



UNIVERSITY · OF
STRATHCLYDE

Department of Design, Manufacture
and Engineering Management



PROCESTEKNISK INSTITUT

TECHNICAL UNIVERSITY
OF DENMARK

INSTITUTE OF MANUFACTURING
ENGINEERING

DIE-CAVITY ELASTICITY CONSIDERATIONS FOR NETT-FORMING

SYNTHESIS REPORT

MAY 1995

Contract No.: BREU-392

BRITE/EURAM Project No.: BE-4129
MM. No.: 9460

DIE-CAVITY ELASTICITY CONSIDERATIONS FOR **NETT-FORMING**

C. Andersen, R. Balendra*, B. Hansen, X. Lu", A. Rosochowski", T. Wanheim, Y. Qin*

Abstract

The research explored the scope for the manufacturing of **nett-formed** components by designing the tools 'right first time'. The off-line procedure for designing such tools is based on the prediction of the elastic effects which occur at the **tool/component** interface under the prescribed load distribution. This distribution is provided by the **Physical Modelling** experiments. The scaled-up load serves as the boundary condition for the subsequent elastic Finite Element analysis of the **tools** and the component. Superimposing the FE results of tool deflection and component **springback** on the nominal form, enables the prediction of the real shape of the components. A **CAD-based geometric modeller** is used to derive the tool compensation required to negate the **tool/component** elastic effects. Industrial tests validated the proposed technique.

Introduction

Competition among various metal conversion technologies has prompted the evolution of cold metal forming into **nett-forming**. There are three aspects of **cold metal** forming which affect the accuracy of the component form; these are the plastic flow of material, the die/component elastic effects and the machine stiffness. The reported research refers to die deflection and component springback as the sources of inaccuracy **in** a component.

Dies are designed with materials which operate within the **elastic** range of their **behaviour**; the elasticity of the die-cavity is, therefore, likely to cause a variation of the component shape from that defined by the die-cavity. **Springback** of the component due to unloading adds further inaccuracy to the component.

The manufacture of die-cavities for **nett-forming** technologies invariably undergoes iterations which require the use of production machinery; this is necessary as currently available **design-support** is **unable** to define the form compensation required to negate the die/component elastic effects. Achieving the required accuracy through production trials is time consuming and expensive.

One possibility for overcoming this deficiency, in the traditional industrial approach, is to carry out a Finite Element (FE) simulation of the metal forming operation. This would include an elastic-plastic analysis of the material flow, coupled with an elastic analysis of the tools. Unfortunately, the method has some disadvantages which undermine its usefulness. The results depend on an appropriate constitutive model of the work-material and friction - both lack common acceptance. Further, one has **often** to deal with 3D components and non-stationary processes. In such cases the elastic-plastic FE analysis requires powerful computers and calculations are time-consuming.

Technical University of Denmark, Institute of Manufacturing Engineering,
Building 425, 2800 Lyngby, Denmark.

* University of Strathclyde, Department of Design, Manufacture and Engineering Management,
James Weir Building, 75 Montrose Street, Glasgow G 1 1XJ, Scotland, UK.

The proposed alternative for designing die-cavities for **nett-shape** forming is based on the FE analysis of **die/component** elastic effects. This approach is based on the distributed load at the die/component interface determined by **Physical Modelling** (PM) experiments. This enables the separate analysis of die deflection and component unloading - both elastic phenomena. Die compensation derived from such an analysis enables designing dies "right-f **first-time**".

This **fundamental** research into the feasibility of the above-mentioned method of designing die-cavities for **nett-shape** forming concentrated on three **areas**: **Physical Modelling**, development of software and validation trials. Chapter 1 deals with measurement of load distribution in **modelling** dies and refers to **modelling** considerations. Chapter 2 describes the methodology for FE simulation of die/component elastic effects and the software for geometric comparison to support the design of compensated tools. Chapter 3 presents the results of PM experiments, FE simulation and comparative measurements for a particular engineering component. The research demonstrates that the proposed method is an effective means of designing die-cavities for **nett-shape** forming.

1. Physical Modelling for the measurement of pressure contours

Physical Modelling (PM) is recognised as a **useful** method of material flow visualization in metal forming operations. PM, which uses **modelling** materials, lubricants and tools, has been successfully employed **in** designing preforms, die-cavity forms and the operation sequence. It **also** enables a significant insight to critical phenomena taking place in bulk metal forming processes.

The proposed method of designing die-cavities for **nett-shape** forming requires a new approach to the use of PM; it is used to provide information on the load distribution at the tool/component interface. Successful measurement of the force contour on the surface of model tools is thus a critical feature of the approach.

Modelling media have to be selected carefully to mimic the **behaviour** of real materials; this selection is more important in the current application of PM which has to provide load distribution data for use in subsequent analysis.

The following points describe the work done to establish a load distribution measuring system and to improve the simulation of real process conditions by selecting proper **modelling** media.

1.1 Measurement of load distribution

New transducer

Load distribution is to be defined by measuring the magnitude and attitude of forces at critical locations on the model die-cavity. This will enable the definition of **the** force contour by interpolation between these locations. To realise this task, the load distribution measuring system has to use many transducers capable of measuring forces in three directions. There have been previous attempts to measure the normal component of the force on the **modelling** dies. However, the current approach required a new transducer which could measure not only the normal force but also both components of the friction force.

The first consideration referred to the principle on which such a transducer could be based. Apart from measuring forces in three directions on the model tool surface, the main requirements were: high sensitivity, small dimensions and simple design. After evaluating various concepts it has been decided to design a transducer which uses the **piezoelectric** principle.

Piezoelectric properties occur naturally in quartz or can be induced in some ceramic materials and certain polymer films. Piezoelectric ceramics appeared to be best suited for the transducer. Piezoelectric ceramics are very versatile; their geometry and piezoelectric characteristic can be tailored to the specific application. In this application, high piezoelectric sensitivity was of prime importance.

The selected piezoelectric ceramic was used in the form of rectangular plates. After preliminary trials it was concluded that the “thickness shear effect” of piezoelectric plates may be used for measuring both, normal and tangential forces. Piezoelectric plates were glued together to form a measuring pair. Three such pairs were oriented parallel to the transducer axis and at 120° to each other. These were attached to the load carrying elements of the transducer. The normal component of the force was measured as the sum of the responses from the three measuring elements. The two tangential components were derived from the same responses by solving two equations of momentum equilibrium. The measuring elements are placed into the open type transducer housing. Fig. 1 shows the transducer design and the technique used to embed it into a model die.

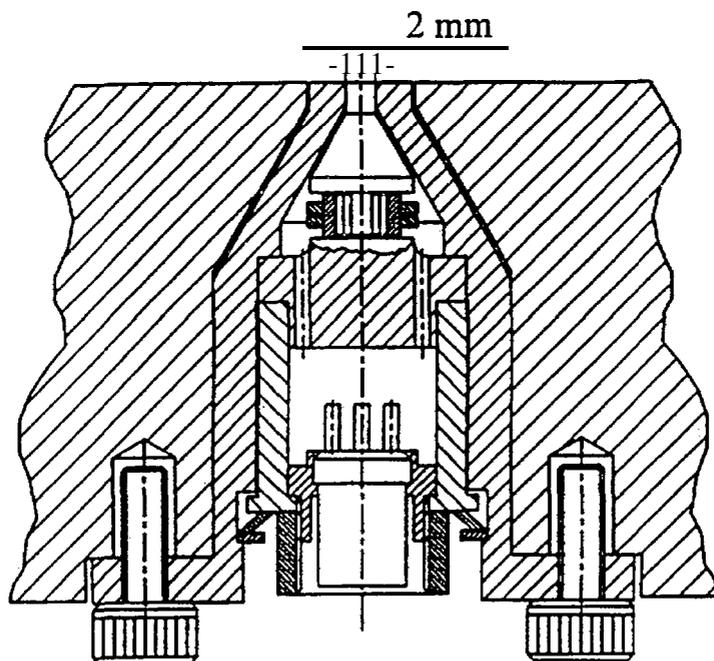


Fig. 1 Transducer design and embedding technique.

Three cables (not visible in Fig. 1) are connected, through the socket at the rear of the transducer, to three charge amplifiers for converting the charge generated into voltage. The

electronic circuit for charge **amplification** is well-known. Nevertheless, one has to be aware of potential problems associated **with quasistatic** measurements when using poorly designed and manufactured electronic circuit. To avoid excessive **drift** of the output signal, the quality of the electronic elements has to be **high**. High insulation resistance of the transducer and the cables is recommended. Proper **shielding** and **grounding** techniques are also important to the acquisition of meaningful results.

Calibration is the constituent part of building a transducer. The proposed transducer was calibrated by applying the known normal and tangential forces independently to the transducer pin. To establish tangential characteristic of the transducer the tangential force was applied every 20°. Fig.2 **illustrates** the three angular calibration curves found for the three measuring elements of the transducer.

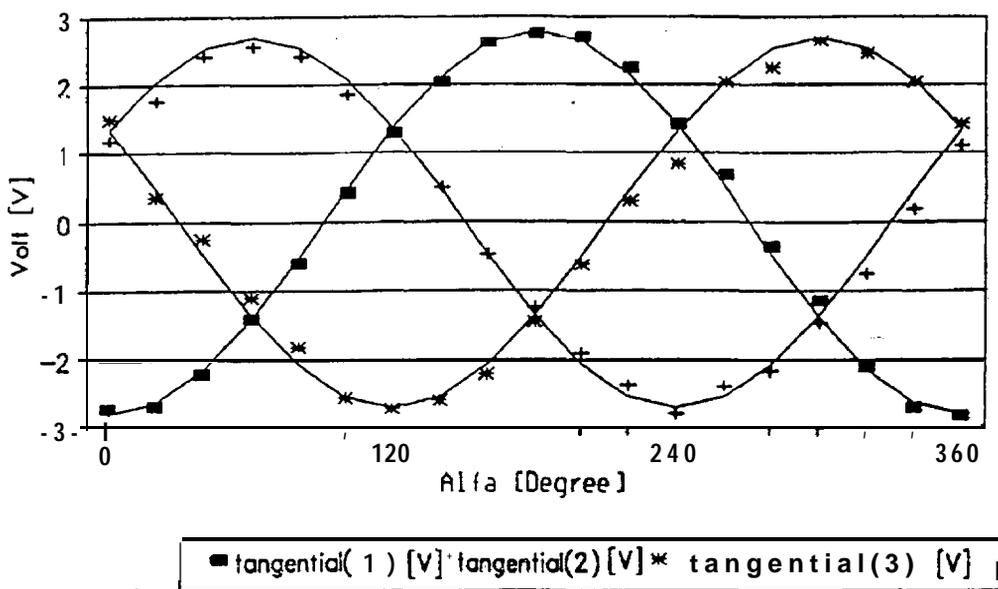


Fig.2 Angular calibration curves for measuring tangential force.

This design of a **triaxial** force transducer was proven to operate successfully. It fulfilled its intended **function** of measuring, accurately, small normal and tangential forces in the **modelling** die cavities. Several original elements were incorporated into the design which enabled the patenting of the transducer [1].

Data acquisition system

To enable definition of the force contour the number of transducer locations should be as large as possible. On the other hand, despite small dimensions of the transducer, even large model tools can accept only a finite number of transducers. The transducers were, therefore, located in the strategic positions on the tools. The envisaged minimum number of transducers is ten. Since each transducer measures three components of the force, the number of measuring channels would be, at least, thirty.

The analysis of data from a large number of channels required systematic approach; appropriate hardware and software were developed to enable this. It was decided to use the data acquisition system based on a plug-in PC data acquisition board with a sampling frequency 50 kHz and a resolution of 12 bits. Since the original number of channels was only 8, it was **necessary** to use a special expansion panel which increased the number of available channels to 128; the large number of channels reduced the effective sampling rate. **Despite** the fact that PM experiments can be regarded as slow processes, the sampling speed **should** be high enough to enable all the results to be referred to same instant in time. No **special** conditioning of the **analog** input signals was required since this **function** was assigned to the charge amplifiers which were coupled directly to transducers.

An indispensable part of the data acquisition system is appropriate software. Special software was required to deal with PM experiments with a large number of channels. This task was accomplished by developing software in C-code and using the library of procedures delivered with the board. One of the **functions** of the data acquisition software was to structure, format and scale the output data into a form which is directly usable in the subsequent FE analysis.

Another option, which was tested, was to use that data as attributes of the tool model in CAD system. Information stored there, could be then extracted to **simplify** and speed up the pre-processing part of the FE analysis.

1.2 **Modelling** media

Data acquired from the cluster of transducers used in PM experiments has to be scaled from the PM level to the engineering level of metal forming. This required knowledge of the characteristics of **modelling** materials and lubricants as well as the properties of conforming engineering materials and lubricants.

Modelling materials

To assess whether the **modelling** material replicates properties of a specific engineering material, the **Hollomon** expression, which represents mathematically the **uniaxial** yield stress of the material, is considered:

For engineering material
$$\sigma = C_e \epsilon^{n_e}$$

For modelling material
$$\sigma = C_m \epsilon^{n_m}$$

The parameters **C** and **n** for both materials have to be determined for the same strain-rate and the same temperature.

The **modelling** materials were compounded to achieve the equivalence of the exponents n_m and n_e , which means it is only the C-values which defines the difference between the two Hollomon expressions and thereby the two materials. Thus, the difference between the two strain hardening curves can be defined as a scalar relationship $K_s = C_e / C_m$. The value of K_s for each pair of **modelling** and engineering material, was used to scale PM results to the engineering level.

Modelling lubricants

The material flow and load in metal forming operations are affected by friction. Therefore, it was important that friction conditions in PM experiments are similar to those in corresponding engineering processes.

The friction conditions at the tool/specimen interface can be described by the general friction law:

$$\tau = f\alpha k$$

where: f is the friction factor

α is the ratio of contact area

k is the yield stress of the material in shear

The ring test was used to evaluate the friction factor f in the real processes and their modelling equivalents; the friction factor f of modelling lubricants can be varied between $0.07 < f < 1$, by mixing different proportions of Vaseline, Zincstearate and Kaolin. The modelling lubricants were compounded to replicate the engineering counterparts by ensuring $f_m = f_e$. The shear yield stress k is related to the yield stress σ in uniaxial tension/compression.

For the von Mises material this relationship is $k = \sigma / \sqrt{3}$.

To make sure that the recipe for the modelling lubricants was correct, a comparative study was performed using the double cup test. The engineering materials and lubricants, as well as their modelling equivalents, were subjected to such trials.

2. Software support for nett-forming

2.1 Methodology of FE simulation

Uniaxial commission analogy

To explain the basic concept let us consider a simple example of the uniaxial elastic-plastic compression of a cylindrical billet. Fig.3 shows, schematically, the component shape and the compression force history (OF). The nominal position of the tool face at the end of the forming cycle is (N). Knowing the force (F) and the elastic characteristic of the tool (NF), one can establish the actual position of the tool (L) due to its elastic deflection (NL). The point (L) also defines the position of the loaded component face. Applying the same force (F) to the loaded component, which has the elastic characteristic (UF), it is possible to find out the final component position (U) which results from its elastic unloading (LU).

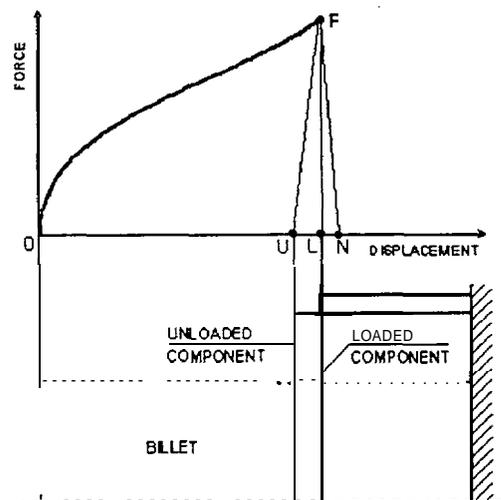


Fig.3 Establishing the final height of the component produced by uniaxial compression.

The distance between the nominal product position N and the final product position U is the basis for tool compensation.

General Procedure

The procedure described for **uniaxial** compression can be extended to any bulk forming process [2]. Fig.4 illustrates the general procedure in the form of a diagram.

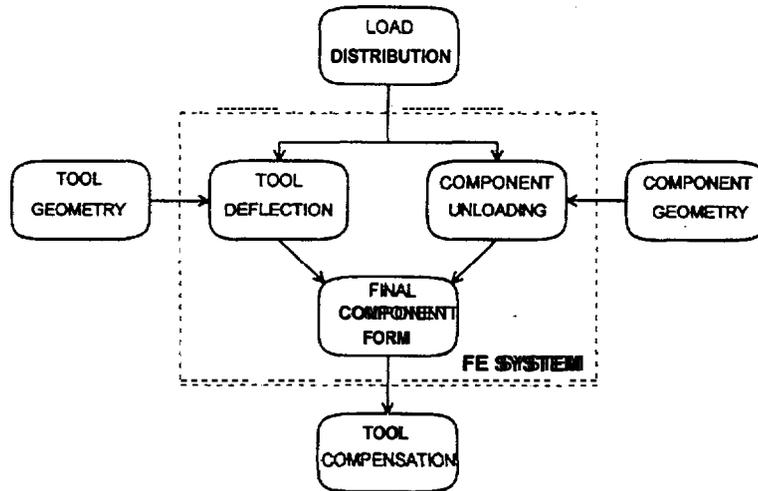


Fig.4 Tool compensation procedure.

The simulation of component unloading is carried out using the linear elastic FE analysis. The same type of analysis can be used for the simulation of tool deflection, although tools which are in contact and can move relative to each other, require non-linear elastic analysis. A commercial FE package, with well-developed pre- and post-processing facilities, was employed for this purpose. Functions of the pre-processing part of the FE package, which were customised by this application, referred to construction of the geometric model, meshing and establishing boundary conditions.

The most convenient way of dealing with geometric modelling is to import CAD data files (in IGES format) which describe the geometry of tools and the component (Fig.4). The tool and component forms imported were nominal descriptions with the tool forming surface conforming to the relevant part of the component shape. The geometry of the component for unloading calculations should be derived from the deflected tools (series system). The difference between the parallel system adopted here and the series system is, however, negligible, as is the tool deflection when compared with the component dimensions.

The main consideration with meshing refers to the die nodes and the component nodes which, for convenience, should coincide at the die/component interface.

Boundary conditions represent the displacement and traction (surface stress) fields imposed on the body. The displacement field results from the geometrical constraints (supports). The FE package used provided the capability of applying continuously varying traction using a function which interpolates between the discrete values of traction determined experimentally.

The pre-processing phase is followed by the FE calculations which were performed separately

for the tools and the component. The post-processing facilities enabled reviewing and interpretation of the results. Fig.5 shows, for example, the displacement results for the final stage of the forward extrusion process of a cylindrical rod. The die deflection and the component springback due to unloading were magnified 50 times to enable visualization.

To obtain the final component form one has to superimpose the displacements due to tool deflection and component springback. A proprietary programming language, available within the FE package, and the FE data base was used to accomplish this task. The final component form, reflecting both the tool deflection and the component springback, is exported to the external system for tool compensation. The tool compensation system is based on the CAD application software described in section 2.2.

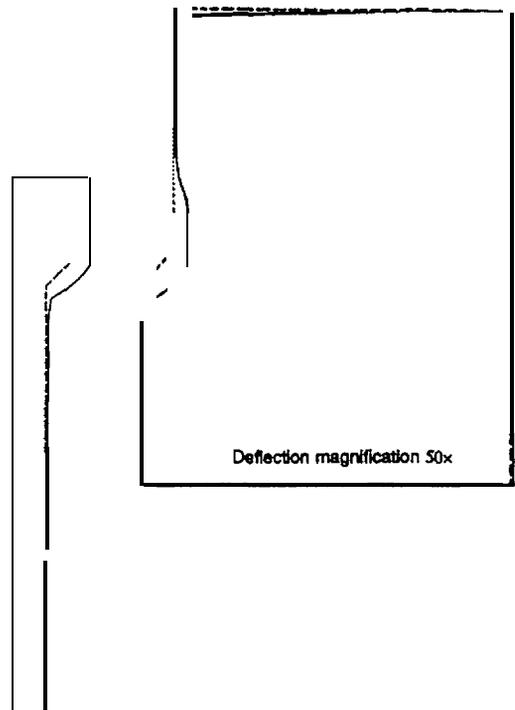


Fig.5 Die deflection and component unloading in forward extrusion.

Complementary remarks

It is possible to distinguish three types of surfaces, each of which contributes in a different manner to the component shape: a free flowing surface, a surface created by the instantaneous tool contact and a surface created at the final stage of the process. Since a free surface is not created by direct tool contact, it is not affected by tool deflection.

A component surface due to instantaneous tool contact is created in stationary processes which is characterised by consecutive portions of the material being subjected to plastic deformation. Thus, accuracy of the component form depends on the instantaneous and local tool deflection as well as the springback of the component material as it exits from the plastic deformation zone. In Fig.5, this type of surface corresponds to the surface of the extrudate. Accuracy of the rod diameter can be improved by compensating a single dimension of the tool which is the die orifice diameter. The so-called stationary process may be slightly dependent on time. This can be analyzed by gathering experimental data in the course of the modelling process and performing FE calculations for several stages of the process. Since compensation of the time dependent part of tool deflection is difficult, information about the die deflection history would be used for prescribing the dimensional tolerance of the component.

During forging, the tool/material contact increases continuously until the final stage of the process. Subsequently, the component form undergoes elastic unloading. The form of the head of the extruded component in Fig. 5 is an example of the surface created at the final stage of the process.

In some processes which are characterised by a high “punch pressure to yield stress” ratio (say

about 3) one can encounter secondary yielding subsequent to the elastic unloading; this results from the contracting die [3]. In this case, the compensation procedure should be based on the surface stress distribution measured after partial unloading (after punch retraction) rather than on that measured at the final loading point.

The last consideration refers to the heat generated during the cold forming operations. The work of plastic deformation is converted into heat; temperature increases in the component and, by convection, in the tools. This, in turn, leads to dimensional changes in all interacting parts and affects component accuracy.

Further research into secondary yielding and temperature effect will be carried out to refine the tool compensation technique for nett-shape forming,

2.2 Form conversion software

The FE analysis of die deflection and component springback was supported by a tool compensation software system. The system had to deal with creating, importing and exporting geometrical models. It processed geometrical data in order to manipulate the different models, compare and modify them. The main role of the system was to act as an interface between the FE package and a CAM package. The final component form imported from the FE system was to be compared with the nominal form to define the geometry of the compensated tools. Such geometry could be then exported to a CAM system to generate the NC codes for the production of metal forming tools.

The wide range of functions available enabled the system to be also used as a means of geometry comparison for the validation trials (see Chapter 3). This involved importing form data from a 3-axis CMM and comparing it with the FE results. Since the system was not solely for the tool compensation purpose, it was referred to as the form conversion software.

The ACIS solid modeller was chosen as a means for developing the form conversion software. ACIS is a new generation 3-dimensional modeller supporting the wide range of geometric

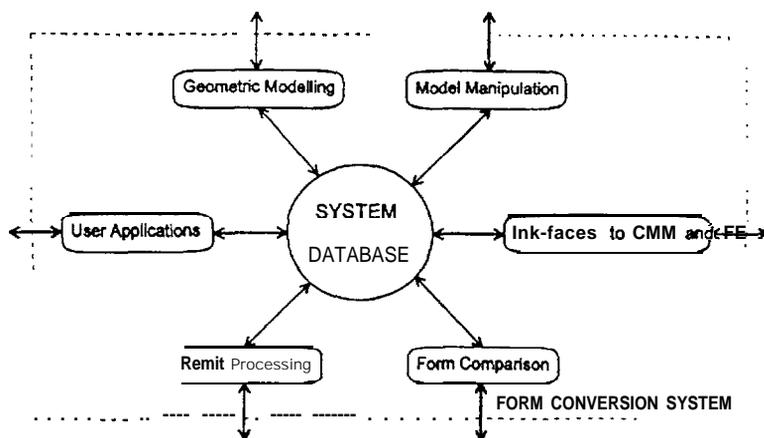


Fig.6 Modules of the form conversion software.

reasoning, **model** manipulation and flexible extraction of information. It is a set of CAD/CAM tools which can be implemented into a user application software.

The functional modules of the form conversion software are built around the system database (Fig.6) which contains the ACIS database and a user's database. Geometric **modelling** employs several methods of creating models. Model manipulation refers to manipulation of geometry as well as manipulation of attributes. Interfaces provide the means for importing **FE** and **CMM** results. The form comparison module contains the means to **classify** and recognise imported **data**, to perform the calculation of errors, error analysis and attribute attachment. The results processing **module** enables the visualization of errors and the generation of an output file.

Building **all** these modules has been based on the use of **C++** which conforms with **object-oriented** programming and object oriented data structure which are used in **ACIS**. A GUI was created using **X-windows** programming tools.

The software developed proved the feasibility of employing a geometric **modeller** in the form conversion application. The software can be developed **further** to cover new functions; it is considered to be an appropriate basis for the development of a commercial tool compensation system with the **FE** module as a core.

3. Validation trials

PM experiments and FE simulations were carried out for five components (Fig.7), provided by the industrial endusers, to validate the proposed approach for the compensation of

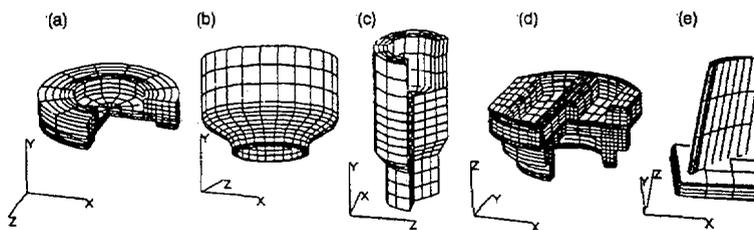


Fig.7 Geometric models of the components used in validation trials.

elasticity dependent errors. The component produced by combined forward-backward extrusion (Fig.7c) has been selected to illustrate the validation results.

Physical Modelling experiments

PM technique was used to measure the force contours on die-cavities of the components subjected to validation trials. The procedure for each component could be divided into the following stages: component selection, component analysis, design of experiment, tool manufacture, performing of experiments and analysis of results. Standard tables have been designed to assist the designer in this process.

The main criterion for selecting components for validation trials **was variety** of shapes these represented and benefits which could be achieved by reducing the product tolerances. The dimensional inaccuracy of a surface of the component was analyzed with a view to identifying causes; only those due to elastic tool deflection and component **springback** were taken into account.

Since the basis of the development was PM, it was necessary that the design of the **modelling** experiment satisfied the similarity conditions for the modelling material and lubricant. The location and quantity of three-axis pressure transducers were also decided on the basis of prior knowledge of process characteristics; an example of the modelling rig for the selected component is shown in Fig.8.

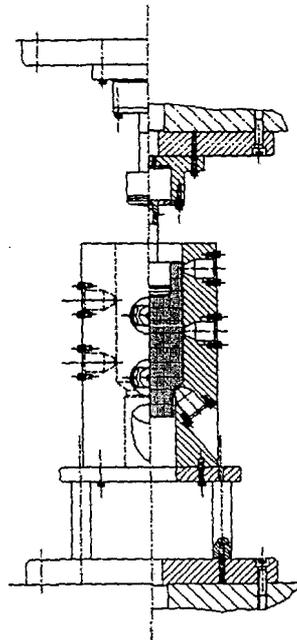


Fig.8 Modelling tools for forward-backward extrusion.

The tools were manufactured on a CNC machine, from CAD-codes established during the design stage. The PM experiments were performed according to the parameter selected during the design stage. A preliminary analysis of the PM experiments covered material flow, strain distribution, die filling and dead zones. When these were acceptable, the signals from the pressure transducers were extracted for analysis; the force contour on the die-cavity was then established and scaled to the engineering level of the process.

FE simulation

Geometric model of both the tool and the component were defined by representing their relative positions when the component was formed. The FE meshes were generated with reference to the purpose of the analysis and the requirements of the relevant boundary and load definitions. FE meshes of the selected 2D axisymmetric component and tool are shown in Fig.9.

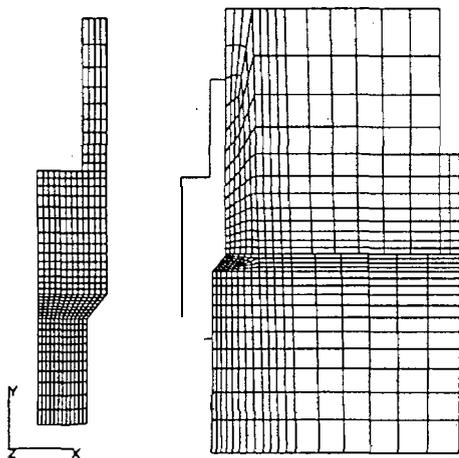


Fig.9 FE model of the component (left) and the tool (right)

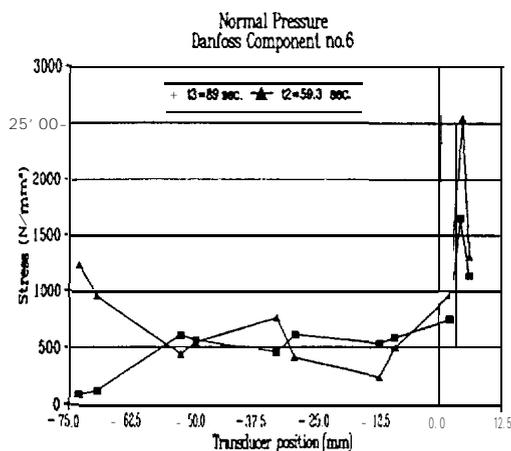


Fig. 10 Load pressure distribution obtained from physical modelling

Displacement boundary conditions are defined to represent both the axisymmetric characteristics and the die supports; the triangles in Fig.9 indicate the fixed support.

Load data obtained from the PM experiments were transformed to a surface stress distribution at the interface of the tool and the component. Fig. 10 shows the distribution of the normal pressure.

The materials of the analyzed bodies were characterized by the Young's modulus 210000 MPa and Poisson ratio 0.3.

Deflection of the tool and component under the defined load distribution are illustrated in Fig.10.

It was found that the maximum radial deflection of the tool was 0.088 mm, and the maximum deflection of the component was 0.041 mm. The maximum value obtained by superposition of these deflections was 0.129mm.

The various compensation values for tool diameters may not be suitable for the manufacture of the tool, therefore, a unified compensation value was used - the mean value of the superimposed deflections of the tool and component. In this case, the mean compensation value for the die diameter was 0.15mm.

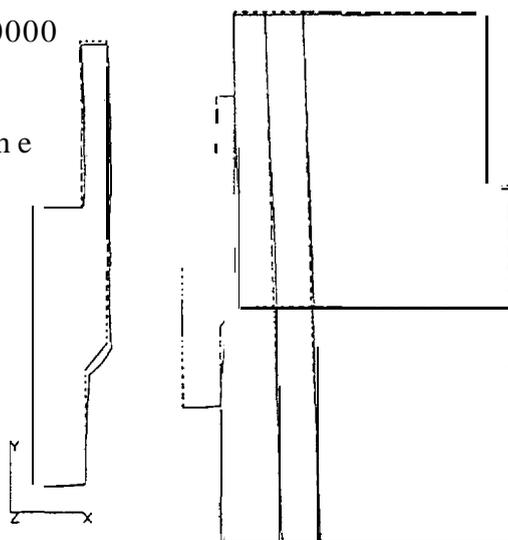


Fig. 11 Elastic deflection of component (left) and tool (right) with 30 times of deflection magnification

Another example of the application of the method has been presented in [5].

CMM measurements

Engineering products were produced and measured on a 3-axis CMM to verify the approach adopted to eliminate the errors in components due to elastic phenomena. A datum was defined on both the product and the tool to relate measurements to each other and to compare measurements with predictions.

Fig. 12 illustrates the comparison for the selected component (the nominal diameter 43.75); the correlation between predicted and measured product shape is good except the region adjacent to the cone part, where the pressure was high and secondary yielding could take place.

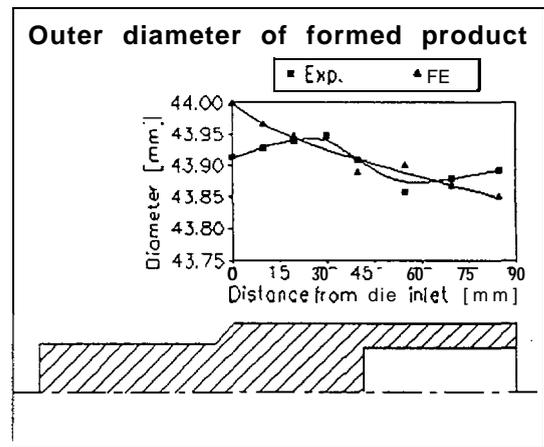


Fig. 12 Comparison of the predicted and measured product diameter.

Conclusions

The aim of this work was to propose, develop and test a new approach for the design of tools for net-forming. The procedure was intended to be entirely off-line to be able to eliminate industrial trials and produce tools “right first time”. The concept was based on two off-line activities - Physical Modelling experiments and Finite Element simulations.

The PM technique required the development of a new transducer which could measure the normal, as well as the friction force on the tool surface. The concept of new transducer was based on a set of piezoceramic plates which were subjected to shear. Surface stress data from a cluster of such transducers were acquired through a multichannel system for subsequent processing. This involved, scaling of the measured stresses up to the level expected from the real processes. Accuracy of this scaling depended on the selection of the modelling materials and lubricants.

As a result of this work a new application for PM was developed. The capability of measuring the distribution of all components of surface stress, in the course of the modelling process, enabled evaluation of the actual tool load. Information of this sort would prevent tool overload and reduce tool wear. For this programme of research the measurement of load distribution defined the boundary condition for FE calculations.

The methodology of FE calculations for tool compensation was established. It was based on two elastic analyses, one referring to the tool and the other referring to the component. The same load distribution, imported from PM, was applied to both bodies. After superimposing the results of tool deflection and component springback, one could obtain the resulting component form. The details of this methodology may differ depending on the type of the component surface concerned and the process involved.

Because of elasticity, the resulting component form does not conform to the nominal form. To compare both forms and derive a proper geometry of the tool, a special CAD-based

commercial die compensation system.

The predictive capability of the proposed system of tool compensation was subjected to industrial trials on a number of cold formed components. Physical Modelling experiments with these components provided the load distribution data employed in the FE analyses. The calculated forms were compared with the real forms measured on a 3-axis CMM. The obtained results confirmed feasibility of the proposed approach. It is possible, using PM and FE techniques, to predict the real component form and thus design tools which negate elastic effects. The economic gains, which result from shorter lead time, are clear.

Further development of this technique will concern secondary yielding due to the contracting die and temperature related geometry changes during cold metal forming. This fundamental research will be accompanied by building a user friendly tool compensation system in the form of a commercial software package.

Acknowledgements

The Technical University of Denmark and University of Strathclyde would like to thank Philips, Eindhoven (NL), Rolls Royce plc, Hillington (UK) and Danfoss a/s, Nordborg (DK) for endorsing this project.

The Technical University of Denmark and University of Strathclyde would like to acknowledge BRITE-EURAM for funding provided under project number DE 4129.

References

- [1] Pressure Transducer Invention by C.B. Andersen, R. Balendra, B. Post-Pedersen, A. Rosochowski, T. Wanheim.
- [2] A. Rosochowski, R. Balendra, "Compensation of die-surfaces for nett-shape forming", 10th International Conference on Computer Aided Production Engineering, Palermo, June 7-9, 1994.
- [3] A. Rosochowski, R. Balendra, "Effect of secondary yielding on nett-shape forming", Int. Journal of Materials Processing (to be published).
- [4] Y. Qin, R. Balendra, A. Rosochowski, "Evaluation of errors in nett-formed components by form comparison", Conference on Metal Science and Technology of Metals, Rouse, October 12-14, 1994.
- [5] X. Lu, R. Balendra, "Elastic deformation analysis of die/workpiece for die-cavity compensation", Int. Journal of Materials Processing (to be published).
- [6] C. B. Andersen, "Smedeværktøjers elastiske udbøjningers indflydelse på emnetolerancerne", Teknisk Nyt Special hefte no. 4 "Mekanik", Teknisk Nyt 1995.
- [7] N. Bay, T. Wanheim, M. Arentoft, C.B. Andersen, B. Bennani, "An Appraisal of Numerical and Physical Modelling for Prediction of Metal Forming Processes", (to be published spring 1995).
- [8] C. B. Andersen, "Elastisk deformation af værktøjerne ved nett-shape forrningivning", Inst. of Manuf. Engn., Tech. Univ. of Denmark, (to be published June 1995).