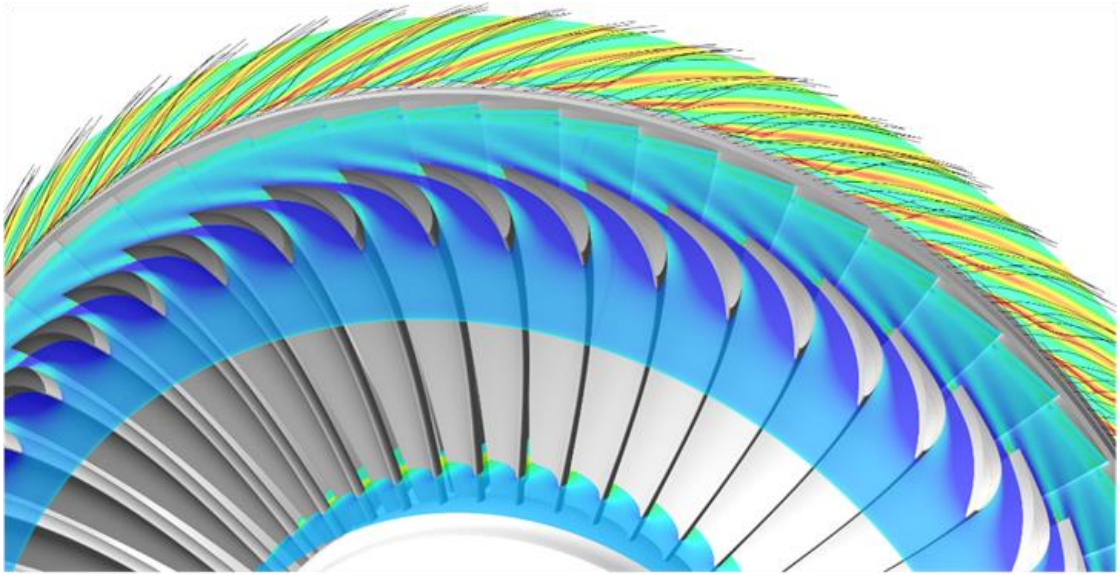
FLTR 2.0 - automatic tool for blade flutter prediction in ANSYS CFX

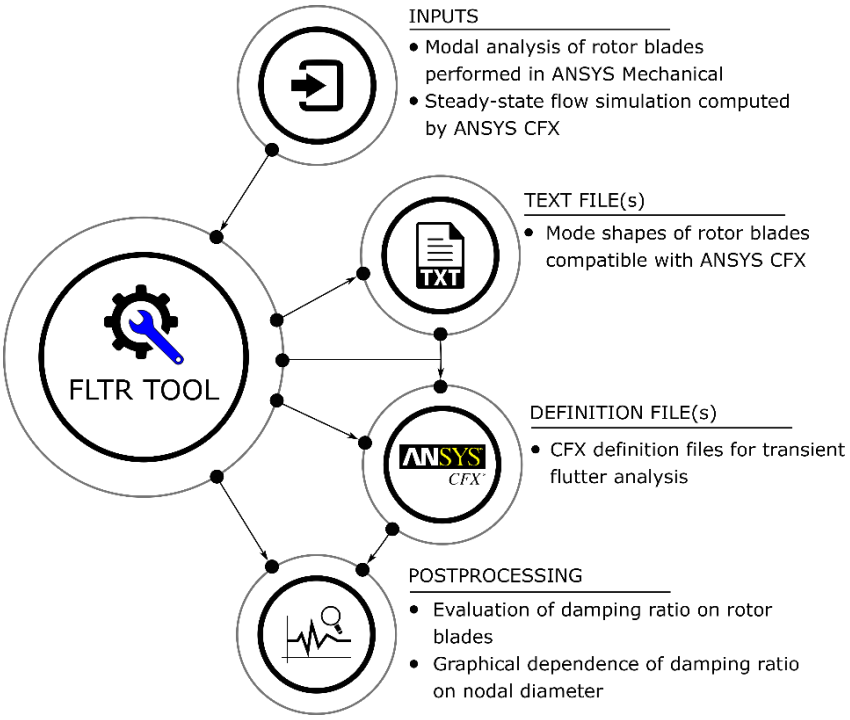
KEY PURPOSES:

- Automation of blade flutter prediction in ANSYS CFX
- Complex unsteady CFD analysis with all geometric details
- Significant reduction of manual preprocessing and postprocessing – 95%
- Reduction in simulation time – more than 60%
- Comparability of flutter results – same methodology



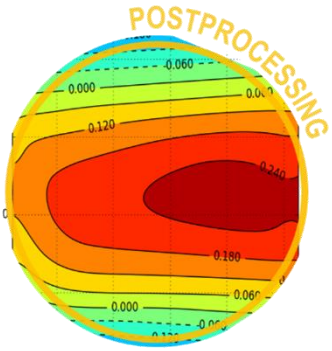
DESCRIPTION:

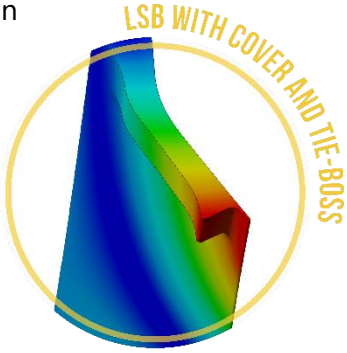
Design of the rotor blades for rotating machines is a complex procedure involving the study of the interaction between structural dynamics and aerodynamic forces – flutter analysis. Flutter analysis involves several steps that are constantly repeating: modal analysis, aerodynamic work calculation and damping ratio evaluation. We would like to introduce our tool for the automation of flutter analysis in ANSYS CFX. The tool automates the generation of transient input files for CFD simulations and the evaluation of damping ratio from aerodynamic work done on blades. Are you performing flutter analysis on daily basis and would you like to significantly reduce time of your analyses? We can help you with our FLTR tool.

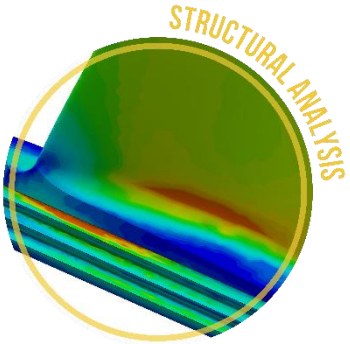


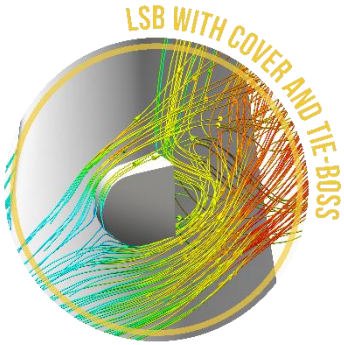
OTHER BENEFITS:

- Excellent support with long time flutter knowledge
- Methodology validated against an experimental measurement
- Efficient flutter results storage (S-shapes, Flutter maps)
- FLTR tool training with a practical tutorial (onsite/online)
- 3D unsteady viscous flow/structural analysis
- Stage geometry with cover and tie-boss
- Stator/Rotor unsteady interaction
- Complex mode shapes
- Mode shapes precheck
- All types of turbomachines



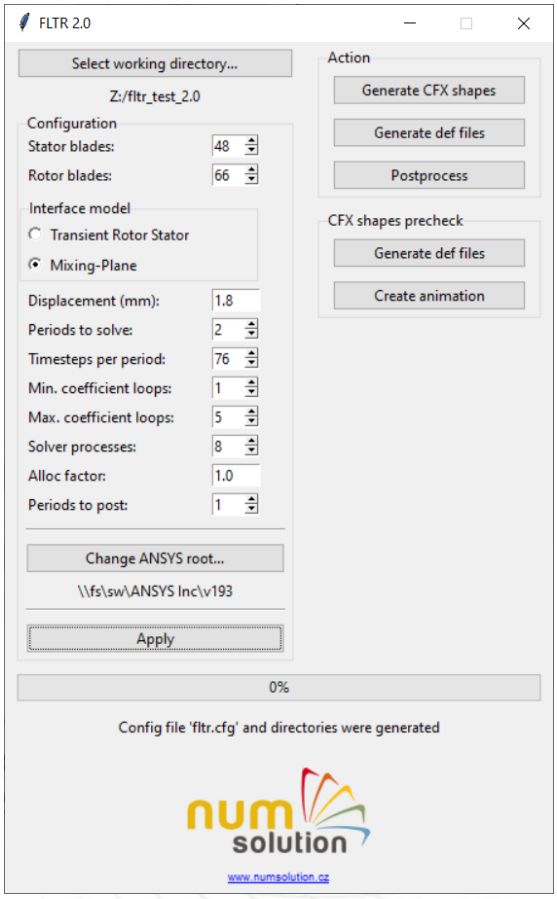






SOFTWARE/HARDWARE REQUIREMENTS

- Supported OS: Windows/Linux
- ANSYS 2019 R3
- Python 3.6



CONTACT:

Do not hesitate to contact us for more information, a flutter training or a demo license > info@numsolution.cz or visit our web > www.numsolution.cz

NUM solution s.r.o.

Corporate Headquarters & Office:

U Pergamenky 1145/12

Prague, 170 00

Czech Republic