ANSA AS A PRE-PROCESSOR FOR FLUID-SOLID COUPLING SIMULATIONS

Chen H.Y.*, Liu Wei, Lv X. M., Zhang L.Z.
Beijing FEAonline Engineering Co., Ltd, China

ABSTRACT: Recently, the important significance of the fluid-solid coupling is generally recognized. Due to the importance of the coupled problems, considerable effort has been devoted in many engineering applications.

The preparation of the mesh for CFD analysis and FEA analysis is complex laborious process that usually involves the combination of different software for the specific tasks. This paper presents the workflow of the fluid-solid coupling simulation with a model under complex load including thermal load， bolt load and large numbers of contact relationship. The workflow involves mesh generation with ANSA, CFD analysis with CFX, FEA Non-linear analysis with ABAQUS, CFX data interfacing with ABAQUS by SimuFSI (in-house software of FEAonline).

KEYWORDS: fluid-solid coupling, mesh generation, Non-linear analysis, Thermal-structural coupling, FEA, CFD

1. Background

Many structures are exposured to fluid pressure and temperature field, such as high-pressured cylinder, low-pressured cylinder, and turbine rotor. These structures show in Figue1 contains various components which fastened together by bolts or connected each other by complex contact. The fluid pressure and the thermal load are the primary loads. High stress may occur due to fluid pressure. Bolt connecting surface separation may lead to leak tightness problem under the thermal load.

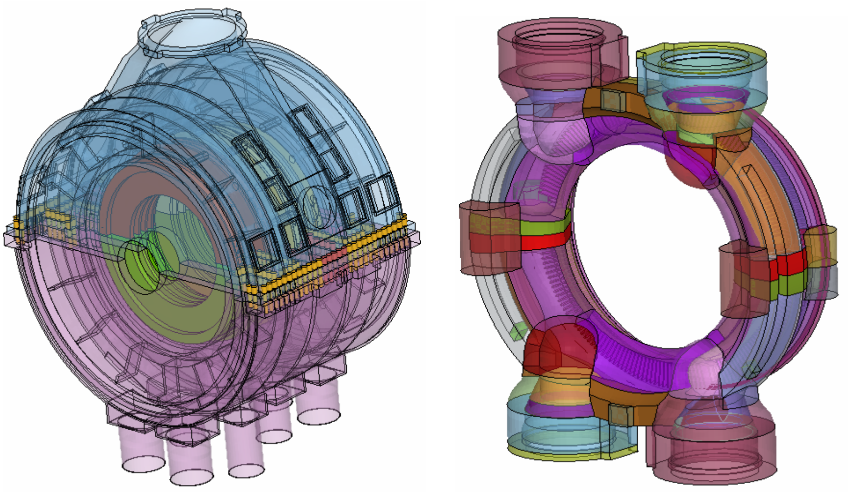


Figure 1 CAD model

In the traditional FEA analysis, uniform pressure and convection heat transfer coefficient were assumed on the multiple surface of the structure under the premise of splitting multi-surface and rough knowledge of the wall flow parameters. The traditional method was known to be gross approximate and time-consuming. To conduct a high-fidelity simulation, FSI model is employed. ABAQUS simulates the displacement and stress of the model, while CFX calculate the pressure and temperature. SimuFSI is used in our effort to transfer the CFD data to FEA data. To reduce the time-cycle, ANSA is adopted as a pre-processor for fluid-solid coupling.

2. FSI modeling approach

For many industrial applications, such as flow in turbine machinery, high temperature vessel, and motion of valves, FSI is sequentially coupled. The governing equation for solid and fluids are solved separately. The interaction between the two solution domains is realized by the interface between the solid and the fluid. The structural solution provides the displacements at the interface surface, and the flow solution provides the pressure and heat flux loading at the interface surface. Different mesh types are applied for the model, consisting of the solid structure on one side and the fluid on its other side. The structure mesh geometry is modeled by mixed tetrahedral and hexagonal langrangian finite elements; the fluid domain is modeled by eulerian finite-volume cells.

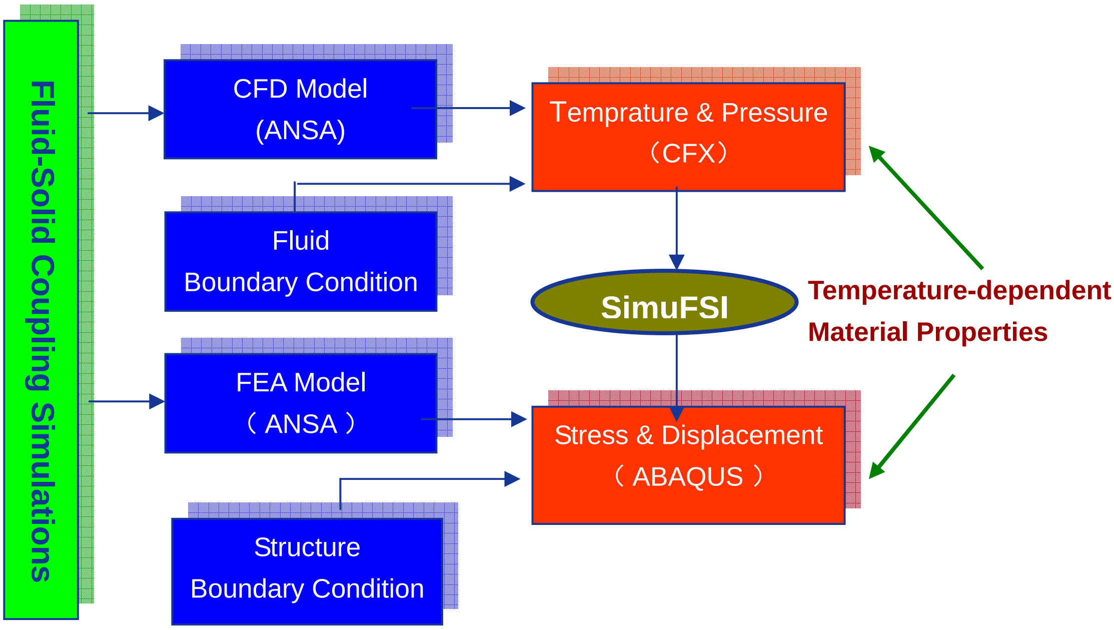


Figure 2 the general flow chart of FSI

Take the Fluid-solid coupling simulation of the low-pressured cylinder as an example, following the generic procedure for FSI simulation (Figure 2), detailed steps are as follows:

1. ANSA is used to build high quality FEA Model for inner casing & blade rings and create turbulent flow model for the fluid domain.
2. Setting of boundary conditions, materials, bolt load and contact among all components are undertake by ANSA
3. CFX solves the fluid domain to get the pressure, heat flux loading, wall temperature. Based on the heat flex loading and wall temperature, CFX solve the solid domain of the low-pressure cylinder to obtain the temperature of the whole model (Figure 3). No heat transfer step in ABAQUS shall be required for the thermal-structural coupling, so you can use only one *.inp file to complete the simulation with high efficiency
4. Then SimuFSI transfer the temperature from CFX software to ABAQUS software as the predefined field and transfer the pressure as the distributed load on the surface.
5. Based on an overall consideration of steam pressure, steam temperature, bolt load and contact load, 3D thermoelastic contact model on cylinder pressure tight will be solved by ABAQUS to get the S.tersca of the whole model, Cpress& Copen. of the cylinder mid-separate surface.

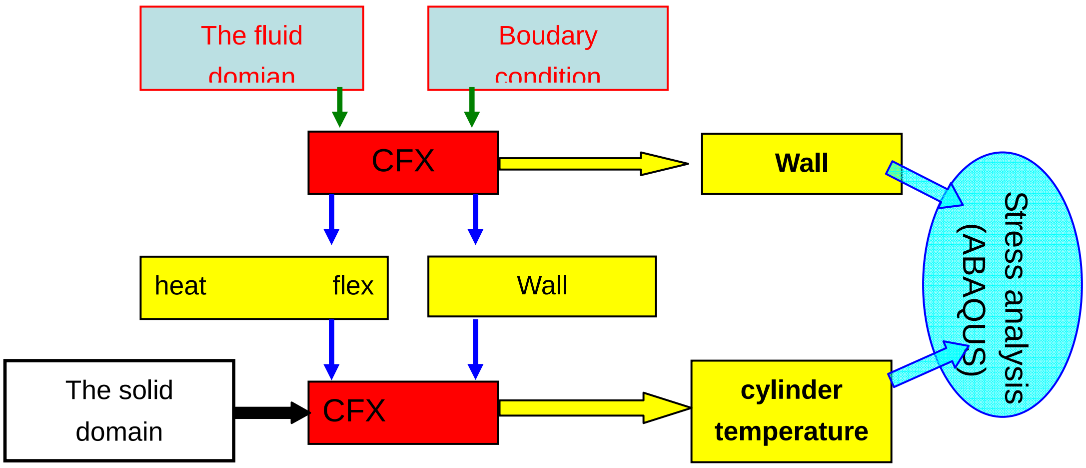


Figure 3 the workflow of obtaining wall pressure and cylinder temperature

3. Pre-processing with ANSA

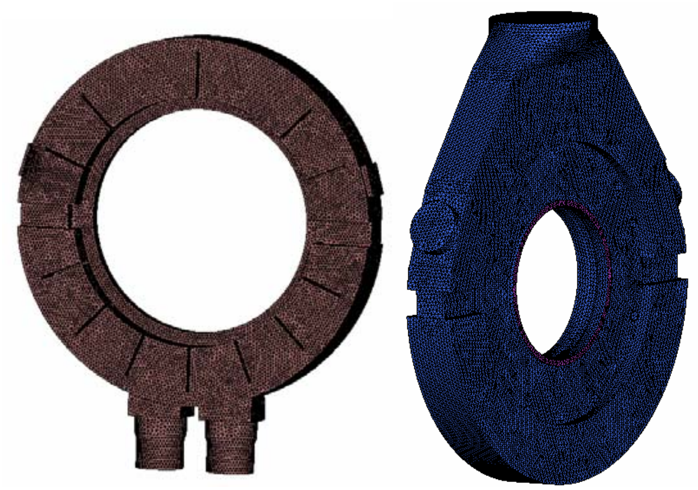


Figure 4 Partial fluid domains of the low-pressured cylinder

As a very flexible and effective tool for preprocessing, especially for complex geometry, ANSA supports output of the model to ABAQUS format and CFX format. Sets, boundary conditions, complex contact and material can be set in DECK>ABAQUS.

According to the specific characteristics of the model, as well as the characteristics of air flow path, four fluid domains are modeled (Figure 4). In order to satisfy the forced condition of the inlet and the outlet, the inlet domain and outlet domain were extended to avoid back flow of the inlet and the outlet during the CFD simulation(forced flow can only flow out at the outlet, otherwise disobeys forced boundary condition which will possibly affect convergence)

There are more than 200 complex contact relationships. Many parts of the low-pressured cylinder are fixed by flex contact relationships, such as bolt, and blade rings. So high-quality hexahedral mesh is required for non-linear analysis as shown in figure 5. To create a Hex mesh of high quality, you must decompose the solid part into sub-volume blocks that can be meshed by the Volume MAP algorithm. Element around transition areas like fillet and bolt holes were modeled to sufficiently capture the stress concentration. There is one washer around the bolt hole. The mesh must be generated with the principle that: 1.) It models the topology accurately. 2) Usage of tetras should be avoided around the contact region. 3) There are enough elements in the region of mid-separate surface. Alignment of node of slave and master surface is appreciated. 4) The bolt should preferably be modeled as solid mesh.

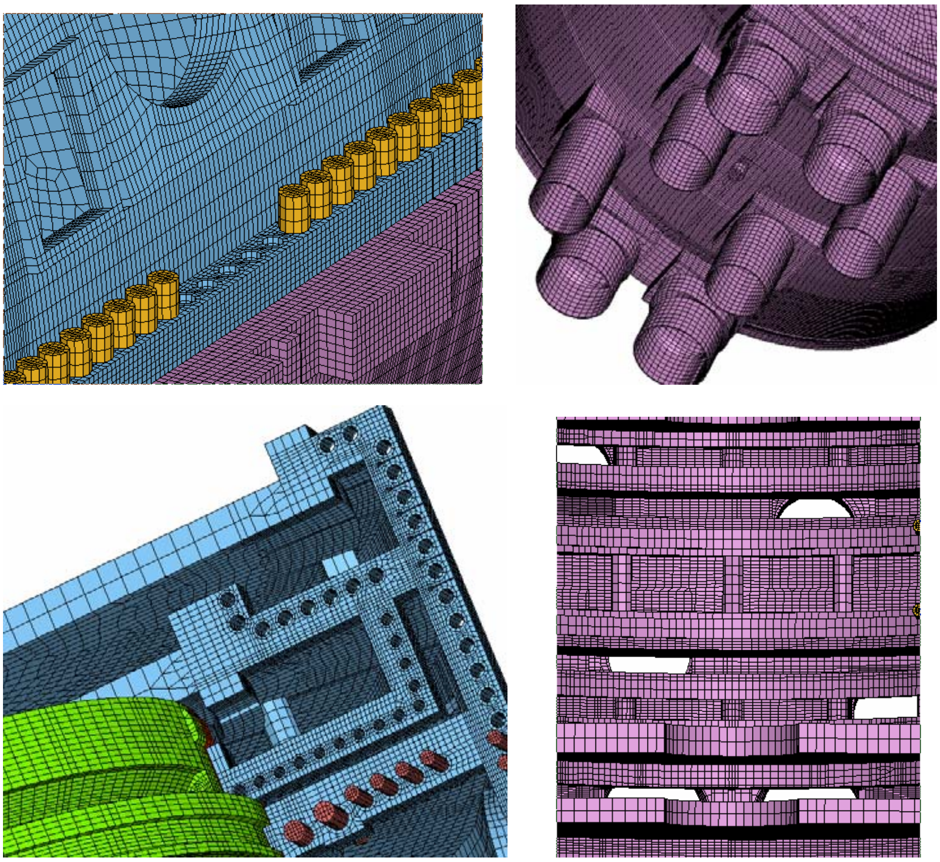


Figure 5 the mesh of the low-pressured cylinder

The total element number is 823,731. Table 1 lists the proportion of various types of element. Table 2 lists the quality of the mesh.

Table 1 Mesh Characteristics of the low-pressured cylinder

Element type	number	proportion（%）
Total NO. of the element	823,731	100
hexas	837,531	96.9331
pentas	25686	3.0668
tetras	0	0

Table 2 the qualities of the mesh

Quality criterion	class	proportion（%）
Aspect Ratio	1~5 （good）	88.7281
	5~10 （medium）	11.1719

	10~16 （bad）	0
Solid Warping	0~6 （good）	94.49
	6~12 （medium）	3.60
	12~30 （bad）	1.91
Solid Jacobian	0.35~0.5（good）	0.015
	0.5~0.6 （medium）	3.22
	0.6~1 （bad）	96.7885

4. FSI software——SimuFSI

The main function of the SimuFSI is as follows:

1. Transforming the result from commercial CFD software to the correspondence node of the FEA model.

2. Given one user-defined grid & the variable values corresponding to the nodes and the second grid, the corresponding value of the second grid can be interpolated. (The input file format of the user-defined grid is shown in Figure 6),

Figure 6 the input file format of the user-defined grid

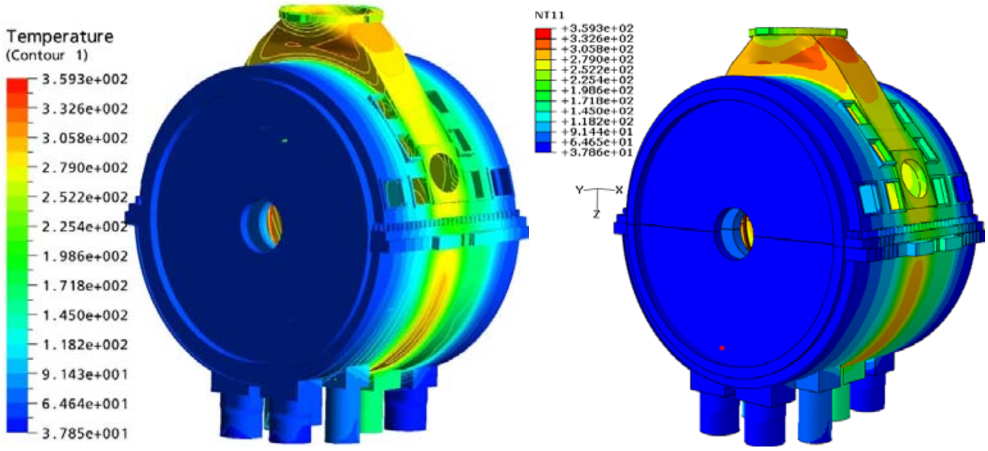
Figure 7 the main interface of the SimuFSI

The main interface of the software is shown in Figure 7, take "CFX to ABAQUS" as an example, the detailed steps are as follows:

1. Choice of commercial software, after the choice of interpolation type, set the interpolation parameters
2. ✓CFD software: CFX
   ✓FEA software: ABAQUS
   ✓Variable type: temperature, pressure, convection heat transfer coefficient
   ✓result file of the CFD simulation
   ✓Node coordinate file of the FEA model
   ✓The name of the output file
   ✓Check the output file

5. Results

In order to verify the accuracy of the SimuFSI, comparison with the temperature field obtained by FEA and that obtained by FSI is done in this paper. Temperature field from the two methods is fully consistent as shown in figure 8.



(a) temperature obtained by CFX (b)temperature transferred into ABAQUSby SimuFSI

Figure 8 temperature comparison

Figure 8 lists the pressure of fluid domains. With the sudden change of the geometry, and the area reduction, flow expansion occurs at the outlet, the pressure decreases, the velocity of flow increases.

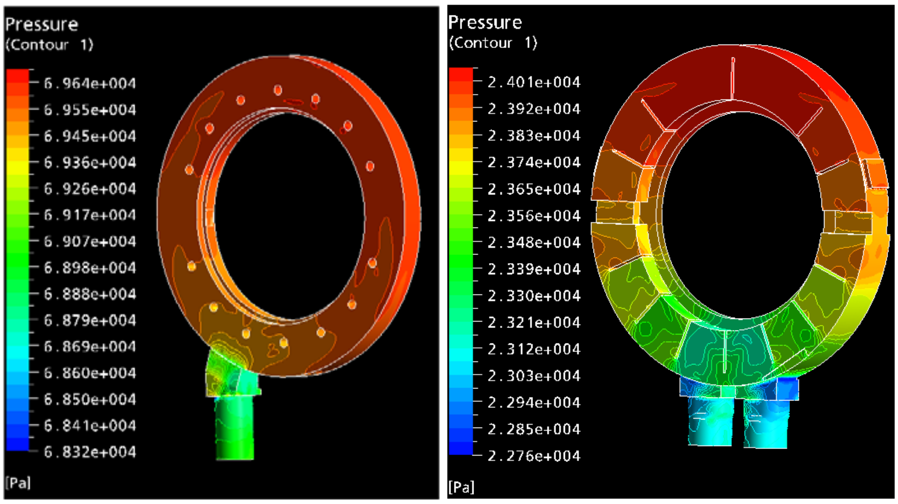


Figure 8 pressure of the wall

Stress plots (Tersca stress) for this analysis are shown in figures31. High stress areas are observed on cylinder near the inlet and support device. High stress mainly caused by temperature load.

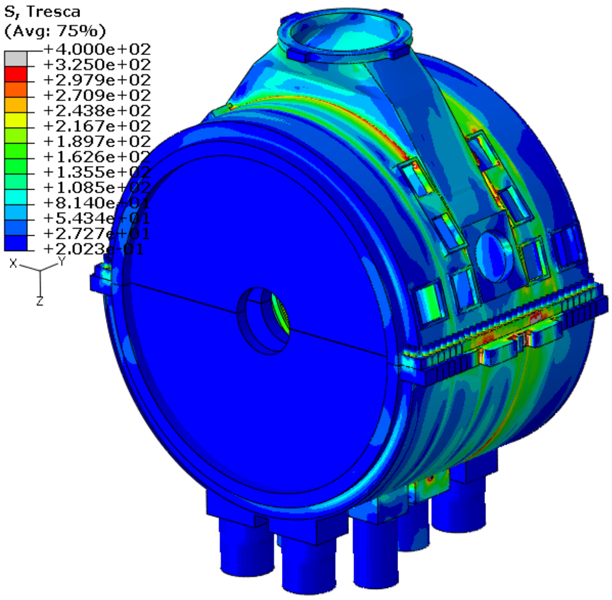


Figure 10 Stress Contour of the whole model

Results for contact stress & contact displacement are also studied. Lack of space forbids further treatment of the topic here.

6. Conclusion

The applied methodology involves a coupling between the FEA code ABAQUS and the CFD finite-volume code CFX, based on SimuFSI between the both codes. This process is a convenient solution for current industries in various engineering sectors where reduction of time-cycle and improved cost efficiency.