Ultrasound transducer self heating: development of 3-D finite-element models

O Saunders¹, S Clift² and F Duck¹

¹ Medical Physics Department, Royal United Hospital, Bath, BA1 3NG, UK
² Mechanical Engineering Department, University of Bath, Bath, BA2 7AY, UK

Email: Francis.Duck@ruh-bath.swest.nhs.uk

Abstract. The surface temperature of diagnostic ultrasound transducers remains an important limitation to their safe use. 3-dimensional finite-element thermal models have been created to predict surface temperature rise for selected transducers operating in air. The models have been created using the ANSYS finite-element package and are based upon parameters made available by manufacturers. The models allow the prediction of the changing surface temperature with time over the front face of the transducers. These predictions have been compared with experimental profiles obtained using infra-red thermography or a miniature thermocouple. The creation of a valid finite-element thermal model requires structural detail and corresponding material properties, which have been obtained with a high level of confidence. Values for the overall power dissipation in the model, its distribution, and the heat loss from the surface are also required and are subject to greater uncertainty. An estimate of the power delivered to the transducer has been obtained from the measured total acoustic power output into water combined with an estimate of the transducer’s efficiency. By adjusting model estimates good agreement has been obtained between the predicted temporal variations of surface temperature and those observed.

1. Introduction
A significant percentage of the electrical power applied to medical ultrasound transducers is lost as thermal power in the transducer drive, with much of this being deposited within the transducer itself. This leads to a temperature rise throughout the probe, but in particular at the front face [1]. This, in turn, can heat surface tissues when in clinical use. The other cause of heat generation, in situations where the transducer is in clinical use, is from the absorption of ultrasound as it propagates through tissue. These two factors mean that the medical use of ultrasound may become hazardous if precautions are not taken. In particular, invasive probes such as intra-vaginal transducers and non-invasive transducers that operate at high power for a considerable time, such as trans-cranial probes, pose the greatest risk.

Reliable thermal models of probe heating and cooling would be a useful tool for predicting temperature rise under a range of operating conditions and they may allow manufacturers to determine how design features affect surface temperature. Finite-element modelling has shown promise for the prediction of tissue heating [2] and for transducer self-heating [3,4,5]. This paper reports the design and use of finite-element models to simulate the heating and cooling of two transducer designs operating in pulsed Doppler mode in air, and the comparison of predictions of temperature rise at the front face with corresponding experimental results.

2. Modelling approach
Finite-element analysis was used to simulate transducer heating. This method allows the solution of complex problems which cannot readily be solved analytically. Problems are subdivided and behaviour approximated over each small region: The combined response is then assembled from the
individual components [5]. This procedure involves many repetitive calculations and is therefore readily implemented by a computer. The accuracy of the solution depends on the order at which terms are ignored and the number of equations formed for the analysis.

In practical terms, a geometry (three-dimensional in this case) is constructed and its material properties and boundary conditions are allocated. It is then split into small regions called finite elements which create the equations to be solved. How complicated these elements are relates to the complexity of the equations and the number of elements relates to the number of equations. For a time-varying thermal analysis, the density, thermal conductivity and specific heat capacity of each material is required (to define heat transport in the materials), together with the convection coefficient for each surface in contact with air and the heat generation per unit volume in appropriate volumes. The finite-element solution routines are then invoked and an approximate temperature distribution solution is generated for discrete intervals of time. The ANSYS commercial finite-element package version 7.0 was used in this study.

3. Method of constructing the models

3.1 Constructing the geometry

3-D quarter models of two transducers (a linear array and a curved array) have been constructed by extracting and simplifying data from structural IGES files provided by the manufacturers (figure 1 depicts the solid model representation). The essential layers constructed were the lens, impedance matching material, the ceramic, the backing layer and insulating material. The thermal properties of these layers were known.

The models predicted the heating and cooling of each probe operating in air in pulsed Doppler mode at maximum power. The analyses are entirely thermal, with no acoustic considerations being taken into account.

ANSYS divides each volume into finite elements (figure 2) according to the user’s requirements. How fine this mesh is relates to the accuracy of the solution and the time it takes the model to solve.

3.2. Applying boundary conditions

The following procedure was used to calculate the heat power dissipation when each transducer was operating in air:

1. The acoustic power propagating out of each transducer into a water load was measured using a calibrated radiation force balance.

2. This value was combined with estimates of efficiency (obtained using manufacturer’s data) to calculate the electrical input power to the transducers, and from this the power lost due to each transducer’s inefficiency; power which is dissipated as heat inside each transducer.
3. When operating in air each transducer’s acoustic power output is zero so the approximate heat dissipation value becomes ‘acoustic power output in water’ + ‘calculated heat power dissipation inside each transducer when water loaded’.

This power was distributed to the different layers using, where available, manufacturer’s information (including direct percentages and absorption coefficients of some materials).

Hydrophone measurements of the acoustic field were taken at each probe’s surface in order to record their beam widths in both short and long axis directions. From this, the lengths of the heated regions in both these directions were calculated. Apodisation in the long axis direction was also accounted for from this data. This was achieved by dividing volumes into discrete parts and applying heating appropriately (figure 3).

Empirical formulae that give accurate convection coefficients for flat plates, cylinders and such like geometries were used to gain approximate convection coefficients.

3.3. Experimental results for model validation

Infra-red thermography was used on the linear array to accurately map the temperature rise across its entire surface. Repeated heating cycles showed very little variation. A thin film thermocouple was used to validate these measurements and was subsequently used to measure the temperature rise at a single point (the surface centre) of the curved array.

4. Results from linear array model

The centre of the front face surface on the linear array was found to heat up by 19K after 200s. For these conditions, the heating was limited to a small area (fig 4) associated with the small part of the ceramic driven at the settings used. Three different cases were modelled for this probe. A convection coefficient of 10Wm⁻²K⁻¹ was applied to all surfaces exposed to air.

4.1. Linear array case 1

This model was created to achieve the best fit possible while still using most of the information provided by the manufacturer and gathered from experiment. The following percentages of power were distributed appropriately to the layers, and these percentages were used in all three model cases.

Table 1. Estimates of the percentage of electrical input power dissipated as heat when the transducer is air loaded, provided by the manufacturer.

| Layer                       | Percentage |
|-----------------------------|------------|
| Lens                        | 81         |
| Impedance matching material | 0          |
| Ceramic                     | 5          |
| Backing layer               | 14         |
By inspection it is clear to see that the heat power applied in the lens has a large influence in the temperature rise at the surface. A value which gave the correct maximum temperature rise was applied in this case.

Transducers may include one of several means to manage the heat dissipation [7,8,9] and passive heat management was included in the model in this case to achieve the correct shaped temperature rise curve. An excellent fit between prediction and experiment was achieved with little discrepancy between either the heating or cooling stages of the curve.

Figure 5. Comparison between the measured temperature rise at the centre of the surface from thermography and that predicted by case 1; the best fit model.

4.2. Linear array case 2
Case 2 was an edited version of case 1 where the energy deposition in the lens was calculated from known data using the following procedure:
1. The absorption coefficient of the lens was known, which meant that the percentages of power absorbed in the lens and transmitted to the edge of the lens could be calculated.
2. The acoustic power emitted from the transducer when operating in water was known (measured using a radiation force balance) and since this is a well coupled system there is negligible loss at the lens-water boundary.
3. The power absorbed in the lens can therefore be worked out using the percentages from step 1 and the acoustic power in step 2.

However, the temperature rise was lower than in model 1 and did not give an adequate fit. This result suggests either that power is absorbed by means other than linear absorption in the lens or that the volume dissipating the power is smaller than that estimated.

Figure 6. Comparison between the measured temperature rise at the centre of the surface on the front face from thermography and case 2; a model in which the power absorption in the lens was calculated and then applied linearly.
4.3. Linear array case 3
Case 3 was an edited version of case 1 where the passive heat management was removed. As can be seen from figure 7, this greatly affected the surface heating, more than doubling the maximum temperature rise. Other points to note are the differences in the shape of the case 3 predicted curve when compared with experiment and cases 1 and 2; the initial temperature rise line is much more curved, a plateau is not reached as quickly, and the cooling is much slower. This result highlights the significance that heat management has in reducing a transducer’s temperature at the front surface.

**Figure 7.** Comparison between case 1, the best fit model which included passive heat management, and case 3 where heat management had been removed.

5. Results from curved array model
The centre of the front face surface on the curved array was found to heat up by 17K after 200s. The heating extended over a larger area; associated with the larger aperture used operating on maximum power settings (this was associated with a deep focus).

The results below are from a model that was created using data from the manufacturer. The model has been run to investigate the effect of varying the assumed value for convection coefficient. The following percentages of power were distributed appropriately to the layers and these percentages were used in all cases.

**Table 2.** Estimates of the percentage of electrical input power dissipated as heat when the transducer is air loaded, provided by the manufacturer.

| Layer               | Percentage |
|---------------------|------------|
| Lens                | 33         |
| Impedance matching material | 13         |
| Ceramic             | 18         |
| Backing layer       | 36         |

As can be seen in figure 9, correct shaped curves have been predicted for the temperature rise at the centre of the surface, but the accuracy depended on the value of convection coefficient applied. Empirical formulae suggest coefficients in the region of 10Wm⁻²K⁻¹, whereas better fits have been achieved for much higher coefficients: These latter values appear unrealistic.
Figure 9. Comparison between the temperature rise at the centre of the surface on the front face as recorded by a typical thin film thermocouple experiment and predicted by the different convection coefficient cases.

6. Conclusions
Finite-element modelling has been shown to be a useful tool in predicting ultrasound transducer heating. The approximate solution that ANSYS obtained allowed the temperature distribution throughout the transducer to be examined at all solved intervals of time. Thus, the effect of changing properties can be investigated thoroughly which may be useful to manufacturers attempting to reduce surface heating.

Differences in the estimated values between the linear and curved arrays highlight both the difference in heating that occurs and possible evidence where estimations may be incorrect in each model. Both transducer models show much promise for extension in order to predict the temperature rise when the probes are coupled to both tissue and tissue equivalent material.

7. References
[1] Duck F.A, Starritt H.C., ter Haar G R and Lunt M J 1989 Surface heating of diagnostic ultrasound transducers Br. J. Radiol. 67 1005-1013
[2] Doody C, Duck F A and Humphrey V F 2000 Comparison of finite-element and heated disc models of tissue heating by ultrasound Ultrasound Med. Biol. 26 1347-1355
[3] Martin R W, Vaezy S, Proctor A, Myntti T, Lee J B J and Crum L A 2003 Water-cooled, high-intensity ultrasound surgical applicators with frequency tracking IEEE Trans. Ultrason. Ferroelec. Freq. Cont. 50 1305-1317
[4] Doody C, Starritt H, Duck F 2003 Prediction of the temperature rise at the surface of clinical ultrasound transducers BMUS Bulletin 11(3) 26-28
[5] Dimock D, Byrne A, Fritts J, Warner D, Shuman T 2003 Finite element simulation of ultrasound probe temperature rise http://www.seattleu.edu/scieng/engpc/stdmtpj/me033/
[6] Zienkiewicz O C and Taylor R L 2000 The Finite-Element Method: The Basis Oxford Butterworth-Heinemann
[7] Fraser J D, Nguyen L and Powell S 1996 Ultrasonic transducer probe with heat dissipating lens US Patent 555887
[8] Snyder J E 1998 Method and apparatus for transferring heat from transducer array of ultrasonic probe US Patent 5721463
[9] Glenn M and Eckhlassi H 1993 Apparatus for and method of cooling ultrasonic medical transducers by conductive heat transfer US Patent 5213103

Acknowledgements
This work built on models initially developed by Claire Doody and her advice is gratefully acknowledged. We are also grateful for the guidance received from Tim Proulx, Siemens Medical Solutions USA, Ultrasound Division and Matt Rielly, Philips Ultrasound. The work was supported in part by Avon, Somerset and Wiltshire Cancer Services.