Modeling and Optimization of Coolant Channel Cross Sections in Induction Heating Coils

K. Svendsen(1, 2) and S. T. Hagen(1)

(1)Telemark University College,
P.O. Box 203, 3901 PORSGRUNN, Norway

(2)EFD Induction AS,
P.O. Box 363, 3701 SKIEN, Norway

ABSTRACT. In induction heating applications where high currents and intermittent power application dominate, thermal fatigue can be a limiting factor of the service life of the induction coil. The outer geometry of the coil is often set by the application, hence the temperature in the cross section has to be affected by the inner shape of the coil, coolant velocity, and thermal properties of the coolant. In this paper a model for estimating the temperature of the cross section is presented. The model is also applied to an optimization problem where a high power loss, copper region is surrounding a wedge shaped cooling channel. The point of the wedge was replaced by a radius that was optimized. The optimum was considered where the thermal fatigue service life is maximized, i.e. where the peak deviation from mean temperature in the cross section was at a minimum. The results show that the optimum corner radius is very small, typically 0.3-0.5 mm.

INTRODUCTION

In induction heating applications the induction coil is a component of crucial importance. The life expectancies of these units are depending on the environment the coils are exposed to and application where they are used. There are numerous fault mechanisms limiting the service life [1, 2]. A properly designed coil in a less demanding application may have a very long service life limited by normal wear. In other applications, there are coils that have much shorter service life, even if they seem to have been designed well. Typically coils used in applications that imply high current densities and short heat cycles have shown to fail prematurely. In these cases thermal fatigue is expected to be a limiting factor of service life. The fatigue life of an induction coil is determined by the thermal stresses in the coil. The thermal stresses are generated by temperature differences in the copper of the coil. It has been shown that there is a relation between the thermal fatigue based service life and the difference between the peak temperature and the mean temperature in the cross section of an induction coil [3, 4]. A manual temperature model of an induction coil was previously developed and compared to experiments [5]. This model is developed further in this paper. When the temperature profiles are found, the optimum solution can be determined by investigating the peak temperature difference from the mean temperature of the different cases. This paper first describes the fully automated model. The second part describes the use of this model to investigate the influence of the radius of the coolant channel wall in a corner of a triangular cross section induction heating coil.

MODELING TEMPERATURE

The model is based on iteration between a Fluent®-based model and a Flux®-based model. Fluent is a CFD (Computational Fluid Dynamics) software package. Flux is a package for calculations in the electromagnetic domain. They are both capable of including thermal calculations. The packages are based on finite volume method and the finite element method, respectively, and they are both commercial codes. The combined model handles a 2D plane-symmetric geometry as shown in Figure 1.

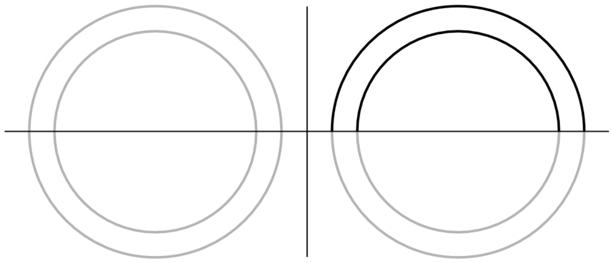

Figure 1 2D plane-symetric geometry with horizontal and vertical symmetry lines. Parts removed from the computational domain by the symmetries are greyed.

The geometry modeled consists of two anti-parallel coil conductors separated by an air gap. The principle behind the model was suggested and validated by experiments in [5]. The electromagnetic domain is modeled in 2D. Due to the anti-parallel structure two symmetry lines can be introduced. The vertical symmetry plane is located between the two tubes where no magnetic flux is crossing the symmetry plane. The second, horizontal symmetry line goes through the middle of the two tubes where all magnetic flux lines are perpendicular to the symmetry plane. After introducing the symmetry-lines the model is reduced to one quadrant of the total domain as shown in Figure 1. Due to the nature of electromagnetics the current is concentrated in the part of the cross section close to the vertical symmetry plane. This will also be where most of the joule losses are dissipated. The heat is conducted to the inner wall of the tube where the heat enters the coolant.

Since the flow in a tube develops along the tube the fluid dynamic model was chosen to be the realized in 3D. The shape of the fluid dynamic model is an extruded version of the 2D plane symmetric geometry and consists of two parts as shown in Figure 2. The first part is a preheat volume where the coolant enters the tube with a flat velocity profile. This preheat volume is made long enough for the velocity and temperature profile to enter a fully developed state before the coolant enters the second part; the measurement volume.

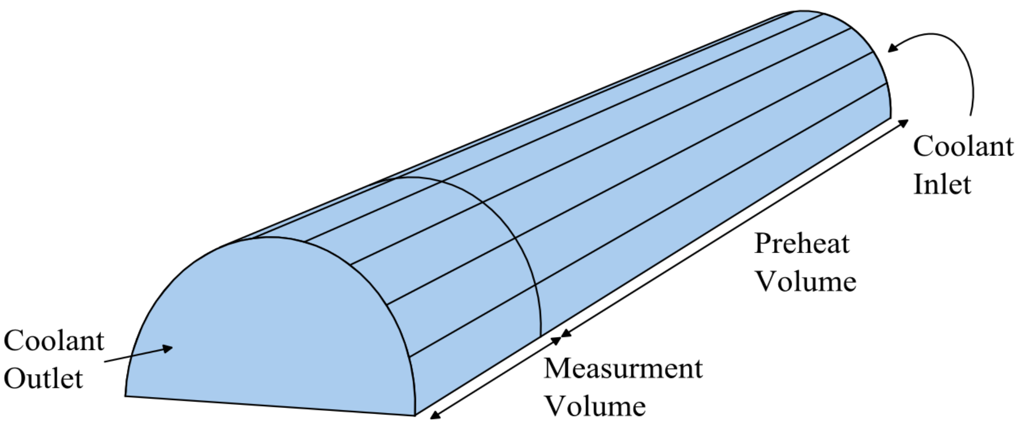

Figure 2 The fluid dynamic geometry. The length of the preheat volume is shortened in this illustration.

Normally the length would be in the range of 50 times the width of the coolant volume. The measurement volume is a relatively short volume where the heat transfer coefficient of the fluid is calculated. The length of the measurement volume is long enough to calculate the heat transfer coefficient without being too much influenced by back flow or geometrical inaccuracies.

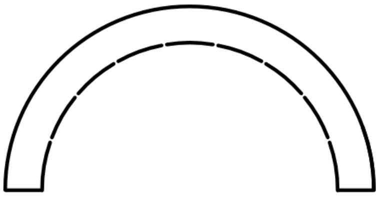

Figure 3 Heat exchange regions on the inner surface of the electromagnetic model.

The two domains are connected at the inner wall of the tube, the wall of the cooling channel. This wall is divided into heat exchange regions along the (inner) wall as shown in Figure 3 (and on the outer surface of Figure 2). The number of heat exchange regions is depending on the application. They must be sufficiently small, so that the temperature and the heat flux can be considered constant over the region.

The flow sheet of the Python program controlling the solving process is shown in Figure 5. The solving process is started by estimating the heat transfer coefficients of the heat exchange regions in the electromagnetic domain. Since there is no water region in the electromagnetic domain, boundary conditions are set on the heat exchange regions; a heat exchange coefficient and a temperature are specified for each region. Here a constant temperature of 20°C is used for all regions. To start the solving process, the heat transfer coefficient of all exchange regions is initialized. The electromagnetic domain is solved and the temperatures and the heat fluxes of the heat exchange regions are recorded. The heat flux of each region is then applied to the corresponding heat exchange regions in the fluid dynamic model, both the corresponding region in the preheat volume and in the measurement volume. The model is solved and the temperatures of the heat exchange regions in the measurement volume are collected. The temperatures from the electromagnetic and the fluid dynamic domains are compared. If they are all within a predetermined difference the solution is considered converged.

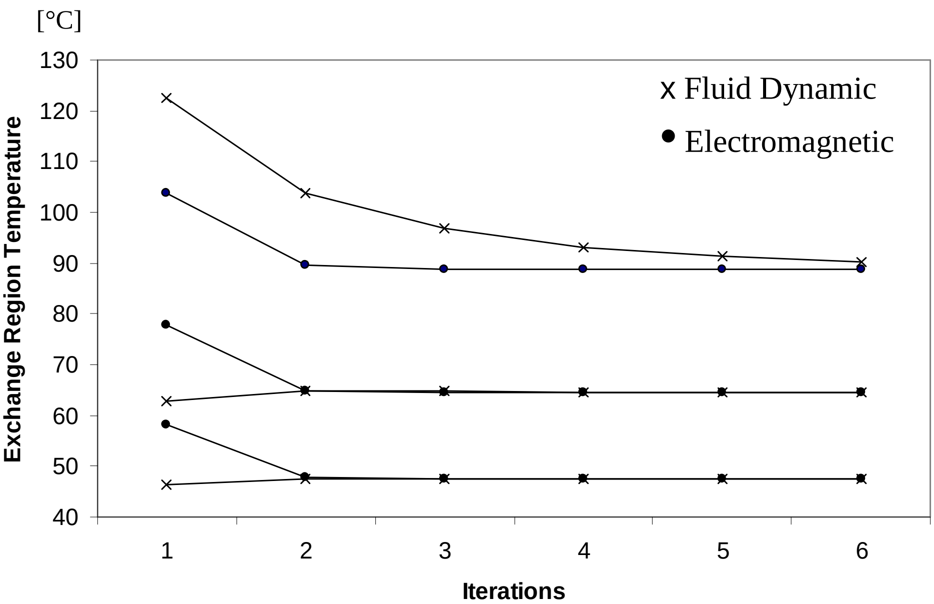

Figure 4 Iteration. Temperature of three different exchange regions in the electromagnetic and in the fluid dynamic domain converging.

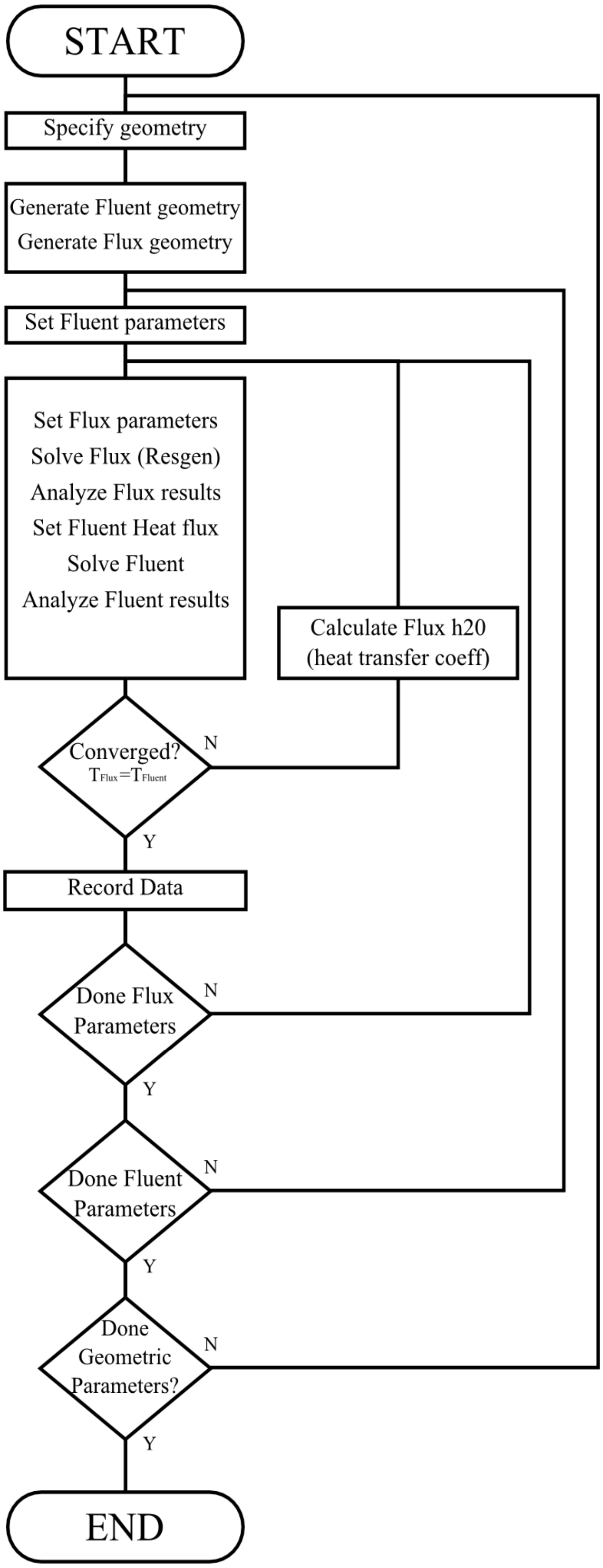

Figure 5　Flow sheet of the solving process.

If the difference is too large, new heat transfer coefficients calculated from the heat flux and the temperature of the regions. A relaxation factor was introduced so the new estimated heat transfer coefficients were chosen a certain factor between the new estimate and the previous value. It was found that this would generally not improve the converging process. A typical way the solution converges over a few iterations is shown in Figure 4. The convergence criterion is that the highest temperature difference between two corresponding regions must be less than 2°C.

INVESTIGATION OF CORNER EFFECT IN COIL

The previously described model is utilized to investigate the effect of the radius of a corner of the cooling channel of a triangular induction coil cross section. The cross section is shown in Figure 6.

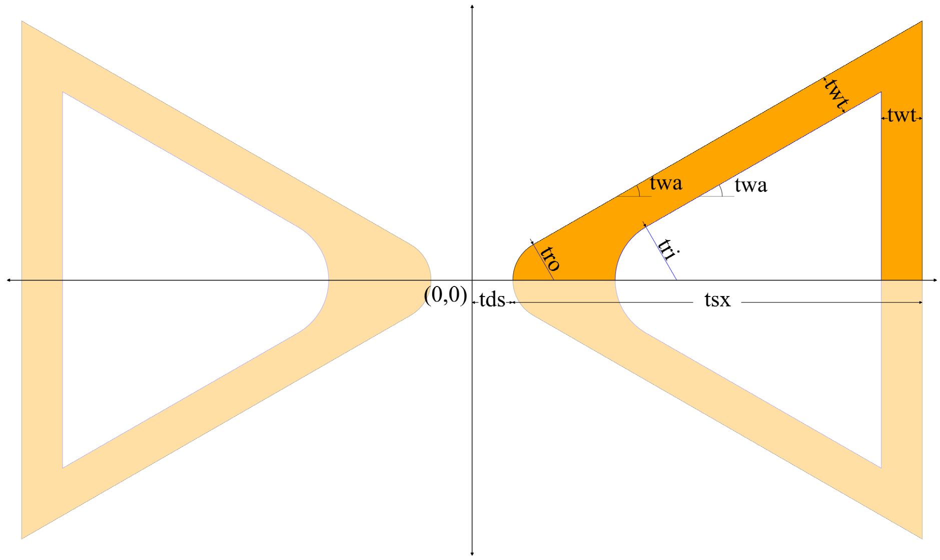

Figure 6 Geometry for investigation of effect of corner inner radius. (**tri** – Tube Radius Inner, **tro** – Tube Radius Outer, **twa** – Tube Wall Angle, **twt** – Tube Width, **tsx** – Tube Size X-direction, and **tds** – Tube distance to vertical Symmetry line.

In this cross section two effects are expected to influence the heat transfer. When the inner radius is increased the surface area of the cooling channel is reduced. This is expected to reduce the heat transfer from the wall to the water. On the other hand, if the corner is to sharp the water velocity in the corner is reduced with a potential of reducing the heat transfer in this important region. The inner radius, '**tri**', and the tube wall angle, '**twa**', of the cross sections are changed to see how this would influence the temperature of the cross section.

The model is set up with the default Fluent properties of water for the fluid. The water inlet has a constant velocity boundary condition of 10 m/s. The water is entering the domain at 20 °C. There is a pressure outlet boundary condition on the outlet and kε-modeling is used for turbulence. In the electromagnetic domain a current of 2.5 kA at 10 kHz is applied to a solid conductor region. The standard copper material properties of Flux are used for the copper region. Thermal insulation is applied to the outer boundaries of the coil, i.e. no conduction, convection, nor radiation through the outer surface. All heat generated in the domain leaves through the heat exchange regions. A transient thermal solver allowed to stabilize is used in the electromagnetic domain and a stationary solver in the fluid dynamic domain.

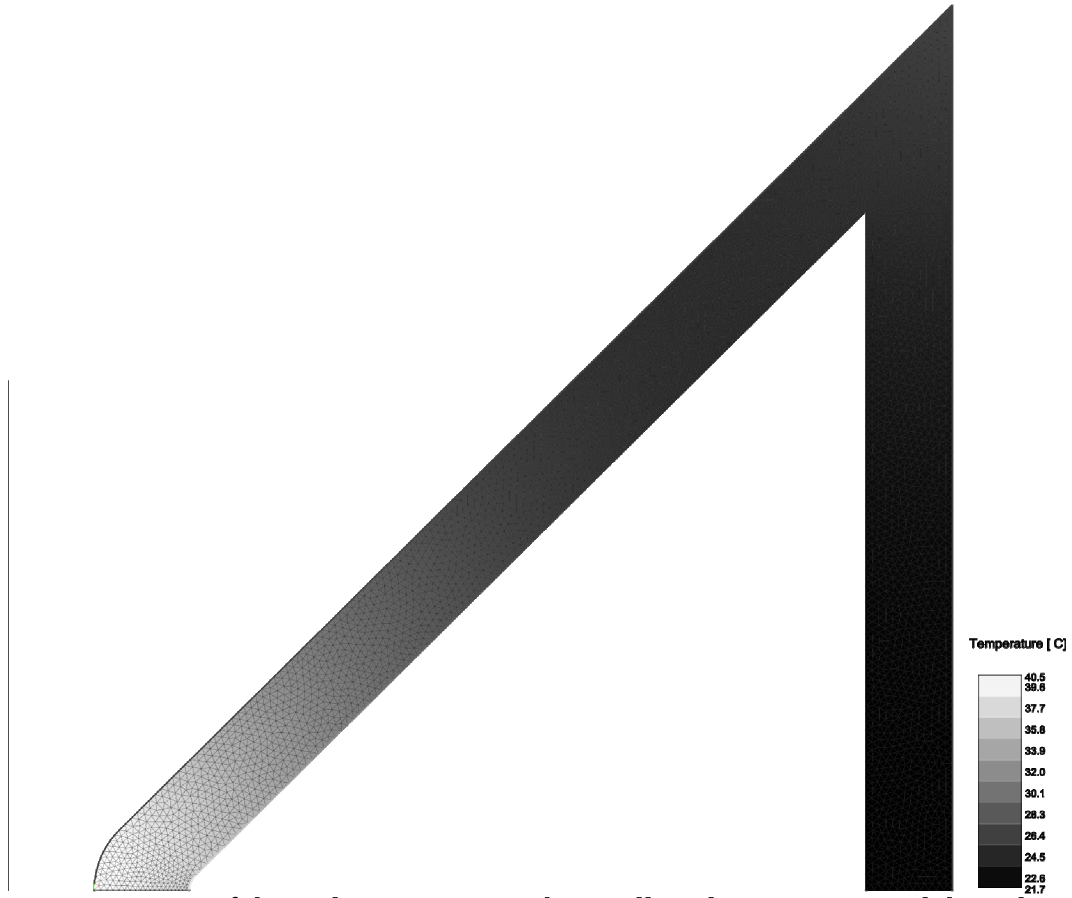

Figure 7 Temperature of the coil cross section when wall angle, **twa** = 45° and the radius of the inner corner, **tri** = 0.25 mm.

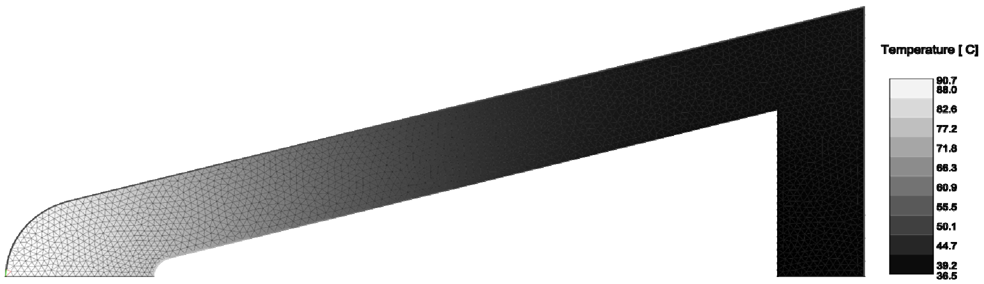

Figure 8 Temperature of the coil cross section when wall angle, **twa** = 15° and the radius of the inner corner, **tri** = 0.25 mm.

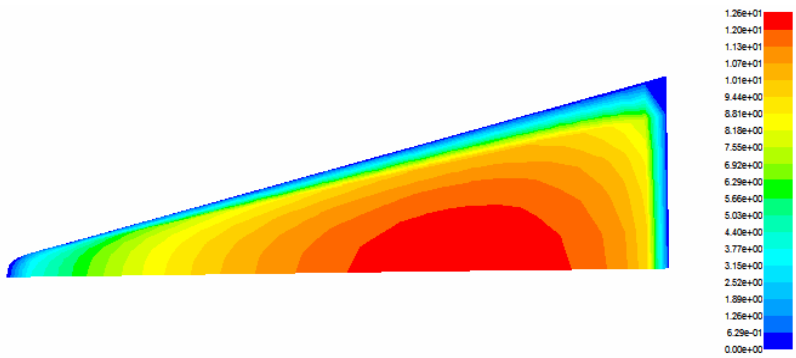

Figure 9 Velocity inside the coil cross section when wall angle, **twa** = 15° and the radius of the inner corner, **tri** = 0.25 mm. Coolant velocity in = 10m/s.

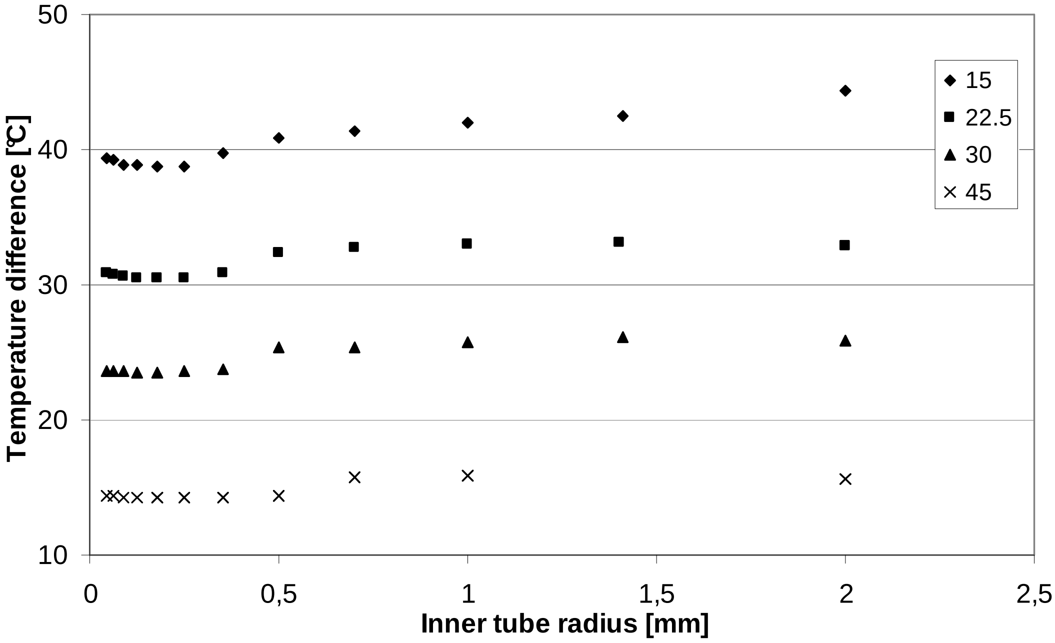

Figure 10 Temperature difference (peak-mean) per tube with different angles.

Two selected cross sections are showed in Figure 7 and in Figure 8. The coolant velocity profile corresponding to the tube cross section shown in Figure 8 is presented in Figure 9. The highest velocity region is located in the right, central side of the cooling channel, away from the region where most of the heat is generated. The velocity closer to the radius is significantly lower than the free stream velocity.

Since the fatigue life of the coil is related to the difference between the peak and the mean temperature of the coil cross section [3], this is the factor that is extracted form the model. The model is investigated with the following settings **tds** = **tro** = **twt** = 1mm and **tsx** = 10mm. The wall angle, **twa**, is set at 15, 30, and 45 degrees and the inner radius of the corner, **tri**, is varied gradually up to 2 mm. Depending on the geometry a range of 13 to 41 heat exchange regions have been utilized. From the generated electromagnetic data files information about the elements in the domain is collected. This information is used to calculate an area weighted mean temperature. In the same process the maximum and minimum temperature are found. From these values the maximum temperature deviation from the mean temperature is found. The resulting temperature deviations are shown in Figure 10.

As seen from the graph there is an optimal radius or a radius range for the different cross sections. The temperature difference is lowest for all the different tube wall angles, **twa**, when the inner tube radius is around 0.2mm. This effect gets more pronounced as the angle of the corner is reduced i.e. when the tube wall angle, **twa**, is reduced.

CONCLUSIONS

A script based model feeding an electromagnetic (Flux) and a fluid dynamic (Fluent) solver is realized. The model obtains a solution that is valid in the electromagnetic and in the fluid dynamic domain through iteration until an agreement of the temperature of two sets of exchange regions (one set in each domain) is obtained. The concept has been utilized to investigate the effect of the corner radius of a triangular cross section used in an induction coil. An optimum was considered where the thermal fatigue service life is maximized, i.e. where the peak deviation from mean temperature in the cross section was at a minimum. It was found that for the higher corner angles there was a maximum recommended corner radius in the 0.25 to 0.5mm range, and for smaller corner angles there seems to be an optimum corner radius about 0.25 mm.

REFERENCES

[1] V. I. Rudnev (2005-2007): Systematic analysis of induction coil failure. Part 1 – 11. Heat Treating Progress.

[2] W. I. Stuehr, D. Lynch (2007): How to Improve Inductor Life, Part 2, Proceedings of the 24th ASM Heat Treating Society Conference, Detroit

[3] K. Svendsen, S.T. Hagen (2008): Thermo-Mechanical Fatigue Life Estimation of Induction Coils, International Scientific Colloquium of Modeling for Electromagnetic Processing, Hanover, 139-144

[4] V. Nemkov, R. Goldstein (2007): Influence of Cooling Conditions on Induction Coil Copper Temperatures. Proceedings of the International Symposium on Heating by Electromagnetic Sources, Padova, 191-199

[5] K. Svendsen, S. T. Hagen, M. C. Melaaen (2007): Temperature distribution in selected cross sections of induction heating coils. Proceedings of the International Symposium on Heating by Electromagnetic Sources, Padova, 357-364