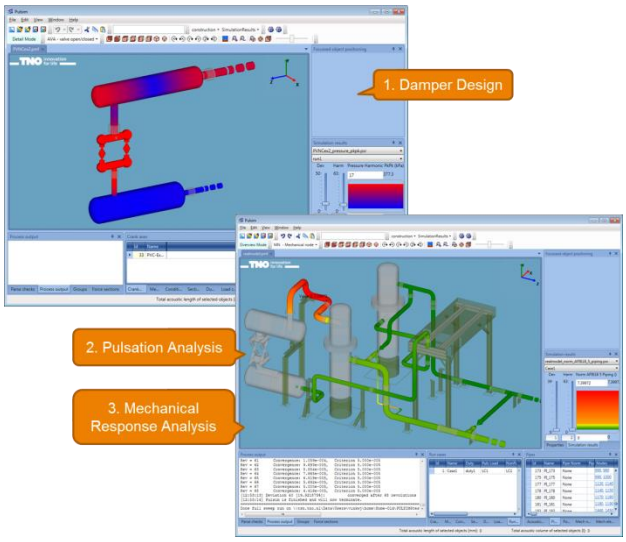
PULSIMSUITE NEWSLETTER

This is the PulsimSuite newsletter of Q1, 2017, presenting version 2.1.4 that was released in January 2017. We give you the latest developments in our API 618/674 pulsation & vibration software toolbox. You are also invited to check out our website: pulsim.tno.nl.



Introducing version 2.1.4:
The mechanical response simulations have been improved in speed and robustness (e.g. by adding more validity checks). As a beta-release, you have the option to select “Modern” element types for the ANSYS model*, which improves accuracy and calculation speed of the ANSYS model. When using this option, do verify your results carefully, as this is a beta-release of the PulsimSuite part of this functionality.

	Element Type	
Component	**Legacy**	**Modern**
Acoustic Pipe	PIPE16	PIPE288
Mechanical Pipe	PIPE16	BEAM188
Elbow	PIPE18	ELBOW290
Straight Beam	BEAM4	BEAM188
Tapered Beam	BEAM44	BEAM188

This table shows the ANSYS element types that PulsimSuite generates by default (“Legacy”) and when setting the option to “Modern”.
The process of translating a model into a mechanical finite-element model has been made faster, and the calculations of eigenmodes/frequencies and forced harmonic responses are also faster than before, now that the new element types are used in the ANSYS model.

MRA cases

Id	Name	Ac Model	AcCaseID	MRA Group	Fstart [Hz]	Fend [Hz]
1	mra1	PAmodel	1		0.01	80
2	mra2	PAmodel	5		0.01	80
3	mra3	PAmodel	12		0.01	80

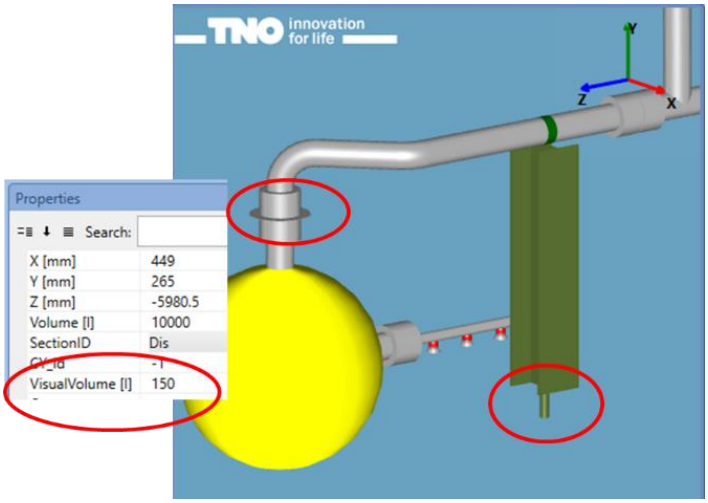
Save ANSYS result files	Element Types	Modal Damping [%]
no	Legacy	2
no	Legacy	2
no	Legacy	2

In version 2.1.4, the mechanical response can now be calculated for specific acoustic run cases that you have found to be critical. You can now also more straightforwardly modify the mechanical information in your model (add supports, add structures), and rerun the analysis only for the cases you are most interested in. For advanced ANSYS users, we have retained the possibility of saving the raw ANSYS result files as an option that can be set in the run case definition. PulsimSuite only needs the post-processed .psr files (Pulsim Simulation Result).

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

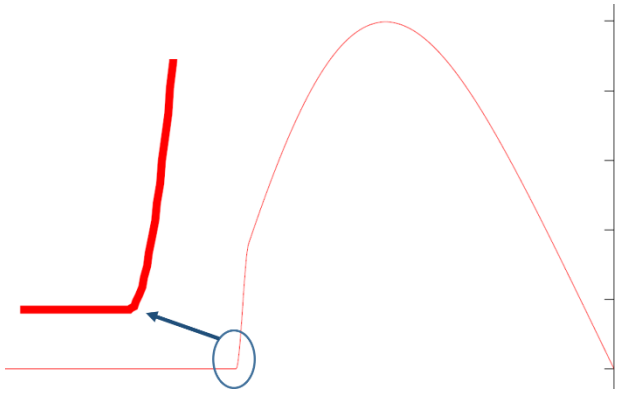
```
ADSForceCriterion;TNO; # ADS : Force selection criterion [TNO,API618_5th].
ADSTake;0.75; # ADS : Take Threshold [kN/4 in.].
ADSSkip;0.25; # ADS : Skip Threshold [kN/4 in.].
ADSMaxDev;10; # ADS : Max. number of deviations per harmonic [-].
ADSDivSize;0.025; # ADS : Multiplier to Deviation Range [-].
ADSMinDivSize;1; # ADS : Minimum Division Size [%].
AllowDispXYZ;-1; # Allowable Level for Vibration Displacement Results in X,
AllowDispT;-1; # Allowable Level for Vibration Displacement Results in T [-
AllowVelXYZ;-1; # Allowable Level for Vibration Velocity Results in X, Y, Z
AllowVelT;-1; # Allowable Level for Vibration Velocity Results in T [-1 for
AllowAccelXYZ;-1; # Allowable Level for Vibration Acceleration Results in X
AllowAccelT;-1; # Allowable Level for Vibration Acceleration Results in T [
AllowStress;-1; # Allowable Level for Stress Results [-1 for default or val
```

The advanced user can tailor the mechanical response analysis even further to his needs. By setting the parameters in a configuration file (see the snippet in the picture here), you have an influence on the automatic force selection algorithm and on the specified limit levels for displacement, velocity, acceleration, and stress.



Visualization of the 3-D geometry has been improved:

- Orifice plates (OR) are visualized larger, so that you can better see them in between a flange pair
- The volume node (VO) of course has a “Volume” property. You can now also define a “VisualVolume”: if that value is 0, the node will be shown as large as the “Volume” is, but if “VisualVolume” has a value different from 0, the node will be shown as large as the “VisualVolume”. The calculations will of course be done with the “Volume” value.
- The joint element (JNT) is better visible, by an increased diameter of the visualization of it.



For pump systems, the flow pulse shape now represents the inertia of cylinder valve and liquid, at the moment when the cylinder valve opens. This greatly improves the convergence and stability of calculations for liquid systems. By faster convergence, the simulations have also become faster.
This picture shows how the start of the pulse is no longer a sudden “step”. This more realistic representation of valve opening thus strongly reduces the (irrelevant) high-frequency content of the time signals.

By the end of May, 2017, PulsimSuite 2.1.5 will be introduced at the first PulsimSuite User Meeting. You are warmly invited to attend this meeting, possibly share your experiences in one of the user’s presentations, and get to know fellow-users during the informal networking-session-with-drinks. A short workshop on compressor manifold analysis with PulsimSuite will be part of the program. Please contact us if you would like to attend this meeting, which will take place in Düsseldorf, Germany. We shall give more information on the meeting date and location in February.

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