Acoustic Prediction Made Practical: Process Time Reduction with Pre/SYSNOISE, a recent joint development by MSC & LMS

L. Cremers, O. Storrer and P. van Vooren

LMS International NV
Interleuvenlaan 70
B-3001 Leuven
Belgium
Tel: +32 (0)16 384500
E-mail: Luc.Cremers@lms.be

ABSTRACT

Finite element models for structural dynamic analysis and boundary elements models for acoustic radiation analysis have different meshing requirements. Acoustic boundary element analysis requires a mesh of the sound radiating surface with a uniform discretization of about six degrees of freedom per shortest acoustic wavelength whereby small details, relative to the acoustic wavelength, can be omitted. In most cases the acoustic analyst needs to generate the acoustic boundary element mesh from the original detailed structural finite element mesh as no geometry information is available for the model. This so-called mesh coarsening process involves in general four phases, i.e. mesh processing, subdomaining, creation of surfaces and re-meshing. Pre/SYSNOISE, a joint-development by MSC and LMS, is a powerful tool to help the acoustic analyst in this tedious task. It combines both the geometry and finite element meshing tools of PATRAN v8.0 and an advanced set of automatic mesh coarsening routines. The different techniques involved in the mesh coarsening process will be explained along with a practical real-life example.

INTRODUCTION

Boundary element (BE) techniques are well-accepted numerical techniques in vibro-acoustic modeling and design. On of the main obstacles towards an even more widespread use of these techniques is the difficulty to produce an adequate acoustic BE mesh from an existing detailed structural FEA mesh. This paper describes a new mesh coarsening tool, Pre/SYSNOISE, a joint development by MSC and LMS.

PROBLEM DEFINITION

FE meshes for structural dynamic analysis and BE meshes for acoustic radiation analysis have different meshing requirements. A structural FE model usually consists of a very detailed mesh with different types of 1D beam, 2D shell and 3D solid elements. Very often the structural model is built-up from different mesh components connected to each other through so-called multi-point constraints. 3D acoustic BE analysis of this model requires a 2D mesh of the sound radiating surface with a uniform discretization of about six degrees of freedom per shortest acoustic wavelength whereby small details, relative to the acoustic wavelength can be omitted.

The BE method is a *global* method in the sense that each degree of freedom is connected to all others. On the other hand FE based methods are *local*, i.e. each degree of freedom is only connected to its neighboring degrees of freedom. As a result BE formulations lead to full (symmetric, for indirect variational formulations) complex system matrices as opposed to sparse system matrices for FE methods. The fully populated matrices for the BE method quickly result in computational intensive analysis sequences. It's therefore clear that the number of degrees of freedom for an acoustic BE model needs to be kept to a minimum.

Very often the acoustic BE mesh needs to be generated from the original structural FE mesh as no geometry information is available for the model. The creation of an acoustic BE mesh from a FE mesh is often referred to as mesh coarsening. This mesh coarsening exercise is in general a very difficult and time-consuming task and usually can take several weeks for real-life problems. Different tools have been developed to speed up this process. In the past these tools have been limited to simplifying structural FE meshes rather than actual mesh coarsening, i.e. reducing the number of degrees of freedom. Pre/SYSNOISE is a new software tool that allows to take this extra step and provides the possibility to truly coarsen a structural FE mesh. A special so-called tesselated surface algorithm has been developed to reconstruct the geometry, i.e. points, lines and surfaces, based on the 2D FE grid information.

In general the full mesh coarsening exercise involves four phases. Starting from the structural FE grid the mesh firstly goes through a mesh simplification and cleaning step. During this *mesh processing* step the grid is reduced to only 2D elements modeling the radiating surface. These 2D elements are then grouped into a set of subdomains in a second phase. Subdomains are a set of interconnected 2D elements boarded by either free edges, junction or feature lines. The subdomains are then processed by the re-surfacing algorithm to create tesselated surfaces matching the geometry of the group of elements. In a fourth and final stage the full geometry of the model, consisting of a set of tesselated surfaces which are adequately edge-matched, can be re-meshed with a desired grid size to yield the acoustic BE mesh of the radiating surface. In the following these different stages will be discussed in more detail and illustrated using some real-life examples.

DISCUSSION

The different phases of the mesh coarsening process are clearly laid out in the main window of Pre/SYSNOISE, depicted in Figure 1. From left to right the different modules, i.e. Mesh Processing, Subdomaining, Surfacing, Geometry and Finite Elements, refer to respectively the mesh simplification and cleaning step, the creation of element subdomains, the creation of surfaces and the re-meshing step.

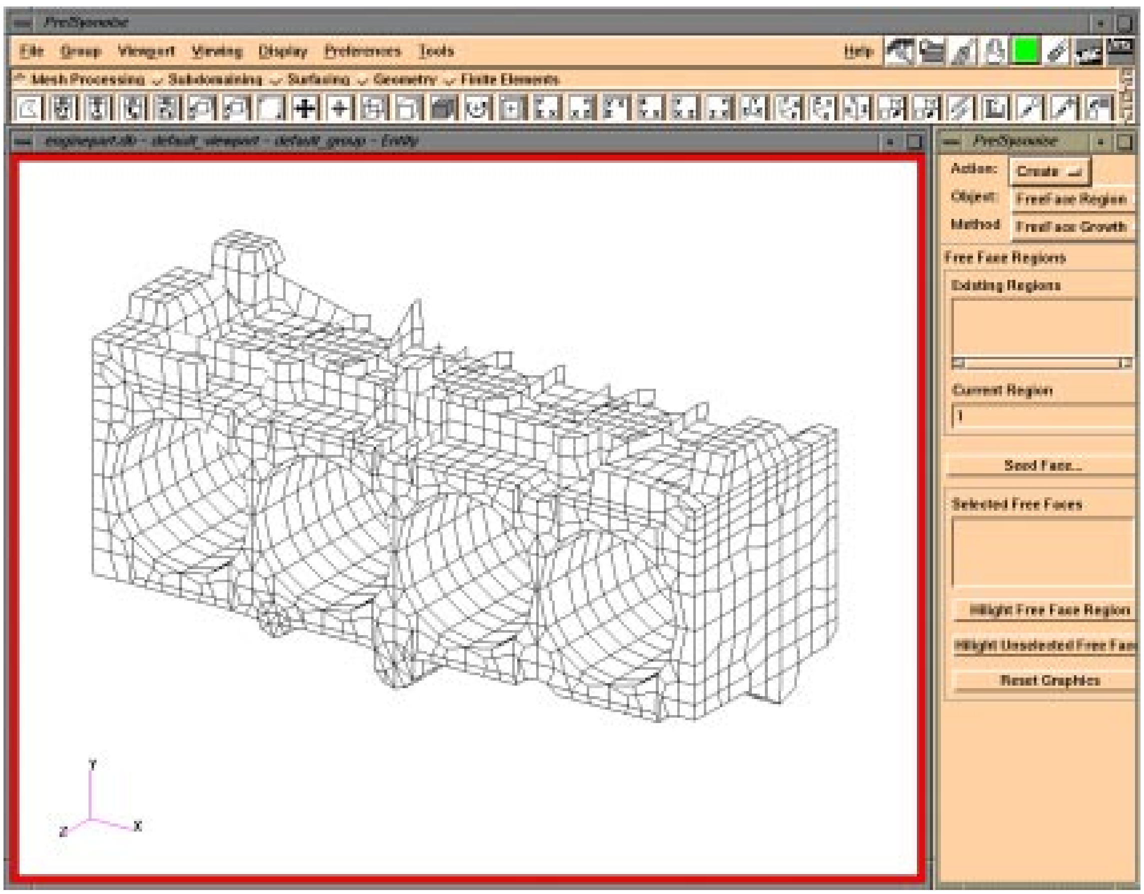

Figure 1: Pre/SYSNOISE main window with the original structural FE mesh of the engine block part.

The different steps of the full mesh coarsening exercise will now be illustrated using an example of an engine part. Figure 1 and 2 feature the structural FE mesh of the upper part of an engineblock. The original mesh consists of 3609 nodes and 3495 elements. The exercise is now to generate an acoustic BE mesh of the radiating surface of this model which is valid for frequencies up to

1000Hz. With the common rule of thumb of at least six degrees of freedom per shortest acoustic wavelength this leads to a maximum element edge size of about 0.05m, i.e. $\frac{1}{6}\left(\frac{c}{v}\right)=\frac{1}{6}\left(\frac{340}{1000}\right)\cong 0.05$ (acoustic medium is air).

In a first phase different tools will be used in order to clean or fix and simplify the mesh. 1D elements are simply identified and deleted from the mesh. The 3D solid elements are skinned such that the external skin or free faces are converted in equivalent 2D elements and the original 3D elements can be removed. At this stage the mesh contains only 2D elements. These 2D elements now need to be checked for compatibility with the acoustic BE method. A special mesh checking routine searches for incompatibilities and fixes them where possible. An incompatibility is for example a quadrilateral element that is connected to another element across its diagonal. The algorithm identifies such occurences and fixes them by simply splitting the original quadrilateral elements by a set of two triangualar elements. Another incompatibility that can be fixed is different elements that are superimposed or the identification of isolated grid points.

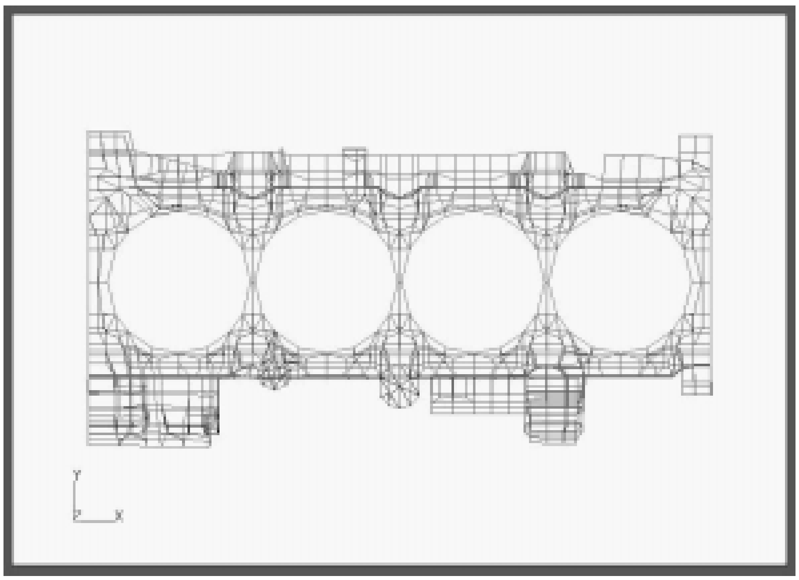

Figure 2: Structural FE mesh of an engine part.

To further simplify the mesh different small details can be identified and removed from the grid. An example of this is the so-called rib removal process. Ribs are considered as a detail in an acoustic sense if they are small compared to the smallest acoustic wavelength. An automatic procedure determines and highlights all possible sets of elements that can be considered as a rib. A set of elements can be considered as a rib if the ratio of grid points, belonging to either free edges or junction edges, with respect to the total set of grid points is larger than a user-specified value; whereby at least one grid point belongs to a free edge. The user than needs to decide which ribs can be classified as an acoustic detail. Figure 3 shows on the left a set of ribs that are identified and consequently removed, yielding the simplified radiating surface shown on the right.

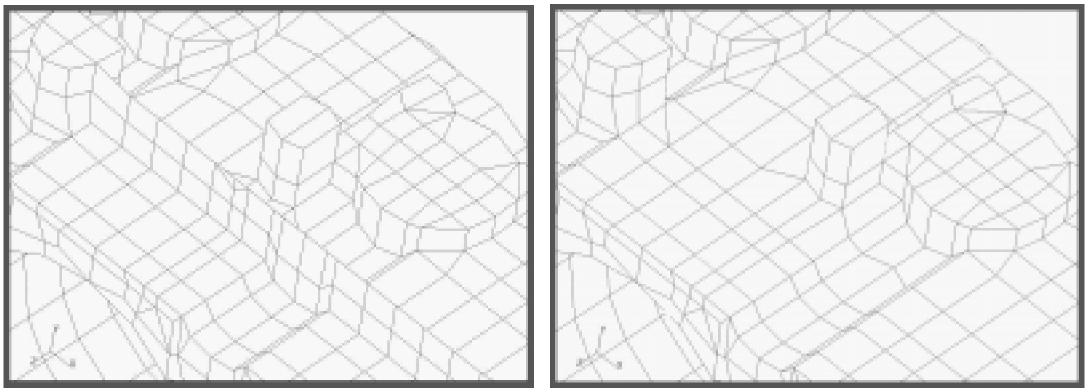

Figure 3: Automatic rib removal.

Another important step in the mesh simplification process is the identification of the radiating surface or, in other words, the removal of elements that are located inside the object and that are not directly in contact with the acoustic medium. The interior elements will not contribute to sound radiation within the acoustic domain and are thus preferreably removed. The so-callled *walker* algorithm is designed to detect this radiating surface. Starting from a user specified element and element side the algorithm explores the surfaces by *walking* across the connected elements while keeping the element normal consistent with the normal of the starting element. The radiating surface is then the ensemble of elements that has been *walked* upon. It's clear that such an algorithm will fail if small holes permit the *walker* to enter the interior of the object. If this is the case an automatic hole fitting alogithm is available to close this access into the interior (as discussed in a following paragraph).

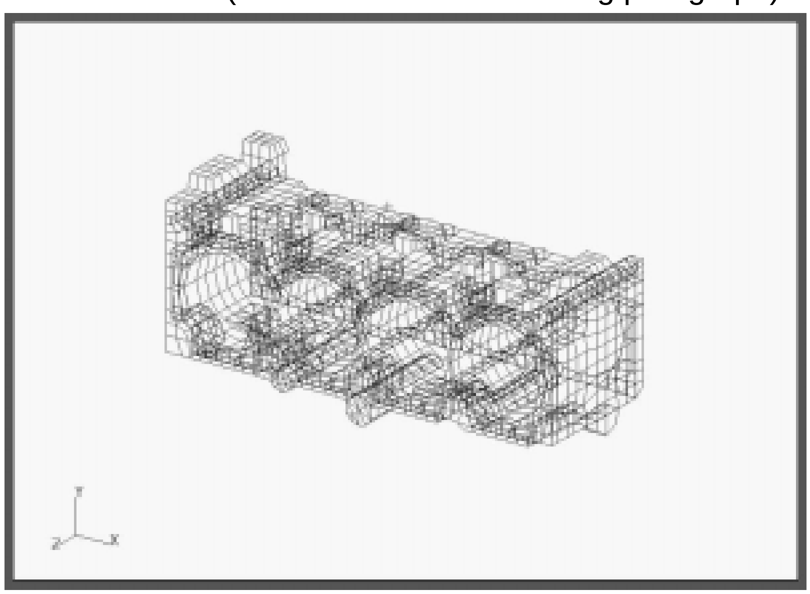

Figure 4: Removal of interior elements based on the automatic walker algorithm started at the exterior side of an element on the radiating surface.

As soon as a minimal set of 2D elements has been obtained the subdomaining phase can be started. The idea here is to subdivide the whole mesh in a set of subdomains reflecting the overall geometric features of the model. The automatic subdomaining algorithm is launched with a user specified feature angle. The feature angle is the angle between the element normals of two connected elements. The algorithm groups the elements in sets of connected elements that are boarded by either a free edge, a junction edge or a so-called feature line. A feature line is built from the common edges of elements for which the feature angle exceeds the predefined value, as depicted in figure 5. Figure 6 display the different subdomains for the partial engine block model for a feature angle of 15 degrees.

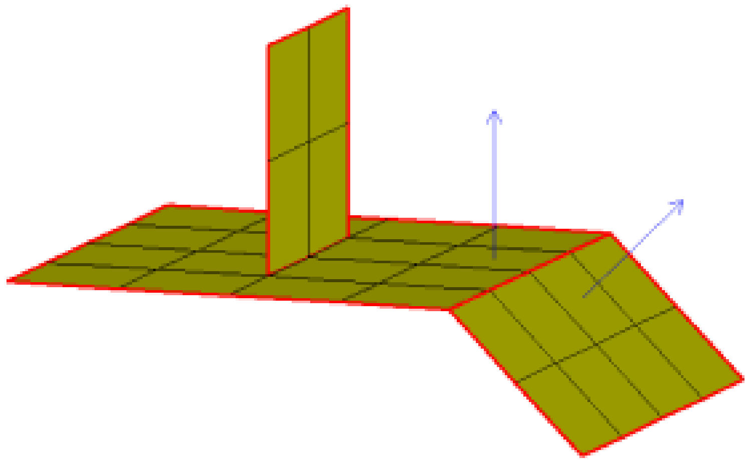

Figure 5: Feature angle definition.

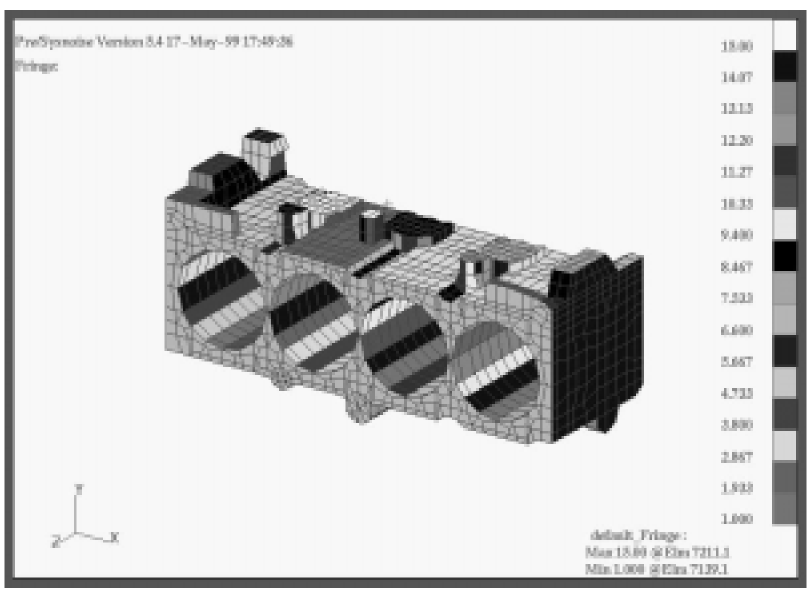

Figure 6: Subdomains based on a feature angle of 15 degrees.

As already mentioned before closing a hole in the object can be beneficial in an acoustic sense, i.e. when ignoring the acoustic radiation from interior parts of the model. Again the decission on wheter or not a hole can be closed is up to the engineering judgement of the user. In the example at hand we decide that for the considered exterior acoustic radiation analysis the cylinder holes can be closed off. The hole filling algorithm identifies the four cylinder holes automatically and fills them up with temporary 2D elements. As soon as the cylinders have been closed off the radiating surface detection algorithm is launched once again in order to remove the elements of the four cylinder on the inside of the enigine part, as shown in figure 7 and 8.

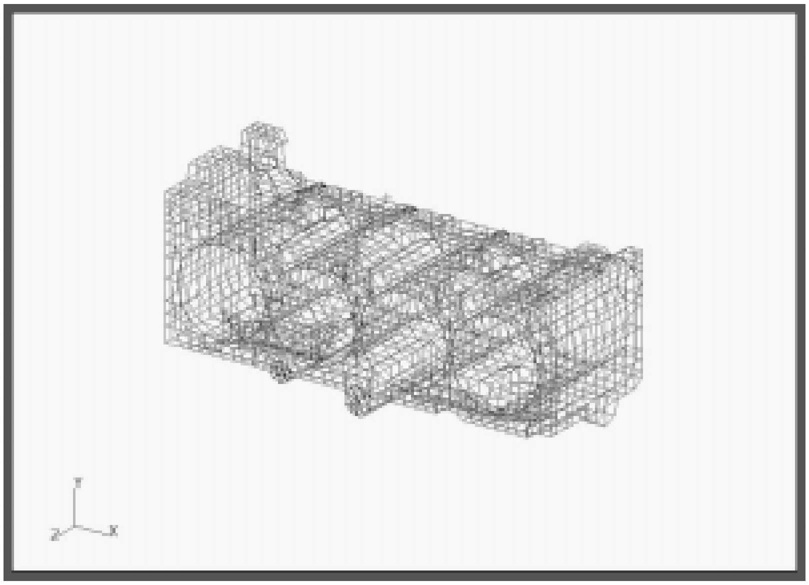

Figure 7: Hole identification.

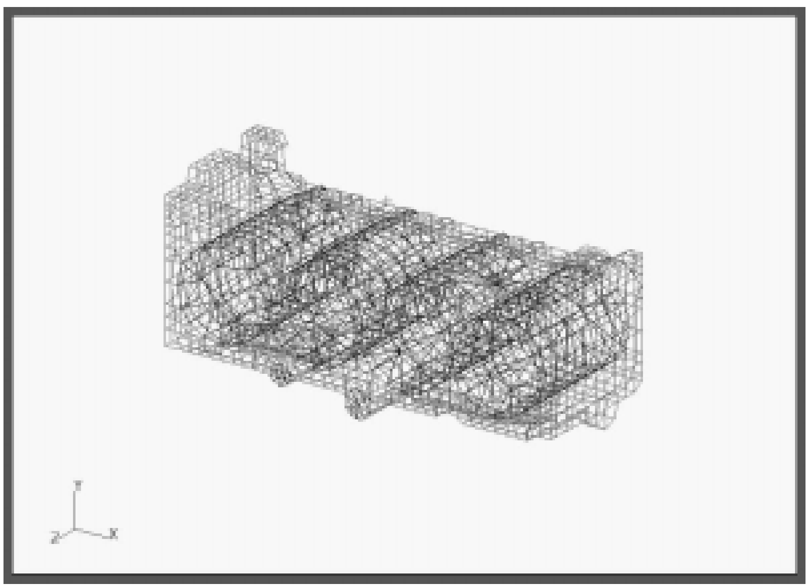

Figure 8: Filled holes and removal of interior elements.

The model is now ready for geometry re-creation. The tesselated surface algorithm is launched for all subdomains. A tesselated surface is fitted through the elements of the subdomain and the original elements are removed from the grid. The tesselated surface replace the subdomain of elements. While creating the tesselated surfaces care is taken to properly match the edges of neighbouring surfaces to ensure element compatibility across surfaces in a subsequent meshing phase. Figure 9 show the geometry on the right created starting form the subdomain of the mesh shown on the left.

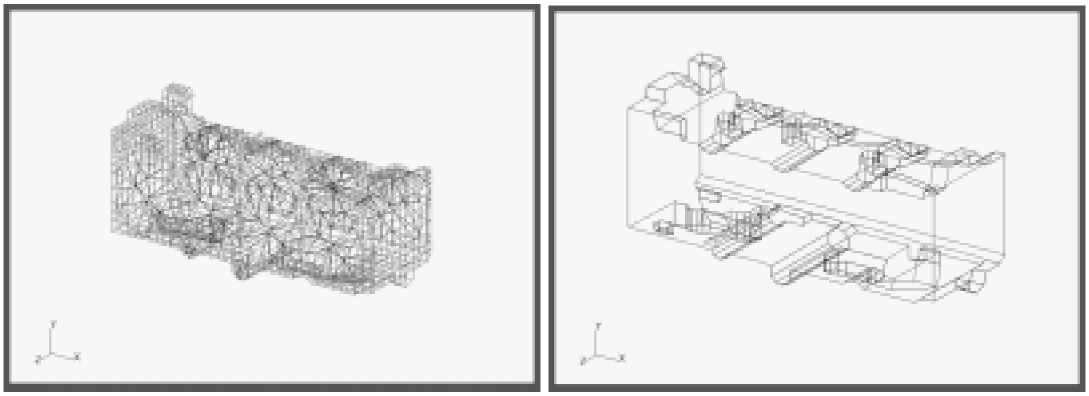

Figure 9: Geometry creation using an automatic tesselated surface algorithm.

This geometric model is now ready to be re-meshed. The automatic tesselated surface algorithm ensures edge matching between the different surfaces. If necessary this edge matching can be double-checked using the edge matching function available in the geometry module. Based on the generally accepted rule of thumb of six degrees of freedom per smallest acoustic wavelength, a uniform mesh seed is now imposed on all edges as shown in figure 10.

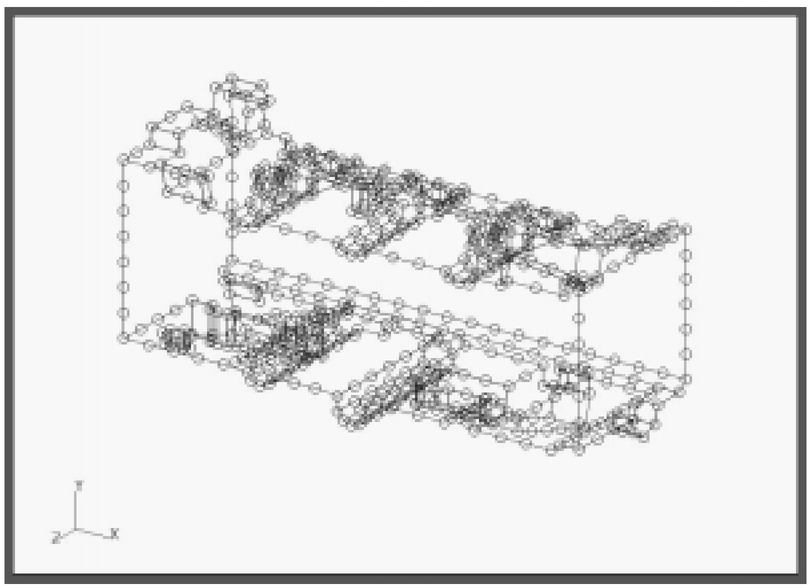

Figure 10: Uniform mesh seed according acoustic rule of thumb of six degrees of freedom per shortest wavelength.

The geometry is then meshed based on this mesh seed using the PATRAN automatic mesh algorithms yielding a coarse acoustic BE mesh as depicted in figure 11. This BE mesh now consists of only 1104 nodes and 1162 elements.

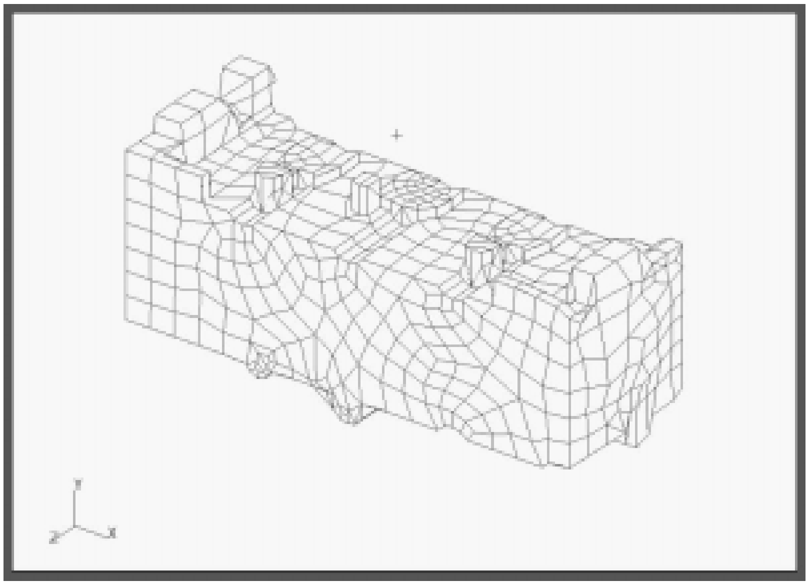

Figure 11: Acoustic BE mesh of the radiating surface.

Figure 12 shows both the structural FE mesh and the acoustic BE mesh imported in SYSNOISE Rev 5.4. A typical vibro-acoustic analysis, in the weak-coupling sense, would proceed by generating acoustic normal velocity boundary conditions on the boundary elements based on the structural response calculated by a structural FEA code. The generation of the acoustic normal velocity boundary conditions includes an interpolation algorithm to account for the transfer of data between the two incompatible grids.

For a fully coupled vibro-acoustic analysis, whereby the interaction between the structure and the acoustic medium is taken into account, a typical approach is to model the structure using a modal basis. The structural modal basis is first transferred onto a compatible grid using a similar geometric interpolation algorithm. The so-called surrogate structural mesh is often identical to the acoustic BE mesh. The modal system matrices of the structure can then simply be coupled to the acoustic BE matrices and solved simultaneously.

A fully coupled analysis is in general only necessary when the acoustic loading on the structure by the acoustic medium is important. This is the case when studying the dynamics of rather flexible structures within a relatively heavy acoustic medium. In other cases the acoustic loading by the structure can be neglected within a one-way or weakly coupled solution scheme.

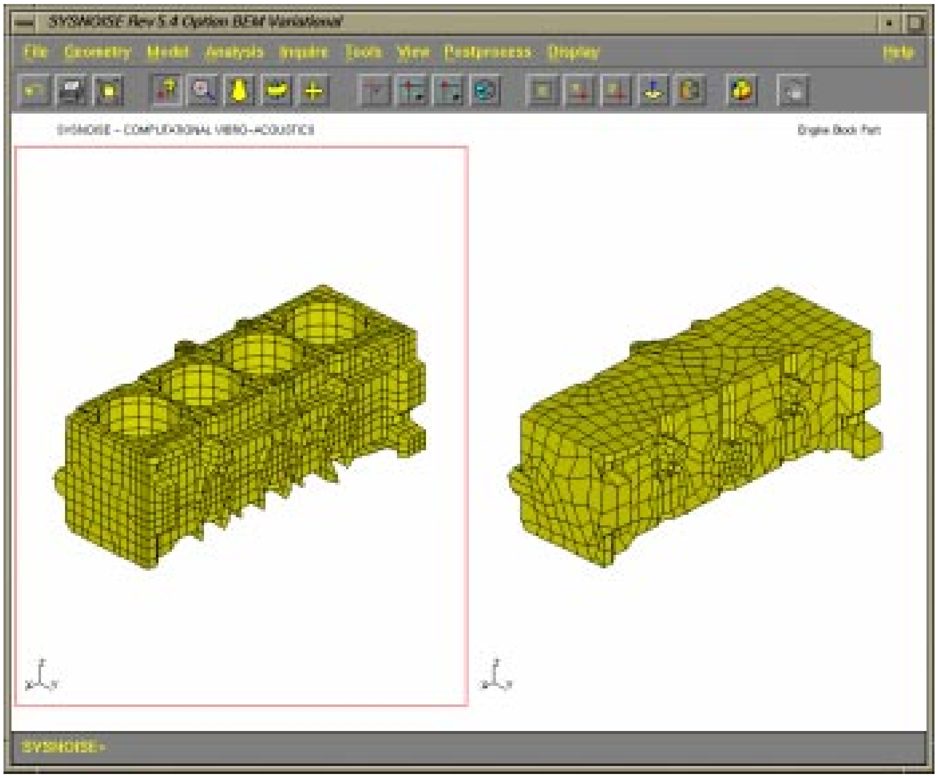

Figure 12: Structural FE and acoustic BE mesh in SYSNOISE Rev 5.4.

CONCLUSIONS

Pre/SYSNOISE, a new mesh coarsening tool jointly developed by MSC and LMS, has been presented. The software is targeted at facilitating the difficult and time consuming exercise of acoustic BE mesh generation through structural FE mesh coarsening. Automatic procedures to generate acoustic BE meshes from detailed structural FE meshes have been presented. Different tools aid the user in simplifying and fixing the structural FE mesh while a unique automatic tesselated surface algorithm is available to completely reconstruct the geometry of the model only based on original grid data. The geometry can then automatically be meshed using a mesh density best suited for the vibro-acoustic analysis in the desired frequency range. The BE model size can therefore be minimized, in turn leading to superior calculation times and computer memory requirements, and thus making acoustic prediction more practical.

REFERENCES

[1] 'Pre/SYSNOISE – a tool for reducing design cycle times', SYSNOISE User's meeting, Leuven, Belgium, (1997).

[2] 'Pre/SYSNOISE – Getting Started Manual', LMS International.