



Self Financed online Training Program on

Computational Fluid Dynamics (CFD) with OpenFOAM

July 23 – 28, 2020

About VNIT Nagpur:

Visvesvaraya National Institute of Technology, Nagpur (VNIT Nagpur), formerly Visvesvaraya Regional College of Engineering, Nagpur (VRCE), is a public engineering and research institution India. It is located in Nagpur, Maharashtra, in central India. It was established in June 1960 by the Government of India and later named in honor of engineer, planner, and statesman, Sir Mokshagundam Visvesvaraya. VNIT Nagpur is centrally-funded and belongs to the National Institutes of Technology (NIT) system. In 2007, the institute was conferred the status of Institute of National Importance by an Act of Parliament of India. The institute awards Bachelors's, Masters and Doctoral degrees in engineering, technology, and architecture.

Objectives:

- The objective of the present online training program is to appreciate the open-source CFD code OpenFOAM and simplify OpenFOAM use.
- This training gives a platform to the participant to understand OpenFOAM from basics to advanced to programming level.

Important Instructions:

- Last date of registration: July 20, 2020.
- The Participant has to ensure internet connectivity during the event.
- Venue/ Platform - **Google Meet**: Meeting links will be shared through registered Gmail-ID
- **Registered participants will get access to pre-recorded videos.**

Timings:

- Teaching and Discussion: 10:00 – 1:00 hrs. Hands-On: 14:00 – 17:00

Registration:

- Last date for online application: July 20, 2020. Course Fees: Rs 2,500/-.
- Kindly register through Gmail-Id only.
- Registration form link:

https://docs.google.com/forms/d/139vBhmAMqPLnIZ1HXQiMHKVK65_WNtd4lj5CUzzA2qI/edit?usp=sharing

Coordinator:

Dr. Trushar B. Gohil, Assistant Professor

Mobile: 88888 72132, Tel: (0712) 2801169, Email: trushar.gohil@gmail.com

Organised by

Department of Mechanical Engineering,

Visvesvaraya National Institute Of Technology (VNIT), South Ambazari Road, Nagpur-440010 (INDIA)

Disclaimer & Usage of trade marks

OpenFOAM® and OpenCFD® are registered trademarks of ESI. This offering is not approved or endorsed by OpenCFD Limited, the producer of the OpenFOAM software.