

# Pranav Bapat

CFD Engineer



Pune, India • [www.linkedin.com/in/pranav-bapat](http://www.linkedin.com/in/pranav-bapat) • +919422044222 • [pranav.bapat09@gmail.com](mailto:pranav.bapat09@gmail.com)

---

## PROFESSIONAL SUMMARY

Multilingual computational engineer with the skill and the mindset to ensure consistent top quality. Professional, highly motivated team player comfortable performing tasks independently and proactively achieving results. Seeking opportunity to join a dynamic team in a growth organization that cultivates a close-knit and collaborative corporate culture where my skills and knowledge can continue being polished.

## TECHNICAL SKILLS

- **CFD simulation software:** Ansys Fluent, Ansys CFX, Star CCM+
- **FEA software:** Simulia Abaqus
- **CAD modelling software:** Ansys SpaceClaim, Ansys DesignModeler, Ansys Workbench, Autodesk Inventor
- **Programming software:** Matlab, Python (Certification: [ude.my/UC-KZ0X8XUV](https://ude.my/UC-KZ0X8XUV))
- **Python IDE:** Jupyter-notebook, Pycharm
- **MS Office utilities:** MS Word, MS PowerPoint, MS Excel, MS Outlook
- **Document editor tool:**  $\text{\LaTeX}$
- **Operating systems:** Linux (Ubuntu), Windows OS, Mac OS

## EDUCATION

**Master's degree in Aeronautical Engineering** 08/2016 – 04/2019

Linköping University, Linköping, Sweden

**Area of focus:** CFD, Aerodynamics, CHT, Computational Mechanics, Aircraft Design

**Bachelor's degree in Mechanical Engineering** 08/2011 – 07/2015

Savitribai Phule Pune University, Pune, India

**Area of focus:** Fluid Mechanics, Heat Transfer, CAD Modelling

## PRACTICAL EXPERIENCE

**Technische Universität Berlin (TU Berlin)** 05/2018 – 02/2019

*Design and Fluid Simulation of a Fluidic Growth Chamber (Master's Thesis)*

- Analysis of fluid flow over the backwards-facing step design chamber using Computational Fluid Dynamics (CFD).
- Used Large Eddy Simulation (LES) turbulence model to solve the flow in the chamber. Star CCM+ software used for simulations.
- Study provides base for research work in understanding the filamentous fungal morphology.
- Thesis work performed independently. Report available: <https://bit.ly/2w1A4pi>.

## WORK EXPERIENCE

**Entercoms Solutions Pvt. Ltd., Pune, India** 06/2015–06/2016

*Process Specialist*

- Integral member of 12-person team which provided after sales logistics solutions for Dell Inc.
- Placement of bulk orders for customers located in Americas, EMEA and AP regions.

- On daily basis analyze and provide solutions for defected parts and place an order to replace them if required.
- Conduct weekly interactions with the client and briefly give stock of the project's development.

## PUBLICATION

**A Numerical Model to Obtain Temperature Distribution During Hard Turning of AISI 52100 Steel (2015)**  
*ScienceDirect (Materials Today: Proceedings)*

- Detailed work available: <https://doi.org/10.1016/j.matpr.2015.07.150>.

## ACADEMIC PROJECTS

### **Aerodynamic Analysis of an Ultralight Aircraft | Project team of 6**

- Aircraft aerodynamic analysis using CFD.
- Geometry editing for simulation done using Ansys Spaceclaim; Ansys Fluent for CFD simulations.
- SST  $k-\omega$  turbulence model used solving the flow over the aircraft.
- Adjoint Solver tool used for optimization of the aircraft design.

### **Assessment of finite volume schemes for convection-diffusion problems | Project team of 2**

- Used finite volume method for solving the transport equation for 1D diffusion and convection-diffusion problems; solver developed in Matlab.

### **Rotor aerodynamics study | Project team of 2**

- Using Blade Element Momentum Theory (BEM) study and design of wind turbine rotor to generate 2MW electricity for average wind speed of 11 m/s; solver developed in Matlab.

### **Driven Cavity flow solver | Project team of 2**

- 2D Finite Volume Solver using SIMPLE algorithm to study Driven Cavity flow; solver developed in Matlab.
- Matlab solver results were validated by simulating the flow in Ansys Fluent software.

### **Heat transfer through a wall insulation | Project team of 3**

- Insulation performance study of wall configuration designed for cold weather.
- 1D Steady and time dependent solvers developed in Matlab.

### **Aerodynamic analysis of the Ahmed Body | Project team of 2**

- 2D CFD simulation to study flow over the body.
- Ansys DesignModeler was used for geometry creation; Ansys fluent for CFD simulations.
- Design optimization was performed to reduce drag generated due to body shape.

## LANGUAGES

- English (Native or bilingual proficiency)
- Hindi and Marathi (Mother tongue)
- German (Limited working proficiency)
- Swedish (basic proficiency)