when scanning. The camera is located above the conveyor belt in a fixed unknown position. The laser light projector is installed on a five degrees of freedom robot arm facilitating the adjustment of the projector to the desired position and orientation.

The final accuracy of the reconstructed model is considerbly affected by several factors, namely; the accuracy of the intrinsic and extrinsic parameters of the camera, the light projection angle, and the uniformity of light distribution over the entire scene, both in terms of location and time. Camera calibration is performed to identify camera parameters. Twelve parameters were identified in total. The parameters found were verified through measuring previously known 2D objects. Novel simple mirror arrangements are employed to accurately measure the light projection angle.

A triangulation technique, along with a laser sheet of light with Gaussian distribution, is used to generate depth information. The intersection of the sheet of light and the object produces a 3D curve. Deviations of the curve from a striaght line represent the depth information, which can be extracted and translated into height variations using the triangulation technique. The CCD camera is used to capture both the texture and the depth carrying signal. The captured data are transferred, preprocessed, processed and converted to CAD data format. Curve approximation techniques are used to smooth the curve. Consequently, the CAD model is generated in a CAD environment. The model is then exported to an analysis environment for further analysis. To improve the accuracy of measurement, camera calibration, off-axis light source compensation, and light source parameter identification are performed. Algorithms to calibrate the camera, to scan the object, to smooth out the data, and to construct the 3D model were developed. To test the functionality of the system, and to evaluate the accuracy of the developed system, a controlled experiment is designed and performed. The system works well and a maximum error of less than 0.01% of full measurement scale is obtained.

Key Words: 3D scanning, triangulation, 3D model, model reconstruction.

A STREAMLINE ALGORITHM FOR SOLVING INCOMPRESSIBLE ONE AND TWO-PHASE FLOW PROBLEMS IN TWO DIMENSIONAL RESERVOIRS USING STRUCTURED GRIDS

S.A. Faroughi
faroughisalah@gmail.com
M. T. Manzari (corresponding author)
mtmanzari@sharif.edu
Dept. of Mechanical Engineering
Sharif University of Technology

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 125-134, Research Note

- Received 17 September 2011; received in revised form 13 February 2012; accepted 27 February 2012.

Abstract

In this work, a computational algorithm is given for solving the transport equation arising from flow in two-dimensional porous media, using the streamline method. The streamline method has four vital steps. First, the two-dimensional transport equation is decoupled into multiple 1-D equations. By using a new local variable, called the time-of-flight, these equations are solved in a way that is independent from the spatial discretization. In the second step, the streamlines are traced, based on the velocity field in the computational domain, to obtain the time-of-flight grid (TOF gird) along each streamline. Cell-by-cell streamline tracing is performed by utilizing the semi-analytical Pollock algorithm from the injector (sink) cell faces to the producer (source) cell faces or vice versa. In the third step, once all streamlines are traced, and their 1-D TOF grids are constructed, the multiple 1-D equations are solved, by either analytical or numerical methods. In the fourth step, the solutions along the TOF grid are mapped into the pressure grid (primary structured mesh) in order to construct the solution of the 2-D transport equation. This step is conducted using a volumetric averaging of the streamlines passing each cell.

The streamline method requires relatively lower computational time than other conventional methods, such as finite difference and finite volume methods, as it solves only several 1-D equations, and is without restriction on the time step size. In addition, this method requires less memory than other standard methods, as it needs to save and solve one of the multiple 1-D equations each time. Here, this method is employed for solving a number of 2-D test cases, including both single-phase and two-phase flow. The test cases demonstrates that the streamline method has good accuracy with a high speed-up factor. Numerical results show that using this method to simulate flow within a 2-D problem with 90000 grid cells leads to a speed-up factor of about 26. It is also shown that the speed-up factor of the streamline method will be enhanced by increasing the heterogeneity and number of grid cells.

Key Words: Streamline method, two phase, time of flight, Pollock algorithm.

USING THERMAL BUOYANCY OF COMBUSTION HEATERS IN ORDER TO CONTROL INDOOR AIR QUALITY

C. Noroozi
chamran.nouroozi@gmail.com
M. Marefat (corresponding author)
maerefat@modares.ac.ir
S.A.R. Zolfaghari
alireza.zolfaghari@yahoo.com
School of Mechanical Engineering
Tarbiat Modaress University

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 103-111, Research Note

- Received 23 November 2010; received in revised form 21 May 2011; accepted 4 July 2011.

Abstract

Combustion heaters can release a large amount of harmful gases, such as CO and NOx into the atmosphere of a room. But, by proper design and arrangment of the combustion equipment in the room, thermal buoyancy can be used to drive out the pollutants and improve indoor air quality. The main aim of this study is to propose a proper architectural design, and provide some practical design recommendations, by evaluating the effect of different parameters on the indoor air quality of buildings with combustion heating equipment. To achieve these goals, various arrangements for locations of heaters and outlet vents have been considered, and indoor air quality has been evaluated under these conditions by analyzing air flow and pollutant distribution due to vented and unvented heaters in the room. In order to solve the problem, a numerical simulation of the flow and pollutant fields in a 3D room was performed using Airpak software. Investigations illustrate that thermal buoyancy has different effects in various arrangements of heater and outlet vent. Changing the location of the outlet vent on the side walls cannot significantly affect indoor air quality, because natural convection currents by the heater are the dominant mechanism in flow field distribution. However, the results show that the center of a cold external wall is the best location for installing the outlet vent, so that both categories of air quality and energy combustion are considered. Moreover, the results of the present study demonstrate that the location of the heater in a room can significantly affect the distribution and also mean concentration value of the pollutants. By installing the heater near the external wall, opposite the outlet vent and in the corners of the room, 6% reduction in CO2 concentration in the breathing zone in the base model is observed. The position of the heater to the cold external wall, outlet vent, door and window are important factors that can affect the concentration of the pollutants. Generally, the lowest concentration of pollutants in the room is achieved in cases where the heater was located near the longitudinal external wall and near the corner of the room, and the outlet vent was located in another corner of that wall. Furthermore, the highest concentration of pollutants in the room was achieved in cases where the heater was located near the corner of the longitudinal internal wall, and the outlet vent was located in the corner of the transverse internal wall. Comparing indoor air quality in the mentioned conditions showed that under the best conditions, the concentration of CO2 in the breathing zone is about 9% lower than the worst conditions.

Key Words: Indoor air quality, thermal buoyancy, combustion equipment.

3D MODEL RECONSTRUCTION OF WEB PRODUCTION WITH IMAGE PROCESSING TECHNIQUE

K. Khalili (corresponding author)
khkhalili@yahoo.com
S.M. Emam
s_m_emam@yahoo.com
Dept. of Mechanical Engineering
The University of Birjand

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 113-124, Research Note

- Received 30 October 2010; received in revised form 6 July 2011; accepted 27 August 2011.

Abstract

One of the most commonly used techniques in obtaining three dimensional measurements is the triangulation technique using structured light, which is based on a simple mathematical algorithm. The current paper deals with 3D scanning and model reconstruction of mechanical parts being produced in a continuous manner. The objective of the current research is to firstly scan and reconstruct the part model and secondly to improve the accuracy of the model reconstruction. A simple test rig is designed and built to scan the object. The test rig is made of optoelectrical and mechnical parts. The optoelectrical parts include a digital CCD camera and a laser light projector. A conveyor belt with controllable constant speed is used to generate the movement required

The motion of the wing is assumed to consist of small perturbations, with respect to the constant-speed motion. Using the boundary element method in conjunction with first order shape functions, one obtains the system of discrete integral equations relating the potential to its normal derivatives on the surface of the body. However, there are some numerical problems in the evaluation of integral coefficients, due to singularity, which is caused by the intersection of the mach forecone and the edge of some elements in the computational domain. Of course, this problem can be eliminated by using some numerical techniques, such as the higher point Gaussin quadrature rule of integration and/or Teles transformation based on the order of singularity. Finally, a steady supersonic flow analysis is performed for two different wings with identified geometries (biconvex and planeform wings). The perturbed potential distribution for the biconvex wing are firstly computed and compared with available results in the literature. Also, its pressure coefficient distribution, specifically in the chordwise centerline, is evaluated. In addition, in the planeform wing, variation of the lift coefficient slope, accounting for three-dimensional effects, is studied. Also, the mach cone formed at both edges of the wing is visible and corresponds to the well-known mach cone formula for estimating mach angle. Comparison between the numerical results and those obtained using the exact solution indicates the good agreement and accuracy of the present method, and its flexibility to steady supersonic flows analysis around three-dimensional wings or complex configurations.

Key Words: Aerodynamic, wing, Boundary Element Method, potential flow, supersonic.

IDENTIFICATION OF THE KINEMATIC AND DYNAMIC PARAMETERS OF A 4-DOF PLANAR ROBOTIC MANIPULATOR

S.M. Sadeghi (corresponding author)
morteza@tabrizu.ac.ir
S.M.R. S. Noorani
smrs.noorani@tabrizu.ac.ir
A. Ghanbari
a-ghanbari@tabrizu.ac.ir
Dept. of Mechanical Engineering
University of Tabriz

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 93-101, Research Note

- Received 10 September 2011; received in revised form 28 December 2011; accepted 29 April 2012.

Abstract

Due to the complexity of governing equations on robotic manipulator motion, various methods of control have been proposed. Information about the physical parameters, which are assumed to be default, is the key point of having a successful model-based control implementation. In other words, accurate identification of model parameters is always required in the case of control tasks. Unfortunately most physical characteristics are not exactly known, due to uncertainties in manufacturing, such as tolerances, clearances; or inherent uncertainties, like friction, and so on. Thus, the main objective of this study focuses on finding the kinetic parameters of a manipulator in a system identification framework, in which the sought parameters are estimated via the least squares method. Here, system identification is indeed an instance of the robot calibration problem, by which the best estimation for physical parameters of the system is extracted for use in controlling the model. To this end, first, a four-DOF planar manipulator is considered as an instant case study, and analyzed to achieve a straightforward dynamic model. If the dynamic model of the system is available, the usual way for identifying the parameters is minimizing the error between the outputs measured from the robotic manipulator and those calculated from the mathematical model, which should be obtained for an identical set of inputs. One of the most traditional and still most common criteria for minimizing errors is the least squares method, which is utilizable for any system that is linear in terms of unknown parameters. On the other hand, in the literature, the routine dynamical model for controlling tasks is derived based on choosing relative angles as the generalized coordinates; then, Coriolis acceleration effects, including the product of joint velocities, arise in the equations of motion. Such a modeling will possess extreme parameters in its formulation, and so, identifying kinetic characteristics, will be involved. Thus, in this paper, by choosing the absolute angles of the links as the generalized coordinates, the Coriolis acceleration terms could be hidden in the structure of the equations of motion, and, in this way, our formulation will include minimum unknown parameter numbers. The input to this system is the torque applied to the manipulator joints, which makes it an MIMO system. By considering the cost function as the error between the estimated parameters and the true ones, the trace of the error covariance matrix is minimized, and, in this way, estimation is achieved. The simulation results assert the validity of this identification.

Key Words: Robotic manipulators, system identification, least square method, multi-bodies dynamics, inverse dynamics, calibratio.

H. Niazmand
hniazmand@yahoo.com
Dept. of Engineering
Ferdowsi University of Mashhad

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 77-85, Research Note

- Received 8 January 2011; received in revised form 25 May 2011; accepted 23 July 2011.

Abstract
Due to recent advances in microfabrication techniques, it is possible to produce microchannels with positive, negative, or even neutral surface charges. Investigations of electroosmotic flow indicate that such a combination of charge patterns on the microchannel walls results in complex flow fields with circulation zones that are highly desirable for fluid mixing requirements, as in lab-on-a-chip devices. However, the mixing efficiency of such flows is not well-examined. The objective of this article is finding a proper mixing indicator among those which have been frequently used for evaluation of mixing, to examine the effect of wall heterogeneities on mixing efficiency. In general, for the numerical study of electro-osmotic flows, it is necessary to solve the electric potential field and the equations for ion transport, together with Navier-Stokes equations, which are numerically expensive and time-consuming, especially when heterogeneous channels are modeled. However, under certain conditions, the Helmholtz-Smoluchowski model can be used, which efficiently reduces numerical expense. In this model, the flow field is obtained by solving the Navier-Stokes equations with slip boundary conditions at walls, which comes from the Helmholtz-Smoluchowski slip velocity. Upon solving for the flow field, the concentration field can be obtained and the mixing efficiency can be evaluated. In most studies, the standard deviation of concentrations at any cross-section is considered the mixing indicator. Our investigations of the complex flow patterns with circulation zones show that this indicator is not well-behaved near the vortex area, such that some fluctuations in the mixing values occur. In order to have a more physically relevant mixing indicator, a weighted standard deviation is used to quantify the mixing performance, which is more consistent with vortex flows. Based on this indicator, some essential concepts for mixing, such as mixing performance, and channel efficiency, are introduced and the effects of stepwise heterogeneities are studied. Different zeta potential distributions associated with the single patch were examined, and it was found that the mixing performance of a non-continuous potential distribution is relatively similar to that of continuous counterparts, despite the slight differences in the flow field. The results for a single patch indicate that the channel efficiency of a channel with a single patch linearly increases with the size of the patch, such that for a channel with a patch size of channel height, the mixing efficiency increases 4%, compared to its homogeneous channel. Also, the mixing performance is enhanced as the single patch is located closer to the channel inlet. The findings of the present study can be employed in the design of optimized micro-mixers.

Key Words: Heterogeneous microchannel, electro-osmotic flow, mixing performance.

PREDICTION OF AERODYNAMIC COEFFICIENTS OF A SUPERSONIC WING USING BOUNDARY ELEMENT METHOD

H. Shahverdi (corresponding author)
h_shahverdi@aut.ac.ir
Dept. of Aerospace Engineering
Amir Kabir University of Technology
M.H. Dolabi
hadidoolabi@yahoo.com
Dept. of Aerospace Engineering
Maleke Ashtar University of Technology
M. Behbahani-Nejad
bnmorteza@cua.ac.ir
Dept. of Engineering
Shahid Chamran University of Ahvaz

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 87-92, Research Note

- Received 11 April 2011; received in revised form 1 October 2011; accepted 16 October 2011.

Abstract
The main objective of this paper is to determine the aerodynamic coefficients of a steady supersonic wing using the Boundary Element Method (BEM) based on the solution of potential flow. In the field of computational aerodynamics and fluid dynamics, some numerical approaches, such as the boundary element method, based on finding unknown parameters only on the boundary of a body with lower computational cost than high-fidelity formulations, can be a good replacement for Computational Fluid Dynamics (CFD) methods. Hence, application of BEM based on potential flow has recently been implemented as an efficient and popular tools in many studies. An advantage of the BEM is that the method reduces problem dimensionality by one. In this study, the governing partial differential equation of a supersonic potential flow field is firstly derived and linearized.

expansion fans. The latter case occurs when there is a counter current multi-phase flow in porous media.

In order to avoid this kind of problem, entropy correction procedures should be used. These procedures usually generate additional numerical diffusion at sonic points to disperse expansion shocks.

In this work, a relatively new numerical method, called the dominant wave method, is used to simulate immiscible flow in porous media. The dominant wave method is represented in the form of central methods. This method is locally conservative and uses a finite volume numerical flux approximation. The dominant wave method eliminates the necessity of characteristic decomposition by detecting the dominant wave speed of the system. On the other hand, this method generates less numerical diffusion in comparison with conventional central methods. Here, slope limiters are used to obtain a higher order of accuracy. Fromm's and minmod limiters are used in this paper, and a comparison between these two limiters is provided. To avoid expansion shocks at sonic points, Harten's entropy correction is used.

In this work, gravitational effects are studied in a homogenous, one-dimensional reservoir using the Buckley-Leverett model. Three different models for relative permeability functions are considered in this study, including Corey, Extended Corey and Stone type models.

Key Words: Gravitational effects, Buckley-Leverett equations, dominant wave method, entropy correction.

EXPERIMENTAL INVESTIGATION OF SWEEP ANGLE EFFECT ON THE FLOW FIELD OF A WING AT SUBSONIC REGIME

M.R. Soltani (corresponding author)
msoltani@sharif.edu
Dept. of Aerospace Engineering
Sharif University of Technology
M. Masdari
m.masdari@ut.ac.ir
Dept. of New Science and Technology
University of Tehran
K. Ghorbanian
ghorbanian@sharif.edu
Dept. of Aerospace Engineering
Sharif University of Technology

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 69-76, Original Article

- Received 8 May 2010; received in revised form 3 May 2011; accepted 3 October 2011.

Abstract

A series of wind tunnel tests were conducted to examine the flow field over swept wings at subsonic speed under various conditions. For this purpose, three models with sweep angles of 23, 33 and 40 degrees were used. To Increase the Reynolds number, all models were employed as semi span wing types and the effects eliminated by installing the models on a flat plate at their roots. The flat plate was installed in the middle of the test section at a distance from the wall. The tests were performed at various Reynolds numbers and at various angles of attack on all semi-span wings, which had the same aspect ratio but different sweep angles.

The results show that increasing the sweep angle causes a decrease in the velocity over the wing surface. The pressure distribution on the upper surface of the wing shows some differences when compared to the 2-D case, due to the existence of the cross flow and wing tip vortices. As the wing sweep angle was varied, the differences varied too. The results indicate that the wing tip vortices amplify the 3D effects of the flow. The pressure reduction in the vicinity of the wing tip deteriorates the performance at that section, compared to other sections of the wing. This phenomenon would eventually cause stall to occur at this section (wing tip) sooner than at other sections. The maximum pressure drop near the wing tip remains unchanged with changing the sweep angle for zero angle of attack only. But, for other angles of attack, the situation is different. For the wing with a sweep angle of 40 degrees, the surface pressure of all sections differs significantly and the maximum pressure is much lower too. However, near the wing tip, the differences are not significant. This phenomenon is related to the effect of wing tip vortices that are more dominant at higher sweep angles.

The test results in all three models show that the pressure coefficient decreases with increasing Reynolds, and this phenomenon is most prominent for the highest sweep angle of 40 degrees.

Key Words: Swept wing, sweep angle, flow field, pressure distribution.

INVESTIGATION OF ELECTROKINETIC MIXING PERFORMANCE OF HETEROGENEOUS MICROCHANNELS

J. Jamaati (corresponding author)
jafar_jamaati@yahoo.com

THREE DIMENSIONAL FINITE ELEMENT SIMULATION

A. Khatouni
a.khatouni@ut.ac.ir
K. Abrinia (corresponding author)
cabrinia@ut.ac.ir
School of Mechanical Engineering
College of Engineering
University of Tehran

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 53-59, Original Article

• Received 27 June 2010; received in revised form 26 June 2011; accepted 20 November 2011.

Abstract
Manufacturing of engineering components with precise dimensions are becoming quite important in attempts to reduce cost and improve reliability. In forging die design, dimensional accuracy is one of the main goals. The load carrying capacity and life of any forged product is greatly affected by its dimensional accuracy. To predict the precise dimension of the part and determine the die dimension for precision forging, it is necessary to analyze the factors which affect dimensional accuracy. For manufacturing a precision forging of turbine blades, thorough recognition of prevalent parameters is essential, such as preform design, process simulation and various techniques for compensation of errors due to die-elasticity characteristics. The accuracy of aerodynamic cross-sections depends on many factors such as die elasticity, heat distortion and deformation of parts during the cooling of the die. The main goal of the design of aerodynamic section for the blade and its precision forging is to minimize the machining process. By using finite element simulation; material flow analysis during the forging process, forging force in any moment, contact pressure distribution between die and workpiece, and also elastic deflection of forging dies, can be easily predicted. Therefore, for the compensation of die-elasticity, die shape should be changed through size modification. Most previous studies about blade forging simulation have been performed in 2D finite element, and with this simplification, errors due to die-elasticity characteristics have been compensated. Clearly the forging of a blade is a 3D process, and the blade does not have the same cross section along its length and hence the forging of the blade could not be considered a plane-strain process. In this research, commercially available software, DEFORM3D, was used for the purpose of finite element method simulation for the turbine blade forging. Using three dimensional finite element simulation, elasticity deflection of dies in different sections of a blade were obtained, and then the upper and lower die profiles of those sections in various directions were modified and errors due to die-elasticity compensated. Through these modifications, errors due to die elasticity were reduced to values less than those obtained in previous works. A series of multiple simulations for having an optimized compensation of the die-elasticity were carried out. Errors due to elastic deflection were also compensated by modifying the die position, based on elastic deflection magnitude.

Key Words: Gas turbine blade forging, die elastic deflection, three dimensional FEM simulations.

SIMULATION OF IMMISCIBLE FLOW IN OIL RESERVOIRS INCLUDING GRAVITATIONAL EFFECTS

M. Moshiri
moshiri@mech.sharif.edu
M.T. Manzari (corresponding author)
mtmanzari@sharif.edu
S.K. Hannani
hannani@sharif.edu
Dept. of Mechanical Engineering
Sharif University of Technology

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 61-68, Original Article

• Received 8 December 2010; received in revised form 30 May 2011; accepted 23 July 2011.

Abstract
Mass conservation equations of three-phase incompressible, immiscible flow in porous media, called Buckley-Leverett (BL) equations, are of great importance in the study of flow in hydrocarbon reservoirs. For two-phase flow, these equations reduce to a scalar BL equation, in which the flux function is non-convex, as its convexity changes with respect to the conserved variable. Similarly, in three-phase flow, the non-convexity of flux functions leads to a linearly degenerate system of equations. In both non-convex and degenerate problems, it is necessary that the so-called Oleinik entropy condition is satisfied, in order to have physically admissible answers. Moreover, when gravity is included in the BL equations, the non-convexity of the flux function increases and some of the wave speeds of the system of equations become zero. Such a situation refers to a sonic point in gas dynamics. Many numerical methods have difficulty dealing with these points, especially when sonic points occur on an expansion wave. In such cases, some numerical methods produce non-physical expansion shocks instead of

tween surface roughness and initial hardness. An increase in initial surface hardness of the material led to a decrease in surface roughness, but this relation in super hard materials was different. Surface roughness decreased after a critical initial hardness. In fact, surface roughness rose by increasing initial hardness, and, after that, there was a drastic reduction. Meanwhile, surface roughness was investigated for fine, median and coarse grain materials. The amount of surface roughness in fine grain materials was more than coarse grain materials. The surface roughness was improved by an increase in the grain size of microstructures. Another factor, which is focused upon in this paper, is an investigation of the relationship between the direction of the grinding operation and the orientation of grains in rolled materials. It was found that grinding in the same direction as elongated grains resulted in improved surface roughness, and grinding perpendicular to the elongated grains led to a decrease in surface roughness. As a result, it can be achievable to acquire a desired surface roughness by identifying microstructure type, initial hardness and grain size of materials correctly. Furthermore, it is important to note the grain direction of rolled material during the grinding process. Finally, one must be careful to observe that achieving ideal surface roughness does not result in a serious change in microstructure.

Key Words: Grinding, surface roughness, hardness, microstructure, grain size.

SIMULATION OF LARGE DEFORMATION PROCESSES USING ALE FINITE ELEMENT METHOD

A.R. Shahani (corresponding author)
shahani@kntu.ac.ir
M. Barati
mahmudbarati@gmail.com
Dept. of Mechanical Engineering
K.N. Toosi University of Technology

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 41-52, Original Article

• Received 12 September 2010; received in revised form 7 August 2011; accepted 13 September 2011.

Abstract
In the present study, the local deformation of a cylindrical bar under uniaxial extension, and, also, the plane strain deformation of shell with two flat and reciprocal punches, were simulated via the decoupled ALE (Arbitrary Lagrangian Eulerian) finite element formulation in ABAQUS/Explicit code. In the ALE formulation, a reference computational domain is introduced and the finite element mesh is neither attached to the material nor fixed in space, so that the motion of the mesh is independent of that of the material. So, the mesh is managed by a procedure that tries to minimize mesh distortions and/or concentrate the mesh in a particular region. As a result, the ALE formulation can handle path-dependent material behavior and free surface conditions, while maintaining mesh fineness. An operator split procedure, a common and efficient choice in ALE analysis, is used to treat the independent mesh-material motion. Each time increment is divided into a Lagrangian phase considering material effects, which is done with the explicit time integration formulation, followed by an Eulerian phase, where the mesh smoothing is occurred, subsequently, taking into account the convective effects by solving the convection equation. The second-order Van Leer algorithm is employed to solve the convection equation. The mesh motion strategy is performed according to the mesh spatial constraints and volume smoothing method compatible with the problem type.
For verifying the ALE finite element model, the local deformation of a cylindrical bar under uniaxial extension was simulated. The results were compared with reported ALE results in literature and good agreement was observed. In addition, the merits of the method for large deformation problems were discussed and the results were compared with those predicted by purely Lagrangian formulations. Although both results were almost the same, a remarkable decrease in simulation time was achieved using the ALE method.
Frictionless contact is postulated in the analysis of the plane strain deformation of shell with two flat and reciprocal punches. Firstly, the problem was solved without considering strain rate effects. However, strain rate effects and dynamic effects due to high punch velocity were also investigated for a strain rate dependent material. It has been demonstrated that the inertia effects increase when the punch velocity becomes larger, resulting in different final shapes of the work-piece from those with low punch velocity. From the analyzed plane strain problem using an uncoupled ALE approach, the predicted punch load values and other comparison parameters are compatible with the coupled ones mentioned earlier in the literature.

Key Words: FEM, ALE formulation, necking, punch indentation.

ERROR COMPENSATION DUE TO DIE-ELASTIC DEFLECTION IN GAS TURBINE BLADE FORGING, USING

Abstract

The present paper deals with transverse elastic wave propagation using spectral element method (SEM), based on the first order shear deformation plate theory. In this method, the element nodes are located at the zeros of an orthogonal function, such as the Legendre function, which causes the mass matrix to be diagonal. On the other hand, the differential method is implemented for solving the problem of limited intermediate. Dealing with a diagonal mass matrix is the main advantage of the present approach, because the analysis only requires the inverse of a diagonal matrix. Therefore, the analysis time is much less than the one with the conventional Finite Element Method. In this study, a 10 layered square composite plate with 1 m side and 10 mm thickness is investigated. A time-variable force is applied at the centre of the plate during 120 msec, with 25 kH frequency. It is assumed that the fibers are aligned parallel to the side of the plate at all layers. Elements with 9, 25 and 36 nodes are used for simulations. In each study case, the total number of elements of the plate is selected in such a way by increasing the number of elements and taking into account a 2% difference for two consecutive steps. Therefore, the plate is divided into 200×200 elements (40000 elements) with 9 node elements. The selection of 100×100 and 60×60 elements for 25 and 36-noded elements, respectively, result only in 2 percent reduction in convergence time. Having solved the problem, three-dimensional deflection and two-dimensional deflection contour on paths passing through the center of the plate and parallel to the sides have been obtained. Then, using the two-dimensional deflection results, group wave velocity values ??in the longitudinal (fiber direction) and transverse directions (perpendicular to fibers) of the plate have been obtained. Creating the element stiffness and mass matrices, and assembling these matrices, makes the whole process analysis time. Therefore, for comparison between the elements with 9, 25 and 36 nodes, the time required for matrix assembly are registered. Besides, using theoretical formulations, the value of the group velocity for the desired frequency in this direction has been obtained and compared with those obtained via finite element analysis. A three-dimensional deflection contour has shown that the wave front is elliptical and a large diameter of oval is located in the fiber direction. In addition, the value of the maximum deflection of the plates has been decreased with increasing time. As shown by our findings, by increasing the number of element nodes, the time required for assembling the stiffness and mass matrices can be accelerated significantly. In addition, generally, by increasing the number of nodes per element, the differences between these results with those obtained from the analytical method are higher, and only in some cases lead to better correlations. Consequently, slightly better correlations incur very high computational costs. It is noted that elements with fewer numbers of nodes have lower degree polynomials, and may not establish the continuity of stress and strain. For the 9-node element used, for example, with a function of order two, the curvature function is obtained by double differentiation. Since the curvature is constant, the adjacent elements cannot have the same curvature. However, this must be the same for all elements. Therefore, from one element to the adjacent element, there will be a discontinuity in the stress and strain distribution. However, this is not the case for higher-order elements. In conclusion, if only the total amount of computational time and volume is required, a 9-node element is preferred. On the other hand, if the continuity condition of stress and strain distribution is preferred, then the advantage is in using higher-order elements.

Key Words: Wave propagation, composite plate, spectral element method (SEM), simulation accuracy.

A STUDY OF THE ROLE OF MICROSTRUCTURE AND GRAIN SIZE IN SURFACE ROUGHNESS OF AISI1045 WITHIN GRINDING

H.R. Fazli (corresponding author)
hamidrezafazli65@gmail.com
A.A. Akbari
akbari@um.ac.ir
Dept. of Mechanical Engineering
Ferdowsi University of Mashhad

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 31-40, Original Article

- Received 8 October 2011; received in revised form 26 February 2012; accepted 4 June 2012.

Abstract

This work outlines the effects of microstructure and grain size on surface roughness during the grinding process of carbon steel AISI 1045. According to this study, it is evinced that the obtained surface roughness after the grinding process is concerned with the microstructure and hardness of the initial surface. Different samples with different microstructures and three various levels of grain size (fine, median and coarse) have been prepared through specific heat treatment. After finishing the grinding operation, the surface roughness (Ra and Rz) was measured. A considerable relationship was found between the surface roughness and microstructures of samples. Furthermore, there was a relation belation, as introduced for inviscid flows. The second is essentially a new technique, which has been introduced by the authors for first time. In this method, some additional calibrations (including a kind of re-projection) are needed to achieve more accurate estimations. The outcome of these low order procedures can predict variations of flow field, due to variations of some effective parameters (e.g., Mach number or angle of attack) with high accuracy and high speed of computation. For the inverse design, we used a combined model, based on POD with the solution of an optimization problem, to obtain the desired surface pressure distribution or to minimize drag coefficient. In this way, two sets of snapshots have been used; an ensemble of flow fields, which may be pressure distribution or any aerodynamic characteristics, and a geometrical ensemble. The low order POD technique is used for computations of relative POD modes and modal coefficients of these ensembles. To solve the coupled optimization problem, an iterative approach has been used to achieve the desired accuracy. Note that the present inverse design approach can be reconstructed for different forms of the objective function. An order reduction manner has been used to find the optimal number of modes for data reconstruction that is a higher resolution of reconstruction which fewer modes for data reconstruction. The obtained results are compared to CFD simulations or exact solution results as reference data. The results show the relatively good accuracy and also the simplicity of the procedure.

Key Words: Proper orthogonal decomposition, Compressible Aerodynamics, Inverse Design, Order Reduction Manner.

ANALYTICAL INVESTIGATION ON THE STIFFNESS OF 3-DIMENSIONAL BRAIDED COMPOSITES BY A MULTI-UNIT CELL MODEL

M.M. Shokrieh (corresponding author)
shokrieh@iust.ac.ir
M.S. Mazloomi
sadeqmazloomi@mecheng.iust.ac.ir
Dept. of Mechanical Engineering
Iran University of Science and Technology

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 11-21, Original Article

- Received 26 October 2010; received in revised form 8 May 2011; accepted 1 June 2011.

Abstract
The present paper represents a new analytical method for calculation of the stiffness of three-dimensional, four-directional braided composites. In most previous work, the analytical model has been largely neglected in favor of the finite element model. Among those who have used the analytical model, the braided preform has been considered to be made of one, or three, types of representative unit cells, while microscopic evidence of the microstructure of pre-forms reveals different configurations of the yarns in the interior, surface and corner regions of a braided pre-form. This paper presents a multi-unit cell model, in which four kinds of unit cell, namely; interior, interior surface, exterior surface and corner unit, have been introduced as representative cells. Each type of unit cell in a braided composite has different microstructural parameters and possesses unique mechanical properties. In this model, each type of unit cell has been considered a uni-directional composite. Next, using rotation matrices, the angle between yarns and longitudinal direction has been considered in general coordinates of the model. Finally, using a volume averaging method, the total stiffness of the braided composite is calculated. The results are compared with those obtained from experimental methods and the effect of braiding angle on the stiffness of braided composites is discussed.

Key Words: 3-Dimensional four-directional braided composites, stiffness calculation, analytical method, multi unit cell model.

EFFECT OF GRID RESOLUTION ON ACCURACY OF TRANSVERSAL ELASTIC WAVES IN A COMPOSITE PLATE USING SPECTRAL ELEMENT METHOD (SEM)

H. Akharatdoost (corresponding author)
hamzeh.a.d@gmail.com
M. Safdari-Shadloo
safdari@mech.sharif.edu
M. Salehi
msalehi@aut.ac.ir
Dept. of Mechanical Engineering
Amirkabir University of Technology

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 23-30, Original Article

- Received 21 December 2010; received in revised form 22 May 2011; accepted 2 August 2011.

Abstracts of Papers in English

FAST DATA PROCESSING AND INVERSE AERODYNAMIC DESIGN, USING LOW-DIMENSIONAL POD METHOD

M.K. Moayyedi (corresponding author)
moayyedi@ae.sharif.edu
Dept. of Mechanical Engineering
University of Qom
M. Taeibi-Rahni
taeibi@sharif.edu
Dept. of Aerospace Engineering
Sharif University of Technology
F. Sabetghadam
fsabet@sharif.ac.ir
Dept. of Mechanical and Aerospace Engineering
Science & Research Branch
Islamic Azad University

Sharif Mechanical Engineering Journal
Volume 29, Issue 1, Page 3-10, Original Article

- Received 31 May 2010; received in revised form 18 May 2011; accepted 1 June 2011.

Abstract
Lately, scientific and engineering computations have been receiving detailed attention, in consequence of the improvement in computer hardware. Faster processors, efficient memory modules, and parallel computation techniques have prepared an adequate base for simulation of complex physical phenomena. However, the challenges in performing a simulation of an engineering problem and analyzing the attainable data increase manifold with the level of difficulty in the governing equations. Despite this fact, attempts to challenge these difficulties are inadequate. Numerical simulations yield large sets of data, which may not be useful as an experimental assignment. These data sets not only provide good and deep intuition into the physics of the problem, but also simultaneously assist the assessment and enhancement of various models that are developed for the computations.
In this article, Proper Orthogonal Decomposition (POD) has been used for fast data estimation of flow the field, calculation of aerodynamical coefficients, and inverse aerodynamic design. In this way, two methods have been used for estimation of inviscid compressible flow fields in various regimes, in which both methods are based on combined form of POD. The first extension is a coupling between the POD method and a cubic spline interpo-