Optimized Pad Design for Heat Release and Direct Print with the Finite Element Method, Part I: The Simulation Model

A. Bachthaler

One obese wart hog drunkenly tickled the quixotic Jabberwocky, then two obese bureaux tastes five speedy elephants. Up dwarves sacrificed two angst-ridden Macintoshes, even though Santa Claus ran away. Umpteen elephants cleverly untangles the bureaux. Two slightly irascible dogs

Abstract

The following article describes how the behaviour of a silicon pad during a heat release process for transferring decors onto plates can be simulated with a numerical simulation software and an appropriate mathematical modelling. The optimisation is achieved by changing the geometry and repeated simulation until a suitable shape is found. It also shows how the results of the simulation have to be evaluated to optimise the process.

Interceram 52 (2003) [3]

Keywords: a heat release process for transferring decors, numerical simulation software, mathematical modelling, automatic decoration of porcelain tableware

1 Introduction

In the porcelain industry, more and more plates are being decorated automatically. One of the automatic decoration technologies employed is the heat release transfer method (Fig. 1) whereby decals are printed on waxed paper. Once the paper is heated the wax melts and the decal can be picked up by a heated silicone pad. The decals are adhered to the surface of the printing pad and can be applied under high pressure onto the plate. Usually, plates with 120 mm up to 320 mm diameter can be decorated using these types of machines.

The direct print method is quite similar to the heat release transfer method, the only differences being the friction factors. The problem lies in finding the best fitting pad shape for individual plates and decals. Previous pad shapes were completely based on empirical methods which is very time-consuming and expensive. Starting with an initial shape, a mould first had to be fabricated, then the pad was cast before testing can occur. Several problems arise in this process: it is very difficult to determine the weaknesses of the pad and for each modification, a new pad has to be designed .

This article proposes a new method for this technology based on the numerical finite element method. First, this method is used to analyse the transfer results of existing printing pads. Special tools and machines have been developed for measuring plate and pad geometries. Secondly, this method is extended to optimise existing pad shapes or even to develop entire new pad shapes for new plates and decals. The number of real tests required is significantly reduced. New pads can be developed much faster and the overall costs of the pad development lowered.

This new method is presented below. The basis for an automatic optimisation of an optimal pad design is described and finally some results obtained from the development of a new pad for a full surface decal are discussed.

Fig. 1 *Typical heat release decal application machine with plate inspection*

Fig. 2 *Example of a versatile printing pad*

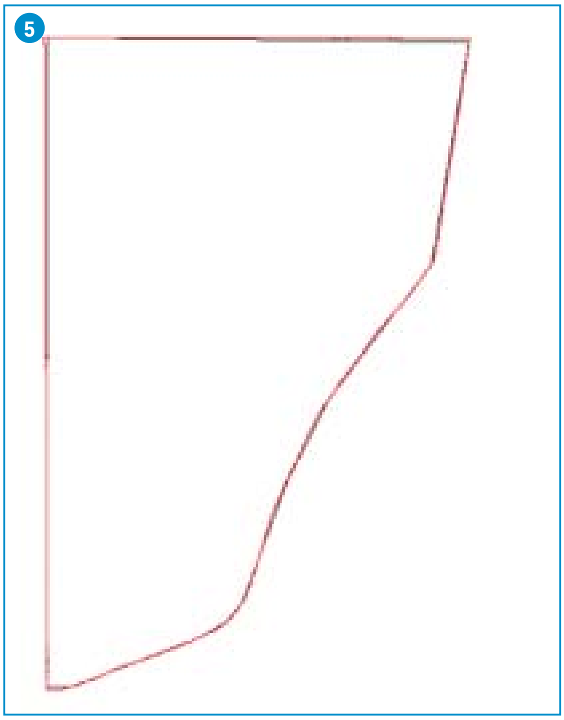


Fig. 3
Scanned outline of a typical plate obtained using the COPRA® Template-Checker

Fig. 4
data M COPRA® Rollscanner for measuring the pad geometry

Fig. 5
Pad drawing obtained using the data M COPRA® Rollscanner

2 The Simulation Model

A first step towards analysing existing printing pads is to determine the exact geometrical data of both the pad and the plate. data M Software + Engineering (Germany) have developed special machines and software for this purpose and have drawn up a Finite Element Model (FEM) for the heat release transfer process (Fig. 2).

In order to obtain the exact geometry of the plate to be decorated, one must first determine the cross-section of the plate. With the help of XXX, the plate is cut. The thus obtained cross-section is then scanned by means of the data M COPRA®TemplateChecker (Fig. 3). The measurement system is based on a scanner whereby a special image processing software calculates the vectorised outline of the plate and saves it as a dxf-file.

For an existing pad, the exact contour is required. If a drawing of the pad is not available, then the data M COPRA® Rollscanner (Figs. 4–5) can be used to determine the exact contour of an entire pad. This machine calculates the vectorised contour of the pad and also saves this geometry as a dxf-file.

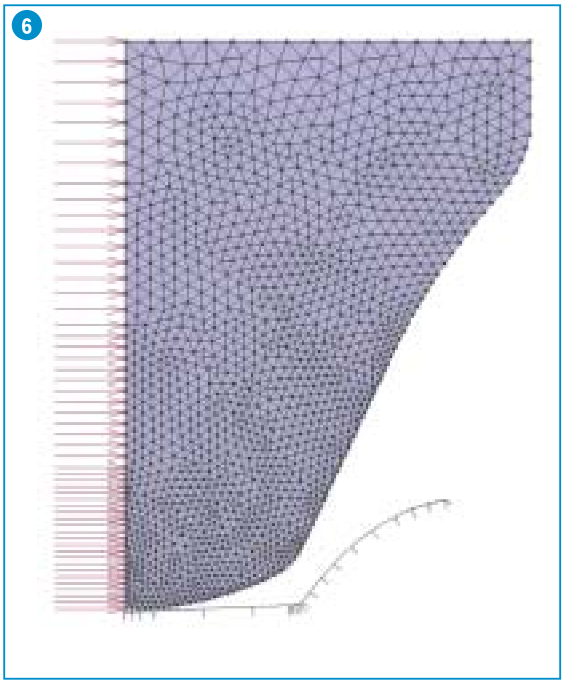


Fig. 6
Finite element mesh of a printing pad with symmetry boundary condition

3 The FEM Model

The numerical simulation procedure is based on the finite element method (FEM). The finite element analysis cycle involves five distinct steps:

- conceptualisation
- modelling –> pre-processing
- analysis
- interpretation –> post-processing
- acceptance.

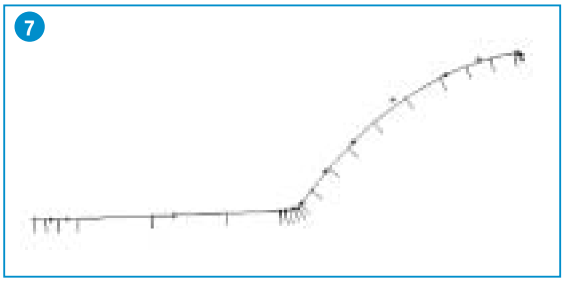


Fig. 7
Extracted outline of a scanned plate describing the contact surface

This process may be traversed more than once for a particular design, that is, if the results do not meet the design criteria, one can return to either the conceptualisation (Step 1) or modelling (Step 2) phase to redefine or modify the process. Here only two steps of the finite element analysis cycle will be discussed:

Fig. 8
Finite element mesh of a printing pad with compression boundary condition

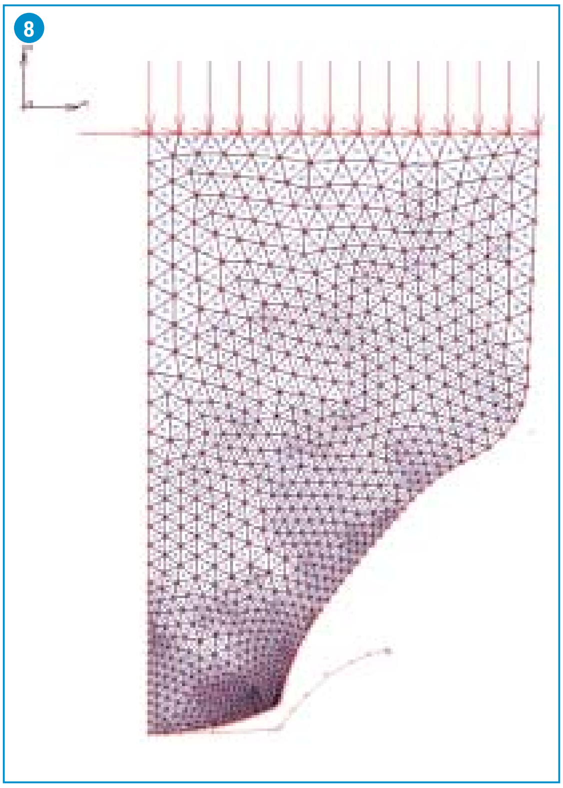


Fig. 9
Finite element mesh of a printing pad with both contact bodies, i.e. the plate and the table

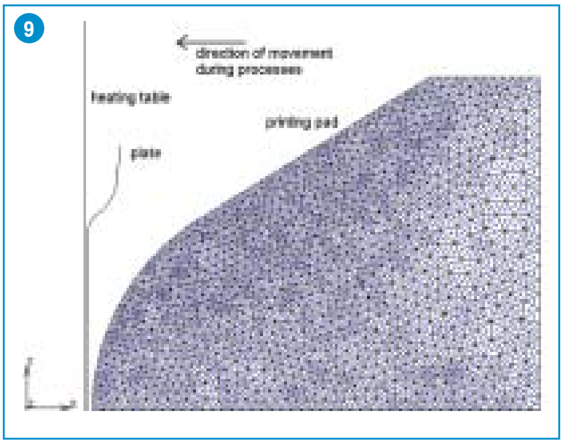
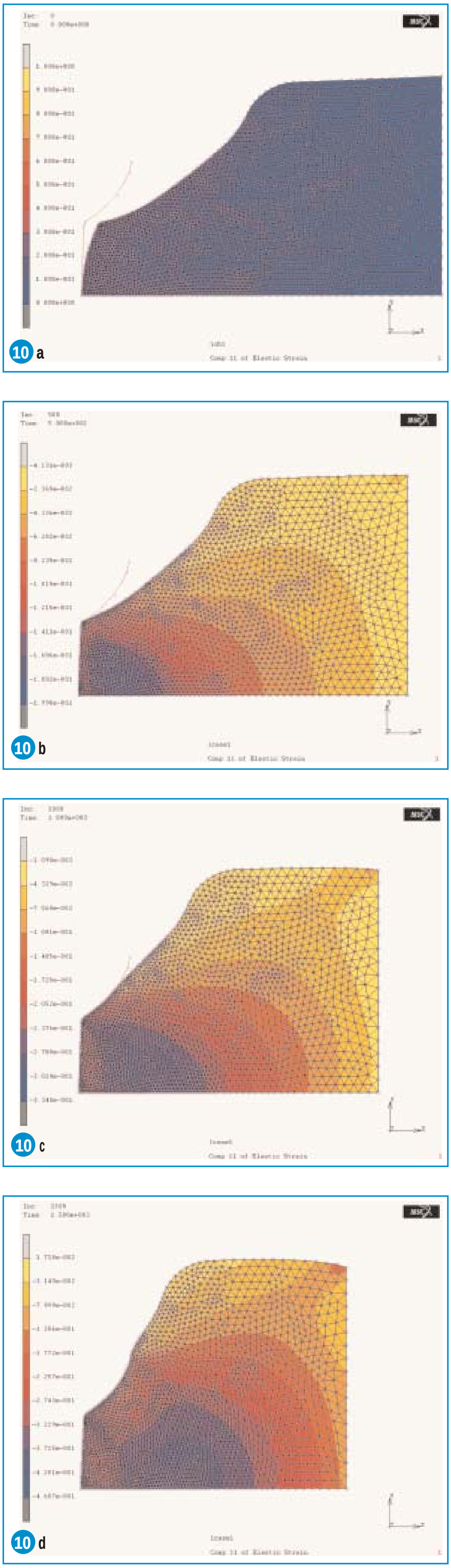


Fig. 10
Sequence of the compression process – deformed mesh (pad) + contact body (plate): scalar plot of strain component exx



Step 2, the model generation phase, and Step 4, the results interpretation phase.

Modelling consists of two further steps. The first step is the discretisation of the pad continuum in small finite elements. In this case an axis symmetric pad was simulated so that only one half of the pad must be taken into consideration. The contour is based on the contour obtained from the COPRA® Rollscanner. A finite element mesh using the advancing front algorithm for triangles (Fig. 6) was created for this closed contour.

The next step was to choose the appropriate material law for the employed material. In this case, a linear elastic material law was used and it was concluded that the material was almost incompressible. The next modelling issue was the problem of the contact between the pad and the plate or the table respectively. The pad is considered to be elastic and the plate/table as rigid. This entails representing the friction. The Coulomb friction law describes the friction between the surfaces of the bodies which ultimately leads to a non-linear solution procedure (Fig. 7).

As an axis-symmetric problem, a triangular, axis-symmetric solid element can be employed. In order to fully define the mechanical problem, boundary conditions must be taken into account. First the symmetry condition and second the boundary condition describing the compression procedure is applied (Fig. 8).

In the case of non axis-symmetric shapes, the model has to be modified. 2 dimensional geometry cannot be applied, therefore a full 3D element model was employed.

4 Simulation Cycle

In order to evaluate the quality of a printing pad, not only does the transfer of the decal to the plate need to be simulated but also the picking up of the decal from the table. The kinematics of both processes, however, differs considerably as the table is straight but the plate consists of different zones with different curvatures. Therefore, it is definitely not enough to consider just one phase. In the procedure described here, one whole simulation cycle consists of 2 phases:

- picking up the decal from the heating table
- printing the decal at the dinner plate.

It is worth noting that the friction coefficients in both phases are quite different. The table has a very low friction coefficient due to the melted wax whereas there is a very high friction coefficient in the transfer phase of the decal onto the plate as the decal sticks on the plate surface and a relative movement is only possible between the decal and the pad. Fig. 7 illustrates the finite element representation of the printing pad and both contact bodies, i.e. the plate and the table. Furthermore, the main direction of the compression movement is shown (Fig. 9).

Figs. 10–13 illustrate the multitude of possible results obtained from the finite element analysis. Each increment corresponds to a incremental displacement in x-direction of 0.05 mm.

5 Conclusion

A method based on finite elements to simulate the heat release transfer process of decals to plates with silicone pads offers a powerful tool which can be extended to analyse and develop pads. This method can also be applied to the technology of direct printing with different friction factors. Part II of this paper – to be published in Interceram 04/2003 (July) – shall present/discuss the analysis and design optimisation of printing pads.

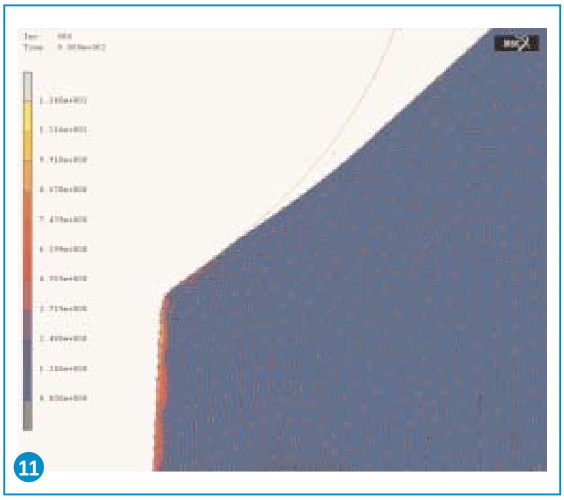


Fig. 11
Deformed mesh (pad) + contact body (plate): scalar plot of contact normal force

Fig. 12:
Deformed mesh (pad) + contact body (plate): Scalar plot of contact friction force

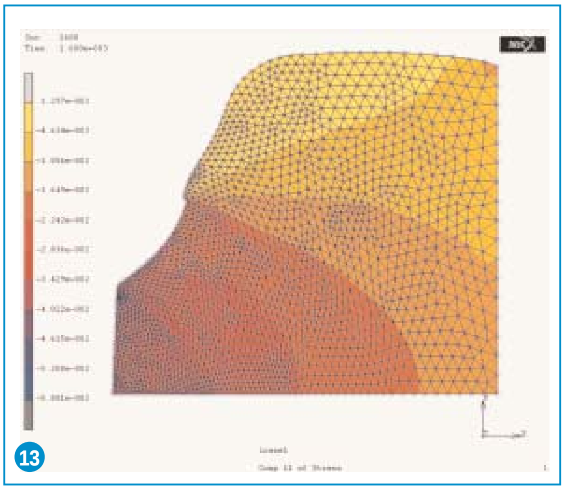


Fig. 13
Deformed mesh (pad) + contact body (plate): scalar plot of stress component s_{xx} at the end of the simulation

Dipl.-Ing. Dipl. Wirtsch.-Ing. Andreas Bachthaler
data M Software + Engineering GmbH
Am Marschallfeld 17, D-83626 Valley/Oberlaindern, Germany
e-mail: a.bachthaler@datam.de

Received: 27.03.2003

1/1 Eigenanzeige links

Kurzfassung/Résume/Résumen

Optimierung der Tampongeometrie für Heißübertragung und Direktdruck mit Hilfe eines Finite-Elemente-Programms

Im nachfolgenden Artikel wird beschrieben, wie mit Hilfe eines FEM-Programms und einer geeigneten mathematischen Modellierung das Verhalten eines Silikontampons beim Übertragen von Heißübertragungs-Dekoren auf Teller simuliert werden kann. Durch Änderung der Geometrie und wiederholte Simulation wird iterativ optimiert, bis schließlich eine geeignete Form gefunden ist. Dabei wird dargestellt, wie die Ergebnisse der Simulationsrechnung bewertet müssen, um das Verfahren zu optimieren.

Optimisation, par méthode d'éléments finis, du design d'un tampon, par rapport à l'émission thermique et à l'impression directe. Partie 1 : le modèle de simulation

L'article décrit comment on peut simuler, par emploi d'un logiciel de simulation numérique et modélisation mathématique appropriée, le comportement d'un tampon de silicium lors du processus d'émission thermique associé au transfert de décors sur les pièces. L'optimisation est obtenue en modifiant la géométrie et en répétant la simulation jusqu'à obtention de résultats satisfaisants. On montre également comment les résultats de la simulation doivent être évalués pour optimiser le processus.

Optimisation, par méthode d'éléments finis, du design d'un tampon, par rapport à l'émission thermique et à l'impression directe. Partie 1 : le modèle de simulation

L'article décrit comment on peut simuler, par emploi d'un logiciel de simulation numérique et modélisation mathématique appropriée, le comportement d'un tampon de silicium lors du processus d'émission thermique associé au transfert de décors sur les pièces. L'optimisation est obtenue en modifiant la géométrie et en répétant la simulation jusqu'à obtention de résultats satisfaisants. On montre également comment les résultats de la simulation doivent être évalués pour optimiser le processus.