Chapter 5
MODELING, ERRORS AND SIMILARITY

5.1. Introduction

The method of finite elements is a numerical method with inevitable errors due to improper modeling or the calculating operations. The result obtained with FEA, is approximately compared to the real one and the correct estimation of the obtained result is one of the important aspects in the application.

The accuracy is the criterion of obtaining the correct result. The FEA includes numerical procedures in which the result may approximate the exact one but also may give result completely different from the real ones. The increase of the error done during the numerical procedures defines the stability of the solution. Unstable is the solution which we lost due to the accumulated errors in the numerical procedures.

Similarity of the solution is when the numerical solution approximates the real one. The term similarity may be used in the iterations as well, in which the results from a given task are used as an input data for the next one. In the similar procedure for defining of some parameters (for example degree of approximating polynomial or size of the element) the difference in the results in the subsequent calculations are decreasing and in the border case can result in zero. On fig.5.1. can be seen that if the process is similar while settling the parameters of the numerical procedures, the accuracy increases and if the process is unsimilar, the accuracy decreases. In this way in FEA the analysis of the errors and the reasons for them is brought down to the examination of the similarity.

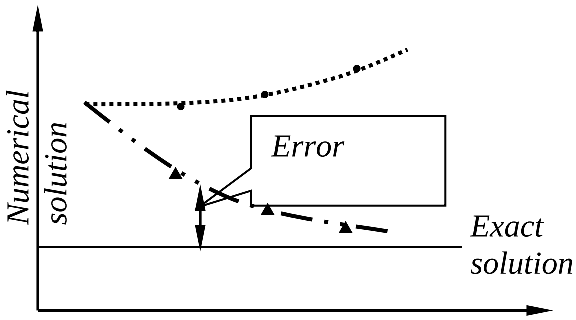


Fig. 5.1

The similarity can be numerically estimated with mathematical problems of which the solution is precisely known or with comparison of results obtained through the different methods for separation of the body to finite elements. It must be taken under consideration that it is not always possible to estimate the accuracy of the solution through tests or in some other way. Some general questions of the similarity can be estimated theoretically.

Most commonly the absolute energy of the system is estimated when we use the theoretical analysis of the similarity of the solution. If in its border cases of change of the parameters (size of the elements, degree of the approximation) it approximate the correct value follows that the displacement, the deformation and the stresses will also approximate the correct one in each segment of the body.

Each elastic body has infinite large number of degrees of freedom. Their restriction is equivalent to introduction of additional inner nodes, which leads to increase of the rigidity of the body compared with its real one. Considering this for FEA, it means that the obtained displacements through this method will be smaller than in comparison to their correct values.

Compressing the mesh of the finite elements, we increase the number of degree of freedom. Of utmost importance in this case is to clarify under what conditions this leads to improvement or similarity of the solution. Another important part is the speed of the similarity. Having a higher speed of similarity in FEA we may obtain a satisfactory solution even with a rather rough mesh of finite elements.

If with $\{u\}$ we designate the displacement matrix, determined in FEA, and with $\{u_o\}$ - the displacement matrix corresponding to the correct solution ad taking into account that the actual displacement minimize the full energy of the system Π, than it follows that $\Pi(\{u\}) \geq \Pi(\{u_o\})$. If for a given deformed condition of the body, constituted of finite elements we designate with $\{u^*\}$ the matrix of the nodal displacements corresponding to the exact values (they are higher than the obtained with FEA), it follows that $\Pi(\{u^*\}) \geq \Pi(\{u\})$. From here is seen $\Pi(\{u^*\}) \geq \Pi(\{u\}) \geq \Pi(\{u_o\})$ and if $\Pi(\{u^*\}) \rightarrow \Pi(\{u_o\})$, than $\Pi(\{u\}) \rightarrow \Pi(\{u_o\})$ as well.

From the above mentioned follows that if in the model of FEA we make nodal displacements corresponding to the correct solution, a proof for the similarity of the solution can be the fact that the full energy of the system approximates to the correct value if there is a decrease in the size of the finite elements.

The examination of the displacements $\{u\}$ and $\{u^*\}$using Tailor's rows shows that the speed of the similarity increase with the increase of the degree of the polynomial with which are approximated the displacements in the finite elements. Apart from that it turns out that the error of the solution is in the sequence

$$E = O\left(h^{p+1-r}\right), \tag{5.1}$$

where h is the size of the element, p is the degree of the approximating displacement polynomial and $r = 0$ the error calculated for the displacements and $r=1$ for calculation of deformation and stresses if and only if the polynomial is complete. For example, for $p=3$ the approximating polynomial should contain the following members x^3, $x^2 y, x, xy, y, xy^2, y^3$ and than the displacement error will decrease around h^4. This means that if the size of the element is decreased twice, the error will decrease eight times. The rejection of whichever of these members (or introduction of constraints for the polynomial coefficients) will result in a decrease in the sequence of the error to h^3. For example, if for the approximation of the displacements is used polynomial of type

$$a_1 + a_2 x + a_3 y + a_4 x^2 + a_5 y^2 + a_6 x^3 + a_7 x^2 y + a_8 x y^2 + a_9 y^3, \tag{5.2}$$

from which the members *xy*, is missing, it follows that the error will be in about h^2 and the speed of the similarity will be considerably less than in its presence. It must be noted also that in the use of polynomials with different degrees for approximation of the separate components of displacement (as it is in the case of the modeling of the shell), the speed of the similarity will be determined by the lowest degree of approximation.

The above-presented facts may be illustrated with the help of a compatible 2D rectangular element. The displacements are presented with full polynomial of first sequence

$$\begin{aligned} u(x,y) &= a_1 + a_2 x + a_3 y + a_4 xy \\ v(x,y) &= a_5 + a_6 x + a_7 y + a_8 xy. \end{aligned} \tag{5.3}$$

The error will be about h^2, where h is the length of the longer side of the rectangle.

For the improvement of the characteristics of the element can be introduced non-nodal degrees of freedom in the equations in (5.3), for example the expression

$$\left(1-\xi^2\right)\left(1-\eta^2\right), \tag{5.4}$$

in non-dimensional coordinates with arbitrary multipliers, where $\xi = \frac{x}{a}$ and $\eta = \frac{y}{b}$. As a result in the equations (5.3) will appear the squared members of the coordinates as well. Because of the fact that the multiplier before the members are equal, turns out that full polynomials of second degree can't be obtained and the error of the approximation stays about h^2. In this case the added functions do not lead to substantial improvement of the finite elements, which is confirmed from the numerical calculations. The addition of terms $\left(1-\xi^2\right)$ and $\left(1-\eta^2\right)$ with arbitrary multiplier leads to polynomial of second degree. In this case, although the compatibility the element is interrupted, the obtained element is with much better characteristics.

It is determined that in the cases of a mesh of compatible elements, the minimum requirement necessary for similarity is that each segment of displacement within the element can be expressed as a polynomial with degree not less than first. This requirement is known as a condition for the corpulence of the finite element. If the elements meet the requirement for completeness and are compatible, the similarity of the solution according to the energy will be monotonous. This means that with the compression of the mesh the full energy of the system will decrease, in which case it will remain higher than the exact value.

From the already examined elements, only the triangle one with a linear plane of the deformation and the compatibility rectangular element, meet the condition for corpulence. In both cases the error of approximation of the displacements decreases as h^2. The numerical examples illustrate that the speed of the similarity in the case with the rectangular element is greater.

Concerning the isoparametric plane finite elements, can be proved that they meet the condition of corpulence and are compatible as well. So their use guarantees the monotonous similarity of the solution in case of compressing the mesh. The case with 1D and 3D isoparametric finite elements is analogous.

In the cases of non-compatible finite elements the similarity of the solution can be obtained if and only if in the border case (of the compression of the mesh) in the approximate functions the members that cause non-compatibility disappear. In other words, a similarity can be guaranteed if the elements in the border case can recreate the linear plane of deformations and so become compatible. A strict requirement for these elements is the condition of recreation of the linear plane of deformations as well as the providing continuity of the body under arbitrary sizes of the elements.

5.2 Classification of the errors

According to the origin of the error of the numerical solution, they can be divided into:

1. Error due to modeling – they are due to differences between the physical and mathematical model. For example, the mathematical model of slab is constructed based on a series of precondition that simplifies the mathematical model but at the same time these could result in errors. Further on the numerical analysis is based on the mathematical model. It must be noted that in the approximation of the plane of displacements in the element can be eliminate some of the physically possible forms of the deformation, which renders the model inadequate.

2. Errors due to discretisation – they are due to presentation of the mathematical model with an infinitely large number of degrees of freedom as a model with a finite number of degrees of freedom. It must be taken into consideration that the error of discretisation depends on the number of degrees of freedom and is also influenced mainly by the average size h of the elements and by the degree p of the polynomial which approximates the plane of displacements. The quality of the mesh is very important as well.

3. Error due to the approximation – they are due to the presentation of the numbers as a finite number of digits.

4. Common error – it is due to the sum of errors of approximation on the given stage of the numerical solution.

5. Error of the algorithm – it is due to the error of approximation, introduced by the accepted algorithm of the numerical operations.

5.3. Errors due to discretisation. Similarity with improvement of the mesh.

The change from the mathematical model to a model of the FEA is related with the choice of number, type and shape of the finite elements, constructing the mesh, giving the kinematics and static border conditions. The model made by this process has errors, commonly known as errors of discretisation. Later on these implemented defects can cause problems in the calculations.

From the above mentioned the error in the calculated value may be represented as $E = O\left(h^{q-r}\right)$, where $q=p+1$, $r=0$ for the error of displacement, $r=1$ for the error of deformations and stresses. One approximate determination of the error can be done using the following formula

$$E \approx Ch^{q-r}, \qquad (5.5)$$

where h is once more the characteristic size of the element and C – constant, depending on the ratio between the longest and the shortest side of the element, on the ratio between the dimensions of the largest and the smallest element of the mesh, the sequence of the square surface for the numerical integration, of the $(q-p+1)$-sequence of the derivative of the correct solution and other factors.

If the precise value of the calculating quantity is Φ, and the calculated is Φ^h, it follows that the error can be determine from

$$E = \left|\Phi - \Phi^h\right|. \qquad (5.6)$$

In (5.6) Φ^h can also be some standardized value of the calculating quantity. The relative error is calculated using the formula

$$E_o = \frac{\left|\Phi - \Phi^h\right|}{\left|\Phi\right|} . \qquad (5.7)$$

On the basis of the so called remaining part of the nodal loads

$$\{\Delta R\} = \{R\} - [K]\{D\} \qquad (5.8)$$

can be determine a scale amount of the error of type

$$E = \frac{\{D\}^T\{\Delta R\}}{\{D\}^T\{R\}} , \qquad (5.9)$$

where $\{\Delta D\} = [K]^{-1}\{\Delta R\}$. The so calculated error represents the ratio of the work done by the remaining loads to the work done by the actual loads, carried out during the displacements $\{D\}$. If the absolute value of this quantity $|E|$ is about 10^{-8} or less, it can be considered that the error of approximation during the calculations is negligible.

One rough indicator of the error of dicretisation can be represented with the expression:

$$E_\Gamma = \rho_1\rho_2 h^{q-r} , \quad h = \frac{1}{N^{1/n}} . \qquad (5.10)$$

In (5.10) ρ_1 is the largest ratio between the long and the short side of the element of the mesh, ρ_2 is the ratio between the general sizes of the biggest and the smallest element of the mesh, *h* is sizeless characteristic length of one element, *N* is the number of elements in the mesh and n – space dimension ($n=1$ in linear task, 2 in 2D and so on). For example, if for 2D area are used 100 6-noded elements with $\rho_1 = \rho_2 = 1$ for the error in the calculations ($p=0$, a $r=0$) we will have $E_\Gamma \approx \left(\frac{1}{\sqrt{100}}\right)^2 \approx \frac{1}{100}$.

The explanation of E_Γ is the following. If E_Γ is $0,1$ times from the acceptable percent error (for example $E_\Gamma = 1$%, and the acceptable percent error is 10 %) the result can be considered as acceptably correct. If it is considerably less, the mesh may be considered as much finer, than the required. If E_Γ is closed t the acceptable error or surpasses it, more calculations are necessary. This means change of the mesh without changing the number of the elements or improvement, increasing the number of elements.

The solution using FEA is approximate to the correct improvement of the mesh, if there are no big errors in the model and if the elements have passed through the quality test. The solution is similar as seen from the below shown figures because of the higher rigidity of the model of FEA in comparison to the real object. Пif there is a improvement in the first mesh (fig. 5.2, a) by splitting of the elements without change of the existing nodes (fig. 5.2, б), the similarity will be monotonous from below.

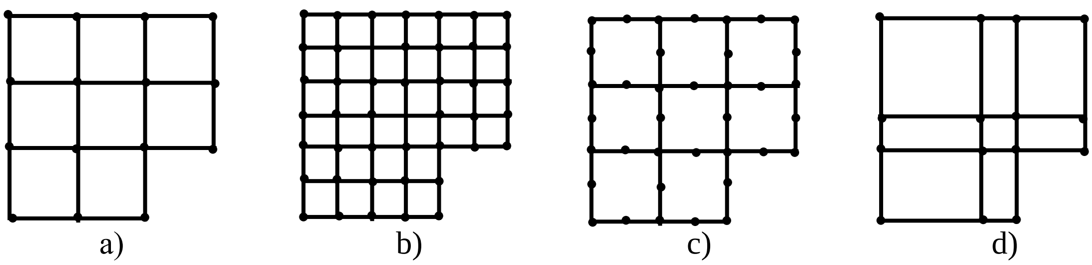


Fig. 5.2

The improvement of the mesh related only o the change of the number of elements and therefore only with change of the characteristic size *h* of the elements, is known as *h*- version of FEA. In this case the degree *p* of the approximating polynomial of the displacement does not change. The so called *p*- version of FEA is related with the change of the degree of the approximating polynomial, without change of the sizes and the number of the elements (fig. 5.2, b). In this case we add only nodes to the elements of the mesh. The third possibility is the so called *r*- version. The improvement in this case is connected to displacement of the nodes, without change of the number of the elements (fig. 5.2, d). Such a change in the mesh without increasing of the number of the elements can only bring to a limited improvement in the accuracy.

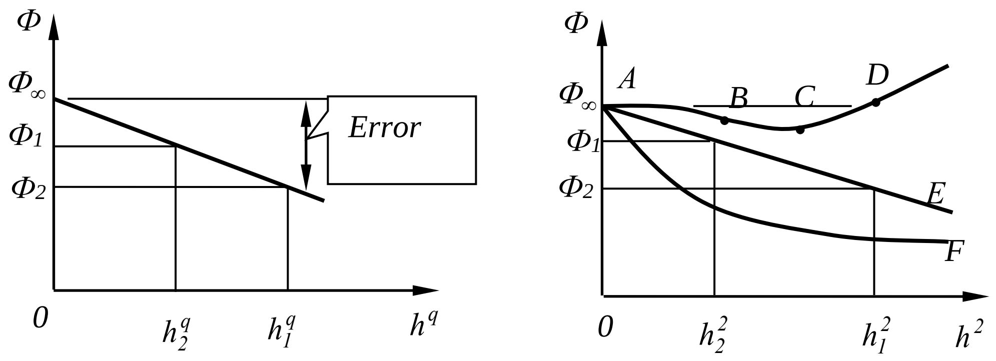
In practice, none of the above version is used independently. It is estimated, that in tasks containing singularity, for example inner angels or cracks, the *p*- version gives a quicker similarity than in *h*- version, mainly in combination with *r*- improvement, which in its turn leads to quick increase of the density in the area of singularity.

The software products of FEA have the possibilities to estimate the error of the solution and to carry out adaptation of the mesh through revision, analysis and automatic repetition of the cycles until we obtain the given tolerance for the similarity. The adaptation may be achieved with *h*- , *p*- or *hp*- improvement of the mesh.

If the speed of the similarity is unknown and we assume monotonous similarity, the improved value of the calculated quantity can be found using linear extrapolation (fig. 5.3, a)

$$\Phi_\infty = \frac{\Phi_1 h_2^q - \Phi_2 h_1^q}{h_2^q - h_1^q} \ , \qquad (5.11)$$

where h_1 и h_2 are the characteristic sizes of the elements of two solutions, Φ_∞ is the value of the calculated quantity in an infinite dense mesh (fig. 5.3, a). it is supposed that the error is proportional to h^q. Conclusion for the fact whether the similarity is linear, quadratic or of higher order, or whether is the order of *q*, can be reached from the results of the three solutions with different meshes. For example, if we decrease twice the size of the element, the error decreases four times, it follows that the error is proportional to h^2. It must be noted that in the case of extremely rough mesh the tendency can be seen with great difficulty.



a) b)

Fig. 5.3

On fig. 5.3, b is visible that if $q=2$ and the similarity is non-monotonous using the values of Φ in points *C* and *D,* the mentioned extrapolation may bring to a worse result. Such danger may be expected with the non-compatible elements, which often show non-monotonous similarity. On fig. 5.3, b the line *AE* corresponds to an error, proportional to h^2, while the line *AF* may become straight if *h* is projected on the abscissa.

It is difficult to accept the value of Φ_∞ as an exact solution, but however the estimation for the percentage error of the mesh with h_2 may be calculated according the formula

$$E = \frac{\Phi_2 - \Phi_\infty}{\Phi_\infty} 100\,\%. \qquad (5.12)$$

5.4. Testing and shape of the elements

The behavior of the different types of elements under loads can be easily defined using numerical tests, by choosing problems to which the solution is known. There are two tests for checking the function of the elements and their quantities.

The first type of testing the work is the estimation of small segments of the finite elements. On fig. 5.4 is shown plane task and a model of FEA of a small group pf 4-noded elements. The thickness is *1*, the force *F* having value $F = \frac{1}{4} 4.1.\sigma_0 = \sigma_0$.

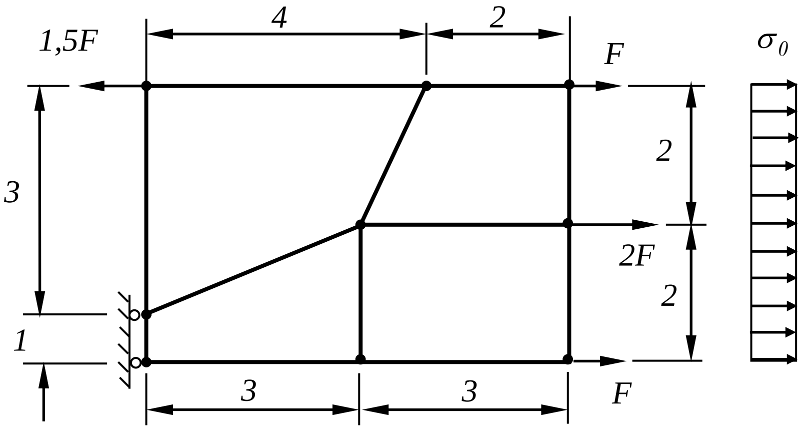


Fig. 5.4

If the solution of FEA gives $\sigma_x = \frac{2F}{at}$ for each point, in which the stress is calculated and the other kinds of stresses are zero, it follows that the test for σ_x is passed. A successful test means that an element used in the mesh is be able to show:

- State of constant deformation;
- A transfer of the body as it is rigid without deformation;
- Compatibility with the neighboring finite element when the deformation is constant in this area

The passing of such a test guarantees that with the improvement of the mesh, the solution will tend to the to the accurate one, despite the speed of the similarity.

The other important test is for the proper values. This test may show the forms in deformed state with zero energy of the deformation, and also the possibility of showing transfer as a rigid body.

If loads are applied to the nodes of the elements, proportional to multiplier λ on the nodal displacement, it can be written as

$$[k]\{d\} = \{\bar{r}\} = \lambda\{d\} \text{ or } ([k] - \lambda[I])\{d\} = \{0\}. \qquad (5.13)$$

This is the task for proper values. The proper values of the stiffness matrix λ_i must be the same as the degrees of freedom in $\{d\}$. To each λ_i corresponds proper vector $\{d\}_i$. If each proper vector is normalized by multiplying with $\{d\}_i^T$ in the following way that $\{d\}_i^T \{d\}_i = 1$, can be written as

$$\{d\}_i^T [k]\{d\}_i = \lambda_i \text{ или } 2U_i = \lambda_i, \qquad (5.14)$$

where U_i is the strain energy of the element.

For one unfixed element, during the test must be obtained such number of zero proper values as are the possible displacements of this element as a rigid body (for example, for plane 3, for the space 6 etc.). less zero proper values means that the element will have displacements as rigid body, which will cause deformation and vice verse if the zero proper values are more, this means that the element will show condition of unstability. The ideal case is when the element is without these defects.

One element may be tested individually. On fig. 5 are shown different cases of this type of testing.

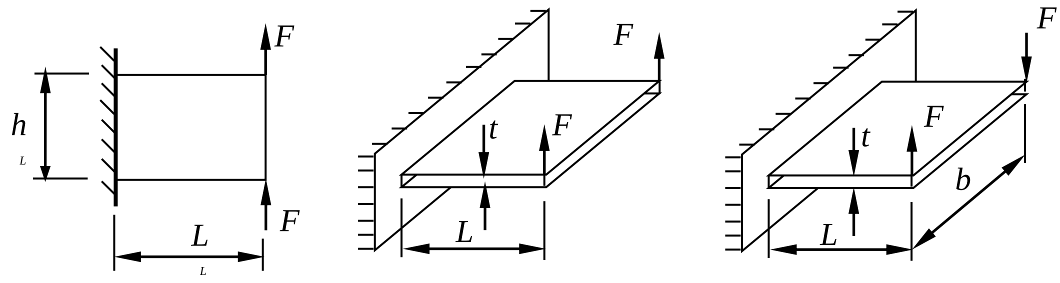


Fig. 5.5

With change of the ratio *L/h* (fig. 5.5, a) can be defined the behavior of the element during its stretching. While testing according to fig. 5.5, b can be estimated the tendency of 'locking' of the element while increasing the ratio *L/t* or the adequate modeling of the tangential stresses when *L/t* become smaller. From the testing using the scheme of fig. 5.5, c the influence of *L/t* и *L/b* can be simultaneously examined over the behavior of the finite element.

Through such test is determined that one element functions best if it has a compact and regular shape. The element becomes more rigid and decreases the accuracy of the solution when the ratio between its sides is increased, when the angles between its sides begin to vary considerably from each other, when the sides become curved or there are irregularly placed nodes along the sides. On fig. 5.6 are presented elements with undesirable shape:

a) – a big ratio between the sides;
b) – shape of a rectangle, resembling a triangle;
c) – a node displaced from the centre;
d) – a strongly inclined rectangle;
e) – a rectangle crooked into triangle;
f) – a strongly curved side;

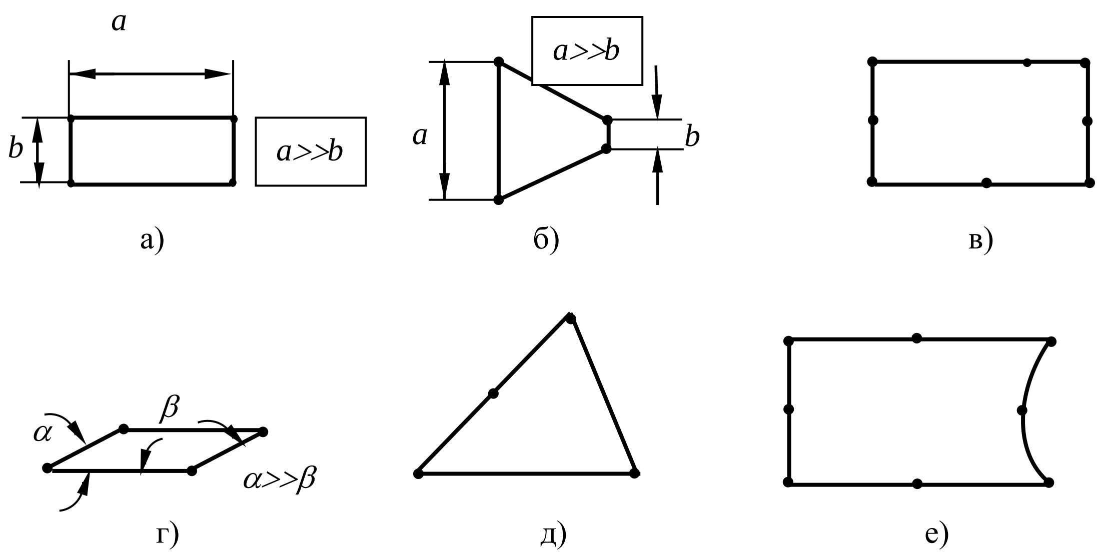


Fig. 5.6

The different types of elements have different sensitivity to the curvature, but the following rules must be generally observed:

- The ratio between the sides of the elements is to be about 1;
- The angles of the rectangular elements are to be about 90^0;
- The nodes along the sides are to be in the middle;
- The sides of the elements are to be straight.

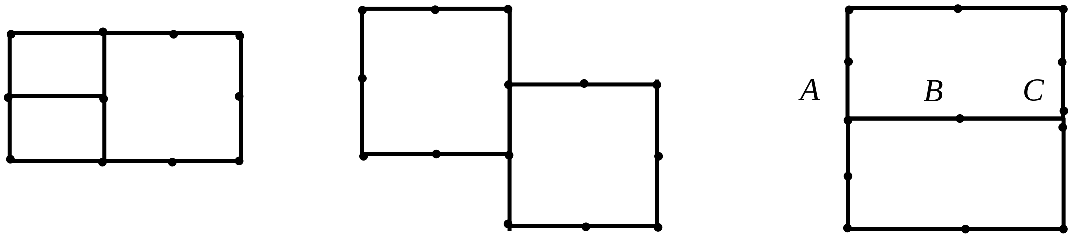
Of course, there are also exceptions. Elements with big ratio between the sides can be used in the cases when the gradient of the area of deformation is small. In the modeling of cracks, to avoid the singularity we change the position of the node on the side in such way that a special type of finite element for the estimation of the crack is obtained. The sides of the elements may be curved according to the curving linear border of the field.

Special attention should be paid also to the joining of the separate types of elements. On fig. 5.6 are shown variations of elements badly connected:

а) – two bi-linear and one quadratic;

б) – two quadratic elements;

в) – two quadratic elements joined in A и B, but not in C.



а) б) в)

Fig. 5.6

During the transfer from rough to fine mesh must not be allowed great miscorrespondance in the sizes of the neighboring elements. A big difference in the rigidity of the neighboring elements must not be accepted. To illustrate this with an example the 3D task can be used in which the ratio between the modulus of elasticity and the volume of the element *E/V* must not surpass 3.

5.5. Estimation of the error. Adaptive solution with FEA.

In one solution of FEA there is enough information for the estimation of the error of discretisation. The present software products have post-processing programs able to make an inspection of the mesh of finite elements, in such way that the next solution is more accurate. The cycles of analysis with inspection of the mesh can be go on until a previously acceptable quantity for the similarity of the solution is obtained. As a criterion for the estimation of the error of the solution the strain energy of the deformations can be used. Such a variant can be explained using the simple case of beam loaded with an uniformly distributed load, shown on fig. 5.7.

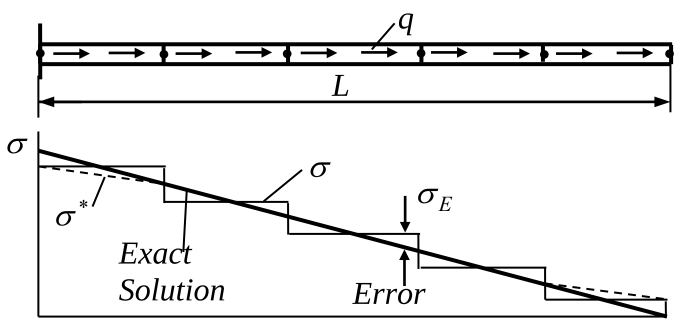


Fig. 5.7

The stress in each element is calculated and a discrete picture, because of the changes of the stresses on the border of the elements. The area of stress obtained using the average values of the border between the elements are constant and with the exception of the two finite elements it coincides with the correct one. Despite the way in which the constant area of stress is obtained, it could be regarded as a solution much closer to the correct one, which is allowed from the given mesh of elements. The difference between the element area of stress and the constant one can be accepted as an approximate estimation of the area of errors. The symbols on fig. 5.7 are as follows: σ - stresses, calculated of the elements; σ^* - the average area of the stresses (constant); $\sigma_E = \sigma - \sigma^*$ - stress due to the error. On the basis of the so defined areas of stress can be determined the strain energy of the deformations, namely

$$U = \sum_{1}^{n} U_i \text{ , where } U_i = \int_0^{L_i} \frac{\sigma^2}{2E} A dx, \qquad (5.15)$$

$$U^* = \sum_{1}^{n} U_i^* \text{ , where } U_i^* = \int_0^{L_i} \frac{(\sigma^*)^2}{2E} A dx, \qquad (5.16)$$

$$U_E = \sum_{1}^{n} U_{E,i} \text{ , where } U_{E,i} = \int_0^{L_i} \frac{\sigma_E^2}{2E} A dx. \qquad (5.15)$$

In the above dependence *A* is the cross section area, L_I is the length of the element, and *E* is the modulus of elasticity. The summing up is from all elements of the mesh.

In the general case of mesh from arbitrary types of elements, the expressions under the integral are of type $\frac{1}{2}\{\sigma\}^T [E]^{-1} \{\sigma\} dV$, where $[E]$ is a matrix of the elastic constants. The energy $U_{E,i}$ is not a criterion for the accuracy of the calculated stresses, but can be used for inspection of the mesh of finite elements. As a criterion for the similarity of the solution can be used the quantity which gives the relative energy error, namely

$$\eta = \left(\frac{U_E}{U + U_E} \right)^{\frac{1}{2}}, \text{ where } 0 \le \eta \le 1. \qquad (5.18)$$

The denominator in (5.18) represents approximation to exact strain energy, taking into account, that the energy *U* is less than the exact strain one, because of the greater rigidity of the model of FEA in comparison to the real construction. Instead of $U + U_E$ can be used energy U^*. The square root serves to connect the quantity η with the area of stresses. As a global quantity η can't be estimation for the accuracy in a separate point.

If the obtained value of η for a given solution in FEA is less than the prescribed one, the procedure is stopped. In the contrary case the calculated value of $U_{E,i}$ for each element is used for adaptive change of the mesh of finite elements. In the inspection of the mesh according to *h*- or *p*- method the number of elements increases correspondingly or more nodes are placed on the elements in the points where $U_{E,i}$ has relatively higher values. The next cycle of the procedure starts with analysis in FEA of the already inspected model. The procedure continues with a definite number of cycles carried out, after which the results are checked. On fig. 5.8 is shown an example of adaptive change of the mesh carried out for the COSMOS/M program.

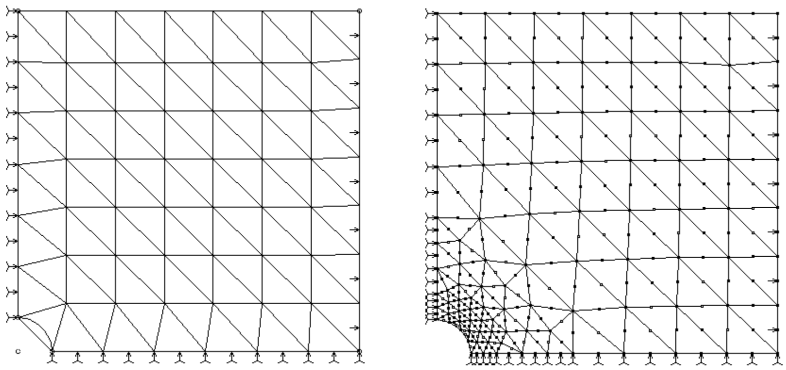


Fig. 5.8

The above described adaptive procedure leads to a mesh of finite elements in which the strain energy of the deformations $U_{E,i}$ can become the same in all elements.