

# Computational Fluid Dynamics

2019 autumn, 1st week

- Tamás Benedek
- [benedek \[at\] ara.bme.hu](mailto:benedek[at]ara.bme.hu)
- [www.ara.bme.hu/~benedek/CFD/icem](http://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 )

Another important rule:

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

CAD model  
(**ICEM**/Design  
Modeler/Import)

Mesh generation  
(**ICEM**)

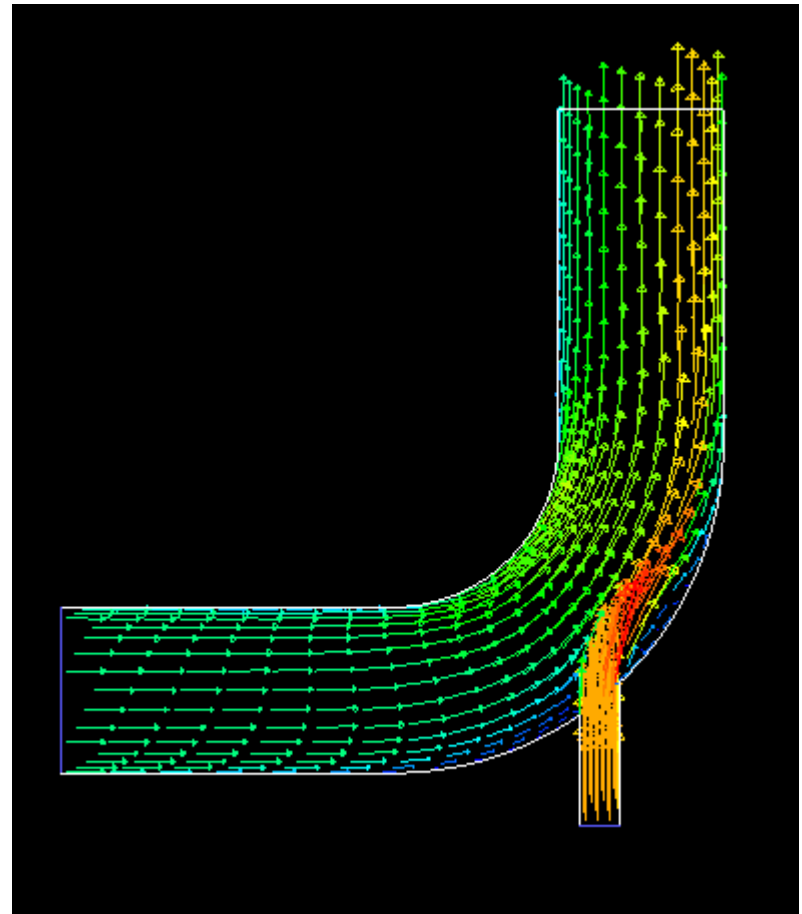
Solver  
(FLUENT)

Postprocessing  
(FLUENT/CFD post/...)

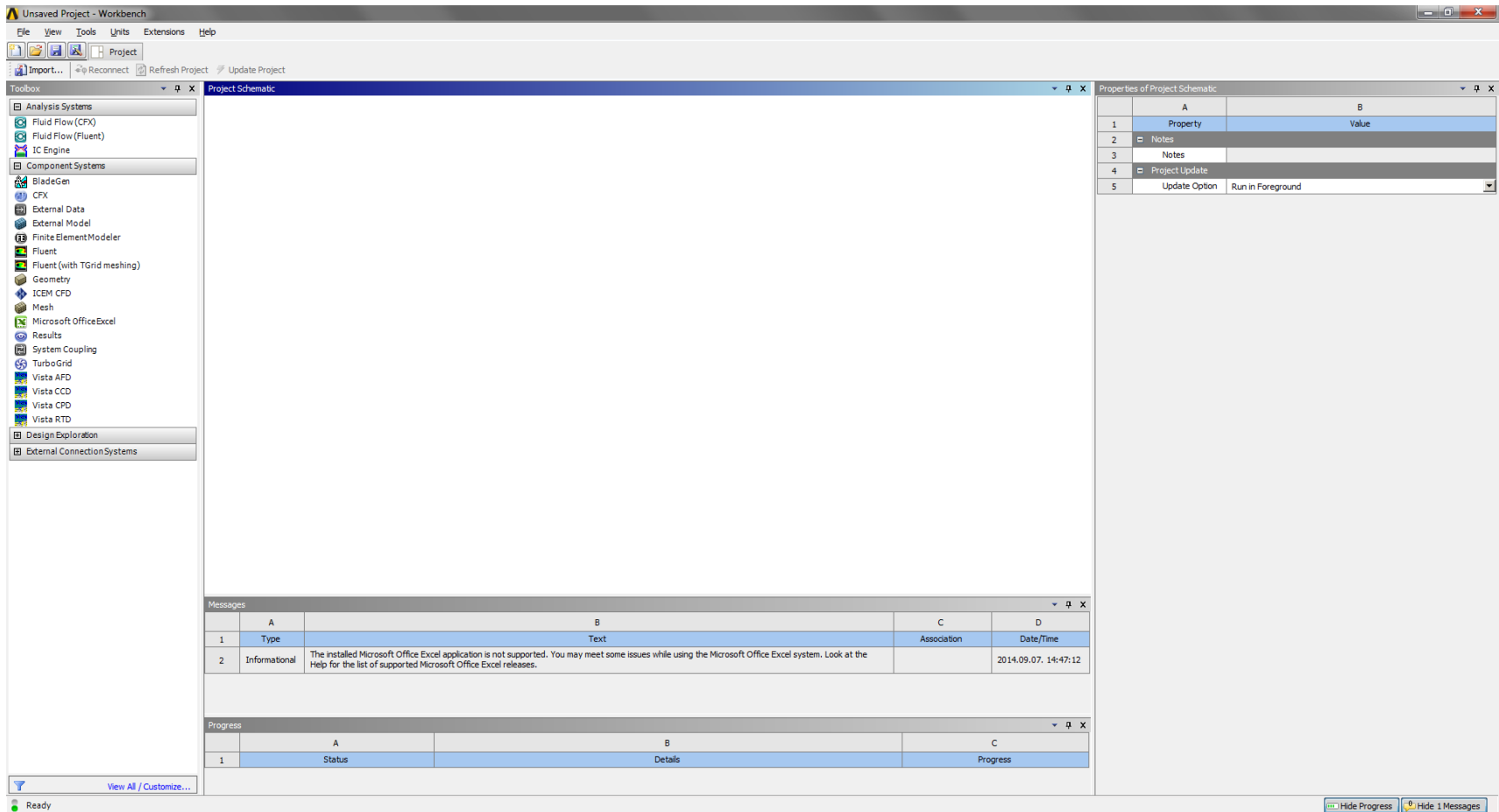
WORKBENCH

# Goal of the present practice

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



# Start the Ansys Workbench





Unsaved Project - Workbench

File View Tools Units Extensions Help

Import... Reconnect Refresh Project Update Project

Toolbox

- Analysis Systems
  - Fluid Flow (CFX)
  - Fluid Flow (Fluent)
  - IC Engine
- Component Systems
  - BladeGen
  - CFX
  - External Data
  - External Model
  - Finite ElementModeler
  - Fluent
  - Fluent (with TGrid meshing)
  - Geometry
  - ICEM CFD
  - Mesh
  - Microsoft Office Excel
  - Results
  - System Coupling
  - TurboGrid
  - Vista AFD
  - Vista CCD
  - Vista CPD
  - Vista RTD
- Design Exploration
- External Connection Systems

Project Schematic

1 A  
2 ICEM CFD Model ?  
ICEM CFD

Properties of Project Schematic

	A	B
1	Property	Value
2	Notes	
3	Notes	
4	Project Update	
5	Update Option	Run in Foreground

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

Messages

	A	B	C	D
1	Type	Text	Association	Date/Time
2	Informational	The installed Microsoft Office Excel application is not supported. You may meet some issues while using the Microsoft Office Excel system. Look at the Help for the list of supported Microsoft Office Excel releases.		2014.09.07, 14:47:12

Progress

	A	B	C
1	Status	Details	Progress

Ready

Hide Progress Hide 1 Messages

Unsaved Project - Workbench

File View Tools Units Extensions Help

Import... Reconnect Refresh Project Update Project

Toolbox

- Analysis Systems
  - Fluid Flow (CFX)
  - Fluid Flow (Fluent)
  - IC Engine
- Component Systems
  - BladeGen
  - CFX
  - External Data
  - External Model
  - Finite ElementModeler
  - Fluent
  - Fluent (with TGrid meshing)
  - Geometry
  - ICEM CFD
  - Mesh
  - Microsoft OfficeExcel
  - Results
  - System Coupling
  - TurboGrid
  - Vista AFD
  - Vista CCD
  - Vista CPD
  - Vista RTD
- Design Exploration
- External Connection Systems

Project Schematic

A

- ICEM CFD
- Model

B

- Fluent
- Setup
- Solution

Fluent

Properties of Project Schematic

	A	B
1	Property	Value
2	Notes	
3	Notes	
4	Project Update	
5	Update Option	Run in Foreground

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

Messages

	A	B	C	D
1	Type	Text	Association	Date/Time
2	Informational	The installed Microsoft Office Excel application is not supported. You may meet some issues while using the Microsoft Office Excel system. Look at the Help for the list of supported Microsoft Office Excel releases.		2014.09.07, 14:47:12

Progress

	A	B	C
1	Status	Details	Progress

Ready

Hide Progress Hide 1 Messages

Unsaved Project - Workbench

File View Tools Units Extensions Help

Import... Reconnect Refresh Project Update Project

Toolbox

- Analysis Systems
  - Fluid Flow (CFX)
  - Fluid Flow (Fluent)
  - IC Engine
- Component Systems
  - BladeGen
  - CFX
  - External Data
  - External Model
  - Finite ElementModeler
  - Fluent
  - Fluent (with TGrid meshing)
  - Geometry
  - ICEM CFD
  - Mesh
  - Microsoft OfficeExcel
  - Results
  - System Coupling
  - TurboGrid
  - Vista AFD
  - Vista CCD
  - Vista CPD
  - Vista RTD
- Design Exploration
- External ConnectionSystems

Project Schematic

ICEM CFD

Fluent

Properties of Schematic A2: Model

	A	B
1	Property	Value
2	General	
3	Component ID	ICM
4	Directory Name	ICM
5	Notes	
6	Notes	
7	Used Licenses	
8	Last Update Used Licenses	
9	Upstream	
10	Subsets From Named Selections	<input type="checkbox"/>

1) Connect them

2) Double click on ICEM CFD/Model

Messages

	A	B	C	D
1	Type	Text	Association	Date/Time
2	Informational	The installed Microsoft Office Excel application is not supported. You may meet some issues while using the Microsoft Office Excel system. Look at the Help for the list of supported Microsoft Office Excel releases.		2014.09.07, 14:47:12

Progress

	A	B	C
1	Status	Details	Progress

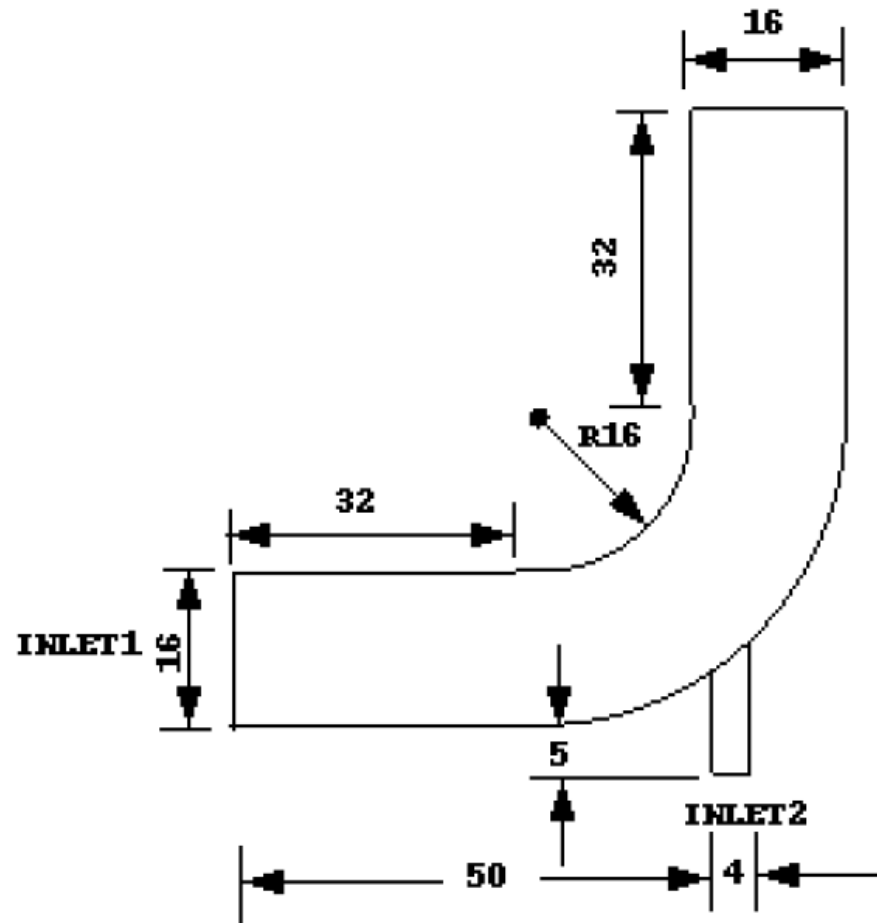
View All / Customize...

Double-click component to edit

Hide Progress Hide 1 Messages

# Geometry

# The original geometry (in cm-s)



# Settings

- Settings/Selction/Auto pick mode: turn ON

# Point creation

The screenshot displays the ANSYS ICM CFD 15.0 software interface. The top menu bar includes File, Edit, View, Info, and Settings. The main toolbar contains various icons for geometry, meshing, and simulation. A red box highlights the 'Geometry' tab in the toolbar, with a red arrow pointing to it from a green callout box. The green callout box contains the text '1.) Select: Geometry, Create Point'. Below the toolbar, the 'Create Point' dialog box is open. A red box highlights the 'Explicit Locations' section in the dialog, with a red arrow pointing to it from a green callout box. The green callout box contains the text '2.) Select: Explicit Location'. Inside the 'Explicit Locations' section, there are input fields for X, Y, and Z coordinates. A red box highlights these input fields, with a red arrow pointing to it from a green callout box. The green callout box contains the text '3.) Type the coordinates here'. Below the input fields, there is a green callout box with the text 'The coordinates of the first point should be: 0,0,0 → Apply'. At the bottom of the dialog, there are buttons for 'Apply', 'OK', and 'Dismiss'. A green callout box at the bottom right of the dialog contains the text 'You can work in any dimension, because you can scale your mesh in the solver'. The ANSYS R15.0 logo is visible in the top right corner of the interface.

1.) Select: Geometry, Create Point

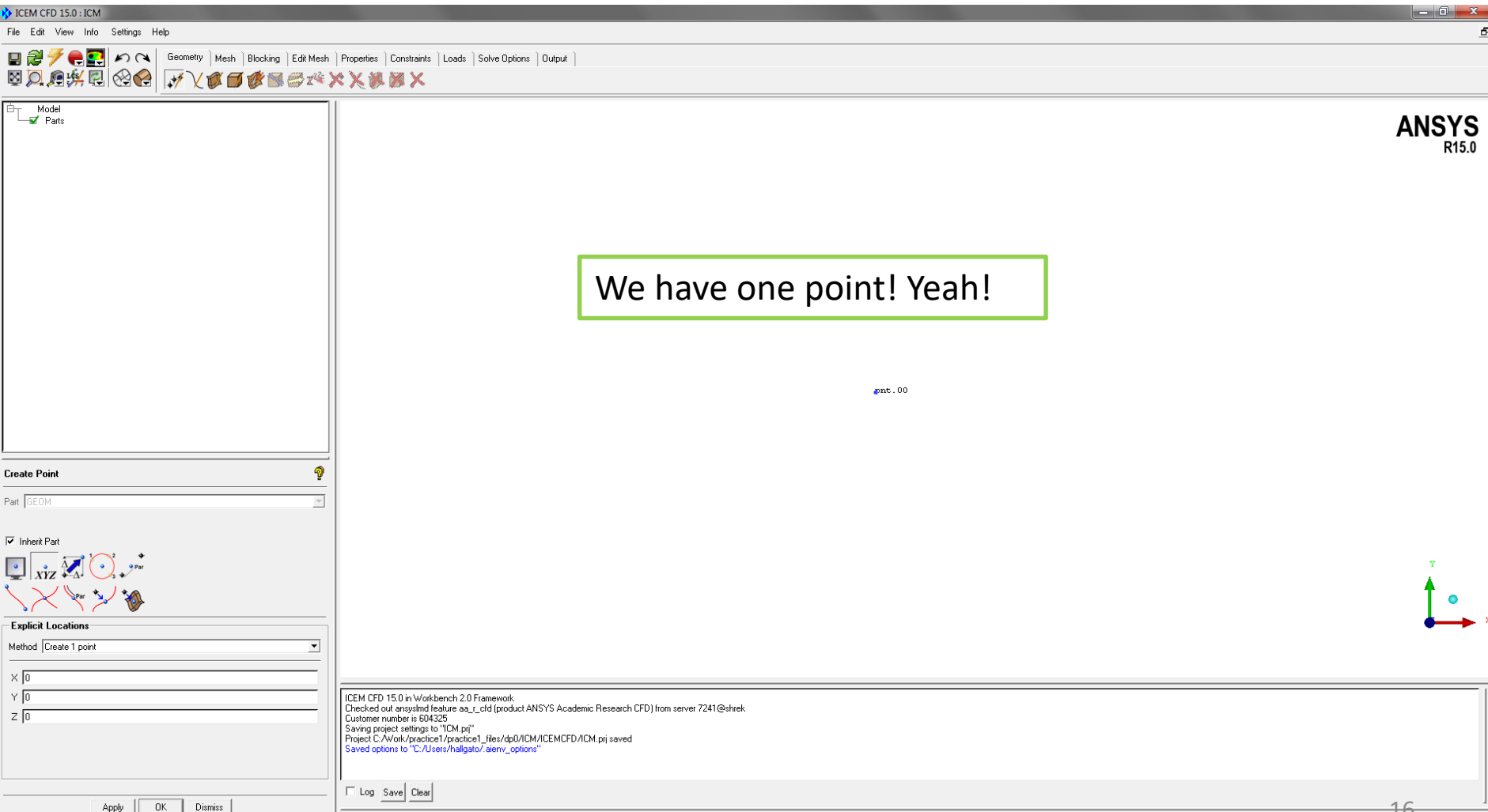
2.) Select: Explicit Location

3.) Type the coordinates here

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

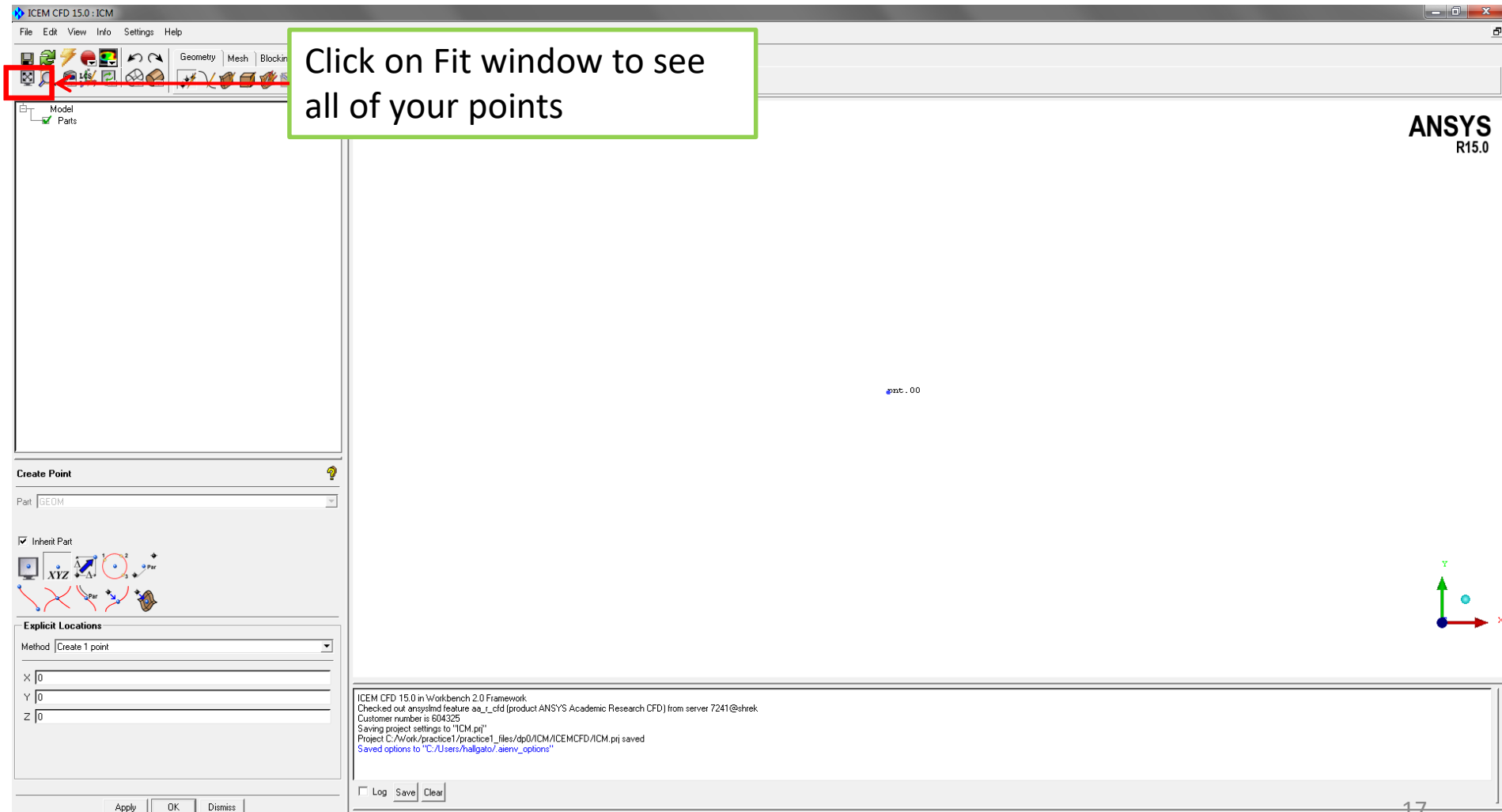
You can work in any dimension, because you can scale your mesh in the solver

# Point creation

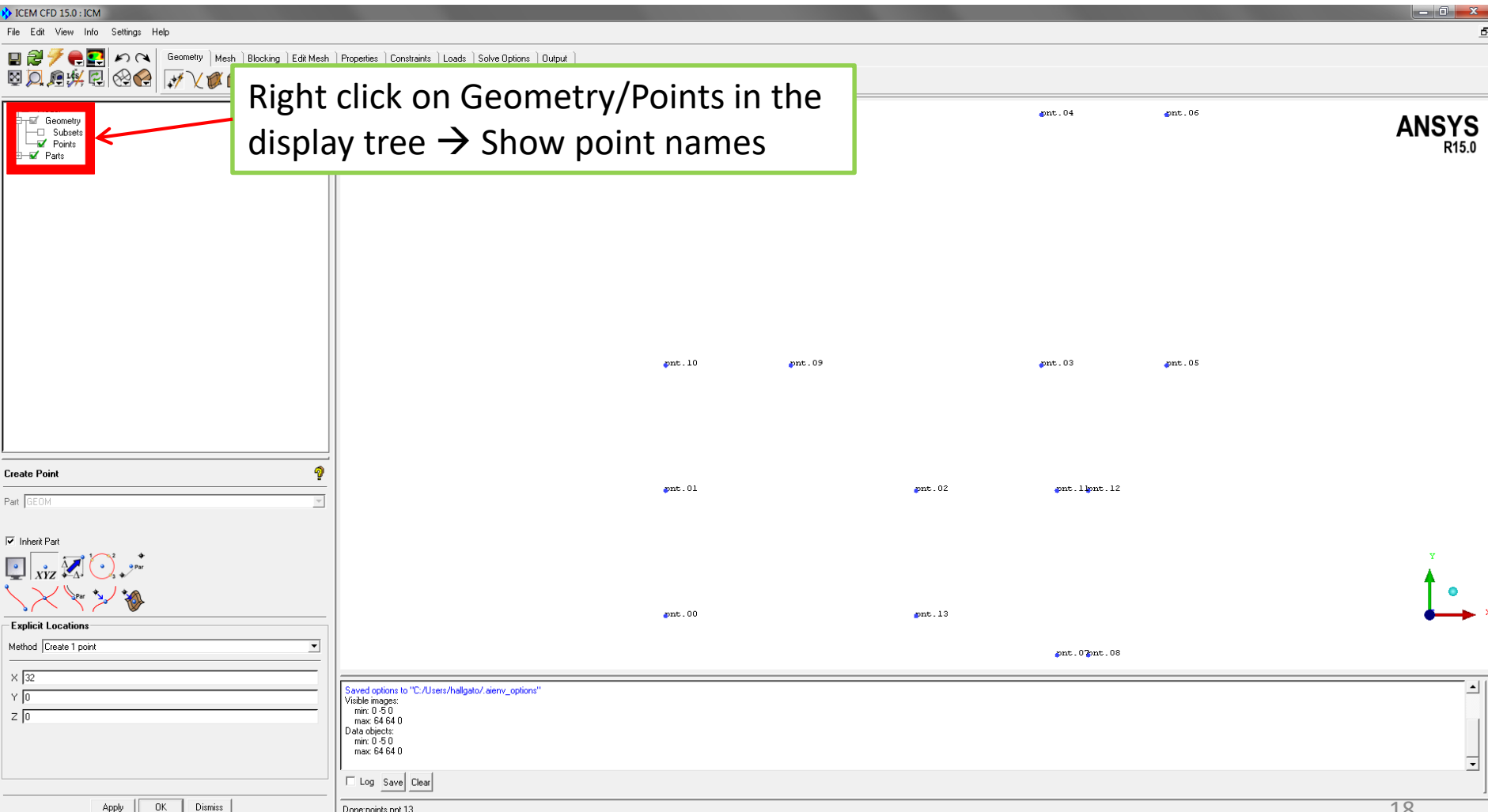




# Point creation



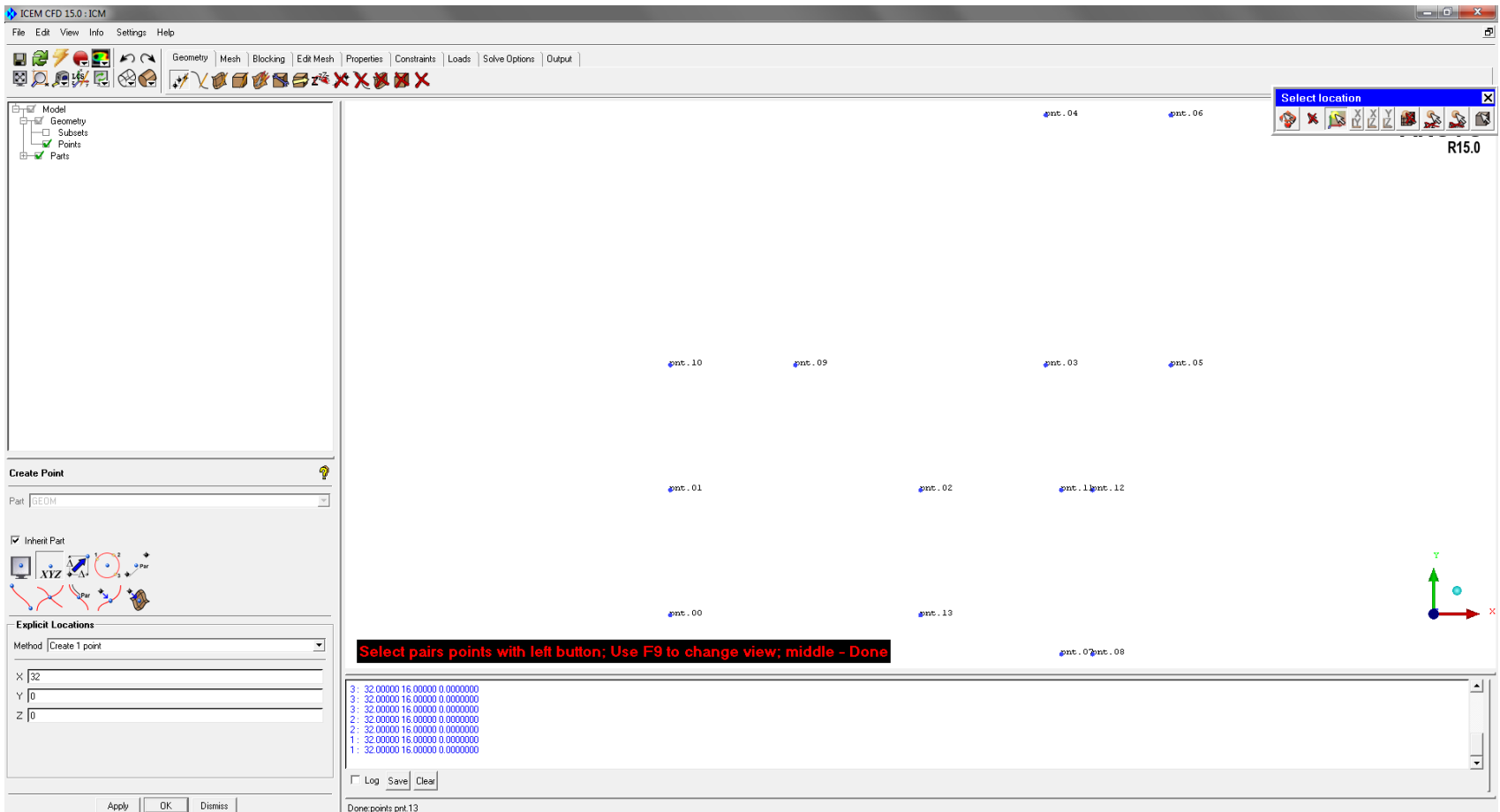
# Point creation



# Point creation – the coordinates of the points

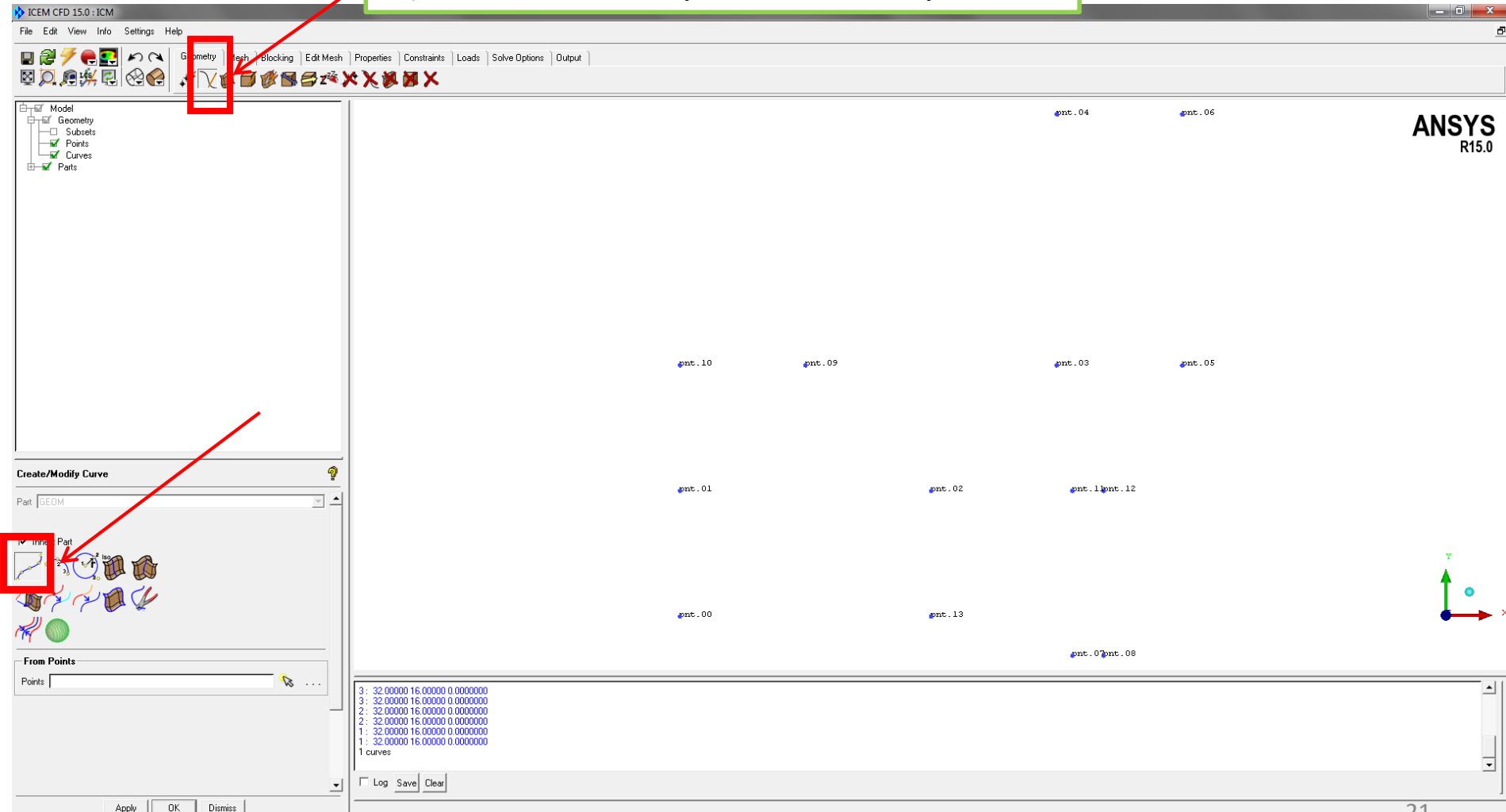
Nr.	Coordinates (x,y,z)
0	0,0,0
1	32,0,0
2	32,16,0
3	48,32,0
4	48,64,0
5	64,32,0
6	64,64,0
7	50,-5,0
8	54,-5,0
9	16,32,0
10	0,32,0
11	50,16,0
12	54,16,0
13	0,16,0

# Point creation



# Curve creation (straight line)

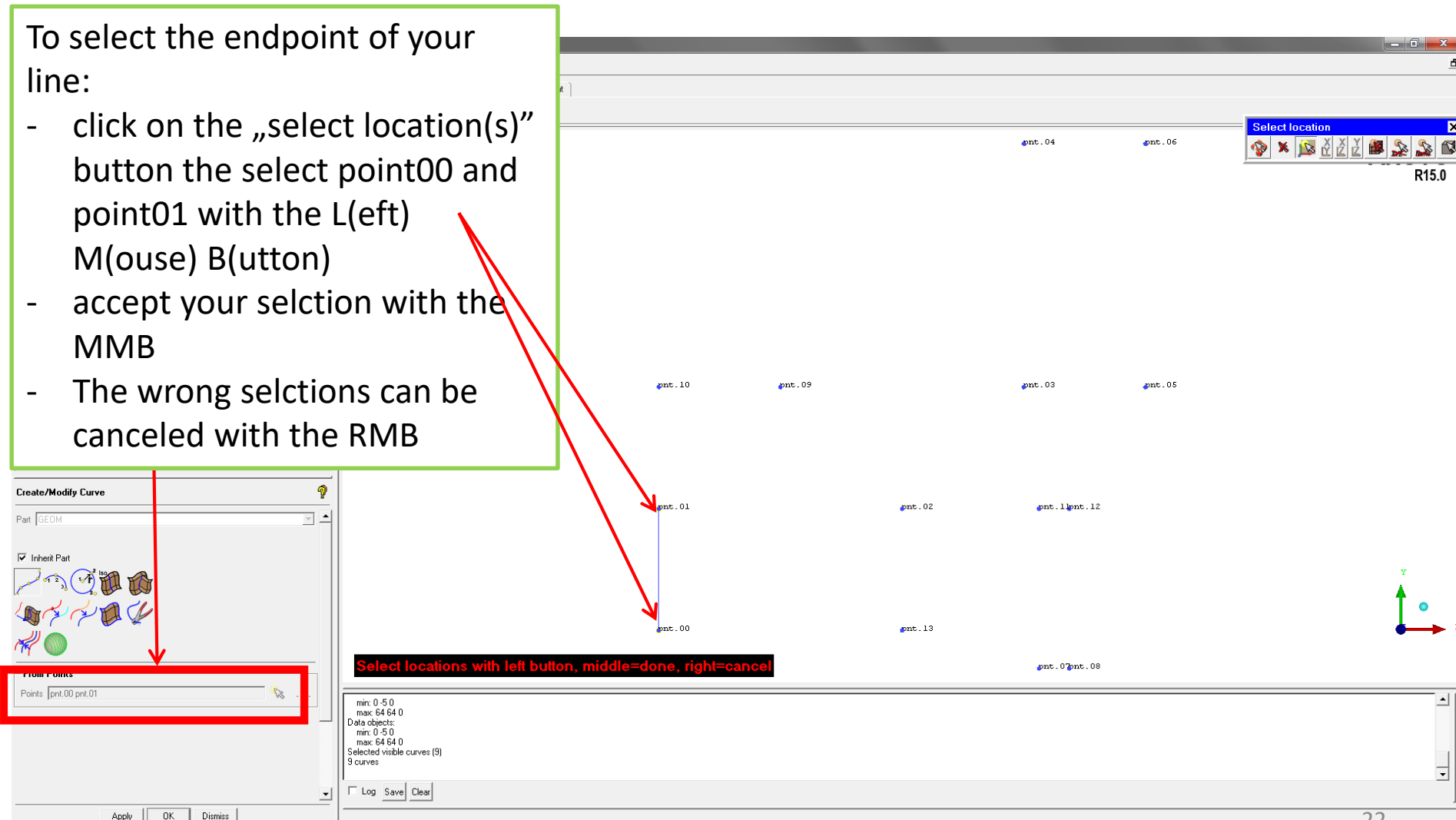
1.) Select: Geometry, Create/Modify Curve



# Curve creation (straight line)

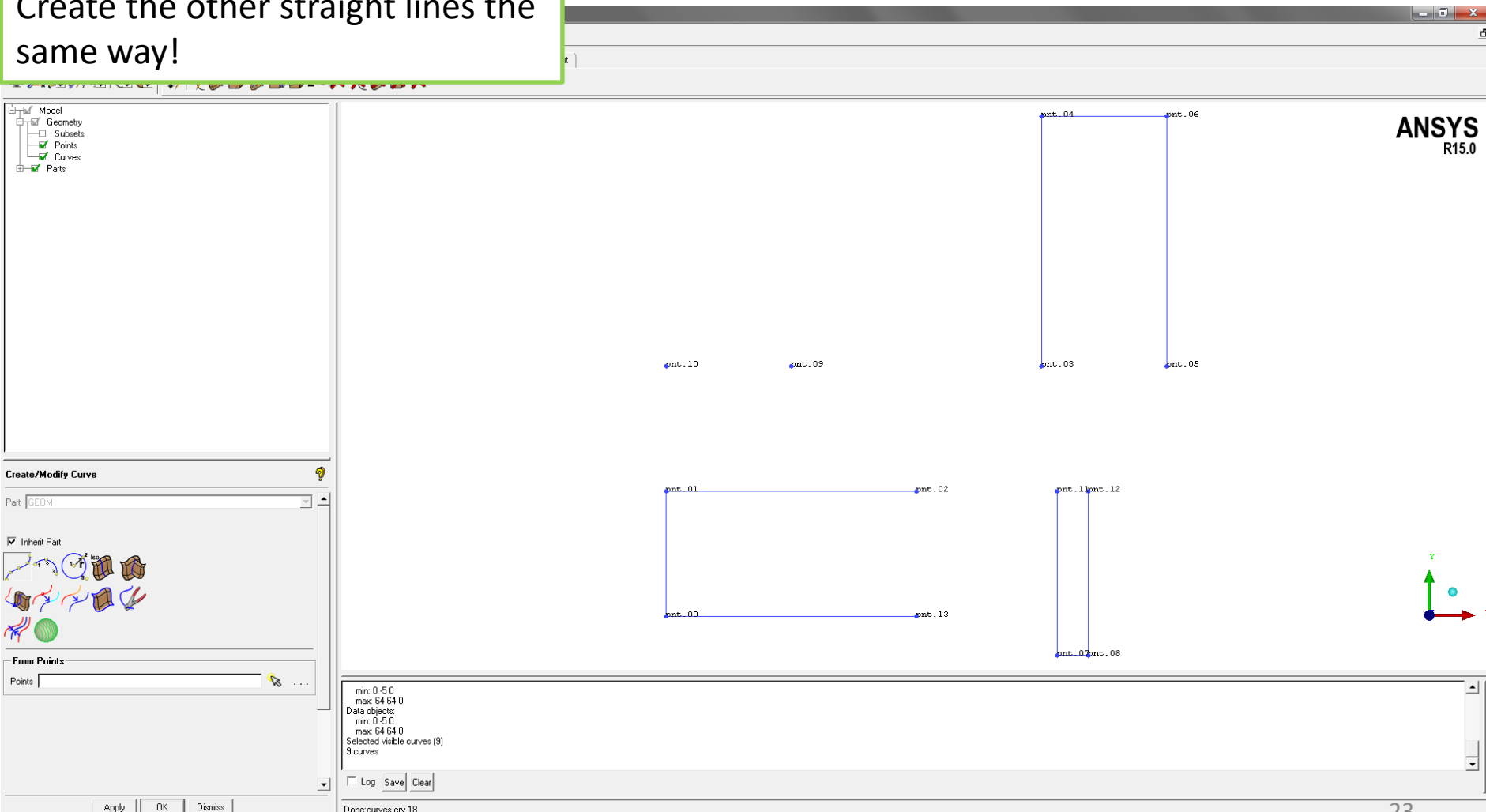
To select the endpoint of your line:

- click on the „select location(s)” button the select point00 and point01 with the L(ef) M(ouse) B(utton)
- accept your selection with the MMB
- The wrong selections can be canceled with the RMB



# Curve creation (straight lines)

Create the other straight lines the same way!



# Curve creation (Arcs)

Create arcs, which fit on three points

1.) Select: Arc

2.) Method: From 3 Points

Arc 2

Arc 1

Select 3 location with left button, right=cancel

min: 0.5 0  
max: 64.64 0  
Data objects:  
min: 0.5 0  
max: 64.64 0  
Selected visible curves (9)  
9 curves

Log Save Clear

Done: curves: crv.20

Apply OK Dismiss



# Create intersection points

1.) Select: Geometry/Create Point

2.) Select: Curve intersections

3.) Select the large arc and the straight line with the LMB (you can cancel with the RMB)

4.) Repeat it with the arc and the other straight line

Select 2 curves with the left button; middle = done, right = back up / cancel, Shift-left = deselect, '?' = list options.

max: 64 64 0  
Selected visible curves (9)  
9 curves  
2 curves  
2 curves  
0 curves  
0 curve

☐ Log ☐ Save ☐ Clear

Apply OK Dismiss

# Segment curves

1.) Select: Geometry/Create – Modify curves

2.) Select:  
Segment curves

Segment your arcs at  
the middle points

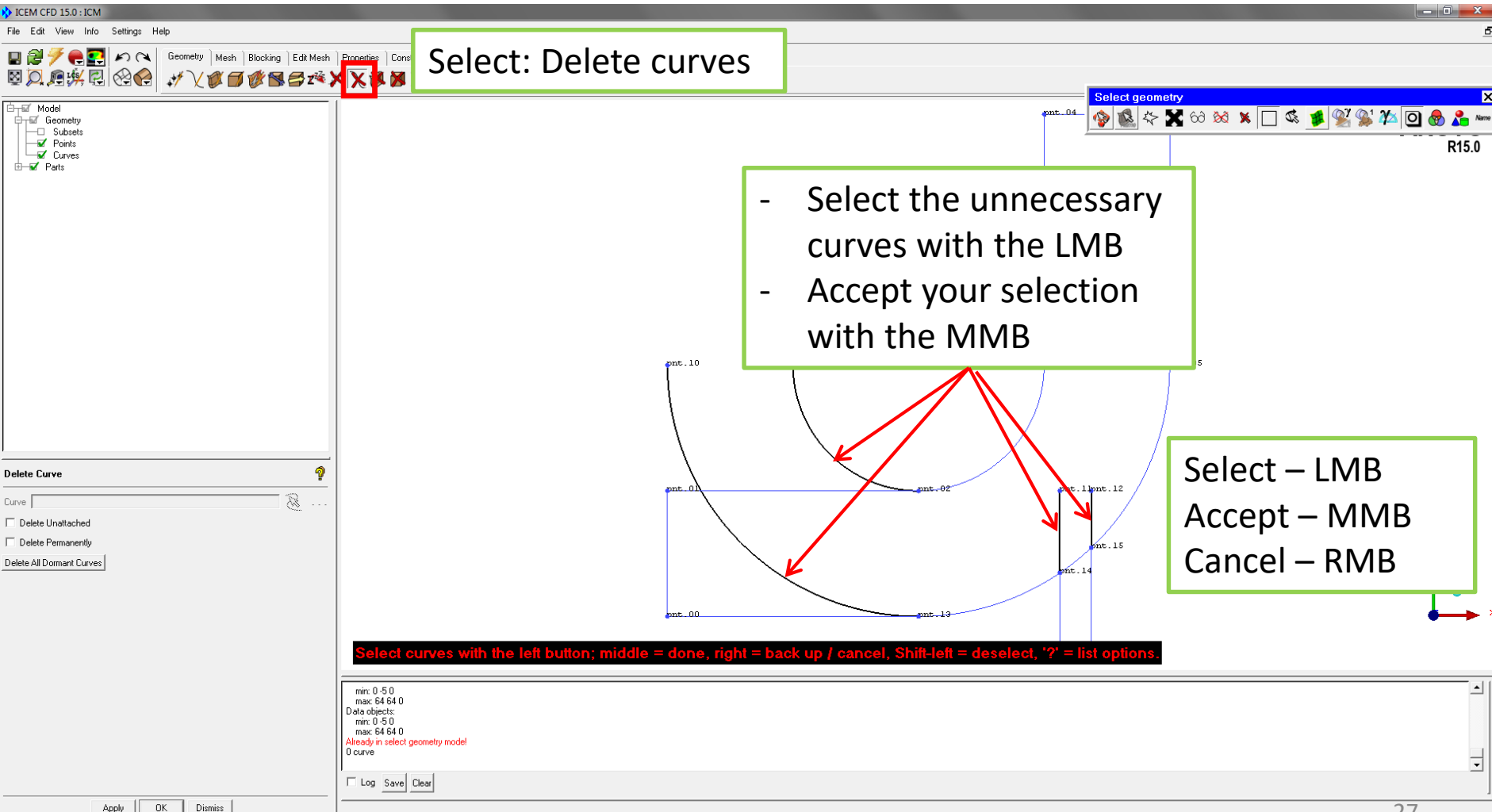
Segment your large arc at the  
intersection points which was  
created in the previous step (you  
can select the points together)

Select – LMB  
Accept – MMB  
Cancel – RMB

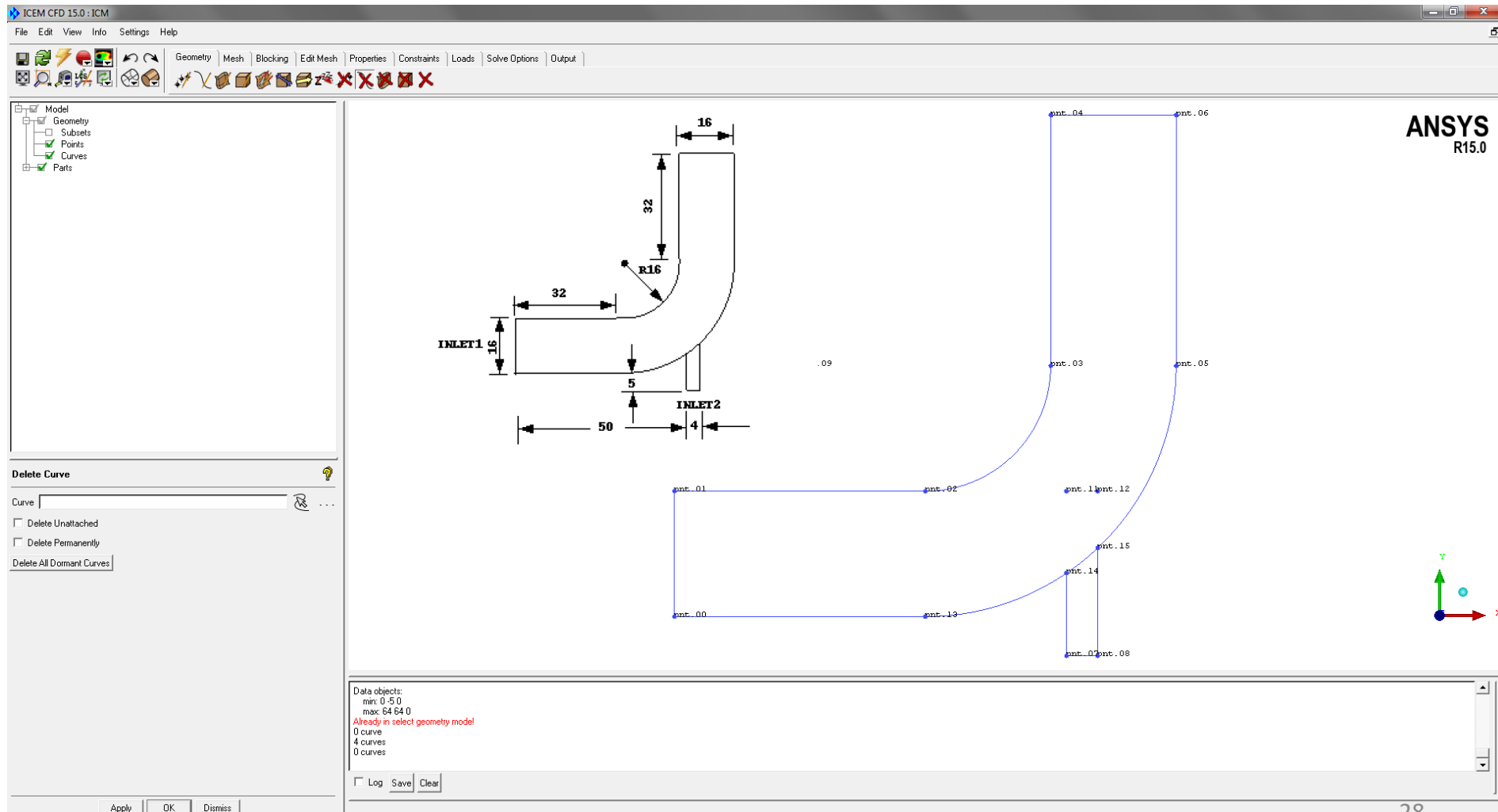
Select 1 curve with the left button; middle = done, right = back up / cancel, Shift-left = deselect, ? = list options.

Segment your straight lines at  
the intersection points as well

# Delete curves



# Delete curves – The result



# Create Parts

The screenshot shows the ICEM CFD 15.0 interface. On the left, the 'Model' tree has 'Curves' and 'Geom' highlighted. The 'Create Part' dialog is open, showing 'Part: velocity\_inlet1' and 'Create Part' button. Below it, the 'Create Part by Selection' section has 'Entities' listed. The main workspace shows a 2D geometry with points (e.g., ent.00, ent.01, ent.02, ent.03, ent.05, ent.09, ent.10, ent.12, ent.13, ent.14, ent.15) and lines. A red arrow points from the 'Entities' list to the vertical line on the left. A 'Select geometry' toolbar is visible at the top right.

1) Right click on the parts in the display tree, Create part

The parts will be your boundaries and fluid zones in the FLUENT (like the named selections in the WB Mesher)

2) The part name should be: Velocity\_inlet1 (dont use space)

3) The selected entity: the vertical straight line on the left hand side

Select – LMB  
Accept – MMB  
Cancel – RMB

Part: velocity\_inlet1  
Create Part

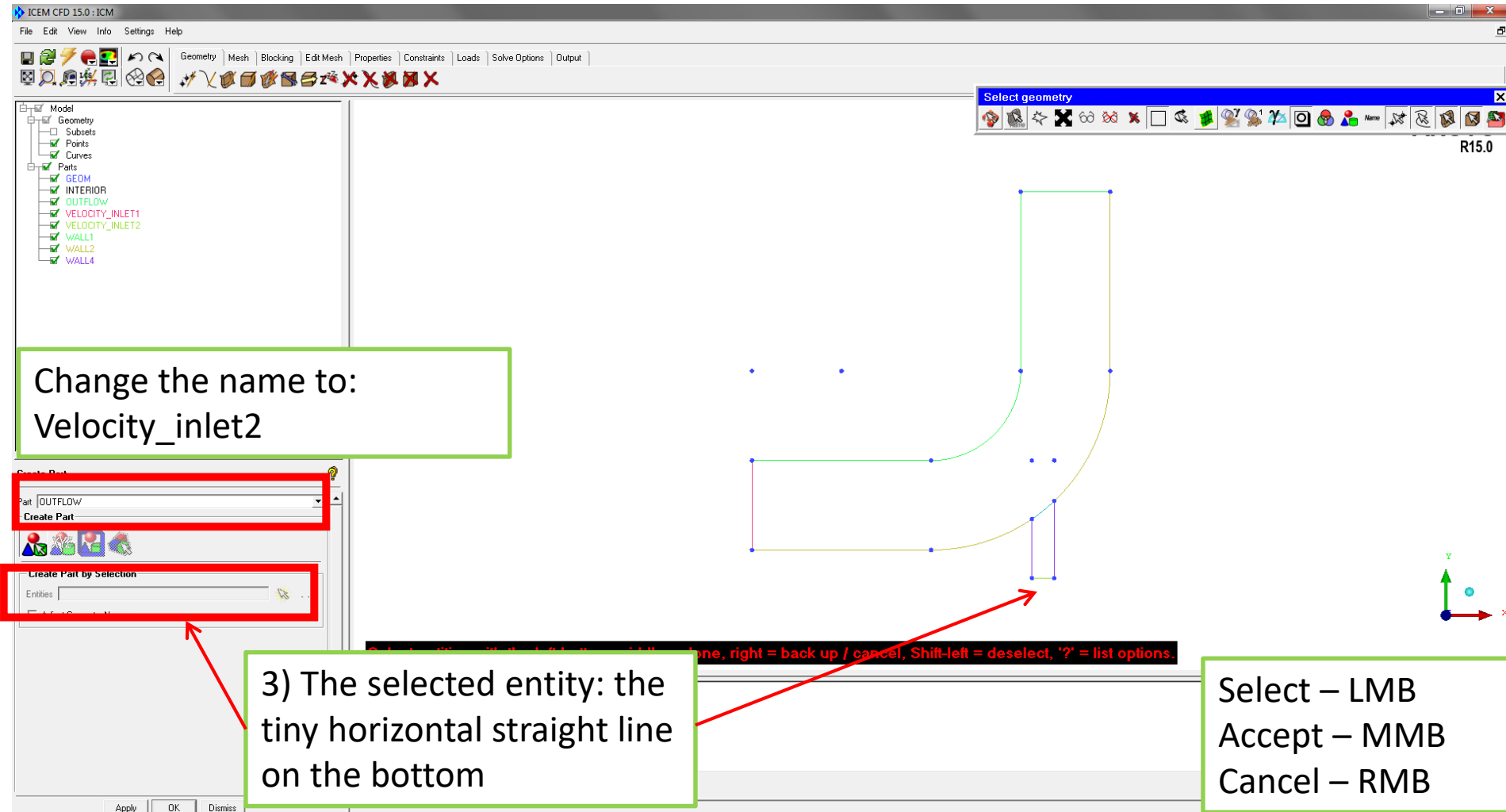
Create Part by Selection  
Entities

ent.10 ent.09 ent.03 ent.05 ent.01 ent.02 ent.12 ent.15 ent.14 ent.13 ent.00

Select geometry

Apply OK Dismiss

# Create Parts



# Create Parts

**Create the other parts!**

**Outflow**

**Wall1**

**Interior (the tiny curve)**

**Wall3**

**Wall2**

**Select – LMB  
Accept – MMB  
Cancel – RMB**

**When you finished save your work!**

Select entities with the left button; middle = done, right = back up / cancel, Shift-left = deselect, '?' = list options.

Selecting geometry.  
2 entities  
Selecting geometry.

Model  
Geom  
Subs  
Point  
Curve  
Parts  
GEOM  
INTERIOR  
OUTFLOW  
VELOCITY\_INLET1  
VELOCITY\_INLET2  
WALL1  
WALL2  
WALL4

Create Part  
Part [OUTFLOW]  
Create Part  
Create Part by Selection  
Entities  
☐ Adjust Geometry Names

File Edit View Info Settings Help  
Geometry Mesh Blocking Edit Mesh Properties Constraints Loads Solve Options Output

R15.0

# Blocking, Meshing



# Block structured mesh

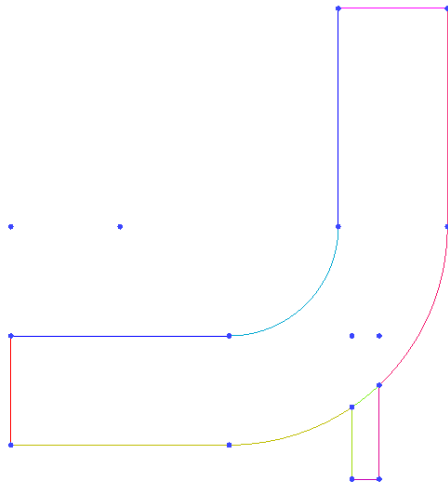
- The mesh is build up from rectangular (2D) or cuboid (3D) segments
- The blocks are divided to quad (2D) or hexa (3D) cells
- The vertices, the edges and the faces of the blocks are associated and shaped to the points, curves and surface of the geometry

# Definitions in ICEM

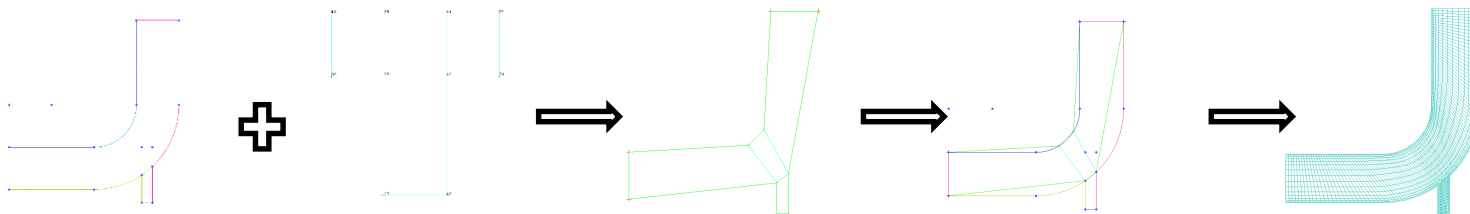
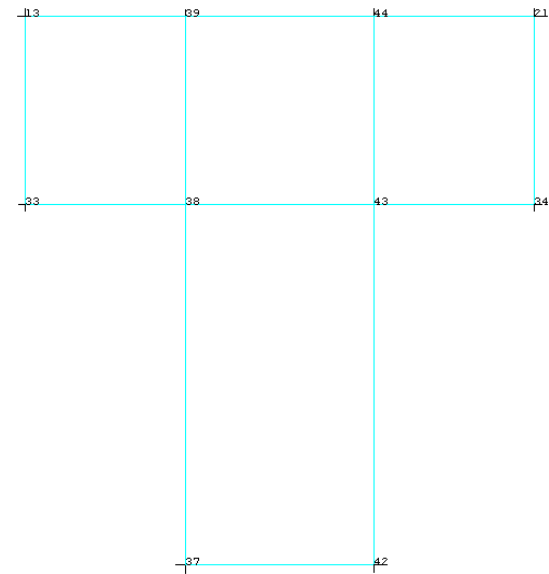
Geometry	Blocking
Point	Vertex
Curve	Edge
Surface	Face

# Sample (the present problem)

Geometry

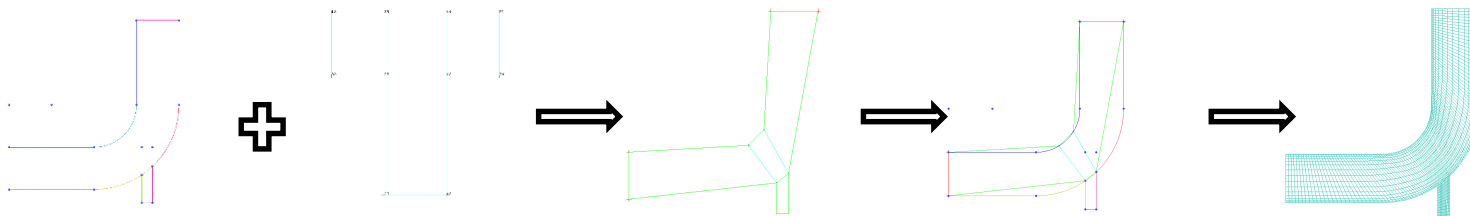
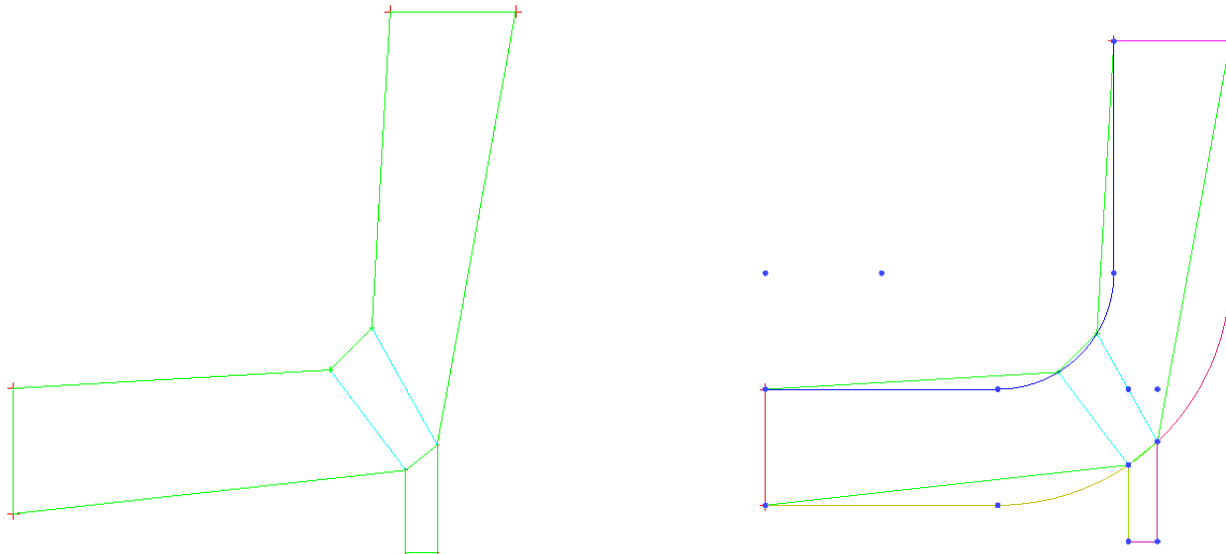


Initial blocking



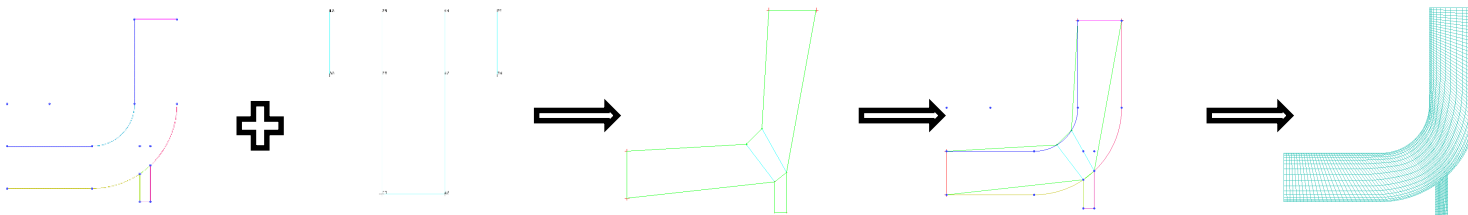
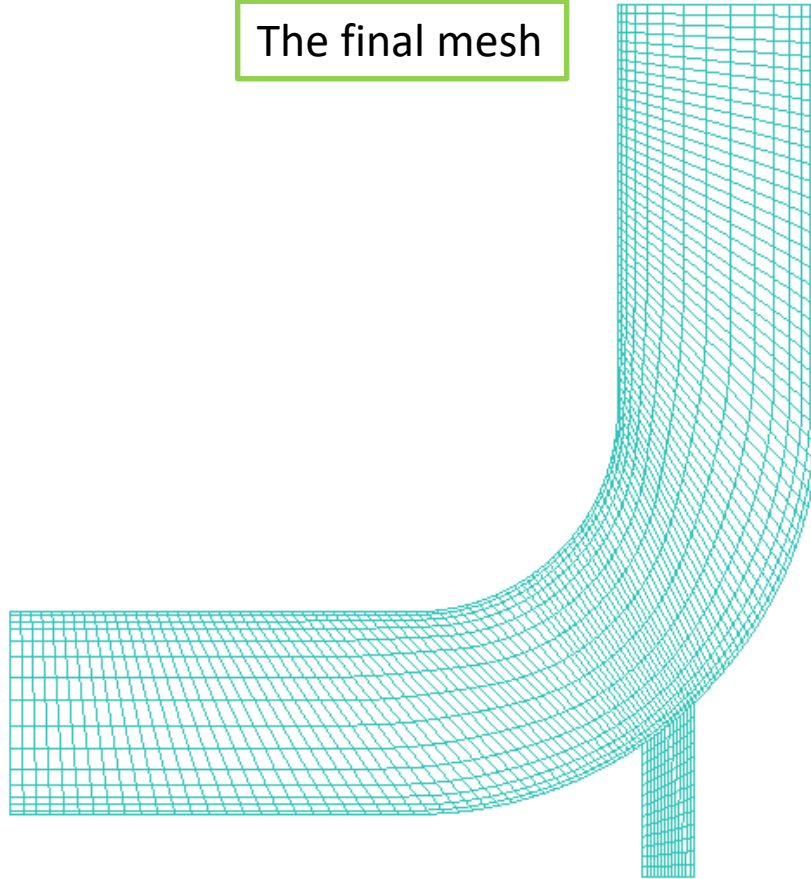
# Sample (the present problem)

The associated blocking



# Sample (the present problem)

The final mesh

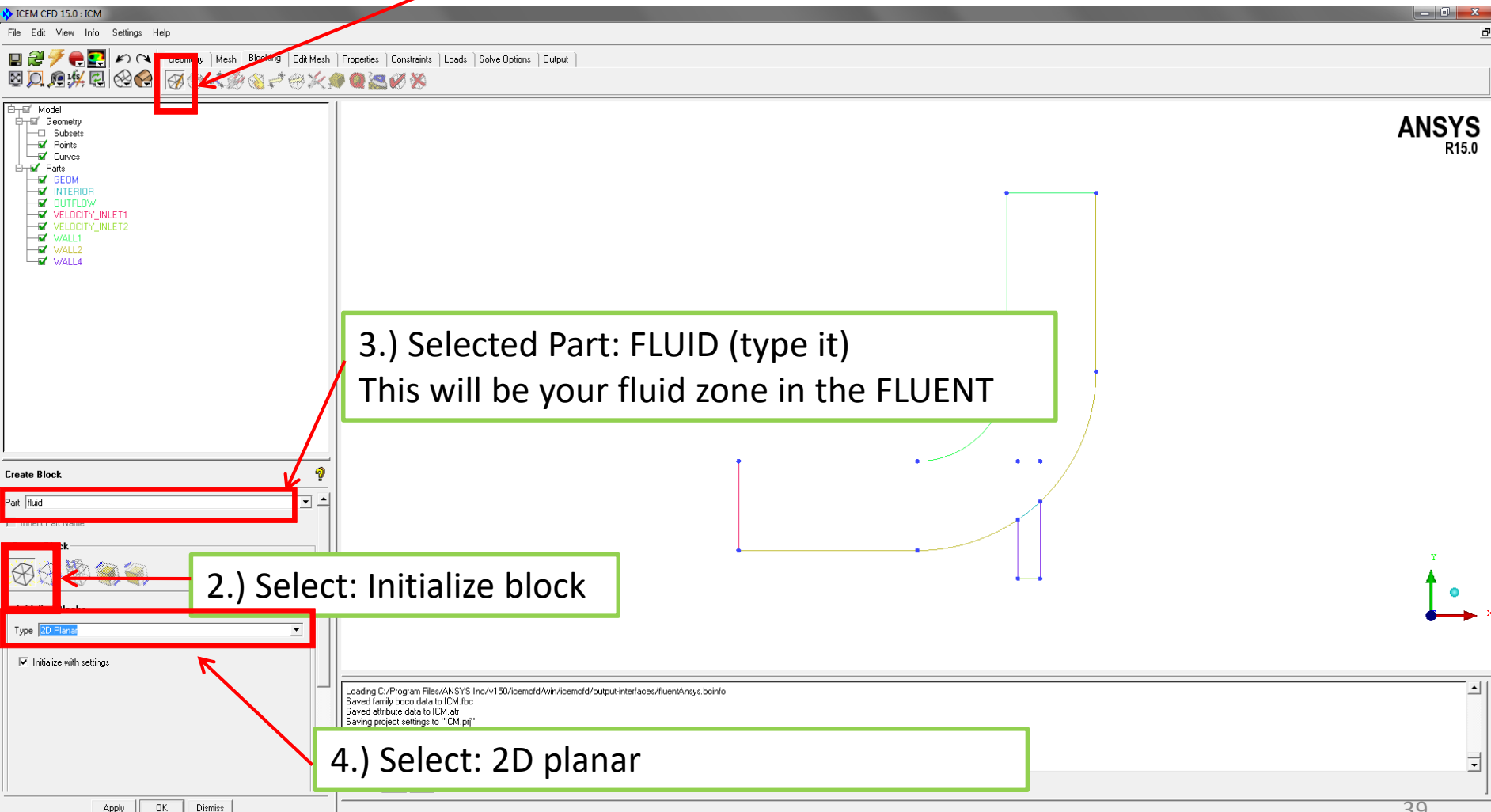


# Let's start blocking

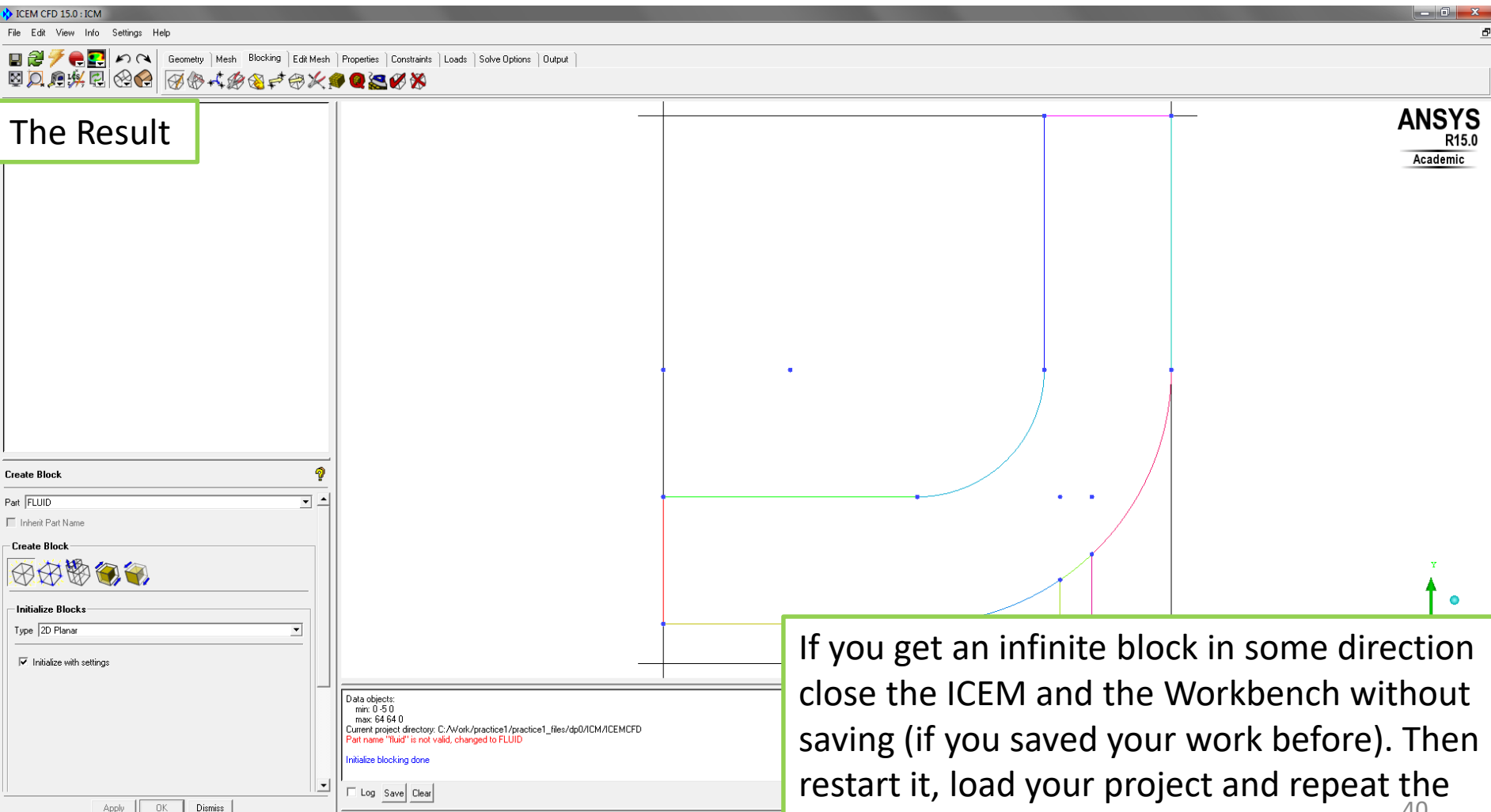
- Save your work before start the blocking!

# Initialize Blocking

1.) Select: Blocking, Create Block



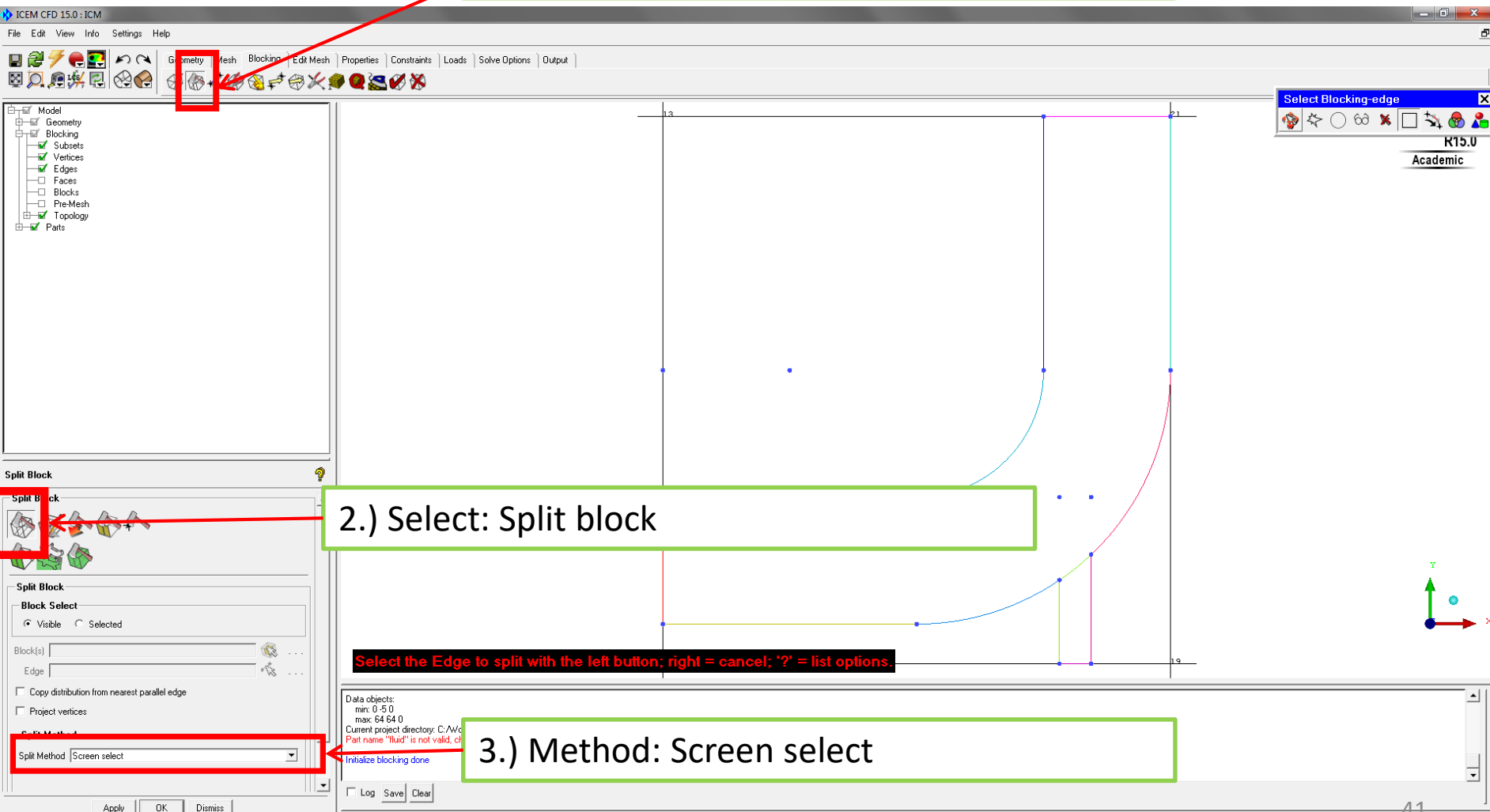
# Initialize Blocking





# Split block

## 1.) Select: Blocking, Split Block



# Split block

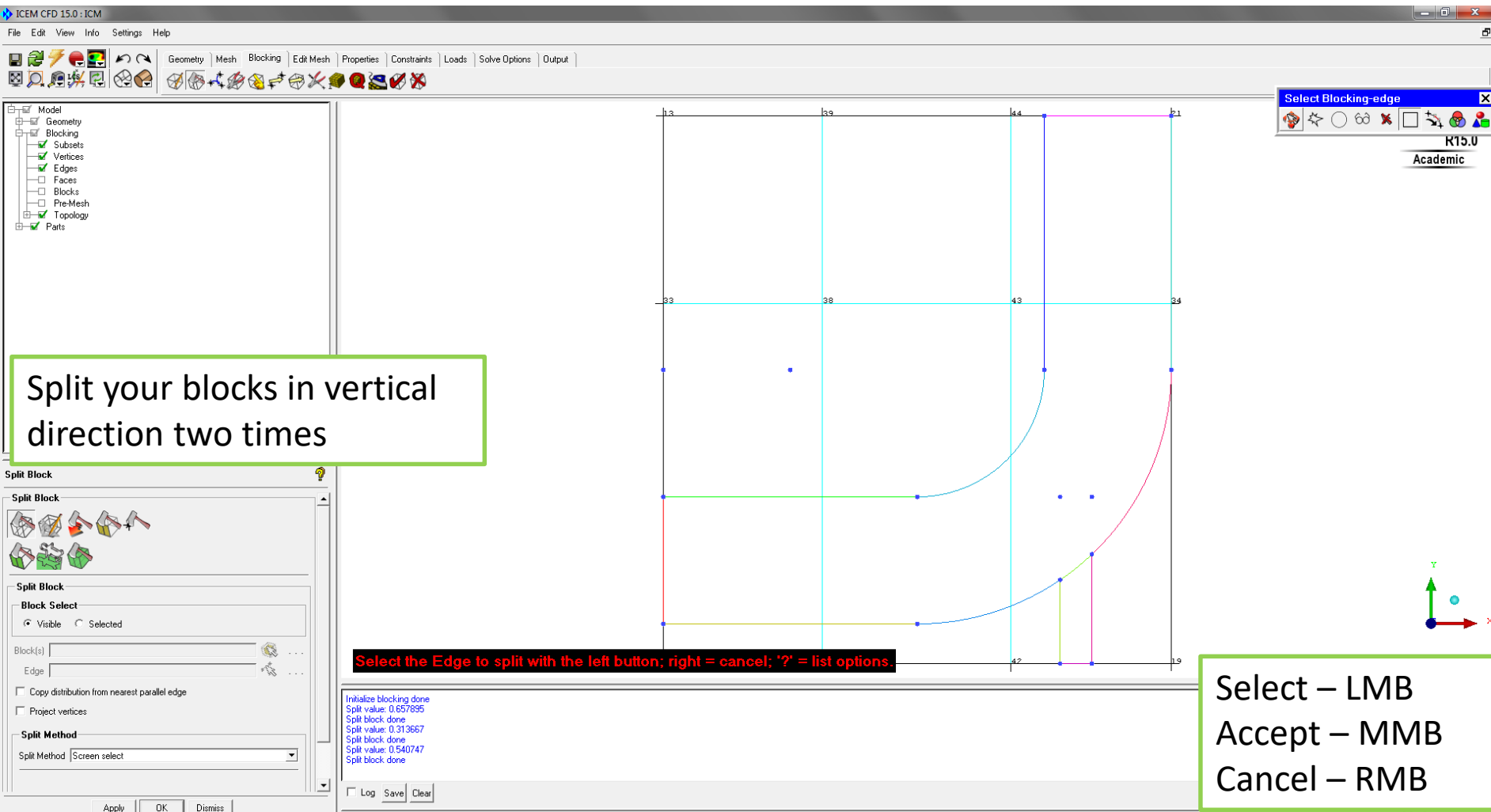
The screenshot displays the ICEM CFD 15.0 software interface. The main workspace shows a 2D block with a curved internal boundary. A red arrow points from a text box to the left vertical edge of the block, indicating the selection point. Another red arrow points from a text box to the 'Block(s)' input field in the 'Split Block' dialog. The dialog has tabs for 'Visible' and 'Selected'. Below the input field, there are checkboxes for 'Copy distribution from nearest parallel edge' and 'Project vertices', and a 'Split Method' dropdown set to 'Screen select'. A status bar at the bottom shows 'max: 64 64 0', 'Current project directory: C:\work\practice1\practice1\_files\dp0\ICM\ICEMCFD', and 'Part name "fluid" is not valid, changed to FLUID'. A console window at the bottom left shows 'Initialize blocking done', 'Split value: 0.657895', and 'Split block done'. A 'Select Blocking-edge' toolbar is visible in the top right, and a coordinate system is in the bottom right.

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

Select the Edge to split with the left button; right = cancel; '?' = list options.

Select – LMB  
Accept – MMB  
Cancel – RMB

# Split block



# Split block

The screenshot displays the ICEM CFD 15.0 software interface. The main window shows a 2D block mesh with numbered blocks (4, 10, 13, 14, 16, 17). A green box labeled "Select: Delete blocks" points to the "Delete blocks" icon in the "Constraints" toolbar. A green box labeled "Delete the two bottom blocks on the left and right hand side" has red arrows pointing to blocks 4 and 17. A red box highlights the "Delete Block" dialog box in the bottom left, which includes a "Blocks" list and a "Delete permanently" checkbox. A black box at the bottom center contains the text: "Select Hexa block with the left button; middle = done, right = back up / cancel, '?' = list options." A green box in the bottom right corner contains the text: "Select – LMB", "Accept – MMB", and "Cancel – RMB". The status bar at the bottom shows "Split value: 0.657895", "Split block done", "Split value: 0.313667", "Split block done", "Split value: 0.540747", "Split block done", and "Already in select block model".

ICEM CFD 15.0 - ICM

File Edit View Info Settings Help

Geometry Mesh Blocking Edit Mesh Properties Constraints Loads

Select: Delete blocks

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

Delete Block

Blocks

☐ Delete permanently

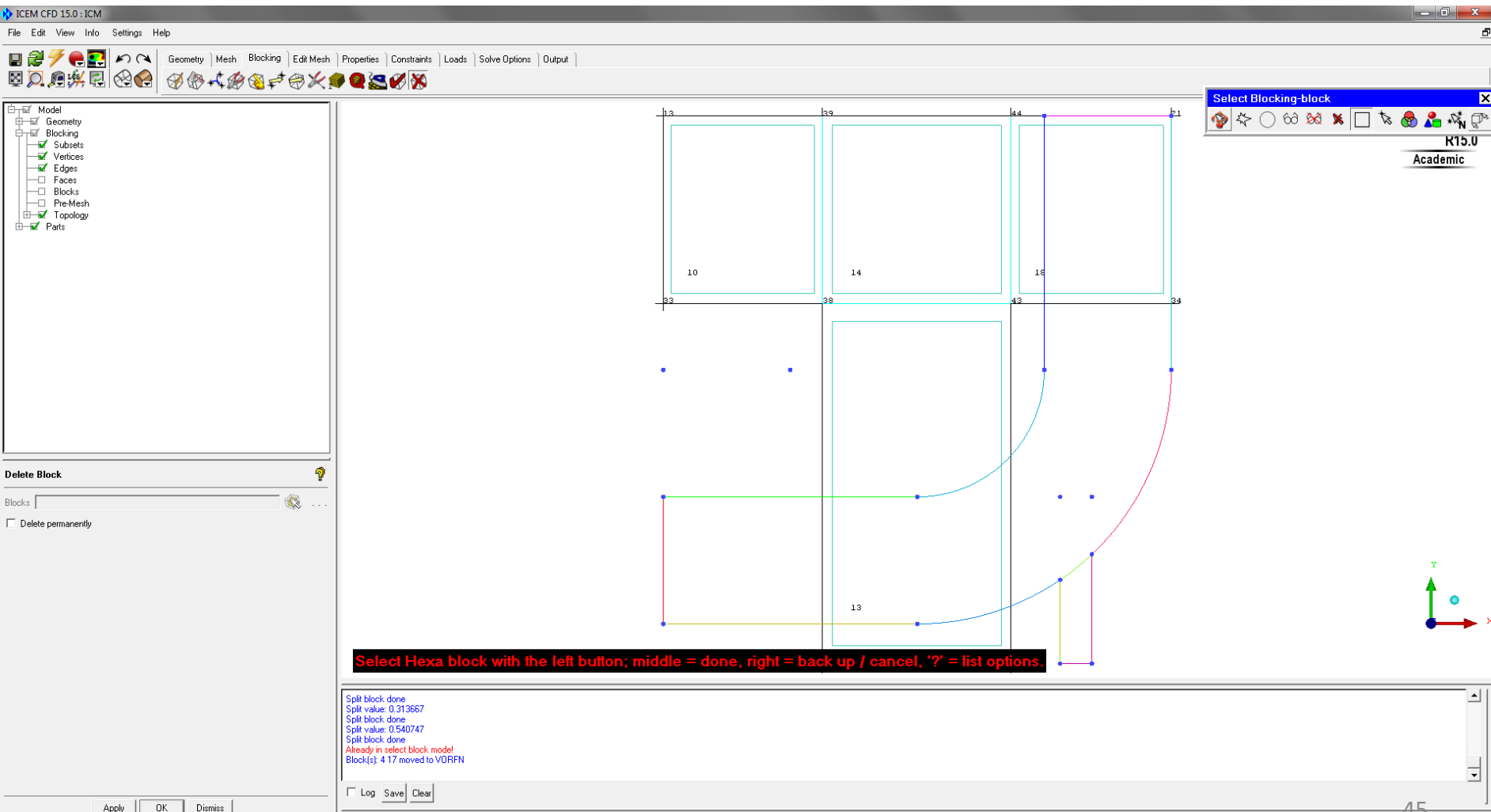
Select Hexa block with the left button; middle = done, right = back up / cancel, '?' = list options.

Split value: 0.657895  
Split block done  
Split value: 0.313667  
Split block done  
Split value: 0.540747  
Split block done  
Already in select block model

Log Save Clear

Select – LMB  
Accept – MMB  
Cancel – RMB

# Split block



# Associate the edges to the curves

ICEM CFD 15.0.1 ICM

File Edit View Info Settings Help

Geometry Mesh Blocking Edit Mesh Properties Constraints Load

Select: Associate

Select the Associate edge to curve than associate the marked edge to the marked curve

Blocking Associations

Edit Associations

Associate Edge -> Curve

Edge(s)

Curve(s)

☐ Project vertices

☐ Project to surface intersection

☐ Project ends to curve intersection

Select Hexa edge with the left button; middle = done, right = back up / cancel, '?' = list options.

Select - LMB  
Accept - MMB  
Cancel - RMB

Split block done  
Split value: 0.313657  
Split block done  
Split value: 0.540747  
Split block done  
Already in select block model  
Block(s): 4 17 moved to VORFN

Log Save Clear

Apply OK Dismiss

Total Commande... practice1 - Workb... ICEM CFD 15.0.1 L...

# Associate the edges to the curves

The screenshot shows the ICEM CFD 15.0 interface. The main window displays a 2D grid with various edges highlighted in different colors (green, blue, red, yellow). A green box with a red arrow points to a green edge, indicating a successful association. The 'Blocking Associations' dialog is open, showing the 'Associate Edge -> Curve' section. The 'Edge(s)' field contains '1,3' and the 'Curve(s)' field contains '3,3'. The 'Project vertices' checkbox is checked. The status bar at the bottom shows the following text: 'Already in select block model', 'Block(s): 4.117 moved to VORFN', 'Done "Edge -> Curve" association', 'Grouped curves: crv.15 crv.20', 'Done "Edge -> Curve" association', 'Grouped curves: crv.25 crv.14', 'Done "Edge -> Curve" association'. The 'Log Save Clear' buttons are visible.

If the association is successful, the edge will be green

Select Hexa edge with the left button; middle = done, right = back up / cancel, '?' = list options.

Already in select block model  
Block(s): 4.117 moved to VORFN  
Done "Edge -> Curve" association  
Grouped curves: crv.15 crv.20  
Done "Edge -> Curve" association  
Grouped curves: crv.25 crv.14  
Done "Edge -> Curve" association

Select – LMB  
Accept – MMB  
Cancel – RMB

# 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

Select Hexa edge with the left button; middle = done, right = back up / cancel, '?' = list options.

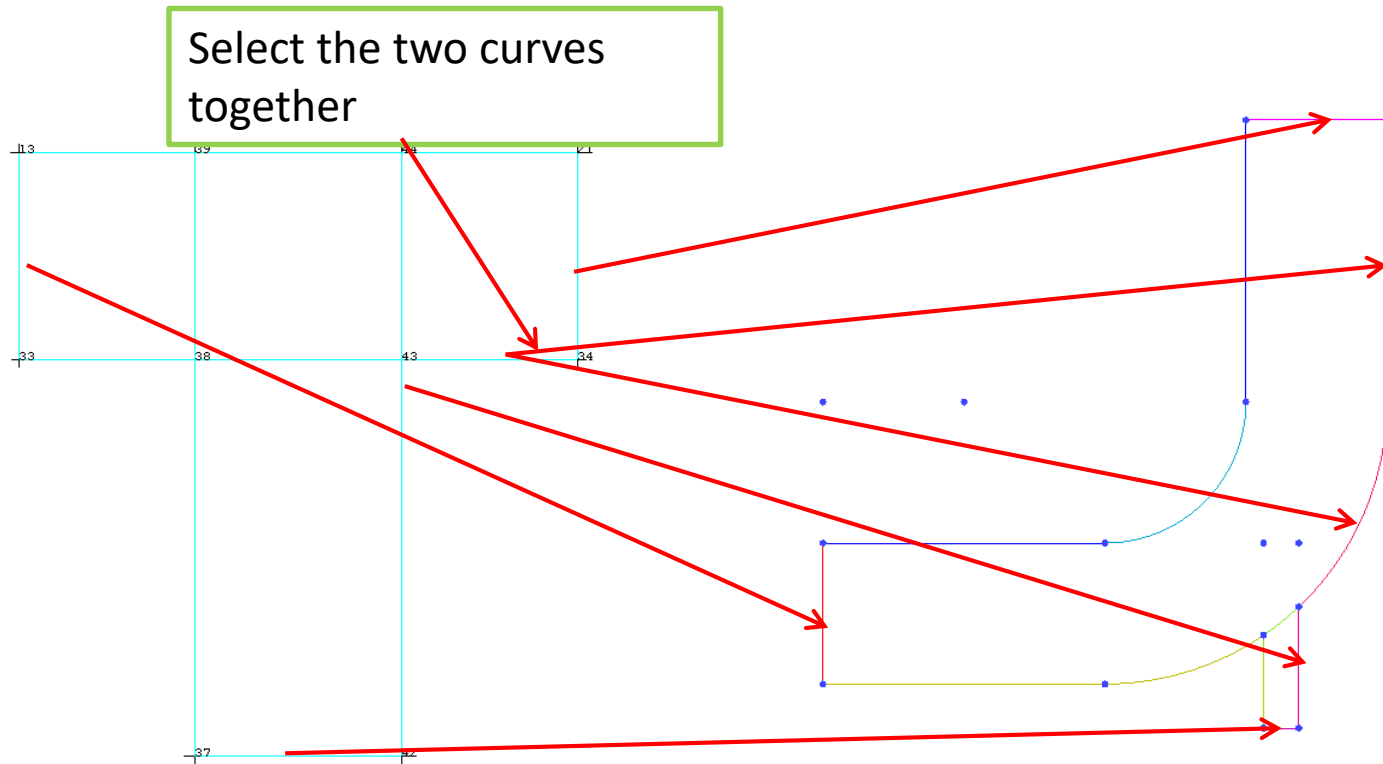
Done "Edge -> Curve" association  
Done "Edge -> Curve" association  
Grouped curves: cny.12 cny.21  
Done "Edge -> Curve" association  
Done "Edge -> Curve" association  
Done "Edge -> Curve" association  
Done "Edge -> Curve" association  
Done "Edge -> Curve" association

Apply OK Dismiss

Log Save Clear



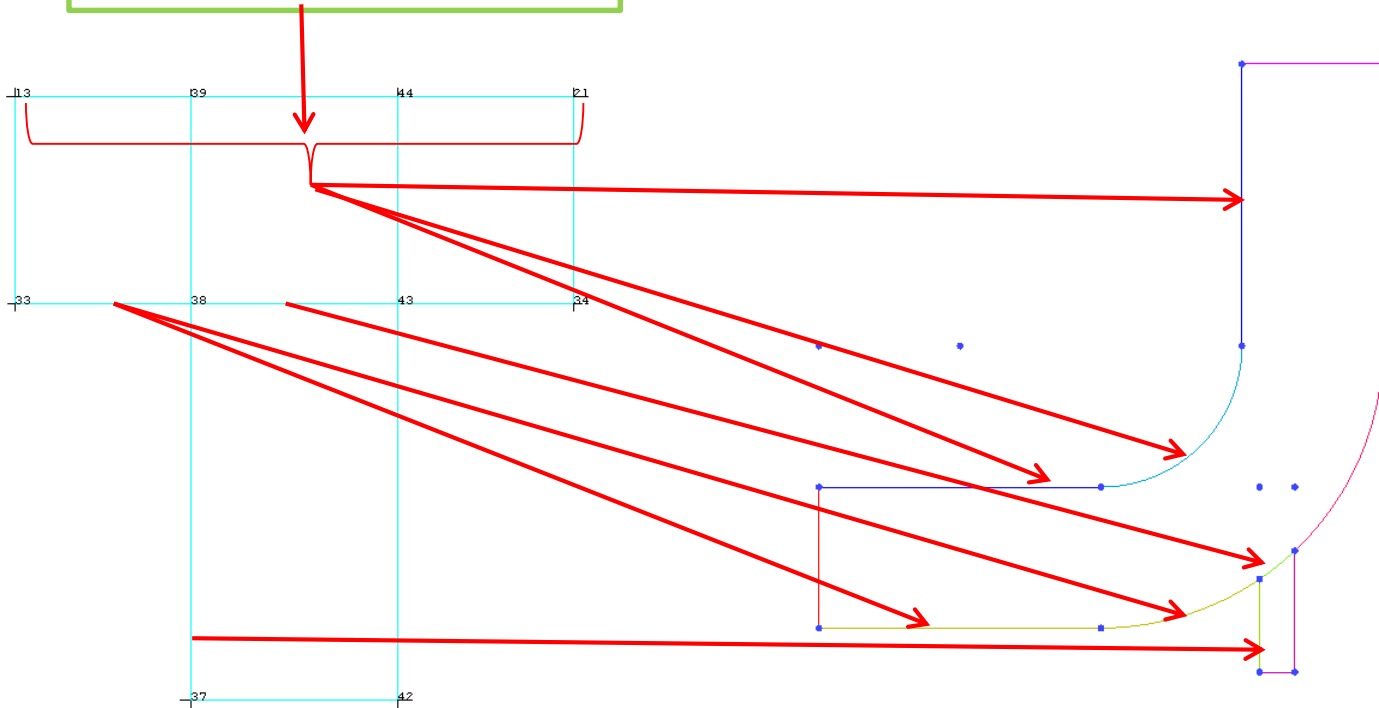
# Associate the edges to the curves



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

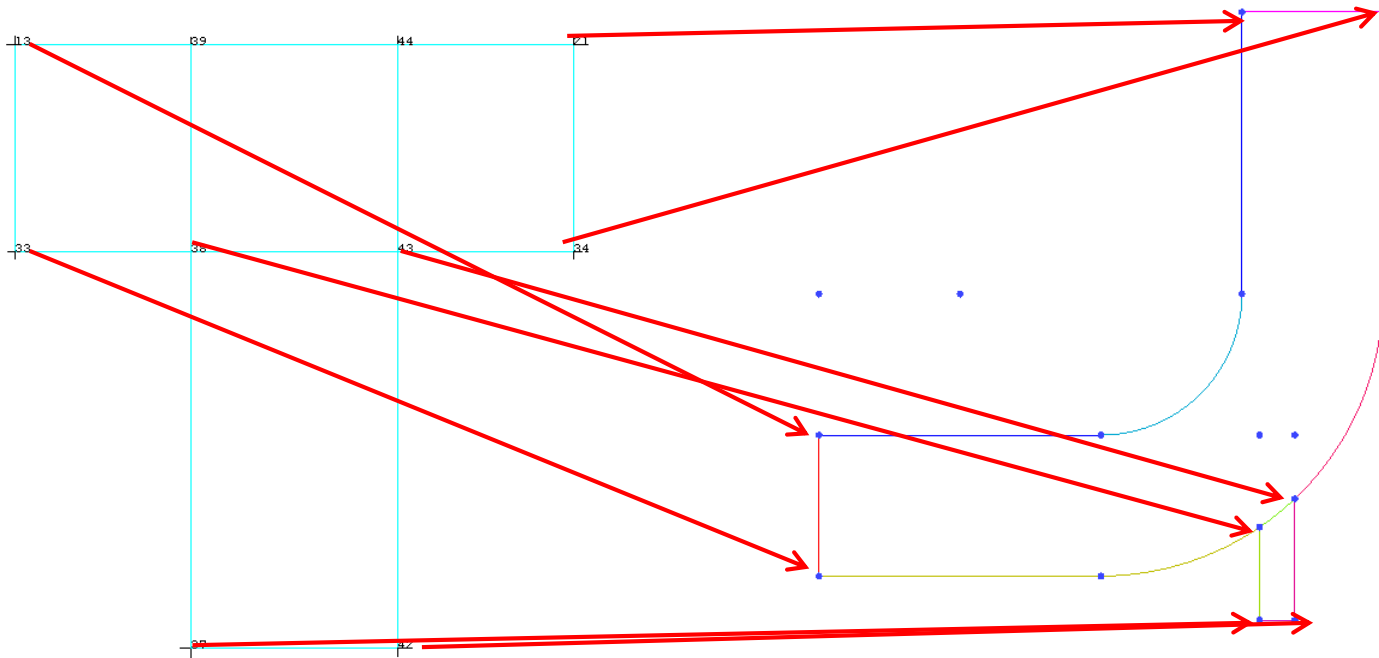
1) Select associate vertex

2) Entity: point

Select Hexa vertex with the left button; right = cancel; '?' = list options.

Select – LMB  
Accept – MMB  
Cancel – RMB

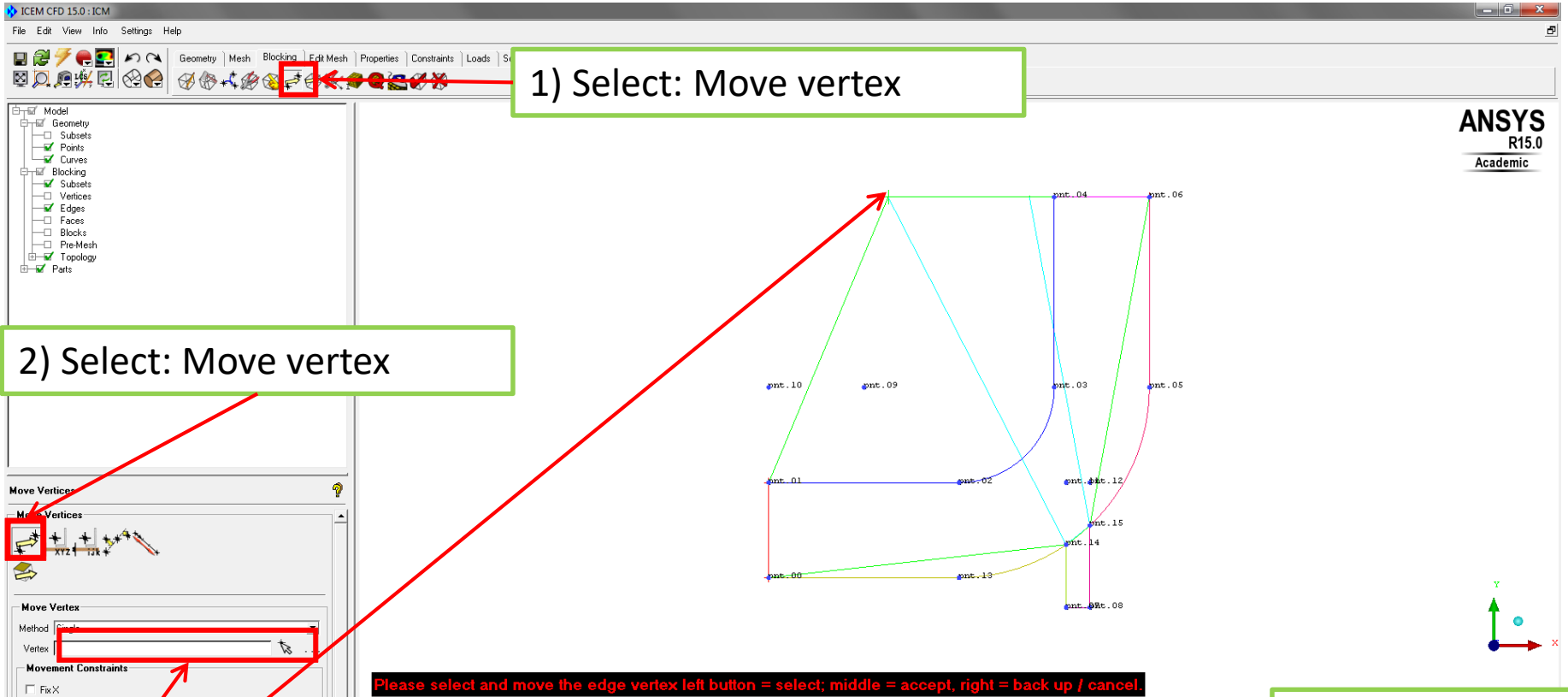
# Associate the vertices to points



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

Select – LMB  
Cancel – RMB

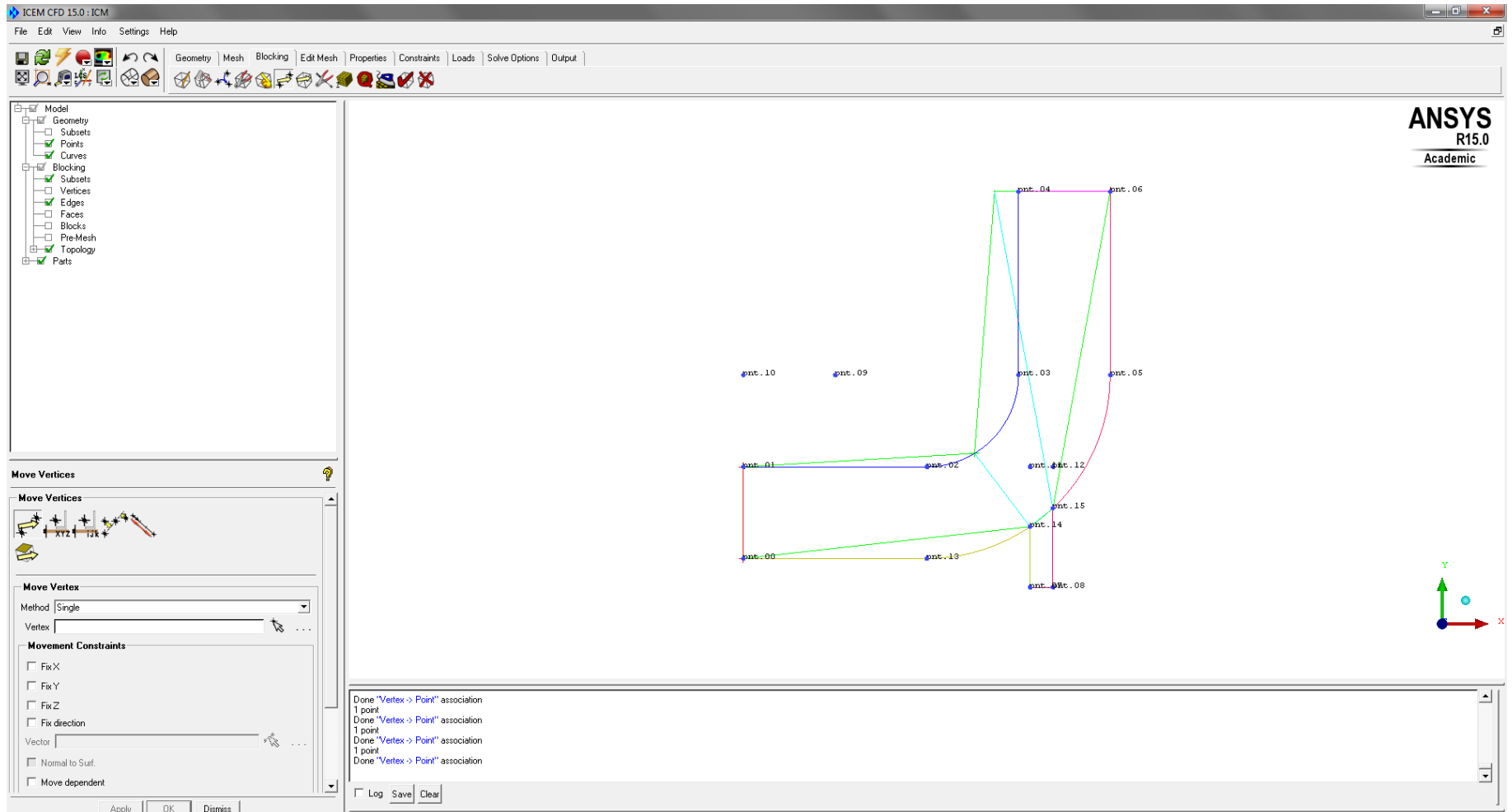
# Move unassociated vertices



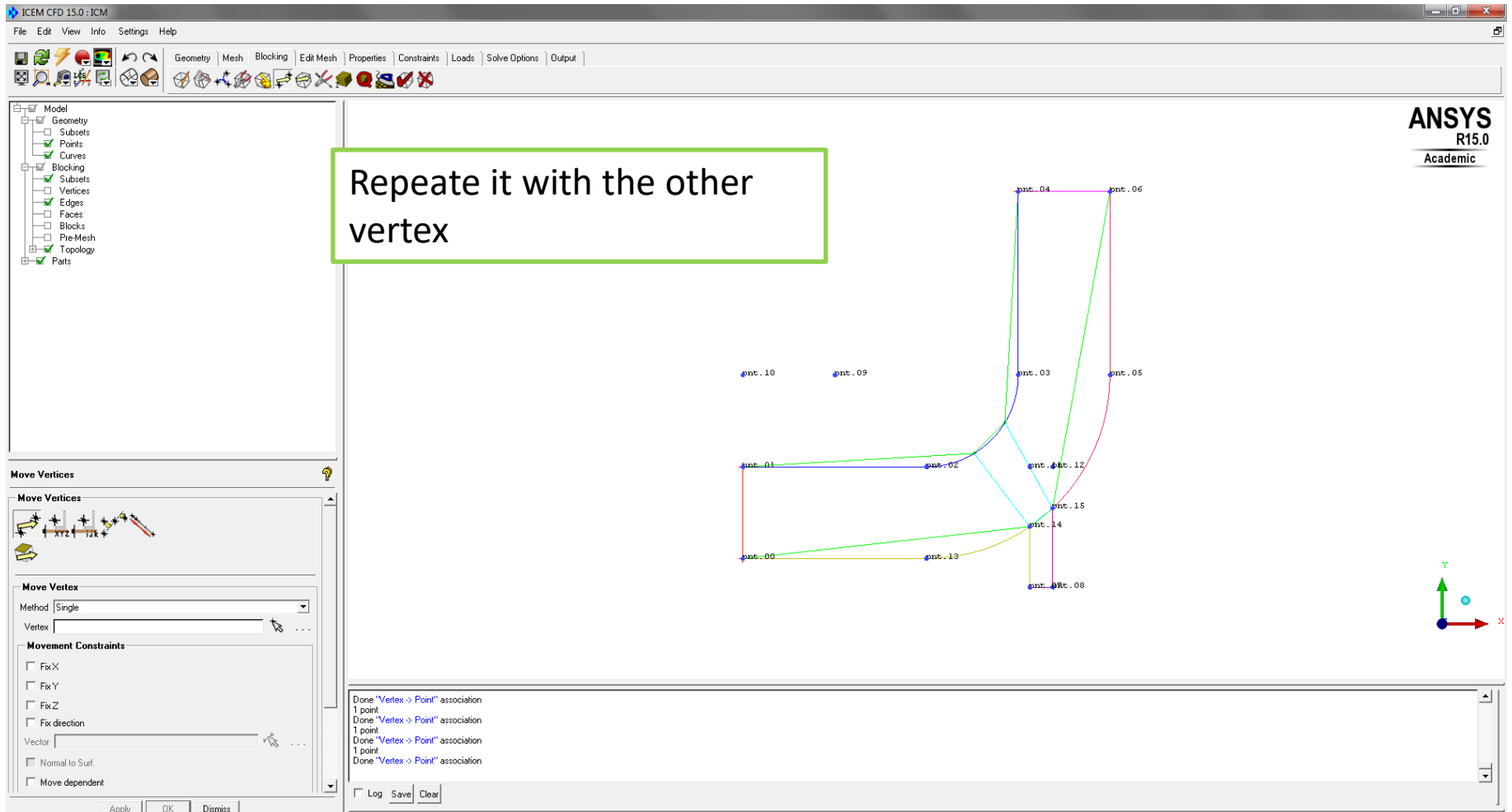
3) Grab (hold the LMB) the marked vertex and drag it to the right position (see it on the next slide)

- Select – LMB
- Accept – MMB
- Cancel – RMB

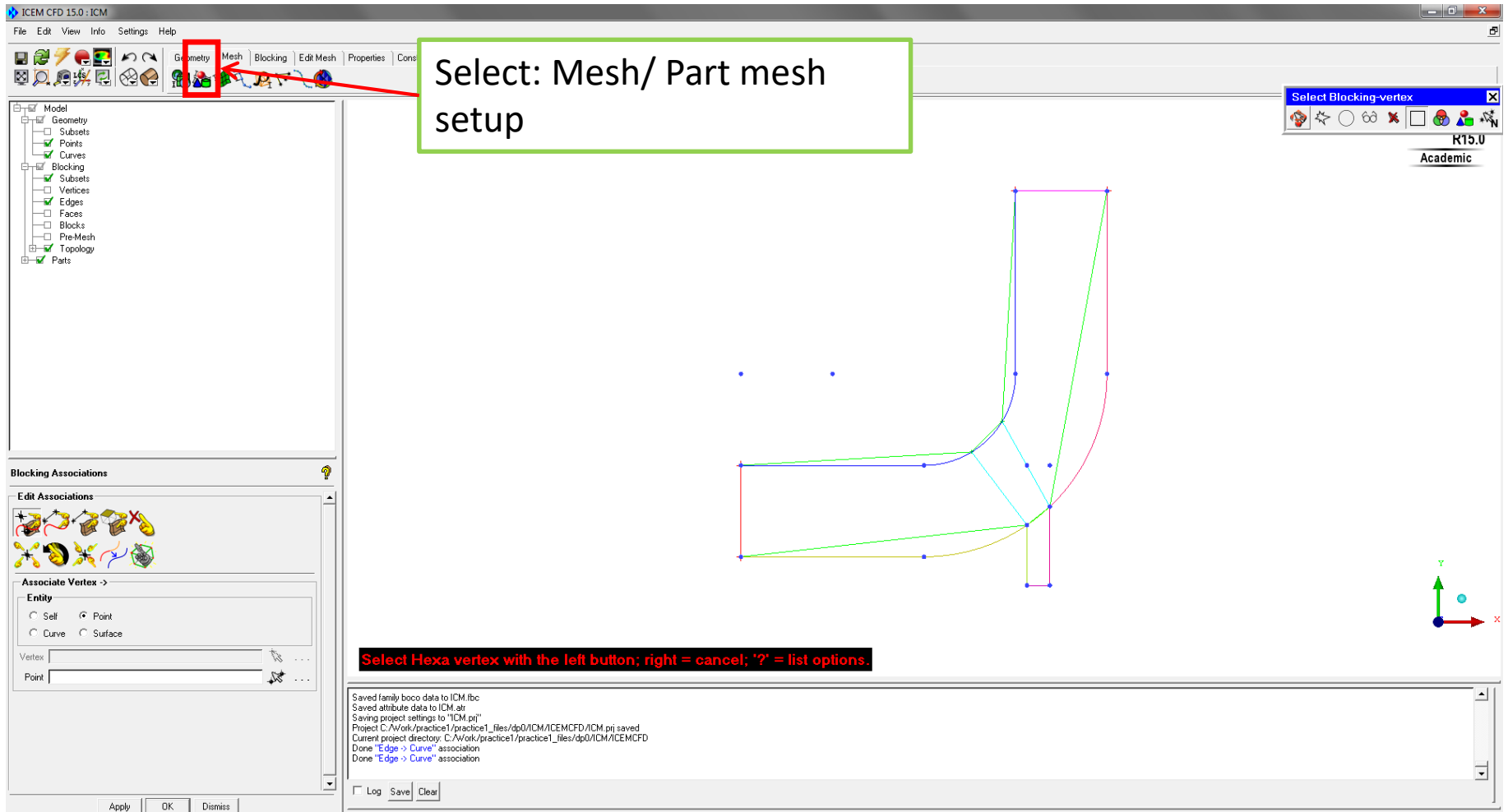
# Move unassociated vertices



# Move unassociated vertices

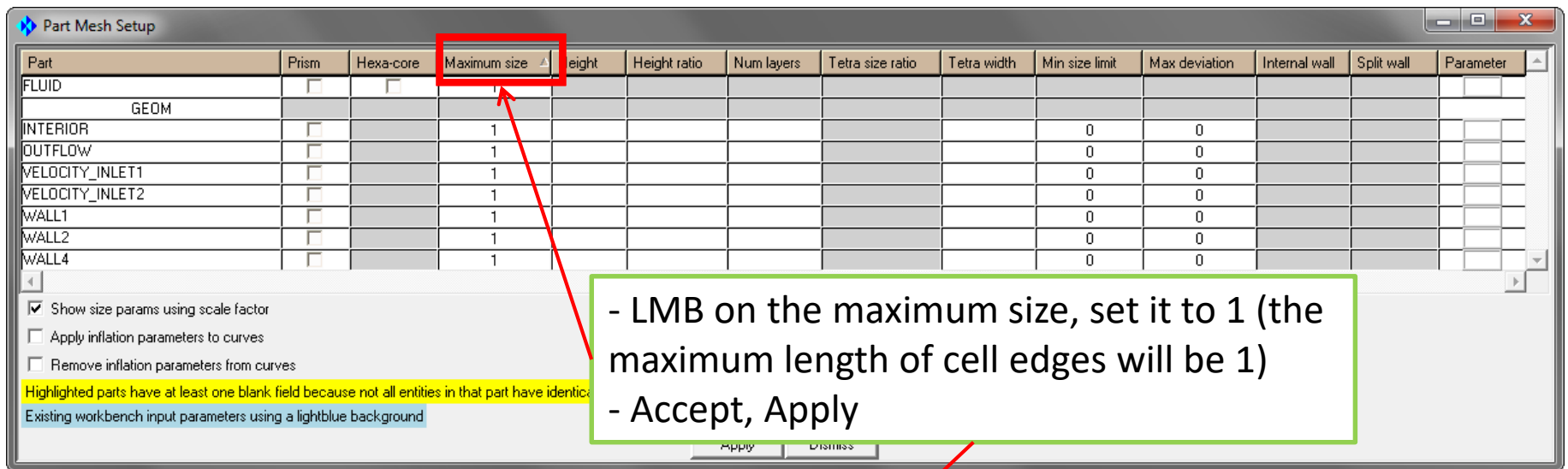


# Setting mesh parameters





# Setting mesh parameters



Part Mesh Setup

Part	Prism	Hexa-core	Maximum size	Height	Height ratio	Num layers	Tetra size ratio	Tetra width	Min size limit	Max deviation	Internal wall	Split wall	Parameter
FLUID	<input type="checkbox"/>	<input type="checkbox"/>											
GEOM													
INTERIOR	<input type="checkbox"/>		1						0	0			
OUTFLOW	<input type="checkbox"/>		1						0	0			
VELOCITY_INLET1	<input type="checkbox"/>		1						0	0			
VELOCITY_INLET2	<input type="checkbox"/>		1						0	0			
WALL1	<input type="checkbox"/>		1						0	0			
WALL2	<input type="checkbox"/>		1						0	0			
WALL4	<input type="checkbox"/>		1						0	0			

☒ Show size params using scale factor  
☐ Apply inflation parameters to curves  
☐ Remove inflation parameters from curves

Highlighted parts have at least one blank field because not all entities in that part have identical parameters.  
Existing workbench input parameters using a lightblue background

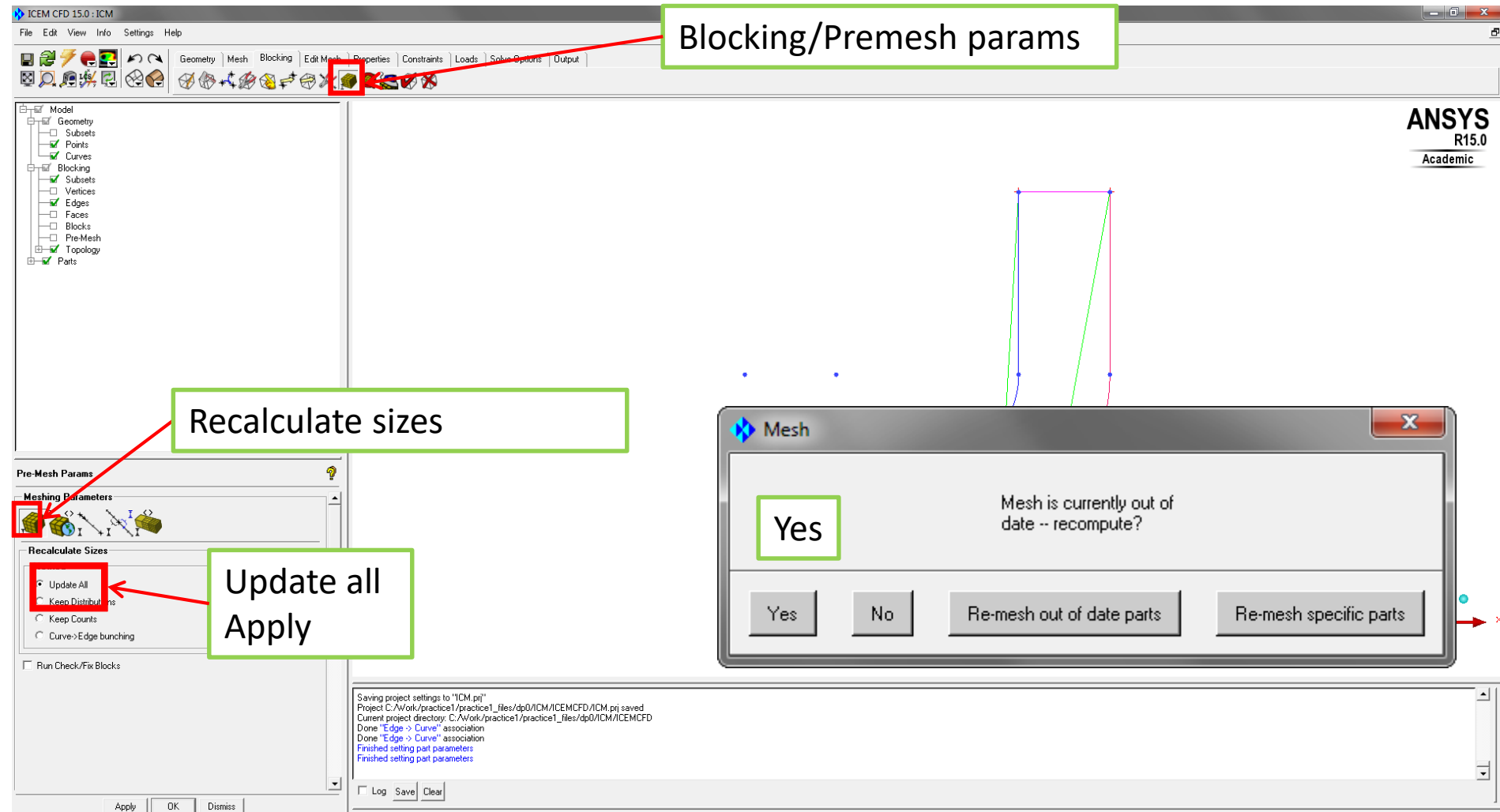
- LMB on the maximum size, set it to 1 (the maximum length of cell edges will be 1)  
- Accept, Apply

MAXIMUM...

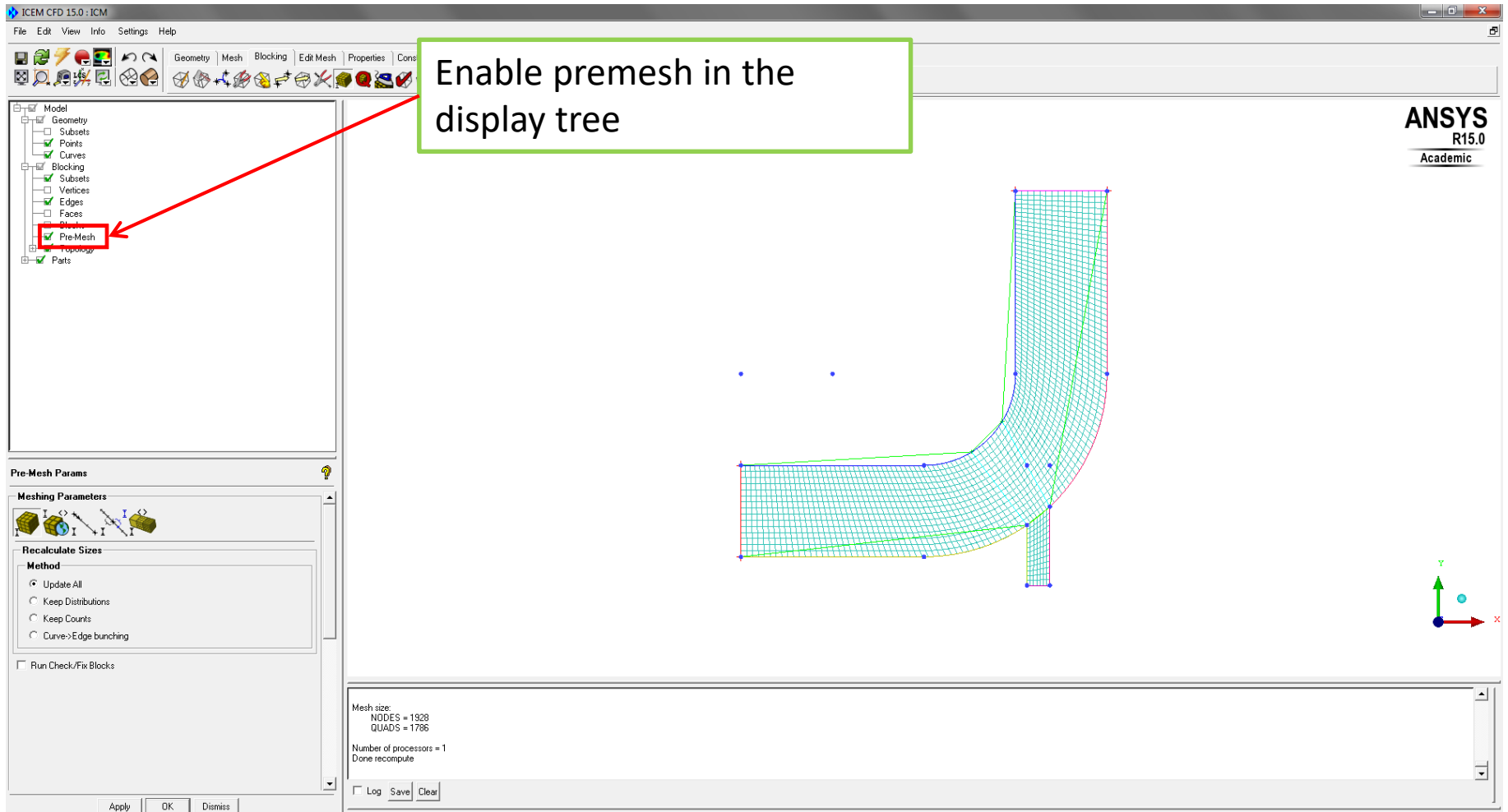
Maximum size 1

Accept Cancel

# Setting mesh parameters



# The premesh



# Refine the mesh

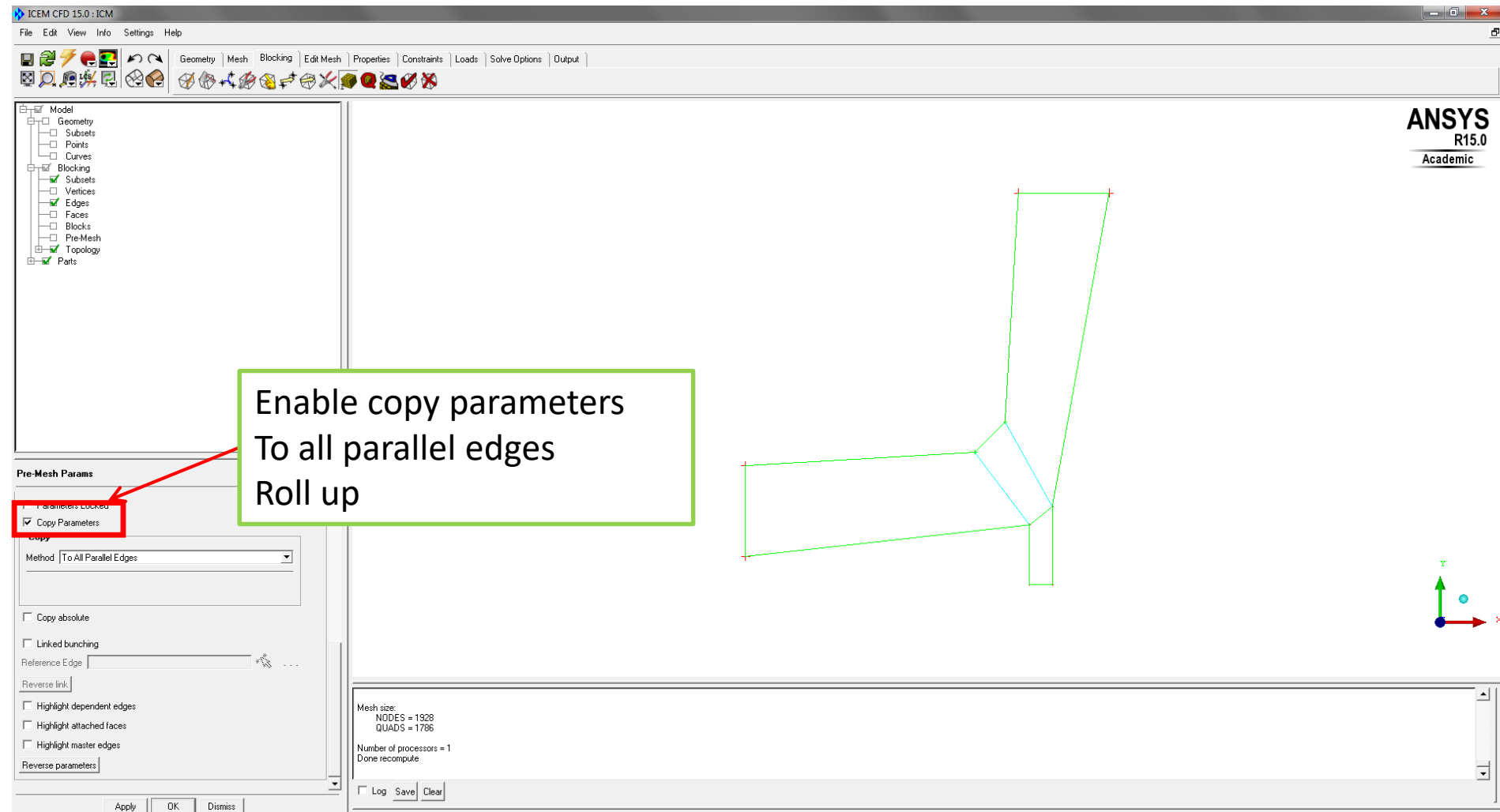
Blocking/Premesh params

Edge parameters, Scroll down

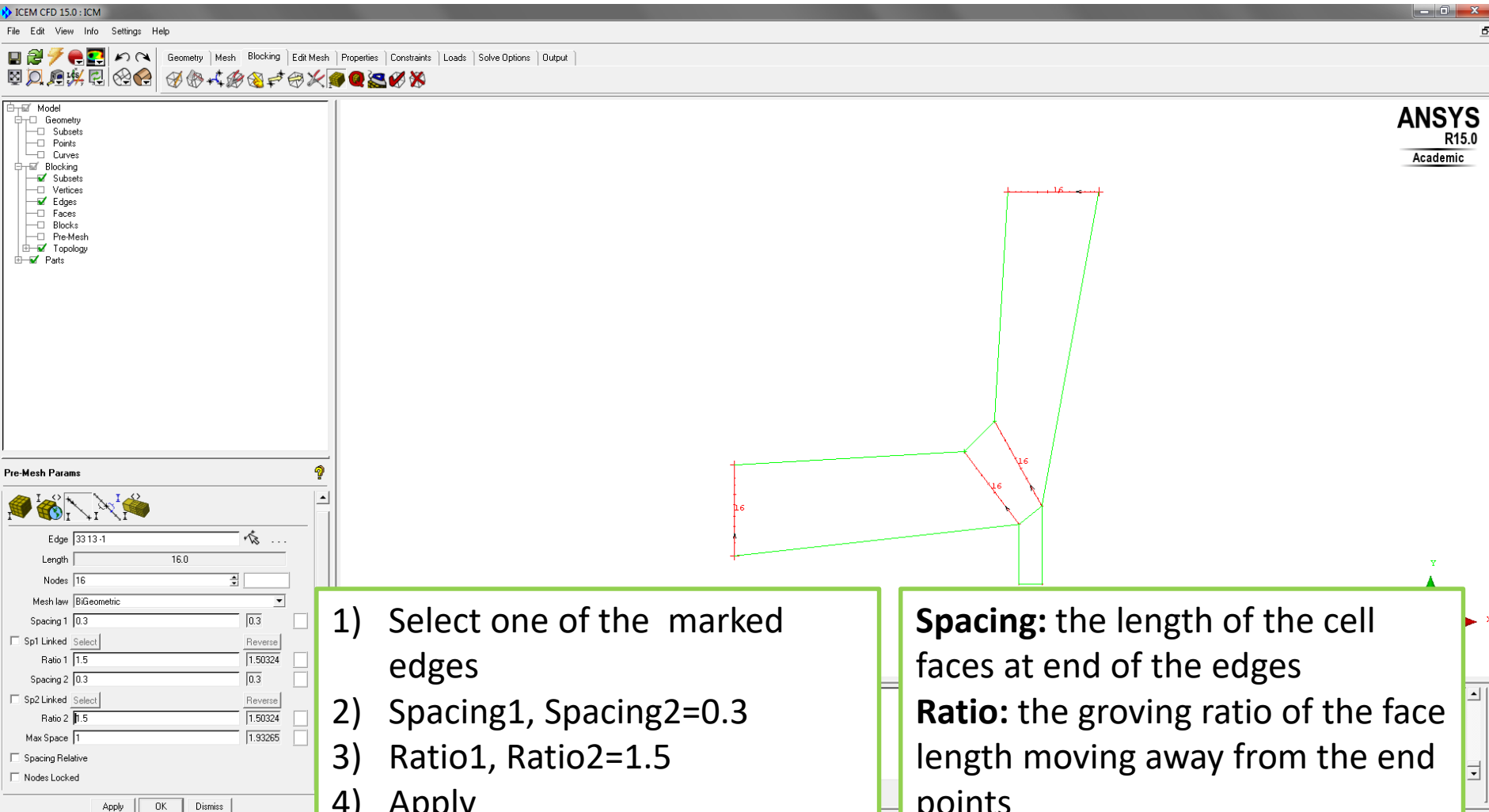
Select edge to set params with the left button; right = cancel; '?' = list options.

Mesh size:  
NODES = 1928  
QUADS = 1786  
Number of processors = 1  
Done recompute

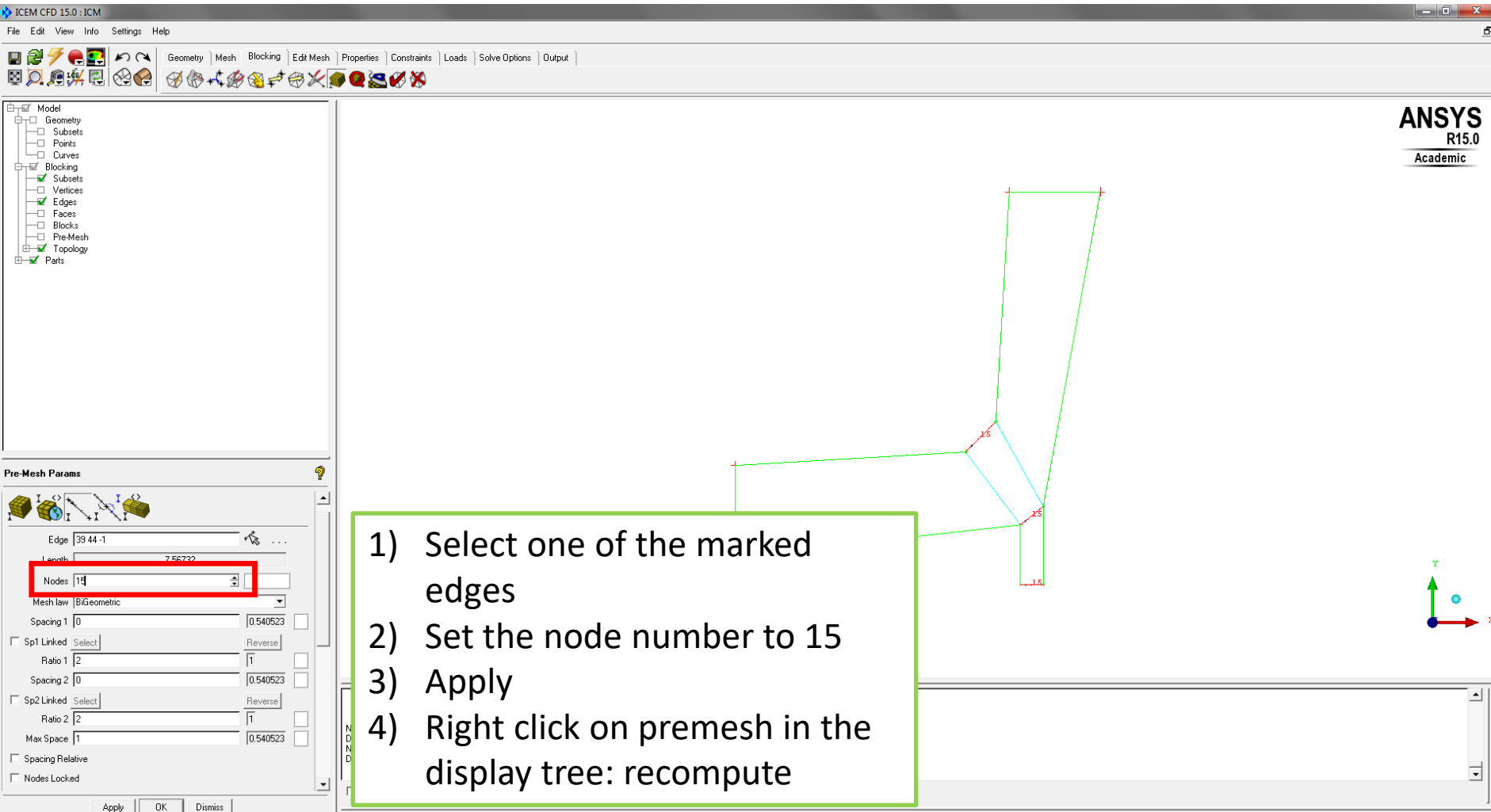
# Refine the mesh



# Refine the mesh



# Refine the mesh



ANSYS R15.0 Academic

Model

- Geometry
  - Subsets
  - Points
  - Curves
- Blocking
  - Subsets
  - Vertices
  - Edges
  - Faces
  - Blocks
  - Pre-Mesh
- Topology
- Parts

Pre-Mesh Params

Edge: 39.44.1

Length: 7.56732

Nodes: 15

Mesh law: BiGeometric

Spacing 1: 0 [0.540523]

Sp1 Linked: Select Reverse

Ratio 1: 2 [1]

Spacing 2: 0 [0.540523]

Sp2 Linked: Select Reverse

Ratio 2: 2 [1]

Max Space: 1 [0.540523]

Spacing Relative

Nodes Locked

Apply OK Dismiss

- 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

The screenshot shows the ICEM CFD 15.0: ICM software interface. The main window displays a 3D model of a wing-like geometry with several edges highlighted in red, indicating they are selected for matching. The left sidebar shows the Model tree with various entities like Geometry, Subsets, Points, Curves, Blocking, Subsets, Vertices, Edges, Faces, Blocks, Pre-Mesh, Topology, and Parts. The bottom-left panel shows the Pre-Mesh Params dialog, with the Match Edge section highlighted by a red box. The Match Edge section includes a Method dropdown set to 'Selected', a Reference Edge field containing '38 43 -1', and a Target Edge(s) field. The bottom status bar shows the number of processors and recomputation status.

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

select target edge(s) with the left button; middle = done, right = back up / cancel, '?' = list options.

Select – LMB  
Cancel – RMB



# Reducing the cell size change

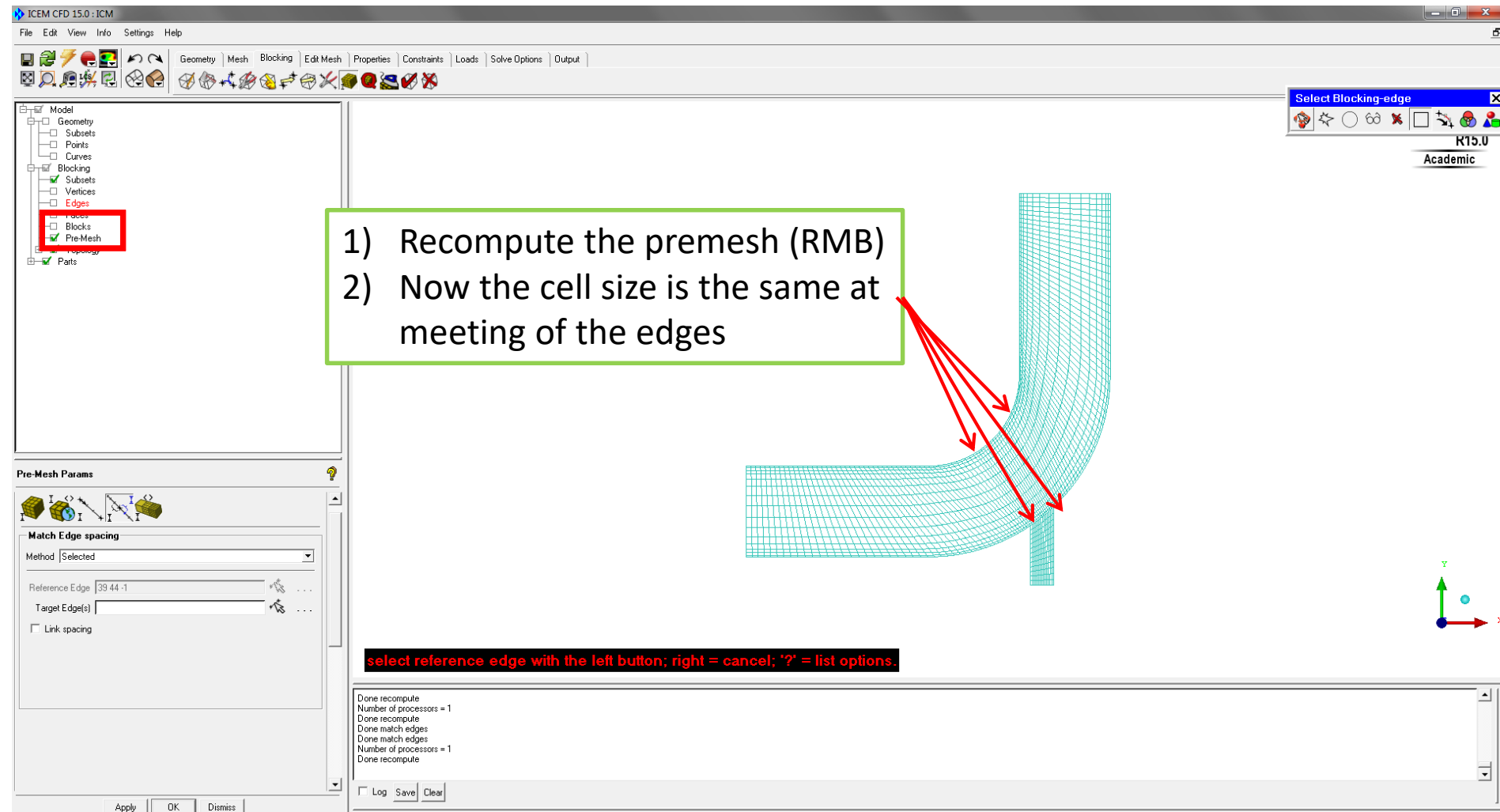
The screenshot shows the ICEM CFD 15.0 interface. The main window displays a 3D model of a mechanical part with several edges highlighted in green. A green box contains the following instructions:

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

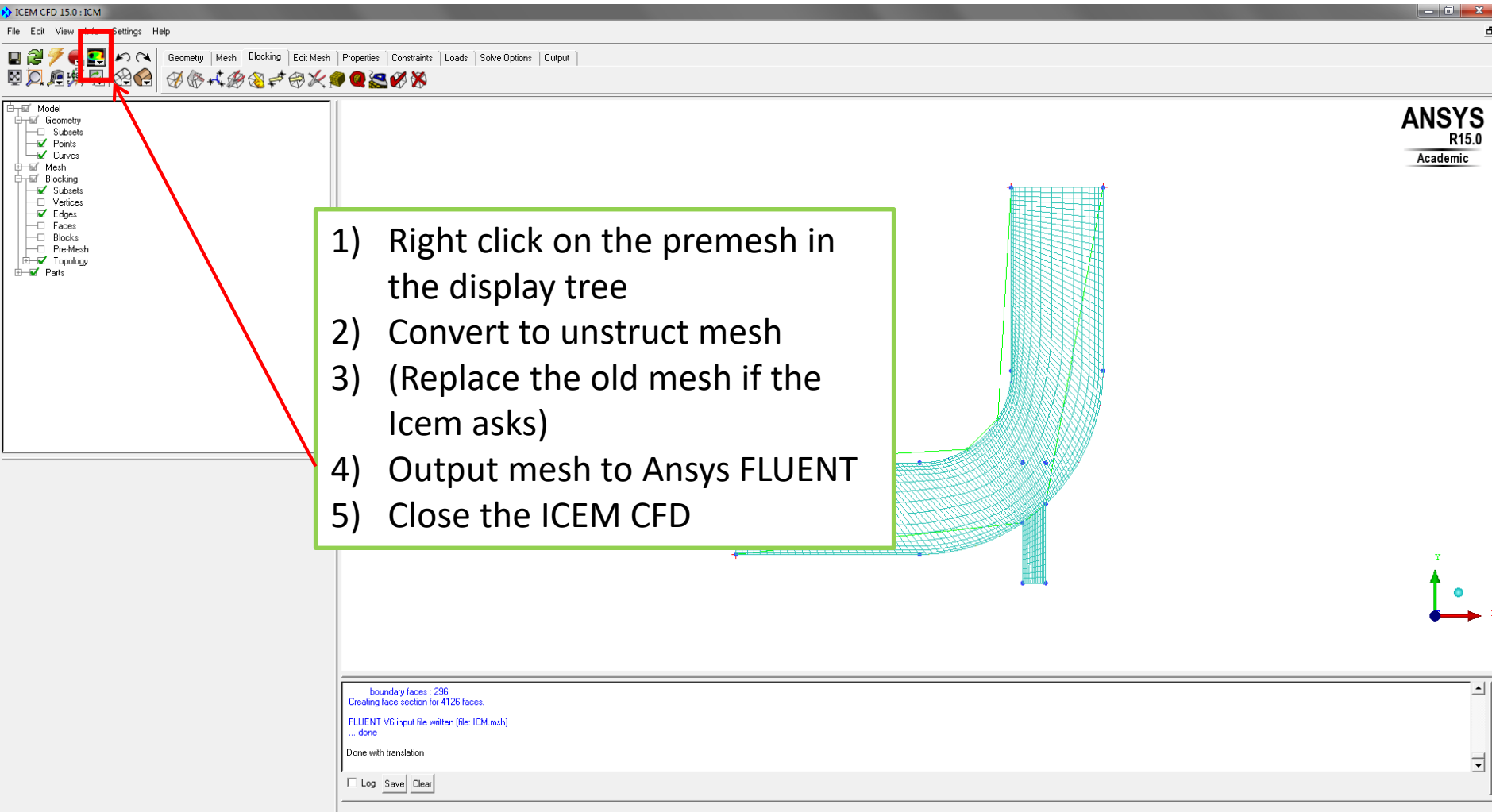
Red arrows point from these instructions to the corresponding fields in the Match Edge dialog. The dialog is titled "Match Edge" and has a "Method" dropdown set to "Selected". The "Reference Edge" field contains "38 43 -1". The "Target Edge(s)" field is empty. The "Link spacing" checkbox is unchecked. The "Pre-Mesh Params" panel is visible on the left, and the "Select Blocking-edge" panel is on the right. A red box highlights the "Match Edge" button in the "Pre-Mesh Params" panel. A red text box at the bottom of the dialog area reads: "select target edge(s) with the left button; middle = done, right = back up / cancel, '?' = list options." The status bar at the bottom shows "Number of processors = 1" and "Done recompute".

**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

boundary faces : 236  
Creating face section for 4126 faces.  
FLUENT V6 input file written (file: ICM.msh)  
... done  
Done with translation  
Log Save Clear

ANSYS R15.0 Academic

# 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](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

The screenshot shows the ANSYS Workbench interface. On the left is the 'Toolbox' containing various analysis systems. The 'Project Schematic' area in the center shows a workflow with two main components: 'ICEM CFD' (labeled A) and 'Fluent' (labeled B). The 'Fluent' component has a sub-component 'Setup' which is highlighted with a red rectangular box. A red arrow points from a callout box to the 'Setup' component. The callout box contains the text '1) Double click on Setup'. The 'Properties of Schematic A2: Model' panel on the right shows a table with columns 'A' and 'B'. The 'Messages' panel at the bottom shows a message about Microsoft Office Excel. The 'Progress' panel at the bottom shows a table with columns 'A', 'B', and 'C'.

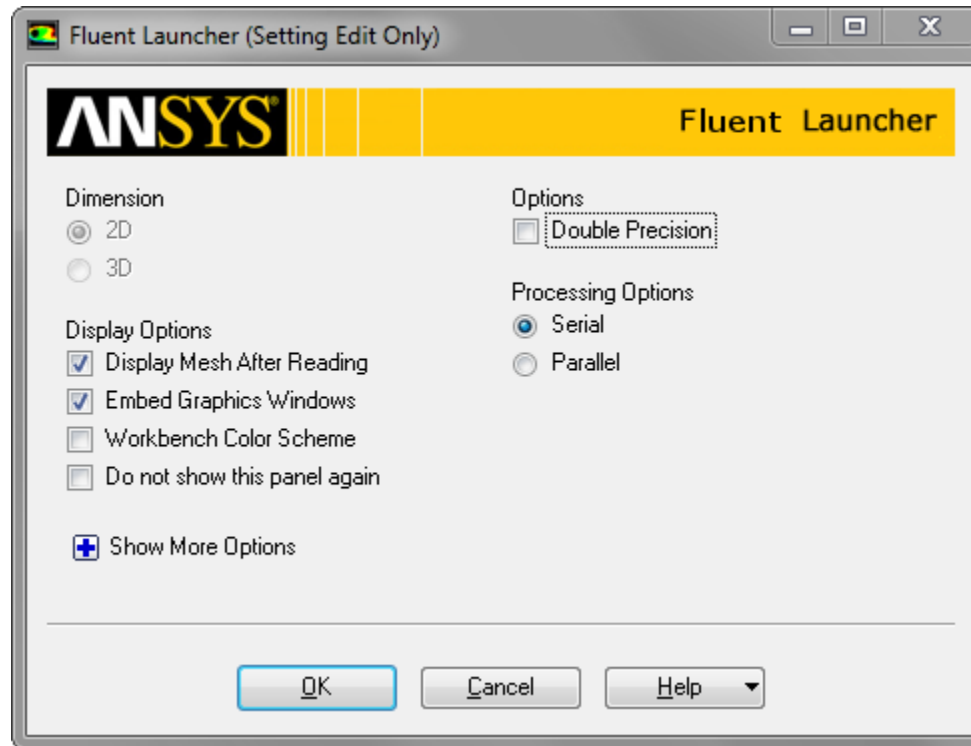
1) Double click on Setup

A	B
Properties	Value
	ICM
	ICM

A	B	C	D	
1	Type	Text	Association	Date/Time
2	Informational	The installed Microsoft Office Excel application is not supported. You may meet some issues while using the Microsoft Office Excel system. Look at the Help for the list of supported Microsoft Office Excel releases.		2014.09.07. 14:47:12

A	B	C	
1	Status	Details	Progress

# Starting FLUENT



OK

# Scaling the mesh

B:Fluent Fluent@CFDLAB-08 [2d, pbns, lam] [ANSYS Academic Research CFD]

File Mesh Define Solve Adapt Surface Display Report Parallel View Help

Meshing

Mes Generation

Solution Setup

General

Models

Materials

Phases

Cell Zone Conditions

Boundary Conditions

Mesh Interfaces

Dynamic Mesh

Reference Values

Solution

Solution Methods

Solution Controls

Monitors

Solution Initialization

Calculation Activities

Run Calculation

Results

Graphics and Animations

Plots

Reports

General

Mesh

Scale... Check Report Quality

Display...

Solver

Type

Pressure-Based Density-Based

Velocity Formulation

Absolute Relative

Time

Steady Transient

2D Space

Planar Axisymmetric Axisymmetric Swirl

Gravity

Units...

Help

1: Mesh

1) Mesh/Scale

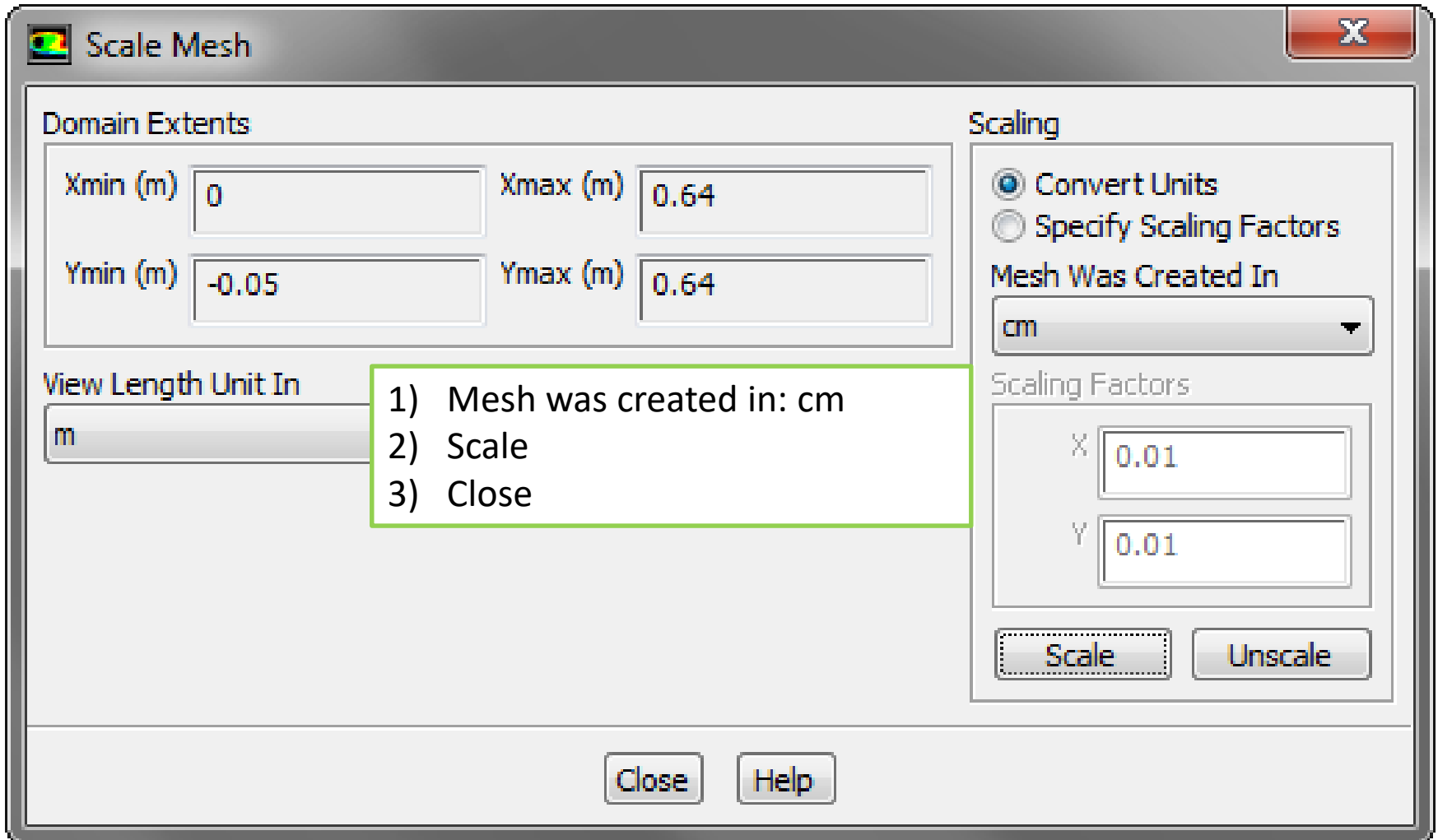
Mesh

Sep 07, 2014  
ANSYS Fluent 15.0 (2d, pbns, lam)

```
int_fluid
fluid
Done.
Preparing mesh for display...
Done.
Preparing mesh for display...
Done.
Writing Settings file "C:\Work\practice1\practice1_files\dp0\FLU\Fluent\ICM.set"...
writing rp variables ... Done.
writing domain variables ... Done.
writing fluid (type fluid) (mixture) ... Done.
writing int_fluid (type interior) (mixture) ... Done.
writing velocity_inlet1 (type velocity-inlet) (mixture) ... Done.
writing velocity_inlet2 (type velocity-inlet) (mixture) ... Done.
writing wall1 (type wall) (mixture) ... Done.
writing wall2 (type wall) (mixture) ... Done.
writing wall4 (type wall) (mixture) ... Done.
writing int_interior (type interior) (mixture) ... Done.
writing outflow (type outflow) (mixture) ... Done.
writing zones map name-id ... Done.
```



# Scaling the mesh



The image shows a software dialog box titled "Scale Mesh". It is divided into two main sections: "Domain Extents" and "Scaling".

**Domain Extents:**

- Xmin (m): 0
- Xmax (m): 0.64
- Ymin (m): -0.05
- Ymax (m): 0.64

**View Length Unit In:** m

**Scaling:**

- ☒ Convert Units
- ☐ Specify Scaling Factors
- Mesh Was Created In: cm
- Scaling Factors:
  - X: 0.01
  - Y: 0.01
- Buttons: Scale, Unscale

At the bottom of the dialog are "Close" and "Help" buttons.

A green rectangular box highlights the following steps:

- 1) Mesh was created in: cm
- 2) Scale
- 3) Close

# Turbulence Model

B:Fluent Fluent@CFDLAB-08 [2d, pbns, lam] [ANSYS Academic Research CFD]

File Mesh Define Solve Adapt Surface Display Report Parallel View Help

Meshing

Mesh Generation

Solution Setup

General

Models

Materials

Phases

Cell Zone Conditions

Boundary Conditions

Mesh Interfaces

Dynamic Mesh

Reference Values

Solution

Solution Methods

Solution Controls

Monitors

Solution Initialization

Calculation Activities

Run Calculation

Results

Graphics and Animations

Plots

Reports

Models

Models

Multiphase - Off

Viscous - On

Radiation - Off

Heat Exchanger - Off

Species - Off

Discrete Phase - Off

Solidification & Melting - Off

Acoustics - Off

Edit...

Help

1: Mesh

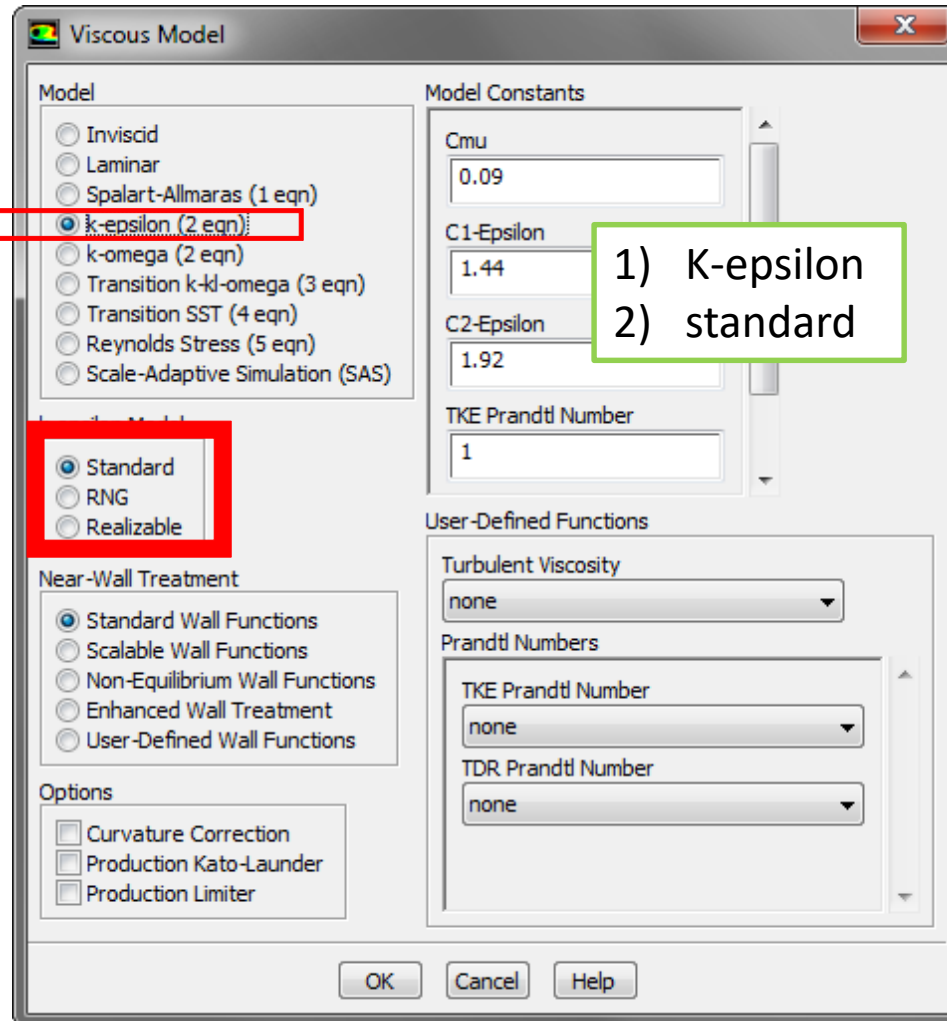
1) Double click on Model/Viscous

Mesh

Sep 07, 2014  
ANSYS Fluent 15.0 (2d, pbns, lam)

```
int_fluid
fluid
Done.
Preparing mesh for display...
Done.
Preparing mesh for display...
Done.
Writing Settings file "C:\Work\practice1\practice1_files\dp0\FLU\Fluent\ICM.set"...
writing rp variables ... Done.
writing domain variables ... Done.
writing fluid (type fluid) (mixture) ... Done.
writing int_fluid (type interior) (mixture) ... Done.
writing velocity_inlet1 (type velocity-inlet) (mixture) ... Done.
writing velocity_inlet2 (type velocity-inlet) (mixture) ... Done.
writing wall1 (type wall) (mixture) ... Done.
writing wall2 (type wall) (mixture) ... Done.
writing wall4 (type wall) (mixture) ... Done.
writing int_interior (type interior) (mixture) ... Done.
writing outflow (type outflow) (mixture) ... Done.
writing zones map name-id ... Done.
```

# Turbulence model



# Boundary conditions

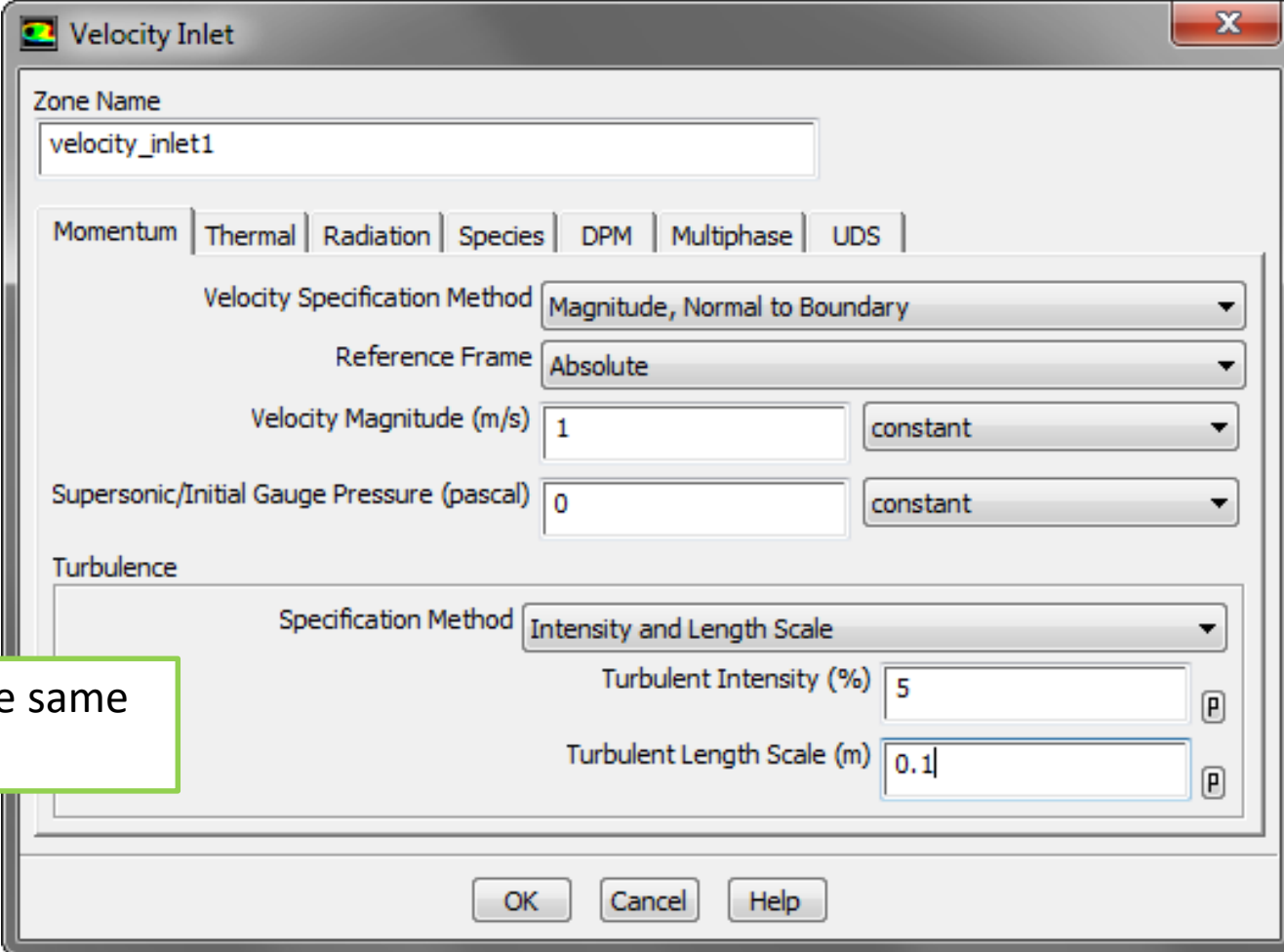
The screenshot displays the ANSYS Fluent 15.0 interface. On the left, the 'Boundary Conditions' panel is active, showing a list of zones: 'int\_fluid', 'int\_interior', 'velocity\_inlet1', 'wall1', 'wall2', and 'wall4'. The 'velocity\_inlet1' zone is selected. Below the list, the 'Type' dropdown is set to 'velocity-inlet'. The 'Edit...' button is highlighted. A red box also highlights the 'Cell Zone Conditions' option in the left-hand menu. The main window shows a green wireframe mesh of a U-bend. A text box with a green border contains the following steps:

- 1) Boundary conditions
- 2) Select: velocity\_inlet1
- 3) Type: velocity-inlet
- 4) Edit

The bottom console window shows the following output:

```
int_fluid
fluid
Done.
Preparing mesh for display...
Done.
Preparing mesh for display...
Done.
Writing Settings file "C:\Work\practice1\practice1_files\dp0\FLU\Fluent\ICM.set"...
writing rp variables ... Done.
writing domain variables ... Done.
writing fluid (type fluid) (mixture) ... Done.
writing int_fluid (type interior) (mixture) ... Done.
writing velocity_inlet1 (type velocity-inlet) (mixture) ... Done.
writing velocity_inlet2 (type velocity-inlet) (mixture) ... Done.
writing wall1 (type wall) (mixture) ... Done.
writing wall2 (type wall) (mixture) ... Done.
writing wall4 (type wall) (mixture) ... Done.
writing int_interior (type interior) (mixture) ... Done.
writing outflow (type outflow) (mixture) ... Done.
writing zones map name-id ... Done.
```

# Boundary conditions



The image shows a 'Velocity Inlet' dialog box from a software application. The 'Zone Name' field is set to 'velocity\_inlet1'. The 'Momentum' tab is selected, showing options for 'Velocity Specification Method' (Magnitude, Normal to Boundary), 'Reference Frame' (Absolute), 'Velocity Magnitude (m/s)' (1), and 'Supersonic/Initial Gauge Pressure (pascal)' (0). The 'Turbulence' section shows 'Specification Method' (Intensity and Length Scale), 'Turbulent Intensity (%)' (5), and 'Turbulent Length Scale (m)' (0.1). The 'OK', 'Cancel', and 'Help' buttons are at the bottom.

Zone Name  
velocity\_inlet1

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

Velocity Specification Method: Magnitude, Normal to Boundary

Reference Frame: Absolute

Velocity Magnitude (m/s): 1 constant

Supersonic/Initial Gauge Pressure (pascal): 0 constant

Turbulence

Specification Method: Intensity and Length Scale

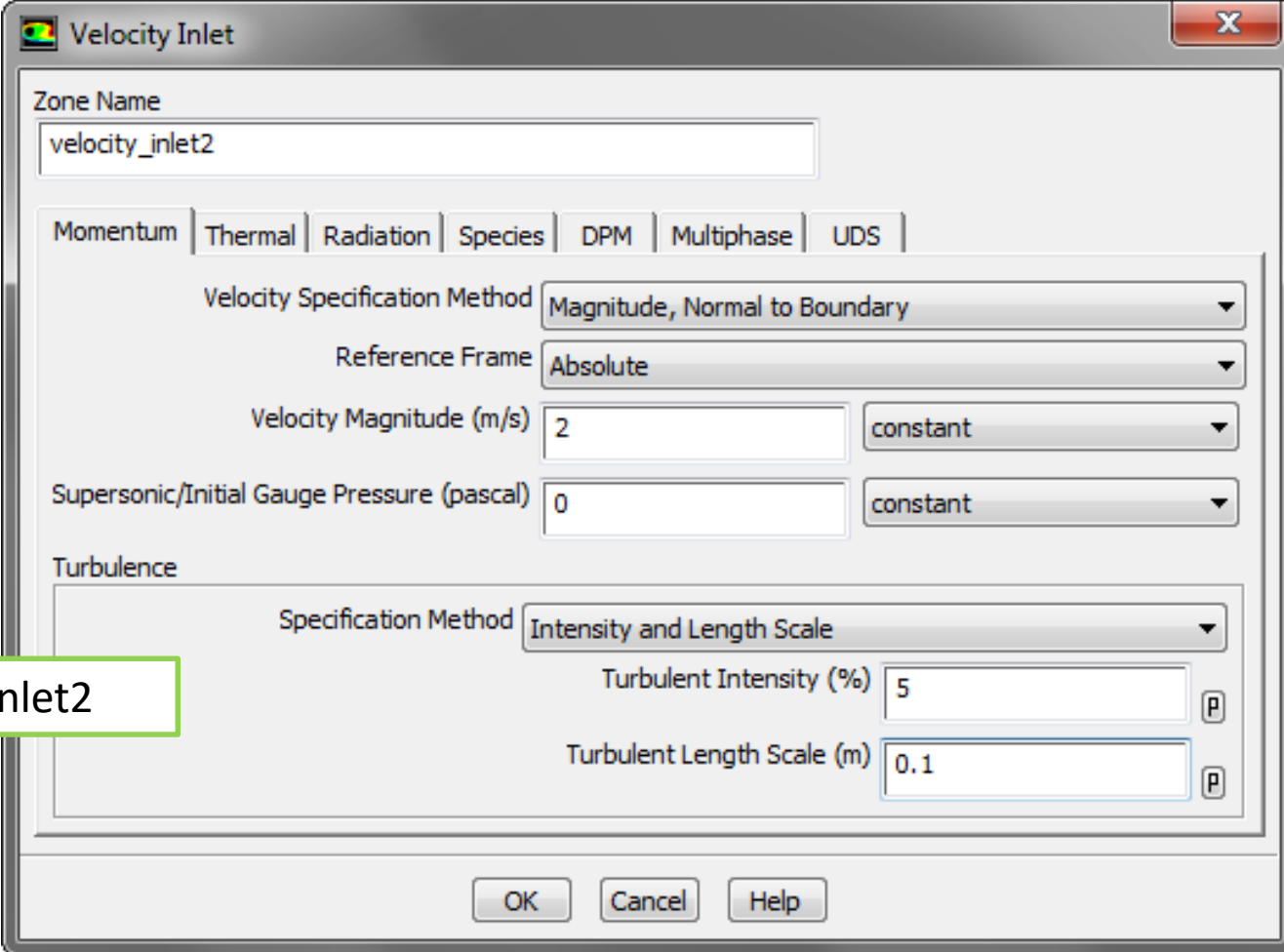
Turbulent Intensity (%): 5 P

Turbulent Length Scale (m): 0.1 P

OK Cancel Help

- 1) Set the same
- 2) OK

# Boundary conditions



The image shows a 'Velocity Inlet' dialog box from a software application. The 'Zone Name' field is set to 'velocity\_inlet2'. The 'Momentum' tab is selected, showing options for 'Velocity Specification Method' (Magnitude, Normal to Boundary), 'Reference Frame' (Absolute), 'Velocity Magnitude (m/s)' (2), and 'Supersonic/Initial Gauge Pressure (pascal)' (0). The 'Turbulence' section shows 'Specification Method' (Intensity and Length Scale), 'Turbulent Intensity (%)' (5), and 'Turbulent Length Scale (m)' (0.1). The dialog has 'OK', 'Cancel', and 'Help' buttons at the bottom.

Zone Name  
velocity\_inlet2

Momentum | Thermal | Radiation | Species | DPM | Multiphase | UDS

Velocity Specification Method: Magnitude, Normal to Boundary

Reference Frame: Absolute

Velocity Magnitude (m/s): 2 constant

Supersonic/Initial Gauge Pressure (pascal): 0 constant

Turbulence

Specification Method: Intensity and Length Scale

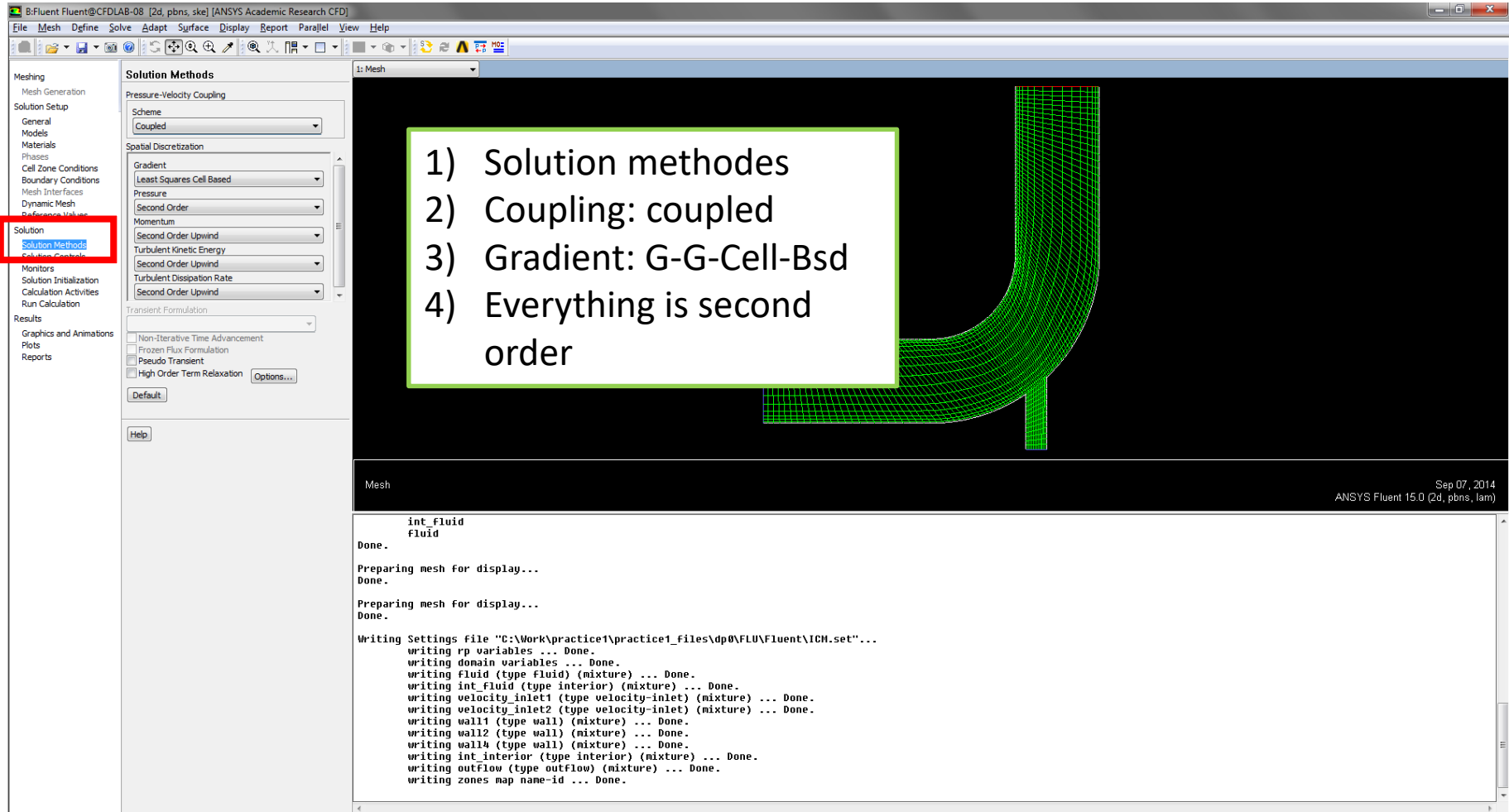
Turbulent Intensity (%): 5 P

Turbulent Length Scale (m): 0.1 P

OK Cancel Help

Velocity\_inlet2

# Discretization Schemes



The image shows the ANSYS Fluent 15.0 interface. On the left, the 'Solution Methods' panel is active, showing the following settings:

- Pressure-Velocity Coupling: Scheme: Coupled
- Spatial Discretization:
  - Gradient: Least Squares Cell Based
  - Pressure: Second Order
  - Momentum: Second Order Upwind
  - Turbulent Kinetic Energy: Second Order Upwind
  - Turbulent Dissipation Rate: Second Order Upwind
- Transient Formulation: Non-Iterative Time Advancement (checked), Frozen Flux Formulation (checked), Pseudo Transient (checked), High Order Term Relaxation (checked)

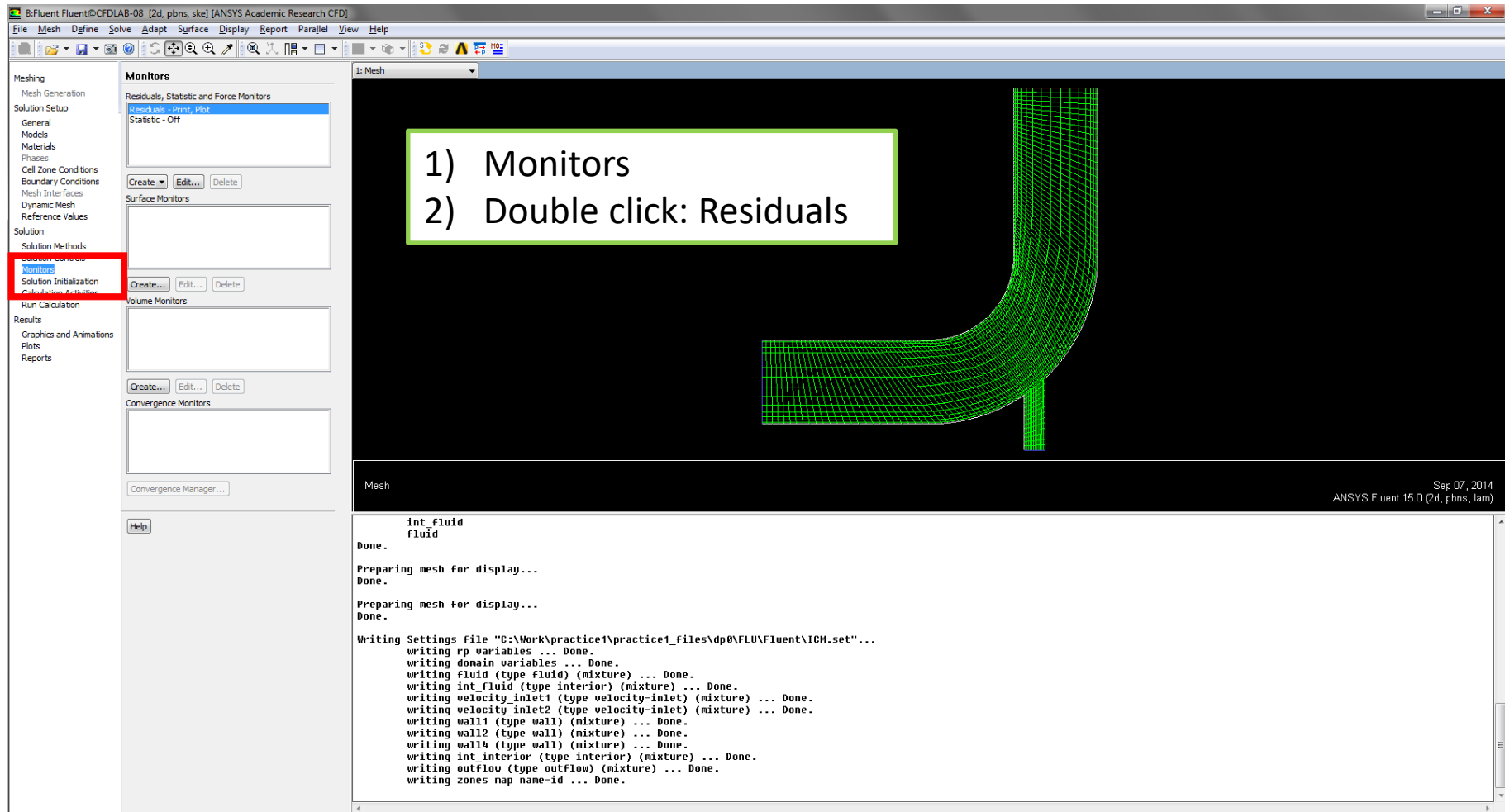
On the right, a 3D mesh of a U-bend is displayed. A text box with a green border contains the following list:

- 1) Solution methods
- 2) Coupling: coupled
- 3) Gradient: G-G-Cell-Bsd
- 4) Everything is second order

At the bottom, the console window shows the following output:

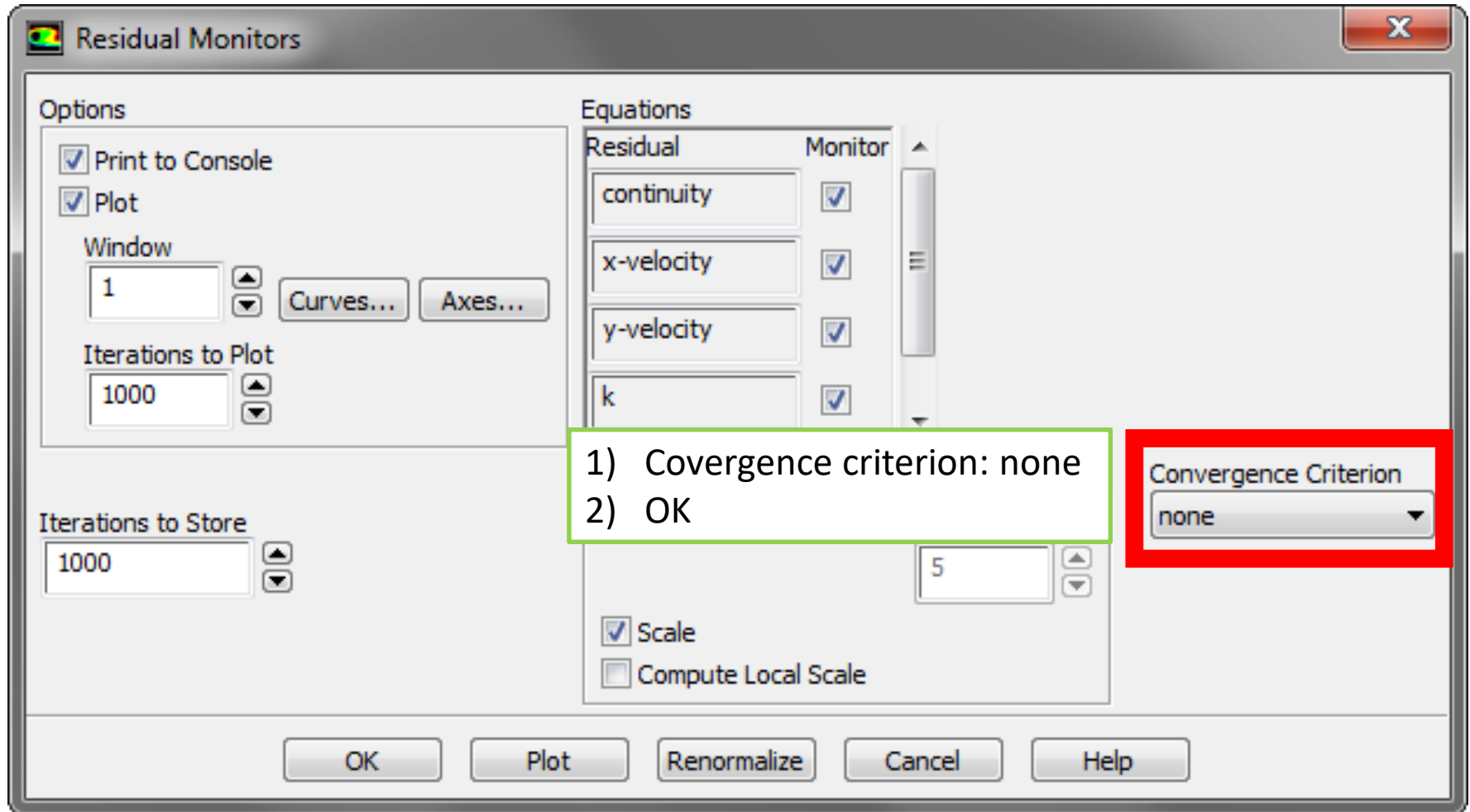
```
int fluid
fluid
Done.
Preparing mesh for display...
Done.
Preparing mesh for display...
Done.
Writing Settings file "C:\Work\practice1\practice1_files\dp0\FLU\Fluent\ICM.set"...
writing rp variables ... Done.
writing domain variables ... Done.
writing fluid (type fluid) (mixture) ... Done.
writing int fluid (type interior) (mixture) ... Done.
writing velocity_inlet1 (type velocity-inlet) (mixture) ... Done.
writing velocity_inlet2 (type velocity-inlet) (mixture) ... Done.
writing wall1 (type wall) (mixture) ... Done.
writing wall2 (type wall) (mixture) ... Done.
writing wall4 (type wall) (mixture) ... Done.
writing int interior (type interior) (mixture) ... Done.
writing outflow (type outflow) (mixture) ... Done.
writing zones map name-id ... Done.
```

# Turn off convergence criterion

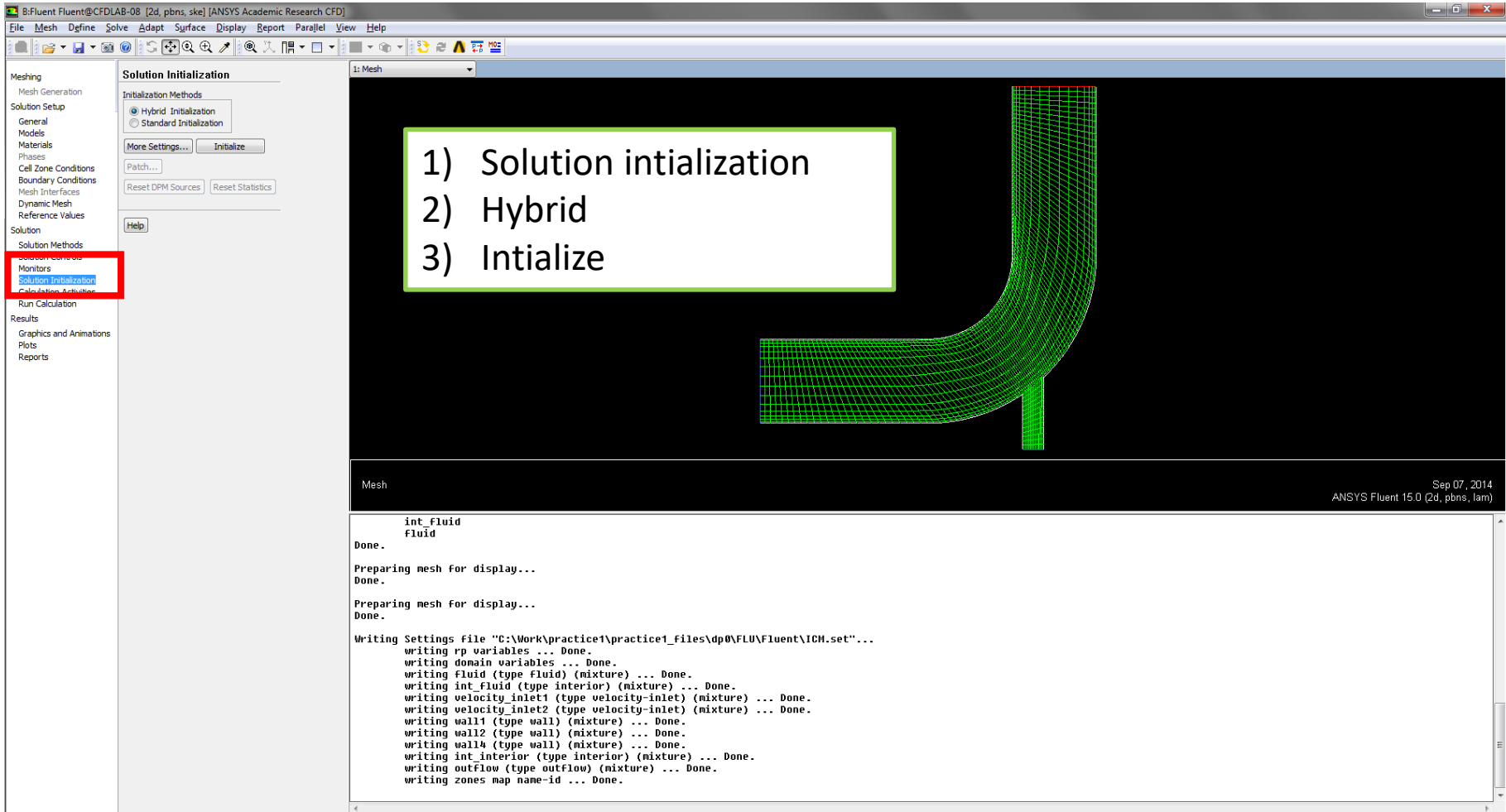




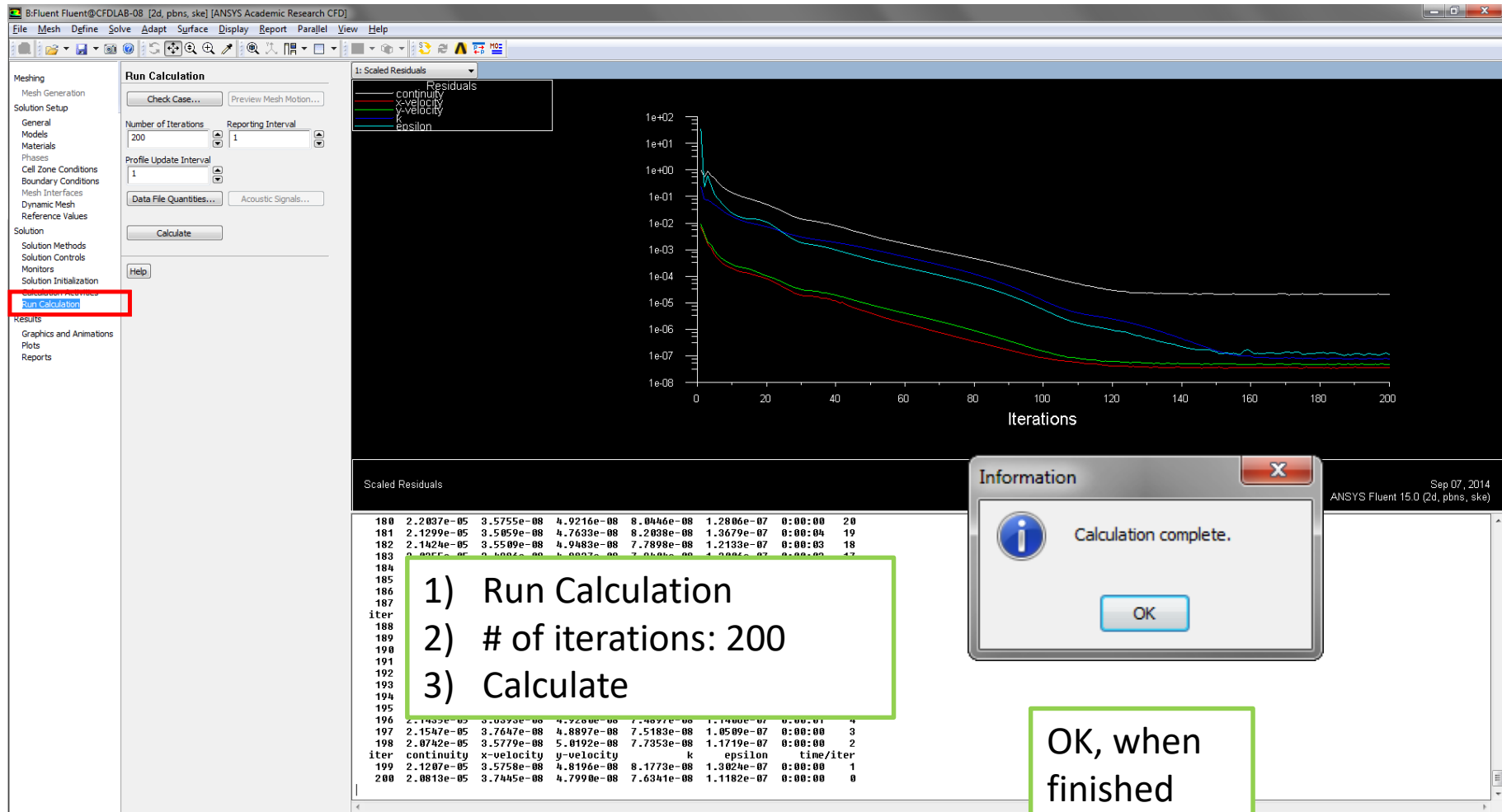
# Turn off convergence criteria



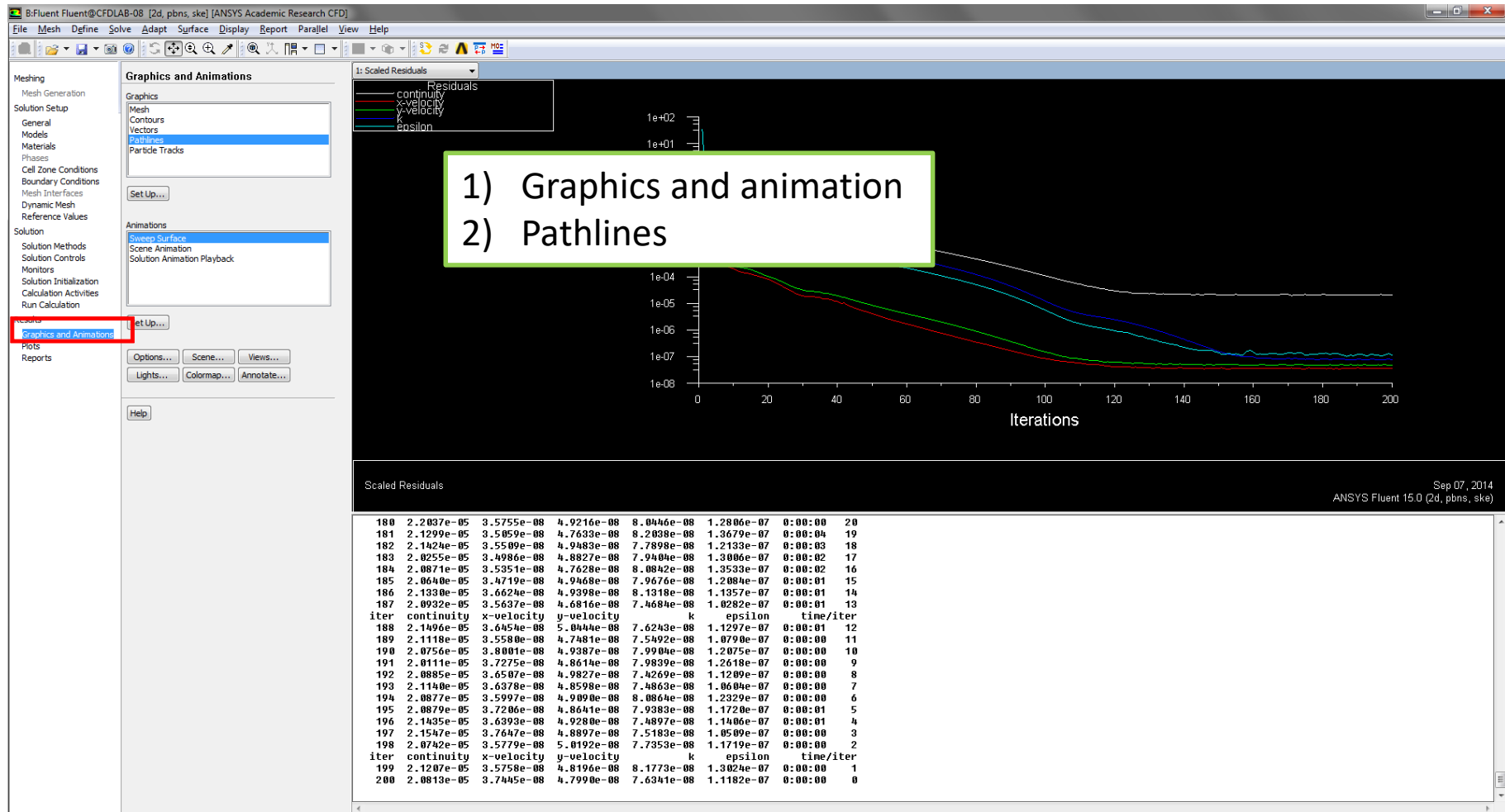
# Intialization



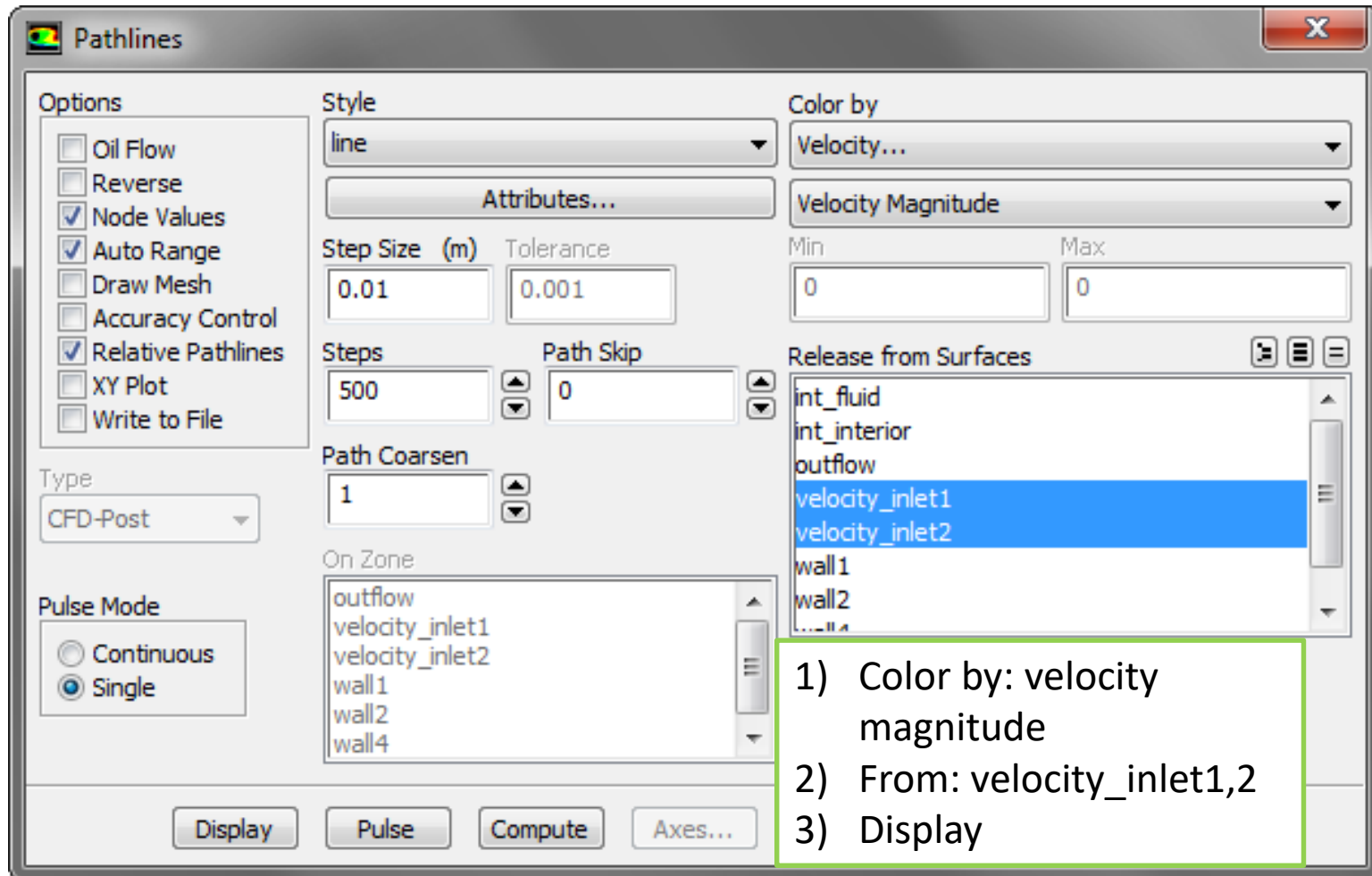
# Calculation



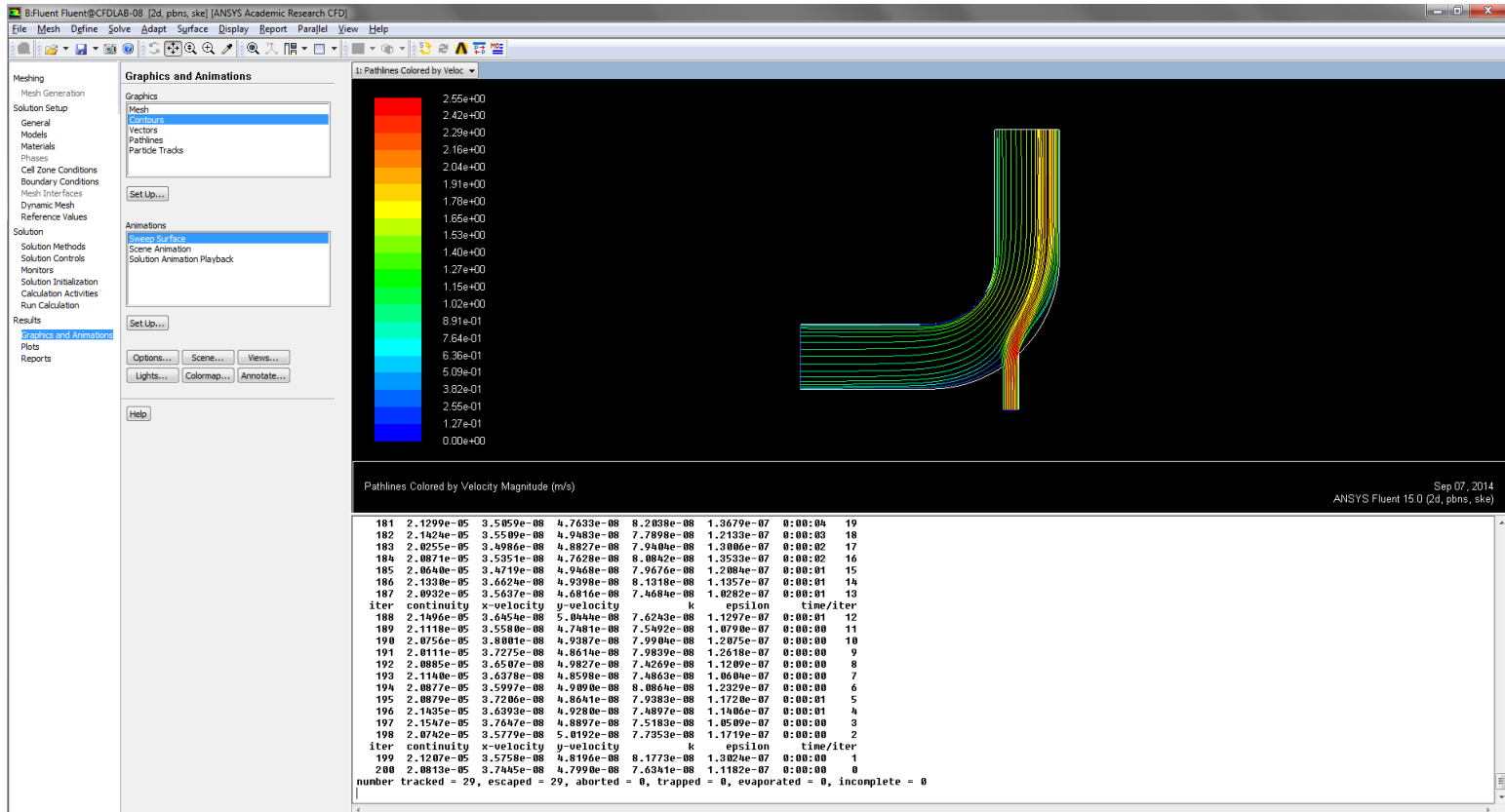
# Drawing pathlines



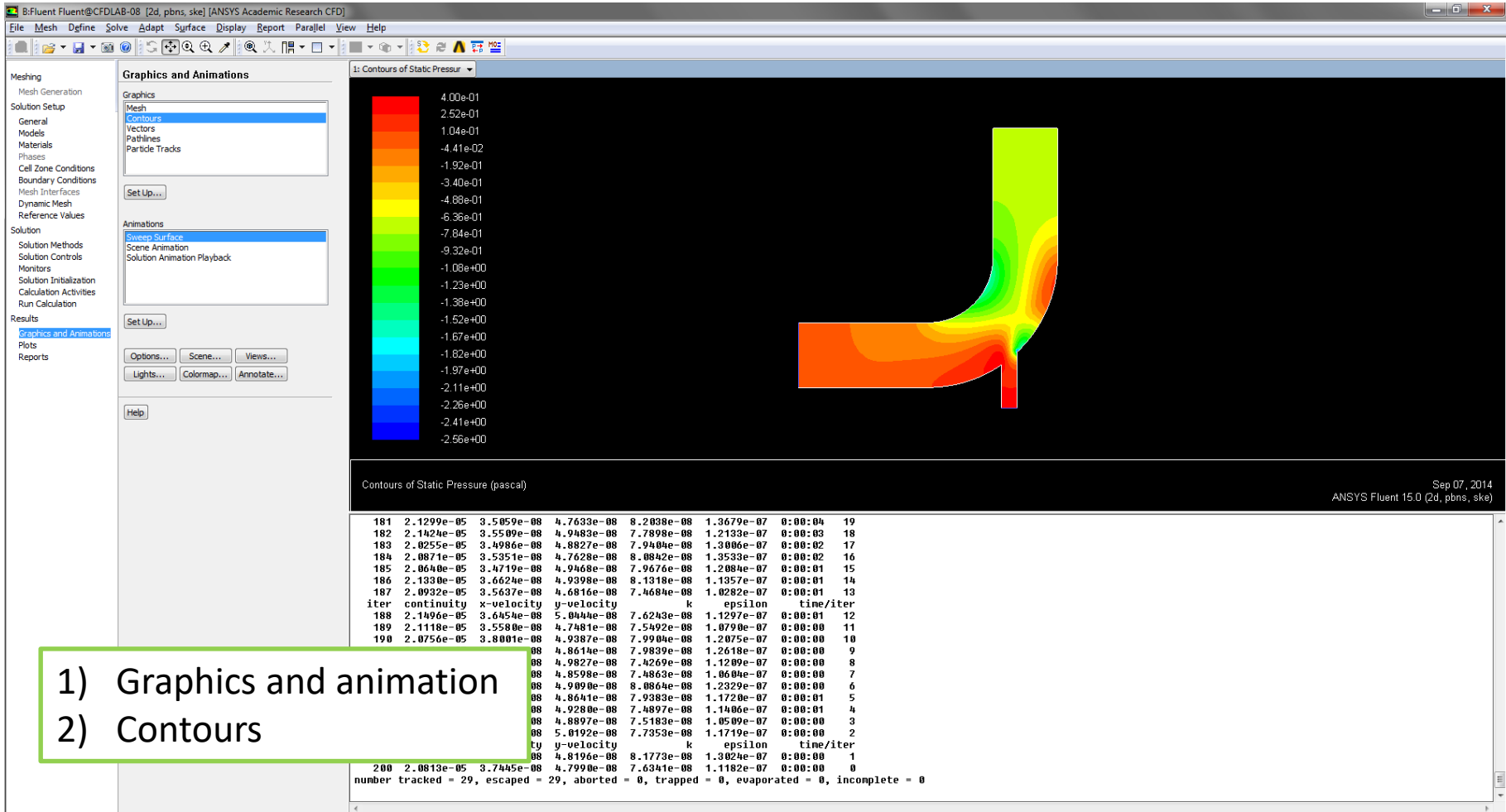
# Drawing pathlines



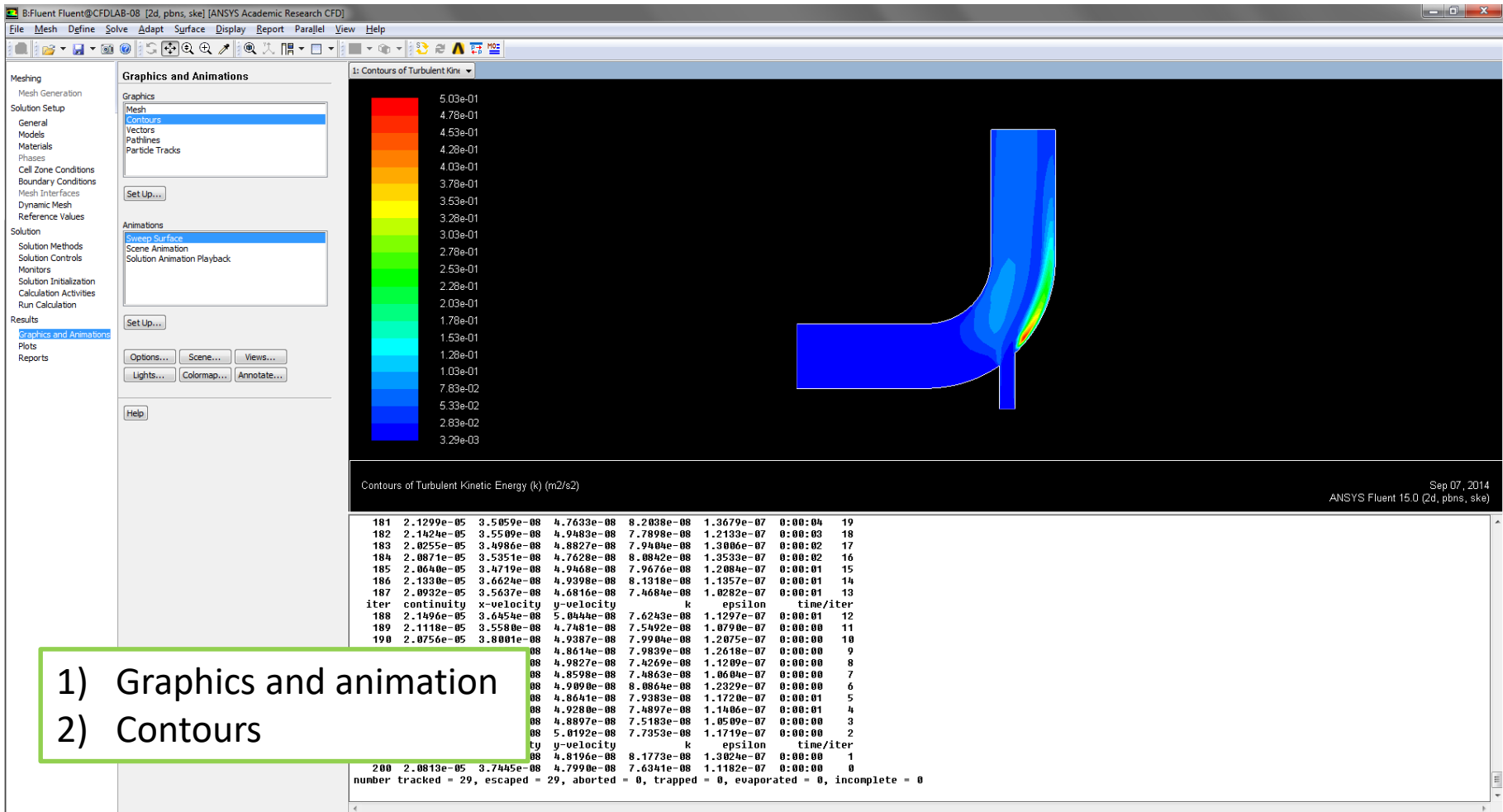
# Drawing pathlines



# Contour plots: pressure

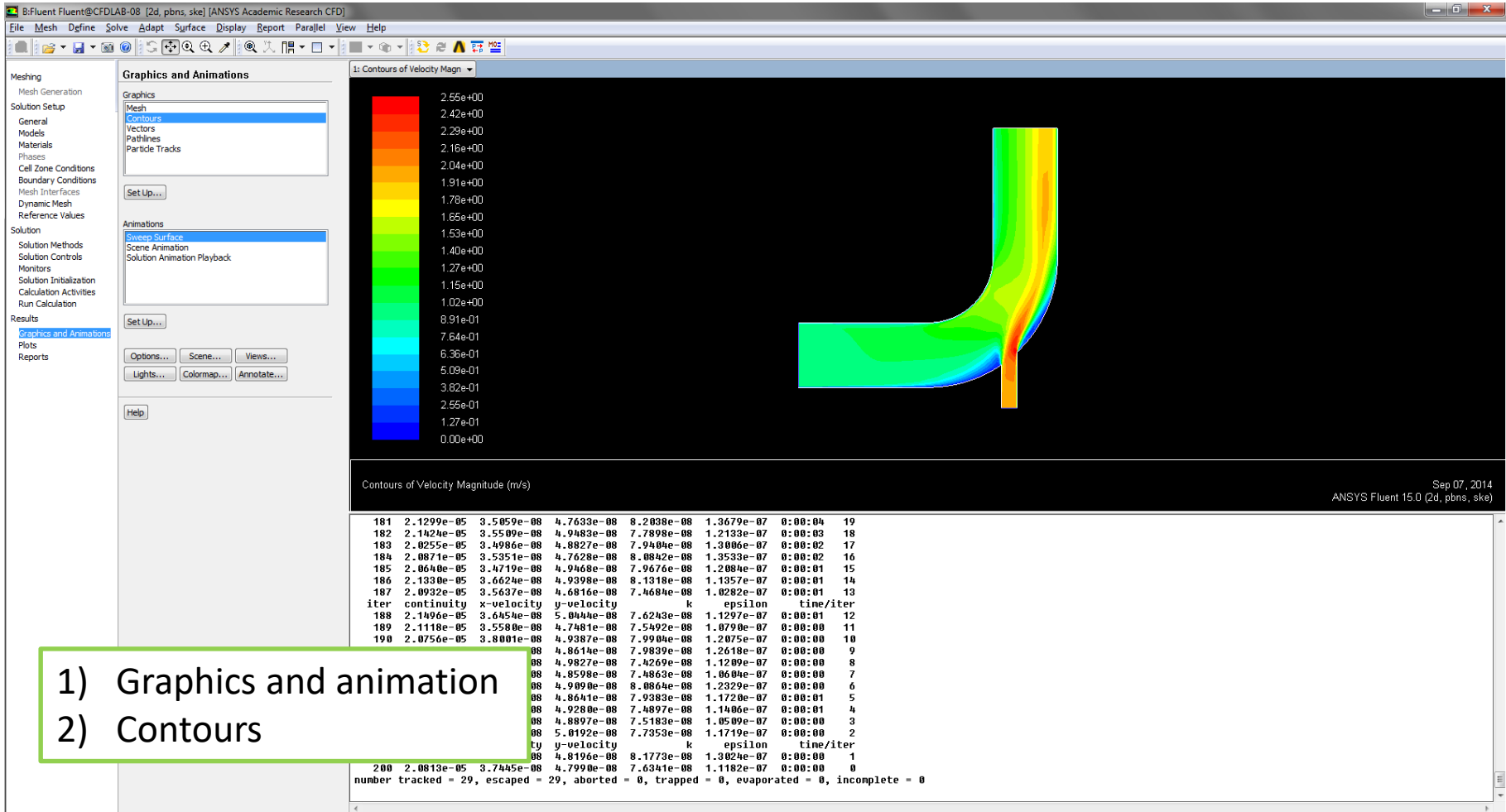


# Contour plots: k

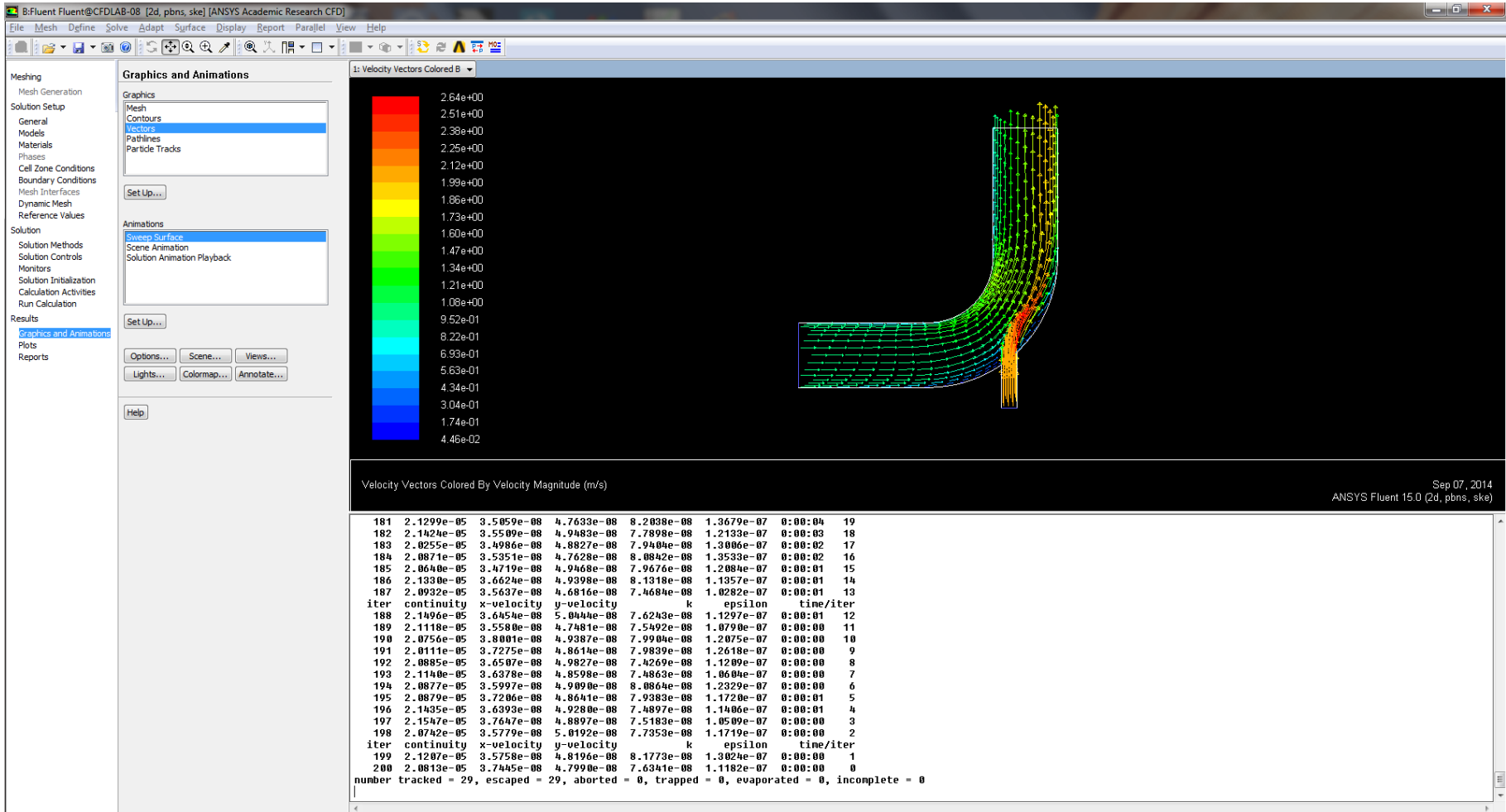




# Contour plots: velocity



# Velocity vectors





*That's all Folks!*