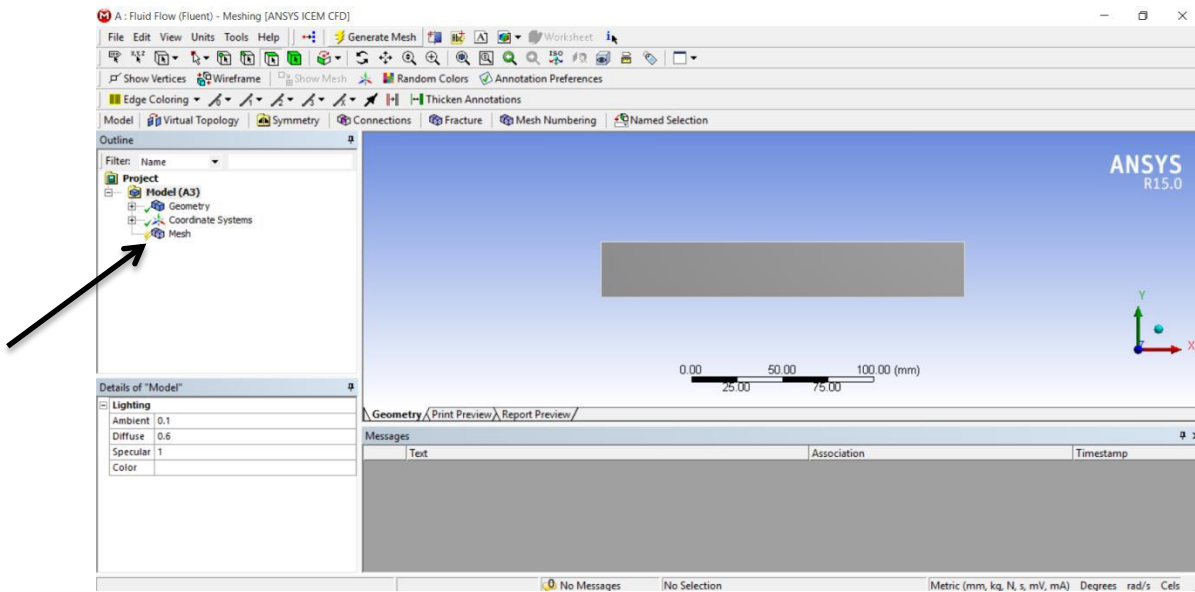
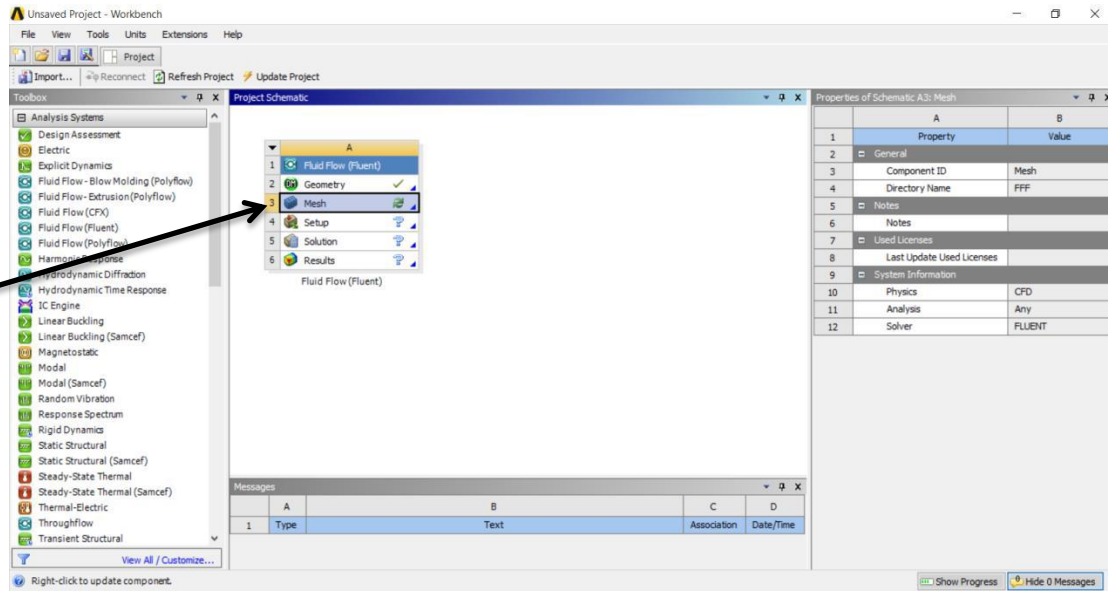


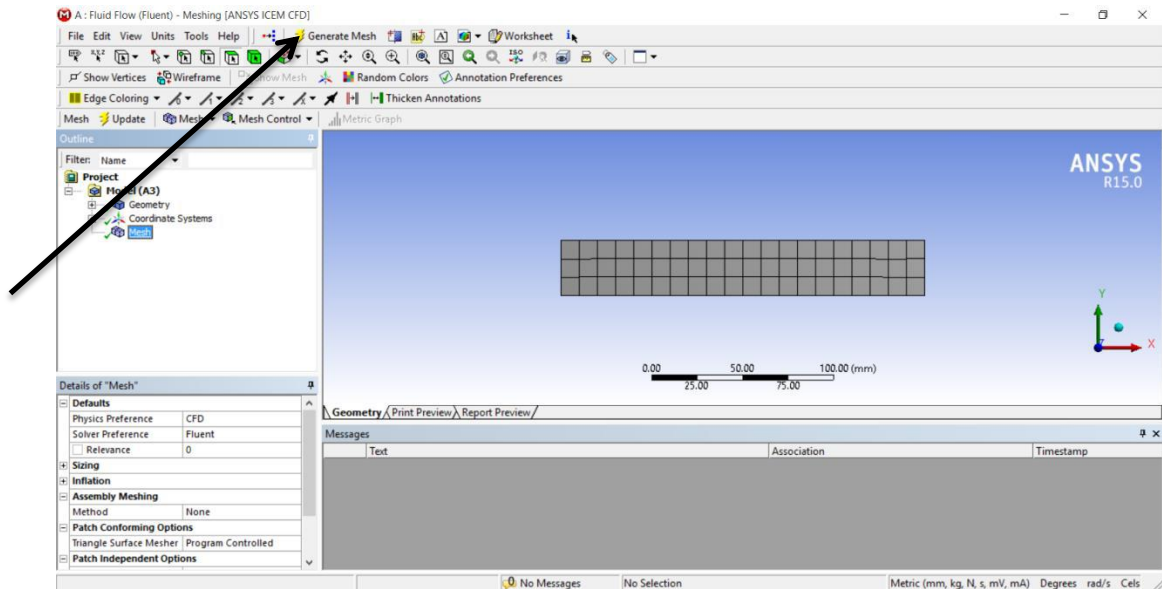


9- Analysis systems → fluid flow (fluent) → Mesh

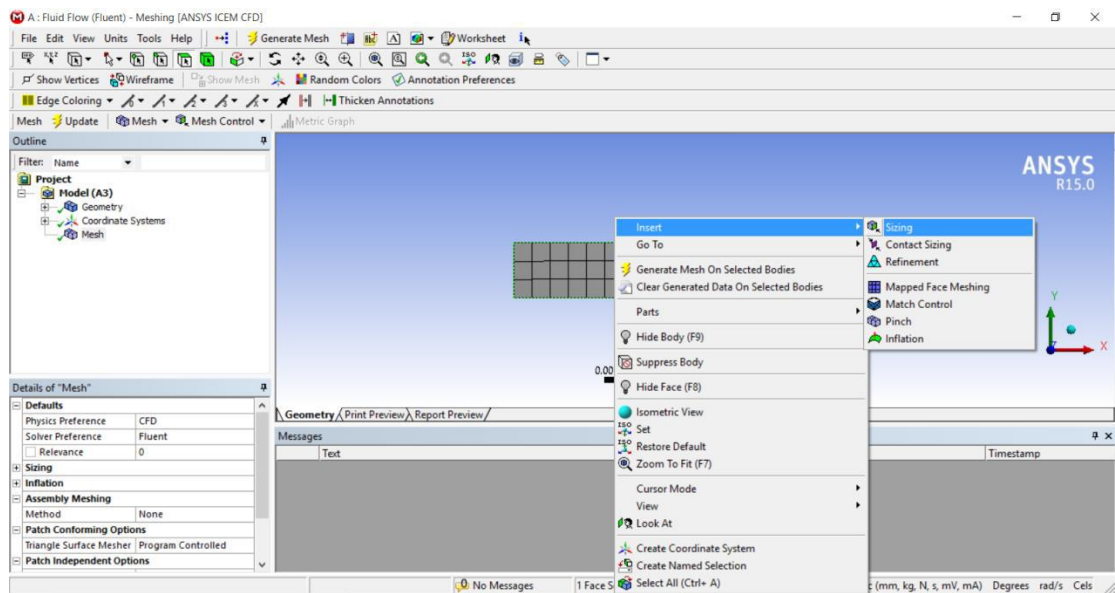




10- Mesh → generation mesh → Mesh

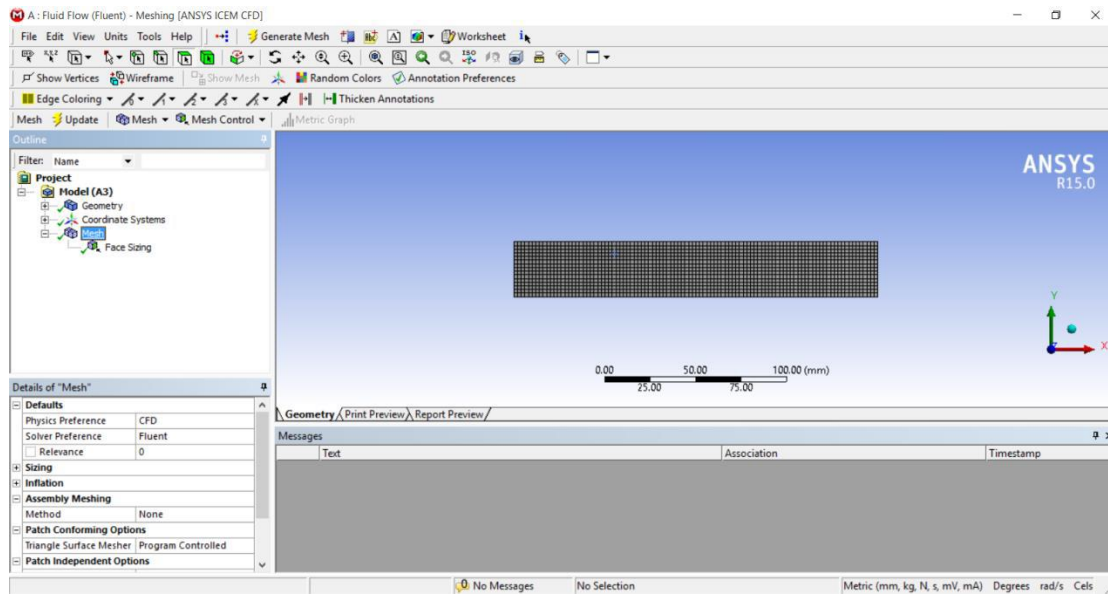
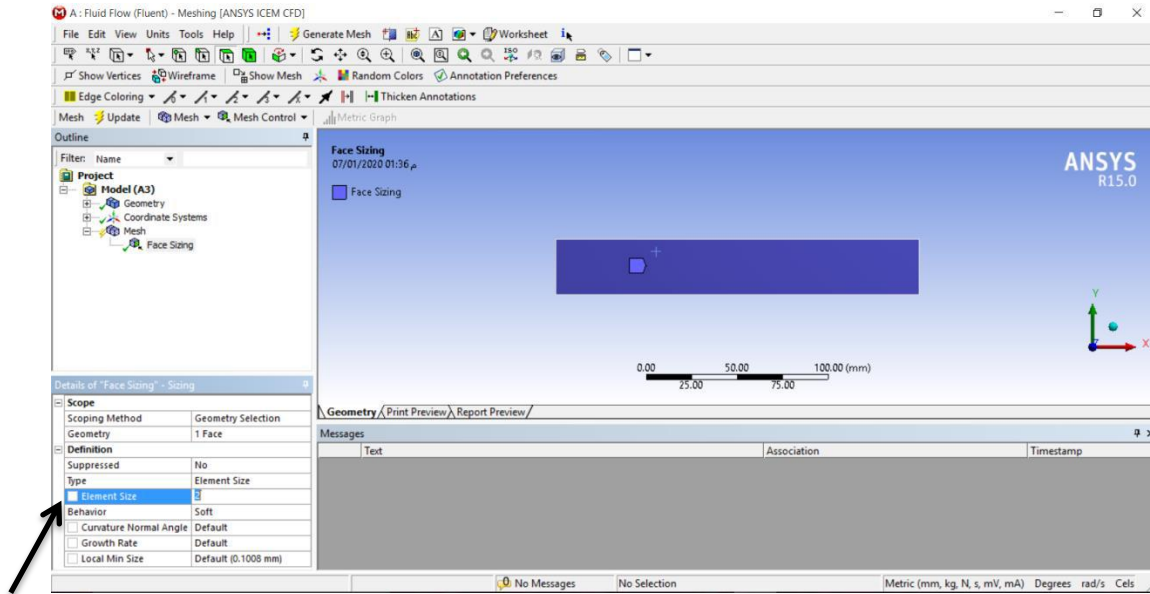


11- Select of geometry mesh (right click) → insert → sizing



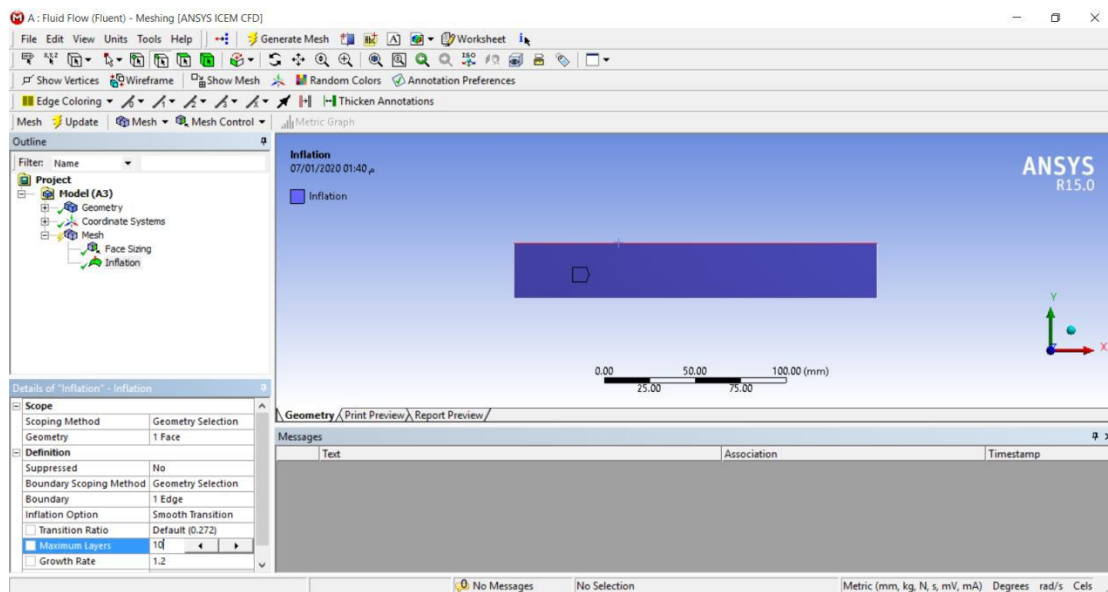
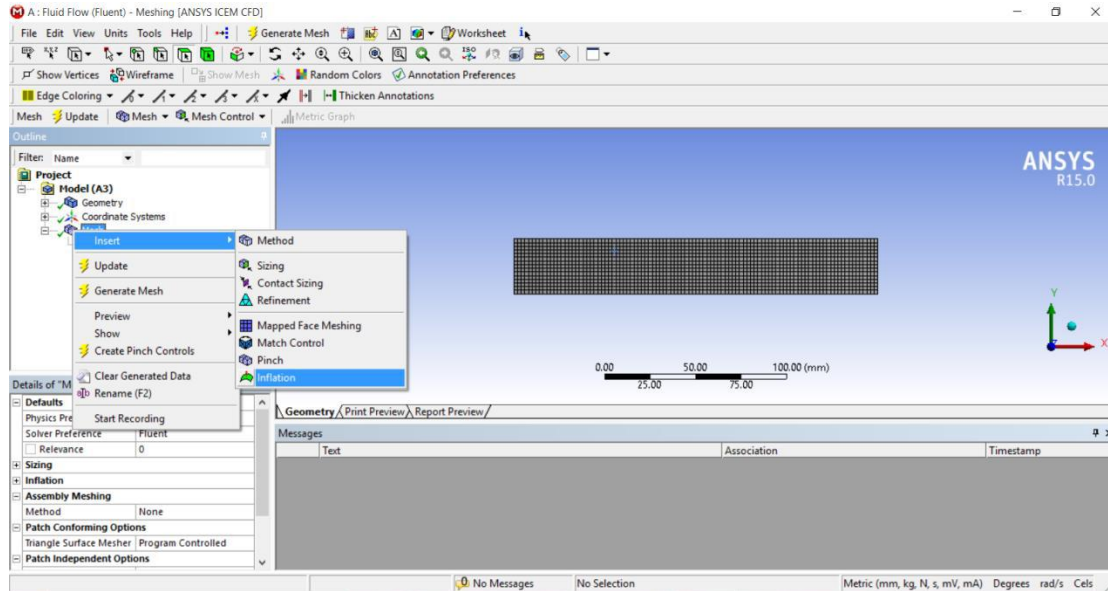


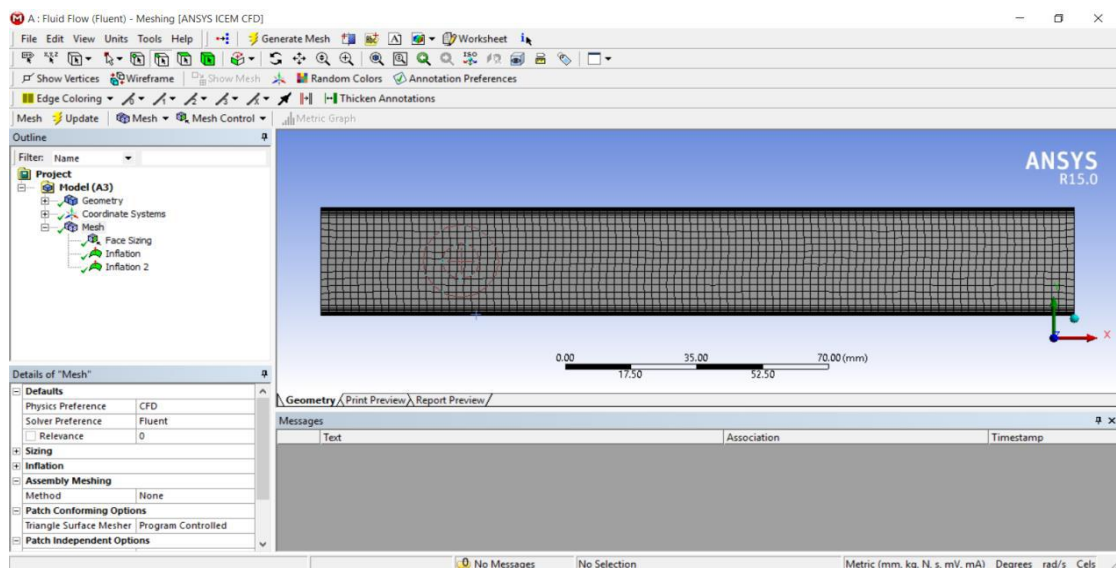
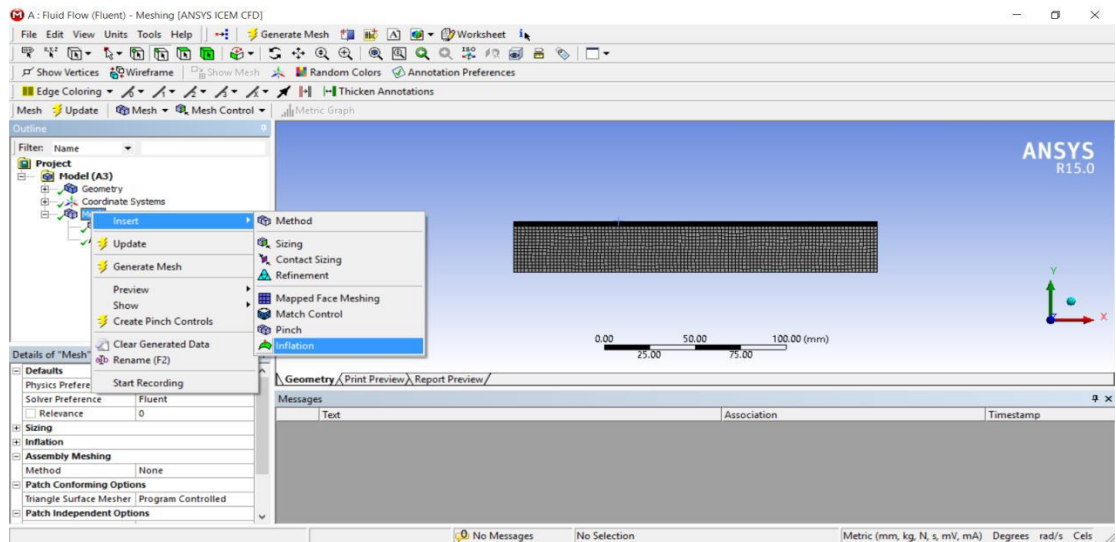
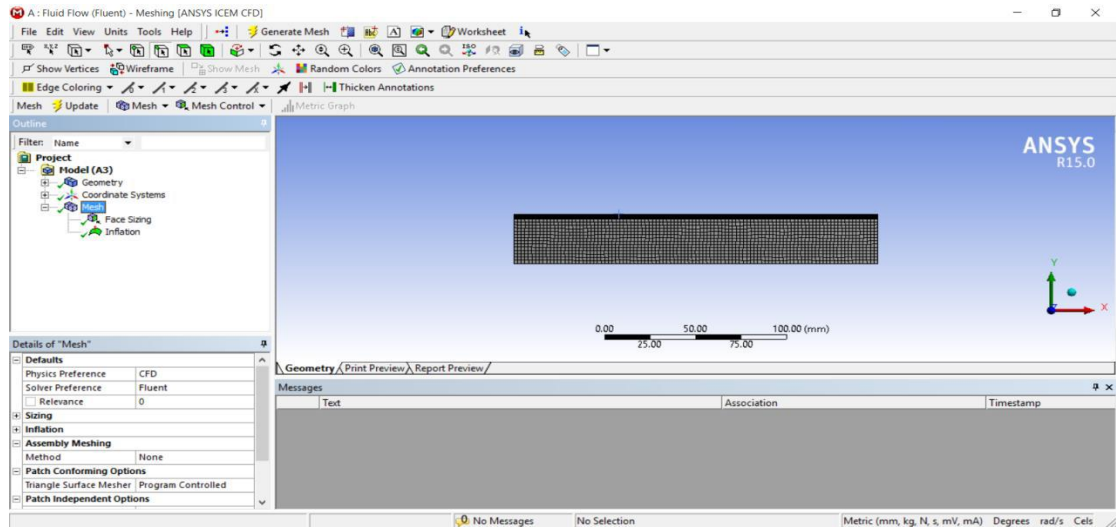
12- Select geometry \longrightarrow apply \longrightarrow element size \longrightarrow generate





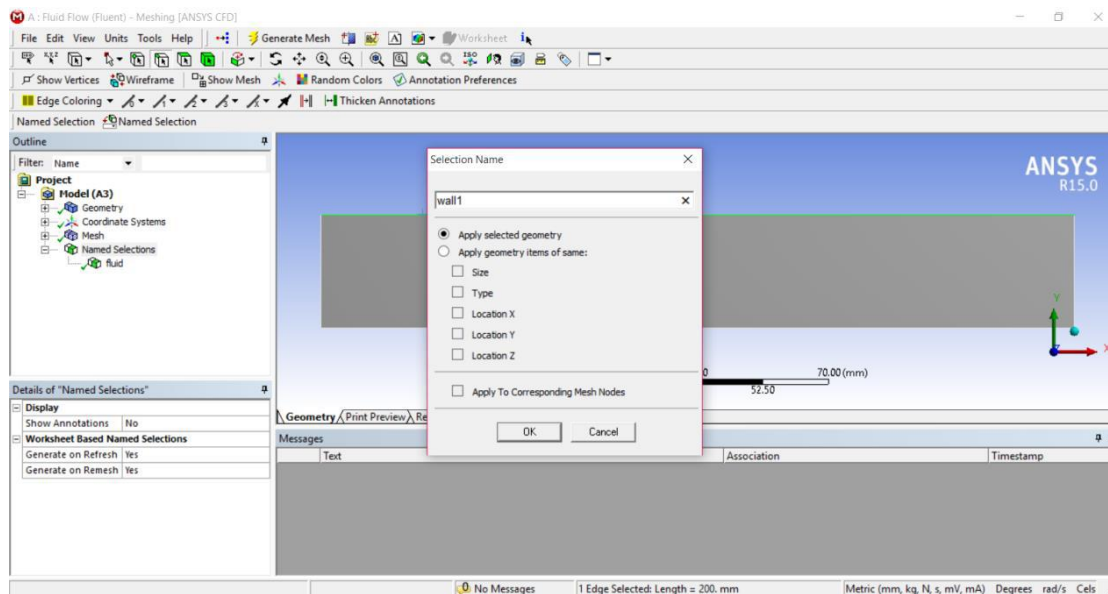
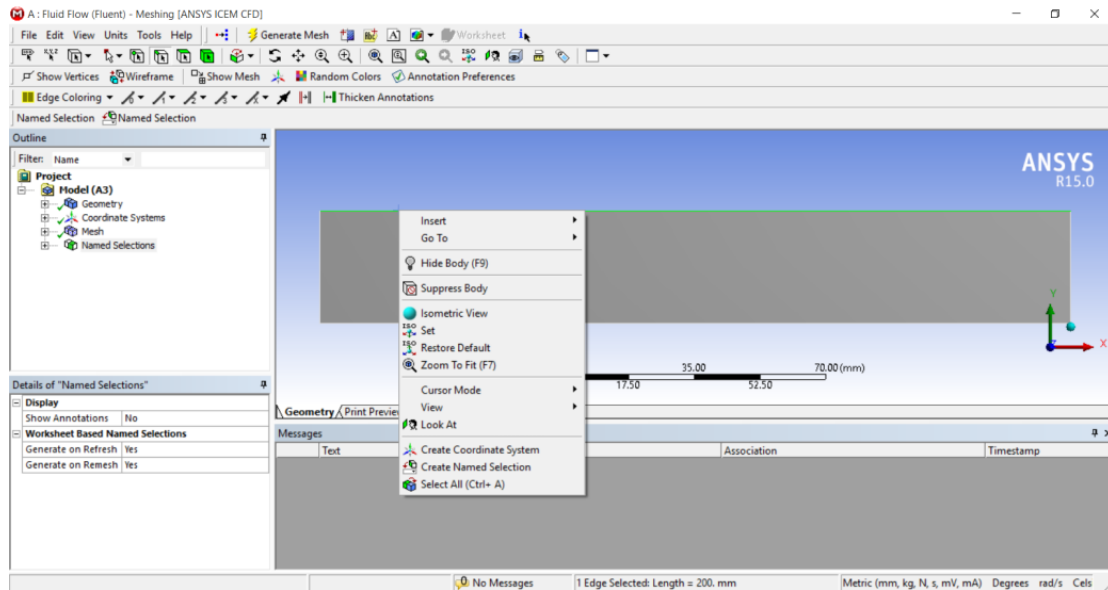
13- Mesh (right click) → insert → inflation → select geometry → apply
→ Select Boundary → apply → Maximum layers → generate (repeat this
steps for all boundaries) → close





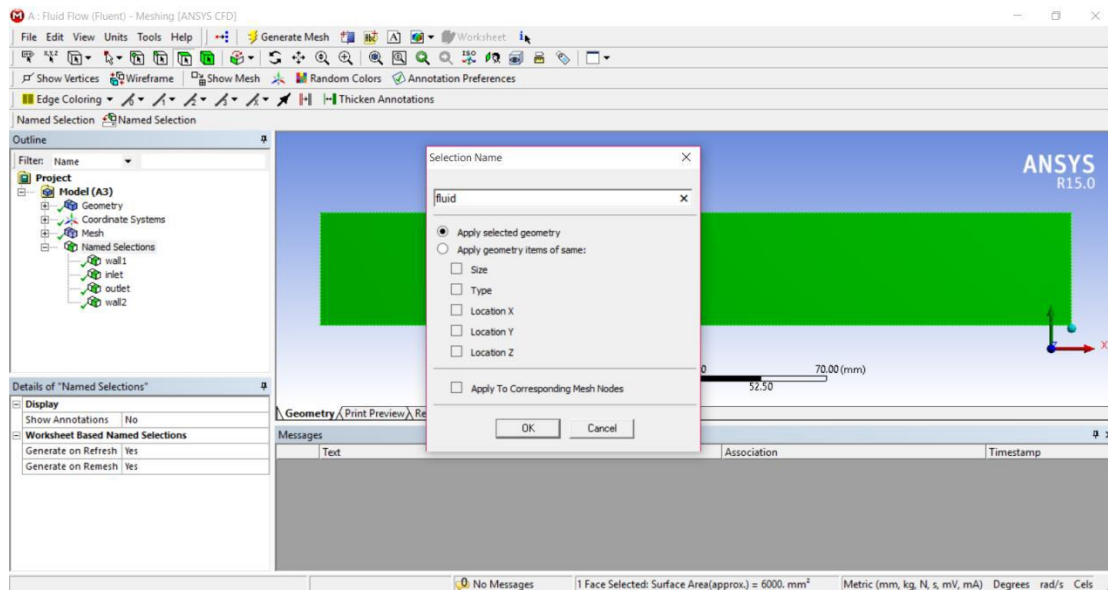
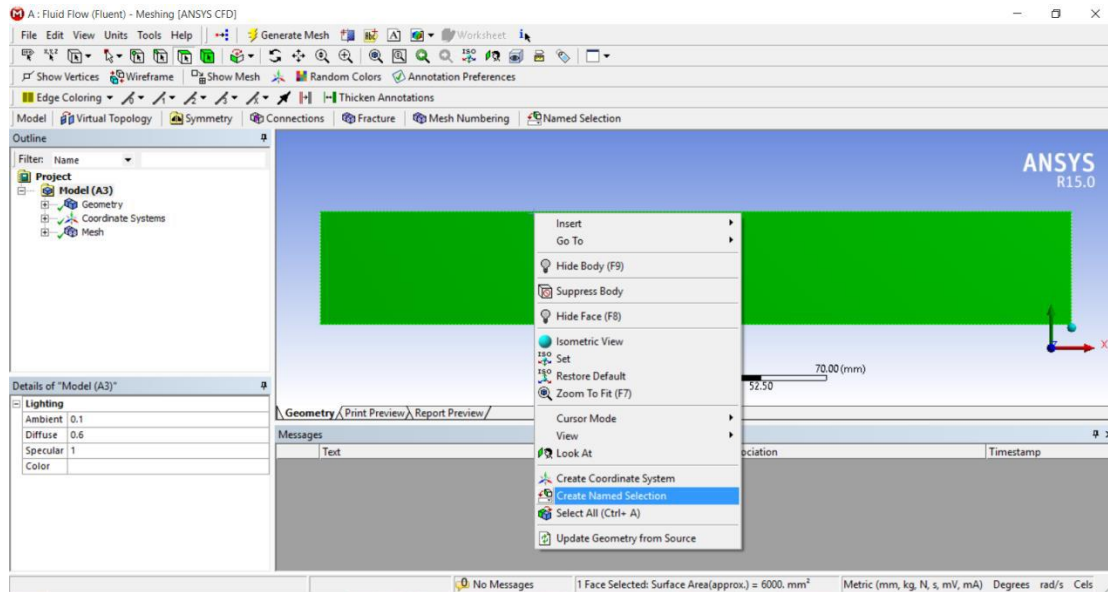


14- Select Boundary (right click) → create named selection → ok



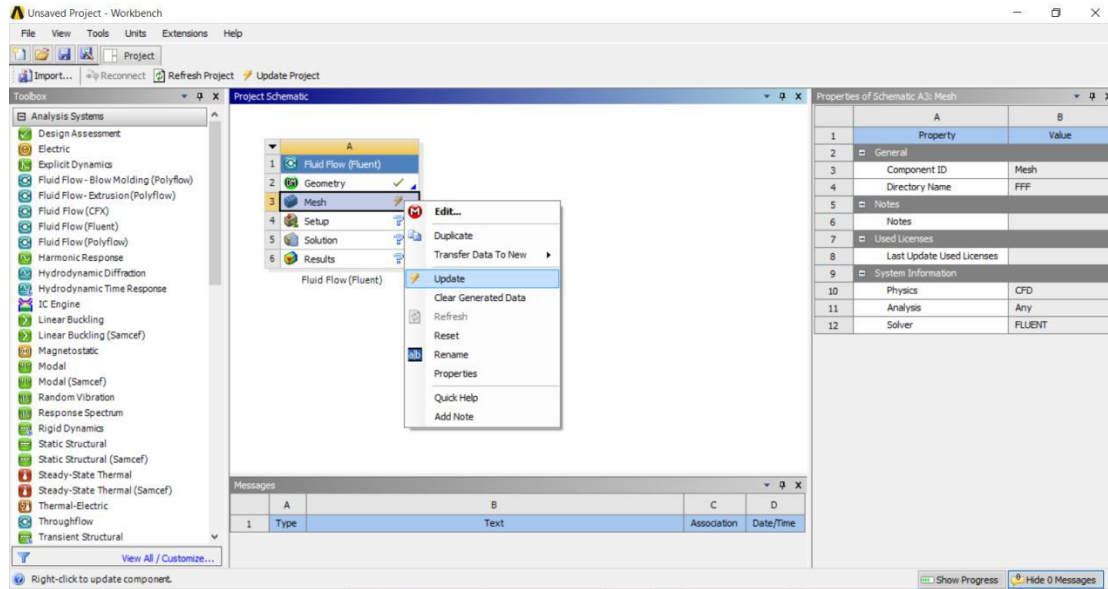


15- Select fluid (right click) → Create named selection → ok

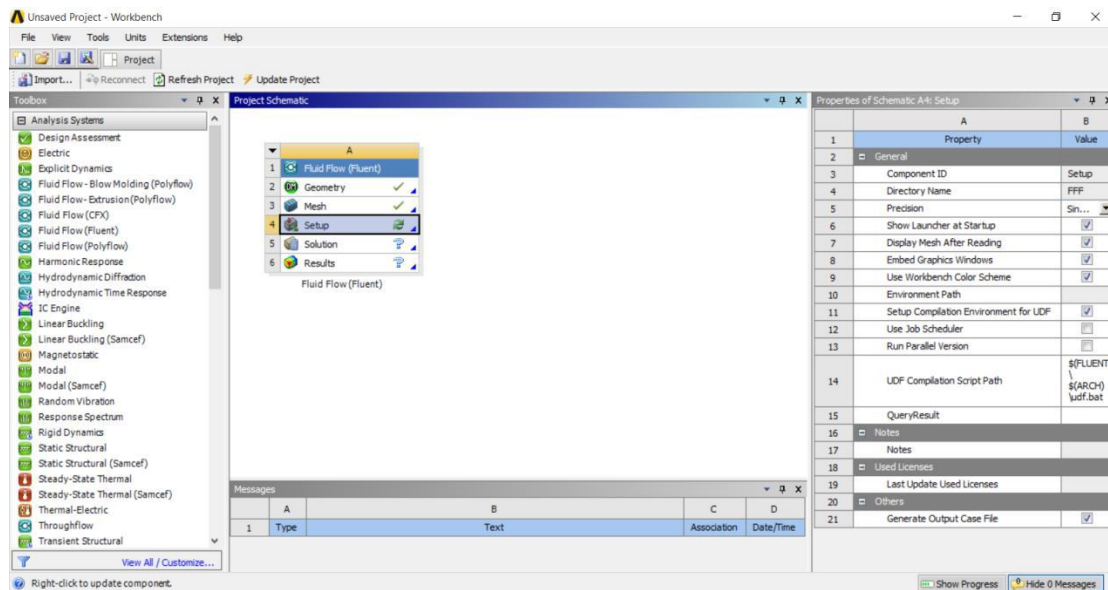




16- Mesh (right click) → update



17- Analysis systems → fluid flow (fluent) → setup





The screenshot displays the ANSYS Workbench environment. The 'Project Schematic' window shows a project named 'Fluid Flow (Fluent)' with steps: 1. Fluid Flow (Fluent), 2. Geometry, 3. Mesh, 4. Setup, 5. Solution, and 6. Results. The 'Fluent Launcher (Setting Edit Only)' dialog box is open, showing options for Dimension (2D/3D), Display Options, Processing Options (Serial/Parallel), and Solver Processes. The 'Schematic A4: Setup' window is also visible, showing a table of properties and values.

Property	Value
Component ID	Setup
Directory Name	FFF
Precision	Sim...
Show Launcher at Startup	<input checked="" type="checkbox"/>
Display Mesh After Reading	<input checked="" type="checkbox"/>
Embed Graphics Windows	<input checked="" type="checkbox"/>
Use Workbench Color Scheme	<input checked="" type="checkbox"/>
Environment Path	
Setup Compilation Environment for UDF	<input checked="" type="checkbox"/>
Use Job Scheduler	<input type="checkbox"/>
Run Parallel Version	<input type="checkbox"/>
UDF Compilation Script Path	\$(FLUENT)\\$(ARCH)\udf.bat
Query/Result	