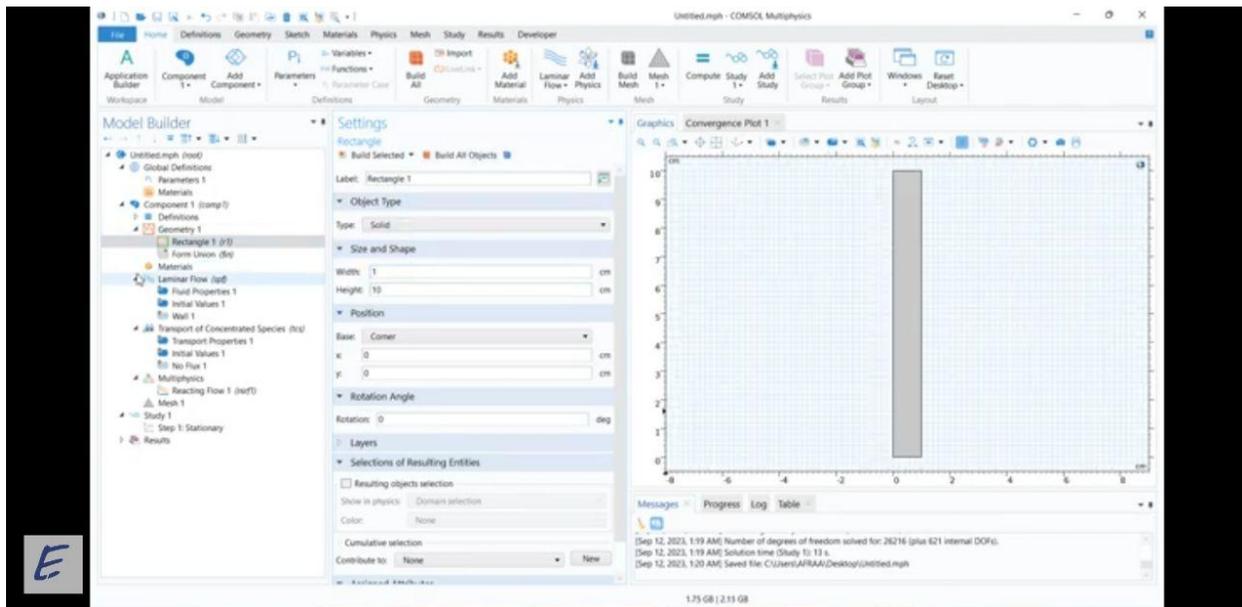




Computing chemical reaction flow

Start by drawing the geometry as shown below:



Apply the inlet boundary condition to the geometry:



Al-Mustaqbal University / College of Engineering & Technology

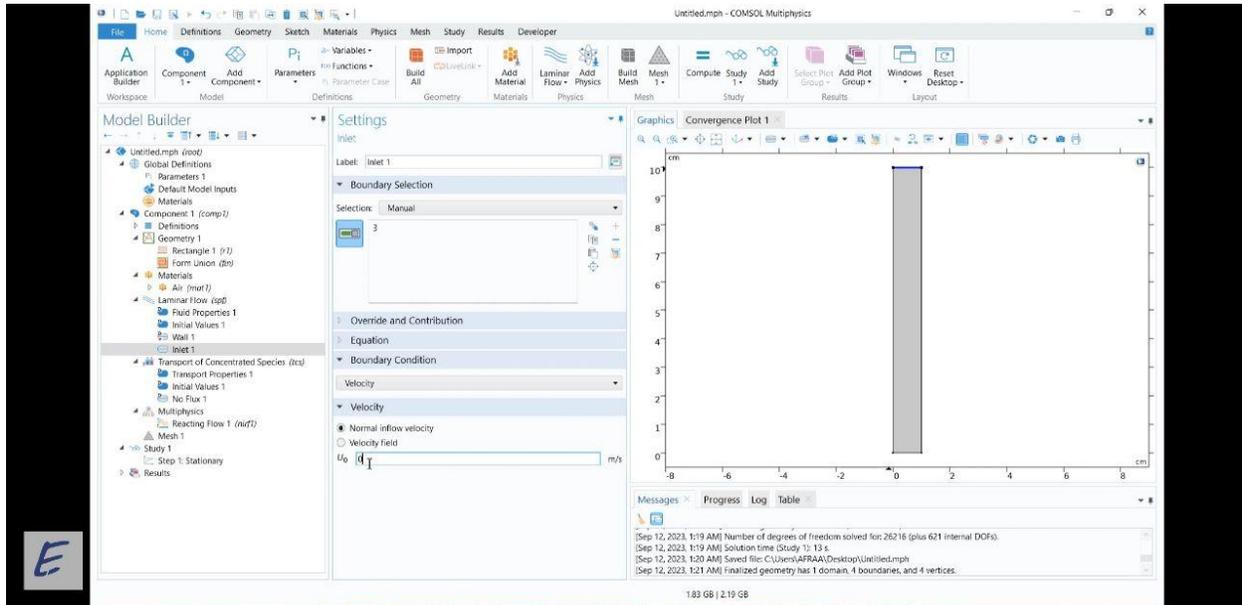
Department :Mechanical Power Engineering

Class : Third

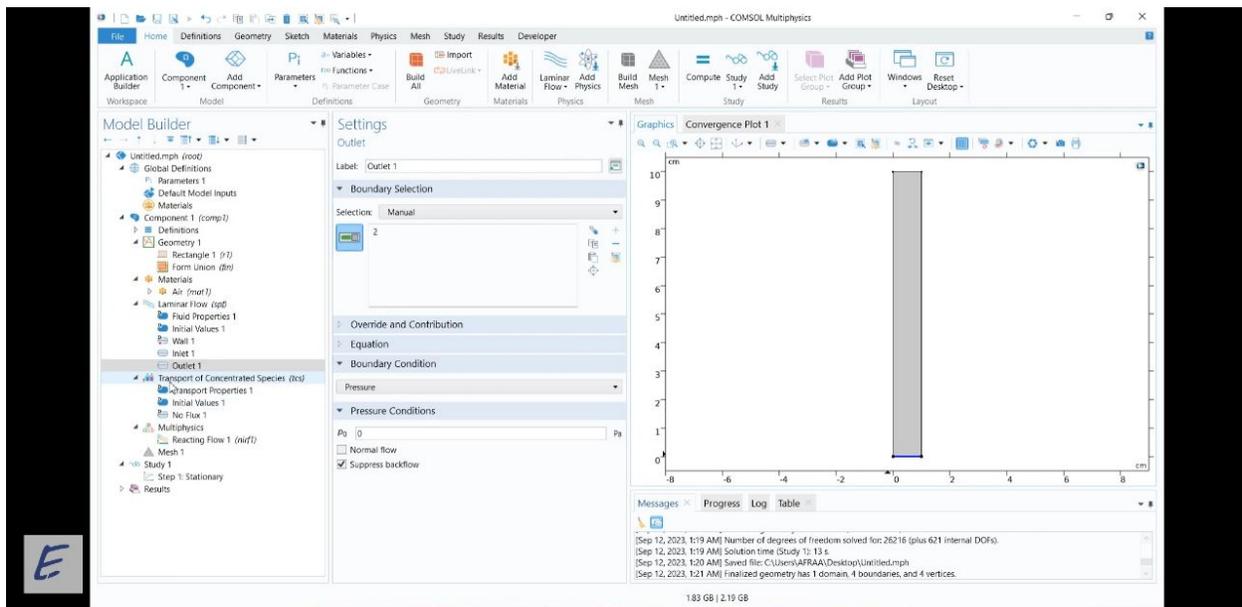
Subject : Computer Applications3 / Code : UOMU0000033

Lecturer: M.Sc Abrar Abdulkareem

2nd term – Lecture No. 11 & Lecture Name Eleventh Lecture



Also apply the outlet boundary condition within the laminar flow physics:

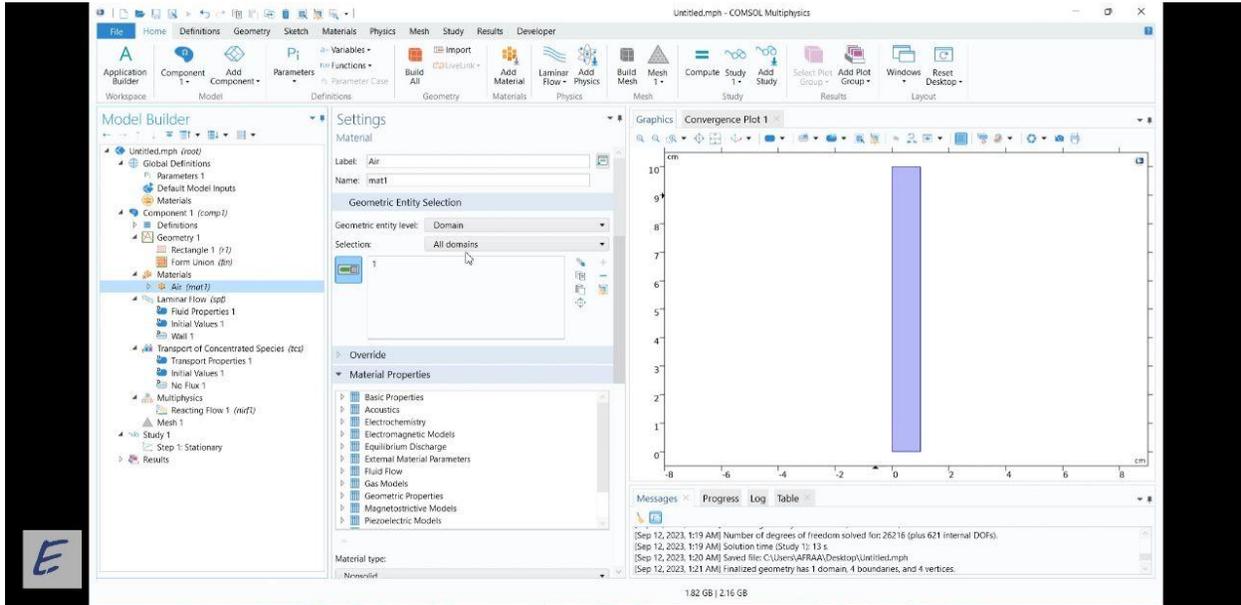




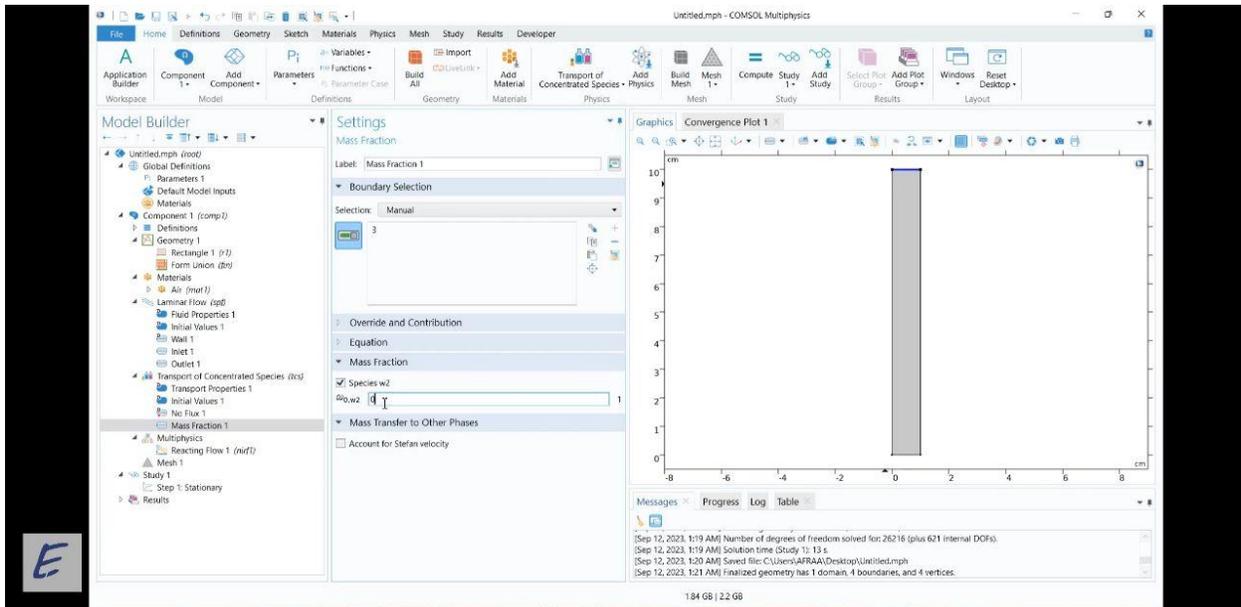
AI-Mustaqbal University / College of Engineering & Technology
Department :Mechanical Power Engineering
Class : Third
Subject : Computer Applications3 / Code : UOMU0000033
Lecturer: M.Sc Abrar Abdulkareem
2nd term – Lecture No. 11 & Lecture Name Eleventh Lecture



Add material to the geometry, we are choosing air here.



Apply mass fraction boundary condition:

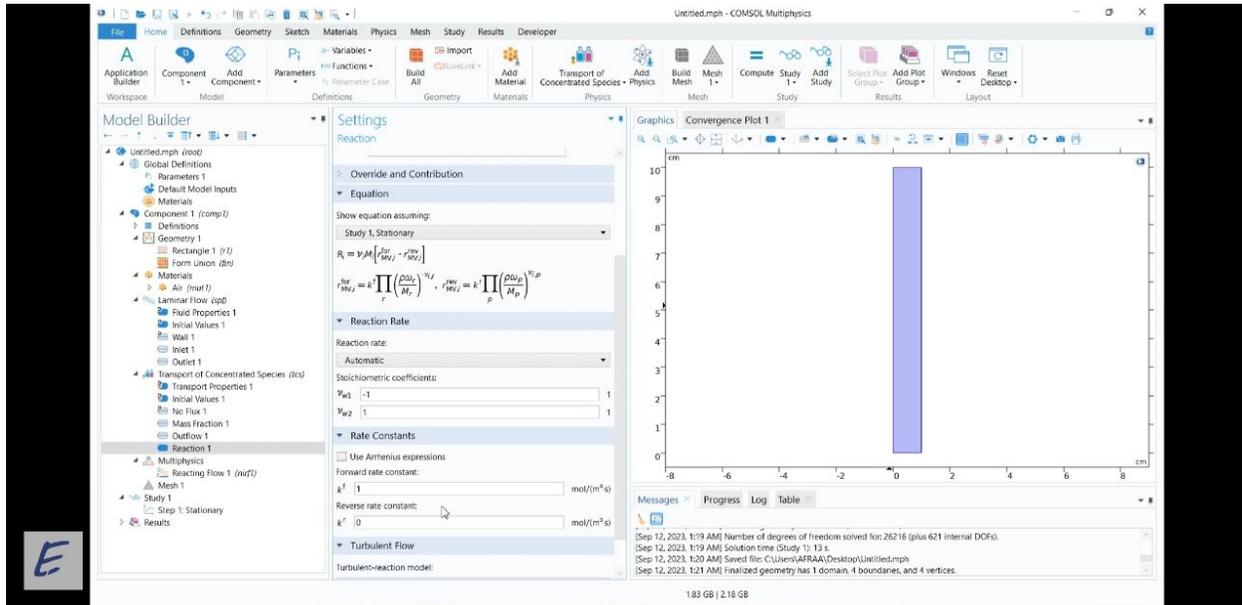




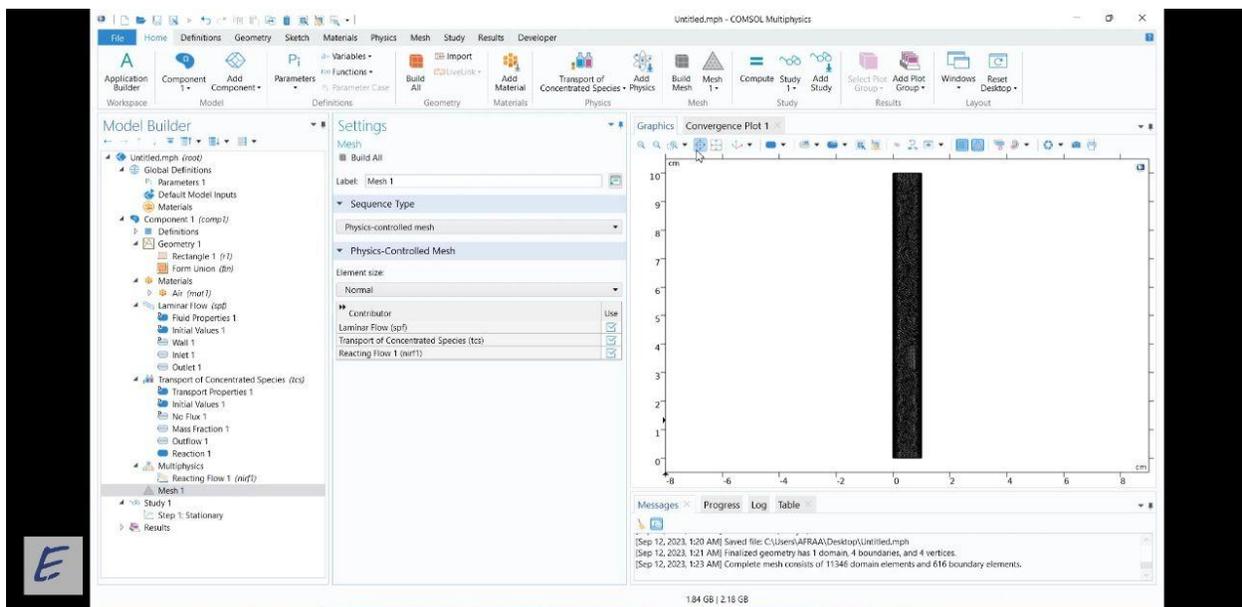
Apply reaction boundary condition:

The screenshot displays the COMSOL Multiphysics software interface. The 'Settings' window for a 'Reaction' boundary condition is open. The 'Equation' section shows the reaction rate equation: $R_i = \nu_i M_i \left[\frac{r_{for}}{M_{for}} - r_{rev} \right]$. The 'Reaction Rate' section is set to 'Automatic'. The 'Reaction Rate' section also shows the reaction rate equation: $r_{for} = k_f \prod_r \left(\frac{\rho_{M_r}}{M_r} \right)^{\nu_r} - r_{rev} = k_r \prod_p \left(\frac{\rho_{M_p}}{M_p} \right)^{\nu_p}$. The 'Reaction Rate' section is set to 'Automatic'. The 'Rate Constants' section shows the forward rate constant $k^f = 1$ mol/(m³s) and the reverse rate constant $k^r = 0$ mol/(m³s). The 'Turbulent Flow' section is set to 'Turbulent-reaction model'. The 'Graphics' window shows a 'Convergence Plot 1' with a vertical bar at 0 on the x-axis. The 'Messages' window shows the following text: [Sep 12, 2023, 1:19 AM] Number of degrees of freedom solved for: 26216 (plus 421 internal DOFs). [Sep 12, 2023, 1:19 AM] Solution time (Study 1): 13 s. [Sep 12, 2023, 1:20 AM] Saved file: C:\Users\AFRAA\Desktop\Untitled1.mph [Sep 12, 2023, 1:21 AM] Finalized geometry has 1 domain, 4 boundaries, and 4 vertices.

Apply reacting flow in Multiphysics:

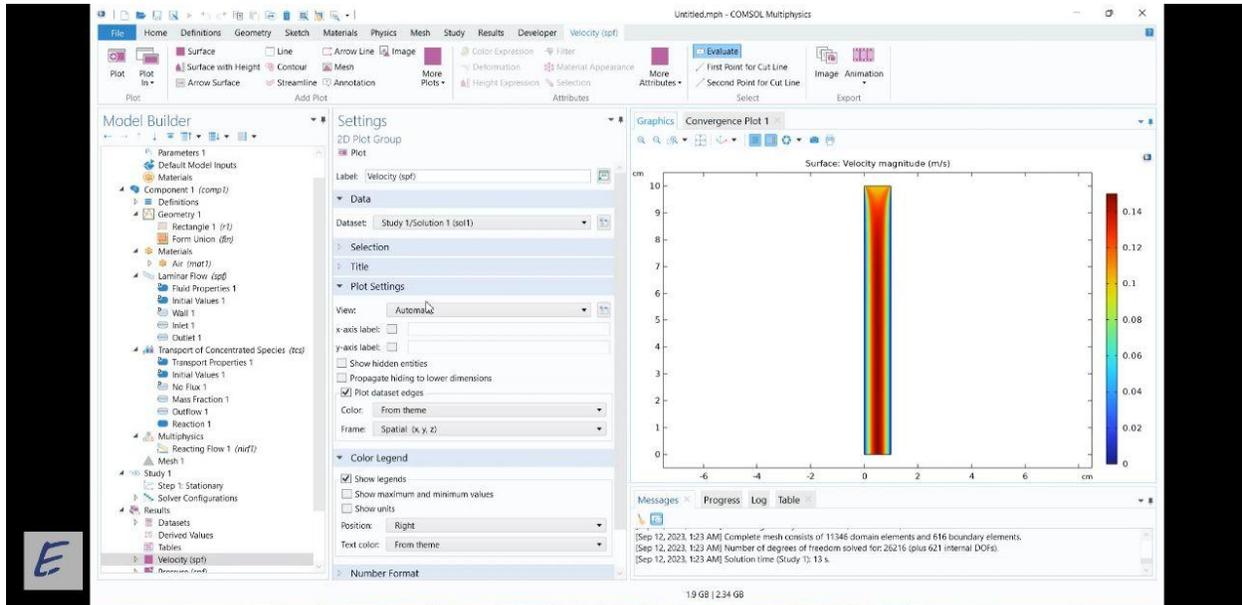


Apply the mesh to the geometry:





Compute the results:



The results shows the fluids velocity and pressure.