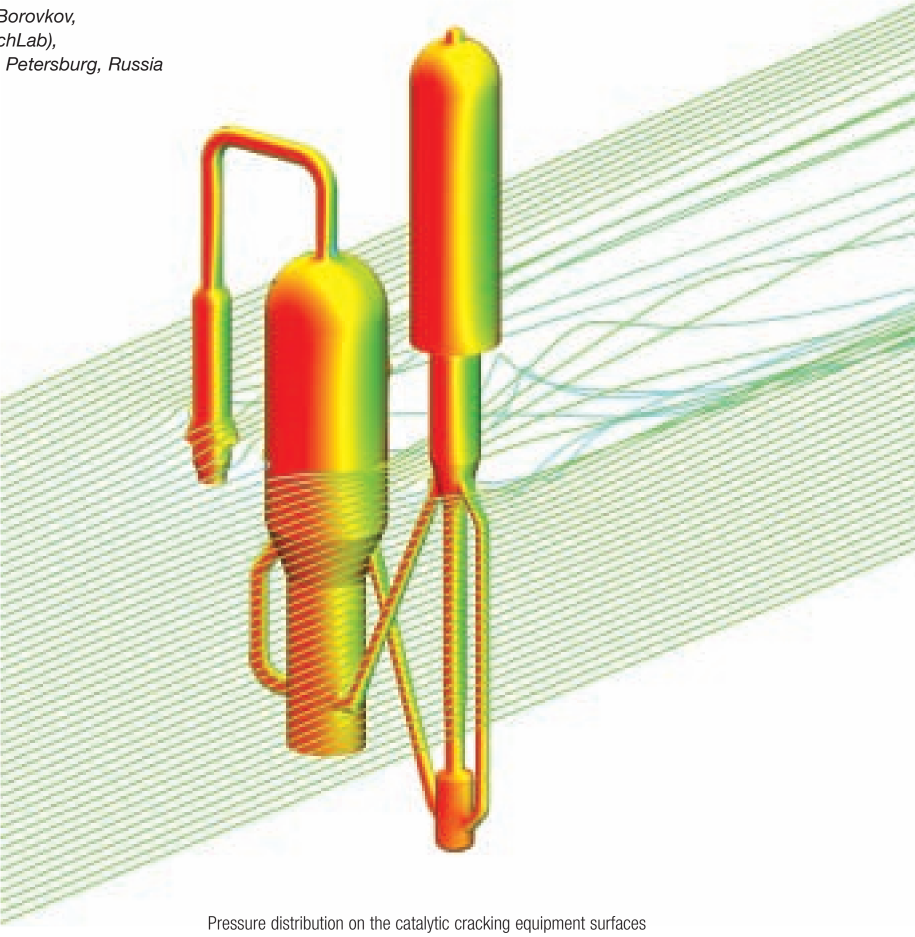
Designing **Safe Crackers**

Fluid and structural simulation combine to help researchers analyze a variety of stresses on catalytic cracking equipment.

By Alexander Michailov, Igor Voinov and Alexey Borovkov,
Computational Mechanics Laboratory (CompMechLab),
St. Petersburg State Polytechnical University, St. Petersburg, Russia



Pressure distribution on the catalytic cracking equipment surfaces due to the external air flow (represented by the streamlines)

In the petrochemical industry, catalytic cracking is one of the major steps in the process of splitting large hydrocarbon molecules into smaller, more useful components for gasoline and jet fuel. The cracking system itself consists of a reactor and a regenerator that are interconnected by a catalyst pipeline network. During the cracking process, the system undergoes mechanical loading — from wind, internal pressure buildup and the weight of the catalyst material — and experiences thermal stresses caused by the repeated temperature changes to the system's walls.

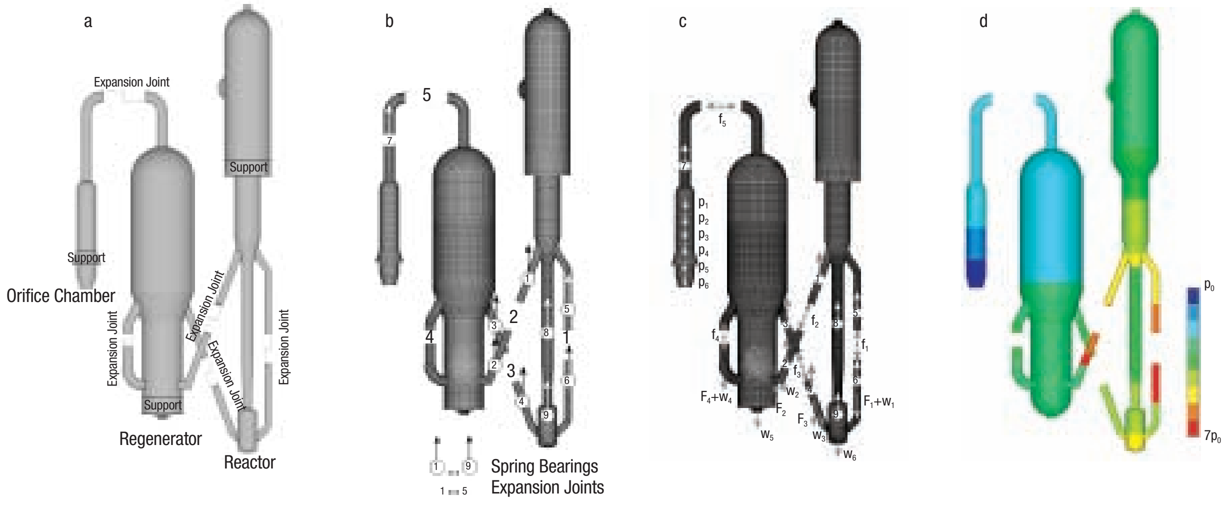
At the request of chemical equipment manufacturer JSC Neftehimproekt, the Computational Mechanics Laboratory (CompMechLab) at St. Petersburg State Polytechnical University in Russia performed a 3-D structural analysis of a catalytic cracker, taking into account the effects of external fluid flow as well as overall mechanical and thermal stresses. In their evaluation of how these effects would impact the cracking system, the CompMechLab engineers chose software from ANSYS for the selection of the cracker construction and materials.

A primary goal of the simulation was to choose wall thickness values for the reactor, regenerator and pipeline connections, taking into consideration the physical effects on the system structure at all operating conditions. By extension, this would allow creation of a list of requirements for third-party suppliers of structural components.

Within ANSYS Mechanical software, CompMechLab used multi-layer shell elements, including SHELL131 to perform thermal analysis and SHELL181 for structural analysis. The reactor and regenerator walls consisted of two layers: an external layer of steel and an internal concrete lining. Applying these shell elements also led to a reduced number of degrees of freedom, which saved computational resources.

The weight of the catalyst pipeline network is about half as much as the reactor and regenerator vessels. It was very important to take these components — including connecting nose pieces, bellows expansion joints and spring bearings — into consideration. Bellows expansion joints are used in the construction to compensate for the displacement variations caused by changing temperature loads on the catalyst pipelines and to reduce structural loads on the nose pieces. These joints are deformable parts that independently function in an elastic manner when undergoing axial, lateral or rotational movement. For the global model, the CompMechLab team simulated these components using MASS21 point mass elements in appropriate locations relative to the reactor and regenerator vessels.

CompMechLab's simulation process focused on analyzing the stiffness of the bellows expansion joints and also the



(a) 3-D solid model of the cracking system, (b) location of the spring bearings and bellows expansion joints, and vector (c) and contour (d) representations of the mechanical loads that are caused by internal and external pressures, gravity and catalyst density, and that are acting on the equipment

forces on the system's spring bearings, which are used to decrease the gravitational loads acting on the nose pieces. The forces on the spring bearings can be counteracted by varying seven thickness and dimensional parameters. Each of the five bellows expansion joints also has three stiffness values to vary, giving 22 independent parameters in total. For each parameter variation, CompMechLab engineers analyzed two sets of operating conditions: the normal working conditions (temperature range 521 degrees C to 740 degrees C, maximum pressure 0.3 MPa) and the design limit conditions (temperature range 555 degrees C to 790 degrees C, maximum pressure 0.8 MPa).

By varying the 22 parameters, researchers performed a series of computations that focused on decreasing the load on the nose pieces. Included in this process was a computational fluid dynamics (CFD) analysis of the wind's impact on the system's external pressure distribution. Using ANSYS CFX software, the analysis team simulated the air flow around the cracker with the built-in shear stress transport (SST) k-ω turbulence model, which is a robust model reliable for a wide class of air flow situations. The wind-induced pressure data from ANSYS CFX output was then interpolated onto the elements of the structural model in ANSYS Mechanical software, thus adding to the load contributions from internal pressure distributions, gravity, thermal stress and forces contributed from the catalyst weight.

The global structural analysis model, which included computing stress distributions on the system and their equivalent displacement vectors, assumed linear behavior of the concrete and steel material. Using codes and standards of the Russian oil and gas industry, CompMechLab engineers determined the maximum allowable stresses for the different parts of the cracking system at particular wall thickness values, taking into consideration safety factors for different operating regimes including startup, normal working conditions and shutdown. Comparing the stresses calculated by ANSYS Mechanical software to those allowed by industry standards, the analysis team was able to validate whether a particular wall material thickness was acceptable or whether the stresses were too high, in which case another analysis iteration would be required using an increased thickness.

Following the global model analysis, CompMechLab created a more detailed submodel for thermostructural and cyclic loading analysis of the upper part of the reactor with consideration of the welded joint between the reactor casing and plenum. The team carried out this step to obtain the reinforcement ring thicknesses in the zones of the nose piece connections to the reactor, which are areas of high stress concentration. Based on these detailed analysis results, engineers selected the zone with the highest stress — known as the critical zone — and perfomed cyclic strength analysis on that zone.

Following the analysis process using software from ANSYS, the design team was able to select appropriate dimensions for all of the structural component parameters. In summary, the CompMechLab engineers utilized the simulation method developed here for analyses of two different cracking systems and, as a result, were able to perform their analyses and issue their technical report for a greatly pleased client in a period of just six weeks. ■