Customer Training Material

# Workshop 5

## Using Moving Reference Frames and Sliding Meshes

# Introduction to ANSYS FLUENT

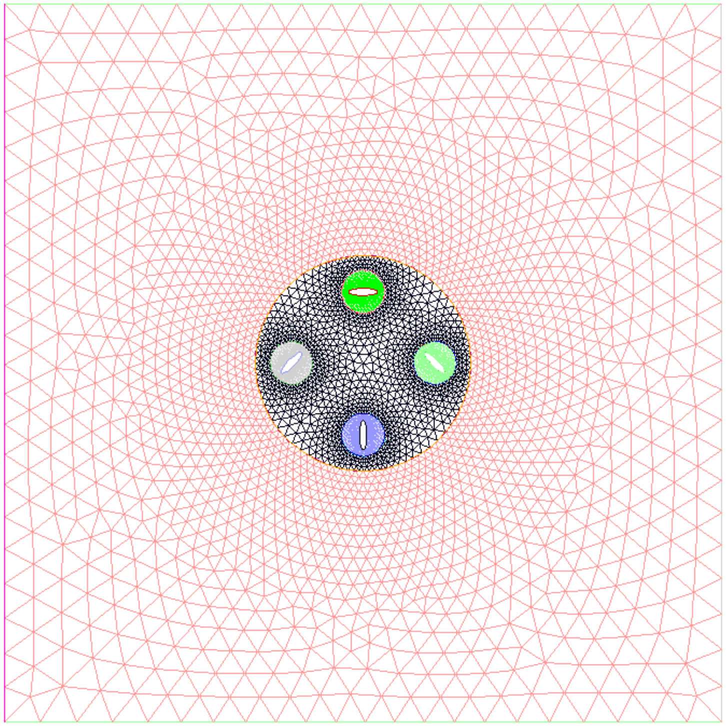
# Introduction [1]



- Several solution strategies exist when there are moving parts in the domain. This workshop will show two such strategies on the same model.

- The application is a vertical axis wind-turbine. This CFD model is a horizontal plane cut through the 4 vertical blades.

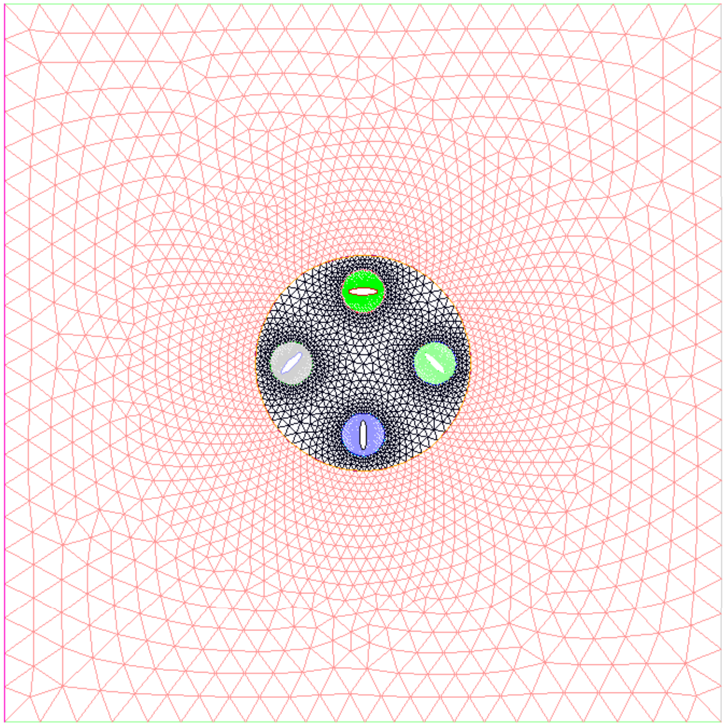
- To help understand the motion, look in the project folder and play the included movie file '`ws5-mesh-animation.avi`'



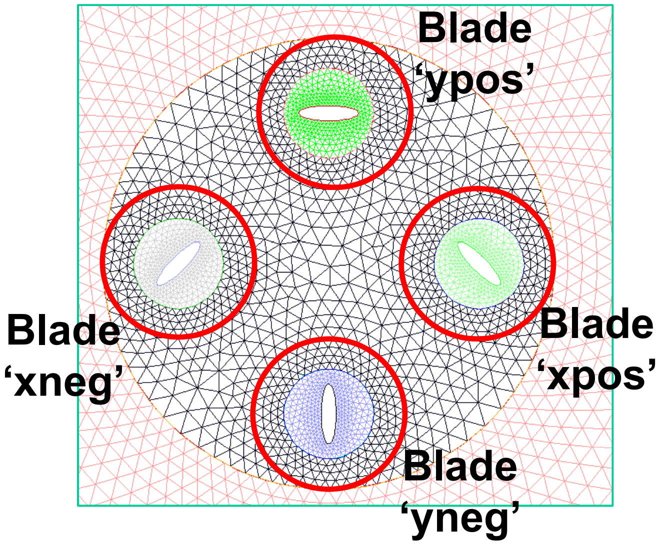
# Introduction [2]



- **Part 1:** This motion will be simulated using the steady-state solver, defining the motion using **Moving Reference Frames (MRF)**



- **Part 2:** The same problem will be solved in a transient manner, moving the mesh using **Sliding Meshes**

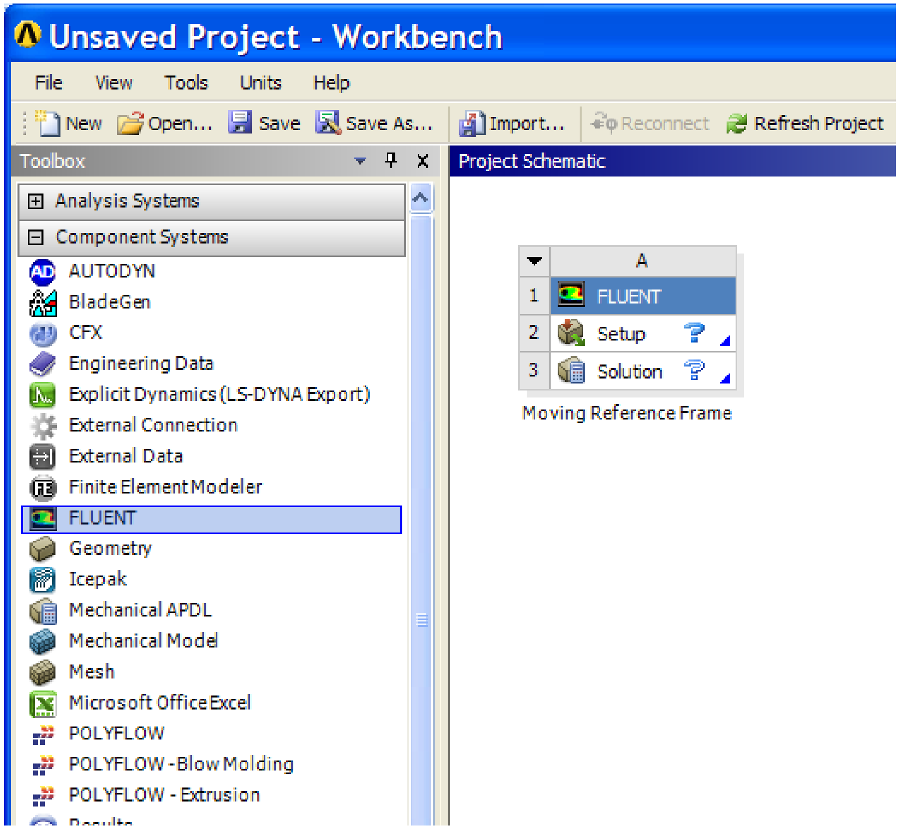


# Part 1 : Using Moving Reference Frames

# Starting FLUENT in Workbench

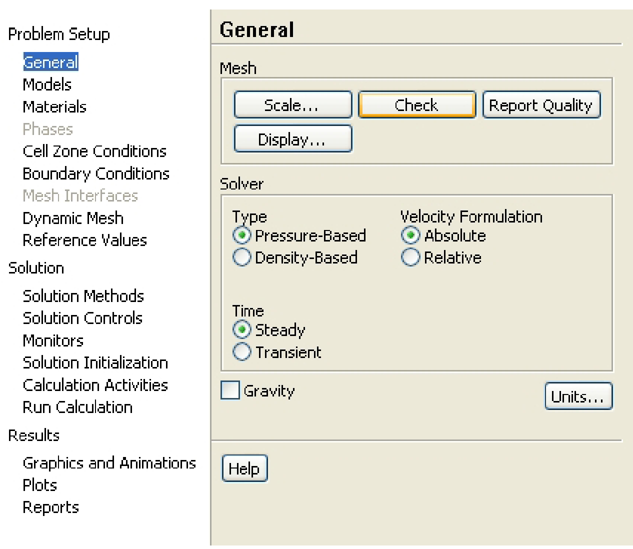
ANSYS
Customer Training Material

1. **Open the Workbench**
   Start > Programs > ANSYS 13.0 > ANSYS Workbench
2. **Drag FLUENT** ('Component Systems') into the project schematic
3. Change the name to ***Moving Reference Frame***
4. Double click on **Setup**
5. Choose ***2D*** and ***"Double Precision"*** under *Options* and retain the other default settings



# Import Mesh

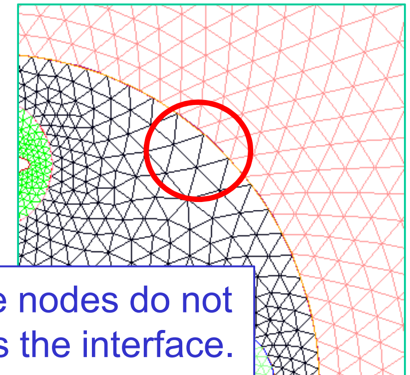
- **This starts a new FLUENT session and the first step is to import the mesh that has already been created:**
  1. Under the ***File*** menu select ***Import → Mesh***
  2. Select the file ws*5-simple-wind-turbine.msh* and click ***OK*** to import the mesh and check the scaling by using ***Scale*** under ***Problem Setup → General.***
     *The blades are 1m radius from the origin (0,0), with the outer bounding box +/- 5m in all directions.*
  3. After reading the mesh, check the grid using ***Mesh > Check*** option or by using ***Check*** under ***Problem Setup > General***



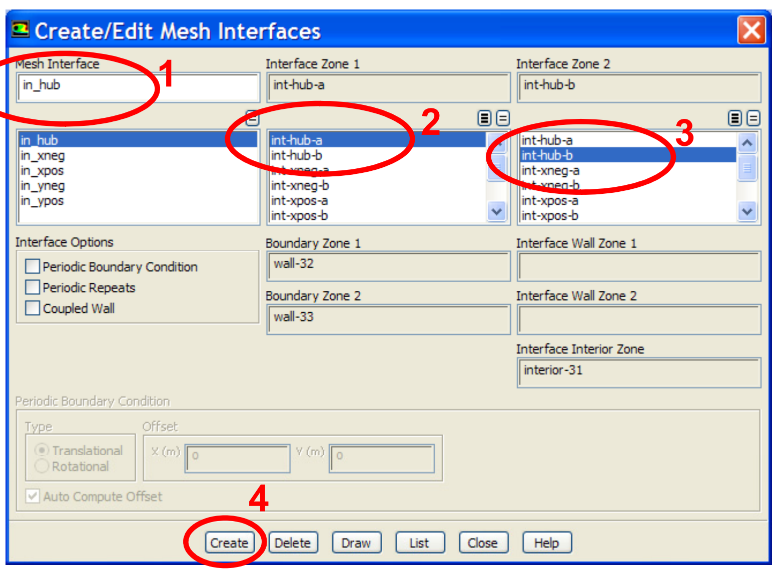

The mesh check will fail!
To allow for the motion later in this workshop, there are intentional non-conformal interfaces (where the mesh nodes do not match across an interface). These need to be paired up in the solver so that interpolation across the interface can occur – so fluid can flow across this join.
More generally, if in DesignModeler you produce several different parts, the mesh will also be non-conformal, and you will need to perform the next step to make sure the solver interpolates across the interface.

# Mesh Interfaces

- **Under Problem Setup > Mesh Interfaces**
  - Click ***Create/Edit***
  - For each row in the table below:
    1. *Enter the name of the mesh interface (column 1)*
    2. *Click on the name of Interface Zone 1 (column 2)*
    3. *Click on the name of Interface Zone 2 (column 3)*
    4. *Click Create*
  - Repeat the mesh check. It should now pass



Note how the nodes do not match across the interface.
• The boundary on the black side is 'Int-hub-a' and the red side is 'Int-hub-b'




| Mesh Interface | Interface Zone 1 | Interface Zone 2 |
|---|---|---|
| in_hub | int-hub-a | int-hub-b |
| in_xneg | int-xneg-a | int-xneg-b |
| in_xpos | int-xpos-a | int-xpos-b |
| in_yneg | int-yneg-a | int-yneg-b |
| in_ypos | int-ypos-a | int-ypos-b |

# Setting up the Models

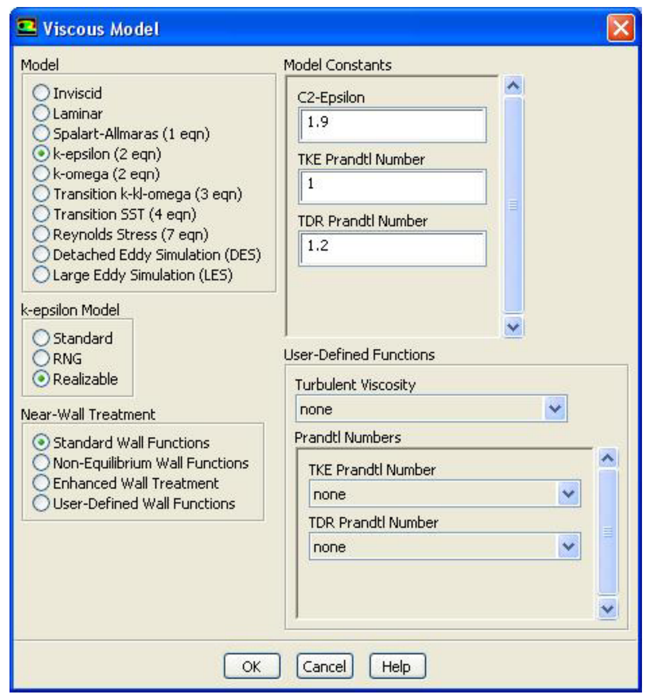


- **Keep the Pressure Based, Steady State solver**
  – Problem Setup > General > Solver
- **Specify Turbulence model**
  – Problem Setup > Models > Viscous
  – Double click and select ***k-epsilon (2 eqn)*** under *Model* and ***Realizable*** under *k-epsilon model* and retain the default settings for the other parameters
- For **Materials**, keep the default

  properties of the material *air*

  **density: 1.225 kg/m³**

  **viscosity: 1.7894e-5 kg/(ms)**

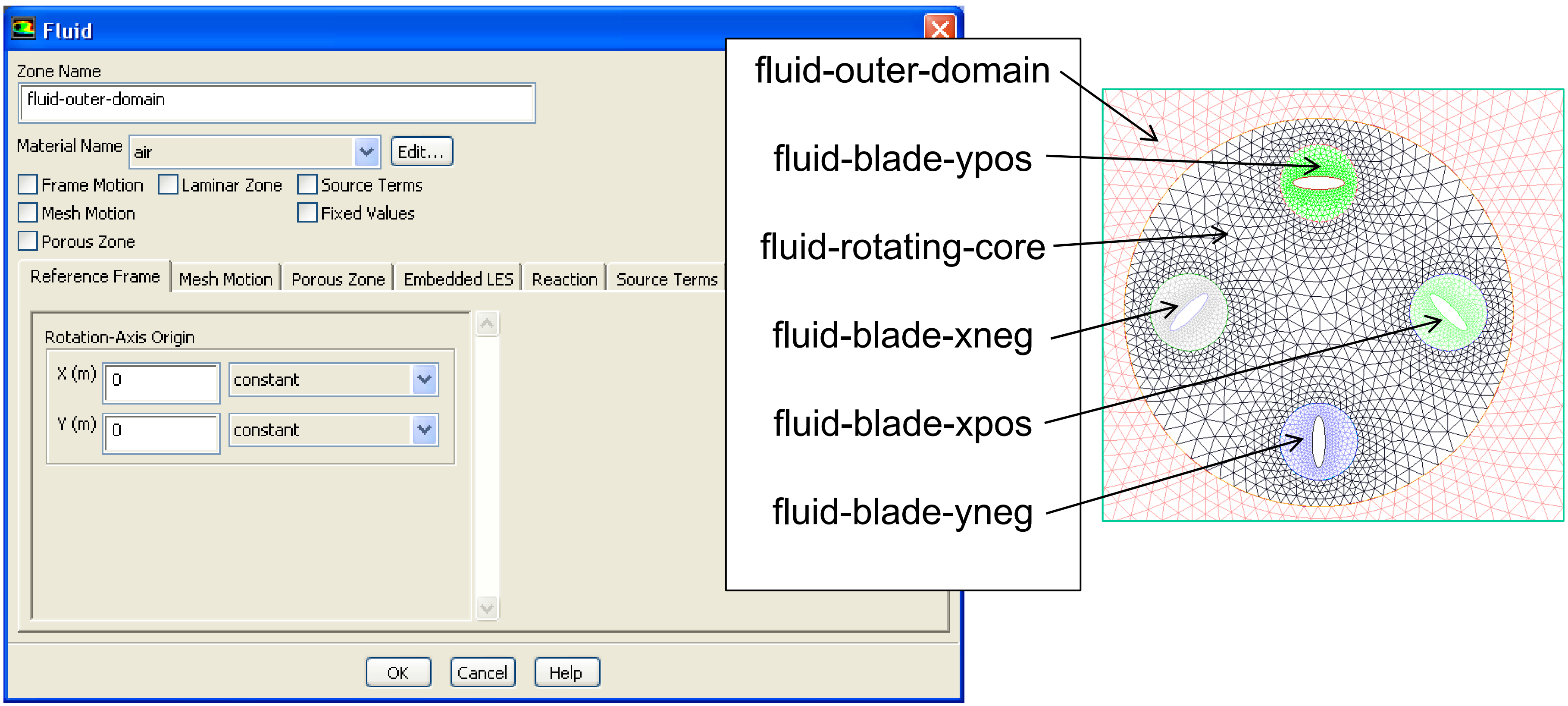


# Cell Zone Conditions (1)

Customer Training Material

- **Under Problem Setup > Cell Zone Conditions**
  - Select ***fluid-outer-domain*** and click ***Edit***
  - Observe ***air*** is already selected and click ***OK***

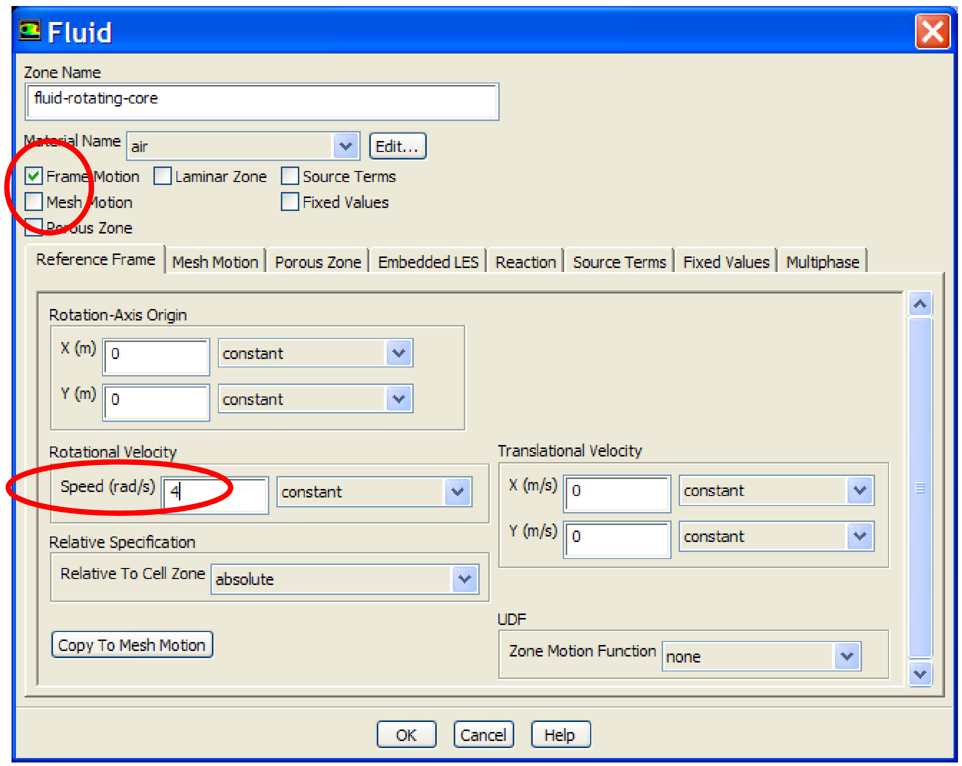
This zone is a stationary zone. It is not necessary to define a motion.



# Cell Zone Conditions (2)



- **Select *fluid-rotating-core* and click *Edit***
  - Observe ***air*** is already selected
  - Click ***Frame Motion***, to activate the Moving Reference Frame model
  - Retain the ***(0 0)*** as *Rotational-Axis Origin*
  - Select ***4 rad/s*** as *Rotational Velocity* and click ***OK***

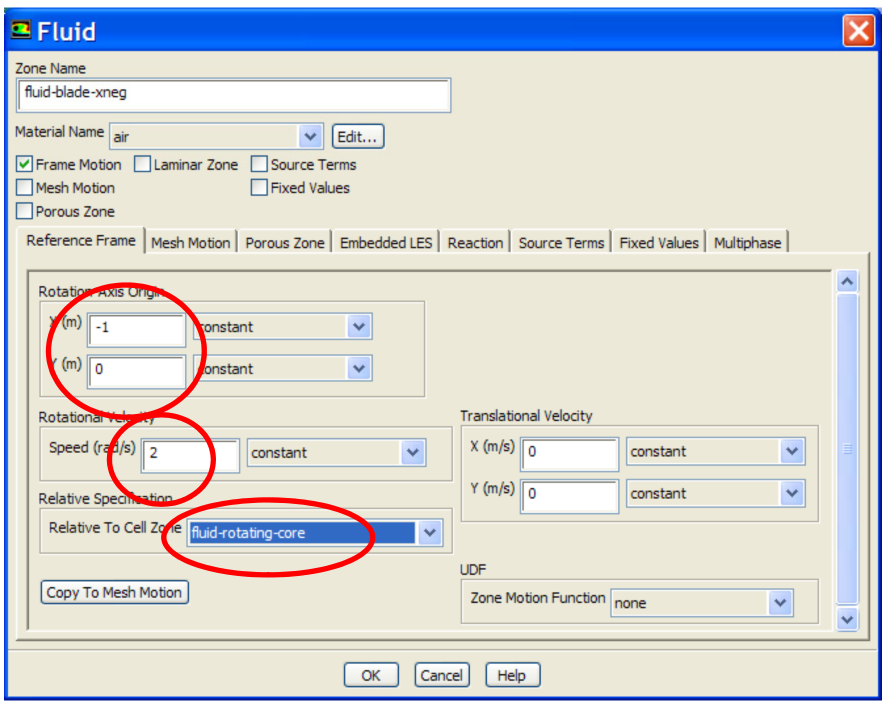



We can account for the motion of the parts, even in a steady state solver by using this technique. By specifying the rotation of the core, all the grid cells are given an additional source term to account for the local acceleration. This is known as using a moving reference frame.

# Cell Zone Conditions (3)



- **Select *fluid-blade-xneg* and click *Edit***
  - Observe ***air*** is already selected
  - Click ***Frame Motion***, to activate the Moving Reference Frame model
  - Set the Rotational-Axis Origin to [-1 0]
  - Set the Rotational Velocity to 2 rad/s
  - Select ***fluid-rotating-core*** as *Relative Specification* and click ***OK***

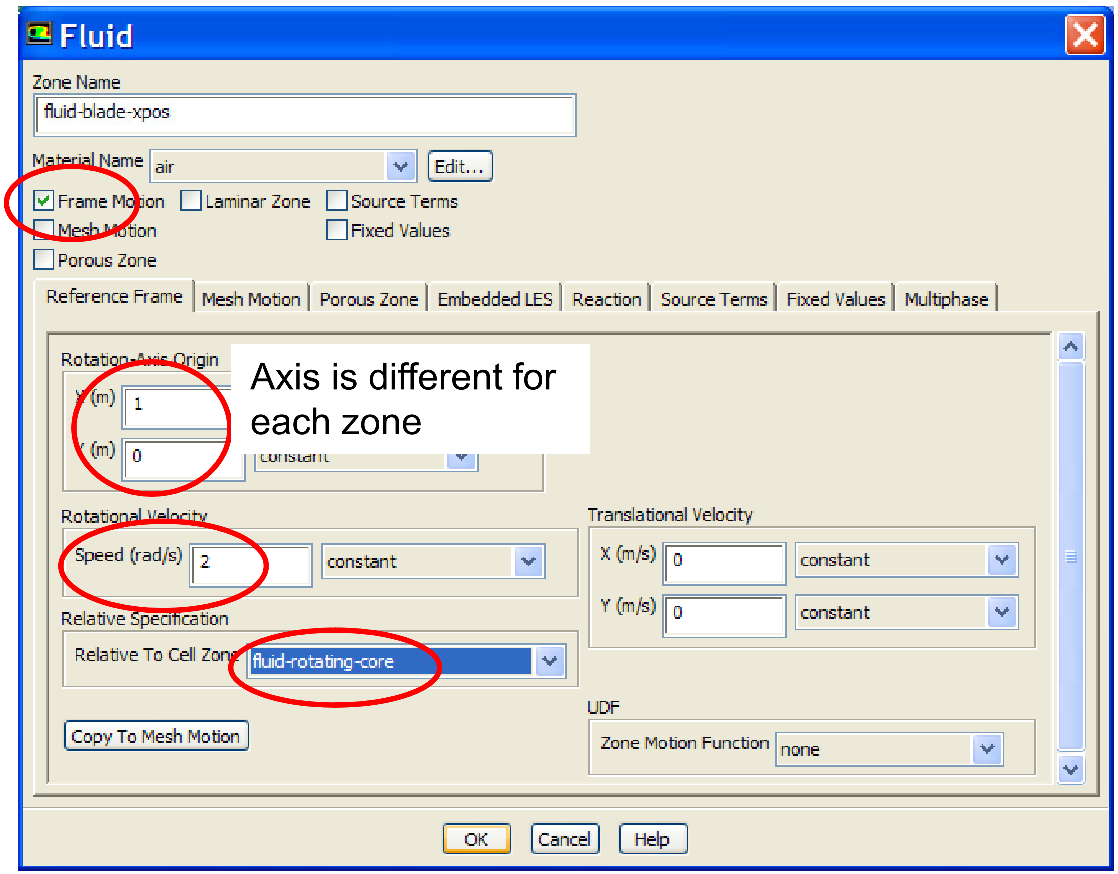



This zone is rotating about its own axis, which is 1m away from the global (hub) axis. The rotation speed is half that of the outer hub.

# Cell Zone Conditions (4)



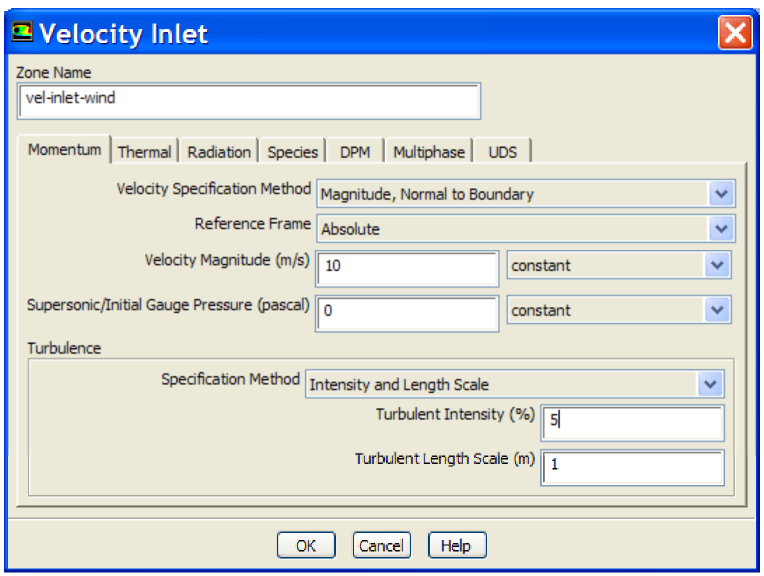
- **Repeat the instructions on the previous Slide for the other 3 blades:**
  - **Zone** fluid-blade-xpos **Axis** [1 0] **Speed** 2rad/s **Relative** fluid-rotating-core
  - **Zone** fluid-blade-yneg **Axis** [0 -1] **Speed** 2rad/s **Relative** fluid-rotating-core
  - **Zone** fluid-blade-ypos **Axis** [0 1] **Speed** 2rad/s **Relative** fluid-rotating-core

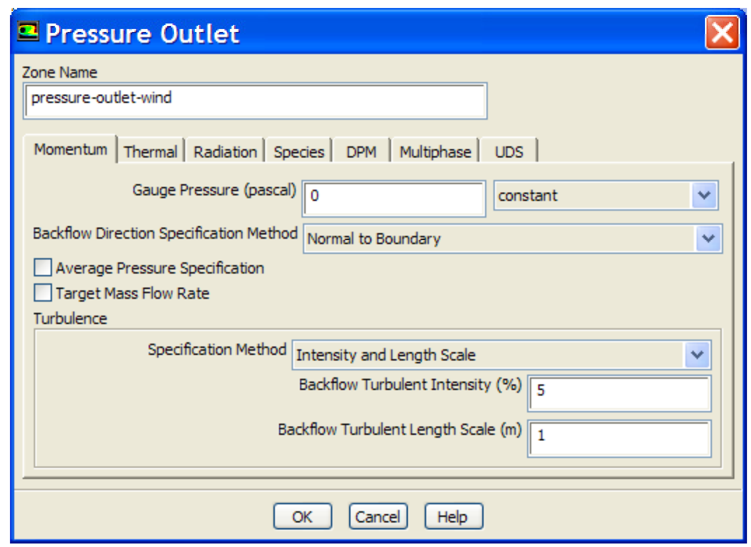


# Boundary Conditions (1)



- **Under Problem Setup > Boundary Conditions**
  - **Vel-Inlet-Wind**
    - Select ***vel-inlet-wind***, click ***Edit*** and set **10 *m/s*** as *Velocity Magnitude*
    - Choose ***Intensity and Length Scale*** under *Turbulence*
    - Set *Turbulence Intensity* to ***5 %*** and *Turbulent Length Scale* to ***1 m*** and click ***OK***
  - **Pressure-Outlet-Wind**
    - Select ***pressure-outlet-wind***, click ***Edit*** and set ***0 Pa*** as *Gauge Pressure*
    - Choose ***Intensity and Length Scale*** under *Turbulence*
    - Set *Turbulence Intensity* to ***5 %*** and *Turbulent Length Scale* to ***1 m*** and click ***OK***

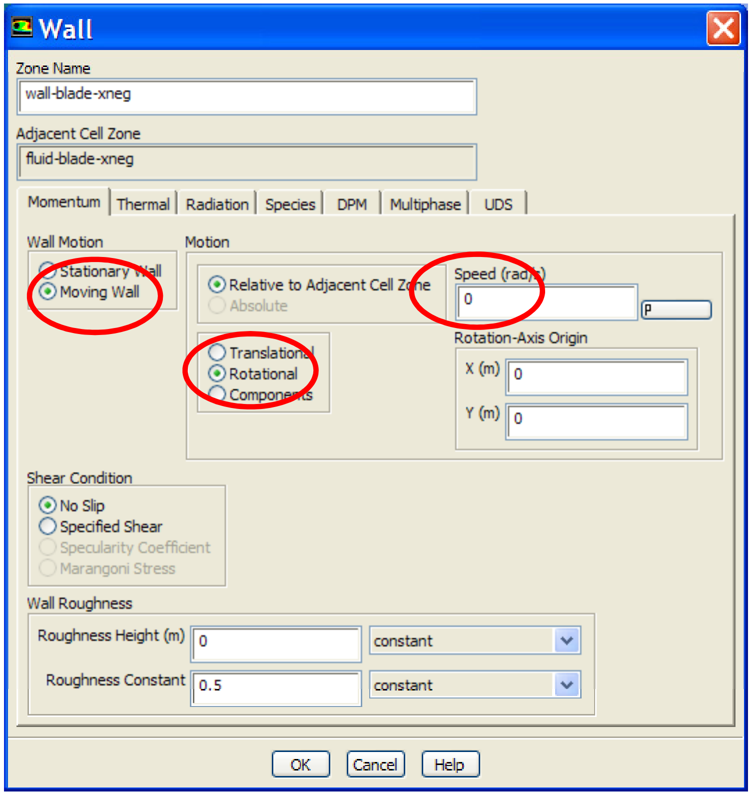


# Boundary Conditions (2)



- **Under Problem Setup > Boundary Conditions**
  - **Rotating wall**
    - Double-click ***wall-blade-xneg*** and select ***Moving Wall*** under *Wall Motion*
    - Select ***Rotational*** under *Motion* and retain ***0 rad/s*** as *Speed relative to cell zone*




The solver needs to know the speed of the wall so as to properly account for wall shear. Since the motion has been set in the cell zone, we will simply tell the boundary condition to use the same conditions (that is, zero velocity with respect to the cell zone).

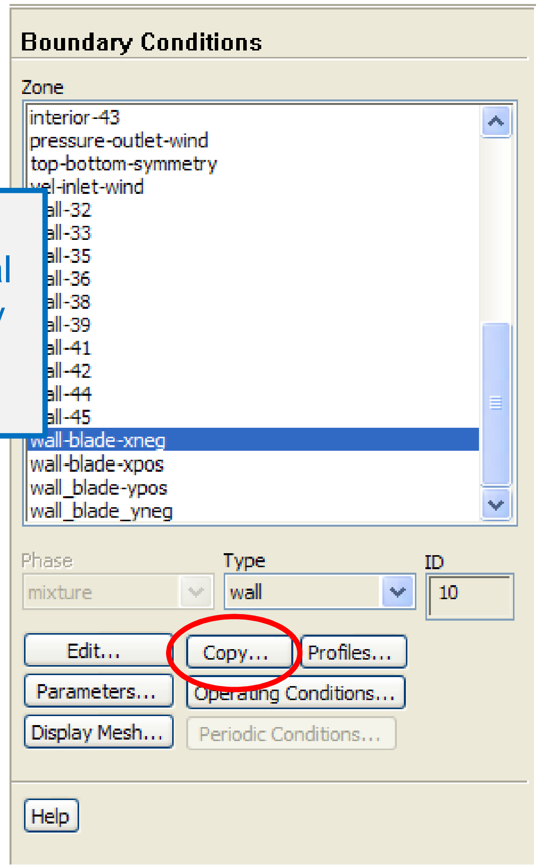
Note that this panel will tell you which cell zone is adjacent to this wall – look at the greyed-out box on the second line.

# Boundary Conditions (3)

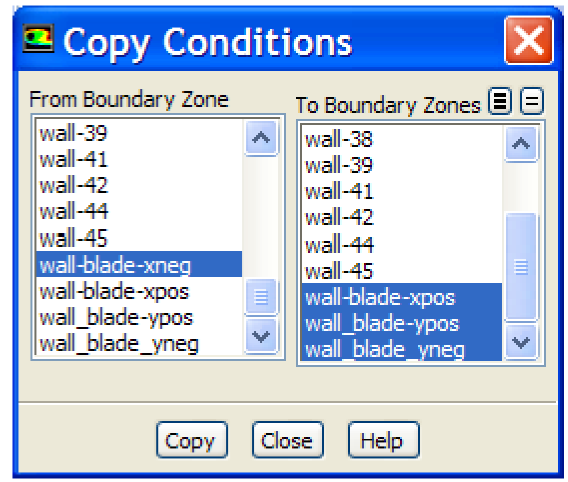


- **Copy the boundary conditions to other zones**

To save time, when boundaries have identical parameters, we can copy from one to (many) others.




1. Select *Copy*

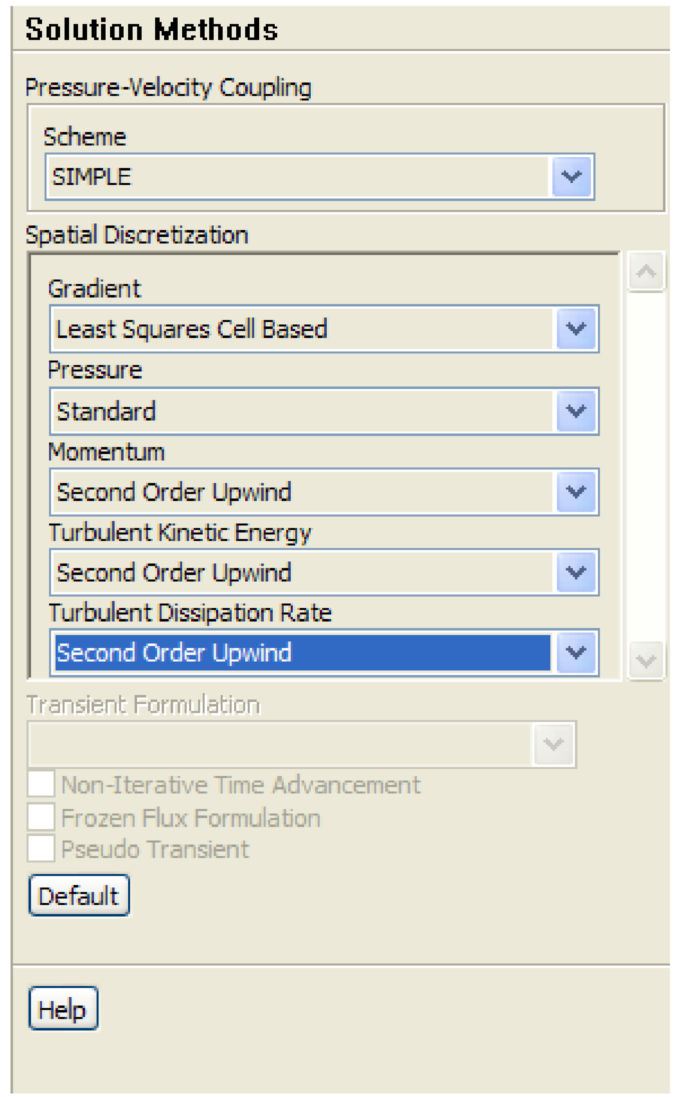



2. Select *wall-blade-xneg*
3. Select the three other blades zone (*wall-blade-xpos*, *wall-blade-yneg*, *wall-blade-ypos*)
4. Click ***Copy, then OK***

# Solution Methods



- **Under Solution > Solution Methods**
  – Select “Second Order Upwind“ for Momentum, and the turbulence equations

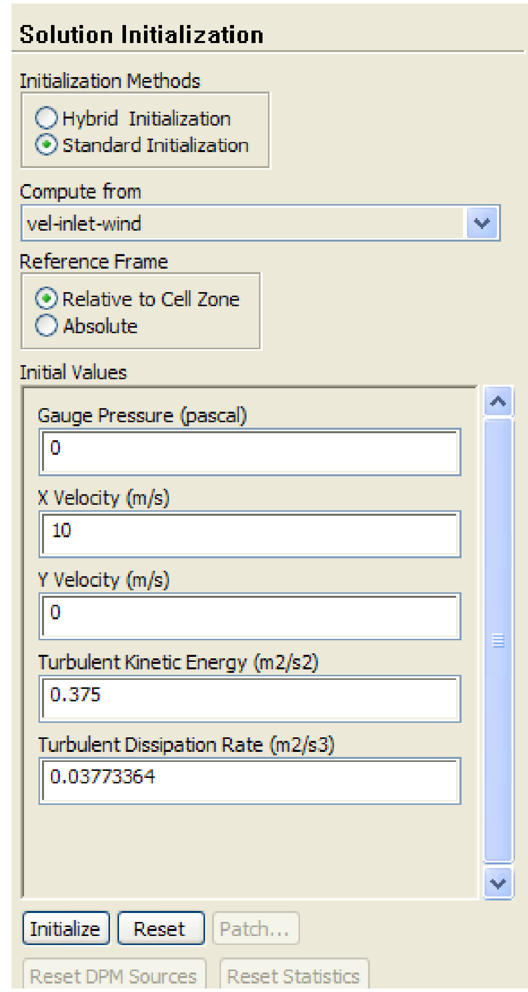



- **Under Solution > Solution Controls**
  – Retain the default settings

# Initialization



- **Under Solution > Solution Initialization**
  - Select ***Standard Initialization***
  - Choose ***vel-inlet-wind*** under *Compute from*
  - Select ***Absolute*** under *Reference Frame* and click ***Initialize***



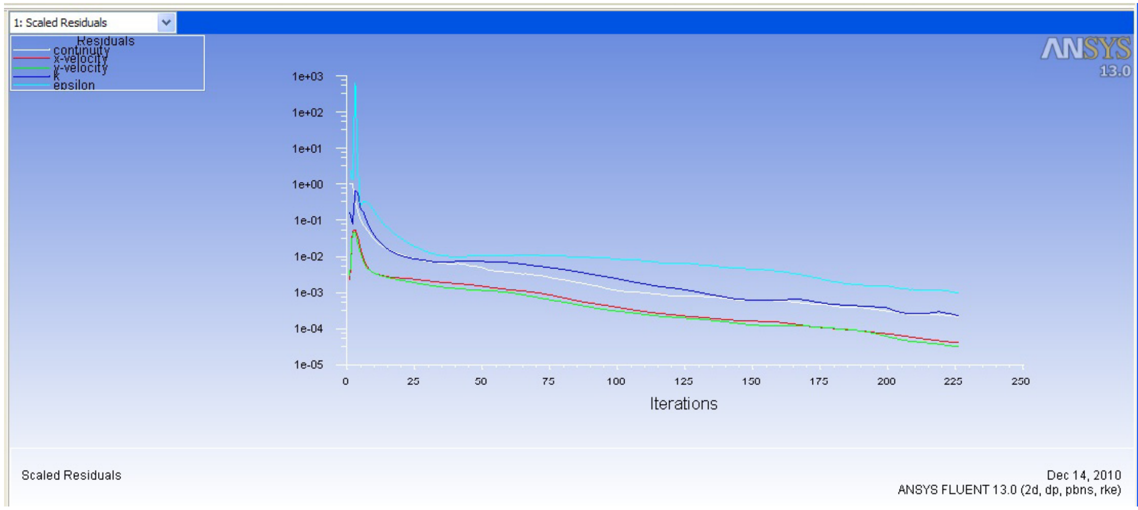
# Write Case and Data File



- **The model is now ready to run.**

- **First Save the Project to your normal working directory**
  - ***File > Save Project***

- **Then Run the Calculation**
  - Set ***500*** as *Number of Iterations*
  - Click ***Calculate*** to start the steady state simulation
  - It will reach the default convergence criteria in just over 200 iterations

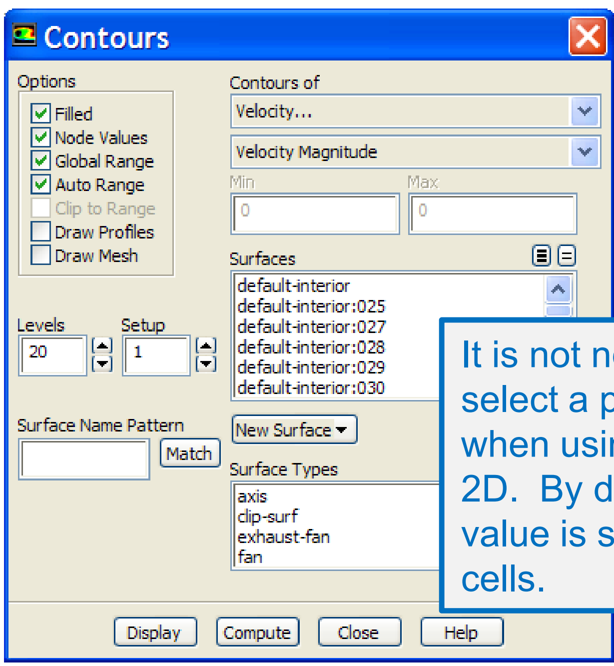



Generally you should always do further checks to determine convergence. However to save time in this example we will simply assume the default convergence criteria are sufficient.

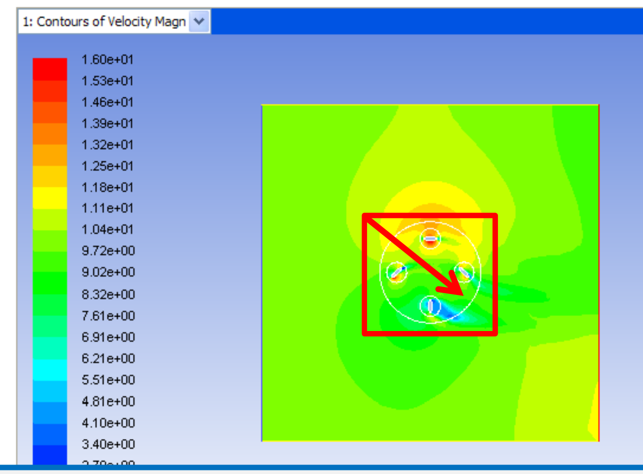
# Post Processing



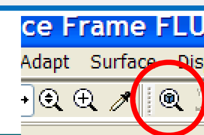
- **You can post process the results in FLUENT**
  - Results > Graphics and Animations
  - Select Contours and click ***Set Up***
  - Select ***Velocity*** under *Contours of* and ***Velocity Magnitude*** below that
  - Tick the box “Filled“, but **do not select any surfaces,** then **Display**
  - Zoom in on the hub using the middle mouse button, by drawing a box from top-left to bottom-right




It is not necessary to select a plotting surface when using FLUENT in 2D. By default the plot value is shown on all cells.



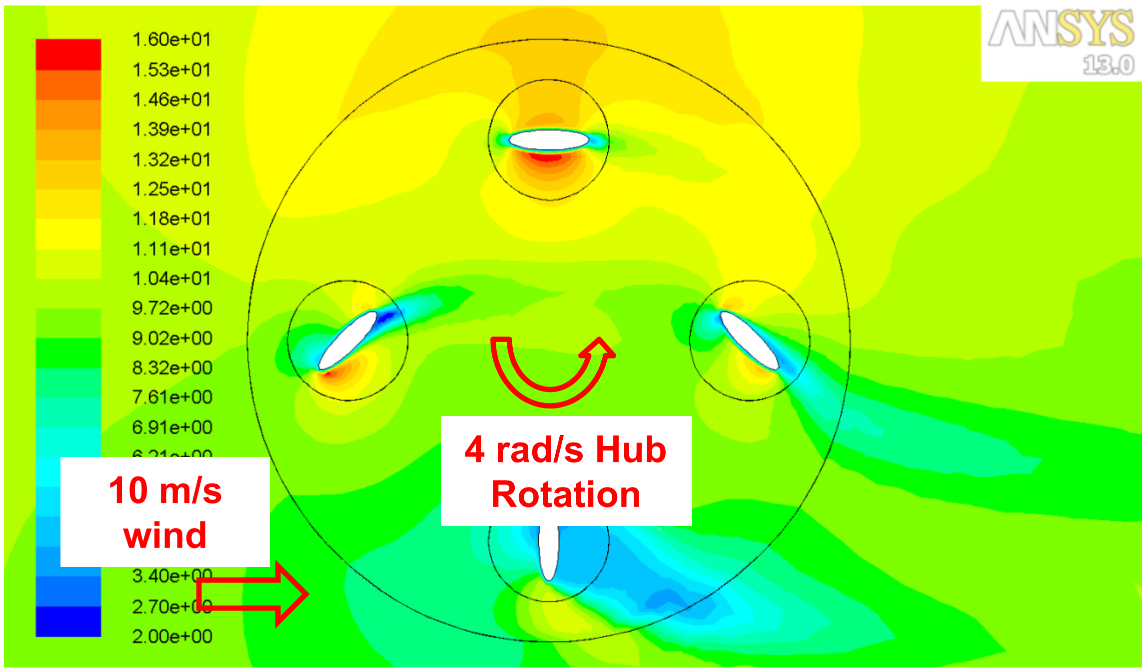

- Drag zoom box with middle mouse button.
- Opposite direction (right to left) will zoom out
- Use ‘Fit To Window’ to reset if necessary



# Post Processing [2]



- **The Velocity contours should look like this:**




Without the cell zone motion (MRF) this would have looked much more symmetrical between top and bottom

You can save this image from FLUENT for use in a report:
- File > Save Picture
- TIFF, and Colour
- Resolution 1200 x 1200
- 'Save'

# Write Case and Data File



- **Save the project**
  - **File > Save Project**

- **Close FLUENT, and return to the ANSYS Workbench window.**

# Part 2 : Using Sliding Meshes

# Sliding Meshes - Introduction



- There are times when the MRF assumption used in Part 1 is an over-simplification of the problem.
  - Only one position of the hub relative to the incoming wind was simulated
  - There will also be some vortices that affect each blade from other blades that have just passed upwind.

- In this next part, we will actually move the relative positions of all the components within FLUENT, and solve this in a transient (time-dependant) manner.

# Preparing Workbench

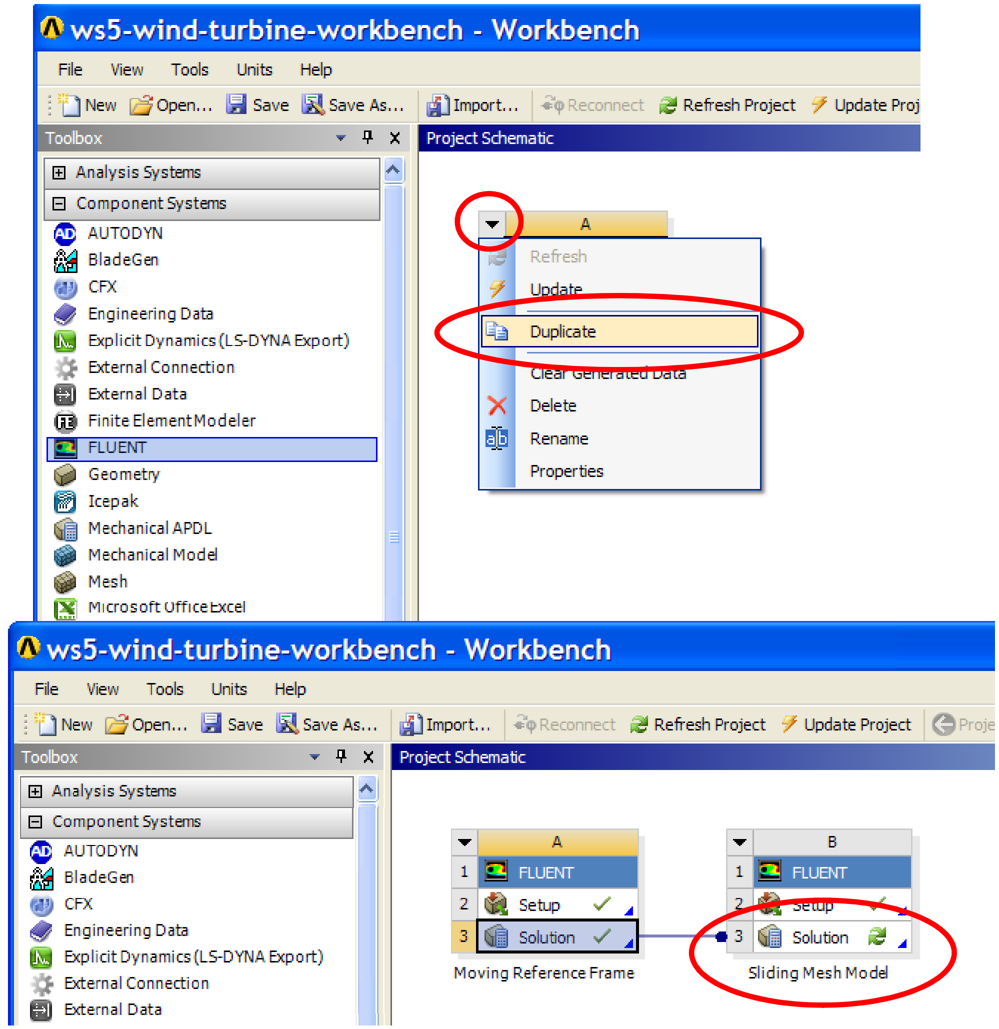


- On the Workbench Project Page, right-click on the arrow at the top of the FLUENT model, and select '**Duplicate**'

- Name the newly created copy 'Sliding Mesh Model'

- Drag the results over from the MRF model (drag a line from cell A3 to B3)
  - *This is so we can use the steady-state solution as a starting point for the new Sliding Mesh computation.*

- Launch FLUENT by double clicking on the new **Solution** cell (B3)



# Model Setup – General Comments

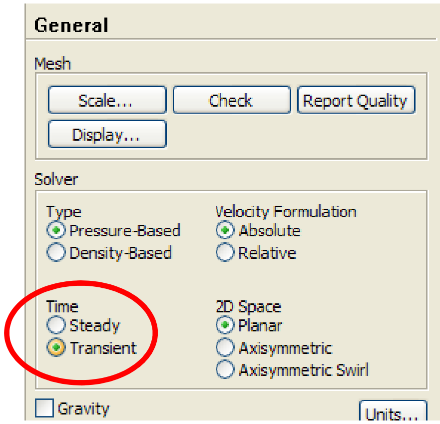


- All the model setup values (boundary conditions, etc) are available in the new FLUENT session.
- You may want to have a look (boundary conditions, plot velocity contours etc) to observe this for yourself.
- The following slides will show how to change this model to a sliding mesh case.
- Note that whereas the motion of the hub is easy to define (rotates about a fixed axis), it is harder to define the motion of the blades because their axis moves, in addition to the rotation of the blade.
- Supplied with this case is a FLUENT User Defined Function (UDF) to apply the rotation and translation to each blade.

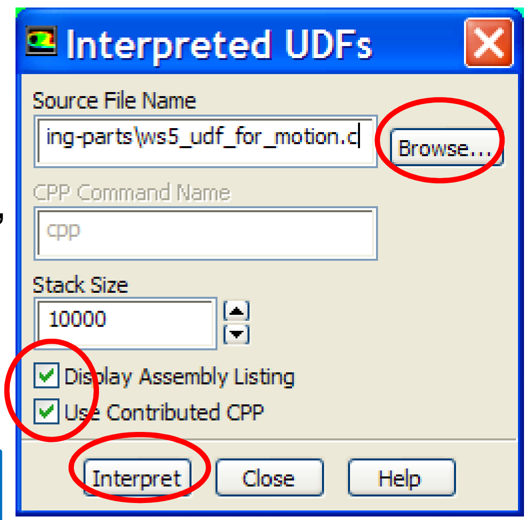
# Solver Setup



- Select **Transient solver**
  - Problem Setup > General > Solver




- Read in the supplied UDF
  - **Copy the UDF** **`ws5_udf_for_motion.c`** to the '**`user_files`**' subdirectory of your project
  - **Define → User Defined → Functions → Interpreted**
  - Select Browse, and pick this .c file from **`user_files`**
  - Enable '**Display Assembly Listing**' and '**Contributed CPP**'
  - Watch the FLUENT TUI window as you select '**Interpret**'
  - If successful, you will see no errors in the TUI window, just about 700 lines of machine code scroll past.

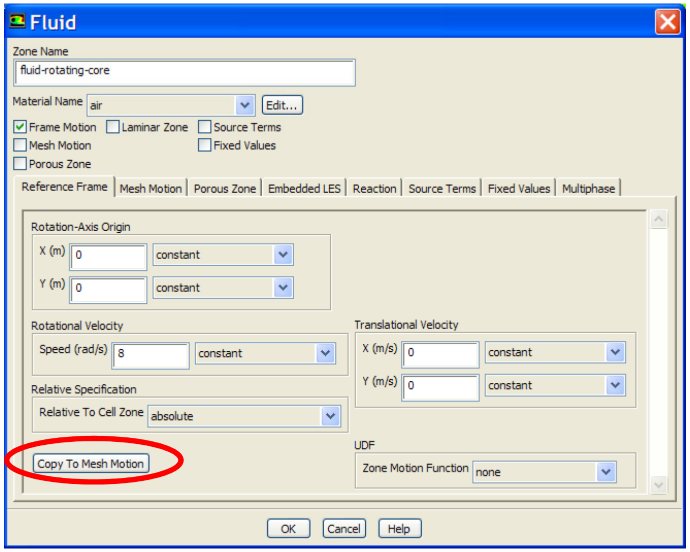


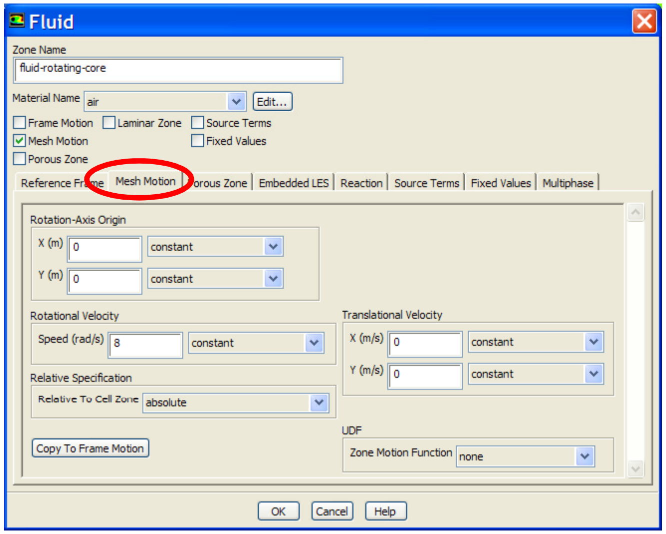

UDFs will be covered in more detail in their own lecture. There are two basic approaches, Interpreted or Compiled. We have opted for Interpreted here because this is a simple and quick to execute UDF (so run speed is not an issue), and because no additional software is needed on your computer.

# Setting up Cell Zones [1]



- **Under Problem Setup > Cell Zone Conditions**
- **Select *fluid-rotating-core* and click *Edit***
  - Click ***Copy to Mesh Motion*** in the *Reference Frame* Tab to activate the Sliding Mesh model
  - The motion type is changed from 'Frame Motion' to 'Mesh Motion'
  - Move to the *Mesh Motion* Tab and observe the rotation speed has been transferred
  - click ***OK***

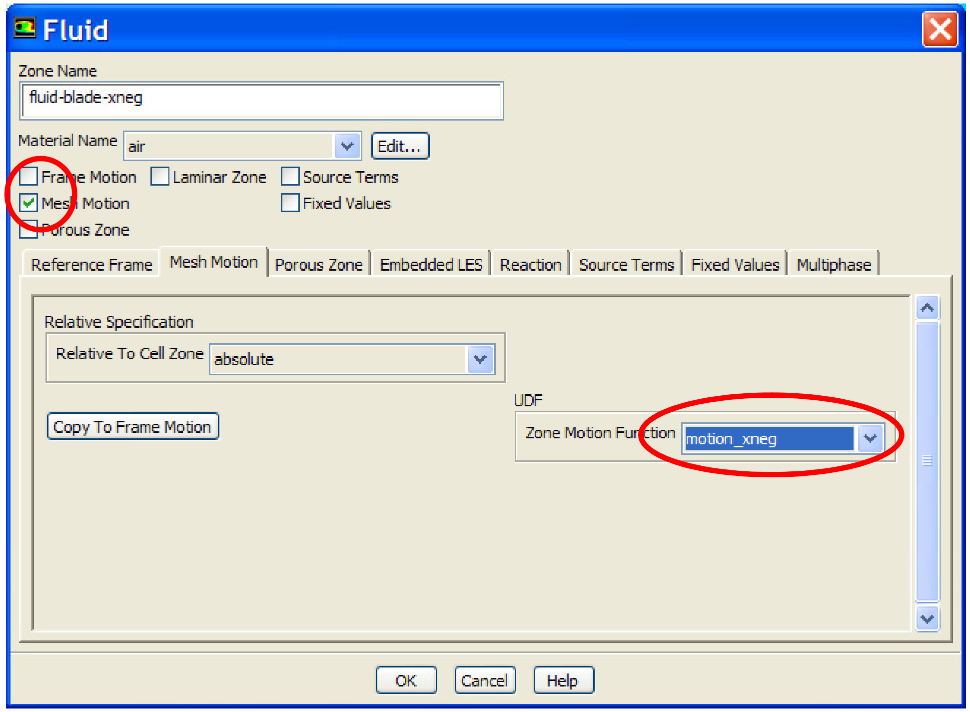


# Setting up Cell Zones [2]

- The setup will be slightly different for the 4 blades.
- **Select *fluid-blade-xneg* and click *Edit***
  - The motion will be set maually via the UDF, we do not need to copy values from the MRF settings (The UDF has hard-coded the axis position and speed)
  - **Un-select** *'Frame Motion'*
  - ***Select*** *'Mesh Motion'*
  - On the *Mesh Motion* Tab, for UDF select 'motion_xneg'

    *be careful to pick the correct UDF and only select 'Mesh Motion'!*
  - *Select 'OK'*




- **Repeat the above** for the other zones:
  - fluid-blade-xpos (motion_xpos)
  - fluid-blade-yneg (motion_yneg)
  - fluid-blade-ypos (motion_ypos)

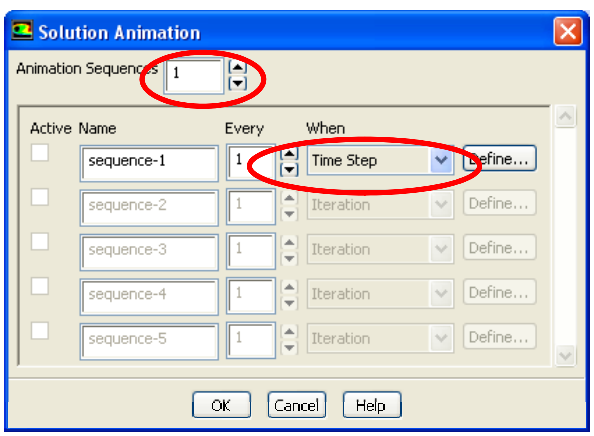
    *be careful to pick the correct corresponding UDF. Re-visit each cell zone and verify the settings are correct*

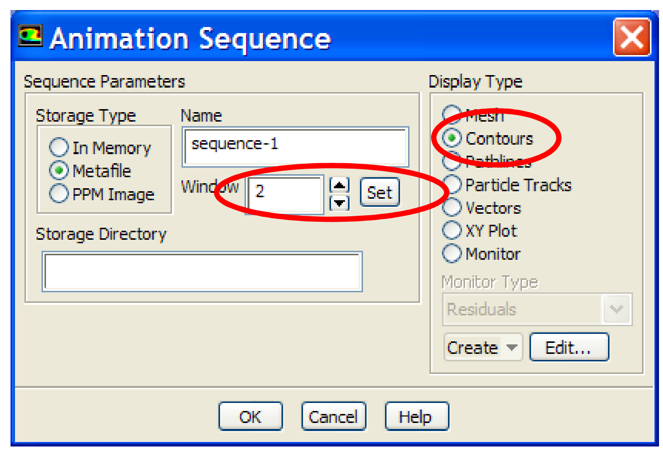
# Setting up an Animation



- **Solution > Calculation Activities > Solution Animations > *Create/Edit***
  - In the Solution Animation Panel increase the *Animation Sequences* to ***1***
  - Select ***Time Step*** under *When* and click ***Define***
  - In Animation Sequence Panel select Window **2, SET**, then ***Contours***

It is very useful with transient simulations to record key-frames of the solution progress. Not only does this help understand the result, but also aids de-bugging if the settings are not as intended.



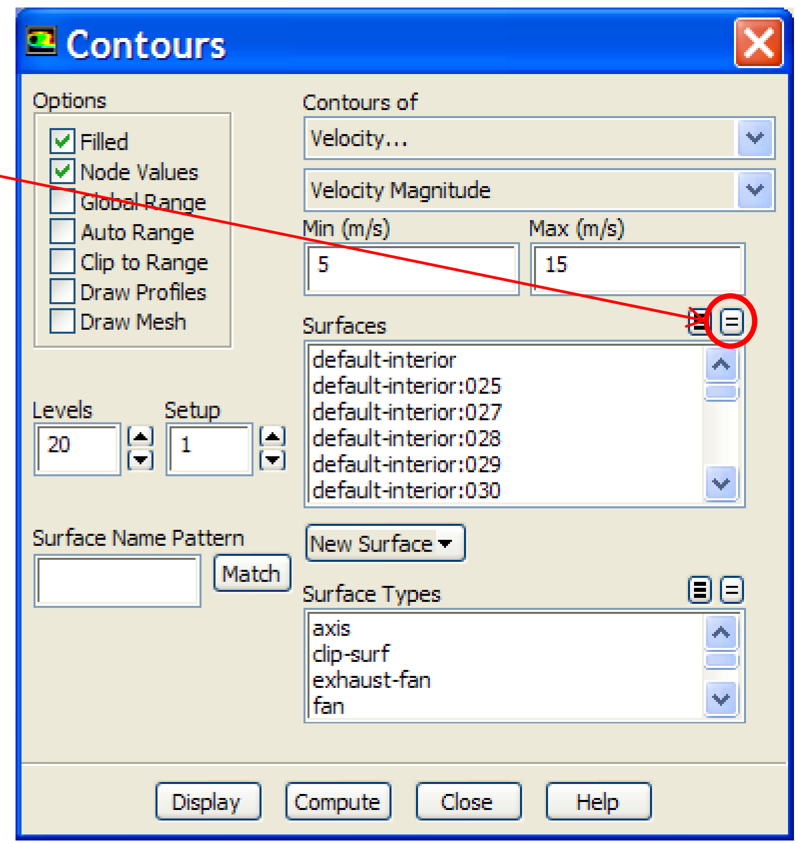



# Setting up an Animation [2]

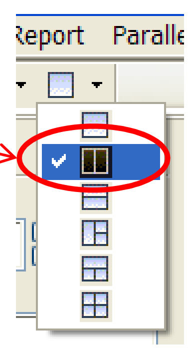
Customer Training Material

- On the Contour Panel:
  – Set  Contours of Velocity > Velocity Magnitude
  – Select ‘Filled’
  – Deselect Global Range, Auto Range and Clip to Range
  – Enter Min=0, Max=20
  – Deselect all surfaces
  – Click ‘Display’
  – Close the Contour Panel
  – **OK** the Animation Sequence Panel
  – **OK** the Solution Animation Panel

Forcing a Max and Min value will ensure all frames in the animation are consistent.




- Graphic layout:
  – Enable 2-window display
  – Use the middle mouse button to zoom in on the blades in graphic window 2



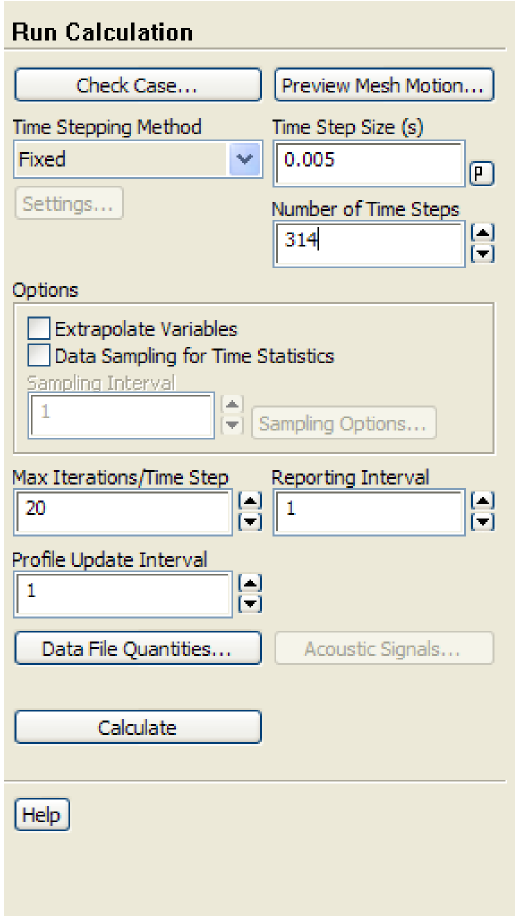
# Write Case and Data File



- **The model is now ready to run.**

- **First Save the Project to your normal working directory**
  - ***File > Save Project***

- **Then Run the Calculation**
  - Set **0.005s** as time step size
  - Set **314** as number of time steps
  - *4 rads/sec equates to 1.57secs/rotation.* 0.005s x 314 = 1.57s
  - Click ***Calculate*** to start the transient simulation



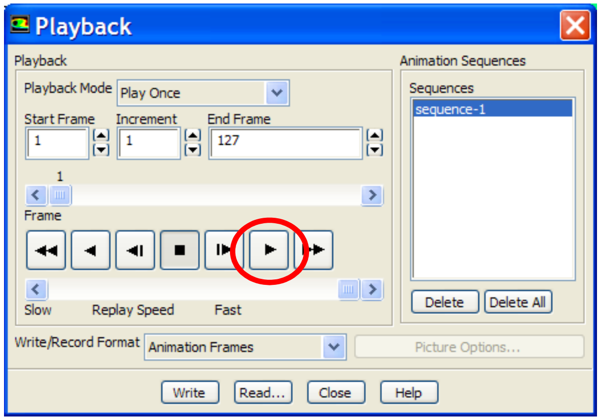

An Initialization of this case is not necessary because we want to continue the simulation with "start condition" of the Moving Reference Frame

# Reviewing the Solution



When the calculation is complete:

- Graphics and Animations > Solution Animation Playback > Setup
  - Selecting the 'Play' button will let you review the animation



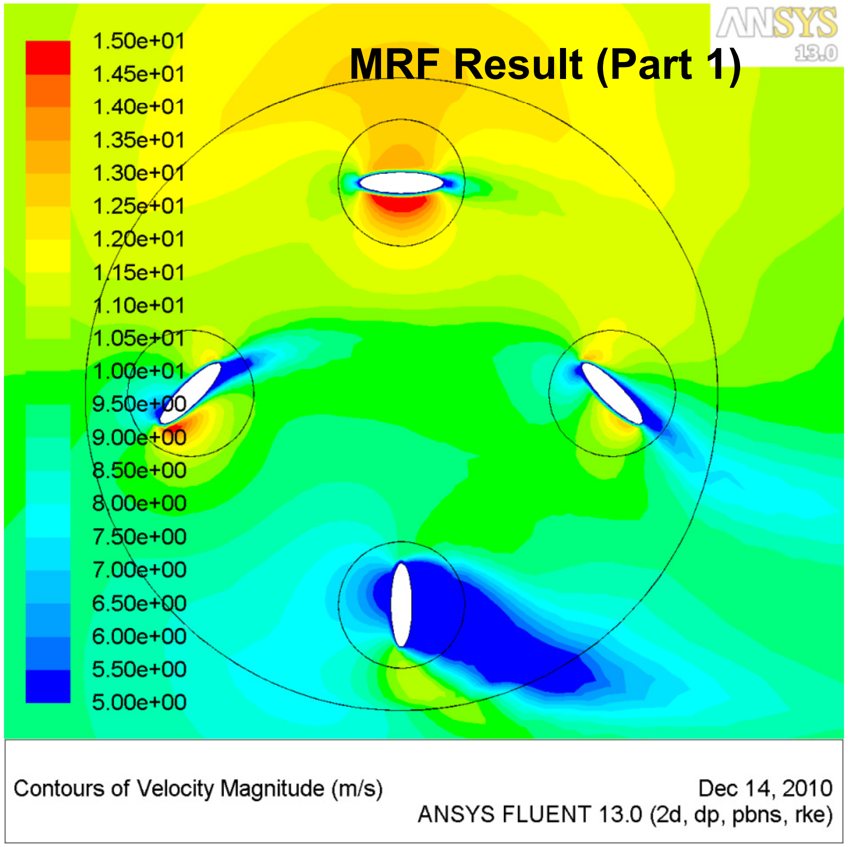

  - Alternatively you can select Write/Record Format, and type MPEG. (this will appear in the folder

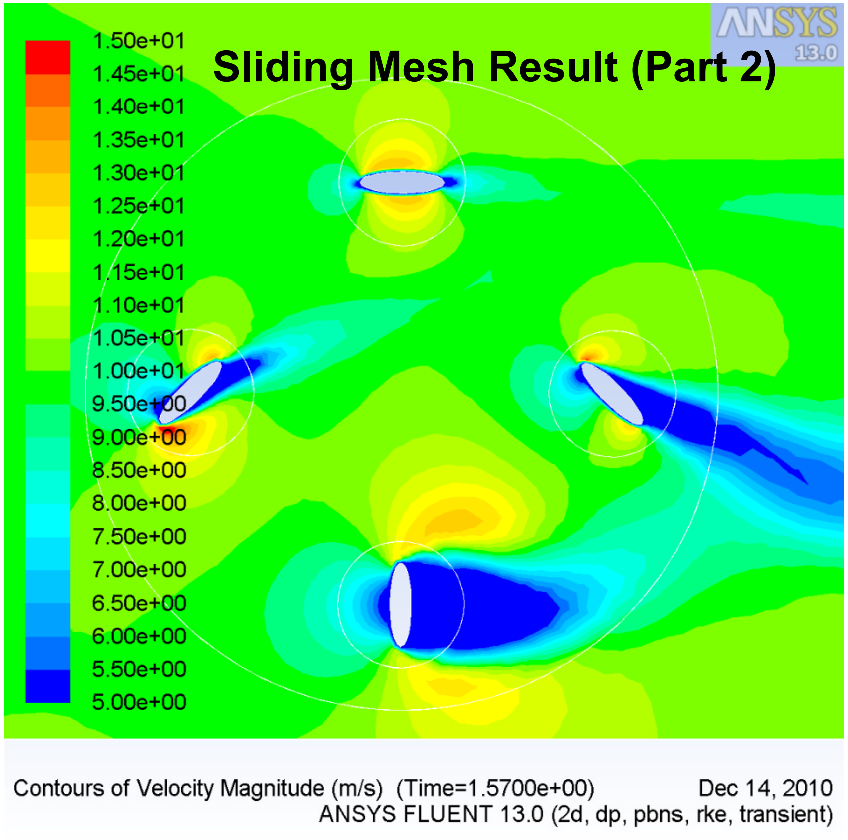
    `working directory\workbench_project_name\dp0\FLU-1\Fluent`)

# Comparing the Results



- The images below show a comparison of the result from the MRF and sliding mesh approaches.
- Although the MRF approach gave a useful early indication of the velocity field and wake behind the blades, a full Transient approach was needed.



# Closing Comments



- **Save the Project**
  - **File > Save Project**

- This workshop has shown how the fluid region can be modified by the solver at every timestep. There is no need to have to go back to the pre-processor to generate a fresh mesh at each step.

- The key feature needed in the mesh in order to do this was having different, disconnected (non-conformal) cell zones. Since they are disconnected, FLUENT could move them as a rigid body

- If the parts actually change shape, there are further tools available in FLUENT (the Dynamic Mesh Model) which allow much greater changes to be made to the mesh, adding and removing grid cells where necessary.

# Optional Extras



- If you have finished early, ahead of the rest of the class, you may want to investigate reporting the turning moment on the device.

  - Use **Report → Forces → Moments** (and set appropriate axes)
  - Pick the 4 blades
  - Compare the result from the two cases computed.

  - The Moment Coefficient can also be plotted as a graph during the solution process **Monitors → Moment.**
  - Click the 'Help' button on this panel and follow the link to 'Monitoring Force and Moment Coefficients' to find out more.
  - Note that this reports a coefficient, which uses the values set in

    **Report → Reference Values** (see help pages).
  - Once set, if you run the solver on for further timesteps, you will see how the moment varies sinusoidally as the device rotates.