher from Ansys UK for providing Ansys CFX, the CFD tool that made this work possible, as well as to Prof. PhD. Eng. Adrian Olaru for the invitation and for proposing me as a reviewer for the papers involving CFD and heat transfer work at this international conference.

References

[1] IAI, Ultra Compact Cleanroom SCARA Robot, May 2006

[2] A. Ristea: *Towards a predicting model for heat transfer inside an industrial robot arm* (UPB, Romania, 2010).

[3] A. Ristea: *Thermal Yields in The Mechanical Elements of Clean Room Industrial Robots*, Optirob 2009, Constanta, 2009.