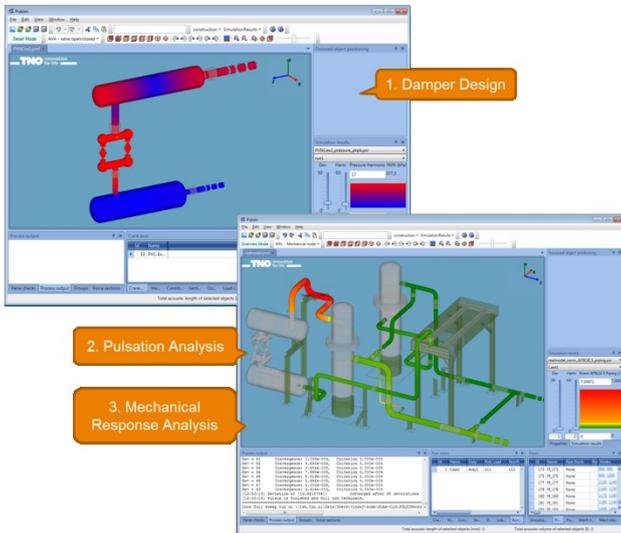# PULSIMSUITE NEWSLETTER

This is the PulsimSuite newsletter of Q3, 2016, presenting version 2.1.3 that was released in July 2016. We give you the latest developments in our API 618/674 pulsation & vibration software toolbox. You are also invited to check out our brand new website: pulsim.tno.nl.

1. Damper Design

2. Pulsation Analysis
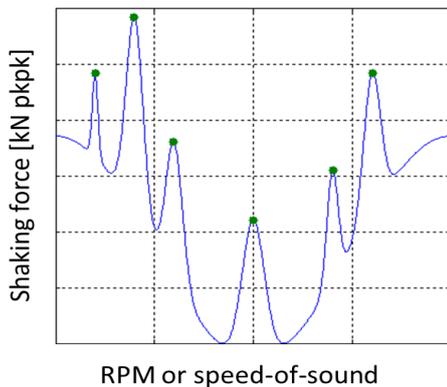
3. Mechanical Response Analysis

Introducing version 2.1.3:
PulsimSuite covers the full API 618/674 workflow. It starts with the damper design, followed by a pipe system pulsation analysis, and a full study of the mechanical response of the pipe system including supporting structures and equipment. All this is modeled, simulated, and analyzed in our unequalled PulsimSuite graphical user interface: One model is used for calculating pulsations and vibrations/stresses.

The mechanical simulations, calculating vibrations and cyclic stresses due to pulsation-induced forces, have now been fully automated: conversion to ANSYS, modal analysis, selection of worst-case shaking forces, simulating, and post-processing are all done by just one click on the "run mechanics" button.
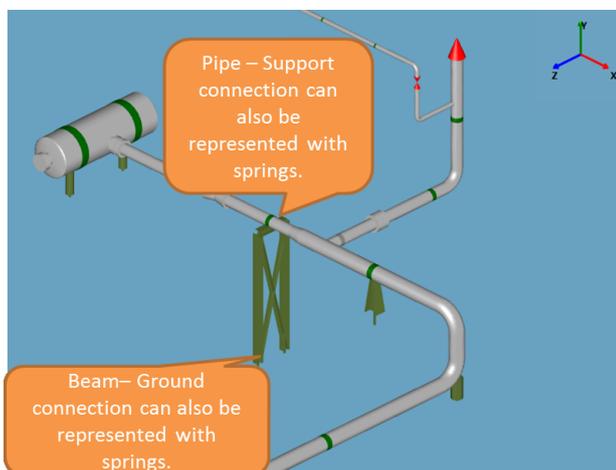
Selecting worst case conditions

Shaking force [kN pkpk]

RPM or speed-of-sound

PulsimSuite's Mechanical Module uses ANSYS on the background*. The input to the forced harmonic response analysis are the pulsation-induced forces that were calculated during the pulsation run(s).

When you press the "run mechanics" button, a smart algorithm scans the pulsation results for the operating conditions where the pulsation-induced forces are most significant. These forces are selected, and used in the mechanical response run.

The resulting vibrations and cyclic stresses are automatically compared to the API recommended levels, so that you can see the compliance check results everywhere in the pipe system immediately .

Pipe – Support connection can also be represented with springs.

Beam– Ground connection can also be represented with springs.

The functionality that converts a PulsimSuite model into an ANSYS model* has been further extended. Supports can now support pipes, valves, elbows, flanges, caps, and tees. Supports can be given 1 to 6 fixed degrees of freedom, with optionally a spring stiffness/damping.
A special element is the nozzle. As the ANSYS model generated by PulsimSuite is built up from beam/pipe elements, nozzles are represented by special vessel-pipe connections with flexibilities (springs) and Stress Intensification Factors (SIFs). These flexibilities and SIFs must be obtained from a detailed solid-element finite element model (built in ANSYS or another tool). In PulsimSuite, you can then model your nozzle straightforwardly, as a NZ point on the nozzle's pipe that connects to the vessel centerline. PulsimSuite's translator takes care of a proper conversion to the ANSYS pipe&beam model.
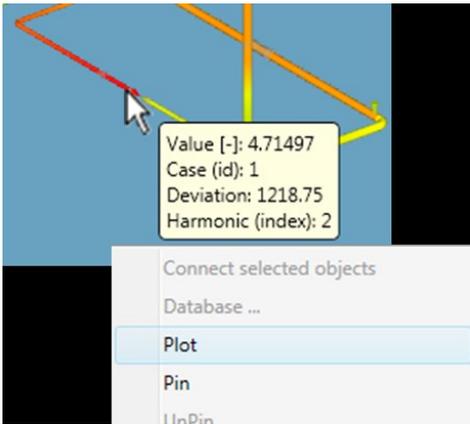
The PulsimSuite "hover-and-click" functionality makes it possible to quickly find the most relevant resonance conditions and standing waves, by a simple hover-and-click action from the view of maximum pulsation levels all over the system.

Apart from this, we have now added a new functionality to plot the result at the location where the mouse hovers over the piping
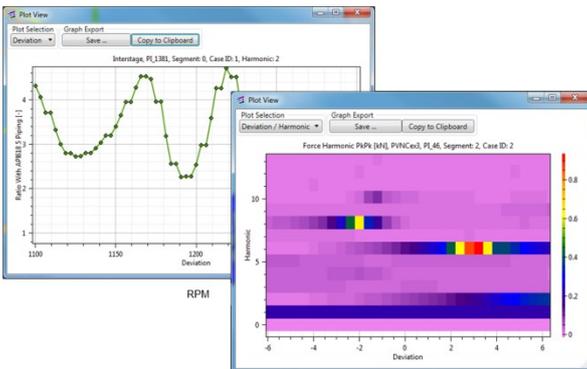

In these plots you can present the local result as function of RPM/speed-of-sound, as function of frequency (the frequency spectrum of the signal), or as function of both. This latter 2-D color plot shows you which frequencies and which deviations from the nominal operating condition are dominant at this pipe location.
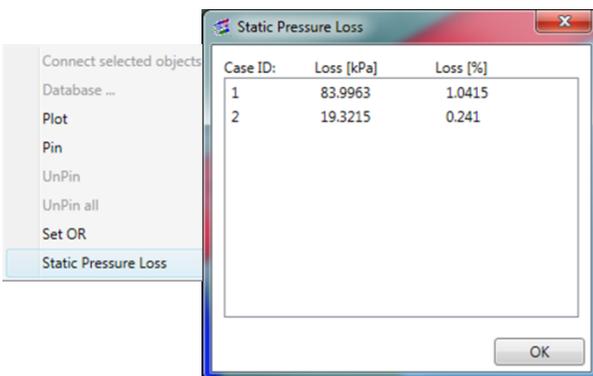
One of the results of a PulsimSuite simulation is the total (static+dynamic) pressure loss over each pipe segment. This result can always be visualized on the pipe system. For orifice plates, it is customary to evaluate the static pressure loss.

When hovering with the mouse over the location of an orifice plate, you can now right-click and select "Static Pressure Loss". This function shows you the component of the orifice's pressure loss caused by the mean flow only: the static pressure loss.
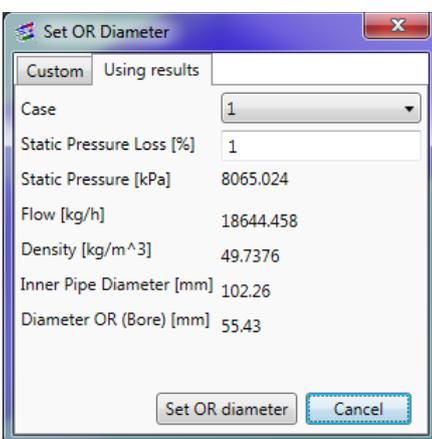
Alternatively, when hovering over an orifice plate in your model, you can choose to set the static pressure loss to a specified percentage of the local mean pressure. To do this, you select the option "Set OR". You will now see a window that shows the mean pressure and flow. Specify the required percentage of pressure loss, and the corresponding required orifice bore is calculated for you. By a click on the Set OR button, the diameter property of the orifice in your model is set to the calculated value.

* The PulsimSuite license does not include the ANSYS license, which needs to be procured separately from ANSYS.

For more information or your feedback, please contact us: pulsimsupport@tno.nl   pulsim.tno.nl
                                                                  wilbert.vink@tno.nl   t: +31.88.8668609