Intelligent Initial Finite Element Mesh Generation for Solutions of 2D Problems

Lina VASILIAUSKIENĖ, Romualdas BAUŠYS

Fundamental Science Faculty, Vilnius Gediminas Technical University
Saulėtekio 11, 2040 Vilnius, Lithuania
e-mail: lina.vasiliauskiene@fm.vtu.lt

Abstract. The new strategy for non-uniform initial FE mesh generation is presented in this paper. The main focus is set to a priori procedures that define the sizing function independent on the mesh generation algorithm. The sizing function used by the mesh generation algorithm is established by control sphere and control space concepts and fully controls mesh gradation in the complex 2D problem domains.

Key words: mesh generation, element size, control space.

1. Introduction

The advent of modern computer technologies provided a powerful tool in numerical simulations for a range of partial differential equations. One of the most frequently used methods for the discretization of physical domain is Finite Element Method (FEM). One of the main concerns in finite element analysis is the adequacy of the finite element mesh. Since the quality of the finite element approximated solutions directly depends on the quality of meshes, an additional process to improve the quality of meshes is necessary for reliable finite element approximations. In order to perform a reliable finite element simulation a number of researches have made efforts to develop an adaptive finite element analysis method which integrates the finite element analysis with error estimation and automatic mesh modification (Baušys, 1999; Lee, 1993; Stupak, 2000). Traditionally adaptive mesh generation process is started from coarse mesh, which gives large discretization error levels and takes a lot of iterations to get a desired final mesh. This happens because the quality of generated FE mesh is judged only from the element shape. With the growing awareness of adaptive finite element strategies much effort is devoted to improve the speed and quality of meshes (Lee, 1999; Löhner, 1996). Usually generators do not account the boundary conditions or material properties of the domain and start from the mesh, which requires more steps for the convergence of desired accuracy and the process is the more time consuming.

The aim of this article is to provide an algorithm for the generation of two dimensional (2D) initial (before the finite element analysis is started) mesh, which will be more refined around the critical regions (singularities, re-entrant corners) of the problem domain.

When incorporated into adaptive FE analysis and compared with traditional approach, the proposed method, which starts the solution process with near optimal initial mesh, will significantly reduce the number of iterations to get the final mesh. The main focus is set to the grid generation of the planar multimaterial and multidomain systems by the Advancing Front technique. The concepts of control space and control sphere are applied in order to control mesh size. Mesh generation examples will demonstrate that the proposed method can discretize the problem domain into almost optimal initial mesh.

2. General Mesh Generation Aspects

The method used for mesh generation can greatly affect the quality of the resulting mesh. Usually the geometry and physical problems of the domain direct the user which method to apply. Real 2D problems involve the complex topology, distribution of the boundary conditions. Such situation requires to use the automatic mesh generator and reduce user influence to this process as much as possible. The Delaunay triangulation and Advancing front method are the most popular mesh generation methods that can be used for the adaptive FE mesh strategies.

Both methods involve the creation of points and the relevant connectivity's. This is usually achieved through different stages that can be summarized as follows:

Step 1. Definition of the domain boundaries;
Step 2. Specification of an element size distribution function;
Step 3. Generation of a mesh respecting the domain boundaries;
Step 4. Mesh quality enchantment (diagonal swapping, mesh smoothing, node reposition).

In general, good quality meshes cannot be obtained directly from the meshing techniques. An additional step is required to optimize the mesh with respect to the element shape. To describe all model features without generating huge numbers of elements, large transitions in element sizes may be required. Many authors have described the use of some form of element size control in the literature (Owen, 1992; Owen, 2000). This is often done in the context of the presentation of the specific meshing algorithm. A background mesh, made from the set of vertices selected from the geometry, is the most commonly used to define an element sizing function. For the Delaunay triangulation the sizing information at all vertices are provided directly from the user input or from the heuristic criteria based on surface curvature, feature size, or physically based phenomena such as boundary layers or error norms from the previous finite element solution. The recent work by Borouchaki *et al.* (1998) uses the modified form of the Delaunay algorithm. In this work a continuous field of element sizing tensors or metric is interpolated throughout planar or three-dimensional domain. Metric ability to take into account the error norms from the previous solution is also demonstrated. For the Advancing Front technique Peraire (Peraire *et al.*, 1987; Peraire, 1988) introduce the concept of the background mesh in the context of the adaptive re-meshing. They describe the background mesh consisting of only few elements or take the actual finite element mesh which has

been previously used for analysis. Interpolation of the additional mesh parameters is also discussed. Numerical improvements of mesh sizing function for the AFT were proposed first by R. Löhner and P. Parikh (1988), more recently by J. Frykestig (1994). Both generators (Delaunay triangulation and Advancing front technique) provide good quality meshes, but using Dealunay triangulation ill-shaped elements can be constructed along the boundaries. The Advancing Front technique tends to produce high quality elements and nicely graded meshes in whole region. In contrast to other techniques, boundary integrity is always preserved, as the discretization of the domain boundary constitutes the initial front, which is not the case for some other mesh generation methods.

3. Sub-optimal Initial Mesh Generation

The efficiency of the adaptive finite element strategies straightforward depends on the number of iterations until optimal mesh is constructed. Traditionally mesh generation process starts with a coarse user-constructed background mesh and mesh sizing function is defined manually by giving its value for each element of the background mesh and is judged only from element shape without any information about boundary or loading conditions. During the finite element meshing process the target element size at the new point is defined commonly from a linear interpolation. After that a corresponding solution is computed and the discretization error estimate analysis is performed so as to redefine mesh sizing function from a posteriori error estimate. This process is the most time consuming and requires a lot of steps for the convergence to the desired accuracy. Some weakness has been noted with the linear interpolation method also (Owen, 2000). Poor results can arise when the triangles of the background mesh are tessellated and skinny. As a result, abrupt changes in element size are common resulting in less than desirable element quality and transitions. To overcome these difficulties either the more perfect initial mesh should be taken as a covering triangulation Δ or the linear interpolation method should be changed.

The best mesh for a given finite element analysis problem can be defined as compromise between the need for accurate results ant the desire for the computational efficiency. Accuracy can be equated with the size and number of elements in the mesh. As element size decreases, the geometry of the problem region is more closely approximated. However, the increased number of elements causes an increase of the cost of the computational expenses. The compromise between accuracy and efficiency is usually achieved by grading the mesh, but in this place we have one problem – the solution gradients are unknown in the first iteration step of the adaptivity strategies and a tools of estimating mesh density requirements is needed before the automatically graded meshes can be produced. In order to overcome this difficulty, a new method for controlling mesh gradation is presented in this section. An automatic mesh generation procedure uses all information about the object geometry, boundary conditions and material distribution data to generate a priori mesh, which is more refined around regions, where high stress concentrations are expected. This approach incorporated into adaptive FE analysis will result in less time and less computational cost.
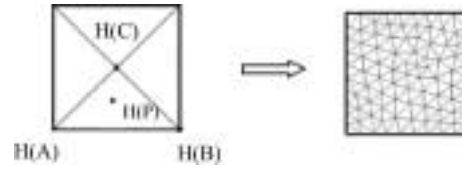
Fig. 1. Construction of control space: the size $H(P)$ at point P is obtained by linear interpolation between sizes $H(A)$, $H(B)$ and $H(C)$ at A, B and C respectively.

Mesh gradation control usually is performed using control space (Frey, 2000) notion.

DEFINITION 1. (Δ, H) is a control space for the mesh T of a given domain Ω if

- $\Omega \in \Delta$, where Δ covers the domain Ω,
- a mesh sizing function $H(P)$ is associated with every point $P \in \Delta$ and obtained from the linear interpolation

$$H(P) = \sum_{i=0}^{2} w_i H(P_i), \tag{1}$$

where w_i are the respective area or barycentric co-ordinates of P within the triangle.

The control space includes two related ingredients: first a covering triangulation Δ is defined, and then a function H is posed (Fig. 1). The proper selection of this pair allows to determine specific geometric or physical properties for mesh elements.

Linear interpolation problems and coarse covering triangulation obligates us to seek an improved mesh generation method. In order to overcome these difficulties the extended control space is implemented in the proposed method. In contrast to the Definition 1, the mesh size function is assigned not to the every point of Δ, but with the most critical region of the given domain Ω. This improvement allows us reflect more precisely all topological and physical domain features to the resulting finite element mesh and gives us more flexibility in mesh gradation control.

For the mesh size determination around each critical region we propose to use the control sphere concept:

DEFINITION 2. The sphere

$$S(P, R) = \{x| \, \|P - x\| = R\} \tag{2}$$

is a control sphere for any critical point P if radius R is defined as follows:

$$R = \sum_{i=1}^{n} (1.2)^i k h', \tag{3}$$

where h' is the largest desirable mesh size. $0 < k \leqslant 1$ is a factor for the controlling the smoothness of the node distribution and involves an information from loads, materials

and boundary. Low values of k result in smoother meshes, while higher values produce coarse meshes. n is the number of spheres from a critical point P, the number n can be determined as the integer part from the expression $\frac{d}{h'}$, where d is the diagonal length of the problem region. Radius R defines the spacing between any two consecutive spheres, generated from this point P.

In the same manner we can define control sphere for segment AB:

$$S(AB, R) = (1 - t)S(A, R) + tS(B, R), \quad t \in [0, 1], \tag{4}$$

where $S(A, R)$ and $S(B, R)$ are control spheres for the begin and end points of segment AB respectively.

The control sphere definition could be simply extended to the general case.

DEFINITION 3. The set of spheres

$$\left\{ S\bigl(\gamma(t), R\bigr) = \sum_{i=0}^{n} \varphi_i S(P_i, R) \right\} \tag{5}$$

is said to be the control sphere along the curve $\gamma(t) = \sum_{i=0}^{n} \varphi_i P_i$, if the centers of all spheres lies on this boundary curve and the radius are defined by Definition 2.

The last control sphere definition involves previously defined control spheres around points and segments can be used to define mesh sizes around any free form curve.

In the proposed approach control space is defined as follows (see Fig. 2).

DEFINITION 4. (Δ, S) is a control space for the mesh T of a given domain Ω if

- $\Omega \in \Delta$, where Δ covers the domain Ω,
- S is a control sphere, associated with every critical region of a given domain Ω.

In practice the control sphere for a point is used to define mesh size around singularities or re-entrant corners, and control sphere for a segment or a curve is useful for the boundary and loading conditions.
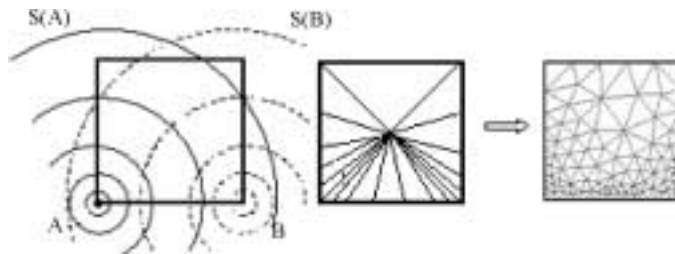

Fig. 2. Construction of control space: control spheres around critical points A and B, covering triangulation and the mesh.

Generating sets of spheres from each critical region, we can fully control smoothness of the node distribution. From the intersection of these spheres with region boundary we obtain a boundary segments, which are used as an initial front for Advancing Front triangulation procedure. The covering triangulation Δ is determined from this modified initial front and completes the control space (Δ, S) construction. Mesh generation procedure continues with the field point creation designed in accordance with the information encoded in this control space.

The control sphere concept allows us fully control mesh grading, but we still have one problem left. An additional check on mesh quality should be performed to avoid poorly shaped triangular elements. Engineers commonly access the shapes of triangles via aspect ratios, determined by dividing length of edges, altitudes, etc. Different measures (Babuska, 1976; Field, 2000) have been independently discovered numerous times and clearly identify perfect or degenerated mesh elements. In the proposed strategy the element shape measure, reported in (Bank, 1996; Bhatia, 1990) is implemented. This measure has a circular contours and is compatible with the proposed control sphere definition:

$$q(T_i) = \frac{4\sqrt{3}A_i}{l_{i1}^2 + l_{i2}^2 + l_{i3}^2},\tag{6}$$

where T_i is the ith triangle of the mesh T, l_{i1}, l_{i2} and l_{i3} are the edges of the triangle T_i, A_i – area of the triangle T_i. This measure allows us to eliminate ill shaped elements and rapid changes of the resulting mesh size.

So, the pair (Δ, S) contains the global information related to different aspects: geometry of the domain, material properties and loading conditions and allows us to construct sub-optimal initial mesh, which is more refined around areas where high solution gradients are expected. In such case adaptive FE strategy is started with almost optimal initial mesh and results in lower levels and better estimates of errors and in less iterations number.

4. Intelligent Initial Mesh Generation Procedure

In this section we present an algorithm, which performs good quality sub-optimal initial mesh for a given 2D problem. This algorithm starts working from the analysis part, which prepares data for the AFT procedure, i.e., decompose initial domain Ω into more simple substructures, then constructs control spaces from the geometry, boundary conditions or material properties encoded in these substructures. After that the sub-optimal initial meshes are generated for each part separately and general mesh is obtained combining these generated mesh parts.

The main algorithm steps can be summarized as follows:

Step 1. Preliminary definitions: object geometry, boundary conditions and material data input.

Step 2. Analysis part:

(a) *Object geometry check*: analysing all consecutive geometric feature or primitive (line, segment, arc, etc.) identify 'candidate' regions (points, segments, curves), that could be detected as critical. This includes regions with holes, notches, cracks or re-entrant points, in which significant stress gradient may be developed. Form a list $\{CR_1, CR_2, \ldots, CR_n\}$, where CR_i identifies 'candidate' critical region and n is the total number of these regions.

(b) *Boundary condition analysis*: check all candidate critical regions CR_i on load influence, remove CR_i from a list if there is no load, which produce any stress concentration to this critical region.

(c) *Domain subdivision*: decompose the original structure (or domain) into several substructures (or standart cases) for which an aproximate stress calculations can be performed. Each substructure contains only one critical region. Form a list $\big\{\{D_1, CR_1\}, \{D_2, CR_2\}, \ldots, \{D_m, CR_m\}\big\}$, where D_i identifies current substructe and CR_i is the corresponding critical region for this D_i, $m \leqslant n$ the number of realy critical regions.

(d) *Control space definition*: using Definition 4, determine control spaces for each substructure D_i. Form a list $\big\{\{D_1, CR_1, CS_1\}, \{D_2, CR_2, CS_2\}, \ldots, \{D_m, CR_m, CS_m\}\big\}$, where control space CS_i determines mesh size distribution in domain part D_i and incorporates information about critical region CR_i.

Step 3. Mesh generation: using Advancing Front Technique, perform sub-optimal initial mesh T_i for each substructure D_i:

(a) Using information encoded in control space CS_i, define the initial front F_i for the AFT triangulation.

(b) Analyze front F_i: select the front entity f (based on a specific criterion). Create an optimal point P_{opt} based on the entity. Determine whether a mesh vertex V exists that should be used instead of P_{opt}. If such point exists, set V to P_{opt}. Form a new element K with f and P_{opt}. Check for element intersection, element size, to validate the above choice.

(c) If front is not empty, return to (b).

Step 4. General mesh generation: from the obtained set of initial meshes $\{T_1, T_2, \ldots, T_n\}$ determine the general initial mesh T, which covers initial domain Ω.

The general framework of intelligent initial mesh generation procedure is shown is shown in Fig. 3.

5. Numerical Examples

Two mesh generation examples are given to test and demonstrate the proposed mesh generation scheme. User interaction is limited to providing the necessary object description,
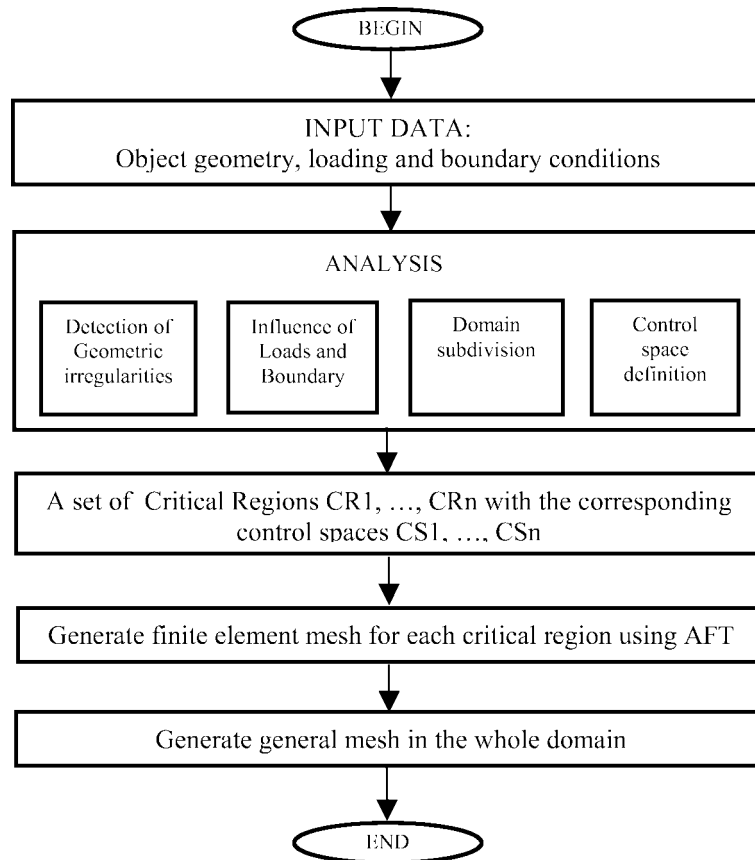
Fig. 3. Outline of the algorithm.

boundary and loading conditions. To illustrate the mesh generation possibilities two 2D domains with different topologies and complexity are analyzed. For each domain pure triangular and quadrilateral meshes are generated. The mesh generator uses extended control space notion to generate a priori sub-optimal initial mesh. Element size information is obtained from control spheres around point and a segment. For each problem we take different isotropic materials. First material is described by Young's modulus $E = 2.06 \cdot 10^5$ MPa and Poison's ratio $\vartheta = 0.3$. To define the second material we take $E = 2.2 \cdot 10^5$ MPa and $\vartheta = 0.28$. The essential boundary condition $p = 1000$ kN is set at the top of the both structures. The outline of dimensions and applied natural boundary conditions are shown in Fig. 4.

According procedure, proposed in Section 4, the sub-optimal initial meshes are performed for both structures A and B (see Fig. 5). For the structure (a) we have 490 elements and for structure (b) the total number of elements is 415. For the sake of comparison we generate uniform meshes for these structures with the same number of elements by the traditional meshing approach (see Fig. 6).
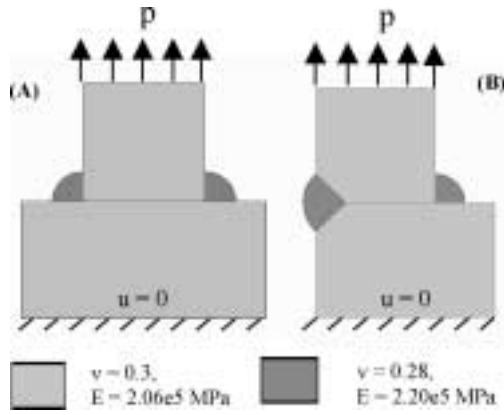
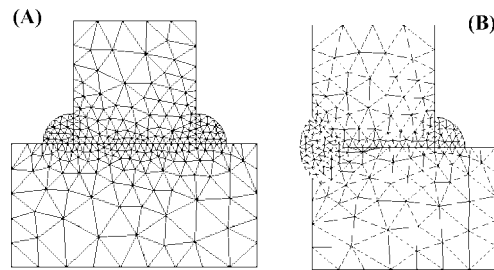Fig. 4. Structure A and structure B with different materials.

Fig. 5. Triangular sub-optimal meshes for structure A and B.
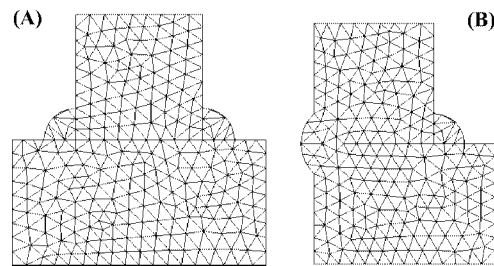

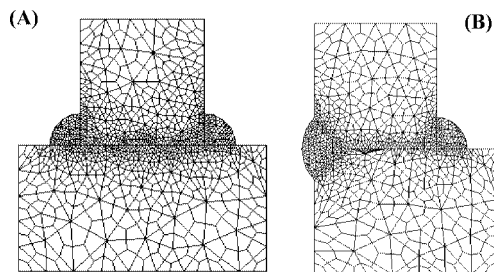Fig. 6. Triangular uniform meshes for structure A and B.

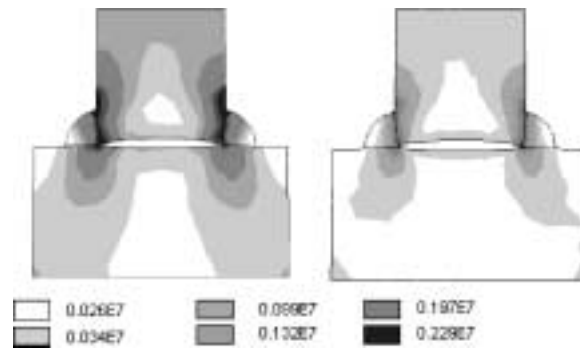Fig. 7. Quadrilateral sub-optimal meshes for structure A and B.

Fig. 8. Structure A: stress distribution for triangular uniform and sub-optimal meshes.
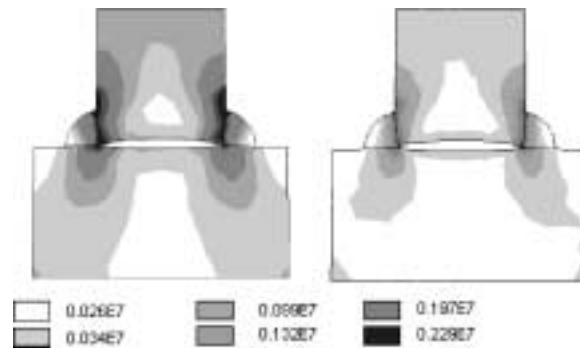

Fig. 9. Structure B: stress distribution for triangular uniform and sub-optimal meshes.

The corresponding solutions are computed and presented in Figs. 8–9. Comparing results from the sub-optimal and uniform meshes we detect that stress distribution areas are quite different for these two samples. The von Misses stresses, obtained from the sub-optimal mesh, are more concentrated around critical regions than the results, obtained from the uniform mesh. For example, for the structure A we obtain such stresses: $\sigma_{\max} = 0.197e7$ Pa by the sub-optimal mesh and $\sigma_{\max} = 0.227e7$ Pa by the uniform mesh. For the structure B the von Misses stresses are $\sigma_{\max} = 0.213e7$ Pa and $\sigma_{\max} = 0.250e7$ Pa by the sub-optimal and uniform meshes respectively. It means that in the first adaptive analysis step we have significantly reduced error and can expect that the optimization stage will take less number of iterations to complete mesh generation.

In order to study a full class of finite elements we do the same procedure with quadrilateral element mesh. The sub-optimal meshes are shown in Fig. 7, stress distribution for the structure A is shown in Fig. 10 and for the structure B in Fig. 11 respectively. As in the previous sample, the stress values are more concentrated for the sub-optimal meshes.

Presented examples show that proposed mesh generation procedure gives us more flexibility in mesh gradation control around areas with different topological incompatibilities and allows us to choose the most suitable mesh size for them.
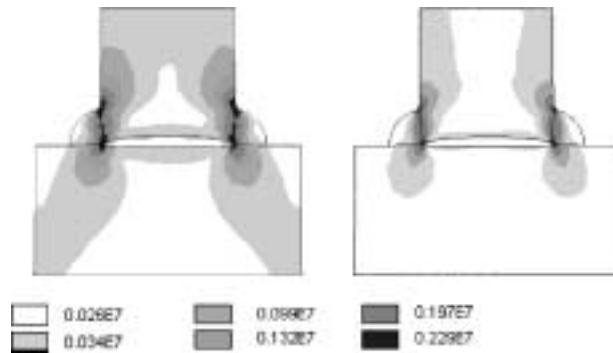
| | 0.026E7 | | 0.099E7 | | 0.197E7 |
| | 0.034E7 | | 0.132E7 | | 0.229E7 |

Fig. 10. Structure A: stress distribution for quadrilateral uniform and sub-optimal meshes.

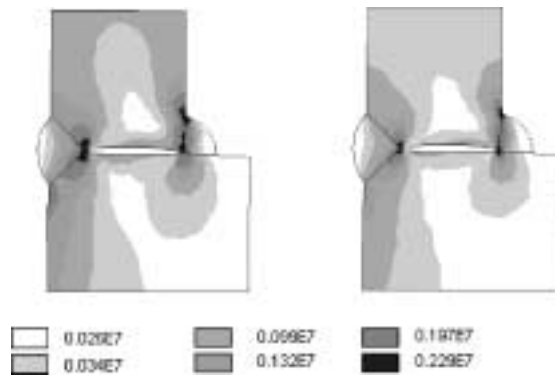| | 0.026E7 | | 0.099E7 | | 0.197E7 |
| | 0.034E7 | | 0.132E7 | | 0.229E7 |

Fig. 11. Structure B: stress distribution for quadrilateral uniform and sub-optimal meshes.

6. Conclusions

In this article a new strategy for non-uniform initial FE mesh generation have been presented. This strategy enables us to control element sizes for different areas. The main focus is set to a priori procedures that define the sizing function independent on the mesh generation algorithm. The sizing function used by the mesh generation algorithm is established by control sphere and control space concepts and fully controls mesh gradation in the complex domains. Also the new methodology for the developing the automatic sub-optimal initial mesh in adaptive FE strategy is presented. It is not difficult to observe that proposed method allows good quality mesh generation with the minimum number of ill-shaped elements. Constructed initial meshes are expected to be a good start for the adaptive FE analysis.

References

Babuska, I., A. Aziz (1976). On the angle condition in finite element method. *SIAM Journal of Numerical Analysis*, **13**, 214–226.

Bank, R., J. Xu (1996). An algorithm for coarsening unstructured meshes. *Numerische Mathematik*, **73**, 1–36.

Baušys, R., N.E. Wiberg (1999). Adaptive finite element strategy for acoustic problems. *Journal of Sound and Vibration*, **226**, 905–922.

Bhatia, R., K. Lawrence (1990). Two-dimensional finite element mesh generation based on stripwise automatic triangulation. *Computers & Structures*, **36**, 309–319.

Borouchaki, H., F. Hecht, P. Frey (1998). Mesh gradation control. *International Journal of Numerical Methods in Engineering*, **43**, 1143–1165.

Field, D.A. (2000). Qualitative measures for initial meshes. *International Journal for Numerical Methods in Engineering*, **47**, 887–906.

Frey, P.J., P.L. George (2000). *Mesh Generation; Application to Finite Elements*. Hermes Science Publishing, Oxford.

Frykestig, J. (1994). Advancing-front mesh generation techniques with application to the finite element method. *Research Report*, Chalmers Univ. of Technology, Göteborg, Sweeden.

Lee, C.K, S.H. Lo (1993). A new scheme for the generation of a graded quadrilateral mesh. *Computers & Structures*, **52**(2), 847–857.

Lee, C.K., R.E. Hoobs (1999). Automatic adaptive finite element mesh generation over arbitrary two-dimensional domain using advancing front technique. *Computers & Structures*, **71**, 9–34.

Löhner, R., P. Parikh (1988). Three-dimensional grid generation by the advancing front method. *Int. J. Numer. Methods Fluids*, **8**, 1135–1149.

Löhner, R. (1996). Extensions and improvements of the advancing front grid generation technique. *Communications in Numerical Methods in Engineering*, **12**, 683–702.

Owen, S.J. (1992) An implementation of natural neighbor interpolation in three dimensions. *Master's Thesis*, Brigham Young University.

Owen S.J., S. Saigal (2000). Surface mesh sizing control. *International Journal for Numerical Methods in Engineering*, **47**, 497–511.

Peraire, J., J. Peiro, L. Formaggia, K. Morgan, O.C. Zienkiewicz (1988). Finite element Euler computations in three dimensions. *International Journal for Numerical Methods in Engineering*, **26**, 2135–2159.

Peraire, J., M. Vahdati, K. Morgan, O.C. Zienkiewicz (1987). Adaptive remeshing for compressible flow computations. *Jour. of Comput. Phys.*, **72**, 449–466.

Stupak, E., R. Baušys (2000). Nestruktūrinių BE tinklų generavimas sudėtingiems dvimačiams objektams. *Statyba*, **6**(1), 17–24.

L. Vasiliauskienė is a PhD student of the Graphical Systems Chair of the Vilnius Gediminas Technical University. The main research interest is connected with the adaptive finite element strategies in static and dynamics, computational mechanics.

R. Baušys is a professor of Vilnius Gediminas Technical University at the Graphical Systems Chair. At present possesses the Habit. Doctor's Degree. He is a head of Graphical Systems of the Vilnius Gediminas Technical University. His main research interests include problems of adaptive finite element strategies, spatial information systems, computer graphics and image processing.

Pagerintas pradinio optimalaus baigtinių elementų tinklo generavimas sprendžiant 2D uždavinius

Lina VASILIAUSKIENĖ, Romualdas BAUŠYS

Šiame straipsnyje pagrindinis dėmesys skiriamas pagerinto pradinio nestruktūrinio baigtinių elementų tinklo generavimui naudojant Plintančio fronto metodą. Tikslui pasiekti, įvedamos kontrolinės erdvės ir kontrolinės sferos sąvokos, leidžiančios kontroliuoti tinklo elementų tankumą tokiose kritinėse srityse kaip singuliarūs taškai, kampai ar skirtingų medžiagų sąlyčio sritys. Lygindami gautus pagerintus tinklus, matome, kad jau po pirmos iteracijos gaunamas beveik optimalus tinklas, todėl galima konstatuoti, kad toks pusiau optimalus tinklas sumažins kompiuterines prisitaikančiųjų strategijų sąnaudas.