THE QUALITY OF MESHES AND FEM COMPUTATIONS

A. Kolcun

Institute of Geonics, Czech Academy of Sciences
Studentská 1768, 708 00 Ostrava, Czech Republic
kolcun@ugn.cas.cz

ABSTRACT

The paper concerns the mesh quality in context of the finite element method (FEM) computations. The importance of the structured meshes is described. Apriori and aposteriori error estimators are mentioned. Adaptive mesh refinement is involved.

Keywords: FEM, structured mesh, error estimator

1 INTRODUCTION

The mesh generation is usually only a part of pre-processing for any type of numerical analysis. The results of this analysis is essential for us. Its quality depends on the used mesh, so the quality of the mesh means, that the solution based on it has good properties. Here we must consider:

geometry – the mesh must be able to *describe the shape* of the analysed domain. Sometimes it is necessery to analyse many variants of the problem which differ for example in material distribution (very often in geomechanical modelling). To be able to compare the results of such variants, it is useful to use the same mesh for all analysed variants – *consistency of variants*;

computation – the solver is usually the most time consuming part of the FE analysis. So the mesh must be constructed in the way enabling *the solver to work in optimal manner*;

solution – the numerical solution is required to describe the behaviour of analysed phenomena properly. It means that in the area of expected great nonlinear changes of solution *the refinement of the mesh* is required;

memory – although, nowadays it seems that the memory is not a limitation factor for the computer modelling, large 3D problems solved in several variants with variant consistency need huge memory capacities. So *the reduction of memory demands* can be considered as another optimality factor.

The issues mentioned above, 'geometry' and 'solution' are connected with the ability of the mesh to fit the shape of analysed domain, the other aspects described above depend on structure of node coincidency in the generated mesh.

2 DISCRETIZATION IN FEM AND STRUCTURED MESHES

Instead of classical formulation

$$\mathcal{L}u = \mathcal{F}, \qquad u \in \Omega$$

with boundary conditions

$$u = u_0 \quad u \in \Gamma_0$$

$$lu = f_0 \quad u \in \Gamma_1$$

where $\mathcal{L}$ is a differential operator in the area Ω, $\Gamma_0 \cup \Gamma_1 = \partial(\Omega)$ is a boundary of the area Ω,

we solve the problem (weak formulation)

$$(\mathcal{A}u, v) = (f, v) \quad \forall v \in V$$

where $\mathcal{A} : V \to V$ and $(\ ,\ )$ is a scalar product in a functional space V. Continuous formulation is replaced with discretized one:

$$(\mathcal{A}u_h, v_h) = (f_h, v_h) \quad \forall v_h \in V_h$$

which leads to the system of equations

$$Ac = b \qquad (*)$$

where $A = (a_{ij})_{i,j=1}^n$ is the stiffness matrix, $a_{ij} = (\mathcal{A}v_i, v_j)$, $b^T = (b_1, \ldots, b_n)$ is the load vector, $b_j = (f, v_j)$.
FE-solution u_h is then in the form $u_h = \sum_i c_i v_i$ where $\{v_i\}_{i=1}^n$ are basic functions.

In the FEM the basic functions are constructed to make the stiffness matrix A sparse, i.e. there are only few nonzero elements per row here.

If we solve $(*)$ by direct solvers it is convenient, if the A is a band matrix, i.e. the nonzero elements are near the principal diagonal:

$$\exists r\, \forall i, j \quad (|i - j| > r \Rightarrow a_{ij} = 0).$$

We can transform sparse matrix to band one finding suitable node numbering. However, minimization of the bandwidth r for general mesh is NP-complete problem. But for the *structured* meshes (i.e. meshes which are equivalent to index grids) the minimization of bandwidth is trivial [Fied81].

The next benefit of the band matrix with minimal bandwidth is low memory demand. So, even if we use the iterative solvers, in the case of the large tasks the structured meshes are more suitable than general ones. Hence, it is shown in [Kol94], [Kol99], that the restrictions due to structured meshes concerning to the ability of the structured mesh to fit the prescribed geometry, are not so severe as one can require.

3 MESH QUALITY

To design a suitable mesh for FEM we need to assess discretization error of the solution $e = \|u - u_h\|$ where u is exact solution and u_h is a numerical one.

3.1 Apriori error estimation

Let us consider $u \in C^1$, for example $u(x, y) = y^2$, and its piecewise linear interpolation from a triangular mesh. Let the triangulation contains the triangle $P_1P_2P_3$, $P_1 = (0, h/2)$, $P_2 = (0, -h/2)$, $P_3 = (\frac{h}{2}\tan\gamma, 0)$, fig. 1.

In the case of the $H^1(\Omega)$ norm the error depends on the term (see [StrFix73])

$$\max|\frac{\partial}{\partial x}u - \frac{\partial}{\partial x}u_h| = \frac{h}{2\tan\gamma}.$$

This fact leeds to Delaunay triangulation

$$\min_{\gamma\in\triangle}\gamma \to \max.$$

But more detailed analysis [Kri92] shows, that bad approximation is a consequence

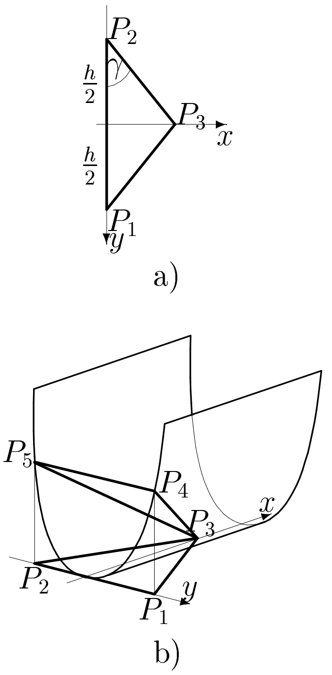

Figure 1: *Linear interpolation in 2D. a) Triangle element. b) Interpolation of the function $u(x, y) = y^2$ with linear one $u_h(x, y)$.*

of too great angle at node P_3. So, the strategy

$$\max_{\gamma \in \triangle} \gamma \to \min$$

seems to be better than Delaunay one.

3.2 Aposteriori error estimation

More powerful strategies are based on adaptive mesh refinement (AMR), which is based on the following algorithm:

```
finish = FALSE
Produce a starting mesh.
repeat
  Carry out FE analysis.
  Compute the error
  indicator q.
  if (q is acceptable) then
   finish = TRUE.
  else
   Refine the mesh
   on information from
   the error estimator.
  end if
until finish.
```

For the elasto-static problems the error indicator based on energy norm is used:

$$|e| = \|u - u_h\|_E =$$

$$= \left(\frac{1}{2} \int_{\Omega} (\sigma - \sigma_h)^T D^{-1} (\sigma - \sigma_h) d\Omega \right)^{\frac{1}{2}}$$

where $\sigma = \sigma(u)$ is stress tensor and D is the elasticity matrix.

ARM is based on the following algorithm:

For the orthogonal meshes and smooth solution the averaged stress gives the better approximation of the exact solution, [Kri85] where the averaged stress is defined as

$$\bar{\sigma}_{\mathbf{i}} = \frac{1}{m} \sum_{\mathbf{j}} \sigma_{\mathbf{j}}$$

where $\mathbf{j}$ are indexes of adjoint elements to element $\mathbf{i}$ and m is the number of these adjoint elements.

The error indicator $|e|$ is defined for all elements on the whole domain, so the global error indicator can be evaluated

$$|q|^2 = \sum_{\mathbf{i}} |e_{\mathbf{i}}|^2.$$

In AMR mentioned above q is acceptable if

$$|q|^2 \leq T|w|^2$$

where T is a threshold of acceptability, $|w|^2 = \sum |w_{\mathbf{i}}|^2$ is the strain energy

$$|w_{\mathbf{i}}| = \left(\frac{1}{2} \int_{\Omega} (\sigma_h)^T D^{-1} (\sigma_h) d\Omega \right)^{\frac{1}{2}}.$$

4 EXAMPLES

4.1 Single structured mesh

We shall illustrate the application of the AMR on a simple example of the square footing problem, [BlaKoh95], fig. 2.

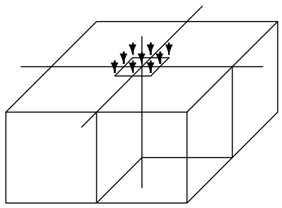

Figure 2: *Square footing problem.*

Due to symmetry we show only one index cut of the mesh. Arrows in the fig. 2, 3 point to the area of the force boundary conditions.

The sequence of generated meshes is shown in the fig. 3.

In the tab. 1 we can find the information about the concentration of local error indicator $|e_{\mathbf{i}}|$. Refinement process of ARM involves the domain where the local error is great.

vol	0	1	2	3	4
50	.25	.11	.08	.03	.03
75	.56	.22	.13	.04	.04
85	.86	.31	.16	.05	.04
90	1.21	.35	.19	.05	.05
95	1.71	.39	.25	.06	.05
100	2.74	1.05	.39	.14	.07

Table 1: *Error distribution according to the volume. For example, in the variant No.2. the local error indicators* $e_{\mathbf{i}} < 0.16$ *in 85 % of the volume of the computational domain.*

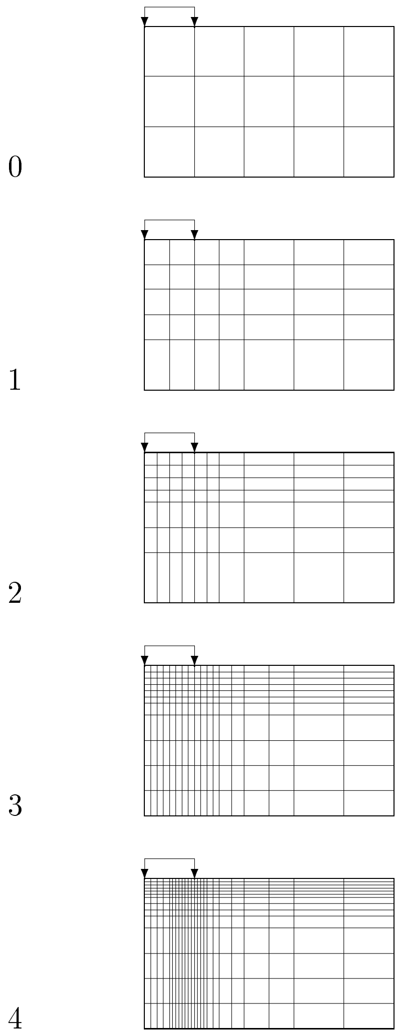

Figure 3: *Process of generating of rectangular mesh for the Square footing problem.*

4.2 Composite structured mesh

The greatest disadvantage of structured meshes is, that we have to refine the mesh not only in the area of big discretisation error see fig. 3.

The use of the iterative local refinement method, [McCor86], [Bla98],successfully solves this problem. Here in AMR loop the FE analysis is realised separately on the coarse mesh and the refined submeshes, where submeshes are made like octree structures.

The second example illustrates the application of large scale 3D FE-modelling for the assessment of the existing stress state and its comparision with the stress which will arise during the supposed future mining, [Bla97], [HajRak98]. Mining process is localised in three nearly parallel veins and is divided to several steps, fig. 4.

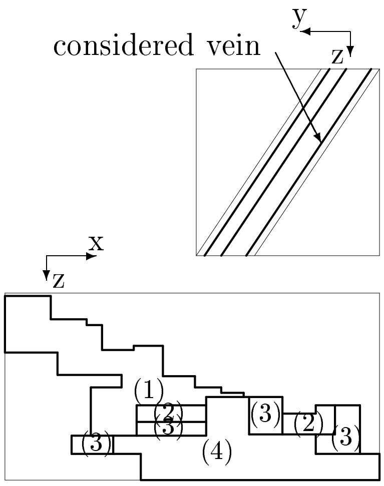

Figure 4: *Four stages of mining process in one ore vein. (1) - the oldest stage ... (4) - the newest one.*

The size of the problem domain is approx. 1400x600x600 m. Owing to the consistency of variants the mesh is very large: 124x137x76 – i.e. 1 291 088 nodes – it means 3 873 264 degrees of freedom. This leads to large memory requirements: about 1.75 GB of outer disk memory and about 80 MB fo the system memory is necessary for running the programs.

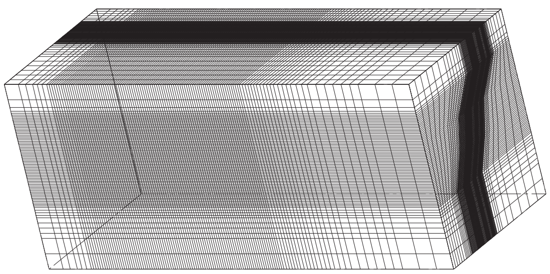

Figure 5: *Boundary faces of the mesh. TGRID-2 generator [Kol97] is used.*

Area of interest is fraught with high discretisation error, so more detailed analysis in this area leads to the composite structured mesh, fig. 6.

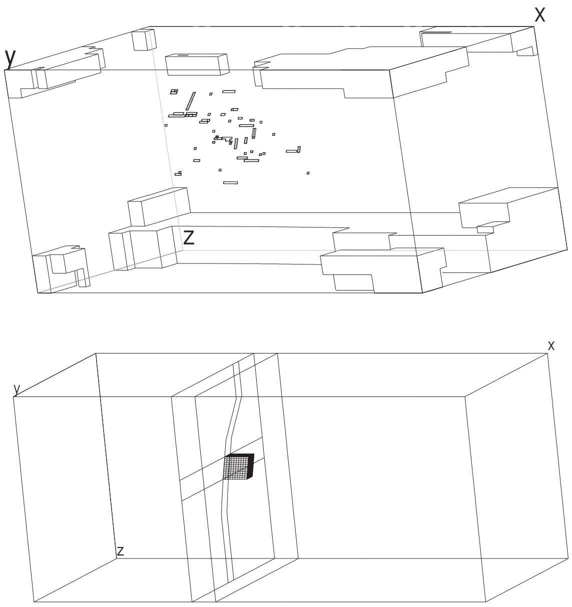

Figure 6: *Elements with high local discretisation error e_i and the localisation of the refined submesh.*

Submesh with 13x29x17 nodes in localised area is refined to the resulting 49x113x65 node mesh. Here we can analyse the problem in more details (to $10^{-1}m$). Hence, the improvement of the solution on the composite mesh is achieved, table 2.

vol(%)	(I)	(II)	(III)
50	.1518	.0428	.0056
75	.4160	.1044	.0112
85	.6893	.1751	.0174
90	.9505	.2418	.0242
95	1.2663	.4600	.0369
100	3.1127	1.0418	.5904

Table 2: *Error distribution according to the volume. (I) - global problem, (II) - subdomain of global problem, (III) - refined mesh in the subdomain.*

5 CONCLUSIONS

There are many aspects of the FE analysis which depends on the quality of the mesh. Some of them need adaptive approach to the mesh generation connected with suitable aposteriori mesh characteristics.

6 ACKNOWLEDGEMENTS

The present work has been supported by grants CP 94 0820 and MŠMT OK 157.

REFERENCES

[BlaKoh95] Blaheta, R., Kohut, R.: *Benchmarks and programs for testing iterative solvers for 3D problems of geomechanics.* IGAS-DAM Report9502, DAM-IGAS Ostrava 1995.

[Bla97] Blaheta, R. et al.: *An application of large scale mathematical modelling in geomechanics.* Proc. of conf: Contemporary Math. Methods in Engineering, Dolní Lomná, 1997.

[Bla98] Blaheta, R., Kohut, R.: *Iterative local refinement methods for nonlinear problems. Software for iterative local refinement methods.* HIPERGEOS Report, DAM-IGAS Ostrava 1998.

[Fied81] Fiedler, M.: *Special matrixes and their use in numerical mathematics* (in czech). SNTL, Praha 1981.

[HajRak98] Hájek, A., Rakowski, Z.: *Role of large scale math. models in development of new mining technology.* HIPERGEOS Report, IGAS Ostrava 1998.

[Kol94] Kolcun, A.: *Semiregular grids in 2D and 3D.* Proc. of conf. WSCG94, Univ.of West Bohemia, Plzeň 1994.

[Kol97] Kolcun, A.: *TGRID-2: Preprocessor for structured nonrectangular FE grids arising in geomechanics.* IGAS-DAM Report, IGAS Ostrava 1997.

[Kol99] Kolcun, A.: *Preprocessing for FEM in geomechanics.* Institute of Geonics ASCR, Ostrava 1999.

[Kri85] Křížek, M.: Superconvergence results for linear triangular elements. In: Equadiff 6, (J. Vosmanský and M.Zlámal ed.), Springer Berlin 1985.

[Kri92] Křížek, M.: On the maximum angle condition for linear tetrahedral elements. *SIAM J. NUMER. ANAL.,* Vol 29, No 2. (1992), (513–520).

[McCor86] McCormick, S., Thomas, J.: *The fast adaptive composite grid method for elliptic equations.* Math. Comp. 46 (1986), (439-456).

[StrFix73] Strang, G., Fix, G.J.: *An analysis of the finite element method.* Prentice Hall, Englewood Clifts, NJ 1973.