

# Computational Fluid Dynamics (CFD)

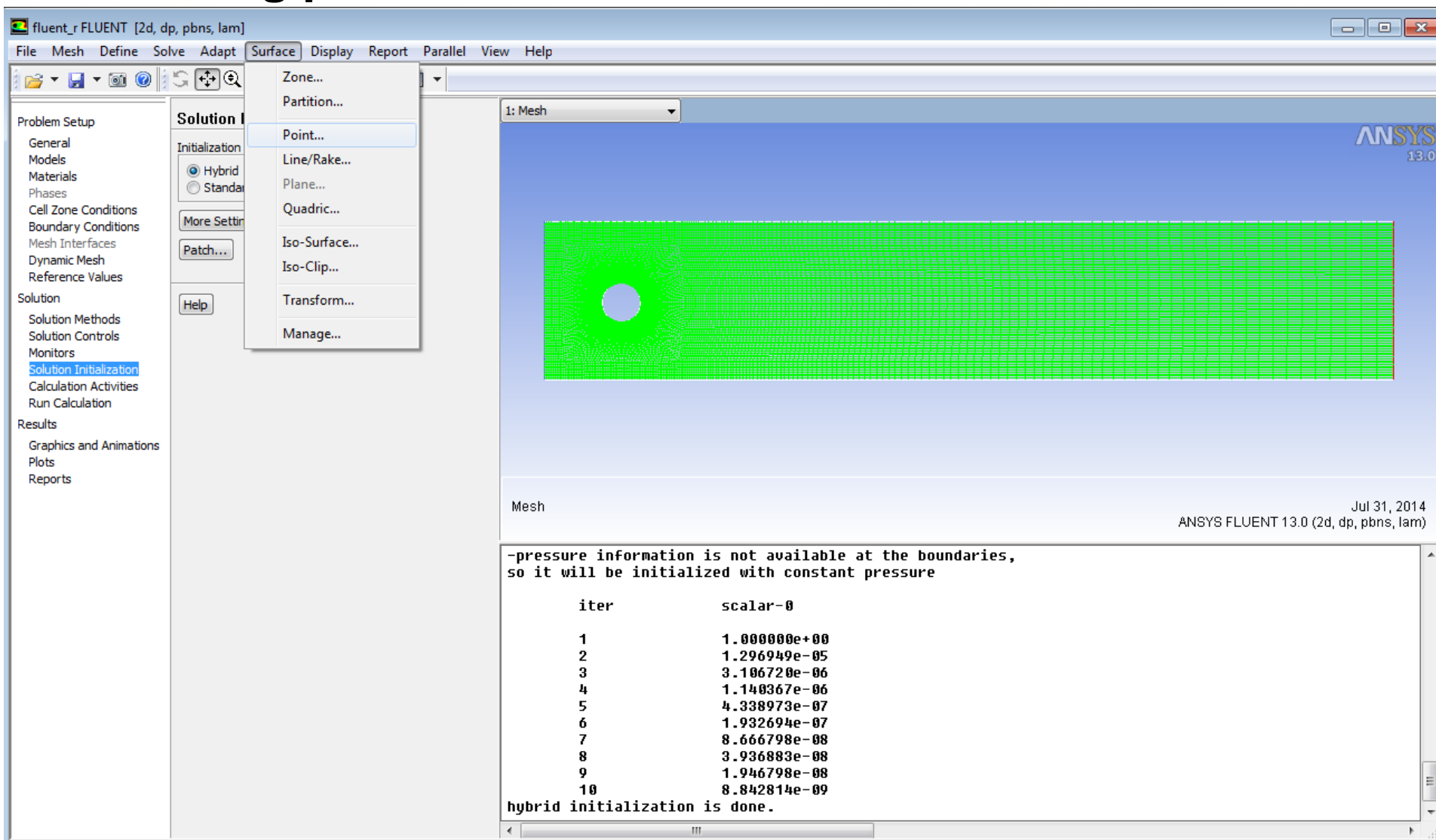
## - Exercises VIII -

Jörg Franke

Master Program Computational Engineering  
Vietnamese-German University (VGU)  
Ho Chi Minh City, Viet Nam

April 2018

# Defining points in FLUENT



fluent\_r FLUENT [2d, dp, pbns, lam]

File Mesh Define Solve Adapt Surface Display Report Parallel View Help

Problem 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

Solution Initialization

Initialization

☒ Hybrid

☐ Standard

More Settings

Patch...

Help

Surface

- Zone...
- Partition...
- Point...**
- Line/Rake...
- Plane...
- Quadric...
- Iso-Surface...
- Iso-Clip...
- Transform...
- Manage...

1: Mesh

ANSYS 13.0

Mesh

Jul 31, 2014  
ANSYS FLUENT 13.0 (2d, dp, pbns, lam)

-pressure information is not available at the boundaries,  
so it will be initialized with constant pressure

iter	scalar-0
1	1.000000e+00
2	1.296949e-05
3	3.106720e-06
4	1.140367e-06
5	4.338973e-07
6	1.932694e-07
7	8.666798e-08
8	3.936883e-08
9	1.946798e-08
10	8.842814e-09

hybrid initialization is done.

# Defining points in FLUENT

The screenshot shows the ANSYS FLUENT 13.0 interface. The main window displays a mesh of a rectangular domain with a circular hole. The mesh is colored green. The left sidebar shows the 'Solution Initialization' tab selected. The 'Point Surface' dialog box is open, showing the 'Options' tab with 'Point Tool' checked and 'Reset' button. The 'Coordinates' section shows x0 (m) = 1.1, y0 (m) = 0.205, and z0 (m) = 0. The 'New Surface Name' field contains 'point-velocities-1'. The 'Create' button is highlighted. A text box with arrows points to the 'Coordinates' and 'New Surface Name' fields, containing the text: 'Enter coordinates (in m!) Give a good name'.

Problem 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

**Solution Initialization**

Initialization Methods

- ☒ Hybrid Initialization
- ☐ Standard Initialization

More Settings... Initialize

Patch...

**Point Surface**

Options

- ☒ Point Tool
- Reset

Coordinates

x0 (m) 1.1

y0 (m) 0.205

z0 (m) 0

Select Point with Mouse

New Surface Name

point-velocities-1

Create Manage... Close Help

Enter coordinates (in m!)  
Give a good name

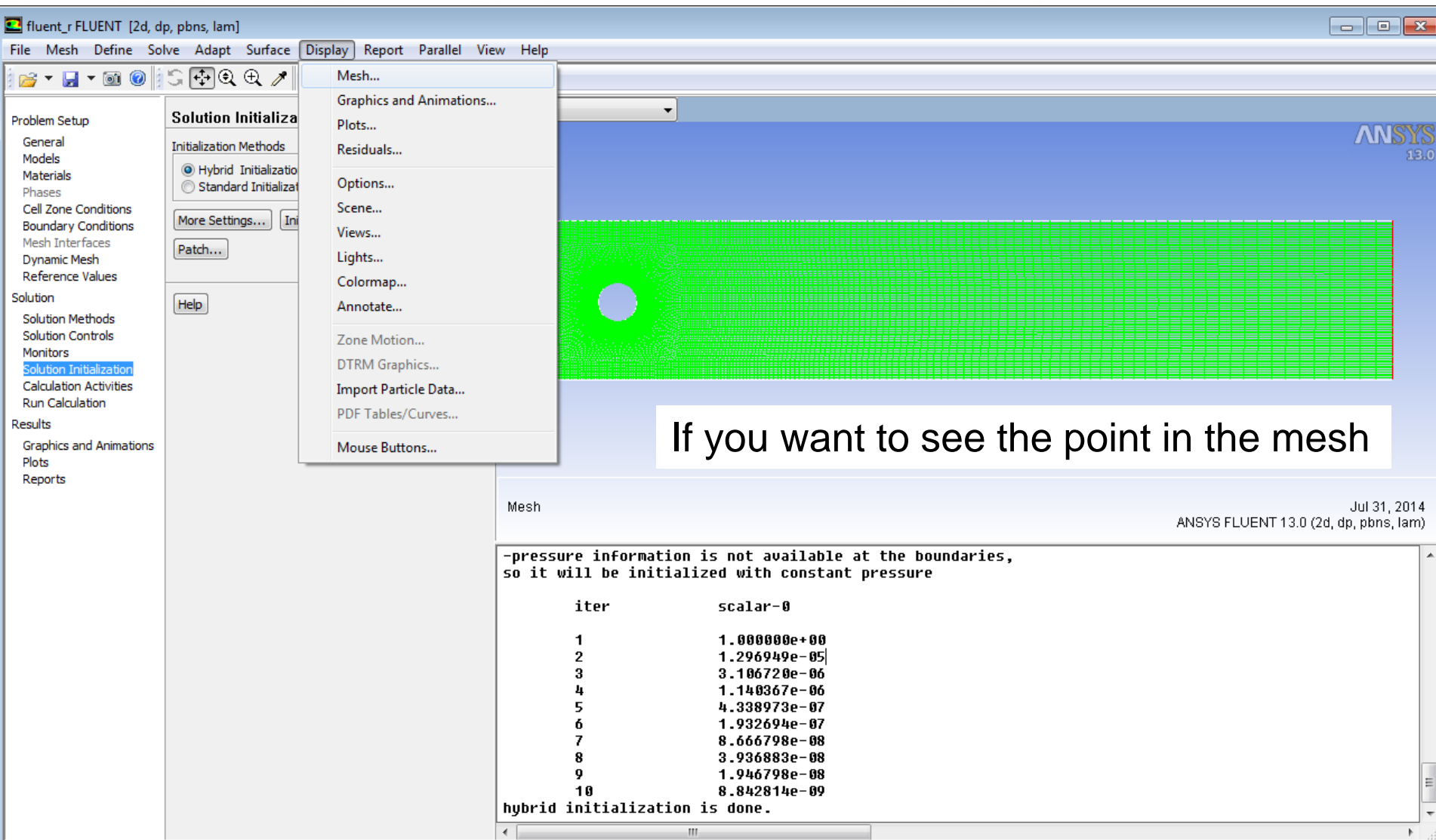
Jul 31, 2014  
ANSYS FLUENT 13.0 (2d, dp, pbns, lam)

-pressure information is not available at the boundaries,  
so it will be initialized with constant pressure

iter	scalar-0
1	1.000000e+00
2	1.296949e-05
3	3.106720e-06
4	1.140367e-06
5	4.338973e-07
6	1.932694e-07
7	8.666798e-08
8	3.936883e-08
9	1.946798e-08
10	8.842814e-09

hybrid initialization is done.

# Defining points in FLUENT



fluent\_r FLUENT [2d, dp, pbns, lam]

File Mesh Define Solve Adapt Surface Display Report Parallel View Help

Problem 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

**Solution Initialization**

Initialization Methods  
☒ Hybrid Initialization  
☐ Standard Initialization

More Settings... In...

Patch... Help

Mesh...  
Graphics and Animations...  
Plots...  
Residuals...  
Options...  
Scene...  
Views...  
Lights...  
Colormap...  
Annotate...  
Zone Motion...  
DTRM Graphics...  
Import Particle Data...  
PDF Tables/Curves...  
Mouse Buttons...

If you want to see the point in the mesh

Mesh

Jul 31, 2014  
ANSYS FLUENT 13.0 (2d, dp, pbns, lam)

-pressure information is not available at the boundaries,  
so it will be initialized with constant pressure

iter	scalar-0
1	1.000000e+00
2	1.296949e-05
3	3.106720e-06
4	1.140367e-06
5	4.338973e-07
6	1.932694e-07
7	8.666798e-08
8	3.936883e-08
9	1.946798e-08
10	8.842814e-09

hybrid initialization is done.

# Defining points in FLUENT

fluent\_r FLUENT [2d, dp, pbns, lam]

File Mesh Define Solve Adapt Surface Display Report Parallel View Help

Problem Setup  
General  
Models  
Materials  
Phases  
Cell Zone Conditions  
Boundary Conditions  
Mesh Interfaces  
Dynamic Mesh  
Reference Values

Solution  
Solution Methods  
Solution Controls  
Monitors  
Solution Initialization

**Solution Initialization**

Initialization Methods  
☒ Hybrid Initialization  
☐ Standard Initialization

More Settings... Initialize  
Patch...  
Help

1: Mesh

ANSYS 13.0

If you want to see the point in the mesh

Mesh

Jul 31, 2014  
ANSYS FLUENT 13.0 (2d, dp, pbns, lam)

-pressure information is not available at the boundaries,  
so it will be initialized with constant pressure

iter	scalar-0
1	1.000000e+00
2	1.296949e-05
3	3.106720e-06
4	1.140367e-06
5	4.338973e-07
6	1.932694e-07
7	8.666798e-08
8	3.936883e-08
9	1.946798e-08
10	8.842814e-09

hybrid initialization is done.

Mesh Display

Options  
☐ Nodes  
☒ Edges  
☐ Faces  
☐ Partitions

Edge Type  
☒ All  
☐ Feature  
☐ Outline

Surfaces  
 in  
 int fluid  
 out  
 point-velocities-1  
 wbot  
 wcyl  
 wtop

Shrink Factor: 0  
Feature Angle: 20

Surface Name Pattern:  Match

Surface Types  
 axis  
 clip-surf  
 exhaust-fan  
 fan

# Output data at points – method A

**Monitors**

Residuals, Statistic and Force Monitors

- Residuals - Print, Plot
- Statistic - Off
- Drag - Print, Write
- Lift - Print, Write
- Moment - Off

**Surface Monitors**

**Volume Monitors**

**Console:**

```
-pressure information is not available at the boundaries,
so it will be initialized with constant pressure
```

iter	scalar-0
1	1.000000e+00
2	1.296949e-05
3	3.106720e-06
4	1.140367e-06
5	4.338973e-07
6	1.932694e-07
7	8.666798e-08
8	3.936883e-08
9	1.946798e-08
10	8.842814e-09

hybrid initialization is done.

# Output data at points – method A

The screenshot shows the ANSYS FLUENT 13.0 interface. The **Surface Monitor** dialog box is open, showing the configuration for monitoring the X Velocity at a specific point. The **Name** is set to "surf-mon-1". The **Report Type** is set to "Vertex Average". The **Field Variable** is set to "Velocity..." and the **X Axis** is set to "X Velocity". The **Surfaces** list includes "in", "int\_fluid", "out", "point-velocities-1", "wbot", "wcyl", and "wtop". The **Options** section has "Print to Console" checked and "Plot" unchecked. The **File Name** is "surf-mon-1.out". The **Get Data Every** is set to "1" and "Iteration".

The console output shows the following text:

```

on is not available at the boundaries,
lized with constant pressure

scalar-0
1.000000e+00
1.296949e-05
3.106720e-06
1.140367e-06
4.338973e-07
1.932694e-07
8.666798e-08
3.936883e-08
1.946798e-08
8.842814e-09
hybrid initialization is done.
  
```

An annotation "Point = Vertex ⇒ Average, Min and Max are the same!" points to the "Vertex Average" report type in the dialog box.

# Output data at points – method A

The screenshot shows the ANSYS FLUENT 13.0 interface. The left sidebar contains the 'Problem Setup' tree with 'Monitors' selected. The 'Monitors' panel is active, showing 'Residuals, Statistic and Force Monitors' and 'Surface Monitors'. The 'Surface Monitors' list includes 'surf-mon-1 - Vertex Average, X Velocity vs. Iteration'. The main window displays a 2D mesh of a rectangular domain with a circular hole, colored green. The bottom panel shows the following text:

```

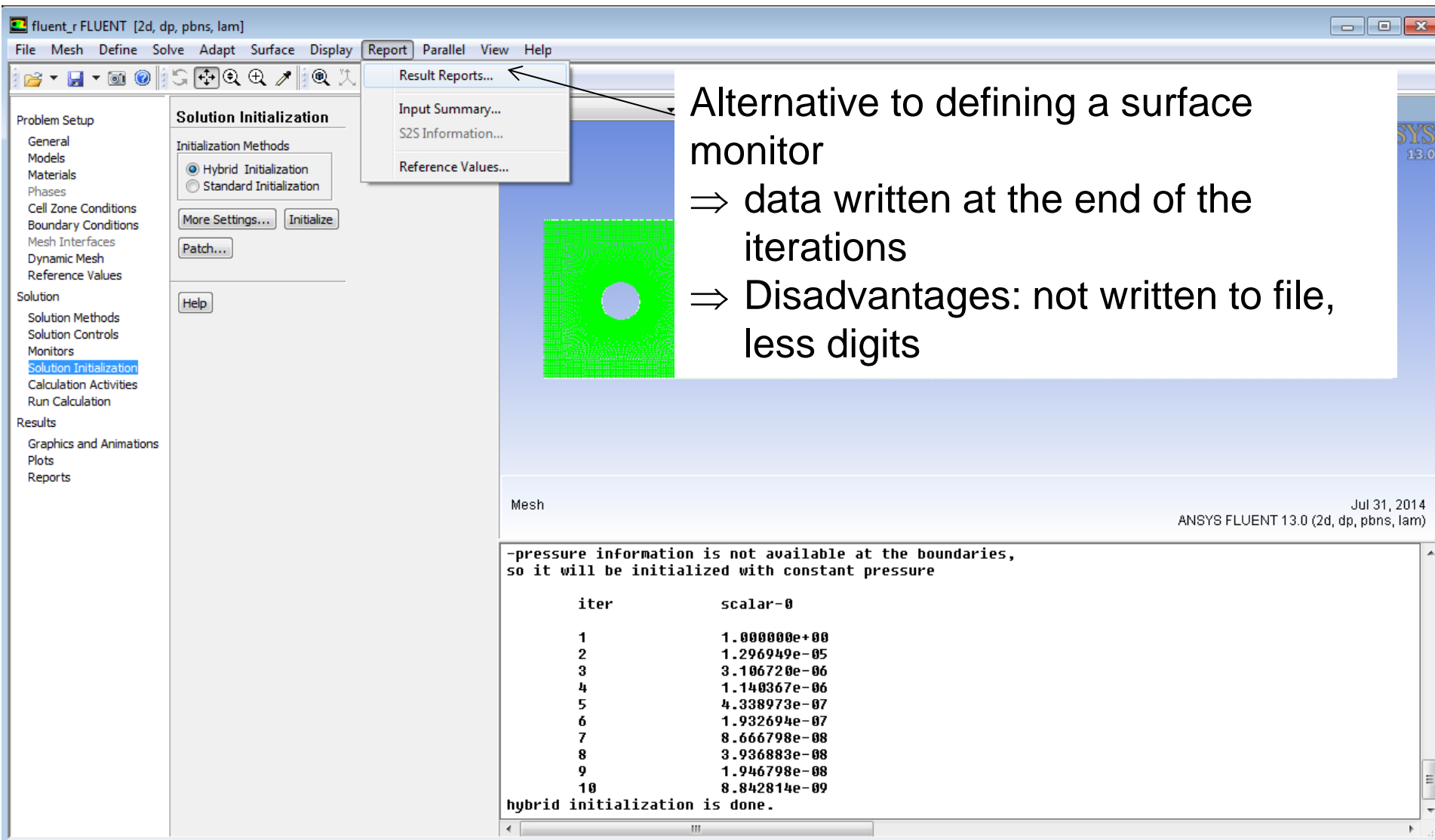
- pressure information is not available at the boundaries,
so it will be initialized with constant pressure

iter          scalar-0
1             1.000000e+00
2             1.296949e-05
3             3.106720e-06
4             1.140367e-06
5             4.338973e-07
6             1.932694e-07
7             8.666798e-08
8             3.936883e-08
9             1.946798e-08
10            8.842814e-09
hybrid initialization is done.
  
```

The bottom right corner of the interface shows the date 'Jul 31, 2014' and the version 'ANSYS FLUENT 13.0 (2d, dp, pbns, lam)'.



# Output data at points – method B



Alternative to defining a surface monitor

- ⇒ data written at the end of the iterations
- ⇒ Disadvantages: not written to file, less digits

Mesh

Jul 31, 2014  
ANSYS FLUENT 13.0 (2d, dp, pbns, lam)

```
-pressure information is not available at the boundaries,
so it will be initialized with constant pressure
```

iter	scalar-0
1	1.000000e+00
2	1.296949e-05
3	3.106720e-06
4	1.140367e-06
5	4.338973e-07
6	1.932694e-07
7	8.666798e-08
8	3.936883e-08
9	1.946798e-08
10	8.842814e-09

hybrid initialization is done.

# Output data at points – method B

Alternative to defining a surface monitor

- ⇒ data written at the end of the iterations
- ⇒ Disadvantages: not written to file, less digits

Surface Integrals

Report Type: Vertex Average

Field Variable: Velocity...

Surface Types: axis, clip-surf, exhaust-fan, fan

Phase: mixture

Surfaces: in, int\_fluid, out, point-velocities-1, wbot, wcyl, wtop

Average of Surface Vertex Values (m/s): 0.2000145

Save Output Parameter...

Compute Write... Close Help

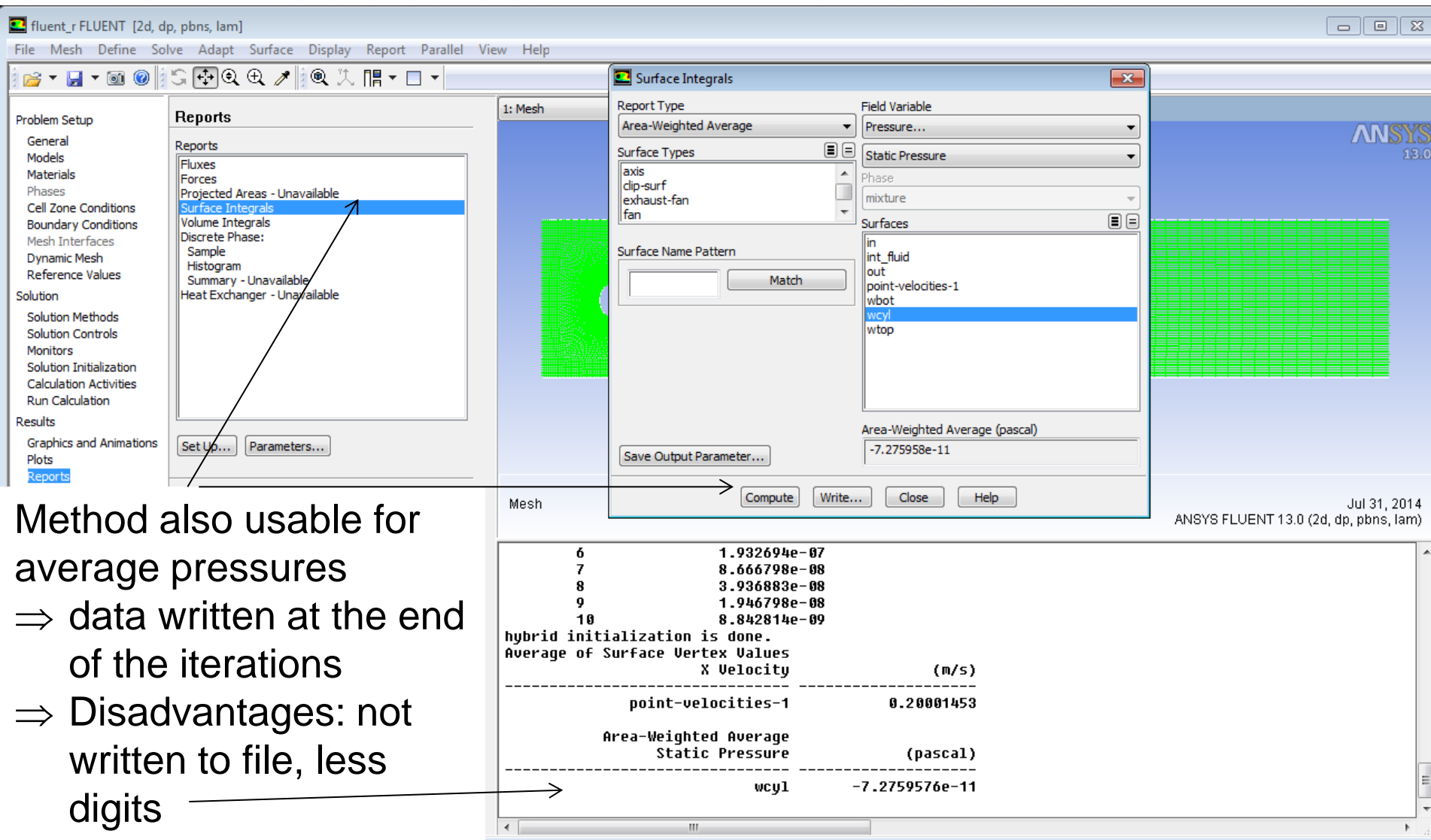
Mesh

hybrid initialization is done.

Average of Surface Vertex Values

	X Velocity	(m/s)
point-velocities-1	0.20001453	

# Output data at surfaces – method B



The screenshot shows the ANSYS FLUENT 13.0 interface. The **Surface Integrals** dialog box is open, showing the following settings:

- Report Type:** Area-Weighted Average
- Field Variable:** Pressure...
- Static Pressure:** Static Pressure
- Phase:** mixture
- Surfaces:** wcy1 (selected)
- Area-Weighted Average (pascal):** -7.275958e-11

The **Results** window shows the output data for the surface integral. The data is as follows:

Surface	Value
6	1.932694e-07
7	8.666798e-08
8	3.936883e-08
9	1.946798e-08
10	8.842814e-09

hybrid initialization is done.  
Average of Surface Vertex Values

Surface	X Velocity (m/s)
point-velocities-1	0.20001453

Area-Weighted Average Static Pressure (pascal)

Surface	Value
wcy1	-7.2759576e-11

Method also usable for average pressures  
 ⇒ data written at the end of the iterations  
 ⇒ Disadvantages: not written to file, less digits

# Output data at surfaces – method A

The screenshot shows the ANSYS FLUENT 13.0 interface. The **Monitors** panel on the left has **Monitors** selected. The **Surface Monitor** dialog box is open, showing the configuration for **surf-mon-2**. The **Report Type** is set to **Area-Weighted Average**, the **Field Variable** is **Pressure...**, and the **Static Pressure** is selected. The **Surfaces** list includes **wcyl**, which is highlighted. The **Options** section shows **Print to Console** and **Write** are checked. The **File Name** is **surf-mon-2.out**, and the **X Axis** is **Iteration**. The **Get Data Every** is set to **1** **Iteration**.

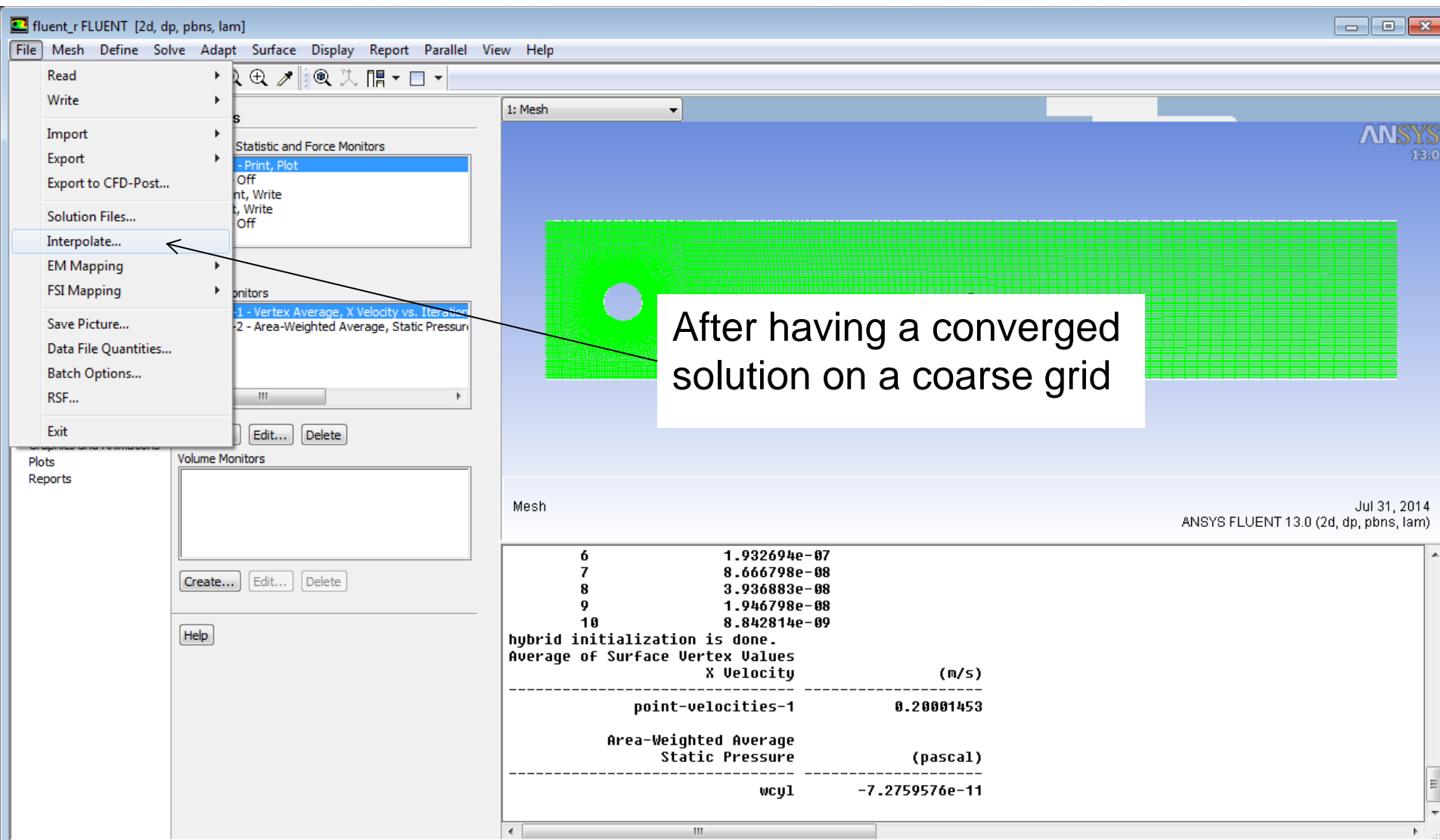
The console output at the bottom shows the following data:

```

6
7      8.666798e-08
8      3.936883e-08
9      1.946798e-08
10     8.842814e-09
hybrid initialization is done.
Average of Surface Vertex Values
      X Velocity                      (m/s)
-----
point-velocities-1                  0.20001453
Area-Weighted Average
      Static Pressure                (pascal)
-----
wcyl                               -7.2759576e-11
  
```

But for pressures also  
surface monitor possible

# Interpolation of results from coarser grids



After having a converged solution on a coarse grid

Mesh

Jul 31, 2014  
ANSYS FLUENT 13.0 (2d, dp, pbns, lam)

6	1.932694e-07
7	8.666798e-08
8	3.936883e-08
9	1.946798e-08
10	8.842814e-09

hybrid initialization is done.

Average of Surface Vertex Values

	X Velocity (m/s)
point-velocities-1	0.20001453

Area-Weighted Average

	Static Pressure (pascal)
wcyl	-7.2759576e-11

# Interpolation of results from coarser grids

fluent\_r FLUENT [2d, dp, pbns, lam]

File Mesh Define Solve Adapt Surface Display Report Parallel View Help

Problem 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

**Monitors**

Residuals, Statistic and Force Monitors

- Residuals - Print, Plot
- Statistic - Off
- Drag - Print, Write
- Lift - Print, Write
- Moment - Off

Edit...

Surface Monitors

- surf-mon-1 - Vertex Average
- surf-mon-2 - Area-Weighted

Create... Edit... Delete

Volume Monitors

Create... Edit... Delete

Help

**Interpolate Data**

Options

- ☐ Read and Interpolate
- ☒ Write Data

Fields

- Pressure
- X Velocity
- Y Velocity

Cell Zones

- ads-0
- ads-1
- fluid

Write Close Help

Write results to interpolation file

Jul 31, 2014  
ANSYS FLUENT 13.0 (2d, dp, pbns, lam)

```

6          1.932694e-07
7          8.666798e-08
8          3.936883e-08
9          1.946798e-08
10         8.842814e-09
hybrid initialization is done.
Average of Surface Vertex Values
      X Velocity                      (m/s)
-----
point-velocities-1                   0.20001453
Area-Weighted Average
      Static Pressure                 (pascal)
-----
wcy1                                -7.2759576e-11
  
```

# Interpolation of results from coarser grids

The screenshot shows the ANSYS FLUENT 13.0 interface. The 'Interpolate Data' dialog box is open, with 'Read and Interpolate' selected under 'Options'. The 'Fields' list includes Pressure, X Velocity, and Y Velocity. The 'Cell Zones' list includes 'fluid'. The 'Read...' button is highlighted. The console output at the bottom shows the results of the interpolation process.

**Interpolate Data Dialog Box:**

- Options:**
  - ☒ Read and Interpolate
  - ☐ Write Data
- Fields:**
  - Pressure
  - X Velocity
  - Y Velocity
- Cell Zones:**
  - ads-0
  - ads-1
  - fluid

**Console Output:**

```

6          1.932694e-07
7          8.666798e-08
8          3.936883e-08
9          1.946798e-08
10         8.842814e-09
hybrid initialization is done.
Average of Surface Vertex Values
      X Velocity                      (m/s)
-----
point-velocities-1                0.20001453
Area-Weighted Average
      Static Pressure                (pascal)
-----
wcyl                             -7.2759576e-11
  
```

- Then set up the case with the next finer grid
- Initialize
- Load the saved interpolation file
- Start your calculation

# Boundary condition moving wall

The screenshot shows the 'Wall' dialog box in ANSYS Fluent. The 'Zone Name' is 'wbot1' and the 'Adjacent Cell Zone' is 'fluid'. The 'Momentum' tab is selected. Under 'Wall Motion', the 'Moving Wall' radio button is selected. In the 'Motion' section, 'Relative to Adjacent Cell Zone' is selected, and the 'Speed (m/s)' is set to 0. Under 'Direction', 'Translational' is selected, and the 'X' direction is set to 1 and the 'Y' direction is set to 0. Under 'Shear Condition', 'No Slip' is selected. Under 'Wall Roughness', the 'Roughness Height (m)' is 0 and the 'Roughness Constant' is 0.5, both with 'constant' dropdown menus. The 'OK', 'Cancel', and 'Help' buttons are at the bottom.

Zone Name  
wbot1

Adjacent Cell Zone  
fluid

Momentum Thermal Radiation Species DPM Multiphase UDS Wall Film

Wall Motion  
☐ Stationary Wall  
☒ Moving Wall

Motion  
☒ Relative to Adjacent Cell Zone  
☐ Absolute

Speed (m/s)  
0

Direction  
☒ Translational  
☐ Rotational  
☐ Components

X 1  
Y 0

Shear Condition  
☒ No Slip  
☐ Specified Shear  
☐ Specularity Coefficient  
☐ Marangoni Stress

Wall Roughness  
Roughness Height (m) 0 constant  
Roughness Constant 0.5 constant

OK Cancel Help