
A Mesh-Geometry Based Approach for Mixed-Dimensional Analysis

Sylvain Bournival, Jean-Christophe Cuillière, and Vincent François

Université du Québec à Trois-Rivières
3351 Boulevard des Forges, CP 500 Trois Rivières G9A 5H7
Québec, Canada
bournisy@uqtr.ca, cuiller@uqtr.ca, françois@uqtr.ca

Summary. When conducting a finite element analysis, the total number of degrees of freedom can be dramatically decreased using finite elements such as beams and shells. Because of geometric complexities, entire models (or portions of models) must be meshed using volume elements in order to obtain accurate simulation results. If however some parts of these models fit the description of shells or beams, then a mixed-dimensional model containing shell, beam and volume elements side by side can be used. This approach can significantly reduce the time needed to mesh and solve the system. Unfortunately, problems arise when trying to connect elements of different dimensions in part due to the incompatible degrees of freedom and the independently created meshes. This paper presents a solution to these problems based on the generation of a compatible mesh composed solely of basic elements and without the requirement of constraint equations.

Keywords: mixed-dimensional analysis, model reduction, mesh, shell elements, beam elements.

1 Introduction

When conducting a finite element analysis (FEA) the engineer is often confronted with long calculation time or with a limit imposed on the number of elements. Dimensional reduction is one of the many techniques employed to reduce the computing time and/or the number of elements. Using plate, shell or beam elements can considerably decrease the total number of degrees of freedom (DOF) without loss of accuracy.

However, not all models are suitable for dimensional reduction. Indeed, portions of a model with a complex geometry must be simulated using volume elements. Some models, often structures, feature portions of their geometry which can be modeled by reduced-dimensional elements while other portions must be modeled using volume elements. Examples of these can be found in [1] and [2]. In these situations it is possible to use mixed-dimensional models which are models that contain more than one type of element. It is indeed possible to combine in a model shell and volume elements for example. The stiffness matrix has no restriction on the type of elements it can contain.

These mixed-dimensional analyses combine the advantages of each type of elements, a reduced computing time and decreased number of nodes for 1D and 2D element types and the ability to mesh complex geometries with volume elements.

Another advantage is the ability to easily modify the model when evaluating design scenarios. For example, in an optimization process, if a shell thickness is modified, only one parameter needs to be changed in a mixed dimensional analysis while, in a full 3D analysis, the CAD model must be changed.

Two major steps are needed to extract results from a mixed dimensional analysis. The first one is determining which component of an assembly, or which portion of a component can be modeled by using reduced dimensional elements. To make this task simpler and more automated, methods have been developed in order to determine automatically what can be reduced. The medial axis transform [3-5] and the work done by Chong in [2] are examples of the progress being made on this subject. Unfortunately, this task usually referred to as geometric idealization is far from being fully automated and a lot of work remains to be done on this subject. In the work presented here, idealized models are built by the designer in a CAD system.

Fig. 1 illustrates an example of a mixed dimensional model (or idealized model). A component is partly modeled by using 3D (volume) elements for complex geometry, beam elements for long and slender portions of the geometry and shell elements for portions of the geometry which can be accurately described by a face associated with a small wall thickness.

The next step in the mixed-dimensional analysis process is to take the idealized model, mesh it and solve the resulting model to obtain analysis results.

Unfortunately these analyses also face some problems, mainly at the interface between elements featuring different dimensions. In this paper, a zone where 1D (beams), 2D (shells) and 3D (volume) elements join is referred to as a *dimensional interface*. We will demonstrate here how these problems can be overcome solely by generating a mesh featuring specific arrangements of classical 1D, 2D and 3D finite elements. This paper focuses on these arrangements when coupling shell elements with volume elements.

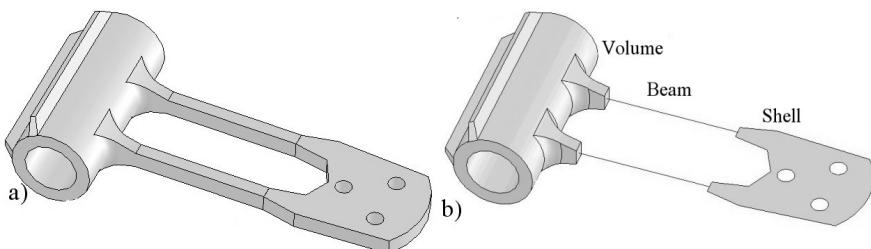


Fig. 1. Example of a mixed dimensional model a) Original model b) Idealized model

2 Problems with Mixed-Dimensional Analysis

When trying to join elements featuring different dimensions, two major problems arise. The first problem is meeting requirements of mesh continuity across a dimensional interface and the second is incompatible degrees of freedom (DOF) across a dimensional interface.

2.1 Mesh Continuity

A mixed dimensional model is created in a CAD system as a group of parts modeled with various dimensions. The mesh generator sees these parts as different entities and meshes them independently. This results in an incompatible mesh at dimensional interfaces. Even if the different parts are in contact with each other, if nodes are not co-incident, these parts are not practically bounded together from a FEA perspective, which implies that the analysis will fail or that results will be inaccurate.

Problems also arise when only some nodes are coincident while others are not. Fig. 2 illustrates this specific situation. A W-beam is modeled using three 2D faces. These faces are then meshed independently with shell elements. Nodes of each face are not always coincident with other nodes at the interfaces. In the process, coincident nodes are merged whereas many nodes at the interfaces are left unmerged. Of course, such a model leads to inaccurate results at the interfaces. High stress concentrations appear near merged nodes since the model represents a situation where faces in contact are bounded together only at merged nodes locations. We will explain below how discontinuous binding between parts (with identical or different dimensions) can be avoided.

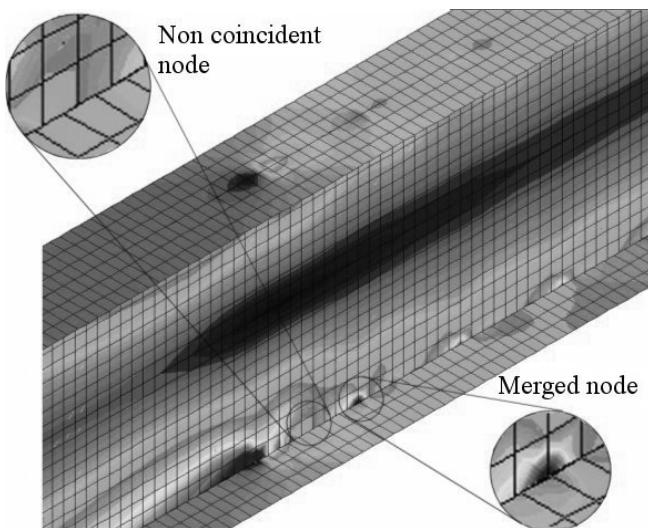


Fig. 2. Examples of a mesh continuity problem, coincident nodes are merged while others are not

2.2 Incompatible Degrees of Freedom

The second major problem faced when using mixed-dimensional analysis is incompatible degrees of freedom (DOF) at a dimensional interface. Typical 3D elements only feature three DOF (3 translations) per node while shell elements typically feature five or six DOF per node (three translations and two or three rotations).

Consequently, when coupling volume and shell elements, even if nodes of both parts in contact are merged, problems can still arise in some circumstances. Fig.3 illustrates a

straight line contact between a shell and a 3D volume. The forces applied on the shell induce a bending moment in the shell. This bending moment is transmitted through the shell thanks to the shell's rotational DOF. Nodes located at the straight line contact (common to shell and volume elements) bring about FEA modeling problems as volume elements do not "follow" rotations of shell element. Assembly of the stiffness matrix as is leads to a singular matrix or to inconsistent results.

Thus, coincident nodes directly merged this way are akin to ball-joint links. Since in the FE model, the shell and volume intersect as a series of coincident nodes, the incompatibility between DOF results in a situation where the shell and volume are joined by a hinge.

We will also explain below how this second type of inconsistency at a dimensional interface can be avoided.

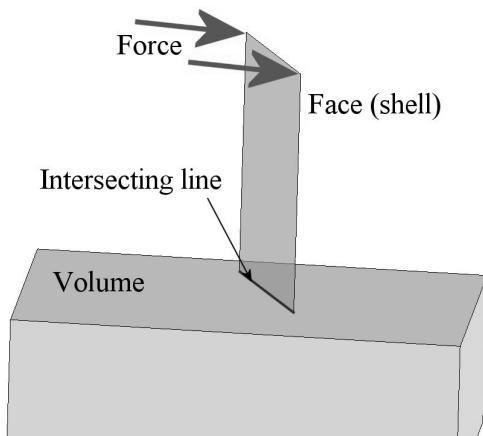


Fig. 3. A shell – volume intersection

3 Alternate solutions

Some techniques were designed to circumvent these problems. Shim [6] describes a way to couple shell and volume elements by adding constraints equations to the solver. These equations are based on equating the mechanical work for elements located on both sides of a dimensional interface and tend to overcome inconsistencies between DOF. Some FEA software feature functions used to couple shell and volume elements. These functions add constraints equations akin to those described above, but accuracy issues related to the use of this type of approaches have been outlined [7].

Another approach to overcome inconsistencies between DOF is to use volume elements with rotational DOF. Indeed, 3D elements have been designed, based on Allmann's triangle with rotational DOF [8]. This triangle, designed for 2D analysis, is a quadratic element with only three nodes, each with 3 DOF (two translations and one referred to as a pseudo-rotation). Since then, many tetrahedron element types have been designed, based on the same principles [9]. Sze [10] has referenced a large number of such elements. It should be noted however that these elements are designed to

improve on the classical linear tetrahedral element and the additional rotational DOF is not designed with the intent of coupling elements of different dimensions. The use of this type of elements implies that the connection between beams or shells and solid elements is concentrated at one point or one line. Therefore, stress distributions in solid parts around these interfaces are completely independent of beam section shapes and shell thicknesses, which introduce significant inaccuracies.

4 Prior Work

Prior to the work presented here, a method was developed to couple beams and volume elements using only specific arrangements of typical 1D, and 3D FEA elements [11]. This method came from an adaptation and combination of two suggestions made by Craveur [12] and it results in solving both problems mentioned above (mesh continuity and incompatible DOF) through the two following steps: *face splitting* and *adding mini-beams*.

Face splitting aims at solving the mesh continuity problem with the objective of finding a solution that is independent from both the mesh generator and the solver. Therefore the best way to force the mesh generator to create nodes at correct locations is to use a technique which consists in splitting a face located at a dimensional interface, in several pieces. This operation generates one or more faces that fill the space of the original face. This is done to add at least one vertex at the intersecting point between the beam and the volume. A standard Delaunay or advancing front unstructured mesh generator will then automatically place a node at the newly created vertex. In order to improve accuracy, the shape of face splitting is the same as beam's section. This is illustrated in Fig. 4; a W-shaped beam intersects a volume at a specific location represented by a point. Two small internal faces (colored in the figure) are cut inside the volume's intersecting face. As it can be seen in Fig. 4, a vertex is located at the intersecting point with the beam, therefore insuring that the mesh generator will create a node at that location.

Adding mini-beams aims at solving the problem of incompatible DOF. Beam elements called mini-beams are placed between the main beam's node and nearby nodes on the volume's boundary. While there is still the ball-joint effect explained earlier, the fact that these supplementary beams are linked with the main beam solves the problem. Of course, the mini-beams should not all be collinear (located along a straight line) which introduces a free rotation around this line. Theoretically, a non-singular stiffness matrix can be obtained with only a few mini-beams and in the example illustrated in fig. 4, this would mean adding only four mini-beams on top of each triangular edge touching the intersection point. Nevertheless, as will be explained just below, the distribution of mini-beams has been studied carefully and optimized in order to obtain analysis results that are as close as possible to the real connection's mechanical behavior.

No simple equation could be found to adapt the mini-beams' elastic properties to the behavior of the beam's section at the dimensional interface. Therefore, in an attempt to improve accuracy, a great number of tests have been made to establish the best possible configuration with the layout of the mini-beams and their properties. However, no configuration was found to perfectly recreate the behavior in every circumstance.

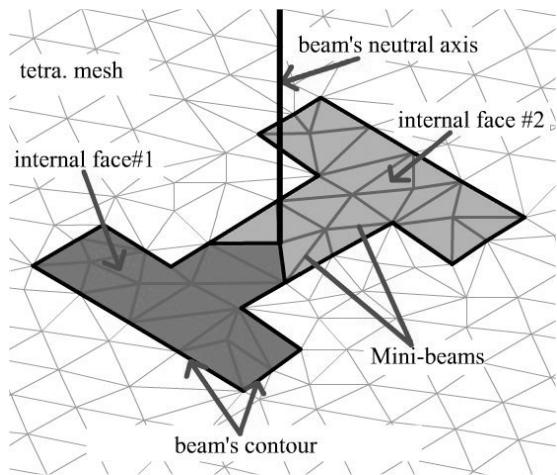


Fig. 4. A beam-volume intersection. Two faces are cut in the original face and mini-beams are overlaid on top of them.

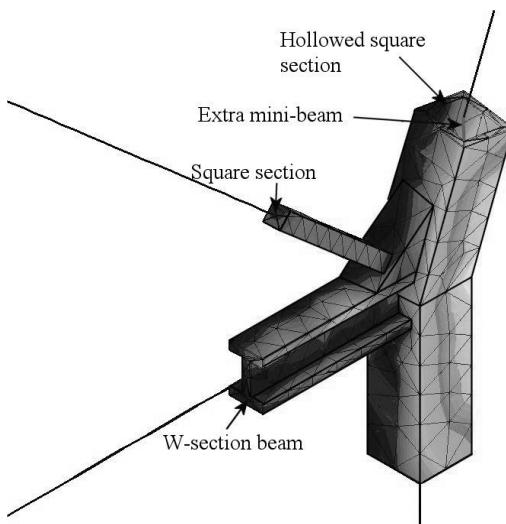


Fig. 5. Local stress results (Von Mises) for an idealized part with several beam-volume intersections

Nonetheless, an acceptable solution was found when using mini-beams with infinite stiffness. This hypothesis meets the Bernoulli-Euler beam element's formulation which states that a beam's section does not change under stress.

To make the entire beam's section rigid, mini-beams of infinite stiffness are laid on every edge of the mesh at the dimensional interface. In the example of fig. 4, this means that every triangle edge inside the two colored faces (obtained from the splitting face

process) is overlaid with a mini-beam of infinite stiffness. This effectively translates in faces of infinite stiffness.

Fig. 5 illustrates a sample stress result on a model composed of four beams intersecting at odd angles. The geometry at the intersection of these four beams is modeled using 3D elements while most of the beams themselves are modeled using beam elements. Fig. 5 also illustrates how the specific case of hollow beams is processed. In fact, a specific scheme was developed in order to deal with beams of which neutral axis does not lie in the material of the beam, which is the case of most hollow beam as well as channel type beams. A supplementary mini-beam of infinite stiffness is used to link the node at the beam's extremity and a random node on the face representing the beam's section. This can be observed in fig. 5 at the intersection with the hollow square section. In this particular case no face splitting is required.

5 Extension to Shell-Volume Intersections

The method proposed in this paper aims at solving the same types of problems (mesh continuity and incompatibility of DOF) for shell-volume intersections. Thus, the work presented below is based on an extension of beam-volume intersections procedures to the case of shell-volume intersections. In the case of shell-volume intersections, dimensional interfaces are still processed following the two steps mentioned above (face splitting and mini-beams generation) but these steps are slightly modified.

5.1 Step One: Face Splitting

The aim of this operation is now to generate a common edge at the intersection between the shell and the volume. An example is illustrated in fig. 6 where a shell intersects a volume on one of its faces. Face splitting consists in cutting two smaller faces along the intersection between the shell and the volume (colored in the figure). Like in the case of beam-volume intersections, the shape of splitting face is consistent with the shell's section, which means the shell's thickness. In the CAD software, two edges are situated at the dimensional interface, one belonging to the volume (on both smaller faces) and one belonging to the shell. These edges and their associated vertices are merged to form only one edge. This results in a non-manifold model since this intersection edge features three adjacent faces.

Once launched, the mesh generator will start by generating nodes on all vertices. Then it will mesh all edges of the model, including the common edge at the dimensional interface, only once. Then, faces are meshed starting from the edges' mesh and therefore the nodes located at the intersection are necessarily coincident between the volume and the shell.

In some cases this procedure cannot be used directly in order to create a common edge between the shell and the volume. For example, this is the case when the intersection edge between the shell and the volume does not entirely lie on a single face bounding the volume. An example of this can be seen in fig. 7. A shell intersects a volume but the intersection edge lies at two different location and extends beyond the volume.

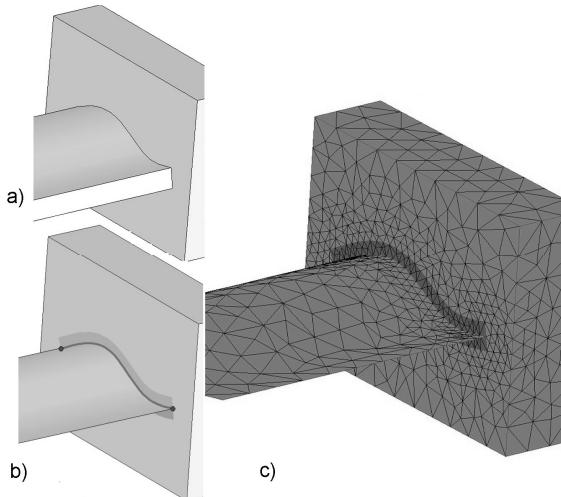


Fig. 6. The method applied to a shell-volume intersection. a) 3D Model b) Model after face splitting operation c) mesh with mini-beams (mini-beams and bounding triangles are represented using different colors also for clarity).

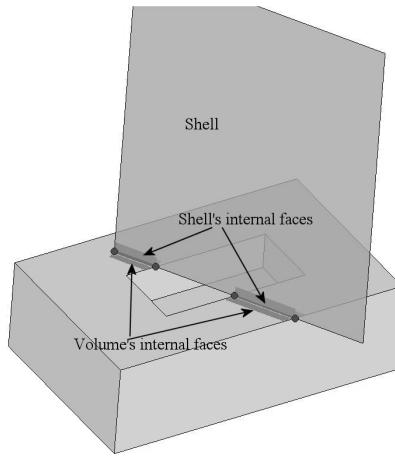


Fig. 7. In order to obtain a series of common edges, the shell's face has to be cut several times

In this case, the mesh continuity problem remains even if volume's intersection face is split correctly because the shell's edge at the intersection cannot be merged as is with an edge on the volume's boundary.

Since the Application Programming Interface (API) we are using does not feature any tool to split an edge of a shell, the solution in this case is to split the shell's face as well, as illustrated in fig. 7. Splitting the shell introduces two rectangular sub-faces, which leads to the division of the shell's intersection edge into four smaller edges. Two of these edges can now be merged with a coincident edge on the volume's

boundary at the intersection. The height of these shell sub-faces is chosen with regard to the element size as specified in the size map¹ imposed in the dimensional interface. This is done in order to obtain well shaped triangular elements when meshing the shell, as the automatic mesh generation process is constrained by splitting faces.

5.2 Step Two: Adding Mini-beams

To prevent the hinge effect described earlier (a consequence of incompatible DOF) mini-beams are also added to both internal faces once the mesh is performed. While the FE solver would not fail in the case illustrated in fig. 6 as the edge is not a straight line, adding these mini-beams first models a rigid connection instead of a hinge connection at the intersection curve. Also, like in the case of beam-volume connections, it models the connection between the shell and the volume more accurately as the connection is made on a face (representing the shell thickness) instead of a curve, reducing by the way stress concentrations at the interface. The operation is rather straightforward as the list of all triangular segments belonging to the connecting faces (contour and inside) can be easily obtained. A mini-beam is then added between both nodes of each segment so that the entire connecting face is overlaid with mini-beams.

The principle of mini-beams of infinite stiffness used in the case of beam-volume intersections cannot be used here however as the shell's section at the interface must be able to bend. Fig. 8a illustrates this type of specific cases where three forces act to

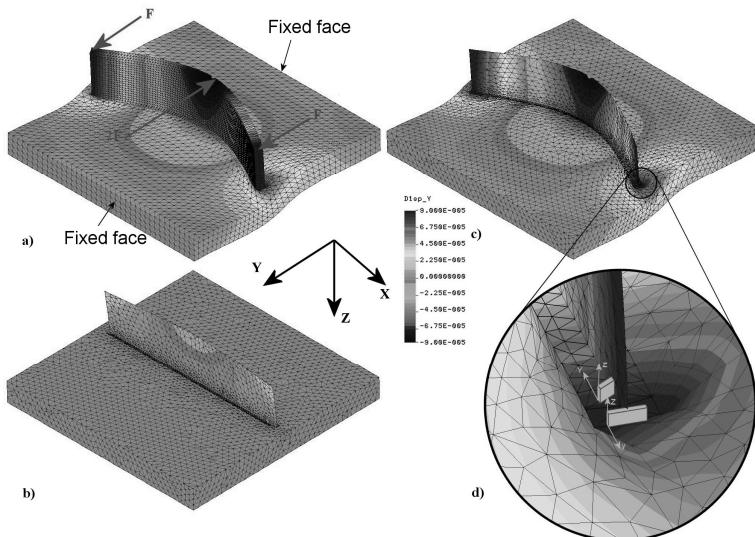


Fig. 8. Displacements results (Y direction) for a shell-volume intersection. a) 3D model b) Model with mini-beams of infinite stiffness c) Model with oriented mini-beams d) Illustration of two mini beam's orientation.

¹ In our work, the size map is a tensorial function describing the desired mesh size at each point in space.

bend the shell, but a section of infinite stiffness prevents the shell from deforming properly (Fig. 8b). In order to solve this problem, oriented mini-beams are used instead of mini-beams with infinite stiffness. Oriented mini-beams are configured with an infinite moment of inertia in one direction, but with no inertia in the other direction and a null section area. In the case of fig. 8, this means that the mini-beams have a moment of inertia I_{yy} infinitely large (see figure 8d). The stiffness in the other directions is supplied directly by the shell itself.

Fig. 9 illustrates Von Mises stress results obtained for a simple model where a portion of a shell is modeled using shell elements while the remaining portion uses volume elements. As it can be seen, the Von Mises stress field is quite continuous across

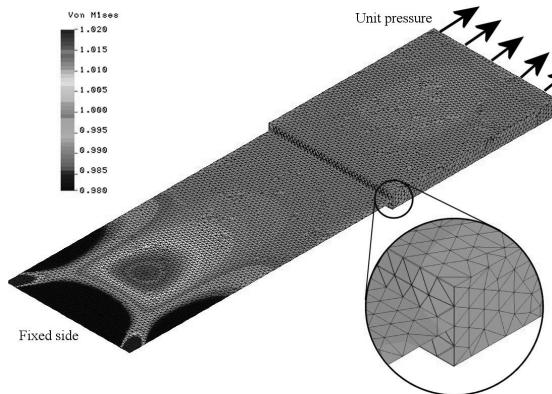


Fig. 9. Von Mises stress results on a shell (tension). Top face stresses are illustrated for shell elements.

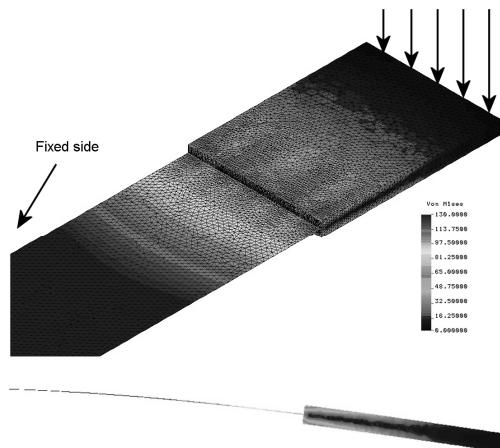


Fig. 10. Von Mises stress results on a shell (bending). Top face stresses are illustrated for shell elements. 3D view and side view pointing out the deformed shape (at the bottom).

the part except for very small discontinuities near the dimensional interface between volume and shell elements. In this particular case these discontinuities appear because of the inability to obtain mini-beams of real zero and infinite stiffness. Fig. 10 features the same model under a bending load.

6 Examples

The method described above has been successfully implemented through the design of a computer program. Practically, the whole process starts from an idealized model, designed in a commercial CAD system. Technical data such as wall thicknesses (for shells), section properties and orientation (for beams), boundary conditions and mechanical properties are also integrated into the model using an interactive interface, as well as an imposed size map for mesh generation. After this, the remainder of the process is fully automated. This automatic process starts with modifications to the idealized model (essentially face splitting) which are led directly in the CAD system using an Application Programming Interface (API). This API is also used to provide the geometric model with technical data mentioned above. This API is first used to process the identification of intersections between shells and volumes (for what concerns the scope of this paper), and then to apply face splitting algorithms. It is also used to flag internal faces to be overlaid by the mesh generator with mini-beams and to merge edges and vertices. Then, the addition of mini-beams is carried out by specific procedures integrated in the automatic mesh generation system developed by our research team [13-15].

It can also be mentioned here that the size map constraining mesh generation can be either specified by the analyst or by an automated process (referred to in our work as automatic mesh pre-optimization [16-18]). Also, in the context of the work presented here, the size map is adapted according to the shape of internal faces (resulting from face splitting).

Table 1. Comparison of the different models

	6 legged support	Coat hanger	Ventilation gate
Mixed dimensional analysis			
Nb. elements	23 171	20 121	42 310
Nb. of DOF	20 295	17 900	47 646
Max displacement (m)	2.85 e-9	2.242 e-9	1.589 e-7
Full 3D model (quadratic tetrahedron)			
Nb. elements	88 048	26 608	34 373
Nb. of DOF	410 316	129 201	200 982
Max displacement (m)	2.995 e-9	2.248e-9	1.497 e-7

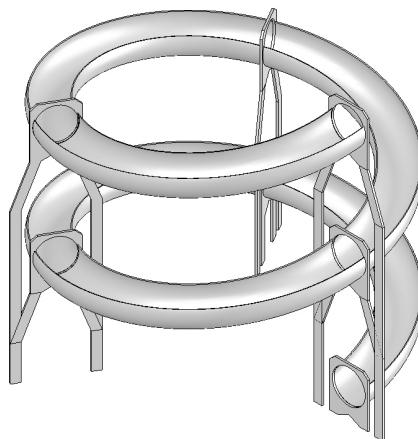


Fig. 11. Solid model of the waterslide

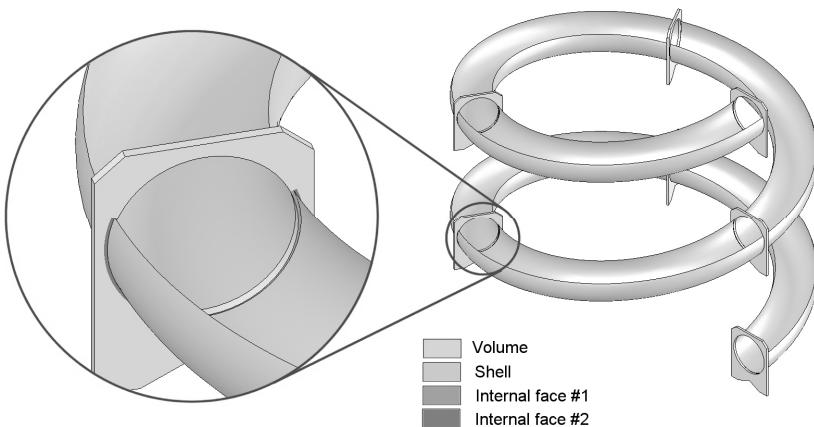


Fig. 12. The idealized model after face splitting

Table 1 summarizes a few simple examples modeled with the method described in this paper. The three different models are compared with results obtained from a full 3D model. The mixed-dimensional models use linear tetrahedrons, linear thin shells and oriented mini-beams while the 3D models use only quadratic tetrahedron. We were unable to compare these results with mixed-dimensional models using constraints equations as we do not have access to that code.

The use of oriented mini-beams is particularly noticeable in the case of the ventilation gate as the faces at the dimensional interface need to bend in order for the model to deform correctly. Note that the 3D models are not necessarily more accurate as tetrahedron elements are ill suited to model thin shells.

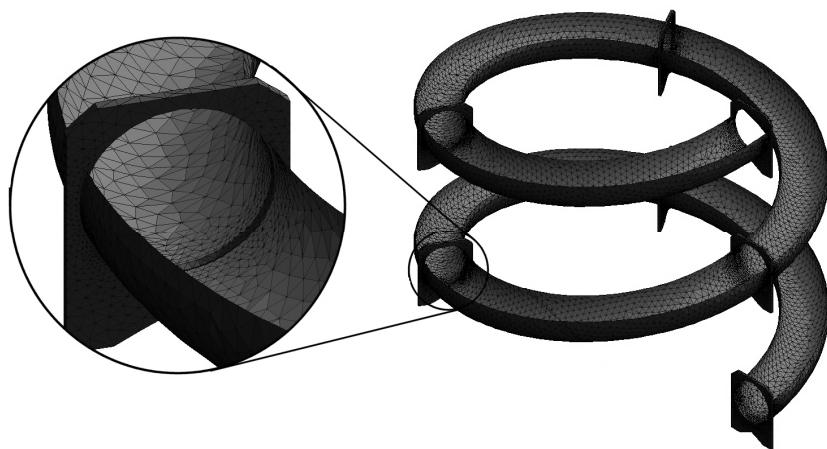


Fig. 13. Mixed-dimensional mesh of the waterslide



Fig. 14. Von Mises Stress results (on external faces for shell elements)

The next example illustrates the analysis of a waterslide structure in details. Fig. 12 illustrates the solid CAD model of the waterslide. This model is then idealized and provided with technical data (boundary conditions, shell thickness and material properties). For this example, support parts are modeled using 3D elements and the half-tube using shell elements. Fig. 13 illustrates this model after face splitting. As seen with the zoom, face splitting results are illustrated using different colors.

Fig 14 illustrates the resulting mixed-dimensional mesh. The 7 volumes and 6 shells are meshed independently. Oriented mini-beams are then added applying the proper orientation and the mesh is automatically graded around the dimensional interfaces.

The mesh, material properties, boundary conditions and shell properties are finally exported to a commercial FEA solver. The resulting Von Mises stress field can be observed in fig. 15. From this model, various design scenarios can now be easily investigated.

7 Conclusions

This paper presents a method intended to couple shell and volume elements for mixed-dimensional analysis only using specific arrangements of classical 2D and 3D finite elements. Since the technique is mesh-geometry based it presents the advantage of being independent from both the mesh generator and the solver. The mesh generator used in this work is based on advancing front techniques, but a Delaunay mesh generator could be used as well.

The software implementation of this work has proved that the concepts presented here are consistent and that the process can be fully automated from an idealized model added with relevant technical data (shell thicknesses and beam sections properties, boundary conditions and element size constraints).

In a design optimization context, this method generates reliable FEA results and in many occasions, the use of this method can provide more accurate results than using only tetrahedral elements. This is particularly the case when shell thickness is small in comparison with the size map. Generally, when using only tetrahedral or hexahedral meshing in the case of mechanical structures with beams and shells, resulting 3D meshes are either very poor (in terms of elements quality) by the way leading to poor FEA results, or excessively large (in terms of DOF), if compared to mixed-dimensional meshes.

Of course, the efficient use of mixed-dimensional analysis is also directly related to the ability to derive idealized models from 3D parametric and feature-based CAD models. Consequently, a significant research work is presently ongoing in order to provide efficient and robust tools for CAD models preparation for FEA and deriving idealized models is one of the subjects that are presently investigated.

Acknowledgments

This study was carried out as part of a project supported by research funding from Quebec Nature and Technology Research Fund and by the Natural Sciences and Engineering Research Council of Canada (NSERC).

References

1. Armstrong, C.G., Bridgett, S.J., Donaghy, R.J., McCune, W., McKeag, R.M., Robinson, D.J.: Techniques for Interactive and Automatic Idealisation of CAD Models. In: Proceedings of the 6th International Conference on Numerical Grid Generation in Computational Field Simulations, July 1998, pp. 643–662 (1998)
2. Chong, C.S., Kumar, A.S., Lee, K.H.: Automatic solid decomposition and reduction for non-manifold geometric model generation. Computer Aided Design 36(13), 1357–1369 (2004)
3. Donaghy, R.J., McCune, W., Armstrong, C.G.: Dimensional Reduction of Analysis Models. In: Proceedings of the 5th International Meshing Roundtable, October 1996, pp. 307–320 (1996)
4. Monaghan, D.J., Doherty, I.W., McCourt, D., Armstrong, C.G.: Coupling 1D Beams to 3D Bodies. In: Proceedings of the 7th International Meshing Roundtable, October 1998, pp. 285–293 (1998)
5. Suresh, K.: Generalization of the Kantorovich Method of Dimensional Reduction. In: Proceedings of the 12th International Meshing Roundtable, September 2003, pp. 261–270 (2003)
6. Shim, K.W.: Mixed Dimensional Coupling in Finite Element Stress Analysis. In: Proceedings of the international meshing roundtable, October 2001, pp. 269–277 (2001)
7. McCune, W., Armstrong, C.G., Robinson, D.J.: Mixed-dimensional coupling in finite element models. International Journal for Numerical Methods in Engineering 49(6), 725–750 (2000)
8. Allman, D.J.: A compatible triangular element including vertex rotations for plane elasticity analysis. Computers and Structures 19(1), 1–8 (1984)
9. Pawlak, T.P., Yunus, S.M.: Solid elements with rotational degrees of freedom: Part II - tetrahedron elements. International Journal for Numerical Methods in Engineering 31(3), 593–610 (1991)
10. Sze, K.Y., Pan, Y.S.: Hybrid stress tetrahedral elements with Allman's rotational D.O.F.s. International Journal for Numerical Methods in Engineering 48(7), 1055–1070 (2000)
11. Bournival, S., Cuilli  re, J.C., Francois, V.: A Mesh based Method for Coupling 1D and 3D Finite Elements. In: Proceedings of the 2nd World Congress of Design and Modelling of Mechanical Systems (CMSM 2007) (March 2007)
12. Craveur, J.C., Marceau, D.: De la CAO au calcul, Paris, Dunod (2001)
13. Cuilli  re, J.C.: An adaptive method for the automatic triangulation of 3D parametric surfaces. Computer Aided Design 30(2), 139–149 (1998)
14. B  chet, E., Cuilli  re, J.C., Trochu, F.: Generation of a Finite Element Mesh from Stereolithography (STL) Files. Computer Aided design 34(1), 1–17 (2002)
15. Francois, V., Cuilli  re, J.C.: An a priori adaptive 3D advancing front mesh generator integrated to solid modeling. Recent Advances in Integrated Design and Manufacturing in Mechanical Engineering, 337–346 (2003)
16. Francois, V., Cuilli  re, J.C.: Automatic mesh pre-optimization based on the geometric discretization error. Advances in Engineering Software 31(10), 763–774 (2000)
17. Cuilli  re, J.C., Maranzana, R.: Automatic and a priori refinement of three-dimensional meshes based on feature recognition techniques. Advances in Engineering software 30(8), 563–573 (1999)
18. Boulet, A., Cuilli  re, J.C., Francois, V.: Automatic size map calculations for mesh pre-optimization in the context of CAD-FEA integration. In: Proceedings of the 2nd World Congress of Design and Modelling of Mechanical Systems (CMSM 2007) (March 2007)

