GAMBIT and FLUENT 6.3

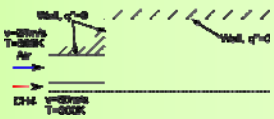
ChE641 Class
Combustion Modeling
Spring 2009

Establish the connection to CAEDM Linux **system**

- Download Cygwin from http://www.cygwin.com/ and install cygwin
- Download Putty from http://www.chiark.greenend.org.uk/~sgtatham/putty/download.html and install Putty
- Start the xwin program in Cygwin by typing "startx" from the command prompt
- Start Putty program
- Make sure that the X11 forwarding in the SSH of Putty configuration window is enabled
- Input 'linuxdesktop.et.byu.edu' in the host name and load it for a connection
- Input username and password to login in CADEM
- Input 'gambit' to go to the gambit environment
- Input 'fluent classwork' to start up FLUENT.

Problem Description

- The example considered here is methane combustion
- As the model is symmetric, only half of the domain width is modeled.
- The inlet of the 2D duct is split into two streams. A stream near the center of the duct is methane. The other stream is air.



GAMBIT

- Draw the geometry of object with the tool boxes of vertex, edge, face and volume
- Specify boundary types (Velocity-inlet, Wall etc)
- Specify continuum type (Fluid or Wall)
- Mesh lines, faces and volumes with certain grid schemes

Basic Steps to Solve a Problem with FLUENT

- **Step 1: Grid**
- **Step 2: Models**
- **Step 3: Materials and Operation Condition**
- **Step 4: Boundary Conditions**
- **Step 5: Solution**
- **Step 6: Postprocessing**

Setup and Solution
Step1 Grid

- Read the grid file
- Check the grid
- Smooth (and swap) the grid
- Scale the grid
- Display the grid

Setup and Solution
Step 2: Models

- Specify solver settings (Default Value)
- Turn on the Energy Equation
- Turn on the standard k- turbulence model
- Turn on the Species Transport model
- Check volumetric box in reaction tab and Eddy –Dissipation
- Check Full Multicomponent Diffusion
- Select Methane-Air Mixture from mixture material

Setup and Solution
Step 3: Materials and Operation Condition

- Check the materials and properties of mixture
- Change or Keep the default operating conditions

Setup and Solution
Step 4: Boundary Conditions

- Set the following conditions for air-inlet zone,fuel-inlet zone and walls

	v(m/s)	T(K)	O2	CH4	q"(W/m^2)
Inlet-Air	24	300	0.23	/	/
Inlet-Fuel	84	300	/	1	/
Wall	/	/	/	/	0

Setup and Solution
Step 5: Solution

- Select method of Pressure-Velocity Coupling, Discretization method and set the value of Under-Relaxation Factors
- Initialize the flow field using conditions at Inlet-air
- Enable the display of residuals during the solution process
- Save the case file
- Begin the calculation by requesting 500 iterations.
- Save the converged flow data

Setup and Solution
Step **6: Postprocessing**

- Display the predicted flow field
- Display the predicted temperature field
- Display the distribution of reactant and product species

Questions?