Abstract Submitted
for the DFD15 Meeting of
The American Physical Society

RANS and LES simulations of the airflow through nasal cavities GIACOMO LAMBERTI, Columbia Univ — The prediction of detailed flow patterns in nasal cavities using computational fluid dynamics (CFD) can provide essential information on the potential relationship between patient-specific geometrical characteristics and health problems. The long-term goal of the OpenNOSE project is to develop a reliable open-source computational tool based on the OpenFOAM CFD toolbox that can assist surgeons in their daily practice. The objective of this study was to investigate the effect of the turbulence model and boundary conditions on simulations of the airflow in nasal cavities. The geometry, including paranasal sinuses, was reconstructed from a carefully selected CT scan, and RANS and LES simulations were carried out for steady inspiration and expiration. At a flow rate near 20 l/min, the flow is laminar in most of the domain. During the inspiration phase, turbulence develops in nasopharynx and oropharynx regions; during the expiration phase, another vortical region is observed down the nostrils. A comparison between different boundary conditions suggests the use of a total pressure condition, or alternatively a uniform velocity, at the inlet and outlet. In future work the same geometry will be used for setting up a laboratory experiment, intended to cross-validate the numerical results.

Giacomo Lamberti
Columbia Univ

Date submitted: 30 Jul 2015 Electronic form version 1.4