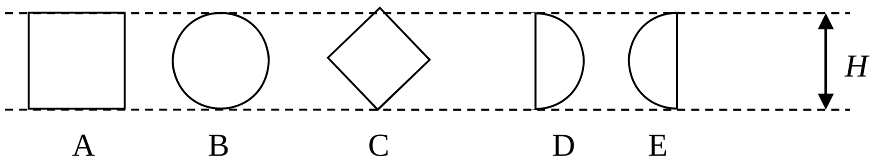
Individual task

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 μ and density ρ) 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 (28/3): 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 (28/3): 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 18/4 12:00:
    - o 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 10-11 April, but even better if you start doing this before the Fluent tutorial.
    - o Create two figures, one with the grid and one with the velocity field, and give the drag coefficient. Upload the results on bilda before 12:00 Thursday 18 April. Post your results in the Discussion Forum "Individual Task" (see the example by me).
- Feedback: Lecture 5, 23/4

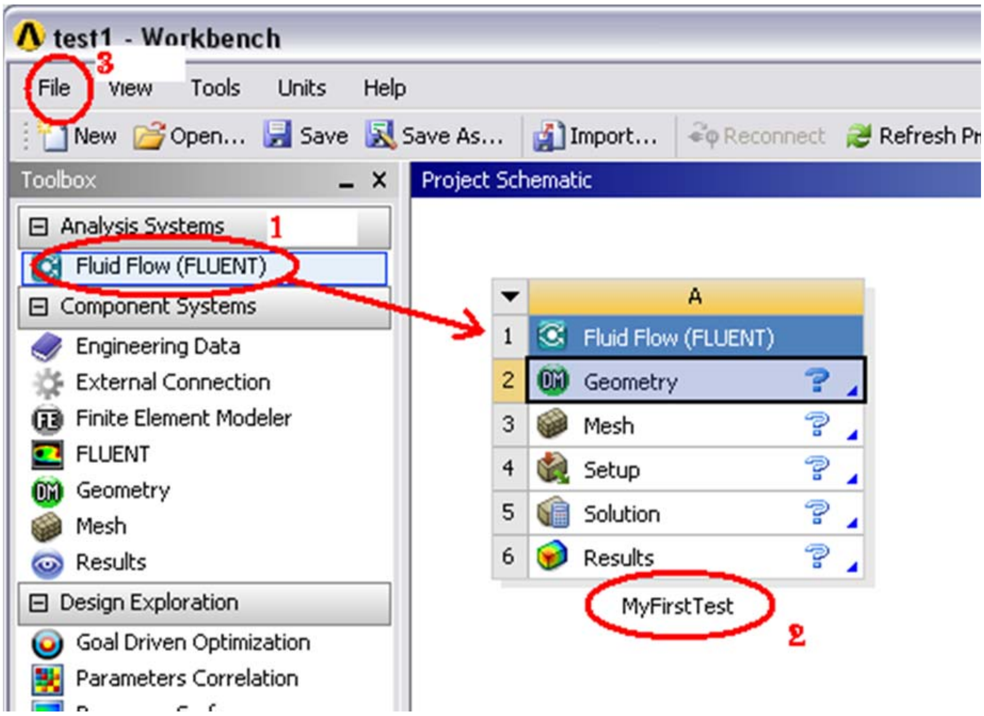
Step-by-step instruction

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 14.5 -> Workbench

(All figures here are from Ansys 13.0 and there are some differences compared with the current version 14.5. If there are differences, the text should be followed).



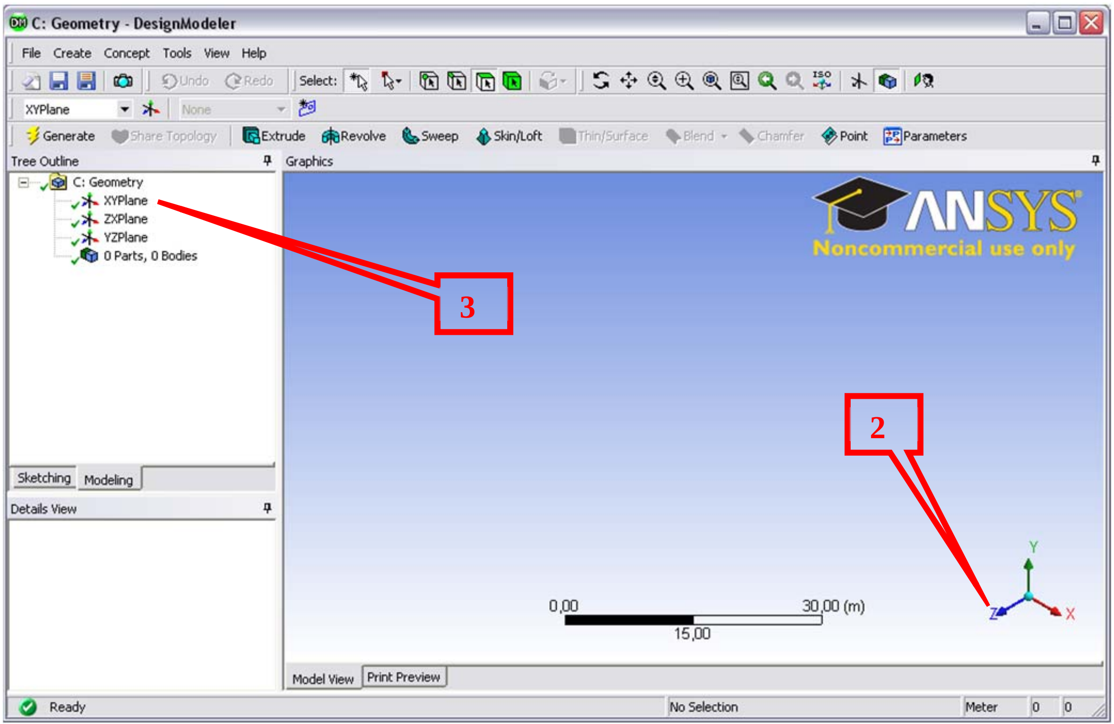
<u>Setup a new computation</u>

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

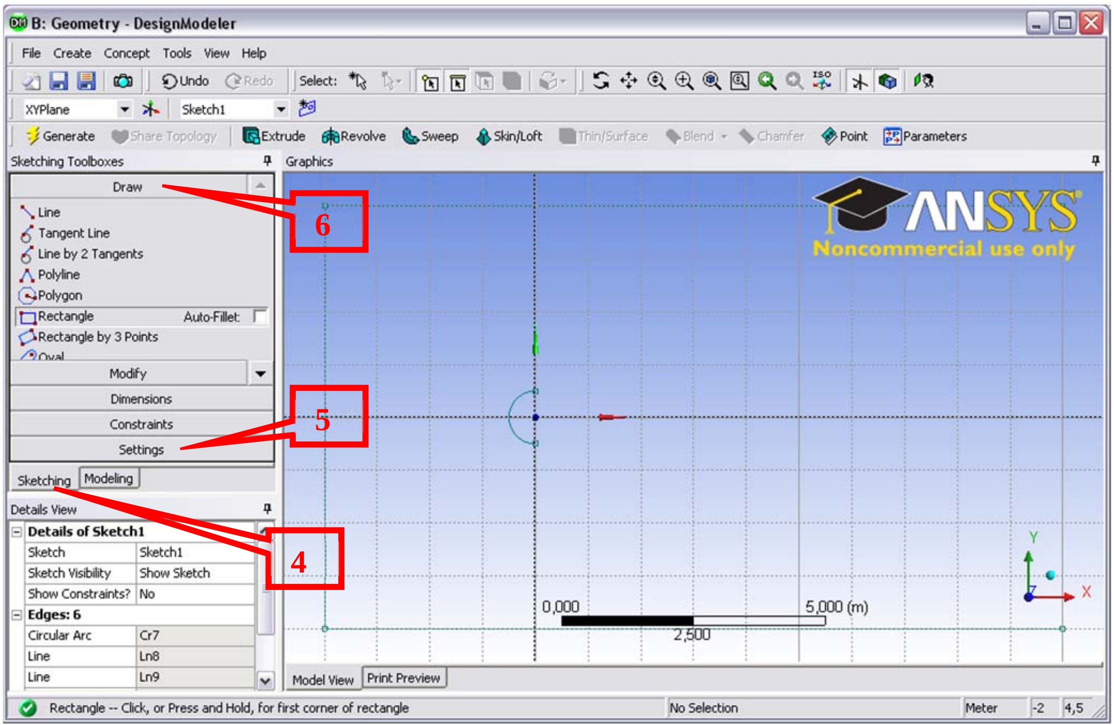
How to create the geometry – DesignModeler

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”

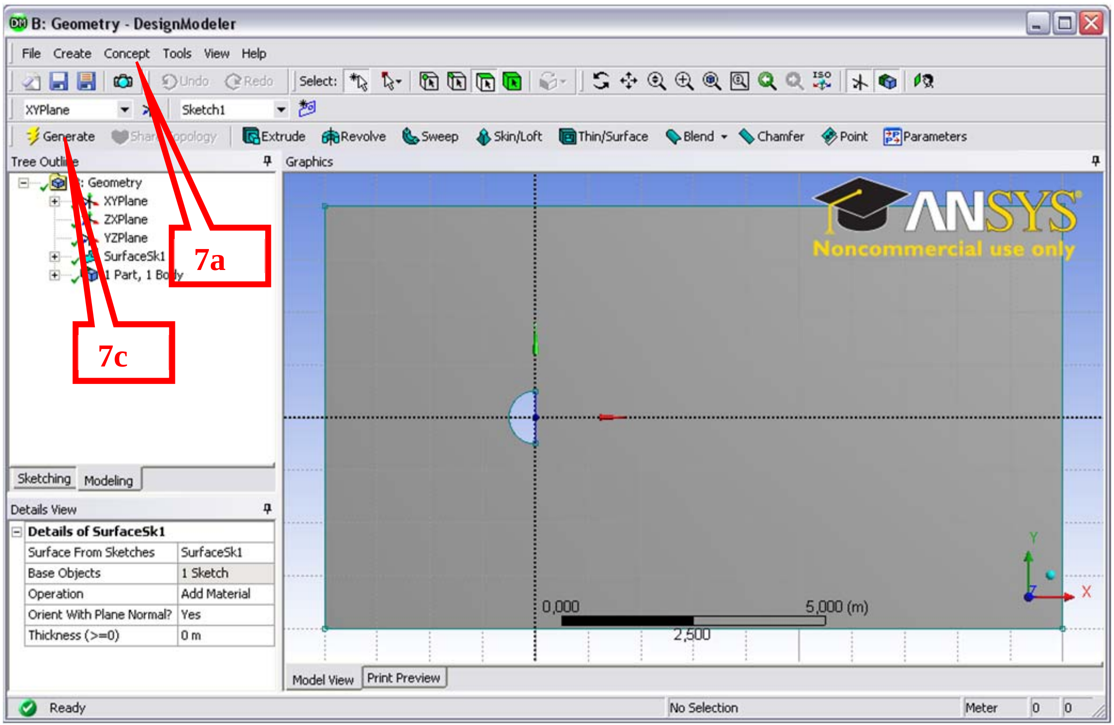
Sketch your geometry



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.

<u>Create surface (computational domain)</u>



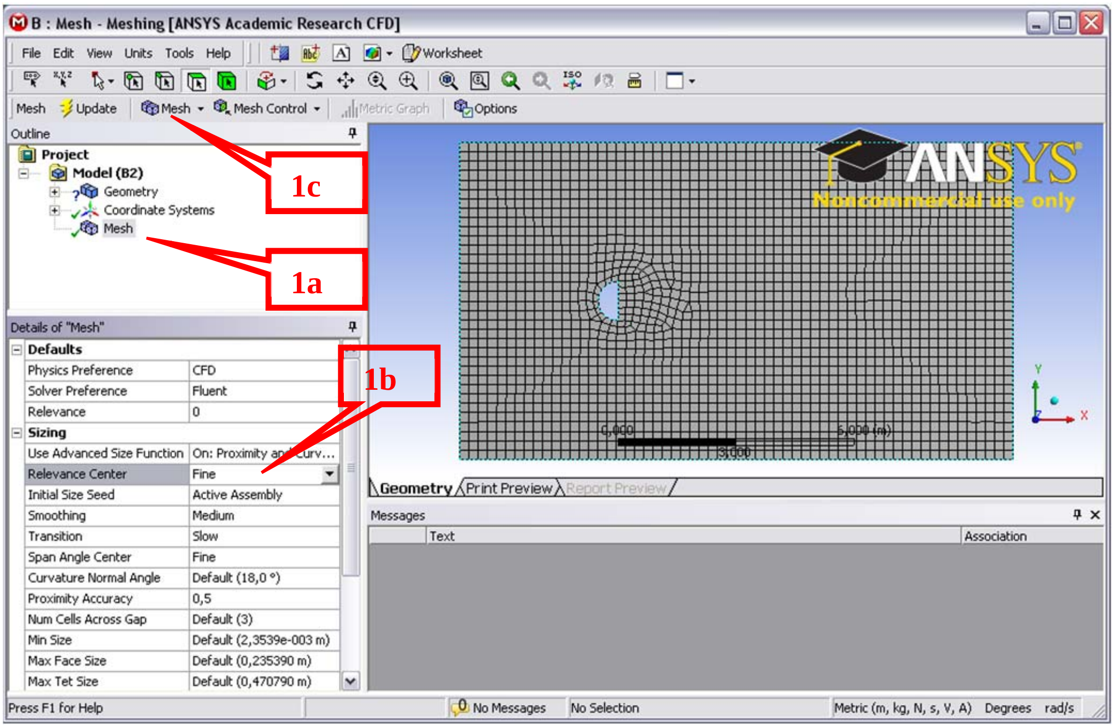
7. Concept -> Surfaces from Edges ->
   a. Concept -> Surfaces from Sketches ->
   b. Select all edges <CTR> Click
   c. Click Generate
8. Close “DesignModeler”

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

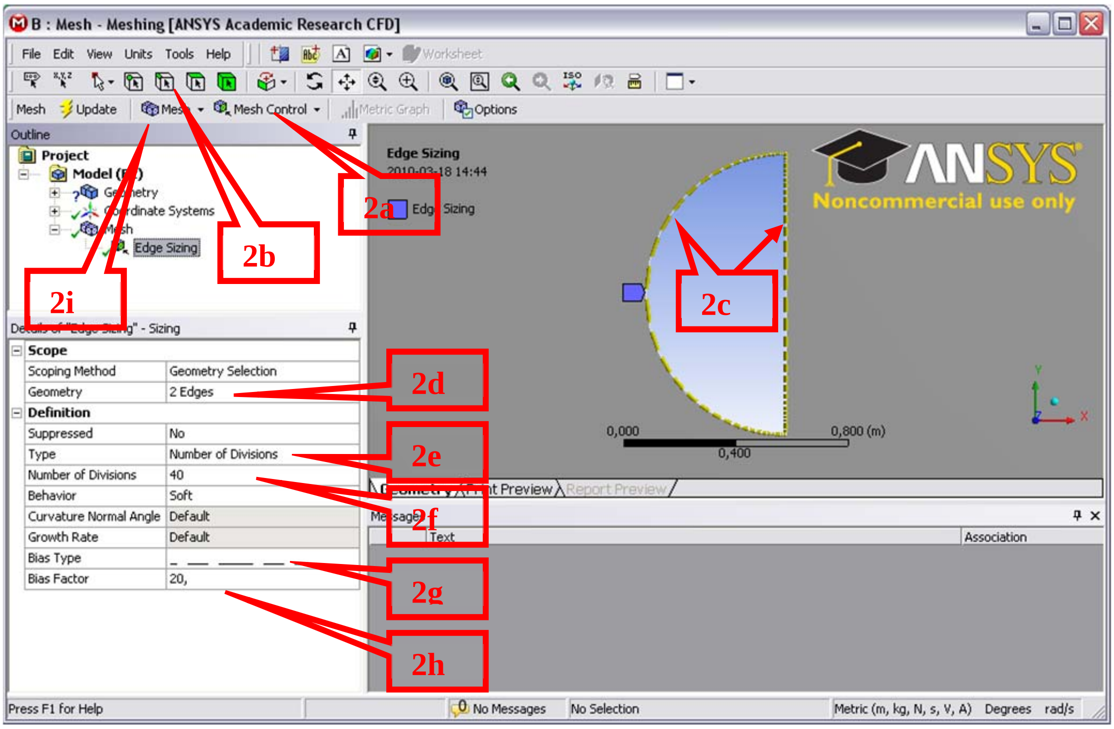
Documentation: Help -> ANSYS Workbench Help -> DesignModeler ->

How to create the mesh –Meshing

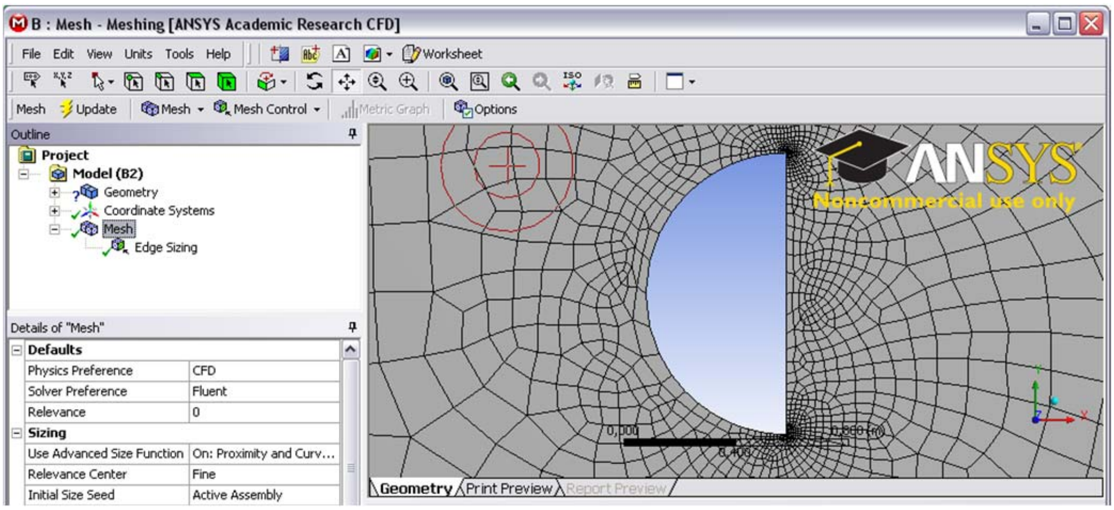
Start Mesher: DoubleClick on “Mesh” in the Workbench Scheme

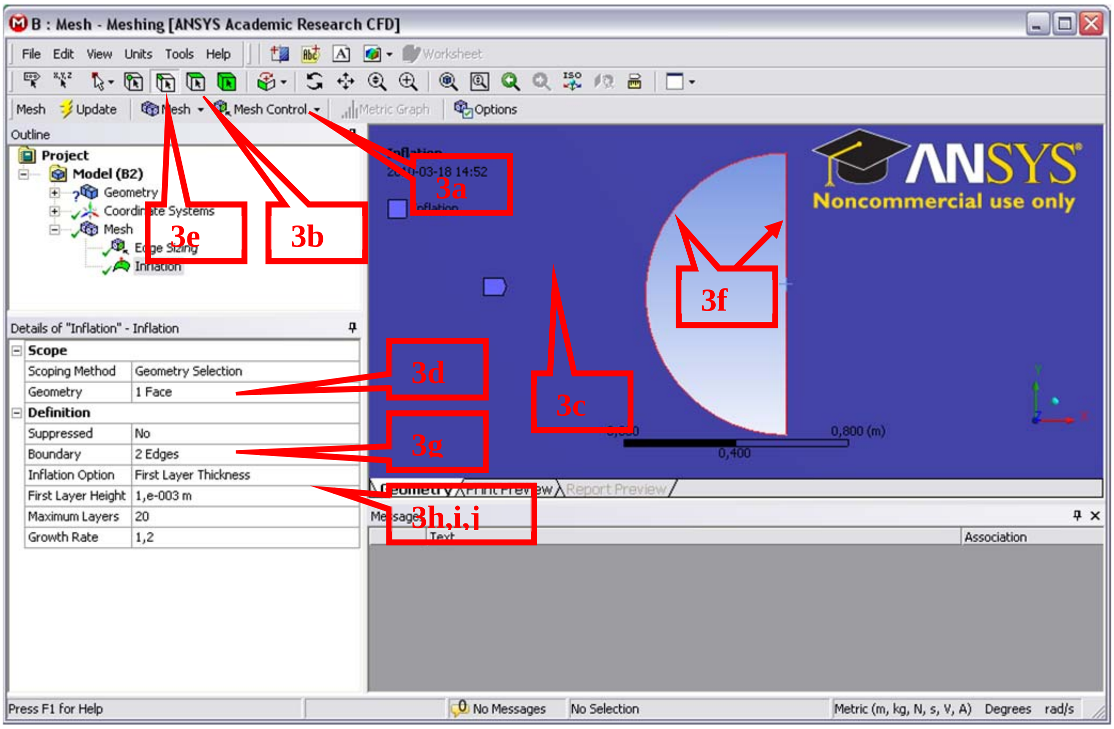


1. Make a first mesh
    a. Click on Mesh
    b. Set Sizing -> Relevance Center to “Fine”
    c. Generate Mesh: Mesh -> Generate Mesh

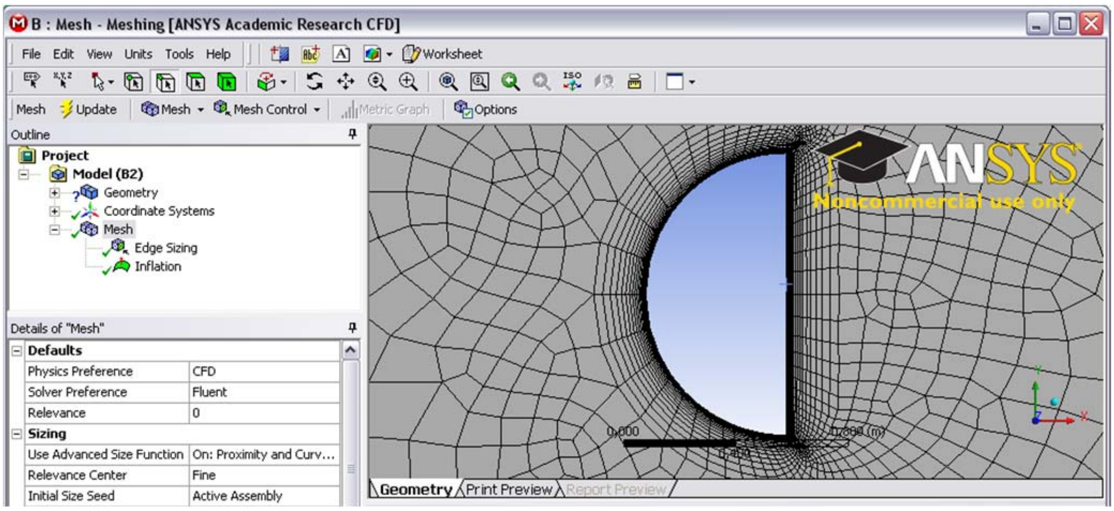


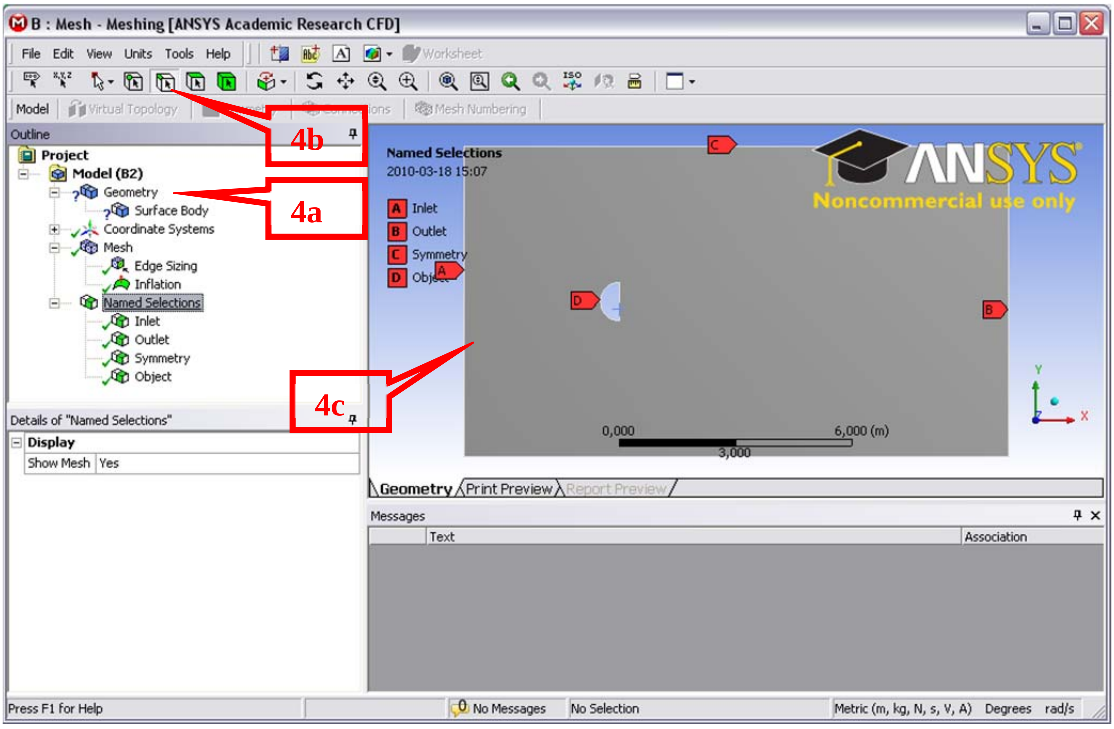
2. Refine mesh around half sphere
    a. Click on Mesh Control -> Sizing
    b. Click on edge symbol
    c. Select the half sphere: <CTR> Click on arc and line
    d. Click on Geometry -> Apply
    e. Set Type = Number of Divisions
    f. Set Number of Divisions = 40
    g. Set Bias Type = “.-_-.”
    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



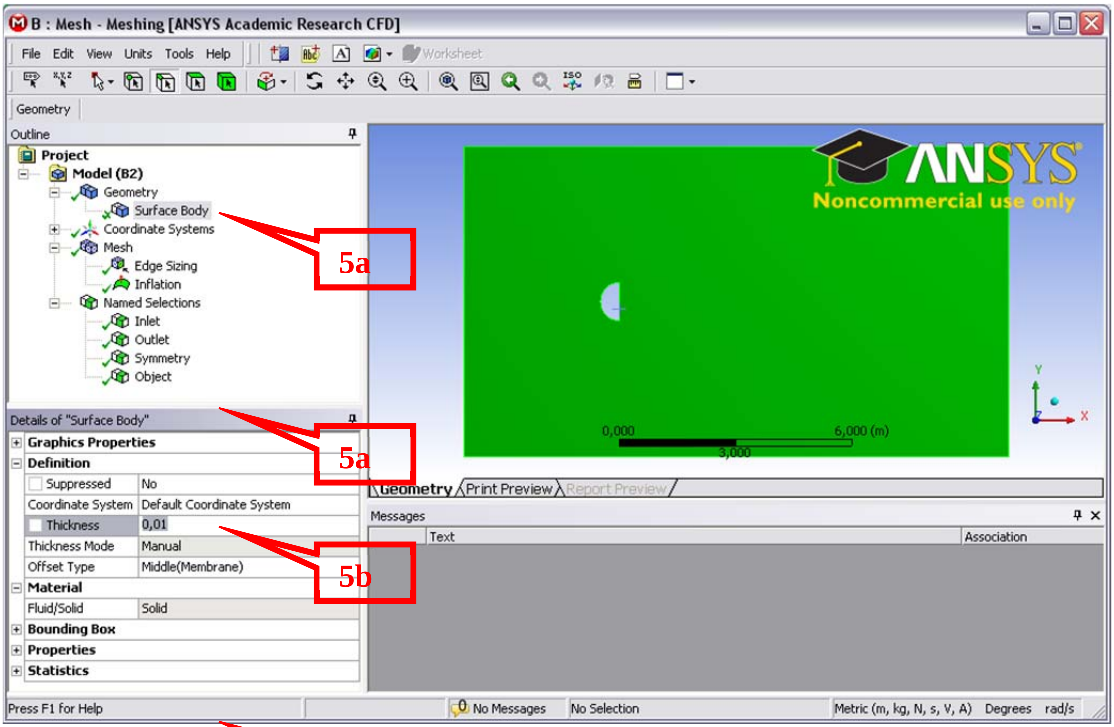


3. Make a boundary layer mesh around the half sphere
    a. Click on Mesh Control -> Inflation
    b. Click on face symbol
    c. Select the computational domain
    d. Click on Geometry -> Apply
    e. Click on edge symbol
    f. Select the half sphere: <CTR> Click on arc and line
    g. Click on Boundary -> 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.



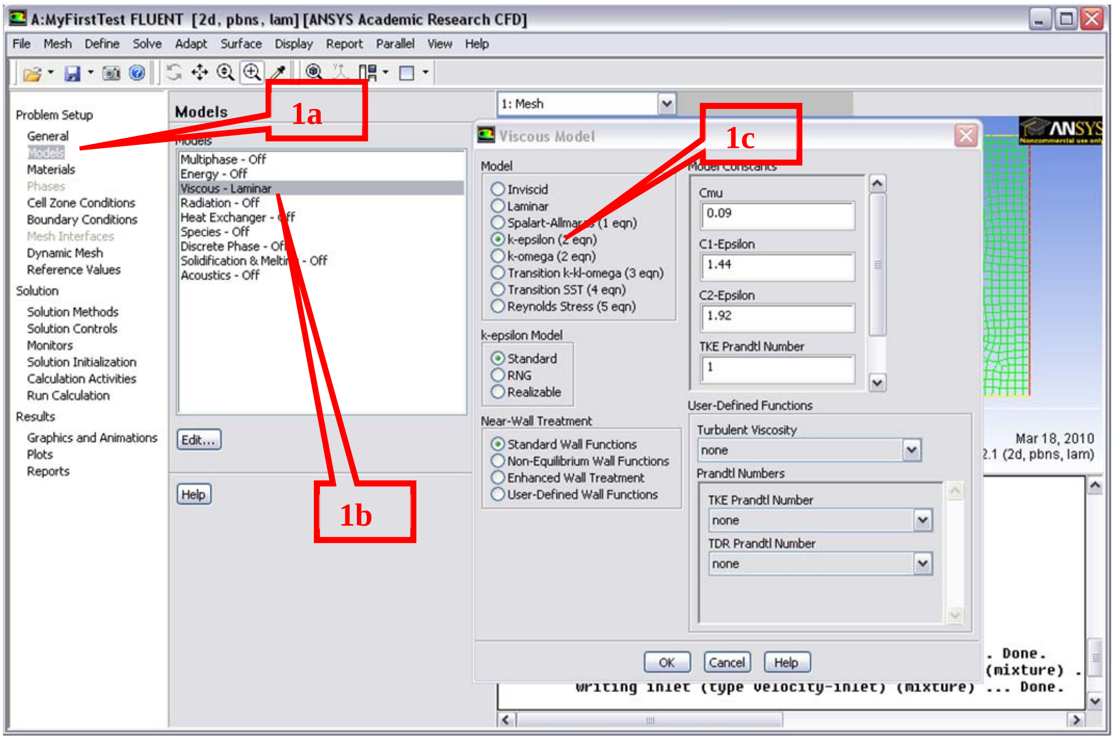
5. Set thickness. The mesh must be 3D, but could be a thin slice that will be transformed back to 2D in Fluent.
    a. Click on Geometry -> Surface Body
    b. Set thickness to e.g. 0,01
6. Finish.
    a. Click on Mesh
    b. Click on Update
    c. Close Mesher

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

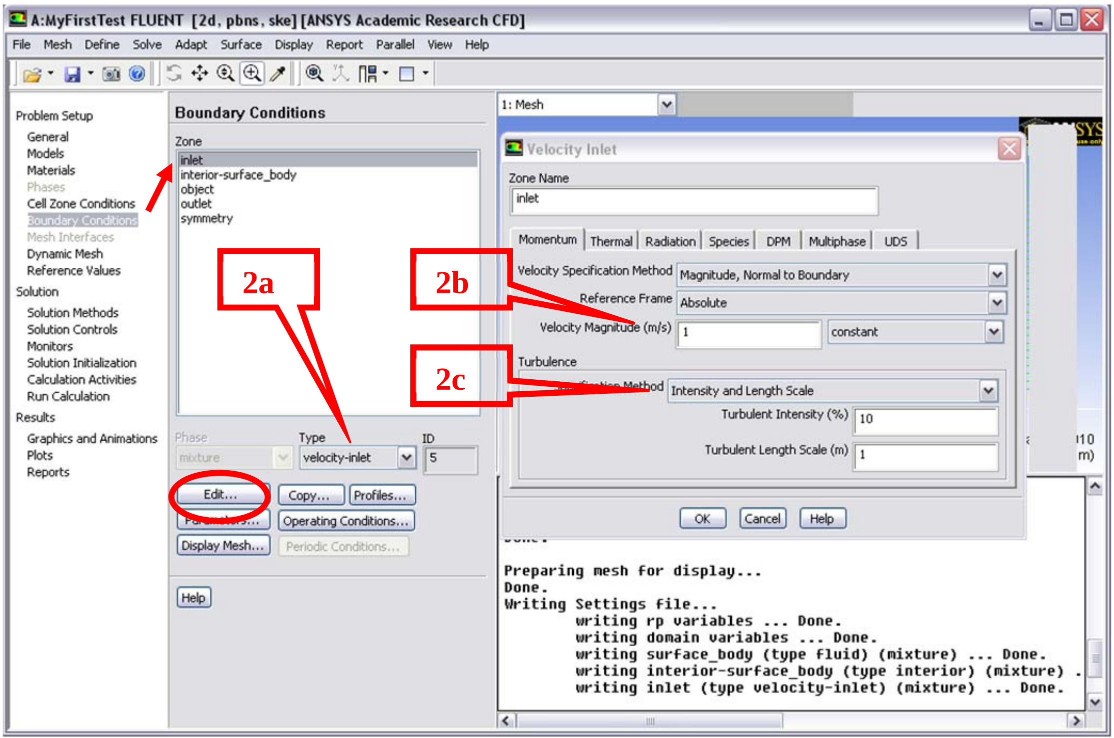
Documentation: Help -> ANSYS Workbench Help -> Meshing ->

How to compute the case using Fluent

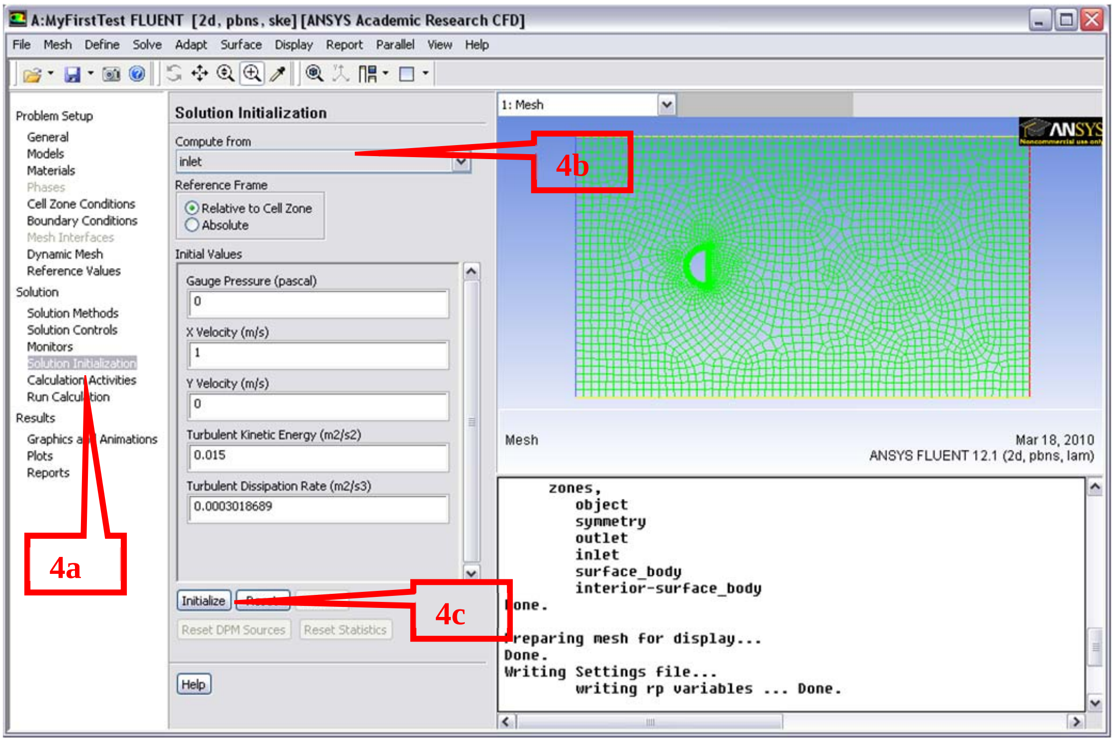
Start Fluent: DoubleClick on “Setup” in the Workbench Scheme



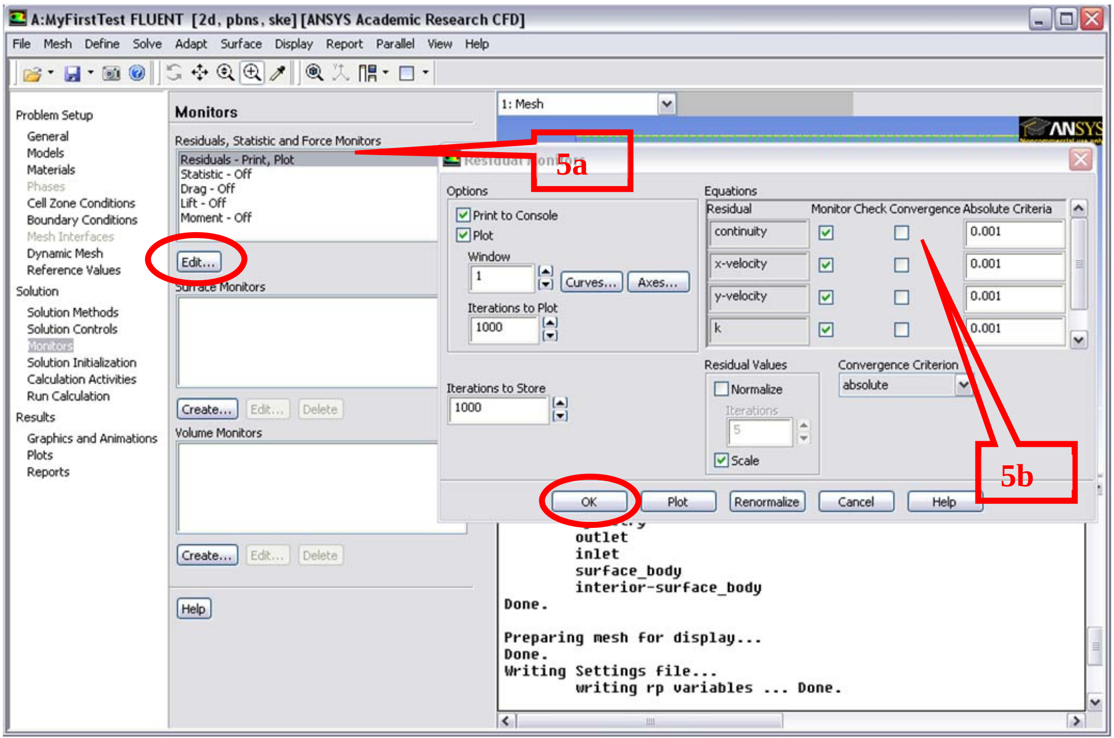
1. Set Standard k-eps model
    a. Click on Model
    b. Click Viscous – Laminar – Edit
    c. Choose k-epsilon



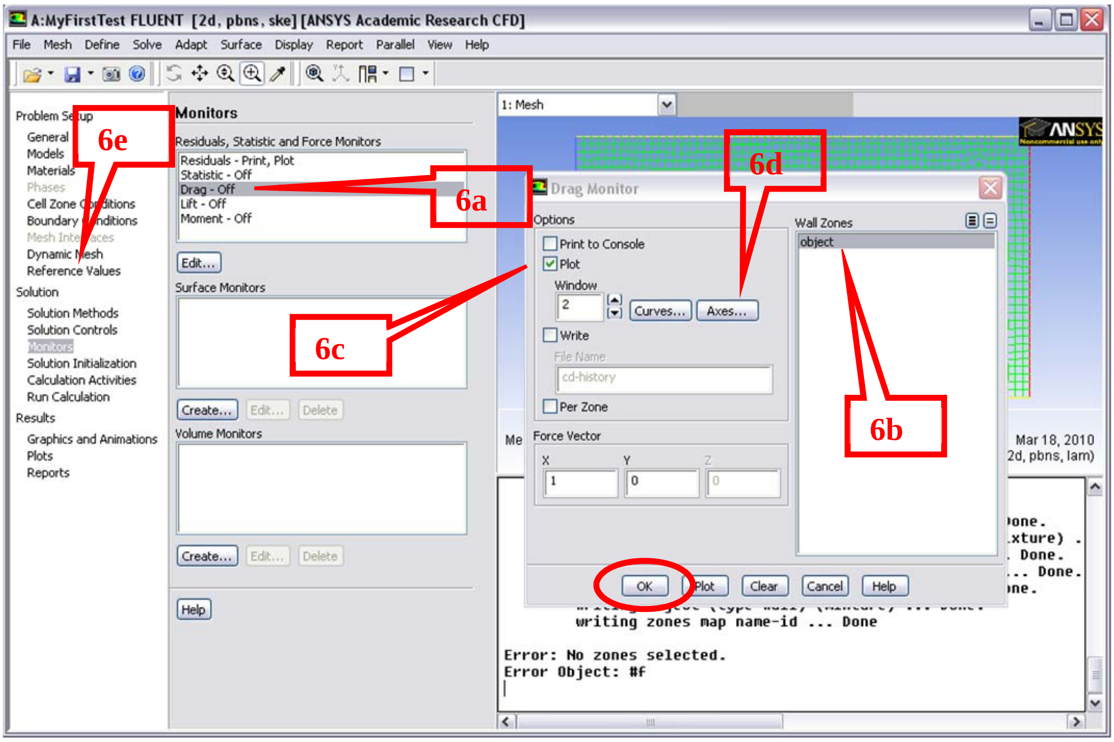
2. Inlet boundary conditions
    a. Boundary Conditions -> Inlet: Type = velocity-inlet. Edit
    b. Set velocity
    c. Set turbulence intensity and length scale
3. Other boundary conditions
    a. Outlet: Type=pressure-outlet, Gauge Pressure=0
    b. Upper and Lower: Type=symmetry
    c. Object: Type=wall



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

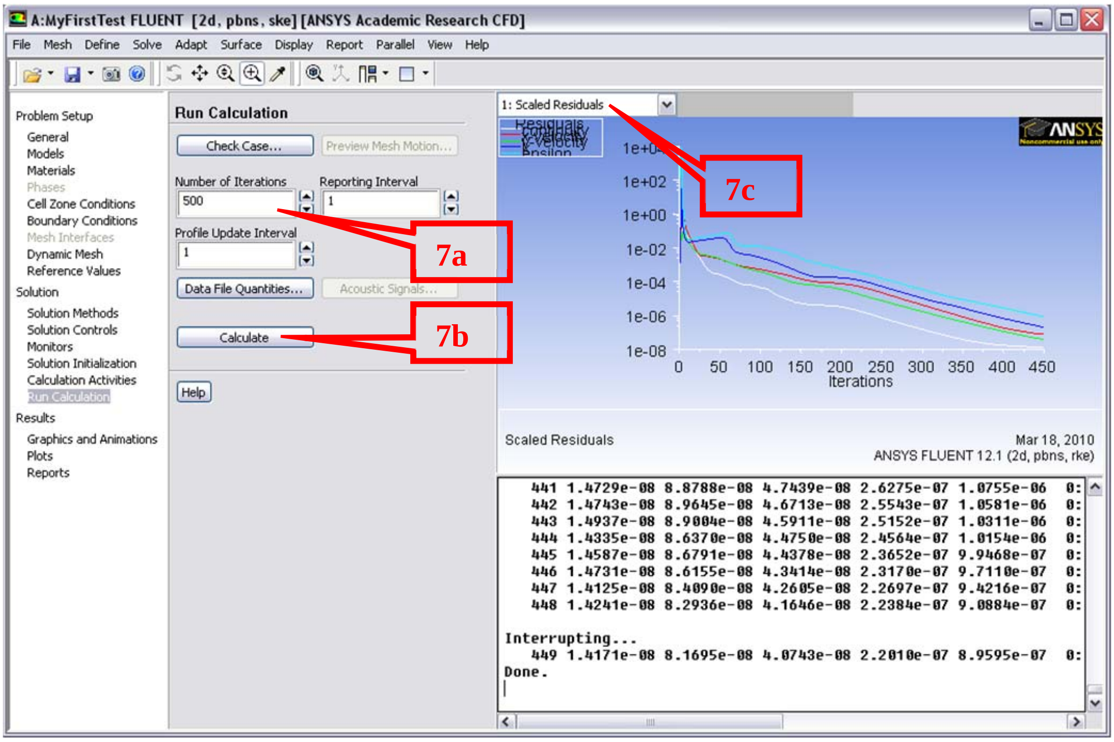


5. Monitor convergence
    a. Monitors -> Residuals -> Edit
    b. Uncheck “Check convergence” for all equatios (also epsilon, you need to scroll down).

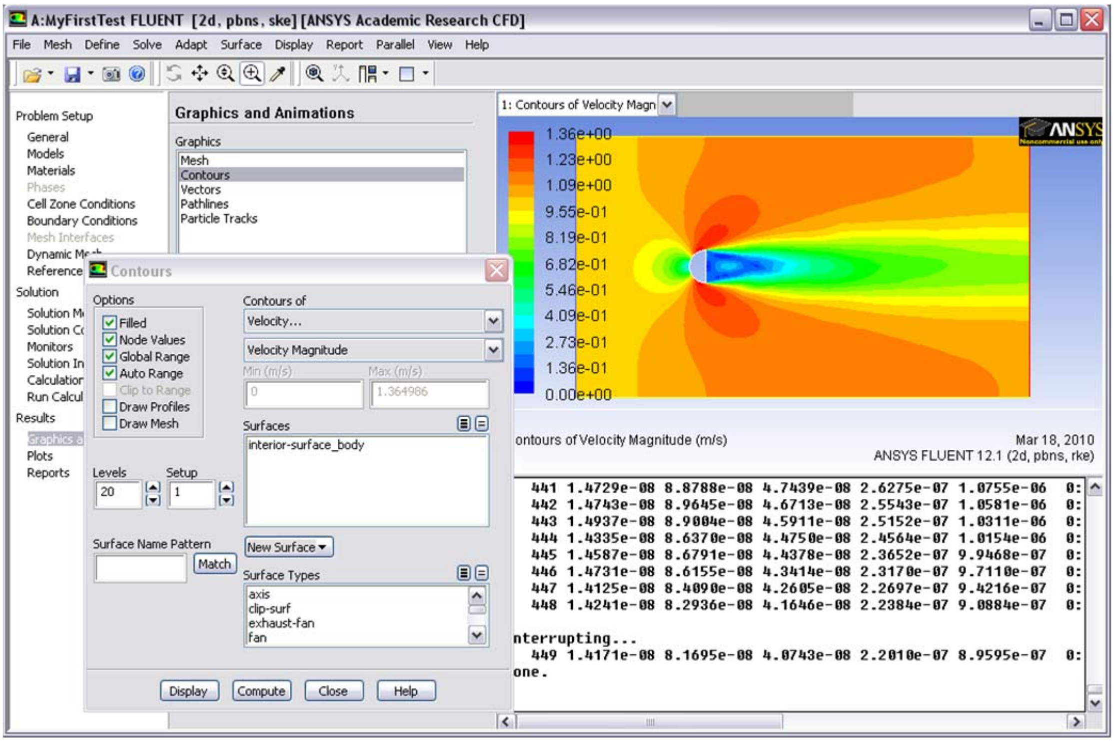


6. Monitor drag
    a. Monitor -> Create -> Drag
    b. Choose object
    c. Check "Plot"
    d. Axes -> Y -> Uncheck Auto Range, set Range 0 to 2
    e. Also set correct Reference Values:
        i. "Area" is reference are for Cd in x-direction.
        ii. For 2D it is per meter in z-direction
        iii. Example: Height=1.0m gives Area=1.0m^2

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



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

9. Try to refine calculation:
    a. Solution -> Solution Methods -> Momentum: Second Order Upwind
    b. 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.
    c. Adapt -> Gradients of Velocity
        i. Press “Compute”: Gives min and max values of velocity gradient
        ii. Set “Refine Threshold” to something smaller than max
        iii. Press “Mark”: Gives no of cell marked for refinement
        iv. Press “Adapt”: Refines the mesh
        v. Look at the mesh: Graphics and Animations -> Mesh -> Set Up

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