المادة: تطبيقات الحاسوب3

التدريسي: م.م. ابرار عبد الكريم سعيد

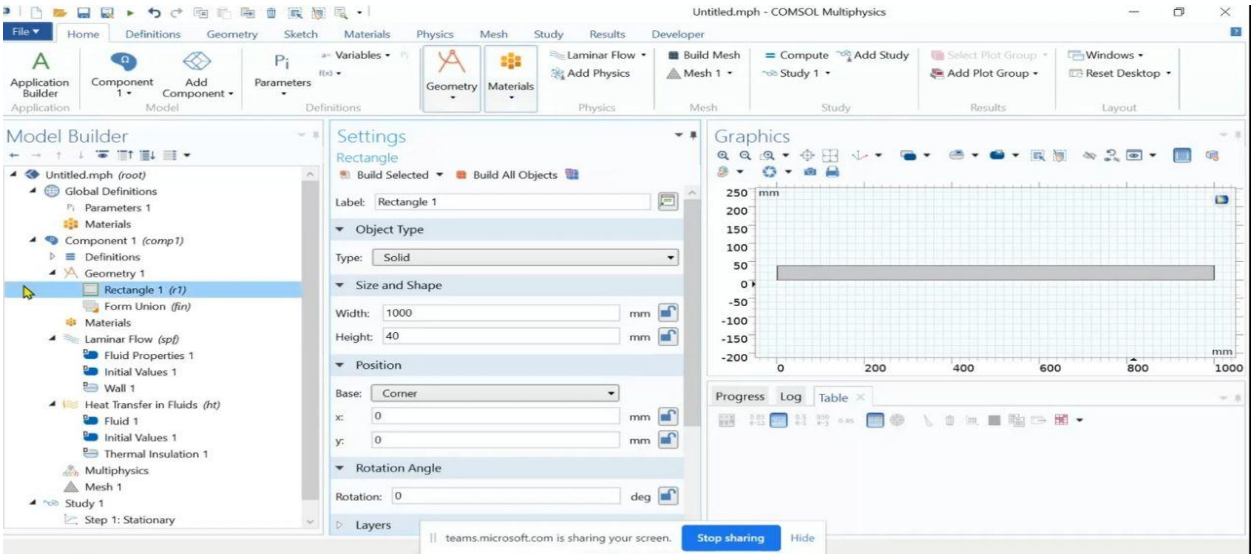
المرحلة: الثالثة

عنوان المحاضرة: المحاضرة الثانية

Computing the fluid flow properties through a channel:

To build a model we need to characterize all the properties and details of the geometry, materials, and values of the fluid properties.

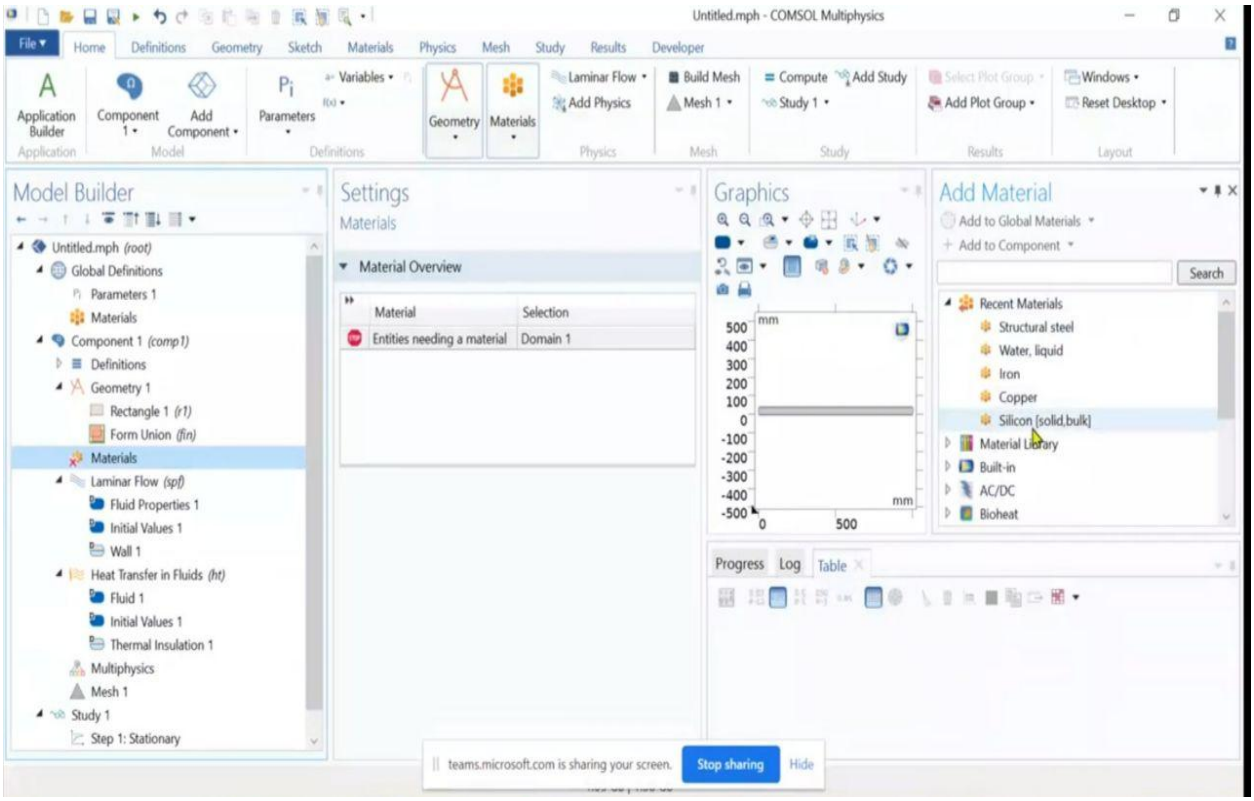
Here we are starting by establishing the geometry by drawing a side view of the channel:



E-mail: Abrar.Abdulkareem.Saeed@uomus.edu.iq

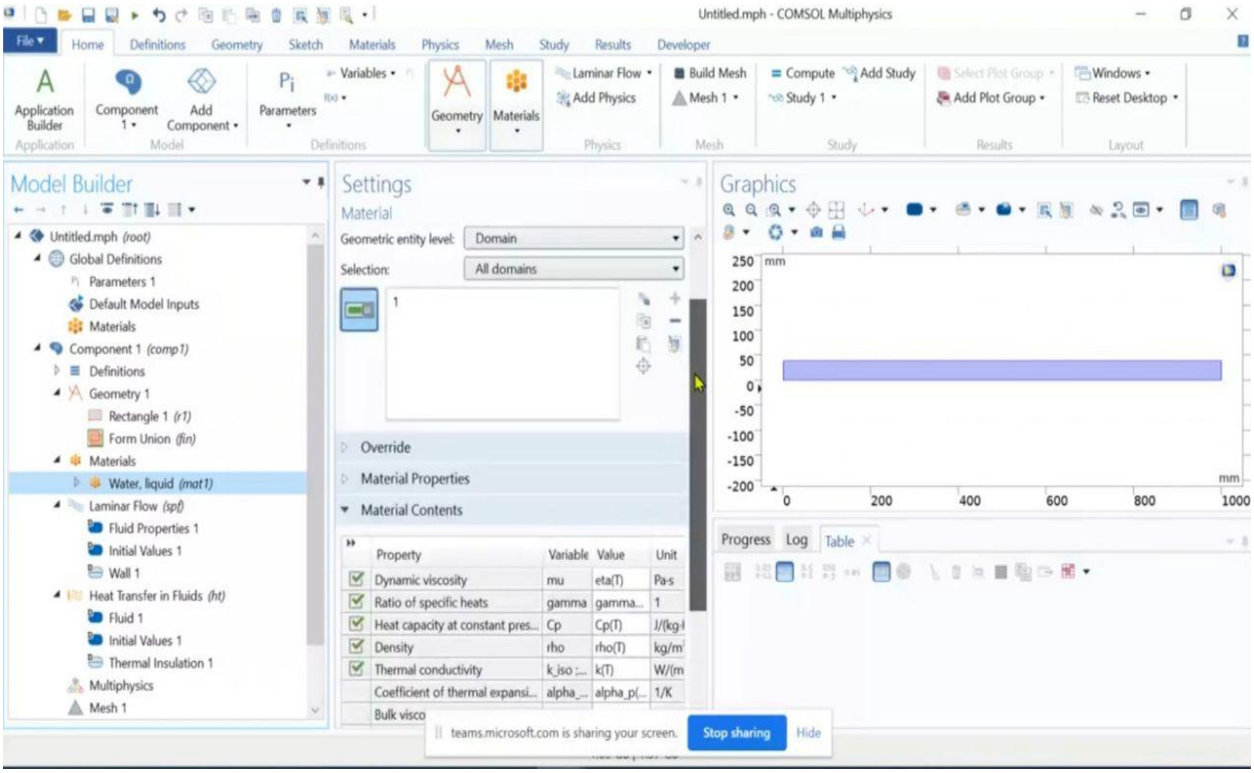
A side view of a cylindrical channel will be a rectangle in the two dimensional system. Also we insert the dimensions of the geometry (width and height). Here we are using the units of millimeters.

The next step is to choose the material of the channel from the library that already exists in the software.



Each material has its own properties which are listed in the software. However, we can choose to fill the properties manually in case we need to insert a specific value.

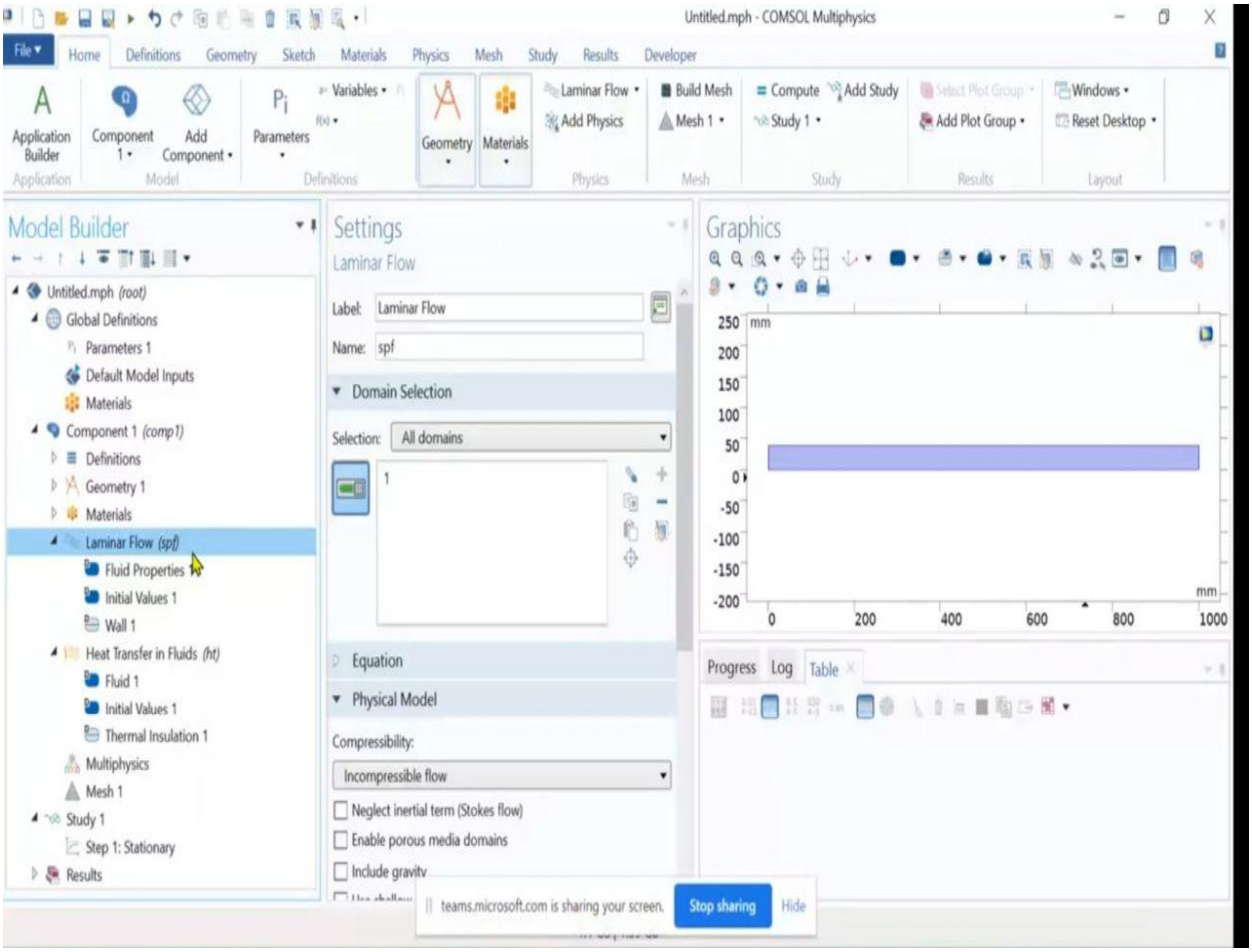
Here we are choosing the water as the fluid material inside the channel.



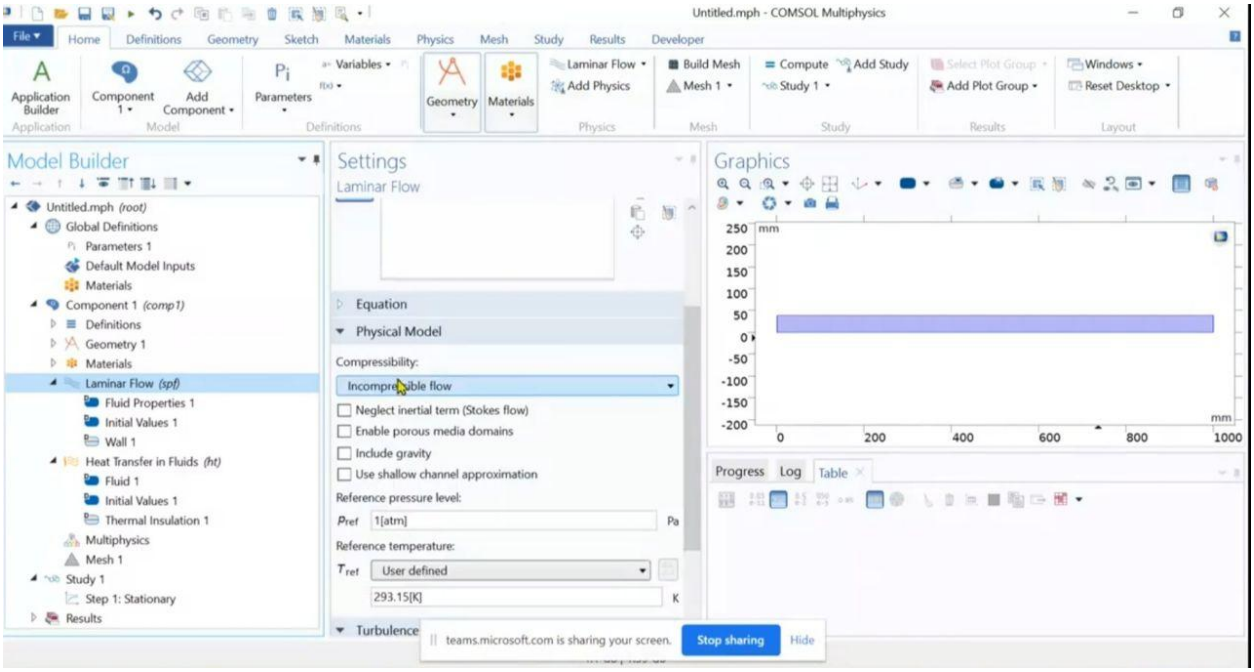
All the properties of the liquid (water) are illustrated in the picture involving dynamic viscosity, heat capacity, density, thermal conductivity and etc…

The next step is adding the type of physics. Here we have fluid flow which is usually divided into two types, laminar and turbulent.

The chosen type here is laminar flow. A fter choosing the physics we apply it to the specefied domain as demonstrated in the figure which will be in purple color.

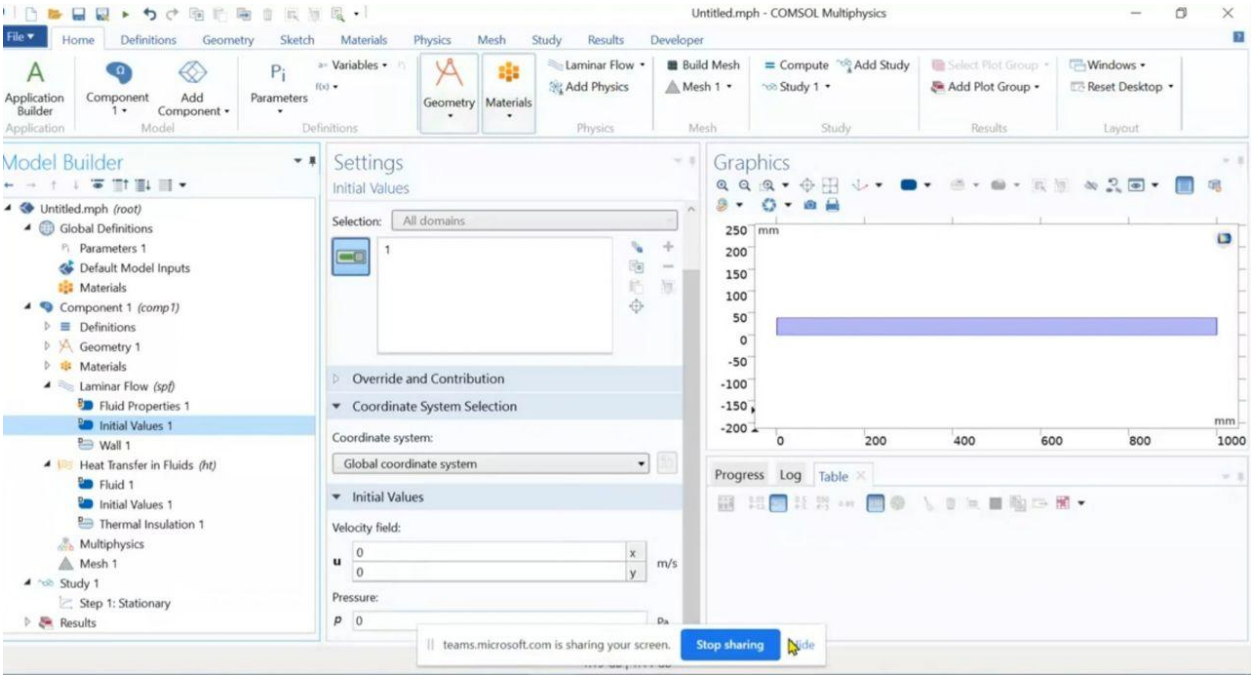


After that we need to specify the compressibility of the flow by choosing whether it is compressible or incompressible as illustrated below.

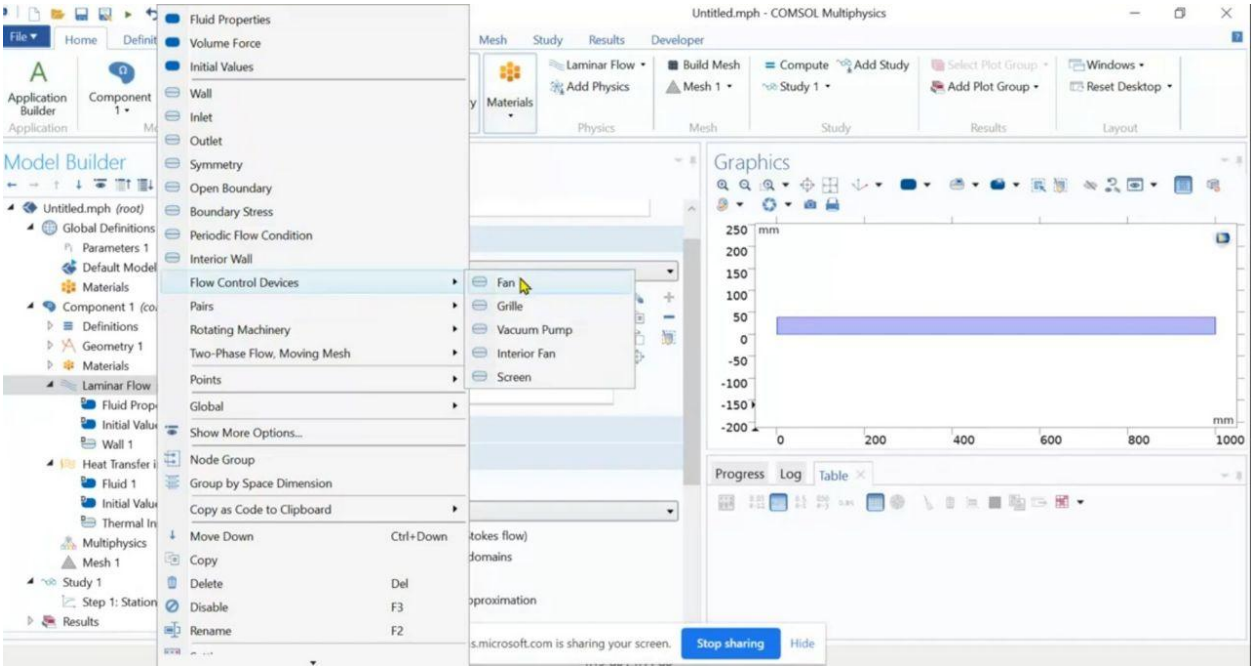


Also we must insert the reference temperature and reference pressure as shown above.

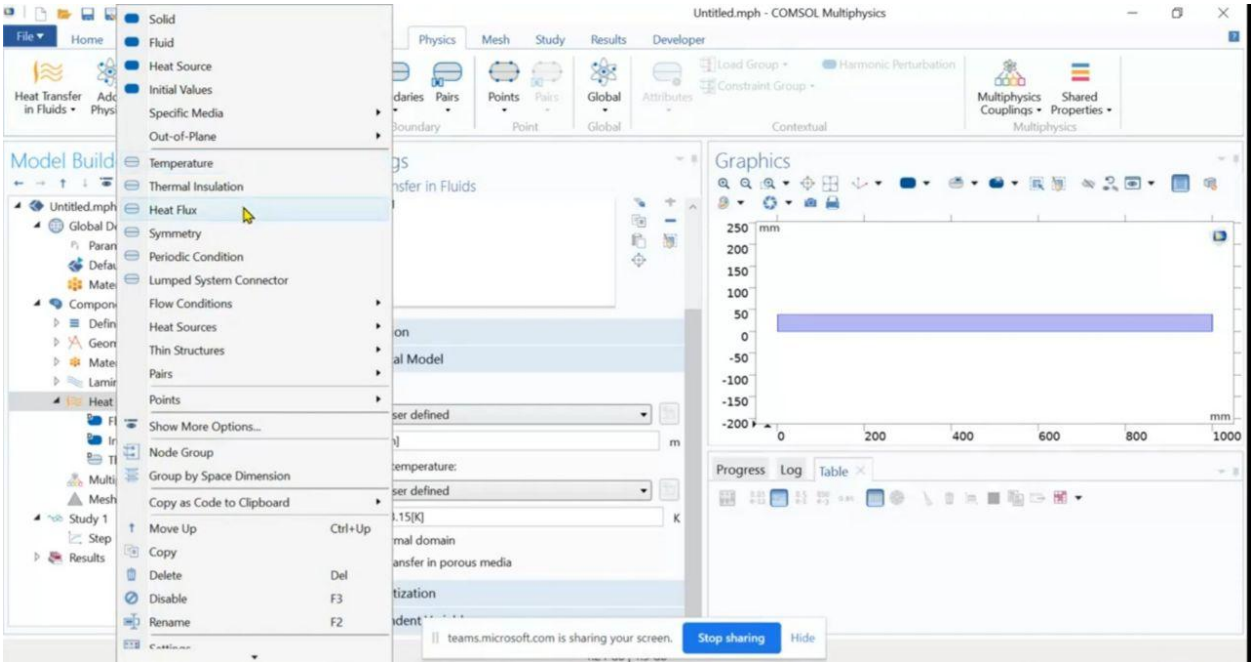
We need to insert the initial values of the fluid such as velocity components which are considered as boundary conditions.



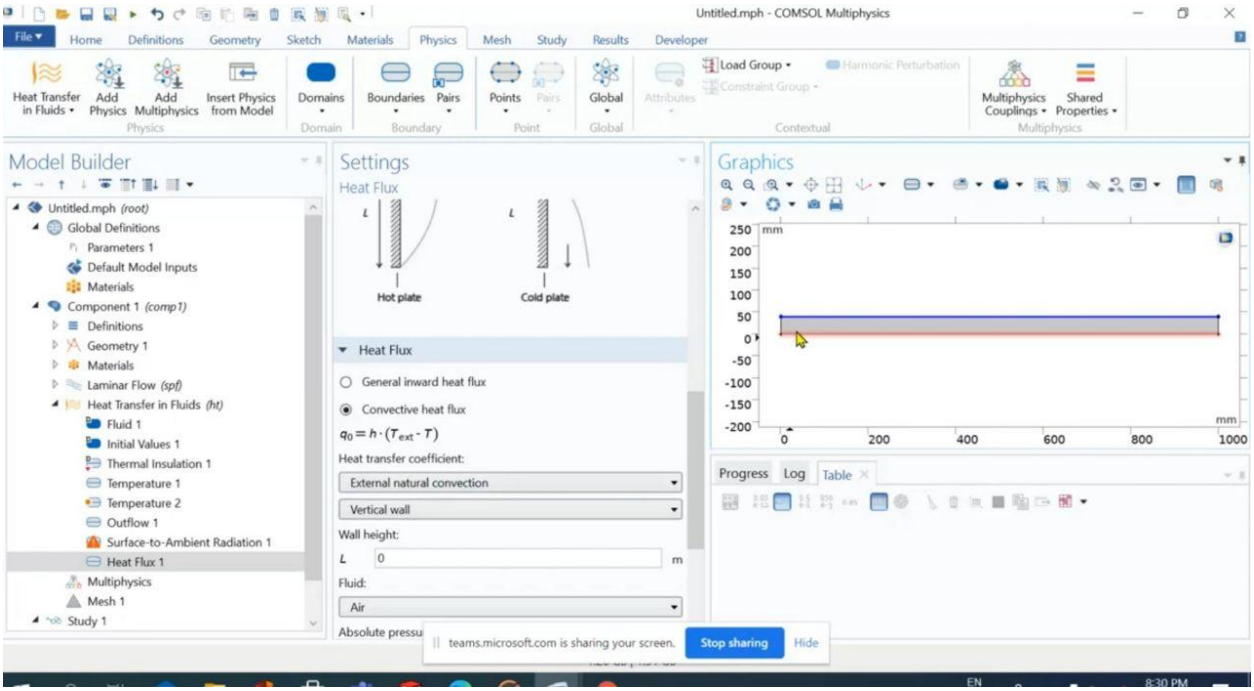
Also we have auxiliaries that can be added to the flow such as fan and pump.

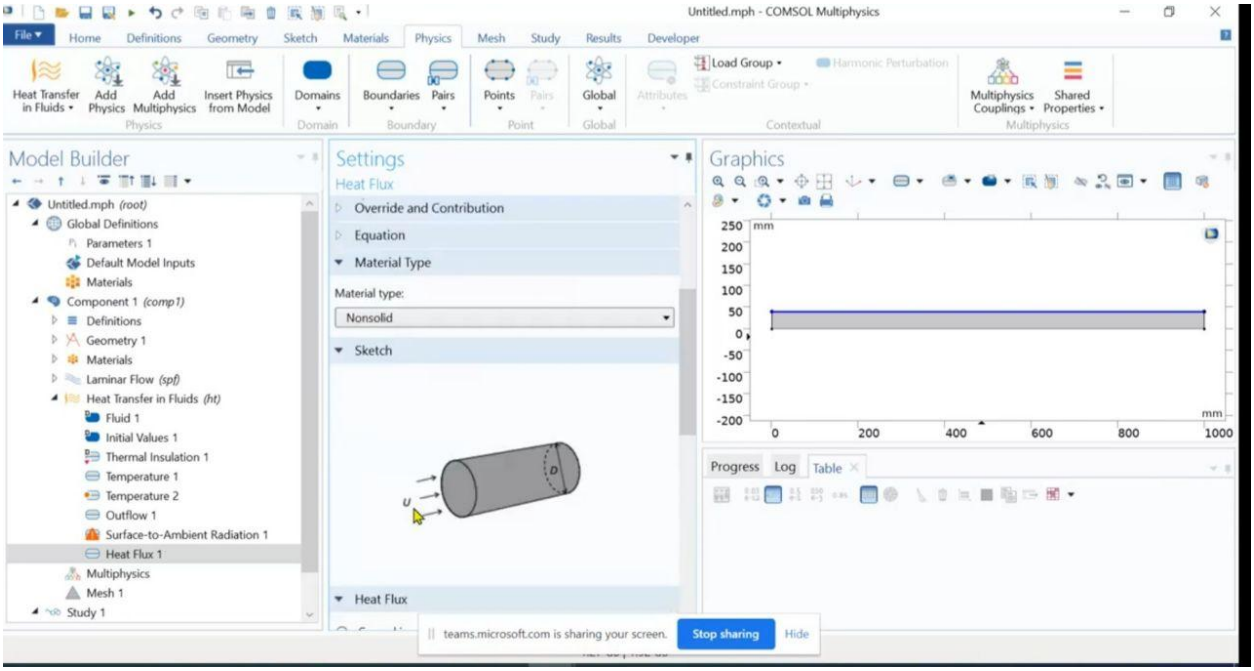


The boundary conditions of the heat transfer physics must be added too such as heat flux, temperature, and insulation.

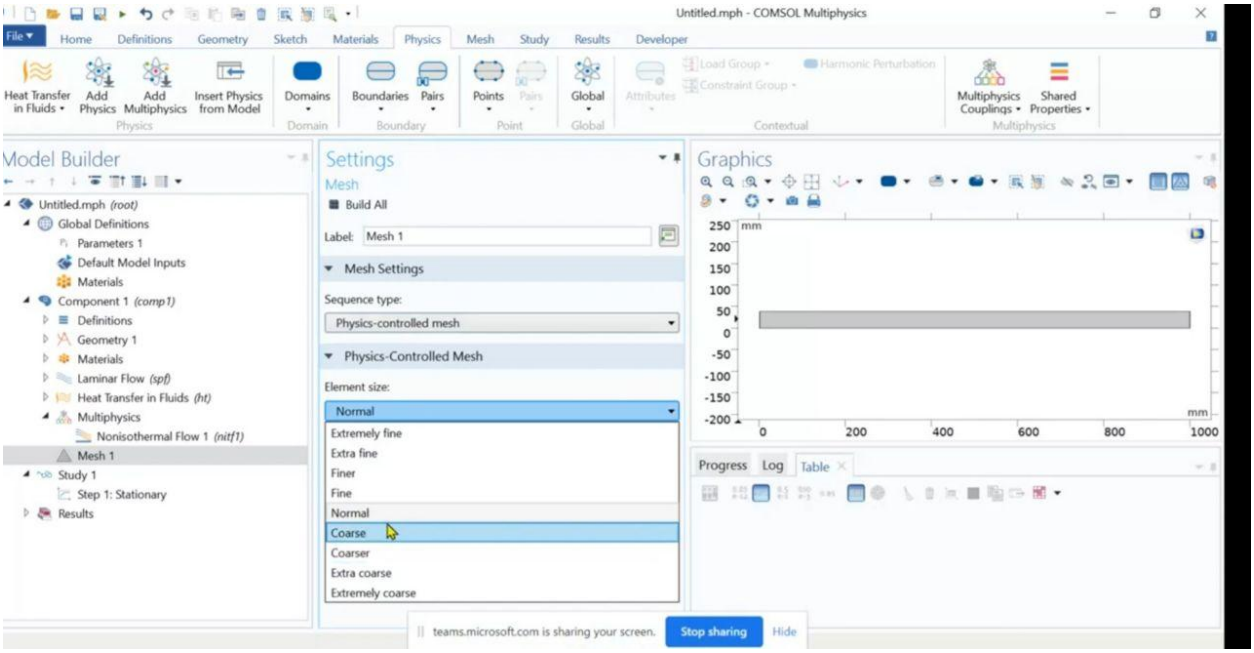


Applying the heat flux as boundary condition as demonstrated below:

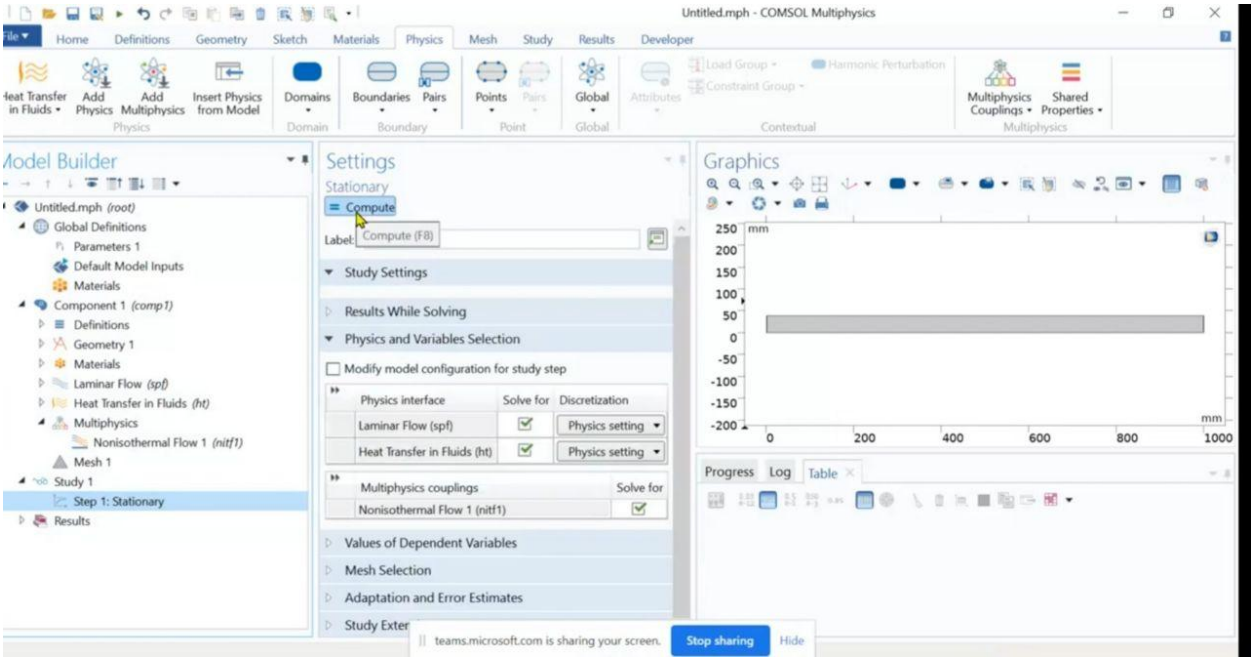


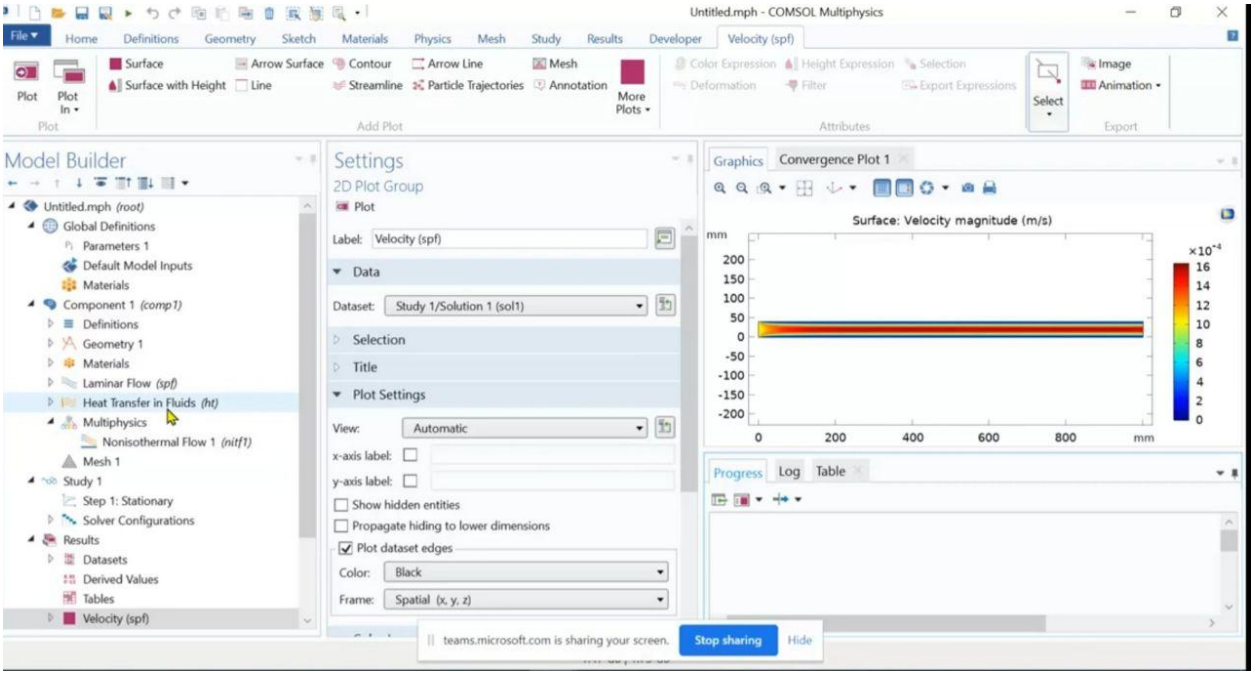


After applying all the boundary conditions and inserting the values of the parameters in the PDE we can apply the mesh which has different types:



After applying the appropriate type of the mesh we can press (compute) to attain the results:





The results demonstrates the velocity of the fluid flow inside the channel.