



ANSYS v16.0 Fluids update: New Features and Enhancements

March 31, 2015 | 11.00am IST | 60 minutes

Presented by: Mahendra Joshi, Technology Specialist, ANSYS India

[CLICK TO REGISTER](#)

In this one hour webinar, you will learn about the new fluid modeling capabilities offered with ANSYS release 16.0. The update will include a presentation of the key features on robustness and workflow improvements.

Furthermore, details on Turbomachinery, Enhancement in Numeric, Adjoint solvers, Comprehensive physics modeling, Maximizing CFD productivity, HPC performance improvements, etc. will also be covered.

About the presenter:



Mahendra Joshi holds a Master's Degree in Thermal Engineering from Indian Institute of Technology Madras, Chennai and a Bachelor's degree in Mechanical Engineering from Walchand College of Engineering, Sangli. He has over 8 years of experience out of total 10 years in Thermal and Fluid simulation. He is associated with ANSYS for last 2 and half years. His areas of interest are moving deforming meshing and User Defined Functions with ANSYS FLUENT. at ANSYS, he is responsible for customer training coordination in his region in addition to customer technical support and presales. Prior to joining ANSYS, he has been designing and analysing aircraft engines for their thermal management. Prior to that, he was also with R&D of one of the top Indian automobile OEM located in Pune.

Follow us



www.ansys.com

We welcome you to multiple avenues of accessing Technical Support from ANSYS

Web based portal: <https://support.ansys.com/portal/site/AnsysCustomerPortal>

Phone Number: 1800 209 3475 or +91 20 665 4300

Business Hour: 10:00AM – 06:00PM (Monday – Friday)