COMPUTATIONAL MECHANICS
WCCM VI in conjunction with APCOM'04, *Sept. 5-10, 2004, Beijing, China*

Fixed Grid Finite Element Analysis for 3D Linear Elastic Structures

M. J. Garcia [1*], O. E. Ruiz [1], L. M. Ruiz [1], O. Querin [2]

[1]*Department of Mechanical Engineering, EAFIT University, Cr 47 No. 7 sur 50, Medellin, Colombia*
[2] *School of Mechanical Engineering, University of Leeds, Leeds LS2 9JT, UK.*
e-mail: {mgarcia, oruiz, lruizram}@eafit.edu.co, ozz@mech-eng.leeds.ac.uk

Abstract Fixed Grid (FG) methodology was first introduced by García and Steven [7] as an engine for numerical estimation of two-dimensional elasticity problems. The advantages of using FG are simplicity and speed at a permissible level of accuracy. Two dimensional FG has been proved effective in approximating the strain and stress field with low requirements of time and computational resources. Moreover, FG has been used as the analytical kernel for different structural optimisation methods as Evolutionary Structural Optimisation [9], Genetic Algorithms (GA), and Evolutionary Strategies [4]. FG consists of dividing the bounding box of the topology of an object into a set of equally sized cubic elements. Elements are assessed to be inside (I), outside (O) or neither inside nor outside (NIO) of the object. Different material properties assigned to the inside and outside media transform the problem into a multi-material elasticity problem. As a result of the subdivision NIO elements have non-continuous properties. They can be approximated in different ways which range from simple setting of NIO elements as O to complex non-continuous domain integration. If homogeneously averaged material properties are used to approximate the NIO element, the element stiffness matrix can be computed as a factor of a standard stiffness matrix thus reducing the computational cost of creating the global stiffness matrix. An additional advantage of FG is found when accomplishing re-analysis, since there is no need to recompute the whole stiffness matrix when the geometry changes

This article presents CAD to FG conversion and the stiffness matrix computation based on non-continuous elements. In addition inclusion/exclusion of O elements in the global stiffness matrix is studied. Preliminary results shown that non-continuous NIO elements improve the accuracy of the results with considerable savings in time. Numerical examples are presented to illustrate the possibilities of the method.

Key words: Fixed Grid Finite Element Analysis, Interactive Design

INTRODUCTION

Fixed Grid (FG) is a methodology to solve elasticity problems was first introduced by García and Steven [2, 7, 6] as an engine for numerical estimation of stress and displacement fields. The advantages of using FG are simplicity and speed at a permissible level of accuracy. In FG the stress error was seen to increase near the region of stress concentration, with a maximum stress error being approximately 10% for a reasonably-sized mesh. However, the average stress error was found to be about 5% or below and the displacement field error was even lower, around 1% [5]. Thus, the FG method was deemed as appropriate for interactive design and structural optimisation where highly accurate analysis is not needed.

A Fixed Grid is generated by superimposing a rectangular grid of equal-sized elements on the given structure instead of generating a mesh to fit to the structure. In this way, elements are either inside(I), outside(O), or on the boundary (NIO) of the structure. An O element is given a material property of a non-interactive media. That is, its value is significantly less than the property of an I element. This transforms the problem into a bi-material one. NIO elements are constituted by two types of material and therefore their properties are not continuous over the element. Different methods can be used to approximate NIO elements. That includes from dropping them as O elements to complex non-continuous domain integration. If averaged material properties are used to approximate the NIO element properties, then the element-stiffness matrix can be computed as a factor of a standard stiffness matrix. therefore the assembly of the system matrix can accomplished efficiently. An additional advantage is found when the shape of the structure is changed in response to a previous analysis. The global structure of the FG is maintained, and recomputation of the new stiffness matrix can be accomplished by changing only the positions affected by the elements whose I, O, and NIO state has changed [8].

The application of FG-FEA to two-dimensional linear elastic problems has been a research topic during the last years, see [2]. It has been proved effective in approximating the strain and stress field with low requirements of time and computational resources. Moreover, FG-FEA was used as the mathematical kernel for different structural optimisation methods like: Evolutionary Structural Optimisation (ESO) [10], Genetic Algorithms (GA) [14], and Evolutionary Strategies (ES) [4].

Previous work has been done with three-dimensional linear elastic structures by Suzuki and Ohtsubo [13]. Boundary conditions are applied by further subdividing the boundary elements (multi-scale voxel) thus requiring a less general resolution. The present approach presents some improvements to the local stiffness matrix and force vector calculations for NIO elements by considering them as non-continuous elements. Also the inclusion or exclusion of O elements in the global stiffness matrix is studied.

1. Definitions

Let Ω_{FG} denote the smallest bounding box that completely encloses the domain Ω and is oriented along the axes of the coordinate system, that is

$$\Omega_{\text{FG}} = \left\{ \mathbf{x} \,\middle|\, \min_{\mathbf{y}\in\Omega}(y_i) \leq x_i \leq \max_{\mathbf{y}\in\Omega}(y_i) \right\},$$

then Ω_{FG} is called in this study the **fixed grid domain**. A point $\mathbf{x} \in \Omega_{\text{FG}}$ is considered *inside* if $\mathbf{x} \in \Omega$. A point $\mathbf{x} \in \Omega_{\text{FG}}$ is considered *outside* if $\mathbf{x} \notin \Omega$. In order to preserve characteristics of the original problem, the material properties of an outside point are the properties of a non-interactive medium. The object is embedded into a box made of non interactive material. Notice that this definition transforms the elastic problem into a bi-material problem. Figure 1 shows examples of a fixed grid domain for two-dimensional and three-dimensional cases. The domain is completely defined by the points $\mathbf{x}_{\text{min}}$ and $\mathbf{x}_{\text{max}}$ that define the maximum and minimum points that belong to Ω_{FG}.

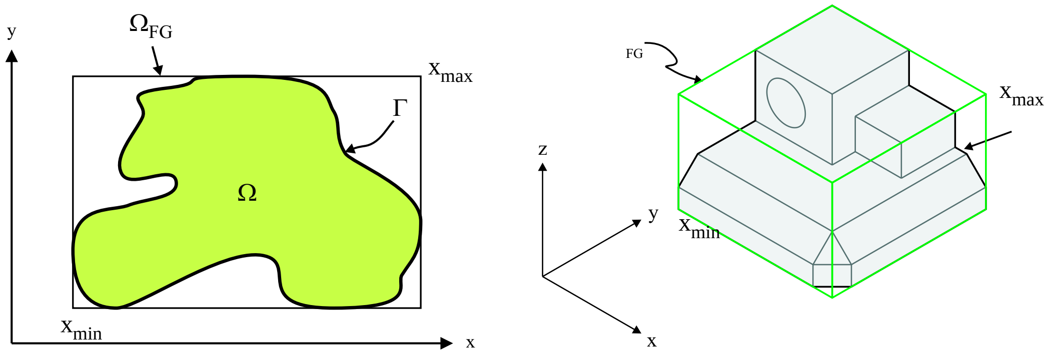


Fig. 1 Typical two- and three-dimensional fixed grid domains

In order to obtain the FG, the fixed grid domain, Ω_{FG}, is subdivided into a set of cubic elements with dimensions h_1, h_2, h_3. See figure 2.

An **element** is each one of the cells, of dimension $h_1 \times h_2 \times h_3$, in which Ω_{FG} is subdivided. The elements $e_m, m = 0, \ldots, n_e - 1$ are numbered in ascending order from the one containing the minimum point $\mathbf{x}_{\text{min}}$ in the direction of the x axis, then in the y axis, and finally in the z axis. An element e_m can be associated by its index in each dimension $e_m = e_{ijk}$. That is row, column, and floor of the element in the grid. The element domain Ωe_m is defined as $\Omega e_m = \left\{ \mathbf{x} \middle| \mathbf{x} \in e_m \right\}$, and its boundary is denoted as $\partial\Omega e_m$.

The **nodes** are the vertices of the elements in the FG. Each element $e_m = e_{ijk}$ has eight nodes n_j, with $j = 0..7$. A node in the FG is classified as emphinside node if $n_j \in \Omega$ and **outside node** if $n_j \notin \Omega$.

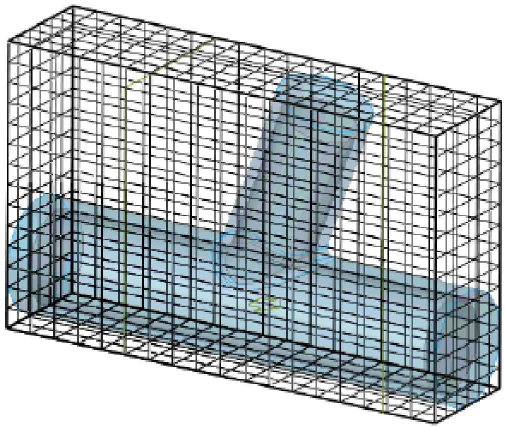


Fig. 2 Three-dimensional fixed grid

2. Element Classification

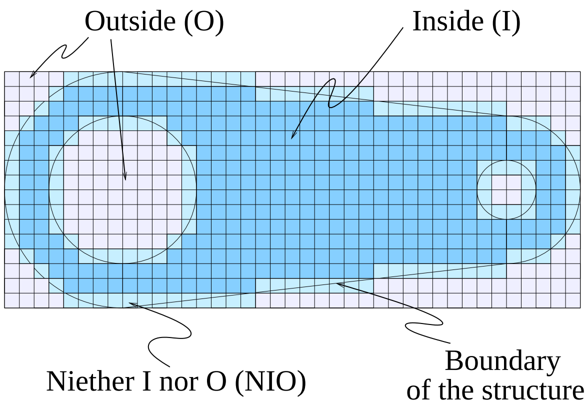
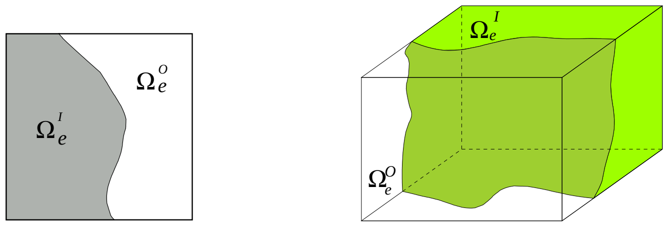


Fig. 3 Fixed grid approximation of the geometry of a structure showing the different types of finite elements

According to the position of the elements with respect to the structure, they can be classified in three different types: an element e_m is called **Inside** (I) if for all $\mathbf{x} \in e_m, \mathbf{x} \in \Omega$. An element is called **Outside** (O) if for all $\mathbf{x} \in e_m, \mathbf{x} \notin \Omega$, that is, the element consists of only points external to Ω. An element is called **Neither Inside nor Outside** (NIO) if there exist points $\mathbf{x}, \mathbf{y} \in e_m$ such that $\mathbf{x} \in \Omega$ and $\mathbf{y} \notin \Omega$. That means, the element has points inside as well as outside of Ω (these are elements on the boundary of the structure). Figure 3 illustrates the different types of elements for a given structure. These elements differ only in their material properties. Elements I have the material properties of the structure, elements O the material properties of a non-active medium, and elements NIO have both material properties.
This representation of the domain facilitates the process of analysis as it does not require sophisticated and high order algorithms to generate the mesh. Furthermore, it will be shown that the local stiffness matrix is the same for all the elements and it only needs to be computed once for the whole analysis
According to the type of nodes, n_j, in an element it can be classified as

$$\text{type}(e_m) = \begin{cases} I & \text{if } n_j \in e_i \quad \Rightarrow n_j \in \Omega \\ O & \text{if } n_j \in e_i \quad \Rightarrow n_j \notin \Omega \\ NIO & \text{Otherwise} \end{cases} \tag{1}$$

This criteria is not exact as there are cases where the element is classified as I or O when it actually is NIO. These cases are presented when the B–Rep of the object intersects the element at the face of the cube



(a) Two-dimensional NIO element

(b) Three-dimensional NIO element

Fig. 4 NIO Element. An element with discontinuous material properties

without touching any of the vertices. For example, a cone end or a pyramid vertex penetrating the face of an element. These cases are the result of small details in the geometry of the object which are not captured by the fixed grid resolution. The important point to notice here is that it may be desirable or not to capture the detail of the geometry for analysis purposes. In general, CAD data needs to be *defeatured* by removing excessive detail that only add noise to the analysis. This is of particular interest at the initial stages of the design [11].

GENERAL DESCRIPTION OF FIXED GRID GENERATION

A Finite Element preprocessor takes the geometry of a structure usually represented by its boundary representation (B–Rep) and subdivides it into a set of finite elements of different shapes and sizes. A fixed grid preprocessor is a specialised version of a finite element preprocessor where all the elements have fixed geometry but different physical properties. Furthermore, the boundary conditions must be properly codified into this new representation. The fixed grid preprocessing or conversion of the B–Rep geometry into a fixed grid representation is accomplished in the following steps: (*i*) Fixed grid domain computation, (*ii*) node classification, (*iii*) B–Rep subdivision to suit the element size, (*iv*) element classification (*I*, *O*, *NIO*), (*v*) computation of NIO geometry and volume, and (*vi*) boundary condition assignment.

FIXED GRID FINITE ELEMENT ANALYSIS

The discrete form of finite element formulation for linear elastic material can be stated as

$$[K]\{u_S\} - \{F\} = 0 \tag{2}$$

where $[K]$ is the stiffness matrix of the system, u_S is the vector of displacements and F represent the vector of forces. Additionally, the stiffness matrix $[K]$ can be constructed from element stiffness matrices $\left[K^{(e)}\right]$ as,

$$[K] = \sum_{e=1}^{E}\left[K^{(e)}\right] = \sum_{e=1}^{E}\int_{\Omega^{(e)}}[B]^T[C][B]\,d\Omega. \tag{3}$$

where $[B]$ is a matrix containing derivatives of the shape functions and $[C]$ is the tensor of material properties. For isotropic materials $[C]$ is given by

$$[C] = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{2}-\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{2}-\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{2}-\nu \end{bmatrix} \tag{4}$$

with E the Young modulus and ν the Poisson's ratio.
Computation of integrals in (3) depends upon the shape and material of the element. Due to its complexity the integrals are usually calculated by using iso-parametric elements and numerical integration [1].

1. The Fixed Grid method

In traditional FEA the domain is subdivided into a set of elements that fit the its shape. The most common algorithm to subdivided the domain is known as Delaunay triangulation. The result of such algorithms is a set of irregular shaped elements. Construction of stiffness matrix $[K]$ implies computation of the integral defined in (3) for each elements in the mesh. In contrast, all the elements of a FG have the same shape but different material properties. However, as it was show in previous section there are only three types of elements: I, O, and NIO. The following section develops the computation of the stiffness matrix for the homogeneous I and O elements and then for the non-homogeneous NIO elements.

2. Stiffness matrix for I and O elements.

From (4) it can be observed that $[C]$ depends only from the Young modulus and the Poisson's ratio. Then, it is possible to express $[C]$ as the sum of two matrices $[F']$ and $[G']$ in the following way,

$$[C] = k\left(\nu\,[F'] + [G']\right) \tag{5}$$

where $k = \dfrac{E}{(1+\nu)(1-2\nu)}$, and

$$[F'] = \nu \begin{bmatrix} -1 & 1 & 1 & 0 & 0 & 0 \\ 1 & -1 & 1 & 0 & 0 & 0 \\ 1 & 1 & -1 & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 & 0 & 0 \\ 0 & 0 & 0 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 & 0 & -1 \end{bmatrix}, \qquad [G'] = \begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{2} \end{bmatrix}. \tag{6}$$

Because of (5) it is possible to redefine (3) as

$$\begin{aligned} \left[K^{(e)}\right] &= \int_{\Omega^{(e)}} [B]^T \left[k\left(\nu\,[F'] + [G']\right)\right][B]\,d\Omega \\ &= k\left\{\nu \int_{\Omega^{(e)}} [B]^T [F'][B]\,d\Omega + \int_{\Omega^{(e)}} [B]^T [G'][B]\,d\Omega\right\} \\ &= k\left\{\nu \int_{\Omega^{(e)}} [F]\,d\Omega + \int_{\Omega^{(e)}} [G]\,d\Omega\right\} \end{aligned} \tag{7}$$

where $[F] = [B]^T[F'][B]$ and $[G] = [B]^T[G'][B]$. Aditionally ν is taken out of the integral because it is constant over $\Omega^{(e)}$.
The computation of $\left[K^{(e)}\right]$ by (7) allows the separation between the mechanical properties of the material and the geometry of the element. Given that all the elements in the FG have the same geometry then the integrals over $[F]$ and $[G]$ are the same and need only to be computed once.

3. Stiffness matrix for NIO elements.

A typical NIO element is shown in figure 4. These elements intersect the boundary of the structure so they are subdivided between two parts $\Omega_e = \Omega_e^I \cup \Omega_e^O$. Here Ω_e^I represents the part of the element with I material properties and Ω_e^O the part of the element with O material properties. This discontinuity in $[C]$ represent a complexity in the computation of $[K_e]$. However, computation of $[K_e]$ can be approximated in different ways. Two possible approximations are presented next.

1) Discrete approximation - A0   This is the simplest way of computing the integrals and consist of approximating the NIOs elements as I or O depending of the amount of element inside the structure. Because the material properties are allowed to take only discrete values (I or O material properties) this approximation is referred as discrete approximation.
If $V = \text{volume}(\Omega_e)$, and $V_I = \text{volume}(\Omega^I_e)$ then, according to the A0 approximation, the material properties, E and ν, of an NIO element are given by

$$(E_{NIO}, \nu_{NIO}) = \begin{cases} (E_I, \nu_I) & \text{if} \quad V_I/V > 1/2 \\ (E_O, \nu_O) & \text{if} \quad V_I/V \leqslant 1/2 \end{cases} \tag{8}$$

Numerical experiments in the two dimensional case have showed that AO approximation presents a large error in the displacement and stress fields. To redue the error it is necesary to decreasing the size of the elements in the grid. As a consequence of this mesh refinement there will be an increasing computational cost of the analysis [2, 3].

2) Weighted average approximation - A1   This is a more precise, but still an approximate method to represent the elements on the boundary. NIO elements are constituted by two different materials (I and O materials). A1 approximation transforms the bi-material element into a homogeneously isotropic element with material properties that best simulate the non continuous element. Thus a property will be the weighted average of the I and O properties:

$$\nu_{NIO} = \nu_I \xi + \nu_O (1 - \xi) \tag{9}$$

$$E_{NIO} = E_I \xi + E_O (1 - \xi) \tag{10}$$

where ξ is equal to the radio $\xi = V_I/V$.

Finally, A0 and A1 approximate a NIO element as homogeneous elements by applying (8) and (9). Therefore, (7) can be used in both cases to compute the their element stiffness matrix.

NUMERICAL TEST

These examples show the capabilities of the FG method. Due to the lack of analytical solutions the FG method is compared with solutions obtained using commercial finite element software (COSMOS/M). For simplicity in all the cases the properties of the material were chosen to be $E = 1 \times 10^9$ and $\nu = 0.2$
In order to compare the Fixed Grid Finite Element Analysis (FG–FEA) against classical FEA avector$\mathbf{d}$ of displacements is defined as $d_i = \sqrt{u_1^2 + u_2^2 + u_3^2}$. That is d_i is the norm of the displacement $\mathbf{u}$ at node i. The displacement error can be defined as

$$E_r = \frac{\|\mathbf{d}_{\text{FG}} - \mathbf{d}_{\text{real}}\|_\infty}{\|\mathbf{d}_{\text{real}}\|_\infty}, \tag{11}$$

where $\mathbf{d}_{\text{real}}$ correspond to a theoretical result of the same experiment. This value can be obtained based on convergence analysis of the FEA solution and using a technique as Richardson extrapolation [2].

1. L beam example

This example consist of an L shaped beam. It is fixed at one extreme and loaded with a shear force at the other end as shown in Figure 5. The maximum displacement was found to be located along the line formed by points $(75, 0, 100)$ and $(75, 25, 100)$. Applying Richardson extrapolation to the displacement found by Cosmos/M, a value of 5.84930×10^{-5} was obtained. This value was compared with the results obtained with different densities of grids. These are are shown in table 1.
In this test, the only difference with classical FEA is the inclusion of O elements into the stiffness matrix. In spite of the fact that the material properties of the Oelements are chosen to be those of non-active media, they can not be chosen as zero because this will result into singularities of the system matrix. As a consequence, inclusion of O elements will increase the stiffness of the overall structure. Nevertheless, it is observed that the error decreases as the density of the mesh does.

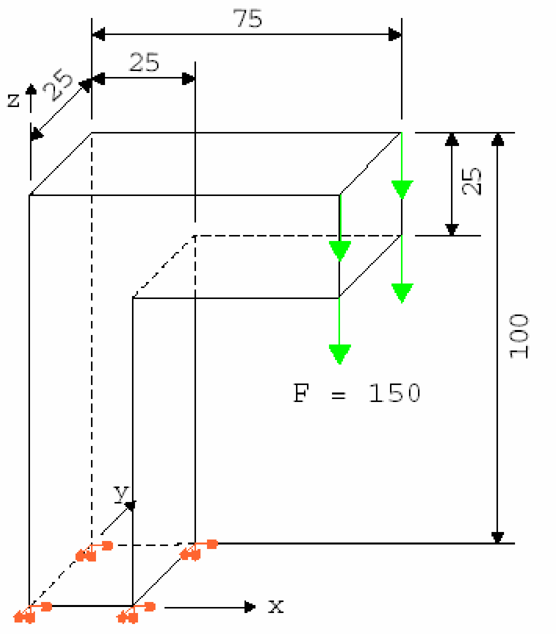


Fig. 5 Boundary conditions for the L beam

Table 1 Displacement error and comparison with classical FEA for the example of the L shaped beam

Mesh size	dof	$\|\mathbf{d}\|_\infty$ COSMOS/M	$\|\mathbf{d}\|_\infty$ FG–FEA	Error
3x1x4	108	6.322×10^{-6}	6.037×10^{-6}	89.68%
6x2x8	540	1.866×10^{-5}	1.766×10^{-5}	69.81%
12x4x16	3240	5.724×10^{-5}	5.385×10^{-5}	7.93%

1) Exclusion of *O* elements Due to the increased stiffness caused by the presence of the *O* elements the same test was accomplished without considering the *O* elements. As it was expected in this case, the result of the FG were equal to the obtained with classical FEA. However, when comparing the solution time of inclusion-exclusion of *O* elements, a reduction of 50% was obtain when *O* elements were excluded [12]. There are two factors that explain this time reduction: one is the reduction in the degrees of freedom of the system, and second, the reduction in the condition number of the stiffness matrix. The condition number is an indicator of how close to singularity a matrix is and has a consequence in the number of iterations used to find the solution when a preconditioned conjugate gradient method is used. The condition number for a system of equations $[K]u = f$ is defined by $\kappa(K) = \|K\|\|K^{-1}\|$. If $\kappa(K)$ is close to one then the matrix is well conditioned. Otherwise, if $\kappa(K)$ is large then the matrix is ill-conditioned. This condition number was calculated for different stiffness matrices of the L beam example. The results are shown in Table 2. It is observed a severe increase of the condition number when including the *O* elements in the solution.

Table 2 Comparison of the condition number for the stiffness matrix when including and excluding the O elements

	$\kappa_2(K)$	
Mesh size	Including O elements	Excluding O elements
2x2x1	9240.1790	79.9036
4x4x2	25438.5042	406.5693
8x8x4	66744.1386	1469.0489
16x16x8	256789.2165	5325.2153

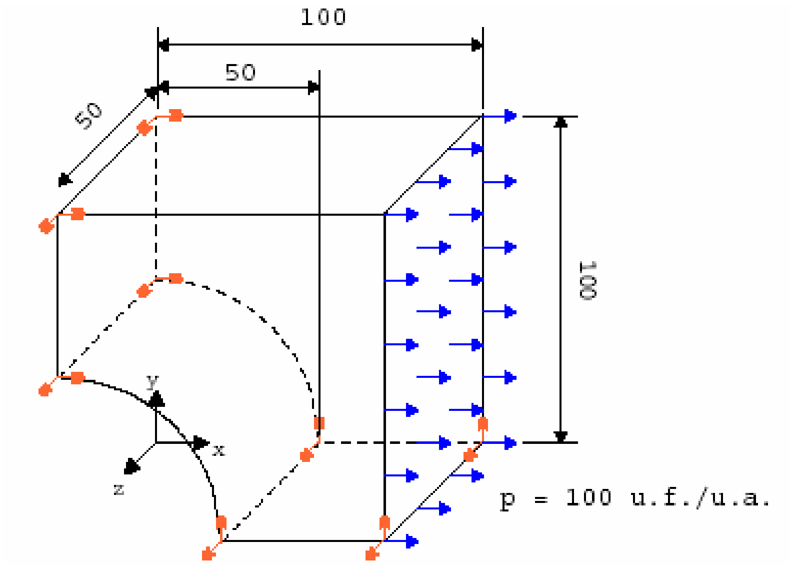


Fig. 6 Schematic view of a quarter of the square plate with a circular hole

Table 3 A0 approximation of NIO elements

	with O elements			*without O elements*		
Mesh size	dof	$\|\mathbf{d}\|_\infty$	Error	dof	$\|\mathbf{d}\|_\infty$	Error
4x4x2	189	3.7155×10^{-5}	3.1%	153	2.7838×10^{-5}	27.4%
8x8x4	1115	4.5586×10^{-5}	18.8%	890	4.0438×10^{-5}	5.4%
10x10x5	2034	4.4306×10^{-5}	15.5%	1638	4.4306×10^{-5}	15.5%
16x16x8	7479	4.0994×10^{-5}	6.8%	5967	3.6998×10^{-5}	3.5%
20x20x10	14069	4.3810×10^{-5}	14.2%	11231	4.3811×10^{-5}	14.2%

2. Square plate with a circular hole This numerical experiment uses a square plate with a circular hole. Figure 6 shows the dimensions and boundary conditions of the structure. Due to the symmetry of the problem, only a quarter of the object is analysed.

The test was intended to observe the behaviour of the method when modelling as tructure with NIO elements. The test also considers the inclusion of O elements into the construction of the stiffness matrix. Using a FEA solver, a maximum displacement of $3.835{\times}10^{-5}$ was found along the line formed by points (100, 0, 0) and (100, 0, 50). (The FEA solver used a mesh of 6750 elements, 7936 nodes and 22784 degrees of freedom). This calculated value was used to determine the displacement error of the FG method. The results are presented in Table 3 for A0 approximation and in Table 4 for A1 approximation. Similar results were obtained in both cases. It is observed that the error has a marked oscillatory behaviour. However, it does decreased as the element size decreased.

Table 4 A1 approximation of NIO elements

	with O elements			*without O elements*		
Mesh size	dof	$\|\mathbf{d}\|_\infty$	Error	dof	$\|\mathbf{d}\|_\infty$	Error
4x4x2	189	3.4258×10^{-5}	10.6%	153	2.5990×10^{-5}	32.2%
8x8x4	1115	4.0578×10^{-5}	5.8%	890	3.0810×10^{-5}	19.6%
10x10x5	2034	4.4306×10^{-5}	15.5%	1638	4.4306×10^{-5}	15.5%
16x16x8	7479	3.8112×10^{-5}	0.6%	5967	3.1666×10^{-5}	17.4%
20x20x10	14069	4.3810×10^{-5}	14.2%	11231	4.3811×10^{-5}	14.2%

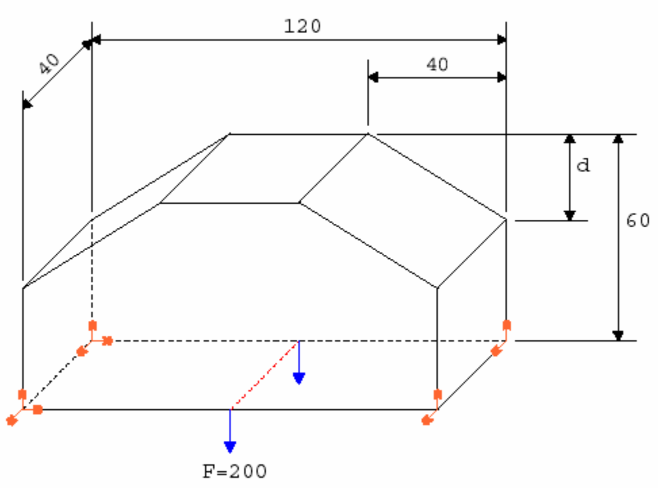


Fig. 7 Geometry and boundary conditions for structure used to test reanalysis capabilities of the FG method

3. Re-analysis The last numerical experiment deals with re-analysis using the FG method. Figure 7 shows the structure and boundary conditions used. The geometry of the object is expressed in terms of the parameter *d*. Variations to its dimension will result into different geometries. Initially a value of *d* = 10 was used. Then it was change to *d* = 20 and *d* = 30. The problem was solved for the initial case and then the stiffness matrix was modified to suit the structure with *d* = 20 and *d* = 30. The results are summarised in table 5. They are compared with the results obtained using Cosmos/M FEA software. The reanalysis procedure showed savings in time of 45% in the first case and 28% in the second case.

Table 5 Re-analysis test and comparison with FEA

case	$u_{r-\text{Cosmos}}$	u_r	Error	Time [s]
1	2.02801×10^{-7}	2.01282×10^{-7}	0.7493%	9.28519
2	2.08286×10^{-7}	2.17141×10^{-7}	4.2511%	5.07962
3	2.17406×10^{-7}	2.17094×10^{-7}	0.1437%	6.67140

CONCLUSIONS

This article presents a method for numerical analysis using a fixed grid three-dimensional domain. The program developed takes a structure previously constructed with a conventional solid modeller program and produces its fixed grid representation. Special care is taken when obtaining the intersection of the object with the grid in such a way that the elements preserve the geometry of the object. The stiffness matrix of the system is obtained as a function of a unique element stiffness matrix thus saving time in its assembling.

The displacement error obtained in the numerical test was found to be from 8 to 15%. Therefore, the usefulness of method as a fast estimator of the displacement and stress fields is observed. However to obtain a good accuracy of the solution a large number of elements is required.

Inclusion of *O* elements in the construction of the stiffness matrix increases the degrees of freedom of the system and produces ill-conditioned matrix. The method is not suitable to analyse structures whose shape is similar to a shell or thin plate. In these cases the number of elements required to properly model the geometry is too large and makes impractical its applicability. Finally, the results presented here are considered preliminary and therefore it is necessary to accomplish a more extensive testing.

Acknowledgements The support by EAFIT-COLCIENCIAS under contract 572-2003 is gratefully acknowledged.

REFERENCES

[1] M. J. Fagan, *Finite element analysis*. Longman, (1992).

[2] Manuel J. Garcýa. *Fixed grid finite element analysis in structural design and optimization*, Aeronautical engineering, The University of Sydney, Sydney, Australia, (1999).

[3] Manuel J. Garcýa, Grant P. Steven, *Interactive aerospace design using fixed grid finite element analysis*, in International Aerospace Congress, Sydney, February, (1997).

[4] M. J. Garcýa, C. A. Gonzalez, *Shape optimization of continuum structures via evolution strategies and fixed grid finite element analysis*, Journal of Structural and Multidisciplinary Optimization, 26(1-2), (2004), 92-98.

[5] Manuel J. Garcýa, Grant P. Steven, *Displacement error for fixed grid finite FEA elasticity problems*, in III Congreso Colombiano en Elementos Finitos y Modelación Matemática, Medellin, October 10-11, (1996).

[6] Manuel J. Garcýa, Grant P. Steven, *Optimisation of structures by using fixed grid representation of the finite element domain*. In Australasian Conference on Structural Optimisation, Sydney, Australia, February, (1998).

[7] Manuel J. Garcýa, Grant P. Steven, *Fixed grid finite elements in elasticity problems*, Engineering Computations, 16(2), (1999), 145-164.

[8] Manuel J. Garcýa, Grant P. Steven, *Fixed grid finite element analysis in structural design and optimization*, Second ISSMO/AIAA Internet Conference on Approximations and Fast Reanalysis in Engineering Optimization, Delft, 25 May-2 June, (2000).

[9] H. Kim, M. J. Garcýa, O. M. Querin, G. P. Steven, Y. M Xie, *Fixed grid finite element analysis in evolutionary structural optimization*, Engineering Computations, 17(4), (2000), 427-439.

[10] H. Kim, M. J. Garcýa, O. M. Querin, G. P. Steven, Y. M Xie, *Fixed grid finite element analysis in evolutionary structural optimization*, Engineering Computations, 17(4), (2000), 427-439.

[11] Anton V. Mobley, Michael P. Carroll, Scott A. Canann, *An object oriented approach to geometry defeaturing for finite element meshing*, In 7th International Meshing Roundtable, Sandia National Labs, Dearborn, Michigan, U.S.A., October 26-28, (1998), pp. 547-563.

[12] Luis Miguel Ruiz, *Elementos Finitos de Malla Fija en tres Dimensiones*, Tesis Pregrado Universidad EAFIT, (2001).

[13] Katsuyuki Suzuki, Hideomi Ohtsubo, Kenjiro Terada, *The analysis of 3d solid using multi-scale voxel data*, in IV World Congress on Computational Mechanics, Buenos Aires, Argentina, June, (1998).

[14] S. Y. Woon, O. M. Querin, G. P. Steven, *Application of the fixed-grid FEA method to step-wise GA shape optimisation*, In Engineering Design Optimization, Proceedings of second ASMO–UK /ISSMO conference, Swansea, UK, (2000), pp. 265-272.