A FINITE ELEMENT FORMULATION IN BOUNDARY REPRESENTATION FOR THE ANALYSIS OF HETEROGENEOUS STRUCTURES

S. Klinkel[1] and R. Reichel[1]

[1]*Chair of Structural Analysis and Dynamics, RWTH-Aachen University, Mies-van-der-Rohe-Str. 1, Aachen, Germany*

e-mail: klinkel@lbb.rwth-aachen.de

1. Introduction

The contribution is concerned with a numerical element formulation in boundary representation. It results in a polynomial element description with an arbitrary number of nodes on the boundary. Scaling the boundary description determines the interior domain. The scaling approach is adopted form the so-called scaled boundary finite element method (SB-FEM). The SB-FEM is a semi-analytical formulation to analyze problems in linear elasticity, see [1]. Within this method, the basic idea is to scale the boundary with respect to a scaling center. The boundary denoted as circumferential direction and the scaling direction span the parameter space. In the present approach, interpolations in scaling direction and circumferential direction are introduced. The interpolation in circumferential direction is independent of the scaling direction. The formulation is suitable to analyze problems in non-linear solid mechanics. The displacement degrees of freedom are located at the nodes on the boundary and in the interior element domain. The degrees of freedom located at the interior domain are eliminated by static condensation, which leads to a polygonal finite element formulation with an arbitrary number of nodes on the boundary. The element formulation allows per definition for Voronoi meshes and quadtree mesh generation, see [2]. The present approach considers a quadtree mesh to model the heterogeneous structure. The heterogeneous structure includes voids and inclusions located in the interior domain. Curved intersection boundaries are modeled by using a special trimming algorithm. It avoids also staircase approximation of curved boundaries. Numerical examples give rise to the performance of the present approach in comparison to other polygonal element formulations, like the virtual element method (VEM). Some benchmark tests show the capability of the element formulation and a comparison to standard and mixed element formulations is presented.

2. Numerical example

The example is concerned with an heterogenous version of the Cook's membrane. The membrane is bounded by the polygon with the (x,y) tuple: (0mm; 0mm); (48mm; 44mm); (48mm; 60mm); (0mm; 44mm), see Fig.1.

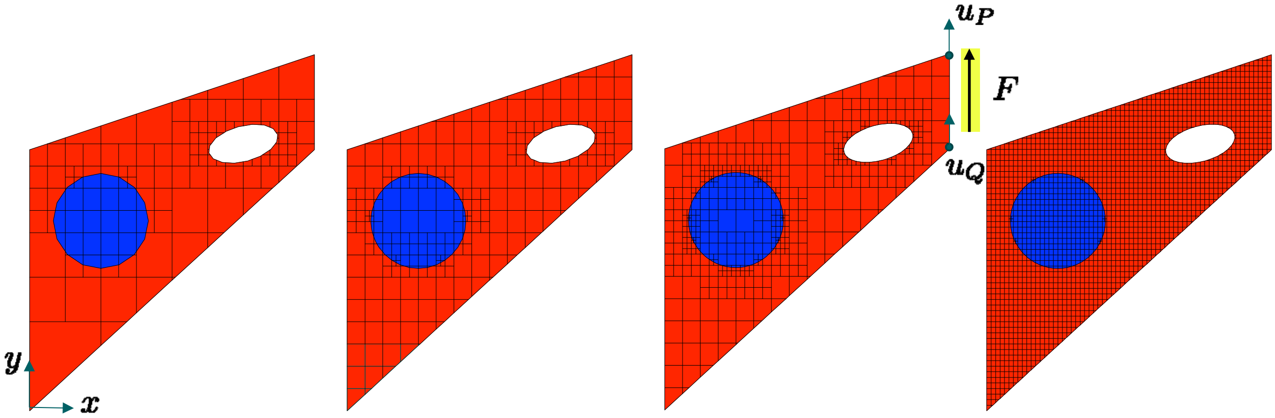

Figure 1: The Cook's membrane with an inclusion and a void. The color red denotes the Neo-Hooke material, blue the St. Venant-Kirchhoff material. The structure is modeled by different quadtree meshes.

The material is described by an Neo-Hook material model, defined with the strain energy function $\Psi = \frac{\mu}{2}(\mathrm{tr}\boldsymbol{F}^{\mathrm{T}}\boldsymbol{F} - 1) - \mu\ln(\mathrm{J}) + \frac{\Lambda}{4}(\mathrm{J}^2 - 1 - 2\ln(\mathrm{J}))$ with the Lame constants $\Lambda = 400889\mathrm{N/mm}^2$ and $\mu =$

80.1938N/mm^2. Here $\boldsymbol{F}$ denotes the deformation gradient and J is its determinant. The boundary of the inclusion is described by a circle with a radius of 8mm and a center point defined by the co-ordinates (12mm, 23mm). The material behavior of the inclusion is determined by the St. Venant-Kirchhoff material with the Young's modulus $E = 2400$N/mm^2 and the Poisson's ratio $\nu = 0.3$. The void is defined by an ellipse with a center located at (36mm, 45mm). The principal axis are rotated with 15 anticlockwise. The largest and smallest radii are defined by 8mm, 3mmm.

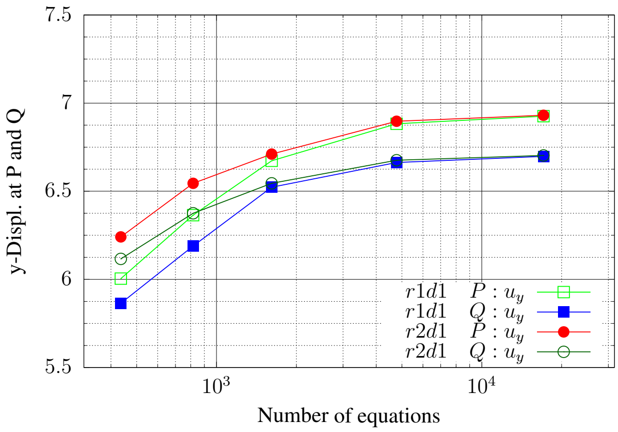

Figure 2: The deflection versus the number of equations (degrees of freedom) for the meshes shown in Fig. 1.

The left side of the membrane is clamped and a force of 100N is applied uniformly to the left hand side. The load is applied in ten steps. The vertical displacement of the upper right corner P and the lower right corner Q are calculated. Therefore the heterogeneous structure is modeled by the quadtree meshing. The meshes with four steps of refinement are shown in Fig.1. Fig.2 shows the deflection versus the number of equations. A nice convergence with mesh refinement is observed. The deformed structure at F=100N is depicted in Fig.3.

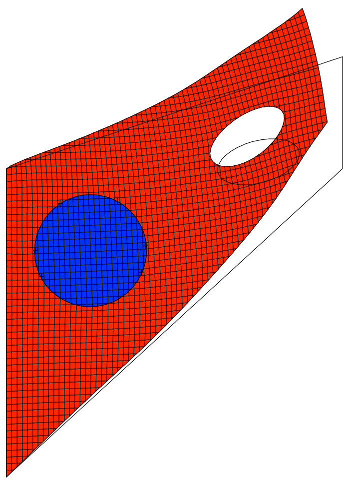

Figure 3: Deformed configuration at F=100N.

Acknowledgments The authors acknowledge the financial support of Exploratory Research Space by RWTH Aachen University.

References

[1] C. Song and J. P. Wolf. The scaled boundary finite-element method-alias consistent infinitesimal finite-element cell method-for elastodynamics. *Comput. Meth. Appl. Mech. Engrg.*, 147:329–355, 1997.

[2] M. Yerry and M. Shephard. A modified quadtree approach to finite element mesh generation. *IEEE Comput. Graph. Appl.*, 3(1):39–46, January 1983.