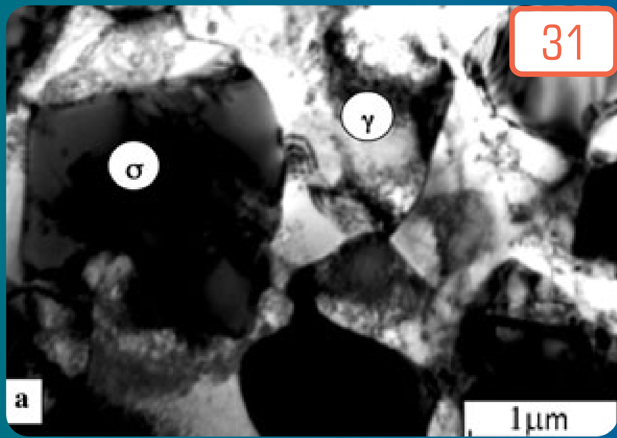
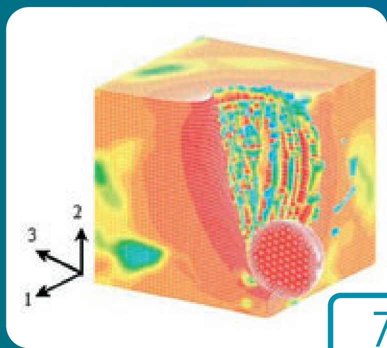
Contents

Selected materialographical photo



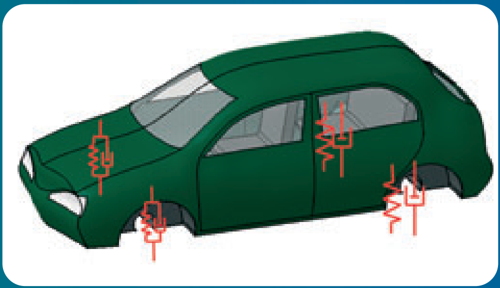
31

The Materials area is shown in the paper on "The impact of deformation on structural changes of the duplex steel" by G. Niewielski, K. Radwański and D. Kuc on a **page 31**. Despite of the many years' research on the plasticity of duplex steels, it was impossible to determine conclusively the mechanisms for structure recovery during the plastic deformation. The paper will attempt to provide explanations for the changes taking place in the steel structure during the superplastic flow. After a solution heat treatment at 1250°C, the steel was subjected to cold deformation through rolling with the total 70% reduction. The specimens were tensioned in the "Instron" strength-testing machine at temperature 850°C at a rate of $v_r = 15\times10^{-3}$ mm/s in a 0.005Pa vacuum. Structural examination was carried out using a light and electron microscopy. The microdiffraction technique was applied to provide diffraction images with Kikuchi lines. The results obtained are vital for designing an effective thermo-mechanical processing technology for the investigated steel. The capacity for increased deformability through combined thermo-mechanical processes, requiring a precise selection of the deformation parameters, was indicated.



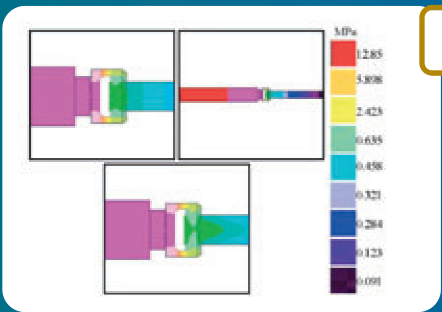
7

Authors: P. Gudimetla and P.K.D.V. Yarlagadda in the paper entitled "Finite element analysis of the interaction between an AWJ particle and a polycrystalline alumina ceramic" on a **page 7** inform that the finite element approach is a good tool to study the nature of interaction between a microscopic particle and a brittle material and accurately predict the erosion mechanisms in such interactions. A 3-dimensional FE model was set up using PATRAN. The alumina ceramic was modelled as a 1-mm cube while a 0.1mm diameter half sphere was used to model a single abrasive particle. The system was imported into ABAQUS and an explicit analysis was performed. The element deletion method was used after invoking a failure criterion to estimate the number of elements removed due to a single impact. The aggregate volume of eroded material was then calculated by multiplying the number of elements removed with the volume of each element. The results of the FEA were compared with the brittle model proposed by Kim & Zeng. The results of the FEA indicate that mixed-mode failure is the most common form of failure in such interactions. The volume of material removed per impact from the FE results is close to 16% of those predicted by Kim & Zeng's model. The finite element framework presented is idealised for the case of regular cubes based on a set of assumptions.



23

The paper from Analysis and Modeling area made by J. Świder, G. Wszołek, K. Foit, P. Michalski and S. Jendrysik on "Vibration analysis of mechanical systems with utilisation of GRAFSIM and CATGEN software" on a **page 23** describes the method and corresponding software used for modelling and vibration analysis of mechanical systems. As an example the study on the modelling of a vibrating system in a form of a passenger car was discussed with the determination of the characteristics describing its vibrations and resonance zones. The paper constitutes the source of information into the theory and practice of mechanical vibrations as well as professional computer software as a tool enabling the determination of the time responses and the a-f-p characteristics of machine and equipment parts. The software described in the paper uses the matrix hybrid graphs method and matrix block diagrams method for making analysis of mechanical systems vibrations. For numerical calculations the system is represented in the form of a matrix block diagram, and is analysed in a detailed way in the Matlab-SIMULINK environment. Numerical software packages give possibility to analyse vibrations of mechanical systems or of their parts exposed to kinematic and dynamic excitations.



67

The paper entitled "Modified design for the poppet in check calves" by S.W. Lee, D.Y. Shin, C. W. Byun, H.K. Yang and I.C. Paek on a **page 67** informs that a new design for the poppet was analysed and modified by CFD and FEM. So the modified poppet was verified through the real experiment and is available in a practical product. The check valve, which is composed of sleeve, connector and poppet, is the one direction valve that blocks fluid flow. The sleeve and connector are constrained and fixed. But the position of the poppet is swiftly moved by the direction of the fluid pressure. In this check valve, water hammer is applied to the poppet by rapid pressure change. Impact of the water is a reason why the fracture of the poppet is occurred. Using computational fluid dynamics (CFD) and finite element method (FEM), the design of the poppet was verified and modified to avoid the fracture. The diameter of the flow path in the poppet decreased from 6.0mm to 5.0mm. By CFD, differential pressure of the modified design was compared with differential pressure of the initial design. So, safety for the structure of the poppet was analysed and verified using available commercial software MSC.MARC. Based on the numerical results, differential pressure increased about 8.7%. However, Von Mises stress of the old poppet with 6.0mm was two times that of the new poppet. It is verified and disclosed from the experiment results that the newly modified poppet had no problem being used in a practical product. When the design is modified, the number and expense of the experiment is reduced.

Analysis and modelling

Manufacturing and processing

Cleaner production

Industrial management and organisation

Indexes