Mecánica Computacional Vol. XXI, pp. 3037–3047
S. R. Idelsohn, V. E. Sonzogni and A. Cardona (Eds.)
Santa Fe–Paraná, Argentina, October 2002

OPTIMIZATION OF A MESH GENERATION TECHNIQUE FOR PIPE CONNECTIONS

Claudio E. Jouglard[*] and José M. Pereiras[†]

[*] Laboratorio de Mecánica Computacional. Departamento de Física.
Facultad de Ingeniería. Universidad de Buenos Aires.
Av. Paseo Colón 850, (1063) Buenos Aires, Argentina
e-mail: cjougla@fi.uba.ar.

[†] Departamento de Computación.
Facultad de Ingeniería. Universidad de Buenos Aires.
Av. Paseo Colón 850, (1063) Buenos Aires, Argentina
e-mail: jpereira@fi.uba.ar.

Key words: Mesh Generation, Refinement Indicators, Pipe Connections.

Abstract. *In this work we present an optimization of a mesh generation technique for pipe connections. Several stress analyses are made on these types of connections using a commercial finite element code with meshes supplied by the user. To facilitate the generation of these meshes a program was developed that generates a basic parameterized mesh composed of quadrilateral superelements, which are refined for each analysis. The user must specify the number of subdivisions in each superelement direction for each superelement. To optimize this process, minimizing user interventions, we employ isotropic and anisotropic refinement indicators. In the oil industry an important point is the analysis of the contact pressures at the connections. In the numerical examples presented we can appreciate the importance of the mesh refinement procedure to obtain accurate values of these pressures.*

1 INTRODUCTION

Although nowadays we have powerful CAD software that simplifies the generation of finite element meshes, except for very simple cases the construction of these meshes can be a very laborious and time consuming task. So, in general, an initial mesh is provided with a resolution according to the *user's experience* and this mesh is used till the end of the analyses. Most popular commercial codes have few tools, if any, to make *automatic* mesh refinements. So, It is very infrequent to refine this initial mesh to verify the accuracy of the results. A simple strategy could be to refine uniformly all the elements of the initial mesh, but this can be very costly, especially in 3D problems. A more rational strategy consists in refine the regions of the mesh that have more *error*. But to implement this adaptive refinement approach we need error and refinement indicators, which are not, in general, provided by standard finite element codes.

The analysis of pipe connections is very important in the oil industry. Several numerical studies are needed to evaluate the quality of these connections using finite element programs with highly nonlinear capabilities. A preprocessor has been specifically developed for the generation of the meshes needed to model pipe connections. This preprocessor generates a basic parameterized mesh composed of quadrilateral superelements. The user must specify the number of subdivisions in each superelement direction for each superelement.

To optimize the definition of a mesh with sufficient resolution a series of indicators have been added to this preprocessor. In this work we describe the implementation of these indicators and present some examples showing the improvement of the results.

2 MESH REFINEMENT INDICATORS

The *discretization error* is defined as the difference between the exact and finite element solutions, i.e.

$$\boldsymbol{e} = \boldsymbol{u} - \boldsymbol{u}_h \tag{1}$$

The specification of a local error in this manner is generally not convenient to identify the overall quality of the solution. For this reason various ‘norms’ representing some integral scalar quantity are often introduced to measure the error. A common measure is the ‘energy norm’. The error in the energy norm is defined as

$$\| \mathbf{u} \| = \left(\int_{\Omega} \boldsymbol{\varepsilon}^T \mathbf{D}\, \boldsymbol{\varepsilon}\, d\Omega \right)^{1/2} = \left(\int_{\Omega} (\mathbf{Su})^T \mathbf{D} (\mathbf{Su})\, d\Omega \right)^{1/2} \tag{2}$$

where $\boldsymbol{S}$ is the first order strain differential operator which defines the strains $\boldsymbol{\varepsilon}$ as

$$\varepsilon = \boldsymbol{S}\, \boldsymbol{u} \tag{3}$$

and where $\boldsymbol{D}$ is the matrix of elastic constants which relates the strain vector $\boldsymbol{\varepsilon}$ with the stress vector $\boldsymbol{\sigma}$ as

$$\sigma = \boldsymbol{D}\varepsilon = \boldsymbol{DS}\, \boldsymbol{u} \tag{4}$$

Then the error in the energy norm is

$$\|\mathbf{e}\| = \left(\int_{\Omega} (\mathbf{Se})^T \mathbf{D} (\mathbf{Se}) \, \mathrm{d}\Omega \right)^{1/2} \tag{5}$$

The approximate displacements $\boldsymbol{u}_h$ gives an approximate stress field $\boldsymbol{\sigma}_h$ as

$$\boldsymbol{\sigma}_h = \boldsymbol{DS}\, \boldsymbol{u}_h \tag{6}$$

Then the error in the energy norm can be written as

$$\|\mathbf{e}\| = \left(\int_{\Omega} (\boldsymbol{\sigma} - \boldsymbol{\sigma}_h)^T \mathbf{D}^{-1} (\boldsymbol{\sigma} - \boldsymbol{\sigma}_h) \, \mathrm{d}\Omega \right)^{1/2} \tag{7}$$

The error in the energy norm can be evaluated by summing over all the elements as

$$\|\mathbf{e}\|^2 = \sum_{i=1}^{nel} \left(\|\mathbf{e}\|_i \right)^2 \tag{8}$$

In an optimal constructed mesh the error is equally distributed over all the elements, this is known as the *principle of equidistribution of error* [1,2,3]. Following this principle the error must be approximately equal on each element. Taking into account that for energy norms only the square of these norms are additive, then for an optimal mesh of *ne* elements we must have for each element an ideal mean error e_m

$$e_m \approx \|\mathbf{e}\| / \sqrt{ne} \tag{9}$$

Then we can define for each element *i* a refinement ratio ξ_i as

$$\xi_i = \frac{\|\mathbf{e}\|_i}{e_m} \tag{10}$$

which defines the elements to be refined when $\xi_i > 1$.

It is found [4] that for an optimal mesh if the mesh is uniformly refined with the size of the element, *h*, tending to zero while the polynomial degree *p* of the approximation is fixed the following estimate can be established

$$\|\mathbf{e}\| \le C h^p \tag{11}$$

To construct an optimal mesh we need a *mesh-size function h(x,y)* which predicts the optimal mesh-size h_{new} for each element. If the current element size is h_{old} , and we assume the rate of convergence of the error to be optimal $O(h^p)$ then the predicted element size should be

$$h_{new} = h_{old} / (\xi_i)^{1/p} \quad , \qquad \text{for } \xi_i > 1 \tag{12}$$

This expression can be used too when $\xi_i < 1$ indicating that in some areas a coarser subdivision is permissible.

It should be noted that the indicators could be limited only to certain regions of the mesh by limiting the sum in (8) to the elements in these regions.

These indicators are isotropic, since they do not give any preferential direction to the refinement. But it is possible to obtain directional indicators that give for each element the main directions of refinement, that is, how to the element must be stretched to minimize the error of the approximation.

In the neighborhood of a point x_0 the function u can be expressed by a Taylor's series as

$$u(x) = u(x_0) + \nabla u(x_0)(x - x_0) + \tfrac{1}{2}\boldsymbol{H}\big(u(x_0)\big)(x - x_0)(x - x_0) + \cdots \tag{13}$$

where $\boldsymbol{H}$ is the *Hessian matrix* defined as

$$\boldsymbol{H}(u) = \begin{bmatrix} \dfrac{\partial^2 u}{\partial x^2} & \dfrac{\partial^2 u}{\partial x \partial y} \\ \dfrac{\partial^2 u}{\partial x \partial y} & \dfrac{\partial^2 u}{\partial y^2} \end{bmatrix} \tag{14}$$

and

$$\boldsymbol{H}(u)(x - x_0)(x - x_0) = \frac{\partial^2 u}{\partial x^2}(x - x_0)^2 + 2\frac{\partial^2 u}{\partial x \partial y}(x - x_0)(y - y_0) + \frac{\partial^2 u}{\partial y^2}(y - y_0)^2 \tag{15}$$

Assuming that u_h is a good approximation to the linear and constant part of the exact solution, that is

$$u(x) \approx u(x_0) + \nabla u(x_0)(x - x_0) \tag{16}$$

then we have

$$u - u_h \approx \tfrac{1}{2}\boldsymbol{H}\big(u(x_0)\big)(x - x_0)(x - x_0) \tag{17}$$

From this expression we can see that the error depends on the direction (x- x_0), that is, there exist directions where the error is maximum and others where the error is minimum. These directions are associated with the eigenvectors of matrix $\boldsymbol{H}$. But this matrix is not positive definite, so it can not be used as a metric, then is convenient to use a tensor $\boldsymbol{G}$ defined as [5]

$$\boldsymbol{G} = \boldsymbol{Q}\,\Lambda\,\boldsymbol{Q}^t \tag{18}$$

where

$$\boldsymbol{Q} = [\mathbf{d}_1 \quad \mathbf{d}_2] \tag{19}$$

is the matrix of eigenvectors of matrix $\boldsymbol{H}$ and

$$\Lambda = \begin{bmatrix} |\lambda_1| & 0 \\ 0 & |\lambda_2| \end{bmatrix} \tag{20}$$

is the matrix of the absolute values of the eigenvalues λ_1, λ_2 of the Hessian matrix.

The direction $\mathbf{d}_1$ is associated with the direction of minimum curvature or minimum error, and the direction $\mathbf{d}_2$ is associated with the direction of maximum curvature or maximum error. So, ideally the element must be stretched along direction $\mathbf{d}_1$ to optimize the error distribution on the mesh.

For elasticity problems the solution $\boldsymbol{u}=(u,v)$ is vectorial, so we use instead of $\boldsymbol{G}$ the next tensor

$$\boldsymbol{G} = \sqrt{\boldsymbol{H}_1^t \boldsymbol{H}_1 + \boldsymbol{H}_2^t \boldsymbol{H}_2} \tag{21}$$

where $\boldsymbol{H}_1 = \boldsymbol{H}(u)$ and $\boldsymbol{H}_2 = \boldsymbol{H}(v)$.

3 ERROR ESTIMATORS

Since the exact solution is unknown we must estimate the error of the finite element solution u_h. Here we adopt error estimators based on the use of post-processing techniques to recover stress fields that are expected to be more precise than those obtained by direct derivation of the finite element displacement fields.

Then we can replace the exact stress σ by the recovered stress $\boldsymbol{\sigma}^*$ to obtain an error estimator $\mathbf{e}^*$ in the energy norm as

$$\| \mathbf{e}^* \| = \left(\int_\Omega \left(\boldsymbol{\sigma}^* - \boldsymbol{\sigma}_h \right)^T \mathbf{D}^{-1} \left(\boldsymbol{\sigma}^* - \boldsymbol{\sigma}_h \right) \mathrm{d}\Omega \right)^{1/2} \tag{22}$$

An easy way of obtaining a smoothed stress field for linear elements consists in averaging nodal stresses from the elements connected at each node

$$\boldsymbol{\sigma}^* = \frac{\left(\sum_i^{nelem} \boldsymbol{\sigma}_h^i \right)}{n} \tag{23}$$

where n is the number of elements surrounding the node.

The recovered nodal values are influenced by the size of the elements attached to the node and better results are obtained when these elements have similar sizes, especially in regions of high gradients.

To obtain the directional indicators we must approximate the second derivatives of the displacement field. In this case we proceed in two steps, first we obtain a smoothed field for the first derivatives by nodal averaging of the centroidal derivatives of each element surrounding the node, that is

Then we interpolate these values of nodal derivatives with the same nodal functions used for displacements, and the second derivatives are obtained by deriving these fields at the element center.

4 NUMERICAL EXAMPLES

The connection analyzed consists of two pipes with threaded ends connected by a threaded box. Due to symmetry only one half of the connection is modeled. In figure 1 we can see a mesh for the complete model. Two loading cases are analyzed: make up and compression. During make up the pipe is subjected to an axial displacement due to the adjustments during the mounting of the connection and in the compression case the pipe is subjected to compression forces in service.

Figure 1: Complete model.

One important point in the analysis of pipe connections is the study of the sealing, and for this we must known the contact pressures developed in the zone of the seal. In figure 2 we can see a detail of the region of the connection and in figure 3 we can see a detail of the region of the seal.

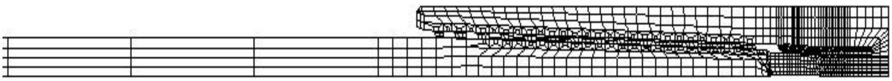

Figure 2: Detail of the region of the connection.

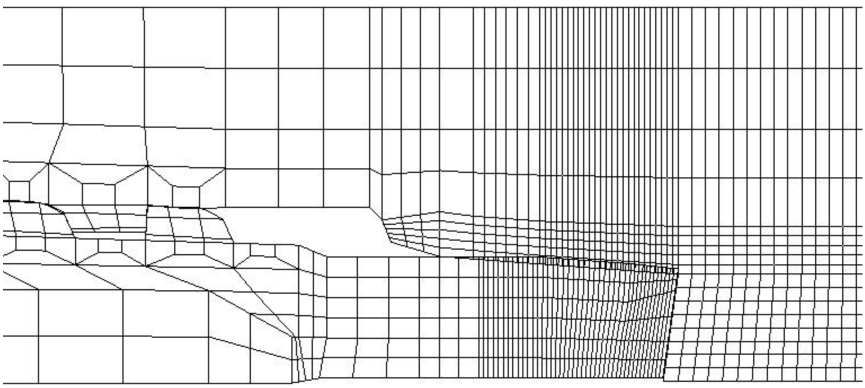

Figure 3: Detail of the region of the sealing.

In figure 4 we can see the variation of the refinement indicator ξ_i for the original mesh and two refinements for make up loading. Only are indicated the elements that must be refined (ξ_i>1), and all the mesh has been considered to equidistribute the error.

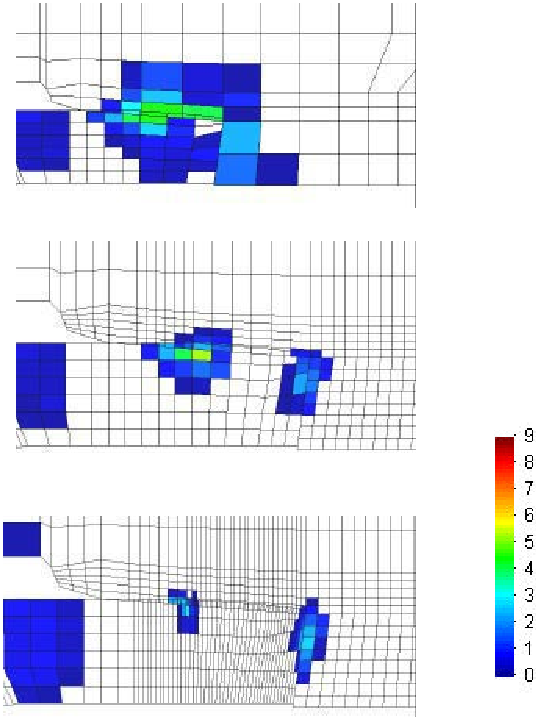

Figure 4: Refinement indicator for all the mesh. Detail in the region of the sealing.

In figure 5 we can see the variation of the refinement indicator ξ_i for the original mesh and two refinements for make up loading. Only are indicated the elements that must be refined (ξ_i>1), but in this case only the elements of the region of the seal has been considered to equidistribute the error. The meshes obtained in this case are similar to those of the previous case indicating a negligible influence of the error in other regions of the mesh for equidistribution of the error in the region of the seal.

Of utmost importance is the distribution of the contact pressures in the region of the seal. We can see in figure 6 the region considered for the analyses, and in figure 7 we can see the variation of the distribution of these pressures with the refinements, and we can appreciate the great influence of the refinements in this distribution.

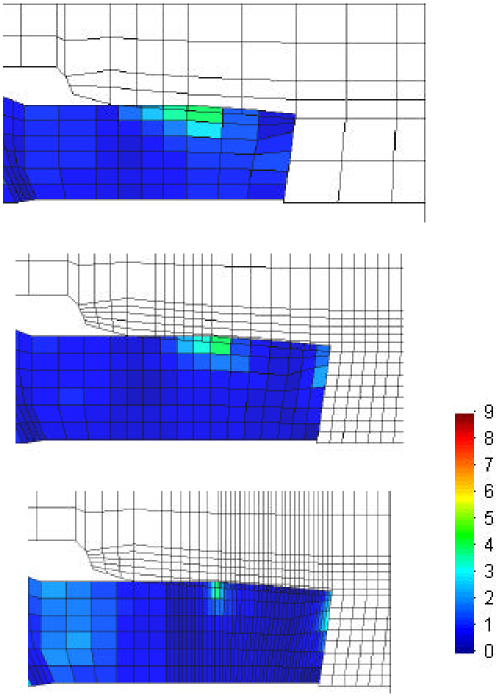

Figure 5: Refinement indicator for the region of the sealing.

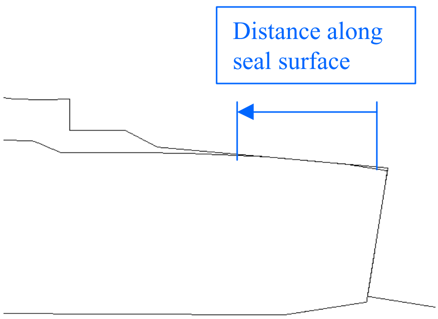

Figure 6: Region for distribution of contact pressures.

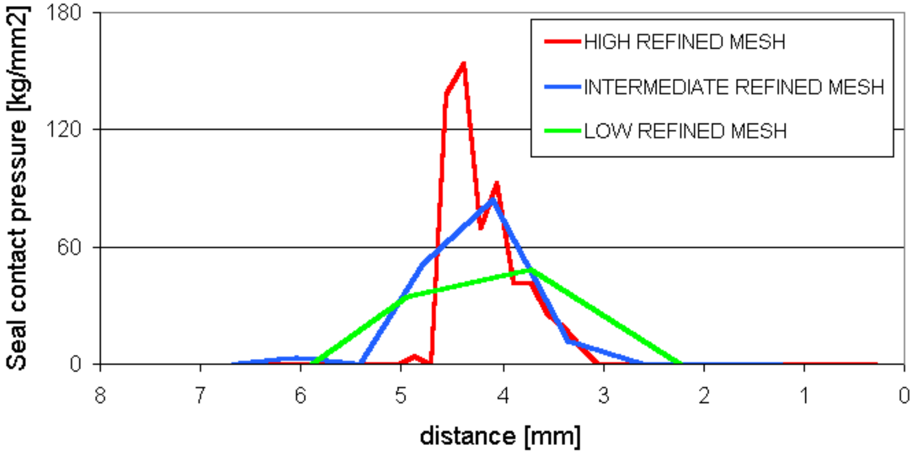

Figure 7: Variation of contact pressures with refinements.

Other region of interest is the zone of the threads where the refinement is important to capture the necking effect when subject to traction at large deformations. In figure 8 the refinement indicators for this region are shown.

Figure 8: Refinement indicators in the region of the threads.

In figure 9 the directions of minimum curvature weighted with the refinement indicator are shown for the elements in the region of the seal for the coarse mesh and in figure 10 the same indicators are shown for the refined mesh. The arrows indicate the direction of stretching of the elements for an optimal distribution of the error in the mesh.

5 CONCLUSIONS

The results show that the use of mesh refinement is necessary to obtain a precise description of the distribution of the contact pressure in the region of the seal. Much work is needed in relation with the directional indicator to implement anisotropic refinement of the elements, that is with different subdivisions of its sides. In the present implementation of the mesh refinement procedure we have not made any changes to the data structures of the finite element solver. In this sense the current implementation is solver independent, but for a complete automatic implementation of adaptive mesh refinements the data structures of the solver should be modified.

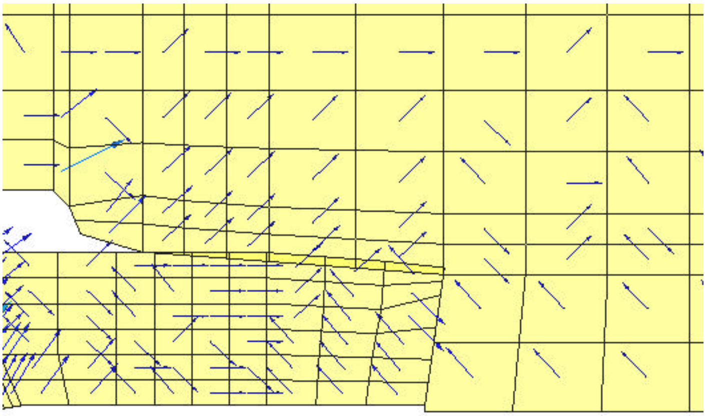

Figure 9: Directional refinement indicators for the coarse mesh.

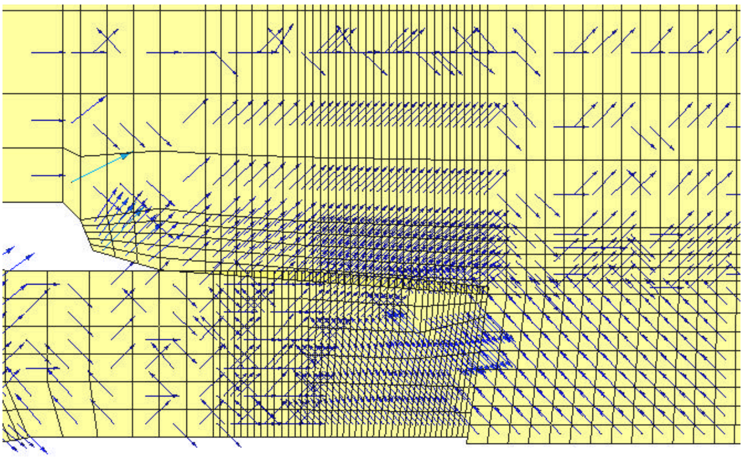

Figure 10: Directional refinement indicators for the refined mesh.

6 REFERENCES

[1] L. Demkowicz, P. Devloo and J.T. Oden, "On h-type mesh-refinement strategy based on minimization of interpolation error", *Comput. Meths. Appl. Mech. Eng.*, **53**, 67-89 (1985).

[2] I. Babuska and W.C. Rheinboldt, "Analysis of optimal finite-element meshes in R^1", *Math.*

Comp., **33**, 435-463 (1979).

[3] A.R. Diaz, N. Kikuchi and J.E. Taylor, "A method of grid optimization for finite element methods", *Comput. Meths. Appl. Mech. Eng.*, **41**, 29-45 (1983).

[4] J.Z. Zhu and O.C. Zienkiewicz, "Adaptive techniques in the finite element method", *Commun. Appl. Numer. Methods,* **4**, 197-204 (1988).

[5] J. Peraire, K. Morgan and J. Peiró, "Unstructured mesh methods for CFD", I.C. Aero Report 90-40, Department of Aeronautics, Imperial College of Science Technology and Medicines (1990).