

## **FLUID FLOW AND PARTICLE DEPOSITION IN THE HUMAN ORAL CAVITY AND OROPHARYNX**

*J. Steffens, R. Robinson\*, Department of Mechanical Engineering, [jts3034@rit.edu](mailto:jts3034@rit.edu), [rjreme@rit.edu](mailto:rjreme@rit.edu)*

Computational fluid dynamics (CFD) is used in order to better understand the movement and deposition of carcinogenic particles in the oral cavity and oropharynx resulting from smoking. A realistic computer model is created using a throat model created in MAYA (by Valerie Henry, Medical Illustration Department, RIT) using cryosectioned images of a cadaver combined with a dental cast of a 22 year old male volunteer. The dental cast is digitized using a Model Maker Z140 3D Scanner. This model is assembled in SolidWorks, meshed in Gambit, and exported into Fluent for CFD analysis. The flow is considered turbulent and unsteady. The flow field is computed using an Eulerian approach to solve the Navier Stokes equations. Smoke particle position is updated at each time step using a discrete phase model which uses a Lagrangian approach to solve the force balance equation. The turbulence solver chosen is the Shear Stress Transport  $k-\omega$  turbulent solver. The inlet condition consists of a slow puff where smoke particles are injected followed by a faster fresh air inhale. A time step of 0.01 seconds and inert particles of  $1000 \text{ kg/m}^3$  density and 9 micron diameter are used for this study. The grid is repeatedly refined in areas of high wall stress to converge the velocity field solution in order to increase the accuracy of the results. Location of each particle is tracked in order to determine deposition location. This research was sponsored by a grant from the American Cancer Society.