Computational Fluid Dynamics

2019 autumn, 1st week

- Tamás Benedek
- benedek [at] ara.bme.hu
- www.ara.bme.hu/~benedek/CFD/icem

The most important rule:

Dont use space or specific characters in:

File names, path, part names, etc. (nowhere)

- Working directory: C:\Work → create your own, for example: Work\quentin_tarantino (if you are Quentin Tarantino)

<u>Another important rule:</u>

Your work will be deleted, if you turn off the computer. → When you finished, save your work on a flash drive or send it to yourself attached to an e-mail

Instructors of ICEM courses:

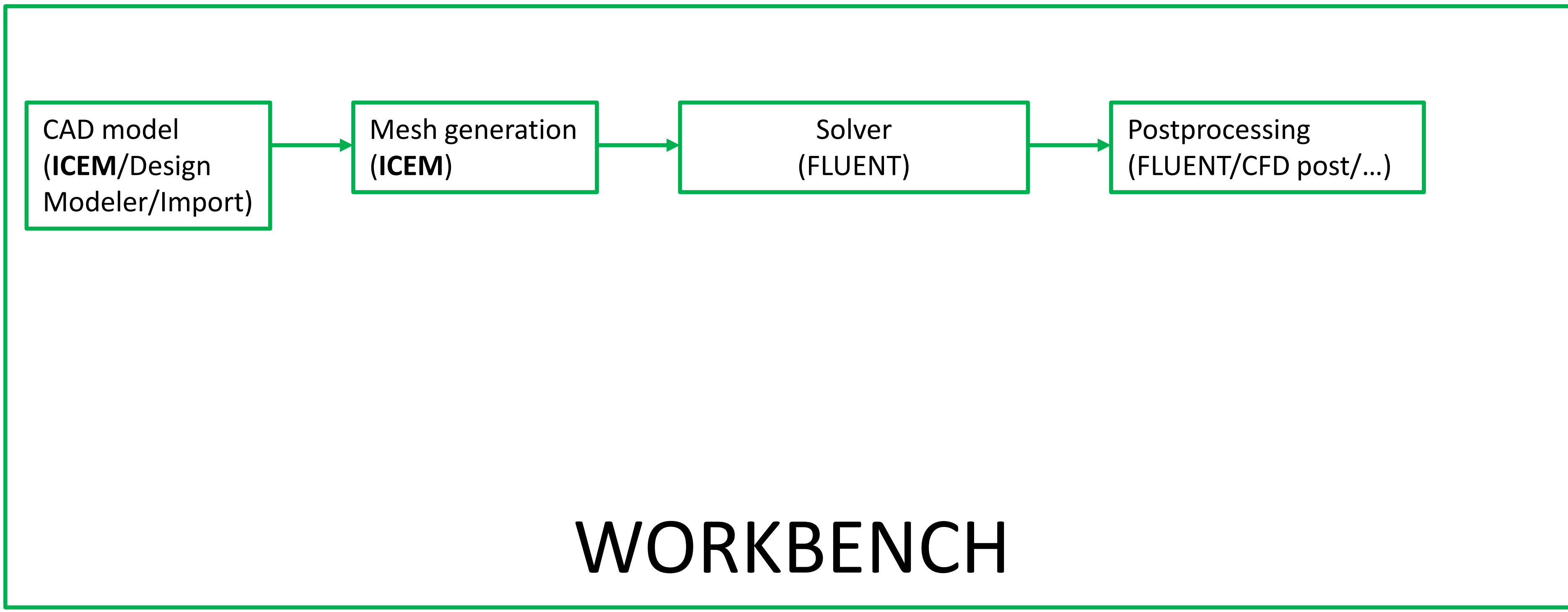
- Dr. Tamás Benedek (benedek [at] ara.bme.hu)

- Kristóf Tokaji (tokaji [at] ara.bme.hu)
- András Tomor (tomor [at] ara.bme.hu)

Agenda

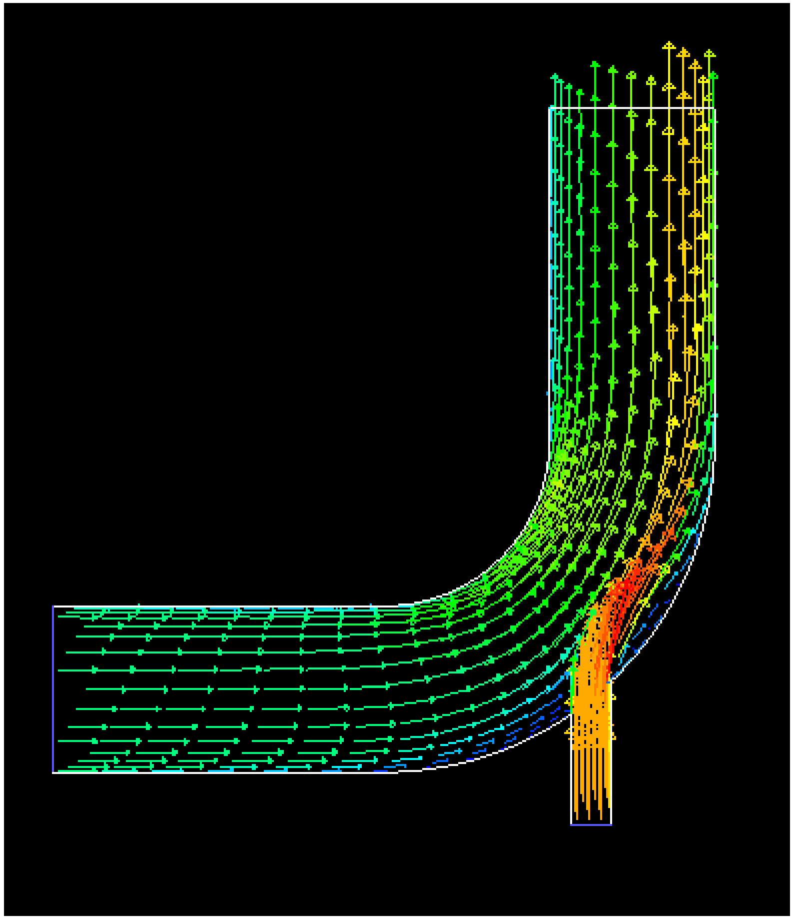
- Week1-5: ICEM and FLUENT practice
- Week6-8: Individual project
- Week9-11: Group project
- Week12-13: Tutorials of specific problems
- Week14: Presentation of the group project,

Solving a problem with CFD

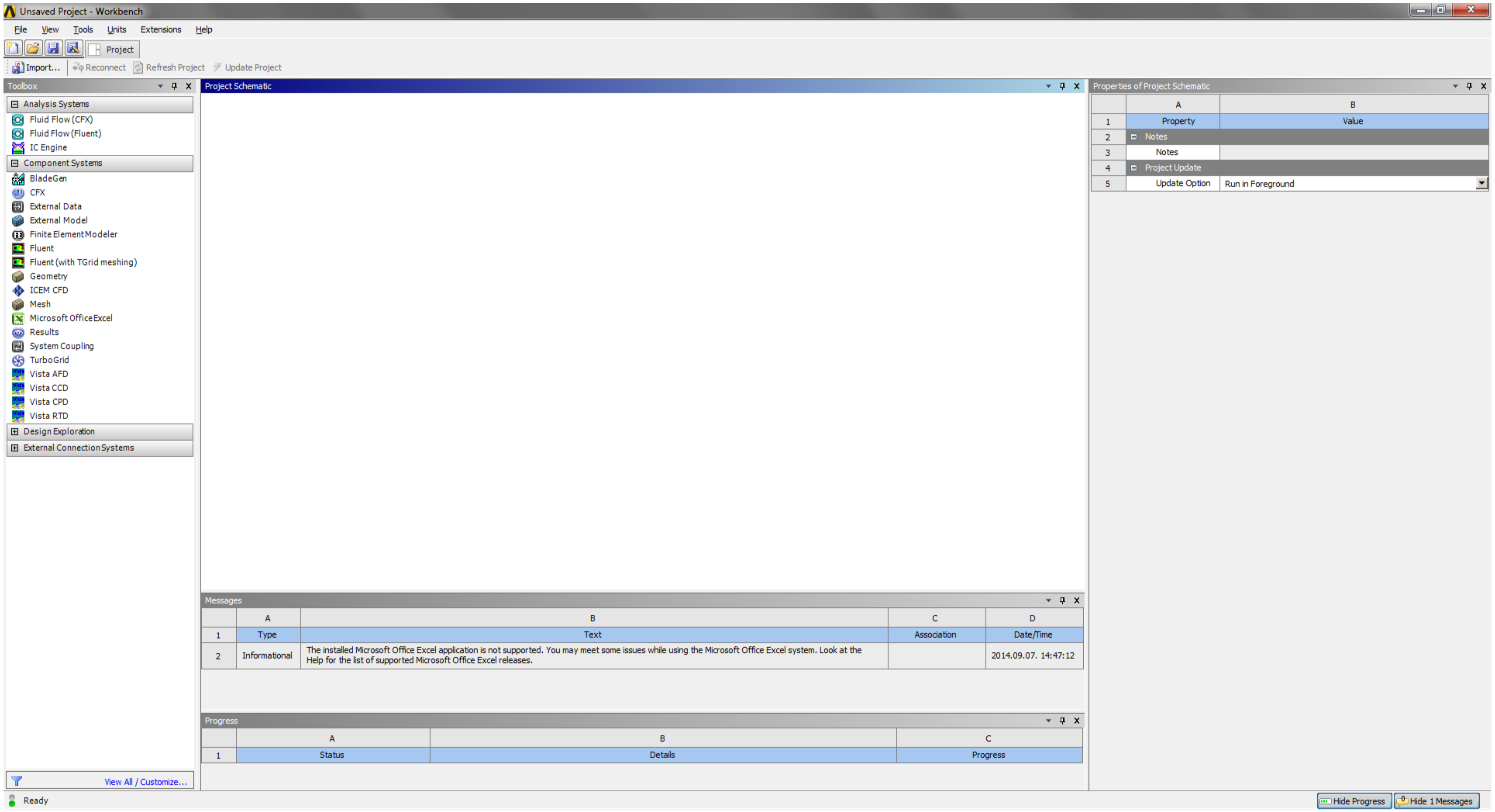


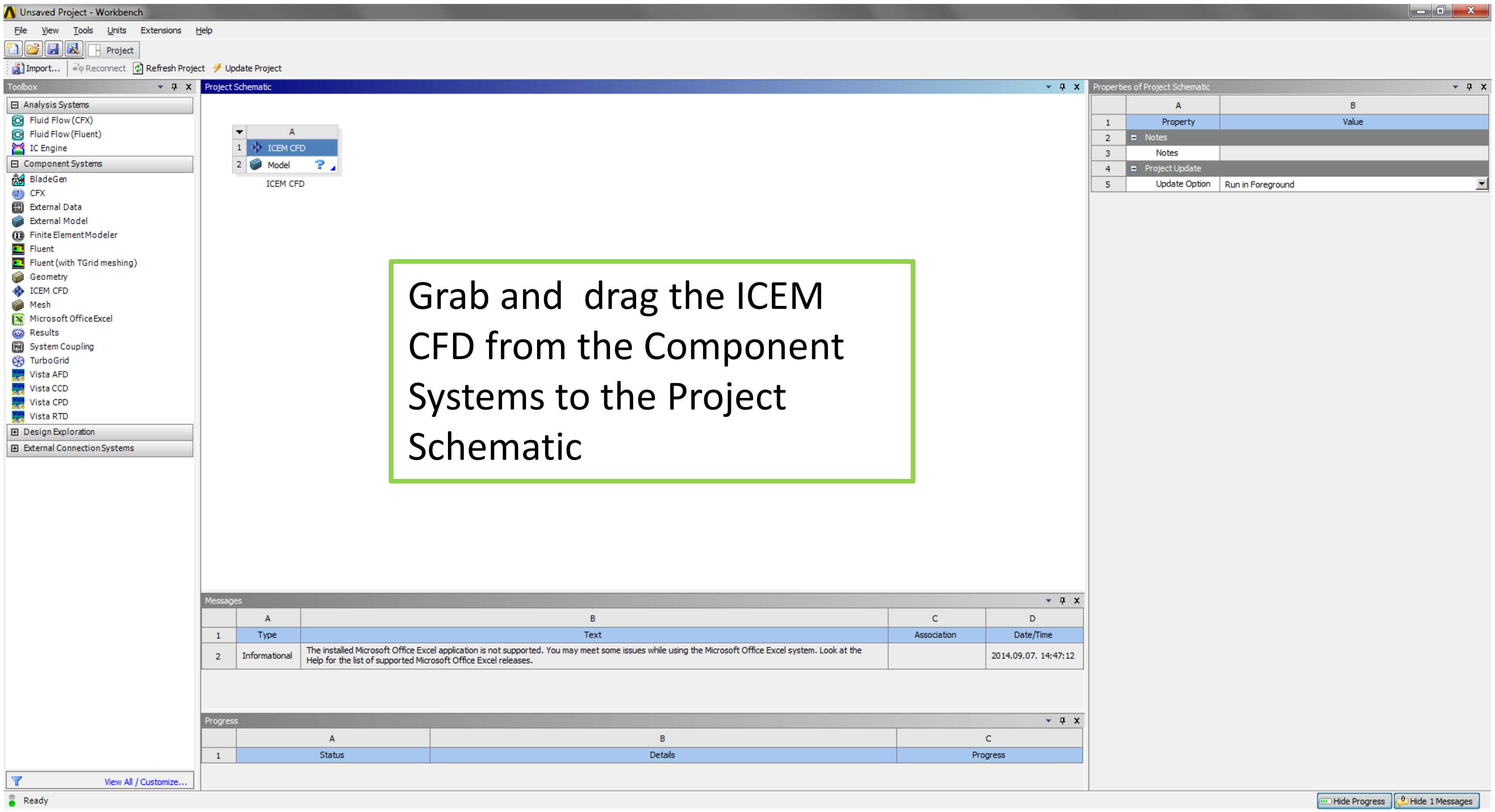
Goal of the present practice

- Make your first mesh in ICEM through the sample of a 2D pipe

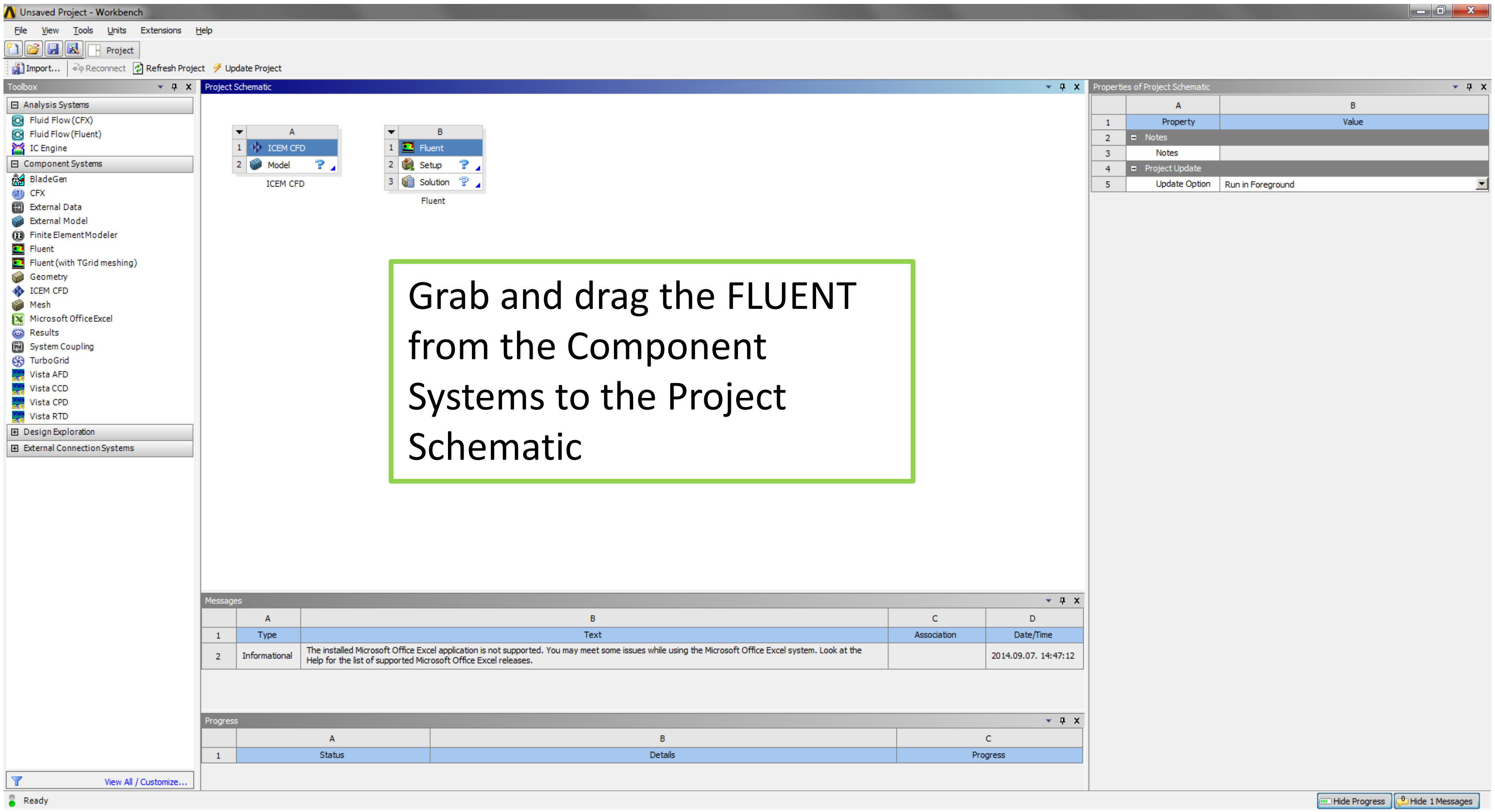


Start the Ansys Workbench

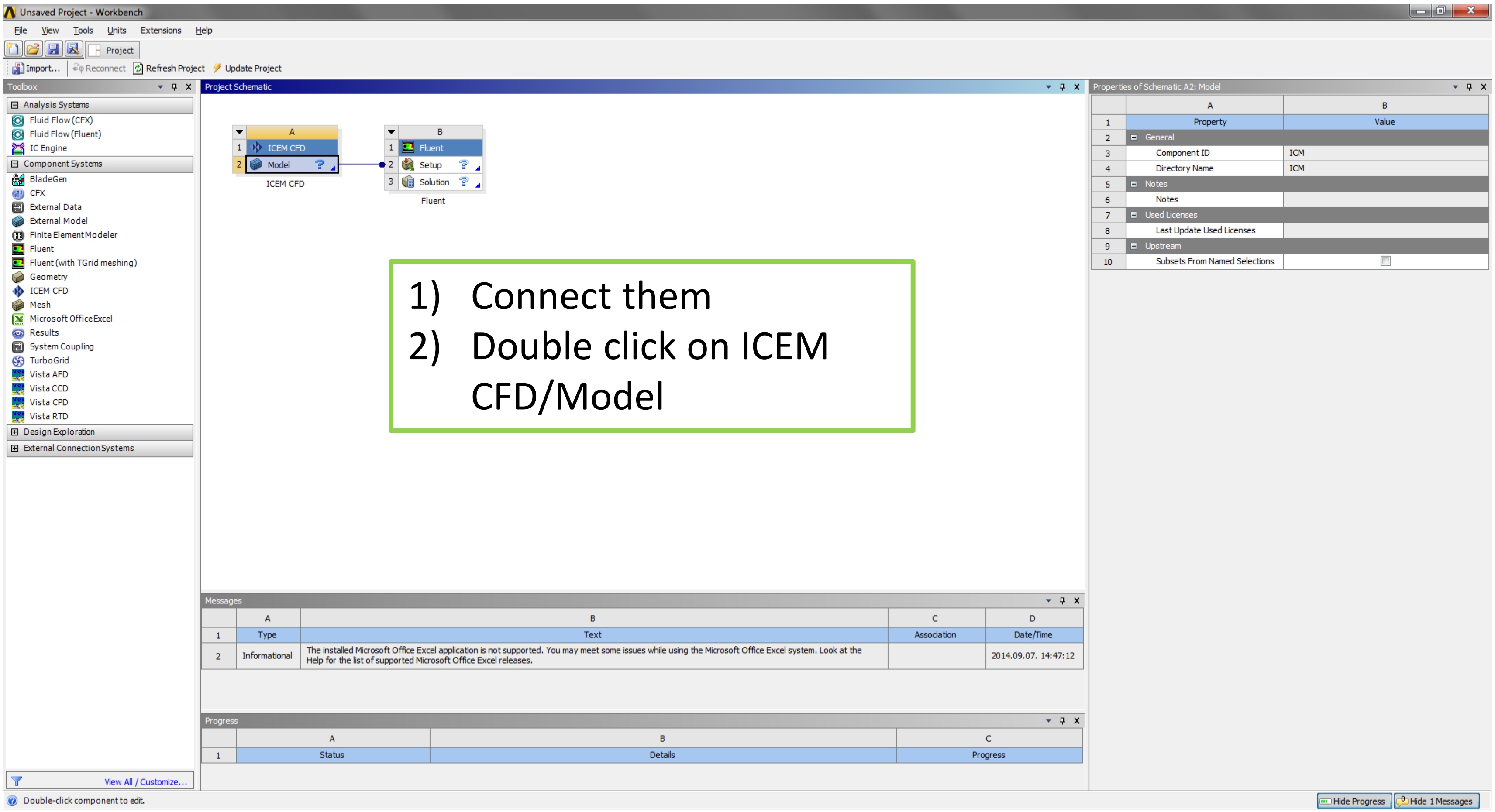




Grab and drag the ICEM CFD from the Component Systems to the Project Schematic



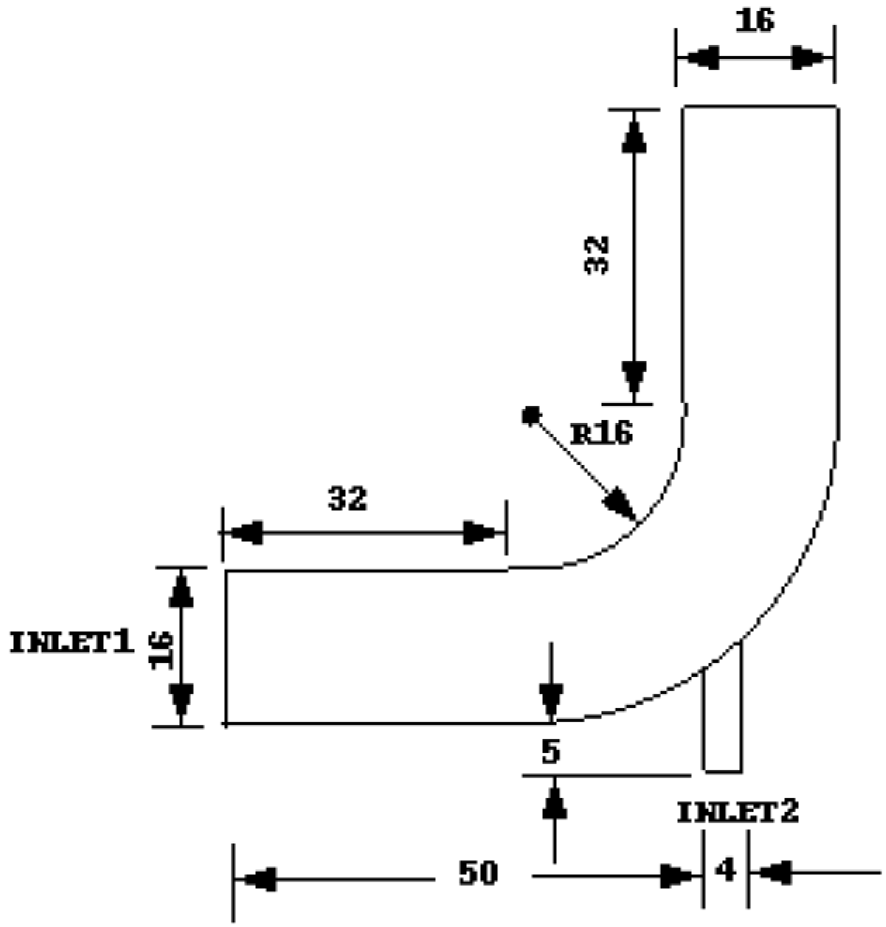
Grab and drag the FLUENT from the Component Systems to the Project Schematic



1) Connect them
2) Double click on ICEM CFD/Model

Geometry

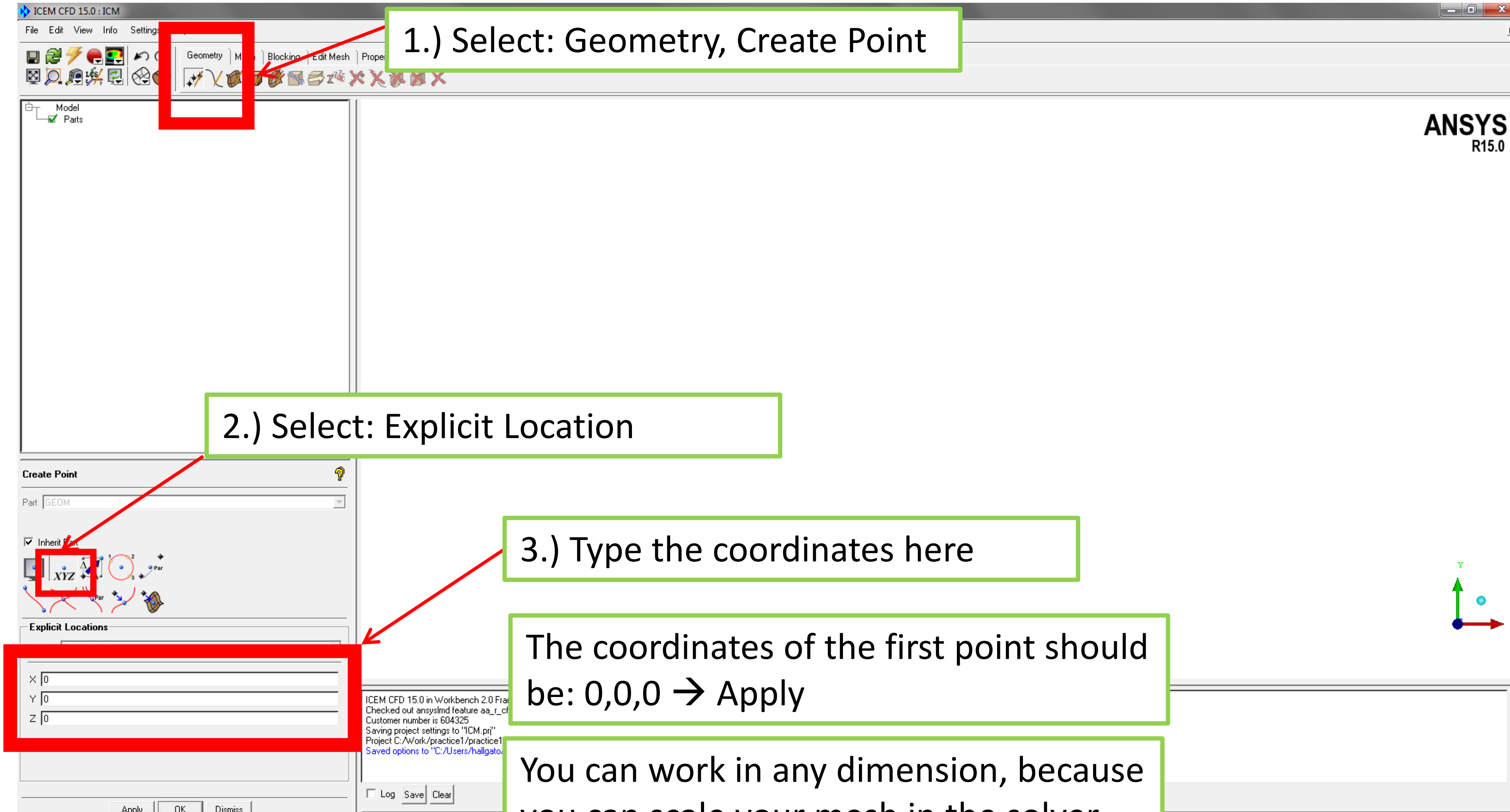
The original geometry (in cm-s)



Settings

- Settings/Selction/Auto pick mode: turn ON

Point creation



1.) Select: Geometry, Create Point

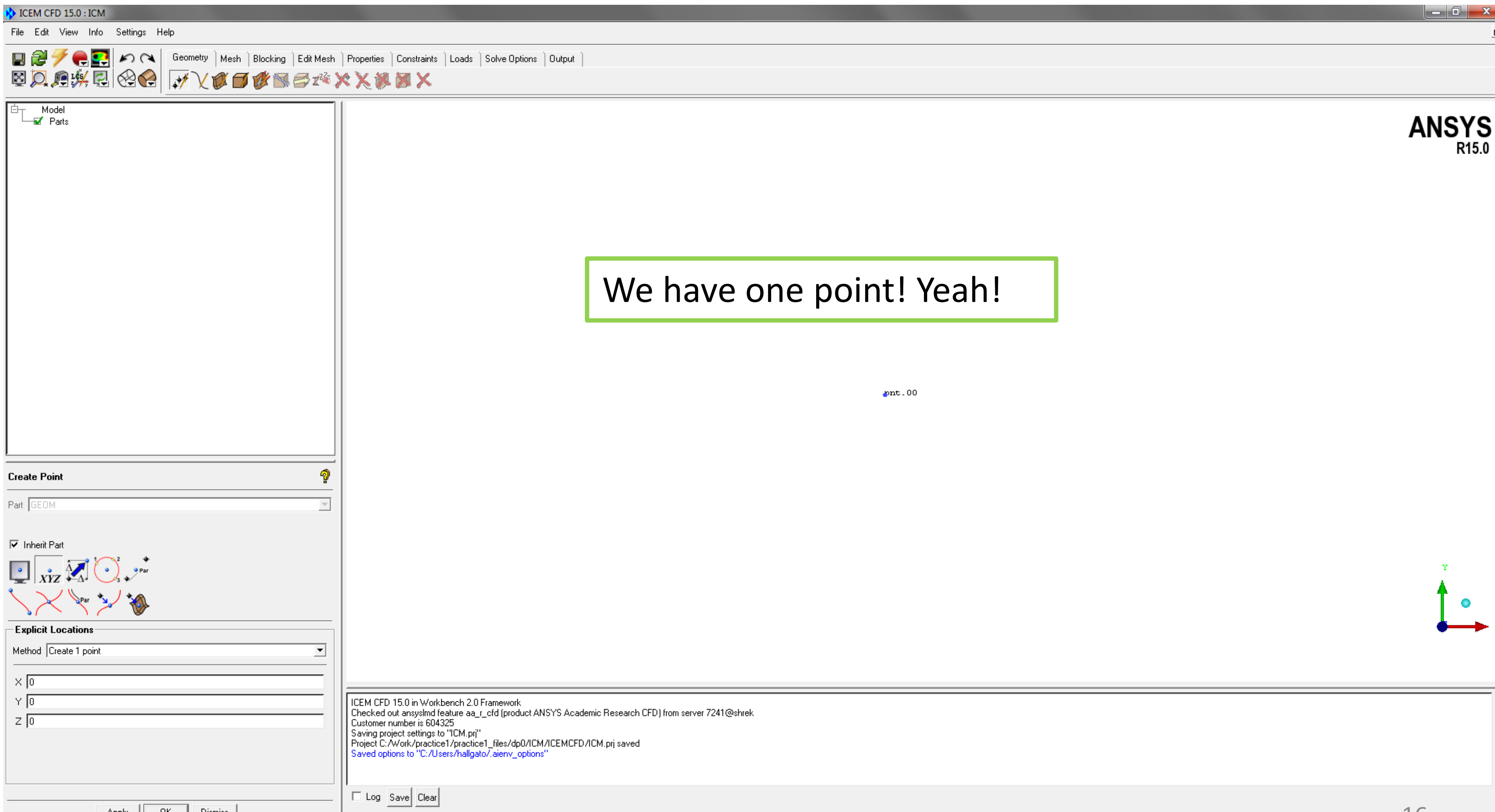
2.) Select: Explicit Location

3.) Type the coordinates here

The coordinates of the first point should be: 0,0,0 → Apply

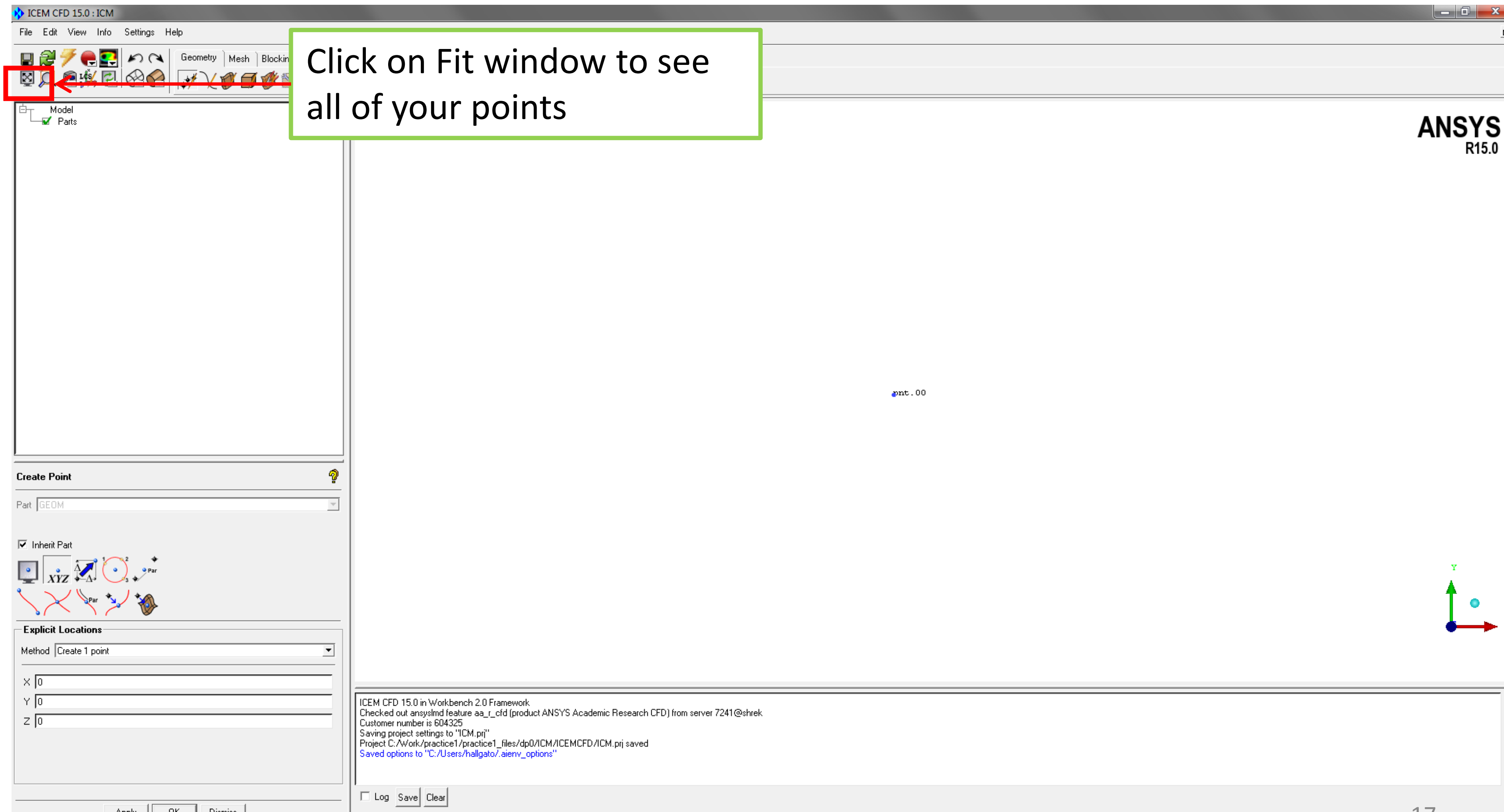
You can work in any dimension, because you can scale your mesh in the solver

Point creation

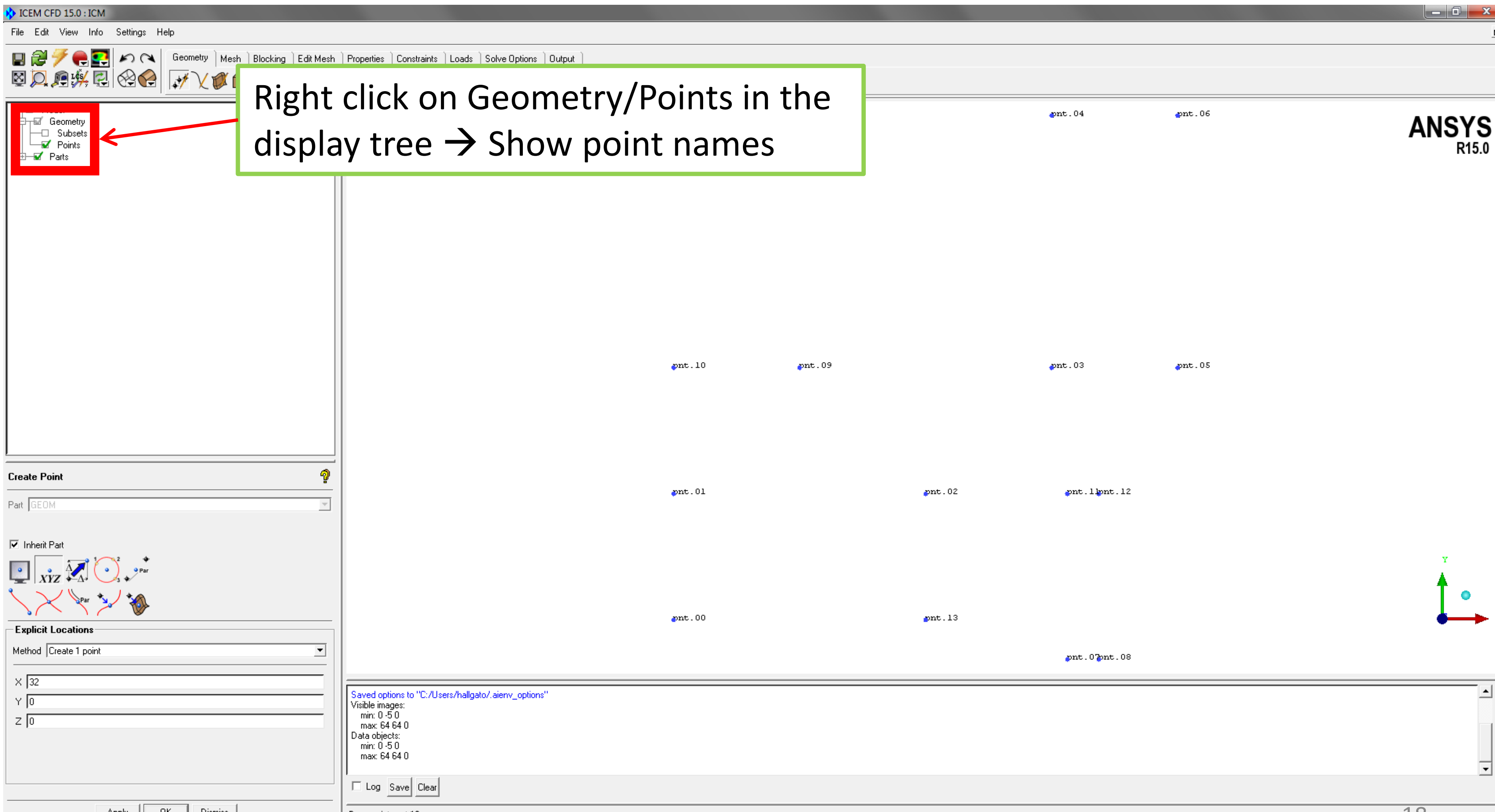


Point creation

Click on Fit window to see all of your points



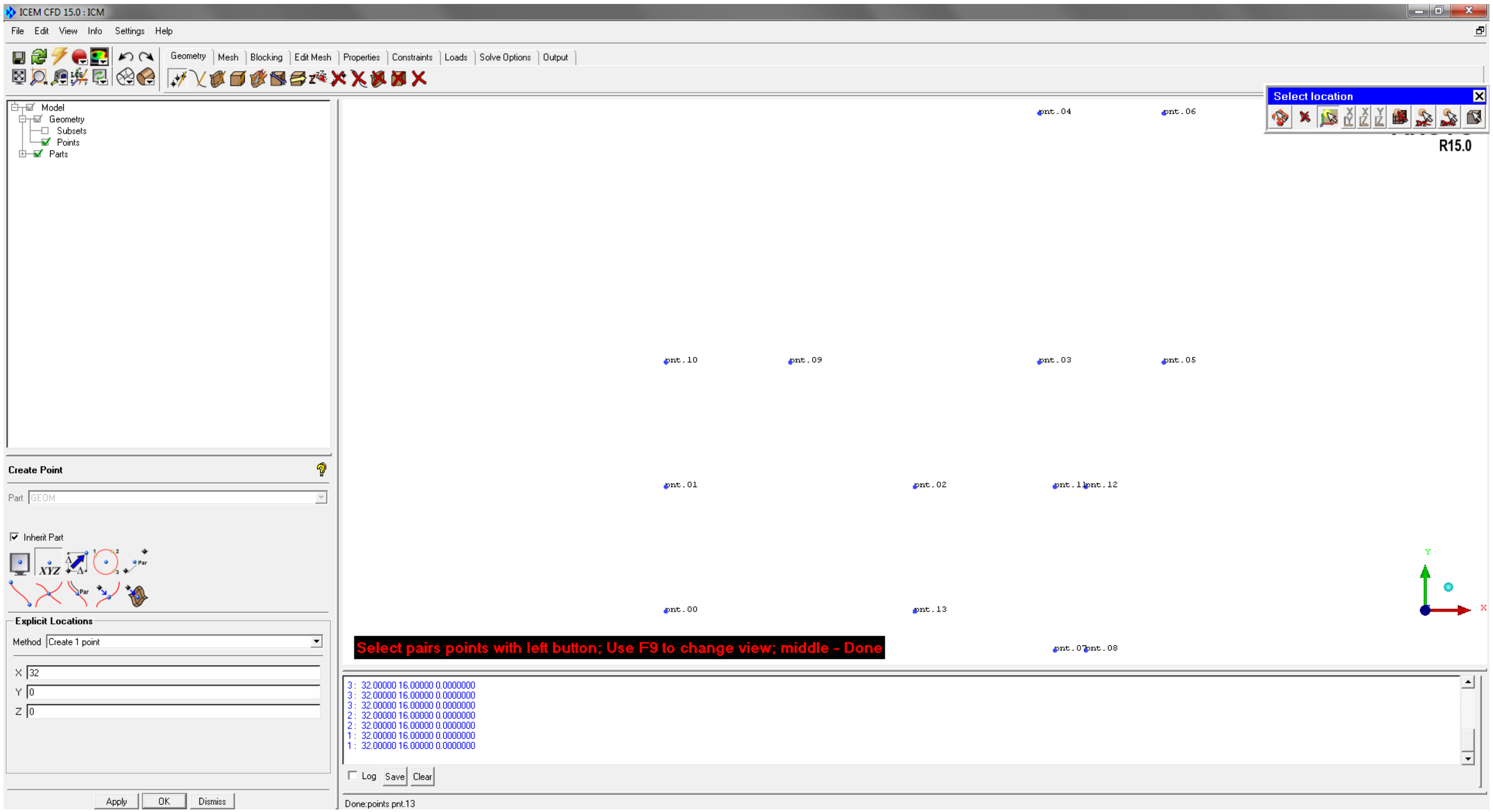
Point creation



Point creation – the coordinates of the points

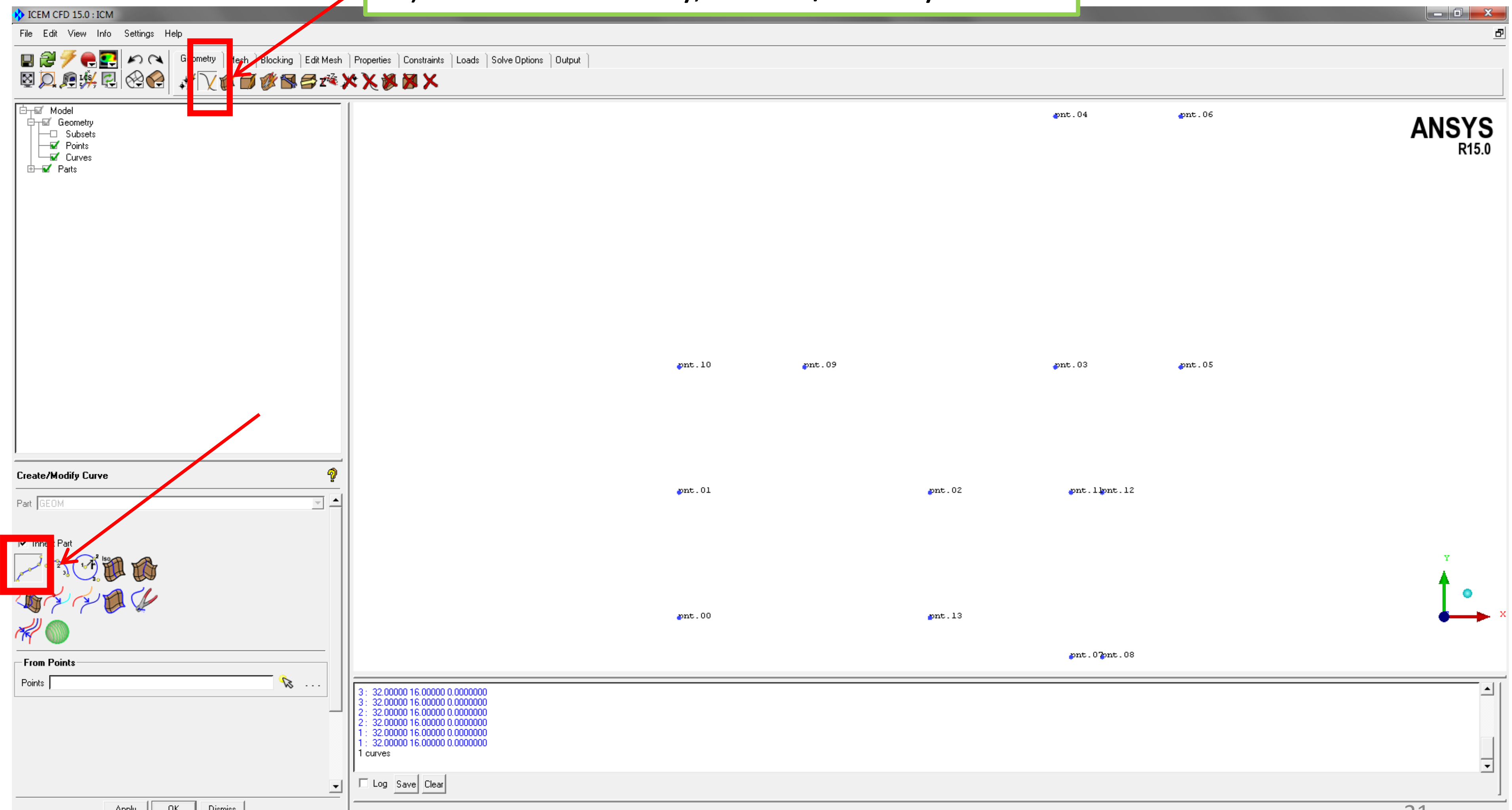
Nr.	Coordinates (x,y,z)
0	0,0,0
1	32,0,0
2	32,16,0
3	48,32,0
4	48,64,0
5	64,32,0
6	64,64,0
7	50,-5,0
8	54,-5,0
9	16,32,0
10	0,32,0
11	50,16,0
12	54,16,0
13	0,16,0

Point creation



Curve creation (straight line)

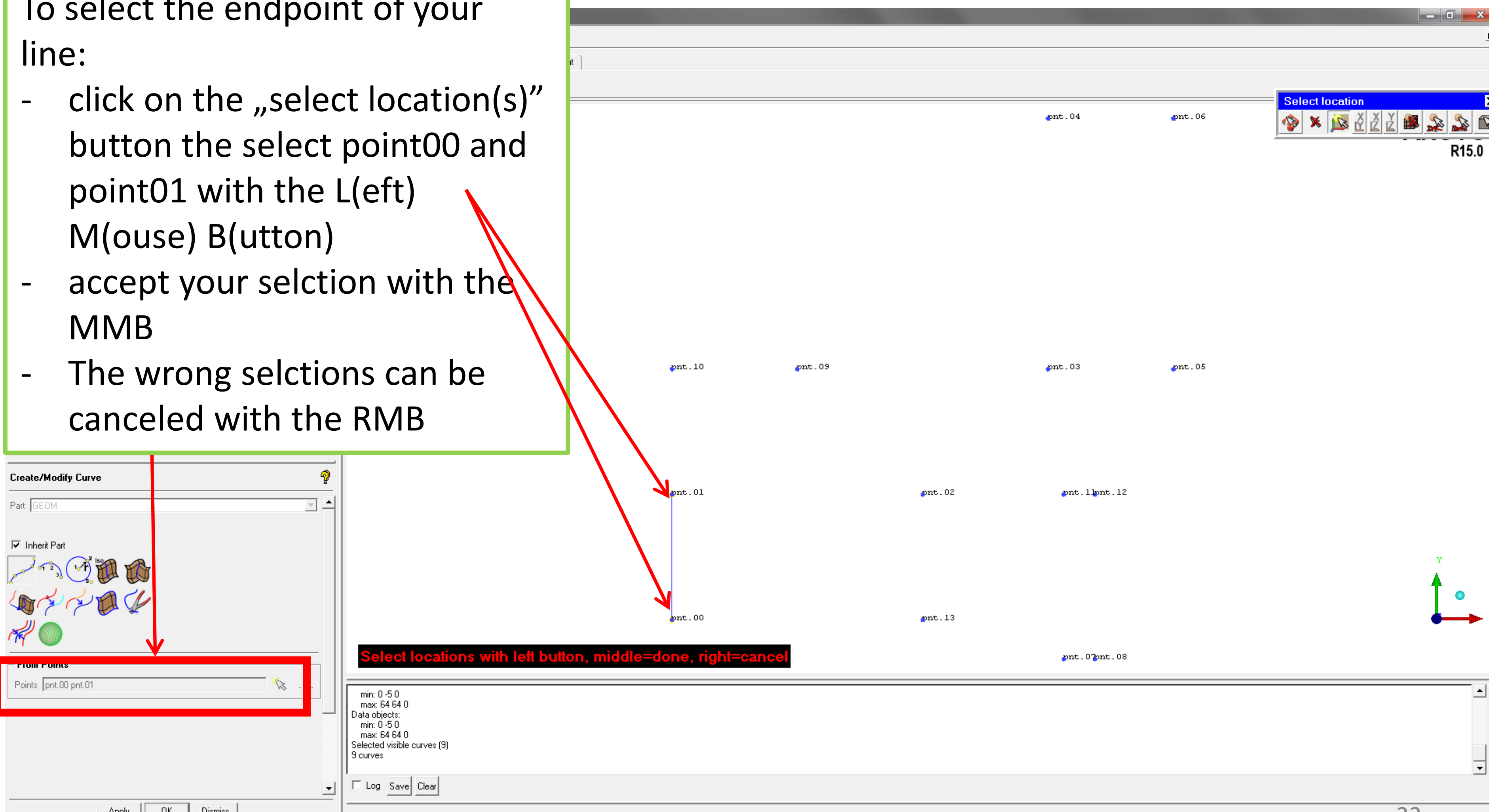
1.) Select: Geometry, Create/Modify Curve



Curve creation (straight line)

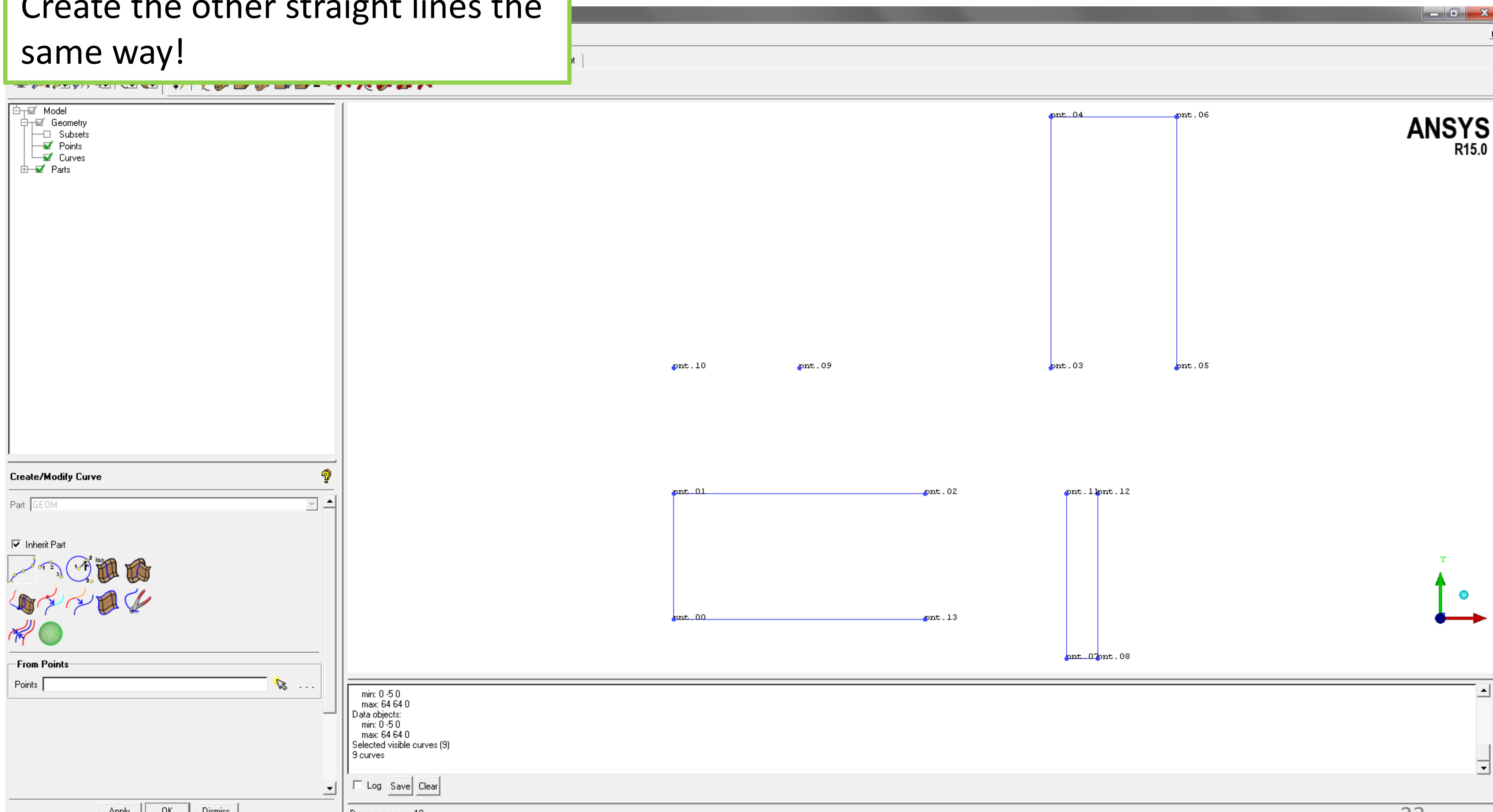
To select the endpoint of your line:

- click on the „select location(s)" button the select point00 and point01 with the L(eft) M(ouse) B(utton)
- accept your selction with the MMB
- The wrong selctions can be canceled with the RMB

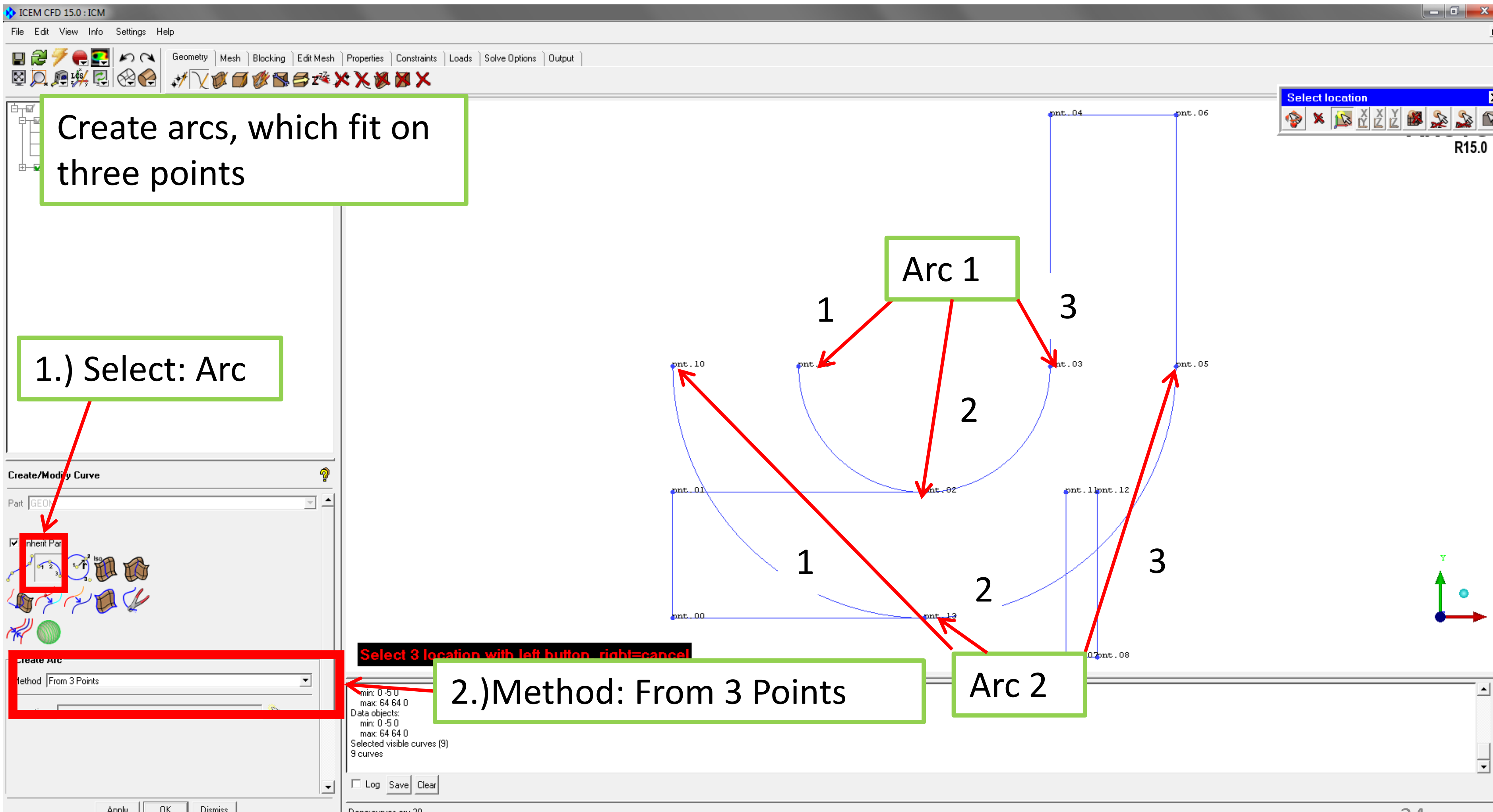


Curve creation (straight lines)

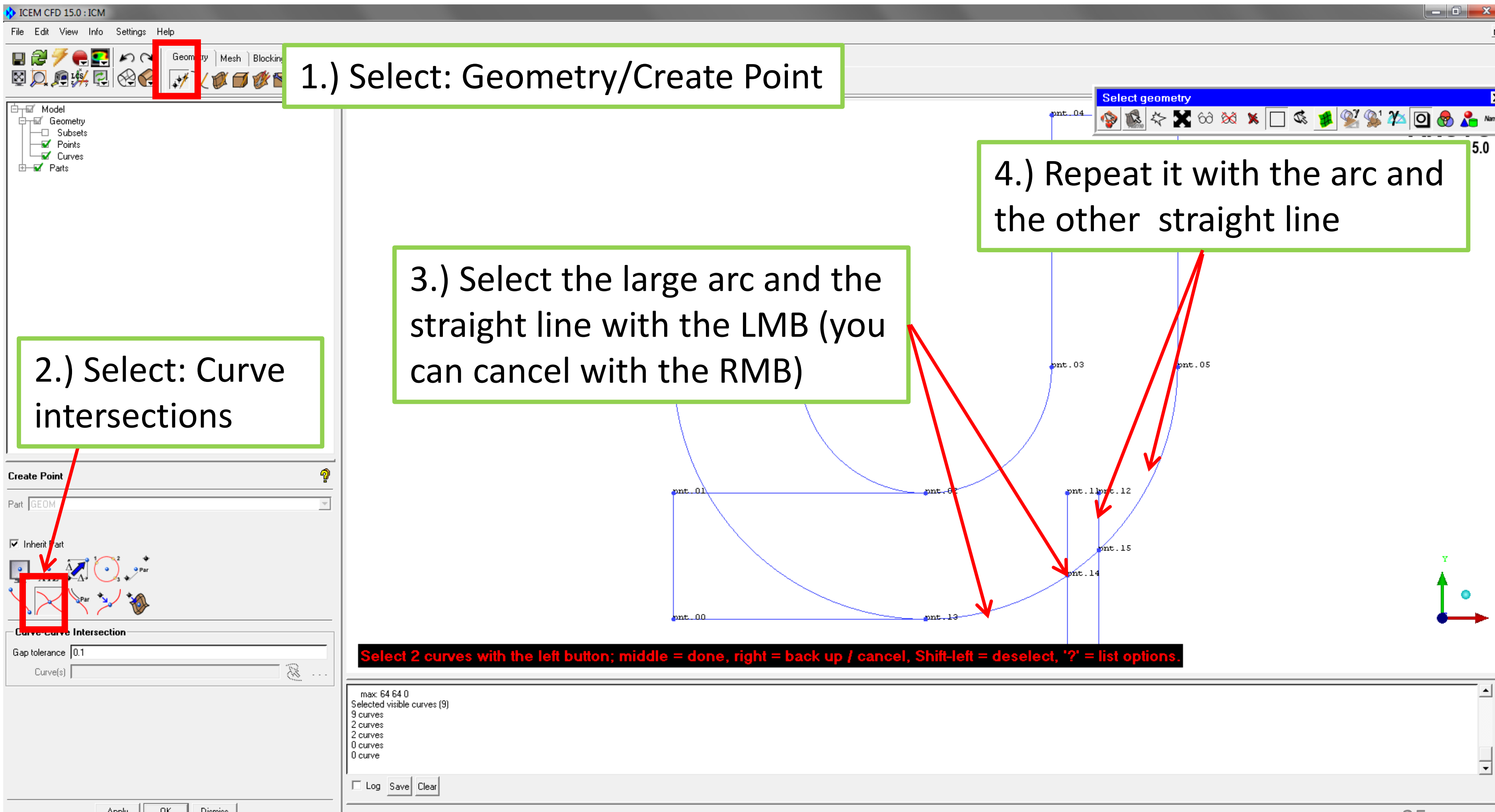
Create the other straight lines the same way!



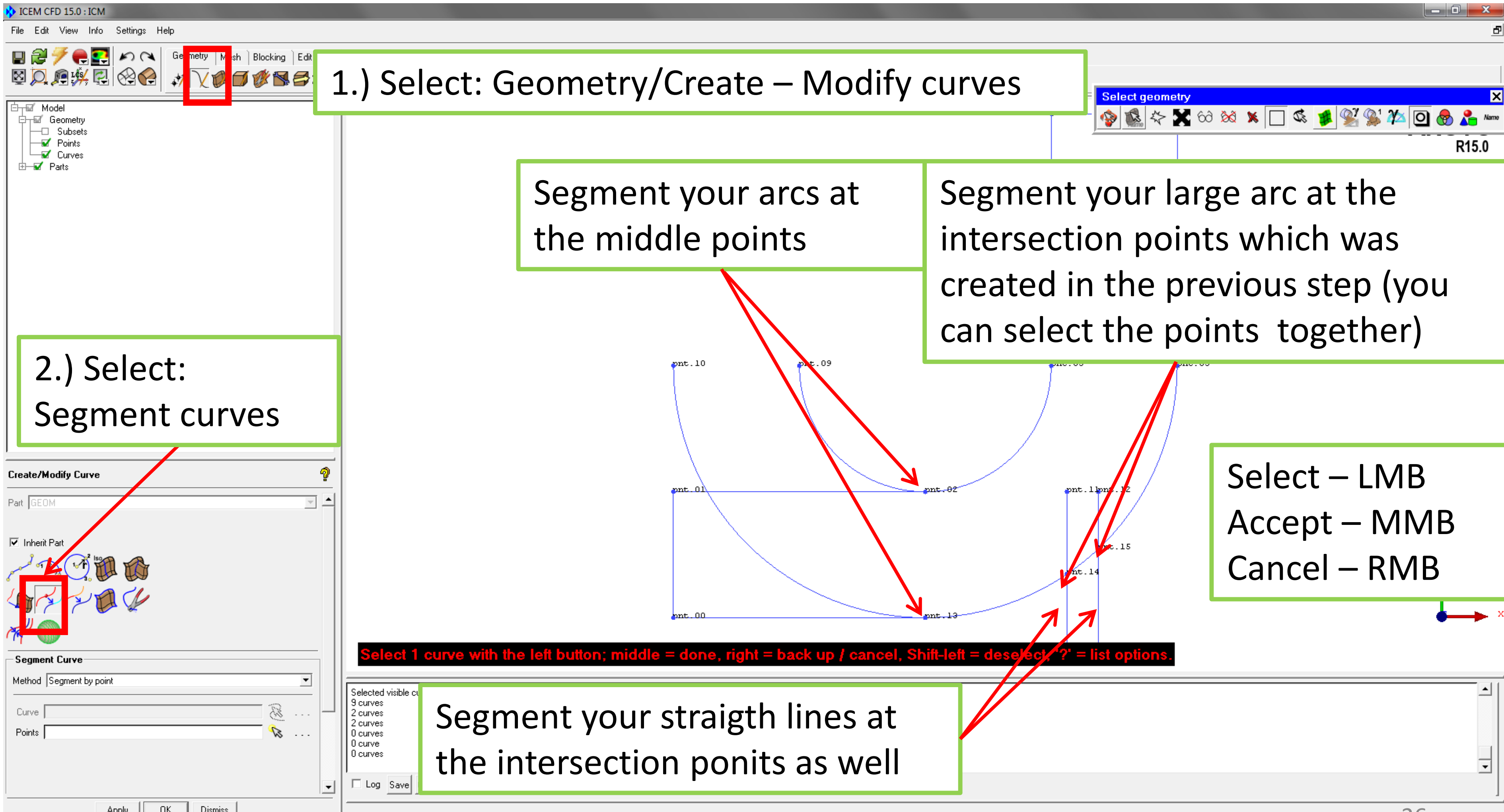
Curve creation (Arcs)



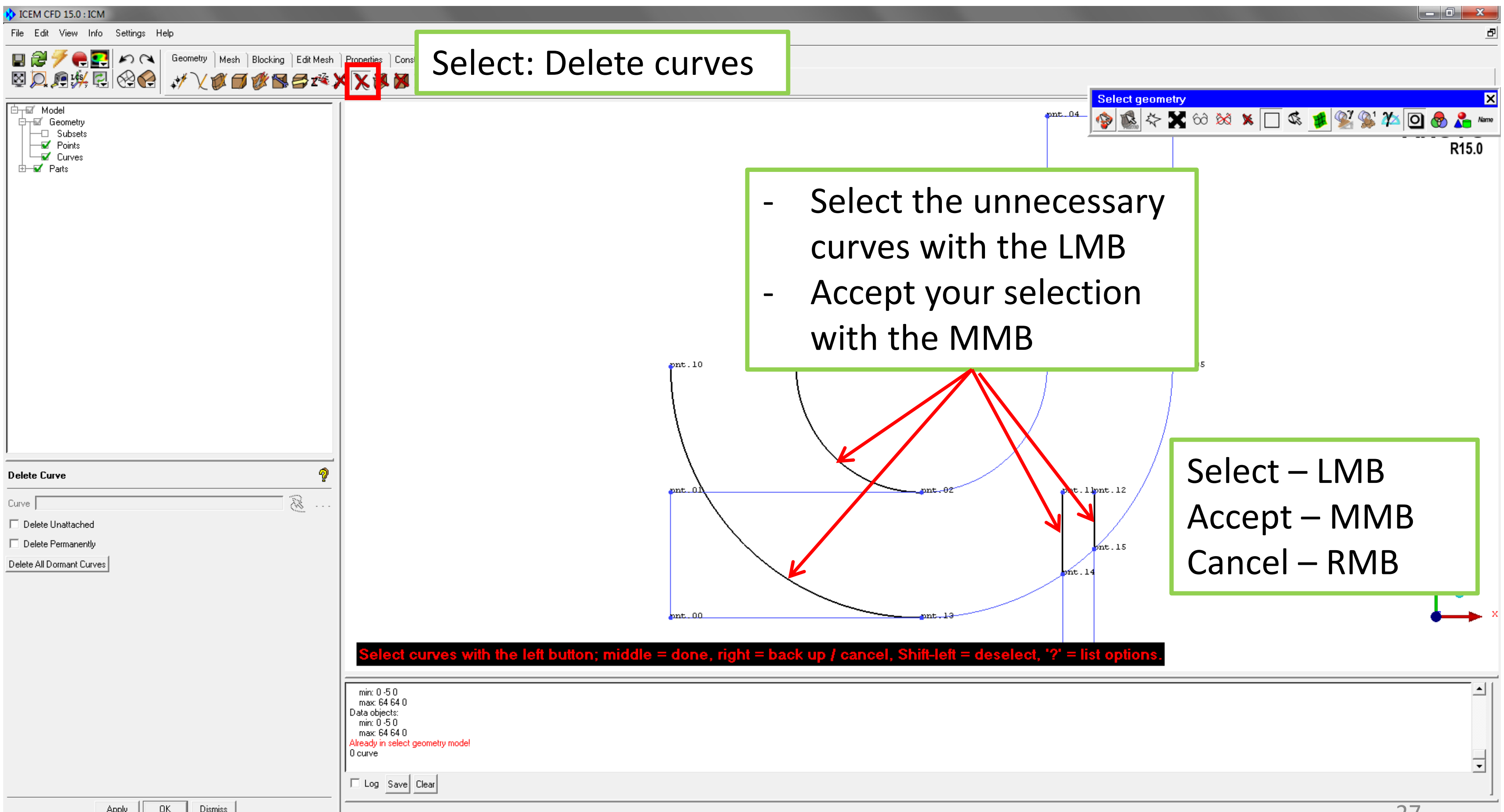
Create intersection points



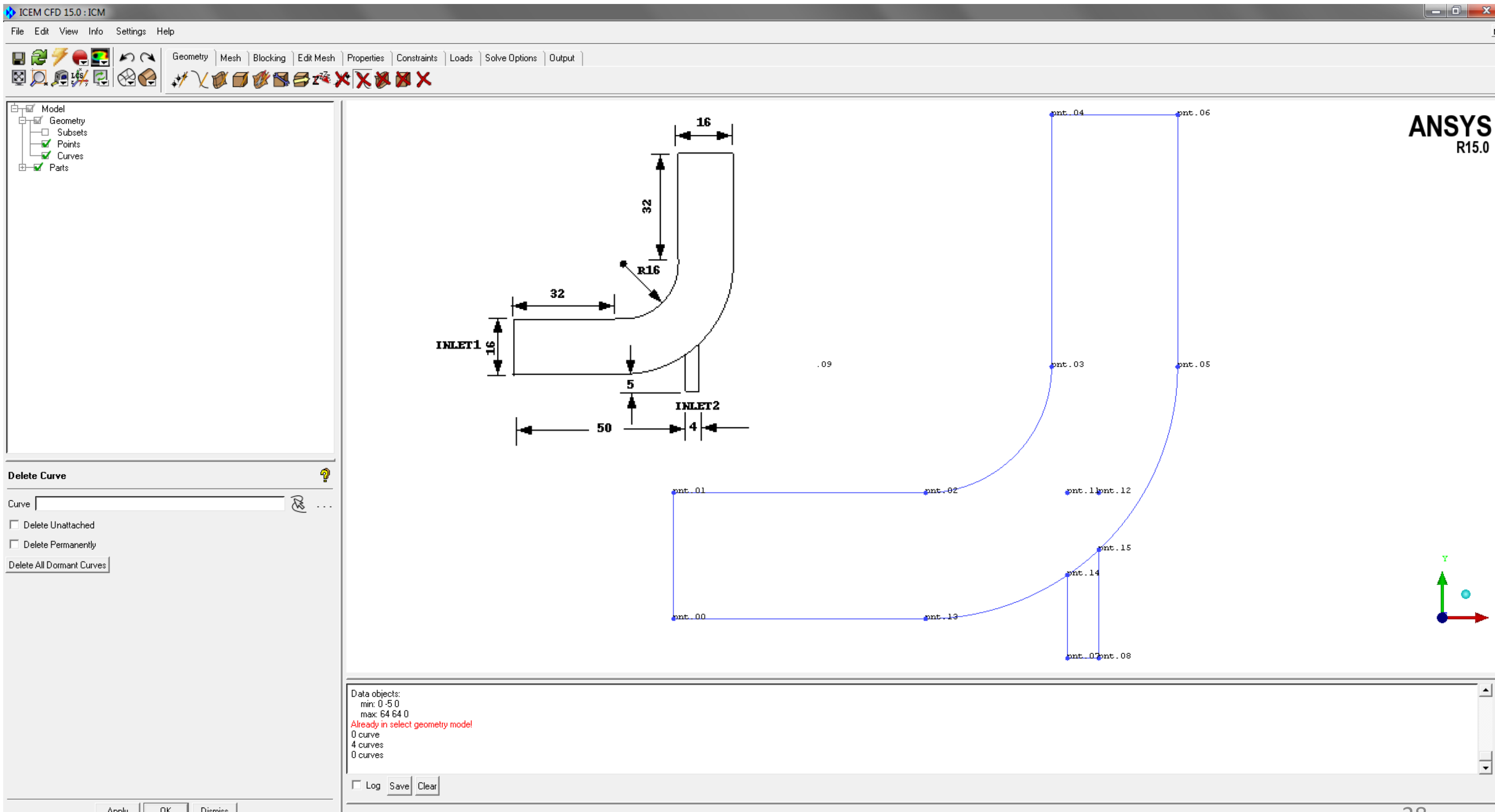
Segment curves



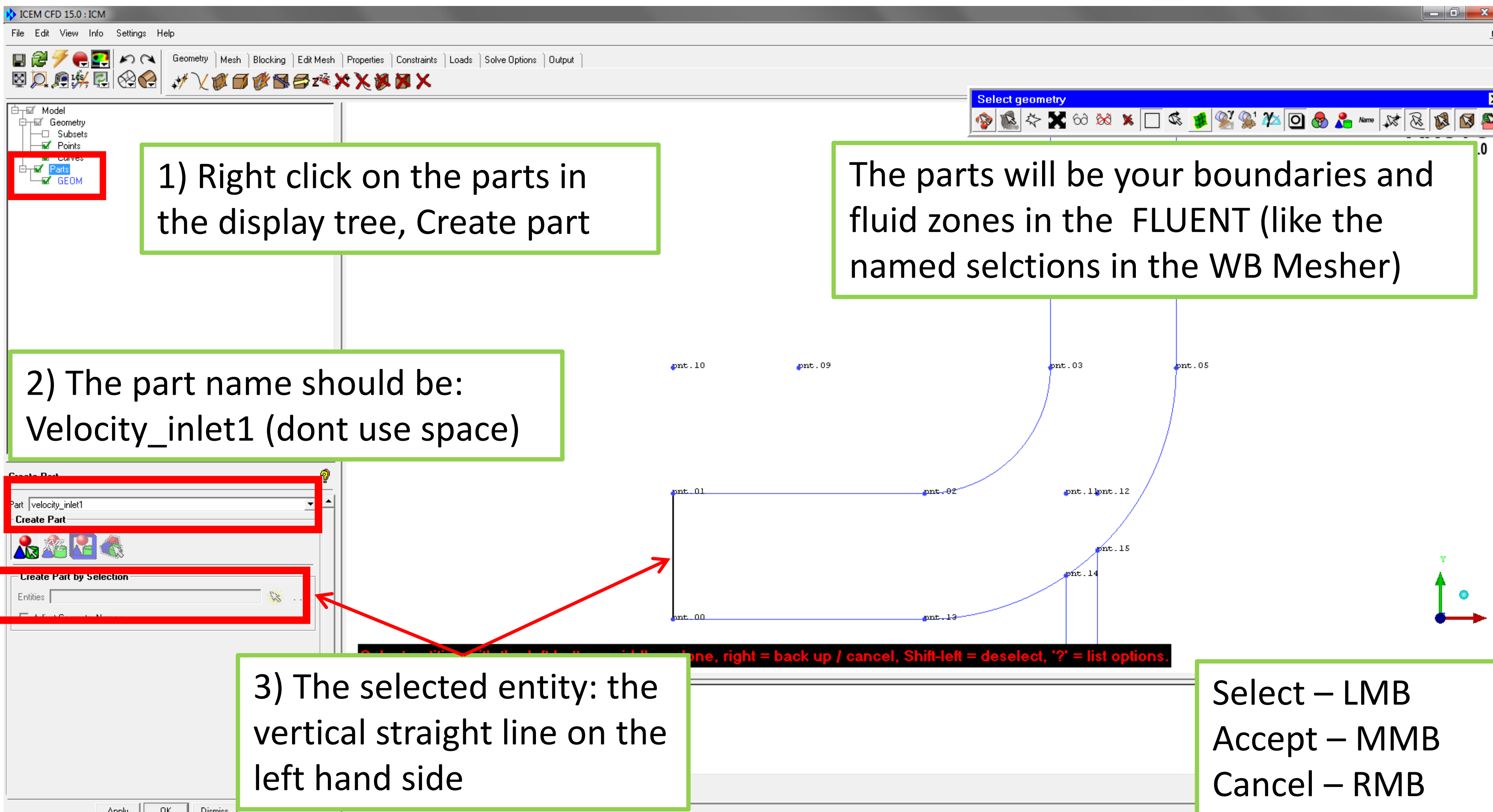
Delete curves



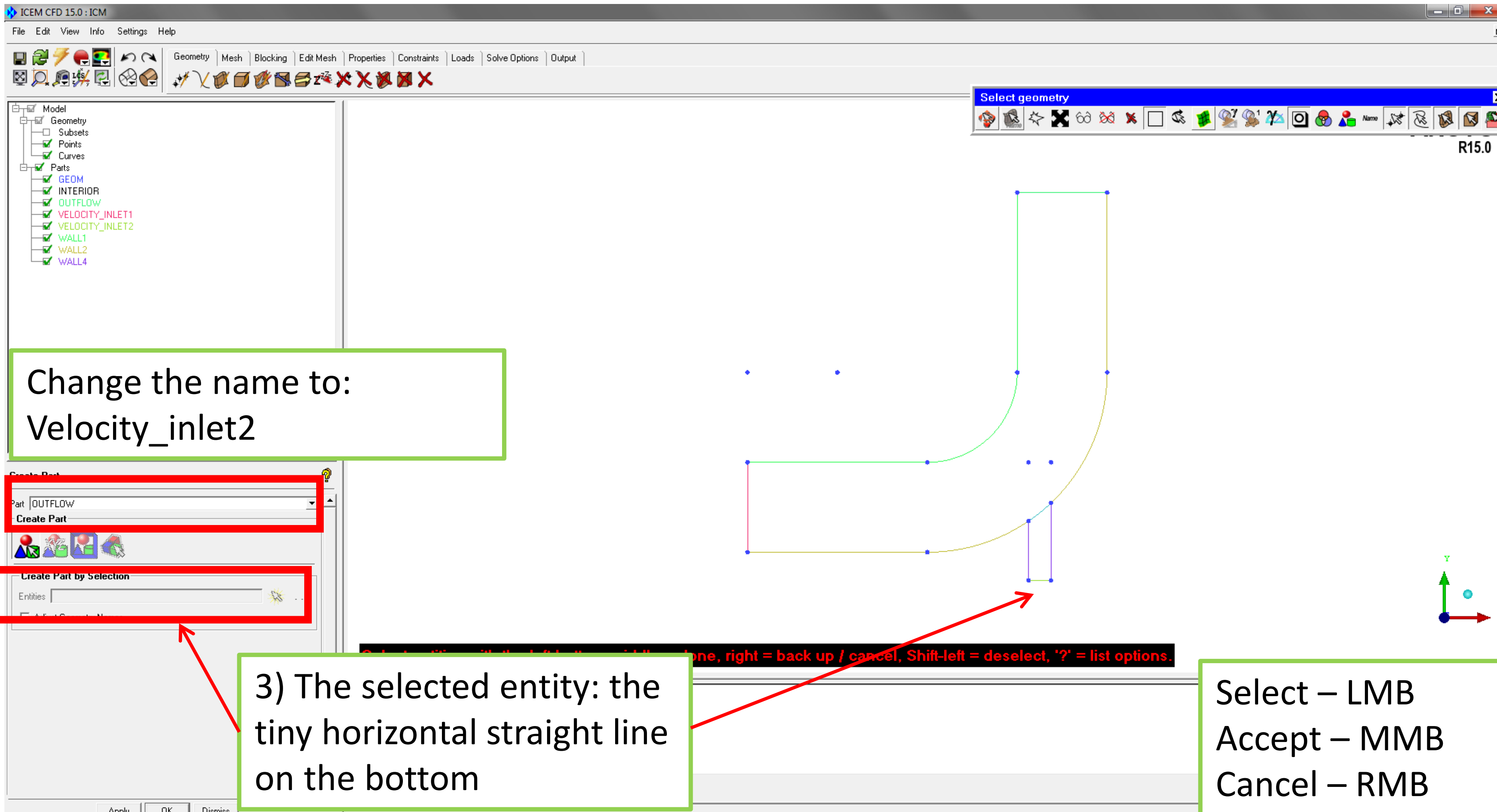
Delete curves – The result



Create Parts



Create Parts

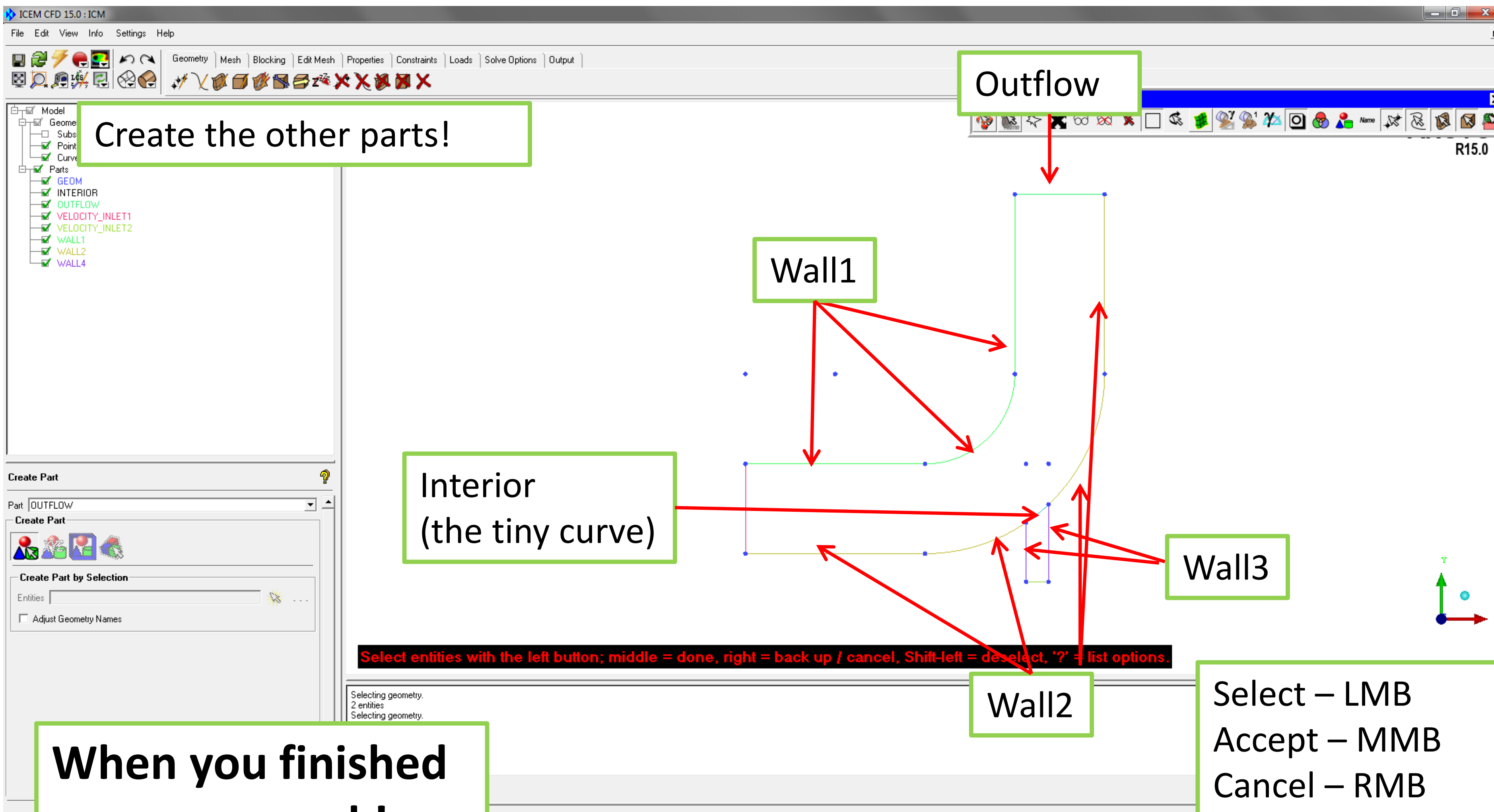


Change the name to: Velocity_inlet2

3) The selected entity: the tiny horizontal straight line on the bottom

Select – LMB
Accept – MMB
Cancel – RMB

Create Parts



When you finished save your work!

Select – LMB
Accept – MMB
Cancel – RMB

Blocking, Meshing

Block structured mesh

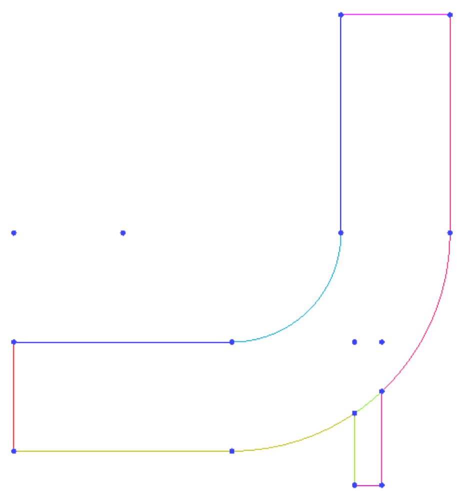
- The mesh is build up from rectangular (2D) or cuboid (3D) segments
- The blocks are divided to quad (2D) or hexa (3D) cells
- The vertices, the edges and the faces of the blocks are associated and shaped to the points, curves and surface of the geometry

Definitions in ICEM

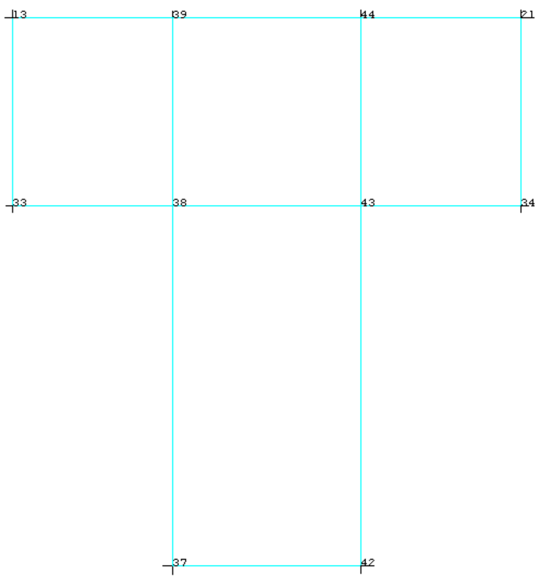
Geometry	Blocking
Point	Vertex
Curve	Edge
Surface	Face

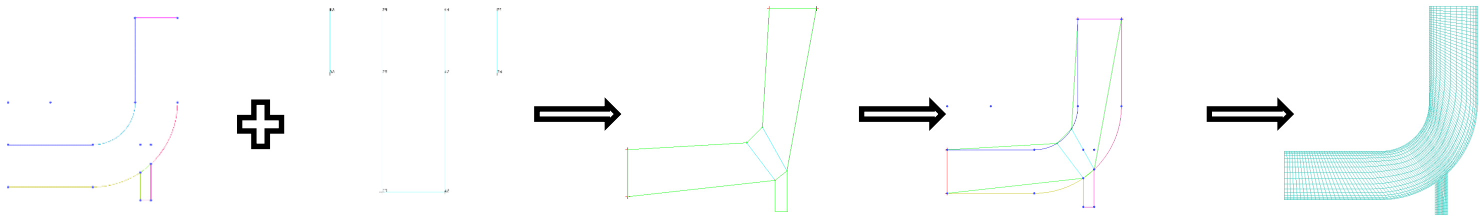
Sample (the present problem)

Geometry



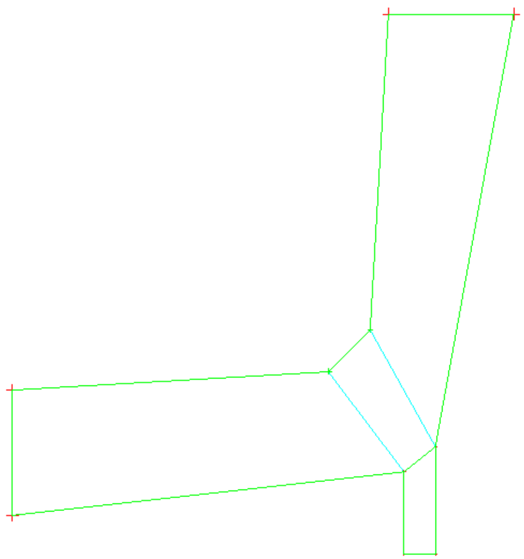
Initial blocking

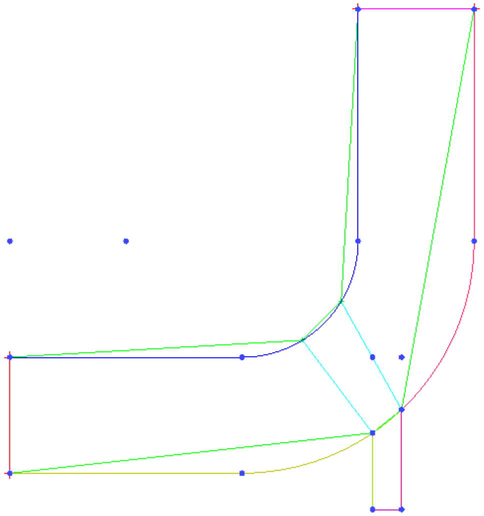


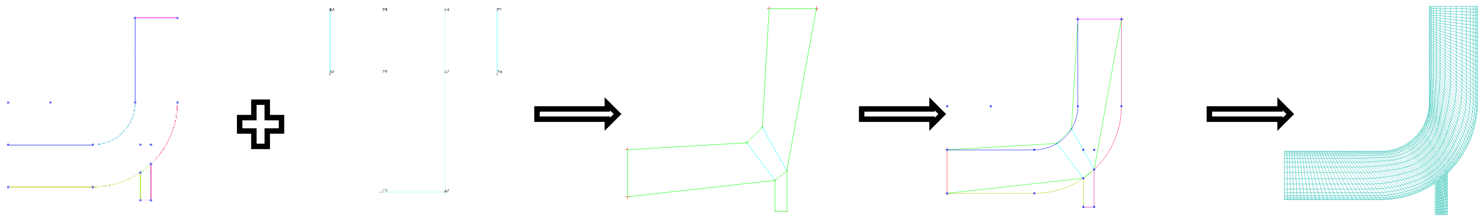


Sample (the present problem)

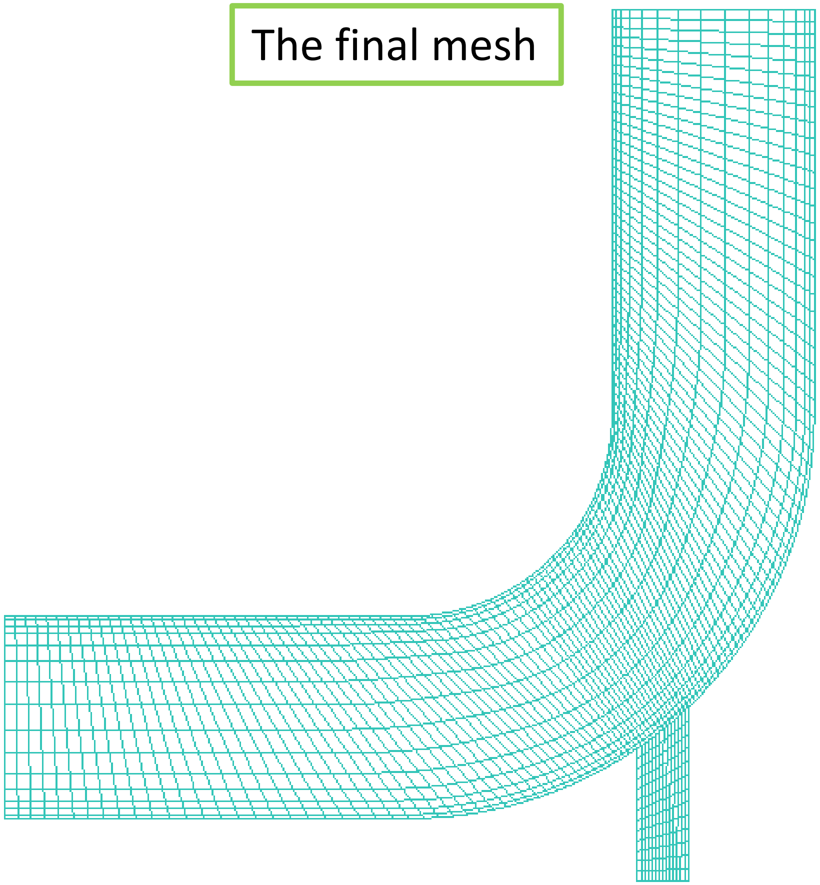
The associated blocking

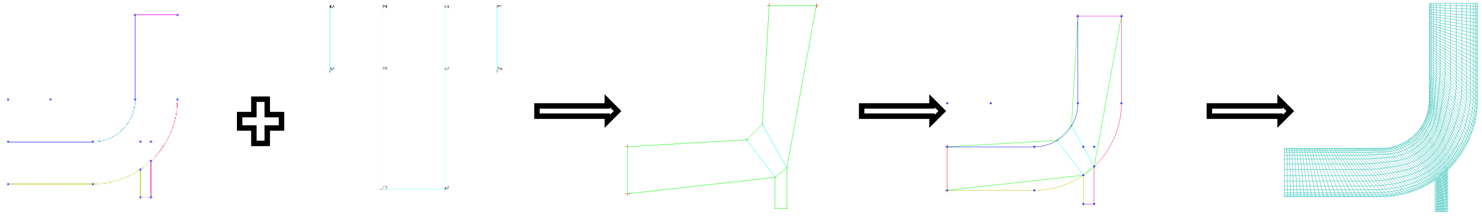






Sample (the present problem)

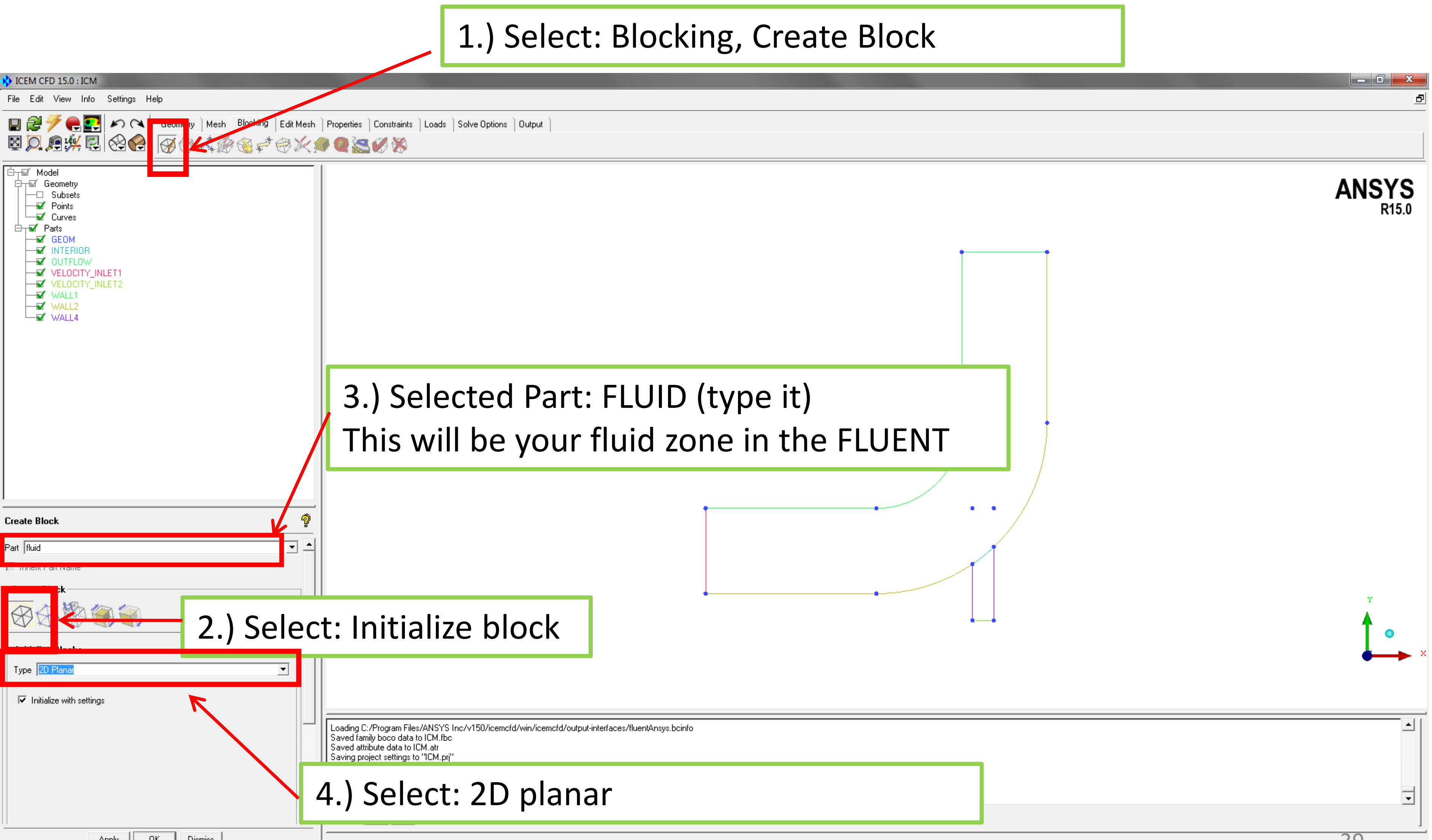




Let’s start blocking

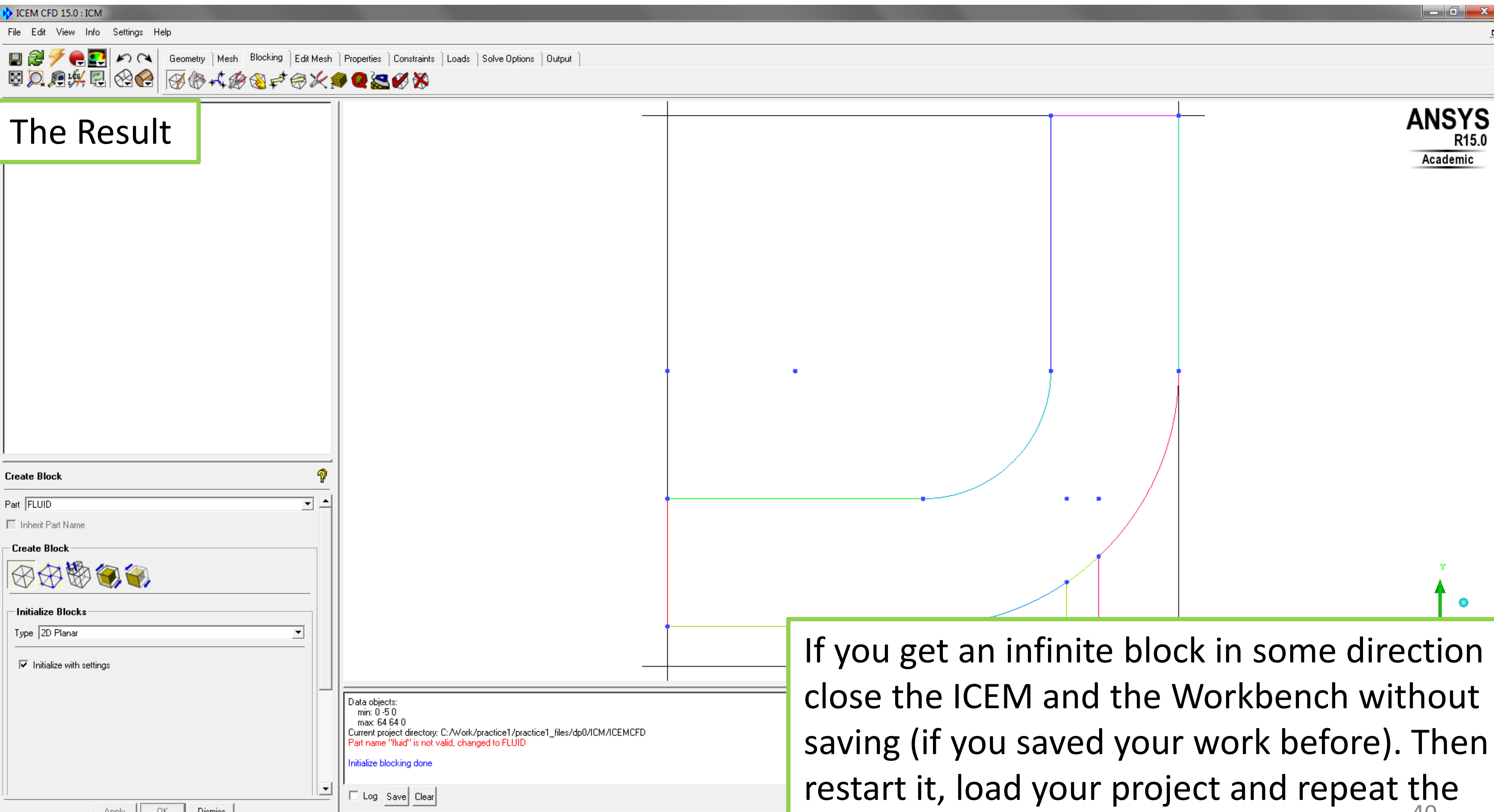
- Save your work before start the blocking!

Initialize Blocking



Initialize Blocking

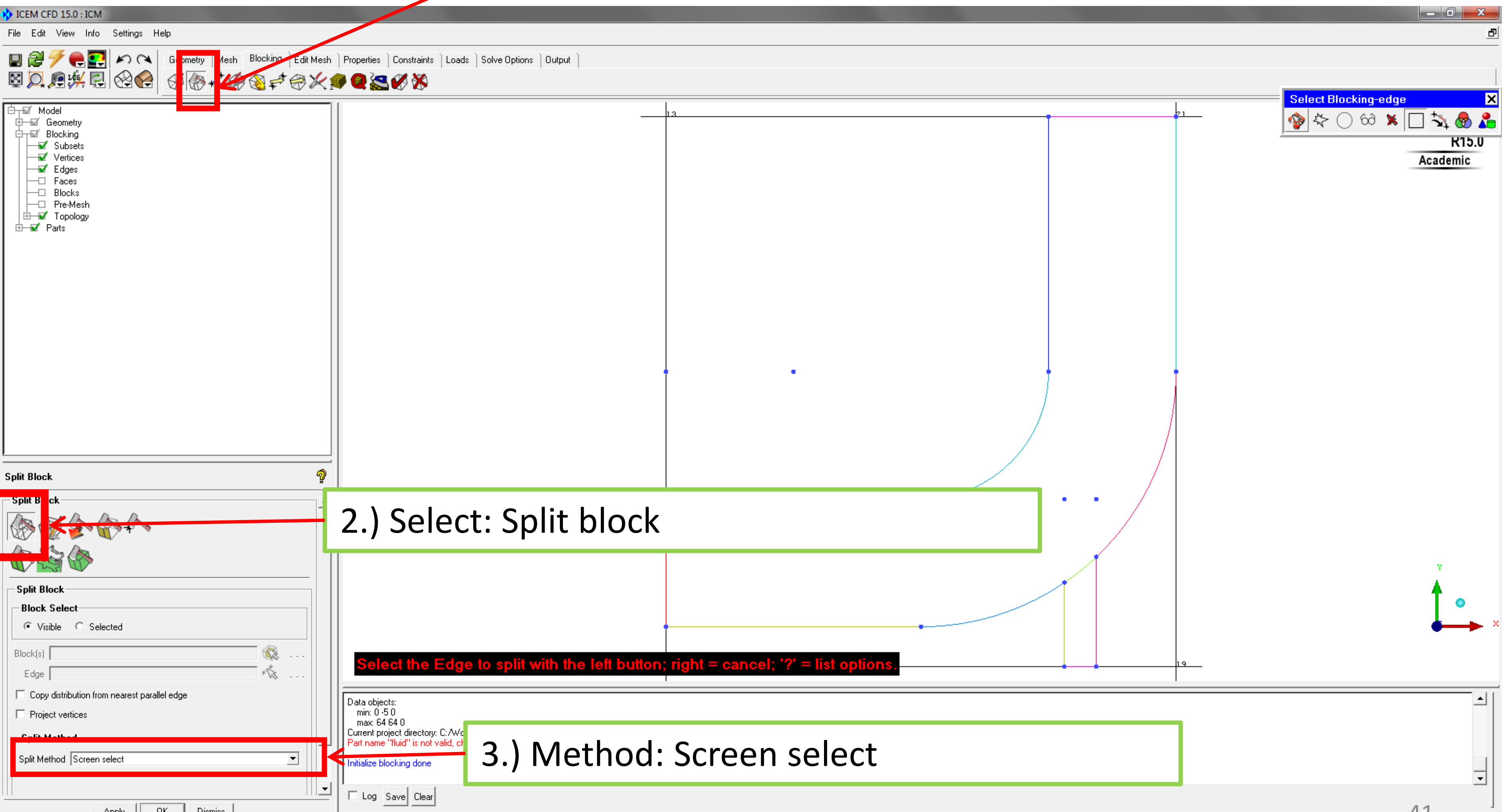
The Result



If you get an infinite block in some direction close the ICEM and the Workbench without saving (if you saved your work before). Then restart it, load your project and repeat the whole blocking procedure

Split block

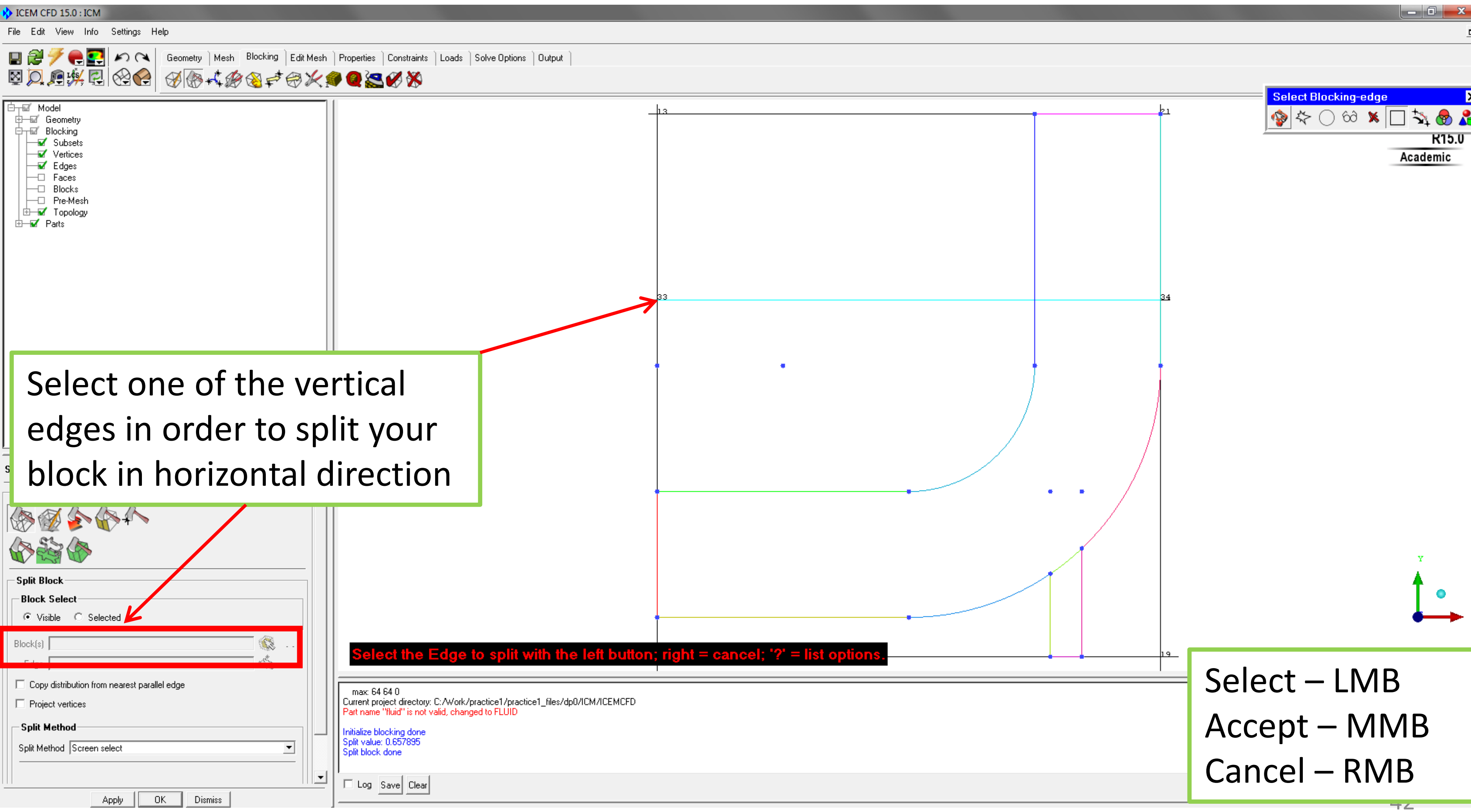
1.) Select: Blocking, Split Block



2.) Select: Split block

3.) Method: Screen select

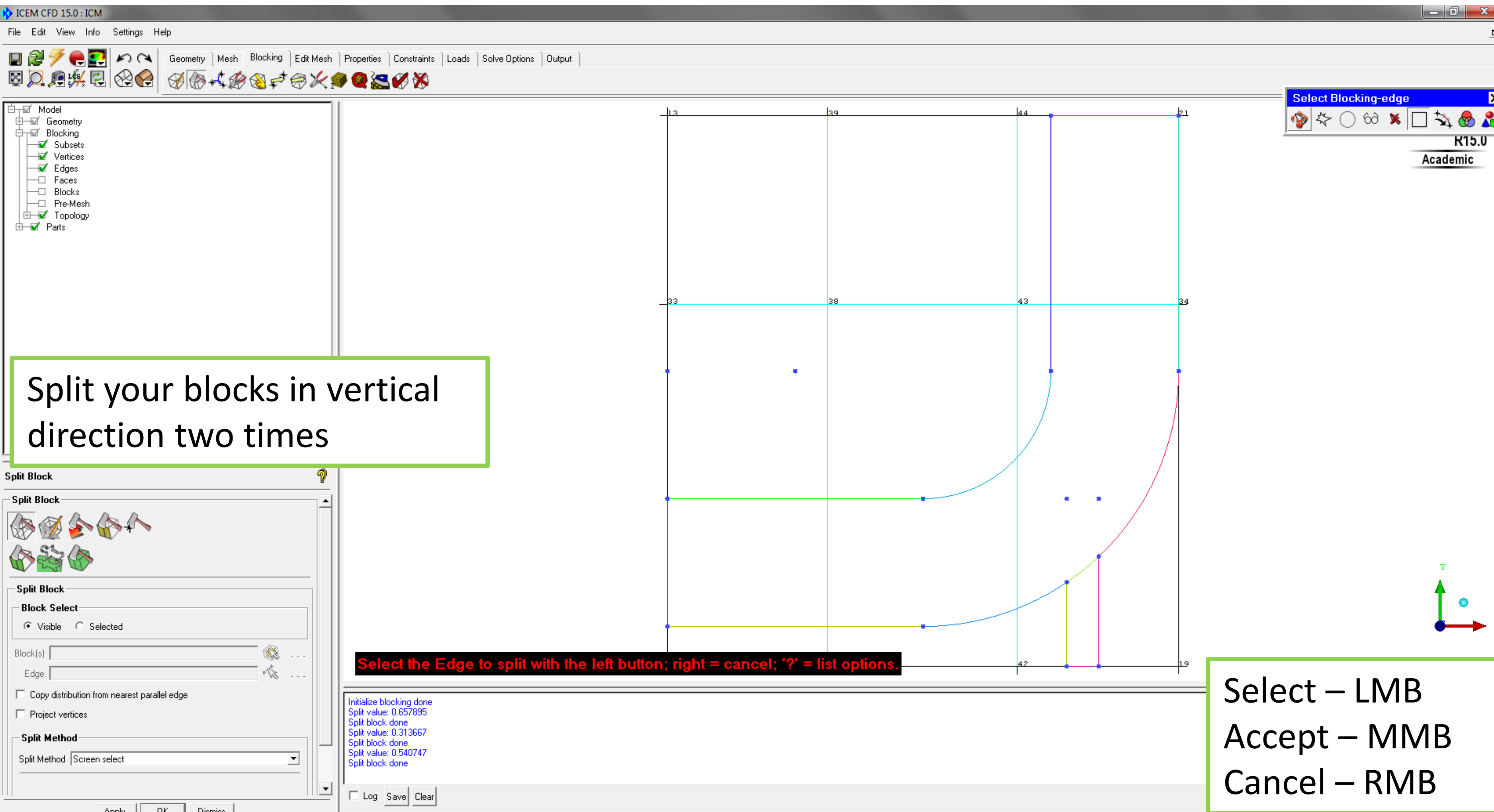
Split block



Select one of the vertical edges in order to split your block in horizontal direction

Select – LMB
Accept – MMB
Cancel – RMB

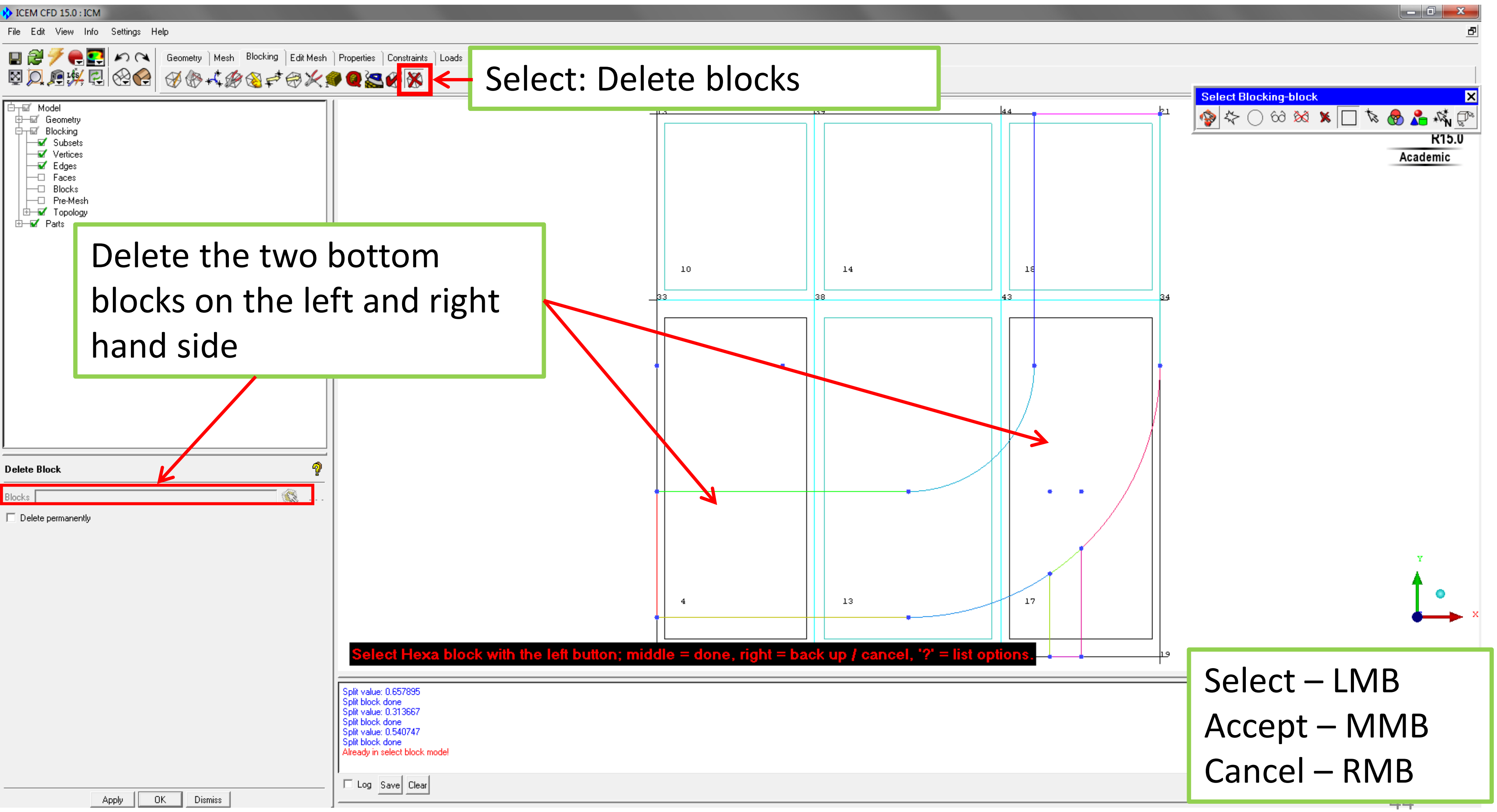
Split block



Split your blocks in vertical direction two times

Select – LMB
Accept – MMB
Cancel – RMB

Split block

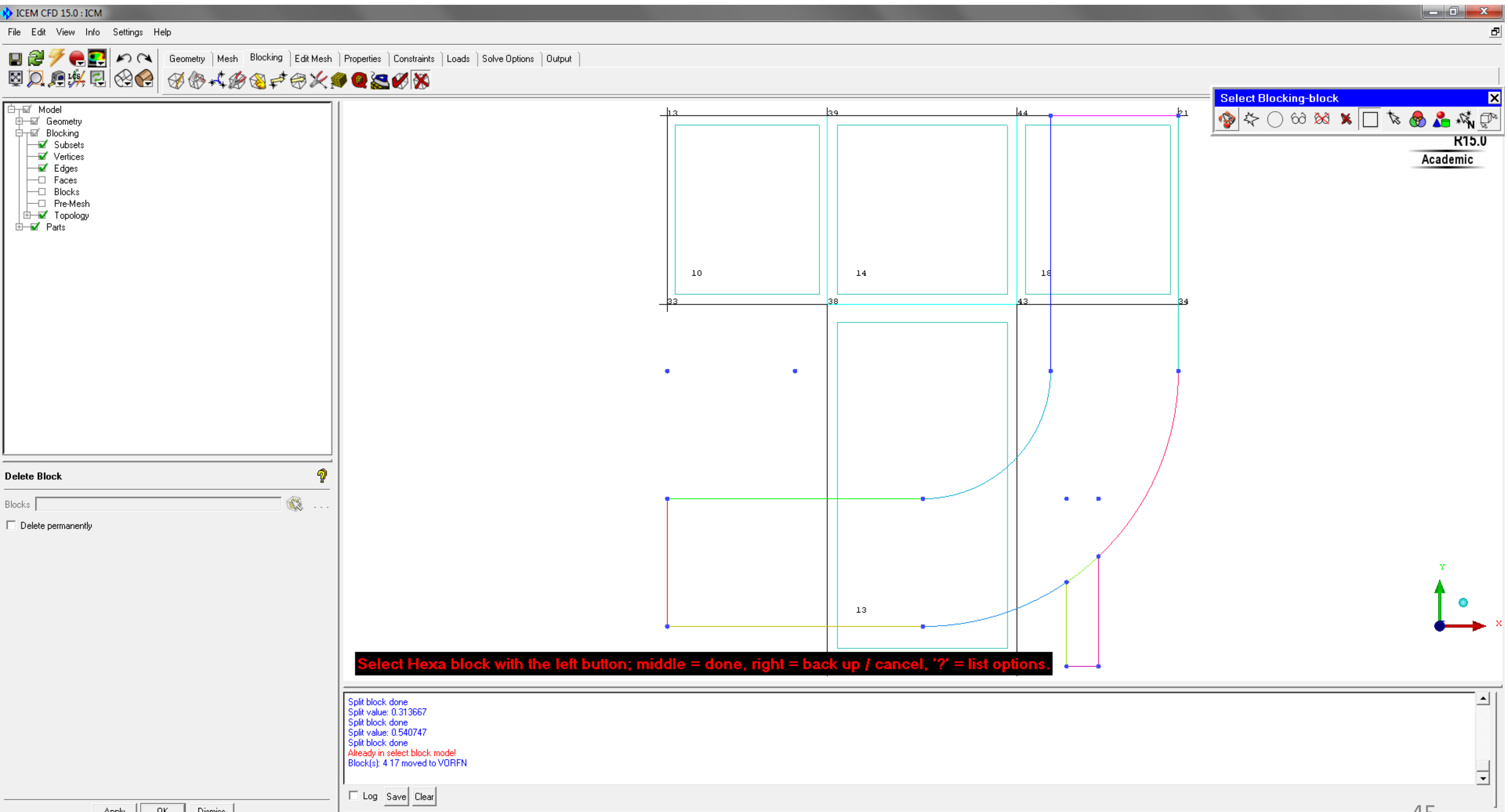


Select: Delete blocks

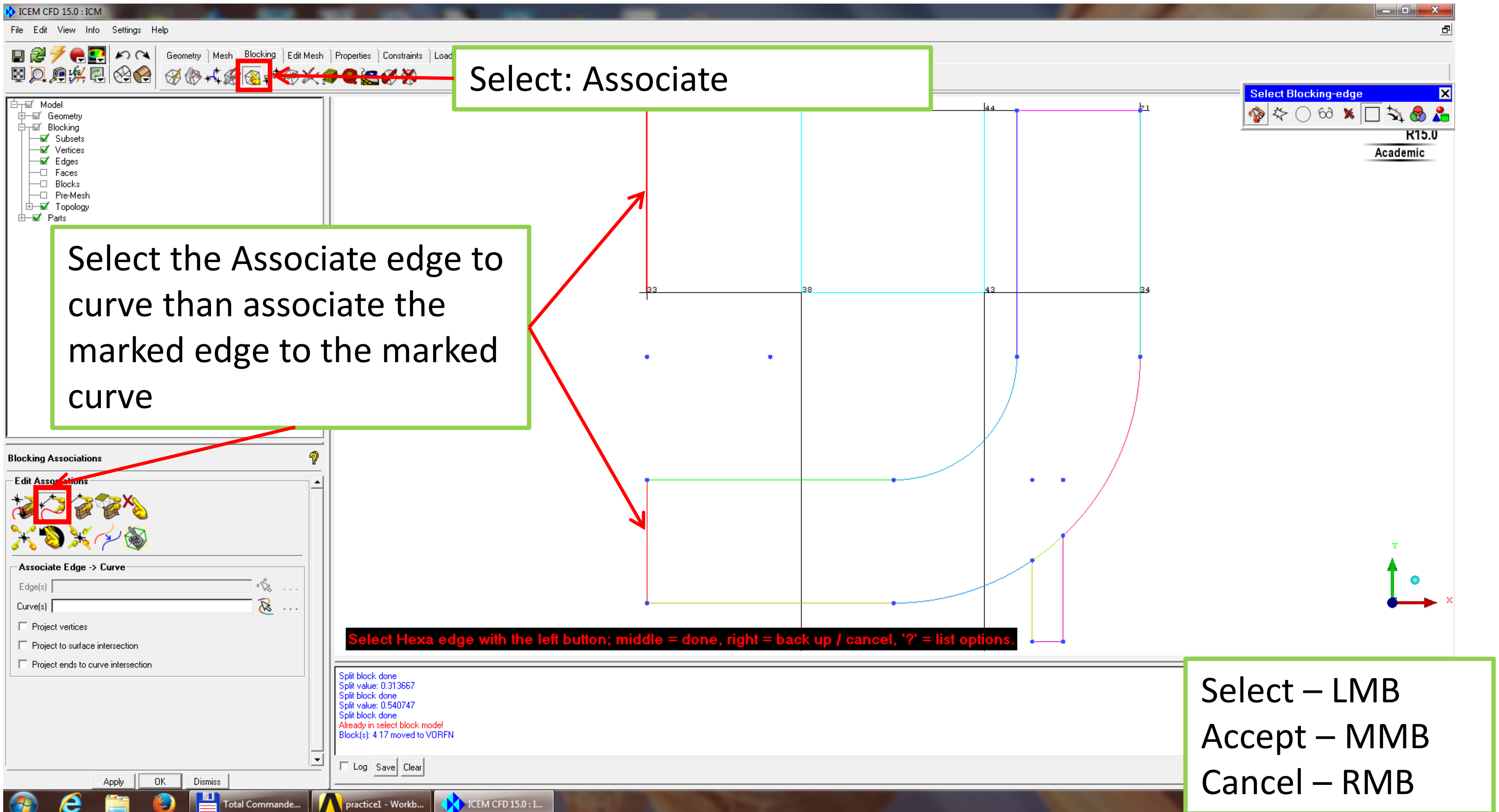
Delete the two bottom blocks on the left and right hand side

Select – LMB
Accept – MMB
Cancel – RMB

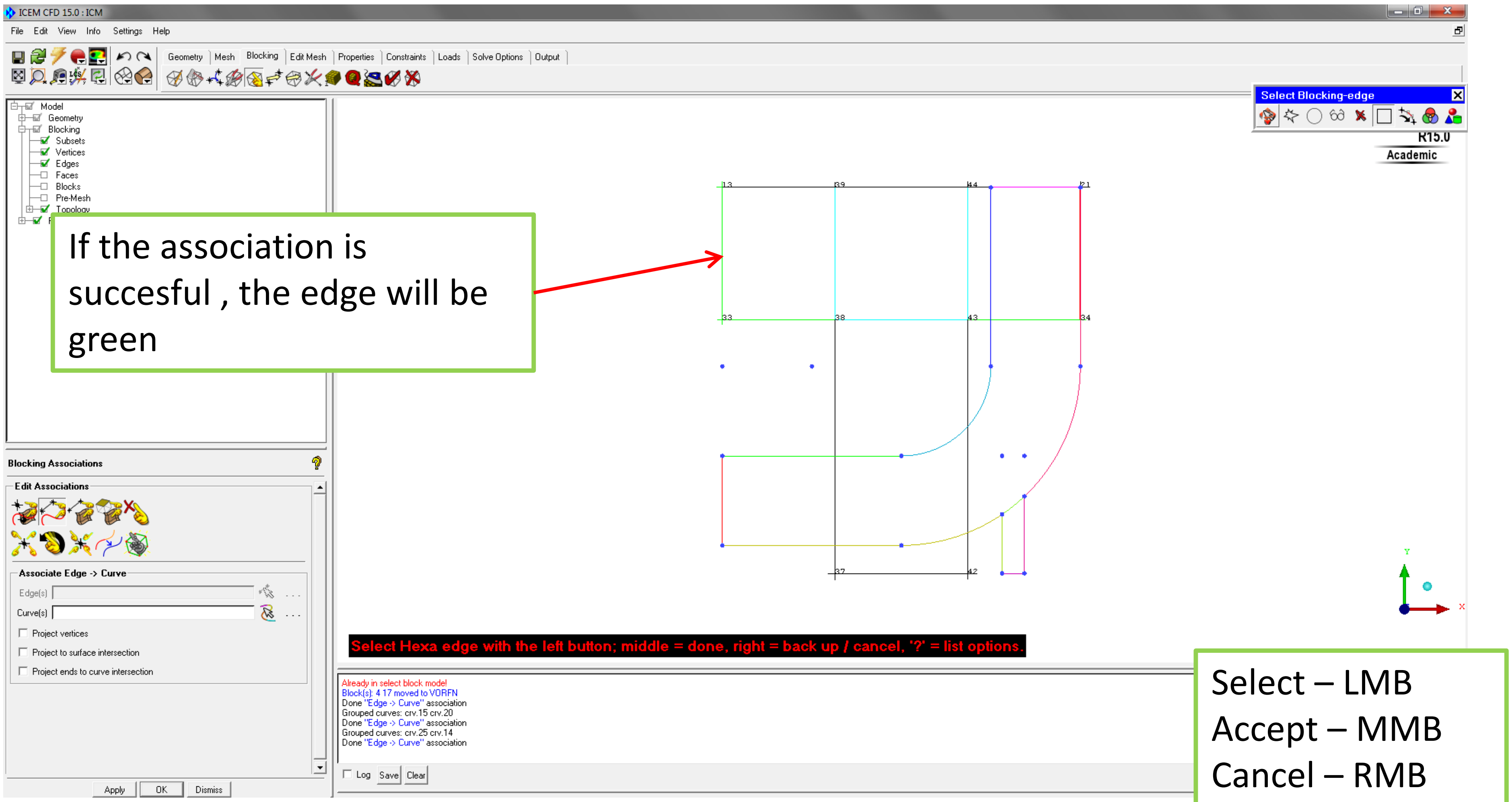
Split block


ICEM CFD 15.0 : ICM
File Edit View Info Settings Help
Geometry Mesh Blocking Edit Mesh Properties Constraints Loads Solve Options Output
Model
Geometry
Blocking
Subsets
Vertices
Edges
Faces
Blocks
Pre-Mesh
Topology
Parts
Delete Block
Blocks
Delete permanently
Apply OK Dismiss
Select Blocking-block
R15.0
Academic
Select Hexa block with the left button; middle = done, right = back up / cancel, '?' = list options.
Split block done
Split value: 0.313667
Split block done
Split value: 0.540747
Split block done
Already in select block mode!
Block(s): 4 17 moved to VORFN
Log Save Clear

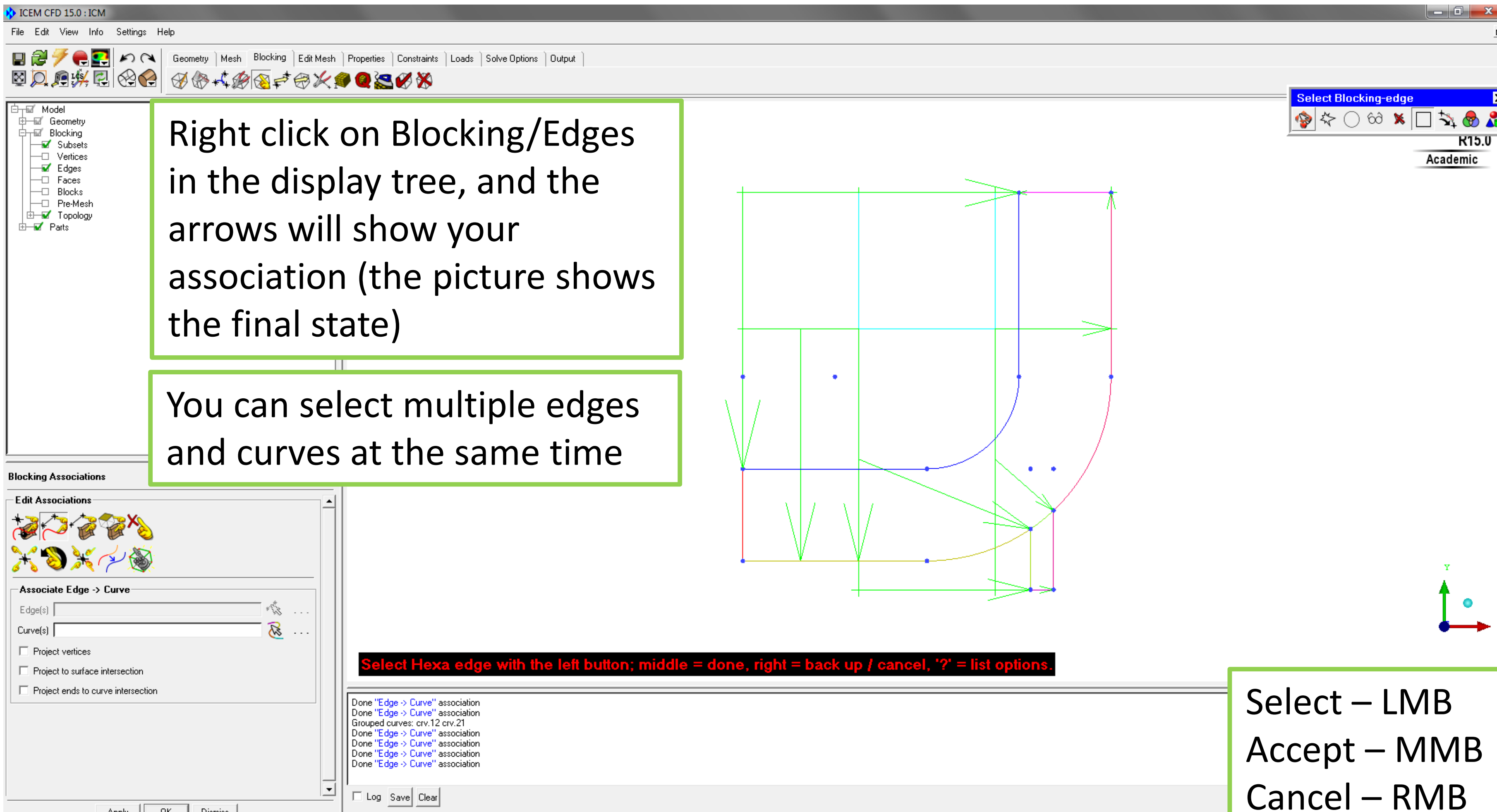
Associate the edges to the curves



Associate the edges to the curves



Associate the edges to the curves



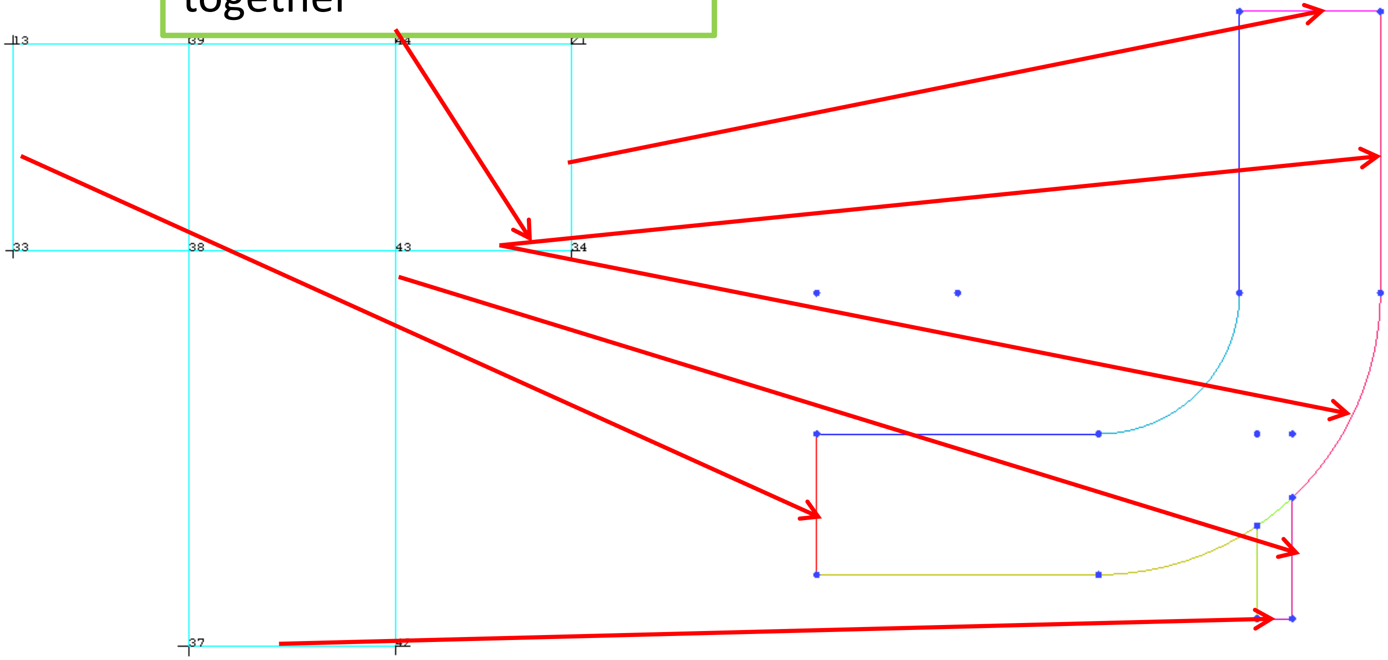
Right click on Blocking/Edges in the display tree, and the arrows will show your association (the picture shows the final state)

You can select multiple edges and curves at the same time

Select – LMB
Accept – MMB
Cancel – RMB

Associate the edges to the curves

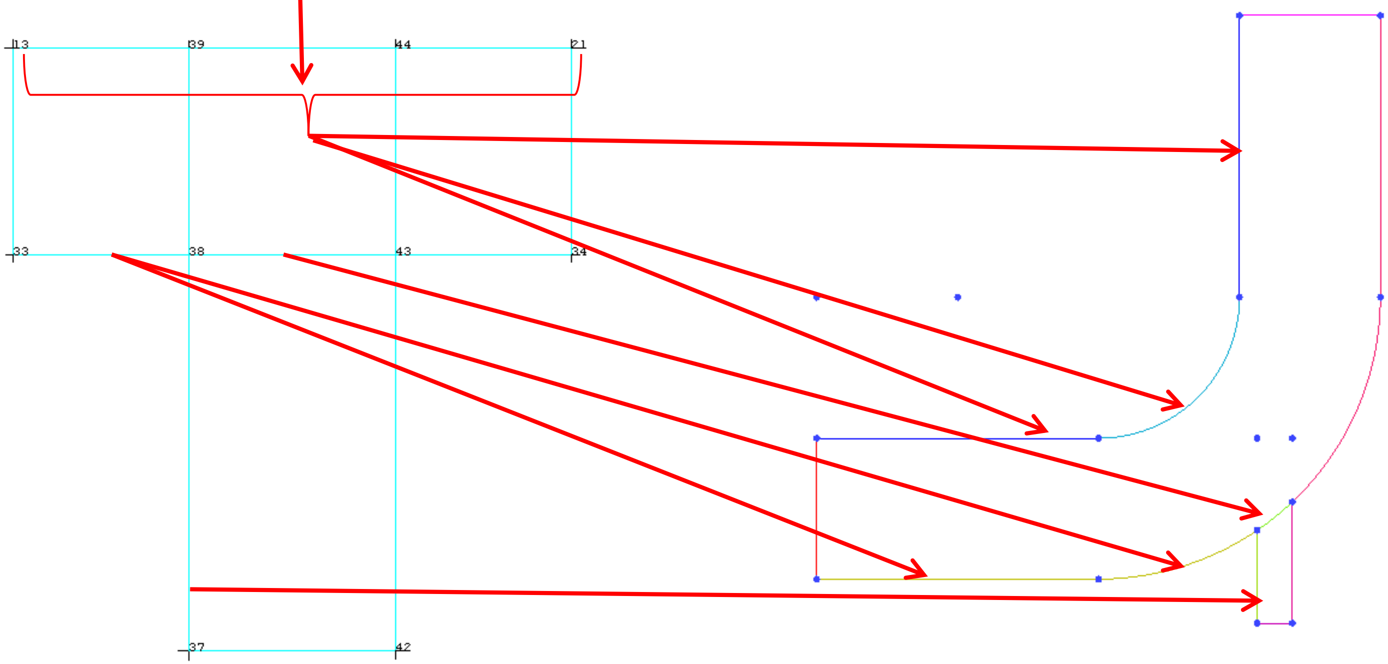
Select the two curves together



Select – LMB
Accept – MMB
Cancel – RMB

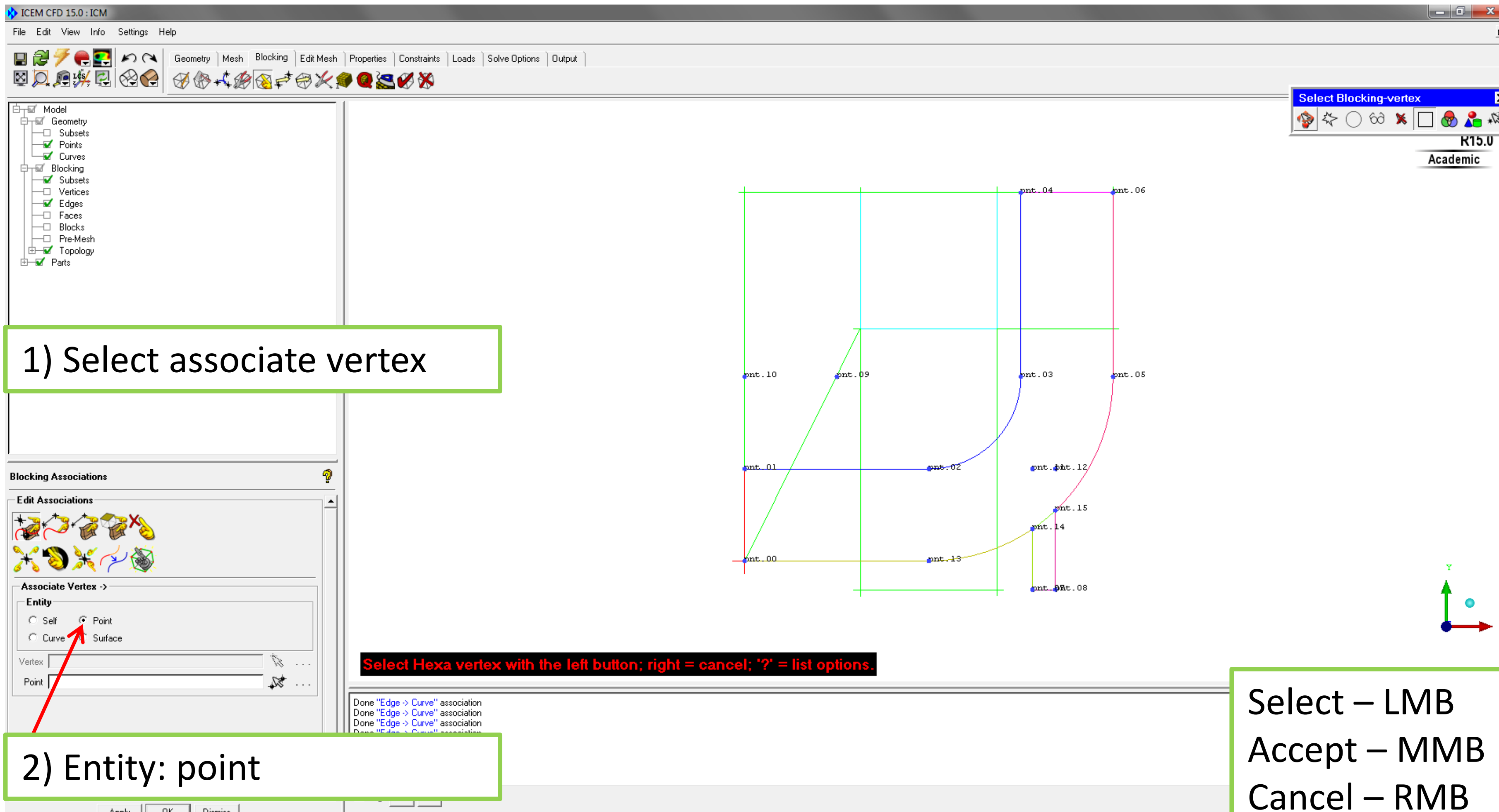
Associate the edges to the curves

Select the three edges together



Select – LMB
Accept – MMB
Cancel – RMB

Associate the vertices to points

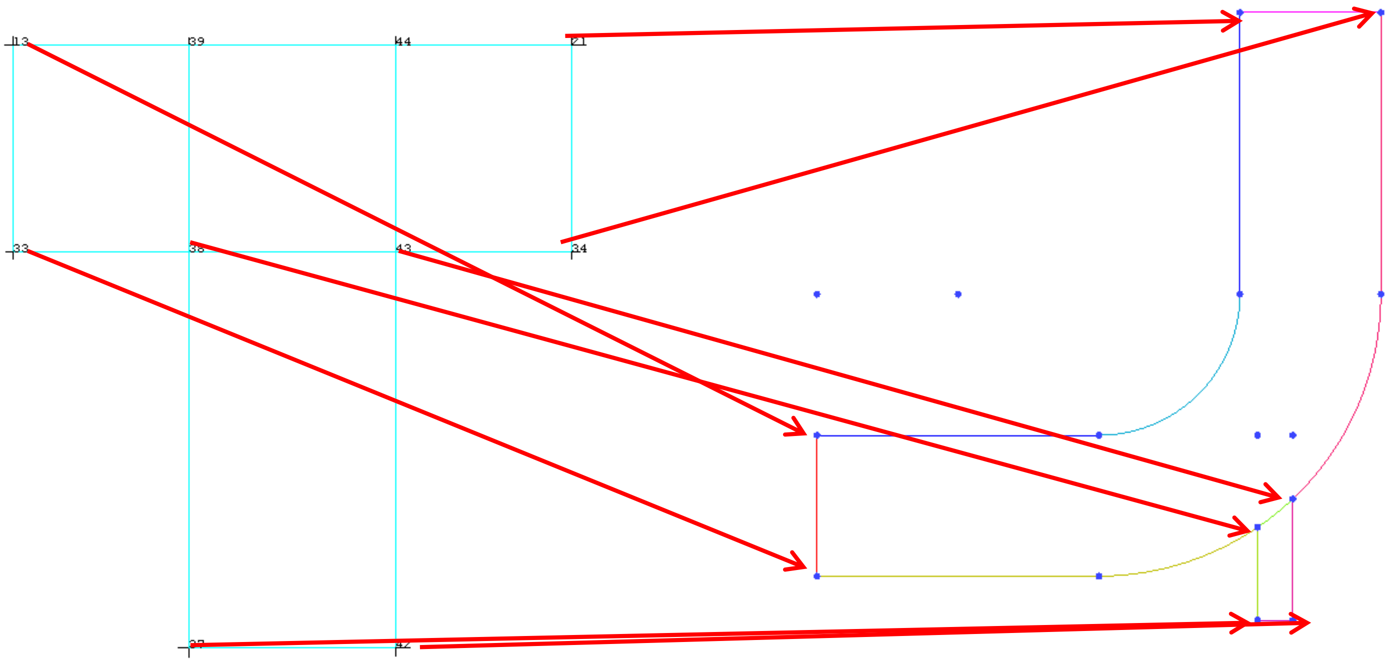


1) Select associate vertex

2) Entity: point

Select – LMB
Accept – MMB
Cancel – RMB

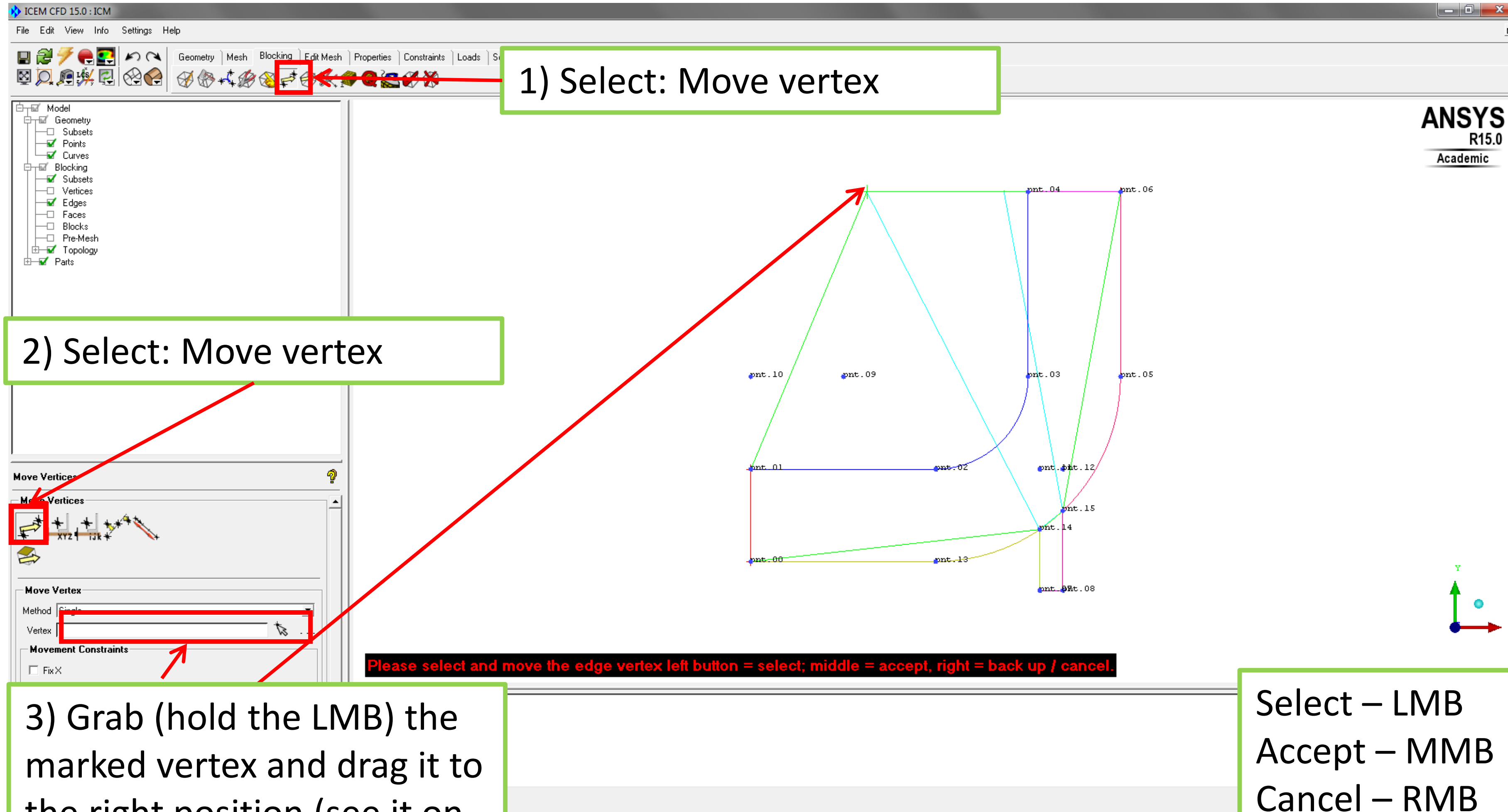
Associate the vertices to points



Associate the vertices to the points as the arrows show
You can see the result on the next slide

Select – LMB
Cancel – RMB

Move unassociated vertices



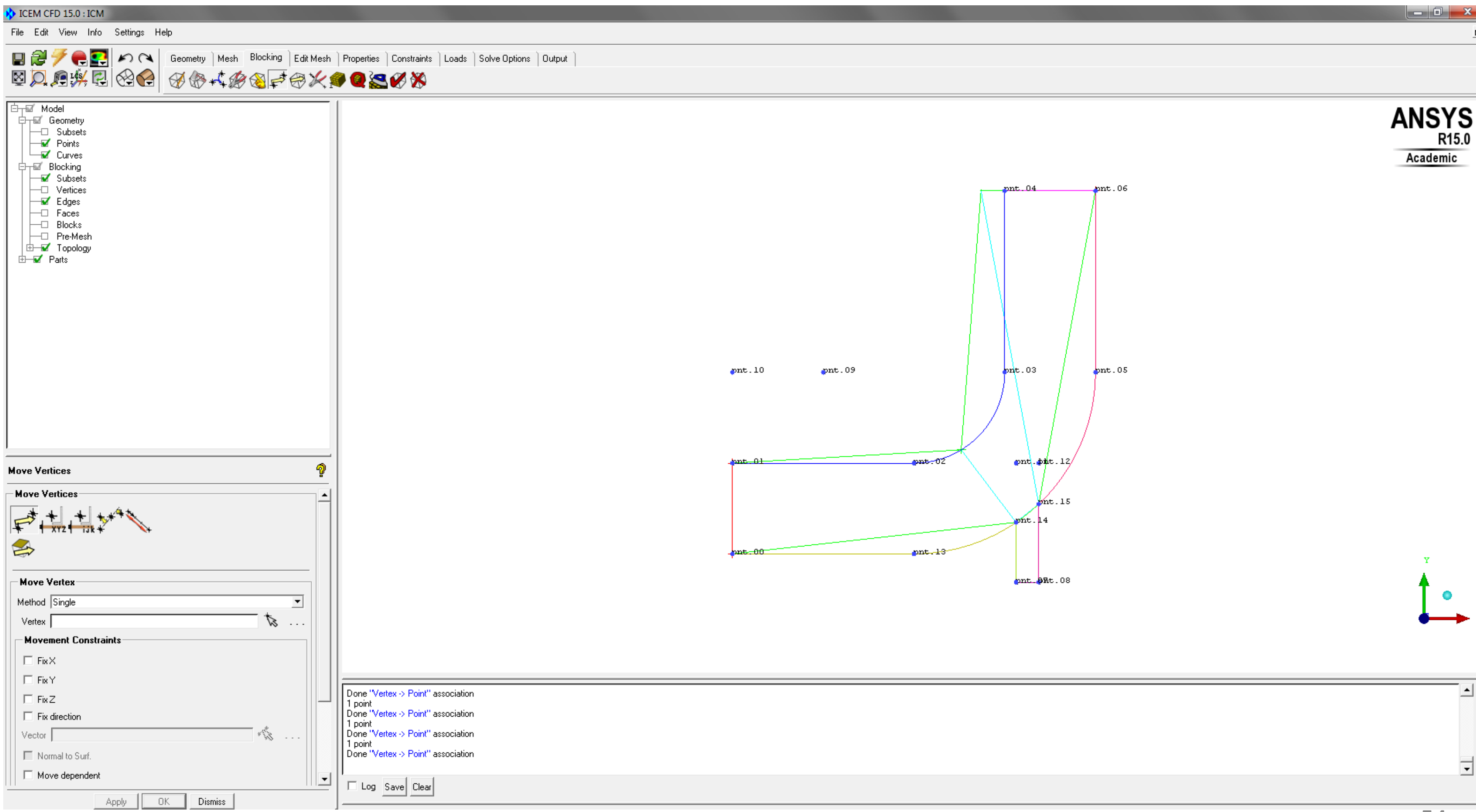
1) Select: Move vertex

2) Select: Move vertex

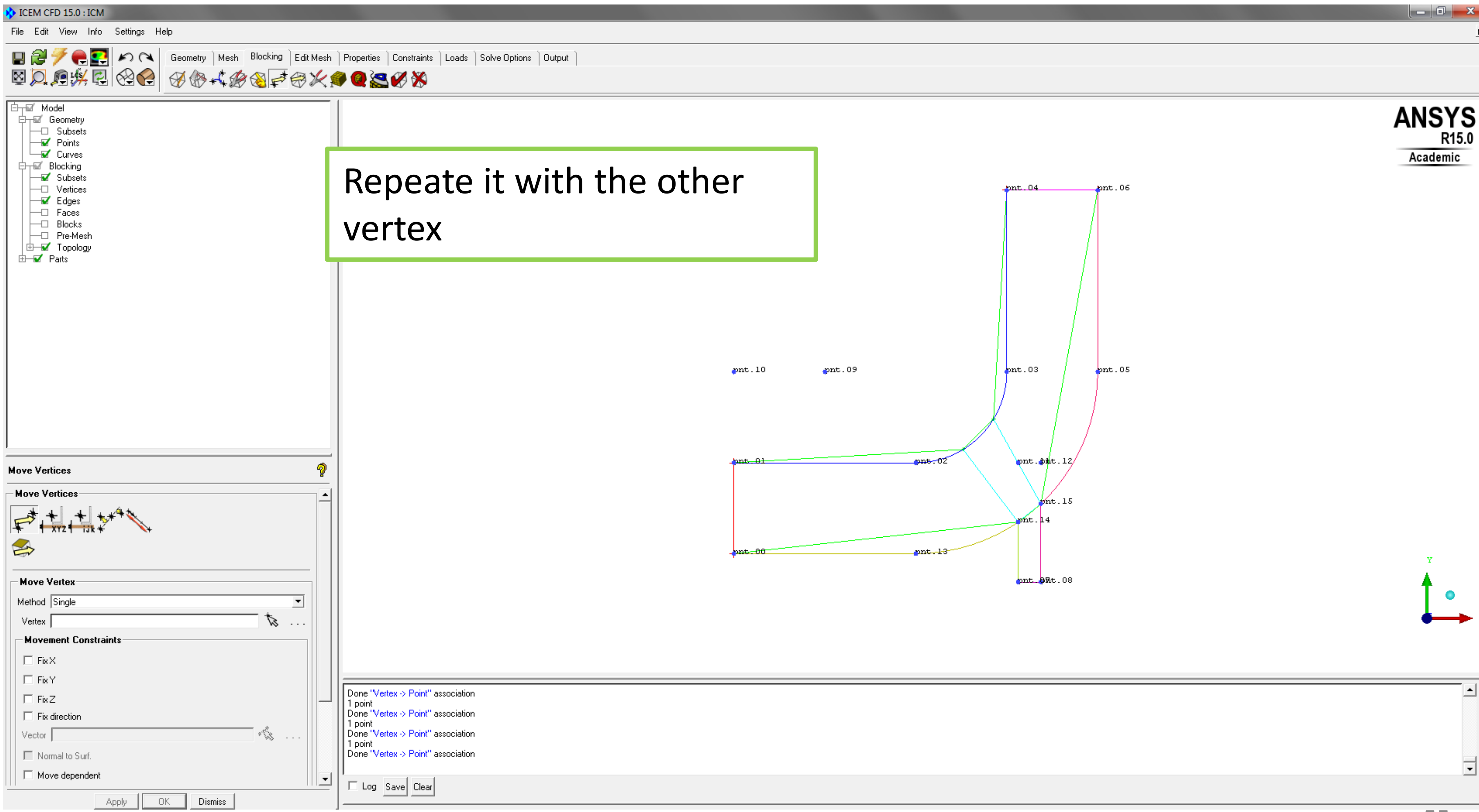
3) Grab (hold the LMB) the marked vertex and drag it to the right position (see it on the next slide)

Select – LMB
Accept – MMB
Cancel – RMB

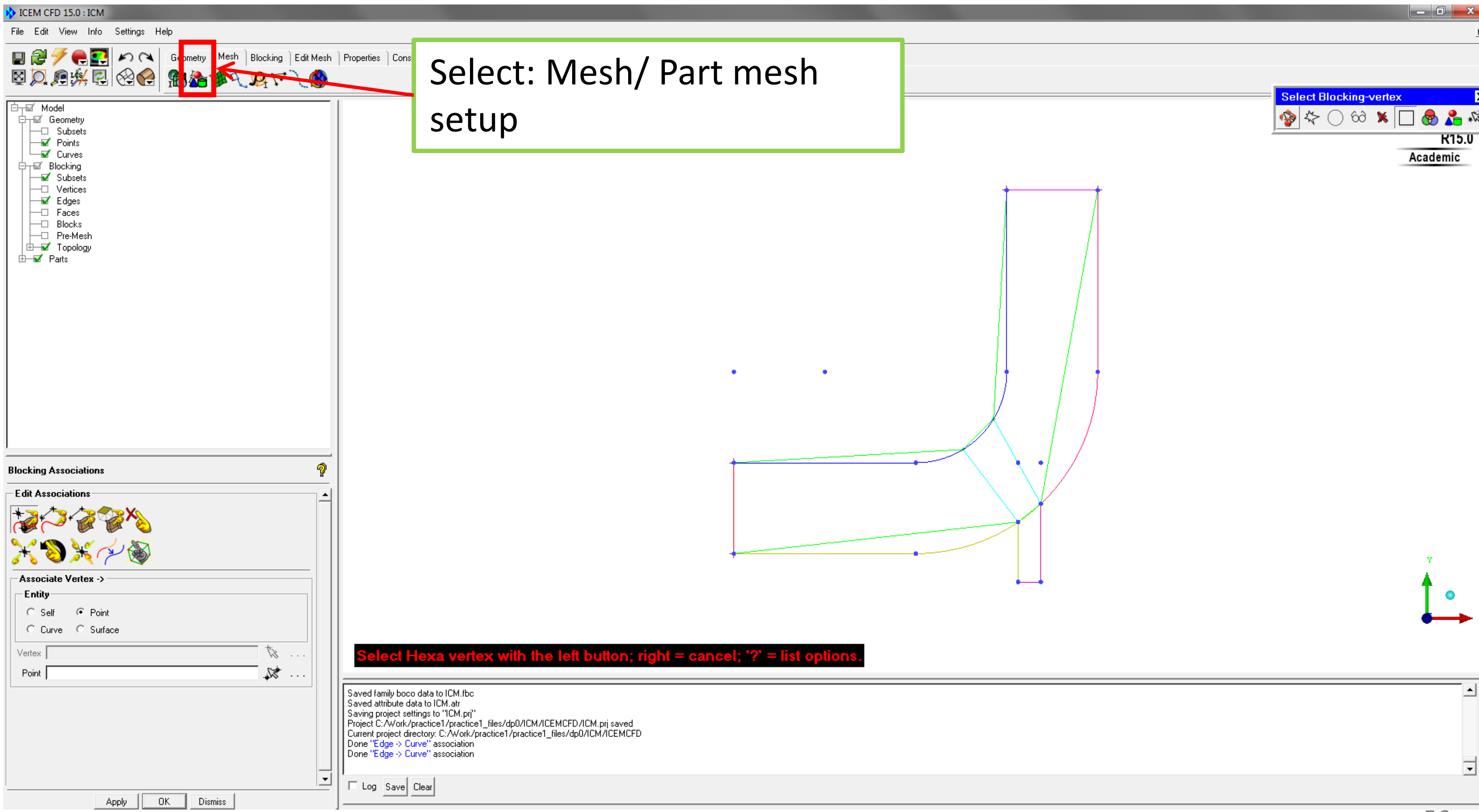
Move unassociated vertices



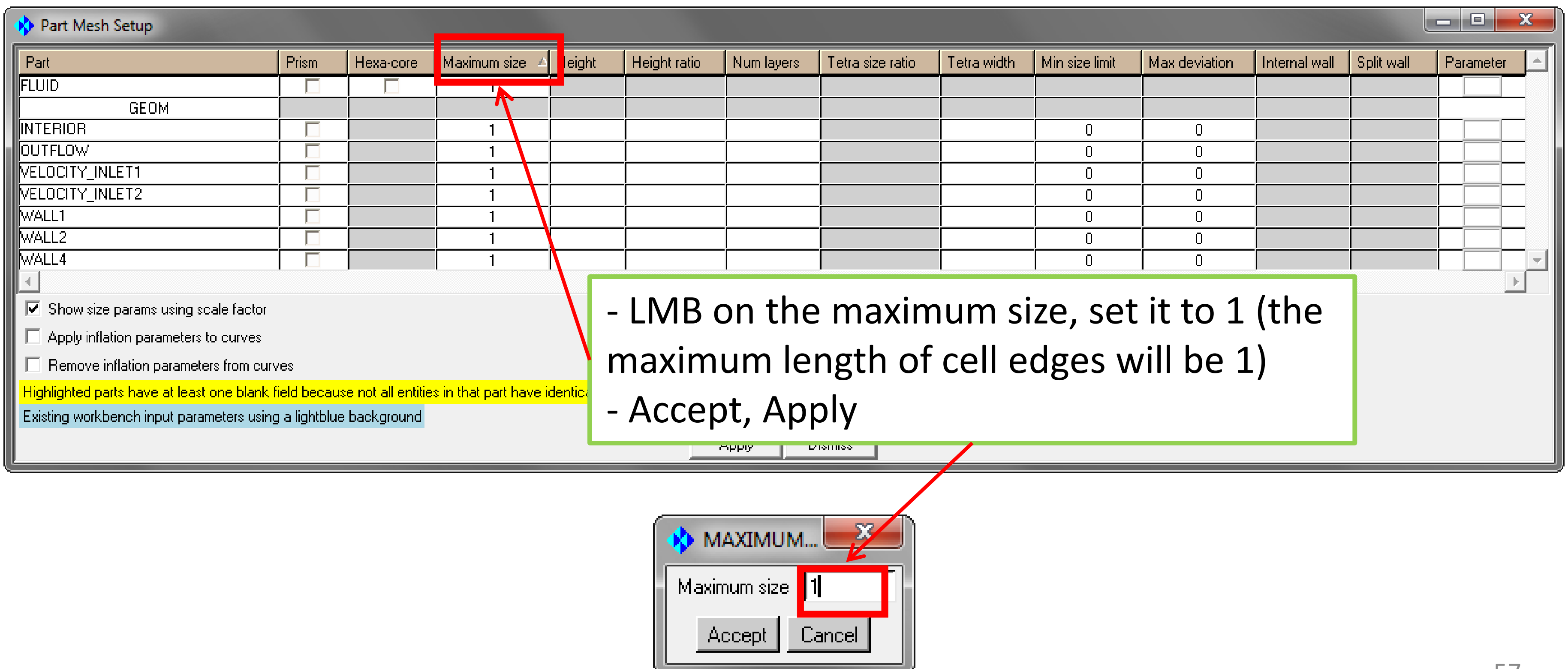
Move unassociated vertices



Setting mesh parameters

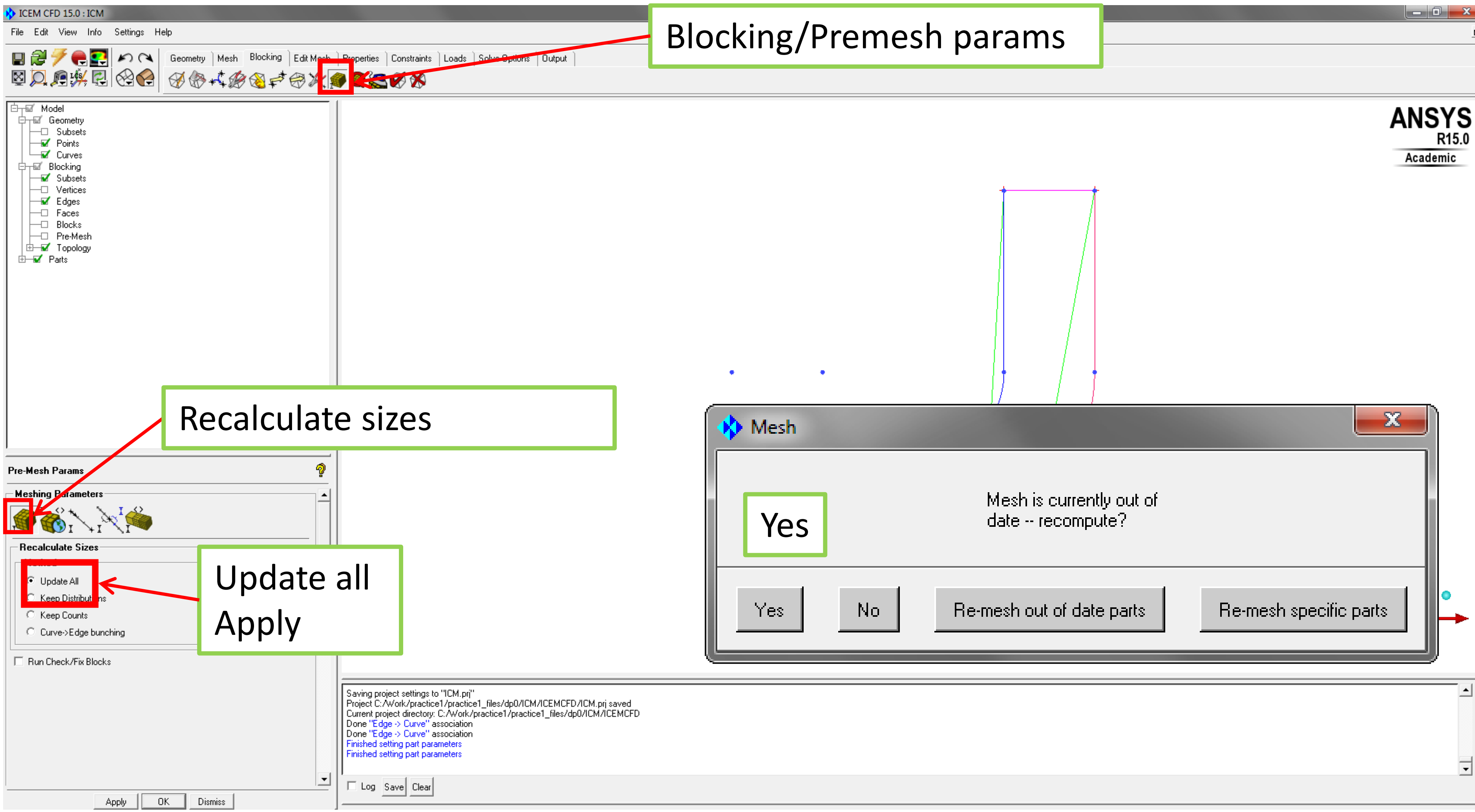


Setting mesh parameters

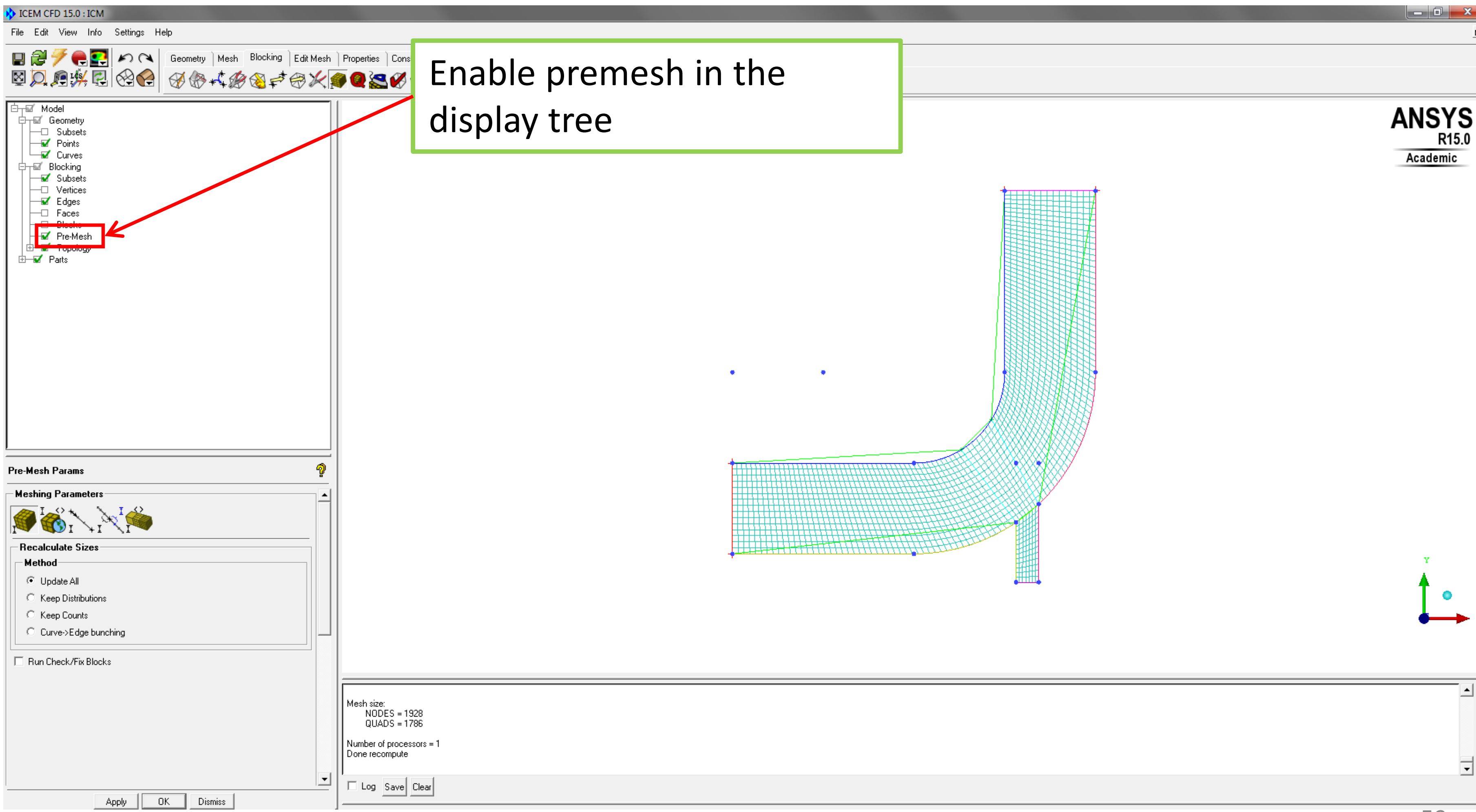


- LMB on the maximum size, set it to 1 (the maximum length of cell edges will be 1)
- Accept, Apply

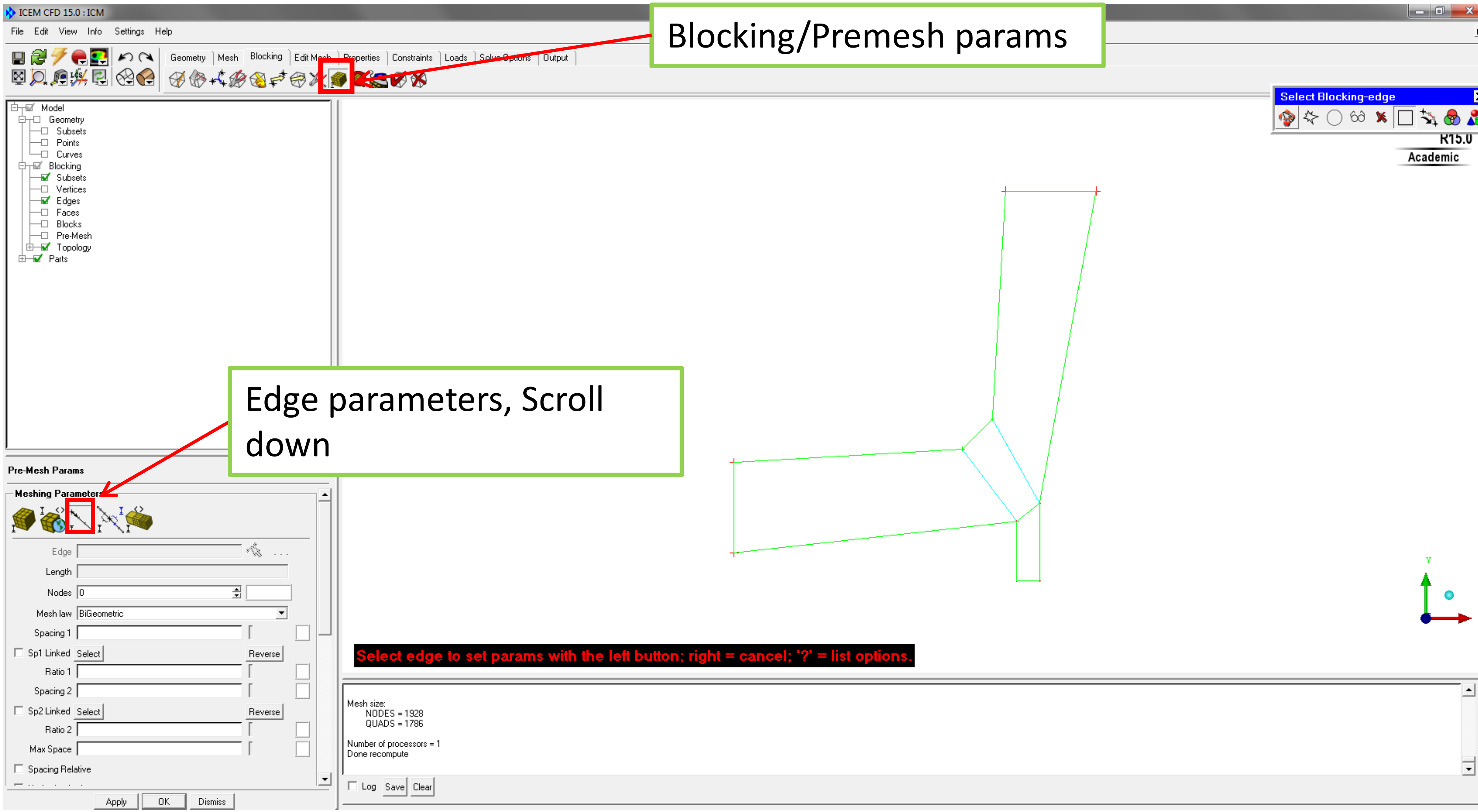
Setting mesh parameters



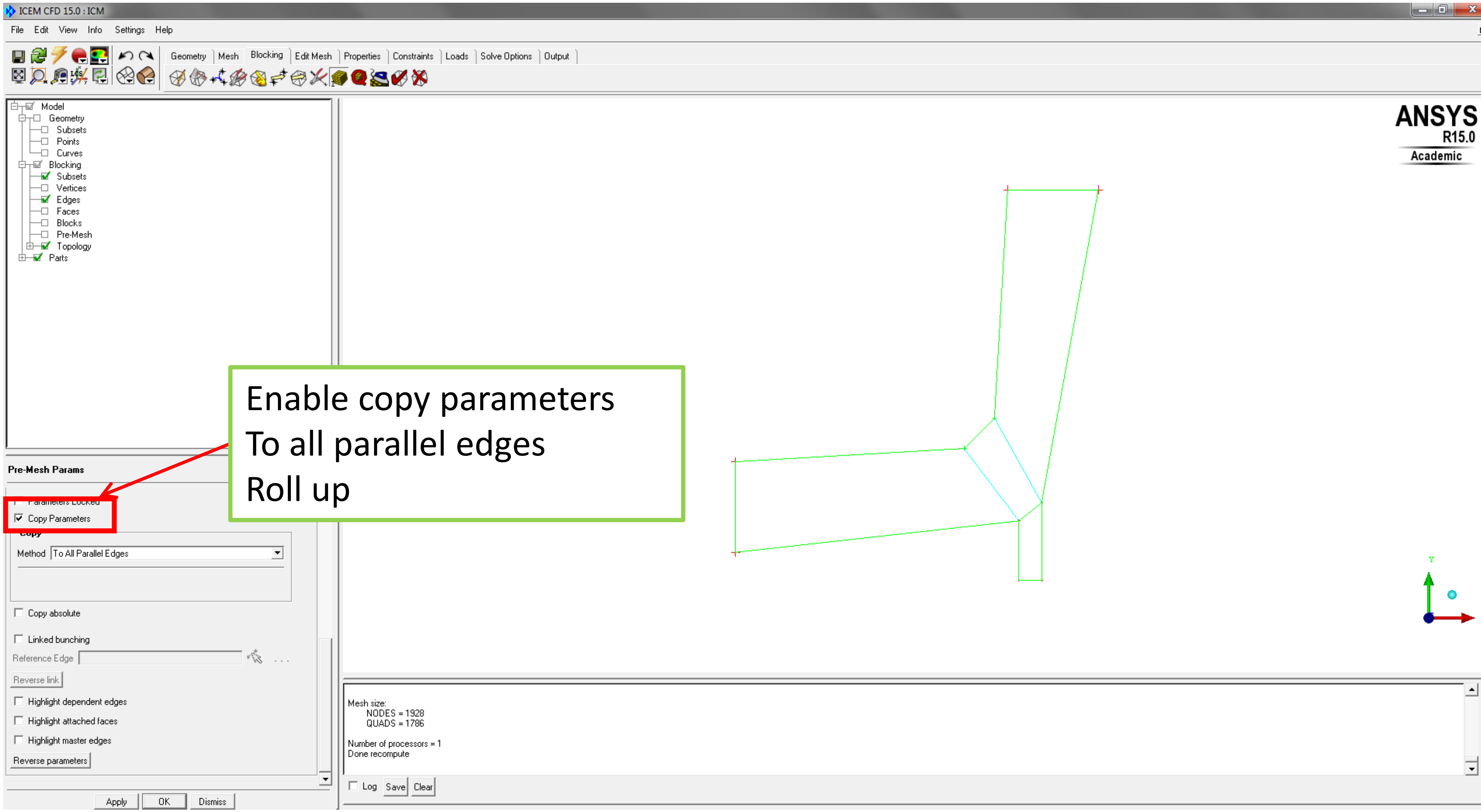
The premesh


ICEM CFD 15.0 : ICM
File Edit View Info Settings Help
Geometry Mesh Blocking Edit Mesh Properties Cons
Model
Geometry
Subsets
Points
Curves
Blocking
Subsets
Vertices
Edges
Faces
Pre-Mesh
Parts
Pre-Mesh Params
Meshing Parameters
Recalculate Sizes
Method
Update All
Keep Distributions
Keep Counts
Curve->Edge bunching
Run Check/Fix Blocks
Apply OK Dismiss
ANSYS
R15.0
Academic
Mesh size:
NODES = 1928
QUADS = 1786
Number of processors = 1
Done recompute
Log Save Clear
Enable premesh in the display tree

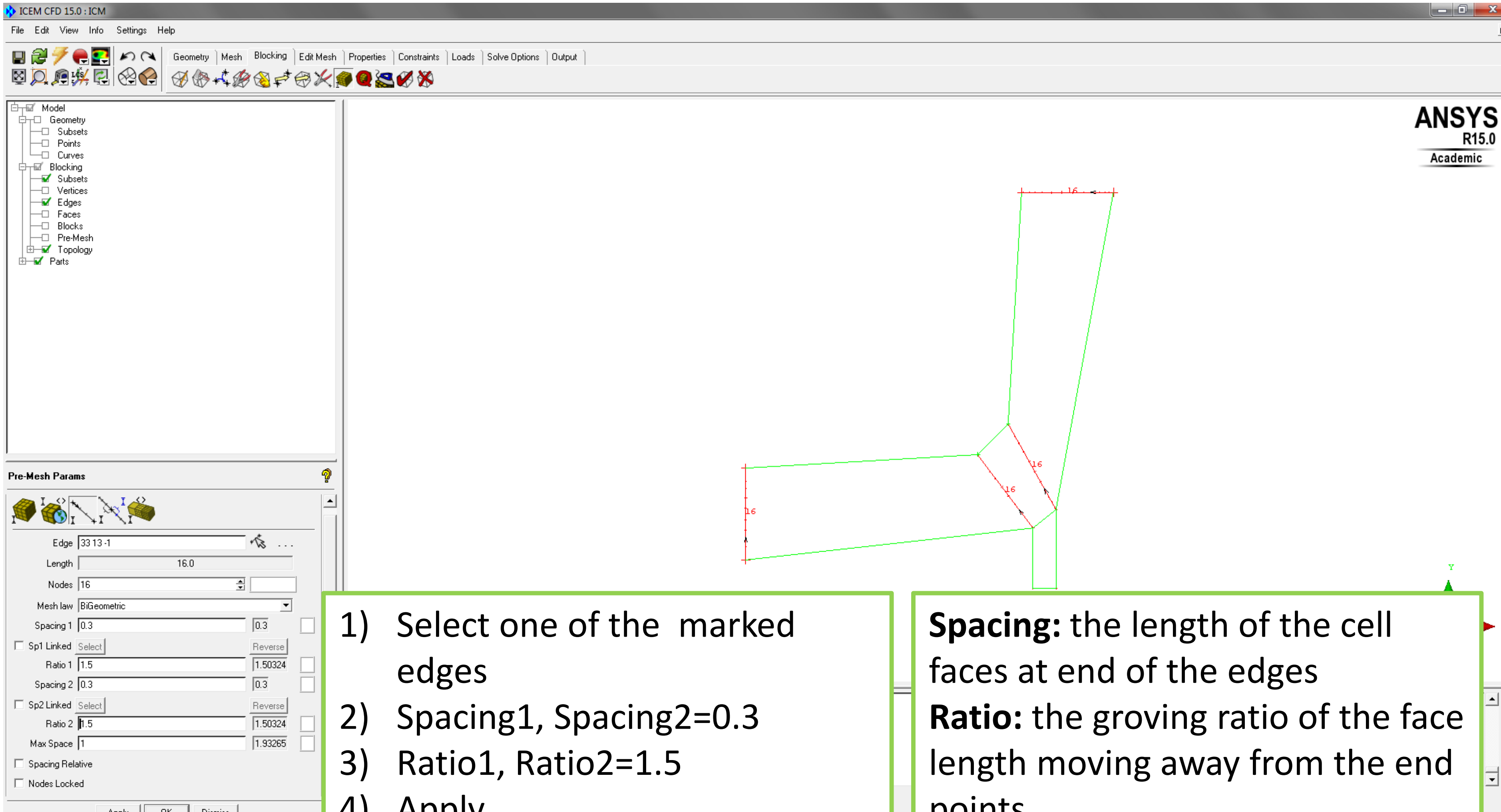
Refine the mesh



Refine the mesh



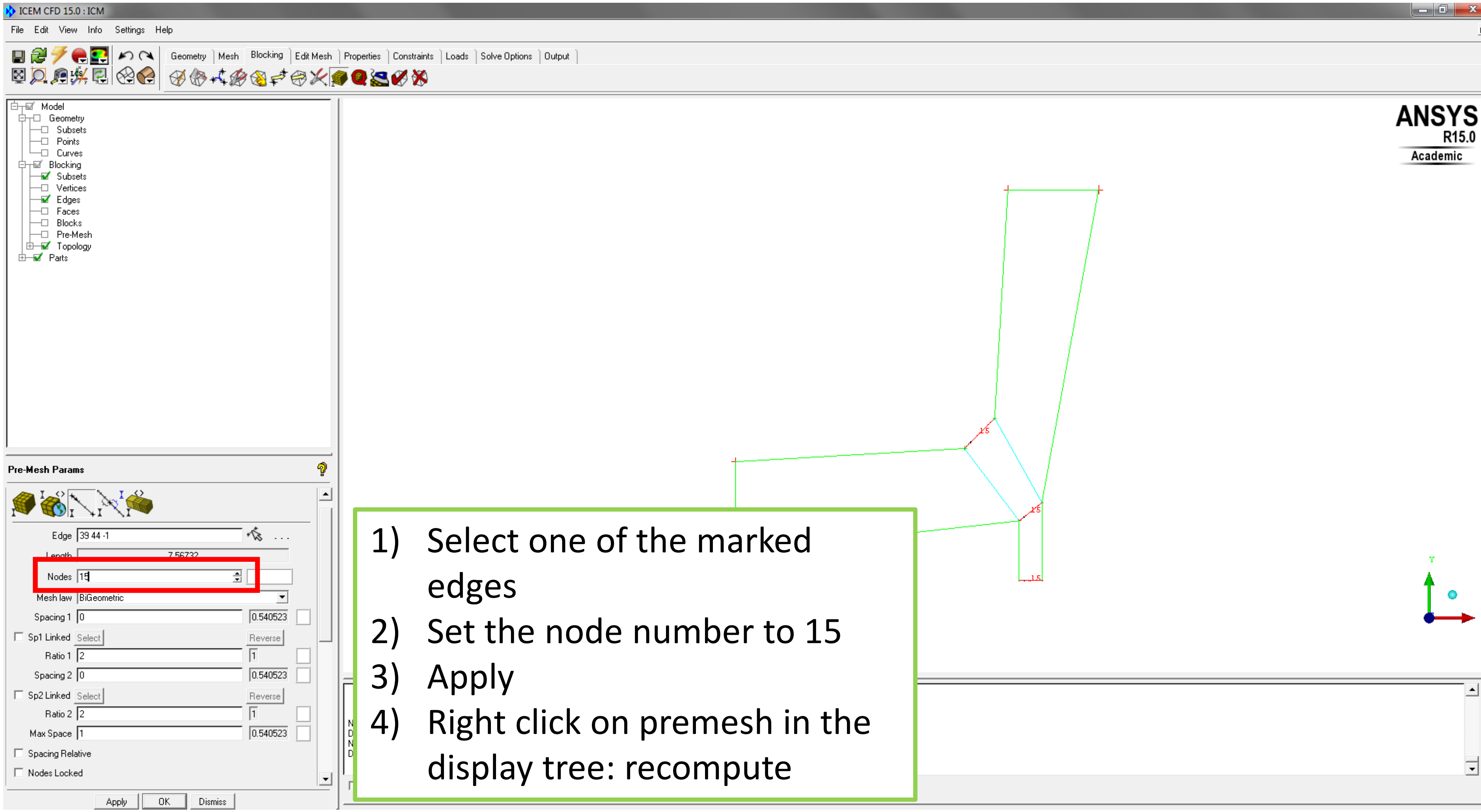
Refine the mesh



1) Select one of the marked edges
2) Spacing1, Spacing2=0.3
3) Ratio1, Ratio2=1.5
4) Apply
5) Right click on premesh in the display tree: recompute

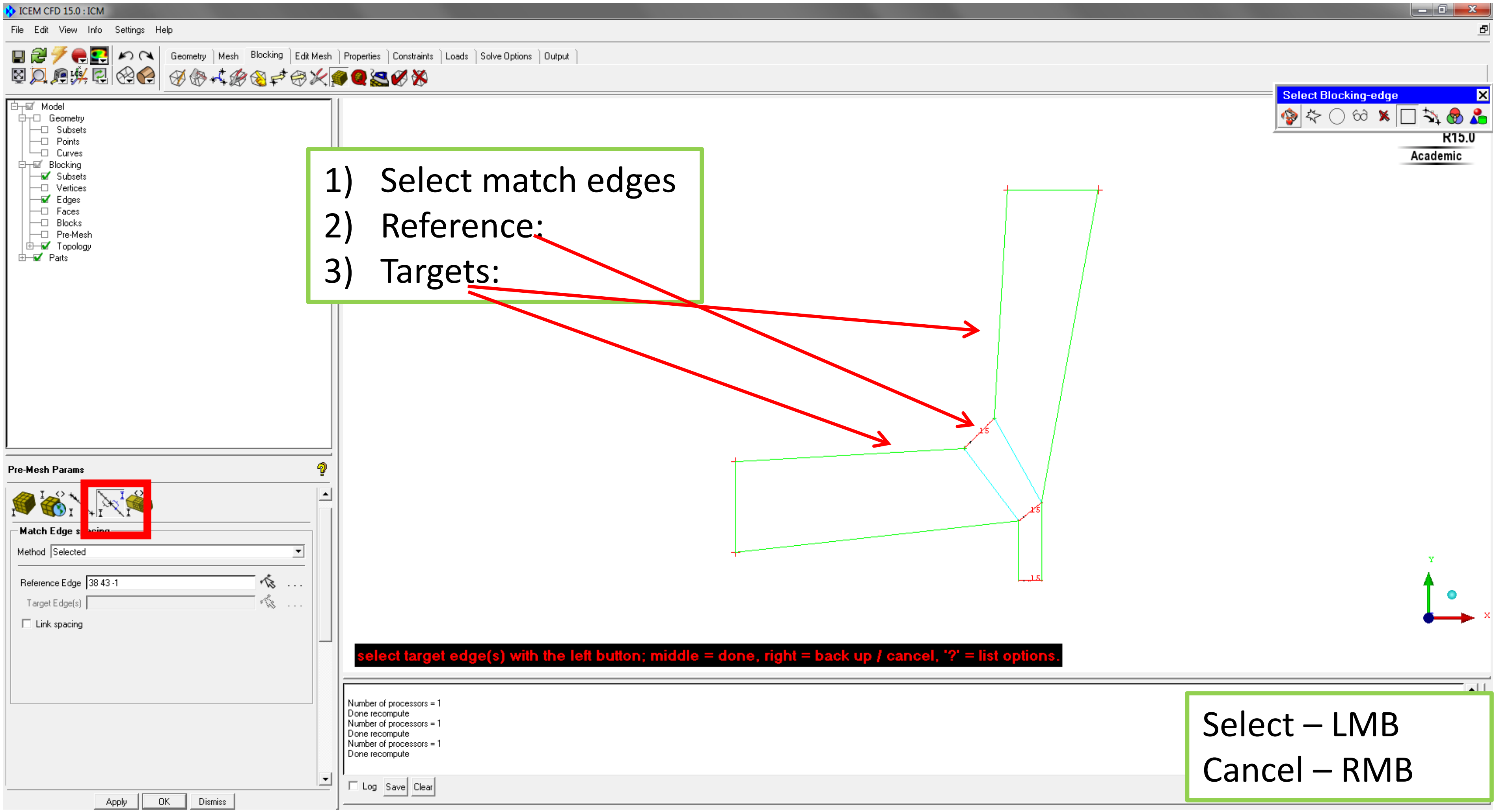
Spacing: the length of the cell faces at end of the edges
Ratio: the groving ratio of the face length moving away from the end points

Refine the mesh

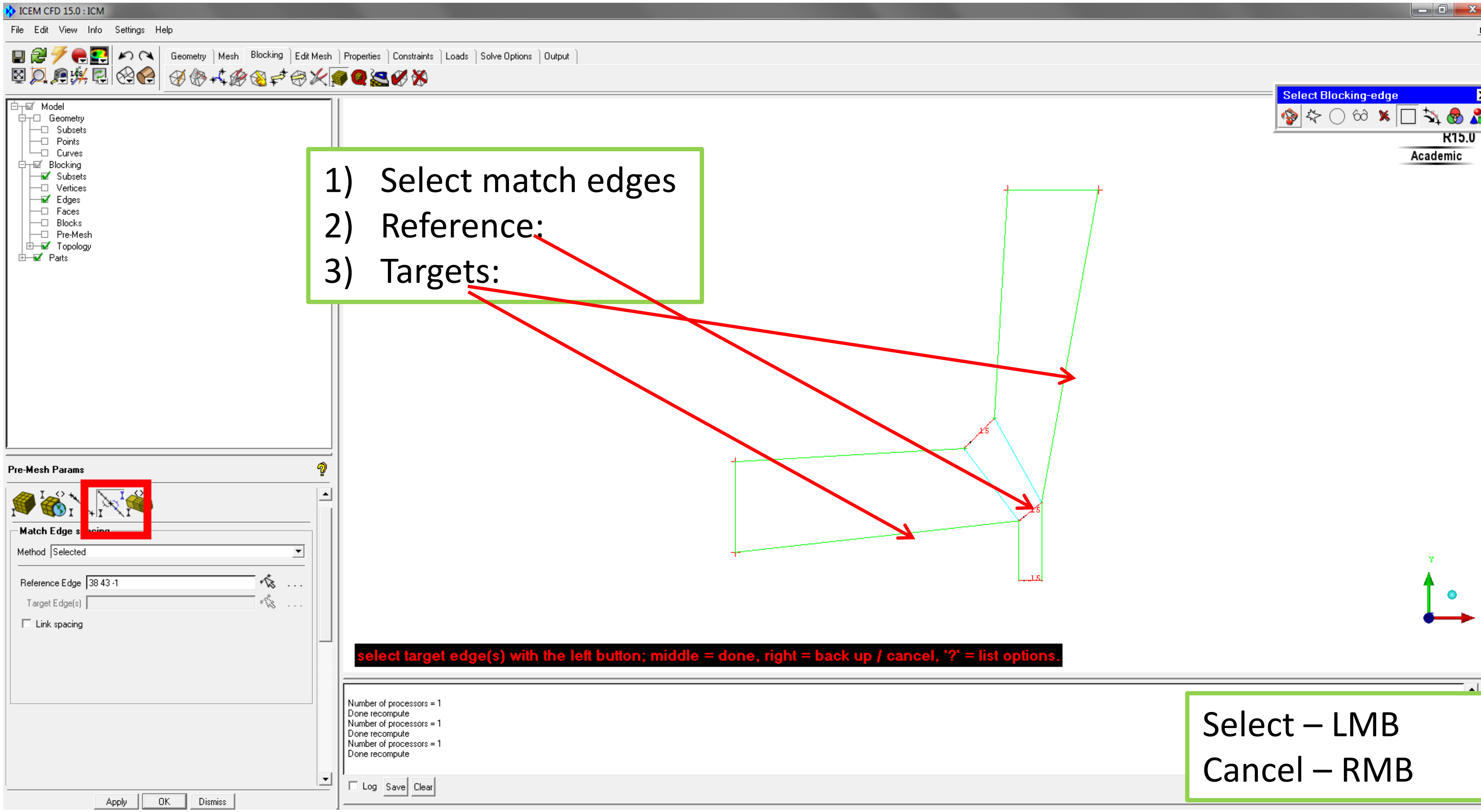


1) Select one of the marked edges
2) Set the node number to 15
3) Apply
4) Right click on premesh in the display tree: recompute

Reducing the cell size change



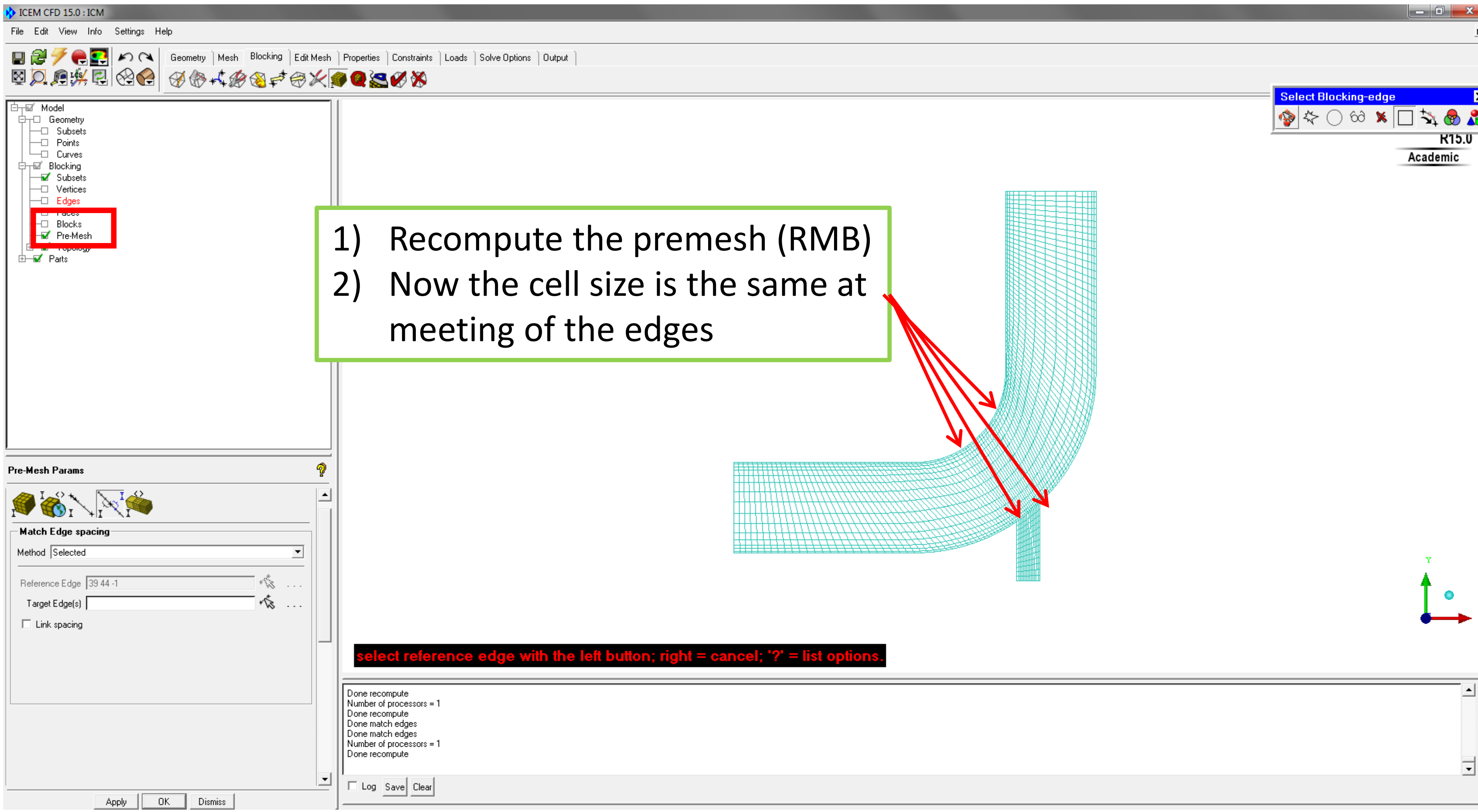
Reducing the cell size change



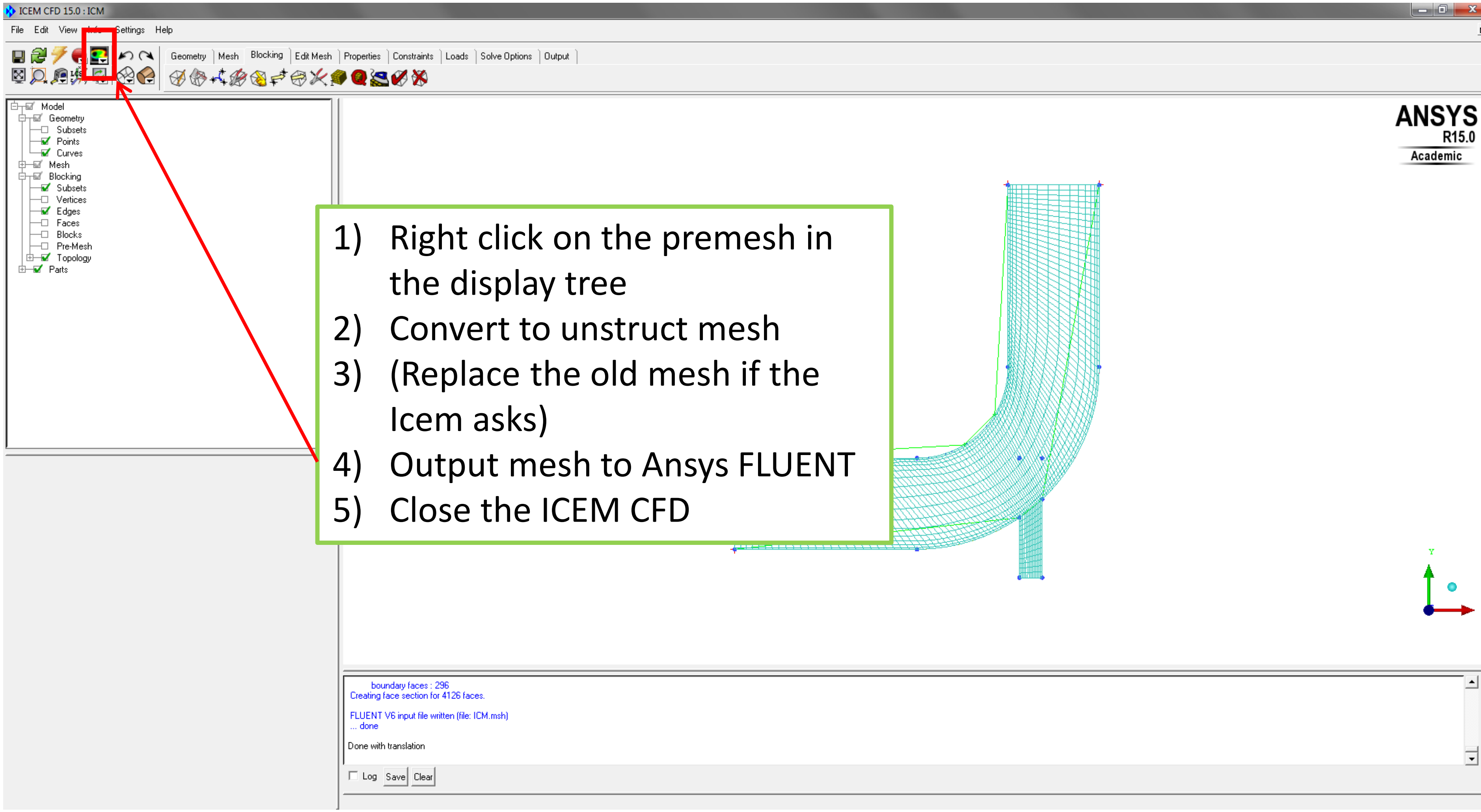
1) Select match edges
2) Reference:
3) Targets:

Select – LMB
Cancel – RMB

Reducing the cell size change



Create FLUENT input file



1) Right click on the premesh in the display tree
2) Convert to unstruct mesh
3) (Replace the old mesh if the Icem asks)
4) Output mesh to Ansys FLUENT
5) Close the ICEM CFD

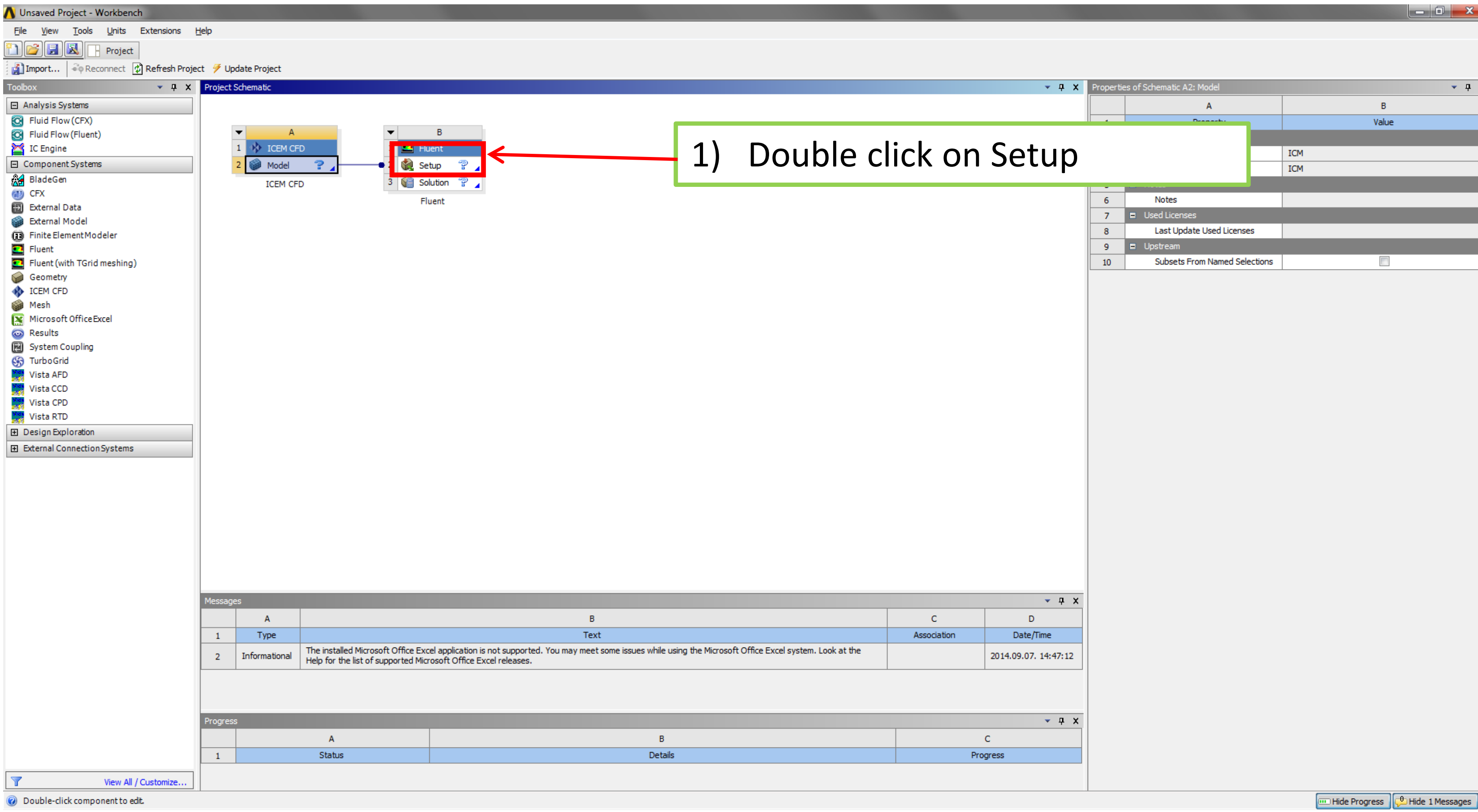
Physical model, Solution of the problem

- For more detailed descriptions of FLUENT settings check (from page 36):

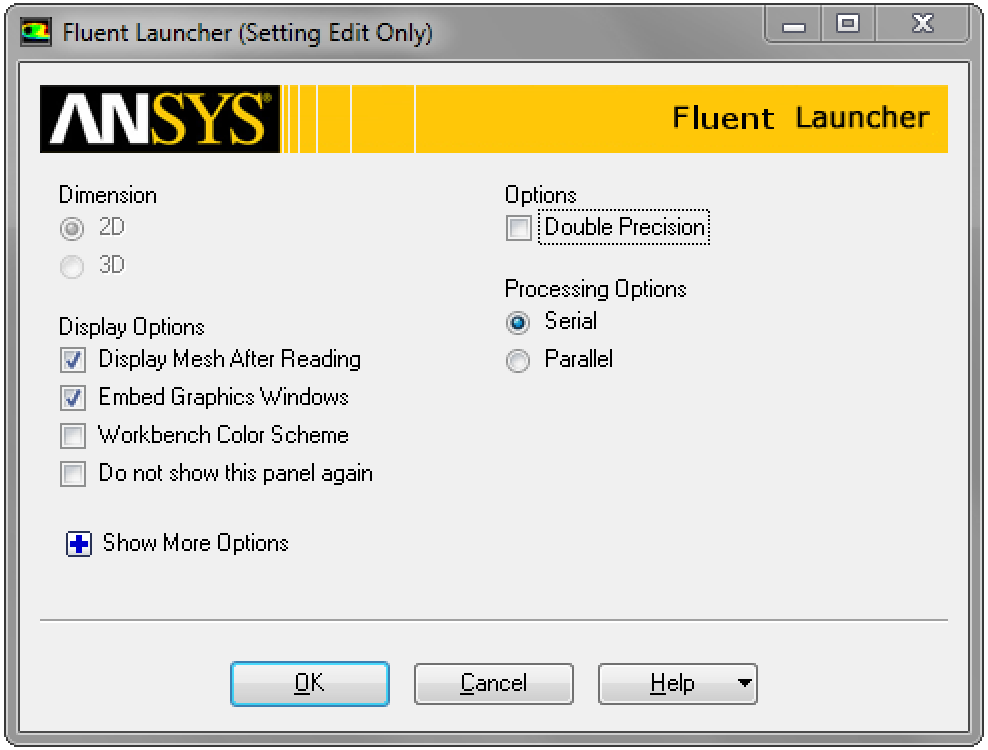
http://www.ara.bme.hu/~benedek/CFD/workbench/workbench_1st_practice/orifice_1415_eng.pdf (orifice.pdf in hungarian)

- Or ask your instructor
- Or ask the lecturer

Starting FLUENT

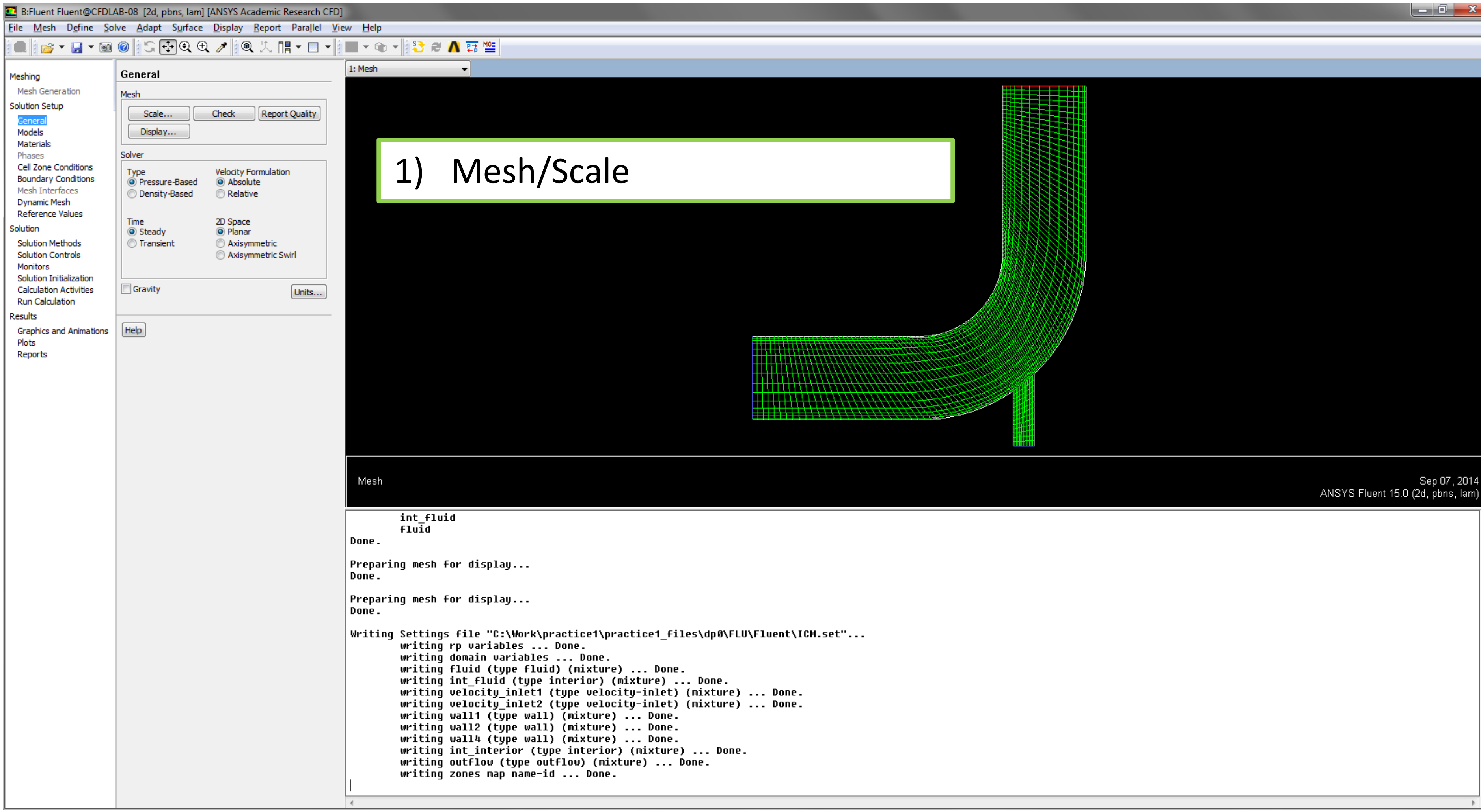


Starting FLUENT

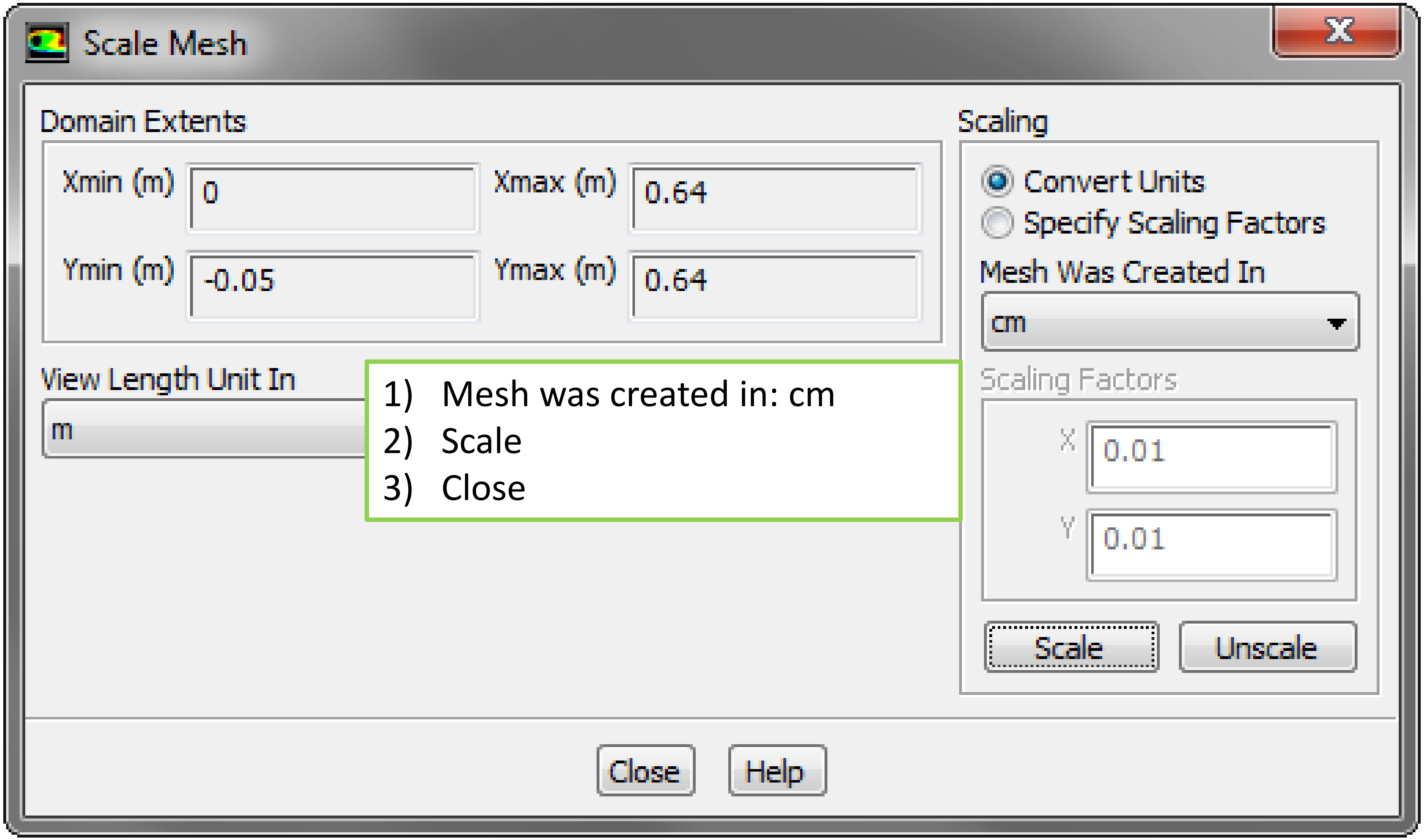


OK

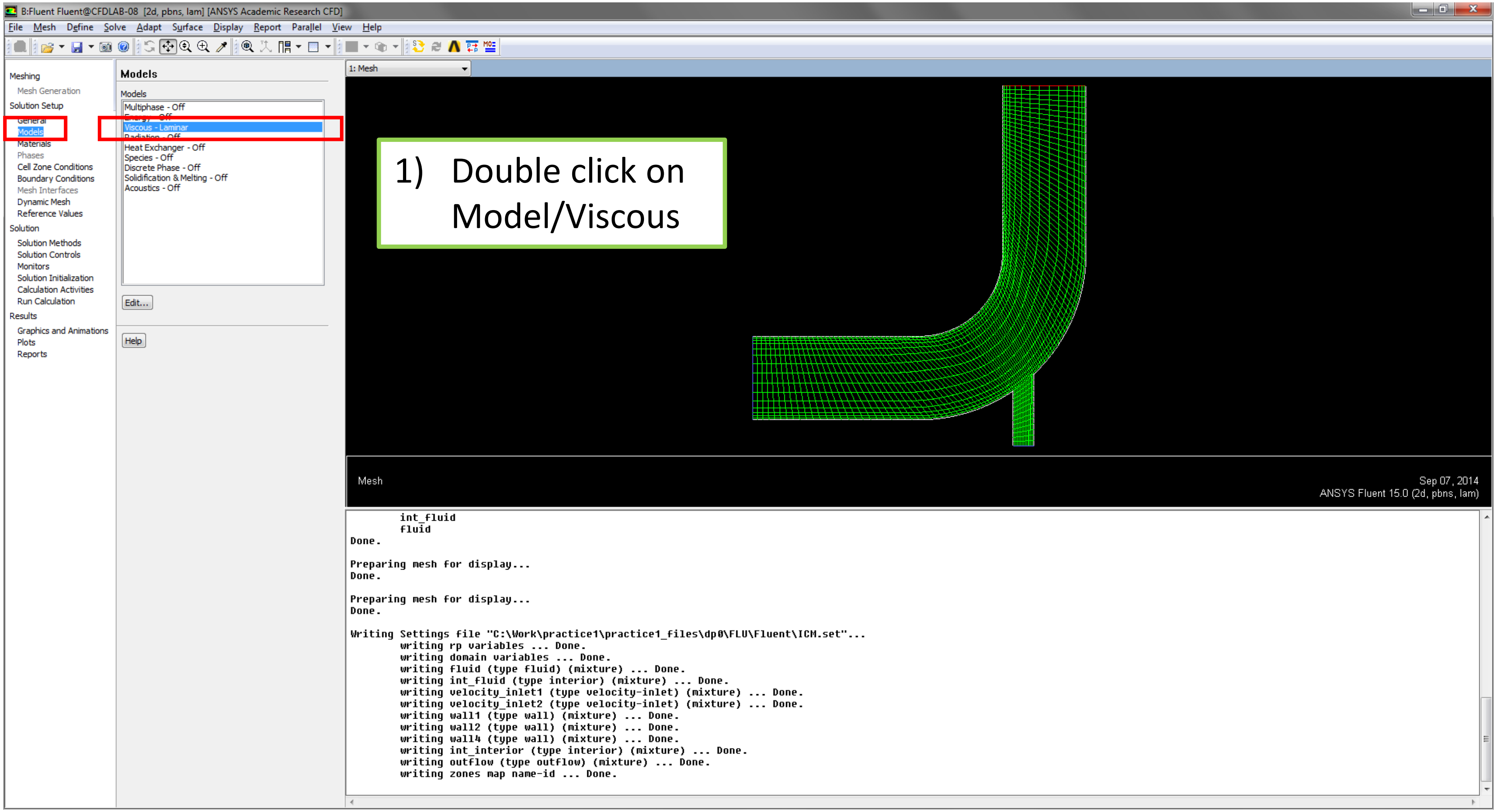
Scaling the mesh



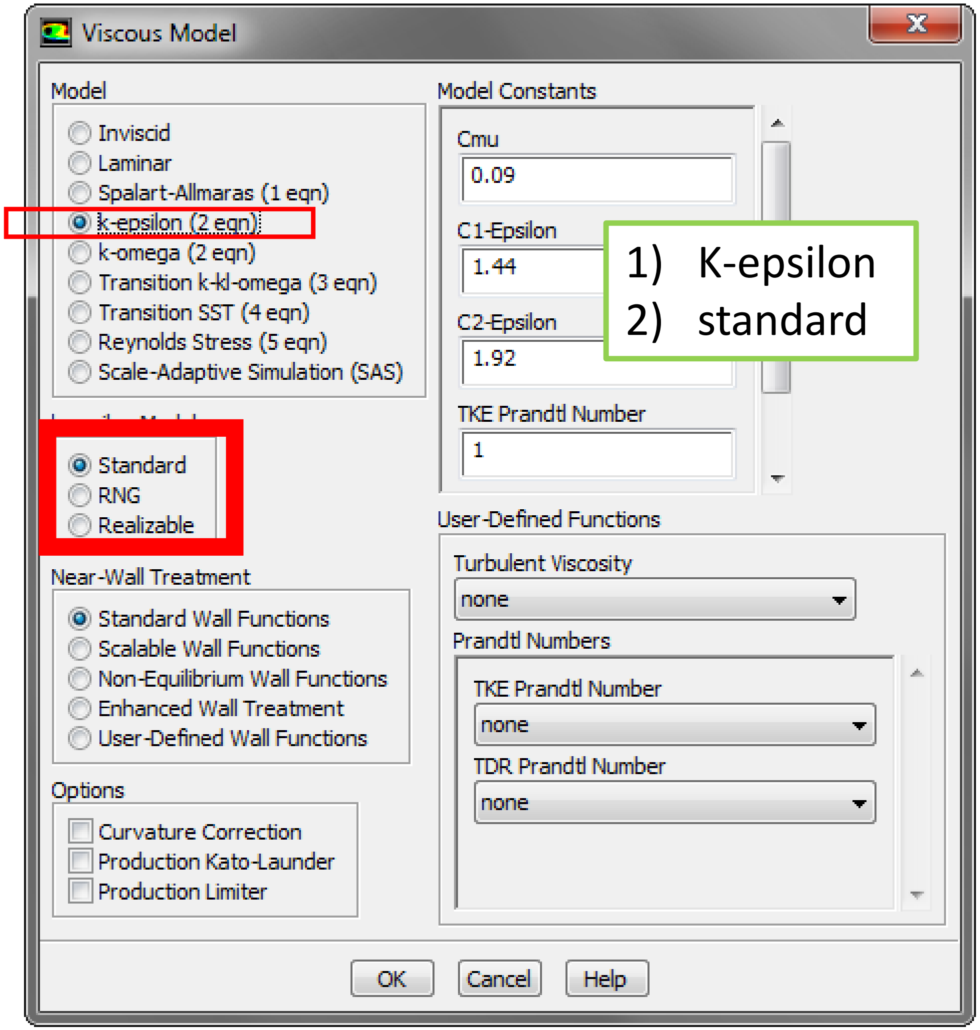
Scaling the mesh



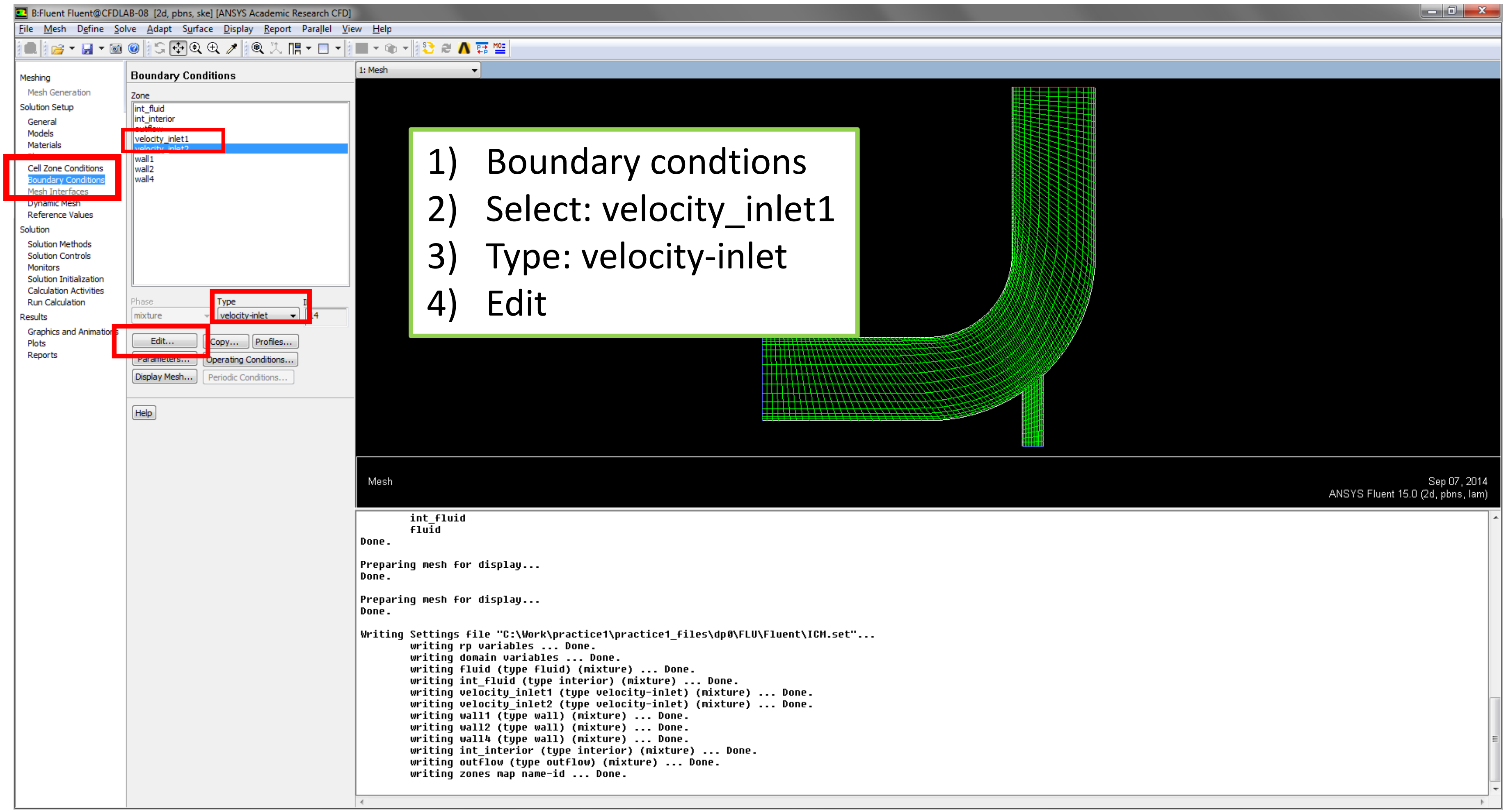
Turbulence Model



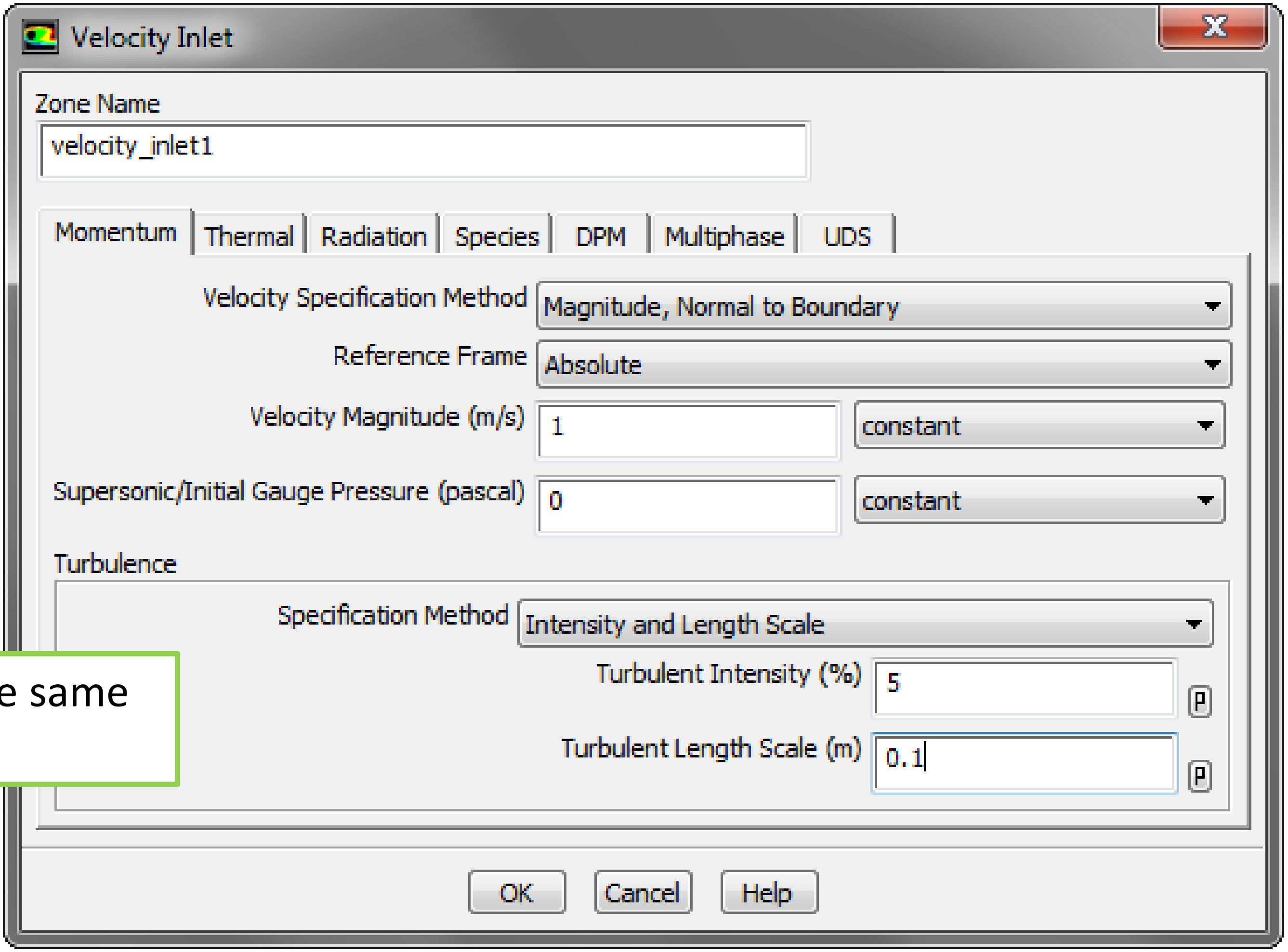
Turbulence model



Boundary conditions

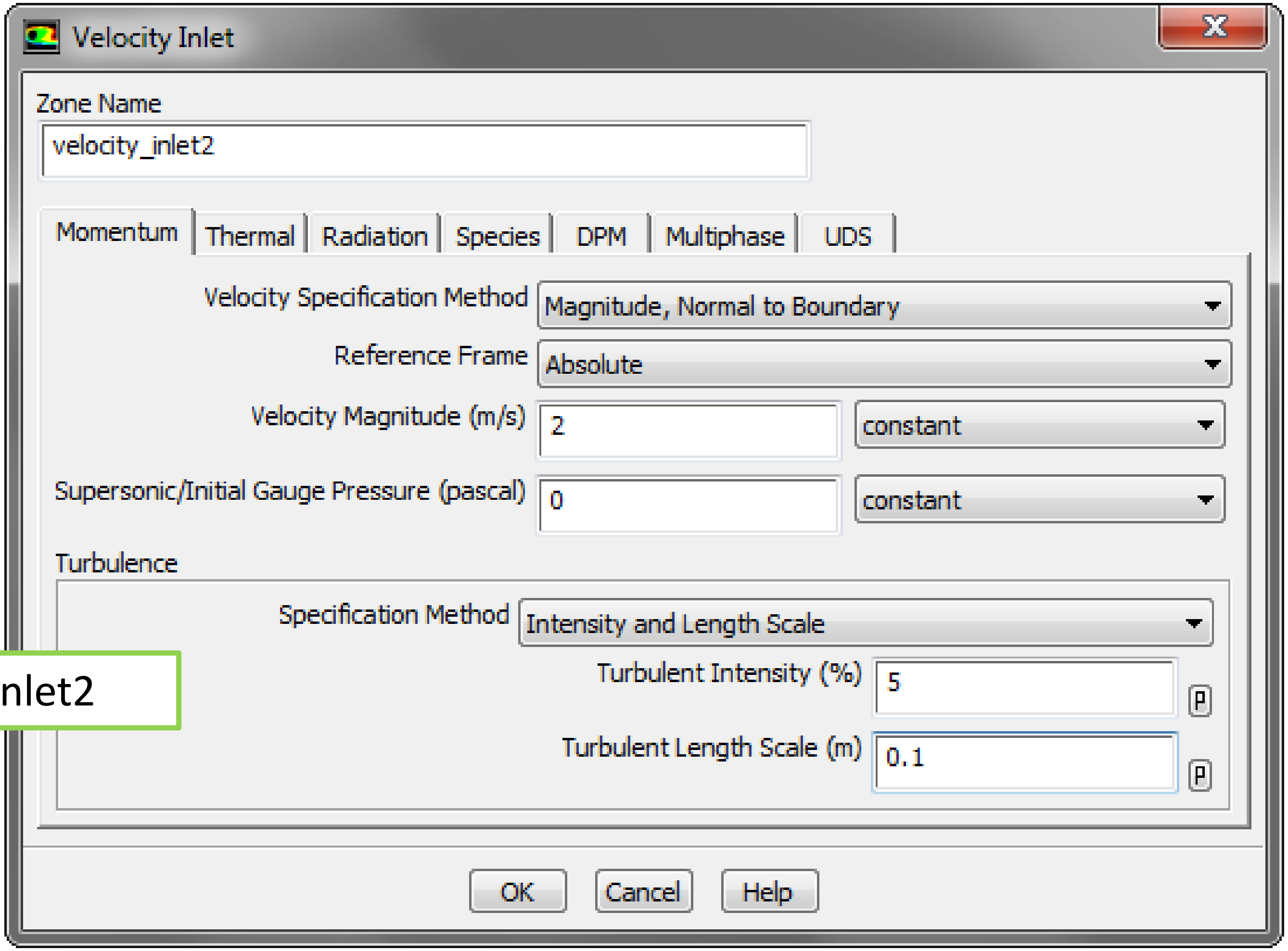


Boundary conditions



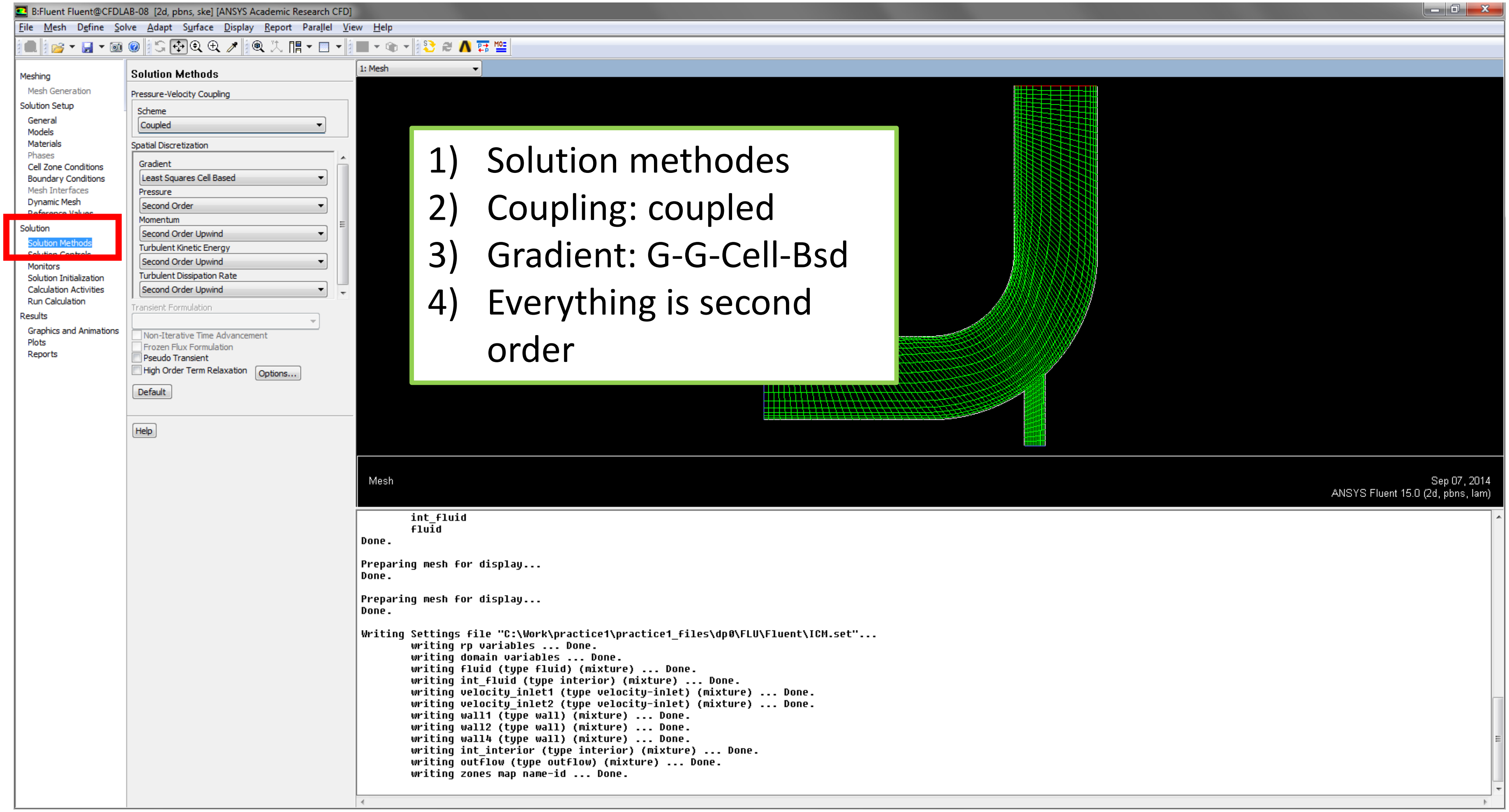
1) Set the same
2) OK

Boundary conditions

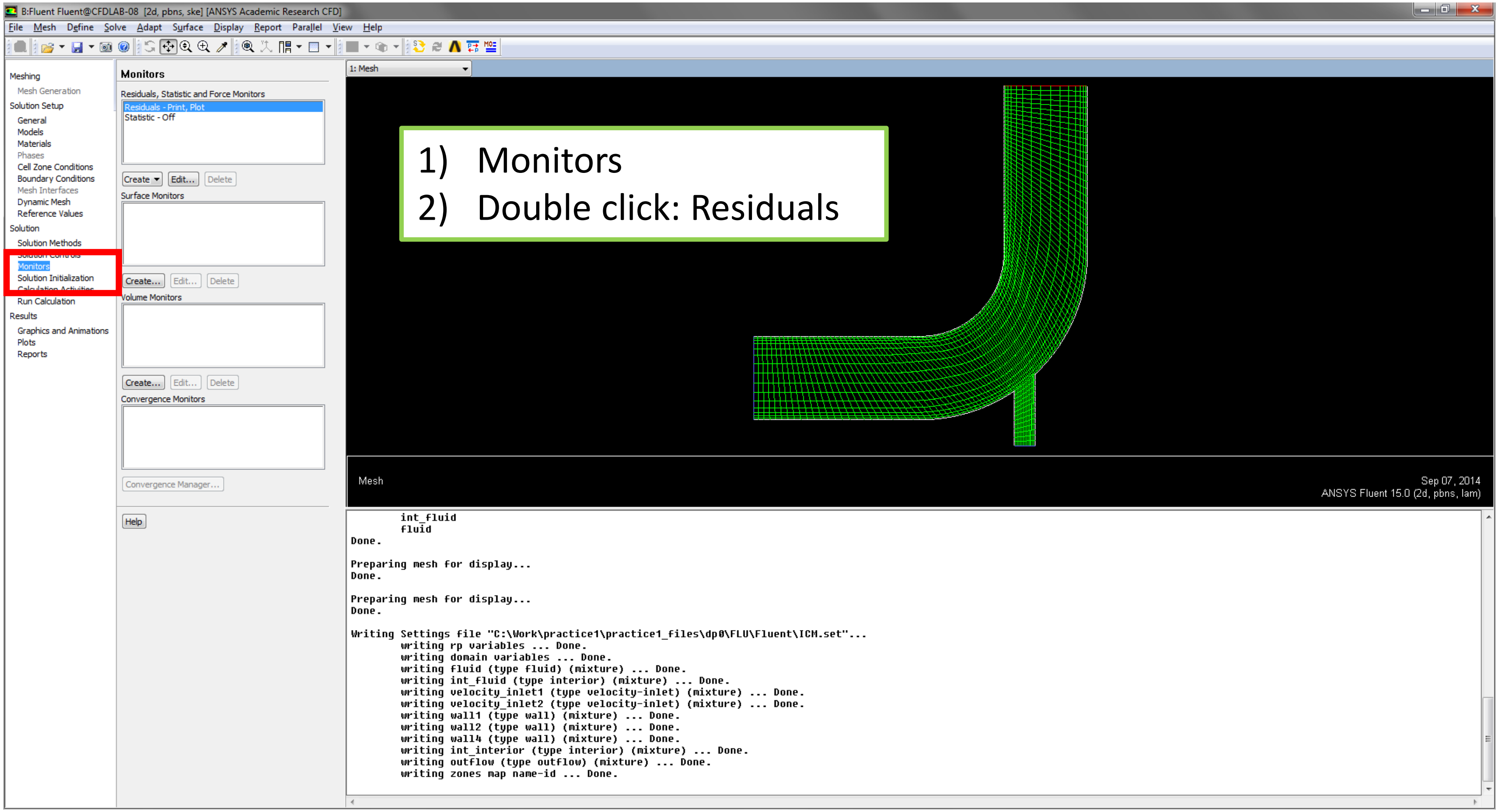


Velocity_inlet2

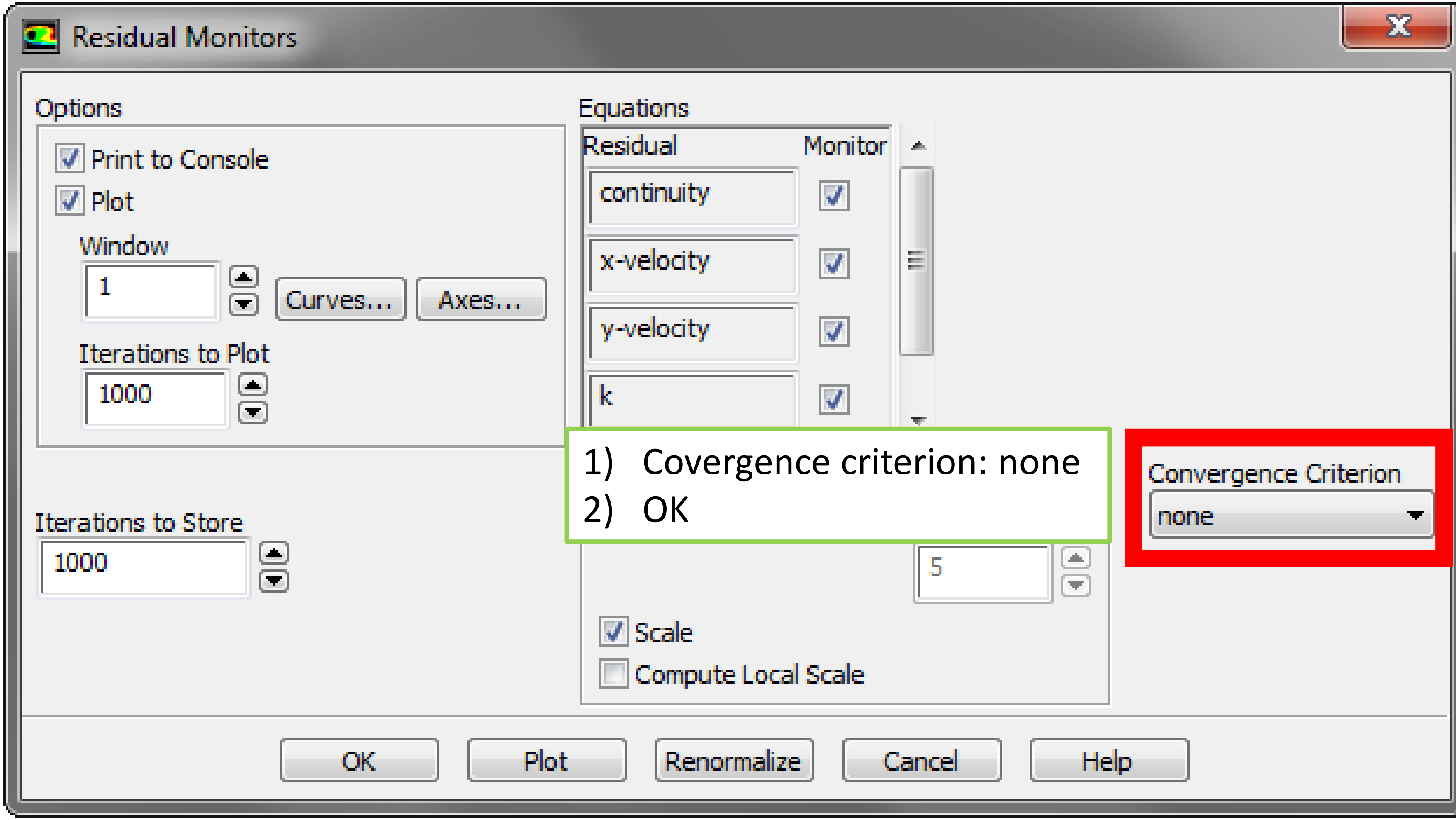
Discretization Schemes



Turn off convergence criterion

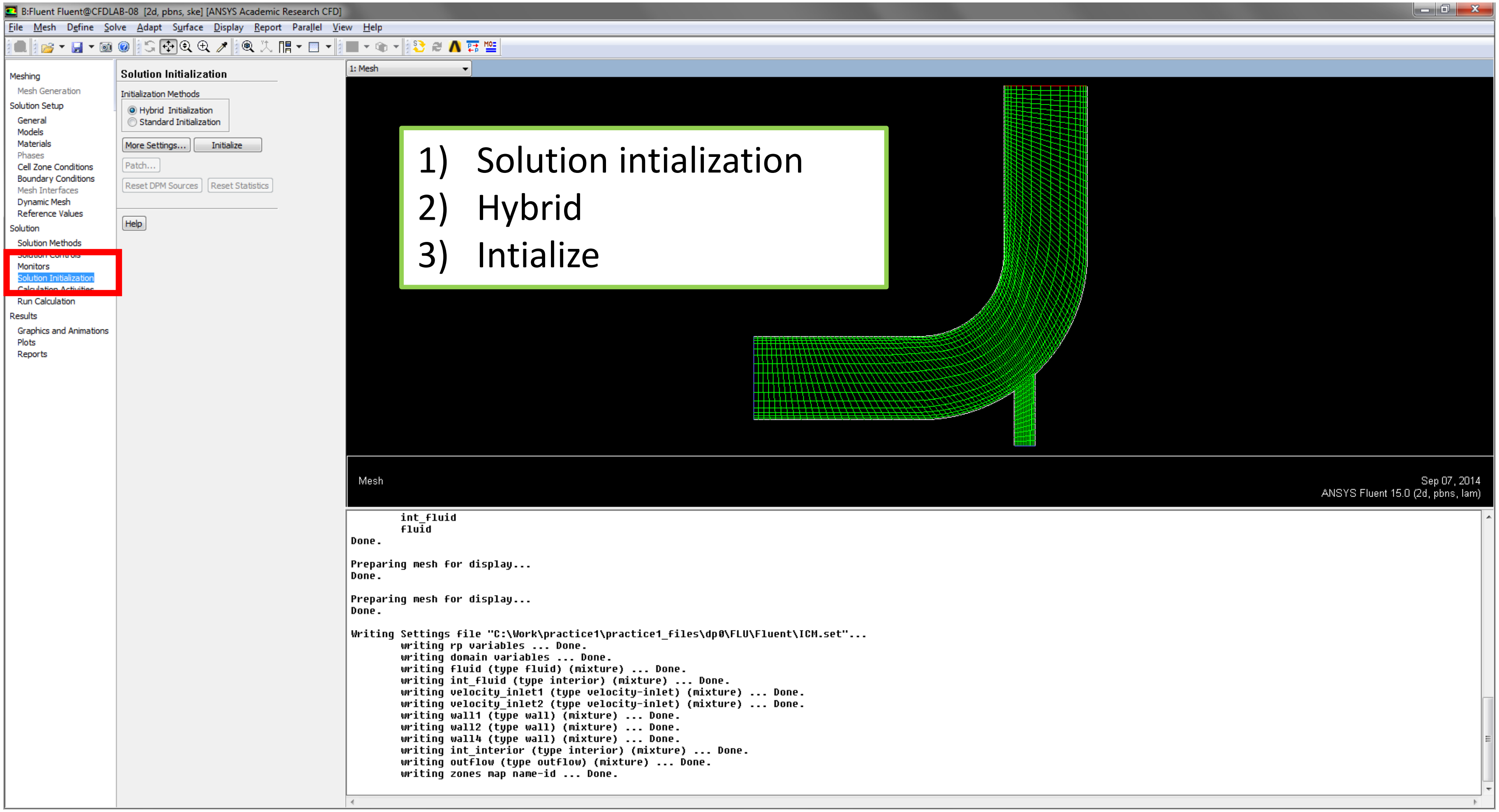


Turn off convergence criterions

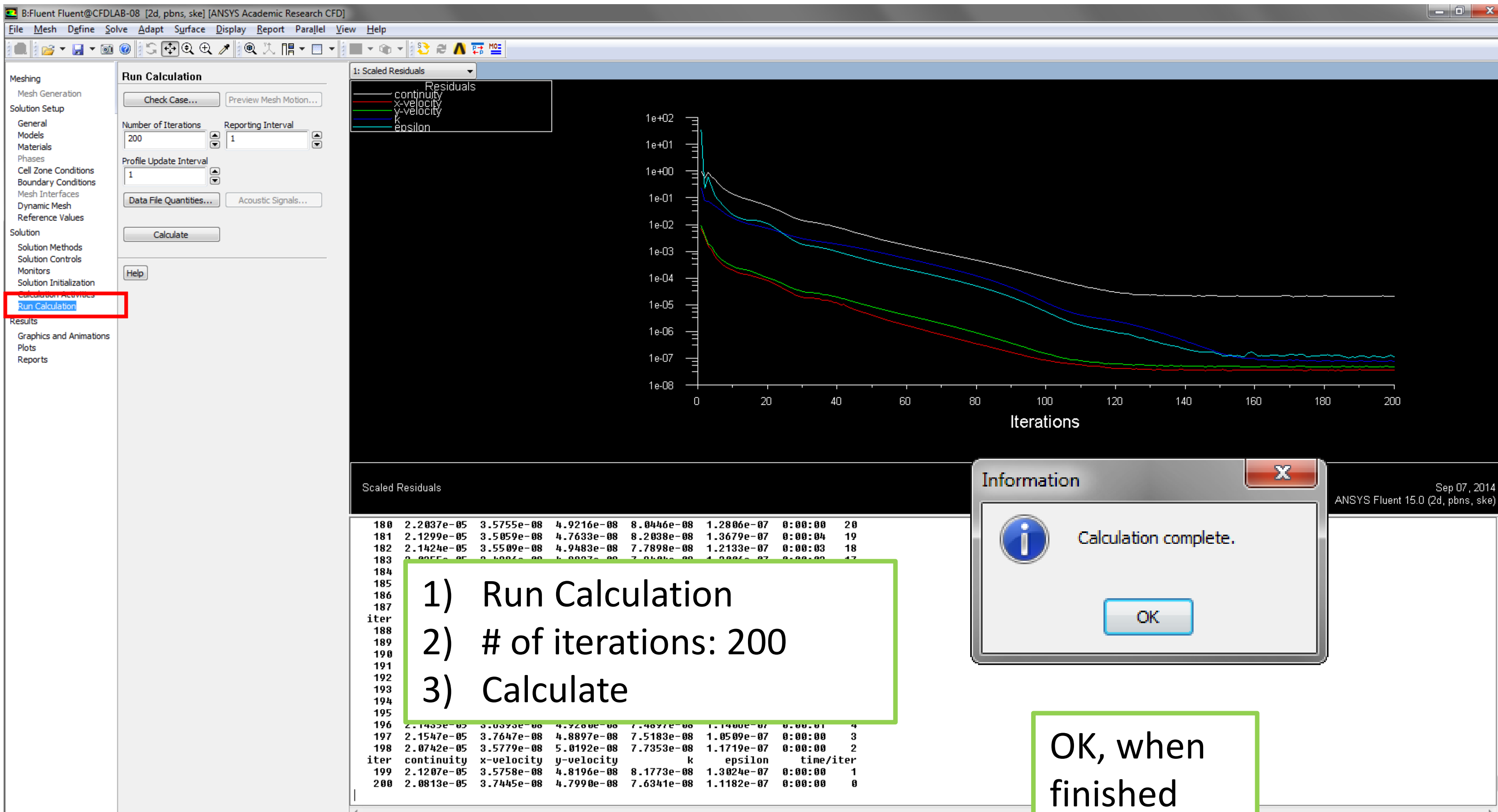


1) Covergence criterion: none
2) OK

Intialization



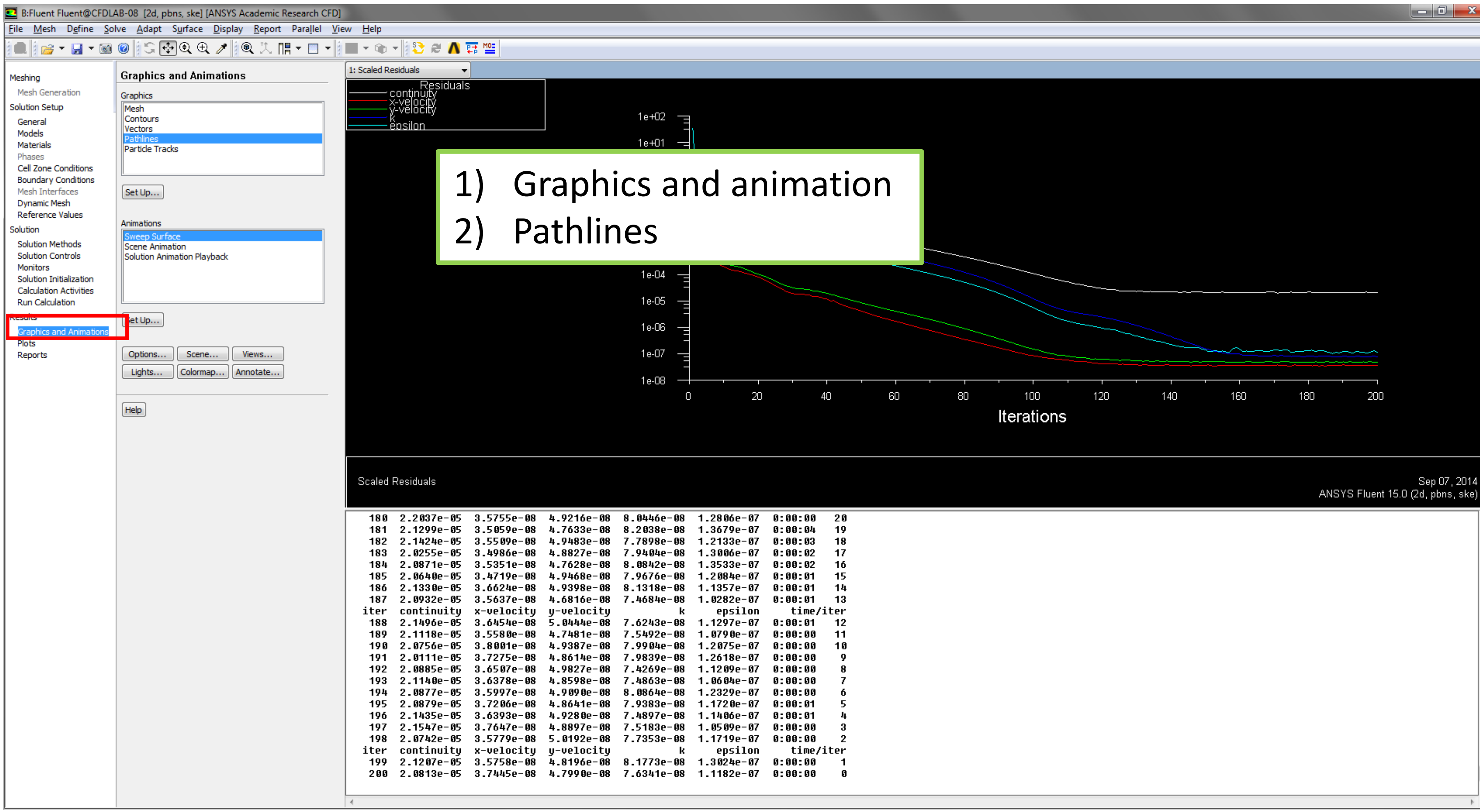
Calculation



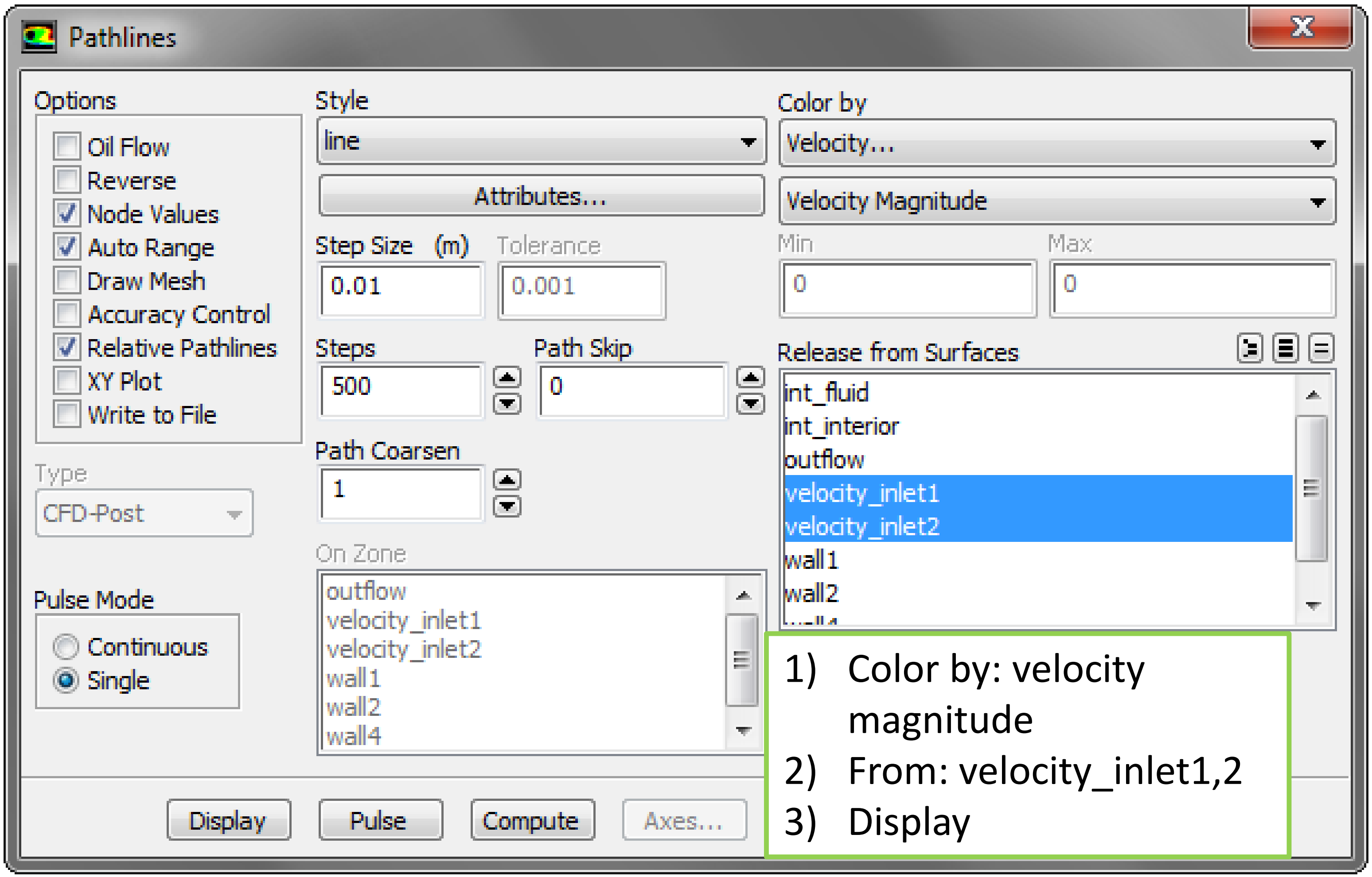
1) Run Calculation
2) # of iterations: 200
3) Calculate

OK, when finished

Drawing pathlines

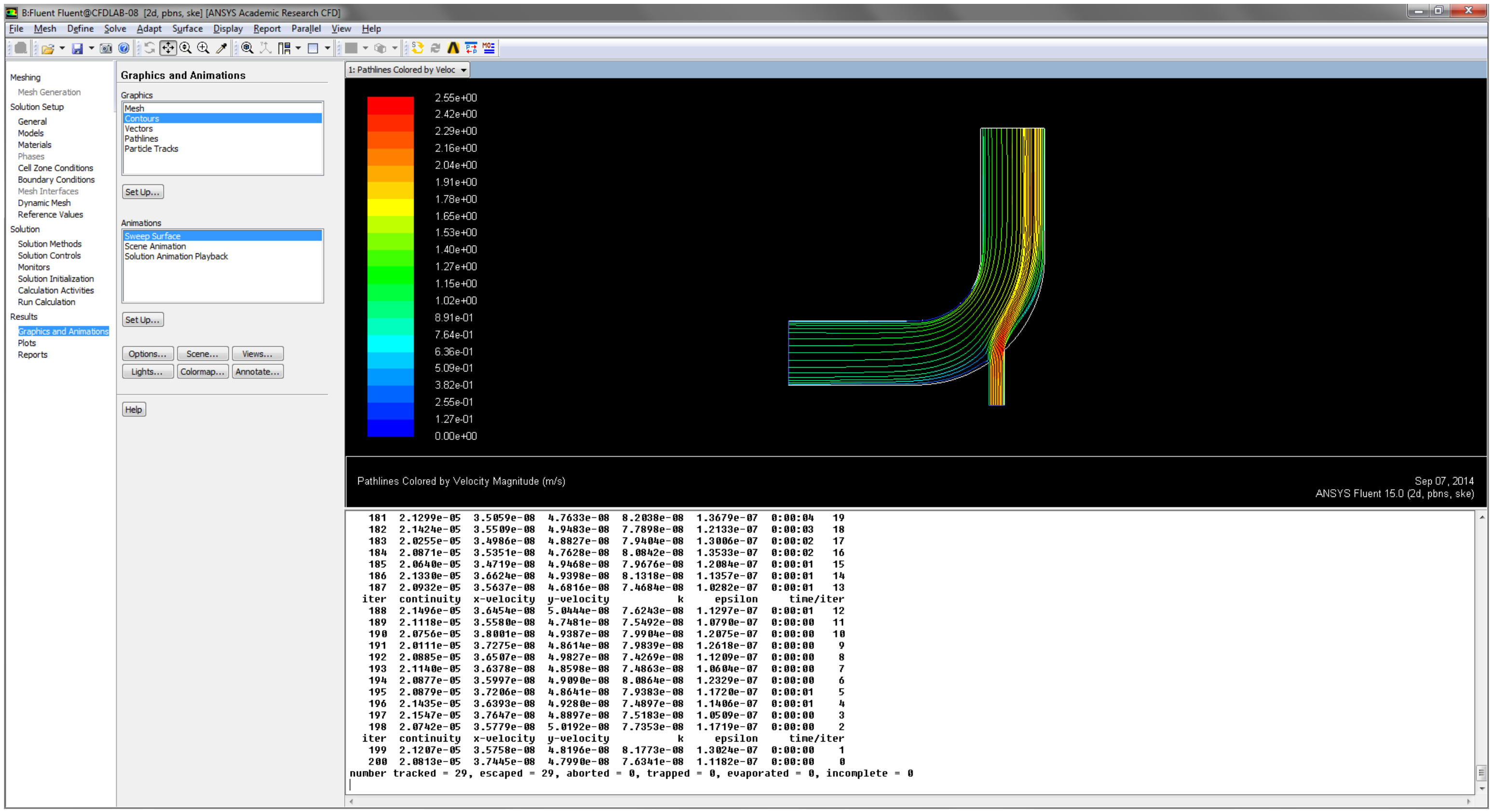


Drawing pathlines

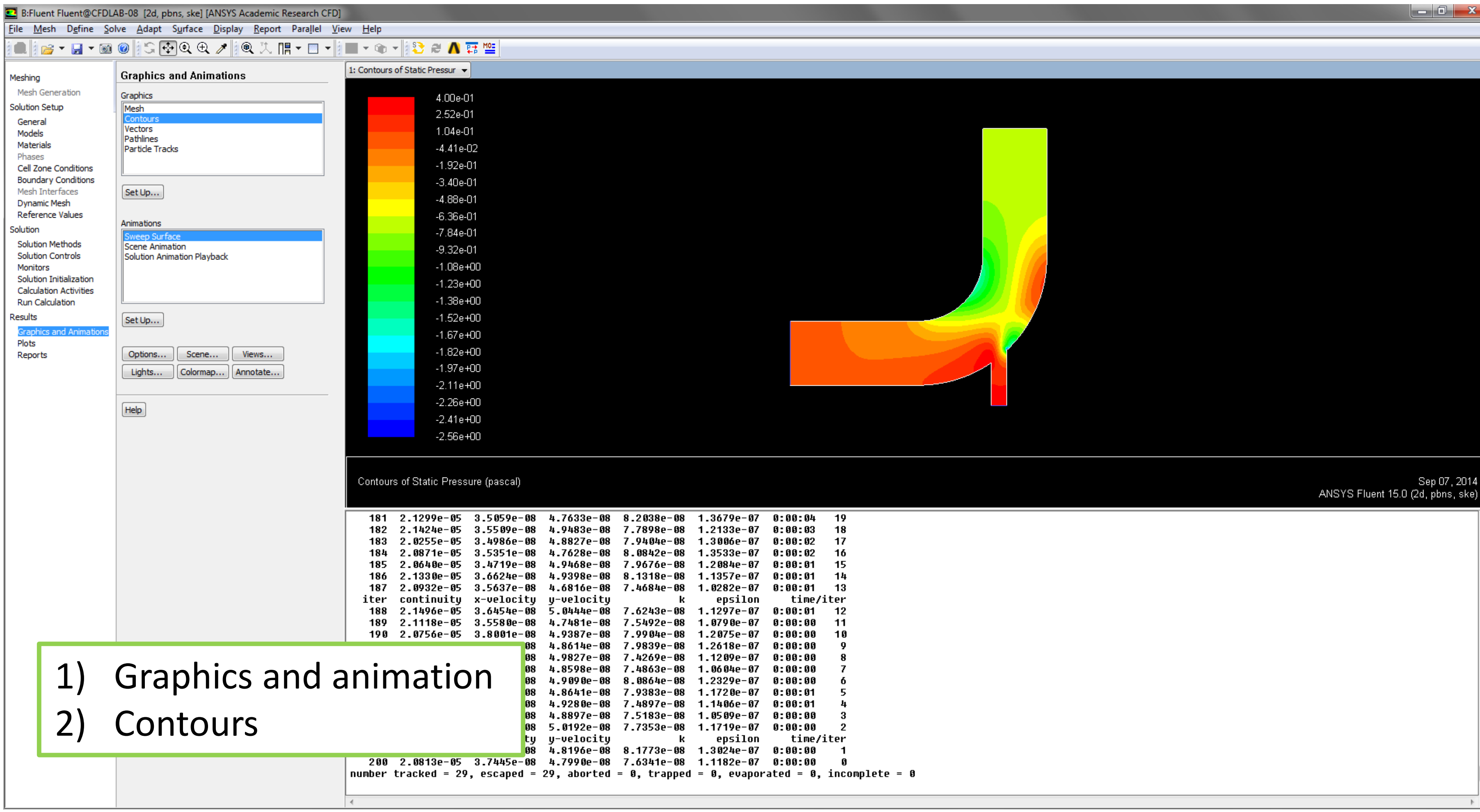


1) Color by: velocity magnitude
2) From: velocity_inlet1,2
3) Display

Drawing pathlines

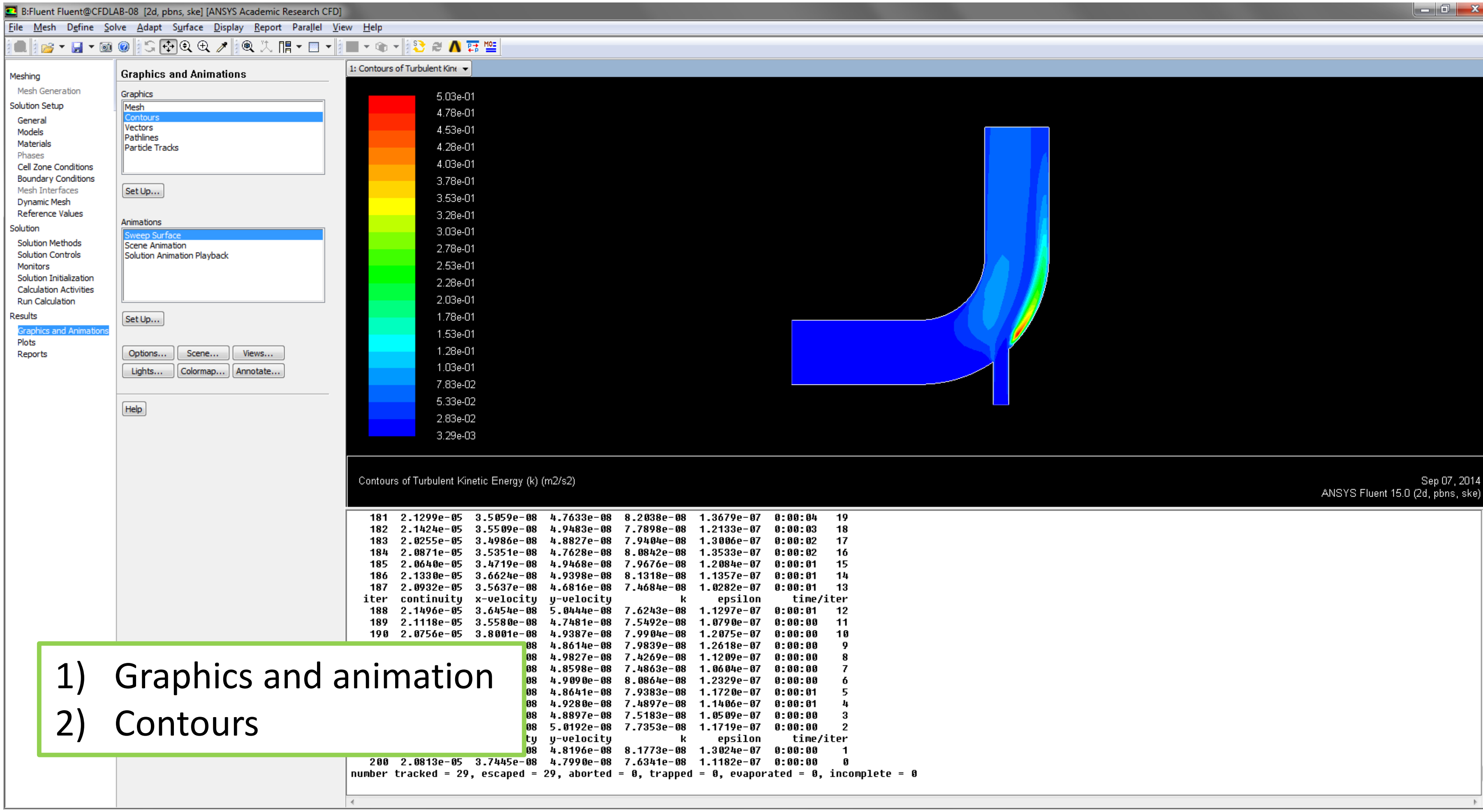


Contour plots: pressure



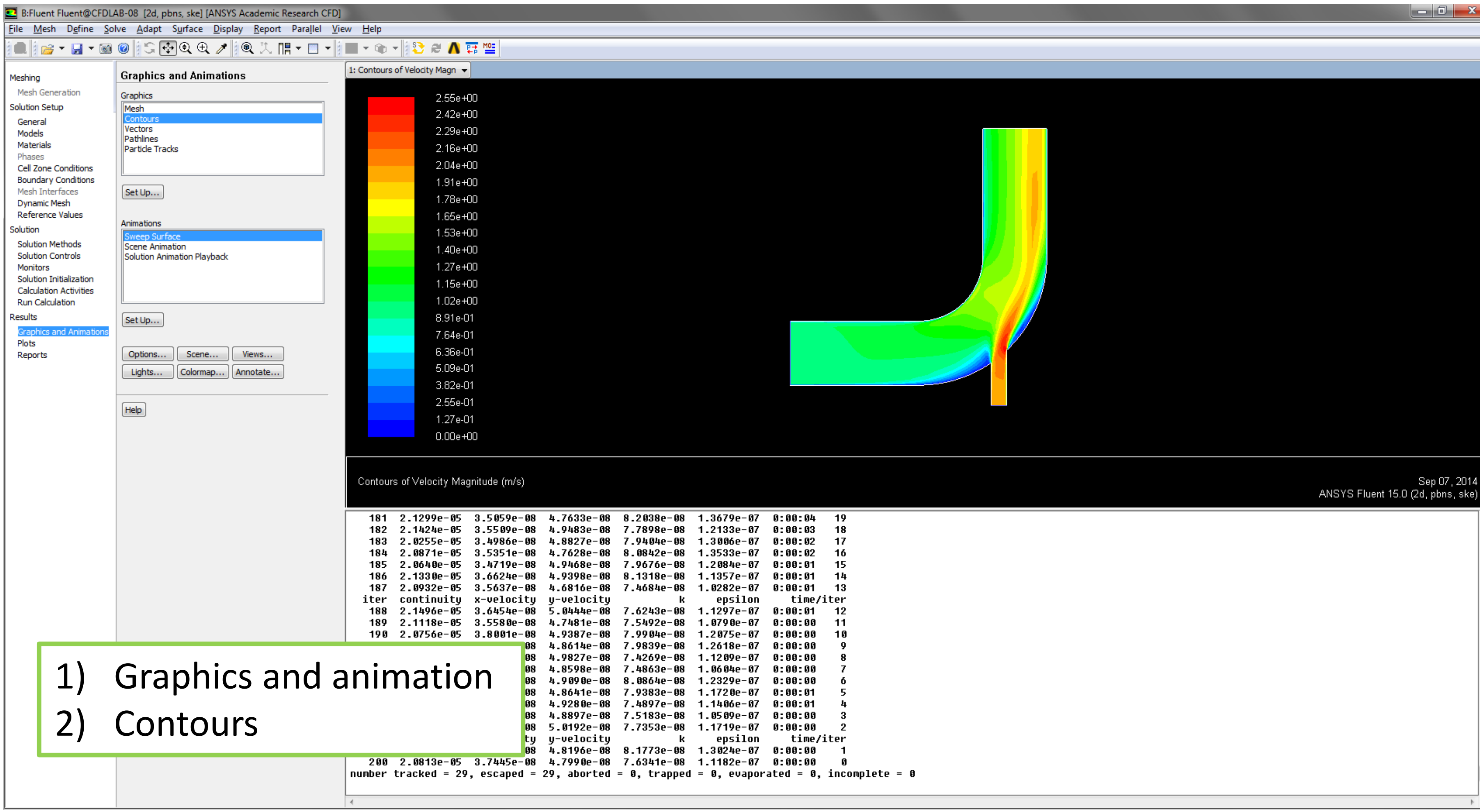
1) Graphics and animation
2) Contours

Contour plots: k



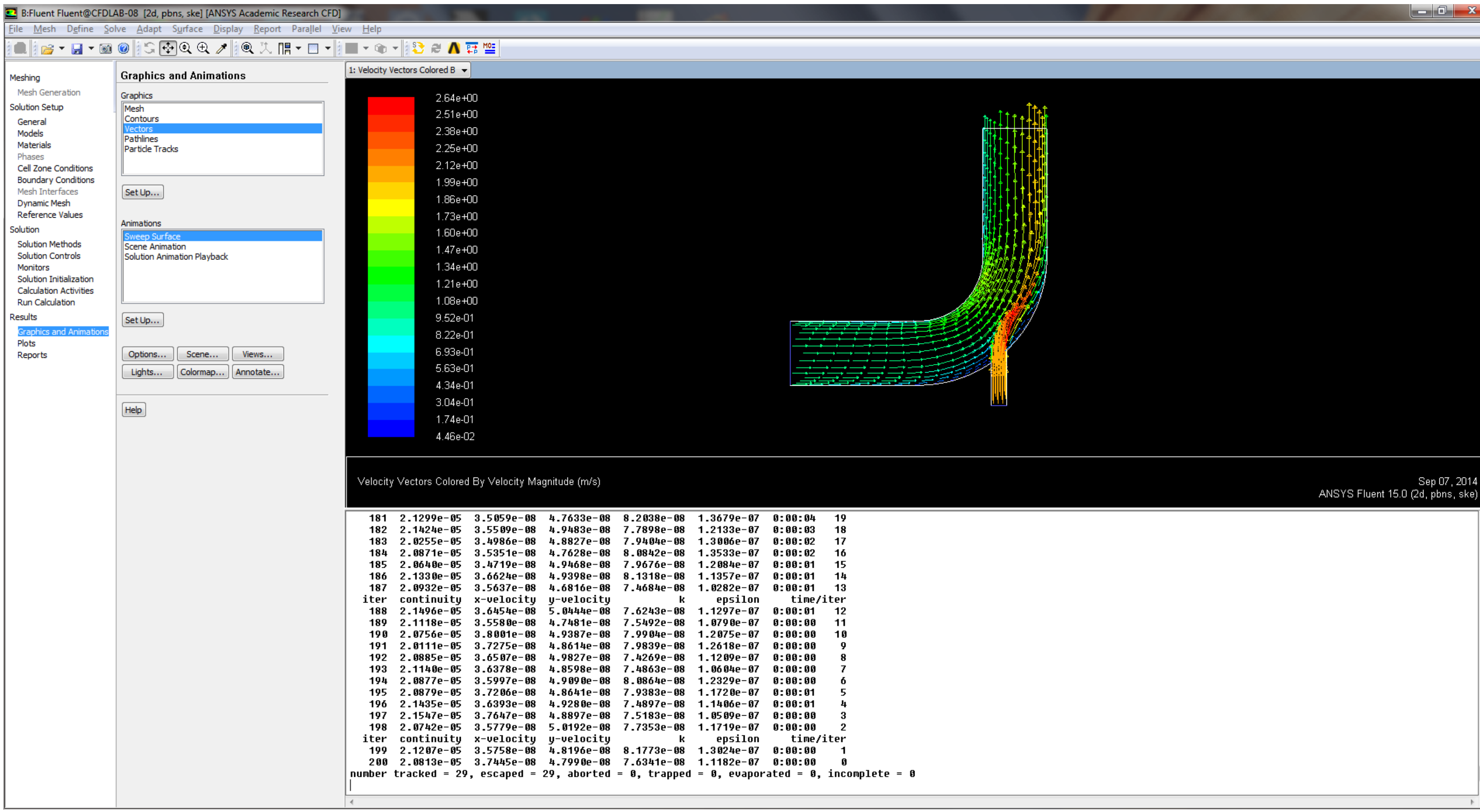
1) Graphics and animation
2) Contours

Contour plots: velocity



1) Graphics and animation
2) Contours

Velocity vectors




That's all Folks!