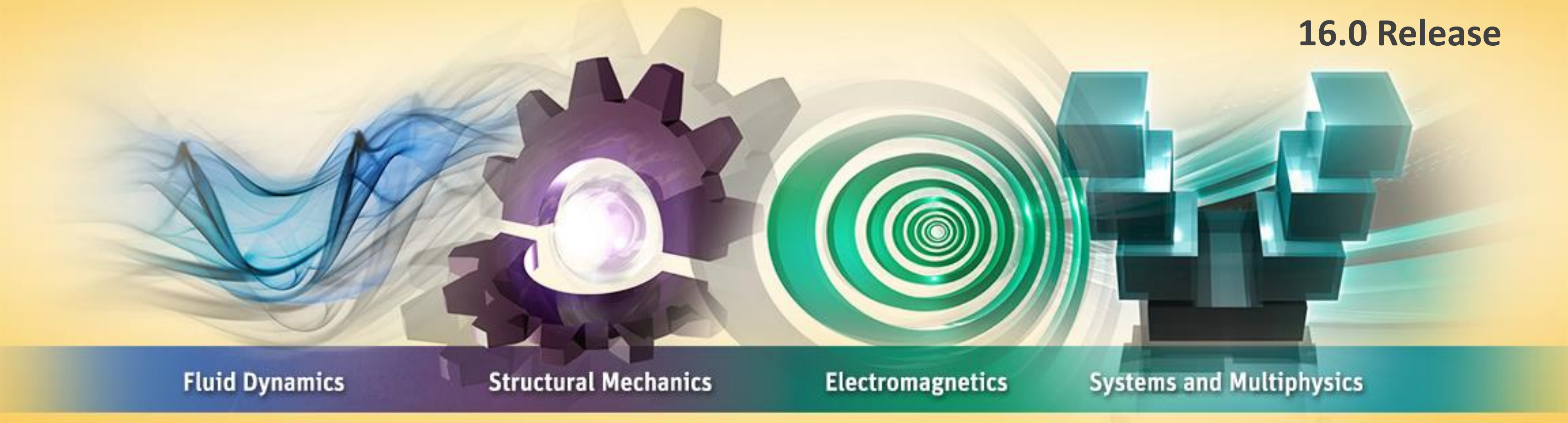
ANSYS

Lecture 6:
Solver Settings and Output File

16.0 Release



Introduction to ANSYS CFX

Realize Your Product Promise®

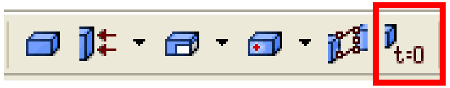
Introduction

- **Lecture Theme:**
  - CFX requires inputs which tell it how to calculate the solution. By introducing the concepts of accuracy, stability and convergence, the purpose of each setting can be understood. Emphasis will be placed on convergence, which is critical for the CFD simulation
- **Learning Objectives:**
  - You will be able to choose appropriate solver settings for your CFD simulation

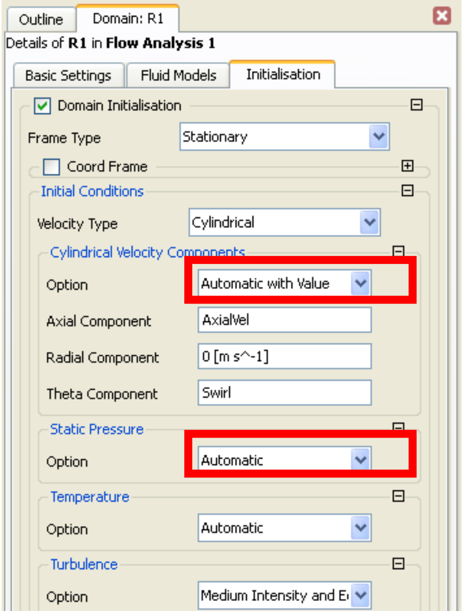
- **Outline**
  - Initialization
  - Solver:
    - Convergence Control
    - Residuals
    - Additional convergence checks
    - Output Control
    - CFX Solver Manager
  - CFX Solver Output File
  - Summary

Initialization

- **All solution variables must be assigned initial values**
- **A good initial guess can reduce solution time**
- **A poor initial guess may cause solver failure**
- **The initial values can be set in 3 ways:**
  - **Automatic option**
    - **CFX-Solver calculates initial values based on boundary condition values and domain settings**
  - **Automatic with Value option**
    - **User specified value will be used**
    - **Can use a constant value or an expression**
  - **From a previous solution *(see next slide)***

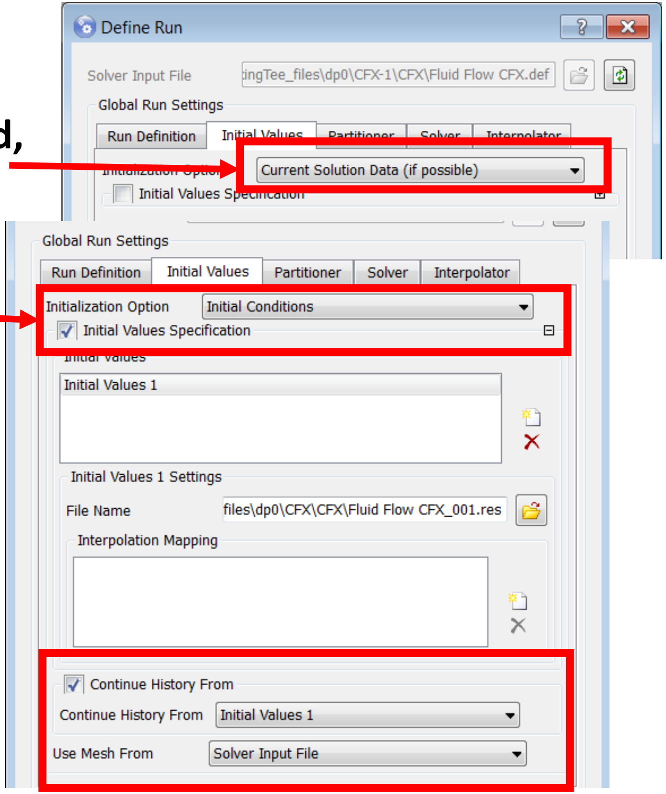


Insert Global Initialization from the toolbar or by right-clicking on Flow Analysis 1



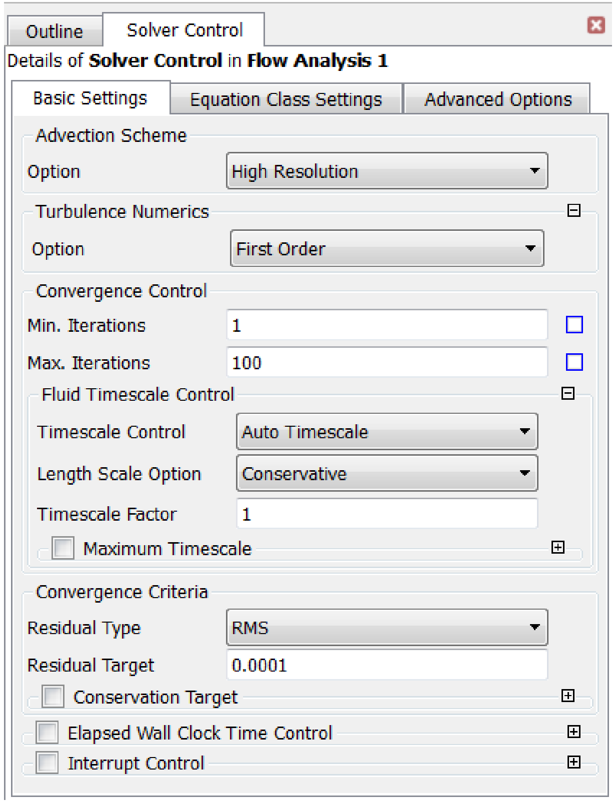
Initialization – Using a Previous Solution

- **Initial Values tab**
  - **by default in Workbench results from a previous run are used, Current Solution Data**
  - **to specify results file**
    - **set option to Initial Conditions**
    - **switch on 'Initial Values Specification'**
    - **Select the file (res, .bak or _full.trn)**
  - **Continue History From carries over convergence history & iteration counters**
  - **Use Mesh From:**
    - **Solver Input File → Initial Values interpolated on to input file**
    - **Initial Values → only physics from the Solver Input File used**



Solver Control – Options

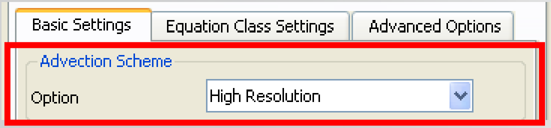
- **The Solver Control panel contains various controls that influence the behavior of the solver**
- **These controls are important for the accuracy of the solution, the stability of the solver and the length of time it takes to obtain a solution**



Introduction | Initialization | Solver | Output File | Summary

ANSYS

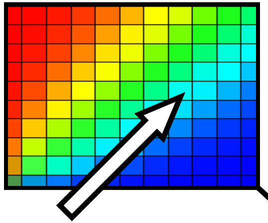
Advection Scheme Theory



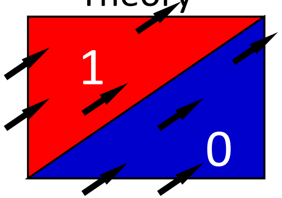
$$\phi_{ip} = \phi_{up} + \beta \nabla\phi \cdot \Delta\vec{r}$$

- ***Upwind*** **advection scheme → β = 0**
  - **robust but only first-order accurate**
- ***Specified Blend*** **scheme → 0< β < 1**
  - **when the correction is included it can overshoot or undershoot what is physically possible**
- ***High Resolution*** **scheme maximizes β throughout the flow domain while keeping the solution bounded**

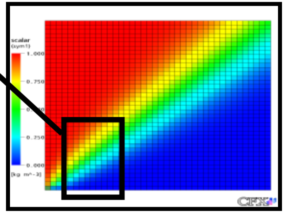
Flow is misaligned with mesh



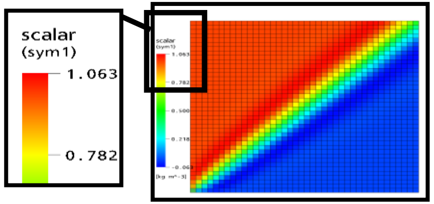
Theory



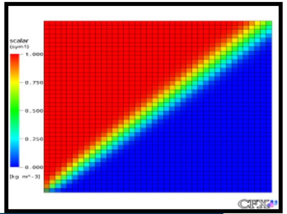
Upwind Scheme



β=1.00



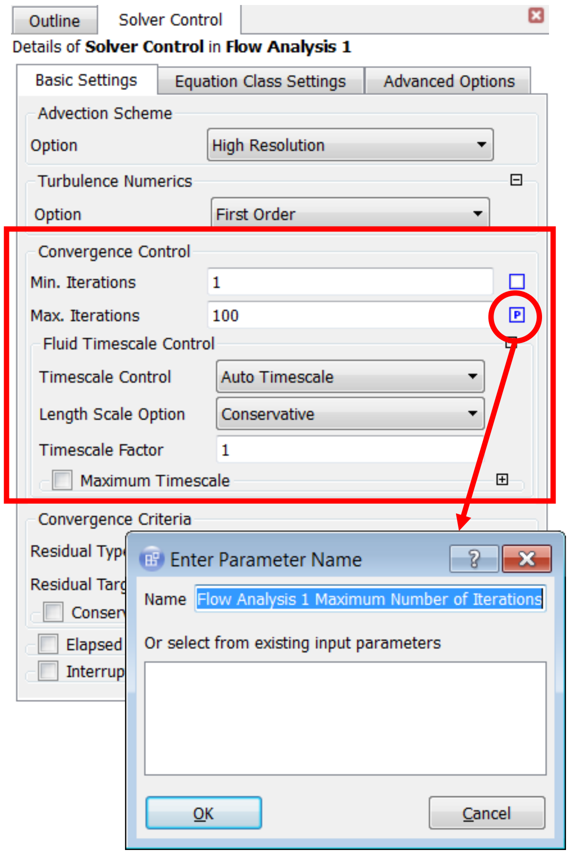
High Resolution Scheme



Introduction | Initialization | Solver | Output File | Summary

Convergence Control

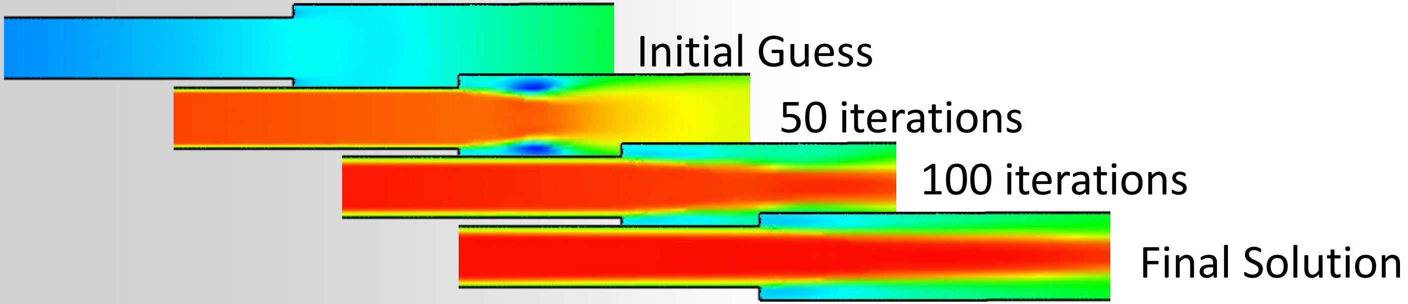
- **The Solver finishes when it reaches Max. Iterations or convergence**
  - **If Max. Iterations is reached, you may not have a converged solution**
  - **Min. and Max. Iterations can be Workbench input parameters**
- **When the Solver finishes you should always check *why* it finished**
- **Fluid Timescale Control sets the timescale in a steady-state simulation**



Introduction | Initialization | Solver | Output File | Summary

Timescale Background

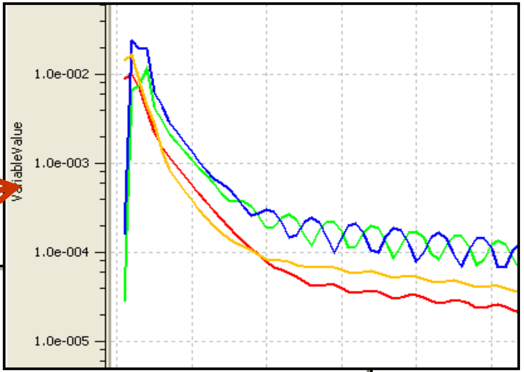
- **ANSYS CFX employs the so-called False Transient in steady-state simulations**
  - **A timescale is used to move the solution towards the final answer**
- **The timescale provides relaxation of equation non-linearities**
- **A steady-state simulation is a 'transient' evolution of the flow from the initial guess to the steady-state conditions**
  - **Converged solution is independent of the timescale used**

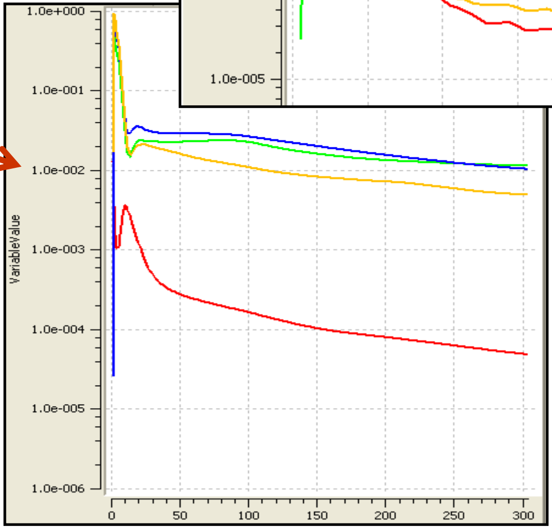


  March 13, 2015

Timescale Selection

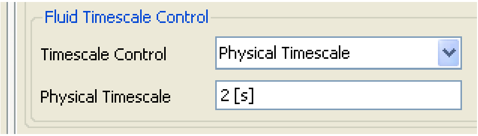
- **For obtaining good convergence, the selection of the timescale plays an important role**
  - Timescale: too large → convergence becomes bouncy or may even lead to the failure
  - Timescale: too small → convergence will be very slow
- **1/3 of (Length Scale / Velocity Scale) is often optimal**
- **Rotating machines: use 0.1/ω to 1/ω (ω in rad/s)**
- **Free Convection:** $\Delta t_{\max} \approx \sqrt{\dfrac{dl}{\beta g \Delta T}}$



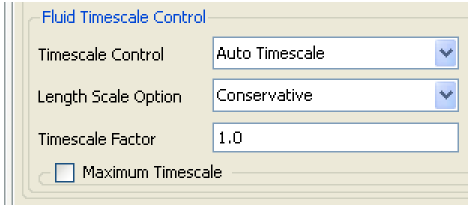


Timescale Control

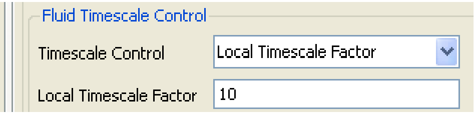
- Physical Timescale
  - constant value or expression
  - Often better than Auto Timescale → faster convergence



- Auto Timescale
  - Solver calculates a timescale based on boundary / initial conditions or current solution and domain length scale
  - Use a *Conservative* or *Aggressive* estimate for the domain length scale, or a specified value
  - Timescale is re-calculated as the flow field changes
  - Set *Maximum Timescale* to provide an upper limit
  - Timescale factor (default = 1) is a multiplier which can be changed to adjust the automatically calculated timescale

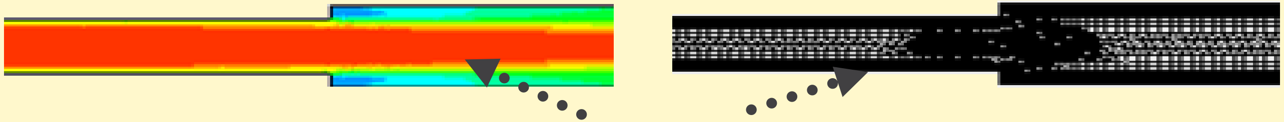


Timescale Control



- **Local Timescale Factor**
  - **Timescale varies throughout the domain**

$$\textit{Local Timescale} = \frac{\textit{Local Mesh Length Scale}}{\textit{Local Velocity Scale}}$$

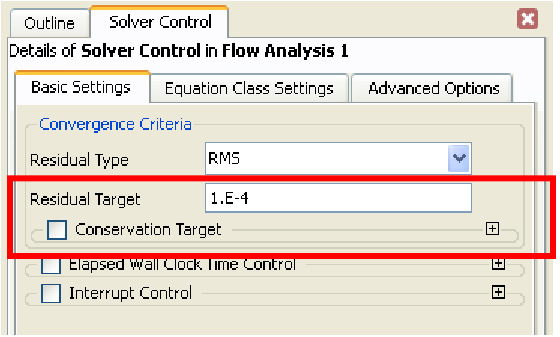


Smaller Timescale in high velocity and/or fine mesh regions

  - **Can accelerate convergence when vastly different local velocity scales exist**
    - **e.g. a jet entering a plenum**
  - **Best used on fairly uniform meshes**
  - **Never use as final solution; always finish off with a constant timescale**

Convergence Criteria

- **Convergence Criteria determine when the solution is considered converged and so when the Solver stops**
  - **Assuming *Max. Iterations* is not reached**
- **Residuals are a measure of how accurately the set of equations have been solved**
  - **Solver iterates towards a solution → never reaches exact solution**
  - **Lower residuals = more accurate solution**



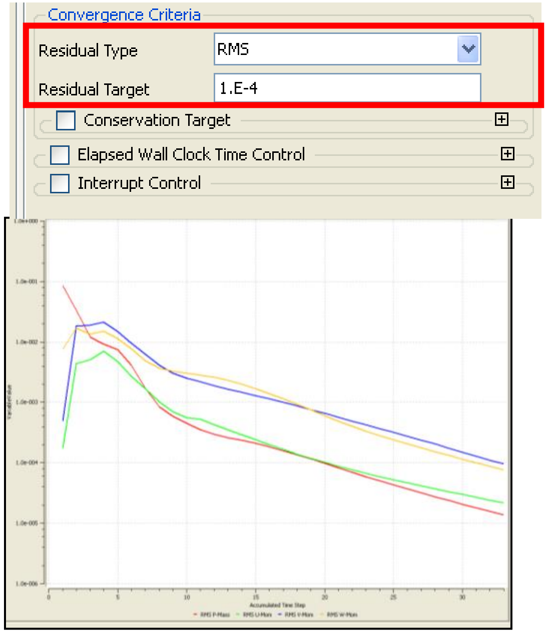
- Equations solved exactly: [A] [Φ] - [b] = [0]
- Iteratively solved: [A] [Φ] - [b] = [R]

Residual vector [R]= error in the numerical solution

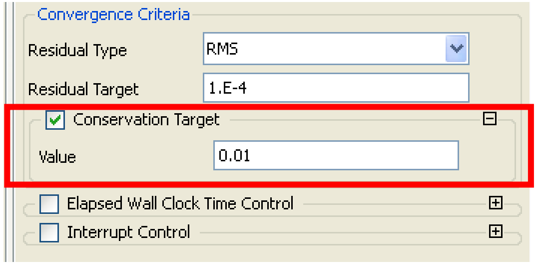
- **Do not confuse accurately solving the equations with overall solution accuracy – the equations may or may not be a good representation of the true system!**

Residuals

- **Residual Type**
  - **MAX: maximum residual anywhere**
  - **RMS: gives a value typical for the whole model**
    - **Root Mean Square (RMS) =** $\sqrt{\frac{\sum_i R_i^2}{n}}$
- **Residual Target**
  - **dependent on the accuracy needed**
    - **MAX residuals < 1.0E-3**
    - **RMS residuals < 1.0E-4, 1.0E-5 or 1.0E-6**



Conservation Target



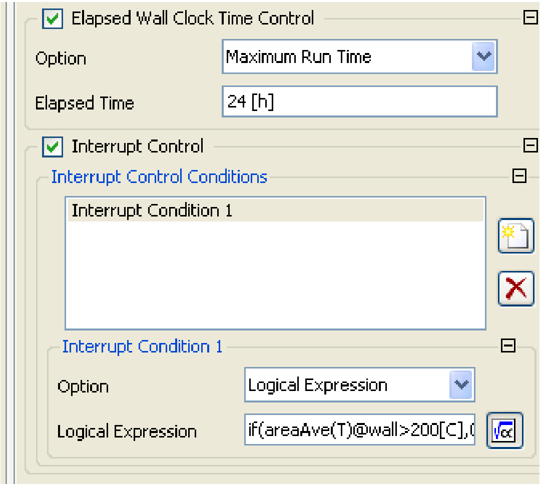
- **Sets a target for the global imbalances**

$$\% \text{ Imbalance} = \frac{\text{Flow Rate In} - \text{Flow Rate Out}}{\text{Maximum Flow Rate}}$$

- **The imbalances measure the overall conservation of a quantity (mass, momentum, energy) in the entire flow domain**
- **Clearly in a converged solution Flow Rate In = Flow Rate Out**
- **It's good practice to set a *Conservation Target* and/or monitor the imbalances**
- **Set a target of 0.01 (1%) or less**
  - **For some applications < 0.01%**
  - **Solver must meet both the *Residual* and *Conservation Target* before stopping**

Elapsed Time and Interrupt Control

- **Elapsed Time Control**
  - Specify the maximum wall clock time for run
  - Solver will stop after this period regardless of whether it has converged

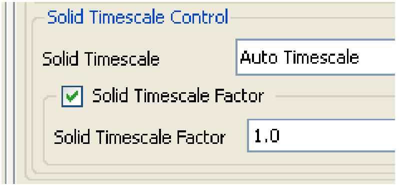


- **Interrupt Control**
  - Criteria based on logical CEL expressions
  - expression returns *true* → solver stops
    - Any value >= 0.5 is true
  - Examples:
    - If temperature exceeds a specified value
      - areaAve(T)@wall>200 [C]
    - If mesh quality drops below a specified value in a moving mesh case

Introduction | Initialization | Solver | Output File | Summary

Solid Timescale Control

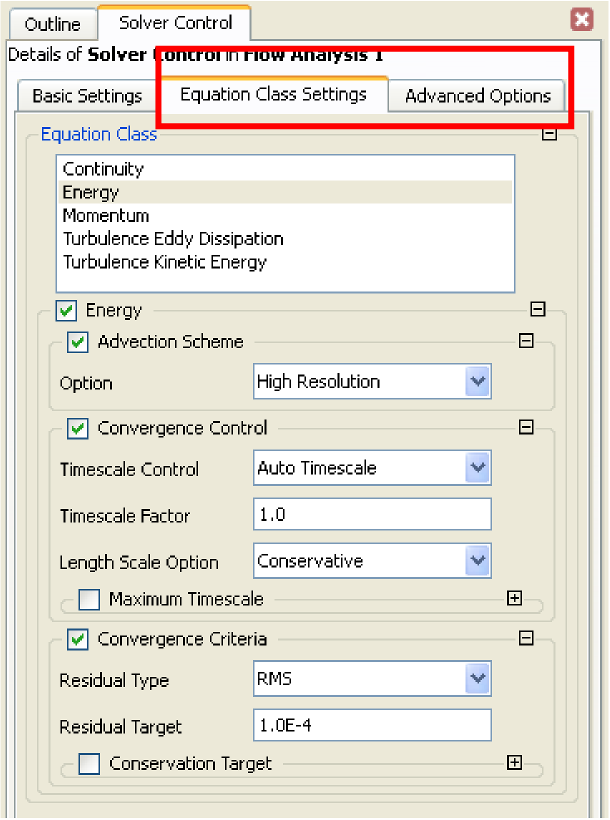
- Only available when a solid domain is included in the simulation
- Solid Timescale should be MUCH larger than the fluid timescale
  - 100 times larger is typical
  - energy equation is usually very stable in the solid



- The fluid timescale is estimated using Length Scale / Velocity Scale
- Auto Timescale calculates solid timescale as a function of the length scale, thermal conductivity, density and specific heat capacity
  - Or you can choose the Physical Timescale option and provide a timescale directly

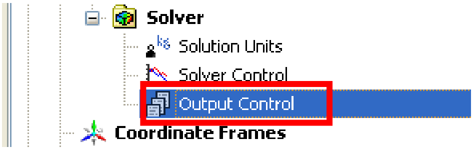
Equation Class Settings

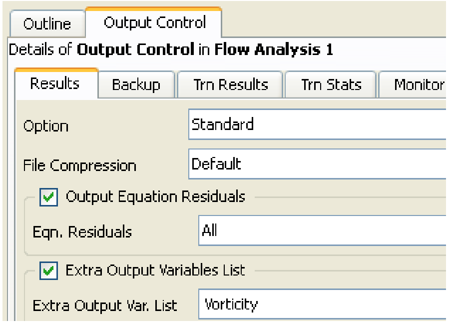
- The Equation Class Settings tab is an advanced option that can be used to set Solver controls on an equation-specific basis
  - Not usually needed
  - Will override the controls set on Basic Settings for the selected equation
- Advanced Options
  - Advanced solver control options
  - Rarely needed



Output Control

- **Control the output produced by the Solver**
- **Results tab controls the final .res file**
  - ***Selected Variables* probably insufficient information for a restart**
  - ***Output Equation Residuals* set to all and check where convergence problems occur**
  - ***Extra Output Variables List* variables not written to standard results file e.g. vorticity**

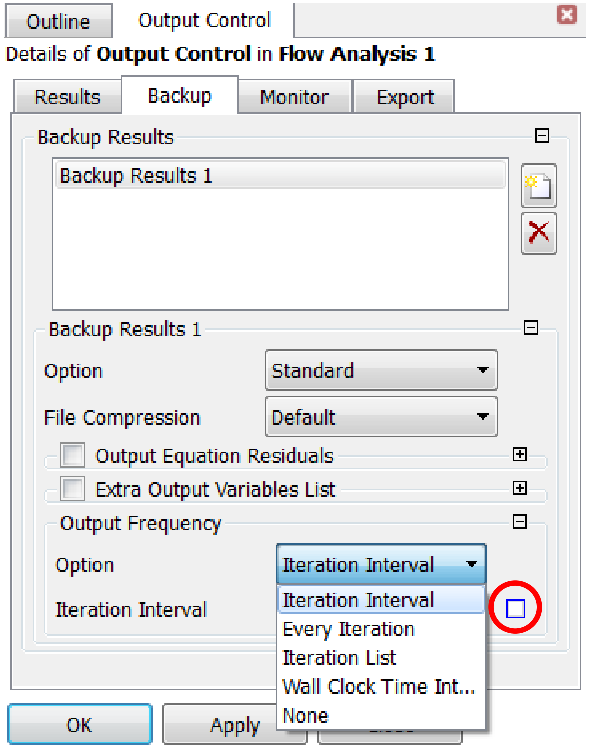




Introduction | Initialization | Solver | Output File | Summary

Output Control

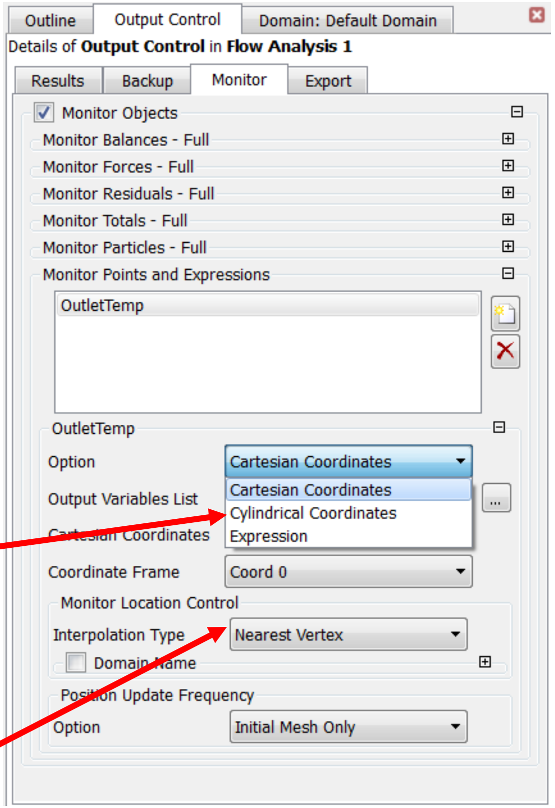
- Backup controls if/when backup files are written
  - Recommended in case of power failure, network interruptions, etc.
  - Essential: Allows a clean solver restart
  - Smallest: Can restart the solver, with a residuals jump
  - If Output Frequency is set to Iteration Interval, it can be a Workbench input parameter



Introduction | Initialization | Solver | Output File | Summary

Output Control – Monitor

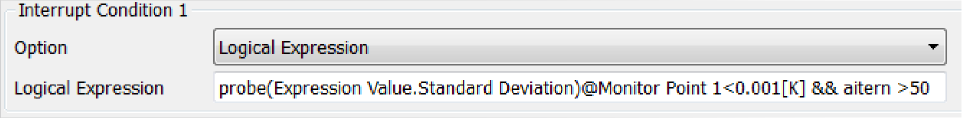
- Allows you to create the Monitor Points
  - Track values of interest as the Solver runs
- In steady-state simulations you should create monitor points for quantities of interest
  - One measure of convergence is when these values are no longer changing
- Can track variable value at location defined by Cartesian or Cylindrical Coordinates or monitor a CEL Expression
  - For Coordinates options, data can be interpolated to Nearest Vertex or use weighted-average of vertex values for containing element (Trilinear)



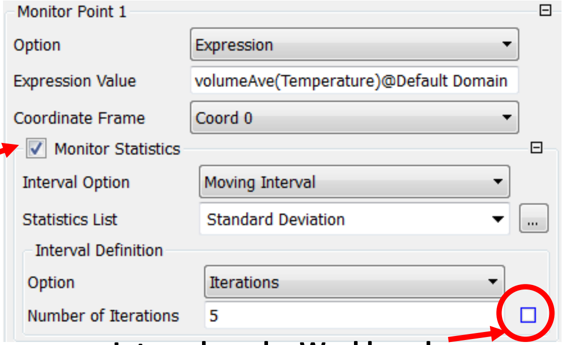
Introduction | Initialization | Solver | Output File | Summary

Output Control – Monitor an Expression

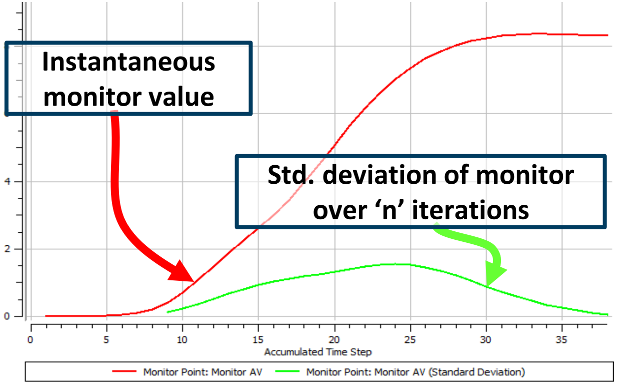
- Expression Option to monitor a CEL expression
- Additional option for expression to Monitor Statistics
- Can probe the statistics for an Interrupt Condition (Solver Control) to stop run once the value of a target quantity:
  - remains below a threshold value for a given period
  - no longer changes significantly



  - Useful for automation

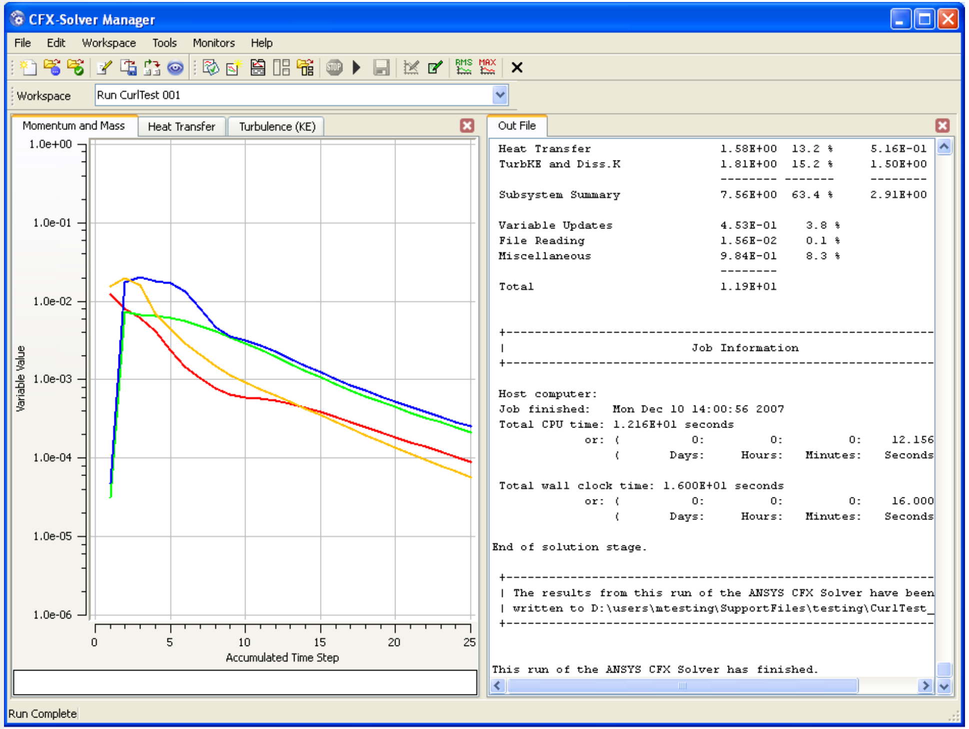


Interval can be Workbench input parameter



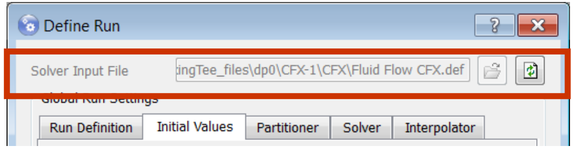
Solver Manager

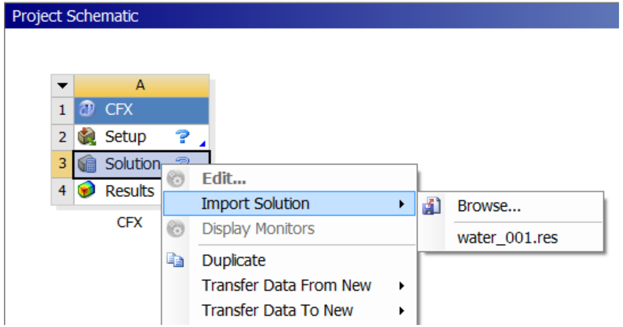
- **The CFX-Solver Manager is a graphical user interface used to:**
  - **Define a run**
  - **Control the CFX-Solver interactively**
  - **View information about the emerging solution**
  - **Export data**



Defining a Run

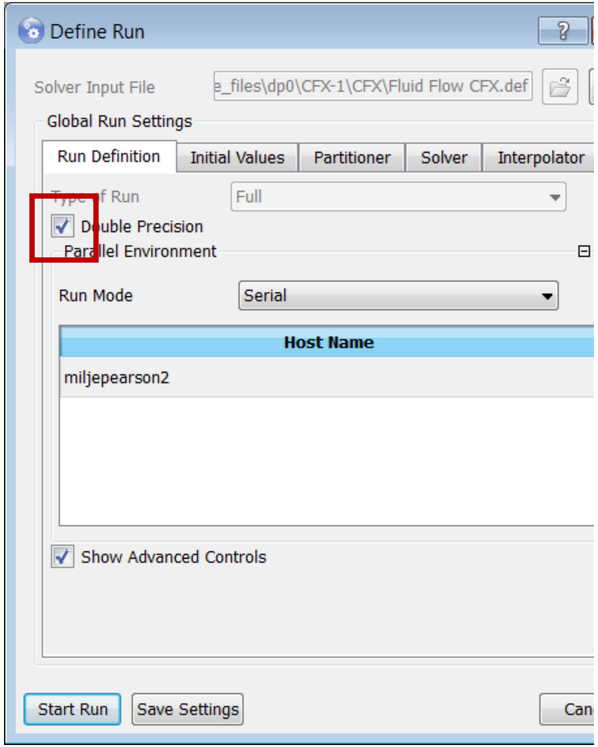
- Solver Input File is usually the .def file (automatically set in WB)
- Can import a .res, .bak or _full.trn files into a new CFX system to restart a previous incomplete run
- To restart with changed physics, create a new .def file and initialise from previous solution as shown earlier





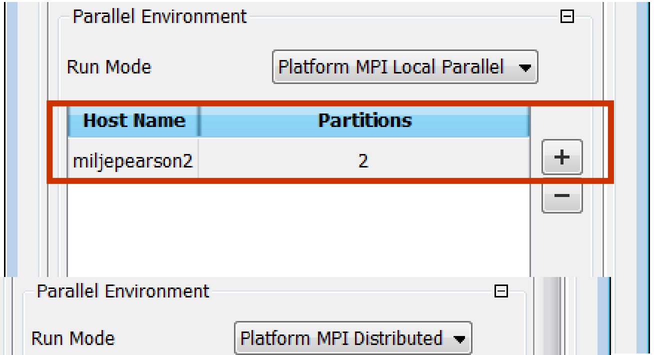
Defining a Run

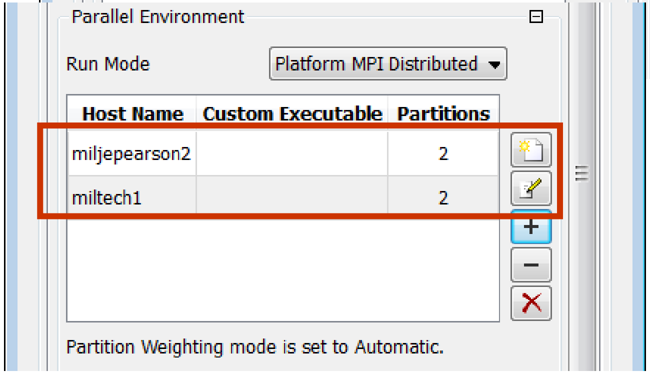
- **Define a new Solver run (contd…)**
  - **Double Precision: will use more significant figures in its calculations**
    - **Use when round-off error could be a problem – if ‘small’ variations in a variable are important, where ‘small’ is relative to the global range of that variable, e.g.:**
      - **Many Mesh Motion cases, since the motion is often small relative to the size of the domain**
      - **If you have a wide pressure range, but small pressure changes are important**
    - **Doubles solver memory requirements**
    - **Small values by themselves do not need double precision**



Defining a Parallel Run

- By default Solver will run in serial
  - Single solver process runs on a local machine
- Set the Run Mode to a parallel option to use of multiple cores/processors
  - Requires parallel licenses
  - Allows you to divide a large CFD problem into smaller partitions
    - Faster solution times & solve larger problems by making use of memory (RAM) on multiple machines
- Local Parallel = running on a single machine

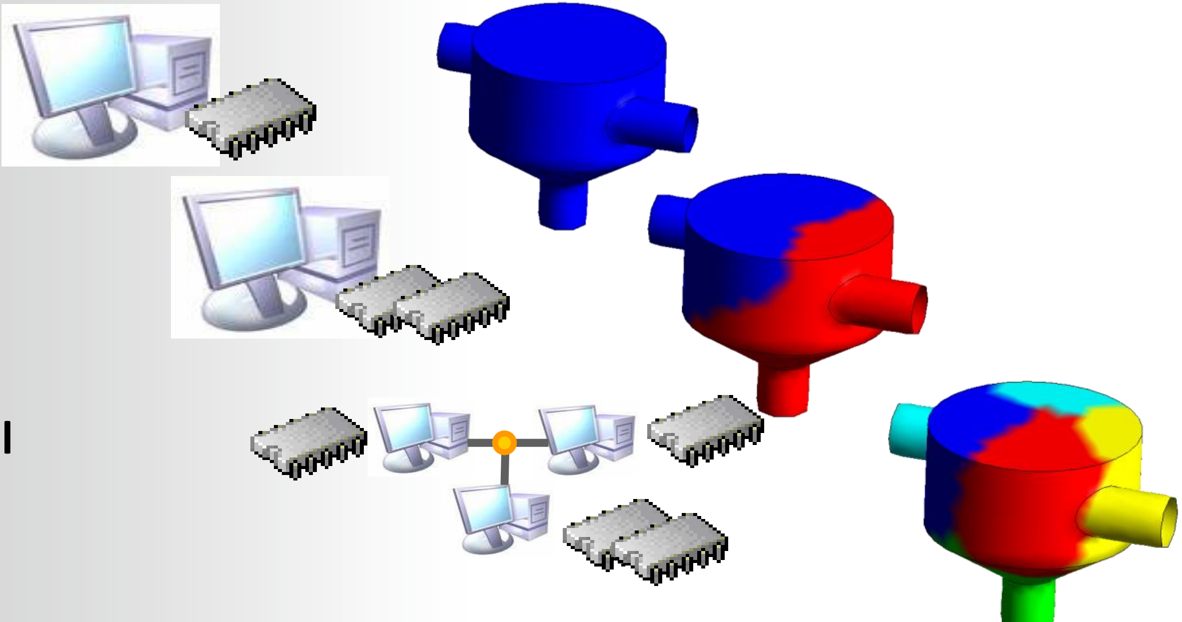




Introduction | Initialization | Solver | Output File | Summary

Defining a Parallel Run

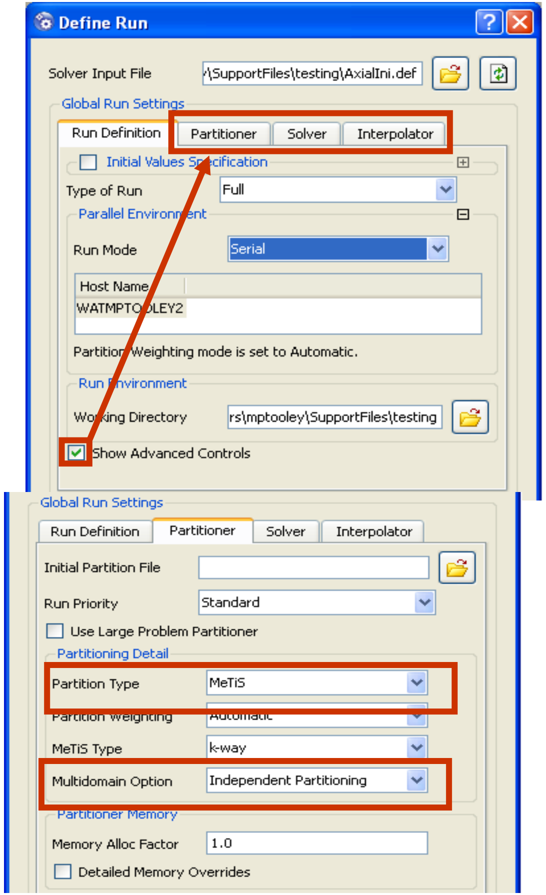
- Serial
- Local Parallel
- Distributed Parallel



- **Different communication methods are available, Platform MPI is recommended in most cases**

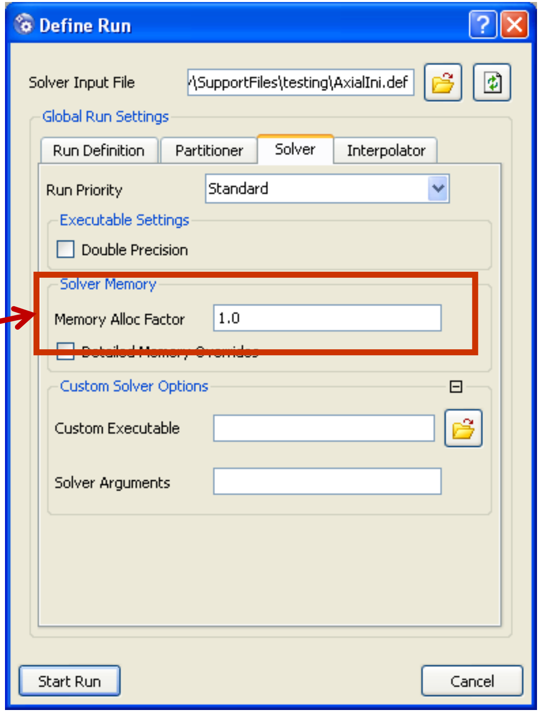
Advanced Controls

- ‘Show Advanced Controls’ toggle enables the Partitioner, Solver and Interpolator tabs
- ‘Partitioner’ tab
  - Partitioning is always a serial process
  - Can be a problem for very large cases
    - MeTiS (default) uses more memory than others. Change method to reduce memory required (see documentation)
    - Use Large Problem Partitioner (64-bit only)
  - Multidomain Option:
    - Independent Partitioning: Each domain is partitioned into ‘n’ partitions
    - Coupled Partitioning: Domains are combined and then partitioned into ‘n’ partitions



Advanced Controls

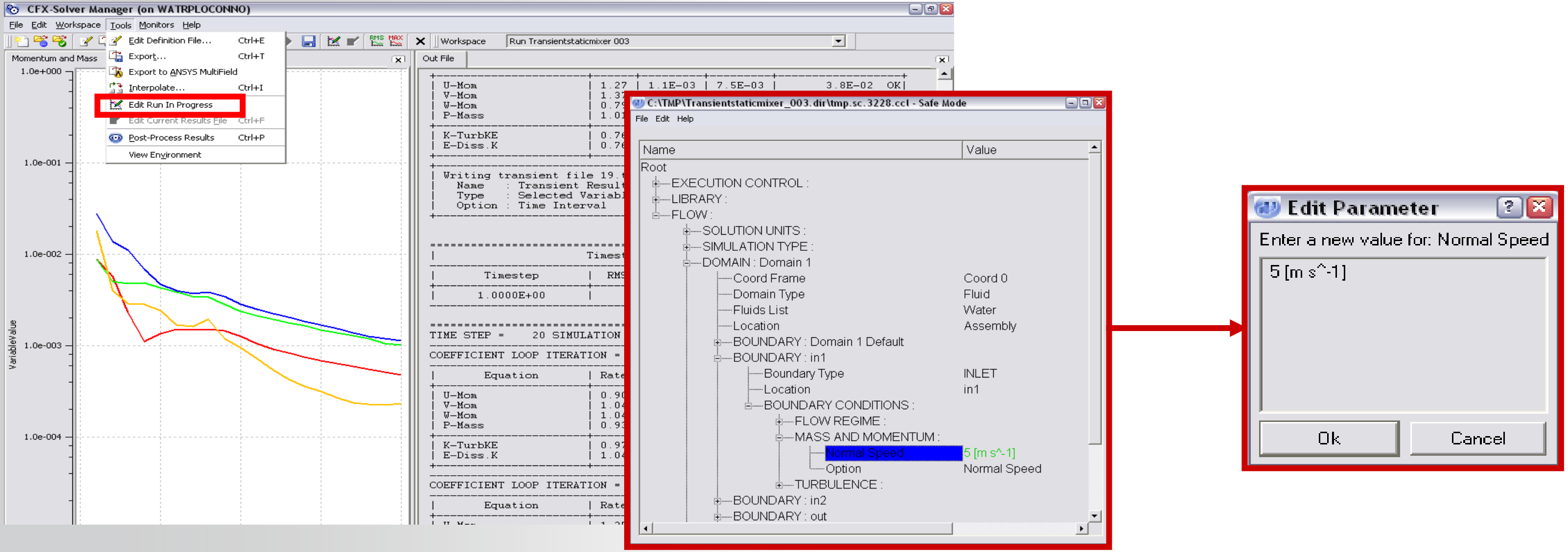
- On the Solver tab you adjust Solver Memory settings
  - The Solver estimates its memory requirements
  - Memory Alloc Factor is a multiplier for this estimate
    - Use when the solver stops with an ‘Insufficient Memory Allocated’ error
    - Can provide individual factors for each stack



Introduction | Initialization | **Solver** | Output File | Summary

Interactive Solver Control

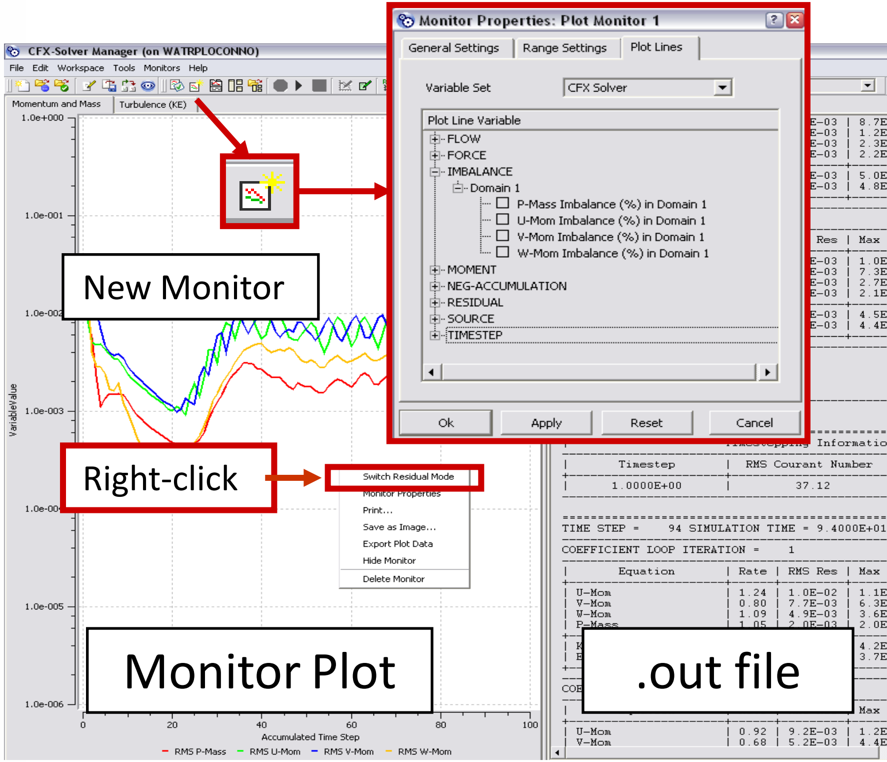
- During a solution, 'Edit Run in Progress' lets you make changes on the fly
  - Models generally cannot be changed, but numeric values can



Introduction | Initialization | Solver | Output File | Summary

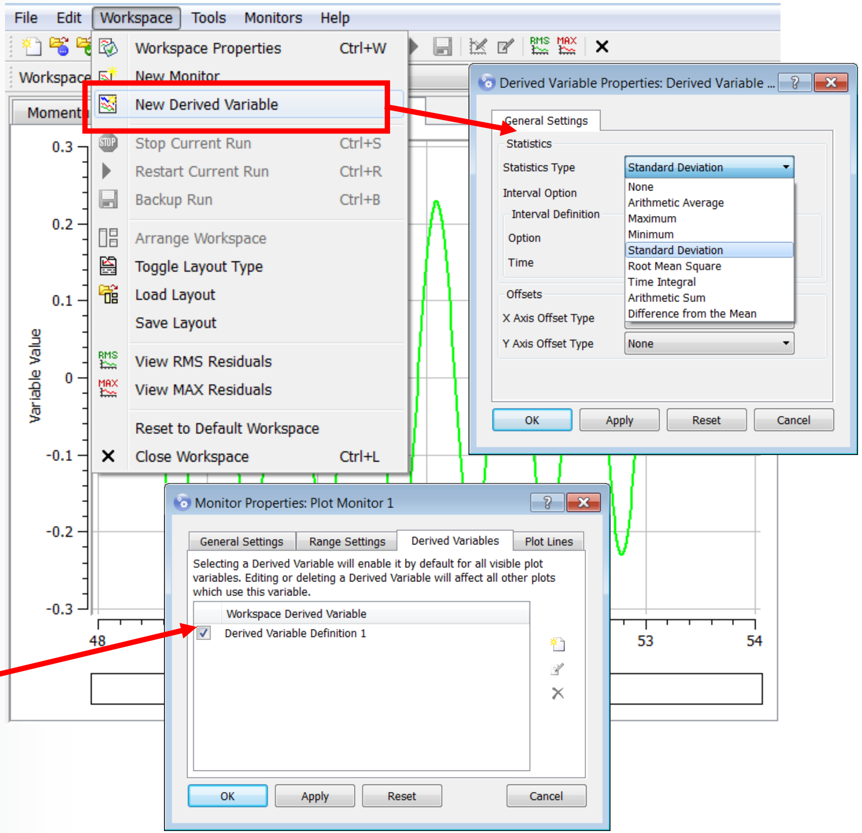
Additional Solution Monitors

- **By default, monitor plots are created showing the RMS residuals for each equation solved, plus one plot for any monitor points**
- **Right-click to switch between RMS and MAX**
- **Additional monitors:**
  - **Imbalances**
  - **Boundary fluxes (FLOW)**
  - **Boundary forces**
    - **Tangential (viscous)**
    - **Normal (pressure)**
  - **Source terms**



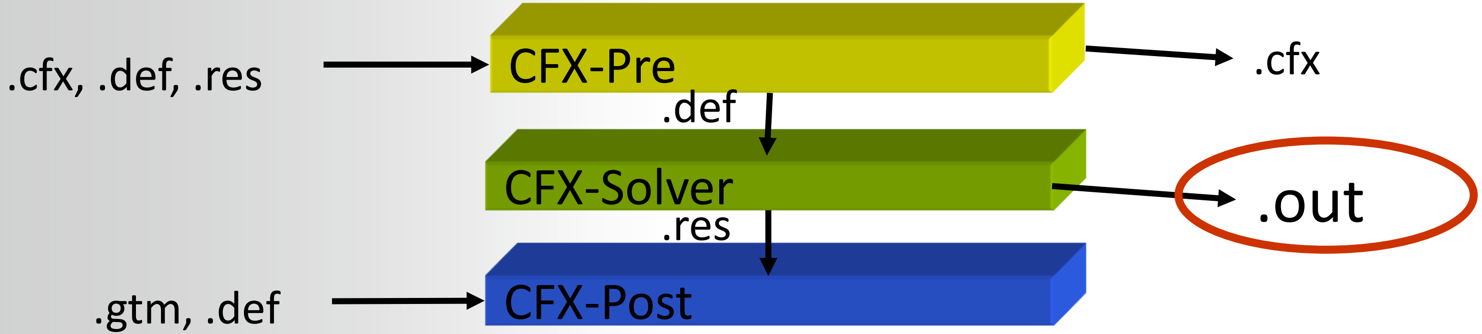
Monitoring Derived Variables

- Visualise trends using Derived Variables
  - Statistics (similar to option for monitored Expressions)
  - Offset from current plot
- Not written to results file
- To create
  - Workspace > New Derived Variable
- To activate
  - Workspace Properties > Derived Variables



Solver Output File

- Produced by the ANSYS CFX-Solver and contains information about simulation:
  - Model setup
  - The state of the solution during the run
  - Job statistics for the particular run



- Now lets take a detailed look at an out file

Model Setup

```
This run of the CFX-14.5 Solver started at 16:53:52 on 04 Dec 2012 by
user jepearso on MILJEPEARSON2 (intel_xeon64.sse2_winnt) using the
command:

 "C:\Program Files\ANSYS Inc\v145\CFX\bin\perllib\cfx5solve.pl"
   -stdout-comms -batch -ccl -

Setting up CFX Solver run ...

+--------------------------------------------------------------------+
|
|                    CFX Command Language fo
|
+-------------------------------------------

LIBRARY:
  CEL:
    EXPRESSIONS:
      PDiff = PInlet -POutlet
      PInlet = massFlowAve(Total Pressure )@R1 Inlet
      POutlet = massFlowAve(Total Pressure )@S1 Outlet
      myPhysicalTimescale = 1 [rad]/myRotationSpeed
      myRotationSpeed = 2880 [rev min^-1]
      nRotor = 30
      power = (torque_z()@R1 Blade + torque_z()@R1 Hub)*myRotationSpeed \
        /1[rad]*nRotor
    END
  END
  MATERIAL: Air Ideal Gas
```

Solver Information

The out file

Introduction | Initialization | Solver | Output File | Summary

Model Setup

```
This run of the CFX-12.0 Solver sta
user djhyde on WATDJHYDE2 (intel_xe

 "C:\Program Files\Ansys Inc\v120\C
   D:\users\djhyde\support\Mahesh\A

Setting up CFX Solver run ...
```

Physics definition
CFX Command Language (CCL)

```
+--------------------------------------------------------------------+
|                                                                    |
|                     CFX Command Language for Run                   |
|                                                                    |
+--------------------------------------------------------------------+

LIBRARY:
  MATERIAL: Air Ideal Gas
    Material Description = Air Ideal Gas (constant Cp)
    Material Group = Air Data, Calorically Perfect Ideal Gases
    Option = Pure Substance
    Thermodynamic State = Gas
    PROPERTIES:
```

Interpolator

```
+--------------------------------------------------------------------+
|                                                                    |
|                 Interpolation of Initial Values                    |
|                                                                    |
+--------------------------------------------------------------------+
```

…

…

```
+--------------------------------------------------------------------+
|       Memory Allocated for Run  (Actual usage may be less)         |
+--------------------------------------------------------------------+

 Data Type          Kwords   Words/Node   Words/Elem       Kbytes  Bytes/Node

 Real               2040.1        27.15        12.04       7969.0      108.62
 Integer            2915.8        38.81        17.20      11389.7      155.24
 Character           200.0         2.66         1.18        195.3        2.66
 Logical              10.0         0.13         0.06         39.1        0.53
 Double              115.1         1.53         0.68        899.1       12.25

======================================================================
                    Interpolating Onto Domain "R1"
======================================================================

Total Number of Nodes in the Target Domain                =         38360
Bounding Box Volume of the Target Mesh                    =   8.59948E-04


Checking all source domains from the source file:
  Target mesh is the same as domain "R1".

Start direct copying of variables from domain "R1".
```

If an Initial Values file is provided, the Interpolator will run to copy / interpolate variables

Memory requirements shown here are for the interpolator only

Details of the interpolation process are shown

Partitioner

```
+------------------------------------------------------------------+
|                                                                  |
|                          Partitioning                            |
|                                                                  |
+------------------------------------------------------------------+
```

…

…

```
+------------------------------------------------------------------+
|        Memory Allocated for Run  (Actual usage may be less)      |
+------------------------------------------------------------------+

Data Type          Kwords   Words/Node   Words/Elem      Kbytes  Bytes/Node

Real               2128.7        28.33        12.56      8315.0      113.33
Integer            4911.7        65.38        28.98     19186.4      261.51
Character          3111.1        41.41        18.35      3038.2       41.41
Logical              65.0         0.87         0.38       253.9        3.46
Double             1200.5        15.98         7.08      9378.9      127.84
```

…

…

```
+------------------------------------------------------------------+
|                    Vertex Based Partitioning                     |
+------------------------------------------------------------------+

Partitioning of domain: R1

- Partitioning tool:          MeTiS multilevel k-way algorithm
- Number of partitions:                 2
- Number of graph-nodes:            37520
- Number of graph-edges:           216288

Partitioning of domain: S1

- Partitioning tool:          MeTiS multilevel k-way algorithm
- Number of partitions:                 2
```

If solving in parallel the Partitioner will run to divide up the simulation into multiple partitions

Memory requirements shown here are for the partitioner only

Details of the partitioning process are shown

  March 13, 2015

Solver

```
+--------------------------------------------------------------------+
|                                                                    |
|                              Solver                                |
|                                                                    |
+--------------------------------------------------------------------+
```

…

…

```
+--------------------------------------------------------------------+
| Memory Allocated for Run  (Actual usage may be less)               |
+--------------------------------------------------------------------+

 Data Type        Kwords   Words/Node  Words/Elem     Kbytes Bytes/Node

 Real            34643.8      461.13      204.39   135327.5    1844.52
 Integer         10024.9      133.44       59.14    39159.8  [illegible]
 Character        3539.7       47.12       20.88     3456.8  [illegible]
 Logical            65.0        0.87        0.38      253.9       3.46
 Double           1208.0       16.08        7.13     9437.5     128.63
```

…

…

```
+--------------------------------------------------------------------+
|                          Mesh Statistics                           |
+--------------------------------------------------------------------+
| Domain Name           | Orthog. Angle | Exp. Factor  | Aspect Ratio |
+-----------------------+---------------+--------------+--------------+
|                       | Minimum [deg] |   Maximum    |   Maximum    |
+-----------------------+---------------+--------------+--------------+
| R1                    |     30.1  ok  |     42    !  |     383  ok  |
| S1                    |     31.4  ok  |    196    !  |      12  OK  |
| Global                |     30.1  ok  |    196    !  |     383  ok  |
+-----------------------+---------------+--------------+--------------+
|                       |  %!  %ok  %OK | %!  %ok  %OK | %!  %ok  %OK |
+-----------------------+---------------+--------------+--------------+
```

The start of the Solver process

Memory requirements shown here are for the Solver. If memory errors occur check if it's the Interpolator, Partitioner or Solver process

Mesh quality checks, mesh statistics and GGI intersection details are shown here

State of the Solution

Diagnostics shown as the solver iterates towards a solution

```
=======================================================================
OUTER LOOP ITERATION =    31                      CPU SECONDS = 2.709E+02
-----------------------------------------------------------------------
|       Equation       | Rate | RMS Res | Max Res |  Linear Solution |
+----------------------+------+---------+---------+------------------+
| U-Mom                | 0.91 | 2.7E-05 | 8.1E-04 |       2.1E-02  OK|
| V-Mom                | 0.85 | 9.5E-05 | 4.1E-03 |       2.3E-02  OK|
| W-Mom                | 0.87 | 8.1E-05 | 5.1E-03 |       2.3E-02  OK|
| P-Mass               | 0.87 | 1.7E-05 | 3.3E-04 |  9.5  5.3E-02  OK|
+----------------------+------+---------+---------+------------------+
| H-Energy             | 0.85 | 8.9E-05 | 3.4E-03 |  5.7  1.1E-02  OK|
+----------------------+------+---------+---------+------------------+
| K-TurbKE             | 0.86 | 7.4E-05 | 2.5E-03 |  5.7  4.9E-03  OK|
| E-Diss.K             | 0.86 | 1.2E-04 | 3.7E-03 |  8.0  2.5E-03  OK|
+----------------------+------+---------+---------+------------------+
```

$$Rate = \frac{R_n}{R_{n-1}}$$

"OK" = good , "ok" = on the limit, "F" = failed to solve the equation. Reducing the timestep may help avoid "ok" and "F". "ok" and "F" can be ignored if they go away

Introduction | Initialization | Solver | **Output File** | Summary

Final Imbalances

When the solution finishes, the Imbalances are shown

Normalised Imbalance Summary		
Equation	**Maximum Flow**	**Imbalance (%)**
U-Mom	7.4643E+02	-0.0013
V-Mom	7.4643E+02	0.0135
W-Mom	7.4643E+02	0.0064
P-Mass	1.1330E+02	0.0005
H-Energy	6.4831E+06	-0.1837

Other Solution Data

- Other data printed at the end of the out file include Viscous and Pressure forces and torques on walls, min/max variable values and detailed CPU requirements

```
+-------------------------------------------------------------------+
|                    Pressure Force On Walls                        |
+-------------------------------------------------------------------
                                        X-Comp.        Y

Domain Group: R1

R1 Blade                             1.6108E+00    1.0
R1 Hub                               4.0476E+01    4.7
R1 Shroud                           -4.8755E+01   -5.7
S1 Blade                            -3.8723E-01   -1.5
S1 Hub                               8.9493E+00    6.8
S1 Shroud                           -8.7473E+00   -4.7
                                     -----------   ---
Domain Group Totals :               -6.8535E+00   -5.1
```

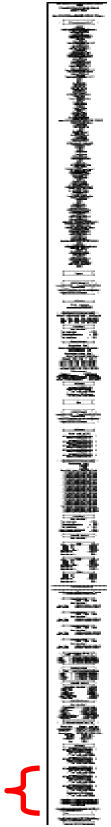
```
+-------------------------------------------------------------------
|                    Variable Range Information
+-------------------------------------------------------------------

Domain Name : R1
+-------------------------------------------------------------------+
|       Variable Name                          |    min     |    max    |
+-------------------------------------------------------------------+
| Density                                     |  1.44E-01  |  2.00E-01 |
| Specific Heat Capacity at Constant Pressure|  1.00E+03  |  1.00E+03 |
| Dynamic Viscosity                           |  1.83E-05  |  1.83E-05 |
```

```
+-----------------------------------------------------------------------+
|          CPU Requirements of Numerical Solution - Total              |
+-----------------------------------------------------------------------+

Subsystem Name                 Discretization        Linear Solution
                               (secs.    %total)     (secs.    %total)
-----------------------------------------------------------------------
Momentum and Mass              5.59E+00   22.6 %     1.91E+00    7.7 %
Heat Transfer                  1.72E+00    7.0 %     3.59E-01    1.5 %
TurbKE and Diss.K              1.55E+00    6.3 %     7.19E-01    2.9 %
                               --------  -------     --------  ------
Subsystem Summary              8.86E+00   35.9 %     2.98E+00   12.1 %

Variable Updates               1.08E+00    4.4 %
GGI Intersection               3.12E-02    0.1 %
File Reading                   1.12E+00    4.6 %
File Writing                   3.02E+00   12.2 %
Miscellaneous                  7.61E+00   30.8 %
                               --------
Total                          2.47E+01
```



Introduction | Initialization | Solver | Output File | Summary

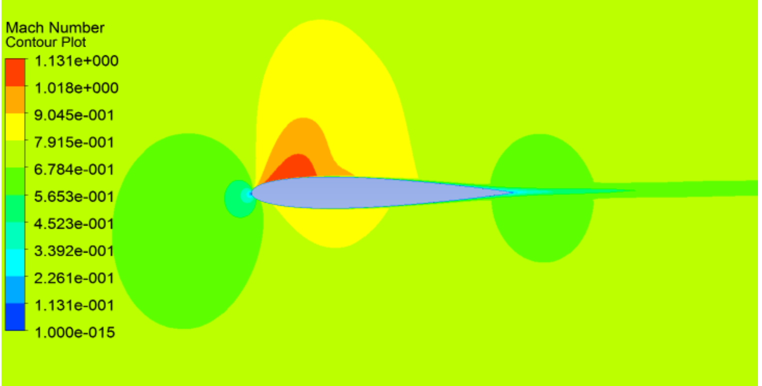
ANSYS

Summary

- This section has covered the following important points:
- Initialization is key to providing a stable solution, especially for complex physics or flows with high solution gradients
- Convergence is one important part of judging solution progress.
  - Don't forget that the solver will terminate when the 'Max Iterations' has been reached, regardless of solution convergence levels
- Output Control:
  - Always monitor imbalances to ensure conservation
  - Use backups so that data can be rescued if the solver/hardware fails
  - Monitor quantities of interest as an additional aid when judging steady-state behaviour
- The output file gives important information about the physics, resource usage and solution progress

Workshop 03 Airfoil

- Flow around a NACA0012 airfoil
  - Assessing Y+ for correct turbulence model behavior
  - Modifying solver settings to improve accuracy
  - Reading in and plotting experimental data alongside CFD results
  - Producing a side-by-side comparison of different CFD results
- This workshop includes a best practice study. If you do not complete it today, there will be time on day 2.



Introduction | Initialization | Solver | Output File | Summary