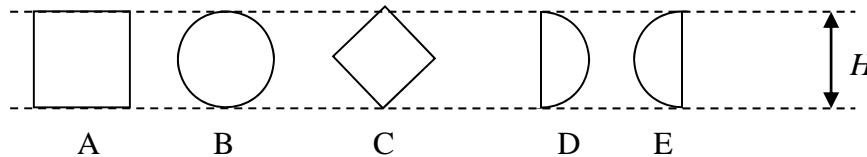


## Individual task

### Objective

To derive the drag coefficient for a 2D object, defined as  $C_D = \frac{2D}{\rho U^2 H}$  where  $D$  (N/m) is the aerodynamic drag force (per unit length in the third direction) acting on the object.

The object could be one of the following (or an object of your choice).



### Settings

- Choose height,  $H$ , free stream velocity,  $U$ , and fluid (dynamic viscosity  $\mu$  and density  $\rho$ ) so that:
- Reynolds number  $Re = \frac{\rho U H}{\mu}$  (choose  $10^4$ ,  $10^5$  or  $10^6$ )
- Mach number  $Ma = \frac{U}{a} < 0.1$  (where  $a$  is speed of sound)
- Observe, the size of the object  $H$  is the projected size in the streamwise direction (see figure).

### What to do (read carefully)

- Preparation for lecture 2 (27/3): Make a sketch of your case and the estimated flow field around it. What physical model (inviscid, laminar, turbulent, compressible) should be used? What is the Reynolds number? Make a sketch of the computational mesh.
- During (and after) lecture 2 (27/3): Determine the grid resolution requirements. What is a suitable near-wall grid size (thickness of the first grid cell) in the wall normal direction considering  $y^+$  requirements for a log-law boundary conditions ( $y^+ = 30-100$ )?
- Before 26/4 23:59:
  - Run the case using Fluent for obtaining the drag coefficient following the step-by-step instruction. If you like, you can do this as an exercise during the Fluent tutorial 20-21 April, but even better if you start doing this before the Fluent tutorial.
  - Create two figures, one with the grid and one with the velocity field, and give the drag coefficient. Upload the results on bilda before 23:59 Sunday 26 April. Post your results in the Discussion Forum “Individual Task” (see the example by me).
- Feedback: Lecture 6, 28/4

### Step-by-step instruction

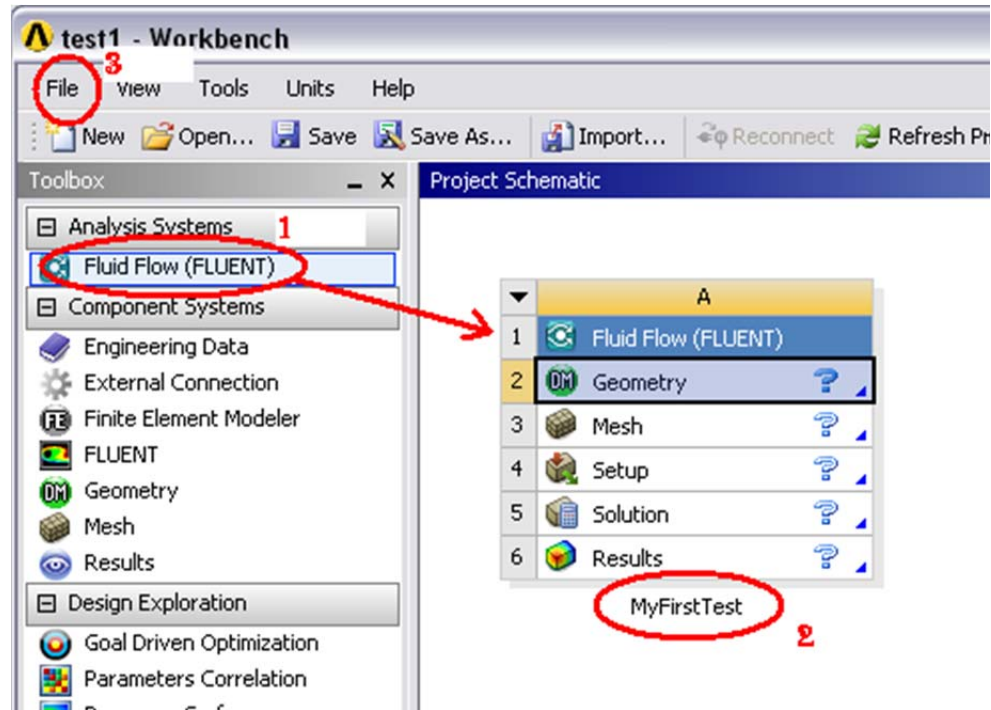
Follow the detailed step-by-step instruction at the course web:

<https://www.kth.se/social/page/course-documents>: Individual Task

## How to setup this case in Fluent – Workbench

Start ANSYS Workbench: All Programs -> Ansys 15.0 -> Workbench

(All figures here are from Ansys 13.0 and there are some differences compared with the current version 15.0. Try to use the most appropriate when differences).



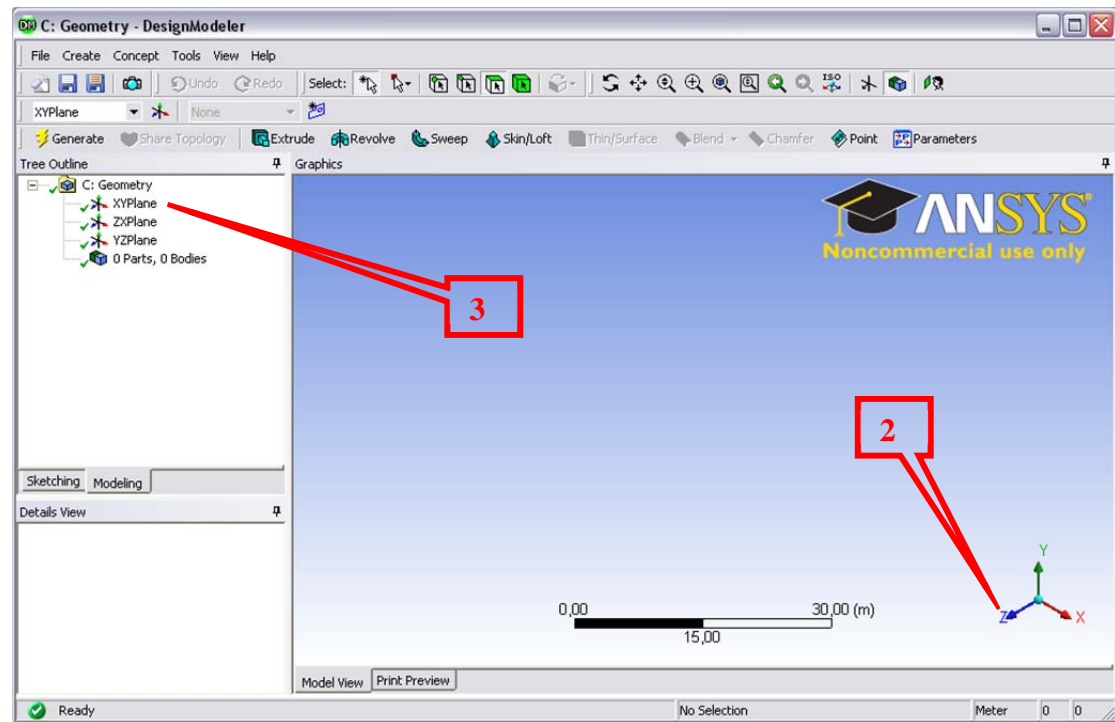
### Setup a new computation

1. Drag and drop “Fluid Flow (FLUENT)” from Toolbox to Project Schematic
2. Set a name in the box
3. Save the project: File -> Save As...

Documentation: Help -> ANSYS Workbench Help -> Workbench -> ANSYS Workbench User's Guide -> Getting Started in ANSYS Workbench -> The ANSYS Workbench Interface

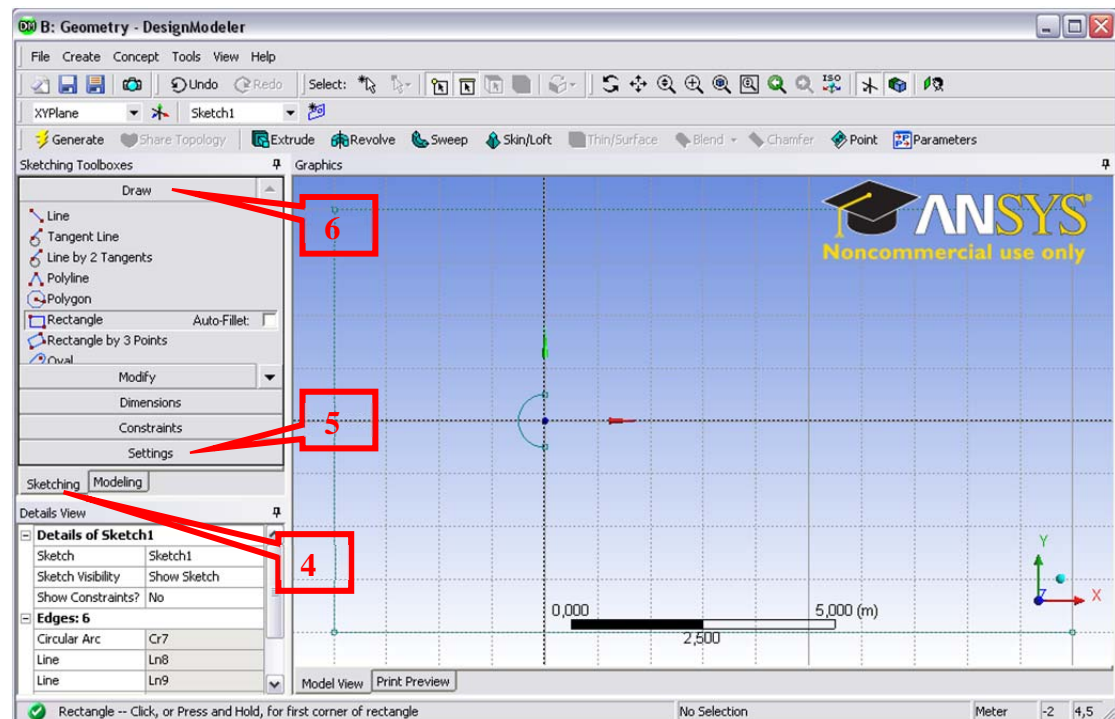
## How to create the geometry – DesignModeler

Start DesignModeler: DoubleClick on “Geometry” in the Workbench Scheme



1. Select length unit: Meter
2. To see a x-y view: Click on the “z” coordinate.
3. To work in the x-y plane: Click on “XYPlane”

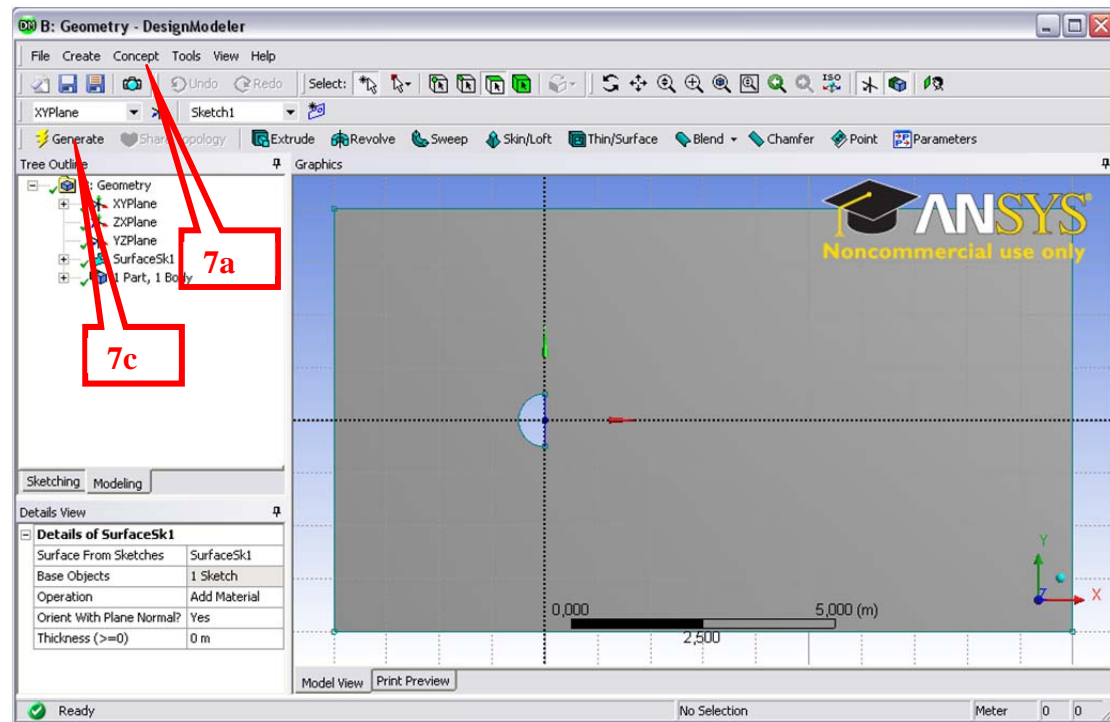
### Sketch your geometry



4. To work in the Sketching mode: Click on “Sketching”
5. Sketching Toolboxes -> Settings ->
  - a. Grid: Show and Snap on
  - b. Major Grid Spacing: 1m

- c. Minor-Step per Major: 5
- d. Snaps per Minor: 2
6. Sketching Toolboxes -> Draw ->
  - a. Arc by Center: Click on origo, first and second arc points (radius = 0.5)
  - b. Line: Click on first and second arc points.
  - c. Rectangle: Drag a rectangle  $x$  from -4 to 10 and  $y$  from -4 to 4.

Create surface (computational domain)



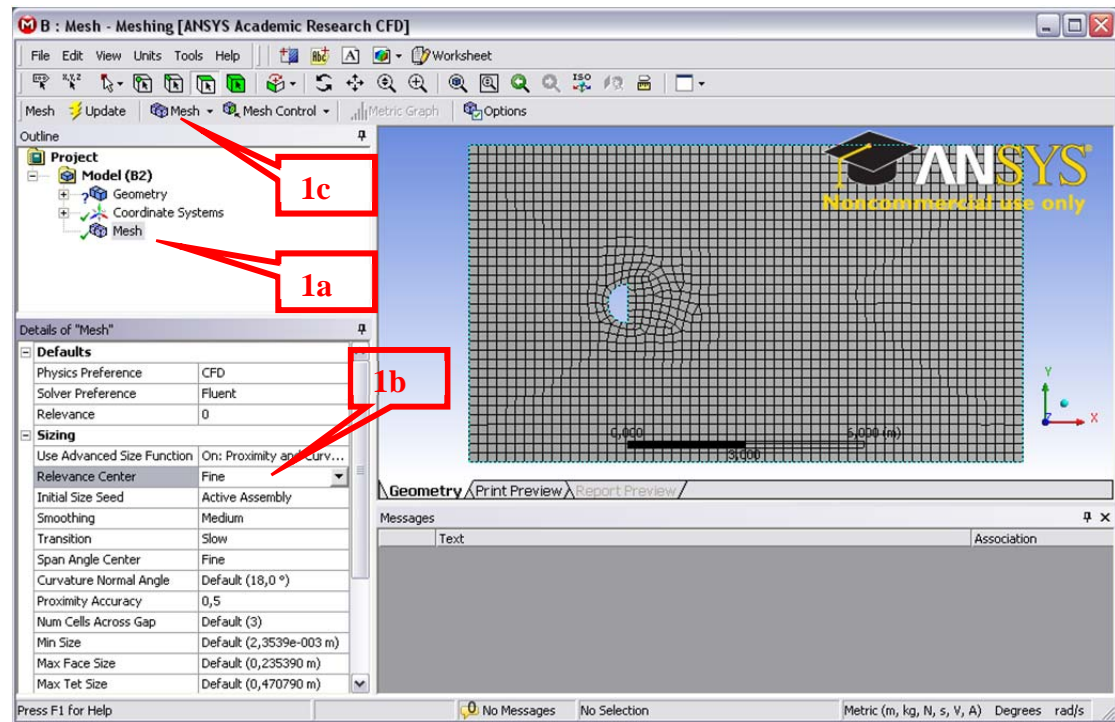
7. Concept -> Surfaces from Edges ->
  - a. Concept -> Surfaces from Sketches ->
  - b. Select all edges <CTR> Click
  - c. Click Generate
8. Close “DesignModeler”

Now, the Geometry cell in the Workbench Scheme should be checked green.

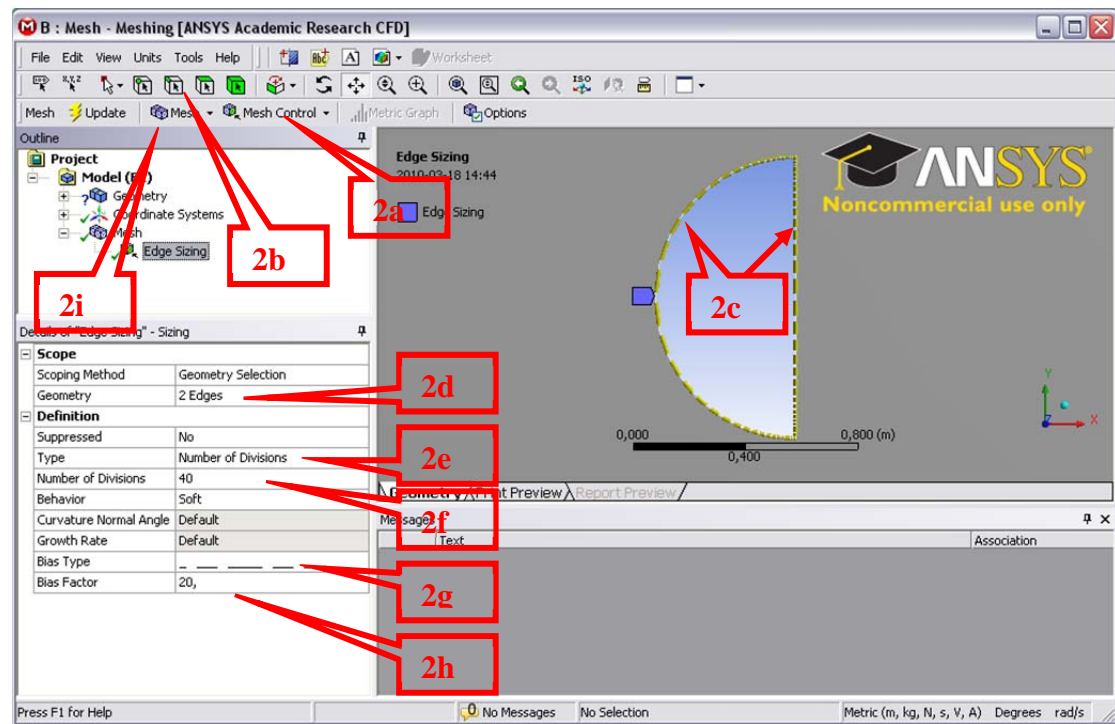
Documentation: Help -> ANSYS Workbench Help -> DesignModeler ->

## How to create the mesh –Meshing

Start Mesher: DoubleClick on “Mesh” in the Workbench Scheme

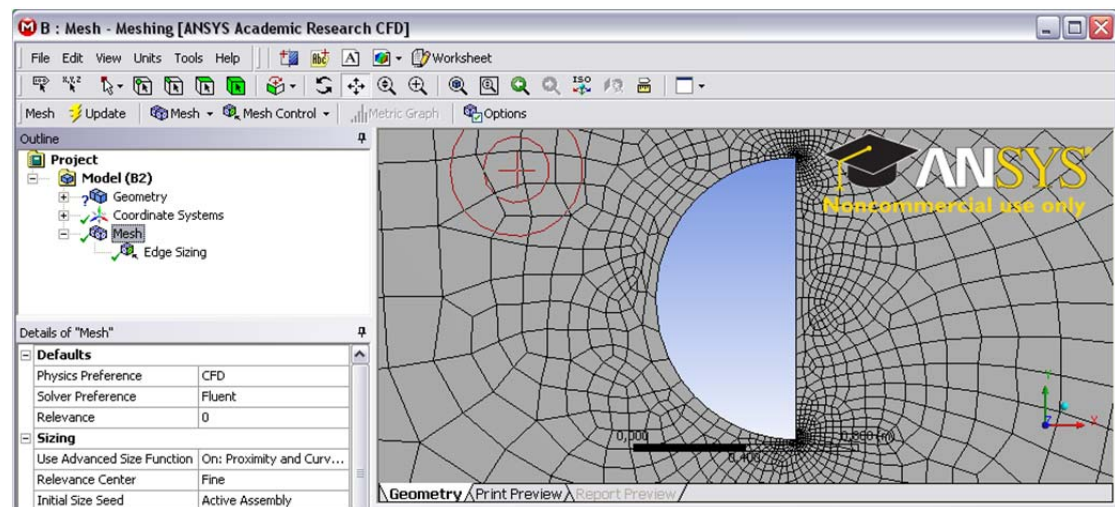


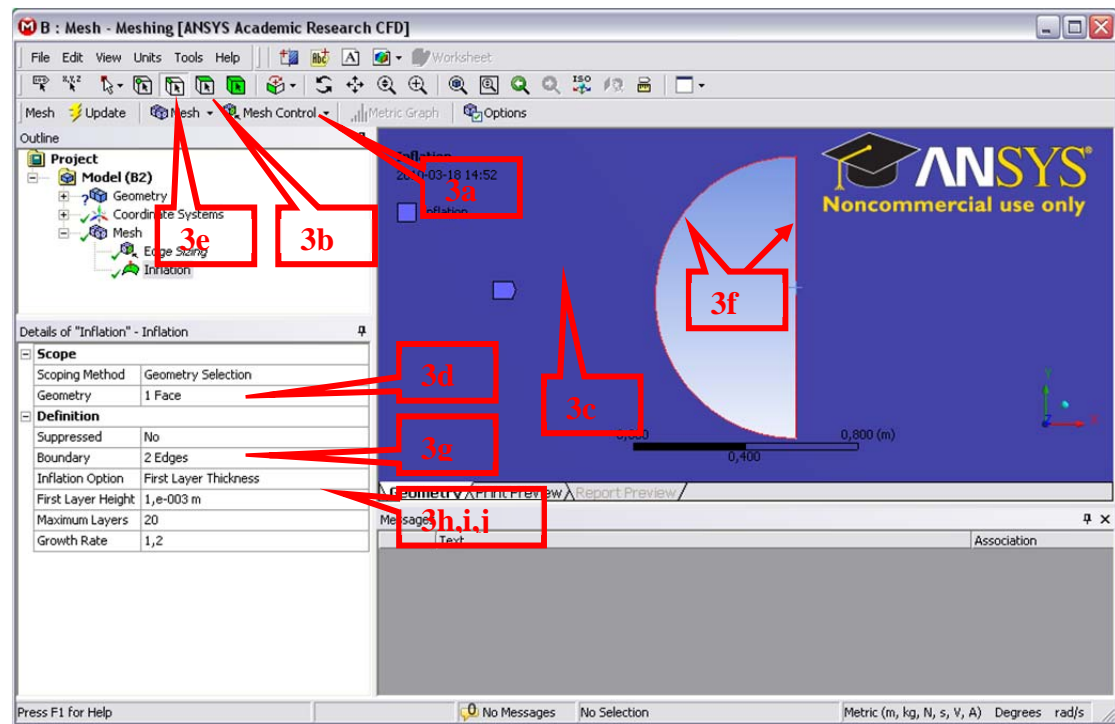
1. Make a first mesh
  - a. Click on Mesh
  - b. Set Sizing -> Relevance Center to “Fine”
  - c. Generate Mesh: Mesh -> Generate Mesh



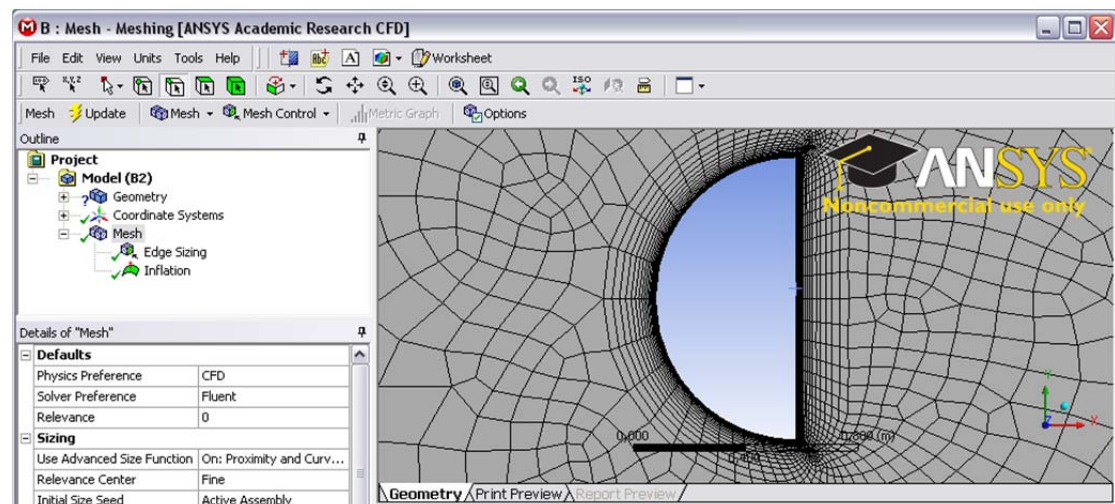
## 2. Refine mesh around half sphere

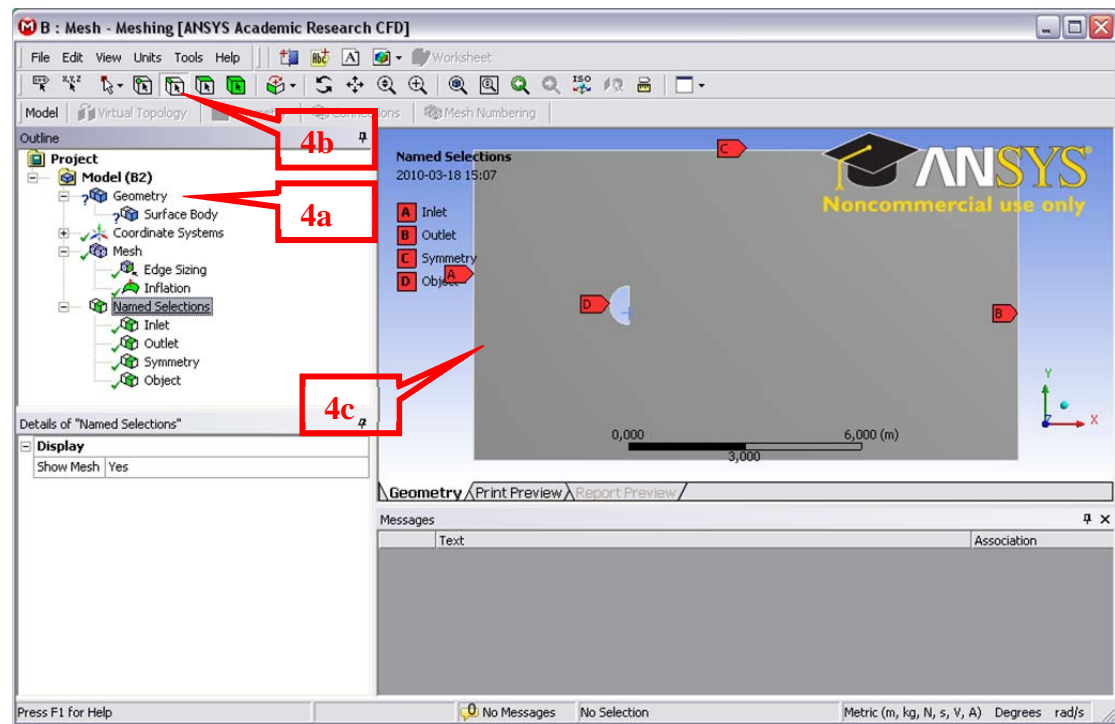
- a. Click on Mesh Control -> Sizing
- b. Click on edge symbol
- c. Select the half sphere: <CTR> Click on arc and line
- d. Click on Geometry -> Apply
- e. Set Type = Number of Divisions
- f. Set Number of Divisions = 40
- g. Set Bias Type = “.-\_.”
- h. Set Bias Factor = 10 (20 as in the figure does not work)
- i. Generate Mesh: Mesh -> Generate Mesh
- j. Click on Mesh to see it





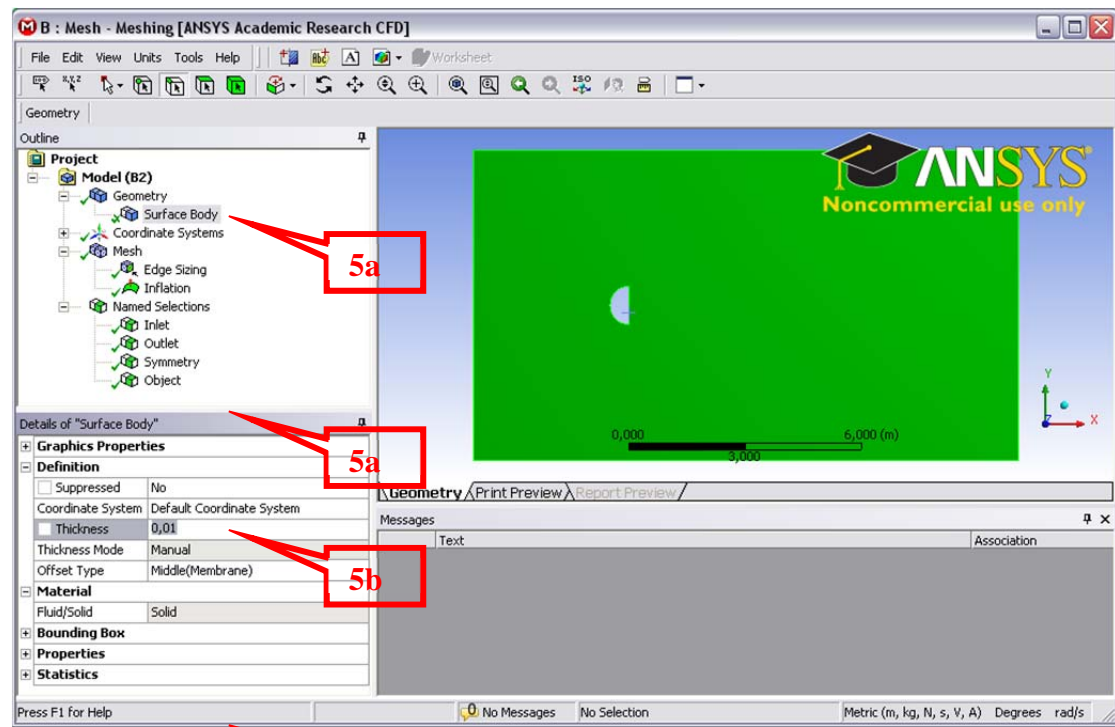
3. Make a boundary layer mesh around the half sphere
  - a. Click on Mesh Control -> Inflation
  - b. Click on face symbol
  - c. Select the computational domain
  - d. Click on Geometry -> Apply
  - e. Click on edge symbol
  - f. Select the half sphere: <CTR> Click on arc and line
  - g. Click on Boundary -> Apply
  - h. Inflation Option = First Layer Thickness
  - i. First Layer Height = 0,001 m
  - j. Maximum Layers = 20
  - k. Generate Mesh: Mesh -> Generate Mesh
  - l. Click on Mesh to see it





4. Set name on the boundaries
  - a. Click on Geometry
  - b. Click on edge symbol
  - c. Select the inlet boundary
  - d. Right-click on the selected boundary -> Create Named Selection
  - e. Give boundary name
  - f. Do this for "Inlet", "Outlet", "Upper", "Lower" and "Object". Use <CTR> click for choosing multiple boundaries. Names are arbitrary.





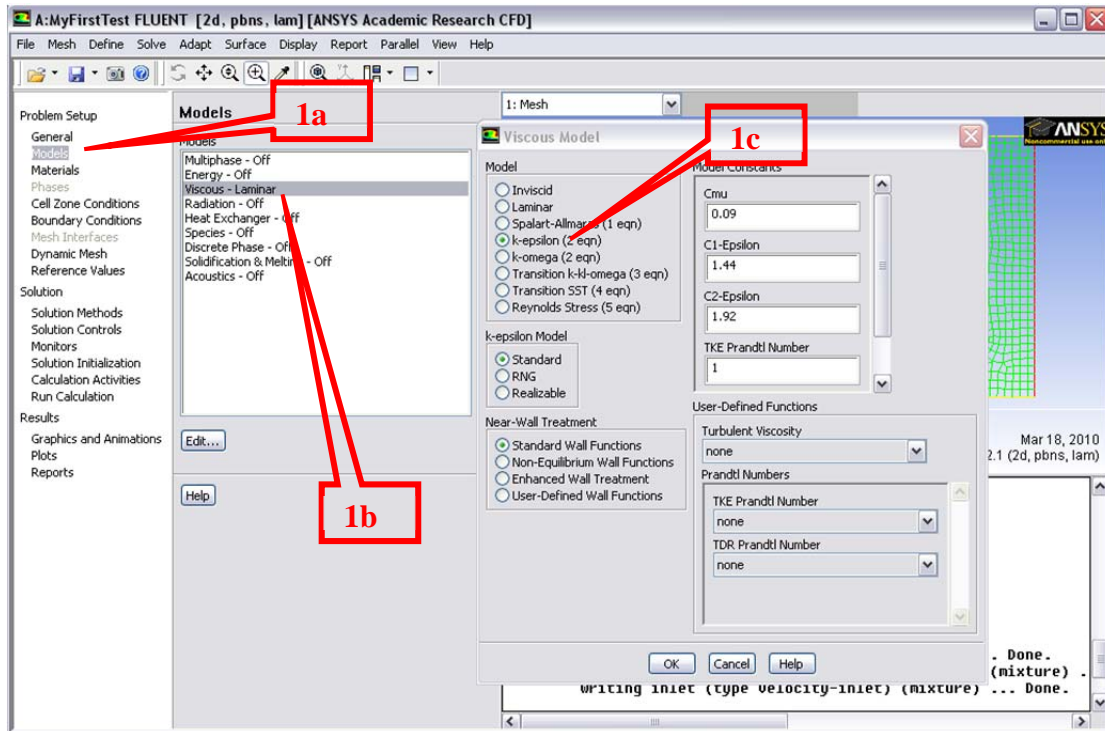
5. Set thickness. The mesh must be 3D, but could be a thin slice that will be transformed back to 2D in Fluent.
  - a. Click on Geometry -> Surface Body
  - b. Set thickness to e.g. 0,01
6. Finish.
  - a. Click on Mesh
  - b. Click on Update
  - c. Close Mesher

Now, the Mesh cell in the Workbench Scheme should be checked green.

Documentation: Help -> ANSYS Workbench Help -> Meshing ->

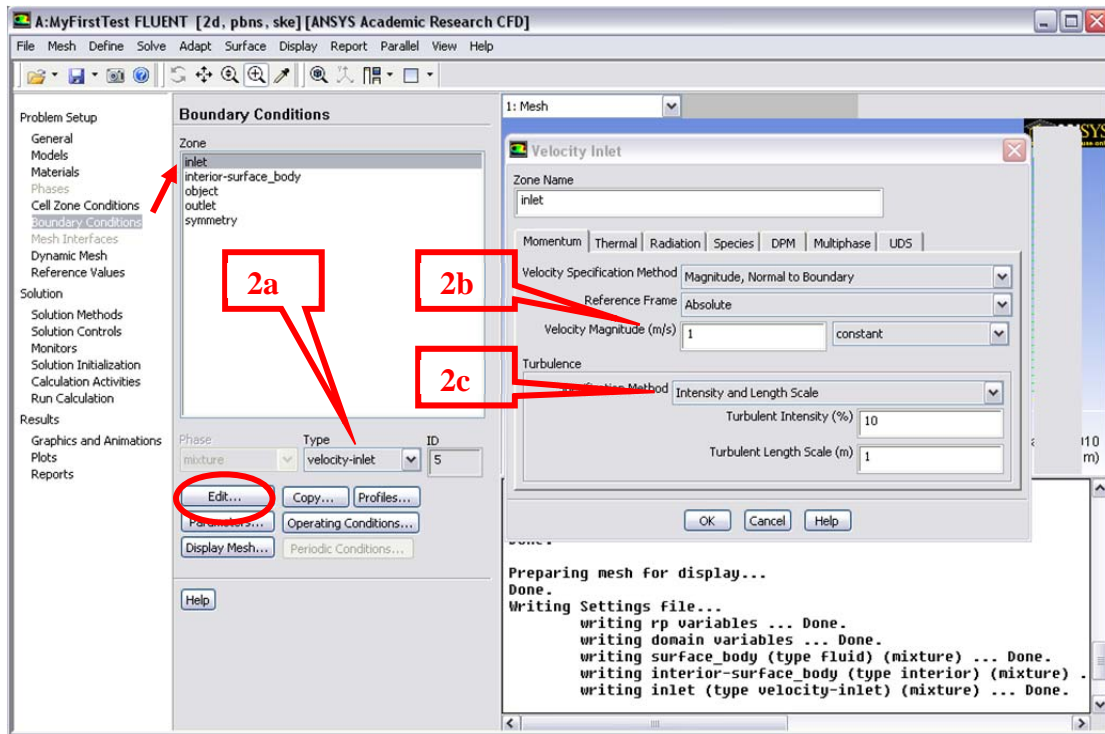
## How to compute the case using Fluent

Start Fluent: DoubleClick on “Setup” in the Workbench Scheme



### 1. Set Standard k-eps model

- a. Click on Model
- b. Click Viscous – Laminar – Edit
- c. Choose k-epsilon

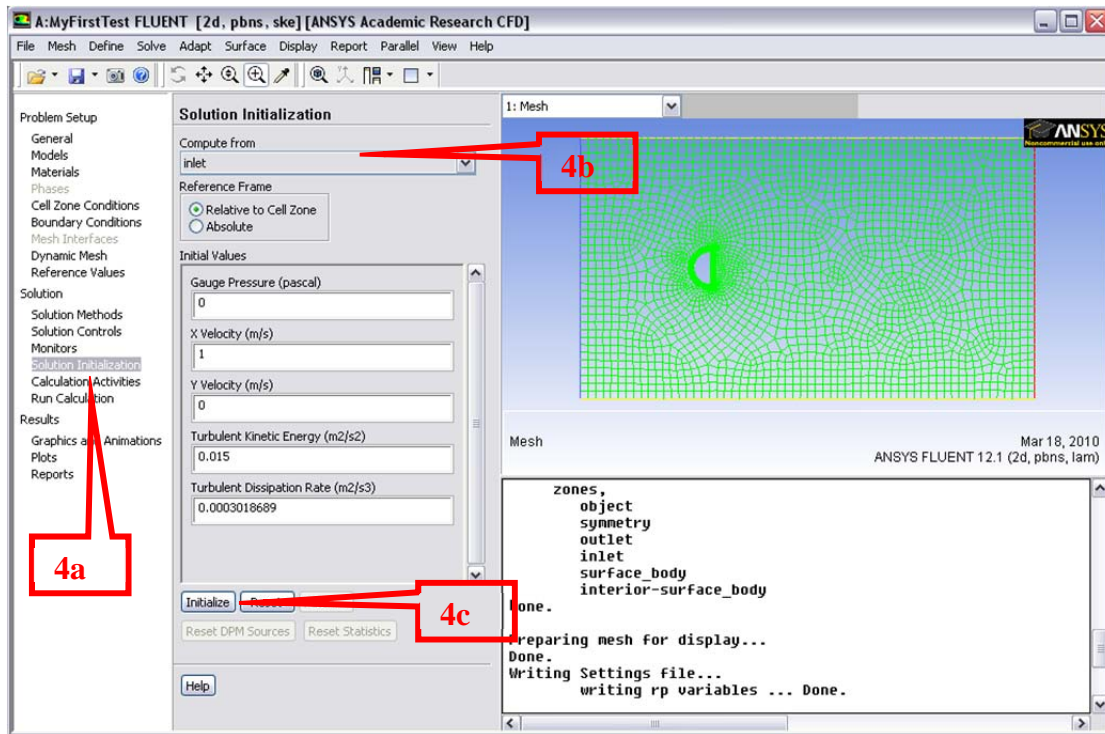


## 2. Inlet boundary conditions

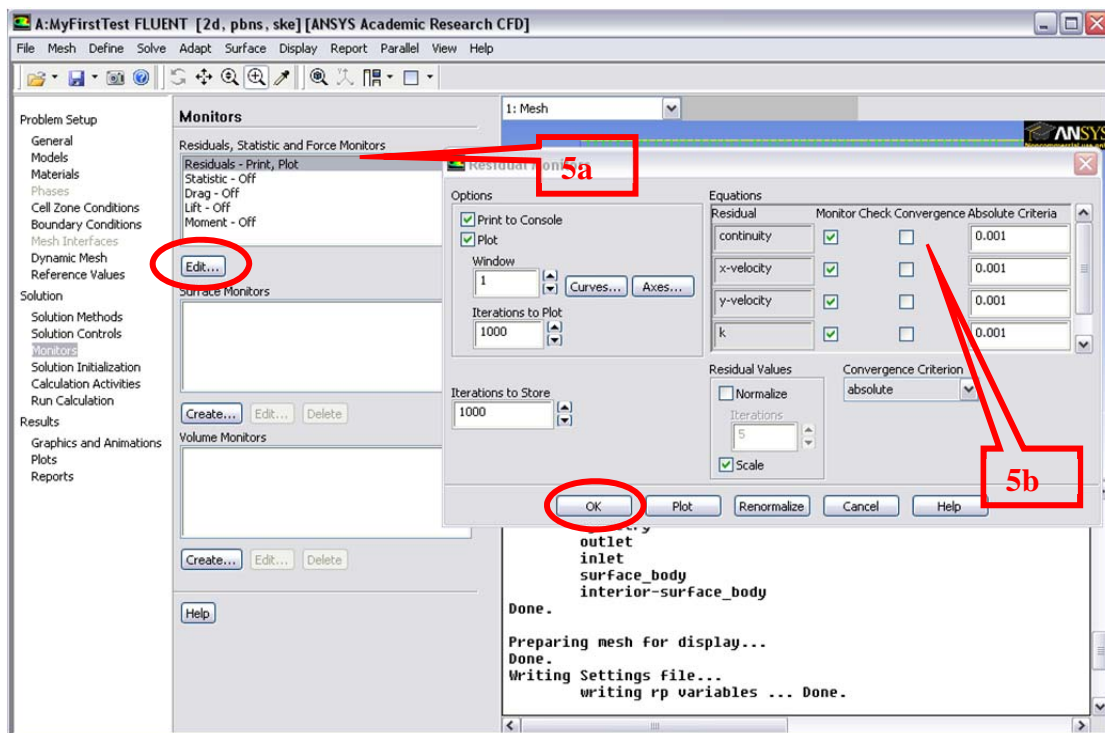
- a. Boundary Conditions -> Inlet: Type = velocity-inlet. Edit
- b. Set velocity
- c. Set turbulence intensity and length scale

## 3. Other boundary conditions

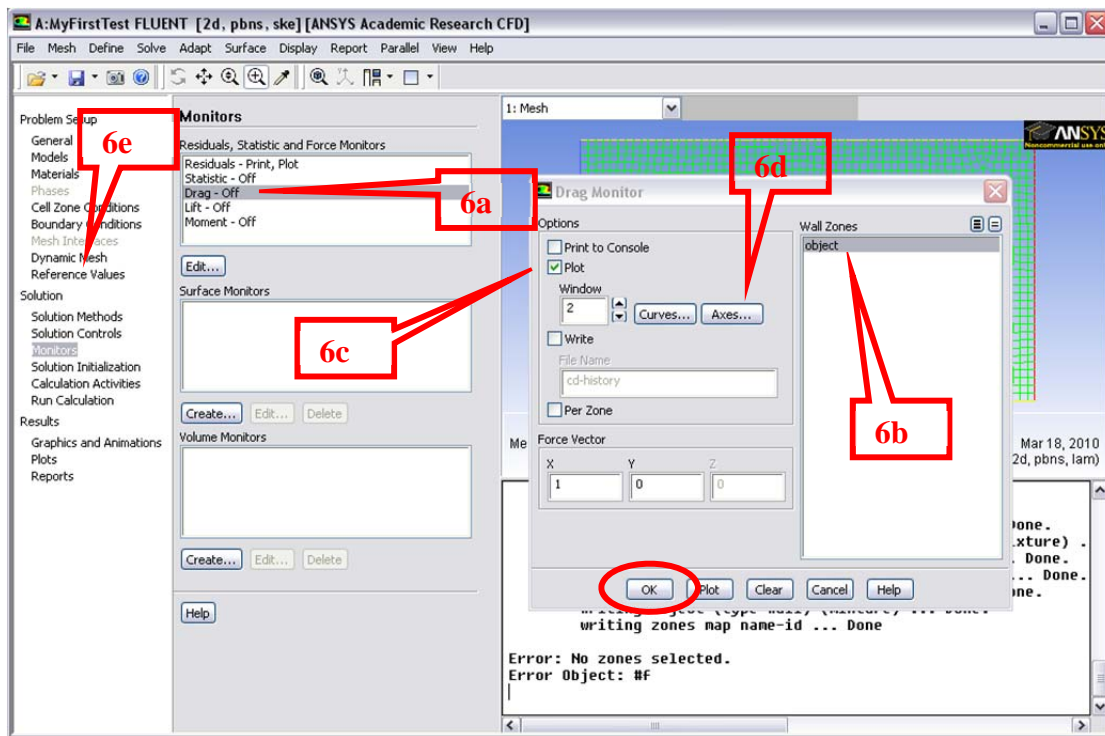
- a. Outlet: Type=pressure-outlet, Gauge Pressure=0
- b. Upper and Lower: Type=symmetry
- c. Object: Type=wall



4. Initialize solution to the value at the inlet boundary
  - a. Solution Initialization: Use “Standard Initialization”
  - b. Compute from: inlet
  - c. Press Initialize

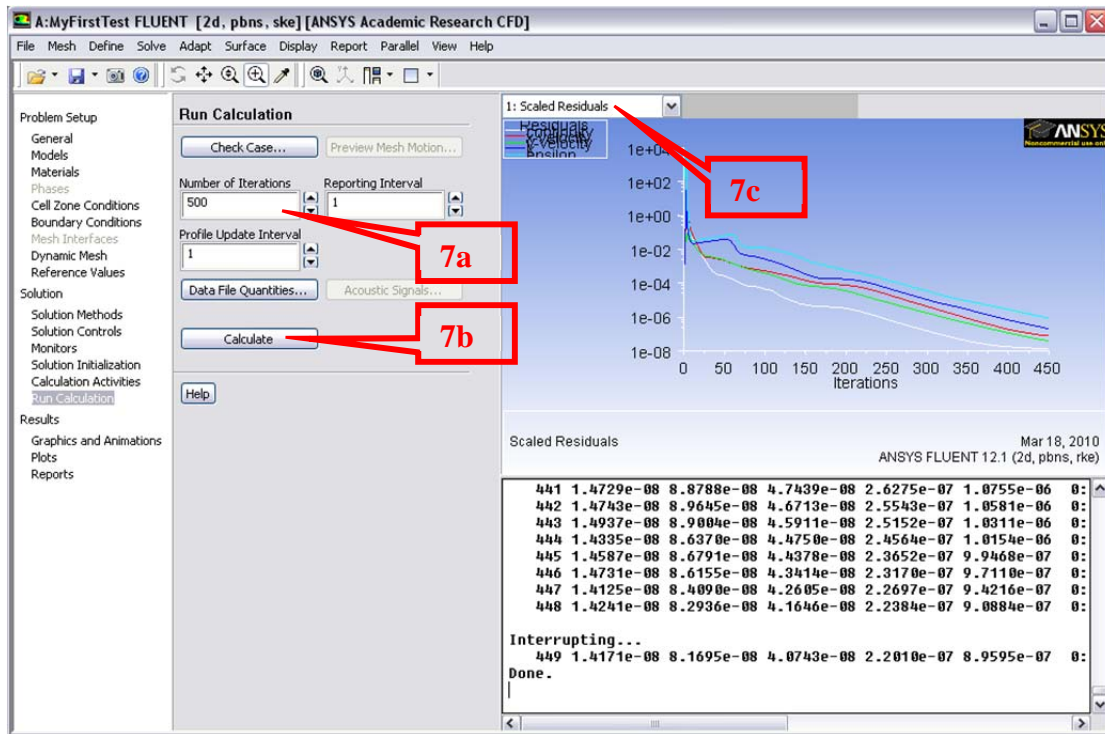


5. Monitor convergence
  - a. Monitors -> Residuals -> Edit
  - b. Uncheck “Check convergence” for all equations (also epsilon, you need to scroll down).



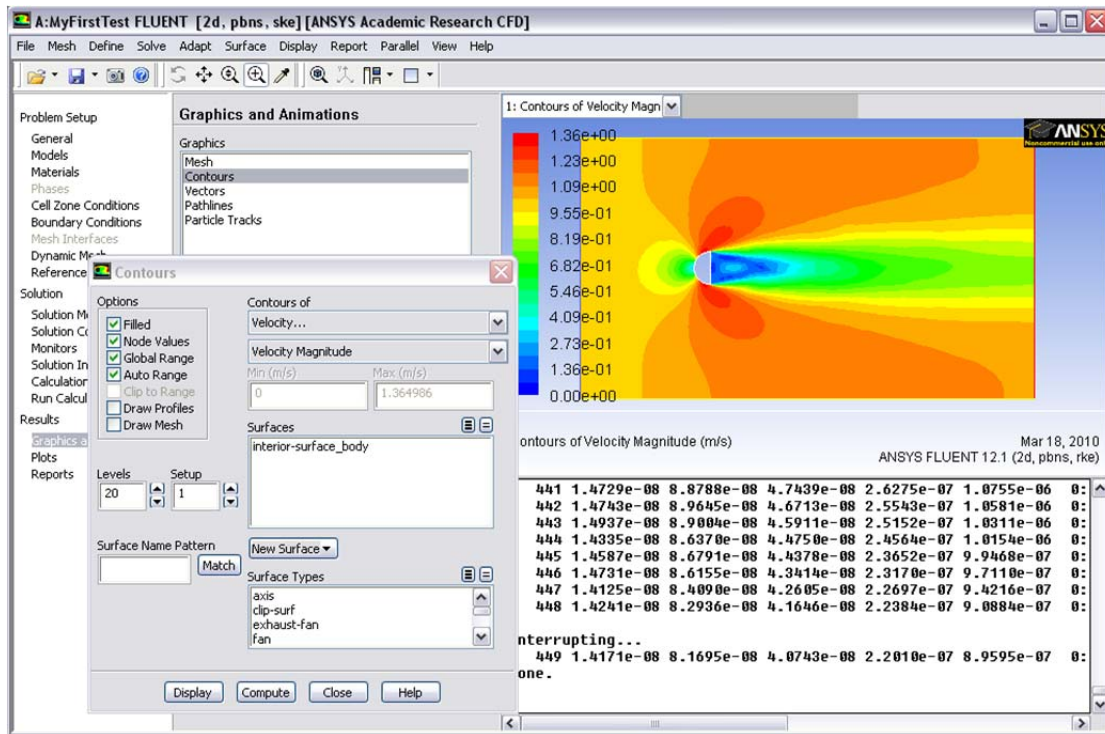
## 6. Monitor drag

- a. Monitor -> Create -> Drag
  - b. Choose object
  - c. Check "Plot"
  - d. Axes -> Y -> Uncheck Auto Range, set Range 0 to 2
  - e. Also set correct Reference Values:
    - i. "Area" is reference area for Cd in x-direction.
    - ii. For 2D it is per meter in z-direction
    - iii. Example: Height=1.0m gives Area=1.0m<sup>2</sup>
- Set first order upwind (too dissipative for giving a good solution, but might be useful for the first try since it improves the stability)
    - Solution Methods:
    - Spatial Discretization -> Momentum -> First Order Upwind



## 7. Run Calculation

- Set Number of Iterations = 500 (or 1000)
- Press “Calculate”, Iterate until Cd levels out
- Check both “Scaled Residuals” and “Drag Convergence History”



8. Plot Velocity field:

- Graphics and Animations -> Contours -> Set Up
- Plot Velocity Magnitude. Check “Filled”
- Zoom by dragging using middle mouse. Zoom out by dragging to upper left.
- Try to plot other properties (Turbulence Kinetic Energy, Velocity arrows)

9. Try to refine calculation:

- Solution -> Solution Methods -> Momentum: Second Order Upwind
- Models -> Viscous: Realizable k-epsilon
  - In particular, look at the amount of turbulence around the stagnation point and the differences between std k-eps and realizable k-eps.
- Adapt -> Gradients of Velocity
  - Press “Compute”: Gives min and max values of velocity gradient
  - Set “Refine Threshold” to something smaller than max
  - Press “Mark”: Gives no of cell marked for refinement
  - Press “Adapt”: Refines the mesh
  - Look at the mesh: Graphics and Animations -> Mesh -> Set Up

Documentation: Help -> User’s Guide Contents (Getting Started Guide, User’s Guide, et.c.)