CRYOGENIC INDUSTRIES
FROSTBYTE
A NEWSLETTER FROM CRYOGENIC INDUSTRIES SUMMER 2014

Application of FSI to Design and Analysis

Engineering simulation software packages continue to evolve, incorporating ever expanding feature sets and capabilities. These programs are used by design and analysis groups within engineering firms for everything from printed circuit board (PCB) cooling optimization to complex phenomena such as aircraft wing flutter. The latter is accomplished using a recent and highly useful capability of simulation software called "Fluid-Structure Interaction," or, FSI. This FSI capability allows engineers to model the effects of fluid and mechanical loads simultaneously, along with the coupled effects of one type of load upon the other.

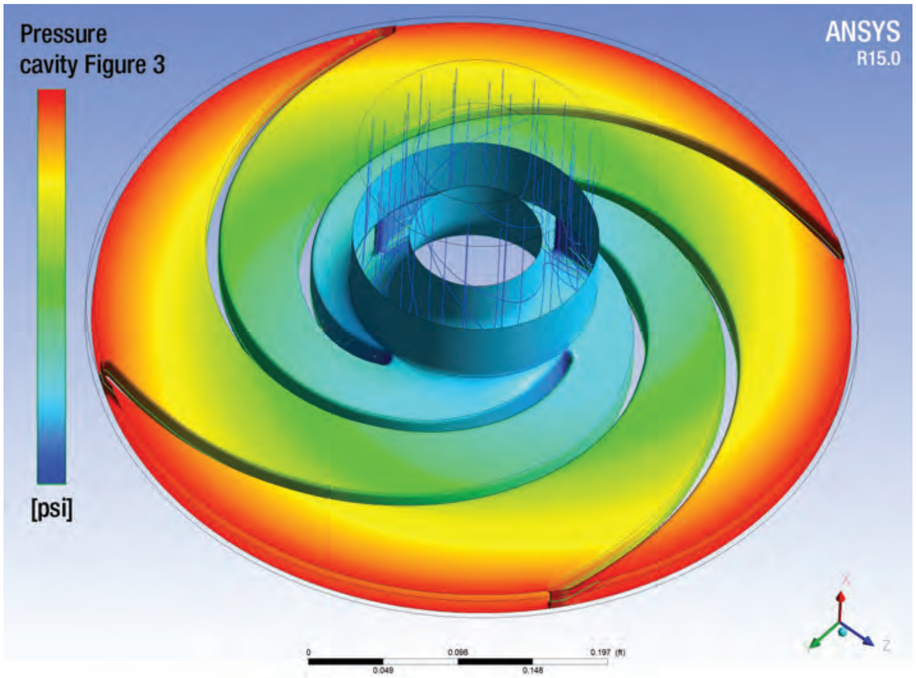

Figure 1: Impeller Internal Pressure Profile

In the wing flutter case, loads calculated by computational fluid dynamics (CFD) modeling of time-dependent pressures around a wing are applied to a mechanical model of the wing, and structural analysis is conducted for each time step of the CFD analysis to determine the stresses and deflections. This is called One-Way FSI. Deflections may also be fed back to CFD to modify the boundary walls in the fluid domain, allowing the mechanical model to influence the fluid flow. This is called Two-Way FSI. Two-Way FSI is typically only necessary when structural deflections are great enough to have a significant effect on the fluid flow.

ACD two years ago expanded its engineering analysis toolset with the introduction of ANSYS simulation software to its engineering workflow. A study using ANSYS FSI capabilities has recently been completed for an existing, legacy pump impeller design used in ACD's TC-50 and TC-30FC two-stage pumps. Stresses due only to centrifugal loading were calculated first, as a baseline case. A one-way FSI analysis was then run, applying CFD-calculated pressure loads (Figure 1) to the impeller mechanical model both within the impeller flow passages, and outside of the impeller in the pump cavity.

A 3D CAD model of the solid impeller was imported to ANSYS. The solid model itself was imported to the Mechanical solver for finite element analysis (FEA), and a fluid domain was generated from the impeller model with matched faces in order to accurately match pressure loads calculated in CFD to the solid model. Flow simulation was completed using ANSYS CFX with a simplified temperature and pressure-dependent fluid model of liquid argon based on National Institute of Standards & Technology (NIST) physical property data. This fluid model was sufficiently accurate for the pressure and temperature range expected in the simulation. Bounding flow conditions (maximum pressure) were applied to find the maximum

continued on page 2

from page 1

expected stresses, and these were applied to a rotating domain turning at the maximum speed the impeller will spin in operation. Flow external to the impeller in the pump cavity was modeled as an axisymmetric wedge of the full 360 degree fluid space to allow rapid, accurate calculation of flow rate through the impeller labyrinth seals with a fine mesh that can correctly resolve the boundary layer (Figure 2). An accurate calculation of flow rate through the seals is required for the correct pressure profile to develop outside of the impeller.

Results from the FSI analysis (Figure 3) indicated that the pressure loading from the fluid flow is the dominant source of stress in the impeller. Additionally, the CFD simulation of the internal flow through the impeller captures the pressure differential between the suction and pressure surfaces of the blading (the blade loading), seen in Figure 4. This greatly improves the accuracy of stresses calculated at the blade roots. With FSI, this study showed that the maximum equivalent stresses in the impeller body are acceptably below the yield strength of the material, and consequently validated the original design of the impeller. Efficient use of computing resources with reasonable assumptions for the boundary conditions and optimized meshing allows the completion time for this type of study to be reduced below two weeks. FSI can then be more easily accommodated in the design workflow for future R&D.

For additional information go to www.acdllc.com.

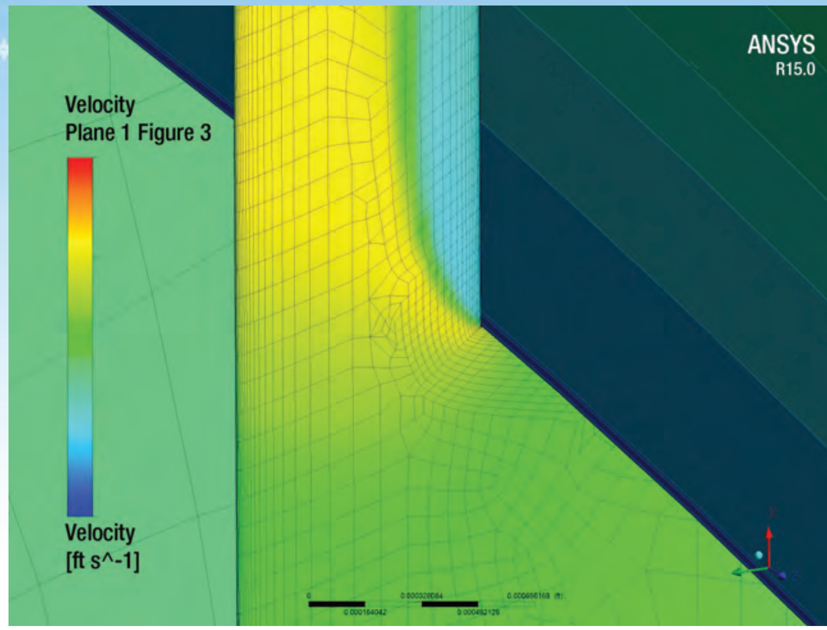

Figure 2: Boundary Layer at Impeller Laby Seal Entrance

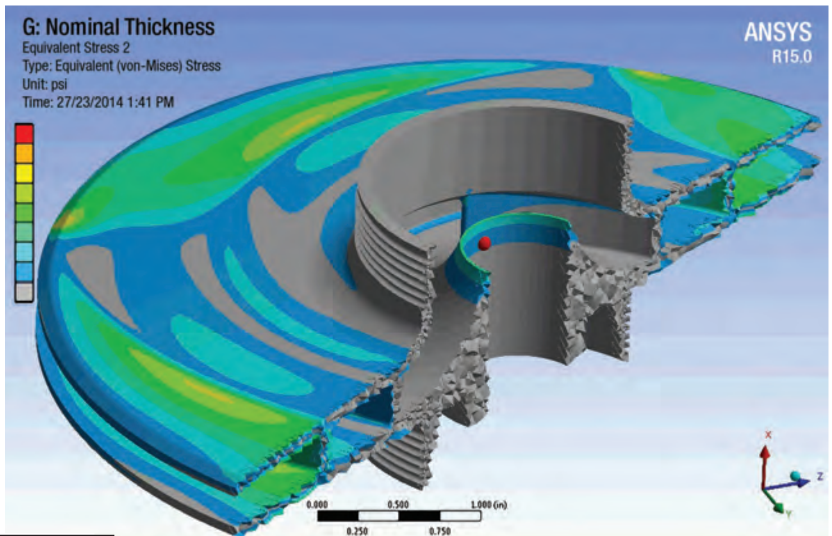

Figure 3: FSI Stresses - Combined Centrifugal and Pressure Load

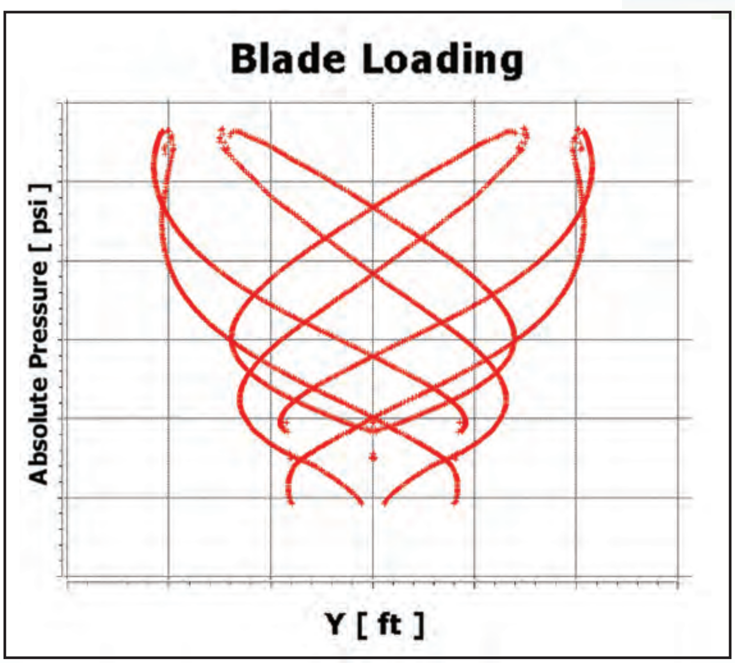

Figure 4: Blade Loading of Pump Impeller