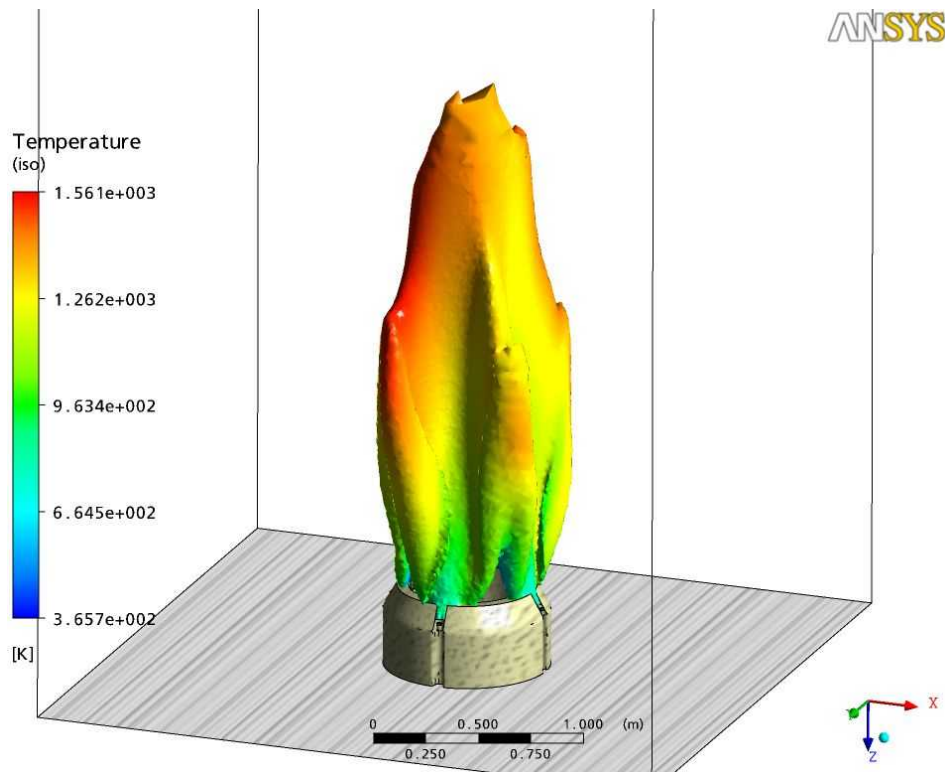


Computational Fluid Dynamics

Hamworthy combustion has the capability to carry out flow modelling of Boiler/Burner systems. Our Talented team of professional Engineers use both Physical and CFD modelling to check on air distribution, flue gas distribution, flue gas temperatures etc. CFD modelling is used by Hamworthy Combustion to develop new burner equipment and develop Low-Nox systems.

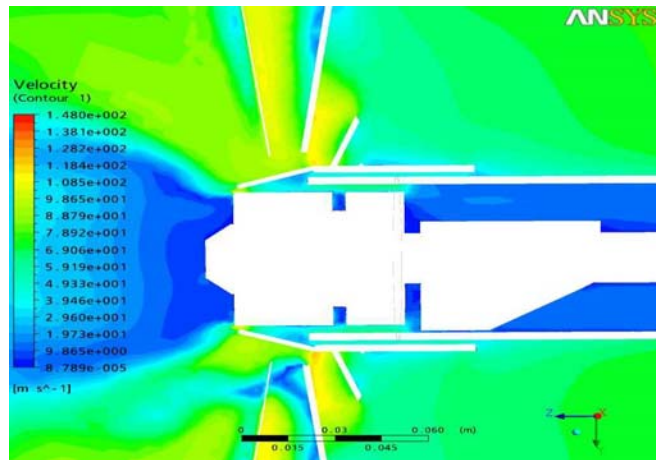
Key Features

- Extensive experience of applying CFD techniques to combustion systems, to improve plant efficiency and reduce emissions.
- CFD modelling of Combustion air systems to improve plant efficiency and reduce emissions.
- Extensive experience of implementing results of analysis to solve on-site problems.
- Ansys CFX advanced computer simulation software used.
- Geometry is modelled using Designmodeler
- The Geometry is Meshed using CFX mesh comprising approx 1 million cells.
- Unstructured mesh using tetrahedral and prism elements.
- Use of inflation layer cells.
- Use of a higher density of cells where variation of flow is great.

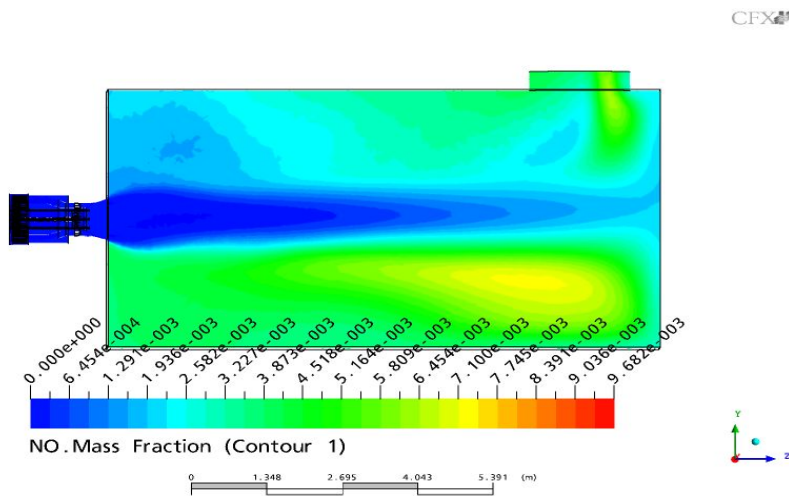


Steps to Create a CFD Simulation

- Build Geometry
- Mesh the Computational domain
- Apply Boundary conditions
- Run solver
- Check for convergence
- Post process results



Velocity profiles used to design nozzles



Prediction of NO production by CFD

For further information on combustion equipment please contact the head office:

Hamworthy Combustion Engineering Limited
 Fleets Corner
 Poole Dorset BH17 0LA
 Tel: +44 1202 662700
 Fax: +44 1202 665333
 Email: info@hamworthy-combustion.com
 Website: <http://www.hamworthy-combustion.com>

©2008 Hamworthy Combustion Engineering Limited – All rights reserved

Power/CFD/February 2008/Rev. 1

Hamworthy Combustion Engineering Limited reserve the right to make changes and improvements which may necessitate alteration to the specification without prior notice

HAMWORTHY
 COMBUSTION

Incorporating:
 PEABODY ENGINEERING
 AIROIL - FLAREGAS
 CHENTRONICS