GESELLSCHAFT für
ANGEWANDTE MATHEMATIK und MECHANIK e.V.
INTERNATIONAL ASSOCIATION of APPLIED MATHEMATICS and MECHANICS

86th Annual Meeting
of the International Association
of Applied Mathematics and Mechanics

March 23-27, 2015

Lecce, Italy

Book of Abstracts - Extract
2015

UNIVERSITÀ DEL SALENTO

jahrestagung.gamm-ev.de

Scientific Program - Timetable

Time	Sunday 22	Monday 23	Tuesday 24	Wednesday 25	Thursday 26	Friday 27
9:00			Contributed sessions (15 in parallel)	Plenary Lecture Moritz Diehl	Contributed sessions (15 in parallel)	Contributed sessions (14 in parallel)
10:00		Registration		von Mises prize lecture		
11:00			Coffee Break	Coffee Break	Coffee Break	Coffee Break
11:30			Plenary Lecture Thomas Böhlke	General Assembly	Plenary Lecture Ferdinando Auricchio	Contributed sessions (11 in parallel)
12:30			Lunch	Lunch	Lunch	
13:00		Opening				
13:30		Univ. Chorus Performance				Closing
14:00		Prandtl Lecture Keith Moffatt	Plenary Lecture Enrique Zuazua	Contributed sessions (15 in parallel)	Plenary Lecture Daniel Kressner	
15:00		Plenary Lecture Giovanni Galdi	Plenary Lecture Nikolaus Adams		Plenary Lecture Stanislaw Stupkiewicz	
16:00	Registration pre-opening	Coffee Break	Coffee Break Poster session	Coffee Break	Coffee Break Poster session	
16:30		Minisymposia & Young Reseachers' Minisymposia (10 in parallel)	Contributed sessions (14 in parallel)	Contributed sessions (15 in parallel)	Contributed sessions (15 in parallel)	
18:30			Public lecture Francesco D'Andria			
19:00		Opening reception at Castle of Charles V				
20:30				Conference dinner at Hotel Tiziano		
21:00						

GAMM 2015

Università del Salento

Table of contents

S04: Structural mechanics

The section will focus on advanced theoretical, numerical and experimental models for the evaluation of the behavior of structures. The diffusion of innovative materials characterized by high strength, anisotropy and unconventional mechanical responses (metamaterials) pose new challenges to the design and the performance of various structural elements like beams, plates and shells. In particular, structural issues may appear at different scales when materials with an internal architecture are employed. Particularly welcome are models and algorithms for structures that address nonlinear material behaviors and investigate structural stability at different scales.

T-splines discretizations for large deformation contact problems

Rossana Dimitri, Giorgio Zavarise
Dipartimento di Ingegneria dell'Innovazione, Università del Salento, ITALY

The isogeometric analysis (IGA) represents a new method of computational analysis that merges design and analysis into one model by using a unified geometric representation. NURBS (Non-Uniform Rational B-Splines) and T-Splines are the most widespread technologies in today's CAD modelling tools and therefore are adopted as basis functions for analyses. In this work the isogeometric concept [1] is applied to study the large deformation multi-body contact problems, which still represent a significant challenge for the analysts in terms of robustness and stability of solutions. For this reason, the development of more efficient, fast and stable finite element contact discretizations is still a hot topic, especially due to the fact that engineering applications become more and more complex. Among the most important challenges that have to be met with respect to finite element discretization is the sensitivity of contact problem to the geometry accuracy.

Non-smooth, C^0-continuous finite element basis functions lead to convergence problems in the analysis of sliding contact and to highly oscillatory contact interactions even when convergence is achieved. Various contact smoothing techniques have been proposed in the literature to address this issue [2-6] which consider the smoothing of the master and slave surfaces as achieved by high-order finite element interpolation based on Lagrange, hierarchic, spline or NURBS interpolations. Within the isogeometric framework, a contact surface possessing C^1 or higher continuity is easily achieved and significant advantages over conventional finite element descriptions have been demonstrated in the last years by applying NURBS based isogeometric discretizations [4-7] to frictionless and/or frictional multi-patch contact problems.

A key problem of multivariate NURBS basis functions, in any case, is their rigid tensor product structure, which implies that refinement is a global process propagating throughout the domain. A possible way to improve the quality of contact results in terms of local pressures and global time-history curves with limited increase in the computational effort is represented by local refinement. This has been recently considered in [8] for frictionless contact applications by using analyis-suitable T-splines discretizations and here extended to large deformation Coulomb frictional contact problems. A Gauss-point-to-surface (GPTS) formulation is combined with the penalty method to treat the contact constraints in the discretized setting, as done in [9]. Using the Bèzier extraction, the suitable T-splines isogeometric discretizations are automatically generated for any analysis-suitable CAD geometry and easily incorporated into the finite element framework [10]. Some numerical examples show that the proposed contact formulation deliver accurate and robust predictions and demonstrate the potential of T-spline-based IGA to solve challenging contact problems in 2D and 3D.

References

[1] T.J.R. Hughes, J.A. Cottrell, Y. Bazilevs, Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement, Comput. Methods Appl. Mech. Eng., 194(2005) 4135-4195.

[2] M.A. Puso, T.A. Laursen, J. Solberg, A segment-to-segment mortar contact method for quadratic elements and large deformations, Comput Methods Appl. Mech. Eng., 197 (2008) 555-566.

[3] D. Franke, A. Düster, V. Nübel, E. Rank, A comparison of the h-, p-, hp- and rp-version of the FEM for the solution of the 2D Hertzian contact problem, Comput. Mech., 45 (2010) 513-522.

[4] L. De Lorenzis, I. Temizer, P. Wriggers, G. Zavarise. A large deformation frictional contact formulation using NURBS-based isogeometric analysis, Int. J. Numer. Meth. Eng. 87(13) (2011) 1278-1300.

[5] L. De Lorenzis, P. Wriggers, G. Zavarise, A mortar formulation for 3D large deformation contact using NURBS-based isogeometric analysis and the augmented Lagrangian method, Comput. Mech. 49(1) (2012) 1-20.

[6] I. Temizer, P. Wriggers, T.J.R. Hughes, Contact treatment in isogeometric analysis with NURBS, Comput. Methods Appl. Mech. Eng. 200 (2011) 1100-1112.

[7] I. Temizer, P. Wriggers, T.J.R. Hughes, Three-dimensional mortar-based frictional contact treatment in isogeometric analysis with NURBS, Comput. Methods Appl. Mech. Eng. 209 (2012) 115-128.

[8] R. Dimitri, L. De Lorenzis, M. Scott, P.Wriggers, R.L. Taylor, G. Zavarise, Isogeometric large deformation frictionless contact using T-splines, Comput. Methods Appl. Mech. Eng. 269 (2014) 394-414.

[9] KA. Fischer, P. Wriggers, Frictionless 2D contact formulations for finite deformations based on the mortar method, Comput. Mech. 36 (2005) 226-244.

[10] M.A. Scott, X. Li, T.W. Sederberg, T.J.R. Hughes, Local refinement of analysis-suitable T-splines, Comput. Methods Appl. Mech. Eng. 213-216 (2012) 206-222.

Following forces as an inverse contact algorithm

A. Konyukhov, R. Izi, K. Schweizerhof
Karlsruhe Institute of Technology

On of the essential finding withing the the geometrically exact theory for contact interactions [1], is that the theory can be understood and implemented in a very simple fashion starting with 2D examples, [2]. The geometrically exact theory for the computational contact mechanics is described in step-by-step manner, using examples of strict derivation from a mathematical point of view. The final goal of the theory is to construct in the independent of approximation form /so-called covariant form/ including application to high-order and isogeometric finite elements.

The essential findings within this reduction are several cases, which allowing to verify all computational algorithms step-by-step as well as to solve another famous computational mechanics.

One of this is representation of the algorithm for the following forces as an inverse contact algorithm. This computational algorithm is constructed as contact algorithm in a covariant geometrically exact form, in which the contact force is assumed as an external following force, while the corresponding tangent matrices are just selected from the already derived in closed form matrices. Such a method is independent on the approximation and is easily applied to the isogeometric finite elements.

Another example is FE modeling numerical modeling of the generalized Euler-Eytelwein problem, the analytically solution is recently obtained in [3].

References

[1] A. Konyukhov, K. Schweizerhof. Computational contact mechanics: geometrically theory for arbitrary shaped bodies, Springer, Heildelberg, New York, Dordrecht, London. (2013), 443 P.

[2] A.Konyukhov, R. Izi Introduction to Computational Contact Mechanics: A Geometrical Approach, Wiley, (2015), 304 P.

[3] A. Konyukhov. Contact of ropes and orthotropic rough surfaces. ZAMM Z. Angew. Math. Mech., 1–18 (2013).

Application of the virtual element method to non-conforming contact interfaces

W. T. Rust[1], P. Wriggers[1], B. D. Reddy[2]
[1]G. W. Leibniz Universität Hannover
Institute of Continuum Mechanics
Appelstraße 11, 30167 Hannover, Germany
Email: rust@ikm.uni-hannover.de
[2]University of Cape Town
Center of Computational Mechanics
Cape-Town

In the literature numerous formulations for classical contact and different discretization methods for the contact zone are available. Widely used are the penalty and Lagrange multiplier method to enforce contact constraints in the finite element environment. But especially non-conforming contact interfaces require a high effort in discretizing the contact surface in order to properly link the degrees of freedom. In this work we adapt the standard contact approaches penalty and Lagrange multiplier method to be used with virtual elements in the contact zone. This allows for an easy and robust contact algorithm for non-conforming meshes with a nodal enforcement of the contact constraint.

The main idea of the recently developed virtual element method is to find a single function that can project the nodal values on the element area while being compatible with the interpolated values on the boundary. This gives the advantage over classical finite elements that it is easily possible to discretize a geometry using convex or non-convex polygons with arbitrary number of vertices. Additionally it offers a simple formulation, easy integration and the possibility to achieve higher continuity.

For this work the virtual elements are implemented for the case of linear elasticity. Therefore a suitable polynomial decomposition has to be chosen. Furthermore, the elements in a contact problem generally can be subject to different loading states along the surface. Because the pure virtual ansatz function only represents an interpolation of the nodal values, the method has to be stabilized with an additional term to give good results. It will be shown in which way this was done in the present work in order to prevent effects such as locking.

The actual contact element consists mainly of a nodal projection algorithm. Here the feature of the virtual elements is used that arbitrary nodes can be inserted along the element boundary. Since this can be done without the need to recompute or change the element ansatz, an easy and robust algorithm is created that can actually transform a non-conforming mesh in a conforming mesh during the computation by projecting and inserting nodes on the opposing surface where needed. After matching the meshes the well known node-to-node contact procedures can easily be applied. In this talk the used methods are described and numerical examples are presented that show the behaviour in small deformation problems with and without friction.

Non-unique Equilibria of a Statically Indeterminate System with Coulomb Friction

W. Steiner
University of Applied Sciences Upper Austria
Stelzhamerstrasse 23, 4600 Wels, Austria

The goal of this contribution is to show that the Coulomb friction model in a statically indeterminate system can result in ambiguous states of equilibrium. We demonstrate that a simple framework under specific external loading is in equilibrium and satisfies all conditions of Coulomb friction in the support points for several states of stress and deformation. The presented problem is motivated by the example of a two dimensional elastic body discussed in [1] where non-unique static solutions were found, too. In our presentation, a new method [2] which extends Castigliano's theorem to problems with Coulomb friction is applied to efficiently analyze numerous states of sticking and sliding of the framework's supports.

References

[1] P. Hild, An example of nonuniqueness for the continuous static unilateral contact model with Coulomb friction, C. R. Acad. Sci. Paris, Ser. I 337 (2003).

[2] W. Steiner, The use of Castigliano's theorem in Coulomb friction problems, Acta Mechanica, September 2014, Volume 225, Issue 9, pp 2471-2483.

Efficient computation of surface-dominated problems using isogeometric finite elements

Christopher Zimmermann, Roger A. Sauer

Aachen Institute for Advanced Study in Computational Engineering Science (AICES)
RWTH Aachen University
Schinkelstr. 2, 52062 Aachen, Germany

Physical problems are often dominated by local surface effects that determine the entire problem behaviour. Geometrically exact membrane and shell formulations can be used to model a wide range of mechanical applications [1]. An example is a water droplet coming into contact with a rigid substrate, where the contact boundary varies due to the deformation of the droplet. Another example is rough surface contact, where at very small asperities local stress peaks can appear. The global behaviour of the entire system is in both examples governed by the behaviour of local, critical surface domains. The accuracy of the computational results strongly depend on the surface discretization especially in local contact regions. Refinement of the entire mesh leads to high computational costs due to a great number of degrees of freedom located at surface domains of minor interest.

In this work, a technique for efficient computation of surface-dominated problems using isogeometric finite elements is presented. The tensor product structure of standard NURBS-based isogeometric analysis lacks local refinement [2]. To overcome this drawback, and to achieve highly locally refined meshes, the Locally Refined B-splines (LR B-splines) were introduced [3]. This refinement is directly performed in the parameter space that is represented locally. The LR B-splines still achieve the high accuracy of isogeometric analysis due to the smoothness of the basis functions that describe the geometry and approximate the solution field. An adaptive refinement indicator is used to determine isogeometric finite elements for refinement which are located at critical surface domains. By the use of the Bézier extraction operator [4], the isogeometric elements are included into an existing finite element simulation framework conveniently.

Numerical results are presented for computations of surface-dominated problems, involving frictional contact and multiple contact interactions. The technique of adaptive local refinement exhibits a decrease of the computational costs, while still achieving high accuracy in comparison to uniformly refined models.

References

[1] R. A. Sauer, T. X. Duong, and C. J. Corbett. A computational formulation for constrained solid and liquid membranes considering isogeometric finite elements. Computer Methods in Applied Mechanics and Engineering, 271(0) (2014), 48–68.

[2] T.J.R. Hughes, J. Cottrell, and Y. Bazilevs. Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement. Comp. Meth. Appl. Mech. Engrg. 194 (2005), 4135–4195.

[3] T. Dokken, T. Lyche, and K. F. Pettersen. Polynomial splines over locally refined box-partitions. Computer Aided Geometric Design, 30(3) (2013), 331–356.

[4] M.J. Borden, M.A. Scott, J.A. Evans, and T.J.R. Hughes. Isogeometric finite element data structure based on Bézier extraction of NURBS. Int. J. Numer. Meth. Engng. 87 (2011), 15–41.

Evaluation of a Finite Element Approach for Damping Determination

Olaf Peter Hentschel, Marius Bonhage,
Lars Panning-von Scheidt, Jörg Wallaschek
Institute of Dynamics and Vibration Research,
Leibniz University Hannover,
Appelstrasse 11, Hannover, 30167, Germany

Due to irreversible processes within the material, mechanical energy in vibratory systems is partly converted into thermal energy. This energy dissipation is known as damping. In freely oscillating systems, damping is characterized by a decreasing vibration amplitude, while in externally excited systems, the vibration amplitude is limited by the damping process. One of the primary objectives in many fields of mechanical engineering is the prediction and reduction of vibration amplitudes to reduce the risk of HCF. In this context, mechanical damping is of essential relevance for the dynamic behavior. Nonlinear calculation tools are used to calculate vibration amplitudes. To enable such a calculation, the specification of structural damping due to dissipations within the material is necessary.

In the full length paper, a well-known law for material damping description (see [1], [2] and [3]) is verified. Thereby, material damping is characterized as a function of local stress amplitudes. To identify the resulting structural damping, a calculation of local stresses is necessary. In a first step, an analytical approach is used to calculate the damping on the basis of the stress distribution. In a second step, the local stress distribution of the described structure is calculated using a finite element approach. In this context, damping is calculated using the identified discrete stress values through the structure. Hereby, an equivalent tensile stress theory is applied. Varying the mesh density, the resulting damping is compared to the calculated damping using the analytical results. Finally it can be seen that the quality of the calculated damping value depends on the quality of the calculated stress value and therefore on the mesh density.

References

[1] B. J. Lazan. Damping of Materials and Members in Structural Mechanics. Pergamon Press (1968). New York.

[2] Y. Kume, F. Hashimoto. Material damping of cantilever beams. Journal of Sound and Vibration 80(1) (1982). 1 –10.

[3] G. D. Gounaris, N. K. Anifantis. Structural damping determination by fnite element approach. Computers and Structures (1999). 445 –452.

Modeling of non-stationary vibration signals based on the modified Kronecker sequences

Iulian Girip and Ligia Munteanu
Institute of Solid Mechanics, Romanian Academy, Bucharest

A vehicle is a multi degree of freedom system and its vibration behaviour (ride comfort) is often random with highly non-stationary characteristics. The vibration behaviour depends on the natural frequencies and mode shape of the vehicle. The paper proposes a model to represent non-stationary random vibration signals based on the modified Kronecker sequences implemented into quasi-Monte Carlo method. The modified Kronecker sequence constructed via generalizing golden ratio is one of the special types of low discrepancy sequences which have better dimensional projections [1]. The actual modeling and simulation of non-stationary random data is more suitable for seismological signals but not proper for the vehicle vibrations [2], [3]. Under these circumstances, this paper presents a new algorithm for finding the modified Kronecker sequences in order to generate non-stationary vehicle vibration signals which mostly withhold the amplitude-frequency-time distribution of the sample signal. The simulated signals hold the similar waveform and their fluctuations of energy as those of the sample signal [4]. Some examples to prove the effectiveness of the model conclude the paper.

References

[1] H.Chi. Generation of parallel modified Kronecker seequences. Monte Carlo Methods Appl., 19 (2013), 261-271.

[2] L.Munteanu, V.Chiroiu, C.Brişan, D.Dumitriu, T.Sireteanu, S.Petre. On the 3D normal tire/off-road vibro-contact problem with friction. Mechanical Systems and Signal Processing (in press) (2014).

[3] C.Brişan, R.V.Vasiu, L.Munteanu. A modular road auto-generating algorithm for developing the road models for driving simulators. Transportation Research part C: Emerging Technologies, 26 (2013), 269-284.

[4] I.Girip. Computational and experimental Monte Carlo algorithms for modeling the systems with hysteretic characteristics. PhD thesis Institute of Solid Mechanics, September 2014.

A homotopy method for the eigenvalue analysis of circular saw blade with inner slits

Junjie Luo, André Schmidt, Lothar Gaul
University of Stuttgart

Vibration reduction is one of the main concerns in the design of circular saw blades because it causes such problems as the deterioration of surface quality and the inaccuracy of sizing. Some studies indicate that the inclusion of inner slits with some viscoelastic media can reduce vibration and noise. Therefore, in this contribution we investigate how inner slits influence the structural behaviour of the saw blades under consideration, particularly the critical rotational speed, above which buckling occurs. Furthermore, inner slits filled with viscoelastic media cause internal damping, when relative deformation of the saw blade occurs. As internal damping can destabilize a system, stability needs to be examined.

This work is focused on how the stability of circular saw blades is influenced by different parameters, such as the rotational speed and the damping properties. This question leads to solving a parameter-dependent eigenvalue problem. Since bifurcation may occur in the spectrum of the eigenform, it is necessary to follow the paths of eigenvalues with respect to some specific parameter in both stable and unstable regions. Because the Newton-Raphson method is extremely sensitive to initial conditions, an algorithm based on homotopy is used to solve this problem [1]. The homotopy method, which is sometimes called the numerical continuation method, in principle, is to construct a homotopy from a trivial function to the one of interest. Smooth curves starting from the trivial solutions will then lead to the desired solutions under some circumstances. The method is applied to a perfect circular saw blade model and then is compared to the one with inner slits, which enables us to study the effects of different geometries. The results can contribute to the further optimization in the design of circular saw blades.

References

[1] J. Hadian, A.H. Nayfeh. Modal interaction in circular plates. Journal of Sound and Vibration (1990), 142(2), 279–292.

Meta-structures for Cloaking Bending Waves

M. Gei[1], D.J. Colquitt[2], M. Brun[3], A.B. Movchan[4], N.V. Movchan[4], I.S. Jones[5]
[1]University of Trento, Trento, Italy
[2]Imperial College, London, U.K.
[3]University of Cagliari, Cagliari, Italy
[4]University of Liverpool, Liverpool, U.K.
[5]Liverpool John Moores University, Liverpool, U.K.

The talk addresses an important issue of cloaking transformations for fourth-order partial differential equations representing flexural waves in thin elastic plates [1]. It is shown that, in contrast with the Helmholtz equation, the general form of the partial differential equation is not invariant with respect to the cloaking transformation. The significant result of this paper is the analysis of the transformed equation and its interpretation in the framework of the linear theory of pre-stressed plates. The paper provides a formal framework for transformation elastodynamics as applied to elastic plates. Furthermore, an algorithm is proposed for designing a broadband square cloak for flexural waves, which employs a regularised push-out transformation. Illustrative numerical examples show high accuracy and efficiency of the proposed cloaking algorithm. In particular, a physical configuration involving a perturbation of an interference pattern generated by two coherent sources is presented. It is demonstrated that the perturbation produced by a cloaked defect is negligibly small even for such a delicate interference pattern.

References

[1] D.J. Colquitt, M. Brun, M. Gei, A.B. Movchan, N.V. Movchan, I.S. Jones. Transformation elastodynamics and cloaking for flexural waves. J. Mech. Phys. Solids 72 (2014), 131–143 .

Shrink fit with FGM-hub subject to heating and rotation

Eray Arslan, Werner Mack,
Department of Mechanical Engineering, Inonu University, Malatya, Turkey
Institute of Mechanics and Mechatronics, Vienna University of Technology, Vienna, Austria

Shrink fits find wide-spread use in mechanical engineering as an efficient means of transfer of moment; examples are shrunk-on rings, armature bandages in rotating machines, or tires of railway wheels [1]. Since the transferable moment essentially depends on the interface pressure between inclusion and hub, it should be as large as possible. This may be facilitated by a partially plastic design, which however also has some drawbacks like a possible permanent redistribution of the stresses after operating at high angular speeds and temperatures [2]. Hence, an interesting alternative (or at least supplement) to admitting partial plasticization is the use of a functionally graded material (FGM), particularly for the hub. As is well known, in a machine part of FGM the material properties vary continuously and can - to a certain extent - be tailored in an appropriate way [3]. Therefore, the aim of the present study is to investigate the essential features of a purely elastically designed shrink fit under plane stress conditions with solid homogeneous inclusion and functionally graded hub, taking both rotation and an elevated temperature into account. The material properties are presupposed to vary according to a power law in the radial direction. The basic grading law is however not postulated for the volume fractions of the constituents, but for the modulus of elasticity, and the dependence of the other physical quantities on the radius then is derived by the rule of mixture. This gives rise to a unique grading index for all the physical quantities (except for Poisson's ratio), and a closed-form solution of the differential equations can be found. Thus, a purely analytical discussion of the problem is possible.

It is shown that in case of radially decreasing density for a sufficiently large ratio of outer surface radius of the hub to interface radius a considerably better performance at rotation may be achieved, accompanied by a substantial saving of weight as compared to a homogeneous hub [4]. These two significant advantages must be weighted against the fact that a marginally worse evolution of the interface pressure with increasing temperature may occur. To discuss these features, a comprehensive comparison of the interface pressure in a shrink fit with homogeneous hub and with FGM-hub for arbitrary combinations of loading by rotation and by heating is provided.

References

[1] I. Dolezel, V. Kotlan, B. Ulrych. Design of joint between disk and shaft based on induction shrink fit. J. Comp. Appl. Math. 270 (2014), 52–62.

[2] W. Mack, M. Plöchl. Transient heating of a rotating elastic-plastic shrink fit. Int. J. Engng. Sci. 38 (2000), 921–938.

[3] J.J. Sobczak, L. Drenchev. Metallic functionally graded materials: a specific class of advanced composites. J. Mater. Sci. Technol. 29 (2013), 297–316.

[4] E. Arslan, W. Mack. Shrink fit with solid inclusion and functionally graded hub. Comp. Struct. 121 (2015), 217–224.

A projection approach to optimal control of elastic beam dynamics

Georgy Kostin
Institute for Problems in Mechanics RAS, Moscow, Russia

A projection approach is presented to model and optimize the controlled lateral motions of an elastic beam. The time T of control process is fixed and the beam has the shape of a long rectangular prism which height $2a_1$ is sufficiently greater than the sizes $2a_2$ and $2a_3$ of its cross section so that

$$\{t, \boldsymbol{x}\} \in \Omega = (0,T) \times V \ , \quad \boldsymbol{x} = \{x_1, x_2, x_3\}^{\mathrm{T}} \in \mathbb{R}^3 \ , \quad V = \{\boldsymbol{x} : \ x_i < a_i \ , \ i = 1,2,3\} \ .$$

The approach is based on an integro-differential statement [1] of the original initial-boundary value problem in linear elasticity with the velocity-momentum and stress-strain relations generalized according to

$$\int_\Omega \boldsymbol{v} \cdot \boldsymbol{q} \,\mathrm{d}\Omega = 0 \quad \text{for} \quad \forall \boldsymbol{q} \in L^2(\Omega) \quad \text{and} \quad \int_\Omega \xi : \tau \,\mathrm{d}\Omega = 0 \quad \text{for} \quad \forall \tau \in L^2(\Omega) \ ;$$

$$\boldsymbol{p}_t = \nabla \cdot \sigma \ , \quad \varepsilon = \frac{1}{2}(\nabla \boldsymbol{w} + \nabla \boldsymbol{w}^{\mathrm{T}}) \ , \quad \boldsymbol{v} = \boldsymbol{w}_t - \rho^{-1} \boldsymbol{p} \ , \quad \xi = \varepsilon - \mathsf{C}^{-1} : \sigma \quad \text{for} \quad \{t, x\} \in \Omega \ ;$$

$$\sigma \cdot \boldsymbol{n} = \boldsymbol{0} \quad \text{for} \quad x_1 = a_1 \quad \text{and} \quad x_i = \pm a_i \ , \ \ i = 2,3 \ ; \quad \boldsymbol{w} = \{0, 0, u(t)\}^{\mathrm{T}} \quad \text{for} \quad x_1 = -a_1 \ ;$$

$$\boldsymbol{w} = \boldsymbol{p} = \boldsymbol{0} \quad \text{for} \quad t = 0 \ .$$

Here, the volume density ρ and the elastic modulus tensor C are mechanical parameters for the beam of homogeneous isotropic material, $\boldsymbol{n}$ is the output normal to the beam boundary. The displacement vector $\boldsymbol{w}(t,x)$, the momentum density vector $\boldsymbol{p}(t,x)$, and the stress tensor $\sigma(t,x)$ are unknown variables, whereas $\boldsymbol{q}(t,x)$ is a vector of virtual momentum density and $\tau(t,x)$ is a tensor of virtual stresses. The function $u(t) \in \mathbb{R}$ is a boundary control input.

On the basis of piecewise polynomial approximations of both the trial variables $\boldsymbol{w}$, $\boldsymbol{p}$, σ and the test functions $\boldsymbol{v}$, τ, a semi-discretization scheme is worked out by taking into account the properties of the beam symmetry [2]. An explicit energy criterion of solution quality is given by

$$\Delta = \Phi \Psi^{-1} < \delta \ll 1 \quad \text{with} \quad \Phi = \frac{1}{2} \int_\Omega (\rho \, \boldsymbol{v} \cdot \boldsymbol{v} + \xi : \mathsf{C} : \xi) \,\mathrm{d}\Omega \quad \text{and} \quad \Psi = \frac{1}{2} \int_\Omega (\rho^{-1} \boldsymbol{p} \cdot \boldsymbol{p} + \varepsilon : \mathsf{C} : \varepsilon) \,\mathrm{d}\Omega \ ,$$

where $\Phi[u]$ defines the global energy error of approximate solutions and $\Psi[u]$ is the time integral of the total mechanical energy stored by the beam during the control process.

The resulting ODE system is used to design the optimal control $u^*(t)$ that minimizes the mean energy of the elastic beam [3] as follows:

$$J[u^*] = \min_{u \in H^1(0,T)} J[u] \quad \text{with} \quad J = T^{-1} \Psi$$

subject to the terminal constraints

$$\boldsymbol{w}(T,x) = \{0, 0, w_T\}^{\mathrm{T}} \quad \text{and} \quad \boldsymbol{p}(T,x) = \boldsymbol{0} \quad \text{for} \quad \boldsymbol{x} \in V \ .$$

The numerical results obtained for the spatial case are analyzed and compared with the conventional Bernoulli beam model.

References

[1] G.V. Kostin, V.V. Saurin. Integrodifferential Relations in Linear Elasticity. De Gruyter, Berlin, 2012.

[2] G.V. Kostin, V.V. Saurin. Modelling and analysis of the natural vibrations of a prismatic elastic beam based on a projection approach. J. Appl. Math. Mech. 75 (2011), 700–710.

[3] G.V. Kostin. The construction of optimal control for the motion of elastic bodies by using the method of integro-differential relations. J. Comp. Sys. Sci. Int. 46 (2007), 532–542.

Energy-momentum conserving discretization of mixed shell elements for large deformation problems

Alexander Janz, Peter Betsch
Institute of Mechanics
Karlsruhe Institute of Technology
Otto-Ammann-Platz 9, 76131 Karlsruhe, Germany
peter.betsch@kit.edu, alexander.janz@kit.edu
www.ifm.kit.edu

In the present work we consider structure-preserving integration methods in the context of mixed finite elements. Low-order mixed finite elements such as the shell element proposed in [1] typically exhibit improved coarse mesh accuracy. On the other hand energy-momentum (EM) consistent time-stepping schemes have been developed in the realm of nonlinear structural dynamics to enhance the numerical stability properties. EM schemes typically exhibit superior robustness and thus offer the possibility to use large time steps while still producing physically meaningful results. Accordingly, combining mixed finite element discretizations in space with EM consistent discretizations in time shows great promise for the design of numerical methods with superior coarse mesh accuracy in space and time.

Starting with a general Hu-Washizu-type variational formulation we develop a second-order accurate structure-preserving integration scheme. The present approach is applicable to a large number of mixed finite element formulations. As sample application we will deal with the mixed shell element [1]. The resulting method can be viewed as mixed extension of the EM method proposed in [2]. Numerical examples dealing with large deformations will show the improved coarse mesh accuracy in space and time of the advocated approach.

References

[1] W. Wagner and F. Gruttmann. A robust non-linear mixed hybrid quadrilateral shell element. *Int. J. Numer. Meth. Engng*, Vol. **64**, 635–666, 2005.

[2] P. Betsch and N. Sänger. On the use of geometrically exact shells in a conserving framework for flexible multibody dynamics. *Comput. Methods Appl. Mech. Engrg.*, Vol. **198**, 1609–1630, 2009.

The Strong Formulation Finite Element Method Applied to Structural Mechanics Problems

Francesco Tornabene, Nicholas Fantuzzi, Michele Bacciocchi, Erasmo Viola
DICAM Department, School of Engineering and Architecture, University of Bologna, ITALY

The Strong Formulation Finite Element Method (SFEM) [1] is a numerical approach that can be used for solving civil, environmental, mechanical, aerospace and naval engineering problems. Generally, practical engineering problems are complex due to geometry, material and load discontinuities. For solving them, it is necessary to divide the whole domain into finite elements of arbitrary shape. The mapping technique is introduced at this level to transform an arbitrarily shaped element to a parent element (computational element). The classic Finite Element Method (FEM) uses the above procedure and the problem at the parent element level is solved by means of weak (variational) formulation. On the contrary, the SFEM summarises a class of methods that is able to approximate total and partial derivatives at discrete points, thus the solution is found in its strong form. The SFEM has its own roots in the Differential Quadrature Method (DQM), which was introduced in the early 1970s. Nevertheless, DQM does not allow to treat arbitrarily shaped domains and problems where discontinuities are present. These features are proper of finite element approaches, in which the global domain is divided into several smaller elements, and after the assembly procedure they solve the complete system. Therefore the SFEM is an hybrid scheme given by the DQM and the FEM. The most significant difference between these two methodologies lays on the formulation used for solving the parent element. In order to clarify the idea about the fact that the SFEM comprehends several numerical techniques, the reader can review a class of methods in the article [2], where it has been clarified that the most important and wide-spread numerical approaches are a sub-class of the method of weighted residuals. Moreover a former review article about DQM can be found in [3] where a state of the art of that time was given. Unfortunately the authors limited their analysis to DQM and they did not focus their attention on the generalization of the DQM concepts to a wider prospective. As far as the authors are concerned, the first paper regarding the present topic was presented in [4]. The discussion was extended in a survey paper published recently [5], where a significant historical review about strong and weak numerical tools was carried out. The authors provided stability and accuracy of one-dimensional and two-dimensional problems when compared to classic exact solutions related to structural problems, such as rods, beams, membranes and plates. The first application of the SFEM regarding one-dimensional in-plane multi-stepped and multi-damaged arches was published in [6]. The authors investigated the vibration of thin membranes in a review paper [7], where several well-known numerical applications were compared to the literature. Some other applications were presented concerning the behavior of elastostatic and elastodynamic plane structures in [8, 9, 10, 11]. Later the authors presented the SFEM applied to the modal analysis of Reissner-Mindlin plates [12, 13]. The SFEM based on DQM and Radial Basis Function (RBF) method has been presented in the work [14]. Finally in the works [15, 16] a particular emphasis has been given to the stress recovery procedure for the evaluation of the three dimensional strain and stresses at all the physical points of the problem. As a definition the SFEM is a numerical procedure that decomposes the physical domain or problem in finite elements and used the strong formulation inside each element mapped on the parent (or computational) element. When in the above procedure the weak formulation is used (instead of the strong form), the WFEM is defined. The latter is well-known in literature as FEM. This paper aims to investigate the application of the SFEM to structural mechanics problems. Since its numerical solutions depend on the number of collocation points, the basis functions used, the location of the points and the number of domain divisions, the authors report in graphical form the stability, accuracy and reliability of the present technique. In this way several aspects are raised and remarks are given as closure.

References

[1] F. Tornabene, N. Fantuzzi (2014) - Mechanics of Laminated Composite Doubly-Curved Shell Structures. The Generalized Differential Quadrature Method and the Strong Formulation Finite Element Method, Esculapio, Bologna. ISBN: 978-88-7488-687-6.

[2] B.A. Finlayson, L.E. Scriven, The method of weighted residual: a review, Appl. Mech. Rev., 19 (1966) 735-748.

[3] C.W. Bert, M. Malik, Differential quadrature method in computational mechanics, Appl. Mech. Rev., 49 (1996) 1-27.

[4] F. Tornabene, N. Fantuzzi, M. Bacciocchi, The Strong Formulation Finite Element Method: Stability and Accuracy, Fract. Struct. Integr., 29 (2014) 251-265.

[5] F. Tornabene, N. Fantuzzi, F. Ubertini, E. Viola. Strong Formulation Finite Element Method Based on Differential Quadrature: A Survey, Appl. Mech. Rev., (2014) In Press. doi:10.1115/1.4028859.

[6] E. Viola, M. Dilena, F. Tornabene, Analytical and Numerical Results for Vibration Analysis of Multi-Stepped and Multi-Damaged Circular Arches, J. Sound Vib. 299 (2007) 143-163.

[7] N. Fantuzzi, F. Tornabene, E. Viola, Generalized Differential Quadrature Finite Element Method for Vibration Analysis of Arbitrarily Shaped Membranes, Int. J. Mech. Sci. 79 (2014) 216-251.

[8] E. Viola, F. Tornabene, E. Ferretti, N. Fantuzzi, On Static Analysis of Composite Plane State Structures via GDQFEM and Cell Method, CMES 94 (2013) 421-458.

[9] E. Viola, F. Tornabene, E. Ferretti, N. Fantuzzi, Soft Core Plane State Structures Under Static Loads Using GDQFEM and Cell Method, CMES 94 (2013) 301-329.

[10] E. Viola, F. Tornabene, E. Ferretti, N. Fantuzzi, GDQFEM Numerical Simulations of Continuous Media with Cracks and Discontinuities, CMES 94 (2013) 331-369.

[11] N. Fantuzzi, New insights into the strong formulation finite element method for solving elastostatic and elastodynamic problems, Curved Layer. Struct. 1 (2014) 94-127.

[12] E. Viola, F. Tornabene, N. Fantuzzi, Generalized Differential Quadrature Finite Element Method for Cracked Composite Structures of Arbitrary Shape, Compos. Struct. 106 (2013) 815-834.

[13] N. Fantuzzi, F. Tornabene, E. Viola, Four-Parameter Functionally Graded Cracked Plates of Arbitrary Shape: a GDQFEM Solution for Free Vibrations, Mech. Adv. Mat. Struct. (2014) In Press. doi: 10.1080/15376494.2014.933992.

[14] N. Fantuzzi, F. Tornabene, E. Viola, A.J.M. Ferreira, A Strong Formulation Finite Element Method (SFEM) Based on RBF and GDQ Techniques for the Static and Dynamic Analyses of Laminated Plates of Arbitrary Shape, Meccanica 49 (2014) 2503-2542.

[15] N. Fantuzzi, F. Tornabene, Strong Formulation Finite Element Method for Arbitrarily Shaped Laminated Plates - I. Theoretical Analysis, Adv. Aircraft Space. Sci. 1 (2014) 124-142.

[16] N. Fantuzzi, F. Tornabene, Strong Formulation Finite Element Method for Arbitrarily Shaped Laminated Plates - II. Numerical Analysis, Adv. Aircraft Space. Sci. 1 (2014) 143-173.

Weighted overconstrained least-squares mixed finite elements for hyperelasticity

Alexander Schwarz, Karl Steeger, Jörg Schröder

Institute of Mechanics, Faculty of Engineering,
University Duisburg-Essen, Universitätsstr. 15, 45141 Essen, GERMANY
alexander.schwarz@uni-due.de, karl.steeger@uni-due.de, j.schroeder@uni-due.de

The main goal of this contribution is the improvement of the approximation quality of geometrically nonlinear elastic problems solved by the least-squares finite element method (LSFEM). Compared with other variational approaches as for example the Galerkin method, a general drawback of least-squares formulations is the unsatisfying approximation quality especially for lower-order elements, see e.g. [1]. In the current work we present a mixed element based on a first-order stress-displacement formulation resulting from a classical least-squares method. By including the stress symmetry in an explicit form, similar to [2], an overconstrained system is derived. We consider different weights for this additional constraint and use a mixed least-squares formulation with a maximal cubic polynomial interpolation order. For the continuous approximation of the stresses Raviart-Thomas elements are used, while for the displacements standard conforming elements are employed. We provide some benchmark problems with the main focus on bending-dominated hyperelastic problems. The proposed formulation is compared to recently developed classical LSFEMs and Galerkin FEMs, in order to show the improvement of performance and accuracy.

References

[1] A. Schwarz, J. Schröder, G. Starke, A modified least-squares mixed finite element with improved momentum balance, Int. J. Numer. Meth. Eng. 81 (2010) 286–306.

[2] A. Schwarz, K. Steeger and J. Schröder. Weighted overconstrained least-squares mixed finite elements for static and dynamic problems in quasi-incompressible elasticity, Comp. Mech. 54 (2014) 603–612.

A stress-velocity least-squares mixed finite element formulation for incompressible elastodynamics.

Carina Nisters[1], Alexander Schwarz[1], Jörg Schröder[1]

[1]Institute for Mechanics, Faculty of Engineering,
University Duisburg-Essen, Universitätsstr. 15, 45141 Essen, GERMANY
carina.nisters@uni-due.de, alexander.schwarz@uni-due.de, j.schroeder@uni-due.de

The purpose of the presented work is the development and implementation of a stress-velocity least-squares mixed finite element formulation under the assumptions of small strain elastodynamics. The idea of implementing a stress-velocity formulation is well-known in fluid-dynamics and is the basic motivation of the $\sigma - v$ formulation in solid dynamics. The formulation of the stress-velocity approach in fluid dynamics is found in [1]. The L_2-norm minimization of the time-discretized residuals of the given first-order system of partial differential equations leads to a functional depending on the stresses $\boldsymbol{\sigma}$ and the velocities $\mathbf{v}$

$$\mathcal{F}(\boldsymbol{\sigma}, \mathbf{v}) = \frac{1}{2} \left(||\alpha_1 (\operatorname{div} \boldsymbol{\sigma} + \mathbf{f} - \rho\, \mathbf{a})||_0^2 + ||\alpha_2 (\mathbb{C}^{-1} \boldsymbol{\sigma} - \nabla^s \mathbf{u})||_0^2 \right),$$

with $\alpha_i\,(i = 1, 2)$ denoting weighting factors, wherein $\mathbf{f}$ describes the body forces, ρ is the density of the solid and $\mathbb{C}$ is the fourth-order material tensor. The accelerations are denoted with $\mathbf{a} = \mathbf{a}(\mathbf{v})$ and the displacements with $\mathbf{u} = \mathbf{u}(\mathbf{v})$, both values are depending on the velocities $\mathbf{v}$ due to the applied time discretization. The stresses are interpolated with vector-valued Raviart-Thomas interpolation functions, while for the approximation of the velocities a standard polynomial interpolation is chosen, see e.g. [2]. First, we examine the influence of different time integration schemes and secondly investigate weighting factors α_i for each residual, which are tested for the spectrum of numerical functionality, compared with the stress-displacement formulation in [3]. In the numerical examples, the proposed stress-velocity mixed least-squares finite element formulation is tested and the results are compared to alternative standard approaches.

References

[1] Z. Cai, B.Lee, and P.Wang, Least-squares methods for incompressible Newtonian fluid flow: Linear stationary problems, SIAM J. NUMER. ANAL. Vol. 42, No. 2 (2004) 843–859.

[2] A. Schwarz, J. Schröder, and G. Starke, A modified least-squares mixed finite element with improved momentum balance, Int. J. Numer. Meth. Eng. 81 (2010) 286–306.

[3] K. Steeger, A. Schwarz, and J. Schröder. A weighted least-squares mixed finite element formulation for quasi incompressible elastodynamics. PAMM. Vol. 11 (2011) 287–288.

Comparison of a mixed least-squares formulation using different approximation spaces.

Karl Steeger[1], Jörg Schröder[1], Alexander Schwarz[1]

[1]Institute of Mechanics, Faculty of Engineering,
University Duisburg-Essen, Universitätsstr. 15, 45141 Essen, GERMANY
karl.steeger@uni-due.de, j.schroeder@uni-due.de, alexander.schwarz@uni-due.de

The main goal of the present work is the comparison of the performance of a least-squares mixed finite element formulation where the solution variables (displacements and stresses) are interpolated using different approximation spaces. Basis for the formulation is a weak form resulting from the minimization of a least-squares functional, compare e.g. [1]. The stresses are approximated in $H^1(\mathcal{B})$ or in $H(\text{div}, \mathcal{B})$, whereas the displacements are always interpolated in $H^1(\mathcal{B})$. As suitable functions for $H^1(\mathcal{B})$ standard interpolation polynomials of Lagrangian type are chosen. For the conforming discretization of the Sobolev space $H(\text{div}, \mathcal{B})$ vector-valued Raviart-Thomas interpolation functions, see also [2], are used. The resulting elements are named as $P_m P_k$ and $RT_m P_k$. Here m (stresses) and k (displacements) denote the approximation order of the particular interpolation function. For the comparison we consider two-dimensional structural mechanical problems under plain strain conditions and small strain assumptions.

References

[1] Cai, Z. and Starke, G., [2004], "Least-squares methods for linear elasticity", SIAM J. Numer. Anal., Vol. 42, pp. 826-842.

[2] Raviart, P. A. and Thomas, J. M., [1977], "A mixed finite element method for second order elliptic problems", Lectures Notes in Math., Vol. 606, Springer Verlag.

A surface oriented solid formulation based on a hybrid Galerkin-collocation method

Sven Klinkel[1], Lin Chen[1], Wolfgang Dornisch[2]
[1]Institute of Structural Analysis and Dynamics, RWTH Aachen University
[2]Institute of Applied Mechanics, University of Kaiserslautern

The contribution is concerned with a numerical method to analyze the mechanical behavior of 3D solids. The method employs directly the geometry defined by the boundary representation modeling technique, which is frequently used in CAD to define solids. It combines the benefits of the isogeometric analysis methodology [1] with the scaled boundary finite element method [2]. In the present approach, only the boundary surfaces of the solid is discretized. No tensor-product structure of three-dimensional objects is exploited to parameterize the physical domain. The weak form is applied only on the boundary surfaces. The governing partial differential equations of elasticity are transformed to an ordinary differential equation (ODE) of Euler type. For the numerical analysis the isogeometric Galerkin approach is employed to approximate the displacement response at the boundary surfaces. It exploits the two-dimensional NURBS objects to parameterize the boundary surfaces. To solve the Euler type ODE, the NURBS based collocation approach is applied, see [3]. The accuracy of the method is validated against the analytical solutions. The presented method is able to analyze solids, which are bounded by an arbitrary number of surfaces. Numerical examples will show the capabilities of the presented method.

References

[1] T. J. R. Hughes, J. A. Cottrell, Y. Bazilevs. Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement. Comput. Meth. Appl. Mech. Engrg. 194 (2005), 4135–4195.

[2] C. Song, J. P. Wolf. The scaled boundary finite-element method-alias consistent infinitesimal finite-element cell method-for elastodynamics. Comput. Methods Appl. Mech. Engrg. 147 (1997), 329–355.

[3] S. Klinkel, L. Chen, W. Dornisch. A NURBS based hybrid collocation-Galerkin method for the analysis of boundary represented solids. Comp. Meth. Appl. Mech. Engrg. 284 (2015), 689–711.

Application of Discontinuous Galerkin Finite Element Method for Discontinuities in Small Deformation Regimes

H. R. Bayat, S. Reese, S. Wulfinghof
Institute of Applied Mechanics (IFAM) RWTH Aachen University, Germany

In this paper, finite element formulation is defined in the frame work of discontinuous Galerkin method. Discontinuous Galerkin methods (dGm) are classically used in fluid mechanics, however recently their application in solid mechanics has become more vivid among scientists. Of special interest is their application in problems tackling with cracks [1], in elliptic problems with constraints such as incompressibility which leads to volumetric locking phenomenon and also in some structural models of shells, plates and beams with constraints for compatibility relation between the degrees of freedom which brings about shear locking [2].

While classical continuous Galerkin methods must be piecewise affine conforming (continuous), dG methods can be applied for discontinuities across element boundaries by application of piecewise constant approximates within each element and use of slope limiters to ensure stability for higher order schemes [2]. In the present work, we implement the Q1SP [3] element in dG formulation and investigate its behaviour for small deformation regimes.

References

[1] J. Mergheim, E. Kuhl, P. Steinmann. A hybrid discontinuous Galerkin/interface method for the computational modelling of failure. Communications in Numerical Methods in Engineering (2004), 511–519.

[2] A. Lew, A.T. Eyck, R. Rangarajan. Some Applications of Discontinuous Galerkin Methods in Solid Mechanics. IUTAM Symposium on Theoretical, Computational and Modelling Aspects of Inelastic Media. (2008), 167-189.

[3] S. Reese, P. Wriggers, B.D. Reddy. A new locking-free brick element technique for large deformation problems in elasticity. Computers and Structures (2000), 291–304.

A New Mixed Finite Element for the Analysis of Structures with Material and Geometric Nonlinearities

Nicola A. Nodargi and P. Bisegna
Department of Civil Engineering and Computer Science
University of Rome "Tor Vergata", via del Politecnico 1, 00133 Rome, Italy
E-mail: nodargi@ing.uniroma2.it, URL: www.dicii.uniroma2.it

The analysis of structural behavior in engineering applications often requires to take into account material and geometric nonlinearities. Aim of the present work is the formulation of a finite element which allows an accurate and effective treatment of such problems.

In the recent literature, mixed finite elements have been proved to be especially appropriate to overcome well-known limits of the classical displacement-based formulation (for an overview on mixed elements, see [1]). In a nonlinear material framework, usually involving direct strain-stress relationships, the natural derivation of such elements is based on the Hu–Washizu functional (e.g., see [2]). A modified version of such functional, in which internal equilibrium is a priori enforced in strong form (for instance, see [3]), is here adopted. Accordingly, the displacement field interpolation is only required on the element boundary and square-integrable interpolations can be selected for stresses and strains. In particular, an interpolation which is continuous only within the individual element is selected for the self-equilibrated stress field (for instance, see [4, 5]), while the strain field is expanded by a linear combination of Dirac delta measures centered on element Gauss points. As a consequence, stress and strain interpolation parameters can be condensed at element level by solving the local compatibility and constitutive equations, whereas the equilibrium is imposed at structural level with respect to the unknown nodal displacement degrees of freedom.

In the present work, an original procedure to carry out the element state determination is proposed. More specifically, the local equations are regarded as the stationarity conditions associated to a linearly constrained minimization problem involving an appropriate incremental element energy function. An efficient minimization algorithm, based on a Newton-Raphson iterative scheme in conjunction with a suitable line-search method, is then discussed.

Structural scale applications are carried out for assessing the performances of the proposed finite element. Geometric nonlinearities are treated by means of the the polar decomposition based corotational formulation proposed in [6]. Besides elastic case studies, applications involving elasto-plastic materials or shape memory alloys are considered. In particular, numerical results show that the present element is free from locking and spurious zero-energy modes, and exhibits a good response in presence of mesh distortion.

References

[1] D. Boffi, F. Brezzi, M. Fortin. Mixed finite element methods and applications. Springer-Verlag, Berlin, (2013).

[2] P. Wriggers, J. Korelc. On enhanced strain methods for small and finite deformations of solids. Comput. Mech. 18(1996) 413–428.

[3] R.L. Taylor, F.C. Filippou, A. Saritas, F. Auricchio. A mixed finite element method for beam and frame problems. Comput. Mech. 31(2003), 192–203.

[4] A. Madeo, G. Zagari, R. Casciaro. An isostatic quadrilateral membrane finite element with drilling rotations and no spurious modes. Finite Elem. Anal. Des. 50(2012), 21–32.

[5] R. Grimaldi, D. Addessi, V. Ciampi. Localization and regularization behavior of mixed finite elements for 2D structural problem with damaging material. Comput. Methods Appl. Mech. Engrg. 197(2007), 255–264.

[6] F. Caselli, P. Bisegna. Polar decomposition based corotational framework for triangular shell elements with distributed load. Int. J. Numer. Methods Eng. 95(2013), 499–528.

Discussion of the Particle Finite Element Method in the Context of Strength of Materials

Matthias Sabel, José York Duran, Christian Sator, Ralf Müller
Institute of Applied Mechanics
University of Kaiserslautern,
Gottlieb-Daimler-Str.,
D-67663, Kaiserslautern, Germany

The particle finite element method (PFEM) combines the benefits of discrete modeling techniques and continuum based methods. While discrete models are well suited for tracking large changes in the topology, they are computationally expensive for problems with large length- and time scales. Standard finite element methods (FEM) are a common way to solve problems with macroscopic length and time scales. However, simulations with large configurational changes like a separation of material require frequent remeshing. The PFEM algorithm used for this work first determines the boundary of a set of particles which is accomplished with the so called α-shape method. This method originates from the field of computer graphics and is named after its crucial parameter α. After detecting the boundary, the region can be meshed with finite elements. Subsequently, a FEM problem can be solved, where the deformation data of previous load steps have to be included. Finally, the resulting displacements are used to update the particle coordinates, and the history variables are stored for the following load step. A central part in the analysis is the determination of the boundary. Although the PFEM has been applied to a number of complex applications with free boundaries (liquids), so far only experience based values for α are provided in literature. The numerical examples in this contribution show that the choice of α affects the material behavior. A simple tensile test shows the influence of α on the material response. High values represent tough materials, while a low α is associated with brittle behavior. Other examples studied in this work include machining simulations. Especially the formation of chips leads to large deformations and topological changes, which is why the machining simulations are well suited to demonstrate the advantages of the PFEM. In these machining simulations, the cutting force is studied for a varying α and again a dependency of the material response on α is observed.

Nonlinear SFEM with fluctacting input parameters

I. Caylak, Alex Dridger, Rolf Mahnken
University of Paderborn, Chair of Engineering Mechanics

In many engineering applications the materials, e.g. adhesives, are heterogeneous. This heterogeneity leads often to uncertainty in the material properties and to uncertainty in the mechanical response. Therefore, macroscopically heterogeneous adhesives should be modelled by a stochastic approach instead a deterministic approach. This contribution presents a nonlinear stochastic finite element method (SFEM). Mostly used SFEMs are *Monte Carlo* (MC), *Galerkin* and *polynomial chaos expansion* (PCE) [1, 2].

The key idea of our contribution is to consider the uncertainity by random material parameters, which are modeled as stochastic fields. Then, from experimental data the distribution of the random variables, i.e. elastic and inelastic material parameters, are known. Consequently, elastic and inelastic material parameters are expanded with the PCE. Furthermore, in the context of material nonlinear problems the stresses and the material tangent, respectively, are functions of stochastic variables. The collacation method is used in order to calculate the PC coefficients of functions. In addition, during the local *Newton* algorithm in order to calculate the plastic corrector $\Delta\lambda$ also a stochastic *Newton* method is considered.

As a numerical example we consider the static problem for uniaxial tension of the rectangular plate. This problem is investigated under plane strain conditions with suitable boundary conditions and some stochastic material parameters. For the material description the non-linear elasto-plastic material model of adhesives as developed in *Mahnken* and *Schlimmer* [3] is used. Results of the deterministic solution and the influence of the distribution of the material parameters are presented.

References

[1] H. G. Matthies and A. Keese. Galerkin methods for linear and nonlinear elliptic stochastic partial differential equations. Comput. Methods Appl. Mech. Engrg., 194:1295-1331, (2005), 491–503.

[2] R. G. Ghanem and P. D. Spanos. Stochastic Finite Elements: A Spectral Approach. Springer-Verlag, New York, (1991)

[2] R. Mahnken; M. Schlimmer: Simulation of strength difference in elasto-plasticity for adhesive materials. Int. J. Numer Met Eng 63, Nr. 10, 1461-1477, (2005)

A high-order enrichment strategy for the finite cell method

Meysam Joualain, Alexander Düster
Numerical Structural Analysis with Application in Ship Technology (M-10)
Hamburg University of Technology, Schwarzenbergstr. 95C, 21073 Hamburg, Germany

The modeling and the computation of structures made of heterogeneous materials is very challenging. Developing a suitable finite element model of such structures involves a lot of efforts and is generally very time consuming. Moreover, the resulting numerical models are usually very large and thus require considerable computational resources. The finite cell method (FCM), which is an immersed boundary method based on finite elements, is considered as a possible method for numerically handling these kinds of problems, especially for homogenization applications and multiscale problems [1, 2, 3]. The main characteristic of the FCM is that – thanks to the application of the immersed boundary method – the mesh can be defined independently from the geometry. This allows to employ Cartesian grids for the purpose of meshing, for instance. In addition, the employment of high-order shape functions together with the FCM makes it possible to achieve high-order convergence rates and, in consequence, to save computational resources.

Although the nonconforming meshes in the FCM lead to a significant simplification in the modeling part, they might cause several difficulties on the solution part. One of the possible problems will occur if the solution exhibits a kink inside an element, for instance at material interfaces. In such a case, the regularity of the solution is lost and so the optimal convergence rate might be deteriorated. In this presentation we will address this issue and show some examples where this can be problematic. We will also explore different remedies by considering the partition of unity method as well as domain decomposition techniques [4]. The proposed solution is based on defining high-order enrichment functions with the help of a high-order implicit representation of the material interface. To this end, we will apply Lagrange shape functions defined on Babuška-Chen points together with the level-set function. Several numerical examples will serve to illustrate the performance of the proposed method.

References

[1] J. Parvizian, A. Düster, E. Rank. Finite cell method – h- and p-extension for embedded domain problems in solid mechanics. Comput. Mech. 41 (2007), 121–133

[2] A. Düster, J. Parvizian, Z. Yang, E. Rank. The finite cell method for three-dimensional problems of solid mechanics. Comput. Methods in Appl. Mech. Eng. 197 (2008), 3768–3782

[3] A. Düster, H.-G. Sehlhorst, E. Rank. Numerical homogenization of heterogeneous and cellular materials utilizing the finite cell method. Comput. Mech. 50 (2012), 413–431

[4] M. Joulaian, A. Düster. Local enrichment of the finite cell method for problems with material interfaces. Comput. Mech. 52 (2013), 741–762

An efficient and robust Reissner–Mindlin shell formulation for isogeometric analysis

Wolfgang Dornisch[1], Ralf Müller[1] , Sven Klinkel[2]
[1]Institute of Applied Mechanics, University of Kaiserslautern
[2]Chair of Structural Analysis and Dynamics, RWTH Aachen University

Isogeometric analysis was introduced in 2005 by Hughes et al. [4]. The novel idea is to use the basis functions of the geometry description of the design model also for the analysis. Thus, the geometry is represented exactly on element level. A closer integration of design and analysis is fostered by the usage of one common geometry model for design and analysis. A prevalent choice for the geometry description in isogeometric shell analysis is Non-Uniform Rational B-spline (NURBS) surfaces, which are commonly used in industrial design software to model thin structures. In order to directly compute structures defined by NURBS surfaces, an efficient isogeometric shell formulation is required. One of the main advantages of NURBS-based isogeometric analysis is the higher continuity between elements, which occurs if k-refinement is used for order elevation. The higher continuity requires special efforts to arrive at an efficient rotational-based shell formulation. In [1] it is shown that concepts which are adapted from common Lagrange-based Reissner–Mindlin shell formulations potentially yield divergent deformation convergence behavior for order elevation. A new rotation formulation adapted to isogeometric analysis is presented in [1]. This formulation makes high-order computations competitive in comparison to low-order computations. An extension to geometries with kinks and sharp interfolds is presented in [2]. However, this formulation lacks efficiency in comparison to standard Lagrange-based shell formulation.

In this contribution an isogeometric Reissner–Mindlin shell formulation derived from the continuum theory is presented. The shell body is described by a shell reference surface, which is defined by NURBS surfaces, and a director vector. The director vector in the reference configuration is interpolated from nodal values, which are determined in a pre-process by a patch-wise L_2-projection. The director vector in the current configuration is computed by an orthogonal rotation using Rodrigues' tensor in every integration point. The interpolation of the axial vector of the rotation is performed according to an idea mentioned in [5] and applied in [3]. In the framework of C^0-continuous shell formulations the higher numerical effort does not pay off. But the more accurate interpolation of the rotations entails significantly more precise results for NURBS-based isogeometric analysis with high continuity. A multiplicative update formulation for the rotations accounts for finite rotations. Several benchmark examples show the superior accuracy of the presented shell formulation for linear and nonlinear computations. The convergence behavior is shown to be correct for k-refinement. The computational cost of the new formulation are compared to standard formulations, assessing both costs for a given mesh size and costs to fall below a predefined error bound.

References

[1] W. Dornisch, S. Klinkel, B. Simeon. Isogeometric Reissner–Mindlin shell analysis with exactly calculated director vectors. Comput. Meth. Appl. Mech. Engrg. 253 (2013), 491–504.

[2] W. Dornisch, S. Klinkel. Treatment of Reissner–Mindlin shells with kinks without the need for drilling rotation stabilization in an isogeometric framework. Comput. Meth. Appl. Mech. Engrg. 276 (2014), 35–66.

[3] F. Gruttmann, S. Klinkel, W. Wagner. A finite rotation shell theory with application to composite structures. Rev. Européenne Élém. 4 (1995), 597–631.

[4] T.J.R. Hughes, J.A. Cottrell, Y. Bazilevs. Isogeometric analysis: CAD, finite elements, NURBS, exact geometry and mesh refinement. Comput. Meth. Appl. Mech. Engrg. 194 (2005), 4135–4195.

[5] J.C. Simo, D.D. Fox. On a stress resultant geometrically exact shell model. Part III: Computational aspects of the nonlinear theory. Comput. Meth. Appl. Mech. Engrg. 79 (1990), 21–70.

Corotational flat triangular elements for the nonlinear analysis of thin shell structures

Federica Caselli, Paolo Bisegna
Department of Civil Engineering and Computer Science,
University of Rome "Tor Vergata"

This contribution shows the effectiveness and versatility of the corotational formulation in the development of shell finite elements for geometric and material nonlinear analysis of thin structures [1, 2, 3].

The corotational approach is based on the idea of separating rigid body motions from strain producing ones. The fundamental idea is to find a transformation that operates like a filter on the global displacements, removing the rigid-body contribution before the finite element kernels use them. The advantage is that most of the geometric nonlinearity of the problem is transferred to the transformation relating the filtered displacements to the total displacements. The strain energy thus obtained proves to be objective with respect to rigid body motions. In addition, the corotational approach leads to a separation of the material and geometrical nonlinearities.

For problems with arbitrarily large displacements and rotations but small strain response, existing elements formulated in the small strain regime can be reused as core elements in the geometrically nonlinear context, after large rigid body motions have been filtered out. This feature has been exploited in [3] in order to develop a new Shape Memory Alloy (SMA) shell finite element. SMA-based devices typically undergo significant configuration changes in their operation, however large rotations rather than large strains are most often involved. By resorting to the corotational formulation, it is possible to adopt constitutive models formulated in the small strain regime, which are simpler and computationally less expensive with respect to finite strain approaches. In particular, a small strain plane-stress SMA model based on the thermodynamically consistent formulation proposed in [4] and able to account for the pseudo-elastic as well as the shape memory effect was considered in [3].

Although the hypothesis of large displacements and rotations but small strains is often suitable, there are many practical situations where shell structures undergo large strains, like in metal forming and applications involving rubber-like or biological materials. The restriction to small strain has represented a long standing limitation for the corotational approach. One of the main reason is that, in order to allow the extension to large strain problems, the correct identification of the element rigid motion plays a crucial role. In particular, as first pointed out by Crisfield [5], a polar decomposition based corotational approach is required. In [2], a corotational flat triangular element for large strain analysis of thin shells with applications to soft biological tissues has been presented, which is based on the polar decomposition based corotational framework developed in [1], combined with an original kinematic description of the core-element displacement field, based on the multiplicative (instead of additive) superposition of membrane and bending actions.

References

[1] F. Caselli, P. Bisegna. Polar decomposition based corotational framework for triangular shell elements with distributed loads. Int. J. Numer. Methods Eng. 95(6) (2013), 499–528.

[2] F. Caselli, P. Bisegna. A corotational flat triangular element for large strain analysis of thin shells with applications to soft biological tissues. Comput. Mech. 54(3) (2014), 847–864.

[3] P. Bisegna, F. Caselli, S. Marfia, E. Sacco. A new SMA shell element based on the corotational formulation. Comput. Mech. 54(5) (2014), 1315–1329.

[4] V. Evangelista, S. Marfia, E. Sacco. Phenomenological 3-D and 1-D consistent models for SMA materials. Comput. Mech. 44(3) (2009), 405–421.

[5] G. F. Moita, M. A. Crisfield. A finite element formulation for 3-D continua using the co-rotational technique. Int. J. Numer. Methods Eng. 39(22) (1996), 3775–3792.

Flexure Hinge Mechanisms Modeled by Nonlinear Euler-Bernoulli-Beams

Robert Friedrich, Rolf Lammering
Helmut-Schmidt-University/University of the Federal Armed Forces,
Institute of Mechanics, Holstenhofweg 85, 22043 Hamburg, Germany

A flexure hinge is an innovative engineering solution for providing relative motion between two adjacent stiff members by the elastic deformation of a flexible connector. Compliant mechanisms based on flexure hinges find application in aeronautics, positioning systems and medical-technical devices. Major drawback of these mechanisms is their limited motion range. Also modeling and optimization are highly demanding steps in the design process. This study aims for a simplification of the design process by means of optimization methods.

Regarding the modeling of flexure hinges, mostly linear approaches have been investigated. A recent study demonstrates that higher order Bernoulli beam elements of variable cross section can be employed to model the linear static and dynamic behavior of flexure hinges accurately [1, 2]. In this study the element is applied with regard to the geometric nonlinear nature of deformation. Axial displacement and transverse deflection are approximated by higher order shape functions. The element stiffness matrix and element tangential stiffness matrix are established in accordance with [3]. The integrals are solved numerically taking the variable cross section of the flexure hinge into account. The solution of nonlinear equations requires an iterative Newton-Raphson procedure where the unknown state $\mathbf{x}^{(r)}$ is computed from the known state $\mathbf{x}^{(r-1)}$ and its increment $\delta\mathbf{x}^{(r-1)}$. After assembly of the stiffness matrix $\mathbf{K}\left(\mathbf{x}\right)^{(r-1)}$ and the tangential stiffness matrix $\mathbf{T}\left(\mathbf{x}\right)^{(r-1)}$ and for a given load vector $\mathbf{F}$ the nonlinear system of equations is solved. The solution procedure is implemented in Matlab. This modeling approach captures the geometric nonlinear behavior of flexure hinges, which is demonstrated by selected benchmarks where the convergence and accuracy of the solution is regarded. Compared to commonly applied finite element methods, this approach reduces the model's degrees of freedom enormously.

Size and computational time privilege the application of the model to geometric design parameter optimization. The maximum output motion serves as objective function, which is constrained by the mechanical stress in the flexure hinge region while geometric design parameters are optimization variables. It is shown that the proposed design process is simplified, accelerated and even more reliable than established methods based on continuum models.

References

[1] R. Friedrich, R. Lammering, M. Rösner. On the modeling of flexure hinge mechanisms with finite beam elements of variable cross section. Prec. Eng. 38 (2014), 915–920.

[2] R. Friedrich, R. Lammering. An improved modeling approach for circular flexure hinges with application to geometry optimization. Proc. Appl. Math. Mech. 13 (2013), 95–96.

[3] J.N. Reddy. An introduction to nonlinear finite element analysis. Oxford Univ. Press (2010).

A layer-wise theory for the structural analysis of glass and photovoltaic laminates

K. Naumenko, H. Altenbach, V. A. Eremeyev
Otto-von-Guericke-University Magdeburg, Institute of Mechanics,
PF 4120, D-39016 Magdeburg, Germany

Laminated plates with glass skin layers and a core layer from soft polymers are widely used in civil engineering. Photovoltaic panels currently available on the market are composed from stiff front and back layers and a core layer. The core layer comprises the solar cells in a soft polymeric encapsulant.

This contribution presents a layer-wise theory for the structural analysis of glass and photovoltaic laminates. Governing equations for the individiual layers, kinematical constraints, and appropriate interaction forces represent the starting point to deduce a twelfth order system of partial differential equations. The Airy stress function, the deflection function, and the vector of relative in-plane displacements of the skin layers are the primary variables in this theory.

A system of differential equations with respect to scalar potentials is presented for symmetric laminates. Three of these differential equations correspond to the first order shear deformation plate theory. The additional second order differential equation represents a correction for laminates with soft core layer. In order to demonstrate the importance of this correction, closed form analytical solutions for a plate strip are derived. Moreover, the significance of additional boundary conditions is shown, using two examples of free and framed plate edges. Since the edges of photovoltaic laminates are usually fixed by frames to restrict the relative sliding of skin layers, the boundary conditions for framed edges are of importance for practical applications.

A user-defined element based on a layer-wise theory for laminated glasses and photovoltaic panels

J. Eisenträger, K. Naumenko, H. Altenbach
Otto-von-Guericke-University Magdeburg, Institute of Mechanics,
PF 4120, D-39016 Magdeburg, Germany

Laminated plates and solar modules are composed of three layers, whereas the core layer, comprising the solar cells and their encapsulation, is much more shear-compliant than the skin layers. If the difference in stiffnesses becomes too large, first-order shear deformation theories cannot be applied to these laminated plates. For this reason, a layer-wise theory for plates has been introduced in [1].

This contribution presents a user-defined quadrilateral Serendipity element with quadratic shape functions, based on the layer-wise theory. In order to determine the element stiffness relation, the principle of virtual work is deduced starting with the governing equations of the layer-wise theory. The element possesses nine degrees of freedom including two components of the in-plane displacement vector of the laminate, two components of the relative displacement vector of the skin layers, the deflection, two components of the cross-section rotation vector of the laminate, and two components of the relative rotation vector of the skin layers. It is implemented into the Abaqus FE code using the subroutine “User Element”. Closed form analytical solutions are referred to the layer-wise theory in order to verify the results of the user-defined element. To assess the deformation state of a photovoltaic panel in practice, two different kinds of boundary conditions are taken into account: free supports and a rigid frame.

References

[1] K. Naumenko, V. A. Eremeyev. A layer-wise theory for laminated glass and photovoltaic panels. Composite Structures 112 (2014), 283–291.

On a mathematical problem of cusped double-layered plates

Natalia Chinchaladze
I. Vekua Institute of Applied Mathematics & Faculty of Exact and natural Sciences
I. Javakhishvili Tbilisi State University

Investigations of cusped elastic prismatic shells actually takes its origin from the fifties of the last century, namely, in 1955 I.Vekua raised the problem of investigation of elastic cusped prismatic shells, whose thickness on the prismatic shell entire boundary or on its part vanishes ([1], [2], [3]). The survey of the elastic shells and plates is given in [4]. Using I. Vekua's dimension reduction method hierarchical models for elastic layered prismatic shells are constructed in [5], [6]. In the symmetric case of the prismatic shells we have to do with plates of variable thickness (see, e.g. [4]). The present talk is devoted to the system of degenerate partial differential equations arising in the zero approximation of hierarchical models of layered plates. The well-posedness of boundary value problems under the reasonable boundary conditions at the cusped edge and given displacements at the non-cusped edge is studied. Some numerical resalts are also given.

Acknowledgement. The work was supported by the Shota Rustaveli National Science Foundation (SRNSF) grant No. 30/28.

References

[1] I. Vekua, On a method of computing prismatic shells, Akad. Nauk Gruzin. SSR. Trudy Tbiliss. Mat. Inst. Razmadze, 21 (1955), 191–259, in Russian.

[2] I. Vekua, Theory of thin shallow shells of variable thickness, Akad. Nauk Gruzin.SSR Trudy Tbiliss. Mat. Inst. Razmadze, 30 (1965), 3–103, in Russian.

[3] I. Vekua, Shell Theory: General Methods of Construction, Monographs, AdvancedTexts and Surveys in Pure and Applied Mathematics, 25. Pitman (Advanced Pub-lishing Program), Boston, MA. 1985.

[4] G.V. Jaiani, Cusped shell-like structures, Springer Briefs in Applied Science and Technology, Springer, Heidelberg, 2011.

[5] G. Jaiani, On a model of layered prismatic shells, Proceedings of I. Vekua Institute of Applied Mathematics, 63 (2013), 13–24, for electronic version see: http://www.viam.science.tsu.ge/publish/proceed.html.

[6] N. Chinchaladze, On a cusped double-layered prismatic shell, Proceedings of I. Vekua Institute of Applied Mathematics, 64 (2014, in print), for electronic version see: http://www.viam.science.tsu.ge/publish/proceed.html.

A membrane finite element formulation for woven fabrics using the generalized polyconvex hyperelastic model

Vu Ngoc Khiêm [1], Mahmood Jabareen [2], and Mikhail Itskov [1]
[1] Department of Continuum Mechanics, RWTH Aachen University,
Kackertstr. 9, 52072 Aachen, Germany
[2] Faculty of Civil and Environmental Engineering Technion, Israel Institute of Technology,
Haifa 32000, Israel
vu@km.rwth-aachen.de

Due to the light weight and the high degree of drapability, textile composite has become a prominent replacement for metals in automotive and aeronautic industry. In this contribution, we propose a membrane finite element formulation for woven fabrics, and implement it into the commercial FEM software package ABAQUS via user subroutine UEL. The constitutive law for woven fabrics is based on the generalized polyconvex hyperelastic model [1], and is formulated with respect to a geodesic basis. In contrast to the usual approach via the user-defined material in commercial FEM softwares, no Green-Naghdi stress rate formulation is needed and the components of the deformation gradient always lie in the element plane. Thus, important features of textile composites such as deformed fiber orientations can be efficiently captured. Robustness and convergence properties of the proposed membrane element formulation are demonstrated by various numerical examples. Predictions of the constitutive model and finite element formulation are comparable to experimental data of picture frame and double dome test.

References

[1] A. E. Ehret, M. Itskov. A polyconvex hyperelastic model for fiber-reinforced materials in application to soft tissues. Journal of Materials Science. 42 (2007), 8853–8863.

On the Mechanics of Ultralight Hollow Microlattices

Bernhard Eidel
Lehrstuhl für Numerische Mechanik, Universität Siegen, Germany
Institut für Mechanik, Universität Duisburg-Essen, Germany

Recent experimental work on the fabrication and mechanical testing of hollow-tube microlattices [1, 2] has attracted considerable interest in industry and academia for several outstanding mechanical properties of these structural materials; a lattice with a unit cell of bcc type is (i) ultralight with $\rho \approx 0.9$ mg/cm^3, (ii) shows considerable energy dissipation and (iii) exhibits almost full recovery of the initial lattice shape even after compressions up of 50%. Hollow-tube microlattices are a novel class of cellular materials, which (iv) allow by their periodic, quasi-deterministic composition for a highly effective topology optimization in contrast to foams, which are stochastic in nature.

The objective of the present work is a thorough understanding of the mechanics of microlattices by finite element analysis. For that aim, we establish a thin-shell model along with an isotropic material law of elasto-plasticity. Already the reduced unit-cell model representing the total lattice captures significant characteristics of compression experiments very well and reveals the interplay of geometrical features such as buckling and folding with inelastic phenomena of plasticity and damage. We show in terms of Ashby-diagrams, that the simulation results follow universal scaling laws for effective stiffness as well as for strength as a function of density, which is in excellent agreement with the experiments. We propose a modified, bcc-derived unit cell that exhibits an increase in elastic stiffness and strength by 1-2 orders of magnitude compared to the original bcc lattice of the experiments in [1, 2], thus making a step towards ultralight and ultratight properties of tailored structural materials.

References

[1] T.A. Schaedler, A.J. Jacobsen, A. Torrents, A.E. Sorensen, J.Lian, J.R. Greer, L. Valdevit, W.B. Carter. Ultralight metallic microlattices. Science 334 (2011), 962–965.

[2] A. Torrents, T.A. Schaedler, A.J. Jacobsen, W.B. Carter, L. Valdevit. Characterization of nickel-based microlattice materials with structural hierarchy from the nanometer to the millimeter scale. Acta Mater. 60 (2012), 3511–3522.

Mechanical Analysis of Metallic SLM-Lattices on Small Scales: Finite Element Simulations versus Experiments

S. Didam[1], B. Eidel[1], A. Ohrndorf[2], H.-J. Christ[2]
[1] Lehrstuhl für Numerische Mechanik, Universität Siegen, Germany
[2] Lehrstuhl für Materialkunde und Werkstoffprüfung, Universität Siegen, Germany

There is an ever more increasing need for lightweight materials in mechanical structures on different length scales. Foam materials are inherently stochastic in nature, which limits the effectivity of optimizing their composition. Quite recently, high-resolution selective laser melting (SLM) has enabled the fabrication of metallic lattices on small scales. As a consequence, lightweight cellular materials with tailored properties with respect to stiffness, strength, and ductility can be fabricated with high fidelity and applied in promising applications as e.g. for bone replacement in orthopedic surgery.

The present contribution deals with the mechanics of metallic SLM-micro-lattices made of stainless steel as a prototype material. Finite element analyses with material parameters identified in experiments investigate the structural load bearing behavior of different unit cell topologies as well as global deformation of entire lattices. Typical failure modes like local buckling as well as global localization in shear bands are analyzed in simulations of compressive, shear and mixed loading conditions and are compared to corresponding results of mechanical tests. Ashby diagrams [1] for the scaling behavior of stiffness and strength at various densities are determined from the simulations and experiments, respectively. Characterization by scanning electron microscopy accompany the deformation analyses and provide detailed insights into laser-melting microstructure in single struts and their nodal connections. Perspectives of future topology optimization of SLM-microlattices are addressed with respect to product-specific properties.

References

[1] L.J. Gibson, M.F. Ashby. Cellular Solids. Cambridge University Press (1999).

Investigation of elastoplastic effects of cables under large spatial deformation

Vanessa Dörlich[1,2], Stefan Diebels[1], Joachim Linn[2]
[1]Lehrstuhl für Technische Mechanik, Universität des Saarlandes
[2]Department Mathematical Methods in Dynamics and Durability,
Fraunhofer Institute for Industrial Mathematics

Cables are flexible, slender components with a complex multi-layer structure. They can be described physically correct by the Cosserat rod theory [1] which basically consists of three parts: geometrically exact kinematics relating configuration variables and objective strain measures, balance equations that govern the dynamic equilibrium of the sectional kinetic quantities and constitutive equations which yield the sectional forces and moments in terms of the deformation.

Finding an appropriate constitutive model is necessary to enable a realistic simulation of a structure under load. While a standard linear elastic description may be sufficient to represent small deformations in academic test examples, every day experience shows that for example electric cables behave quite differently under large deformations. Since they are composed of various layers, effects like friction, delamination and pull-out can occur in practice. Additionally, the particular layers consist of different classes of materials including metallic wires which may reach the yield stress under large spatial deformations. Consequently, inelastic material behavior cannot be neglected in the model. A viscoelastic constitutive model formulated for the sectional force and moment quantities and objective strain measures of the Cosserat rod theory was already presented in [2] and [3]. Our work aims at a similar approach for plasticity. Coupling between the single stiffnesses can be expected for complex, multiaxial loading and should eventually be included in the description extending the approach of Simo et al. [4].

Furthermore, it is important that the model parameters are accessible. Our contribution focuses on the design of suitable experiments. Classical standardized tests to measure bending, tensile and torsional stiffnesses cannot cover the mentioned effects and are only sufficient to determine the parameters for elastic models. Consequently, finite deformation experiments have to be developed which provide access to parameters in the inelastic range. Information about the coupling between the single stiffnesses can be provided by multiaxial experiments combining bending, torsion and tension of cables.

References

[1] J.C. Simo. A finite strain beam formulation: the three dimensional dynamic problem - Part I. Comp. Method. Appl. M. 49 (1985), 55–70.

[2] H. Lang, J. Linn, A. Tuganov. Geometrically exact Cosserat rods with Kelvin–Voigt type viscous damping. Mech. Sci. 4 (2013), 79–96.

[3] O.A. Bauchau, Z. Lao, M. Lyu, S. Brändle, J. Linn. Formulation of viscoelastic constitutive laws for beams in flexible multibody dynamics. Proc. IMSD, Busan, Korea (2014).

[4] J.C. Simo, K.D. Hjelmstad, R.L. Taylor. Numerical formulations for finite deformation problems of beams accounting for the effect of transverse shear. Comp. Method. Appl. M. 42 (1984), 301–330.

Determination of a Constitutive Friction Law Using an Elastic-Plastic Half-Space Model

F. Beyer, K. Willner

Chair of Applied Mechanics, University of Erlangen-Nuremberg, Egerlandstraße 5, 91058 Erlangen, Germany

This contribution presents a procedure, how contact interaction of rough surfaces are examined with a half-space approximation for the deduction of a constitutive friction law. Roughness is composed of fine random irregularities of a surface structure which affect the functionality of surfaces. Due to this fact, technical surfaces get into contact only with their peaks for low to moderate contact load. The resulting real contact area is smaller than the apparent contact area. Furthermore, the contact load is distributed locally on the surface peaks which are exposed to plastic deformation. For the investigation of friction of surfaces it is necessary to determine this deformation as frictional shear stress is transferred in the real contact.

In addition, the mutli-scale character coming with roughness demands on the one hand a very fine resolution for the accurate reproduction of such irregularities and surface peaks. On the other hand, the contact region has to be large enough to be representative to show waviness or the surface structure itself.

The usage of the Finite-Element-Method is a broadly used possibility to treat such a complex matter numerically due to the continuously rising computing capacity. However, computing power is still a limiting factor. Therefore, a canny alternative is taken into account with the application of a half-space approximation. It only depends on the two-dimensional surface boundary which consumes less computing effort than FEM with its 3-dimensional volume approach for the same surface resolution and area.

The elastic-plastic half-space model based on a variational principle of the minimization of the complementary energy is validated and calibrated with experiments. The simulations of several contact configurations are combined for the determination of a constitutional friction law consisting of two equations. The law is implemented into the framework of the commerical FEM-software Simufact.forming to see the impact in a metal forming process which is commonly simulated with Tresca's law of friction.

Application of fibre Bragg grating sensors for residual stress analysis

Susann Hannusch, Dr. Martin Stockmann, Prof. Jörn Ihlemann

Professorship of Solid Mechanics, Faculty of Mechanical Engineering,
Technische Universität Chemnitz, Chemnitz, Germany

All components exhibit residual stresses which might be caused by the manufacturing process. Especially casting is a procedure in which influences of residual stresses, like the warpage of components after being removed from the mould, become visible. Obviously these effects are related to metals, plastics or reinforced plastics. Additionally, the residual stresses can influence life expectancy or the capacity to withstand stresses of the component in a positive or negative way.

In order to profit from the positive or to minimize the negative influences of the residual stresses it is important to know which kind of residual stresses are emerged. Therefore, the measurement of residual stresses is necessary although they can only be measured indirectly. Moreover, most common methods have been developed for metals and not all of them can be applied to plastics. One well-investigated method is the hole-drilling method which was developed by Josef Mathar [1]. This method varies the stress state by drilling a hole into the specimen. The resulting deformations are measured and converted into the desired residual stresses by employing a mathematical model. At the present state the hole-drilling method is standardised in ASTM E837-08 [2] for the application on metal materials in which the strains are measured by strain-gauge rosettes. This method measures the strains solely on the surface while changing the stress state which is one main disadvantage. In consequence the method provides only less information on the distribution of the residual stresses in the material.

The present work describes a new approach which allows the measurement of strains in several plains while the stress state is changed. Therefore, a measurement sensor is necessary which can be embedded into the specimen. In order to investigate reinforced or pure plastics, fibre Bragg grating sensors are most suitable because these are glass fibres which are also embedded as reinforcing fibres. By the application of fibre Bragg grating sensors the experimental set-up has to be changed in a way that strains are measured in tangential and not radial direction as typically. This alternation of the set-up is necessary because of technological aspects. The usage of the new approach requires an adapted experimental set-up as well as a new method to calculate the residual stresses based on the tangential strains of different plains.

References

[1] J. Mathar. Determination of initial stresses by measuring the deformation around drilled holes. Trans. ASME 56 (1934), 249–259.

[2] ASTM E 827-08. Standard Test Method for Determing Residual Stresses by the Hole-Drilling Strain-Gage Method (2008).

Kink banding in laminated composite structures

Christina Völlmecke
Technische Universität Berlin, Institut für Mechanik,
Stabilität und Versagen funktionsoptimierter Strukturen

The formation of kink bands in fibre-reinforced composite components can yield a significant reduction of the stiffness of a structure. Subsequently it may also lead to further failure modes such as matrix damage, delamination *etc.* [1].
Thus herein an analytical model is proposed for kink band deformations occurring in laminated composites whilst the underlying formulations are based on geometric and potential energy principles. Therefore the total potential energy $V(Q_i, \Delta)$ of the system is derived from the strain energy $U(Q_i)$ minus the work done of the load $P\Delta(Q_i)$. The equilibrium paths are subsequently derived by minimizing the energy with respect to the generalized coordinates Q_i.
The model originates from a previous pilot study on kink banding in unidirectional laminae [2]. Earlier formulations are adapted and enhanced such that nonlinear material behaviour and imperfections stemming from initial fibre waviness can be allowed for [3, 4].
Furthermore the model is significantly extended to simulate a symmetric, multi-directional laminate lay-up whilst the kink band is assumed to occur in the $0°$ laminae on the neutral axis only [5]. The surrounding, outer layers are modelled with homogenized material properties derived using Classical Laminate Theory and a Rayleigh-Ritz approach is utilized to approximate the displacement of the remaining layers.
Parametric investigations are undertaken and the results are in very good agreement with the literature. The proposed analytical approach thus encourages further studies on systematic, phenomenological investigations of kink banding in composite structures using energy minimization principles.

References

[1] Prabhakar, P., Waas, A. M. Interaction between kinking and splitting in the compressive failure of unidirectional fiber reinforced laminated composites. Compos. Struct. 98 (2013), 85–92.

[2] Wadee, M. A., Völlmecke, C., Haley, J. F., Yiatros, S. Geometric modeling of kink banding in laminated structures. Philos. Trans. R. Soc. Lond., A 370 (2012), 1827–1849.

[3] Zidek, R. A. E., Völlmecke, C. On the influence of material non-linearities in geometric modeling of kink band instabilities in unidirectional fiber composites. Int. J. Nonlinear Mech. 62 (2014), 23–32.

[4] Zidek, R. A. E., Völlmecke, C. Analytical studies on the imperfection sensitivity and on the kink band inclination angle of unidirectional fiber composites. Compos. Part A-Appl. S. 64 (2014), 177–184.

[5] Völlmecke, C., Zidek, R. A. E. Geometric modelling of kink banding in multidirectional composites. J. Eng. Math. (2014), accepted manuscript.

Interlaminar stress recovery for arbitrarily curved laminated shells

Robert Winkler[1], Herbert Haller[2], Johannes Gerstmayr[1]
[1]Institute of Mechatronics, Leopold-Franzens-Universität Innsbruck, Austria
[2]Intales GmbH, Natters, Austria

Interlaminar shear (ILS) and normal (ILN) stress components play an important role in turn of strength analyses of laminated composite structures, in particular, if they involve strongly curved components such as stiffening members like channel section beams. Conventional shell theories and corresponding finite element formulations do not provide accurate distributions of stress components along the thickness of laminated composites. Therefore, post processing techniques have been developed to calculate the stress distribution from equilibrium and boundary conditions based on the membrane strains obtained from a conventional finite element analysis, see [1], [2], and [3], for plate bending problems. A recovery technique for ILN strains of curved shells has been presented in Ref. [4].

Here a recovery technique for both, ILN and ILS strains, valid for arbitrarily curved shells is described. Therefore, the shell reference surface is parametrized by curvilinear coordinates (ξ^α), $\alpha = 1, 2$. To parametrize the shell body, a third coordinate, ξ^3, is introduced which measures the distance perpendicular to the reference surface. Provided that the in-plane stress components $\sigma^{\alpha\beta}(\xi^3)$ are known, the ILS and INS components $\sigma^{\alpha 3}(\xi^3)$ and $\sigma^{33}(\xi^3)$, respectively, can be obtained by numerical integration from Cauchy's equilibrium conditions. For this purpose the equilibrium conditions are expressed in curvilinear coordinates (ξ^α, ξ^3). Consequently, they involve the covariant derivatives of stress components and the stress components themselves. Projecting these vector equations to the reference surface, two equations are obtained determining $\sigma^{\alpha 3}(\xi^3)$. Projection onto the transverse (ξ^3) direction yields the governing equation for $\sigma^{33}(\xi^3)$. If the required functions $\sigma^{\alpha\beta}(\xi^3)$ are obtained from a conventional shell finite element (FE) model, special attention has to be payed to the determination of the covariant in-plane derivatives $\sigma^{\alpha\beta}|_\gamma$. Under certain assumptions, a well-known procedure allows to determine the latter approximately from the shear forces. For plates, this procedure is elaborated in Ref. [1], e.g. Here we adopt this approach for curved shells and, in particular, if normal stress components become relevant. Further it is demonstrated how the zero-stress condition at the lower *and* at the upper boundary of the laminate can be fulfilled using a certain freedom in the choice of unspecified terms involving the in-plane derivatives $\sigma^{33}{}_{,\alpha}$ and $\sigma^{\beta 3}|_\alpha$. Due to the application of general curvilinear coordinates, no restrictions are imposed on how the curvature information is provided to the algorithm. In particular, there is no need to determine principal directions and curvatures even if the finite elements are arbitrarily oriented or distorted.

The procedure has been verified numerically comparing the results obtained from a conventional shell FE analysis via our postprocessing technique with a reference solution obtained from a three-dimensional (3D) FE analysis performed with Abaqus. For the shell analysis we apply a four node curved shell element. The curvature information is provided by the normal directions at the nodes. For the reference solution the laminate is modeled with the 27 node hexahedral continuum element C3D27. Thereby each ply is represented by a layer of elements. The example problems range from a curved beam bending test to a doubly curved shell model representing a curved channel section beam. Thereby an excellent coincidence of the results is observed for σ^{33} in general (including very low curvature radius-to-thickness ratios) and a good one for $\sigma^{\alpha 3}$ in the case of dominating shear forces. In the case of dominating bending moments the ILS stresses are relatively small and the results of the 3D analyses do no longer serve as a proper reference since they strongly depend on the exact type of the load introduction.

References

[1] R. Rolfes, K. Rohwer, Improved transverse shear stresses in composite finite elements based on first order shear deformation theory. Int. J. Numer. Meth. Engng. 40 (1997), 51–60.

[2] R. Rolfes, K. Rohwer, M. Ballerstaedt, Efficient linear transverse normal stress analysis of layered composite plates. Computers and Structures. 68 (1998), 643–652.

[3] M. Schürg, W. Wagner, F. Gruttmann. An enhanced FSDT model for the calculation of interlaminar shear stresses in composite plate structures. Computational Mechanics. 44 (2009), 765–776.

[4] R. Roos, G. Kress, P. Ermanni, A post-processing method for interlaminar normal stresses in doubly curved laminates. Composite Structures 81 (2007), 463–470.

Optimization of two-layered steel/aluminum hollow cylinders under combined load

Tunc Apatay, Werner Mack
Department of Mechanical Engineering, Gazi University, Ankara, Turkey
Institute of Mechanics and Mechatronics, Vienna University of Technology, Vienna, Austria

Thick-walled long hollow cylinders are structural elements found frequently in various industrial fields like power engineering and chemical engineering, where pressure vessels, fluid conveying tubes or centrifuges, e.g., are widely used devices. Typically, these elements are loaded by a radial temperature gradient, by internal pressure or by rotation, and often also by a combination thereof. In some cases, it is advantageous not to use a homogeneous tube, but a layered one consisting of two or more materials; reasons for this may be, amongst others, different chemical and/or thermal requirements at the inner and outer surface, improved strength or the demand for a reduction of the weight of the device. While there exist many investigations of the stress fields in thermally loaded layered hollow cylinders for given material data and geometry (e.g.[1]-[3]), in particular for given ratios of the thicknesses of the layers, the problem of finding the optimum composition for given (maximum) combined load was rarely addressed, however (compare [4] and the related study of shrink-fitted vessels without rotation [5]).

Hence, subject of the present study is a two-layered tube under generalized plane strain subject to combinations of rotation, internal pressure, and elevated temperature at the inner surface. Since centrifugal forces are proportional to material density and radius, it is presupposed that the inner cylindrical layer consists of the heavier material, whereas the outer layer is made of a material with lower density. As criterion for the maximum allowable stresses the yield criterion by von Mises is applied, and the device is optimized with respect to its weight. The present investigation not only gives a comprehensive overview of the elastic limits of composite tubes of the above type (where plasticization may start at different radii) but also provides a straightforward procedure for determining the optimum composition. While particularly steel/aluminum composite hollow cylinders are discussed, the results nevertheless are applicable to any material combinations with similar ratios of the material properties.

References

[1] Y.-C. Yang, C.-K. Chen. Thermoelastic transient response of an infinitely long annular cylinder composed of two different materials. Int. J. Engng. Sci. 24 (1986), 569–581.

[2] Z.-Y. Lee, C.K. Chen, C.-I. Hung. Transient thermal stress analysis of multilayered hollow cylinder.Acta Mech. 151 (2001), 75–88.

[3] A.N. Eraslan. Thermally induced deformations of composite tubes subjected to a nonuniform heat source. J. Thermal Stresses 26 (2003), 167–193.

[4] J. Huang, Y. Lu, C. Shen. Thermal elastic-plastic limit analysis and optimal design for composite cylinders of ceramic/metal functionally graded materials. Mater. Sci. Forum 423-425 (2003), 681–686.

[5] G. Yuan, H. Liu, Z. Wang. Optimum design for shrink-fit multi-layer vessels under ultrahigh pressure using different materials. Chin. J. Mech. Eng. 23 (2010), 582–589.

Experimental investigations on PP-PE foil specimens

Carmen Sguazzo, Stefan Hartmann
Institute of Applied Mechanics, Clausthal University of Technology,
Clausthal-Zellerfeld, Germany

Multi-layered composite are adopted in numerous structures related to different engineering fields. When they are employed into the automotive field, characteristics as lightweight together with good formability are very important. Consequently, it becomes essential to investigate how each material contributes with its own mechanical characteristics to the behaviour of the final product. In the three layered composite, object of the present investigation, an internal polymer core is used in order to enhance the damping characteristics, while external steel layers improve its ductility. Particularly, the three-layered sandwich is a combination of external layers of non-alloy low carbon mild steel and a polymer core of Polypropylene and Polyethylene (PP-PE). In the present work, the experimental characterization of the PP-PE core is presented. The experiments, have been carried out within the large deformation range and have been monitored by means of a Digital Image Correlation system. The rate-dependence and the temperature-dependence have been investigated. The experimental work has been conducted in order to interpret the resulting physical evidence by means of the Continuum Mechanics approach. This will give the possibility to model the behaviour of the polymer core in an appropriate way with the respect to the process in which the material is used.

Stress concentration control in the problem of plane elasticity theory

Nana Odishelidze, Francisco Criado Aldeanueva, Francisco Criado,
Iv. Javakhishvili Tbilisi State University, Faculty of Exact and Natural Sciences
Malaga University,Campus El Ejido, Department of Applied Physics II
Malaga University,Campus Teatinos, Department of Statistics and Operational Research

In engineering practice, one of the important problem is the problem of stresses concentration investigation near the hole-contour. In the plate with a hole, at some point the tangential-normal stresses (in case of plane elasticity theory) and the tangential-normal moments (in case of bending of thin plates), can reach such values that cause destruction of plates or the formation of plastic zones near the hole.

It is proven that in case of infinite domains the minimum of maximum values of tangential-normal stresses (tangential-normal moments) will be obtainedon such holes, where these values maintain constant. Such holes are called the full-strength ones. These problems are considered in the [1], [2].

For finite domains the axis-symmetric and cycle symmetric problems of the plane theory of elasticity and plate bending with partially unknown boundaries are studied in [3], [4], [6], [7].

The paper addresses a problem of plane elasticity theory for a doubly connected domain S on the plane $z = x + iy$, which external boundary is a rhombus boundary, whose diagonals lie at the coordinate axes OX and OY, the internal boundary required full-strength hole, whose symmetric axes are the rhombus diagonals.

Let to every link of the broken line (outer boundary of the given body) apply the absolutely smooth rigid stamps with rectilinear bases which is displaced under the action of concentrated normally compressive forces P, applied to the stamp midpoints. There is no friction between the given elastic body and stamps.

Under the above assumptions, the tangential stresses $\tau_{ns} = 0$ are equal to zero along the entire boundary of the domain S and the normal displacements of every link of external boundary $v_n = v = const$ are the piecewise constant functions.

The most effective methods used for such problem investigation are the methods of analytical functions theory. On the basis of the well-known Kolosov-Muskelishvili's formulas [5], the shape of the unknown full-strength hole and the stress state of the body are determined. Using numerical analysis method the corresponding graphs are constructed.

References

[1] N. B. Banichuk. Optimization of elastic solids. (Russian) Nauka, Moscow, 1980.

[2] G. Cherepanov, *Inverse problem of plane theory of elastisity.* (Russian) Prikl. Mat. Mekh. No. 6, 38(1974), 963 –980.

[3] R. Bantsuri., *On one mixed problem of the plane theory of elasticity with a partially unknown boundary ,* Proc. A. Razmadze Math. Inst., Vol. 140, (2006), 9 –16.

[4] R. Bantsuri., *Solution of the mixed problem of plate bending for a multi-connected domain with partially unknown boundary in the presence of cyclic symmetr,* Proc. A. Razmadze Math. Inst.,Vol. 145, (2007), 9 –22.

[5] N. I. Muskhelishvili., *Some Basic problems of mathematical theory of elasticity,* (Russian), Nauka, Moscow 1966.

[6] N. Odishelidze, F. Criado-Aldeanueva, F. Criado, J. M. Sanchez. *On one contact problem of plane elasticity for a doubly connected domain: application to a hexagon.* Zeitschrift fï¿œr Angewandte Mathematik und Physik, Vol. 64, No.1,(2013), 193 –200.

[7] N. Odishelidze, F. Criado-Aldeanueva, J.M. Sanchez . *A mixed problem of plate bending for a regular octagon weakened with a required full-strength hole.* Acta Mechanica, vol. 224, No. 1, (2013), 183 –192.

Stress analyses of multi-layered composite pipes subjected to internal pressure

İsmail Yasin Sülü[1], Şemsettin Temiz[2]
[1]Inonu University, Department of Mechanical Engineering, Turkey
[2]Batman University, Department of Mechanical Engineering, Turkey

The stress analyses of four, six and eight layered composite pipes with different orientation angles, under internal pressure, was investigated. The code of a numerical model was created in ANSYS software for numerical analyses, and the numerical results of four layered composite pipes were confirmed by experimental results with different orientation angles. Each layer of composite pipes was modeled with the same characteristics. The problems were studied using a computational tool based on the Finite Element Method (FEM). Each layer of the composite pipes was examined with different orientation angles. Failure loads of four layered composite pipes were obtained from experimental tests and compared with the numerical results. The hoop and shear stresses were obtained numerically for each layer. Radial, tangential and axial stresses were determined in the radial direction of the composite pipes. The shear extension coupling was considered because the lay-up angles with $+\theta$ and -θ layers were in the different radii

A model reduction approach for hyperelastic materials based on Proper Orthogonal Decomposition

André Hürkamp, Michael Kaliske
Institute for Structural Analysis, TU Dresden

Computational models in all fields of engineering applications tend to get more detailed and complex. The usage of nonlinear Finite Element (FE) methods is well established in engineering problems and it has been proven that it provides reliable results. Nevertheless, for complex models a lot of time and computational resources are required. Especially when parameter studies are carried out, similar problems are solved repetitively. For solving such kind of problems more efficiently, sophisticated model reduction techniques are needed. Those techniques are even essential, when we deal with high dimensional problems, as in the field of uncertainty quantification, or with real time simulations in the field of surgery training and support.

By the Proper Orthogonal Decomposition (POD), the most relevant information of a set of data is extracted. For example, with the aid of the POD we obtain singular values and corresponding base vectors from a matrix of snapshots, e.g. the FE solution of the displacement field calculated at several instants of time. Hence, using only base vectors corresponding to the largest singular values leads to a subspace of the problem with a lower dimension than the original one. This procedure is similar to a principal component analysis or Karhunen-Loève decomposition. Once the subspace for a certain type of problem is identified, it is possible to solve similar problems with a reduced number of degrees of freedom much smaller than for the original problem. Furthermore, we are able to compute time and space functions from the subspace, so that e.g. the displacement over a certain time domain can be approximated by a series expansion of a few functions depending only on time and functions depending only on space. Hence, it is possible to reproduce complete FE solutions only by matrix multiplication.

In this contribution, we present a novel model reduction approach for hyperelastic materials based on the separation of time and space dependent functions. Several computational examples involving large deformations, complex geometries and different constitutive models are investigated and the results obtained from the reduced computation are compared with detailed FE computations.

XFEM for Deformation Theory of Plasticity

Samir Omerović, Thomas-Peter Fries
Graz University of Technology

In plasticity theory, in general two different kinds of discontinuities in the displacement field appear. There are weak discontinuities in the transition zone between elastic and elastoplastic behaviour (plastic interface), as shown in [1]. For perfectly plastic behaviour, even strong discontinuities can occur, therefore corresponding to a jump in the displacement field [2]. The location of the interface is not known a priori, but constitues an additional unknown for the boundary value problem. This introduces additional difficulties when discretizing the weak form as the plastic interface is within elements. This poses problems in terms of the convergence properties, the convergece rate changes drastically, and also the absolute error increases.

Up to now different strategies were applied to improve the performance of the numerical methods. This includes e.g. adaption of the element size (h-refinement), increasing the approximation space over each element (p-refinement) [3], reallocation of the nodes (r-refinement) and also combinations of the preceding (e.g. hp and rp-extensions [1]). However, these strategies improve the accuracy by using extensive remeshing procedures, that are numerically expensive and/or time consuming.

We propose to enrich the approximation space in the presence of weak discontinuities at the plastic interface with modified abs functions [4]. As commonly done within the framework of the Extended Finite Element Method (XFEM), the plastic interface is tracked using the level set method [5]. It is stressed that no remeshing whatsoever is necessary. An other approach is also shown using the Heaviside enrichment and enforcing continuity using Lagrange multipliers. The numerical examples confirm that with minimal effort, the absolute error is significantly reduced. Using the same enrichment, it is also possible to model strong discontinuities (occuring for example in slip lines) within the same framework.

References

[1] V. Nübel, A. Düster, E. Rank. An rp-adaptive finite element method for the deformation theory of plasticity. Computational Mechanics 39(5) (2007), 557–574.

[2] C. Johnson, R. Scott. A Finite Element Method for Problems in Perfect Plasticity Using Discontinous Trial Functions. Nonlinear Finite Element Analysis in Structural Mechanics (1981), 307–324.

[3] A. Düster, Rank E. The p-Version of the Finite Element Method Compared to an Adaptive h-Version for the Deformation Theory of Plasticity. Int. J. Numer. Methods Eng. 52 (2001), 673–703.

[4] N. Moës, M. Cloirec, P. Cartraud, J.-F. Remacle. A computational approach to handle complex microstructure geometries. Comput. Method Appl. M. 192 (2003), 3163–3177.

[5] T.-P. Fries, T. Belytschko The extended/generalized finite element method: An overview of the method and its applications. Int. J. Numer. Meth. Engng. 84(3) (2010), 253–304.

Advanced FE-analysis of metal-to-metal seals considering fluid pressure penetration at two scales

Yevgen Gorash, William Dempster, William D. Nicholls and Robert Hamilton
Department of Mechanical and Aerospace Engineering, University of Strathclyde,
75 Montrose Street, Glasgow G1 1XJ, Scotland, UK

This numerical study investigates the behaviour of the contact faces in the metal-to-metal seal of a typical pressure relief valve [1] in the commercial FE-package ANSYS. The valve geometry is simplified to an axisymmetric problem, which comprises a simple representative geometry consisting of only three components. A cylindrical nozzle, which has a valve seat on top, contacts with a disk, which is preloaded by a compressed linear spring. All the components are made of the steel AISI type 316N(L) [2] defined using the multilinear kinematic hardening material model based on monotonic and cyclic tests at 20°C. Analysis considerations include the effects of the Fluid Pressure Penetration (FPP) across the valve seat which exists at two different scales. In-service observations show that there is certain limited fluid leakage through the valve seat at operational pressures about 90% of the set pressure [3], which is caused by the fluid penetrating into surface asperities at the microscale [4]. At the macroscale, non-linear FE-analysis using the FPP technique available in ANSYS revealed that there is also a limited amount of fluid penetrating into gap, which is caused primarily by the global plastic deformation of the valve seat. Plastic strain is assessed using multilinear kinematic hardening model based on the monotonic stress-strain curve obtained from a monotonic tensile test and the cyclic stress-strain curve obtained from a number of tests with stabilised cyclic response [5]. Accurate prediction of the fluid pressure profile over the valve seat is addressed in this study by considering the FPP interaction on both scales. The shape of this pressure profile introduces an additional component of the spring force, which needs to be considered to provide a reliable sealing. Moreover, the analysis showed that the evolution of the profile, which is caused by the isotropic softening of the material, is significant during the cyclic operation of the valve. It was also demonstrated that the type of working fluid affects the shape of the pressure profile in the multiscale. Finally, suggestions to improve the leakage tightness of the metal-to-metal seals are formulated.

References

[1] Malek, M.A., *Pressure Relief Devices: ASME and API Code Simplified.* McGraw-Hill: New York, USA, 2006.

[2] Gorash, Y., Altenbach, H. & Lvov, G., Modelling of high-temperature inelastic behaviour of the austenitic steel AISI type 316 using a continuum damage mechanics approach. *Journal of Strain Analysis*, 47(4), pp. 229-243, 2012.

[3] Hellemans, M., *The Safety Relief Valve Handbook: Design and Use of Process Safety Valves to ASME and International Codes and Standards.* Butterworth-Heinemann: Oxford, UK, 2009.

[4] Müller, H.K. & Nau, B.S., *Fluid Sealing Technology: Principles and Applications.* Marcel Dekker, Inc.: New York, USA, 1998.

[5] Chaboche, J.-L., Dang Van, K. & Cordier, G., Modelization of the strain memory effect on the cyclic hardening of 316 stainless steel. *Trans. 5th Int. Conf. on Structural Mechanics in Reactor Technology*, IASMiRT: Berlin, Germany, number L11/3 in SMiRT5, pp. 1-10, 1979.

Investigations on clamping effects with Die-Less-Hydroforming-Structures

Andreas Metzger, Daniel C. Ruff, Thomas Ummenhofer
Karlsruhe Institute of Technology, KIT Steel & Lightweight Structures, Research Center for Steel, Timber & Masonry

The development of objects, items and structures produced by a special forming technology called "Die-Less Hydroforming" is currently very popular among many designers as well as artists and is also focused by some researchers (e.g. see [1]). Within the scope of "Die-Less Hydroforming", two or more CNC-cut thin flat 2-D steel blanks with arbitrary geometry are seal-welded at their edges to generate a closed initial chamber for inflation by a medium such as air, water or oil. When inflating the initial structure, a transformation into a 3-D object is performed showing plastic deformations combined with local high plastic strain. No auxiliary die or punch is used, i.e. only the initial geometry, combined with the internal pressure, determines the shape of the resulting body, which also shows stability phenomena like wrinkling and buckling.

When using a special blank geometry with openings or cutouts ("pie slices") at the edges, it is possible to generate a clamping effect due to the deformation of the flat blanks during inflation because of the multidimensional forming process of the cutouts at the edge. The openings or cutouts at the edges transform to a 3-D shape and will close like a clamp.

As an example, we want to refer to a nice stool called "Puff", designed and built by the Israeli designer Moran Barmaper. The seating of this stool initially consists of two thin flat seal-welded blanks with special openings at the edge. Before inflating, wooden chair legs are inserted in the openings. By inflating, the seal-welded blanks transform into something like a 3-D-pillow and at the same time, the openings are closing and fixing the stool legs and a bearing hybrid stool is generated (c.f. [2]).

In this contribution, we want to present numerical and experimental results of a study of these clamping effects, investigated on tests with special Die-Less-Hydroforming-samples. We will focus on a special geometry that looks like a "Pac-Man". We investigate the closing and clamping effect of the "mouth" of the "Pac-Man" in practical inflating tests and compare the results with numerical forming simulations using LS-DYNA.
After finishing the ongoing investigations, we might be able to present some first technical application for the use of clamping effects of Die-Less-Hydroforming-Structures.

References

[1] A. Metzger, D.C. Ruff and T. Ummenhofer. FEM-Simulation of "Die-Less Hydroforming". Conference Contribution (2014), Presented at the 85th Annual Meeting of GAMM, Erlangen-Nürnberg, PAMM Publication in process

[2] "metal furniture inflated using a bike pump puff by moran barmaper", *Andrea Chin, www.designboom.com*, Oct 5, 2012 retrieved on Dec 19, 2014 from
http://www.designboom.com/design/metal-furniture-inFLated-using-a-bike-pump-puFF-by-moran-barmaper

On one contact problem of plane elasticity theory with partially unknown boundary

Magda Tsintsadze, Nana Odishelidze
Iv. Javakhishvili Tbilisi State University, Faculty of Exact and Natural Sciences, Georgia

Applications of elastic plates weakened with full-strength holes are of a great interest in several mechanical constructions (building practice, mechanical engineering, shipbuilding, aircraft construction, etc).

It's proven that in case of infinite domains the minimum of tangential normal stresses (tangential normal moments) maximal values will be obtained on such contours, where these values maintain constant. These holes are named full-strength holes. The solvability of these problems allow to control stress optimal distribution at the hole boundary via appropriate hole shape selection. These problems are considered in the [1],[2]. It has been proved that in frames of the same load, the weight of a plate weakened by a hole with full-strength contours is 40% less than the weight of the one weakened by a circular hole. [2].

For finite domains the above mentioned problems are studied in [3], [4],[5],[6]. The paper addresses a problem of plane elasticity theory for a doubly connected domain S on the plane $z = x + iy$, which external boundary is an isosceles trapezoid boundary; the internal boundary is required full-strength hole including the origin of coordinates. The trapezoid axis coincides with the OY axis. To every link of the given body broken line , the absolutely smooth rigid stamps with rectilinear bases are applied and they are under the action of P normally-compressive forces. There is no friction between the surface of given elastic body and stamps. Uniformly distributed normal stress is applied to the unknown full-strength contour. Tangential stresses along the entire boundary $\tau_{ns} = 0$ are equal to zero and normal displacements are the piecewise constants $v_n = v = const$. Linear segments are endowed with the boundary conditions of the third problem. The most effective methods for studying these problems are the methods of analytical function theory. These problems are both mechanical and geometrical character since the shape of hole is required and the conformal mapping is used to define it. The formulas analogous to Kolosov-Muskhelishvili [7]. are used for investigation of these problems. The unknown full-strength contour and stressed state of the body are determined.

References

[1] N. B. Banichuk, *Optimization of elastic solids.* (Russian) Nauka, Moscow, 1980;

[2] G. Cherepanov, *Inverse problem of plane theory of elastisity.* (Russian) Prikl. Mat. Mekh. No. 6, 38(1974),963–980.

[3] R. Bantsuri., *On one mixed problem of the plane theory of elasticity with a partially unknown boundary ,* Proc. A. Razmadze Math. Inst., Vol. 140, (2006),9 –16.

[4] N. Odishelidze and F. Criado-Aldeanueva., *A mixed problem of plane elasticity for a domain with a partially unknown boundary,*International Applied Mechanics, Vol. 42, No. 3, (2006),342 –349.

[5] N. Odishelidze, F. Criado-Aldeanueva., *Some axially symmetric problems of the theory of plane elasticity with partially unknown boundaries ,* Acta Mech., Vol. 199, (2008),227 –240.

[6] N. Odishelidze, F. Criado-Aldeanueva, F. Criado, J. M. Sanchez. *On one contact problem of plane elasticity for a doubly connected domain: application to a hexagon.* Zeitschrift fï¿œr Angewandte Mathematik und Physik, (2013), Vol. 64, No.1, 193 –200.

[7] N. I. Muskhelishvili., *Some Basic problems of mathematical theory of elasticity,* (Russian), Nauka, Moscow 1966.

Frequency veering and mode degeneration of a rectangular disc

F. Brouet, J. Twiefel, J. Wallaschek
Institut für Dynamik und Schwingungen, Leibniz Universität Hannover, Hannover, Germany

The modal properties of an elementary rectangular disc with four slots, aimed to be driven in the longitudinal mode, are investigated. The eigenfrequencies are plotted as a function of the geometrical aspect ratio height-width of the disc. Curve veering effects occur in some regions of close frequencies and the modal properties are strongly perturbed.

The veering phenomenon in dynamic systems is well known for many applications like turbine blades, bridges with aeroelastic effects or rotating cantilever beams. In some regions the frequency loci approach each other closely and suddenly veer away again, each one taking on the trajectory of the other. Then the mode shapes are swapped in a smooth transition.

By deriving the eigenvalue sensitivity with respect to a variation parameter, Fox and Kapoor [1] calculated the eigenvector sensitivity for the undamped dynamic eigenvalue problem. J.L. du Bois, S. Adhikari, N.A.J. Lieven [2] introduced indices and criterions to quantify the veering intensity.

The eigenvalue veering phenomenon is often associated with irregularities or mistuning because it occurs when this mistuning is of the same order or greater than the coupling, see Afolabi [3]. Yang and Griffin [4] projected the mode shapes in a new basis of nominal modes by defining a coefficient matrix of participation factors.

In this contribution, one veering zone affecting the longitudinal mode and one spurious mode is investigated. The coupling between both modes is estimated at one critical aspect ratio and the longitudinal mode can be retrieved after tuning the disc geometry.

The spurious mode can be eliminated by coupling both end edges of the disc with a rigid spring in the simulation and the longitudinal mode is retrieved in an artificial way. The new pseudo-cylic state defines the basis of nominal modes and the mistuning of the system can be seen as the removal of the coupling spring. Then the shapes of both localized longitudinal and spurious modes can be spanned by the new created basis containing the unperturbed longitudinal mode. Since the nominal modes and the mistuning matrices are now determined, the modal coupling can be calculated.

The modal coupling between the eigenvectors of the nominal basis can be reduced thanks to several methods like linearized sensitivity studies or minimizing functions, by adapting the disc design. The resulting modes (without the rigid coupling spring) remain uncoupled and the longitudinal mode is retrieved in the critical region.

References

[1] R.L. Fox, M.P. Kapoor. Rates of changes of eigenvalues and eigenvectors. AIAA Journal (1968), Vol. 6(12) 2426–2429.

[2] J.L. du Bois, S. Adhikari, N.A.J. Lieven. On the quantification of eigenvalue curve veering: a veering index. Journal of Applied Mechanics (2011), Vol. 78.

[3] D. Afolabi. Modal interaction in linear dynamic systems near degenerates modes. National Aeronautics and Space Administration (1991).

[4] M.T. Yang, J.H. Griffin. A normalized modal eigenvalues approach for resolving modal interaction (1997), Vol. 119 647–650.