

Satya Sasank K

Bachelor of Engineering, Mechanical

Hyderabad, Telangana, India

Mob.: +91 8095231308

Email : kssasank43@gmail.com

Links

*follow links in hyper bold

LinkedIn : **Satya Sasank**

GrabCAD : **Aytas**

Projects : **CodeBlue**

Skills

CAX

SolidWorks, FreeCAD,
OpenFOAM, ParaView,
Converge Studio 3.0,
Ansys-Fluent, ANSA
GrabCAD Print

OS

Linux{Ubuntu, Manjaro},
Windows

SCRIPTING

C++, Python, Octave/Matlab

OTHERS

Office Suite, Git, Latex,
Blender, Atlassian Jira

Education

2015-2019

Bachelors in Mechanical Eng.
Visvesvaraya Technological
University, Belagavi
CGPA : 6.47/10

2020-2021

Masters certification in
External aerodynamics
Skill-lync

Coursework

Mathematics
Thermodynamics
Heat Transfer
Fluid Mechanics
Energy Engineering
Additive Manufacturing
Mechatronics

Experience

- 2020 MAR **AstroSoc Corp Applied Sciences Division** **R&D Intern**
Designed various mechanical components and UAV frames, conducted flow simulation over a fuselage, analyzed the results, suggested suitable changes in geometry.
- 2021 AUG **Freelancer** **Multidisciplinary**

Certifications/Achievements

- 2020 JULY **NASA's RMA Challenge** **Online Challenge**
Won First Place in NASA's GrabCAD Challenge on Restraint and mobility aid design. [Challenge Link](#)
- 2017 AUG **Training on 3D printing** **Workshop**
Completed a project based training program on 3D printing
Certificate : **36770008**
- 2017 JUNE **Training at Bangalore Aircraft Industries Limited** **Workshop**
completed a 10 day workshop on flight dynamics, structures, Cad modelling, stress analysis, fabrication, and material testing.

Projects

- 2021 JAN **1D supersonic nozzle flow (Mac-Cormac Method)** **Matlab**
Developed a OCTAVE code to simulate isentropic flow through quasi 1D nozzle at subsonic-supersonic velocity.
- 2021 MAY **FSAE car project** **Converge**
Simulated flow over an FSAE car model in CONVERGE CFD and analysed the aerodynamics of wings and body.
- 2021 JUNE **Flow over Ahmed Body** **Ansys Fluent**
Modeled the Ahmed body and simulated flow over it in a steady state condition in ANSYS FLUENT.
- 2021 AUG **CFD meshing of a Tesla Cyber Truck** **Meshing**
A CAD model of a Tesla cyber truck has been cleaned and pre-processed using ANSA.
- 2021 SEPT **Creating a new OpenFOAM solver** **C++**
Using the code of the existing openfoam solvers a new solver has been created and tested upon a simple case.

Languages

English : Full professional proficiency | Telugu : Native proficiency