6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
1115 June 2018, Glasgow, UK

STABILIZATION METHOD FOR CONTACT PROBLEMS UNDER THE CARTESIAN GRID-BASED FINITE ELEMENT METHOD FRAMEWORK

E. Nadal, J.M. Navarro-Jiménez, M. Tur, J. Albelda and J.J Ródenas

Centro de Investigación en Ingeniería Mecánica (CIIM), Universitat Politècnica de València, Camino de Vera s/n (Edificio 5E), 46022 Valencia, Spain. {ennaso,jonaji}@upvnet.upv.es, {matuva,jalbelda,jjrodena}@mcm.upv.es

Key words: Immersed boundary, Cartesian grid, Stabilization, Contact problems

Abstract. When using Immersed Boundary methods for solving the elasticity problem, some nodes are outside of the problem domain with a small associated stiffness in comparison to the nodes placed into the problem domain. This characteristic issue of the Immersed Boundary methods yields ill-conditioning problems when solving the global system of equations. The main reason behind this behaviour is the fact that the energy contribution of the pathological nodes is small, therefore the global energy of the problem is only slightly affected by the solution of these nodes. This contribution proposes a method that adds an extra term to the formulation that stabilizes the solution of those pathological nodes. This new term consists in i) a stiffness-type matrix involving only these nodes (which value is related to the element size) for the LHS of the system and ii) a force directly applied to these nodes. The results show an improvement of the system matrix condition number and thus, a better performance of the iterative solvers. In addition, in the case of contact problems the ill-conditioning of the system matrix prevents the convergence for the contact problem. The numerical results show that the addition of the proposed stabilisation term allows to alleviate these kind of problems.

1 INTRODUCTION

Last decades in the XXth century, a parallel concept to the Finite Element Method (FEM) emerged, the Immerse Boundary Method (IBM) which according to [7] has its origins in the paper published by VK Saul'ev in Russian *Solution of certain boundary value problems on high-speed computers by the fictitious domain method* (Mat.Z. 1963.4:912-925). In the FEM framework the mesh is conforming with the geometry of the component. Therefore, the mesh generation complexity is directly related with that of the geometry. Besides the existence of advanced and automated mesh generators algorithms [10, 17], the meshing process is one of the most tedious processes of the FEM. On the contrary, the IBM completely separates the mesh used for solving the Finite Element (FE) problem from the geometry of the component, therefore the geometrical complexity is completely unrelated with the mesh generation process, which is, in fact, usually octree-based. Since the FE mesh is not related with the geometry, an special treatment of the boundary is needed in the IBM. This important issue is a key ingredient to differentiate the IBM approaches such as the CutFEM [6], using the Level Set method to define the geometry of the component, the Finite Cell Method [15] using special integration methods or the cgFEM [21, 19] which is able to take into account the exact geometry by using special integration algorithms.

Independently of the approach, most of these methods increase some difficulties with respect to classical FEM in similar aspects: i.e. imposing Dirichlet boundary bonditions, numerical conditioning, accuracy over the boundary, etc. CutFEM [6] proposes a robust methodology to guarantee the stability when Dirichlet boundaries cut the mesh resulting in very small element subregions. cgFEM [21, 19] also uses stabilization methods for imposing Dirichlet boundary conditions guaranteeing the coercitivity. These stabilization procedures are able to guarantee the solvability of the problem at hand for direct solvers. However, when iterative solvers are needed, not only the solvability must be guaranteed but also the condition number should be controlled in order to guarantee the convergence of the iterative solver.

In this paper we propose a procedure to keep under control the condition number of the system of equations under the cgFEM framework. The proposed method uses a displacements recovery procedure over the boundary to control the solution of the nodes outside the physical domain, thus avoiding degrees of freedom with a small stiffness associated. The proposed method has been tested for bilinear and biquadratic elements for the linear elasticity problem and for the contact problem, showing an improvement of the solution convergence.

2 PROBLEM STATEMENT

This paper is devoted to solve the 3D contact problem by means of the cgFEM. The notation used all along the contribution is settled at this point. The Cauchy stress field is denoted as $\boldsymbol{\sigma}$, the displacement field as $\mathbf{u}$, and the strain field as $\boldsymbol{\varepsilon}$, all these fields being defined over the domain $\Omega \subset \mathbb{R}^3$, with boundary denoted by $\partial\Omega$. Prescribed tractions denoted by $\mathbf{t}$ are imposed over the part Γ_N of the boundary, while displacements denoted by $\bar{\mathbf{u}}$ are prescribed over the part Γ_D of the boundary. Body loads are denoted as $\mathbf{b}$.

The linear elasticity problem takes the following variational from

$$
\begin{aligned}
&\text{Find } \mathbf{u} \in (V + \{\mathbf{w}\}) : \forall \mathbf{v} \in V \\
&a(\mathbf{u},\mathbf{v}) = l(\mathbf{v}) \qquad \text{where} \\
&a(\mathbf{u},\mathbf{v}) = \int_\Omega \boldsymbol{\varepsilon}(\mathbf{u})^T \mathbf{D} \boldsymbol{\varepsilon}(\mathbf{v}) \, \mathrm{d}\Omega \\
&l(\mathbf{v}) = \int_\Omega \mathbf{b}^T \mathbf{v} \, \mathrm{d}\Omega + \int_{\Gamma_N} \mathbf{t}^T \mathbf{v} \, \mathrm{d}\Gamma,
\end{aligned}
\tag{1}
$$

where $V = \{\mathbf{v} \mid \mathbf{v} \in [H^1(\Omega)]^3, \mathbf{v}|_{\Gamma_D} = \mathbf{0}\}$, $\mathbf{w}$ is a particular displacement field satisfying the Dirichlet boundary conditions and here the matrix $\mathbf{D}$ contains the elasticity coefficients of the usual linear isotropic constitutive law relating the stress field with strain field. For the contact problem, considering large deformations, we denote as Γ_C the contact surface, being $\Gamma_C^{(1)}$ the slave contact surface and $\Gamma_C^{(2)}$ the master contact surface. We use a ray-tracing technique [20] to define the contact point pairs, i.e. we intersect the master contact surface $\Gamma_C^{(2)}$ at $\mathbf{x}^{(2)}$ with a line emanating from $\mathbf{x}^{(1)}$ in the direction of the normal vector to the slave surface $\mathbf{n}^{(1)}$. Then the normal contact gap can be defined as:

$$g_N = \left(\mathbf{x}^{(2)} - \mathbf{x}^{(1)}\right) \cdot \mathbf{n}^{(1)}. \tag{2}$$

The complete formulation for the contact problem used in this manuscript is completely described in [20, 14].

2.1 BOUNDARY CONDITIONS IN cgFEM

When using the cgFEM for numerically solving the problem defined in section 2, we find out that the Dirichlet boundary conditions cannot be directly applied as in the standard FEM since, in general, there are not any nodes over the boundary. Therefore, a mortar method is used which weakly imposes the essential boundary conditions. In order to do that, a Lagrange multipliers discretization over the Dirichlet boundaries is needed. The choice of the Lagrange multipliers space is crucial for the well behaviour of the proposed method. Several works on that sense came out in the last years. Barbosa and Hughes [2, 3] propose stabilization methods in order to guarantee the Ladyzhenskaya-Babuška-Breezi (LBB) condition. Other authors [4, 13] instead, propose the Vital Vertex Method which *a prioiri* defines an appropriate discretization for the Lagrange multiplier space; however this procedure is not trivial for the 3D case. Also Hansbo *et. al.* [12] and Burman and Hansbo [8] propose an adaptation of the Nitsche's method to the IBM framework. In the context of the Finite Cell Method, the authors propose the use of a Nitsche's based approach for imposing the essential boundary conditions [18]. More recently Tur *et. al.* [21] propose a stabilization technique which makes use of recovery procedures easing the implementation of the method specially for the 3D case. This last method is adopted in this contribution. The weak imposition of Dirichlet boundary conditions via a mortar method implies the use of Lagrange multipliers, therefore solving problem (1) is equivalent to solve the following problem:

$$\mathscr{L}(\mathbf{v}^h, \boldsymbol{\mu}^h) = \frac{1}{2} a(\mathbf{v}^h, \mathbf{v}^h) + b(\boldsymbol{\mu}^h, \mathbf{v}^h - \mathbf{g}) - l(\mathbf{v}^h), \tag{3}$$

where $\mathbf{v}^h \in V^h$ is the discrete counterpart of the space V and $\boldsymbol{\mu}^h \in M^h$, a suitable discretized space for the Lagrange multipliers. Note that, in general, the appropriate Lagrange multipliers space is not easy to find since it is problem-dependent and it also depends on the way the mesh and geometry intersect. The approach followed in this contribution introduces a Lagrange multiplier at each integration point of the surface with a constant approximation space. This discretization is not suitable in general, therefore an additional stabilization term is added:

$$\mathscr{L}(\mathbf{v}^h, \boldsymbol{\mu}^h) = \mathscr{L}(\mathbf{v}^h, \boldsymbol{\mu}^h) - \frac{h}{k_1} \int_{\Gamma_D} \boldsymbol{\mu}^h \cdot (\boldsymbol{\lambda}^h - \mathbf{T}(\hat{\mathbf{u}}^h)) \, \mathrm{d}\Gamma, \tag{4}$$

where $k_1 > 1$ is parameter defined by the user and h is the characteristic mesh size. The interested reader is addressed to [21] for further details.

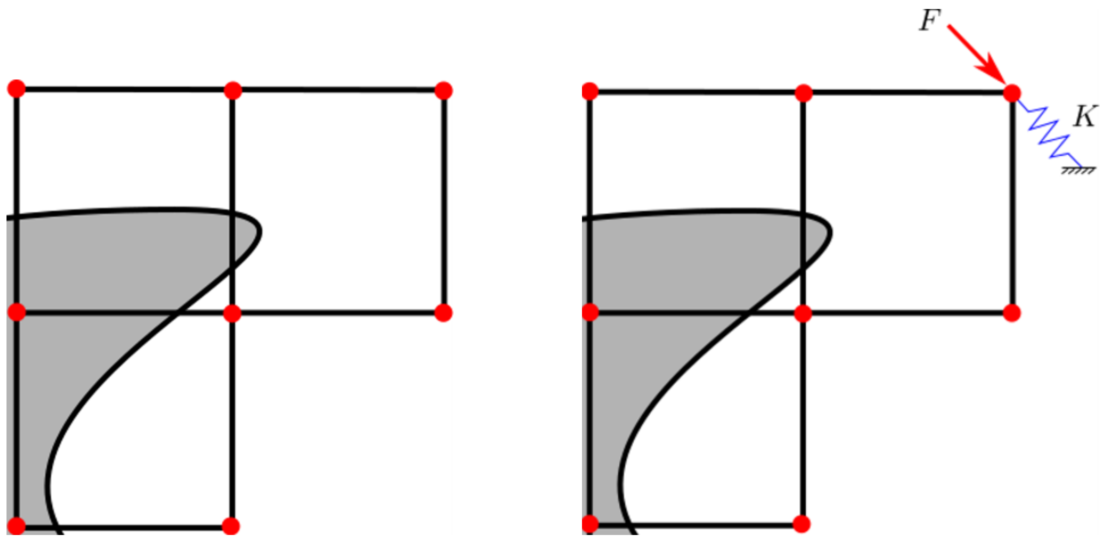
Expression (4) is similar to that used in the Nitsche's method in which the operator $\mathbf{T}(\hat{\mathbf{u}}^h)$ corresponds to the tractions over the Dirichlet boundary. Because of that, Nitsche's method results hard to implement. In the proposed approach, the operator $\mathbf{T}(\hat{\mathbf{u}}^h)$ is a post-process of the solution, guaranteeing the correct convergence of the method [21]. Further details about the evaluation of this operator are given in section 3.2. Problem (4) is already solvable and stable since essential boundary conditions are properly imposed.

However, depending on how the mesh and geometry intersect, it could happen that the problem becomes ill-conditioned. For small problems, this is not an important issue since direct solvers are able to solve them without major difficulty. In the case of bigger problems iterative solvers are used, and their convergence is strongly affected by the condition number of the system to solve [1]. In the case of IBM in general, and in particular in the case of cgFEM, there are some mesh configurations in which the position of the geometry boundary with respect to the nodes, specially the external nodes, provokes ill-conditioning issues, preventing the use of iterative solvers. Figure 1a shows an example in which the numerical problem is suitable to be ill-conditioned. If the stiffness associated to the nodes outside the domain is small this leads to an ill-conditioning of the system of equations. This is because the solution of those pathological nodes does not affect to the energy of the problem. That is, almost independently of the solution of those nodes the global energy remains the same. In other words, the sensibility of the energy to the variation of the solution of those nodes is small. To the authors knowledge, there is not any technique to face this problem in bibliography. Therefore in this contribution we pretend to introduce a new technique based on the use of recovered fields with the aim of, at least, partially solve the issue, common for the IBM. The proposed technique adds an additional term to the formulation according to equation (5)

$$\mathscr{L}(\mathbf{v}^h, \boldsymbol{\mu}^h) = \mathscr{L}(\mathbf{v}^h, \boldsymbol{\mu}^h) - \frac{h}{k_1} \int_{\Gamma_D} \boldsymbol{\mu}^h \cdot (\boldsymbol{\lambda}^h - \mathbf{T}(\hat{\mathbf{u}}^h))\, \mathrm{d}\Gamma - \frac{k_2 E}{h^2} \int_{\Omega^\star} \mathbf{v}^h \cdot (\mathbf{u}^h - \mathbf{S}(\hat{\mathbf{u}}^h))\, \mathrm{d}\Gamma, \quad (5)$$

where E is the Young's modulus and $k_2 > 0$ is also a user defined parameter.

The additional term in equation (5) penalises the "free" displacement of those pathological nodes. In fact, it introduces and artificial stiffness which is compensated with an artificial force as shown in figure 1b. In this term, the integration domain $\Omega^\star$ corresponds to the element containing the pathological node, not to the surface but to the volume. Thus, the integration domain now considers also the part of the element falling outside the problem domain. The operator $\mathbf{S}(\hat{\mathbf{u}}^h)$ corresponds to a displacements field obtained as a post-process of the FE solution $\mathbf{u}^h$. Further details of operator $\mathbf{S}$ are in section 3.4.



(a) Ill-conditioning issue due to mesh-geometry intersection.

(b) Effect of the additional term.

Figure 1: Scheme of an element subject to ill-conditioning issues. The stiffness of the nodes far from the domain (in grey) is small.

2.2 RESOLUTION ALGORITHM

In this contribution we consider the Conjugate Gradient Squared Method (CGS) provided by Matlab for solving problem (5). As you can appreciate, the additional terms require the solution to be available, being now a non-linear problem. This non-linear problem is solved by using the fixed point algorithm. The procedure follows the next steps:

1. Set an initial guess, $\mathbf{u}_0^h$.
2. Evaluate terms $\mathbf{T}(\mathbf{u}_0^h)$ and $\mathbf{S}(\mathbf{u}_0^h)$.
3. Solve problem (5) using the CGS method up to a given tolerance, τ_0, obtaining $\mathbf{u}_1^h$.
4. Evaluate $\epsilon \propto \|\mathbf{u}_0^h - \mathbf{u}_1^h\|$.
5. Set a new tolerance $\tau_1 = \epsilon\tau_0$.
6. Evaluate terms $\mathbf{T}(\mathbf{u}_1^h)$ and $\mathbf{S}(\mathbf{u}_1^h)$.
7. Solve problem (5) using the CGS method up to a given tolerance, τ_1, obtaining $\mathbf{u}_2^h$.
8. Evaluate $\epsilon \propto \|\mathbf{u}_1^h - \mathbf{u}_2^h\|$.
9. If $\epsilon <$ Tol, then the solution is obtained. In other case, continue with step 5.

Note that it could seem that this fixed point algorithm is expensive, however the corrections introduced by the stabilization terms are local and concentrated in the boundary. This fact allows to reuse the solution of the previous iterations as initial guess for the CGS solver in the next iteration, speeding up the calculations. The benefits of the proposed method are crucial for iterative solvers due to notorious decreasing of the condition number, allowing the right convergence of the iterative solver. Numerical results will illustrate the increase on the performance when the proposed method is used for both, linear elasticity and contact problems considering large deformations.

3 RECOVERY TECHNIQUES

Recovery procedures arise from error indicators techniques developed during the last decades [24, 25, 5, 9, 22, 16, 23], just to cite some. Among them we can highlight the Superconvergent Patch Recovery technique developed by Zienkiewicz and Zhu [25] which provides a robust, efficient and easy-to-implement error indicator. The recovery procedure used in this error indicator proposed by Zienkiewicz and Zhu is the basis of the recovery techniques proposed in this contribution.

3.1 Stress recovery technique

The recovered field at each patch of elements i, $\hat{\boldsymbol{\sigma}}_i$, is obtained by minimizing the following functional:

$$\mathcal{F}_i^{\sigma}(\hat{\boldsymbol{\sigma}}_i) = \int_{\hat{\Omega}_i} (\boldsymbol{\sigma}^h - \hat{\boldsymbol{\sigma}}_i)^2 \, \mathrm{d}\Omega. \tag{6}$$

A patch of elements consist in the elements attached to the node i, also called *assembly node*. Additionally, we can also add extra terms which improve the quality of the recovered stress field at each patch, $\hat{\boldsymbol{\sigma}}_i$, such as boundary and internal equilibrium and compatibility, see for instance [11]. In this contribution the field $\hat{\boldsymbol{\sigma}}_i$ is approximated by a polynomial of the same order than the FE solution. After obtaining the recovered stress field at each patch of elements, valid only in the patch surrounding the node i, the recovered field in the whole domain is obtained by using the Conjoint Polynomial Enhancement [5], which is nothing but the weighted sum of the contribution of each patch at a given position $\mathbf{x} \in \Omega$:

$$\hat{\boldsymbol{\sigma}} = \sum_{j=1}^{Nvn} N(\mathbf{x})_j \hat{\boldsymbol{\sigma}}_j(\mathbf{x}), \tag{7}$$

where Nvn is the number of vertex nodes in a element and the weighting functions are the linear shape functions of the elements.

3.2 Operator T

The operator $\mathbf{T}$ is the projection of the field $\hat{\boldsymbol{\sigma}}$ to the boundary, that is:

$$\mathbf{T}(\mathbf{u}^h) = \mathbf{n} \cdot \hat{\boldsymbol{\sigma}} \tag{8}$$

Note that for practical purposes since operator $\mathbf{T}$ is only needed along the Dirichlet boundaries, it only requires the recovery process to be applied in the patches cut by the boundary, therefore it can be considered as computationally inexpensive.

3.3 Displacement recovery technique

As a difference from the stress recovery procedure, in which one of the main objectives is to obtain a continuous stress field from the discontinuous one provided by the FEM, the displacement recovery procedure improves the existing solution by locally increasing the degree of approximation. Thus, the recovered displacement field $\hat{\mathbf{u}}_i$ at each patch i is obtained by minimizing the following functional:

$$\mathcal{F}_i^u(\hat{\mathbf{u}}_i) = \int_{\hat{\Omega}_i} (\mathbf{u}^h - \hat{\mathbf{u}}_i)^2 \, \mathrm{d}\Omega. \tag{9}$$

The recovered displacement field $\hat{\mathbf{u}}_i$ is approximated by a polynomial of one degree higher than the FEM solution. In this case, the functional $\mathcal{F}_i^u$ can be enriched by adding extra terms to enforce equilibrium via a collocation method:

$$\mathcal{F}_i^{u,eq} = \mathcal{F}_i^{u,eq} + \sum_{k=1}^{N_{ie}} \lambda_k (\nabla \cdot \hat{\boldsymbol{\sigma}}_i^u - \mathbf{b}) + \sum_{k=1+N_{ie}}^{N_{ie}+N_{be}} \lambda_k (\mathbf{n} \cdot \hat{\boldsymbol{\sigma}}_i^u - \mathbf{t}), \tag{10}$$

where λ_k is the k^{th} Lagrange multiplier, N_{ie} is the number of collocation points in which the internal equilibrium is enforced at each patch, N_{be} is the number of points in which the boundary equilibrium is enforced. The stress field $\hat{\boldsymbol{\sigma}}_i^u$ is evaluated at each patch as follows:

$$\hat{\boldsymbol{\sigma}}_i^u = \mathbf{D}\varepsilon(\hat{\mathbf{u}}_i). \tag{11}$$

Finally, the displacement and stress fields over the whole domain are approximated by using the Conjoint Polynomial Enhancement as follows:

$$\hat{\boldsymbol{\sigma}}^\sigma = \sum_{j=1}^{Nvn} N(\mathbf{x})_j \hat{\boldsymbol{\sigma}}_i^u, \tag{12}$$

and

$$\hat{\mathbf{u}}^u = \sum_{j=1}^{Nvn} N(\mathbf{x})_j \hat{\mathbf{u}}_i. \tag{13}$$

Note that the difference between $\hat{\boldsymbol{\sigma}}^\sigma$ and $\hat{\boldsymbol{\sigma}}^u$ is that the former is locally equilibrated and the later is compatible with $\hat{\mathbf{u}}^u$. Note that the superindex $^\sigma$ indicates that the field is locally equilibrated and the superindex ^{u} indicates that the field is kinematically admissible or it directly comes from a kinematically admissible field.

3.4 Operator S

The operator **S** is directly the displacement field $\hat{\mathbf{u}}^u$. Note that for practical purposes since operator **S** is only needed along the boundaries, it only requires the recovery process to be applied in the patches cut by the boundary, therefore it can be considered as computationally inexpensive.

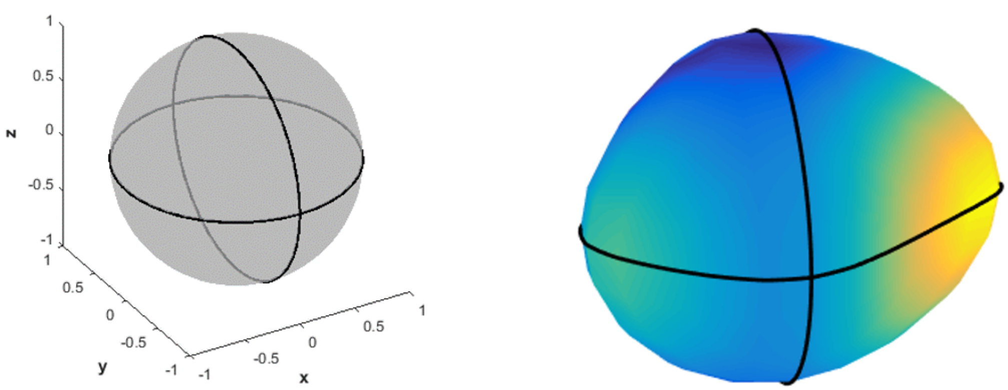
4 NUMERICAL RESULTS

In this section, two academic problems are used to show the numerical results obtained with the proposed method. The first problem is a Dirichlet problem with a spherical domain. The second presents two bodies under contact.

4.1 Problem 1. Sphere

This problem is defined in an spherical domain of diameter 2 centred at origin, as shown in Figure 2a. The displacement exact solution (14) is artificially generated. Therefore, the problem prescribes the displacement field (14) over the external boundary and the corresponding body loads numerically evaluated from the displacement field. The Young modulus is set to $E = 1000$ and the Poisson ratio $\nu = 0.3$. The numerical solution obtained with the proposed method is shown in Figure 2b.

$$
\begin{aligned}
u_x(x,y,z) &= x + x^2 - 2xy + x^3 - 3xy^2 + x^y \\
u_y(x,y,z) &= -y - 2xy - 3x^2y + y^3 - xy^2 \\
u_z(x,y,z) &= 0
\end{aligned}
\tag{14}
$$



(a) Geometry of the problem.

(b) Solution obtained with the proposed method.

Figure 2: Problem 1. Geometry and solution.

Figure 3 shows the convergence of the problem, obtaining for all the cases right convergence rates for a linear discretization. This fact, numerically demonstrates that the

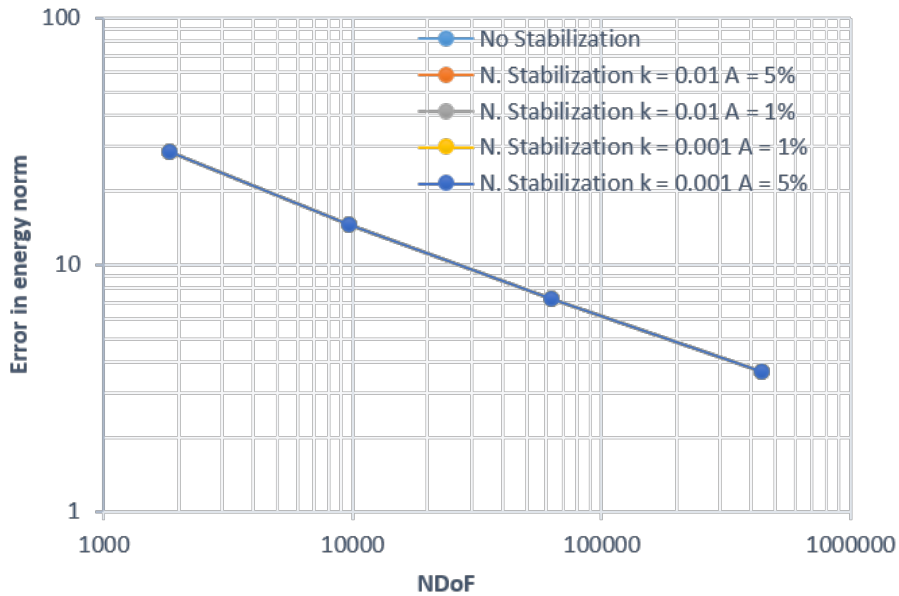


Figure 3: Problem 1. Convergence of the exact error in energy norm.

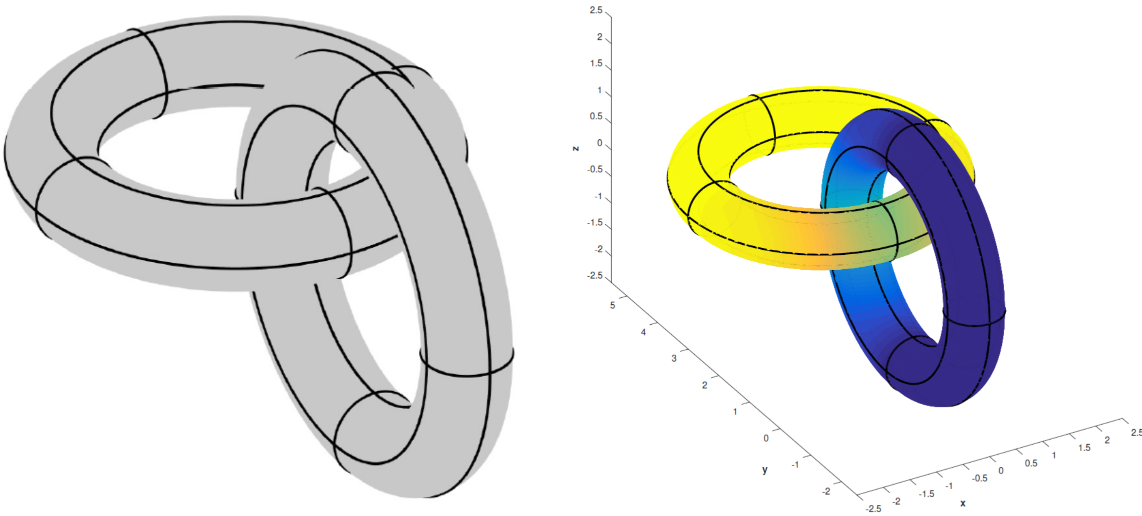
additional stabilization term does no modify the convergence of the numerical problem to the exact solution. Table 1 shows the condition number of the problem without the stabilization term (first row) and the condition number when the stabilization term is used, for different mesh levels, i.e. level 4 means 2^4 elements in each direction. We can observe that, as a difference with the first row, the condition number is considerably smaller and slightly changes between the different meshes.

Level 4		Level 5	
$7.42 \cdot 10^8$		$7.42 \cdot 10^{18}$	
$k_2 = 0.01$ $A = 1\%$	$3.32 \cdot 10^6$	$k_2 = 0.01$ $A = 1\%$	$5.08 \cdot 10^6$
$k_2 = 0.01$ $A = 5\%$	$3.32 \cdot 10^6$	$k_2 = 0.01$ $A = 5\%$	$5.08 \cdot 10^6$
$k_2 = 0.001$ $A = 1\%$	$3.32 \cdot 10^6$	$k_2 = 0.001$ $A = 1\%$	$5.09 \cdot 10^6$
$k_2 = 0.001$ $A = 5\%$	$3.32 \cdot 10^6$	$k_2 = 0.001$ $A = 5\%$	$5.09 \cdot 10^6$

Table 1: Condition number on the system of equations for different meshes and configuration parameters. The parameter A corresponds to the ratio between the volume of the element and the intersected volume.

4.2 Problem 2. Contact problem

This problem shows the contact between two annulus. The annulus placed at right has prescribed homogeneous Dirichlet boundary condition over the half left of the domain, whereas the left annulus has a prescribed displacement of 0.5 in y direction over the left half of the surface. 5 load steps are considered. The Young modulus is set to $E = 1000$ and the Poisson ration is $\nu = 0.3$. Large displacements are also considered.



(a) Geometry of the problem.

(b) Solution obtained with the proposed method.

Figure 4: Problem 2. Geometry and solution.

Table 2 shows the accumulative iterations of the CGS iterative solver during the contact problem resolution. It can be appreciated, for coarse meshes, that the stabilization techniques does not provide better results, but when the element size decreases, the number of iterations without stabilization considerably increases. This is related with the increase of the condition number for finer meshes observed in the previous example.

Mesh Level	With stabilization	Without stabilization
3	1223	1161
4	2171	2276
5	4296	5698

Table 2: Accumulative iterations for the iterative solver.

CONCLUSIONS

This manuscript presents a stabilization algorithm in charge of controlling the condition number of the system of equations. This algorithm is applied into the cgFEM framework considering the linear elastic problem and a contact problem with a large displacements formulation. The proposed method needs a fixed point iterative solver which is embedded into the iterative solver for the global system of equations, thus preserving the efficiency of the method. The obtained results show that the proposed method effectively controls the condition number of the system of equations maintaining the right convergence of the numerical solution to the exact one. Additionally, as a consequence of the control of the condition number, the number of iterations needed for the iterative solver decreases.

ACKNOWLEDGEMENTS

The authors acknowledge the support of Ministerio de Economía, Industria y Competitividad of Spain (DPI2017-89816-R) and the Generalitat Valenciana (PROMETEO/2016/007).

REFERENCES

[1] Owe Axelsson. Iterative Solution Methods. 1994.

[2] Helio J C Barbosa and Thomas J R Hughes. The finite element method with Lagrange multipliers on the boundary: circumventing the Babu??ka-Brezzi condition. *Computer Methods in Applied Mechanics and Engineering*, 85(1):109–128, 1991.

[3] Helio J C Barbosa and Thomas J R Hughes. Boundary Lagrange multipliers in finite element methods: Error analysis in natural norms. *Numerische Mathematik*, 62(1):1–15, dec 1992.

[4] Eric Béchet, Nicolas Moës, Barbara Wohlmuth, Éric Béchet, Nicolas Moës, and Barbara Wohlmuth. A stable Lagrange multiplier space for stiff interface conditions within the extended finite element method. *International Journal for Numerical Methods in Engineering*, 78(8):931–954, may 2009.

[5] Ted Blacker and Ted Belytschko. Superconvergent patch recovery with equilibrium and conjoint interpolant enhancements. *International Journal for Numerical Methods in Engineering*, 37(3):517–536, 1994.

[6] Erik Burman, Susanne Claus, Peter Hansbo, Mats G. Larson, and André Massing. CutFEM: Discretizing geometry and partial differential equations. *International Journal for Numerical Methods in Engineering*, 104(7):472–501, nov 2015.

[7] Erik Burman and Peter Hansbo. Fictitious domain finite element methods using cut elements: I. A stabilized Lagrange multiplier method. *Computer Methods in Applied Mechanics and Engineering*, 199(41-44):2680–2686, 2010.

[8] Erik Burman and Peter Hansbo. Fictitious domain finite element methods using cut elements: II. A stabilized Nitsche method. *Applied Numerical Mathematics*, 62(4):328–341, 2012.

[9] Pedro Díez, J J Ródenas, O C Zienkiewicz, D Pedro, Pedro Díez, J J Ródenas, and O C Zienkiewicz. Equilibrated patch recovery error estimates : simple and accurate upper bounds of the error. *International Journal for Numerical Methods in Engineering*, 69(August 200610):2075–2098, 2007.

[10] Christophe Geuzaine and Jean-François Remacle. Gmsh: a three-dimensional finite element mesh generator with built-in pre-and post-processing facilities. *International Journal for Numerical Methods in Engineering 79(11)*, 0:1309–1331, 2009.

[11] O. A. González-Estrada, E. Nadal, J. J. Ródenas, P. Kerfriden, S. P. a. Bordas, and F. J. Fuenmayor. Mesh adaptivity driven by goal-oriented locally equilibrated superconvergent patch recovery. *Computational Mechanics*, 53(5):957–976, oct 2013.

[12] Peter Hansbo, Carlo Lovadina, Ilaria Perugia, and Giancarlo Sangalli. A Lagrange multiplier method for the finite element solution of elliptic interface problems using non-matching meshes. *Numerische Mathematik*, 100(1):91–115, mar 2005.

[13] Martin Hautefeuille, Chandrasekhar Annavarapu, and John E. Dolbow. Robust imposition of Dirichlet boundary conditions on embedded surfaces. *International Journal for Numerical Methods in Engineering*, 90(1):40–64, apr 2012.

[14] José Manuel Navarro-Jiménez, Manuel Tur, José Albelda, and Juan José Ródenas. Large deformation frictional contact analysis with immersed boundary method. *Computational Mechanics*, pages 1–18, jan 2018.

[15] Jamshid Parvizian, Alexander Düster, and Ernst Rank. Finite cell method. *Computational Mechanics*, 41(1):121–133, sep 2007.

[16] J J Ródenas, M Tur, F J Fuenmayor, and A Vercher. Improvement of the superconvergent patch recovery technique by the use of constraint equations: the SPR-C technique. *International Journal for Numerical Methods in Engineering*, 70(6):705–727, 2007.

[17] E. Ruiz-Girons and J. Sarrate. Generation of structured hexahedral meshes in volumes with holes. *Finite Elements in Analysis and Design*, 46(10):792–804, oct 2010.

[18] Dominik Schillinger and Martin Ruess. The Finite Cell Method: A Review in the Context of Higher-Order Structural Analysis of CAD and Image-Based Geometric Models. *Archives of Computational Methods in Engineering*, 22(3):391–455, 2015.

[19] M Tur, José Albelda, E Nadal, and J J Ródenas. Imposing Dirichlet boundary conditions in hierarchical Cartesian meshes by means of stabilized Lagrange multipliers. *International Journal for Numerical Methods in Engineering*, 98(6):399–417, may 2014.

[20] M. Tur, F. J. Fuenmayor, and P. Wriggers. A mortar-based frictional contact formulation for large deformations using Lagrange multipliers. *Computer Methods in Applied Mechanics and Engineering*, 198(37-40):2860–2873, 2009.

[21] Manuel Tur, José Albelda, Onofre Marco, and Juan José Ródenas. Stabilized method of imposing Dirichlet boundary conditions using a recovered stress field. *Computer Methods in Applied Mechanics and Engineering*, 296:352–375, aug 2015.

[22] Q Z Xiao and B L Karihaloo. Statically admissible stress recovery using the moving least squares technique. In B H V Topping and C A Mota Soares, editors, *Congress in Computational Structures Technology*, pages 111–138, Stirling, Scotland, 2004. Saxe-Coburg Publications.

[23] Zhimin Zhang. A posteriori error estimates on irregular grids based on gradient recovery. *Advances in Computational Mathematics*, 15:363–374, 2001.

[24] O C Zienkiewicz and J Z Zhu. A simple error estimator and adaptive procedure for practical engineering analysis. *International Journal for Numerical Methods in Engineering*, 24(2):337–357, 1987.

[25] O C Zienkiewicz and J Z Zhu. The superconvergent patch recovery and a posteriori error estimates. Part 1: The recovery technique. *International Journal for Numerical Methods in Engineering*, 33(7):1331–1364, 1992.