

ANSA and META for exterior noise analyses with ACTRAN

Introduction

In the field of exterior noise analysis resides ACTRAN, an acoustic radiation modeller within the MSC software suite, which is widely used in the automotive, aircraft, aerospace and audio industries. As in many other CAE disciplines, ANSA's capabilities can considerably facilitate the model build-up and mesh creation processes for analyses in ACTRAN.

One of the main features of ANSA, that the engineer can benefit from when preparing a model for an ACTRAN analysis, is the various available methods for the creation of the exterior cavity - the so called acoustic mesh- according to the analysis requirements. The model exchange from ANSA to ACTRAN is well arranged through the bilateral supported NASTRAN format. The results of the ACTRAN analyses are supported by META, in 2D as well as 3D animations.

Additionally, ANSA offers an interface for RADACT (ACTRAN's sub-module) which runs in batch mode and thus, does not have any GUI.

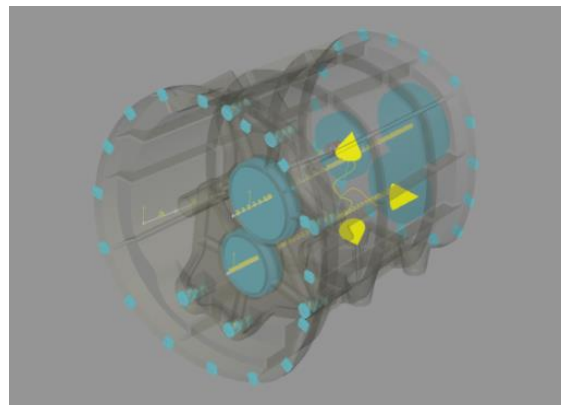
Preparing a model in ANSA for ACTRAN

As an example, a gearbox structure is prepared for an exterior noise analysis. In the first phase, the structure is prepared for a typical frequency response analysis (SOL111). These results are later imported into ACTRAN as boundary conditions. Then, the acoustic fluid mesh is prepared according to the analysis requirements.

- Model preparation for the structural Modal Response analysis

In this case, the acoustic radiation based on a tri-axial dynamic load excitation on its main shaft is analysed up to a frequency of 1 kHz.

As a first step in this exterior noise analysis with the ACTRAN solver, the displacement results on all nodes of the structure - taken from a NASTRAN SOL 111 modal response analysis (in op2 format) - are used as boundary conditions. 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 introduced in version v14.0.0.

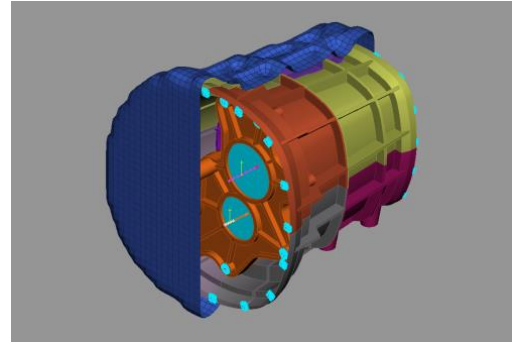




- Model preparation for the acoustic fluid mesh creation

In order to create the required mesh for the exterior noise analysis and to analyse the radiation characteristics of the structure for that load-case, the engineer can use the meshing capabilities of the ANSA pre-processor. Once the structure is ready, the next step is the creation of a wrap mesh around the gear box.

The wrap mesh will act later in the ACTRAN analysis as structure and will be identified as the so called BC_Mesh.

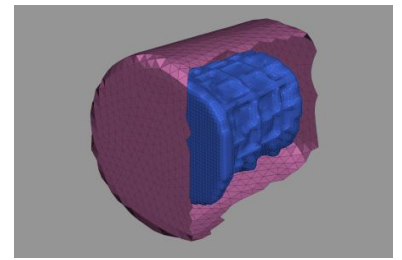
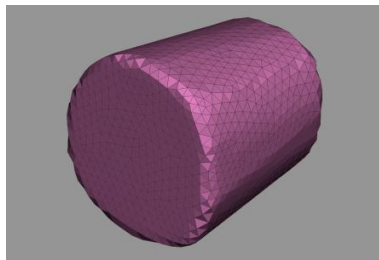
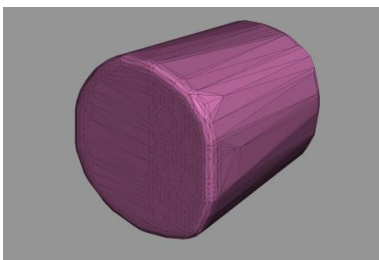


Furthermore, it represents the surface of the structure where the normal directions of the displacement results, from the modal response analysis, will act as boundary conditions in ACTRAN.

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

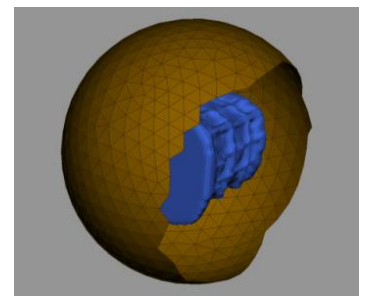
- Convex mesh creation

The first way to create the infinite area is by producing an outer envelope, based on an STL type mesh. It is possible to "smoothen" the produced mesh and to assign a user defined mesh size to it, according to parameters, such as the maximum frequency in the analysis.



- Sphere creation

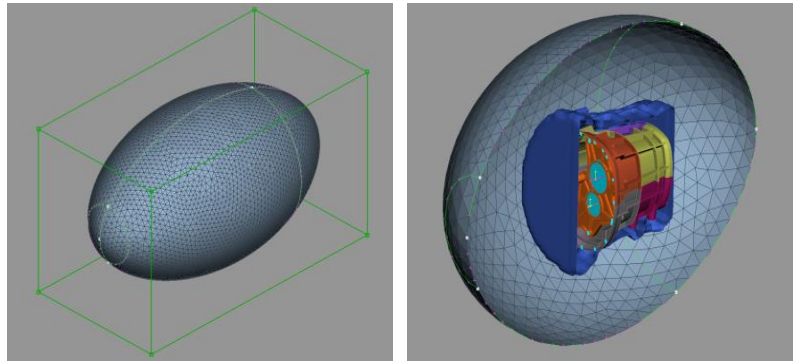
Another method is by using the build-in standard surface features of ANSA. It is possible to create a sphere surface directly, based on the COG of the wrapped surface. Then, this sphere can be meshed with the desired mesh type and size.



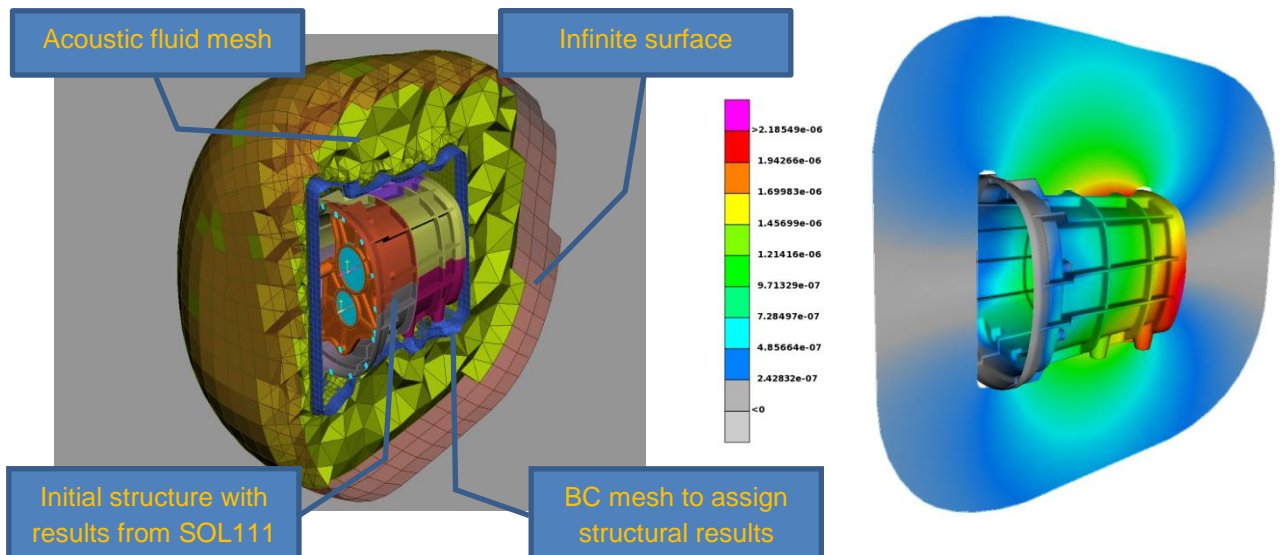


- Using an adaptive generic envelop

The third way is by using a generic envelop with an initial ellipsoid shape. This can also be combined with a script function that adapts the envelope to any user-given structure. The infinite area's size and surface mesh type can be adapted to the demands of the given structure and analysis set up.



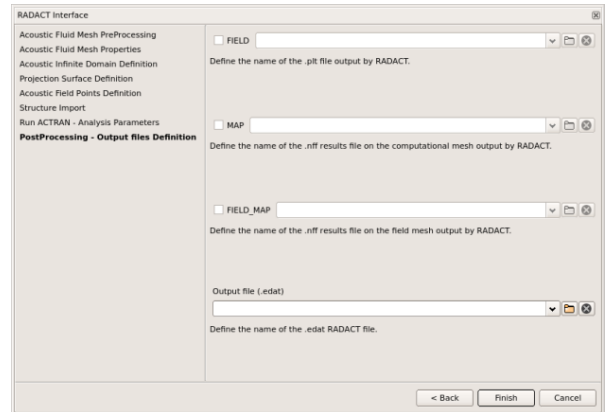
After the creation of the inner and outer boundary of the acoustic mesh, it is easy to define the volume between those two boundaries and mesh it with solid elements. Once the ACTRAN analysis has been completed, the results of the exterior noise analysis can be reviewed in the META post-processor, taking advantage of its designated post-processing capabilities.





The ANSA interface for RADACT

The RADACT interface embedded in ANSA provides the user with the opportunity to create and export a ready-to-solve RADACT input file. It also assigns the respective structural results and sets up all necessary solver and analysis parameters required. Moreover, using the selection capabilities embedded in ANSA, it is now very easy to assign the particular model entries to the respective acoustic properties for the exterior noise analysis.



Conclusions

Through the exterior cavity modelling capabilities in ANSA the engineers can create an input file for ACTRAN in a very convenient way. The functionalities greatly facilitate the set-up of an exterior noise analysis, rendering the whole procedure easy and comprehensive for even the less experienced user. Especially the RADACT interface, designed as wizard, guides the user to the creation of a ready to run RADACT analysis file in an eminently efficient and thus robust way.

For more information contact
BETA CAE Systems
Email: ansa@beta-cae.com