Towards semantic-based 3D mesh modeling

Ruding Lou, Franca Giannini, Jean-Philippe Pernot, Alexei Mikchevitch,
Bianca Falcidieno, Philippe Véron and Raphaël Marc

***Abstract*— Nowadays, most of the numerical simulations in product maintenance are carried out by several loops of the following steps: 1) CAD model creation/optimization, 2) Finite Element (FE) mesh generation, 3) insertion of semantic data for physical simulation (e.g. material behavior laws, boundary conditions) and 4) FE simulation and analysis of the results. The four steps are repeated for the evaluation of each conceived maintenance solution. The semantic data are attached to the mesh through the use of groups of mesh entities sharing the same semantic characteristics. Thus, any modification of the CAD model always implies an update of the mesh as well as an update of the attached semantic data. This is time-consuming and thus not suitable for industrial maintenance. Moreover, the CAD models do not always exist and should therefore be reconstructed starting from scratch or from the scanned physical object. In this paper, we propose a framework towards the definition of semantics based CAD-less operators wherein semantic enriched meshes are manipulated directly. This work also finds interest in the preliminary design phases where alternative solutions have to be quickly evaluated.**

***Index Terms* — Mesh modification, Finite Element analysis Semantics, semantic preservation.**

I. Introduction

In product maintenance, the optimizations of the structure are classically suggested and numerically simulated before performing the modifications on the real product. It avoids expensive physical experimentations when prototyping and assessing new solutions all along the product lifecycle. In industrial production contexte, a crucial issue is related to the need of reducing as much as possible expensive production stops during the maintenance operations. Thus, it is important to be able to provide rapidly a solution improving production machinery characteristics as well as satisfying multiple safety criteria. To do this, experts must have appropriate numerical tools supporting them in rapidly and accurately evaluating different alternative solutions from the mechanical view point. Unfortunately, the existing methodologies for product behavior numerical analysis and solution assessment does not answer to these needs.

Today, most of the product behavior analyses rely on the following steps: a) conceptual solution proposal and its detailed design using Computer-Aided Design (CAD) software, b) complex Finite Element (FE) mesh model preparation, c) specification of multiple FE semantic information useful for FE analysis d), FE simulation, result evaluation and optimization loops (Figure 1). During the optimization steps, geometric modifications are generally performed on the CAD model corresponding to the new solutions, thus requiring a complete regeneration of the FE mesh models and a redefinition of all semantics on it. Figures 1.c_1 and 1.c_2 show two alternative solutions tested to remove a stress concentration phenomenon detected after the first FE analysis (Fig. 1.d). Here, the changes have been performed on the CAD model thus requiring all the above described steps.

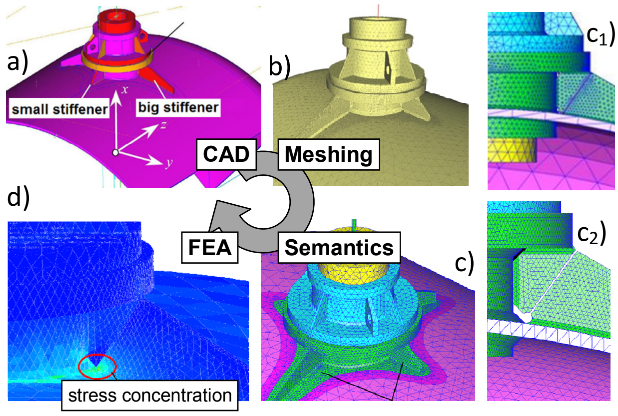

Figure 1 Mainstream methodology for product behavior analysis and optimization (courtesy EDF-R&D).

However, in this context, the CAD models are not always available and/or do not fully fit to the reality that can be measured on the real physical settings using 3D scanning techniques. Thus, the creation of complementary CAD models leads to an additional waste of time and should then be avoided as much as possible. Moreover, such a process is inappropriate to design rapidly alternative solutions since the shape modifications performed on the CAD model require an total re-meshing of the model which needs tuning again the model and makes the previous defined semantics invalid. The re-assignment of all the semantic data is a quite time consuming work. This is especially true when the model contains numerous mesh groups supporting lots of semantic data. For example, the models designed by the EDF (Electricité de France) engineers can contain up to 500 mesh groups. As a consequence, in the prototyping and assessment of structural modifications, even small local changes require expensive complete updating of the simulation model that is critical for

R. Lou, J.P.Pernot and P. Véron (firstname.lastname@ensam.fr) are with Arts et Métier ParisTech, 13100 Aix en Provence, France.

F.Giannini and B.Falcidieno (firstname.lastname@ge.imati.cnr.it) are with the Italian National Council of Research, Institute for Applied Mathematics and Information Technologies, via de Marini, 6 Genova, Italy

A.Mikchevitch and R. Marc (firstname.lastname@edf.fr) are with the Electricité de France Group – R&D, 1 avenue du Général de Gaulle, 92141 Clamart, France.

fast studies. Maintaining and updating the previously defined semantic information while performing geometric modifications would strongly reduce the time required for alternative evaluations. An example of semantic information updating could be the propagation of pressure data to the walls of a hole inserted into an object embedded in a fluid.

To overcome these limits, we propose a fast CAD-less prototyping framework working directly at the level of meshes enriched by semantics of geometric nature relative to mesh groups. In this way, the time necessary for FE model preparation can be strongly reduced. The idea is to remove the "hard" steps of re-meshing and FE model preparation by bringing the necessary local modifications directly onto the meshes while maintaining and possibly propagating the semantic data.

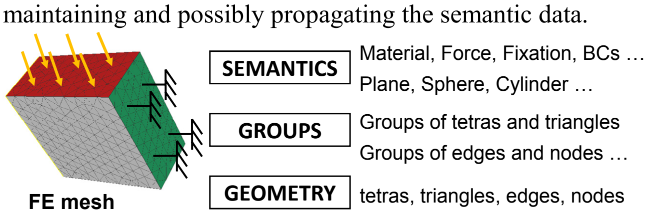

Figure 2 Triple levels information for CAD-less operators.

In particular, we introduce mesh modification operators that work directly on the semantically enriched FE mesh models (Figure 2). They simultaneously act at the geometric level corresponding to the mesh elements (geometry level), at the group one (low level of shape semantics), corresponding to the FE groups collecting mesh elements characterized by high level semantic data represented at the top level. The operator behavior is driven by both the FE and shape semantics. For what concerns the shape semantics, the operators are considering as the operands (i.e. operated mesh and modifying tool) as well as the shape of the FE groups' boundaries. All these data are transformed into a set of constraints that drive a 2D/3D mesh deformation engine during the mesh modification operation.

The paper is organized as follows. Section II summarizes some related works. The types of considered semantic information and the underlying principles related to the semantics preservation during the geometry modification are described in section III. Section IV presents the concept of group and group boundary as support for association and preservation of semantic data. Section V presents geometry operators. Section VI shows some proposal of semantic propagation.

II. State of the Arts

Boolean operations on meshes acting purely on the geometry have been largely studied. In [1]-[3], mesh cutting approaches are proposed. They directly split the elements to follow the cutting trajectory, no real attention is provided to the mesh quality required for FE analysis. The cutting operators presented in [4] and [5] privilege the mesh quality but the cutting profile on the mesh does not perfectly match the cutting tool. In [6], Boolean operations are performed on volumetric meshes by evaluating the intersection of the boundary meshes and re-filling entirely the tetrahedral mesh. This is not acceptable when dealing with tuned FE meshes which should be modified only locally. Lira et al. have been working on the computation of potentially multiple intersections between two intersecting meshes [7]. The modifications solely affect the elements directly involved in the intersection. Thus, degenerated triangles, i.e. triangles defined by one or two small angles, may be created thus affecting the mesh quality. Graysmith and al. have proposed a method to join two meshes [8]. The algorithm consists in four steps: contact detection, shell construction, element creation within the shell and mesh assembly. However, their method is not adapted to the treatment of meshes of heterogeneous sizes, as can be the case of FE meshes, and the quality of the elements in the contact area does not fit the FE requirements.

Semantic aspects have been widely detailed and exemplified in the Aim@Shape European Network of Excellence [9]. In industrial design, the semantic data correspond to all the information that is used to design and manufacture a product, its colors, its material, its decomposition into meaningful areas and so on. In the context of FE simulation, they may also correspond to all the data required before running the simulation properly saying: Boundary Conditions (BCs), geometric parameters (thickness of a group of faces), materials and so on [10]. Actually, semantic aspects can be encountered in all the steps of the product lifecycle. Integration and maintenance of semantic information during the product modeling process has been subject of research since many years [11] and [12]. Some approaches try to take into account this application-dependent information during the manipulation and modification of the underlying geometric models. Hamri et al. [13] and [14] have proposed a unified framework to handle and to process the CAD models and FE meshes through an intermediate polyhedral representation. In their approach, the semantic data are taken into account through the specification of partitions whose boundaries may drive a polyhedral simplification method used to adapt the digital mock-up to the various engineering needs (e.g. visualization, FE simulation, clash detection). This is an interesting example on how semantic information can constrain geometric manipulations.

However, to the best of our knowledge, the preservation and propagation of FE simulation semantics during the mesh manipulation have not been addressed yet. Most of the existing methods and software treat the problem from a geometric point of view while forgetting that the geometry is often considered as a support used also to convey semantic information all along the design process. Hence, a specific effort has to be done in this direction.

In this paper we introduce a framework for CAD-less modifications based on local mesh deformations under constraints maintaining FE semantics (FE groups, FE simulation data). The adopted deformation tool produces the objective shapes while insuring the global mesh quality. The constraints are coming from the shape of the operated model, the modification tool and the groups. In the following section our group treatment and mesh deformation method are presented.

III. Semantic manipulation on FE model

The semantic information attached to a FE model may be of different nature:

- High level FE semantics (material, loading, fixation, displacements, etc.) are associated to a part of a mesh with various dimensions (from 0D to 3D) through the use of groups gathering together geometric elements (nodes, edges, faces or volumes).
- Geometric shapes (edges, plane, sphere, cylinder etc.) corresponding to:
  - the shape of the operated mesh model,
  - the shape of the tool modifying the mesh model,
  - the shape of the FE groups,
  - the shape of the FE group boundaries.

Therefore, the proposed CAD-less framework takes into account the semantic information describing both the FE simulation and the shapes. The semantic data are maintained during the mesh modifications by correctly gathering the mesh elements respecting the original groups and by preserving the correct position of the gathered mesh elements.

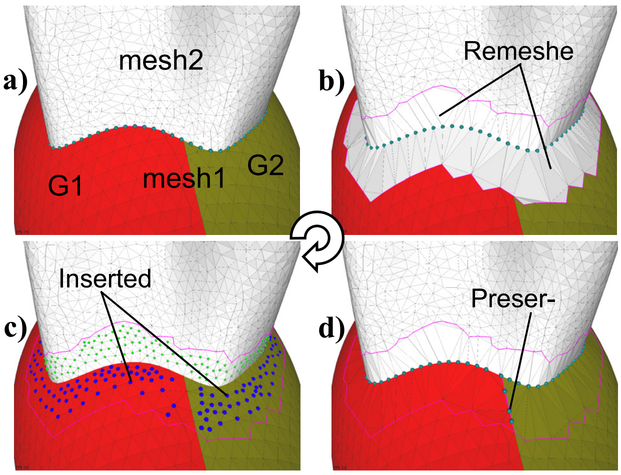

Figure 3 Merging of triangle meshes preserving semantics

Figure 3 shows an example of modification of a 2D mesh containing semantic information. The Figure 3.a shows the initial mesh1 to be operated on which two groups G1 and G2 are defined. The modification consists in merging this model with another model mesh2. The mesh merge result is shown by Figure 3.b and the contact zone is re-meshed to avoid the intersection between the triangles of the two meshes. From this example, we can see that without considering the associated semantics we could not correctly decide which elements should finally belong to G1 and G2. There are even triangles laying on the both zone of G1 and G2. This is because during the re-meshing we have lost a kind of boundary of initial groups which is preserved in the merging result shown by Figure 3.c. The group boundary concept is presented in more details in the next section. The two groups are holding correct elements from space partitioning point of view but the shape defined by the initial groups is not preserved because the new triangles have density much bigger. In the merging result shown by the Figure 3.d new nodes have been inserted and their positions are adjusted according to the shape of the group body. The deformation of mesh by repositioning nodes is presented in detail in the section V.

On the new added part which is originally mesh2 we could imagine to propagate group definition on it according to the different semantic information carried. We suppose that there is a group G3 covering both G1 and G2 on which a fluid pressure is defined. This pressure semantic data could be also propagated onto the new added part if the full model is immersed in the fluid. The group propagation possibilities are discussed in section VI.

IV. Group preservation

Maintaining and propagating semantic information during the mesh modification operation goes through the preserving of the FE mesh groups (here, we suppose that the semantic of the shapes is also associated to the mesh using groups). In particular, this involves an appropriate handling of mesh group characteristics: the group boundaries (geometric and “virtual”), the group shapes (supporting surface or curve type), topologies (nodes, edges, triangles, etc.).

In this work, we adopt the definition and use of the so-called Virtual Group Boundaries (VGB) given in [15]. Roughly speaking, a VGB is defined by a set of 1D and/or 2D elements located at the group boundary enclosing 0D-3D groups (set of nodes or elements). Nodes on VGB are used to apply additional constraints during the local mesh deformation to make these nodes staying on the shape of the group boundary.

Moreover, several semantics can be associated to the same mesh elements. For example, a part of a surface can receive a pressure and may be of a specific material. Therefore groups can overlap. This means that a mesh entity can belong to partially overlapping groups. To easily set up constraints during the shape modifications and make easier the re-assignment of group definition while re-meshing, we have introduced the notion of Elementary Group (EG) to enable the definition of non-overlapping configurations [15]. An elementary group $EG_{k\ldots h}$ is the set of all the mesh entities e such that e belongs to the groups $G_{k\ldots h}$. Thus, the group G_k is formed by one or more elementary groups EG_{ki}.

V. Mesh modification operator

According to various mechanical engineering needs, a first set of FE modification operators has been designed and can be classified with respect to the following types (Figure 4): material addition, material removal and crack/contact insertion. These operators directly act on an initial/reference FE mesh (A) with a surface primitive (B) used as an operating tool. These operations correspond either to Boolean operations on the reference mesh (for material addition and removal) or as a constrained modification of the reference mesh (for crack/contact insertion). Actually, they can be roughly linked to the classical Constructive Solid Geometry (CSG) operators: union and subtraction of meshes. Crack/contact may be seen as a special case of non-regularized operations.

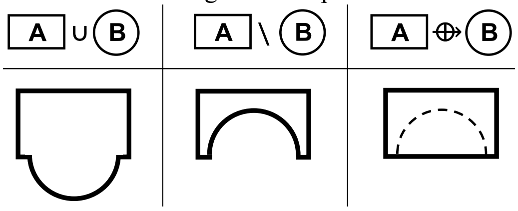

Figure 4 Three categories of considered Boolean operations

For repositioning the nodes under certain shape constraints and at the same time for maximizing the general quality of the mesh elements we have adopted a deformation tool [16] based on the FDM (Force Density Method). With this approach, the mesh modification results from the resolution of an optimization problem defined by a set of linear and non-linear equality constraints, and an objective function φ to be minimized (Eq. 1). The unknowns are the positions of the nodes in the surrounding of the area to be modified. They are gathered together in the unknown vector X. The constraints form a constraint vector G linking some of the node position:

$$\begin{cases} \mathbf{G}(\mathbf{X}) = \mathbf{0}, \\ \min \varphi(\mathbf{X}). \end{cases} \quad (1)$$

Mesh operator modifies on the operated mesh the triangles/tetrahedra intersecting with the tool (mesh or perfect surface) and also optimizes the aspect ratio of both inner and surrounding triangles/tetrahedra. The constraints considered in the deformation are related to:

- the shape of the operated mesh body and its boundary,
- the shape of the tool,
- the shape of the different groups' body and boundary.

VI. Towards Semantic Propagation

The proposed CAD-less operators perform geometric modifications by preserving the semantic data as well as the group definition. It can also handle some semantic data propagation. We details it on one example in the case of mesh drilling. The Figure 5.a shows a cube-like model A on which fluid pressure is defined on the four facets. The arrows represent the pressure produced by fluid. On this example, we want to make a cylindrical hole going through the model. Figure 5.b shows the half of model A in which we can see the half hole realized. Since the pressure is produced by the fluid and the two sides of the hole (top and bottom) received initially the pressure, we can suggest the user to automatically propagate the pressure to the interior cylinder wall of the hole thus saving manipulation times (Figure 5.b).

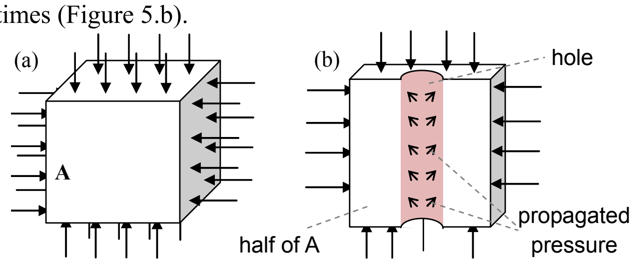

Figure 5 Pressure propagation in case of a drilling

VII. Results

This section gives some results relative to the application of the mesh drilling and crack operators on 2D/3D meshes.

First, the cylindrical drilling operator is applied on the triangle mesh of a half-vase segmented in two groups of triangles G1 and G2 (Figure 6.a). The first step aims at removing all the triangles that are completely inside the cylindrical volume defined by the axis and the radius of the hole tool (Figure 6.b). The Figure 6.c shows the result of the local mesh deformation. The drilling interface nodes (IN) are constrained to stay on the cylinder, VGBN nodes are constrained to stay on the identified VGB, and MBN nodes are constrained to stay on the mesh boundary. The transition nodes (TN) are constrained by the shape of the mesh.

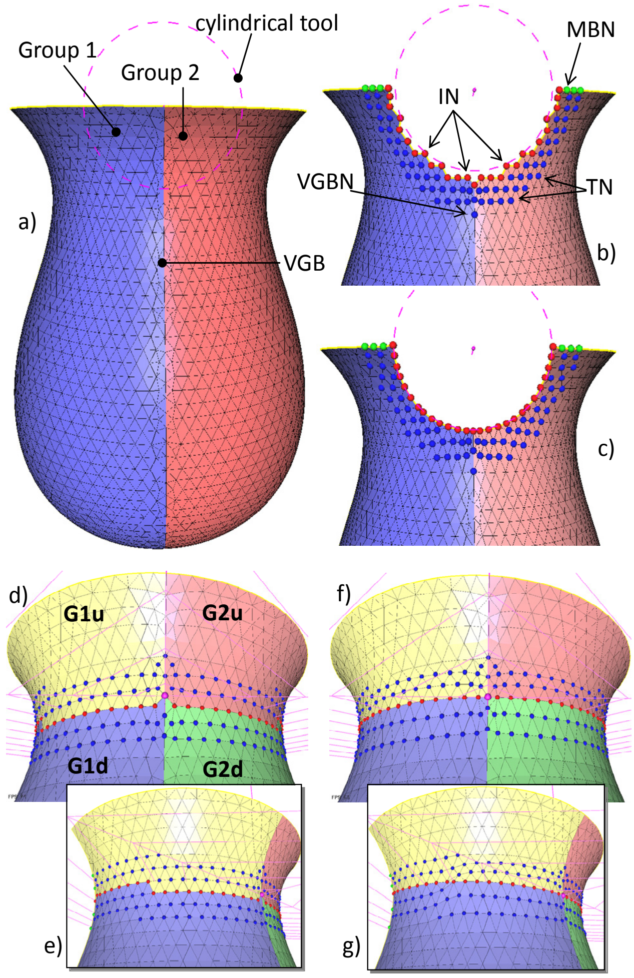

Figure 6 Drilling and cracking on 2D FE mesh

A planar crack/contact zone is applied on the same model presented by Figure 6.a. Figure 6.d and Figure 6.e showing from different point of view the vase that is split into two subparts by a planar cutting tool. Therefore, the initial group G1 (resp. G2) corresponds to the group G1u and G2d (resp. G2u and G2d) in the upper part and the lower part. The red nodes are the ones shared by the two parts. These nodes will be repositioned onto the crack plane through the deformation process. Figure 6.f and Figure 6.g show the result of the deformation so that the vase is cracked along the plane.

Second, the cylindrical drilling operator is applied on a cube-like tetrahedral mesh segmented in three tetrahedral groups (Figure 7.a). Here, the segmentation of the volume has been performed so that the resulting boundary between groups 1 and 2 is cylindrical and the boundary between groups 2 and 3 is spherical. As a consequence, some nodes are constrained to stay on the cylindrical tool, some other on the spherical VGB, some on both, and so on. For 3D mesh, transition nodes are completely free to move inside the volume. Here, mesh boundary nodes have to stay on the faces of the cube. The Figure 7.b_1 shows that tetrahedra inside the drilling cylinder are removed and Figure 7.b_2 shows the result of the local mesh deformation when using the minimization of the external forces applied to the bar network coupled to the mesh.

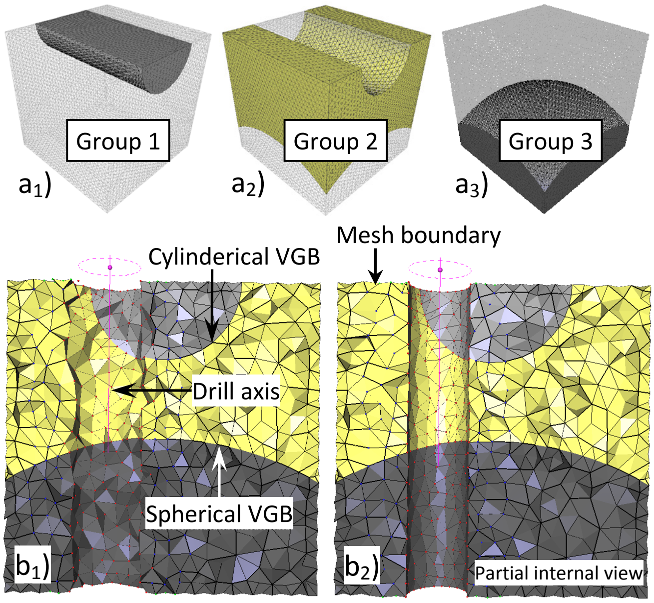

Figure 7 CAD-less operator: drilling on 3D FE mesh

VIII. CONCLUSION

This paper introduces a framework for the definition of semantic-based CAD-less operators. To maintain attached semantics during the geometric modifications of the FE mesh model, we emphasize the need of implementing new geometric modeling operators able to simultaneously handle geometric and semantic data associated to the FE mesh model. Group boundaries have been identified as the key elements for setting the necessary constraints and for preserving semantic association. The propagation of semantics after the geometric mesh modifications is still in progress.

ACKNOWLEDGMENT

This work is partially supported by the Research and Development direction of the EDF Group, and it is partially carried out within the scope of the FOCUS K3D project supported by the European Commission [17]. The newly developed operators will be implemented into SALOME® [18] software platform, a free CAD/Mesh/FE Analysis/Post-processing software co-developed mainly by the Open CASCADE company, the EDF Group and CEA with some contributions of other industrial and university partners.

REFERENCES

[1] G.Turkiyyah, W.B.Karam, Z.Ajami, A.Nasri, 2009 ,Mesh cutting during real-time physical simulation, Symposium on Solid and Physical Modeling, p. 159-168

[2] C. D. Bruyns, S. Senger, A. Menon, K. Montgomery, S. Wildermut, R. Boyl, 2002, *A survey of interactive mesh-cutting techniques and a new method for im-plementing generalized mesh cutting using virtual tools*. The journal of visualization and computer animation, 13, pp. 21- 42.

[3] D. Bielser, A. Volker, M. Gross, 1999, *Interactive cuts through 3-dimensional soft tissue*. Computer Graphics Forum 18-3 (1999), pp. C31–C38

[4] Z.Ji, L.Liu, Z.Chen, G.Wang, 2006, *Easy Mesh Cut-ting. Proceedings of Eurographics*, Vienna, Austria, Computer Graphics Forum, 2006, 25(3): 283-291.

[5] G.Turini, G.Ganovelli, C.Montani, *Simulating Drilling on Tetrahedral Meshes*, 2006, Proceedings of Eurographics Conference - Short Papers, p. 127-131

[6] P, Kršek, 2002, Complex human tissues fem models prepared by boolean operations, In: Biomechanics of man 2002, Praha, CZ, UK FTVS, p. 24-26, ISBN 80-86-317-23-4

[7] W.M. Lira, L.C.G. Coelho, L.F. Martha. *Multiple Intersections of Finite-Element Surface Meshes*. In proceedings of the 11th International Meshing Roundtable, pp. 355-366, 2002

[8] J.L. Graysmith, C.T. Shaw. *Automated procedures for Boolean operations on Finite Element meshes*. Engineering Computation: International Journal for Computer-Aided Engineering and Software, vol. 14(7), pp. 702-717, 1997.

[9] Aim@Shape 2004-08. *Advanced and innovative models and tools for the development of semantic-based systems for handling, acquiring, and processing knowledge embedded in multidimensional digital objects*. European Network of Excellence, Key Action: 2.3.1.7 Semantic-based knowledge systems, VI Framework, http://www.aimatshape.net.

[10] C.G. Armstrong, D.J. Monaghan, M.A. Price, H. Ou, J. Lamont. *Integrating CAE Concepts with CAD Geometry*. in Engineering Computational Technology edited by B. H. V. Topping and Z. Bittnar, Saxe-Coburg Publica-tions, pp. 75-104, 2002.

[11] T.De Martino, B.Falcidieno, F. Giannini. *A Reference Model for Product Information Sharing in Concurrent Engineering Environment, Virtual Prototyping: Virtual environments and the product design process*, IFIP Chapman & Hall, 1995

[12] R. Bidarra, W.F. Bronsvoort. *Semantic feature modeling. Computer-Aided Design*, 32(3), pp. 201–225, 2000.

[13] O. Hamri, J-C. Léon, *Interoperality between CAD and simulation models for cooperative design. CIRP Design Seminar*. May 12-14th, Grenoble, France, 2003.

[14] O. Hamri, J-C. Léon, F. Giannini, B. Falcidieno. *Using CAD models and their semantics to prepare FE simulation*. ASME DETC Int. Comp. in Eng. Conf., Salt Lake City 2005

[15] R.Lou, F.Giannini, J.P.Pernot, A.Mikchevitch, B.Falcidieno, P.Veron, R.Marc, 2009, *Towards cad-less finite element analysis using group boundaries for enriched meshes manipulation*, Proc. of the ASME 2009 Int. Design Eng. Tech. Conf. & Computers and Information in Eng. Conf., DETC09-CIE86575, San Diego, USA.

[16] J-P. Pernot, B Falcidieno, F Giannini, J-C. Léon, *Shape tuning in Fully Free-Form Deformation* Features, J. of Computing and Information Science in Engineering, 5(2):95-103, 2005.

[17] FOCUS K3D (ICT-2007-214993) Foster the comprehension and use of Knowledge intensive 3D media http://focusk3d.eu/

[18] SALOME®. Open Source Integration Platform for Numerical Simulation. http://www.salome-platform.org/