

# A PRE-PROCESSING INTERFACE EMBEDDED IN ANSA FOR ACOUSTIC RADIATION ANALYSES WITH RADACT

<sup>1</sup>Tassos Sarridis, <sup>2</sup>Chryssa Sferidou\*

<sup>1</sup>BETA CAE Systems, Greece, <sup>1</sup>BETA CAE Systems, Greece

KEYWORDS – Acoustic, ACTRAN, FRF, Modal Response, RADACT, Radiation

ABSTRACT – RADACT is a module of ACTRAN, an acoustic radiation modeler, which is able to perform acoustic radiation analysis in batch mode based on NASTRAN formatted structure results, structure and acoustic mesh.

Motivation:

- RADACT does not have any GUI for creation of the ACTRAN input file prior to submitting it to the solver and, therefore, needs to be written manually.
- Users, who are used to build up a model with a common pre-processor like ANSA, have to learn with ACTRAN another GUI, in order set up an acoustic radiation analysis.
- The structural model and the fluid volume mesh have to be created within the pre-processor anyway, so why not set up also the RADACT analysis, which needs the fluid and structure mesh as input file.

Solution:

The RADACT interface embedded in ANSA provides the user with capabilities to easily create and export a RADACT input file which is ready to be solved with the ACTRAN solver. Moreover, it creates automatically or imports all necessary meshes for the acoustic finite and infinite domain by using the convenient selection capabilities embedded in ANSA for the respective properties from the model. It assigns also the respective structural results file and sets up all necessary solver and analysis parameters in order to export the complete RADACT input file.

TECHNICAL PAPER -

## 1. INTRODUCTION

Nowadays, customer demands and expectations regarding acoustic performance, along with the tightening of legal regulations on noise emission levels and human exposure to noise, increases the demands on the noise and vibration properties. Moreover, they turned into important design criteria during the engineering of any device that operates under dynamic load conditions.

In this framework, this challenges the manufacturers to achieve higher-quality products in ever shorter time frames, while at the same time, reduce the costs. This can only be achieved when, during the design evolution, the virtual prototyping and simulation are intensively used in the early design stage, reducing in parallel the traditional and time consuming test phase and the building of expensive physical prototypes in a minimum. As a result, each stage of the design cycle is supported by CAE (Computer Aided Engineering) methodologies, which allow the prediction of the functional performance for attributes like NVH – especially, the exterior acoustics. Moreover, the researchers and software developers are continuously enhancing and developing new modeling methodologies and automations increasing the efficiency and thus, enabling shorter virtual design cycle with lower costs. In the field of exterior noise analysis resides ACTRAN, an acoustic radiation modeler within the MSC software suite, widely used in the automotive, aircraft, aerospace and audio manufacturers. Its complementary sub-module RADACT is able to perform acoustic radiation analyses in batch mode, based on NASTRAN formatted structure results as boundary conditions (so called BC\_MESH boundary condition), the structure and acoustic mesh.

---

One of the main advantages of the RADACT module is that, in order to perform an exterior noise analysis, there is no need to create an ACTRAN model. Just a reduced input ASCII file is used, which comprises of all needed commands, calling the respective model parts and controlling the respective solver and analysis parameters.

However, as any manually written command file, there are some shortfalls that emerge, due to the lack of GUI support. Such shortfalls are the uncertainty during the input of the respective PIDs for each of the structure model parts or acoustic fluid mesh, or other solver specific commands, which could make the whole analysis process error prone increasing the reworking time.

In this paper, a new GUI will be introduced for the first time, which is embedded in ANSA and provides the user with capabilities to easily create and export a RADACT input file which is ready to be solved with the ACTRAN solver. Furthermore, it automatically creates or imports all necessary meshes for the acoustic finite and infinite domain by using convenient selection tools within ANSA. It also assigns the respective structural results file and sets up all necessary analysis and post-processing parameters, in order to export the complete RADACT input file.

## 2. PREREQUISITE MODEL PREPARATIONS

### Model preparation for the structural Modal Response Analysis

The introduction of the RADACT interface will be done based of the application on a real case example. The acoustic radiation, based on a tri-axial dynamic load excitation on its main shaft will be analyzed up to a frequency of 1 kHz.

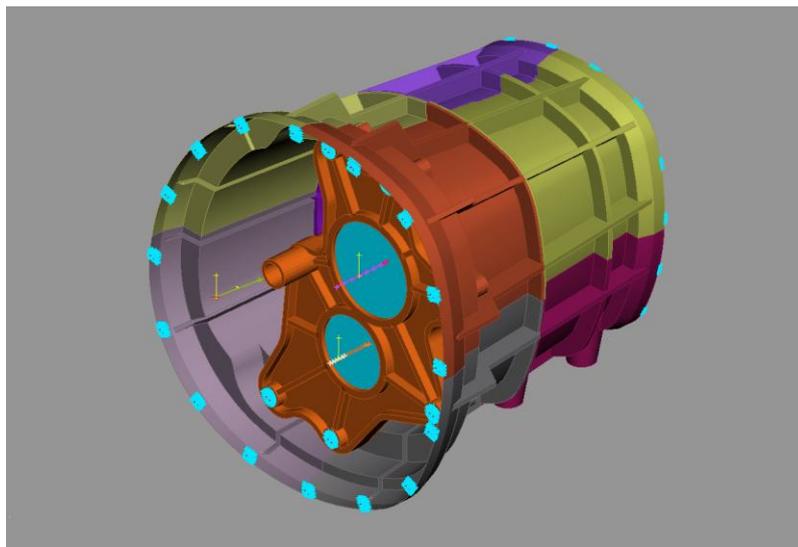


Figure 1 – The gearbox to be analysed, in PID view mode

One of the major steps for this exterior noise analysis with the ACTRAN solver is to have the Displacement result on all nodes of the structure in .op2 format, which are given from a SOL 111 modal response analysis with NASTRAN, as the boundary conditions for RADACT. The load-case set up, which consists of a combined dynamic load excitation, can be fully set up within ANSA, especially with the enhanced NASTRAN header since version v14.0.0.

---

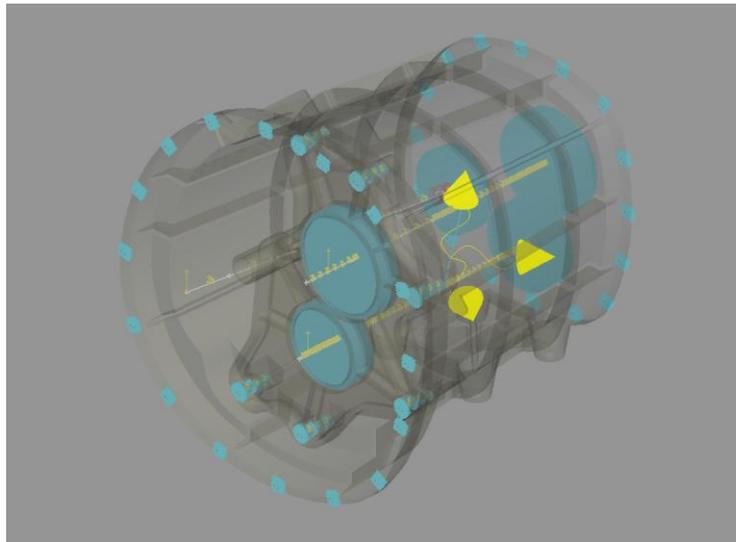


Figure 2 – The tri-axial dynamic load excitation at the centre of the main shaft

#### Model preparation for the acoustic fluid mesh creation

In order to create the mesh needed for the exterior noise analysis and to analyse the radiation characteristics of the structure for that load-case, one can use the meshing capabilities of the ANSA pre-processor. Once having the structure ready, one of the very next steps is to create a wrap mesh around the gear box. It is advisable to close the bigger openings of the gear box with the functions FILL GAP>Openings or Stich and to put the new created shell elements, which serve as auxiliary elements for the wrap functionality, into a separate part and/or property, in order to isolate them easily for the later remove, once the wrap mesh is once done.

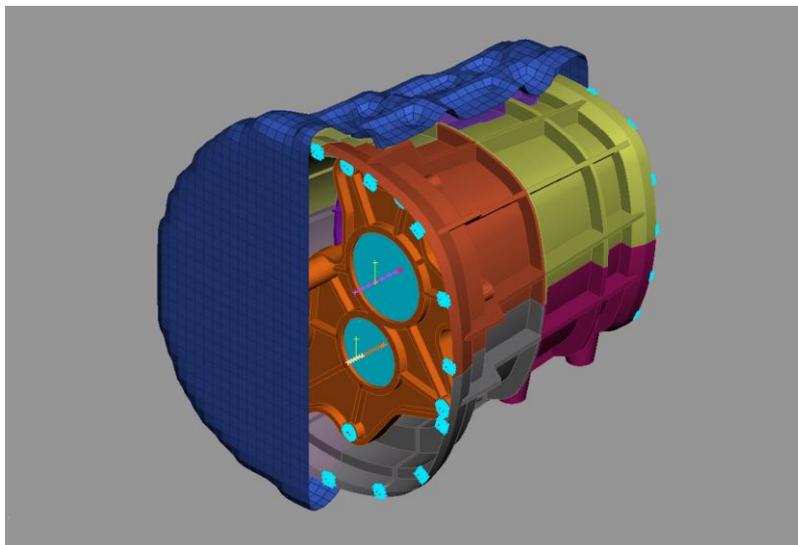


Figure 3 – The smooth wrap mesh (in blue) of the gearbox, which will act as the boundary surface, holding the structural results, later on in the RADACTION analysis

The wrap mesh will act later in the RADACTION/ACTRAN analysis as structure and will be called as BC\_Mesh. Furthermore, it represents the surface of the structure where the normal directions of the displacement results, from the modal response analysis, will act as boundary conditions in ACTRAN.

---

After the creation of the wrap mesh, the next step is to create the outer boundary (infinite surface) which encloses the acoustic cavity mesh, that will be created in the next step, and which can be created in three ways.

1. Creation of a convex mesh:

The function SHELL MESH>ELEMENTs>CONVEX will be used, in order to create an outer envelop with an absolute distance of e.g. 200 from the wrapped surface. The result is an envelope of an STL type mesh. With the aid of the functions RECONS and SUPPRESS it is possible to smoothen the result mesh and to assign the user given mesh size to it.

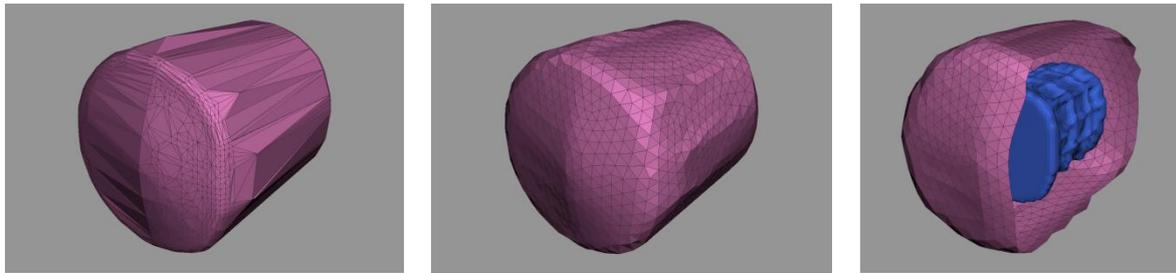


Figure 4 – Smoothing of the outer envelope created by the CONVEX function

Creation of a sphere:

Another way to create the infinite area is to use the build-in standard surface features. With the TOPO function SURFs>VOLUME>Sphere, it is possible to create a sphere surface directly based on the COG of the wrapped surface. The only thing that needs to be further done is to mesh this sphere with the desired mesh type and size.

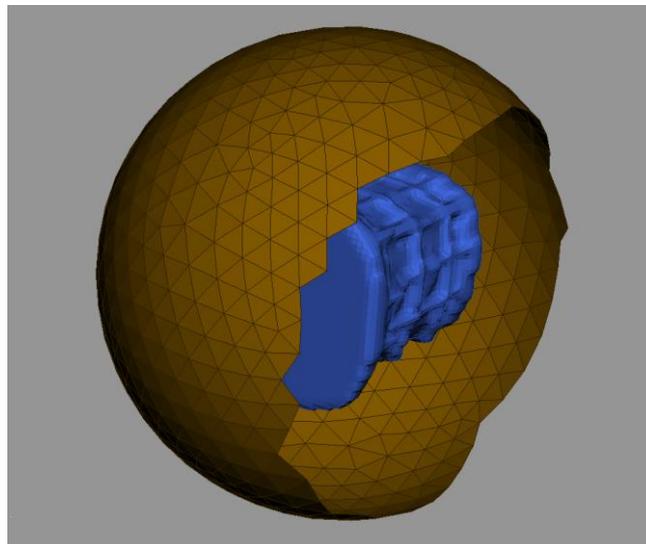


Figure 5 – Fluid mesh envelope created by a standard spherical surface around the COG of the wrapped surface.

2. Using a generic envelop combined with an automated adaption

The 3rd way to create the infinite area, is to use a generic envelop with an initial shape of an ellipsoid which can be combined with a script function that adapts it to any user given structure. The bellow visible infinite area is represented by an ellipsoidal surface which size and surface mesh type can be adapted to the demands of the given structure and analysis set up.

---

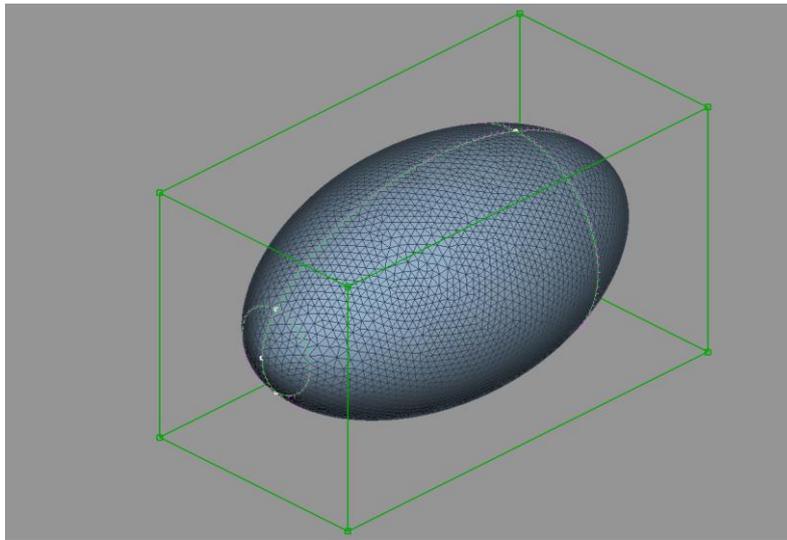


Figure 6 – Standard ellipsoidal infinite area, adaptable to any structure by an automated way and via a morphing box.

By using a simple script it is possible to centre the CoG of this auxiliary ellipsoid with the CoG of the gearbox automatically and to scale its size to a user given value. Additionally, the mesh size of the ellipsoid can be reconstructed to a user given size.

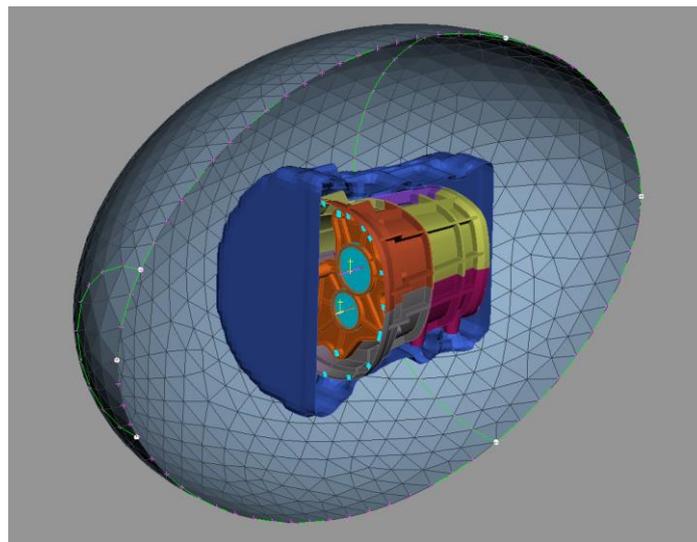


Figure 7 – The ellipsoidal envelope adapted and reconstructed in order to fit the analysis requirements.

However for the purpose of this work, the first approach by using a convex envelop will be followed. After the creation of the inner and outer boundary of the acoustic mesh, it is easy to create the volume definition and mesh with solid elements between those two boundaries.

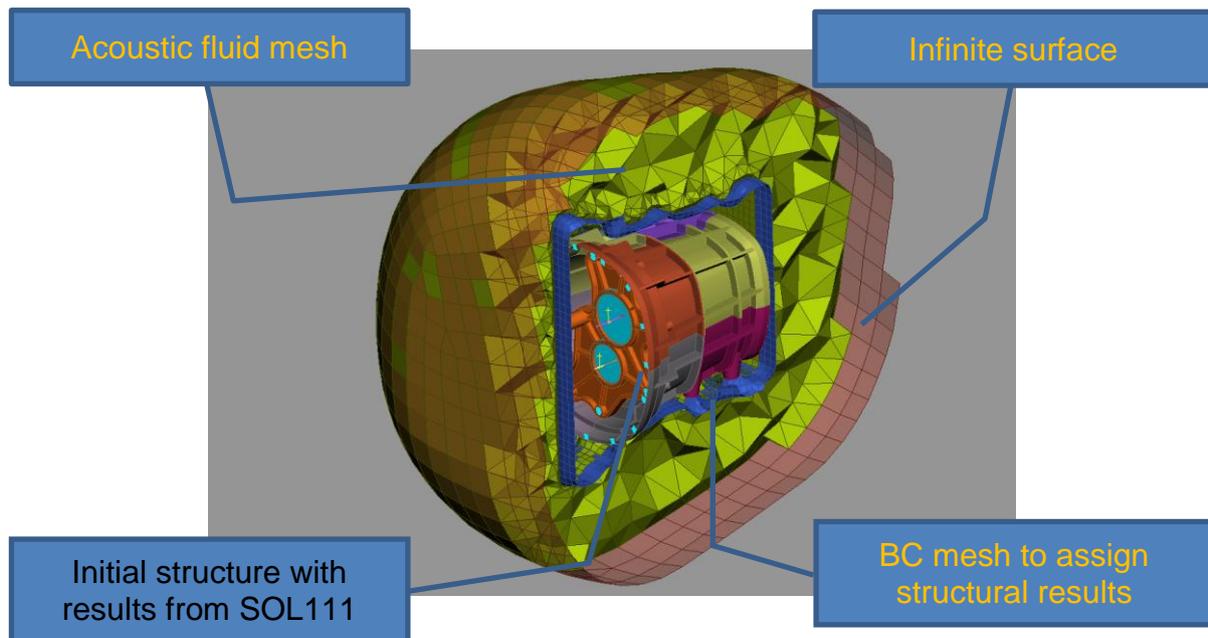


Figure 8 – The acoustic fluid mesh, created by means of the infinite surface and the wrapped surface, as “BC mesh”.

In the figure 8 all the main domains, which are mandatory and need to be assigned to RADACT for the exterior noise analysis, are represented in a clear manner.

### 3. THE RADACT INTERFACE

An interface embedded in ANSA was developed that supports not only the creation of the above meshes, but also the setup of the whole analysis file and exports it to the RADACT format, which can be directly assigned to the ACTRAN solver.

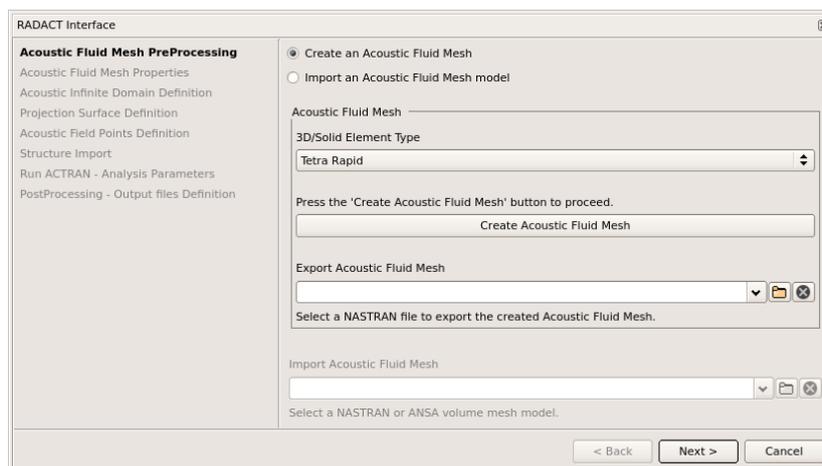


Figure 9 – 1<sup>st</sup> step: Pre-processing for the acoustic fluid mesh creation

The interface comes as a wizard, which will create the ready to run RADACT file in eight steps. In the first step, the user has to decide whether to create the acoustic fluid mesh, with the respective infinite and wrapped surfaces, from scratch or to import it as an external file. The option “Create an Acoustic Fluid mesh” will generate the above mentioned mandatory three domains in a fully automatic way, which is still under development and will be issued as a next step.

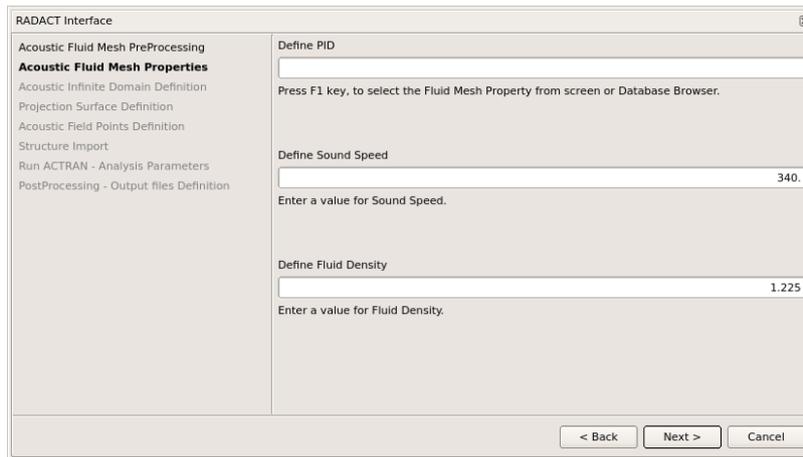


Figure 9 – 2nd step: Assignment of the acoustic fluid mesh properties

Once the model with the three domains is there, then, in the 2<sup>nd</sup> step, the user has to select the respective property for the fluid mesh and to assign the respective physical values for the fluid medium as sound speed and density.

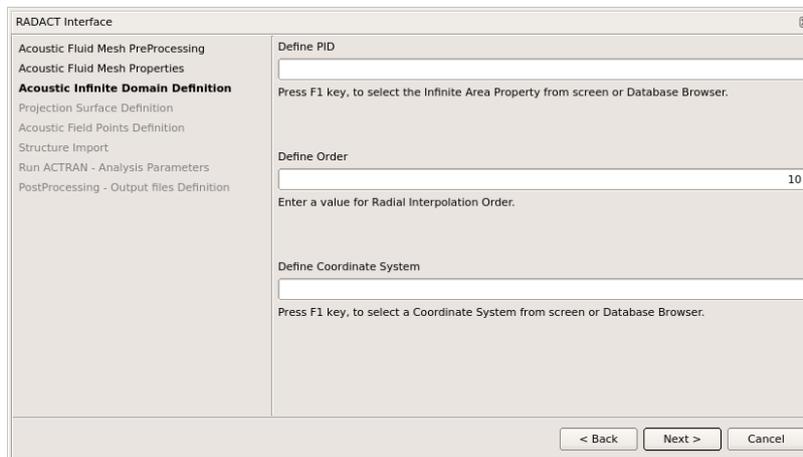


Figure 10 – 3rd step: Assignment of the infinite domain properties

In the 3<sup>rd</sup> step the user has to select which properties will be assigned as infinite domain. This infinite domain represents the free field condition, where infinite elements will be placed within ACTRAN. Additionally, the radial interpolation order within the infinite elements and its local coordinate system are mandatory to be given [1].

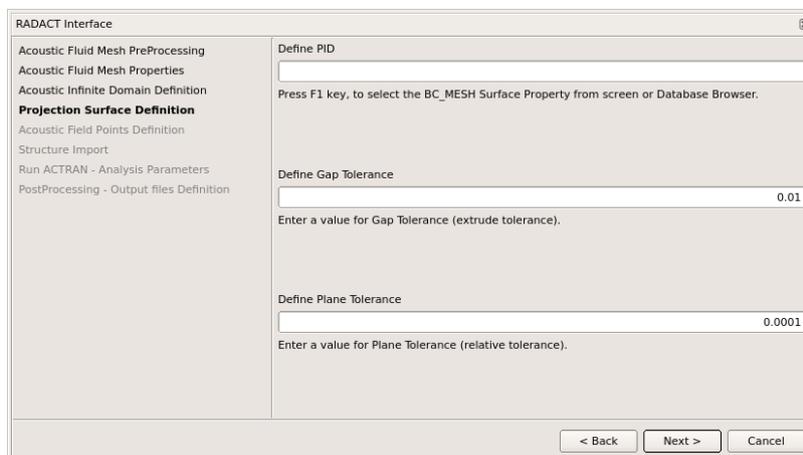


Figure 11 – 4th step: Assignment of the projection surface properties

The acoustic boundary conditions, which are provided by the structure and which hold the frequency response results, are often not part of the computational grids for the acoustic domain. For this reason, the “Projection Surface” will be used, where the structure results will be mapped on as boundary conditions [1]. The smooth wrap mesh will serve as “Projection Surface”, property of which has to be selected here. The input of the Gap and Plane tolerance are optional, as they control the grids to be considered during the projection of the results.

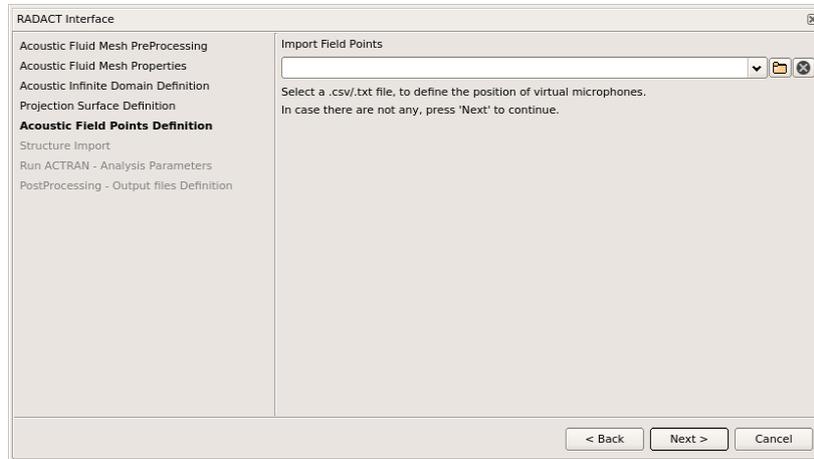


Figure 12 – 5th step: Import the Acoustic Field Points

The acoustic field points, which represent virtual microphones and where the sound pressure levels will be calculated, can be imported as an external file in one of the common .csv or .txt format.

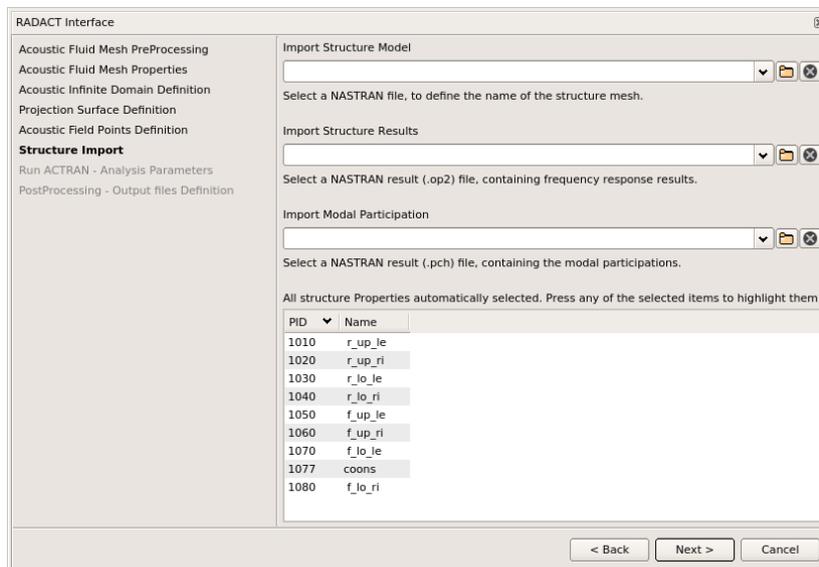


Figure 13 – 6th step: Import of the structure file and its vibrational results as boundary conditions

In the 6<sup>th</sup> step, the assignment of the structure and its results will be defined, based on which the acoustic radiation shall be calculated. The vibrational results file contains the frequency response results at various frequencies. Optionally, the modal participation results, given in .pch file format, can be used, from which the ACTRAN solver recombines the modes for the definition of the boundary conditions [1].

In the lower field all structure properties of the loaded model are listed. This list serves as visibility control in order to highlight the selected properties.

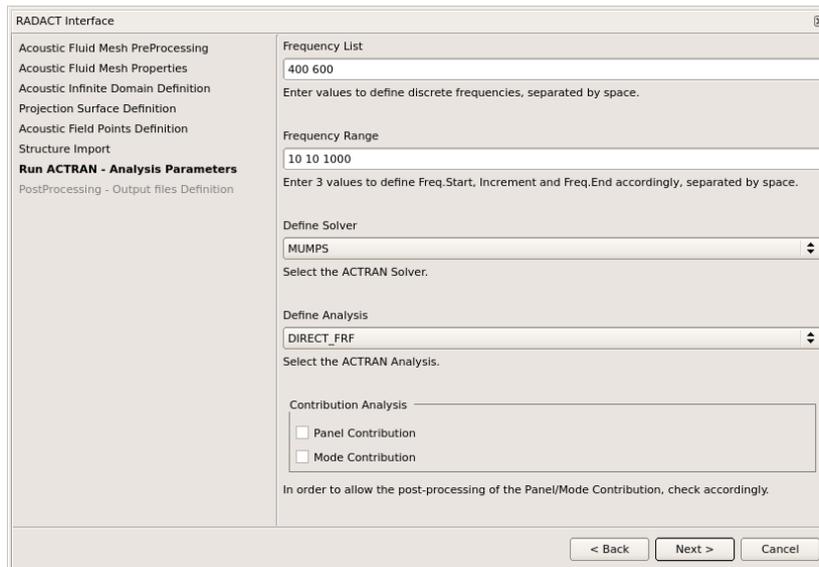


Figure 14 – 7th step: Setup of the analysis parameters for the ACTRAN solver

Prior the completion of the RADACT input file and its export, some analysis parameters need to be defined, such as the frequencies to be analysed, the solver and analysis type. Moreover, optionally it can be defined whether a panel or mode contribution analysis shall be performed.

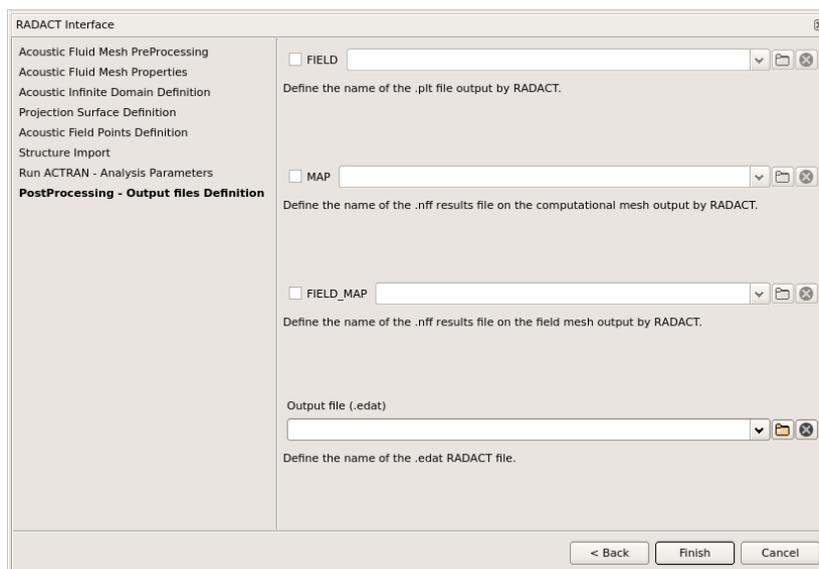


Figure 15 – 8th step: Setup of the analysis parameters for the ACTRAN solver

During the last step of the wizard, besides the definition of the output location of the RADACT input file, some post-processing options can also be determined. The Frequency Response Functions (FRF) can be plotted in the .pls file. The sound pressure results can be plotted at the computational mesh (as created in step 1) and at the field map mesh, which is used only for the visualisation of the results in fringe mode.

Finishing the wizard, a RADACT input is created, which can be directly assigned to the ACTRAN solver for the analysis. The structure of that file can be reviewed below:

```

1 BEGIN RADACT
2
3 COMMENT_START
4 ActranVI output file
5 VERSION : 12.0
6 COMMENT_STOP
7
8 BEGIN ACOUSTIC_MODEL_IMPORT
9
10 BEGIN FINITE_DOMAIN
11 NASTRAN /mnt/raid_disk/titanas/chryssa/ACTRAN/gearbox_full_create.ansa
12 FID 1075
13 SOUND_SPEED 2.2
14 FLUID_DENSITY 2.3
15 END FINITE_DOMAIN
16
17 BEGIN INFINITE_DOMAIN
18 FID 1076
19 ORDER 10
20 COORDINATE_SYSTEM
21 0.00000 0.00000 0.00000
22 4.852162 89.37847 14.41416
23 4.782822 89.36738 19.28553
24 2.296335 89.82349 14.50533
25 END INFINITE_DOMAIN
26
27 BEGIN PROJECTION_SURFACE
28 FID 1076
29 GAP_TOL 10
30 PLANE_TOL 5
31 END PROJECTION_SURFACE
32
33 END ACOUSTIC_MODEL_IMPORT
34
35 BEGIN STRUCTURE_MODEL_IMPORT
36 NASTRAN /mnt/raid_disk/titanas/chryssa/ACTRAN/SOL111_Gearbox_EVO2_ERP.nas
37 NASTRAN_BC_FILE /mnt/raid_disk/titanas/chryssa/ACTRAN/SOL111_Gearbox_EVO2_ERP.op2
38 NASTRAN_MODAL_PARTICIPATION /mnt/raid_disk/titanas/chryssa/ACTRAN/SOL111_Gearbox_EVO2_ERP.pch
39 FID 1020
40 END STRUCTURE_MODEL_IMPORT
41
42 BEGIN RUN_ACTRAN
43 FREQUENCY_LIST 400 600
44 FREQUENCY_RANGE 1 10 1000
45 SOLVER KRIVLOV
46 Model Resp_ANALYSIS
47 PFI_FILE /home/chryssa/test.plt
48 MAP_FILE /home/chryssa/test.nff
49 FIELD_MAP_FILE /home/chryssa/test.nff
50 PANEL_CONTRIBUTION 1
51 MESH_CONTRIBUTION 1

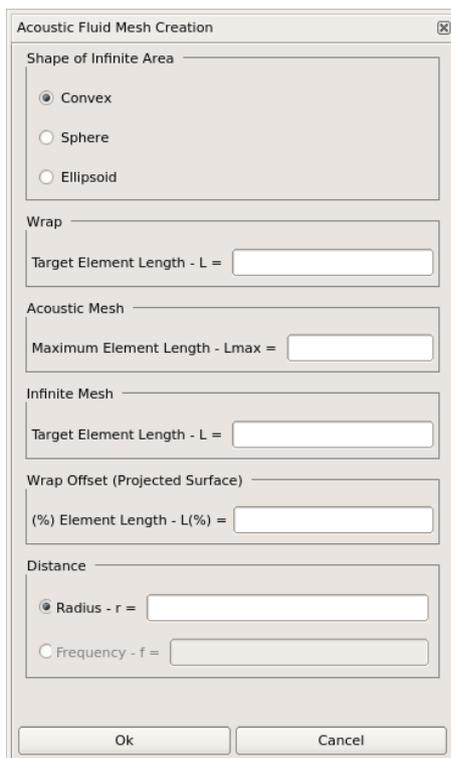
```

Figure 16 – Created RADACT input file format

#### 4. CONCLUSIONS

An interface has been developed, which supports the RADACT user to create an analysis file with a very convenient, efficient and thus robust method. Its main advantages are that it can be applied within an environment which is common for a user who is about to prepare the model for an acoustic radiation analysis. The wizard functionality supports also inexperienced users with ACTRAN, to easily set up an exterior noise analysis.

As next steps, the interface will be enhanced with features that will allow “on the fly” modifications of the acoustic mesh model, without having to re-import it again.



Moreover, the method to create the acoustic model mesh will be provided fully in a fully automation mode. The user will have the options to choose one of the three methods to create the infinite surface.

First, the method will be chosen: Convex, Sphere, or Ellipsoid. Then, a target element length of the projected surface (BC\_mesh) will be assigned.

The element length of the solids and that of the infinite surface will be defined. Optionally, an offset on the wrap surface can be performed.

The Radius (r) or the Frequency (f) will be used to define the size of the outer boundary, the infinite surface.

#### REFERENCES

- (1) ACTRAN 12 User’s Guide